
# Help for EnterVol
# Version 1.84
# October 30, 2013

# Table of Contents

# EnterVol for ArcGIS Overview


*EnterVol for ArcGIS* creates and manipulates true 3D volumetric geology and/or analyte models created in EnterVol Geology, EnterVol GeoStats, or saved in C Tech's EVS Field File (EFF) format. *EnterVol for ArcGIS* performs subsetting (plumes, cuts, slices, cross-sections, contouring, etc.) and analysis (volumetrics) inside of ArcGIS Desktop10.1 3D visualization of these models is performed in ArcScene™.

EVS Field Files can be created in C Tech's EVS or MVS software and in the EnterVol Product Suite components *EnterVol GeoStats* and *EnterVol Geology.*

In ESRI terminology, C Tech's EFFs represent a feature dataset, usually consisting of a single feature class (*a collection of geographic features with the same geometry type (such as point, line, or polygon), the same attributes, and the same spatial reference).* EnterVol extends the definition of feature class to include volumetric geometry types.

A single EFF file can contain multiple features classes in that there can be feature datasets with multiple geometry types. A single EFF can contain any or all of EnterVol's supported geometry types including points, lines, surfaces and volumes.

Each of the above geometry types is composed of nodes which represent endpoints or vertices of the object. Each node has x, y, & z coordinates to specify its location. Unique to EnterVol is the ability to associate with the nodes as well as the features. Nodal data is a concept that is new to ArcGIS desktop just like volumetric geometry types.

We need volumetric geometry types because without them volumes cannot be represented as anything more than the space enclosed by surfaces. These volumetric features allow us to create volumetric grids which can have simple or incredibly complex shapes.

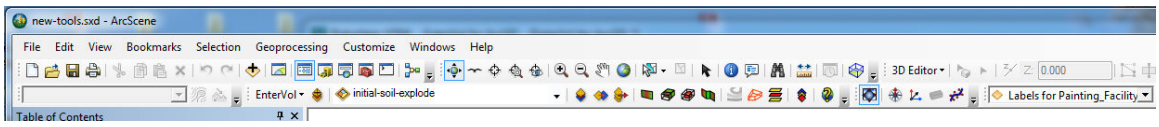
### But why do we need nodal data? What can it do that normal attribute data associated with each feature cannot do?

As you apply the volumetric subsetting operations in EnterVol, you will notice that plumes and arbitrary cuts through volumes have smooth surfaces. Furthermore, when EnterVol cuts or slices through a volumetric dataset, the result incorporates the data throughout the resulting subset.

These smooth subsetting operations are only possible because of nodal data. When attributes (data) are associated with each feature such as a triangle, the resolution or smoothness with which we can subset is limited to the size of each triangle. EnterVol's paradigm of associating continuum data with the nodes (vertices) of each feature allow us to subset to a sub-feature level. This dramatically improves accuracy and allow you to use coarser grids without compromise. It also greatly improves analytical accuracy of computing plume volumes and makes possible the integration of chemical masses.

EnterVol has two toolbars. The primary is the EnterVol toolbar and also EnterVol Tools. The EnterVol toolbar includes options to:

- Layer Operations
  - Adding data
  - Layer Selector
  - Set symbology
  - Copy symbology
  - Export to EVS
- Subsetting Tools
  - Slice
  - Cut
  - Plume
  - Fence
- Display Tools
  - Isolines
  - Edges
  - Exploding geologic layers
- Mass and Volume (Volumetric Analysis)
- Open this Help File




The EnterVol Tools toolbar includes options to:

- Label Features
- EnterVol Raster to Volume
- Project Features to 2D
- Open the EnterVol Tools Scene Control window
- Add Direction Indicator
- Add Axes (ArcScene) & Add Grid (ArcMap)

# Installation of EnterVol

EnterVol's Product Suite runs under ArcGIS Desktop 10.x. Earlier version of ArcGIS (e.g. 9.x) are not supported. EnterVol requires an ArcGIS Desktop 10 license and the ArcScene extension. No other ESRI products or third party applications are required.

The EnterVol Product Suite and support applications and files can be downloaded at www.ctech.com. Without registering and logging in, you will have access to the primary EnterVol files shown below:

The EnterVol for ArcGIS Installer includes EnterVol for ArcGIS (the core product), EnterVol Geology, EnterVol GeoStats and EnterVol Tools. The EnterVol Sample Data is recommended to allow you to duplicate the exercises in the Tutorials which are included in this help file.

## Installation of EnterVol

EnterVol's Product Suite runs under ArcGIS Desktop 10.x. Earlier version of ArcGIS (e.g. 9.x) are not supported. EnterVol requires an ArcGIS Desktop 10 license and the ArcScene extension. No other ESRI products or third party applications are required.

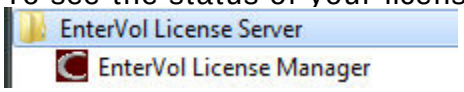
The EnterVol Product Suite and support applications and files can be downloaded at www.ctech.com. Without registering and logging in, you will have access to the primary EnterVol files shown below:


The EnterVol for ArcGIS Installer includes EnterVol for ArcGIS (the core product), EnterVol Geology, EnterVol GeoStats and EnterVol Tools. The EnterVol Sample Data is recommended to allow you to duplicate the exercises in the Tutorials which are included in this help file.

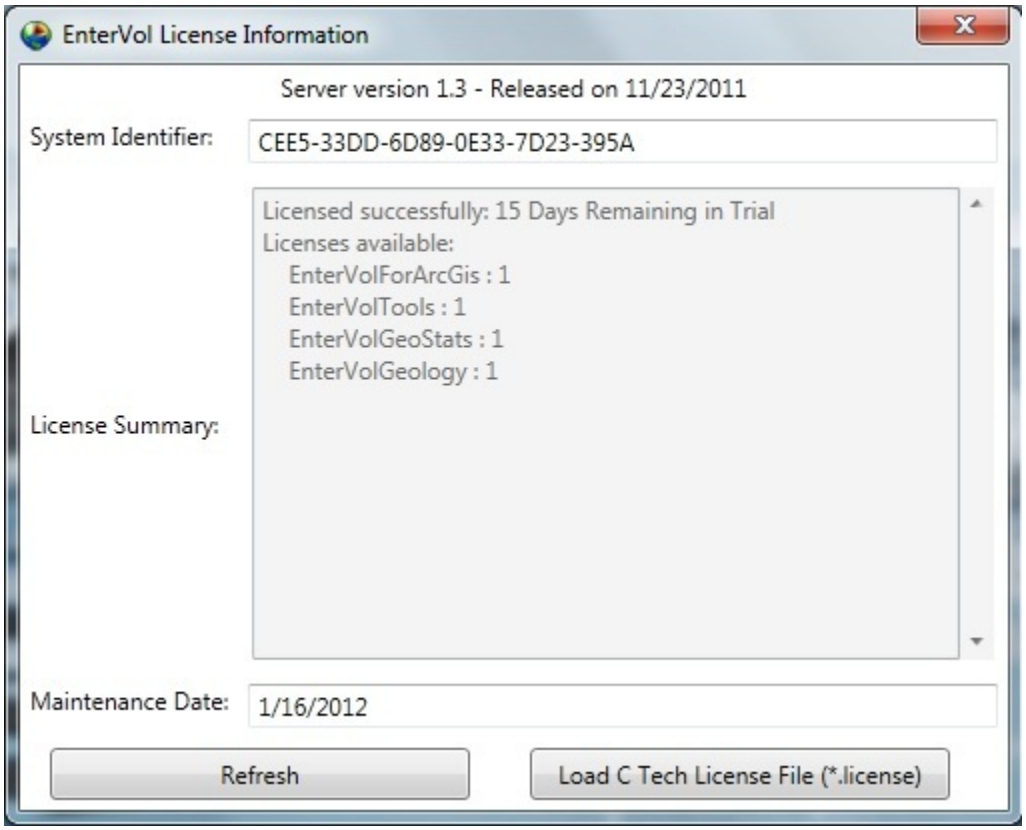
## Trial License

Provided you have never before installed any version of EnterVol on your computer, upon installation you will automatically receive a 15 day fully functional license. Installing a newer version of EnterVol or uninstalling and re-installing will not extend the trial license.

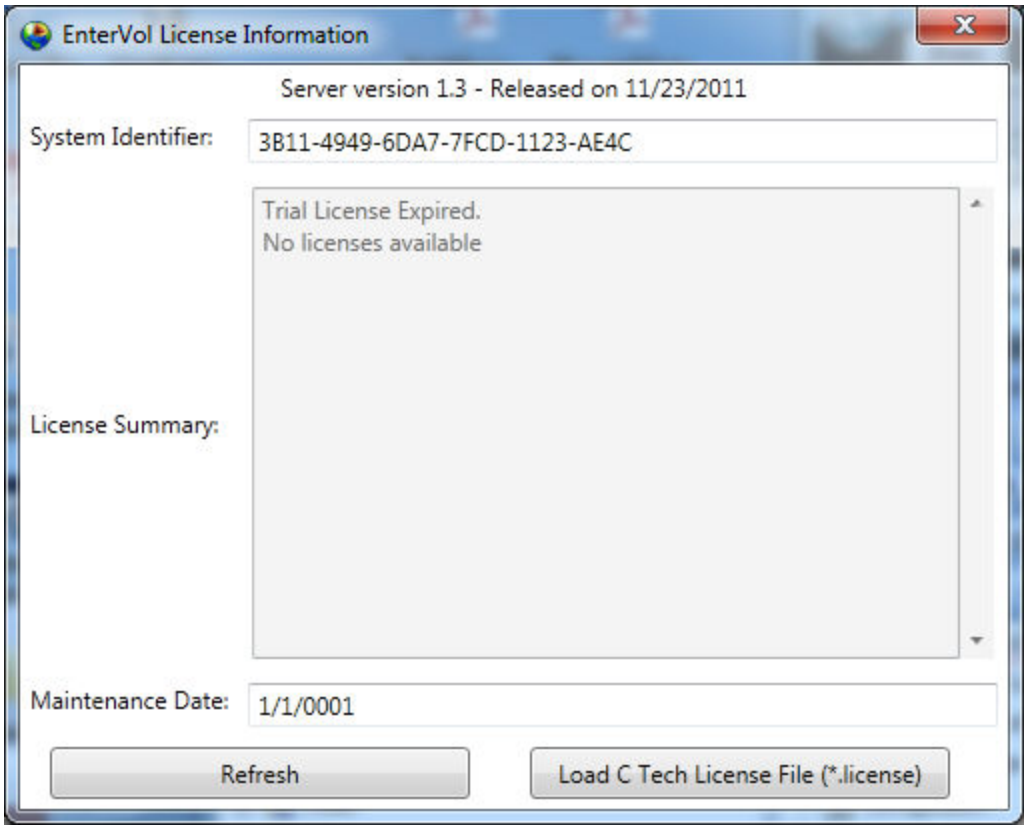
To see the status of your license you can open the EnterVol License Manager




or *Licensing* in the EnterVol menu in ArcGIS.

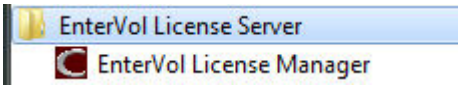



After the trial period, if you have not purchased an EnterVol license, it will show:

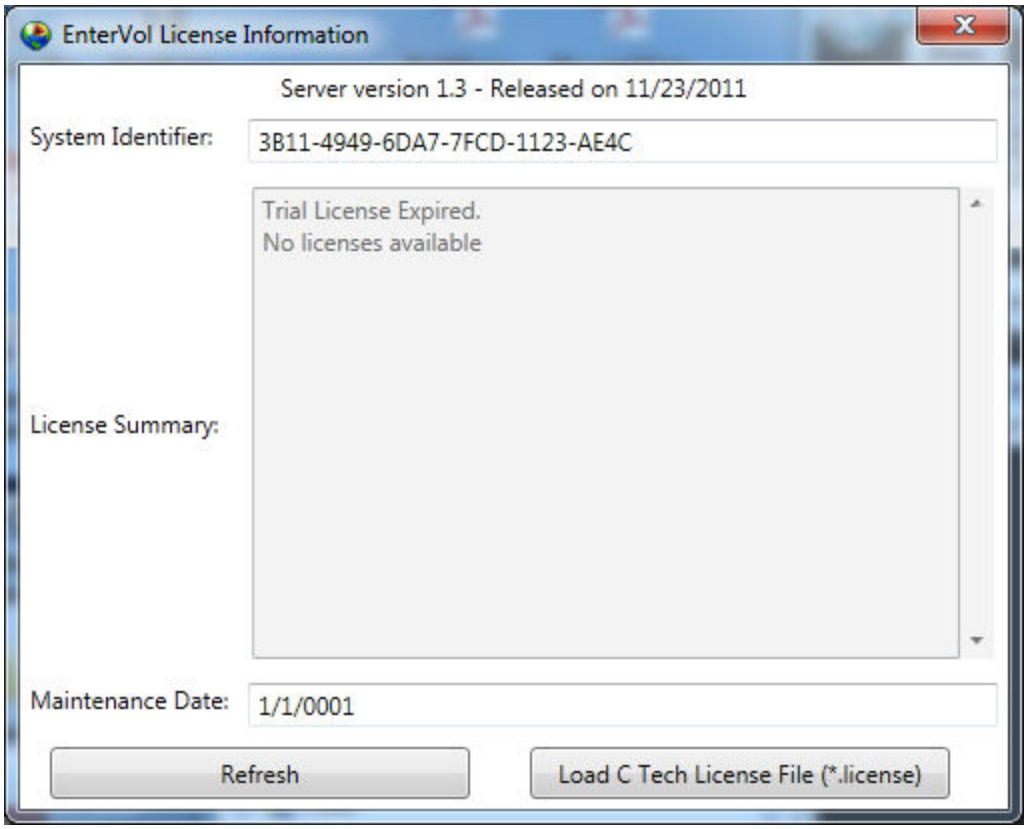



## Installing an EnterVol Fixed License

To purchase and install an EnterVol fixed license you must:

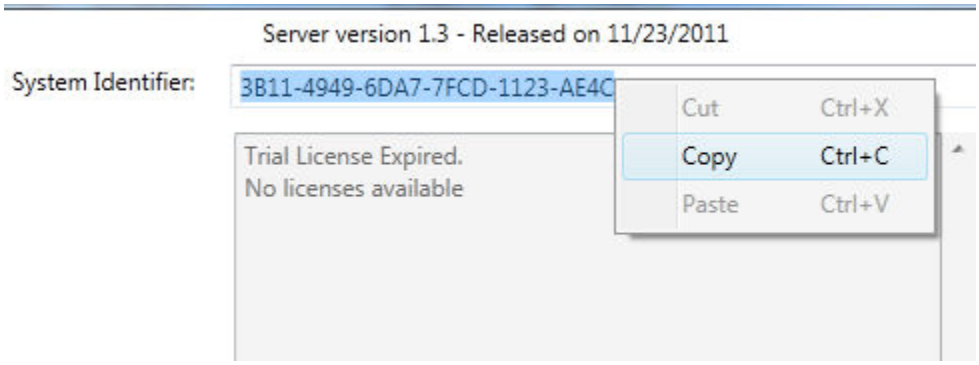
- Select the computer with ArcGIS 10.x and ArcScene already installed to be licensed
- Install EnterVol
- Open the EnterVol License Manager 

or *Licensing* in the EnterVol menu in ArcGIS.




PLEASE NOTE: Your System Identifier will vary and the Trial License shown above may be different.

To create your license, C Tech must receive your System Identifier. DO NOT send us a screenshot of this window. Rather, select the alphanumeric string in the System Identifier window and select Copy as shown below.



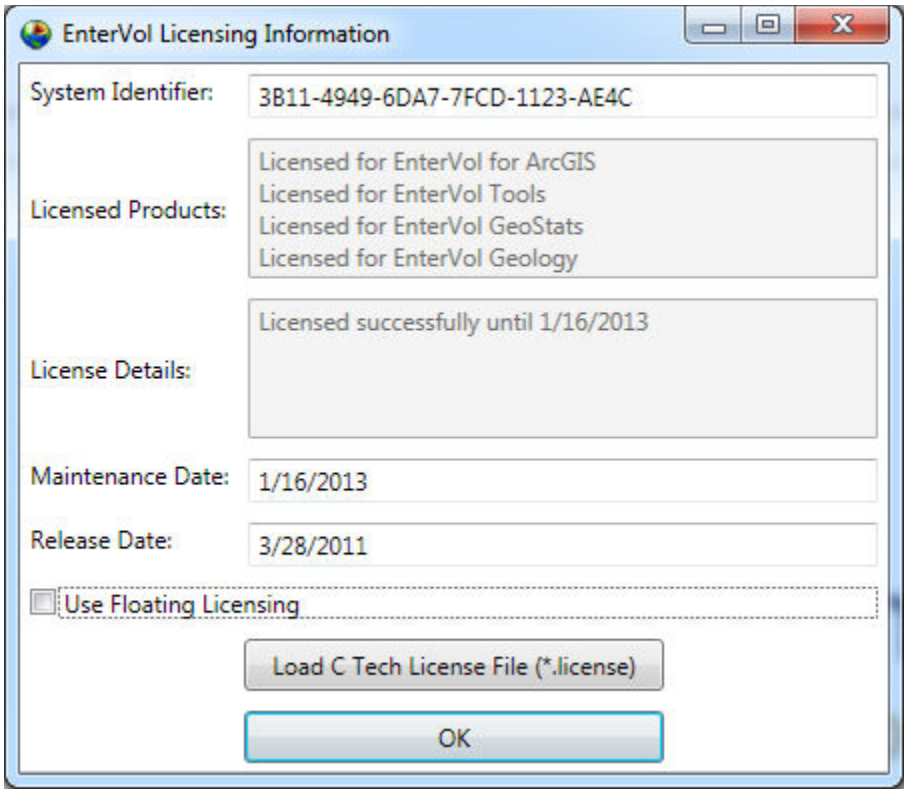

E-mail the System Identifier string to sales@ctech.com along with the following required information:

- **System Identifier:** XXXX-1234-YYYY-5678-ZZZZ-0009
- **Company**: ABC Consultants, Inc.
- **User**: Ms. Jane Doe
- **E-Mail:** j.doe@abc-cons-inc.com

- **Phone**: 505-555-1212
- **C Tech's Invoice number** for your order (this is to ensure that you receive the proper licenses)

Upon verification of the information you provide, C Tech will generate a license file which will be sent to the e-mail address provided above.

Open the EnterVol License Manager and click the *Load C Tech License File (*.license)* button. Browse to the .license file and select it. The License Manager should update to reflect the licenses you have purchased.



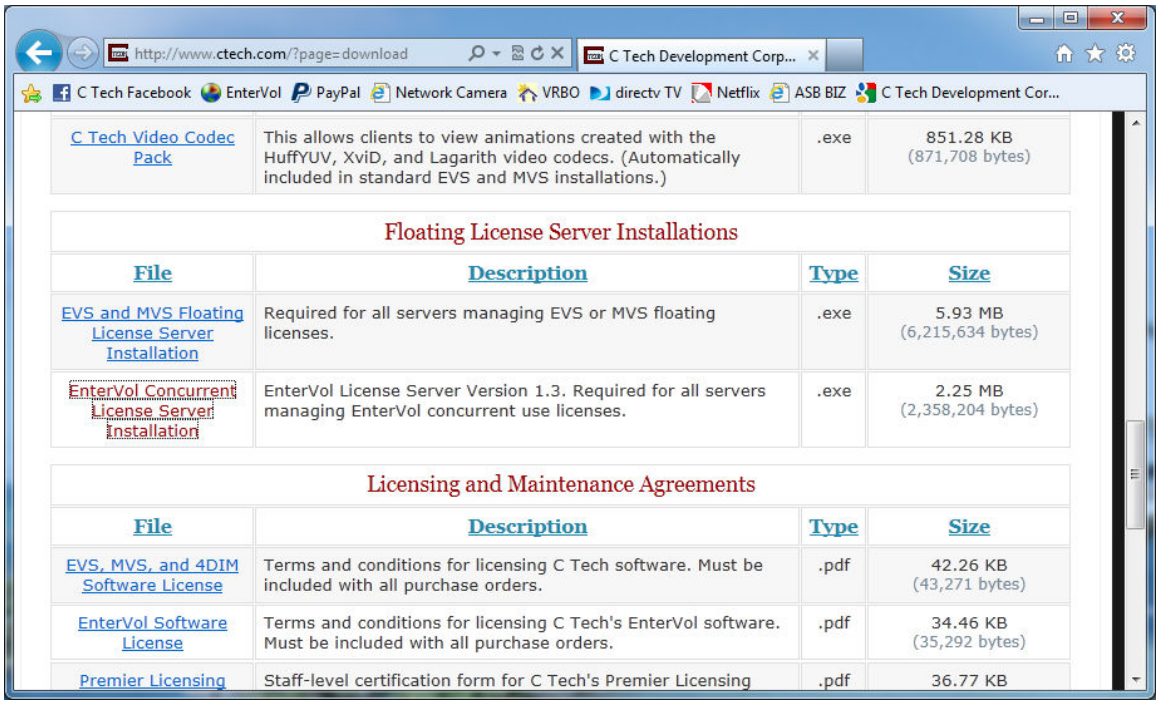

The above window shows a single fixed license for all EnterVol products through January 16, 2013.

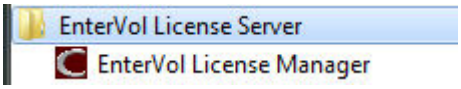
If you start ArcScene, you should see the EnterVol menus enabled. If you have any license problems see the License Troubleshooting Section

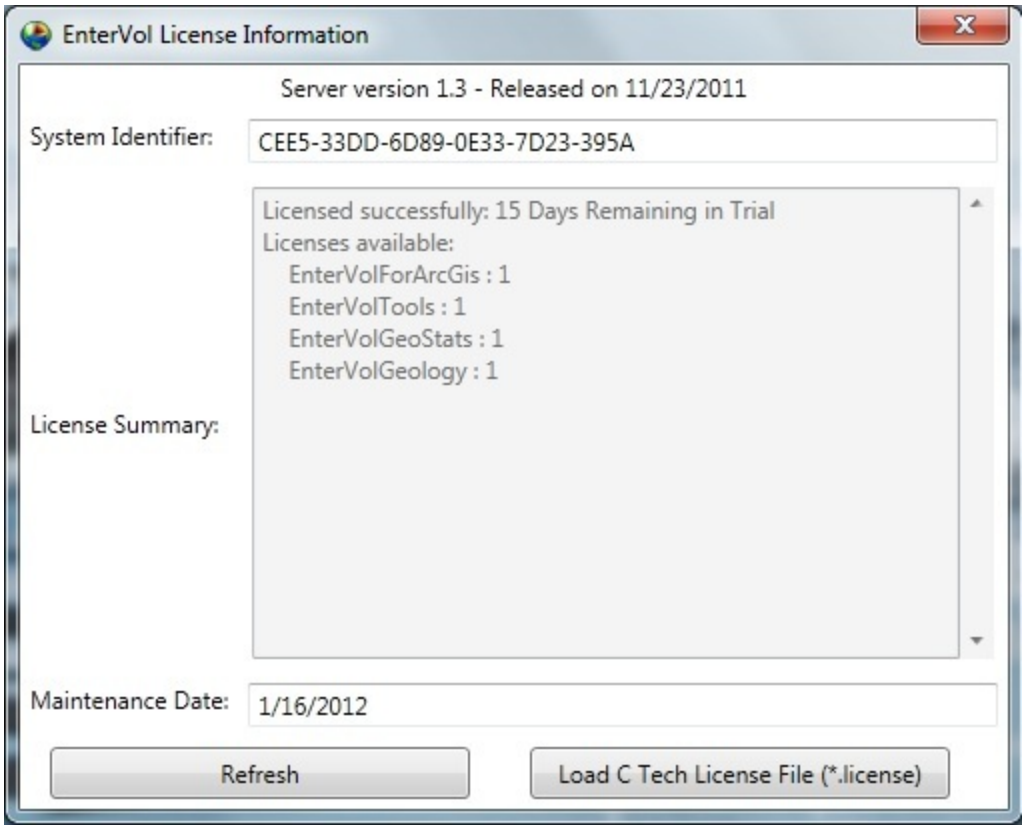
## Server Installation for EnterVol Concurrent Licenses

To purchase and install EnterVol Concurrent Licenses you must:

- Select the computer which will be the **license server** for EnterVol licenses.
  - The computer to be the *license server* does not require ArcGIS 10.x or any third party software.
- Download and install the EnterVol Concurrent License Server Installation from the downloads page ctech.com. Please note that this program is only accessible to registered users at ctech.com who are logged in.
  - Don't miss the **FINAL STEP** at the end of this topic.

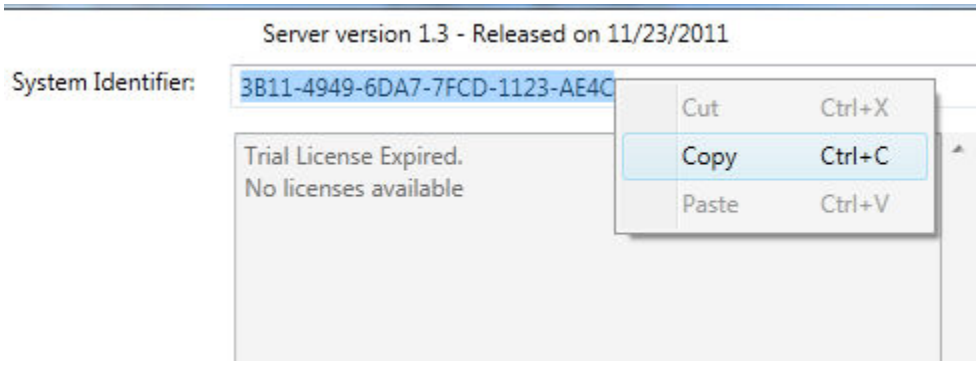



- Open the EnterVol License Manager 




PLEASE NOTE: Your System Identifier will vary and the Trial License shown above may be different.

To create your license C Tech must receive your System Identifier. DO NOT send us a screenshot of this window. Rather, select the alphanumeric string in the System Identifier window and select Copy.



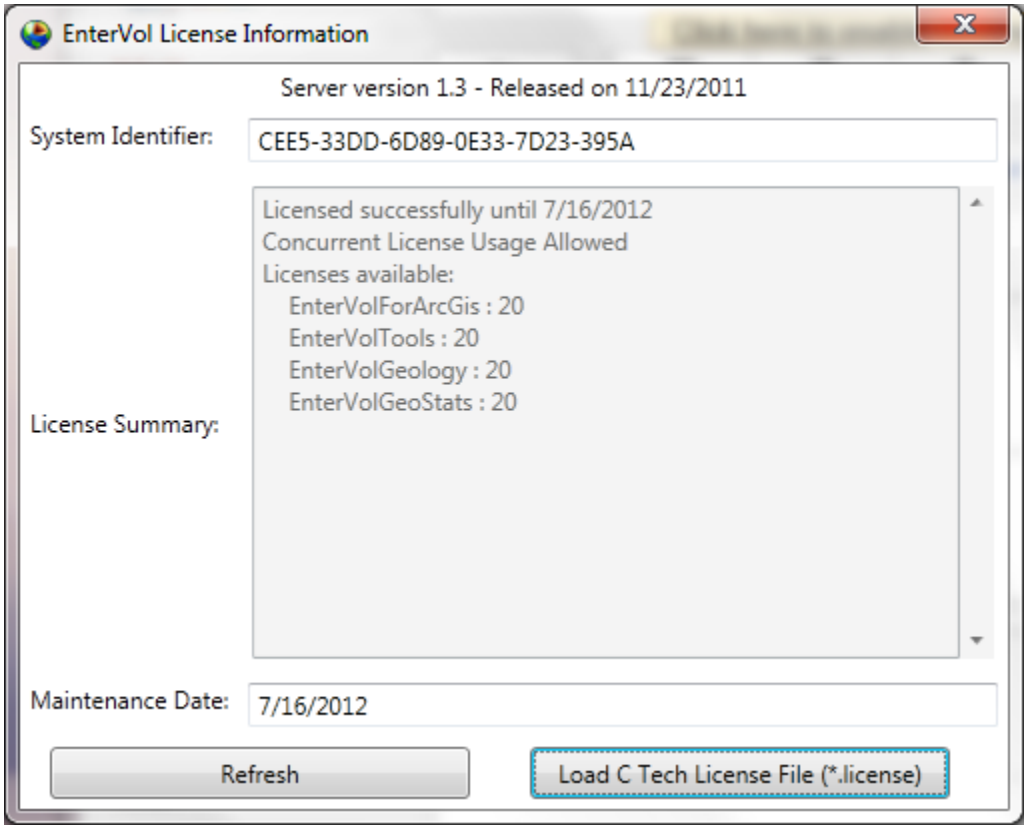

Then send that string to sales@ctech.com along with the following required information:

- **System Identifier:** XXXX-1234-YYYY-5678-ZZZZ-0009
- **Company**: ABC Consultants, Inc.

- **User**: Ms. Jane Doe
- **E-Mail:** j.doe@abc-cons-inc.com
- **Phone**: 505-555-1212
- **C Tech's Invoice number** for your order (this is to ensure that you receive the proper licenses)

Upon verification of the information you provide, C Tech will generate a license file which will be sent to the e-mail address provided above.

Open the EnterVol License Manager and click the *Load C Tech License File (*.license)* button. Browse to the .license file and select it. The License Manager should update to reflect the licenses you have purchased.



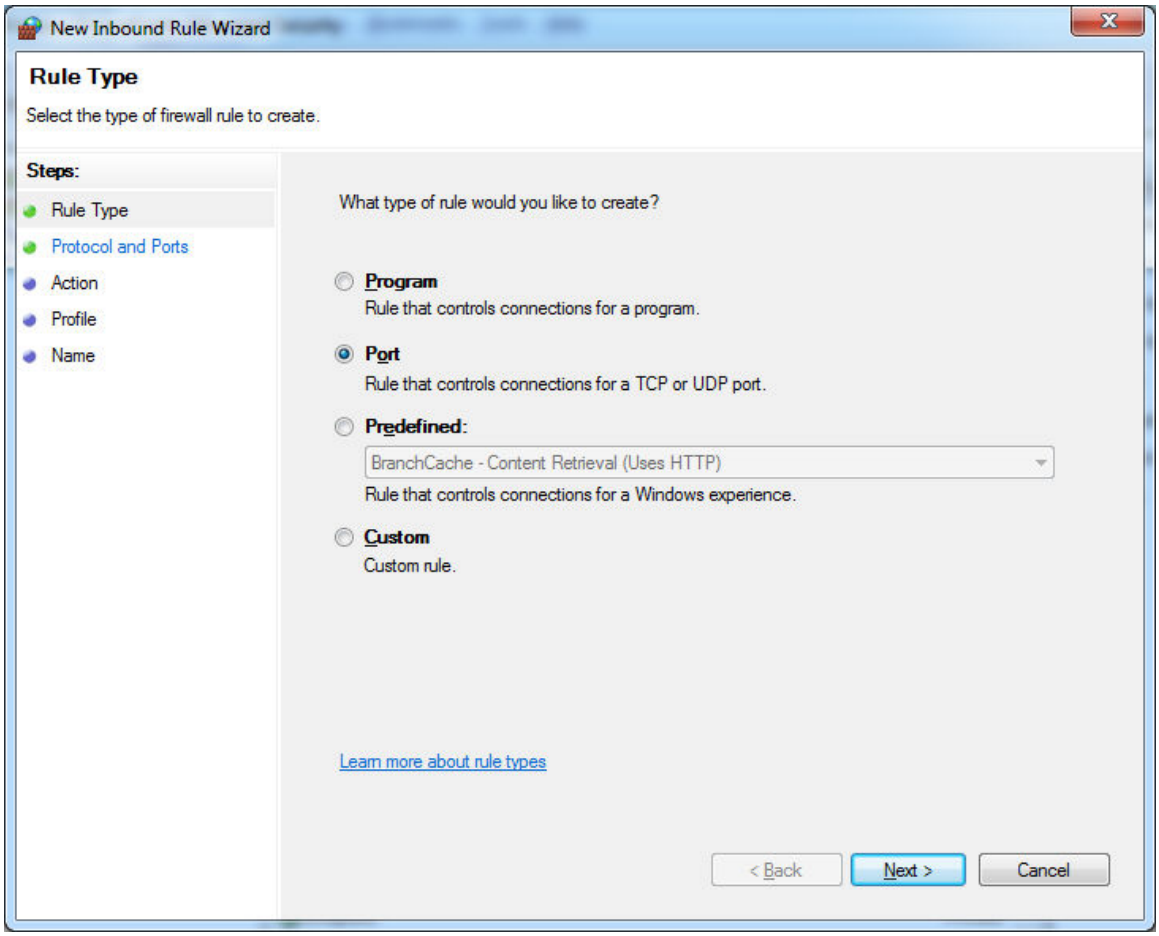

The above window shows a 20 user concurrent license for all EnterVol products through July 16, 2012.

**FINAL STEP:** The last step in the server installation is to unblock port 9347 to inbound TCP queries. The process to do this will depend on what Firewall software your server uses. The example below is for Windows 7.

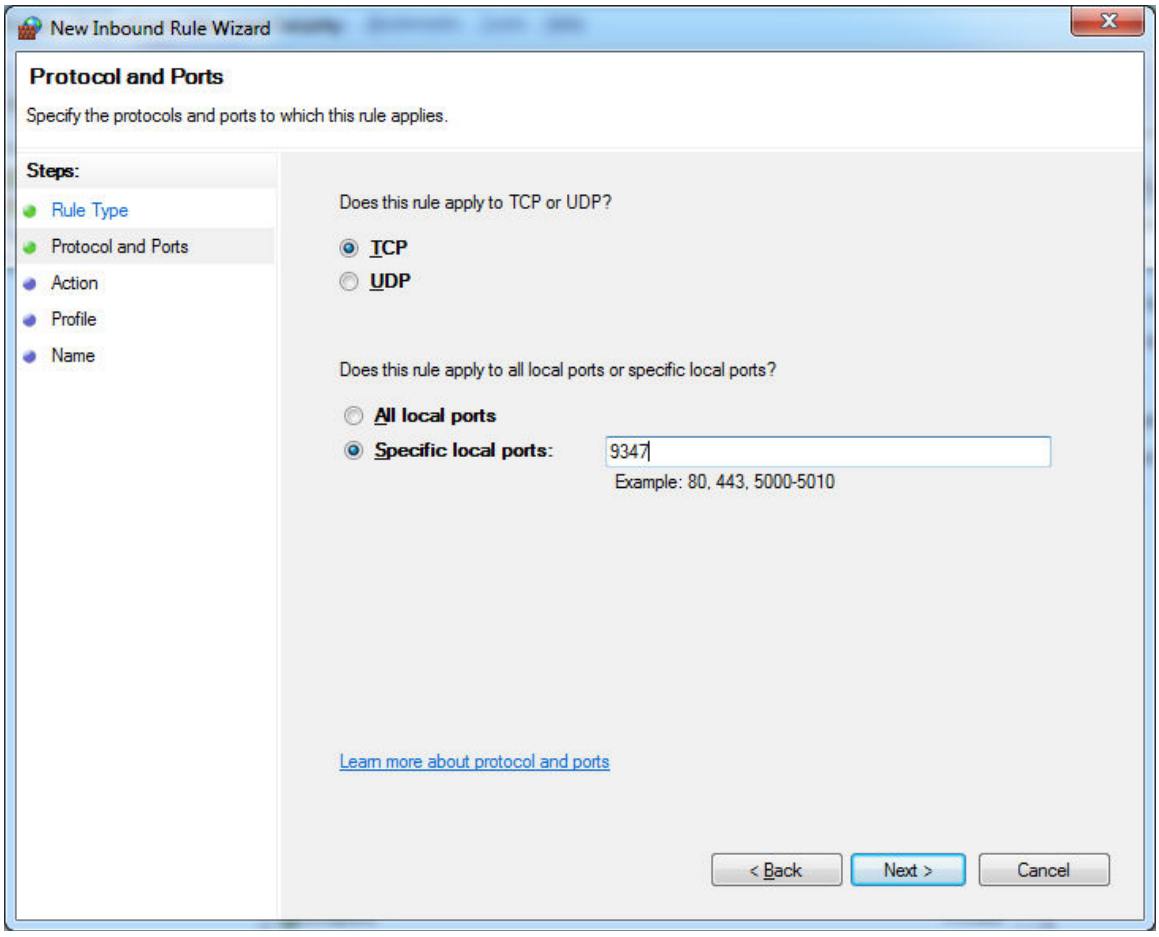
Open *Windows Firewall with Advanced Security*.

Select *Inbound Rules* and create a *New Rule*

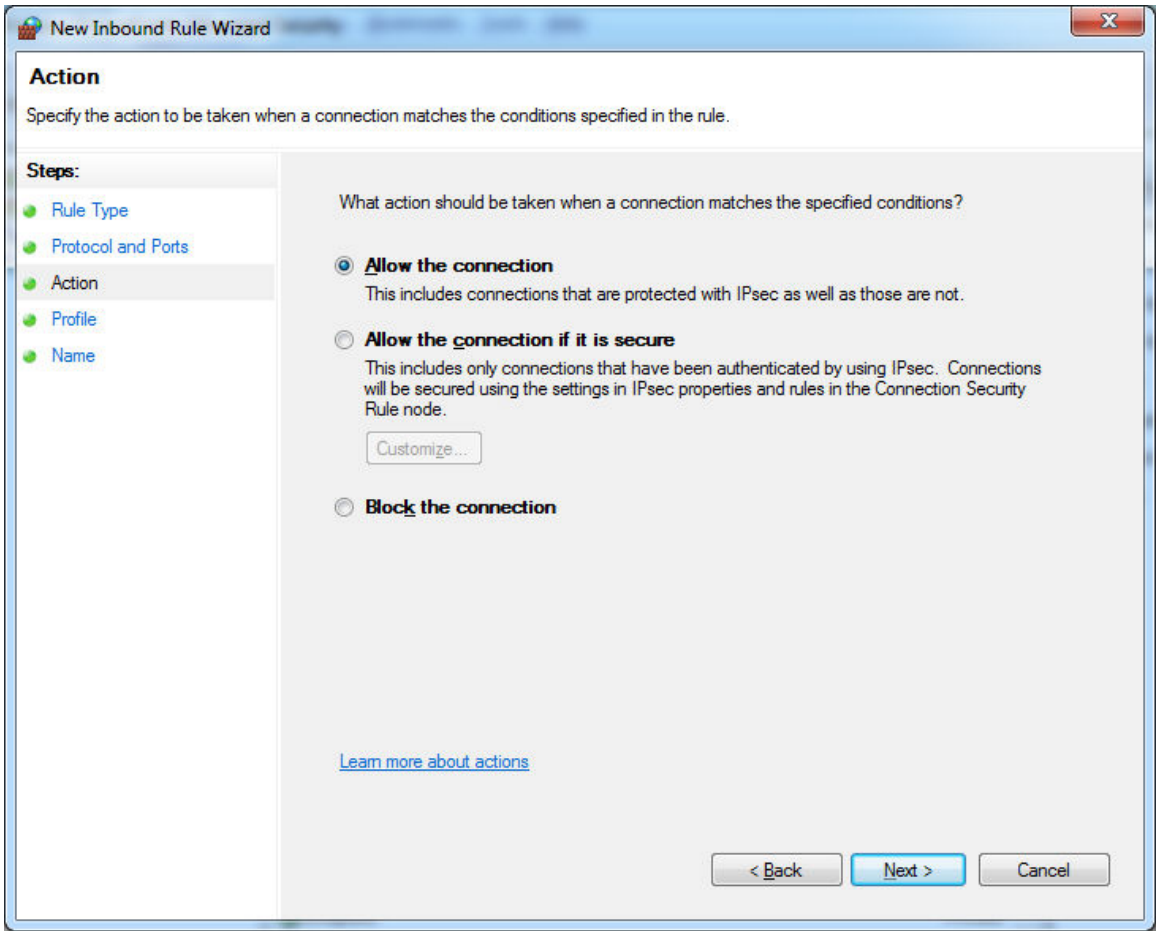
Select *Port Rule*

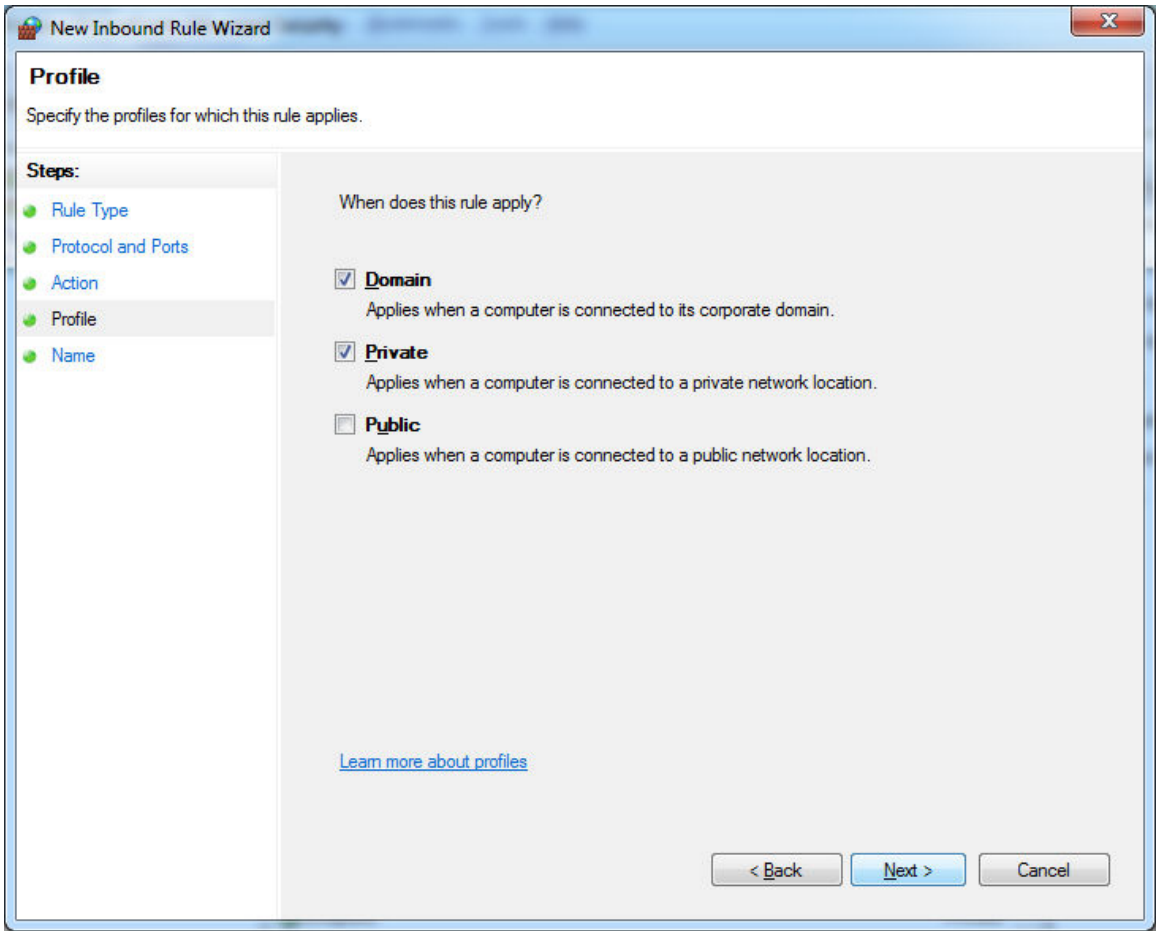


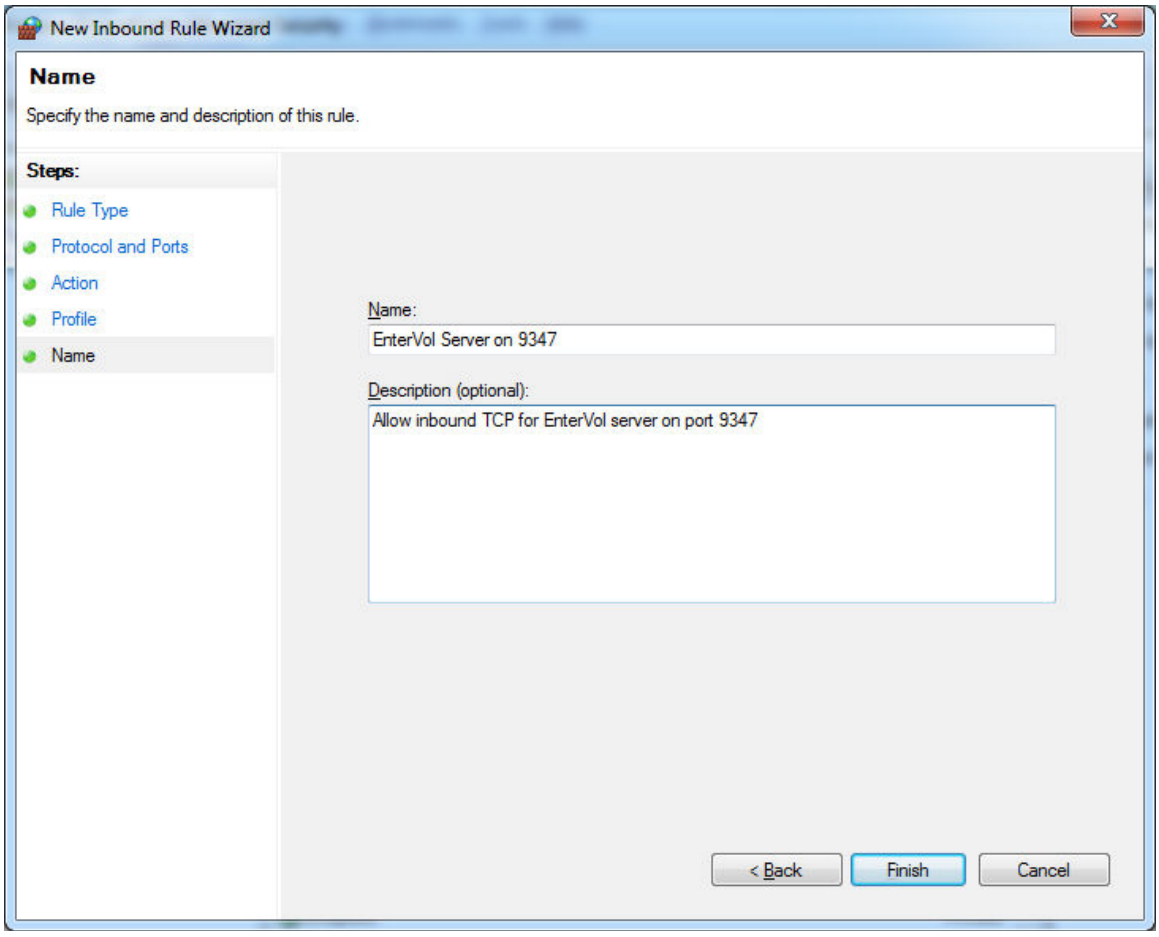

*Choose TCP and set the port to 9347*




Allow the connection with settings that meet your company's IT policies.


New Inbound Rule Wizard
Action
Specify the action to be taken when a connection matches the conditions specified in the rule.
Steps:
Rule Type
Protocol and Ports
Action
Profile
Name
What action should be taken when a connection matches the specified conditions?
Allow the connection
This includes connections that are protected with IPsec as well as those are not.
Allow the connection if it is secure
This includes only connections that have been authenticated by using IPsec. Connections will be secured using the settings in IPsec properties and rules in the Connection Security Rule node.
Customize...
Block the connection
Learn more about actions
< Back
Next >
Cancel

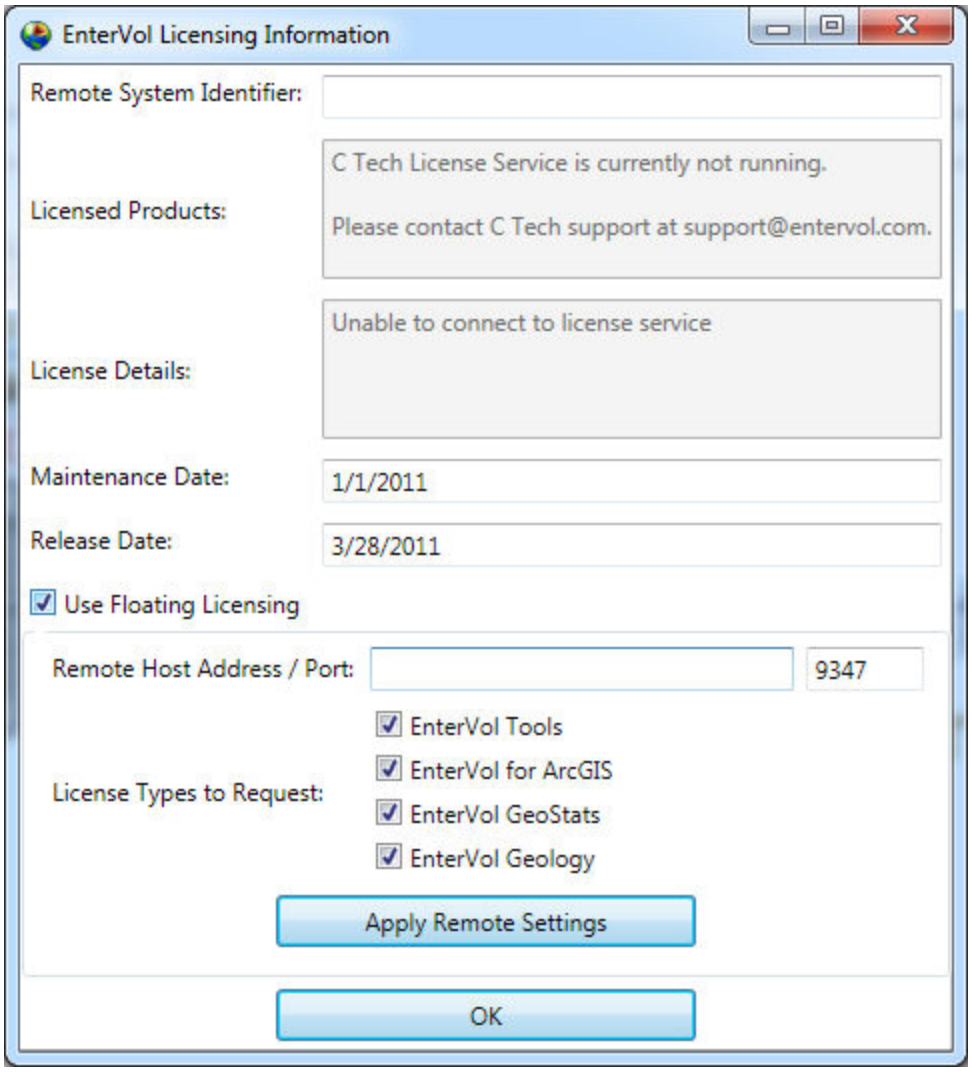

New Inbound Rule Wizard
Profile
Specify the profiles for which this rule applies.
Steps:
Rule Type
Protocol and Ports
Action
Profile
Name
When does this rule apply?
Domain
Applies when a computer is connected to its corporate domain.
Private
Applies when a computer is connected to a private network location.
Public
Applies when a computer is connected to a public network location.
Learn more about profiles
< Back
Next >
Cancel



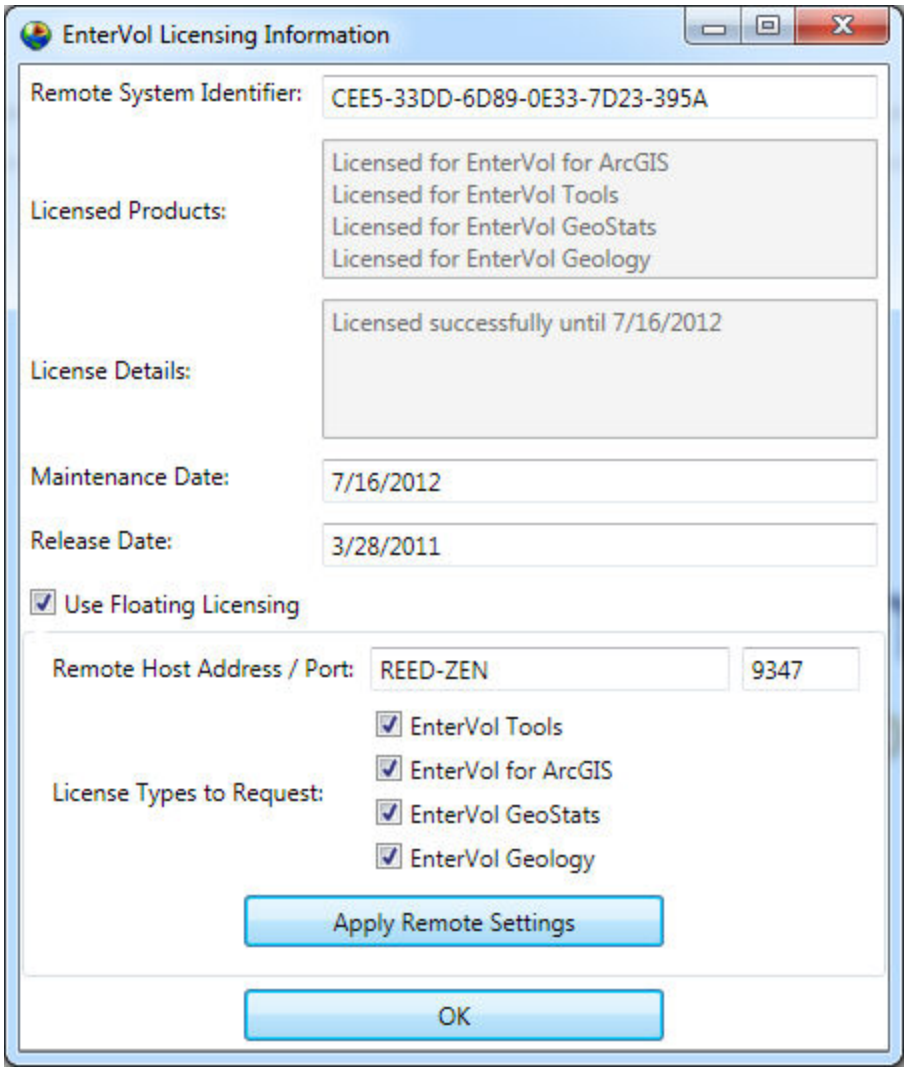

## Client Installation for Concurrent Licenses

If your organization has purchased EnterVol concurrent licenses and properly installed and configured an EnterVol license server on your network, you can follow the simple steps below to configure a client installation for a concurrent license.

- Open the EnterVol Licensing Window and turn on the "Use Floating License" toggle as shown below.




- Enter the Remote Host (Server) Name or IP address
- Select the License Types to Request
- Click "Apply Remote Settings"




If your client computer can connect to a valid server, your licensing window should show something like the above.

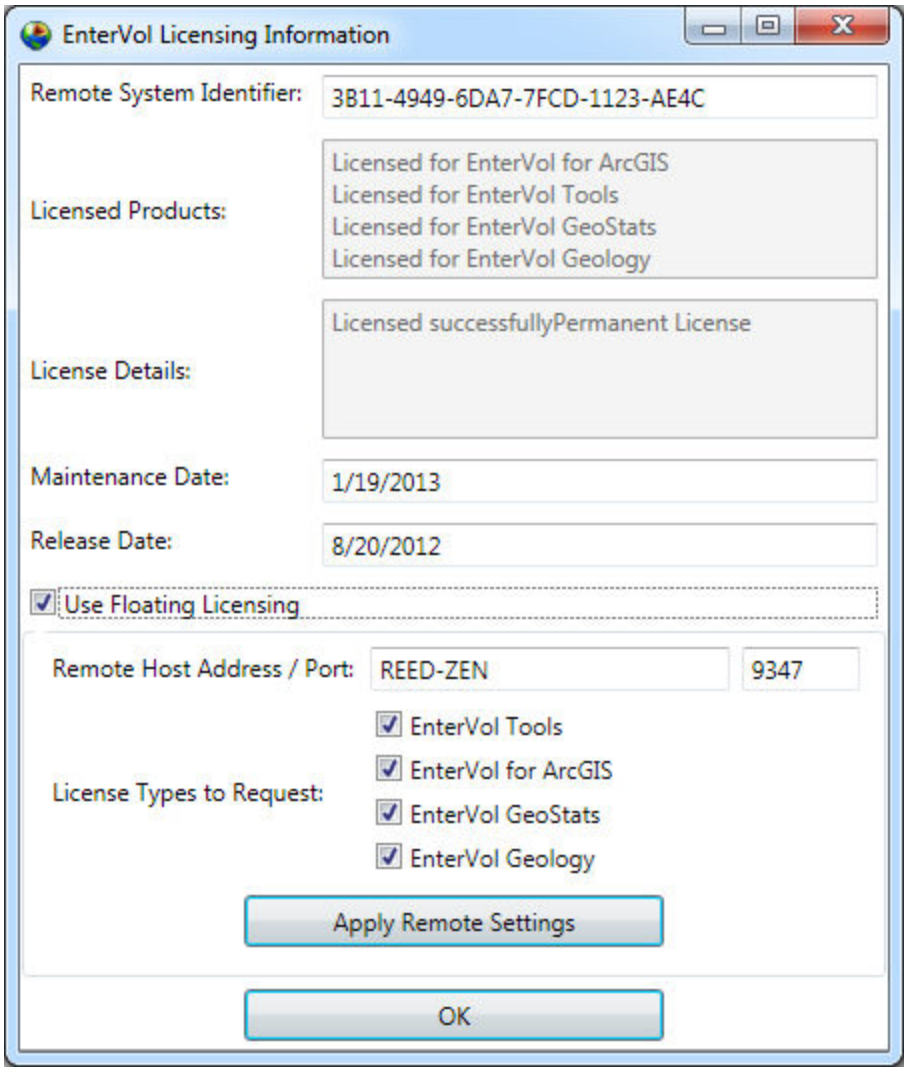
## License Troubleshooting Section

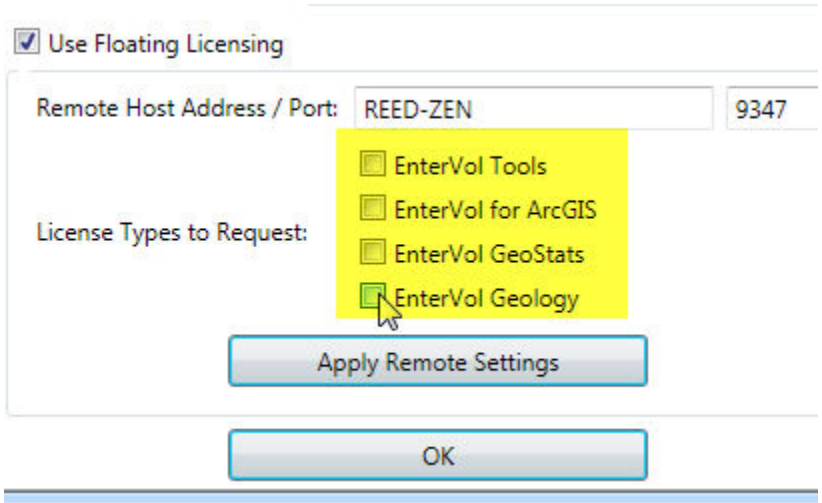
Provided you've followed the installation instructions, you shouldn't need to be here.

However here are a list of common issues:

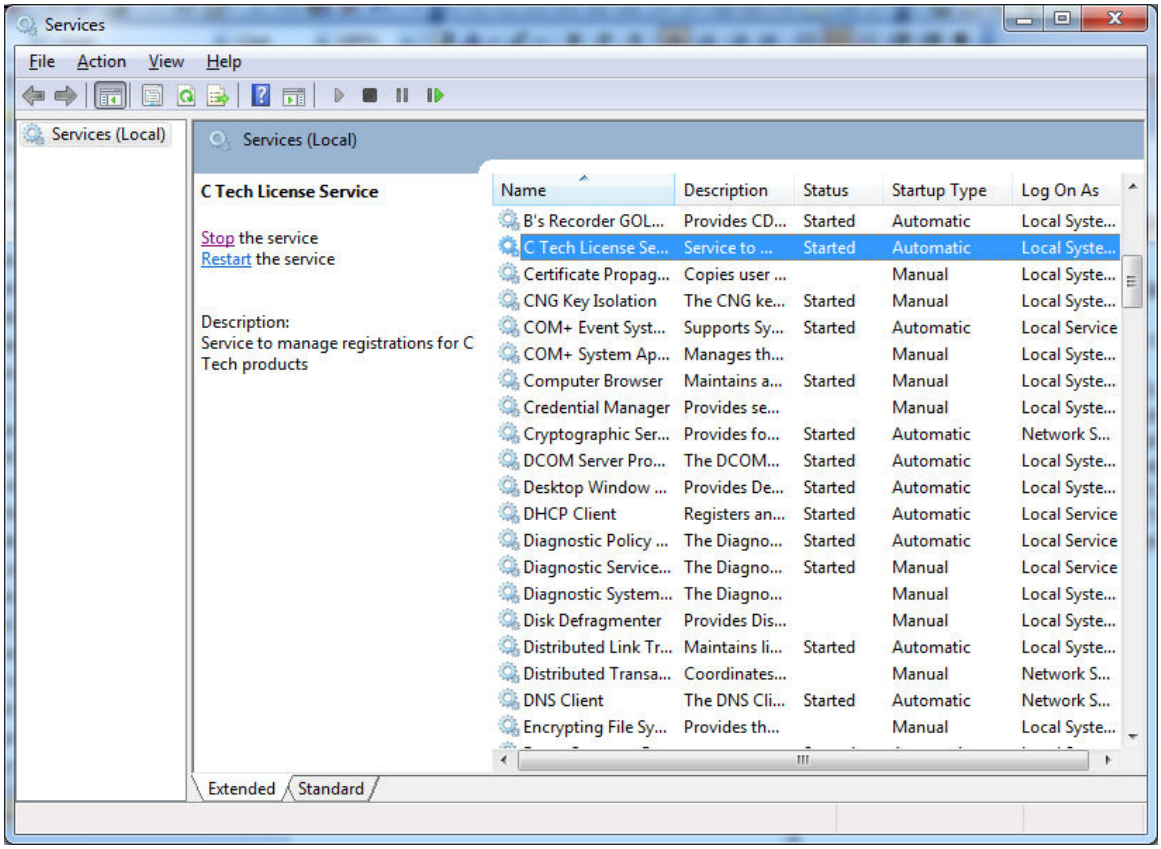
- Fixed Licenses
  - Confirm that the C Tech License Service is started (see image below)
  - Open the EnterVol License window and confirm you have a license that is not expired and has a Maintenance Date compatible with the EnterVol version you are running

    - Please note that you cannot install newer versions of EnterVol during a Trial License period.
- Concurrent Licenses on Server
    - Confirm that the C Tech License Service is started (see image below)
    - Confirm that the Server's Firewall is not blocking port 9347 (see Final Step)
    - Open the EnterVol License Manager and confirm you have licenses that are not expired
- Concurrent Licenses on Client
    - Open the EnterVol License window and confirm:
        - You turn on the toggle: *Use Floating License*
        - You have properly specified your server
        - Your server has licenses that are not expired and has a Maintenance Date compatible with the EnterVol version you are running
        - If you "Cannot connect to the License Server":
            - Try to ping to the server name
            - Confirm that the server's firewall is not blocking port 9347
- Releasing Concurrent Licenses on Client
    - If multiple users are sharing a limited number of EnterVol seats, they must be released.
    - Open the EnterVol License window (see image below)
        - UNCHECK the licenses that you wish to release (see second image below)
        - Click *Apply Remote Settings*
        - Click *OK* to close the window
    - Note: It will take up to 5 minutes for the licenses to be released and available to another user.


EnterVol Licensing Information
Remote System Identifier:
3B11-4949-6DA7-7FCD-1123-AE4C
Licensed Products:
Licensed for EnterVol for ArcGIS
Licensed for EnterVol Tools
Licensed for EnterVol GeoStats
Licensed for EnterVol Geology
License Details:
Licensed successfullyPermanent License
Maintenance Date:
1/19/2013
Release Date:
8/20/2012
Use Floating Licensing
Remote Host Address / Port:
REED-ZEN
9347
License Types to Request:
EnterVol Tools
EnterVol for ArcGIS
EnterVol GeoStats
EnterVol Geology
Apply Remote Settings
OK

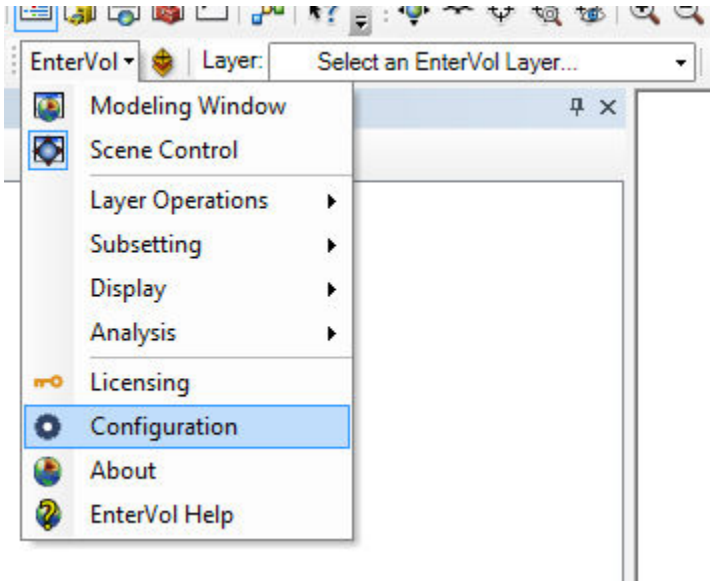

Use Floating Licensing
Remote Host Address / Port:
REED-ZEN
9347
License Types to Request:
EnterVol Tools
EnterVol for ArcGIS
EnterVol GeoStats
EnterVol Geology
Apply Remote Settings
OK

**The *C Tech License Server* must be started in order for EnterVol to run.**



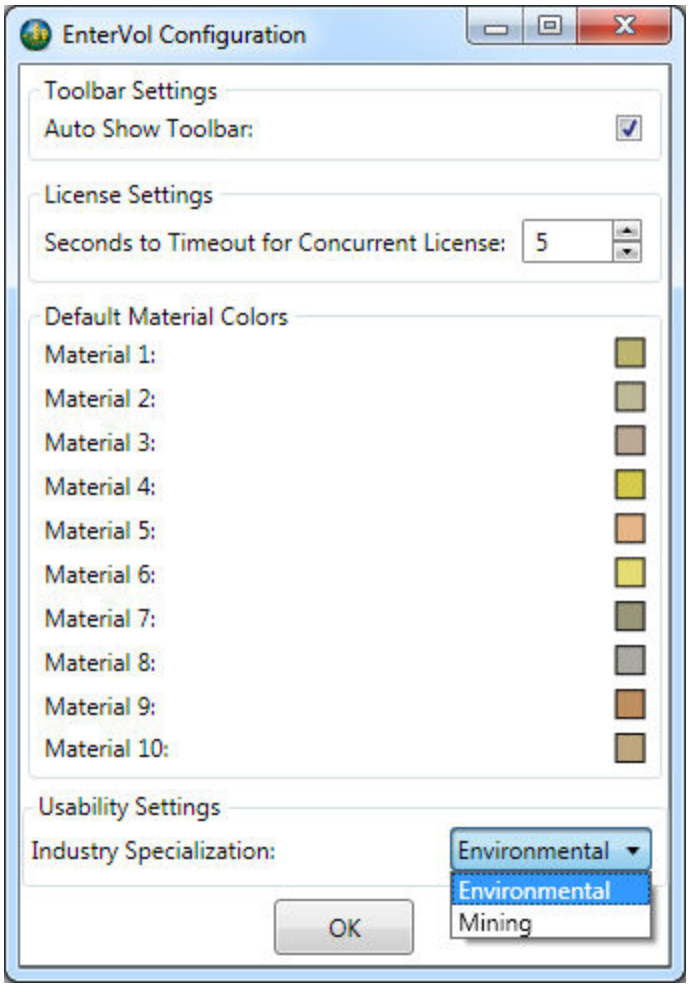
# Configuration

The Configuration window provides some basic settings that affect other operations in EnterVol. Once settings are changed, you must exit ArcScene and restart it for the configuration setting to be set.



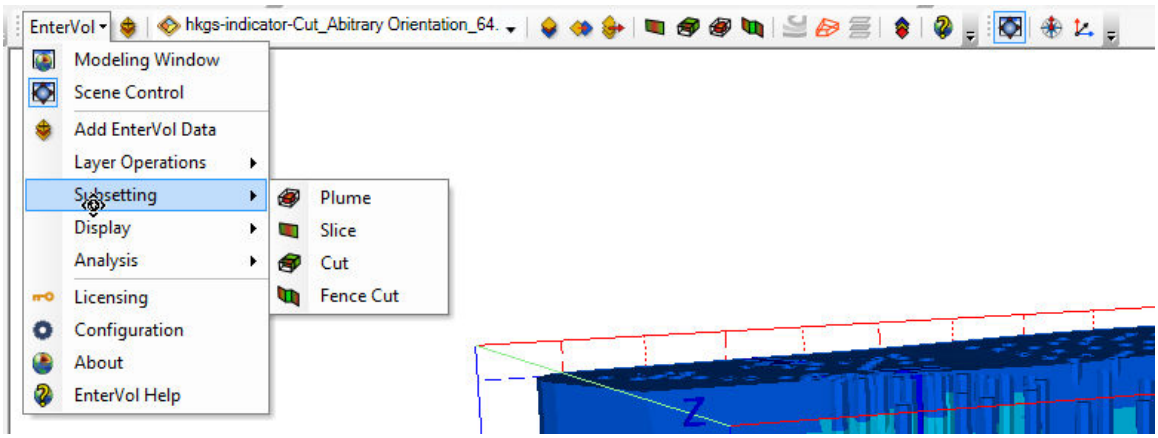

The primary configuration options are:

- "Auto Show Toolbar" which is on by default, shows the EnterVol toolbar by default.
- The Default Material Colors allows you to set standard colors to be used as defaults.
- "**Industry Specialization**" is an important feature. This affects some of the basic terminology (plume vs. ore body) used for tools as well as default units and the reporting in volumetrics. The current options are:
  - Environmental
  - Mining


EnterVol Configuration
Toolbar Settings
Auto Show Toolbar:
License Settings
Seconds to Timeout for Concurrent License:
5
Default Material Colors
Material 1:
Material 2:
Material 3:
Material 4:
Material 5:
Material 6:
Material 7:
Material 8:
Material 9:
Material 10:
Usability Settings
Industry Specialization:
Environmental
Environmental
Mining
OK

# Using EnterVol

EnterVol provides multiple ways to access its functionality. If you turn on the EnterVol Toolbar, it provides access to all EnterVol functionality:




**Tools and operations can be accessed in three different ways:**

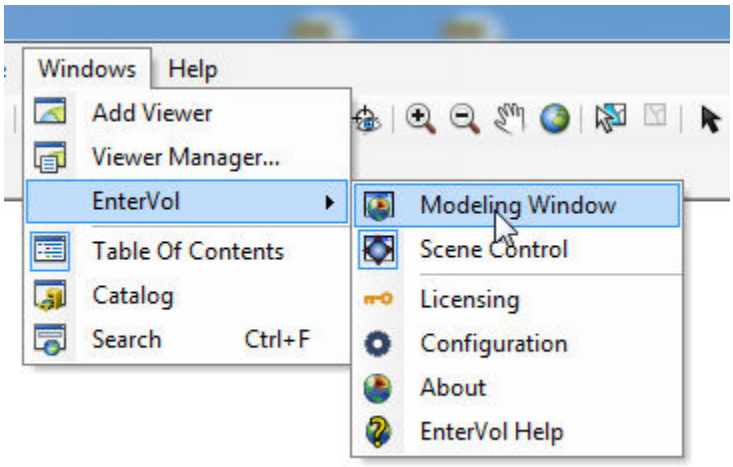
1. The menu structure shown above
2. Individual icons. 
3. By *Right-Clicking* on the Scene or Individual Layers in the Table of Contents (TOC) shown at the end of this topic

The Operations Include:

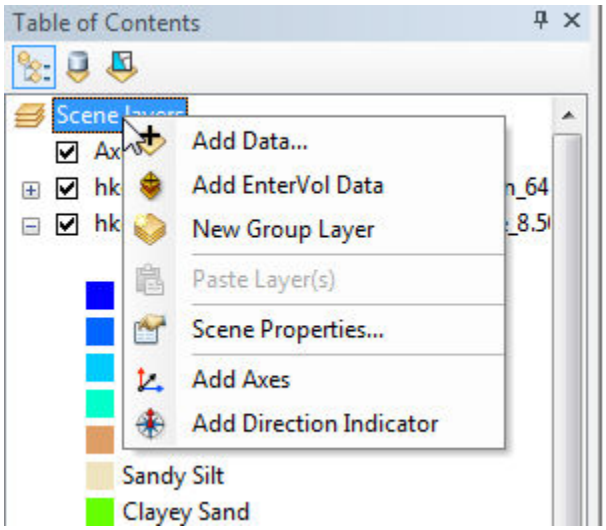
- **Modeling Window**
  - Loading or Creating Projects
  - Importing Data
  - Building 2D, Surface and 3D Grids
  - Building Geologic models with Stratigraphy or Lithology
  - Mapping Analytical Data to 3D Grids
- **Scene Control**: Opens the EnterVol Scene Control window
- **Add EnterVol Data**: Directly import C Tech legacy grids and data files
- **Layer Operations**:
  - Set Symbology
  - Copy Symbology
  - Export to EVS: Outputs objects in C Tech format for use in EVS & MVS
- **Subsetting**:
  - Plume
  - Slice
  - Cut
  - Fence Cut
- **Display**:

  - Contour
  - Isolines
  - Extract Points
  - Extract Edges
  - Explode Layers
- **Analysis**: Mass and Volume
- *Licensing, Configuration, About and EnterVol Help*

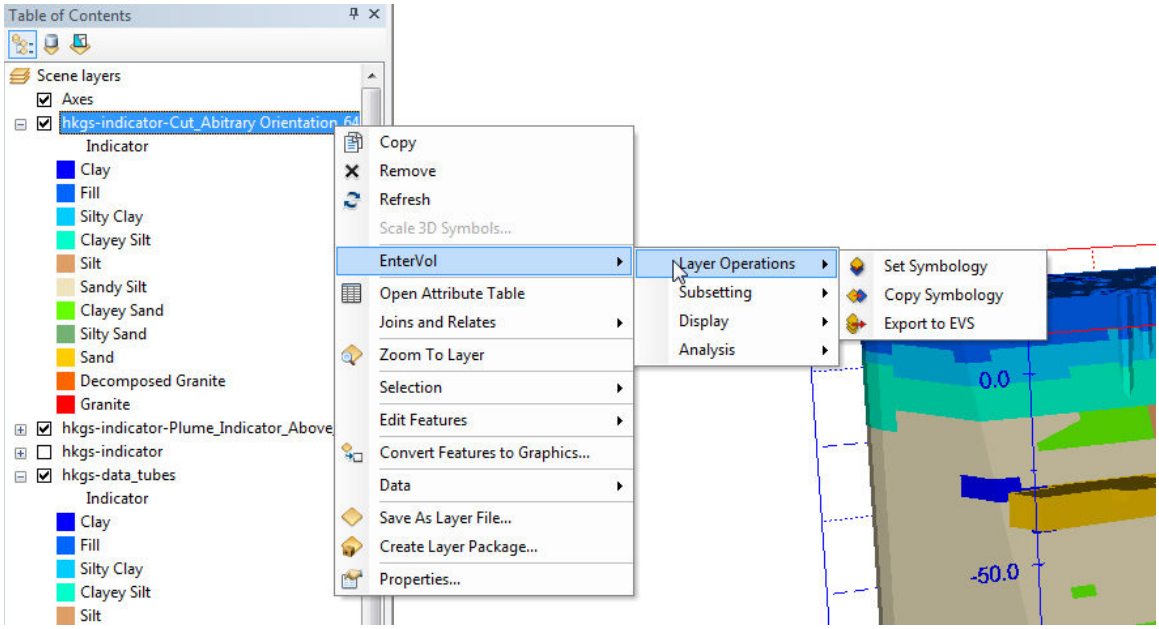
With the EnterVol toolbar not visible, all operations may still be accessed. Access to all EnterVol windows are available in the Pull-Down menus under Windows:

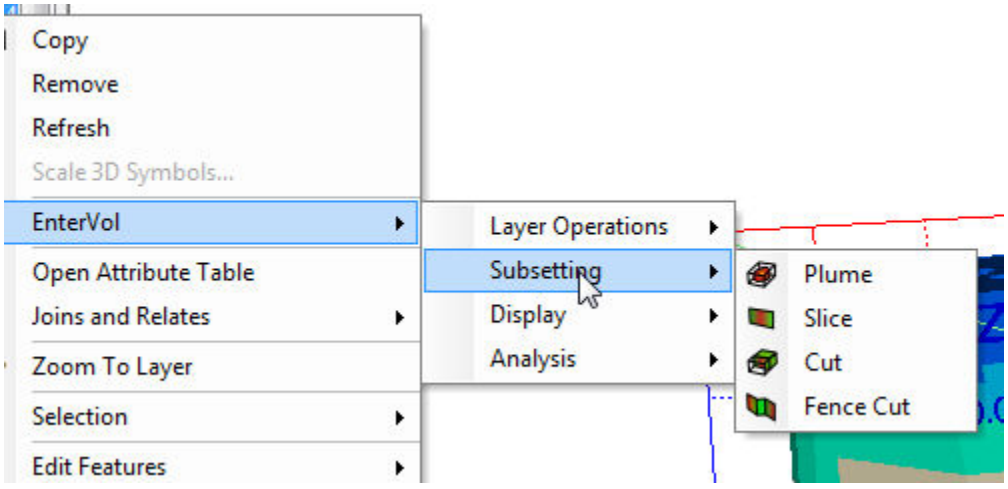


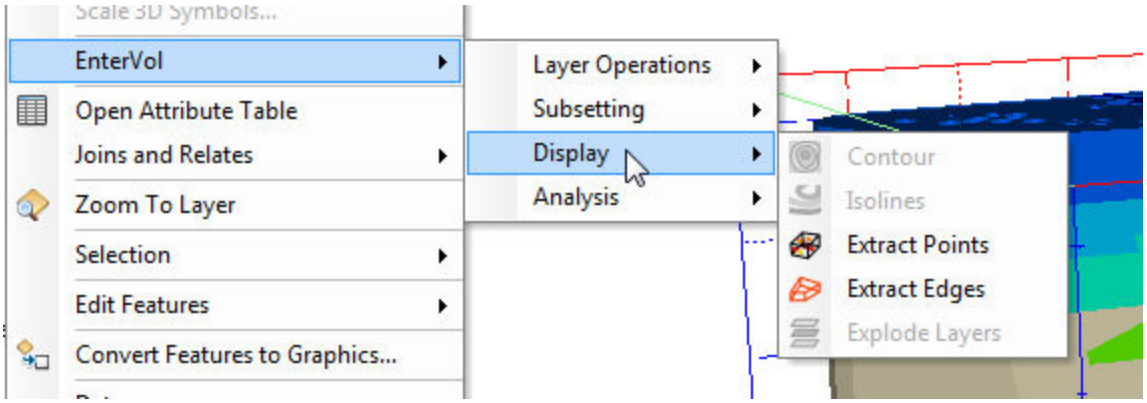

All other operations are accessed with Right Clicking in the TOC. If you Right-Click on *Scene Layers* you can access *Add EnterVol Data, Add Axes and Add Direction Indicator.*

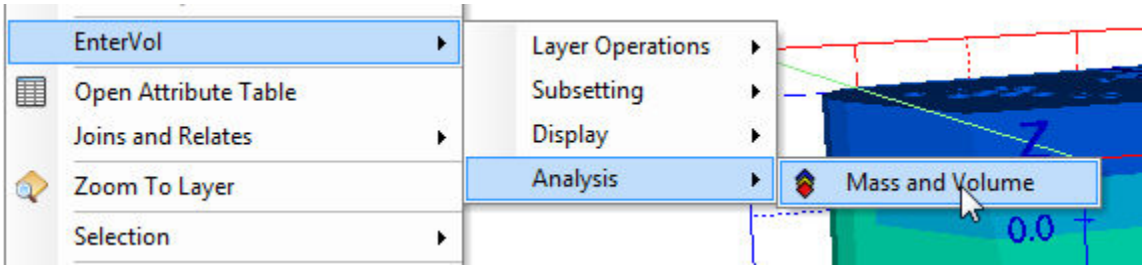


If you Right-Click on Individual *Layers* in the TOC you can access all other EnterVol operations which would be relevant to that layer. Different types of layers will have different functionality available. For example, you can't put isolines on models colored by unique values (like geologic material), but you could on a contaminant plume. Below are some examples:

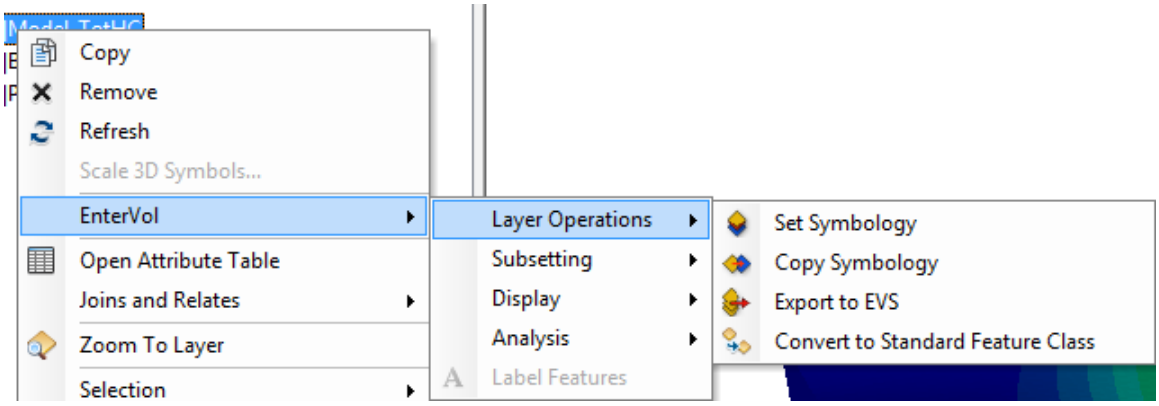



Scale 3D Symbols...
EnterVol
Open Attribute Table
Joins and Relates
Zoom To Layer
Selection
Edit Features
Convert Features to Graphics...
Layer Operations
Subsetting
Display
Analysis
Contour
Isolines
Extract Points
Extract Edges
Explode Layers


EnterVol
Open Attribute Table
Joins and Relates
Zoom To Layer
Selection
Layer Operations
Subsetting
Display
Analysis
Mass and Volume
0.0

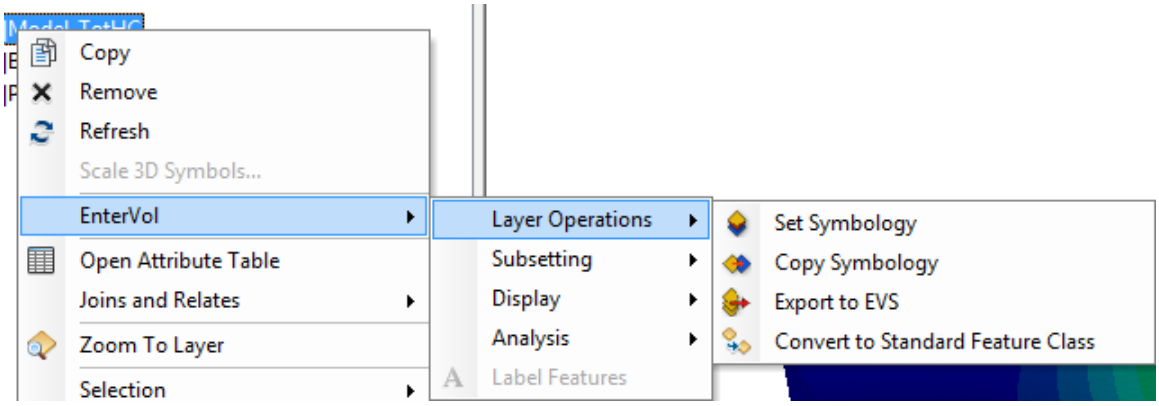
# Layer Operations




Layer Operations allow you to import any element of an EnterVol Project Files (.EVP), EnterVol Model Files (.EVMs), EVS Field Files (.EFF, .EFZ and .EFB) and various other C Tech file formats, and to set and copy the symbology to be applied to those objects.

Most EFFs contain features which can have multiple attributes. In fact, C Tech's EFFs have unique characteristics that set them apart from traditional ESRI objects. EFF objects of all types (points, lines, surfaces and volumes) can contain both cell attributes (traditional ESRI paradigm) and nodal data.

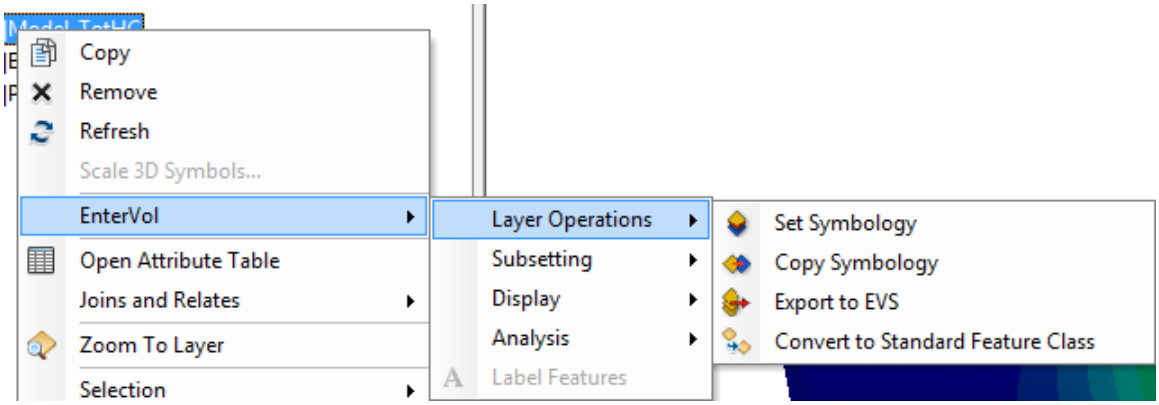
## Layer Operations




Layer Operations allow you to import any element of an EnterVol Project Files (.EVP), EnterVol Model Files (.EVMs), EVS Field Files (.EFF, .EFZ and .EFB) and various other C Tech file formats, and to set and copy the symbology to be applied to those objects.

Most EFFs contain features which can have multiple attributes. In fact, C Tech's EFFs have unique characteristics that set them apart from traditional ESRI objects. EFF objects of all types (points, lines, surfaces and volumes) can contain both cell attributes (traditional ESRI paradigm) and nodal data.

