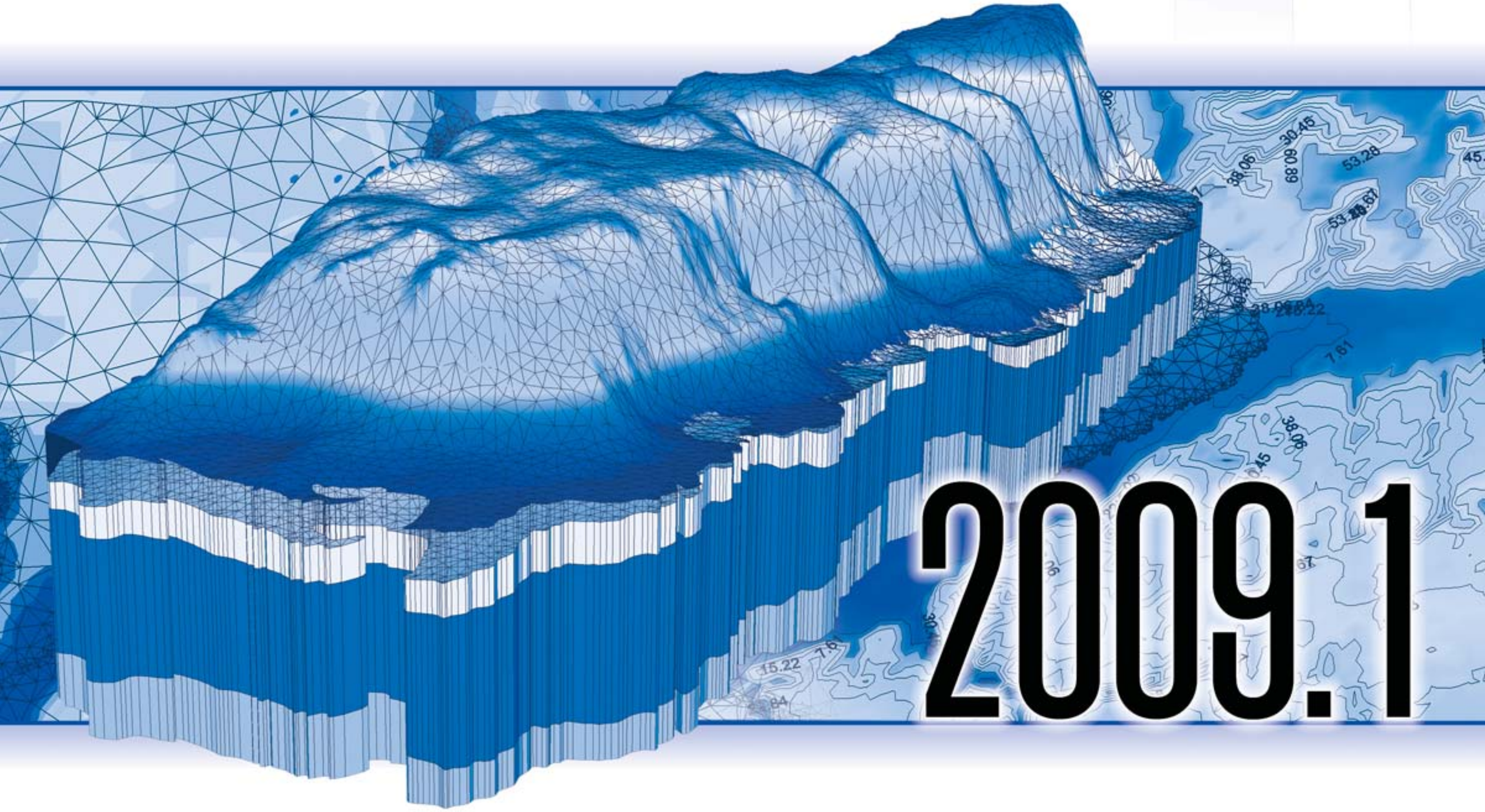
Schlumberger
WATER SERVICES

Demo Tutorial



Hydro GeoBuilder

A flexible, simulator-independent, hydrogeological modeling environment

Hydro GeoBuilder - Demonstration Exercise

Introduction

Introduction

This tutorial provides step-by-step instructions on how to create a simple groundwater flow conceptual model using Hydro GeoBuilder. In this tutorial you will,

- Import data from raw Excel files, Shapefiles and XYZ surface files
- Create surfaces by interpolating XYZ point data
- Visualize imported data in three-dimensions
- Create a new conceptual model
- Define geologic horizons and structural zones using surfaces
- Create conductivity property zones and define initial heads
- Create a Type 4 (Well) boundary condition and recharge boundary condition
- Create a finite element mesh
- Define slice elevations using a deformed mesh type
- Translate the conceptual model to a FEFLOW .FEM file.

Project

The data used in this tutorial is based on the data set used in the FEFLOW demonstration exercise. The dataset includes various files that contain data about a fictitious groundwater flow system in an area north of Friedrichshagen, Germany. A completed version of the conceptual model is copied to your computer during the Hydro GeoBuilder installation to the following directory:

```
C:\Program Files\Hydro GeoBuilder\Hydro GeoBuilder
Projects\Demo Project - Friedrichshagen\suppfiles
```

You can open this project in Hydro GeoBuilder by going to **File** > **Open**, and selecting the **Friedrichshagen.amd** file, located in the above directory.

Note: If you chose to install Hydro GeoBuilder to a non-default location on your computer, please look in this directory for the Friedrichshagen project.

Terms and Notations

For the purposes of this tutorial, the following terms and notations will be used:

Type: - type in the given word or value

⇔ - press the Tab key on your keyboard

<Enter> - press the Enter key on your keyboard

☞ - click the left mouse button where indicated

☞☞ - double-click the left mouse button where indicated

The **bold faced type** indicates menu or window items to click on or values to type in.

[...] - denotes a button to click on, either in a window, or in the side or bottom menu bars.

Starting Hydro GeoBuilder

Before proceeding, please ensure that Hydro GeoBuilder software is installed on your computer along with a valid software license. For more information on installation and licensing, please see the Hydro GeoBuilder Getting Started guide, located in the CD booklet, or on the installation CD.

To start **Hydro GeoBuilder**,

☞ **Hydro GeoBuilder** shortcut from your desktop.

Alternatively, you can start Hydro GeoBuilder by selecting **Start > Programs > SWS Software > Hydro GeoBuilder > Hydro GeoBuilder**.

The Hydro GeoBuilder **main window** will appear on your screen. The various components of the main window are labeled in the following figure.

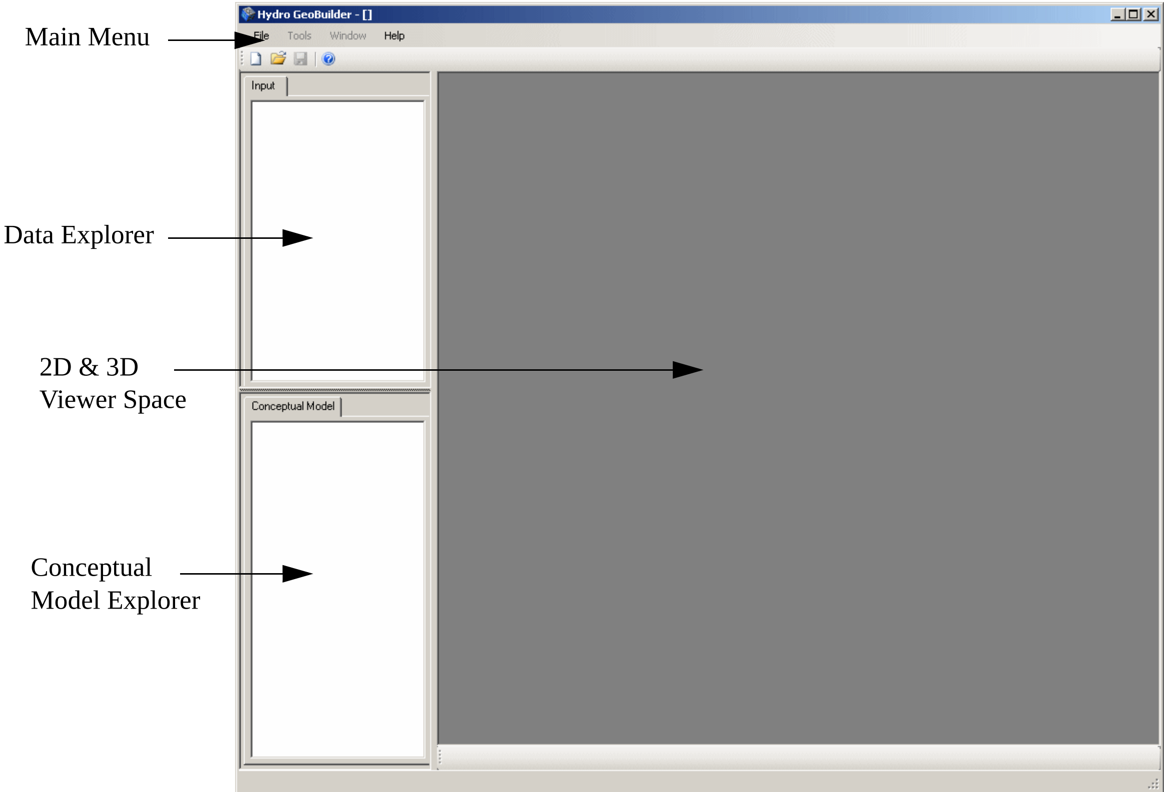


Figure 1 Labeled Hydro GeoBuilder main window

Creating a New Project

First you will create a new Hydro GeoBuilder project, and define the related project settings, including the data repository folder, coordinate system, and default unit settings.

To create a new project,

☞ **File > New > Project...** from the main menu.

The **Create Project** dialog box will appear on your screen.

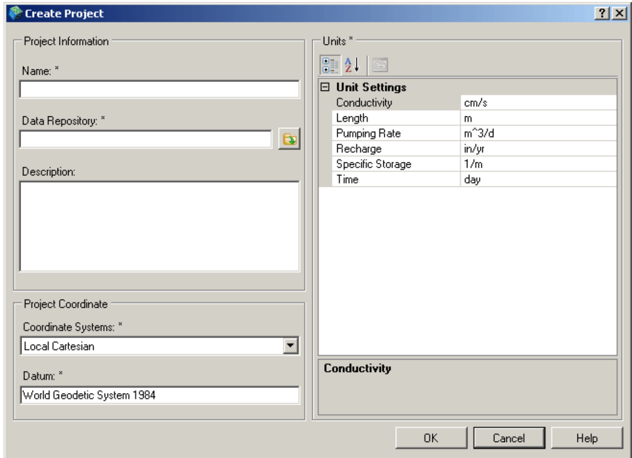


Figure 2 Create Project dialog box

Enter the following information in the **Create Project** dialog box.

In the **Name** field,

type: **Demonstration Exercise**

Beside the **Data Repository** field,

☞ **Browse for Folder** button

The **Browse for Folder** dialog box will appear.

☞ **[+]** button, located beside **Documents**

☞ **Documents**

☞ **Make New Folder** button

type: **Demonstration Exercise**

<Enter>

☞ **OK** button

For the **Project Coordinate System,** you will keep the default setting and use a local cartesian coordinate system for building the conceptual model.

In the **Unit Settings**, for **Conductivity**,

☞ **m/s** from the combo box

For **Recharge**,

☞ **m/yr** from the combo box

You will keep the default settings for the remaining units.

☞ **OK** button

Importing Data

Hydro GeoBuilder supports importing data from various standard raw data types to allow flexibility in building and interpreting your conceptual model. Data can be imported and used in many ways; GIS data can be used to delineate and visualize geometry of structural zones, horizons and features of your conceptual model, while attribute data can be used for assigning properties to structural zones and attributes to boundary conditions.

For an overview of the supported data types and how they can be used in HGB, please refer to “Table 1: Supported Data Types in Hydro GeoBuilder” on page 57.

Importing a Site Map

First you will import a site map of the modeling area from a JPEG file.

To import a site image,

☞ Right-click in the **Data Explorer**

☞ **Import Data...**, from the pop-up menu

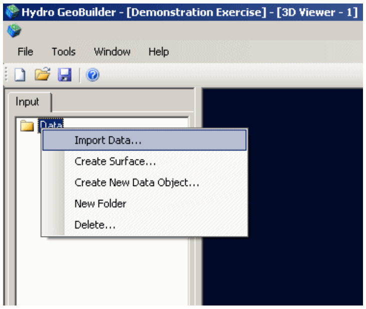


Figure 3 Importing data

The **Data Import** wizard will appear on your screen (Figure 4).

