Contents

Part 1: Introduction

Leapfrog Hydro allows rapid construction of 3D hydrogeological meshes directly from scattered borehole data without the need for manual digitisation. Leapfrog uses implicit modelling to generate geological models in hours rather than days. Leapfrog Hydro uses a rapid 3D interpolation technique, FastRBF™, to construct 3D boundary models from borehole data. Leapfrog was designed especially for geologists and environmental scientists. Its streamlined workflow is easy to learn and use, and basic modelling techniques can be learned in as few as four hours.

About this Manual

This manual is divided into:

- Part 1: Introduction. This part introduces Leapfrog Hydro, outlines system requirements and provides information on getting technical support.
- Part 2: Getting Started. This part describes how to install Leapfrog Hydro, start the application, activate the licence and run a graphics test to ensure your computer can accurately display geological 3D models.
- Part 3: Working with Leapfrog. This part describes the Leapfrog workspace. Topics include navigation in 3D, drawing in 3D and customising the workspace to suit your work environment.
- Part 4: Leapfrog HydroTutorials. A series of tutorials based on a small data set will introduce you to basic concepts in Leapfrog Hydroand get you to the point where you can start processing your own data. The tutorials take two to four hours to complete.
- Part 5: Reference. This part describes Leapfrog Hydro features in greater detailed than described in the tutorials.

To get the most out of the information in this manual, you should be familiar with Microsoft Windows™ XP, Windows™ Vista or Windows™ 7 operating systems and with basic hydrogeological terminology. Experience with geological modelling software is not necessary.

Tutorial Files

Tutorial files are copied to your computer's hard drive when Leapfrog Hydro is installed. [more info required about a specific location, or at least sufficient information for the user to find the files readily]

Conventions Used

The following conventions are used throughout this manual:

- Keys on the keyboard are in `bold monospace text`. For example, press `N`.
- Menu options and dialog items are in **bold normal text**. For example, select **Project > Save**.
- Buttons to be pressed are in **bold normal text**. For example, press the **Cancel** button.

- Text to be entered from the keyboard are in `monospace text`. For example, type `Hello`.
- File and directory paths are in `monospace text`. For example, `C:\Program Files\Leapfrog`.

System Requirements

While a system that meets the minimum requirements will be sufficient for running the small tutorial data set, real-world data is typically large and takes more time to process. More modern systems will be more capable of processing data sets in a reasonable amount of time than older computers with less memory. Minimum system requirements are:

- Windows™ XP (service pack 3) 32-bit/64-bit, Windows™ Vista or Windows™ 7
- 1.5 GHz or higher processor
- 2GB system RAM
- Video card supporting OpenGL 1.3:
    - NVIDIA: GeForce FX series or better, including GeForce 6000~8000, Quadro FX 500~5600 series. GeForce 4 and earlier are not supported.
    - ATI: Radeon 9000 series or better, including Radeon X300~X1950, HD 2400~HD2900 series
    - Intel GMA3100 graphics or better

To ensure efficient processing of real-world data sets, a system that meets the following requirements is recommended:

- Windows™ Vista or Windows™ 7 64-bit
- 2 GHz dual core or higher processor
- 2GB system RAM
- Video card supporting OpenGL 2.0:
    - ATI HD4000 series or mobile versions
    - NVIDA GeForce 8000, GT220, GTX275
    - Intel X4500

Ensure that the correct graphics driver is installed to achieve hardware acceleration for OpenGL. The use of recently released video cards is not recommended as their drivers can be unstable until some time after their commercial release.

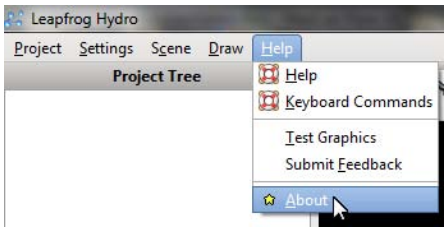
Once Leapfrog is installed, run the graphics test (**Help > Test Graphics**) to ensure that the video card is operating correctly. See Running the Graphics Test for more information.

Getting Support

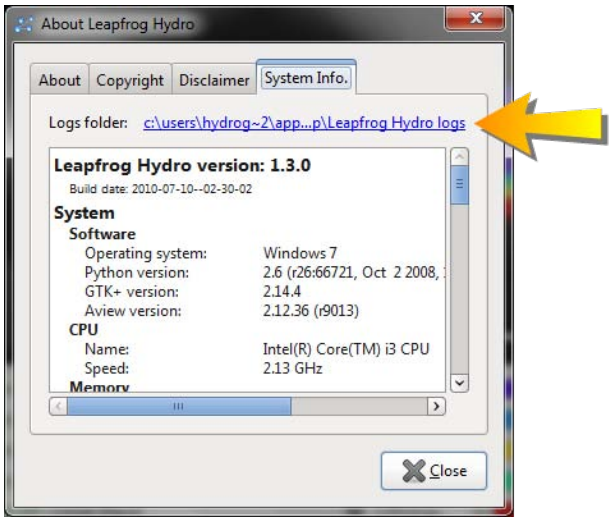
Technical support is available by emailing the address supplied when you purchased Leapfrog Hydro.

When contacting technical support, please include your license number and a full description of the problem or query, including any information provided in system or error messages.

You can access your license information by selecting **Help > About**:



You may also be asked to provide a copy of the log files. To find out where these are stored, select **Help > About**, then click on the **System Info.** tab. Logs are stored in the location given as the **Logs folder**.



Click on the link to open the folder.

Part 2: Getting Started

This part describes how to install Leapfrog Hydro, activate the product license and ensure the application will run correctly on your computer.

Installing Leapfrog Hydro

Before starting the installation, make sure the dongle is not yet connected. You should also disconnect any other dongles you might have.

To install Leapfrog Hydro, double-click on the file LeapfrogHydroSetup.exe and follow the installation wizard.

Once the installation is complete, insert the dongle. Your computer will install drivers for the dongle. Once the drivers have been installed, you will need to activate your license, which is described in Activating the License.

Activating the License

To run Leapfrog Hydro, you must have a dongle connected to one of your computer's USB ports and a valid Leapfrog Hydrolicense.

When your license is issued, you will receive an email containing your license information and instructions on updating the dongle with your new license. To activate the license:

1. Ensure your computer is connected to the internet.
2. Insert the dongle into one of your computer's USB ports.
3. On your computer, select **Start > All Programs > Leapfrog Hydro > License Manager**.
4. Follow the instructions in the email.

If you have not received an email, contact sales@leapfroghydro.com.

Once the license is activated, you can start Leapfrog Hydro.

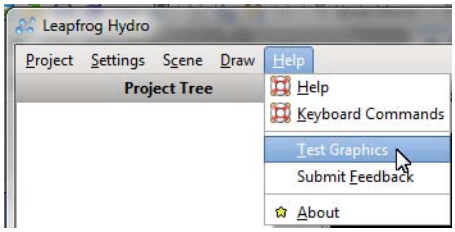
Starting Leapfrog

To start Leapfrog Hydro, you can double-click the Leapfrog Hydro icon on your desktop or select **Start > Programs** or **All Programs > Leapfrog > Leapfrog Hydro**.

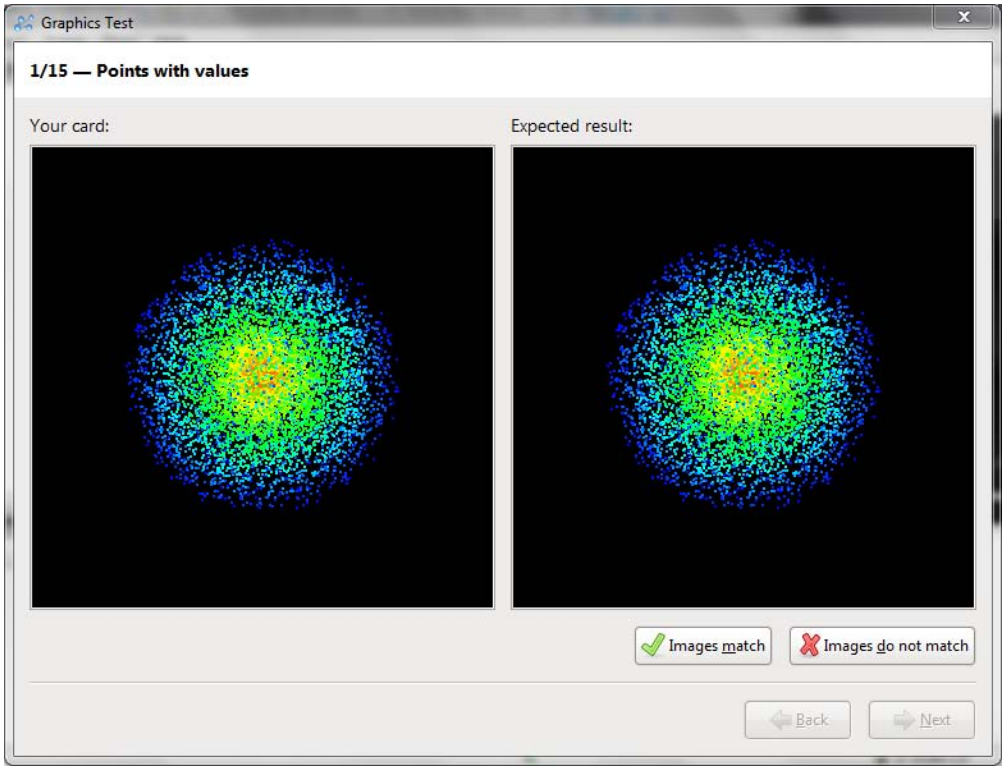
Running the Graphics Test

When you first run Leapfrog Hydro, run the graphics test to test the capability of your computer's video card.

To start the test, open Leapfrog and select **Help > Test Graphics**:

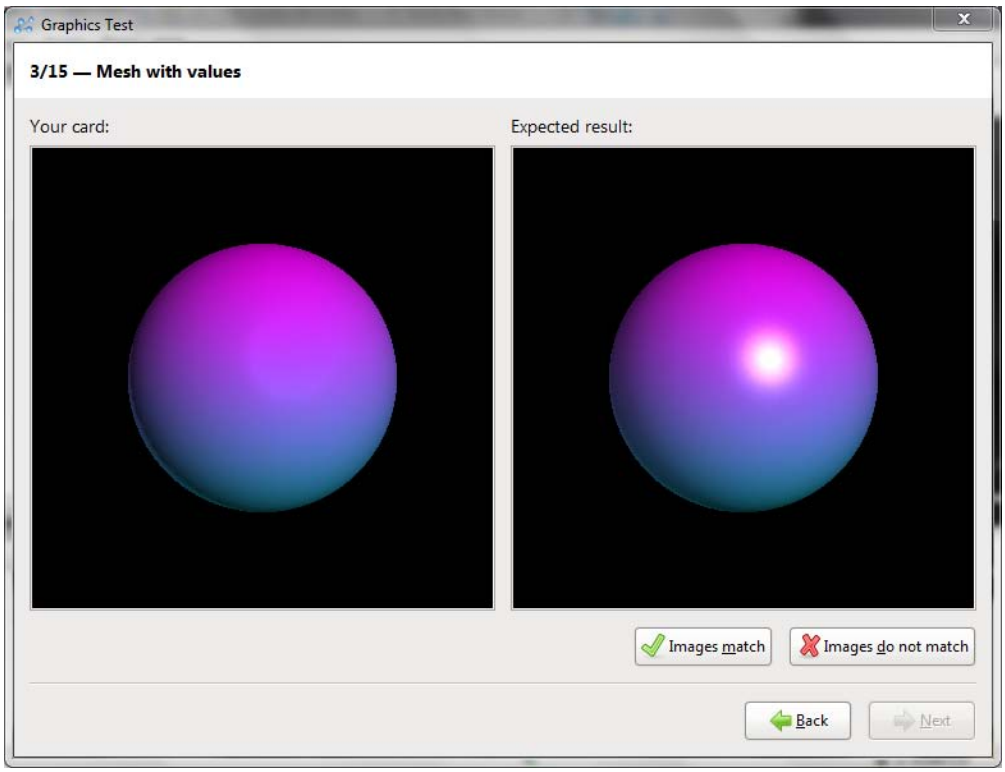


The graphics test displays a series of image pairs that test the capabilities of your video card.

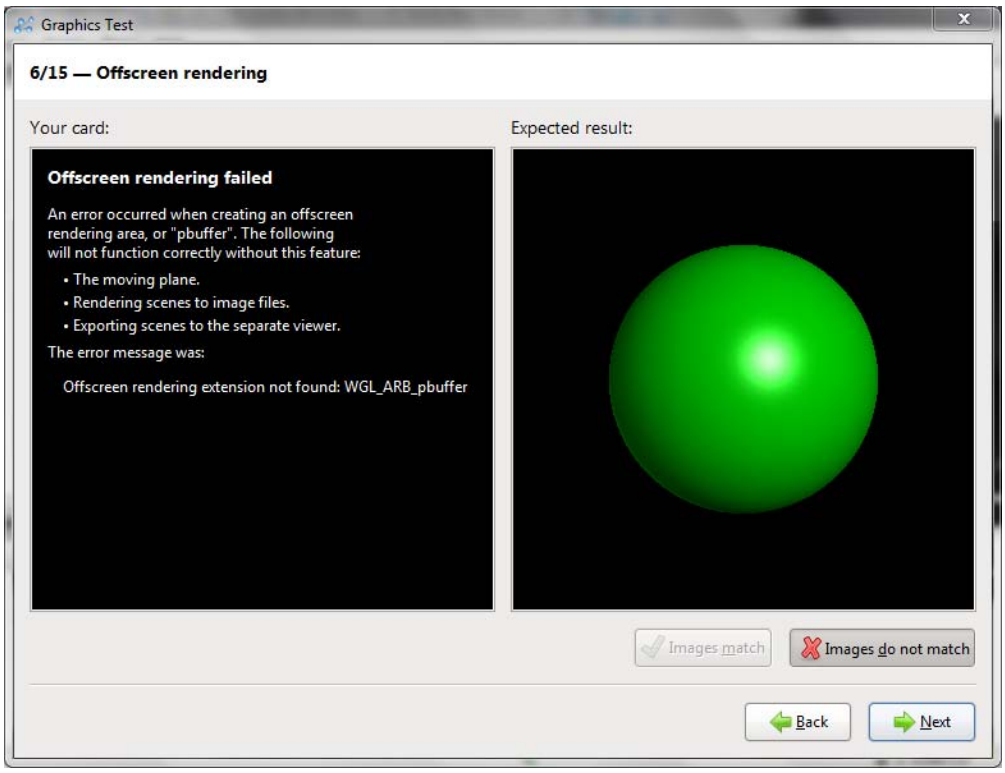


For each pair, click on either the Images match button or the Images do not match button, then on **Next** to proceed to the next step.

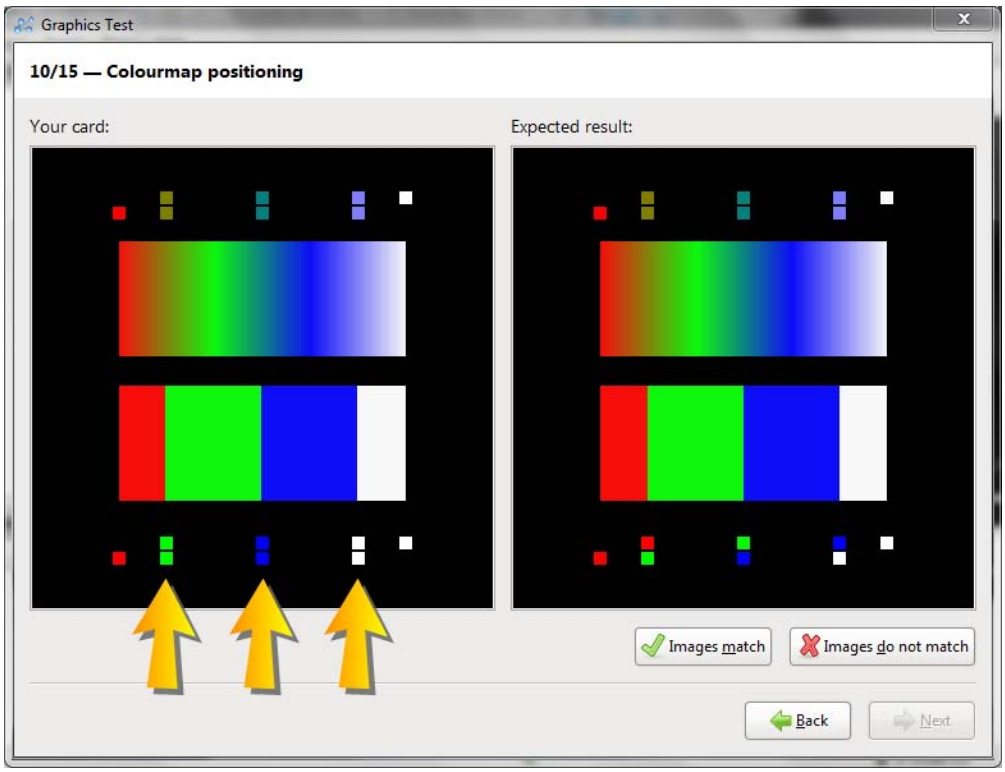
There are different ways in which images do not match. For example, for the pairs below, part of the image on the left has not rendered correctly:



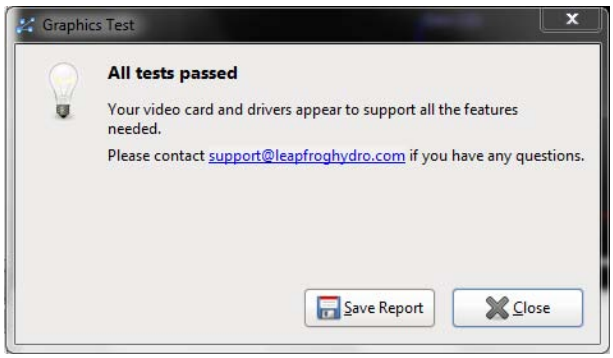
For this pair, the images clearly do not match:



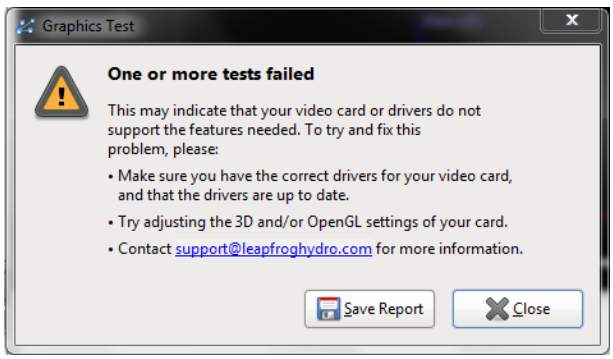
However, for this pair, the difference between the two images is subtle, with two small squares along the bottom of each image displayed incorrectly:



At the conclusion of the test, the result is displayed:



If one or more tests have failed, the message below will be displayed:



If you click on **Save Report**, you will be prompted to save the file on your computer. If you cannot resolve the problem following the steps described in Troubleshooting Video Card Issues, email this

file to the supplied email address, together with information about your system copied from the **Help > About > System Info** tab.

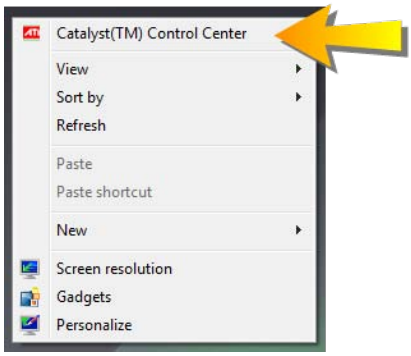
Troubleshooting Video Card Issues

If the graphics test fails, there are two steps you can take to correct the problem:

- Ensure your video driver is up-to-date. See Updating Your Video Card Driver.
- Check your video card's 3D/OpenGL settings. See Checking Your 3D/OpenGL Settings.

Updating Your Video Card Driver

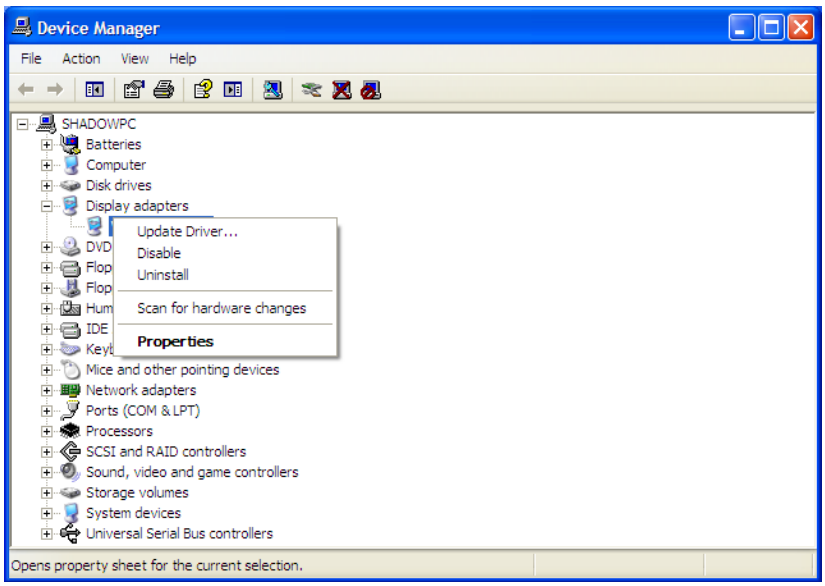
The best way to update your video driver is to do this using any tools that your computer's manufacturer provides. For example, some manufacturers allow you to update drivers using a tool installed in the **Start** menu. Look under **All Programs** (or **Programs** in Windows XP) for items that carry the manufacturer's name. Another option is to right-click on the desktop to see if there is an update tool available. In the example below, the manufacturer has included an option for updating video drivers called "Catalyst Control Center":



If your computer appears to have no manufacturer-installed tools for updating drivers, you can go to the manufacturer's website and see if updates are available there. Look for parts of the website relating to support and driver downloads or driver updates.

If your computer's manufacturer does not provide information to help you update your video driver, you can run the **Hardware Update Wizard** to see if a newer driver is available. To do this:

1. Open the open the **Start** menu and navigate to the **Control Panel**.
2. Open the **System** option.
3. Switch to the **Hardware** tab.
4. Click on the **Device Manager** option.
5. Click to open the **Display adapters** option.
6. Right-click on the display adapter and choose the **Update Driver** option:



The **Hardware Update Wizard** will open:

Follow the steps to complete the wizard. If the wizard cannot update your software because your computer already has the latest software, it may be that your video driver is already up-to-date and that the reason the graphics test fails may lie in the card's 3D/OpenGL settings. See Checking Your 3D/OpenGL Settings.

If your video card's driver has been updated, run the graphics test again to see if the computer passes the tests. If the test still fails, use the **Save Report** option and email the report to the supplied email address, together with information on your system copied from the **Help > About > System Info** tab.

Checking Your 3D/OpenGL Settings

The step or steps at which your computer fails the graphics test may indicate what 3D/OpenGL settings need to be changed. Save the graphics test report, as described in Running the Graphics Test and email it to support@leapfroghydro.com.

Part 3: Working with Leapfrog

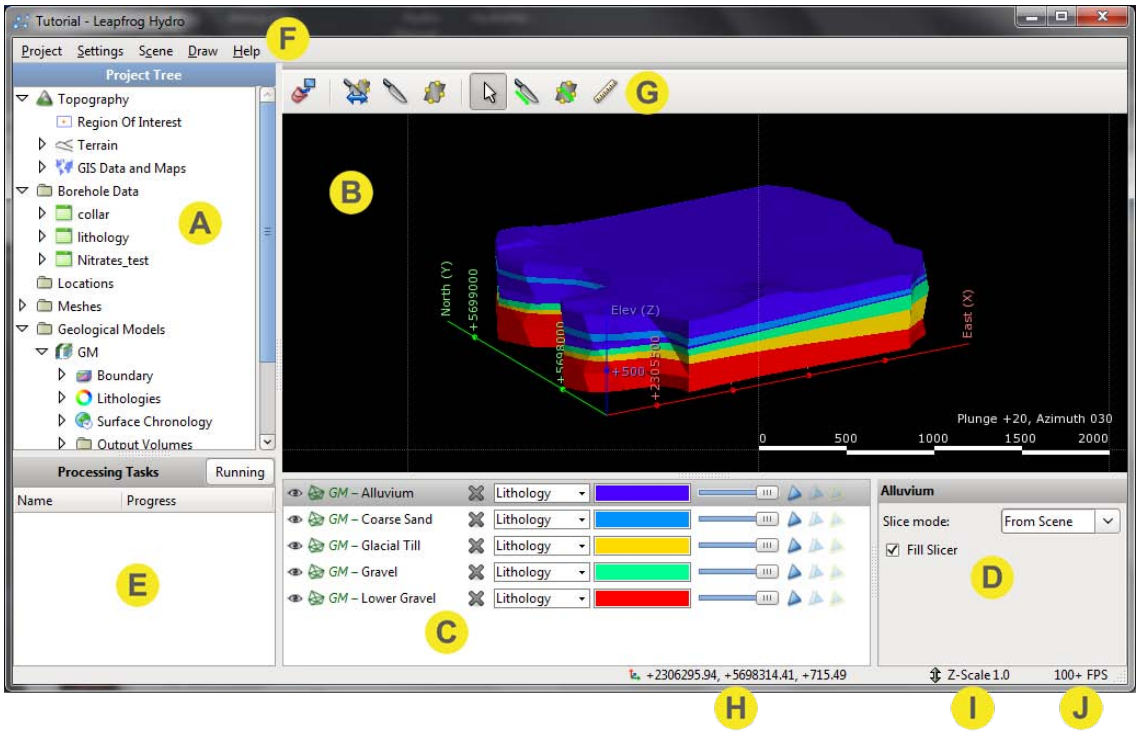
This part describes how to work with Leapfrog Hydro. It covers the following topics:

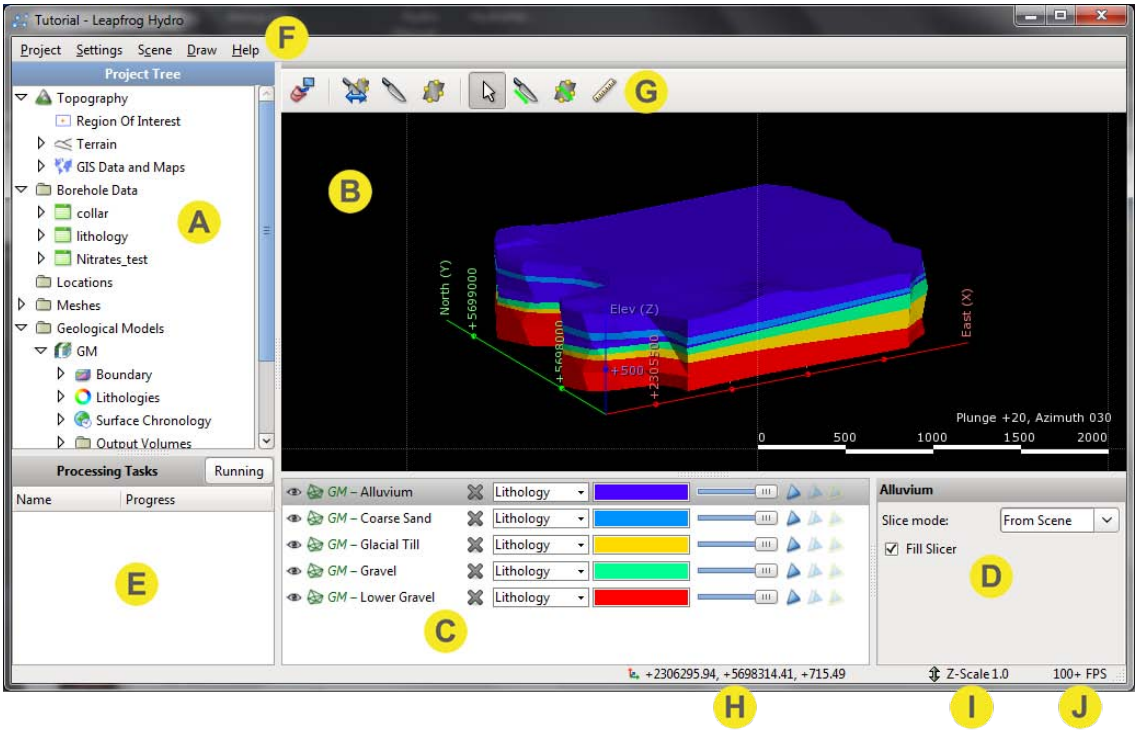
- The Leapfrog Hydro Main Window
- Navigating in the Scene Window
- Working with the Project Tree
- Working with Objects in the Scene
- Viewing Data in the Scene
- Drawing in the Scene
- The Typical Leapfrog Hydro Workflow
- Other Workflows

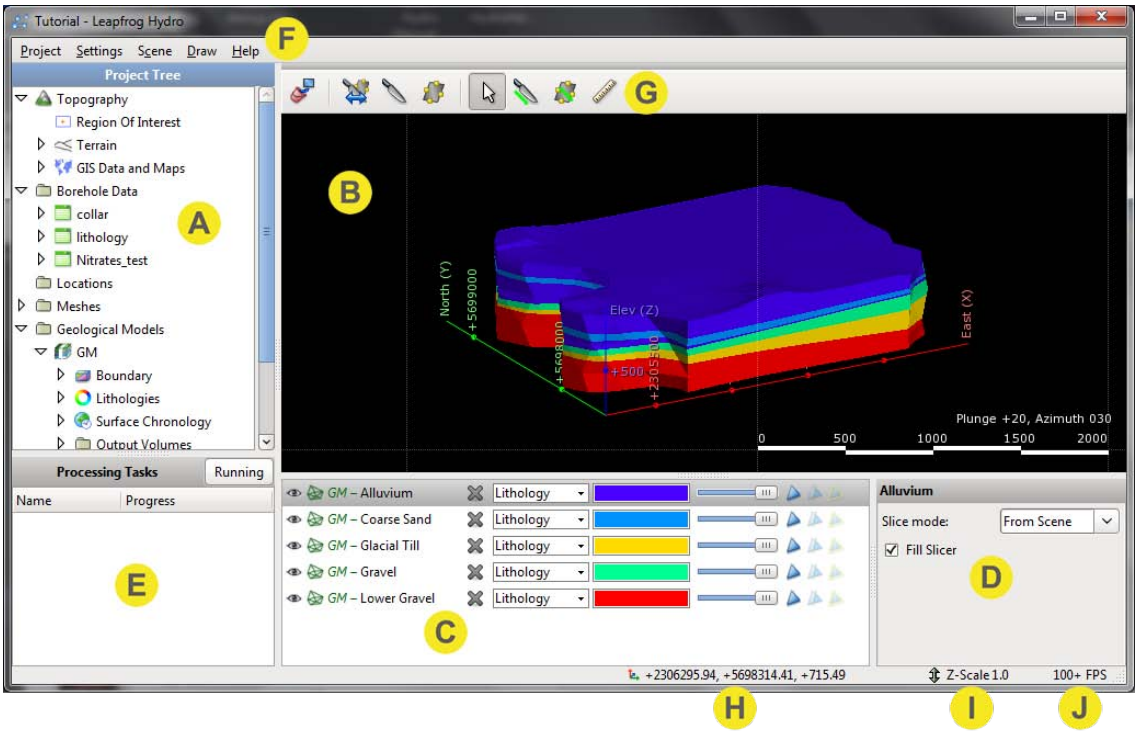
Also described are options for customising the Leapfrog workspace to suit your own needs.

The Leapfrog Hydro Main Window

The Leapfrog Hydro main window is shown below:







A Project tree. Displays all the items (maps, images, data sets) that have been imported to or created in the project. See Working with the Project Tree.

B Scene window. Displays a 3D representation of selected data from the project tree. See Working with Objects in the Scene.

C Shape list. Shows what items from the project tree are displayed in the scene window. See Changing the Appearance of Objects in the Scene Window.

D Shape property frame. Shows parameters of the shape currently selected in the shape list.

E Processing frame. Shows what items are currently being processed or are in the processing queue.

F Main menu

G Scene toolbar

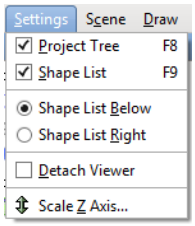
H Coordinates of the mouse in the scene window

I Current z-axis setting. See Scale Z Axis (Settings Menu).

J Current frames per second setting. See Quality (Scene Menu).

Changing the Appearance of the Main Window

The **Settings** menu contains options that let you change the layout of the Leapfrogmain window to suit the way you work:



These settings are described below.

Project Tree (Settings Menu)

The **Settings > Project Tree** option determines whether or not the project tree is displayed in the Leapfrog main window.

If your screen space is limited and you are primarily working viewing and editing data, you may wish to hide the project tree from view. To do so, tick the box or press the `F8` key.

Shape List (Settings Menu)

The **Settings > Shape List** option determines whether or not the shape list is displayed in the Leapfrog main window.

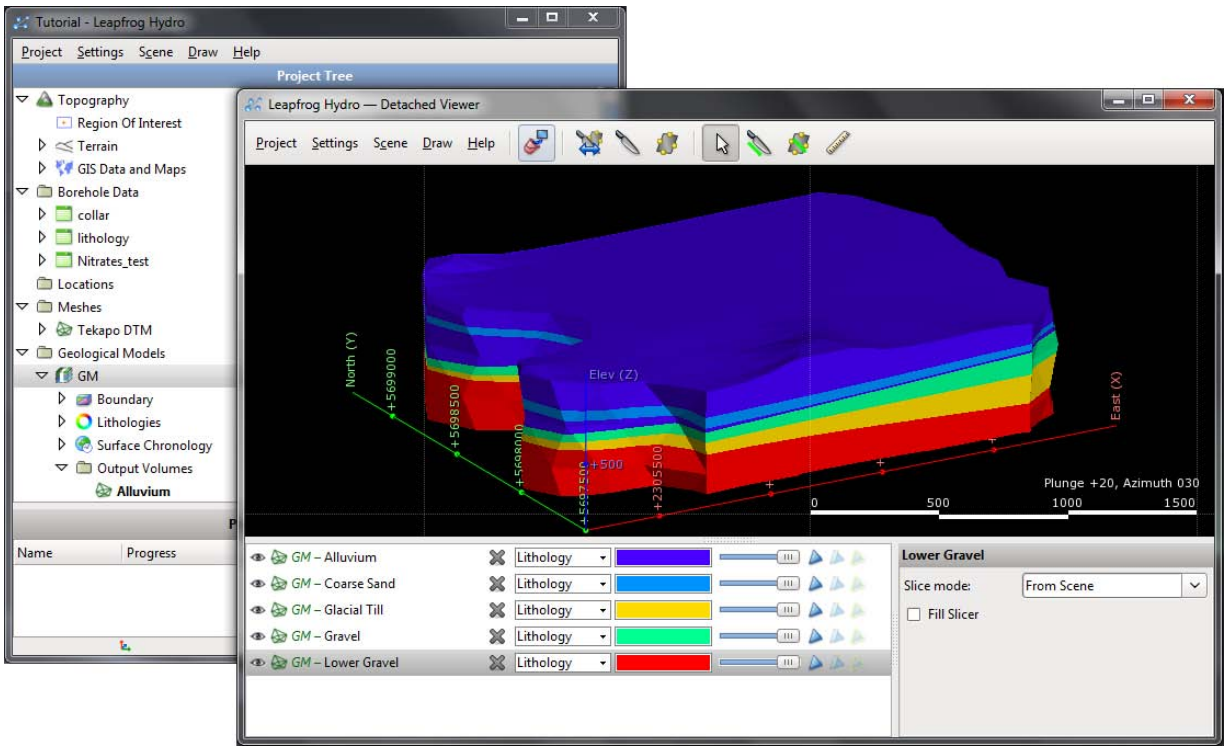
If your screen space is limited and you are primarily working with objects in the project tree, you may wish to hide the shape list from view. To do so, tick the box or press the `F9` key.

Shape List Location (Settings Menu)

The **Settings > Shape List Below** and **Shape List Right** options determine the position of the shape list in the Leapfrog main window. Select the option you prefer.

Detach Viewer (Settings Menu)

The **Settings > Detach Viewer** option lets you split the Leapfrog main window into two separate parts:



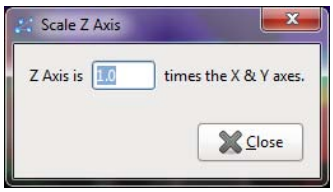
This is useful if you have two or more displays. You could have the scene window maximized on one screen and the project tree and processing frame on the other.

To restore the original layout, select untick the **Detach Viewer** box or press the `Esc` key.

Scale Z Axis (Settings Menu)

The **Settings > Scale Z Axis** option lets set a value greater than 1.0 for the Z-axis relative to the X- and Y-axes. This is useful when the area under study is very planar and extends over a wide area. In such cases, scaling the Z-axis can accentuate the distribution of data along the Z-axis.

To scale the Z-axis, select **Scale Z Axis** from the **Settings** menu. The **Scale Z Axis** window will appear:



Enter a value greater than or equal to 1. The contents of the scene will be automatically scaled, so you can experiment with different values before choosing one that best accentuates the distribution of data. Click the **Close** button to dismiss the window.

Navigating in the Scene Window

The best way to navigate the scene in Leapfrog is using the mouse:

- Rotate the scene by clicking and dragging.
- Zoom in and out using the scroll wheel or by holding the right mouse button and moving the mouse.
- Pan the scene by clicking and holding both mouse buttons, then dragging.
- View more information about any object in the scene by clicking on it.

You can also navigate in the scene using keyboard shortcuts:

- Rotate the scene using the arrow keys. For smaller steps, hold down the shift key while pressing the arrow keys.
- Zoom in and out using the `Page Up` and `Page Down` keys. For smaller steps, hold down the `Shift` key.
- Pan the scene by holding down the `Alt` key, then using the arrow keys.

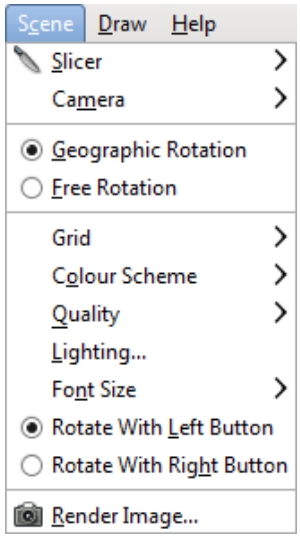
You can also switch to a specific views using keyboard keys:

- Press the `Home` key to view from the northeast
- Press the `N` key to view from the north
- Press the `E` key to view from the east
- Press the `W` key to view from the west
- Press the `S` key to view from the south
- Press the `D` key to view down from the top
- Press the `U` key to view up from the bottom

The current viewing angle and scale are displayed in the lower righthand corner of the scene window.

Changing Scene Window Navigation

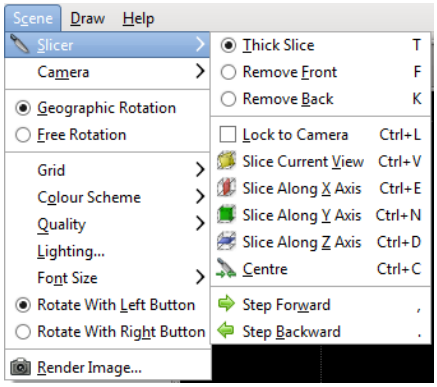
The **Scene** menu contains options that let you change how you navigate and view objects in the scene window:



These settings are described below.

Slicer Settings (Scene Menu)

The **Scene > Slicer** menu contains options that let you change the properties of slices made through the model with the slicer tool:



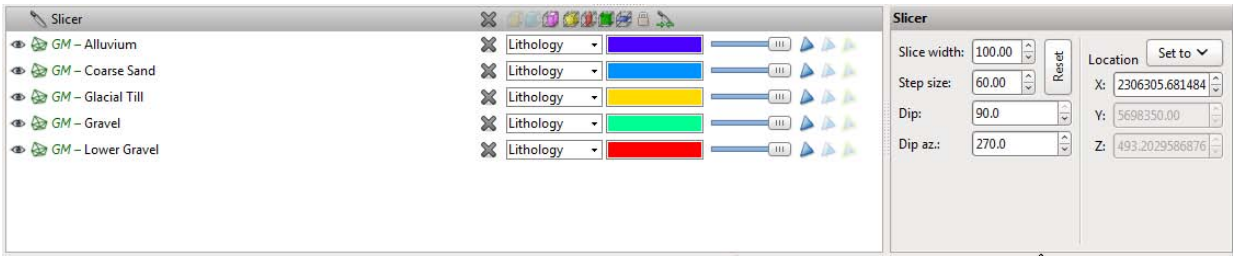
The slicer tool must be active for these settings to have any effect.

The **Thick Slice**, **Remove Front** and **Remove Back** options determine how the model is cut away from the slice.

The **Lock to Camera** option locks the slice to the current view, so that moving the scene changes the direction of the slice.

Other options in the **Scene > Slicer** menu slice the scene in specific directions, and the **Step Forward** and **Step Backward** options allow you to undo and redo slices.

The options in the **Scene > Slicer** menu can also be selected from the shape list:



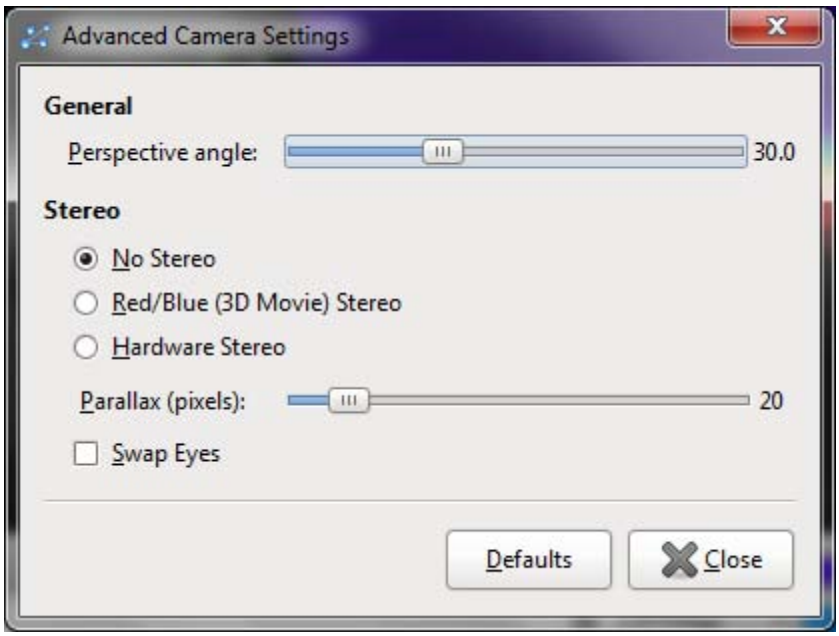
The icons displayed for the **Slicer** object correspond to the different menu options in the **Scene > Slicer** menu. Finer controls over the slice properties are also available in the properties frame.

Camera Settings (Scene Menu)

The **Scene > Camera > Perspective** and **Orthographic** options determine the view of the model in the scene window. Select the option you prefer. If **Perspective** is chosen, you can change the **Perspective angle** in the **Advanced Camera Settings** window, which is described below.

Other options in the **Scene > Camera** menu allow you to switch between various views of the scene.

Advanced camera options are set by selecting **Scene > Camera > Advanced**. The **Advanced Camera Settings** window will appear:



The **Perspective Angle** sets the camera when **Perspective** is chosen as the **Scene > Camera** option. This is similar to adjusting the zoom setting on a camera.

The **Stereo** options can be used when you have special stereoscopic accessories, such as red-blue glasses or special 3D viewing glasses. Pick the option for your accessory type.

The **Parallax** setting can be adjusted for the best 3D effect for your eyes.

If the 3D effect is not apparent, it may be that the 3D projection settings are opposite to those for your glasses. If this seems to be the case, tick the **Swap Eyes** option to see if this is corrected.

Changes made to these settings are automatically applied to the scene. To save the current settings and return to the main window, click **Close**. You can also return to the settings in place when the **Advanced Camera Settings** window was first opened by clicking **Defaults**.

Rotation Settings (Scene Menu)

The **Scene > Geographic Rotation** and **Free Rotation** options determine whether or not the model displayed in the scene window can be "rolled".

With the **Geographic Rotation** option, the azimuth and elevation can be varied, but the Z axis is constrained to be vertical. With the **Free Rotation** option, however, the rotation of the model is not fixed to any axis and the model can be freely rolled.

Select the option you prefer.

Grid Settings (Scene Menu)

The **Scene > Grid** options determine how the model grid is displayed in the scene window.

The **Show Grid** option determines whether or not the model grid is displayed. Select the option you prefer.

The **Loose Spacing**, **Normal Spacing** and **Fine Spacing** options determine the spacing of the grid.

The **Show Axis Lines** option determines whether or not the X, Y, Z coordinates are displayed in the scene. Tick the box to show East (red), North (green) and Elevation (blue) and extents of the data in the scene.

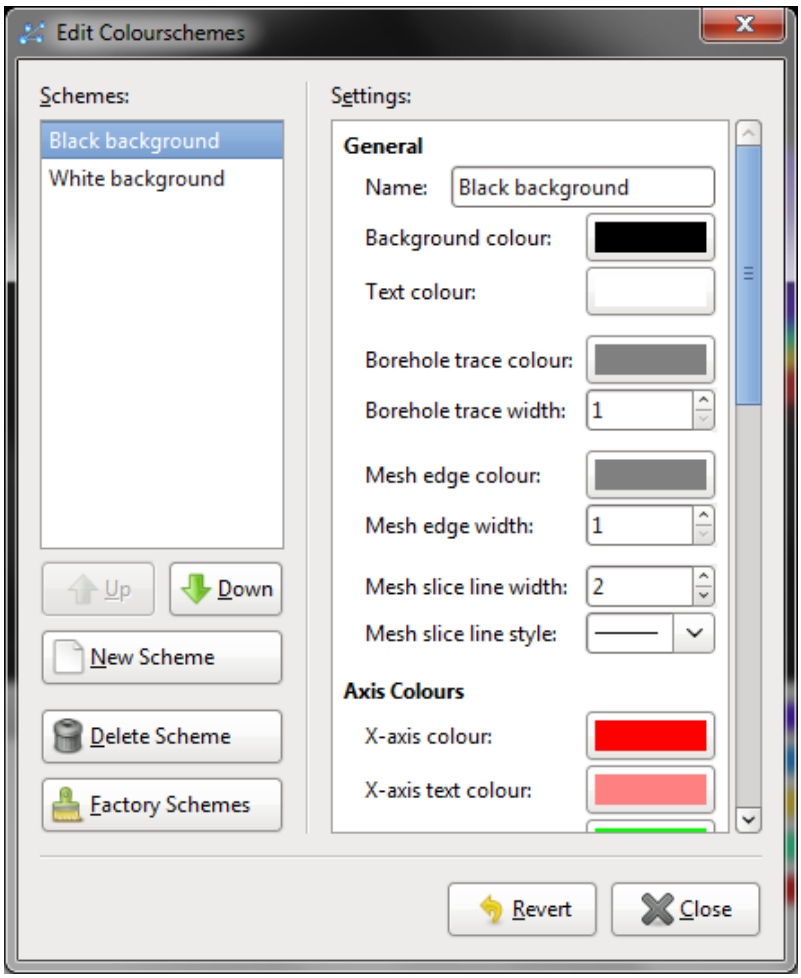
The **Numbers on Axes** option determines whether or not scale ticks are shown along the grid. These are automatically adjusted to fit the extent of the data and the current zoom setting. This option cannot be enabled if **Show Axis Lines** is disabled.

The **Show Axis Box** option determines whether or not the axes box that encloses the current data set is displayed. This option cannot be enabled if **Show Axis Line** is disabled.

Colour Scheme (Scene Menu)

The **Scene > Colour Scheme** options determine the colour scheme used in the scene window. There are two options in the menu, **Black Background** and **White Background**. **Black Background** is the default setting for new Leapfrog projects.

You can also create custom colour schemes by selecting **Scene > Colour Scheme > Edit**. The **Edit Colourschemes** window will appear:



In this window, you can define new colour schemes and set how they appear in the menu. The colour scheme that is placed at the top of the list will automatically be applied to the currently open project and will be applied by default for any new projects.

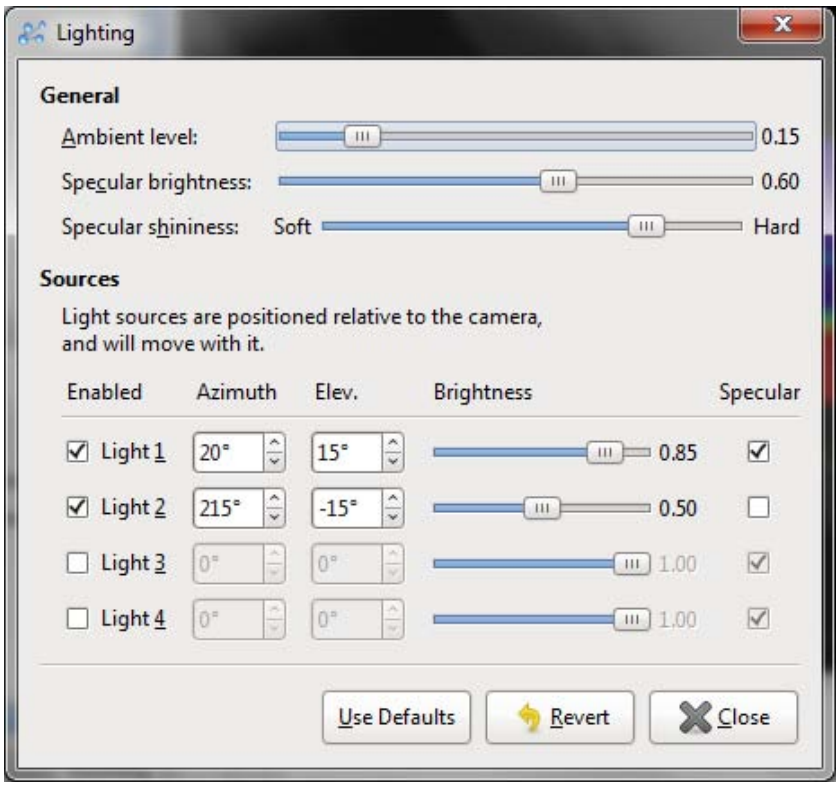
Quality (Scene Menu)

The **Scene > Quality** options determine how the data is displayed. If rotation and zooming seems sluggish, try selecting either the **Fast** or **Fastest** settings. If you have a good graphics card, try either the **High** or **Highest** settings to use the card's capabilities to provide better quality imaging.

The **Lower When Moving** option reduces the quality of the scene displayed when it is being moved. If you notice display problems when moving the scene, tick the box for this option.

Lighting (Scene Menu)

The **Scene > Lighting** options let you adjust preferences for visual effects in the scene window. You may be able to use these settings to emphasise significance in the data displayed in the scene. To adjust these settings, select **Scene > Lighting**. The **Lighting** window will appear:



The **Ambient level** setting determines the overall brightness of the scene.

The **Specular brightness** and **Specular shininess** settings determine how light appears to fall on the surfaces in the scene. The **Specular brightness** setting has a stronger effect when **Specular shininess** is soft.

You can have up to four light sources defined for a project. Select a light source by ticking the tick box for the source definition. By default, two light sources are defined.

Changes made to these settings are automatically applied to the scene. To save the current settings and return to the main window, click **Close**. You can also return to the settings in place when the **Lighting** window was first opened by clicking **Revert**, or restore Leapfrog Hydro's default lighting settings by clicking **Use Defaults**.

Font Size (Scene Menu)

The **Scene > Font Size** options determine the size of text used to display information on the grid and axes. Select the option you prefer.

Rotate Button Options (Scene Menu)

The **Scene > Rotate With Left Button** and **Rotate With Right Button** options determine how the mouse is used to rotate the scene.

When **Rotate With Left Button** is selected, you can rotate the scene by holding down the left mouse button and dragging in the scene. When **Rotate With Right Button** is selected, holding down the right mouse button and dragging rotates the scene.

When **Rotate With Left Button** is selected, clicking and holding the right mouse button can be used to zoom in and out of the scene. However, if **Rotate With Right Button** is selected, zooming and

rotating functions are not swapped. Instead, the left mouse button can only be used for clicking on different objects in the scene.

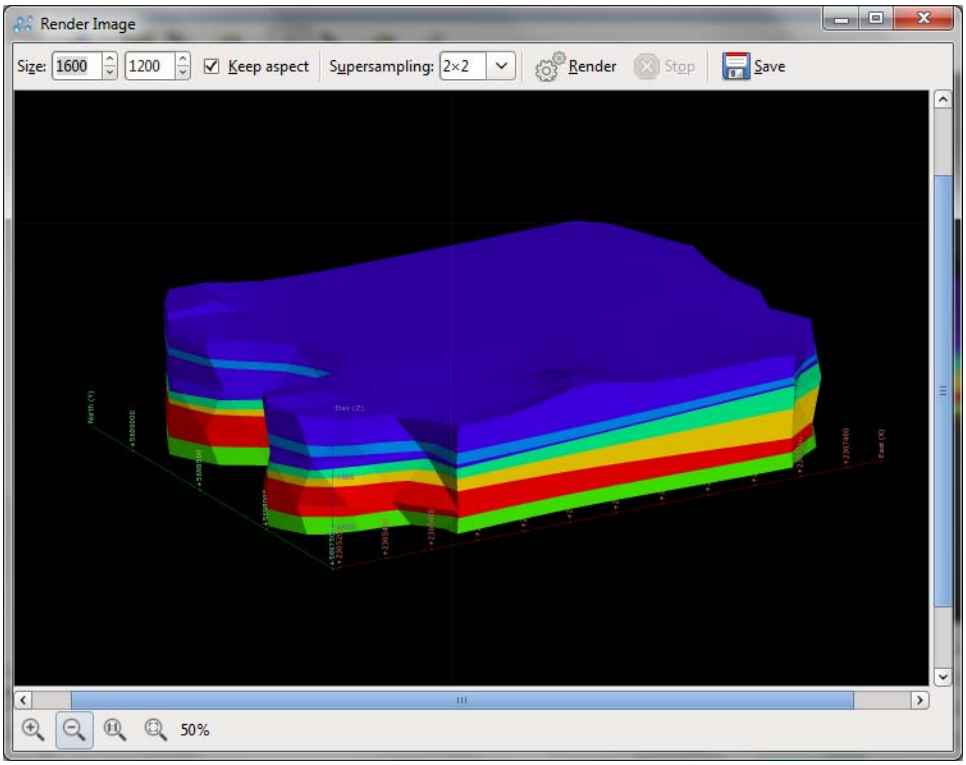
If you find that you tend to accidentally move the scene when you intend to click, consider selecting **Rotate With Right Button.**

Select the option you prefer.

