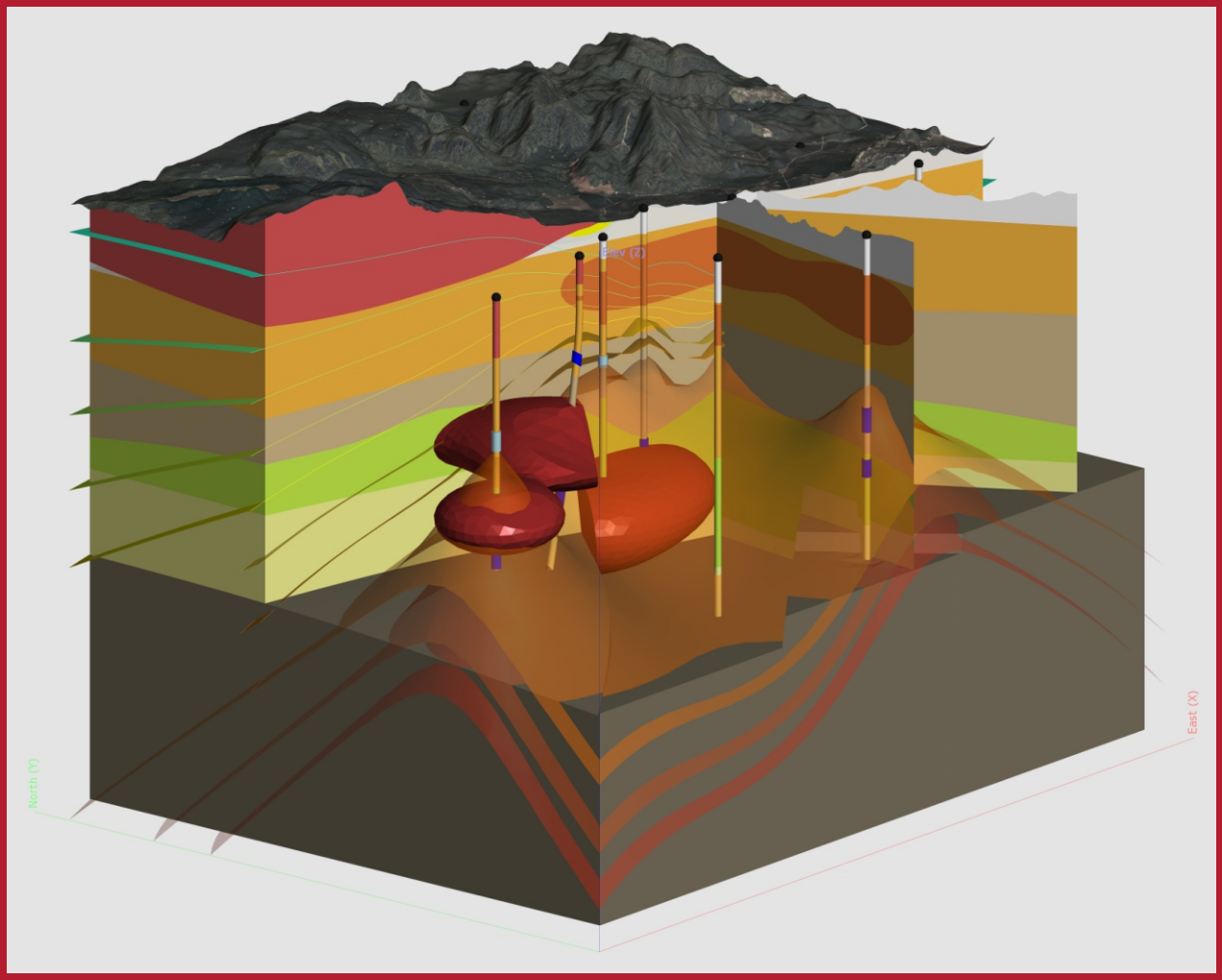
# INTRODUCTORY LEAPFROG GEOTHERMAL TRAINING COURSE

June 2017

Samantha Alcaraz
GNS Science, New Zealand

Location: Taupo, New Zealand

For: Environment Southland




GNS SCIENCE TE PŪ AO

leapfrog®
FAST, DYNAMIC GEOLOGICAL MODELLING

**IMPORTANT NOTICE:** Terms of participation in Leapfrog Geothermal Training Course

The material and data contained in these training notes are the property of GNS Science. Accordingly, these training notes are for the sole use of participants in the Leapfrog Geothermal training course and must be kept confidential to them. The training notes may not be copied, adapted, published or otherwise distributed without the prior written permission of GNS Science.

# INTRODUCTION TO LEAPFROG GEOTHERMAL

## TUTORIAL 1: UNDERSTANDING AND USING A PROJECT

This tutorial is an introduction to the Leapfrog Geothermal software. You will be introduced to the different visualisation tools and options available using an existing project.

## TABLE OF CONTENTS


Introduction 2

The Leapfrog display 3

- Interacting with the scene ... 3
- Customizing the interface ... 5

Leapfrog Geothermal project tree 6

- Project tree structure ... 6
- Expanding the project tree ... 8
- Accessing tools via the project tree ... 9

Investigating an existing project 10

- Using the scenes ... 10
- Locating an object ... 10
- Changing the appearance of an object in the scene ... 12
- Accessing the data ... 13

Visualisation tools 13

- Topographic data ... 13
- Slicing through the data ... 18
- Quantitative measurements ... 20
- Querying data ... 23

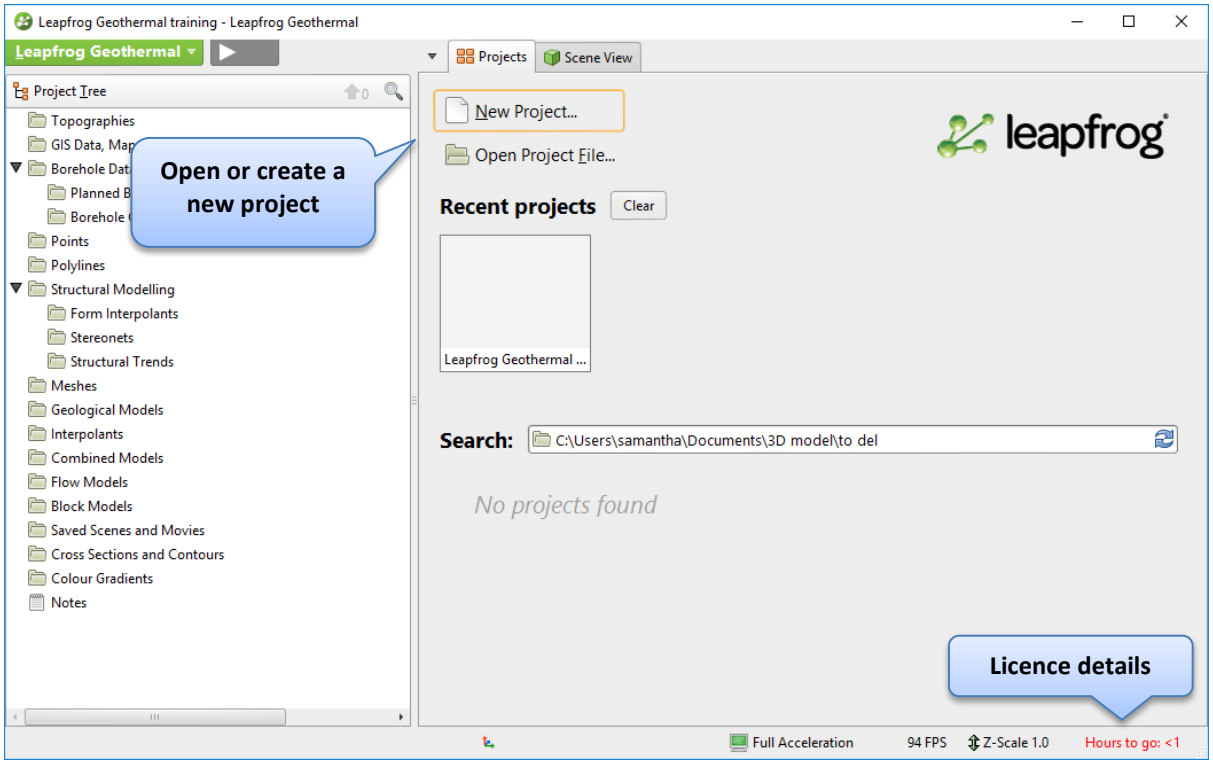
Conclusion 24

## INTRODUCTION

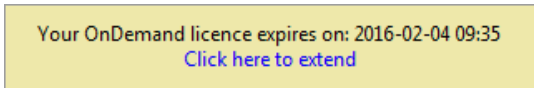
For this tutorial, we are going to navigate through an existing project.

- Open **Leapfrog Geothermal 3.2**

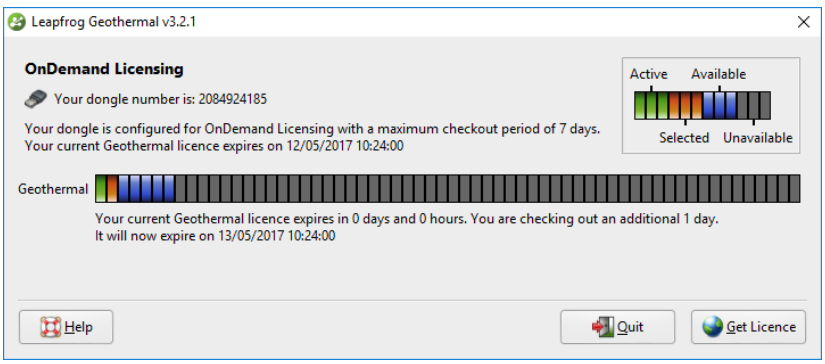
From the opening window you can see the list of your recent projects, if any, you can create a new project, or you can open an existing one. From this window you can also check your OnDemand licensing details.




- Click on the **"Hours to go"** button to get more information on your OnDemand licence.




- Click on **Click here to extend**. The software is contacting the Licence Server. It will bring the **OnDemand Licensing** menu where you can check your licenses for a longer time period. Click on **Quit**.



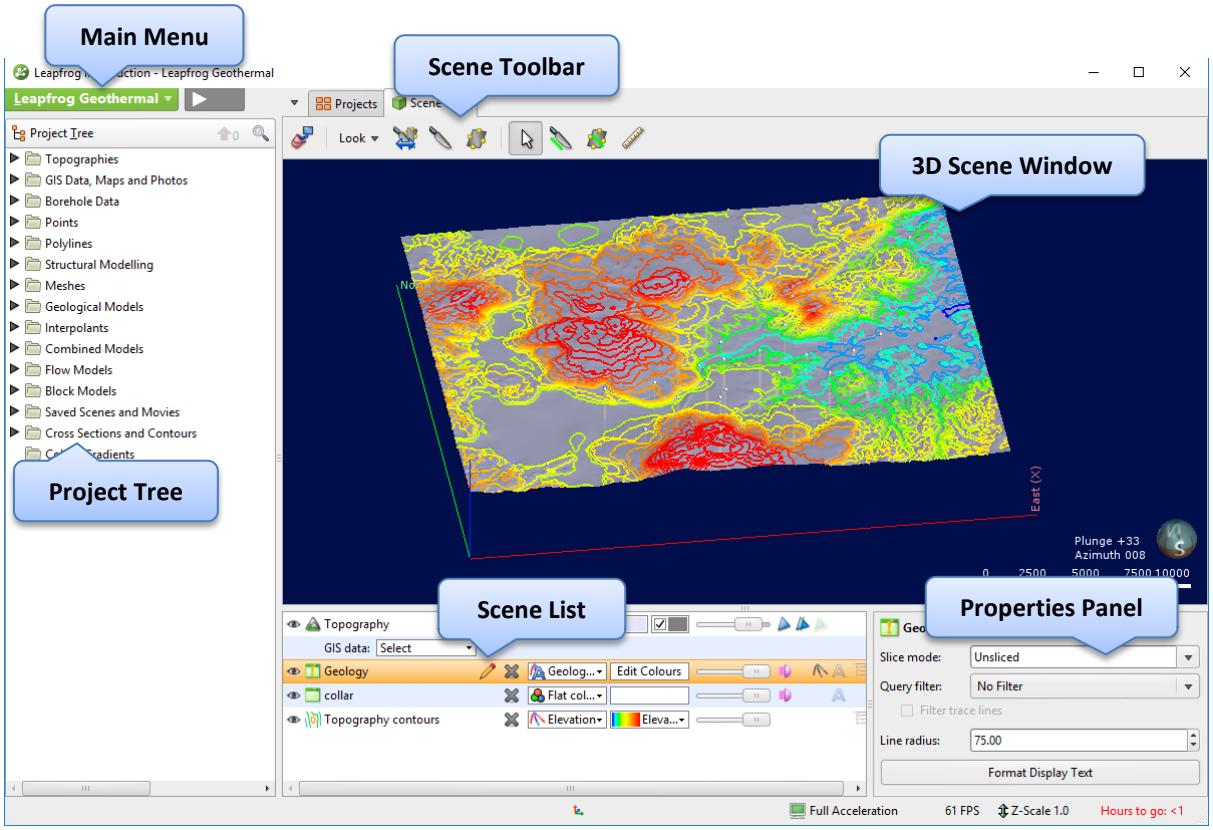

- Select **Open Project File**.
- Navigate to the **Leapfrog Training\Project\Leapfrog Introduction** folder.
- Select the *Leapfrog Introduction.aproj* file.
- Click **Open.**

## THE LEAPFROG DISPLAY

The Leapfrog Scene View display can be divided into 4 main parts:

- The **Project Tree** that contains the project data.
- The **3D Scene Window** which displays selected data in 3D.
- The **Scene List** and **Shape Properties Panel** which control how the data is displayed. They are also used as a working set of the data relevant to the current task.
- The **Processing Panel** which lists objects being processed or waiting processing.

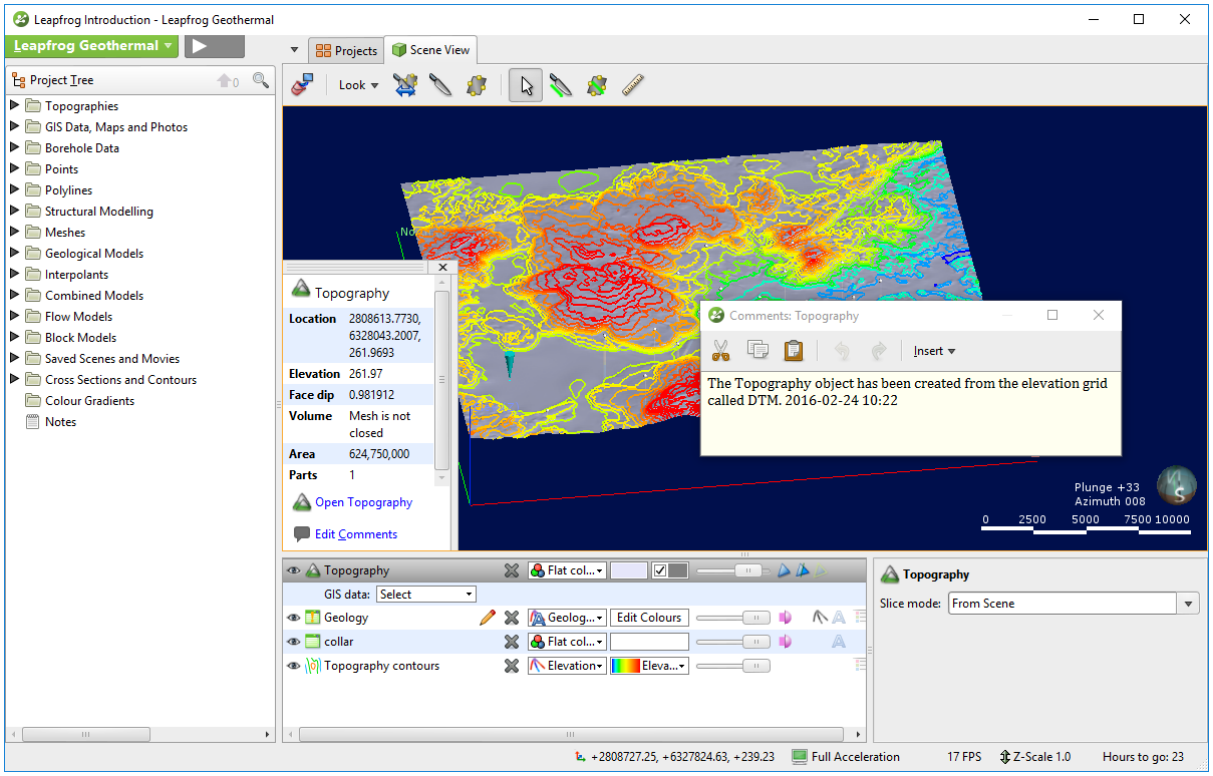
Below is a typical view of what you will see when you start a Leapfrog project, with the project tree and 3D Scene Window left in the state when the project was last opened.




## INTERACTING WITH THE SCENE

When the data is displayed in the scene, you can view and interact with it. Key interactions are mouse operations:

- Click and drag the left mouse button to **rotate** the scene.
- Click and drag the right mouse button to **zoom in and out** the scene. You can also use the scroll wheel to zoom in and out.
- Click and drag the middle mouse button or click the left and right button of your mouse simultaneously to **pan** the scene.
- Left click on an object in the 3D Scene highlights it in the **Scene List** and brings up information about the object. The software also allows the user to enter comments about objects.




Useful keyboard operations include:

- The **home** key sets the view to displayed data.
- Access standard geographical views: **U** for looking **U**p, **D** for looking **D**own, **E** for looking **E**ast, **W** for **W**est, **S** for **S**outh, **N** for **N**orth.

- Take some time to rotate the scene using these simple operations. You can also access these options from the **Look** menu in the Scene toolbar.

The scene window displays a 3D representation of selected data from the project tree. You can add objects from the project tree to the scene in two ways:

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

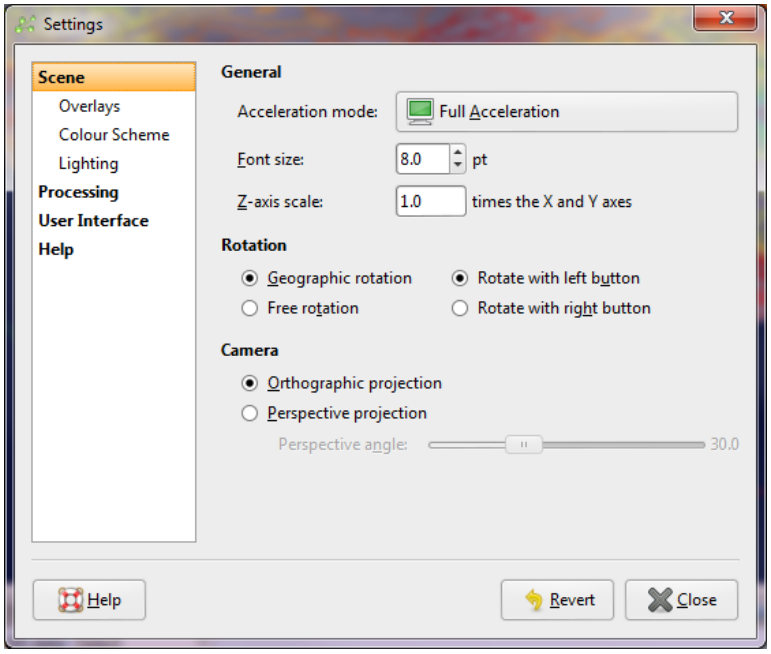
The Scene toolbar provide various options that we will investigate throughout this tutorial:

- Clear scene: remove all the data present in the 3D Scene.
- Look: options to access preset views and define bookmarks.
- Plane and Slicer options.
- Ruler option.
- Standard Select button.

## CUSTOMIZING THE INTERFACE

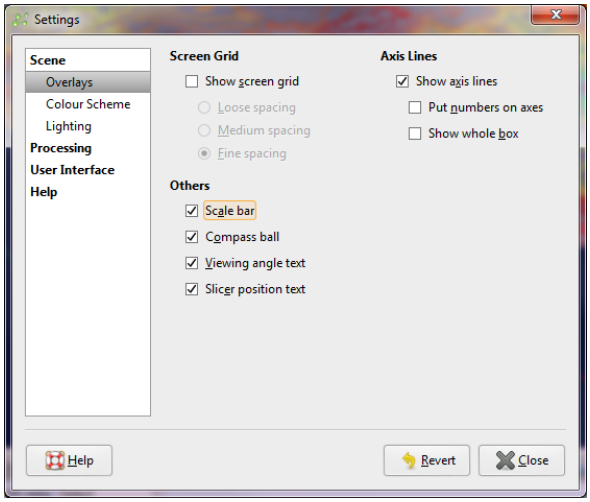
Leapfrog Geothermal provides many options to customize the interface to your liking.

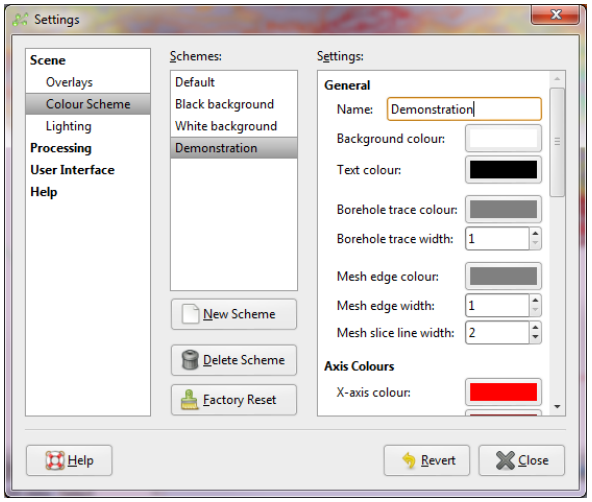
- Click on the main **Leapfrog Geothermal Menu** and select **Settings**.




From this menu you can change the Z-axis scale, your text size, Rotation and Camera options.

- Click on **Overlays** to access the optional display options: Grid, Axis lines, Scale bar, Compass and more. Click on the options to see the differences in the 3D scene window.
- Click on the **Colour Scheme** to change your background colour and other colouring options. You can create your own Scheme.

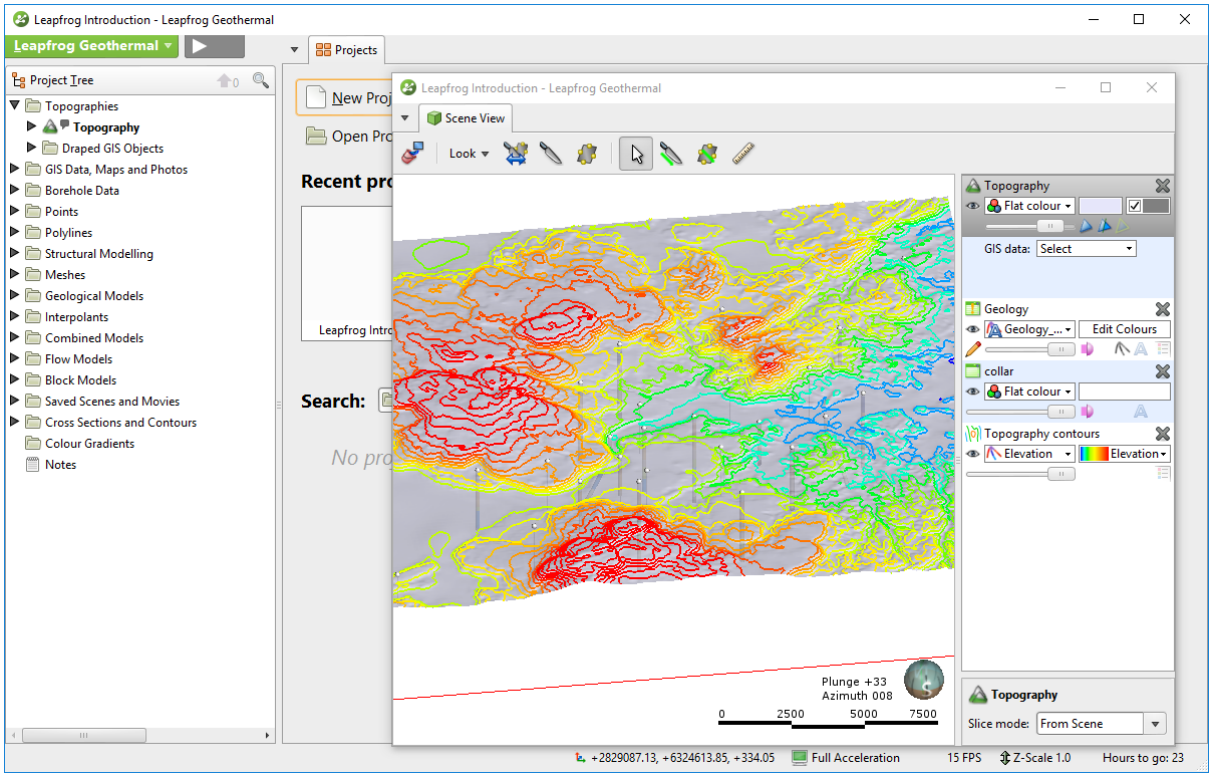



- Click on the **User Interface** and select **Show Tree Lines** and **Show Save Scene** (the latest will be added to your Scene toolbar).
- Investigate the other options. When satisfied, click **Close**.

You can also customize the size and location of your tabs and scene windows.

- Click and drag the grey partitions to change the size of your scene panels.
- Click on the partition of the scene list and drag it to the right hand side of your window to set to the side instead of bottom.
- Click and drag the **Scene View** Tab to open it as an independent window. Follow the same method to bring it back into the main Leapfrog Geothermal interface.

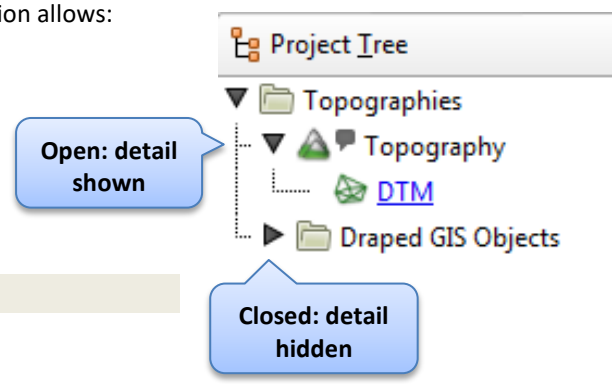



## LEAPFROG GEOTHERMAL PROJECT TREE

In Leapfrog, objects in the project tree are organized **from the top down**. The objects used to build the models, and the models themselves, are located within the project tree folders.

The first step in understanding a project and investigating its contents is to compact the tree by hiding the detail. This is done using the small arrows on the left side of the project tree. If an object does not have a side arrow then there is no detail stored below. This function allows:

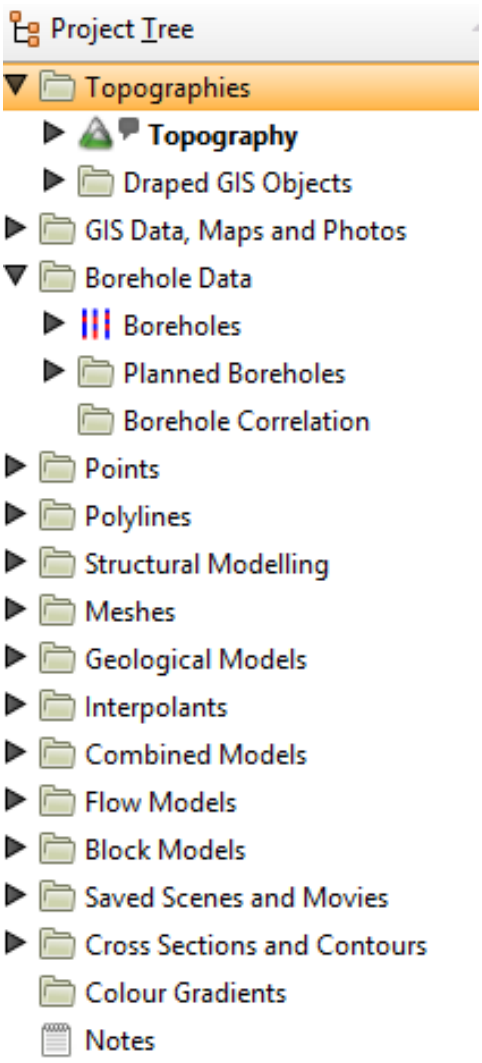
- To hide the details unless you want to focus on the specific construction of an object.
- It supports a workflow where a simple model is successively refined.




- Compact the project tree.

## PROJECT TREE STRUCTURE

The base level of the project tree is made of 16 folders. All the data and tools available within a project are found within these folders:




- **Topographies**: In Leapfrog Geothermal, the topography represents the ground surface and all data associated with it. All objects that make up the topography are stored under this folder. It also hosts the clipping boundary object which defines a region of interest.
- **GIS Data, Maps and Photos**: Vector data can be imported to define land use in two dimensions. The maps and photos can also be imported, and geo-referenced within the interface.
- **Borehole data**: This folder contains the **Boreholes** object which is made of tables defining the physical 3D shape of boreholes. It must contain the location and survey data, and at least one interval table. This folder also hosts the **Planned Boreholes** folder where new wells can be created and evaluated against existing models and the **Borehole Correlation** folder.
- **Points:** Points must have 3D coordinates, with or without associated values. They can be used for visualization only or as input to define surfaces and models.
- **Polylines**: Polylines are created or imported in Leapfrog Geothermal as a source of manual input to generate or constrain surfaces.
- **Structural Modelling:** This folder contains structural measurements, created within Leapfrog or imported and option to model structural data
- **Meshes:** Meshes are used to represent surfaces. A mesh is a collection of vertices and triangles that define the shape of a 3D surface object. Leapfrog Geothermal can import meshes created in other software, and can also export meshes for use in other software.
- **Geological Models:** All geological models created in a Leapfrog project are saved under this folder. A geological model contains several key components (**Boundary**, **Fault System**, **Lithologies**, **Surface Chronology** and **Output Volumes**).
- **Interpolants**: Interpolants are numeric models representing the physical properties of an area from either borehole data or point data. **Interpolant Models** contain five key components (**Boundary**, **Trend**, **Values**, **Isosurfaces**, **and Output Volumes**). **Multi Domain Interpolant** and **Indicator Interpolant** can also be created from this folder.
- **Combined Models**: This folder provides the tools to combine categorical model (Geological model) with interpolant models.
- **Flow Models**: This object can be used to build, import and export flow grids. These grids can be created in Leapfrog Geothermal, or created in external software and visualised in Leapfrog Geothermal. Leapfrog Geothermal currently provides options to work with ModFlow, FEFLOW and TOUGH2 models.
- **Block Models**: This folder can be used to create or import block model grids to evaluate them against your geological and interpolant models. You can also import key standard grid files such as Magnetotelluric Resistivity files, UBC and Gocad models.

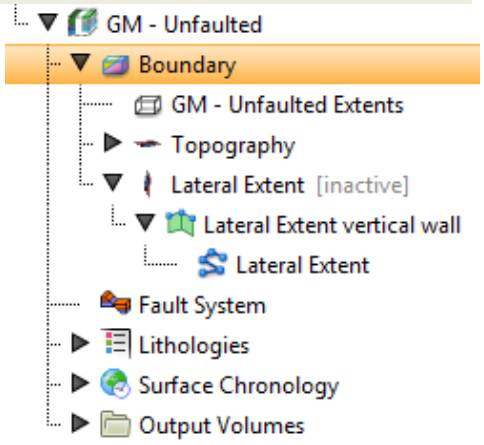
- **Saved Scenes and Movies**: **Scenes** are like bookmark views of the project data; the user can save a scene from any data in the 3D window. Scenes save the objects present in the 3D view, and the display properties, but it doesn't save the data itself. Scenes can be exported for use in the Leapfrog Viewer so you can share your information. **Movies** are created by combining multiple scenes.
- **Cross Sections and Contours**: Cross section images can be imported and geo-referenced in this object. Cross sections of geological or interpolant models within the Leapfrog project can also be created and exported. This folder also provides the tools to contour any surface present in the model.
- **Colour Gradients**: This folder provides the tools to import colour gradient from other software packages.

*Note: While the main project tree structure cannot be altered, it is possible to create subfolders to organise the data imported in the project.*

## EXPANDING THE PROJECT TREE

The Leapfrog Geothermal structure supports a workflow where a simple model is successively refined. Let's investigate how a geological model is built.

- Expand the **Geological Models** folder.
- Expand the geological model called **Geological Model – Unfaulted**. You can see the 5 key components mentioned previously:
  - Boundary.
  - Fault System.
  - Lithologies.
  - Surface Chronology.
  - Output Volumes.
- Expand the **Boundary** Object. You can see that this object is created from 3 items:
  - A Boundary box (**GM - Unfaulted Extents**).
  - The **Topography**.
  - A **Lateral Extent** which is used to define the boundary of the geological model. If one continues expanding the **Lateral boundary**, we can see that it is defined by a **polyline** .



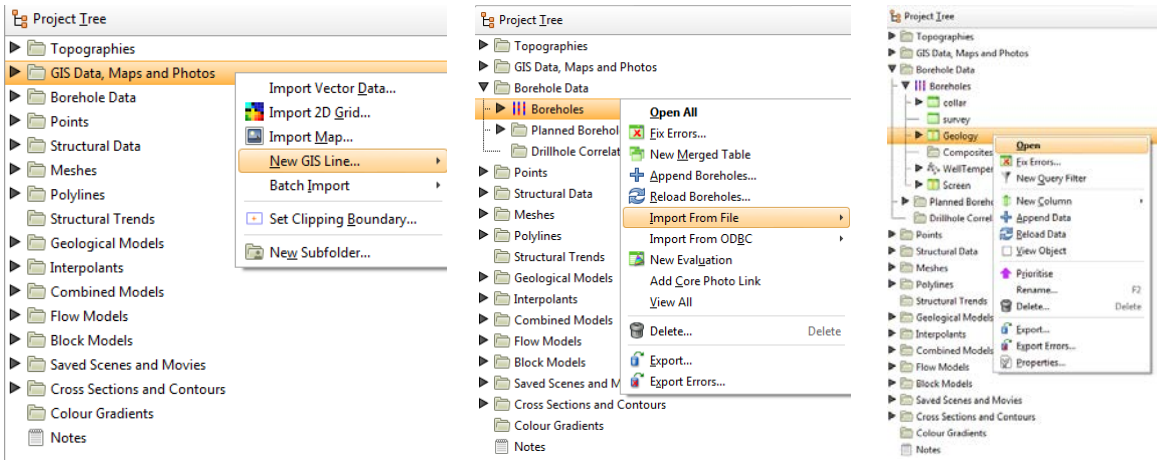

This principle of expanding to reveal the object source applies to all objects in the project tree.

## ACCESSING TOOLS VIA THE PROJECT TREE

- Right-click on each object in the project tree to view the actions you can perform for that object. This works at any level of the structure.

For example:

- The **GIS Data, Maps and Photos** object can be used to import vector data and maps, create new GIS line objects and define the region of interest.
- The **Boreholes** object can be used to add or create new data, delete or export data.
- The **Interval table** object (geology in the following image) reveals a menu showing possible actions specific to this dataset.




When a **bold** option appears in the right-click menu, it means the option can also be selected by double-clicking on the object. For example, the **Open** tool in the images above.

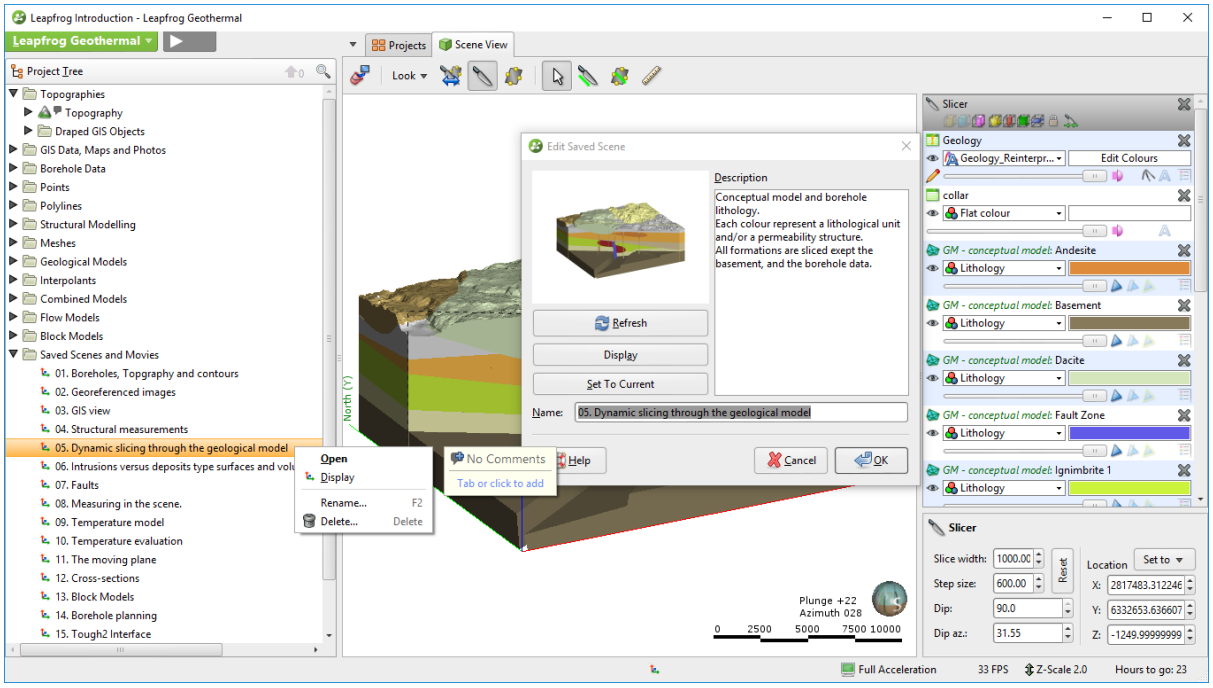
Objects shown in **bold text** in the project tree are currently displayed in the scene window.

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

## USING THE SCENES

To investigate and understand an existing model, it is often useful to look at the scenes saved in the **Saved Scenes and Movies** folder. A saved scene is a way of storing the scene list and settings for later use, like a bookmark. It is very useful for providing a stable point of reference in a project and for explaining important aspects of the project. In a saved scene a collection of objects are displayed, along with a summary of what the scene is showing.

- In the demonstration project you have opened, you can see that several scenes are saved. You can investigate the contents of a scene in two ways:
- Right-click on one scene of interest and select **Open**. This brings up a thumbnail sketch and a brief description of what the scene shows (if it has been documented).
- Dragging the scene into the viewer will allow you to interact with the data in the scene.

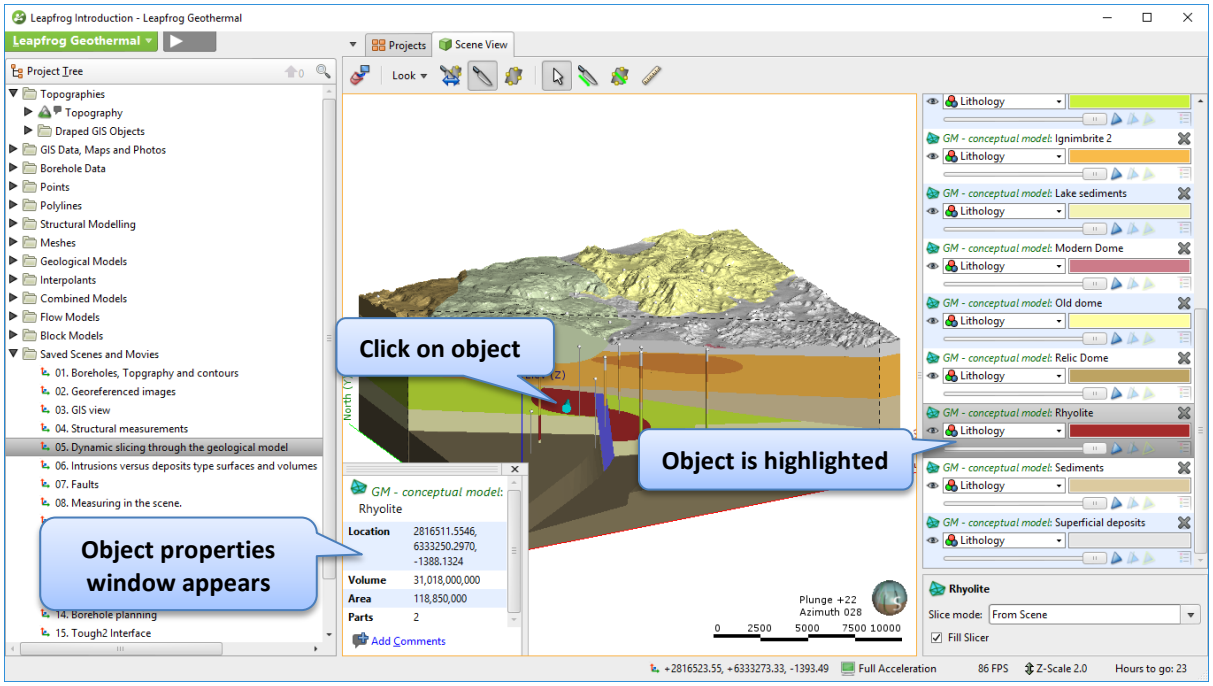



## LOCATING AN OBJECT

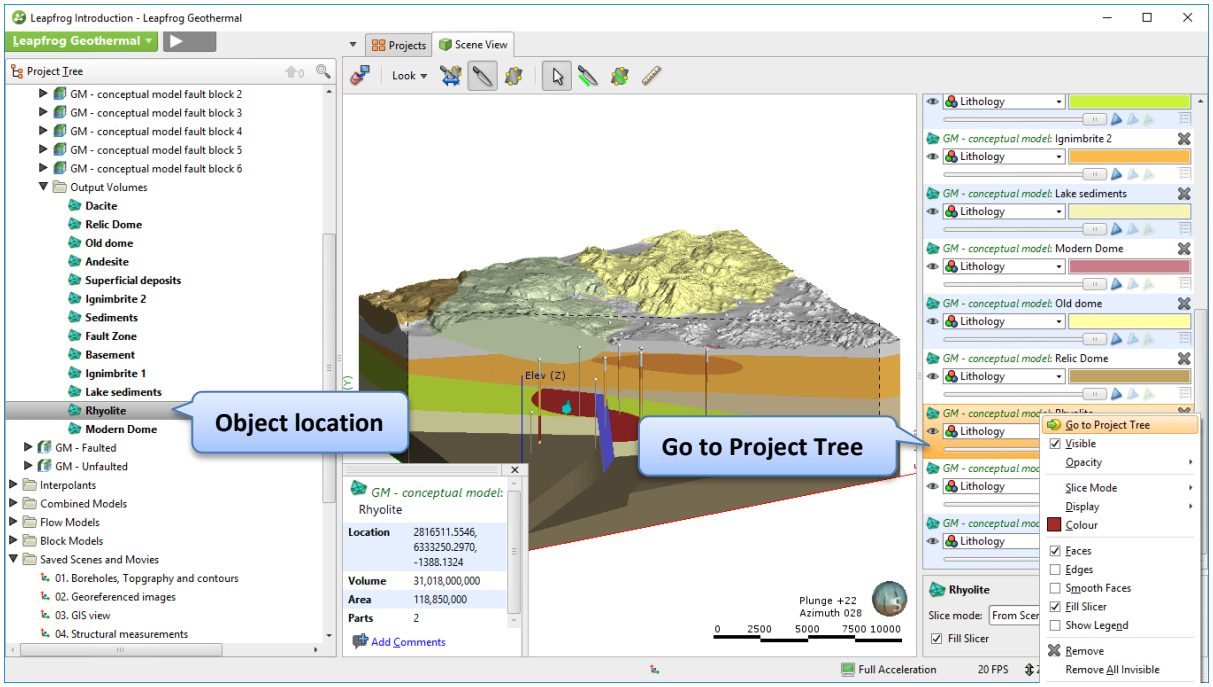
The scene list contains all the objects available for viewing in the 3D viewer. It can be used to store a collection of objects needed for a particular task or within a specific scene.

- It is often easier to make an object temporarily invisible than to remove it from the scene list. To make an object invisible, toggle the “eye” beside the object in the scene list.
- There are a large number of other options for controlling the appearance of an object in the scene, and these vary depending on the object being viewed, and will be described in detail.

The easiest way of locating an object in the scene list is to click on it in the 3D display window. This opens the object properties window and also highlights it in the Scene List, where its appearance can be changed.

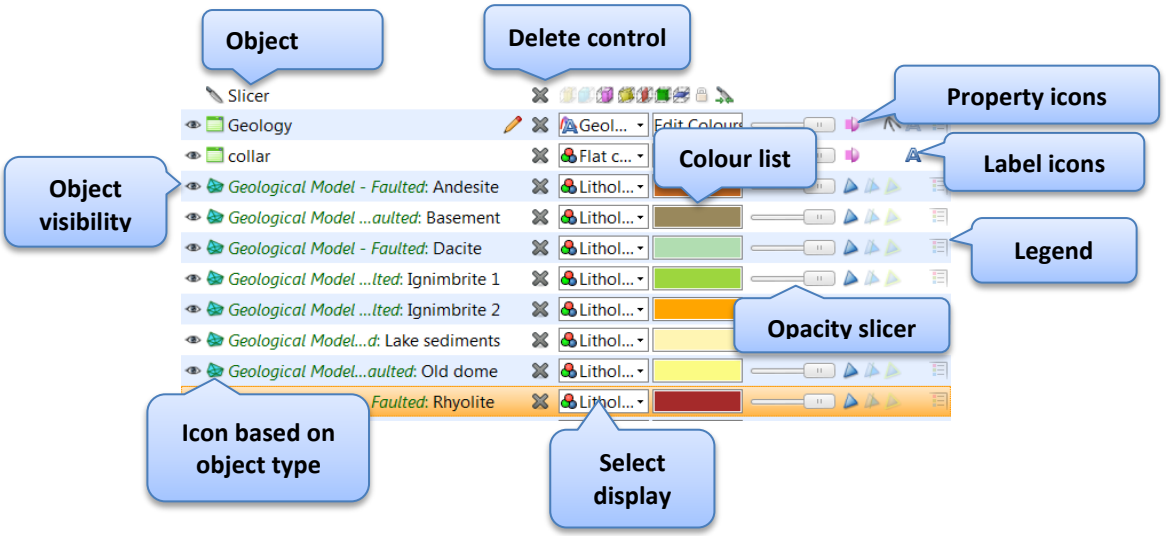



- Right-clicking on the object in the Scene List and selecting **Go to Project Tree** takes you to the location of the object in the Project Tree.



## CHANGING THE APPEARANCE OF AN OBJECT IN THE SCENE

There are two places where the appearance of an object can be changed. The first is in the controls that are associated with the object in the scene list.




- Common features on all objects:
  - **Delete control**: removes the object from the scene. It doesn't delete the data from the project.
  - **Object name.**
- Features available on all except the slicer, ruler and moving plane.
  - **Object visibility:** click to make an object visible or invisible in the scene.
  - **Select display**: controls which parameter is displayed (for example Lithology along a borehole).
  - **Colour list**: controls how numerical values and lithological units are coloured.
  - **Opacity slider**: changes the transparency of the object.
- **Property icons**, specific to an object: these vary with the type of the object. For example, showing the triangles on a mesh or rendering points as spheres.

The second area for controlling how an object appears is in the **properties box**, where more detailed control of the data is provided. This is available only for the selected object.

The main items appearing on the properties box are:

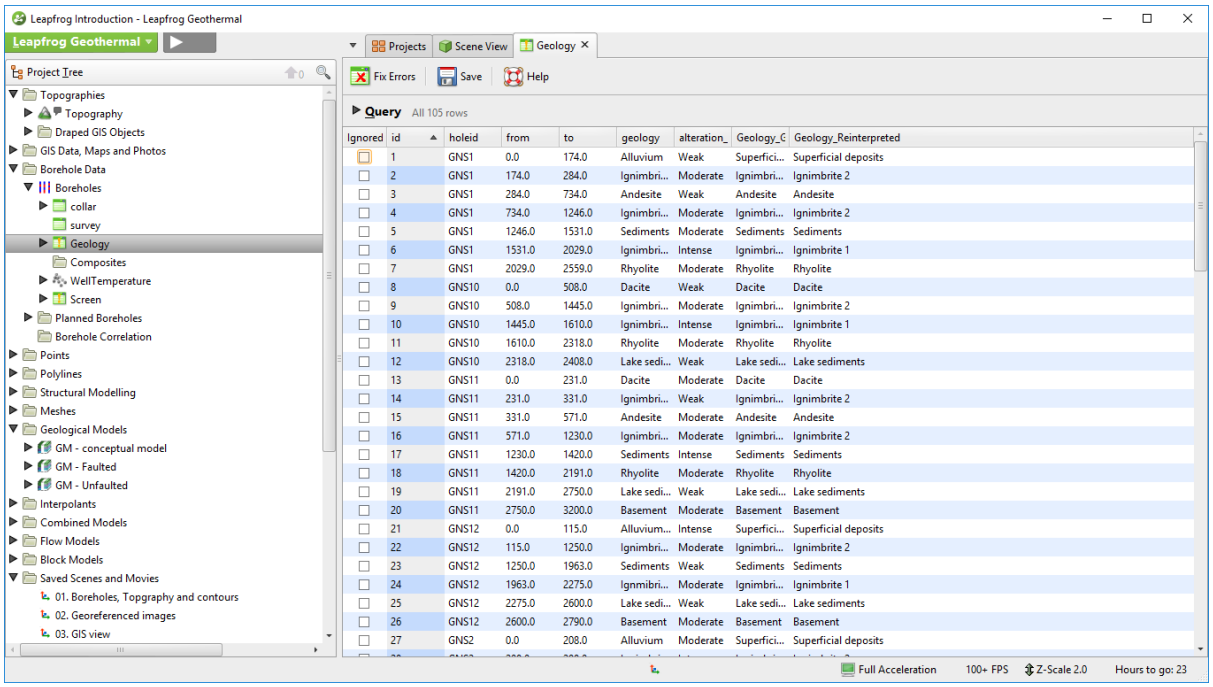
- **Slice mode**: enables the user to change how the slicer interacts with an individual object.
- **Object size**: controls line width, point size or radius of a sphere.
- **Query filter**: allows the user to display only a subset of the data as defined by a database query.
- **Value filter**: displays a subset of the data based on numerical values.

*Note: A saved scene does not maintain an independent copy of the data, so if the data changes then so does the appearance of the scene.*

## ACCESSING THE DATA

Many objects in the project tree are tabulated datasets (e.g., boreholes tables, points, structural measurements, even shapefiles). You can visualise the tables or attributes when opening these objects.