## Add EnterVol Data

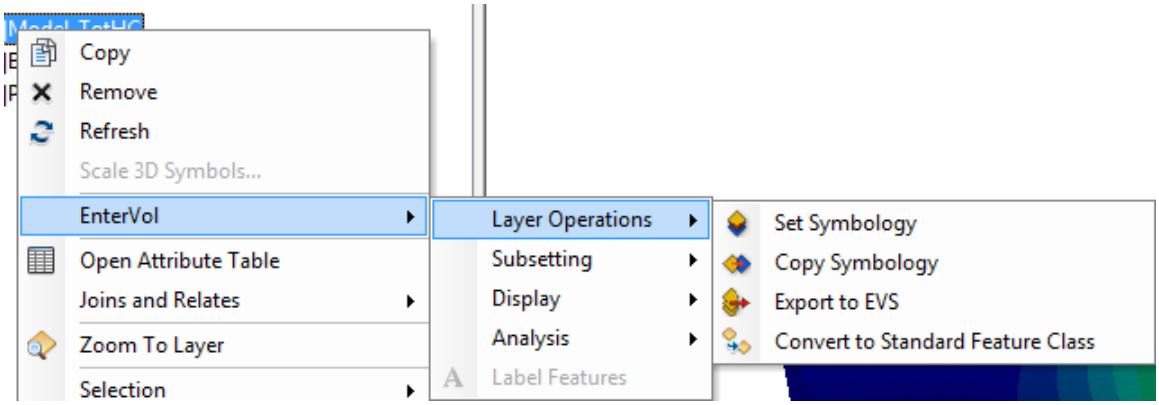



Using this button, you can add various types of data to your scenes:

- EnterVol Project Files (.EVP) can contain all of the file types listed below. Selecting an EVP file allows you to select any one of them.
- EnterVol Model Files (.EVM) are a superset of EVS Field Files and are the default file format for all models created in EnterVol.
- EFF (EVS Field Files) representing both volumetric grids, surfaces and/or lines. Includes .EFF (ASCII), .EFZ (zip archive) and .EFB (binary) formats
- GEO: Geologic boring data representing hierarchical geology data collected in vertical borings
- GMF: C Tech's Geology Multi-File allows you to import a cloud of points corresponding to uncorrelated points on one or more surfaces representing hierarchical geology data
- PGF: Pre-Geology data representing 3D boring logs (can be non-vertical)
- apdv: C Tech's chemistry (analyte) file format for contamination and ore grade data representing point samples (can be non-vertical)
- aidv: C Tech's chemistry (analyte) file format for contamination and ore grade data representing samples collected over an interval (can be non-vertical)

When EnterVol data is added, a summary of the data and settings used to create it is added to the Layer Properties under General.Description.

## Add EnterVol Data



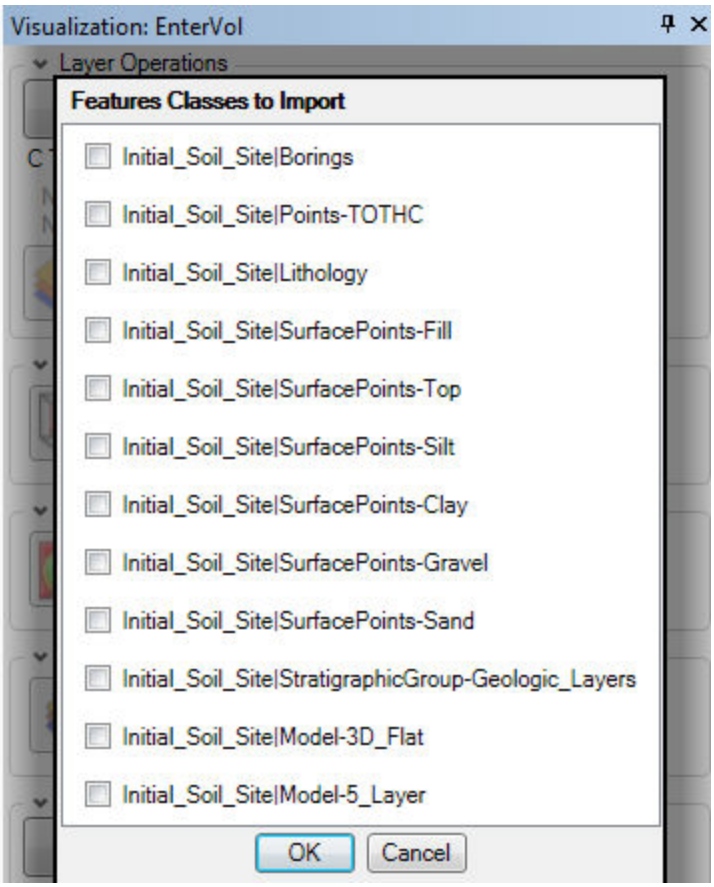

Using this button, you can add various types of data to your scenes:

- EnterVol Project Files (.EVP) can contain all of the file types listed below. Selecting an EVP file allows you to select any one of them.
- EnterVol Model Files (.EVM) are a superset of EVS Field Files and are the default file format for all models created in EnterVol.
- EFF (EVS Field Files) representing both volumetric grids, surfaces and/or lines. Includes .EFF (ASCII), .EFZ (zip archive) and .EFB (binary) formats
- GEO: Geologic boring data representing hierarchical geology data collected in vertical borings
- GMF: C Tech's Geology Multi-File allows you to import a cloud of points corresponding to uncorrelated points on one or more surfaces representing hierarchical geology data
- PGF: Pre-Geology data representing 3D boring logs (can be non-vertical)
- apdv: C Tech's chemistry (analyte) file format for contamination and ore grade data representing point samples (can be non-vertical)
- aidv: C Tech's chemistry (analyte) file format for contamination and ore grade data representing samples collected over an interval (can be non-vertical)

When EnterVol data is added, a summary of the data and settings used to create it is added to the Layer Properties under General.Description.

### Add EnterVol Project File Data

EnterVol Project Files (.EVP) can contain all of the file types listed here. Selecting an EVP file allows you to select any one of them.




When EnterVol data is added, a summary of the data and settings used to create it is added to the Layer Properties under General.Description.

### Add EnterVol Models EVM

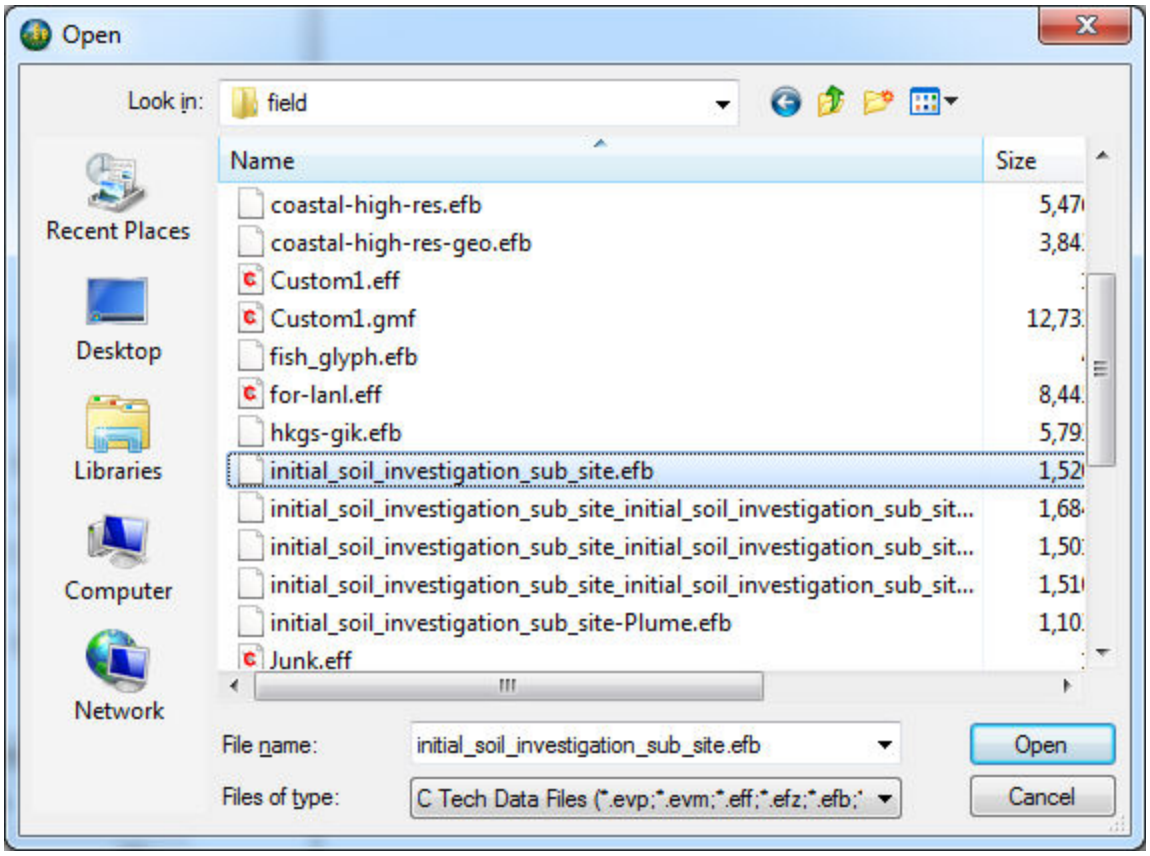
EnterVol Model Files (.EVM) are a superset of EVS Field Files and are the default file format for all models created in EnterVol. They are typically stored in the EnterVol Project File and can be selected from the check list that opens when an EVP file is selected.

In general loading a .EVM file is equivalent to loading a .EFF (or .EFB) file.

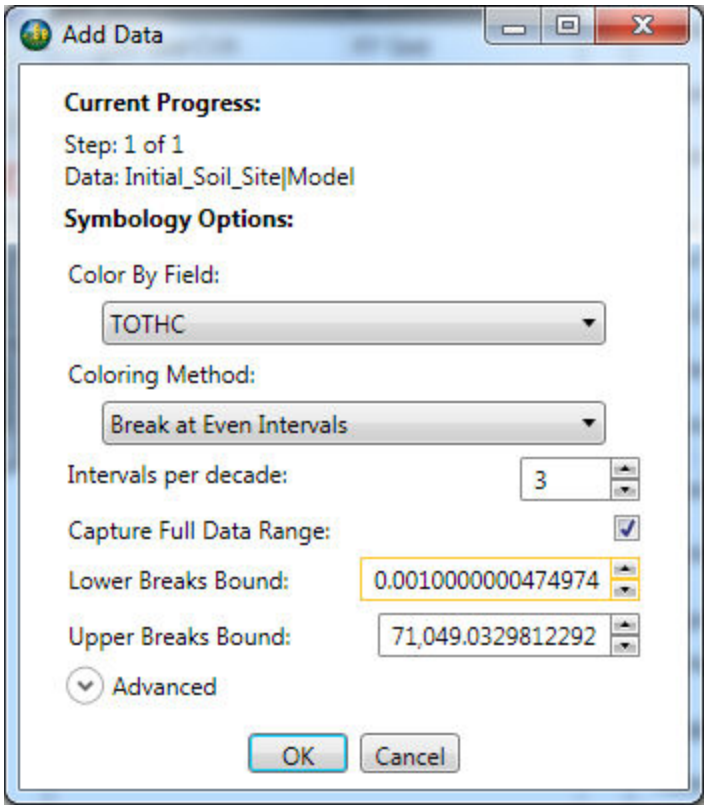
When EnterVol data is added, a summary of the data and settings used to create it is added to the Layer Properties under General.Description.

### Add C Tech EFF

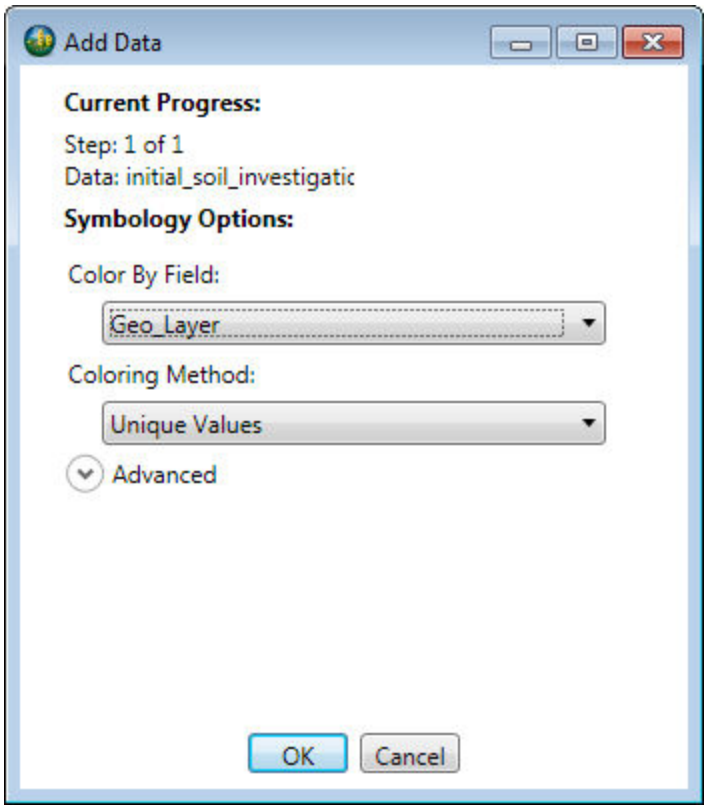

Select the *Add EnterVol Data* button above to add a new EFF to your ArcGIS project.



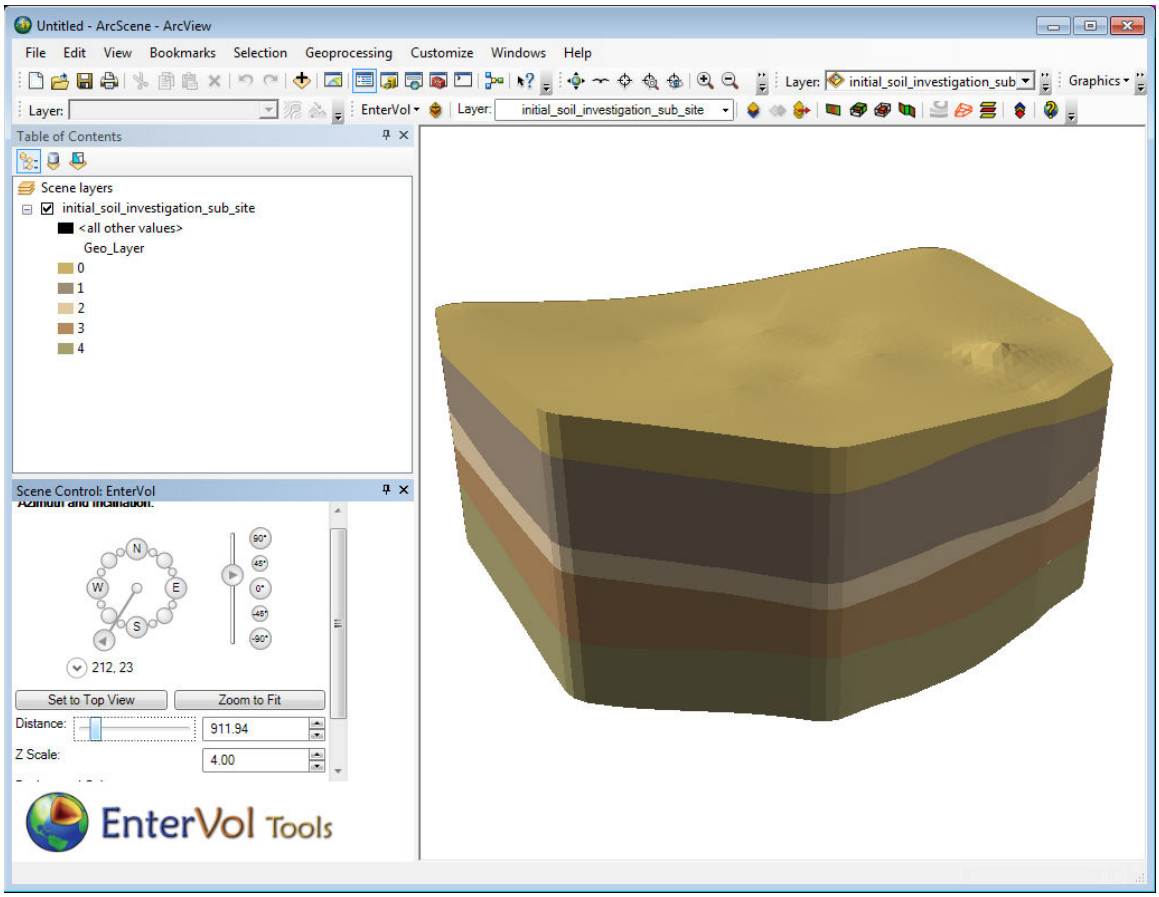

After the EFF loads, a new window will appear to allow you to set the initial symbology:



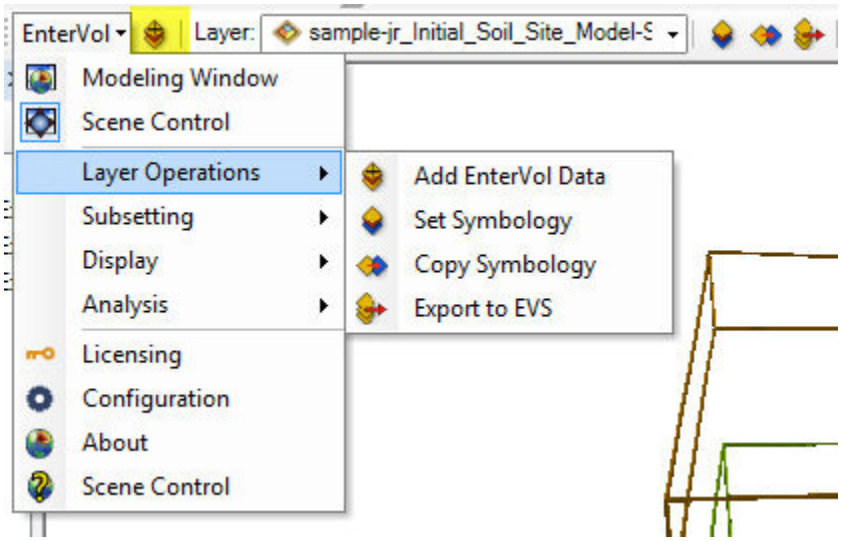

If we choose the first pull-down menu we can see that this EFF has several attributes which can be used for coloring:



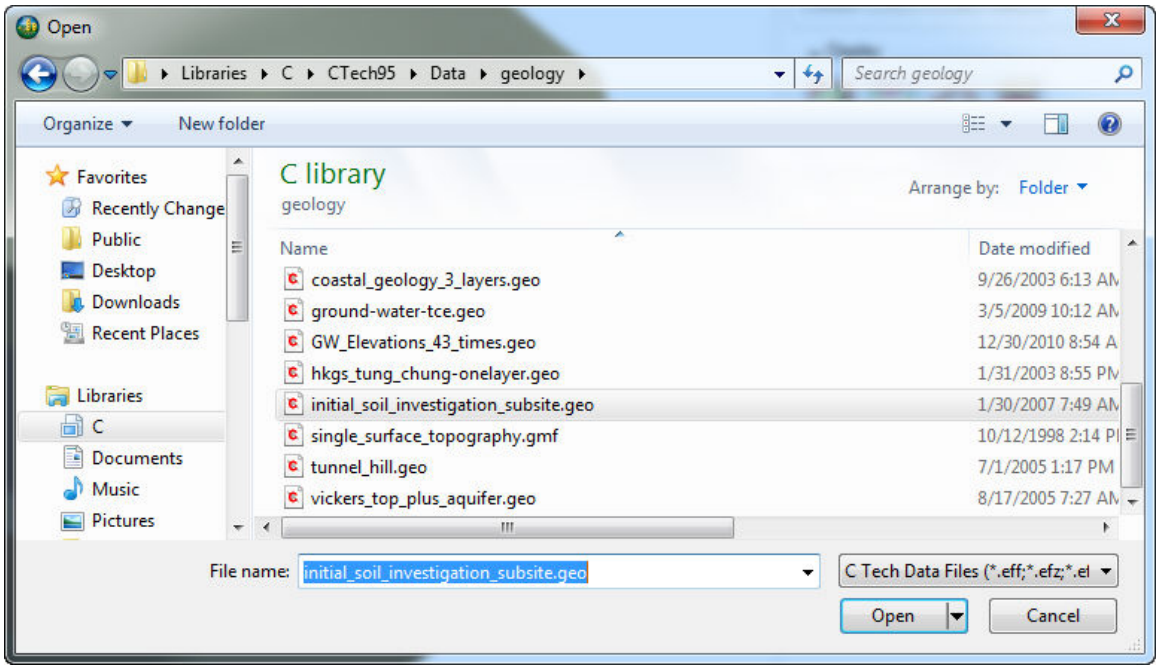

However, if we choose Geo_Layer, the results in our ArcScene window will be:

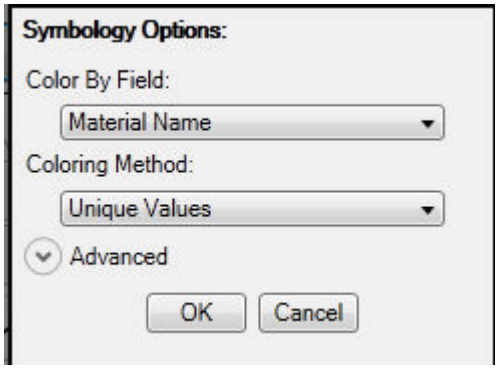



## Add C Tech GEO: Geology File




Beginning with the scene at the end of the Add C Tech EFF topic, press the Add EnterVol Data button and select the file initial_soil_investigation_subsite.geo.

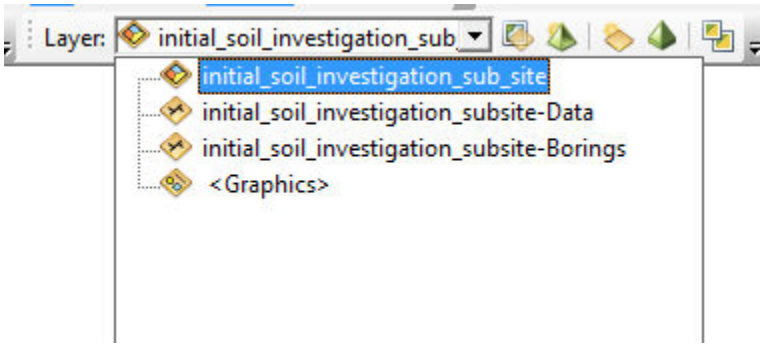



Accept the default values 

and geologic borings will be added to your scene.

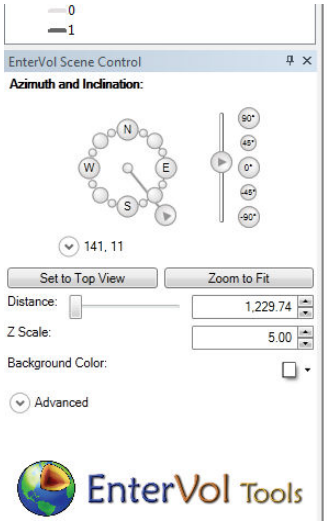
However, since the volumetric model is opaque, we can only see a hint of them on the surface.

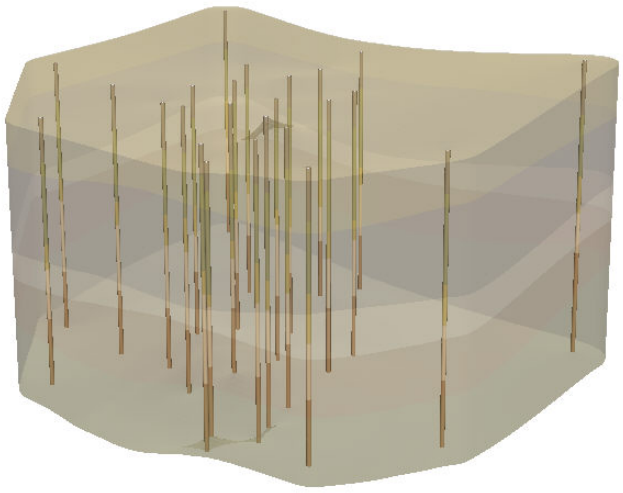
From the *3D Effects* toolbar select the volumetric model



and then adjust its transparency to 65% 

to reveal the borings

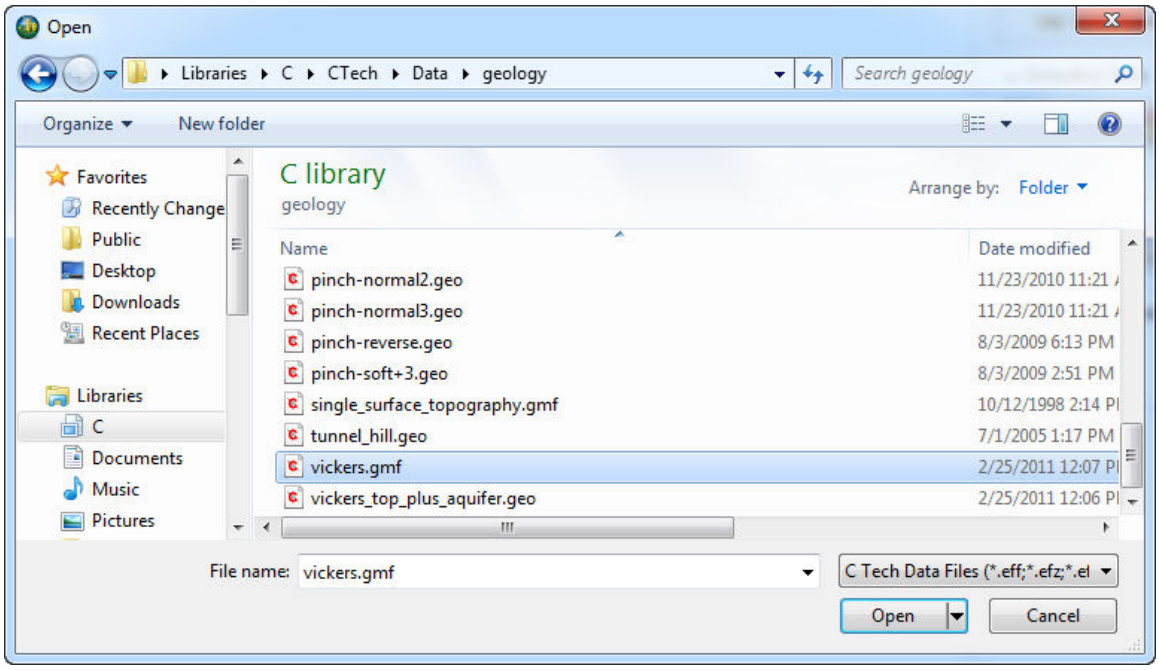


**Add C Tech GMF**

Let's begin with a new project (scene).

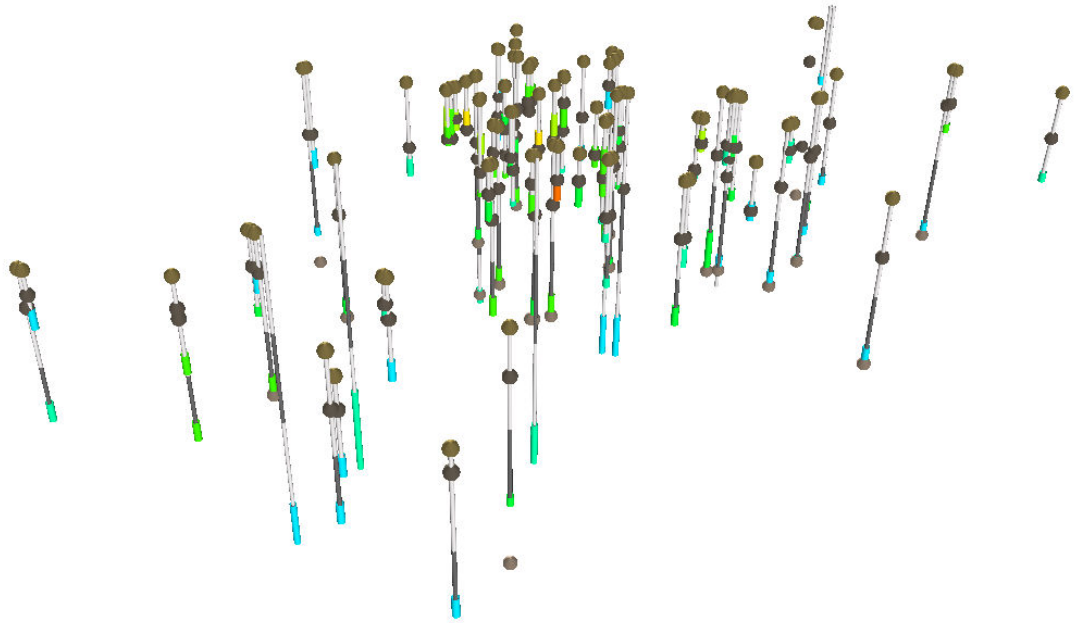
Click the Add EnterVol Data button  and browse to find the file vickers.gmf




Let's accept the default coloring by surface number to get

This will get more interesting if we add the aidv file from the Add C Tech aidv: Groundwater Chemistry topic.

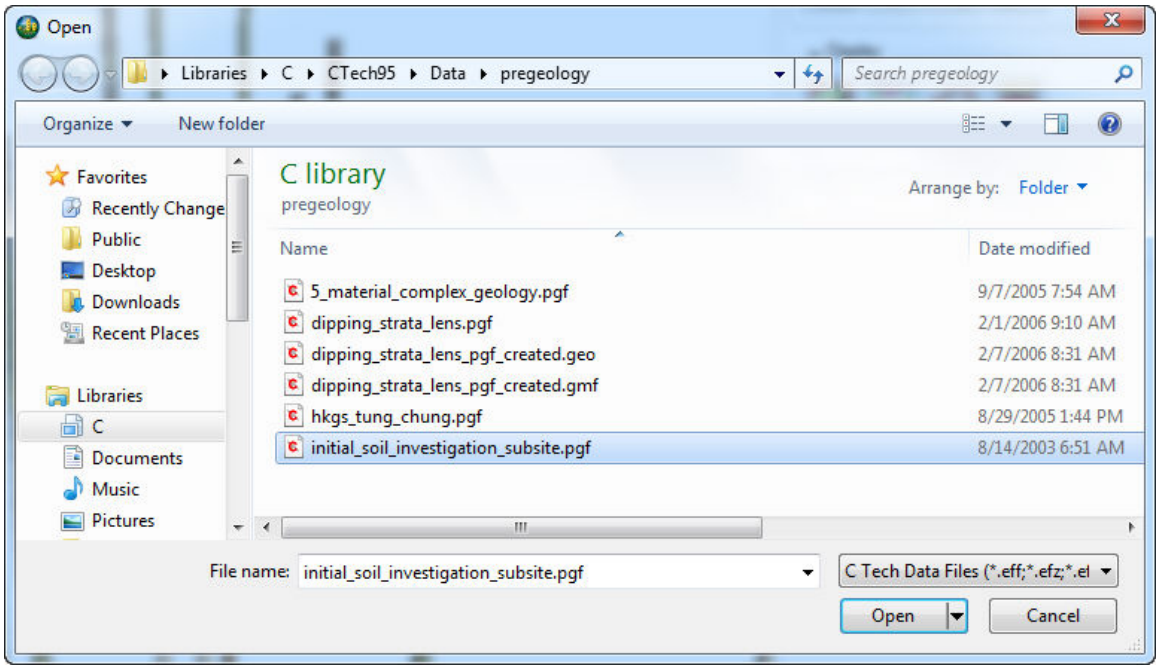
With that added we'll see:

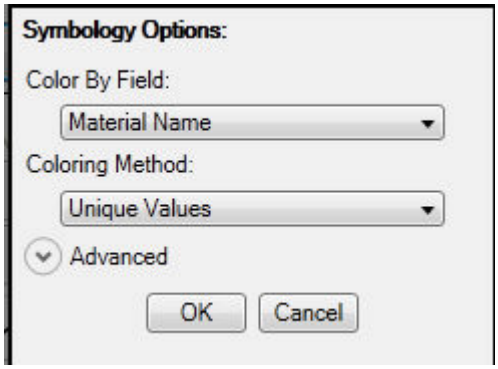


This give us the ability to see how the geology data correlates with the groundwater data.

### Add C Tech Pre-Geology File

Beginning with the scene at the end of the Add C Tech EFF topic, press the Add EnterVol Data button and select the file initial_soil_investigation_subsite.pgf.

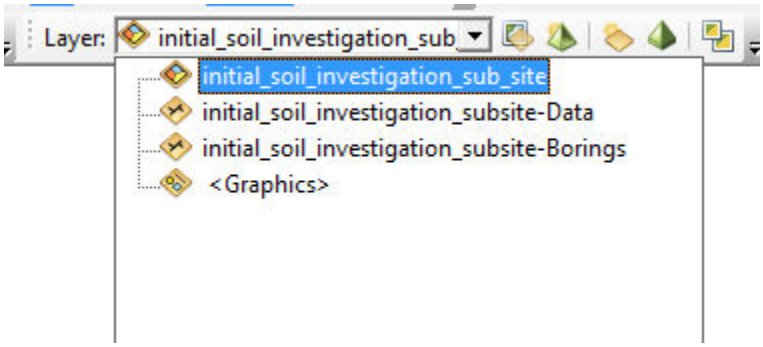



Accept the default values 

and geologic borings will be added to your scene.

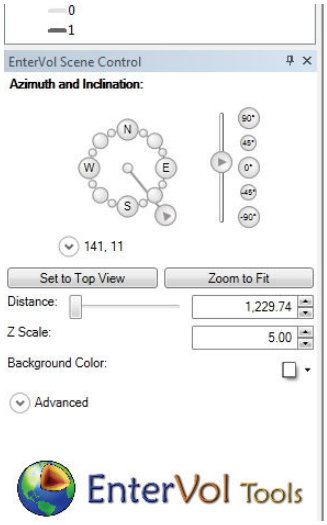
However, since the volumetric model is opaque, we can only see a hint of them on the surface.

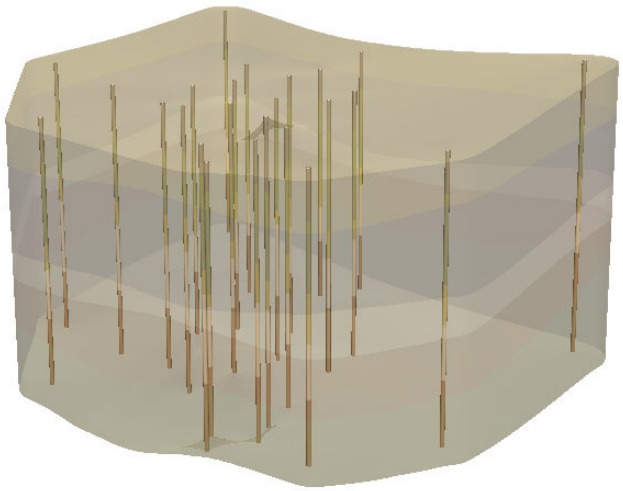
From the *3D Effects* toolbar select the volumetric model



and then adjust its transparency to 65% 

to reveal the borings



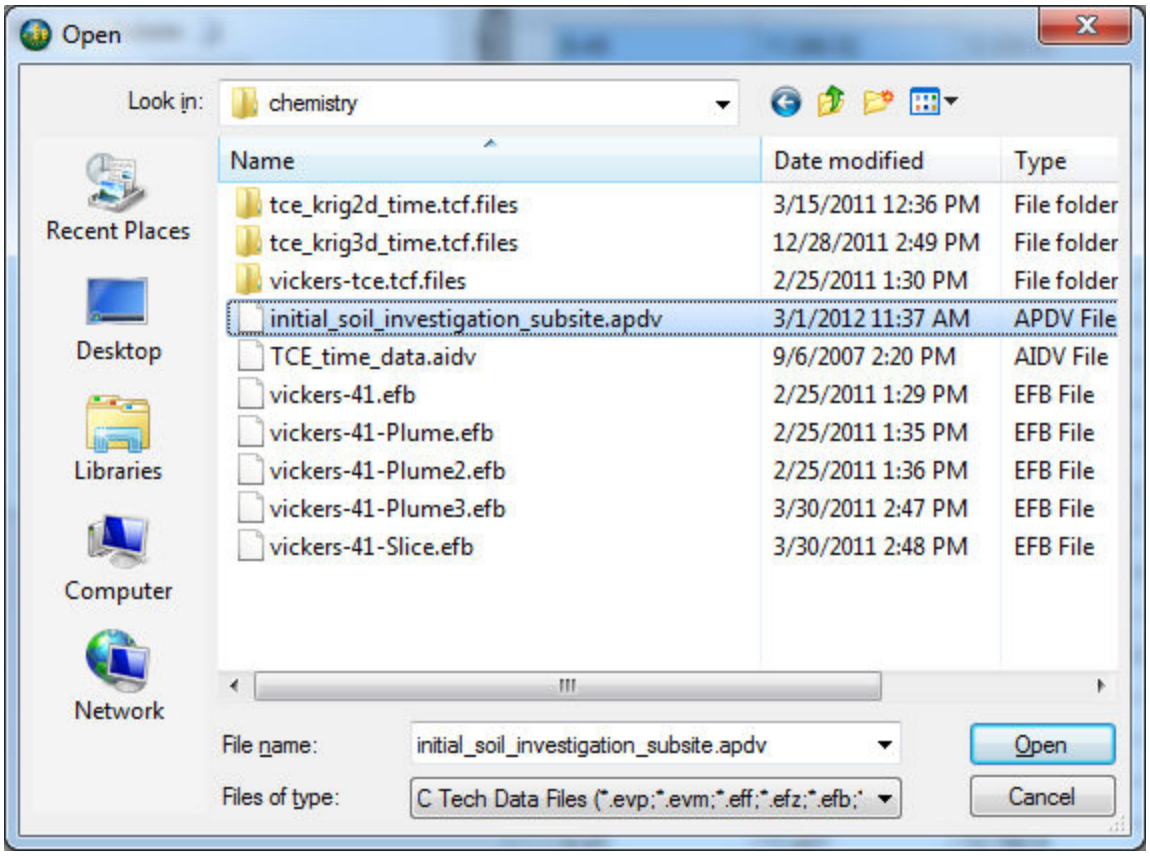



## Add C Tech APDV: Analytical Point Data Value

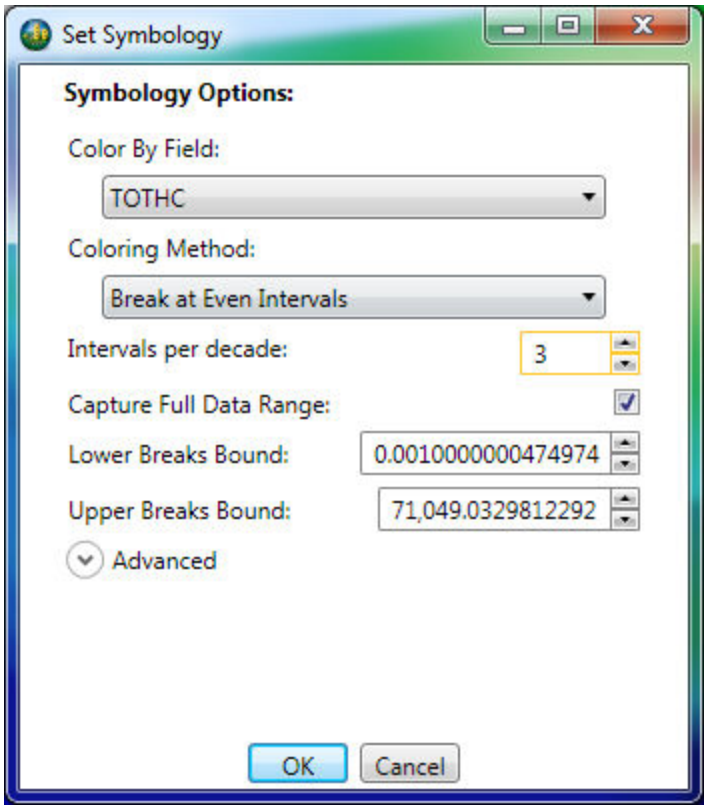
If you start with the project state at the end of the Set Symbology topic, we're ready to add our sample data to the scene.

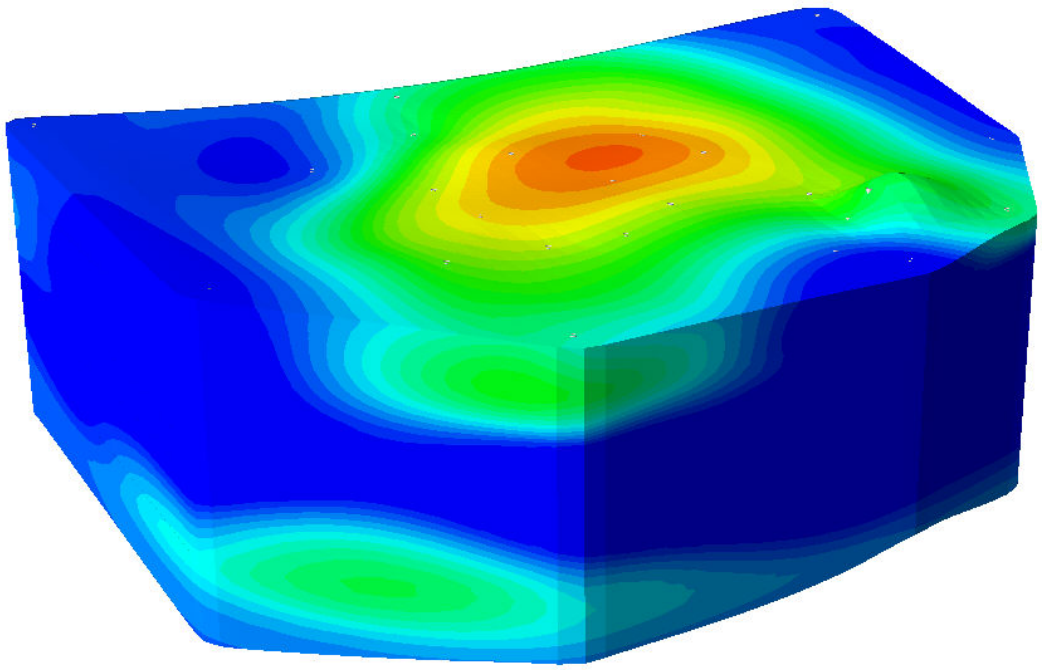
The model that we've added was created with two C Tech Data files. One for the geology (5 stratigraphic layers) and the other for Total Hydrocarbon soil contamination (TOTHC). Our apdv chemistry file format can be used for any volumetrically measured parameter where one or more measurements are taken at a point vs. over a length of a boring or well screen (in that case use our aidv format).

Press the Add EnterVol Data button and browse to find the file initial_soil_investigation_subsite.apdv




We'll accept the defaults since they match the EFF data previously added:

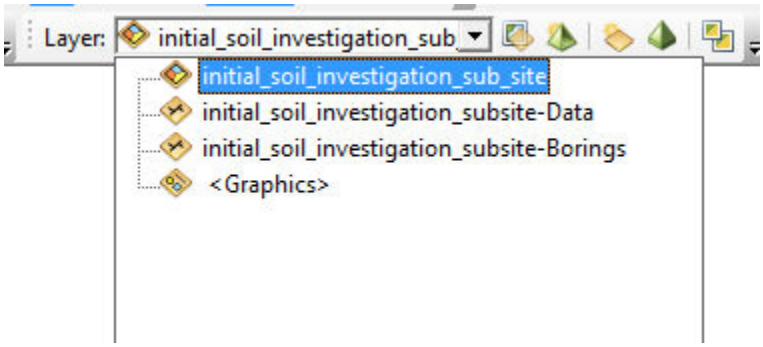

Set Symbology
Symbology Options:
Color By Field:
TOTHC
Coloring Method:
Break at Even Intervals
Intervals per decade:
3
Capture Full Data Range:
Lower Breaks Bound:
0.0010000000474974
Upper Breaks Bound:
71,049.0329812292
Advanced
OK
Cancel



Has our data actually been added? You can see some evidence of boring peaking out of the ground surface since the tubes representing them have horizontal tops which partially extend from the non-level surface.

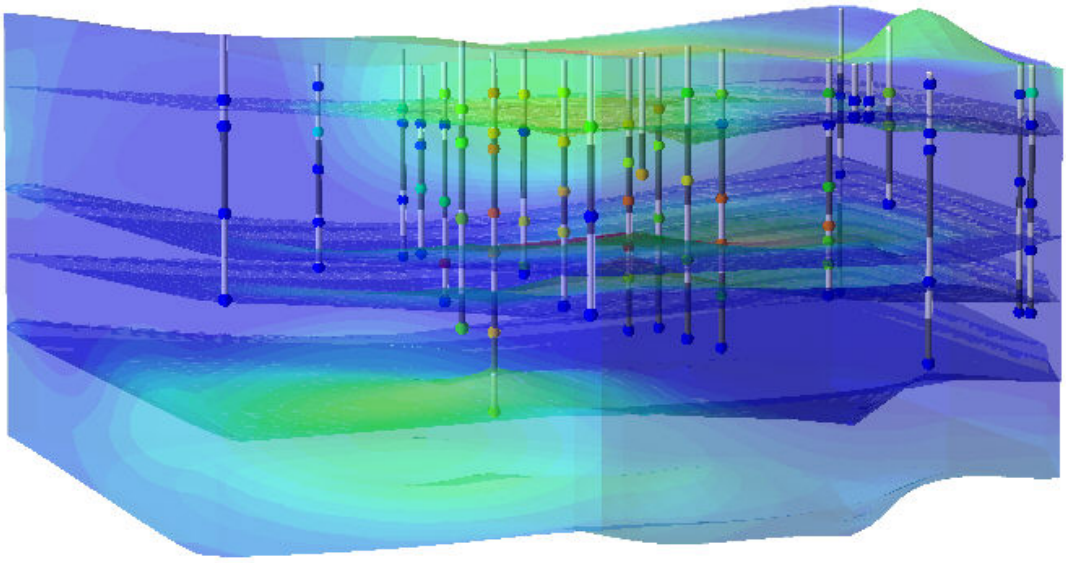
However, since the volumetric model is opaque, we can only see a hint of them on the surface.

From the *3D Effects* toolbar select the volumetric model




and then adjust its transparency to 65% 

to reveal the borings



Please note that the borings are actually two independent layers in your scene

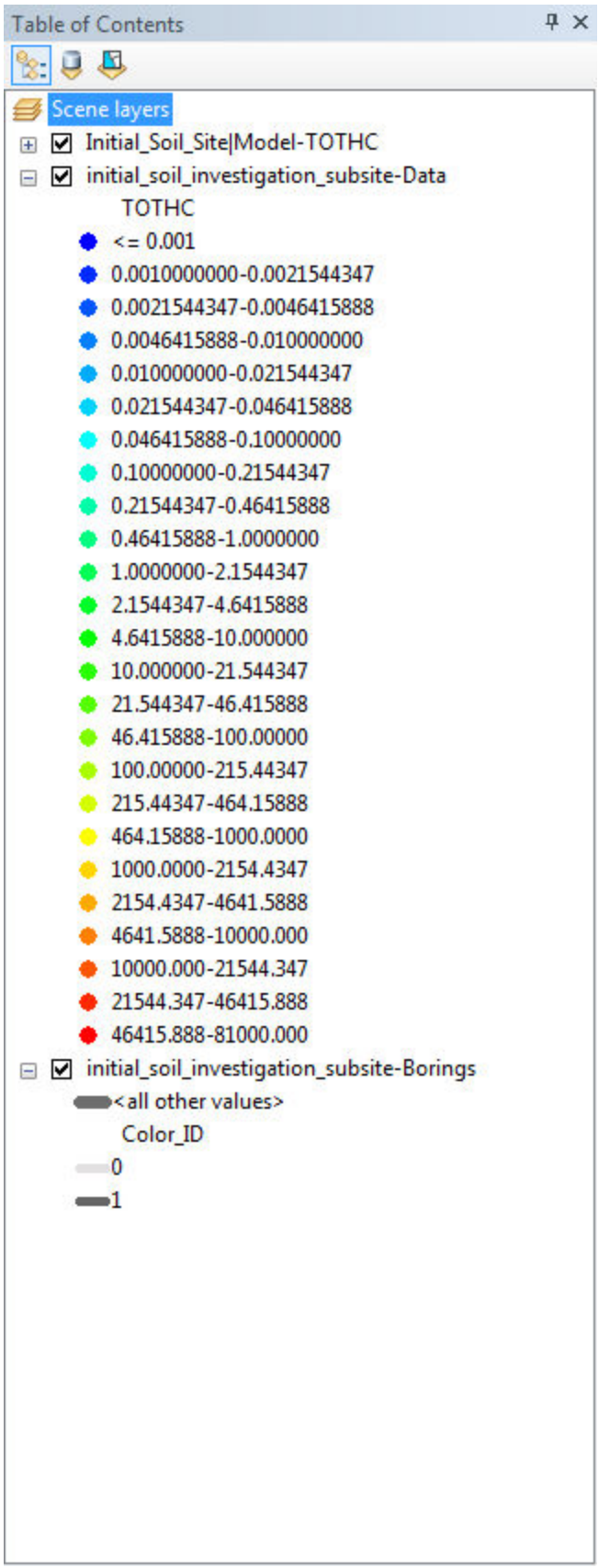


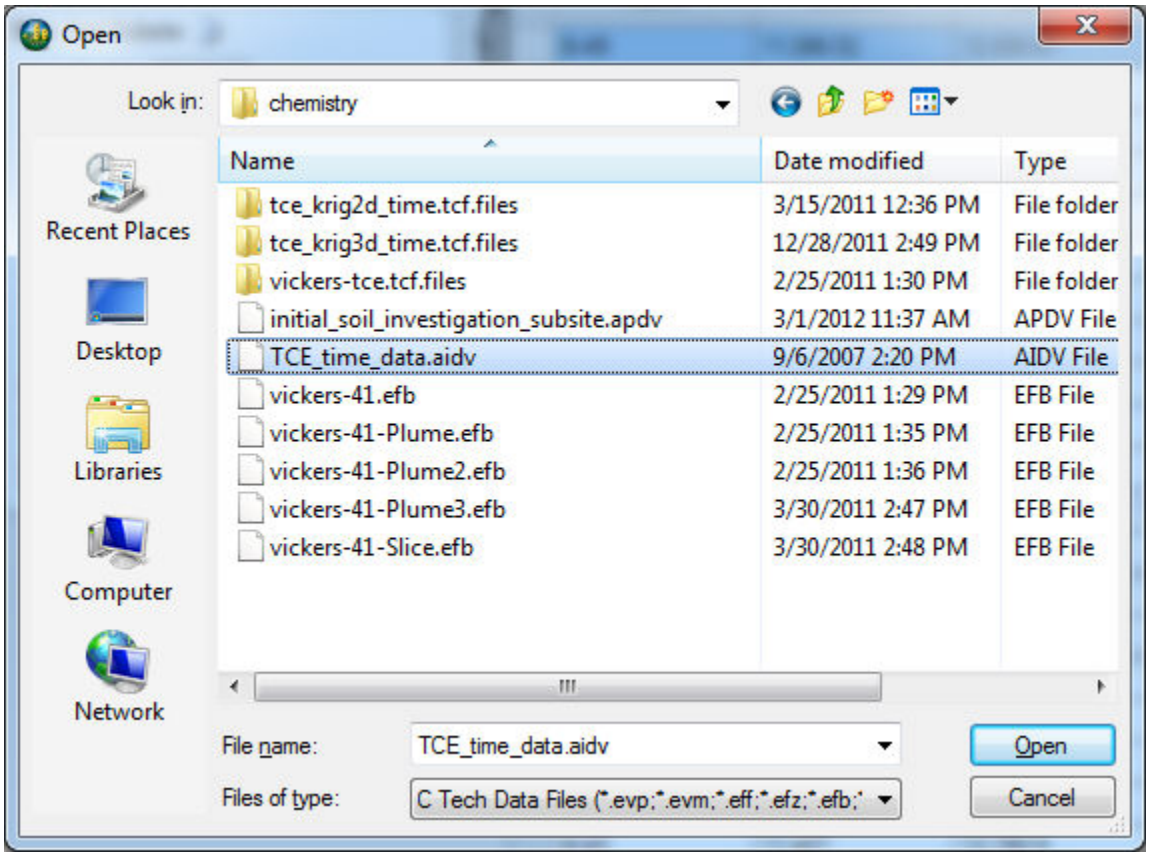
Table of Contents
Scene layers
Initial_Soil_Site|Model-TOTHC
initial_soil_investigation_subsite-Data
TOTHC
<= 0.001
0.0010000000-0.0021544347
0.0021544347-0.0046415888
0.0046415888-0.010000000
0.010000000-0.021544347
0.021544347-0.046415888
0.046415888-0.10000000
0.10000000-0.21544347
0.21544347-0.46415888
0.46415888-1.0000000
1.0000000-2.1544347
2.1544347-4.6415888
4.6415888-10.000000
10.000000-21.544347
21.544347-46.415888
46.415888-100.00000
100.00000-215.44347
215.44347-464.15888
464.15888-1000.0000
1000.0000-2154.4347
2154.4347-4641.5888
4641.5888-10000.000
10000.000-21544.347
21544.347-46415.888
46415.888-81000.000
initial_soil_investigation_subsite-Borings
<all other values>
Color_ID
0
1

The first layer is the data represented as spheres. The second is the borings represented as lines, with a radius...making them tubes.

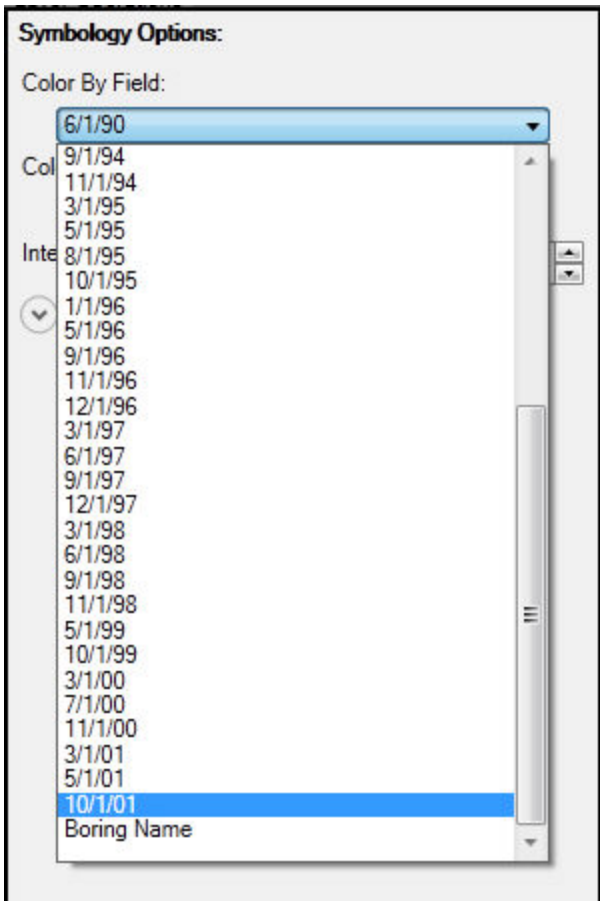
### Add C Tech AIDV: Analytical Interval Data Value

Let's begin with a new project (scene).

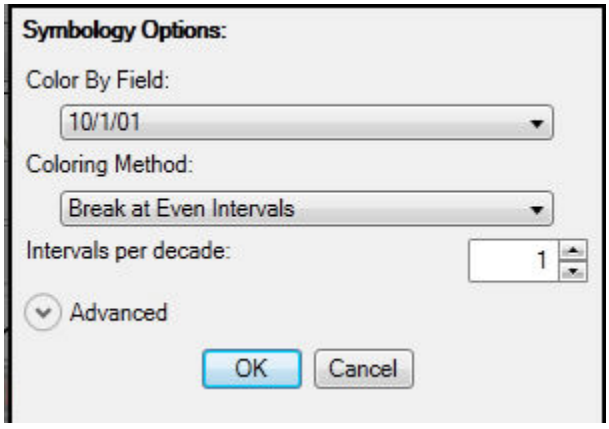
Click the Add EnterVol Data button and browse to find the file TCE_time_data.aidv



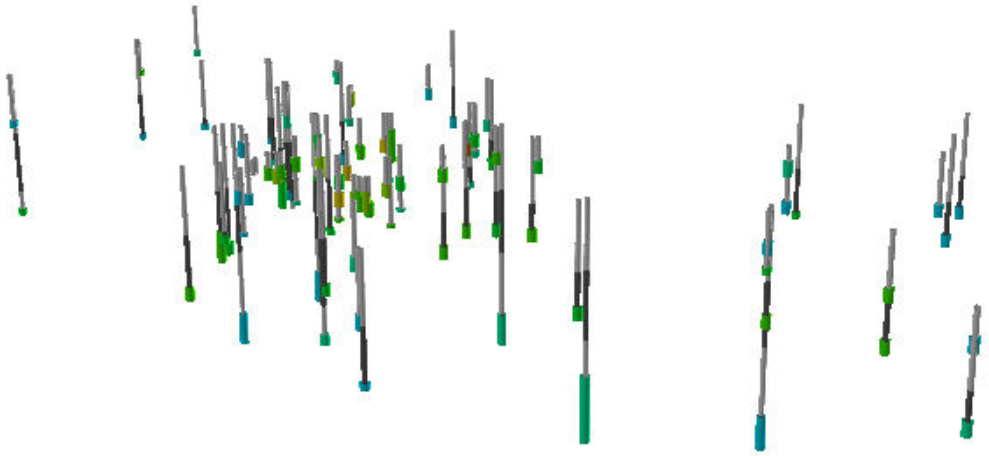

In the Color By Field pull-down, select the last time listed of 10/1/01



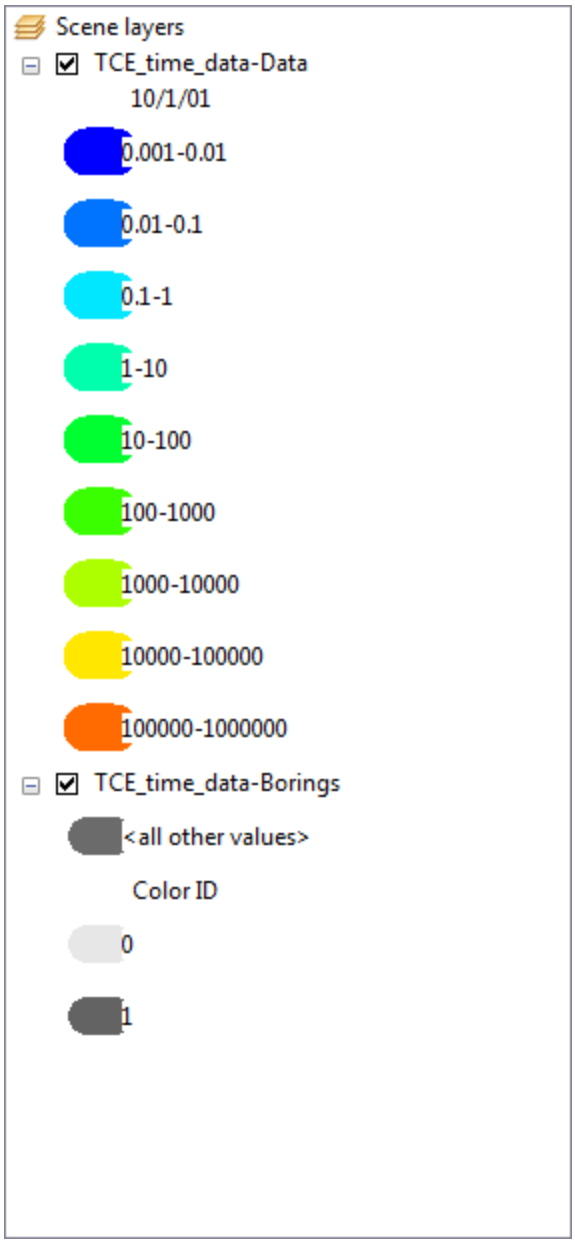

Since this data spans several orders of magnitude, let's choose the Break and Even Intevals coloring method.



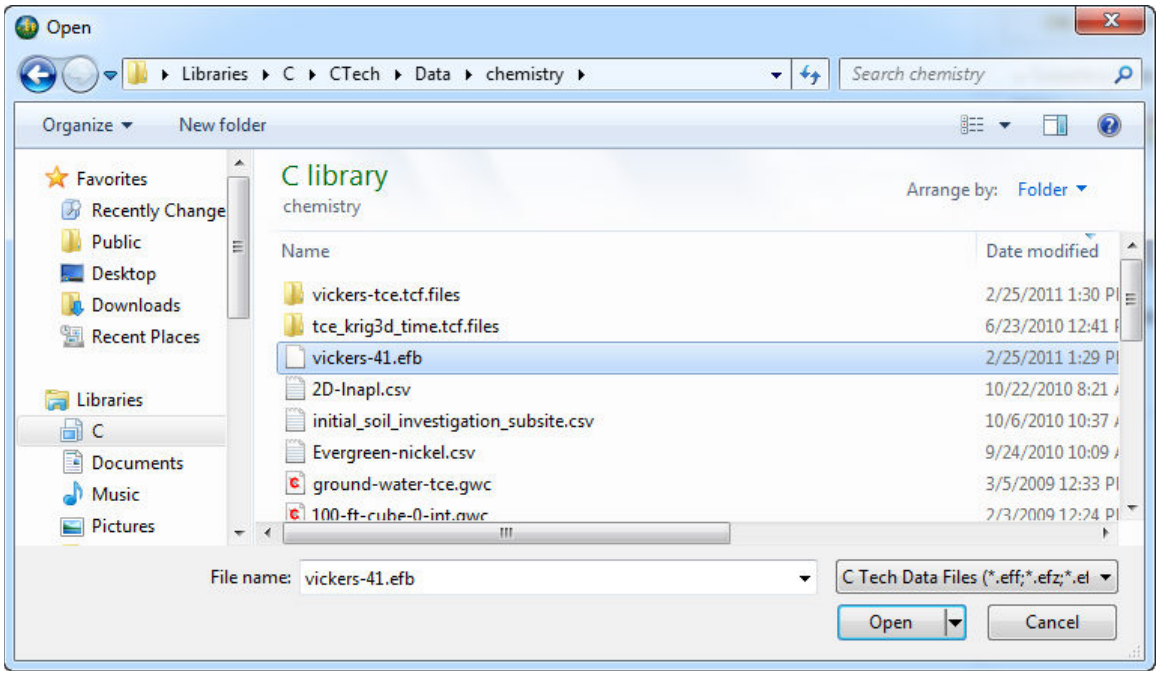

Upon clicking OK you'll see this in your scene.



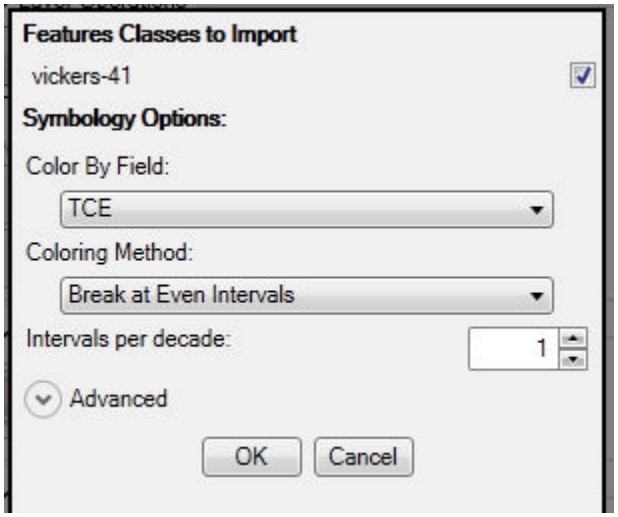
Note that there are two layers for the colored screen intervals and the borings.



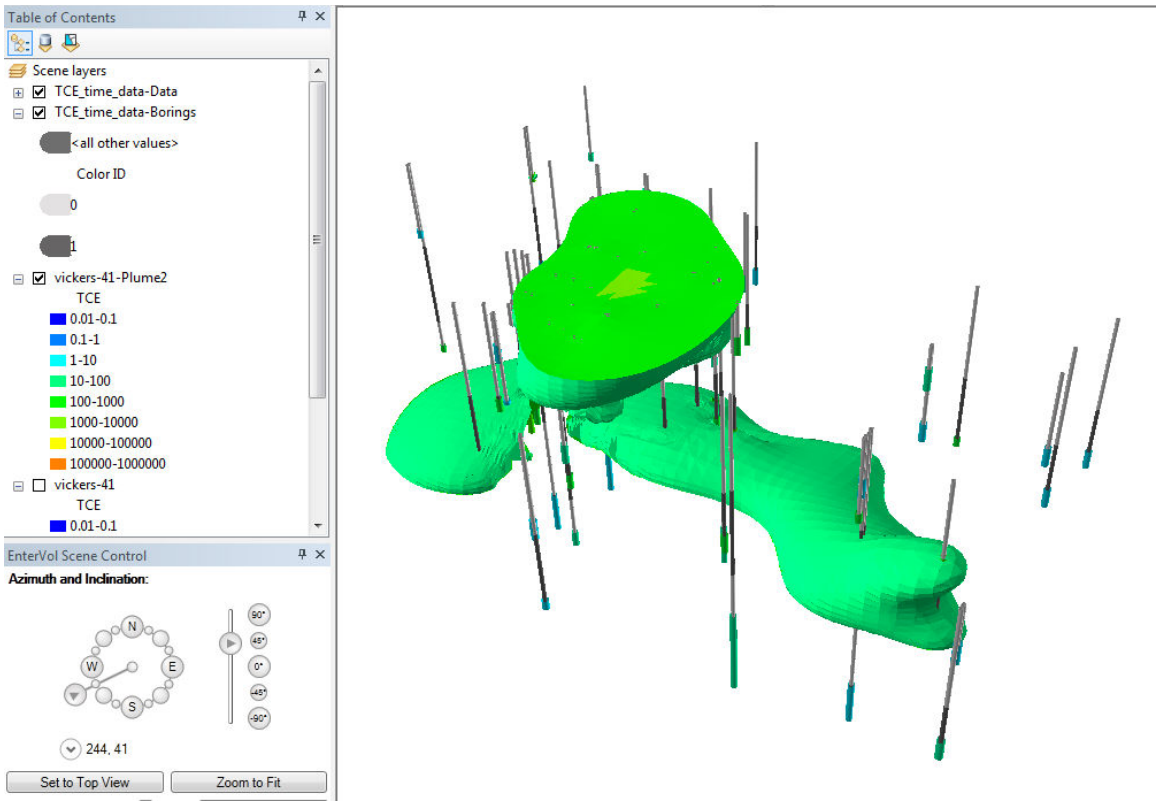

If we add the corresponding EFF




Select appropriate settings:




And now make a plume at 100 mg/l

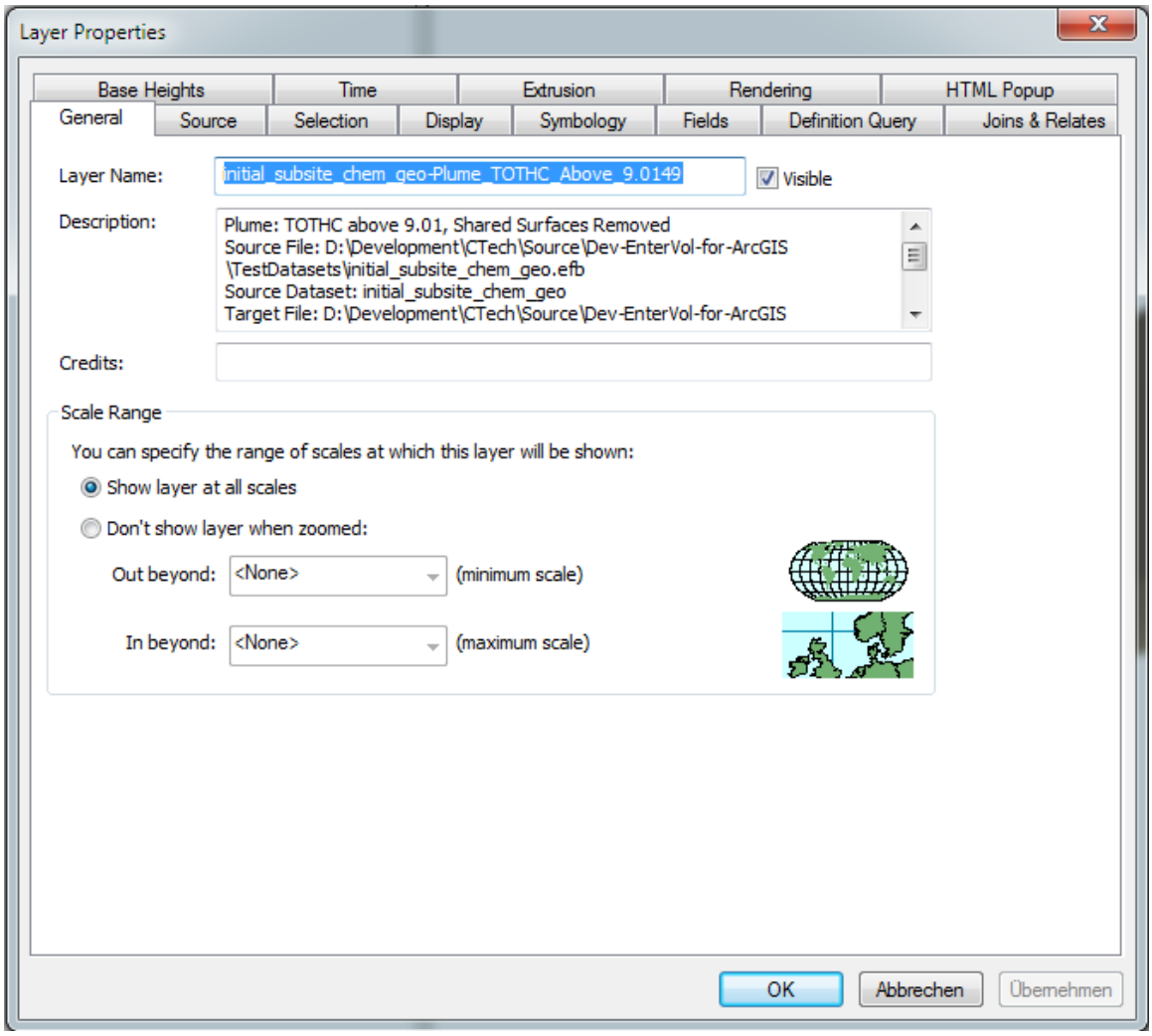



## Layer Description Summary

When EnterVol (or EVS) data is added, or when subsetting operations are performed, EnterVol adds a summary description of the settings which resulted in the creation of the object. This provides a substantial level of automatic documentation of your work.

The description only gets filled in when you use the EnterVol "Add Data" button or run a subsetting tool (and add the results directly).

Using ArcScene/ArcView's "Add Data" or dragging from ArcCatalog will not show the descriptions, since it's part of the "layer" in the .sxd/.mxd and not part of the feature class.




For the example plume above the full text of the description which is added is:

Plume: TOTHC above 9.01, Shared Surfaces Removed

Source File: D:\Development\CTech\Source\Dev-EnterVol-for-ArcGIS\TestDatasets\initial_subsite_chem_geo.efb

Source Dataset: initial_subsite_chem_geo

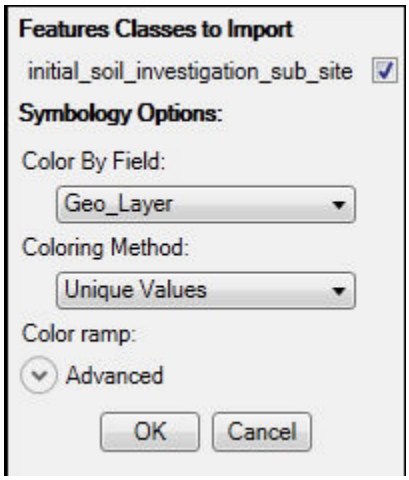
Target File: D:\Development\CTech\Source\Dev-EnterVol-for-ArcGIS\TestDatasets\initial_subsite_chem_geo-Plume8.evm

Created: Fri, 24 May 2013 22:24:28 GMT

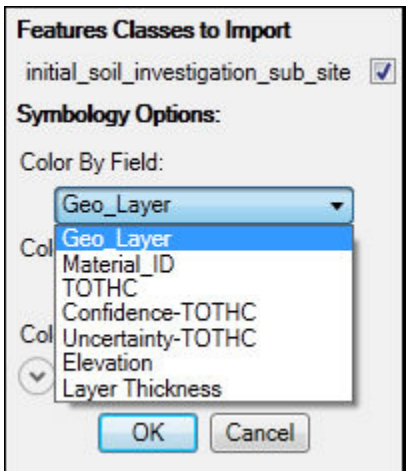
EnterVol for ArcGIS Version 1.8.1 - © 2013 ctech.com

## Set Symbology

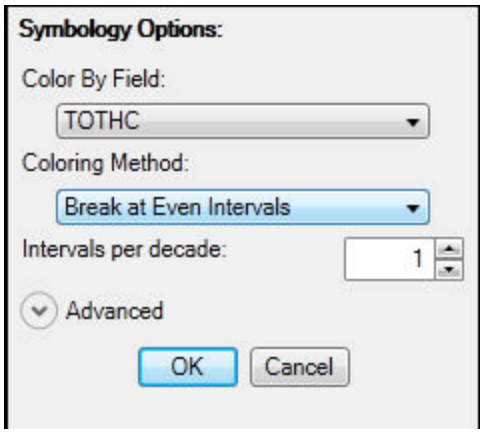
Set Symbology provides a simple menu to assign symbology and coloring to your objects using techniques not available in ArcGIS. However, you can set the symbology of any of the objects created in EnterVol using standard ArcGIS techniques also.




Let's change the data used for coloring

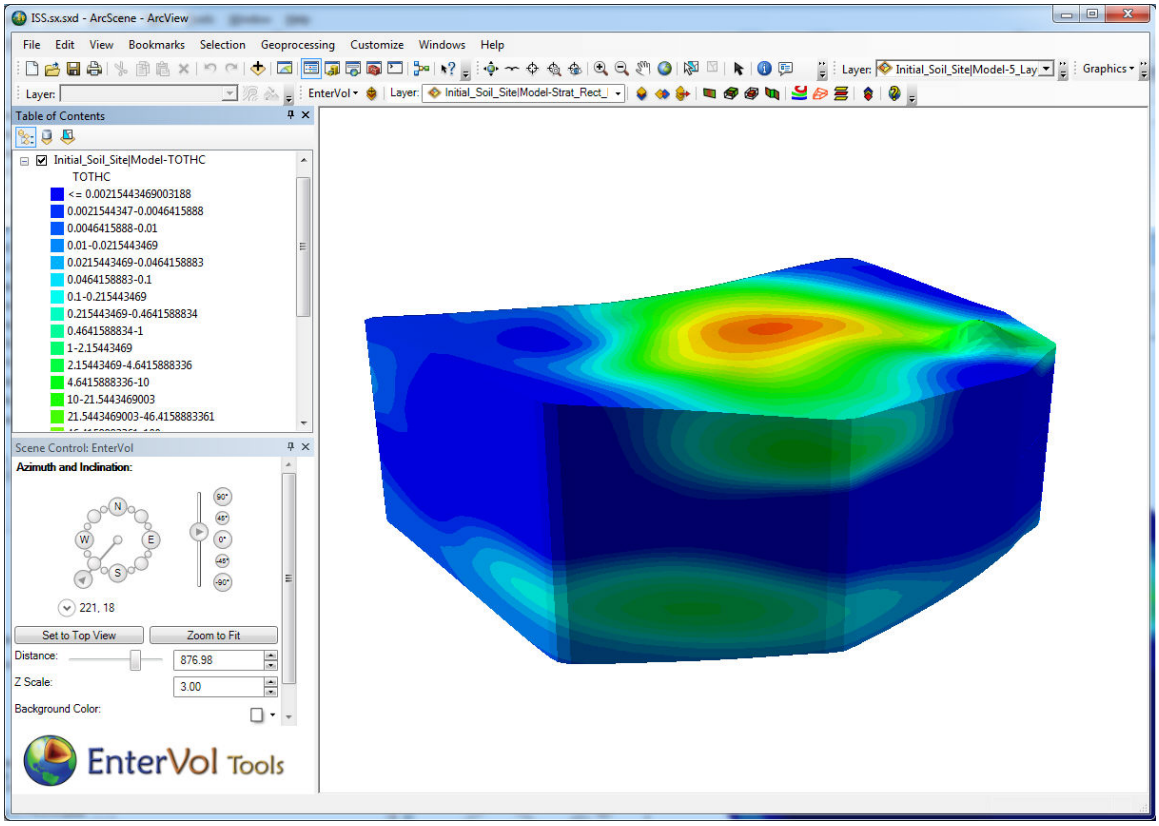



Choose TOTHC and *Break at Even Intervals* with 3 *Intervals per decade*




and press OK

This changes how our initial feature class is colored:

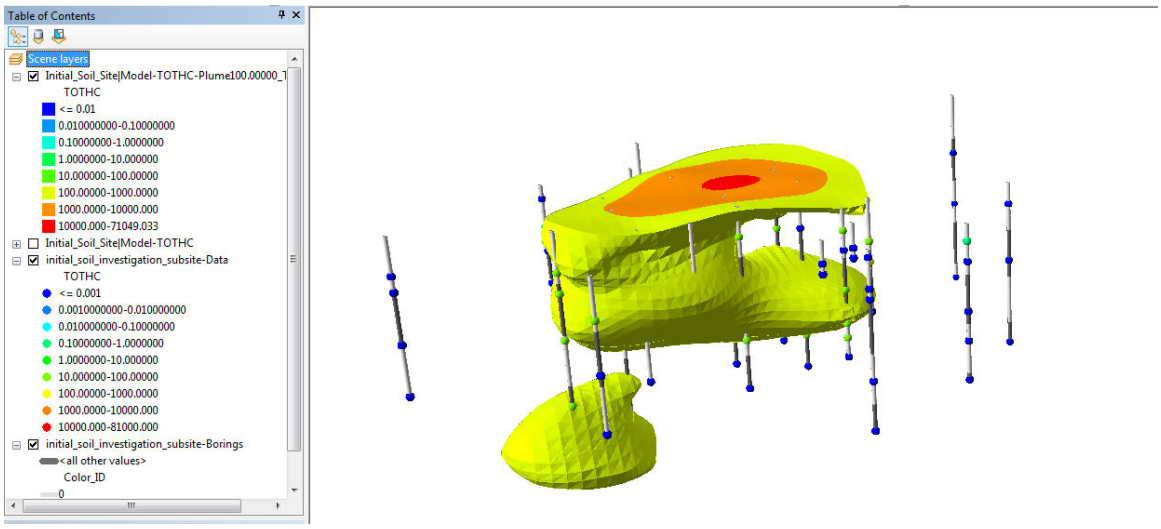



## Copy Symbology

When layers are not created from other layers where you would normally have the symbology copied from the input during creation, it is common that the symbology (coloring and breaks) will not be the same. When different

layers in the same scene represent the same data, but do not share common symbology, your map can be confusing and even deceptive.

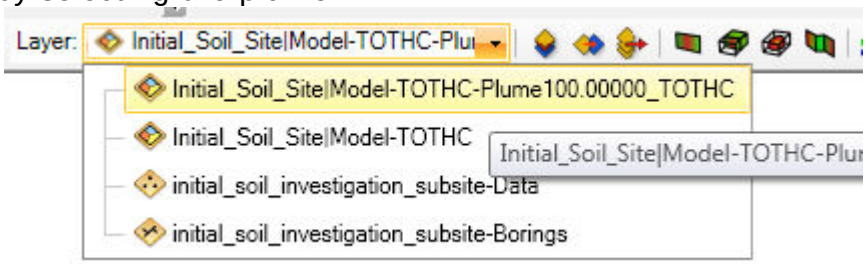
The Copy Symbology tools makes it easy to apply the symbology from one layer to another.




For this example we have a 100 ppm plume and the TOTHC points in the scene, however the minimum and maximum values are not the same.

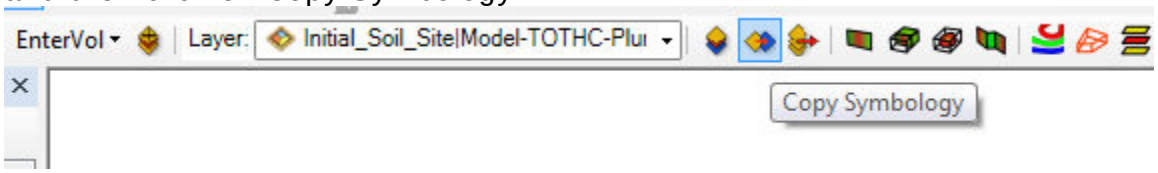
**The color blue (and all in between) have different connotations in each layer.**

Start by selecting the plume

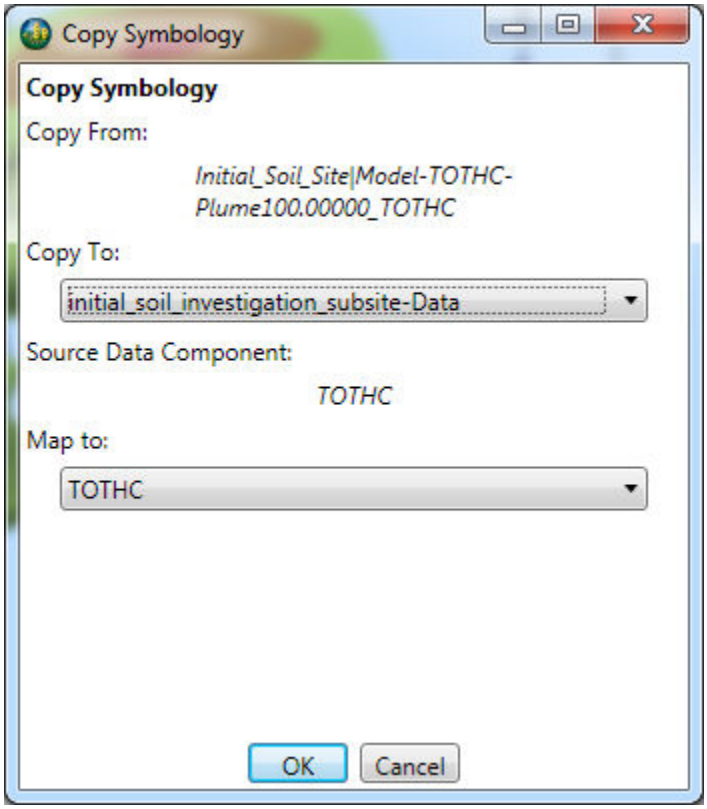



layer

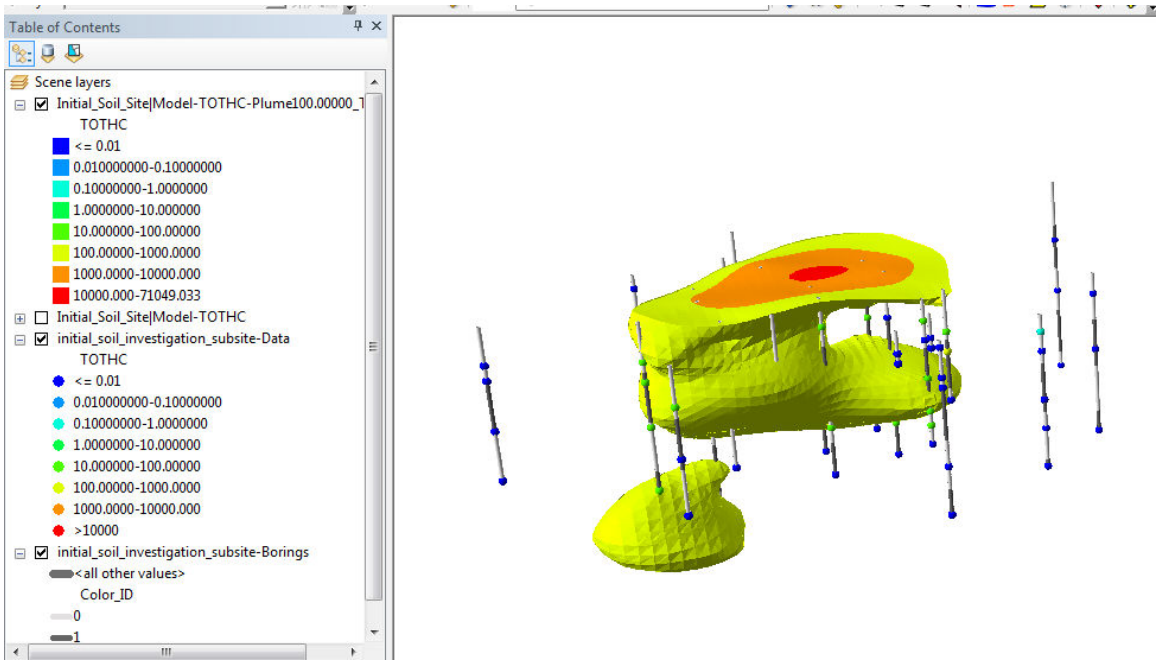
and then click on Copy Symbology



The Copy Symbology opens. For this example, we'll copy TOTHC from the plume to the data points.

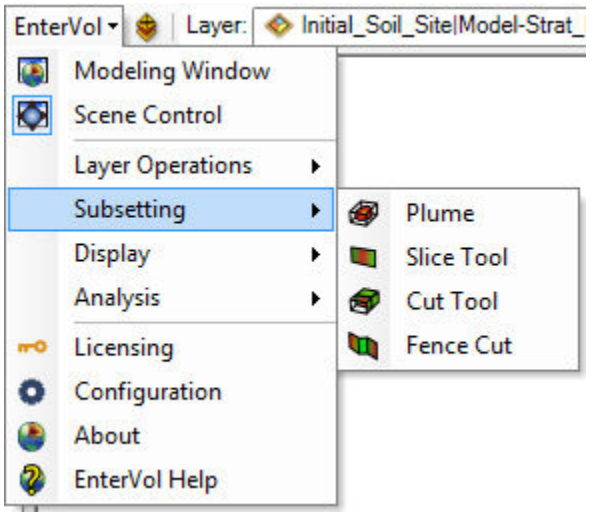



when you press ok, you should see



# Subsetting Tools

## Subsetting Tools




The Subsetting Tools consist of:

1. Slice
2. Cut
3. Create Plume
4. Create Fence Cut

If you start with the project state at the end of the Set Symbology topic, we're ready to apply some subsetting operations. If your Subsetting Tools buttons look faded as compared with the image above, make sure you select the object from EnterVol's Layer Selector to which you want to apply the subsetting operations.

When any subsetting operation is performed, a summary of the data and settings used to create it is added to the Layer Properties under General.Description.

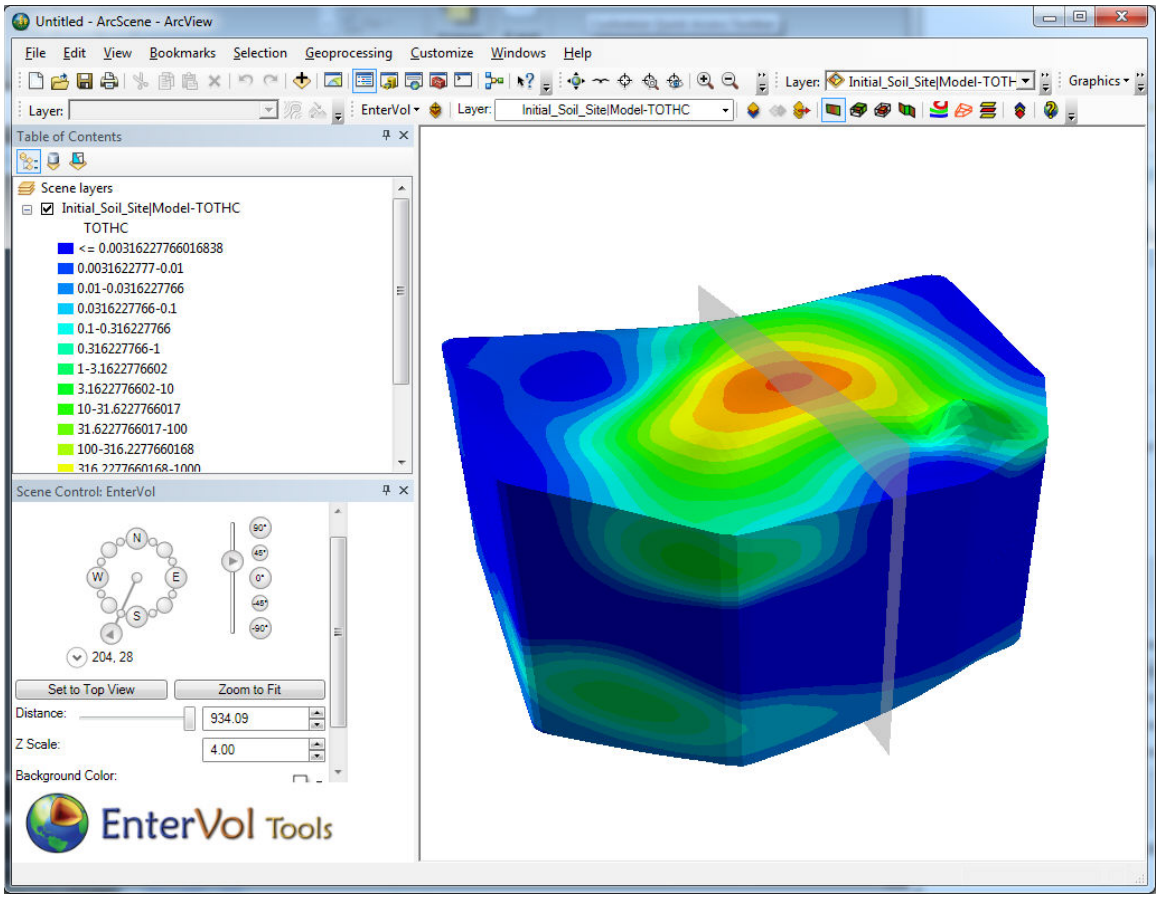
### Slice

Slicing through a volume of data is a common requirement. ESRI tools offer ways to create slices through (hollow) volumes, but lack the ability to map the data within the volume onto the slice. When EnterVol slices through a volume, the data is inherently present in the internal volumetric EFF. This data is accurately mapped to the slice based on either cell attributes or nodal data.

