leapfrog® | geo

Fundamentals training

Session notes

Contents

leapfrog
FAST, DYNAMIC GEOLOGICAL MODELLING

Session 1: Working with Projects

Contents

Goals

In this session, we will cover:

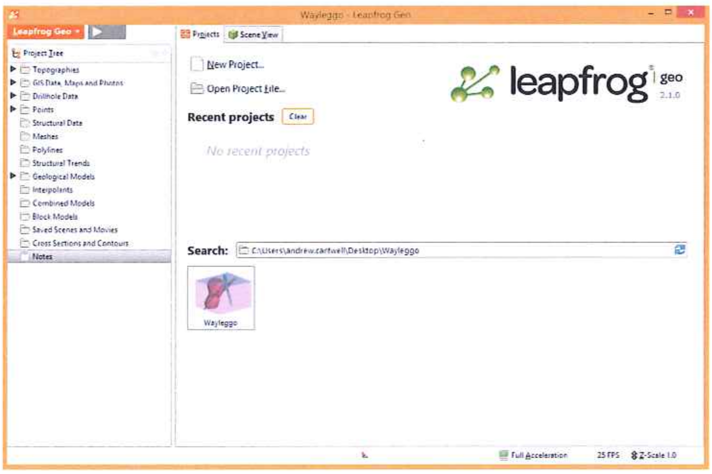
- The basic layout of the Leapfrog Geo screen
- Using the menu and toolbar
- Using the mouse
- Creating, opening and closing projects

The project file for this session can be found in the `Sessions \ Session 1 - working with projects` folder.

Opening and Organising Projects

When you open Leapfrog, the **Projects** tab displays thumbnails of the last five projects opened. Leapfrog also displays the projects saved in the **Search** folder. To see the location of a project file, hold the mouse over the thumbnail. To clear the recent projects, click **Clear**.

You can change the **Search** folder by clicking on the folder button (🗀) and navigating to the folder you wish to use. This is useful if you have one folder in which you keep most of your Leapfrog Geo projects.

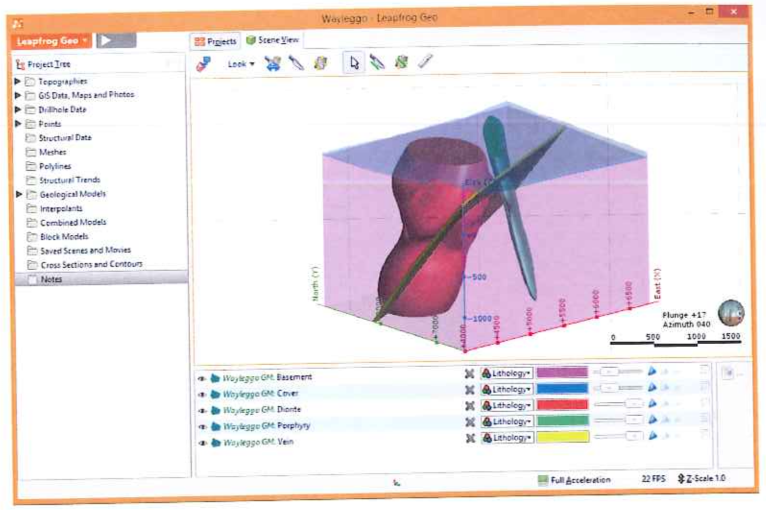


Open a project by clicking on the thumbnail. If the project you want to open is not displayed, click the **Open Project File** button to browse for the project.

There is a project file in the data folder for this session. To open the project:

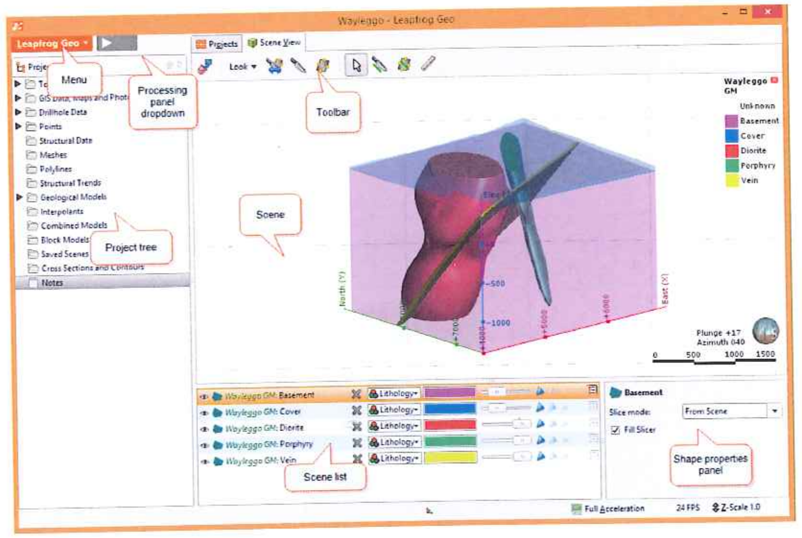
1. In the **Projects** tab, click **Open Project File**.
2. Browse to the folder for this session.
3. Select the project and click **Open**.

This will open the completed project.



Only one project can be open at any time.

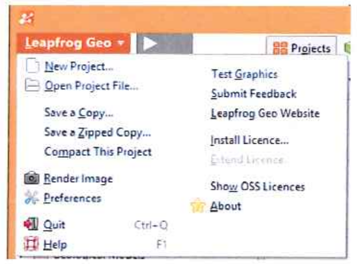
The Leapfrog Geo Main Window



leapfrog
FAST, DYNAMIC GEOLOGICAL MODELLING

The Menu

The Leapfrog Geo menu is rather small:



Don't be fooled by this, as most of the functionality is revealed by right-clicking on folders and objects in the project tree.

Saving and compacting projects

When objects are deleted from a Leapfrog project, the software retains the objects as part of the file, even though the objects are no longer being used in the project. This can cause projects to unnecessarily take up a lot of disk space, especially if large objects such as images have been deleted from the project.

To remove these unused objects, there is a **Compact This Project** option under the menu. This may take a few minutes depending on the size of the project, and Leapfrog will close the project and reopen it once compacting has been completed.

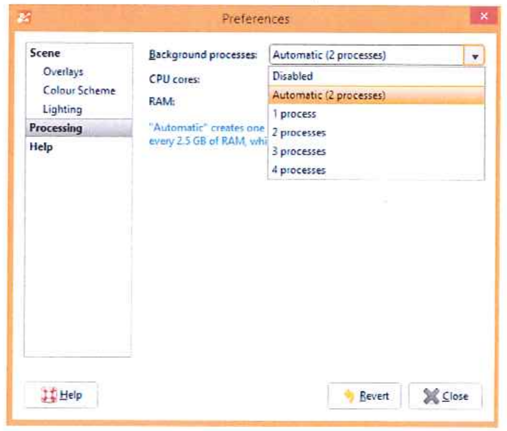
There are options to **Save a Copy**, and **Save a Zipped Copy** of projects. By saving a copy, you can save a separate copy of the project to another location on your disk. Saving a zipped copy lets you save a full copy of the project, including both the project file and the data file which are discussed later in this session.

It is recommended that if a project is being saved to pass on to another user or send via email, the **Save a Zipped Copy** is used.

Preferences

The preferences option allows a number of default settings to be changed. Most of these options are self explanatory, but the processing and help options will be discussed further below.

By selecting the **Processing** tab, you can select how many processes Leapfrog is able to use. By default, this is set to 'Automatic', which creates one process per physical CPU core, or for every 2.5GB of RAM, whichever is smaller, up to a limit of 4 processes. To manually change this, click the drop down menu at the top of the preferences window.



The help menu is accessible either online or via a local folder. If you will be in a remote location away from an internet connection, it is recommended that a local version of the help files is installed on your machine. The local version of the help files is available to download from Leapfrog's website.

The Toolbar

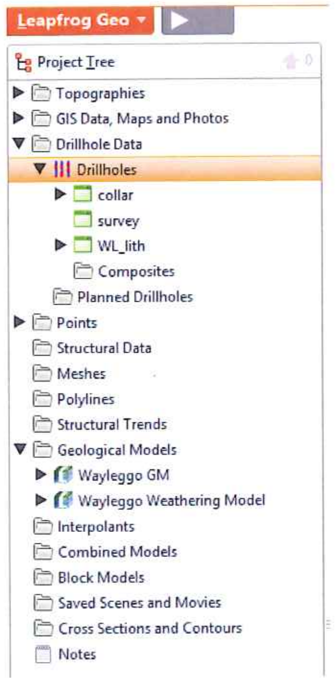
Leapfrog only has a few toolbar buttons.

One is the **Clear Scene** button ( ), which removes everything from the scene window. Another displays the **Look** menu, for changing to different viewing angles. The slicer, plane, and ruler tools are all available in the top toolbar.

Hold the mouse over each tool in the toolbar to view its function.

The Project Tree

The left hand panel contains the project tree:

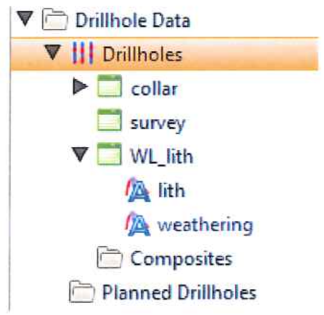


When you open any project, there will be a list of folders and objects in the project tree. This top-level list of folders is consistent for all projects. The way objects are organised in the project tree lets you reveal or hide information about an object to focus on objects you are currently working with.

The project tree is where you import and work with your data. In general, if you want to do anything with the objects in the project tree, right-click on an object to see the available options.

If you remove or change an object, it will affect any related objects that appear underneath it.

Click the triangle in front of **Drillhole Data** folder. When this is open, you can see a **Drillholes** object that also has a triangle in front of it. Open the **Drillholes** object to see the data tables that make up the drillhole data in the project. You can also open up each table to look at the data it contains.



The Processing Panel Dropdown

The **Processing Panel** can be accessed by clicking the dropdown button to the right of the Leapfrog Geo menu. The button is grey when there are no running tasks, and green when tasks are running.

This panel lists all processes which are either running or waiting to run. The number of processes running at any one time depends on your settings in the preferences tab, discussed earlier in this session.

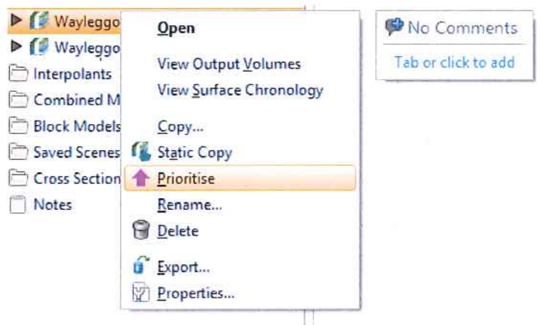
Leapfrog is either able to run all processes, run priority processes only, or pause all processing.

Run All will run all queued processes in the default order.

Priority Only only runs tasks which have been assigned as 'priority'.

Prioritised Running

If you would like a task to be given priority so it is processed before other tasks, you can right click on the object in the project tree and select **Prioritise**.

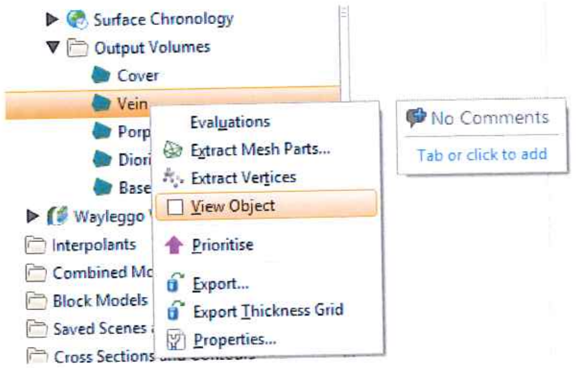


Objects such as geological models are usually dependant on other objects, such as drillholes and topographies. If this is the case, higher level objects (drillholes and topographies) will be processed first to allow the prioritised objects to be processed.

To clear prioritised running for objects, right click on the object in the project tree and select **Clear Priority**. If a number of objects have been given priority and you would like to clear them all at the same time, there is a small pink arrow above the project tree which you can click to highlight all prioritised objects. Once they have been highlighted, right click on one of the objects and select **Clear Priority**.

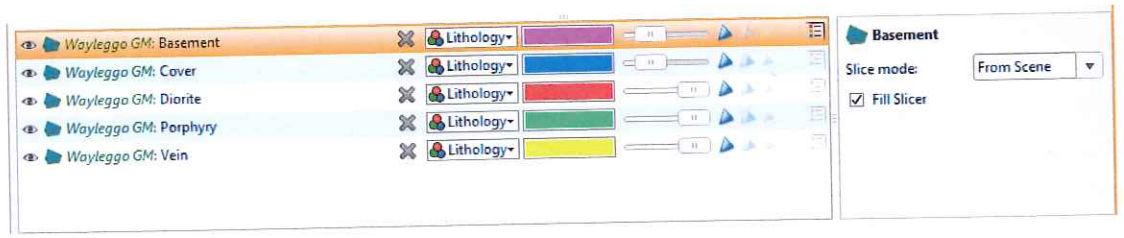
The Scene

To the right of the project tree is the scene. This is the area where objects appear when they are added from the project tree. To add an object to the scene, drag and drop it using the left mouse button from the project tree into the scene. You can also right click on an object and select **View Object**.



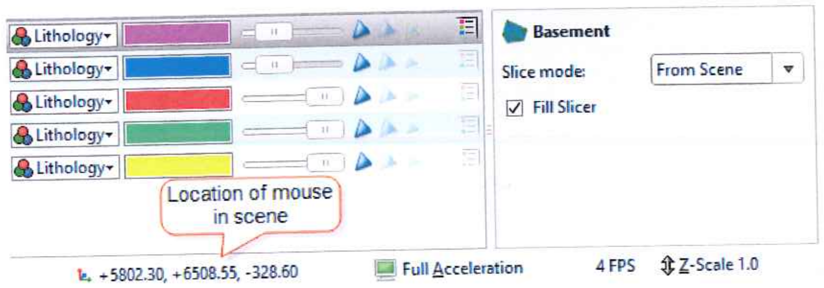
The Scene List and Shape Properties Panel

The scene list below the scene window lists all objects that are active (visible) in the scene window.



The Status Bar

The status bar at the bottom of the window has some useful features.



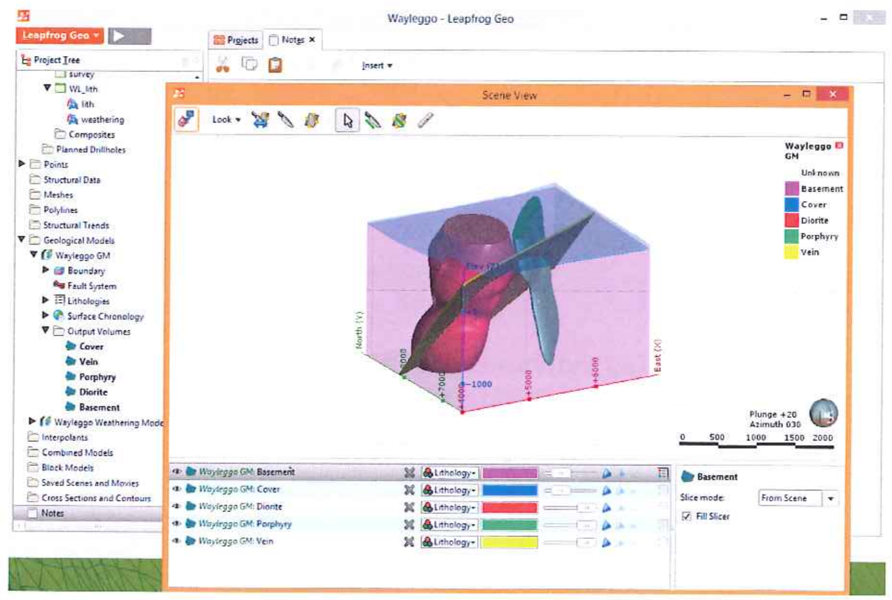
The coordinates that appear in the status bar show the location of the mouse cursor when it is over an object in the scene window. No coordinates are displayed when the cursor isn't over an object as Leapfrog doesn't know where in 3D space you're trying to measure.

The **Z-Scale** button lets you set a value greather than 1 for the Z-axis relative to the X- and Y-axes . This is useful for thin, flat projects that are difficult to visualise with a z-scale of 1.

The **3D Acceleration** option is also important. There are three modes: **Software Rendering** (not recommended), **Partial Acceleration** and **Full Acceleration**. You can find out more about each option by clicking the button and reading the descriptions for each. It's best to test both **Partial Acceleration** and **Full Acceleration** modes to see which one gives you the highest frames per second (FPS) value. The FPS value tells you how quickly objects in the scene can be rendered. If this gets too low (below 10), you should consider updating your graphics card. If this isn't possible, you can reduce the number of 3D objects (e.g. by clicking the **Make lines solid** option for drillholes) or transparent objects in the scene.

Reorganising the Screen Space

You can split the Leapfrog main window into separate parts to make better use of your screen space. To detach a tab, click on it and drag it away from the main window. You can then move and resize the detached tab.



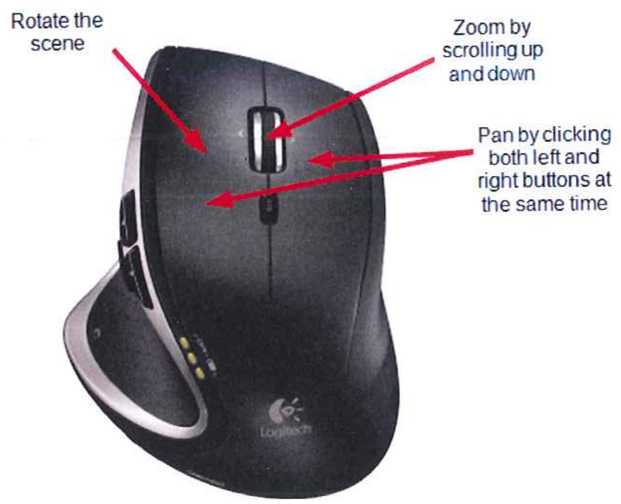
You can also:

- Detach the project tree. Right-click at the top of the project tree and select **Detach**.
- Display the shape list to the right of the scene window. Right-click on the shape list and select the arrangement you prefer.
- Display the shape list apart from the main window. Right-click on the shape list and select **Put List in Separate Tab**. Then you can detach the shape list from the main window as you would any other tab.

You can dock a tab by dragging it onto the main window. Release the mouse button when the tab becomes partially transparent. You can reattach the project tree by closing it, and dock the shape list by right-clicking in it and selecting where you wish to have the shape list.

You can also dock any tab or window by closing it. If you have rearranged your screen space, close all undocked tabs and windows to return to the original screen layout.

Interacting with the Scene



Rotating

Left-click on the mouse (index finger) and move the mouse.

You can also rotate the scene by pressing the arrow buttons on your keyboard.

Zooming

To zoom either move your scroll wheel up and down or click and hold the right mouse button, and move the mouse up and down. You can also use the `Page Up` and `Page Down` keys on your keyboard.

Panning

Click the left and right mouse buttons together then move the mouse around, or click the scroll button.

Centre of Rotation

To centre on something in the view, hover over it. You will see some numbers at the bottom of the screen that indicate that the cursor is on something in the scene window. When you move over empty space, the numbers disappear.

Click with both left and right buttons together on the item that you want in the centre. This will then remain in the centre of the scene. When you rotate and when you zoom in, it zooms in on that item. This can be very useful.

Creating a Project

Next we will build a simple model with five rock types to introduce the basic concepts of geological modelling in Leapfrog Geo. The rock types we will be using are as follows

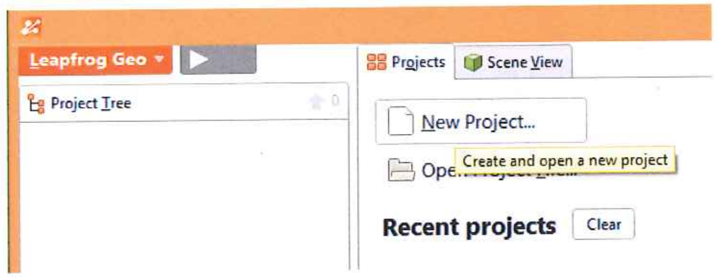
:

- Cover (youngest)
- Vein

- Porphyry
- Diorite
- Basement (oldest)

First, we will set up a folder for all Leapfrog projects.

1. Click on the **Projects** tab.
2. Click the **New Project** button:



3. Click on the **Browse** button to put a folder on your local disk.

Note: The project needs to be on your local disk rather than on a network drive to ensure the database operates consistently.

4. Make a new folder and call it Leapfrog Projects.
5. Click **OK**.
6. Name the new project "Wayleggo" and click **OK**.

Leapfrog will display a blank project called "Wayleggo".

Working with Leapfrog Projects

When your project was created, Leapfrog automatically created a sub-folder called `Wayleggo` that contains the `Wayleggo.aproj` project file and all supporting files.

Leapfrog Project Files

Your Leapfrog project is made up of a number of different folders. In the screenshot below, `Wayleggo.aproj` is the actual project, and `Wayleggo.aproj_data` folder is the database containing the binary code that makes up your project.

It's a good idea to stay away from this folder: You can't change anything in your project by looking in it, and if you move or delete anything from it, you run the risk of corrupting your project.

When the project is open in Leapfrog, a `Wayleggo.lock` file will appear in this folder. The `.lock` file protects the project from being moved while the project is open and from being opened by another instance of Leapfrog, which can happen when projects are saved on shared network drives.

Saving Leapfrog Projects

Leapfrog projects are saved automatically as you work with them. This means if you exit Leapfrog while there are tasks still processing, the tasks will be waiting to resume processing the next time you open that project.

Session 2: Importing Drillhole Data

Contents

Goals

In this session, we will import and validate existing drillhole data for a project. In these early sessions, we are focussing on lithological data.

At the end of this session, you will be familiar with:

- Importing drillhole data
- Checking and labelling survey, collar and interval table data
- Viewing the data

Note: You can import additional data at any point during the modelling process. However, Leapfrog can have only one drillhole data set in a project. Therefore, additional drillhole data that is imported will be added to or will modify the existing drillhole data set.

Techniques for adding new drillholes or updating data down drillholes are covered in Session 5: Dynamic Updating.

The data for this session is in the `Sessions \ Session 2 to 6 - Wayleggo` folder.

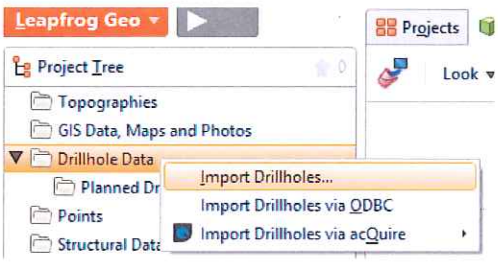
Importing Wayleggo WL campaign data

To start, we need to get data into Leapfrog. The answer to most things in the project tree is to right-click.

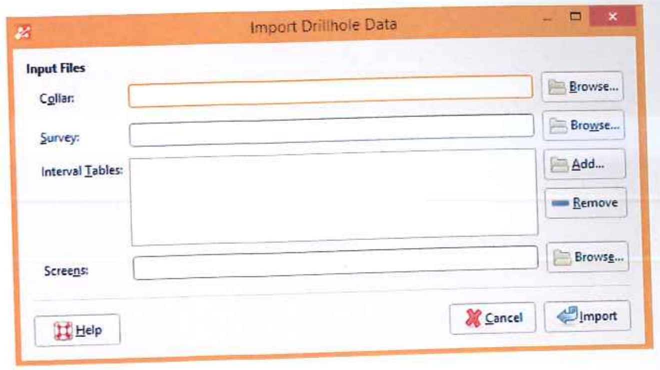
To import drillhole data:

1. Right-click on the **Drillhole Data** folder at the top of the project tree.

2. Select the **Import Drillholes** option:



The **Import Drillhole Data** window will open:



Drillhole data import requires a minimum of three files; a collar file, a survey file and at least one interval table file. Each file is imported as a table and has a certain number of required columns. You can also import additional columns, such as the date when measurements were taken, the name of the geologist who logged the drillhole, etc.

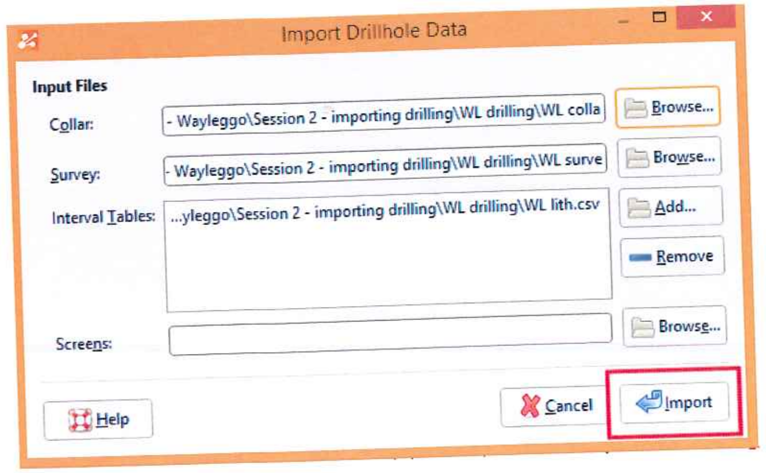
At this point, we need to add the collar, survey and interval tables to the **Input Files** list.

3. Click the **Browse** button for **Collar**.
4. Navigate to the location where the Leapfrog Geo Training Data folder has been saved.
5. Open the `Wayleggo` folder.
6. Open the `Session 2 - importing drilling` folder.
7. Open the `WL drilling` folder.
8. Select the file `WL collar.csv`.
9. Click **Open**.

Leapfrog looks for survey and interval tables in the same folder as the collar file, and if it finds them it will add them to the **Import Drillhole Data** window. If the survey and interval tables are in a different location, or have different file names, you will need to select the additional files. If this is the case, you can add files by clicking the **Browse** button for the survey file, or the **Add** button for **Interval Tables** and selecting the file.

In this case, however, Leapfrog has added all the files and we can start importing them.

10. Click **Import**:



This opens the **Import Drillholes** window with a preview of the data Leapfrog has read from your collar table. When importing and previewing drillhole data, required columns are marked in green, and additional columns are marked in orange.

Leapfrog works through the files being imported one at a time. Progress is shown at the top of the window. The filename in bold is the file currently being displayed (`collar.csv`). There are alsosurvey and lith files.

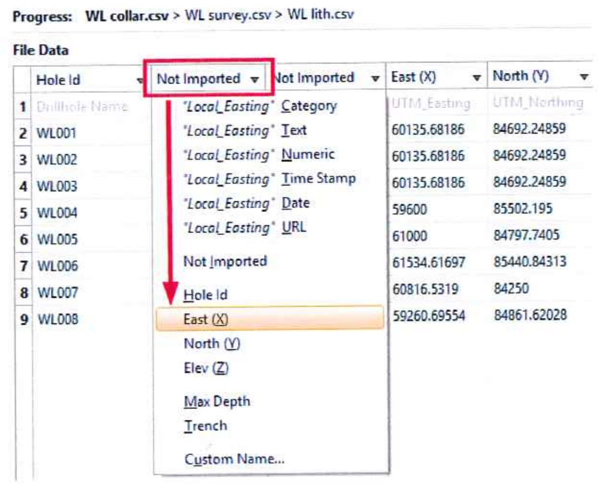
It is possible to choose which encoding you would like Leapfrog to use when importing drillhole data. By leaving the auto encoding option selected, Leapfrog will select the most suitable encoding type. If there are uncommon international characters in your drillhole data, select the **character encoding** dropdown and find the encoding type that matches your data.

The Collar File

The collar file needs at least the following columns: East (X), North (Y), Elev (Z) and Hole ID.

It is recommended that the collar file also has a maximum depth column that specifies the length of the hole. This can be used to validate interval table data with the collar file.

There are a number of data column types to choose from. Leapfrog automatically selects East (X), North (Y), Elev (Z), Hole ID and Max Depth based on the column headers from the database, but in this case the Easting and Northings have been incorrectly selected as the UTM files, rather than the Local files. To change the automatic selection, click the **Not Imported** dropdown above the Easting you would like to import (in this case 'Local_Easting'). By default this column is not imported, but we need to change it to **East (X)**. This will deselect the original, incorrect selection, and select the correct one.



Repeat for the Northing, so both Easting and Northing are using the local columns rather than the UTM columns.

When you import drillhole data, be careful not to get carried away with importing additional columns. When you later append other drillhole data, you will need to have exactly the same columns.

We have now selected all the columns we want to import.

11. Click **Next**.

The Survey File

The survey file defines the orientation of the drillholes at given depths. The survey file must have columns for Hole ID, Depth, Dip and Azimuth. For a straight drillhole, only one row needs to be specified.

In this case, the four required columns have been correctly selected by default.

12. Click **Next** to go to the lith table.

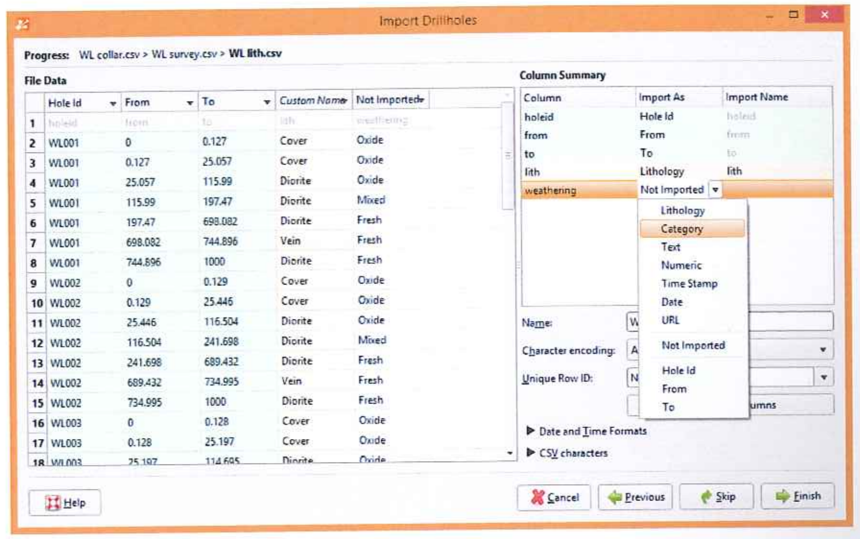
Interval Tables

Interval table data specifies segments down the drillhole with associated values such as lithology codes or assay values.

Interval table files must have columns for the Hole ID, From and To depth and one or more columns of measurements. Measurements can include grade, lithology, date or any required numeric or textual values.

The lith table has two columns we wish to import. To import multiple data columns at once, it is easier to work in the **Column Summary** on the righthand side. The lith column has been correctly set as "Lithology", but we need to set the other column so it can be imported.

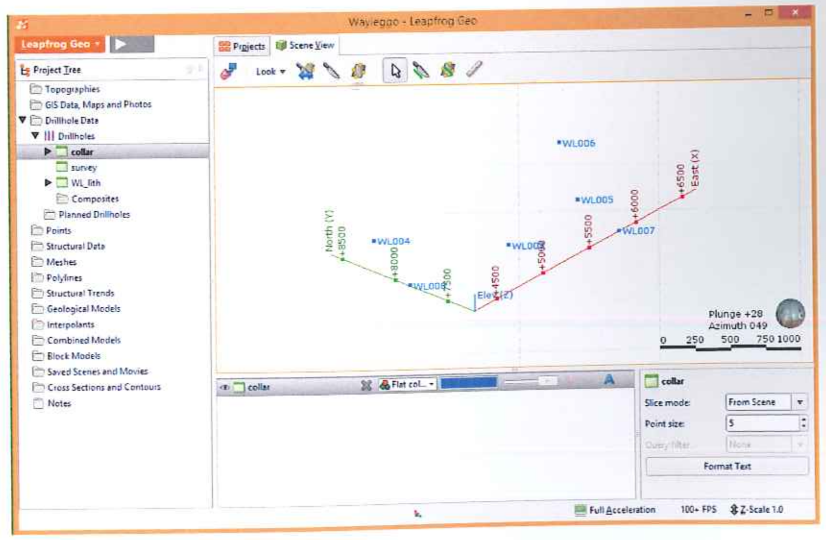
13. In the **Column Summary**, click on the "weathering" column and choose "Category":



Leapfrog will process the data, checking for errors in the database and desurveying the drillholes. Progress will be listed in the processing panel dropdown

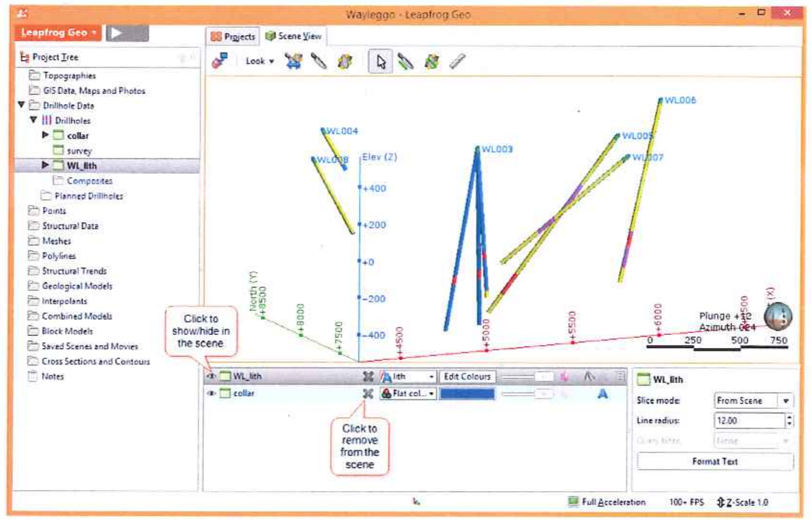
Viewing the Data

To view the imported data, drag and drop the individual tables into the scene window. Here, the collar table has been added to the scene:



There is nothing to view in the survey data but you can add the lith table to the scene to view the data.

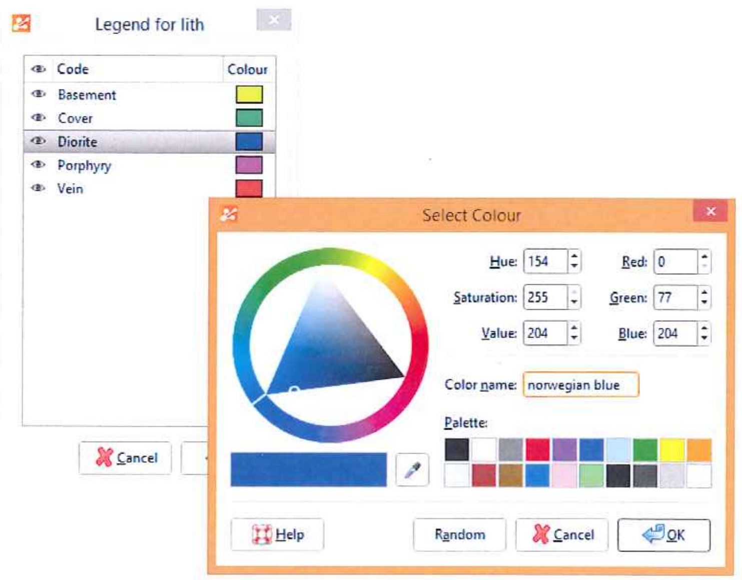
You will see the objects listed in the scene list:



Note that:

- There is an eye button for each object. Click the eye to hide it in the scene.
- The icon for each object indicates what type of item it is. In this case, both objects have the same icon because they are both tables.
- Next to the icon is the name of the object.
- The X allows you to remove the object from the scene. It does not delete the object from the project.
- The first dropdown list shows the object's attributes, giving you different options for displaying the object. For example, lith can be shown as lith, weathering or flat colour.
- The second dropdown list lets you edit colours or select colourmaps.
- A legend button appears for many objects. Click the button to show a legend in the scene.

If you select **Edit Colours** from the second dropdown list, you can change the colours used. Click on a colour to open the colour chart and either select your preferred colour, or use the eyedropper tool to select a colour from your screen:

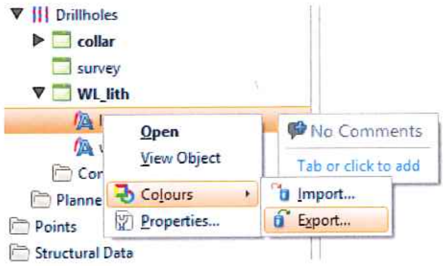


leapfrog
FAST, DYNAMIC GEOLOGICAL MODELLING

Importing and Exporting Colour Files

Once a series of colours have been chosen in Leapfrog Geo, they can be exported as an *.lfc file (Leapfrog Colour File). These can then be reimported to other Leapfrog projects to save time setting the colours up. This is particularly useful if your company has a specified colour scheme which is consistent across different software packages and different projects.

To export an *.lfc, right click on the **Lith** table and select **Colours>Export**



To import a colourmap, repeat the steps above but select **Import** rather than **Export**.

leapfrog

FAST, DYNAMIC GEOLOGICAL MODELLING

Session 3: Creating a Topography

Contents

Goals

In this session, we will import topography points and use them to create a topography. We will then import an image and drape it on the topography. At the end of this session, you will be familiar with:

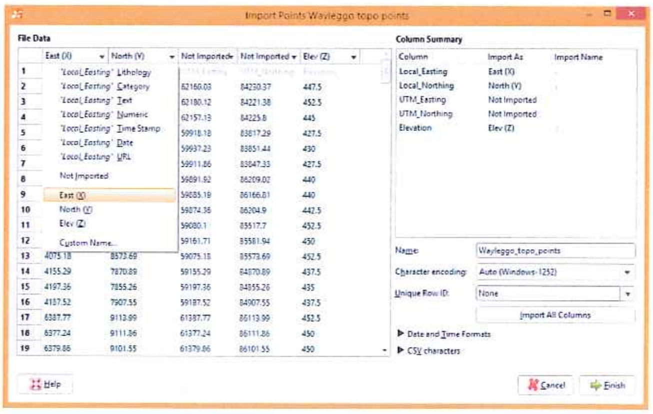
- Importing points that correspond to the topography
- Creating the topography
- Viewing drillhole data in the scene
- Constraining the model to a clipping boundary
- Importing a georeferenced image
- Draping an image and GIS data on the topography

The data for this session is in the `Sessions \ Session 2 to 6 - Wayleggo` folder.

Importing Topography Points

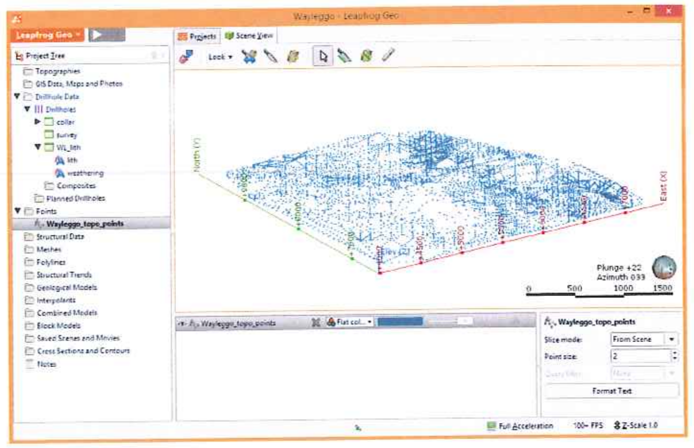
To select the topography points to import:

1. Open the project from Session 2: Importing Drillhole Data.
2. Clear the scene.
3. Right-click on the **Points** folder in the project tree
4. Select **Import Points**.
5. Browse to the folder for this session, and double click on the topography points to import them.
6. Make sure all three columns (East (X), North(Y) and Elev (Z)) are selected for importing. As with the drillhole collars, the UTM Eastings and Northings have been selected instead of the Local Eastings and Northings. Change the selected columns to the Local data rather than the UTM data.



7. Click **Finish**.

8. Add the points to the scene to view them by dragging the Wayleggo_topo_points object from the project tree into the scene:



Creating a Topography

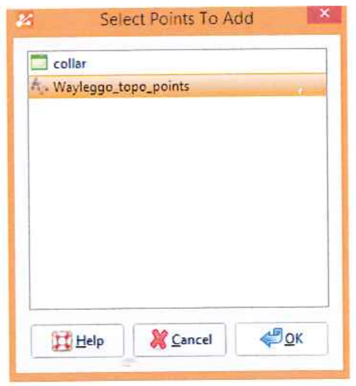
To create the topography:

1. Right click on the **Topographies** folder in the project tree.

2. Select **New Topography > From Points**.

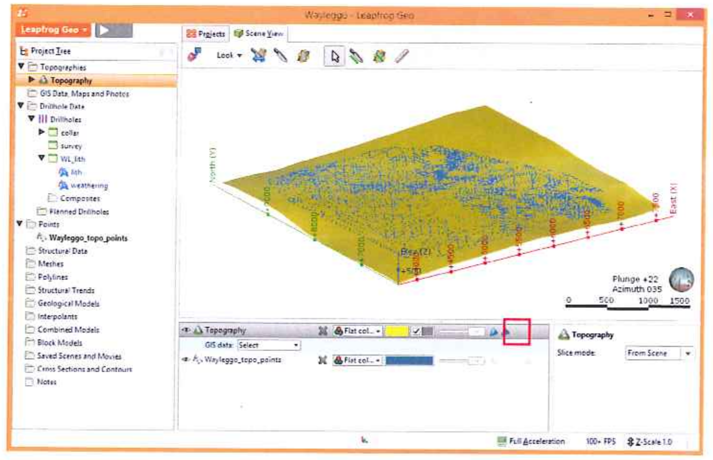
As the collar file contains points (x, y and z), as well as the newly imported topo points, the option to choose either is available.

3. In this case the topo points contain more detail, so select the Wayleggo_topo_points file and click **OK**:

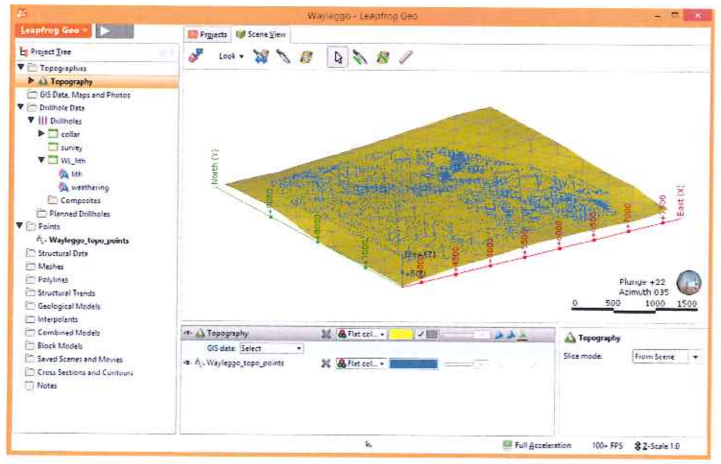


4. In the window that appears, accept the default name for the topography by clicking **OK**.

5. Drag the new topography into the scene:

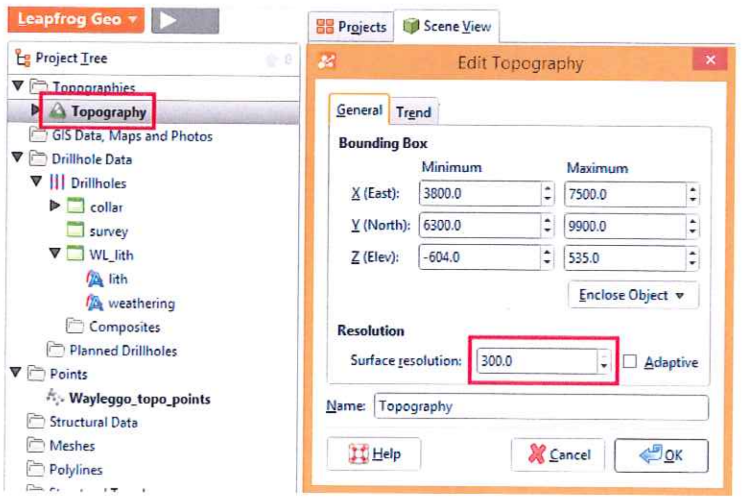


6. At the moment we are looking at the surface, but to view the triangles making up the surface, click the **Show Edges** icon which is highlighted in red in the image above.



We can see that the triangles are reasonably coarse compared to the point spacing. There are a few options in Leapfrog to make the surface represent the points more accurately.

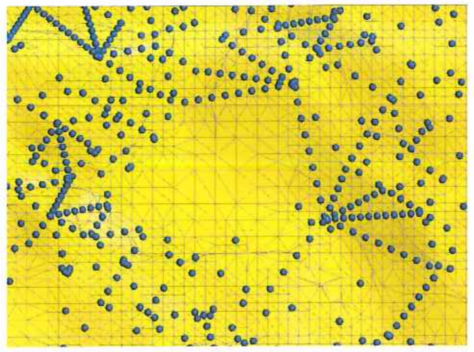
1. Double click on the topography object in the project tree



2. There is a surface resolution box, as well as an adaptive checkbox.

3. By lowering the surface resolution, the triangles making up the surface will become smaller. The resolution is approximately equal to the length of the sides of the triangles. In this case, a resolution of between 25 and 50 is appropriate.

4. Ticking the adaptive checkbox allows a surface to be created which consists of fine resolution where the data spacing is small, and coarse resolution where the data is far apart:



An alternative way to create a topography is from the collar points. To do this, follow the above steps but select the collar data instead of the Wayleggo_topo_large points.

Displaying Drillhole Data with the Topography

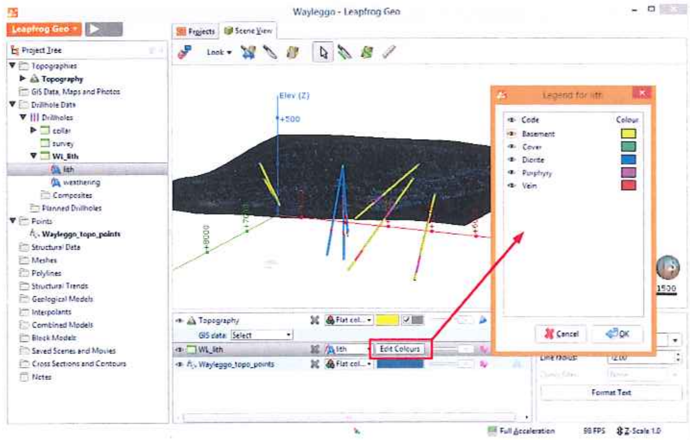
To add the drillhole data to the scene window:

1. Drag the WL_lith table into the scene.

2. Rotate the scene to view the drillholes.

All your drillhole data should be below the topography, although it may not match if the drillhole data is older than the topography data.

3. Click on **Edit Colours** in the scene list:



4. Experiment with hiding units in the scene and changing their display colours.

5. Dismiss the colour legend by clicking **OK**.

Importing and Draping Images and GIS Data

In this section, we will import a georeferenced image and display the data in the scene draped on the topography. First, we will import the image.