- Open the **geology** table from the Project Tree. Double-click on the object or right-click open to see the table.




| Ignored | id | holeid | from | to | geology | alteration_ | Geology_C | Geology_Reinterpreted |
|---|---|---|---|---|---|---|---|---|
| | 1 | GNS1 | 0.0 | 174.0 | Alluvium | Weak | Superfici... | Superficial deposits |
| | 2 | GNS1 | 174.0 | 284.0 | Ignimbri... | Moderate | Ignimbri... | Ignimbrite 2 |
| | 3 | GNS1 | 284.0 | 734.0 | Andesite | Weak | Andesite | Andesite |
| | 4 | GNS1 | 734.0 | 1246.0 | Ignimbri... | Moderate | Ignimbri... | Ignimbrite 2 |
| | 5 | GNS1 | 1246.0 | 1531.0 | Sediments | Moderate | Sediments | Sediments |
| | 6 | GNS1 | 1531.0 | 2029.0 | Ignimbri... | Intense | Ignimbri... | Ignimbrite 1 |
| | 7 | GNS1 | 2029.0 | 2559.0 | Rhyolite | Moderate | Rhyolite | Rhyolite |
| | 8 | GNS10 | 0.0 | 508.0 | Dacite | Weak | Dacite | Dacite |
| | 9 | GNS10 | 508.0 | 1445.0 | Ignimbri... | Moderate | Ignimbri... | Ignimbrite 2 |
| | 10 | GNS10 | 1445.0 | 1610.0 | Ignimbri... | Intense | Ignimbri... | Ignimbrite 1 |
| | 11 | GNS10 | 1610.0 | 2318.0 | Rhyolite | Moderate | Rhyolite | Rhyolite |
| | 12 | GNS10 | 2318.0 | 2408.0 | Lake sedi... | Weak | Lake sedi... | Lake sediments |
| | 13 | GNS11 | 0.0 | 231.0 | Dacite | Moderate | Dacite | Dacite |
| | 14 | GNS11 | 231.0 | 331.0 | Ignimbri... | Weak | Ignimbri... | Ignimbrite 2 |
| | 15 | GNS11 | 331.0 | 571.0 | Andesite | Moderate | Andesite | Andesite |
| | 16 | GNS11 | 571.0 | 1230.0 | Ignimbri... | Moderate | Ignimbri... | Ignimbrite 2 |
| | 17 | GNS11 | 1230.0 | 1420.0 | Sediments | Intense | Sediments | Sediments |
| | 18 | GNS11 | 1420.0 | 2191.0 | Rhyolite | Moderate | Rhyolite | Rhyolite |
| | 19 | GNS11 | 2191.0 | 2750.0 | Lake sedi... | Weak | Lake sedi... | Lake sediments |
| | 20 | GNS11 | 2750.0 | 3200.0 | Basement | Moderate | Basement | Basement |
| | 21 | GNS12 | 0.0 | 115.0 | Alluvium... | Intense | Superfici... | Superficial deposits |
| | 22 | GNS12 | 115.0 | 1250.0 | Ignimbri... | Moderate | Ignimbri... | Ignimbrite 2 |
| | 23 | GNS12 | 1250.0 | 1963.0 | Sediments | Weak | Sediments | Sediments |
| | 24 | GNS12 | 1963.0 | 2275.0 | Ignmibri... | Moderate | Ignimbri... | Ignimbrite 1 |
| | 25 | GNS12 | 2275.0 | 2600.0 | Lake sedi... | Weak | Lake sedi... | Lake sediments |
| | 26 | GNS12 | 2600.0 | 2790.0 | Basement | Moderate | Basement | Basement |
| | 27 | GNS2 | 0.0 | 208.0 | Alluvium | Moderate | Superfici... | Superficial deposits |

## VISUALISATION TOOLS

## TOPOGRAPHIC DATA

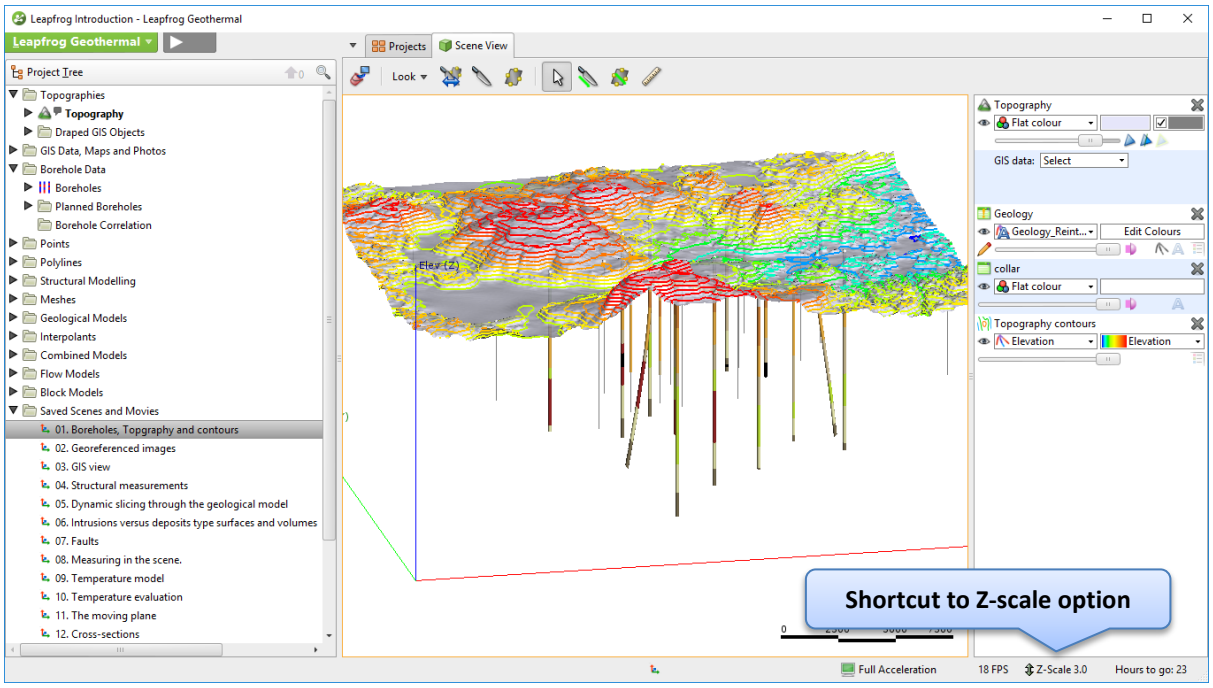
Varying height data in a project can pose problems. It is important for all the data to be consistent and relevant; hence a single elevation field (the Topography) is defined. Several data sources can be combined under this single object.

GIS data and maps typically display a 2D representation of land use at ground level. There are three ways of looking at GIS data and maps:

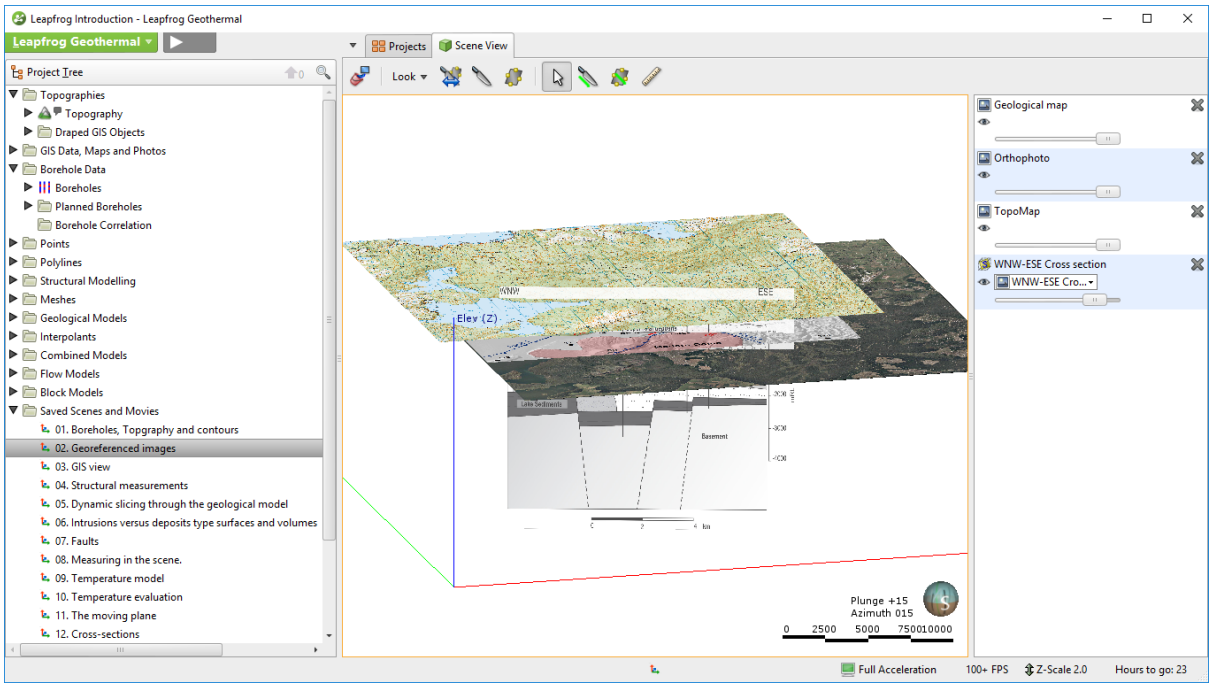
- In its “native” form: This can be either with its own height data or on fixed elevation if none is provided. This is achieved by dragging the GIS/Map data into the scene directly from the project tree.
- GIS objects are automatically draped on the Topography when imported in the project. You can see the object, projected at topography level from **Topographies/Draped GIS Objects.** While this option works well for lines, it is not perfect for polygons if you want to display them at the same time as the topography itself.
- On the topography: this is achieved by dragging the **Topography** object into the scene and selecting which GIS/Map is to be displayed. Maps can also be draped on any mesh.

- Display the saved Scene **01. Boreholes, topography and contours** in the demonstration project. You can turn on and off the Topography object and the contours. The contours are GIS vectors, with their own elevation value displayed in their native form.

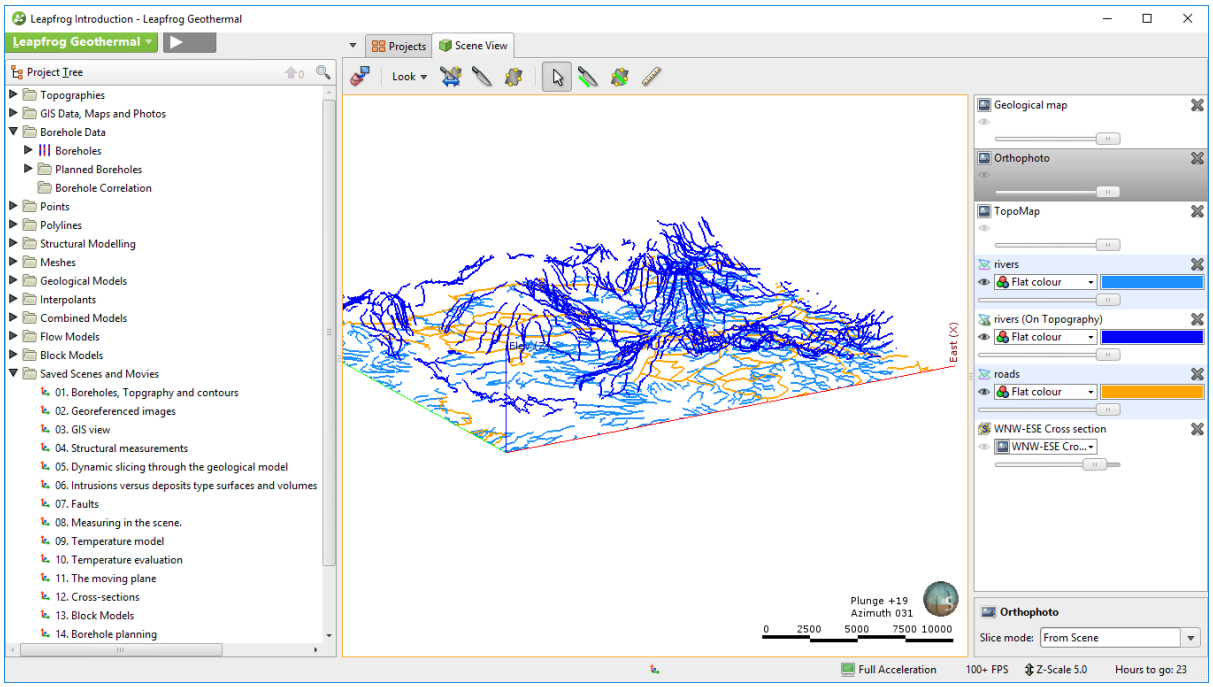
- Exaggerate the Z-scale.




- Display Scene **02. Georeferenced images.** You can see that raster images, either maps or cross section, can be imported in Leapfrog and appropriately geo-referenced. By default, maps are imported at 0 m elevation.



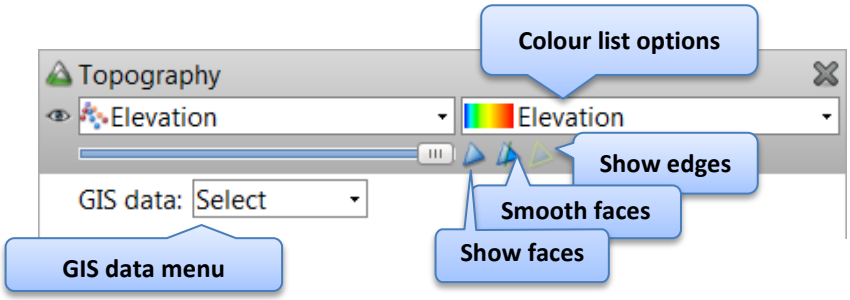

- Add GIS data in its native form, and GIS data draped on the topography to the scene to see the differences.

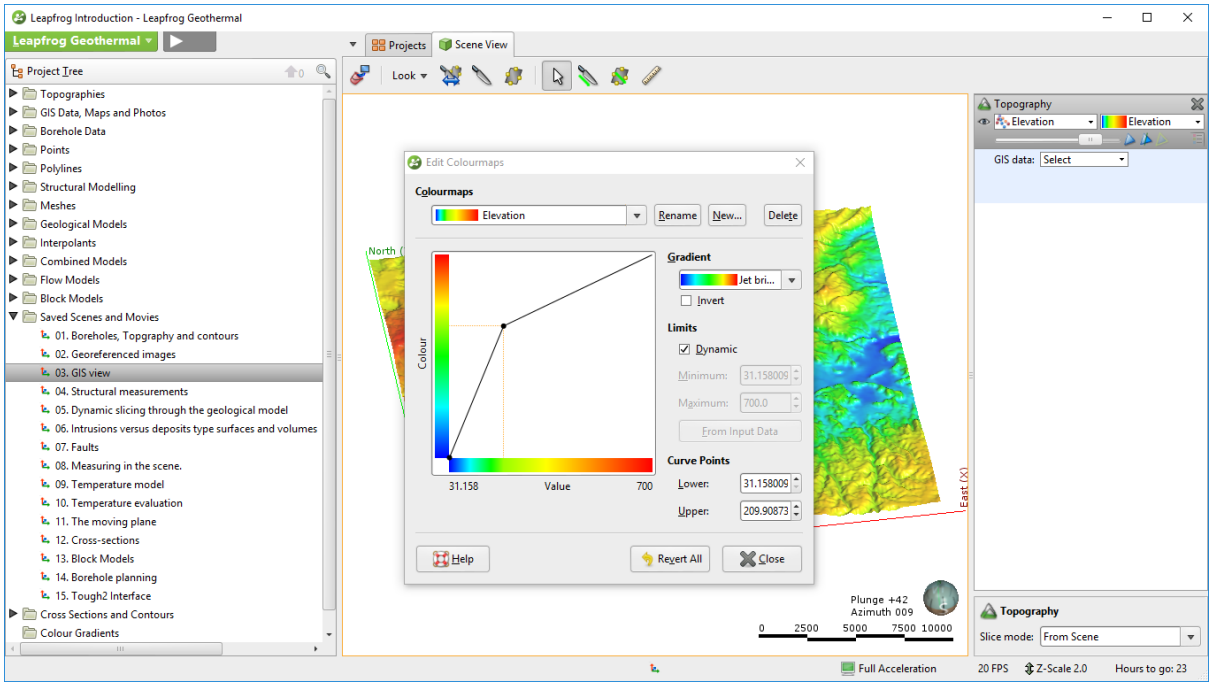



As you can see, the maps have been loaded at different elevations to differentiate them in the 3D Viewer. They are not accurately positioned in the 3D space.

## ELEVATION

- Display scene **03. GIS views**.
- Look at the property icons available for the **Topography** object and test them.
- Change the colour list of the **Topography** object to **Elevation** and set the GIS data menu to **None**.
- Click on the coloured bar and **Edit Colourmaps**.



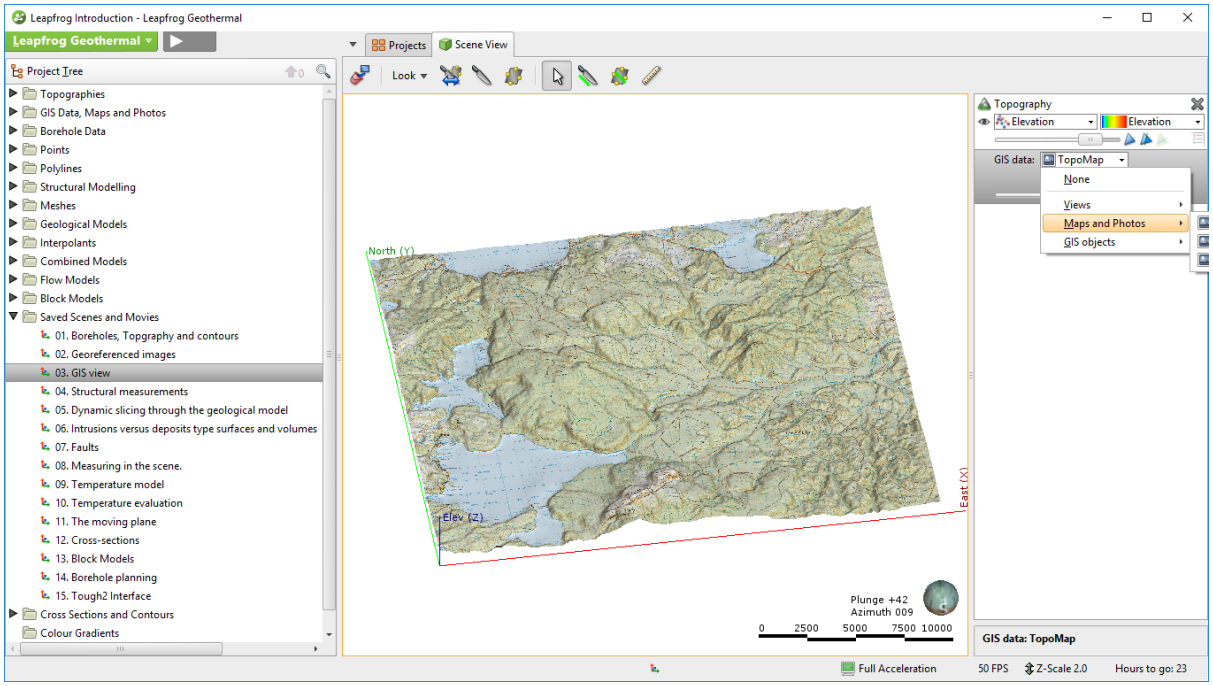



## DRAPING AN IMAGE

One easy way to view the maps in their correct position is to combine them with the Topography object so that it can be “draped” automatically over the terrain.

- In the Scene List, the **Topography** object has a **GIS data** menu, select the **TopoMap** from the **Maps and Photos** option.

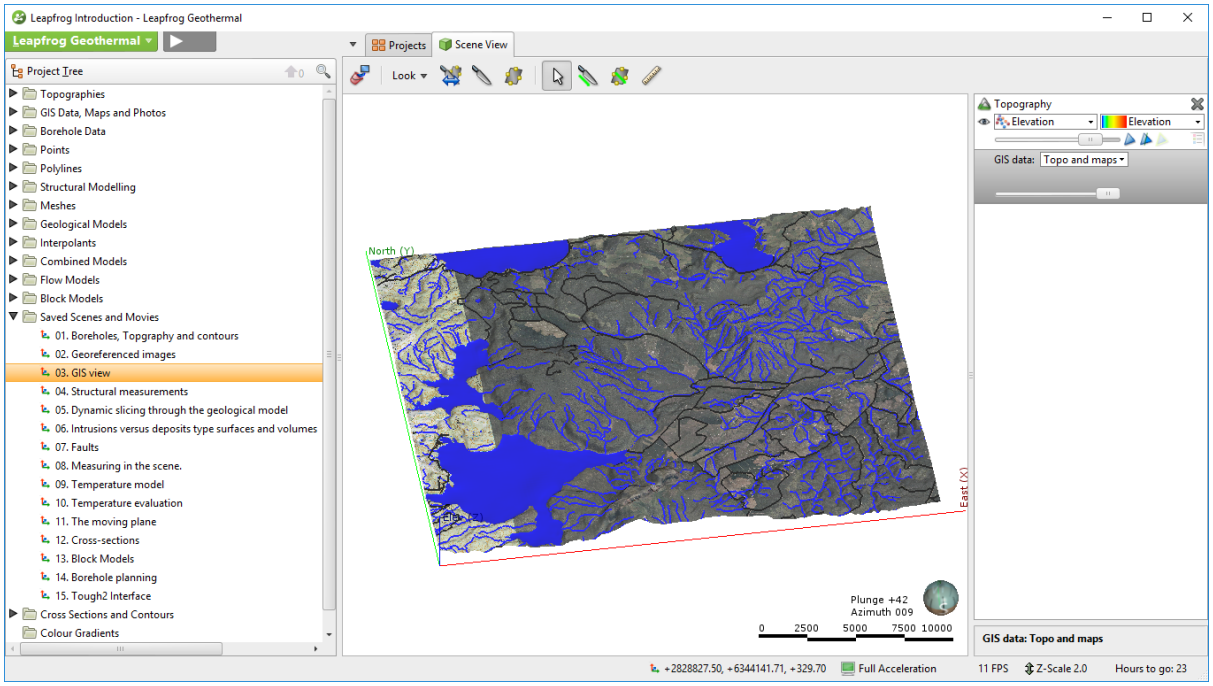
As you can see, the topographic map is now draped over the Terrain. From this menu you have access to all the maps and GIS layers in the project. However you can only display one object at a time on the topography.



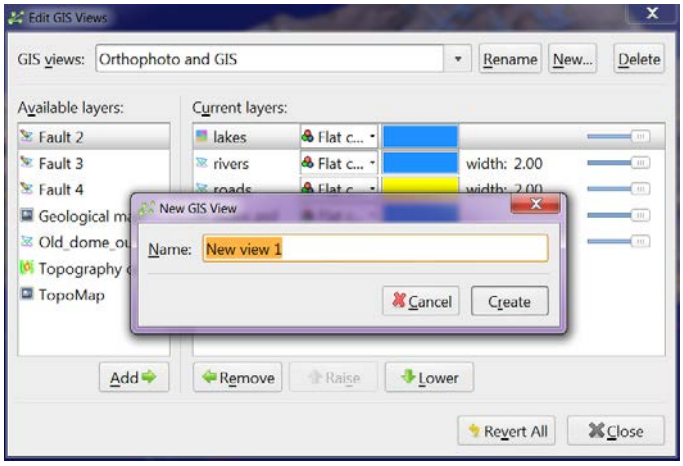
## CREATING VIEWS

It is possible to combine GIS objects and maps into a single layer or theme. This is currently done through the scene list and is referred to as a "View".

- In the **GIS data** menu, select the view called **Topo and maps** and observe the data. As you can see the rivers, lakes and roads have been draped over the Terrain as well as the **Orthophoto** and the **TopoMap** underneath.




- In the GIS menu, select **Views/Edit Views.** Click on **New.**
- Give it a name.




- On the left of the window, there is a list of available layers: maps and GIS data. Select the layers you would like to add to the view and click on **Add** (or double-click on the layer). The layers will appear in the **Current layers** list.
- You can modify the display settings of each object in the view.
- Click on **Close.**
- In the **GIS data** menu of the Scene List, select the view you have just created.

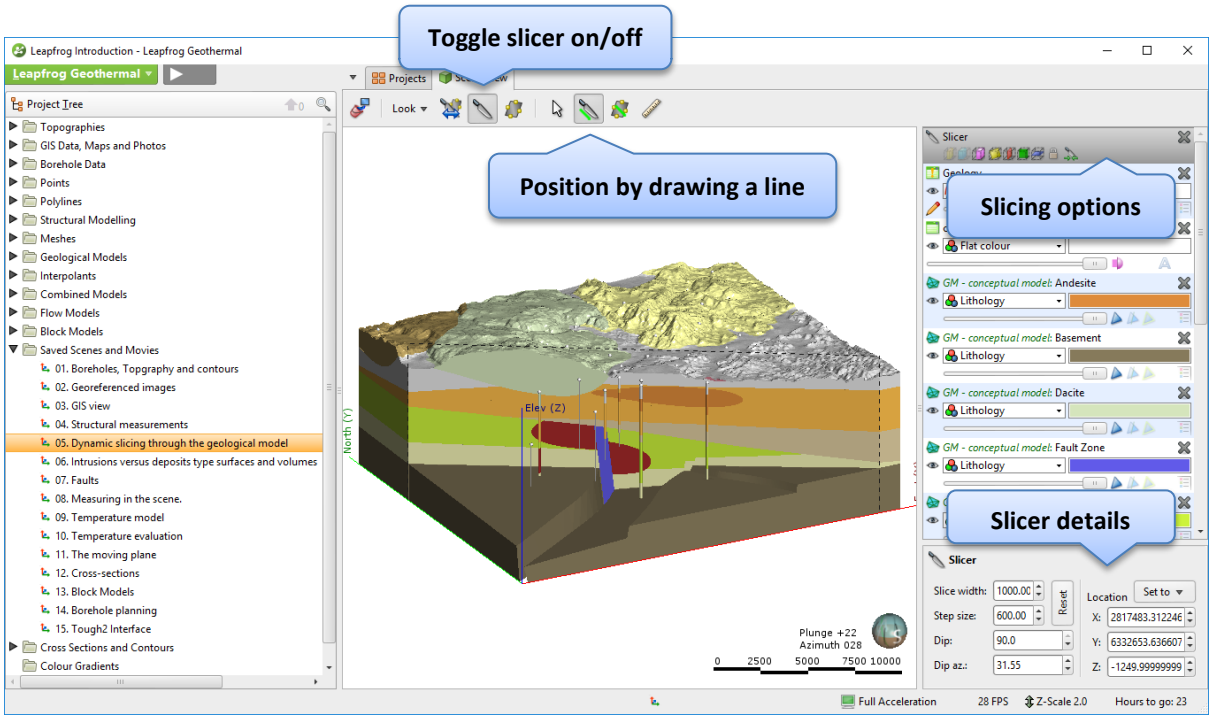
## SLICING THROUGH THE DATA

Slicing is a powerful tool for revealing data that is concealed behind other objects, or for emphasizing data. It can be considered as a form of dynamic sectioning.

The slicer can be turned on in two ways:

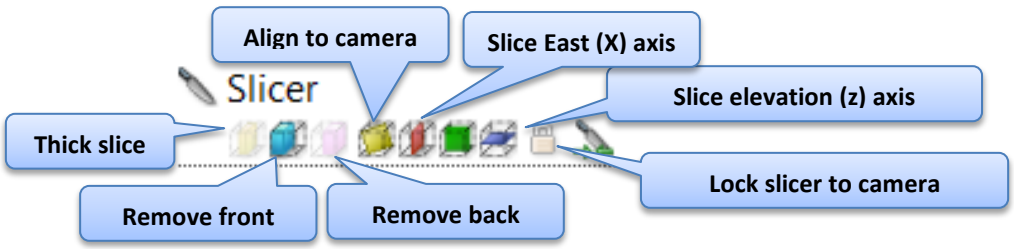
- A toggle switch which turns the slicer on and off
- A line which effectively "cuts" the scene

- Display scene **05. Dynamic slicing through the geological model**.




*The slicing is controllable in several ways:*

- *via the **keyboard** using "," and "." to step Forward and Backward.*
- *via the **keyboard** using <CTRL> + <E> to slice East axis, or <CTRL>+<N> to slice North axis.*
- *via the **mouse**: Holding down the <CTRL> key on the keyboard and dragging the right mouse moves the slicing plane in a direction orthogonal to the slicing plane. Holding down the <CTRL> key on the keyboard and dragging the middle mouse (both buttons together on a two button mouse) changes the thickness of a thick slice.*
- *via the **scene list** as shown in the following images.*



*When creating a vertical slice, to mimic a 2D cross section, we often want to look perpendicular to the section. Use the "L" or "SHIFT + L" shortcuts to visualize one side of the slicer, or the other side. Some easy way to check your slice is vertical:*

- *Control that your slicer dip angle is set to 90°.*
- *If you have the grid overlay activated, when looking straight into your slicer, you should see the Z axis grid appearing.*

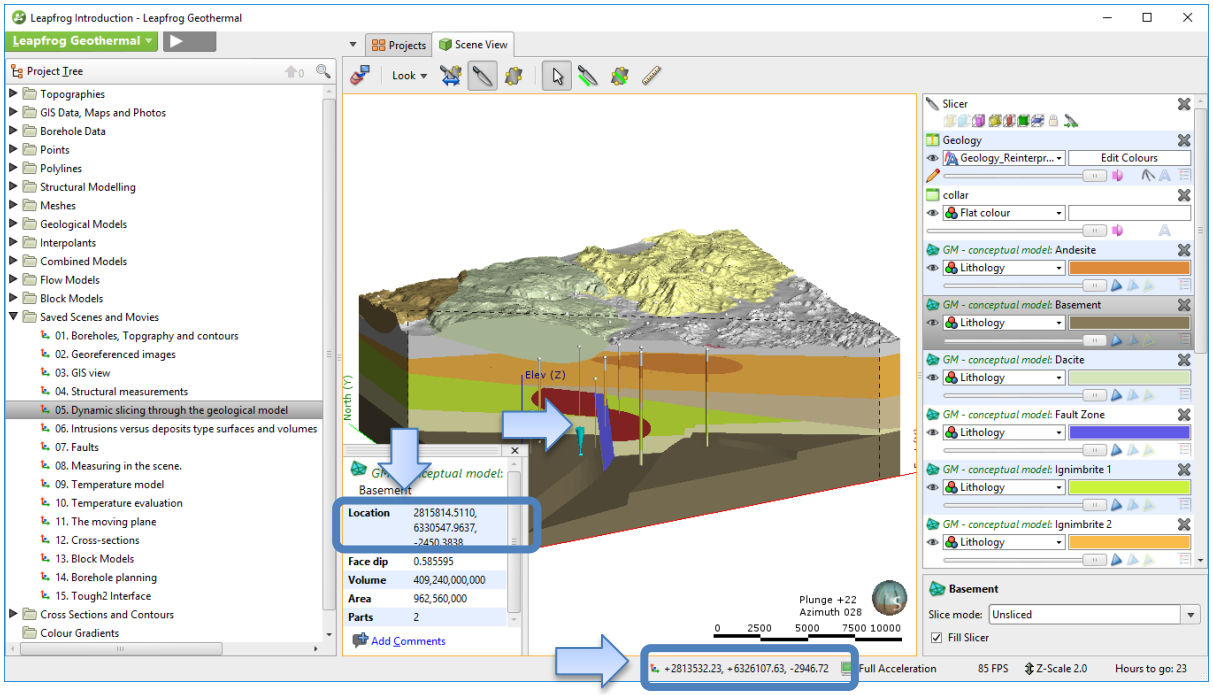
- Use the toolbar to remove the slicer, and re-position it.
- Take some time to look at the icons of the slicer and their properties. Modify the parameters to familiarize yourself with them.
- Change the step size. This corresponds to the distance moved for a single key press, and is also used to control the speed with which the slicer moves when the cursor is dragged.
- On occasion the scene may appear empty because the slicer has removed all the visible data. Either reposition the slicer or turn it off.
- Change the slicer properties of individual objects (e.g., **Basement**).

*Note: If the slicer moves as you are rotating the view it may be that you have locked the slicer to the camera view. Untick the padlock icon.*

## LOCATING A 3D-POINT

The simplest way of getting an approximate location in the scene is to make sure the object of interest is visible in the 3D-viewer and position the cursor over the location. The position is dynamically updated in the toolbar at the bottom of the window. This is calculated using the depth buffer on the graphics card and therefore is not exact.

If an exact position is required the user should click on the object. This causes Leapfrog to compute the exact position from the object, display it in a pop-up window and place a point in the scene to confirm the exact location.



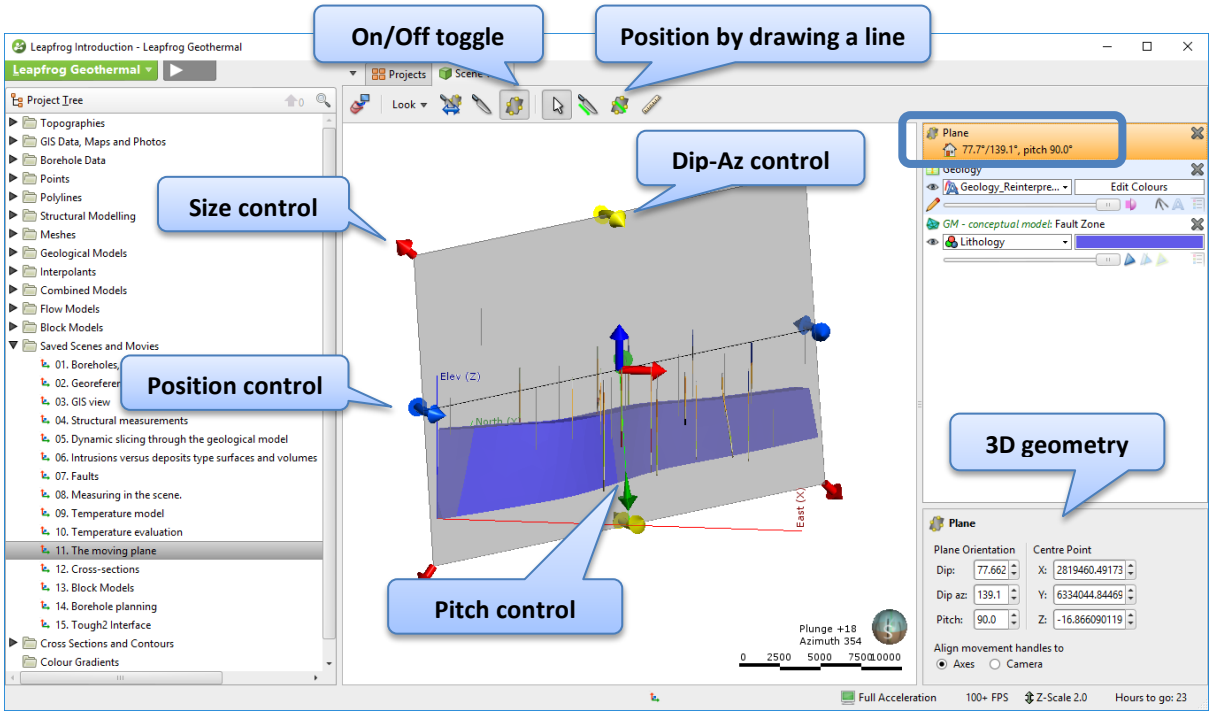

## THE MOVING PLANE

The moving plane is the basic tool within Leapfrog for defining trends and measuring planar structures.

- Open the saved scene **11. The moving plane** from the Demonstration project.
- Click on the plane, observe its properties.

The three axes are defined as follows:

- The first axis is along the green line in the moving plane.
- The second axis is defined to be in the plane and orthogonal to the green line.
- The third axis is perpendicular to the plane.




The plane can be positioned in 3 ways:

- User defined numerical input. Here the user types in a location, dip, strike and pitch.
- The plane can be manually adjusted using handles in the scene. These appear when clicking on the moving plane in the scene.
- The user can set the 3D Viewer to look parallel to the desired plane position and draw a line. This is usually the most convenient method of defining a planar structure.

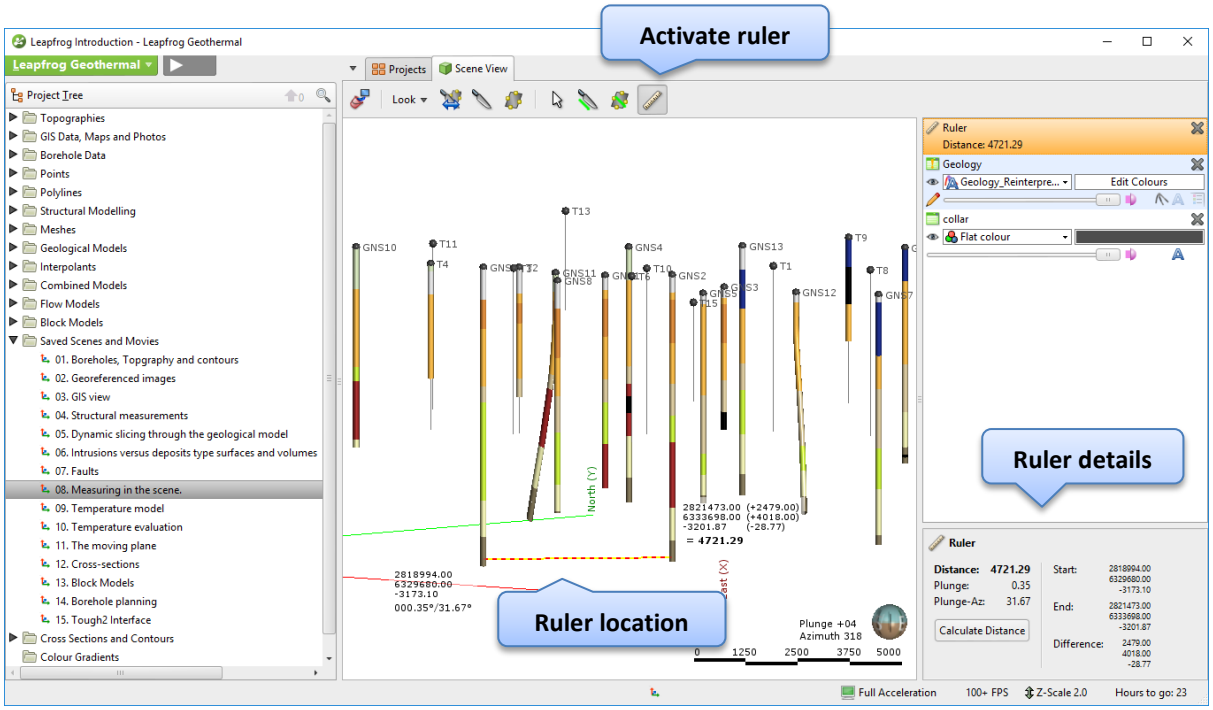
- Practice moving the plane to different positions.

## THE RULER

The ruler is a tool that can be used to measure distances within a three dimensional space.

The measurement is taken by clicking and dragging the ruler between two objects in the scene. When there is a valid measurement the dragged line changes to yellow/red dashes and remains in the scene after the user releases the mouse button.

- Open scene **08. Measuring in the Scene** in the demonstration project.
- Practice making some measurements.

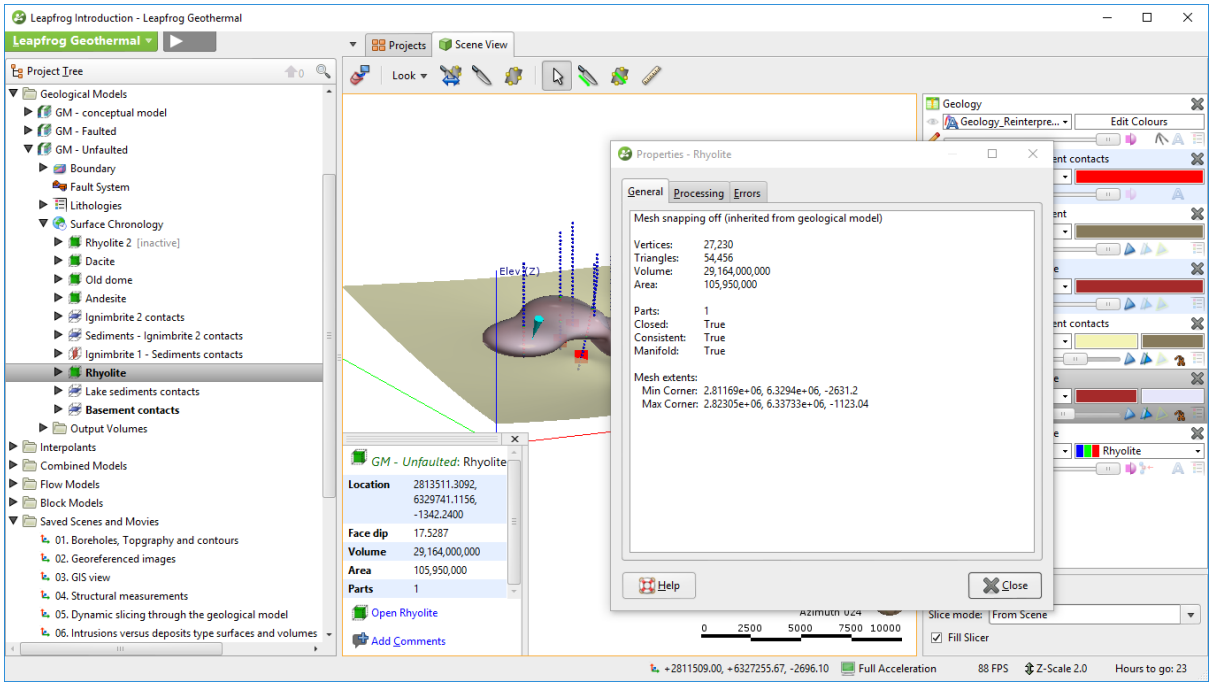



*Important Note: It is recommended to change the viewing angle after making a measurement to control the location of the ruler. It can be easier to use the ruler over a slicer or a moving plane for better accuracy.*

## OBJECT DIMENSIONS

Information on any object can be obtained by right-clicking on this object in the project tree and selecting properties. Alternatively a subset of the data can be obtained by clicking on an object in the scene.

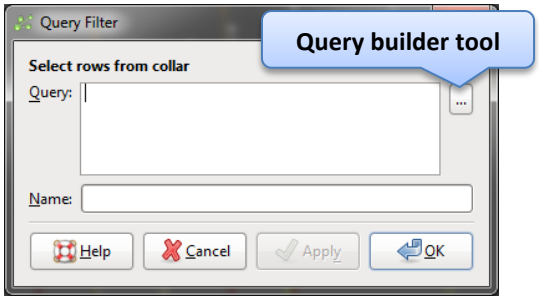
- Open scene **06. Intrusions versus deposits type surfaces** from the demonstration project.
- Click on the Rhyolite surface. A new window appears with some details, including the volume of the formation.
- Right-click on the **Rhyolite** object in the scene list and select **Go to the Project Tree**.
- In the project tree, right-click on the object and select P**roperties**. You will also find volumetric information.




## QUERYING DATA

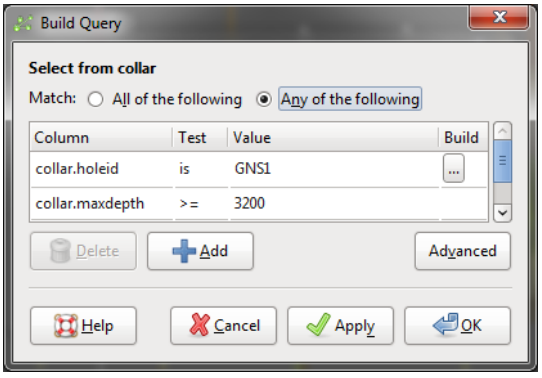
Leapfrog allows you to create queries on objects to visualize and/or model a subset of the original dataset. Queries are SQL based and a Query builder tool is available to help users generate their query in a straightforward way.

- Clear the scene.
- Drag the **collar** and **geology** objects in the 3D scene.
- Right-click on the **collar** object in the project tree.
- Select **New Query Filter** and a new window will appear. Click on the **Query Builder** to build a query.



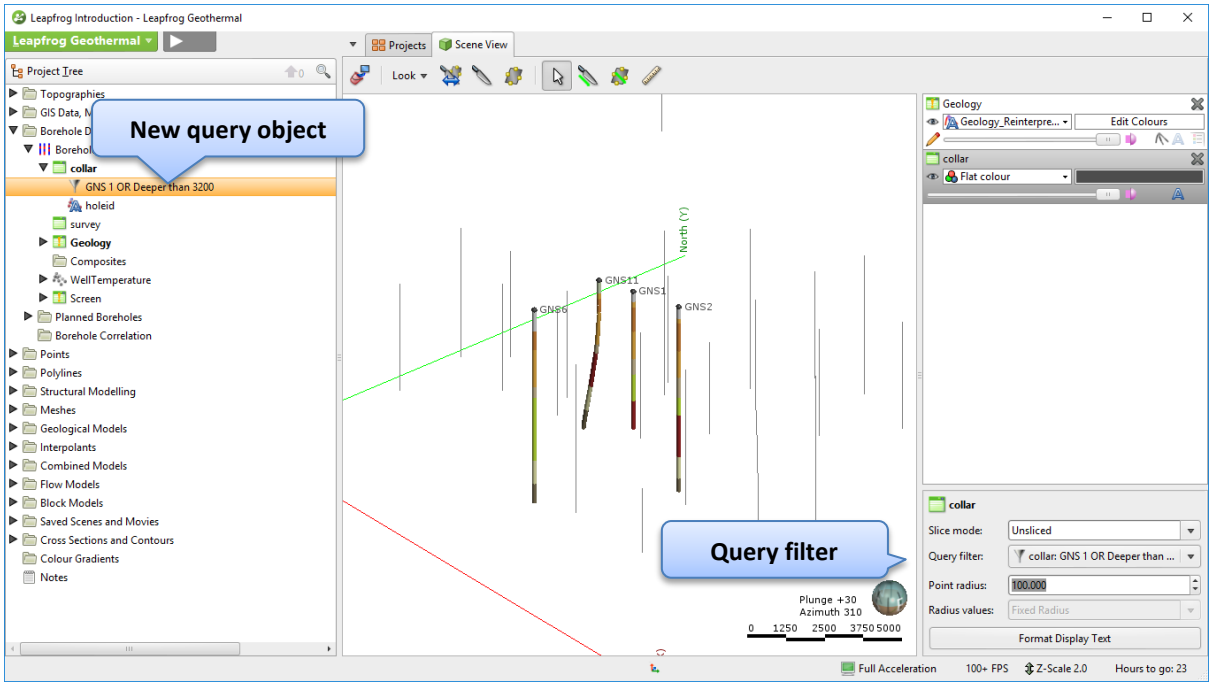

A new window appears to help you build your query.

- In the column window select the collar.holeid from the drop down list.
  - Select the Test as “is”.
  - Enter the value “GNS1” or click on the build icon to get the list of all available options.
- Follow the same procedure for a numeric value as per the example in the image (Collar max depth >=3200 m).
- Select the option “Any of the Following” which is an “OR” statement in SQL language.
- Enter a name for the query and click **OK**.



Under the Collar object in the project tree there is a new query object.

- Drag in the query object in the 3D scene.
- In the scene list property windows, the query filter is now activated and you can choose to use your query.
- You can also link the queries between tables:
    - Click on the **Geology** object in the scene list.
    - In the property panel there is a **Query filter** option: Select the query you have just created and observe the results in the 3D scene.




*If nothing displays on screen, it could be because the query entered has no solutions.*

## CONCLUSION

You are now familiar with the Leapfrog Geothermal visualisation tools and able to investigate an existing project. The following tutorials will show you how to import data and build your own models.

# INTRODUCTION TO LEAPFROG GEOTHERMAL

## TUTORIAL 2: SHARING INFORMATION

Through this tutorial we are looking at different ways of extracting model information and sharing it to promote communication and discussion.

## TABLE OF CONTENTS


Rendering an image ..... 2

Creating sections ..... 4

- Cross-section ..... 4
- Fence Section ..... 9
- Serial section ..... 11

Working with Scenes ..... 13

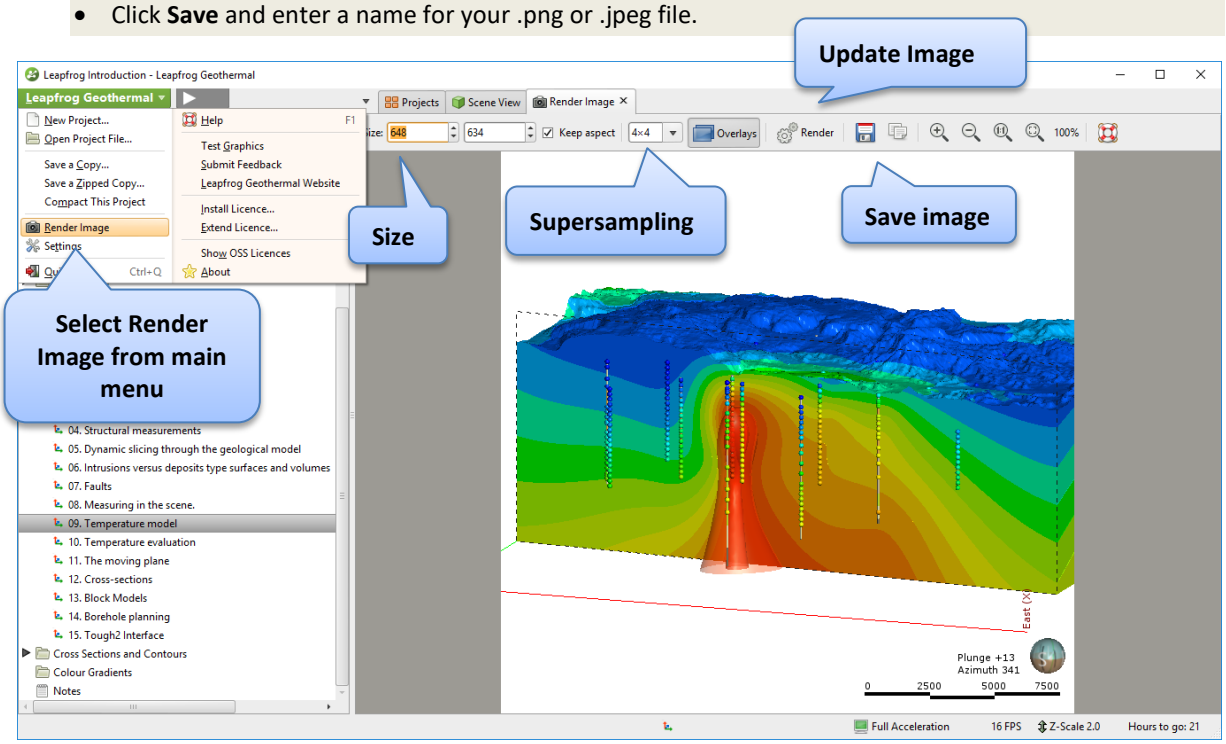
- Exporting scenes to the viewer ..... 13
- Exporting scenes into a movie ..... 15

Conclusion ..... 16

## RENDERING AN IMAGE

Leapfrog supports the ability to render high quality images, through the render image tool. Images can be exported at high resolution.