Render Image (Scene Menu)

The **Scene > Render Image** option lets you save a copy of the current scene window on your computer.

To save an image, select **Scene > Render Image**. Leapfrog Hydro renders an image from the current scene window using current settings, then displays it in the **Render Image** window:

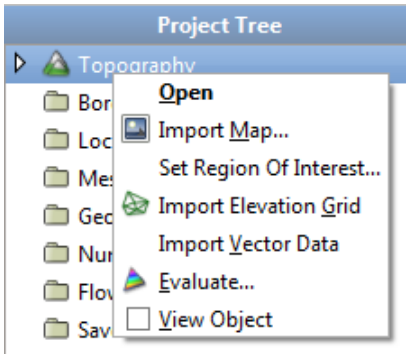


Use the zoom tools at the bottom of the window to view the rendered image. You can change settings, then render a new image by clicking **Render**. Once you are satisifed with the rendered image, click **Save**. You will be prompted to enter a filename and location.

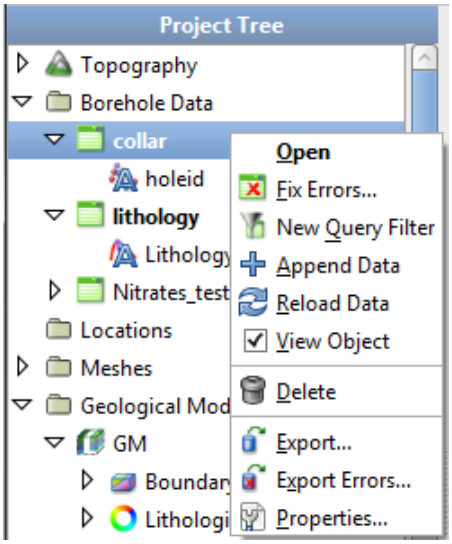
Working with the Project Tree

When a project is created, a series of objects is added to the project tree. These objects are used to organize imported items such as maps, images and data sets into categories. These objects also provide tools that let you import information into the project and generate models.

Right-click on each object to view the actions you can perform for that object. For example, you can use the **Topography** object to import a map, elevation grid, and vector data, set the region of interest, and evaluate topography:



You can also right-click on objects within the top-level objects to view more information on that object or carry out actions specific to that object. For example, right-clicking on a table of imported borehole data reveals a menu showing possible actions:



When a bold option appears in the right-click menu, as **Open** does in the menu above, that option can also be selected by double-clicking on the object.

Objects shown in bold text in the project tree are currently displayed in the scene window. In the image above, the collar and lithology tables are displayed in bold text, indicating that they are visible in the scene window.

Some actions are not available until data has been imported into the project.

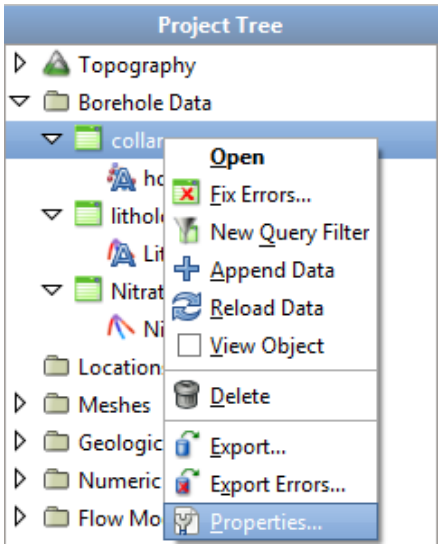
The order in which objects appear in the project tree roughly reflects the typical Leapfrog Hydro workflow. These high-level objects are:

- **Topography**. Stores information used to define the area in which calculations will be carried out, the region of interest. With the **Topography** object, you can import topography and elevation such as maps and digital elevation models. For more information, see Tutorial 2: Defining the Topography and The Topography Object.
- **Borehole Data**. Stores imported borehole data and provides options for viewing and correcting borehole data. For more information, see Tutorial 3: Importing and Working with Borehole Data and The Borehole Data Object.
- **Locations**. Stores location information you can use to define surfaces. You can also import data, such as distribution of contaminants or temperature, for interpolation as part of a numeric model.

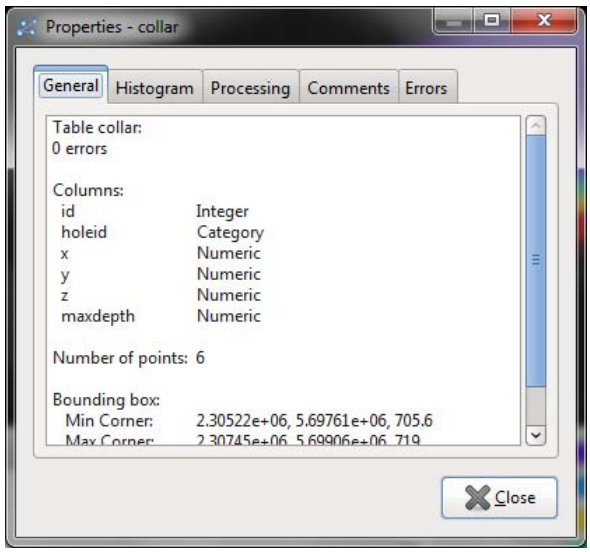
- **Meshes**. Imports and stores meshes and digital elevation models created in other modeling packages.
- **Geological Models**. Builds geological models of a defined area. Geological models are usually based on borehole data.
- **Numeric Models**. Builds models of the physical properties of an area built up from either borehole data that contains information about those properties or location information associated with those properties.
- **Flow Models**. Builds flow models for calculating groundwater flow and/or water distribution. Models can be exported for use in other applications. Leapfrog can also import flow models built using other applications.
- **Saved Scenes and Movies**. Stores saved scenes. With the **Saved Scenes and Movies** object, you can create scenes to highlight an important observations for later comparison, create movies, and export scenes and movies for viewing in other applications.
- **Cross Sections**. Creates cross sections from imported images, georeferenced data or from shapes drawn in Leapfrog.

Viewing Object Properties

You can view the properties of imported and generated objects in the project tree by right-clicking on an object and selecting **Properties**:



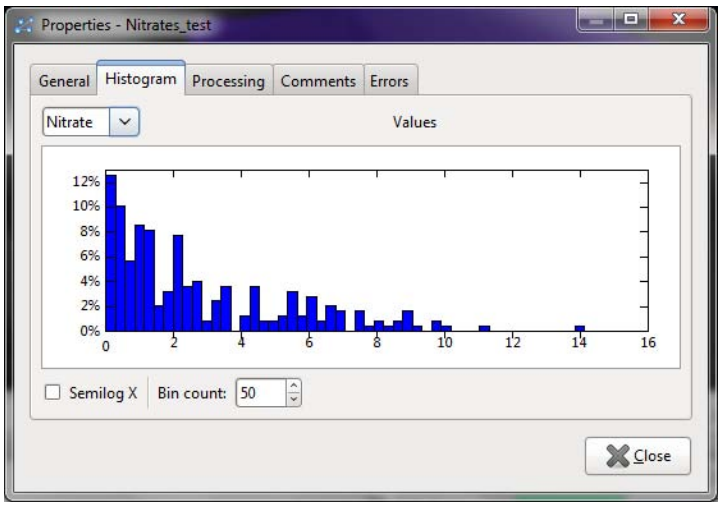
The information displayed in the properties window depends on the object selected. For example, the properties for a collar table will appear as:



The **Processing** tab shows the current status of processing (queued to process, processing, finished). The **Errors** tab shows any errors that occurred while processing the object.

In the **Comments** tab, you can enter information on the object, which can be helpful when the project contains many objects or when several different people are working on a model.

Some objects may display other tabs in the **Properties** window. For example, table objects will have a **General** tab that shows the table's data structure. Interval tables and numeric data tables may also have a **Histogram** tab that shows the distribution of the data in a particular column:



Working with Objects in the Scene

The scene window displays a 3D representation of selected data from the project tree. Objects from the project tree can be added to the scene in two ways:

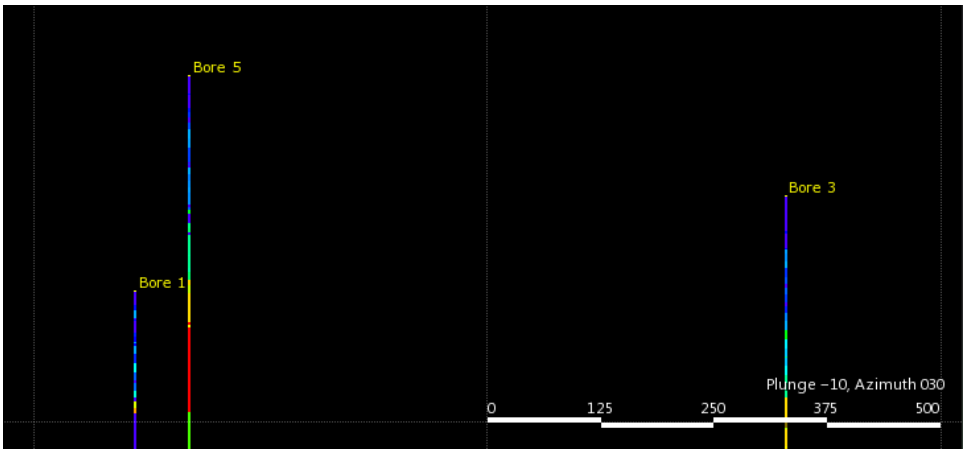
- Right-click on the object and tick the **View Object** box.
- Click on the object and drag it into the scene.

Once objects have added to the scene window, you can use the shape list and the shape properties frame to change the way those objects appear in the scene. See Changing the Appearance of Objects in the Scene Window.

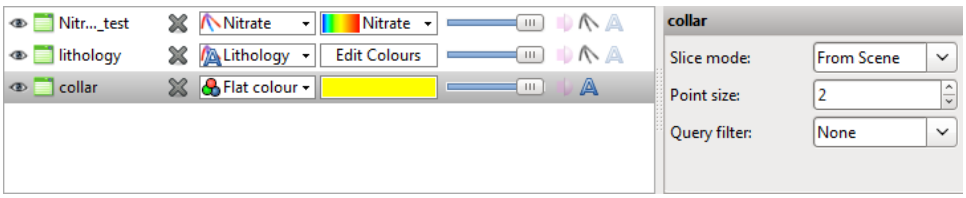
Changing the Appearance of Objects in the Scene Window

Once objects have beenadded to the scene window, you can use controls in the shape list and the shape properties frame to change the way objects are displayed. This is useful in making important data more apparent in the scene.

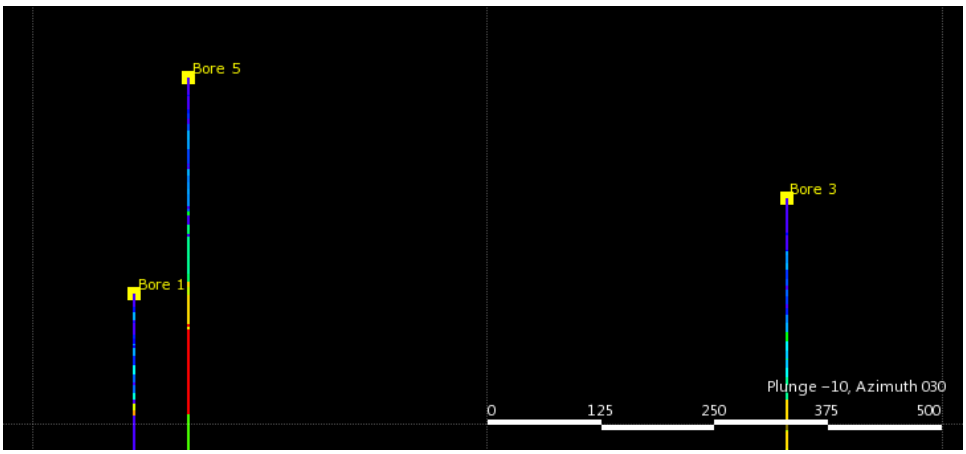
For example, when borehole data is first added to a scene, the collars are displayed as small points:



To change the size of the collars, select the collar object in the shape list:



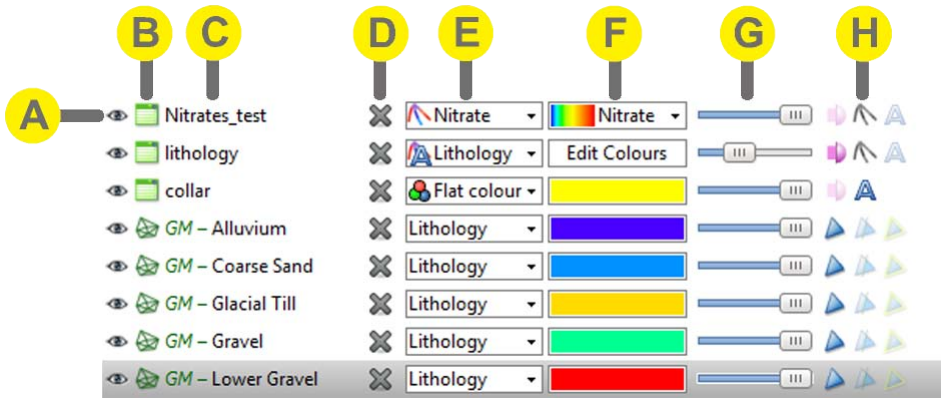
The **Point size** field in the properties window is 2. Increasing it to 10 changes the display of the collars, making them more apparent in the scene:



See Shape List Controls for more information on the properties that can be changed using the shape list.

Shape List Controls

The controls available in the shape list are:



A Object visibility. Click to make the object visible or invisible in the scene.

B Object type

C Object name

D Delete control. Click this to remove the object from the scene.

E View list

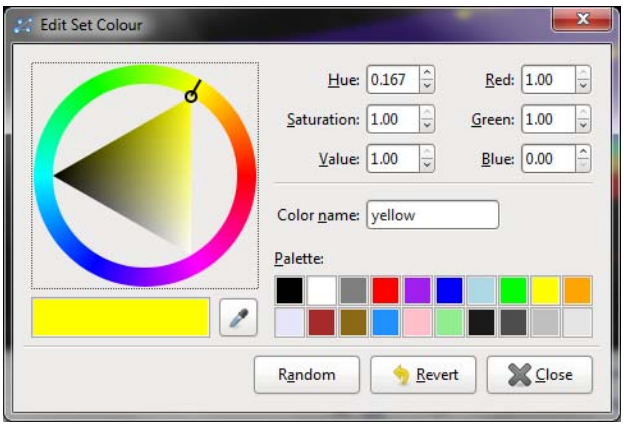
F Colour list

G Opacity slider

H Property icons. The icons that appear depend on the type of object in the scene.

Changing an Object's Display Colour

Many objects viewed in the scene are displayed using a single colour. To change the colour, add the object to the scene window, then click on the colour chip in the shape list. An **Edit Set Colour** window will be displayed:

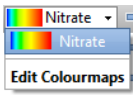


Use the controls in the window to choose a new colour. You can also choose a colour chip from the **Palette** or set a random colour by clicking on the **Random** button.

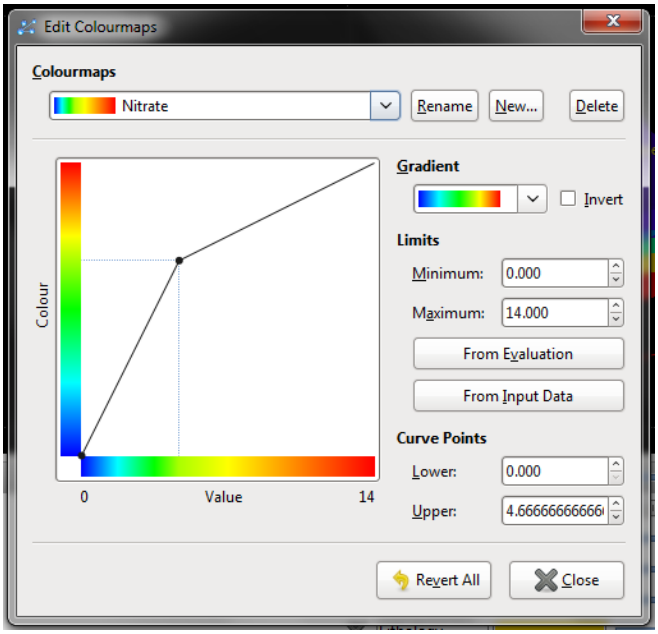
Changes made to the colour are automatically applied to the scene. To save the currently displayed colour and return to the main window, click **Close**. You can also revert to the colour assigned when the window was first opened by clicking **Revert**.

Changing a Colour Gradient

Some objects viewed in the scene are displayed using a colour gradient. To change the gradient, add the object to the scene window. Click on the colour gradient in the shape list, then select **Edit Colourmaps**:



The **Edit Colourmaps** window will appear:

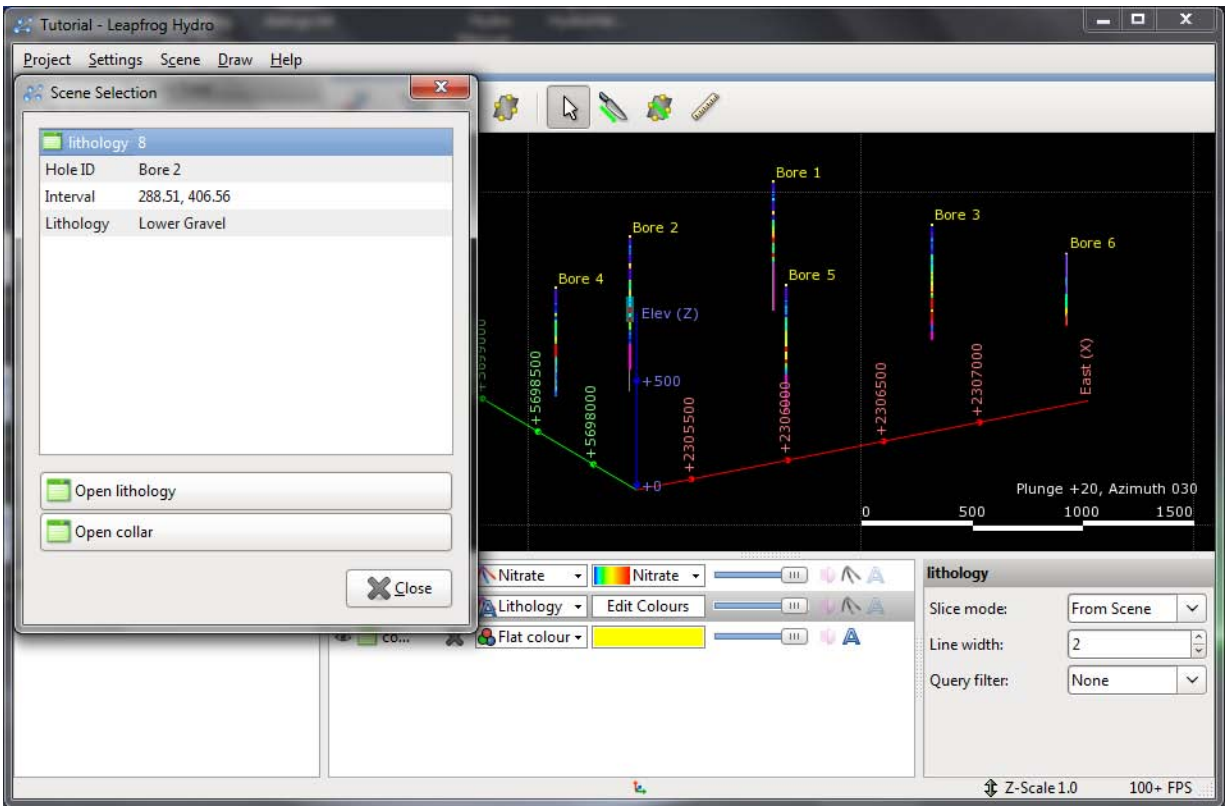


Use the controls in the window to choose new colours for the gradient.

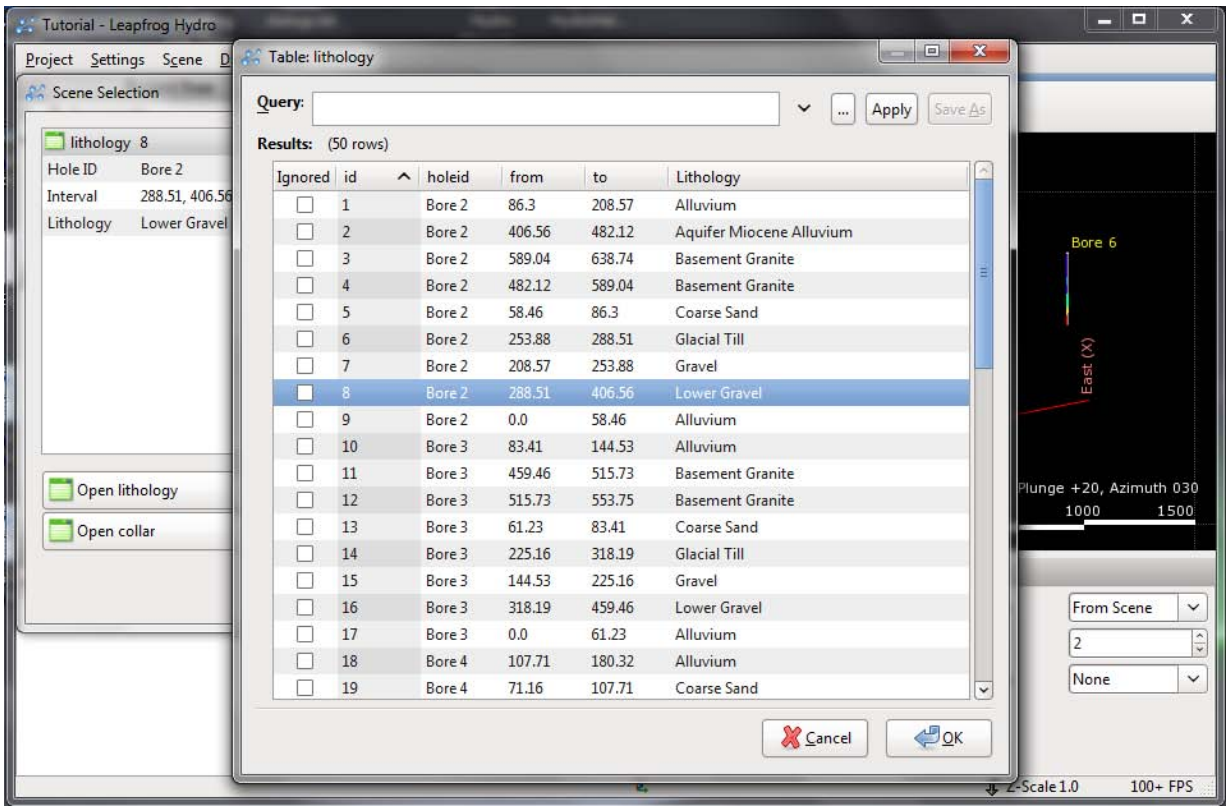
Changes made to the gradient are automatically applied to the scene. To save the currently displayed gradient and return to the main window, click **Close**. You can also revert to the original gradient by clicking **Revert**.

Viewing Data in the Scene

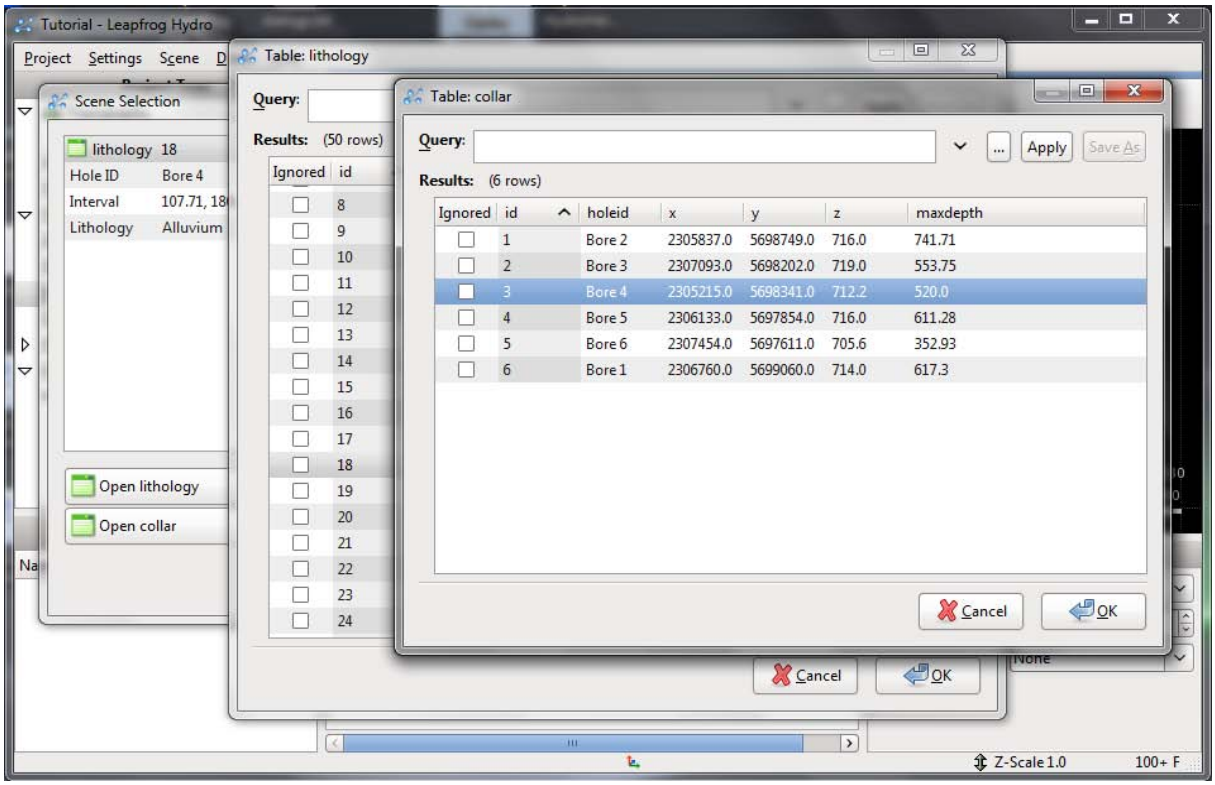
For any object that appears in a scene, relevant data can be accessed by clicking on the object. For example, when borehole data is displayed in the scene, clicking on a segment will display the corresponding data:



These scene selection windows also provide access to the data tables behind the individual data points. For example, in the screen above, the **Open lithology** and **Open collar** buttons open the tables relevant to the selected segment:

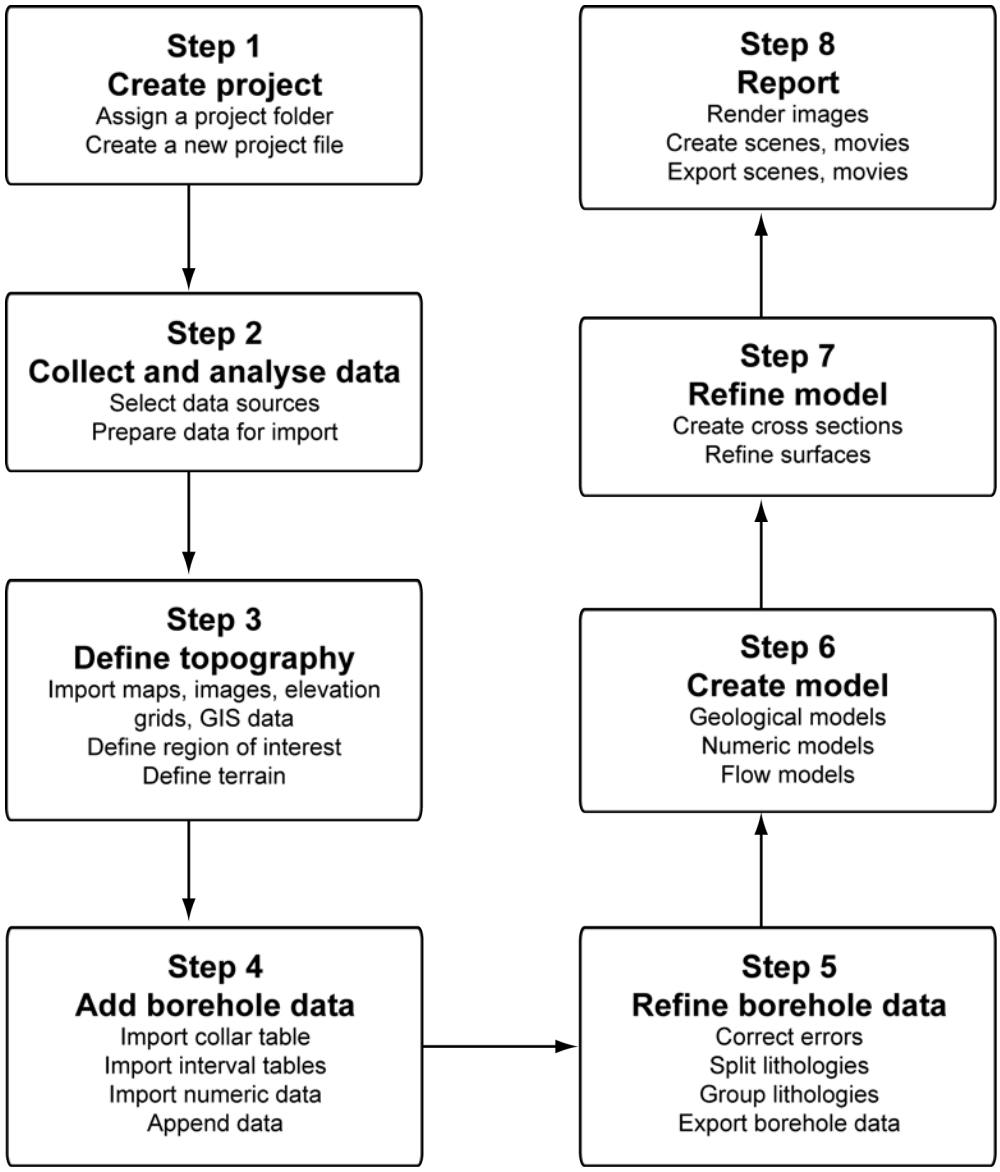


You can have more than one table open at a time, and easily switch between tables:



The Typical Leapfrog Hydro Workflow

There are many possible approaches to creating geological models in Leapfrog Hydro. A typical workflow, using borehole data to create models, is shown below:



These steps are reflected in the arrangement of objects in the project tree.

Each of these steps is described briefly below.

Step 1 – Create Project

In this step, choose a folder to use as the project folder and, in Leapfrog Hydro, create a new project.

Step 2 – Collect and Analyse Data

In this step, select what data to use in the project and do any processing necessary to ensure the validity of the data sources.

Although the validity of data is important in creating geological models, it is not necessary to ensure that data is perfectly valid before importing it, as Leapfrog has tools and techniques for correcting and working with errors in imported data. Information corrected in Leapfrog can also be exported.

Step 3 – Define Topography

In this step, define the topography for the geological model. The topography represents the ground surface and all data associated with it.

Maps and other images, elevation grids and GIS can be used to define the topography.

Step 4 – Add Borehole Data

Borehole data forms the basis of models created in Leapfrog.

Leapfrog expects borehole data that is stored in a collar table and interval tables. Each project can have only one collar file, but multiple interval tables can be imported. It is also possible to append data from various sources to make up the project's collar table and interval tables.

Step 5 – Refine Borehole Data

As borehole data often contains errors that prevent the computation of an accurate model, Leapfrog also has tools that help you to correct errors in the data. You can also export data corrected in Leapfrog.

Step 6 – Create Models

In this step, geological models, numeric models and flow models are created using imported data and images.

Step 7 – Refine Models

Once a basic model has been created, refine the model by adjusting surfaces.

Step 8 – Report

Leapfrog has several tools for reporting the results of modeling. You can render images, create and annotate specific scenes and add a number of scenes together to create a movie.

Images, scenes and movies can be exported for use in other applications, including the stand-alone Hydro Scene Viewer.

Other Workflows

Leapfrog Hydro supports many possible workflows, and it is not strictly necessary to begin building a model by building the topography, then importing borehole data. For example, you may wish to build a geological model from a series of meshes. If this was the case, you could use the **Meshes** object to import the meshes, then work down through the project tree to build a geological model, create cross sections and export scenes and movies.

Part 4: Leapfrog HydroTutorials

In this part, a series of tutorials based on a small data set will introduce you to basic concepts in Leapfrog Hydro and get you to the point where you can start processing your own data. The tutorials take two to four hours to complete.

Tutorial files are copied to your computer's hard drive when Leapfrog Hydro is installed. [more info required about a specific location, or at least sufficient information for the user to find the files readily]

The tutorials are:

- Tutorial 1: Working with Projects
- Tutorial 2: Defining the Topography
- Tutorial 3: Importing and Working with Borehole Data
- Tutorial 4: Building a Simple Geological Model
- Tutorial 5: Building a Simple Numeric Model

Tutorial 1: Working with Projects

In this tutorial, you will create a new project and learn about project files and opening existing projects.

Topics are:

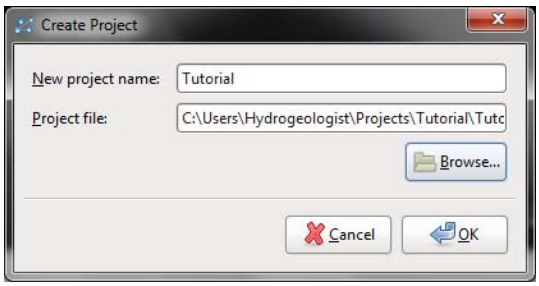
- Creating a New Project
- Project Files
- The Parts of a New Project
- Saving Projects
- Opening Projects
- Compacting Projects

You will need a copy of the tutorial files that were copied to your computer when Leapfrog Hydro was installed. To do this:

- Go to the location where Leapfrog Hydro was installed.
- Copy the Tutorials folder to your My Documents folder.

Creating a New Project

To create a new project, start Leapfrog Hydro and select **Project > New**. The **Create Project** window will appear.

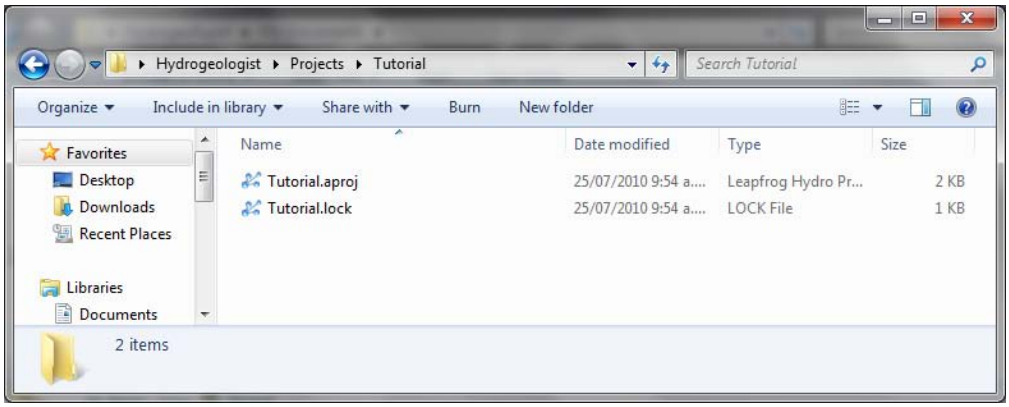


Enter a name for the new project and click **Browse** to select a folder where the project will be saved.

Click **OK** to create the project.

Project Files

If you open the folder you selected, you will see that a new folder has been created using the name you gave for the tutorial. Open that folder to view the project file Leapfrog Hydro has created, which in this case is called `Tutorial.aproj`, as shown below:



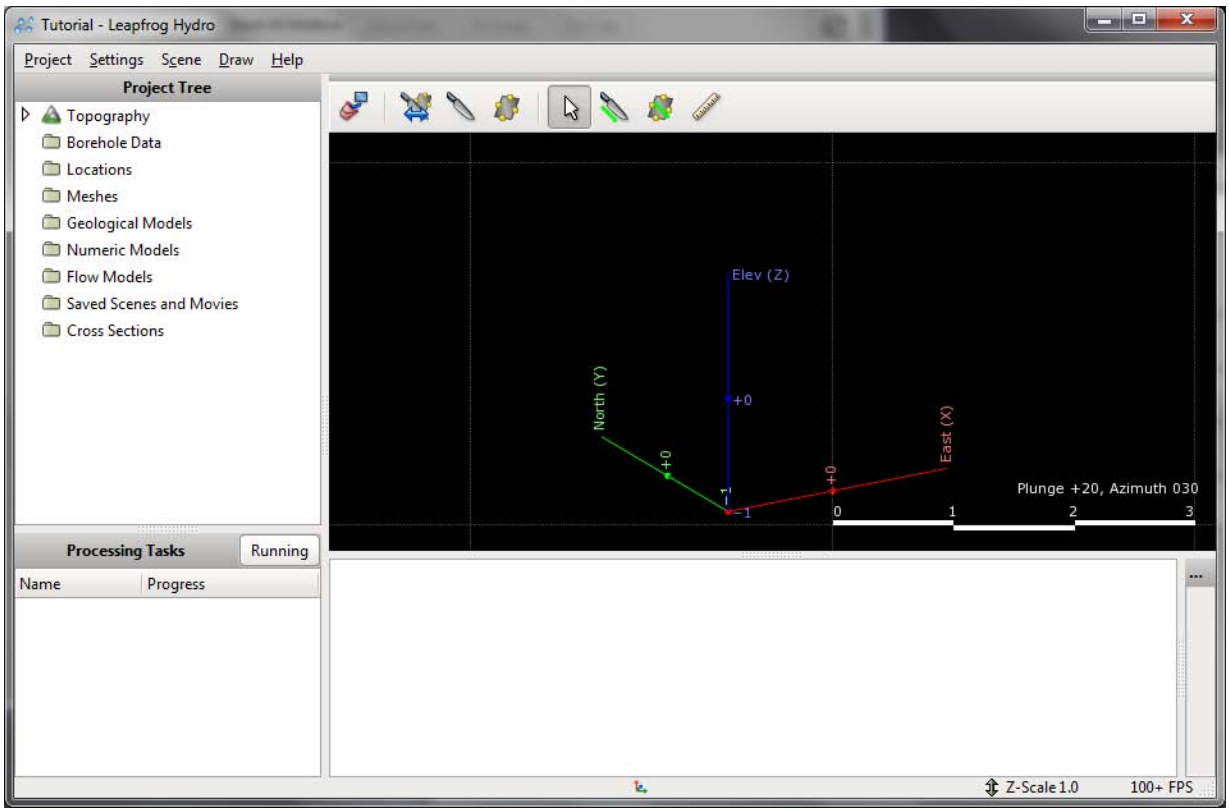
The `Tutorial.lock` file is created when the project is opened and closed when the project is closed. Later, once you have added borehole data to the project, there will be an additional file, ending with .apbd. The `.apbd` file contains all the borehole data imported into a Leapfrogproject.

To learn more see:

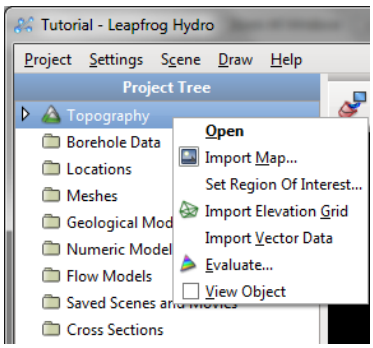
- Compacting Projects

The Parts of a New Project

In the Leapfrog window, you will see the new project with its blank scene window:



A series of objects have been added to the project tree. These objects are used to organize items such as maps, images and data sets into categories. These objects also provide tools that let you import information into the project and generate models. Right-click on each object to view the actions you can perform for that object. For example, from the **Topography** object, you can import a map, elevation grid, and vector data, set the region of interest, and evaluate topography:



The order in which the objects appear in the project tree roughly reflects the typical Leapfrog work-flow. See The Typical Leapfrog Hydro Workflow for more information.

These objects are:

- **Topography**. Stores information used to define the area in which calculations will be carried out, the region of interest. With the **Topography** object, you can import topography and elevation such as maps and digital elevation models.
- **Borehole Data**. Stores imported borehole data and provides options for viewing and correcting borehole data.

- **Locations**. Stores location information you can use to define surfaces. You can also import data, such as distribution of contaminants or temperature, for interpolation as part of a numeric model.
- **Meshes**. Imports and stores meshes and digital elevation models created in other modeling packages.
- **Geological Models**. Builds geological models of a defined area. Geological models are usually based on borehole data.
- **Numeric Models**. Builds models of the physical properties of an area built up from either borehole data that contains information about those properties or location information associated with those properties.
- **Flow Models**. Builds flow models for calculating groundwater flow and/or water distribution. Models can be exported for use in other applications. Leapfrog can also import flow models built using other applications.
- **Saved Scenes and Movies**. Stores saved scenes. With the **Saved Scenes and Movies** object, you can create scenes to highlight an important observations for later comparison, create movies, and export scenes and movies for viewing in other applications.
- **Cross Sections**. Creates cross sections from imported images, georeferenced data or from shapes drawn in Leapfrog.

Saving Projects

Leapfrog Hydro saves an open project each time a processing task has been completed or settings have changed. Projects are also saved when they are closed so that scene settings can be restored when the project is next opened.

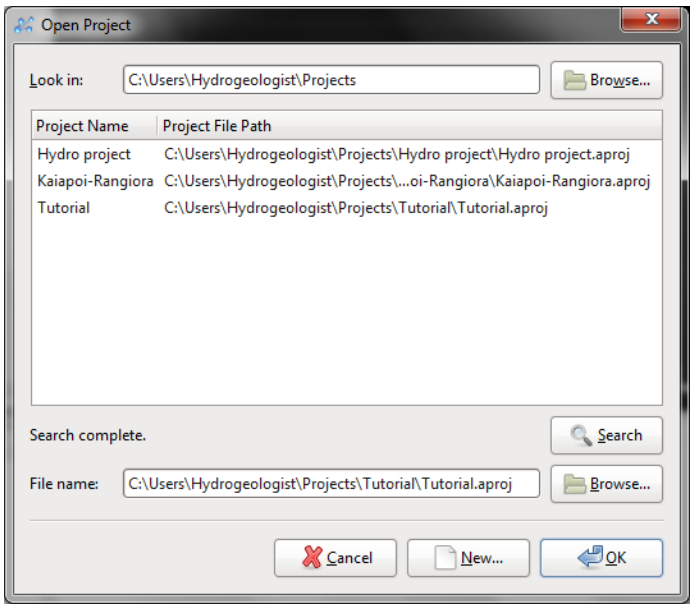
To learn more see:

- Compacting Projects

Opening Projects

If Leapfrog is not already running, you can open a project by doubling clicking on the **.aproj** file.

If Leapfrog is already running and another project is open, open a different project by selecting **Project > Open**. The **Open Project** window will appear:



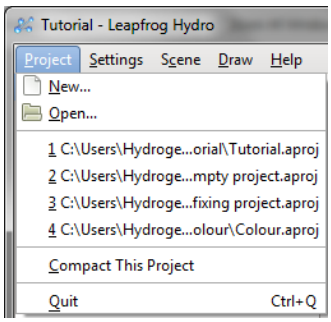
Leapfrog automatically displays a list of all the projects in the folders in the **Look in** folder. You can change this folder by clicking on **Browse**. The **Search** button simply refreshes the list of projects that are found in the **Look in** folder. Click on it if you have copied files to the **Look in** folder since opening the **Open Project** window.

You can also navigate directly to the required project file by clicking on the **Browse** button adjacent to the **File Name**. Choosing a project in this way does not change the **Look in** folder, which means that next time you want to open a project, Leapfrog will first look in that folder.

The **Look in** folder is useful if you have one folder in which you keep most of your Leapfrog projects. You can set that as the **Look in** folder. If, however, you have copied a project to your desktop and you wish to open that project, you can do so using the **Browse** button adjacent to the **File name** field.

The **Look in** folder is also the default folder in which new projects will be created.

You can also choose a project from the list of recently open projects:



Only one copy of Leapfrog can be running at a time, which is the reason why double-clicking on a project when another is already open will not result in the new project being opened.

Compacting Projects

When you delete objects from a project file, Leapfrog retains those objects but notes that they are no longer used. Over time, the project file will grow in size and data stored in the database may become fragmented.

Compacting a project removes these unused objects and any unused space from the database. When you compact a project, Leapfrog will close the project and back it up before compacting it. Depending on the size of the project, compacting it may take several minutes. Leapfrog will then reopen the project.

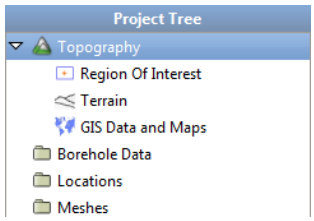
To compact a project, select **Project > Compact This Project**. You will be asked to confirm your choice.

Tutorial 2: Defining the Topography

[pic of workflow with Topography highlighted]

The first step in creating a model is to define the topography, which represents the ground surface and all data associated with it.

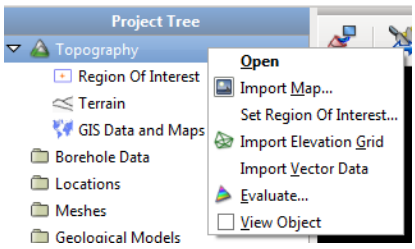
In Leapfrog, the **Topography** object is the first object in the project tree:



It is made up of:

- **Region of Interest**. This object represents the area in which data will be added and models created. The Region of Interest is important in limiting what data Leapfrog uses to perform calculations. Leapfrog automatically sets a region of interest, but for models with large data sets, adjusting the region of interest will reduce the amount of time Leapfrog requires for performing calculations.
- **Terrain**. This object defines the height of the data. The Terrain can be derived from a single DTM, but a variety of sources can be used.
- **GIS Data and Maps**. This object contains all maps, photos and vector data that define land use.

All tools for defining the topography are accessed by right-clicking on the **Topography** object and the items in it:



Imported objects and objects created while building the topography are also stored as part of the topography object, and more complex functions can be carried out by right-clicking on individual objects. In this tutorial, however, all functions required can be accessed by right-clicking on the **Topography** object itself.

In this tutorial, you will define the topography from:

- a map and related georeference data
- a DTM file
- GIS vector data

This information can be found in the `Tutorial` folder you started a project in as part of Tutorial 1: Working with Projects, in the folders "Maps", "GIS data" and "Elevation data".

Topics are:

- Defining the Region of Interest
- Defining the Terrain
- Adding Information About Land Use

Defining the Region of Interest

The first step in creating the topography is to define the region of interest. The best way to start this is to import a map or aerial photograph. The region of interest can also be defined by importing GIS or borehole data, but importing an image is faster and provides a visual reference that is helpful in working with data added to the project in the future.

Leapfrog automatically defines a region of interest, which will grow to fit any data imported. This can, however, result in a large area in which calculations are performed.

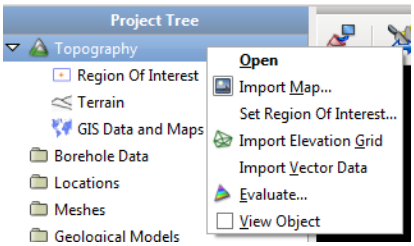
For this tutorial, defining the region of interest requires four steps:

- Importing a Map
- Adding Georeference Data to the Map
- Displaying the Map
- Adjusting the Region of Interest

The files for this part of the tutorial can be found in the `Maps` folder.

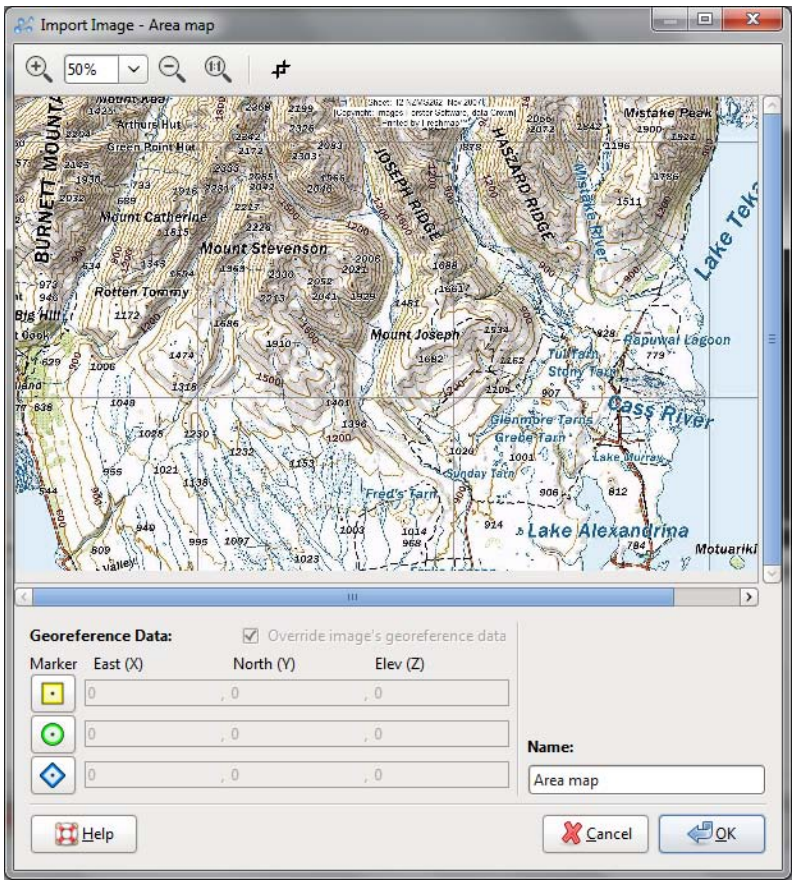
Importing a Map

To import a map, right-click on the **Topography** object and select the **Import Map** option:



In the window that appears, select the file in the `Maps` folder called `Area map.jpg`, then click **OK**.

The **Import Image** window then displays the image:



You can crop the image, if required.

The **Name** field displays the name of the imported file. If you wish to use a more descriptive name in the project file, enter it in this field.

The next step is to add georeference data to the map. See Adding Georeference Data to the Map.

Adding Georeference Data to the Map

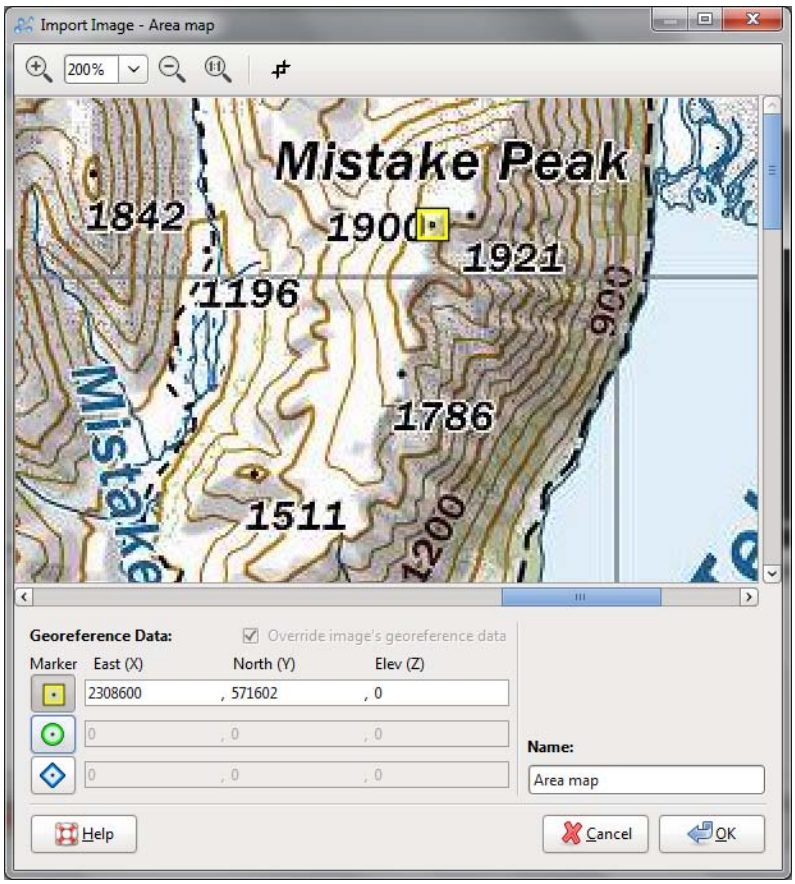
Adding georeference data to the map will set the coordinates for the project as a whole.

The first step in adding georeference data to an image is to add three reference markers. There are two ways to add the markers to the image:

- Click on a marker to add the marker to the image.
- Drag a marker and drop it onto the map.

Whichever method is used, you can move the marker to the required position by dragging it or by using the arrow keys. You can also use the zoom controls for more precise control over the area in which you are positioning a marker.

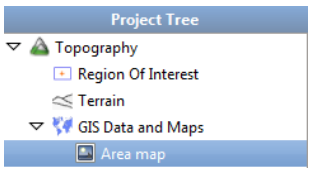
To select a different marker, click on it in the list, then move it using the mouse or the arrow keys. Once each marker is correctly positioned, enter the real-world X and Y coordinates for each marker:



This information is contained in the file `Georeference Information.txt` in the `Maps` folder. The elevation (Z) value should be the same for each marker.

If the image imported contains georeference data, that data will automatically be displayed. See Importing Maps for more information.