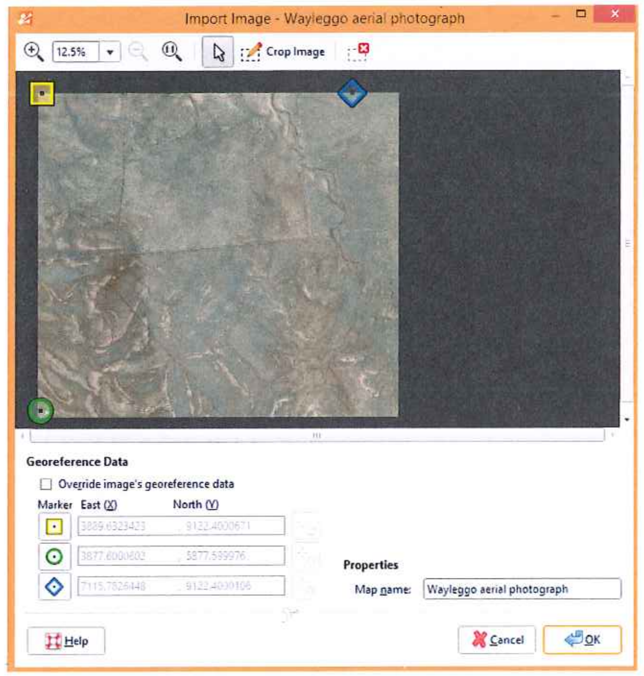
1. Right click on **GIS Data, Maps and Photos** in the project tree.

2. Select **Import Map**.

3. Select the `Wayleggo aerial photograph.tif` image.

4. Click **Open**.

A window will appear in which you can choose to either use the existing georeference data, or override the georeference.



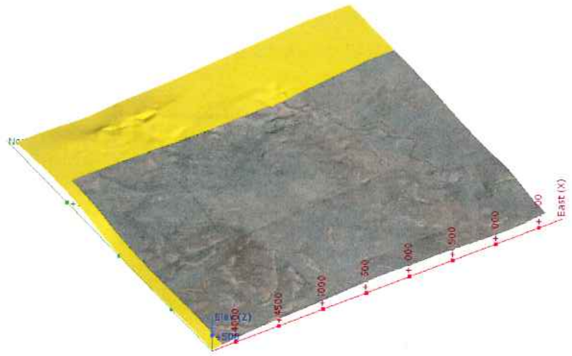
5. This file has been georeferenced, so we can click **OK** and it will be imported into the project.

To drape the image on the topography:

1. Add the topography to the scene.

2. In the shape list, click on the **GIS Data** dropdown list and select the imported image

The image will appear draped on the topography:

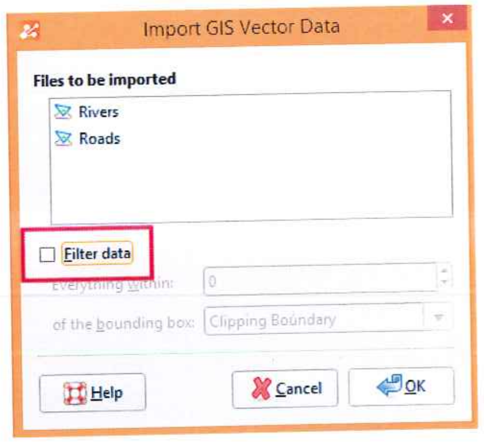


There are also GIS lines representing rivers and roads, which can be imported and draped on the topography.

1. Right click on the **GIS Data, Maps and Photos** folder and select **Import Vector Data**.

2. There are two files which can both be imported at the same time by selecting them both and clicking **Open**.

3. Deselect the **Filter Data** checkbox. This allows you to filter GIS Data to within a certain distance of a bounding box, but in this case we would like to import the complete files.

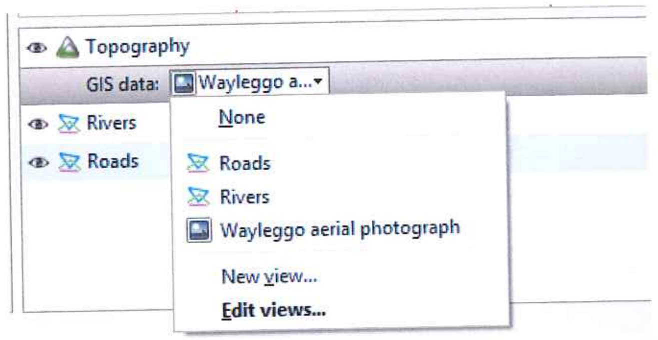


4. Click OK and the files will be imported.

5. Drag the files into the scene to view them, and note that the default elevation is +550 m, which is the elevation which the lines were originally exported in.

The next step is to drape the GIS lines onto the topography and view them alongside the map. To do this, we need to create a new view.

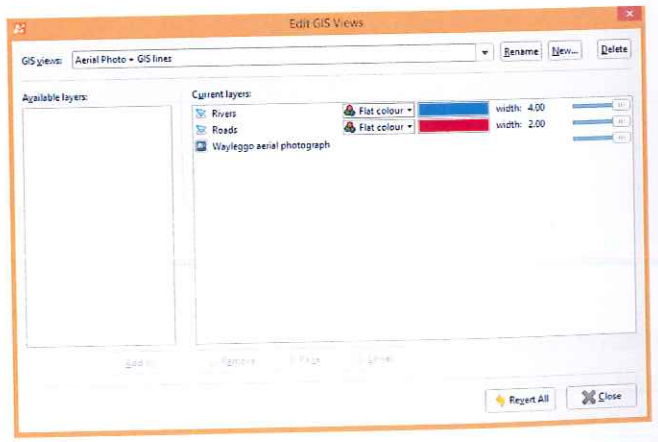
1. Make sure the topography is in the scene, and click the **GIS Data** dropdown in the scene list.



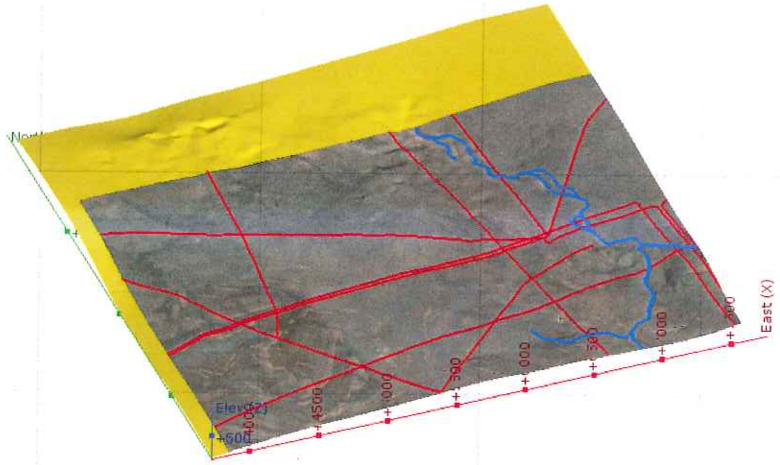
2. Select **New View** and rename the view 'Aerial Photo + GIS lines'.

3. The **Edit GIS Views** window will appear, which gives you the option to add images and GIS data to the **Current Layers** window. Add the two GIS lines and the aerial photograph to the **Current Layers** window.

4. Make sure the GIS lines are listed above the aerial photograph, as this gives them priority so they will be visible on top of the aerial photograph. Change the colours and line widths of the GIS lines appropriately.



5. Click **Close** and the new view will be displayed in the scene.



Session 4: Building a Simple Geological Model

Contents

Goals

In this session, we will model a young vein which cuts an older porphyry intrusion, which in turn cuts an older diorite intrusion within the basement. The intrusions and basement are overlain by cover which is up to 25m thick.

At the end of this session, you will be familiar with:

- How different surface types work in Leapfrog, and when to use each type
- Creating a geological model
- Defining an erosional surface
- Defining veins
- Defining intrusions
- Using the moving plane to adjust a surface
- Defining the chronological order of surfaces
- Changing the resolution of a geological model

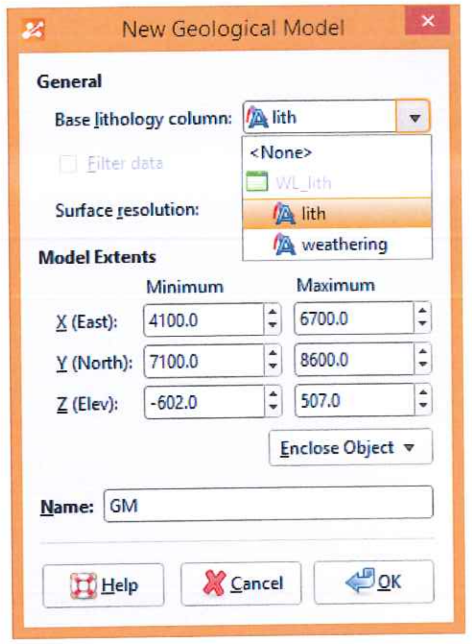
For this session, open the Wayleggo project from sessions 2 and 3.

Creating a New Geological Model

To create a new geological model:

1. Right click on **Geological Models** in the project tree and select **New Geological Model**.

2. In the window that appears, make sure the lith column is selected for the **Base lithology column**.



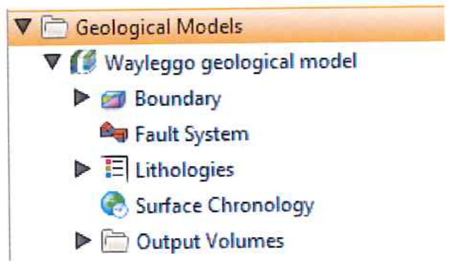
You can change many geological model settings once the model has been created, but you cannot change the lithology column used to create the model.

3. Click the **Enclose Object** dropdown, and select the **WL_lith** table, which will cut the boundary of the geological model to the extents of the drilling.

4. Change the Surface Resolution to 20, and the name of the GM to Wayleggo geological model.

5. Click **OK**.

The new geological model will appear in the project tree made up of several objects:



Different users start their models differently, but one common method is to look at the lithologies which seem more 'obvious'. The best way to look at this is by clearing the scene, dragging your drillholes in and visualising them in 3D. When we do this, it seems that the cover (at the top of each drillhole) and the vein (cutting through 5 of the drillholes) are reasonably obvious, so we should be able to create surfaces representing them easily. We will model the cover first, then the vein, then the two intrusions. The last lithology to be modelled will be the basement, which will be 'everything else', so we don't need to create its surface.

Modelling the Cover

The last lithology we need to create a surface for is the cover. To do this, we will use the erosion tool. Depositional surfaces and erosional surfaces are both created in the same manner, with the difference being in the way that the surfaces are cut when producing volumes. This is discussed further at the end of this session.

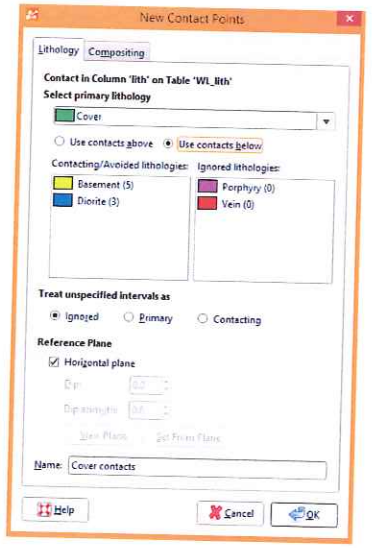
To create the erosional surface:

1. Right click on the surface chronology folder and select **new erosion** > **from base lithology**.

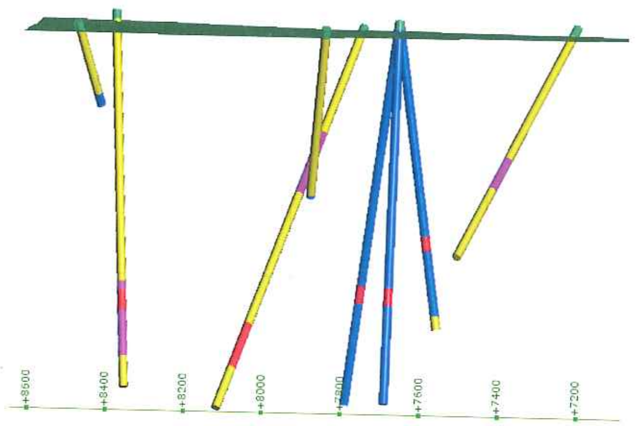
2. The **new contact points** dialogue will appear

3. In this case, we will set the primary lithology to 'cover', and change checkbox from 'use contacts above' to 'use contacts below'.

'Use contacts below' ensure the surface will pass through all instances where a lithology contacts *below* the cover. You will see numbers listed after the basement and diorite; these tell us how many times the cover is contacting with each lithlogy. In this case, the cover contacts with the basement 5 times and the diorite 3 times.



4. Drag the surface into the scene to view it.



Modelling the Vein

To display the vein in the scene:

1. Clear the scene.

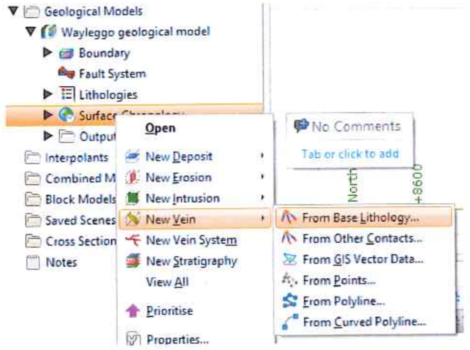
2. Add the WL_lith interval table to the scene.

3. Change the display so you can only see the vein lithology.

There are only five intervals which intercept the vein, but they contain enough information to create a reasonable volume .

4. Right click on the surface chronology folder under the geological model and select **new vein** > **from base lithology**. There is an option to include points at the ends of holes - this is for when a drillhole ends in the vein

lithology, and lets you choose whether or not to create a point representing the end of these drillholes. In this instance it isn't relevant, as there are no drillholes which end in the vein lithology.

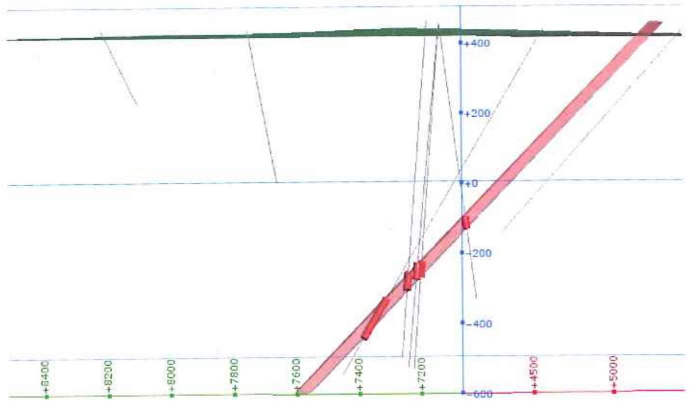


5. Click **OK**, and you will see the vein being created in the processing panel to the bottom left of the Leapfrog window.

6. Drag the vein into the scene.

You will see that the vein is clipped to the topography by default, and does a good job of representing the lithology shown in the drilling considering the limited information.

There are a number of settings which can be changed once the default vein has been produced, but for now we will leave these as they are and come back to them later.



Modelling the two intrusions

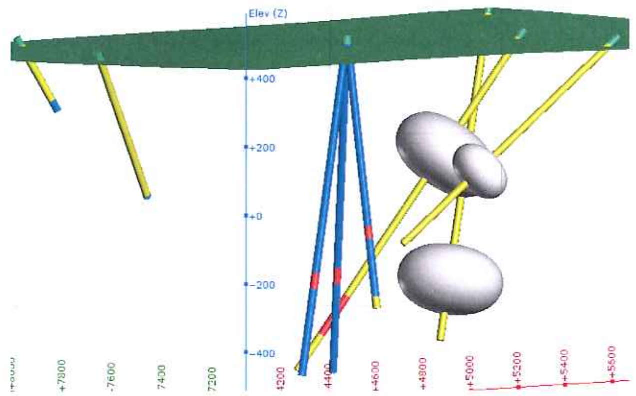
There are two intrusions shown in the drilling - these are the porphyry and the diorite. We will model these using the intrusion tool which is found under the surface chronology folder.

The porphyry is the younger intrusion so we will model it first.

1. Right click on the surface chronology folder and select **new intrusion** > **from base lithology**

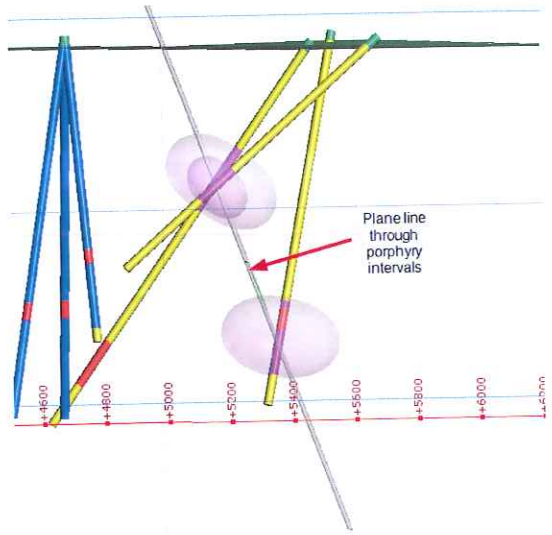
2. Select porphyry as the interior lithology, and drag the cover and vein lithologies across to the 'ignore' window. The lithologies which remain in the 'Exterior lithologies' window will be used in the point generation; whenever the porphyry contacts against either the basement or the diorite, a point will be created. The cover and vein lithologies are ignored here because they are both younger than the porphyry, so didn't exist when the porphyry was emplaced.

3. Drag the porphyry into the scene:



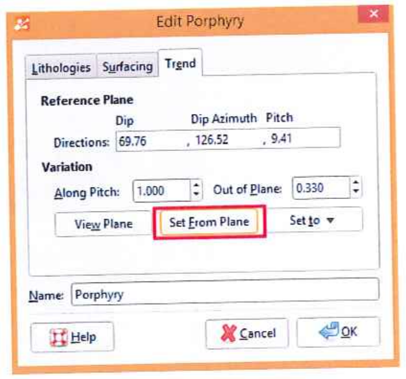
Three volumes are created - we would expect these to link, but there isn't enough data to extend each volume to meet each other. To fix this, we can add a global trend. This allows us to specify a direction of maximum continuity and a direction of minimum continuity, which control the stretch of the intrusion.

4. Draw a plane through the porphyry intervals.



5. Double click on the porphyry intrusion, then click on the **trend** tab.

6. Click **set from plane**, which will copy the dip, dip azimuth and pitch from the plane across to the intrusion.



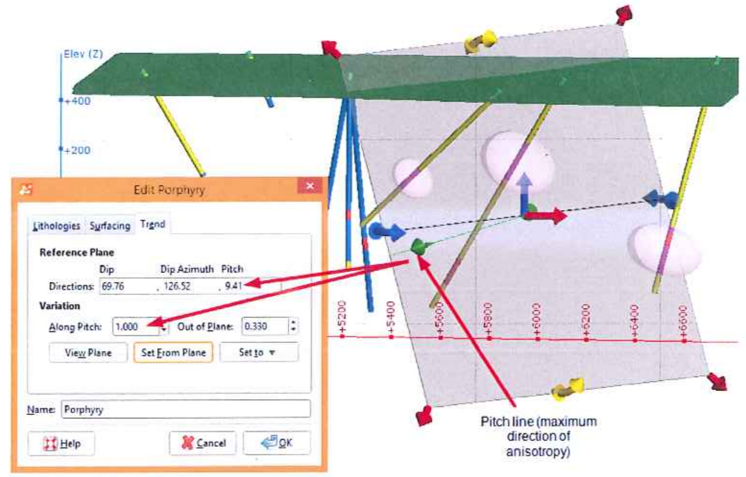
7. The final change we need to make is the **along pitch** and **out of plane** values.

8. Along pitch refers to the maximum direction of continuity, and is represented with the green arrow on the plane.

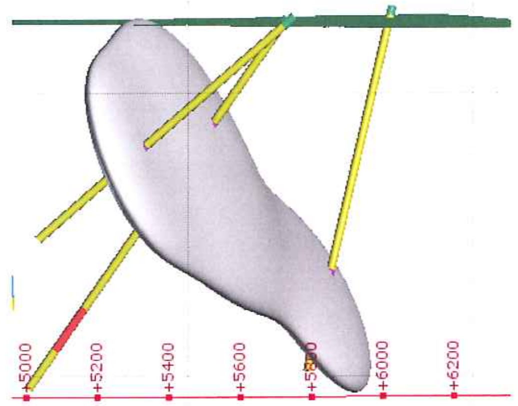
9. Out of plane refers to the minimum direction of continuity, which is the out of plane direction.

10. The intermediate direction of continuity cannot be changed, and is set to 1.

The important point to remember is that the ratio between the maximum, minimum, and intermediate (set to 1), lets us control the surface. In this case, by setting the **along plane** value (maximum) to 1, and the **out of plane** value (minimum) to 0.3, and remembering that the intermediate value is always set to 1, we can make the intrusion look more realistic.



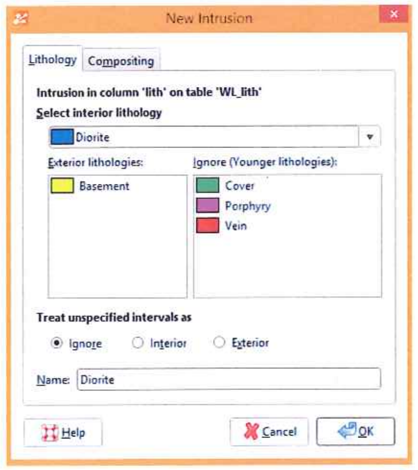
11. Click **OK** and view the new surface in the scene.



Repeat for the diorite intrusion:

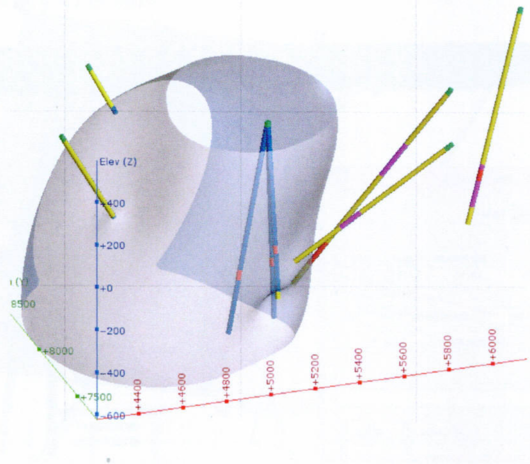
1. Right click on the surface chronology and select **new intrusion** > **from base lithology**

2. Ignore the cover, porphyry and vein lithologies. The only exterior lithology will be the basement.



3. At this stage there is no obvious trend that can be set, as we have a limited number of intercepts. Once we add additional drilling we will be able to more easily see the trend of this intrusion.

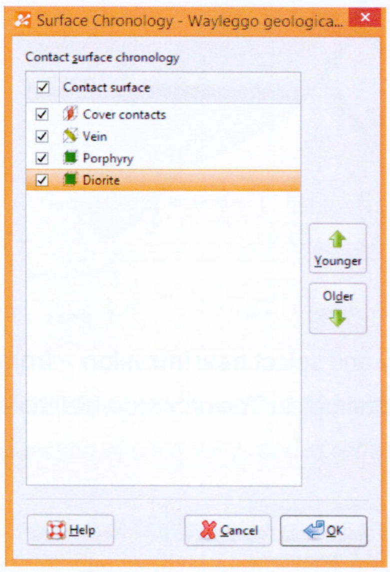
4. Click **OK**, and drag the cover surface into the scene.



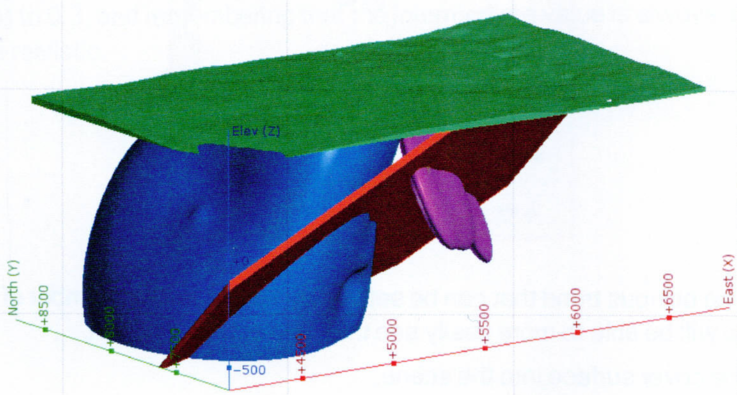
Creating output volumes

We now have four surfaces representing four out of the five lithologies, with the final lithology being the basement.

1. To turn these surfaces into volumes which cut against each other, double click on the surface chronology folder, change the order of the four lithologies so that the youngest is at the top and the oldest is at the bottom:

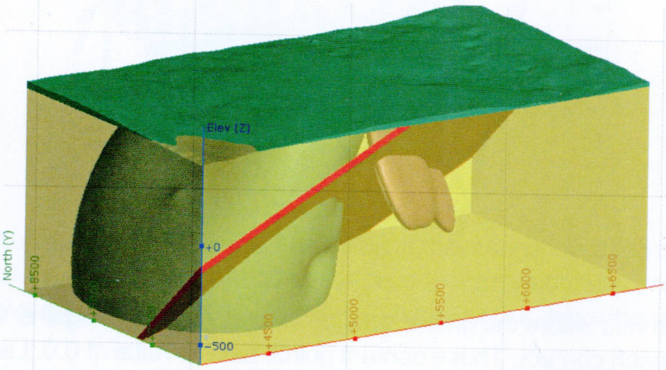


2. Click **OK**, and the surfaces will be cut against each other to produce volumes.



3. The last step is to change the 'unknown' lithology to 'basement'.

4. Double click on the diorite intrusion surface (the second to oldest lithology), and change the second lithology (which is the older lithology) to basement.

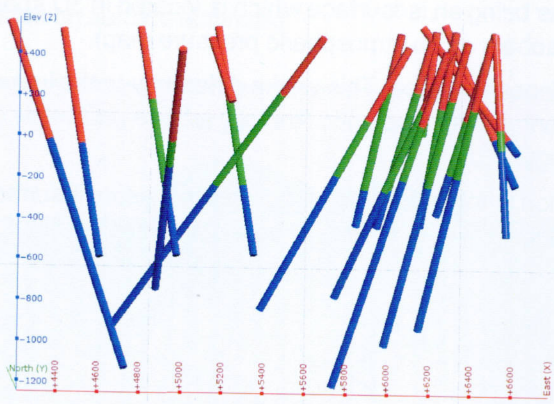


Surfaces in Leapfrog Geo

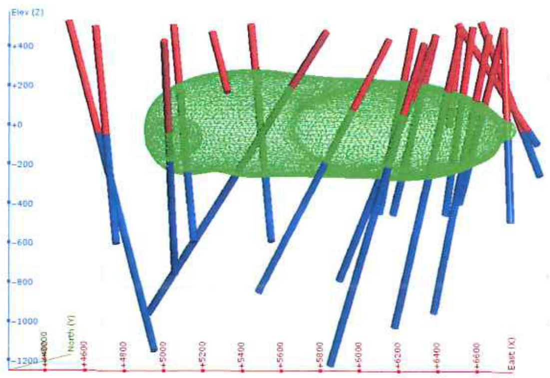
As we have seen, there are a number of different ways to create a surface in Geo. We have focused on creating surfaces within the **Geological Models** folder, where we have been using the **Surface Chronology** subfolder.

When you right click on the Surface Chronology subfolder, all the available options appear. These include deposit, erosion, intrusion, vein, vein system, and stratigraphy. It is important to remember that even though each surface has a specific name, it is the method of surface creation which is more important. For example, intrusion surfaces can be used when the user would like to create a surface surrounding a specific lithology (it doesn't necessarily need to be an intrusion). Likewise, a deposit or erosion surface can be used when the user would like to create a surface between one lithology and a number of other lithologies.

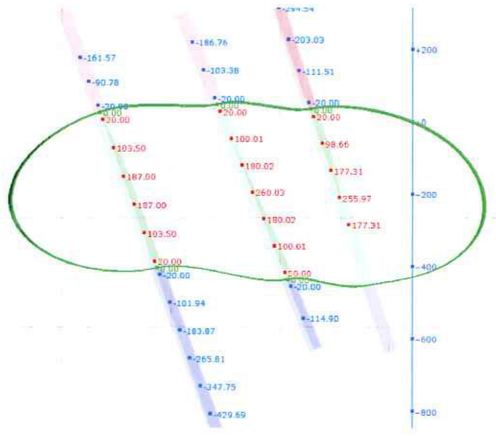
As a very simple example, the below screenshots have been taken using 3 lithologies; upper (red), lower (blue) and intrusion (green). By creating and changing the settings of the geological model, we will demonstrate how each surface type works.



Firstly we will use an intrusion surface to represent the green intrusion. Here is the result:



Leapfrog has found the intervals where the green intrusion lithology contacts against either the red or blue, and has put 'contact points' at each contact. These contact points have a value of 0.0. Leapfrog has also added 'volume points' up and down the drillholes. As we move progressively further away from the intrusion, the value of the volume points decreases in the negative direction. As we get further inside the intrusion, the value of the volume points increases in the positive direction. Here is a section view of the volume points:

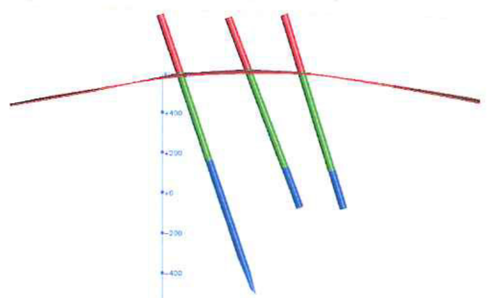


Think about the actual surface as being an isosurface which is located in 3D space where ever a value of 0.0 exists (a 2D analogy would be isobars on an atmospheric pressure map).

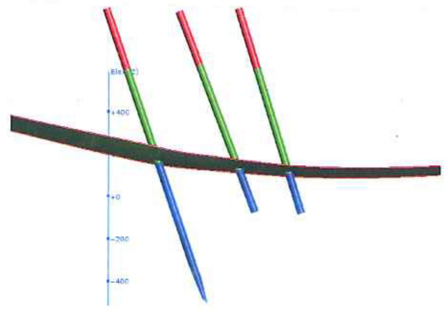
Next we will look at creating a deposit surface. This works differently to the intrusion surface, as instead of Leapfrog creating points at all contacts for a specific lithology (above *and* below it), it creates points either above *or* below a specific lithology.

If we use the same green intrusion lithology as above, but create a deposit surface rather than an intrusion surface, we get the following (looking at the same view as the previous image).

Using 'contacts above':



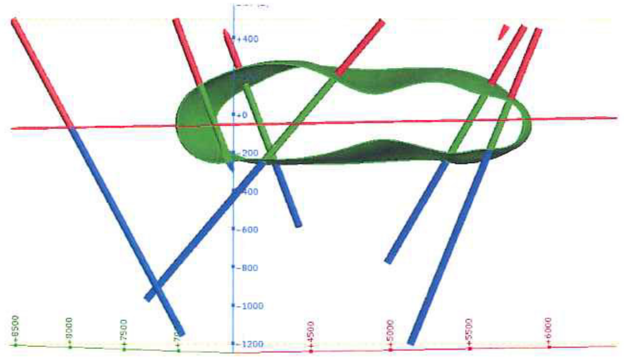
And using 'contacts below':



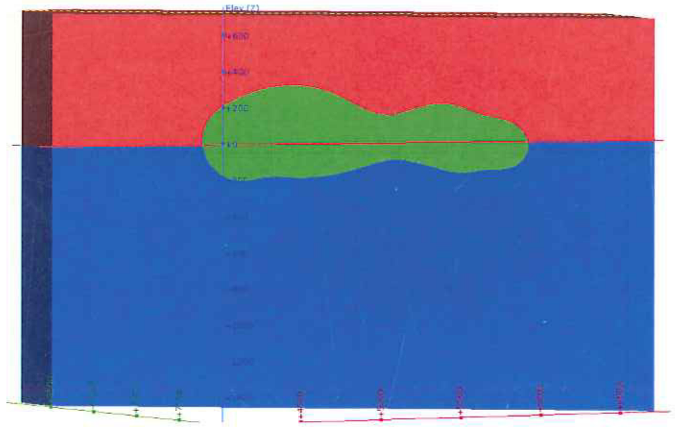
Deposit surfaces and Erosion surfaces are exactly the same in the manner in which they are constructed, but differ in the manner in which they are cut against other surfaces to form volumes. Erosion surfaces act as unconformities, so 'cut' any surfaces they cross. Deposit surfaces act as depositional layers, so 'stack' on top of any surfaces they cross.

For a simple demonstration, the contact between the upper (red) and lower (blue) lithologies has been modelled using both erosion and deposit surfaces, and the interaction between these surfaces and the intrusion surface when all are activated and turned into volumes becomes apparent (noting that the erosion and deposit surfaces are exactly the same shape).

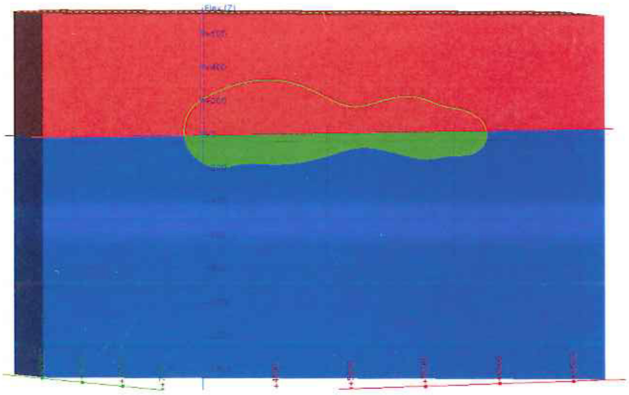
Here are the surfaces shown in section view:



With the deposit surface and intrusion surface activated:



With the erosion surface and intrusion surface activated:



You can see the difference between the two images; in the top image the upper layer is a deposit, so stacks on top of the green intrusion. The the bottom image the upper layer is an erosion, so cuts across the green intrusion.

Remember that there are a huge number of options available when creating surfaces which we will discuss as we move through the training course, and the above example is only a simple demonstration of the most basic surfacing options.

Session 5: Dynamic Updating

Contents

Goals

In this session, we will append drillhole data by adding in a new drilling campaign. We will create an updated geological model and check the updated model against the original.

At the end of this session, you will be familiar with:

- Appending drillhole data
- Using a query filter
- Copying a geological model
- Evaluating a new model against the original

For this session, open the Wayleggo model project from earlier and clear the scene.

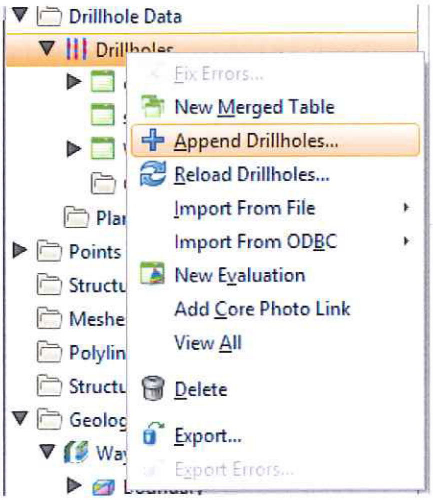
The data to append is in the `Sessions \ Session 2 to 6 - Wayleggo` folder.

Appending Drillhole Data

First, we will import the data from the new drilling campaign.

1. Right click on the **Drillholes** object in the project tree.

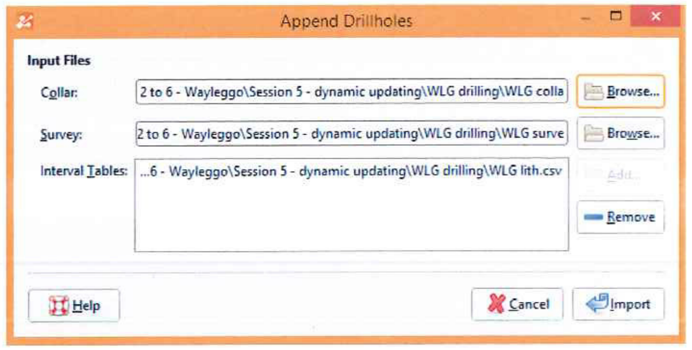
2. Select **Append Drillholes**:



3. Click **Browse** to locate the collar file for the WLG campaign.

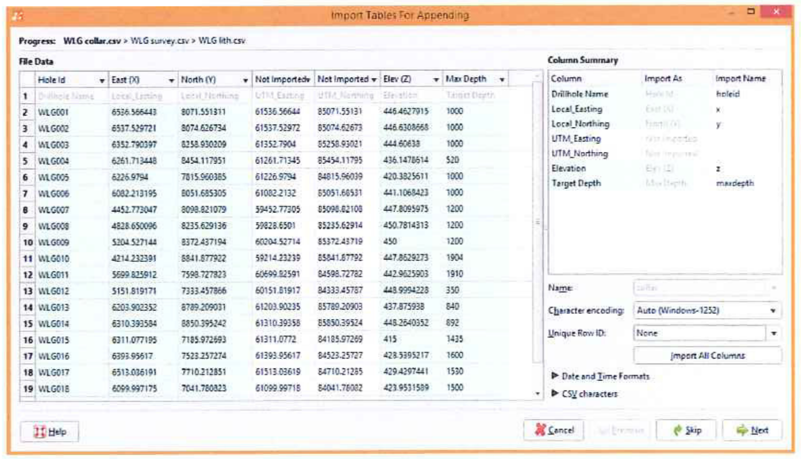
leapfrog
FAST, DYNAMIC GEOLOGICAL MODELLING

4. Select the `WLG Collar.csv` file and click **Open**.



5. Click **Import** to start working through the files, checking that the new data aligns with the existing data tables.

In the **Column Summary** part of each import window, notice that each file will be importing the same columns as the original drillhole campaign, including the changes we made in Session 2 to import the Local Easting and Northing columns rather than the UTM columns:



You are unable to add any "new" columns at this stage.

6. Work through the files, then select **Finish**.

Your drillhole data will now contain information from both drilling campaigns. Because Leapfrog works in a top-down manner, the geological model depends on the drillhole data and will also be updated when the drillhole data is appended.

Updating the Geological Model Extents

Although the geological models in the project have been updated with the new drillhole data, the geological model extents remain those set when the models were created. We will change the extents for the simple geological model created in the earlier sessions.

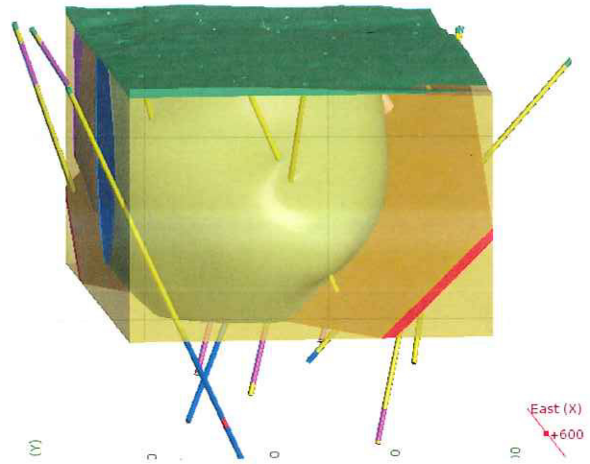
1. Clear the scene.

2. Add drilling to the scene

3. Add the existing geological model to the scene

You will see that a number of the drillholes are visible outside the extents of the geological model.

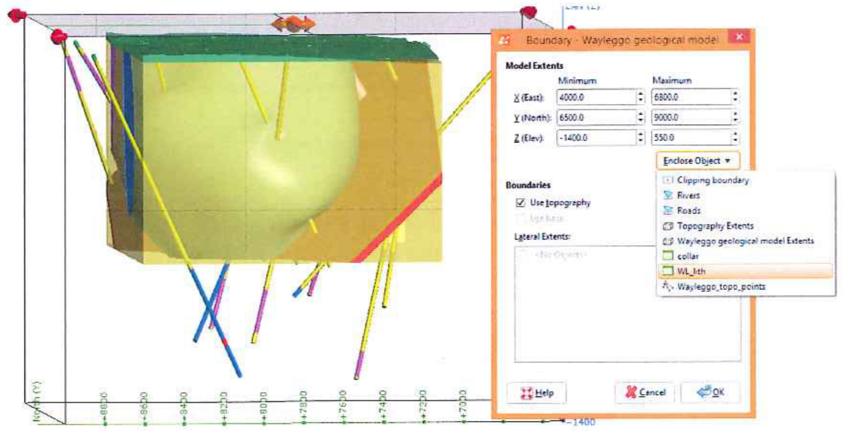
leapfrog
FAST, DYNAMIC GEOLOGICAL MODELLING



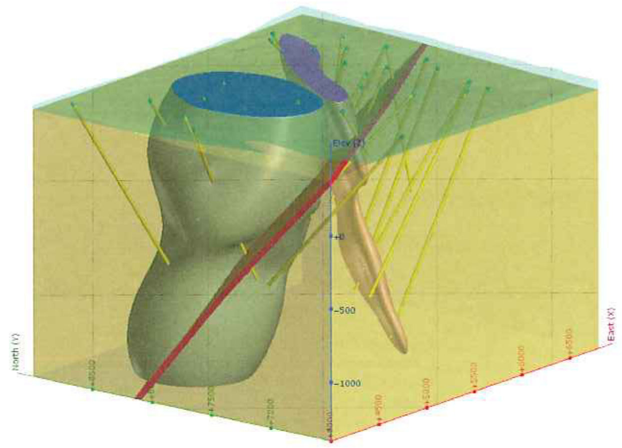
4. Double click on the **boundary** folder under the geological model

5. Click the **enclose object** dropdown, and select the WL_lith table, which will extend the boundary so it encloses the updated WL_lith data.

6. You will see in the scene that the boundary with red arrows has expanded to include the new drillholes.



7. Click **OK** and the model will re run and extend to the edges of the new boundary:

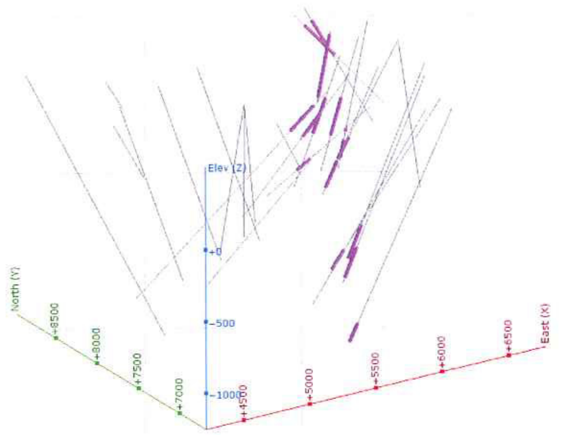


Examining the Geological Model

While Leapfrog automatically updates models, it doesn't do the same for the geological interpretation. Once the project has updated, we need to see whether or not the updated model fits our original interpretation.

To examine your Porphyry unit

1. Clear the scene.
2. Add the WL_lith table to the scene.
3. Keep the porphyry lithology visible, and turn the others off



4. Add the porphyry output volume to the scene.
5. Examine the volume to ensure it still honours the drillhole data, and that the trend which was set earlier is still suitable. To view the plane which was used to set the trend, double click on the porphyry surface, then click on the trend tab, and click 'view plane'.
6. In this case, the trend is still reasonable so we don't need to make any changes.
7. Clear the scene.

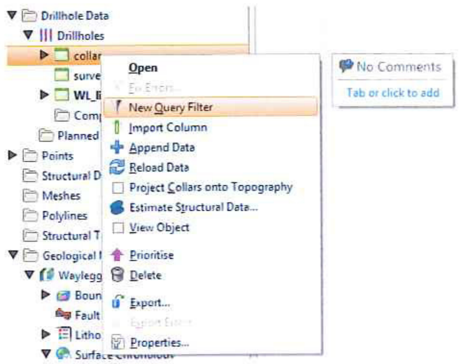
Evaluating the New Model Against the Original

A useful feature when appending drillhole data is being able to evaluate a new model against the original. This helps you gain a better understanding of the difference any new drilling has made.

The first step of this process is being able to distinguish the original drilling campaign from the new one. To do this we need to create a query filter, which we will base on the names of the drilling campaigns

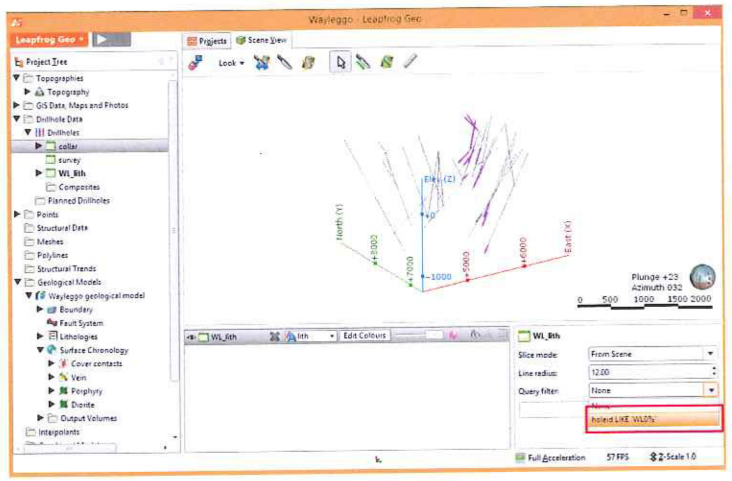
To create a query filter:

1. Right click on the collar table and select **New Query Filter**:



FAST, DYNAMIC GEOLOGICAL MODELLING

2. Select the '...' button to the right of the window which appears. This will open the **build query** window.
3. By clicking in the top row of the window, a series of dropdown arrows will appear.
4. Click on the dropdown arrow labelled **column**, and select **collar.holeid**.
5. Change the **test** value to **starts with**, and type 'WL0' (using the number 0, rather than the letter O).
6. Click **OK**, give the query filter a name, and click **OK** again.
7. In the shape properties panel to the bottom right of the Leapfrog window, change the query filter dropdown so it displays the recently created filter.

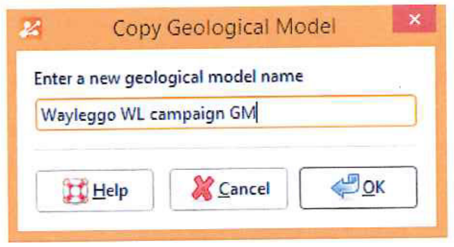


The drillhole traces for all drillholes will still be visible, but only the intervals from the original WL campaign are displayed.

You can display the whole drillhole dataset by selecting None from the **Query filter** dropdown list in the properties panel.

Next, we will make a copy of the geological model.

8. Make a copy of the geological model by right-clicking on it and selecting **Copy**.
9. Name the new model "Wayleggo WL Campaign GM".

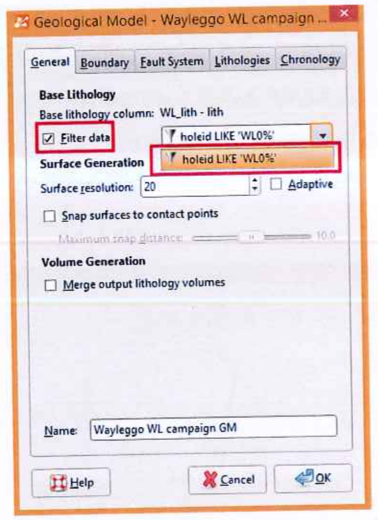


10. Click **OK**.

Next, we will use the query filter to define the new geological model.