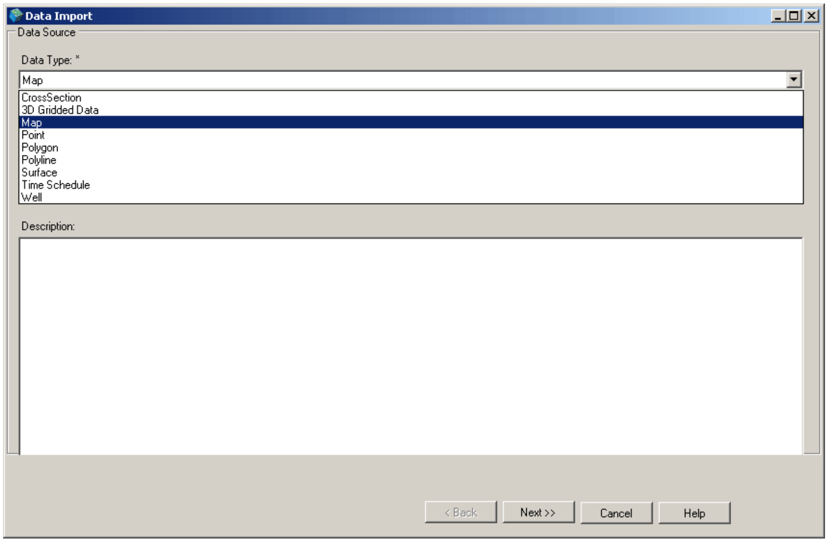


Figure 4 Import map: Selecting the data source

From the **Data Type** combo box,

☞ **Map**

Beside the **Source File** field,

☞ **[...]** button

The **Select Data Source file!** dialog box will appear on your screen. The supporting files for this tutorial can be found in the Hydro GeoBuilder installation folder. If you changed the path during installation, please look in this directory for these files. If you chose the default path, navigate to the following directory,

```
C:\Program Files\Hydro GeoBuilder\Hydro GeoBuilder Projects\Demo
Project - Friedrichshagen\suppfiles\Images
```

☞ **SimulationArea.jpg**

☞ **Open** button

All images must be georeferenced before they can be used in Hydro GeoBuilder. If an image has not been georeferenced, you will be prompted to do so during the import process. The **SimulationArea.jpg** image has already been georeferenced for you. For more information on georeferencing raster images, please refer to the Hydro GeoBuilder User's Manual.

In the **Data Import** dialog,

☞ **Next** >> button

The next screen allows you to select the coordinate system of the georeferenced raster image. If the coordinate system of the image is different than that of your project, Hydro GeoBuilder will automatically perform a geotransformation to ensure the image is displayed correctly. Because this project uses a Local Cartesian coordinate system, no conversion is required.

☞ **Next** >> button

A preview of the site map will appear on your screen, along with the georeferenced map coordinates (Figure 5).

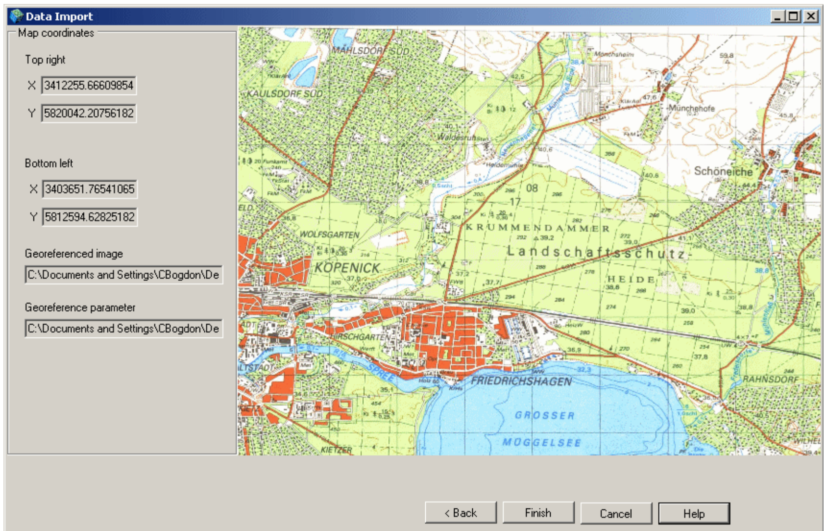


Figure 5 Preview map and view georeferenced map coordinates

☞ **Finish** button

The imported map will now be listed as a **Data Object** in the **Data Explorer** (Figure 6). A **Data Object** refers to any data set or data element that has been imported, or created manually using the 2D drawing tools.

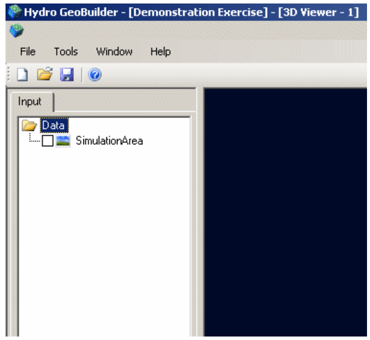


Figure 6 SimulationArea added to Data Explorer

By right-clicking on the **SimulationArea** data object, you can view the data in a *2D Viewer, 3D Viewer, load the data object settings* or *delete* the data object from the project (Figure 7).

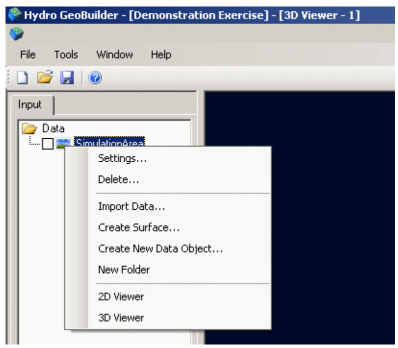


Figure 7 Right-click on Data Object to access various options

By default, the raster image is assigned an elevation value of 0. However, using the data object settings, you can assign a new constant elevation, or drape the image over a surface data object to visualize relief. For more information on data object settings, please refer to the Hydro GeoBuilder User's Manual.

You will now view the imported map data object in the 3D Viewer window.

In the **Data Explorer**,

☞ ☐ beside the **SimulationArea** data object

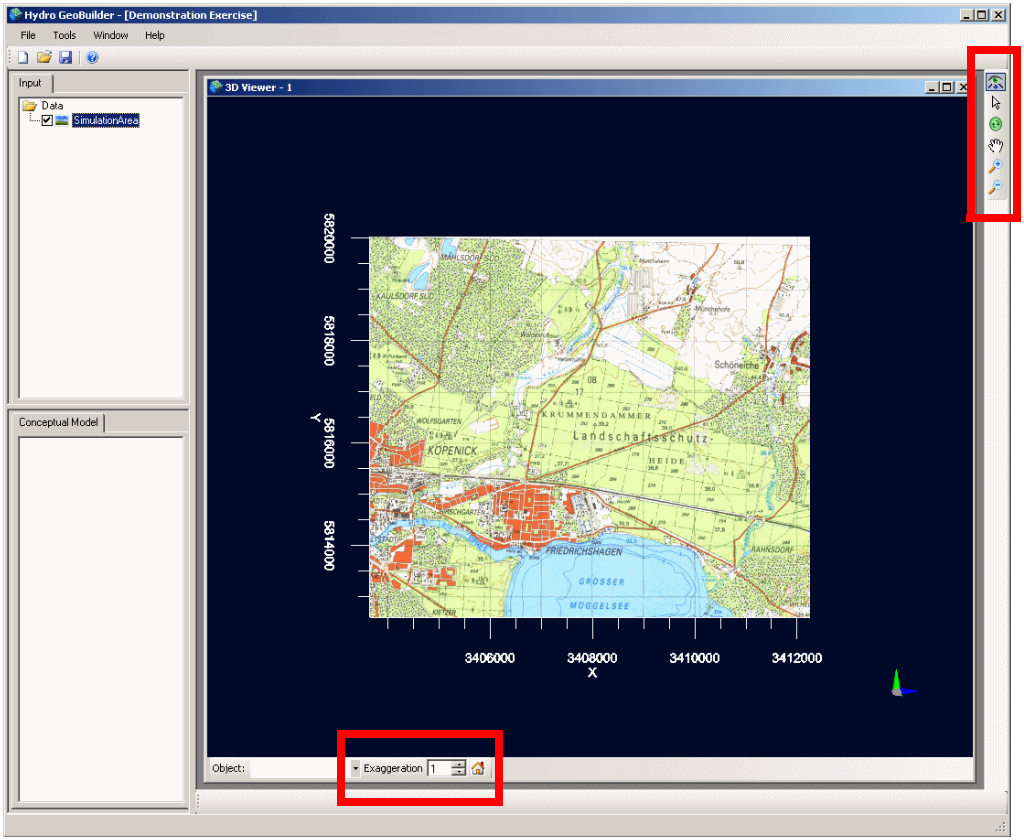


Figure 8 Show map data object in 3D Viewer window

You can **Rotate** the map by clicking anywhere inside the viewer window and dragging the mouse. You can zoom in and zoom out using the scroll wheel on your mouse, or by using the  **Zoom In** and  **Zoom Out** buttons located along the right side of the main window.

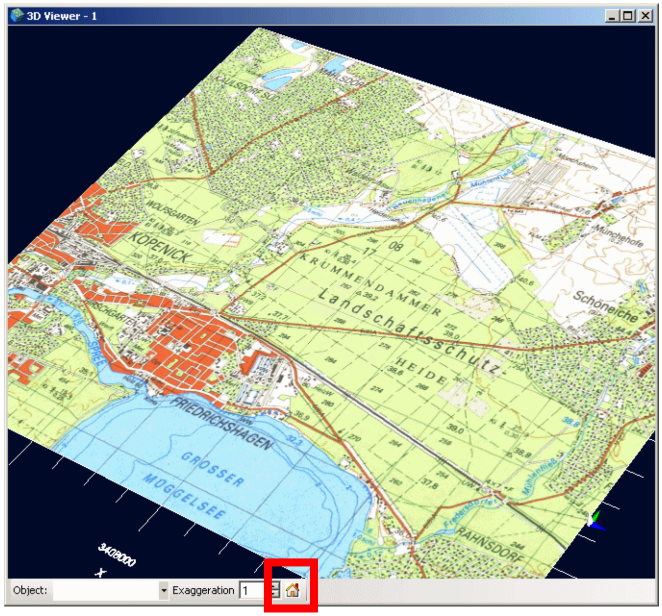


Figure 9 Zoom in and rotate map data object in 3D Viewer window

You can reset the viewer configuration back to planar view by selecting the **Reset Screen Position** button (indicated in Figure 9), located along the bottom of the 3D Viewer window.

Importing the Model Boundary

The model boundary, *i.e.*, the X-Y extents of the model, will be defined using an imported Shapefile (*.SHP) polygon. To import a polygon shapefile,

☞ Right-click in the **Data Explorer**

☞ **Import Data...**, from the pop-up menu (Figure 3)

The **Data Import** dialog box will appear on your screen (Figure 4).

From the **Data Type** combo box,

☞ **Polygon**

Beside the **Source File** field,

☞ **[...]** button

The **Select Data Source file!** dialog box will appear on your screen. Navigate to the following directory on your computer.

```
C:\Program Files\Hydro GeoBuilder\Hydro GeoBuilder Projects\Demo
Project - Friedrichshagen\suppfiles\Polygons
```

Note: If you did not install HGB to the default directory, please look for a similar path in the chosen installation directory.

- ☞ **Model_area_ply.shp** file
- ☞ **Open** button

From the **Data Import** dialog box,

- ☞ **Next>>** button

The geographic information of the source data will appear on your screen.

- ☞ **Next>>** button

The **Data Mapping** screen will appear. No data mapping is required for this data.

- ☞ **Next>>** button

The final step in the import process checks the source file for errors and invalid data. No errors or warnings were detected.

- ☞ **Finish** button

The imported polygon data object will now appear in the **Data Explorer**, along with the imported map (Figure 10).

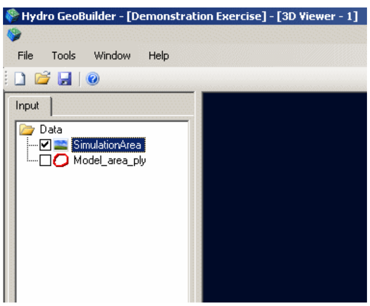


Figure 10 Imported polygon in Data Explorer

You may view the polygon in the opened **3D Viewer** window by selecting the ☐ check box beside the **Model_area_ply** data object.

By default, the imported polygon is assigned an elevation value of 0. However, using the data object settings, you can define a new constant elevation, or drape the polygon over a surface data object to visualize relief. For more information on data object settings, please refer to the Hydro GeoBuilder User's Manual.

Importing Borehole Data

Next you will import borehole data from a Microsoft Excel (.XLS) spreadsheet. The spreadsheet contains the borehole XY location, ground surface elevations, well contacts elevations (elevations of contact points where the borehole intersects the bottom of geologic layers), and Kx values for each borehole in the model region. This data will be imported as a point data object, and will be used later to create surfaces for defining structural zones and property zones.

To import a point data object,

- ☞ Right-click in the **Data Explorer**
- ☞ **Import Data...**, from the pop-up menu (Figure 3)

The **Data Import** dialog box will appear on your screen (Figure 4).

From the **Data Type** combo box,

- ☞ **Point**

Beside the **Source File** field,

- ☞ **[...]** button

The **Select Data Source file!** dialog box will appear on your screen. Navigate to the following directory on your computer.

```
C:\Program Files\Hydro GeoBuilder\Hydro GeoBuilder Projects\Demo
Project - Friedrichshagen\suppfiles\Wells
```

- ☞ **boreholes.xls** file
- ☞ **Open** button

From the **Data Import** dialog box,

- ☞ **Next>>** button

A preview of the data in the source file will appear on your screen.

- ☞ **Next>>** button

The geographic information of the source data will appear on your screen.

- ☞ **Next>>** button

The data mapping screen will appear. In this step, you will map the fields in the source data to the required target fields in Hydro GeoBuilder.