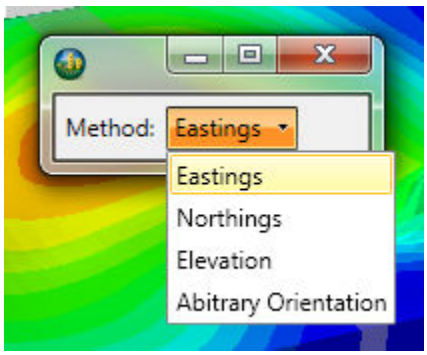
When a slice is performed, a summary of the data and settings used to create it is added to the Layer Properties under General.Description.

Let's begin with the scene layer created at the end of the Set Symbology topic and slice through the volume.

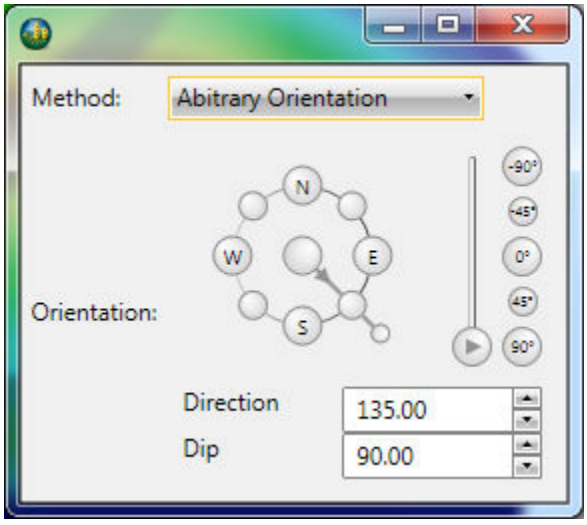

Select the scene layer you want to slice through and click on the Slice icon




If you right-click on the grey slice, there will be four choices for how you will specify the location and orientation of the slice plane.



Slice at an Easting (X) coordinate is the first (default), but for this exercise we will choose "Arbitrary Orientation" which is the most flexible.




The arbitrary slice menu has many parameters. The initial two are the slice Direction and Dip which defines the orientation. Note that with the default Dip of 0.0 (zero), the plane direction become irrelevant (changing it will have no effect), however for any other Dip, the Direction rotates the plane through 360 degrees.

With these two angles the slice can be fully defined by providing any X-Y-Z point through which the slice passes. You specify that point by clicking with the left mouse button.

Once you do that an expanded menu allows you to refine the precise location for the slice.

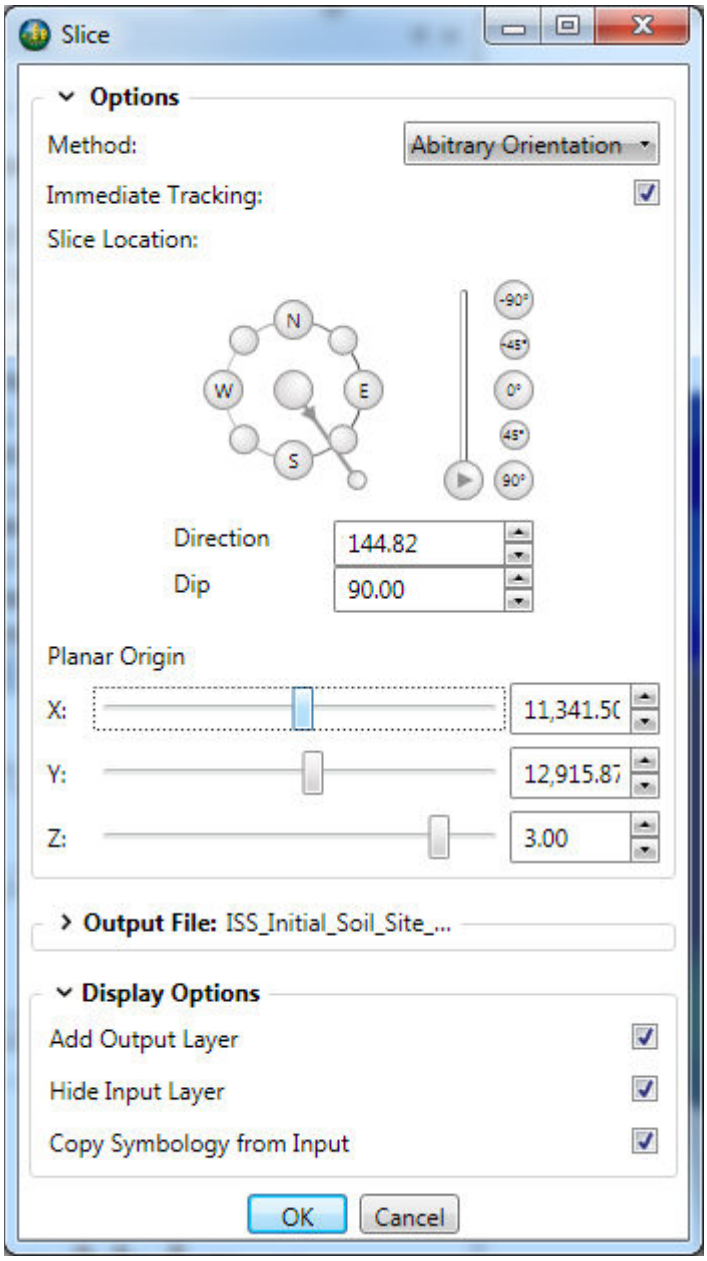
You'll notice that as you adjust the Dip, Direction and Origin, a transparent gray slice will be shown in the view to display precisely where the slice will be created.

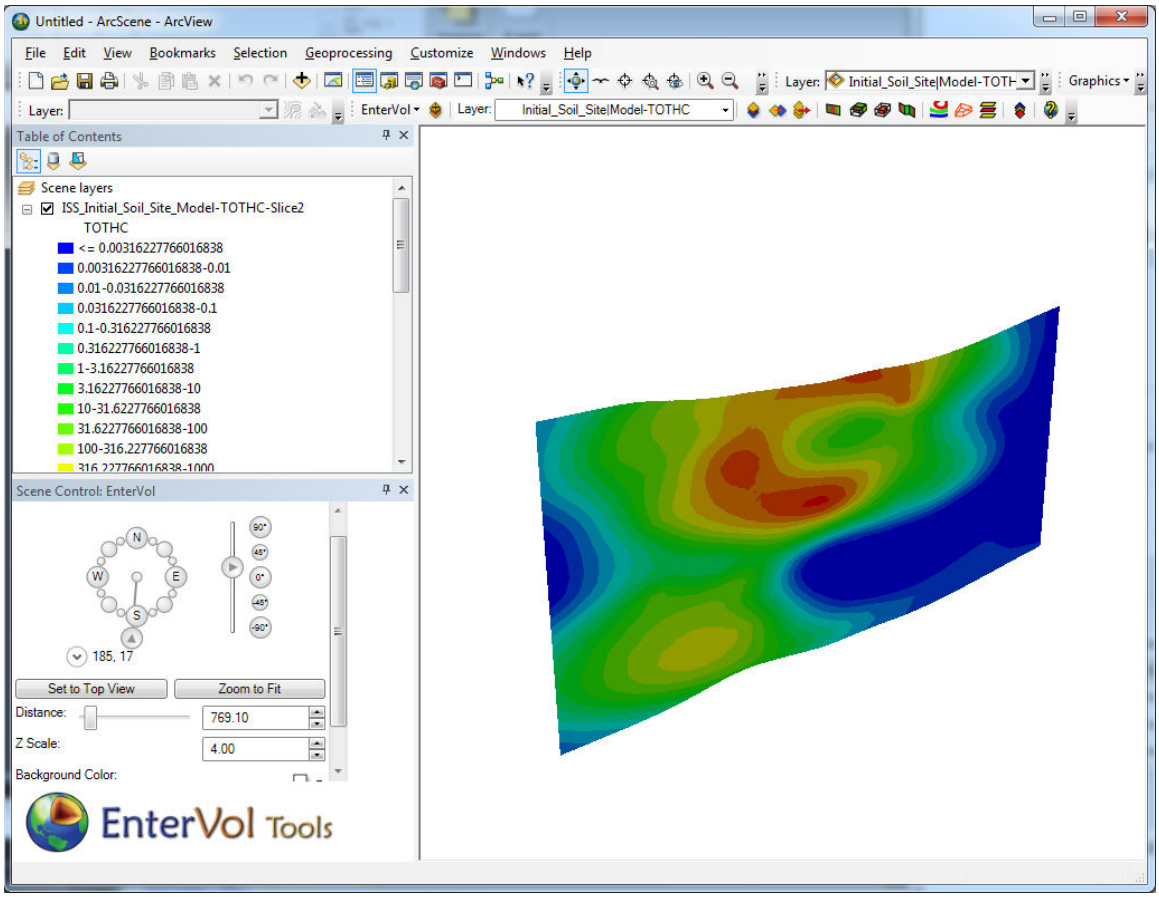
Choose the options you want such as:

- Add Output Layer:
  - If this is not checked, the EFF will be created but not added to your scene or Table of Contents.
- Hide Input Layer
  - if not checked, your slice will exist but will be inside the input layer and not visible unless the input layer is turned off or made transparent.

- Copy Symbology from Input
  - use the same coloring as the object being sliced

Below we have made some small changes and then click OK.






There are many other things we'll likely want to do. Four interesting ones would be:

- Add External Edges
- Add a Cut
- Add Isolines

**Cut**

Cutting through a volume of data is a common requirement. A cut is distinctly different than a slice, though the cutting plane is defined in an identical manner to slice planes. When you cut a volume (or any object), everything on one side of the cutting plane is "cut" away. Cutting through a volume results in a volumetric output, whereas slicing through a volume creates a slice surface.

When a cut is performed, a summary of the data and settings used to create it is added to the Layer Properties under General.Description.

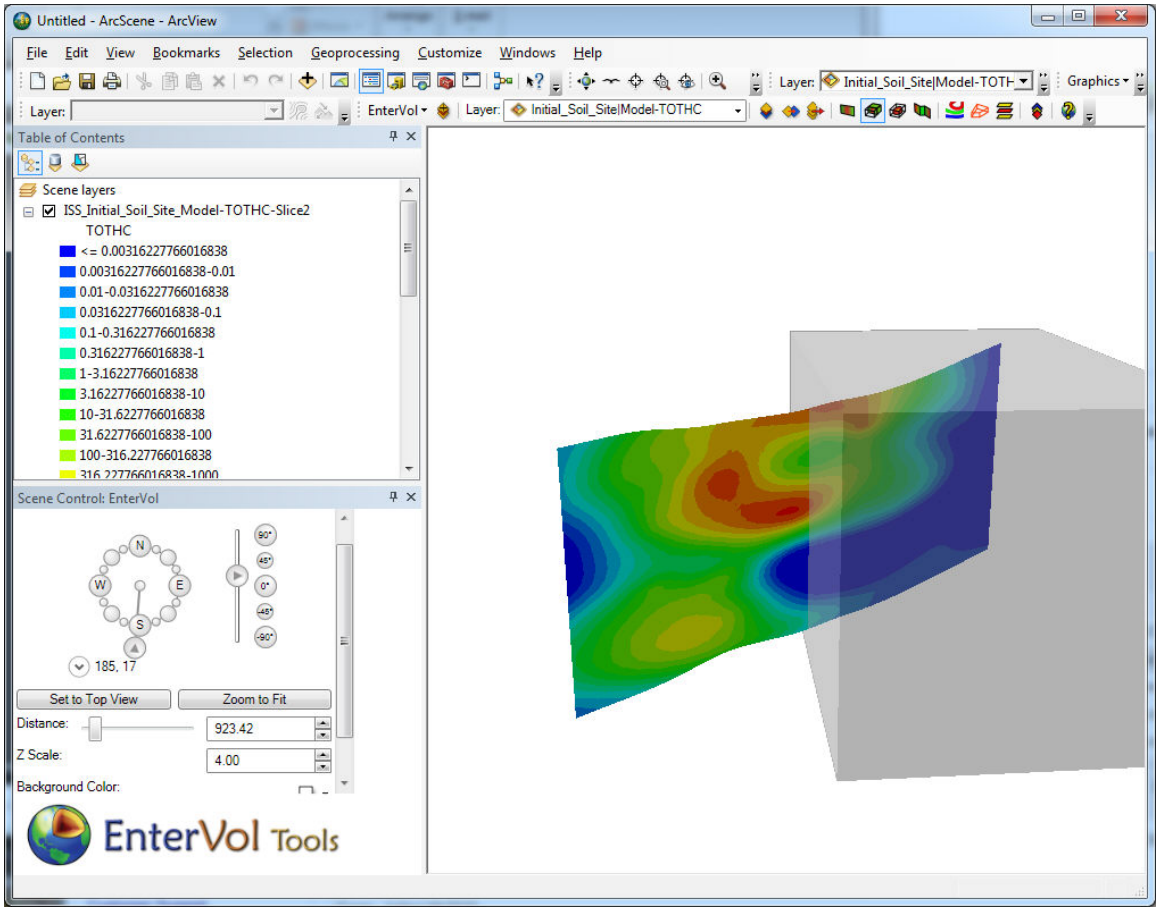
ESRI tools offer ways to cut through (hollow) volumes, but lack the ability to map the data within the volume onto the remaining portion. When EnterVol

cuts through a volume, the remaining portion of the volume inherits all data present in the parent volumetric object.

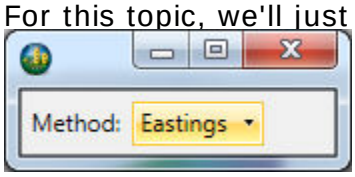
Let's begin with the scene layer created at the end of the slice topic and add a cut through the volume.

Select the scene layer representing the original EFF and click on the Cut icon.

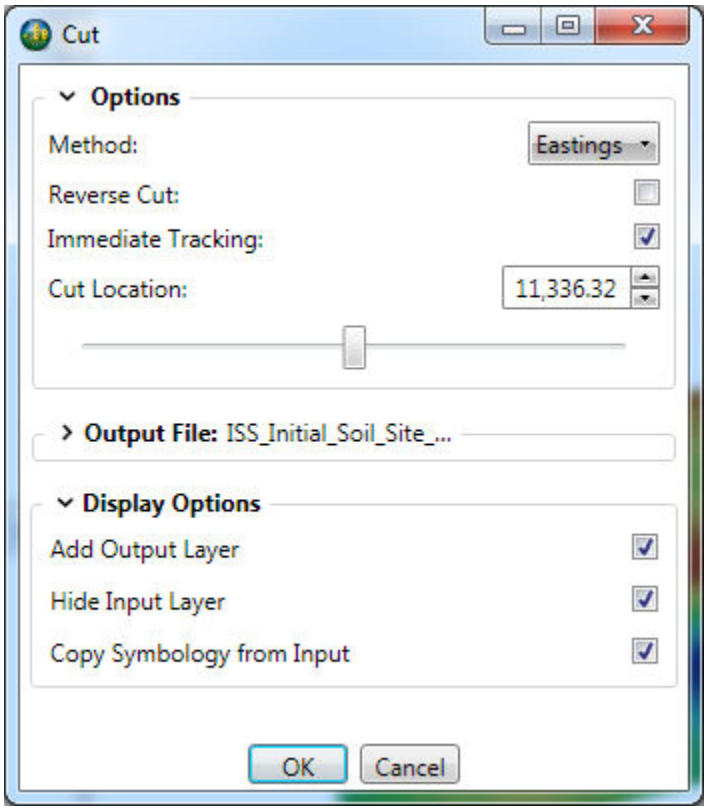

the scene will display the cutting volume:

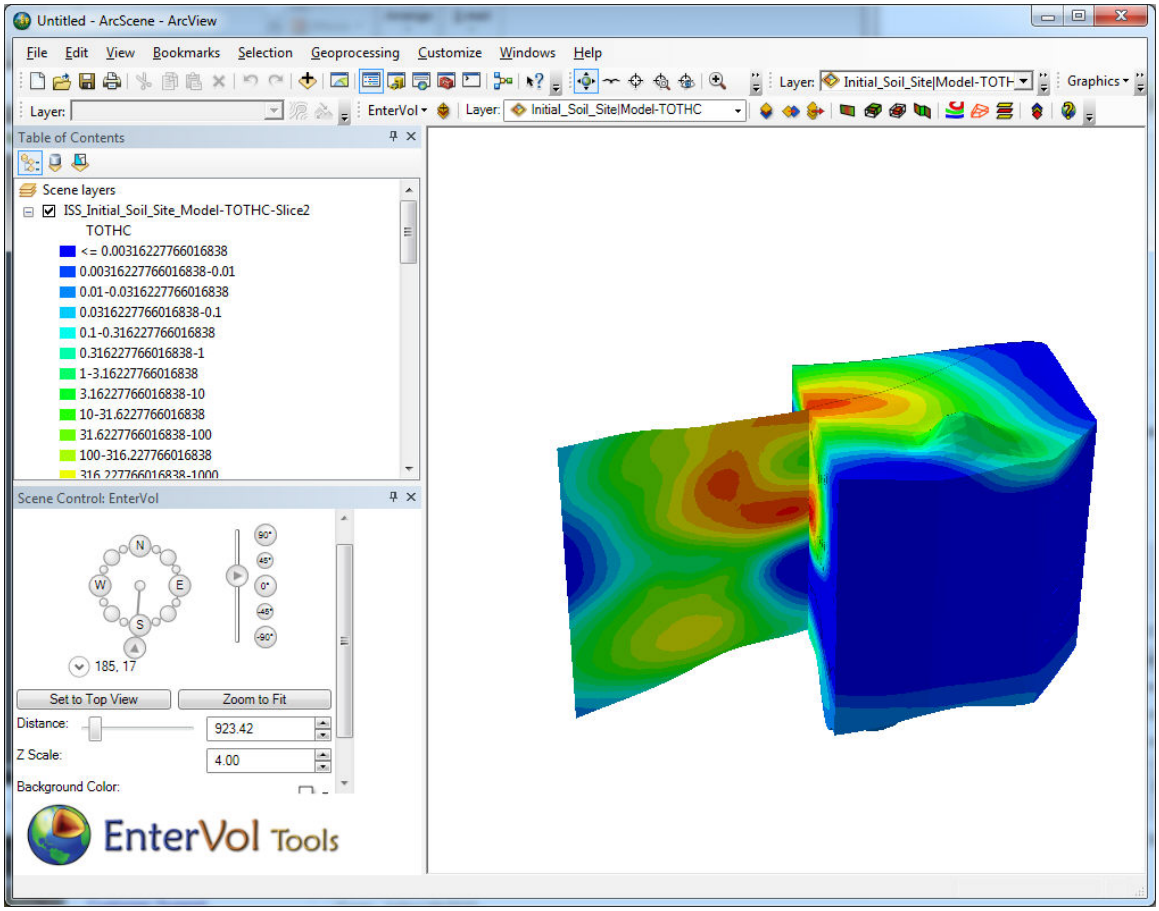



Cut has all the same options for controlling the cutting plane as Slice. You can right click to choose the cutting method and a left click will initially position the cut.

For this topic, we'll just use the default position of an Eastings 

cut.





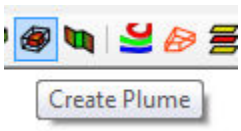

**Plume (volumetric subset) of Analytical Data**

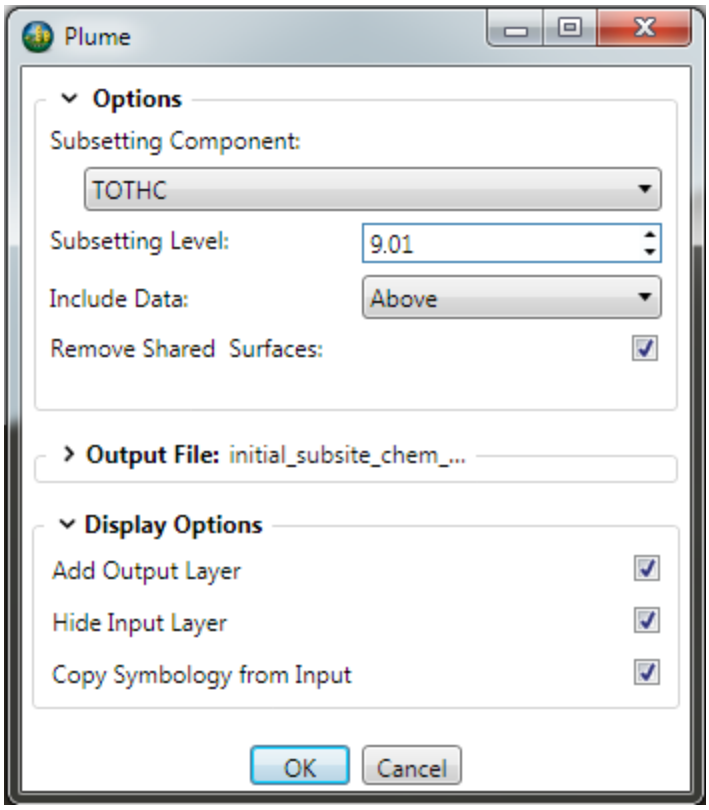
The display of Plume icon vs. Ore Body is dependent on your Configuration settings.

When a plume is created, a summary of the data and settings used to create it is added to the Layer Properties under General.Description.

Let's begin with the scene layer created at the end of the Set Symbology topic.

Since the object we're using has several attributes including Total Hydrocarbon Concentrations (TOTHC) which have a continuum of values ranging from 0.001 to ~100,000 (8 orders of magnitude), let's create a plume at 0.2 mg/kg.



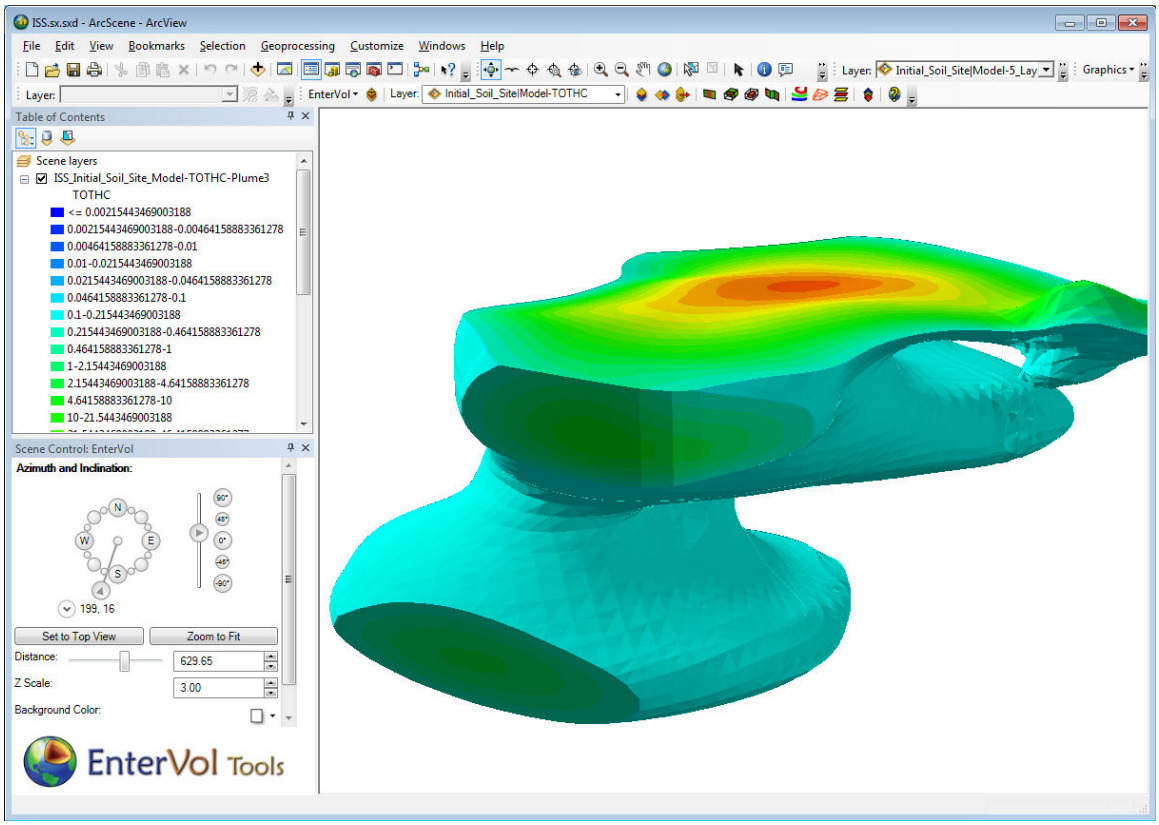



Choose the options you want such as:

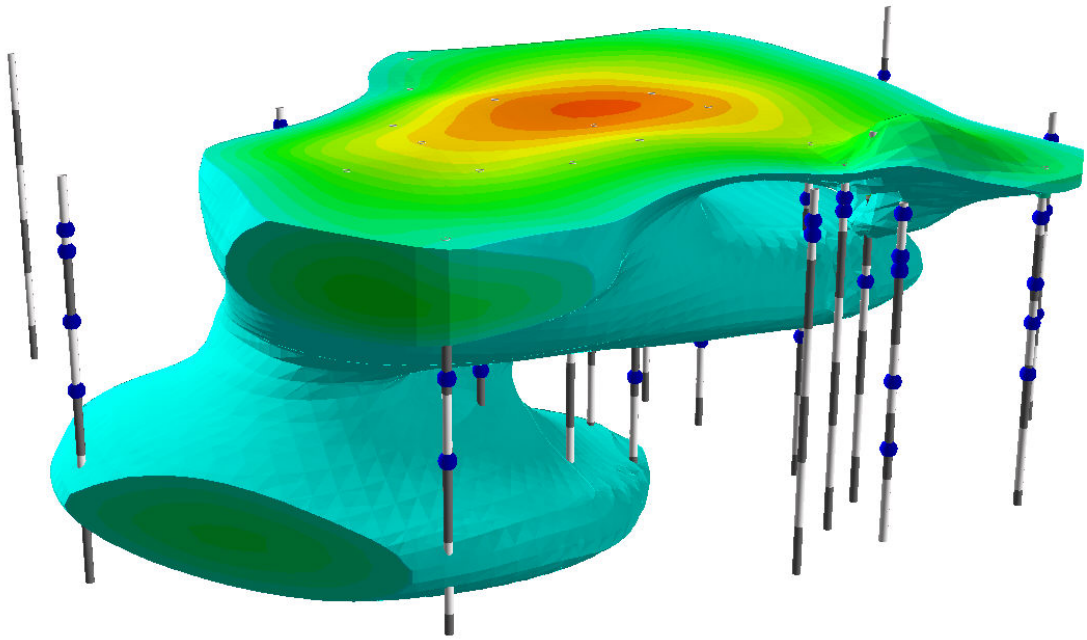
- **Subsetting Component:** the data used for subsetting
- **Include Data:** Options are *Above* or *Below*
- **Remove Shared Surfaces:** when creating plumes in models with stratigraphic layers or lithology, this option removes the internal surfaces.
- **Add Output Layer:** if this is not checked, the EFF representing the plume will be created but not added to your scene or Table of Contents
- **Hide Input Layer:** if not checked, your plume will exist but will be inside the input layer and not visible unless the input layer is turned off or made transparent.
- **Copy Symbology from Input:** use the same coloring as the input layer

By default new layers will be written to the same folder as their input layers and their name will identify the type of subsetting or display operation being performed.

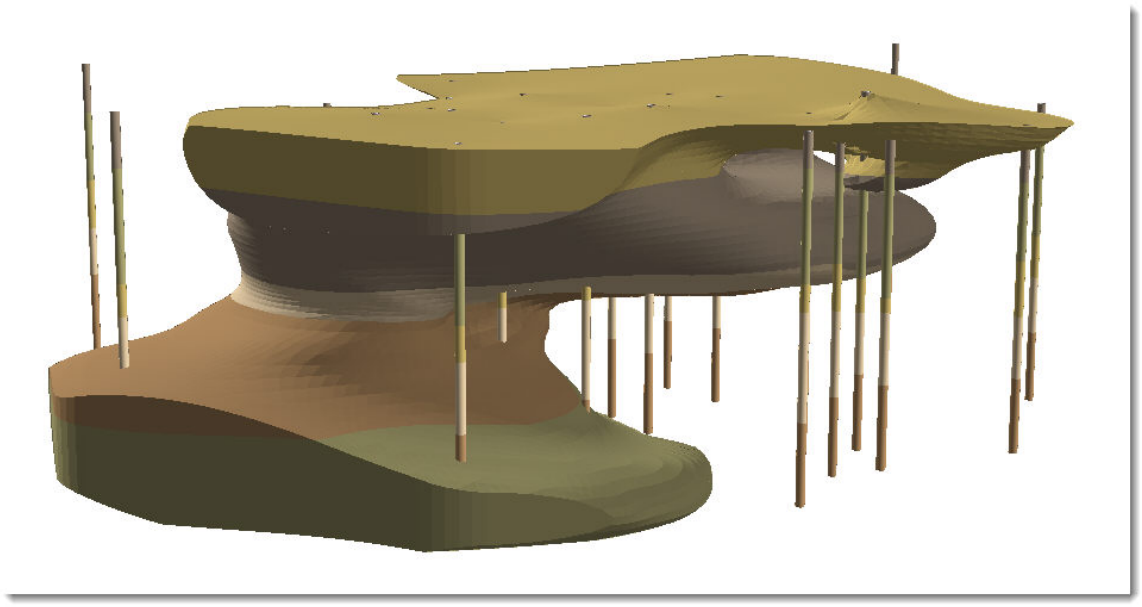
Upon pressing OK, and waiting several seconds, our initial scene layer is unselected and our plume should appear.




Adding in the analyte data and borings we have



The same plume level colored by geologic material with borings will give us: (Z Scale is 3.0)



Other examples of using the Plume tool are at the bottom of this topic.

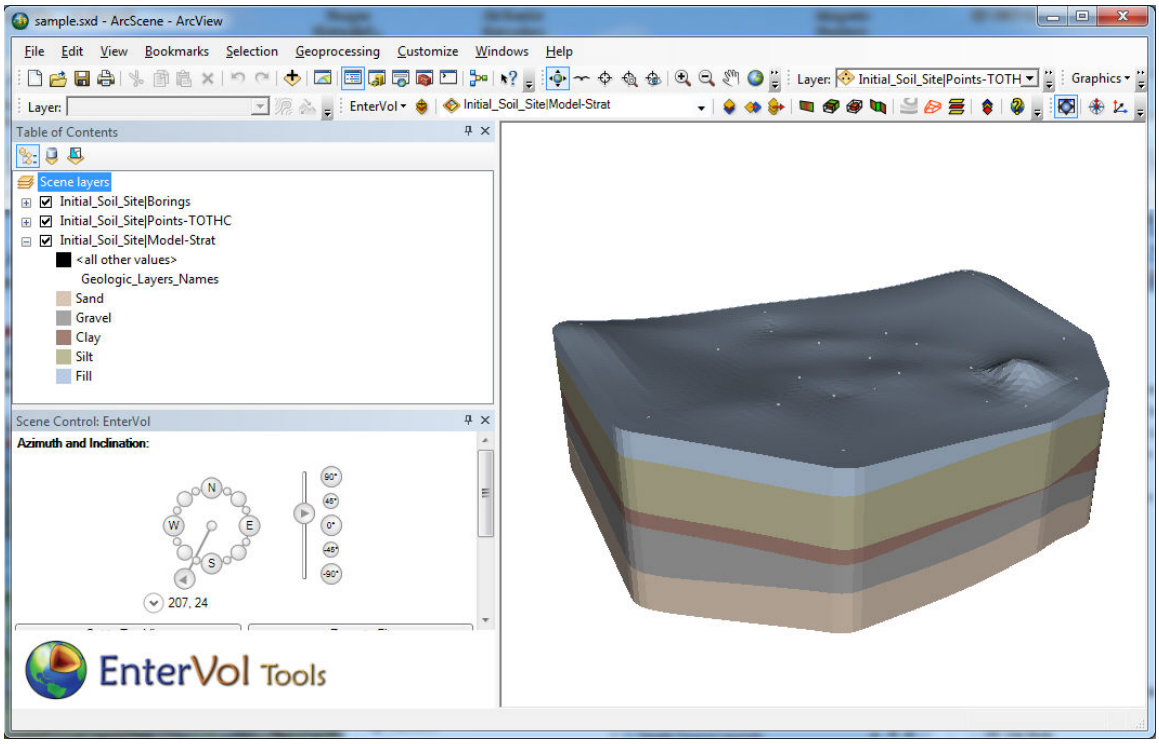
**Plume for Unique Values**

The Plume tool can also be used to subset models containing unique values. This includes attributes such as:

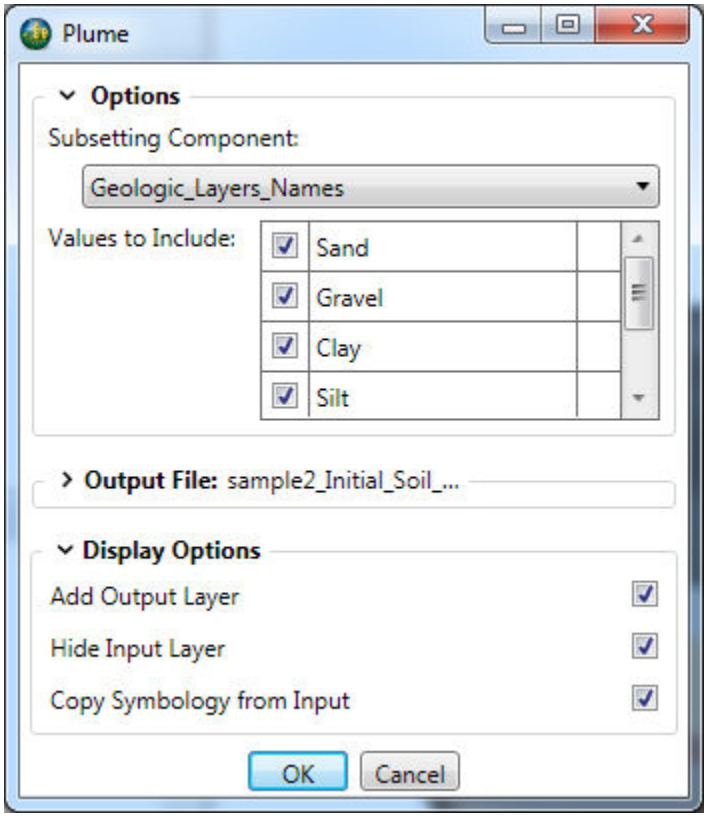
- Material IDs
- Geologic Layers Order
- Geologic Layers Names
- Lithology Names

When a plume is created, a summary of the data and settings used to create it is added to the Layer Properties under General.Description.

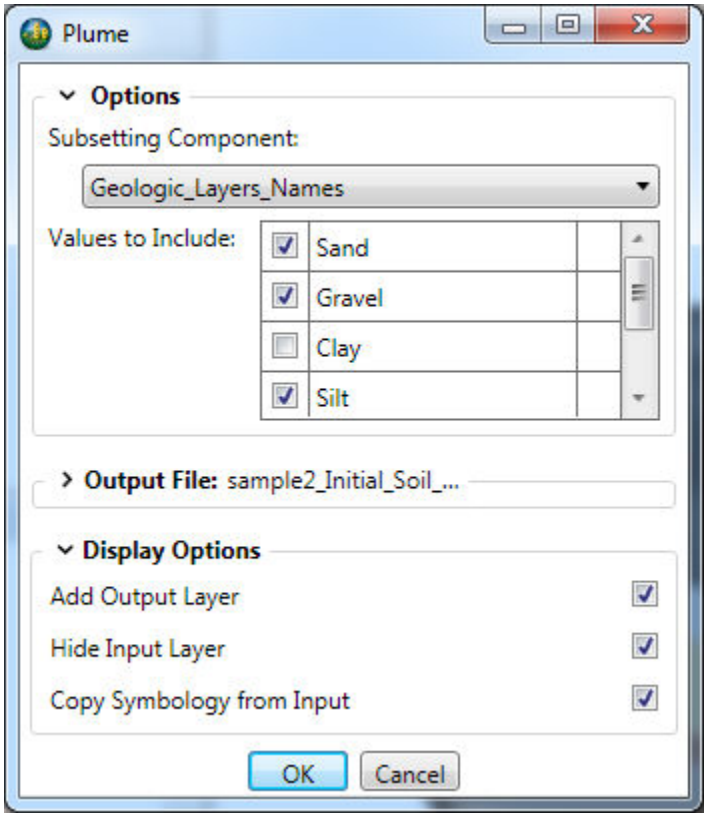
Beginning with the project below, which has stratigraphic layers with analytical data:

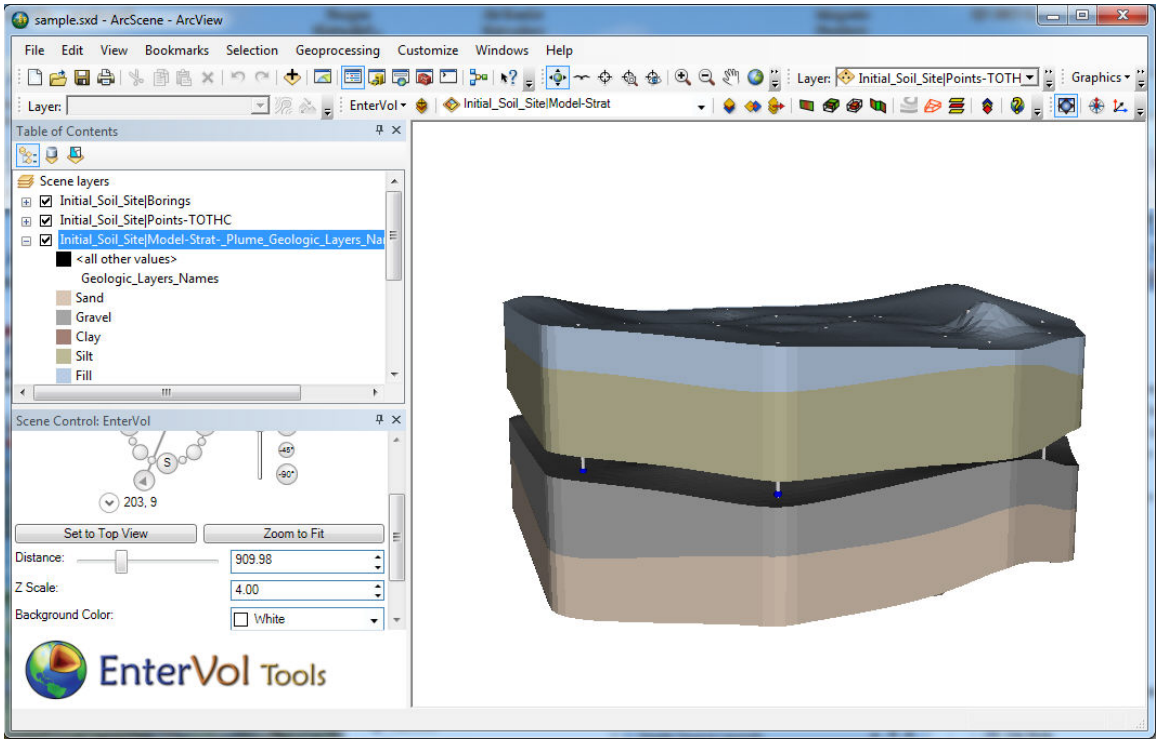



We can select the Plume tool and the default will show the *Geologic Layer Names*:

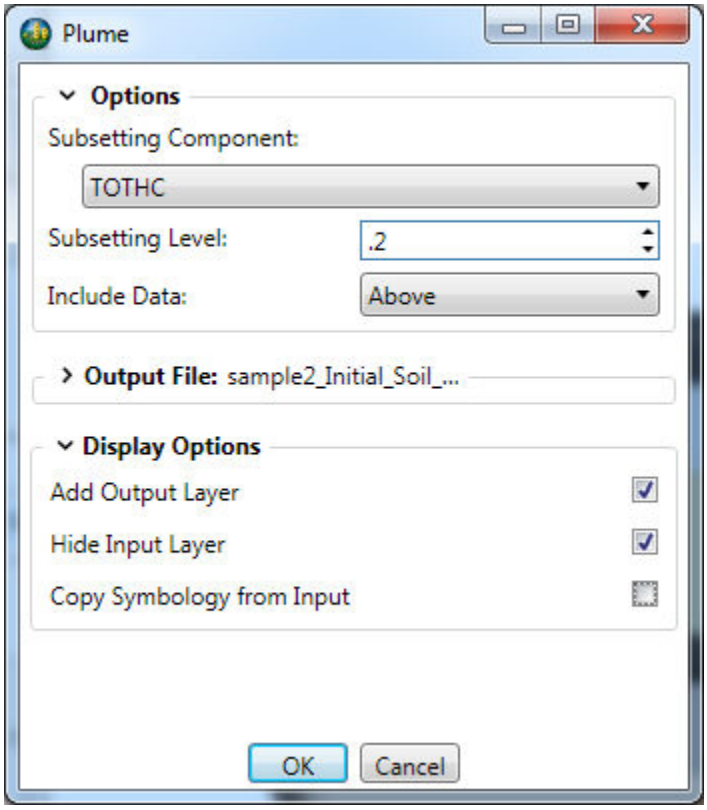


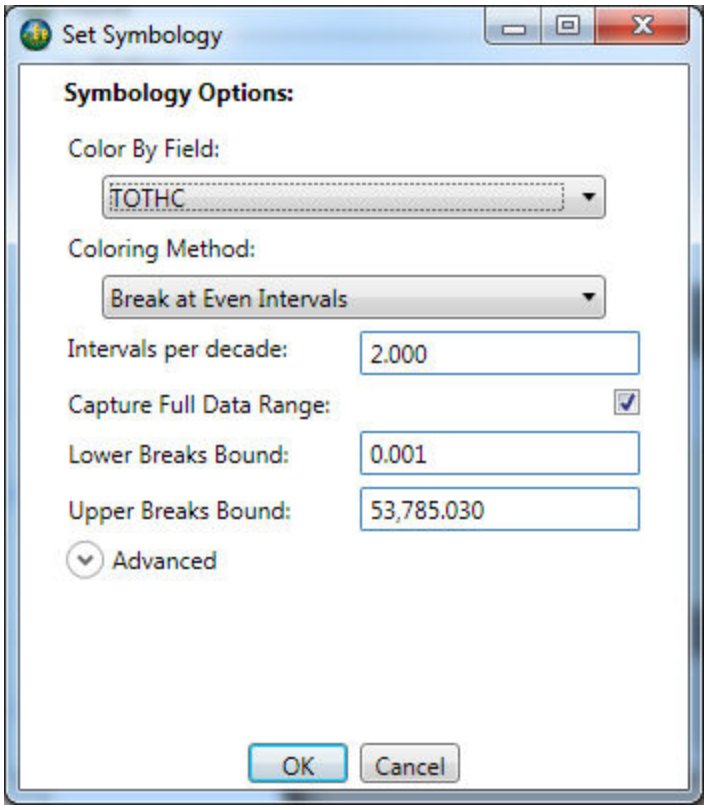
*If we turn off Clay, which is the middle (third) layer, the resulting model will show:*

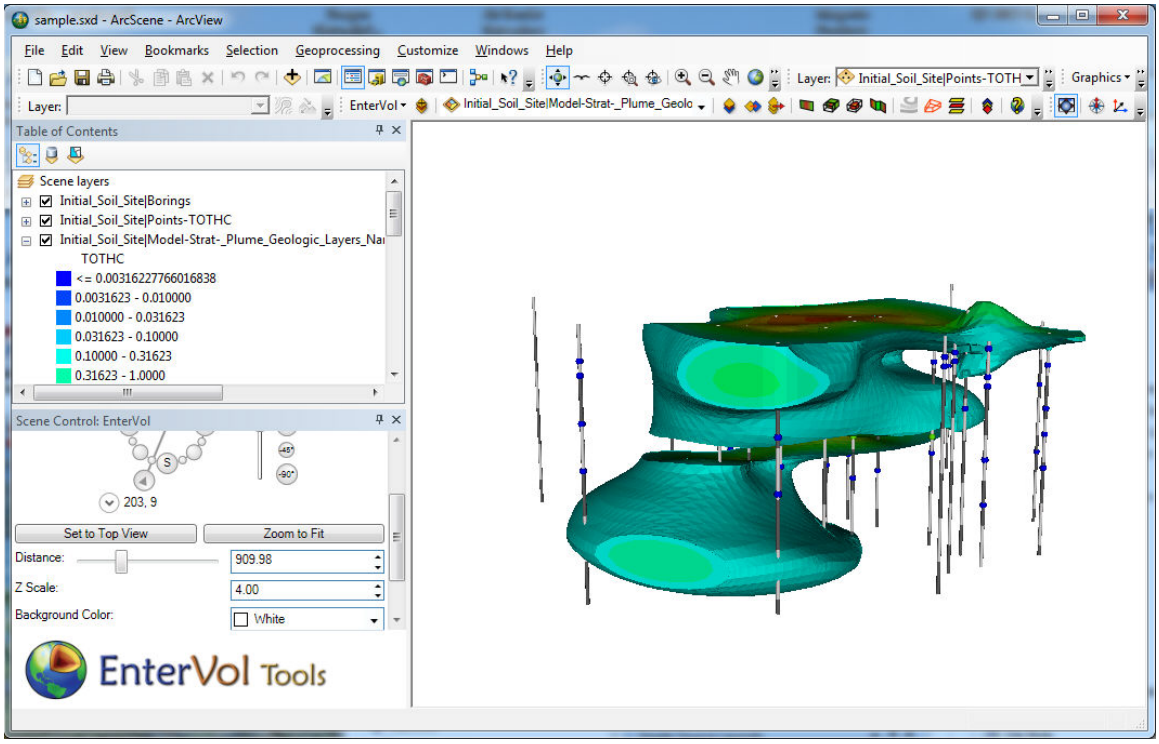


We can then subset this layer by TOTHC to show a plume with the Clay layer absent:




Set Symbology
Symbology Options:
Color By Field:
TOTHC
Coloring Method:
Break at Even Intervals
Intervals per decade:
2.000
Capture Full Data Range:
Lower Breaks Bound:
0.001
Upper Breaks Bound:
53,785.030
Advanced
OK
Cancel

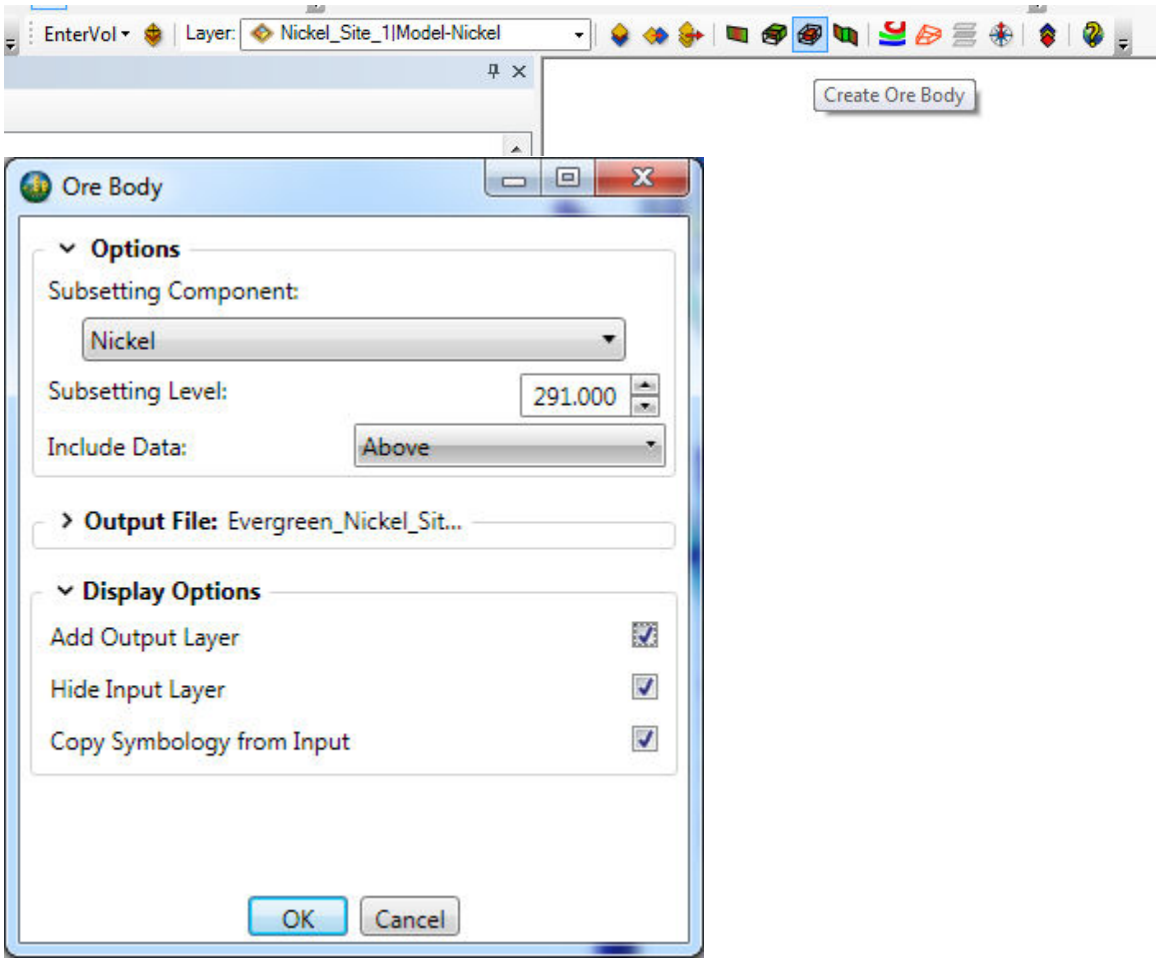

sample.sxd - ArcScene - ArcView
File Edit View Bookmarks Selection Geoprocessing Customize Windows Help
Table of Contents
Scene layers
Initial_Soil_Site|Borings
Initial_Soil_Site|Points-TOTHC
Initial_Soil_Site|Model-Strat-_Plume_Geologic_Layers_Na
TOTHC
<= 0.00316227766016838
0.0031623 - 0.010000
0.010000 - 0.031623
0.031623 - 0.10000
0.10000 - 0.31623
0.31623 - 1.0000
Scene Control: EnterVol
Set to Top View
Zoom to Fit
Distance:
909.98
Z Scale:
4.00
Background Color:
White
EnterVol Tools

## Ore Body

The Ore Body tool is nearly identical to the plume tool, except that its option are tailored to mining.

When an ore body is created, a summary of the data and settings used to create it is added to the Layer Properties under General.Description.

The display of Ore Body (vs. Plume) is dependent on your Configuration settings.




Choose the options you want such as:

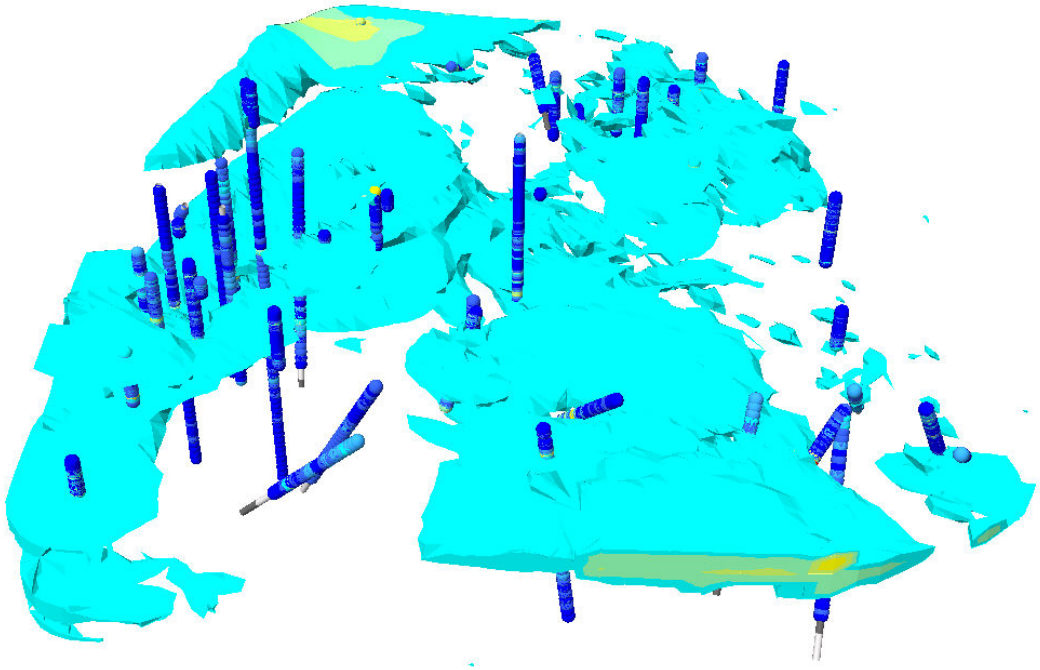
- Add Output Layer:
  - If this is not checked, the feature class representing the plume will be created but not added to your scene or Table of Contents.
- Hide Input Layer

    - if not checked, your plume will exist but will be inside the input layer and not visible unless the input layer is turned off or made transparent.

- Copy Symbology from Input
    - use the same coloring as the input layer

By default new layers will be written to the same folder as their input layers and their name will identify the type of subsetting or display operation being performed.

Upon pressing OK, and waiting several seconds, our initial scene layer is unselected and our plume should appear.



**Create Fence Cut**

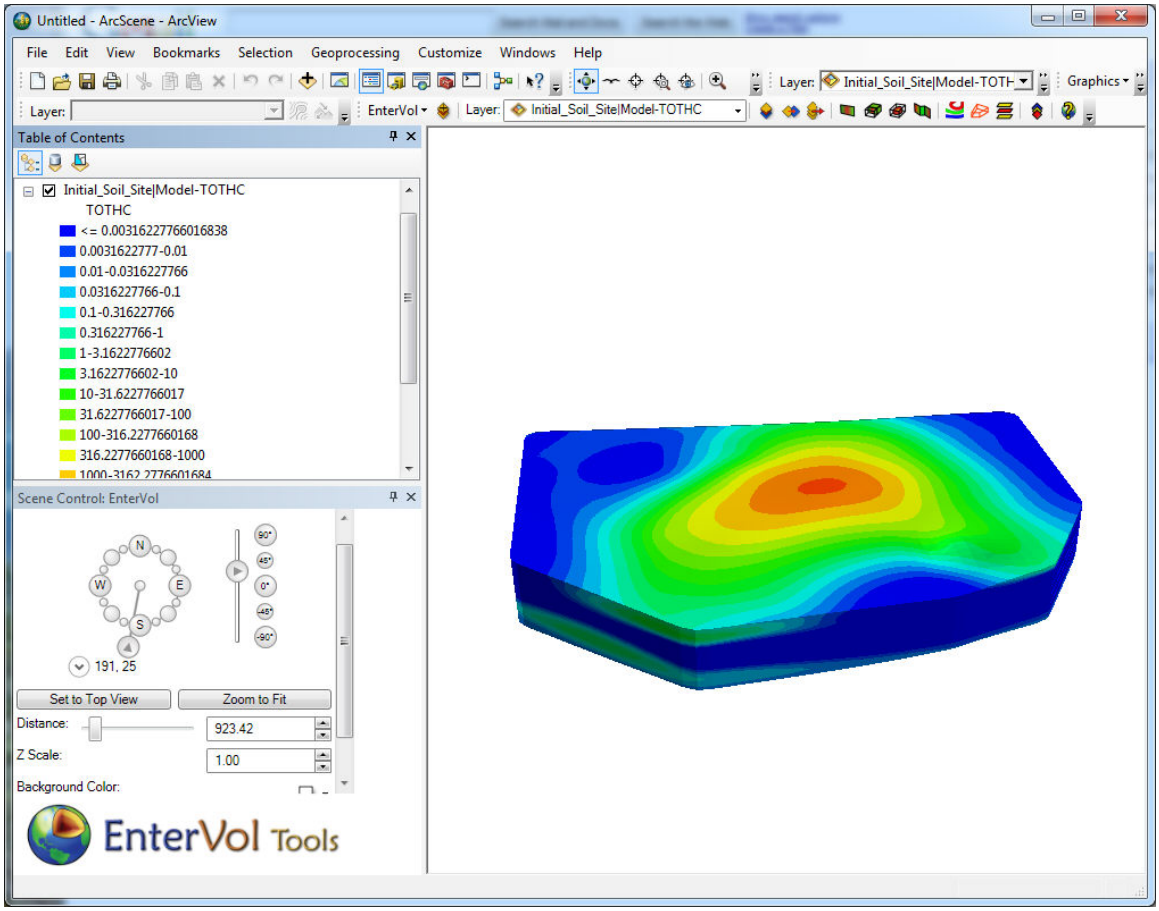
Generation of true fence diagrams cut from your 3D layers is easy with EnterVol. The process is simple:

1. Read, specify or draw the path for the fence
2. Select the layer to be used as input
3. Create the fence

When a fence operation is performed, a summary of the data and settings used to create it is added to the Layer Properties under General.Description.

For the example below we will draw our fence paths in ArcScene and use ArcMap to convert the graphics object to a shapefile.

Begin with the scene from the end of the Set Symbology topic making sure that the topmost layer (which is the original EFF) is on.

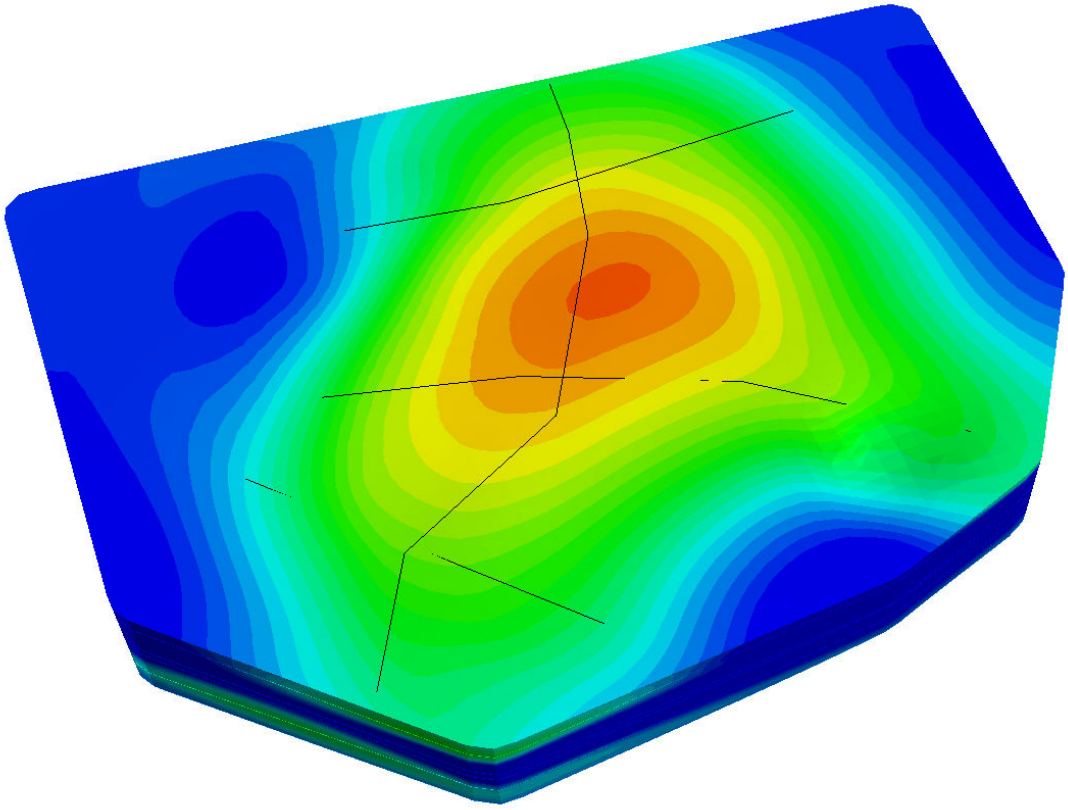



Set the Z Scale in the EnterVol Scene Control to be 1.0

Make the Graphics toolbar visible and choose the line tool.


It is best to create a new Graphics Layer. This is the only way to later control visibility of that layer other than deleting it.

Draw each segment of your fence path on the top surface of the layer, completing each segment with a left-double click.

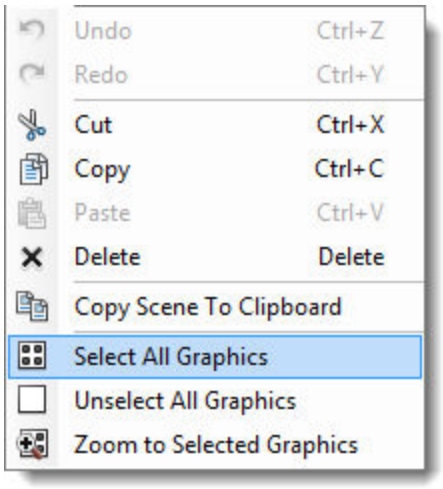


Note that the lines look broken because they are passing through the 3D top surface as we draw points that fall in low spots.

There are two options at this point:

1. You can use the new graphics layer to directly create the fence. This does not require the additional steps in ArcMap.
2. You can copy the graphics into ArcMap and save them as a shapefile. This extra step is important if you wish to straighten your fence to a 2D projection and want to use our advanced axes (grid) operations.

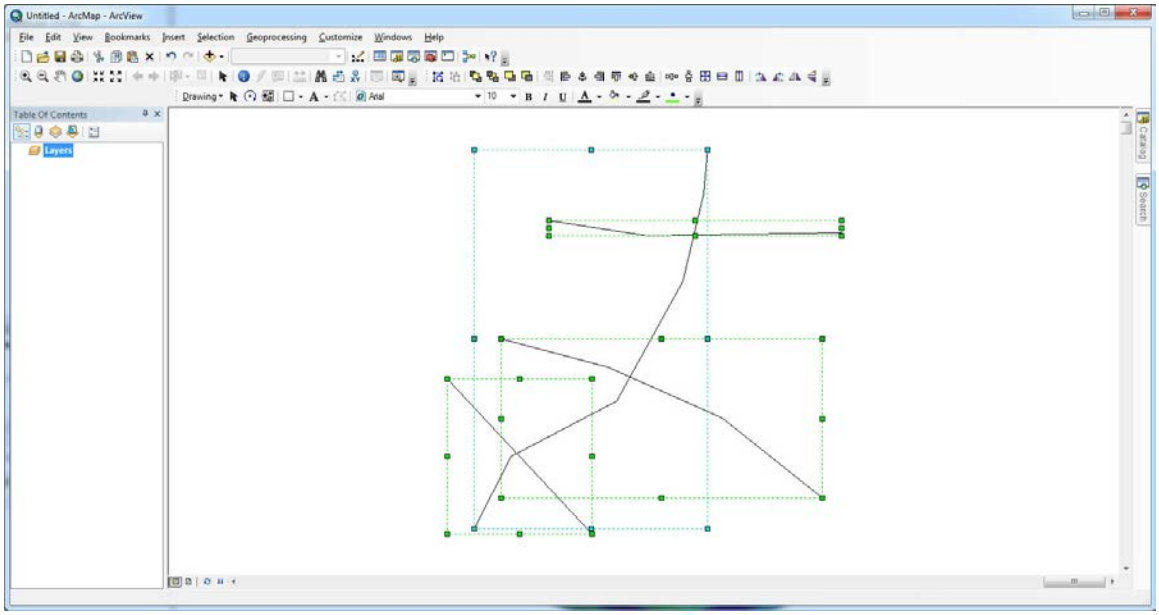
From the edit menu, choose Select All Graphics



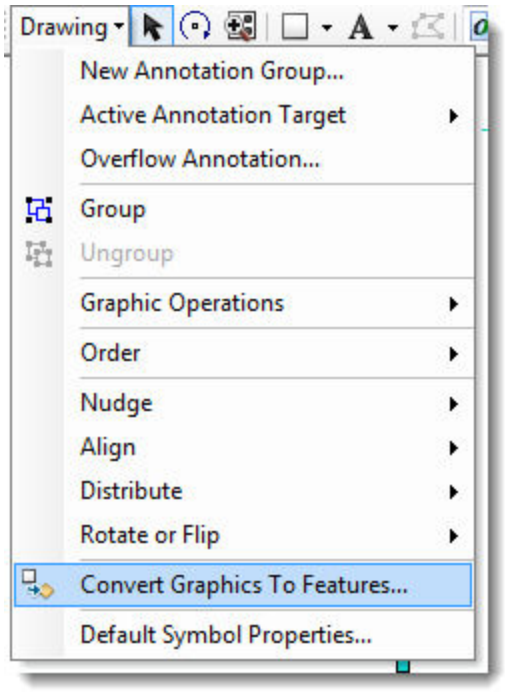

and then Copy (Ctrl+C)

At this point we want to convert our path to a shapefile. We do this in ArcMap.

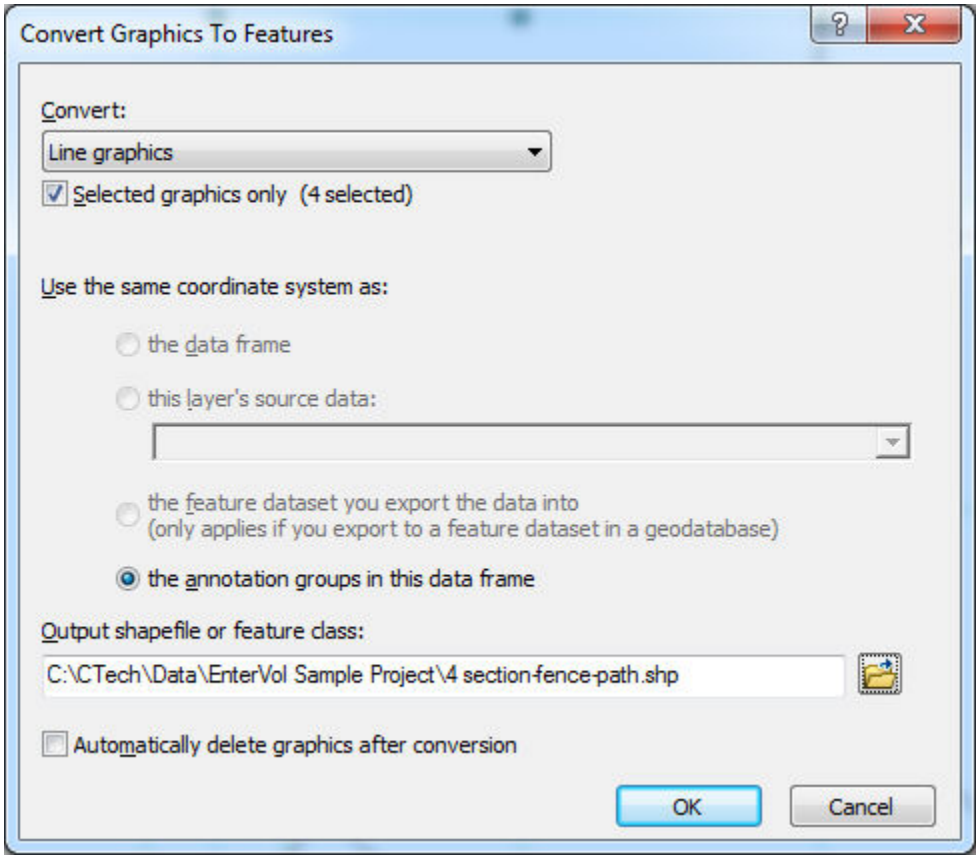
Open ArcMap with a New Blank Map. Highlight Layers and Paste (Ctrl+V) into ArcMap and from the Edit menu choose "Zoom to Selected Elements". You should see:



From the Drawing toolbar, choose "Convert Graphics to Features"



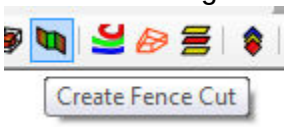

Provide a suitable name and folder

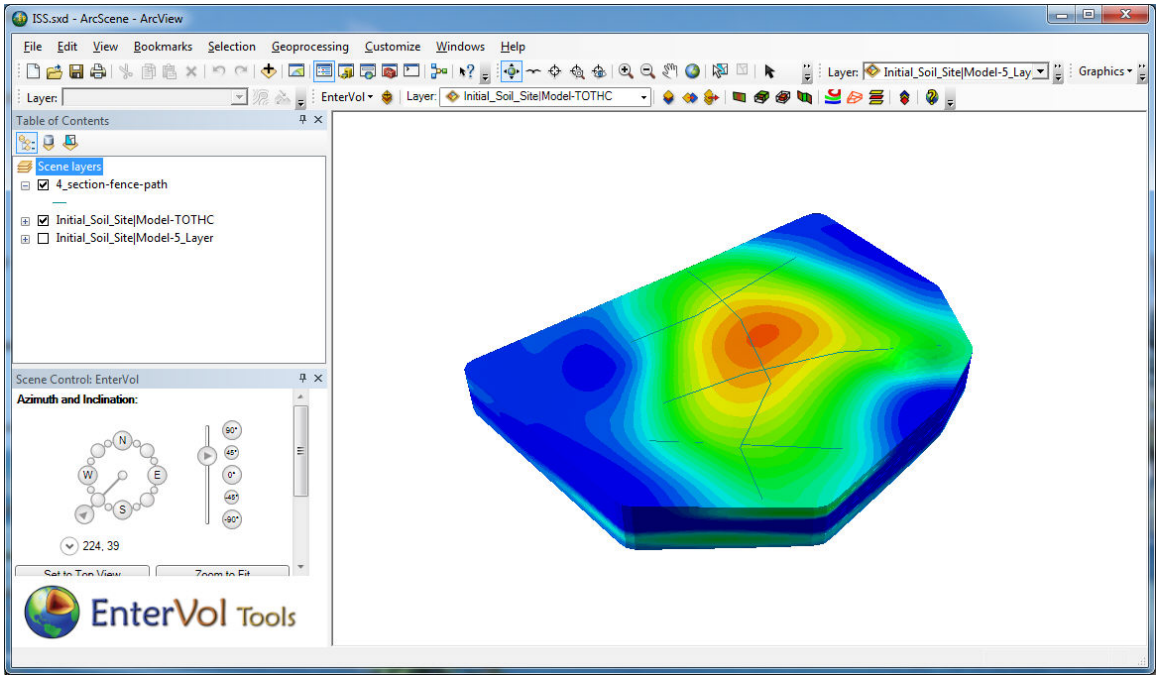



and click OK.

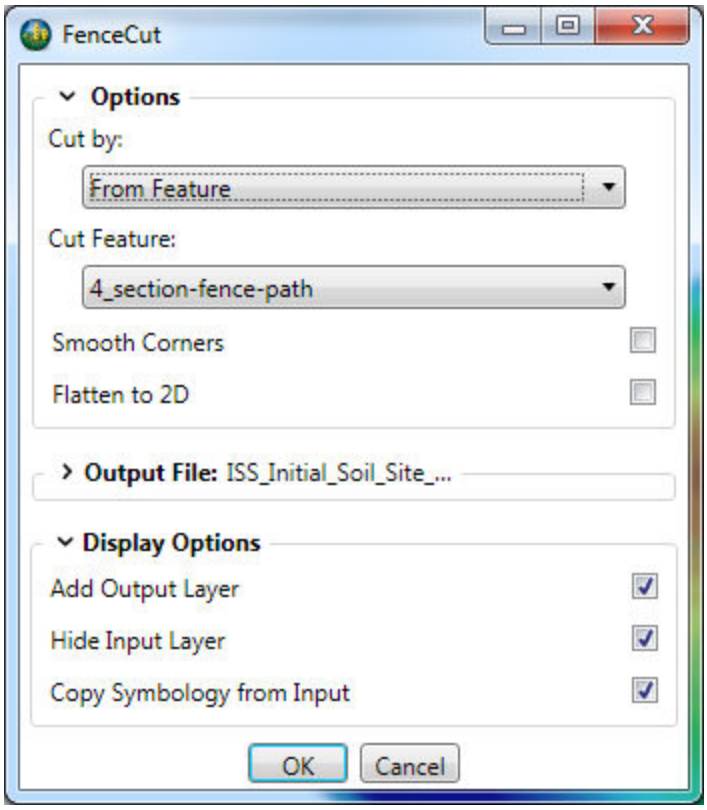
Back in ArcScene, delete the "graphics object" we just drew and click the Add Data button and add the shapefile we just created to the project.

Select the original EFF layer and click the Create Fence Cut button



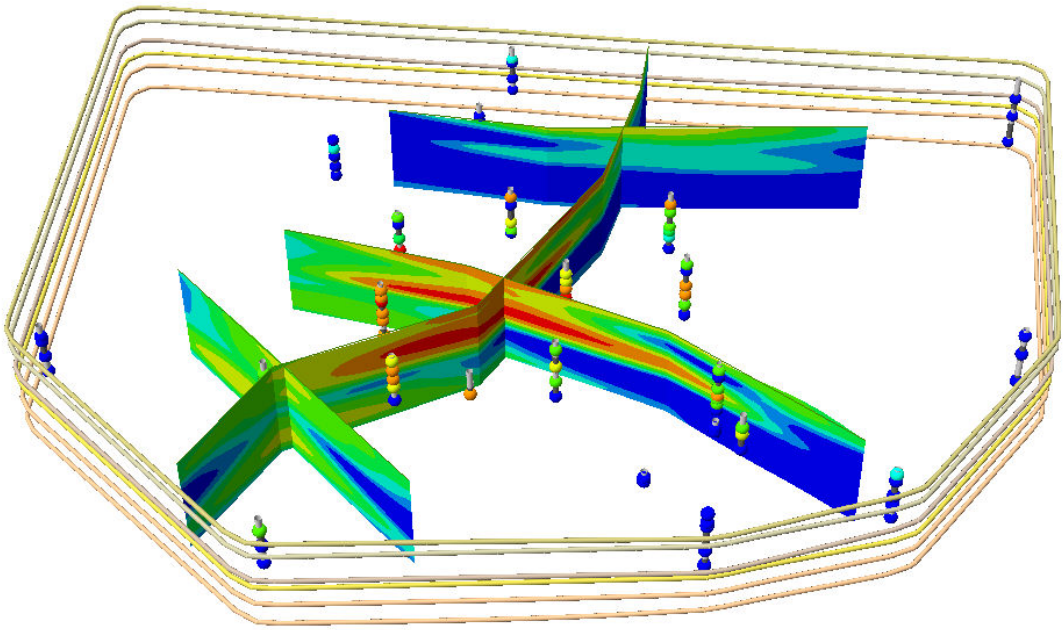

Choose the options you want such as:



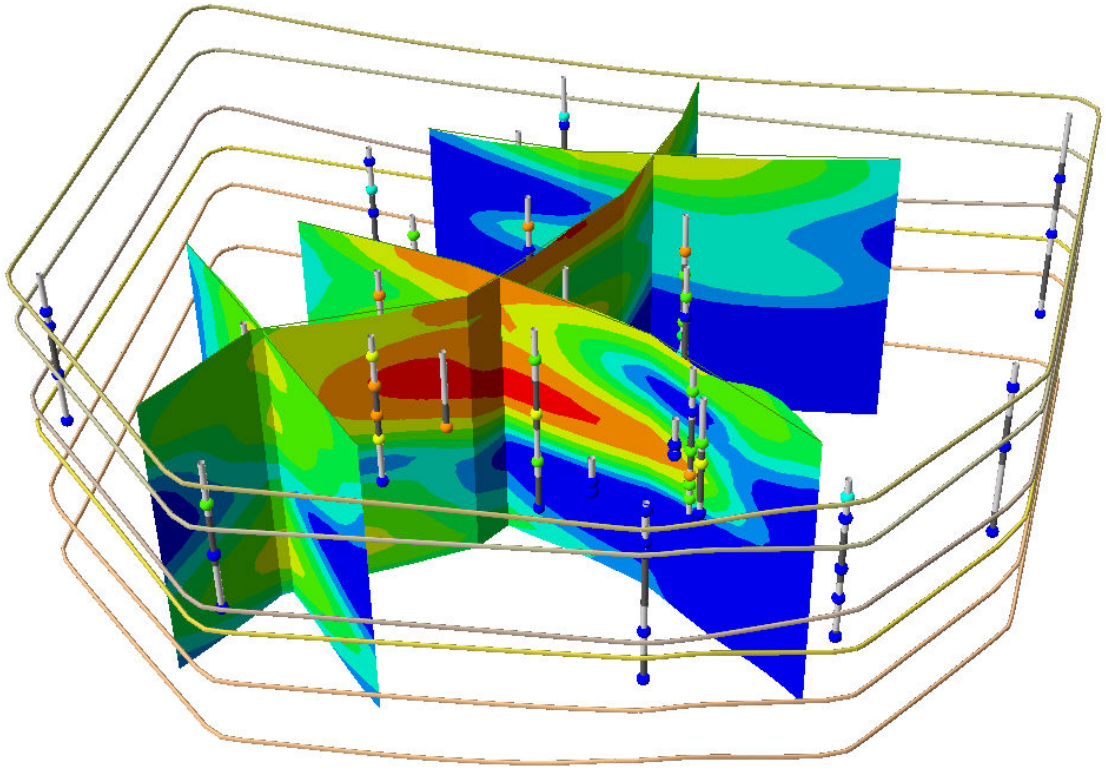

- Add Output Layer:
  - If this is not checked, the EFF will be created but not added to your scene or Table of Contents.
- Hide Input Layer
  - if not checked, your slice will exist but will be inside the input layer and not visible unless the input layer is turned off or made transparent.

- Copy Symbology from Input
  - use the same coloring as the object being sliced

You should see something like the view above and then choose OK. After turning on edges, the view will look like:



With Z Scale at 3.0, it is a bit easier to see details!



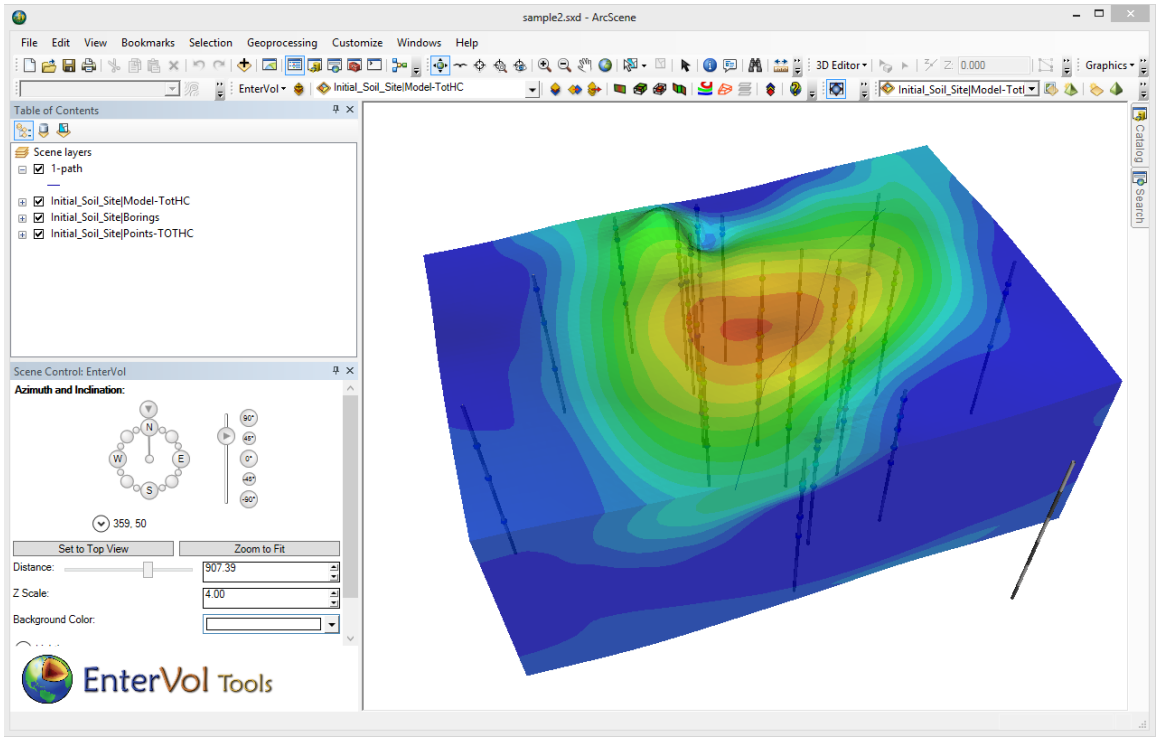
### Fence Cut Flatten to 2D

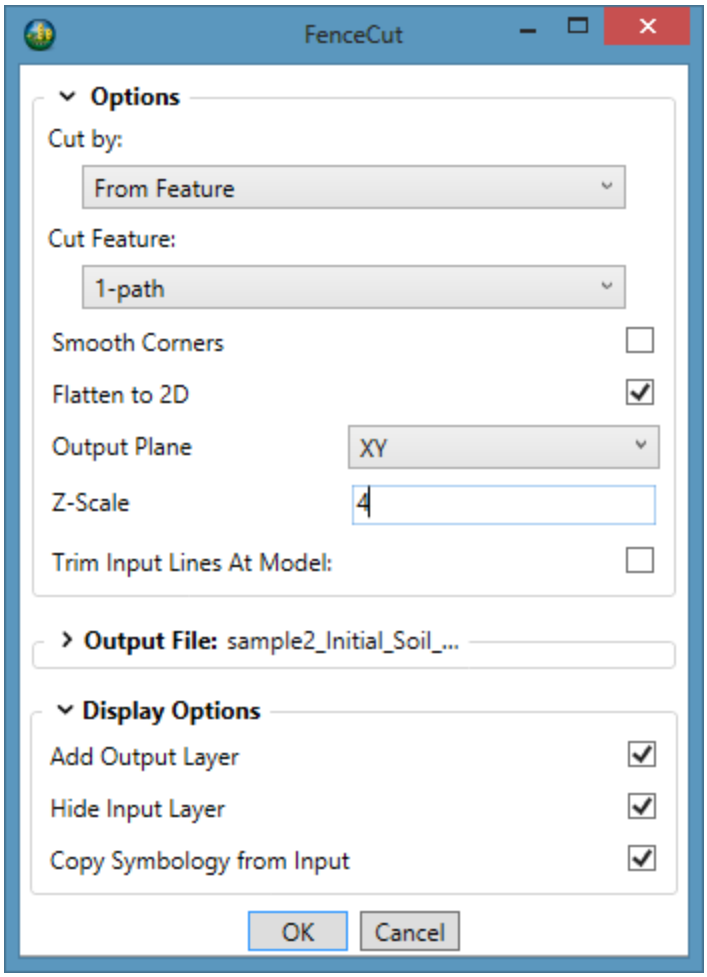
When you create a fence cut with a single path, you have the option to flatten the fence to 2D, which straightens the path after creation and remaps the resulting object to either the XY, XZ or YZ plane.

Note: The path used must contain only a single fence cross section.

In the project below, we have a single path shapefile line and a 3D model of Total HC. The volumetric model is slightly transparent since the fence path falls inside its extents. Remember, the Z coordinates of fence paths are irrelevant.

When any subsetting operation is performed, a summary of the data and settings used to create it is added to the Layer Properties under General.Description.

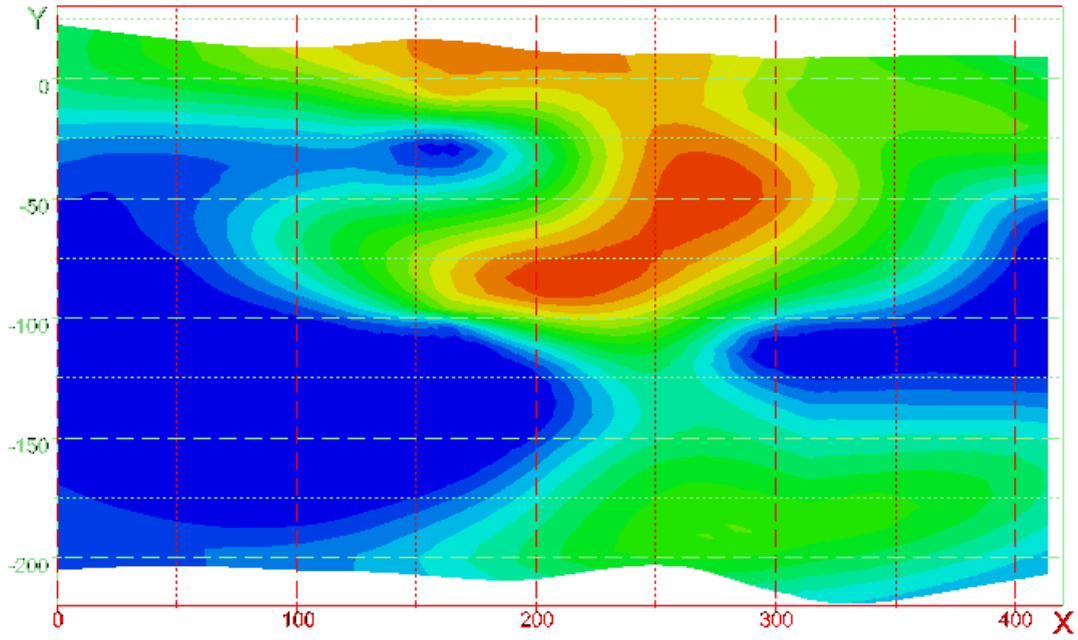





The Output Plane options are:

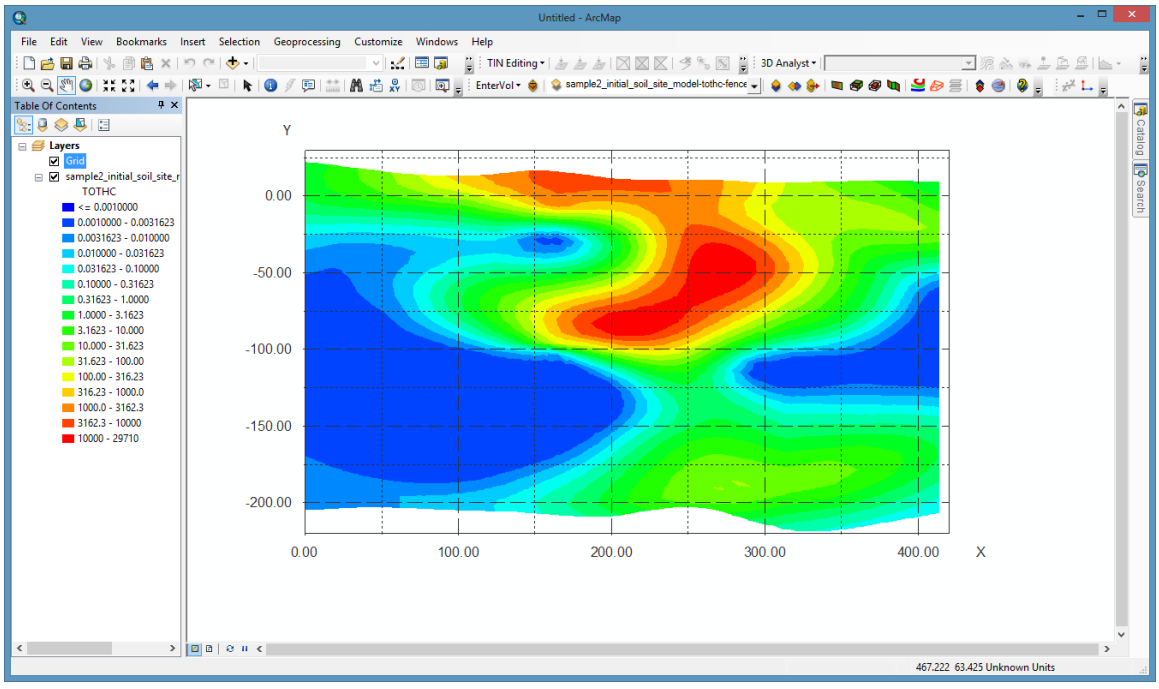
- XY is best for use with ArcMap
- XZ and YZ can be used with ArcScene or exported to EVS, however, please recognize that the coordinates of the flattened fence begin at the origin of the line (path) used to create it, which means that it will not be in the projection of the rest of your project.