Once you have set the coordinates for each marker, click **OK** to return to the main window. Once Leapfrog saves the map, it will appear in the project tree under the **GIS Data and Maps** object:



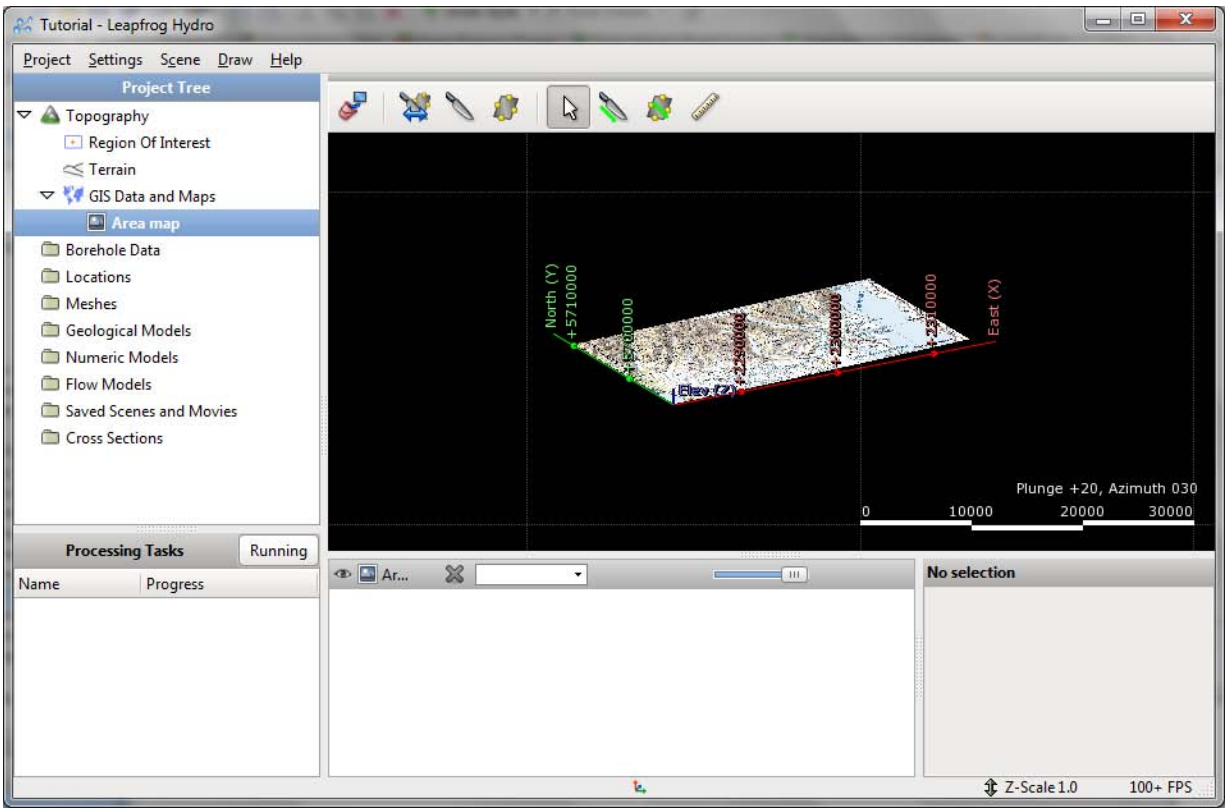
To display the map, see Displaying the Map.

Displaying the Map

There are two ways to display the imported map:

- Click on the image in the project tree and drag it into the scene.
- Right-click on the map in the project tree and tick **View Object**.

The map will then appear in the scene:



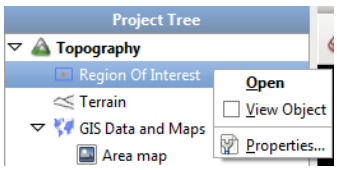
Notice, also, that the image is now listed in the shape list.

The next step is to adjust the region of interest. See Adjusting the Region of Interest.

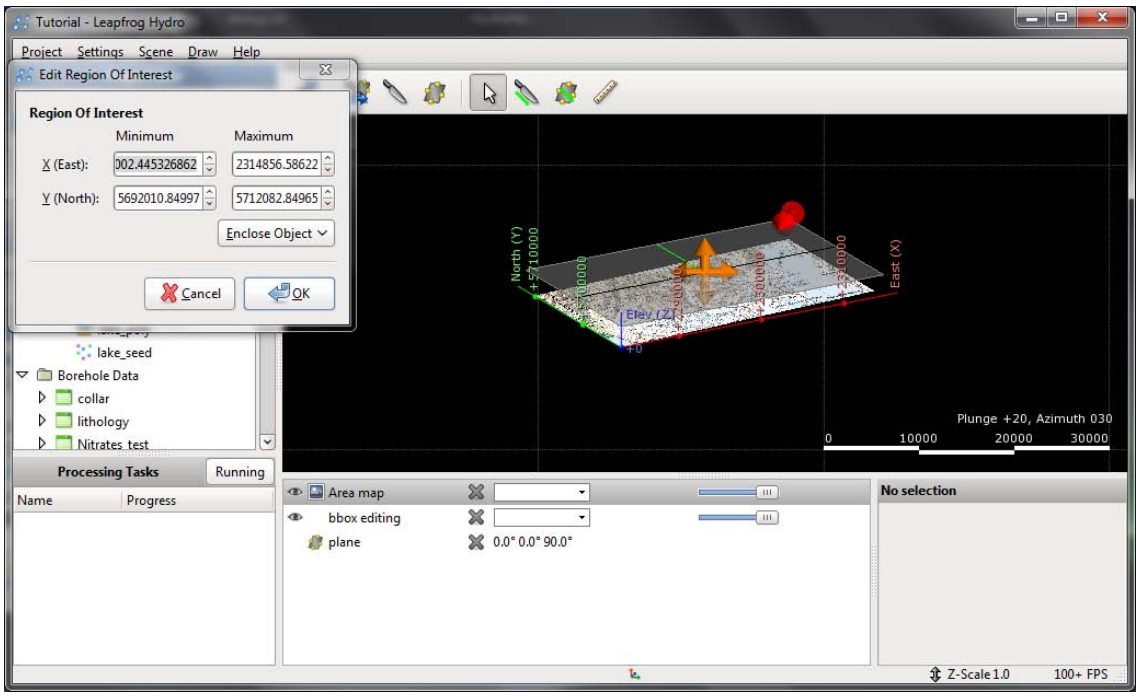
Adjusting the Region of Interest

When a project is created, Leapfrog automatically sets a region of interest. As data is added to the project, the Region of Interest changes to fit that data. For models with large data sets, adjusting the region of interest will reduce the amount of time Leapfrog requires for performing calculations.

To adjust the region of interest, double-click on the **Region of Interest** object in the project tree or right-click and select **Open**:



The **Edit Region of Interest** window will be displayed, together with controls in the scene that will help you to set the region of interest:



There are three ways to define the rectangular volume of interest:

- Enter the coordinates.
- Select **Enclose Object** and choose from the list of objects in the project. The region of interest will be updated to the size of the selected object.
- Use the controls that appear in the scene. The orange handle adjusts the center of the plane and the red handle adjust the size.

When you have finished adjusting the region of interest, click **OK**.

The next step is to define the terrain. See Defining the Terrain.

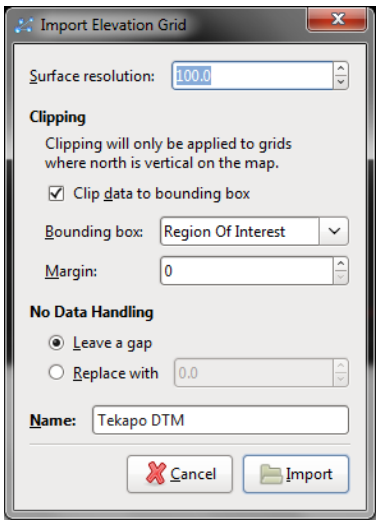
Defining the Terrain

In Leapfrog, the **Terrain** object is used to define the elevation of the ground's surface. The terrain will often be defined by a single digital terrain model (DTM), but there may be circumstances in which data from several sources is combined to ensure the data is consistent.

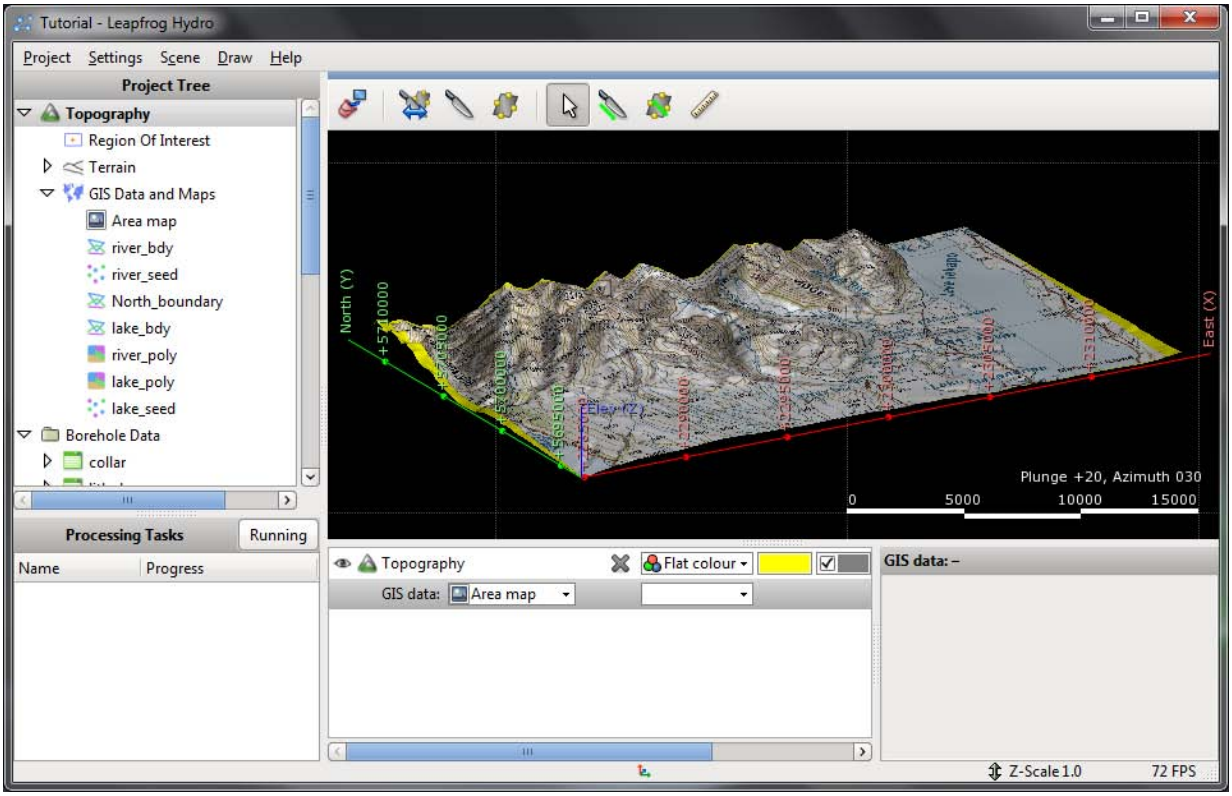
The terrain is used to bound geological models and to position GIS data onto the topography. As a result, changing the terrain will cause geological models to be recomputed and GIS data to be repositioned. This can be time-consuming, which is why the terrain should be defined before GIS data is added to the project and before any models are created.

For this tutorial, a single DTM file will be used to define the terrain. It is called `Tekapo DTM.asc` and can be found in the `Elevation data` folder.

To import an elevation grid, right-click on the **Topography** object in the project tree and select **Import Elevation Grid**. Navigate to the folder that contains the `Tekapo DTM.asc` file and open the file. The **Import Elevation Grid** window will be displayed:



Click **Import**. The topography is displayed in three dimensions:



Now that the terrain is defined, maps will be automatically draped over the topography.
The imported mesh is stored under the **Terrain** object. A copy is also stored in the **Meshes** folder.
The next step is to add information about land use. See Adding Information About Land Use.

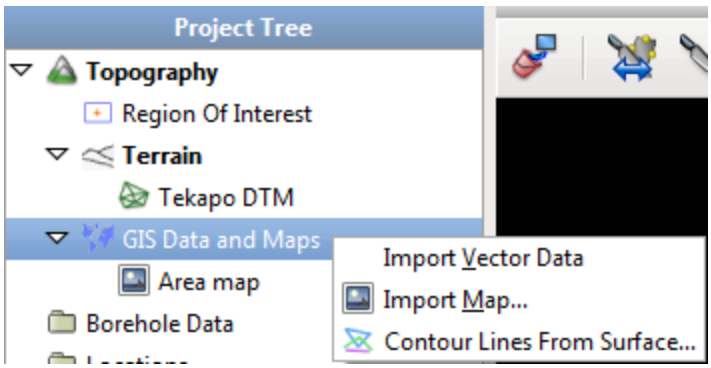
Adding Information About Land Use

In Leapfrog, vector data can be imported to define land use in two dimensions.
For this tutorial, vector data can be found in the `GIS data` folder.
See:

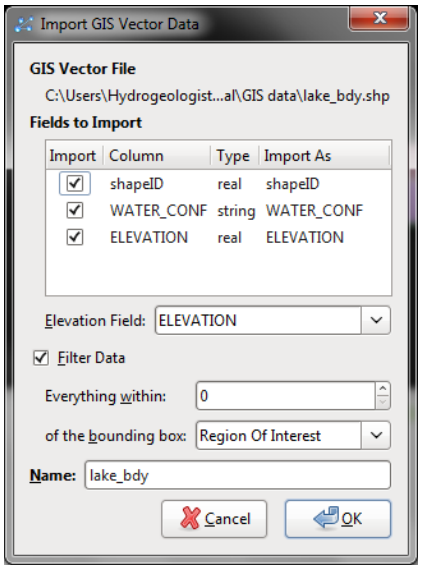
- Importing GIS Data
- Viewing GIS Data

Importing GIS Data

To import vector data, right-click on the **GIS Data and Maps** object and select **Import Vector Data**:

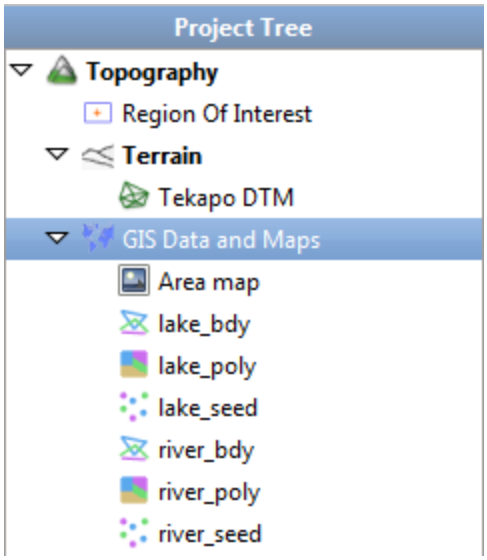


Navigate to the folder that contains GIS data files and import the files, one at a time. For each file, the **Import GIS Vector Data** window will be displayed:



The options in this window are discussed further in Importing Vector Data. Click **OK** to import the file.

The imported objects are stored under the **GIS Data and Maps** object:

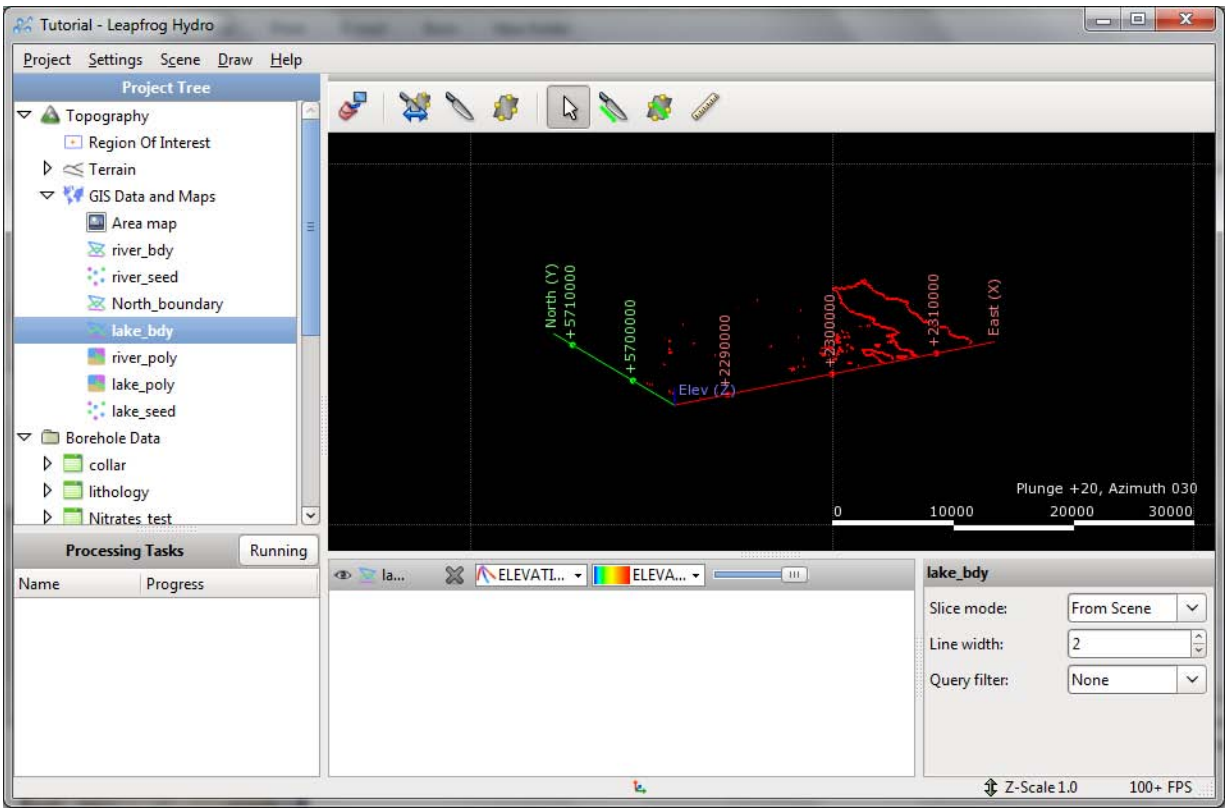


Different icons indicate point, line and polygon data.

Viewing GIS Data

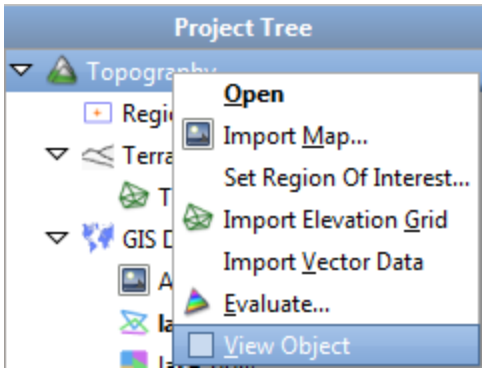
There are two ways to view imported GIS data. The first is to view them independently of the topography.

To view data in this way, either drag and drop the object into the scene or right-click on the object and select **View Object**. In the scene below, only the lake_bdy object is visible in the scene:

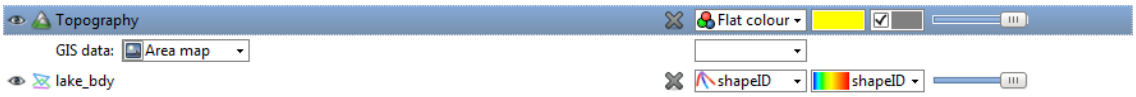


The second way to view GIS data is in combination with selected items of the topography. To do this, add a GIS data object to the scene by dragging and dropping it into the scene or by right-clicking on the object and selecting View Object.

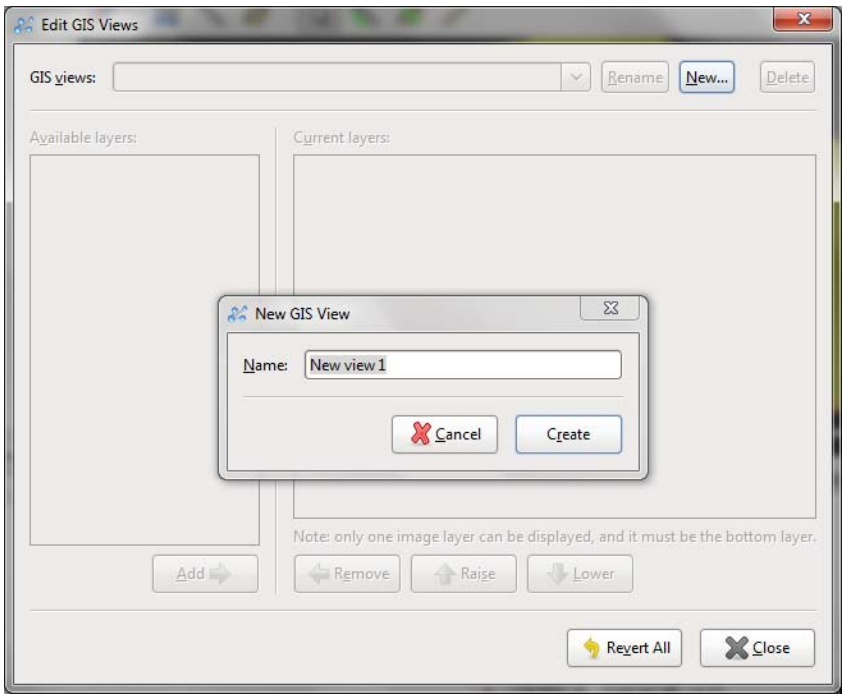
Add the **Topography** object to the scene by dragging and dropping it into the scene or by right-clicking on it and selecting **View Object**:



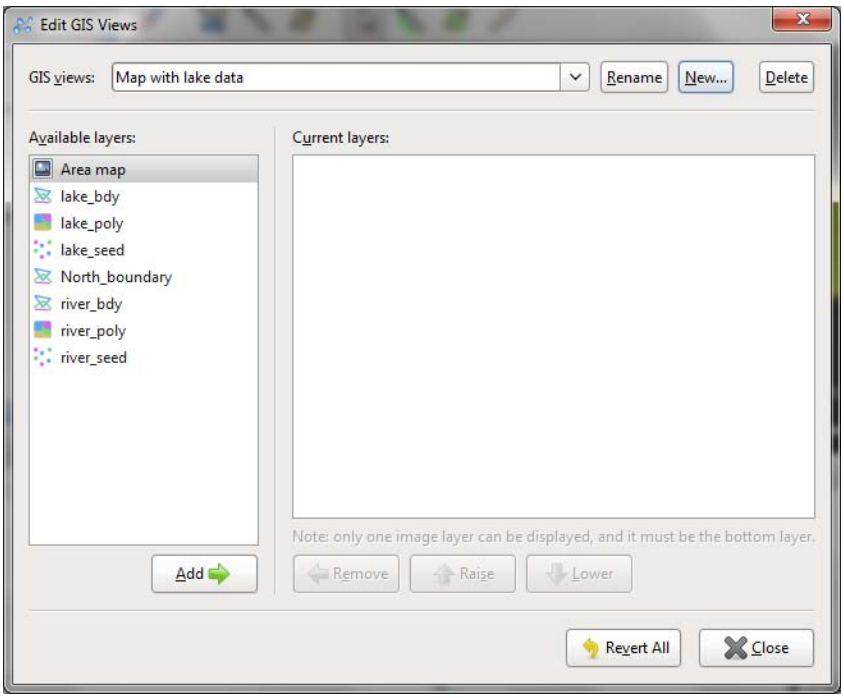
Both the **Topography** and the **GIS data** object will appear in the shape list:



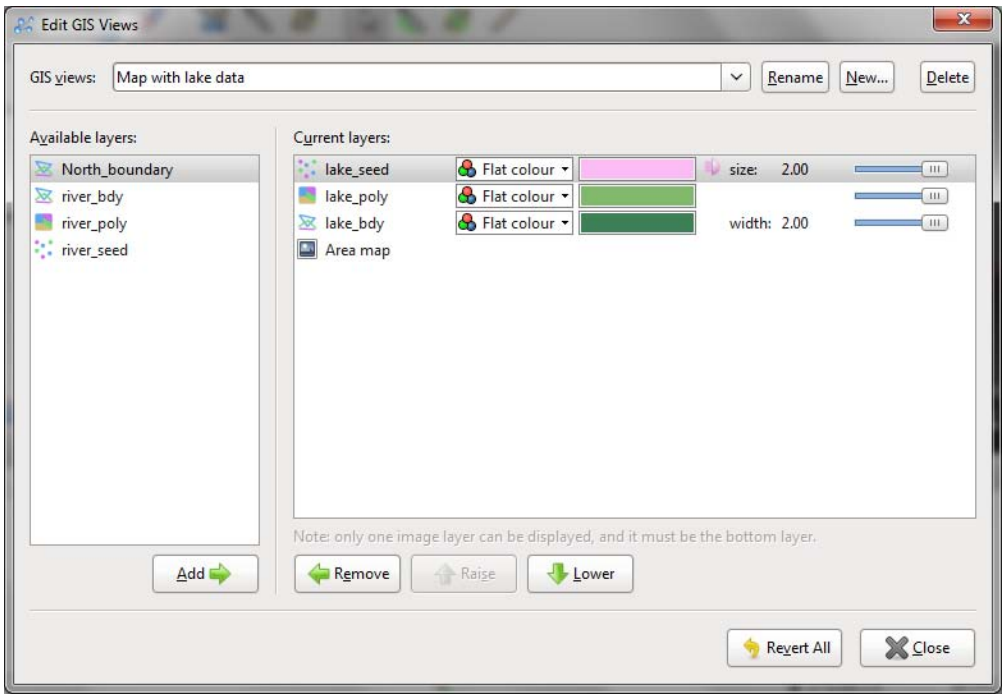
From the **GIS data** drop-down list, select **New View**. The **New GIS View** window will appear:



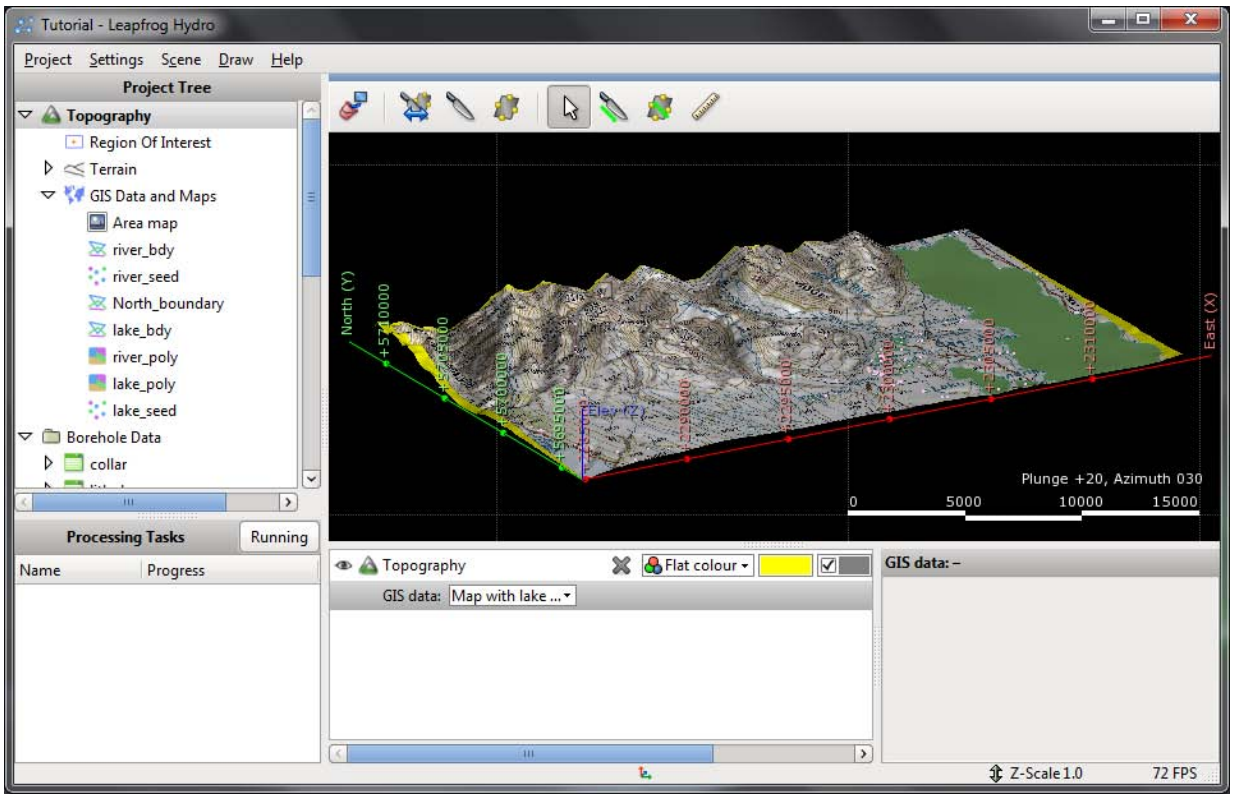
Enter a name for the new view or keep the default name, then click **Create**. The **Edit GIS Views** window will appear.



Add the required layers to the **Current layers** box and use the **Raise** and **Lower** arrows to arrange the layers.



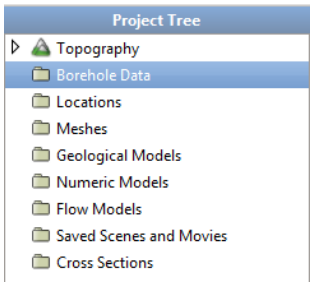
You can also change the opacity and colour of the different layers. Click **Close**. The new view is displayed:



Tutorial 3: Importing and Working with Borehole Data

Borehole data forms the basis of models created in Leapfrog Hydro. As borehole data often contains errors that prevent the computation of an accurate model, Leapfrog also has tools that help you to correct errors in the data.

In Leapfrog, borehole data is managed using the **Borehole Data** object in the project tree:



Until data is imported, the only options available from the **Borehole Data** object are import options.

Leapfrog expects borehole data that is stored in a collar table and interval tables. Each project can have only one collar file, but multiple interval tables can be imported. For the interval tables, Leapfrog expects data corresponding to the hole ID, the start and end depths and at least one column of measurements.

In this tutorial, you will import, correct and display a small set of borehole data that includes information on nitrate levels. The borehole data files can be found in the `Tutorial` folder, under `Borehole data`. There are three files:

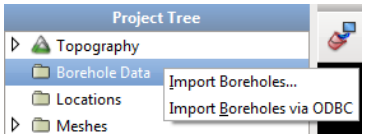
- `Collar.csv` contains the collar table.
- `Lithology.csv` contains the interval table that describes the lithology. Information in this file will be used to build a simple geological model in Tutorial 4: Building a Simple Geological Model.
- `Nitrates test.csv` contains the interval table that describes nitrate levels for each borehole. Information in this file will be used to build a simple numeric model in Tutorial 5: Building a Simple Numeric Model.

Topics covered in this tutorial are:

- Importing Borehole Data
- Correcting Errors in the Data
- Displaying Borehole Data
- Interacting with Borehole Tables via the Scene

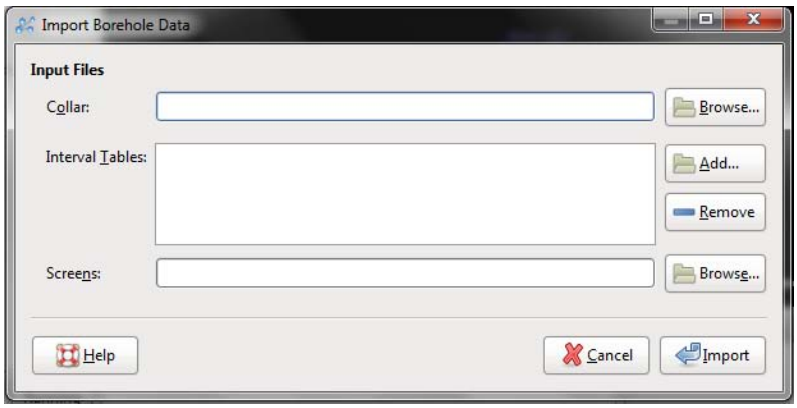
Importing Borehole Data

When you right-click on the **Borehole Data** object, there are two options available:



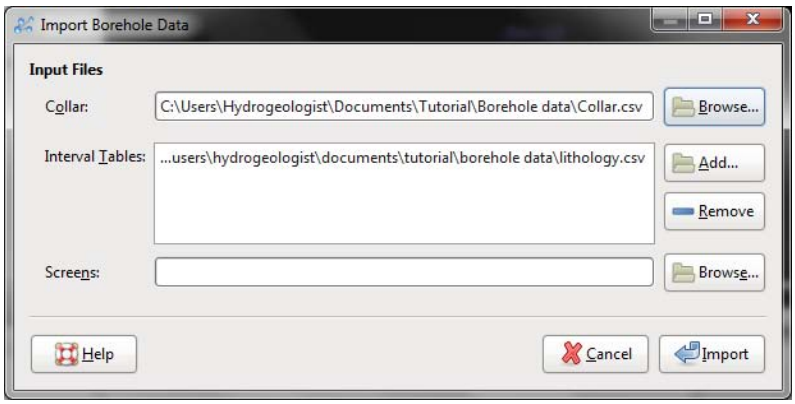
For this tutorial, select the **Import Boreholes** option.

The **Import Borehole Data** window will be displayed:

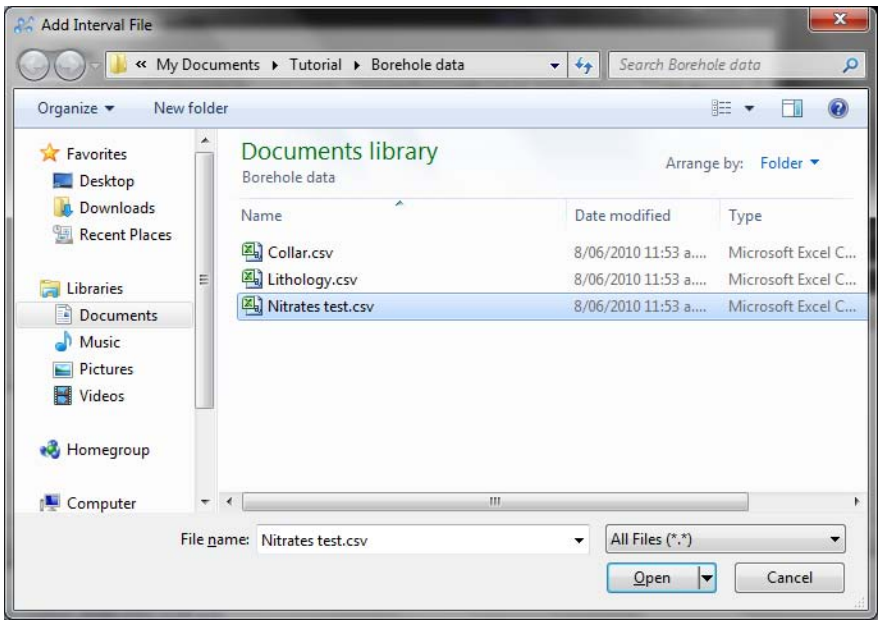


For the **Collar**, click on the **Browse** button to locate the tutorial file called `Collar.csv`. Leapfrog will look for interval tables in the same location and will display them in the **Interval Tables** list.

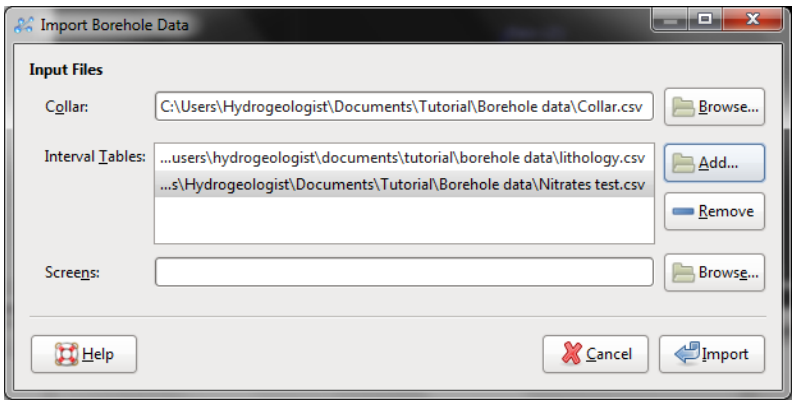
In this example, adding the `Collar.csv` file has resulted in only the `Lithology.csv` file being added to the **Interval Table** list:



To add the nitrates table to the list, click on the **Add** button and navigate to the `Nitrates test.csv` file:



Click **Open** to return to the **Import Borehole Data** window, where the nitrates file has now been added to the **Interval Tables** list:

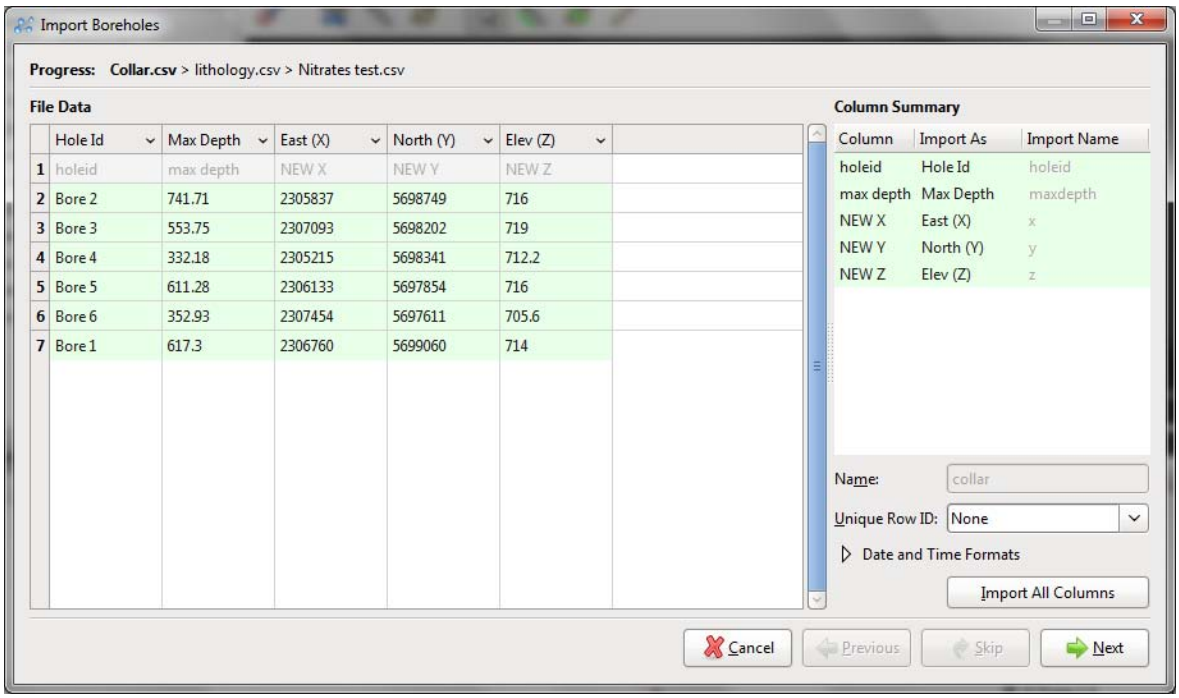


If you do not wish to use any of the files displayed in the **Interval Tables** list, click on the file to highlight it, then click **Remove**.

Once you click on **Import**, Leapfrog will start the process of importing data.

Importing the Collar File

Leapfrog expects four columns for the collar data, a borehole identifier (Hole ID) and the location of the top of the well, in X, Y and Z coordinates. It attempts to match the data in the imported file to the expected format. With the file `Collars.csv`, Leapfrog has correctly mapped the imported data to the expected formats:



The breadcrumb at the top of the window indicates progress in importing the borehole files. In this window, the information in the file `Collar.csv` is being considered.

The **Max Depth** column is not required, but because the information is in the file, Leapfrog will use it to verify the integrity of the data. If that information is not included in a collar file, Leapfrog will determine it from the interval tables.

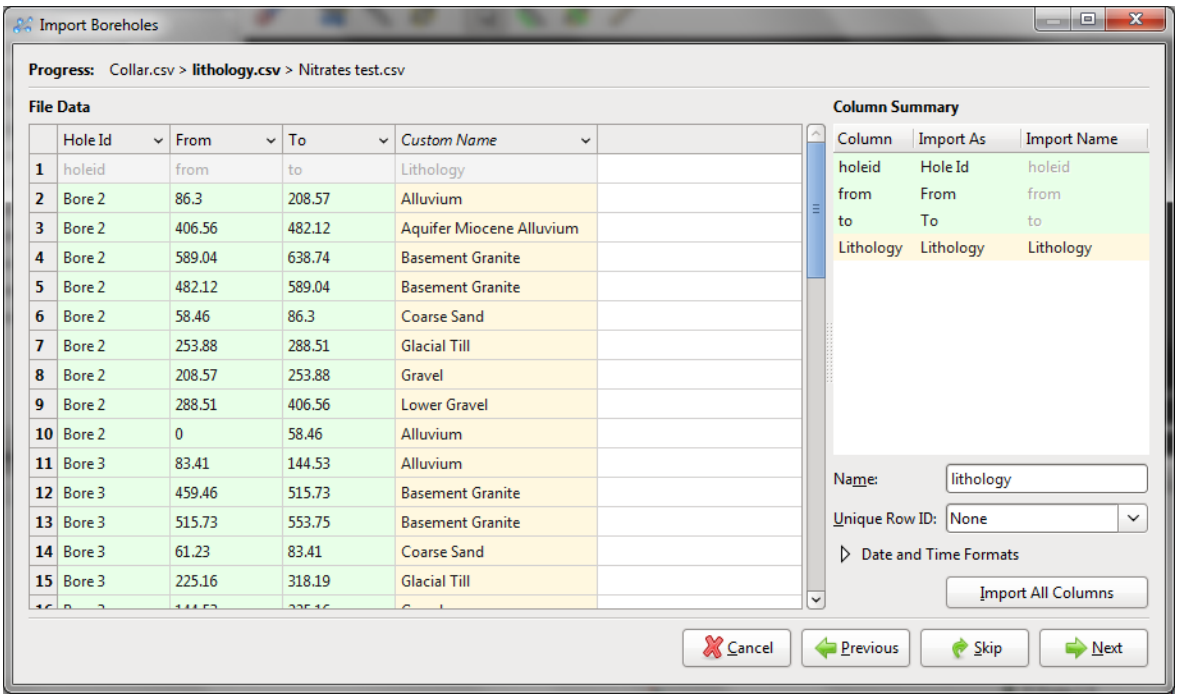
Click on **Next** to move to the next step.

Many data types can be included in the interval tables. The most common types are:

- Lithology data, which describes the geology encountered in drilling the borehole. Lithology data columns can be used to create geological models.
- Numeric data, which describes data such as temperature and concentration. Numeric data columns can be used to model continuously varying distributions by way of interpolations.
- Text data, which can be displayed in Leapfrog as part of a scene.

Importing the Lithology Interval Table

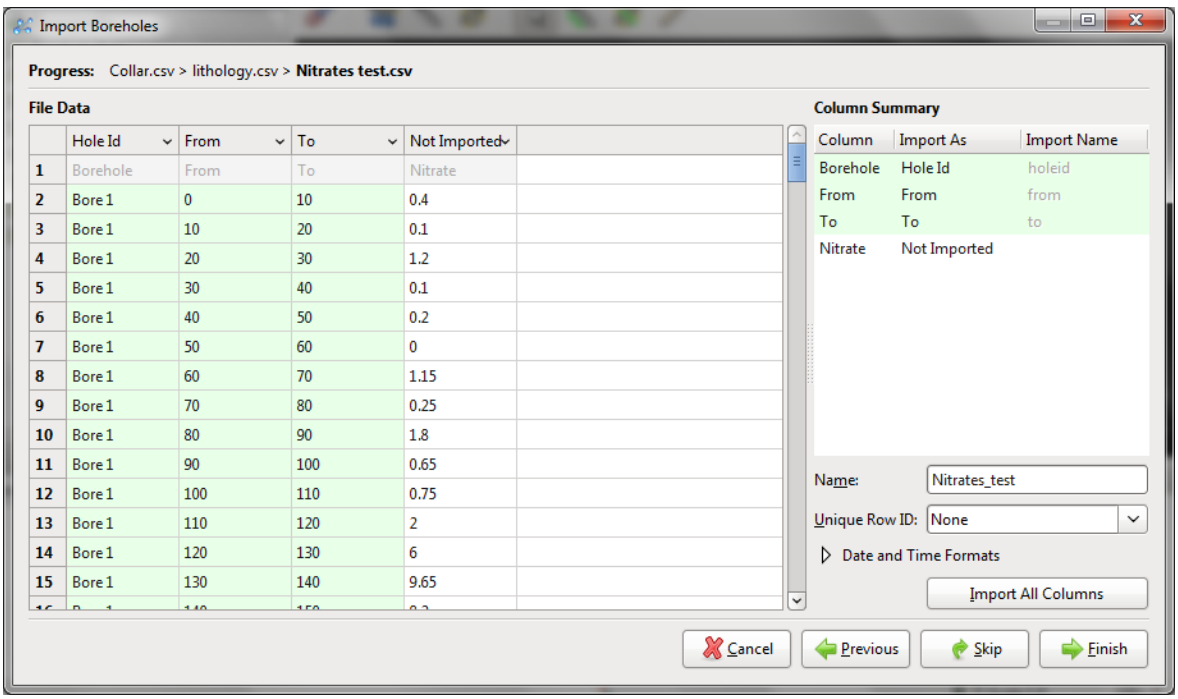
For the file `Lithology.csv`, Leapfrog has correctly determined the lithology column:



Click **Next** to move on to the next file.

Importing the Nitrates Interval Table

For the file `Nitrates test.csv`, Leapfrog has not marked the Nitrate column for importation:

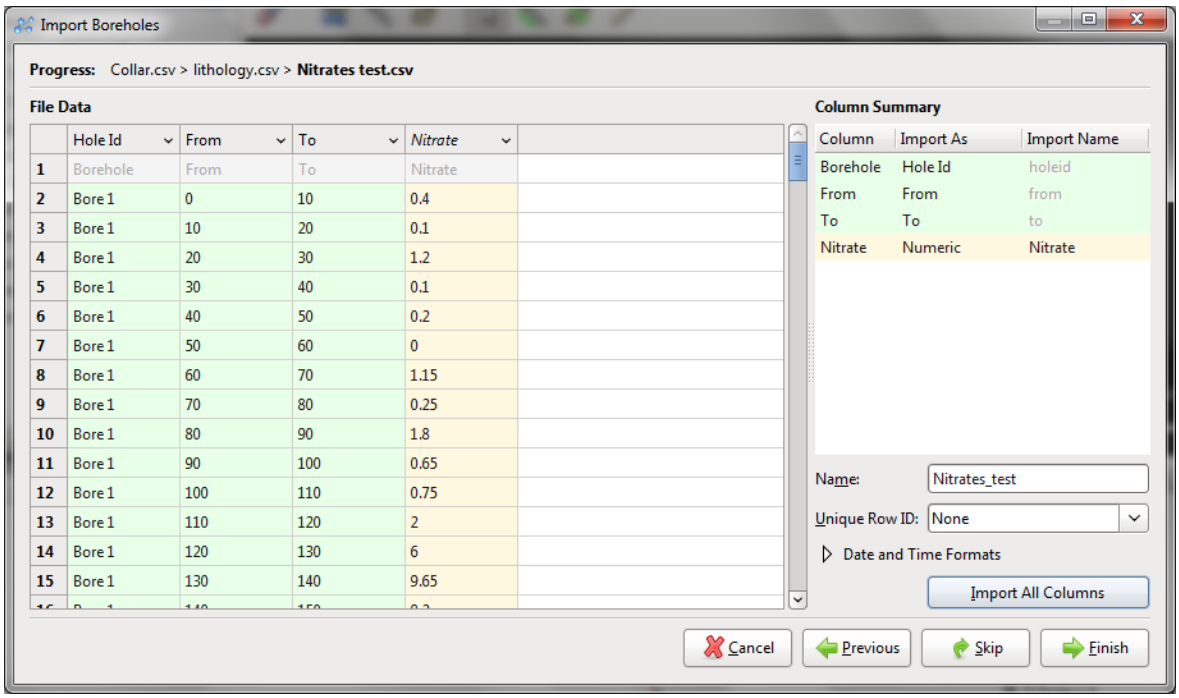


To select it for importation, you can either:

- Select the "Nitrate" Numeric option from the **Not Imported** drop-down list. This identifies the data as numeric data and uses the column title from the file.

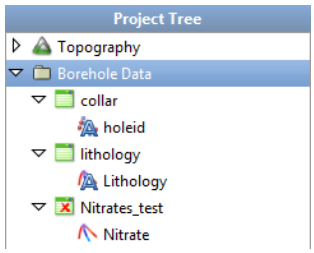
- Click on **Import All Columns**, in which case, Leapfrog will determine the type of data from the properties of the information each column.

When the **Import All Columns** button is clicked on, the **Column Summary** list on the righthand side of the window indicates that the Nitrate column will be imported as numeric data:



Click on **Finish** complete the process.

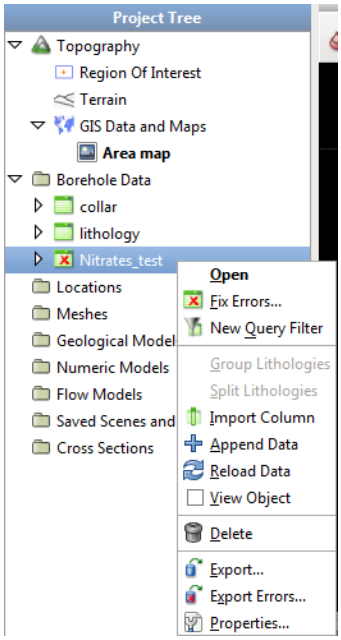
Once Leapfrog has imported the data, it will appear in the project tree under the **Borehole Data** object:



The Nitrates_test object has an X across is because there are errors in the data. See Correcting Errors in the Data.

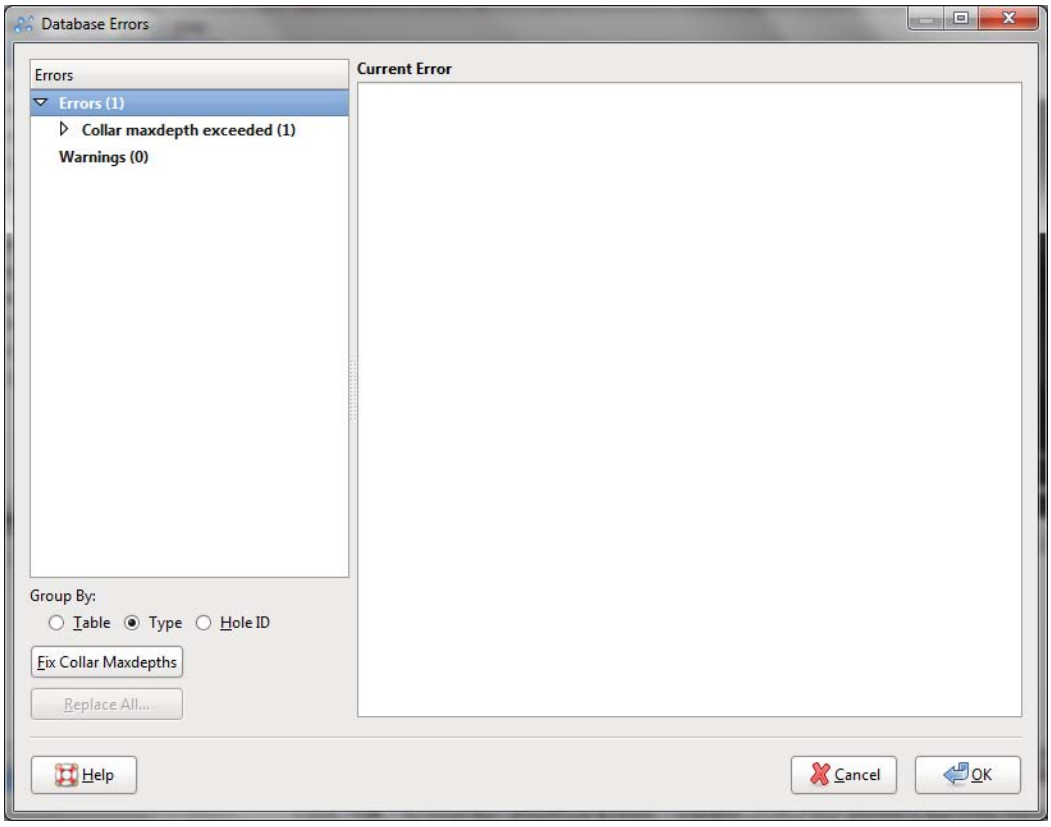
Correcting Errors in the Data

When conflicts or errors are detected in imported borehole data, Leapfrog marks the table containing that data with a red X:

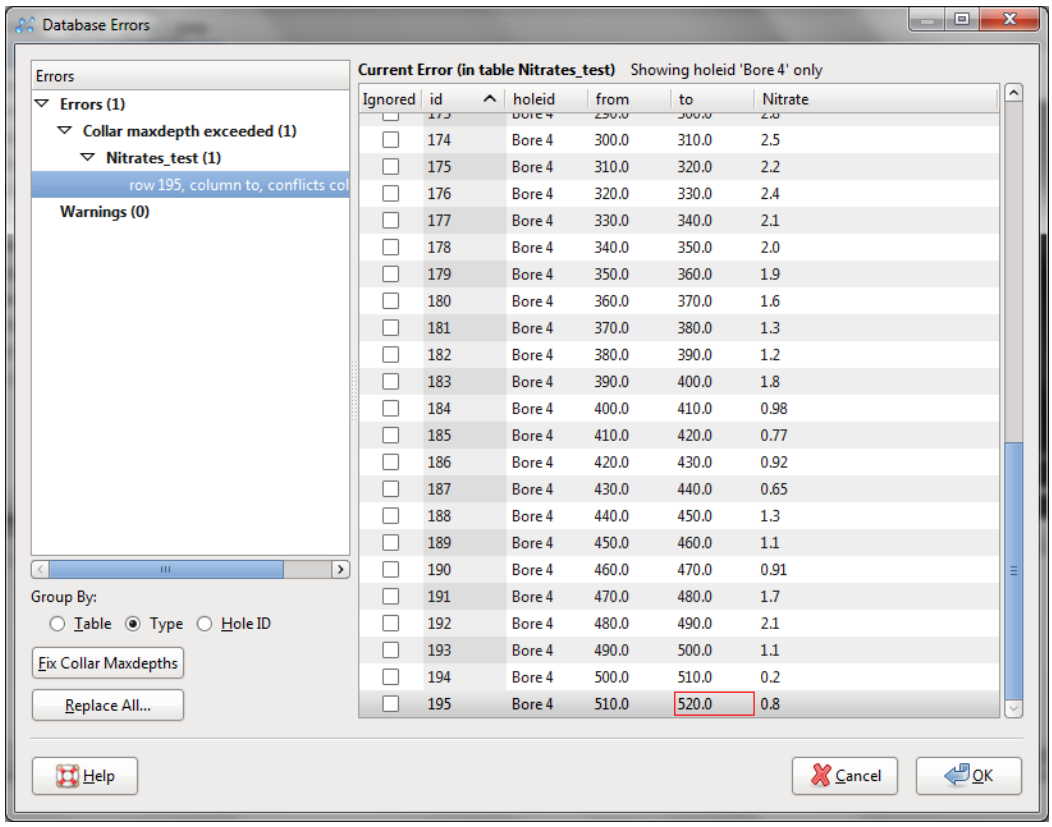


To correct the data, right-click on the incorrect object and select **Fix Errors**.