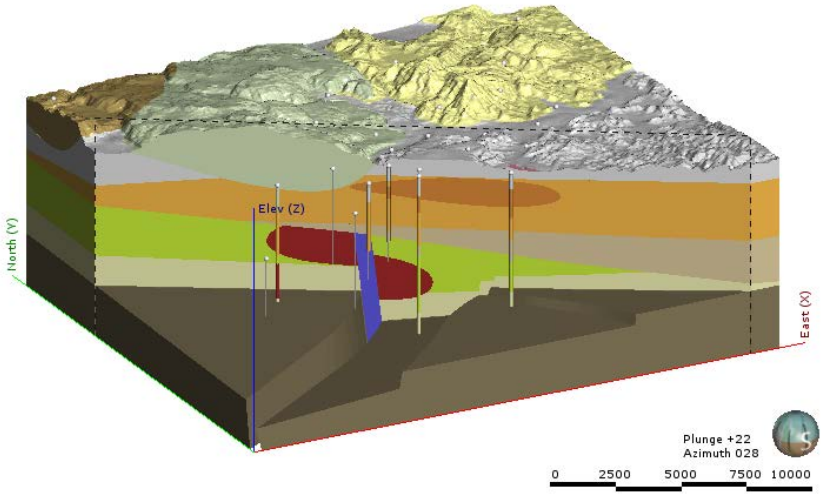
- Display a combination of data on screen. You can use one of the saved scenes.
- Click on the **Leapfrog Geothermal** menu.
- Select **Render Image**.
- A new **Render Image** tab appears.
- Click **Save**. You can save it as a JPEG or a PNG file.
- For better quality images, you can use the **Supersampling** option or increase the size of the image. You will need to click on **Render** again if you change any of the by default parameters.
- Click **Save** and enter a name for your .png or .jpeg file.

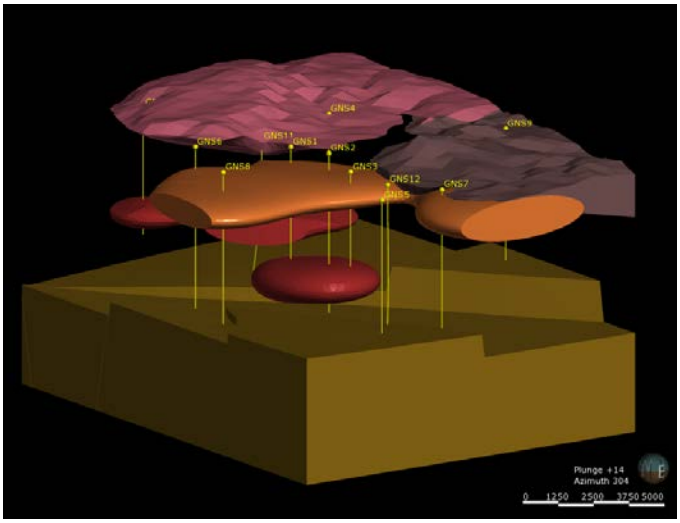


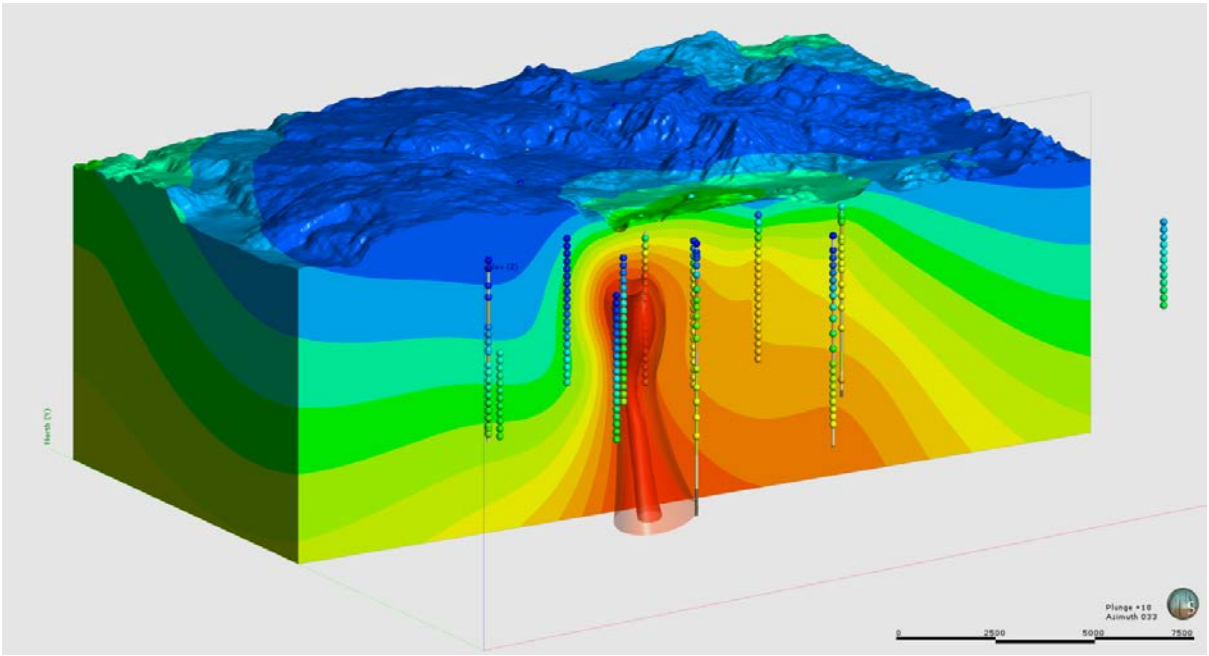

*Remember to:*

- *use the slicing properties of each object.*
- *use transparency.*
- *choose to display or mask the grid and/or axis lines.*
- *change the background colour to suit your purposes.*

Below are some examples of rendered images of the models we just created which could be used for various presentations:


Elev (Z)
North (Y)
East (X)
Plunge +22
Azimuth 028
0 2500 5000 7500 10000


GNS4
GNS9
GNS6
GNS11
GNS1
GNS2
GNS8
GNS3
GNS12
GNS7
GNS5
Plunge +14
Azimuth 304
0 1250 2500 3750 5000


North (Y)
Plunge +10
Azimuth 032
0 2500 5000 7500

## CROSS-SECTION

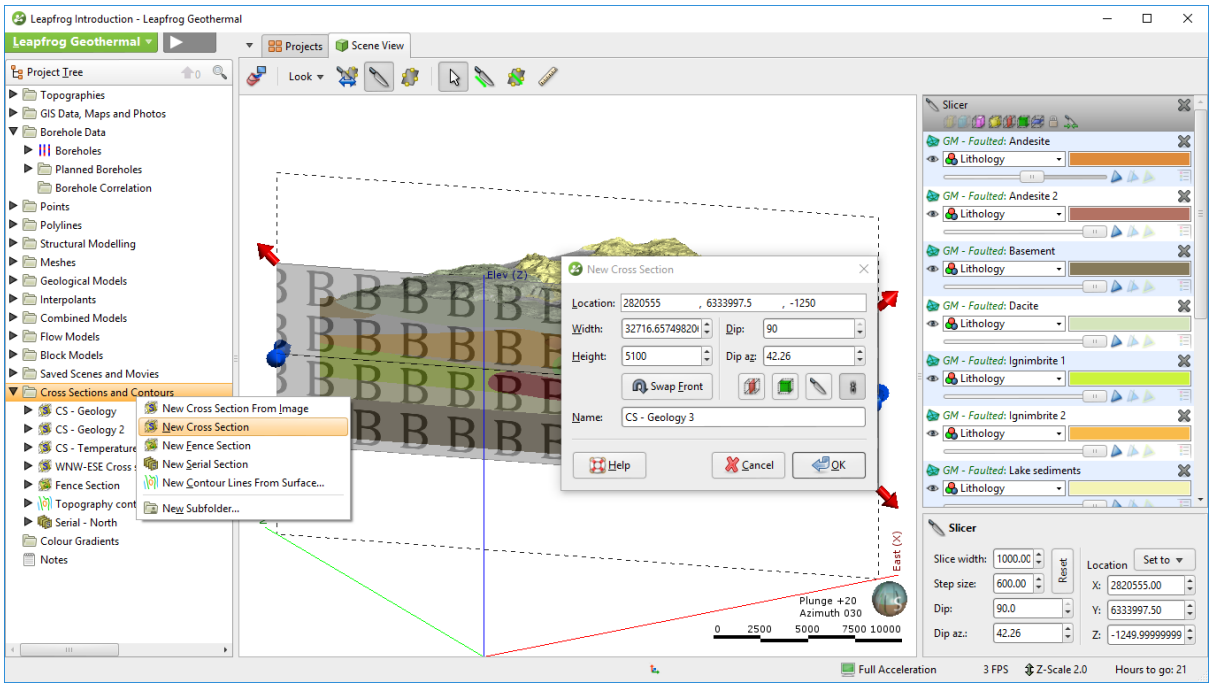
A cross-section is typically a 2D vertical plane representing the geology and/or other parameters. Leapfrog Geothermal provides the tools to automatically generate cross-sections in any direction/orientation.

There are two ways to create cross-sections:

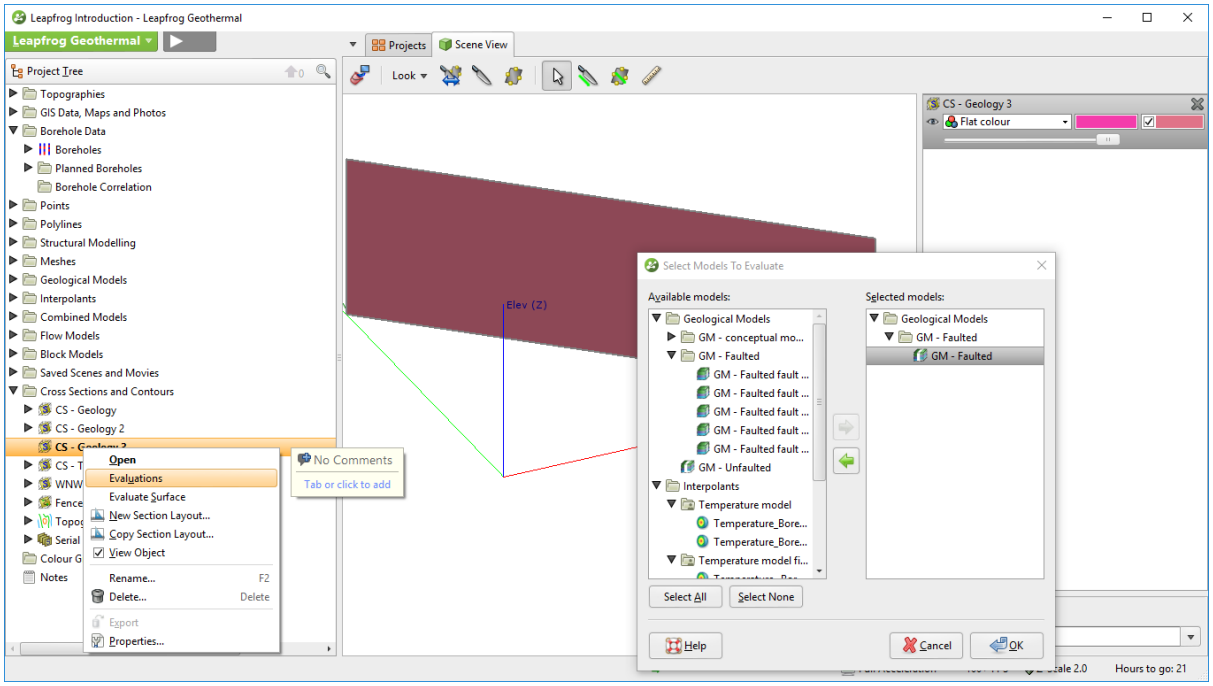
- You can render the image displayed in the 3D scene.
- You can use the cross-section tools imbedded within Leapfrog.

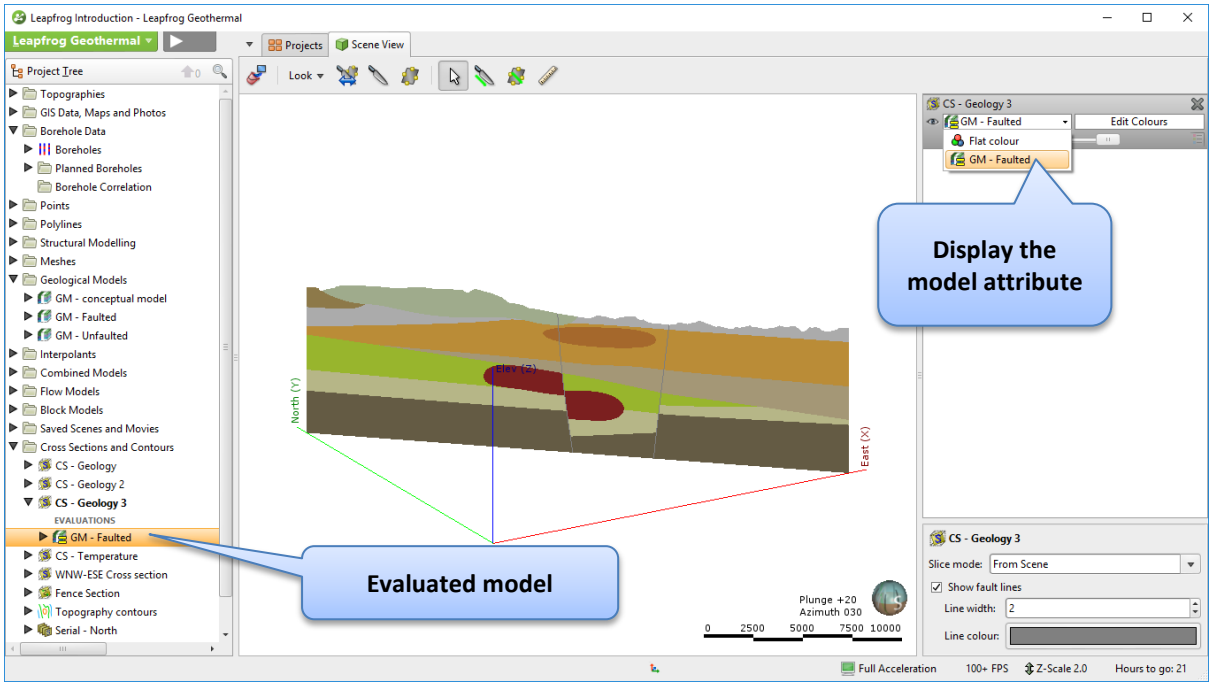
To create a quick section, use the slicer tool to generate a cross-section, and adjust the display effects before rendering the image.

- Display one of the model on screen. You can use of the saved scenes.
- Slice the model.
- Right-click on the **Cross Sections and Contours** folder in the **Project Tree**.
- Select **New Cross-section**.
- A **New Section** window appears.
- Type the **Section Name** and click **OK**.



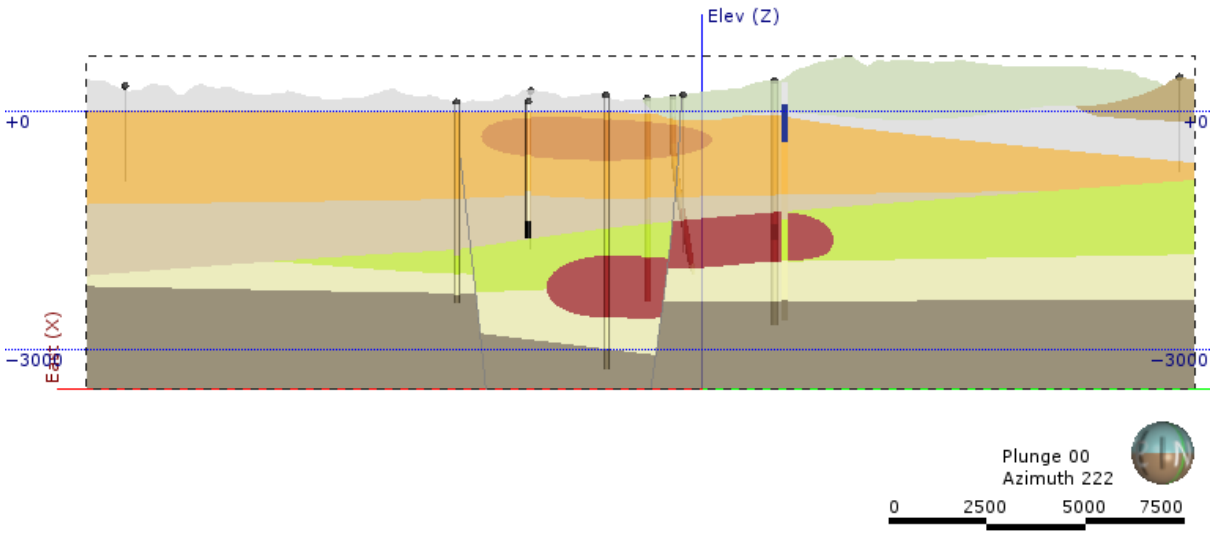

- The new object is currently displaying as a plain plane.
- Right-click on the object in the **Project Tree** and select evaluation. Select the model you wish to evaluate. For the geological model, make sure to evaluate the top level model) which is the full model).
- Click **OK**. Leapfrog will start evaluating the geological model against the cross-section.
- Once finished, you can now display the geological model attribute when displaying the new section.




- Modify your display properties until satisfied. For example, you can:
  - Set the slicer plane to the cross-section. Use “L” or “shift L” to set the view perpendicular to the section.
  - Display borehole data and change its properties so you can see the wells on each side of the cross-section: set the collar and geology to **Thick Slice**, and define the **Slice Width** to 5000 m (that means you will display all the wells within 2500 m on each side of the slicer plane).
  - Add some transparency on the cross-section to see-through.
  - Display the grid if you want to have a scale reference.

- Use the **Rendering** tool to export an image.



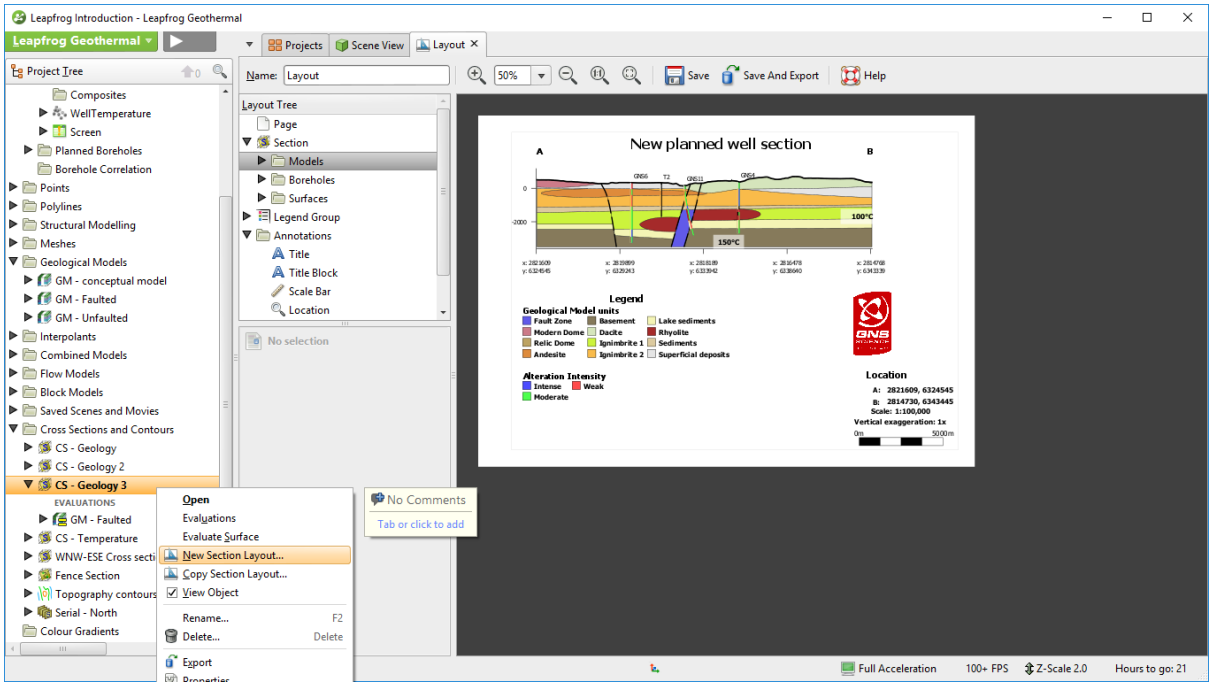

For a more professional rendering, use the cross-section export tool imbedded in Leapfrog Geothermal.

- Right-click on the **cross-section** you have just created and select **New Section Layout**.
- A new tab opens with many options you can modify to create a complete and well-designed image output. Take some time to browse through different menu to look at the available options.

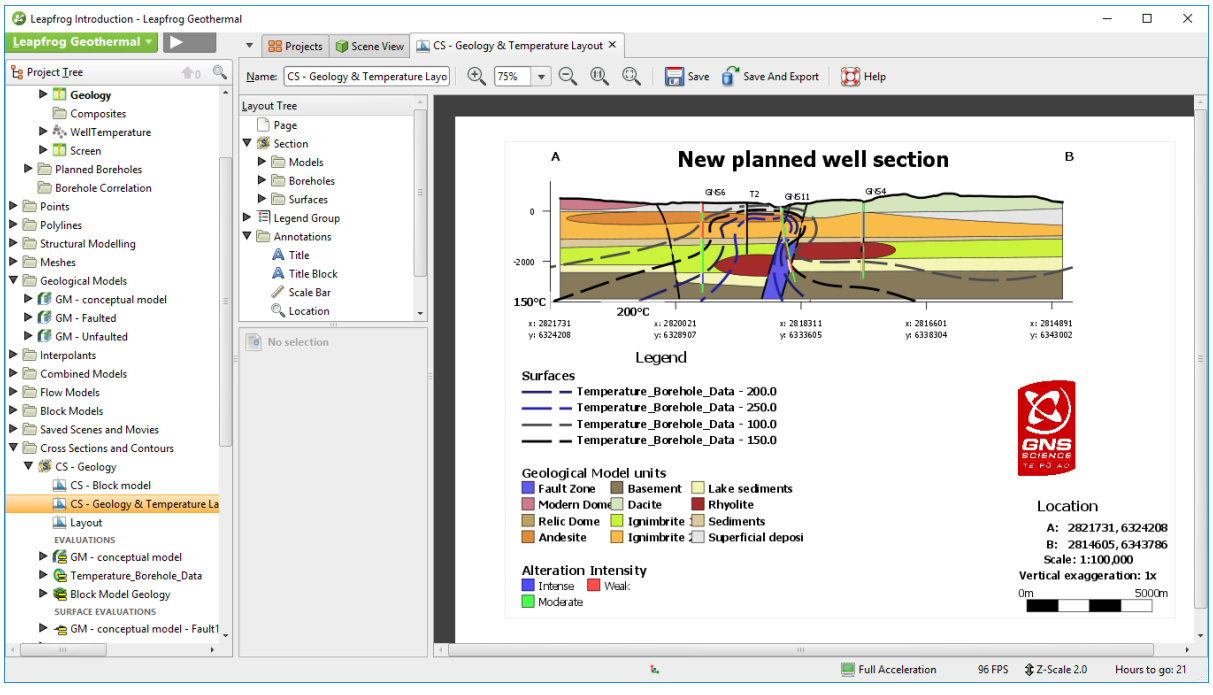
*Notes:*

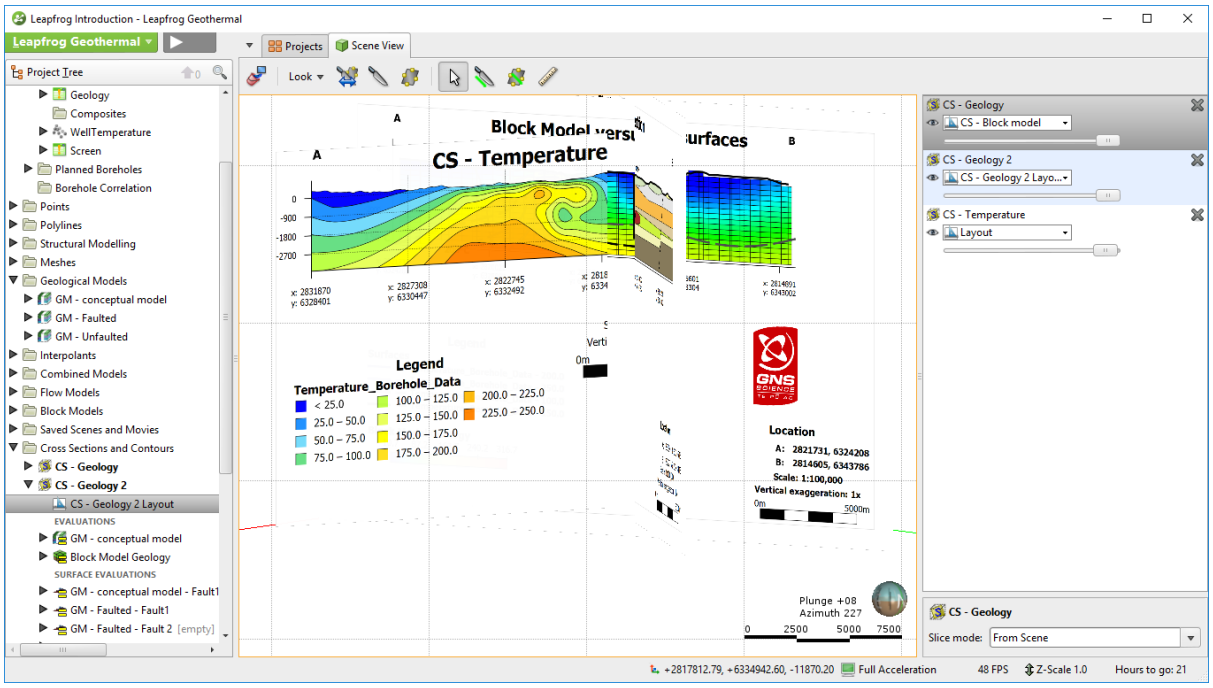
- *To be able to add more information, you first need to evaluate the data (e.g., models and/or surfaces) from the Project Tree.*
- *You can project your borehole on the cross-section. Leapfrog Geothermal gives you a list of the boreholes and their distance to the cross-section. You can also display multiple attributes along the borehole traces (e.g., on the left display, the geology, on the right, the alteration).*
- *You can add annotation and images (e.g., Logo) to complement your image.*

- Once you are satisfied with your preview, click **Save** or **Save & Export**.
- You can export your layout as SVG, PDF, PNG or GeoTiff. The PDF and SVG files can be edited for any final touch up you may require. The GeoTiff option exports a georeferenced image.




- Create a new section at a different location. Copy the layout from the original one (copy Layout option from the Project Tree). In a matter of seconds, you can now create many cross-sections.
- Create a new section and/or layout including your numerical model data as well. You can add the iso-surfaces on top of your geology, or choose to display the temperature volumes instead.






You can export your cross-section as .dxf for use in other packages.

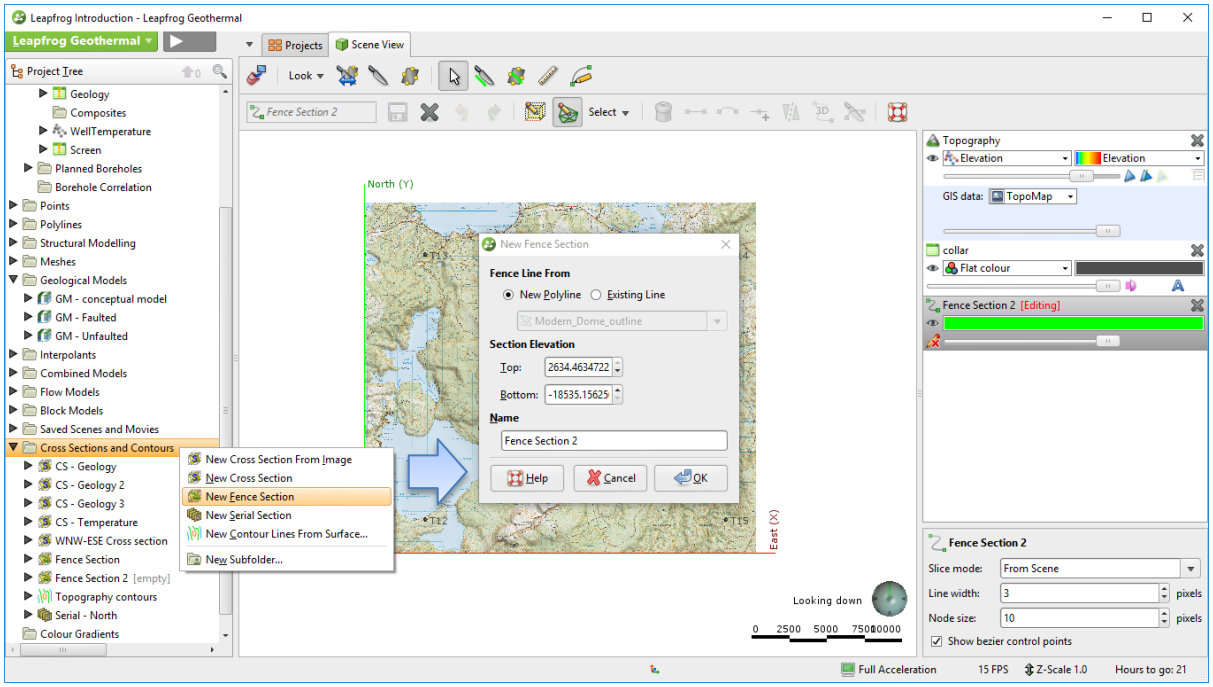
- Right-click on the cross-section in the Project Tree and select **Export**.

## FENCE SECTION

A Fence Section is a segmented cross-section. Its location is defined by drawing a polyline.

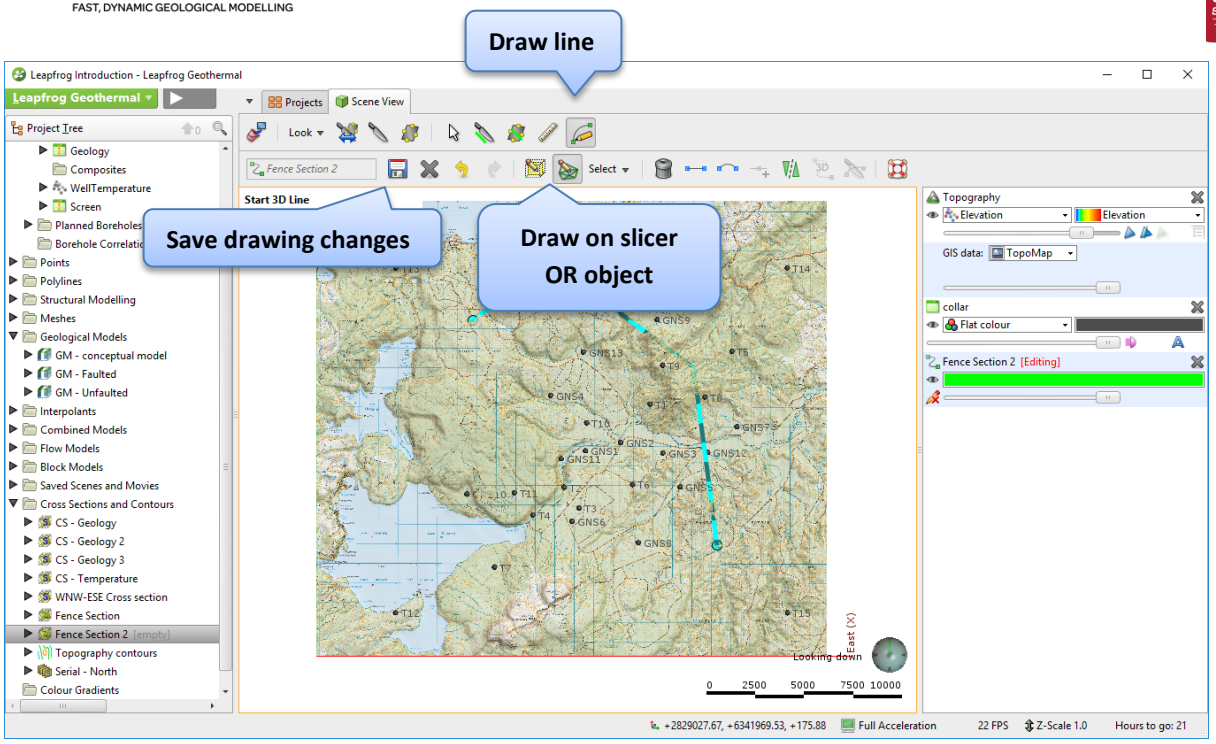
Create a Fence Section showing the geology expected along a predefined path:

- Display the **Topography** and the **collar** table.
- Right-click on the **Cross-Sections and Contours** folder.
- Select the N**ew Fence Section** option. A New Fence Section window appears.
- Select **Fence Line From/New Polyline**. An existing polyline might be used as well.
- Set the top and bottom depths.
- Give the fence section a **Name.**
- Click **OK.**




The drawing tools are activated, and a polyline can be drawn.

- Click on the **Draw lines** icon. By default you will be drawing on an object, in this case the topography layer. You can change to draw on a slicer if you need a 2D plane.
- Start drawing by clicking to add points. If you hold and drag your mouse you will create a tangent at the vertex of the polyline. Right-click to end the polyline, then click on the **Save** icon.




***Rules:***

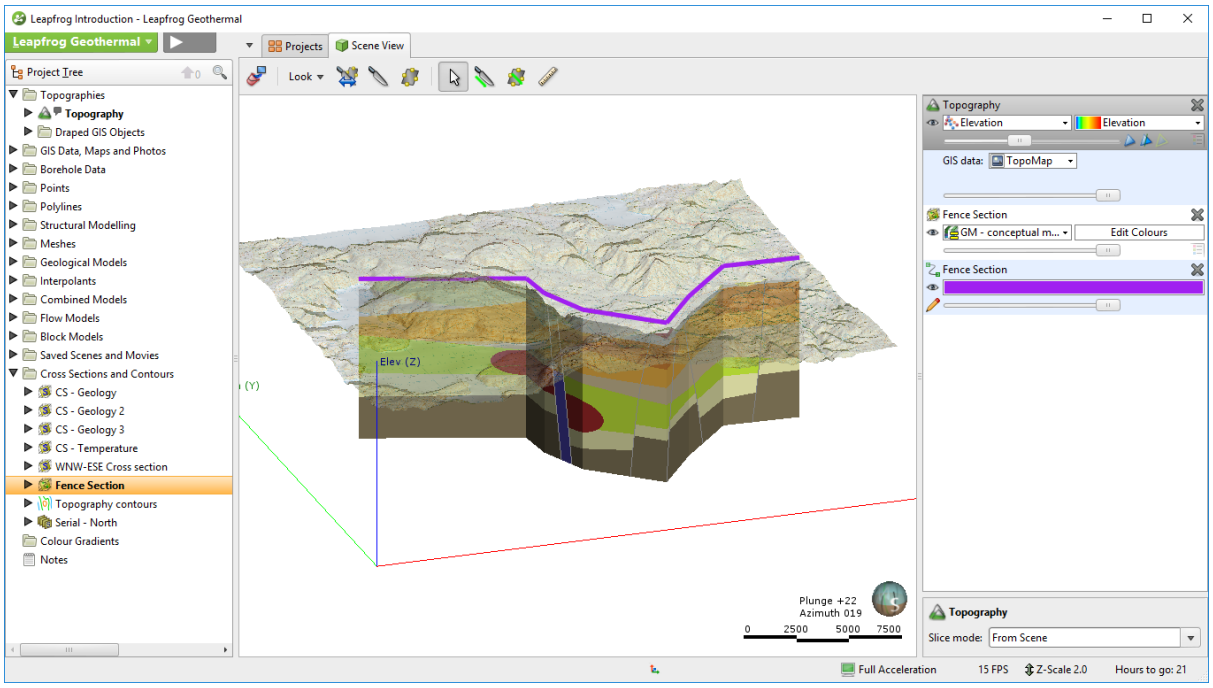
- *A Fence Section polyline cannot self-intersect.*
- *A Fence Section polyline must be made of a continuous single segment.*

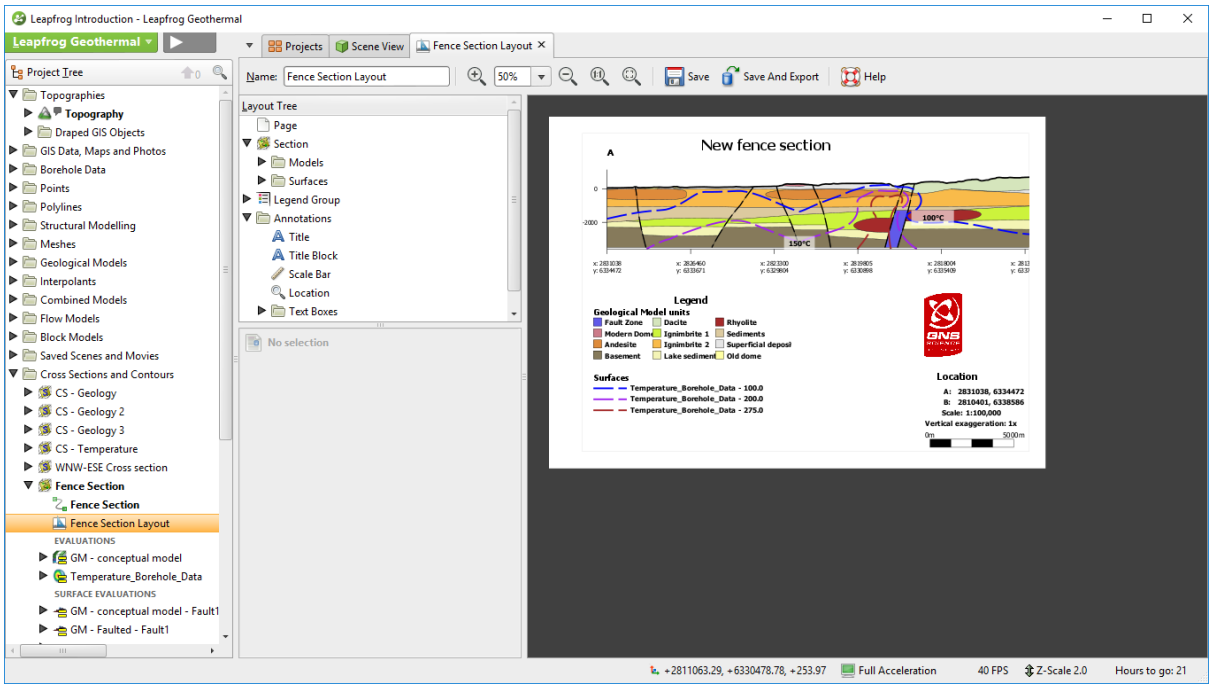
***Drawing tips:***

*Attempting to draw in 3D can lead to unpredictable results.*

- *If the polyline is drawn on the slicer plane, make sure to orient the slicer before starting to draw.*
- *Create vertices of the polylines by left clicks.*
- *Terminate the polyline with a right click.*
- *Modify vertices' locations by clicking on the point, press <CTRL> and drag the point to its new location.*
- *To add vertices, hold <CTRL>, click on a line segment and drag to a new point location.*
- *To delete a point, click on a point and press delete.*
- *To delete the entire polyline, double click on the polyline to select all the segments and vertices and delete.*
- *You need to click on the **Save** icon to save your edits.*

- Save the polyline and close the editor.
- A new section is available in the **Cross-Sections and Contours** folder.
- Right-click on the **new Fence Section** and select **Evaluations**.
- You can also use the **Layout** options as per a standard cross-section. The Fence Section is then unfolded.

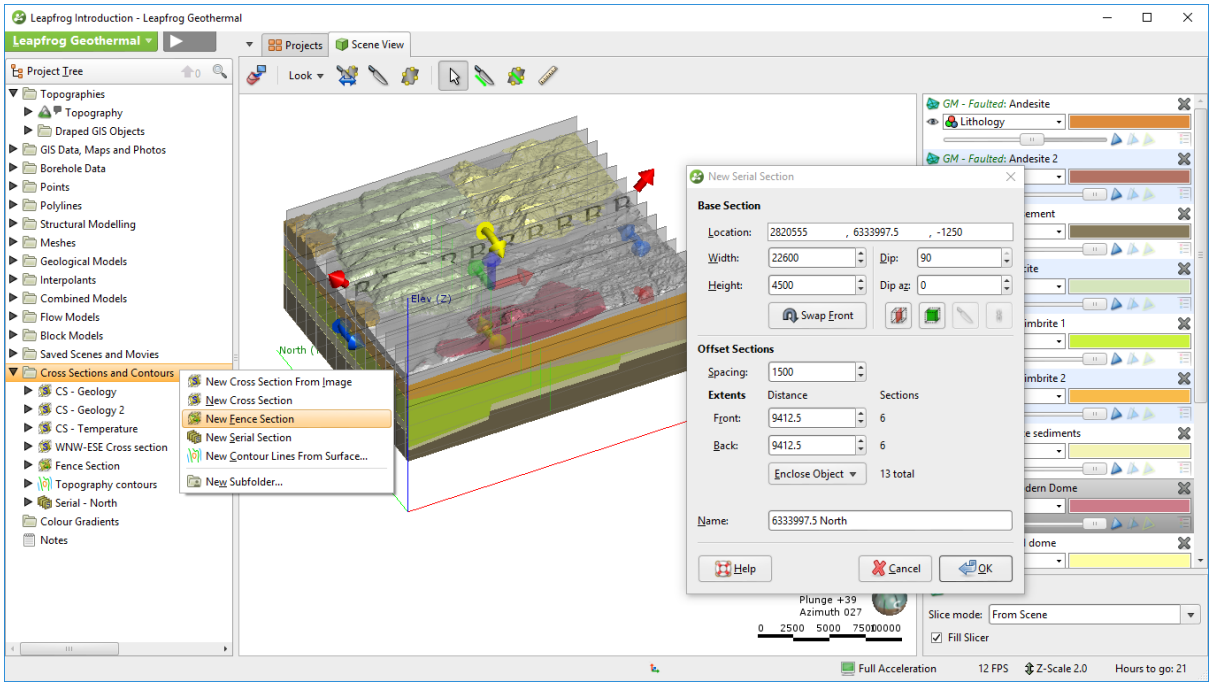


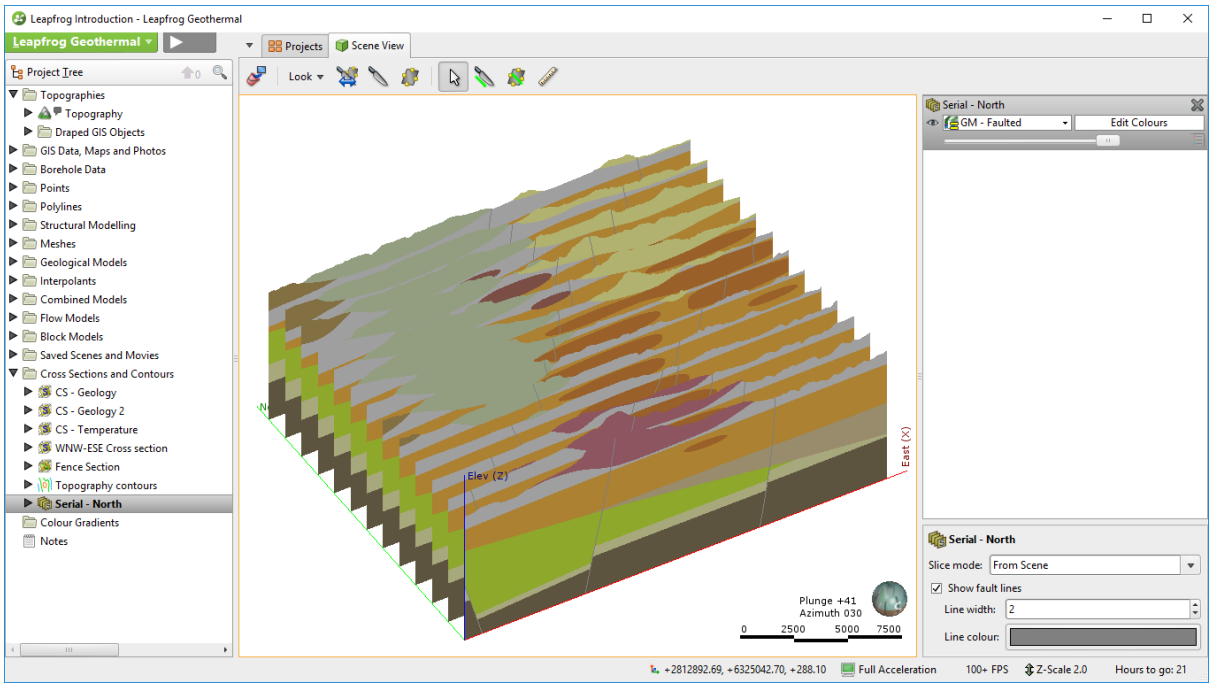

## SERIAL SECTION

This tool allows you to create automatically a series of parallel sections.

- Display a **model**.
- Right-click on the **Cross Sections and Contours** folder in the Project Tree. Select a **New Serial Section**.
- The sections will be defaults cover the extent of your model.
- Change the spacing to 1500 m.
- Enter a name and click **OK**.




- Right-Click on the **new Serial Section** icon to access the **Evaluations** options. It will be applied to all sections.
- Display the sections one by one, or drag the **Serial Section** icon into the 3-D scene for visualisation.
- You can then create new section layouts as for a normal section.

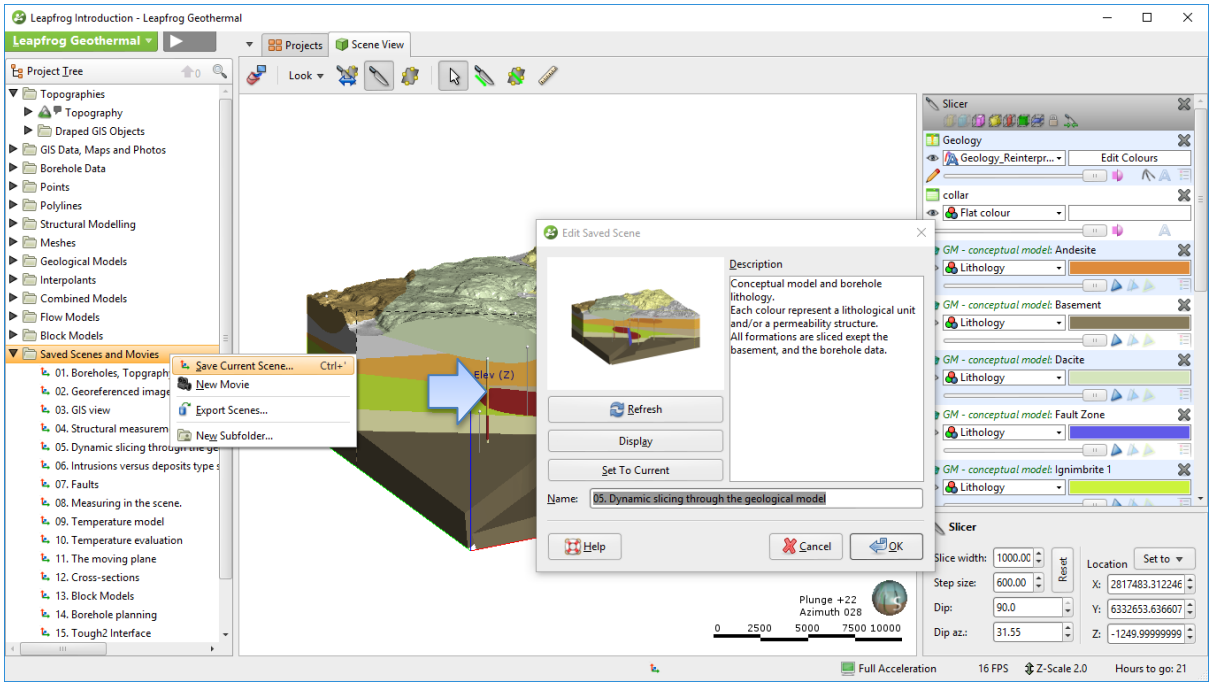


## EXPORTING SCENES TO THE VIEWER

Scenes are like bookmarks. Each scene records the scene list as well as the state of your 3D display window. It does not save the data itself. If a model is updated, the scene will include the updated model. It is very useful for providing a stable point of reference in a project and for explaining key aspects of the project.

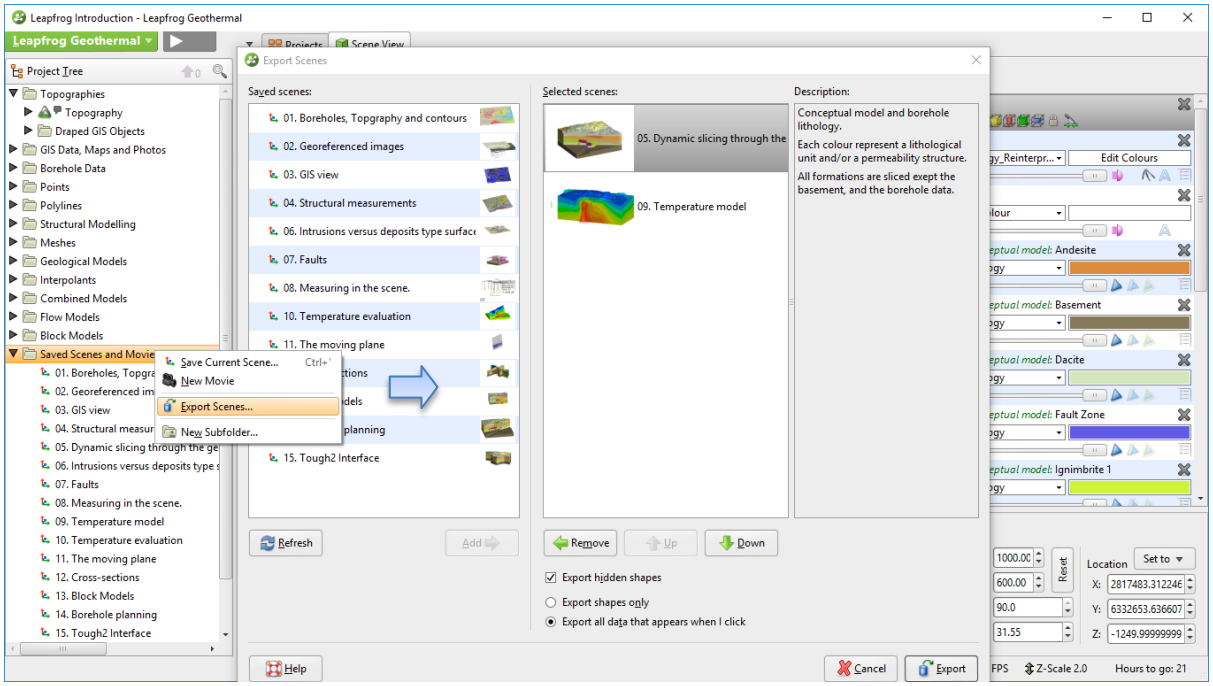
- The saved scene is one of the principal ways of documenting a view in the project.
- Saved scenes are used as a story board element for movie generation.
- Scenes can also be exported as a file for use in the free viewer.