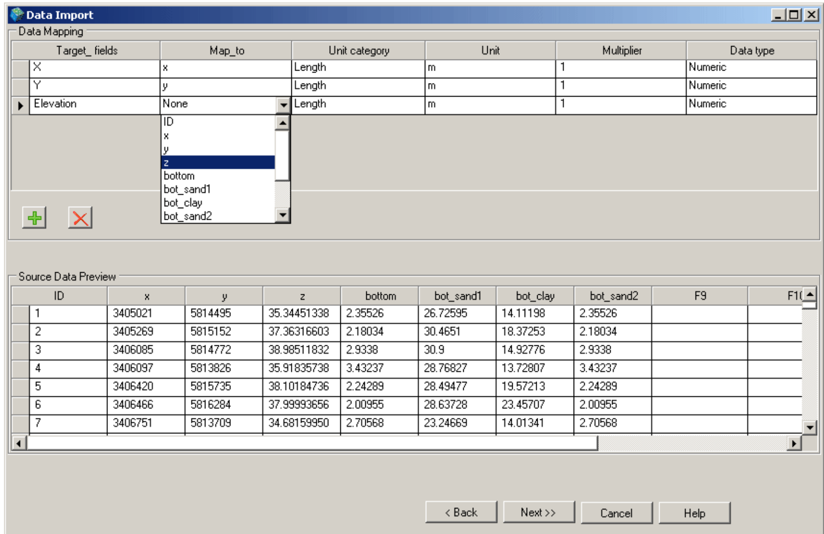


Figure 11 Mapping fields in source data to target fields

The **X** and **Y** fields are automatically mapped, however the **Elevation** field must be mapped manually.

Under the **Map_to** column, beside the **Elevation** field,

☞ **z,** from the combo box (Figure 11).

To include the remaining well tops, *i.e.*, **bot_sand1**, **bot_clay** and **bot_sand2**, and the conductivity values, *i.e.*, **Kx**, you will add new attributes.

☞ **Add new attribute** button

A new row will be added to the **Data Mapping** table.

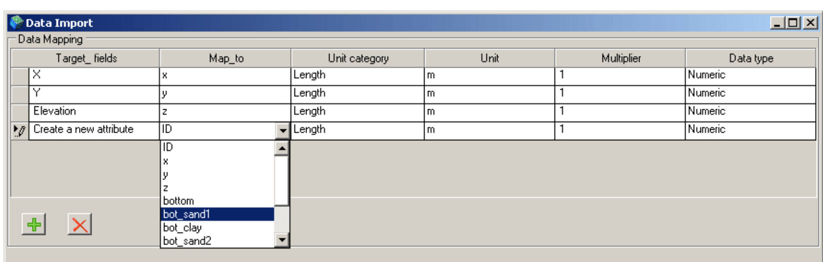


Figure 12 Creating and mapping a new attribute

Under the **Map_to** column, beside the **Create a new attribute** field,

☞ **bot_sand1,** from the combo box (Figure 12)

☞ **Add new attribute** button

A new row will be added to the **Data Mapping** table.

Under the **Map_to** column, beside the **Create a new attribute** field,

☞ **bot_clay,** from the combo box

☞  **Add new attribute** button

A new row will be added to the **Data Mapping** table.

Under the **Map_to** column, beside the **Create a new attribute** field,

☞ **bot_sand2,** from the combo box

Under the **Map_to** column, beside the **Create a new attribute** field,

☞ **Kx,** from the combo box

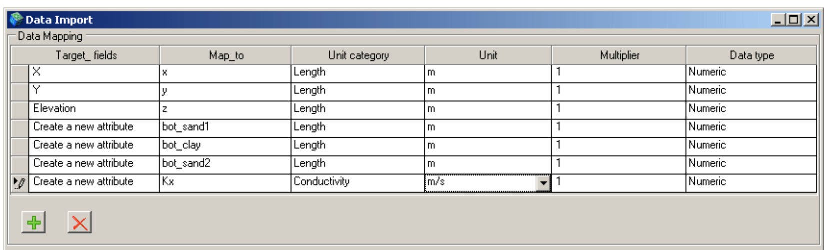
Beside the **Kx** field, under the **Unit Category** column,

☞ **Conductivity,** from the combo box

Beside the **Kx** field, under the **Unit** column,

☞ **m/s,** from the combo box

Your screen should look identical to the image shown below.



Target_ fields	Map_to	Unit category	Unit	Multiplier	Data type
X	x	Length	m	1	Numeric
Y	y	Length	m	1	Numeric
Elevation	z	Length	m	1	Numeric
Create a new attribute	bot_sand1	Length	m	1	Numeric
Create a new attribute	bot_clay	Length	m	1	Numeric
Create a new attribute	bot_sand2	Length	m	1	Numeric
Create a new attribute	Kx	Conductivity	m/s	1	Numeric

☞ **Next>>** button

The final step in the import process checks the source file for errors and invalid data. No errors or warnings were detected.

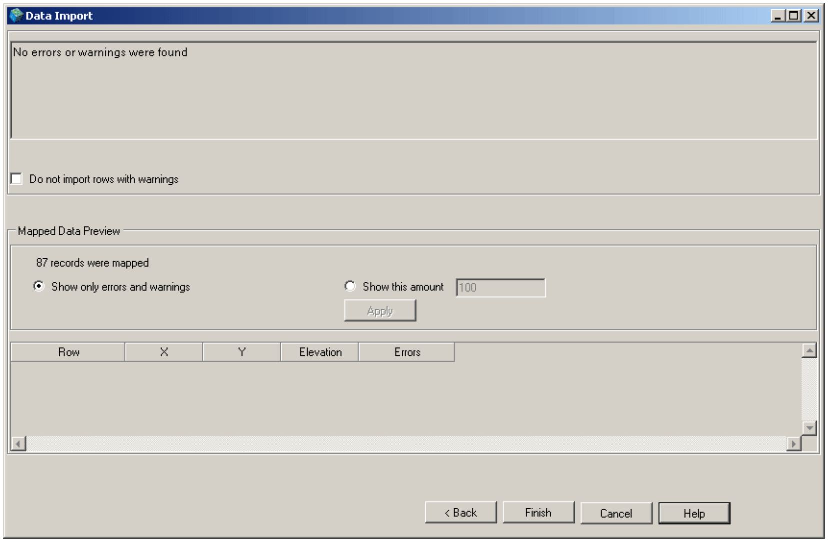


Figure 13 Data validation for mapped fields.

To complete the import process,

☞ **Finish** button

The imported point data object will now appear in the **Data Explorer** (Figure 14).

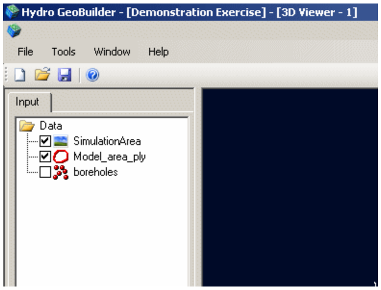


Figure 14 Imported point data object in data explorer

To view the data object in the **3D Viewer** window,

☞ ☐ Check box beside the **Boreholes** data object in the **Data Explorer**

Note: If you have closed the **3D Viewer -1** window, you can open a new viewer window by selecting **Window > New 3D Viewer**, from the Hydro GeoBuilder main menu.

The points will appear in the **3D Viewer - 1** window in the default color. To change the color of the points,

☞ right-click on the **Boreholes** data object in the **Data Explorer**.

☞ **Settings...** from the pop-up menu

The **Settings** window will appear on your screen (Figure 15).

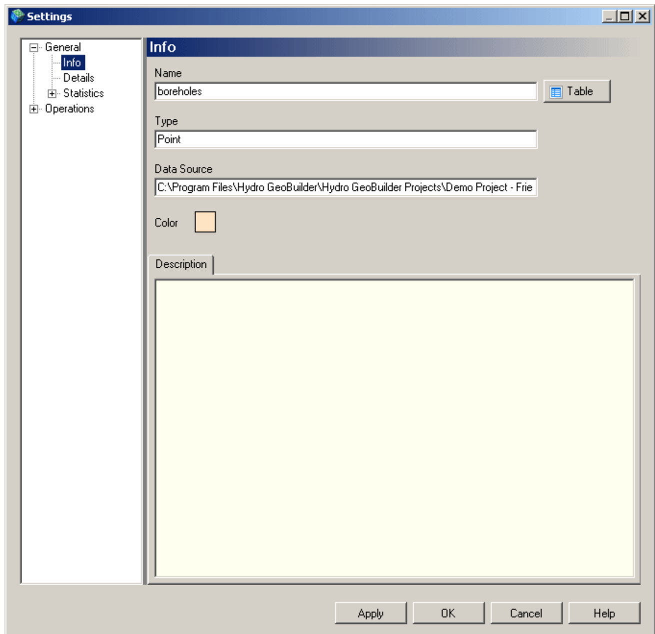


Figure 15   Data object settings window

☞ **Color** swatch

The **Color** box will appear on your screen.

☞ **Red**

☞ **OK** button

☞ **OK** button, to close the **Settings** window.

The points will now appear red in the **3D Viewer -1** window. To gain a better vertical perspective of the data, you may wish to increase the **Vertical Exaggeration** value. To do so,

In the **Vertical Exaggeration** box, located at the bottom of the viewer (Figure 16),

type: **30**

<Enter>

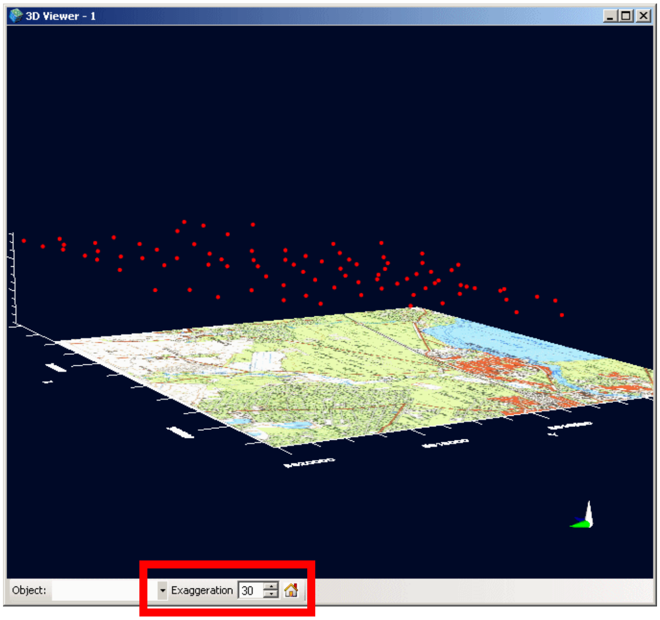


Figure 16 Boreholes data object displayed in 3D Viewer window over site map with increased vertical exaggeration.

Feel free to take some time to zoom in, zoom out, pan and rotate the displayed points data as desired.

Importing Pumping Wells

Two pumping wells are located in the bottom half of the model area. The pumping well data, including XYZ locations, well depth, screen details and pumping schedule data are contained in an Excel (.XLS) spreadsheet. You will import this data and use it later to define a Type 4 (Well) boundary condition. To import well data,

☞ Right-click in the **Data Explorer**

☞ **Import Data...**, from the pop-up menu (Figure 3)

The **Data Import** dialog box will appear on your screen (Figure 4).

From the **Data Type** combo box,

☞ **Well**

Beside the **Source File** field,

☞ **[...]** button

The **Select Data Source file!** dialog box will appear on your screen. Navigate to the following directory on your computer:

C:\Program Files\Hydro GeoBuilder\Hydro GeoBuilder Projects\Demo Project - Friedrichshagen\suppfiles\Wells

☞ **pumpingwells.xls** file

☞ **Open** button

From the **Data Import** dialog box,

☞ **Next>>** button

A preview of the data in the source file will appear on your screen.

☞ **Next>>** button

Various well options will appear on your screen, allowing you to specify the type of wells being imported (Figure 17).

☞ **Well heads with the following data** radio button

☞ **Pumping Schedule** check box

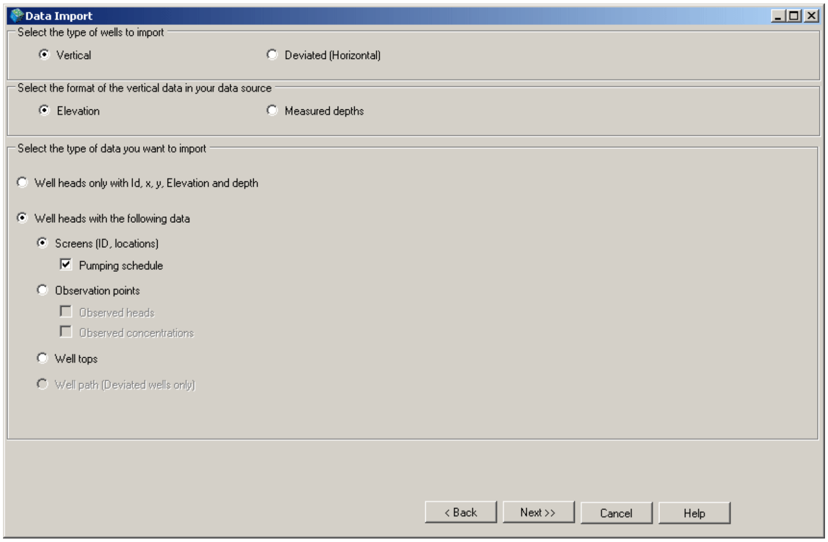


Figure 17 Well import options

Note: Currently, deviated/horizontal well translation is not supported for finite element models (FEFLOW).

☞ **Next>>** button

The geographic information of the source data will appear on your screen.

☞ **Next>>** button again

The data mapping screen will appear. In this step, you will map the fields in the source data to the required target fields in Hydro GeoBuilder.