Zooming to the resulting flattened fence in ArcScene with axes added will show:

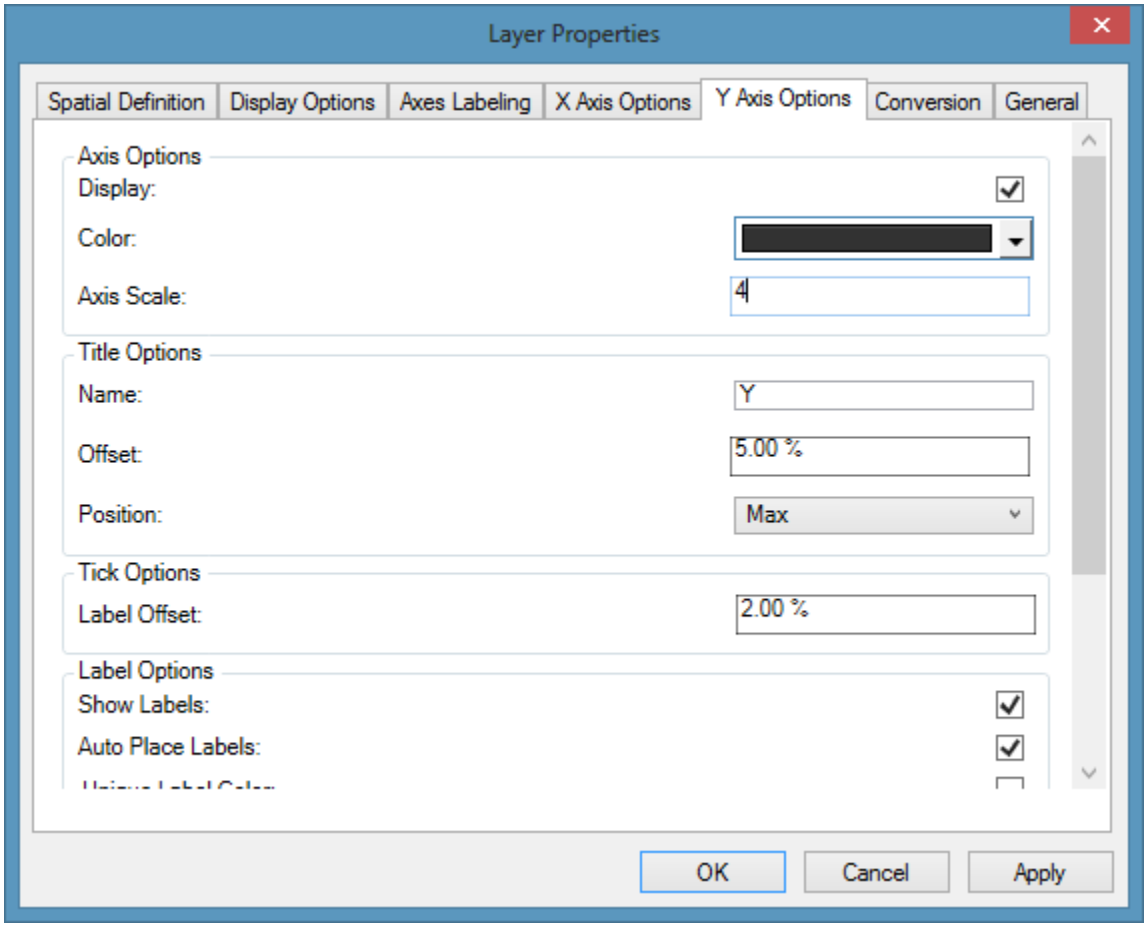



However, because we created this in the XY plane with a Z-Scale of 4, the Y axes coordinates are now 4 times greater than they should be.

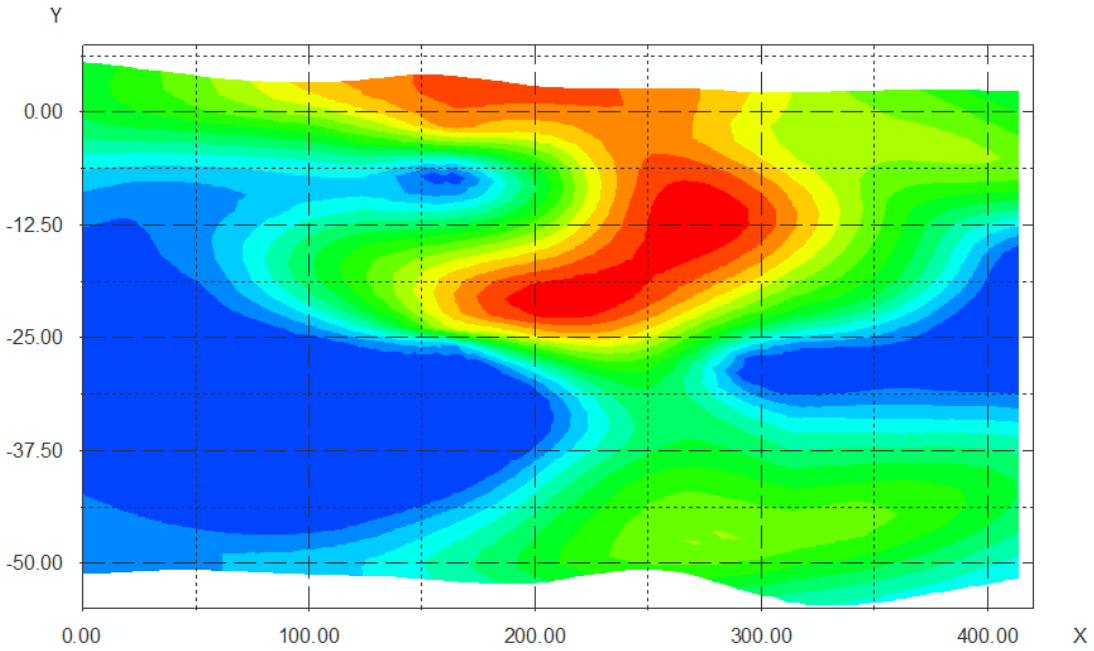
If we import the same 2D fence into ArcMap and add axes (called grid in ArcMap) we get:



However, in ArcMap, the Grid (axes) layer has an option to set Y Scale. If we change that to be 4

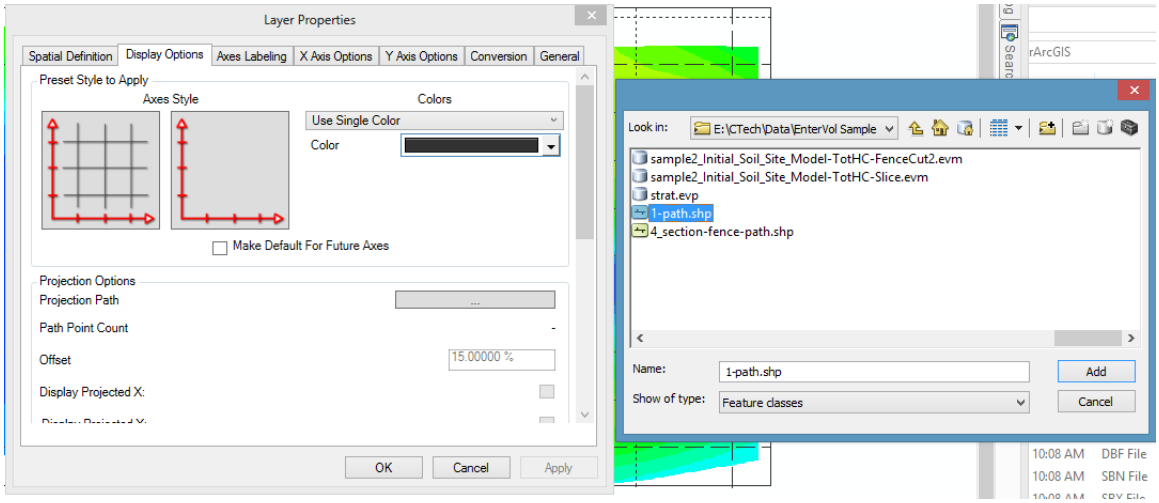



our axes will show correct elevations:




Another important option is "Trim lines at Model". When this toggle is off, the origin of the fence corresponds to the start of the line. The order of the points that define the line therefore makes a significant direction in the resulting fence. If the toggle is ON, the origin is determined by the line as trimmed by the model, however the order of drawing is still relevant.

Finally, our bottom (X) axis is really showing the path length, not an actual X coordinate. Under Grid.Display_Options, we can set the projection path using the same shapefile that was used to generate the fence.

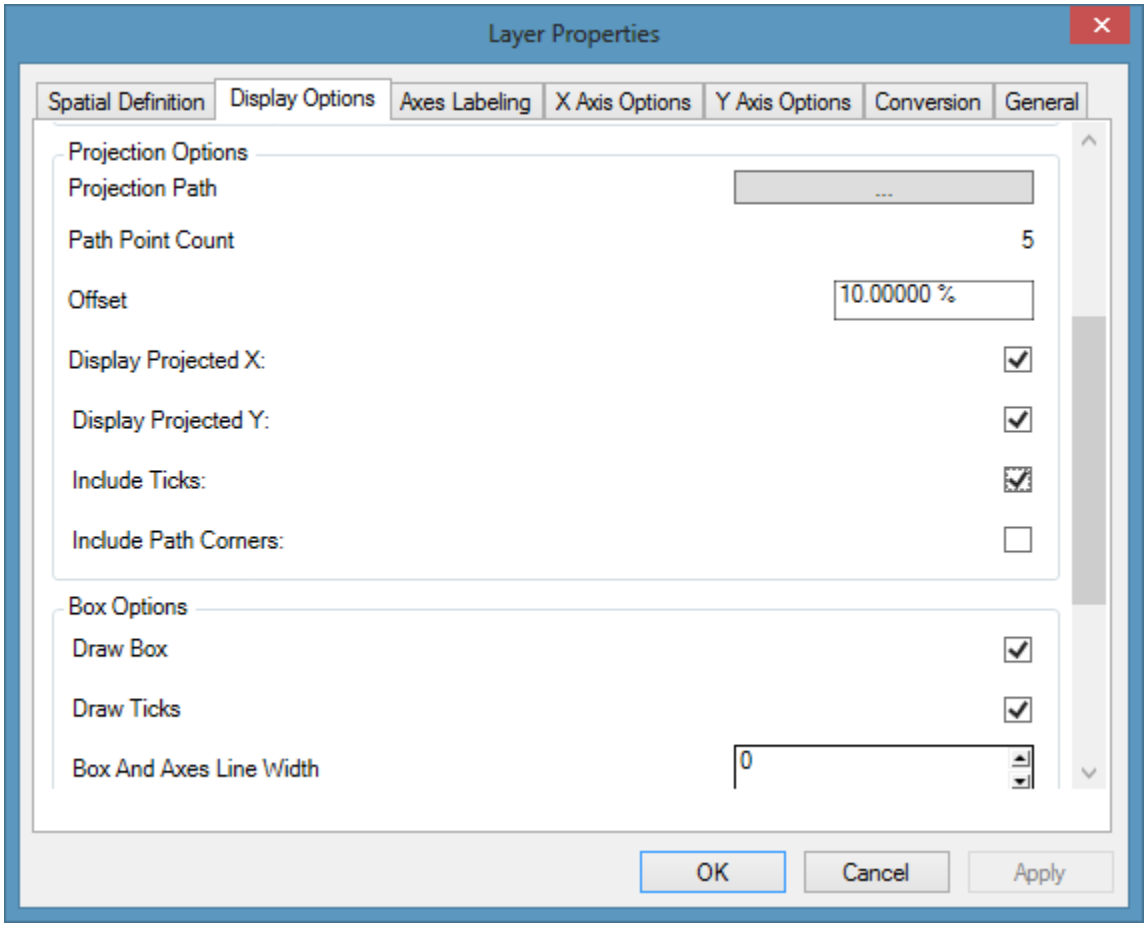



Be sure to turn on Display Projected X (and Y) and we'll change the Offset to 10%.

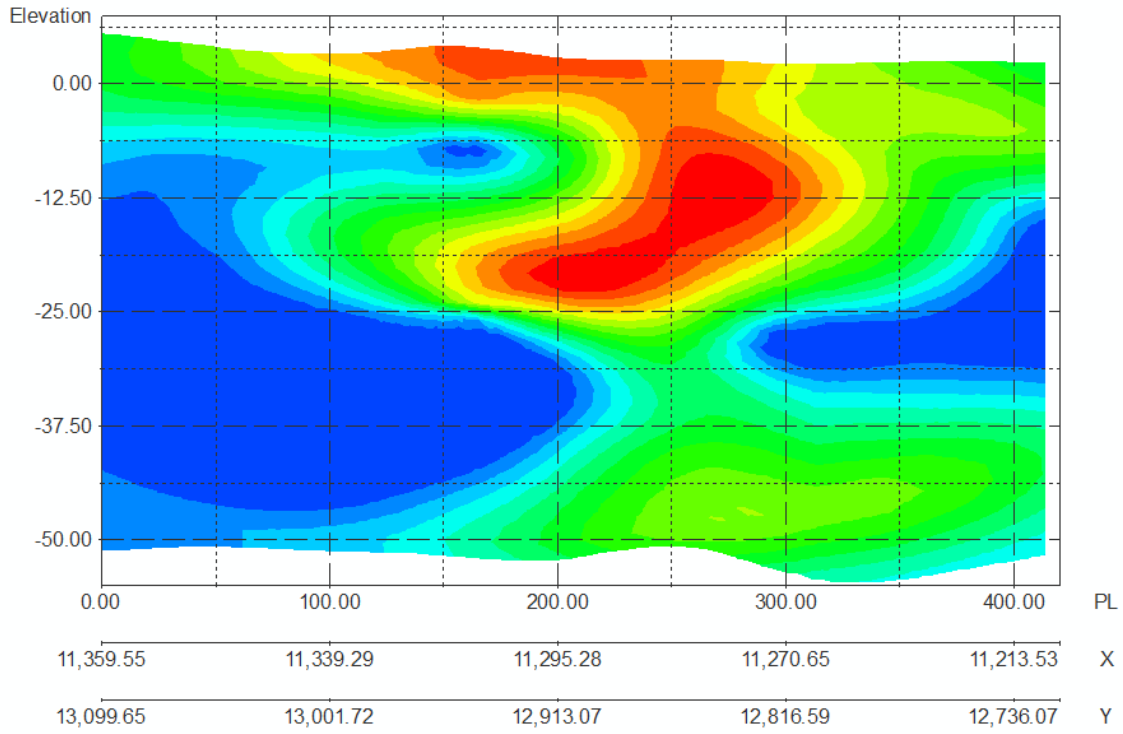
At this point there are two other interesting options. These are:

1. Include Ticks: which will put the X (and Y) true coordinate labels that correspond to the major axis tick distances along the path. In this case 100 feet.
2. Include Path Corners: This will put labels at the endpoints of each segment of the fence. If your fence was drawn from boring-to-boring, these will be your boring coordinates.

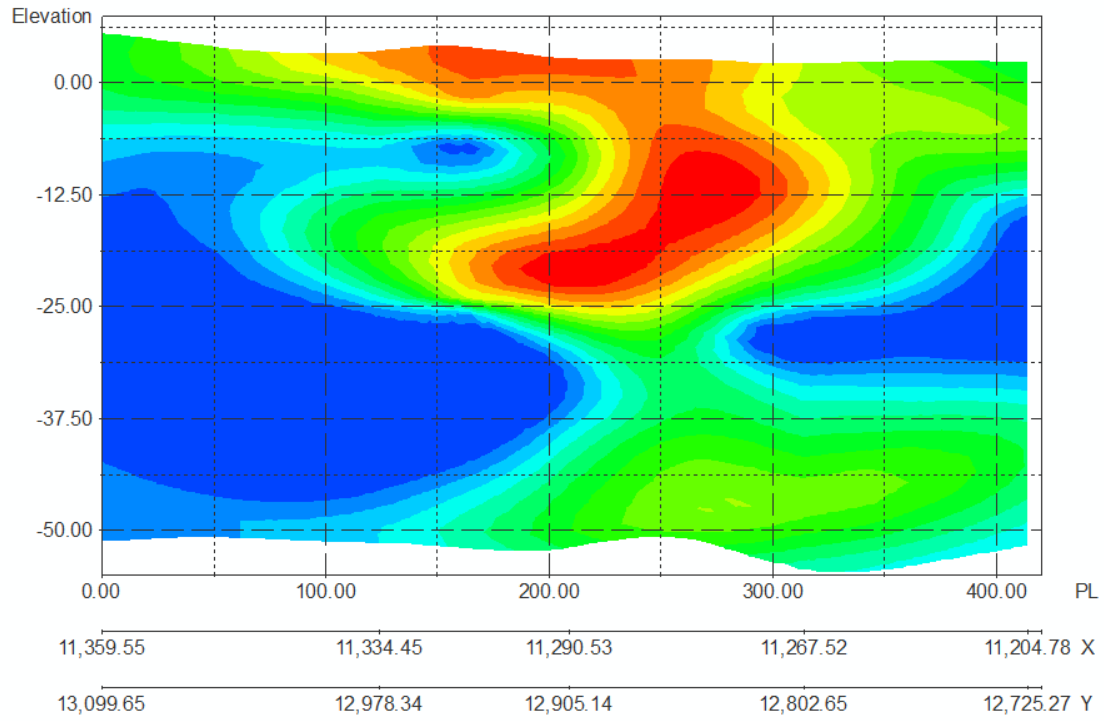
Generally you will not want both of the above options as the labels will tend to be on top of each other.




For the first *Include Ticks* option we get:



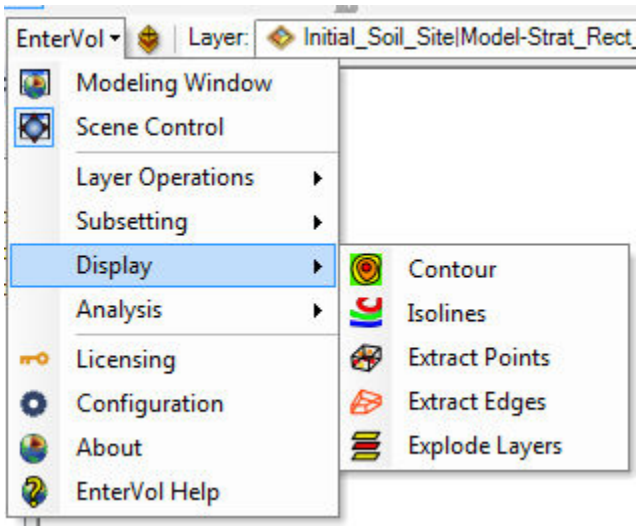

And for the second *Path Corners* option:



note: for the above two figures we've also edited the X and Y Axis Title Names

## Display

### Display



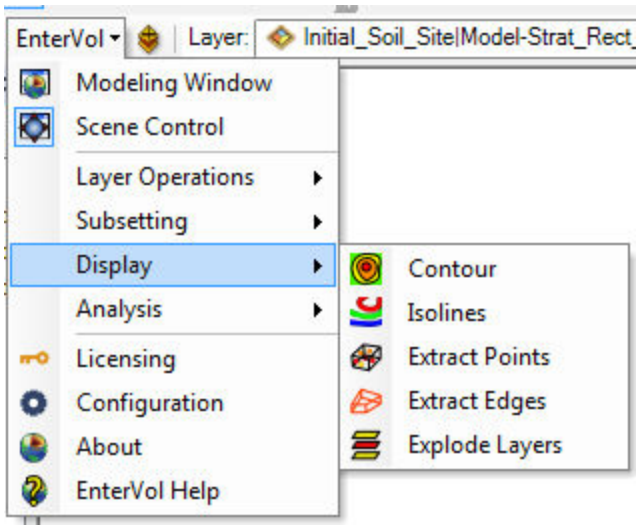

The display tools consist of:

1. Contour
2. Isolines
3. Extract Points
4. Extract Edges
5. Explode (geologic) Layers
6. Set View from Direction (ArcMap only)

### Contour

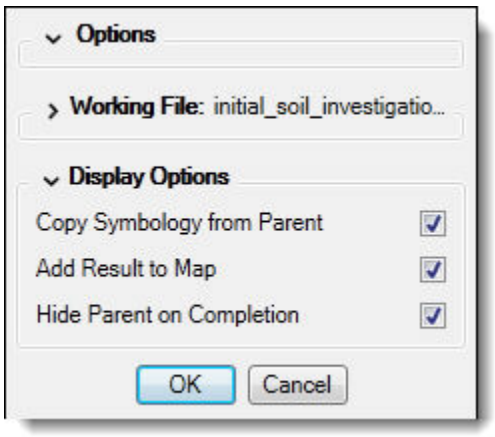
Smooth Contours are automatically generated whenever any subsetting operation is performed. Please note that contours are only possible because of nodal data. The "Contour" tool is accessible from the EnterVol menu:

When a contour is performed, a summary of the data and settings used to create it is added to the Layer Properties under General.Description.




Since contouring of all subsetting displays is automatic, the only time when the contour tool is needed is when you want to export a subsetting operation as a shapefile.

Select the scene layer representing slice output and click on the *Contour* icon.



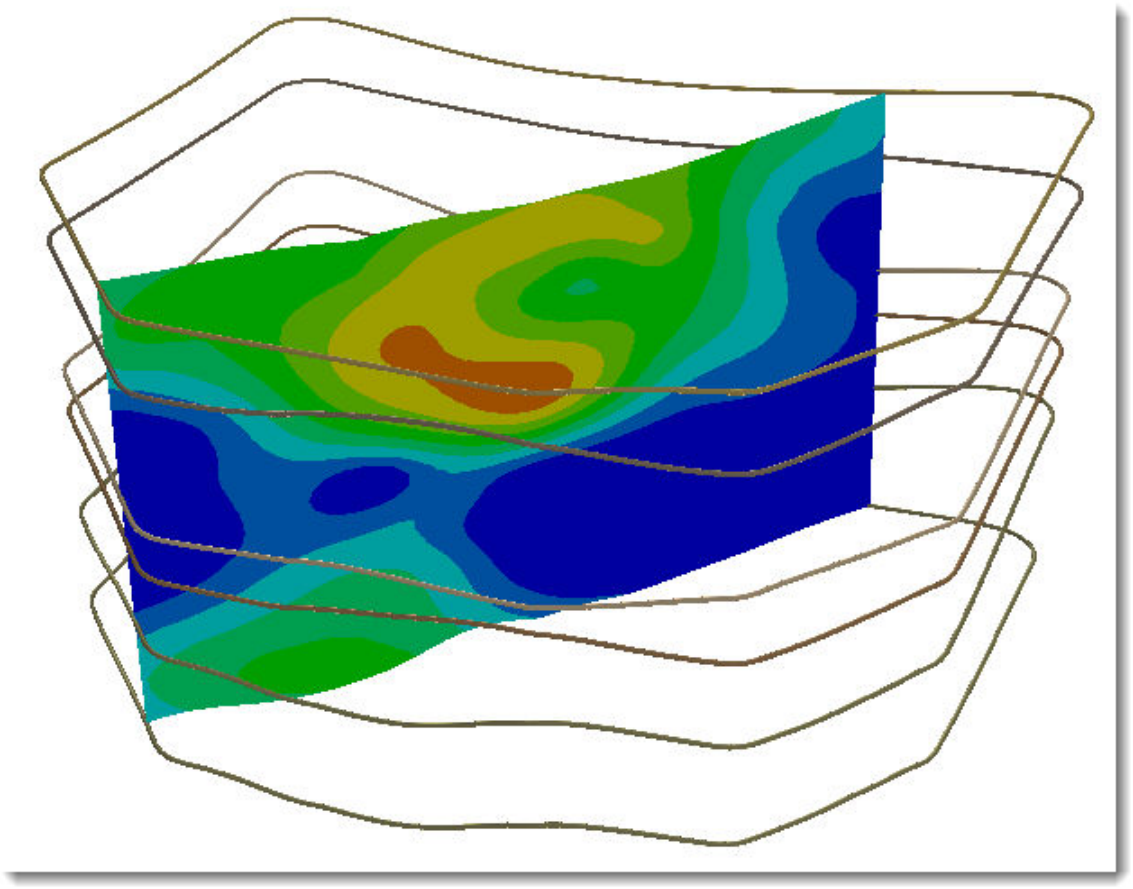

You can specify the layer to be contoured.


Choose the Display Options you want such as:

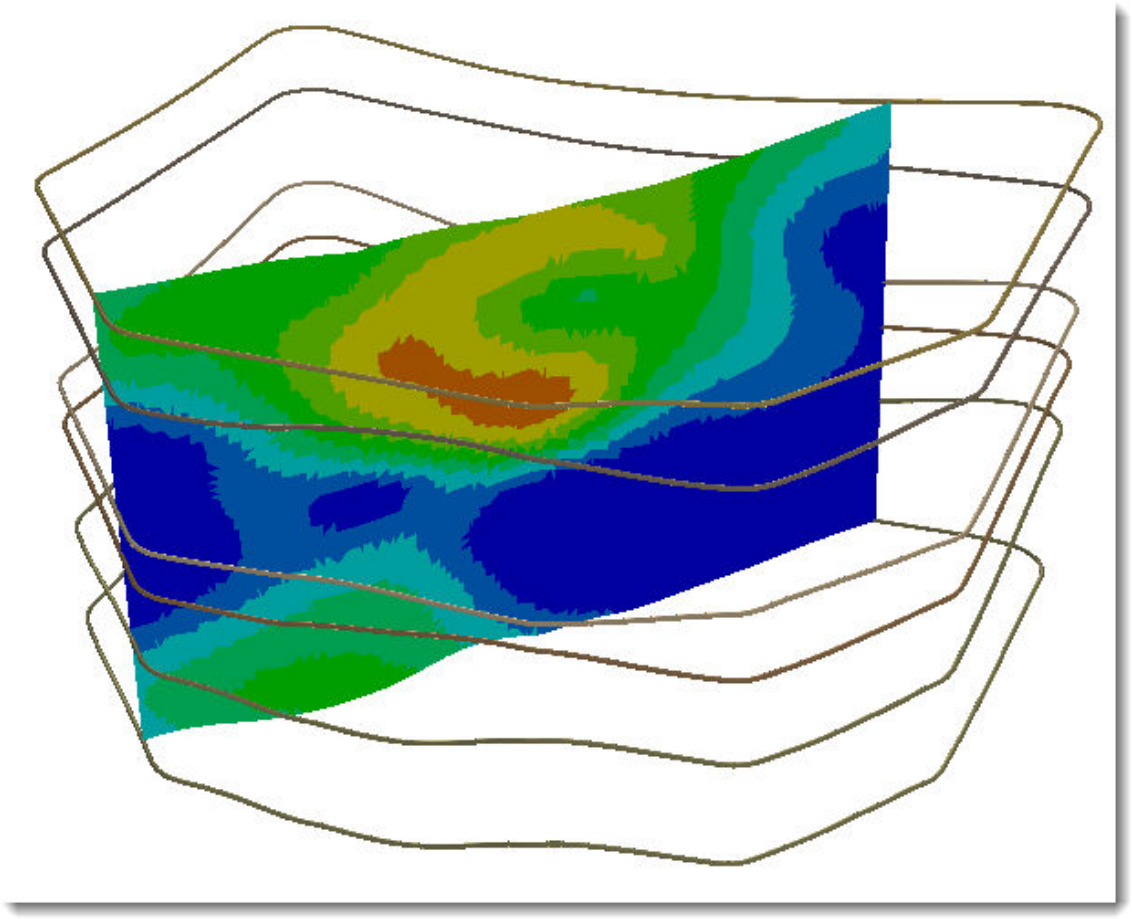
- Add Output Layer:
  - If this is not checked, the EFF will be created but not added to your scene or Table of Contents.
- Hide Input Layer
  - if checked your input layer will be turned off

- Copy Symbology from Input
  - use the same coloring as the object being sliced

Press OK with the default options and you should see:



Compare this to the original slice output below that would be obtained if you were to write a shapefile without contouring first.



The jagged appearance above is the result of having to color each original triangle in the slice surface a solid color. Using the nodal data inherent in the slice, more accurate boundaries between the data regions can be computed and individual triangles are divided to create the more accurate and smoother output.

### Isolines

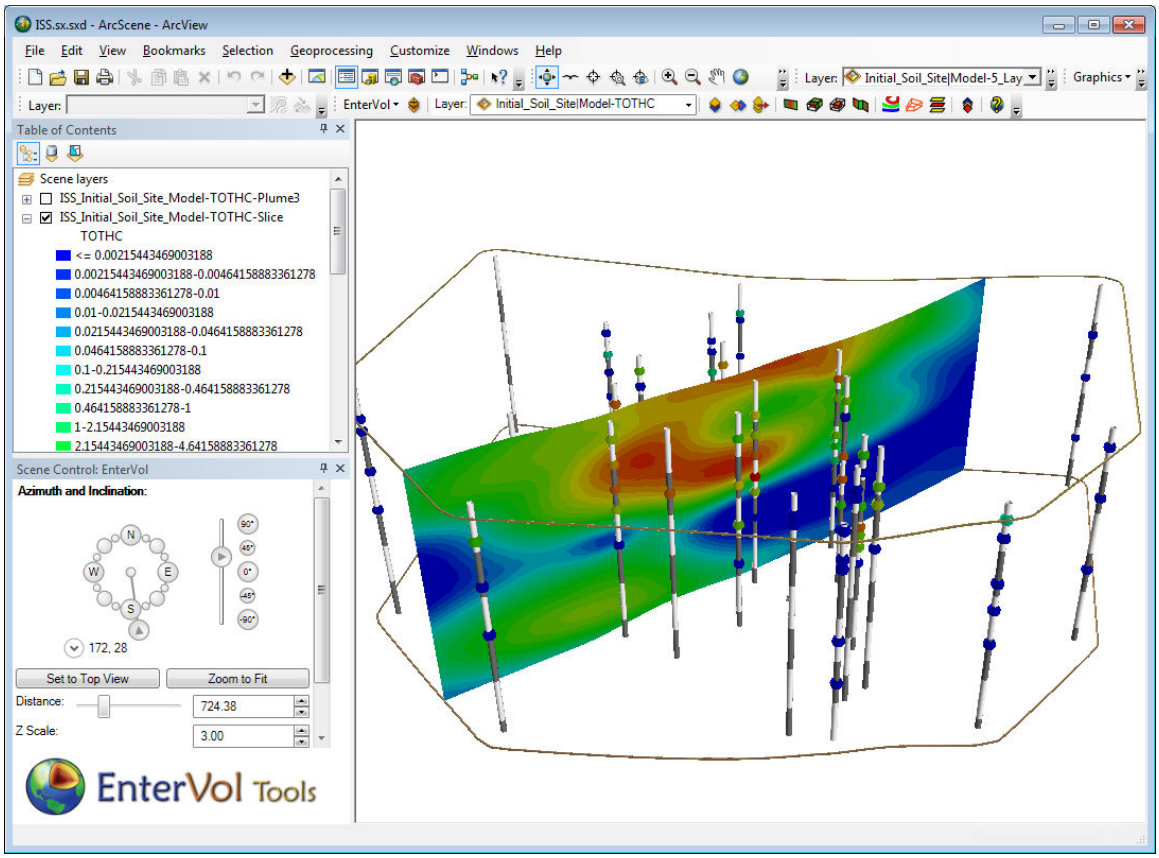
Isolines can be created on planar or complex surfaces. They require that the underlying layer have continuum data such as elevations, concentrations, etc. EnterVol determines the path of the isolines using the nodal data in order to achieve smooth and accurate contours.

When isolines are created, a summary of the data and settings used to create it is added to the Layer Properties under General.Description.

The isolines will be generated using the symbology (data) displayed on the layer used as input. It will also inherit the same breaks. In the case below, this will be TOTHC (total Hydrocarbons) with even decade intervals.

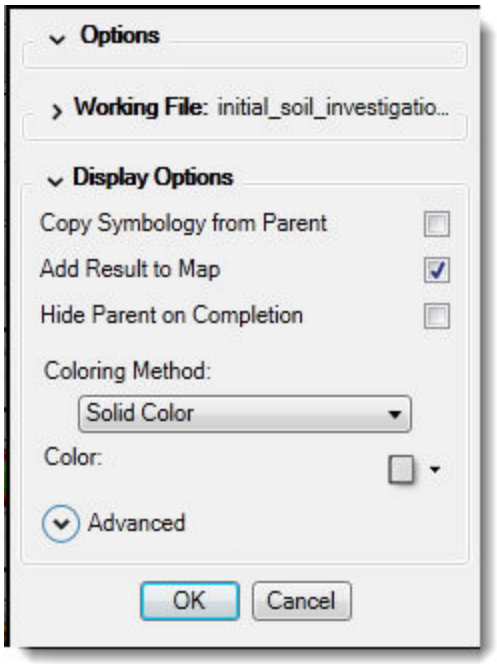
Let's begin with the scene layer created at the end of the Smooth Contours topic and add isolines to our slice.

Begin by choosing the layer on which you want to apply isolines We'll choose the Contour layer,




and press the Isolines button under Display.

Note that the default options are a bit different.

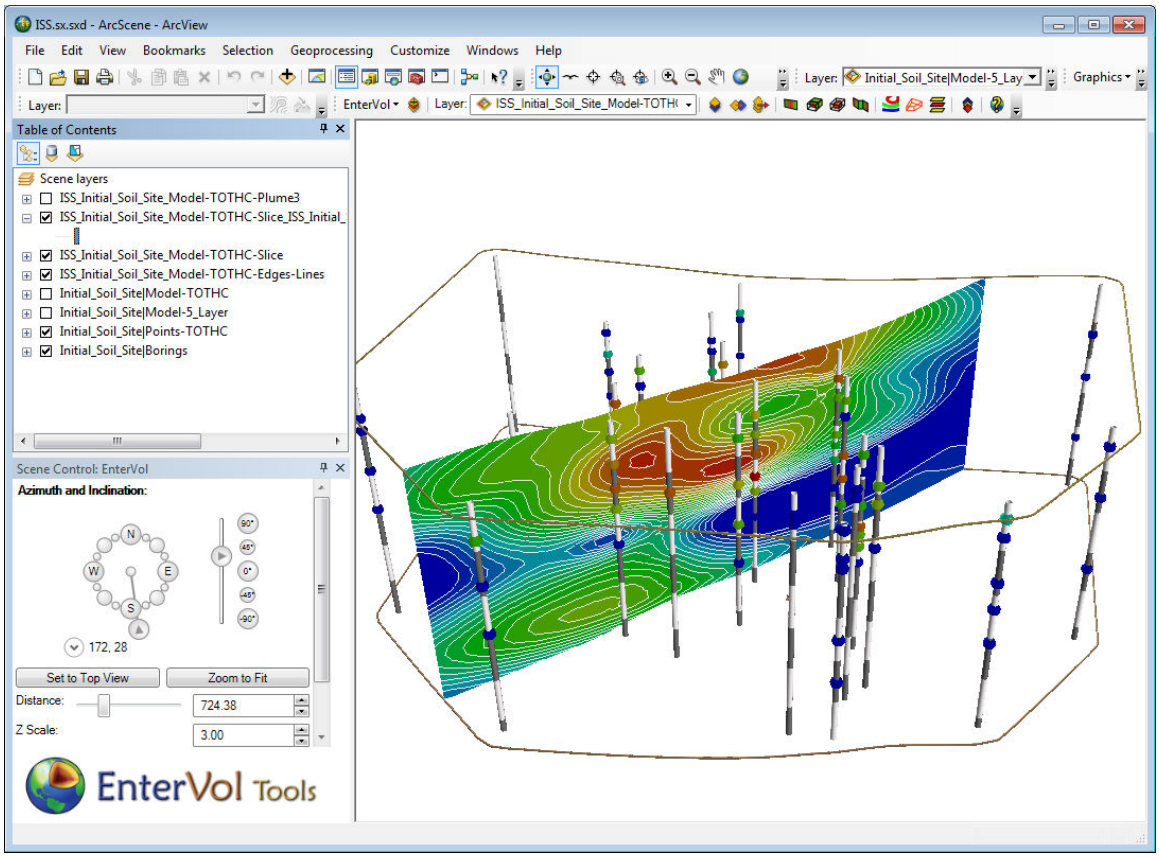



Choose the options you want such as:

- Add Output Layer:
  - If this is not checked, the EFF will be created but not added to your scene or Table of Contents.
- Hide Input Layer
  - if checked your input layer will be turned off

- Copy Symbology from Input
  - Default is off which gives a solid off-white color
  - If checked, use the same coloring as the object being sliced

By default we don't copy the Input symbology, since when our isolines are the same color as the input layer, they are difficult to see. If you want only isolines and not the source, then you might want to copy the symbology and Hide the Input Layer.



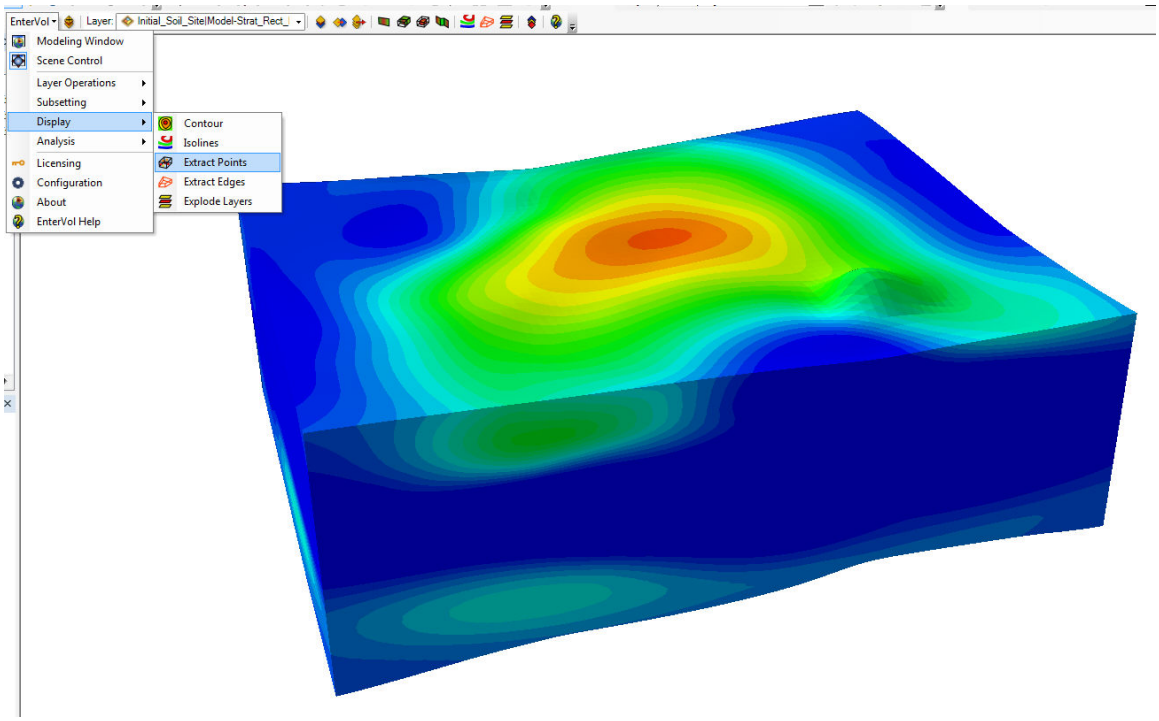

Note that the isolines are on the same breaks as our underlying surface. The isolines would be identical if we had chosen the original slice layer rather than the smooth contour layer. The reason is that both are based on the same nodal data though the cell attributes (data) are quite different.

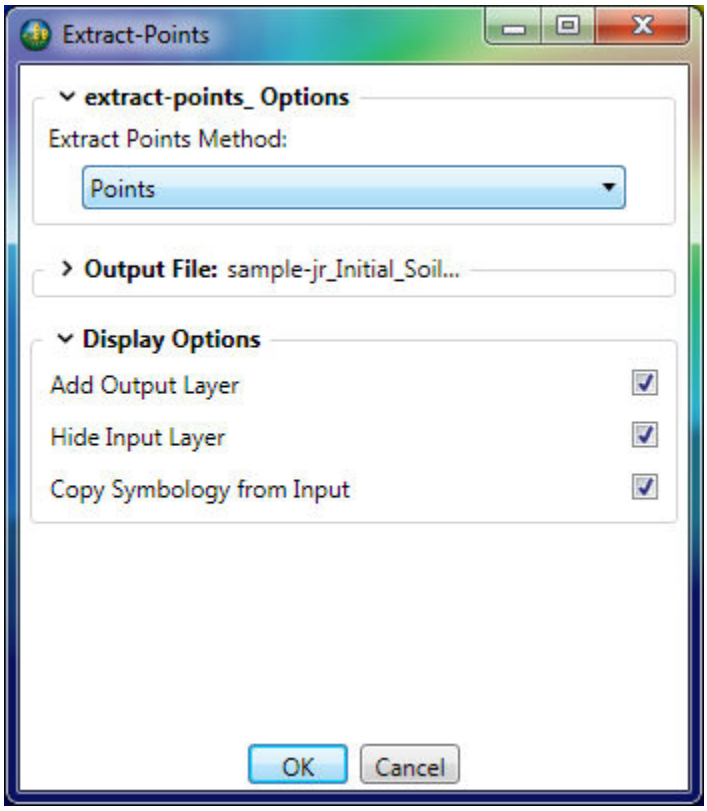
### Extract Points

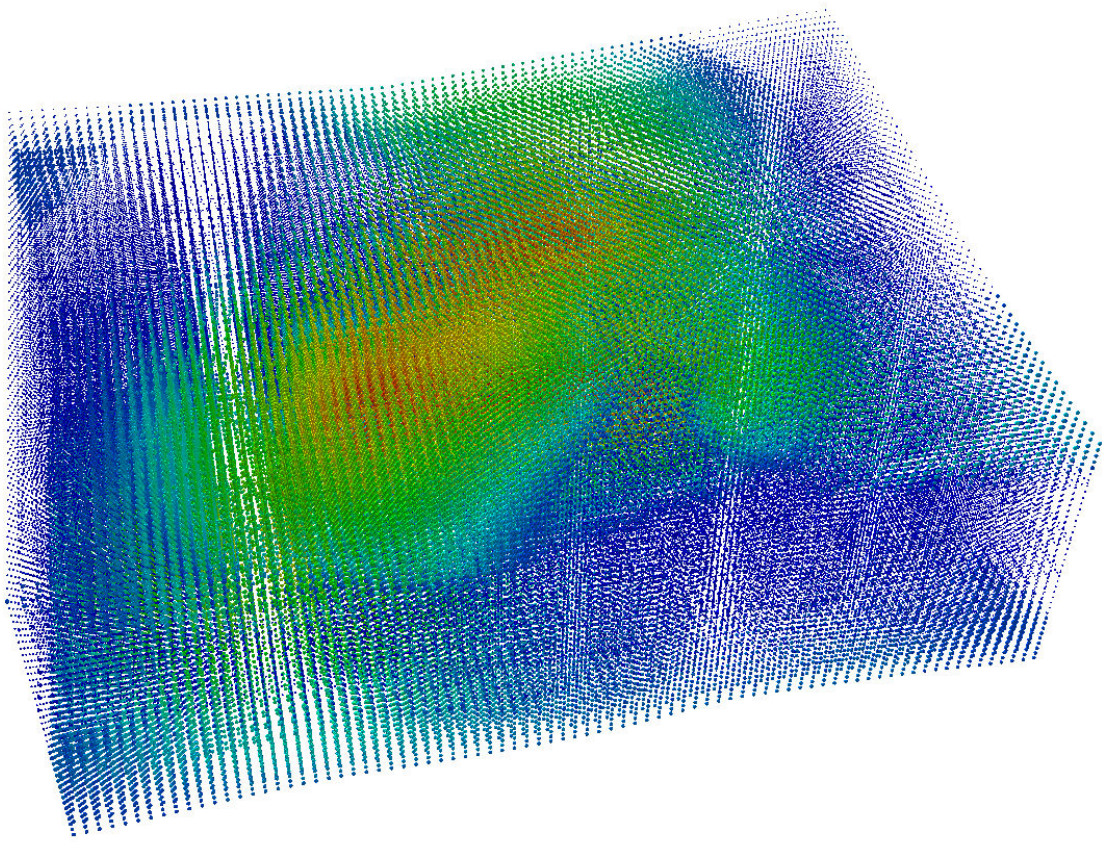
The Extract Points tool allows you to extract and display a point cloud representing:

- The Points (nodes) that define the corners of each cell.
  - When the data is cell data, such as geologic (stratigraphic) material, the data is not there and therefore the Points option will not map the symbology. In this case, use the cell centers option.
- The cell centers as points.
  - If mapping nodal data, the displayed value is the average of all of each cells corners.

When extract points is performed, a summary of the data and settings used to create it is added to the Layer Properties under General.Description.


EnterVol
Layer:
Initial_Soil_Site|Model-Strat_Rect_
Modeling Window
Scene Control
Layer Operations
Subsetting
Display
Analysis
Licensing
Configuration
About
EnterVol Help
Contour
Isolines
Extract Points
Extract Edges
Explode Layers


Extract-Points
extract-points_ Options
Extract Points Method:
Points
Output File: sample-jr_Initial_Soil...
Display Options
Add Output Layer
Hide Input Layer
Copy Symbology from Input
OK
Cancel



**Explode Layers**

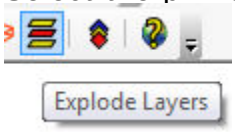
C Tech EFF files representing multiple geologic layers can be exploded, or separated based on the layer number (Geo_Layer) or Material IDs. This allows you to visualize the internal lithology of model and to see how analytes are distributed by geology.

When explode layers is performed, a summary of the data and settings used to create it is added to the Layer Properties under General.Description.

It is generally confusing to have layers visible which represent both exploded and unexploded representations of the same model in your scene. For that reason, it is often best to explode the entire model and then perform subsetting and display operations on the exploded model. Otherwise you will need to perform two operations for each output (e.g. smooth contours, fences, isolines, etc.)

For the example below, we'll begin with the scene at the end of the *Add C Tech Field* topic.

Select the primary layer and click on the Explode Layers button




Select the method to explode 

.

Let's change the default explode distance 

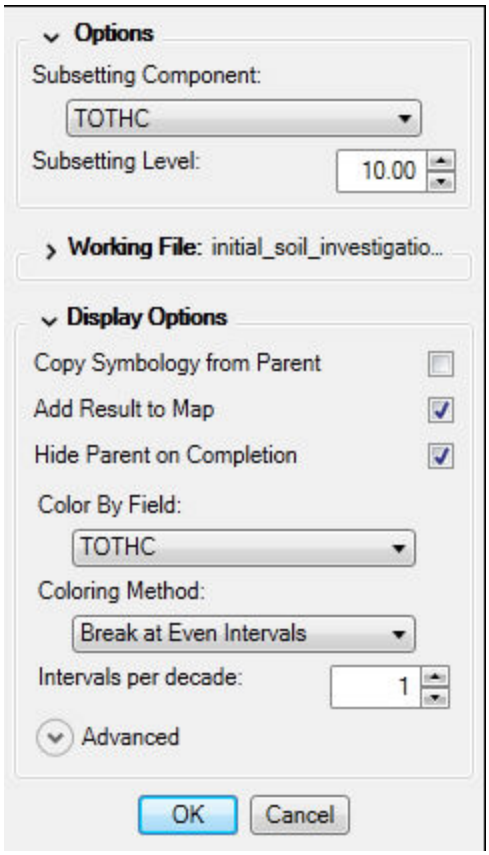
and set the options you want such as:

- Add Output Layer:
  - If this is not checked, the EFF will be created but not added to your scene or Table of Contents.
- Hide Input Layer
  - if not checked, your slice will exist but will be inside the input layer and not visible unless the input layer is turned off or made transparent.

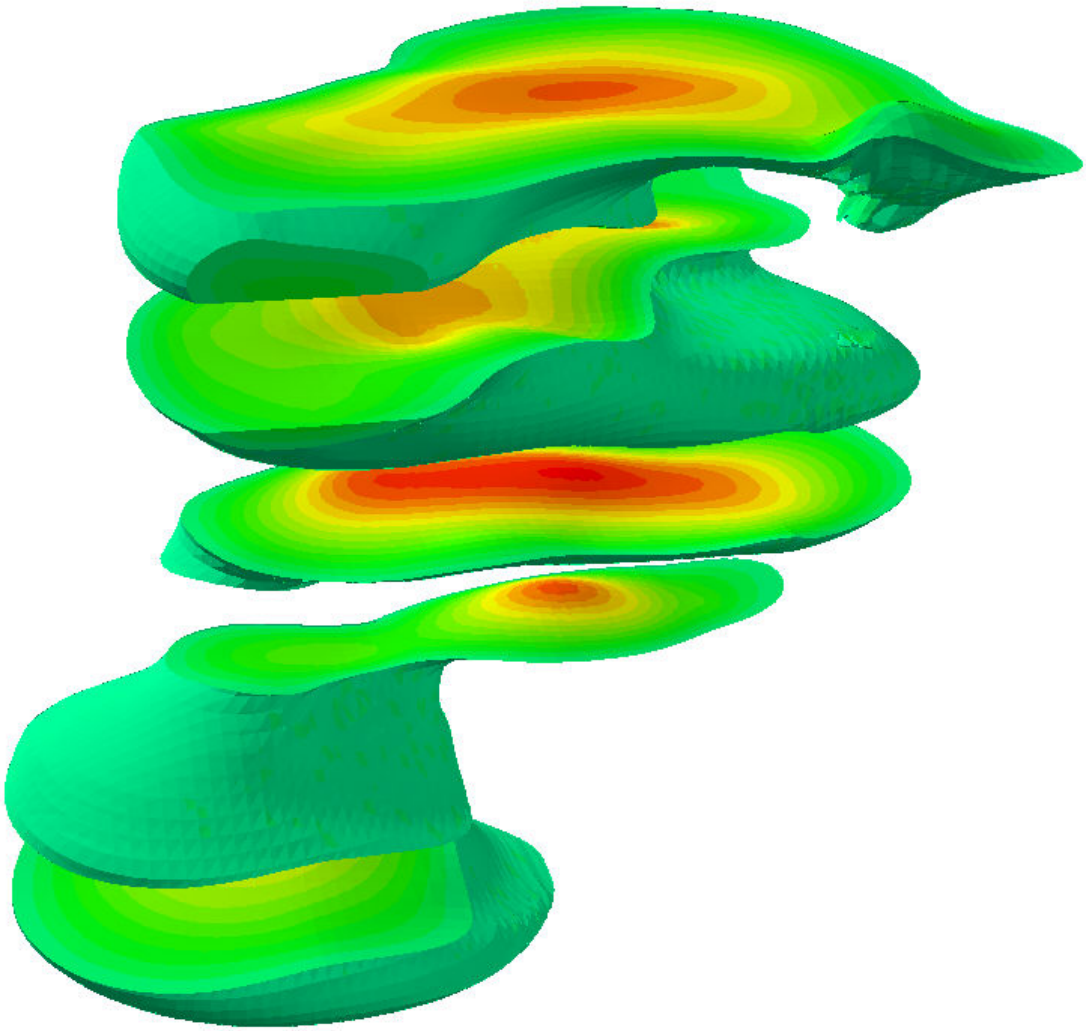
- Copy Symbology from Input
  - use the same coloring as the object being sliced

Upon clicking OK, and setting Z Scale to 5.0 your scene should show.

If we select the exploded layer, and create a TOTHC plume at 10.0 mg/kg




we would have

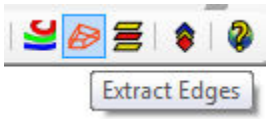


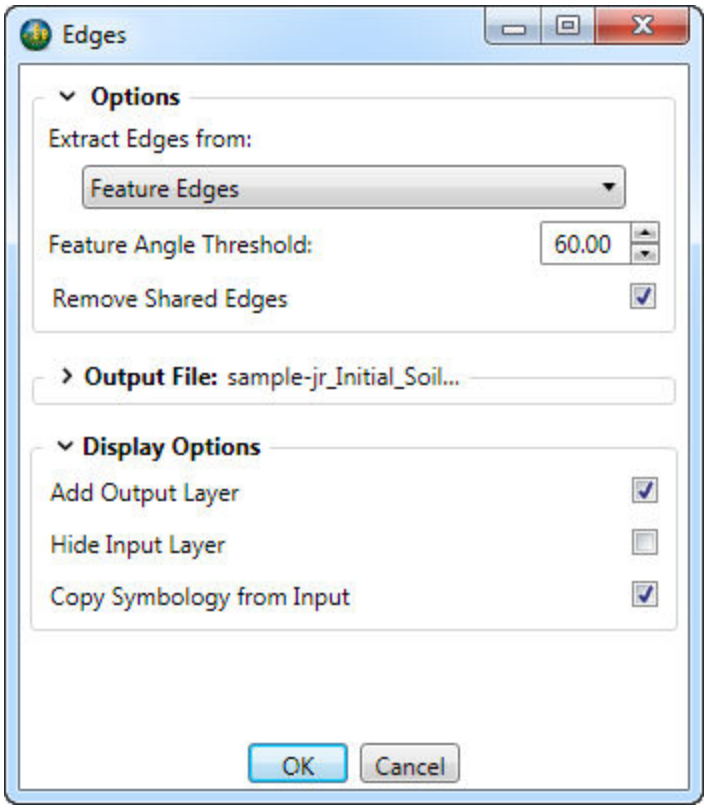
**Extract Edges**

Let's begin with the scene layer created at the end of the Create Fence Cut topic and add the external edges of the volume.

Select the scene layer representing the original volumetric grid and click on the Extract Edges icon.

Note: When extract edges is performed, a summary of the data and settings used to create it is added to the Layer Properties under General.Description.






The following controls are available:

- *Extract Edges from*:
  - Boundary Edges are derived from the inherent grid rather than the topography of the grid
  - Feature Edges (Default) are derived from the topography of the grid
- *Feature Angle Threshold*: is used to determine the slope change between surface elements which results in an edge being constructed.
- *Remove Shared Edges* toggle: The edges between geologic layers which are not exploded will be removed if this toggle is on.

You can also specify the location and name of the EFF file which represents the slice will be created.

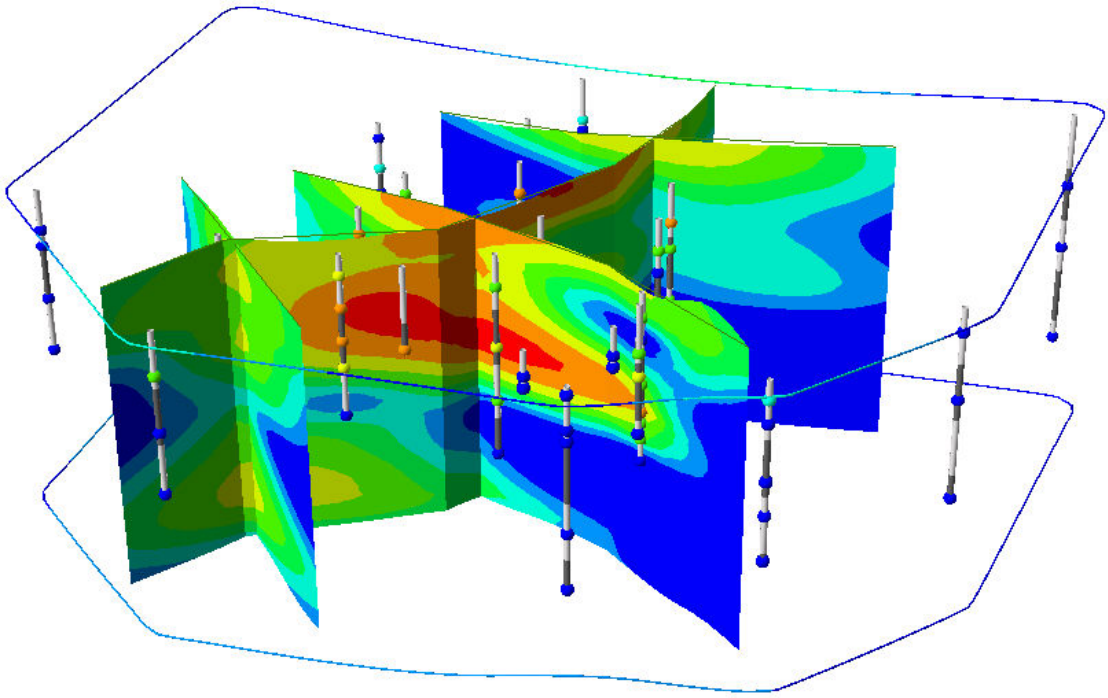

Choose the Display Options you want such as:

- Add Output Layer:

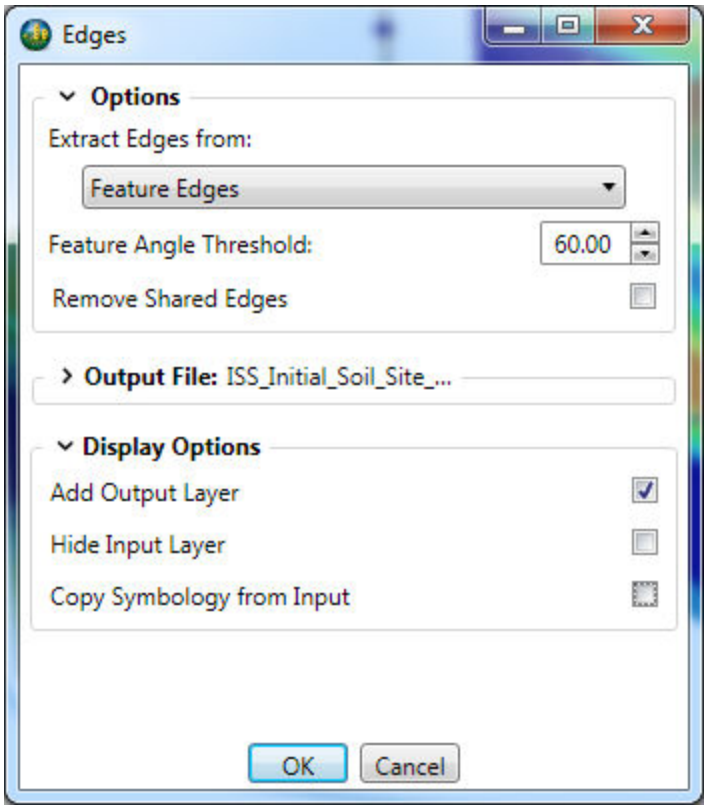
    - If this is not checked, the EFF will be created but not added to your scene or Table of Contents.
- Hide Input Layer
    - if checked your input layer will be turned off

- Copy Symbology from Input
    - use the same coloring as the object being sliced

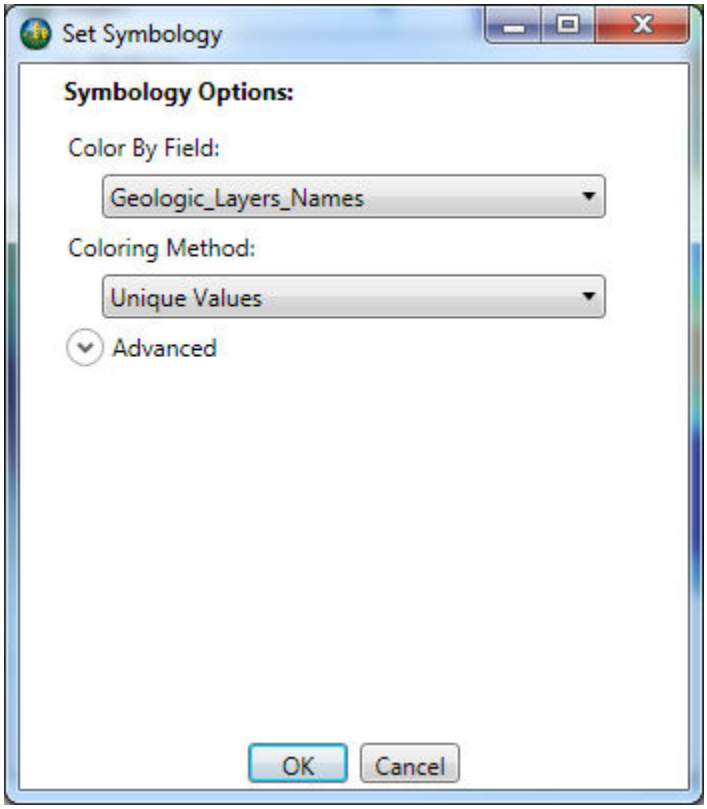
Press OK with the default options and you should see:



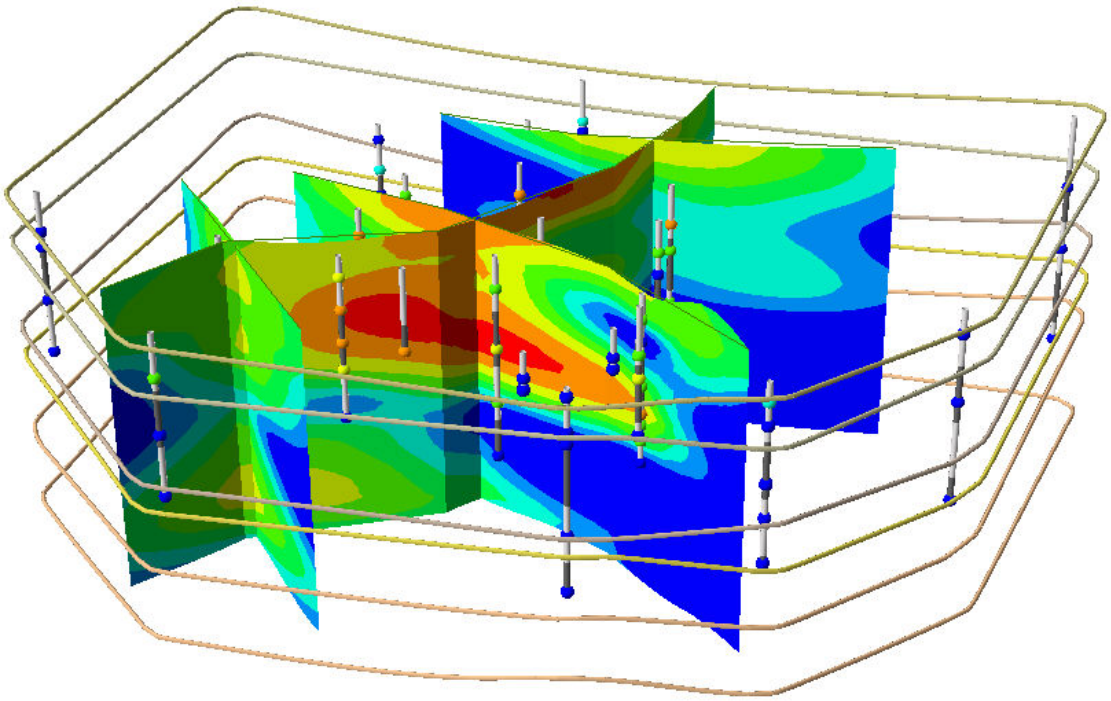
If we choose slightly different options we can get our geologic layer edges:


Edges
Options
Extract Edges from:
Feature Edges
Feature Angle Threshold:
60.00
Remove Shared Edges
Output File: ISS_Initial_Soil_Site_...
Display Options
Add Output Layer
Hide Input Layer
Copy Symbology from Input
OK
Cancel

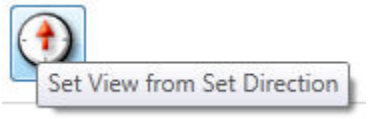
and


Set Symbology
Symbology Options:
Color By Field:
Geologic_Layers_Names
Coloring Method:
Unique Values
Advanced
OK
Cancel

we get:



**Set View from Direction**

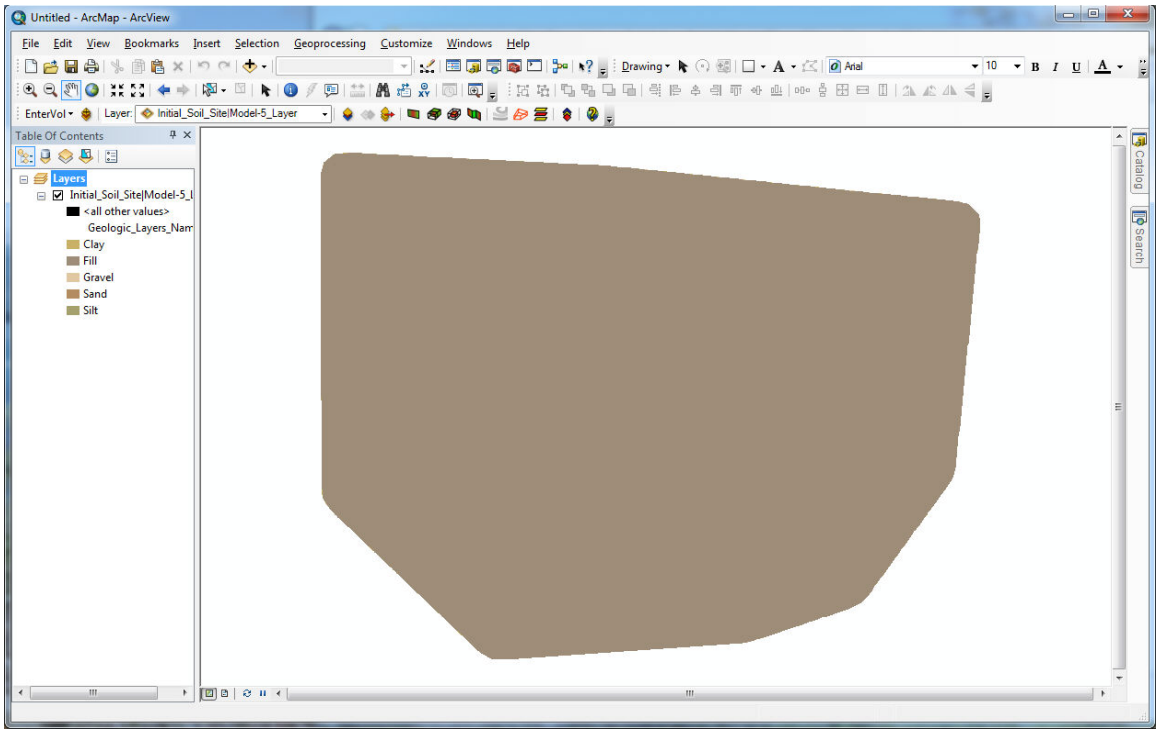



Set View from Direction applies to ArcMap only because it rotates 3D layers making them appear more three dimensional in ArcMap maps.

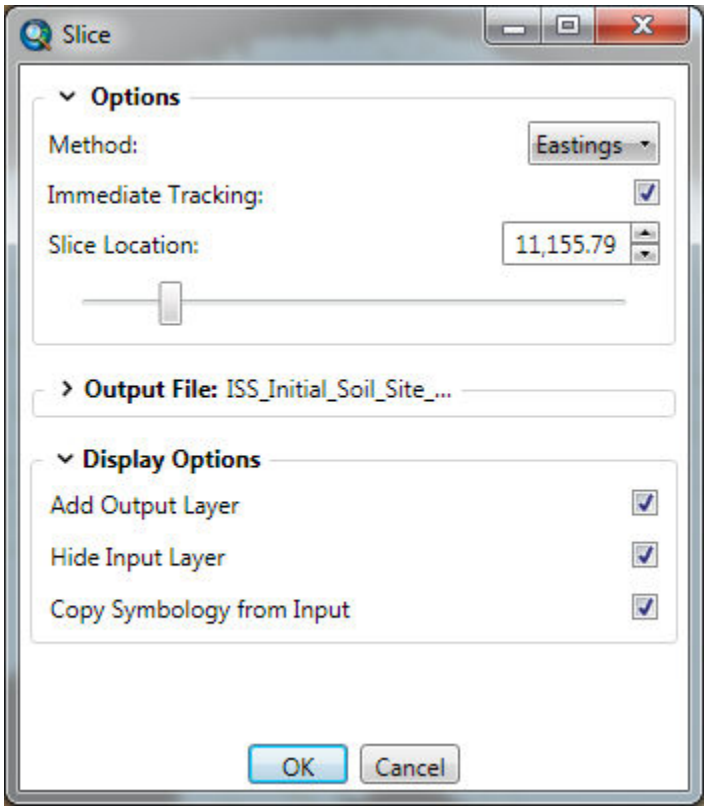
It sets rotations for an individual layer (EFF) for pseudo 3D display in ArcMap. Once this tool is applied, the layer is not in its true position in your map and therefore this should be used only for annotation purposes only because it rotates 3D objects making them appear more three dimensional in ArcMap maps.

To understand this, we will create a slice Easting and view it from the West.

In ArcMap, add the initial_soil_investigation_subsite.evp file with Layer Names as the symbology.

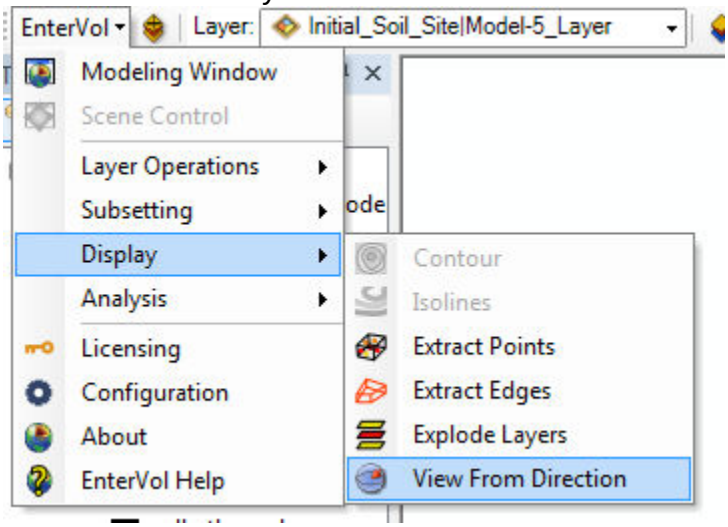



Next we'll create a Easting slice through the centroid (default location)

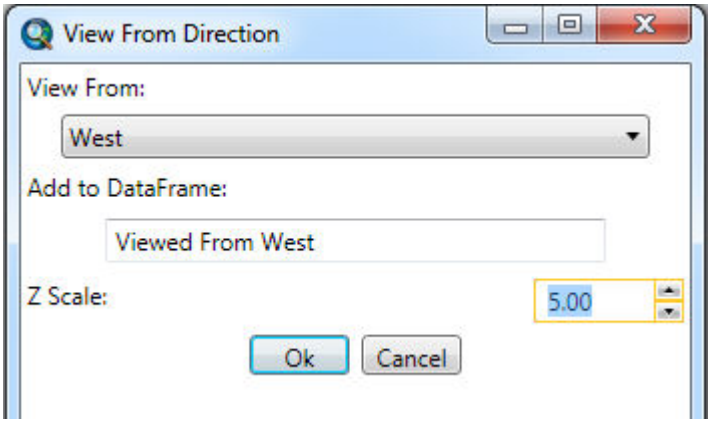


Since this slice has no thickness and is being viewed on edge, our map appears blank (if the slice is the only selected layer).

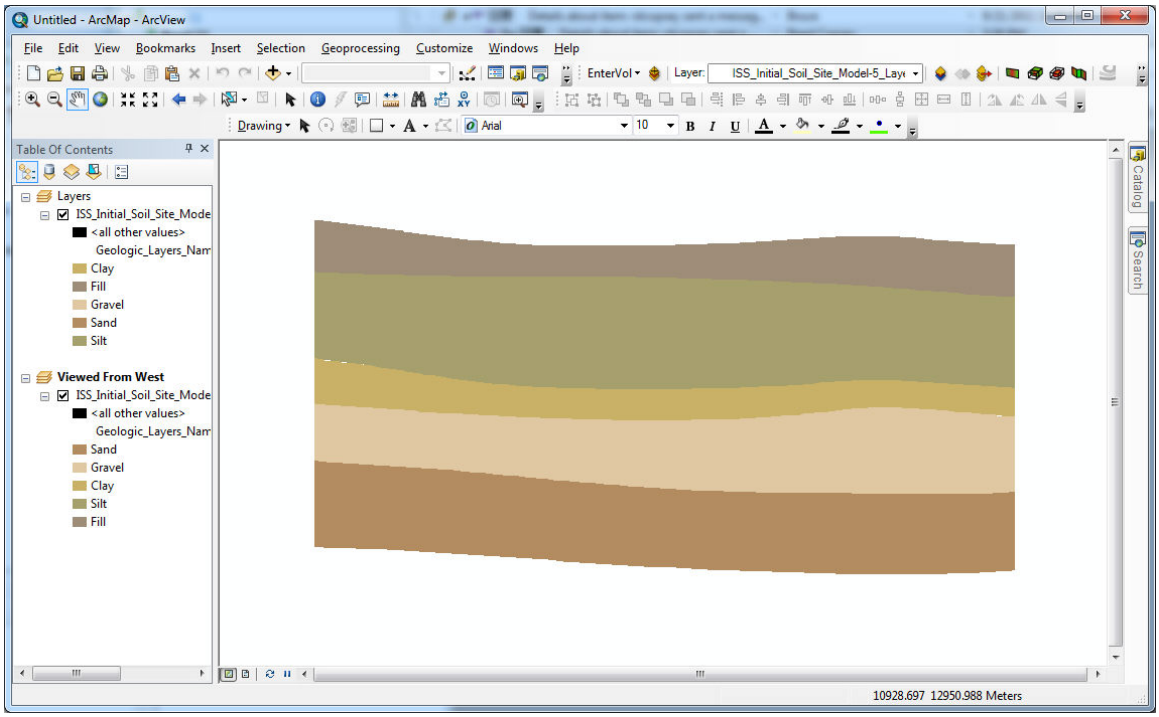
Select the slice layer at click




Choose West with a Z Scale of 5.0




A new data frame is created and will be active by default. This shows the slice in cross-section as below.

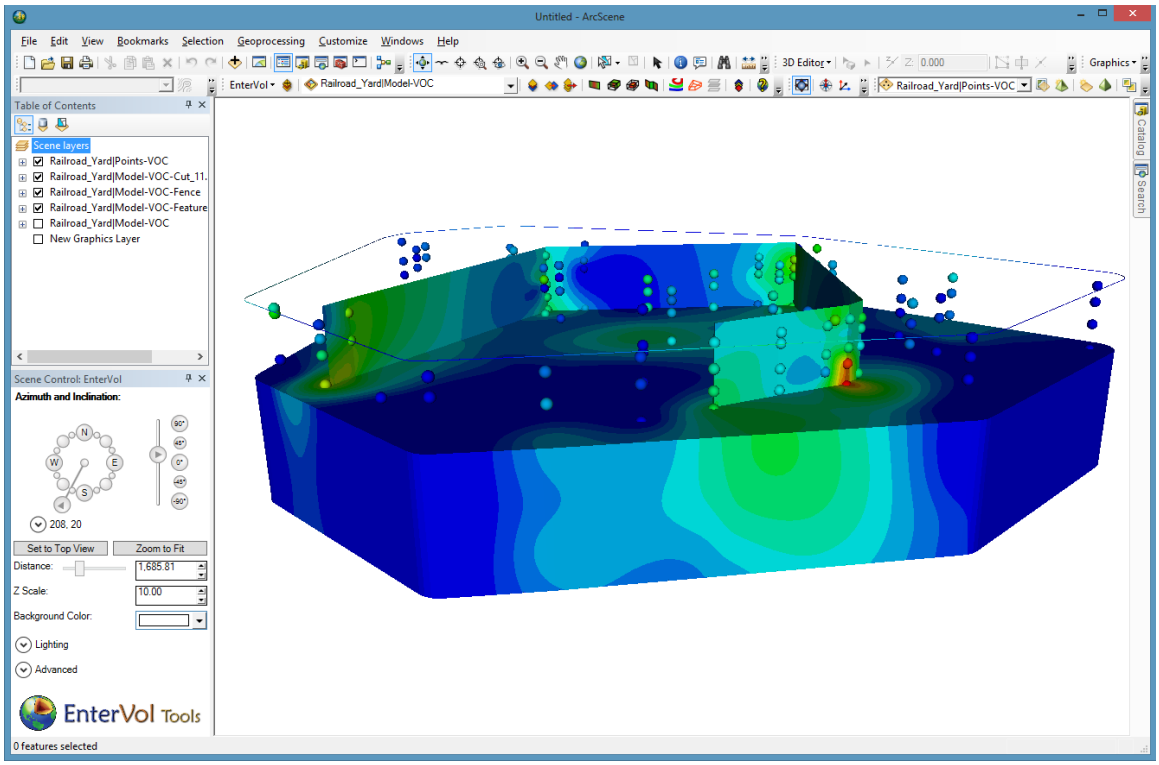



## Exporting EnterVol Models

Since EnterVol models are true volumetric representations and use data associated with their nodes for more accuracy, it is desirable to keep them in EnterVol format.

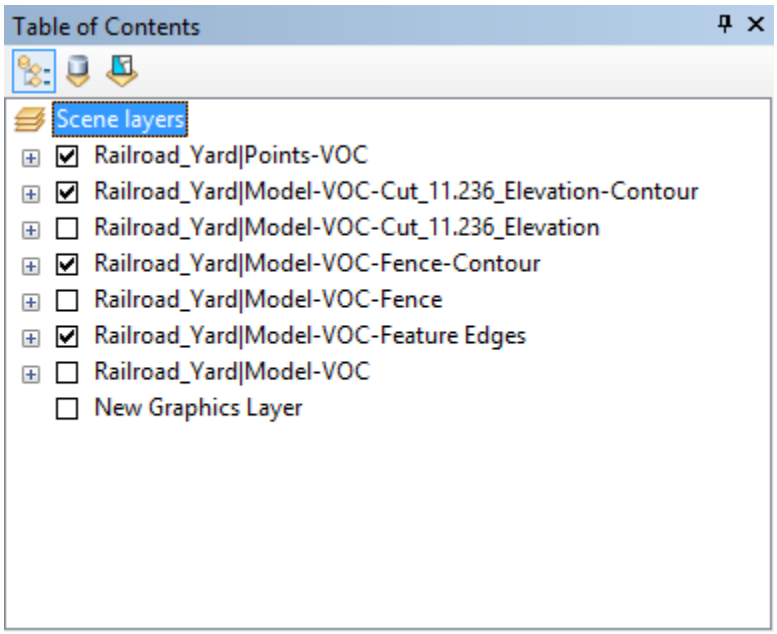
However, we recognize that users will want to export their EnterVol models for use by associates who may not have EnterVol. We make this process simple and painless.

First, make sure that any objects which are displayed with smooth contours, are contoured as in this topic. Even though we display all EnterVol created objects with smooth contours, unless the contour tool is used, the exported feature class will not have smooth contours. This is because the EnterVol objects are internally represented as more complex objects with multiple types of data. For export, we need to simplify them.



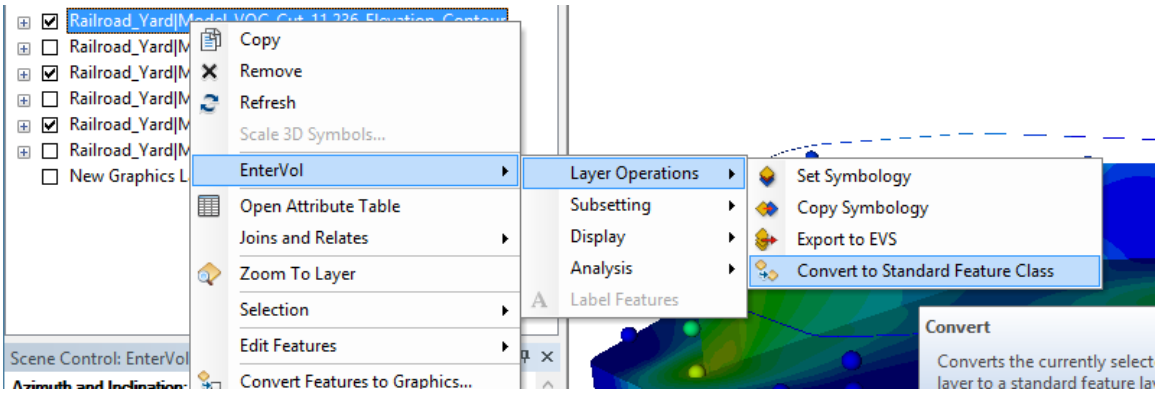

For our model above, we need to contour the cut and fence layers.

Note that we now have contoured layers in our TOC

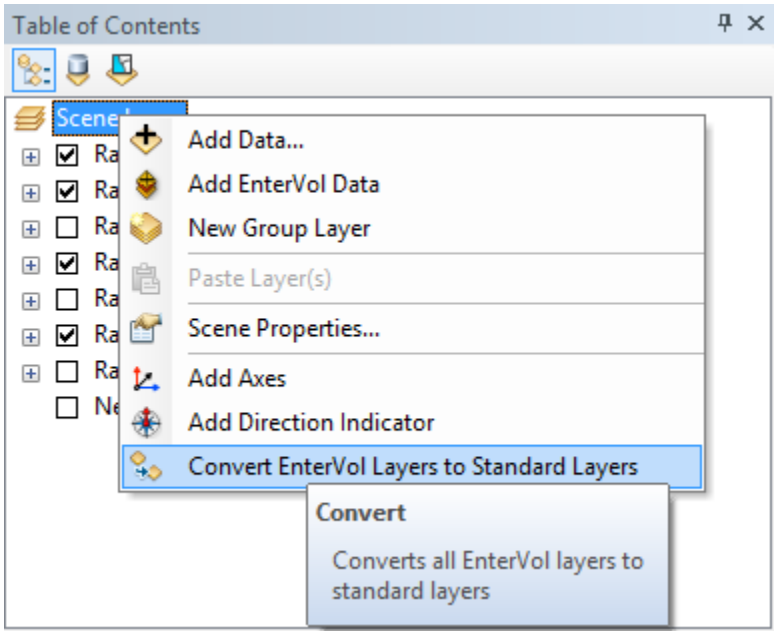



At this point there are two option:

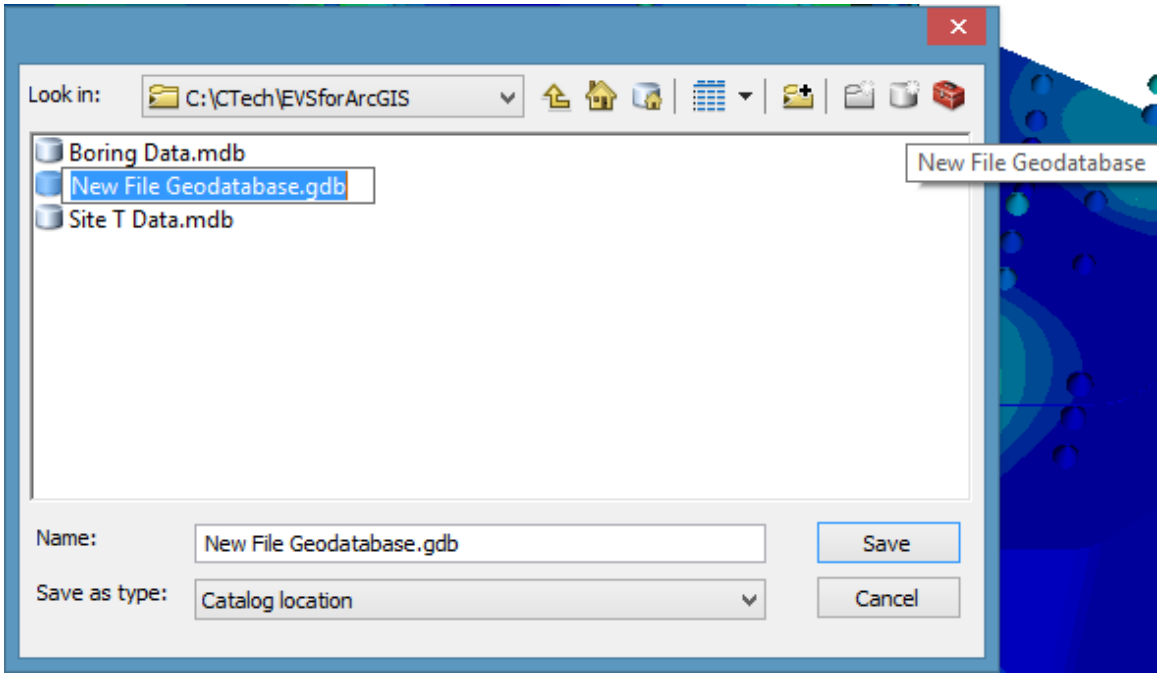
1) You can export individual layers one-at-a-time using the menu shown below:




2) You can Convert all EnterVol Layers to Standard Layers




Then select a geodatabase or create a new one:




Once it completes, all EnterVol layers will be converted to standard (ArcGIS) layers. You can then save your project and deliver the geodatabase to anyone with ArcScene, without their needing to have EnterVol.

You can also select any folder and it outputs each layer as shapefiles in that folder.

# EnterVol Tools

The EnterVol Tools are a licensed product which is included with EnterVol for ArcGIS. The primary licensed products are:

- The EnterVol Tools toolbar includes options to:
  - Label Features
  - EnterVol Raster to Volume
  - Project Features to 2D
  - Open the EnterVol Tools Scene Control window
  - Add Direction Indicator
  - Add Axes (ArcScene) & Add Grid (ArcMap)
- EnterVol Scene Control
  - Provides the ability to quickly set views using Azimuth and Inclination
  - Set Distance to layers, field of view
  - Provides enhanced lighting for true 3D objects
- EnterVol Axes
  - Labeled 3D Axes with auto or user defined extents
  - 3D Views need scale references which show distances along X, Y, & Z. Z axes are often scaled and 3D views distort objects.
- EnterVol Direction Indicator
  - 3D Arrow or Compass Rose
  - Auto or user defined extents and parameters

Without a license some products such as the Scene Control, can still be used but with limited functionality.