- Display data/model of interest.
- Right-click on the **Saved Scenes and Movies** folder. Select **Save Current Scene**.
- Give the scene a name and click **OK.**

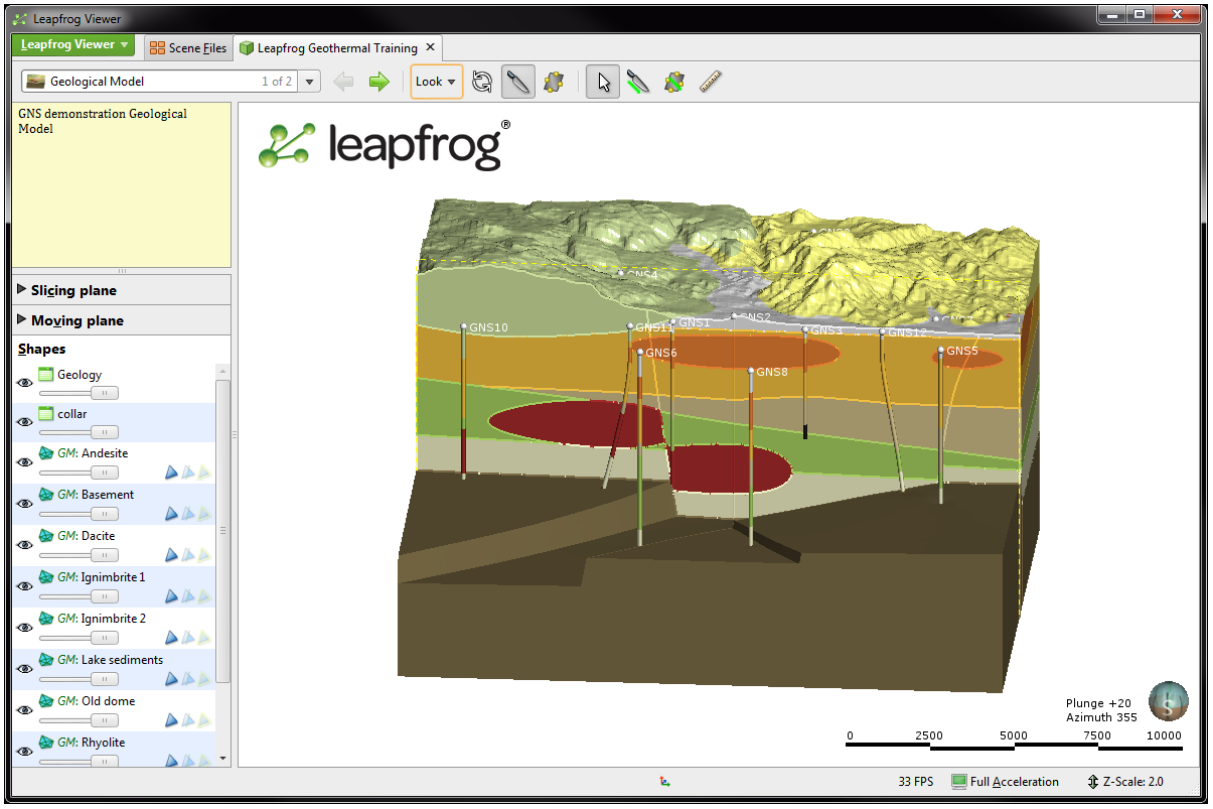



Once you have documented the scenes of interest, you can export them to the Leapfrog Viewer.

- Right-click on the **Saved Scenes and Movies** folder and select **Export Scenes**. A new window appears. Add the scenes you wish to export in the Selected Scene panel**.** If you have several scenes, an option to re-order them is available, using the **Up/Down** tools.
- Tick the **Export hidden shapes** box. This option exports layers in the scene list even if not turned on.
- Select **Export all data that appears when I click.** This option exports the data (e.g., the attributes/names) associated with each shape.
- Click **Export.**
- Select a location and give the file a name. It will save a .lfview file that can be opened with the Leapfrog Viewer.
- Open the new file. Take some time to browse through the viewer.




Now that you are familiar with Leapfrog Geothermal, using the Leapfrog Viewer will be really straightforward. It is a good way to share information in a true 3D interface with people that don't have access to the full software package. You can visualise the information provided, slice it, turn layers on and off, make measurements and draw moving planes.

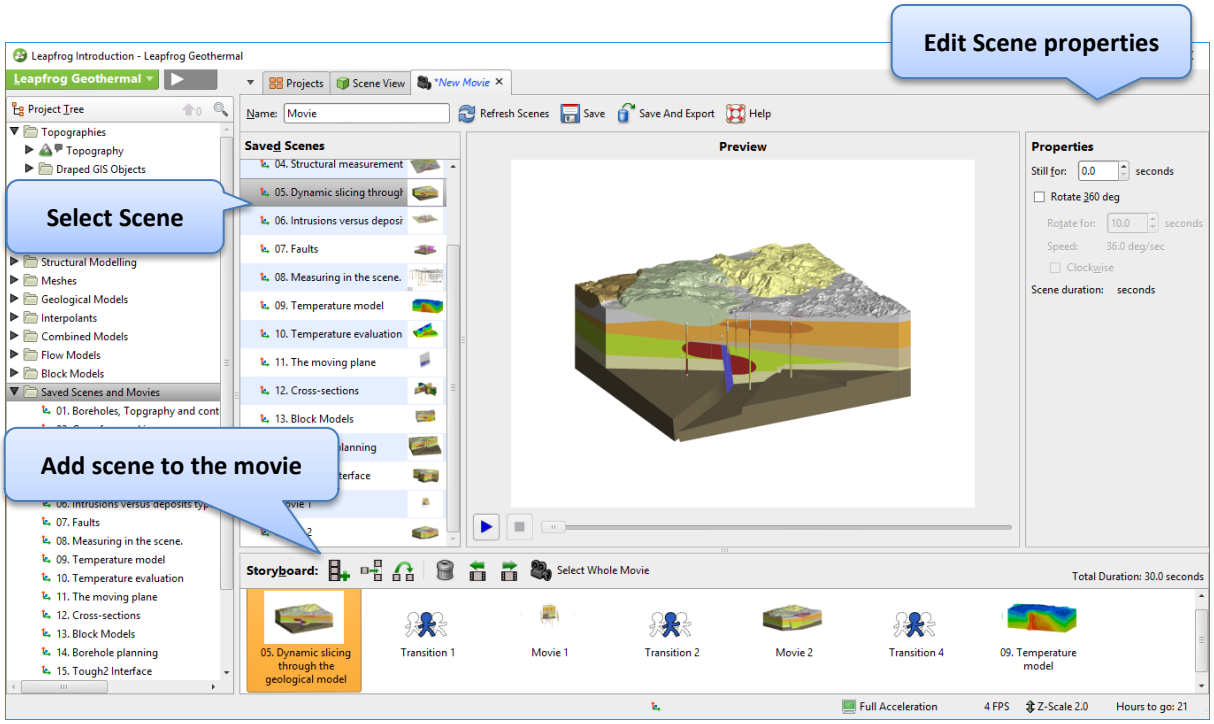



*The Leapfrog Viewer is free and can be downloaded on the Leapfrog website. It is an easy way to display and distribute 3D models for discussion and demonstration.*

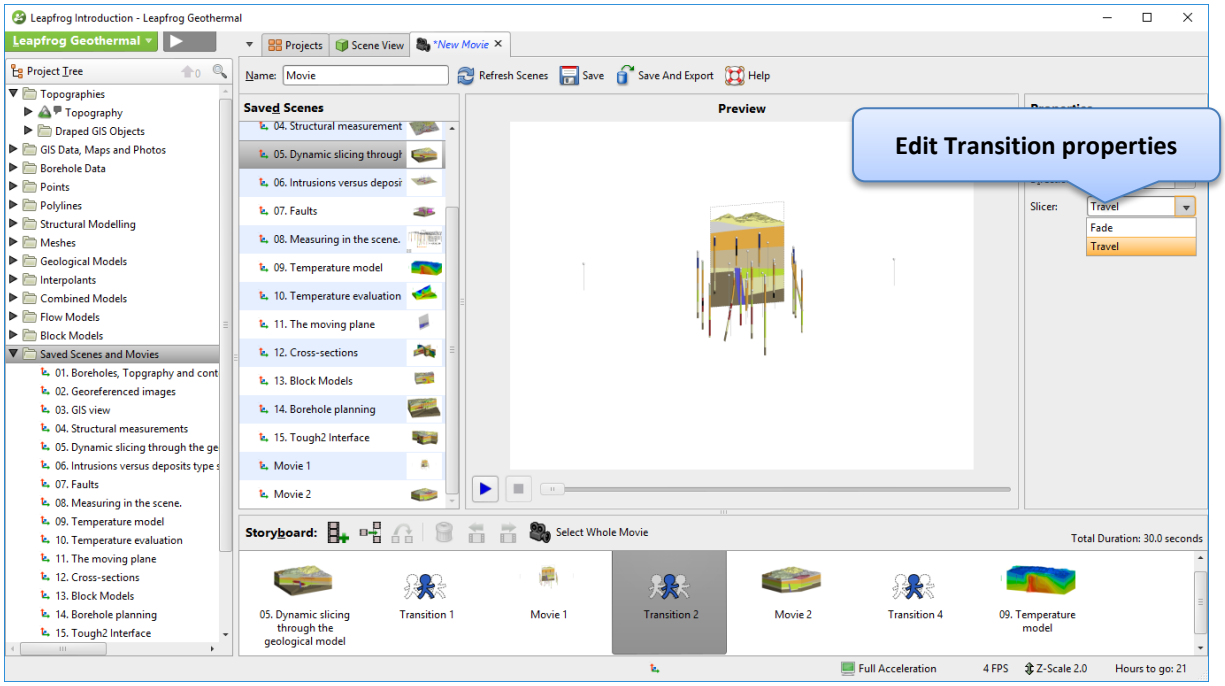
## EXPORTING SCENES INTO A MOVIE

Movie generation is easy and really impressive for demonstration purposes.

- Create two new scenes where the model is sliced in two parallel positions.
- Right-click on the **Saved Scenes and Movies** folder and select **New Movie**.
- In the **New Movie** tab select your scenes and click on **Add Scene** to storyboard.
- Click on the **Scene** in the storyboard and look at the options. Change the **Still for** option to 2 seconds, check the **Rotate 360 deg** box and **Rotate for** 4 seconds.




- Click on the **Transition** icon. Change the transition time to a **Duration** of 4 seconds. When looking at the Transition between 2 views where the slicer has been kept parallel, select the Slicer **Travel** option.
- Click on another scene and try a new combination of parameters. You can change the settings at a later stage to try different scenarios.
- Once satisfied with your settings click **Save and Export**. A new **Movie** object appears under the **Saved Scenes and Movies** folder.
- On the **Export Movie** window, accept the defaults settings and click on **Export** to save the movie.
- Once Leapfrog finishes rendering the images, select **Play Movie**. It will reproduce the movie in your default media player software. Leapfrog saves movies as .wmv files.




*Leapfrog Geothermal will create a movie going from Scene to Scene using smooth transition. It will automatically select the shortest angle between different views (it can be changed) and will progressively fade from one frame to the other. It can also progressively travel through a model when parallel views are set up.*

*Once satisfied with a movie, you can export to high resolution from the Export Movie window by increasing the Movie Size. It will take longer to process but will be high definition.*

## CONCLUSION

In this tutorial we have learnt about tools that facilitate sharing of information from a built model. You can now create 2-D, 3-D and dynamic outputs for your communication needs.

# INTRODUCTION TO LEAPFROG GEOTHERMAL: IMPORTING DATA

## TUTORIAL 3: IMPORTING DATA INTO LEAPFROG

The objective of this tutorial is to create a new Leapfrog Geothermal project and import different types of data, ready to enable subsequent modelling.

## TABLE OF CONTENTS


Creating a new Leapfrog Geothermal project ..... 2

Importing maps and aerial photographs ..... 2

Setting the region of interest ..... 6

Creating the Topography ..... 7

Importing GIS data ..... 9

Importing meshes ..... 10

Importing point data ..... 11

Importing structural data ..... 12

Importing borehole data ..... 14

- Importing collar, survey and iInterval tables ..... 14
- Importing point values ..... 18

Importing an existing section ..... 19

Managing data ..... 20

- Fixing errors ..... 20
- Grouping lithologies ..... 21
- Ignoring data ..... 23

Conclusion ..... 23

## CREATING A NEW LEAPFROG GEOTHERMAL PROJECT

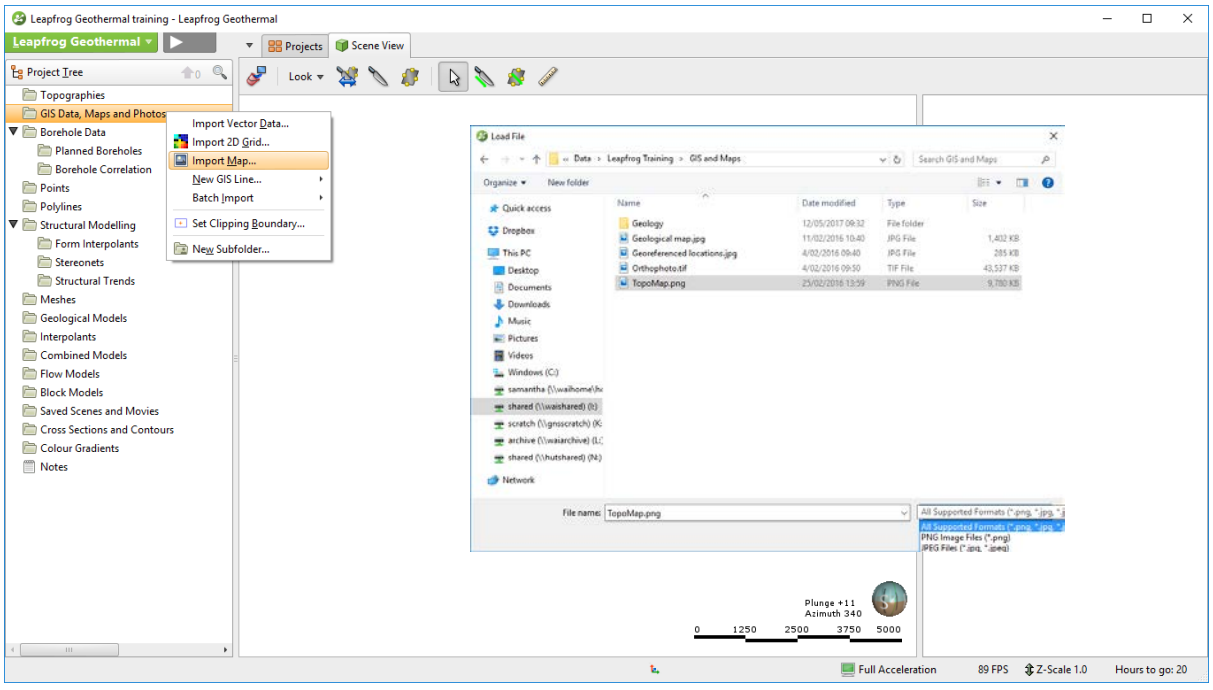
- Open **Leapfrog Geothermal 3.2**.
- In the Projects tab select **New Project**...
- Fill in the name of your new project and save it under the *Leapfrog Training\Project* folder.
- Select **OK.**

The Project Tree, Processing Tasks, Scene list and Visualisation panes appear in a new project, that is currently empty.

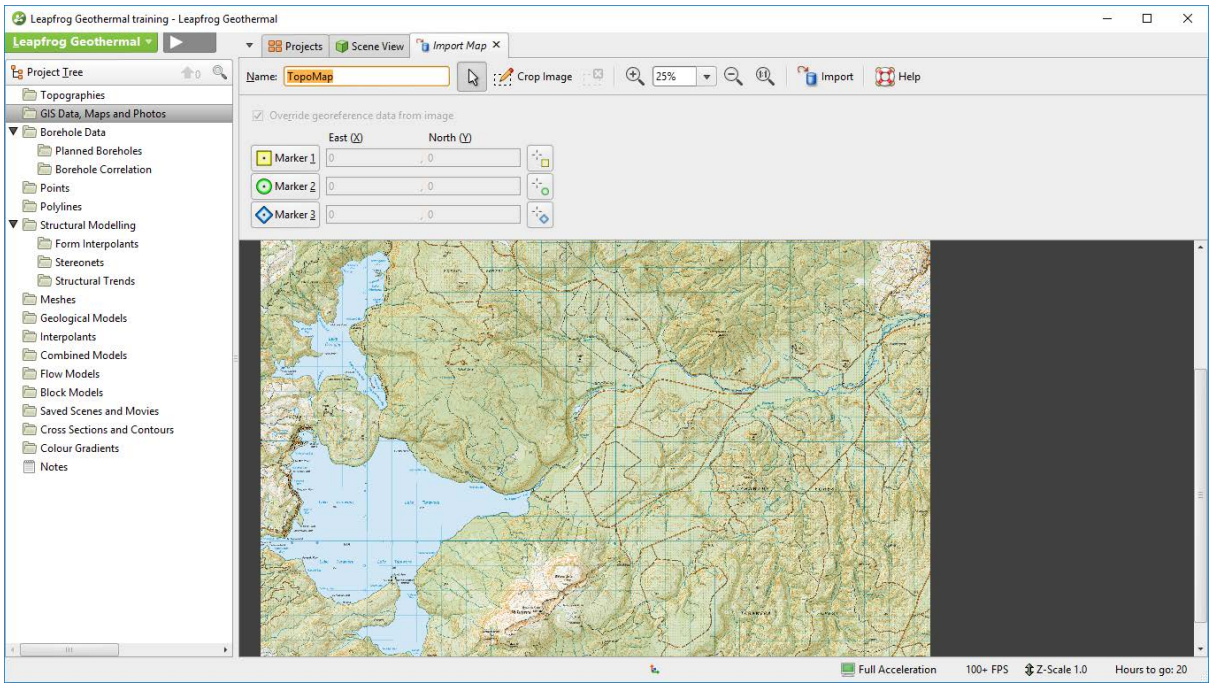
## IMPORTING MAPS AND AERIAL PHOTOGRAPHS

The easiest place to start loading data is with a map or aerial photograph. This allows the user to define the region of interest which can be used to clip large data sets. Leapfrog imports png, tiff, jpeg and bmp images and these can be imported from the GIS Data, Maps and Photos folder as per below.

- Right-click on the **GIS Data, Maps and Photos** folder.
- Select **Import Map**…
- Browse for the *TopoMap.png* in the *Leapfrog Training\GIS and Maps* folder.




A new **Import Map** tab opens.



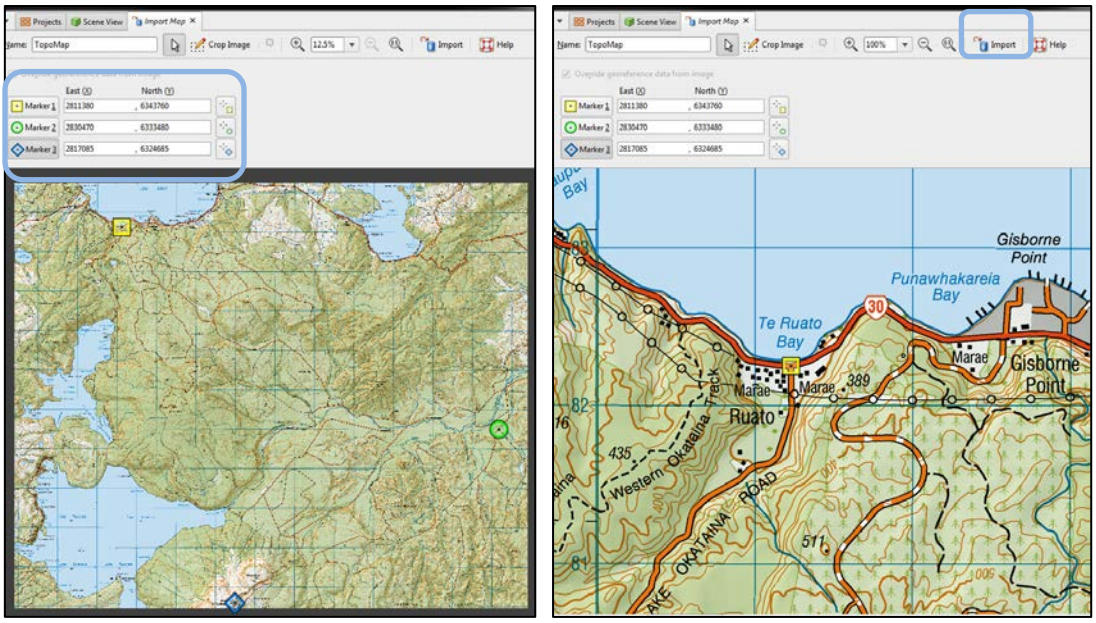

Maps are geo-referenced pictures:

- 3 points of known coordinates are necessary to correctly position the map into the model. These reference points are positioned in the image and then assigned their real world coordinates by the user.
- The Z-locations is identical for the three points. If the map is displayed in the 3D-viewer the Z location is used to define the height at which it is displayed.

You can open the file *Georeferenced Information.txt* and the *Georeferenced locations.jpg* image located in the *GIS and Maps* folder to visualize the position of the markers.

The simplest process for positioning the points in Leapfrog is:

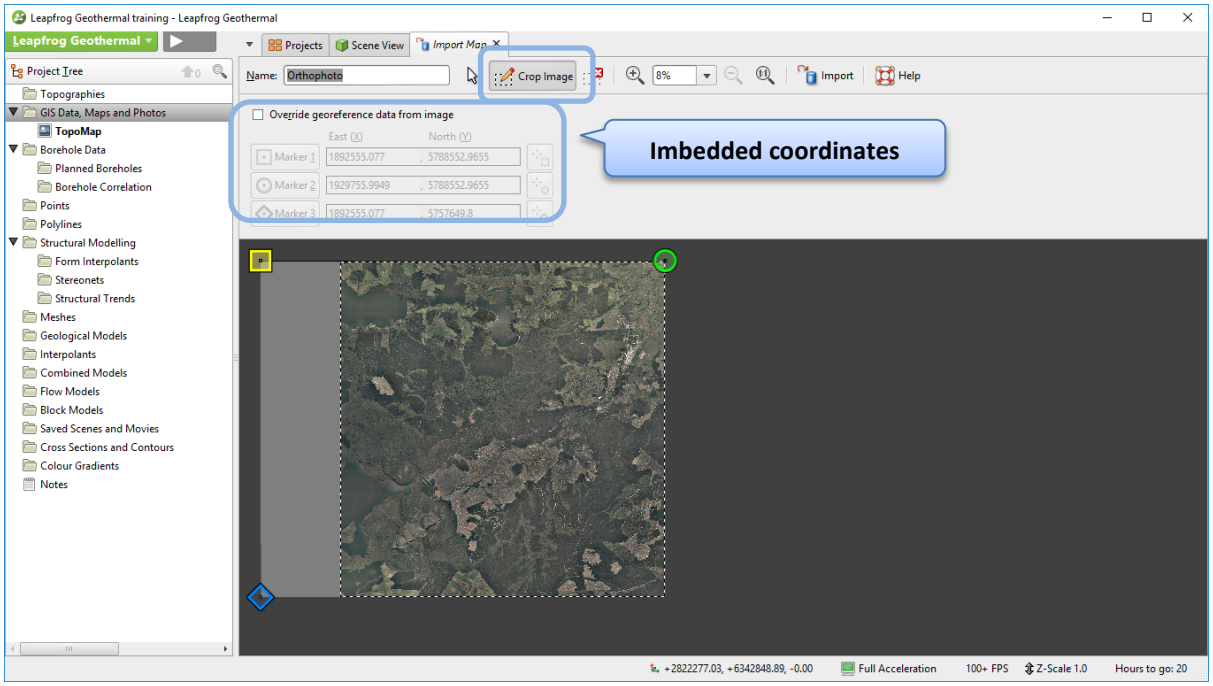
- Click on **Marker 1** and drag the symbol to its approximate location on the map using the *Georeferenced locations.jpg* image as guideline.
- Repeat the same procedure for the two remaining markers. To improve the referencing accuracy, it is important to ensure the markers do not lie on a line.
- Zoom in on the individual icons to accurately locate the markers.
- Type in the coordinates provided in the *Georeferenced Information.txt* file.
- Click **Import** and look at the map in the 3D scene.




*It is easy to make a mistake typing in the coordinates. If the map looks in an unusual position or distorted, the georeferenced points can be adjusted: right-click on the image and select **Open**.*

Maps or photos can have imbedded coordinates associated with them. If this is the case, Leapfrog automatically populates the coordinates. If the geo-referencing is erroneous or refers to another coordinate system, you can choose to override the data.

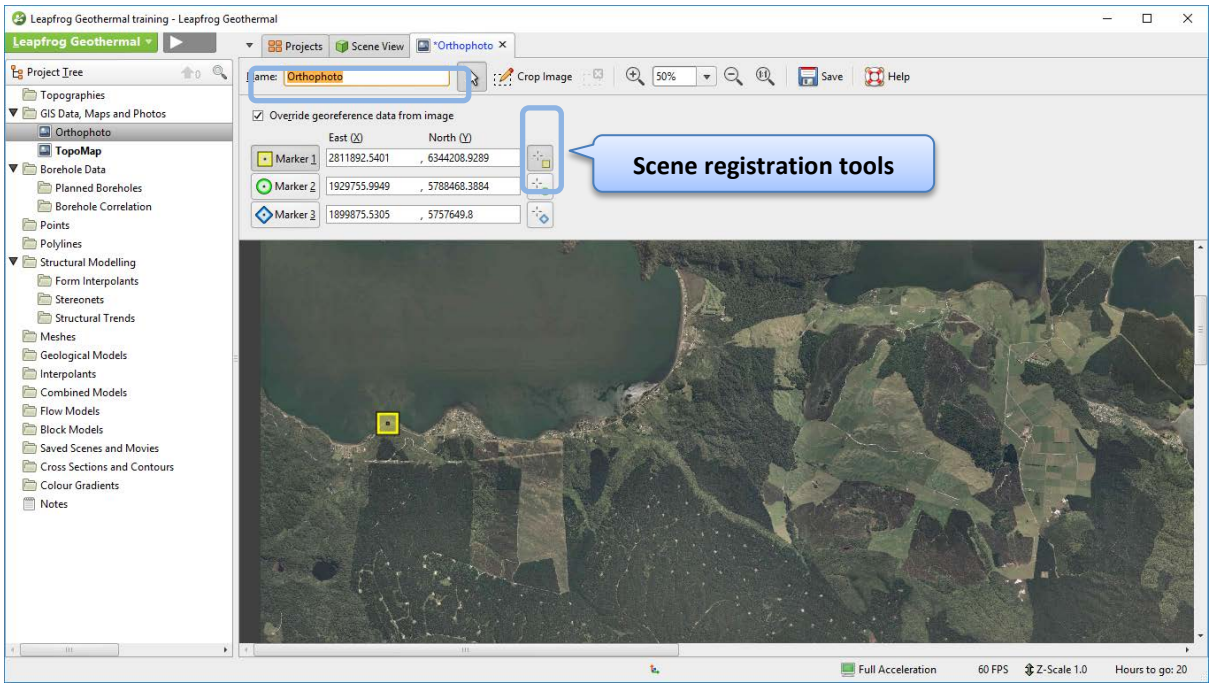
- Import the *Orthophoto.tif* file from the *Leapfrog Training\GIS and Maps* folder.
- Note that this photo has imbedded coordinates associated with it. It also has a white border that we do not wish to import.
- Click on **Crop Image** and select the area of the image you want to import.
- Click **Import**.



- Visualise the **Orthophoto** and the **TopoMap** in the 3D scene (remember to use the **Home** button on the keyboard).

The **Orthophoto** is not referenced using the same coordinate system as the **TopoMap**. Its imbedded coordinates need to be ignored and new coordinates given. This time, the **TopoMap** can be used as a reference map to populate the coordinates automatically.

- Right-click on **Orthophoto** and select **Open**.
- Select **Override georeference data** from image
- Position Marker 1 on a clear landmark (e.g. lake edge). Note that the **TopoMap** is smaller than the **Orthophoto**.
- Select the **Click on the scene to place the registration mark** tool and click on its true location in the map. The coordinates will be automatically populated.
- Repeat for Marker 2 and 3.
- Click **Save**.
- Visualise the **Orthophoto** and the **TopoMap** in the 3D scene. Make adjustments if needed.

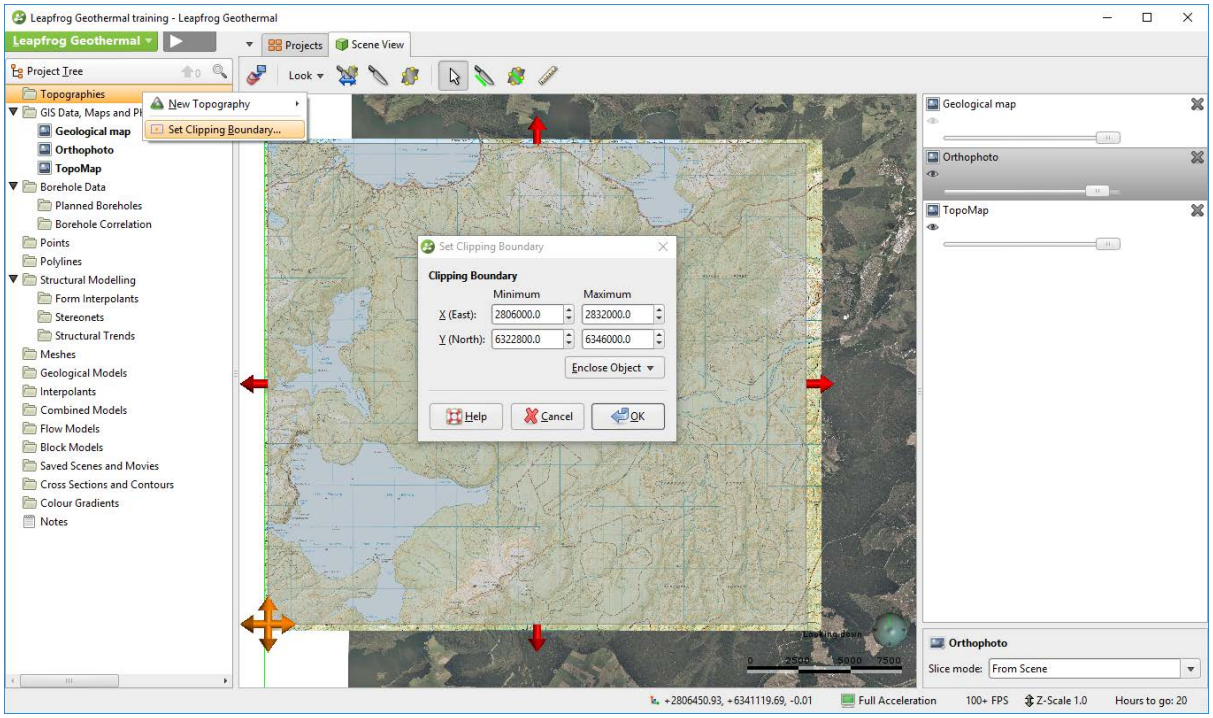



- Import the *Geological map.jpg* file from the *Leapfrog Training\GIS and Maps* folder.

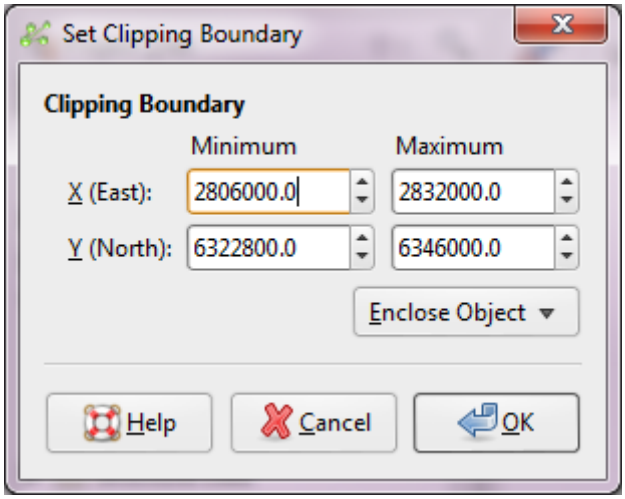
## SETTING THE REGION OF INTEREST

Once a map is imported it is worthwhile setting the region of interest, which can be used to clip some large vector/grid datasets. The clipping boundary can be set on screen to cover the desired region or by typing coordinates directly into the dialog. Alternatively, if other data is already loaded, the region of interest can be set to cover some of the data.

- Right-click on **Topographies** in the Project Tree.
- Select **Set Clipping Boundary**.
- Click in the scene view and look in map view (Press "D" on your keyboard).




- Resize the area of interest using the red and orange arrows and/or adjust the coordinates manually.
- Click **OK**.

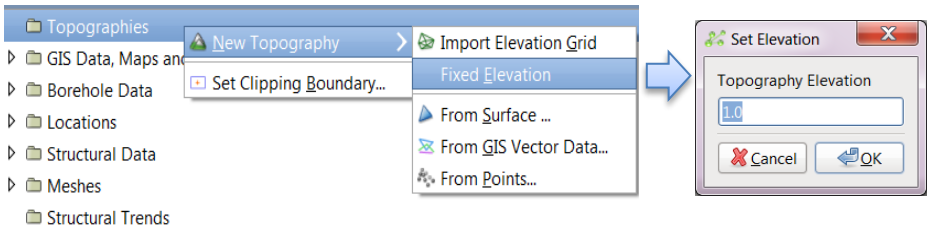



*Notes:*
*Unless explicitly set, the region of interest expands to include all loaded data. This can lead to problems if the user accidently loads data in another coordinate system or large files. For this reason, it is best to set the region of interest early in the modelling process.*
*Set your region of interest to a slightly wider extent than your expected model, to avoid any risk of having to increase it at a later stage.*

## CREATING THE TOPOGRAPHY

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

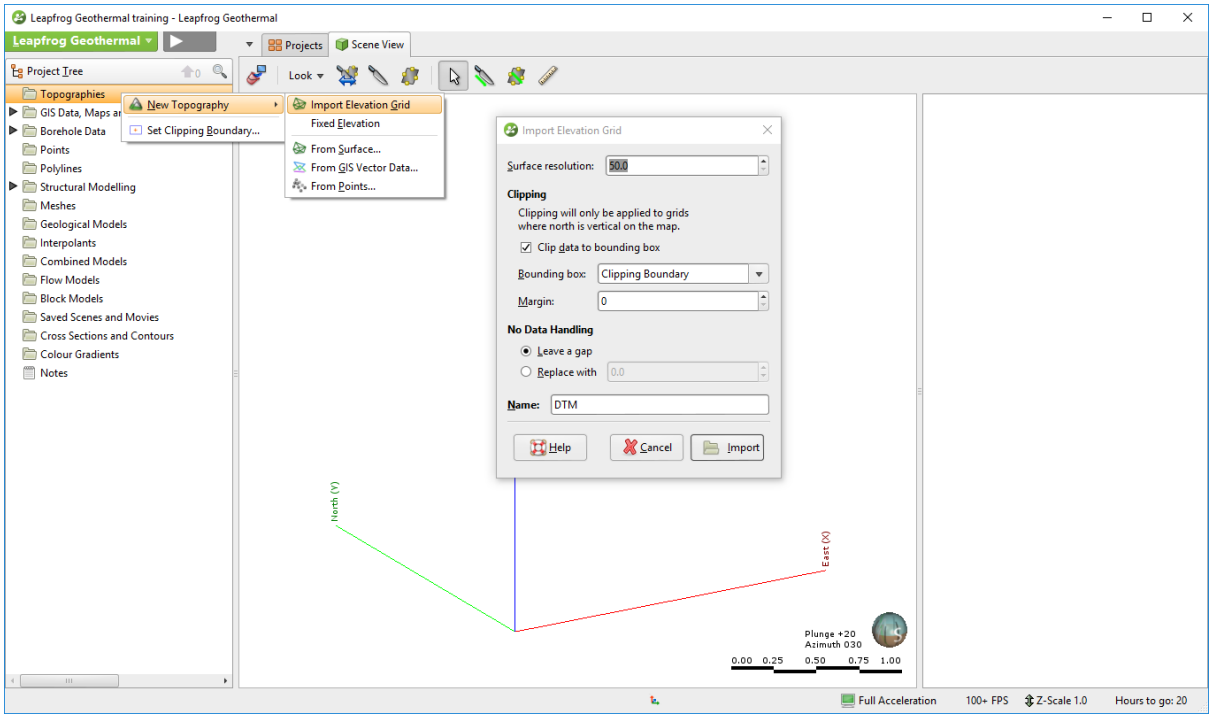
If there is no topography data available it can be set to a constant height by right-clicking on the terrain and setting the elevation.




*The terrain is used to bound geological /interpolant and flow models and to position GIS data onto the topography. As a result, changing the terrain will cause models to be recomputed and GIS data to be repositioned.*

For this tutorial, a single DTM file will be used to define the topography: the *DTM.asc* file.

- Right-click on **Topographies**.
- Select **New Topography** and **Import Elevation Grid**.
- Browse to the *Leapfrog Training\TopoData* folder.
- Open the *DTM.asc* file.

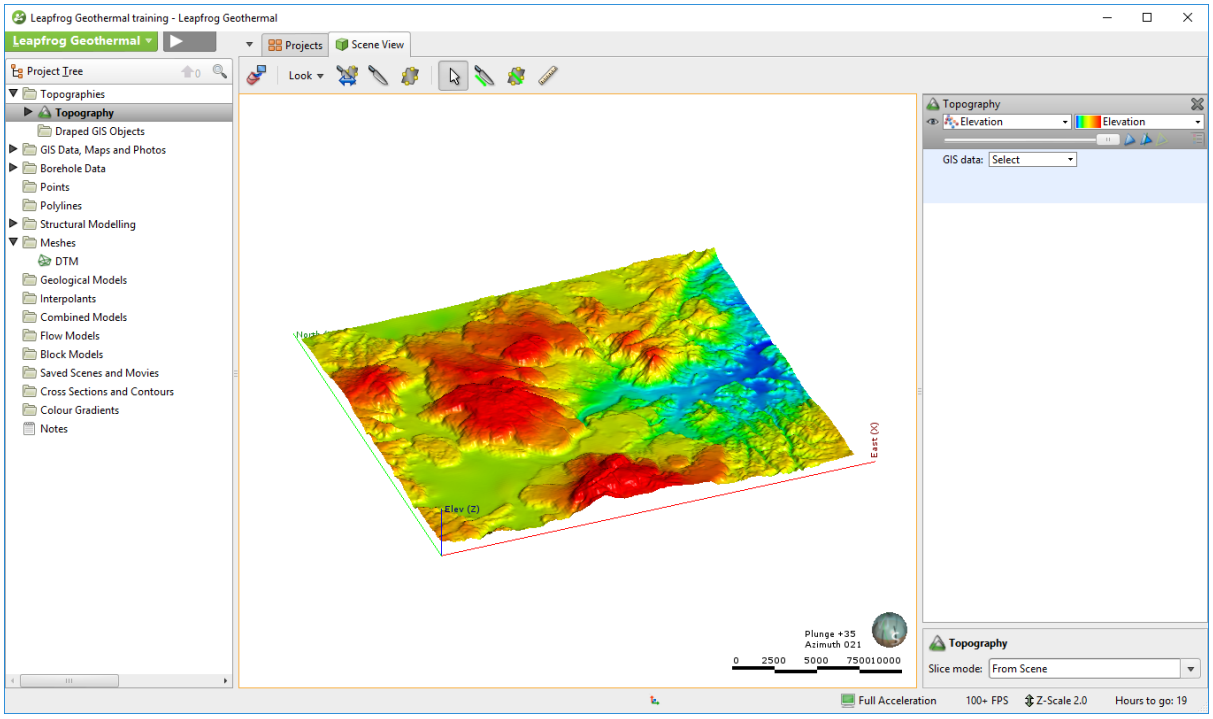



The **Import Elevation Grid** menu will open. Enter the following parameters:

- Surface Resolution: **50 (metres).**
- Tick **Clip Data to bounding box**.

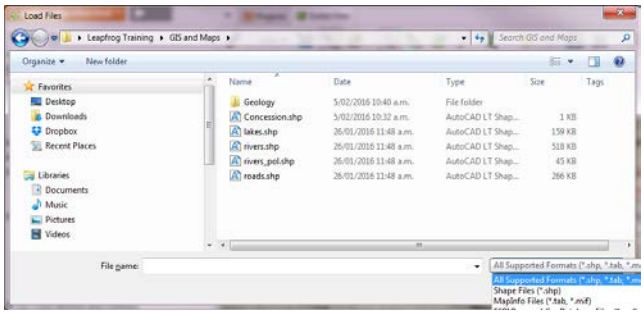
- Enter a name.
- Click on **Import**.
- A new window opens for the **Topography** object. You get the option to rename it, then click **OK**. A new object has been created under the Topographies folder and the DTM file has been converted into a mesh now located under the Meshes folder of the Project Tree.
- Drag the new **Topography** object in the 3D scene. Remember that display settings can be modified by changing **Flat Colour** to **Elevation** (in the Scene List) and adjusting the Colourmap for a better visualisation of height variation.
- Use the **Scale Z Axis** tool to exaggerate the Z scale.

Once the Topography object is defined, remember that you can drape the maps on it.



## IMPORTING GIS DATA

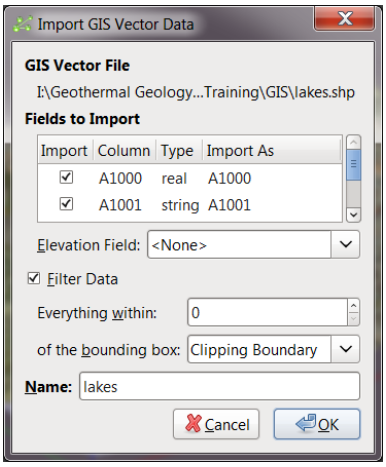
Leapfrog Geothermal allows you to import, create and export GIS data.




- Right-click on **GIS Data, Maps and Photos**, select **Import Vector Data**.
- Under *Leapfrog Training\GIS and Maps* folder select the *lakes.shp* file and click **Open.**

*Note that several GIS formats can be imported.*

A new window appears. You can choose which field you wish to import. You can specify one of the fields as elevation if your GIS data has elevation data associated to it. You can also spatially filter the data. By default, Leapfrog will clip the GIS dataset to the clipping boundary previously defined.



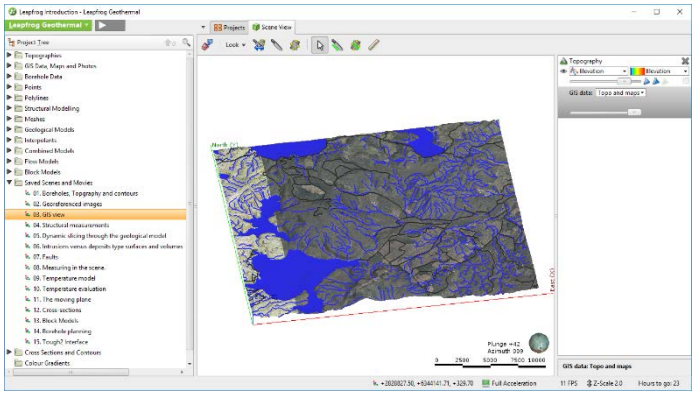

- Filter the data using the **Clipping Boundary**.
- Click **OK.**
- Drag the shapefile in the 3D scene.
- Repeat the operation and import the following shapefiles from the same folder:
  - *concession.shp*
  - *roads.shp*
  - *rivers.shp*
  - *rivers_pol.shp.*
- Note that you can do batch import.
- You can now drag the GIS data into the 3D scene.

Remember that you can visualize the GIS data wrapped on the elevation grid from the *Topographies\Draped GIS Objects* folder, or by creating a **GIS Data View**.

- Drag and drop the **Topography** object into the scene.
- From the GIS data drop-down list in the scene list, select **Views\Edit Views...**

The **Edit GIS Views** window will appear and you can create a new one.

- Combine maps and GIS data to be wrapped on the elevation grid.



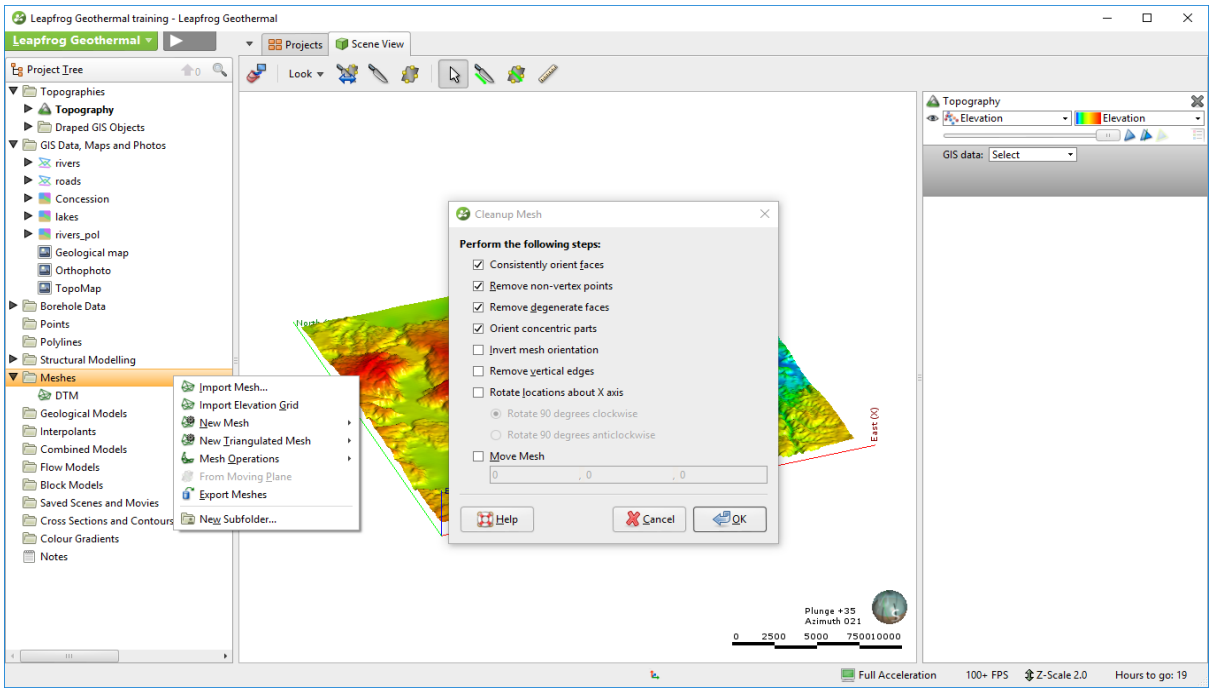
*Maps wrapped on the Topography object are automatically cropped to the extent of the topography.*

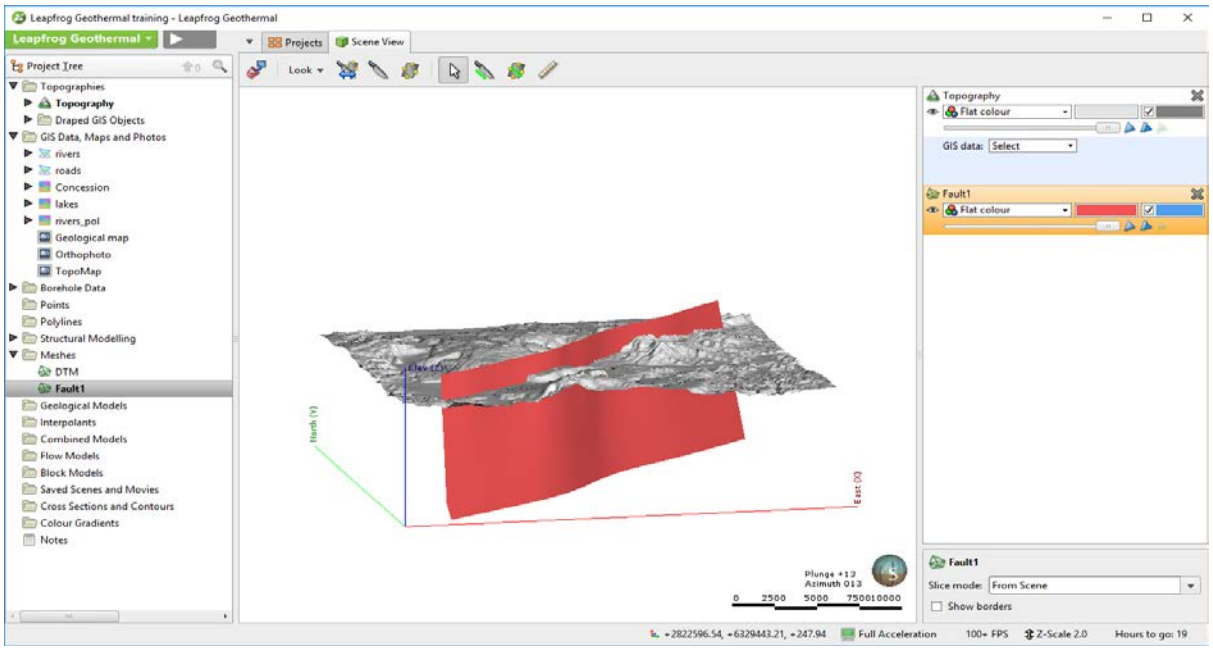
## IMPORTING MESHES

Meshes are used in Leapfrog to represent 3D surface objects. Leapfrog can import various file format containing vertices and triangles combined to define a shape. Meshes generated in Leapfrog use the .msh extension.

- Right-click on the **Meshes** folder in the Project Tree and select **Import Mesh**.
- Browse to the *Leapfrog Training\Mesh* folder and select *Fault1.msh*.
- A new window opens. Click **OK**.
- The new surface will appear in the Meshes folder. Drag the new fault object into the 3D scene.

*From this menu you can offset the imported mesh in 3D space.*



## IMPORTING POINT DATA

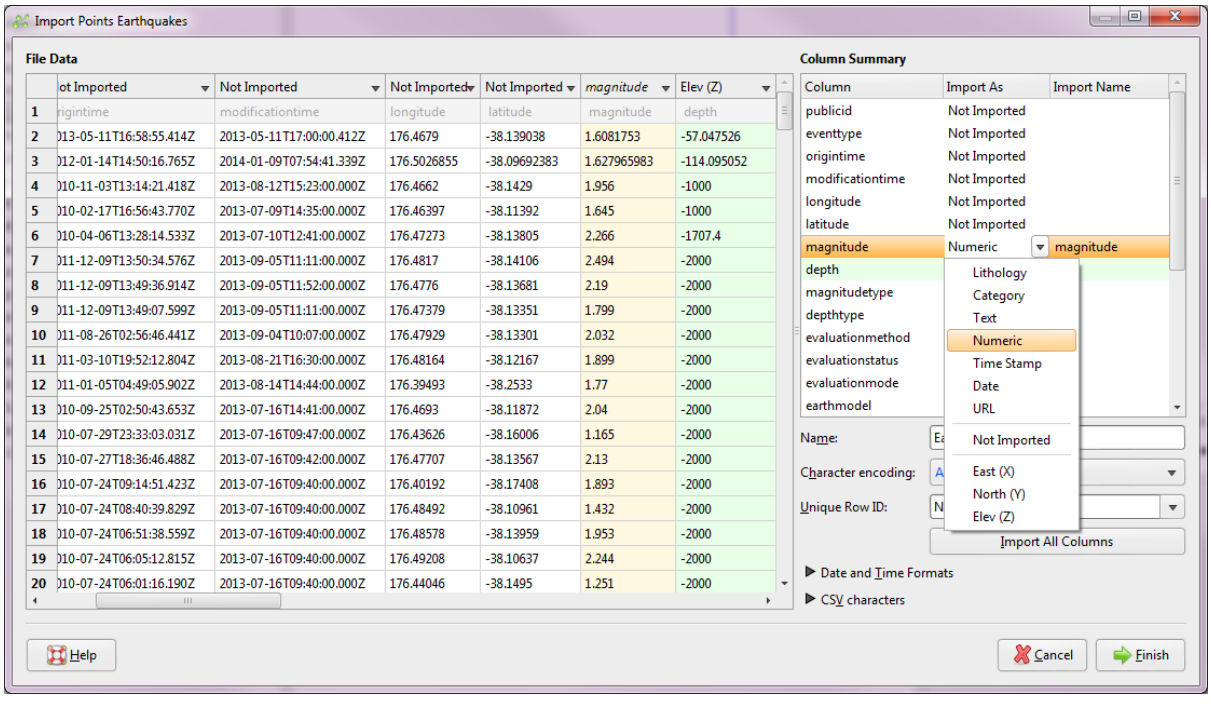
Point data can be loaded in X,Y,Z format and associated to an arbitrary number of columns that contain numerical values. This is a generic format that can be used to load any georeferenced points, geochemical and geophysical data.