The **Database Errors** window will appear:

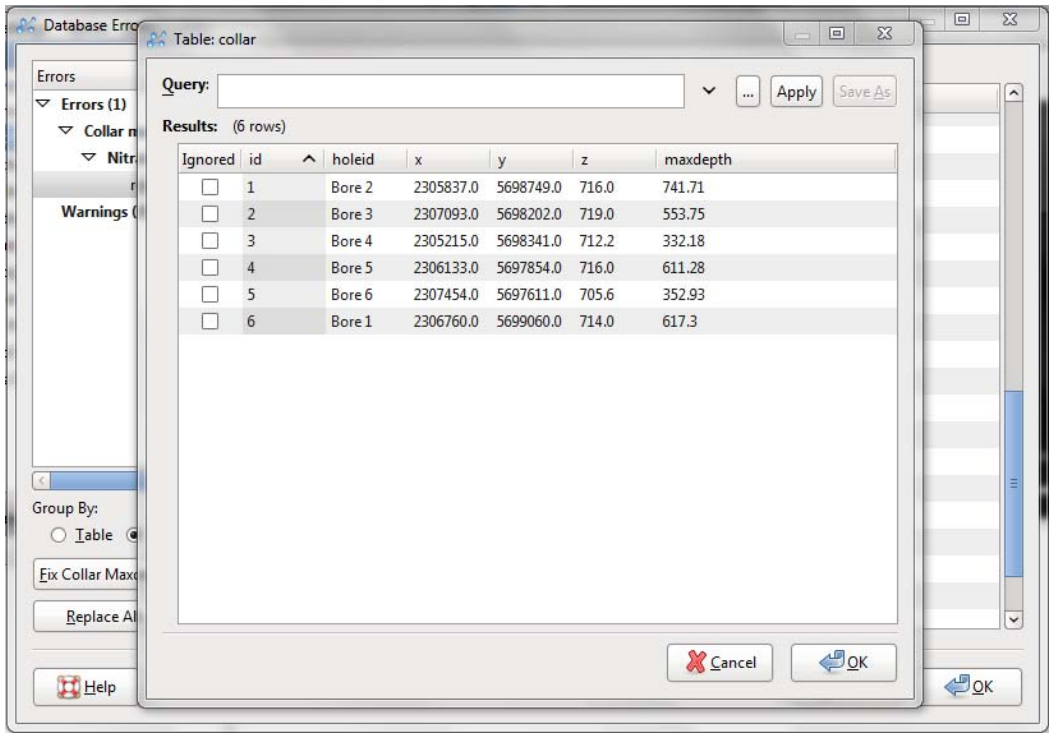


This error has occurred because the collar depth data in one table conflicts with that in another. Click on the arrows to expand the list and view more information about the error:



The location of the error is marked in the table by a red box.

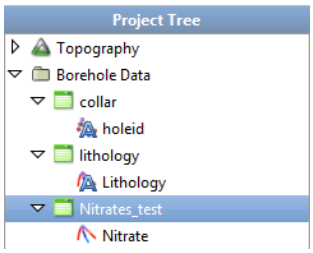
While the **Database Errors** window is open, right-click on the collar table to view the collar data that row 195 conflicts with:



The maximum collar depth specified for Bore 4 is 332.18, which conflicts with the nitrate data collected for Bore 4.

Click **Cancel** to close the collar table. In the **Database Errors** window, click the **Fix Collar Max-depths** button. This will increase the maximum collar depth value in the collar table to match the depth values in the Nitrates_test interval table.

Click **OK** to close the **Database Errors** window. Notice that the red X has been removed from the Nitrates_test object:



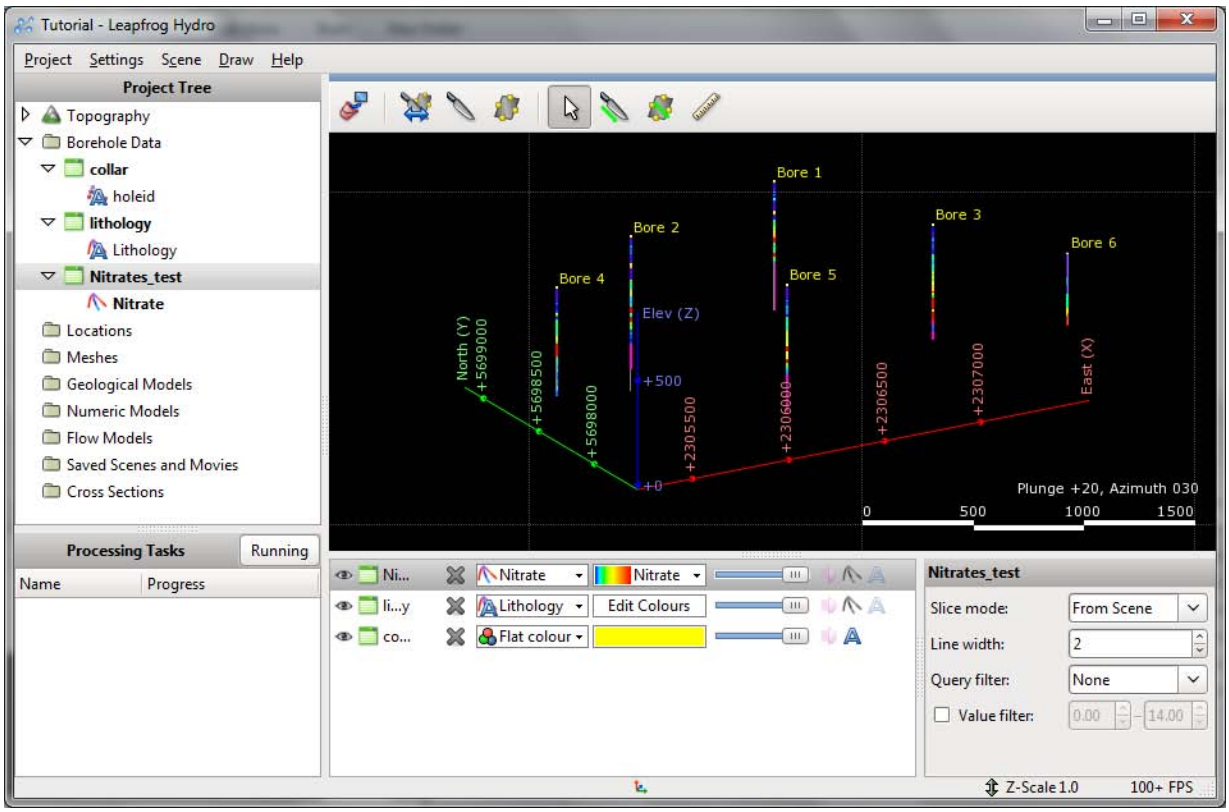
Displaying Borehole Data

At this point, the project contains corrected borehole data, but nothing is visible in the scene.

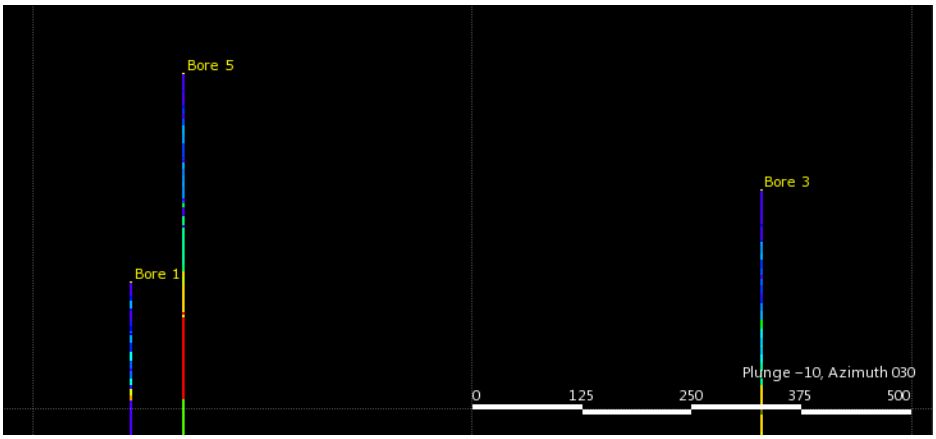
There are two ways to display the borehole data objects:

- Click on each table in the project tree and drag it into the scene.
- Right-click on each table in the project tree and tick **View Object**.

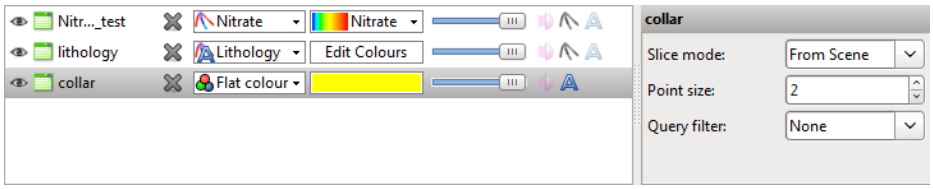
In the image below, the collar, lithology and nitrates test tables have been added to the scene window:



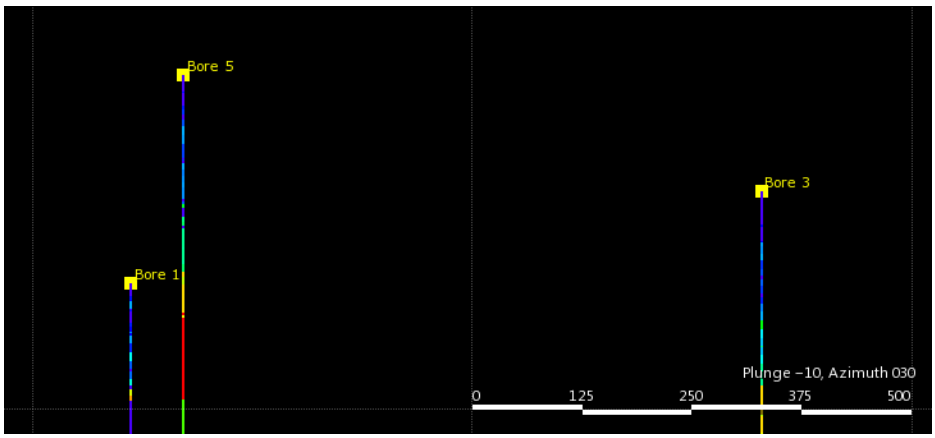
The appearance of the displayed data can be changed by changing the properties of the object in the shape list. For example, when the data is first add to the scene, the collars are displayed as small points:



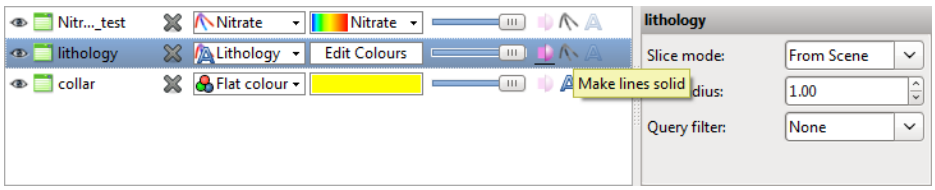
To change the size of the collars, select the collar object in the shape list:



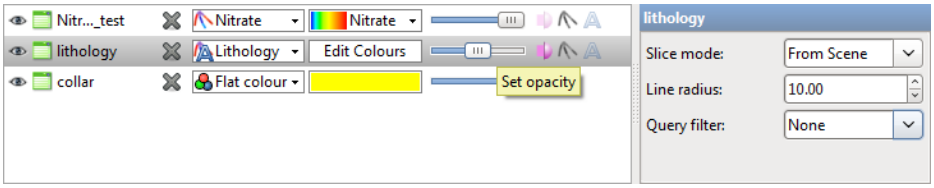
The **Point size** field in the properties window is 2. Increasing it to 10 changes the display of the collars, making them more apparent in the scene:



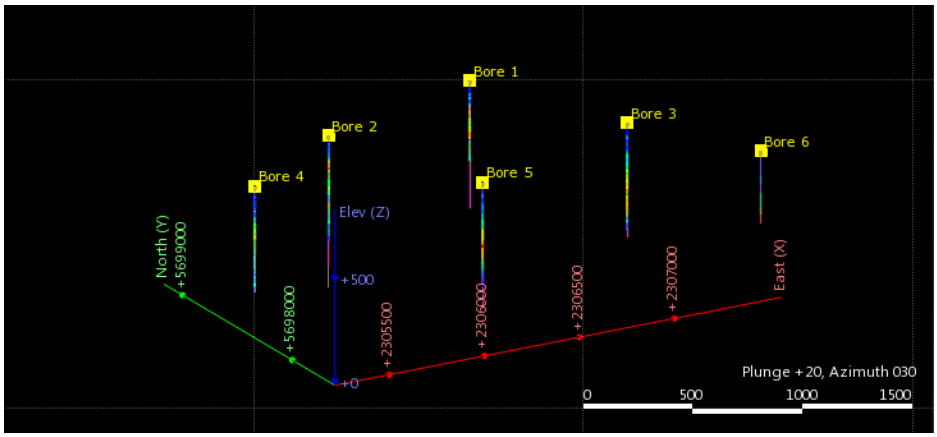
Next, select the lithology object and click on the **Make lines solid** icon:



In the properties window, set the **Line radius** to 10 and move the **Opacity** slider to the left:

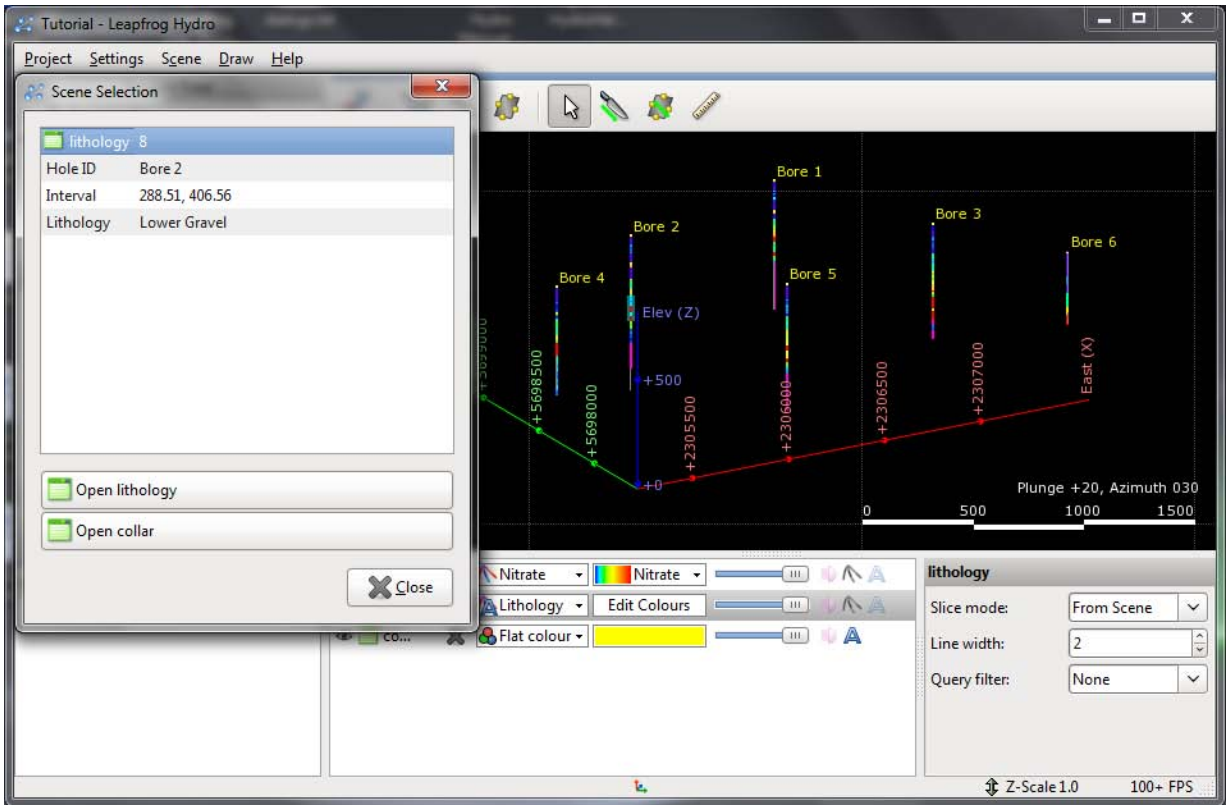


The effect in the scene window is that the boreholes are shown as wells with the nitrate intervals inside:

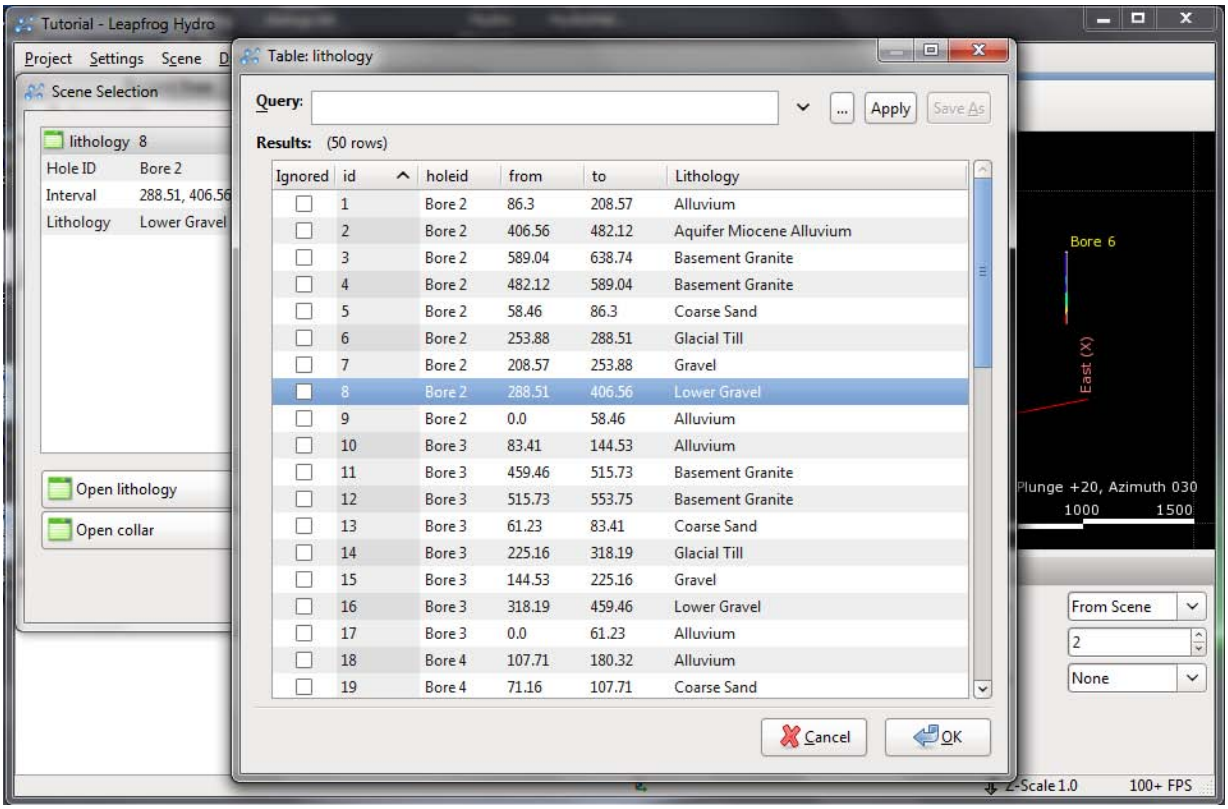


Interacting with Borehole Tables via the Scene

To display data tables using the scene, simply click on a data segment. The corresponding data is displayed:

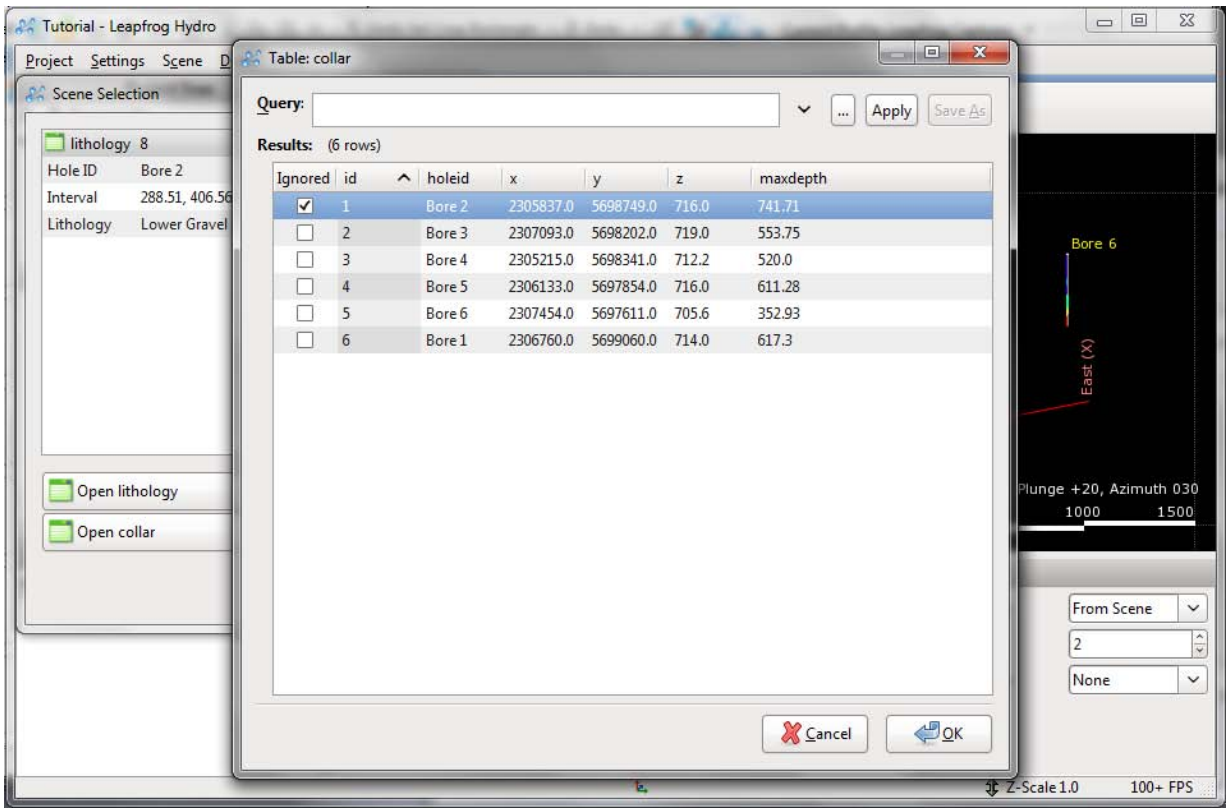


Click the **Open lithology** and **Open collar** buttons to go directly to the corresponding tables, where the selected data segment will be highlighted. In this example, the lithology table is displayed:



In this table, you can change any entry by clicking on the row, then on the value. This is useful when the three-dimensional display of the data has made errors apparent. Edited data can be exported from Leapfrog.

A suspicious value can be ignored by ticking the Ignored box for that value. When the **Ignored** box is ticked in the collar table:



the entire borehole will be removed from the scene and no further processing will be performed on it.

Tutorial 4: Building a Simple Geological Model

A geological model is the fundamental method for describing lithological units in Leapfrog Hydro. A geological model consists of a number of non-intersecting volumes that fit together to exactly fill a set of defined model extents.

Building a geological model is a process of successive refinement made up of two key steps:

- Defining the model extents. These usually correspond to the ground surface and known flow boundaries.
- Defining the internal structure. This involves generating contact surfaces that define the internal divisions within the model boundaries. These internal divisions correspond to the contacts between lithological units.

In Leapfrog, boundaries between lithological units are specified in terms of geological processes such as deposition, intrusion and erosion. Erosions are not actually the geological process of erosion but rather are a surface hierarchy.

In this tutorial, you will create a simple geological model using the borehole data imported for Tutorial 3: Importing and Working with Borehole Data. You will:

- Create a new geological model
- Define the model extents using different methods
- Define the internal structure of the model from borehole data

Some of the model extents will be created from imported data. Files for this part of the tutorial can be found in the `Tutorial` folder, under `Boundary data`.

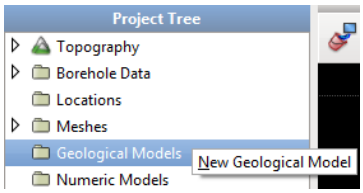
Topics covered in this tutorial are:

- Creating a New Geological Model
- Refining the Model Extents
- Defining the Internal Structure

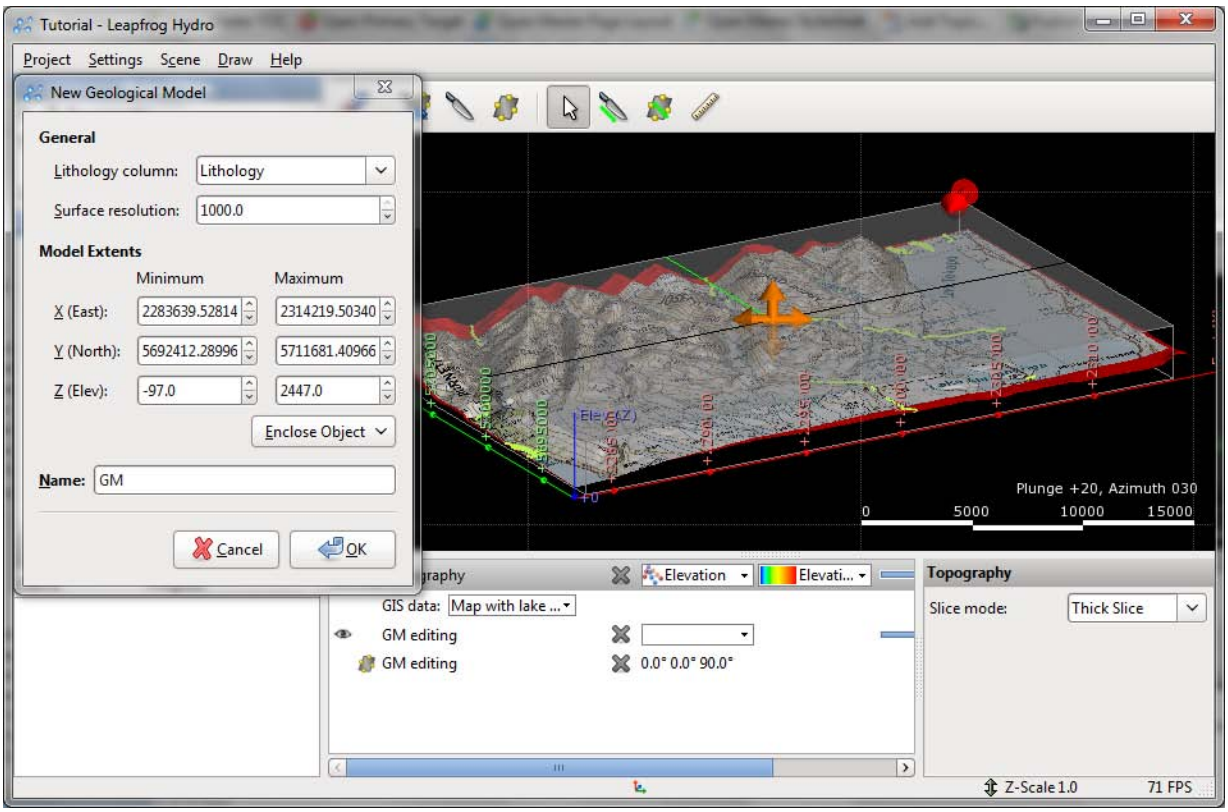
Creating a New Geological Model

Open the tutorial project from Tutorial 3: Importing and Working with Borehole Data and click on the **Clear scene** button to clear any objects from the scene window. Drag the area map imported in Tutorial 2: Defining the Topography into the scene and press `D` to view it from above.

Right-click on the **Geological Models** object and select **New Geological Model**:



The **New Geological Model** window is displayed, together with controls in the scene that will help you to set the model extents:



This window lets you set a basic rectangular set of model extents aligned with the south/north and east/west axes. It is possible to describe irregular-shaped model extents, which is discussed further

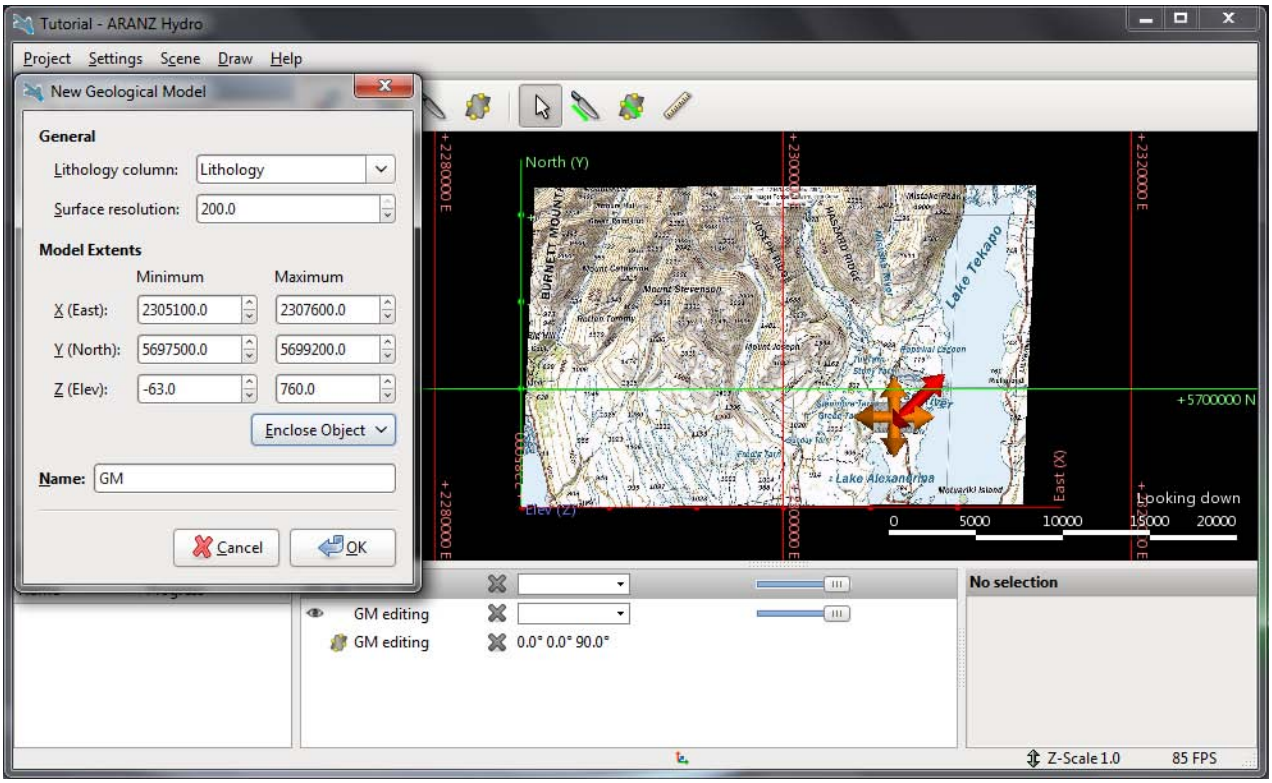
in Refining the Model's Lateral Extents. When creating the geological model, however, it is sufficient to set a rectangle that is similar in size to the final model extents.

There are three ways to define the rectangular volume of interest:

- Enter the coordinates.
- Select **Enclose Object** and choose from the list of objects in the project. The model extents will be updated to the size of the selected object.
- Use the controls that appear in the scene. The orange handle adjusts the center of the plane and the red handle adjust the size.

Experiment with the different methods to see how the controls work.

The **Enclose Object** list can be used to set the size of the rectangular extents to fit the data that will be used to generate the model. In this tutorial, borehole data will be used as the basis of the model, so select "lithology segments" from the list. The model extents will be updated to fit the selected data:

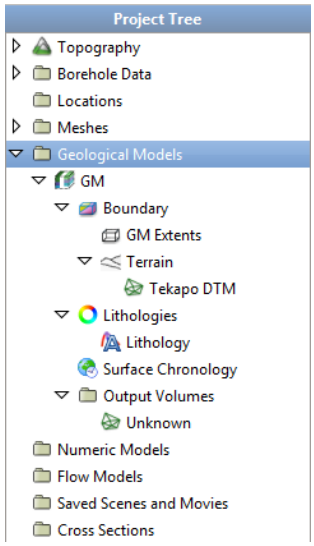


The **Surface resolution** is also set in this window. A lower value will produce more detail, but calculations will take longer. The advantage of Leapfrog over many other geological modeling products is that the surface resolution can be easily changed.

When borehole data is used as the basis of a model, the Lithology column determines what column in the imported borehole data will be used. If more than one column of lithology data is available for creating models, be sure to choose the correct one at this point as it cannot be changed once the initial model has been created.

Click **OK** to create the new model.

Once Leapfrog has generated the new geological model, it will appear in the project tree under the **Geological Models** object:



The major elements of the new geological model are:

- The **Boundary** object. This includes the geological model extents and the terrain.
- The **Lithologies** object. This is the lithology information that describes the different volumes that will make up the internal structure of the geological model. The information included as part of the **Lithologies** object has been generated from the lithology table in the **Borehole Data** object.
- The **Surface Chronology** object. This defines the chronological order of the volumes of the geological model. When the model is first created, the **Surface Chronology** object will contain no data.
- The **Output Volumes** object. This includes the generated volumes that make up the geological model. Initially, it only contains a volume called Unknown.

Although the geological model extents were defined while creating the new model, the first step in working on the model is to further refining those extents. See Refining the Model's Lateral Extents.

Refining the Model Extents

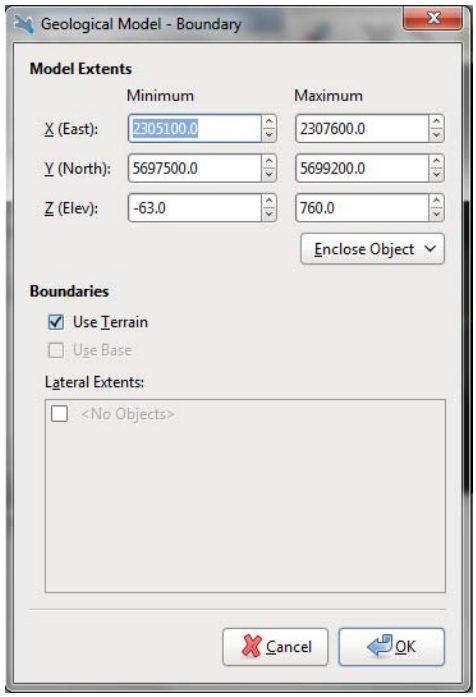
When a new model is created, Leapfrog defines a basic rectangular set of model extents that define the top, base and lateral extents of the model's external boundaries.

A number of different techniques can be used to modify the external boundaries so that they better represent the area being.

- The terrain can be used as the model's upper surface.
- A polyline can be drawn and used to create a new lateral extent or base.
- GIS vector data, points data or surface meshes can be imported and used to create a new lateral extent or base.
- Lithology contacts can be used to create the model's base.

When a geological model's boundaries are changed, the model is recomputed, which can make changing a model's boundaries time-consuming and frustrating. For this reason, Leapfrog allows you to create new lateral extents and refine them before they are applied to the model.

To view the model's extents, double-click on the **Boundary** object or right-click and choose **Open**. The **Geological Model - Boundary** window will be displayed:



The **Lateral Extents** list will remain empty until new extents are defined. For now, click **OK** to close the window.

See Using the Terrain to Define the Top of the Model.

Using the Terrain to Define the Top of the Model

In Leapfrog, the Terrain object is used to define the elevation of the ground's surface. It is, therefore, the most obvious source of information about the top surface of a geological model, which is why terrain information that makes up the **Topography** object is automatically included in new geological models created in Leapfrog.

Refining the Model's Lateral Extents

In Leapfrog, it is possible to generate a surface by merging several different data sources, but this tutorial only considers using a single data source. There are several methods for creating a new lateral extent, but in each case, the lateral extent is created in isolation before being applied to the model. This saves time as it does not require that the model be recalculated upon every change to its boundaries.

You can create multiple lateral extents and easily switch between them using the **Geological Model - Boundary** window.

For this tutorial:

- A new southern boundary will be created from a drawn polyline
- A new northern boundary will be created from imported GIS vector data.

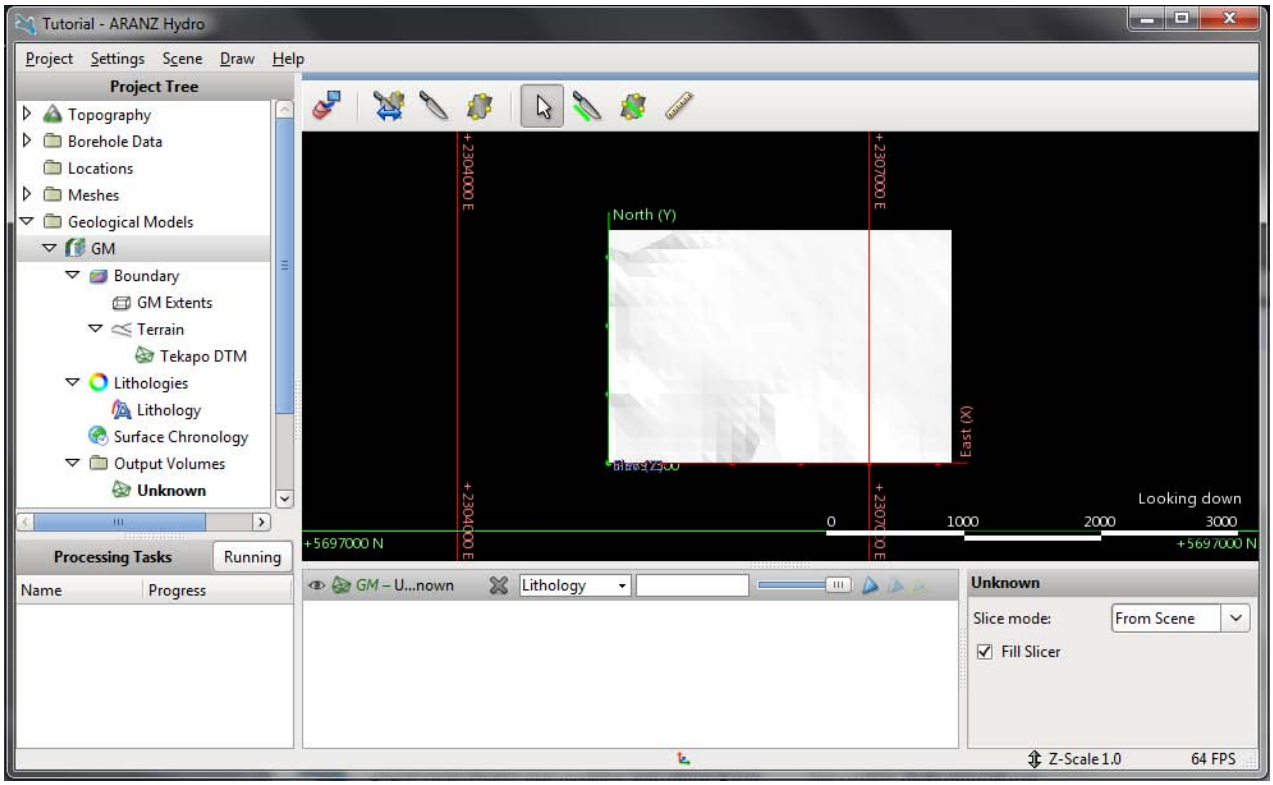
A new lateral extent can only reduce the volume of a geological model defined by the initial rectangular model extents.

See Creating a New Lateral Extent from a Polyline.

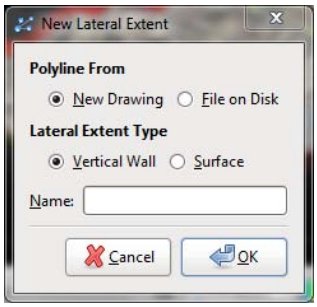
Creating a New Lateral Extent from a Polyline

In this part of the tutorial, a new southern boundary will be created from a polyline.

To start, click on the **Clear scene** button. Add the geological model to the scene by either dragging and dropping the model into the scene or by right-clicking on the model and selecting **View Object**. Then press the D key on your keyboard to view the model from above.

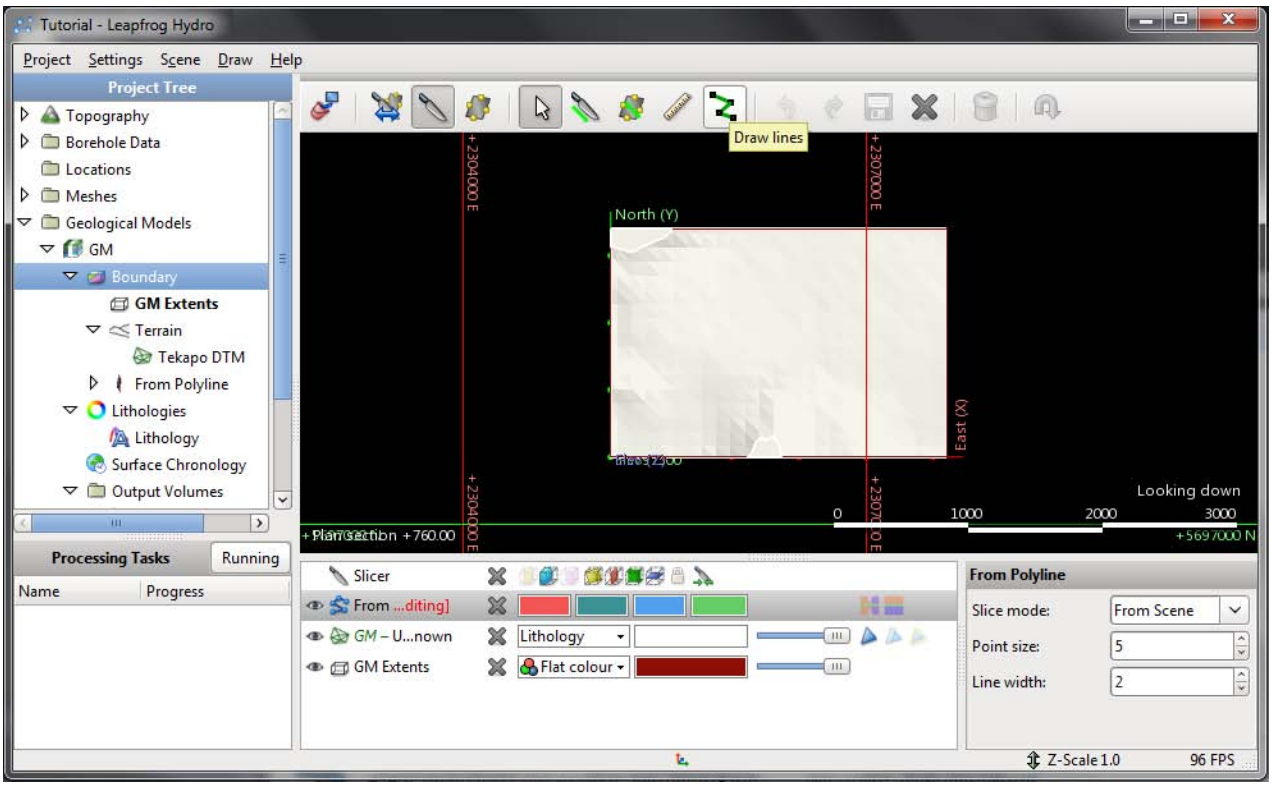


To create a new lateral extent, right-click on the **Boundary** object and select **New Lateral Extent > From Polyline**. The **New Lateral Extent** window will be displayed:

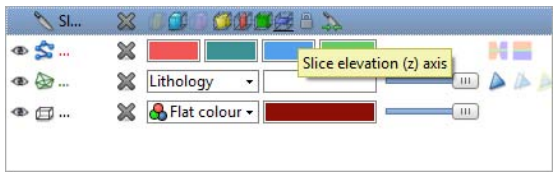


Make sure **New Drawing** and **Vertical Wall** are selected, then click **OK**.

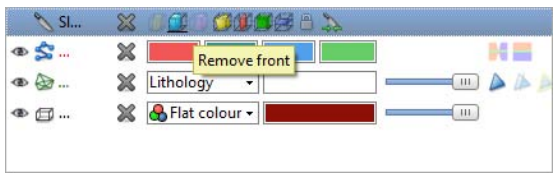
Click on the **Draw lines** icon, then on the **Slicer** tool in the shape list.



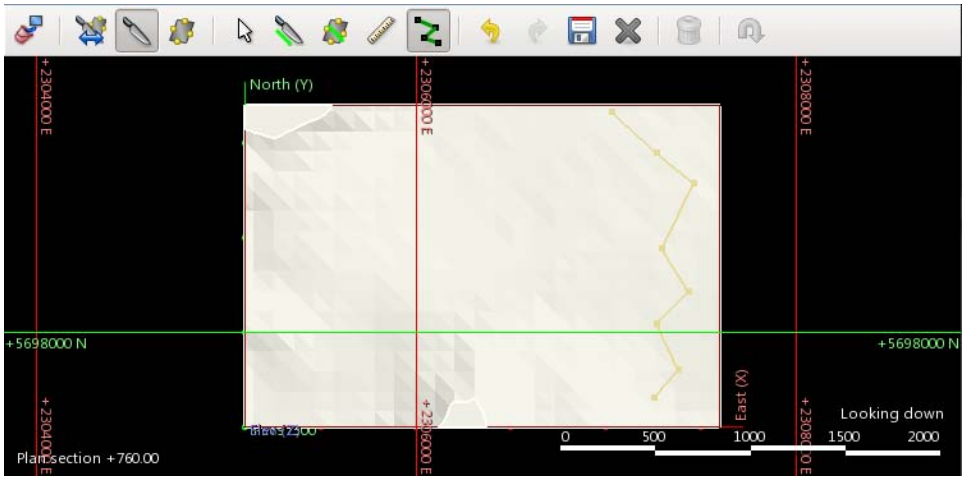
Ensure the **Slice elevation (z) axis** icon is selected by clicking on it:



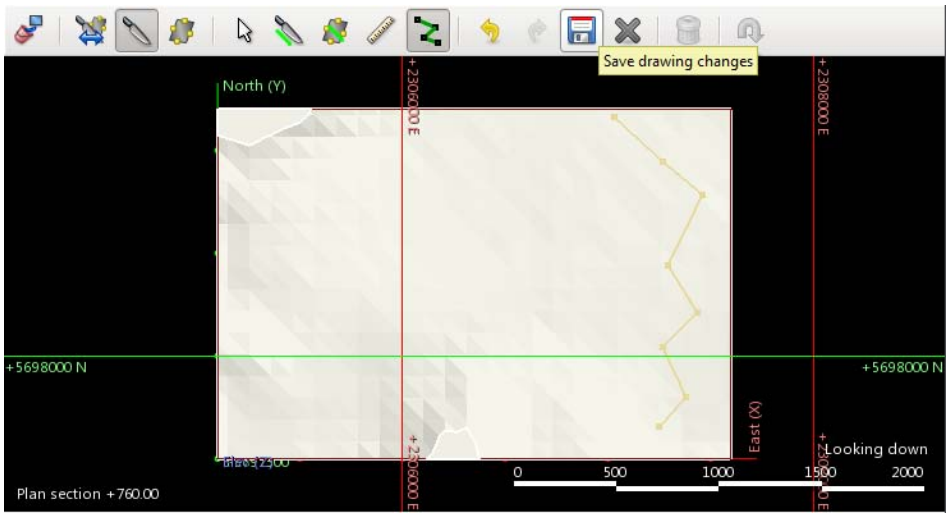
Then click on the **Remove front** icon:



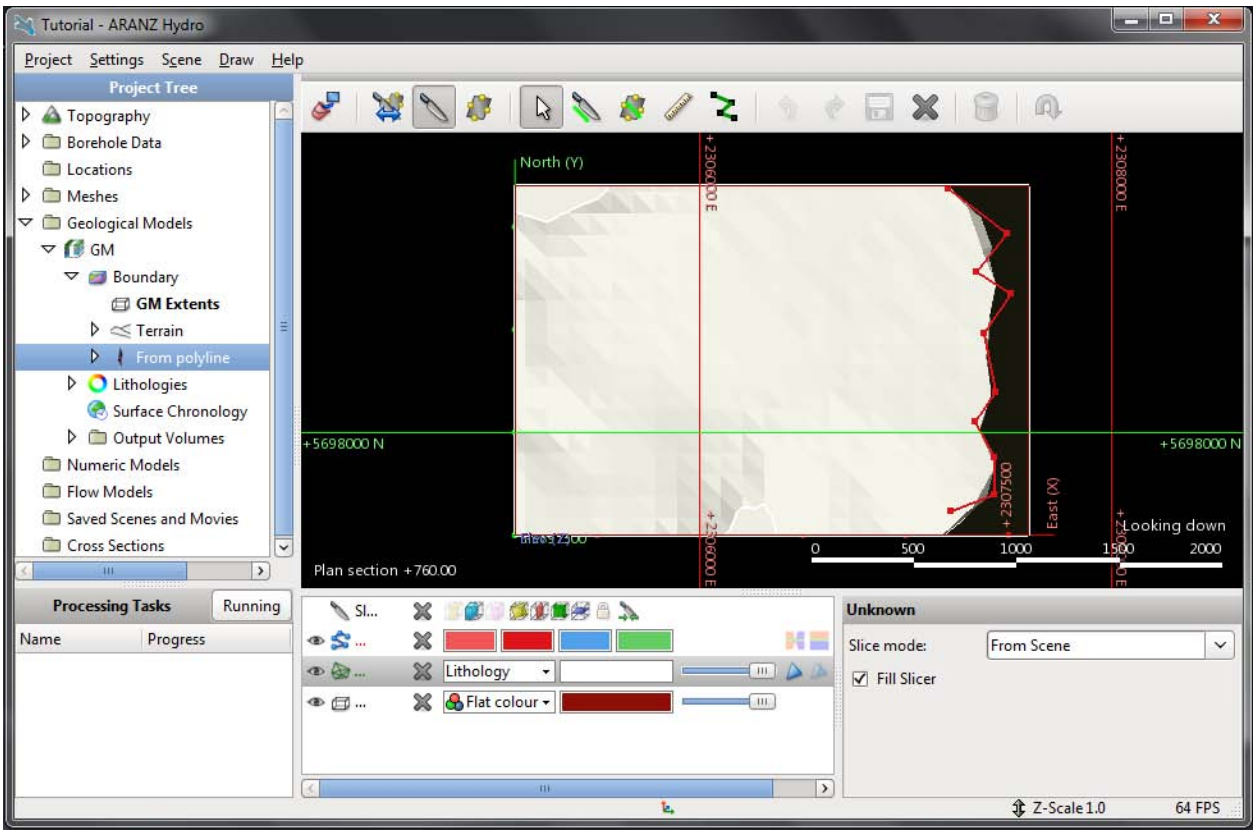
Start drawing from east to west by clicking to add points.



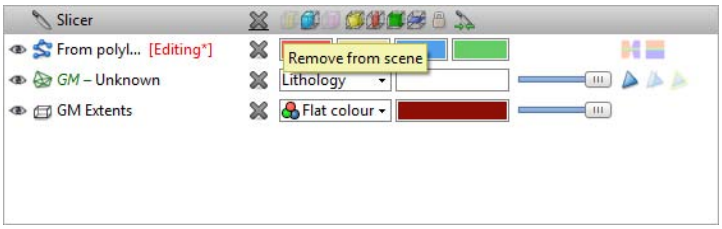
Right-click to end the polyline, then click the **Save** icon:



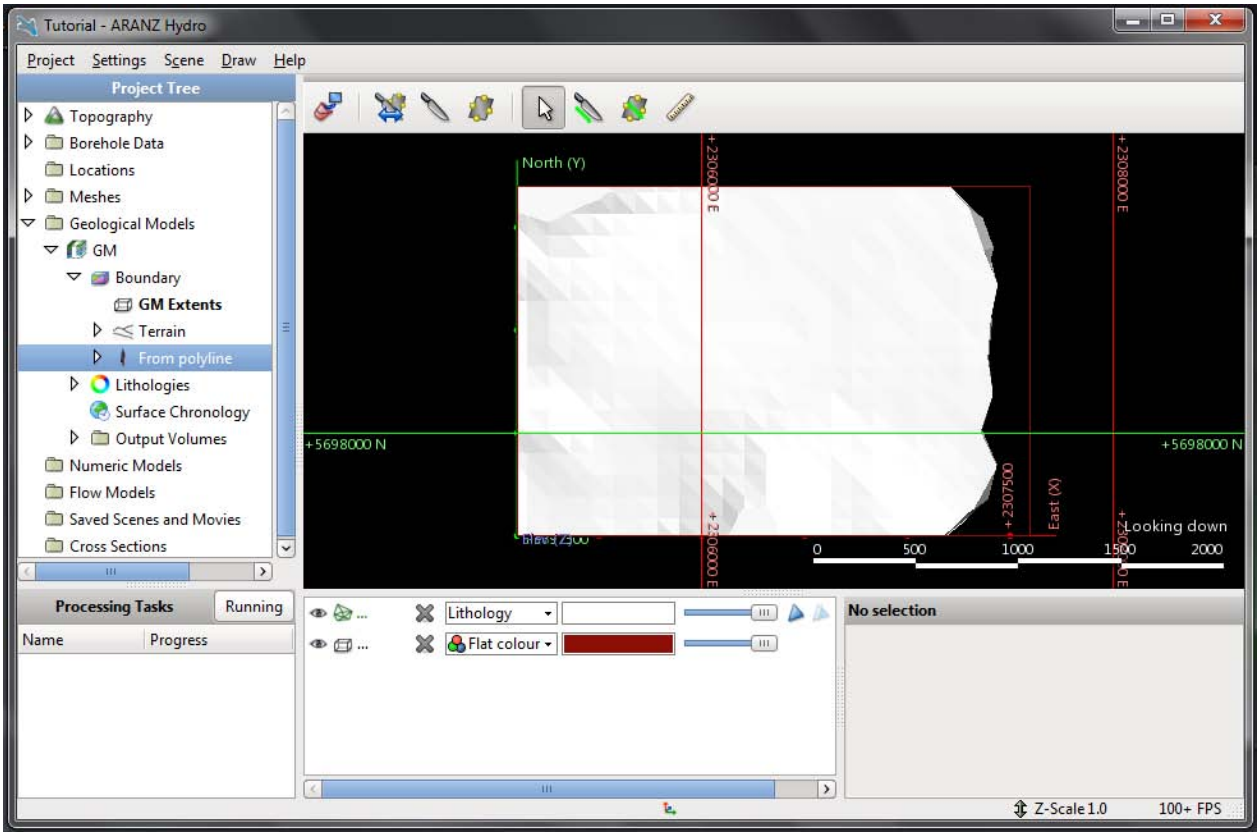
Leapfrog will then generate the new boundary, which will be displayed in the scene and in the project tree as part of the **Boundary** object:



However, the polyline object remains in the scene. Remove it and the **Slicer** from the scene by clicking the X next to each object in the shape list:



The new lateral extent now appears in the scene.