## EnterVol Tools

The EnterVol Tools are a licensed product which is included with EnterVol for ArcGIS. The primary licensed products are:

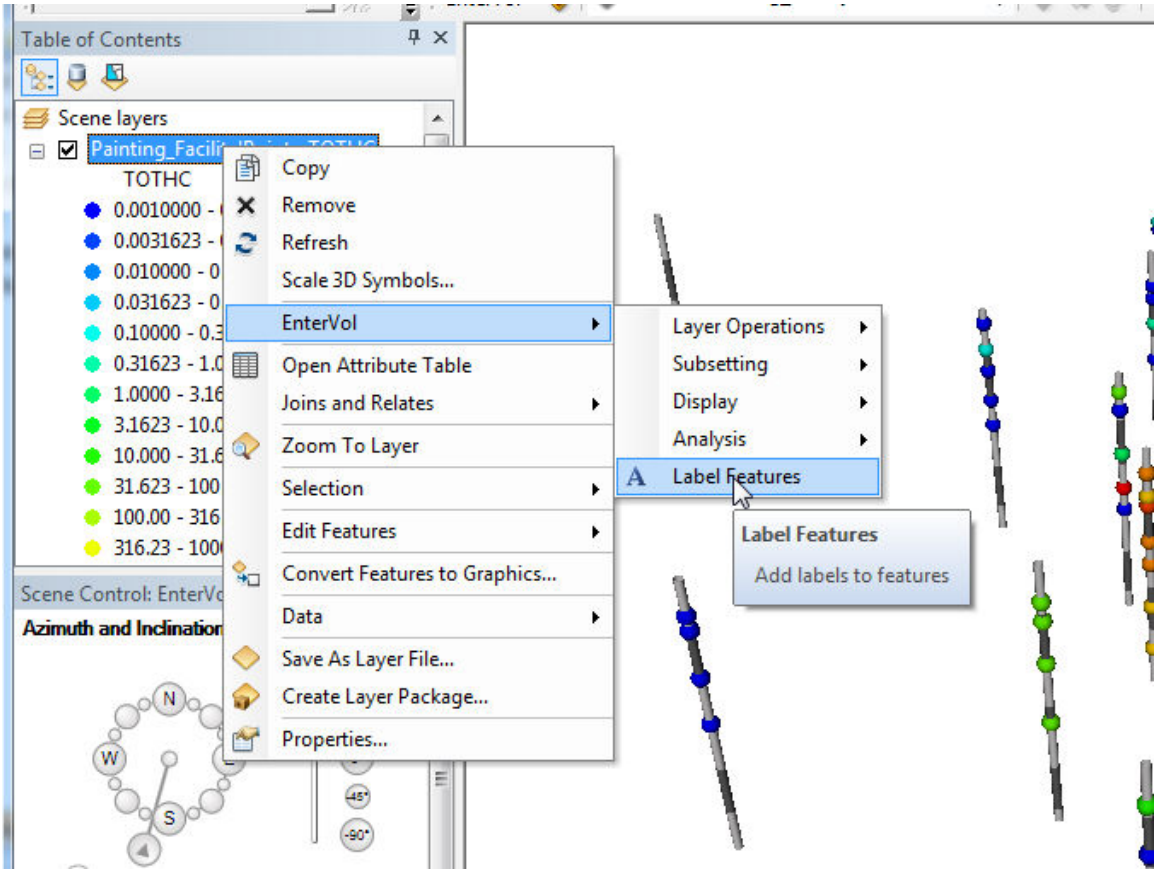
- The EnterVol Tools toolbar includes options to:
  - Label Features
  - EnterVol Raster to Volume
  - Project Features to 2D
  - Open the EnterVol Tools Scene Control window
  - Add Direction Indicator
  - Add Axes (ArcScene) & Add Grid (ArcMap)
- EnterVol Scene Control
  - Provides the ability to quickly set views using Azimuth and Inclination
  - Set Distance to layers, field of view
  - Provides enhanced lighting for true 3D objects
- EnterVol Axes
  - Labeled 3D Axes with auto or user defined extents
  - 3D Views need scale references which show distances along X, Y, & Z. Z axes are often scaled and 3D views distort objects.
- EnterVol Direction Indicator

- 3D Arrow or Compass Rose
- Auto or user defined extents and parameters

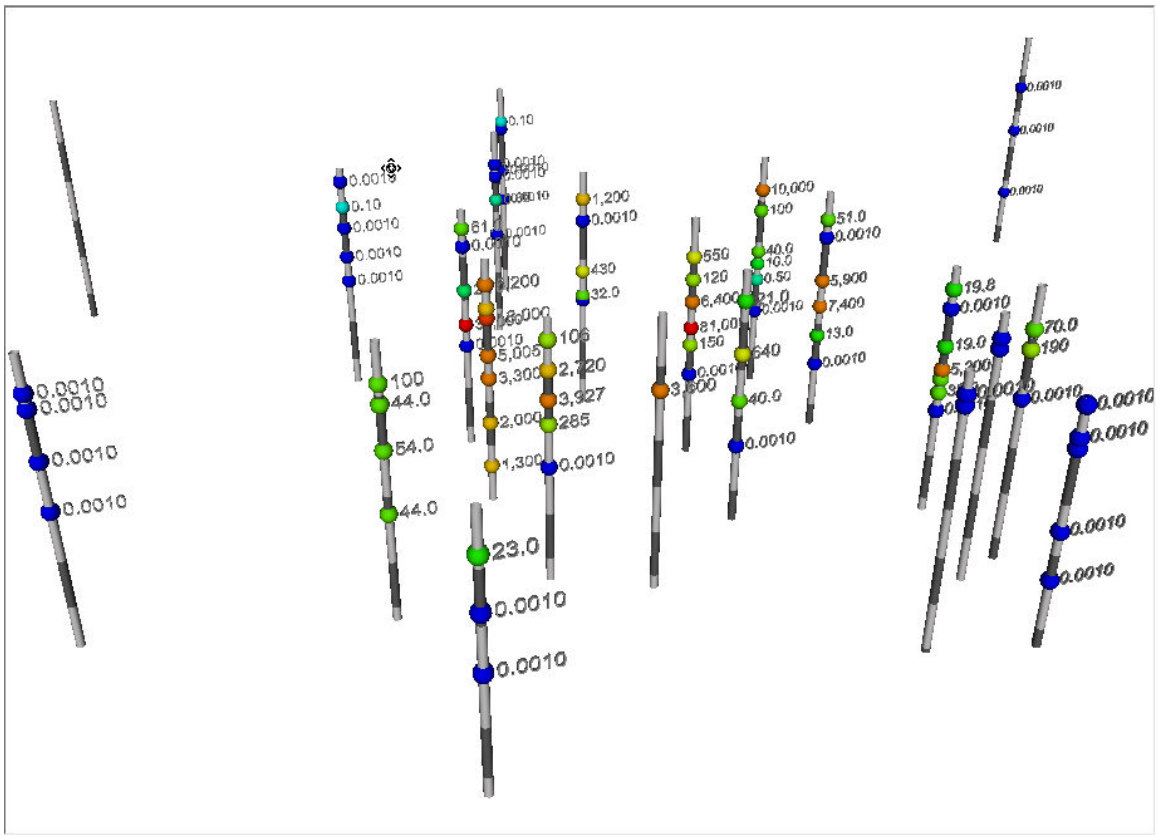
Without a license some products such as the Scene Control, can still be used but with limited functionality.

## Label Features

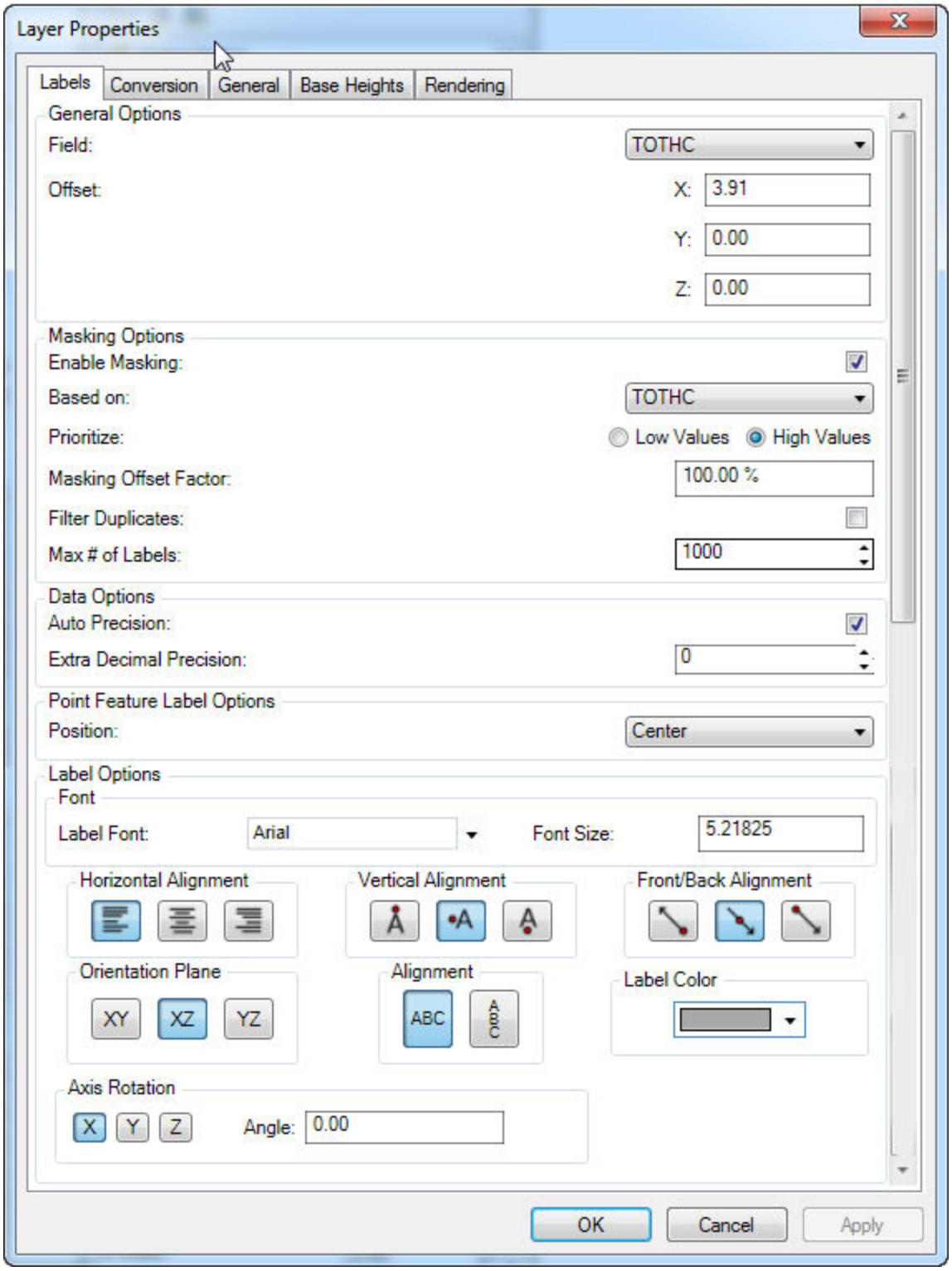
EnterVol Tools includes the ability to place 3D labels on any feature class. Consider the example below




The default settings place 3D labels with intelligent precision on every object in the feature class.



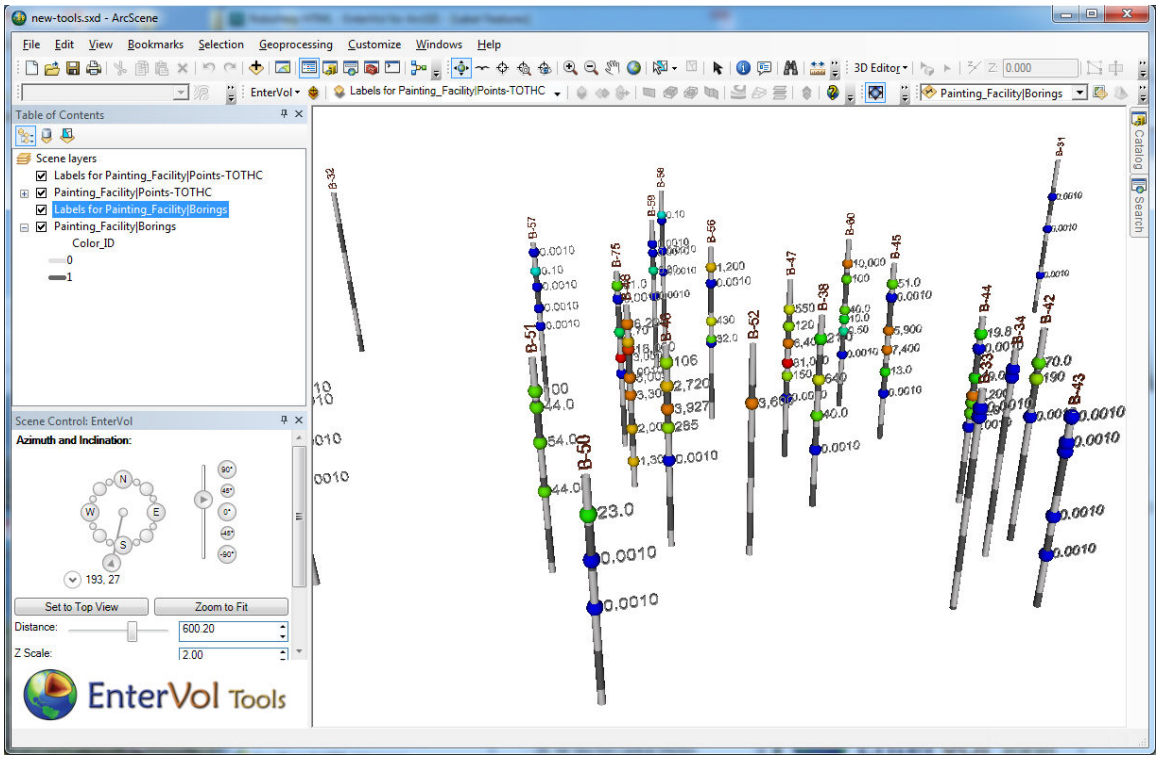

If you right click on the Labels and choose Properties, you can edit the settings:



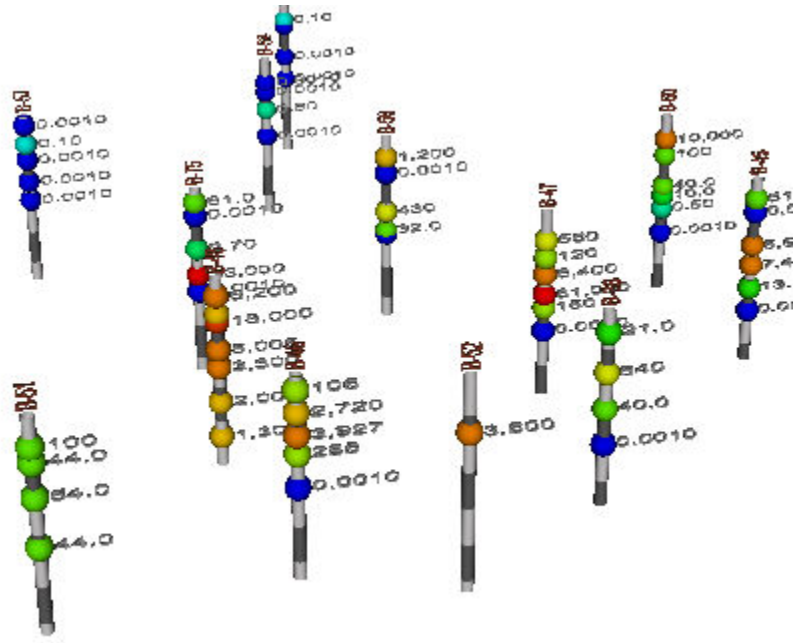

- **Offset**: determines the positioning of each label with respect to its object

    - The default for point samples is about 1/2 of a sphere radius in (+) X
- **Masking Options**
    - *Enable Masking* turns on the ability to cull some labels
    - Masking can be *Based on* any attribute
    - By default, masking is *Prioritized* to favor High Values
    - The *Masking Offset Factor* determines how close labels can be to their neighbors before being masked
    - The *Max # of Labels* prevents accidentally creating huge numbers of labels on dense datasets. Remember that each label is a complex 3D object, and takes time to create and render.
- **Data Options**
    - ***Auto Precision* provides intelligent display of numbers based on the number of significant figures.**
    - ***Extra Decimal Precision* adds additional decimal places**
- **Point Feature Label Options (applies to point features only)**
    - **Center positions labels relative to the center of point features**
    - **Radius positions labels relative to the outside surface of the sphere for point features**
- **FONT options include**
    - **Font & Size (in your project units)**
    - **Alignment options**
    - **Axis of Rotation and Angle**

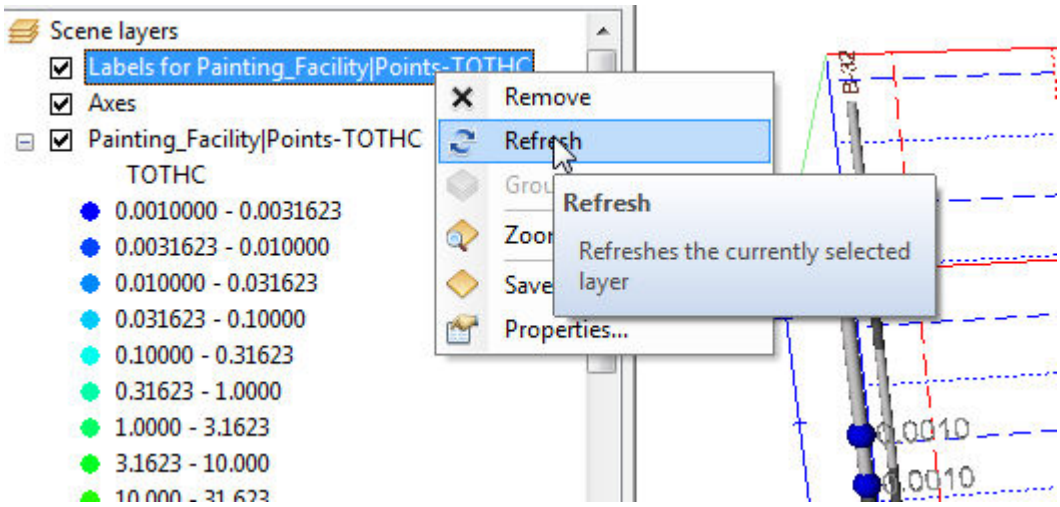
The default labeling for borings is vertical orientation and aligned with the top of each boring. The boring labels below have default options except that the color was changed to brown to make the boring labels more distinct from the Points labels.



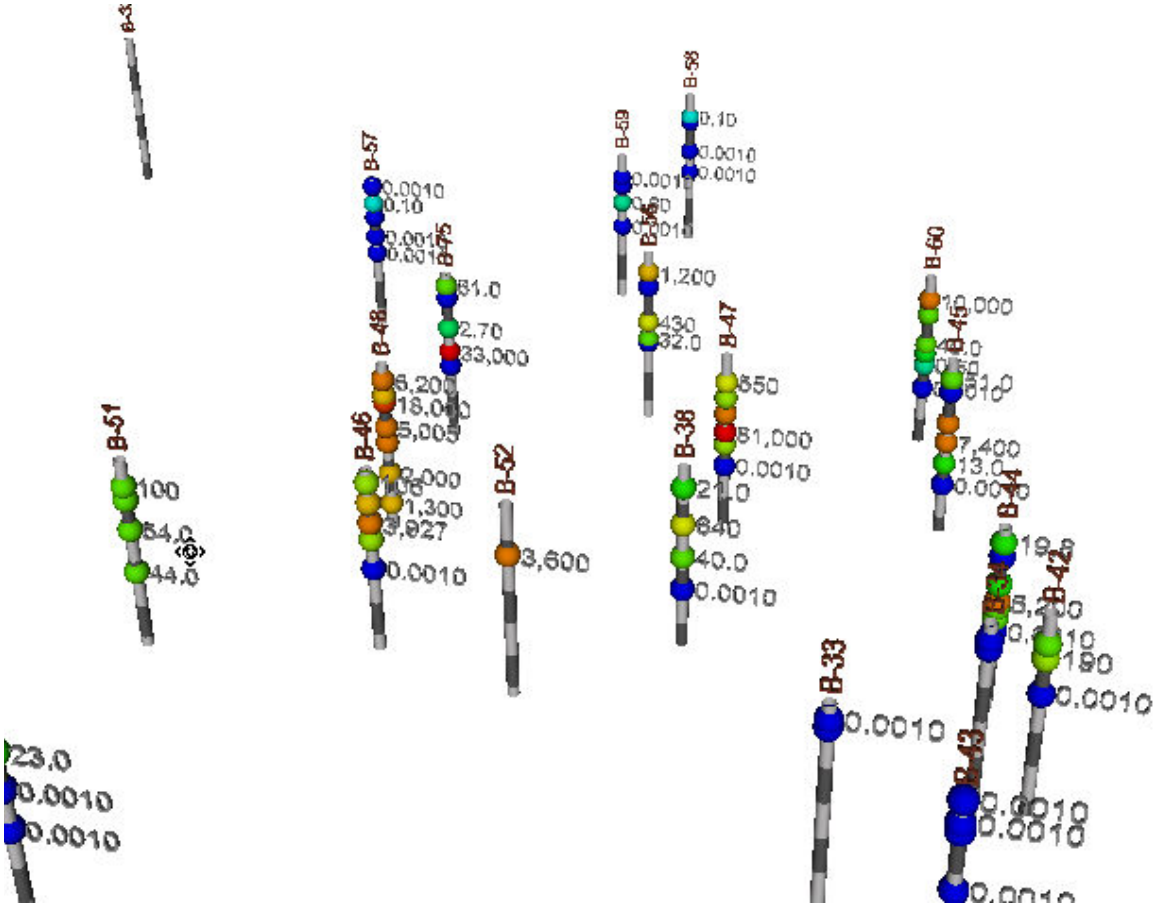

**IMPORTANT NOTE:** Labels, Axes and Direction Indicators are not automatically regenerated when the *Z Scale* is changed in the *EnterVol Scene Controls.* In the example above, if Z Scale is reduced to 1.0, the labels will appear highly compressed (see below)



## Refreshing each label




is required to restore the proper aspect ratio for labels.



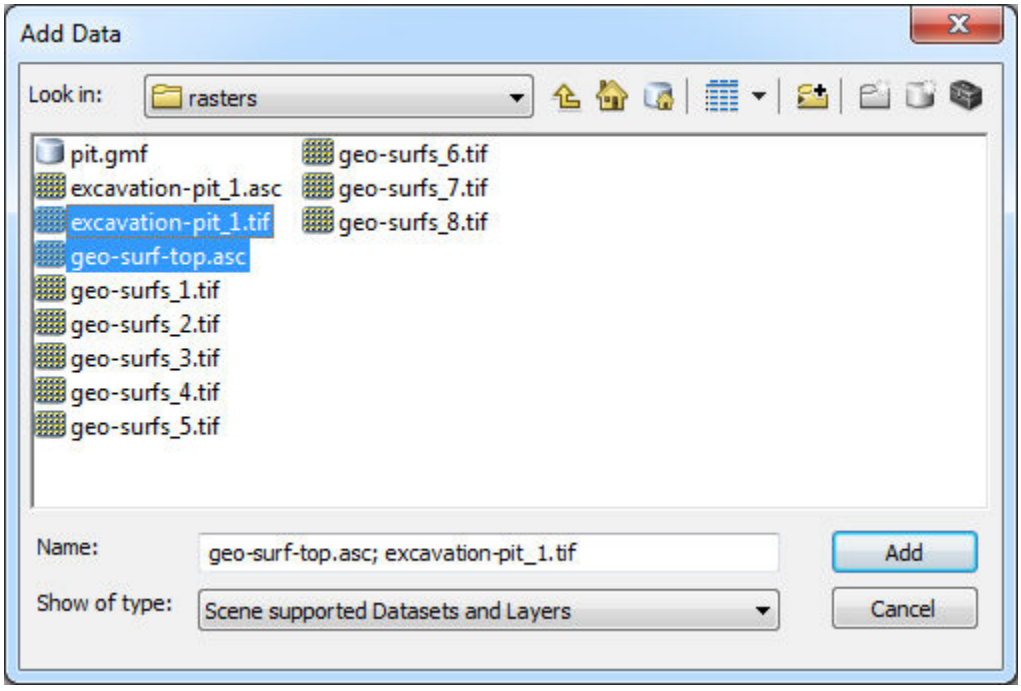
## EnterVol Tools Raster to Volume

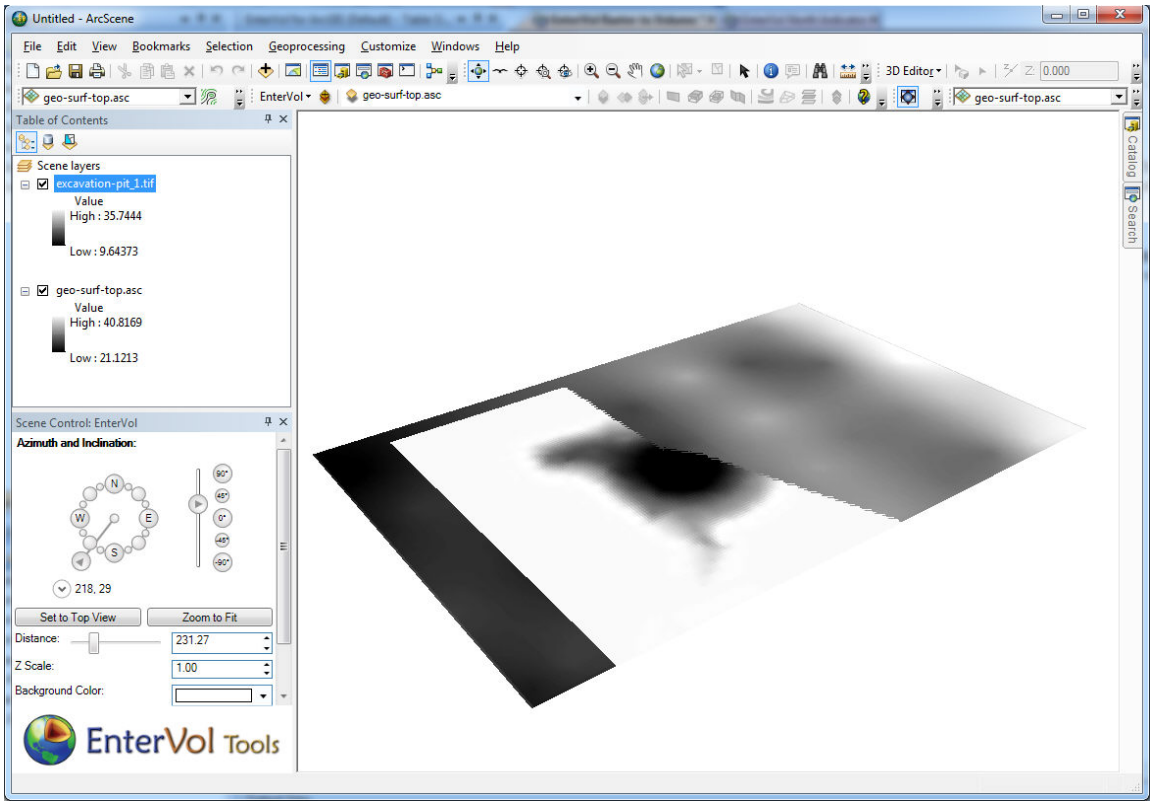
EnterVol Tools includes the ability to create closed multi-patch "volumes" from two rasters. The volume is created from any two rasters in your Scene. Please note that the "volume" created meets ESRI criteria for volumes, but is actually a closed shell which encloses a volumetric region. This is distinctly different from the true volumes created with EnterVol's Geology and GeoStats components.

The process is:

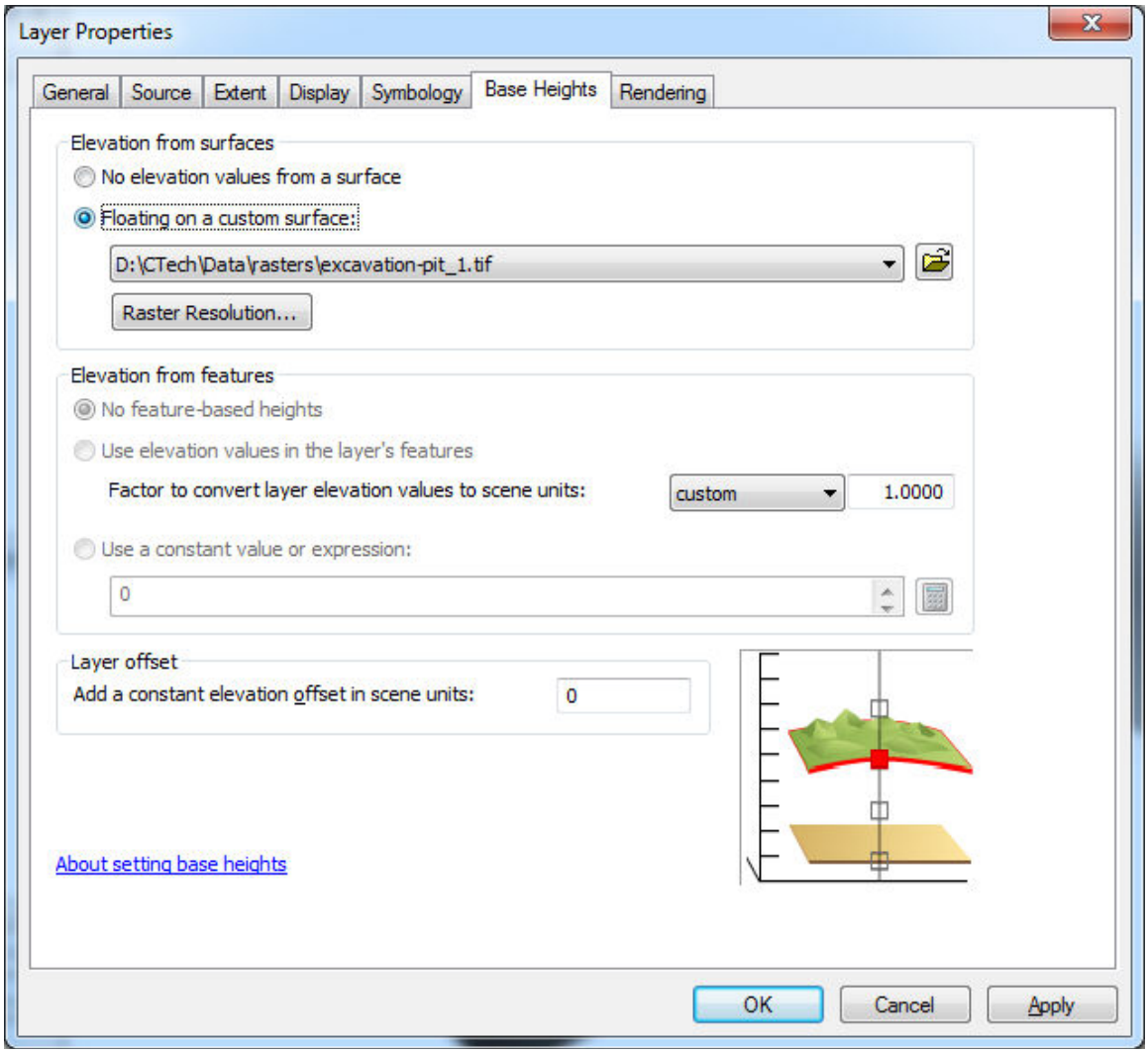
1. Specify which surface is TOP and which is BOTTOM. If the surfaces intersect one another the top/bottom relationship will determine which volume(s) will be created. If the surfaces do not intersect, the common areal extent will determine the extent of the volume created.
2. Specify the output grid resolution and interpolation parameters
3. Provide a name for the resultant multi-patch shapefile
4. Create the volume

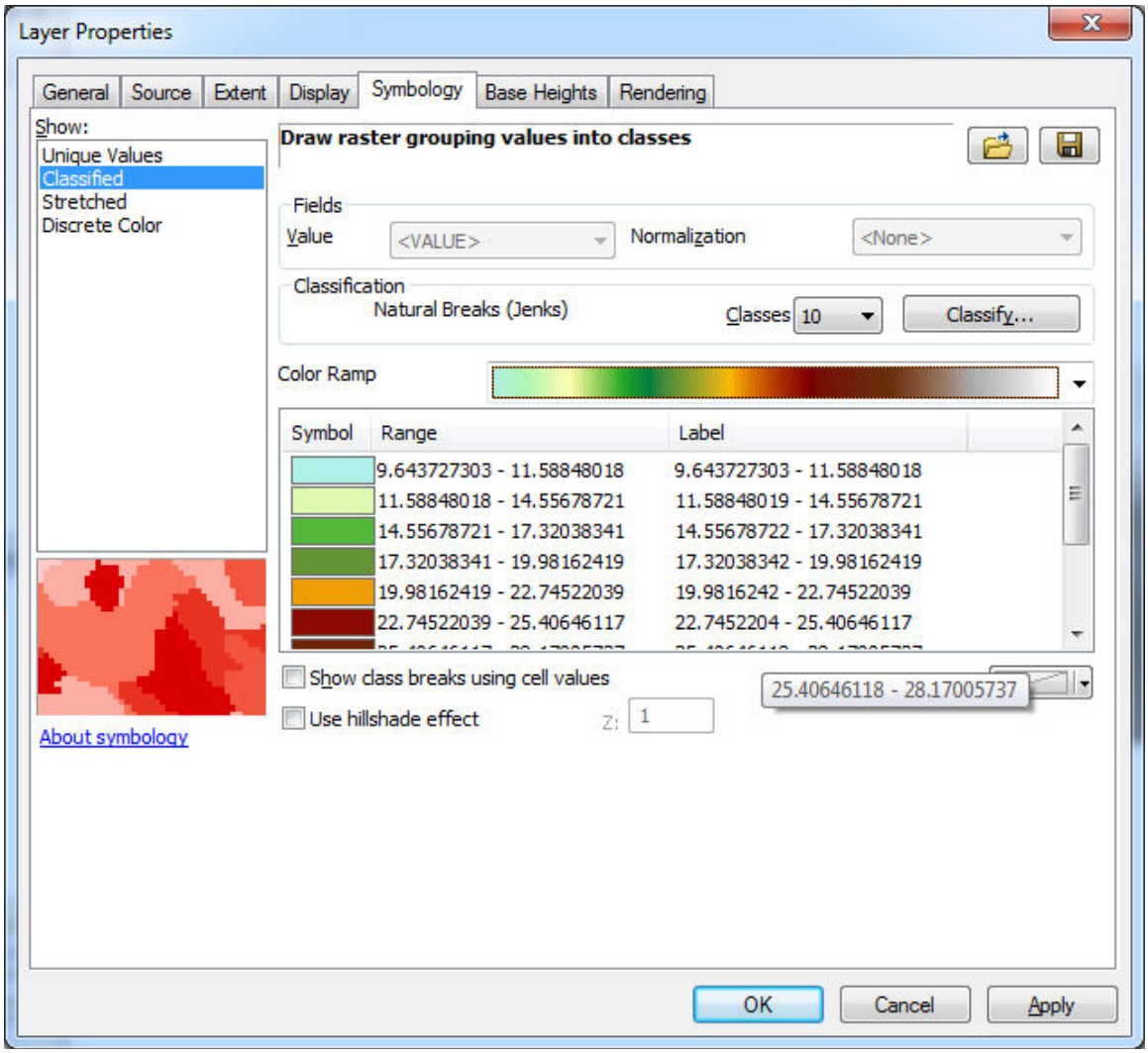
First we'll add two rasters to our scene.



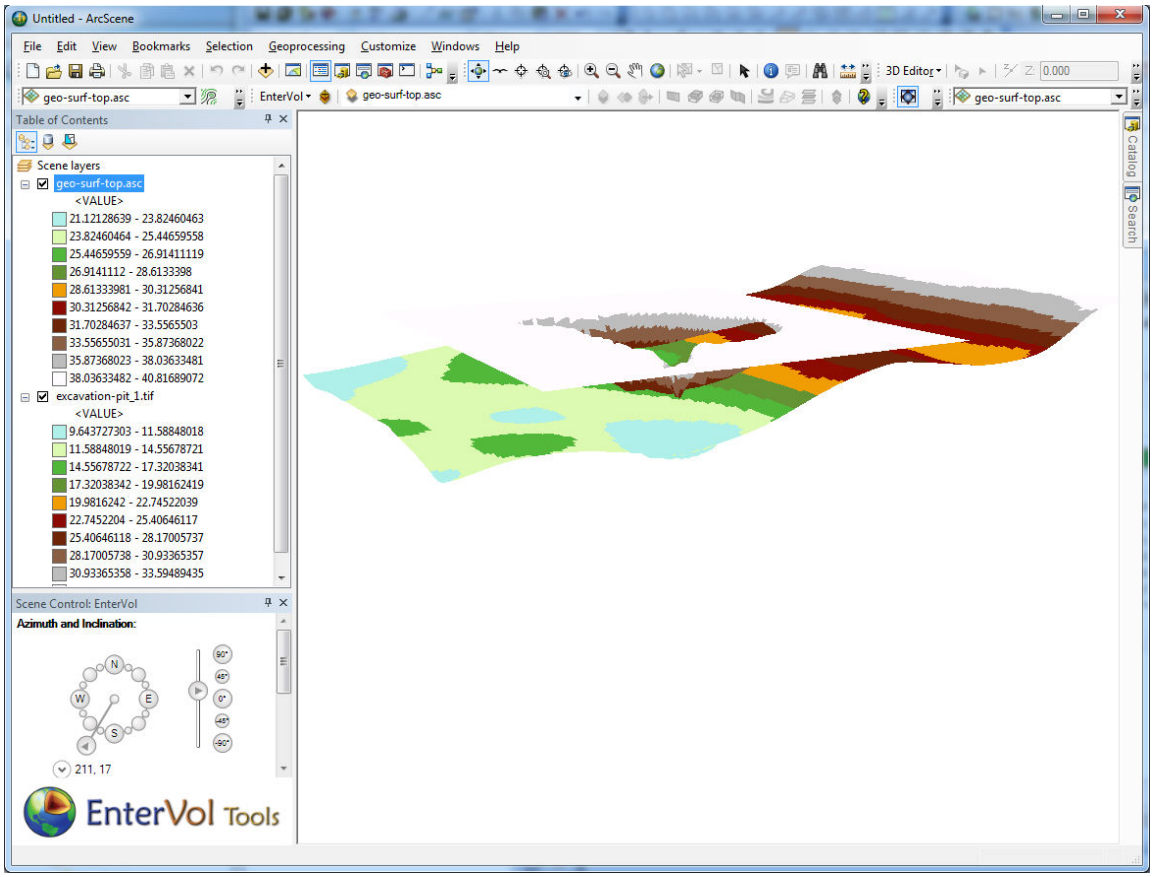



Let's adjust the symbology and base heights to better see each raster:


Layer Properties
General
Source
Extent
Display
Symbology
Base Heights
Rendering
Elevation from surfaces
No elevation values from a surface
Floating on a custom surface:
D:\CTech\Data\rasters\excavation-pit_1.tif
Raster Resolution...
Elevation from features
No feature-based heights
Use elevation values in the layer's features
Factor to convert layer elevation values to scene units:
custom
1.0000
Use a constant value or expression:
0
Layer offset
Add a constant elevation offset in scene units:
0
About setting base heights
OK
Cancel
Apply

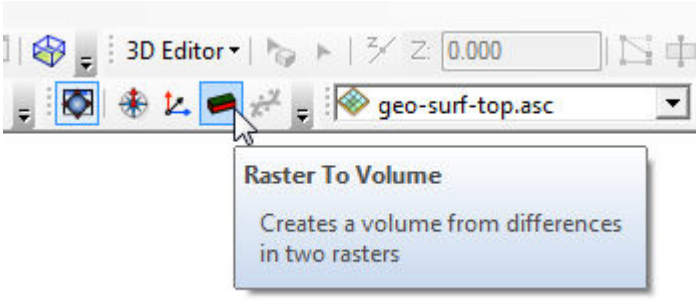



Repeat for the other surface and



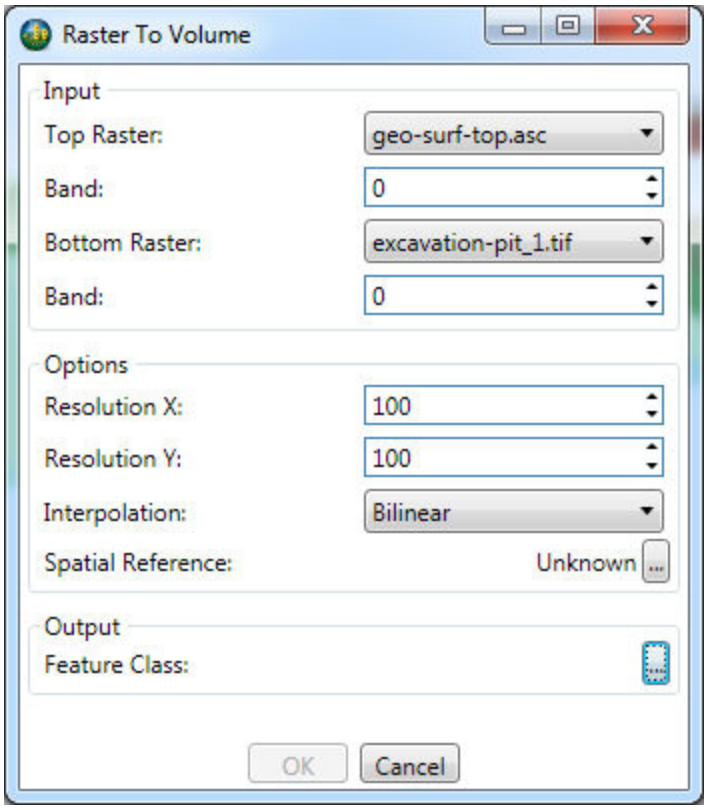

It is clear that the volume between these two surfaces depends on which one we want to be the top. We'll choose the geo_surf_top.asc

Click on the Raster to Volume

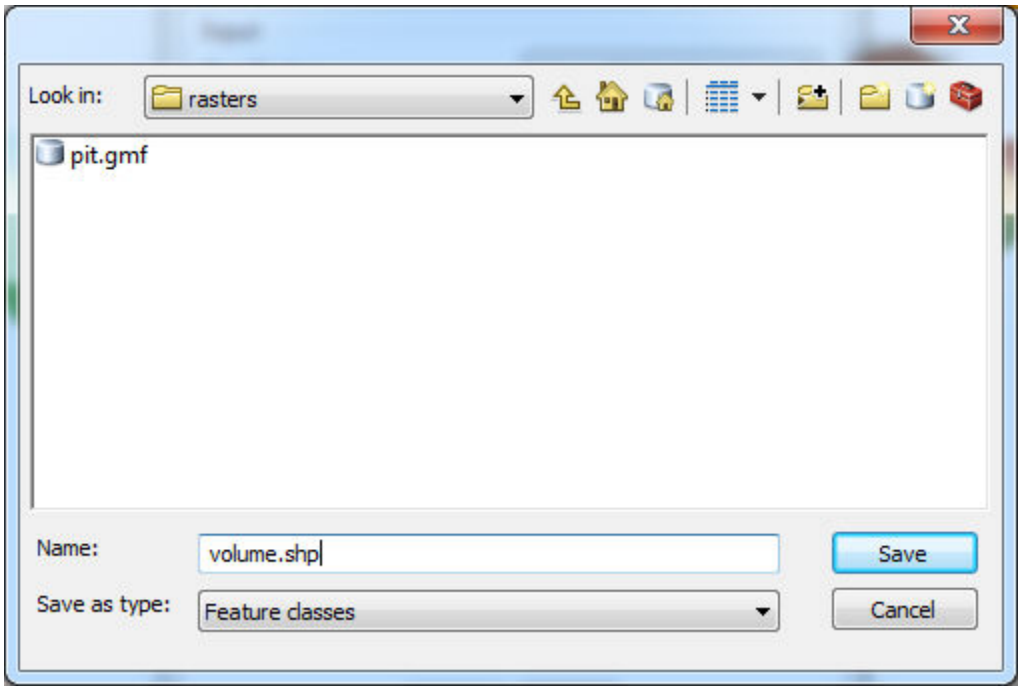


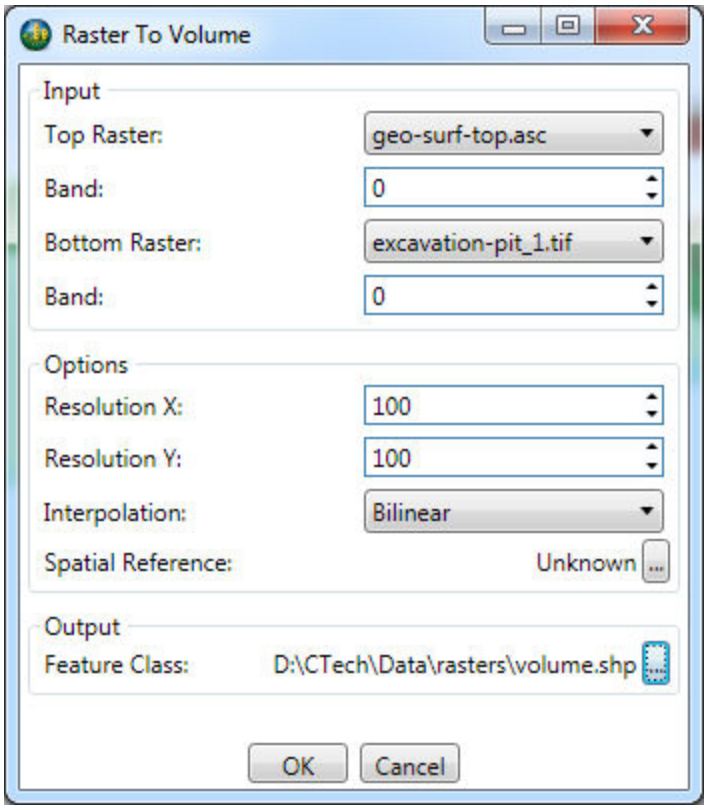

tool:

Set the surfaces and resolution:



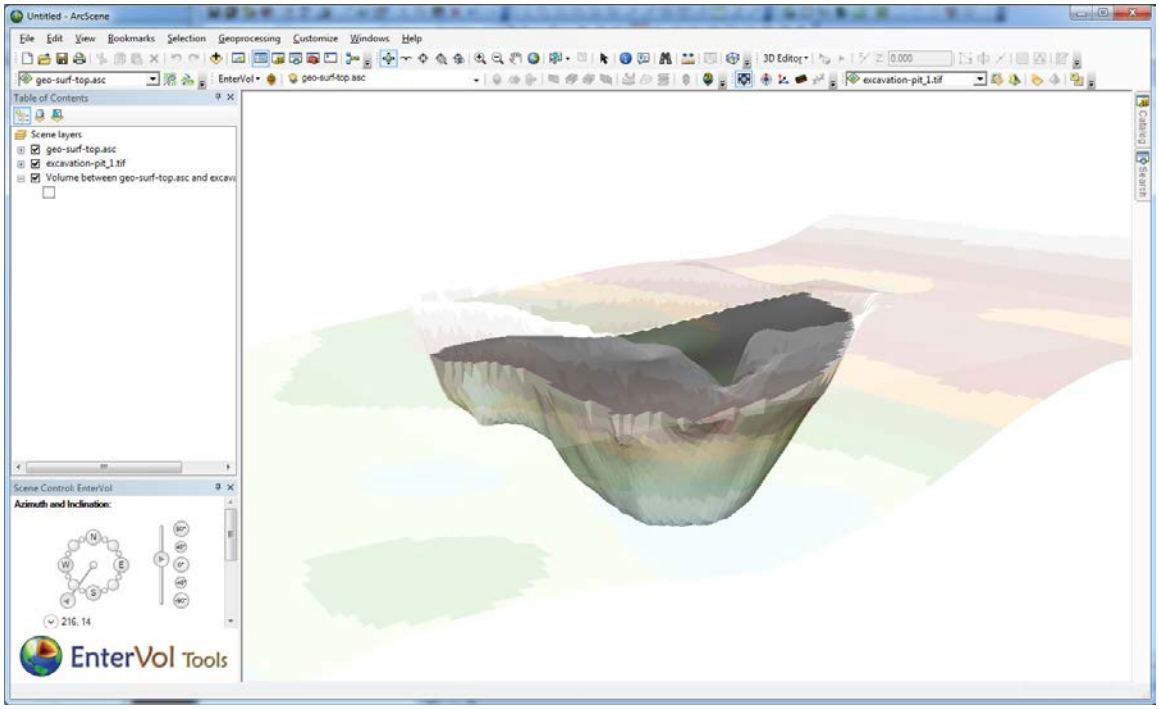

Name the output:






And click OK

I've made both of the rasters transparent so the volume is more obvious:

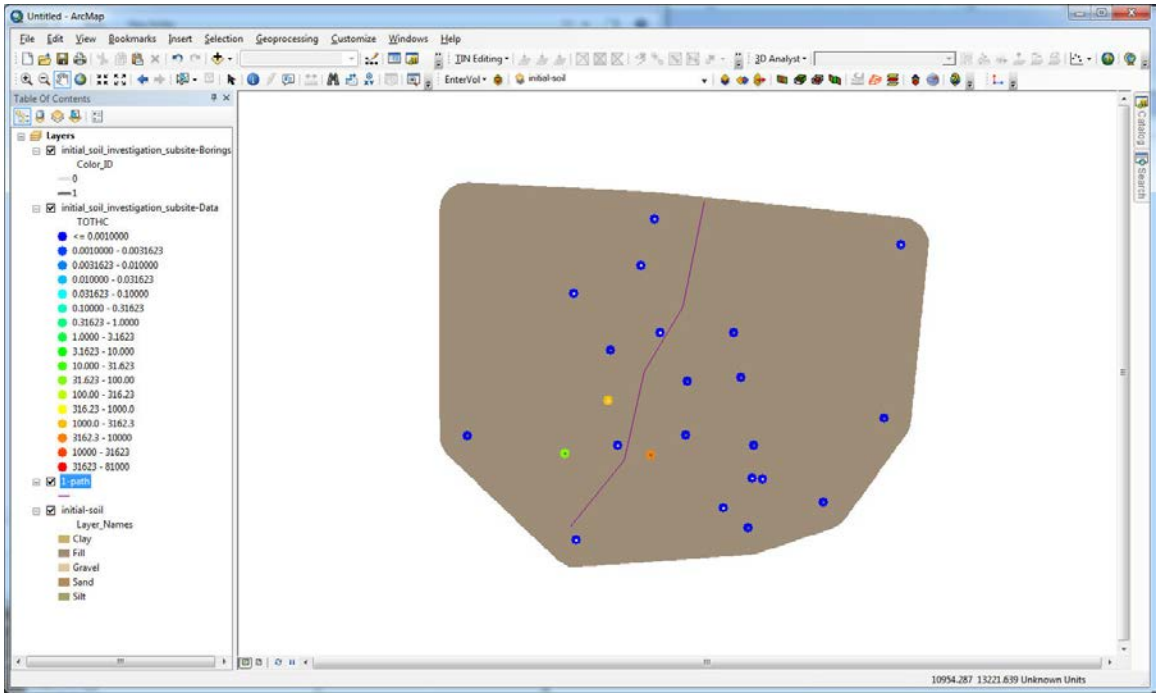


The resultant shapefile is a closed multi-patch

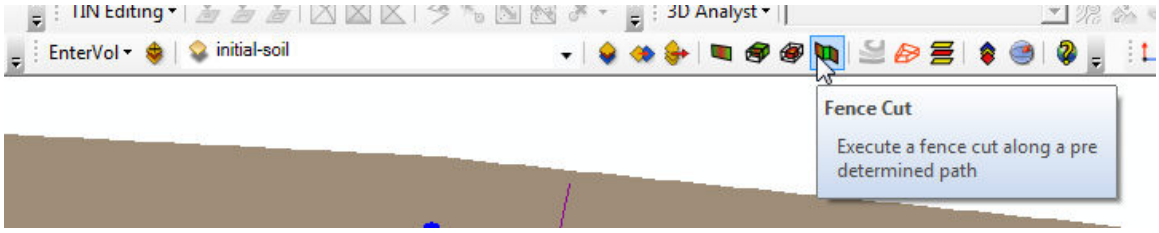
## Project Features to 2D

EnterVol Tools' Project Feature tool allows you to project any line or point feature class onto a 2D fence diagram. To demonstrate this, we'll begin with an ArcMap project with a 3D geologic model, Point contaminant data and borings (from an APDV file), and a shapefile which defines the path of our fence.

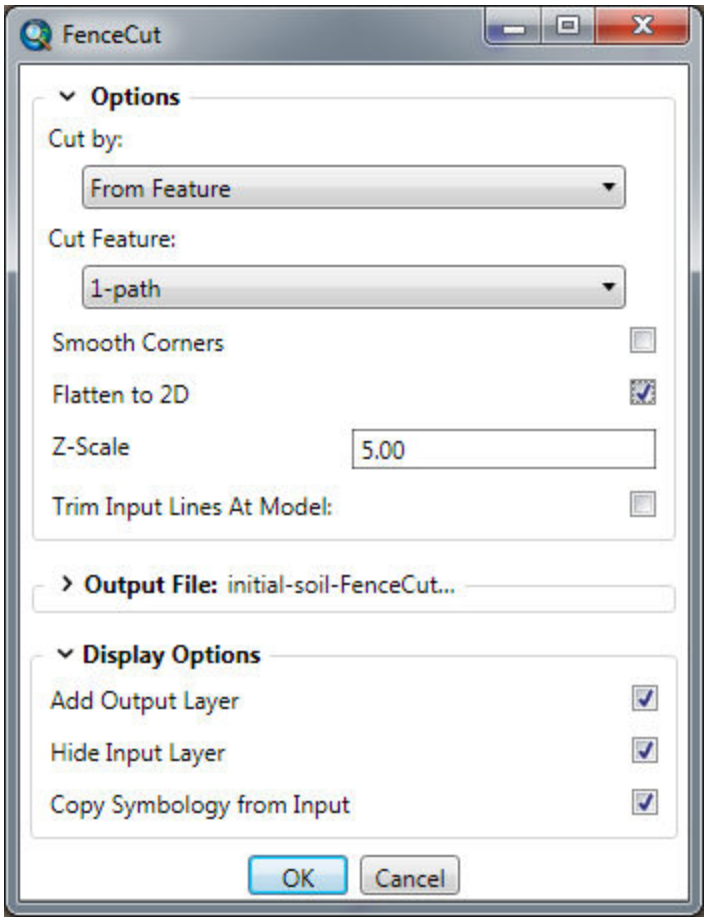
Note: The path used must contain only a single fence cross section.

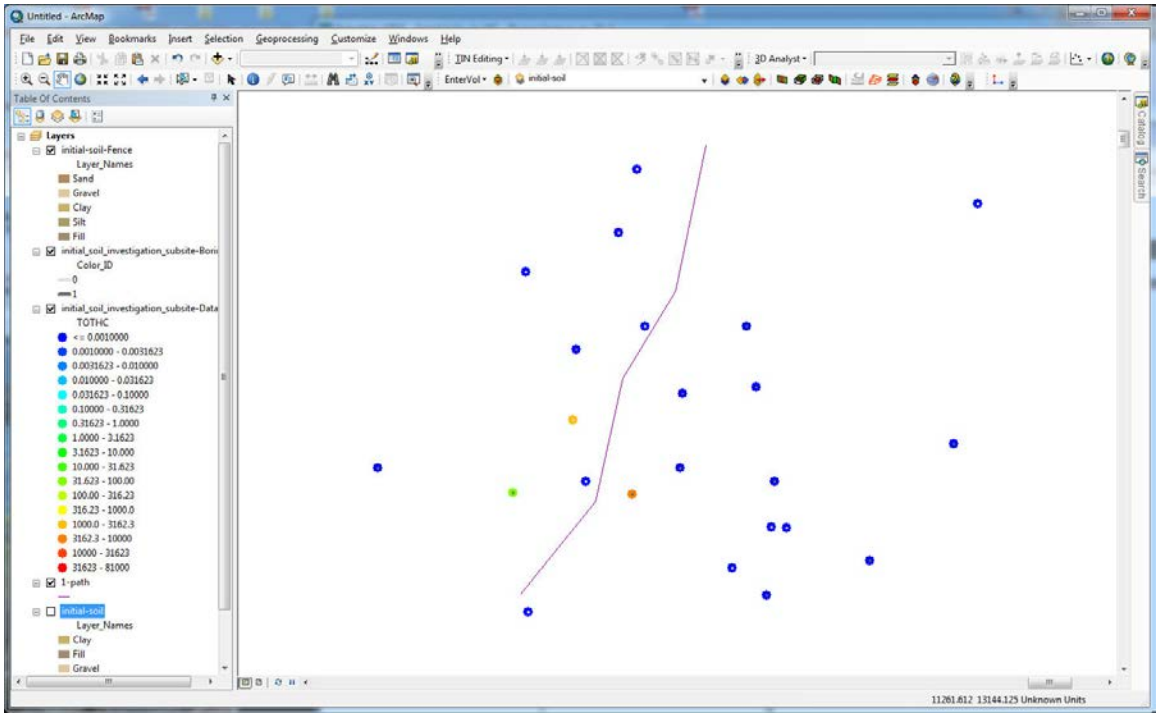



First with our 3D model as the selected EnterVol object, we'll choose the Fence Cut tool to create our fence

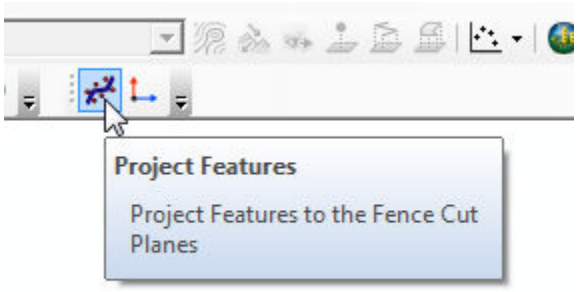



Select a Z Scale and turn on the Flatten to 2D toggle. This will reproject the fence into the X-Y plane making it far more useful in ArcMap.


FenceCut
Options
Cut by:
From Feature
Cut Feature:
1-path
Smooth Corners
Flatten to 2D
Z-Scale
5.00
Trim Input Lines At Model:
Output File: initial-soil-FenceCut...
Display Options
Add Output Layer
Hide Input Layer
Copy Symbology from Input
OK
Cancel


Untitled - ArcMap
File Edit View Bookmarks Insert Selection Geoprocessing Customize Windows Help
Table Of Contents
Layers
initial-soil-Fence
Layer_Names
Sand
Gravel
Clay
Silt
Fill
Color_ID
TOTHC
1-path
initial-soil
Layer_Names
Clay
Fill
Gravel
11261.612 13144.125 Unknown Units

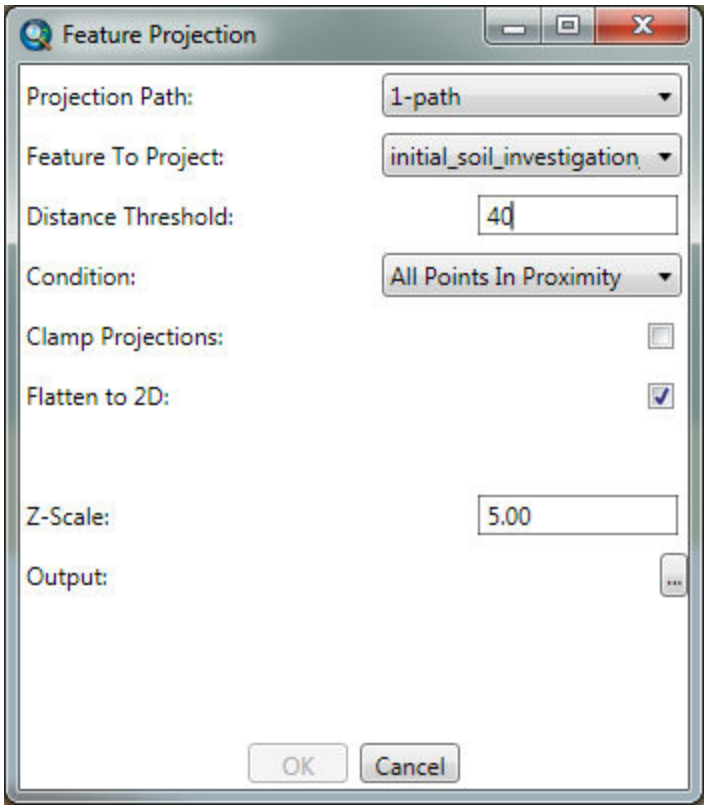
Our 2D fence is created, but is not in the same extents as our current view. We could create a new view, but let's project our borings and samples first. Note that because the Hide Input Layer toggle was ON, our 3D Geologic model is now not visible.



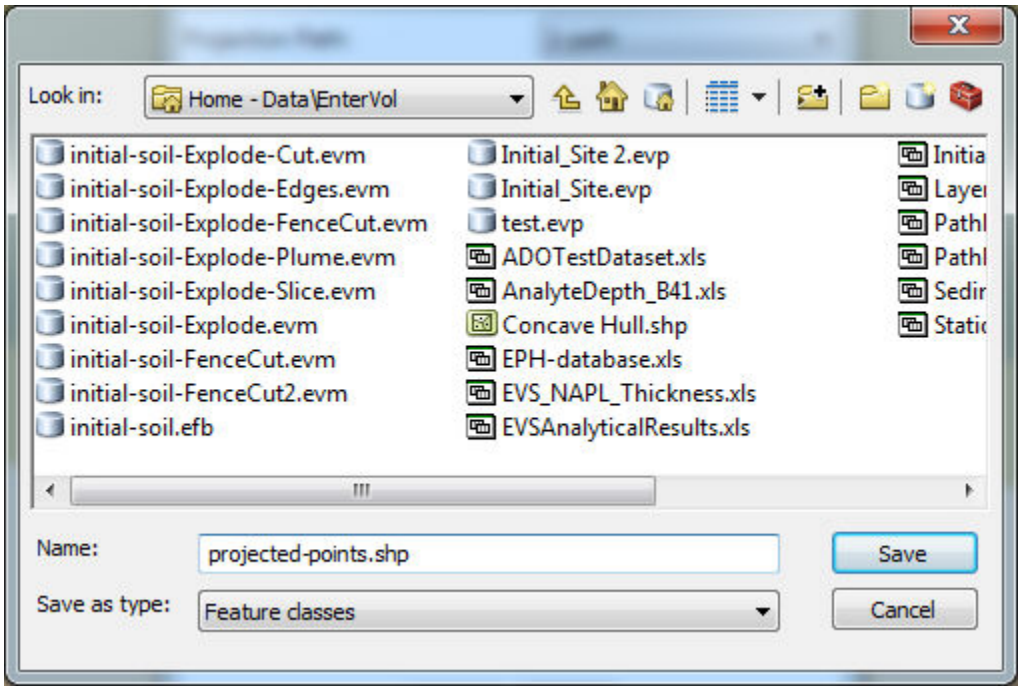

The parameters in the Feature Projection tool are:

- *Projection Path*: Select the line in your scene or view to use for the projection. Normally this is the same line that was used to create a Fence
- *Feature To Project*: Select the feature class to be projected. This can be a point (data points) or line (borings) object
- *Distance Threshold*: This is the distance from the path within which objects will be projected.
- *Condition:* The Condition setting controls which features will be projected.
  - When "Any Points in Proximity" is chosen, a feature will be included in the projected results if any of the points defining the feature are within the Distance Threshold.
  - When "All Points in Proximity" is chosen, every point defining the feature must be within the Distance Threshold for that feature to be included.
- *Clamp Projections*: When OFF, the projection path is extended along the direction determined by either the first or last line segment. When ON, points that would have been off the path are placed at the start or end of the path.
- *Flatten to 2D*: This transforms the projected data onto the X-Y plane, to make the original 3D results more useful in ArcMap.

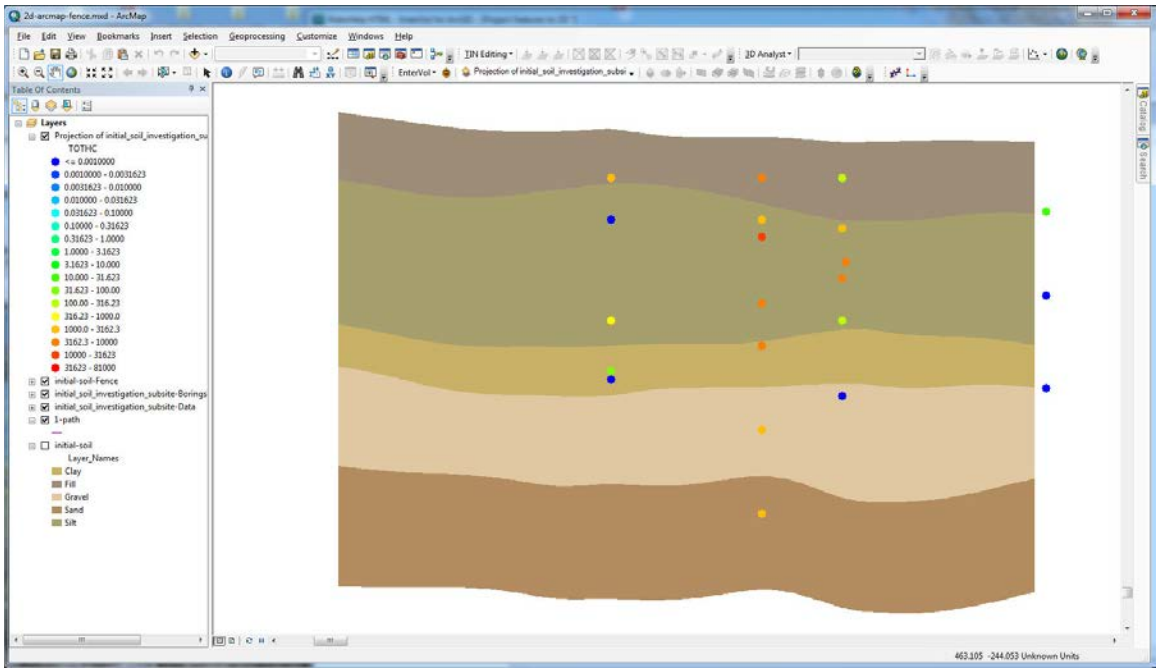
So, when we select the Project Features tool we will want to choose the following options:



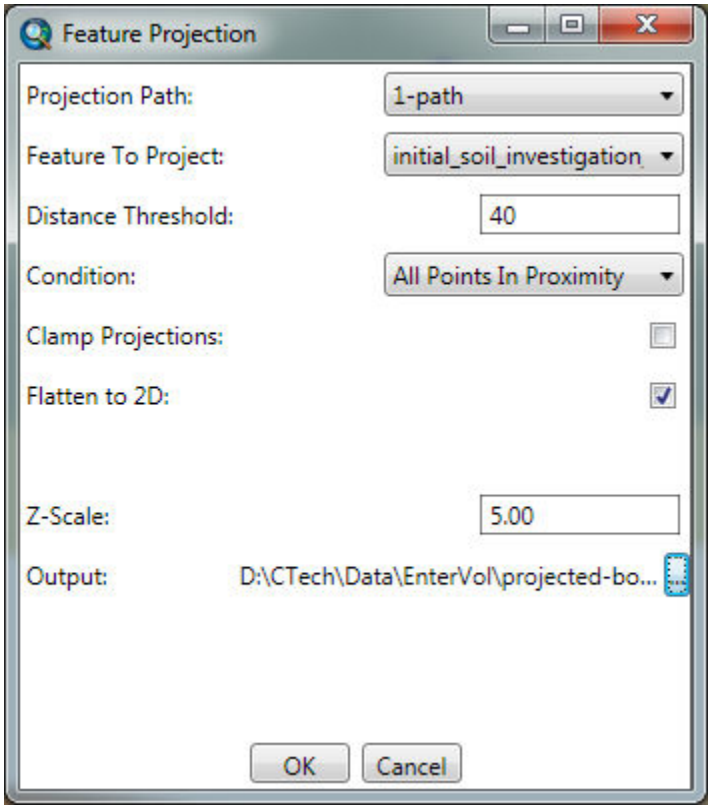

Set the feature class output name:



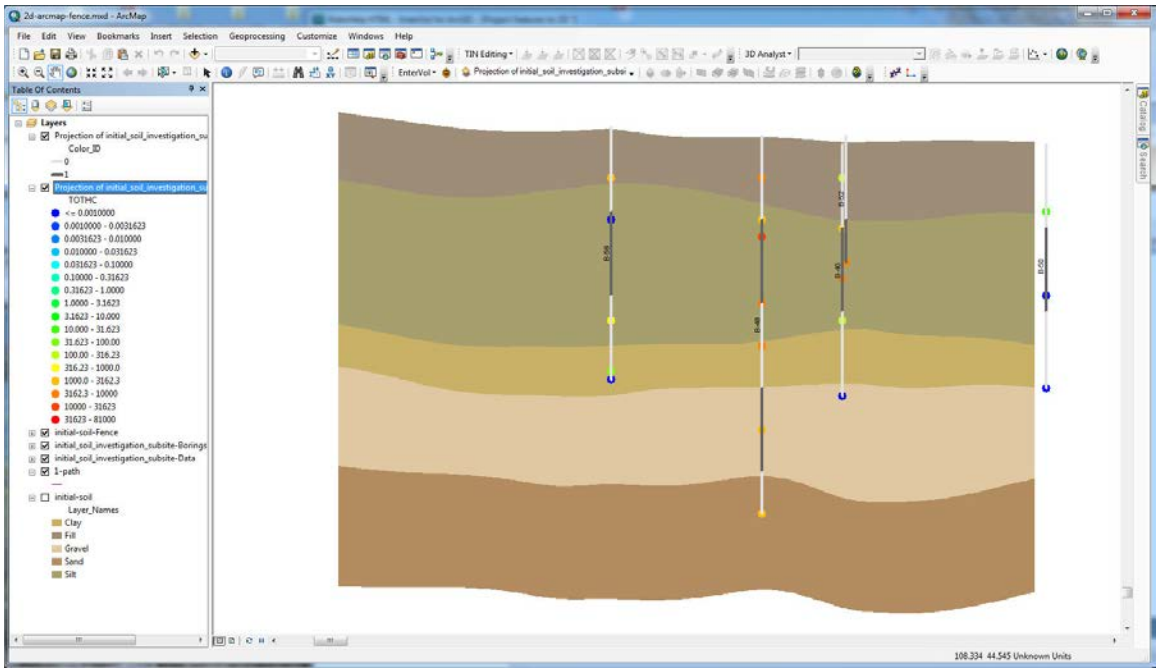

If we zoom to the 2D projected fence we will see:



And if we do the same for the borings



we get:



## Scene Control

EnterVol's Scene Control provide a powerful new way to manipulate and monitor your three-dimensional view in ArcScene. The controls not only allow you to set your view, but provides a display of what your view represents when you manipulate the scene with your mouse.

The EnterVol Scene Control is a licensed product which is included with EnterVol for ArcGIS. Without a license, it can be used as a display of view settings, but cannot be used to set the view.

As a reference for the parameters below, a normal map view is an *Azimuth* of 180 degrees (South) and an *Inclination* of 90 degrees.

ESRI defines your view using four basic parameters which are shown in the image at the bottom of this page:

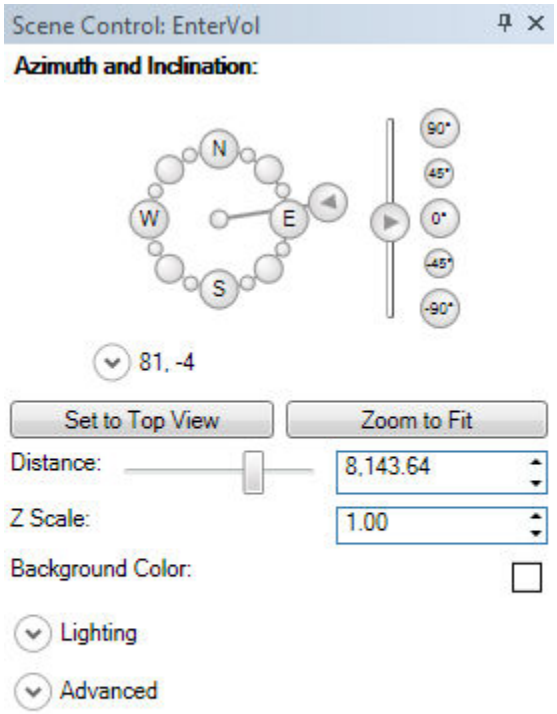
1. The Observer Position (x-y-z)
2. The Target Position (x-y-z)
3. Roll angle in degrees: This is equivalent to the roll of an aircraft or tilting the camera. Please note that when doing normal mouse manipulations, roll is always zero.

4. Viewfield angle in degrees : This is the field of view of the camera lens. Larger values result in "fish eye" effects, smaller values provide an effect like a telephoto lens.

From the 4 inputs above, the *Distance to Target* and *Pitch* (inclination) are determined.

However, please note that ESRI limits inclinations to not be equal to 90 degrees, so Top Views will set Inclination to 89.90 degrees. Even choosing Orthographic (2D view) in ArcScene's View Setting window will not set the observer and target x-y coordinates identical (required to achieve a true top view).

The basic EnterVol Scene Control is shown below.



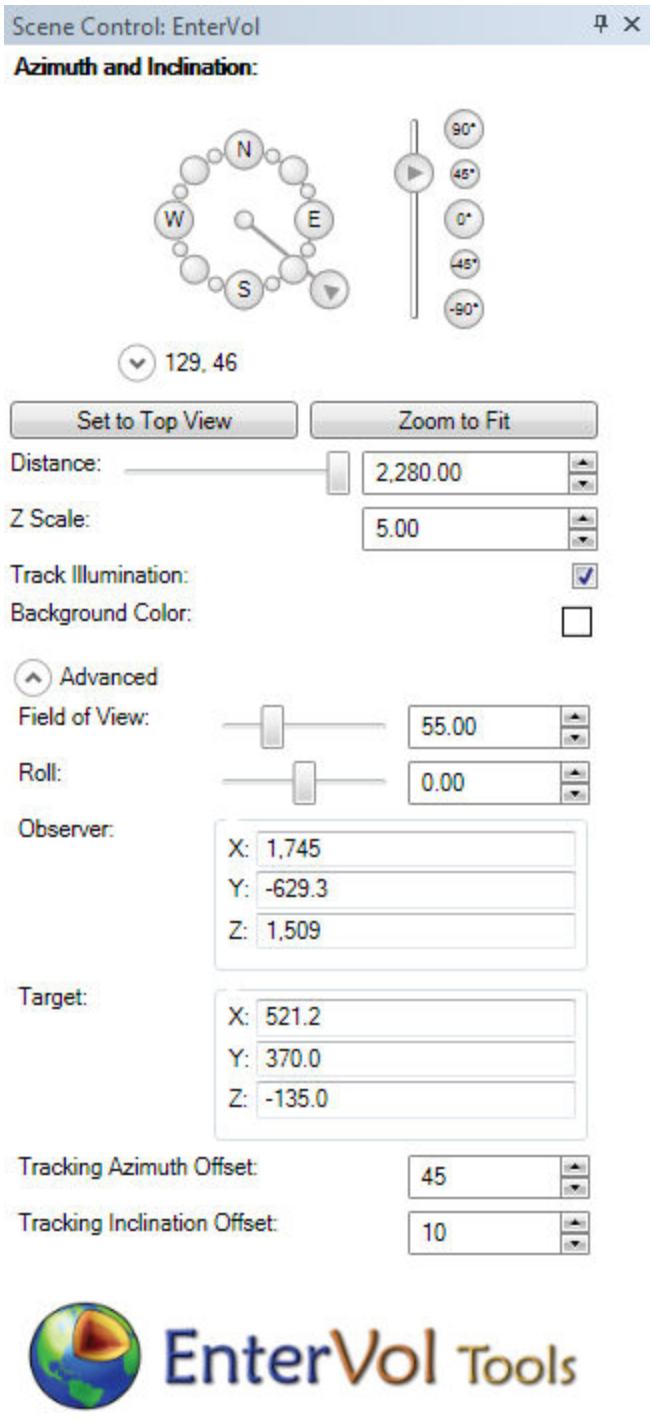


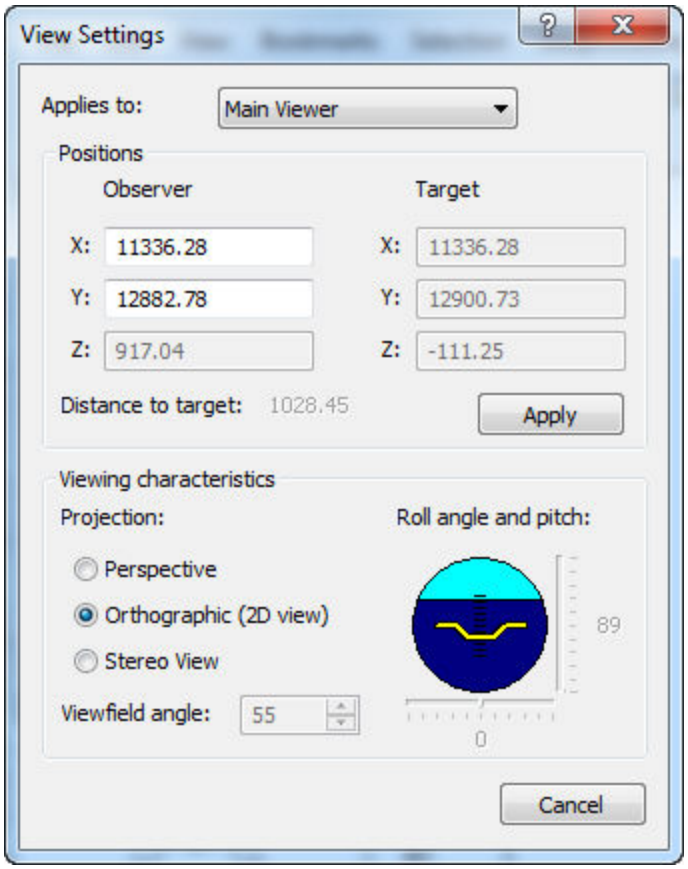
The controls have the following parameters:

- Azimuth dial, buttons and type-in: This sets the compass direction from which you are viewing your layers. Quick buttons are provided at 22.5 degree intervals
  - The Azimuth and Inclination type-ins are not visible by default and are accessed by pressing the v button.
- Inclination (vertical) slider, buttons and type-in: This sets the inclination of your view. 90 degrees (89.9) is looking straight down and

zero is a horizontal view. Quick buttons are provided at 45 degree intervals

- The *Set to Top View* button performs several simultaneous functions.
  - It sets your Azimuth and Inclination to 180,89.9.
  - It resets the view to show all active layers (zoom to fit)
- The *Zoom to Fit* button resets the view to show all active layers
- *Distance* provides a means to zoom.
- *Z Scale* allows you to control the Vertical Exaggeration which would otherwise be set in ArcScene's Scene Properties.
- The  *Lighting* button opens up additional controls. Our tracking headlight ensures that you can see features on all sides of 3D objects.
  - Lighting Mode (EnterVol Headlight or ArcScene Default). The EnterVol Headlight is fixed with respect to the observer...the light follows the camera and is fixed with respect to your eye (not like a sun).
  - Headlight Intensity
- *Background Color* is the same control found in ArcScene's Scene Properties.
- The  *Advanced* button opens up additional controls which are discussed above and are identical to those found in ArcScene's View Setting window.

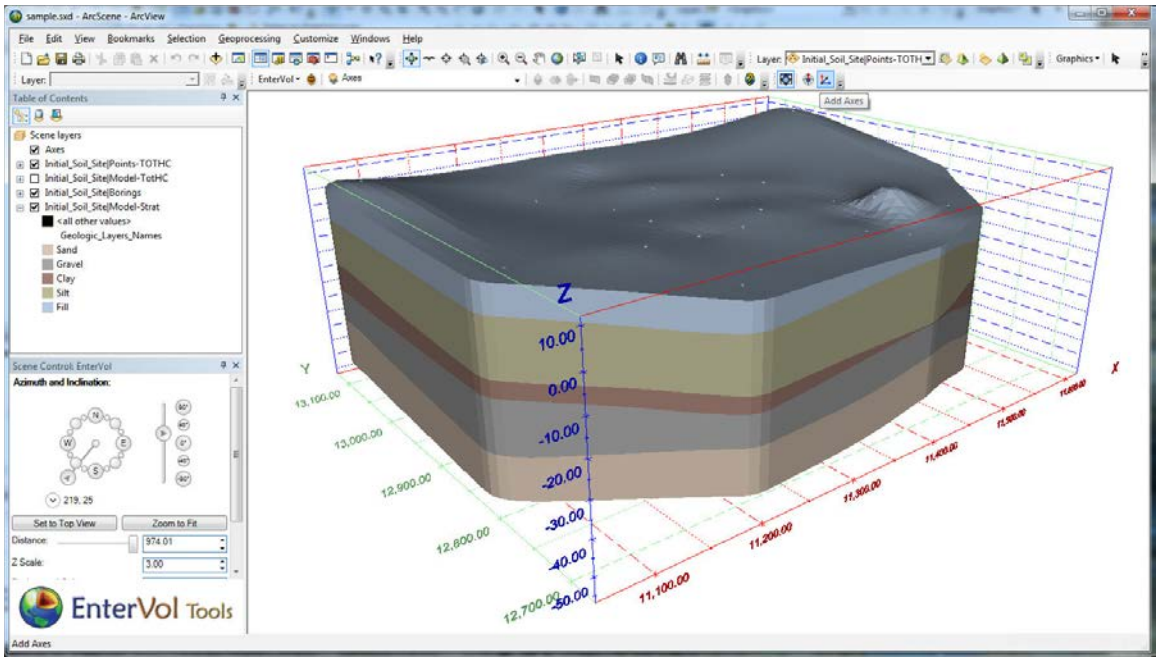

Scene Control: EnterVol
Azimuth and Inclination:
N
W
E
S
90°
45°
0°
-45°
-90°
129, 46
Set to Top View
Zoom to Fit
Distance:
2,280.00
Z Scale:
5.00
Track Illumination:
Background Color:
Advanced
Field of View:
55.00
Roll:
0.00
Observer:
X: 1,745
Y: -629.3
Z: 1,509
Target:
X: 521.2
Y: 370.0
Z: -135.0
Tracking Azimuth Offset:
45
Tracking Inclination Offset:
10
EnterVol Tools




## Axes

The Add Axes button will add axes which serve as a scale reference in all three directions (X, Y & Z)

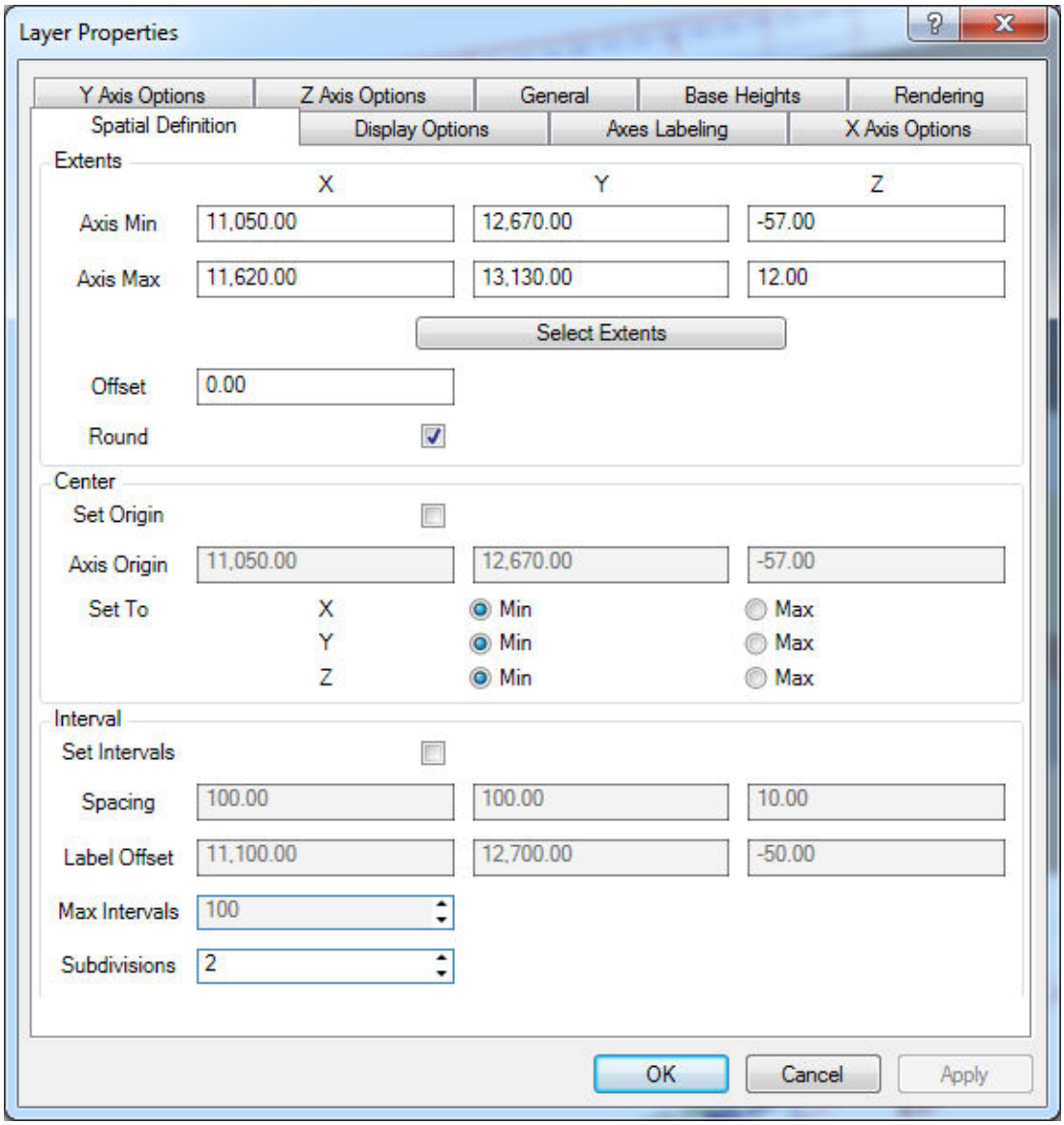
In ArcMap, this same button creates 2D Grid (axes). The relevant unique settings are discussed in the Fence Cut Flatten to 2D topic.




The image above shows the default axes that are obtained by clicking the Add Axes button.