- Right-click on the **Points** folder.
- Select **Import Points**.
- Browse to the *Leapfrog Training\Points* folder.
- Select the *Earthquakes.csv* file.
- Click **Open.**

Leapfrog Geothermal will display a new window where it is trying to match the data found in the files with the expected columns, based on the header row in each file. The user needs to check that the columns are correctly attributed to X,Y, and Z and to define any custom columns.

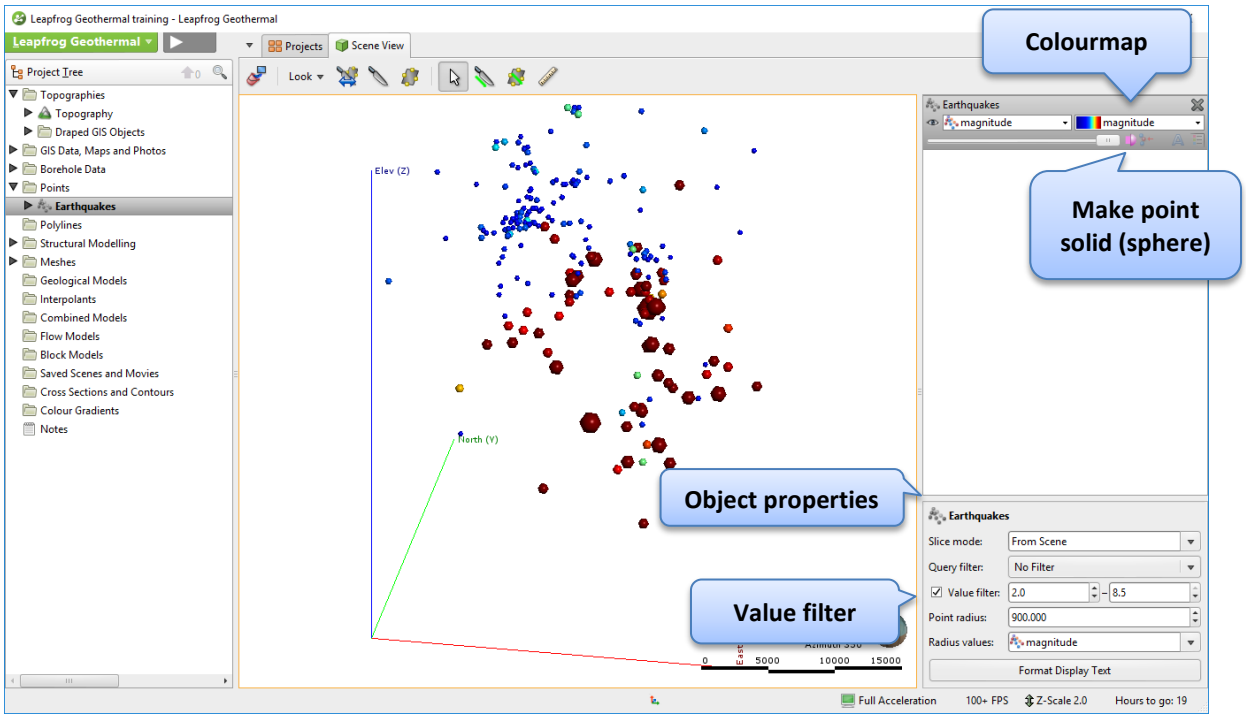
The series of dropdown lists along the top of the table indicates the label Leapfrog Geothermal will assign to the imported data. If Leapfrog Geothermal has incorrectly identified a column, choose the correct assignment from the dropdown list as per the instructions below:

- *Easting* is**: East (X)**.
- *Northing* is: **North (Y)**.
- *Depth* is: **Elev (Z)**. The column is not recognized and will not be imported unless manually selected.
- *Magnitude*: The column is not recognized and will not be imported unless manually selected. Select the Data Type as **Numeric.**
- Choose more columns if you want to.
- Click **Finish**.




- Display the new object in the 3D scene.

Once the data is imported it is often useful to change the colour or display values within a range. Remember to use the object properties in the Scene List including the **Value Filter** tool.




*Reminder: A query filter can also be generated to show a subset of the data. Queries can be applied to several data type (locations, GIS attributes, borehole tables) and used as input when generating geological models or numerical models.*

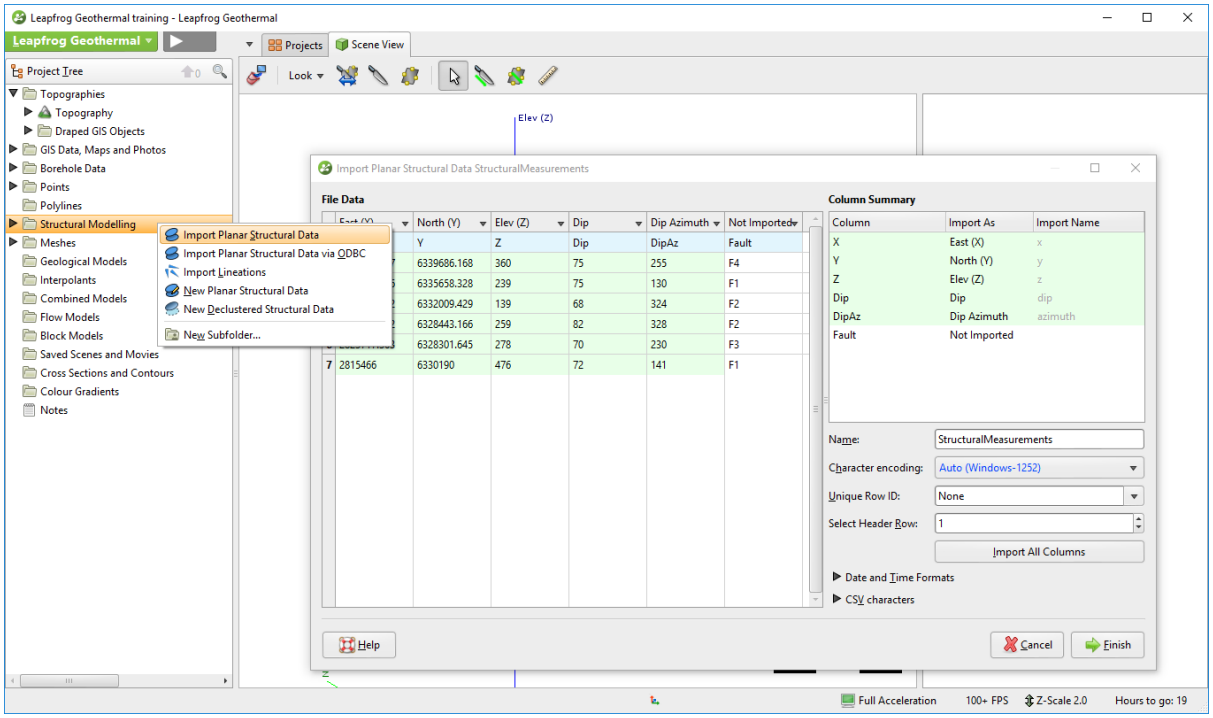
## IMPORTING STRUCTURAL DATA

Leapfrog Geothermal allows to create or to import structural measurements from the structural data folder.

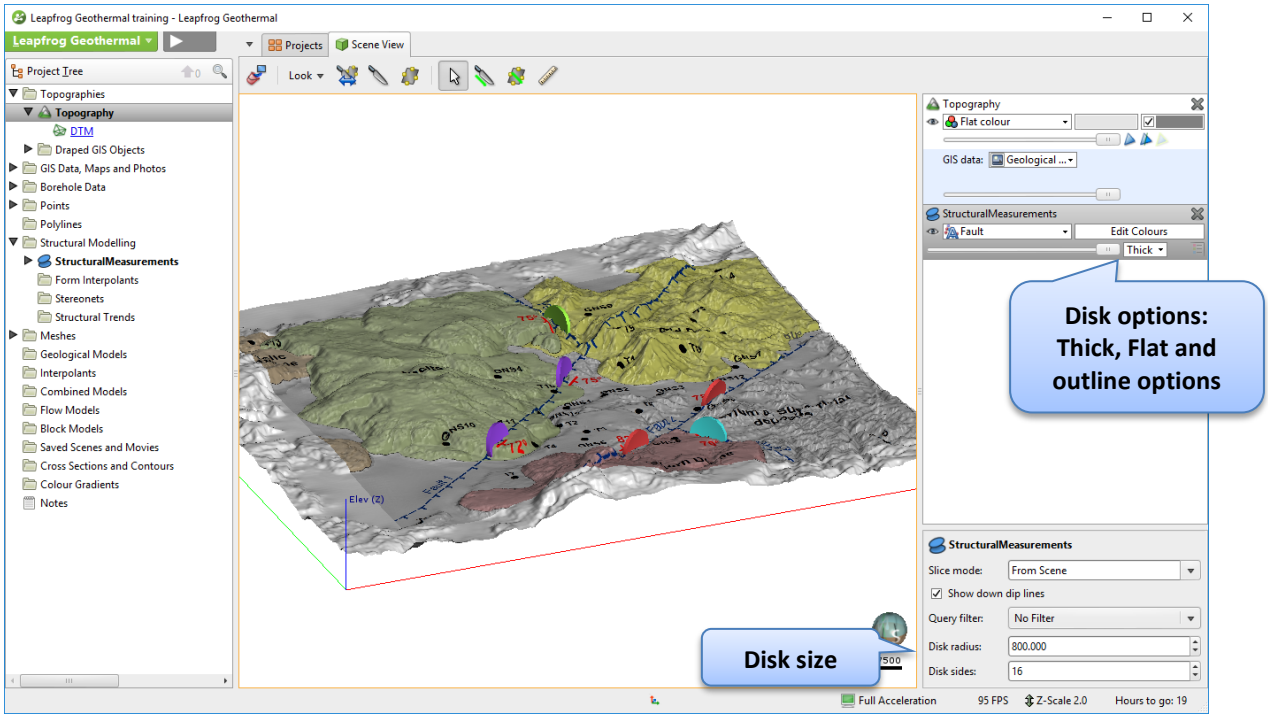
- Right-click on the **Structural Modelling** folder and select **Import Planar Structural Data**.
- Browse to the *Leapfrog Training\Structural Data* folder and select the *Structural Measurements.csv* file. A new Import Structural Measurements window pops-up.

The data consists of:

- X, Y, Z location points.
- Dip and Dip azimuth at the measured point.
- A Fault column (optional), listing the name of the fault associated to the structural measurement.




- Import the Fault column as **Category.**
- Click **Finish**.
- Display the new object in the 3D scene. Structural measurements are represented as disks oriented in the 3D space.
- Use the visualisation properties to modify the size of the disks.
- Use the drop-down menu in the scene list to visualise the disks by their attributes: Dip, Azimuth or Fault.



## IMPORTING BOREHOLE DATA

Borehole data are the main basis to create models in Leapfrog Geothermal. Leapfrog Geothermal expects a collar table, a survey table and interval tables. Each project can have only one collar file and one survey file, but multiple interval tables can be imported.

Each well is defined by a unique ID. It must be similar in each table.

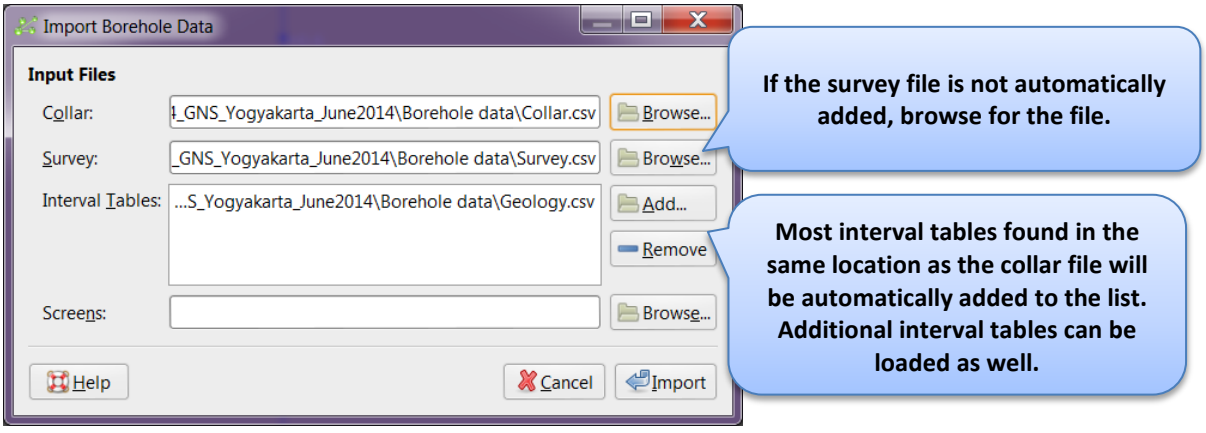
- The collar table contains at least: Well name, X, Y, Z coordinates and the maximum drilled depth (measured depth).
- The survey table contains information that describes the deviation of each borehole: Well name, and a list of depths with associated azimuth and dip. **The dip angle is 90 for vertical and 0 for horizontal.**
- The interval table contains the well name, and the data encountered in a given interval defined by its top depth (*from*) and bottom depth (*to*). The interval tables must contain information on measurements such as lithology, numeric or text values.

*These are the mandatory fields required by the software, but you can load as many columns as you want into your tables.*

Three methods are available to import borehole data in a project:
- Import borehole from .csv tables (demonstrated in this tutorial).
- Establish a connection to an ODBC database.
- Import boreholes via acQuire.

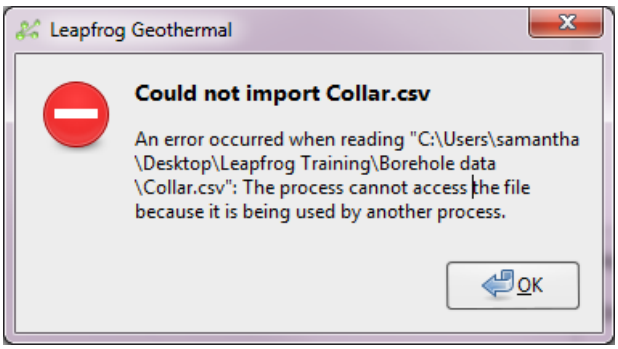
### IMPORTING COLLAR, SURVEY AND IINTERVAL TABLES

- Right-click on the **Borehole Data** folder in the Project Tree and select **Import Boreholes.**
- The Import Borehole Data window appears.
- For the collar data, click on **Browse** to locate the file in *Leapfrog Training\Borehole data.*
- Leapfrog automatically picks up the other tables as they are saved in the same directory.




- Click on **Import** to start the process of importing data.

*If the files are already opened in other software, for example Excel, an error will appear. Close the files you are loading and try again.*



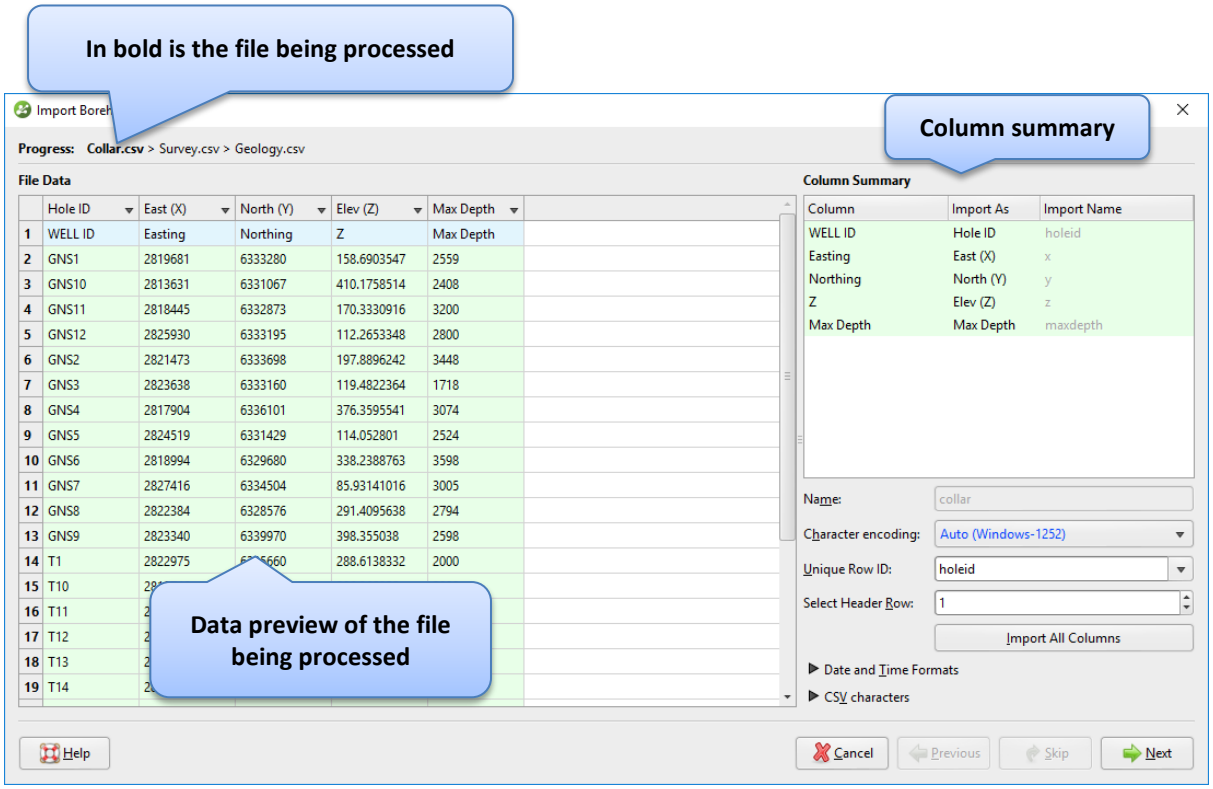
Leapfrog Geothermal will display a series of windows, one for each file, and will attempt to match the data found in the files with the columns expected, based on the header row in each file. It is the same principle as used when importing point data.

Here are a few key points:

- If Leapfrog Geothermal marked a column "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 any of the information Leapfrog Geothermal expects is missing, an error message will appear. Return to the table and correct the problem described.

For the **Collar** table:

- Ensure that the columns have been correctly identified as per the image below.
- Click **Next** to move on to the next file.

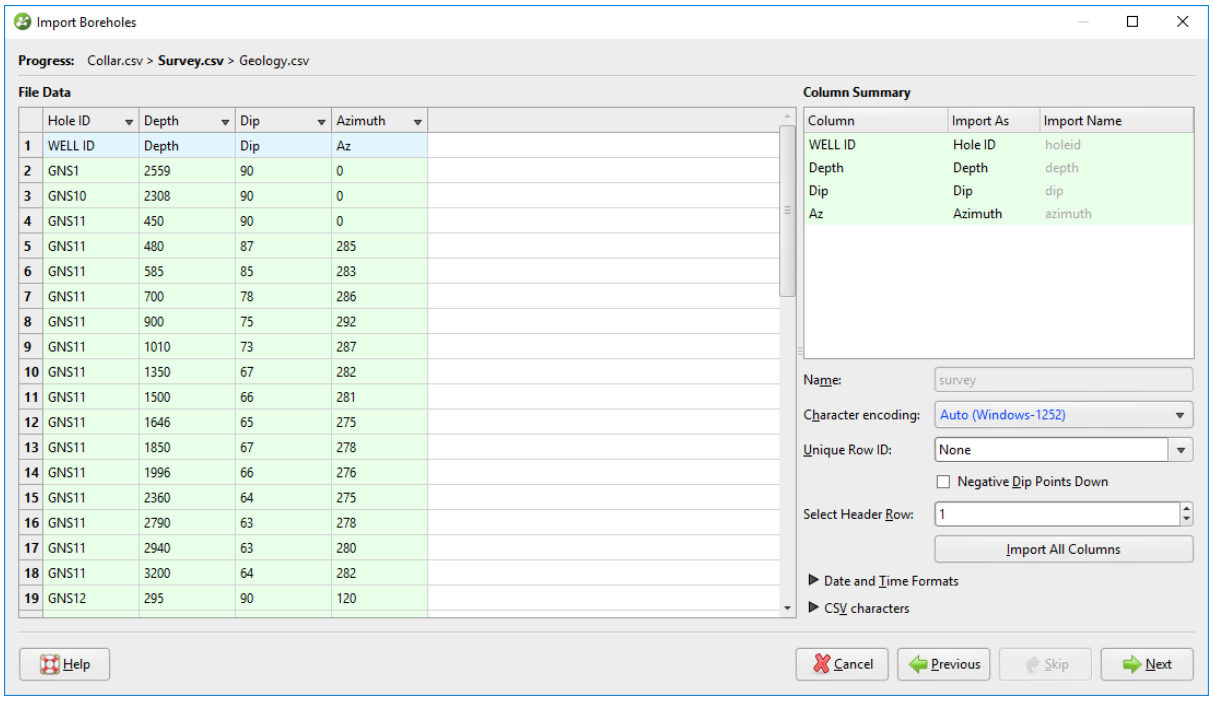



| | Hole ID | East (X) | North (Y) | Elev (Z) | Max Depth |
|---|---|---|---|---|---|
| 1 | WELL ID | Easting | Northing | Z | Max Depth |
| 2 | GNS1 | 2819681 | 6333280 | 158.6903547 | 2559 |
| 3 | GNS10 | 2813631 | 6331067 | 410.1758514 | 2408 |
| 4 | GNS11 | 2818445 | 6332873 | 170.3330916 | 3200 |
| 5 | GNS12 | 2825930 | 6333195 | 112.2653348 | 2800 |
| 6 | GNS2 | 2821473 | 6333698 | 197.8896242 | 3448 |
| 7 | GNS3 | 2823638 | 6333160 | 119.4822364 | 1718 |
| 8 | GNS4 | 2817904 | 6336101 | 376.3595541 | 3074 |
| 9 | GNS5 | 2824519 | 6331429 | 114.052801 | 2524 |
| 10 | GNS6 | 2818994 | 6329680 | 338.2388763 | 3598 |
| 11 | GNS7 | 2827416 | 6334504 | 85.93141016 | 3005 |
| 12 | GNS8 | 2822384 | 6328576 | 291.4095638 | 2794 |
| 13 | GNS9 | 2823340 | 6339970 | 398.355038 | 2598 |
| 14 | T1 | 2822975 | [illegible] | 288.6138332 | 2000 |
| 15 | T10 | 28[illegible] | | | |
| 16 | T11 | 2[illegible] | | | |
| 17 | T12 | 2[illegible] | | | |
| 18 | T13 | 2[illegible] | | | |
| 19 | T14 | 2[illegible] | | | |

| Column | Import As | Import Name |
|---|---|---|
| WELL ID | Hole ID | holeid |
| Easting | East (X) | x |
| Northing | North (Y) | y |
| Z | Elev (Z) | z |
| Max Depth | Max Depth | maxdepth |

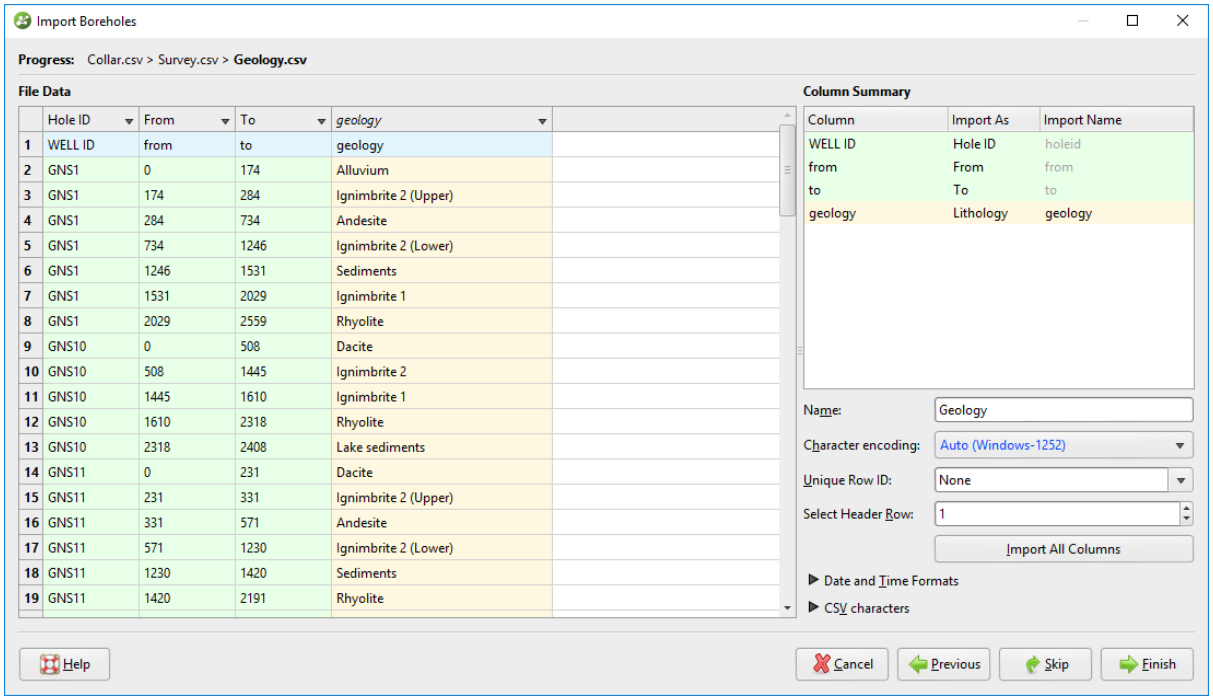
For the **Survey** table:

- Check that the information is correct.
- Make sure that the **Unique Row ID** is set to **none** in this case. There are several azimuth and dip values for each well so there is no unique identifier in this table.
- Click **Next**.




For the **Geology** table:

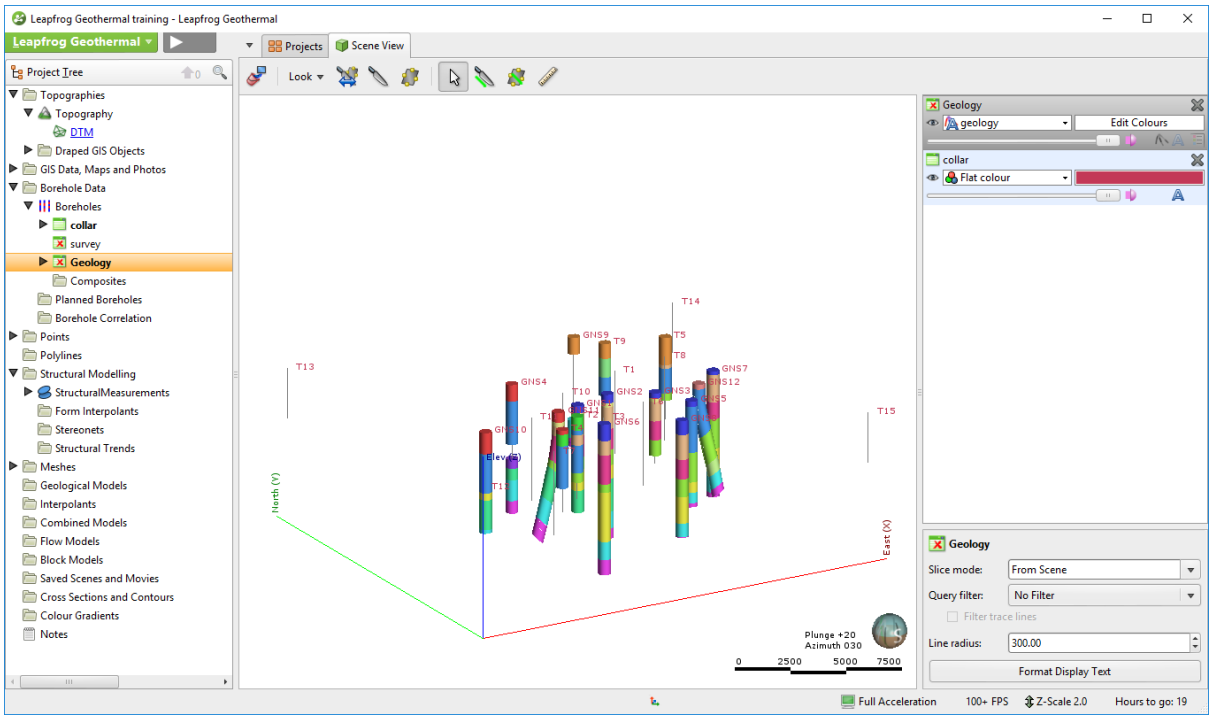
- Ensure that the columns have been correctly identified.
- Click **Next**.




- Once you have checked the last file, click on **Finish** to complete the process.

The Boreholes object is now populated with three tables: **Collar**, **Survey** and **Geology**.

- Select the **Boreholes** object and drag it into the 3D scene to visualise the borehole data. As you can see, only the **geology** and the **collar** object are visible in the Scene List. The survey table cannot be seen independently.

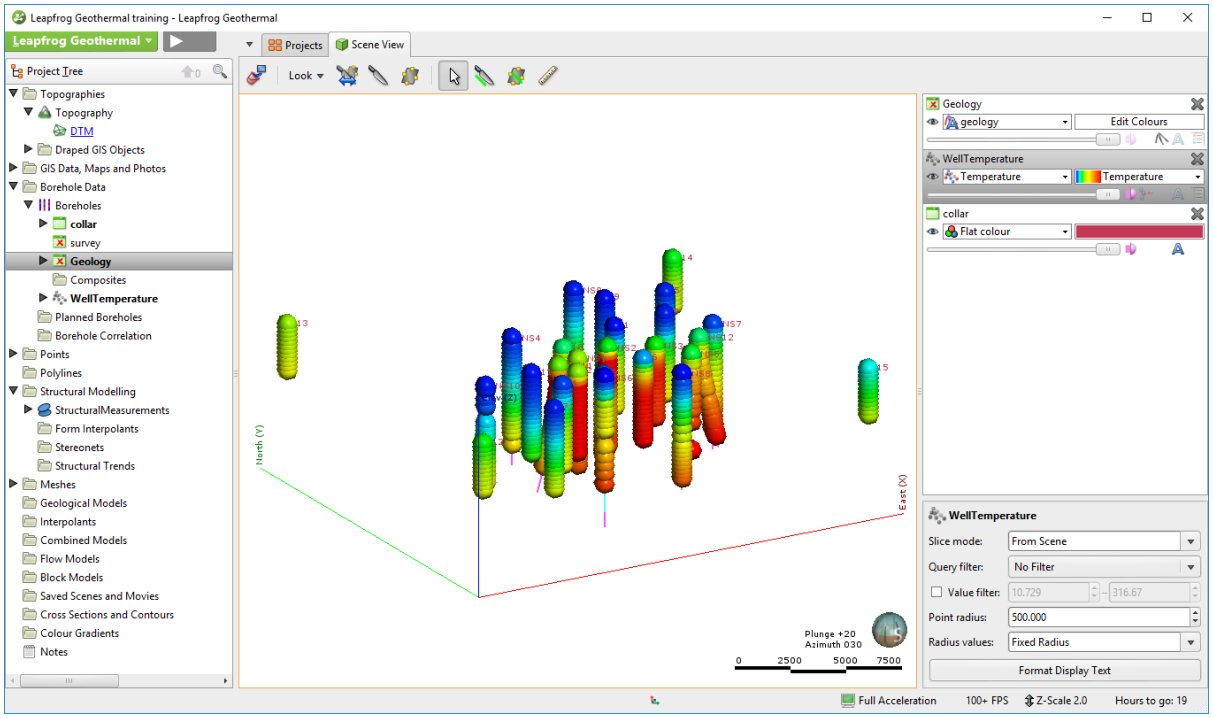



*A red cross appeared on the icons of the **Survey** and **Geology** tables, which means that the software found errors in the data. Errors will be fixed later in this tutorial.*

## IMPORTING POINT VALUES

Leapfrog Geothermal allows importing point data along the borehole trace. The information required is: well name, depth along the well and any subsequent data such as numeric value or categories.

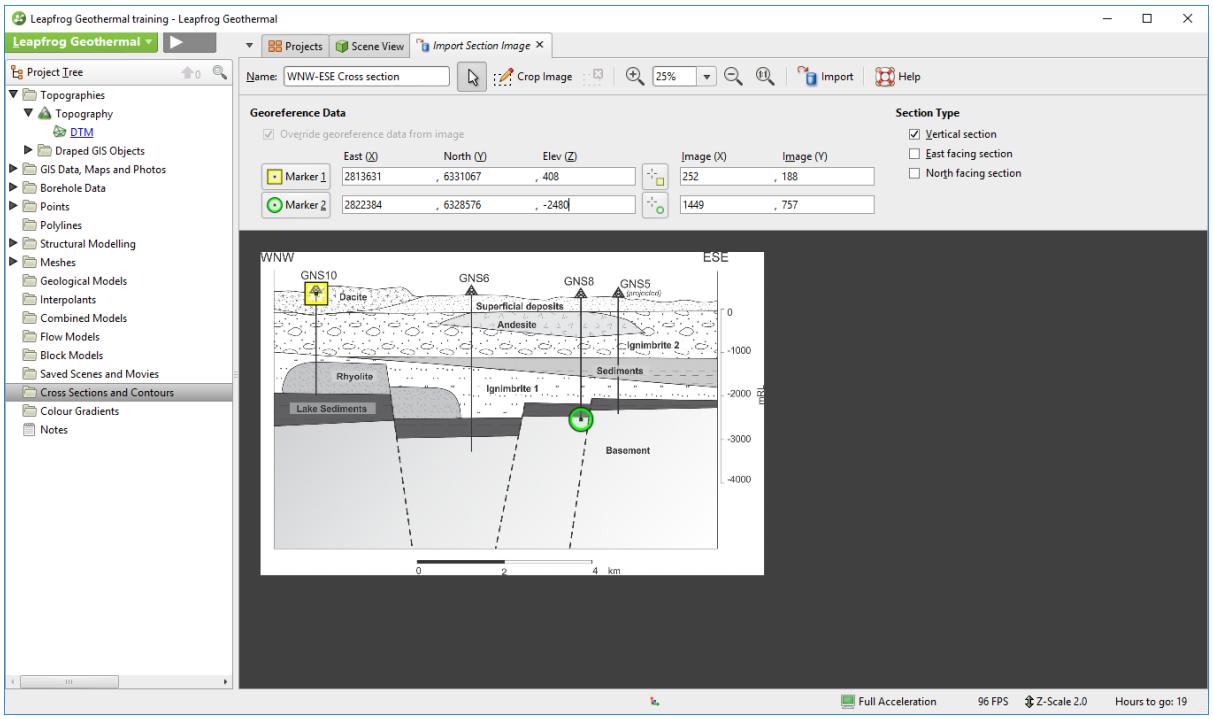
- From the **Borehole Data** folder, right-click on **Boreholes** and select **Import From File/Point Values**.
- Select the *WellTemperature.csv* file.
- The **Import Depth Points** window appears.
- Import the Temperature column as **Numeric** data**.**
- Click **Finish**.
- Visualize the data together with the collar and geology data.



## IMPORTING AN EXISTING SECTION

Importing a section uses the same procedure covered for geo-referencing maps. It requires two or three reference points, which must be located in the image and given their real coordinates.

- Right-click on the **Cross Sections and Contours** folder.
- Select **New Cross Section from Image**.
- Browse to the *Cross section* folder in the tutorial data.
- Select the file *WNW-ESE Cross section.jpg*.
- Check the **Vertical section** type. Only 2 markers are now necessary.
- Position the markers using the well location as guidelines (GNS10 well head location; GNS8 bottom hole location).
- Enter the coordinates manually or click on the scene to extract the coordinates from the scene.
- Click **Import.**
- Visualize the cross-section in the 3D scene.

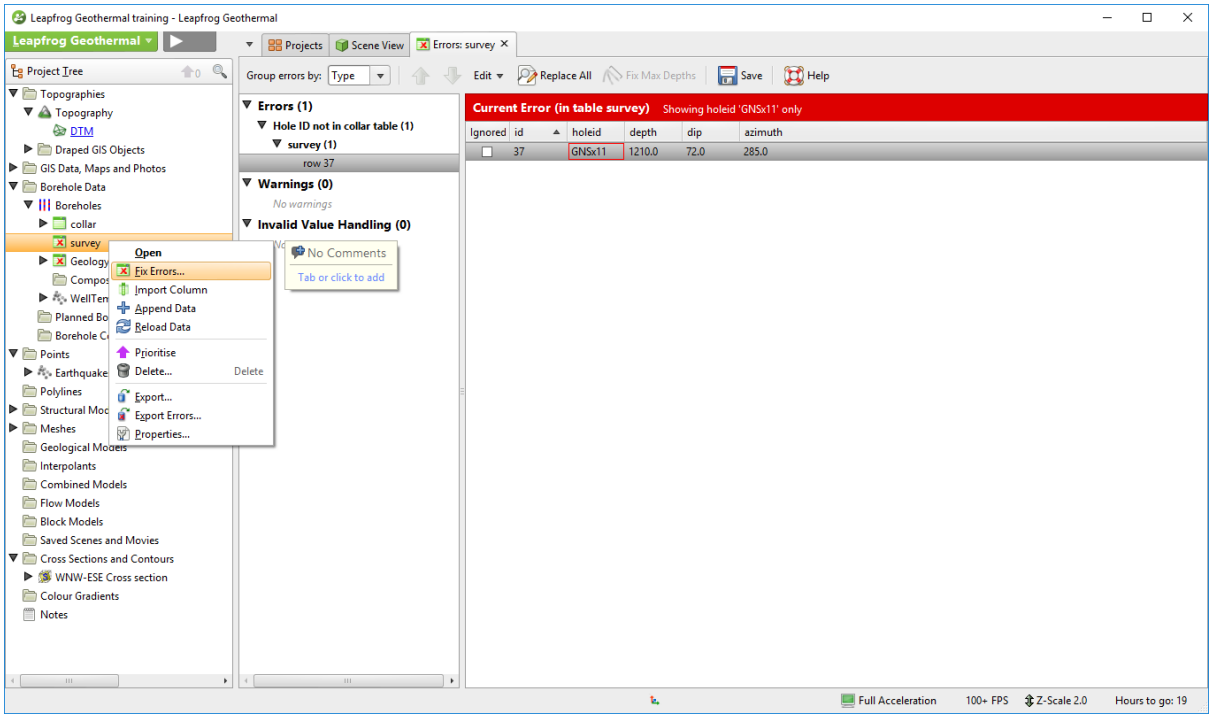


## FIXING ERRORS

Unprocessed data often contains errors that prevent the computation of an accurate model. Leapfrog Geothermal has tools that help the user to correct errors in the data.

A red cross on the icon of the data imported means that Leapfrog identified inconsistencies in the imported data. In this exercise, the **Survey** and **Geology** tables under the **Boreholes** object have a red cross. Errors can be fixed as follows:

- Right-click on the **Borehole Data/Boreholes/Survey** table and select **Fix Errors**.
- The Errors tab appears, with one errors listed:
  - **HoleID not in collar table (1)**
- Expand the error until you find the row number and click on it. On the right hand side, the table appears with the error surrounded in red.
- In this example you can see that the well name has been misspelled. Correct it and click **Save**.
- The red cross has now disappeared. Close the **Errors** tab.



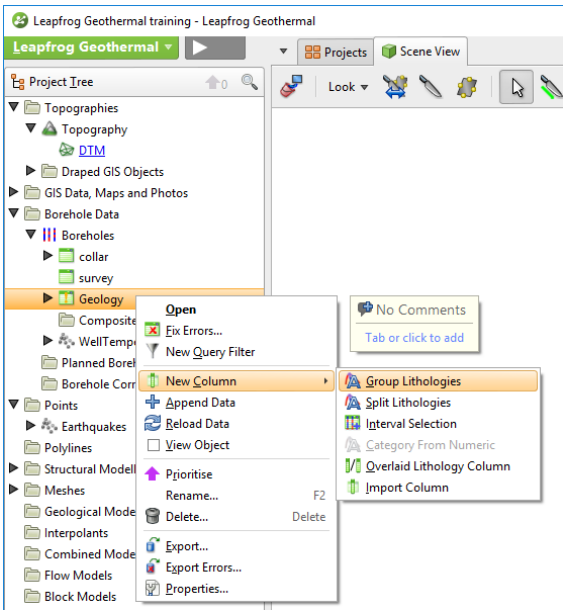

- Right-click on the **Borehole Data/Boreholes/Geology** table and select **Fix Errors**. Three errors are listed:
  - One error is related to the **Collar maximum depth**: the interval within the geology table is deeper than the maximum depth defined in the collar table.
  - Two errors are due to **overlapping segments**: lithology intervals overlap each other.
- Fix all the errors and check the warnings.

*If multiple errors are identified, it may be more appropriate to update the source file and reload the borehole data within the software once all errors are fixed. Right-click on the table and select Reload Data.*

## GROUPING LITHOLOGIES

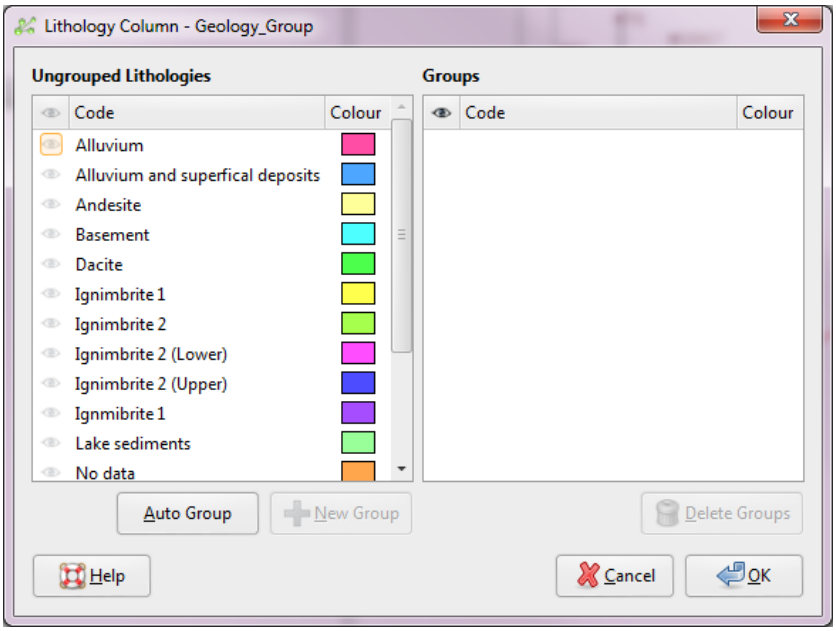
There are often many sub-stratigraphic units defined during the observation of cuttings and cores during the drilling of a well. In order to have a synthetic model, or when building a preliminary model, it is recommended to group the lithology into major units. You can create several grouped lithologies and their associated Geological Models. You will also experience that well databases are not always clean of errors. Misspelling is one of the most frequent examples of data issue. Grouping the data in Leapfrog allows you to ignore it.

- Right-click on the **Geology** table and select **New Column/Group Lithologies**.
- A new window will appear: select the Base Column as **Geology.**
- Give a name to the new column that will be created: "Geology_Group".
- Click **OK**.

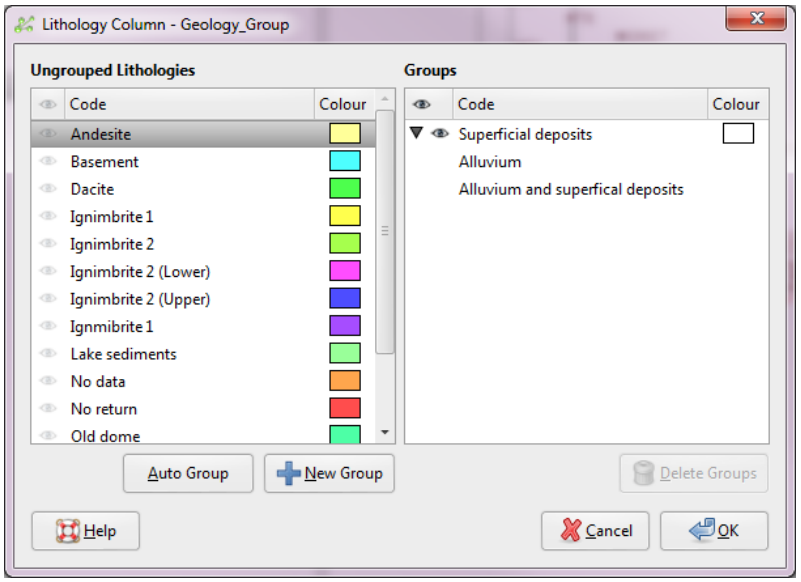



The menu is used to group several Lithologies and to create the simplified stratigraphic column appears. The left hand side displays all the lithologies encountered in the imported Geology table, whereas the right hand side displays the new stratigraphic grouped column.

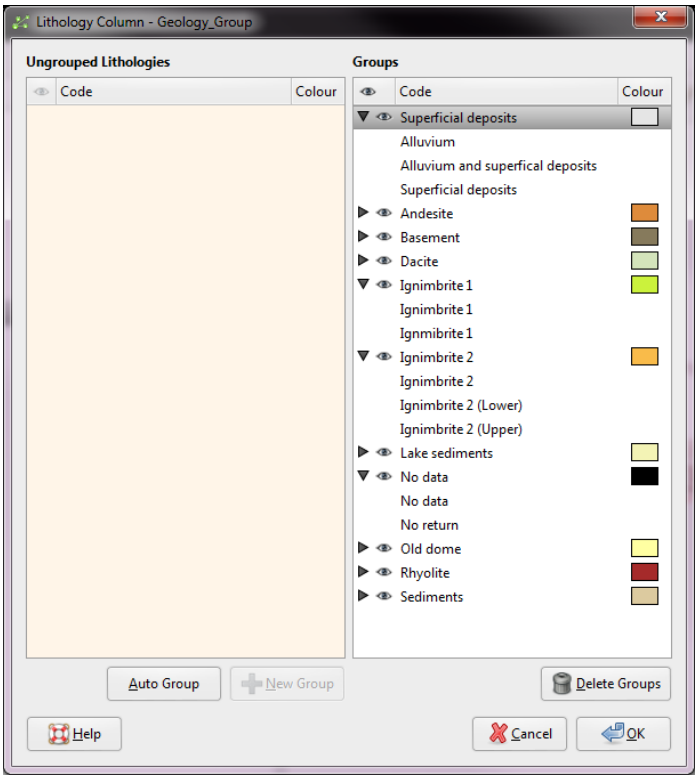
As you can see the first two lithologies are Alluvium, and Alluvium and superficial deposits. We can group these two formations together.



- Select **Alluvium**.
- Click on **New Group**.
- Change its name to **"Superficial deposits"** and expand the group by clicking on the triangle on its left.
- In the left panel, click on **Alluvium and superficial deposits** and drag it into the **Superficial Deposits** group**.**




- Observe the remaining formation and group them together.
- Change the color of each group if desired.
- Visualize the new **Geology_Group** in the 3D scene.

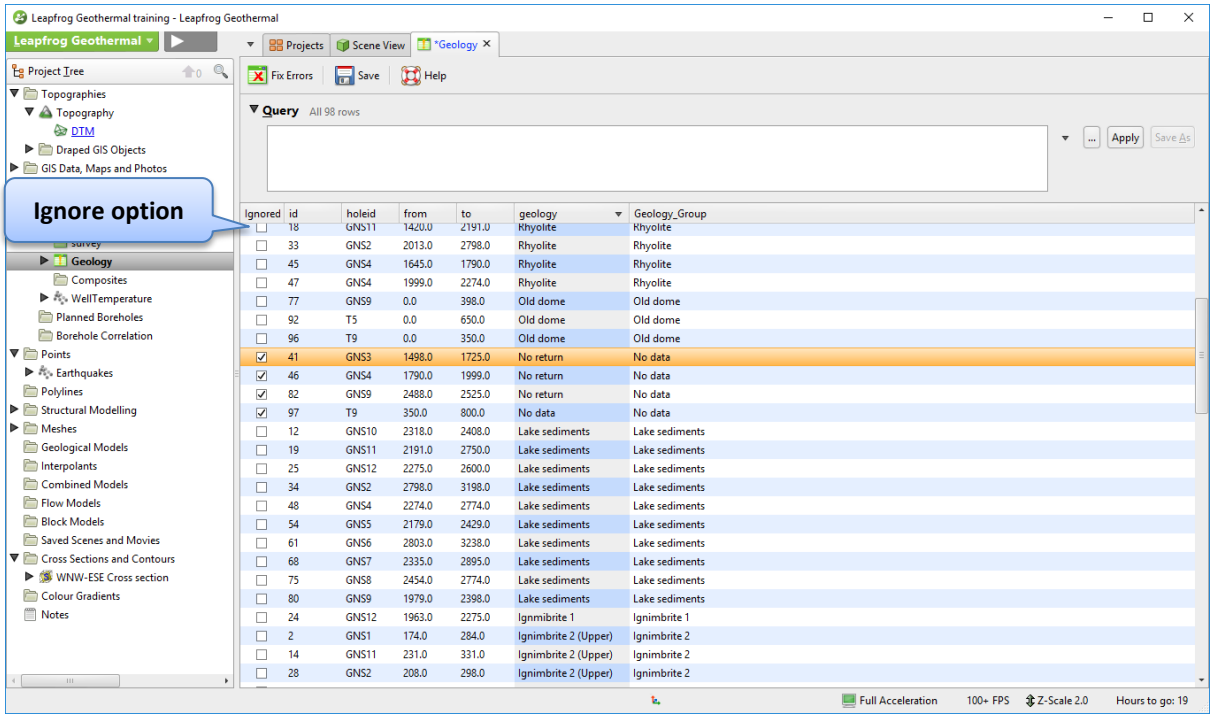



*This process illustrates the importance of starting with a "clean" database, to avoid having to fix too many errors within the interface. Use a consistent naming convention and check for misspelling to avoid duplication of lithologies. Leapfrog Geothermal provides the tool to identify errors and misspelling, but is not a database management interface in itself.*

## IGNORING DATA

In your database you may find some intervals with no data, for example due to circulation losses, or some formation you may want to ignore due to uncertainty in its quality.