11. Double-click on the Wayleggo WL Campaign GM.
12. Tick the **Filter Data** option.

The **Query Filter** default should be the WL0 filter:



13. Click **OK**.

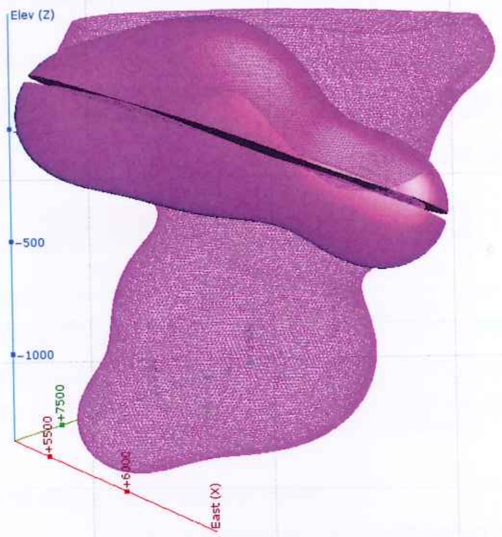
To compare the new and original geological models:

1. Clear the scene.

2. Add the porphyry volume from the original model the scene.

3. Add the porphyry volume from the Wayleggo WL campaign GM.

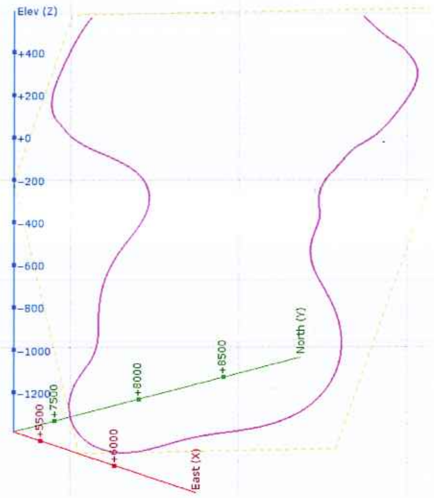
4. Differentiate the GM model volume from the original volume either by turning on the show edges option for one of the volumes, or by changing the transparency of one of the volumes:



We can easily see the difference the additional drilling has made to the size and shape of the porphyry volume. This view also clearly shows that to the south west of the model (on the left of this image), the porphyry intrusion has not been realistically modelled. By inspecting the drilling surrounding this area, we can see that the default surface runs between two drillholes, one with porphyry in it, and the other without. To edit this surface, we will use polylines.

Editing a surface using polylines

1. Put a slicer through the surface so that the area which needs to be edited is sliced through.



2. Right click on the surface which needs to be edited (in this case the porphyry), and select **edit** > with **polyline**.

This brings up the polyline editing toolbar:

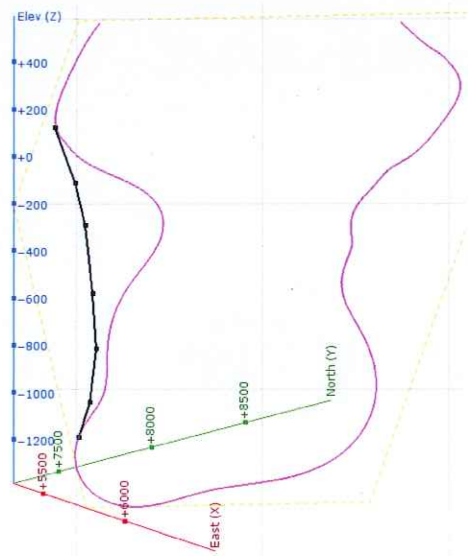
The **draw lines** icon allows a new line to be drawn.

The icons to the right of the toolbar allow the user to either draw on the slicer, or draw on an object.

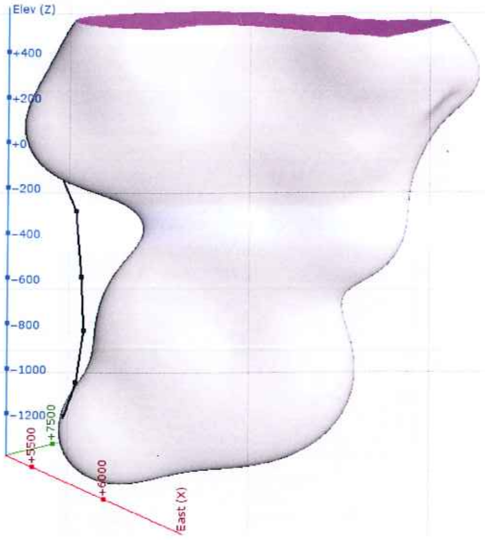
There are options for undo and redo, as well as save and cancel the drawing.

3. To draw a polyline, click on the draw lines icon and left click once on the sliced plane. Continue left clicking to create points which are linked to make a line.

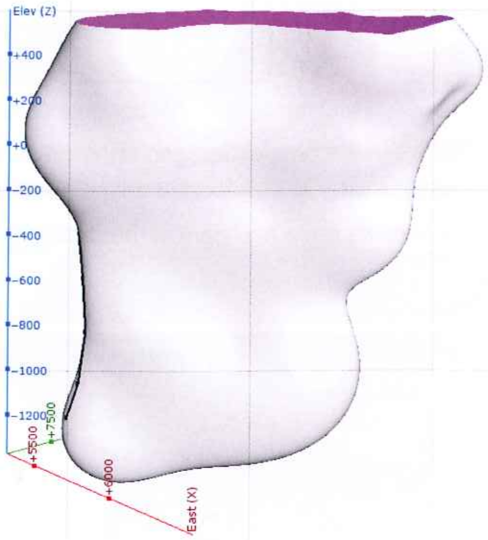
4. To complete the drawing, right click. Then right click again to return to cursor mode.



5. Remove the slice to check that the polyline looks sensible compared with the original surface.



6. When you are happy with the polyline(s), click save in the top toolbar. This will add the polyline to the data which is being used to create the surface.



The polyline which was used to edit the surface can be viewed under the surface in the project tree by expanding the triangles out:

To edit the polyline, right click on it and select **edit polyline**.

Exercise 1: Wayleggo Weathering Model

Contents

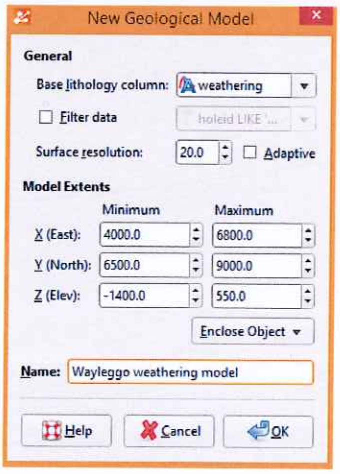
Goals

For this exercise, we will create a weathering model from the Wayleggo dataset which has been used for the previous four sessions.

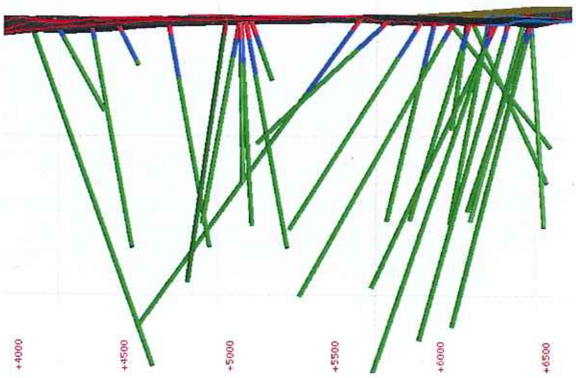
The data for this exercise is available in the project which has already been created, and doesn't require any additional importing.

Creating a geological model

Create a new geological model, making sure to select **weathering** for the base lithology column. Drop the resolution down as appropriate, limit the extent of the surfaces to the drilling, and name the model 'Wayleggo weathering model'

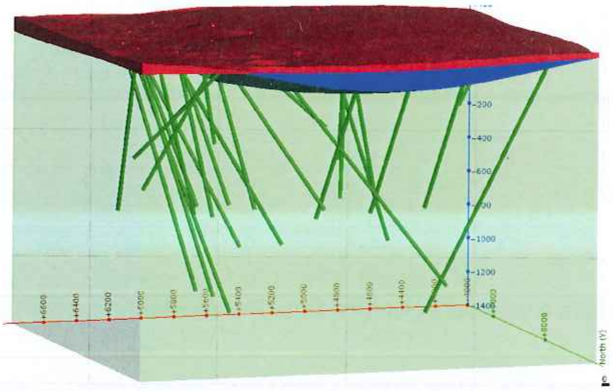


Visualise the drilling in 3D to get an idea of the volumes which will need to be modelled:



1. Decide on the best method of creating the two required surfaces - note that there are several possibilities here.

2. Once the surfaces have been created, activate them in the surface chronology folder and make sure they are in the correct chronological order.



Session 6: Presentation Tools

Contents

Goals

In this session, we will explain how to use display features in Leapfrog for report writing or presentation purposes.

At the end of this session, you will be familiar with:

- Saving and exporting scenes
- Using the Leapfrog Viewer
- Creating movies
- Rendering images

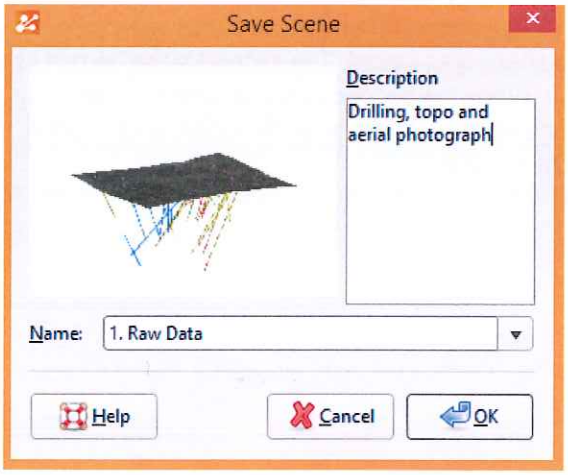
For this session we will use the Wayleggo project from sessions 1 to 6.

Saving and exporting scenes

Start by opening the Wayleggo project, and view the raw data in the scene. This should include the topography with draped aerial photograph, as well as the drilling.

1. Save the scene by right clicking on the **Saved Scenes and Movies** folder in the project tree, and selecting **Save Current Scene**.

2. Give the scene a description, and change the name of the scene. Note that scenes will be listed in the project tree in alphabetical order, so by naming them using a number followed by the descriptive name, they will be listed correctly (eg 1. Raw data, 2. Intrusional surfaces, 3. Other surfaces etc).

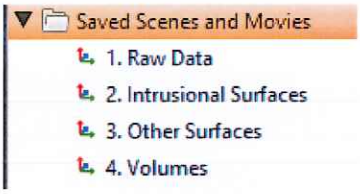


3. Click **OK**, and the scene will be saved.

4. You can now clear the scene, and when the saved scene is dragged back into view, the same settings including objects, slicer and transparencies are remembered.

A good habit to get into is to save scenes as you progress through a model, describing the steps that were taken to build it, and pointing out important or interesting features of the model.

5. Build another three or four scenes describing the steps taken to build the model.



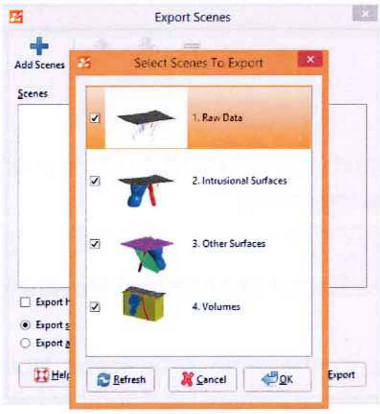
Using the Leapfrog Viewer

Once scenes have been saved in your Leapfrog project, you can export them and open them in the Leapfrog Viewer, which is free software available on the Leapfrog3d website. Leapfrog Viewer allows the user to rotate, zoom, slice, change the transparency and view the details (volume etc) of each object in the saved scene.

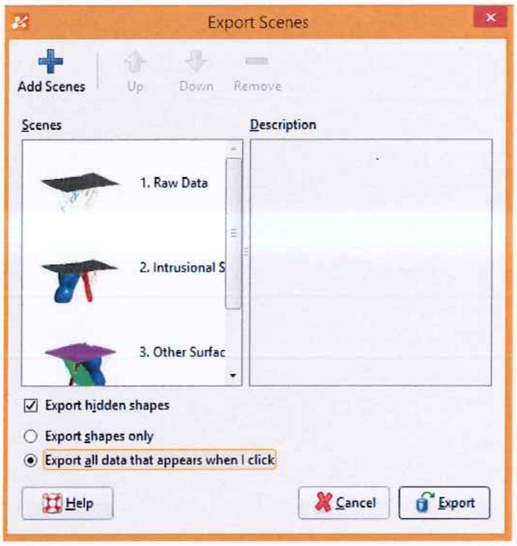
If you have Leapfrog Viewer installed on your computer, you can work through the following steps.

1. Right click on the **Saved Scenes and Movies** folder, and select export scenes.

2. Select the scenes which you would like to export by ticking the checkbox for each, then click **OK**.



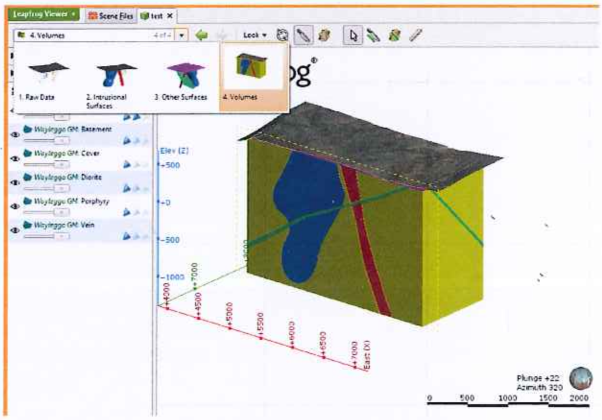
There are two options on the next window which can be selected to change the data which is exported. The first is the "export hidden shapes" checkbox. When selected, this will export any shapes which are in the scene but hidden at the time the scene was saved. The second is the "export shapes only" or "export all data that appears when I click" option. When the second option is selected, data that appears in the Leapfrog scene when an object is clicked on, such as the volume, area, elevation, and location, is exported. When the first option is selected, this data is not exported.



3. Choose the options you would like as above, and click **Export**.

4. Enter a file name, and click save.

5. Open the Leapfrog Viewer, and navigate to your exported saved scenes to open them.



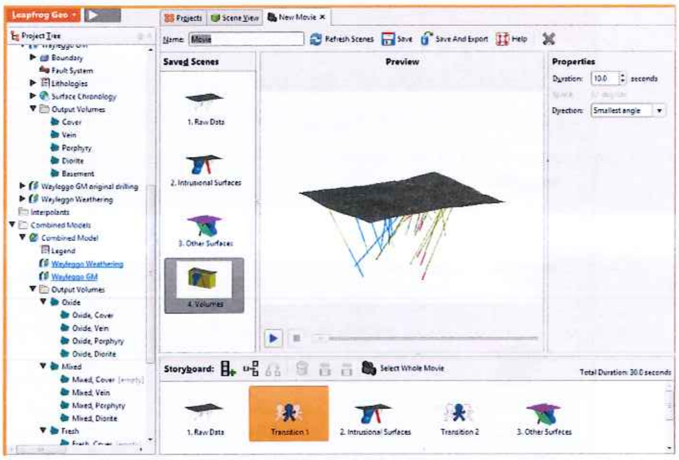
Creating movies

Movies are a great way of displaying your model at a conference or presentation. In Leapfrog, movies can be created by adding saved scenes to a storyboard, then editing the transitions between each scene.

When setting up the scenes to make the movie with, remember that one scene will rotate/fade into the next, so it is useful to imagine what the transitions will look like before saving your scenes.

1. Right click on the **Saved Scenes and Movies** folder in the project tree and select **New Movie**.

A new tab will appear next to the "projects" and "scene view" tabs at the top of the screen.



Along the left tree, the available saved scenes are listed. Double click on a saved scene to add it to the storyboard, which is at the bottom of the scene. When there is more than one saved scene in the storyboard, transitions will also appear in the storyboard.

To edit the properties of an item in the storyboard, click on it once, and properties will appear in the right window. Different properties are available depending on which item in the storyboard is selected.

Once the movie has been completed, save and export it in the top toolbar.

There are several options for export; the quality can be anywhere from HD (1920x1080) to lower quality, the number of frames per second can be changed, and whether or not to export frame images (for import to an external movie editing package) can be selected.

Rendering images

Any scene in Leapfrog can be rendered as either a png or jpg image. To access this feature, click on the Leapfrog Geo menu at the top left of the screen, and select **Render Image**. There are three main features to note here.

- The number of pixels along the x and y axis of the image can be changed to a maximum of 40,000 x 10,000. The image's proportions can be retained by ticking the "keep aspect" checkbox.
- By turning supersampling on, jagged edges caused by pixels will be smoothed. The highest option (4x4) will take longer to process than the lowest option (2x2) or keeping supersampling off.
- Overlays such as the scale bar and grid lines can be turned on or off.

After any change is made, the render button needs to be selected for the image to be updated.

Once the image has been rendered, it can be saved in both png and jpg formats. In general, png is better for images with few colours, like a geological model, and jpg is better for images with many colours, like aerial photographs.

leapfrog
FAST, DYNAMIC GEOLOGICAL MODELLING

Session 7: Combined Models

Contents

Goals

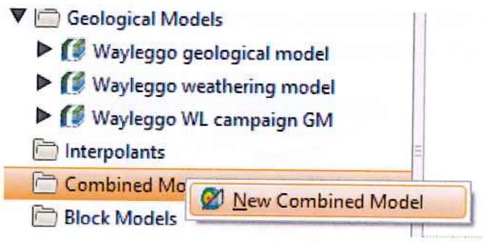
In this session, we will combine the weathering and geological models together to create a combined model.

For this session, open the Wayleggo project from the previous sessions.

Creating a New Combined Model

To create a new combined model:

1. Right click on **Combined Models** in the project tree and select **New Combined Model**

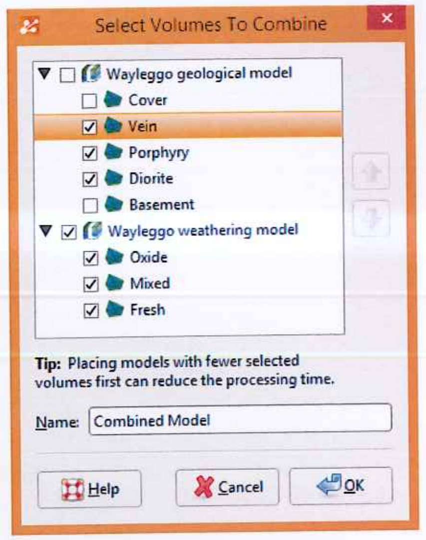


There will be three models available to combine; the weathering model, the full geological model, and the geological model using only the WL campaign.

2. Select the weathering model and the full geological model, then click **OK**.

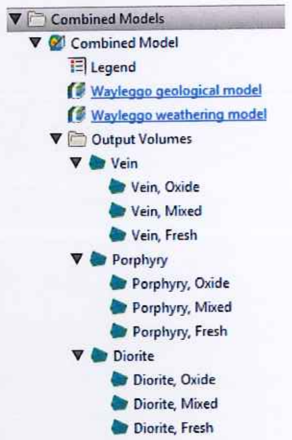
A window will appear with a set of check boxes relating to volumes in each model.

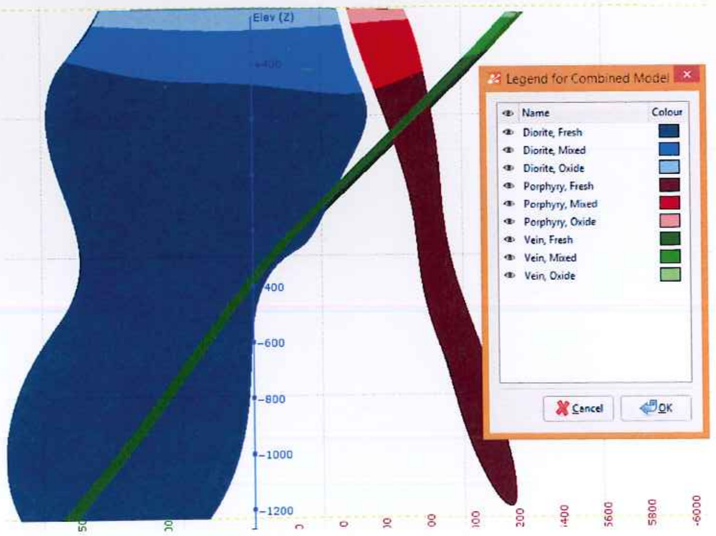
3. Select the porphyry, vein and diorite, as well as each of the weathering units, and click **OK**.



The combined model may take a few minutes to process depending on the processing speed of your computer. By looking in the output volumes of the combined model, you can see which volumes have been created and which are still processing.

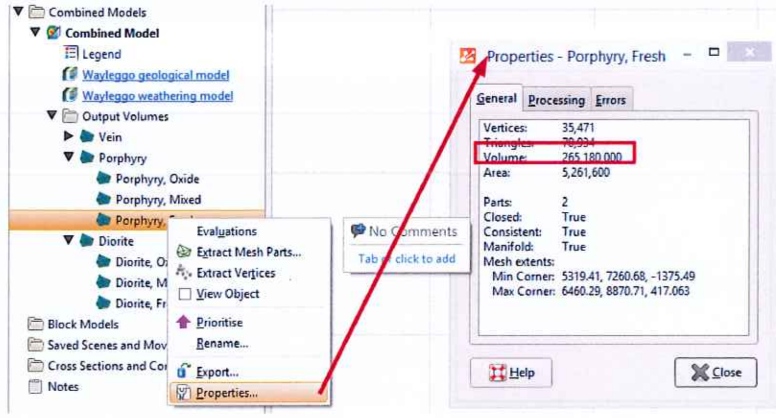
Once the combined model has finished processing, you can see that each geological volume has been split into three sub units, representing each weathering type.





Finding Volumes

To find the volume for a specific combination of volumes, right click on the item of interest, then click properties. The volume as well as a number of other properties are listed.



Session 8: Building from a map using GIS data

Contents

Goals

For this exercise, we will create a geological model using an imported map and topography.

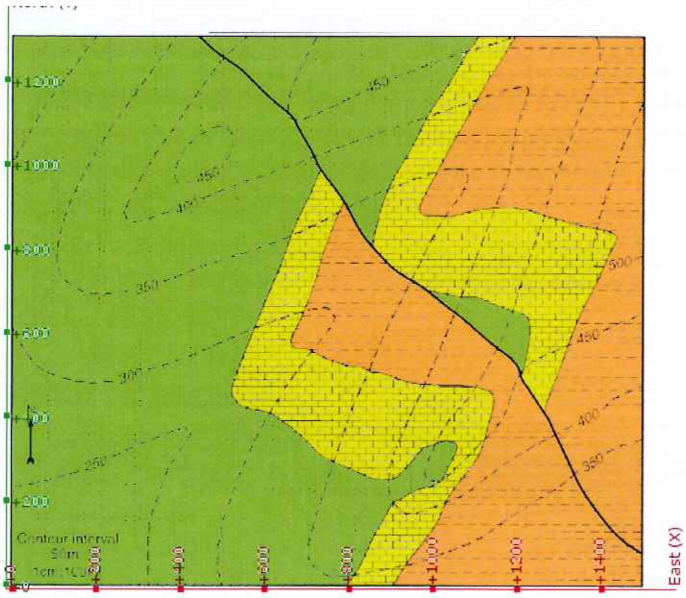
The data for this exercise can be found in the `Sessions \ Session 8 - building from a map using GIS lines` folder.

Setting Up the Project

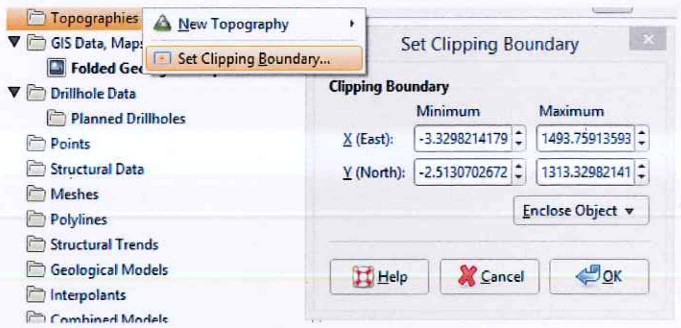
1. Create a new project called **Folded Geological Map**.

2. Import the map, which is called `Folded Geological Map.png`.

3. The map is already georeferenced, so click **OK** and drag the map into the scene to view it. Click **D** to look down.

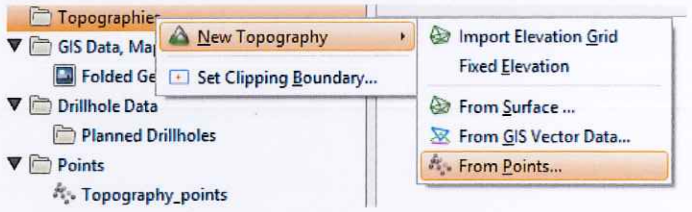


4. Use the controls in the scene to set the clipping boundary to the size of the imported map. Once the clipping boundary has been set, all surfaces which are created will be clipped by default to the clipping boundary.



5. Import the file `Topography_points.csv` to the pointsfolder.

6. Create a topography from the imported points:



7. View the topography surface in the scene, and add the geological map to the view.

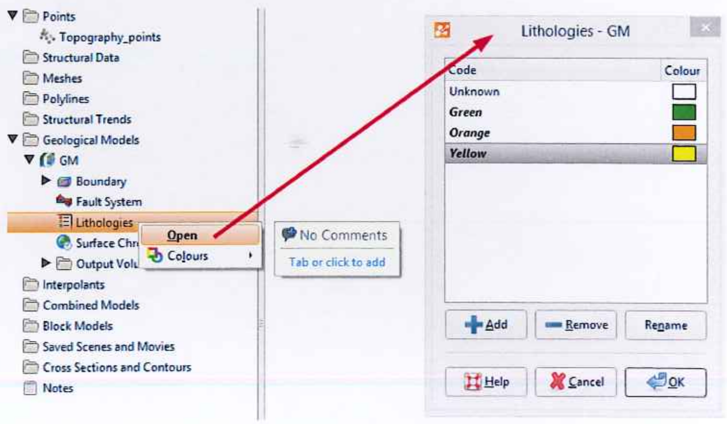
8. Make sure the topography is a reasonable resolution compared to the points.

Creating the Geological Model

1. Create a new geological model, and change the resolution to an appropriate number.

As we don't have drilling to define our lithologies, they need to be manually defined.

2. Double-click on the **Lithologies** object under the Geological Model and create the lithologies represented in the map, naming them Green, Yellow and Orange. Change the colours of the lithologies so they match their names:



Modelling the Contacts

The contacts will be created by drawing GIS lines representing each of the contacts.

1. Clear the scene, then add the topography to the scene with the map displayed.

2. Create a new GIS line and name it "Green Yellow contact"

3. Trace along the entire length of the green to yellow contact. You should end up with three segments; one on the West side of the fault, and two on the East. These will be dealt with later when we add the fault to the geological model:

4. Click Save, then repeat for the yellow to orange contact.

Now that we have defined the surface expressions of the contacts, we can let Leapfrog Geo determine the dip at surface of the contacts. As there is no additional dip information at depth, we will assume the surface dip is continued to depth.

5. Right click on the **Surface Chronology** folder in the geological model.

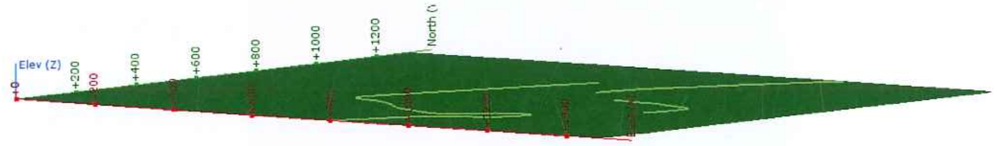
6. Select **New Deposit** > **From GIS Vector Data**

7. Select "Green Yellow contact (On Topography)" as the GIS vector data to use.

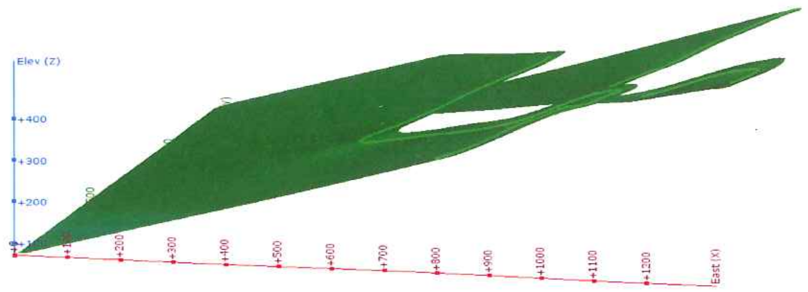
By using the 'On Topography' version of the GIS line, Leapfrog is able to drape the GIS line onto the topography and calculate the dip of the surface. If we didn't use the 'On Topography' version, the surface would firstly be empty, because the GIS line is above the topography which makes it outside the boundary of the geological model. If we increased the boundary size so the GIS line was inside the boundary, the surface would be completely flat, as the GIS line was drawn on a single elevation.

To describe this differently, if you imagine the GIS line as a densely spaced series of points, the 'surface of best fit' for the GIS line is flat (as all the points are on the same elevation).

The image below shows this flat surface:



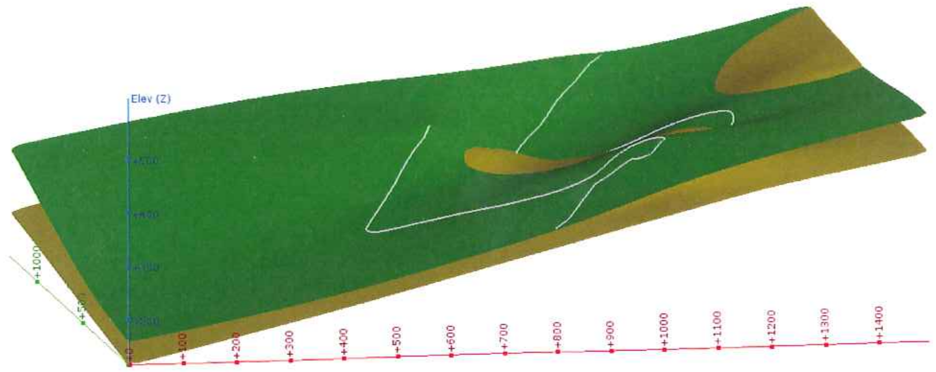
Now if we look at the 'On Topography' version of the GIS line, we can see that as the 'points' of the GIS line are no longer on a single elevation, rather they are draped on the topography, instead of the surface being flat it has a dip. This is shown in the image below:



8. Change the First Lithology to Green, and the Second Lithology to Yellow, then click **OK**.

9. The surface that is created extends through the fault, and links all three segments of the GIS line. We will split this surface so it represents each side of the fault at a later stage in the exercise.

10. Repeat the above steps for the yellow to orange contact.

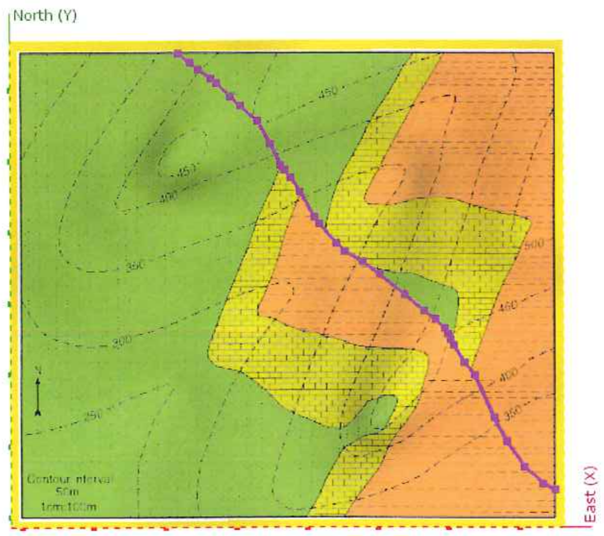


Creating the Fault

We will create the fault by drawing a GIS line, and let the intersection of the GIS line and the topography calculate the dip at surface. Once the fault has been created, we will activate it to split the two surfaces created above into four (two on each side of the fault).

1. Create a new GIS line and name it "Fault".

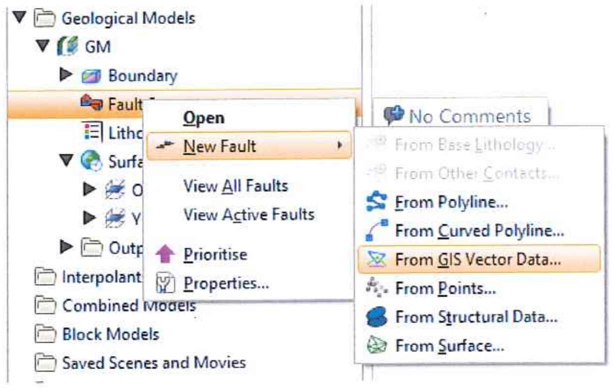
2. Trace the fault in the scene:



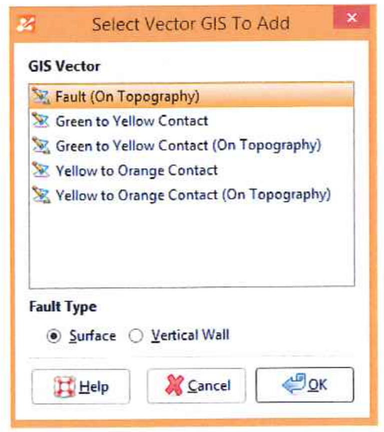
3. Right-click to end the drawing of the fault, then click the **Save** button.

Adding the Fault to the Geological Model

1. In the geological model, create a fault from the GIS line:



2. Select "Fault (On Topography)":



3. Enable the fault in the geological model by double-clicking on the **Fault System** and ticking the box.

4. Clear the scene.

5. Add GM unit 1 to the scene.

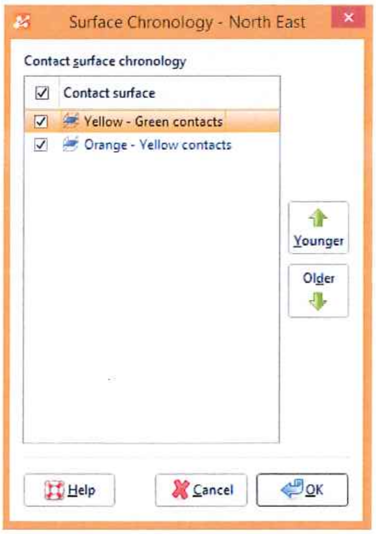
6. Rename GM unit 1 and 2 to "East" and "West".

Enabling the surfaces to create output volumes

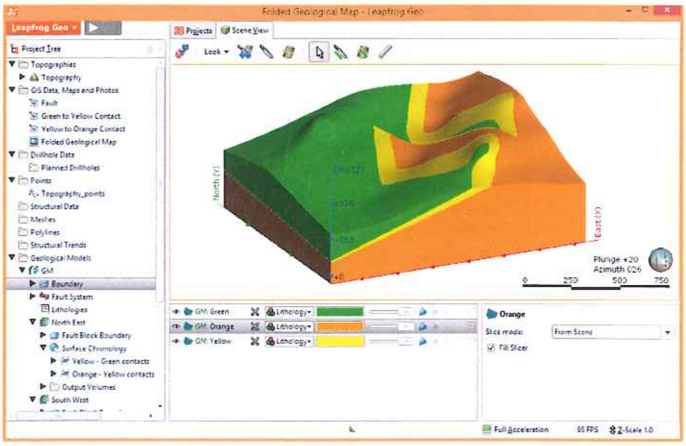
Now that surfaces have been created on each side of the fault, representing all surface contacts, the surfaces can be activated in the **Surface Chronology** folder.

1. Double click on the **Surface Chronology** folder for the East fault block.

2. Activate the surfaces by selecting the checkbox for each contact, and ensure the contacts are in the correct chronological order.



3. Activate the surfaces for the West fault block, then view the completed model in the scene:



Session 9: Building from a Map Using Structural Data

Contents

Goals

In this session, we will build a geological model from a map, structural data and GIS lines.

The data for this session is available from the `Sessions \ Session 9 - building from a map using structural data` folder.

Creating a New Project

To start, we are going to create a new project.

1. Click on the **Projects** tab.
2. Click the **New Project** button.
3. Enter the name "Sagean Valley" for the project and click **OK**.

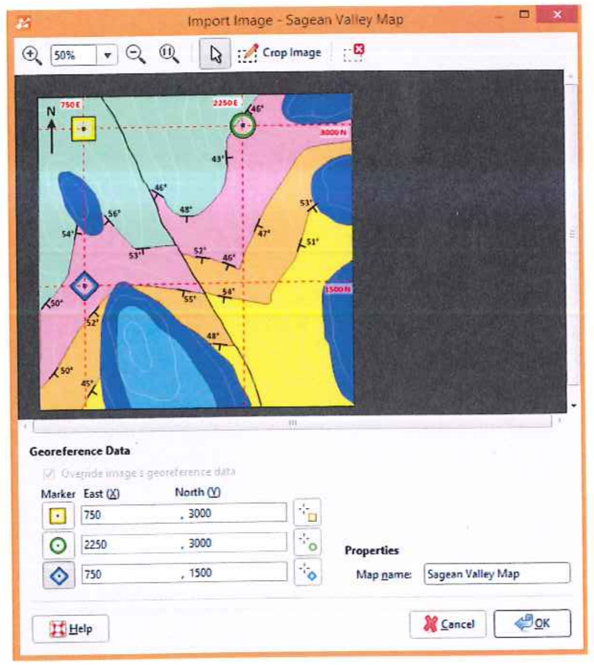
Importing a Non Georeferenced Map

For this project, we will import a non georeferenced map.

1. Right-click on the **GIS Data, Maps and Photos** folder and select **Import Map**.
2. Browse to the folder for this session and select the `Sagean Valley Map.png` file.
3. Click **Open**.

The **Import Image** window will appear. In this window, you enter georeferencing information using three points of reference. These reference points are positioned in the image and then assigned their real-world coordinates.

4. Click on the yellow marker and position it in the scene, as shown below:



5. Enter the coordinates for the yellow marker, read from the map.

6. Repeat the process for the green and blue markers.

7. Click **OK**.

8. Add the image to the scene and press the D key to view it from above.

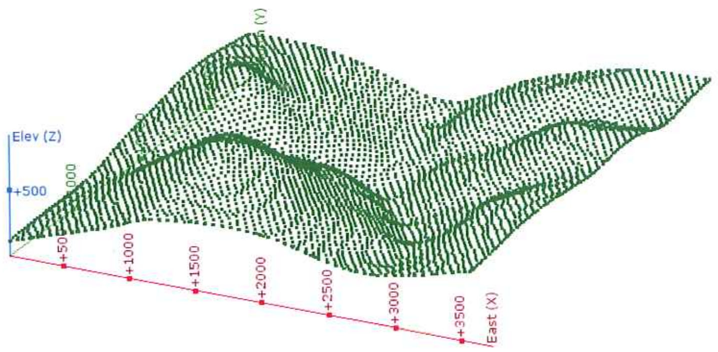
If the image appears skewed in the scene, you may have entered one or more of the co-ordinates incorrectly. To edit the georeference markers, double-click on the image in the project tree.

Creating the Topography from Points

1. Right click on the **Points** folder and select **Import Points**.

2. Select the file `Sagean_Valley_Topography.csv` and click **Open**.

3. Click **Finish**.



4. Right-click on the **Topographies** folder and select **New Topography > From Points**.

5. Select the topography points and click **OK**.

6. Click **OK** to accept the default name for the topography.

7. Make sure the resolution of the topography is appropriate when compared to the topography point spacing.

Digitising the Fault

By drawing a GIS line then draping it on the topography, Leapfrog can automatically calculate the dip of the fault at the surface, based on the topography.

1. Click **D** to look down, and zoom in so you have a clear view of the entire fault.

2. Right click on the **GIS Data, Maps and Photos** folder and select **New GIS Lines**.Enter the name "Fault", and click **OK**.

3. A new toolbar will appear at the top of the screen, with icons to let you draw a GIS line.

4. Hold the mouse cursor over each button to check its functionality.

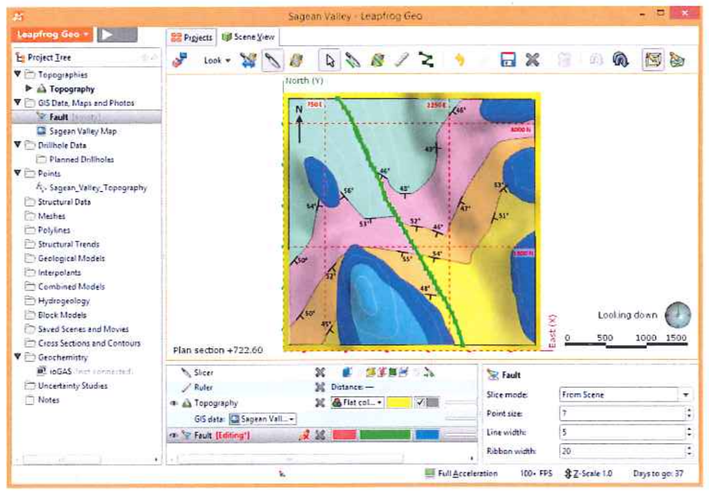
5. Click the **Draw Lines** icon. Note that once this has been selected, the cursor icon changes. To revert back to the standard cursor, either click the cursor icon in the top toolbar, or click the right mouse button anywhere in the scene. For a shortcut to accessing the cursor while in drawing mode, you can hold the **Shift** button, which lets you move the scene around using the normal controls. When the Shift button is released, the drawing icon will return.

6. To draw the fault, click and release the cursor at one end of the fault to add a point. Move along the fault, digitising from one end to the other by clicking and releasing.

7. Once the fault has been digitised, the drawing can be finished by right clicking.

8. There are a number of different edits which can be applied to the completed GIS line. To delete a single point, click on it, then click the **Delete Selected Items** icon on the top toolbar. To delete the entire line, double click anywhere on it, and click the **Delete Selected Items** icon. To add a point to the line, hold the **Ctrl**button on the keyboard and hover the cursor over the line. The cursor icon will change, which means you can click and drag to add a point.

9. Once you are happy with the GIS line, click the **Save** icon in the top toolbar



Importing Structural Data for the Contact Surfaces

The structural data for the contact surfaces will be imported from a series of files. You could, however, create a table for each contact surface using the information on the map.

1. Clear the scene.

2. Right-click on the **Structural Data** folder and select **Import Structural Data**.

3. Navigate to the folder for this session.

4. Select the file `Green Pink Contact.csv`.

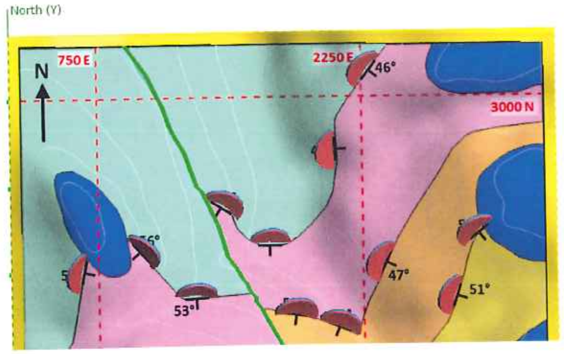
5. Click **Open**, then **Finish**.

The new structural data table will appear in the project tree.

6. Add the topography to the scene.

7. Press the `D` key to view the scene from above.

8. Add the Green Pink Contact.csv structural data table to the scene:



9. Import the files `Pink Orange Contact` and `Orange Yellow Contact` and add them to the scene.

Creating the Geological Model for the Western Side

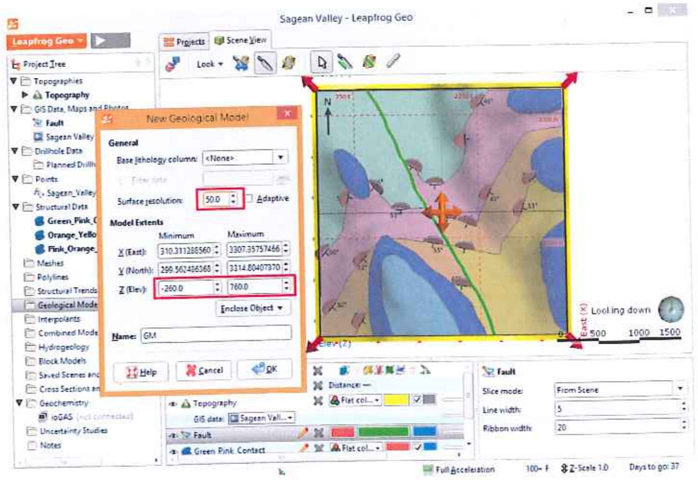
At this point, we have the data necessary to build the depositional layers for the geological model, but we haven't yet imported the GIS lines representing the two blue erosional surfaces. We will do this later, after the geological model has been built and other surfaces modelled.

1. Right-click on the **Geological Models** folder in the project tree and select **New Geological Model**.

2. Use the controls in the scene to set the model extents to enclose the map.

3. In the **New Geological Model** window, set the **Z (Elev)** values to -260 and 760.

4. Set the **Surface Resolution** to 50.



5. Click **OK**.

Adding the Fault to the Geological Model

We will now add the fault to the Geological model to split it into an East block and a West block, so we can model each separately

1. Clear the scene.

2. Drag the topography with surface map into the scene.

3. Press the `D` key to view the scene from above.

4. Right click on the **Fault System** for the new geological model and select **New Fault > From GIS Vector Data**.

When the **Fault Type** is set to **Surface**, the drawn GIS line will appear in the list twice. The Fault (On Topography) option automatically drapes the drawn GIS line on the topography.This is the step which allows the dip of the fault at the surface to be calculated automatically, based on how the drawn GIS line intersects with the topography. See the previous session for further detail on this.

5. Select the Fault (On Topography) object and click **OK**.

6. Expand the **Fault System** in the project tree to see the new fault.

Defining the Lithologies

As we have no drillhole data in the project, we need to define the lithologies for the geological model manually.

1. Double-click on the **Lithologies** object for the geological model.

2. In the window that appears, click the **Add** button and enter a name for the first lithology, Green. Change the colour of the lithology to green. Repeat for the other lithologies.

3. Click **OK** to close the **Lithologies** window.

Defining the Sedimentary Sequence

We will define the sedimentary sequence by building from oldest (Green) to youngest (Yellow), then we will define the two erosional surfaces.

1. Clear the scene.

2. Right-click on the **Surface Chronology** object and select **New Deposit > From Structural Data**.

3. In the window that appears, change the first lithology to Pink (youngest) and the second lithology to green (oldest).

4. Click **OK**, and select the **Existing Structural Data** check box. Select Green_Pink_Contact structural data, and click **OK**.

5. Repeat the process for the Pink_Orange_Contact and Orange_Yellow_Contact structural data tables.

6. Add the topography to the scene.

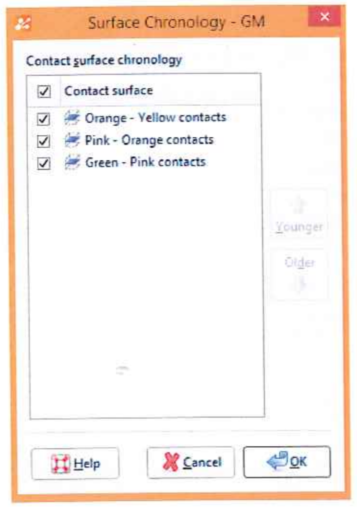
7. Add the new contact surfaces to the scene to confirm that they are correctly oriented.

One of the more difficult tasks in building geological models is identifying the sides of contact surfaces, especially in a complex environment where the geology may be overturned. Leapfrog Geo labels surfaces as having older and younger sides, which is the "younging" direction.

8. Double-click on the **Surface Chronology**.

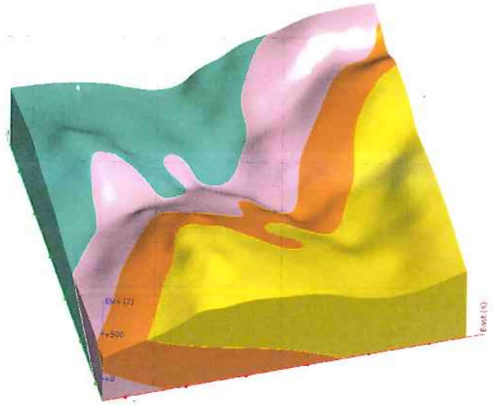
9. Make sure the contact surfaces are in the correct chronological order.