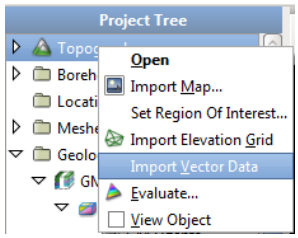


Creating a New Lateral Extent from GIS Vector Data

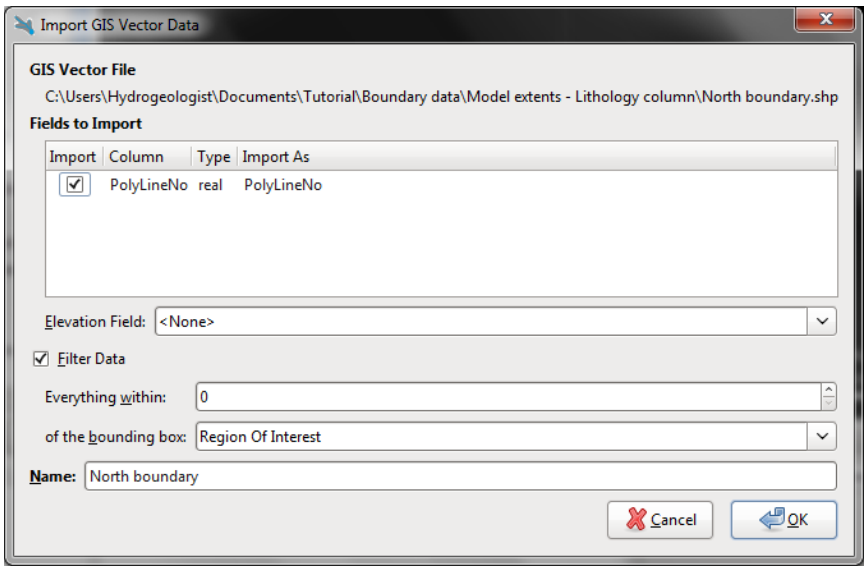
In this part of the tutorial, a new northern boundary will be created from imported GIS vector data. The file required is called `North boundary.dbf` and can be found in the `Boundary data` folder.

To start, reset the view by pressing the `Home` key on your keyboard.

Before a new extent can be added, the data to build it must be imported into the project. From the **Topography** object, select **Import Vector Data**:

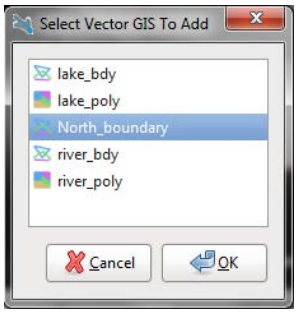


Navigate to the file containing the `North boundary.shp` file and click **Open**. The **Import GIS Vector Data** window will be displayed:



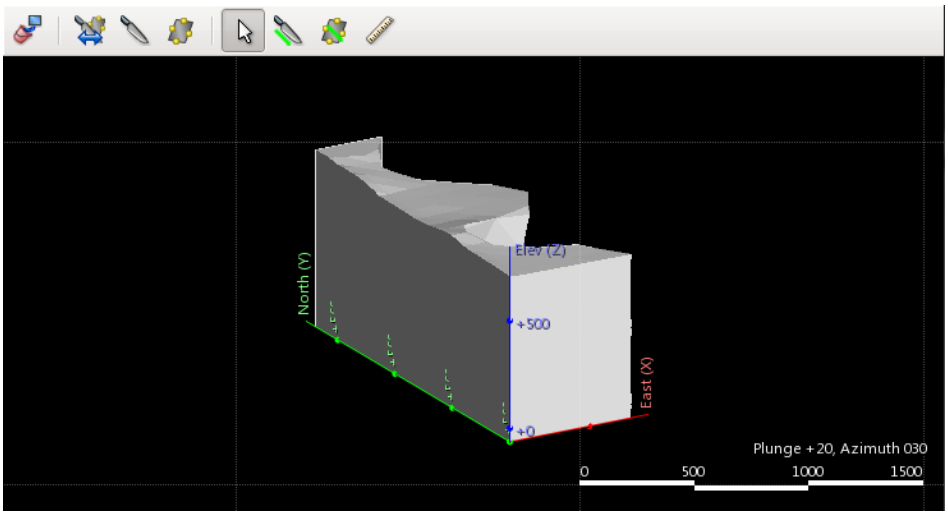
Click **OK**.

Right-click on the **Boundary** object and select **New Lateral Extent > From GIS Vector Data**. The **Select Vector GIS To Add** window will be displayed:

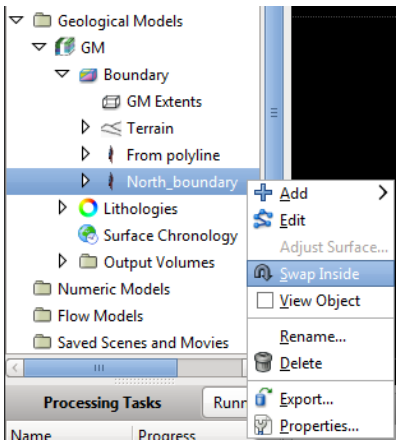


Select North_boundary and click **OK**.

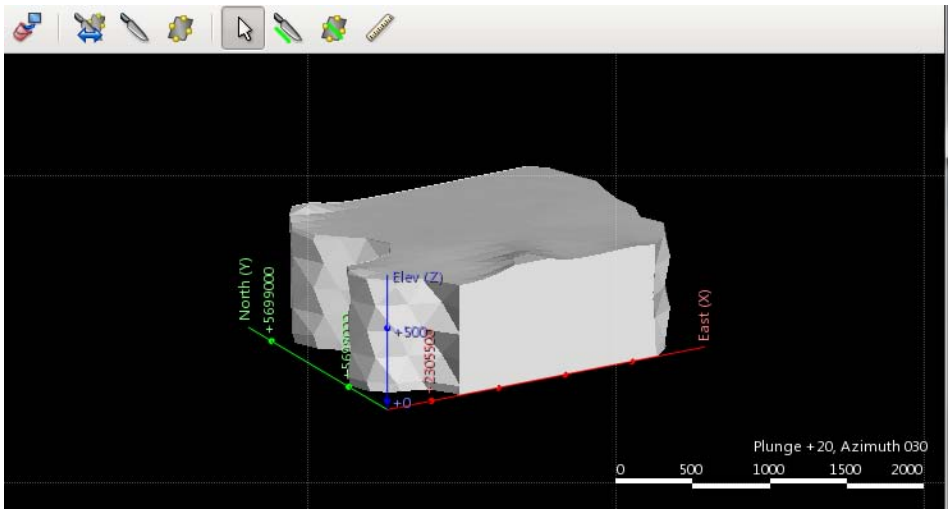
Once the new boundary has been generated, it will appear in the scene:



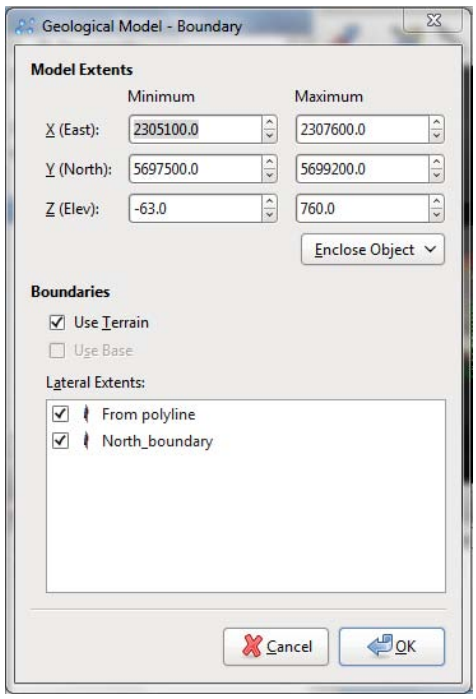
If the wrong side of the boundary has been cut, as it has in the scene above, right-click on the lateral extent in the project tree and select **Swap Inside**:



The boundary will then be recalculated and displayed:



Now that two new extents have been created, double-click on the **Boundary** object to open the **Geological Model Boundary** window. The two lateral extents now appear in the **Lateral Extents** list and can be added to or removed from the model by ticking the box.

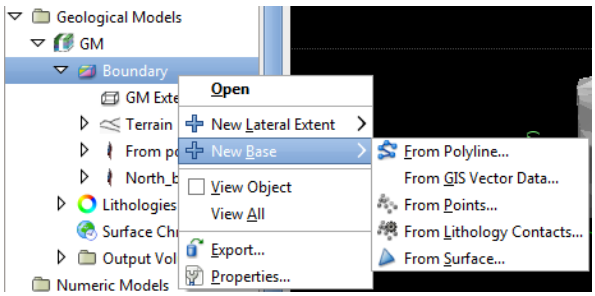


Refining the Model's Base

The model's base can be defined:

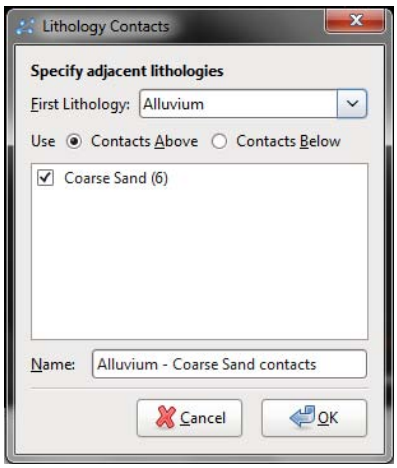
- Using a polyline
- From imported GIS vector data, points data or surface meshes
- From lithology contacts

The process for creating a new base is similar to that for creating lateral extents. Right-click on the **Boundary** object and select **New Base** and the option required:



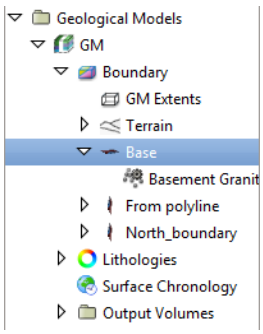
For this part of the tutorial, the base of the model will be defined from lithology contacts.

To start, right-click on the **Boundary** object and select **New Base > From Lithology Contacts**. The **Lithology Contacts** window will appear:

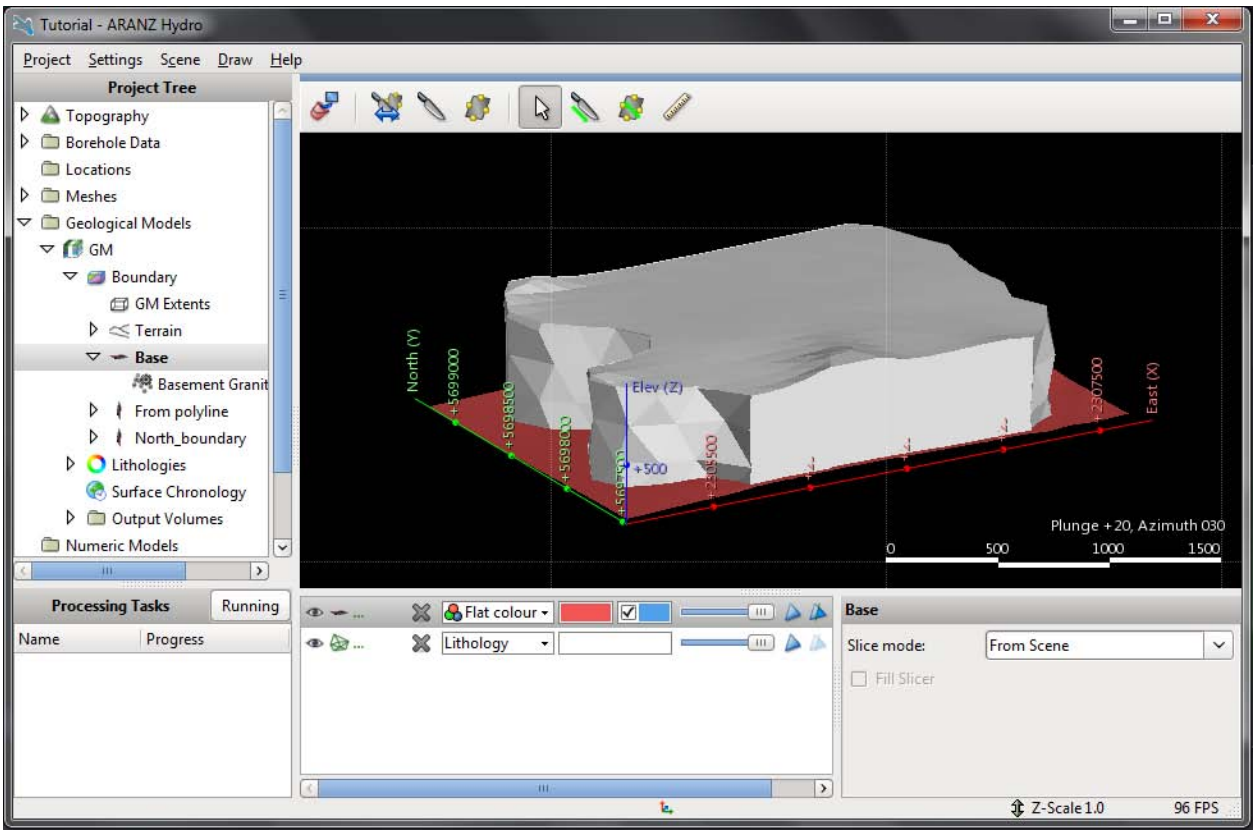


From the **First Lithology** list, choose the lowest lithology in the model, which in this case is "Basement Granite". Click **OK**.

Leapfrog will generate the new base, which will appear under the **Boundary** object:



Drag the new base into the scene:



Defining the Internal Structure

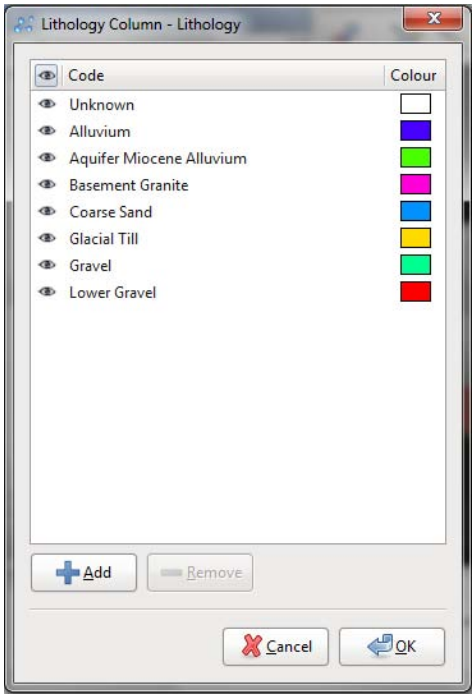
Lithology columns from imported borehole data form the basis of the internal structure of a geological model.

For this part of the tutorial, the volumes that make up the internal structure of the geological model will be generated from a series of contact surfaces derived from imported borehole data. These contact surfaces are then added in chronological order to the **Surface Chronology**.

Viewing Lithologies

When the geological model was first created, the **Lithology Column** field specified what data is used to build the model.

To view the different lithologies identified in the data, open the **Lithologies** object in the geological model. The **Lithology Column** window will be displayed:



All the lithologies found in the specified column in the borehole data have been assigned colors. An additional lithology also appears in the list, called "Unknown". This is automatically added to the list of lithologies so that any unknown volumes generated when the geological model is built can be highlighted as this lithology.

Viewing the Surface Chronology

To view the surface chronology, open the **Surface Chronology** object in the geological model. The **Contact Surface Chronology** window will be displayed:

At this point, there is no information in this window as no contact surfaces have been generated and added to the chronology.

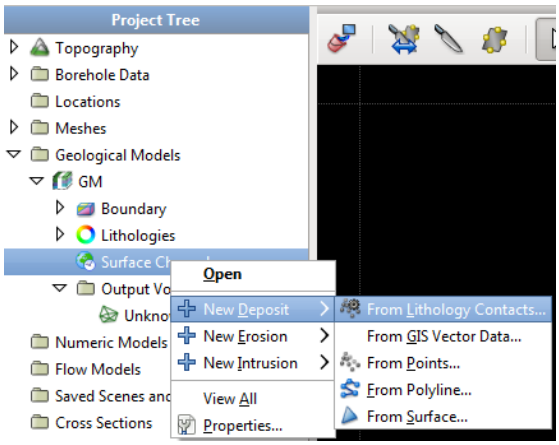
Creating the Contact Surfaces

The first step in building the volumes that make up the model is to create the contact surfaces. For this tutorial, the order of contact surfaces, from bottom to top, is:

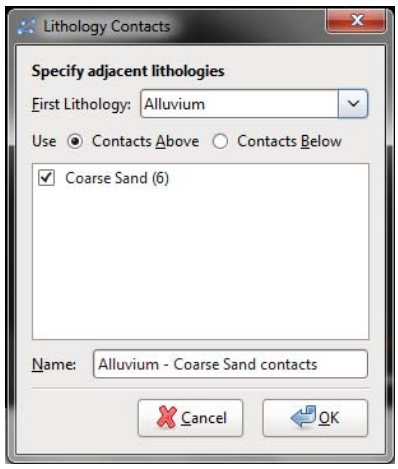
- Lower Gravel – Glacial Till
- Glacial Till – Gravel
- Gravel – Alluvium
- Alluvium – Coarse Sand

Best practice is to build the geological model from top to bottom.

To do this, right-click on the **Surface Chronology** object and select **New Deposit > From Lithology Contacts**:

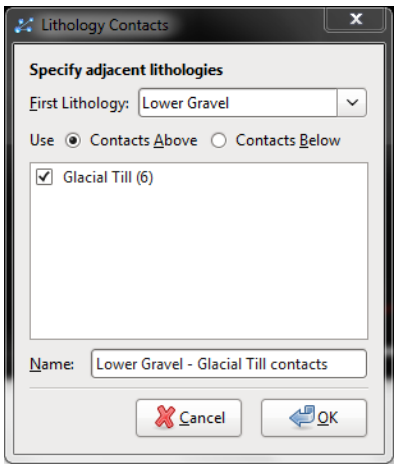


The **Lithology Contacts** window will be displayed:

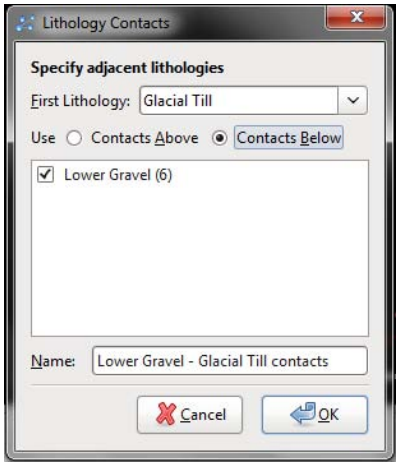


Best practice is to build the geological model from top to bottom, so for this tutorial, build the "Lower Gravel – Glacial Till" contact surface first. To do this in one of two ways:

- Select "Lower Gravel" from the **First Lithology** list and then click the **Contacts Above** button:



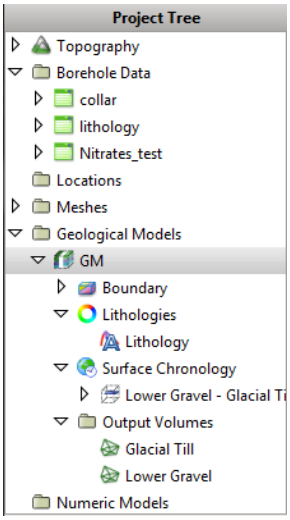
- Select "Glacial Till" from the **First Lithology** list and then click the **Contacts Below** button:



Leapfrog automatically assigns a name for the contact surface that will be created. Click **OK**.

When Leapfrog has finished generating the contact surface, double-click on the **Surface Chronology** object. The **Contact Surface Chronology** window will be displayed:

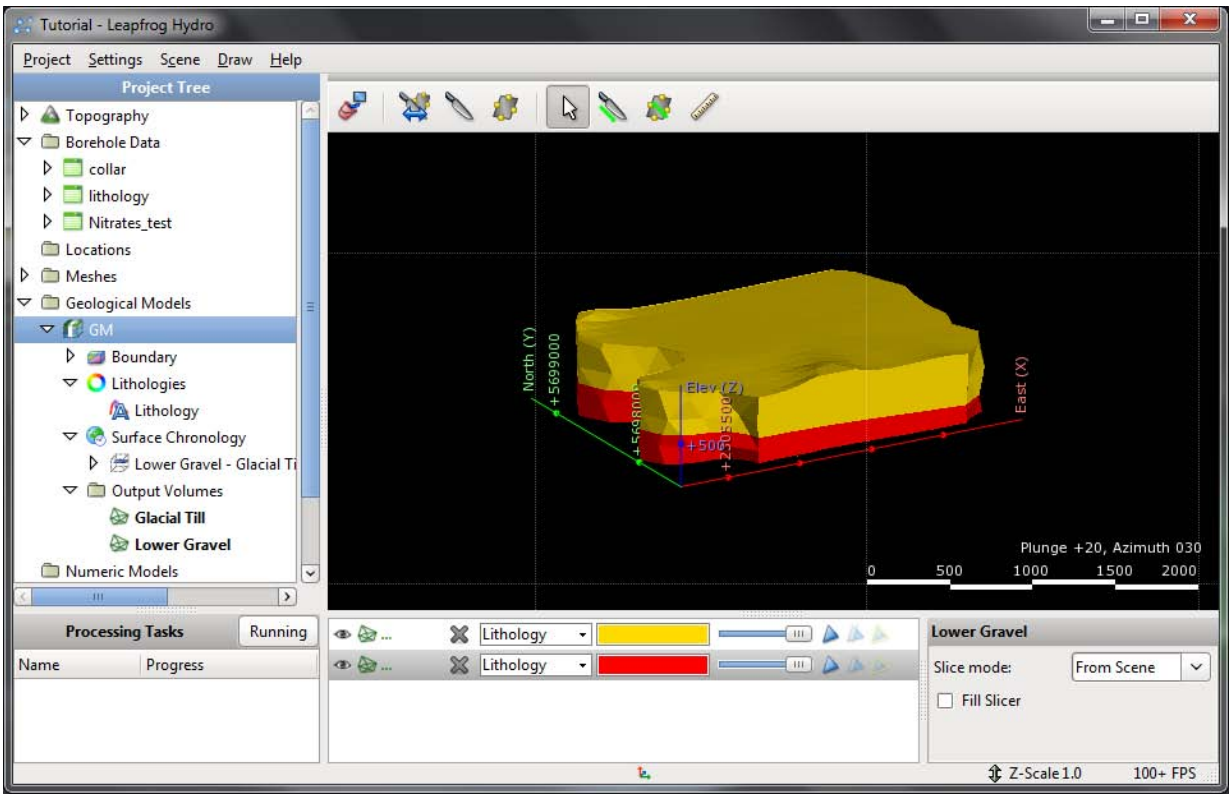
Tick the box to add the Lower Gravel – Glacial Till contact surface to the chronology, then click **OK**. This results in Leapfrog generating the first two volumes, which are added to the **Output Volumes** object:



At this point, the geological model is not yet visible in the scene. See Viewing the Geological Model Volumes.

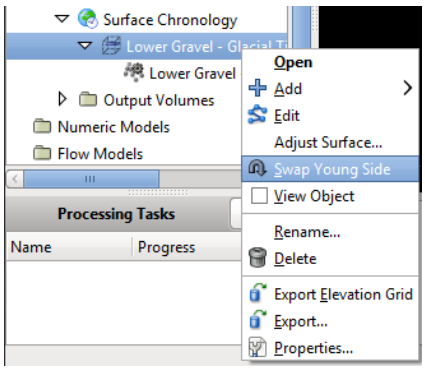
Viewing the Geological Model Volumes

Once this is complete, drag the geological model into the scene to view the volumes:

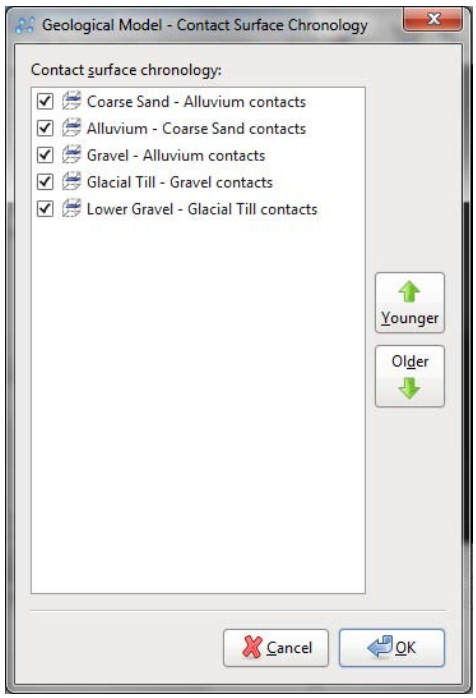


Continue creating contact surfaces and adding them to the **Contact Surface Chronology** table.

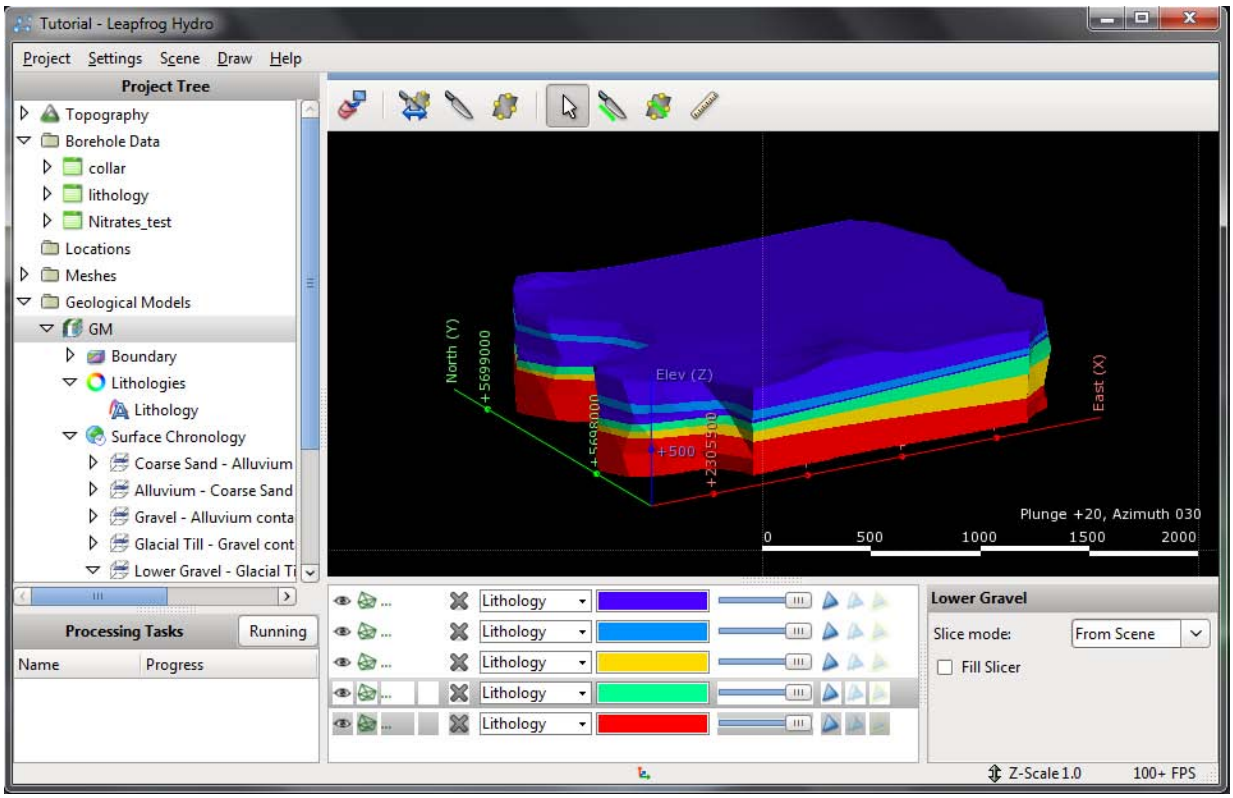
After creating each contact surface, drag the model into the scene to check the order of the processed volumes. Leapfrog will usually orient a surface correctly, but if this is not the case, you can correct the orientation by right-clicking on the surface and selecting **Swap Young Side**:



When all the surfaces have been generated, they should appear in the **Contact Surface Chronology** window as shown:



Once all volumes have been generated and displayed, the model should appear similar to the one below:



Tutorial 5: Building a Simple Numeric Model

A numeric model describes how a real, numerical quantity such as nitrate concentration or temperature varies in three-dimensional space.

Leapfrog Hydro contains a powerful 3D interpolation engine that can be used to interpolate any numeric data (e.g. piezometric head measurements) and build a numeric models. Numeric models are built from either:

- Columns of borehole data imported as numeric values
- Location arrays loaded with associated numeric values

In this tutorial, you will create a simple numeric model using nitrate concentation data imported for Tutorial 3: Importing and Working with Borehole Data. You will:

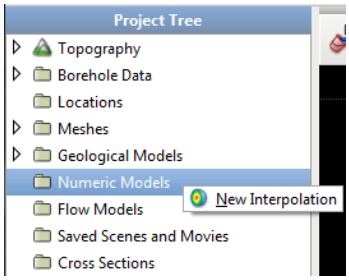
- Create a new numeric model
- View the model
- Change interpolation and surface parameters

Topics covered in this tutorial are:

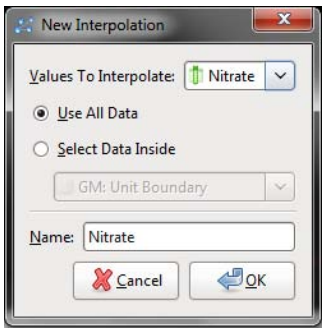
- Creating a New Numeric Model
- Viewing the Numeric Model
- Setting Interpolation and Surfaces Parameters

Creating a New Numeric Model

Open the tutorial project from Tutorial 4: Building a Simple Geological Model and click on the **Clear scene** button to clear any objects from the scene window. The data that will be used to create the model was loaded as part of Tutorial 3: Importing and Working with Borehole Data, so the first step is to right-click on the **Numeric Models** object and select **New Interpolation**:

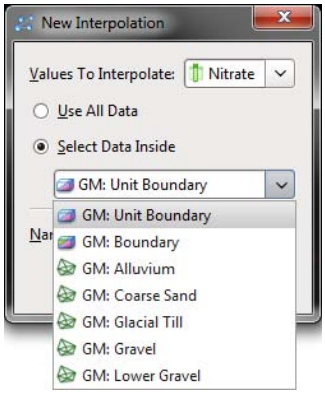


The **New Interpolation** window will appear:



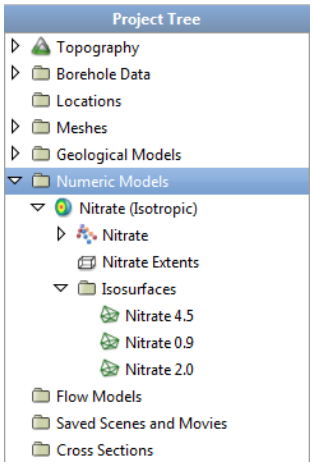
If more than one column of numeric data has been imported, the different columns can be selected from the **Values To Interpolate** dropdown list.

All data available can be used, or the selection can be limited to data inside specific boundaries selected from the dropdown list:



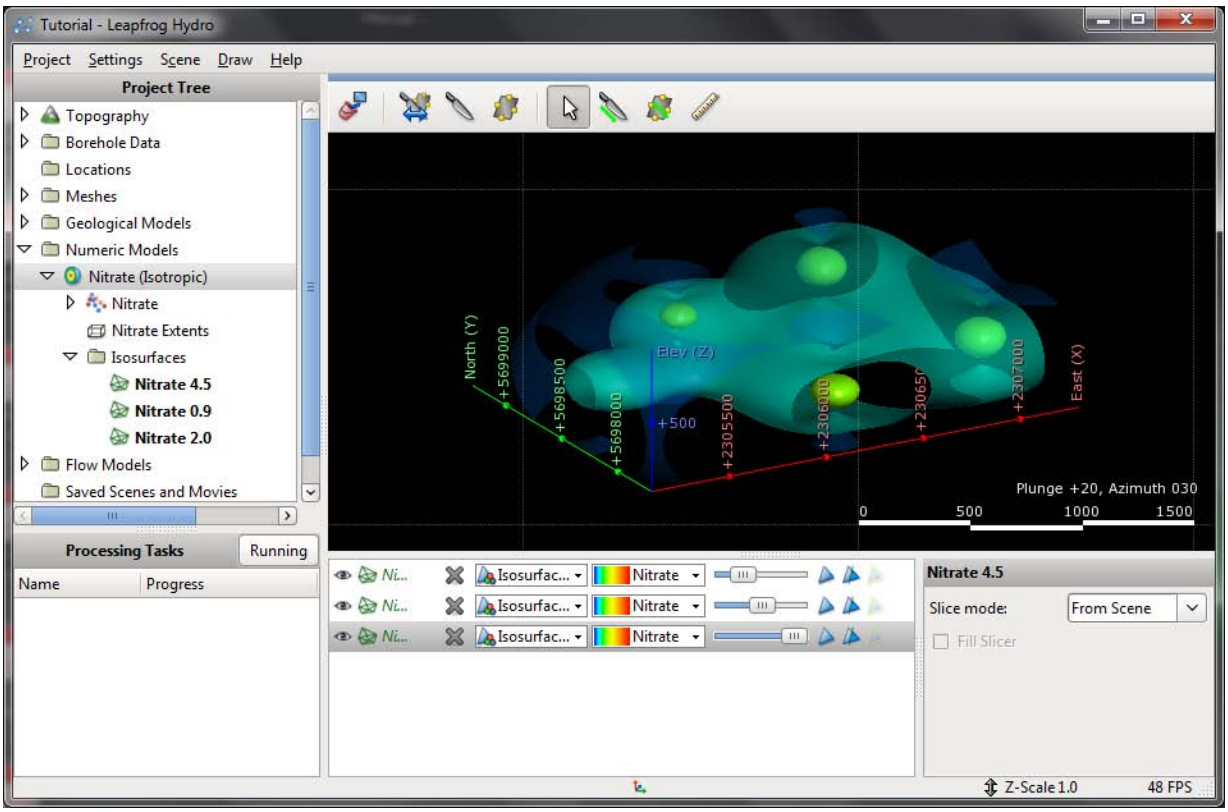
Select **Use All Data** and click **OK** to create the new model.

Hydro automatically generates a bounding box (Nitrate Extents) and and interpolates three iso-surfaces within that bounding box. These appear in the project tree under the **Numeric Models** object:



Viewing the Numeric Model

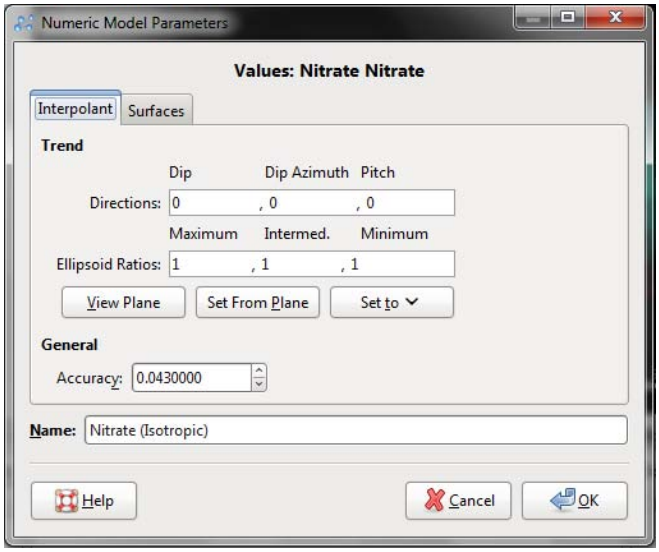
To view the model, add it to the scene by either dragging and dropping the model into the scene or by right-clicking on the model and selecting **View Object**.



A default trend is used when generating the model. This can be adjusted in the **Numeric Model Parameters** window, which is described in Setting Interpolation and Surfaces Parameters.

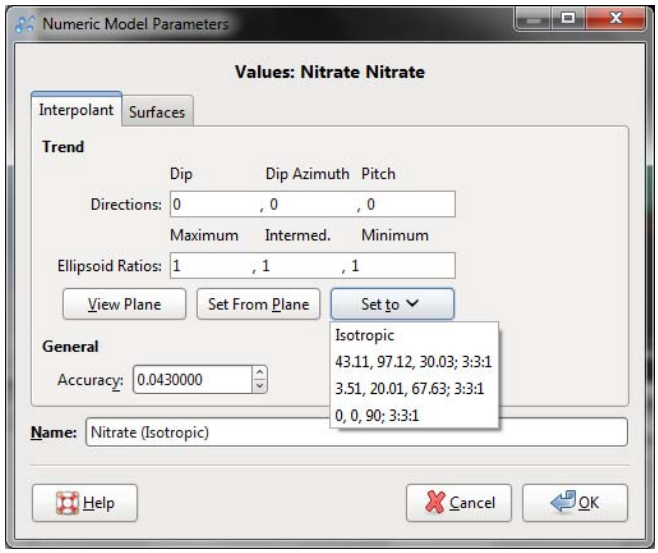
Setting Interpolation and Surfaces Parameters

The surfaces Leapfrog generated can be adjusted by double-clicking the numeric model in the project tree to open the **Numeric Model Parameters** window:

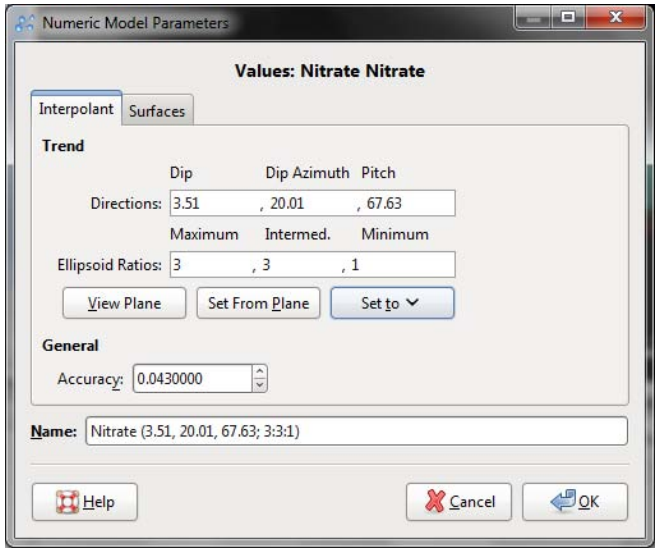


In the **Interpolant** tab, you can define the directional trend (anisotropy) and set the accuracy of the interpolation. For example, if the resulting surfaces weren't as faithful to the data as required, the accuracy could be reduced.

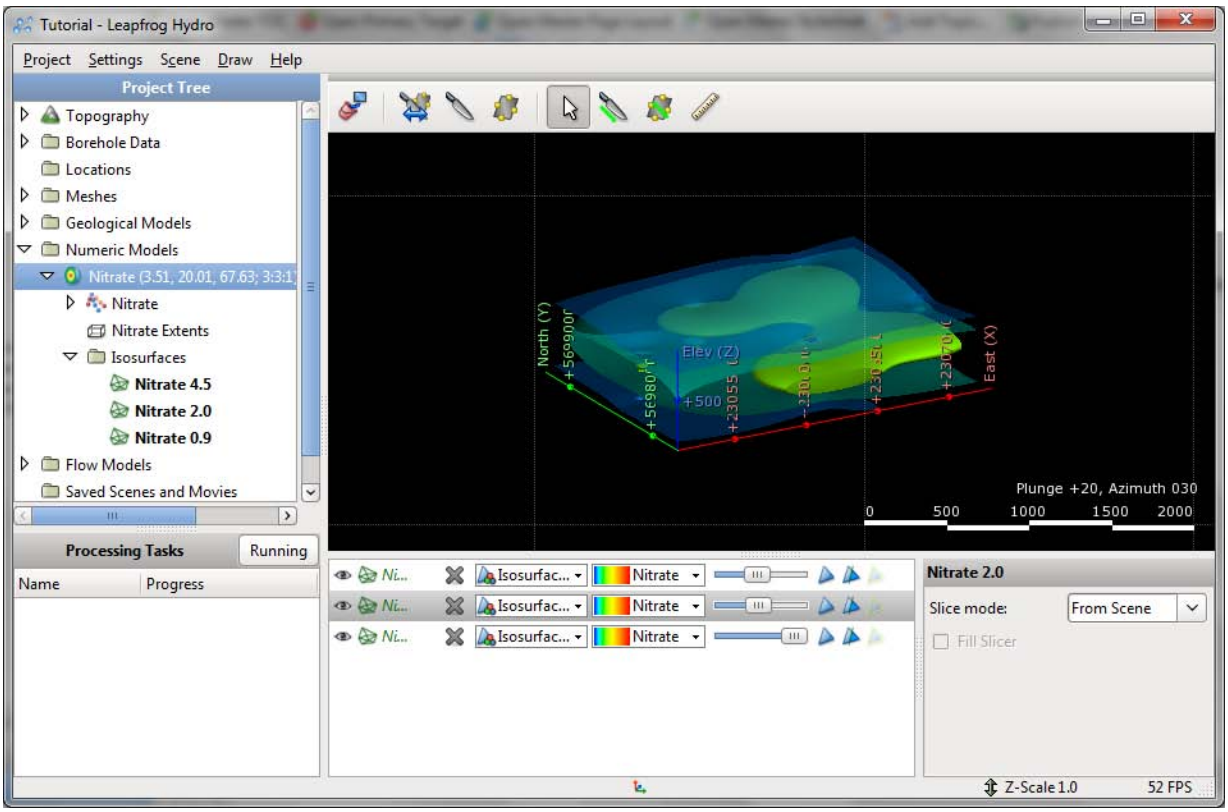
The **Set to list** contains a number of different options Leapfrog has generated based on the data used to build the model:



For the tutorial model, select the **3.51, 20.01, 67.63; 3:3:1** option. The **Trend** values will be updated to reflect the new setting:



Click **OK** to regenerate the model and view changes:



Part 5: Reference

This part describes Leapfrog Hydro features in greater detail than described in Part 3: Working with Leapfrog and in the tutorials. It is intended to help you get more out of Leapfrog once you have learned the basics.

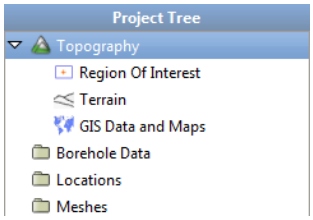
Topics are organised according to the objects in the project tree:

- The Topography Object
- The Borehole Data Object
- The Geological Model Object

The Topography Object

This topic describes the **Topography** object and the objects that are added to it in the process of building a Leapfrog Hydro project.

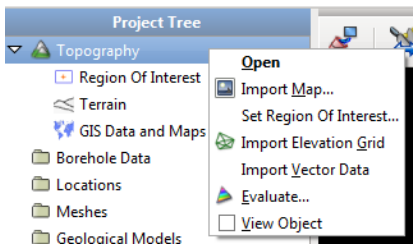
In Leapfrog, the **Topography** object represents the ground surface and all data associated with it. The **Topography** object is the first object in the project tree:



It contains three other objects:

- **Region of Interest**. This object represents the area in which data will be added and models created. The region of interest is important in limiting what data Leapfrog uses to perform calculations. Leapfrog automatically sets a region of interest, but for models with large data sets, adjusting the region of interest will reduce the amount of time Leapfrog requires for performing calculations.
- **Terrain**. This object defines the height of the data. The terrain can be derived from a single DTM, but a variety of sources can be used.
- **GIS Data and Maps**. This object contains all maps, photos and vector data that define land use.

All tools for defining the topography are accessed by right-clicking on the **Topography** object and the items in it:



Imported objects and objects created while building the topography are also stored as part of the **Topography** object, and more complex functions can be carried out by right-clicking on individual objects.

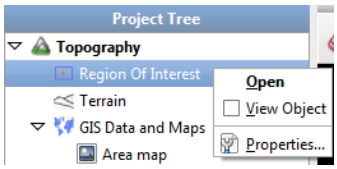
Tutorial 2: Defining the Topography describes how to use these objects to define the topography for a small set of data.

The Region of Interest Object

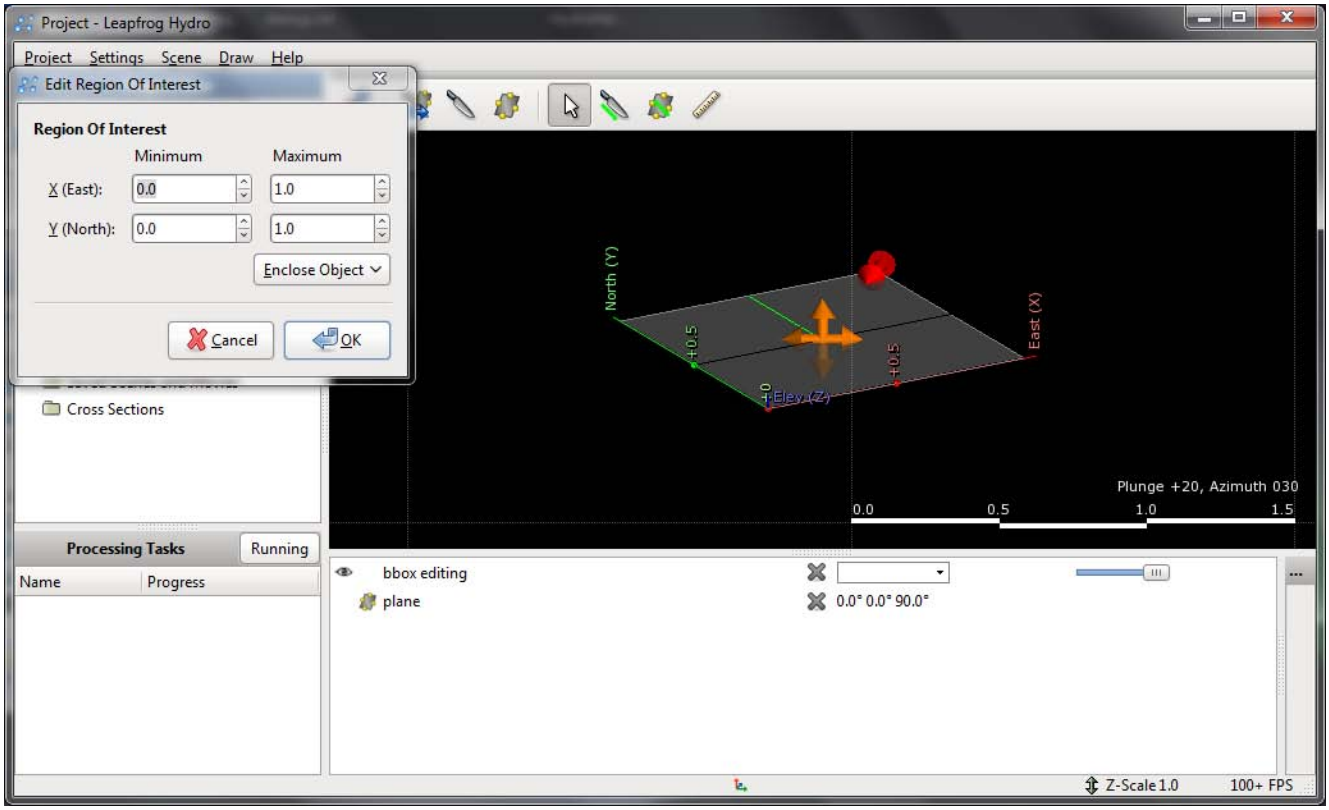
The **Region of Interest** object is used to define the area in which models will be built.

When a project is created, Leapfrog automatically sets the region of interest to a default size. As data is added to the project, the region of interest changes to fit that data. For models with large data sets, adjusting the region of interest will reduce the amount of time Leapfrog requires for performing calculations.

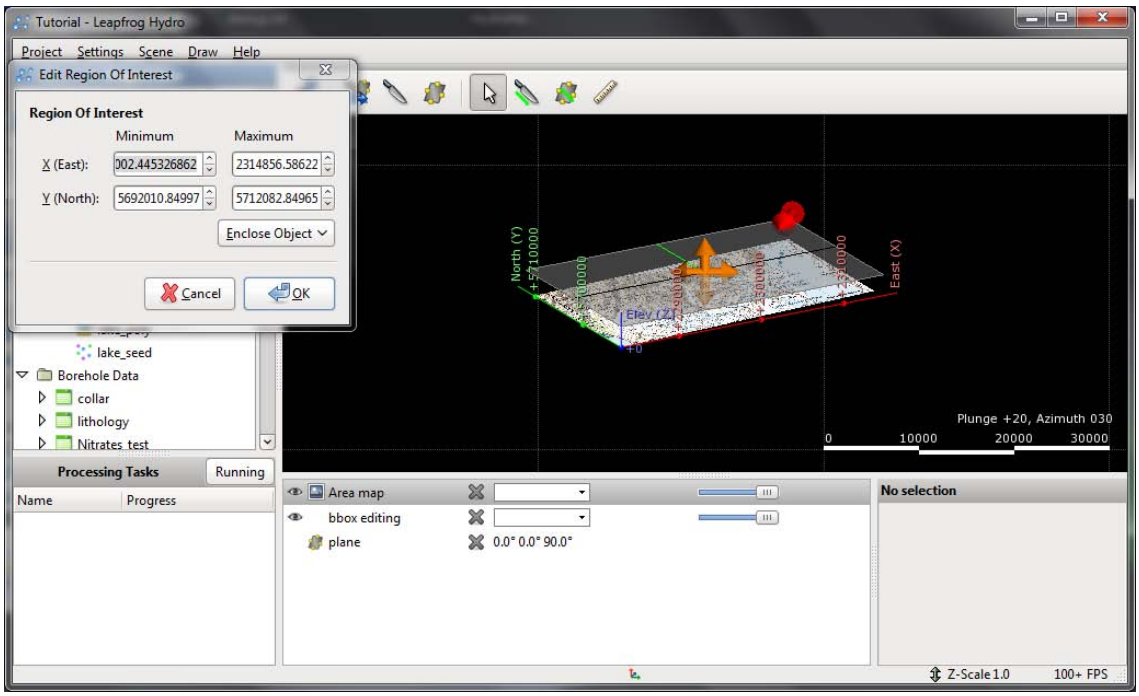
Although the region of interest automatically grows to fit any data that is imported, it can be set manually. To do this, double-click on the **Region of Interest** object in the project tree or right-click and select **Open**:



The **Edit Region of Interest** window will be displayed, together with controls in the scene that will help you to set the region of interest:



In the scene shown above, there is no data in the project, so the region of interest is set to default values. Once data has been added to the project, can be used as the basis for the region of interest, as shown below, where a map has been imported:



There are three ways to define the rectangular volume of interest:

- Enter the coordinates.
- Select **Enclose Object** and choose from the list of objects in the project. The region of interest will be updated to the size of the selected object.
- Use the controls that appear in the scene. The orange handle adjusts the center of the plane and the red handle adjust the size.

When you have finished adjusting the region of interest, click **OK**.

The Terrain Object

The **Terrain** object is used to define the elevation of the ground surface. The terrain will often be defined by a single digital terrain model (DTM), but there may be circumstances in which data from several sources is combined to ensure the data is consistent.

The terrain is used to bound geological models and to position GIS data onto the topography. As a result, changing the terrain will cause geological models to be recomputed and GIS data to be repositioned. This can be time-consuming, which is why the terrain should be defined before GIS data is added to the project and before any models are created.

The terrain can be built from:

- Points data. See Building the Terrain from Points Data.
- A surface. See Building the Terrain from a Surface.
- GIS vector data. See Building the Terrain from GIS Vector Data.
- An elevation grid. See Building the Terrain from an Elevation Grid.

Once the terrain is defined, maps will be automatically draped over the topography.

See Tutorial 2: Defining the Topography for an example of how a single DTM file can be used to build the terrain.

Building the Terrain from Points Data

To build the terrain from points data, that data must first be imported into the project.

Once the points data you wish to use has been imported, right-click on the **Terrain** object and select **Add Height Data > Points**. The **Select Points to Add** window will be displayed, showing points data available in the project:

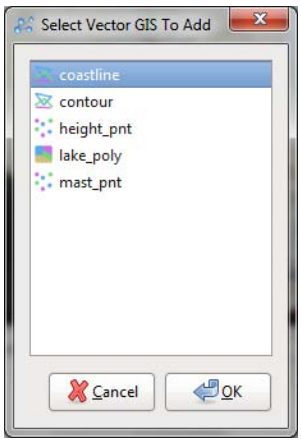
Select the information you wish to use and click **OK**.