- Open the **Geology** table.
- Identify the well or the interval you wish to ignore and tick the **Ignored** box.
- For the purpose of this exercise, do not ignore the segments. Discard any changes.




## CONCLUSION

In this tutorial, we imported a series of topographic and GIS data, a cross section, numeric data, structural data and various borehole data. We performed a quality check of these datasets. They are now ready to be used to build the geological model.

Do not underestimate the time you need to spend for quality check and preparation of your data. The quality of your model depends on the quality of your data.

# BUILDING A COMPLEX GEOLOGICAL MODEL: GNS DEMONSTRATION PROJECT

## TUTORIAL 5: BUILDING A FAULTED GEOLOGICAL MODEL

In this tutorial we are going to build a complex geological model including faults.

## TABLE OF CONTENTS


Structure of the GNS demonstration model....2

The Fault System....3

- Theory about fault modelling....3
- Creating faults from a surface....4
- Creating faults from a geological map....5
- Creating faults using a cross-section as a guideline....9
- Creating faults using the trending plane....10
- Fault interactions....12

Combining faults and lithological surfaces....14

- Ignoring surface data outside a boundary....14
- Final check and fault activation....14

Adjusting surfaces....17

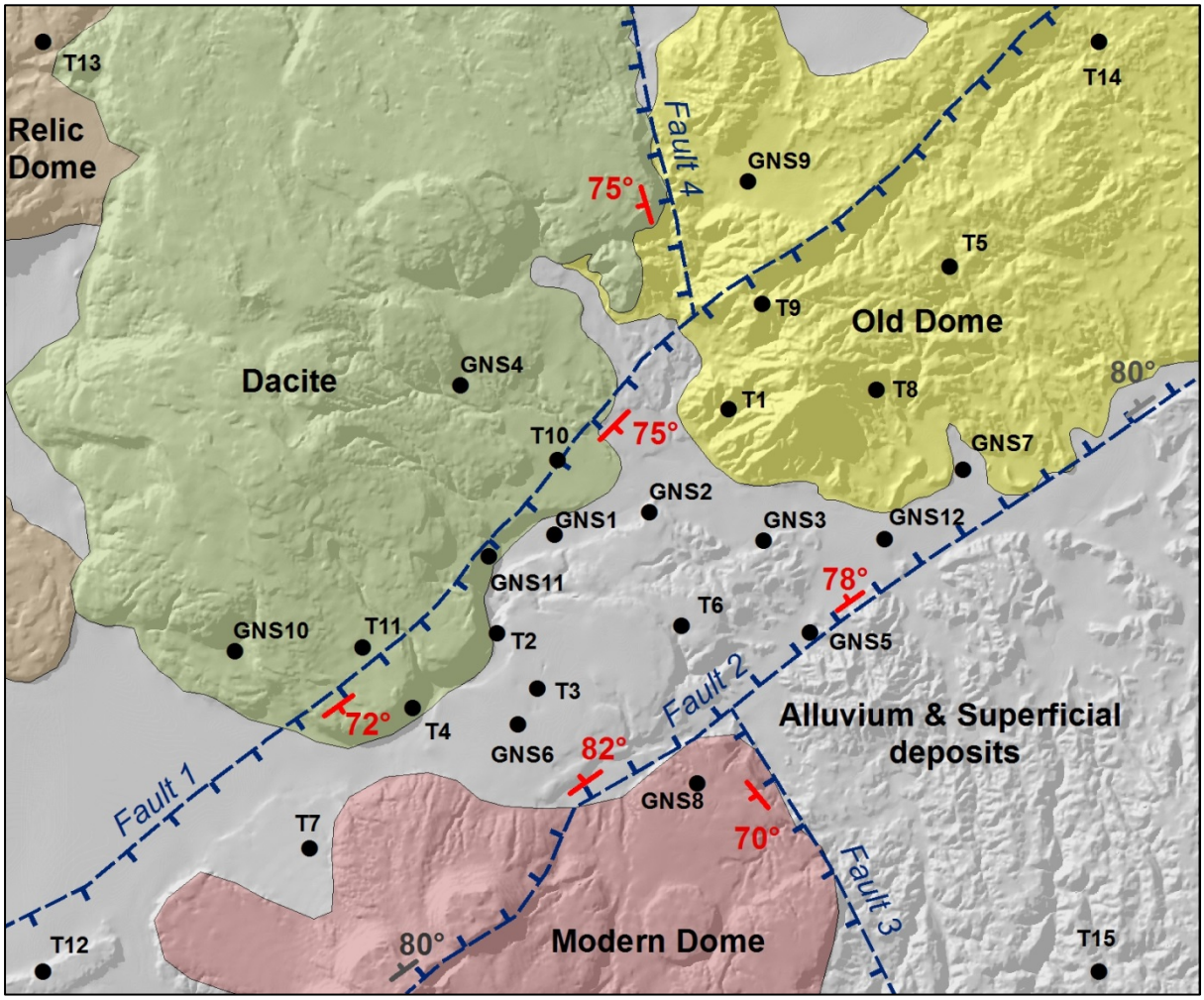
- Adjusting deposits....17
- Adjusting intrusions....22

Conclusion....25

## STRUCTURE OF THE GNS DEMONSTRATION MODEL

In the previous tutorial we built an unfaulted model of the stratigraphic sequence of the GNS demonstration field. You must now be familiar with the different formations and the surface types used by Leapfrog to realistically represent continuous sedimentary layers versus discontinuous lava bodies.

We are now going to introduce faults in the model. In a real case scenario, you would look carefully at the borehole data, identify the offsets in the different formation and integrate more data to locate your faults. Due to time constraints during this training, we are not doing a geological interpretation of the data. We will use the faults inferred locations and geometries from the geological map. We will create the faults using several techniques for you to practice with tools available within the software.



## THE FAULT SYSTEM

Faults are added and managed within the **Fault System** object of the Geological model. Faults are complex components of a geological model and should only be included when you have a good understanding of the basic uses of the software, and a good understanding of your geothermal field.

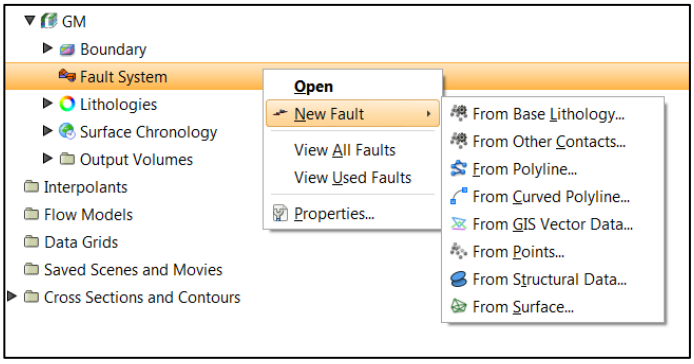
## THEORY ABOUT FAULT MODELLING

Here are a few key points to remember when creating faults:

- The **Fault System** object defines faults and their interactions.
- When a new geological model is created, the **Fault System** object is empty.
- The faults in a geological model are not active until the fault system chronology box is ticked for each fault. This means you can build your structural network without re-computing the geological model, until you are entirely satisfied.
- When the faults are activated:
    - Leapfrog will compute the structural framework according to the fault chronology and interactions: it will generate individual fault blocks on each side of the faults.
    - Leapfrog will replicate the surface chronology from the original model into all the fault blocks using the same surface characteristics. For example, if a deposit surface is created from an existing mesh, it will be created from the same mesh in the sub-models. If a deposit is formed from lithological contacts, then the software re-computes the surface using the contact points, however, by default, it will only use the points located in the fault block boundaries unless stated otherwise. Y**ou need to do some planning before faulting a model.**
- At this stage, Leapfrog does not have a common chronology table between faults and surfaces. Thus, if you create a fault, by default it will affect the entire stratigraphic column.
- If a formation is unfaulted, you need to specifically tell the software to take into account all the contact points before activating the faults.

For you to practice and use as many Leapfrog Geothermal tools as possible we will:

- Create a fault from an existing surface.
- Create a fault from a geological map using fault lines and structural measurements as guidelines.
- Create a fault using the combination of the geological map and existing cross-sections as guidelines.



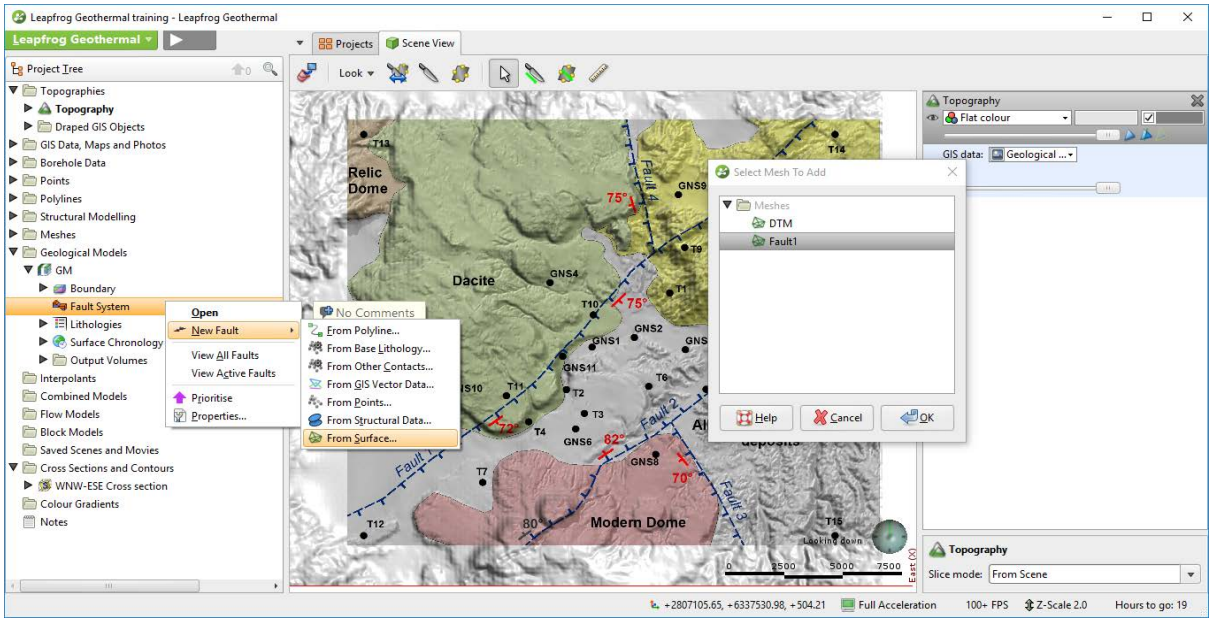

- For clarity, do not use the lateral extent in this exercise.

*The fault surfaces are colour-coded orange and pink. These colours represent the geographic side of the fault. In this example the orange shows the north-west, while the pink shows the south-east.*

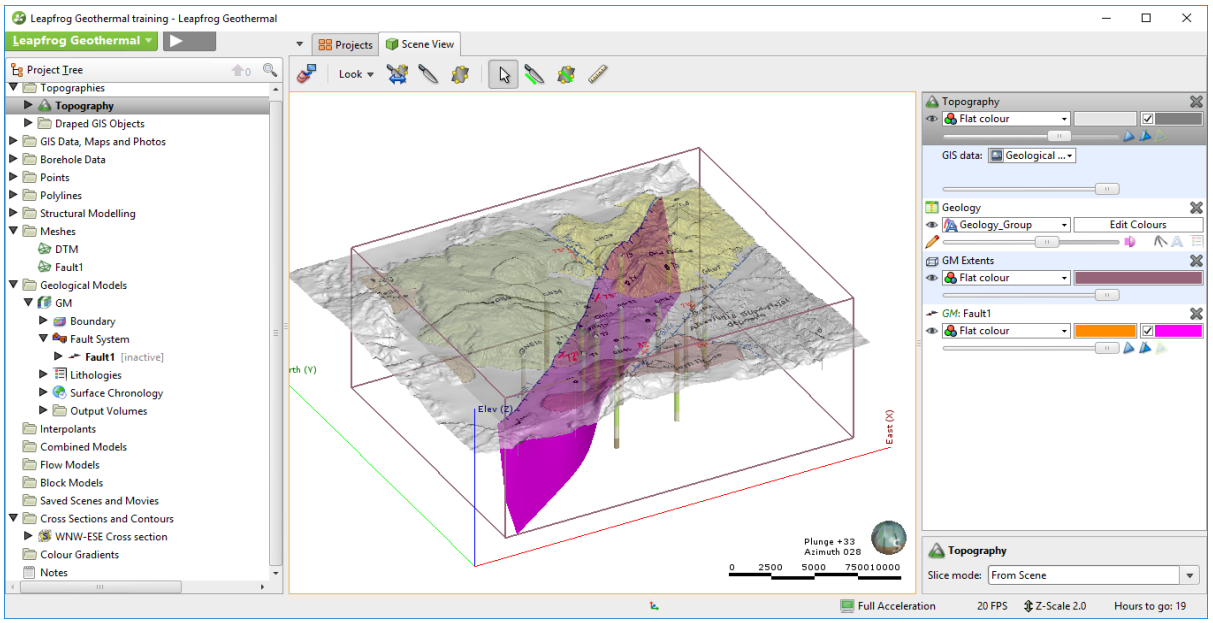
## CREATING FAULTS FROM A SURFACE

This method is used when a fault has already been created in another Leapfrog project, or in another software package. It must be imported into the project before you can use it.

- For this exercise we will use the **Fault 1** mesh that we previously imported in Tutorial 3.
- Create a new fault from Surface in your geological model, using **Fault 1** as input.




- Depending on your model extent, you may have a warning message **"Fault1" is too small**. If this is the case, select **Use 'Fault1' Vertices**. This means the mesh is smaller than your geological model boundaries and it will convert each vertex into a points, and rerun the interpolation between these points. Display **Fault 1**.

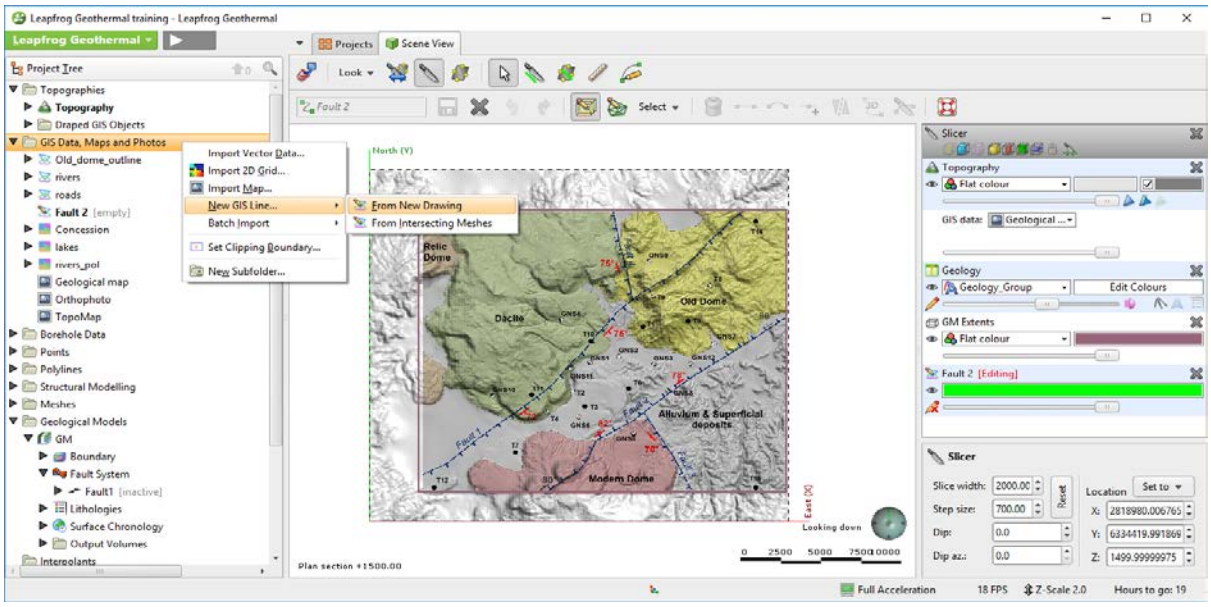



- Double-click on the **Fault System** object. As you see, the Fault 1 is not activated. Do not activate any of the faults until we have finished building the entire structural network.

## CREATING FAULTS FROM A GEOLOGICAL MAP

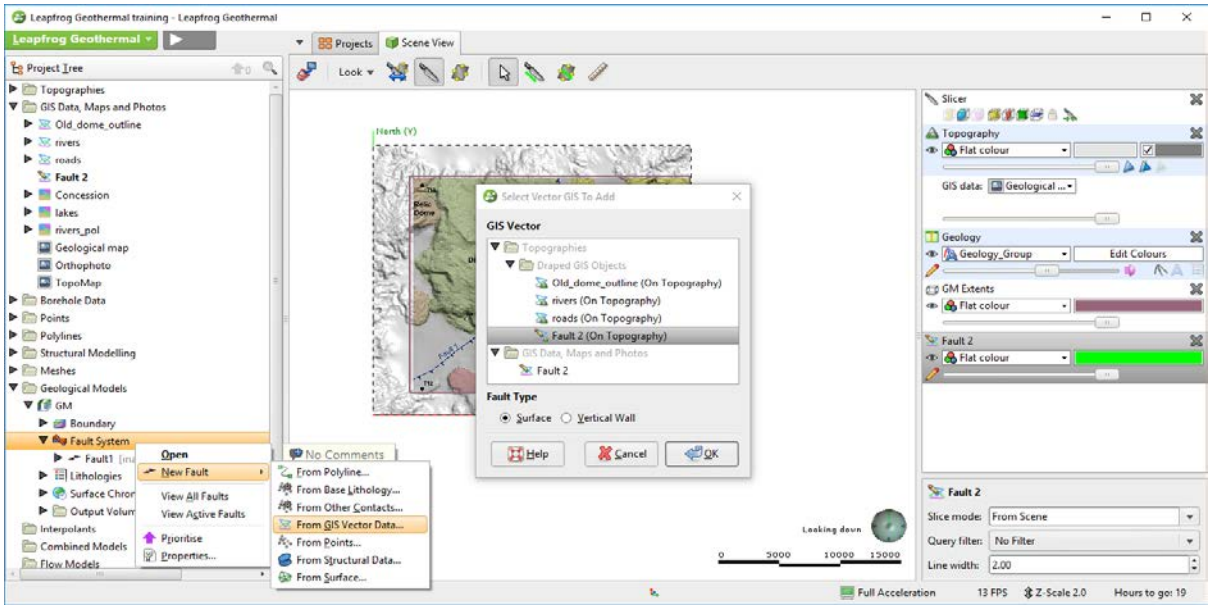
The surface geological map is used as a guideline to build this fault. You will create a GIS line to constrain the surface trace of the fault, and use the structural measurement associated with Fault 2 to constrain the dip orientation and dip angle of this fault.

- From the **GIS Data, Maps and Photos** folder create a new GIS line. By default Leapfrog will bring in the slicer and slice the model horizontally and set the view in Map view (looking down). The drawing toolbar is now active. Display the geological map into the 3D scene if not already present.
- Draw **Fault 2**, following the geological map trace. Alternatively, draw on the **Topography** object.

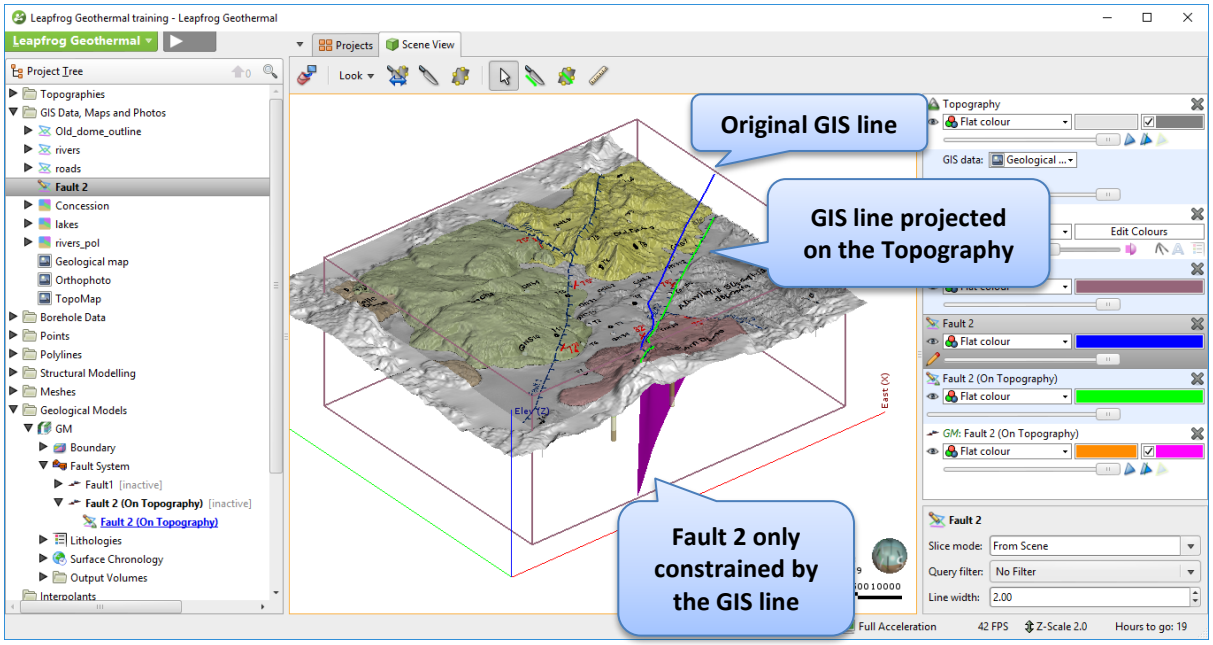



*Note: the same rules apply to draw a polyline or a GIS line.*

- In the **Geological Model**, create a new fault **from GIS Vector Data**.
- Make sure to select the **Fault Type** as a **Surface** (which means it will have a dip angle), and not a vertical wall. Select the **Fault 2 (On Topography)** object and click **OK**.

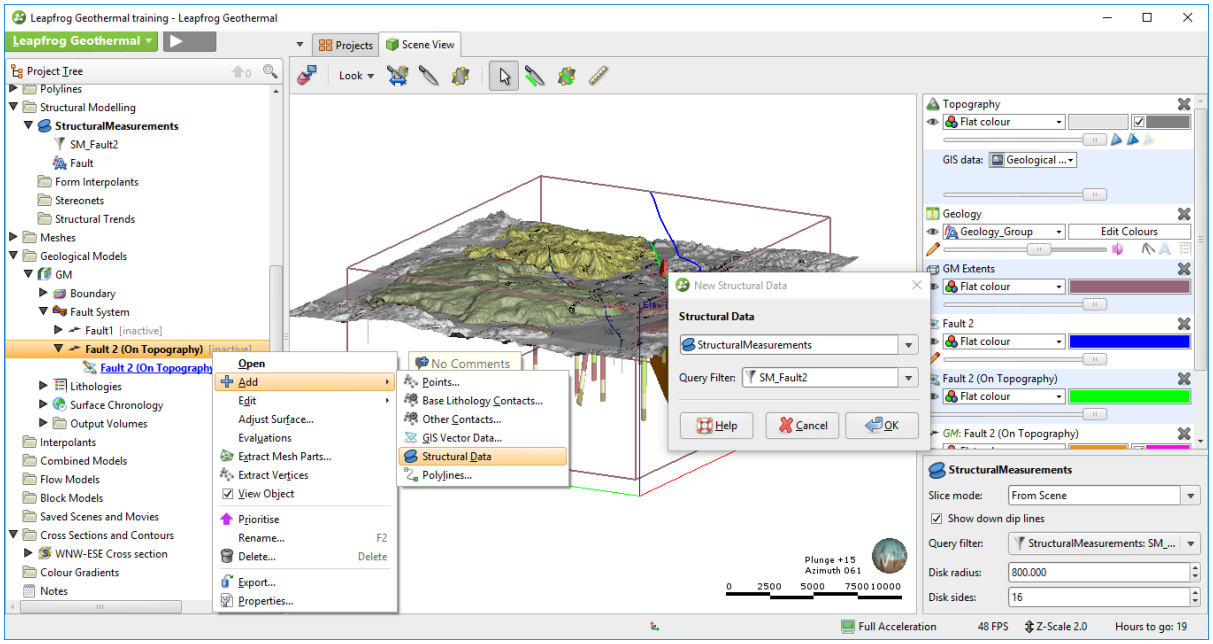


- Visualise the new surface. Its geometry at depth is currently unconstrained.




We will now constrain the dip angle of the fault using structural measurements. We imported structural measurements for all the faults in Tutorial 3, however, we need to isolate the structural measurements for Fault 2 to be able to use it as input for this specific fault. The easiest option is to create a query to isolate Fault 2 and use the result of the query to constrain the dip angle.

- Apply a new query filter on the **StructuralMeasurements** object.
- To adjust the surface of **Fault 2** using the structural measurement right-click on Fault 2 in the Project Tree and select **Add/Structural Data**.
- Select **StructuralMeasurements,** use the Query Filter and click **OK**.



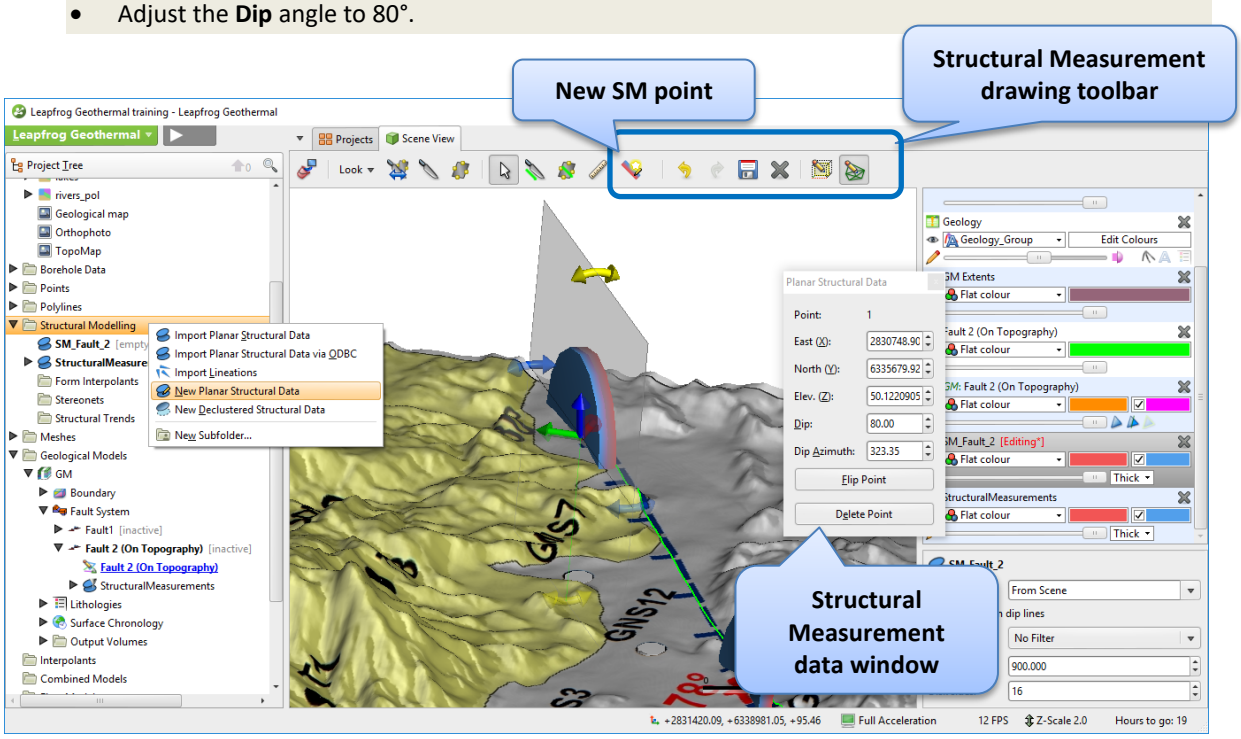
- Double-click on Fault 2 in the project tree and turn the Boundary filter OFF. This will ensure the structural measurement are taken into account even if sitting slightly above topography level.
- Check the geometry of Fault 2. The dip angle/direction is now constrained by the structural measurement.

For practice, this exercise will also take you through the steps involved in creating new structural measurements. We will add structural measurements at the end of the fault line.

- Display the geological map draped on the DTM.
- In the **Project Tree**, right-click on the Structural Modelling folder and select **New Planar Structural Data**.
- Enter a name such as "**SM Fault 2 new data**" and click **OK**.

The structural data toolbar appears above the scene along with a structural data window.

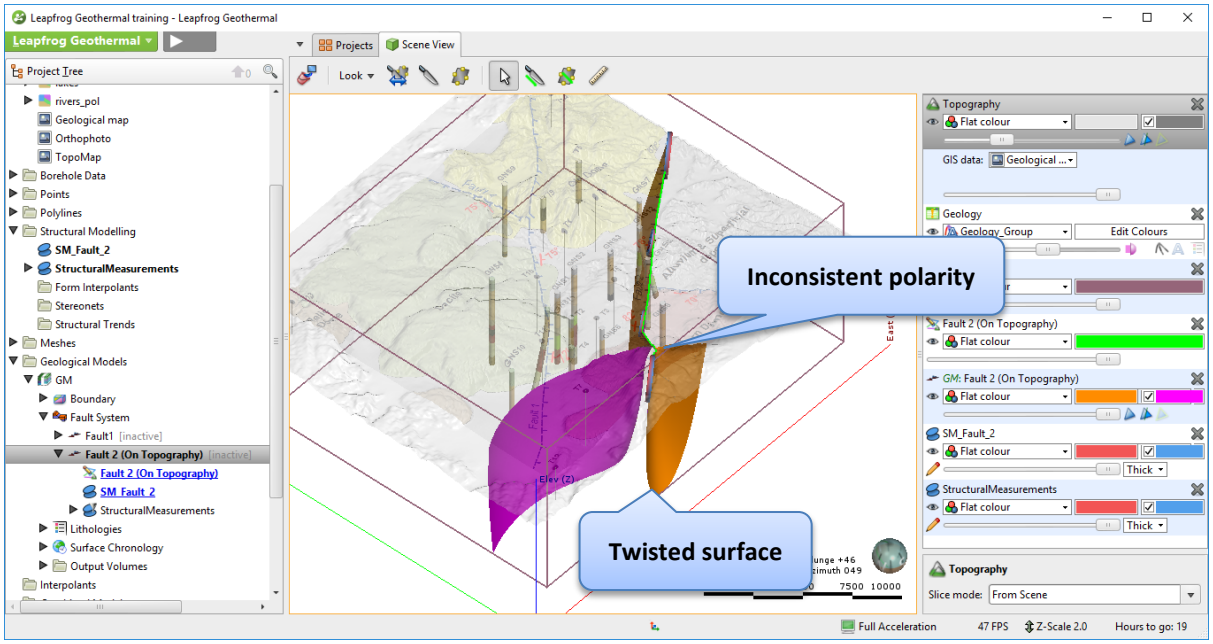
- Zoom in to one end of **Fault 2**. Use the grey structural measurements as a guide.
- Rotate the scene to look in the direction of the dip (dipping west in this example).
- Check that the **Draw on objects** icon is selected in the new toolbar. It will allow you to draw the new structural measurements points directly on the DTM (the other option would be to draw on the slicer).
- Click on the **structural data point** icon and draw the point in the scene by clicking on the centre of the strike and dip, and drag it along the strike direction.
- Adjust the **Dip** angle to 80°.



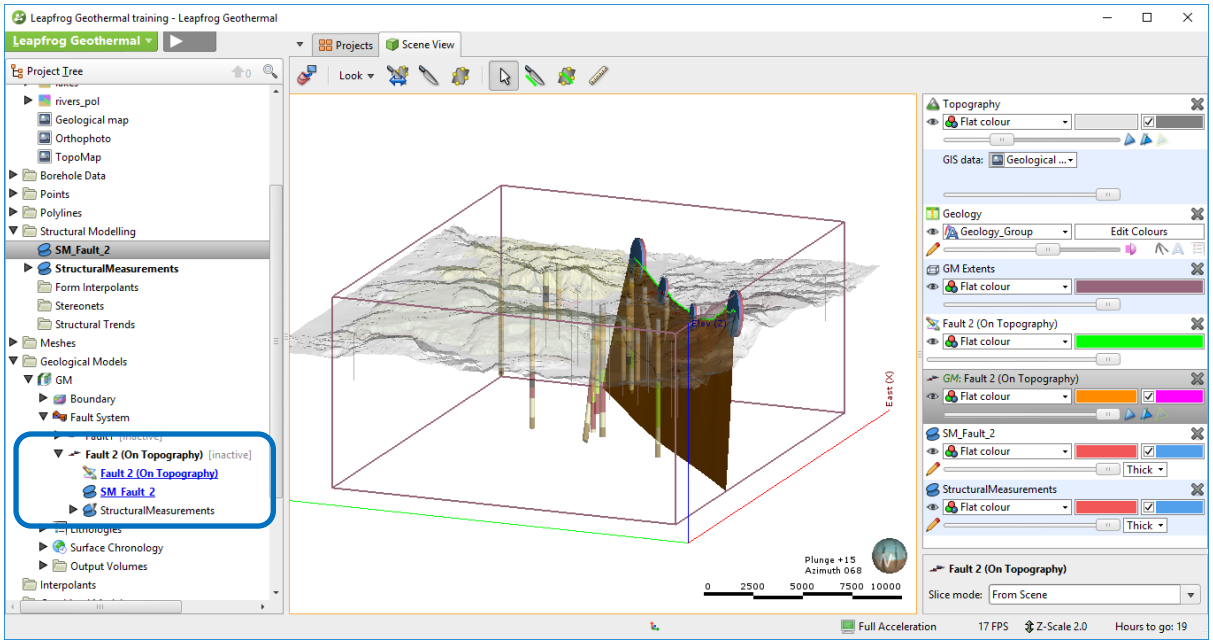

*Draw structural data points in the same direction to ensure the polarities remain constant. Use the Flip Point option if one is not properly orientated.*

- Draw a second point at the other extremity using the same method and save your edits.
- To adjust the surface of **Fault 2** using the new structural measurement right-click on **Fault 2** in the **Project Tree** and select **Add/Structural Data**.
- Display **Fault 2**.

If the fault surface is twisted, check the orientation of your structural measurements. They must be consistent between the two datasets. Edit your structural measurement if needed.



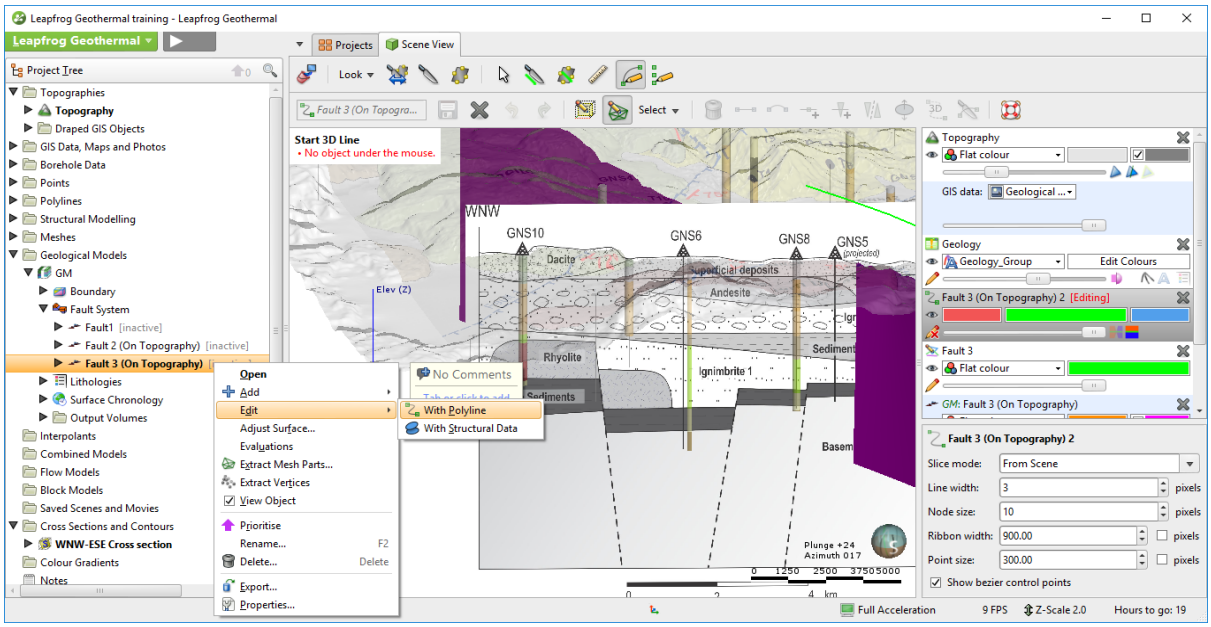

As you can see the fault is now honouring the GIS line at surface as well as honouring the structural measurements information (dip azimuth and dip angle).

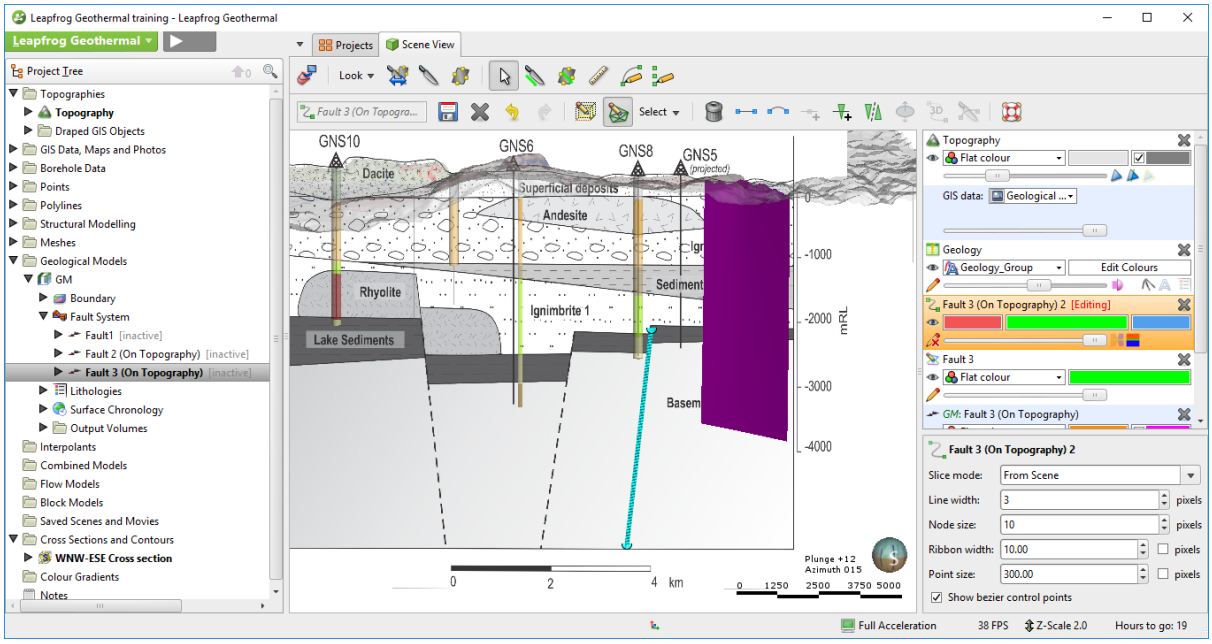


## CREATING FAULTS USING A CROSS-SECTION AS A GUIDELINE

To create Fault 3 we will use the geological map and the cross-sections as guidelines to build the fault geometry. This time, and for practice purposes, we will combine the use of a GIS line (surface trace) and a polyline (fault location at depth).

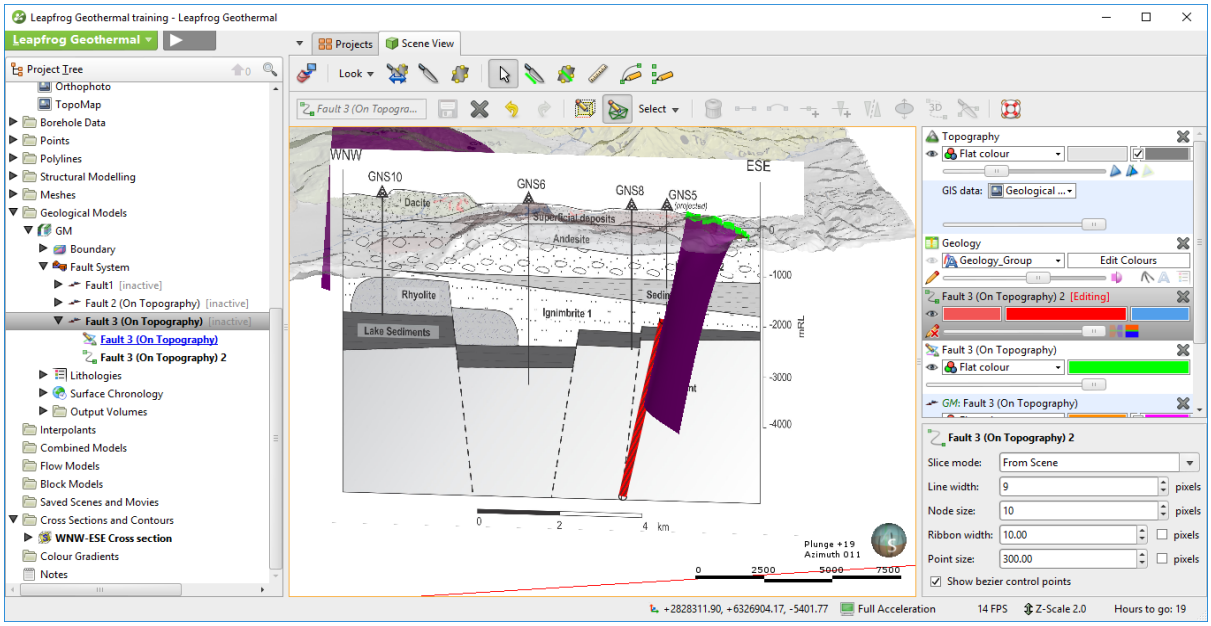
- Clear your scene and bring the **WNW-ESE cross-section** we imported in Tutorial 3 and the **geological map** (draped on the DTM) into the 3D scene.
- Create a new GIS line for **Fault 3** and create a new fault based on this GIS object. The fault will have a dip angle so make sure to select a **surface** and not a vertical wall. Select the GIS line projected on **Topography** (this is representing its true location in space).
- Right-click on **Fault 3** in the **Project Tree** and select **Edit with polyline**.
- Draw a polyline representing the fault at depth as per the cross-section trace. You can draw on the cross-section directly, but make sure to select the draw on object icon.



- Save your edits and display **Fault 3**.

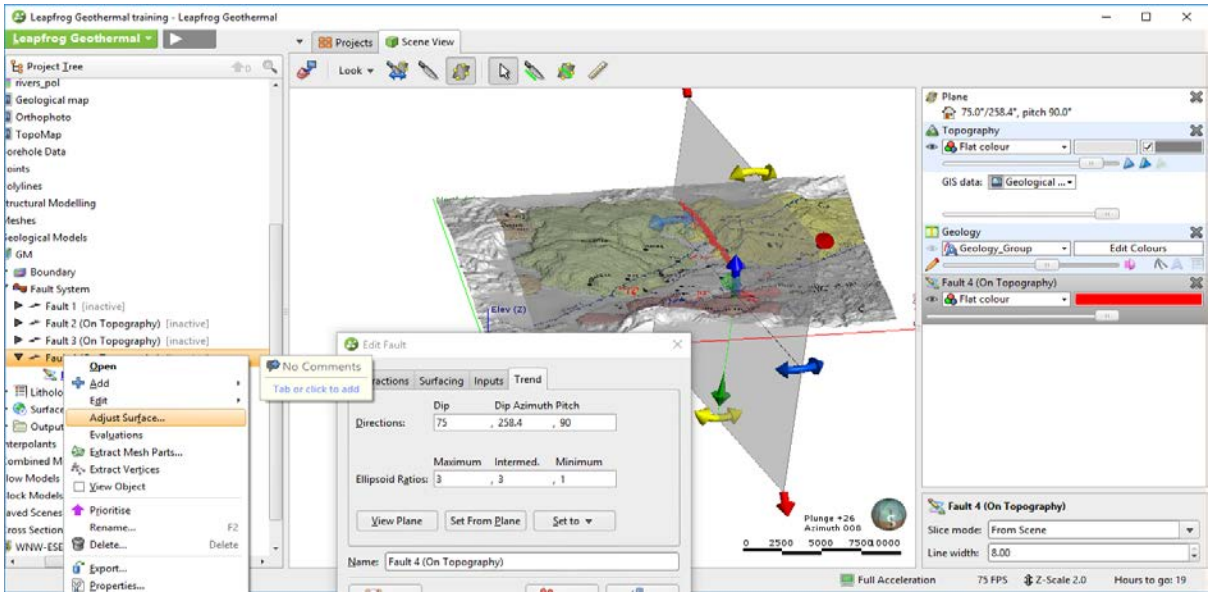
You will notice that this method is approximate and the plane may not look perfect as it is constrained by the vertices of two disjointed lines. You can adjust the polylines location and the trend until you are satisfied.



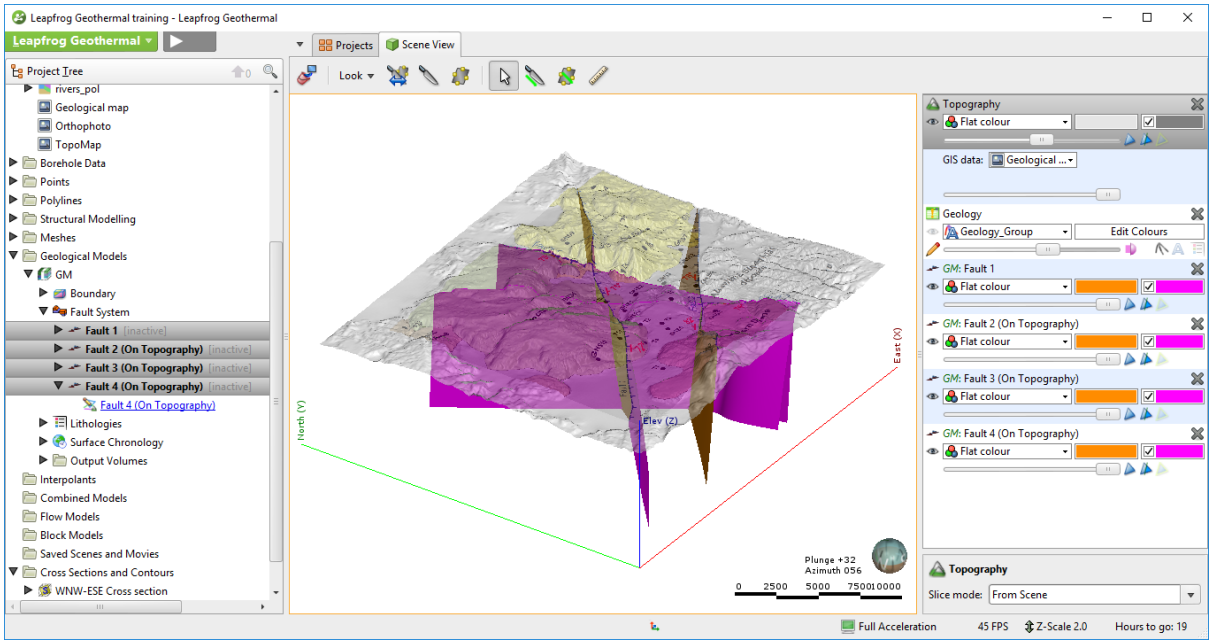

## CREATING FAULTS USING THE TRENDING PLANE

To create Fault 4, we will use the quickest technique available in Leapfrog to generate a dipping surface. We will constrain Fault 4 using a GIS line and constrain the dip angle using the trending plane tool.

- Create a new GIS line for **Fault 4** and create a new fault based on this GIS object.
- Look down and draw a plane line parallel to your fault trace.
- Rotate your view and click on the **plane**. Use the handles to incline the plane towards the west, and adjust the dip angle from the plane property window.
- Right-click on **Fault 4** in the Project Tree and select **Adjust Surface**.
- Click on **Set From Plane**. Make sure the parameters are updated and click **OK**. Check **Fault 4**.



- You can now look at the four faults. Each surface is extended up to the model boundaries at this stage.




*You can edit your faults at any stage in the modelling process.*

*In a real case scenario, try to avoid self intersecting faults at depth. This would add complications and unecessary fault blocks to your final model.*

## FAULT INTERACTIONS

Now that all the faults have been created, we can look at the chronology and the interactions between these faults.

- Double-click on the **Fault System** object.

In this window you will get access to:

- The **Fault system chronology**: you can re-order the faults.
- The **Interaction Type** menu: it is active for all faults except for the youngest one.
- Add an interaction and have a look at the options. The oldest faults can either:
  - Cross-over the youngest faults.
  - Terminate against the youngest faults, in which case you have to choose on which side it is terminating.
- DO NOT ACTIVATE THE FAULTS (the model is not ready to be faulted yet).

*If there are any inconsistencies in your fault interactions and chronology, a warning message will pop up.*

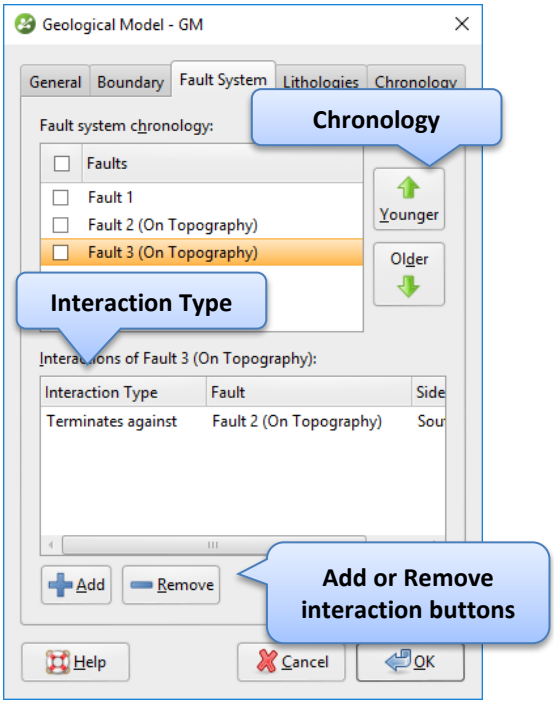
In this example, we do not have an established chronology of the fault system. However, looking at the geological map, we can identify spatial relationships between faults that will influence the chronology order:

- Fault 3 terminates on Fault 2.
- Fault 4 terminates on Fault 1.
- Fault 1 and Fault 2 do not interact.
- Fault 3 and Fault 4 do not interact.