10. Enable the contact surfaces by ticking them



11. Click **OK**.

12. Clear the scene and add the unfaulted geological model to the scene:



Activating the Fault in the Model

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

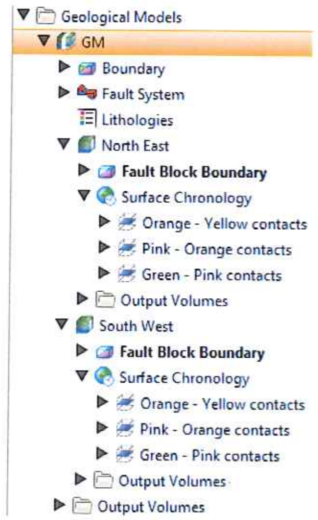
2. Tick the box to activate the fault, and click **OK**.

3. Add the geological model to the scene

The geological model has been split into two blocks, labelled "GM fault block 1" and GM fault block 2".

It's a good idea to rename the fault blocks so they are more easily recognisable.

4. Right click on each of the fault blocks, and rename them "North East" and "South West".



Adding an Erosion Surface from a GIS Line

Next, we will add two erosions to the model, lithologies Dark Blue and Blue. They will both be built from imported GIS lines.

1. Right-click on the **GIS Data, Maps and Photos** folder and select **Import Vector Data**.

2. Navigate to the folder for this session and select the files `Dark_Blue_Outline.shp` and `Light_Blue_Outline_west.shp`.

3. Click **Open**.

4. Deselect the **Filter Data** checkbox, which ensures that all data will be imported to the project without being clipped to a boundary.

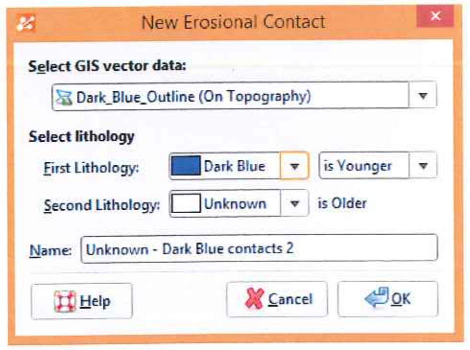
5. Click **OK**.

6. Right-click on the **Surface Chronology** for the South West side of the geological model and select **New Erosion > From GIS Vector Data**.

7. Select Dark_Blue_Outline (On Topography) for the **GIS Vector Data**.

8. Select Dark Blue for the **First Lithology**.

9. Because the older side of the unconformity has more than one lithology contacting it, leave the **Second Lithology** as Unknown:



10. Click **OK**.

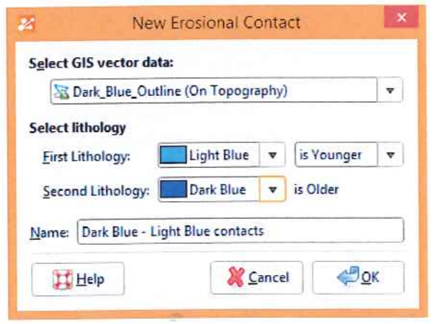
11. Right-click on the **Surface Chronology** for the South West side of the geological model and select **New Erosion > From GIS Vector Data**.

12. Select Light_Blue_Outline(On Topography) for the **GIS Vector Data**.

13. Select Light Blue for the **First Lithology**

14. Because this side of the unconformity has only one lithology contacting it, we can change the **Second Lithology** to Dark Blue.

15. Click **OK**.



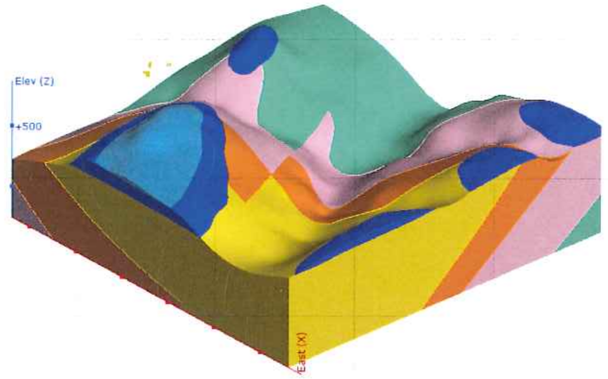
16. Activate the surfaces for both the Dark Blue and Light Blue lithologies under the **Surface Chronology** object, making sure the surfaces are in the correct chronological order.

17. Click **OK**.

18. Clear the scene and add the model to the scene:

19. Use the imported GIS line to add an erosion for Dark Blue.on the Eastern side of the fault.

20. View the model in the scene:



Session 10: Stratigraphic sequence tool

Contents

Goals

In this session we will import drilling through an ODBC link, then create a faulted geological model and use stratigraphic sequences to model the layers in each fault block.

The data for this exercise can be found in the `Sessions \ Session 10 - stratigraphic sequence tool` folder.

Setting Up the Project and importing data via ODBC

1. Create a new project called "Stratigraphic Sequence".

We will import drillholes using an ODBC link, but if your computer isn't running 64-bit Office you may need to import directly from a CSV file, which is available in the folder for this session.

2. Right click on the **drillhole data** folder and select import drillholes via ODBC.

You can either link directly to an ODBC data source using a username and password, or browse to a database file.

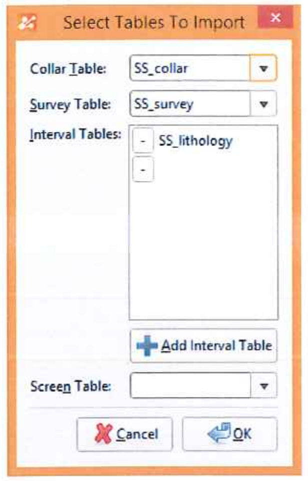
3. In this case, we will browse to a database file which is located in the folder for this session.

4. Click **OK** once the database has been selected.

5. A 'testing connection' window will appear for a few seconds.

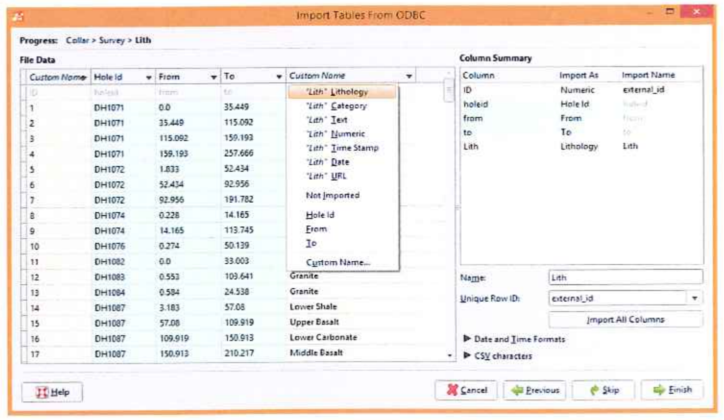
The database should contain a collar file, survey file as well as interval tables.

The files have been correctly selected by Leapfrog using the file names. If the default selections are incorrect, click the dropdown arrows for each to change the selection.

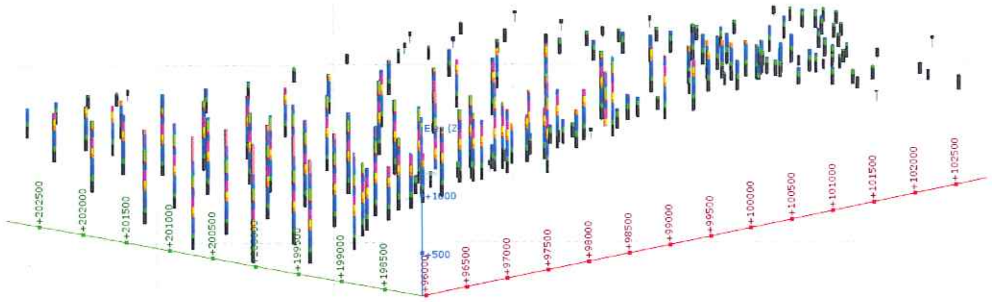


6. Click **OK**.

7. The **import tables from ODBC** window will appear. In this case, all columns are selected correctly by default, but it is possible to change each column using the dropdown arrows.



8. View the drillholes in the scene.



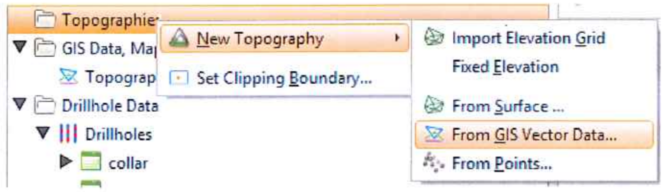
Creating the topography and faults

Next, we will create the topography from surface contours.

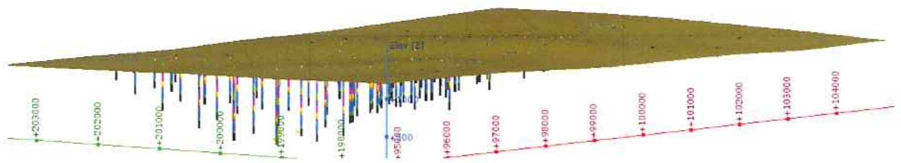
1. Right click on the **GIS data, maps and photos** folder and select **import vector data**

leapfrog
FAST, DYNAMIC GEOLOGICAL MODELLING

2. Select the `Topography contours.shp` file.
3. Click **OK**.
4. Right click on the topographies folder and select **new topography** > **from GIS vector data**



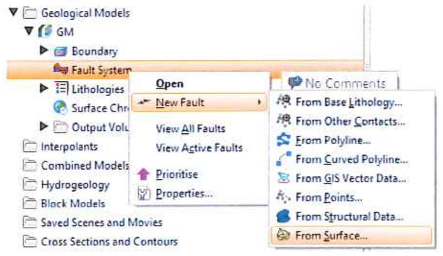
5. Click **OK**, and accept the default name for the topography by clicking **OK** again.
6. Make sure the resolution of the topography is appropriate compared to the data.
7. Add the topography to the scene:



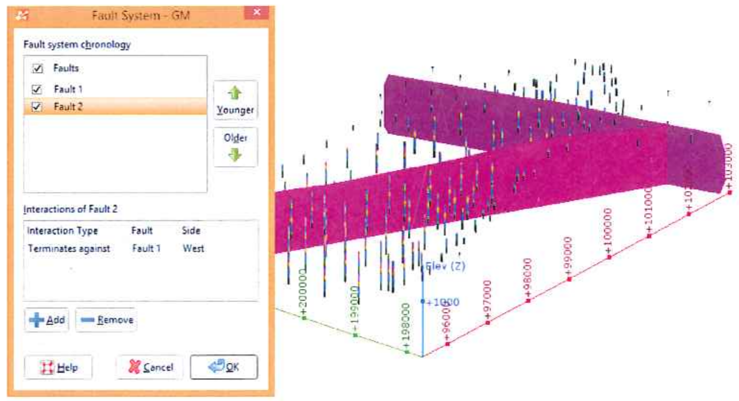
8. Use the **Meshes** folder to import the two fault meshes, labelled `fault 1` and `fault 2`.

Creating the Geological Model and Adding the Faults

1. Create a new geological model, enclosing the Lith table.
2. Drop the resolution down to 200 to provide more definition between drillholes
3. Use the imported meshes to define the faults:



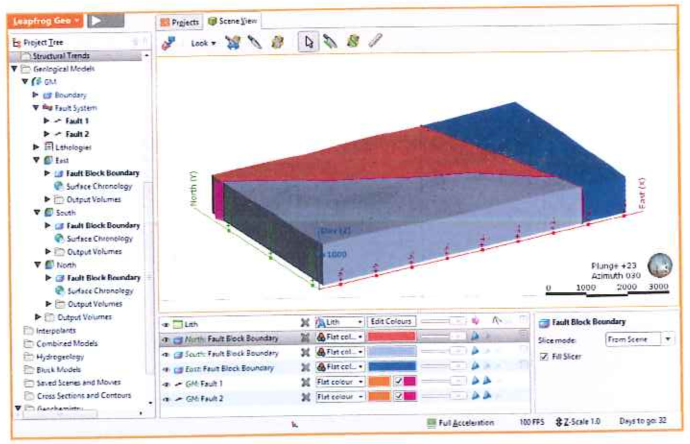
4. Add the **Fault System** object to the scene to view the faults.
5. Double-click on the **Fault System** object to arrange the chronology and set the fault interactions:



6. Activate the faults by ticking the boxes and clicking **OK**.

This will create a series of fault blocks, which will appear in the project tree under the geological model.

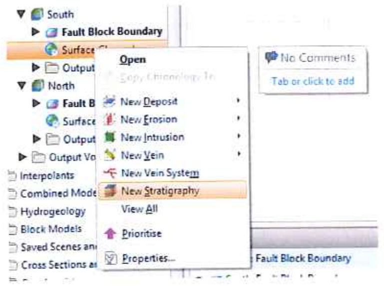
7. Rename the fault blocks to East, North and South.



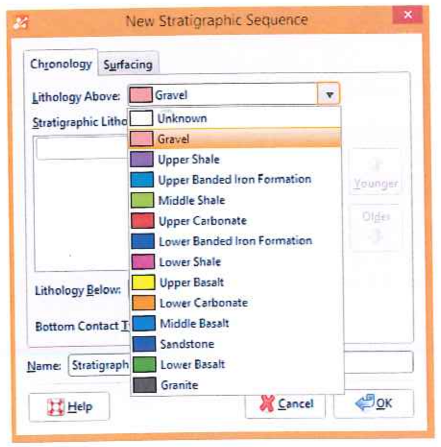
Defining the Stratigraphic Sequence

For this model, the oldest lithology is Granite and the youngest is Gravel. We will use this for the lithology above and lithology below in the stratigraphic sequence.

1. Create a **new stratigraphy** for the South unit.



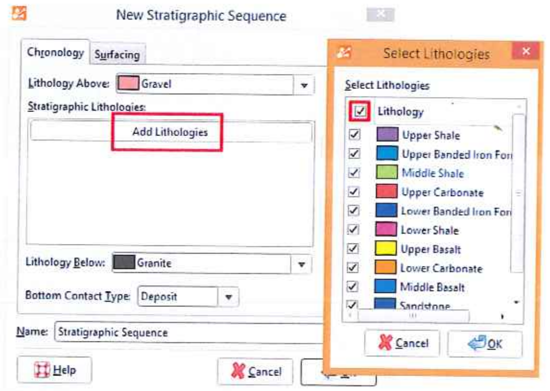
2. Select Gravel for **Lithology Above**:



3. Select Granite for **Lithology Below**.

leapfrog
FAST, DYNAMIC GEOLOGICAL MODELLING

4. Add all other lithologies to the **Stratigraphic Lithologies** list:



5. Click **OK**.

6. Enable the stratigraphic sequence in the **Surface Chronology** folder by double clicking on it, and ticking the checkbox.

7. Clear the scene.

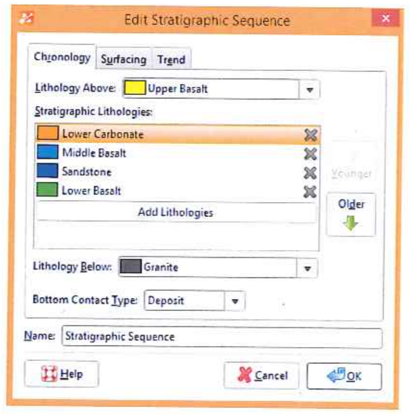
8. Once the stratigraphic sequence has finished processing, add it to the scene.

9. Define the stratigraphic sequences for the remaining fault blocks (north and east). The easiest method of achieving this is to right click on the **surface chronology** folder under the south fault block, then click **copy chronology to**, and selecting the other two units.

When these units have finished running, you will notice that the east fault block has an error. Right click on the stratigraphic sequence and select **list errors**. You will see seven errors listed, each saying that a particular unit is empty. This means there is no data in the east fault block for these seven units.

1. Click **OK** on the errors window

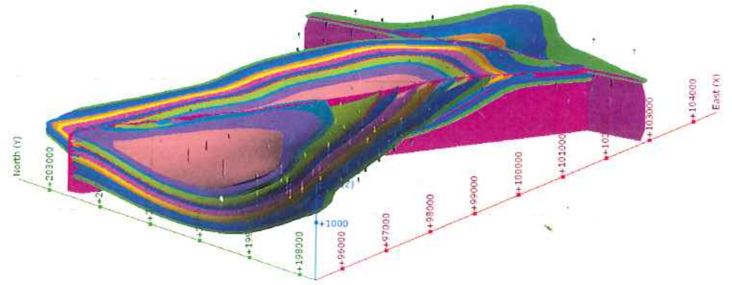
2. Double click on the stratigraphic sequence for the east fault block, and change the **lithology above** and stratigraphic lithologies to remove all units above the upper basalt (which will be the **lithology above**).



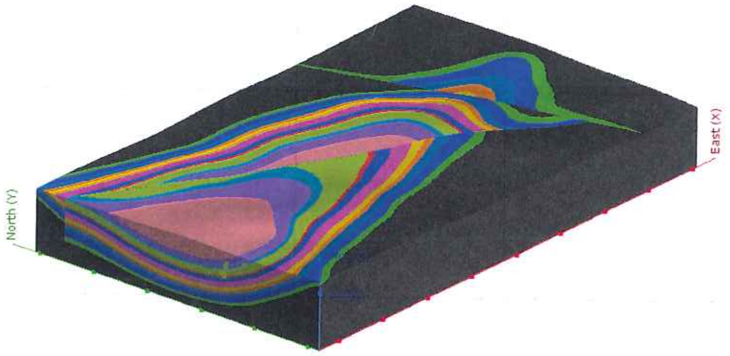
3. Click OK and let the surfaces re-run, then drag each set of surfaces into the scene.

leapfrog

FAST, DYNAMIC GEOLOGICAL MODELLING



4. Double click on the surface chronology under each fault block, and activate each stratigraphic sequence to turn the surfaces into volumes. Drag the geological model into the scene to view it.



leapfrog
FAST, DYNAMIC GEOLOGICAL MODELLING

Session 11: Wolf Pass Introduction

Contents

Goals

For this series of sessions, we will import drillholes and topography, then build a complete set of models and outputs. By the end of these sessions you will know how to:

- Import drillholes and topography points
- Manipulate drillhole data using groupings and interval selection, as well as fix errors
- Model veins
- Build an interpolant and an indicator interpolant
- Create and export cross sections
- Create block models
- Plan drillholes

The data for this exercise can be found in the `Sessions\ Session 10 - Wolf Pass` folder.

Setting up the project

1. Create a new project and call it "Wolf Pass".

We will be importing a drilling dataset which contains lithology and grade, as well as a LIDAR point set representing the topography. The lithology contains 15 different codes which we will group into 5 major codes:

Recent: SAPR, COLLV and ASH

Dacite: DA

Early Diorite: E1, E2, E3, EBX1 and EBX2

Intermineral Diorite: I1, I2 and IBX

Basement: H and SBX

Note that the leftover code (SGNCRLSS) represents significant core loss, so will be left ungrouped for this model.

The Early Diorite intrusion is the oldest intrusion, which was emplaced into the schist basement and contains the highest gold and copper grades. This was followed by the Intermineral Diorite intrusion which also contains gold and copper, then the barren Dacite dykes cut through all three existing lithologies. Weathering and a nearby volcanic eruption formed the recent layer, which is the youngest lithology shown in the logging.

leapfrog

FAST, DYNAMIC GEOLOGICAL MODELLING

Session 12: Wolfpass Importing and manipulating data

Contents

Goals

In this session we will import topography points and use the triangulated mesh feature to create a surface. There are 2D grids representing slope, aspect and elevation. We will then import the drillholes and manipulate them to the point at which we are able to successfully model them.

Creating the Topography using a Triangulated Mesh

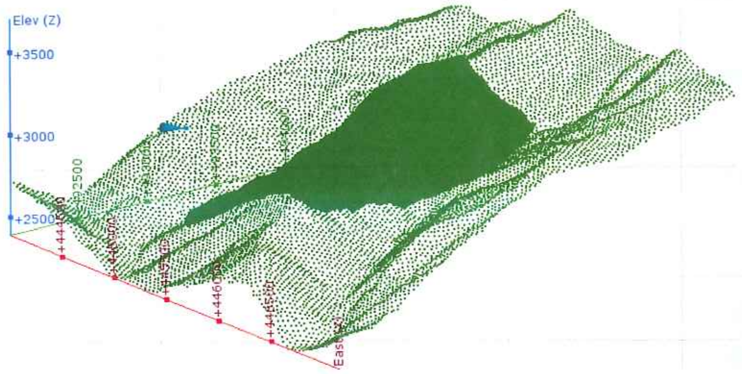
1. Open the Wolfpass project.

In the Wayleggo project which was used at the beginning of the course, we covered the steps involved in building a topography from points in the topography folder. This created a good topographic surface for widely and irregularly spaced data. For this project, there is a very detailed LIDAR dataset which has been merged with a less detailed regional topography dataset, so we will look at a different method of dealing with this densely spaced data.

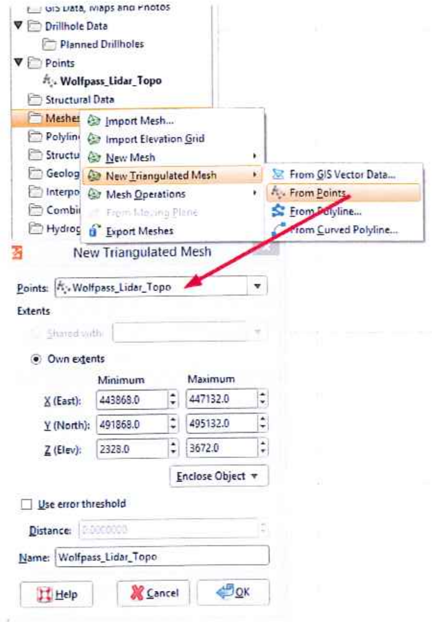
If we were to use the surfacer in the topographies folder to build the topography directly from the points, it would be a very time consuming process. To deal with densely spaced data, a different triangulation algorithm is available in the meshes folder. This works in a similar manner to a standard triangulation by joining the points together with triangles, but also has the ability to apply an error threshold which significantly speeds up the process, while still building a surface which is representative of the quality of the data.

2. Import the `Wolfpass_Lidar_Topo` points into the **points** folder.

3. There are close to 700,000 points so the they may take a minute or so to be imported. If you are finding the import too slow, there is also a lower density data set in the folder for this exercise.



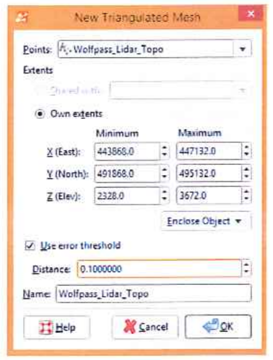
4. Right click on the meshes folder and select **new triangulated mesh** > **from points**

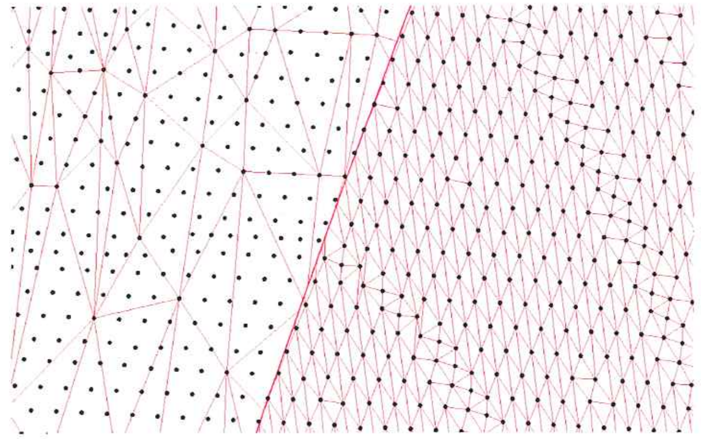


Setting the Error Threshold

The error threshold is a useful tool which can deal with very densely spaced data in a more sensible manner than just sampling the dataset. The error threshold works by excluding points which don't have a vertical offset greater than the threshold. If the threshold is set to 0.5, and the difference between one point and its neighbour is less than 0.5 m, this point will be excluded and the triangle will be fitted to the next point which is above the threshold. There will always be a trade off between the accuracy of the data and the processing time for the surface.

5. The point spacing for this data set is around 2 m around the main project area. Set the error threshold to 0.1, click **OK** and let the surface process. Again, this may take a few minutes as there are close to 700,000 data points.





In the image above, the right side is the triangulation without an error threshold (ie it creates a triangle between all points), and the left side is the triangulation with an error threshold of 0.1 (ie any point shown which is NOT at the vertex of a triangle is less than 0.1 m vertically from the surface).

6. Check the surface against the points to see whether the error threshold applied is producing a reasonable surface.

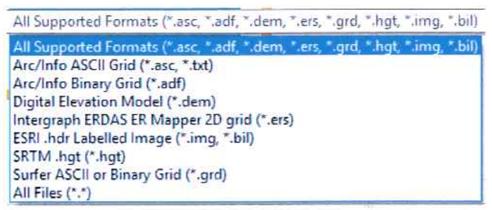
Now that the surface has been created, it needs to be added as a topography under the topographies folder.

7. Right click on the **topographies** folder and select **new topography** > **from surface**, then select the mesh and click **OK**.

8. Accept the default name and click **OK** again.

Importing 2D grids

Leapfrog can import 2D grids in a number of different formats. Once imported, these can be set to a specific elevation. In this case, we will import 2D grids representing the aspect, slope and elevation of the topography surface, then set the elevations of each to the topography.



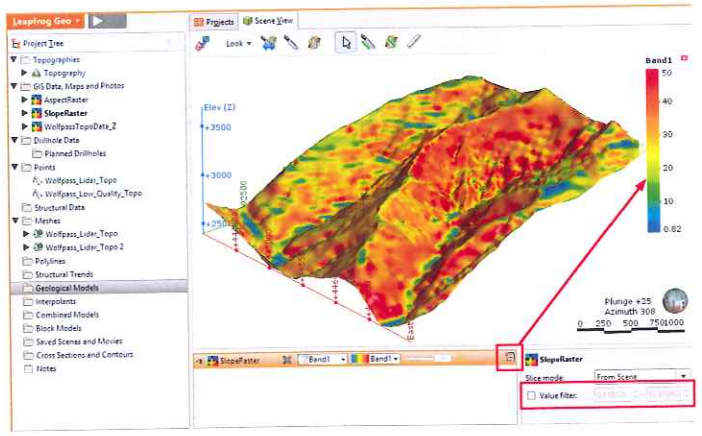
1. Right click on the **GIS Data, Maps and Photos** folder and select **Import 2D grid**.

2. Navigate to the folder containing the 2D grids, and import them one at a time, noting that the grids are georeferenced in this example, but is possible to override or add georeferencing within Leapfrog.

3. View the grids in the scene. They are flat grids at an elevation of 0 m.

4. To set the elevation of the grids to that of the topography, right click on one of the grids and select **Set Elevation**, then select the **From Surface** checkbox, and make sure "Topography" is selected.

5. Click **OK**, and repeat for the other two grids.



In the image above, the grid representing the slope of the surface is being displayed alongside the legend. We can see that the slope varies between 0.82° and 50°. These values can be queried using the value filter in the shape properties panel.

Importing GIS data representing the dykes

There are GIS lines representing the hangingwall and footwall of dyke 1 on the topography.

1. Right click on the **GIS Data, Maps and Photos** folder in the project tree, and select **Import Vector Data**. Navigate to the GIS Data in the folder for this project, and select both the hangingwall and footwall.

2. Click **Open** to import the files to Leapfrog.

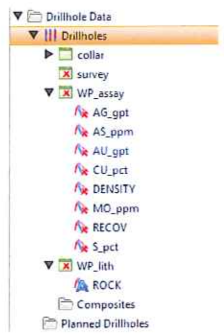
3. In this case we don't want to filter the data to within a certain distance from a bounding box, so untick the 'Filter Data' checkbox.

4. View the GIS lines in the scene.

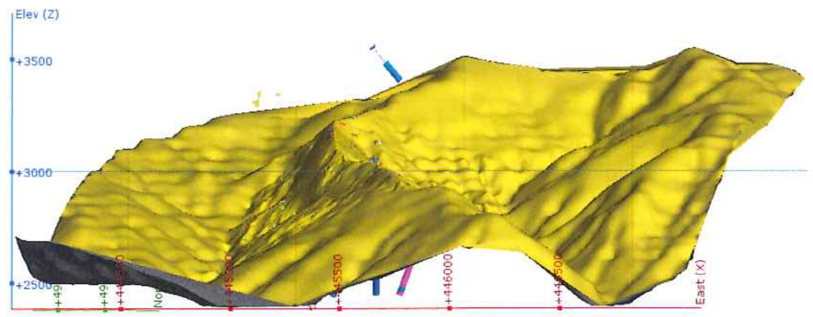
Importing the drillholes and fixing errors

When data is imported into Leapfrog, it runs a validation to check whether there are any inconsistencies in the data. Errors will then be flagged, and can be fixed in Leapfrog with a number of tools, or exported to be fixed in the database.

1. Import the drillholes for the wolfpass project. Make sure all the numeric data is imported as numeric data, and the WP_lith column is imported as lithology data.



You will notice that Leapfrog has flagged a number of errors with the data set. By dragging the drilling into the scene you will immediately notice that one of the drillholes has a collar above the topography.



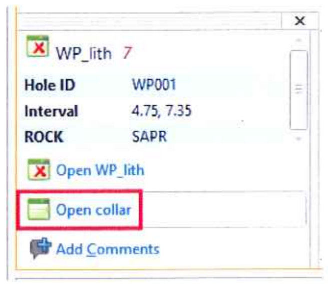
To fix this there are two options:

- We can open the collar table in Leapfrog and manually change the height
- We can project the collar onto the topography.

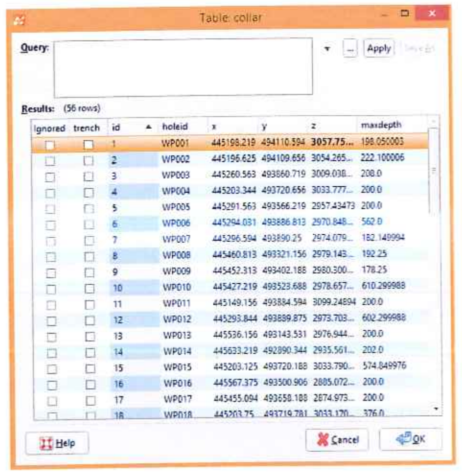
The correct height for this collar is 3057.75. Both methods are outlined below.

Manually changing the height

1. In the scene, click on the drillhole which is floating above the topography, then click open collar.



2. The collar table is opened, with the correct collar highlighted. The height can be changed in the table which is opened.



The text which has been edited remains bold for future reference.

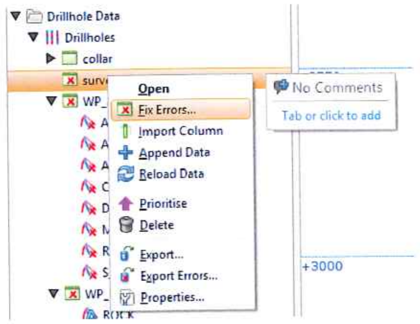
Setting the elevation from topography

1. If you have a topography surface which is more accurate than the survey elevation for the collars, you can project the collars onto topography by right clicking on the drillholes object and selecting **project collars onto topography**. This will project all collars vertically either up or down onto the topography surface.

Fixing Survey Errors

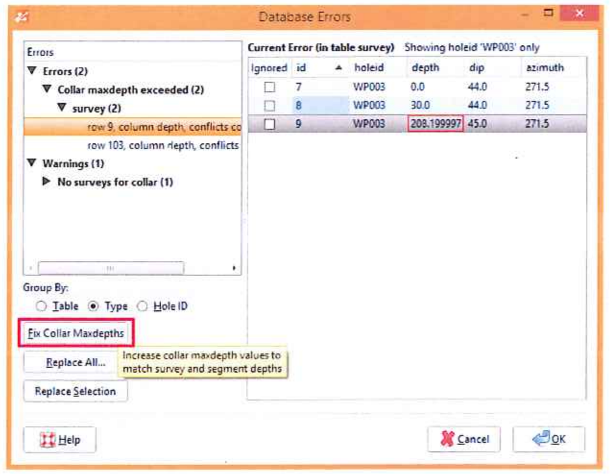
Now that drillholes are in the correct location in space, we will fix the survey data errors.

1. Right click on the survey table and select **fix errors**.

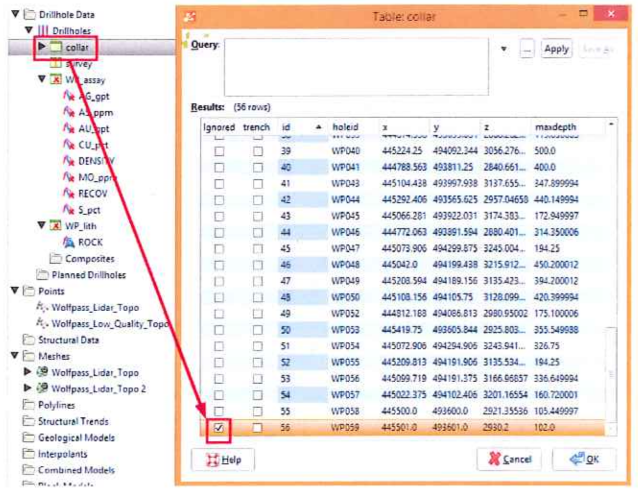


Two errors and one warning have been flagged. The two errors are both "collar maxdepth exceeded" errors, which means the depth listed in the survey is greater than that listed in the collar table. The warning is telling us that there is no trace or downhole data for the listed collar.

2. To fix the "collar maxdepth exceeded" error, click the **fix collar maxdepths** button to the bottom left of the window, which will adjust the max depth listed in the collar table to match that of the survey table.



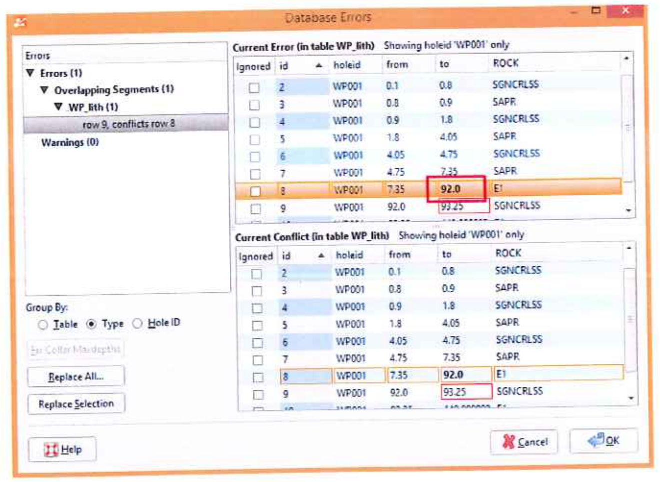
3. To fix the warning, we need to locate the collar in the collar table. Click **OK** for the database errors window and let the table reprocess to remove the collar maxdepth errors, then locate collar WP059 in the collar table.



Errors in the Lithology Table

From the initial validation completed by Leapfrog, one interval has been flagged which has an overlapping segment. To fix this, we can manually edit the from and to values as required. In this case, by editing the 'to' value of the row labelled id 8, we will be able to remove the error.

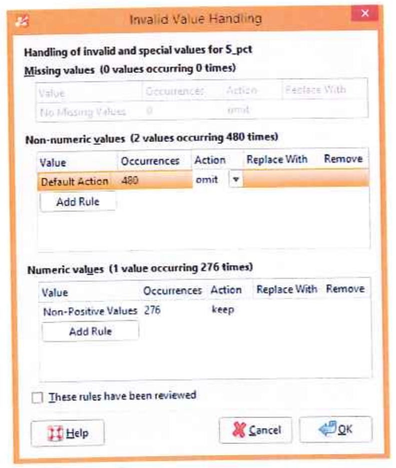
1. Change this value to 92.0, instead of 92.550003. The overlapping intervals have now been fixed. As was the case earlier, the edited value remains bold in the table for future reference.



Errors in the Numeric Data

Leapfrog flags intervals which are missing, contain non numeric values, or contain non positive values. For each item we have the option to either replace, omit, or keep the result. We will start by looking at the errors in the Sulphur table.

1. Double click on S under the assay table. The **invalid value handling** window will appear for Sulphur.

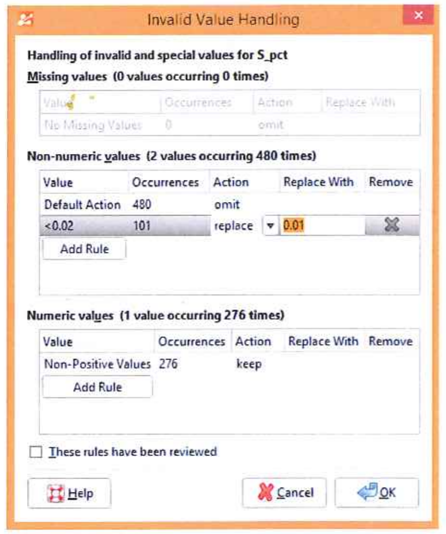


In this window we can see that there are no missing values, 2 non numeric values which occur 480 times, and 1 numeric value which occurs 276 times. We will start by fixing the non numeric values.

2. Select **add rule** under the non numeric values heading. A window with the values in error will appear. We can see that these values are <0.02, and NS. Highlight <0.02 and click **OK**.

This adds <0.02 to the non numeric values window, which we can then use to define the action we would like to take whenever Leapfrog comes across this value in the Sulphur table.

3. In this case, <0.02 is the detection limit, so we will replace it with half the detection limit, which is 0.01.



4. Click **add rule** again, and this time highlight NS.

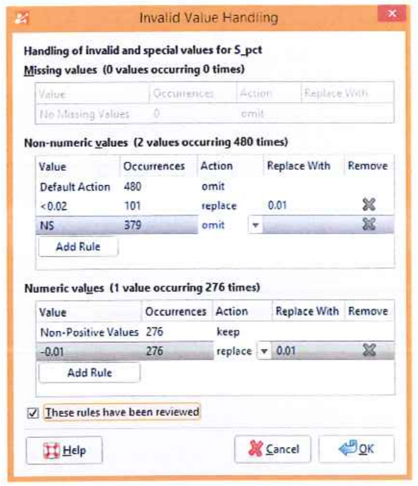
5. For these intervals there are no samples, so change the action to 'omit'.

There are also 276 numerical value errors.

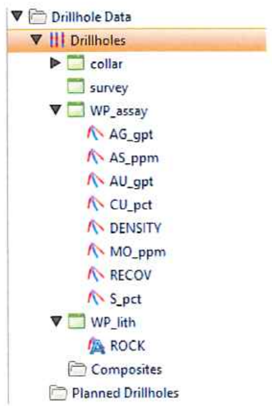
6. Click the **add rule** button to see that the value is -0.01, then click **OK**.

7. Change the action to 'replace', and use 0.01 as the number to replace the value with.

8. Once the rules have been changed, tick the **these rules have been reviewed** checkbox, which will remove the red cross from the assay table in the project tree.



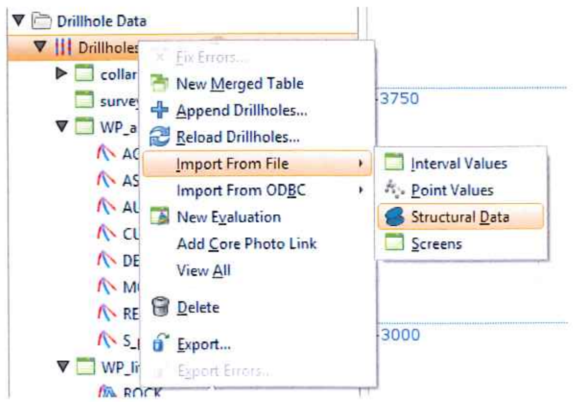
Repeat this process for the other assay tables until the errors have been removed.



Importing Downhole Structural Data

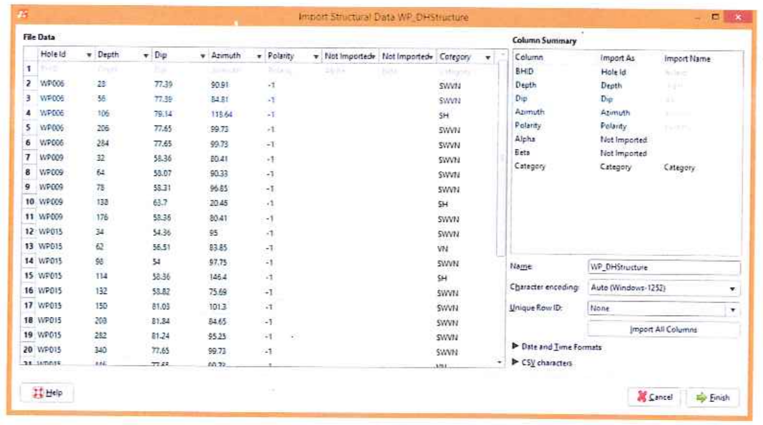
Structural data can be imported into Leapfrog in either dip - dip azimuth with an associated downhole depth and hole ID, or as alpha - beta measurements downhole. We will look at importing some downhole structural data for the Wolfpass project in both formats.

1. Right click on the **Drillholes** folder and select import from file > structural data.

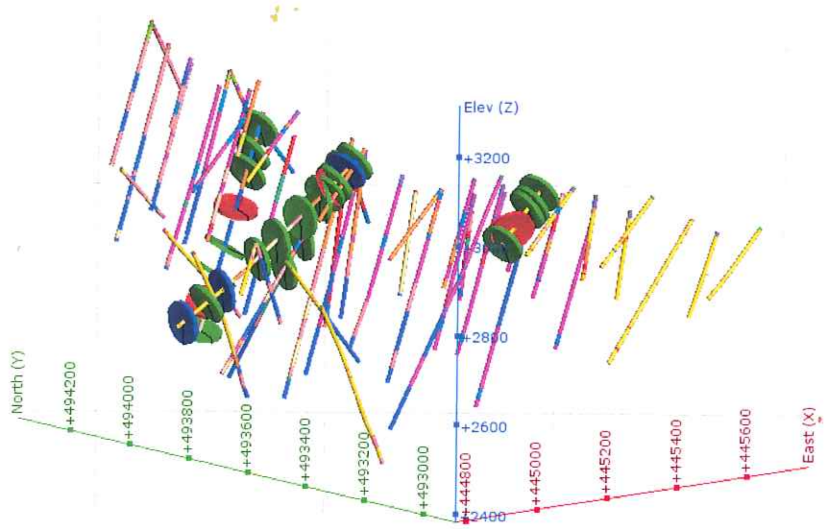


2. Import the `WP_DHStructure.csv` file from the drillholes folder.

3. The columns are selected by default by Leapfrog. In this case, the Alpha and Beta columns have been selected even though they don't have any data. Change these two columns to 'Not Imported'. The Category column needs to be imported as Category data.

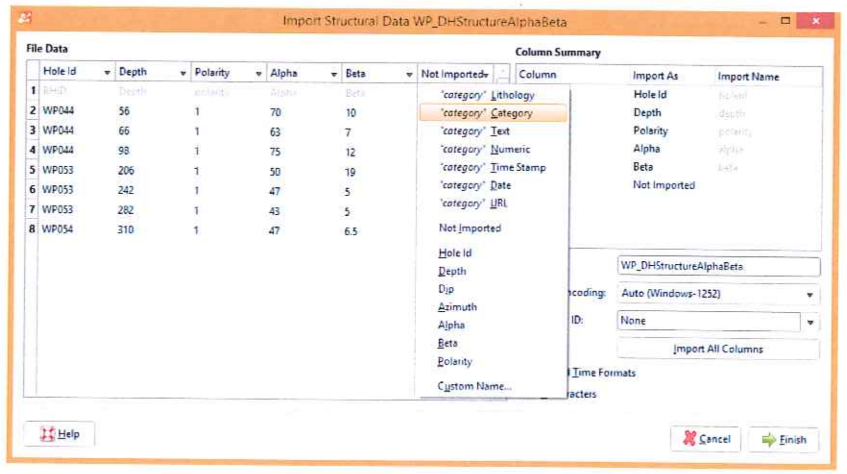


4. Click **Finish** and the structural data will be imported. View the data alongside the drilling.



Next we will import the alpha - beta structural data. This is very much similar to importing dip - dip azimuth structural data, but once imported, the position of the orientation mark must be specified.

1. Import the `WP_StructureAlphaBeta.csv` file following the same steps as above.
2. Again, select the category data type for the category column, then click **finish**.



To specify the position of the core orientation, double click on the data you have imported in the project tree and go to the compatibility tab. Make sure this is set to 'bottom of core' for this dataset.

Grouping lithologies

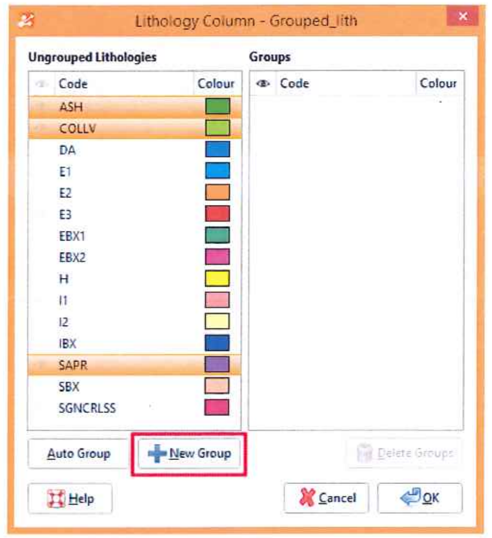
Data often comes from multiple sources, and different geologists (in some cases logging over several decades) use different terminology. When building models, there may be specific reasons for treating multiple lithologies in a single group. In these cases, lithologies can be grouped in Leapfrog without altering the original data source.

Grouping lithologies adds a new column to the lithology table, which allows you to model either with it or with the original data.

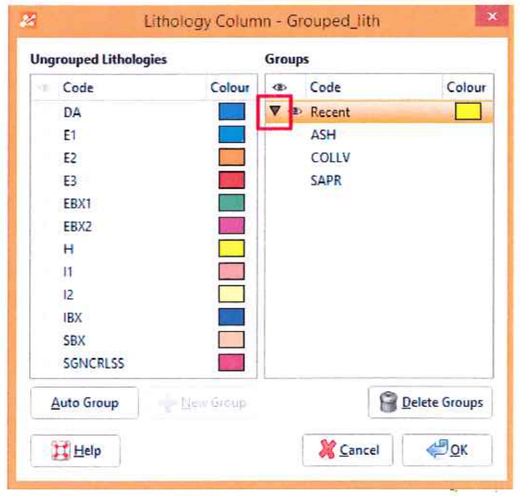
1. Add the lithology table to the scene
2. Right click on the lithology table, and select **New Column** > **Group Lithologies**
3. In the window that appears, make sure ROCK is left as the base column, and change the name to 'grouped lith'. Click **OK**.

In the window that appears, we will create new groups and add lithologies to these groups.