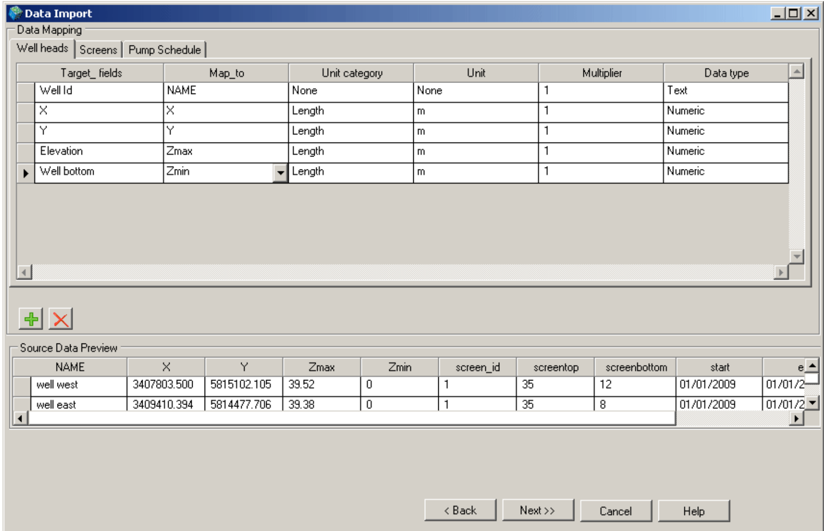


Figure 18 Mapping well heads

Under the **Map_to** column, beside the **Well Id** field,

☞ **NAME,** from the combo box

Under the **Map_to** column, beside the **Elevation** field,

☞ **Zmax,** from the combo box

Under the **Map_to** column, beside the **Well bottom** field,

☞ **Zmin,** from the combo box

Your screen should look similar to the image shown above (Figure 18).

Next you will map the fields for the well screen data.

☞ **Screens** tab (at the top of the window)

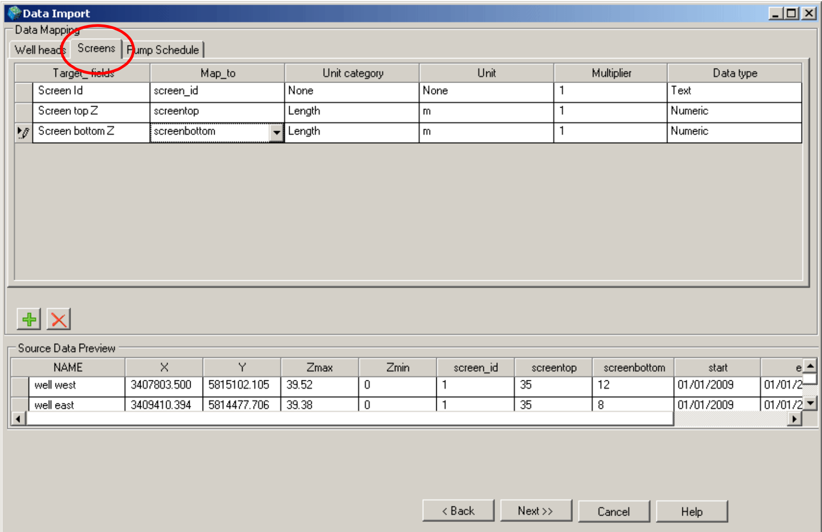


Figure 19 Mapping well screens

Under the **Map_to** column, beside the **Screen Id** field,

☞ **screen_id,** from the combo box

Under the **Map_to** column, beside the **Screen top Z** field,

☞ **screentop,** from the combo box

Under the **Map_to** column, beside the **Screen bottom Z** field,

☞ **screenbottom,** from the combo box

Your screen should look similar to the image shown above (Figure 19).

Next you will map the fields for the pumping schedule data.

☞ **Pump Schedule** tab

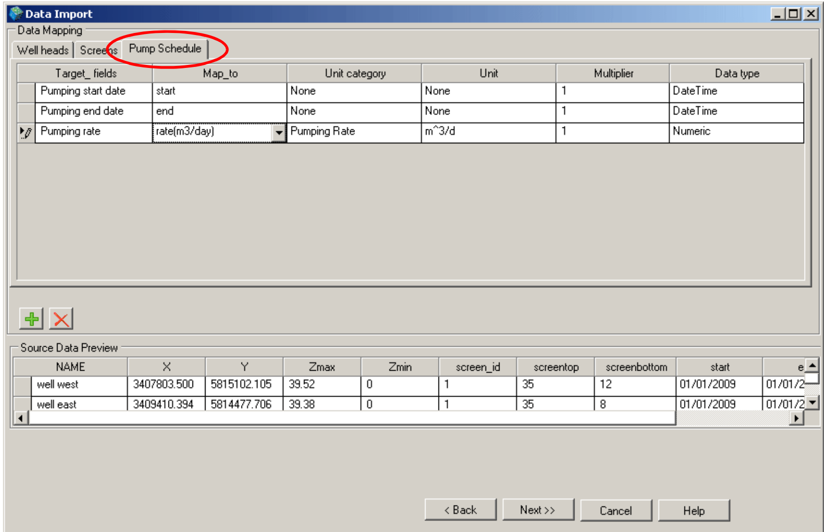


Figure 20 Mapping pumping schedule fields

Under the **Map_to** column, beside the **Pumping start date** field,

☞ **start,** from the combo box

Under the **Map_to** column, beside the **Pumping end date** field,

☞ **end,** from the combo box

Under the **Map_to** column, beside the **Pumping rate** field,

☞ **rate (m3/day)** from the combo box

Your screen should look similar to the image shown above (Figure 20).

☞ **Next>>** button

☞ **Finish** button

The well data object will now appear in the **Data Explorer**.

To view the data object in the **3D Viewer - 1** window,

☞ ☐ Checkbox beside the **pumpingwells** data object in the **Data Explorer**

Feel free to rotate, zoom in or out and pan the data as desired. Your viewer should look similar to the image shown below (Figure 21).

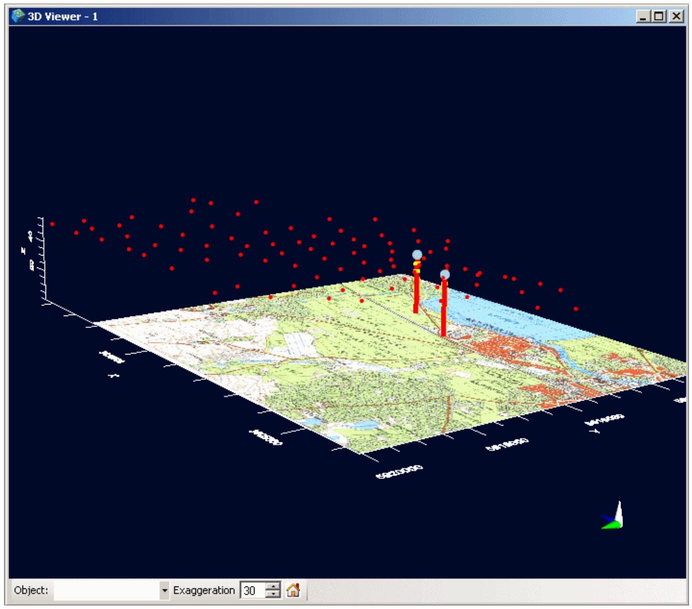


Figure 21 3D Viewer displaying pumping wells, well heads and sitemap

You have imported enough data to begin constructing the conceptual model. However, before doing so, you will create surfaces from the imported points data to represent the vertical boundaries of the geologic structures.

Creating Surfaces from Points

In Hydro GeoBuilder, a surface refers to a set of interpolated points that represent the spatial distribution of an attribute. Surfaces may represent changes in topography or geologic layers over space, e.g., digital elevation models, or the spatial distribution of properties such as conductivity, storage and initial heads.

Surfaces can be imported from surface files using the import utility, or they can be generated from **Point** data objects by interpolating XYZ point data.

In this section, you will interpolate the XYZ points data imported in the previous section (**Boreholes**) to generate surface data objects. These surface will be used later to define the horizons of the conceptual model.

Ground Surface

The first surface you will generate will be interpolated from the **Elevation** attribute in the Boreholes data object, and will represent the ground surface of the conceptual model. To generate a surface data object from points data,

☞ Right-click anywhere in the **Data Explorer** (Figure 3)

☞ **Create Surface,** from the pop-up menu

The **Create Surface** dialog box will appear on your screen (Figure 22)

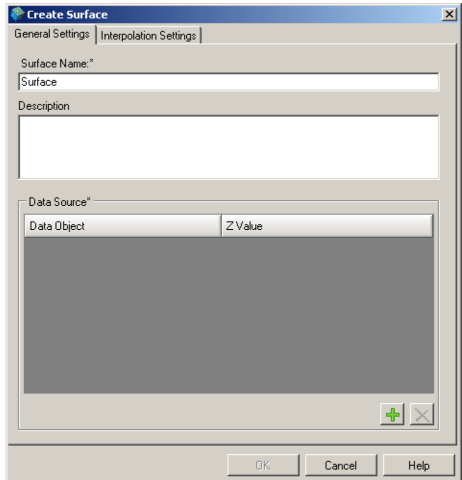


Figure 22 Create Surface dialog box

In the **Surface Name** text box,

type: **Ground Surface**

From the **Data Explorer**,

☞ **Boreholes** data object

At the bottom of the **Create Surface** dialog box,

☞ button

The **Boreholes** point data object will be added to the **Data Source** table in the **Create Surface** dialog box (Figure 23).

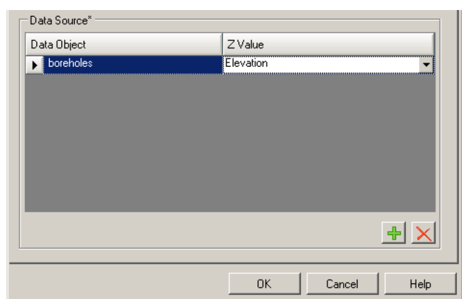


Figure 23 Add points data source

All the imported attributes for the boreholes data object are listed in the **Z Value** combo box. By default the **Elevation** attribute (ground surface elevation) is already selected.

Hydro GeoBuilder provides various interpolation methods for creating surfaces from points. To modify the interpolation method,

☞ **Interpolation Settings** tab

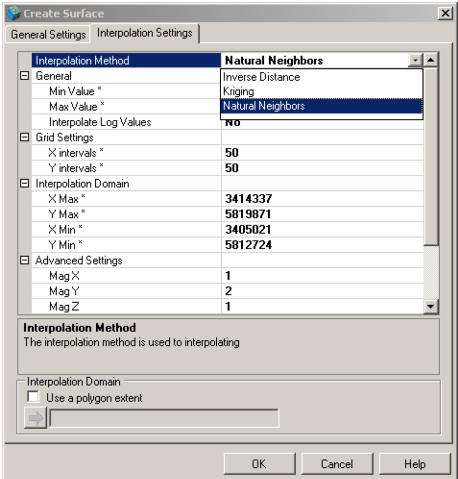


Figure 24 Selecting interpolation method

Beside the **Interpolation Method** field,

☞ **Natural Neighbors** from the combo box.

☞ **OK** button, to generate the surface

An **Information** message will appear on your screen,

☞ **OK** button

A new surface data object will appear in the **Data Explorer.**

To view the data object in the **3D Viewer - 1** window (Figure 25),

☞ ☐ Checkbox beside the **Ground Surface** data object in the **Data Explorer**

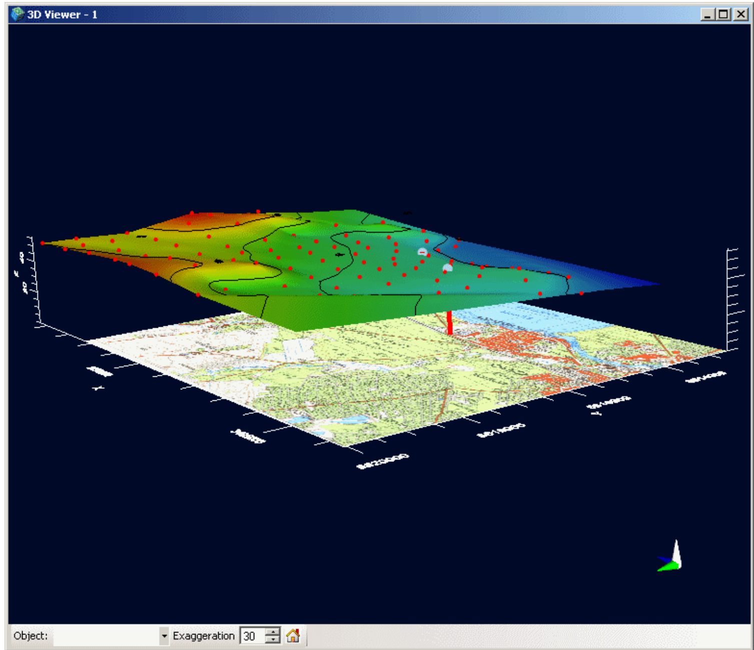


Figure 25 Ground surface displayed in 3D Viewer window

Next, you will repeat the steps above to generate surfaces for the **sand1**, **sand2** and **clay** layers.

Bottom of Sand1 Layer

The next surface you will generate will be interpolated from the **Bot_Sand1** attribute in the **Boreholes** data object, and will represent the bottom of the first layer in the conceptual model.

☞ Right-click anywhere in the **Data Explorer** (Figure 3)

☞ **Create Surface,** from the pop-up menu

The **Create Surface** dialog box will appear on your screen (Figure 22)

In the **Surface Name** text box,

type: **Bot_Sand1**

From the **Data Explorer**,

☞ **Boreholes** data object

At the bottom of the **Create Surface** dialog box,

☞ button

In the **Data Source** table,

☞ **bot_sand1** from the combo box

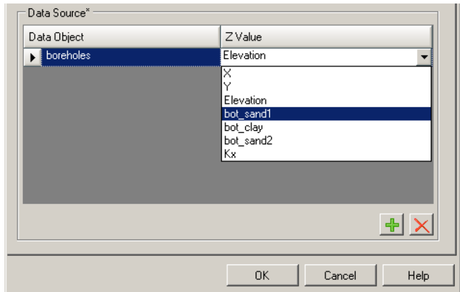


Figure 26 Select **bot_sand1** as Z Value data source

☞ **Interpolation Settings** tab

The interpolation settings will appear on your screen (Figure 24)

Beside the **Interpolation Method** field,

☞ **Natural Neighbors** from the combo box.