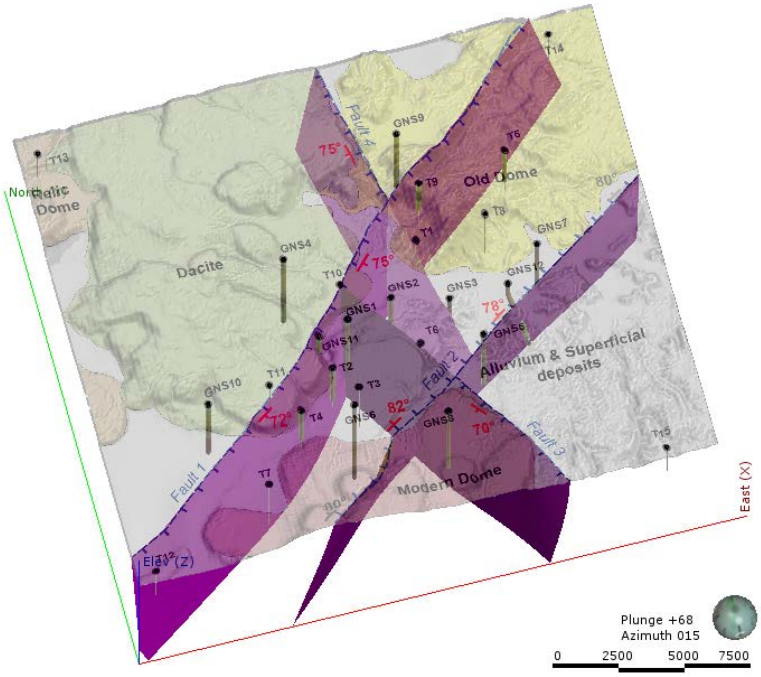
We can conclude that Fault 1 and Fault 2 should be considered as the youngest faults that will affect the oldest ones.

For practice, we will look at two different interaction scenarios. Make sure that you are not activating the faults while doing this.

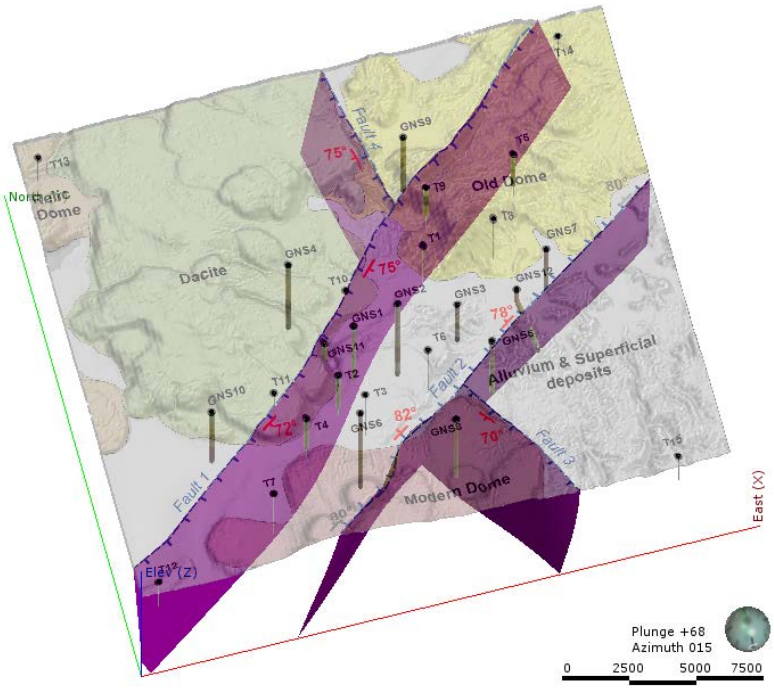
- Re-order the chronology of the faults, making sure that **Fault 3** and **4** are the oldest ones, and **Fault 1** is the youngest.
- Click on **Fault 3** in the fault system chronology window.
  - **Add** an interaction.
  - Select **Crosses over** Fault 2.
  - **Add** an interaction.
  - Select **Terminates against** Fault 1 on South East side.
- Click on **Fault 4**.
  - **Add** an interaction.
  - Select **Crosses over** Fault 1.
  - **Add** an interaction.
  - Select **Terminates against** Fault 2 on North West side.
- Make sure that the faults are not activated (the check box are unticked) and click **OK**.



- Once Leapfrog is finished computing, right-click on the **Fault System** object and select **View All Faults** if not already displayed.




- Now that you understand the principle, adjust your fault interactions in order to match the geological map as per the following image.



## COMBINING FAULTS AND LITHOLOGICAL SURFACES

As we have mentioned earlier, faults will divide the original geological model in fault blocks and will replicate the stratigraphic sequence as built in the original model in each of the blocks.

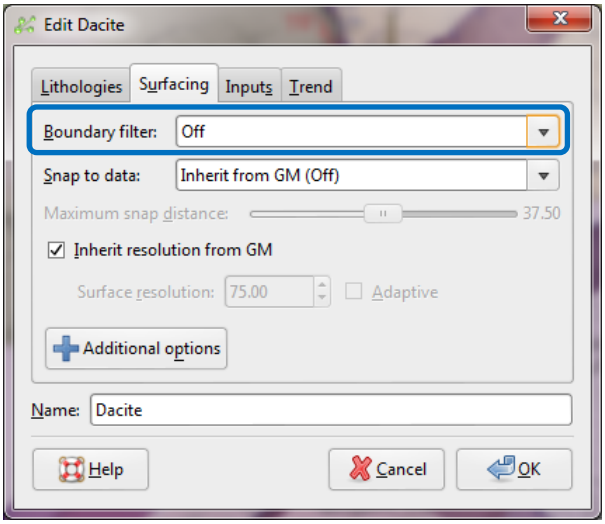
- As you know, we have built all our surfaces using contact points. The unfaulted model takes into account all the contact points from all the boreholes across the geological model boundary.
- Once faults are activated, Leapfrog will rebuild these surfaces using the contact points. However, once faulted, the geological model fault blocks have a smaller boundary (between faults), and the software will by default only use the contact points within this reduced boundary. This means that even though some surfaces might not be faulted, they will still not coincide from block to block as the input data to create them is different.

In the case of this GNS demonstration project only the Basement, the Lake Sediments and the Rhyolite formations are displaced by the faults. The solution is to let Leapfrog know that it needs to use ALL the contact points.

## IGNORING SURFACE DATA OUTSIDE A BOUNDARY

For the unfaulted formations, we need to specifically tell Leapfrog to use all the points within the entire model, not only the points within individual blocks. It is important to do this before activating the faults, otherwise you would need to go in each block to do so at a later stage.

- Double-click on each surface within your geological model (deposits, erosion and intrusions), except the **Basement**, the **Lake Sediments** and the **Rhyolite** surfaces, and set the B**oundary filter** to **Off** within the **Surfacing** tab.




## FINAL CHECK AND FAULT ACTIVATION

*When you work on your own project and wish to test different scenarios, it would be appropriate to make copies of your model. You can do this by right-clicking on the geological model and selecting* ***Copy****. This option will make an exact active copy of your model. Alternatively, you can create a* ***Static Copy****, however you will not be able to update the copied model any further.*

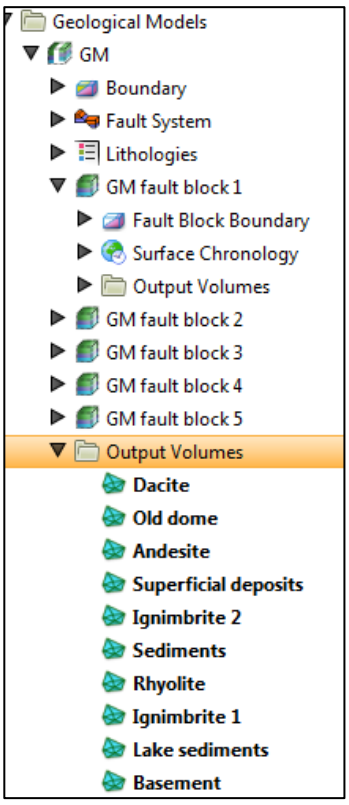
*You can only copy models within one Leapfrog project; you cannot copy active models between projects. Would you wish to visualise an existing model in another project, you can export the model. It will not be editable.*

*It is also advised to back up the project at regular intervals, especially when you know that you will add major changes to it. Within the Leapfrog Geothermal menu,* ***Save A Copy****.*

- In the **General** tab of geological model property window, change the Volume generation option to **"do not merge output volumes"**. This will allow to identify the different fault blocks when clicking on a volume.
- In the **Surface Chronology** window, check all the surfaces to compute the volumes.
- Finally, open the **Fault System** object and activate the faults.

Leapfrog will take some time to compute. It is building a full model now.

- Expand your geological model in the Project Tree.



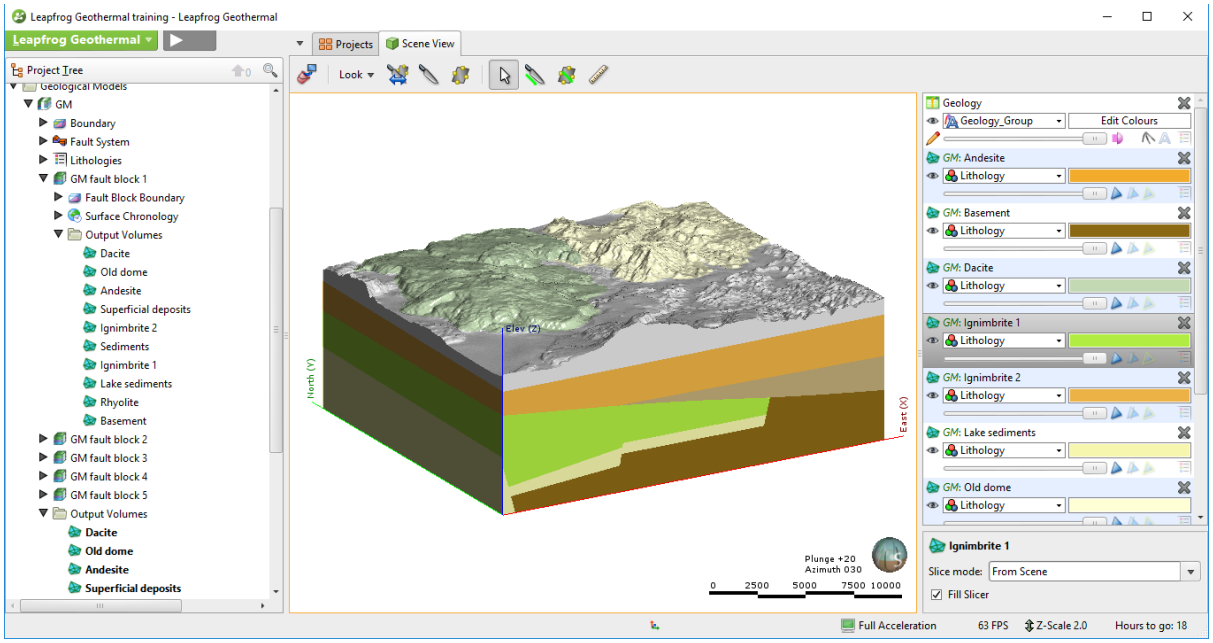

You can see that it has been subdivided in fault blocks. Expand the fault blocks and notice that each one of them is a geological model in itself (including the objects Boundary, Surface Chronology and Output Volumes).

In this model five fault blocks are created. When we display the full model, we will now see one volume per formation.

*Visual tricks:*

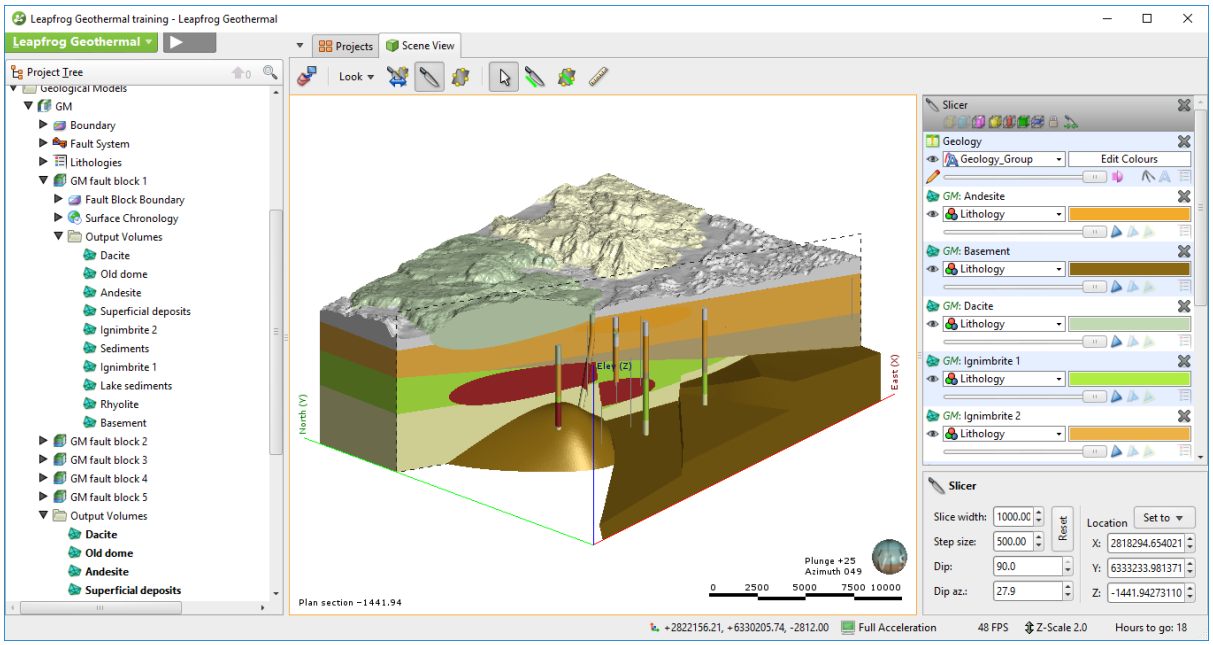
- *If you want to see the full model, with only one volume per formation: right-click on the geological model and select **View Output Volumes**.*
- *If you want to see the surfaces used to create these volumes: select **View Surface Chronology**.*
- *If you want to see the different block extents: select **View Fault Blocks Boundaries**.*
- *If you want to see the surfaces or volumes of one individual fault block: right-click on the geological model fault block of interest and select **View Surface Chronology** or **View Output Volumes**.*

- Spend some time looking at your model, per fault block or as a whole, visualising surfaces and volumes.




*You may get warning messages popping up while it is building the model. Some of the surfaces are not continuous across the entire model, especially the intrusions. If a formation is not present in one of the faulted block, an empty volume will be generated and a warning message will appear. Click OK.*

If you look closely at the generated volumes, you will notice that the model is not perfect, especially for the deepest formations. We will now spend some time editing the surfaces to make the model geologically coherent. The most important at this stage is to confirm that the unfaulted surfaces are identical across the different fault blocks.



## ADJUSTING SURFACES

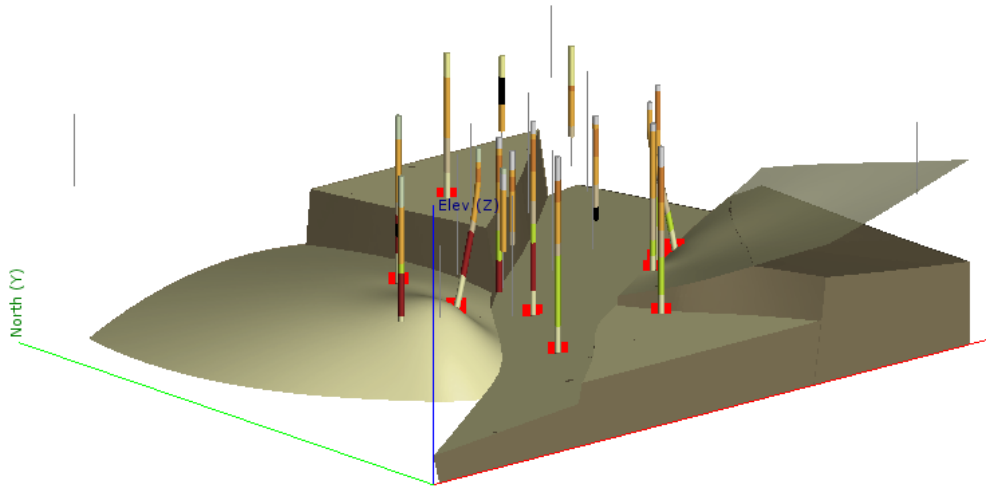
Now that the model is built and faulted, we can look at adjusting some problematic surfaces: the Basement and the Lake Sediments. Remember that they are built using lithology contacts.

- By default, the software is only using the contact points within each fault block boundary. As a consequence it generated different surfaces in each faulted block.
- In a perfect situation, we would have enough contact points to constrain the surfaces adequately within each block. In reality, we don't often have enough contact points, so we will need to add some edits to force the basement surfaces to behave properly.
- Leapfrog Geothermal honours the contact points from the borehole data and the edits manually entered by the modeller. However, the two inputs are kept separately and if the interpretation changes or more factual data become available, user inputs can be modified or deleted at any time without affecting the source data.

## ADJUSTING DEPOSITS

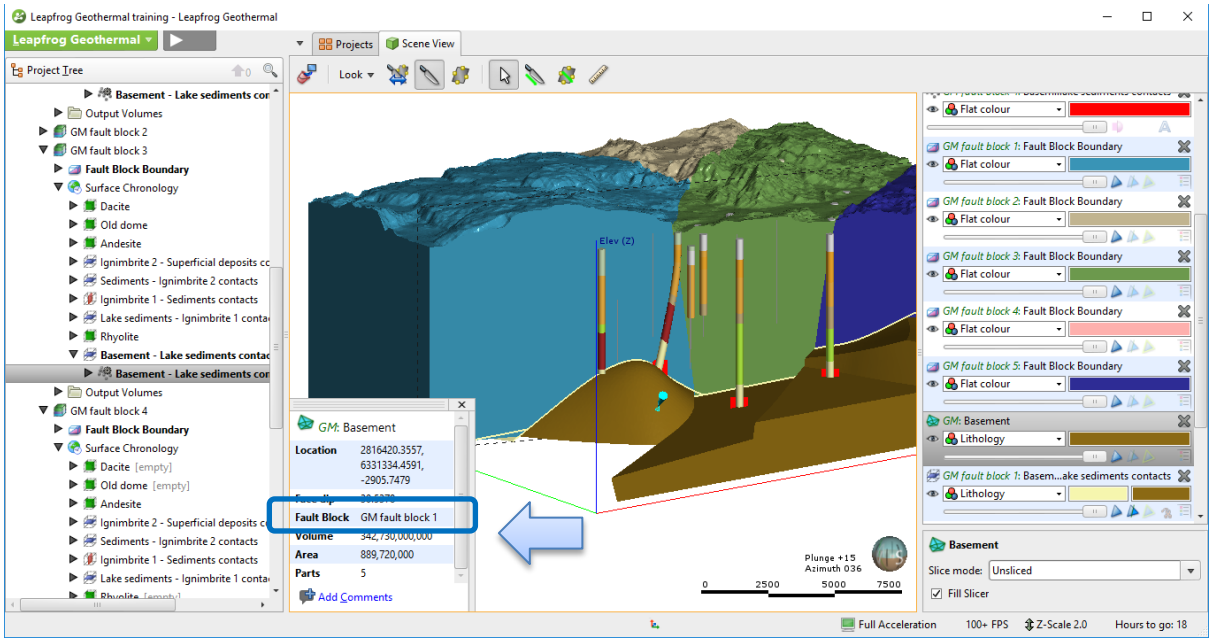
- Clear the scene and drag the borehole **geology** in the 3D scene.
- Display the main **Basement volume**.
- Display the **Basement surfaces** and **Basement contact points** from each model fault block. Clip the surfaces for a better display.
- Spend some time looking at the surfaces and contact points.

As you can see, two of the blocks will need to be edited to create a coherent geological model.



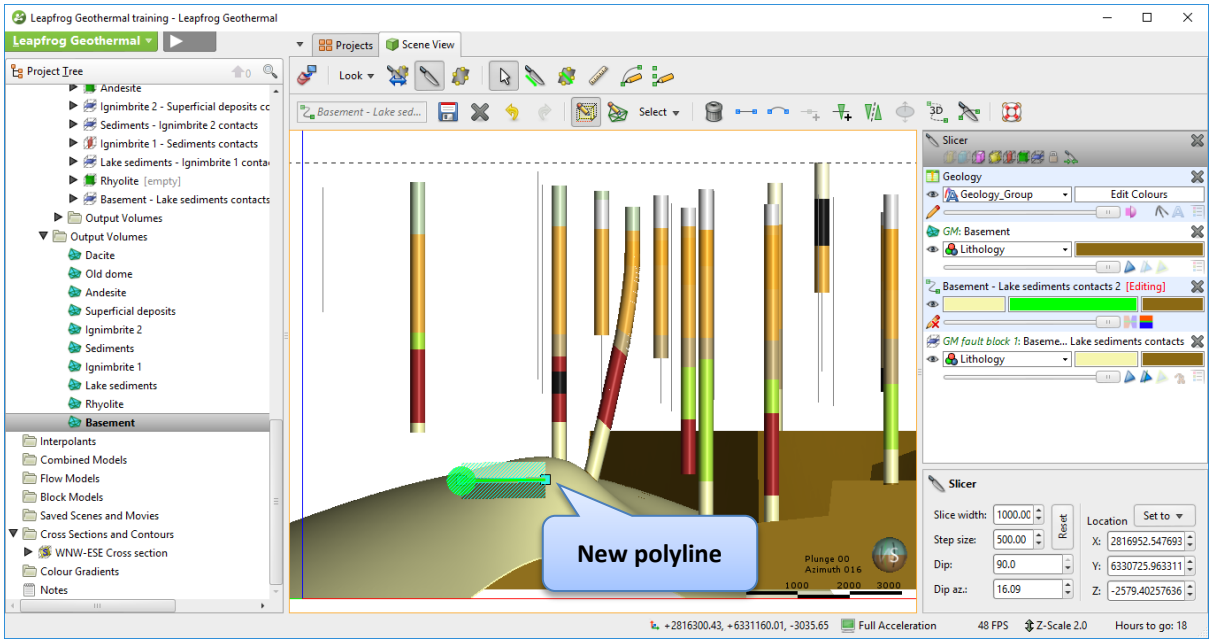

These blocks have contact points from deviated wells. Leapfrog is accommodating the best possible surfaces between a dipping contact (deviated well) and the nearby horizontal contact points from vertical wells and consequently creates curved surfaces.

- You need to identify the fault blocks that need editing by either:
  - Displaying the fault block boundaries;
  - Clicking on a contact surface, contact point or volume and look at the properties.

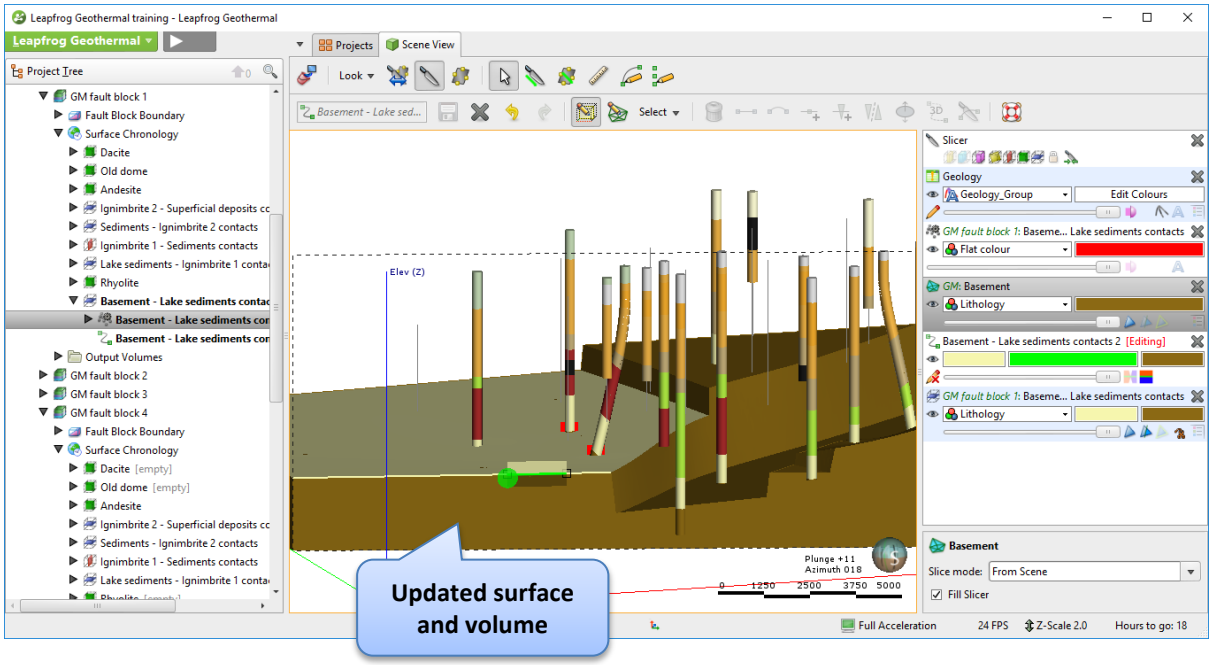



- Once you have identified the fault blocks, edit the **Basement surface** knowing that we aim at sub-horizontal surfaces. Use the following two options to edit the two basement blocks that need to be adjusted.

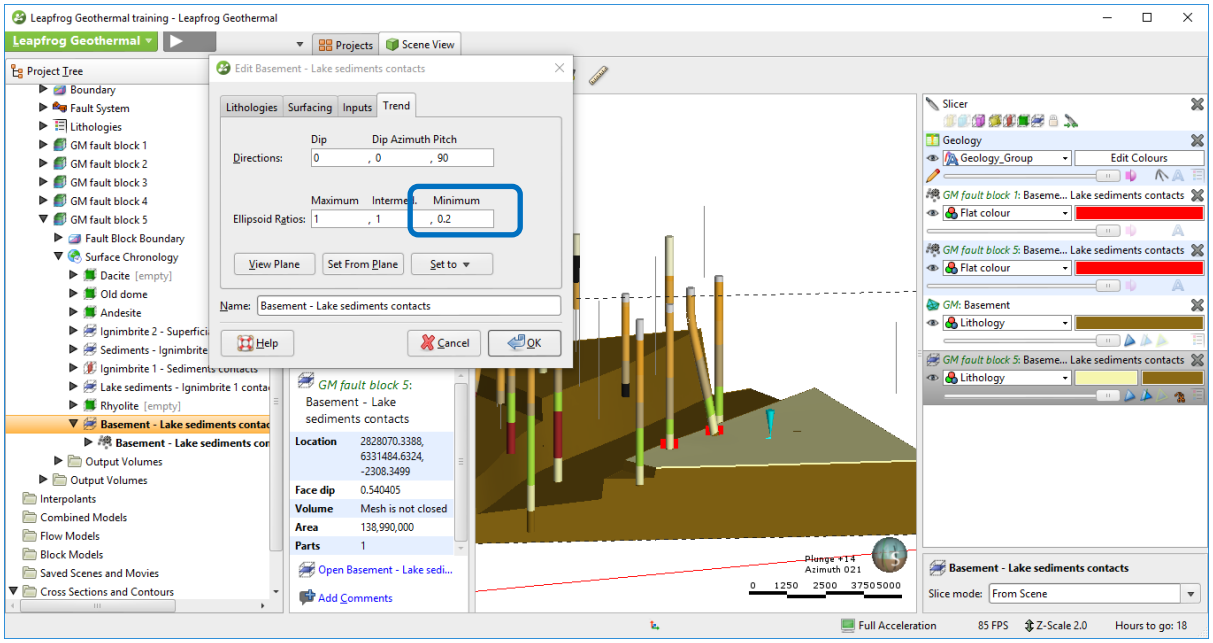
- **Option 1**: Create a polyline on the edges of the model boundary to constrain the surfaces.
  - Right-click on the deposit surface and select **Edit/With Polyline**.
  - Look down and use the **draw slicer line** tool to slice the model vertically.
  - Look straight into the slicer plane (type "L" or "SHIFT L" on your keyboard).
  - Draw a polyline at the required level using the existing contact points as a guideline.



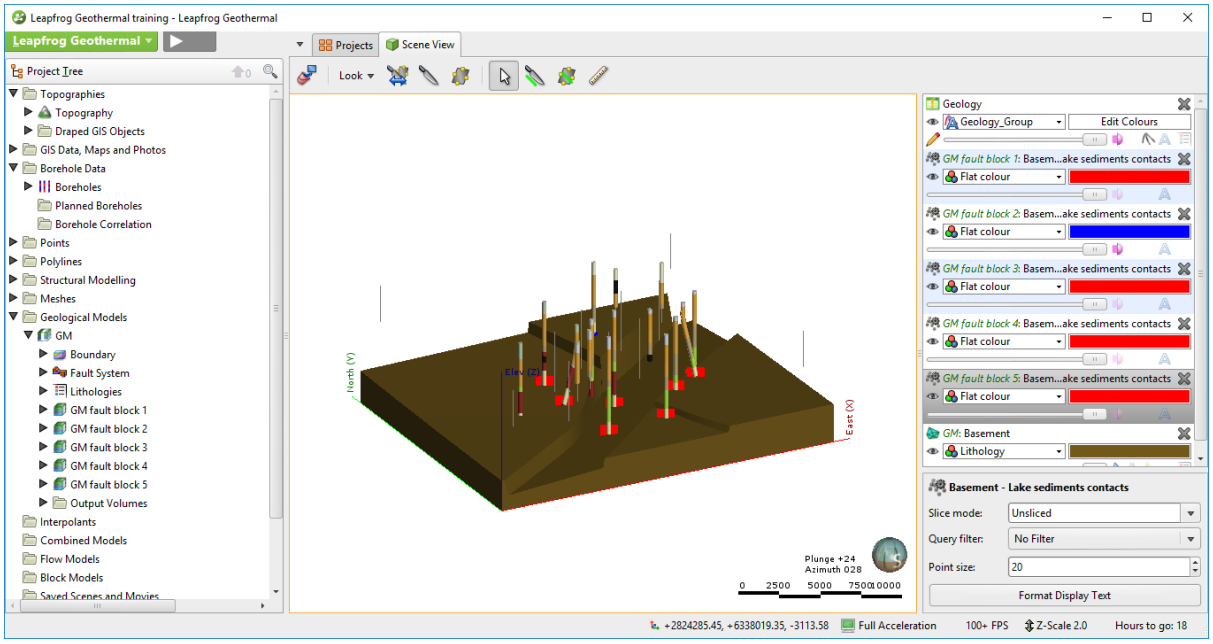

- Save your edits and check the updated surface/volume.



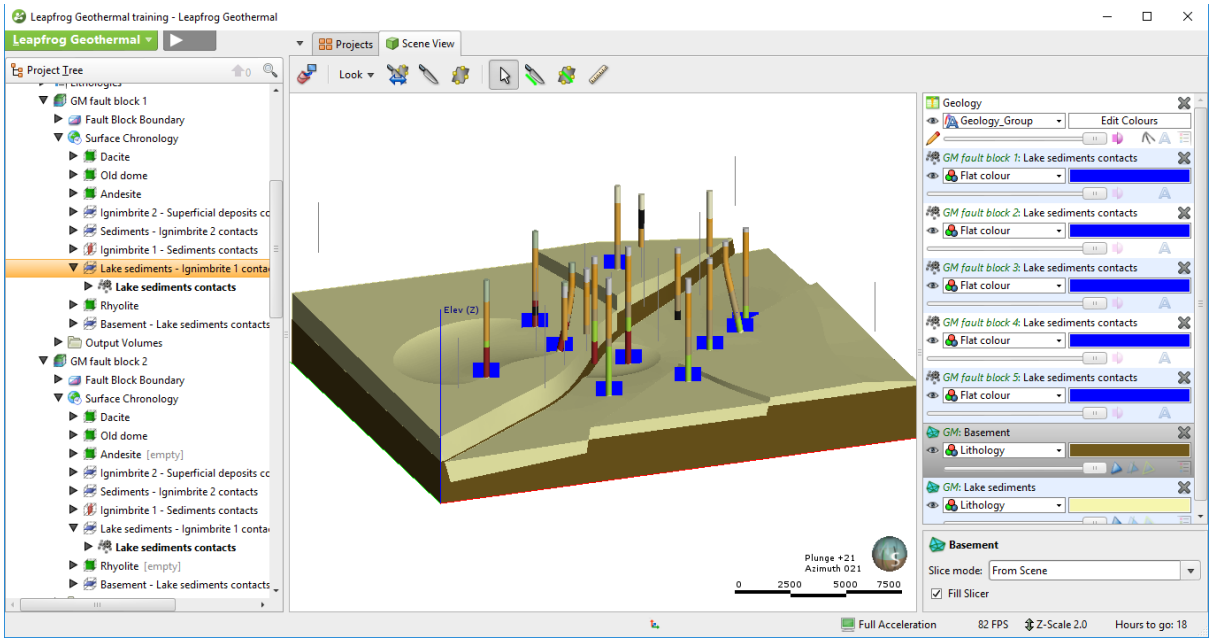
- **Option2**: Adjust the surface to elongate it horizontally.
    - Right-click on the **Basement surface** in the appropriate fault block.
    - Select **Adjust Surface**: Reduce the **Minimum Ellipsoid Ratio** value to force the surface horizontally.



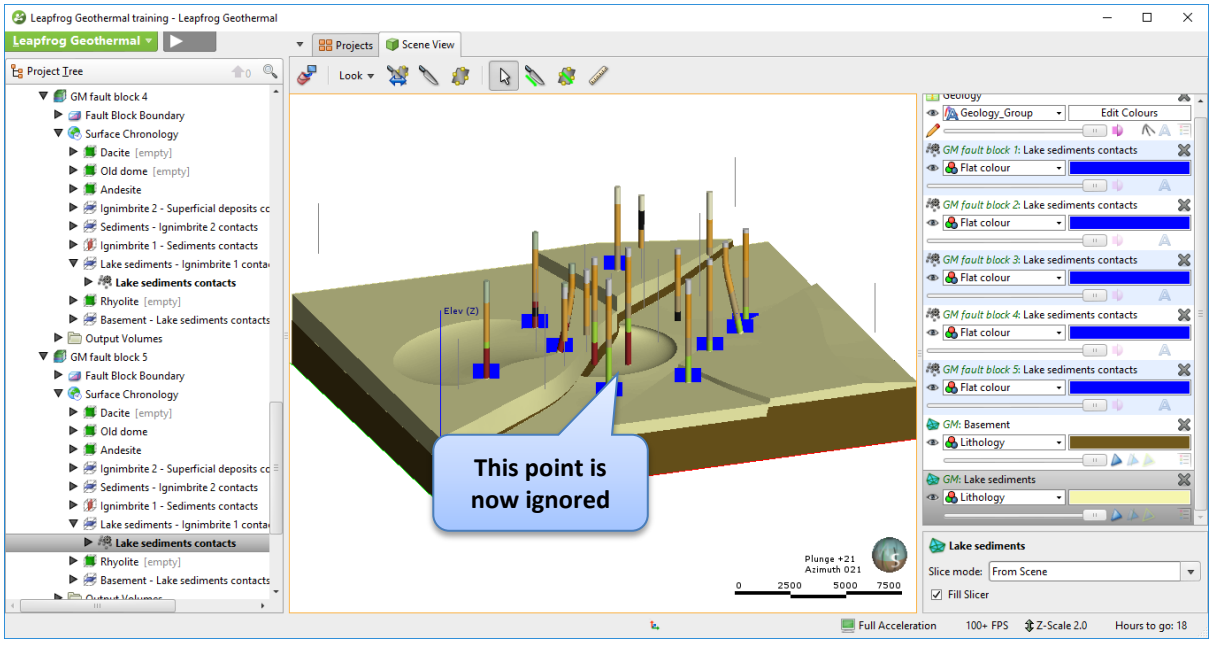

- The final **Basement** should look similar to the following image.




- If time allows, follow the same methodology for the Top of the **Lake Sediments**.



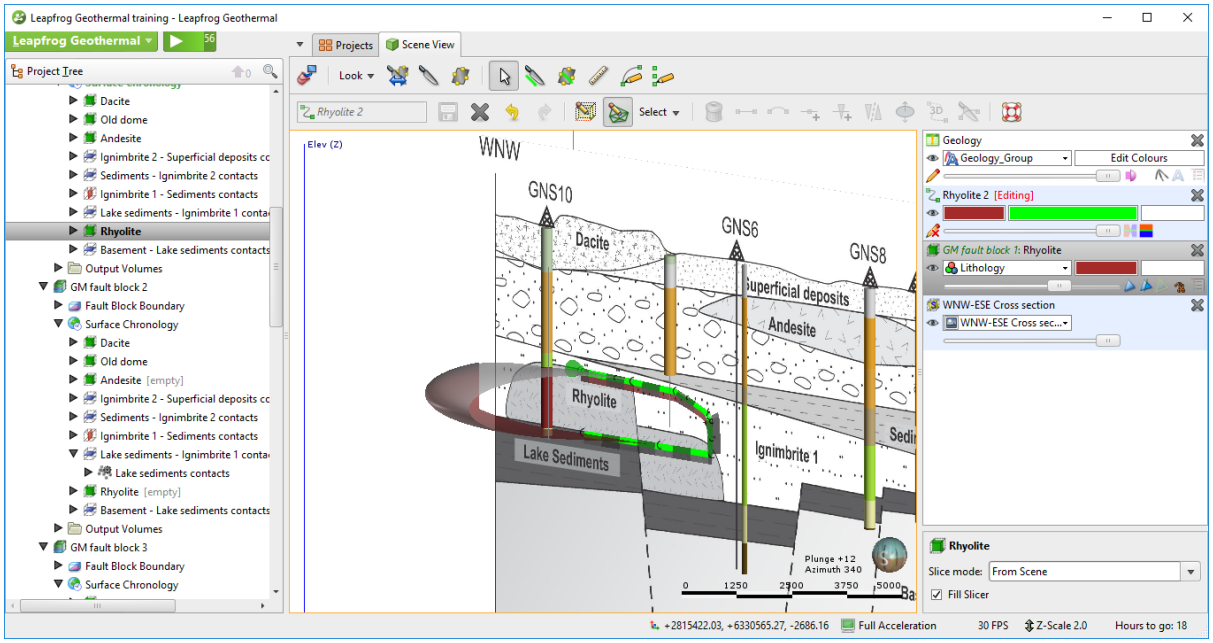

While analysing your model, remember that you can modify the input data used to build surfaces at any time. For example, it would be relevant to ignore the contact point with the rhyolite for the top surface of the Lake sediments within the graben structure. The sediments may have accumulated around the rhyolite dome.



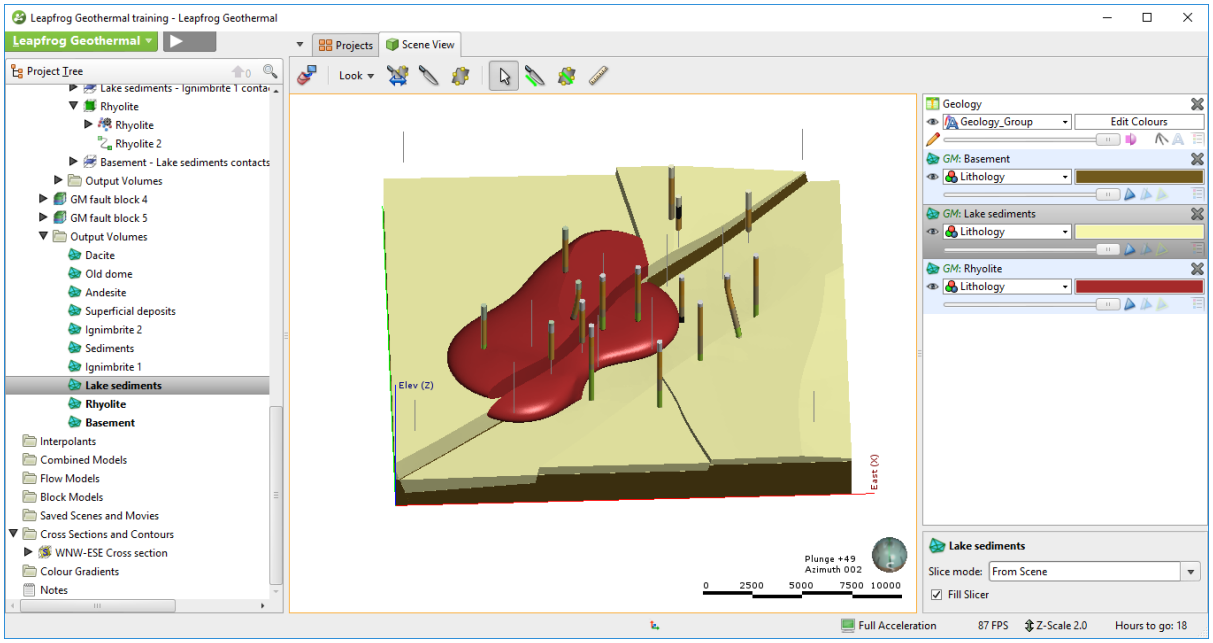
## ADJUSTING INTRUSIONS

If you have time, try to edit the Rhyolite. Follow the same principle, being aware that editing intrusion surfaces is not always an easy task. As you practice you will become more confident.

- As a starting point, bring the cross section in the 3D scene and edit the **Rhyolite** with a polyline. Click and drag each vertex if you want to create tangents.
- Check the orientation of the polyline.



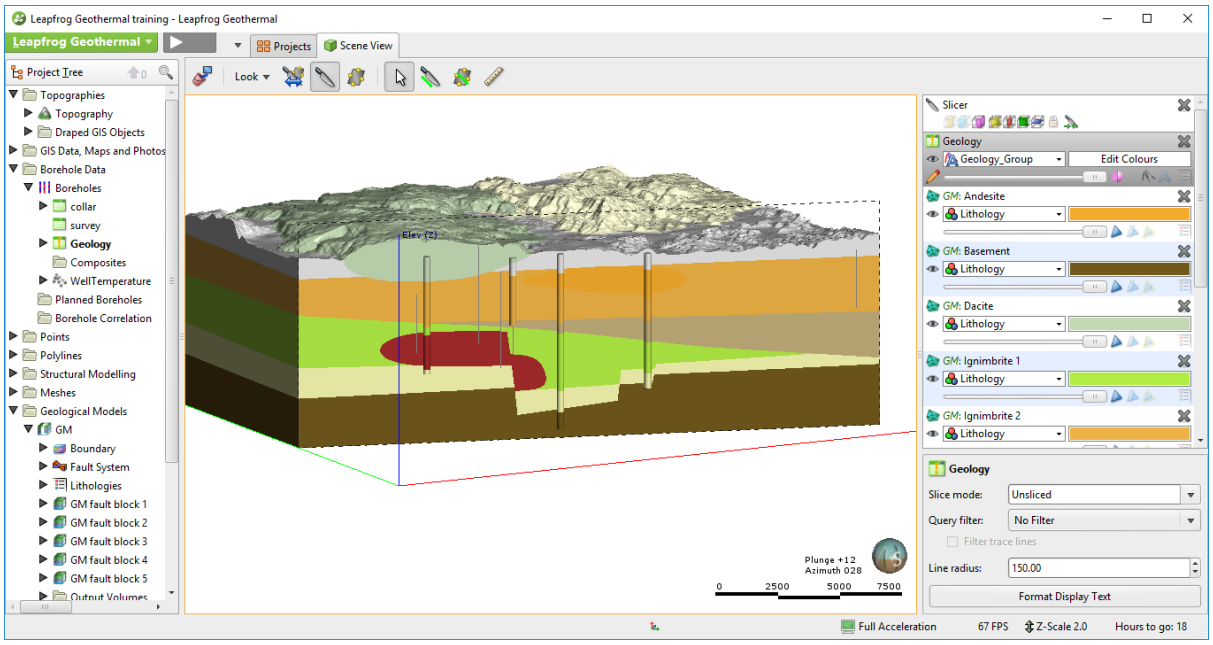

- With more edits, you can constrain the shape of the **Rhyolite**.



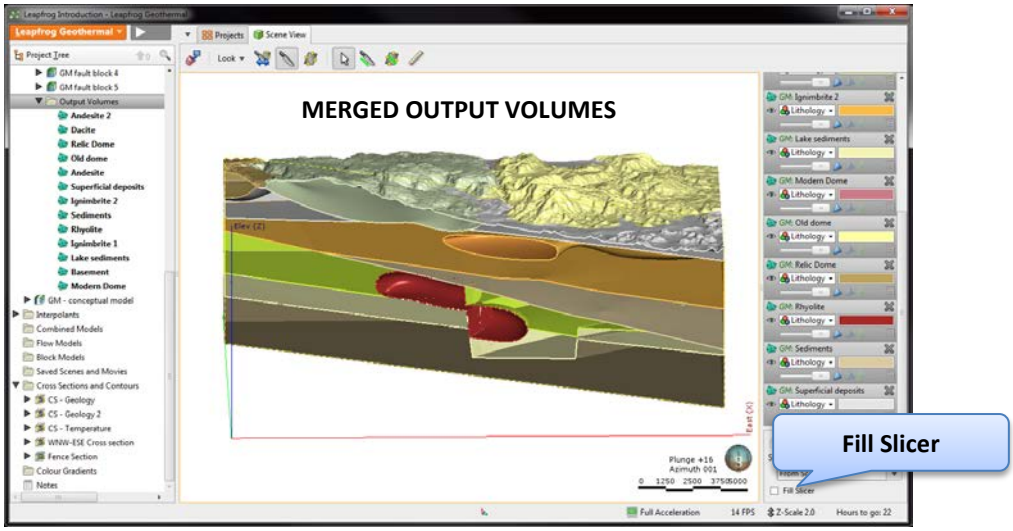
*Hints:*

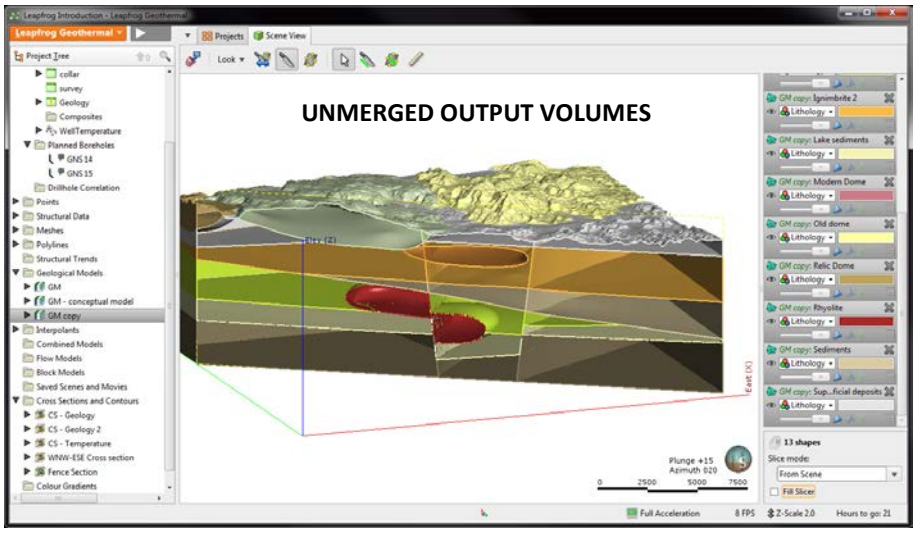
- *Remember that surfaces are constrained by the occurrence of lithological contacts. But the non-occurrence of a formation is also a source of information.*
- *As a rule of thumb, it is best not to overdo the manual editing, and if the surface doesn't snap to the desired position the most likely cause is nearby data conflicting with your edits.*
- *When updating a model, Leapfrog will honour the borehole data as well as your edits. Minimising the amount of manual editing makes the update of the models faster and easier to control.*

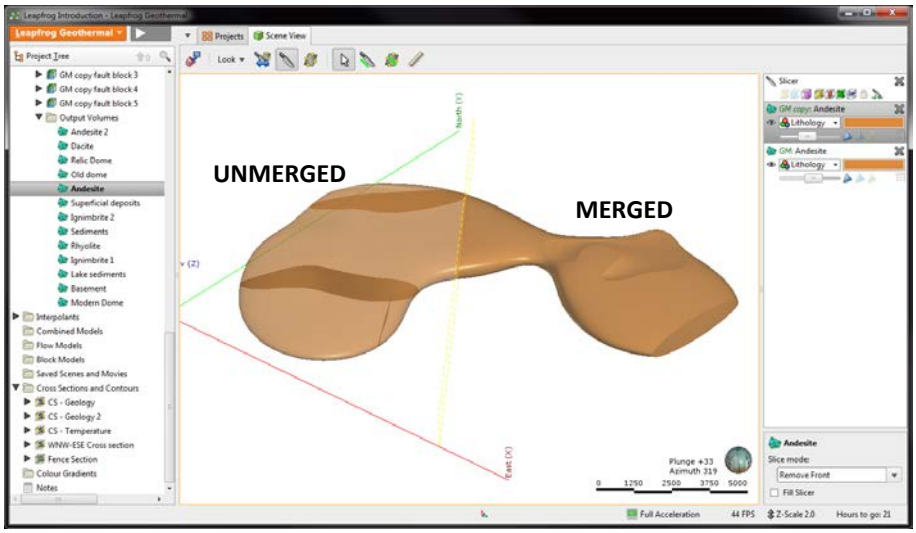
- Once finished take some time to look at model. Your final faulted model is now completed.



- At this stage, once the edits are done, it is often useful to **merge the output volumes** again. This option is available from your geological model **property window**.
- You see the differences when looking at a sliced model with the **Fill Slicer** option unchecked; but also when you have transparency on a unit.



## CONCLUSION

In this tutorial you have learnt different techniques to create faults in a geological model. You now have an understanding of the fault network, including fault chronology and relationships. You understand how faults are affecting a model and the importance to define the contact surfaces properties before activating the faults.

You have also practiced editing surfaces and intrusions to adjust/correct this model to make it structurally and geologically coherent.

# UPDATING A GEOLOGICAL MODEL

## TUTORIAL 6: UPDATING A GEOLOGICAL MODEL

This tutorial will take you through the process of up-dating an existing model. There are several ways of updating a model and we will go through the options based on different possible scenarios.

## TABLE OF CONTENTS


Introduction ..... 2

Updating the Boreholes data object ..... 2

- Minor update to existing wells ..... 2
- Major update to existing wells ..... 3
- Adding attributes to the Interval table ..... 3

Adding a new well to a project ..... 4

Modifying the input in a geological model ..... 8

- Using a subset of the original dataset ..... 8
- Adding lithologies ..... 8

Testing Scenarios ..... 9

- Interval Selection ..... 9
- Creating a mesh from borehole data ..... 11

Conclusion ..... 14

## INTRODUCTION

Updating a model can be required for several reasons:

- One interval has been reinterpreted.
- Series of wells have been reinterpreted.
- A new well has been drilled and must be added to the model.
- New formations have been identified.

Each scenario is handled differently. In most cases, Leapfrog Geothermal allows quick and easy updates. The software will reprocess all the models in the project using the new data. However, a few situations require special attention:

- Manual edits in your model will have to be checked for consistency. The software will always honour the given data (both the borehole tables and the edits from the user). However, it is the user's responsibility to check the validity of manual inputs as the model evolves.
- Any contact deposit built using a fixed surface (mesh) should be reviewed. If the original surface needs any change due to new well information, the mesh will need to be updated.
- Any new **Lithologies** will have to be added to the geological model.

An update will require processing time. Leapfrog Geothermal is designed in such a way that modifications are processed quickly. Depending on the complexity of the project, it might take minutes or (in the worst case) a few hours to rebuild a full model.