4. Highlight ASH, COLLV and SAPR using ctrl or shift select, then click **new group**.

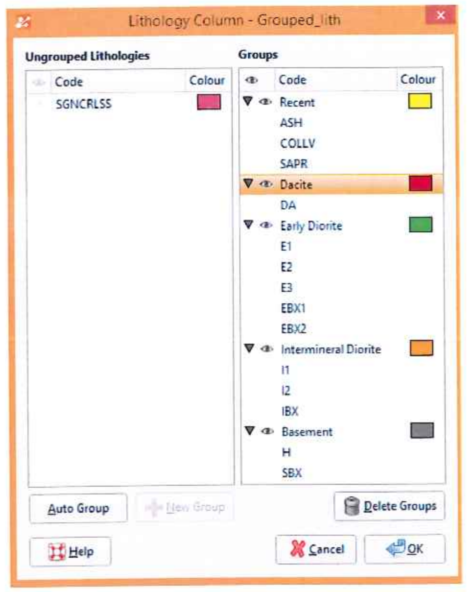


5. Enter the name 'Recent' for the group. Expand the small triangle to the left of the new lithology to see the lithologies which make the 'Recent' group up.

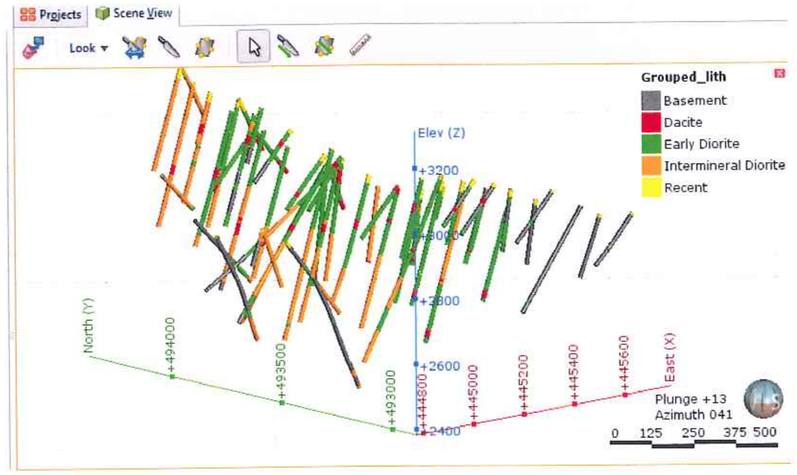


6. Repeat the process to create the following groups as described at the start of this session:
   - Basement: H, SBX
   - Dacite: DA
   - Early Diorite: E1, E2, E3, EBX1, EBX2
   - Intermineral Diorite: I1, I2, IBX

Leave SGNCRLSS ungrouped, as it represents significant core loss so is of no use for modelling.



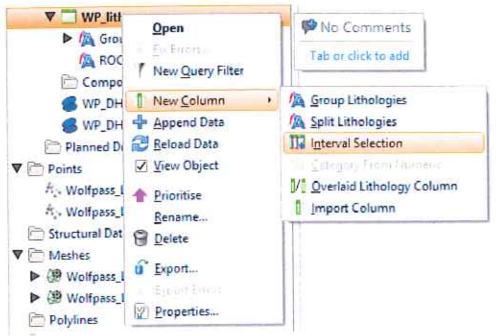
Click **OK**, and the scene will be updated to display the newly grouped lithology.



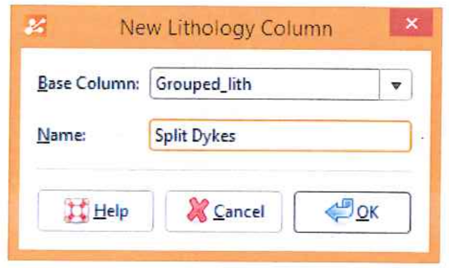
Differentiating the Dykes using the Interval Selection Tool

The Dacite lithology is made up of two dykes that appear as a single lithology, so we will split the dykes into two lithologies to make them easier to model.

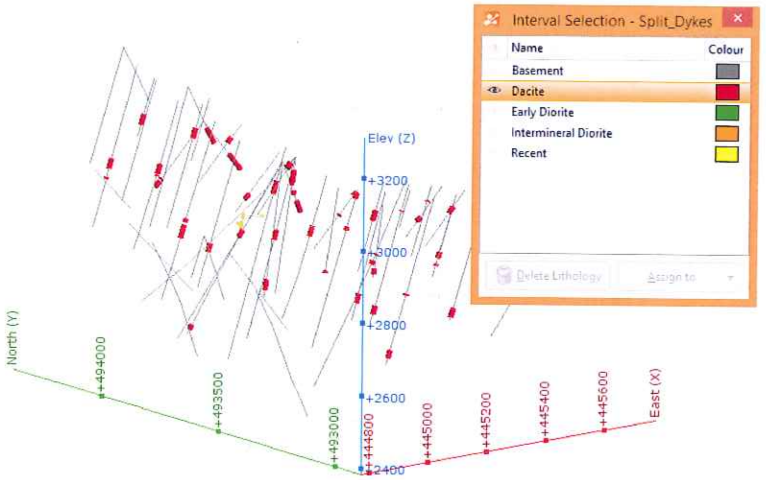
1. Right click on the lithology table in the project tree, and select **New Column** > **Interval Selection**.



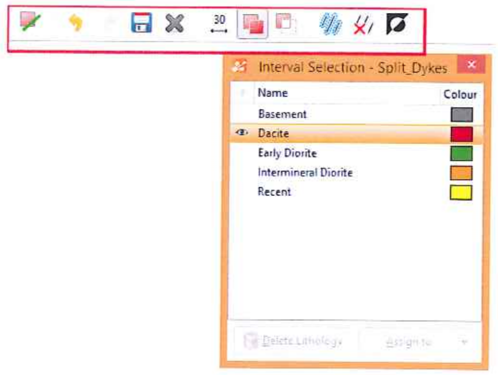
2. Change the base column to 'grouped lith', and enter 'split dykes' for the name, then click **OK**.



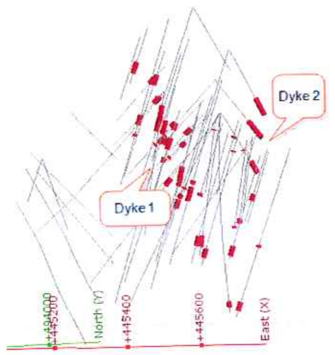
3. Hide all the lithologies apart from the dacite, and change the display of the drillholes to 3D so they are easier to see.



When an interval selection is created, a set of tools for selecting intervals in the scene is displayed in the top toolbar. Hold the cursor over each to view its function:



Line the dykes up in the scene so you can easily see both of them.

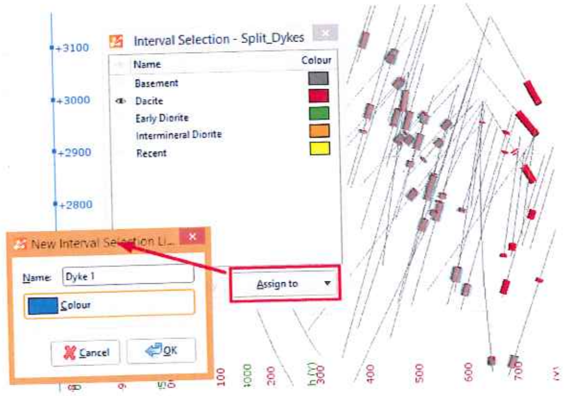


4. Click on the **Select intervals** tool ( ) and draw across the segments for the first dyke.

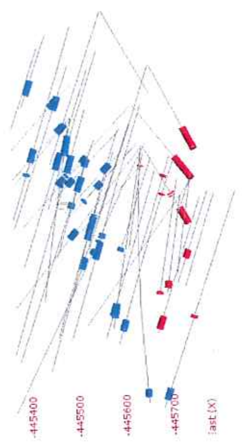
5. You can change the width of the selection tool by clicking on the line width icon in the interval selection toolbar.

6. In the **Interval Selection** window, click **Assign to**, then **Create New Lithology**.

7. In the window that appears, enter the name "Dacite Dyke 1", then click **OK**:



The selected segments will be assigned as the new dkye.



8. Select segments for the second dyke and create a new category for it.

Note that it isn't crucial to get the selections perfect at this stage, as we can always go back later to make adjustments.

9. When you have finished, click the **Save** button in the toolbar.

10. Close the **Interval Selection** window.

The raw data has now been successfully imported, validated and prepared for modelling so we can move on to the next step, which is creating the geological model for the Wolfpass deposit.

Session 13: Wolfpass Geological Model

Contents

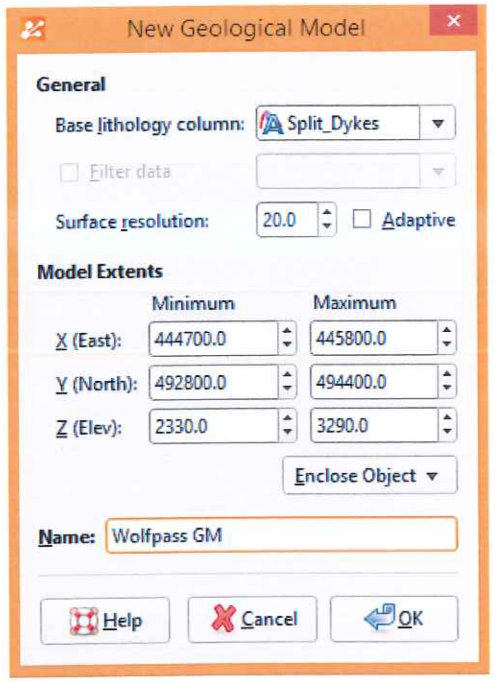
Goals

In this session, we will look at creating a geological model with three dykes, then use the features in Leapfrog Geo to create a vein system to model the dykes with.

The data for this session is in the `Sessions \ Session 10 - Vein Modelling` folder.

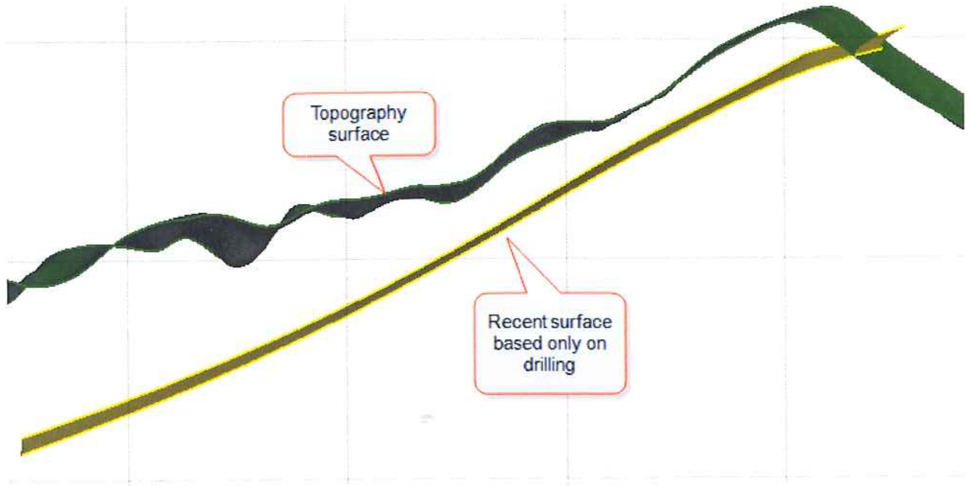
We will start by modelling recent layer, which is reasonably well defined at the top of most drillholes. Then we will work our way from youngest to oldest; from the dykes, then the intermineral diorite, then the early diorite. The last lithology to be modelled is the basement, which will be defined as 'everything else'. This means we won't need to create a separate surface for the basement, as it will fill in the gaps between the other lithologies.

1. Create a new geological model with:
    - Split_Dykes set as the **base lithology column**
    - The WP_lith table as the extents
    - The resolution set to 20
    - The name Wolfpass GM.



Building a Cover Sequence or Weathering Profile using a Surface Offset

By building the Recent lithology using the erosion tool under the surface chronology folder, the recent surface becomes overthickened towards the edges of the model where there is limited data.



To prevent this, we will create a new mesh from offset points.

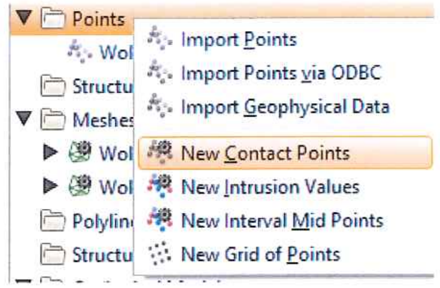
This lets us use the points from the recent intervals combined with an offset topography surface to create a more realistic surface away from drilling data. This technique works particularly well for deposits relating to the topography such as laterites.

The recent unit was created by grouping the ASH, COLLV and SAPR. Of these lithologies, the saprolite is the only weathering feature which is affected by topography. To take this into account, we will offset the topography surface to the SAPR contact points. This will allow us to create a surface which honours the data in the drilling, but away from the drilling it will have a reasonably consistent thickness as would be expected from this type of surface.

The offset surface function is available in the meshes folder. The steps below run through the process involved in creating an offset surface.

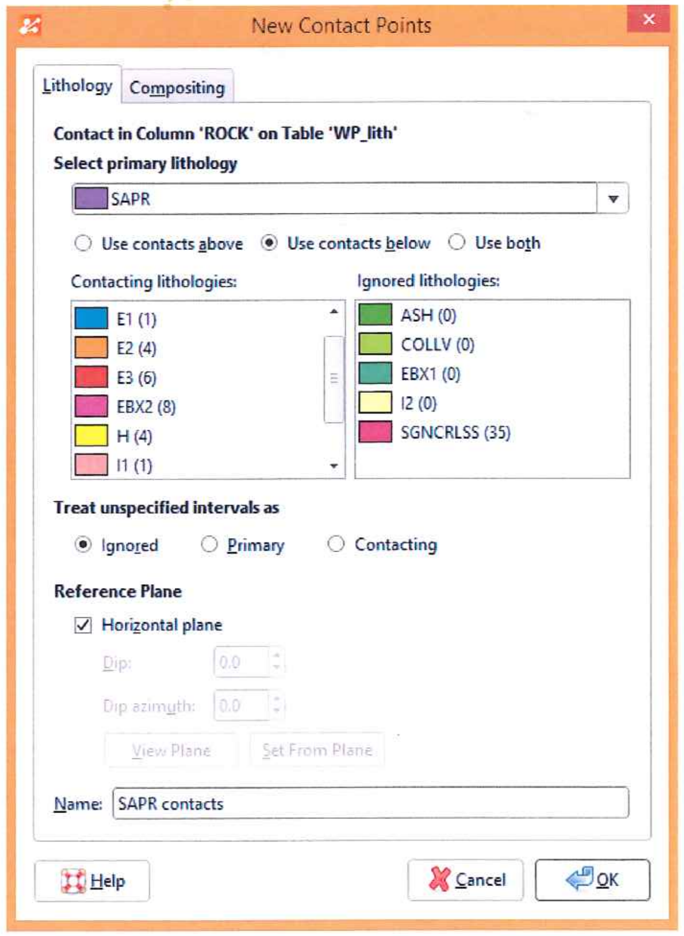
Creating the contact points

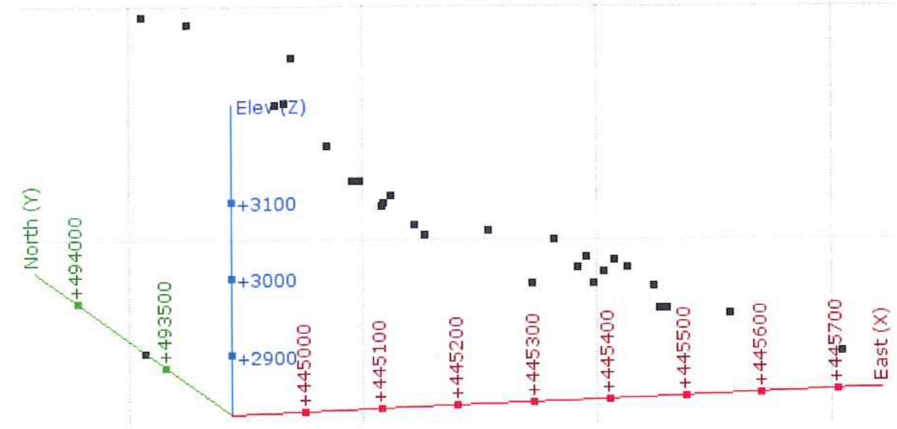
1. Right click on the Points folder and select **new contact points**. This will bring up the same dialogue that appears when a depositional or erosional surface is created, but it will only create the contact points.



2. Select the 'ROCK' column to extract points from

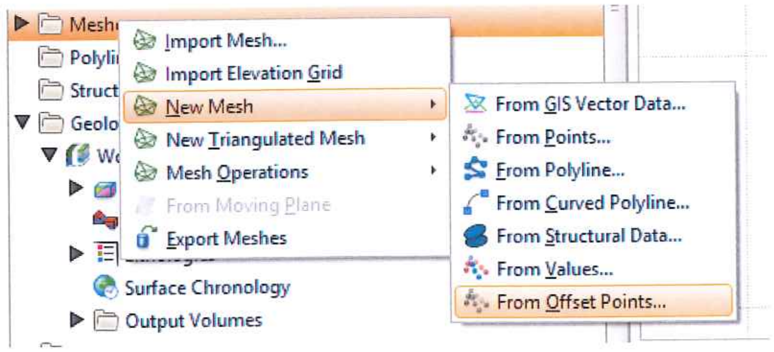
3. Select 'SAPR' as the primary lithology, and change the point generation type to **use contact below**. Make sure to add the SGNCRLSS (significant coreloss) to the ignored lithologies window.





Creating the Offset Mesh

1. Right click on the **meshes** folder and select **new mesh** > **from offset points**. The offset points dialogue will appear.

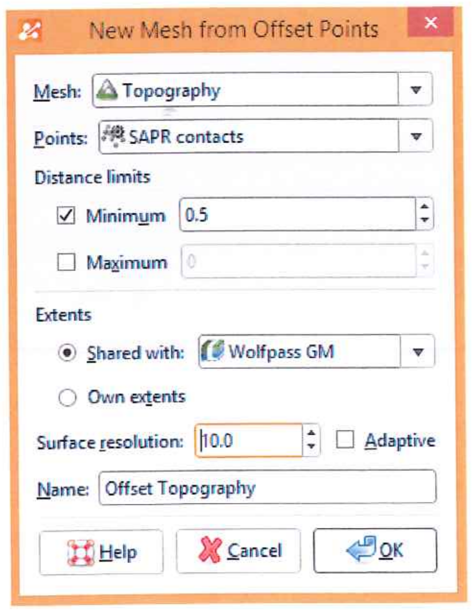


2. Set the mesh to be the topography, and the points to be the SAPR contacts.

The distance limits (minimum and maximum) limit the minimum and maximum offsets of the mesh.

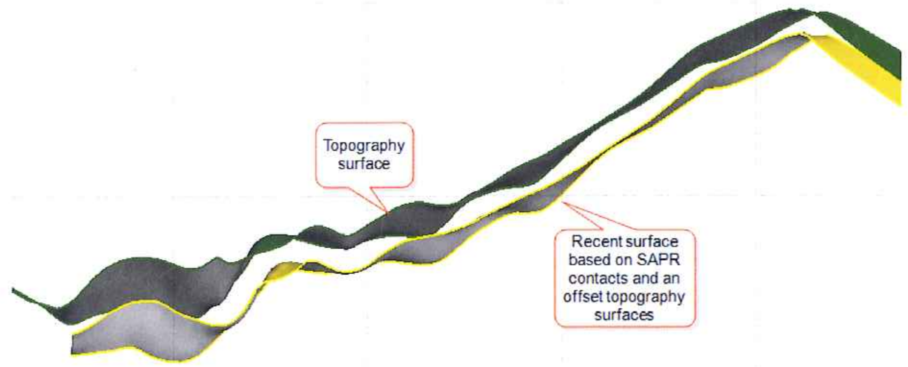
3. Set the minimum distance limit to 0.5, which means the surface will always be offset a minimum of 0.5m.

4. Set the surface resolution to 10, and click **OK**.



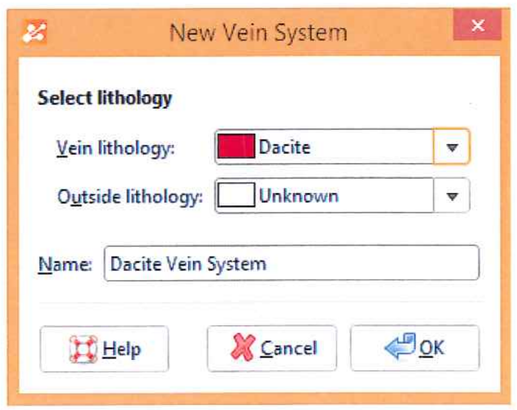
5. Now that the mesh has been created, it needs to be added as a contact.

6. Right click on the surface chronology of the geological model and select **new erosion** > **from surface**. Set the first lithology to 'recent', and click **OK**.

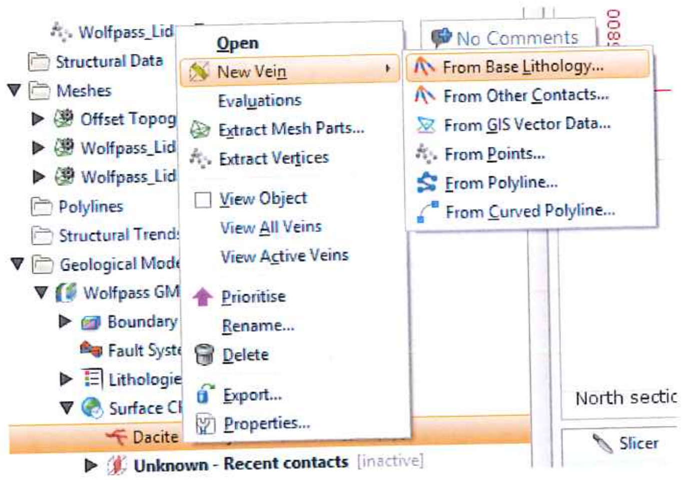


Modelling the Dykes

7. From the **Surface Chronology**, create a new vein system, selecting Diorite as the vein lithology.



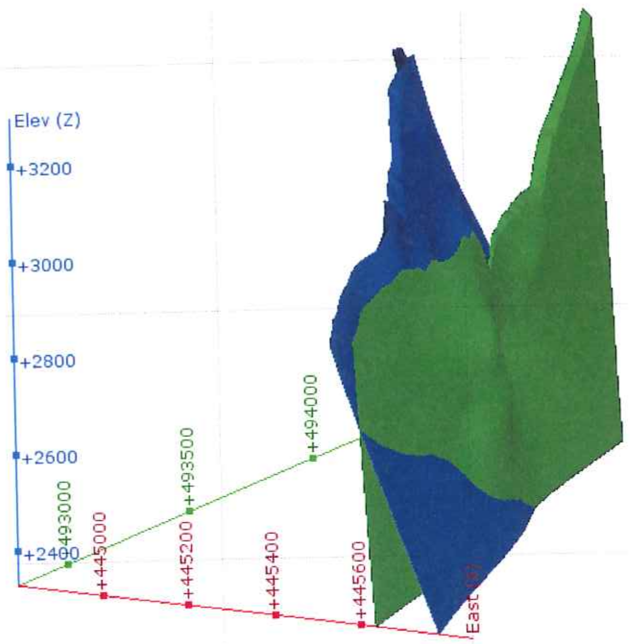
8. Right click on the Dacite Vein System which is created in the scene, and select **new vein** > **from base lithology**



9. Select dyke 1 as the vein lithology, and click **OK**.

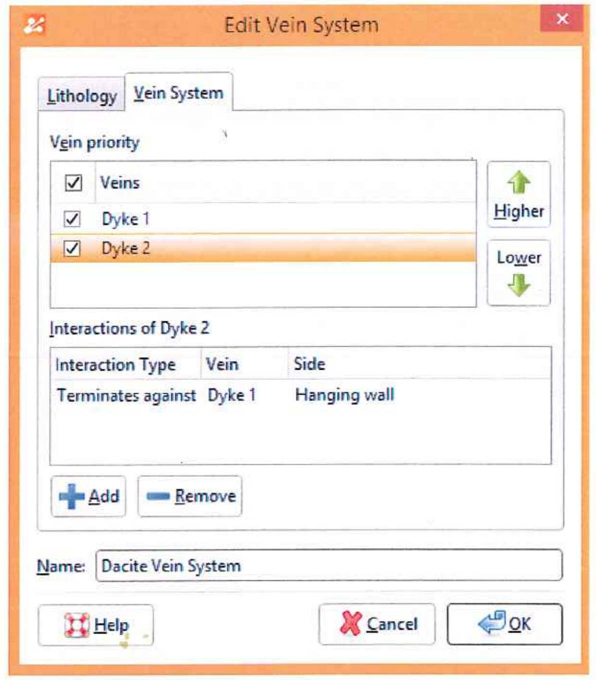
10. Repeat for dyke 2.

11. View the dykes in the scene, and note that they cross over each other at the southern end of the model.

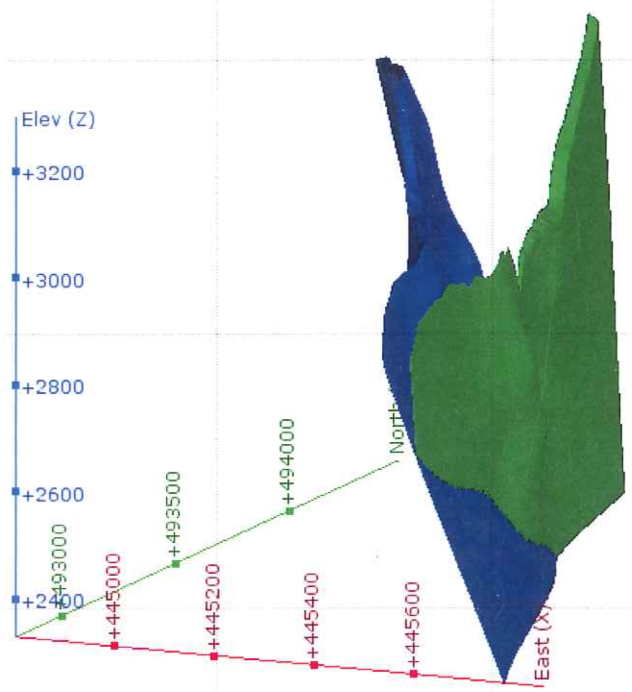


12. Double click on the dacite vein system, then click on the vein system tab.

13. Activate each of the dykes by ticking the checkbox next to each, then add an interaction for dyke 2 - it should terminate against dyke 1 on the hanging wall side.



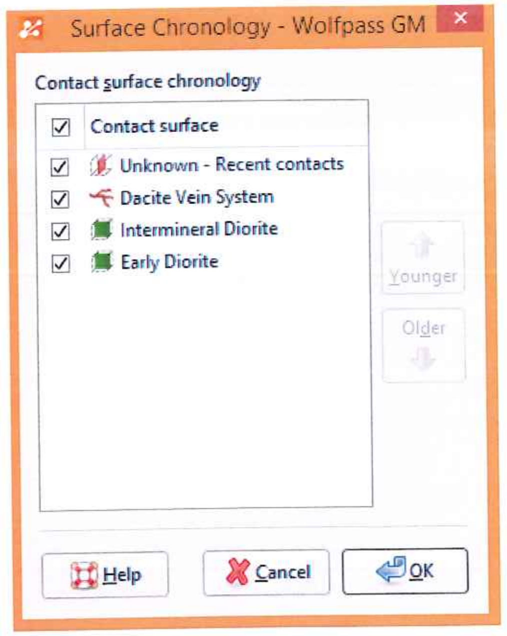
14. Click **OK**, then view the dykes in the scene again. Note that they no longer cross over, and that dyke 2 terminates against dyke 1 on the hanging wall side.



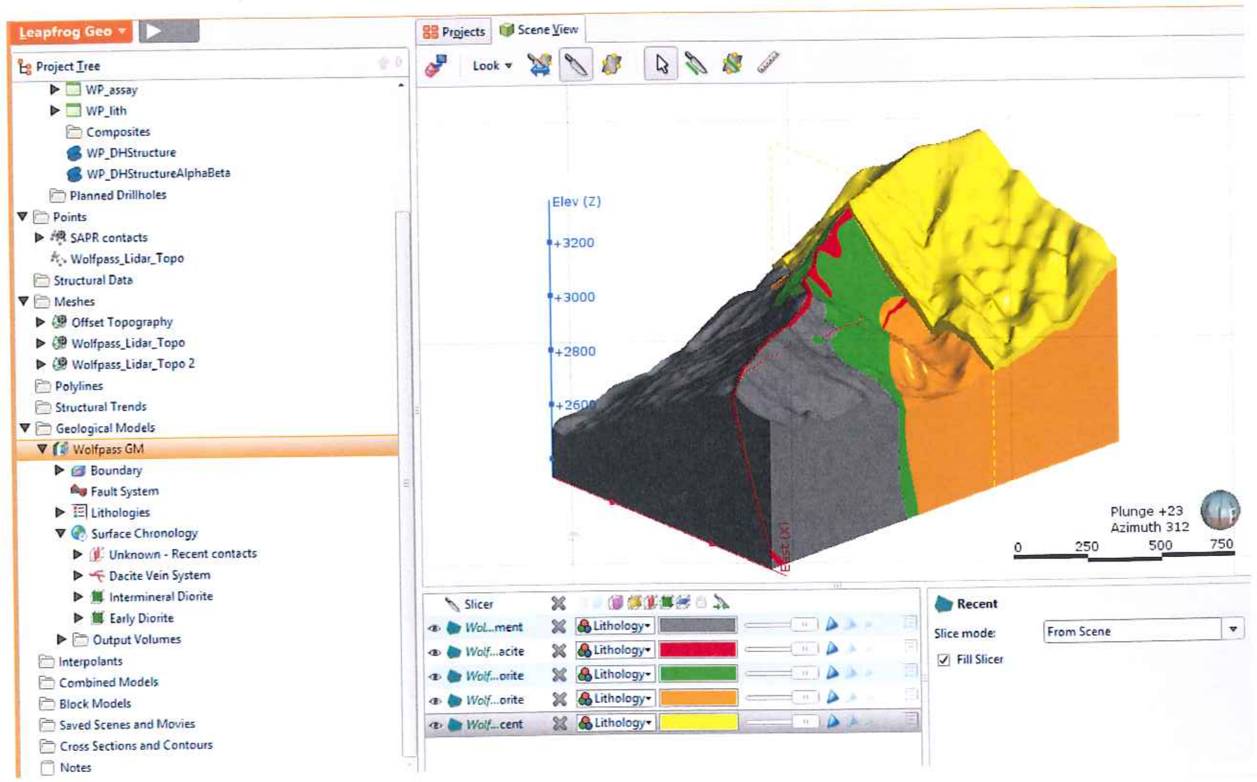
We will look at editing the dykes at the end of this session once the other lithologies have been created.

Modelling the Other Lithologies

Model the other lithologies using the techniques from earlier sessions and add the lithologies to the surface chronology. The Intermineral and Early Diorites will both be modelled as intrusions, then the Basement added as the 'other side' of the Early Diorite. Once the surfaces have been created, the volumes can be created by double clicking on the Surface Chronology folder, making sure the surfaces are in the correct order, and ticking the checkboxes.



The model should appear similar to this:



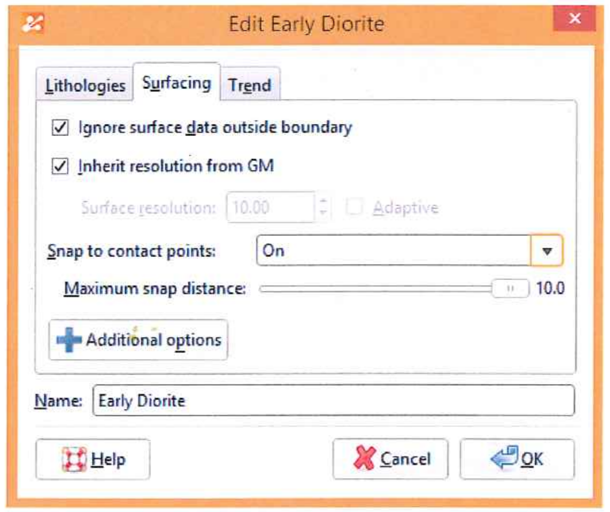
Snapping a Surface to Points

By default, surfaces in Leapfrog don't snap to perfectly to points. This is because the method Leapfrog uses to create surfaces involves turning categorical intervals to numeric values, and putting a surface through the "0" isovalue (with progressively more positive values inside the volume, and progressively more negative values outside the volume).

Default surfaces will always pass through the contact very close to the 0 isovalue, but it is sometimes necessary to force surfaces to exactly honour the contact point.

1. Double click on the surface you would like to snap to contacts, and go to the "surfacing" tab.

2. There is a "snap to contact points" dropdown option - change this to "on". The maximum snap distance controls the distance away from the default surface that the snap will move. By making this as large as possible (which will be the same as the resolution for the surface), the new "snapped" surface is able to move that distance away from its default. By making it as small as possible (zero), the surface won't be able to move away from its default at all.



Editing the dykes

Now that the initial geological model has been completed, we will look at editing the dykes. There are a number of tools available to use - these will be discussed in more detail in the Vein Modelling exercise. For this session, there is a cross section which can be imported to add additional detail to the dykes.

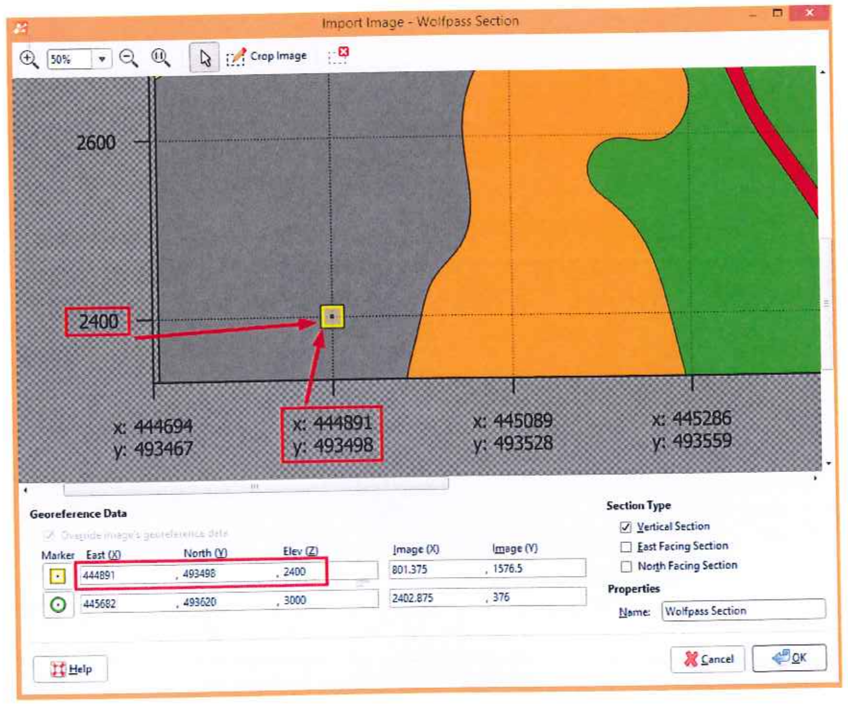
1. To import the cross section to Leapfrog, right click on the **Cross Sections and Contours** folder in the project tree and select **New Cross Section from Image**.

2. Navigate to the cross section named "Wolfpass Section.png", which is in the folder for this session.

3. It is a vertical section, so tick the "vertical section" checkbox in the bottom right corner of the import window.

4. Place the yellow square and green circle markers on two points along the grid of the section.

5. In the "Georeference Data" area, complete the easting, northing and elevation for each marker.



6. Click **OK** and the section will be imported.

7. An easy method of viewing the cross section in relation to the existing GM is to clear the scene, then drag the current geological model and the imported cross section into the scene.

8. By default, the cross section will be displayed using a flat colour - this can be changed to the actual cross section using the dropdown box in the scene list.

9. Now put a slicer through the scene - it can be anywhere and doesn't have to line up with the cross section.

10. Highlight the "Slicer" icon in the scene list, then in the properties panel to the right, click the **Set To** dropdown box. Select "Wolfpass Section", and the slicer will move to be located along the imported section.

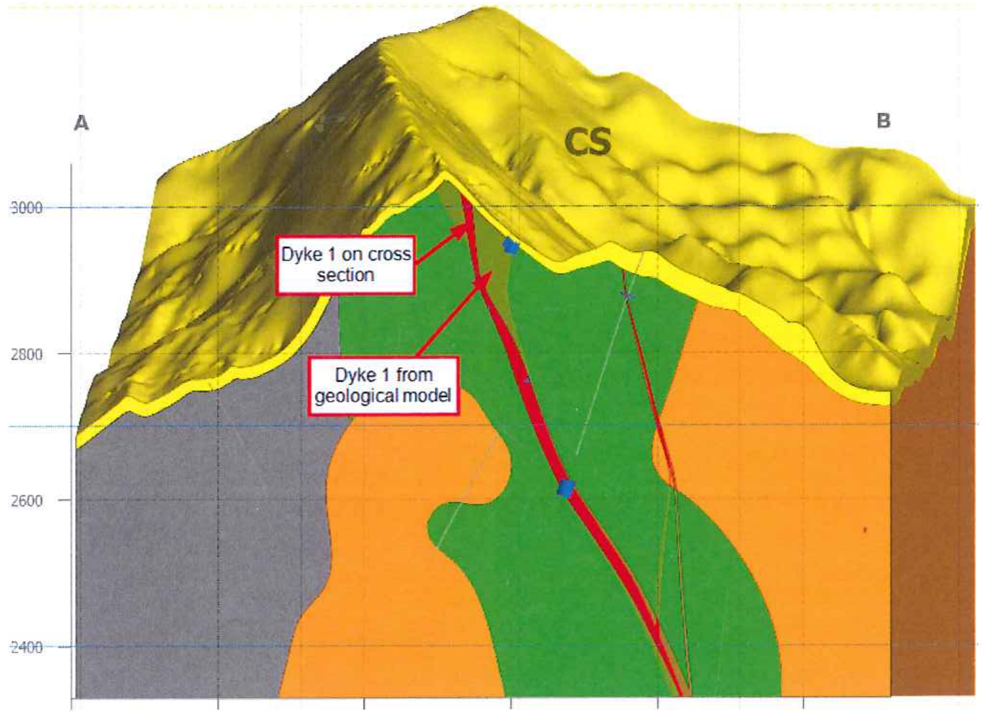
11. If the slicer has removed the wrong side of the model or is showing the thick slice, make sure "Remove Back" is selected as the slice mode.

12. To view the geological model behind the cross section, make the cross section slightly transparent using the transparency slider.

Now that we can see the interpreted section next to our existing geological model, we can use two methods to edit the geological model. Firstly we can edit the interval selections for the dykes, and secondly we can draw polyline edits for the hangingwall and footwall surfaces of the dykes.

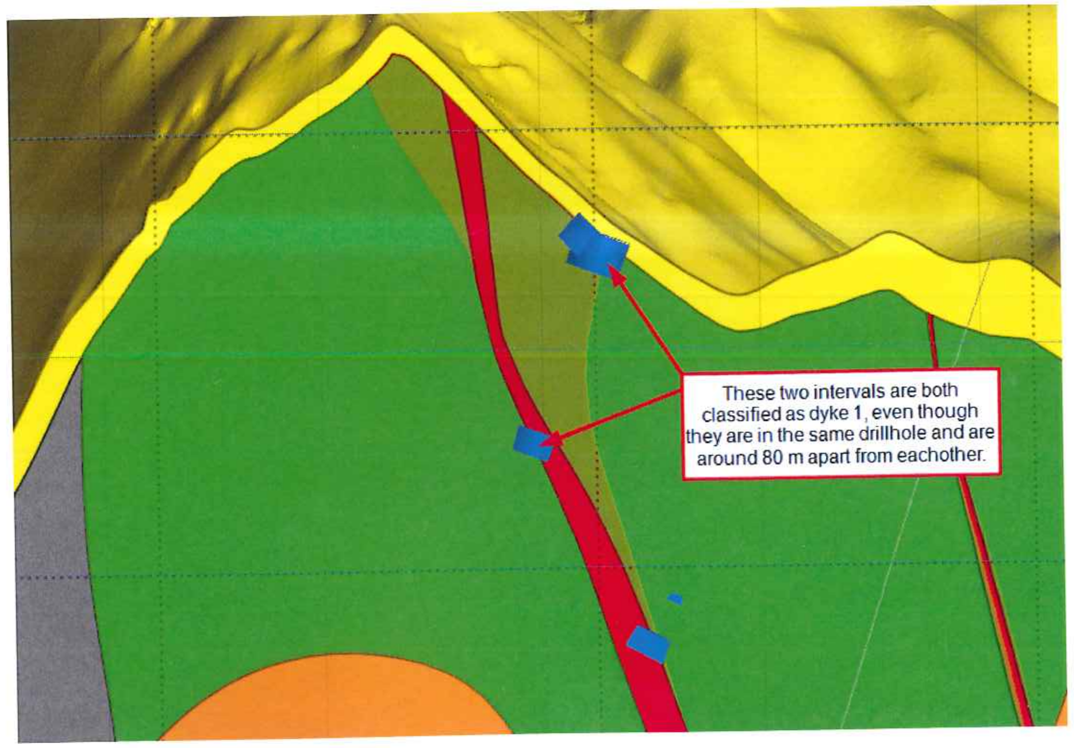
Editing interval selections based on cross section

Keep the geological model and partially transparent cross section in the scene, and drag in the "Split_Dykes" intervals (these are the intervals we used to split the dacite into two separate dykes). Make sure only the dykes are visible, and the drillholes are being displayed in 3D. It's also useful to change the slice mode of the drillholes to "thick slice" so the drillholes are visible in front of the geological model.



If we zoom in on this image, we can see why there is a difference between the dyke in the cross section and the dyke in the geological model:

leapfrog
FAST, DYNAMIC GEOLOGICAL MODELLING



We can see there are two intervals in the same drillhole which have been classified as dyke 1. When Leapfrog creates the hangingwall and footwall surfaces, the bottom of the lowest interval becomes the footwall point, and the top of the highest interval becomes the hangingwall point. In this case, the preferred surface would be created by using the top and bottom of the lowest interval only. This means we need to edit our original interval selection to remove the upper intervals in the image above.

1. Right click on the "Split_Dykes" drilling table and select **Open**.

2. Make sure that only the intervals classified as either Dyke 1 or Dyke 2 are visible, and change the drillholes to 3D view.

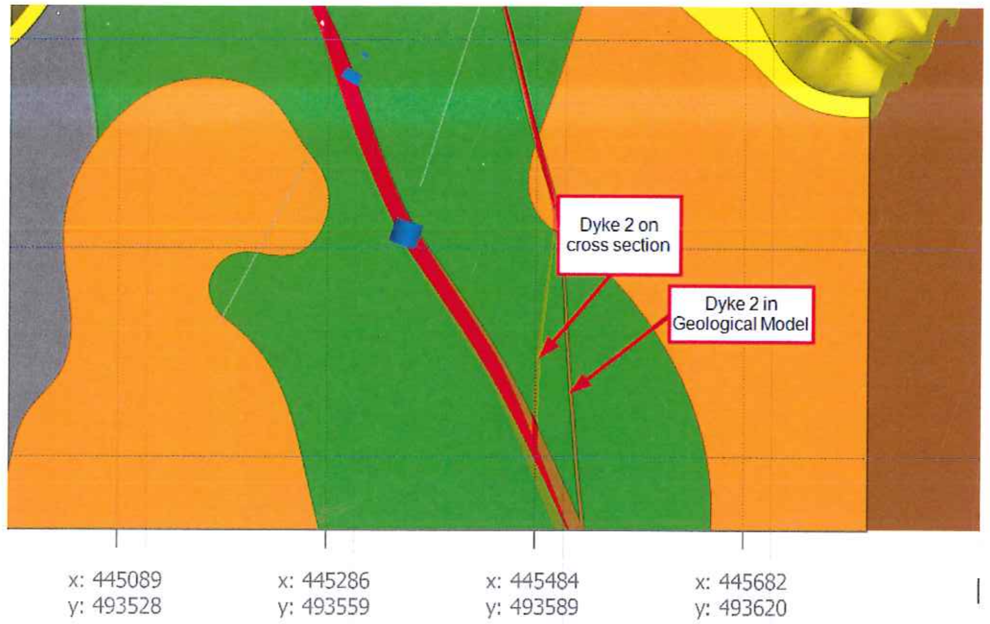
3. Using the interval selection tool, select the upper intervals and assign them to "Unassigned".

4. Click the save button, and the entire model will be re run to include the edited interval selection.

leapfrog
FAST, DYNAMIC GEOLOGICAL MODELLING

5. Once the model has finished processing, it will be displayed in the scene automatically

Depending on the interval selections you made, there may also be an issue with dyke two towards the bottom of the model:



To correct this issue, we need to manually edit the hangingwall and footwall with polylines based on the cross section.

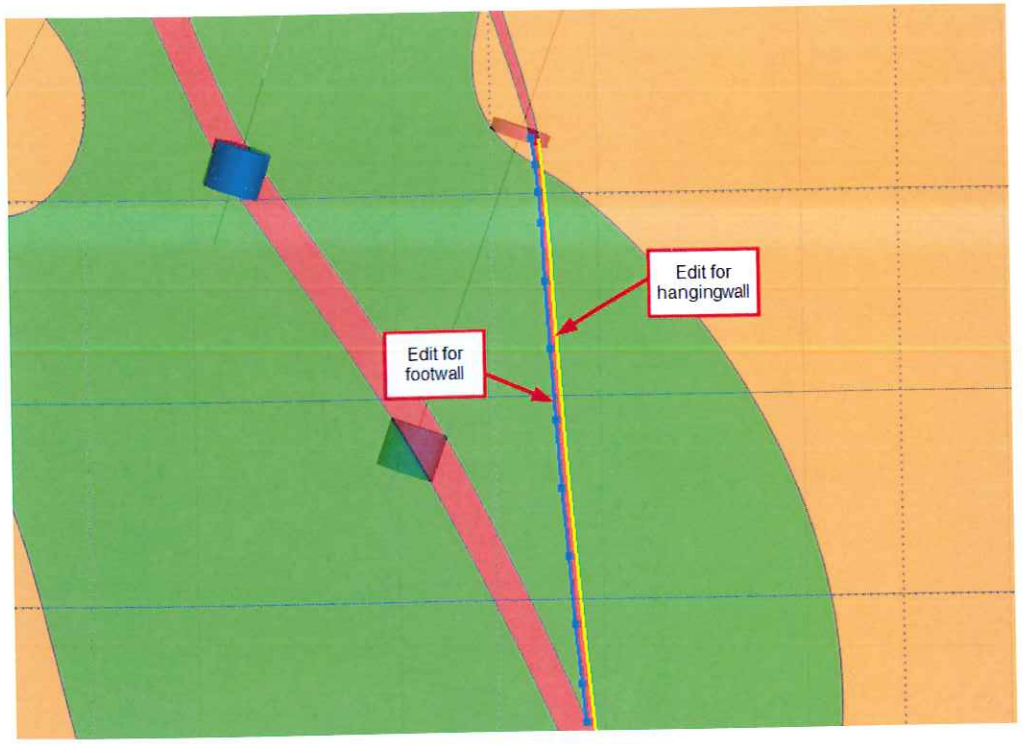
Editing using polylines based on a cross section

1. Keep the same sliced view as earlier, and expand the "Dyke 2" object out in the project tree until the hangingwall and footwall objects are visible (the icon is a blue triangle).