☞ **OK** button, to generate the surface

An **Information** message will appear on your screen,

☞ **OK** button

A new surface data object will appear in the **Data Explorer**.

Bottom of Clay Layer

The next surface you will generate will be interpolated from the **Bot_Clay** attribute in the **Boreholes** data object, and will represent the bottom of the second layer in the conceptual model.

☞ Right-click anywhere in the **Data Explorer** (Figure 3)

☞ **Create Surface,** from the pop-up menu

The **Create Surface** dialog box will appear on your screen (Figure 22)

In the S**urface Name** text box,

type: **Bot_Clay**

From the **Data Explorer**,

☞ **Boreholes** data object

At the bottom of the **Create Surfac**e dialog box,

☞ button

In the **Data Source** table,

☞ **bot_clay** from the combo box

☞ **Interpolation Settings** tab

The interpolation settings will appear on your screen (Figure 24)

Beside the **Interpolation Method** field,

☞ **Natural Neighbors** from the combo box.

☞ **OK** button, to generate the surface

An **Information** message will appear on your screen,

☞ **OK** button

A new surface data object will appear in the **Data Explorer**.

Bottom of Sand2 Layer

The last surface you will generate will be interpolated from the **Bot_Sand2** attribute in the **Boreholes** data object, and will represent the bottom of the third layer in the conceptual model.

☞ Right-click anywhere in the **Data Explorer** (Figure 3)

☞ **Create Surface,** from the pop-up menu

The **Create Surface** dialog box will appear on your screen (Figure 22)

In the S**urface Name** text box,

type: **Bot_Sand2**

From the **Data Explorer**,

☞ **Boreholes** data object

At the bottom of the **Create Surface** dialog box,

☞ button

In the **Data Source** table,

☞ **bot_sand2** from the combo box

☞ **Interpolation Settings** tab

The interpolation settings will appear on your screen (Figure 24)

Beside the **Interpolation Method** field,

☞ **Natural Neighbors** from the combo box.

☞ **OK** button, to generate the surface

An **Information** message will appear on your screen,

☞ **OK** button

A new surface data object will appear in the **Data Explorer**.

View the surfaces in the 3D Viewer window by selecting the check box beside each new surface in the **Data Explorer**.

☞ ☐ beside **Bot_Sand1**

☞ ☐ beside **Bot_Clay**

☞ ☐ beside **Bot_Sand2**

When rotated and zoomed in, your viewer should look similar to the image shown below.

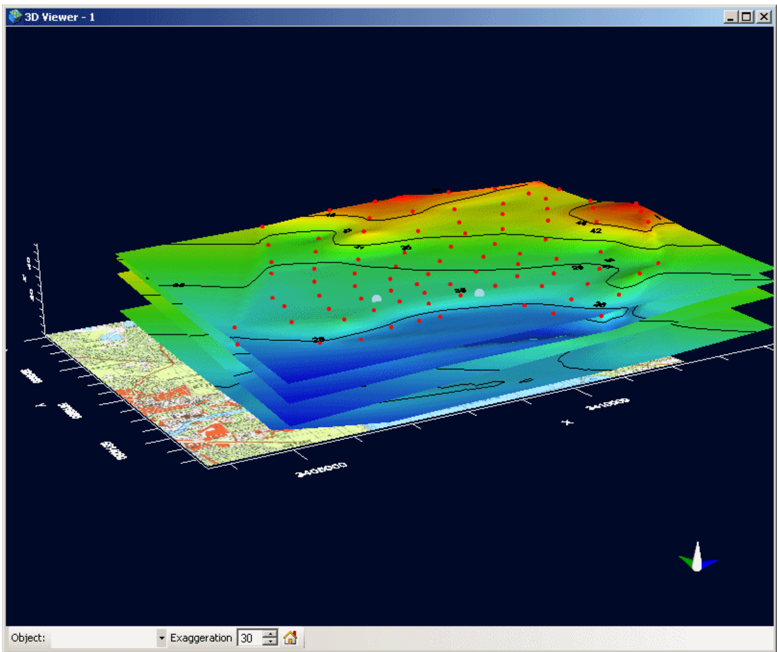


Figure 27 **Ground surface**, **bot_sand1**, **bot_sand2**, **bot_clay** surfaces displayed in 3D Viewer window

Creating a Conceptual Model

You now have the required data objects in the **Data Explorer** to begin constructing the conceptual model.

To create a conceptual model,

☞ **File > New > Conceptual Model...,** from the main menu (Figure 28).

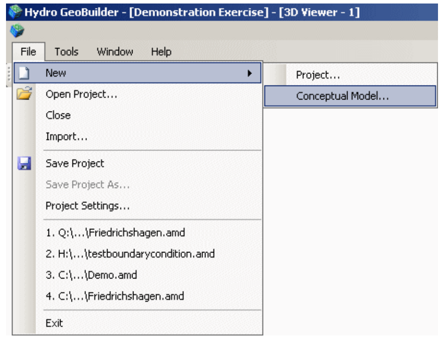


Figure 28 Create a new conceptual model

The **New Conceptual Model** dialog box will appear on your screen (Figure 29).

In the **Name** field,

type: **Demonstration**

You will use today's date as the **Start Date**, and therefore no change is required.

Next, you will define the **Model Area** using the **Model_Area_ply** data object from the **Data Explorer**.

Keeping the **New Conceptual Model** dialog box visible on your screen,

☞ **Model_Area_ply** from the **Data Explorer**

☞ Anywhere on the **New Conceptual Model** dialog box

☞ button, to insert the data object into the **Model Domain** field.

The **New Conceptual Model** dialog box on your screen should look identical to the one shown below (the Start date may be different).

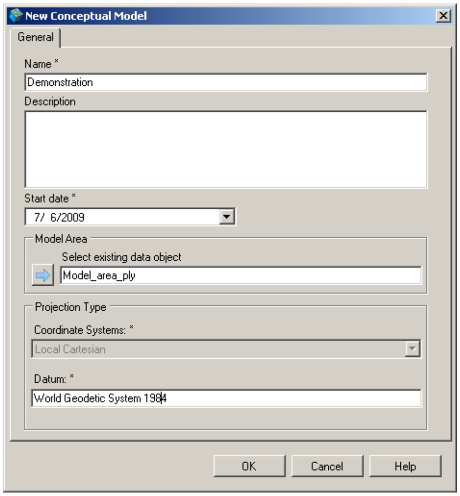


Figure 29 New conceptual model dialog box

☞ **OK** button

A collapsed **Demonstration** folder will be added to the **Conceptual Model Explorer**.

☞ **[+] button,** beside the Demonstration folder

A series of subfolders will display in a hierarchical "tree" structure. This tree sets up the workflow for defining the necessary components of the conceptual model.

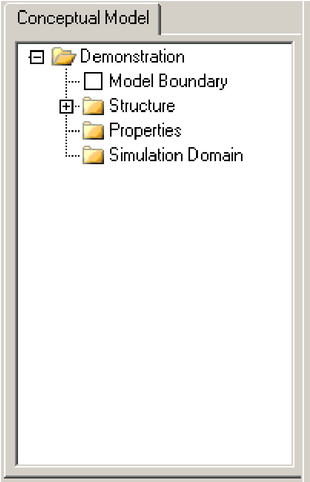


Figure 30 Conceptual Model tree

Defining Horizons

Horizons are stratigraphic layers (2D surfaces with topography) that define the upper and lower boundaries of the structural zones in a conceptual model. In Hydro GeoBuilder, horizons are created by clipping or extending interpolated surface data objects to the conceptual model area. At the same time, horizon rules are enforced so that layers do not intersect. This establishes layers that satisfy both FEFLOW and MODFLOW requirements. You will use the surfaces generated in the previous section (“Creating Surfaces from Points” on page 21) to define the conceptual model’s horizons.

From the conceptual model tree,

☞ Right-click on the **Structure** folder

☞ **Create Horizons** from the pop-up menu (Figure 31)

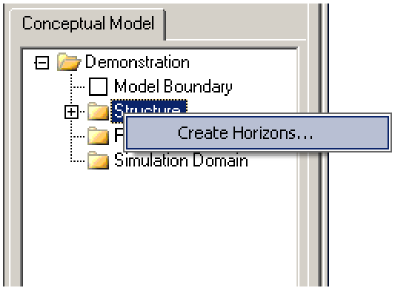


Figure 31 Create horizons

The **Horizon Settings** window will appear on your screen. In this window, you will specify which surface data objects to use to generate the horizons.

The conceptual model will have four horizons. Insert four rows in the **Horizon Settings** window.

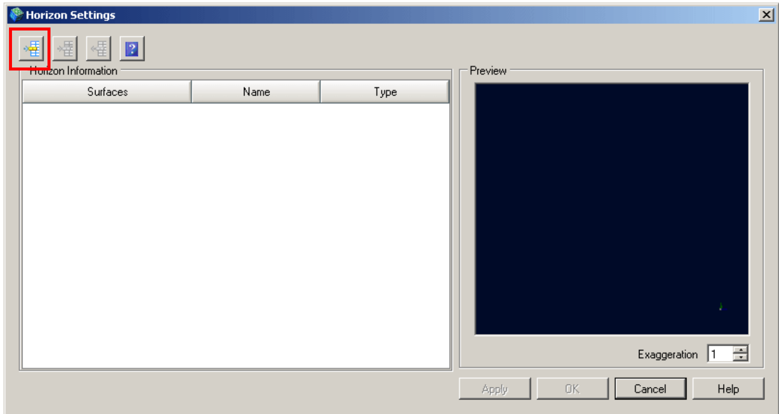


Figure 32 Horizon Settings window

- ☞  **Insert Row** button (Figure 32)
- ☞  **Insert Row** button, again
- ☞  **Insert Row** button, again
- ☞  **Insert Row** button, again

For the top horizon, you will use the **Ground Surface** surface data object. While keeping the **Horizon Settings** window open,

- ☞  **Ground Surface** surface data object from the **Data Explorer**

Back in the **Horizon Settings** window,

- ☞  button, to insert the **Data Object** for **Horizon 1**

Under the **Type** column,

- ☞ **Erosional**, from the combo box

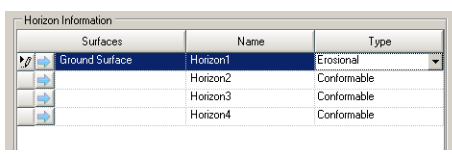


Figure 33 Adding surfaces and defining horizon type

Repeating the steps above, you will add the remaining surfaces, *e.g., bot_sand1, bot_clay and bot_sand2*, to the table.

From the **Data Explorer**,

- ☞ **Bot_Sand1** surface data object

Back in the **Horizon Settings** window,

☞ button, to insert the data object for **Horizon 2**

From the **Data Explorer**,

☞ **Bot_Clay** surface data object

Back in the **Horizon Settings** window,

☞ button, to insert the data object for **Horizon 3**

From the **Data Explorer**,

☞ **Bot_Sand2** surface data object

Back in the **Horizon Settings** window,

☞ button, to insert the data object for **Horizon 4**

Under the **Type** column,

☞ **Base** from the combo box for **Horizon 4**

☞ **Apply** button

The **Horizon Settings** window on your screen should look identical to the one shown below (Figure 34)

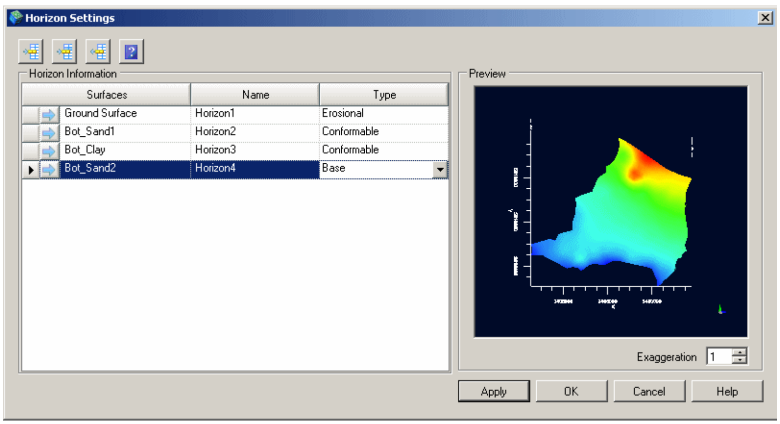


Figure 34 Defining horizons and viewing preview

A preview of the horizons will display in the adjacent 3D window. Feel free to inspect the horizons by rotating, zooming and moving the horizons using your mouse. You may also wish to increase the vertical exaggeration value to gain a better vertical perspective.

To accept the horizon settings,

☞ **OK** button

The horizons will be created and added to the **Conceptual Model** tree, under the **Horizons** folder. Along with the horizons, the structural zones between the horizons are automatically generated and added to the conceptual model tree.

Horizons and **Zones** may be viewed in a 3D Viewer by selecting the corresponding checkbox in the conceptual model tree. To view the zones, you will open a new 3D Viewer window.

From the Hydro GeoBuilder main menu,

☞ **Window > New 3D Window**

A **3D Viewer - 2** window will appear on your screen. Change the vertical exaggeration value of the viewer. In the **Exaggeration** box (located at the bottom of the viewer),

type: **35**

<enter>

From the **Conceptual Mode**l tree, under the **Zones** node,

☞ ☐ **Zone 1**

☞ ☐ **Zone 2**

☞ ☐ **Zone 3**

Note: Information about the structural zones can be viewed in the zone settings. To access these settings, right-click on the zone in the conceptual model tree and select **Settings...** . From here, you can view various *statistics*, including the zone volume and area, and you can change *style settings*, including transparency and color. For more information on zone settings, please refer to the Hydro GeoBuilder User's Manual.