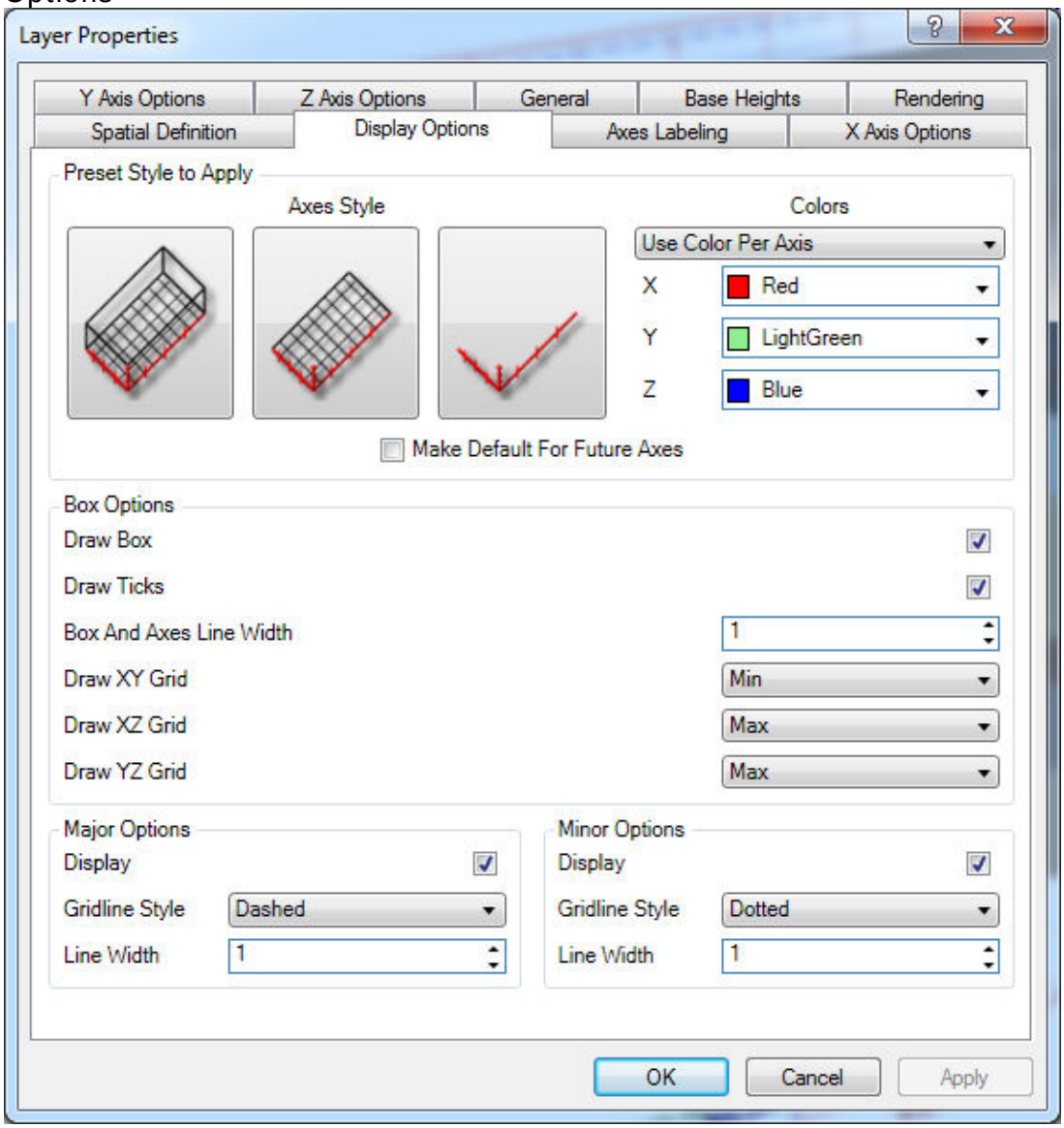
If you right-click on Axes in the Scene Layers and choose properties, the following window will open. There are many options associated with Axes which are accessed through the 6 (non-ArcScene) tabs:

1. Spatial



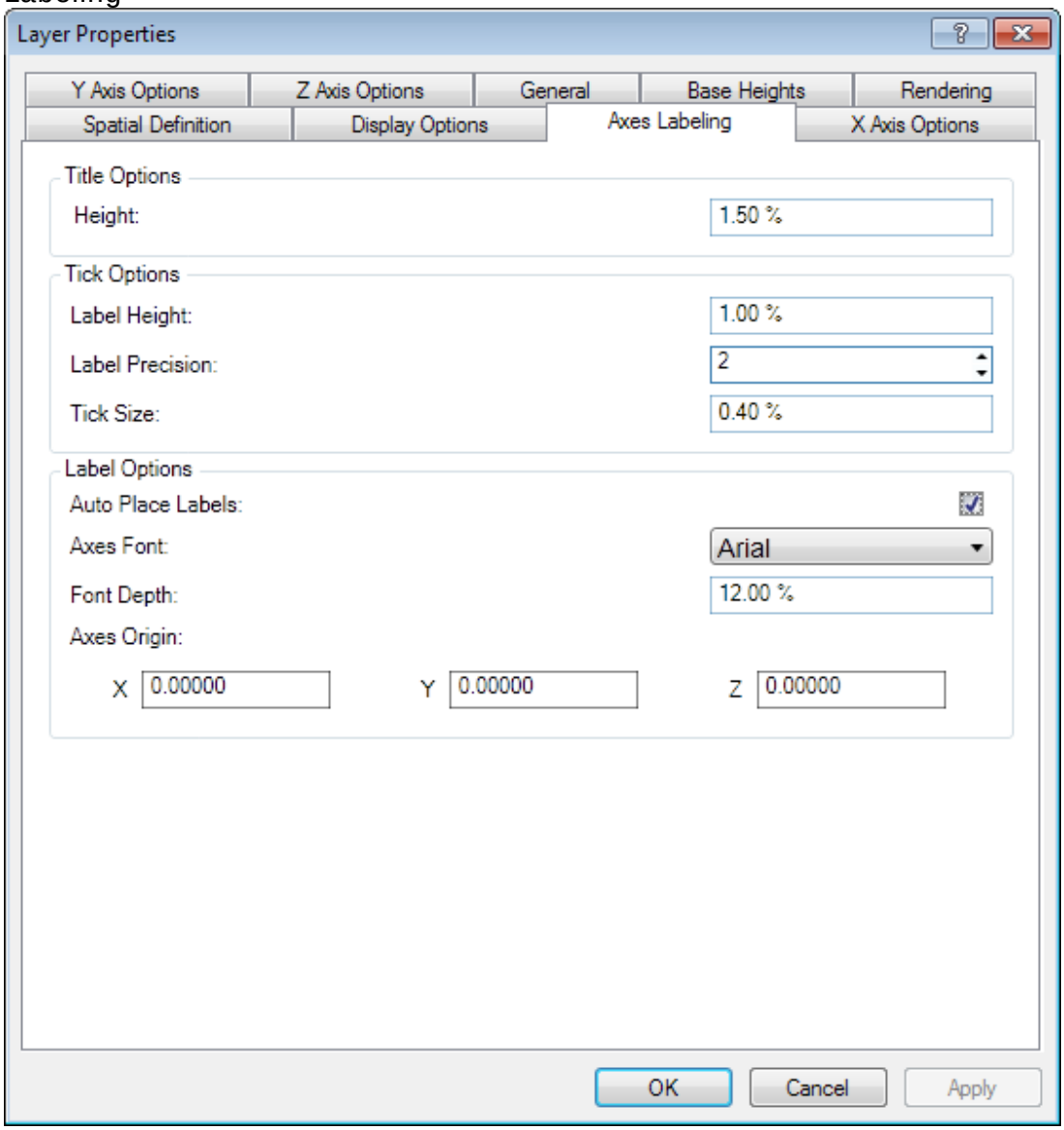

1. The *Extents* of your entire scene are automatically determined, but may be overridden.
2. You can add an offset to make the axes extents larger than the layers in your scene
3. The *Round* toggle uses intelligent rounding of the coordinates
4. The CENTER options allow you to specify where the axes labels occur. This can be the Min or Max of X, Y & Z
5. The INTERVAL options provide control over the labeling spacing

2. Display Options




1. *Axes Style* gives you access to three preset Styles. These can be later customized
2. *Colors* provides control over whether each axis is a unique color
3. *Box Options* determine whether the 8 edged box is fully drawn and whether grids are drawn or certain planes
4. *Major and Minor Options* determine the line width and line style

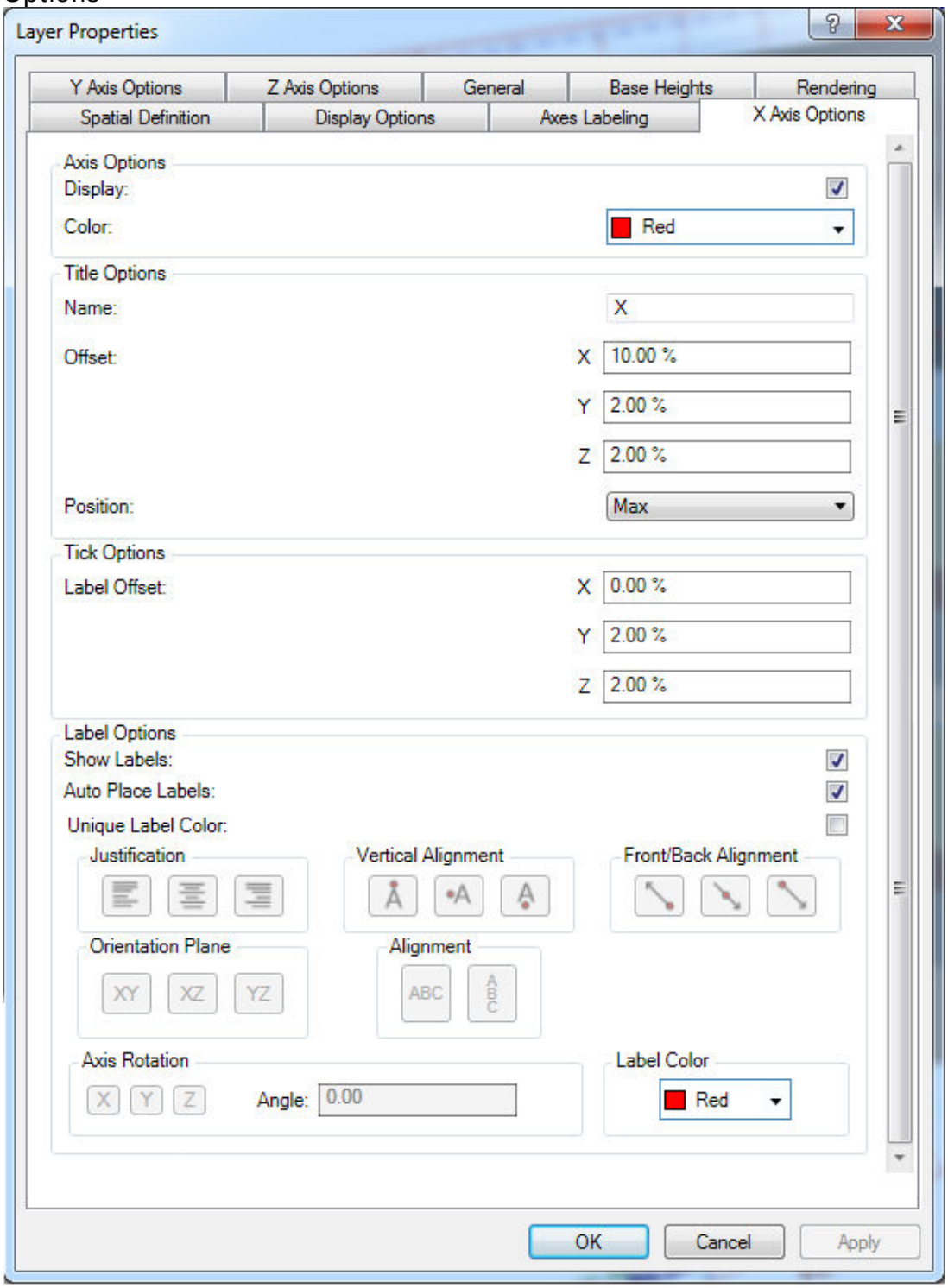
3. Axes
   Labeling




1. Title Options
   1. *Height* is the height of all axes titles are displayed in percent and are proportional to the X-Y-Z extent of all layers in your scene
2. Tick Options
   1. *Label Height* is the height of the coordinate labels are displayed in percent and are proportional to the X-Y-Z extent of all layers in your scene
   2. *Precision* determines the number of decimal points displayed
   3. *Tick Length* is length of ticks are displayed in percent and are proportional to the X-Y-Z extent of all layers in your scene
3. Label Options

1. *Auto Place Labels* is on by default and sets the positioning of labels for you in an intelligent manner. Turn it off to have full control over label placement.
2. *Axes Font* sets the font used for all titles and coordinates
3. *Font Depth* is the extrusion depth as a percentage of label height.
4. *Axes Origin* provides X, Y & Z offsets to the coordinates displayed.

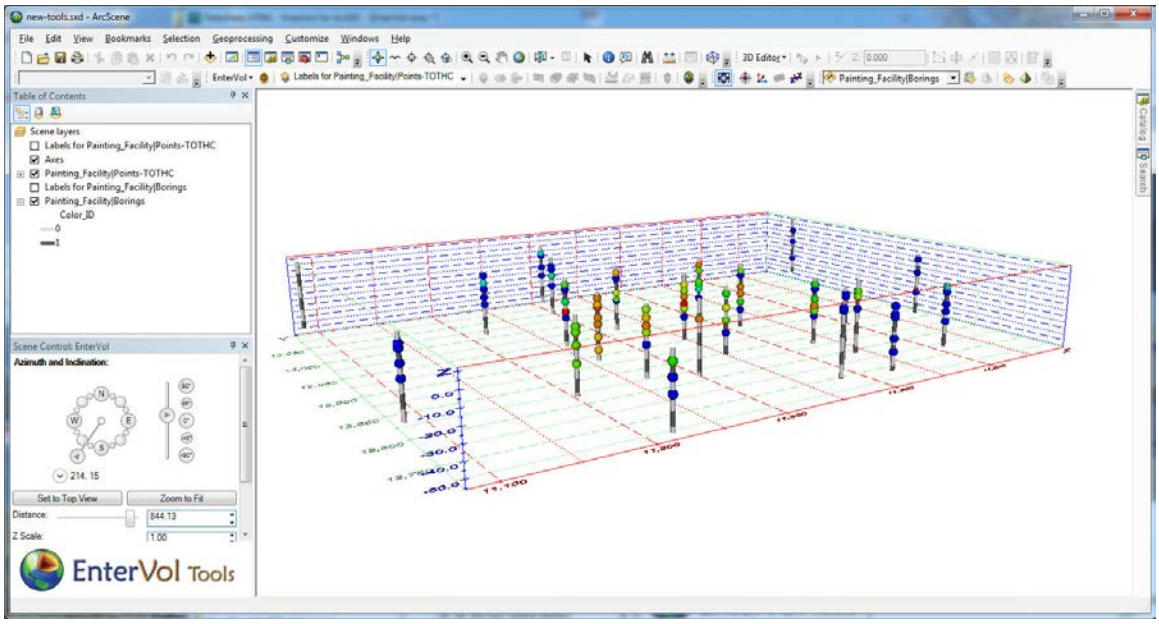
4. X, Y & Z Axis Options




1. The options for all three axes are identical. The
2. Axis Options
    1. *Display* is a toggle whether to display this axis
    2. *Color* sets the color for this axis

3. Title Options
    1. *Name* is the string to display for this axis (e.g. X or Eastings)
    2. *Offset* provides X, Y & Z offsets for the placement of the axis title. These can be positive or negative and are displayed in percent and are proportional to the X-Y-Z extent of all layers in your scene
4. Tick Options
    1. *Offset* provides X, Y & Z offsets for the placement of the axis tick labels. These can be positive or negative and are displayed in percent and are proportional to the X-Y-Z extent of all layers in your scene
5. Label Options
    1. *Show Labels determines if labels are on.*
    2. *Auto Place Labels* is on by default and sets the positioning of labels for you in an intelligent manner. Turn it off to have full control over label placement.
    3. *Unique Label Color* allows you to set the label color different than the axes color.
    4. *Justification* determines the Left-Right justification
    5. *Vertical Alignment* determines the Up-Down justification
    6. *Front/Back Alignment* determines the justification in the extrusion direction
    7. *Orientation Plane* determines in which plane the labels are drawn
    8. *Axis Rotation* and *Angle* determine the rotation of the labels in their plane
    9. *Label Color* is disabled unless unique label color is on

**IMPORTANT NOTE:** Labels, Axes and Direction Indicators are not automatically regenerated when the *Z Scale* is changed in the *EnterVol Scene Controls.* In the example above, if Z Scale is reduced to 1.0, the labels will appear highly compressed (see below)

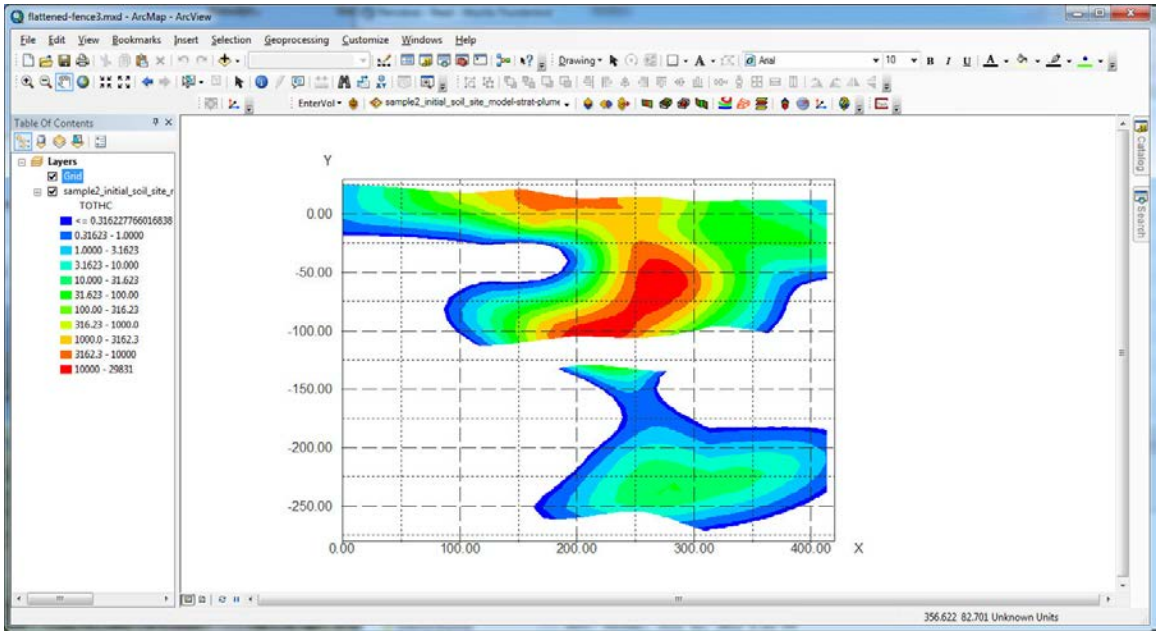



Refreshing the axes is required to restore the proper aspect ratio for labels.

## Grid for ArcMap

In ArcMap, the *Add Grid* button creates 2D Grid (axes). 

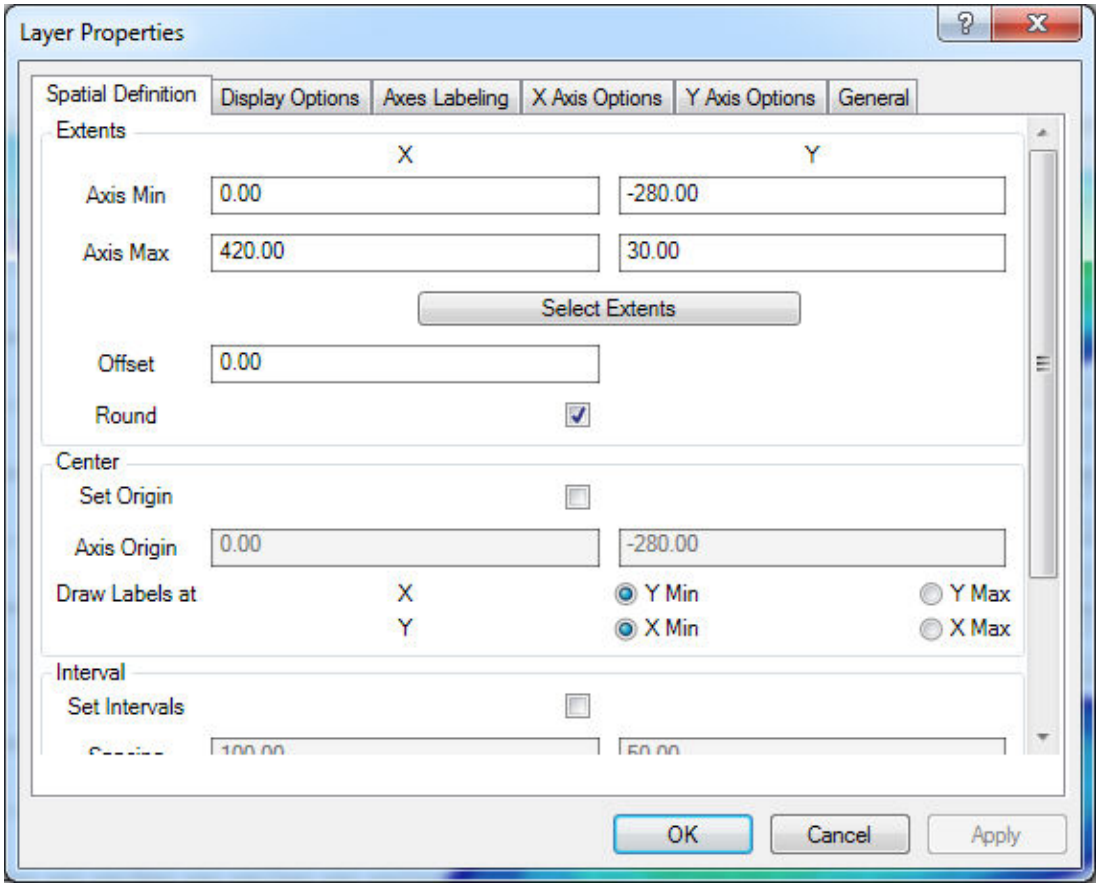

An example using it was discussed in the Fence Cut Flatten to 2D topic.
In general, the path used should contain only a single fence cross section.



The image above shows the default axes that are obtained by clicking the *Add Grid* button.

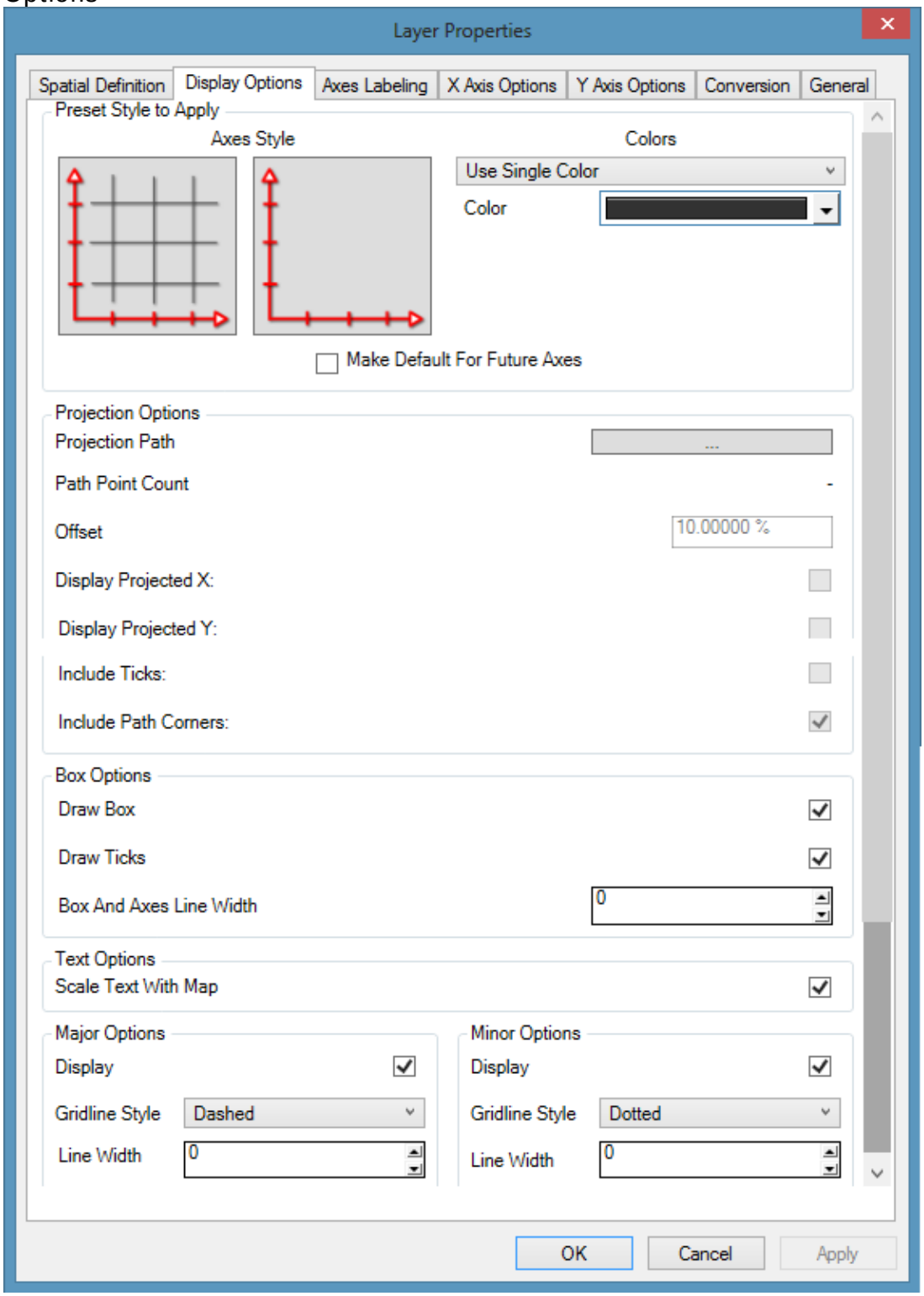
If you right-click on Grid in the Map Layers and choose properties, the following window will open. There are many options associated with Grid which are accessed through the 5 (non-ArcMap) tabs:

1. Spatial Definition




1. The *Extents* of your entire scene are automatically determined, but may be overridden.
2. You can add an offset to make the axes extents larger than the layers in your scene
3. The *Round* toggle uses intelligent rounding of the coordinates
4. The CENTER options allow you to specify where the axes labels occur. This can be the Min or Max of X & Y
5. The INTERVAL options provide control over the labeling spacing

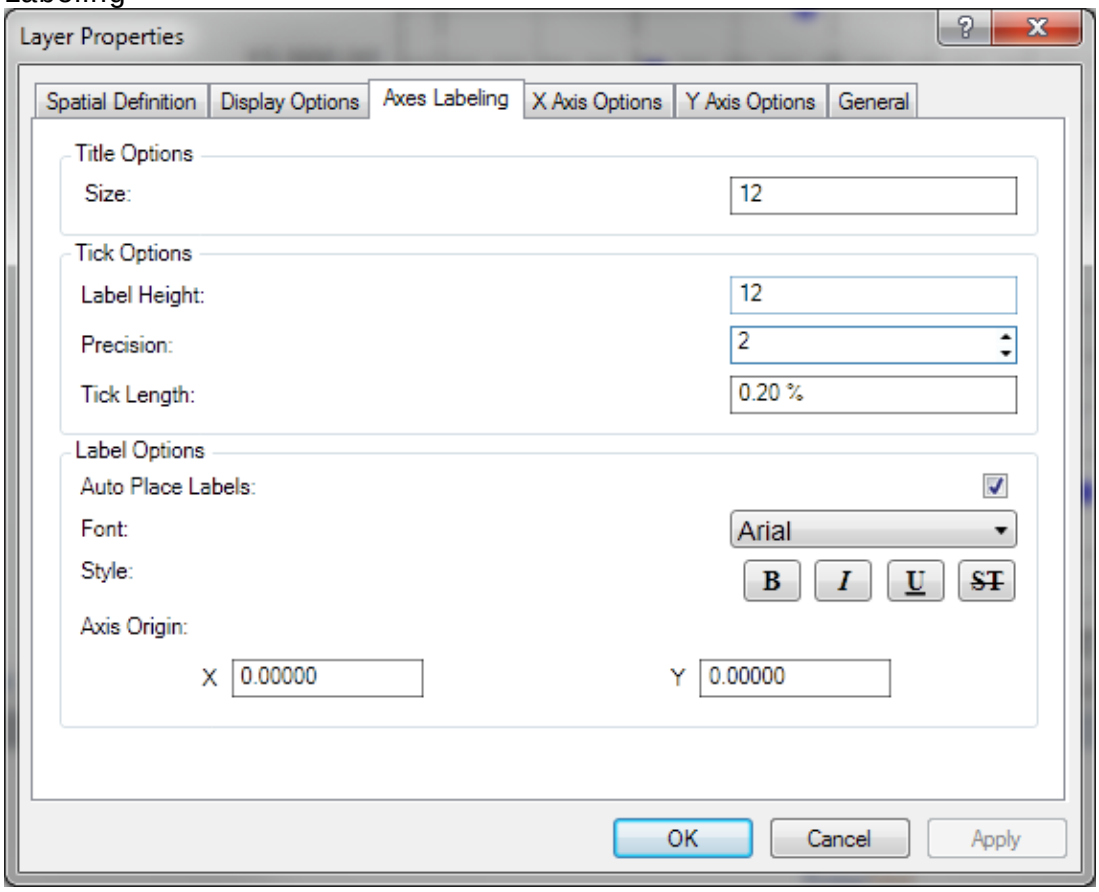
2. Display Options




1. *Axes Style* gives you access to two preset Styles. These can be later customized
2. *Colors* provides control over whether each axis is a unique color

3. Projections Options provides control to import a projection path and labels the grid based on the original X and Y coordinates instead of just the path length. An example of using this is shown in this topic.
4. *Box Options* determine whether the 4 edged box is fully drawn and whether a grid is drawn
5. *Major and Minor Options* determine the line width and line style

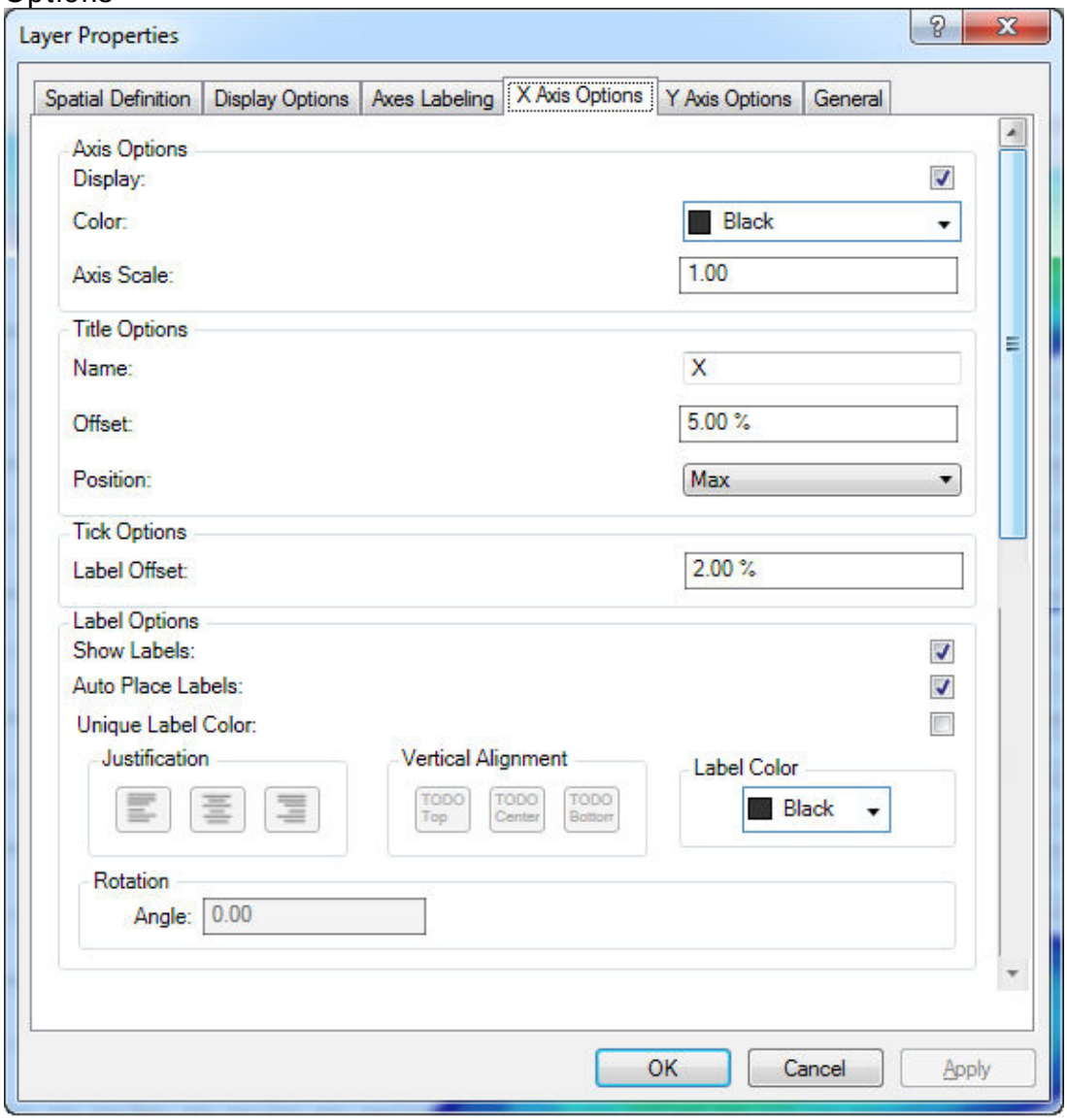
3. Axes Labeling




1. Title Options
    1. *Size* is the height of all axes titles are displayed in percent and are proportional to the X-Y extent of all layers in your map
2. Tick Options
    1. *Label Height* is the height of the coordinate labels are displayed in percent and are proportional to the X-Y extent of all layers in your map
    2. *Precision* determines the number of decimal points displayed
    3. *Tick Length* is length of ticks are displayed in percent and are proportional to the X-Y extent of all layers in your scene
3. Label Options

1. *Auto Place Labels* is on by default and sets the positioning of labels for you in an intelligent manner. Turn it off to have full control over label placement.
2. *Font* sets the font used for all titles and coordinates
3. *Axes Origin* provides X & Y offsets to the coordinates displayed.

4. X & Y Axis Options



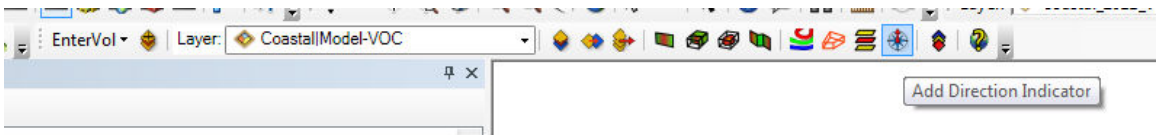

1. The options for both X & Y axes are identical. The X Axis is shown above
2. Axis Options
   1. *Display* is a toggle whether to display this axis
   2. *Color* sets the color for this axis
   3. ***Axis Scale*** is an important and unique feature in EnterVol Tool's Grid. It allows you to apply a scale factor to the labeled coordinates for either axis. For maps representing

something other than X-Y Spatial coordinates, this is often critically important.

3. Title Options
    1. *Name* is the string to display for this axis (e.g. X or Eastings)
    2. *Offset* provides X & Y offsets for the placement of the axis title. These can be positive or negative and are displayed in percent and are proportional to the X-Y extent of all layers in your scene
    3. *Position* is the where the axis title will be placed. Options are Min and Max
4. Tick Options
    1. *Label Offset* provides X & Y offsets for the placement of the axis tick labels. These can be positive or negative and are displayed in percent and are proportional to the X-Y extent of all layers in your scene
5. Label Options
    1. *Show Labels determines if labels are on.*
    2. *Auto Place Labels* is on by default and sets the positioning of labels for you in an intelligent manner. Turn it off to have full control over label placement.
    3. *Unique Label Color* allows you to set the label color different than the axes color.
    4. *Justification* determines the Left-Right justification
    5. *Vertical Alignment* determines the Up-Down justification
    6. *Label Color* is disabled unless unique label color is on
    7. *RotationAngle* determine the rotation of the labels in the X-Y plane

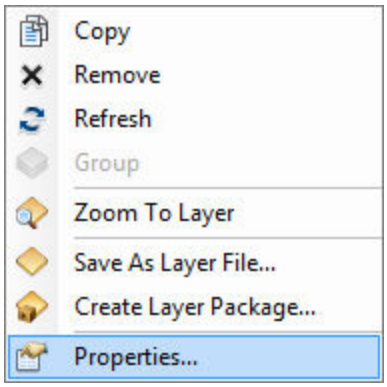
## Add Direction Indicator

EnterVol's Tools includes the ability to add a "North" Direction Indicator

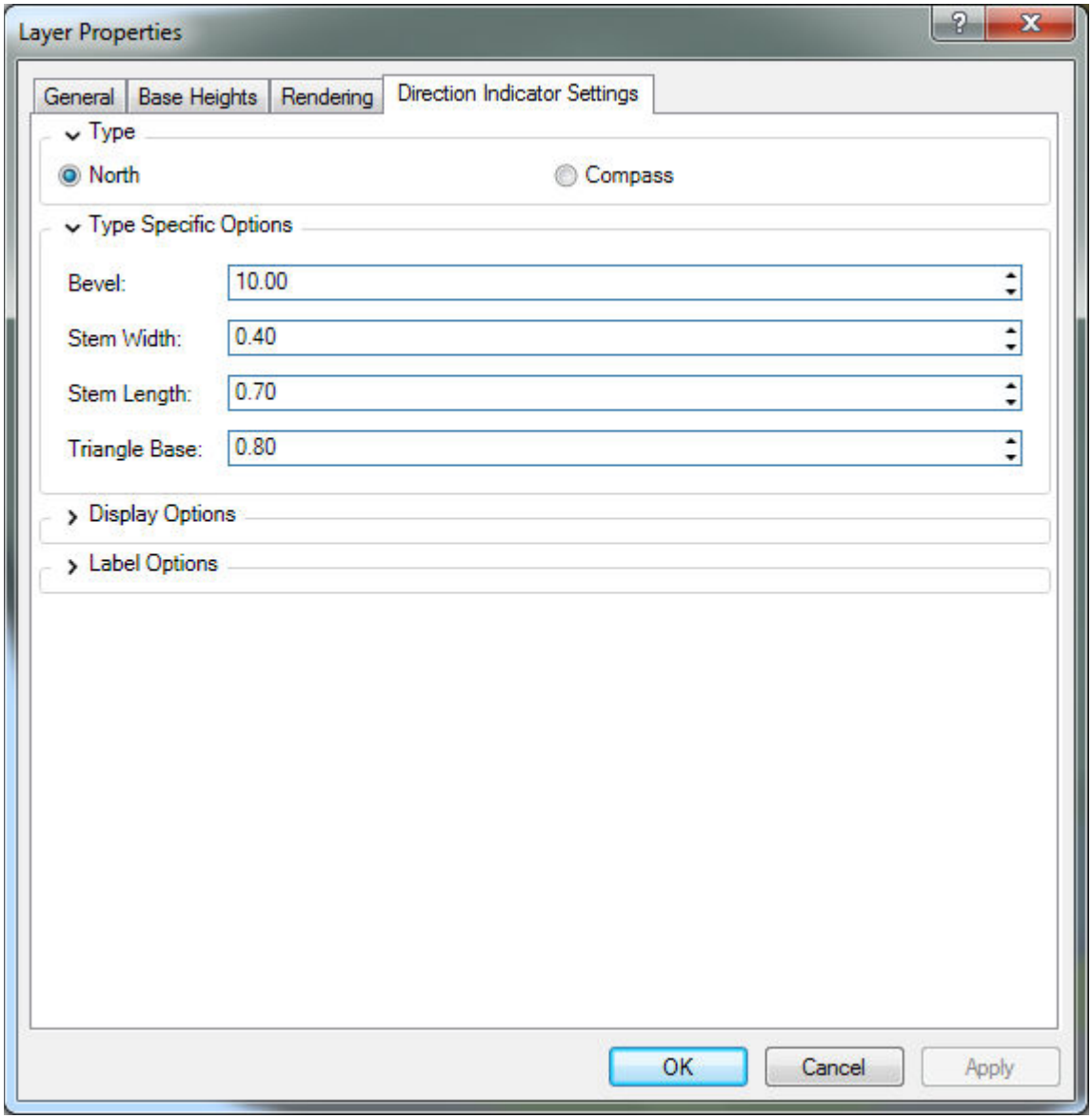



As soon as you press the button, it adds a "North" scene layer to the Table of Contents.

To modify the position and properties of the indicator, right click on the North layer and choose Properties.

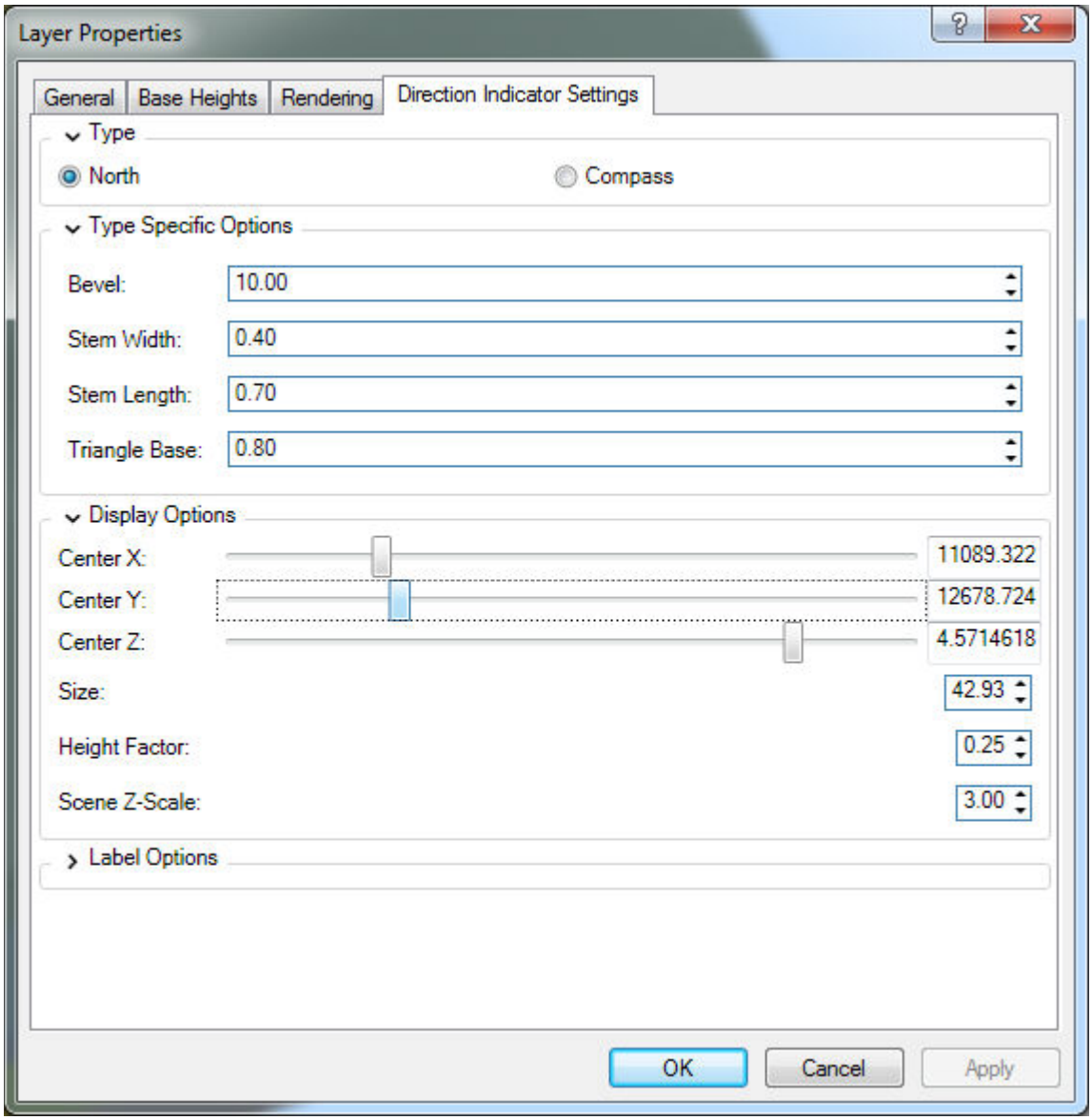


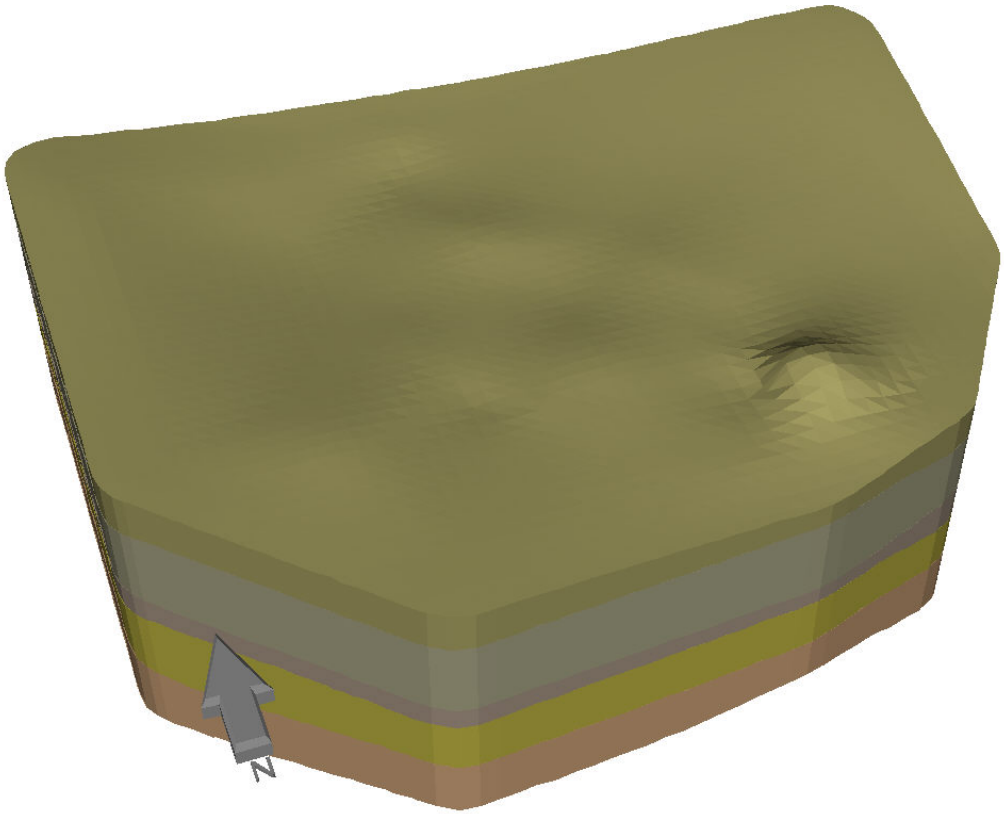

In the Properties window you'll find Direction Indicator Settings:



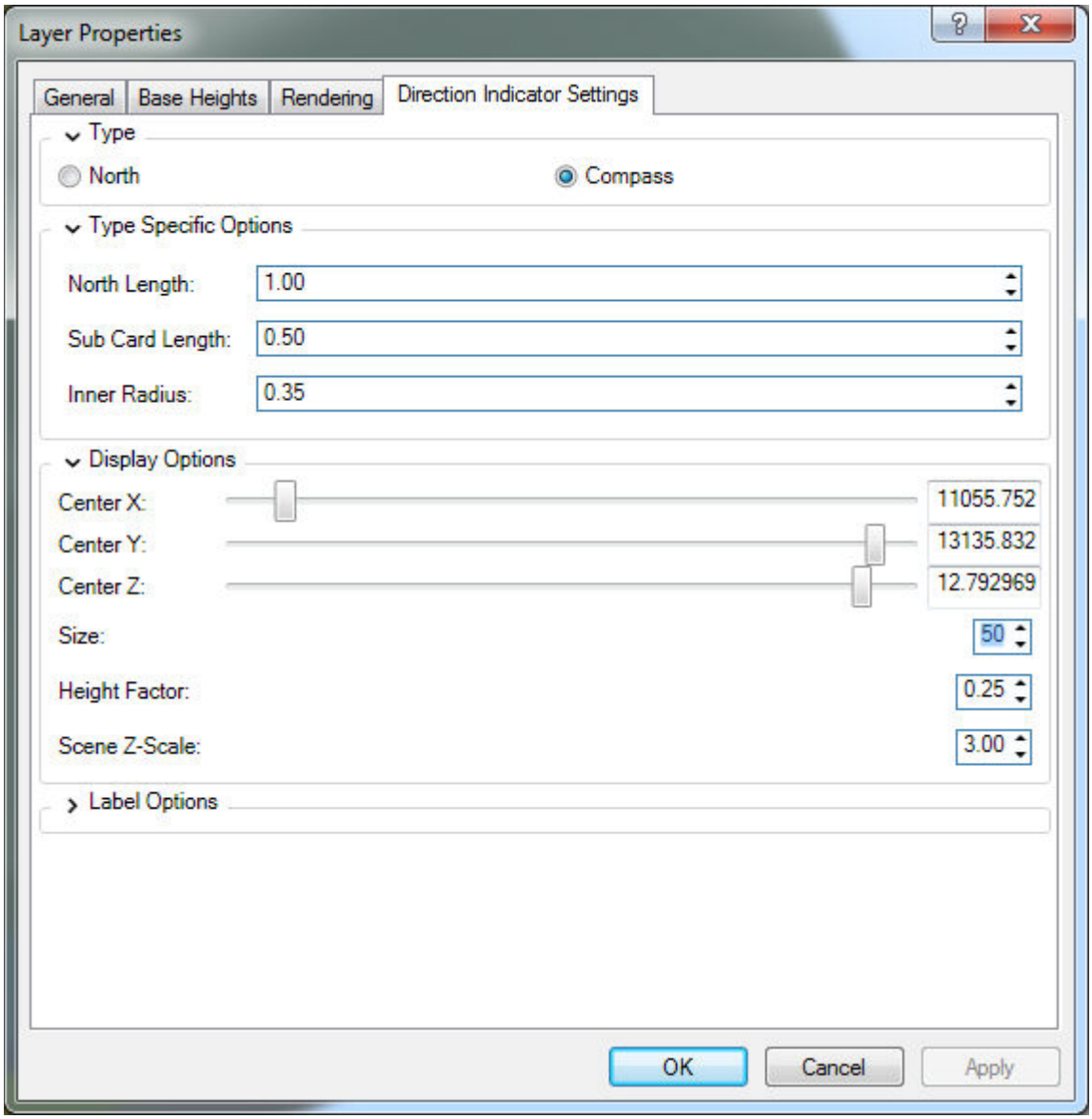
Here you can choose between a North Arrow or Compass Rose indicator and set the size, aspect, and position of the indicator.

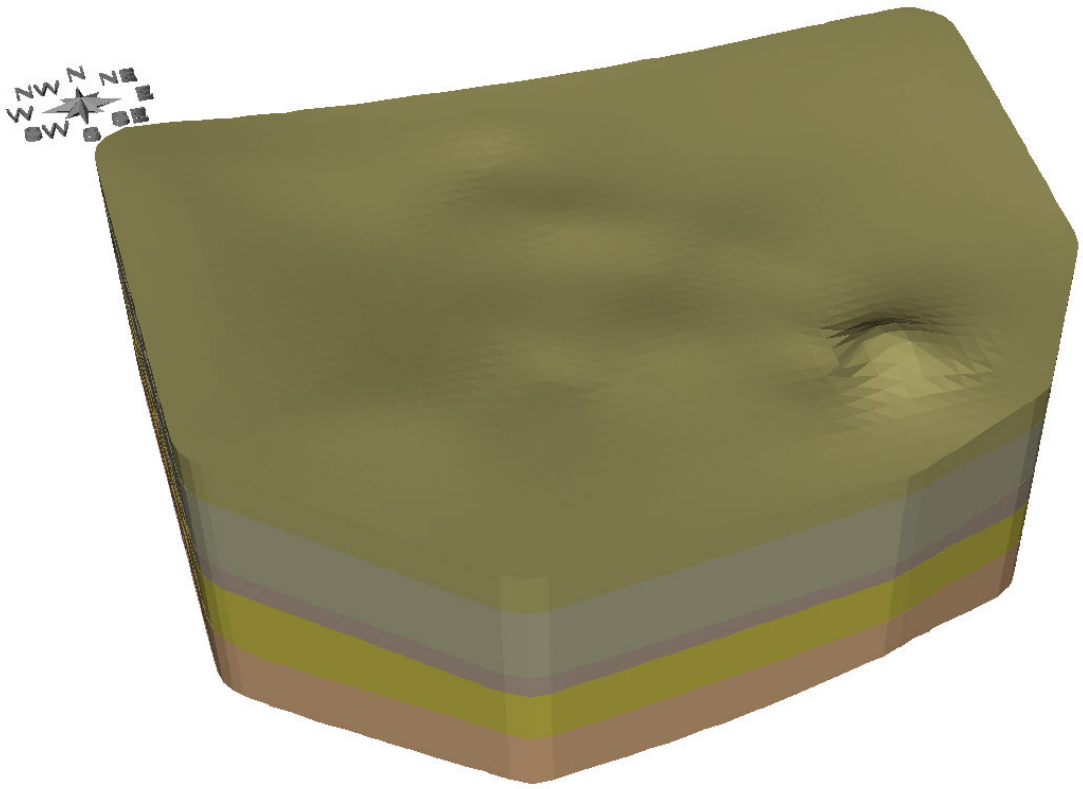
As you make changes, you can click *Apply* and it will update the indicator and view.



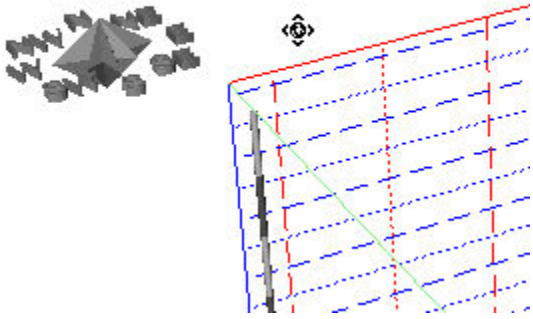



or

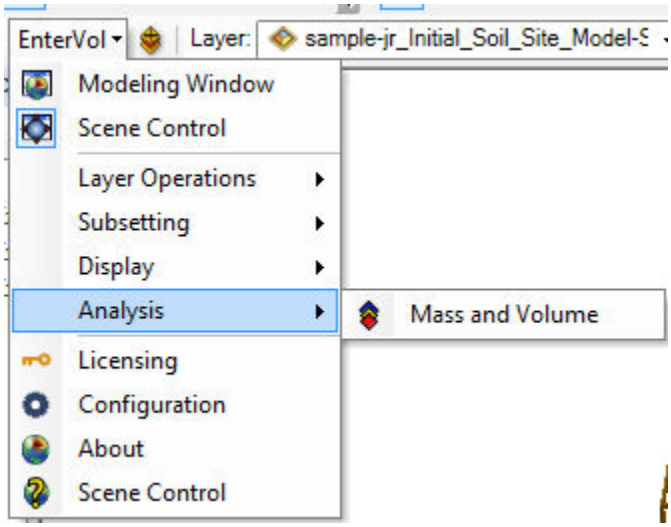

Layer Properties
General
Base Heights
Rendering
Direction Indicator Settings
Type
North
Compass
Type Specific Options
North Length:
1.00
Sub Card Length:
0.50
Inner Radius:
0.35
Display Options
Center X:
11055.752
Center Y:
13135.832
Center Z:
12.792969
Size:
50
Height Factor:
0.25
Scene Z-Scale:
3.00
Label Options
OK
Cancel
Apply



**IMPORTANT NOTE:** Labels, Axes and Direction Indicators are not automatically regenerated when the *Z Scale* is changed in the *EnterVol Scene Controls.* In the example above, if Z Scale is changed, the labels and indicator can appear highly distorted. Refresh the North layer to regenerate the indicator for the new Z Scale.



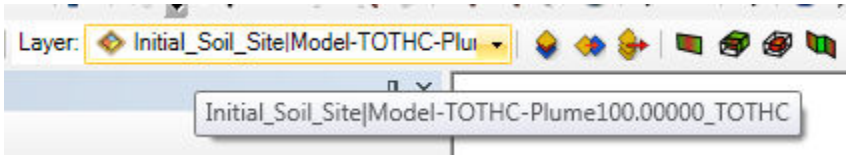
# Mass and Volume for Environmental Sciences



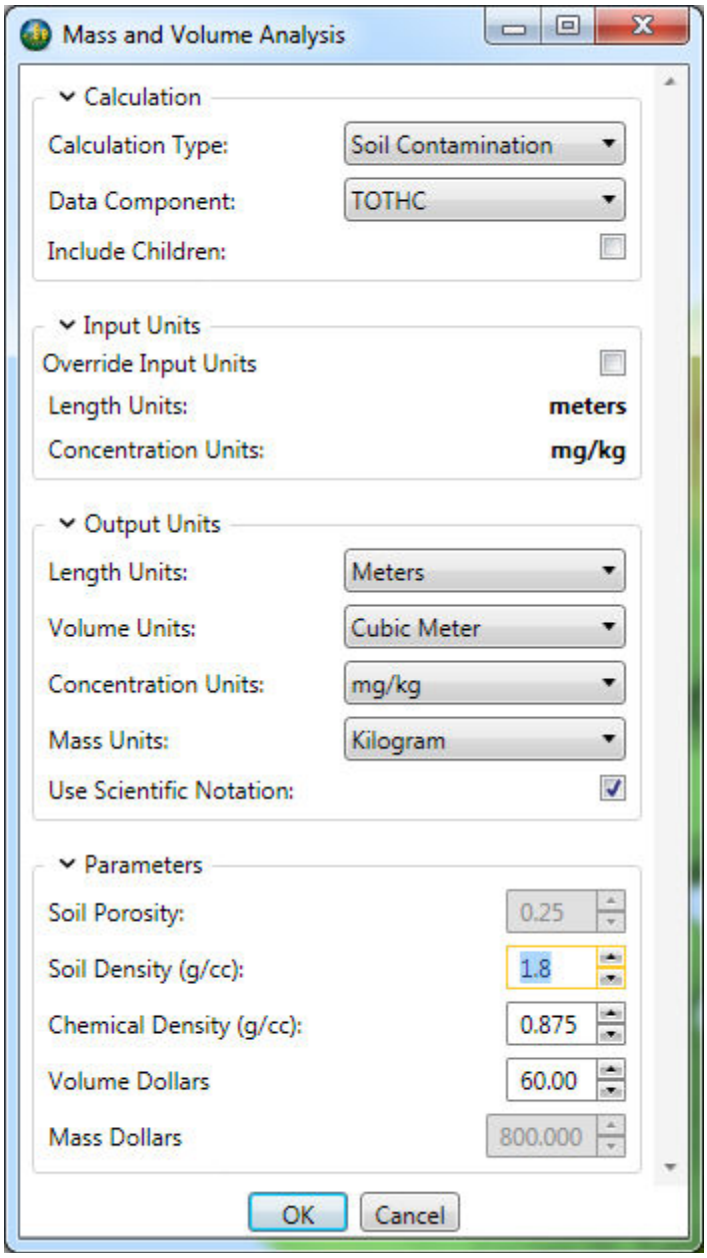

In EnterVol, volumetric analysis is far more than determining the volume of space bounded by surfaces. Since everything we do is truly volumetric with data throughout the volumes, it is possible to compute many important characteristics of your models. This includes plume volume, chemical volume (if distilled or concentrated), soil mass, chemical mass, center of gravity and values ($) associated with both volumes and masses. All calculations are performed for the entire layer and each geologic unit within it.

Let's begin by creating a plume at 100 mg/kg for the TOTHC scene at the end of the Set Symbology topic.

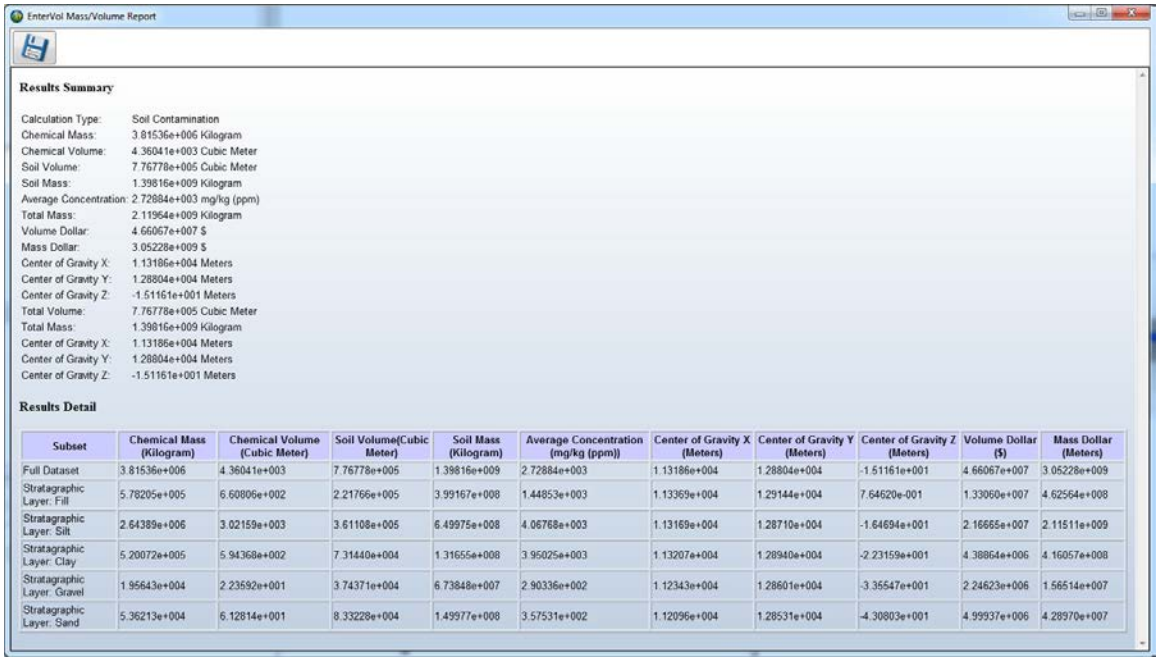
After creating the plume select it in the EnterVol Layer selector




click on the *Mass and Volume* button  and the Mass and Volume Analysis window opens. Let's set the Chemical Density to 0.875 and the Soil Density to 1.8.

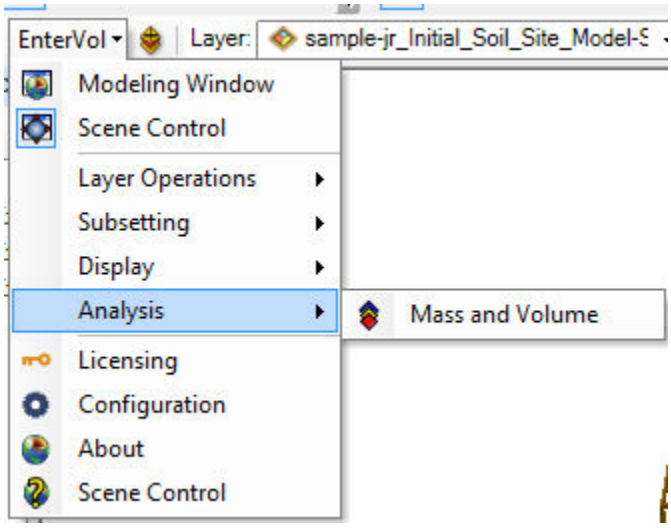



We'll accept these values and when the calculations are complete the following new window appears (please note that we've expanded this window to show all of it). Since the model used to create the plume had stratigraphy, we not only get the analysis for the total plume, but also for each stratigraphic layer.




| Subset | Chemical Mass (Kilogram) | Chemical Volume (Cubic Meter) | Soil Volume(Cubic Meter) | Soil Mass (Kilogram) | Average Concentration (mg/kg (ppm)) | Center of Gravity X (Meters) | Center of Gravity Y (Meters) | Center of Gravity Z (Meters) | Volume Dollar ($) | Mass Dollar (Meters) |
|---|---|---|---|---|---|---|---|---|---|---|
| Full Dataset | 3.81536e+006 | 4.36041e+003 | 7.76778e+005 | 1.39816e+009 | 2.72884e+003 | 1.13186e+004 | 1.28804e+004 | -1.51161e+001 | 4.66067e+007 | 3.05228e+009 |
| Stratagraphic Layer: Fill | 5.78205e+005 | 6.60806e+002 | 2.21766e+005 | 3.99167e+008 | 1.44853e+003 | 1.13369e+004 | 1.29144e+004 | 7.64620e-001 | 1.33060e+007 | 4.62564e+008 |
| Stratagraphic Layer: Silt | 2.64389e+006 | 3.02159e+003 | 3.61108e+005 | 6.49975e+008 | 4.06768e+003 | 1.13169e+004 | 1.28710e+004 | -1.64694e+001 | 2.16665e+007 | 2.11511e+009 |
| Stratagraphic Layer: Clay | 5.20072e+005 | 5.94368e+002 | 7.31440e+004 | 1.31655e+008 | 3.95025e+003 | 1.13207e+004 | 1.28940e+004 | -2.23159e+001 | 4.38864e+006 | 4.16057e+008 |
| Stratagraphic Layer: Gravel | 1.95643e+004 | 2.23592e+001 | 3.74371e+004 | 6.73848e+007 | 2.90336e+002 | 1.12343e+004 | 1.28601e+004 | -3.35547e+001 | 2.24623e+006 | 1.56514e+007 |
| Stratagraphic Layer: Sand | 5.36213e+004 | 6.12814e+001 | 8.33228e+004 | 1.49977e+008 | 3.57531e+002 | 1.12096e+004 | 1.28531e+004 | -4.30803e+001 | 4.99937e+006 | 4.28970e+007 |

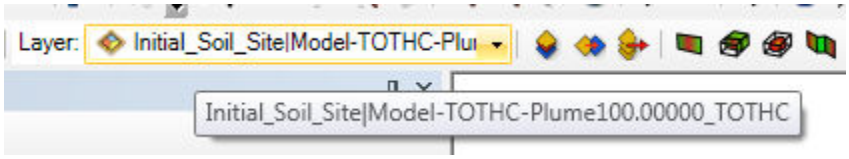
## Mass and Volume for Environmental Sciences



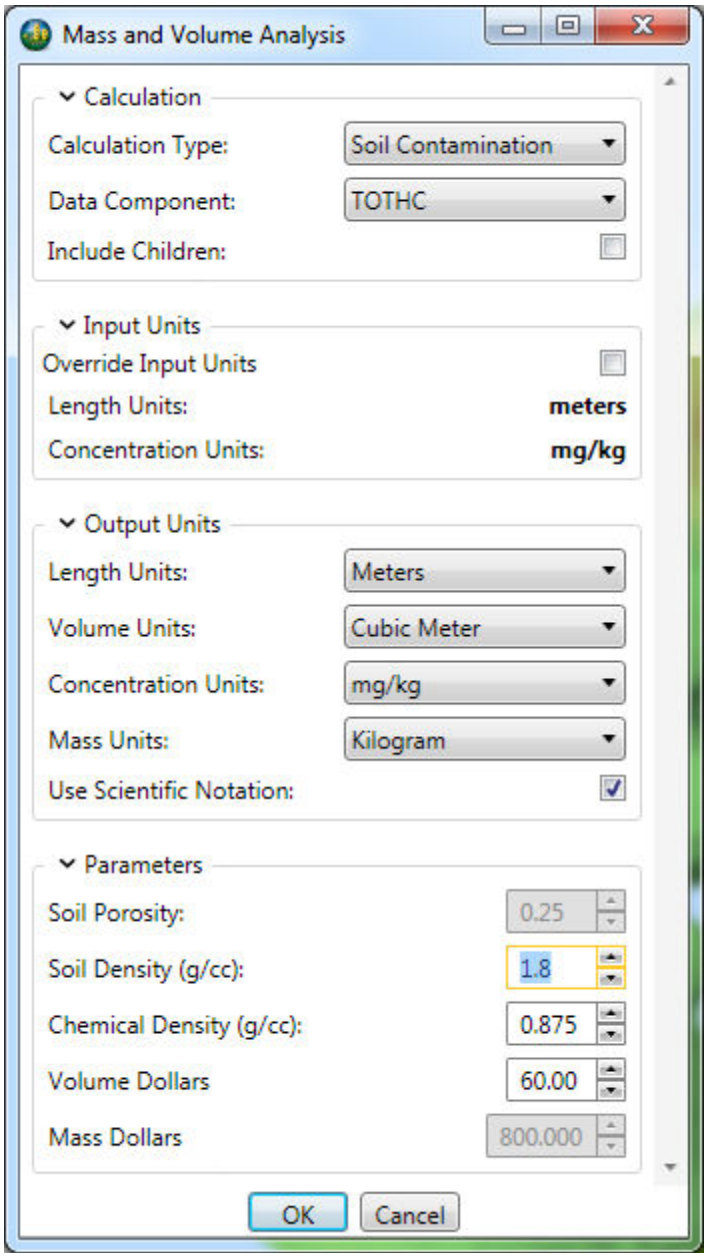

In EnterVol, volumetric analysis is far more than determining the volume of space bounded by surfaces. Since everything we do is truly volumetric with data throughout the volumes, it is possible to compute many important characteristics of your models. This includes plume volume, chemical volume (if distilled or concentrated), soil mass, chemical mass, center of gravity and values ($) associated with both volumes and masses. All calculations are performed for the entire layer and each geologic unit within it.

Let's begin by creating a plume at 100 mg/kg for the TOTHC scene at the end of the Set Symbology topic.

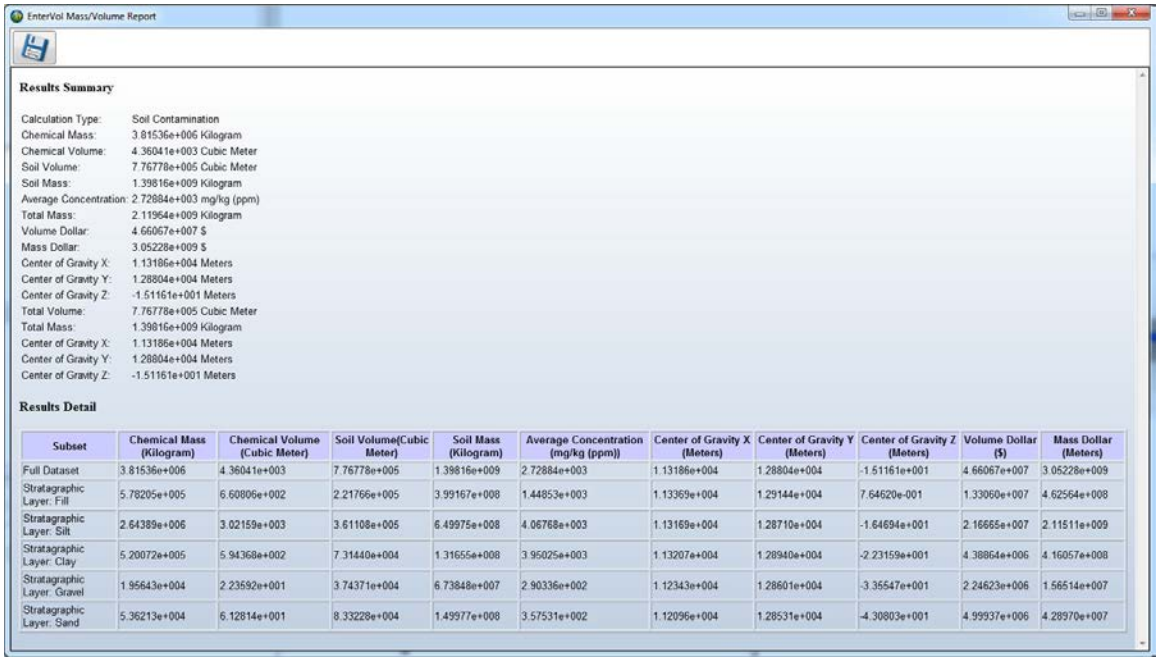
After creating the plume select it in the EnterVol Layer selector



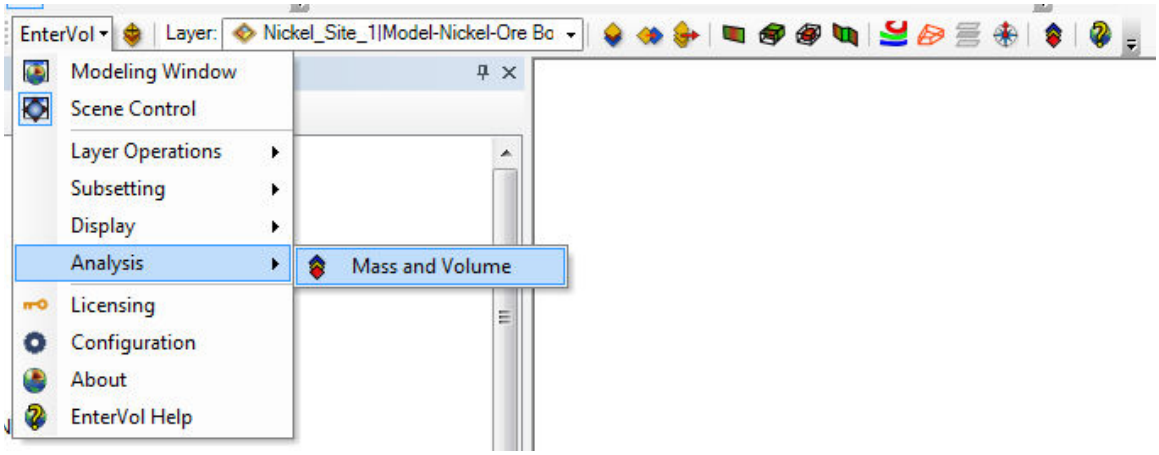

click on the *Mass and Volume* button and the Mass and Volume Analysis window opens. Let's set the Chemical Density to 0.875 and the Soil Density to 1.8.



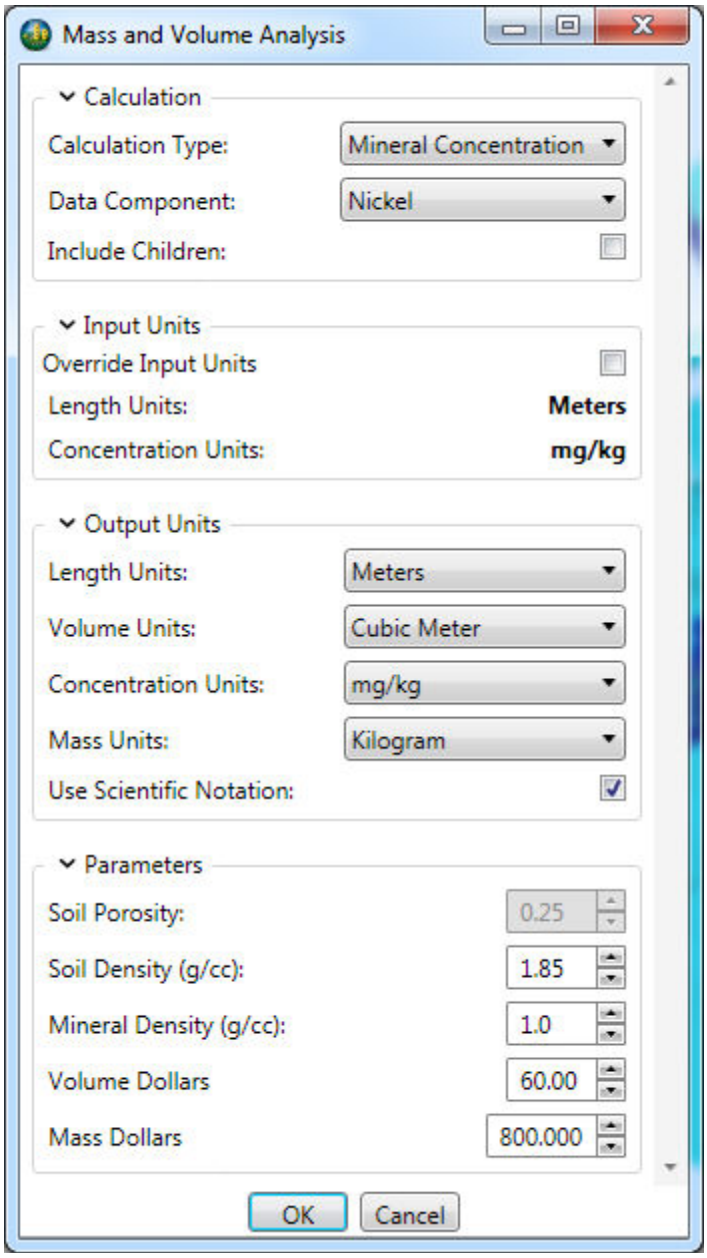

We'll accept these values and when the calculations are complete the following new window appears (please note that we've expanded this window to show all of it). Since the model used to create the plume had stratigraphy, we not only get the analysis for the total plume, but also for each stratigraphic layer.



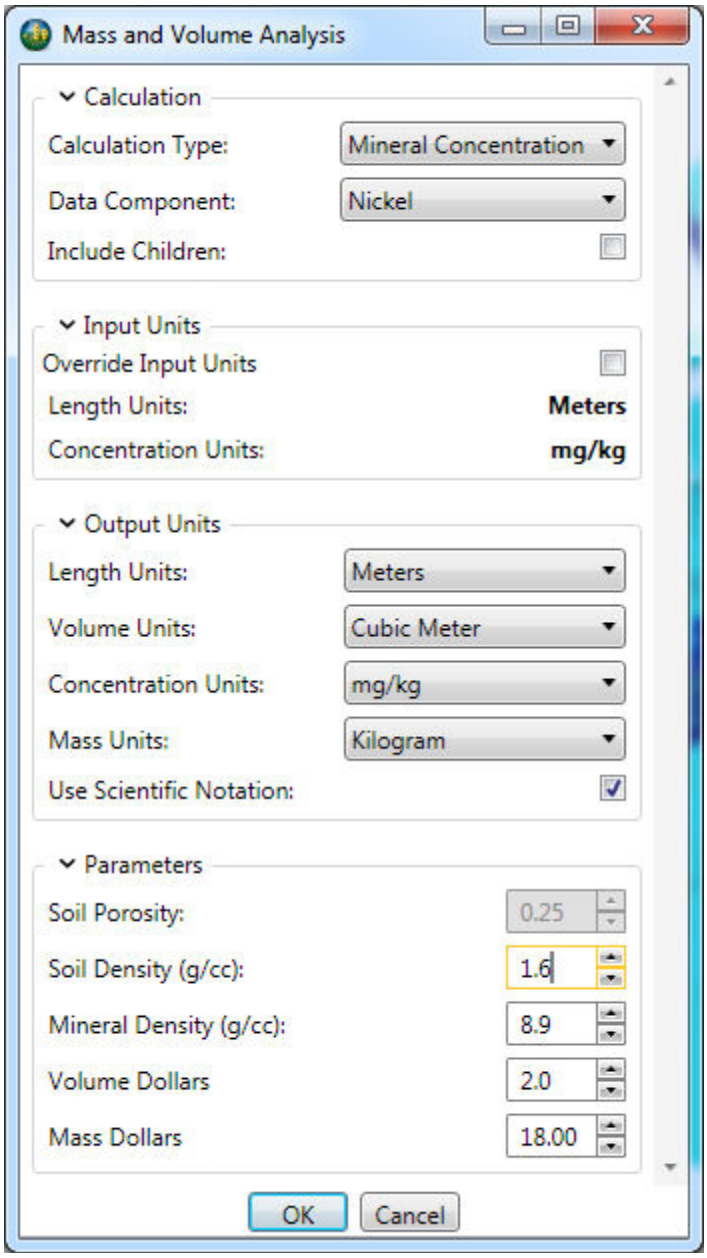

## Mass and Volume for Mining



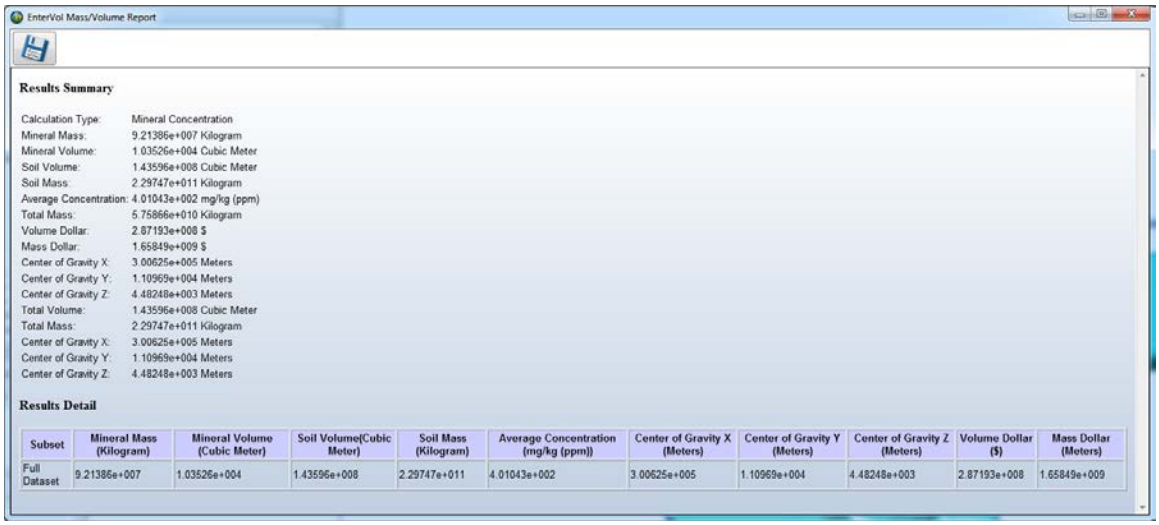

Using the same dataset used for the Ore Body topic, below is the default settings when we choose *Mass and Volume*with a 291 ppm ore body as the selected layer.




For our model of nickel ore, we'll want to change the *Soil Density* to 1.6, the *Mineral Density* to 8.9, and the *Mass Dollars* to $18 (dollars per kg). Let's also estimate the excavation and ore processing cost as the *Volume Dollars* at $2.00 per cubic meter.



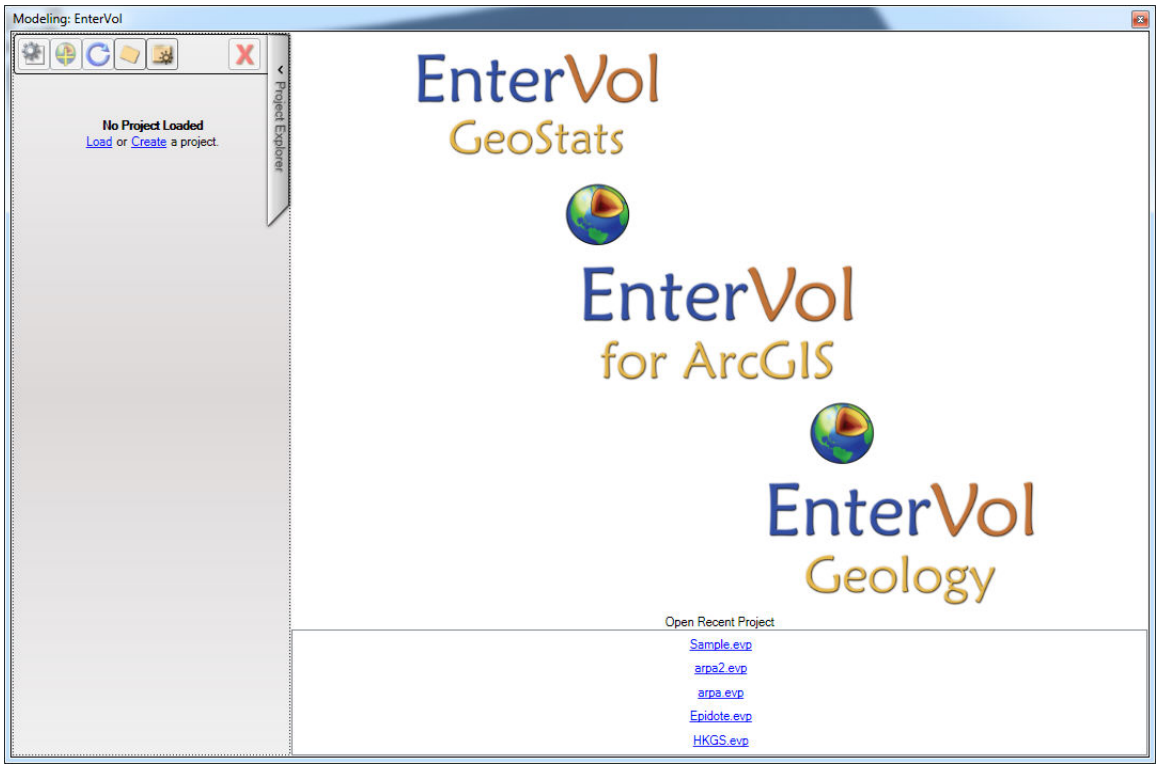

When we click OK we get:



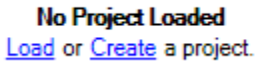

Note that our nickel reserves have a gross value $1.65 billion and our excavation and processing costs (at $2 per sq. m) are $0.287 billion. We can easily determine the maximum processing costs where this ore body would still be profitable.

# Load or Create a Project

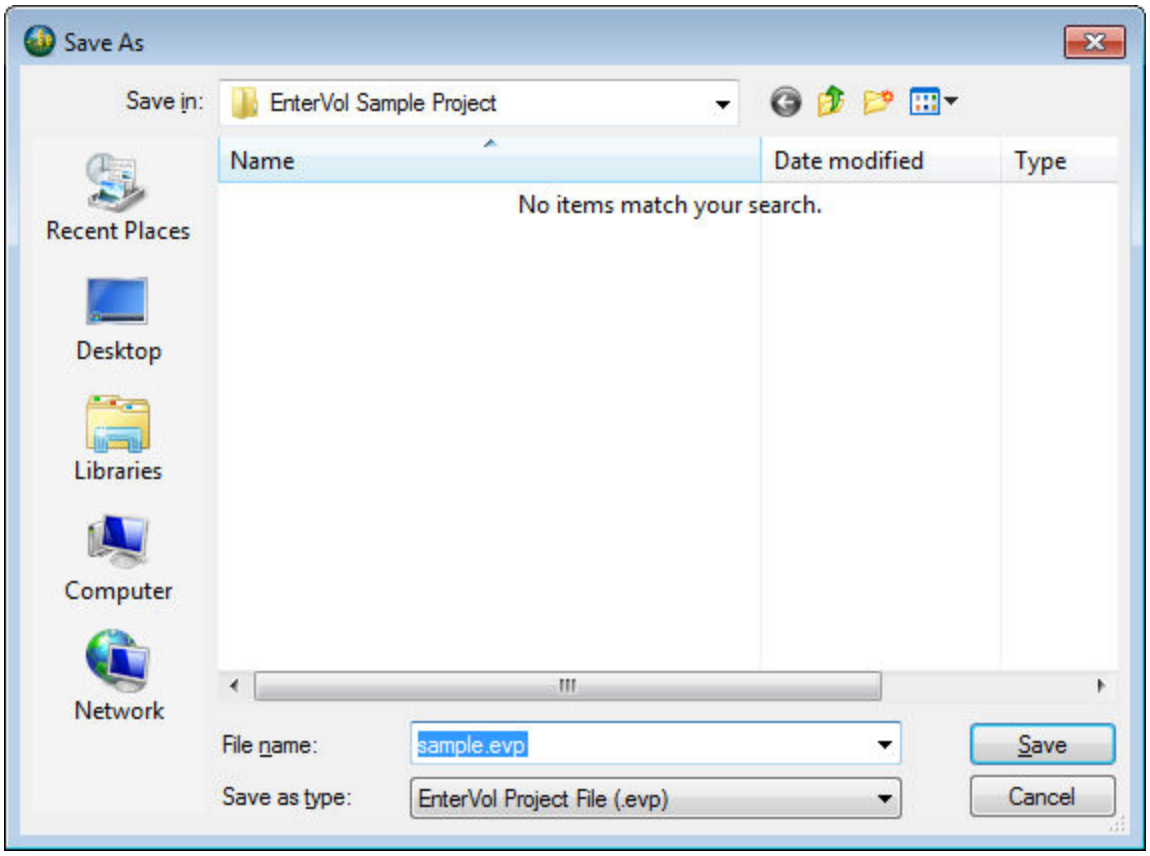
A project is the first step towards creating volumetric models. You can load an existing project (.EVP - EnterVol Project) or create a new project.