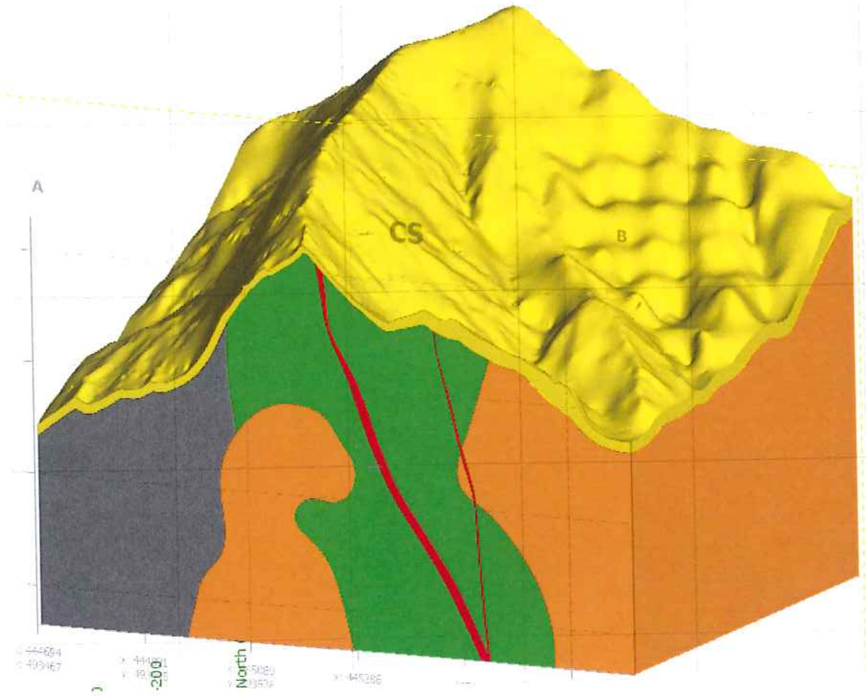
2. Right click on the hangingwall object and select **Edit** > **With Polyline**.

3. The polyline editing toolbar will appear at the top of the screen. Select the **Draw Lines** button, and digitise along the bottom few hundred metres of dyke 2 as shown on the cross section.

4. Click **Save**, then repeat for the footwall.



5. Again, the model will be reprocessed to include the new edits. Compare the edited geological model with the cross section to check that it represents it as expected. If required, edit dyke 2 using polylines.



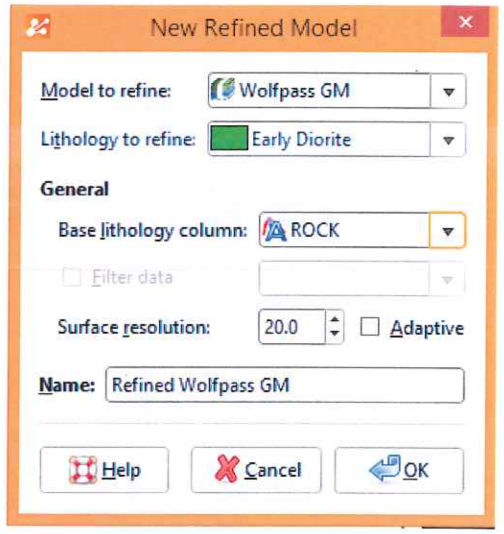
Creating a Refined GM

Now that the grouped lithologies have been modelled, we can look at the original pre-grouped lithologies and see if there is any part of the model which can be created in more detail. In this case, the grouped Early Diorite lithology can be split up and modelled as seperate lithologies.

1. Right click on the Geological Models folder and select **New Refined Model.**

leapfrog
FAST, DYNAMIC GEOLOGICAL MODELLING

2. Select 'Wolfpass GM 'as the model to refine, and Early Diorite as the Lithology to refine. Change the Base lithology column to ROCK (this is the drillhole data prior to grouping).



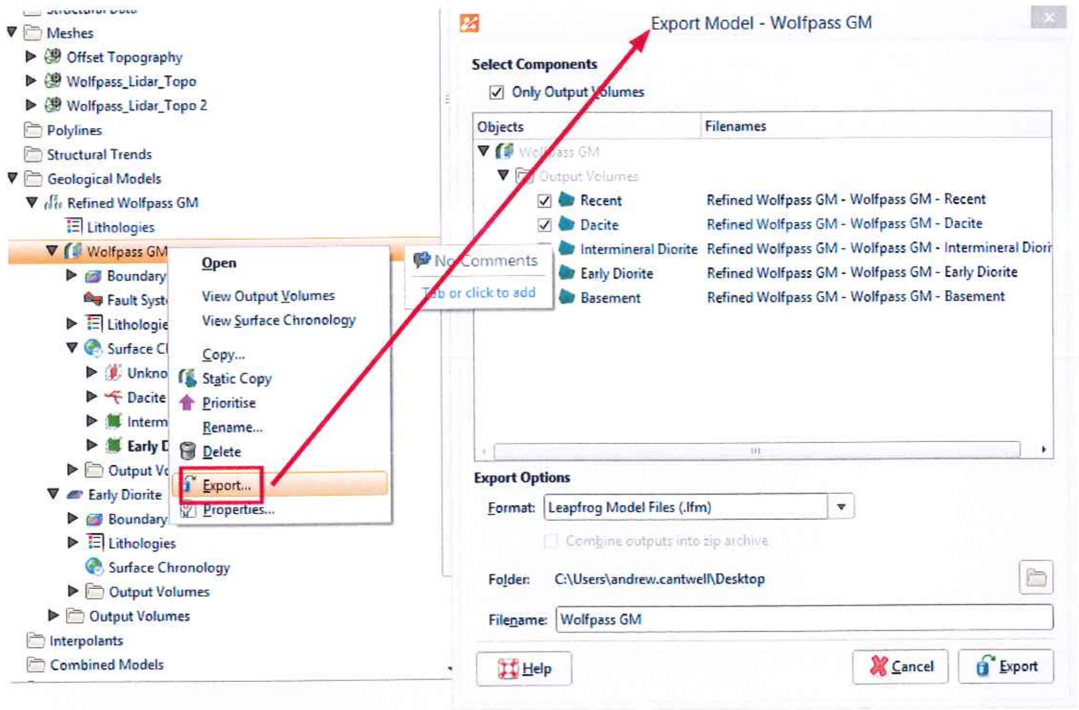
A new sub folder will appear underneath the Geological Models folder. This gives you the same options as you have for the full GM, including the ability to create new boundaries and surface chronologies.

3. Build new surfaces to subdivide the existing Early Diorite volume into its separate components (E1, E2, E3, EBX1, EBX2)

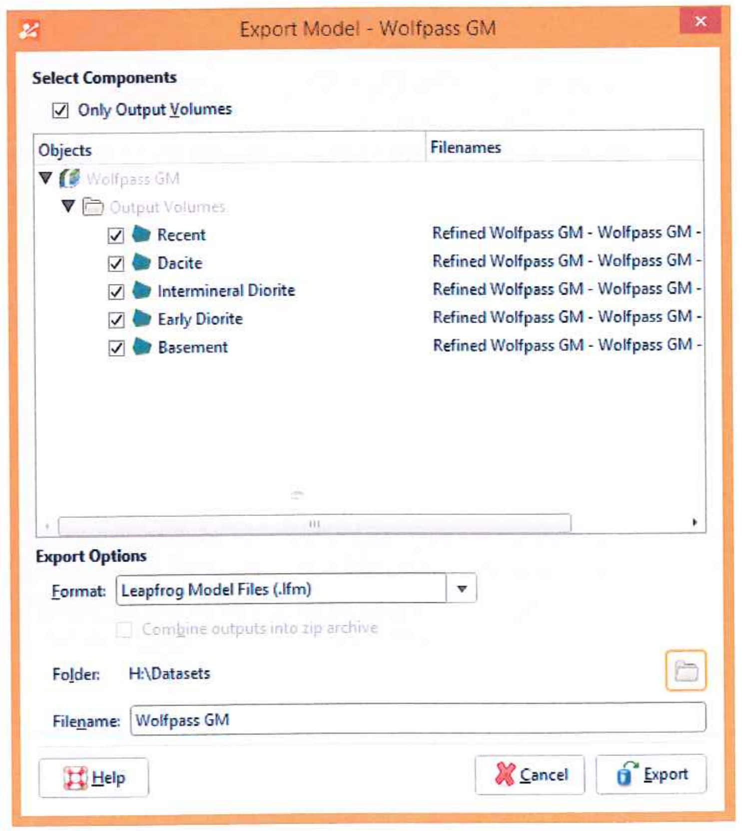
Exporting a Leapfrog Model

You can export an entire Geological Model as a *.lfm file, which will export each surface making up the Geological Model in a single file. This file can then be imported back into Leapfrog Geo under the meshes folder.

1. Right click on the Wolfpass Geological Model and select **Export**. By default the **Only Output Volumes** checkbox is ticked. By unticking this, the surfaces making up the geological model also become available for export. For this example, leave the checkbox ticked.



2. At the bottom of the window, select the format to export the model in, then choose which folder to save the model to.



3. To reimport the meshes to Leapfrog, right click on the **Meshes** folder and select **Import Mesh**. Navigate to the saved Leapfrog model file and select **Open**.

4. The meshes will be added to the Leapfrog project, and can be viewed individually or all at once. To add all the meshes to the scene without having to drag each one from the project tree, use the shift key to multiselect the meshes, and drag them into the scene at the same time.

Session 14: Wolfpass Interpolation

Contents

Creating an interpolant in Leapfrog Geo uses assay data, temperature values, geophysical data or any other numeric data which is sparsely distributed in space to interpolate across a region. Interpolation in Leapfrog is fast and flexible, and the shells produced to represent mineralisation are smoother and more reproducible than traditional hand-drawn meshes.

This session contains a relatively large amount of theoretical background. To understand how Leapfrog creates interpolants, we will need to introduce basic interpolation and geostatistics concepts.

For this session we will be using the project used for the Wolfpass sessions.

Modelling Approach

An interpolant can be built in four steps from a variety of data. Any data that contains points with X,Y,Z coordinates and an associated value can be used for interpolation.

- The first step is to clean the drillhole data by removing inconsistencies in the data. This can be a time-consuming process with some data sets, but it is critical as the quality of any model ultimately depends on the quality of the data. In this example, the data has already been cleaned in the first Wolfpass session so we can proceed to the next step.
- The second step is to select the numeric values then apply appropriate parameters to the values. An interpolant estimates the values over a region from an initial set of point values. The numeric values can be selected directly as points if they have been imported into the **Points** folder. If you are creating your interpolant from drillhole data, Leapfrog will allow you to select the segments used to generate points. When you are adjusting the interpolant later on, you can work directly with point values.
- The third step is to apply a trend. A trend allows the directions and strength of mineralisation to be defined to ensure the resulting interpolant is geologically reasonable. Adding a global or structural trend will alter the isosurfaces. It should be adjusted to ensure these honour the expected mineralisation patterns. This is where we will initially direct our focus in this session.
- The final step is to determine how the isosurfaces are bounded and calculate the volume of mineral within each isosurface.

Iterative Refinement

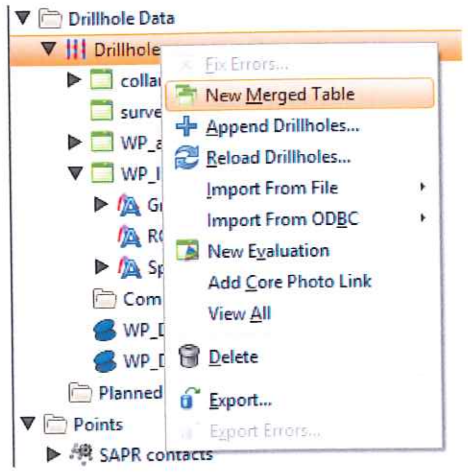
Building an interpolant is a process of successive refinement. This involves:

- Defining the interpolant and basic structures. This usually corresponds to defining the topography and boundaries.
- Refining the internal structure. This involves setting the proper trends and making manual corrections to the point and value data until the resulting surfaces are geologically realistic.

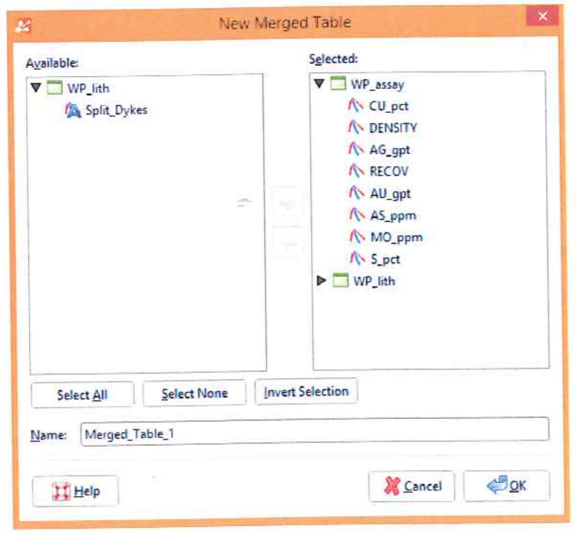
Merged Tables

A good way to find out which lithologies are important to the mineralisation is by creating a merged table.

1. Right click on the drillholes object in the project tree, and click **new merged table**.

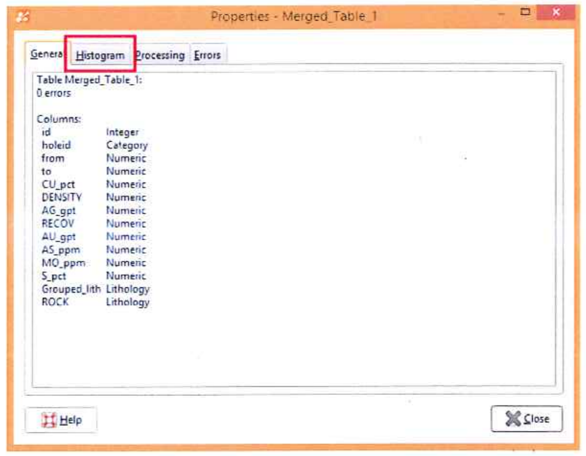


2. Select the tables you would like to merge. In this case, all of the assay columns, plus the grouped_lith and ROCK tables should be selected.

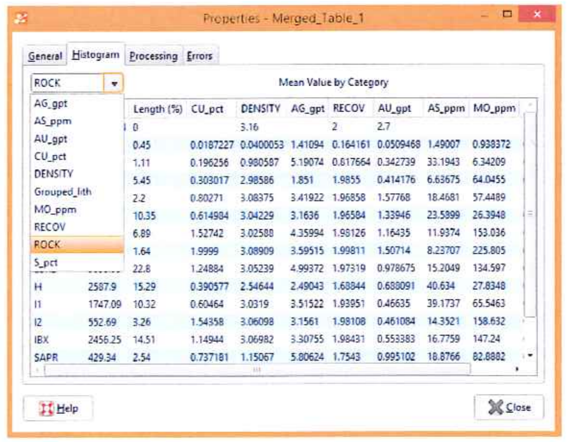


3. Click **OK**.
4. Right click on the merged table in the project tree and select **properties**.
5. Click on the **histogram** tab.

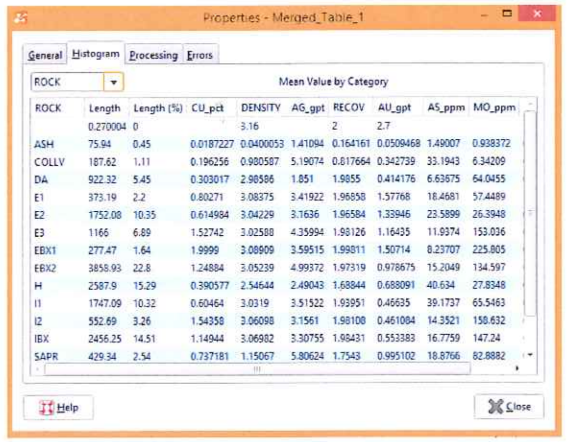
leapfrog
FAST, DYNAMIC GEOLOGICAL MODELLING



6. In the dropdown box which appears, change the selection to ROCK.



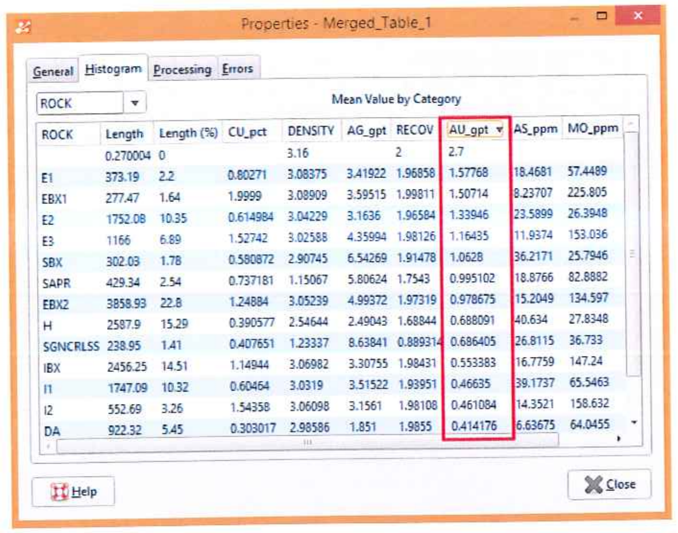
7. A table will appear with the rock type down the left hand side, and the numerical data across the top.



There are a number of useful statistics in this table.

8. Firstly, we can see the total interval length of each lithology. This can be useful when there are hundreds of codes which require grouping - by sorting the list based on interval length, it is easy to start eliminating the irrelevant codes from the grouping.

9. Secondly, we can sort the columns based on numerical values. Click the AU_gpt heading to sort based on gold grade.



ROCK	Length	Length (%)	CU_pct	DENSITY	AG_gpt	RECOV	AU_gpt	AS_ppm	MO_ppm
	0.270004	0		3.16		2	2.7		
E1	373.19	2.2	0.80271	3.08375	3.41922	1.96858	1.57768	18.4681	57.4489
EBX1	277.47	1.64	1.9999	3.08909	3.59515	1.99811	1.50714	8.23707	225.805
E2	1752.08	10.35	0.614984	3.04229	3.1636	1.96584	1.33946	23.5899	26.3948
E3	1166	6.89	1.52742	3.02588	4.35994	1.98126	1.16435	11.9374	153.036
SBX	302.03	1.78	0.580872	2.90745	6.54269	1.91478	1.0628	36.2171	25.7946
SAPR	429.34	2.54	0.737181	1.15067	5.80624	1.7543	0.995102	18.8766	82.8882
EBX2	3858.93	22.8	1.24884	3.05239	4.99372	1.97319	0.978675	15.2049	134.597
H	2587.9	15.29	0.390577	2.54644	2.49043	1.68844	0.688091	40.634	27.8348
SGNCRLSS	238.95	1.41	0.407651	1.23337	8.63841	0.889314	0.686405	26.8115	36.733
IBX	2456.25	14.51	1.14944	3.06982	3.30755	1.98431	0.553383	16.7759	147.24
I1	1747.09	10.32	0.60464	3.0319	3.51522	1.93951	0.46635	39.1737	65.5463
I2	552.69	3.26	1.54358	3.06098	3.1561	1.98108	0.461084	14.3521	158.632
DA	922.32	5.45	0.303017	2.98586	1.851	1.9855	0.414176	6.63675	64.0455

10. We can quickly see the lithologies which contain higher gold grade, as well as those with less. The three lithologies with the highest grade are E1, EBX1 and E2, and the three with the lowest grade are the ash, colluvium and dacite veins.

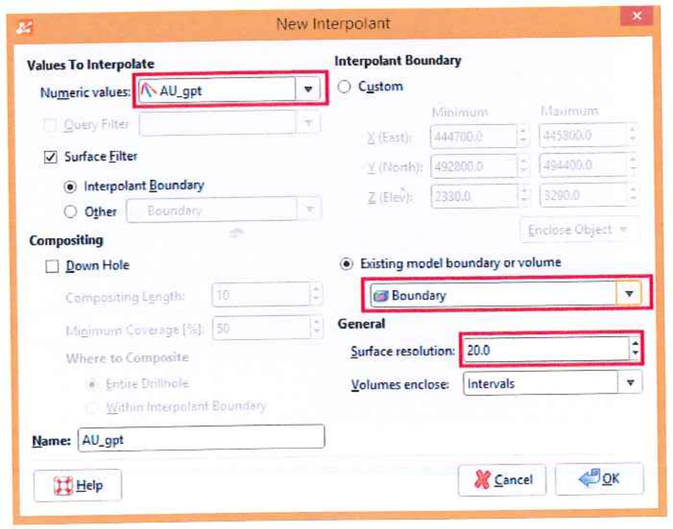
11. Repeat the above for copper, and it becomes obvious that the dacite veins are barren, and the collivium and ash don't hold any grade either. We will take this into account when compositing in Leapfrog.

Creating a First Pass Gold Interpolation

It is a good idea to run a quick interpolant through numeric data to check how the isosurfaces behave. Once this is done, we can go ahead and create an interpolant model with full knowledge of the data.

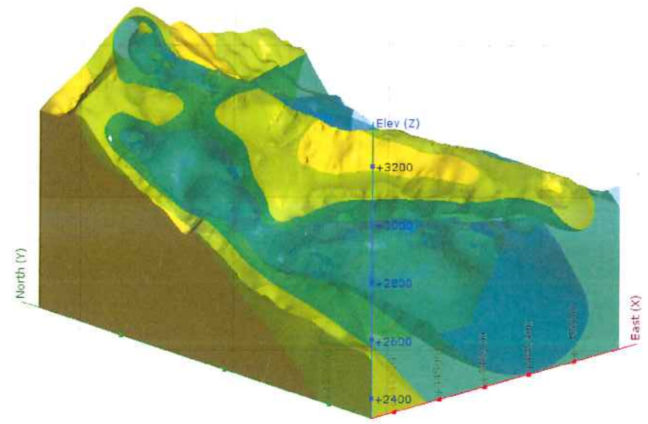
1. Right click on the **interpolants** folder in the project tree, and select **New Interpolant**.

2. The new interpolant window will appear, with a few basic options for defining the interpolant. Change the numeric values dropdown to **AU_gpt**, and the interpolant boundary to **wolfpass GM: boundary**. Change the resolution to 20, then click **OK**.



3. Once this has finished processing, drag the interpolant into the scene.

4. As you would expect for a first pass model which has been created without changing any parameters, it is unrealistic.



We will change a few of the more important parameters, and check how they change the interpolant model.

5. Double click on the AU_gpt interpolant in the project tree, to bring the **edit interpolant** window up.

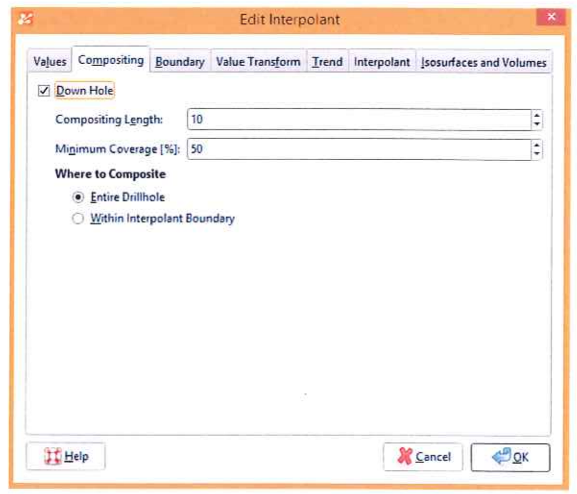
We will start by creating an interpolant model for the entire area, then look at creating a model within the Early Diorite (which is the major mineralised lithology).

For the first model, we will change parameters in the **compositing**, **value transform**, **interpolant**, and **isosurfaces and volumes** tabs. When we create the second model within the Early Diorite, we will also look at the **value**, **boundary** and **trend** tabs.

Compositing tab

Compositing in Leapfrog can be completed either within an interpolant, or in the drillholes folder. We will look at both, but for now will focus on compositing within an interpolant.

1. Click on the **Compositing** tab, and tick the "Down Hole" check box.



2. By default, the compositing length is set to 10 and the minimum coverage is set to 50%.

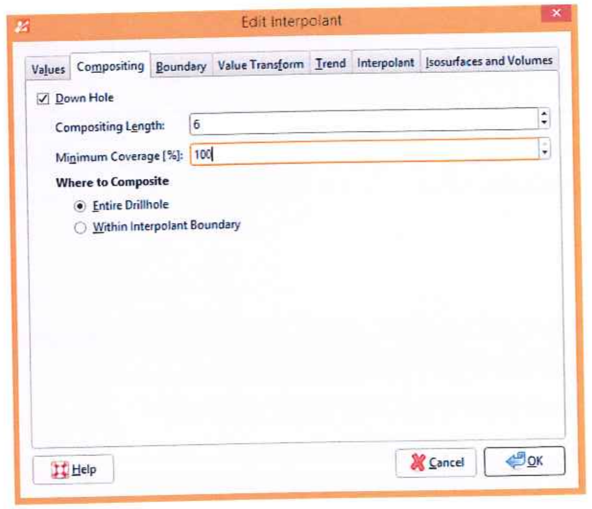
Compositing length will vary on a number of factors, including the deposit style, mining method, and raw sample length. In general, high grade underground mines will require a shorter sample length compared to bulk open pit mines. The other point to note is that if possible, samples shouldn't be "split". For example, if most intervals have a 2 m sample length, choosing a composite of 1 m or 5 m will split the intervals, which will artificially reduce variance as an interval with a single value will be represented in more than one composite. In this case, we will change the composite length to 6 m, as the raw interval lengths are mostly 2 m so the intervals aren't being split. A composite of 6 m is also reasonable for the deposit style we are working with in this case.

3. Change the **Compositing Length** to 6.

The minimum coverage defines what to do at the end of the drillhole. For example, if the total length of drilling is 181 m and we are compositing to 6 m lengths, our composites will go from 0 m - 6 m, then 6 m - 12 m etc. When we get to the bottom of the hole, the last composite will be 174 m - 180 m. There is one metre left over, so the minimum coverage defines the manner in which Leapfrog treats this last section.

- If the minimum coverage is 0 %, this means any length, no matter how short, will be retained at the end of the hole
- If the minimum coverage is 100%, this means the length at the end of the hole must be 100% of the composite length to be retained (ie, anything less than 6 m for this example will be discarded)
- If the minimum coverage is 50%, this means the length at the end of the hole must be at least 3 m to be retained (50% of 6 m).

4. Change the Minimum Coverage to 100% (this will remove any segments shorter than 6m at the end of the hole, ensuring regular sample lengths).



5. Click **OK** to process the changes before moving on to the next step.

Value Transform tab

1. Double click on the interpolant again, and move across to the **Value Transform** tab.

The **Transform Type** can be set either to **None** (default) or to **Log**. For this example, changing the transform type to Log will change the histogram so it is more normally distributed. The **Pre-log shift** option becomes available once the Log transform is selected; this prevents issues when taking the logarithm of zero or negative numbers.

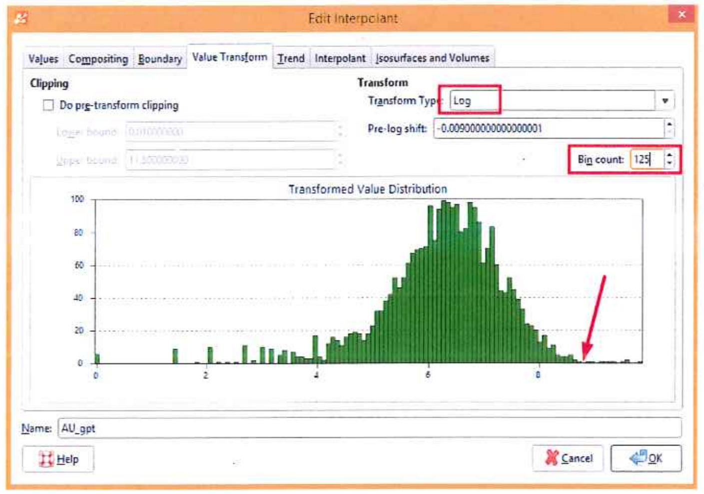
2. Change the **Transform Type** to "Log" by ticking the check box.

The **Lower bound** and **Upper bound** options become available once the **Do pre-transform clipping** check box has been enabled. By setting the Upper bound, all samples with a value greater than the specified Upper bound will be reduced to the value of the Upper bound. This prevents samples with very high grades having an undue influence on the model. As a rule of thumb, an Upper bound can be selected where the histogram starts to break down. A simple method of checking this is by looking at the histogram and finding the value at which the columns start to get gaps between them.

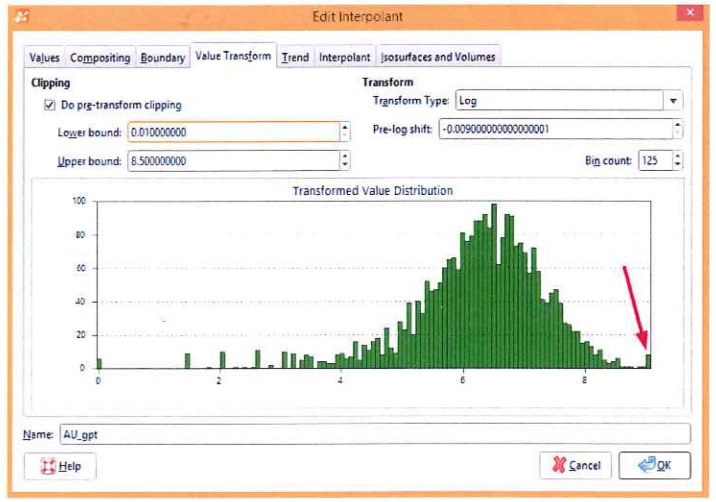
3. Change the bin count to around 125 to split the data into more columns, and note that there is a gap in the data at a value of around 8.5.

4. We will use this value as our Upper bound.

5. Tick the **Do pre-transform clipping** checkbox, and change the value of the **Upper bound** to 8.5.



6. Note that once the Upper bound has been applied, all values greater than 8.5 have been reduced to be equal to 8.5.



7. Click OK again to process the changes before moving on to the next step.

Interpolant tab

1. Double click on the interpolant object to open the **Edit Interpolant** window.

2. Move to the **Interpolant** tab.

There are a number of parameters here which can be set based either on rules of thumb, or by using geostatistical input from packages such as Supervisor or Isatis. For this example, we will look at rules of thumb which work well for a number of examples. The default settings are almost certainly incorrect so the next few paragraphs are important when creating reasonable interpolant models. As you may have figured out by now, Leapfrog is fast at creating models, but that doesn't necessarily mean the models are correct. Understanding how the interpolation works is one of the key topics in the Leapfrog Geo Intermediate course, which is an additional day on top of this course, which covers the fundamentals.

Interpolant

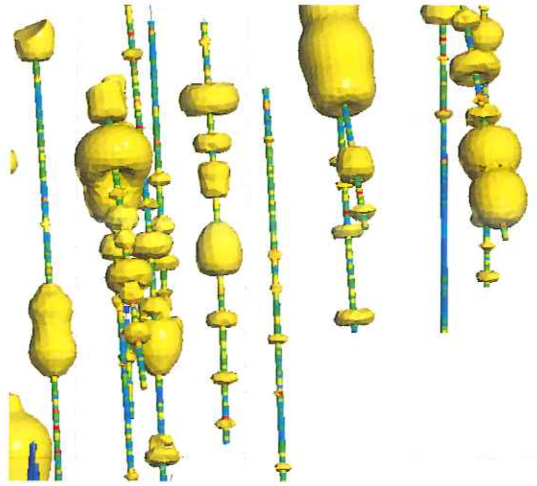
There are two options under the **Interpolant** dropdown - Linear, and Spheroidal. The **Linear** interpolant works well for lithology data, and for quickly visualising data trends. It is not suitable for values with a distinct finite range of influence such as most metallic ore deposits. The **Linear** interpolant assumes that values a certain distance from a particular point have a proportionally greater influence on that point than values further away. The **Spheroidal** interpolant works well when there is a finite range, beyond which the influence of one point upon another should fall to zero. This is the case for most metallic ore deposits.

1. Change the interpolant type to **Spheroidal**, and note that the Interpolant Function in the window changes shape to display the Spheroidal interpolant rather than the Linear interpolant.

Base Range

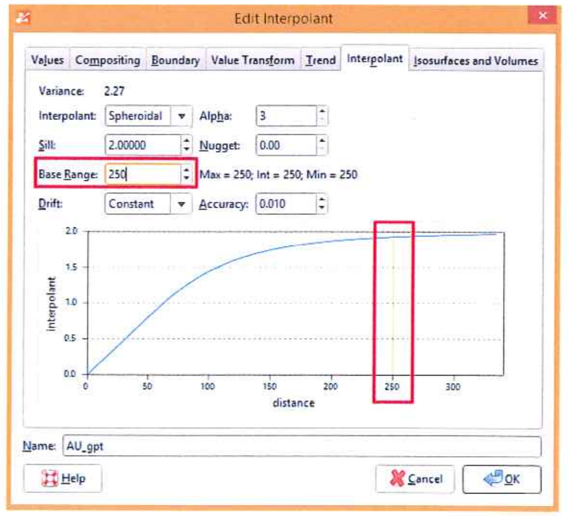
The **Spheroidal** interpolant has a Base Range which represents the distance from the data at which the value equals the sill. As we move away from a specified point, the influence of that point decays in a roughly linear manner up to a distance of around 30% of the range. Past 30% of the Base Range, the influence of the point starts dropping more quickly until it reaches 96% of the value of the Sill.

In simpler terms, the Base Range is the parameter that roughly corresponds to continuity. Leapfrog is essentially creating an isosurface through points of equal value; by increasing the Base Range, the isosurface is able to stretch a further distance between points. The effect of the Base Range can be visualised most obviously when it is too small. For this example, setting a Base Range of around 20 will produce a series of isosurfaces which surround the drilling - these are sometime referred to as "strings of pearls", and are a good indication that the Base Range needs to be increased (it's extremely unlikely that all the drillholes manage to perfectly follow thin pipes of high grade, while missing the surrounding low grade!) See below for an example of this.



As a rule of thumb, the Base Range should be set to 2.0 - 2.5 times the distance between drillholes. In this case, the average distance between holes is around 100 m, so a Base Range of between 200 and 250 should be a good starting point.

1. Change the **Base Range** to 250, and note that the shape of the interpolant function changes to include the range of 250 (which is represented using a vertical yellow line).



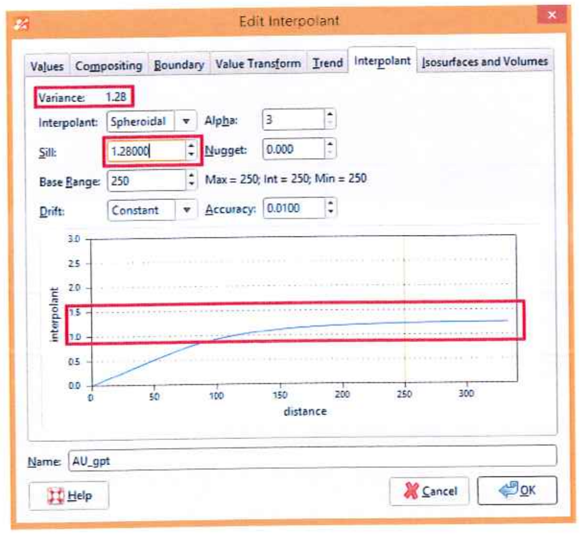
Sill

The **Sill** controls the upper limit of the interpolant function, where there ceases to be any correlation between values. As a rule of thumb, we can use the variance of the data as the **Sill** (listed at the top of the window). This will give a good starting point, but can be refined by running the data through an external geostatistics package such as Supervisor or Isatis. Note that the listed variance will change depending on whether a log transform/value clip has been applied in the previous step.

In this case, the log transform as well as an upper bound of 8.5 have been applied, so the listed variance should be 1.28.

1. Change the **Sill** to 1.28

2. Note that the shape of the interpolant function changes again, and is now limited in the y-direction by the **Sill** (the function approaches the sill, and will be at 96% of the sill when the function crosses the **Range** line).

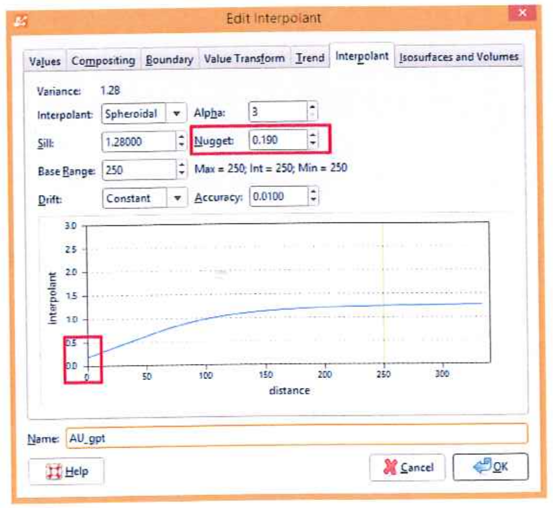


Nugget

The **Nugget** allows for local anomolies in sampled data - where a sample is significantly different to what might be predicted for that point based on the surrounding data. By increasing the value of the nugget, more emphasis is placed on the average values of surrounding samples and less on the actual data point. The nugget can also be used to reduce noise from inaccurately measured samples. The rule of thumb for the nugget changes depending on the deposit type, and geostatistical input is vital. For this deposit (a porphyry gold project), a nugget of 10 - 20% is appropriate. It is important to note that the nugget is a percentage of the sill, so in this case a 15% nugget would be 0.19 (15% of 1.28).

1. Change the nugget to 0.19.

2. Note the change in the interpolant function - the base point (0.0, 0.0) moves up the y axis to equal the value of the nugget.

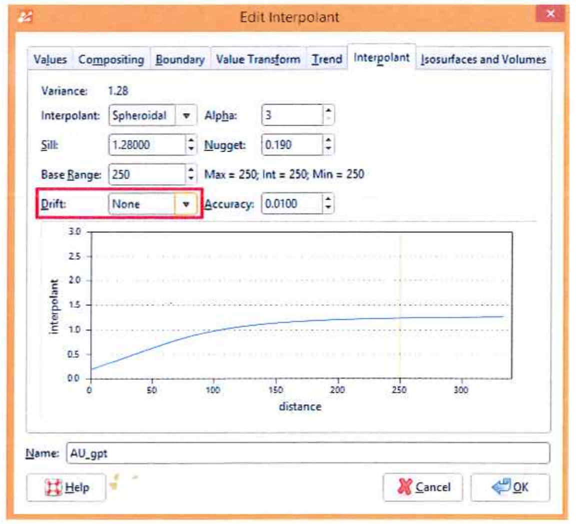


Drift

The **Drift** controls the manner in which the interpolant decays away from the data. A 'constant' drift means the interpolant will decay to the mean value of the data. A drift of 'none' means the interpolant will decay to a value of zero away from the data, so is useful when there are no low grade holes constraining the deposit. A drift of 'linear' means the interpolant decays linearly away from the data.

In this case, as the interpolant model is currently not bounded to any domain (geological, structural, weathering etc), a sensible drift to use will be **None**. This means that as we move away from the data toward the edges of the model, the value of the interpolant will revert to a value of zero.

1. Change the drift to **None**.



Alpha

The **Alpha** determines how steeply the interpolant rises toward the **Sill**. A low Alpha value produces an interpolant function that rises more steeply than a high Alpha value. By looking at the interpolant function while changing the Alpha, we can see that a high Alpha will give points at intermediate distances more influence compared to lower Alpha values. The possible Alpha values are 9, 7, 5 and 3.

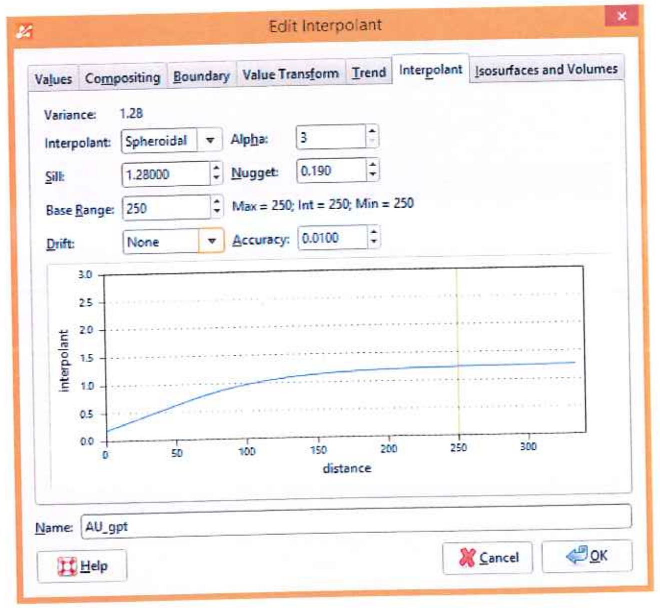
An Alpha of 9 gives the best approximation to the Spherical Variogram, but takes longer to process and in most situations gives a very similar result to using an Alpha of 3.

In this case, we will keep the Alpha at 3 to reduce processing time.

Accuracy

Leapfrog Geo estimates the **Accuracy** from the data by taking a fraction of the smallest difference between measured data values. There is little point in changing the accuracy to be significantly smaller than the errors in the measured data, as the interpolation will run more slowly and will degrade the interpolation result.

The rule of thumb here is to leave the accuracy as it is.



Isosurfaces and Volumes tab

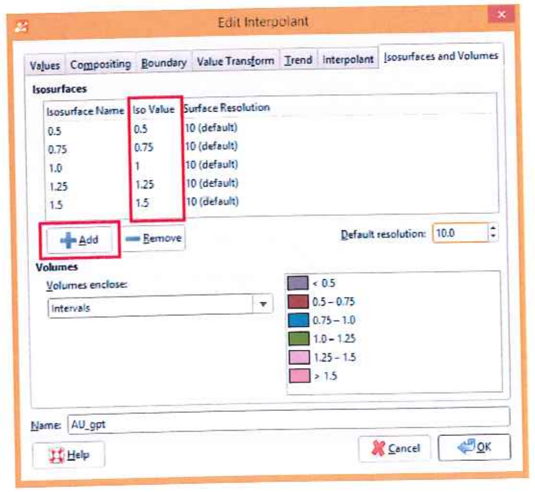
The **Isosurfaces and Volumes** tab lets us choose the values to create isosurfaces at, as well as defining how the isosurfaces create the associated volumes, and the resolution of the isosurfaces.

By default there are three isosurface values, which are at the lower quartile, median and upper quartile of the data which is being used (in this case 0.25, 0.51 and 1.01).

These values are almost always incorrect, but are useful in checking the general shape of the interpolant model. We will go ahead and change them to more reasonable values.

1. Move to the **Isosurfaces and Volumes** tab.

2. Click to highlight one of the default values beneath the Iso Value heading, then click it again to enable the editing mode. Change the values to 0.5, 0.75, 1.0, 1.25 and 1.5. Add additional isosurfaces by clicking the **Add** button.

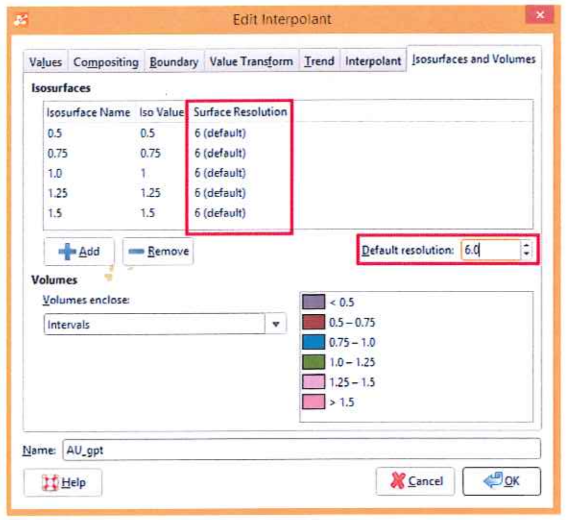


Resolution is important when creating isosurfaces - ideally we would want it to be equal to the composite length (6 m in this case) - a quick test using one of Leapfrog's laptops (16GB RAM, 2.9GHz processor) took 55 seconds to run these isosurfaces at a resolution of 6, but if your laptop is particularly slow it may be worth increasing the resolution to between 12 - 15. This will still give you a reasonable surface, but will process more quickly.

The resolution is important because it determines the size of the triangles making up the surface. If the resolution is 6, the approximate edge length of the triangles will be 6 units in length (remembering that Leapfrog is unit-less). If the edge length of the triangles is 6 units, they will be able to include intervals which are as small as 6 m long. If we were to increase the resolution to 12, the triangles would only be able to include intervals as small as 12 m long (so will miss some of the smaller intervals).

Obviously lower resolution produces a more accurate surface, but can take a lot longer to run. A general guide is that if you half the resolution, the processing time will increase by 4 times.

3. Change the **Default Resolution** to an appropriate value (somewhere between 6 and 20). Note that the resolutions for each surface are set by the default resolution, unless a different resolution for a particular surface is specified.

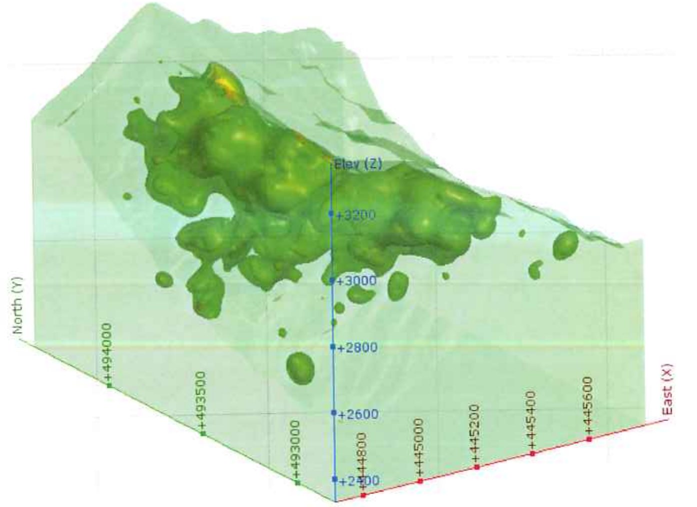


The **Volumes Enclose** dropdown lets you choose from Intervals, Higher Values, and Lower Values.

- **Intervals** will create a series of "donut" shaped shells (in this example the shells will be < 0.5, 0.5 - 0.75, 0.75 - 1.0, 1.0 - 1.25, 1.25 - 1.5, >1.5).
- **Higher Values** will create a series of shells which enclose all higher values within them (in this example the shells will be >0.5, >0.75, >1.0, >1.25, >1.5).
- **Lower Values** will create a series of shells which enclose all lower values within them (in this example the shells will be <0.5, <0.75, <1.0, <1.25, <1.5).

4. In this case we will start by using Intervals, so keep this selected

5. Click **OK**, and wait for the interpolant to finish running. Depending on the resolution chosen, this may take a few minutes.



As a first pass model, this isn't a bad result. We obviously need to add trends to give the mineralisation more of a defined shape, as well as limit the extents of the model to within geological domains (which we made as part of the geological model earlier).

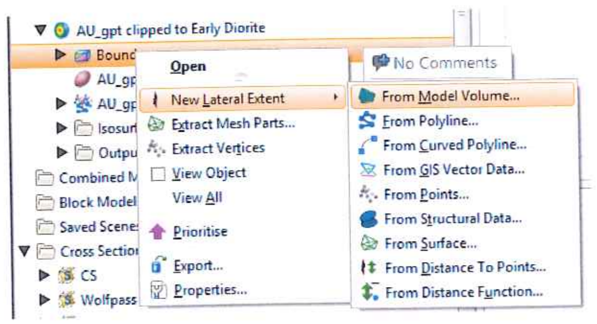
Copying an Interpolant, and Clipping to a Domain

The easiest way to clip an interpolant to a geological domain is to firstly create a copy of it. Once the interpolant has been copied, we can alter some of the parameters in the new interpolant and check the changes against the original interpolant.