The points data will appear in the project tree under the **Terrain** object.

Building the Terrain from GIS Vector Data

To build the terrain from GIS vector data, that data must first be imported into the project. See Importing Vector Data for more information.

Once the data you wish to use has been imported, right-click on the **Terrain** object and select **Add Height Data > GIS Vector Data**. The **Select Vector GIS Data to Add** window will be displayed, showing vector data in the project that contains elevation information:



Select the information you wish to use and click **OK**.

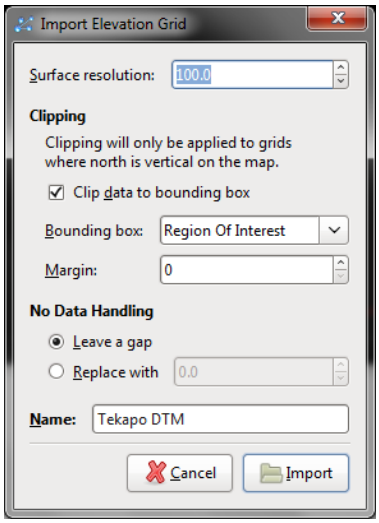
The vector data will appear in the project tree under the **Terrain** object.

Building the Terrain from an Elevation Grid

Elevation grid formats Hydro supports include:

- Arc/Info ASCII Grid files (*.asc, *.txt)
- Arc/Info Binary Grid files (*.adf)
- ESRI.hdr Labelled Image files (*.img, *.bil)
- GeoTIFF Image files (*.tiff, *.tif)
- Surfer ASCII or Binary Grid files (*.grd)
- Digital Elevation Model files (*.dem)

To build the terrain from an elevation grid, right-click on the **Terrain** object and select **Add Height Data > Import Elevation Grid**. Navigate to the folder that contains the elevation grid file and open the file. The **Import Elevation Grid** window will be displayed:



Leapfrog automatically sets a **Surface Resolution** based on the information in the file, but you can change the value if you wish. A lower value will produce more detail, but calculations will take longer.

Data is automatically clipped to the region of interest, but you can change how the data is filtered to suit any bounding box that exists in the project. If you do not wish to clip the data, untick **Clip data to bounding box**.

The **Margin** value determines how far outside the selected **Bounding box** the terrain will extend.

Select the information you wish to use and click **Import**.

The mesh will be added to the **Meshes** object and will also appear in the project tree under the **Terrain** object.

Building the Terrain from a Surface

To build the terrain from a surface, right-click on the **Terrain** object and select **Add Height Data > Surface**. The **Select Mesh to Add** window will be displayed, showing the meshes available in the project:

Select the surface you wish to use and click **OK**.

The surface will appear in the project tree under the **Terrain** object.

The GIS Data and Maps Object

The **GIS Data and Maps** object is used to import and manage maps, images and vector data that define land use in two dimensions.

Using the **GIS Data and Maps** object, you can:

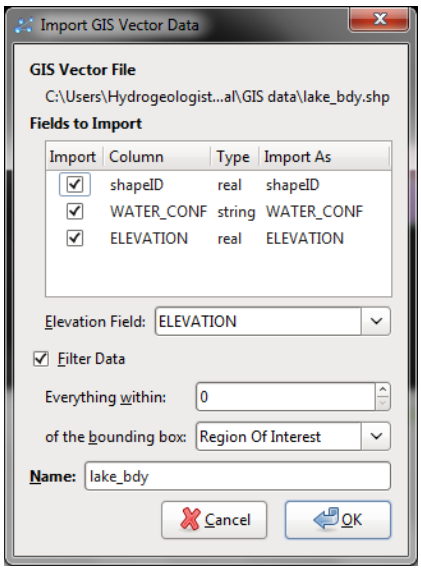
- Import vector data. See Importing Vector Data.
- Import maps. See Importing Maps.
- Create contour lines from a surface. See Creating Contour Lines from a Surface.

Importing Vector Data

Vector data formats Leapfrog supports include:

- ESRI Shape files (*.shp)
- MapInfo files (*.tab, *.mif)
- ESRI Personal GeoDatabase files (*.mdb)

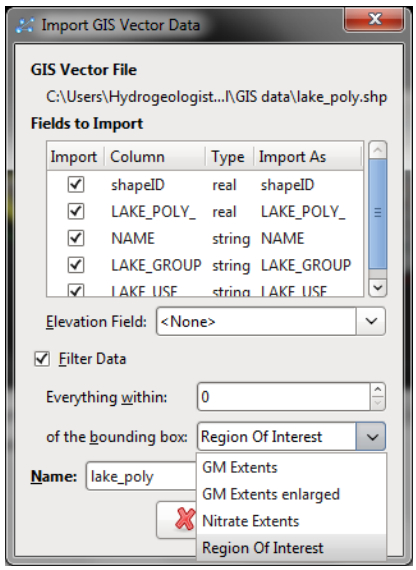
To import vector data, right-click on the **GIS Data and Maps** object and select **Import Vector Data**. Navigate to the folder that contains GIS data file and select the file. The **Import GIS Vector Data** window will be displayed:



A summary of the data is displayed.

You can choose what field in the data to use for the **Elevation Field** from the dropdown list.

Data is automatically clipped to the region of interest, but you can change how the data is filtered to suit any bounding box that exists in the project. For example, for the file being imported below, the bounding box options available include the extents of several models:



If you do not wish to clip the data, untick the **Filter Data** box.

Click **OK** to add the data to the project. The object will appear in the project tree under the **GIS Data and Maps** object.

Importing Maps

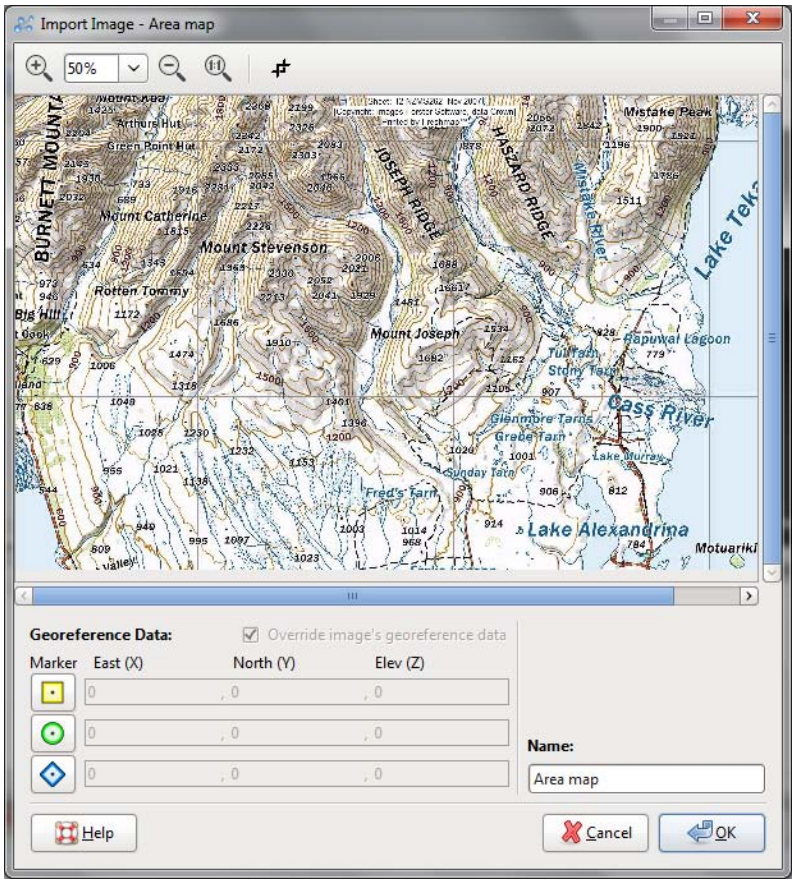
Image formats Leapfrog supports include:

- PNG files (*.png)
- JPEG files (*.jpg, *.jpeg)

- TIFF and GeoTIFF files (*.tiff, *.tif)

If the file contains georeference information, it too will be imported.

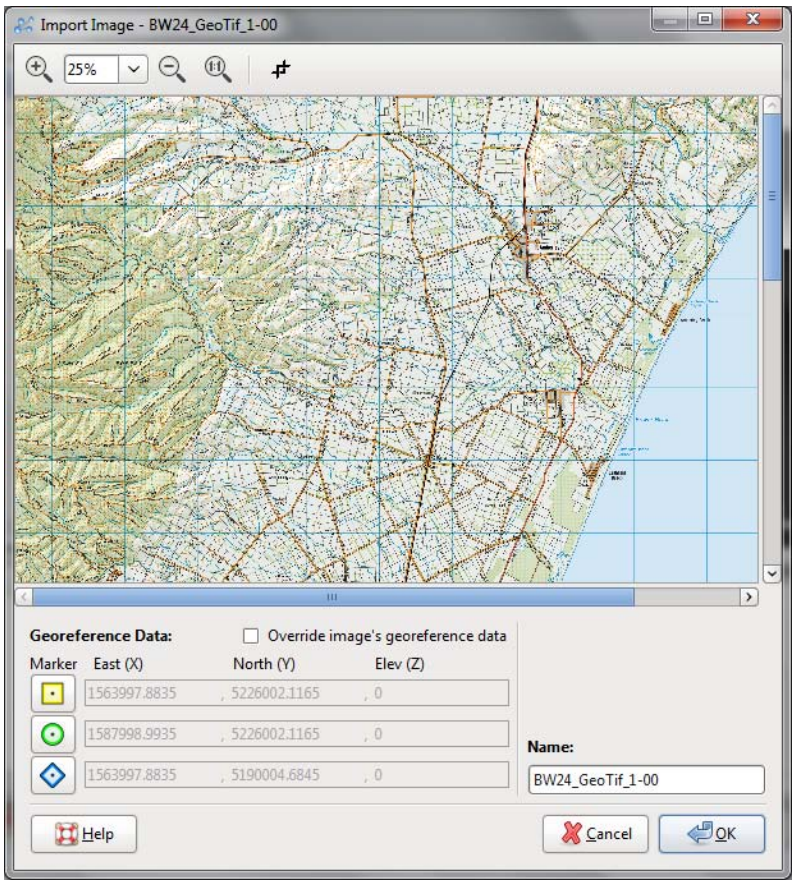
To import a map, right-click on the **GIS Data and Maps** object and select **Import Map**. Navigate to the folder that contains GIS data file and select the file. The **Import Image** window will be displayed:



You can crop the image, if required.

The **Name** field displays the name of the imported file. If you wish to use a more descriptive name in the project file, enter it in this field.

If the image contains georeference data, it will automatically be added to the map and displayed:



You can edit the imported georeference data by ticking the **Override image's georeference data** box, then editing the information.

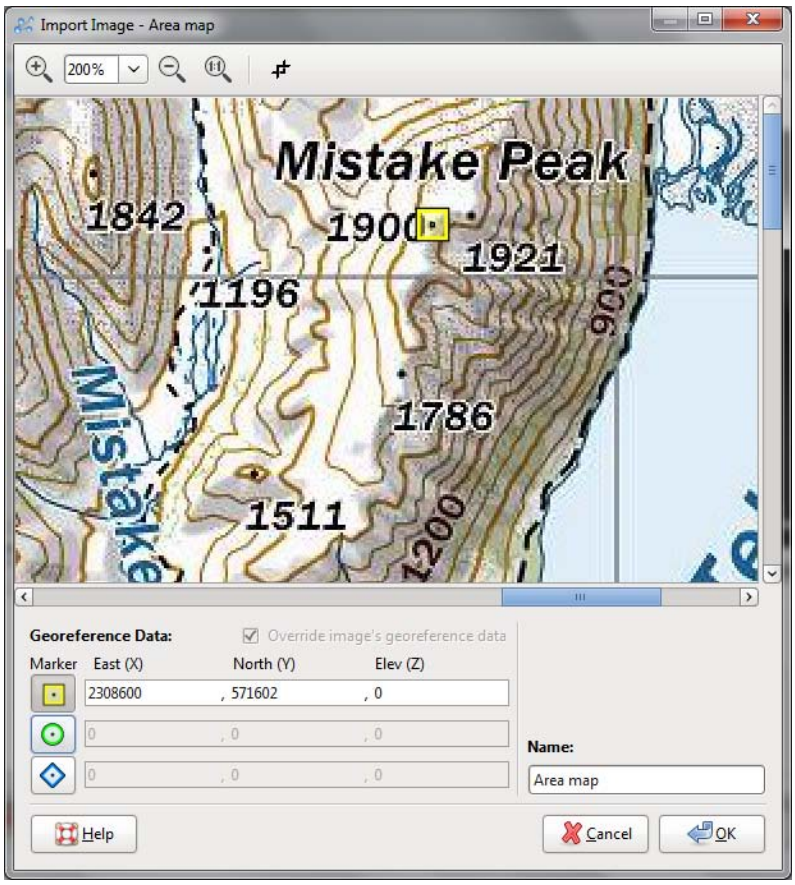
If the image does not contain georeference information, you will need to add it manually by adding three reference markers. To add georeference data to the map, start adding markers to the image. There are two ways to do this:

- Click on a marker to add the marker to the image.
- Drag a marker and drop it onto the map.

Whichever method is used, you can move the marker to the required position by dragging it or by using the arrow keys. You can also use the zoom controls for more precise control over the area in which you are positioning a marker.

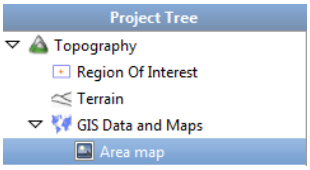
To select a different marker, click on it in the list, then move it using the mouse or the arrow keys.

Once each marker is correctly positioned, enter the real-world X and Y coordinates for each marker:



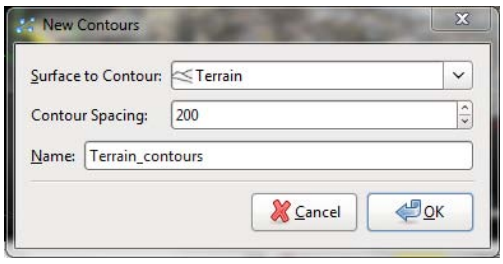
The **Elev (Z)** value can be set to zero, as height information is added to the project as part of creating the terrain. See Georeferencing in Leapfrog.

Once you have set the coordinates for each marker, click **OK** to return to the main window. Once Leapfrog saves the map, it will appear in the project tree under the **GIS Data and Maps** object:



Creating Contour Lines from a Surface

To create contour lines from a surface, right-click on the **GIS Data and Maps** object and select **Contour Lines From Surface**.



The **Surface to Contour** dropdown list contains all the surfaces in the project. Select the surface to use for the new contour and adjust the **Contour Spacing**, if required. Click **OK** to generate the surface.

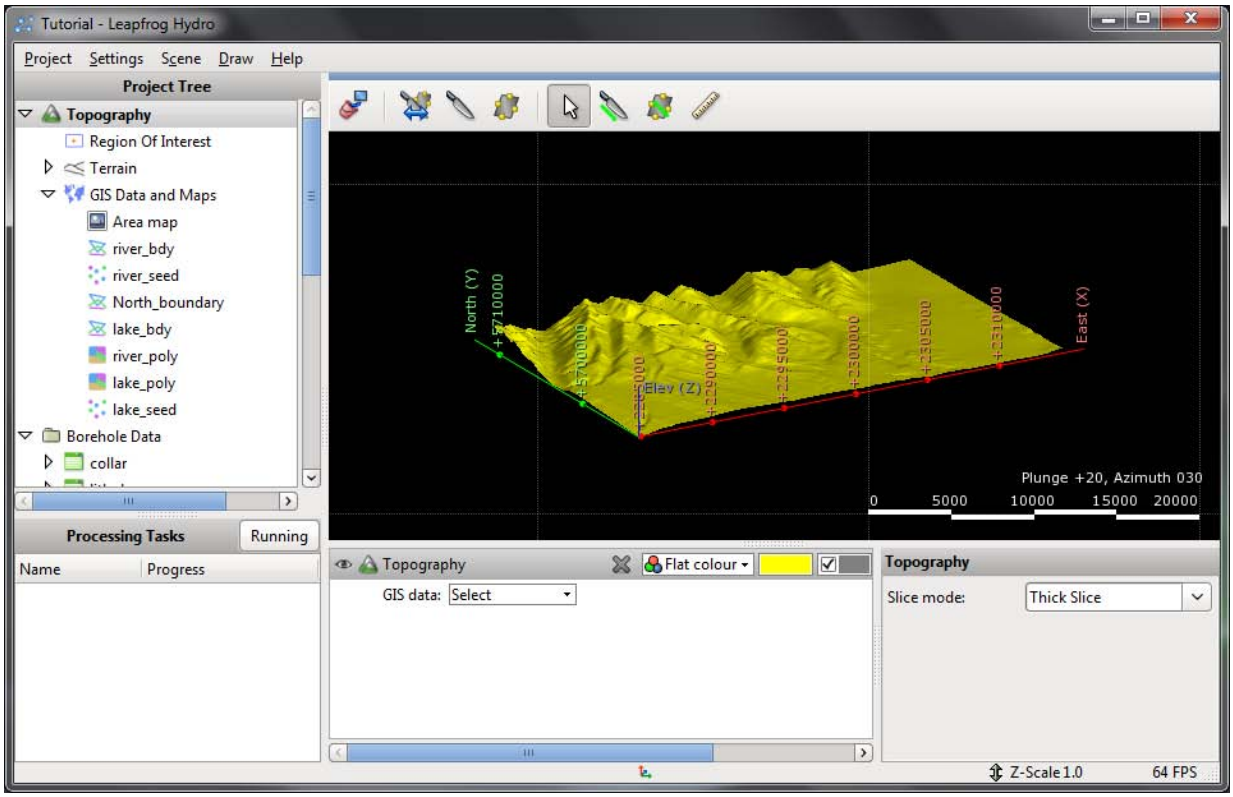
The new contours will appear in the project tree under the **GIS Data and Maps** object.

Displaying Topography in the Scene Window

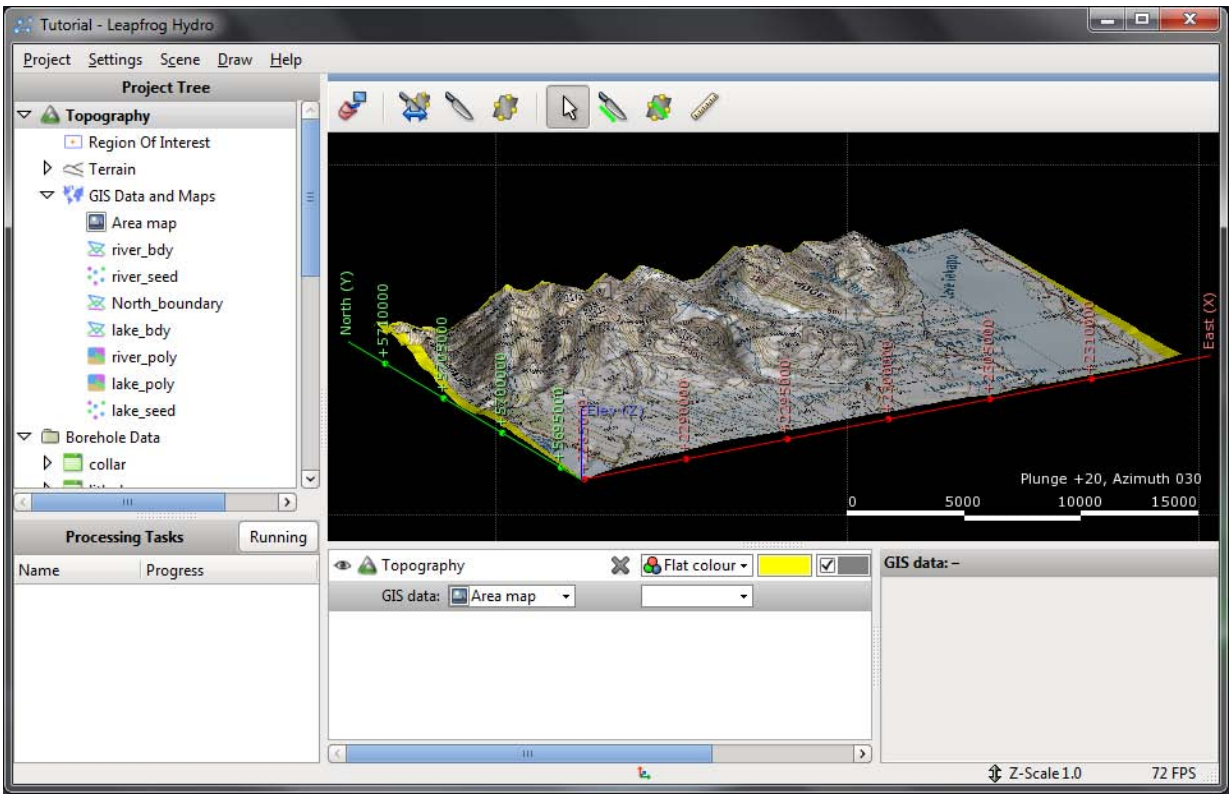
To display the topography in the scene window, either:

- Right-click on the **Topography** object and select **View Object**.
- Click on the **Topography** object and drag it into the scene.

The topography will be displayed in the scene window and in the shape list:



The **Topography** object in the shape list provides additional controls that can be used to change the way the topography is displayed and highlight features of interest. For example, selecting an imported map from the dropdown list results in it being displayed draped onto the terrain:



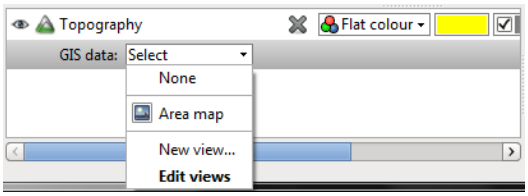
GIS data objects can also be displayed in this manner by selecting them from the **GIS data** list.

You can also create custom views combining multiple GIS data and map objects. See Creating Custom Topography Views for more information.

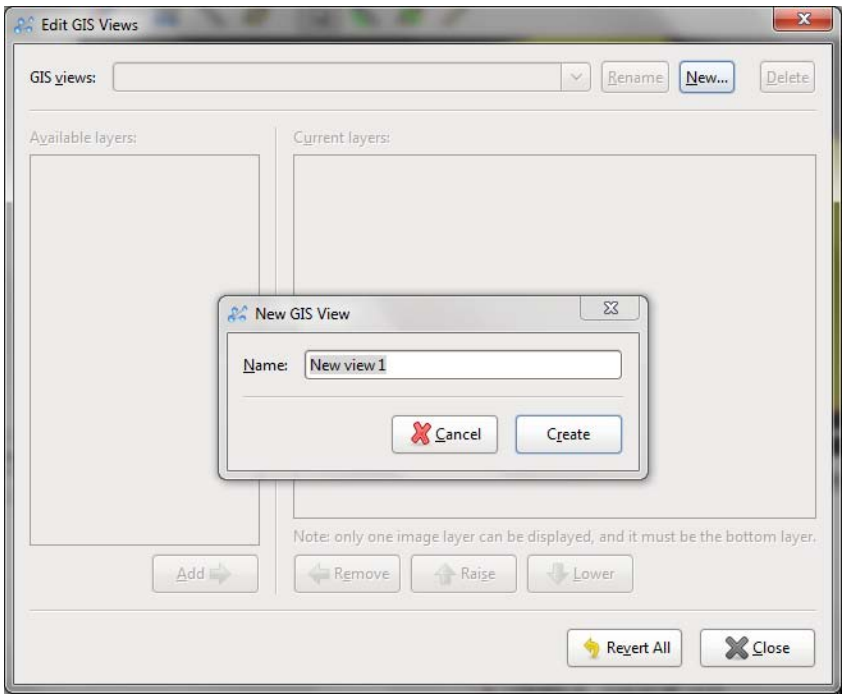
Creating Custom Topography Views

In Leapfrog, you can create custom views of the topography to change the way the topography is displayed and highlight features of interest. To do this, drag the **Topography** object into the scene. The topography will appear in the shape list with a **GIS data** dropdown list.

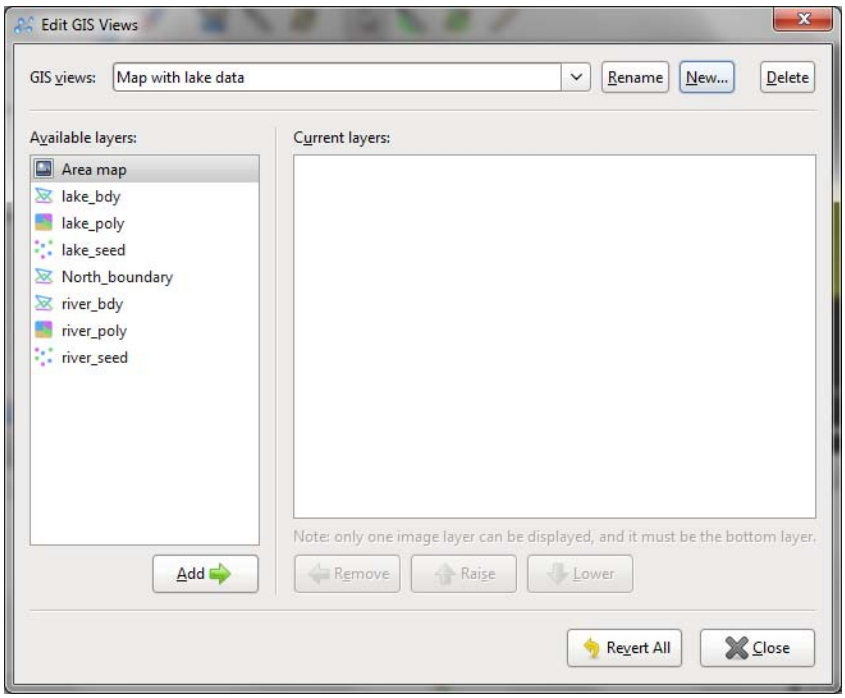
Select **New View** from the **GIS data** list:



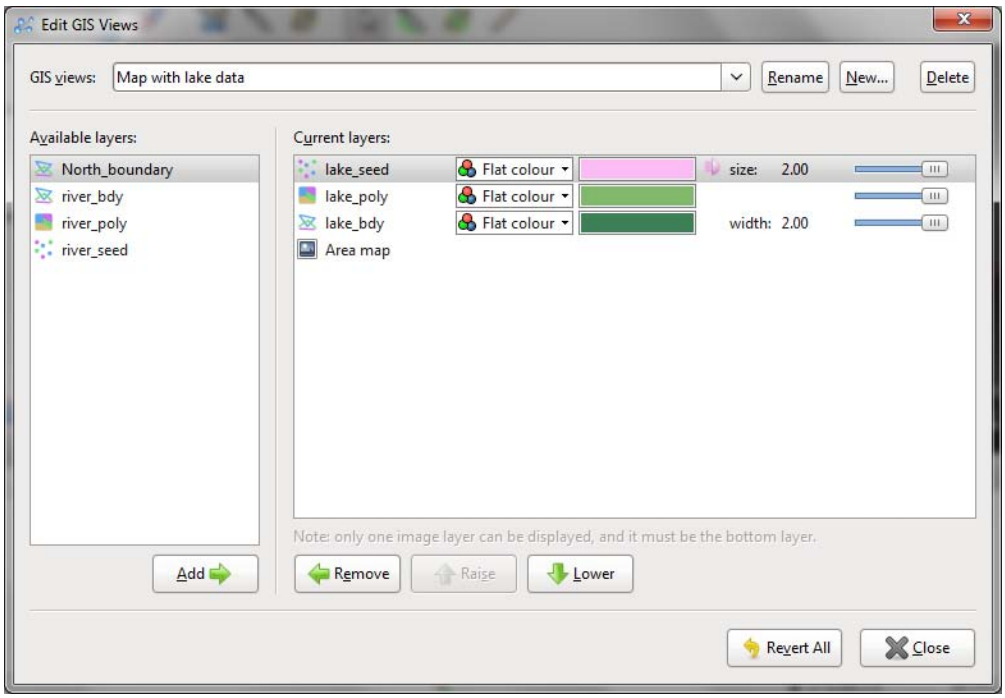
The **Edit GIS Views** window will appear with a prompt asking you to enter a name for the new view:



Enter a name for the view and click **Create**. The new view will be created in the **Edit GIS Views** window, together with a list of **Available layers**:

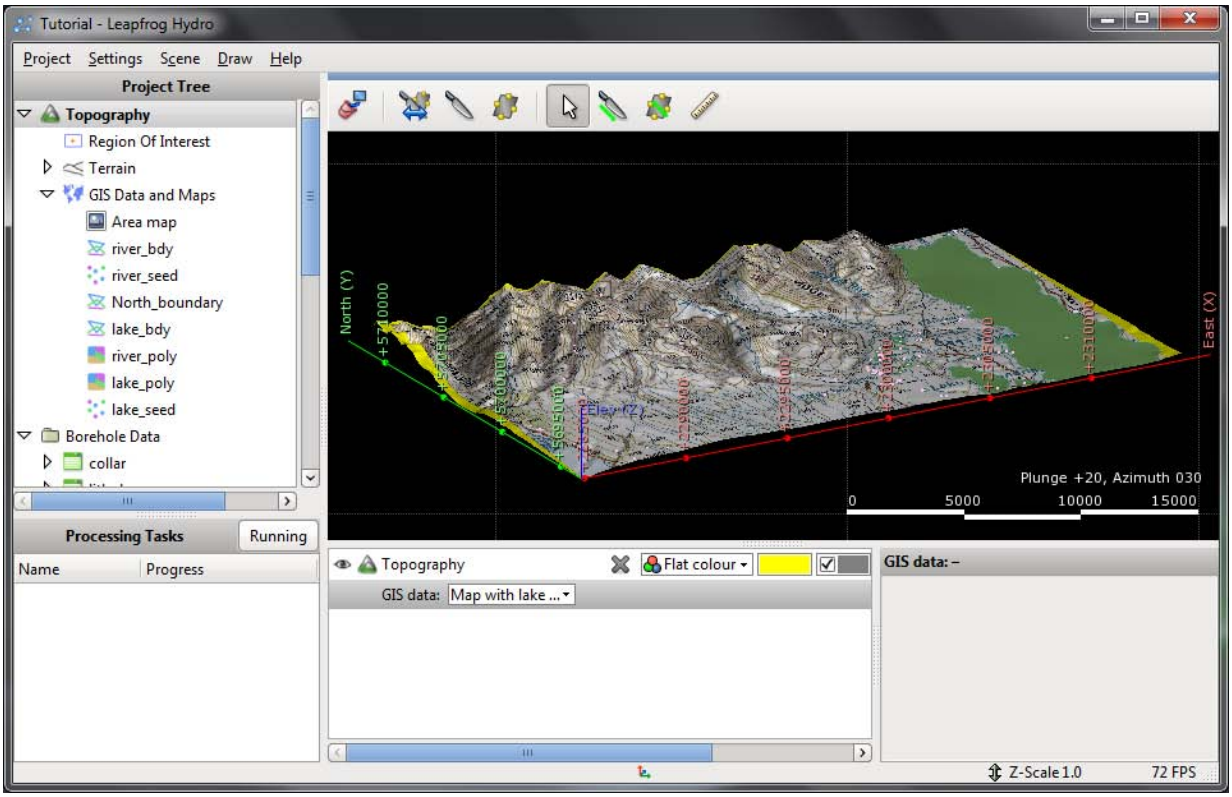


To add a layer to the view, click on it, then click the **Add** button. You can change how the different layers are displayed using the controls in the **Current layers** list and change the order of layers using the **Raise** and **Lower** buttons.



When you have added and arranged all the required layers, click **Close**.

In the main window, the topography displayed in the scene window will be updated to reflect the new view:



Georeferencing in Leapfrog

Leapfrog imports georeference information as X, Y, Z coordinates. It is not necessary to specify what georeferencing standard is being used.

If the data you are importing uses, for example, latitude and longitude for X and Y, but feet for elevation, you can scale the Z axis to a value that provides a better picture of the information being displayed. See Scale Z Axis (Settings Menu) for more information.

If you are importing data that uses different coordinate systems, you will need to process the data so that it is using the same coordinate system.

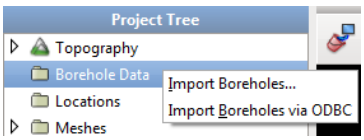
The Borehole Data Object

This topic describes the **Borehole Data** object and the objects that are added to it in the process of building a Leapfrog project.

In Leapfrog, borehole data defines the physical 3D shape of boreholes. It is made up of:

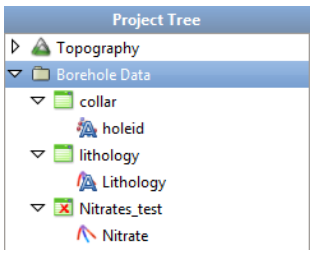
- A collar table, containing information on the location of the borehole.
- At least one interval table, containing information on measurements such as lithology, date or any numeric or textual values. An interval table must also include collar IDs that correspond to those in the collar table and sample start and end depth.

When a project is first created, the **Borehole Data** object is empty and the only options available are for importing data:



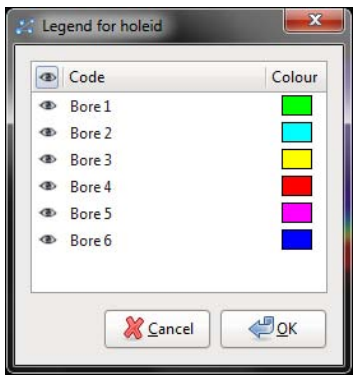
See Importing Borehole Data and Importing Borehole Data via ODBC for more information on these options.

Once data has been added to the project, each table imported appears under the **Borehole Data** object:



When there are errors in the data, the relevant table will be marked with a red X. See Correcting Borehole Data Errors for more information on fixing data errors.

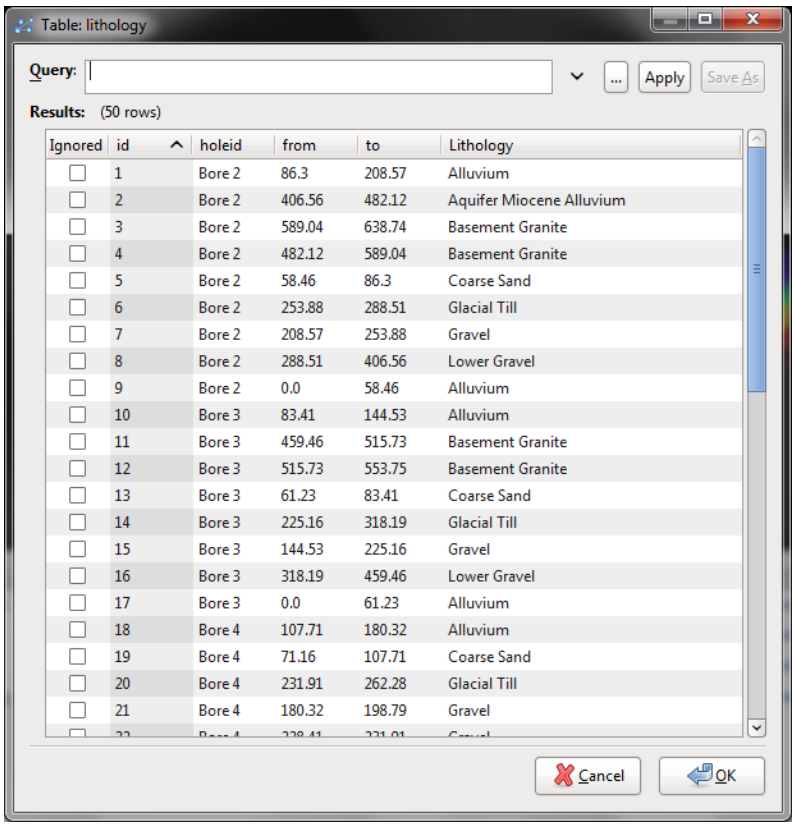
The objects included as part of the collar and lithology tables are the legend for those tables. Double-click on a legend to open it:



You can change the colours used by clicking on the colour chips and selecting a new colour.

The object included as part of the nitrate table indicates that it contains numeric data.

You can open each table by double-clicking on the table icon. The table will be displayed and you can make changes:

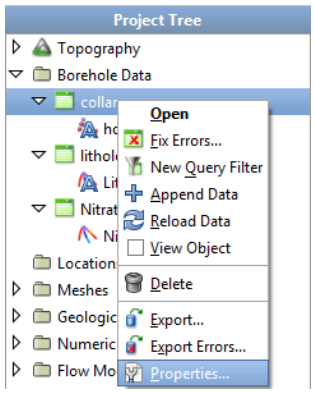


Ignored	id	holeid	from	to	Lithology
	1	Bore 2	86.3	208.57	Alluvium
	2	Bore 2	406.56	482.12	Aquifer Miocene Alluvium
	3	Bore 2	589.04	638.74	Basement Granite
	4	Bore 2	482.12	589.04	Basement Granite
	5	Bore 2	58.46	86.3	Coarse Sand
	6	Bore 2	253.88	288.51	Glacial Till
	7	Bore 2	208.57	253.88	Gravel
	8	Bore 2	288.51	406.56	Lower Gravel
	9	Bore 2	0.0	58.46	Alluvium
	10	Bore 3	83.41	144.53	Alluvium
	11	Bore 3	459.46	515.73	Basement Granite
	12	Bore 3	515.73	553.75	Basement Granite
	13	Bore 3	61.23	83.41	Coarse Sand
	14	Bore 3	225.16	318.19	Glacial Till
	15	Bore 3	144.53	225.16	Gravel
	16	Bore 3	318.19	459.46	Lower Gravel
	17	Bore 3	0.0	61.23	Alluvium
	18	Bore 4	107.71	180.32	Alluvium
	19	Bore 4	71.16	107.71	Coarse Sand
	20	Bore 4	231.91	262.28	Glacial Till
	21	Bore 4	180.32	198.79	Gravel

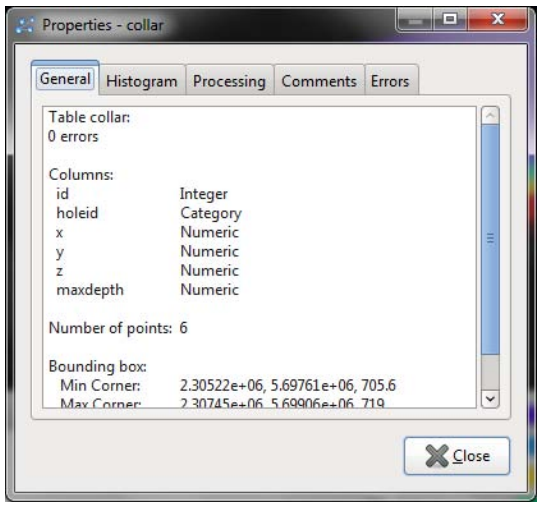
Add the tables to the scene by dragging them into it or by right-clicking on them in the project tree and selecting **View Object**.

Viewing Borehole Data Object Properties

You can also view the properties of each imported object by right-clicking and selecting **Properties** or by pressing `Alt+Enter` on the keyboard:



The information displayed in the **Properties** window depends on the object selected. For example, the properties for the collar table appear as:



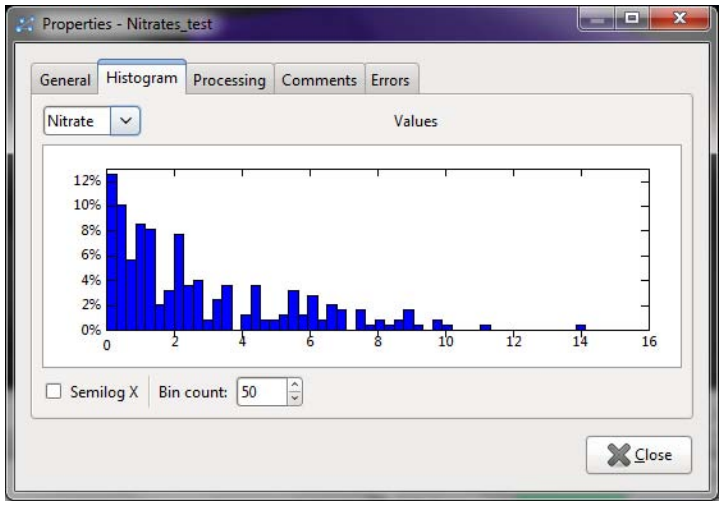
Some objects have tabs for information unique to that object, but generally, there are three tabs:

- The **Processing** tab shows the current status of the processing.
- In the **Comments** tab, you can enter information about the object.
- The **Errors** tab shows errors that have occurred while processing the table.

Information entered in the **Comments** tab are saved when the **Properties** window is closed.

For table objects, the **General** tab shows the table's data structure.

Interval tables and numeric data tables may also have a **Histogram** tab that shows the distribution of the data in a particular column:



If the table contains several columns, you can display histograms for each column. If so, you will be able to choose the other columns from the dropdown box. You can also:

- Adjust the number of intervals in the histogram by changing the **Bin count**.
- Remove values less than zero by ticking the **Semilog X** box.

Changes made to the way the histogram is displayed are not saved when the **Properties** window is closed.

Using the Borehole Data Object to Import Borehole Data

This section describes how to use the **Borehole Data** object to import borehole data. Topics are:

Expected Borehole Data Format

Hydro expects borehole data that is stored in a collar table and at least one interval table. Each project can have only one collar file, but multiple interval tables can be imported.

The collar table should contain five columns:

- A borehole identifier
- The location of the borehole in X, Y and Z coordinates
- The maximum depth of the borehole

The maximum depth value is used to validate the data imported for the interval tables.

For interval tables, Leapfrog expects, at minimum, four columns:

- A borehole identifier
- Start and end depths
- A column of measurements

If a borehole ID in an interval table does not correspond to one in the collar file, the interval table will contain errors.

Supported column types are:

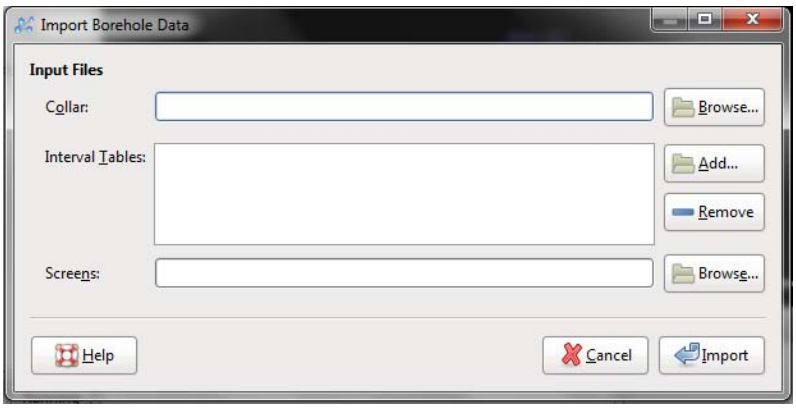
- Lithology column containing lithologic data, which can be used for geological modelling.
- Numeric column containing numeric values, which can be used for numeric modeling. Numeric data columns are not validated when imported.
- Category column, which is text representing categories such as company, geologist, or mineralization.
- Text column containing text data that is not categorical, such as comments. Text columns are not validated when imported.
- Date column containing date data. Custom date and timestamps formats are supported.

Leapfrog imports tables from CSV and text files.

Importing Borehole Data

See Tutorial 3: Importing and Working with Borehole Data for an example of importing borehole data.

To import borehole data, right-click on the **Borehole Data** object and select **Import Boreholes**. The **Import Borehole Data** window will appear:

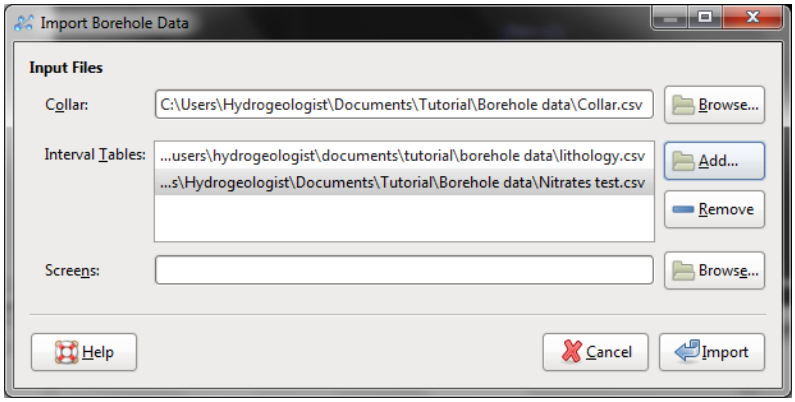


For the **Collar**, click on the **Browse** button to locate the collar file. Leapfrog will then look for interval tables in the same location and will display them in the **Interval Tables** list. If a table you wish to import does not appear, you can add it manually by clicking on the **Interval Tables – Browse** button.

You can also add any screens to import by clicking on the **Screens – Browse** button.

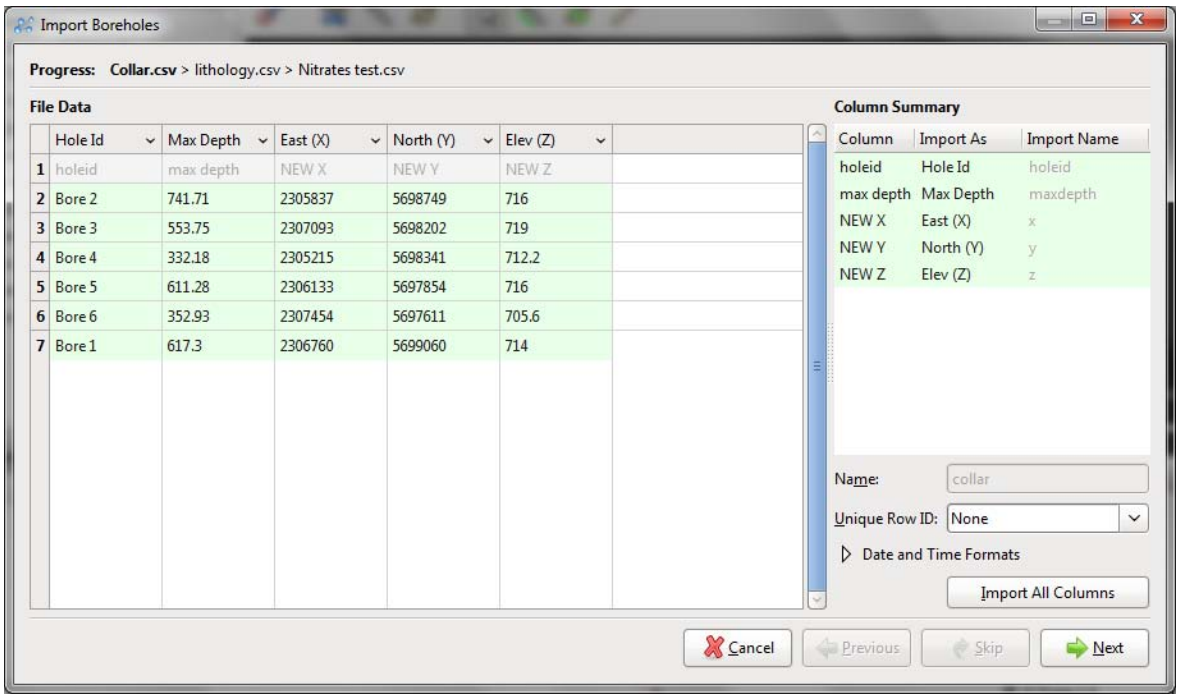
Interval table files can also be removed from the list by clicking on the file to highlight it, then on the **Remove** button.

In the screen below, a collar file and two interval tables have been selected for import:



Clicking on **Import** starts the process of importing data. Leapfrog will display a series of windows, one for each file, and will attempt to match the data found in the files with the columns expected, based on the header row in each file.

For the collar table, a window similar to that shown below will be displayed:



The breadcrumb at the top of the window indicates progress in importing the borehole files.

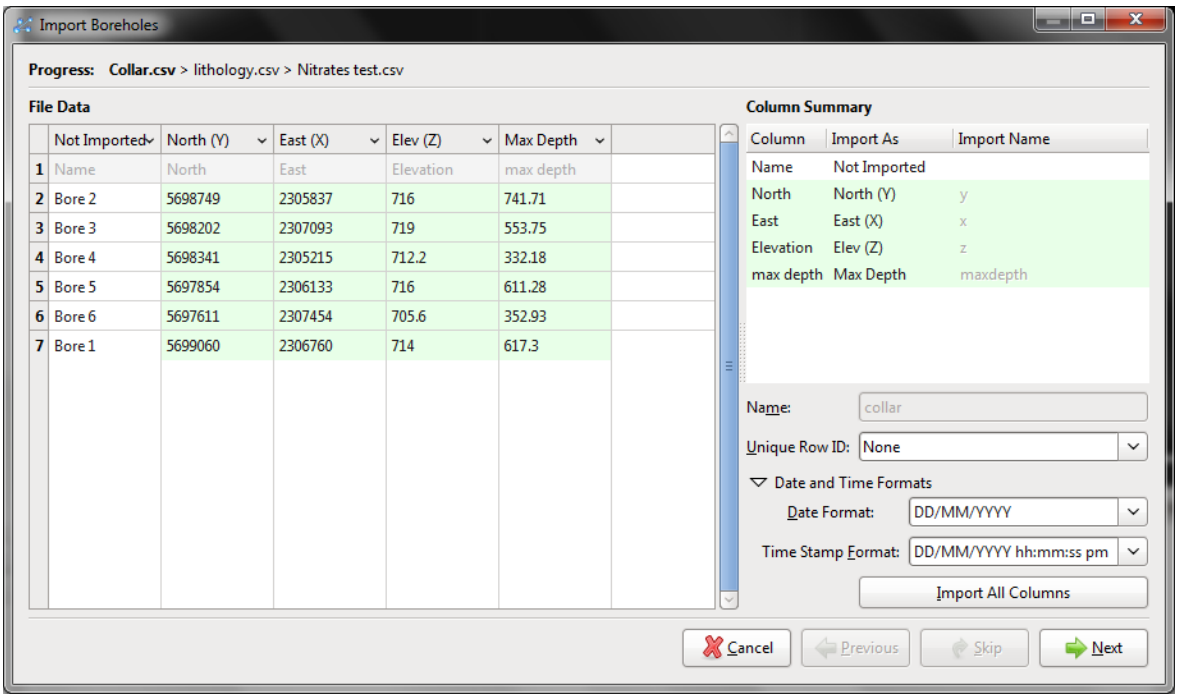
The series of dropdown lists along the top of the table indicates the label Leapfrog will assign to the imported data. If Leapfrog has incorrectly identified a column, choose the correct assignment from the dropdown list. You can also assign a data type using this dropdown list.

If Leapfrog has marked a column as "Not Imported" and you wish to import that column, select a label and data type for that column from the dropdown list. If you wish to import all columns, click on the **Import All Columns** button.

If you choose not to import a column then wish to do so later, you can add it later using the **Import Column** option. See Importing Columns.

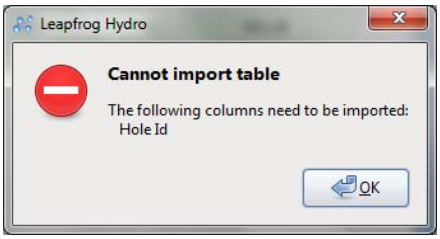
The Max Depth column is not required, but because the information is in the file, Leapfrog will use it to verify the integrity of the data. If that information is not included in a collar file, Leapfrog will determine it from the interval tables.

If the data contains date and time information, click on **Date and Time Formats** to reveal more information and modify the format, if necessary:



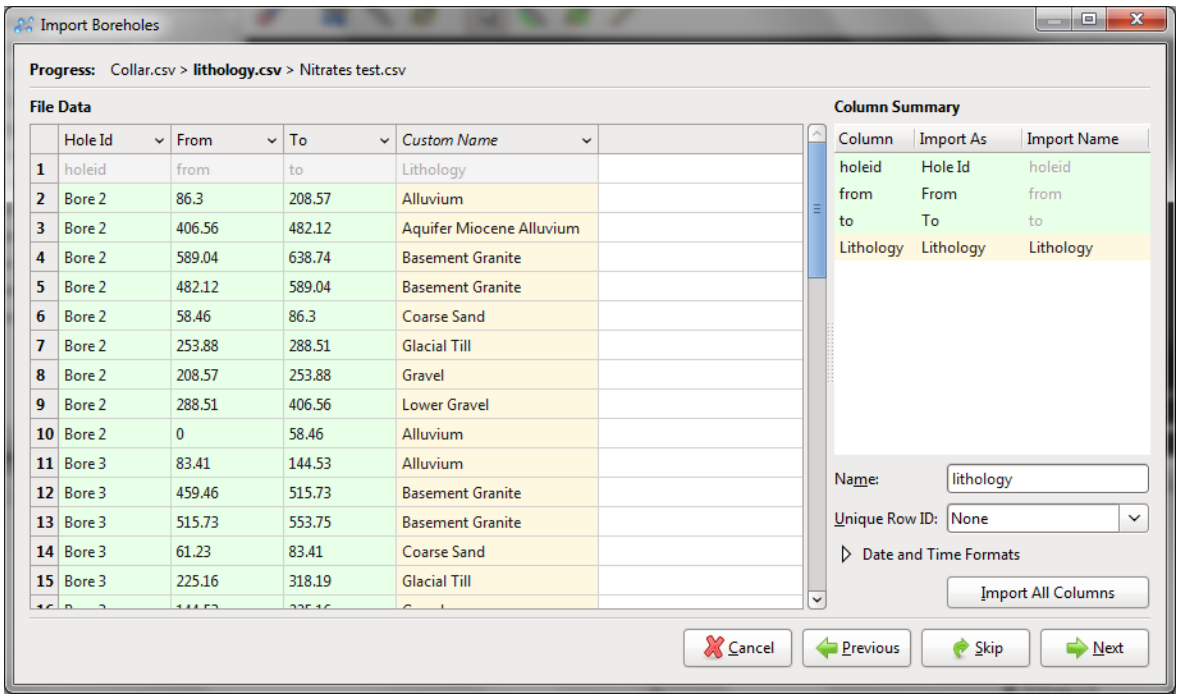
If the date and time format you wish to use is not among the options, you can create a custom format.