Due to time constraints, we will only import a new well within the model as an example, but each option is described below. You will also use additional tools to test different model scenarios.

## UPDATING THE BOREHOLES DATA OBJECT

### MINOR UPDATE TO EXISTING WELLS

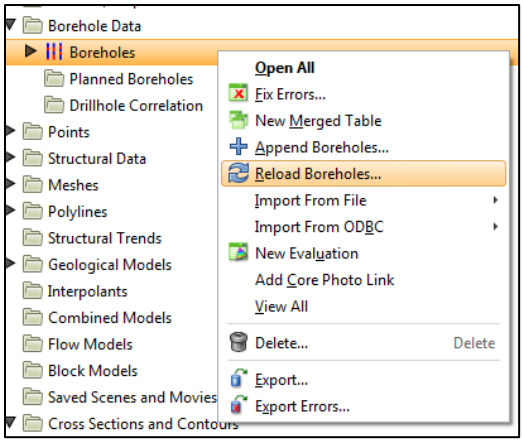
For minor updates to existing wells within the model, it is suggested that you work directly within borehole tables.

- Open the table that needs updating, e.g., the geology table.
- Modify the required from-to intervals and/or the lithology:
    - The grouped column cannot be edited directly, however the software will re-compute the geology table and if any changes in the lithology have been made, it will allocate the modified lithologies to the proper group.
    - If a new lithology is created, it will appear in the ungrouped lithologies window which you can then add to your group categories.
- Once you click OK the software will rerun the project immediately.
- Once the project has run, check the model for consistency, especially in surfaces combining lithology contacts and manual edits, as well as deposits using surface meshes.

## MAJOR UPDATE TO EXISTING WELLS

If a series of wells have been reinterpreted, it is often preferable to return to the original database to avoid confusion and duplication of your edits. In this case, the user will need to reload the data.

- Modify your raw .csv tables or your database, with the appropriate data.
- Keep the table outline identical. The software will ask for the same format as the original tables.
- Right-click on the **Boreholes** object in the Project Tree.
- Select **Reload Boreholes**.
- The **Reload Boreholes** window appears, asking for the file location. Browse to the right file location if different from the one proposed.
- Click **Import**.
- The software will rerun the project immediately.

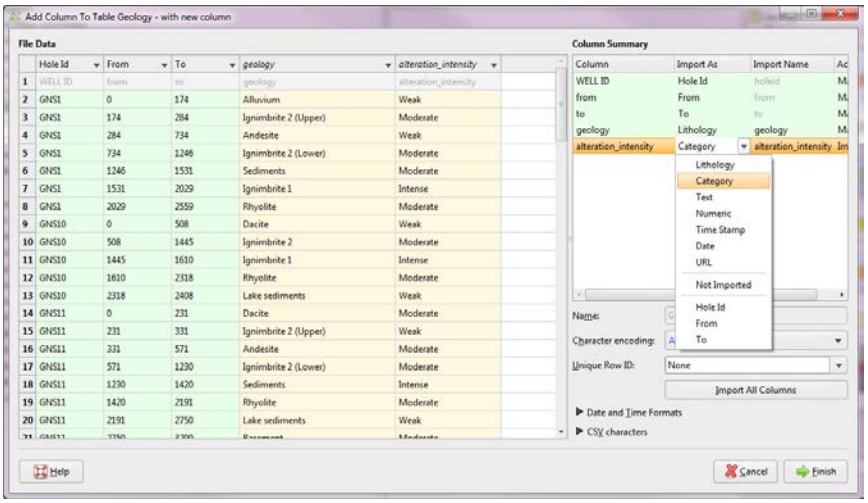



The software will check the data validity once again. If a red cross appears on one of the borehole tables, you may have to fix new errors. Once the project has run, check the model for consistency, especially in surfaces combining lithology contacts and manual edits by users, as well as surfaces using meshes.

## ADDING ATTRIBUTES TO THE INTERVAL TABLE

You can add new columns to your borehole tables. The new column would need to be added within your database first (if not originally there).

- Right-click on the **Geology** table and select **New Column/Import Column**.
- Browse to your *Geology – with new column.csv* file (**Borehole data/add data**).
- Change the new column from **Not Imported** into a **Category**.
- Click **Finish**. Leapfrog will not re-run the geological model as it is not affecting the input column used to build it.



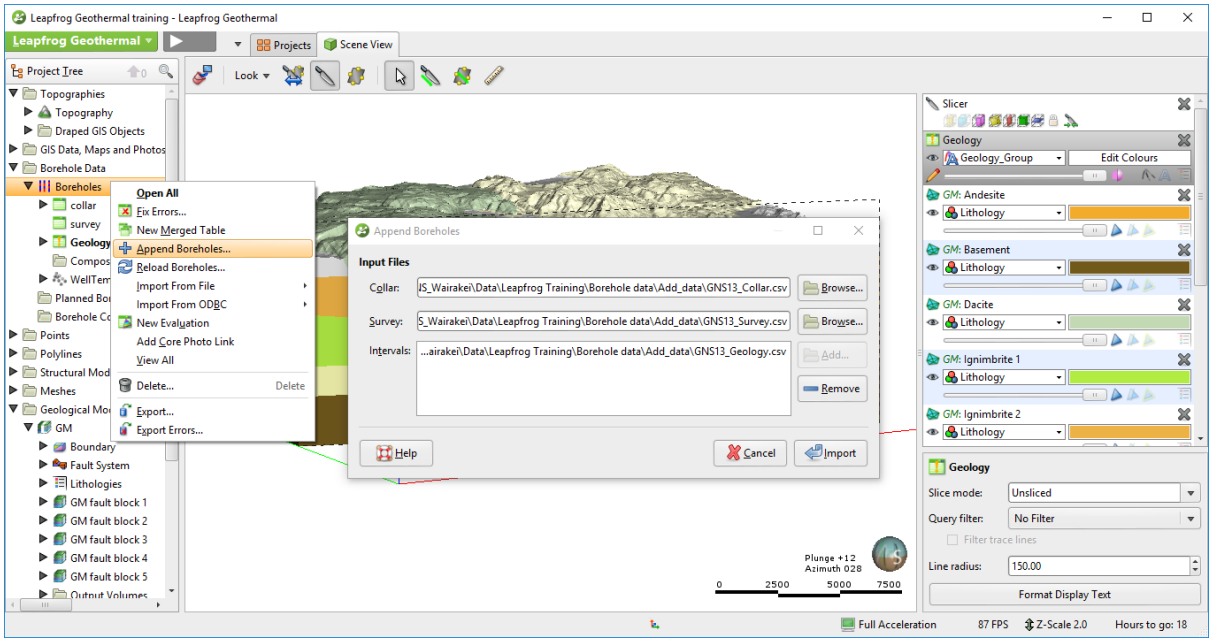

*At this stage we cannot delete columns in Leapfrog. If you import a new column, be aware that you cannot delete it afterwards. Even if you reload the data, Leapfrog will ask you for all the full original datasets.*

## ADDING A NEW WELL TO A PROJECT

As long as drilling proceeds in a field, wells can be added to the model in near real-time. New tables with the information relative to this well need to be created, keeping the same table format as the original ones in the model.

For this exercise, we are going to add GNS13 to our model. Right-click the **Boreholes** object in the Project Tree.

- Select **Append Boreholes**.
- The **Append Boreholes** window appears. It works in the same way as when you originally loaded your borehole data. Browse to the *Leapfrog Training* \ *Borehole Data* \ *Add data* folder and select the GNS13 files.

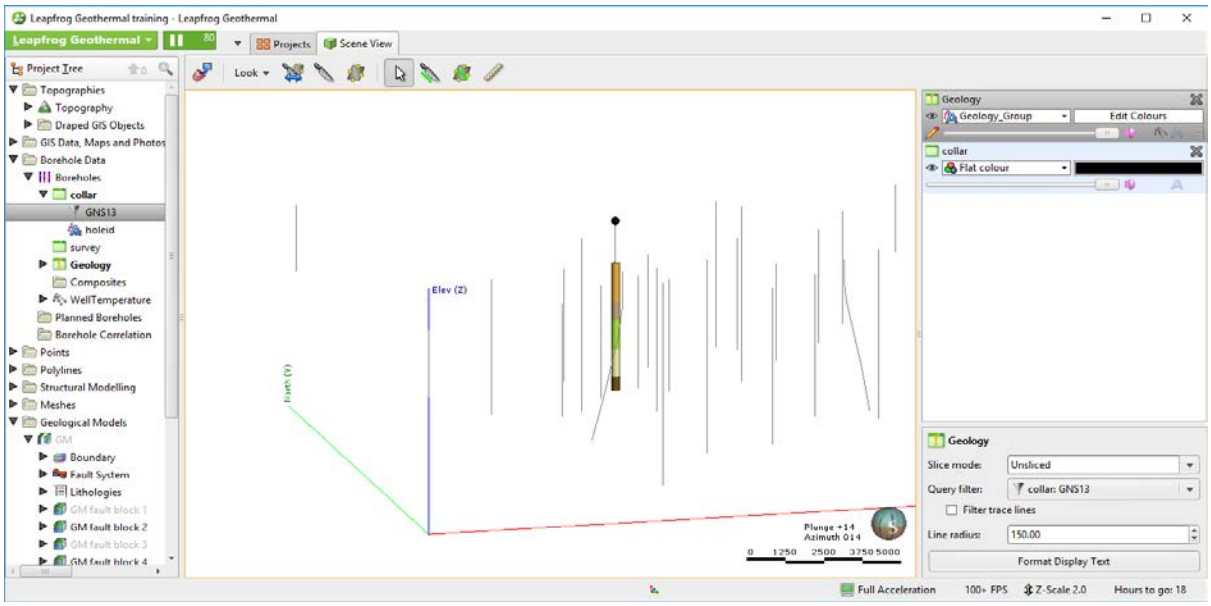



*Instead of appending data, you can always choose to **Reload** the data including the new well in the original tables.*

In this project, our model is mostly based on the borehole data. As we modified the table by adding new information Leapfrog Geothermal is now updating all the models in this project built from the borehole object. Without having to wait for the full project to run, we can already do some checks as soon as the Boreholes object has been processed:

- Check for any error in the table: if Leapfrog finds some inconsistencies or error in the new data, a red cross will appear on your table icon.
- Check your groups to identify any new formation or misspelling introduced with the new wells.

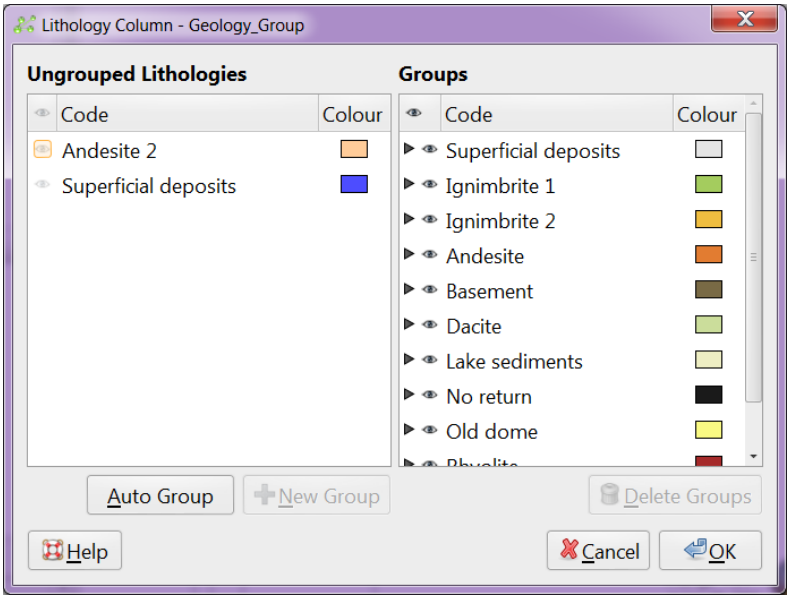
- Create a new query to individualize GNS13 from the other wells.
- Once the **Borehole** object has been processed, you can click on the **pause** icon in the **Processing Tasks.**




- As you can see on your boreholes tables, there are no errdors identified.

From visualizing GNS13 in the 3D scene, you can see that the top of the well has not been recognized (it is not colour coded when displaying the **Geology_Group** attribute). This means that some formations in the GNS13 could not be automatically allocated to the existing groups.

- Expand the **Geology** table object and double-click on the **Geology_Group** attribute. In the grouping window you can see that two formations are not allocated to the groups.

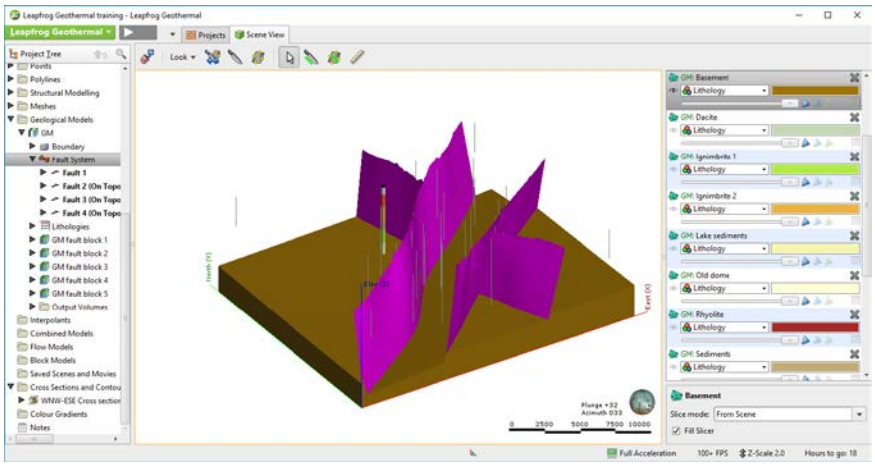



- The **Superficial deposit** lithology is straightforward and can be allocated to the superficial deposits group. On the other hand, the **Andesite 2** lithology is a new formation that was not previously encountered in the other wells. It needs to be added as a new group. Once done, click **OK**.

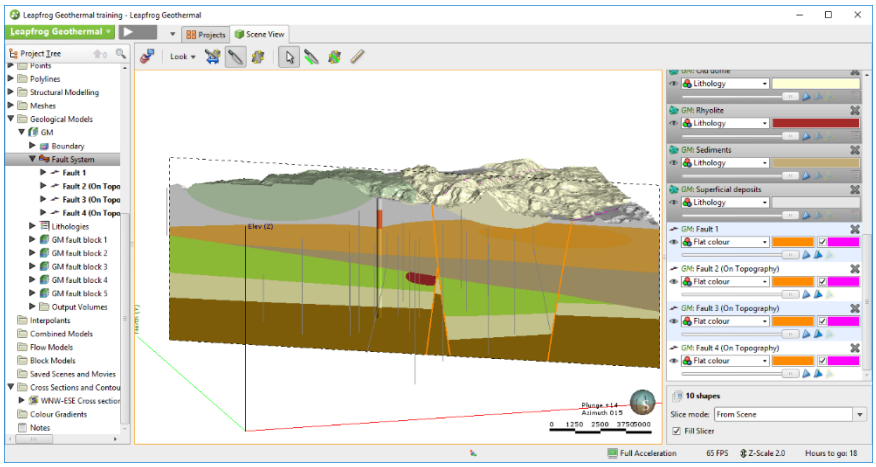
- Let the project run: click on the **Run All** icon in the Processing toolbar. You can now see that the first two intervals are colour coded.

Depending on your computer speed, rerunning the models may vary. Once the processing is finished, we need to check the geological model and make sure that no manual edits are in conflict with the new data. We also need to create a new formation for Andesite 2.

- Identify GNS13 position compared to your existing Fault network.



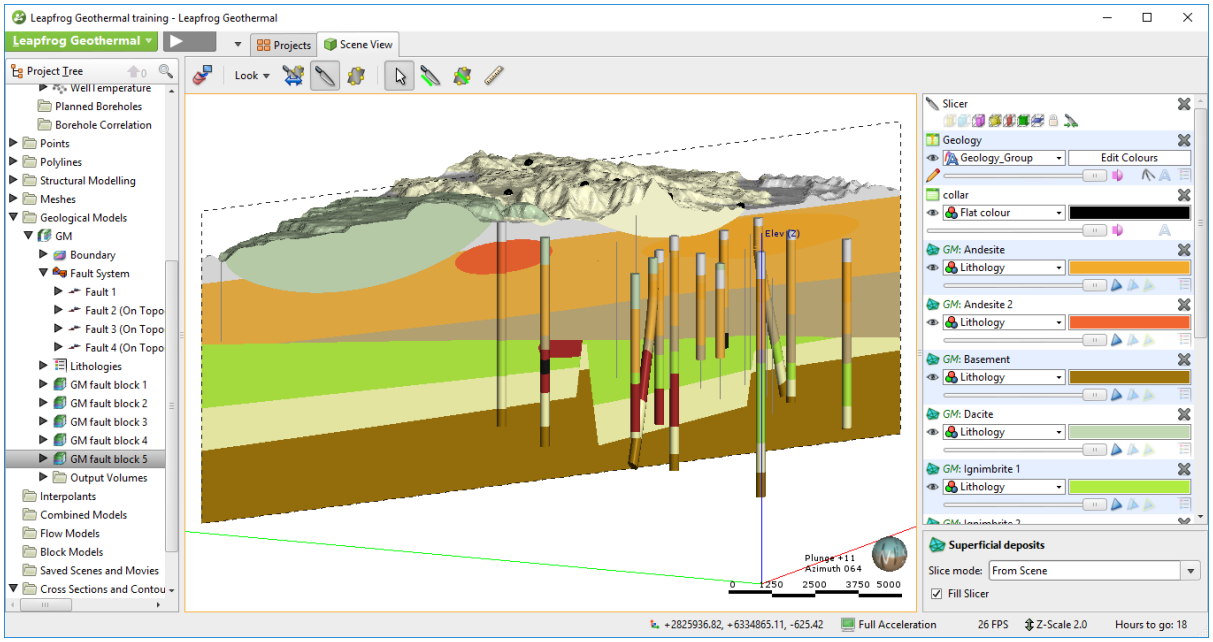
- Slice through GNS13 and compare the borehole Lithology with the model.



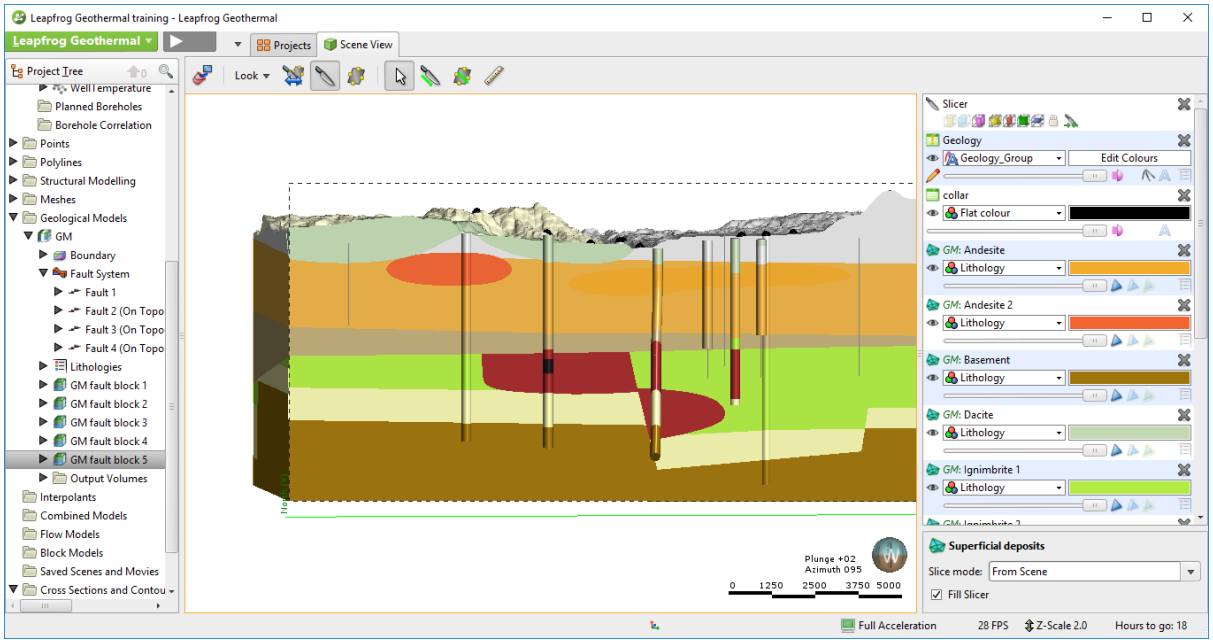
You can see that all the formations have been automatically updated and there is no conflict in the model except for the Ignimbrite 2 top surface and the non-existing Andesite 2 formation.

The software is by default honouring the contact point between Ignimbrite 2 and the new Andesite 2 unit. For this exercise we will model Andesite 2 as an intrusion so that it cross-cut Ignimbrite 2 automatically. The Ignimbrite 2 top surface will also need to be edited.

- Browse to the **Surface Chronology** object of the GNS13 block in the Project Tree and create a new intrusion for **Andesite 2**. Edit or adjust the new intrusion to create a geologically reasonable shape.
- Edit **Ignimbrite 2** top surface to ignore the contact point with **Andesite 2**. You will have to do this in each fault block to be consistent.
- Check the relationship between the new formation and the surrounding intrusions. Adjust the **surface chronology** if necessary and activate the formation. A new volume will be created and your model is up-to-date.


Leapfrog Geothermal training - Leapfrog Geothermal
Leapfrog Geothermal
Projects
Scene View
Project Tree
Look
Slicer
Geology
Geology_Group
Edit Colours
collar
Flat colour
GM: Andesite
GM: Andesite 2
GM: Basement
GM: Dacite
GM: Ignimbrite 1
Lithology
Superficial deposits
Slice mode:
From Scene
Fill Slicer
Boundary
Fault System
Fault 1
Fault 2 (On Topo
Fault 3 (On Topo
Fault 4 (On Topo
Lithologies
GM fault block 1
GM fault block 2
GM fault block 3
GM fault block 4
GM fault block 5
Output Volumes
Interpolants
Combined Models
Flow Models
Block Models
Saved Scenes and Movies
Cross Sections and Contou
Plunge +11
Azimuth 064
0 1250 2500 3750 5000
+2825936.82, +6334865.11, -625.42
Full Acceleration
26 FPS
Z-Scale 2.0
Hours to go: 18

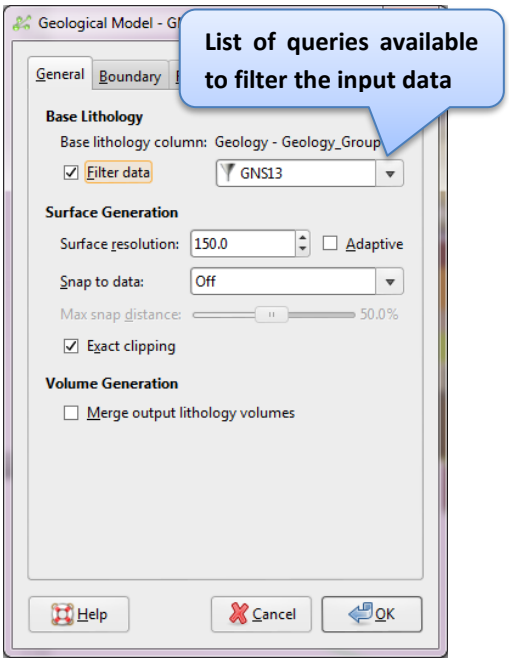
D


Leapfrog Geothermal training - Leapfrog Geothermal
Leapfrog Geothermal
Projects
Scene View
Project Tree
Look
Slicer
Geology
Geology_Group
Edit Colours
collar
Flat colour
GM: Andesite
GM: Andesite 2
GM: Basement
GM: Dacite
GM: Ignimbrite 1
Lithology
Superficial deposits
Slice mode:
From Scene
Fill Slicer
Boundary
Fault System
Fault 1
Fault 2 (On Topo
Fault 3 (On Topo
Fault 4 (On Topo
Lithologies
GM fault block 1
GM fault block 2
GM fault block 3
GM fault block 4
GM fault block 5
Output Volumes
Interpolants
Combined Models
Flow Models
Block Models
Saved Scenes and Movies
Cross Sections and Contou
Plunge +02
Azimuth 095
0 1250 2500 3750 5000
Full Acceleration
28 FPS
Z-Scale 2.0
Hours to go: 18

## USING A SUBSET OF THE ORIGINAL DATASET

If for any reason you want to rebuild the model using a subset of your original dataset, create a query on your borehole data and use this query to filter the data in your geological model.

- Make a copy of your Geological Model.
- Double-click on your new model.
- In the **General** tab, check the **Filter Data** box and select the query to use.
- If you press **OK**, Leapfrog will re-run the model.

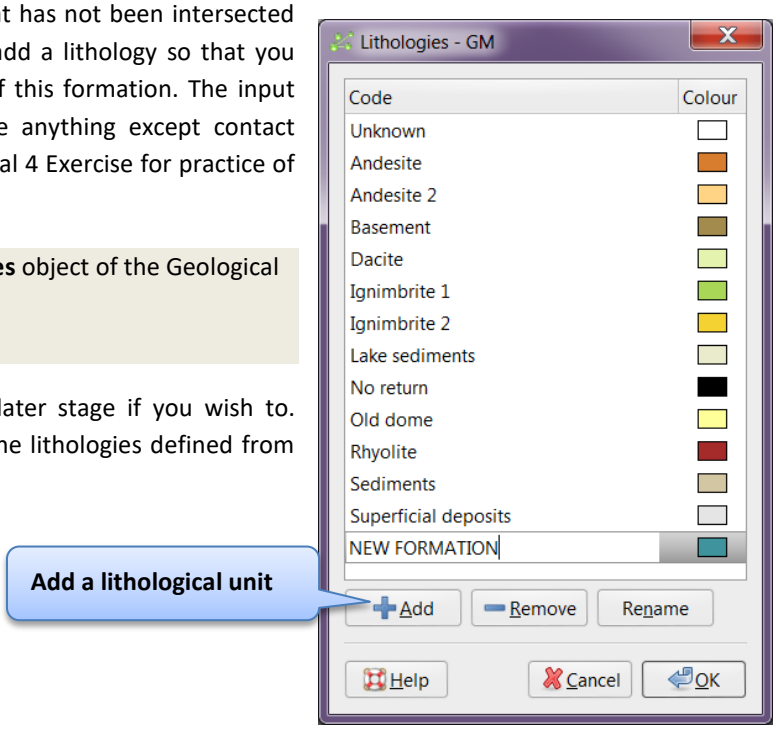



## ADDING LITHOLOGIES

If you want to model a formation that has not been intersected by any borehole, you can manually add a lithology so that you can create a surface and a volume of this formation. The input data to create this surface could be anything except contact points from boreholes (refer to Tutorial 4 Exercise for practice of this method).

- Double-click on the **Lithologies** object of the Geological Model.
- Click on the **Add** button.

You can delete this formation at a later stage if you wish to. However, you cannot delete any of the lithologies defined from the borehole data.



## TESTING SCENARIOS

The workflow in Leapfrog Geothermal allows you to rapidly build several models to test alternative scenarios. You can use the same borehole data to build a different model by altering some faults for example. Or you may want to test a different model with different grouping solutions in case some lithological intervals are ambiguous. In the section below, we present such an example using the interval selection tool to test an alternative to the first model, without changing the primary data and without losing information from your first model.

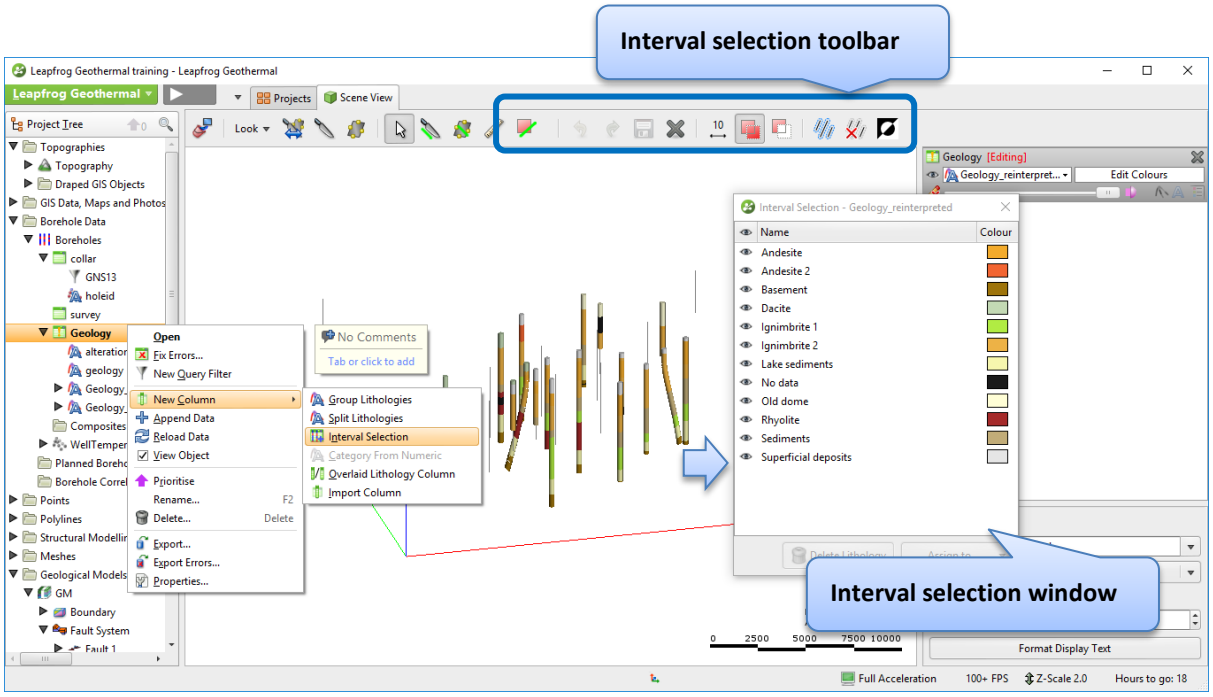
We are going to create a new column in your lithology column, assigning some intervals to a new formation. We will then create a surface of this new formation to compare with the current model to help in your interpretation of the local geology.

## INTERVAL SELECTION

In this exercise we will re-assign some lithological intervals to a new formation. We want to group the Andesite 2 (GNS13), the Old Dome (all wells and surface data) and the Andesite (in GNS7 only).

The following process will create a new column in the interval table, which means that it will not automatically update the existing model.

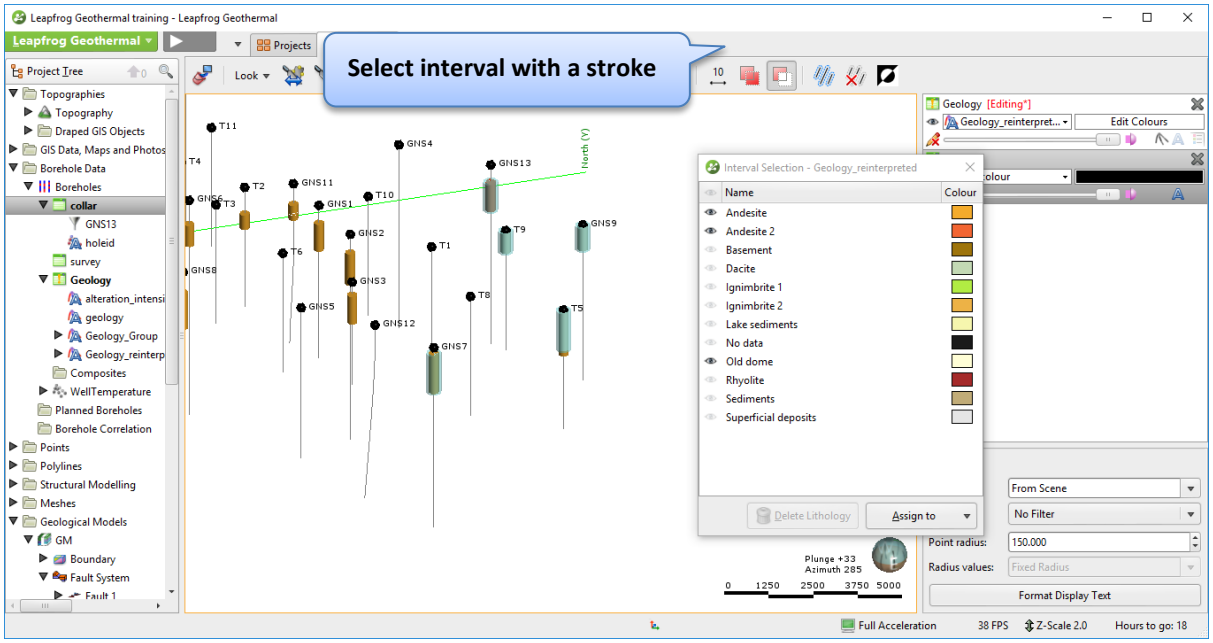
- Right-click on the **Geology** table in the project tree and select **New Column/Interval Selection**.
- Select the **Geology_group** as your **base column** and enter a name such as *"***Geology reinterpreted***"* and click **OK**.
- A new window and new toolbar will appear.



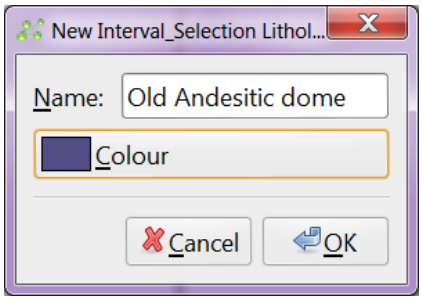

This tool is allowing you to select some intervals on screen and reassign them to a new formation.

- Click on the **Select Interval with a stroke** icon and select the following intervals by clicking on them or click and drag the selection line through them:

- **Andesite 2** in GNS13.
- **Andesite** in GNS7.
- **Old dome** in GNS9, T9 and T5.



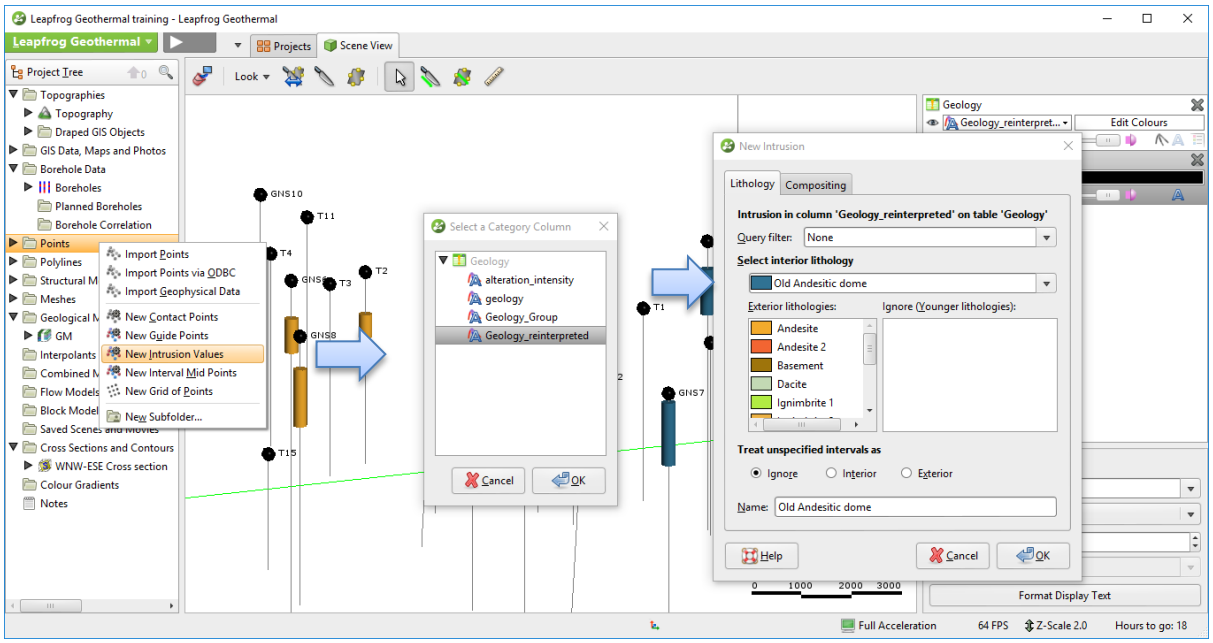

- If you make a mistake, click on the **Remove from the current selection** icon and click on the interval you need to remove.
- Once you are satisfied with the selection, click on **Assign to** on the Interval selection window and **Create a New Lithology**.
- Enter a new name and click **OK**.
- Save your edits. The colour of the intervals of interest will be updated in the **Geology_reinterpreted** attribute. The rest remains unchanged.
- Close the Interval selection session and open the geology table to check that a new column has been created (without modifying the primary data).

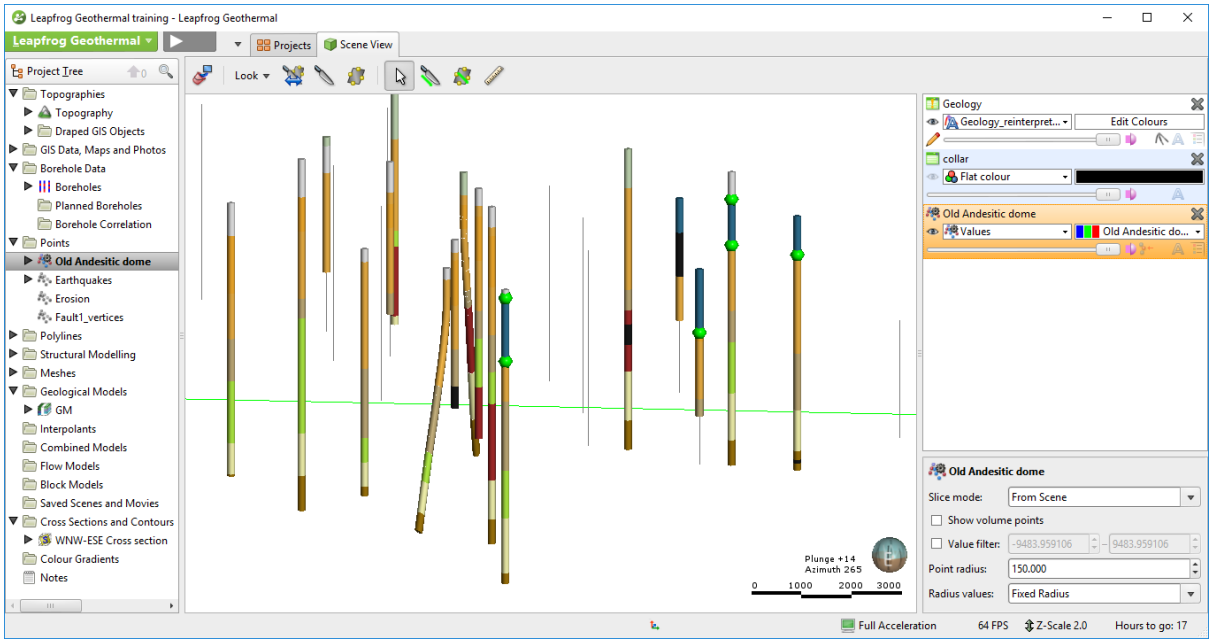


## CREATING A MESH FROM BOREHOLE DATA

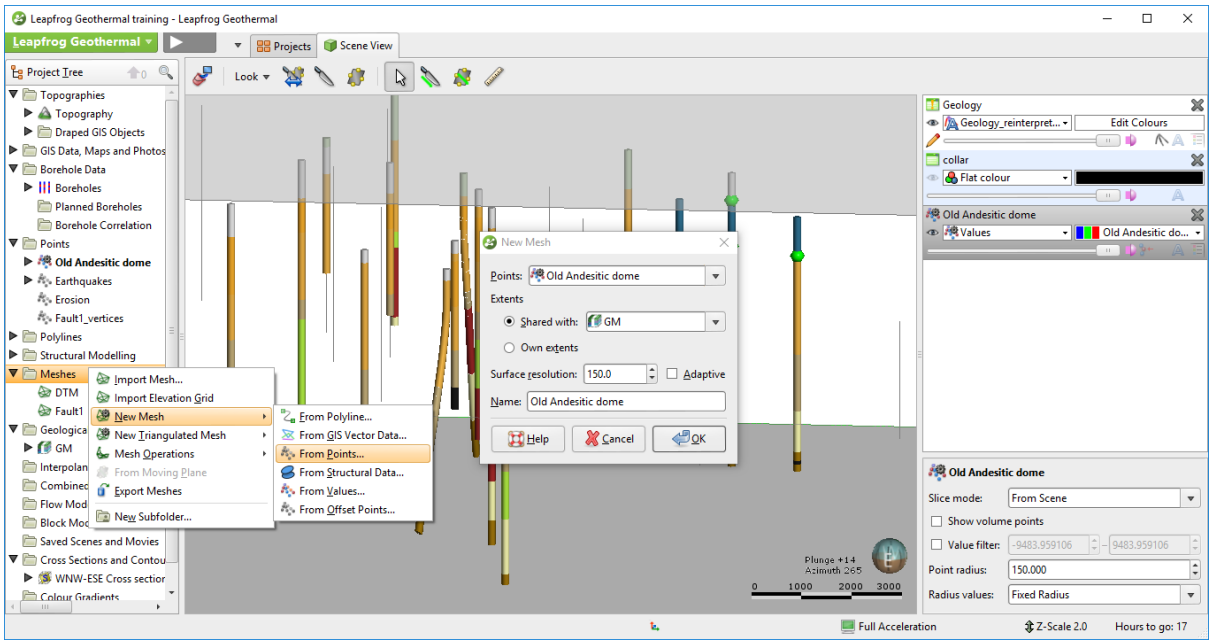
In this section we are going to create a new surface based on the reinterpreted column, using the mesh tool. This can be really useful for modelling surfaces that are not faulted and can be shared between fault blocks. This is a two steps process: first, you need to extract the contact points from the boreholes table; then, use these contacts as input for your mesh.

- Right-click on the **Points** folder in the Project Tree and select **New Intrusions Values**. This tool will allow you to extract contact points (intrusion type contact points) from the borehole table and save them independently in the Points folder.
- Select the **Geology reinterpreted** category column.
- In the **New Intrusion** window, select your new formation as interior lithology and click **OK**. A new set of points appear under the location folder.

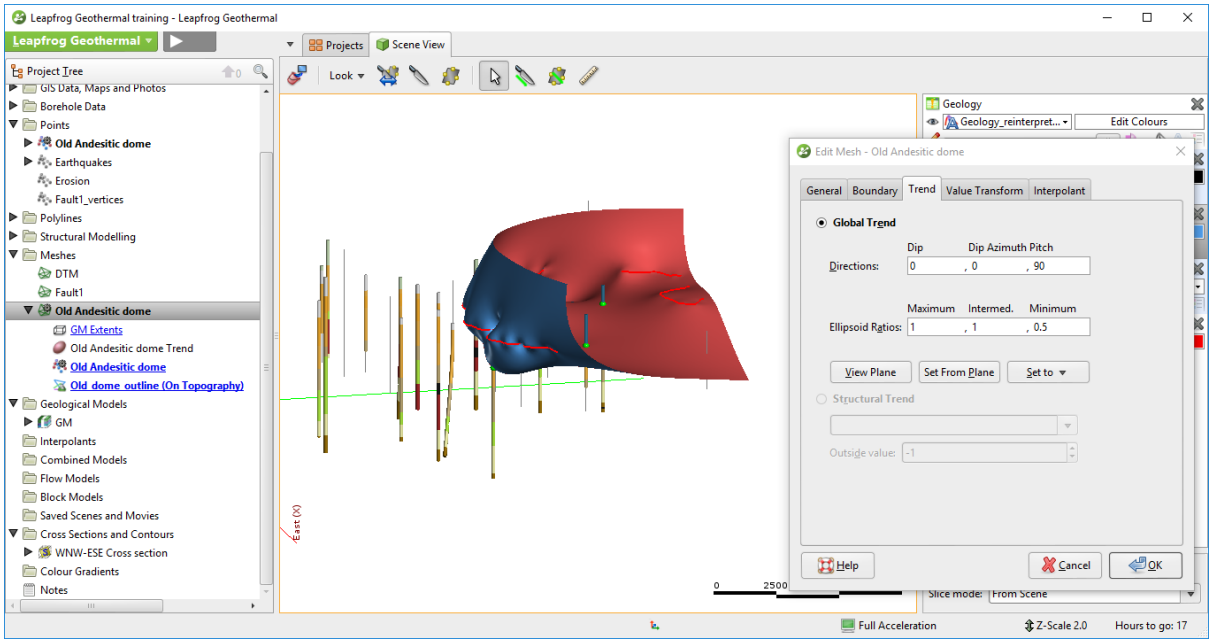


- Right-click on the Meshes folder and select **New Mesh / From Points**.
- Select the points **Old Andesitic dome**.
- Select **Own extents** and make sure it is slightly smaller than the geological model boundary.
- Change the resolution to 150 m and enter a new name before clicking **OK**. A new mesh has been created.

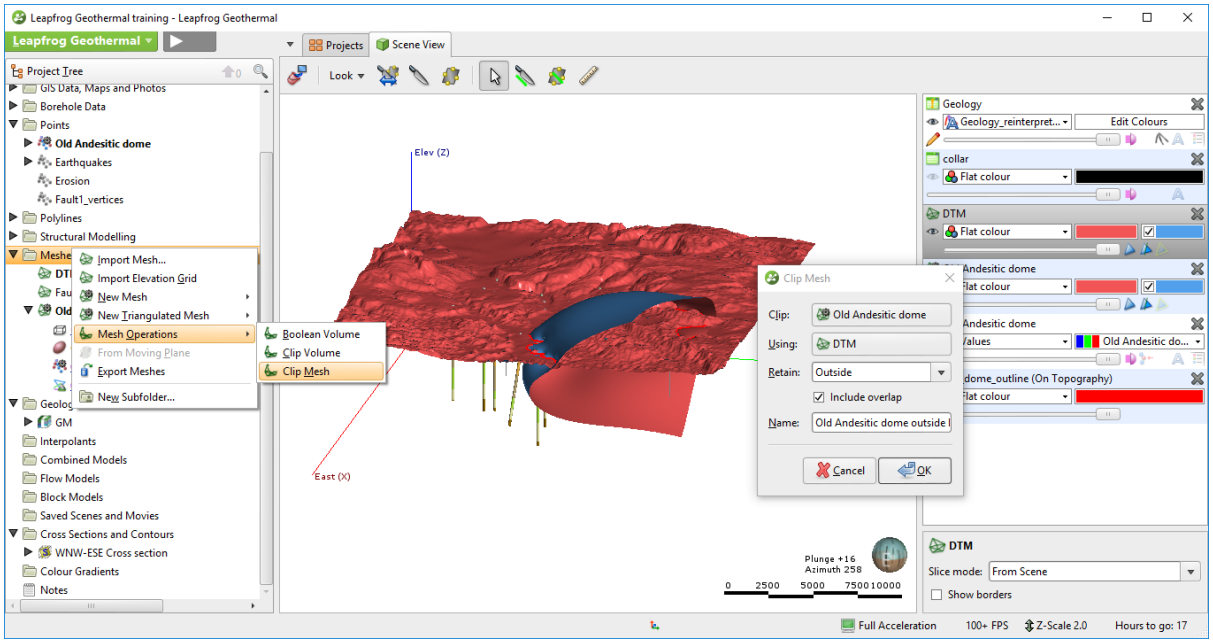



- Remember we also have surface data for the **Old Dome** unit. Use it to further constrain this mesh. Right-click on the mesh and **Add/GIS Vector data**.
- Expand the Mesh in the project tree. You can see it has its own extents, its own trend, the locations points and the GIS outline as input data. Modify the trend to force a horizontal elongation.

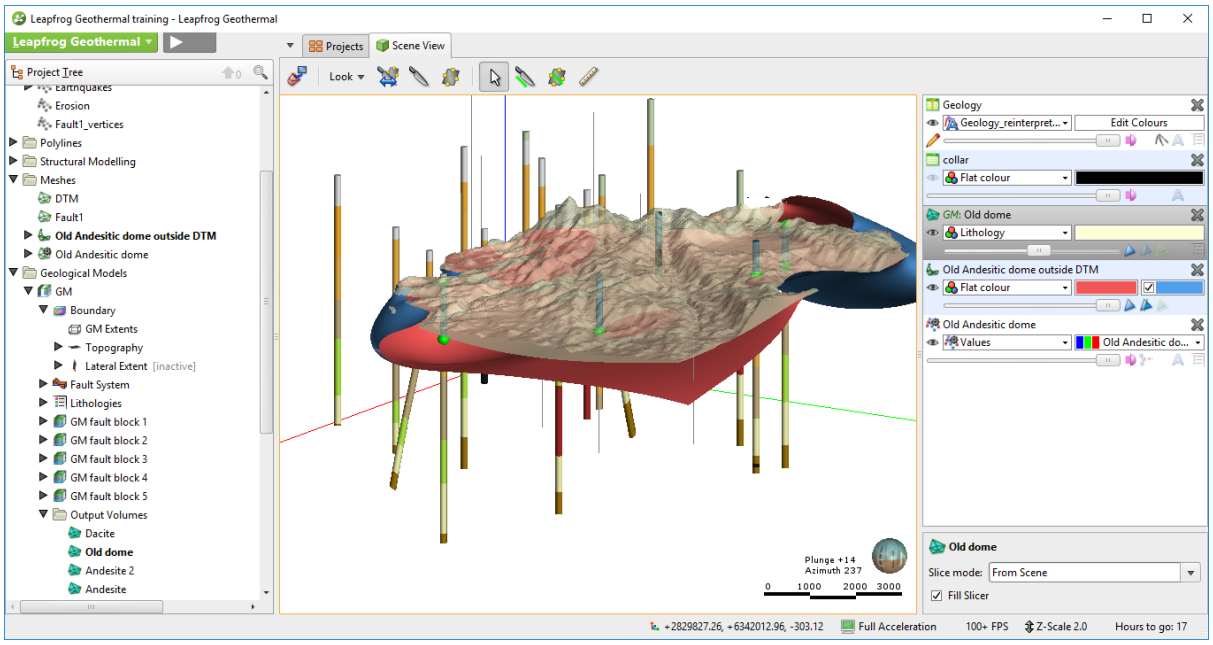



- Right-click on the **Meshes** folder and select **Mesh Operation/Clip Mesh**.

- A new window appears. Select the **Old Andesitic dome** as the mesh to **Clip** and the **DTM** as the **Clipping mesh**. Visualise the **DTM** surface to identify the side to be retained (red colour is inside, blue colour is outside). Click **OK**.




A new mesh has been created, now clipped to the Topography. Compare this new surface with the existing volumes to help in your interpretation. It would then be easy to adjust the existing geological model.




*The two meshes created in this process are dynamic. If we update the borehole table, the intrusion points from the Geology reinterpreted column will be updated, consequently the Old Andesitic dome mesh will be updated, and consequently the clipped mesh will be updated.*

## CONCLUSION

In this tutorial you have learnt different options to update your geological model as new information becomes available. You are now confident with updating your borehole data, adding new information to your table and updating your geological model.

You have tested a method to create new surfaces using borehole data as input, without having to generate a full geological model to test alternative scenario and assist in the geological interpretations.

All these tools will be extremely useful when you start building your own model.