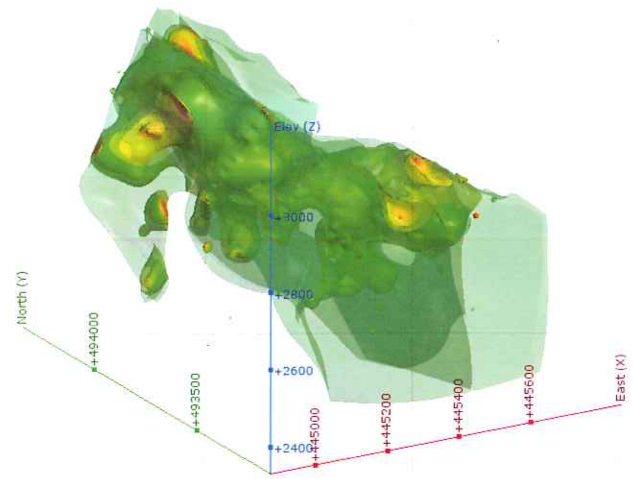
1. To copy an interpolant, right click on the existing interpolant, then click **Copy**.
2. Rename the copied interpolant "Au_gpt clipped to Early Diorite".

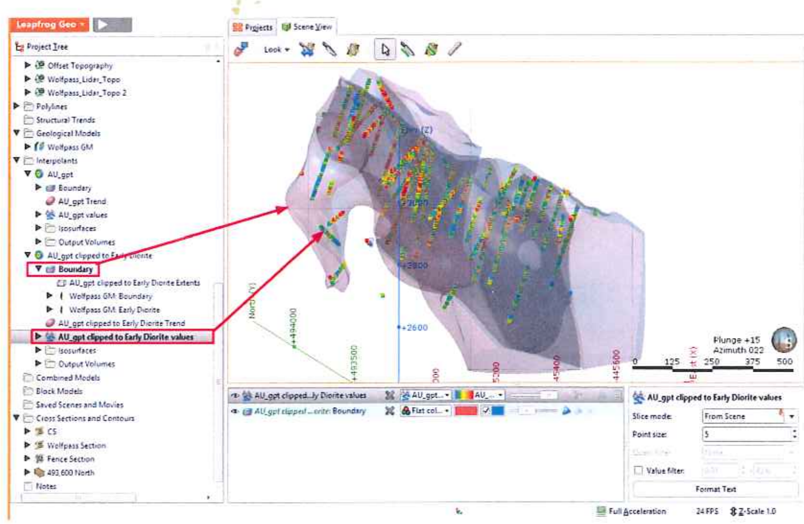
Now that we have a second interpolant model, we can open it up and make appropriate changes to firstly clip the data / surfaces to the Early Diorite, and secondly update the parameters to take into account the changed data (for example, the data will be clipped to the Early Diorite lithology which will change the Upper Bound, Sill, Nugget, and Drift).

3. Expand the new interpolant out until you can see the **Boundary** folder.
4. Right click on the **Boundary** folder, and select **New Lateral Extent** > **From Model Volume**.
5. Select the **Early Diorite** output volume, and click **OK**.



The model will be reprocessed, and will change in two different ways; firstly it clips the isosurfaces to the Early Diorite boundary, and secondly it clips the data to the Early Diorite boundary. We can see the clipped isosurfaces in the image directly below, and the clipped data in the image below that.

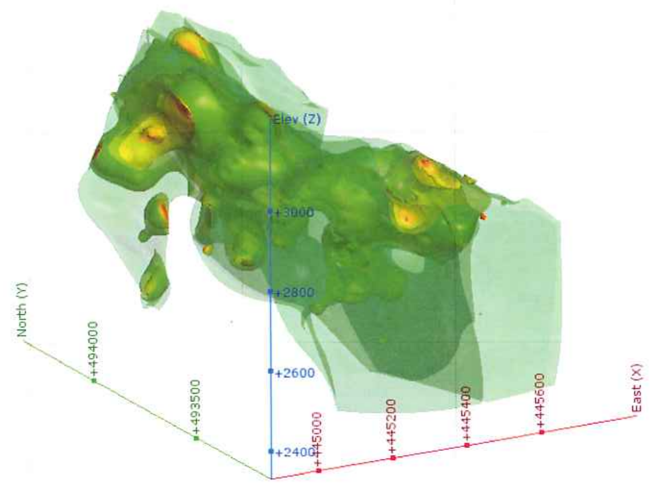




Now that the boundary has been changed, we need to edit the other interpolation parameters as discussed above.

1. Double click on the **Interpolant** object to open the **Edit Interpolant** window. We will again move across each of the tabs making appropriate changes.
2. In the **Values** tab, we can see that the surface filter is set to "Boundary". As the boundary is now set to the Early Diorite volume, this means the surface filter is already using the Early Diorite so we don't need to change anything here.
3. Move across to the **Compositing** tab. Again, we have made all the changes we require so we don't need to change anything here. This is the same with the **Boundary** tab, and the **Trend** tab.
4. The **Interpolant** tab is the first which contains fields we need to change.
5. Using the rules of thumb above, make appropriate changes to the **Sill**, **Drift**, and **Nugget**.

6. The **Isosurfaces and Volumes** tab should be correct as it is, so we can click OK and let the interpolant reprocess.



Adding a Structural Trend to an Interpolant Model

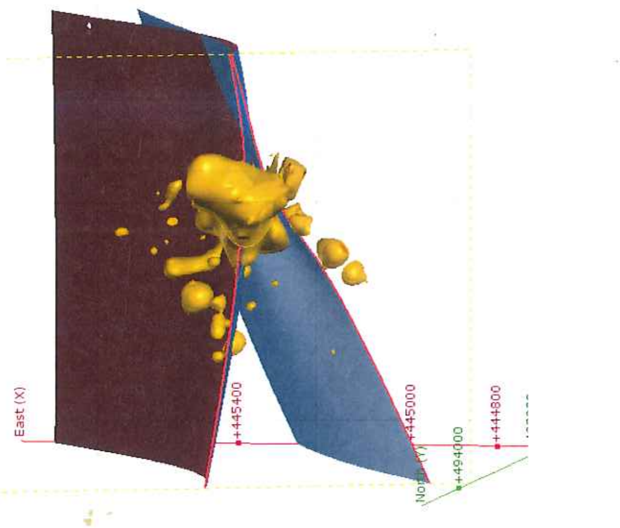
Now that we have changed the parameters of the Interpolant Model to give a reasonable first pass, the final step is to apply a trend in a similar manner to applying a trend to intrusional surfaces. We will have looked at global trends previously in this course (even though we were applying them to Intrusional Surfaces), but for this model global trends won't provide us with enough detail. When this is the case, the answer is to look at Structural Trends. There is not enough time in this course to look at Structural Trends in too much detail, but a more comprehensive look at structural trends is available in the **Leapfrog Geo Intermediate** course.

Creating a Structural Trend

To create a Structural Trend, we need to create meshes which represent the direction of the trend. We can use as many meshes as required, and apply different strengths and ranges to each. A common example of where this might be useful is when dealing with multiple zones of high grade. This could be a syngenetic deposit with an epigenetic zone formed by later weathering, or it could be two separate intrusions each with a different plunge and strike.

There are many methods we can use to define meshes, but an easy option is by using Curved Polylines. Take a look at the existing Interpolant Model and pick out a few trends - two possibilities are shown below. Note that

it can be easier to pick trends by hiding the initial isosurfaces and viewing the points which are being used to make the isosurfaces. Once the points are in the scene, use the value filter in the properties panel to slowly hide the lower grade points - as you do this, the trend often becomes clearer.



1. Create two or more meshes defining your trends. In the example above, there is one named "Vertical Trend" and one named "Dipping Trend". Both were made using Curved Polylines.

2. Once the meshes have been created, right click on the **Structural Trends** folder and select **New Structural Trend**.

3. Click **Add**, near the bottom of the window, and select the two meshes that you created in the previous step.

4. The three options at the top of the window are **Non-decaying** (default), **Blending**, and **Strongest Along Meshes**.

- **Non-decaying** assumes the strength of the trend won't decay away from the meshes.
- **Blending** takes any one point in the model, then uses a combination of the multiple meshes to define the direction and strength of the trend at that point
- **Strongest Along Meshes** takes any one point in the model, then uses the closest mesh at that point to define the direction and strength of the trend at that point.

For this example, the most reasonable type of trend to use is **Strongest Along Meshes**.

5. Select **Strongest Along Meshes**, and the **Range** option becomes available.

- The **Range** defines the perpendicular distance away from each mesh that the mesh has influence on
- The **Strength** defines the strength of the trend (a strength of "5" would be the equivalent of a 5:5:1 ratio in a global trend)
- As we move further away from the mesh, the **Strength** decays until the **Range** is met (unless Non-decaying is selected, in which case the strength does not decay away from the mesh).

6. Keep the Strength and Range at 5.0 and 100 for both meshes, and click **OK**.

There is a short description at the bottom of the window giving instructions to turn the total Units into grams. In this case, we can look at the average density of the Early Diorite lithology, multiply it by the total Units, then divide by 31.1 to give total ounces of Gold.

- Total Units is 109,158,221
- Density of Early Diorite is 3.05 (which can be found in the histogram tab of the merged table)
- (109,158,221 x 3.05) / 31.1 = 10.7 million ounces of Gold within the Early Diorite.

Note that we haven't included a cut off, so we are calculating the grade within the entire Early Diorite domain. There are several things we could do to get a more constrained result, including limiting the interpolant to within a certain distance to the drilling (New Lateral Extent > From Distance Function), using an Indicator Interpolant as a boundary (see below), and creating further refined geological domains (using Refined Models).

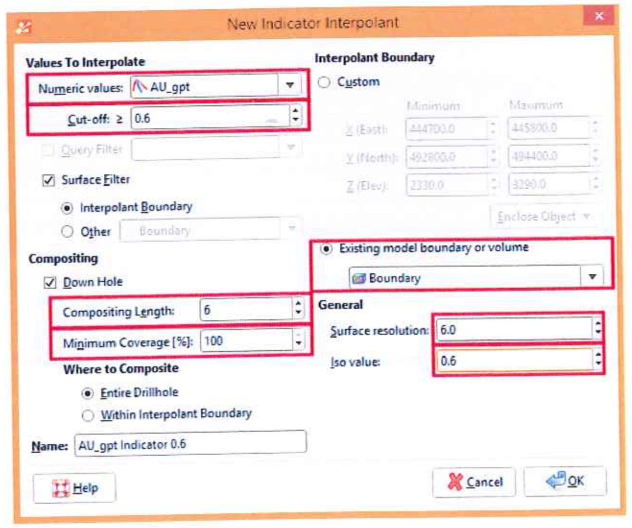
Indicator Interpolants

In simple terms, Indicator Interpolants allow the user to specify a cut-off grade, then assign either a "0" (for grades below the cut-off) or "1" (for grades above the cut-off). Once the cut-off grade has been defined, a surface is created around the "1" values. Small, uneconomic volumes can be automatically removed using a Volume Filter, and a Statistics tab gives detailed statistics on how the model overlaps with other data in the project.

Once the volume has been created, it can be used as a boundary, or region of interest within which further processing can be carried out.

In this case, we will define our cut-off at a value of 0.6.

1. Right click on the **Interpolants** folder and select **New Indicator Interpolant**.
2. Change the **Numeric Values** option to AU_gpt.
3. Change the **Cut-off** to 0.6
4. Keep the **Surface Filter** as the Interpolant Boundary.
5. Change the Compositing so it is the same as for our Interpolant (composite length of 6, minimum coverage of 100%).
6. Change the Interpolant Boundary to the existing Geological Model boundary, to limit the Interpolant to be underneath the topography.
7. Drop the Surface Resolution to 6, and change the Iso value to 0.6.



8. Click **OK**, and the initial interpolation will be processed.

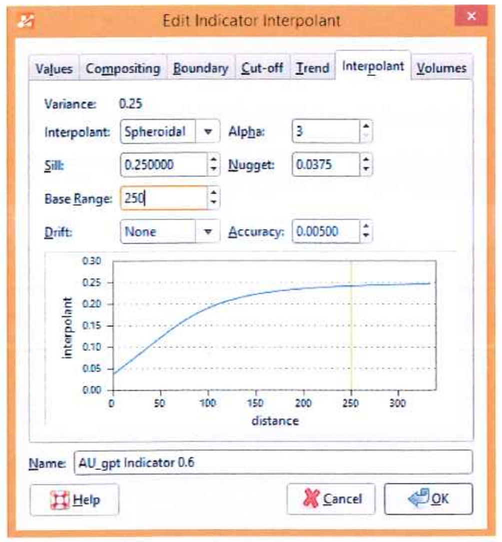
As was the case with the Interpolant Model, we will need to go ahead and make appropriate changes to the default Indicator Interpolant Model. This time we already know most of the rules of thumb, so they will be moved through quickly.

1. Double click on the **Indicator Interpolant** to open the **Edit Indicator Interpolant** window.
2. The **Values** tab can be left without change, as the surface filter is already using the boundary (which is in this case the Geological Model boundary).
3. The **Compositing**, **Boundary** and **Cut-off** tabs can be left as they are as we have already made the appropriate changes.
4. In the **Trend** tab, we can add our Structural Trend following the same process as for the Interpolant Model (tick the check box, and make sure the correct Structural Trend has been selected).

5. There are a number of changes we need to make in the Interpolant tab, following the same rules of thumb as earlier.

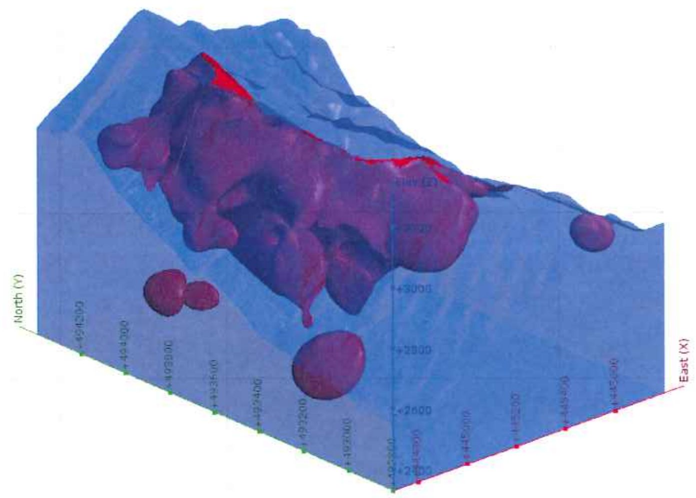
- Change the **Interpolant** to be Spheroidal.
- Change the **Sill** to equal the variance (0.25 in this example)
- Change the **Base Range** to be 2.0 - 2.5x the drillhole spacing (250 in this example)
- Change the **Drift** to None
- Keep the **Alpha** at 3
- Change the **Nugget** to 15% of the Sill (0.0375 in this example)

leapfrog
FAST, DYNAMIC GEOLOGICAL MODELLING



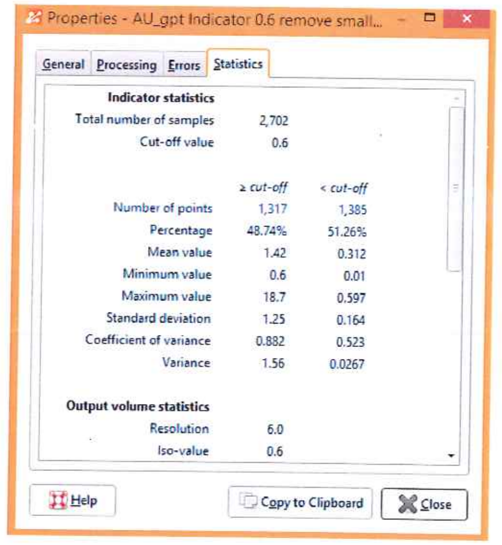
6. Move across to the **Volumes** tab, and select the Volume Filtering check box. Change the **Discard volume parts smaller** than option to 100,000 units cubed (this will remove all volumes smaller than 100,000 units cubed).

7. Drag the Indicator Interpolant into the Scene to view it.



8. Right click on the Indicator Interpolant and click **Properties**, then navigate to the **Statistics** tab. Statistics relating to the Indicator are listed in the window, and can be copied to the clipboard for importing into Microsoft Excel or a similar package.

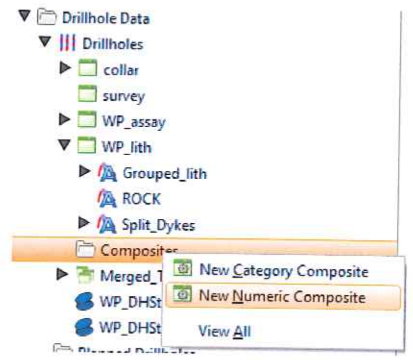
leapfrog
FAST, DYNAMIC GEOLOGICAL MODELLING



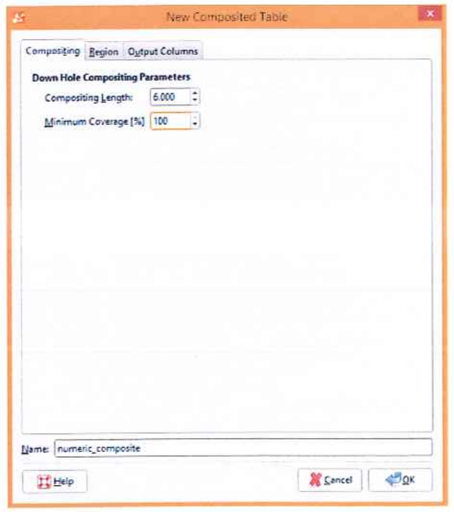
Compositing Outside the Interpolant Model

It is possible to create a set of composited drillholes outside the Interpolant Model. To do this, we go directly to the drillholes folder. This folder gives us more options, including whether to composite the entire drillhole, or only within a particular lithology. Once the composite has been completed, it can be used to create an Interpolant Model.

1. Right click on the **Composites** folder, which is under the **Drillholes** folder in the project tree, and select **New Numeric Composite**.

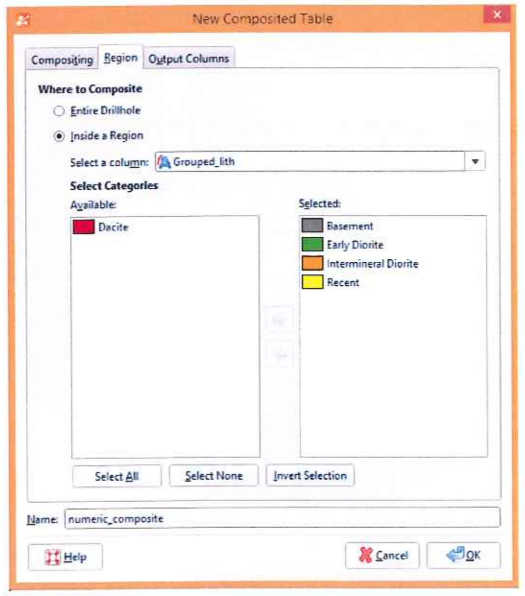


2. We will follow similar rules to earlier; changing the Composite Length to 6 and the Minimum Coverage to 100%.



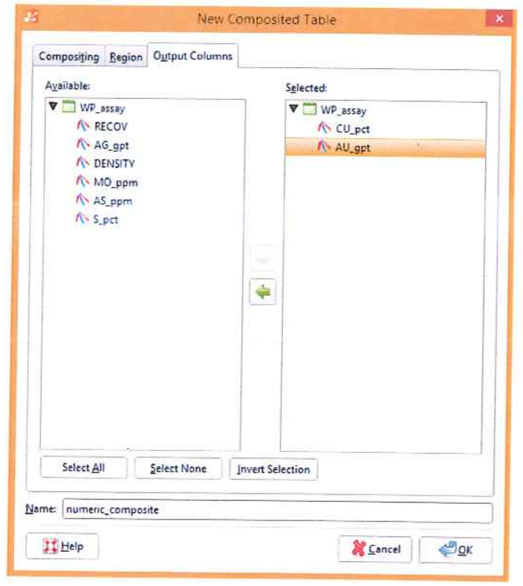
3. Click on the **region** tab, and change the selected option to 'inside a region'.

4. In this case we are looking at this for a demonstration, so there is no need to go any further, however it is worth noting that we can choose to run a numeric composite within a specific region, where a region is determined using categorical or lithological intervals from any drill hole column.



5. Click on the **output columns** tab, and select copper and gold, then click **OK**.

leapfrog
FAST, DYNAMIC GEOLOGICAL MODELLING



The composited table will be available under the **Composites** sub folder, which is beneath the **Drillhole Data** folder in the **Project Tree**.

Exercise: Create an Interpolant Model for the Copper

Use the rules of thumb discussed above to create an Interpolant Model for the Copper.

leapfrog
FAST, DYNAMIC GEOLOGICAL MODELLING

Session 15: Wolfpass Cross Sections

Contents

Goals

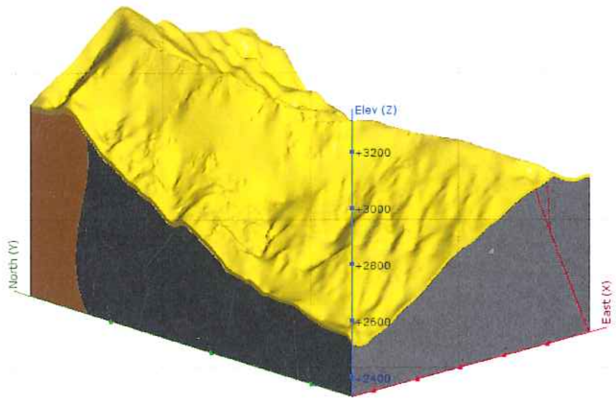
In this session, we will go over the creation of cross sections in Leapfrog Geo, as well as showing a number of related features

For this session open the Wolfpass model.

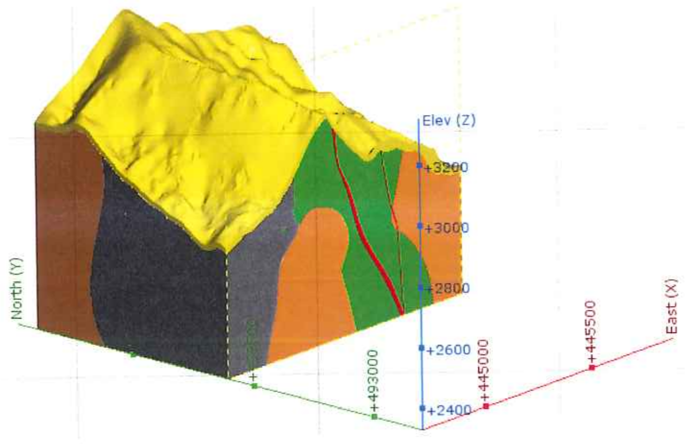
Setting up a cross section

Cross sections can be accessed in the **cross sections and contours** folder in the project tree. Before creating a cross section, it is best to bring a model into the scene. This means the extents and location of the default section will be created in the correct general area. Once the location of the cross section has been set up, any model or surface or drillhole in the project can be evaluated against the cross section. The steps required to set up a cross section are listed below.

1. View a model in the scene. This could be a geological model, an interpolant model or another set of surfaces.



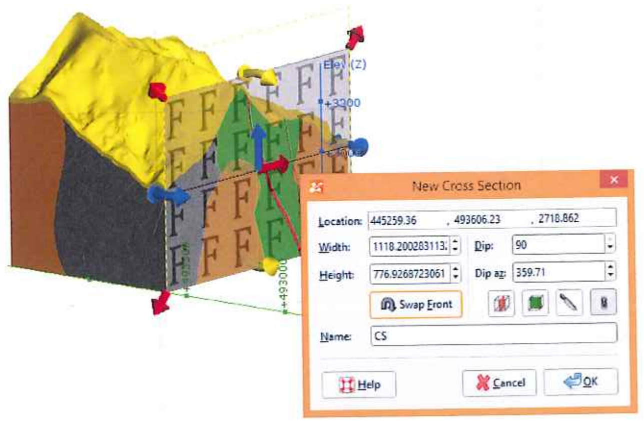
2. Put a slice through the model where you would like the cross section to be located.



3. Right click on the **cross sections and contours** folder in the project tree

4. Select **new cross section**

5. The **new cross section** window will appear in the scene, as well as a plane which can be manipulated to control the size and location of the cross section. The plane will be positioned on the sliced section of the model.



6. If the location of the cross section has been decided upon, the exact location can be entered in the window. If not, the plane in the scene can be moved to a suitable location.

7. The size, as well as the dip and azimuth of the cross section, can be altered both in the window and by using the plane in the scene.

8. There are shortcut buttons in the window which allow the section to be set to either an north-south or east-west orientation.

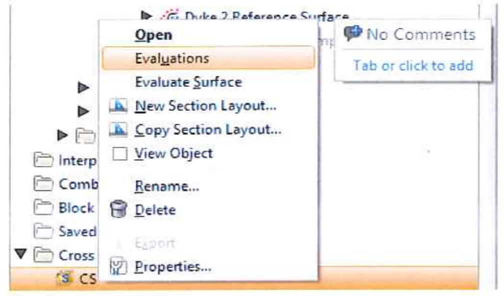
9. The front of the cross section can be changed in the window by selecting the **swap front** button. When clicked, the 'B' for back and 'F' for front will be updated on the plane in the scene. This is important, as it controls which side of the plane is in view when the cross section is being edited.

10. Once the section has been positioned as required, click **OK** to create the cross section. This will appear in the cross sections and contours folder, and can be edited by double clicking on it in the project tree.

Evaluating a geological model against a cross section

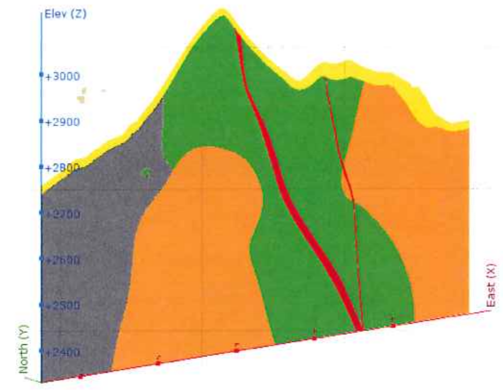
Once the location and size of the cross section have been set up, we can evaluate models and surfaces against it.

1. Right click on the cross section in the project tree, and select **evaluations**.



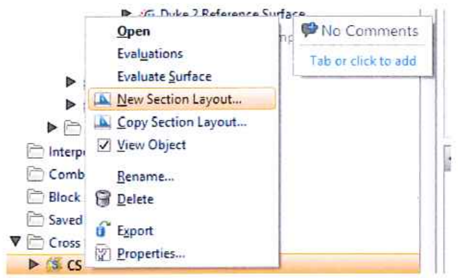
2. Choose the geological model to be evaluated against the cross section by moving it across to the window on the right, then click **OK**. The model will take a few seconds to evaluate onto the section.

3. Drag the evaluated cross section into the scene to view it. By default the display will be flat colour - this can be changed by clicking the display dropdown in the scene list, and changing the display from flat colour to Wolfpass GM.



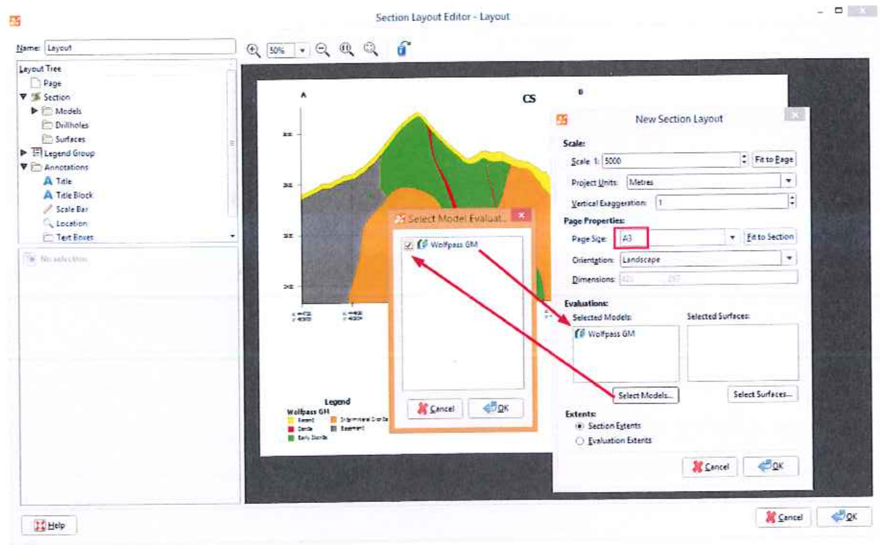
The next step is to create a new section layout and make changes as required to the cross section. This will let us add features such as text, drilling, legends and scale bars to the section.

1. Right click on the cross section in the project tree, and select **new section layout**.

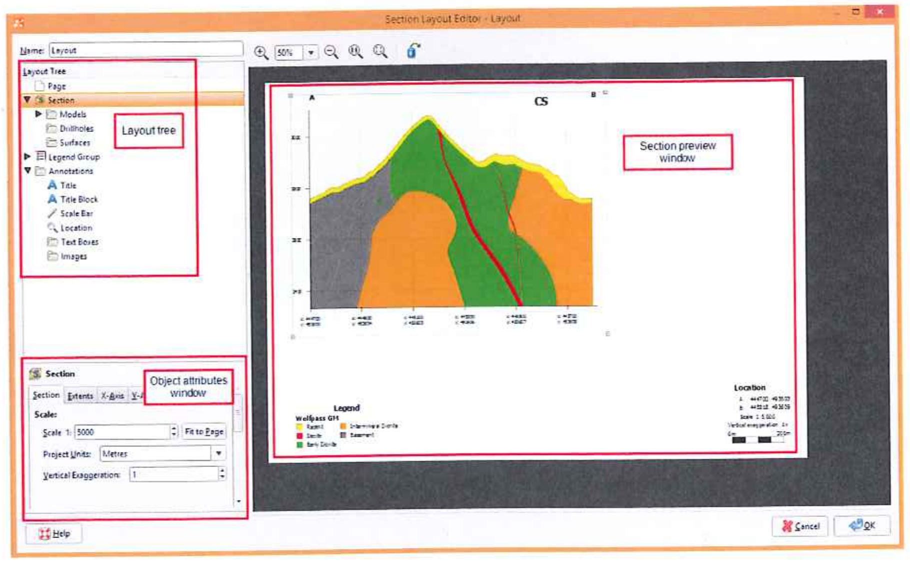


2. The section editor window will appear in the background, as well as a window which allows the basic set up of the section to be changed.

3. In the initial window, change the page size to A3 and add the evaluated model to the section



4. Click **OK**, and you will be able to see the **Section Layout Editor**.



The structure of the editor is similar to Leapfrog; the **layout tree** on the left side of the window shows the different layers which can be added to the cross section, the **object attributes window** displays additional options for the selected object in the layout tree, and the **section preview window** shows a dynamic view of the section, which is updated whenever changes are made in either of the other windows.

The three main folders in the layout tree allow the section, legend group, and annotations to be changed, as shown below.

Changing the set up of the page

1. Click on the **page** folder at the top left of the layout tree.

2. The available options are listed in the object attributes window, including page size, orientation, dimensions (if page size is set to 'custom'), and margins.

3. As each option is changed in the object attributes window, the view is updated live in the section preview window.

Organising the section

The **Section** folder lets us add models, drillholes and surfaces to the cross section.

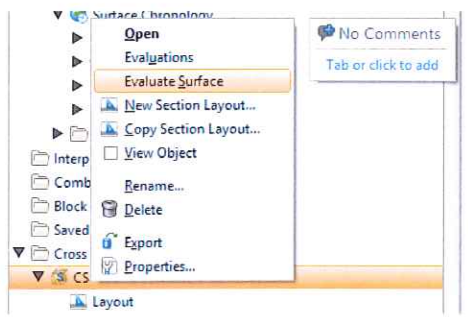
- **Models** include any geological, interpolant, combined or refined models.
- **Drillholes** allow traces within a specified distance from the section to be added, then up to three types of associated data (categorical or numeric) can be viewed and filtered on the drillholes in the section.
- **Surfaces** include any surfaces in the model such as meshes, faults, boundaries, topographies and volumes.

Models and **Surfaces** need to be evaluated against the cross section in the project tree before being added to the cross section in the section layout window. Previously we chose to evaluate the Wolfpass GM against the cross section, so this is the only model available to view. We haven't evaluated any surfaces or drillholes against the cross section yet.

Adding surfaces to the section

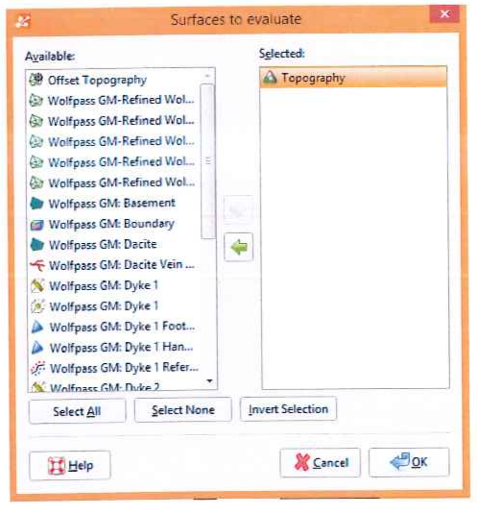
To evaluate a surface against the cross section, we need to go back to the project tree in Leapfrog.

1. Right click on the cross section in the project tree, and select **Evaluate Surface**.



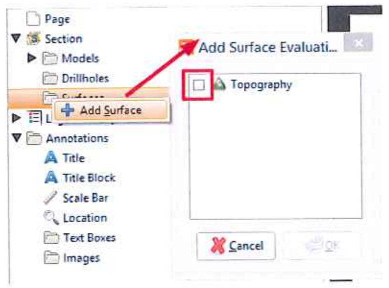
All surfaces in the project are listed. We will evaluate the topography surface against the cross section.

2. Move the Topography across to the "Selected" window on the right, either by double clicking on the Topography, or by clicking and dragging it across.



3. Click **OK**, and the topography surface will be added to the cross section. It won't be displayed yet - this is the next step.

4. To display the topography in the section now that it has been evaluated, right click on the **Surfaces** folder in the **layout tree** and select **Add Surface**.



5. Tick the check box, then click **OK**.

By default the surface is very thin so is difficult to see. Next we will change the colour and thickness of the topography surface.

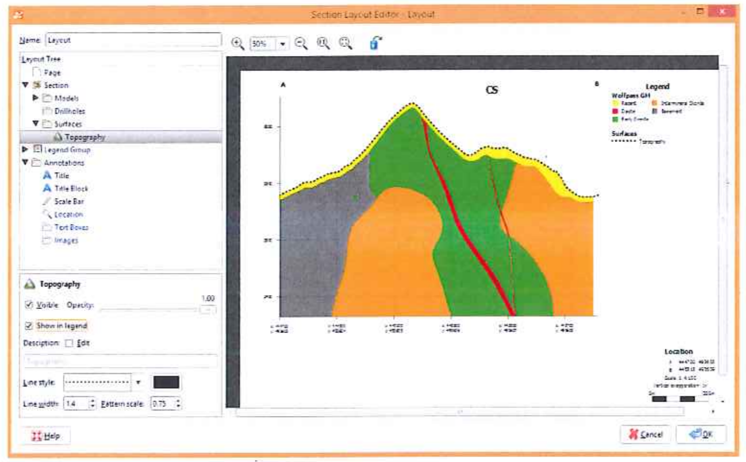
6. Expand the **Surfaces** folder out so you can see the topography surface.

7. Click once on the topography surface to highlight it - this also brings up the **object attributes window** for the topography surface.

At the bottom of this, there are options to change the line width, style, colour, and the pattern scale (depending on the style of line chosen).

8. Change the line width to around 1.5, and the line scale / colour as appropriate.

9. Tick the **Show in legend** check box.



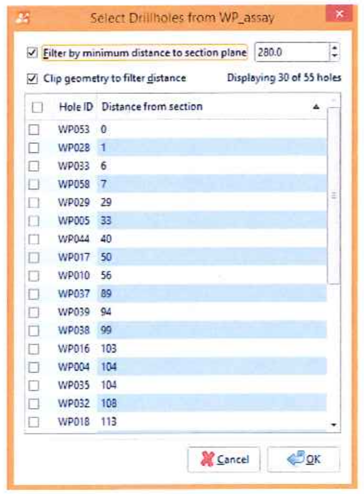
For this example we won't add any additional surfaces, but there are a number of useful surfaces which could be added - a weathering surface, planned pit shell, or existing / planned underground workings.

Adding drillholes to a section

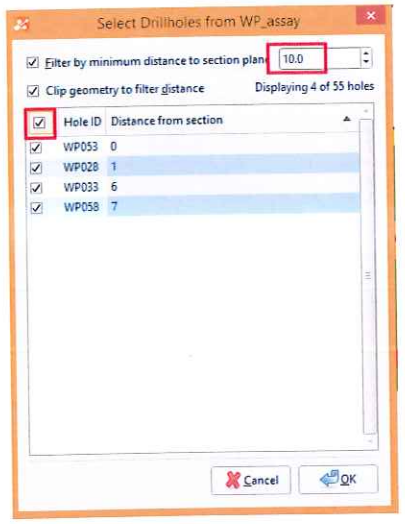
To add drillholes to a section, we can go straight to the drillholes folder in the cross section layout tree, rather than having to go back to the project tree and evaluate them against the section.

1. Right click on the **drillholes** folder in the cross section layout tree and select **Add Drillholes**.

2. Select WP_assay, and click **OK**.



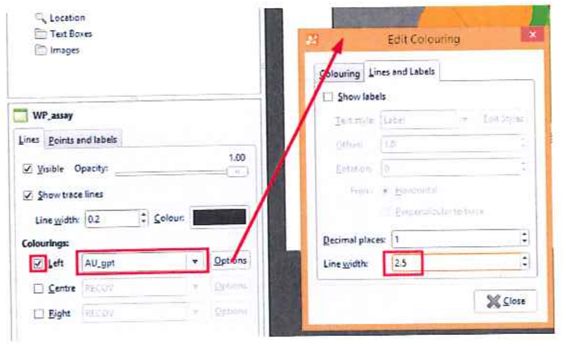
3. Drillholes are able to be filtered based on minimum distance from the section plane. In the top right corner of the window, enter the distance you would like to filter to. For the example above, a distance of 10 m is appropriate. Once the filter has been applied, tick the check boxes to the left of each of the drillholes to add them to the cross section.



4. Click **OK**, and the drillhole traces will appear in the cross section.

To change the display along the traces, we need to look in the object attributes window in the bottom left corner. Up to three different properties can be displayed, along the left, centre and right of the trace.

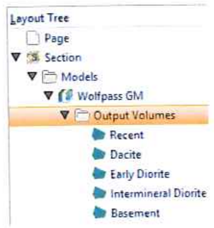
1. Tick the "Left" checkbox, and change the display to "AU_gpt".
2. Click the **Options** button to the right of the dropdown box.
3. Add a value filter with a lower cut off of 0.5.
4. Move across to the **Lines and Labels** tab, and increase the line width to 2.5. In this case it is best to keep the labels off, as there are too many intervals so the text overlaps itself.



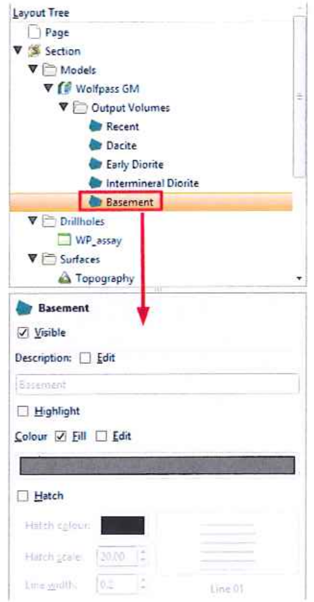
Editing the models in a section

Now that we have added the geological model, drillholes and the topography surface to the section, we will edit the display of the model by changing colours, transparency and hatches.

1. Go to the models folder in the layout tree, and expand the triangle out until you can see each volume.

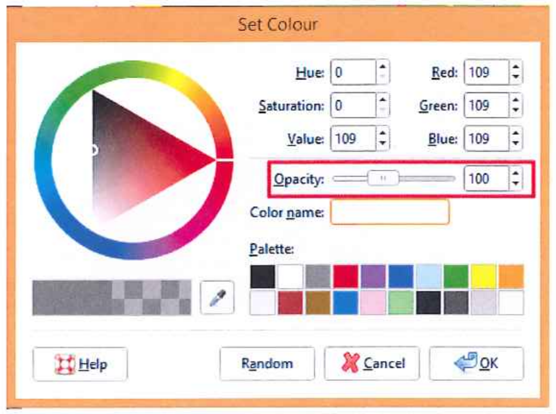


2. Click on the "Basement" volume to bring up the basement editing features in the object attributes window.

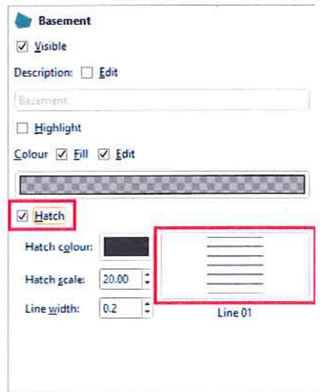


In this window we can change whether the basement is visible, its label in the legend, whether to highlight it, and the colour / hatch of the volume.

3. Keep the basement visible and the description as default.
4. Tick the "Edit" check box, then click on the colour swatch to edit the colour. Keep the colour the same, but change the opacity to 100.



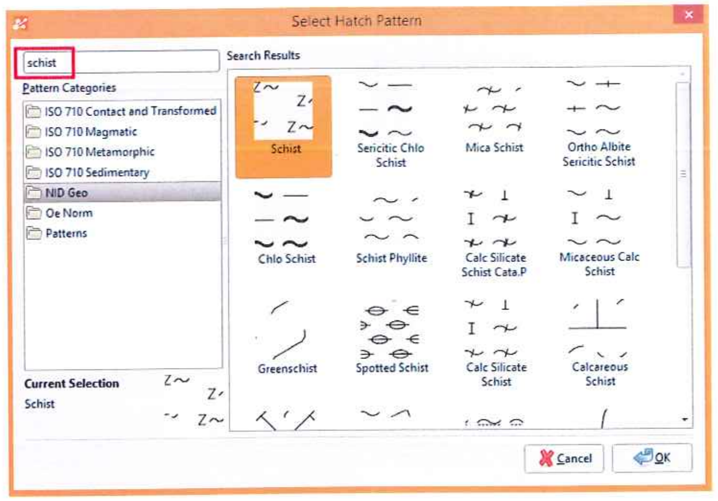
5. Tick the "Hatch" checkbox, then click on the hatch swatch to edit the hatch pattern.



There are a number of common rocktypes that can be searched for in the search window, otherwise you can look through each folder to find an appropriate rock type. For the basement in this model, we will use a schist hatch pattern.

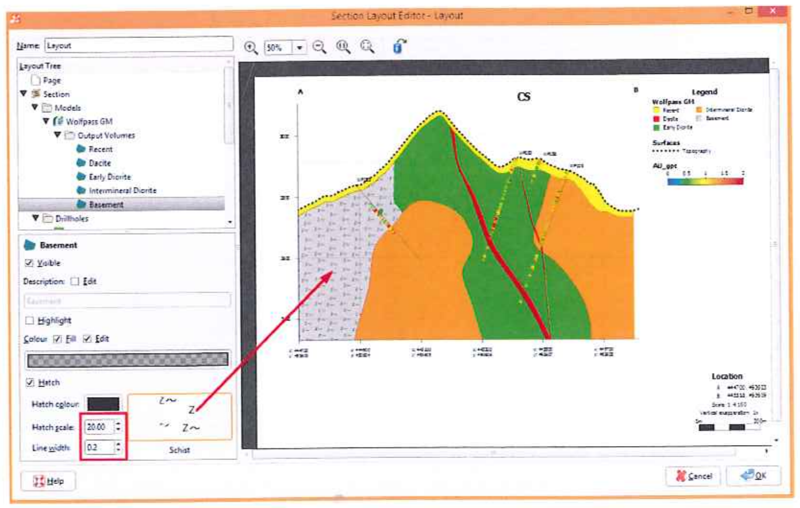
6. Type "Schist" into the search box and 21 different options appear.

7. Choose the first by clicking on it, and click **OK**.



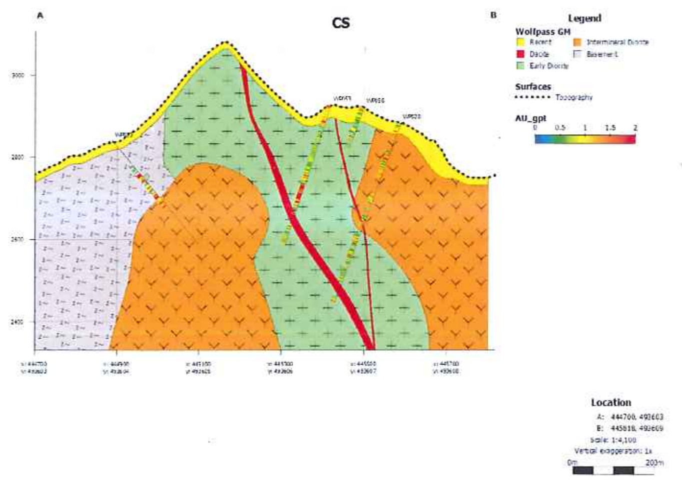
The cross section will be updated to include the hatch pattern.

8. To change the scale and line width of the hatch pattern, use the options in the object attributes window.

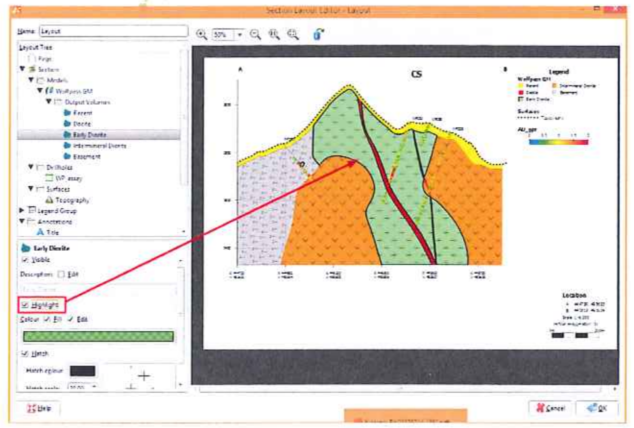


9. Add appropriate hatch patterns to the other lithologies in the section.

leapfrog
FAST, DYNAMIC GEOLOGICAL MODELLING



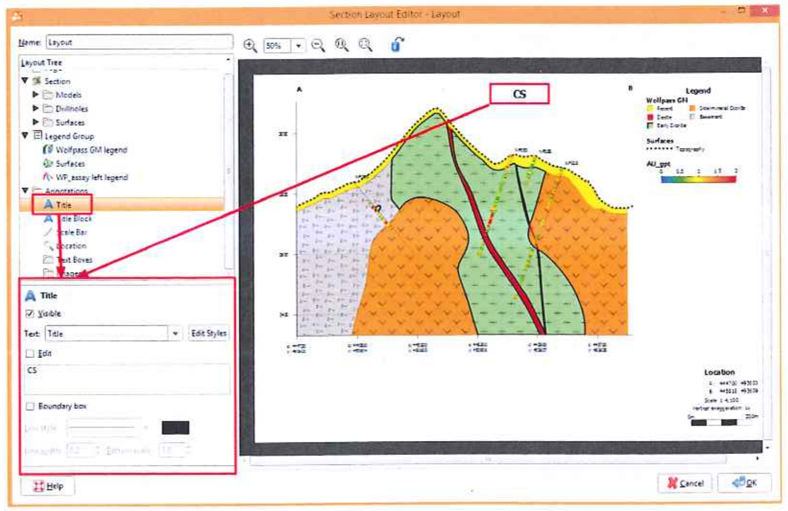
10. To highlight a particular lithology with a bold outline, we can click on it in the layout tree, then tick the "Highlight" check box.