Feel free to add other data objects to the viewer, *e.g.*, sitemap (SimulationArea), model area (Model_area_ply), or Wells (pumpingwells), and to rotate and zoom in on the data.

Your **3D Viewer Window** should look similar to the image shown below (Figure 35)

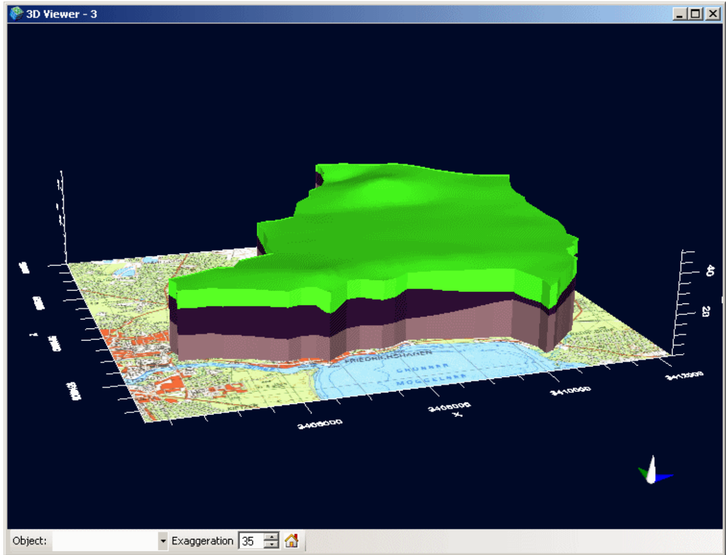


Figure 35 Displaying structural zones and site map in 3D Viewer

Defining Property Zones

This section will guide you through the steps to assign conductivity values to each structural zone in the conceptual model. Initial heads will also be defined.

First, you will start by defining the conductivity values for the Sand1 Layer (Upper Aquifer).

Sand1 Layer (Upper Aquifer)

The conductivity values for the upper aquifer will be defined using a surface data object. The surface will be created by interpolating the Kx attribute from the Boreholes data object.

☞ Right-click anywhere in the **Data Explorer** (Figure 3)

☞ **Create Surface,** from the pop-up menu

The **Create Surface** dialog box will appear on your screen (Figure 22)

In the **Surface Name** text box,

type: **Sand1_Kx**

From the **Data Explorer**,

☞ **Boreholes** data object

At the bottom of the **Create Surfac**e dialog box,

☞  button

In the **Data Source** table,

☞ **Kx** from the combo box (Figure 36)

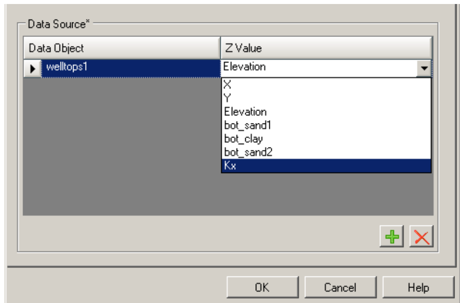


Figure 36 Select **Kx** as ZValue data source

☞ **Interpolation Settings** tab

The interpolation settings will appear on your screen (Figure 24)

Beside the **Interpolation Method** field,

☞ **Natural Neighbors** from the combo box.

☞ **OK** button, to generate the surface

An **Information** message will appear on your screen,

☞ **OK** button

Now you will define the property zone using the generated surface data object.

From the **Conceptual Mode**l tree,

☞ right-click on the **Properties** folder

☞ **Define Property Zone...** from the pop-up menu.

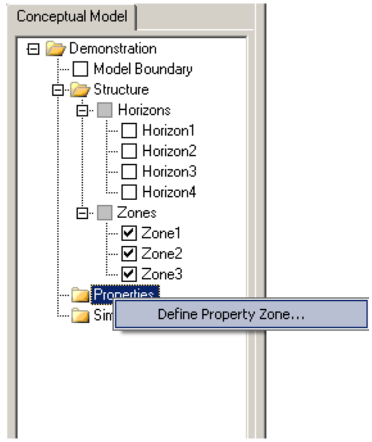


Figure 37 Define a new property zone

The **New Property Zone** dialog box will appear on your screen.

In the **Name** field,

type: **Upper Aquifer**

The property zone geometry will be defined using the **Zone 1** structural zone.

From the **Conceptual Model** tree,

☞ **Zone 1** (located under **Zones** node)

Back in the **New Property Zone** dialog,

☞  button

The bottom half of the **New Property Zone** dialog box should look similar to the image shown below (Figure 38).

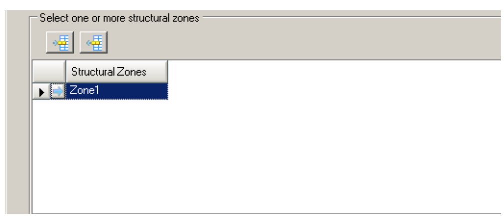


Figure 38 Select a structural zone for defining the property zone geometry

☞ **Next** >> button

The next step involves specifying the conductivity values for the property zone. For the upper aquifer you will use the surface **Sand1_Kx** to define the conductivity values.

In this case, the Kx, Ky, and Kz values will be the same, indicating the assigned property values are assumed horizontally and vertically isotropic.

Under the **Kx (m/s)** column, in the **Method** row,

☞ **Use Surface,** from the combo box.

Under the **Ky (m/s)** column, in the **Method** row,

☞ **Use Surface,** from the combo box.

Under the **Kz (m/s)** column, in the **Method** row,

☞ **Use Surface,** from the combo box.

Your screen should look similar to the image shown below (Figure 39).

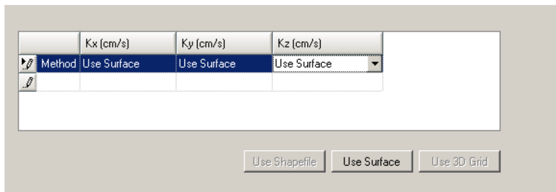


Figure 39 Selecting method for assigning conductivity values

☞ **Use Surface** button

The **Provide Data** dialog box will appear on your screen, containing empty fields for Kx, Ky and Kz.

From the **Data Explorer**,

☞ **Sand1_Kx** data object

In the **Provide Data** dialog box,

☞ button, under the **Kx** column

☞ button, under the **Ky** column

☞ button, under the **Kz** column

The **Provide Data** dialog on your screen should look identical to the image shown below (Figure 40).

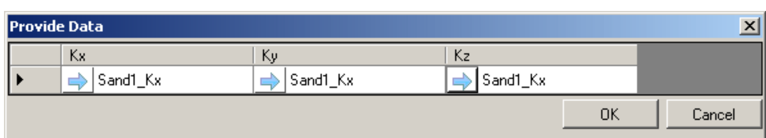


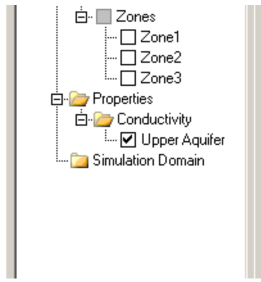
Figure 40 Using surface data object to define Kx, Ky and Kz property values

☞ **OK** button

In the **New Property Zone** dialog box,

☞ **Finish** button

In the conceptual model tree, a **Conductivity** folder will be added, under which the new property zone will appear. Like other data objects, the property zone may be viewed in the 3D Viewer by selecting the checkbox located beside it.



Note: If you have property data stored in surface files, *e.g.*, USGS .DEM, ESRI ASCII Grid (.ASC, .GRD), Surfer .GRD (ASCII or Binary), you can import these files directly as a Surface data object using the import utility, and use for defining spatially-variable property zones.

Clay1 Layer (Aquitard)

For the **Clay1 Layer** (middle layer) you will assign a constant value for Kx, Ky and Kz.

From the **Conceptual Model** tree,

☞ right-click on the **Properties** folder

☞ **Define Property Zone...** from the pop-up menu. (Figure 37)

The **New Property Zone** dialog will appear on your screen.

In the **Name** field,

type: **Aquitard**

The property zone geometry will be defined using structural **Zone 2**.

From the **Conceptual Model** tree,

☞ **Zone 2** (under **Zones** node)

In the **New Property Zone** dialog,

☞  button

☞ **Next** button

In the **Kz(m/s)** field,

type: **0.0001**

☞ **Finish** button

Sand2 Layer (Bottom Aquifer)

For the **Sand2 Layer** (bottom layer) you will assign a constant value for Kx, Ky and Kz.

From the **Conceptual Model** tree,

☞ right-click on the **Properties** folder

☞ **Define Property Zone...** from the pop-up menu. (Figure 37)

The **New Property Zone** dialog will appear on your screen.

In the **Name** field,

type: **Bottom Aquifer**

The property zone geometry will be defined using structural **Zone 3**.

From the **Conceptual Model** tree,

☞ **Zone 3** (under **Zones** node)

In the **New Property Zone** dialog,

☞  button

☞ **Next** button

In the **Kz(m/s)** field,

type: **0.001**

☞ **Finish** button

Defining Initial Heads

The heads at the beginning of the simulation will be defined using a surface data object. First you will import XYZ point data. Then you will interpolate these points to generate a surface, as demonstrated in previous sections in this tutorial.

Importing the Points Data

☞ Right-click in the **Data Explorer**

☞ **Import Data...**, from the pop-up menu (Figure 3)

The **Data Import** dialog box will appear on your screen (Figure 4).

From the **Data Type** combo box,

☞ **Point**

Beside the **Source File** field,

☞ **[...]** button

The **Select Data Source file!** dialog box will appear on your screen. Navigate to the following directory on your computer.

```
C:\Program Files\Hydro GeoBuilder\Hydro GeoBuilder Projects\Demo
Project - Friedrichshagen\suppfiles\Points
```

☞ **demo_head_ini.xls** file

☞ **Open** button

From the **Data Import** dialog box,

☞ **Next>>** button

A preview of the data in the source file will appear on your screen.

☞ **Next>>** button

The geographic information of the source data will appear on your screen.

☞ **Next>>** button again

The data mapping screen will appear. In this step, you will map the fields in the source data to the required target fields in Hydro GeoBuilder.

Under the **Map_to** column, beside the **X** field,

☞ **F1,** from the combo box

Under the **Map_to** column, beside the **Y** field,

☞ **F2,** from the combo box

Under the **Map_to** column, beside the **Elevation** field,

☞ **F3,** from the combo box

☞ **Next** >> button

☞ **Finish** button

Creating the Surface

Next you will generate a surface using the imported points data.

☞ Right-click anywhere in the **Data Explorer** (Figure 3)

☞ **Create Surface,** from the pop-up menu

The **Create Surface** dialog box will appear on your screen (Figure 22)

In the **Surface Name** text box,

type: **Initial Heads**

From the **Data Explorer**,

☞ **Demo_head_ini** data object

At the bottom of the **Create Surfac**e dialog box,

☞  button

In the **Data Source** table,

☞ **Elevation** from the combo box (Figure 36)

☞ **Interpolation Settings** tab

The interpolation settings will appear on your screen (Figure 24)

Beside the **Interpolation Method** field,

☞ **Natural Neighbors** from the combo box.

☞ **OK** button, to generate the surface

An **Information** message will appear on your screen,

☞ **OK** button

Defining the Initial Heads Property Zone

From the **Conceptual Model** tree,

☞ right-click on the **Properties** folder

☞ **Define Property Zone...** from the pop-up menu. (Figure 37)

The **New Property Zone** dialog box will appear on your screen.

In the **Name** field,

type: **Initial Heads**

All three structural zones will be used to define the geometry of the initial heads property zone.

☞  button

☞  button, again.

From the **Conceptual Model** tree,

☞ **Zone1**

From the **New Property Zone** dialog box,

☞  button, beside the top row

From the **Conceptual Model** tree,

☞ **Zone2**

From the **New Property Zone** dialog box,

☞ button, beside the middle row

From the **Conceptual Model** tree,

☞ **Zone3**

From the **New Property Zone** dialog box,

☞ button, beside the bottom row

The bottom of the **New Property Zone** dialog box should appear identical to the one shown below (Figure 41).

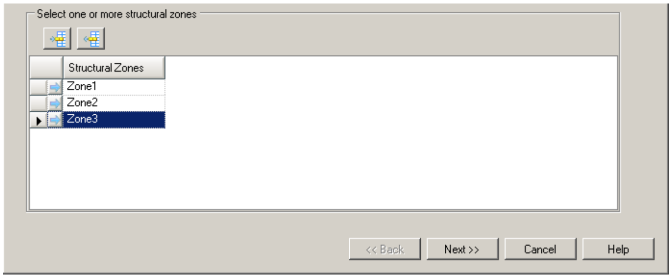


Figure 41 Defining property zone geometry from structural zones.

☞ **Next** >> button

☞ **Initial Heads** radio button

In the **Method** row, under the **Initial Heads** column,

☞ **Use Surface** from the combo box

☞ **Use Surface** button

The **Provide Data** dialog box will appear on your screen.

From the **Data Explorer,**

☞ **Initial Heads** data object

From the **Provide Data** dialog box,

☞ button

☞ **OK** button

☞ **Finish** button

The **Initial Heads** property zone will be added to the **Conceptual Model** tree, under the **Initial Heads** folder.

Generating the Simulation Domain

The next step is to define the region of the conceptual model from which the numerical model will be generated. This region is called the **Simulation Domain.** The simulation domain is generated by combining the **Structural Zones** of the conceptual model.