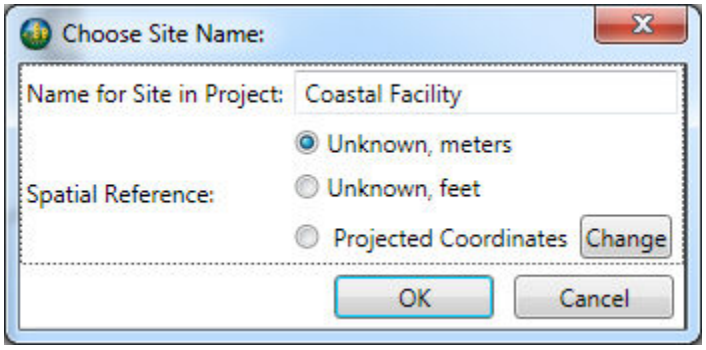

Project files (.evp) are databases into which you load your geologic and analyte data



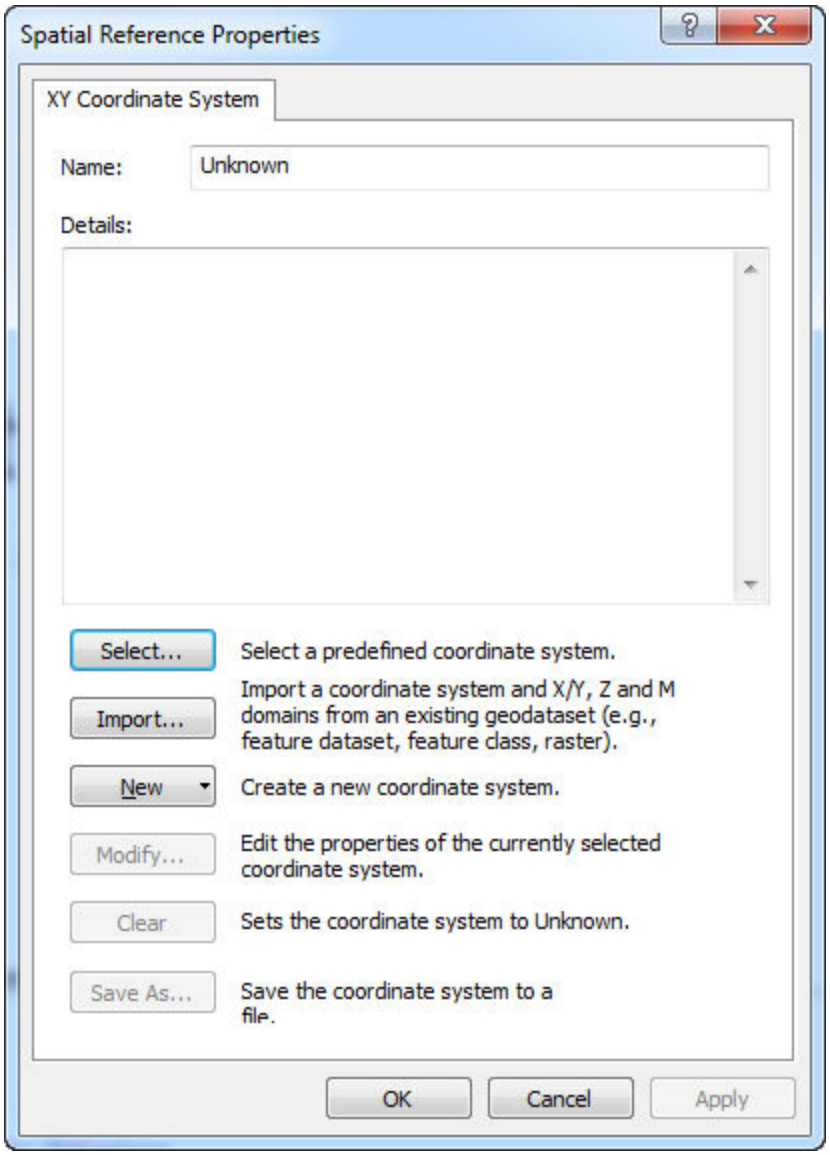

Let's begin by creating a new project:



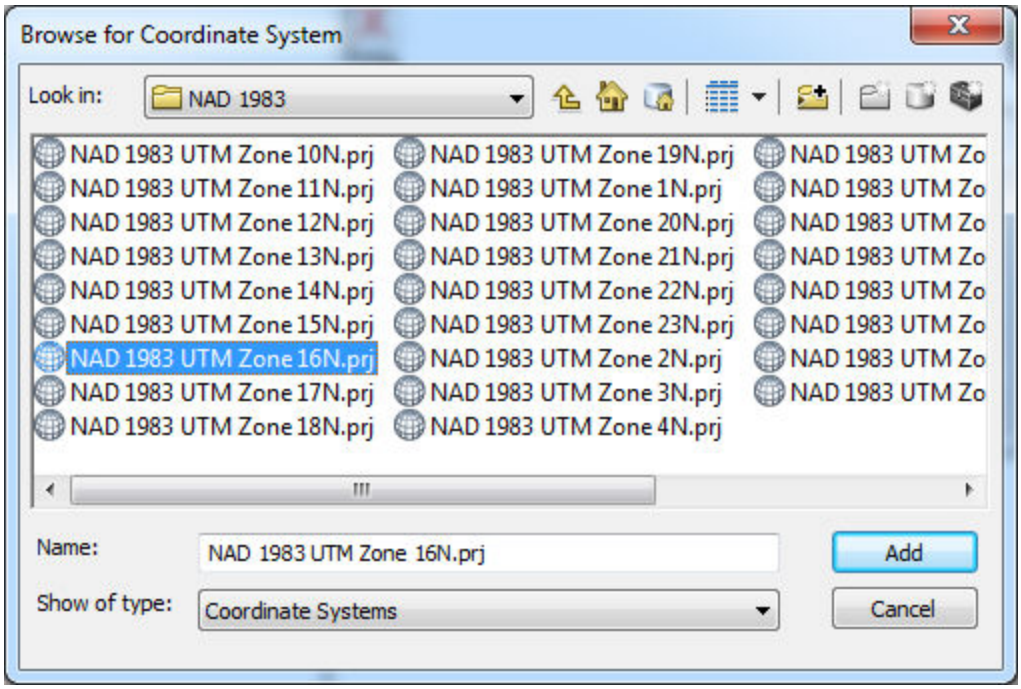

which will then prompt you for the site name.



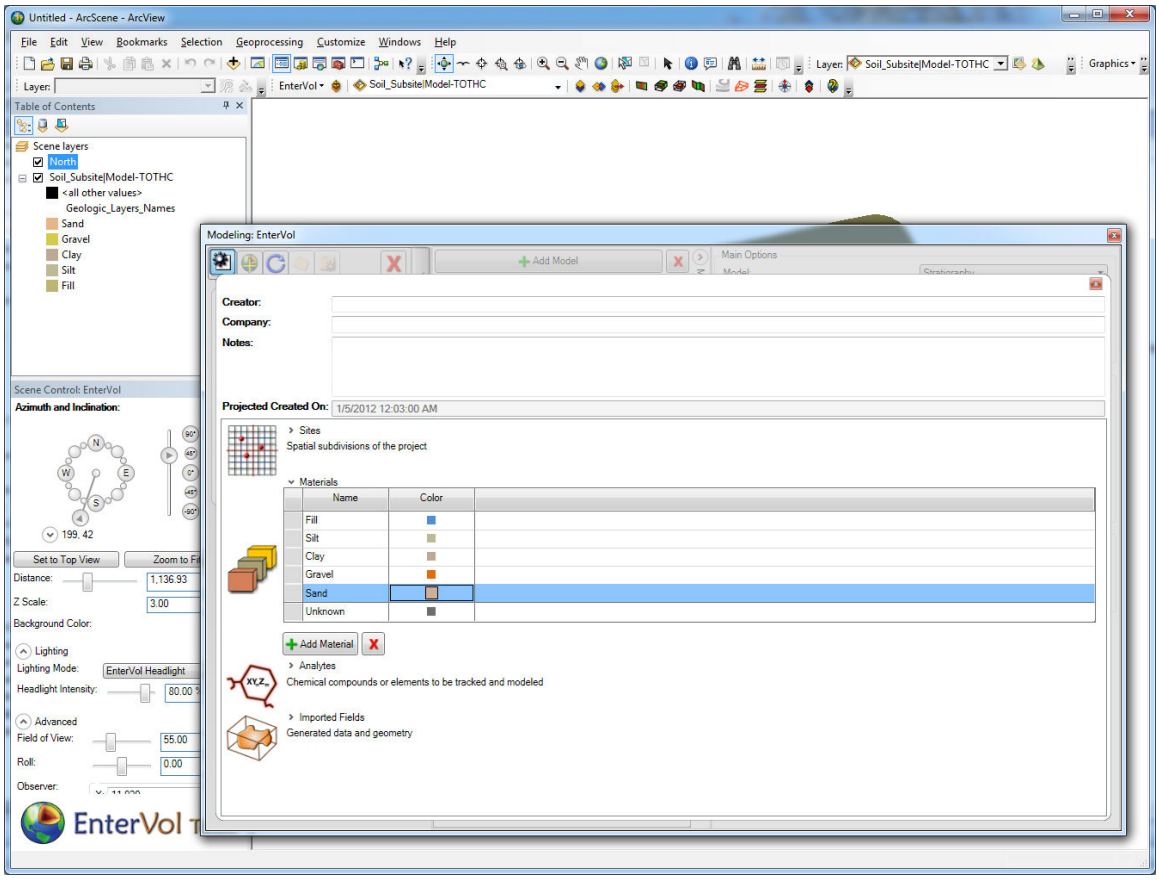

If you know the Project Coordinate System, choose *Projected Coordinates,* and browse for your coordinate system




Choose *Select...*and then browse

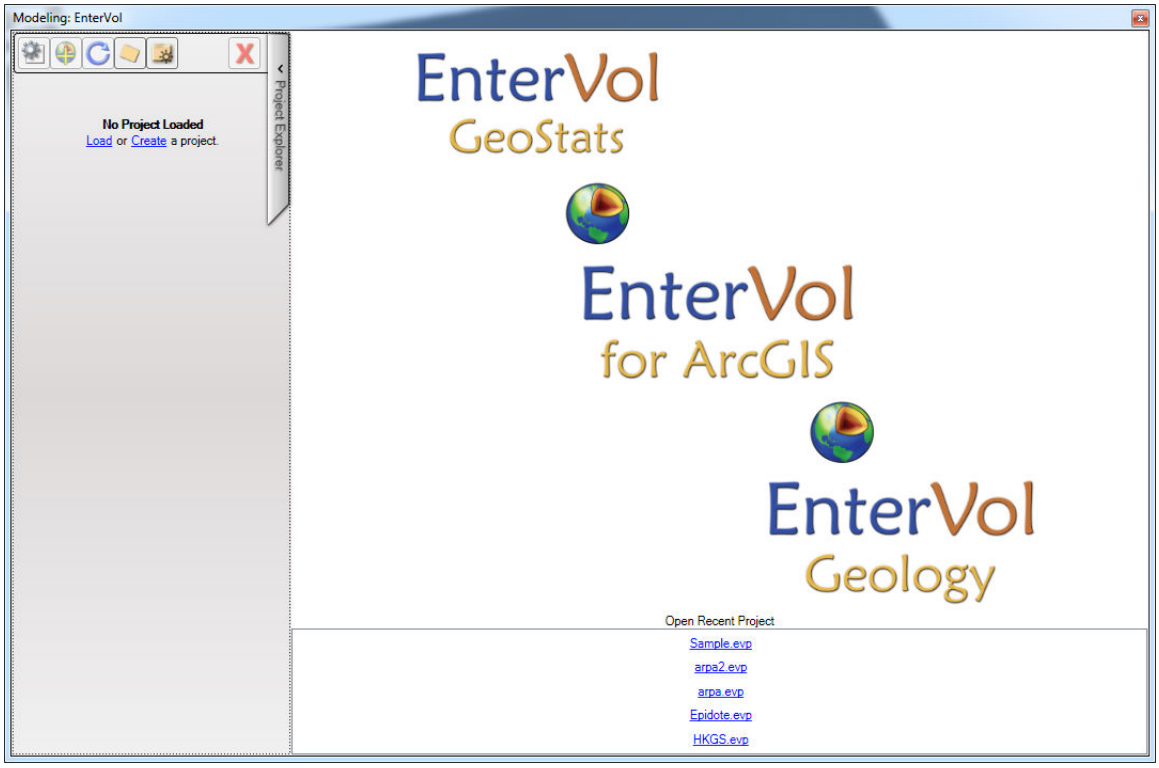



Each project can contain multiple sites, however we will only enter data for a single site for this project. At this point, the Modeling window changes to show the window below. Here you can enter additional information about the project. For example, you can assign colors to Materials which will cause all layers added to the Table of Contents to use these colors.

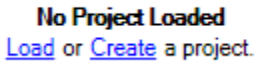



## Load or Create a Project

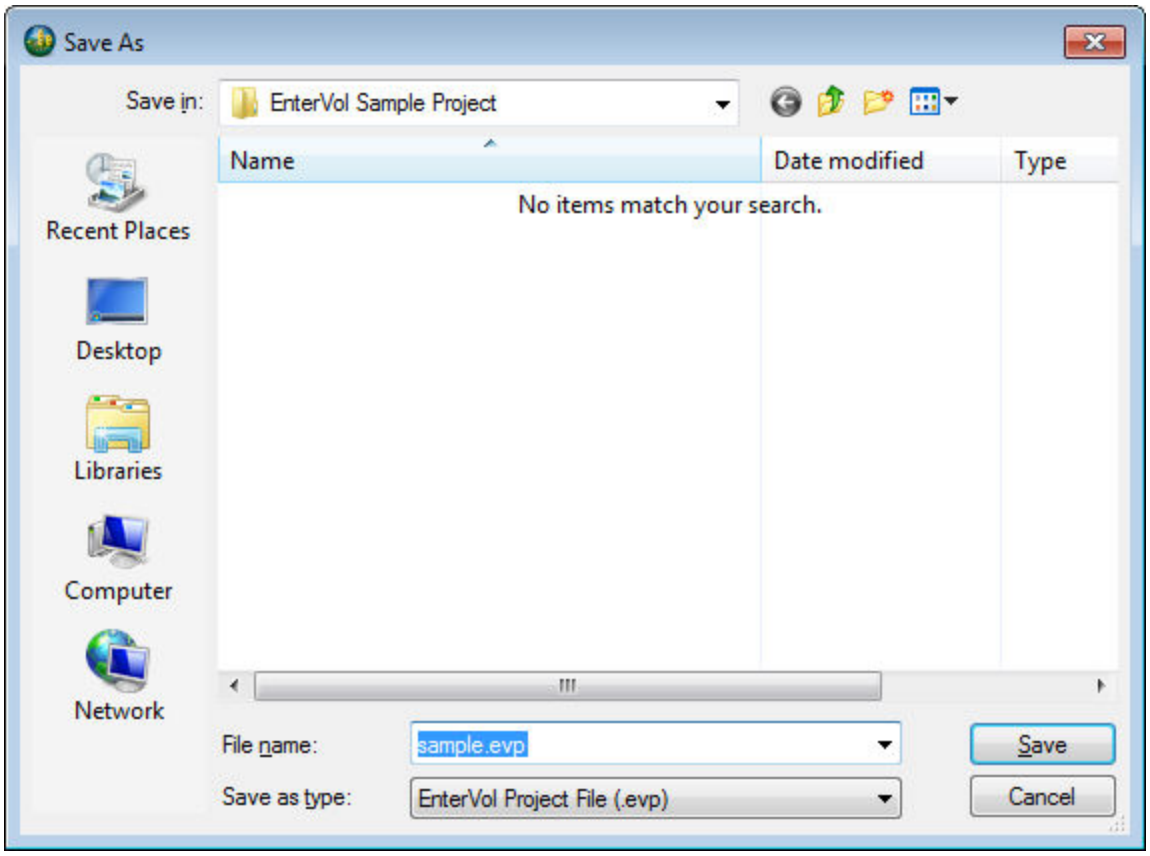
A project is the first step towards creating volumetric models. You can load an existing project (.EVP - EnterVol Project) or create a new project.



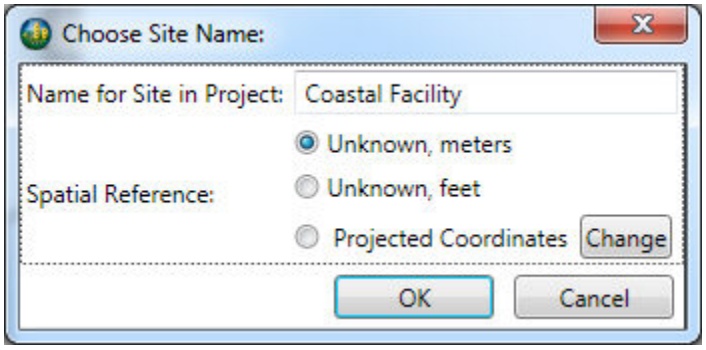

Project files (.evp) are databases into which you load your geologic and analyte data



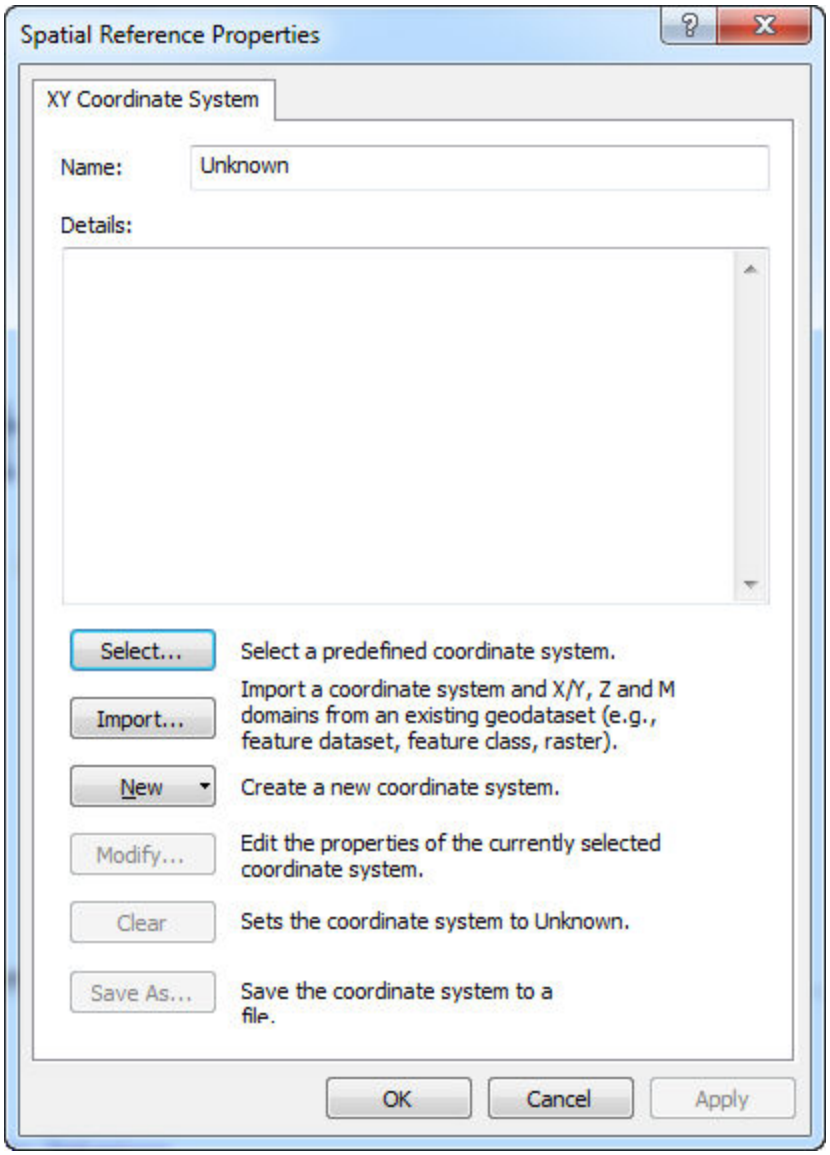

Let's begin by creating a new project:



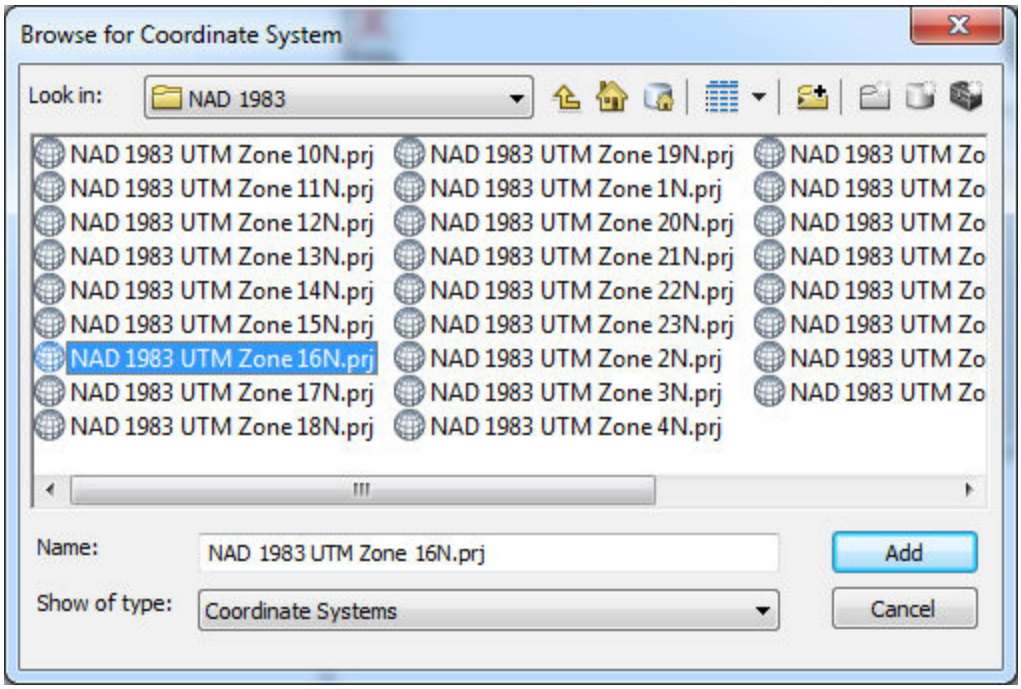

which will then prompt you for the site name.



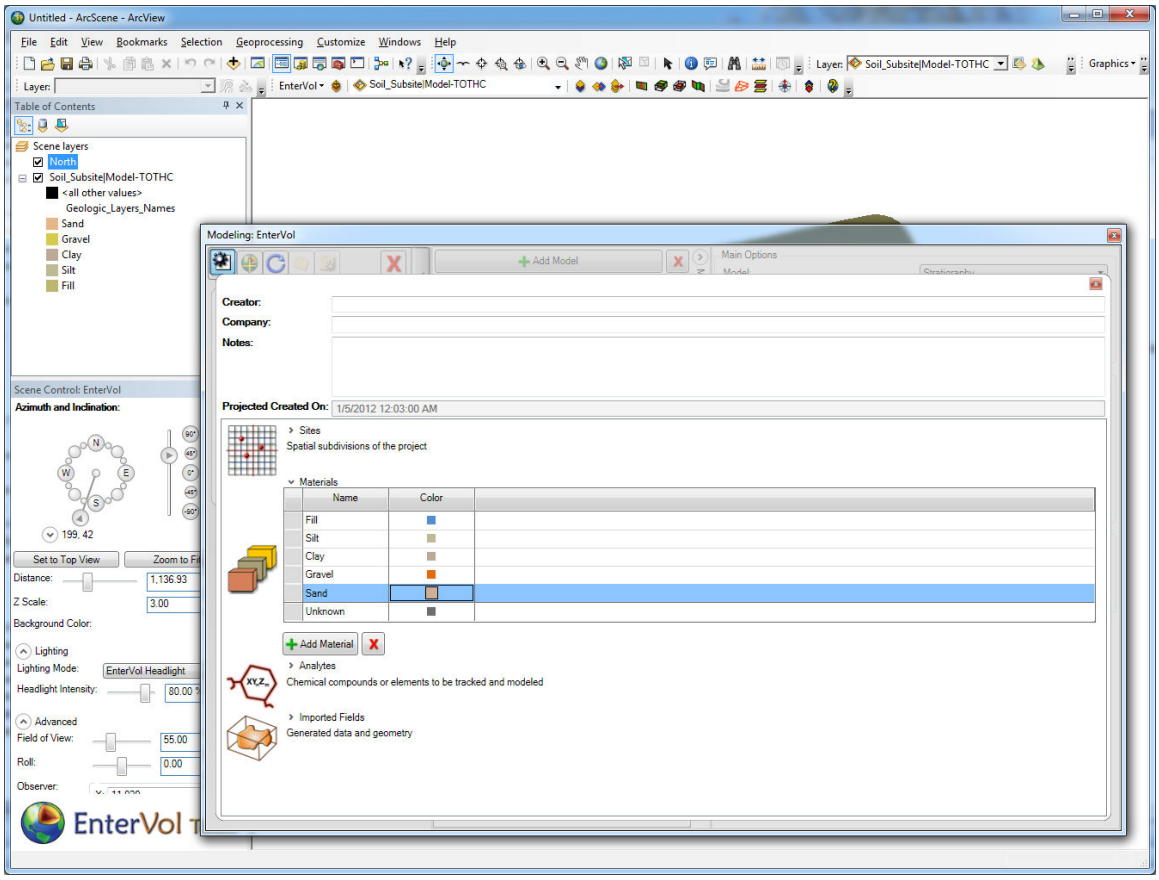

If you know the Project Coordinate System, choose *Projected Coordinates,* and browse for your coordinate system




Choose *Select...*and then browse

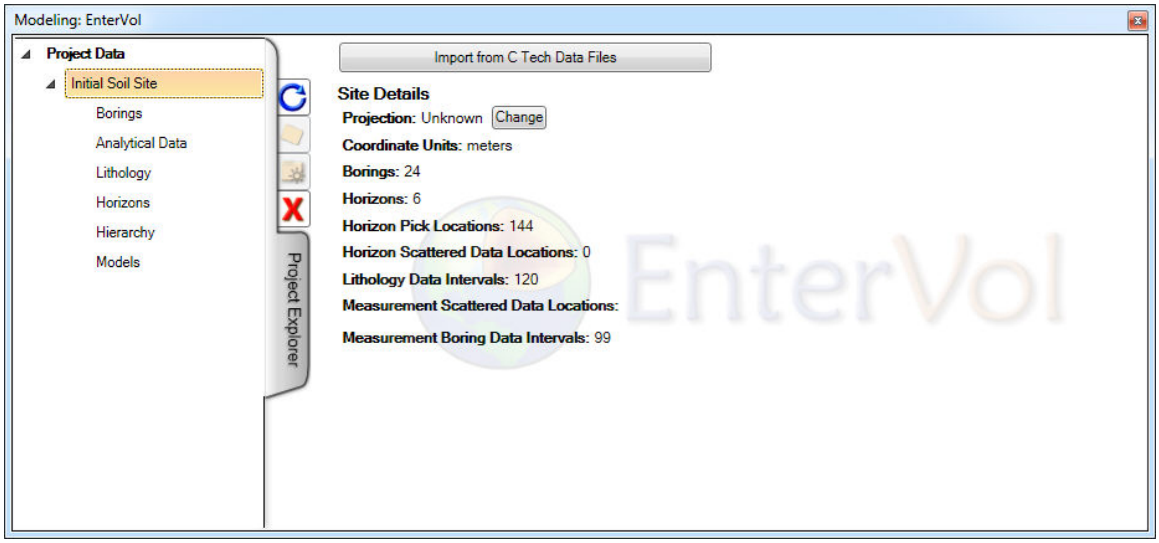



Each project can contain multiple sites, however we will only enter data for a single site for this project. At this point, the Modeling window changes to show the window below. Here you can enter additional information about the project. For example, you can assign colors to Materials which will cause all layers added to the Table of Contents to use these colors.

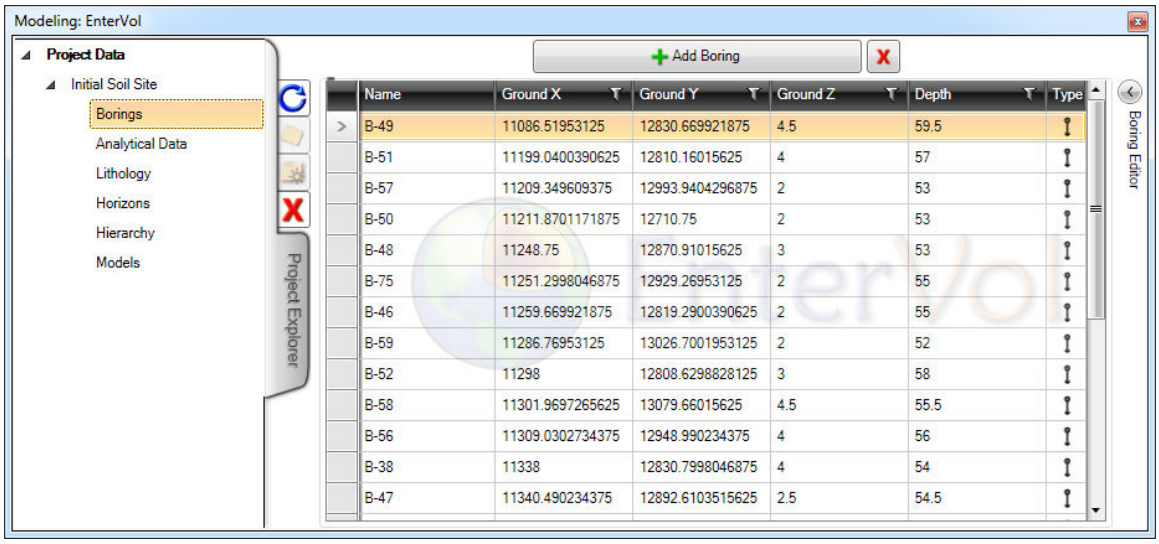



## Site Details and Settings

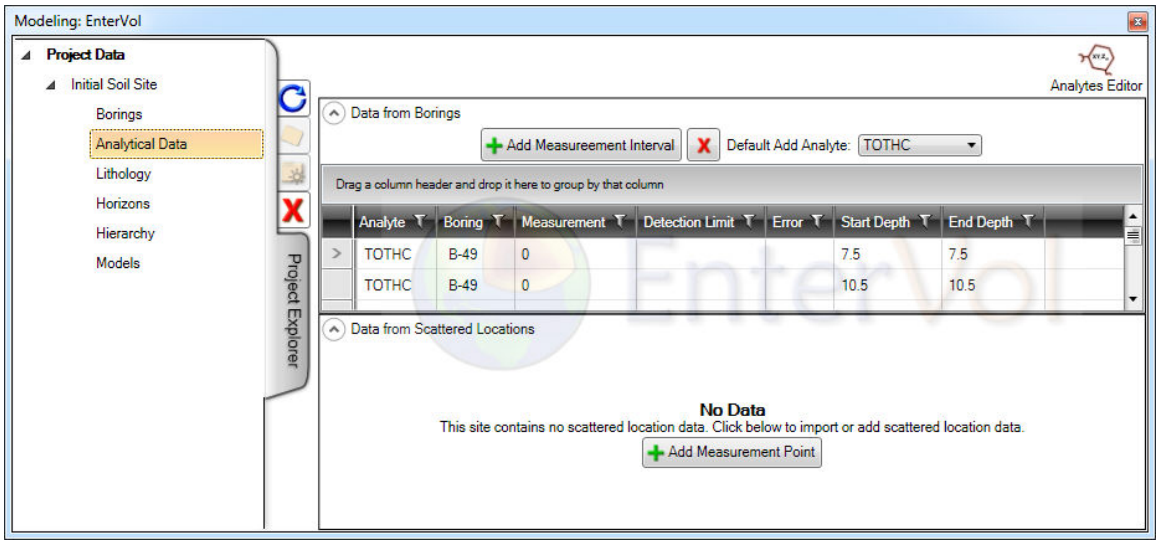
Below your Site are various sections for all site data and models.



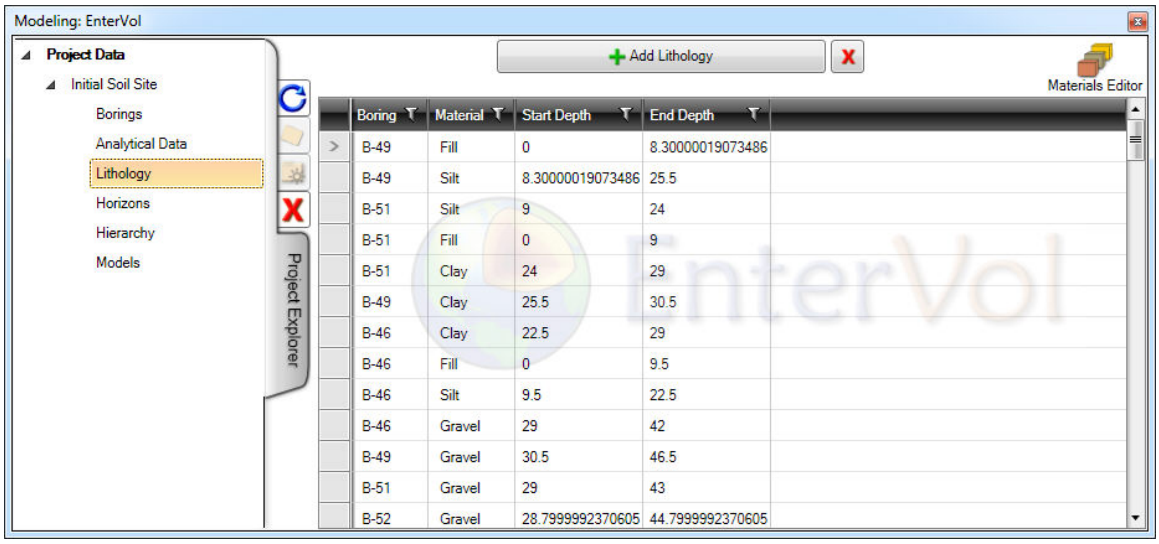

Selecting *Borings* shows all defined borings



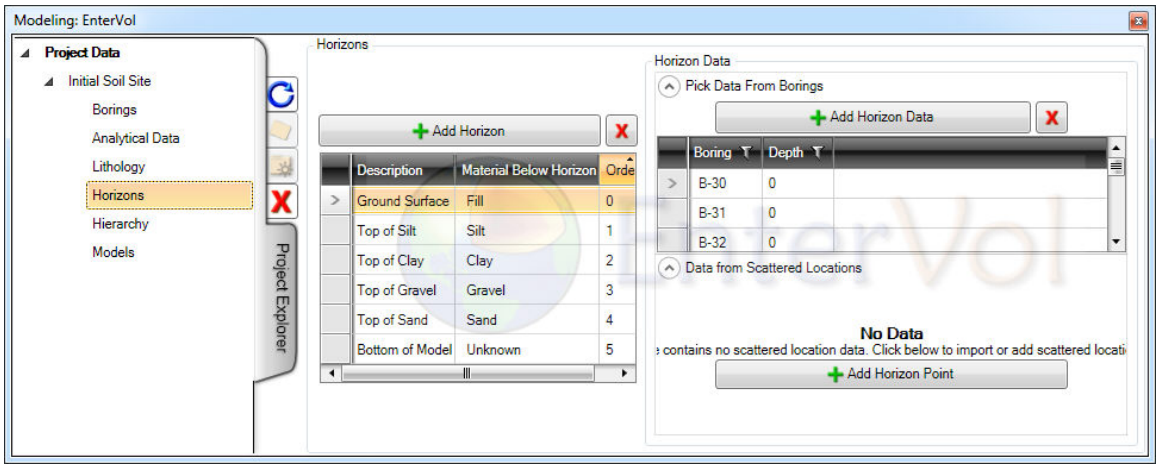

Selecting *Analytical Data* provides and editable spreadsheet style table of all analyte data



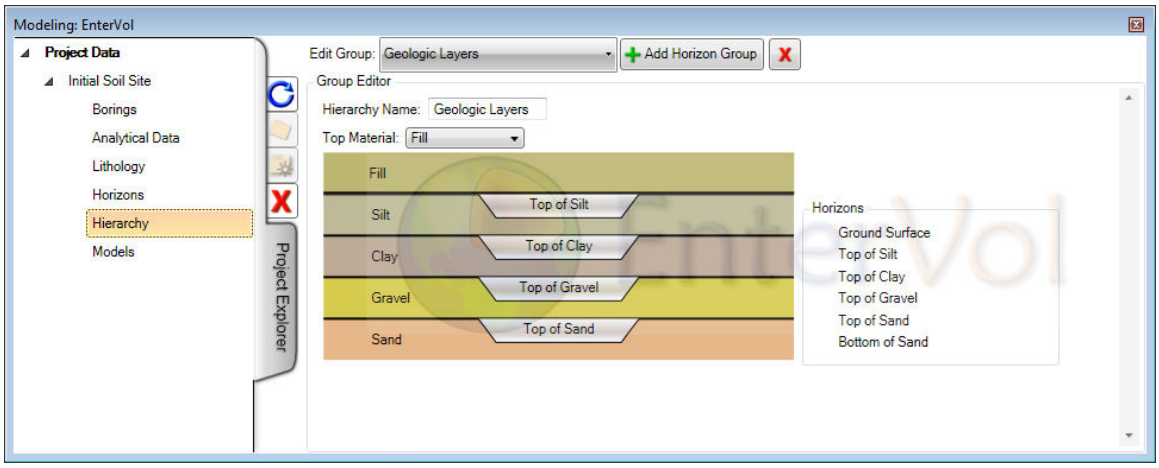

*Lithology* section shows each boring log interval




The *Horizons* section lists all surfaces used to define stratigraphy and all horizon picks from borings.




The *Hierarchy* section lists the stratigraphic layers in order and all horizons used to define stratigraphy. By selecting the horizons in the display below, you can move the order of horizons or eliminate layers in the hierarchy. Removing a horizon eliminates the layer below it.




In the example above, removing the horizon "Top of Clay" eliminates clay from the hierarchy resulting in a Silt layer that continues to the Top of Gravel.

The Models section is discussed in the Concepts and Modeling Process topic.

## Concepts and Database Structure

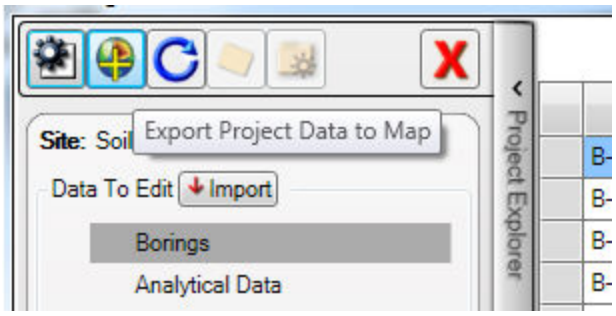
Data is stored in the Project database using one of two data structures:

Data which is collected from borings is stored based on the way that data is collected:

- Borings are defined based upon:
  - The X-Y-Z coordinate of the top of the boring collar
  - A sequence of boring interval definitions, each having Dip, Dip Direction, and Length
- Data, whether it is analytical (e.g. assay, contamination, etc.), lithologic, geophysical, or stratigraphy horizons is stored based upon the *Depth* and *Boring ID*.
  - *Depth* is defined as the distance down the boring, regardless of boring orientation with respect to ground surface or the horizon.

Data which is not collected from borings must have X-Y-Z coordinates and a measurement for each sample. Multiple analytes measured at the same location are each treated as unique data samples.

In the topics which follow the methods for importing data for direct display are shown. However, any data which is imported into the modeling database can be displayed by selecting the "Export Project Data to Map" button.




## Concepts and Database Structure

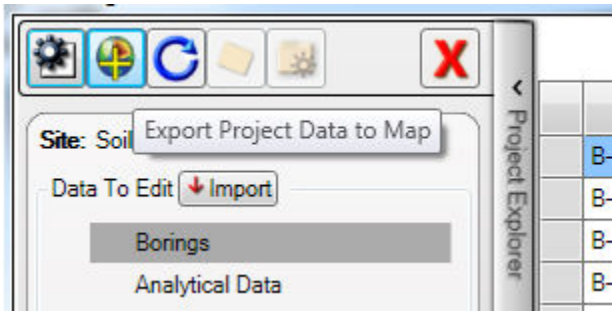
Data is stored in the Project database using one of two data structures:

Data which is collected from borings is stored based on the way that data is collected:

- Borings are defined based upon:
  - The X-Y-Z coordinate of the top of the boring collar
  - A sequence of boring interval definitions, each having Dip, Dip Direction, and Length
- Data, whether it is analytical (e.g. assay, contamination, etc.), lithologic, geophysical, or stratigraphy horizons is stored based upon the *Depth* and *Boring ID*.
  - *Depth* is defined as the distance down the boring, regardless of boring orientation with respect to ground surface or the horizon.

Data which is not collected from borings must have X-Y-Z coordinates and a measurement for each sample. Multiple analytes measured at the same location are each treated as unique data samples.

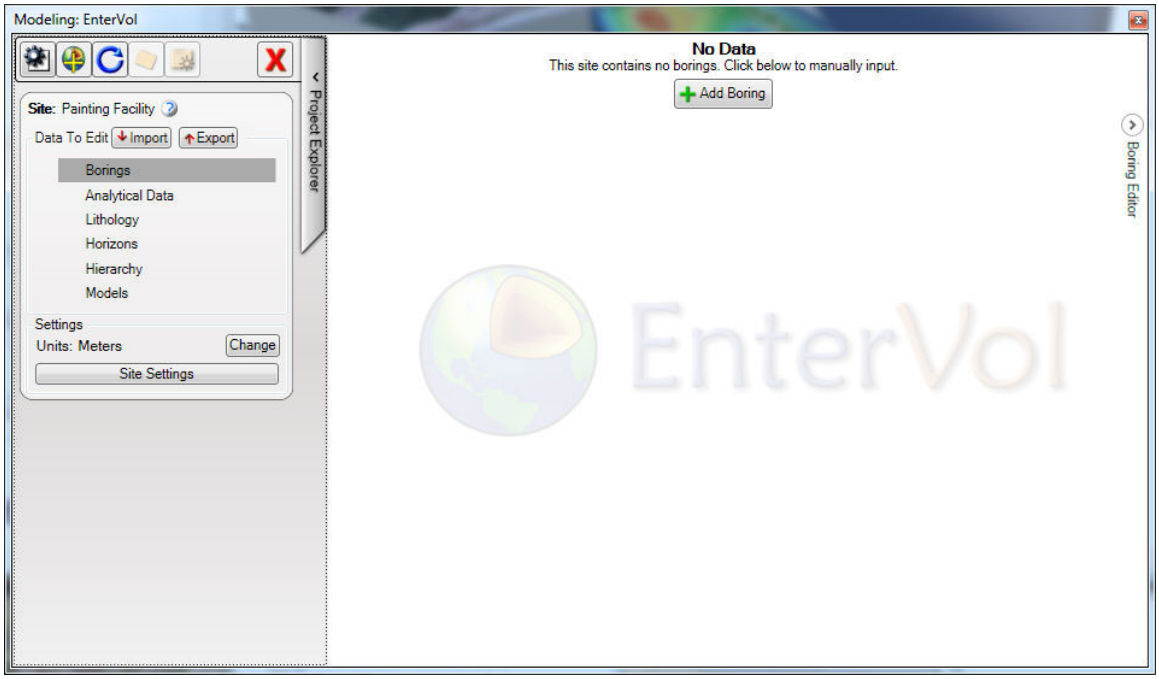
In the topics which follow the methods for importing data for direct display are shown. However, any data which is imported into the modeling database can be displayed by selecting the "Export Project Data to Map" button.



## Importing Data

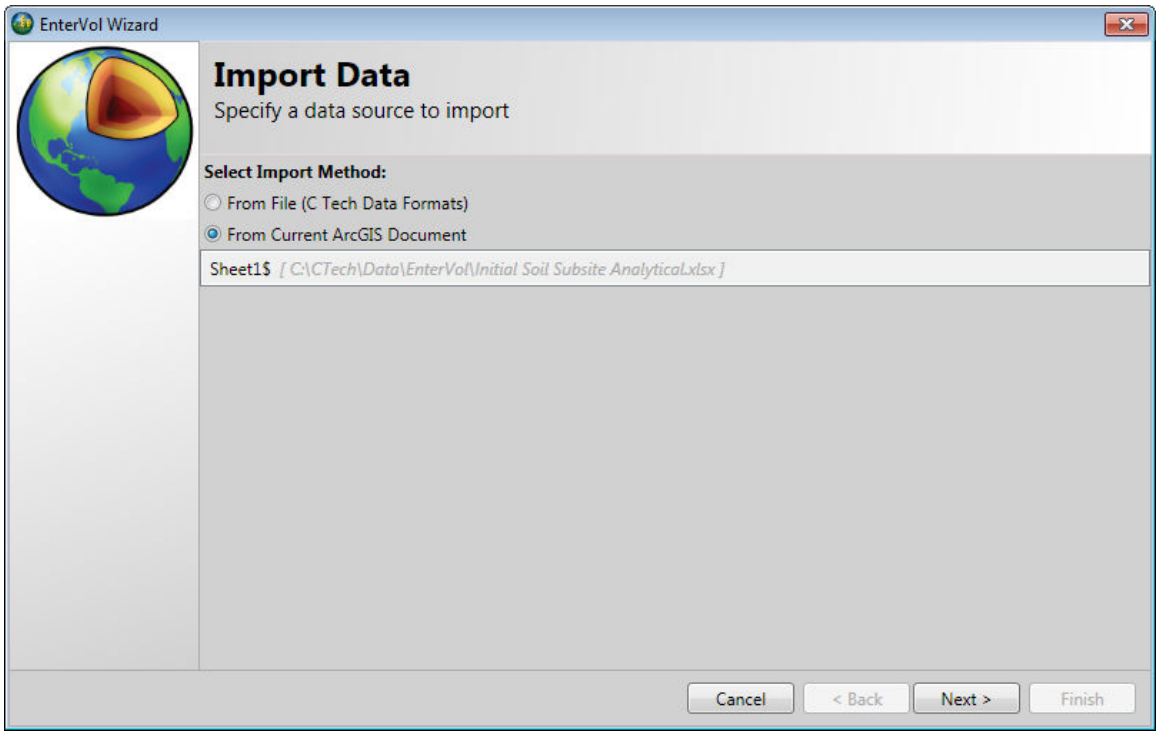
Click on the add data buttons for each data type to enter new records into any of the database categories. Below we will demonstrate importing Borings data from a table. The following procedure is applicable to:

- Borings
- Analytical Data
- Lithology Data
- Stratigraphic Data

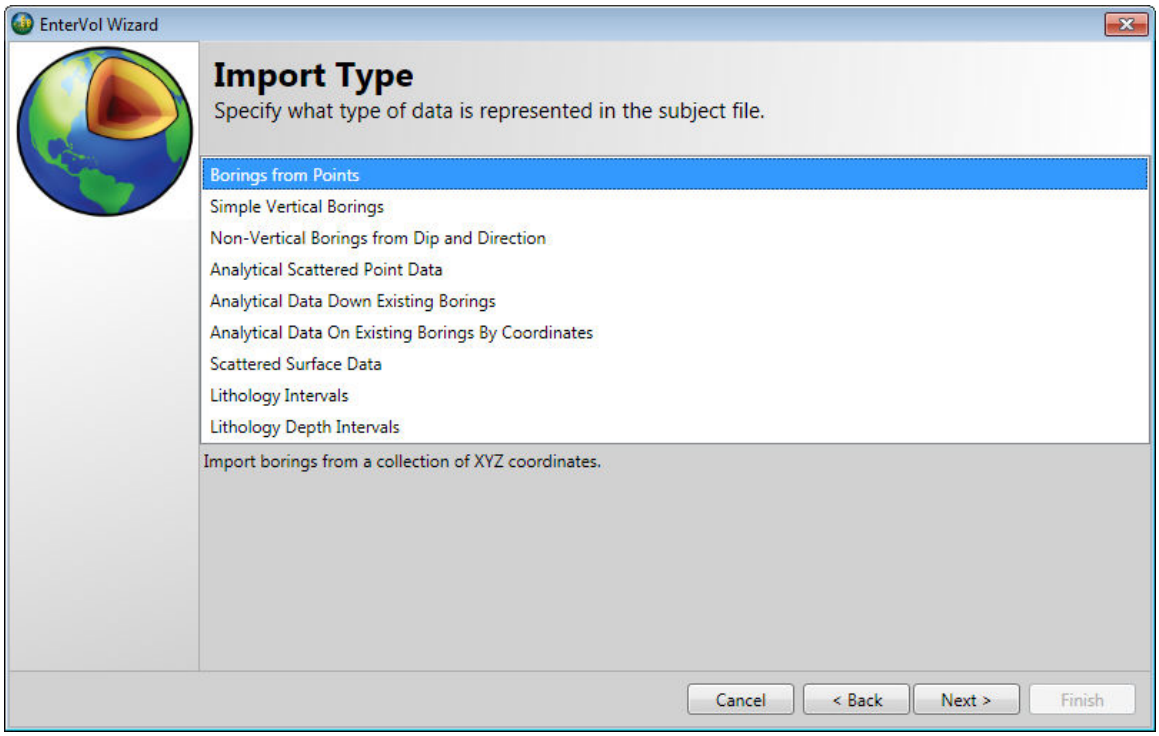


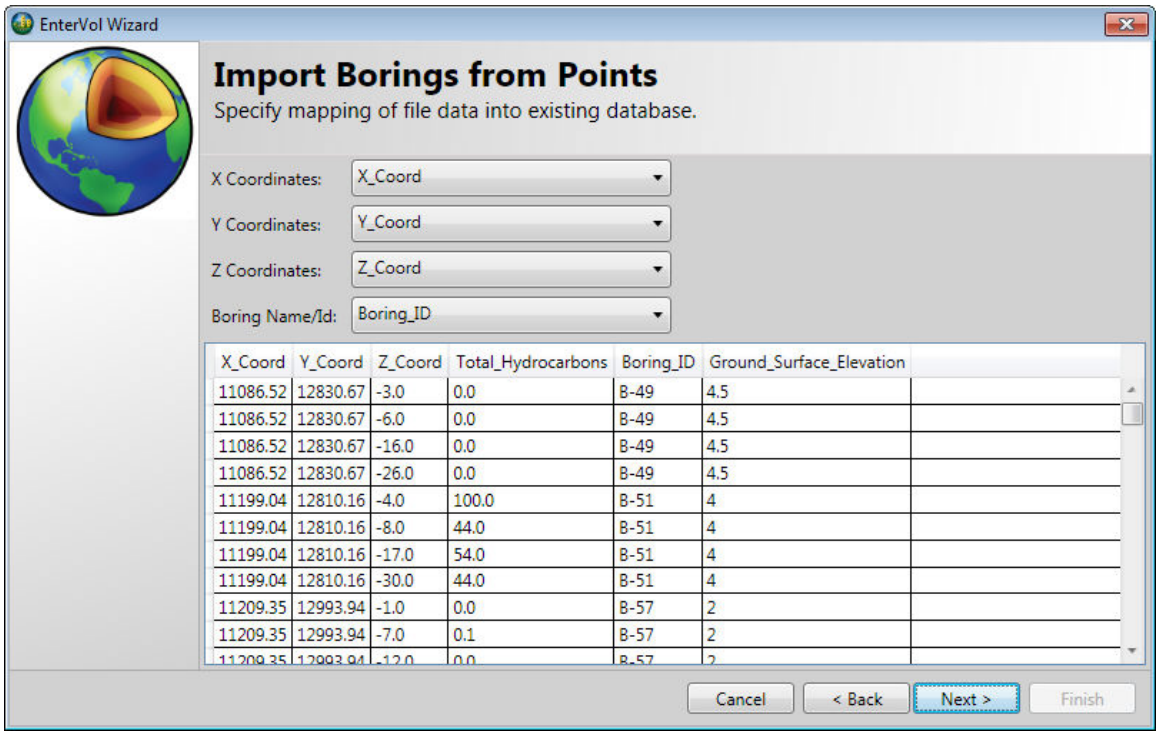

We can enter new borings data. If the boring is a single vertical interval, it can be fully characterized by the ground X-Y-Z coordinates, the total depth and the Boring Name.

Upon clicking the Import button, we have the option to read any C Tech ASCII data file type or use any feature class or table in your current scene or map. Below we will choose an Excel spreadsheet from our scene.

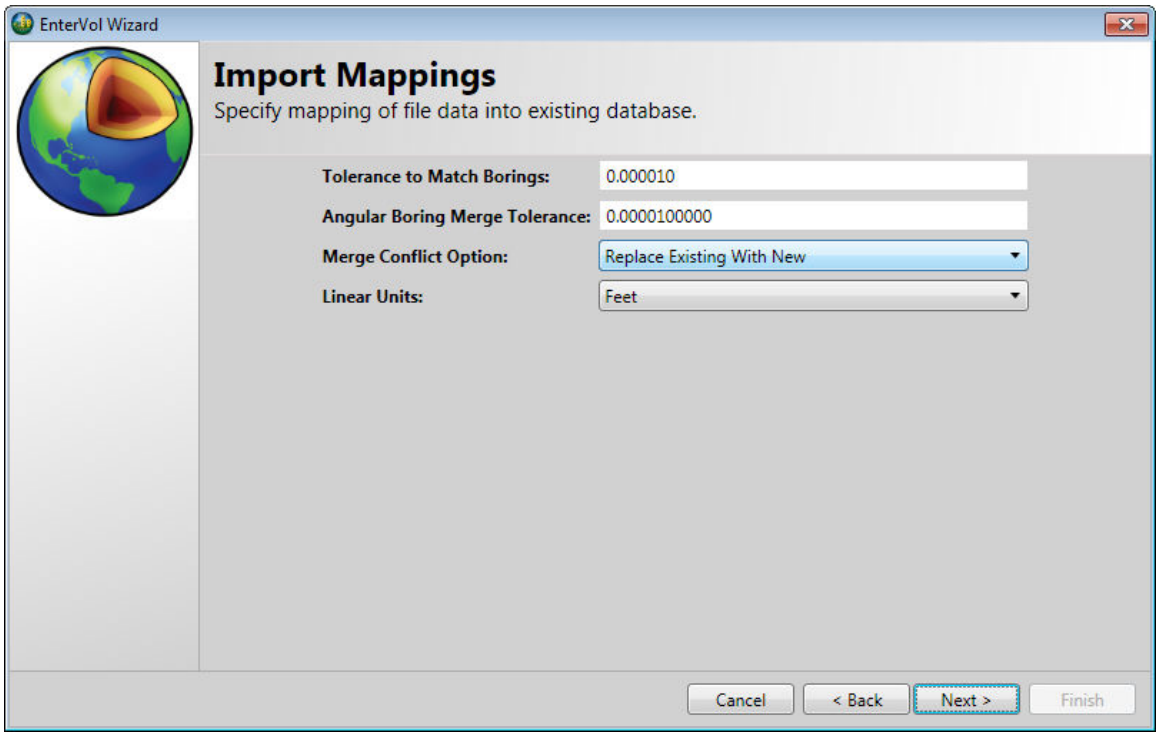



Next we must specify what type of information is in the table selected. We will be choosing *Borings from Points*

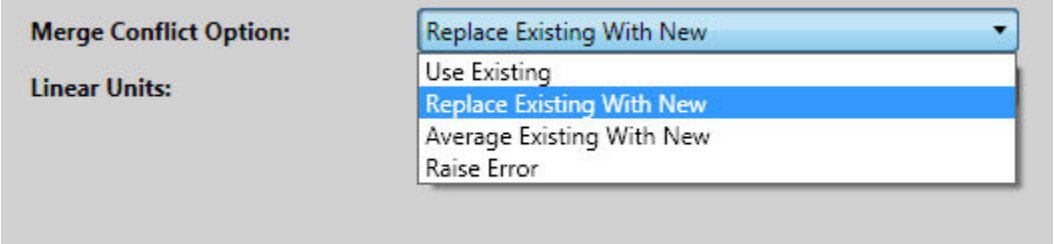




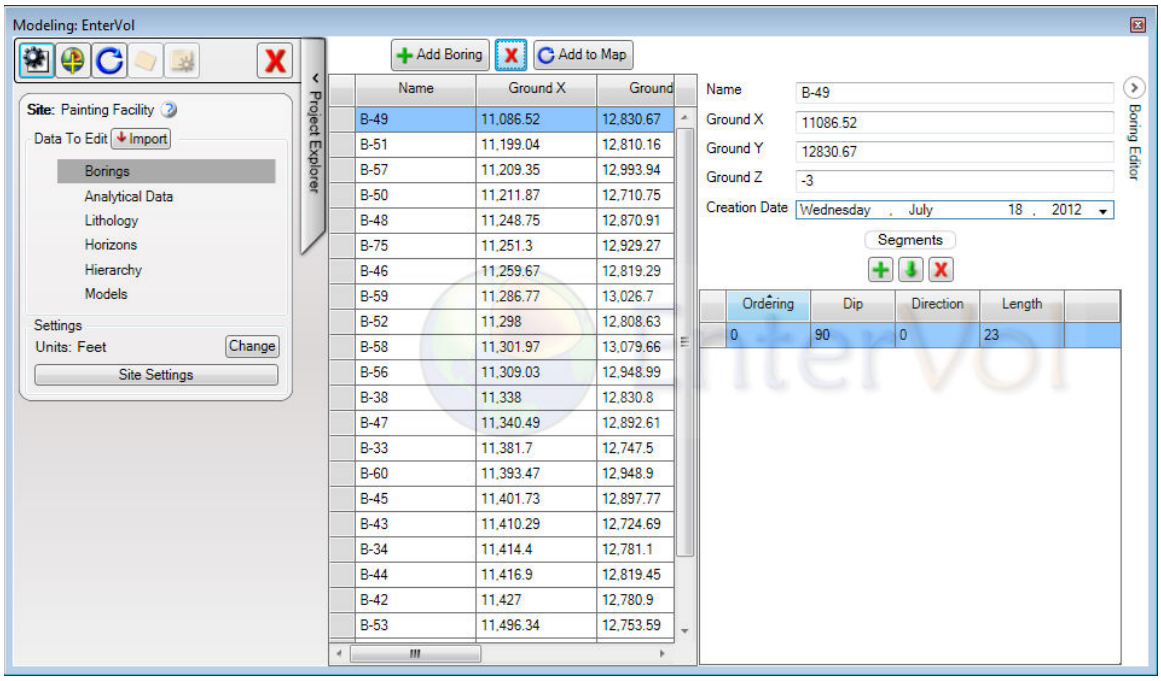

The Tolerance options allow you to automatically merge data that is already in your database.



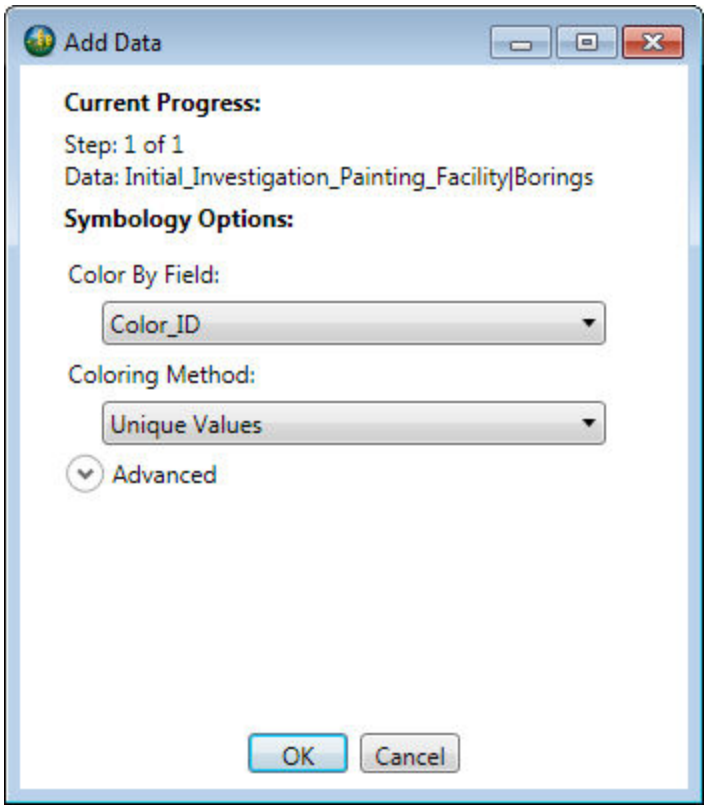
You have options when tolerance conditions are met. These include replacing data, disclosing errors or averaging.

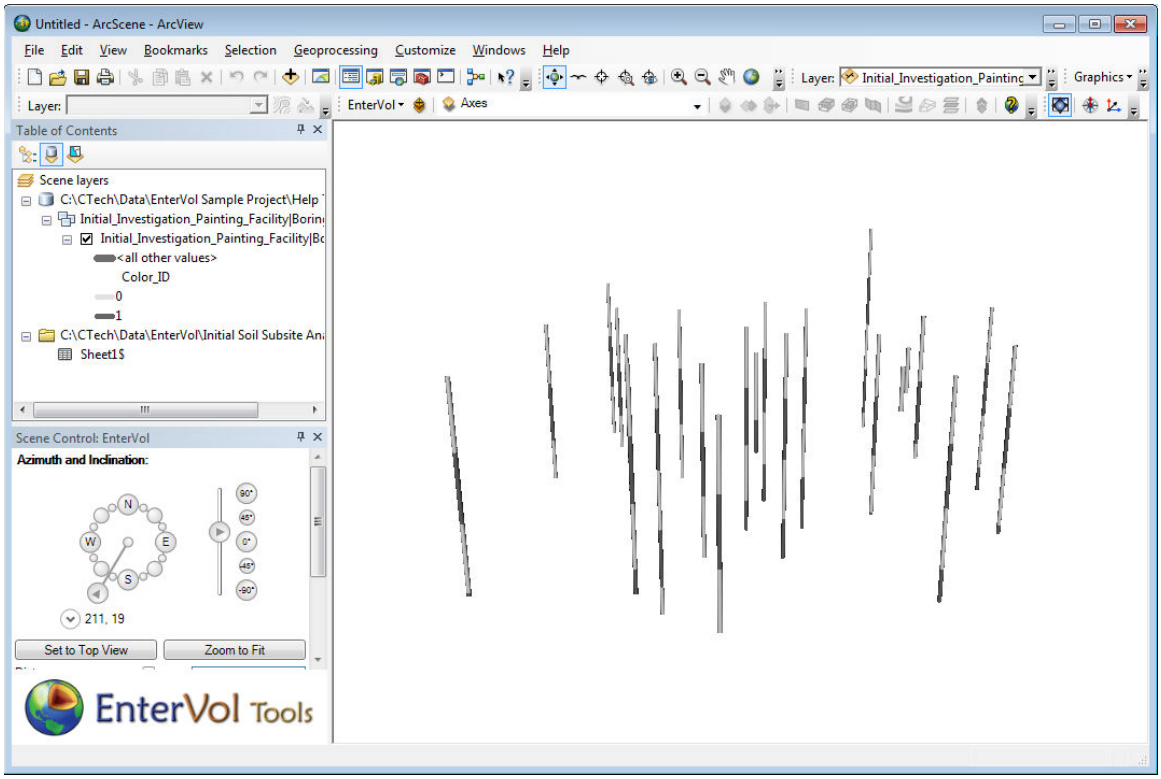



If units in this table or file are different, we will perform conversions automatically.



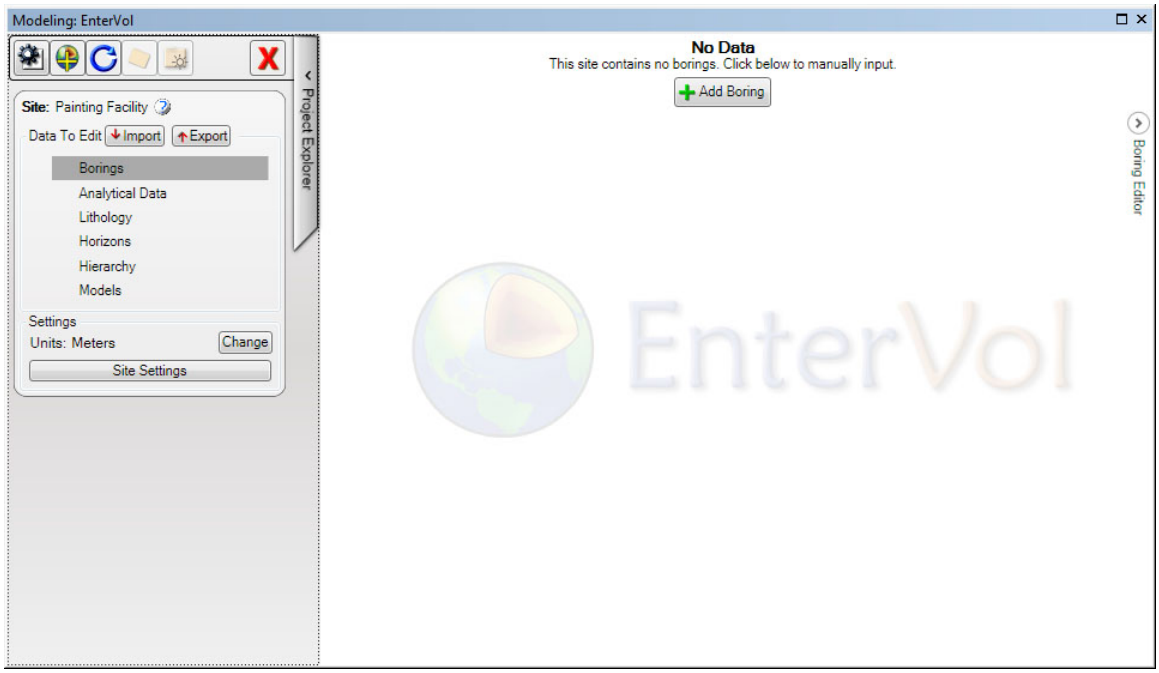

Now that the data has been imported, clicking "Add to Map" will import the data into your scene or map.


Add Data
Current Progress:
Step: 1 of 1
Data: Initial_Investigation_Painting_Facility|Borings
Symbology Options:
Color By Field:
Color_ID
Coloring Method:
Unique Values
Advanced
OK
Cancel


Untitled - ArcScene - ArcView
File Edit View Bookmarks Selection Geoprocessing Customize Windows Help
Table of Contents
Scene layers
Color_ID
Scene Control: EnterVol
Azimuth and Inclination:
Set to Top View
Zoom to Fit
EnterVol Tools

## Importing Analyte Data from C Tech Files

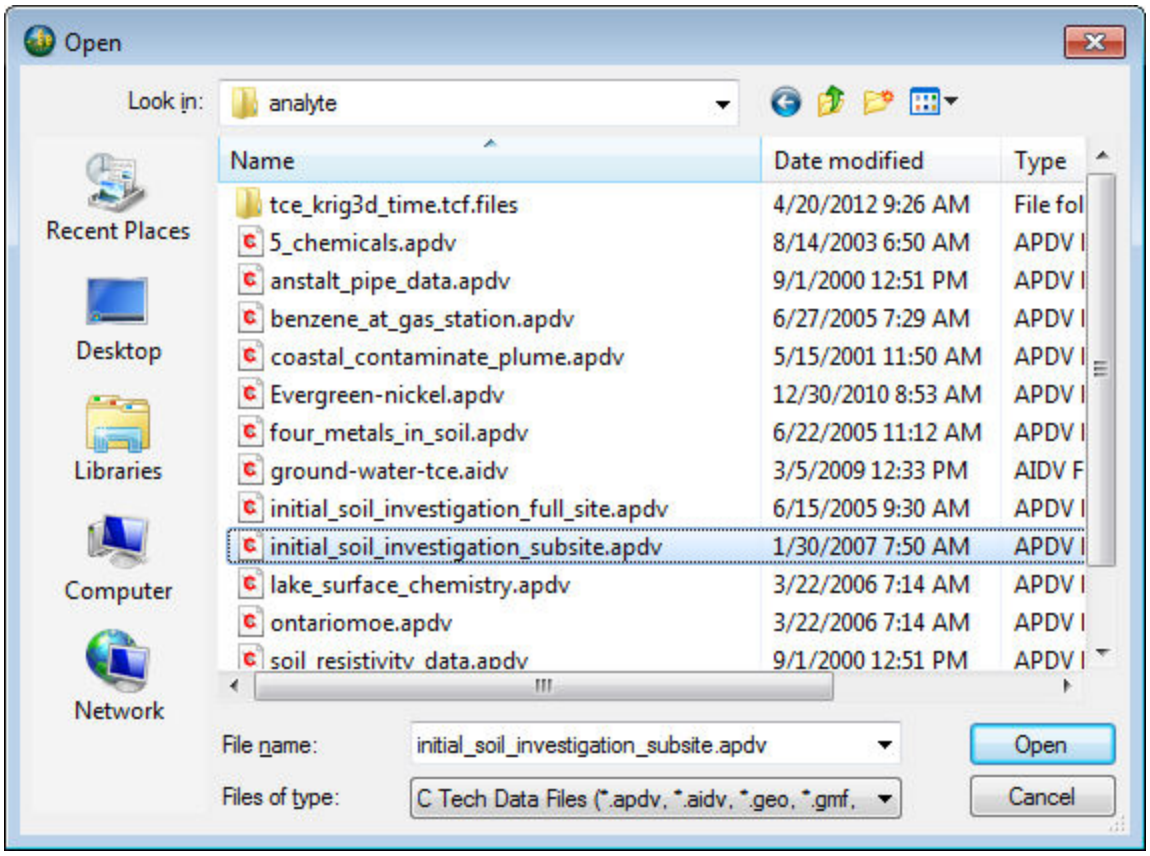
Click on the Import button



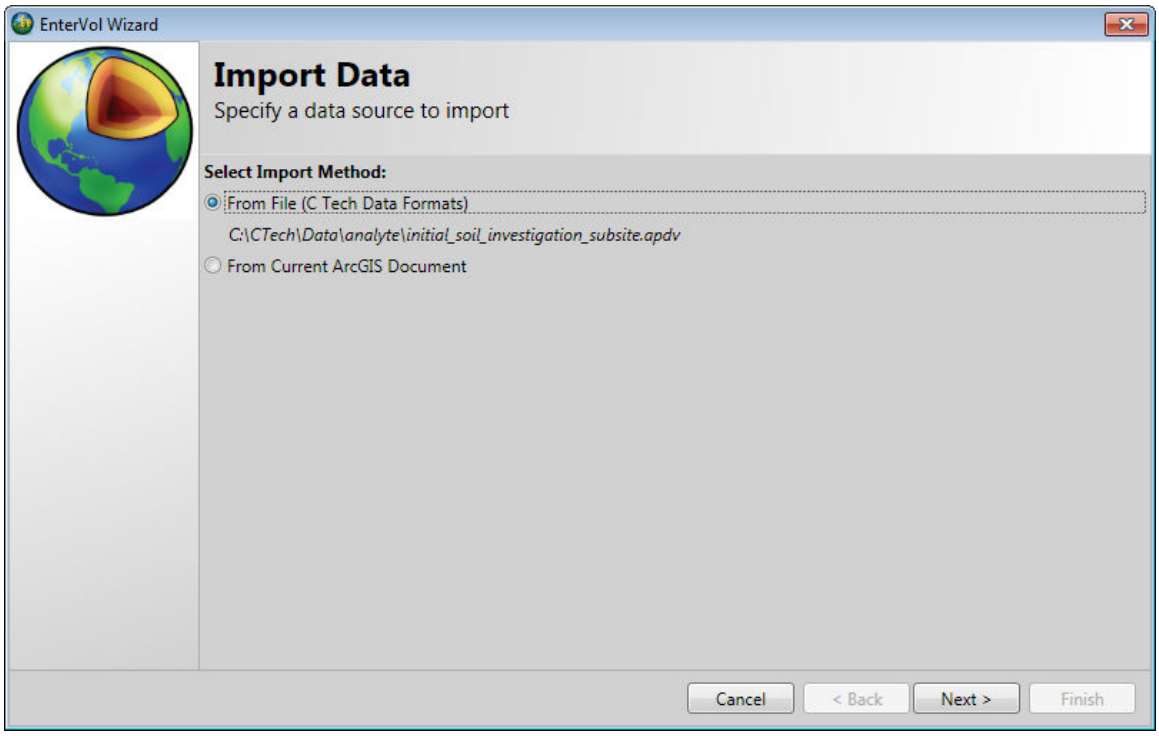

Click on the Import button to open a browser to import analytical or geologic data. The five file types supported are:

1. .APDV: Analyte data measured at a point
2. .AIDV: Analyte data measured over an interval
3. .GEO: Stratigraphy data collected from vertical borings
4. .GMF: Stratigraphy data collected at scattered locations on each surface
5. .PGF: Lithology (boring log) data

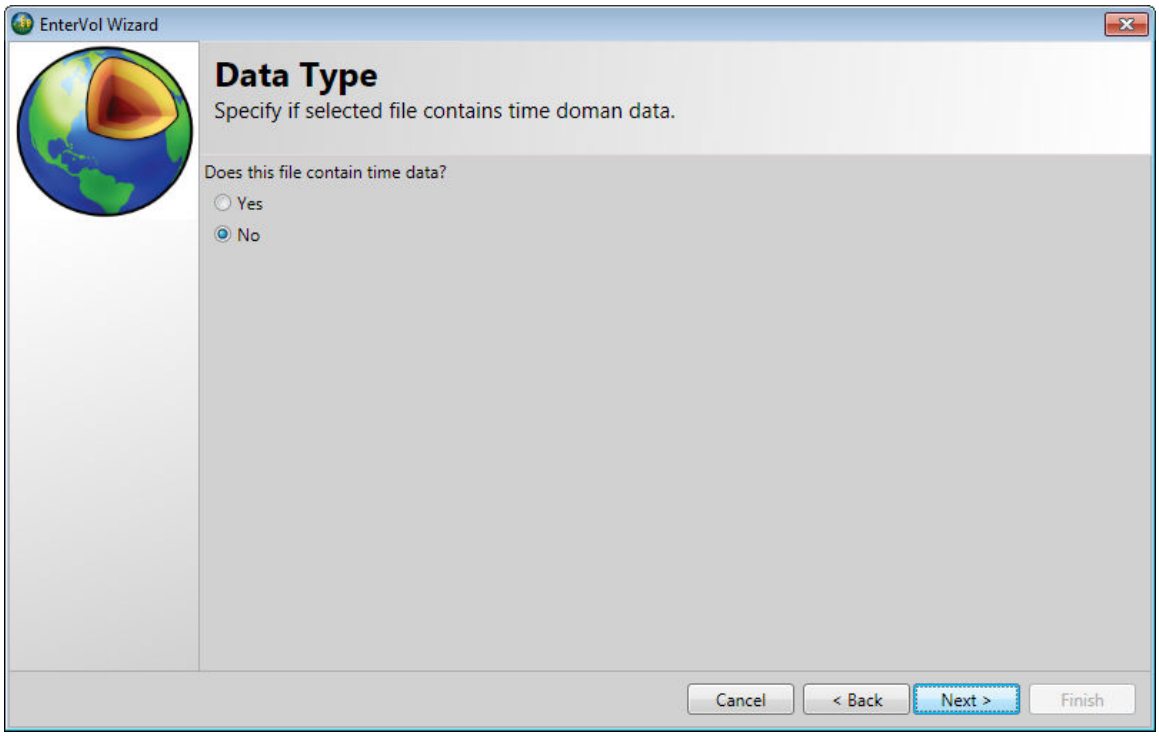
Selecting *From File* will open a browser.




Let's select initial_soil_investigation_subsite.apdv which is Total Hydrocarbon (TOTHC) soil contamination data from a site in California.

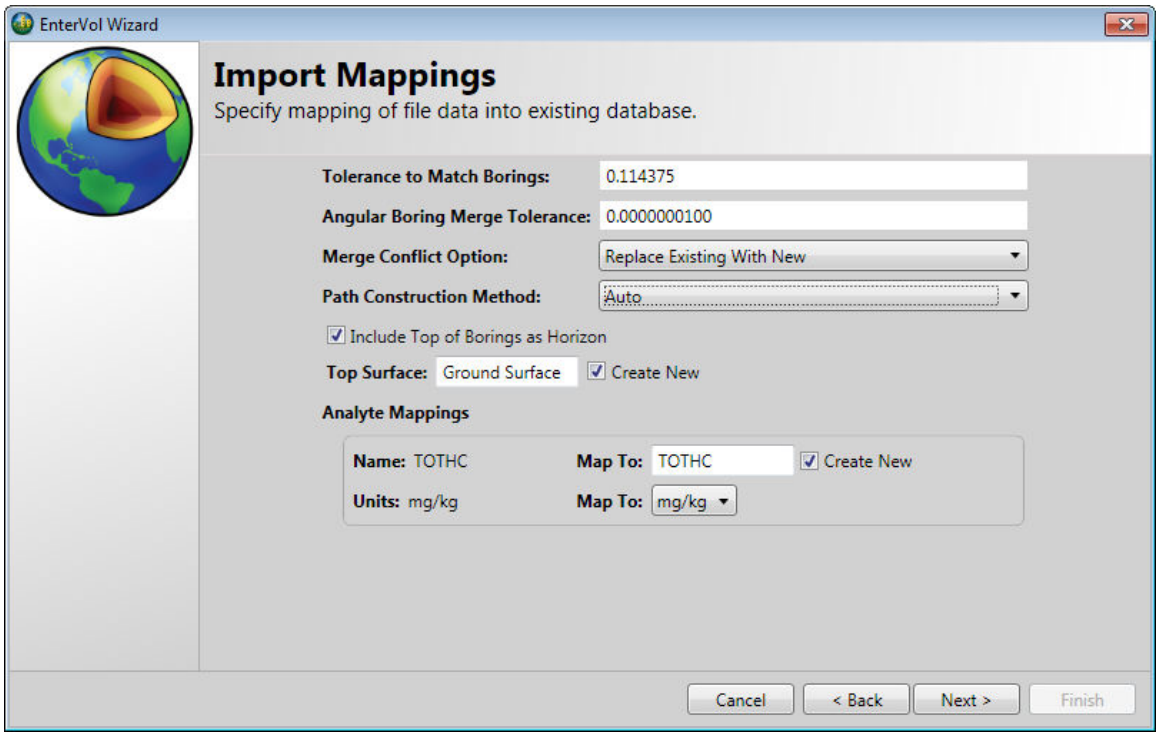



After selecting an analyte file, the following window will open. If your data file represents data measured at different times, select **Yes** at which point you would choose the analyte which is associated with the time states.

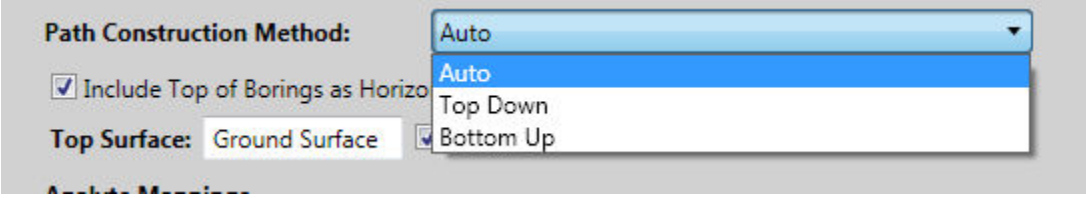


If your data is not measured at different times, just press Next to continue.

The next window allows you to confirm your data units and whether the tops of the borings specified in your .apdv or .aidv file should be used as a horizon surface (such as the ground surface).

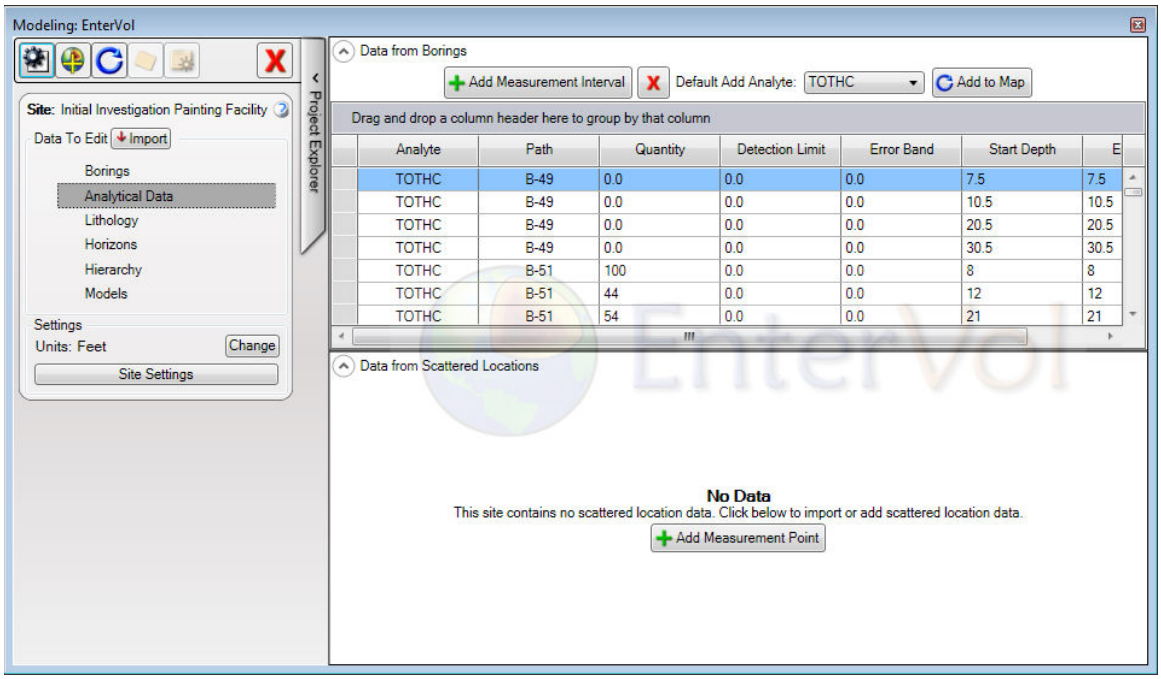



The Tolerances are the same as with [importing borings]

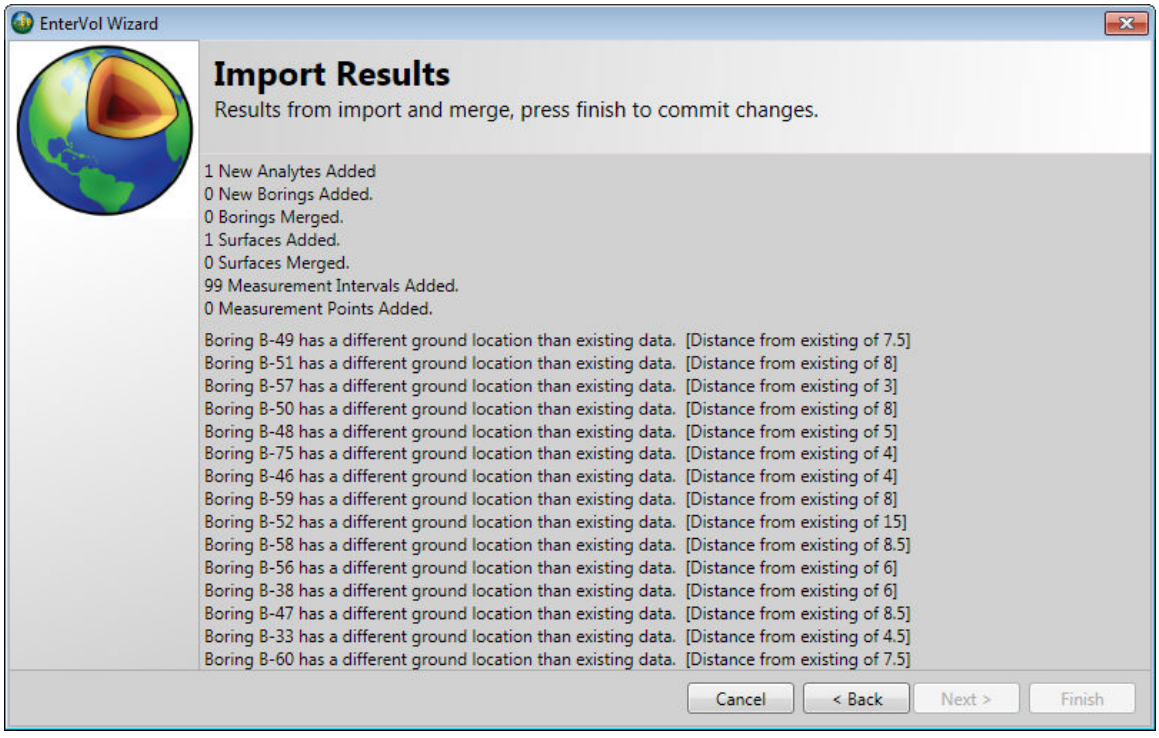



Path Construction Method allows you to force borings to be sorted and defined *Top Down* or *Bottom Up*

When data collected in borings is imported, the samples and the borings data is separately entered into the database. As shown below, the file we've imported represents 23 borings and 99 TOTHC samples.




If your data is inconsistent, upon importing, the wizard will detect the errors and report something like the following:

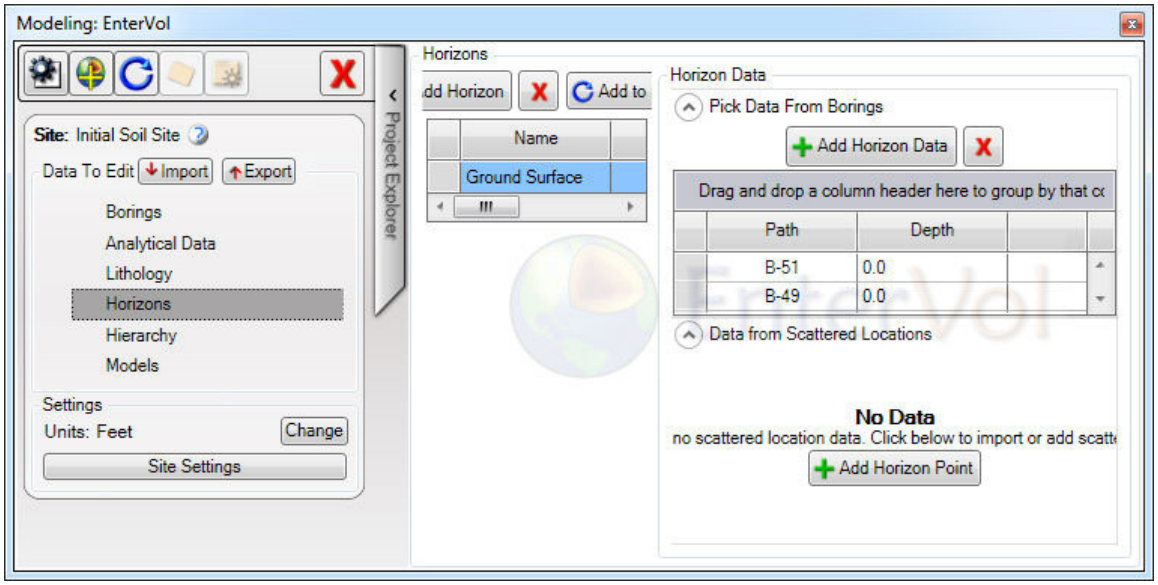


## Importing Stratigraphic Data from C Tech Files

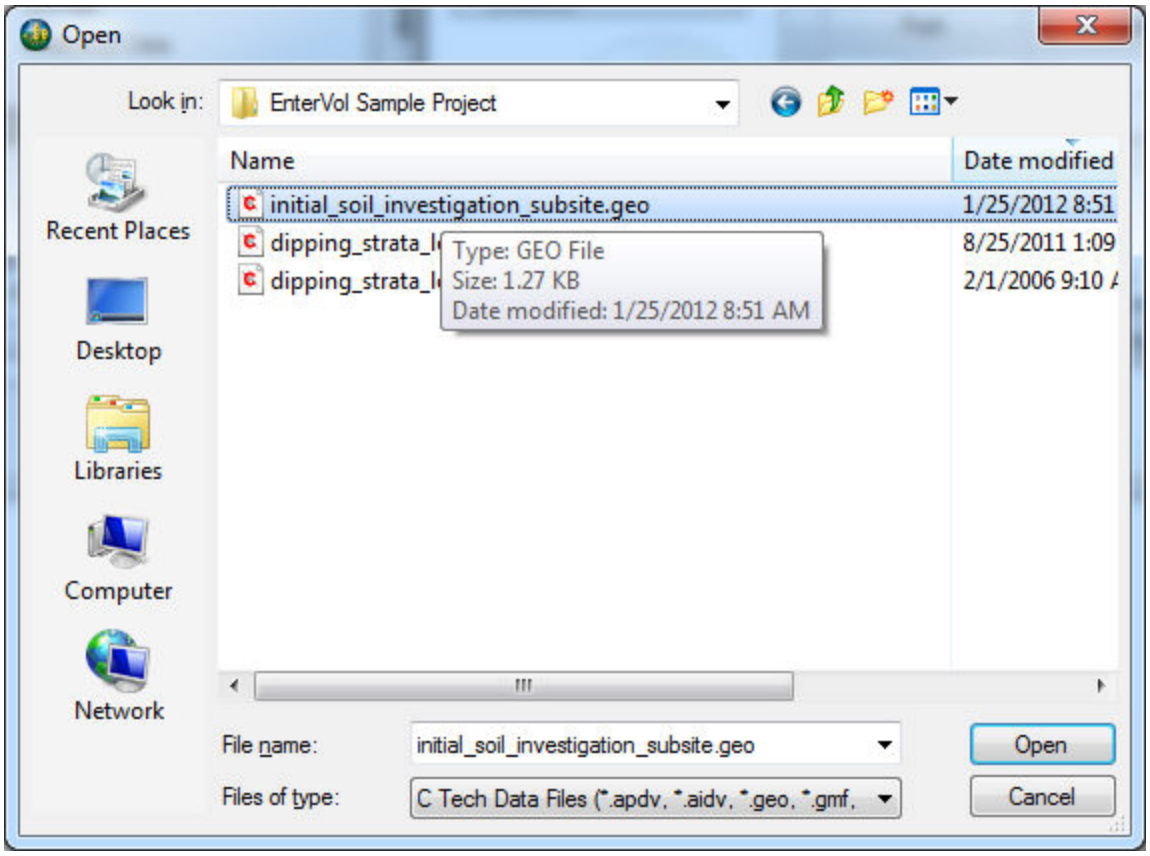
Stratigraphy data is information about geologic surfaces (horizons) which form the tops and bottoms of stratigraphic units which comprise a geologic hierarchy.