Once you are satisfied with the assigned column names, click **Next** to proceed to the next step. If any of the information Leapfrog expects is missing, an error message will appear:



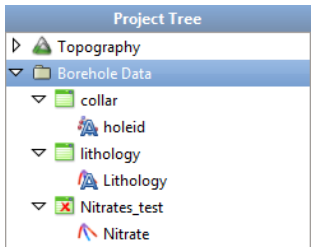
Return to the table and correct the problem described.

Next, the interval table data will be displayed:



Again, Leapfrog attempts to match the data found in the file with the columns expected. Ensure that the columns have been correctly identified, then click **Next** to move on to the next file.

Once you have checked the last file, click on **Finish** complete the process. Once Leapfrog has imported the data, it will appear in the project tree under the **Borehole Data** object, with tables containing incorrect data marked with a red X:



Importing Borehole Data via ODBC

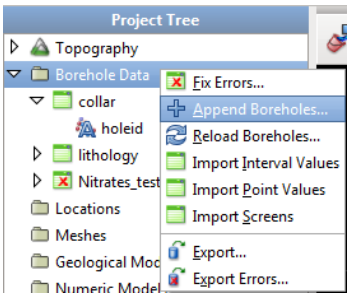
To import borehole data directly from any database that uses an ODBC interface, right-click on the **Borehole Data** object and select **Import Borehole Data via ODBC**. The **Select ODBC Data Source** window will appear:

Enter the information supplied by your database administrator and click **OK**.

Once the data source is selected, the importation process is similar to that described in Importing Borehole Data.

Appending Boreholes

If borehole data is stored in multiple files, you can import other files into the project and add them to the existing borehole data. To do this, right-click on the **Borehole Data** object and select **Append Boreholes**:



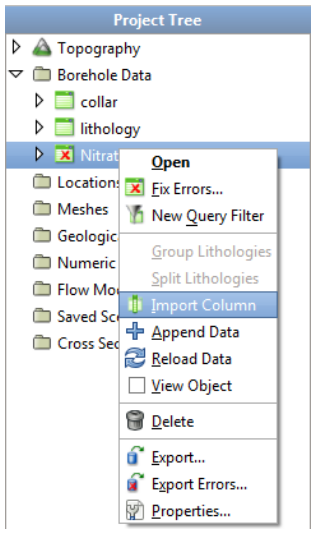
The **Append Boreholes** window is displayed:

Appending borehole data is similar to importing the original collar and interval tables (see Importing Borehole Data). Leapfrog will ask you to specify the file location, and then will display the data in the file in the **Import Tables for Appending** window.

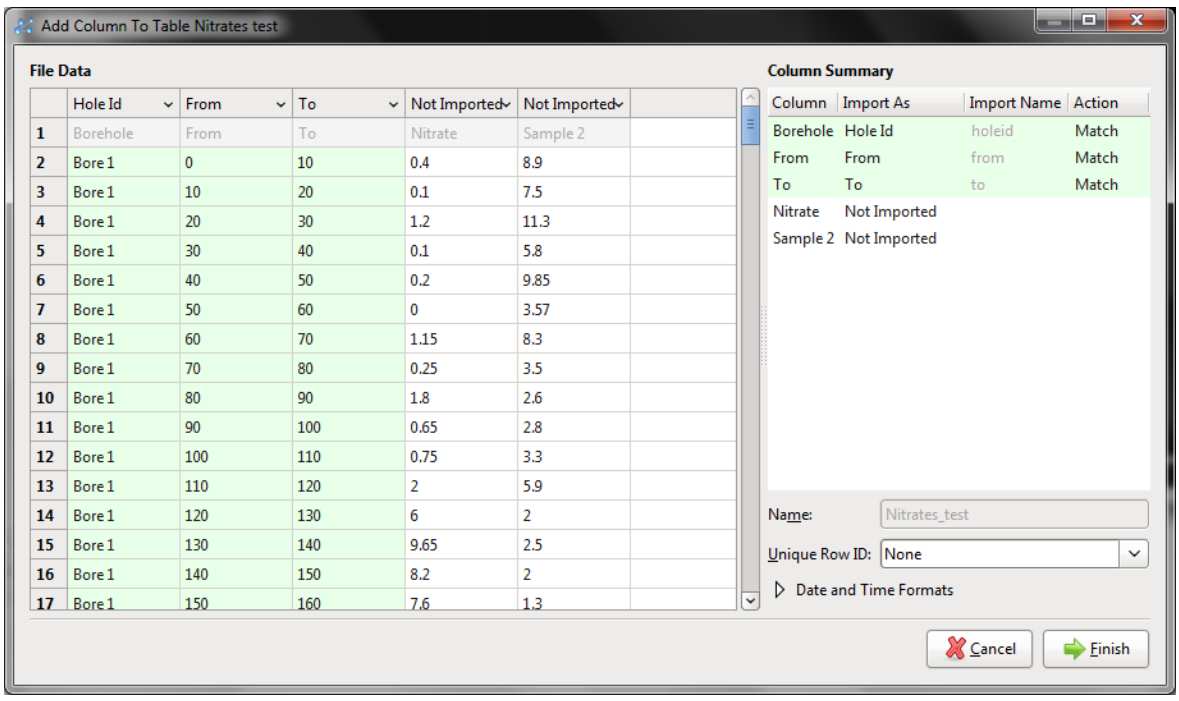
Step through the files, checking the **Column Summary** for each file to ensure that the correct information will be imported, then click **Finish** to add the new files.

Importing Columns

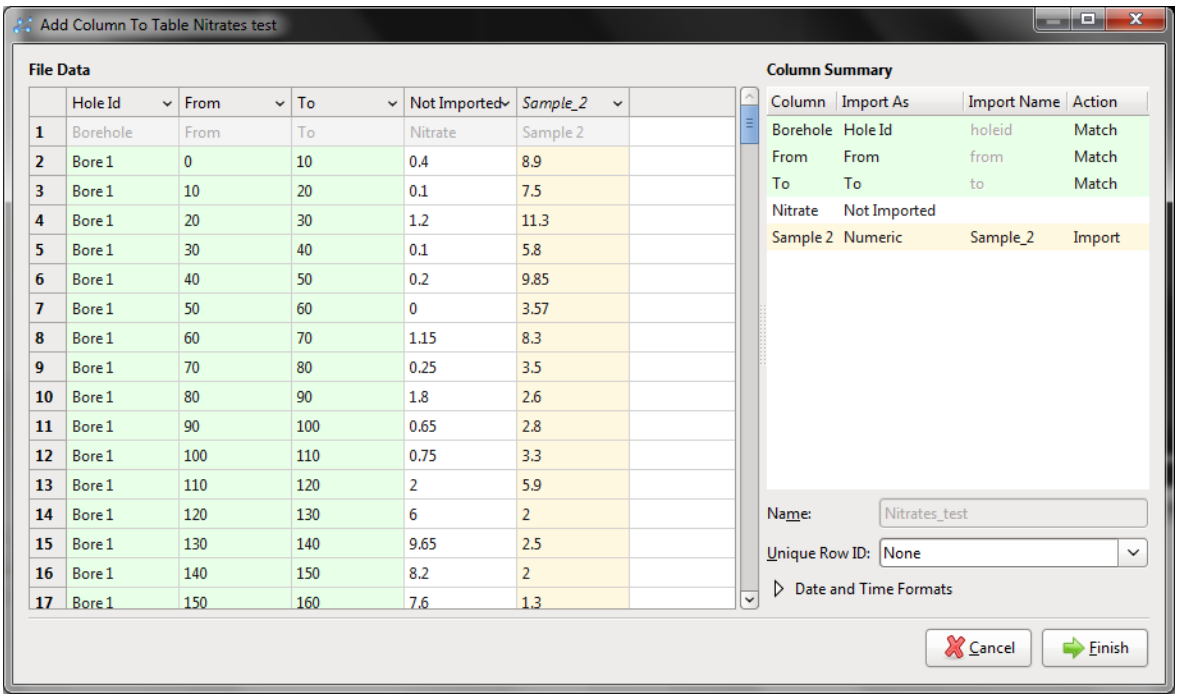
Columns of an interval table that have not been imported during the borehole data import can be added to the table at any time. To do this, right-click on the interval table in the project tree and select **Import Column**:



Importing a column is similar to importing interval tables themselves. Leapfrog will ask you to specify the file location, and then will display the data in the file in the **Add Column To Table** window:



Select as many columns to import as required and assign the appropriate data type. Before finishing, check the **Column Summary** to ensure that the correct information will be imported:



It is not necessary to re-import any columns that are already part of the project. The borehole ID and To and From data columns, however, will be used to check the validity of the new column against existing data.

Click **Finish** to import the new column.

Correcting Borehole Data Errors

Because borehole data often contains errors that prevent the computation of an accurate model, Leapfrog also has tools that help you to correct errors in the data, which can then be exported. In addition, when errors will be corrected in the primary data source, Leapfrog can export a list of errors.

Until errors are corrected, the rows that contain those errors are excluded for all processing that uses the table, including viewing in the scene. For example, if the ID for a collar position contains an invalid coordinate, that collar will not be displayed in the scene as its location is not known. Likewise, any processing will ignore data associated with this borehole.

When conflicts or errors are detected in borehole data, Leapfrog marks the table containing error with a red X.

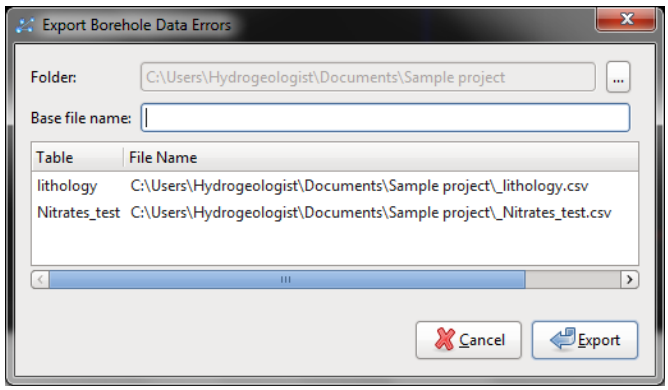
Although errors in borehole data can be fixed in Leapfrog, fixing errors sometimes need to be carried out on the primary data source. If this is the case, you can export a list of all the errors in the borehole data.

The **Export Errors** option only exports errors currently present in the database. Once errors have been fixed in Leapfrog, they cannot be exported, as they no longer exist. For this reason, it is recommended that if you are going to fix a large number of errors in Leapfrog you first export a list of those errors.

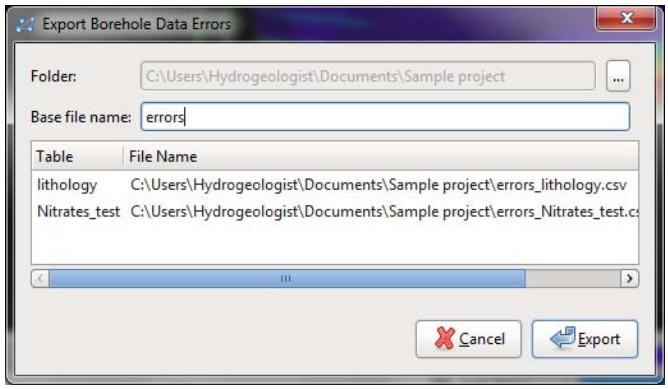
All the errors in borehole data tables can be exported at once or the errors in a single table can be exported. See:

Exporting All Borehole Data Errors

To export all errors in the borehole data, right-click on the **Borehole Data** object and select **Export Errors**. The **Export Borehole Data Errors** window will appear:



The list shows the files that will be created, one for each table that contains errors. Choose a folder where the files will be saved, then enter a **Base file name**, which will be added to the front of each file name. For example, entering "errors" for the **Base file name** results in two files called `errors_lithology.csv` and `errors_Nitrates_test.csv`:



Click **Export** to export the error files.

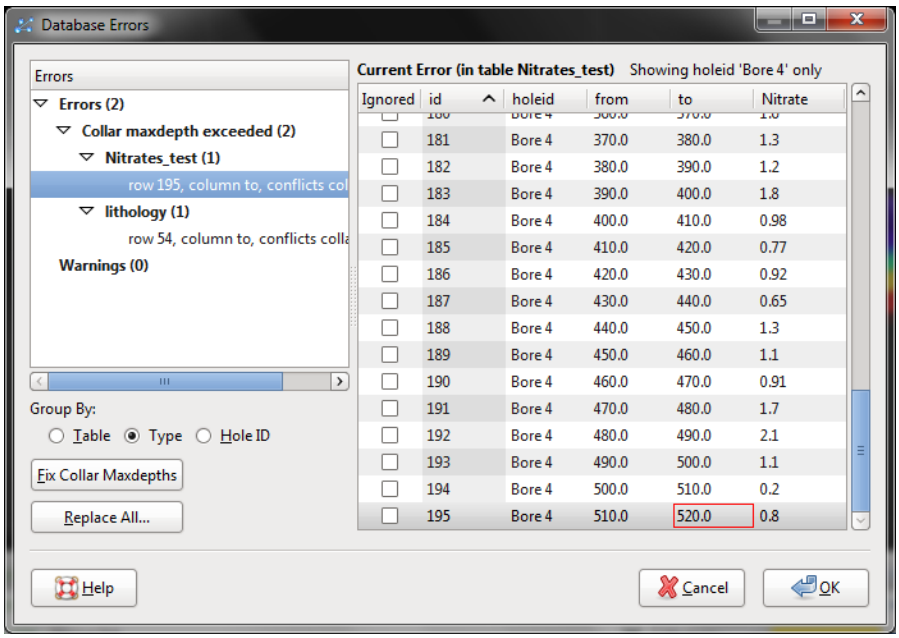
Exporting Errors from a Single Borehole Data Table

To export errors from a single table, right-click on that table and select **Export Errors**. Choose a location where the file will be saved and click **Save**.

Correcting Errors in Leapfrog

If you require a list of errors, select the **Export Errors** option before fixing them. Once errors have been fixed in Leapfrog, they cannot be exported as they no longer exist.

To correct errors in borehole data, right-click on the affected table and select **Fix Errors**. The **Database Errors** window will appear. Expand the list to view the details of the selected error:



Errors are automatically grouped by **Type** when the **Database Errors** window is opened. If there are many errors, you may find it useful to view them grouped by **Table** or **HoleID**.

The red box shows the location of the error, which occurs because the value conflicts with the maximum depth value in the collar file. The data displayed is the tutorial data, which is intended to be virtually free of error, but illustrates how error conditions appear in the **Database Errors** window.

You can change the size of the **Database Errors** window and the columns in the table to view more information easily. When you have finished fixing errors, click the **OK** button. Leapfrog will then update the database.

There are many possible sources of errors. Some useful techniques for fixing errors are described below.

Checking for Missing Files

Sometimes collar and interval table data is spread across several files and not all files have been imported. When you have many errors in freshly imported data, make sure that all necessary files have been imported.

Ignoring Errors

Ticking the **Ignored** box for a row results in that row being omitted from all processing, as though it has been deleted.

Modifying Data

You can double click on any cell to edit its content, except for the id column. If the cell you wish to edit is already selected, press the space bar to start editing.

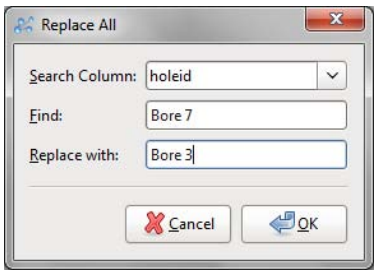
Changes made to data in this way are immediately saved to the database.

Fixing Collar Maxdepths

If there is a borehole segment falls beyond the maxdepth defined in the collar table, a Collar max-depth exceeded error will be reported. To fix this error, click on the **Fix Collar Maxdepths** button. This results in the collar table being updated with a maxdepth value that reflects the data in the interval tables.

Replacing All

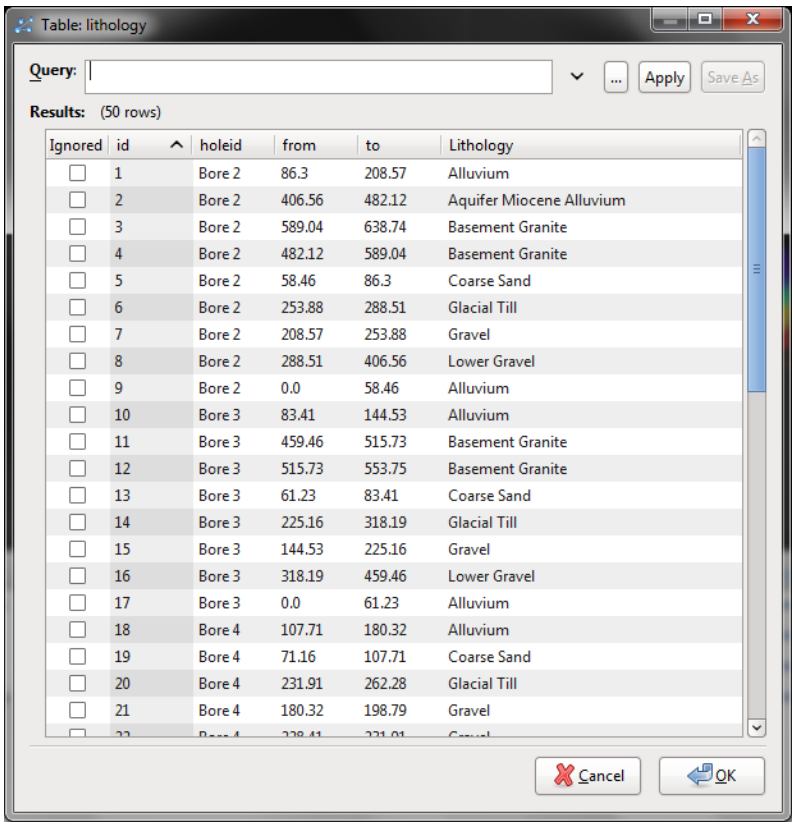
If you know that a certain data value is incorrect and this value occurs multiple times in the data that is currently displayed, click the **Replace All** button. The **Replace All** window will appear:



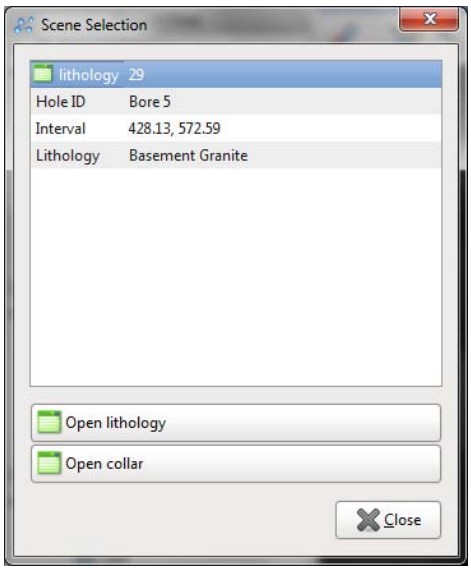
Enter the column to search in, then the **Find** and **Replace with** values. Click **OK** to make the change.

Displaying Borehole Data Tables

You can open borehole data tables by double-clicking on the table icon in the project tree. The table will be displayed:



You can also open a data table by clicking on a segment displayed in the scene. For example, clicking on a borehole segment in the scene will display the **Scene Selection** window and detailed information about the segment clicked on:

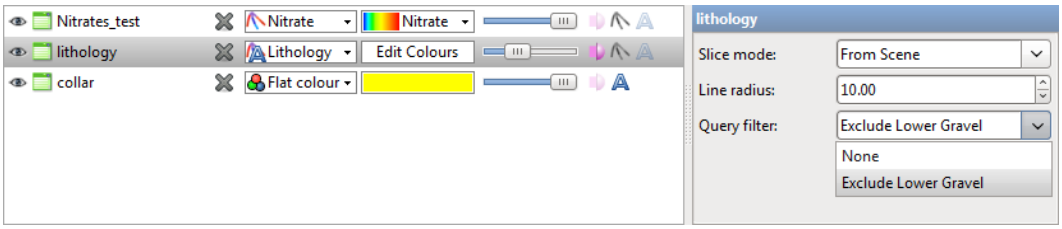


You can open the relevant lithology and collar tables by clicking on the **Open lithology** and **Open collar** buttons, with the selected segment highlighted. This is useful when, for example, displaying borehole data in the scene makes it apparent that the data for that borehole is incorrect.

Using Query Filters to Display Borehole Data

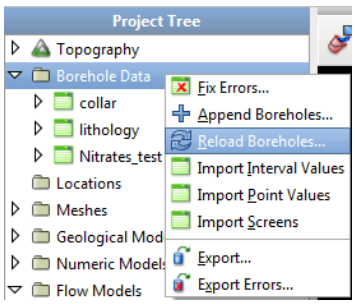
Query filters can be used to control what intervals or boreholes are displayed in the scene window.

To apply a query filter to the scene, select the object in the shape list, then select the filter from the **Query filter** dropdown list in the property frame:



Reloading Borehole Data

Reloading data is necessary when the imported data is modified externally. All borehole data can be reloaded by right-clicking on the **Borehole Data** object and selecting **Reload Data**:



The process is similar to that for importing borehole data.

If you only need to reload a single table, right-click on that table and select **Reload Data**.

When you reload data, Leapfrog retains the table structure and refreshes the data contained in the tables. This ensures that you do not need to reassign the data type for each column and select the columns to be imported.

Deleting Borehole Data

To delete a lithology or numeric data table, right-click on the table in the project tree and select **Delete**. You will be asked to confirm your choice.

When you right-click on the borehole table and select **Delete**, the resulting action will also remove any lithology and numeric data tables from the project. You will be asked to confirm your choice.

Exporting Borehole Data

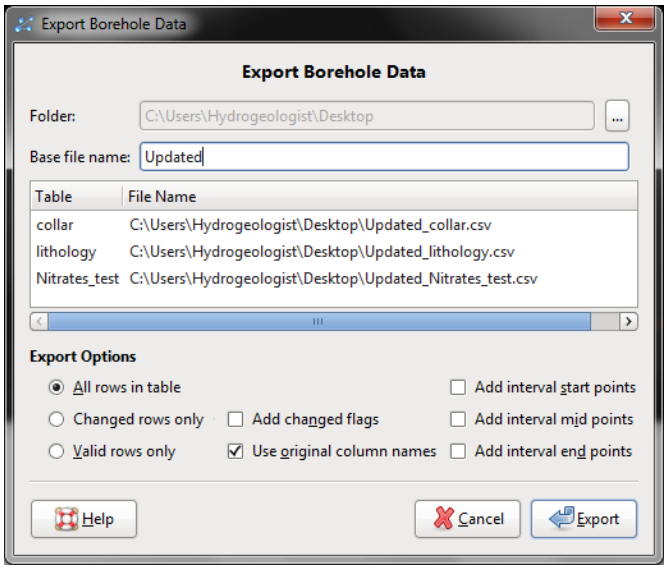
Changes made to the borehole data in Leapfrog only affect the Leapfrog database, not the original data files. You can export Leapfrog borehole data, which is useful if you wish to keep a copy of borehole data outside the project.

You can:

- Export all data. See Exporting All Borehole Data.
- Export only a single data table. See Exporting a Single Borehole Data Table.

Exporting All Borehole Data

To export all borehole data, right-click on the **Borehole Data** object and select **Export**. The **Export Borehole Data** window will appear:

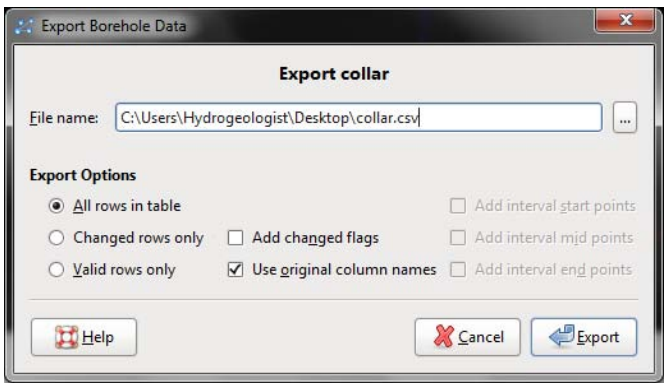


The list shows the files that will be created, one for each data table. Choose a folder where the files will be saved, then enter a **Base file name**, which will be added to the front of each file name. For example, entering "Updated" for the **Base file name** results in files called `Updated _collar.csv`, `Updated _lithology.csv` and `Updated_Nitrates_test.csv`.

Choose the **Folder** in which to save the files. Select the **Export Options** required, then click **Export**.

Exporting a Single Borehole Data Table

To export a single borehole data table, right-click on that table and select **Export**.



Choose a **File name** and location. Select the **Export Options** required, then click **Export**.

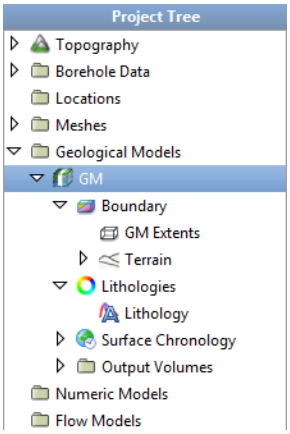
The Geological Model Object

This topic describes the **Geological Model** object and the objects that are added to it in the process of building a geological model.

When a geological model is been created, the object representing that model contains four other objects:

- **Boundary**. This object defines the limits of the geological model.
- **Lithologies**. This object describes all the lithological units to be modelled in the geological model and the colours that are used to display them on the screen. It is generated automatically from all the lithologies identified in the borehole data.
- **Surface Chronology**. This object is a collection of contact surfaces that are combined in a chronological order to produce the volumes of the geological model.
- **Output Volumes**. This object contains all the volumes generated in building the geological model.

All tools for building a geological model are accessed by right-clicking on the **Geological Model** object and the items in it:



Imported objects and objects created while building a model are also stored as part of the **Geological Model** object, and more complex functions can be carried out by right-clicking on individual objects.

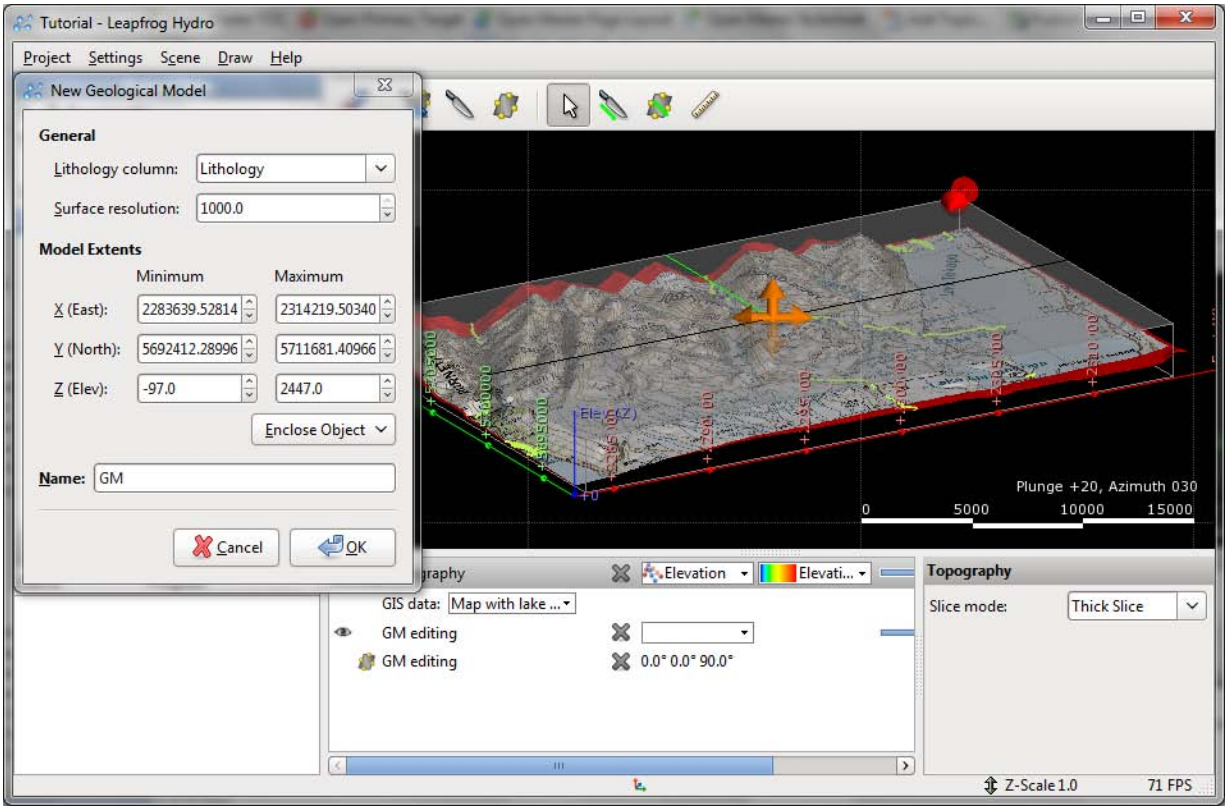
Tutorial 4: Building a Simple Geological Model describes how to use these objects to build a geological model from a small set of data.

New Geological Model Window

In the **New Geological Model** window, you can define the basic properties of a geological model, including the data used to create the model, the surface resolution and the basic model extents.

When defining a new geological model, it is a good idea to add the topography to the scene so you have some idea of where to set the geological model extents.

To define a new geological model, right-click on the **Geological Model** object and select **New Geological Model**. The **New Geological Model** window will open, together with a set of controls in the scene that help in defining the model extents:



This window lets you set a basic rectangular set of model extents aligned with the south/north and east/west axes. There are three ways to do this:

- Enter the coordinates.
- Select **Enclose Object** and choose from the list of objects in the project. The model extents will be updated to the size of the selected object.
- Use the controls that appear in the scene. The orange handle adjusts the centre of the plane and the red handle adjust the size.

The **Enclose Object** list can be used to set the size of the rectangular extents to fit the data that will be used to generate the model.

The **Surface resolution** is also set in this window. A lower value will produce more detail, but calculations will take longer. The advantage of Leapfrog over many other geological modelling products is that the surface resolution can be easily changed.

When borehole data is used as the basis of a model, the **Lithology column** determines what column in the imported borehole data will be used. If more than one column of lithology data is available for creating models, be sure to choose the correct one at this point as it cannot be changed once the initial model has been created.

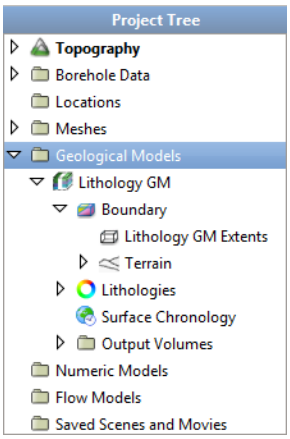
The **Name** you assign for the model should describe the purpose of the model. This **Name** will be used in naming the objects that will be added to the model.

Click **OK** to create the new model.

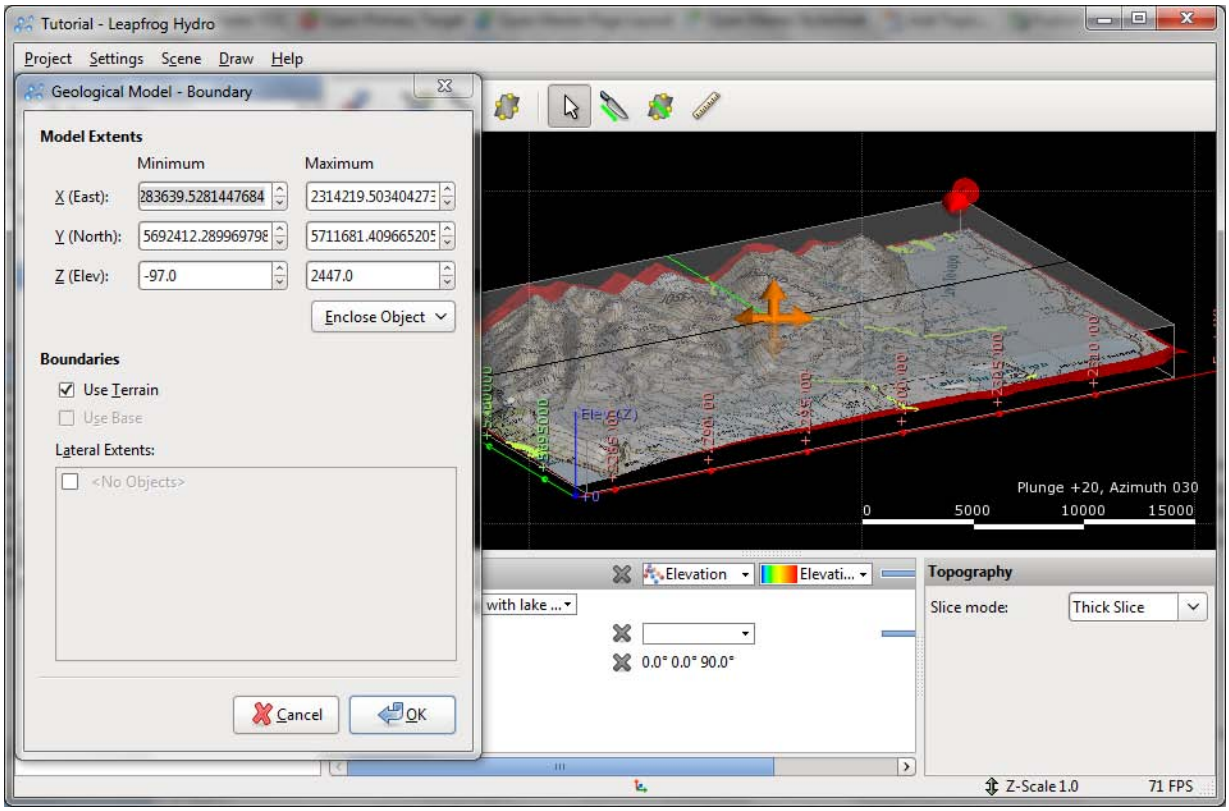
The Boundary Object

The **Boundary** object defines the limits of a geological model.

When a new geological model is first created, the only item under the **Boundary** object will be an object for the model's basic extents, which will be named according to the name you assigned for the model:



Double-click on the **Boundary** object to open the **Geological Model - Boundary** window:



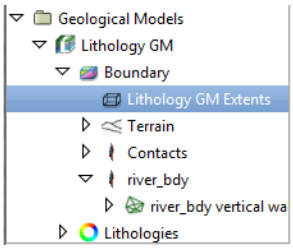
In this window, you can change the boundaries you set when you first created the model.

You can also:

- Select different objects to use as lateral extents. See Creating New Lateral Extents for more information.
- Use a custom base. See Creating a New Base for more information.

The Geological Model Extents Object

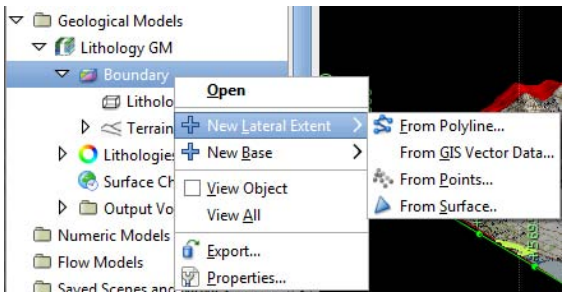
For each geological model created in a project, an object will appear under the **Boundary** object describing the model extents:



Double-clicking on this object accesses the same information accessed by double-clicking on the **Boundary** object itself. See The Boundary Object for more information.

Creating New Lateral Extents

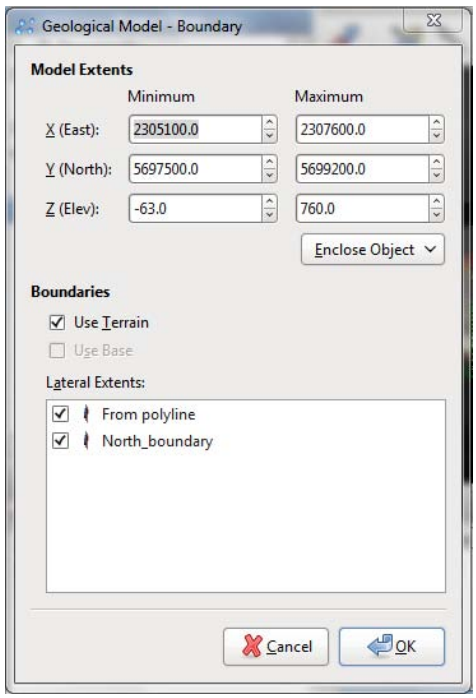
To create a new lateral extent, right-click on the **Boundary** object for the geological model you are working on:



There are four options:

- New Lateral Extent From Polyline
- New Lateral Extent From GIS Vector Data
- New Lateral Extent From Points
- New Lateral Extent From Surface

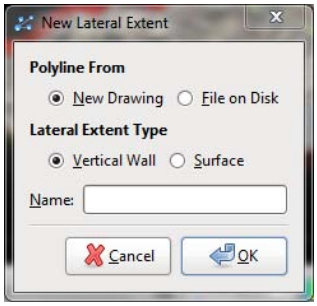
Once a new lateral extent has been created, you can add it to the model by double-clicking on the **Boundary** object. The lateral extents available will be shown in the **Lateral Extents** list:



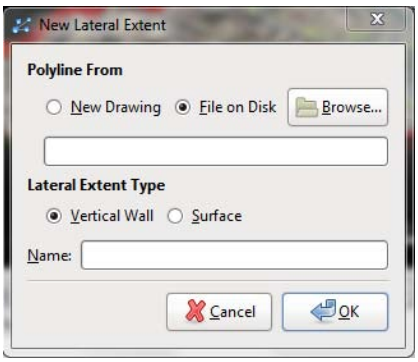
Tick the box next to the lateral extent required.

New Lateral Extent From Polyline

To create a new lateral extent from a polyline you draw or import from a file, right-click on the **Boundary** object for the geological model you are working on and select **New Lateral Extent > From Polyline**. The **New Lateral Extent** window will appear:



You can draw the polyline in the scene directly by selecting the **New Drawing** option. You can also import the polyline from a file by selecting the **File on Disk** option. You will then be prompted for the file location:



Leapfrog Hydro supports the following polyline formats for lateral extents:

- Datamine Polylines (*.asc)
- Surpac String Polylines (*.str)
- Gemcom Polylines (*.asc)
- Micromine Polylines (*.asc, *.str)
- MineSight Polylines (*.srg)
- Gocad Polylines (*.pl, *.ts)
- Drawing Interchange Polylines (*.dxf)
- Leapfrog Hydro 3D Polylines (*.csv, *.txt)

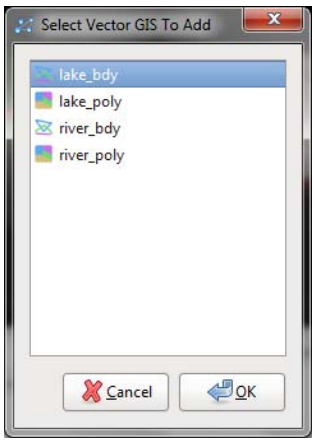
Select whether you wish to create a new **Vertical Wall** or **Surface**.

If you have chosen to create a **New Drawing**, the drawing controls will appear in the scene and you can begin drawing. If you have chosen to import a file, the new polyline will be imported and added to the geological model. In both cases, you can add the new lateral extent to the scene by double-clicking on the **Boundary** object and selecting the lateral extents required from the **Lateral Extents** list, as described in Creating New Lateral Extents.

New Lateral Extent From GIS Vector Data

To create a new lateral extent from GIS vector data, that data must first be imported into the project.

Once the data you wish to use has been imported, right-click on the **Boundary** object for the geological model you are working on and select **New Lateral Extent > From GIS Vector Data**. The **Select Vector GIS Data to Add** window will be displayed, showing vector data available in the project:



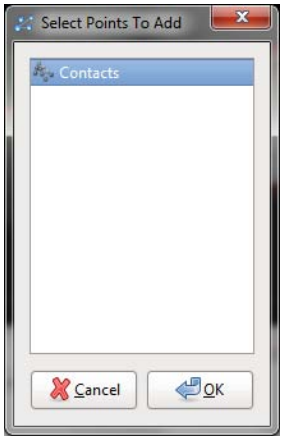
Select the information you wish to use and click **OK**.

The selected vector data object will appear in the project tree under the **Boundary** object. Add the new lateral extent to the scene by double-clicking on the **Boundary** object as described in Creating New Lateral Extents.

New Lateral Extent From Points

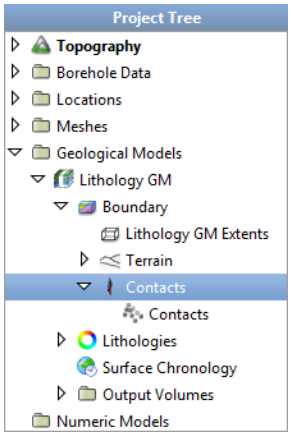
To create a new lateral extent from points, that data must first be imported into the project.

Once the points data you wish to use has been imported, right-click on the **Boundary** object for the geological model you are working on and select **New Lateral Extent > From Points**. The **Select Points to Add** window will be displayed, showing points data available in the project:



Select the information you wish to use and click **OK**.

The selected points data object will appear in the project tree under the **Boundary** object:



Add the new lateral extent to the scene by double-clicking on the **Boundary** object as described in Creating New Lateral Extents.

New Lateral Extent From Surface

To create a new lateral extent from a surface, the surface must first be imported into the project.

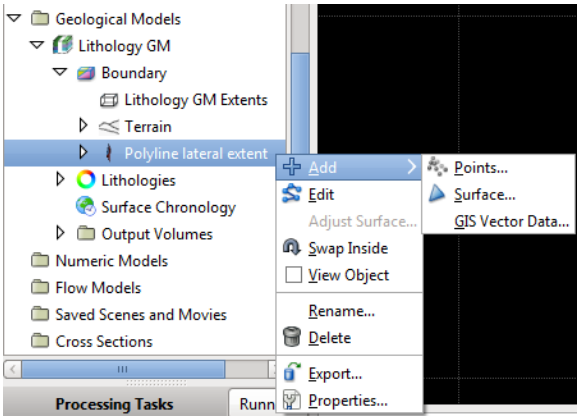
Once the surface you wish to use has been imported, right-click on the **Boundary** object and select **New Lateral Extent > From Surface**. The **Select Mesh to Add** window will be displayed, showing the meshes available in the project:

Select the surface you wish to use and click **OK**.

The surface will appear in the project tree under the **Boundary** object. Add the new lateral extent to the scene by double-clicking on the **Boundary** object as described in Creating New Lateral Extents.

Modifying Lateral Extents

When you have created a lateral extent from a polyline, GIS vector data, points data or surface meshes, you can modify these lateral extents in several ways, which are shown below:



You can add points data, meshes and GIS vector data to any lateral extent. Select the required option and Leapfrog Hydro will display a list of the data that can be used for the lateral extent.

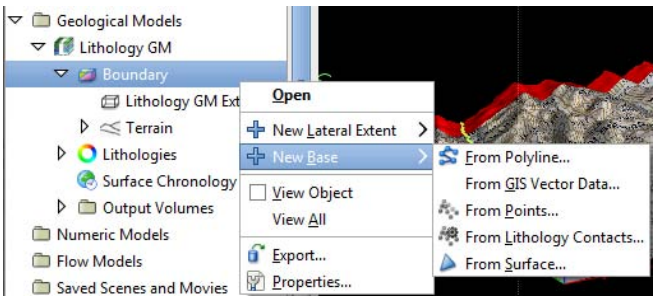
You can also:

- Edit the lateral extent
- Adjust the surface of GIS data and points data lateral extents
- Swap the inside of the lateral extent

You can also export any lateral extent as a mesh. See Exporting a Lateral Extent as a Mesh.

Creating a New Base

To create a new base, right-click on the **Boundary** object for the geological model you are working on:

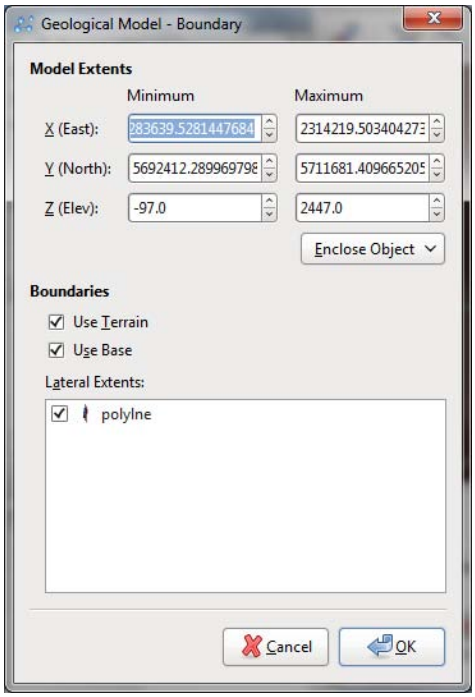


There are five options:

- New Base From Polyline
- New Base From GIS Vector Data
- New Base From Points
- New Base From Lithology Contacts
- New Base From Surface

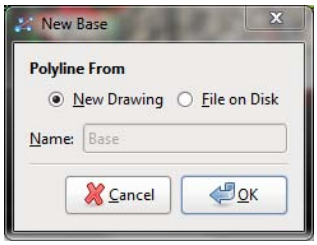
Each geological model can have only one base defined, so if you wish to define a new base, you must first delete the existing base from the model. Do this by right-clicking on the **Base** object and selecting **Delete**.

You can also choose not to use the base you have defined. Do this by double-clicking on the **Boundary** object and unticking the **Use Base** option:

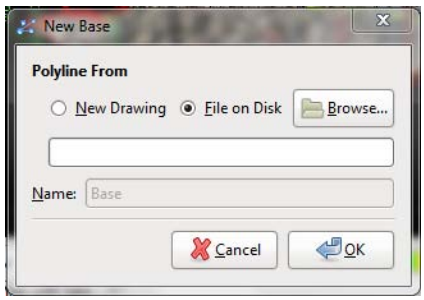


New Base From Polyline

To create a new base from a polyline you draw or import from a file, right-click on the **Boundary** object for the geological model you are working on and select **New Base > From Polyline**. The **New Base** window will appear:



You can draw the polyline in the scene directly by selecting the **New Drawing** option. You can also import the polyline from a file by selecting the **File on Disk** option. You will then be prompted for the file location:



Leapfrog Hydro supports the following polyline formats for the base:

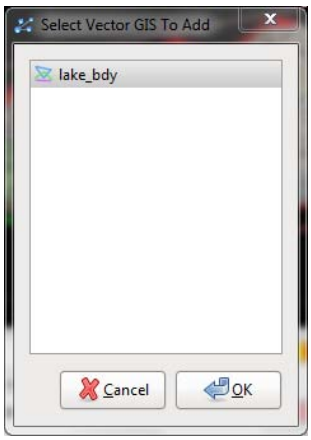
- Datamine Polylines (*.asc)
- Surpac String Polylines (*.str)
- Gemcom Polylines (*.asc)
- Micromine Polylines (*.asc, *.str)
- MineSight Polylines (*.srg)
- Gocad Polylines (*.pl, *.ts)
- Drawing Interchange Polylines (*.dxf)
- Leapfrog Hydro 3D Polylines (*.csv, *.txt)

If you have chosen to create a **New Drawing**, the drawing controls will appear in the scene and you can begin drawing. If you have chosen to import a file, the new polyline will be imported and added to the geological model. In both cases, the base will automatically be added to the model.

New Base From GIS Vector Data

To create a new base from GIS vector data, that data must first be imported into the project.

Once the data you wish to use has been imported, right-click on the **Boundary** object for the geological model you are working on and select **New Base > From GIS Vector Data** The **Select Vector GIS Data to Add** window will be displayed, showing vector data in the project:



Select the information you wish to use and click **OK**.

The selected vector data object will appear in the project tree under the **Boundary** object. The new base will automatically be added to the model.

New Base From Points

To create a new base from points, that data must first be imported into the project.

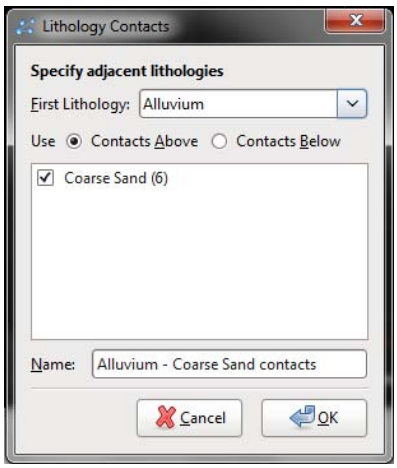
Once the points data you wish to use has been imported, right-click on the **Boundary** object for the geological model you are working on and select **New Base > From Points**. The **Select Points to Add** window will be displayed, showing points data available in the project:

Select the information you wish to use and click **OK**.

The selected points data object will appear in the project tree under the **Boundary** object. The new base will automatically be added to the model.

New Base From Lithology Contacts

To create a new base from lithology contacts, right-click on the **Boundary** object for the geological model you are working on and select **New Base > From Lithology Contacts**. The **Lithology Contacts** window will appear:



Choose the lithology you wish to use as the base and click **OK**.

Leapfrog Hydro will generate the new base, which will appear under the **Boundary** object. The new base will automatically be added to the model.

New Base From Surface

To create a new base from a surface, the surface must first be imported into the project.

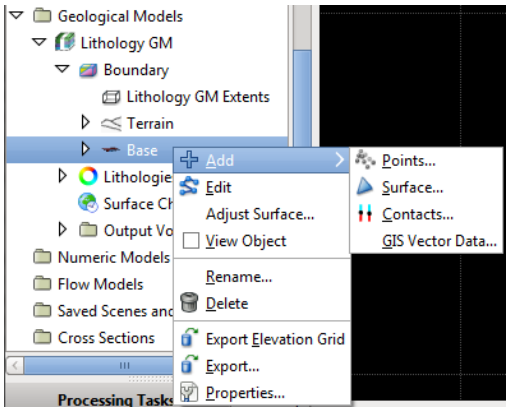
Once the surface you wish to use has been imported, right-click on the **Boundary** object and select **New Base > From Surface**. The **Select Mesh to Add** window will be displayed, showing the meshes available in the project:

Select the surface you wish to use and click **OK**.

The surface will appear in the project tree under the **Boundary** object. The new base will automatically be added to the model.

Modifying the Base

When you have created a new base from a polyline, GIS vector data, points data, surface meshes or lithology contacts, you can modify the base in several ways, which are shown below:



You can add points data, meshes, lithology contacts and GIS vector data to the base. Select the required option and Leapfrog Hydro will display a list of the data that can be used for the base.

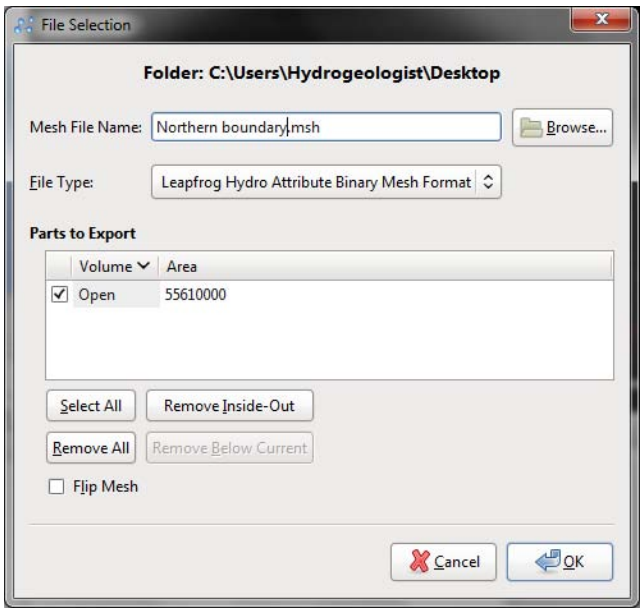
You can edit the base and adjust the surface.

You can also:

- Export the base as a mesh. See Exporting a Lateral Extent or the Base as a Mesh.
- Export the base as an elevation grid. See Exporting the Base as an Elevation Grid.

Exporting a Lateral Extent or the Base as a Mesh

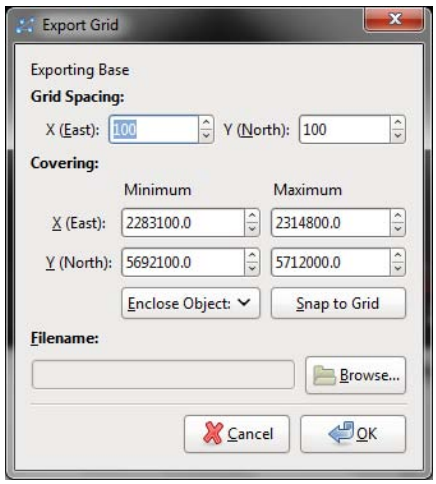
To export a lateral extent or the base as a mesh, right-click on the object in the project tree and select **Export**. The **File Selection** window will appear:



Enter a name for the new mesh file and click **Browse** to choose a location. You can select from a number **File Type** options and choose which parts of the lateral extent to export. Click **OK** to save the mesh.

Exporting the Base as an Elevation Grid

To export the base as an elevation grid, right-click on the **Base** object in the project tree and select **Export Elevation Grid**. The **Export Grid** window will appear:



Enter a name for the new elevation grid file and click **Browse** to choose a location. Choose the **Grid Spacing** and **Covering** properties, then click **OK** to save the grid.

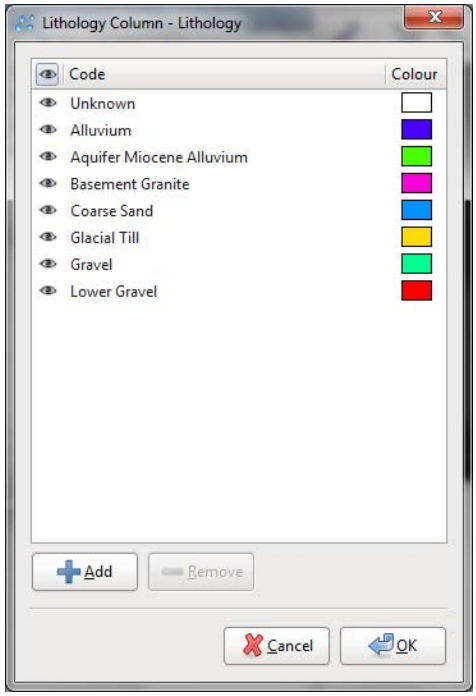
The Terrain Object (Geological Model)

When a new geological model is created, the **Terrain** object that is part of the **Topography** object is used to define the elevation of the ground surface. See The Terrain Object for more information on the properties of the **Topography > Terrain** object.