From the **Conceptual Model** tree,

☞ Right-click on the **Simulation Domain** folder

☞ **Generate Default Simulation Domain...** , from the pop-up menu

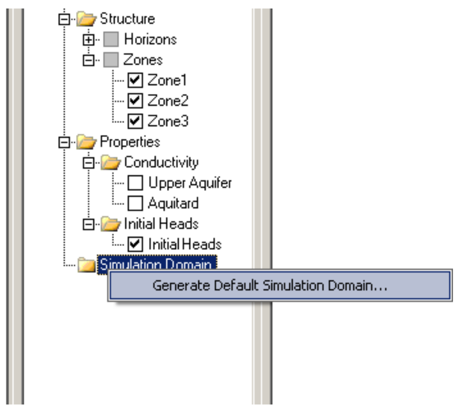


Figure 42 Generating the simulation model domain

This will create child folders under the **Simulation Domain** folder. To view the child folders,

☞ **[+]**, beside the **Simulation Domain** folder.

The **Model Domain** child node will appear.

☞ **[+]**, beside the **Model Domain** node.

This will reveal a child folder called **Boundary Conditions**. From here you can define the model's boundary conditions. This process is described in the following section.

Defining Boundary Conditions

In this section, you will define a **Type 4 (Well)** boundary condition and **Material: In/Out Flow** properties to the top layer (recharge boundary condition) .

Defining a Pumping Well Boundary Condition

You will use the imported **pumpingwells** data object to define a pumping well boundary condition.

From the conceptual model tree,

☞ Right-click on the **Boundary Condition** folder

☞ **Define Pumping Well Boundary Condition...**, from the pop-up menu.

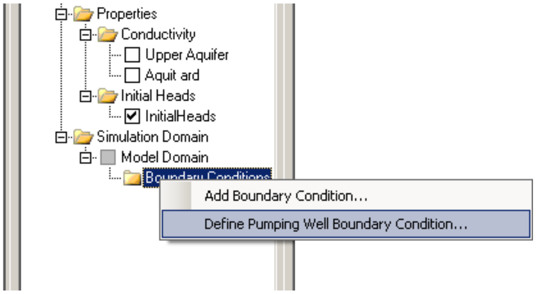


Figure 43 Creating a pumping well boundary

The **Pumping Well Boundary Condition** dialog will appear on your screen.

In the **Name** field,

type: **Pumping Wells**

From the **Data Explorer** tree,

☞  **pumpingwells** data object

In the **Pumping Well Boundary Condition** dialog box,

☞ Anywhere in the dialog box to make the box active.

☞  button

The dialog box should appear identical to the one shown below (Figure 44).

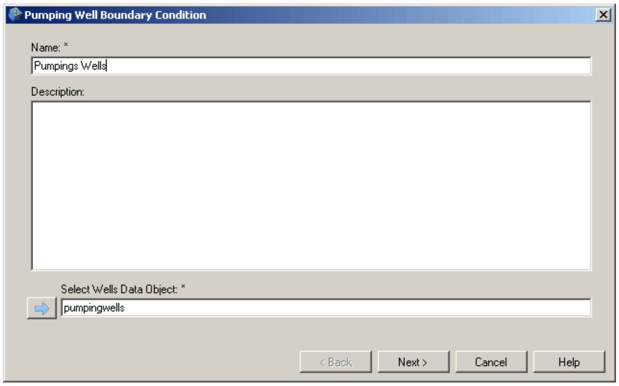


Figure 44 Choosing a wells data object

☞ **Next** button

The next step allows you to select which wells in the specified wells data object to include in the boundary condition. By default, both wells are already selected.

☞ **Next>** button

The next step validates the pumping well data to ensure the required data exists in the specified well data object. In order for a well to be included in a boundary condition it must have screen data and a pumping schedule. Both wells are valid as they meet the data requirements.

☞ **Next>** button

The final step provides a detailed preview of the well geometry, pumping schedules, and screen details of each well in the specified wells data object.

☞ **Finish** button

The boundary condition will now appear under the **Boundary Conditions** folder in the **Conceptual Model** tree.

Defining a Recharge Boundary Condition

In this step, you will define a groundwater recharge boundary condition to the top layer of the conceptual model.

From the **Conceptual Model** tree,

☞ Right-click on the **Boundary Condition** folder

☞ **Add Boundary Condition...**, from the pop-up menu.

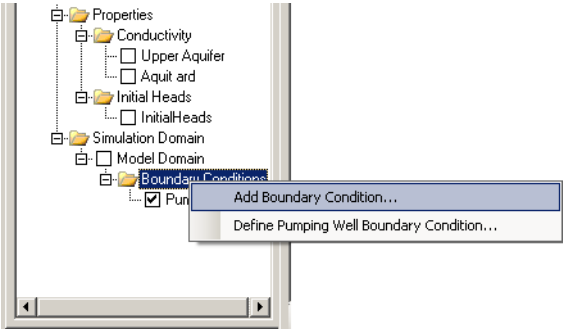


Figure 45 Adding a boundary condition

The **Define Boundary Condition** dialog box will appear on your screen.

From the **Select Boundary Condition Type** combo box,

☞ **Recharge**

In the **Name** field,

type: **Groundwater Recharge**

From the **Data Explorer**,

☞ **Model_area_ply**

In the **Define Boundary Condition** dialog box,

☞ Anywhere in the dialog box to make it active

☞  button

☞ **Next>>** button

The next dialog allows us to define the recharge value.

> **Note:** Hydro GeoBuilder provides various options for defining boundary condition attributes. Attributes can be assigned from attributes stored in **Surface**, **Time Schedule**, **Shapefile** and **3D Gridded** data objects. You can also set attributes as **Static** (no change over time) or **Transient** (changes over time). For more information on the available methods for assigning boundary condition attribute values, please refer to the Hydro GeoBuilder User's Manual.

For this tutorial, you will assign a static constant recharge value.

In the empty field located below the **Constant** field,

type: **0.001**

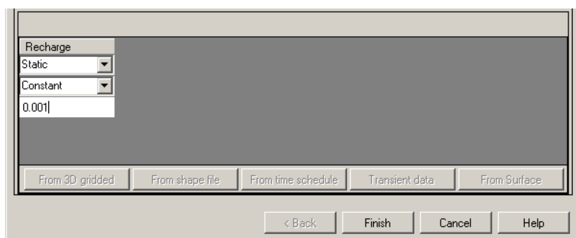


Figure 46 Specifying a recharge value

☞ **Finish** button

Creating the Finite Element Mesh

Hydro GeoBuilder provides two methods for discretizing your conceptual model: the finite difference method and the finite element method. In this tutorial, you will use the finite element method to discretize the conceptual model.

Note: For information on how to create finite difference grids for running MODFLOW simulations, please refer to the Hydro GeoBuilder User's Manual.

In the following sections, you will define add-ins for the 2D superelement mesh, then define slice elevations from the horizons in the conceptual model, resulting in a 3D finite element mesh.

Importing an Add-In Polygon

Before creating the finite element mesh, you will import a polygon which will be used to define a localized area of mesh refinement.

> **Note:** Hydro GeoBuilder allows you to create and edit polygon data objects by digitizing shapes using the 2D Viewer drawings tools. This feature may be useful for delineating boundary condition zones, or for defining localized areas of mesh and grid refinement. For more information the 2D Viewer drawings tools, please refer to the Hydro GeoBuilder User’s Manual.

- ☞ Right-click in the **Data Explorer**
- ☞ **Import Data...**, from the pop-up menu (Figure 3)

The **Data Import** dialog box will appear on your screen (Figure 4).

From the **Data Type** combo box,

- ☞ **Polygon**

Beside the **Source File** field,

- ☞ **[...]** button

The **Select Data Source file!** dialog box will appear on your screen. Navigate to the following directory on your computer.

```
C:\Program Files\Hydro GeoBuilder\Hydro GeoBuilder Projects\Demo
Project - Friedrichshagen\suppfiles\Polygons
```

- ☞ **super mesh.shp** file
- ☞ **Open** button

From the **Data Import** dialog box,

- ☞ **Next>>** button

The geographic information of the source data will appear on your screen.

- ☞ **Next>>** button again

The **Data Mapping** screen will appear. No data mapping is required for this data.

- ☞ **Next>>** button again

The final step in the import process checks the source file for errors and invalid data. No errors or warnings were detected.

☞ **Finish** button

The imported polygon will now appear in the **Data Explorer**.

Creating the Finite Element Mesh

From the **Conceptual Model Tree**,

☞ Right-click on the **Model Domain** node

☞ **Create Finite Element Mesh...**, from the pop-up menu

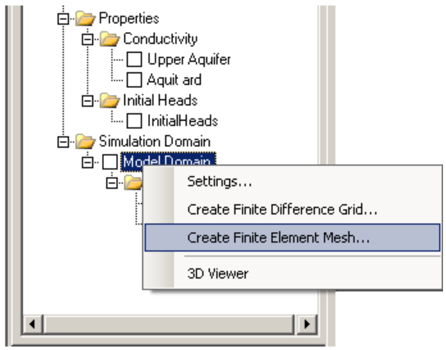


Figure 47 Creating a finite element mesh

The **Define Finite Element Mesh window** will appear on your screen.

In the **Name** field,

type: **Mesh1**

You will see that Hydro GeoBuilder has automatically included the model boundary as the area for the super element mesh, and the **Pumping Wells** well boundary condition as points add-ins, in the adjacent 2D Viewer . You will now add the imported **demo_refine.shp** polygon to the superelement mesh.

From the **Data Explorer**,

☞ **super mesh** data object

From the **Define Finite Element Mesh** dialog box,

☞ **Add-In Lines/Points/Polygons** (located at the bottom-left corner)

The **demo_refine** data object will be added to the **Add-In Lines/Points/Polygons** list, and will appear in the adjacent 2D Viewer window (Figure 48).

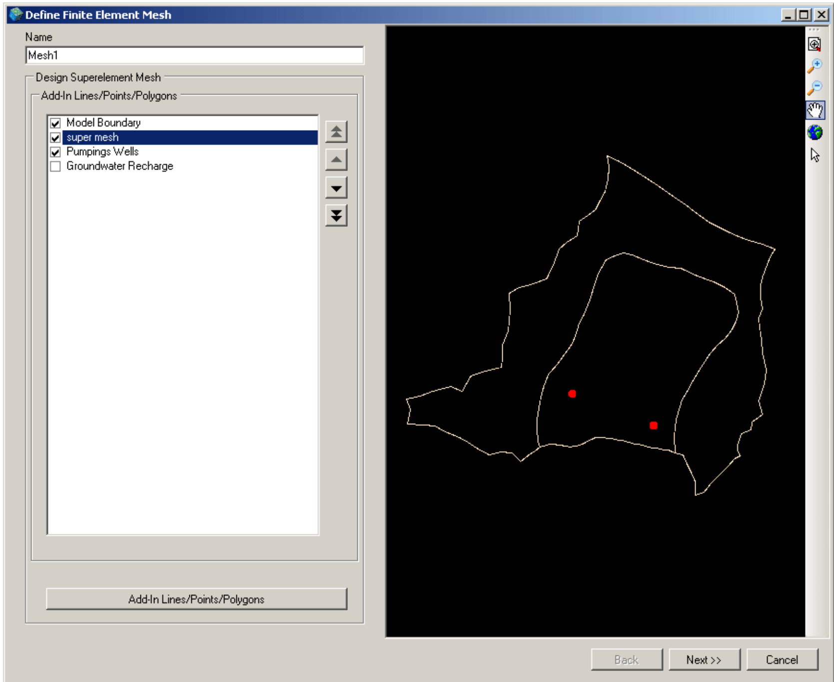


Figure 48 Defining the superelement mesh

☞ **Next>>** button

Various mesh options will appear on your screen.

> **Note:** Hydro GeoBuilder uses Triangle, an advanced algorithm developed by J.R. Shewchuck, to generate exact Delaunay triangulations, constrained Delaunay triangulations and conforming Delaunay triangulations. For more information on the triangulation options available in Hydro GeoBuilder, please refer to the User's Manual.

You will use the default settings to generate the finite element mesh. You will use the **super mesh** polygon to delineate an area of the model domain that will have a more detailed mesh.

Under the **Total Number of Elements** column,

type: **500**

For the **Refinement along all superelement border edges** option,

☞ ☑ , to **disable** the option.

For the **Refinement along line add-ins option**,

☞ ☑ , to **disable** the option.

☞ **Polygons Refinement...** button

The **Polygon Refinement** window will appear on your screen.

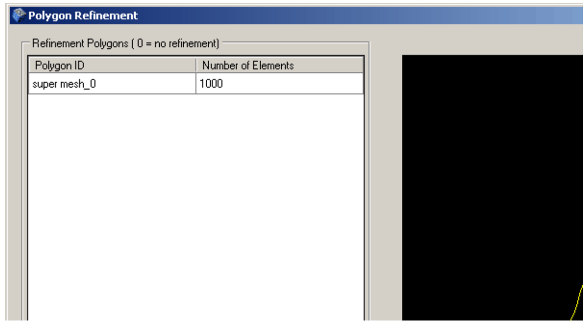


Figure 49 Refining area of mesh using **super mesh**

Under the **Number of Elements** column,

type: **1000**

☞ **OK** button

Now you are ready to generate the finite element mesh. To do so,

☞ **Generate** button (located at the bottom of the Define Finite Element Mesh window)

Your screen should look similar to the image shown below.