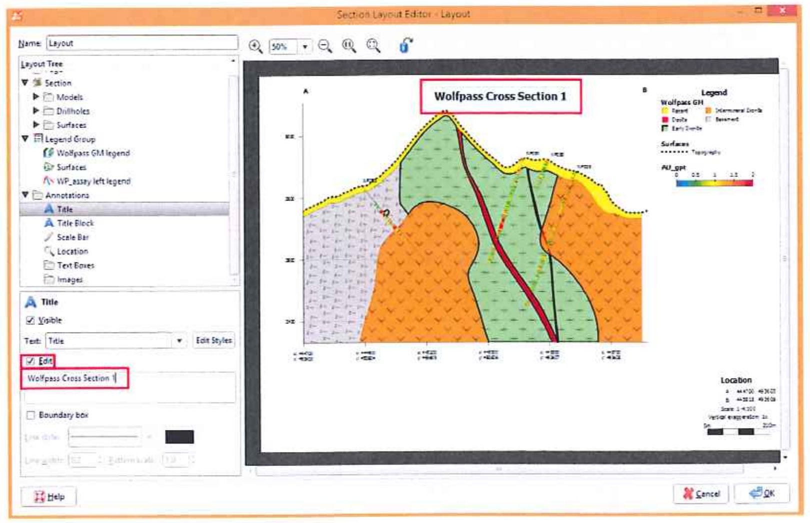
Annotations

Next we will edit the annotations, including the title and title block, as well as scale bar, location labels, text boxes and images.

To edit the title, there are two options. We could either click on the title in the section layout view, or click on the title in the layout tree. Either will highlight the section and display the options in the object attribute table.



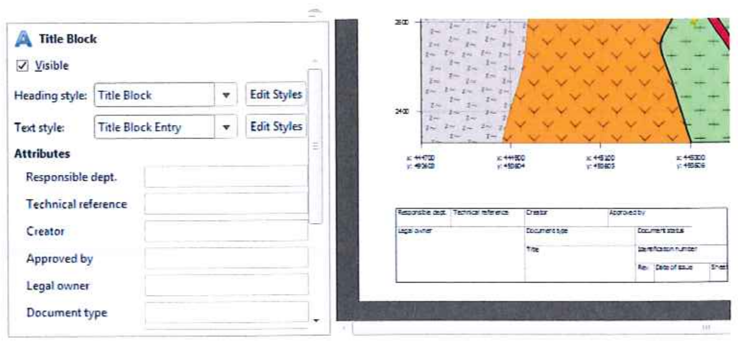
1. To edit the name of the title, tick the "Edit" checkbox which activates the text box editor.



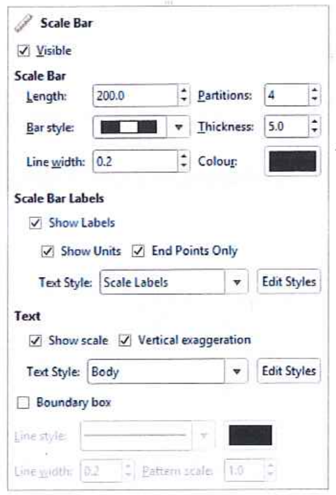
2. Enter an appropriate title for the cross section.

3. Choose whether to display a boundary box around the title, and change the line style, width and colour.

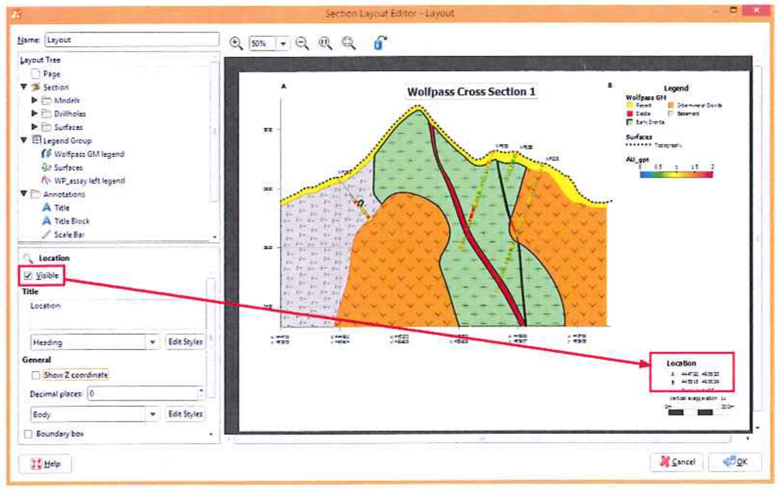
There is one title block available, which can be seen in the section by ticking the "Visible" check box. If required, enter the attributes for each part of the title block and they will be updated in the section layout.



Change the scale bar options as required.



Choose whether or not to view the location of the section

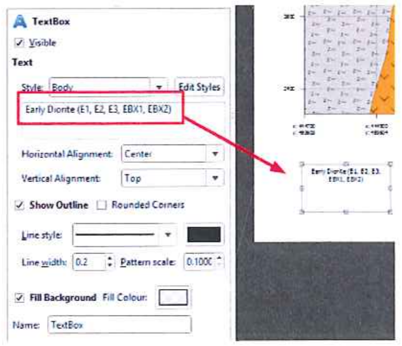


Adding a text box

1. To add a text box to the cross section, right click on the **Text Boxes** folder and select **Add Text Box**.

2. Once the text box is in the cross section, it can be moved by clicking and dragging.

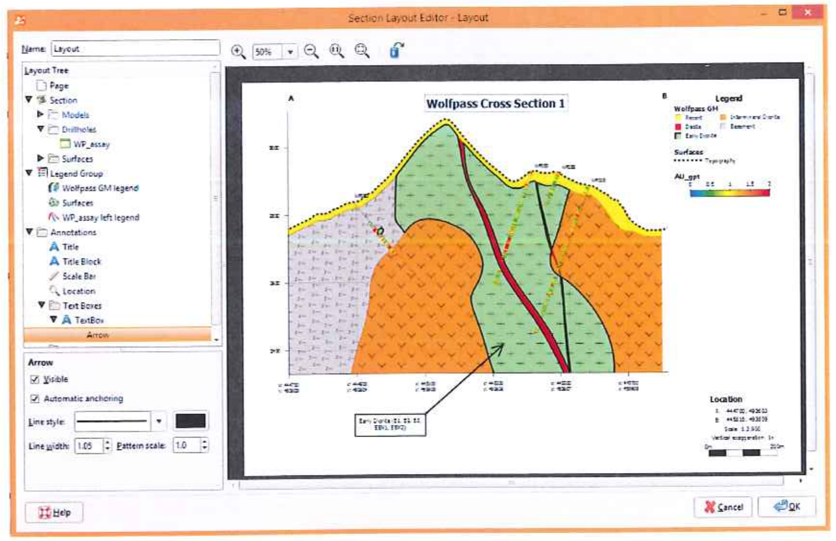
3. To enter text, go to the object attributes window and type in the blank box.



4. Now to add an arrow coming from the text box, right click on the text box in the layout tree and click **Add Arrow**.

An arrow will appear on the text box, which can be dragged around using the node on the end of it.

5. The "Automatic Anchoring" option chooses whether the text box end of the arrow moves around the text box dynamically, or whether it is anchored in one position to the text box. The line style and colour can also be changed.



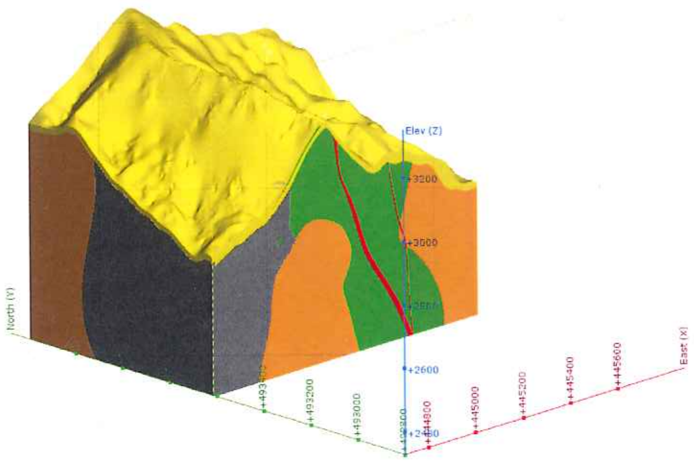
The last option we will look at is the images folder.

Adding an image

Images can be a useful way to describe the location in space of the section. There are multiple possible options, but two common options are to take a screen shot in plan view showing the location of the section, and take a screenshot in oblique view showing the location.

1. To save the section layout, click OK in the bottom right corner of the section layout window.

2. Set the scene up so you can see the location of the cross section in relation to the geological model, and take a screen capture. You can either use the built-in "Render Image" option in Leapfrog (under the orange **Leapfrog Geo** menu), or use any screen capture software such as Snipping Tool, which is installed for free with Windows.



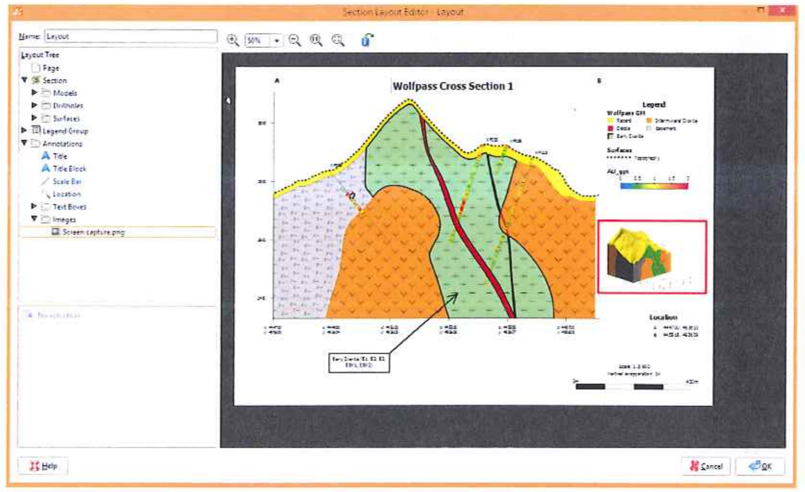
3. To add the image to the cross section, go back to the Cross Sections folder in the project tree and double click on the layout we were working on a few minutes ago. This will open the section as it was when we clicked **OK**.

4. Go to the **Images** folder, right click on it, and select **Add Image**.

5. Navigate to the screen capture that was taken in step 2, and click **Open**.

leapfrog
FAST, DYNAMIC GEOLOGICAL MODELLING

6. The image will be added to the cross section, and can be repositioned and resized as required.

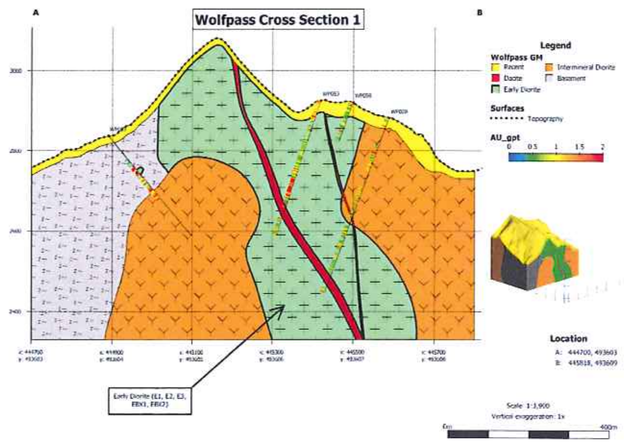


Saving and exporting cross sections

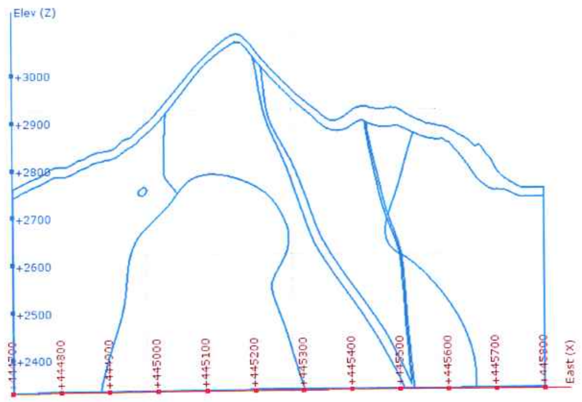
Once the cross section has been set up as required, click **OK** in the bottom right corner of the section layout editor. This will save the layout under the section in the project tree. To access it again, double click on the layout in the project tree.

The section can be saved in *.pdf, *.svg and *.png formats by clicking on the button to the right in the image below.

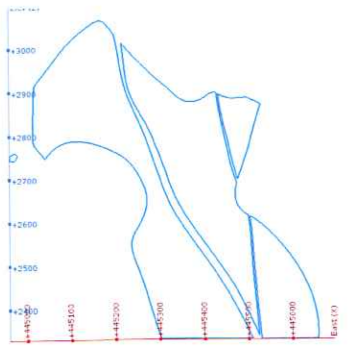
Once the section has been saved in the required format, click **OK**. An option asking whether you would like to open and view the exported section will appear.



The line work for the cross section can be exported as a dxf by right clicking on the cross section and selecting export, choosing which model to evaluate on the section, and clicking **OK**.



In this case the geological model line work has been exported, which means each volume on the cross section will be exported as a separate dxf line. These can be viewed together (as above) or separately (as below):



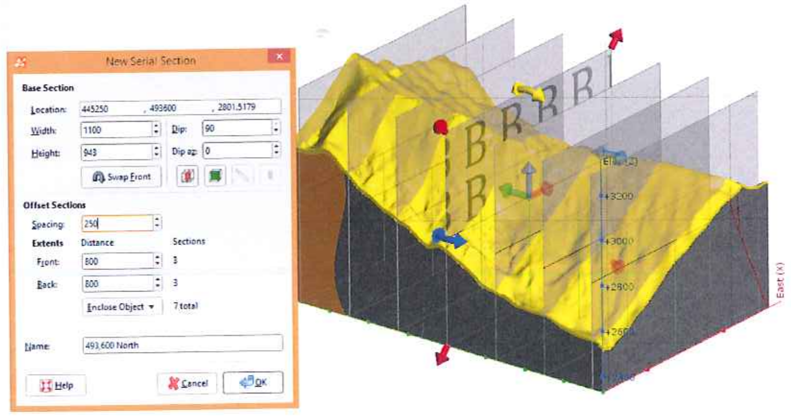
Creating serial sections

Serial sections at specific spacings can easily be created in Leapfrog.

1. Right click on the cross sections and contours folder in the project tree in Leapfrog, and select **New Serial Section**.

2. Choose the location of the base section either by using the arrows in the scene, or typing directly into the **New Serial Section** window.

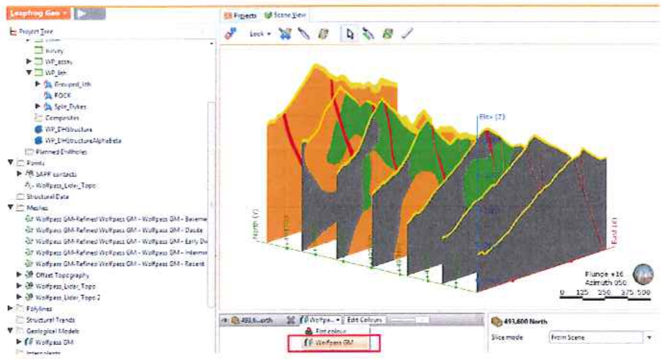
3. Change the spacing between each section to an appropriate distance, and click **OK**.



4. A new serial section will appear under the **Cross Sections and Contours** folder.

5. Right click on the serial section and select **Evaluations**. Select a model to evaluate on the sections, and click **OK**. Depending on the number of sections making up the serial section, the model may take a few minutes to evaluate.

6. Drag the top level serial sections object into the scene, and make sure the view is changed to "Wolfpass GM" rather than "Flat Colour".



Now that the model has been evaluated, we can create a new layout. This needs to be done for each section.

1. Right click on one of the serial sections, and select **New Section Layout**.

2. Change the settings for the section as described above, and click **OK**.

3. Right click on the next serial section, and select either **New Section Layout**, or **Copy Section Layout**.

4. When **Copy Section Layout** is selected, it is possible to copy a layout from an existing cross section (either a serial section or individual section), rather than creating a new layout for each section.

Creating fence sections

Create a new fence section by right clicking on the **Cross Sections and Contours** folder, and selecting **New Fence Section**. A fence section can be created using either an existing polyline, or by drawing a new polyline.

Evaluations can be made on the fence section by right clicking on it, and selecting evaluations. Section layouts can be created as above, then exported.

The fence section can be viewed in the scene, and the linework exported as a dxf by right clicking on the fence section and selecting **Export**.

Session 16: Wolfpass Drillhole Planning

Contents

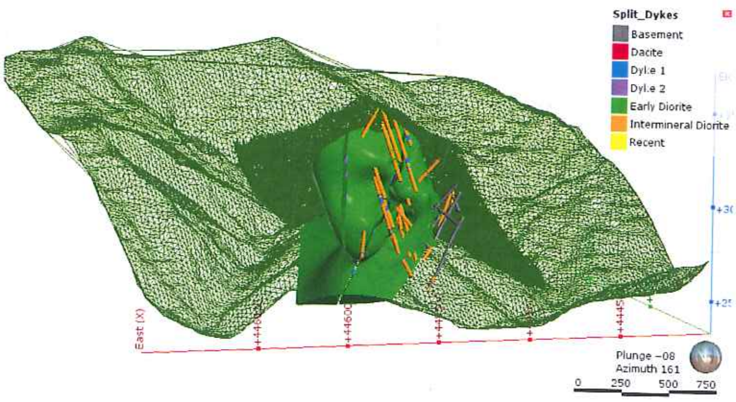
Goals

In this session, we will look at planning drillholes in Leapfrog Geo. This is a simple process, and can include outputs such as expected grade and lithology based on existing models, as well as the standard survey data. We will continue with the Wolfpass dataset for this session.

Set up of a Planned Drillhole

The easiest method of setting up a Planned Drillhole is to view the topography and the drillholes in the scene, as well as the target (which could be the interpolant model or a geological model). We will then specify the collar and target locations, and alter the lift, drift and depth. Once the location of the drillhole has been entered, a prognosis can be made. This takes any existing model and evaluates it against the planned drillhole, letting us look at expected grade and lithology down hole based on current models.

1. The first step is to drag the drillholes, topography and Early Diorite output volume into the scene.



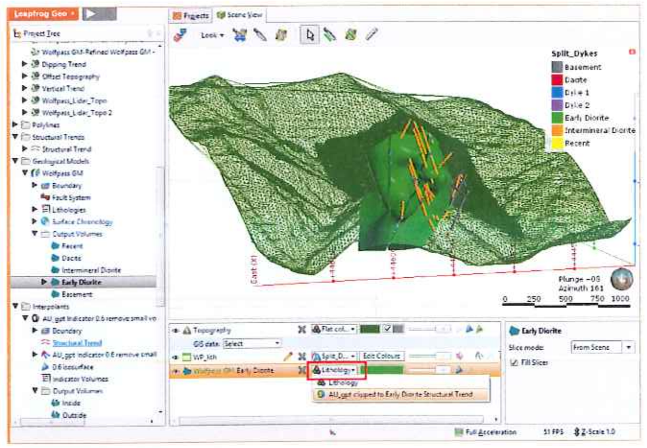
A good way to visualise the location of the high grade based on existing data is to run an evaluation on the Early Diorite.

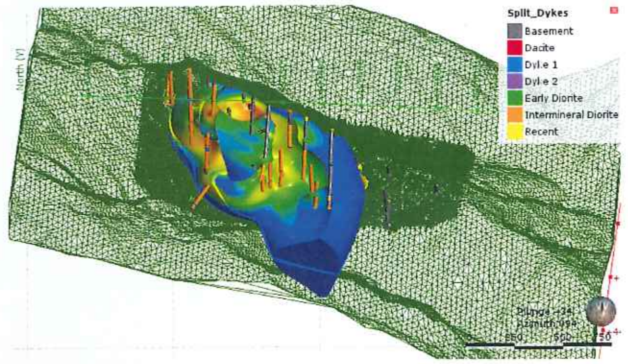
2. Right click on the Early Diorite output volume and select **Evaluations**.

3. Select the **AU_gpt clipped to Early Diorite Structural Trend** interpolant model, and drag it across to the right hand window.

4. Click **OK**, and the interpolant will be evaluated against the Early Diorite surface.

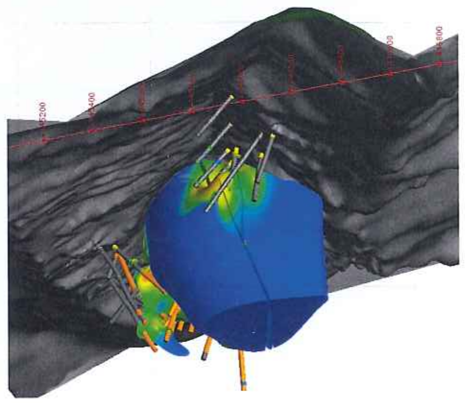
5. Once the evaluation has finished processing, click the display dropdown box in the Scene List, and select the interpolant.





Now that we can see the interpolant evaluated against the mineralised Early Diorite, we can see that there are patched where the high grade is well defined, and also patches where the high grade is poorly defined. To the southern end of the Early Diorite there is a large area of blue, which we can see contains no drillholes. We will plan a drillhole in this area to better define the edge of the deposit.

6. Rotate the scene until you can see both the point on the topography that you would like your collar to be located, and the point on the Early Diorite that you would like the drillhole to target (note that you can be looking underneath the topography to make it easier to visualise)



7. Right click on the **Planned Drillholes** folder, which is beneath the top level **Drillhole Data** folder.

8. Select **Plan Drillhole**, and the **Drillhole Planning** window will appear.

In this window we have several main sections. At the top of the window, we can specify whether to start at the **Collar** or start at the **Target**. Depending on which is selected, either the **Collar** or the **Target** window will be greyed out (in the image above, the **Collar** is selected so the **Target** section is greyed out).

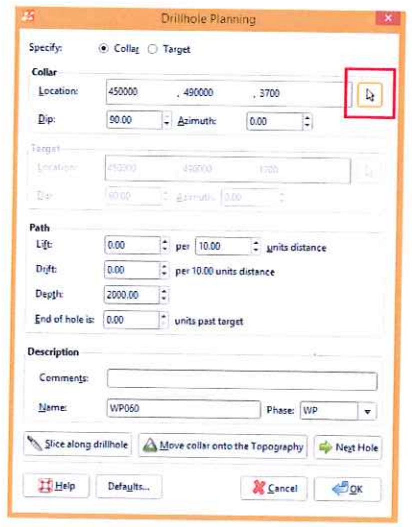
Around halfway down the window there is a **Path** section. Once the start point of the drillhole has been specified, we can change its path by varying the lift, drift, and depth.

At the bottom of the window in the **Description** section, we can add a comment, change the name and change the phase of drilling.

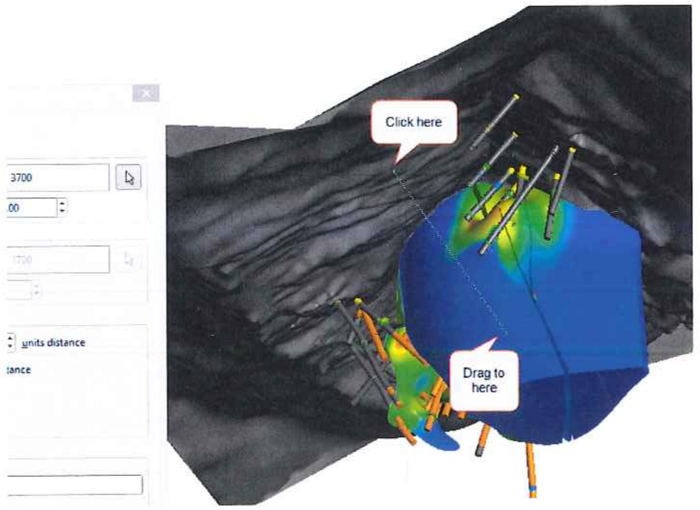
We will look at each of these sections during this session.

9. Firstly, make sure the **Collar** option is selected at the top of the window.

10. Making sure the scene is still in the correct position so that the desired collar location on the topography, as well as the target location are both visible(as in step 6), click the cursor icon in the top right corner of the **Drillhole Planning** window.

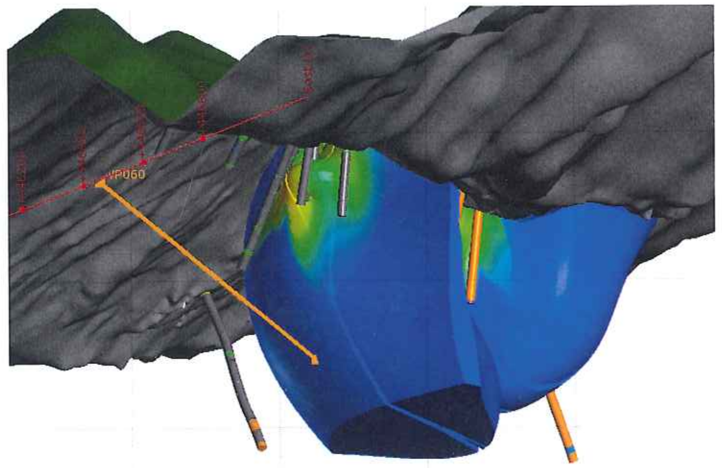


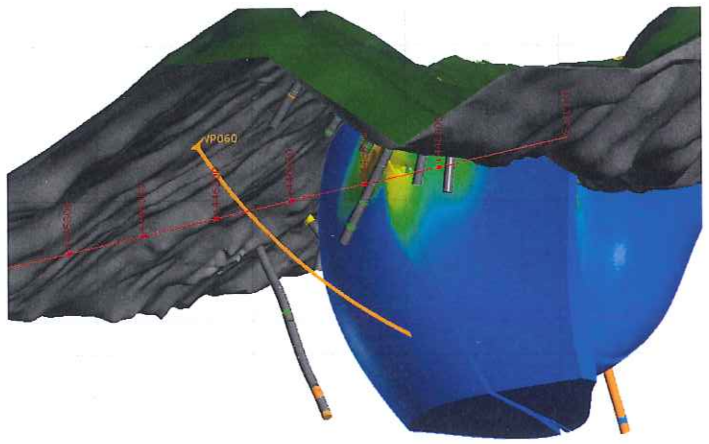
11. Click on the topography where you would like the collar to be located, then drag to the point on the Early Diorite where you would like the target to be located.



12. When you release the cursor, a straight drillhole will be in the scene.

13. It is now possible to edit the lift (vertical movement) and drift (horizontal movement), as well as the total depth, and the distance past the target. The dip and azimuth at the surface can also be edited in the top section.



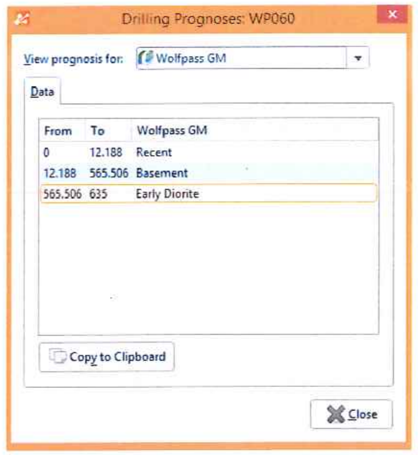


14. Click **OK** and the planned drillhole will be saved under the **Planned Drillholes** folder.

Now we will create a prognosis, which will show the expected grade and lithology at depth based on existing models.

1. Right click on the planned drillhole, and select **Drilling Prognosis**.

2. The default model is the geological model. For the planned drillhole in the example above, the first 12.188 m is expected to be Recent, then from 12.188 m to 565.506 m is Basement, then the Early Diorite intrusion makes up the final 60 m.


Drilling Prognoses: WP060

View prognosis for: Wolfpass GM

Data

From	To	Wolfpass GM
0	12.188	Recent
12.188	565.506	Basement
565.506	635	Early Diorite

Copy to Clipboard

Close

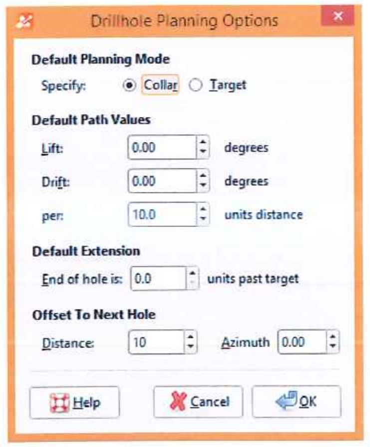
3. To change the prognosis to the interpolant model, click the dropdown at the top of the Drilling Prognosis window and select the most recent interpolant model.

4. The prognosis will take a few seconds to process, then we can see the grade that is expected to be hit down hole. In this example, we are drilling into an area which currently doesn't show any grade so the prognosis isn't very impressive, although it does increase as we move into the Early Diorite which is expected.

Creating a Fence of Drillholes

To create a fence of drillholes at a set distance and azimuth apart, we can edit the **Drilling Planning Options**.

1. Right click on the **Planned Drillholes** folder and select **Edit Planned Drillhole Defaults**.



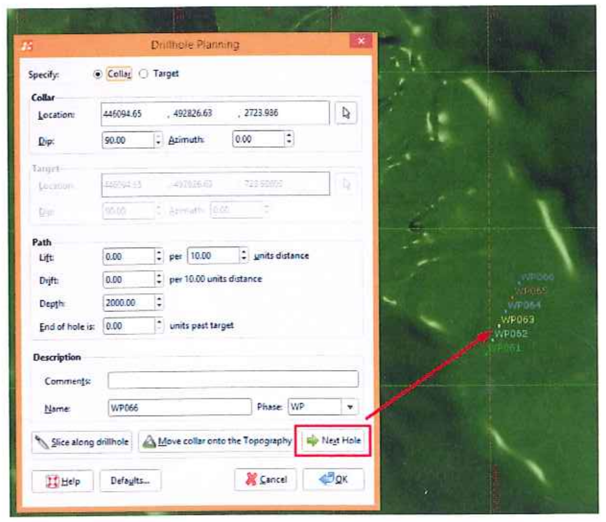
The easiest way to plan a fence is to change the **Offset To Next Hole** options (both the distance and azimuth).

2. For this example, we will specify a distance of 50 m and an azimuth of 25 degrees, then click **OK**.

3. Right click on the Planned Drillholes folder and select **Plan Drillholes**.

4. As we did above, specify the collar location by clicking on the cursor, then clicking on the desired collar location on the topography.

5. Once the initial hole has been created, click **Next Hole** in the bottom right hand corner of the **Drillhole Planning** window.



In the plan view above, we can see that each time Next Hole is clicked, a new hole is created at a distance of 50 m and an azimuth of 25 degrees from the previous hole. Each hole will have a collar on the topography, and will have the lift and drift that were specified in the Drillhole Planning Options (in the example above these were both left at 0.0, so the drillholes are vertical).

Exporting Planned Drillholes

There are two ways to export planned drillholes. The first is to export a csv file, and the second is to export a Viewer file.

1. To export a csv file, right click on the **Planned Drillholes** folder and select **Export Planned Drillholes**.

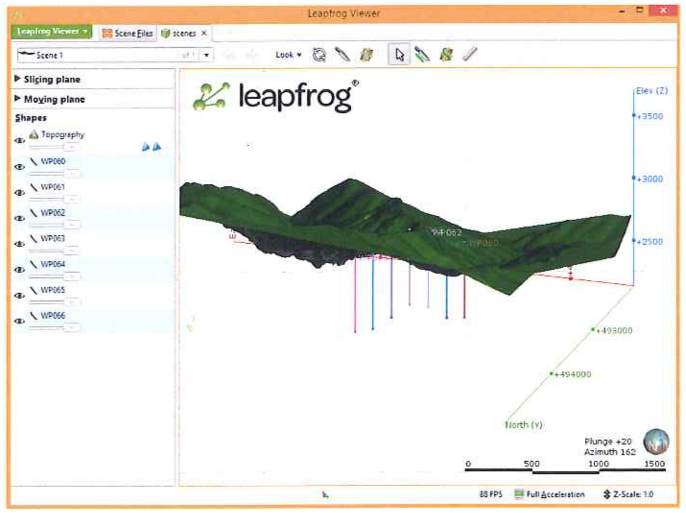
2. Select the drillholes to export using the check boxes, then click **Export**.

3. Browse to an appropriate location, give the file a name, then click **Save**.

4. The planned drillholes will be exported to a csv file which can be opened in Microsoft Excel or any text editor.

	A	B	C	D	E	F	G	H	I	J	K	L
1	Drillhole I	Easting	Northing	Elevation	Azimuth	Dip	Lift Rate	Drift Rate	Distance	Extension	Target De	Comment
2	WP060	445246.5	492933.7	2804.37	25.99119	55	0.6	0	10	0	635	
3	WP061	445989	492600.1	2827.892	0	90	0	0	10	0	2000	
4	WP062	446010.1	492645.4	2811.99	0	90	0	0	10	0	2000	
5	WP063	446031.3	492690.7	2798.888	0	90	0	0	10	0	2000	
6	WP064	446052.4	492736	2781.328	0	90	0	0	10	0	2000	
7	WP065	446073.5	492781.3	2753.92	0	90	0	0	10	0	2000	
8	WP066	446094.7	492826.6	2723.986	0	90	0	0	10	0	2000	

1. To export a Viewer file, save a scene containing the planned drillholes, then export the scene. The scene can then be opened by anyone with the free Leapfrog Viewer installed on their computer.



Exercise 2: Vein Modelling

Contents

Goals

In this exercise, we will look at creating a geological model with three dykes, then use the features in Leapfrog Geo to create a vein system to model the dykes with.

The data for this exercise is in the `Exercises \ Exercise 2 - Vein Modelling` folder.

Importing Data

Import the data following the same steps as have been covered in previous sessions. For this session, we have:

- Topography (as GIS contour lines)
- Drilling data (collar, survey, lith)
- GIS data (surface contacts for breccia, granodiorite and sandstone-marble)
- Structural data (sandstone-marble contact)

Building the geological model

Create a geological model for the project, setting an appropriate boundary and surface resolution. The chronology for the model is as follows:

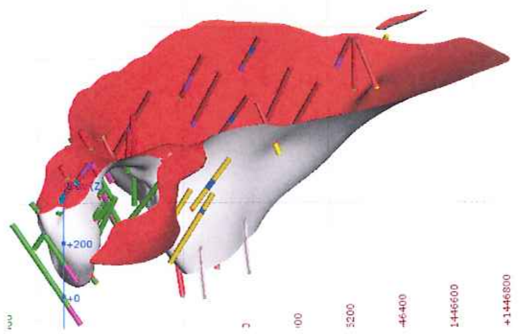
Youngest
Diorite Dykes
Hydrothermal Breccia
Granodiorite
Marble
Sandstone
Oldest

1. Create the above as new lithologies under the geological model.

2. Start by building a depositional surface for the marble-sandstone contact. There are no drillhole intercepts for this surface, so the only option is to build the surface from the GIS line representing the contact, and the structural measurements at the surface.

3. Model the granodiorite contact using an intrusional surface. Include both the drilling intercepts and the GIS line on the surface. Make sure that you ignore the younger lithologies and alter the anisotropy of the surface.

4. Model the hydrothermal breccia using another intrusional surface. Again, there are drilling intercepts as well as a GIS line on the surface delineating the outcrop. Alter the anisotropy of the unit, and also add a root to the breccia using curved polylines.



Modelling the dykes

Leapfrog Geo has an advanced implicit vein modelling tool which allows the user to create a vein network with terminations and pinchouts based on the drilling data and a curved reference surface.

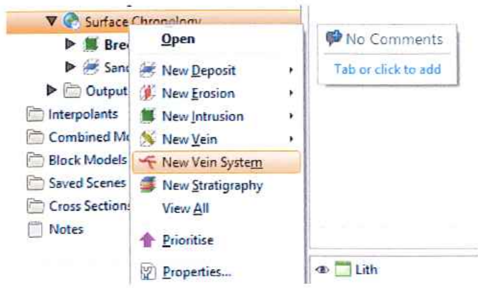
For this project we do not need to use the interval selection tool to define the veins, as they have already been logged in the drilling.

Creating a vein network

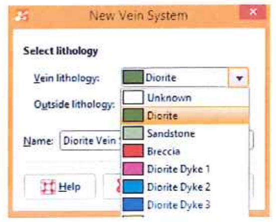
In Leapfrog there are two options for creating veins. We can either create a single vein by selecting **New Vein** under the surface chronology, or we can create a new vein system. The vein system allows a network of veins with one assigned lithology to be created, and interactions between veins added.

1. Double click on the lithologies object under the GM and add a unit called "diorite". This will be the lithology which makes up the three dykes.

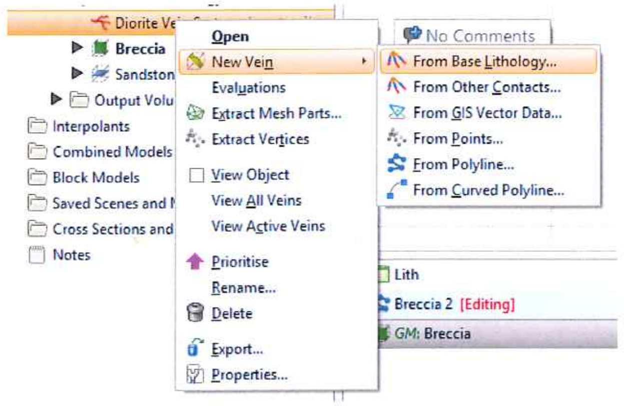
2. Right click on the surface chronology under the GM and select **New Vein System**.



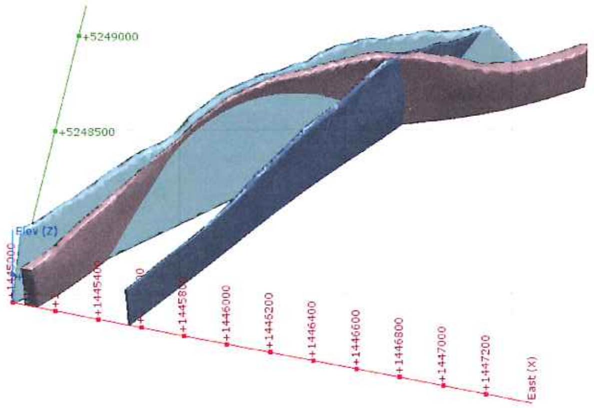
3. The interior lithology for the vein system will be diorite. Select this, then click **OK**.



4. The vein system will have appeared as an object under the surface chronology folder. Build each of the dykes (diorite 1, diorite 2, diorite 3) separately by right clicking on the vein system and selecting **New Vein > From Base Lithology**.

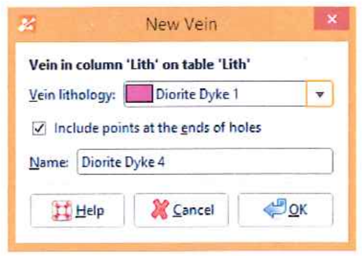


5. The three dykes should look like the following:



6. These are the default surfaces, so we will now go through and make the required edits.

NOTE: If there are drillholes that don't penetrate all the way through the unit that is being modelled, you can choose whether or not to add a point at the end of the drillhole by ticking the **include points at the ends of holes** checkbox.



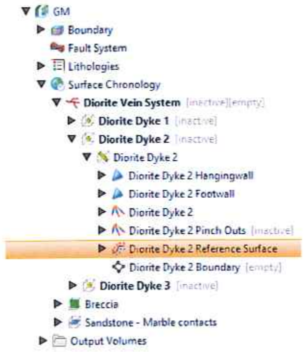
Adjusting reference surfaces

In Leapfrog Geo a vein is built using an offset surface from a reference plane. The plane is interpolated from the centre points of the vein segments. This surface is then offset to create the hanging wall and foot wall surfaces.

There are two options for creating the vein reference plane. Either a curved surface generated from the centre points can be used, or else a flat plane can be used (this was the only available option in the previous versions of Leapfrog Geo). A curved reference plane can easily be edited using polylines and structural data, making it much more flexible than Leapfrog's previous vein modelling tool.

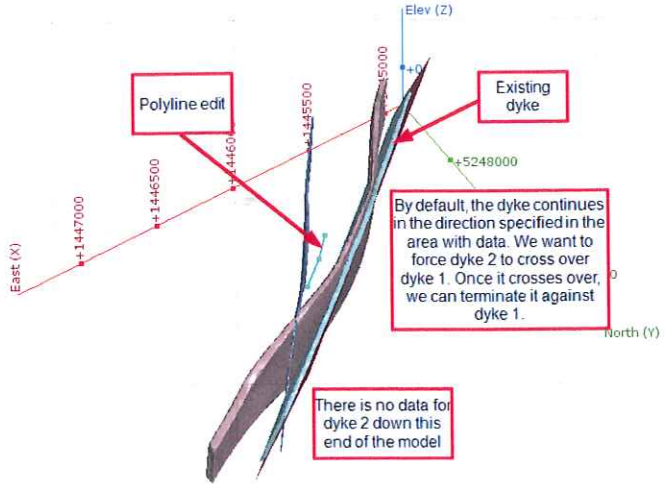
For this model, we will edit the reference plane of dyke 2 which is only present at the south western end of the deposit, as it splays off dyke 1. We need to make this edit as we want dyke 1 to cross over dyke 2 to start with, then later we can terminate it against dyke 2 to create a more realistic splay.

1. Select the triangle to the left of dyke 2 to expose the objects under it. Drag the reference surface into the scene.

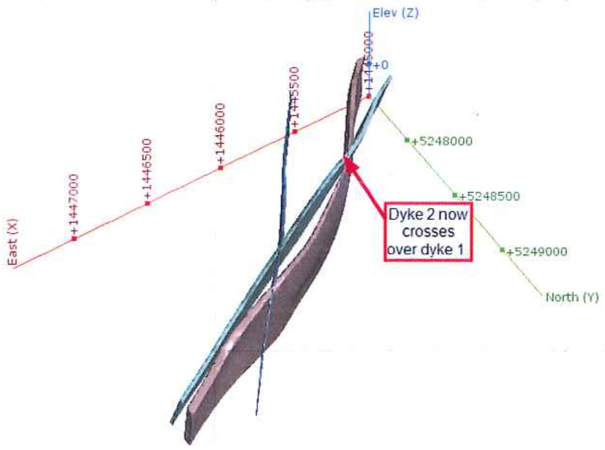


2. Right click on the reference plane and select **edit with curved polyline**. The editing tool here works in the same manner as for any other surface.

3. Drawing on the slicer, add curved polylines until the reference plane is on the Eastern side of dyke 2, similar to the image below.



4. Once the **Save** button is hit at the top of the screen, you will notice that as the dyke moves away from the data, it offsets to follow the form of the reference surface.

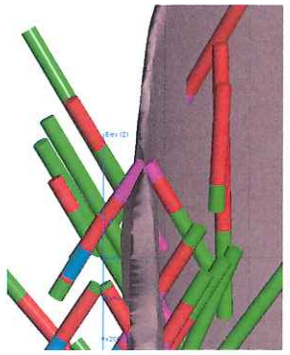


5. If you are dealing with a dyke which is planar, it is still possible to use the standard reference plane by double clicking on the reference surface and selecting the plane.

Editing the segments

When a vein is created, Leapfrog extracts the hanging wall and foot wall segments from the drilling data. Once this has been completed, the foot wall and hanging wall sides are determined so the surface can be offset to the points.

By interrogating dyke 1, you will notice an area which has been pinched out and is not honouring the drilling data. This is because two drillholes are collared within the dyke drilling in different directions so the hanging wall - foot wall segments have been incorrectly assigned. This is not a problem, as they can easily be edited using the process described below.



1. Drag the diorite dyke 1 segments into the scene. You will see that all segments have a blue and red side. Red indicates the hanging wall, and blue indicates the foot wall side.

Editing the hanging wall and foot wall manually

If you would like to edit the hanging and footwall manually you can do this buy right clicking on the hanging wall and foot wall under the vein and selecting edit. You can also add points, structural discs and polyline to the hanging wall or footwall surface. These could be measurements regularly collected from the mine and added to update the model.

Also if you have surface contacts which you would like to honour you can add these by using the draw on surface function and create polyline edits on the topography.

Completed Model

The final geological model including all lithologies and dykes should look like the image below.

Exercise 3: Geothermal System

Contents

Goals

For this exercise, we will create a geological model of a geothermal system from imported drillhole and GIS data.

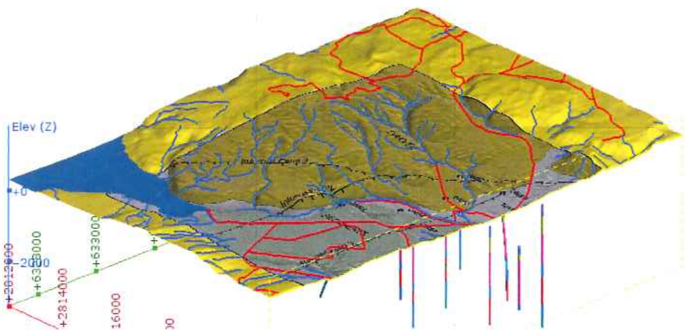
The data for this exercise can be found in the `Exercises \ Exercise 3 - Geothermal System` folder.

Setting up the Project and Importing Data

Create a new project called **Geothermal System**.

Import the data from the folder for this exercise:

- Topography is a DTM (*.asc file)
- There are GIS lines representing roads, rivers and lakes
- There is an ortho photo, geological map and topographical map
- There are boreholes with collar and survey files, as well as geology and well temperatures.



Building the Geological Model

Create a new geological model. Accept the default extents, but drop the resolution down to 200.

Build the sequence from the bottom to the top:

- Basement: New deposit from base lithology, using contacts above, making sure to ignore the 'no return'. Set a horizontal, out of plane trend of 0.1.
- Rhyolite: New intrusion from base lithology, making sure to ignore the 'no return'. Set an appropriate trend based on the drilling / default surface (0.3 out of plane works well)
- Ignimbrite 1: New deposit from base lithology, using contacts above, ignoring the rhyolite (which is a younger intrusion). Set a horizontal, out of plane trend of 0.2.

- Ignimbrite 2: New deposit from base lithology, using contacts above, ignoring the dacite. Import the GIS line 'Sediments - ignimbrite 2 contacts' (don't filter the data), and add it to the default surface by right clicking on the surface > Add > GIS Vector Data. Make sure to choose the 'On Topography' version of the file.
- Andesite: New intrusion from base lithology, ignoring no return and dacite. Set an appropriate trend (0.3 out of plane works well)
- Alluvium: New erosion from base lithology, using contacts below. Import the GIS layer 'superficial deposit outline' and add it to the surface, making sure to use the 'on topography' version. Import the structural measurement 'structural data superficial' and add it to the surface.
- Dacite: New intrusion from base lithology, ignoring the no return. Import the GIS layer 'dacite outline' and add it to the surface, making sure to use the 'on topography' version of the GIS line.

Put the surfaces in the correct chronological order, and activate them by ticking the checkboxes in the Surface Chronology folder.

Check whether any manual edits are necessary. In the South West corner, the sediments - alluvium contact will need to be edited.

Create the Fault System

Create the faults by drawing new GIS lines using the geological map as a guideline. Adjust the surfaces to add dips. There is downhole structural data that can be imported for Fault 2, the dip of Fault 1 is vertical, and the dip of Fault 3 is 77 degrees to the East.

Add interactions to the faults based on the traces on the geological map, then activate the faults to split the model into four fault blocks.