The Lithologies Object

The **Lithologies** object describes all the different units to be modelled in the geological model and the colours that are used to display them on the screen. When a new geological model is created, the information in the **Lithologies** object is generated automatically from all the lithologies defined in the borehole data.

To view the properties of the **Lithologies** object, double-click on it. The **Lithology Column** window will be displayed:



You can:

- Change the colour used to display a lithology. To do so, click on the colour chip and choose a new colour.
- Add a lithology. This is useful in cases where it is known that a lithology has not yet been sampled. To add a lithology, click on the **Add** button, enter a name for the lithology and choose a colour.
- Remove a lithology. To do so, click on the lithology you wish to remove, then click the **Remove** button.

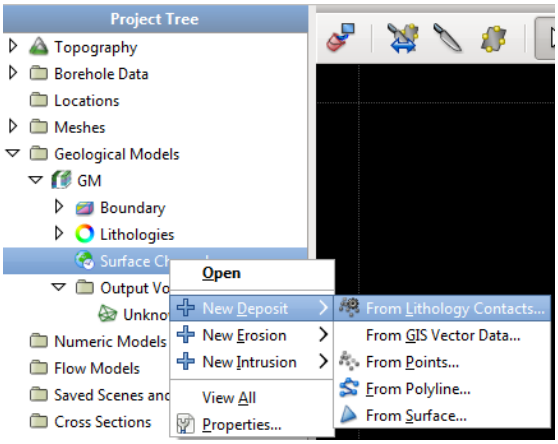
The Surface Chronology Object

The **Surface Chronology** object represents the collection of contact surfaces that are combined in chronological order to produce the volumes of the geological model.

All contact surface have a younger and older side, each with a separate lithological classification. The younging direction determines how contact surfaces cut against each other, so it is important to check that the younging direction is correct. Leapfrog Hydro will by default put the younger side up, but you can change this if, for example, you know the geology is overturned.

Once the volumes are produced they are labelled according to the adjacent contact surfaces. If Leapfrog Hydro is unable to determine the lithology of a volume it will be labelled as "Unknown".

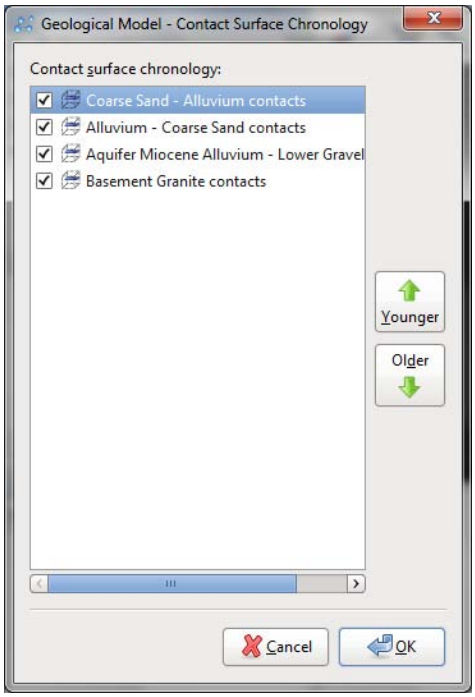
To create contact surfaces, right-click on the **Surface Chronology** object and select from the options:



Three types of contact surfaces can be created. See:

- Creating New Depositional Contacts
- Creating New Erosional Contacts
- Creating New Intrusional Contacts

Once a contact surface has been generated, you can add it to the surface chronology by double-clicking on the **Surface Chronology** object. The **Contact Surface Chronology** window will be displayed:



The **Contact surface chronology** list displays all the contact surfaces that have been generated. To add a surface to the surface chronology, tick the box for it. Use the **Younger** and **Older** buttons to change the order in which surfaces appear in the chronology.

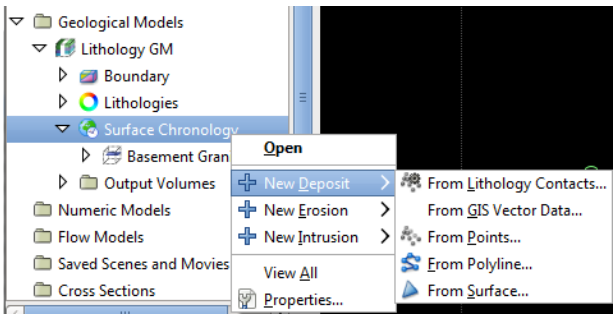
When you click **OK**, Leapfrog Hydro will use the contact surface information to generate the model volumes, which will then appear under the **Output Volumes** object. To view the volumes, you can either:

- Drag the volumes into the scene, one by one.
- Drag the geological model object into the scene.

Creating New Depositional Contacts

A deposit is a type of contact surface that defines a depositional type process. As in a real depositional process the new deposit will appear conformably on top of the underlying older volume(s). This effectively means deposits will not occur in regions defined by older deposits.

To create a new depositional contact surface, right-click on the **Surface Chronology** object and select one of the **New Deposit** options:

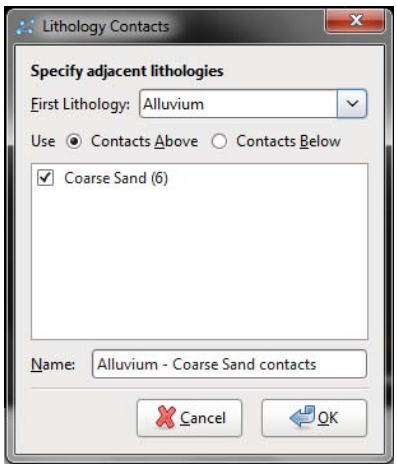


See:

- New Deposit From Lithology Contacts
- New Deposit From GIS Vector Data
- New Deposit From Points
- New Deposit From Polyline
- New Deposit From Surface

New Deposit From Lithology Contacts

To create a new depositional contact surface from lithology contacts, right-click on the **Surface Chronology** object and select **New Deposit > From Lithology Contacts**. The **Lithology Contacts** window will be displayed:



The **Contacts Above** and **Contacts Below** lists display the number of contacts that exist in the borehole data between the **First Lithology** and those it comes into contact with.

Select the **First Lithology** from the list, then choose whether you wish to define a surface for the **Contacts Above** or **Contacts Below** the selected lithology.

The **Name** for the new contact surface is automatically generated from the **First Lithology** and the **Contacts Above** or **Contacts Below** options chosen. You can also enter a different name, if you wish.

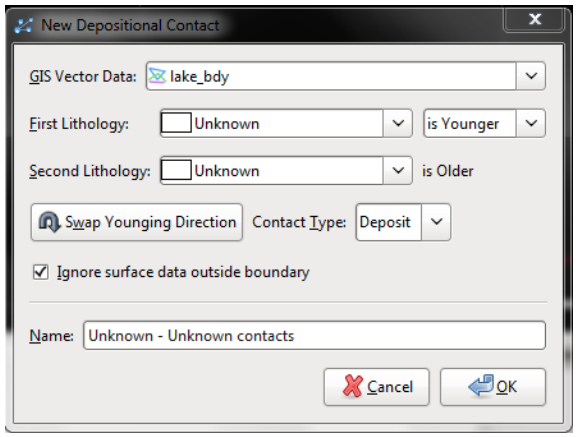
To create the surface, click **OK**.

Once the surface is generated, it will be added to the **Output Volumes** object. To add the volume to the geological model, you must first add it to the **Surface Chronology** object. See The Surface Chronology Object.

See Tutorial 4: Building a Simple Geological Model for an example of how to generate contact surfaces from lithology contacts.

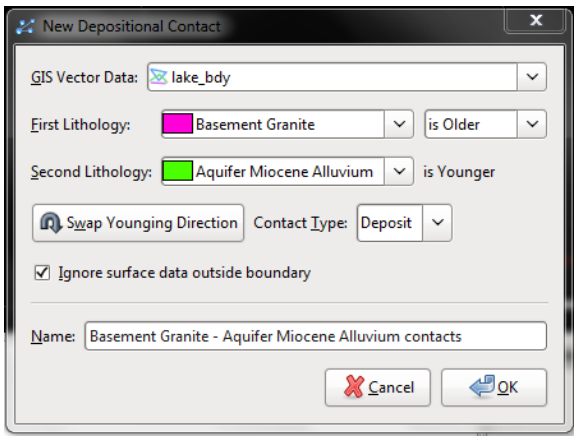
New Deposit From GIS Vector Data

To create a new depositional contact surface from GIS vector data, right-click on the **Surface Chronology** object and select **New Deposit > From GIS Vector Data**. The **New Depositional Contact** window will be displayed:



The **GIS Vector Data** drop-down list displays all vector data in the project that can be used to create a new contact surface. If the data you wish to use does not appear in the list, ensure that it has been imported into the project.

Select the **First Lithology** and **Second Lithology** and make sure the younging direction is correct.



Because the process for generating a depositional contact surface is similar to that for generating an erosional surface, you can change the **Contact Type**.

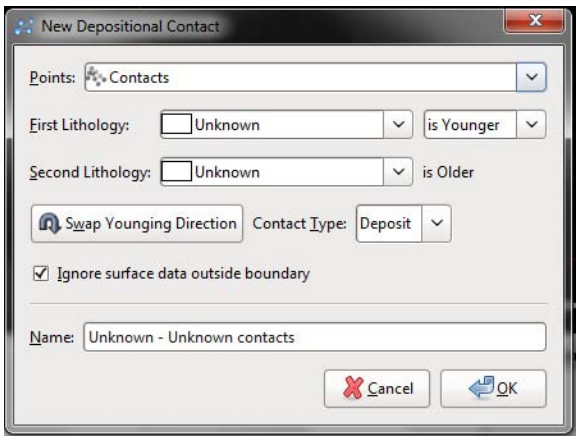
The **Name** for the new contact surface is automatically generated from the **First Lithology** and the **Second Lithology** options chosen. You can also enter a different name, if you wish.

Click **OK** to generate the new contact surface.

Once the surface is generated, it will be added to the **Output Volumes** object. To add the volume to the geological model, you must first add it to the **Surface Chronology** object. See The Surface Chronology Object.

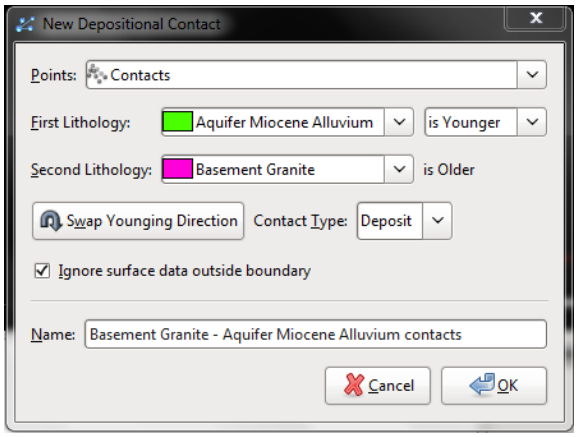
New Deposit From Points

To create a new depositional contact surface from points data, right-click on the **Surface Chronology** object and select **New Deposit > From Points**. The **New Depositional Contact** window will be displayed:



The **Points** drop-down list displays all points data in the project that can be used to create a new contact surface. If the data you wish to use does not appear in the list, ensure that it has been imported into the project.

Select the **First Lithology** and **Second Lithology** and make sure the younging direction is correct.



Because the process for generating a depositional contact surface is similar to that for generating an erosional surface, you can change the **Contact Type**.

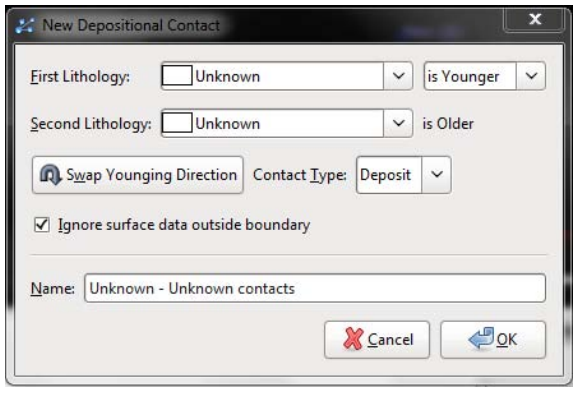
The **Name** for the new contact surface is automatically generated from the **First Lithology** and the **Second Lithology** options chosen. You can also enter a different name, if you wish.

Click **OK** to generate the new contact surface.

Once the surface is generated, it will be added to the **Output Volumes** object. To add the volume to the geological model, you must first add it to the **Surface Chronology** object. See The Surface Chronology Object.

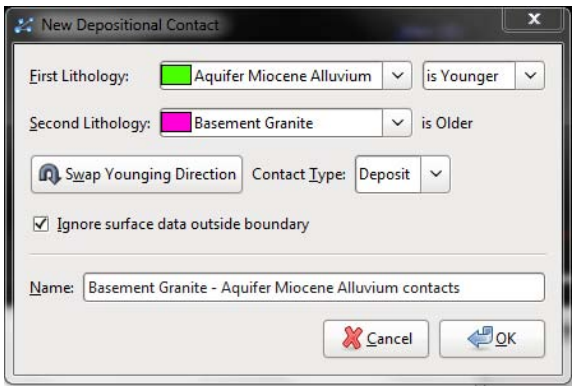
New Deposit From Polyline

To create a new depositional contact surface from a polyline, right-click on the **Surface Chronology** object and select **New Deposit > From Polyline**. The **New Depositional Contact** window will be displayed:



The **Points** drop-down list displays all points data in the project that can be used to create a new contact surface. If the data you wish to use does not appear in the list, ensure that it has been imported into the project.

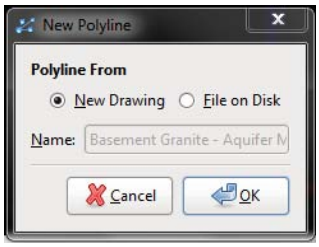
Select the **First Lithology** and **Second Lithology** and make sure the younging direction is correct.



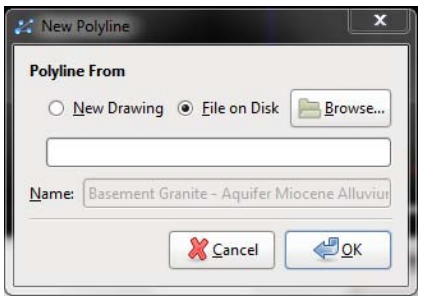
Because the process for generating a depositional contact surface is similar to that for generating an erosional surface, you can change the **Contact Type**.

The **Name** for the new contact surface is automatically generated from the **First Lithology** and the **Second Lithology** options chosen. You can also enter a different name, if you wish.

Click **OK**. The **New Polyline** window will be displayed:



You can draw the polyline in the scene directly by selecting the **New Drawing** option. You can also import the polyline from a file by selecting the **File on Disk** option. You will then be prompted for the file location:



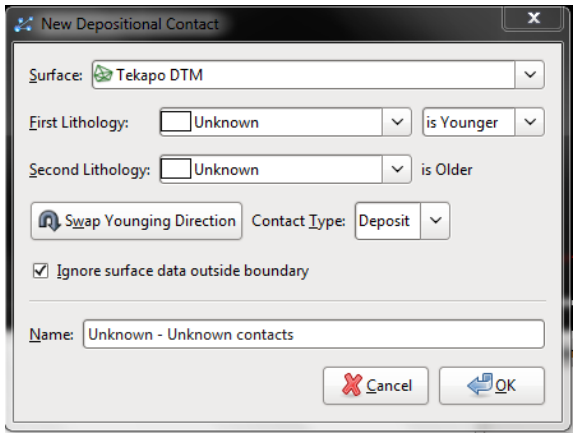
If you have chosen to create a **New Drawing**, the drawing controls will appear in the scene and you can begin drawing. If you have chosen to import a file, the new polyline will be imported and the contact surface generated.

Once the surface is generated, it will be added to the **Output Volumes** object. To add the volume to the geological model, you must first add it to the **Surface Chronology** object. See The Surface Chronology Object.

New Deposit From Surface

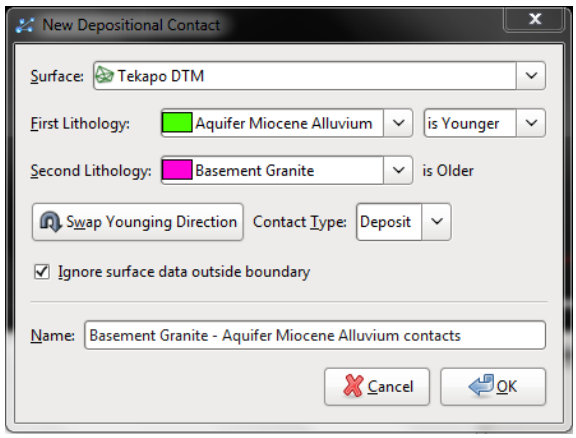
To create a new depositional contact surface from a surface, right-click on the **Surface Chronology** object and select **New Deposit > From Surface**. The **New Depositional Contact** window will be

displayed:



The **Surface** drop-down list displays all the surfaces in the project that can be used to create a new contact surface. If the surface you wish to use does not appear in the list, ensure that it has been imported into the project.

Select the **First Lithology** and **Second Lithology** and make sure the younging direction is correct.



Because the process for generating a depositional contact surface is similar to that for generating an erosional surface, you can change the **Contact Type**.

The **Name** for the new contact surface is automatically generated from the **First Lithology** and the **Second Lithology** options chosen. You can also enter a different name, if you wish.

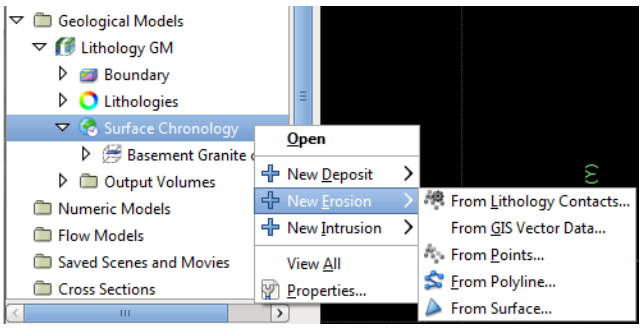
Click **OK** to generate the new contact surface.

Once the surface is generated, it will be added to the **Output Volumes** object. To add the volume to the geological model, you must first add it to the **Surface Chronology** object. See The Surface Chronology Object.

Creating New Erosional Contacts

An erosion is a type of contact surface that removes the existing lithologies that the erosion cuts through. In an erosion the material on the younger side of the contact surface is removed and the lithologic unit deposited on the older side.

To create a new erosional contact surface, right-click on the **Surface Chronology** object and select one of the **New Erosion** options:



These options are similar to those for creating depositional contacts. See:

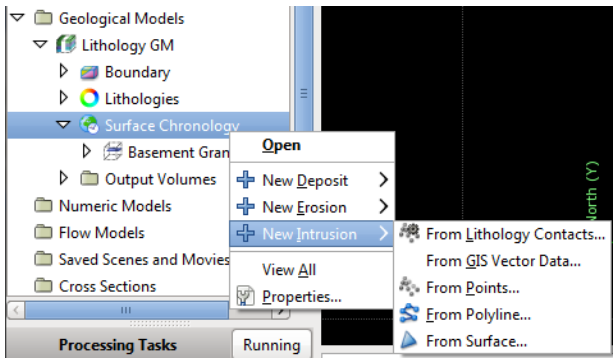
- New Deposit From Lithology Contacts
- New Deposit From GIS Vector Data
- New Deposit From Points
- New Deposit From Polyline
- New Deposit From Surface

Creating New Intrusional Contacts

An intrusion is a type of contact surface that defines the removal of existing lithologies and their replacement by an intrusive rock.

An intrusion removes all the existing material on the younger side of the contact surface. An intrusion should always have the younger side of its surface labelled with the intruded material. Typically the older side will be labelled unknown as an intrusion will usually displace multiple older lithologies.

To create a new intrustional contact surface, right-click on the **Surface Chronology** object and select one of the **New Intrusion** options:



To create a new intrustional contact surface from lithology contacts, see New Intrusion From Lithology Contacts.

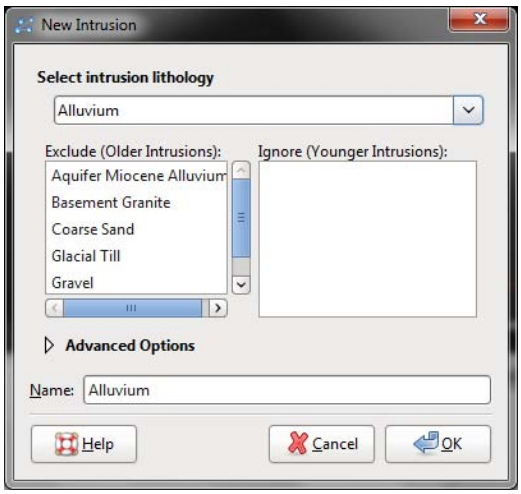
The other options are similar to those for creating depositional contacts. See:

- New Deposit From GIS Vector Data
- New Deposit From Points

- New Deposit From Polyline
- New Deposit From Surface

New Intrusion From Lithology Contacts

To create a new intrusional contact surface from lithology contacts, right-click on the **Surface Chronology** object and select **New Intrusion > From Lithology Contacts**. The **Lithology Contacts** window will be displayed:



Select the lithology that will remove existing materials from the **Select intrusion lithology** drop-down box, then select the options required. Click **OK** to generate the new contact surface.

Once the surface is generated, it will be added to the **Output Volumes** object. To add the volume to the geological model, you must first add it to the **Surface Chronology** object. See The Surface Chronology Object.

Output Volumes Object

The **Output Volumes** contains all the volumes generated in building a geological model.

When a geological model is first created, the only volume that appears under the **Output Volumes** object will be a volume called "Unknown". Once contact surfaces have been generated and added to the **Surface Chronology** object, new volumes will be generated and added to the **Output Volumes** object.

To view the generated volumes in the scene, you can either:

- Drag the volumes into the scene, one by one.
- Drag the geological model object into the scene.

The Cross Sections Object

This topic describes the **Cross Sections** object and the tools available for creating and modifying cross sections.

A cross section is typically a vertical plane with an image or geologic cross section applied to it. A section is positioned in a scene using a plane or by georeferencing a section image. Geological models can be evaluated and images can be loaded on a section.

Options available for the **Cross Sections** object are:

- Creating a New Cross Section from an Image
- Creating a New Blank Cross Section
- Creating a New Cross Section from the Slicer

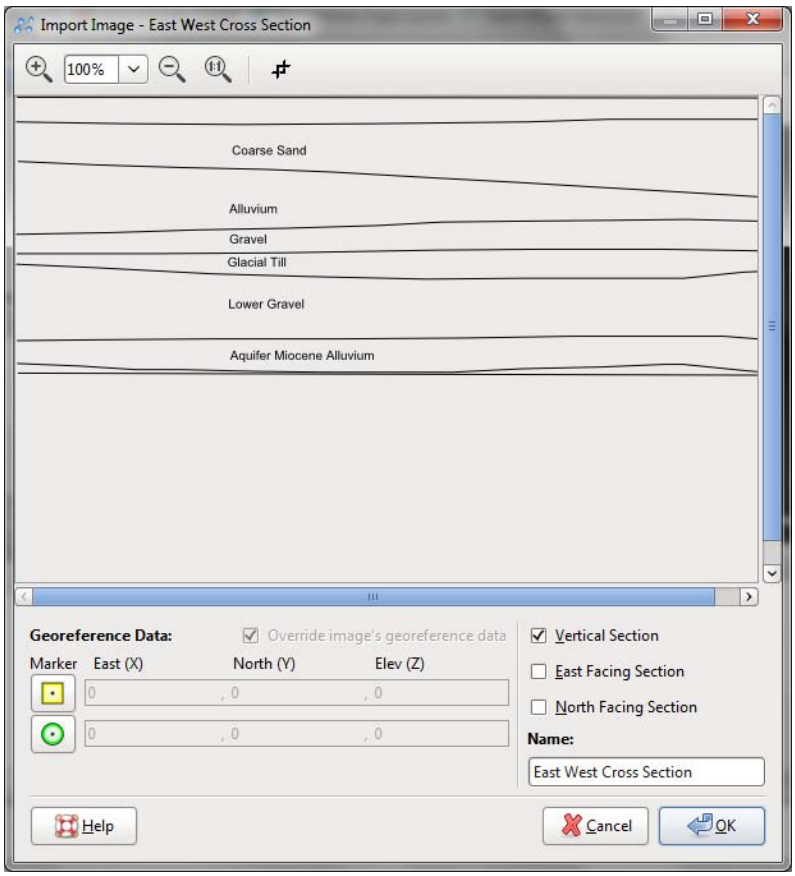
Once a cross section has been created, it can be applied to a specific geological model, then exported as a lithology cross section. For more information, see:

- Editing a Cross Section
- Evaluating a Geological Model
- Exporting a Lithology Cross Section

Creating a New Cross Section from an Image

To create a new cross section from an image, right-click on the **Cross Sections** object and select **New Cross Section From Image**. You will be prompted to select an image.

Next, the **Import Image** window will be displayed:

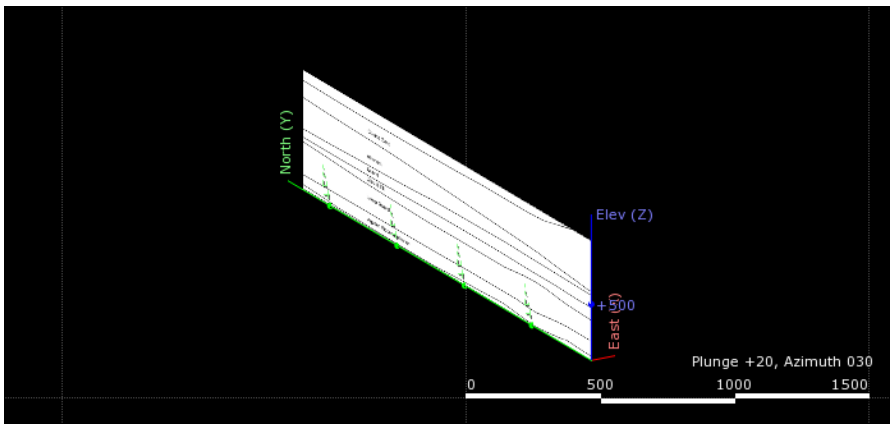


If the image contains georeference data, it will automatically be added to the map and displayed. You can edit the imported georeference data by ticking the **Override image's georeference data** box, then editing the information.

If the image does not contain georeference information, you will need to add it manually by adding reference markers. For an image marked as a **Vertical Section**, only two reference points are required.

Add the markers to the image and enter the coordinates. Note that selecting either **East Facing Section** or **North Facing Section** constrain the points. For example, for an east-facing image, you will only need to enter the **East (X)** coordinates once.

Once you have set the coordinates for each marker, click **OK**. The new cross section will be created and added to the **Cross Sections** object. Display it by dragging it into the scene or by right-clicking on the cross section and selecting **View Object**. The cross section will appear in the scene:

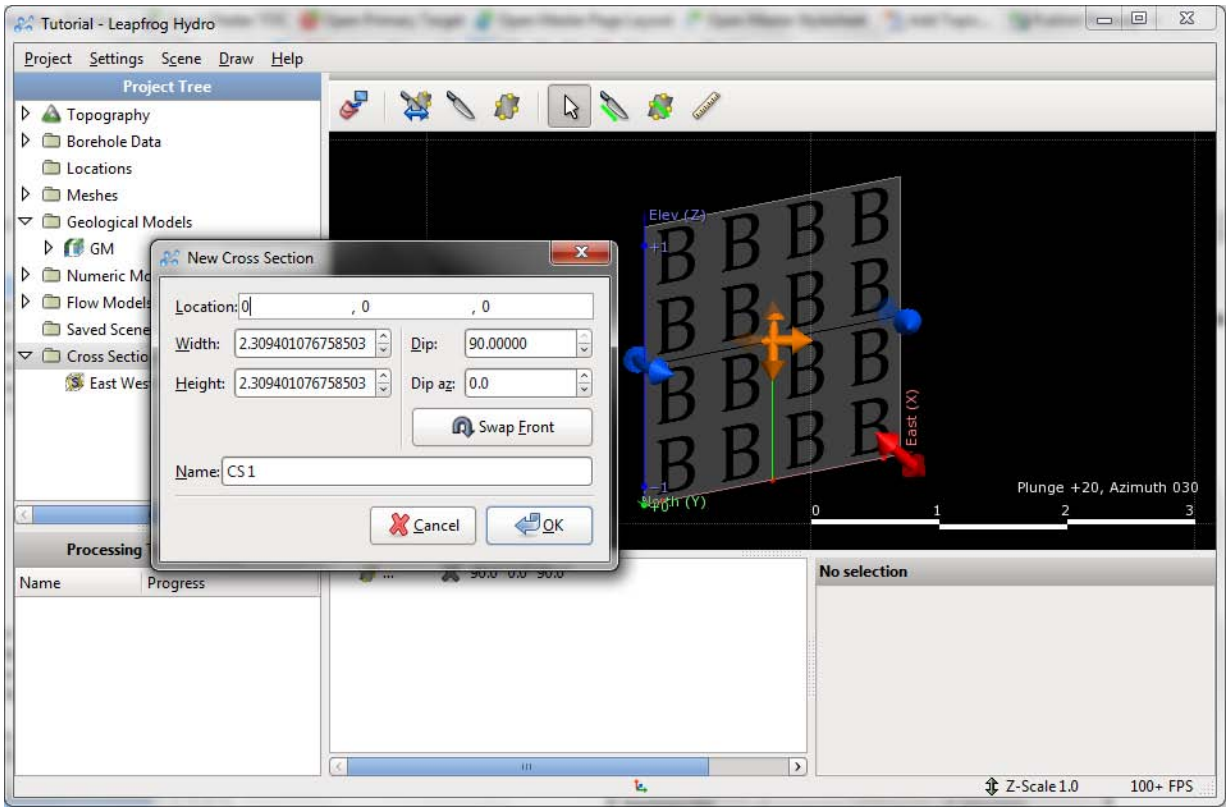


Once you have created a cross section, you can edit it or use it to evaluate a geological model. See:

- Editing a Cross Section
- Evaluating a Geological Model

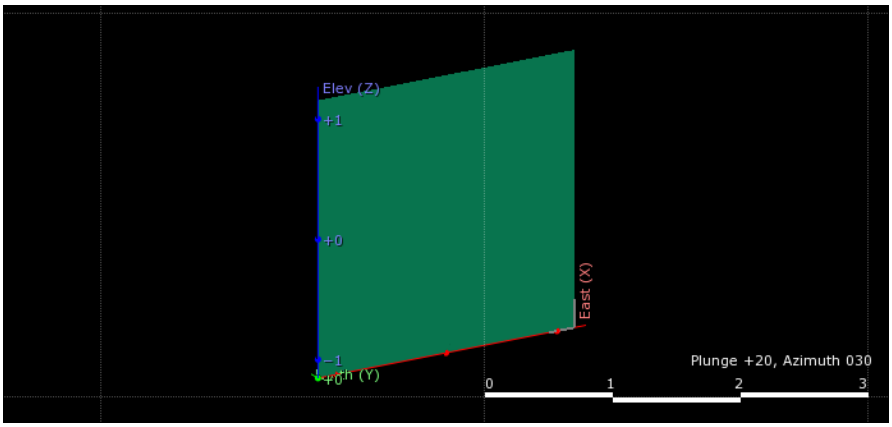
Creating a New Blank Cross Section

To create a new blank cross section, right-click on the **Cross Sections** object and select **New Blank Cross Section**. The **New Cross Section** window will be displayed, together with a set of controls in the scene that will help you to create the cross section:



The back of the cross section is marked with the letters B, while the front of the cross section is marked with Fs.

Use the controls to make the changes required, then click **OK**. The new cross section will be created and added to the **Cross Sections** object. Display it by dragging it into the scene or by right-clicking on the cross section and selecting **View Object**. The cross section will appear in the scene:

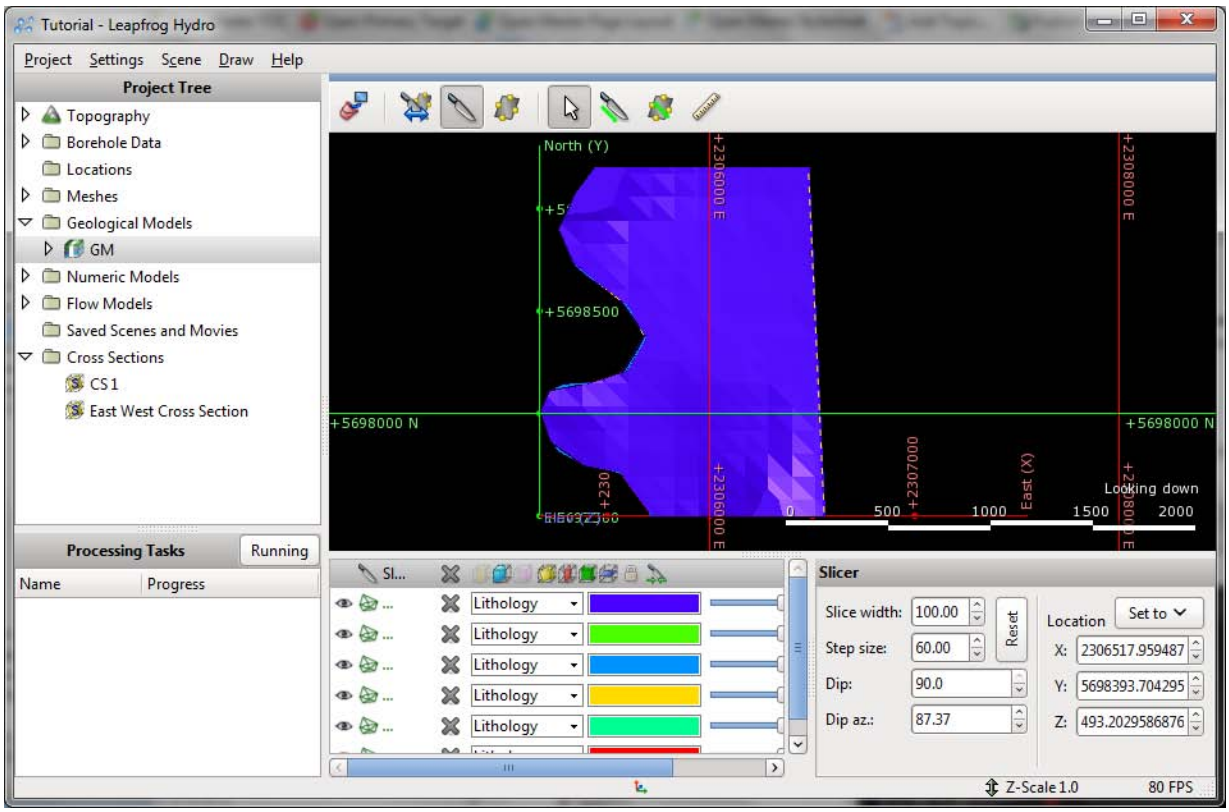


Once you have created a cross section, you can edit it or use it to evaluate a geological model. See:

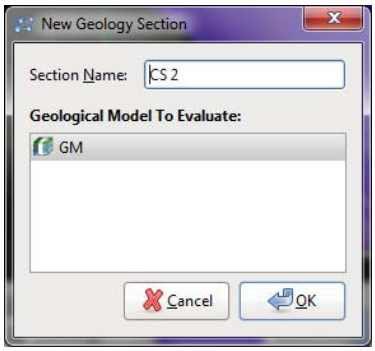
- Editing a Cross Section
- Evaluating a Geological Model

Creating a New Cross Section from the Slicer

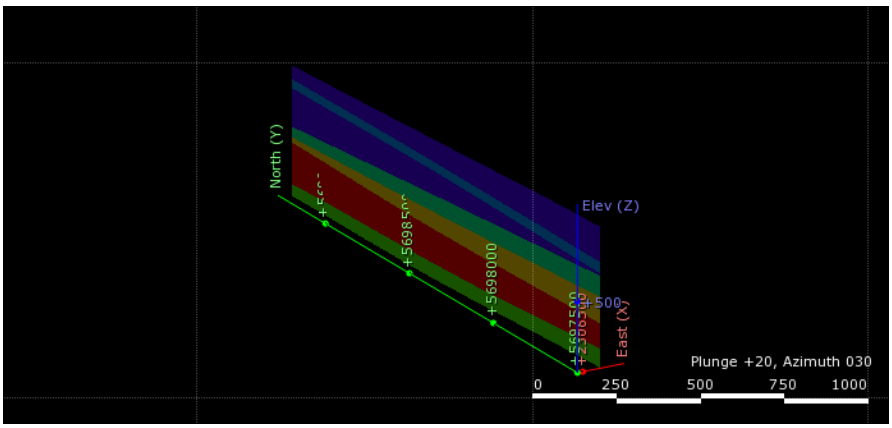
To create a new cross section from the slicer, drag a geological model into the scene and orient it in a suitable direction for the cross section you wish to create. Use the **Draw slicer line** tool to draw the new cross section. In the image below, the geological model is displayed from above and a line has been drawn across it to create a vertical cross section:



Next, right-click on the **Cross Sections** object and select **New Cross Section From Slicer**. The **New Geology Section** window will be displayed:



A list of the geological models created in the project is displayed in the **Geological Model To Evaluate** list. Select the model required, enter a **Section Name** and click **OK**. The new cross section will be created and added to the **Cross Sections** object. Display it by clearing the scene, then adding the cross section to the scene. The cross section will appear in the scene:

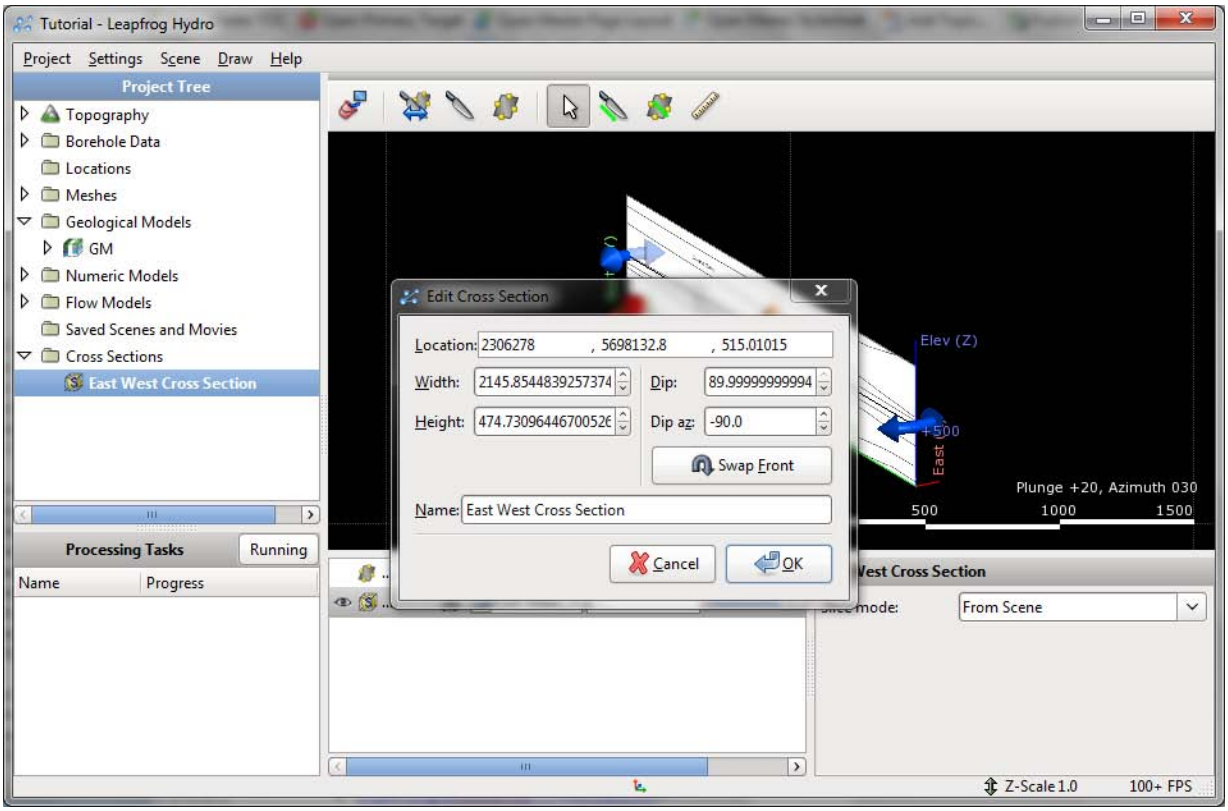


You can then edit the cross section, use it to evaluate a different geological model or export an evaluation as a lithology cross section. See:

- Editing a Cross Section
- Evaluating a Geological Model
- Exporting a Lithology Cross Section

Editing a Cross Section

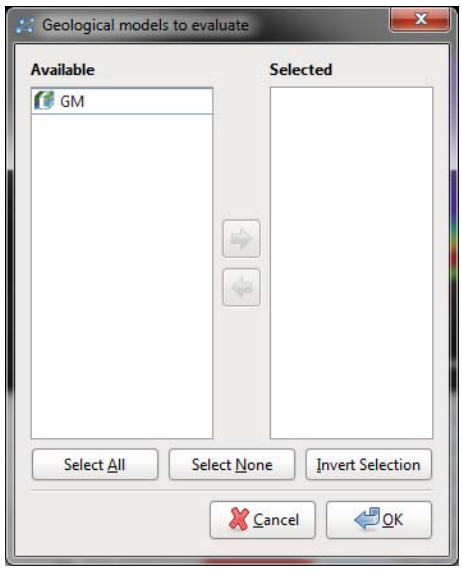
To edit a cross section, you can either double-click on the cross section in the **Cross Sections** object or right-click and select **Open**. The **Edit Cross Section** window will be displayed, together with a set of controls in the scene that will help you to edit the cross section:



Use the controls to make the changes required, then click **OK**. The cross section will be updated.

Evaluating a Geological Model

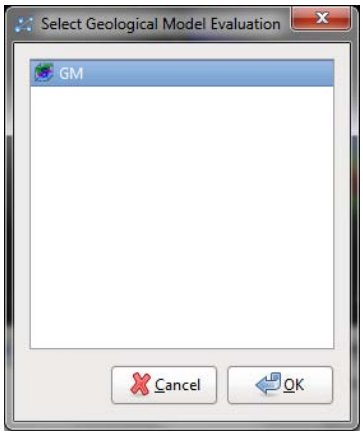
To display a geological model on a cross section, right-click on the cross section in the **Cross Sections** object and select **Evaluate Geological Model**. The **Geological models to evaluate** window will be displayed, with a list of geological models available in the project:



Only one geological model can be evaluated at a time. Move the geological model you wish to display on the cross section into the **Selected** list and click **OK**.

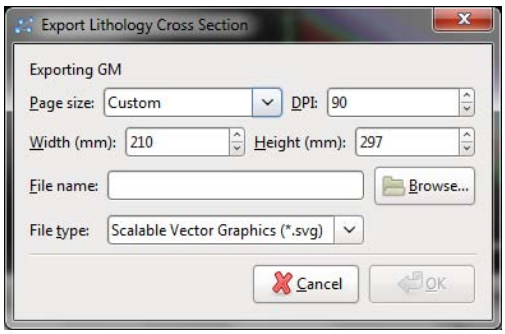
Exporting a Lithology Cross Section

Once you have evaluated a geological model on a cross section (see Evaluating a Geological Model), you can export a lithology cross section. To do this, right-click on the cross section in the **Cross Sections** object and select object and select **Export Lithology Cross Section**. The **Select Geological Model Evaluation** window will be displayed:



Select the evaluation you wish to export, then click **OK**.

The **Export Lithology Cross Section** window will be displayed:



Enter the information required and select the **File type**. Options are:

- Scalable Vector Graphic (*.svg)
- Encapsulated PostScript (*.eps)

Click **OK** to create the file.

The Saved Scenes and Movies Object

This topic describes the **Saved Scenes and Movies** object and the tools available in Leapfrog for creating and exporting scenes and movies.

Options available for the **Saved Scenes and Movies** object are:

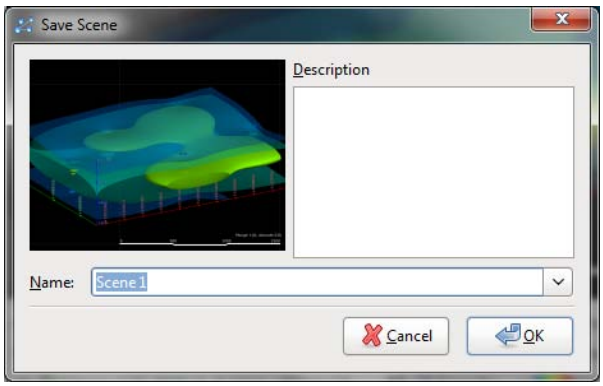
Once a scene has been saved, you can display it by dragging it into the scene or by right-clicking on it and selecting **Display**. You can also edit the scene and combine a series of scenes to create a movie.

For more information, see:

- Saving a Scene
- Editing a Saved Scene
- Creating a Movie
- Exporting a Scene

Saving a Scene

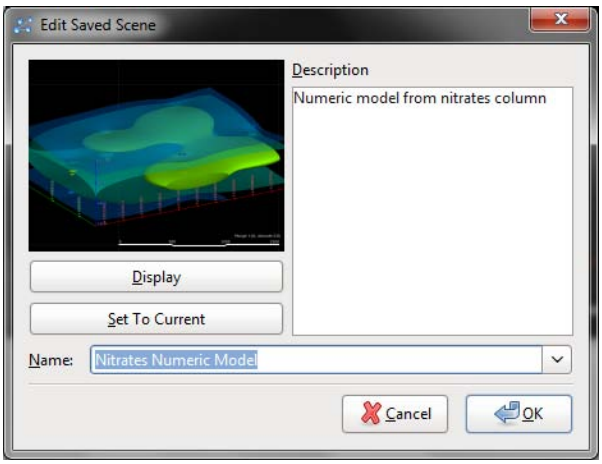
To save the current scene, right-click on the **Saved Scenes and Movies** object and select **Save Scene**. The **Save Scene** window will be displayed:



Enter a name and description for the scene and click **OK**. The scene will be saved and added to the **Saved Scenes and Movies** object.

Editing a Saved Scene

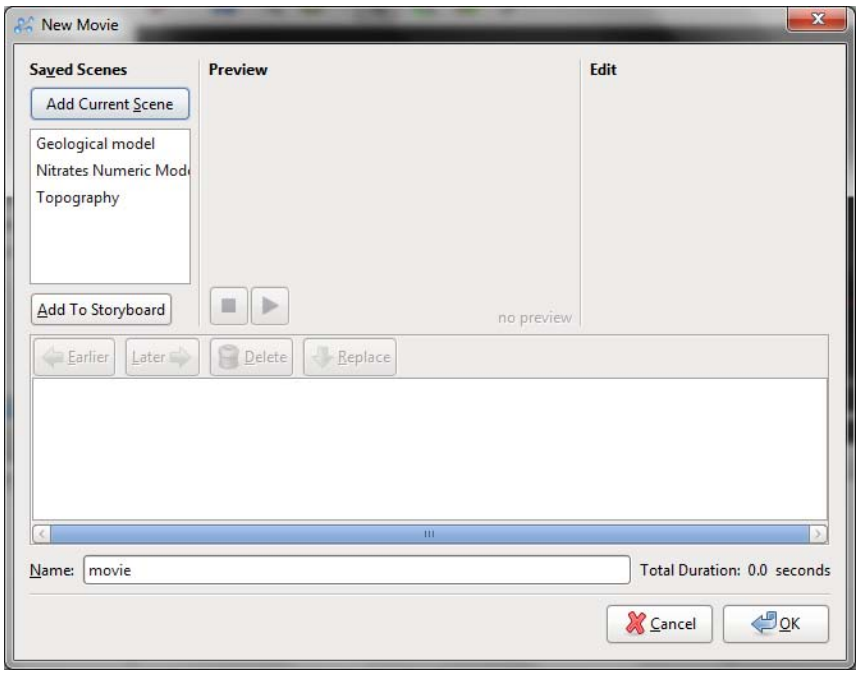
To edit a scene you have saved, right-click on the scene saved in the **Saved Scenes and Movies** object and select **Open**. The **Edit Saved Scene** window will be displayed:



Update the information and click **OK**.

Creating a Movie

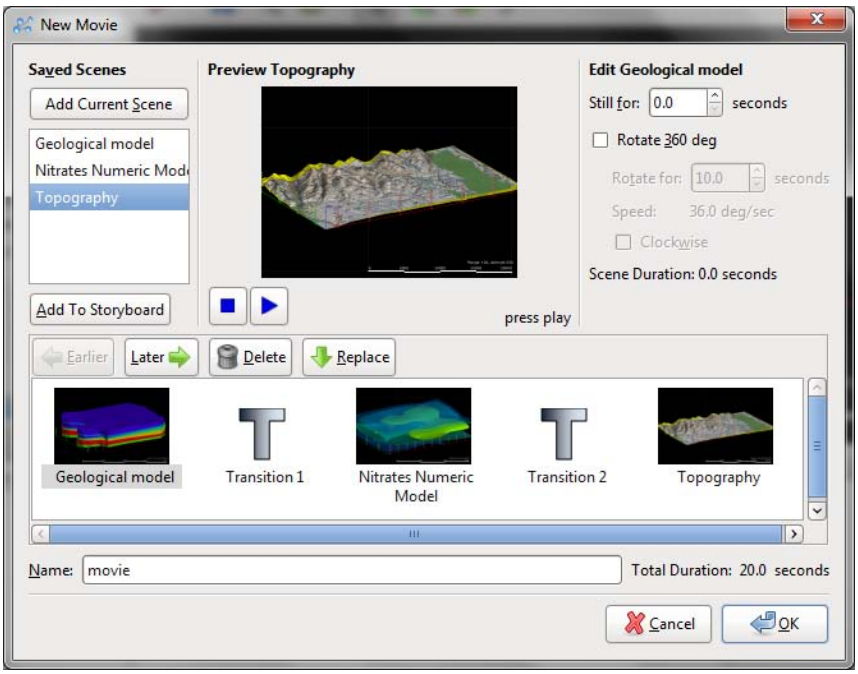
To create a new movie, right-click on the **Saved Scenes and Movies** object and select **Create Movie**. The **New Movie** window will be displayed:



Select the saved scenes you wish to add to the movie by clicking on them in the list, then clicking **Add To Storyboard**. You can also add a scene to the storyboard by double-clicking on it in the scene in the list. Adding a scene to the storyboard in this manner also adds a transition after the scene.

You can add the scene currently displayed in the scene window to the movie by clicking the **Add Current Scene** button.

Once you have added scenes to the storyboard, you can click on a scene to change how it will appear in the movie:

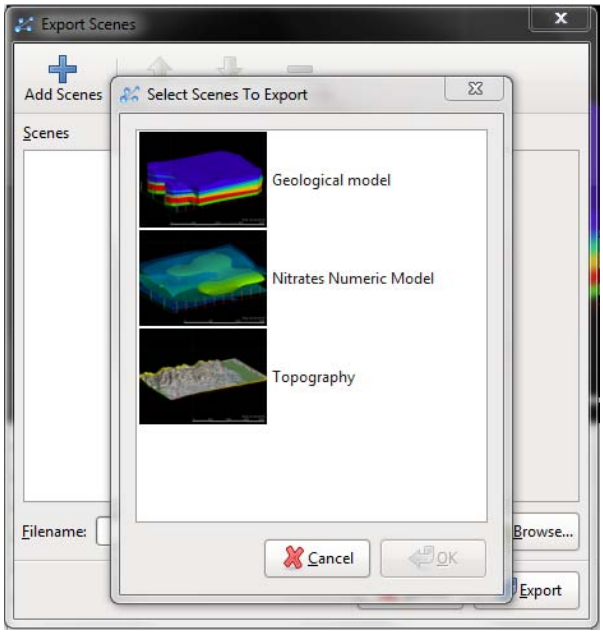


You can also change the transitions by clicking on them.

Enter a name and description for the movie and click **OK**. The movie will be saved and added to the **Saved Scenes and Movies** object.

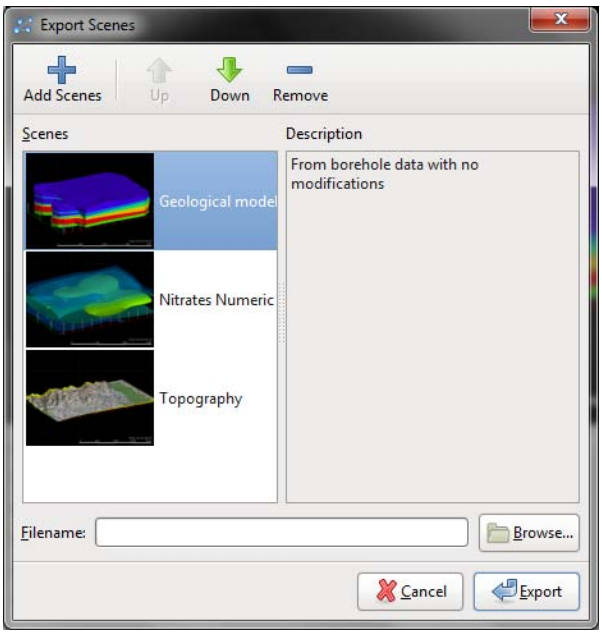
Exporting a Scene

To export a scene, right-click on the **Saved Scenes and Movies** object and select **Export Scenes**. The **Export Scenes** window will be displayed, together with the **Select Scenes To Export** window:



Select the scene you wish to export by clicking on it. If you wish to export more than one, hold down the `Shift` key while clicking on scenes. Click **OK** to close the scene list.

The **Export Scenes** window will be displayed with the selected scenes listed:



You can add more scenes to the list, if you wish, and change the order in which scenes will appear in the exported file. Enter a **Filename** and click **Browse** to choose a location, then click **Export**. Leapfrog will then export the scenes and return to the main window.