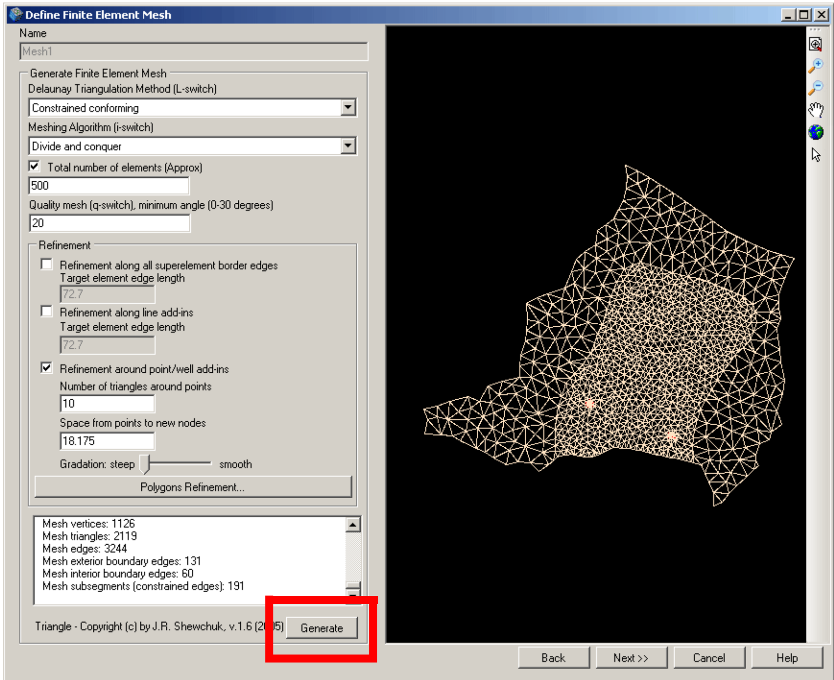


Figure 50 Generated finite element mesh with refinement area

☞ **Next** >> button

The **Define Slice Elevation** options will appear on your screen. You will use a **Deformed** mesh type, which fits the slice elevations to the tops and bottom of the horizons.

In the **Exaggerate** box,

type: 35

<Enter>

The adjacent 3D Viewer preview allows you to inspect the mesh slices by rotating, zooming and panning the mesh.

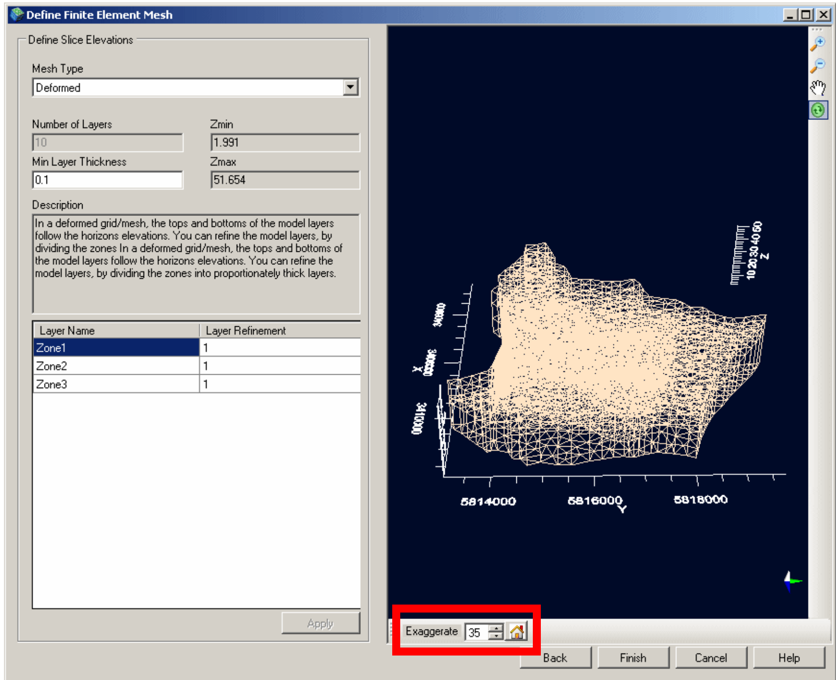


Figure 51 Defining and previewing the vertical slice elevations

You will use the default minimum layer thickness. The grid in the lower left allows for quick vertical refinement of any of the layers. For this tutorial, vertical refinement is not required.

☞ **Finish** button

A **Finite Element Mesh** node called **Mesh 1** will be added to the **Conceptual Model Tree**, under the **Boundary Conditions** folder.

To view the mesh in the 3D-Viewer 1 window,

☞ ☐ **Mesh1**

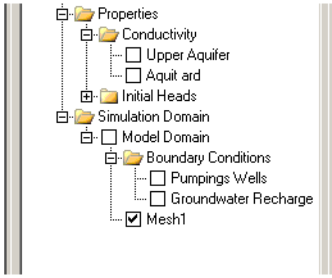


Figure 52 Showing finite element mesh in viewer

You may wish to add other objects to the viewer, such as the structural zones, site map, surfaces and other shapes in the **Data Explorer**.

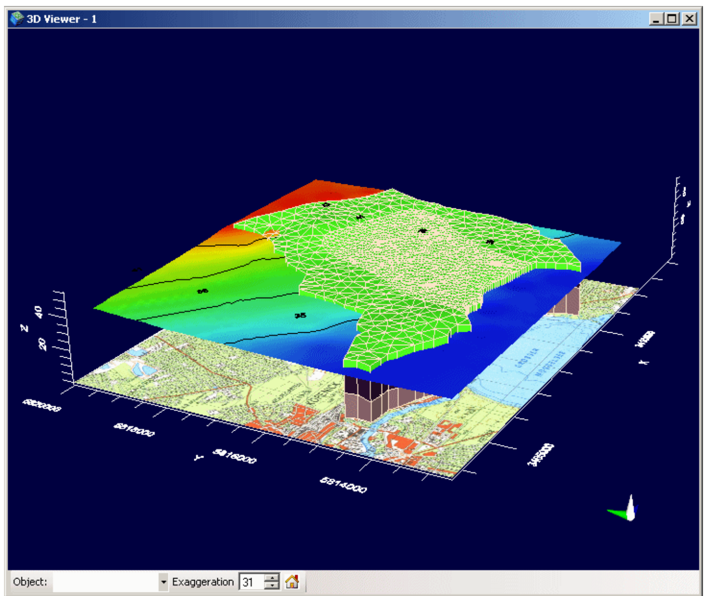


Figure 53 Displaying **Mesh 1**, **Zone 1**, **Zone 2**, **Zone 3**, **Initial Heads** and **SimulationArea** (sitemap) in the **3D Viewer** window.

Translating to Finite Element Numerical Model

In this step, you will generate the FEFLOW ASCII . FEM file from the data in the conceptual model and the 3D finite element mesh.

From the conceptual model tree,

☞ Right-click on the **Demonstration** folder (root folder)

☞ **Translate to Finite Element Model...** from the pop-up menu

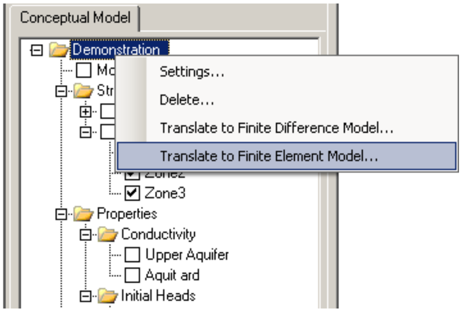


Figure 54 Translating a conceptual model to a finite element numerical model

The **Translate to Finite Element Model** dialog box will appear on your screen. From here you can define various FEFLOW simulation settings, including the **Simulation Type** (steady state or transient), **Flow Type**, **Start Date**, **Start Time** and **Steady State Simulation Time**.

Please note the **Output name** setting. It indicates the directory and filename where the .FEM file will be saved.

For this tutorial, you will keep the default simulation settings.

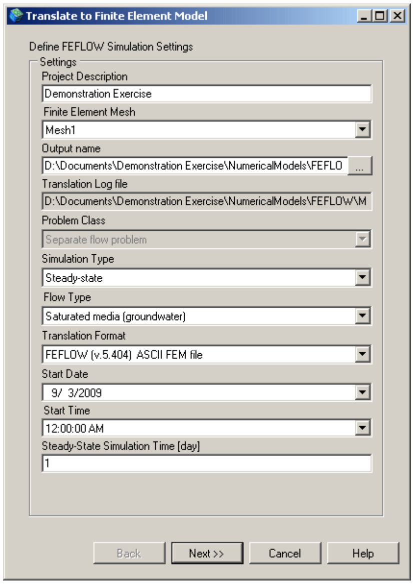


Figure 55 FEFLOW simulation settings

☞ **Next** >> button

The next screen allows us to include or exclude boundary conditions from the numerical model. Two boundary conditions were defined in this tutorial; a recharge boundary and a well boundary. Pumping wells will be translated as Type 4 boundary condition. Recharge will be translated as an In/Out Flow material. Future versions of HGB will provide support for additional types of boundary conditions.

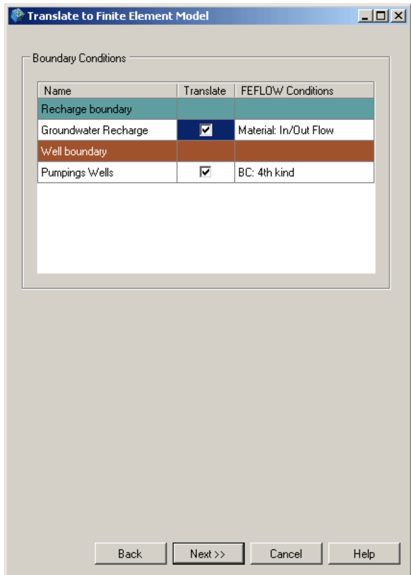
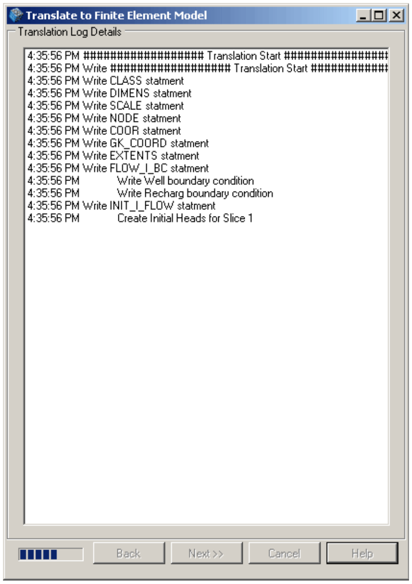


Figure 56 Include/Exclude boundary conditions from translation

☞ **Next** >> button

Hydro Geobuilder will begin to convert the conceptual model and selected mesh, to the FEFLOW ASCII .FEM file, and display the progress details on screen.



Once the translation is finished,

☞ **Finish** button.

Viewing the FEFLOW Input Files

To view the generated input files, open Window's Explorer (right-click on the Start button in your task bar, and select Explore), and navigate to the following folder on your computer.

```
C:\Documents\Demonstration Exercise\Numerical
Models\FEFLOW\MESH1
```

The **Demonstration Exercise.FEM** file can be loaded into FEFLOW for running the flow simulation, or for making further changes.

Table 1: Supported Data Types in Hydro GeoBuilder

Data Type	Supported File Types	Description	How can it be used in HGB?
Points	.XLS, .MDB, .DXF, .TXT, .CSV, .ASC	Discrete data points with known attribute(s), *e.g.*, X, Y, elevation, top/ bottoms of formations, Kx, Initial Heads.	Interpolate the points to generate surfaces, which can be used for defining conceptual model horizons, or distributed parameter values such as Kx, Initial Heads, Recharge, etc.
Polygons	2D/3D ESRI Shapefile, AutoCAD DXF	GIS vector files containing polygon geometry and attributes	Use to define the conceptual model domain Use to delineate property zones Use to define geometry of aerial boundary conditions, *e.g.*, lake, recharge, specified-head.
Polylines	2D/3D ESRI Shapefile, AutoCAD DXF	GIS vector files containing line geometry and attributes	Use to define geometry of linear boundary conditions, *e.g.*, river, drain, general head.
Surfaces	USGS .DEM, ESRI ASCII Grid (.ASC, .FRD), Surfer .GRD (ASCII or Binary)	Files containing an ordered array of interpolated values at regularly spaced intervals that represent the spatial distribution of an attribute, *e.g.*, digital elevation model	Use to define conceptual model horizons Use to assign spatially-variable attributes to boundary conditions and property zones.
Wells	.XLS	Well head coordinates (X, Y, Z) and associated well attribute data such as screen intervals, pumping schedules, observation points and data, well tops (contact points with geological formations), and well path (for deviated wells)	Interpolate well heads to generate surface representing topography. Convert well tops to surfaces representing top/bottoms of geological formations Use to define pumping well boundary conditions.
Time Schedules	.XLS	Attributes measured over time, *e.g.*, hydrographs	Use to define transient data for boundary conditions, such as recharge, river stage elevations etc.
Maps	.JPG, .BMP, .TIF, .GIF	Raster images, *e.g.*, aerial photographs, topographic maps, satellite imagery	Use sitemaps for gaining a perspective of the dimensions of the model, and for locating important characteristics of the model.
Cross Sections	HGA-3D Explorer (.3XS)	Cross sections generated using Hydro GeoAnalyst data management software	Generate surfaces from cross section model interpretation layers and use for defining model horizons/structural zones.

Data Type	Supported File Types	Description	How can it be used in HGB?
3D Gridded Data	TecPlot . DAT, MODFLOW .HDS	3D Grid with attributes at each grid cell.	Use to visualize heads data generated from a MODFLOW run in Visual MODFLOW. Use to assign spatially-variable attributes to boundary conditions and property zones.