C Tech has two ASCII file formats which define stratigraphic data: .GEO and .GMF formats. Normally, you would import one or the other. This is especially true if both file contain the same basic information. However, an exception would be if you'd imported a .GEO file and then also had a high resolution topographic surface. In this case, the GMF will be added to the total set of stratigraphic data allowing you to create a more refined model.

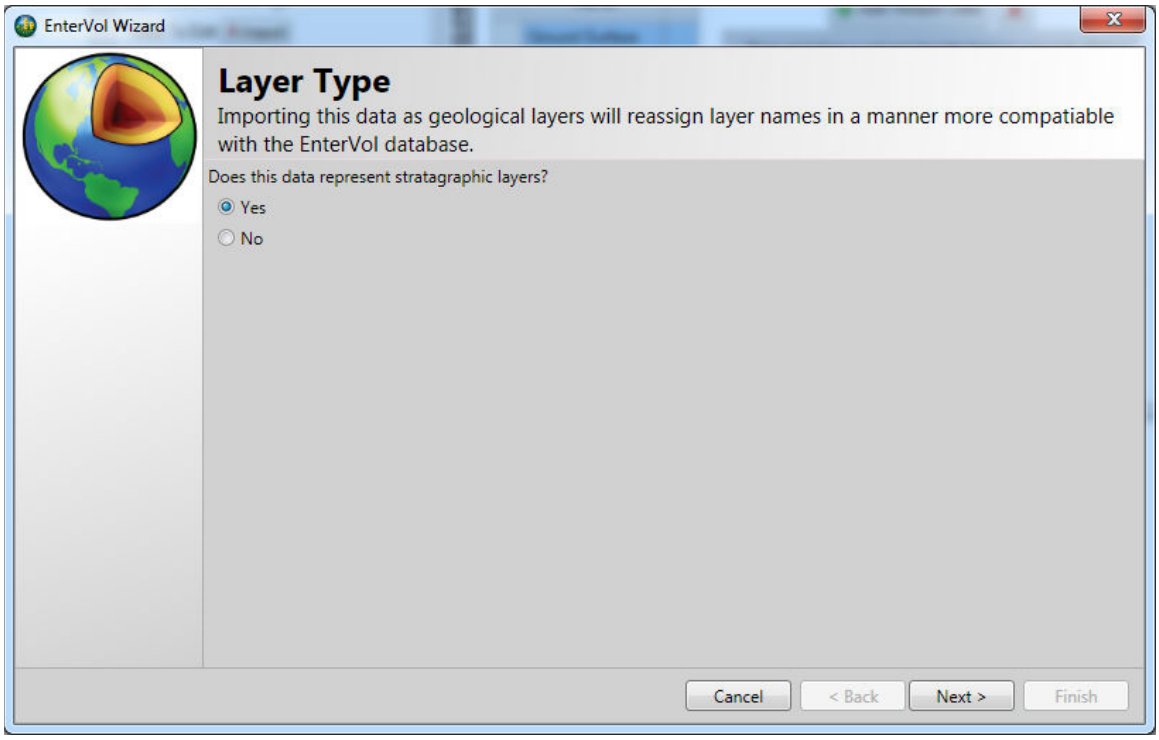
Select the site and click on the Import Button



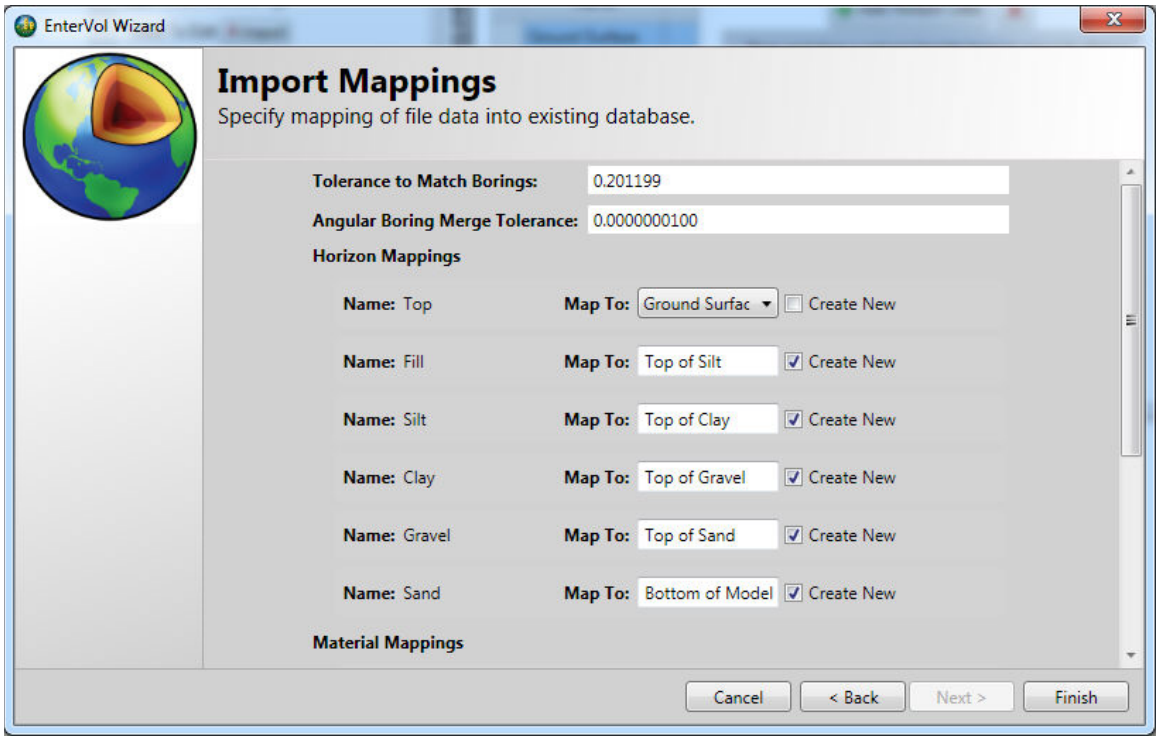

Browse to find a .GEO or .GMF file which contains stratigraphic data.




After selecting the .GEO file above, you are prompted whether the file contains stratigraphic layers. This same type of file can also contain other surface data such as time domain water table elevations, seismic horizons, etc.



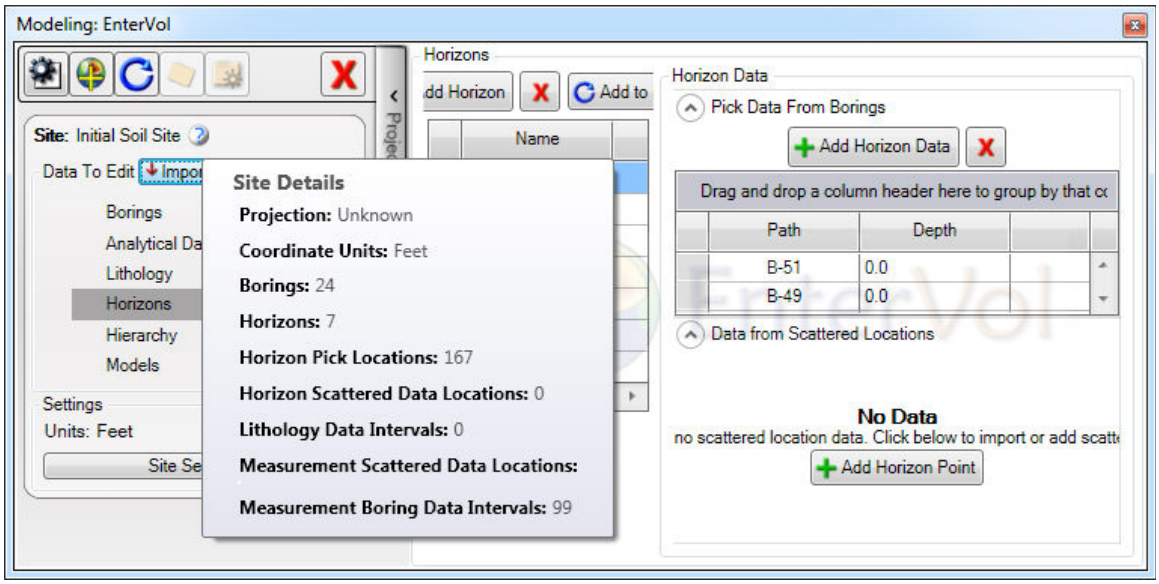

For our example, select Yes. Another window will appear which will have the Horizons and materials renamed to better fit the stratigraphic hierarchy represented by the data.



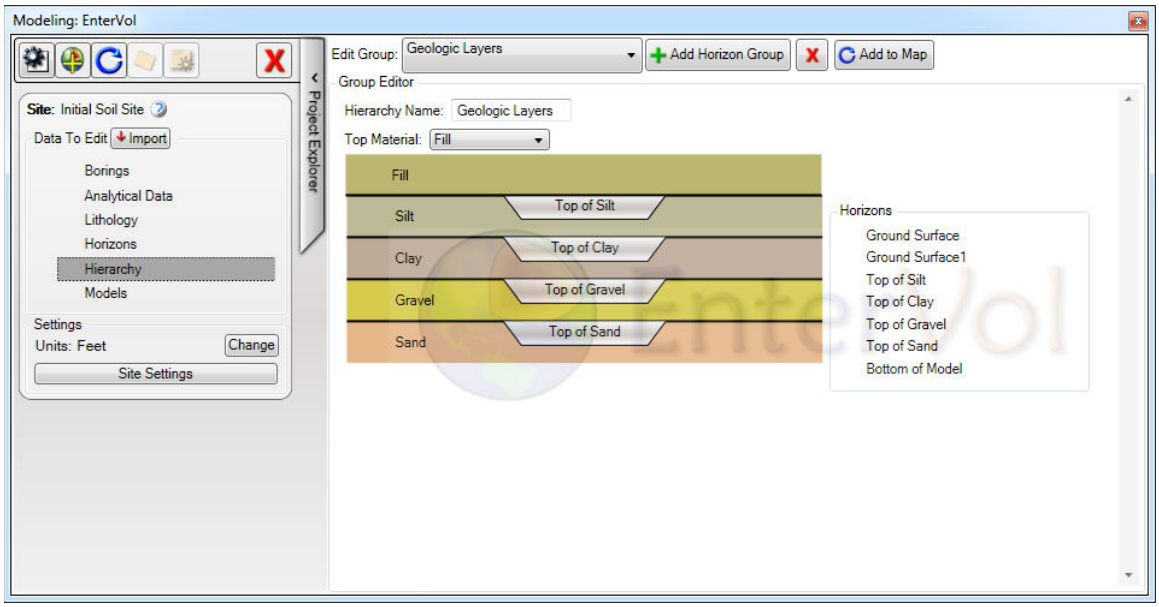
You can override or edit any of the horizon name mappings or material mappings. C Tech's GEO and GMF file formats, traditionally name Horizons based on the layer above them. In other words, the default horizon names assume the horizons represent the Bottom of that layer.

In EnterVol, the process of creating stratigraphy begins with the material found at the ground surface, and your 3D grid is cut by horizons which define the Tops of each new stratigraphic layer. For this reason, the revised horizon names above make it much easier to understand the process and the consequences of horizon reordering or deletion.

After importing the .GEO file, the site tab will update to show 6 Horizons now exist and they will be listed in the Horizons window.



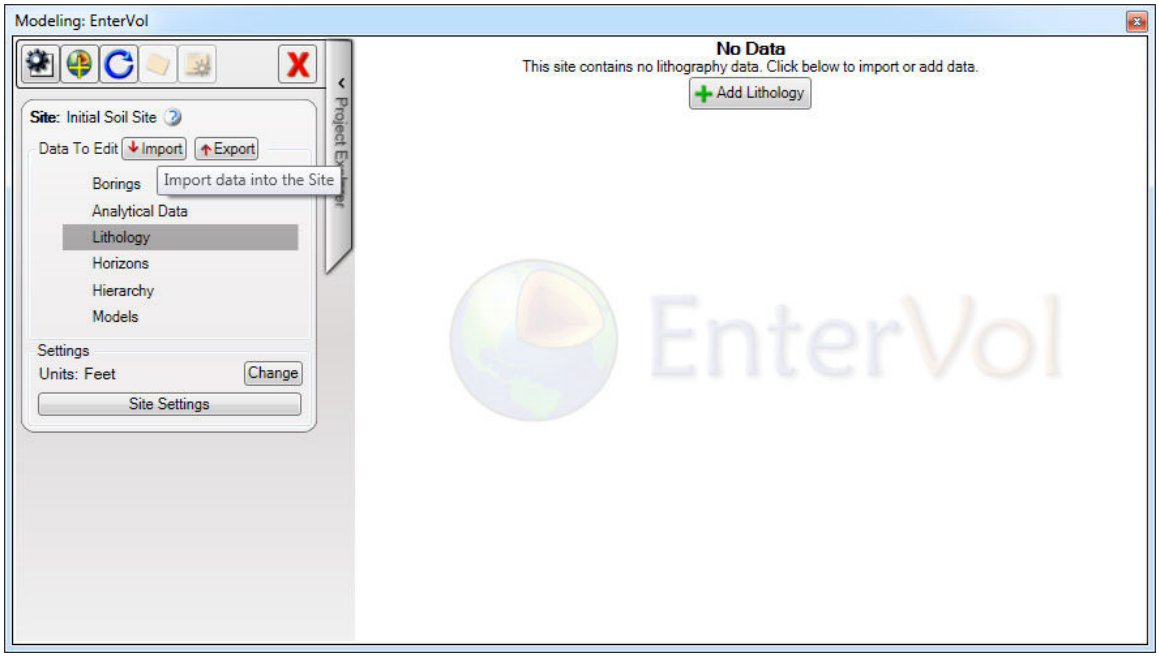

And the Hierarchy section will show:



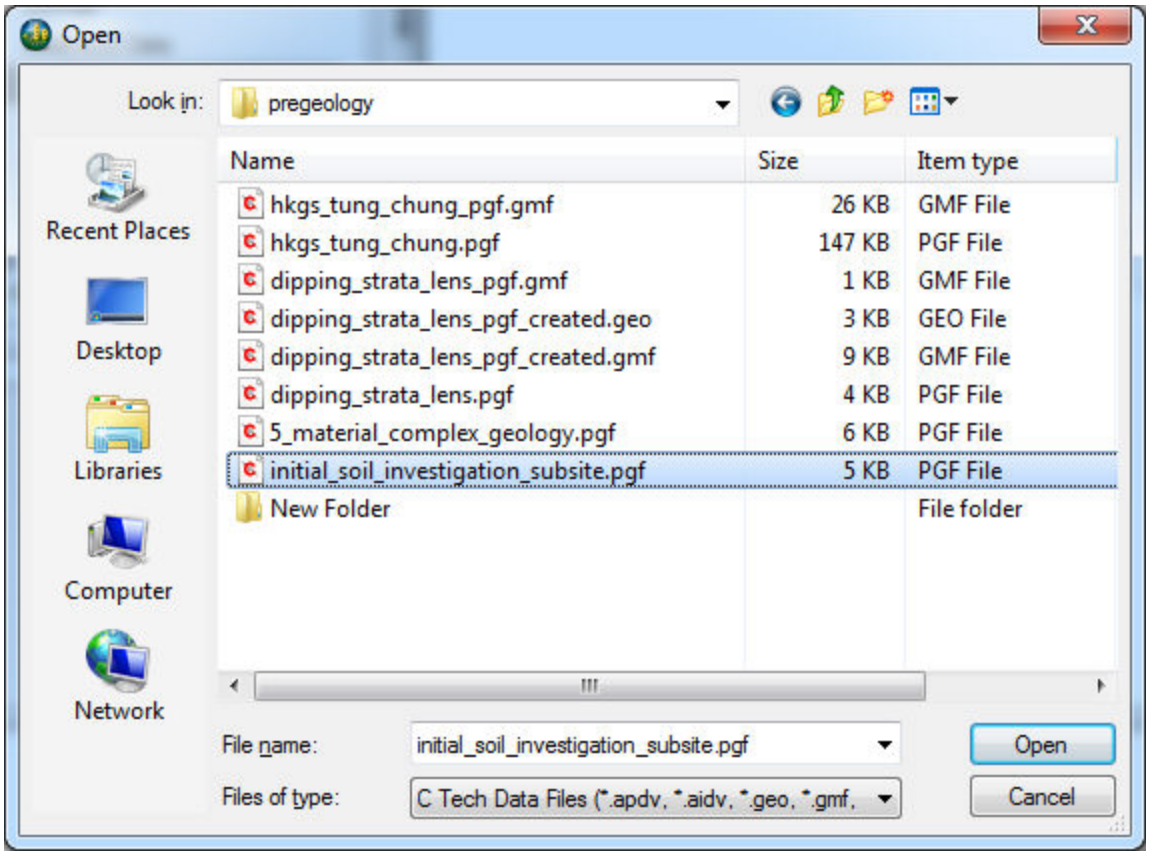

In this example, removing the horizon "Top of Clay" eliminates clay from the hierarchy resulting in a Silt layer that continues to the Top of Gravel.

### Importing Lithology Data from C Tech Files

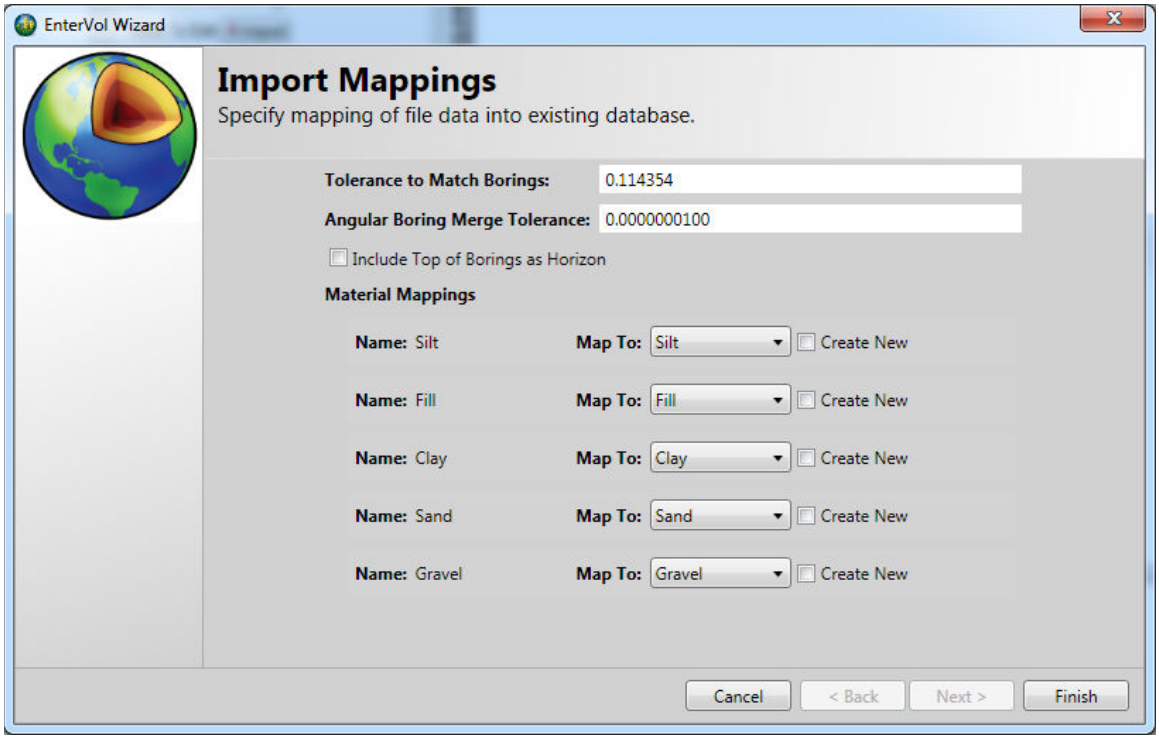
Lithology data defines the materials observed along borings. EnterVol uses C Tech's PGF file format for this type of data.



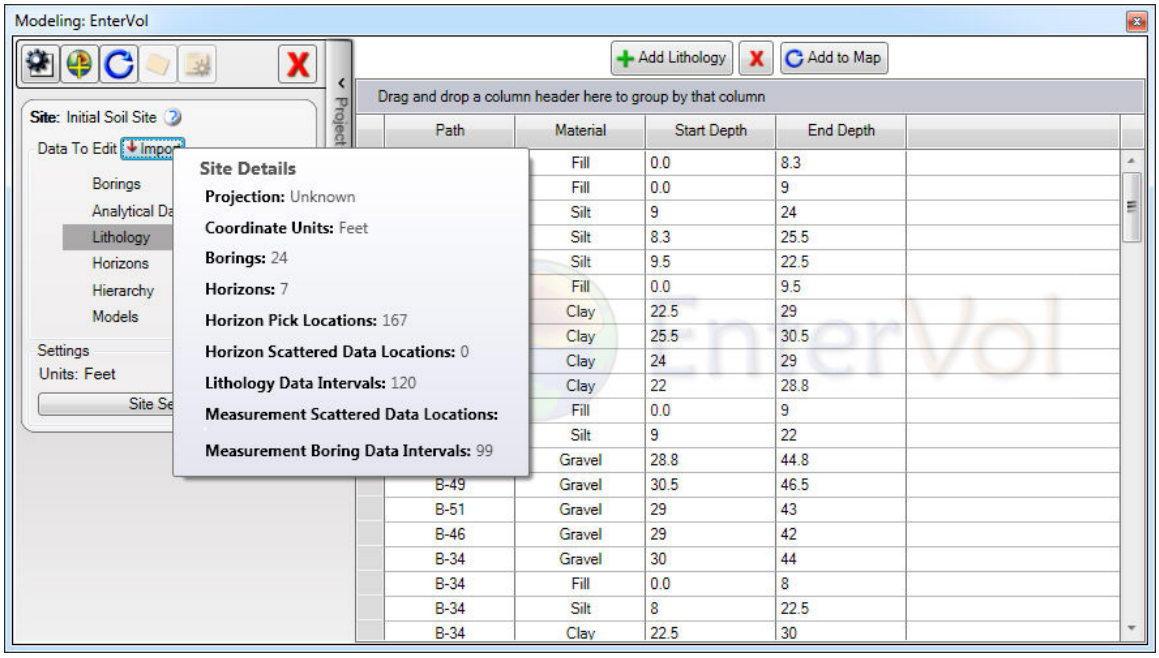

Open the file initial_soil_investigation_subsite.pgf




The next window will allow you to remap material names.

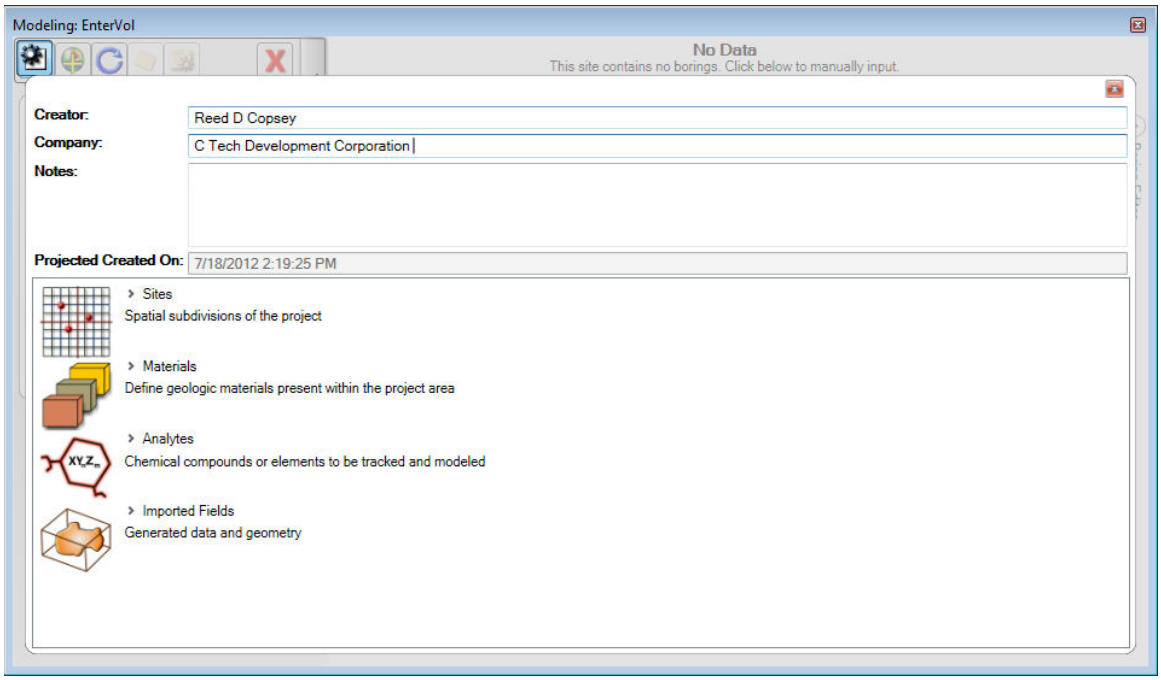


Note that there are now 120 Lithology Data Intervals. For this simple PGF file, there were 24 borings, each having 5 materials observed. 5*24 = 120



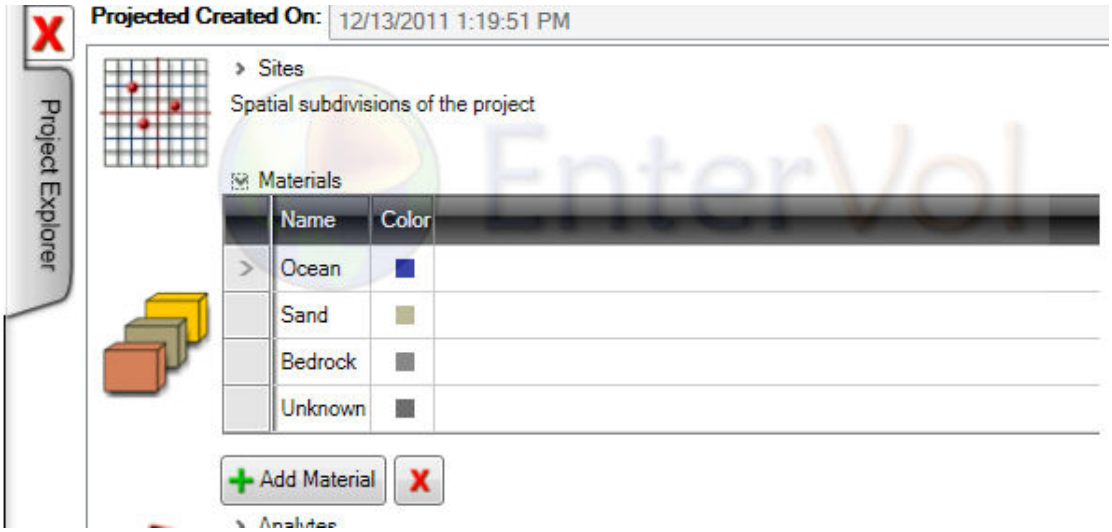

## Project Settings

Once you've imported your data, you should confirm or adjust your Project settings.

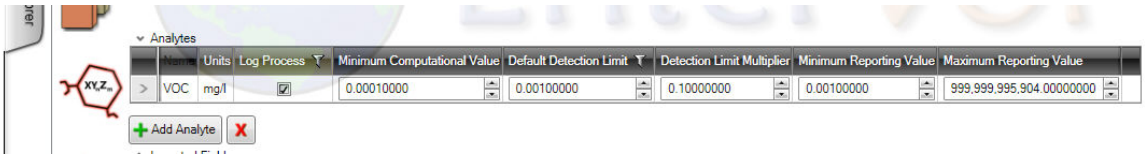


The two important settings are the Materials settings and the Analytes settings.

Expanding Materials gives access to edit the names and colors associated with each geologic material




Expanding Analytes: Allows us to confirm or edit the following important information:



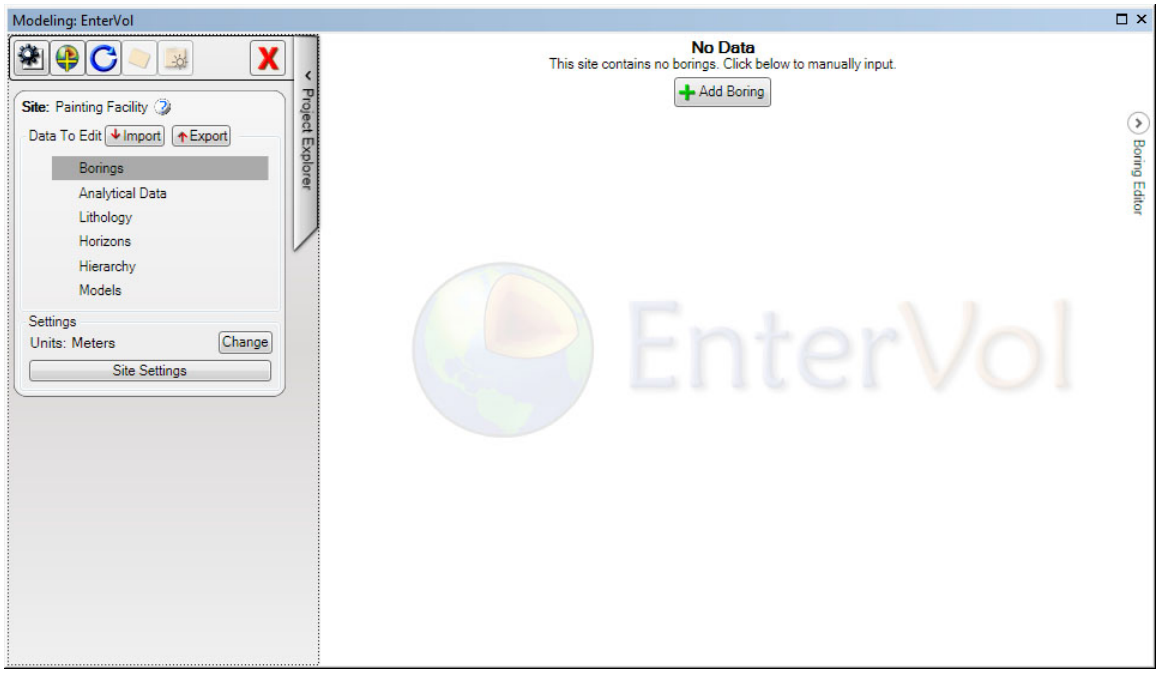

- **Analyte Name** (VOC shown above)
- **Analyte Units (mg/l shown above)**

There are six other parameters which affect how your data is used for estimation and reporting.

- **Log Process**: When this is on, we take Log10 of the Analyte data. Before we can do this, we must be sure that the data has no ZERO or Negative values.
- **Minimum Computational Value** is a master clipping that is performed just before Log Processing or estimation (if Log is off). Any values less than this value are set to this value. If you are not log processing and your data has meaningful negative numbers be sure to properly set this parameter.

- **Default Detection Limit** is the value applied to any Analyte samples which have the value of zero (0.0)
- **Detection Limit Multiplier** is applied to all samples in the database which are flagged as non-detects. For example, when importing C Tech's .apdv (Soil Chemistry) data files, any values which are ZERO or have a value preceded by "<" (e.g. <0.2 ug/l) will be multiplied by this value before estimation.
- **Minimum Reporting Value** is a master clipping that is performed after estimation (e.g. kriging). Often this value is set to match the Default Detection Limit so that the resulting analyte grid does not have any values which are reported to be below the ability to detect.
- **Maximum Reporting Value** is a master clipping that is performed after estimation (e.g. kriging). It is seldom used, which is why the default value is a very large number.

# Exporting Data




The Export button allows you to export data which has been imported into the EnterVol Modeling system in EVS (Earth Volumetric Studio) standard ASCII formats.

The Export button will export all of your project data at one time. This will include:

- Borings (vertical or non-vertical) in APDV format
- Analytical Data

    - If Point data on borings is present, an APDV with borings will be written
    - If Scattered Point data without borings is present, an APDV without borings will be written
    - If Interval data is present, an APDV with borings will be written. During Export, you will be prompted for a *Maximum Gap* parameter which will convert intervals into points.
- Lithology Data will be exported in PGF format
- Horizons data is not exported.
- Hierarchy data is not exported.

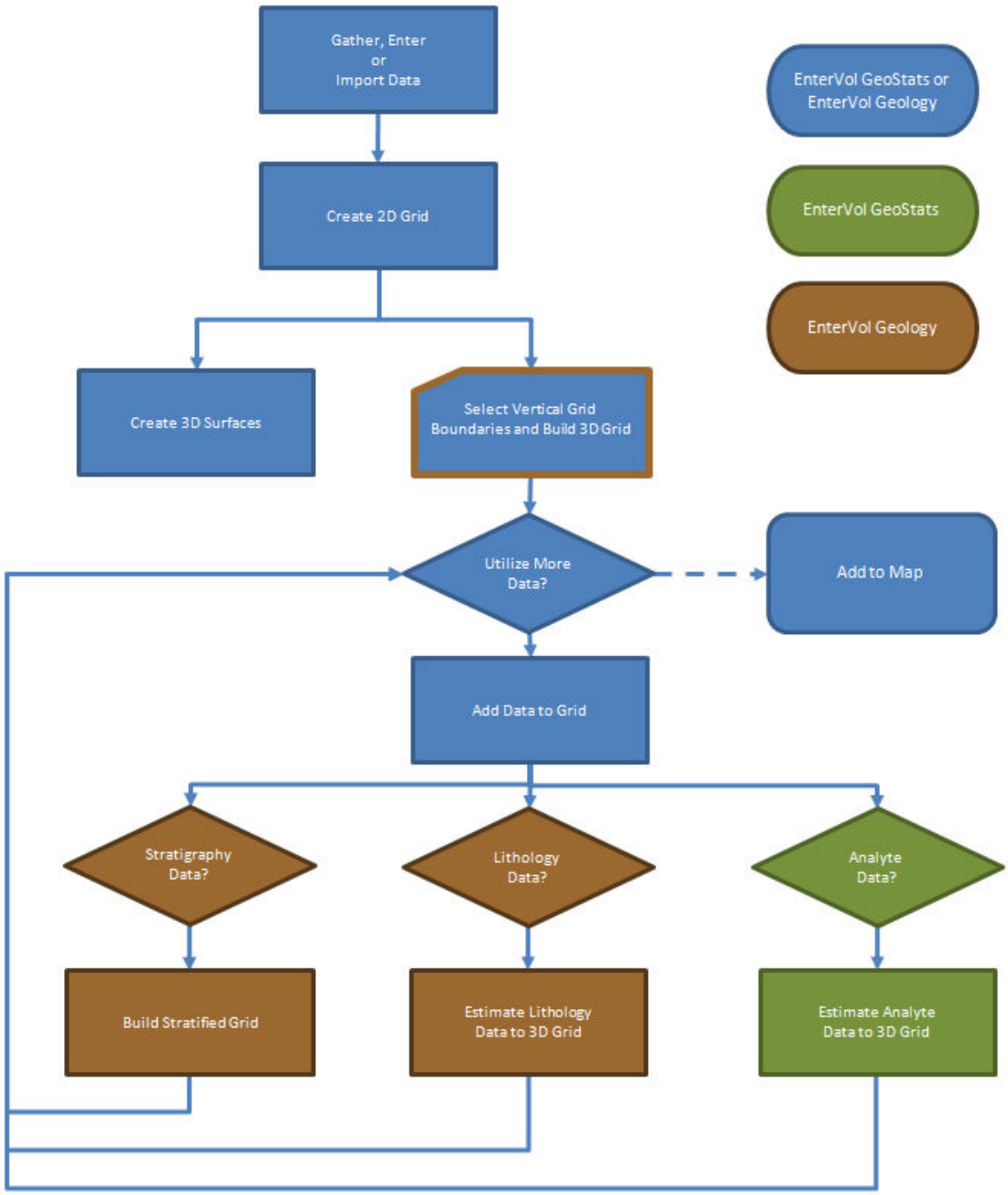
## Concepts and Modeling Process

The EnterVol Modeling Process is shown in the flowchart below.

- Processes shown in blue require either EnterVol GeoStats or EnterVol Geology.
- Those in green require EnterVol GeoStats.
- Those in brown require EnterVol Geology.

The selection of the variable or analytic surfaces that provide the vertical bounding of the 3D grid is a process which has different functionality in EnterVol GeoStats and EnterVol Geology.

- In EnterVol GeoStats, the 3D grid can be bounded by:
    - Two specified elevations
    - An arbitrary (3D) surface and an elevation or thickness
- In EnterVol Geology, the 3D grid can be bounded by:
    - Two specified elevations
    - An arbitrary 3D surface and an elevation or thickness
    - Two arbitrary 3D surfaces



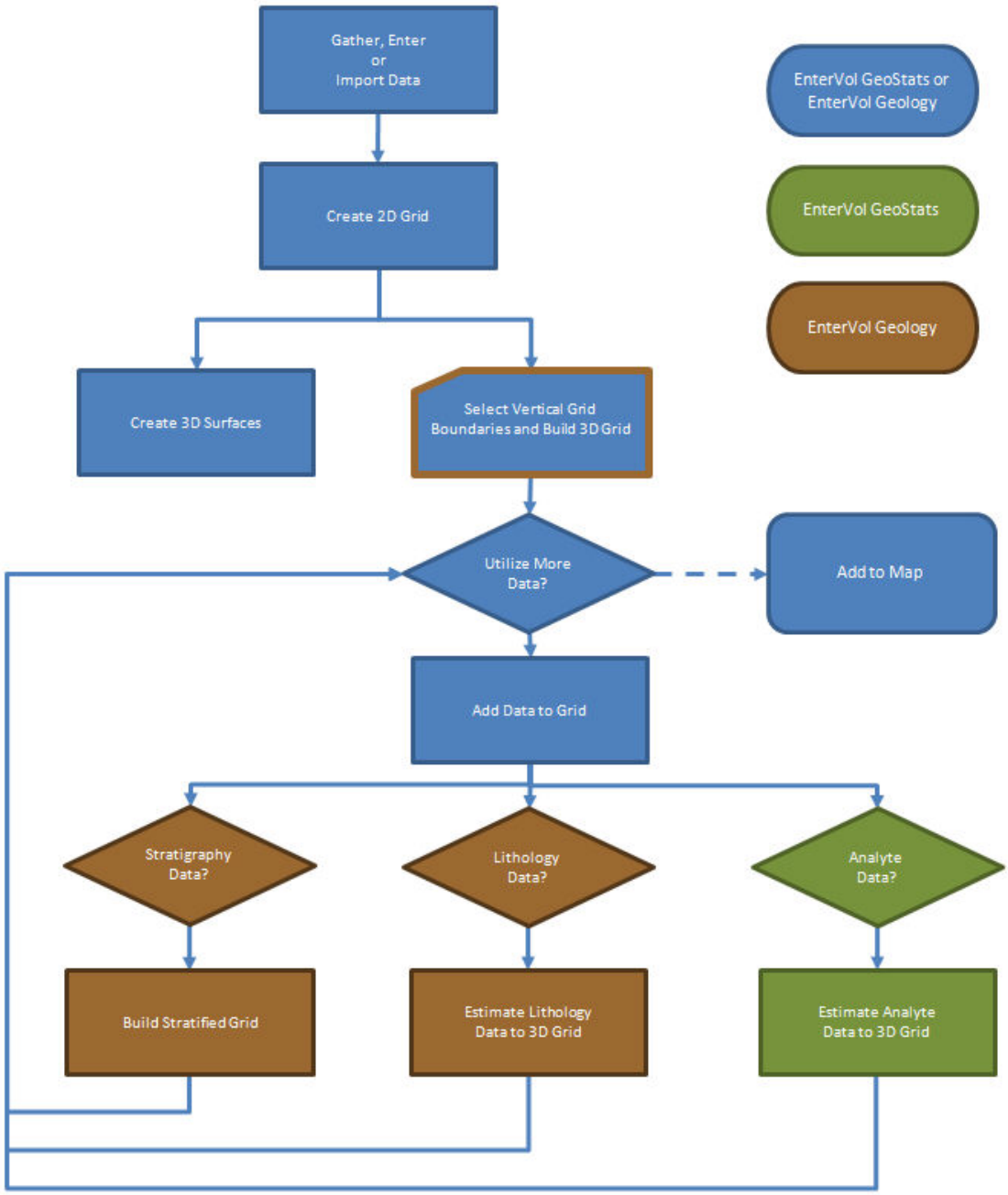

## Concepts and Modeling Process

The EnterVol Modeling Process is shown in the flowchart below.

- Processes shown in blue require either EnterVol GeoStats or EnterVol Geology.
- Those in green require EnterVol GeoStats.
- Those in brown require EnterVol Geology.

The selection of the variable or analytic surfaces that provide the vertical bounding of the 3D grid is a process which has different functionality in EnterVol GeoStats and EnterVol Geology.

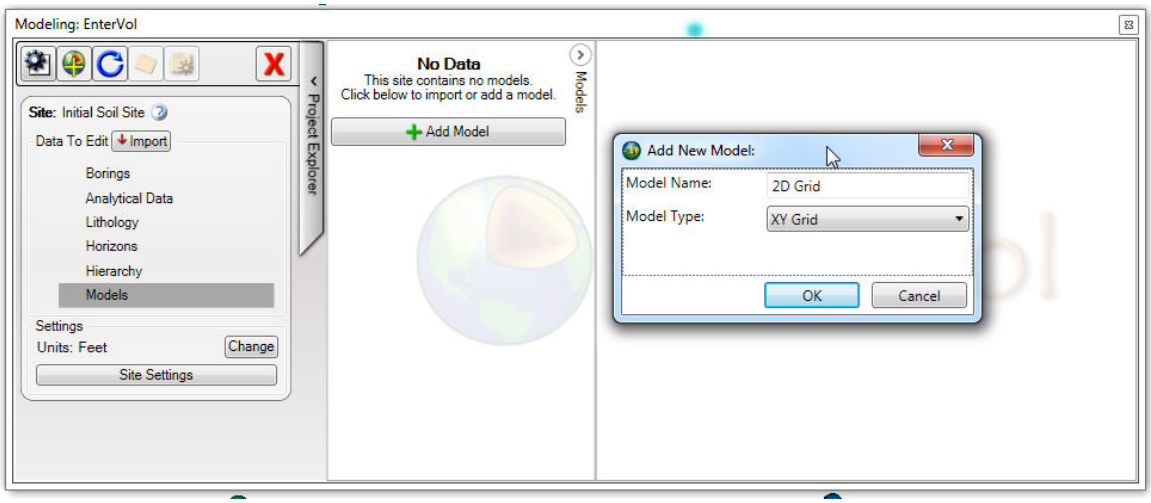
- In EnterVol GeoStats, the 3D grid can be bounded by:
  - Two specified elevations
  - An arbitrary (3D) surface and an elevation or thickness
- In EnterVol Geology, the 3D grid can be bounded by:
  - Two specified elevations
  - An arbitrary 3D surface and an elevation or thickness
  - Two arbitrary 3D surfaces


Gather, Enter
or
Import Data
Create 2D Grid
Create 3D Surfaces
Select Vertical Grid
Boundaries and Build 3D Grid
Utilize More
Data?
Add to Map
Add Data to Grid
Stratigraphy
Data?
Lithology
Data?
Analyte
Data?
Build Stratified Grid
Estimate Lithology
Data to 3D Grid
Estimate Analyte
Data to 3D Grid
EnterVol GeoStats or
EnterVol Geology
EnterVol GeoStats
EnterVol Geology

## 2D Grid Generation

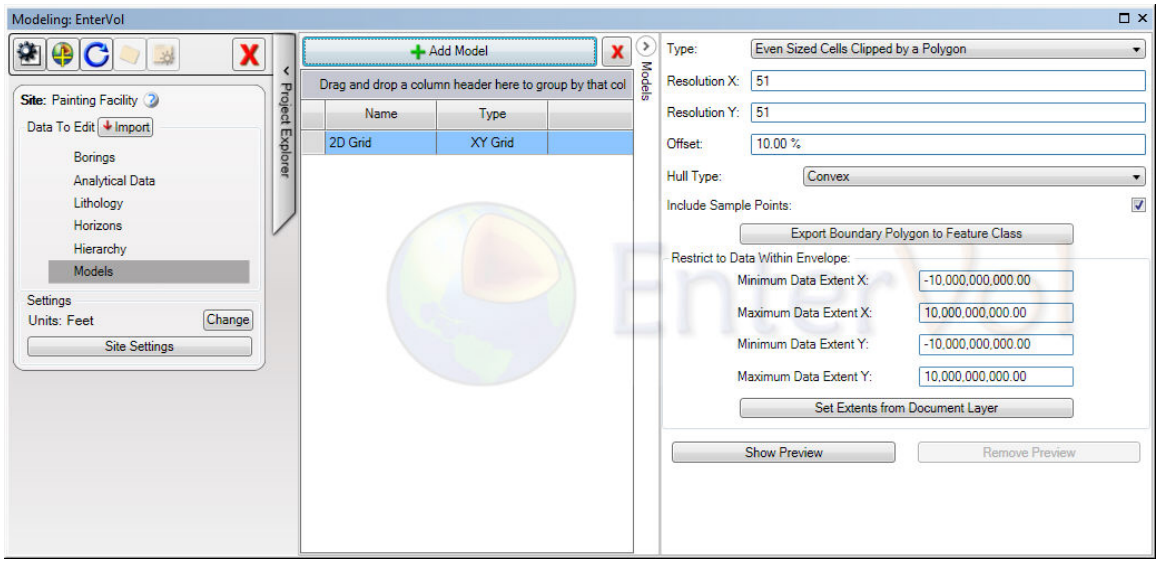
2D Grid Generation is a required step in building a 3D volumetric grid. The 2D grid is the triangles (cells) and the X-Y locations (nodes) that define the corners of the cells in the X-Y Plane

Click on the Add Model button




.

When you choose OK after providing a Model Name, the 2D Grid modeling options window opens



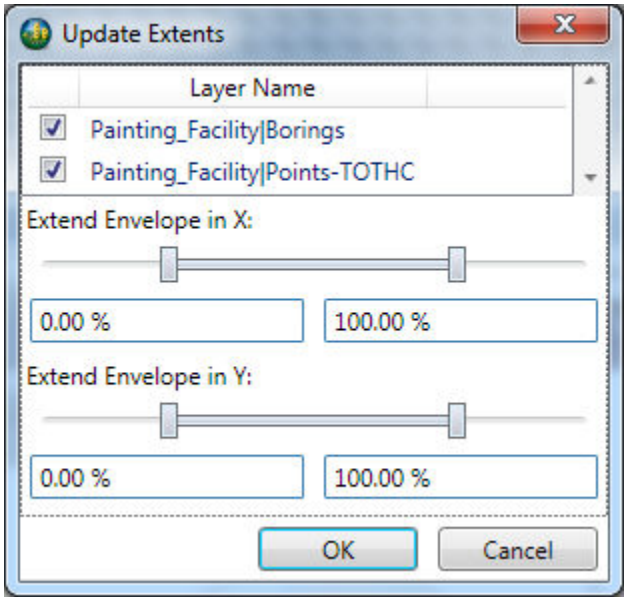

There are three different types of 2D grids.

1. Rectilinear

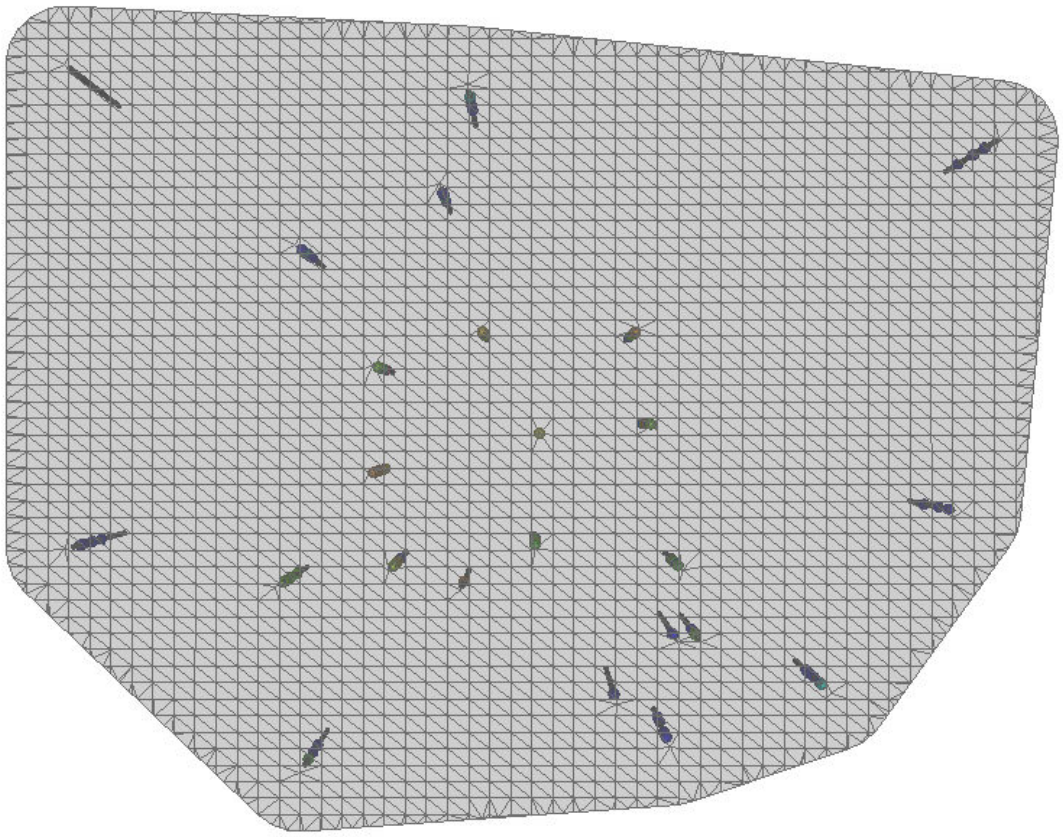
2. Even Sized Cells Clipped by a Polygon
3. Convex Hull with Smoothed Cells

The extents of your grid are determined automatically by the extent of all boring (tops), analytical points and surface data, however you can restrict the Minimum and Maximum X-Y Data extents to limit the data used to determine the grid. This grid restriction does not limit the data used to compute surfaces, horizons, 3D grids or Analyte Estimation. You must have data in your scene to set your extents. This can be one or many of your Scene Layers and you can then crop that extent with interactive sliders.

Press the *Set Extents from Document Layer* button. In the example below, there are two Scene Layer in the project. They are both selected by default.



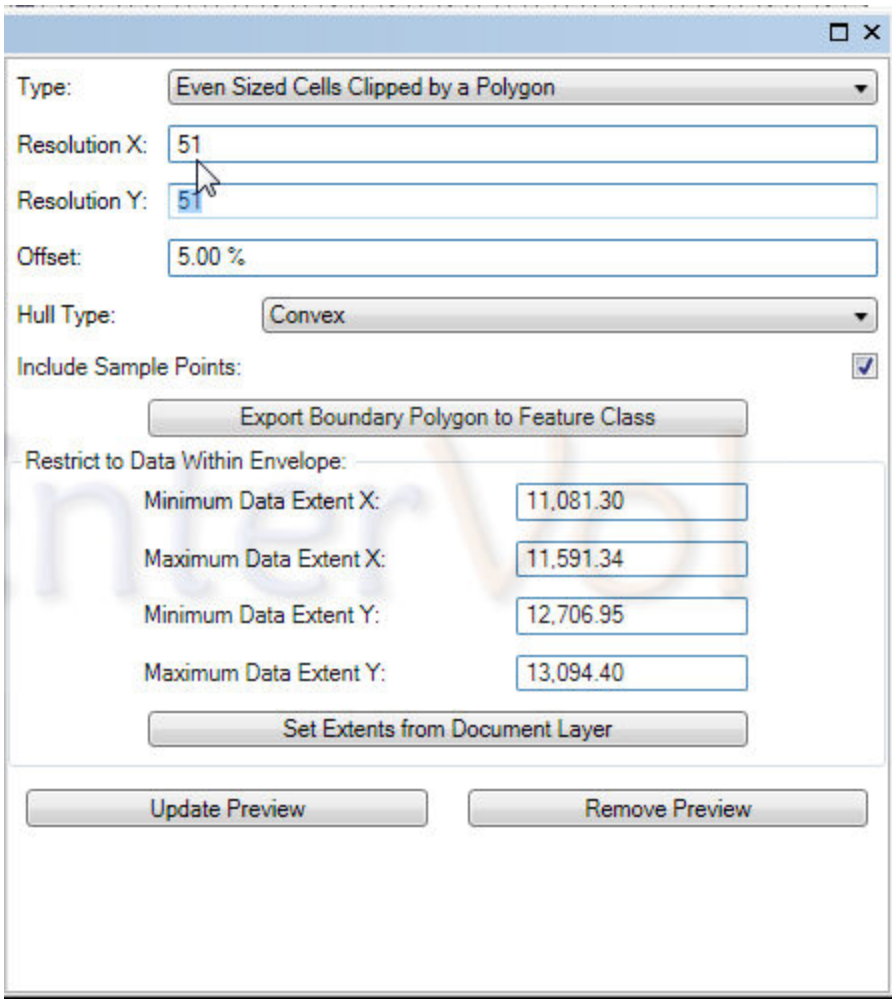

If we click on Show Preview we can see the resulting grid created with the *Even Sized Cells Clipped to a Polygon* option and the default "convex hull":



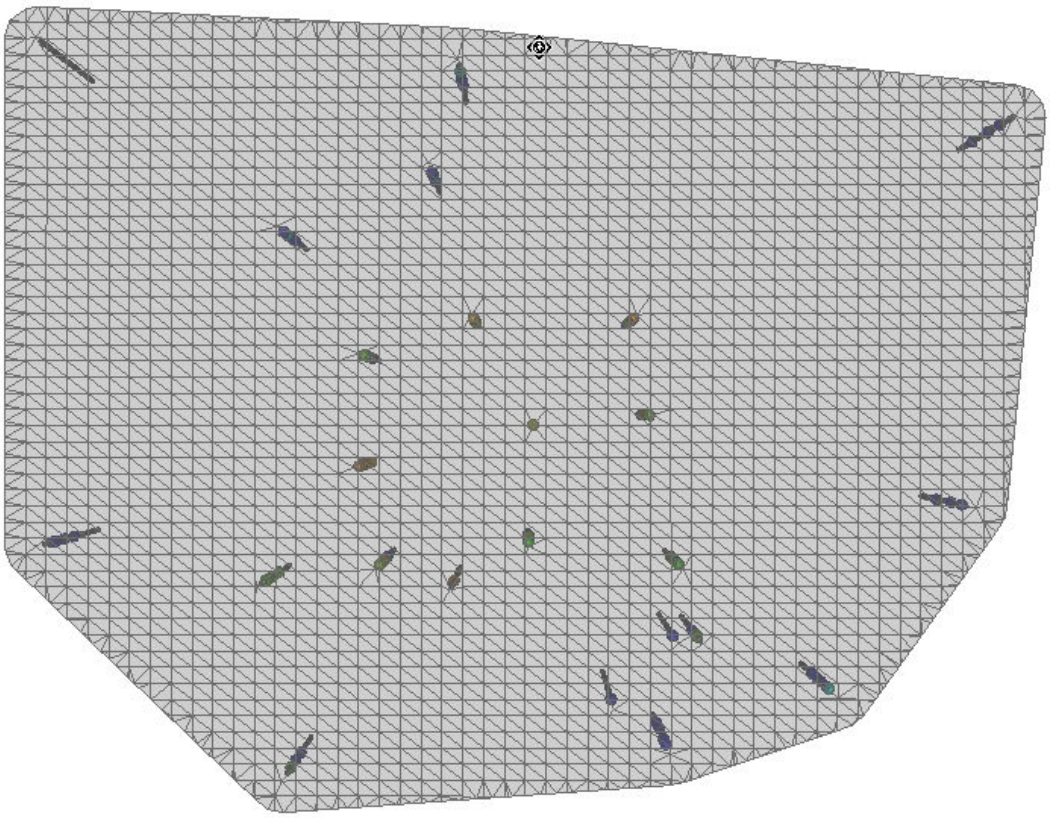
The convex hull of your data is a shape equivalent to a rubber band stretched around all of your surface points. All three grid types can be offset (extended) a percentage of the X-Y diagonal extent. The default value of 10% was used in the example above.

**NOTE:** The 2D grid preview is a Graphic added to the Scene's default Graphics Layer. If you save your project with the preview showing, it will be there when you next load the project. To remove it, select it as a graphic and delete.

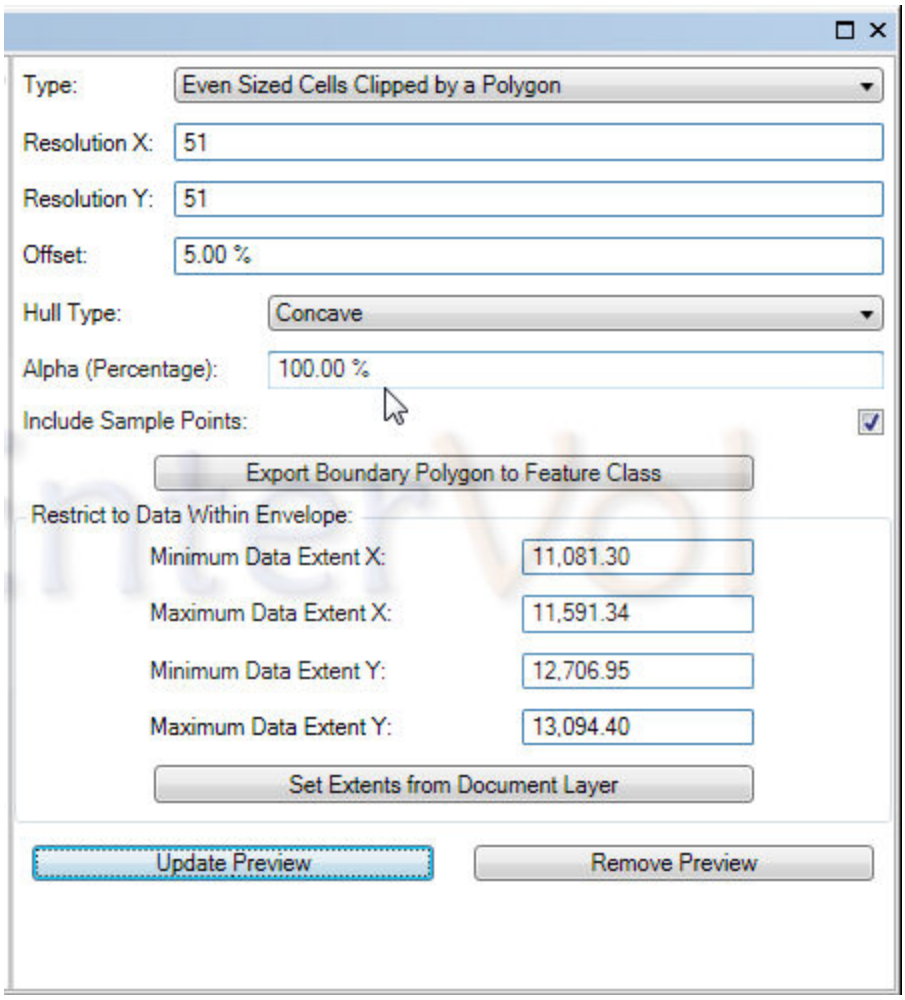
Let's see our grid if we select the other convex hull option and reduce the offset to 5%. Make the changes below and click on Update Preview.

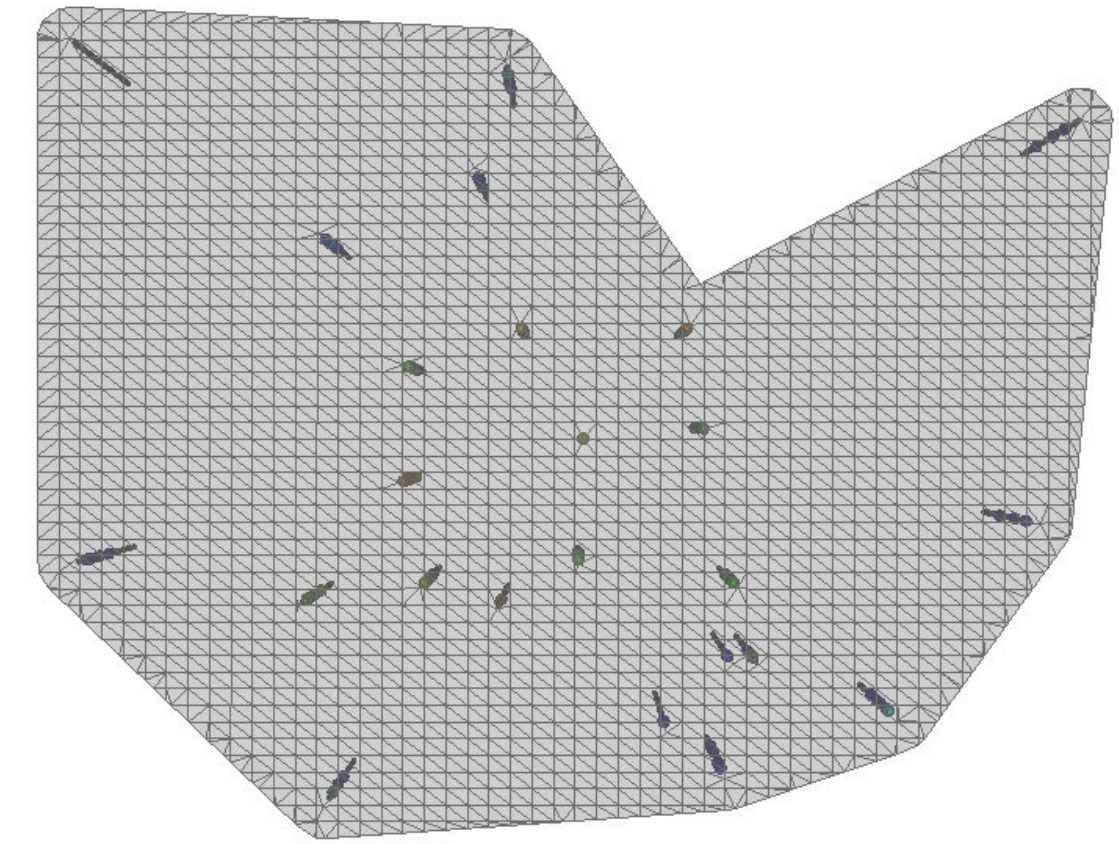



**NOTE:** When the *Include Sample Points* option is toggled on, the grid will include the X-Y coordinates of each point or sample. This helps to ensure that the estimations accurate include these values.

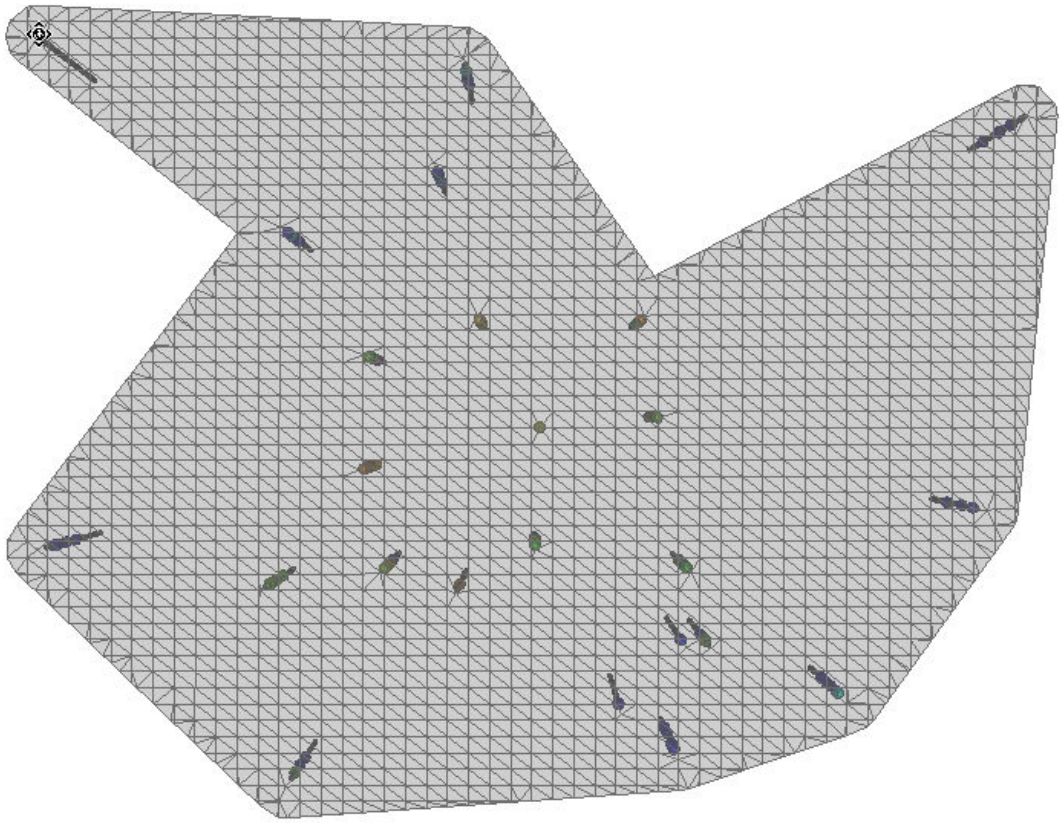


There is another interesting option which is similar to convex hull, but attempts to exclude regions of low data density. It is the *Concave Hull*. Using the default Alpha value of 100% yields


Type:
Even Sized Cells Clipped by a Polygon
Resolution X:
51
Resolution Y:
51
Offset:
5.00 %
Hull Type:
Concave
Alpha (Percentage):
100.00 %
Include Sample Points:
Export Boundary Polygon to Feature Class
Restrict to Data Within Envelope:
Minimum Data Extent X:
11,081.30
Maximum Data Extent X:
11,591.34
Minimum Data Extent Y:
12,706.95
Maximum Data Extent Y:
13,094.40
Set Extents from Document Layer
Update Preview
Remove Preview

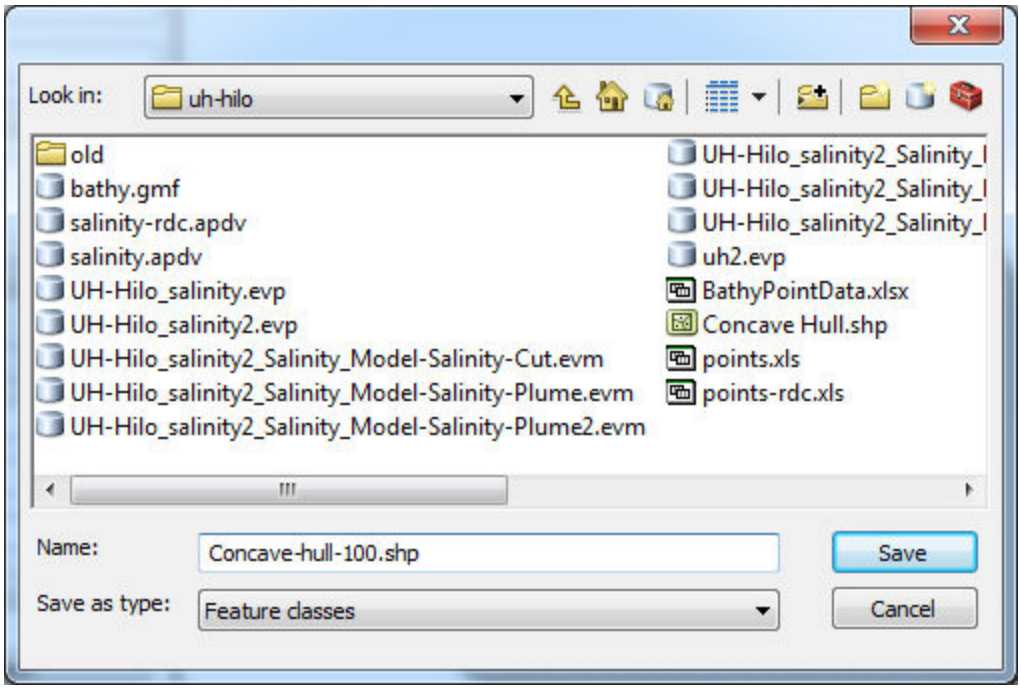


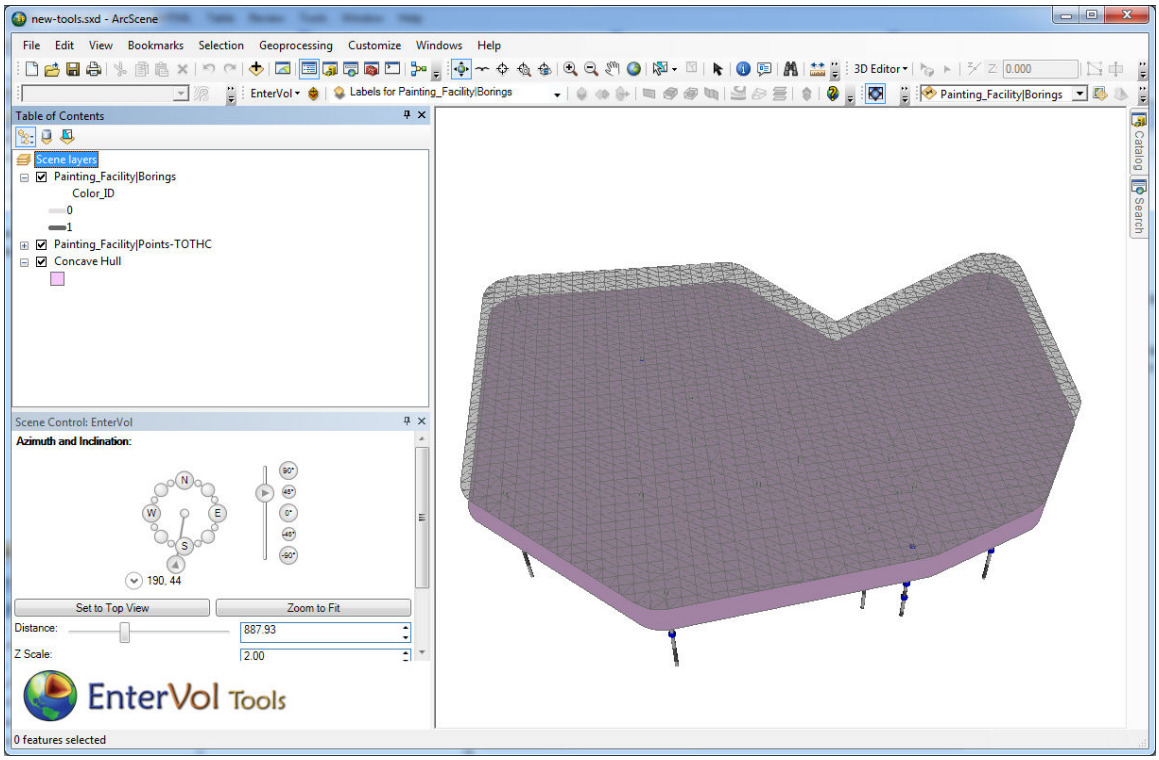
Lowering Alpha to 80% (which is the lowest allowed value) gives:



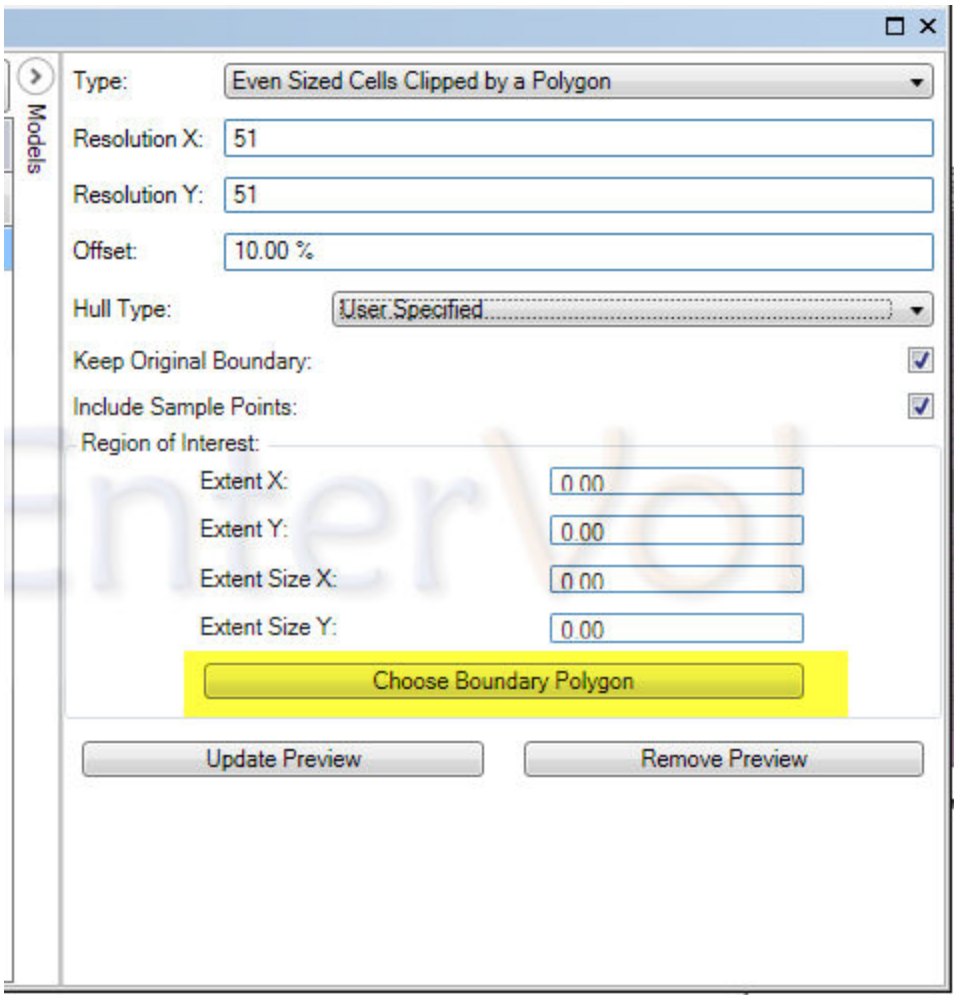
For the convex and concave hull options, you can *Export Boundary Polygon to Feature Class*. This allow you to export the current grid shape and then add more data to your project. You can then use this polygon to restrict your grid in ways that are not possible if the new data has larger extents.

For the Concave hull with 100% alpha and 10% offset, the polygon would be:

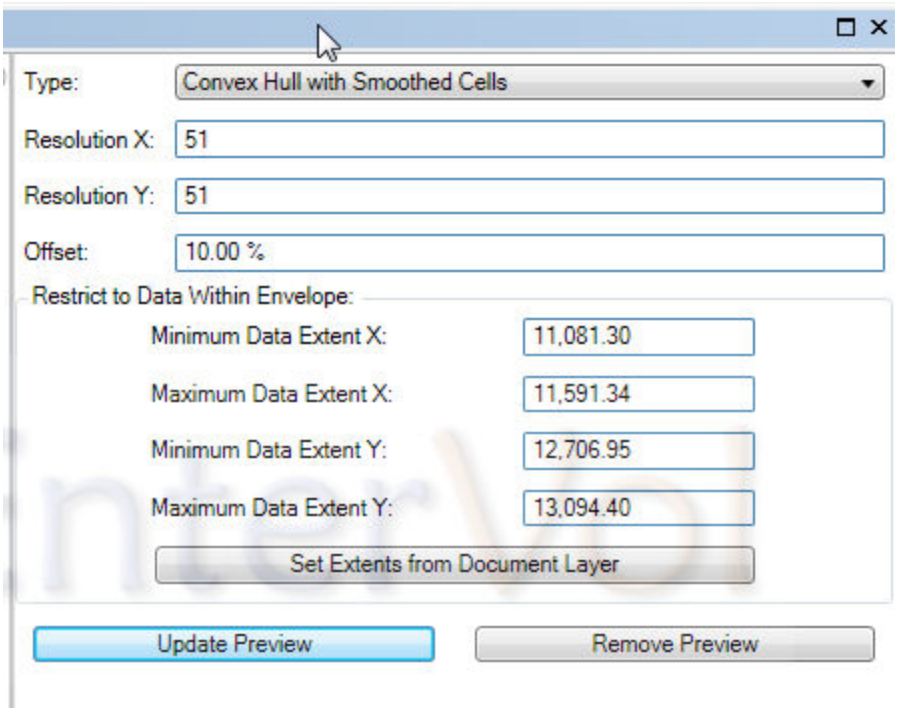


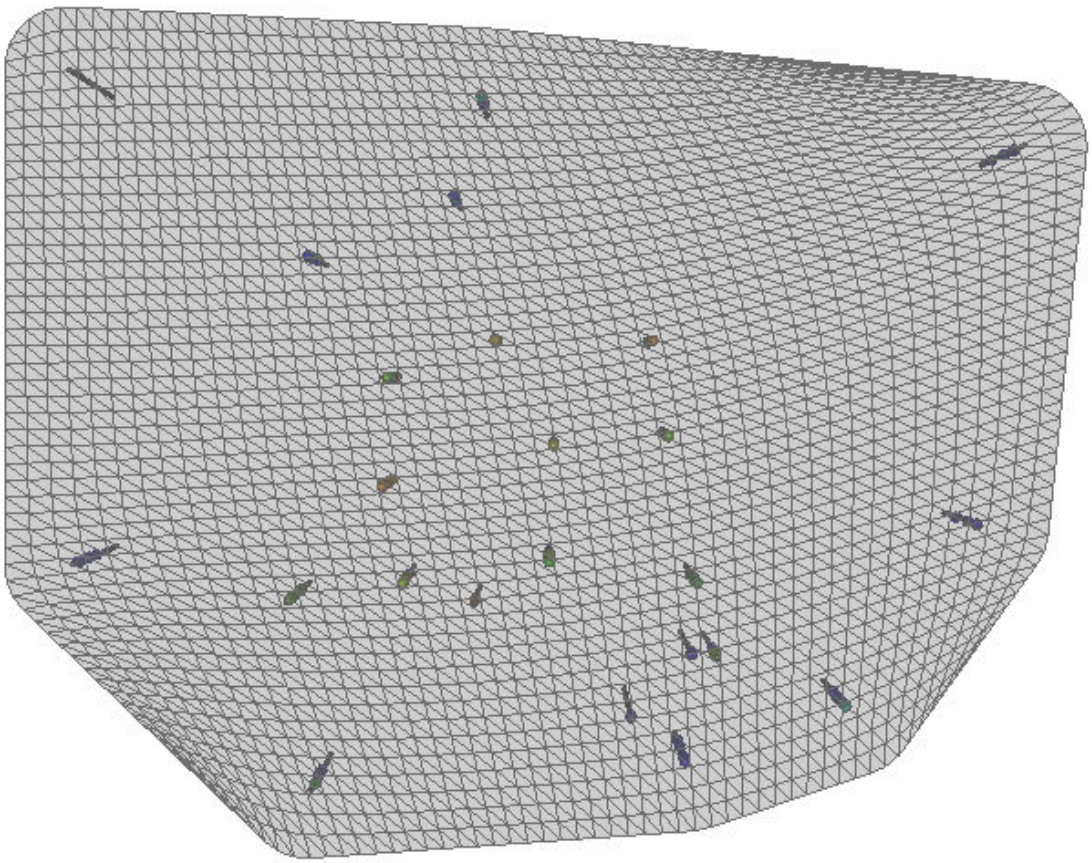




To use this polygon, choose the User Specified boundary polygon options:

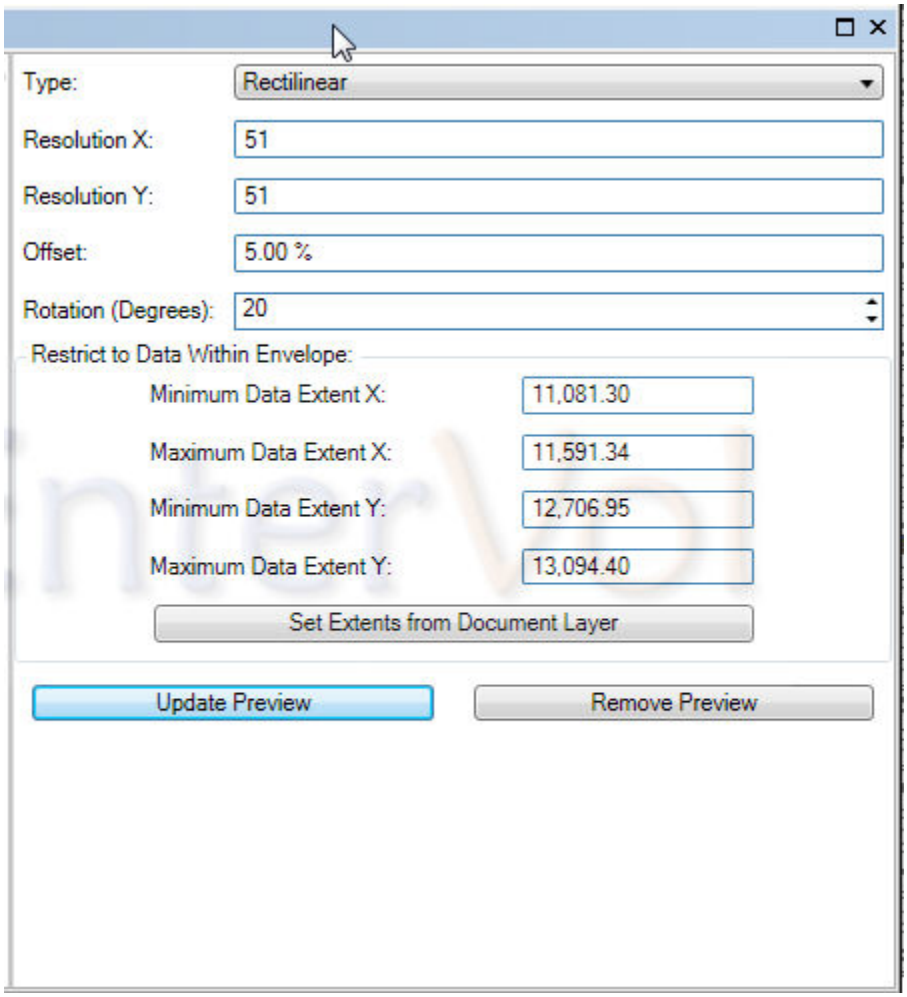


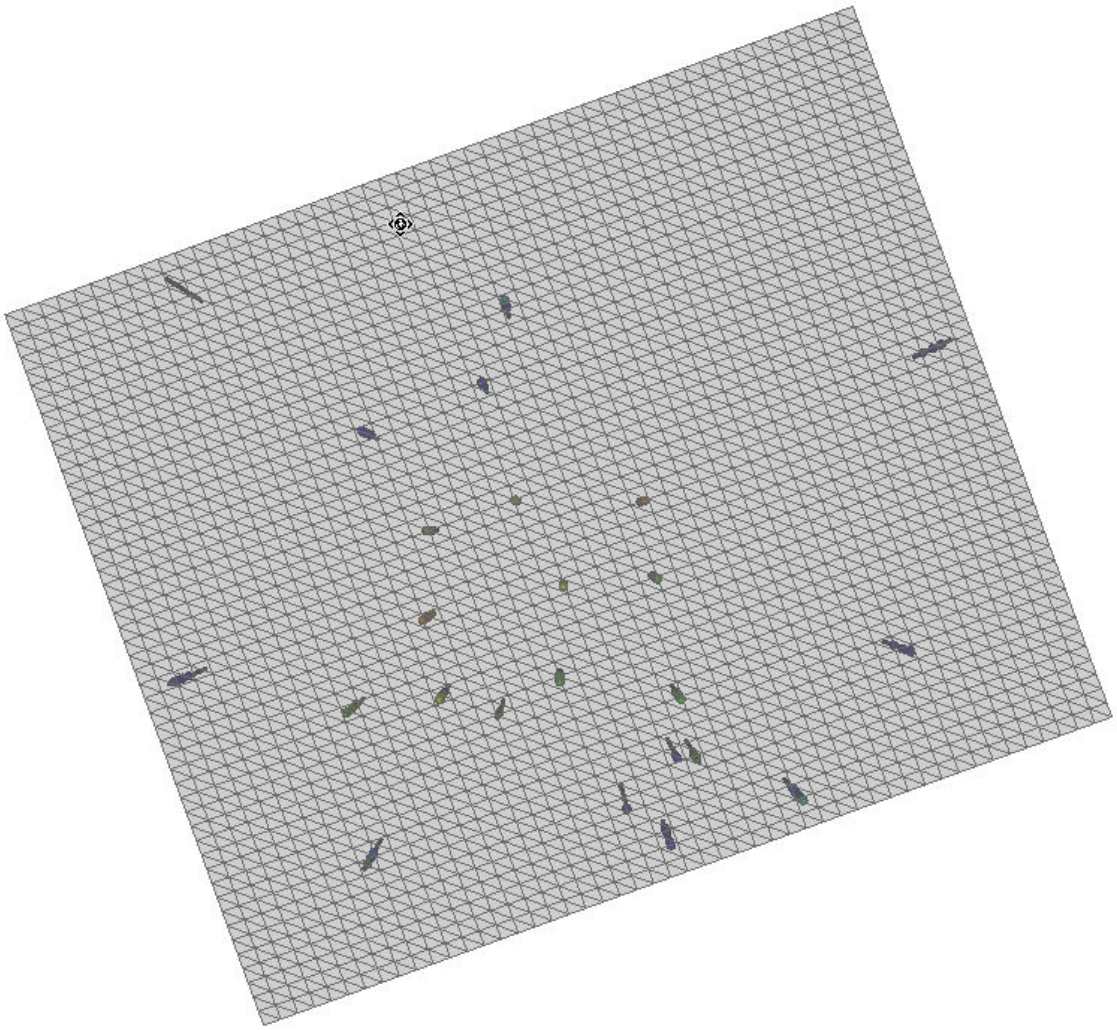

Another option is the Convex Hull with Smoothed Cells. This is very similar to the Convex Hull standard in EVS and MVS.


Type:
Convex Hull with Smoothed Cells
Resolution X:
51
Resolution Y:
51
Offset:
10.00 %
Restrict to Data Within Envelope:
Minimum Data Extent X:
11,081.30
Maximum Data Extent X:
11,591.34
Minimum Data Extent Y:
12,706.95
Maximum Data Extent Y:
13,094.40
Set Extents from Document Layer
Update Preview
Remove Preview



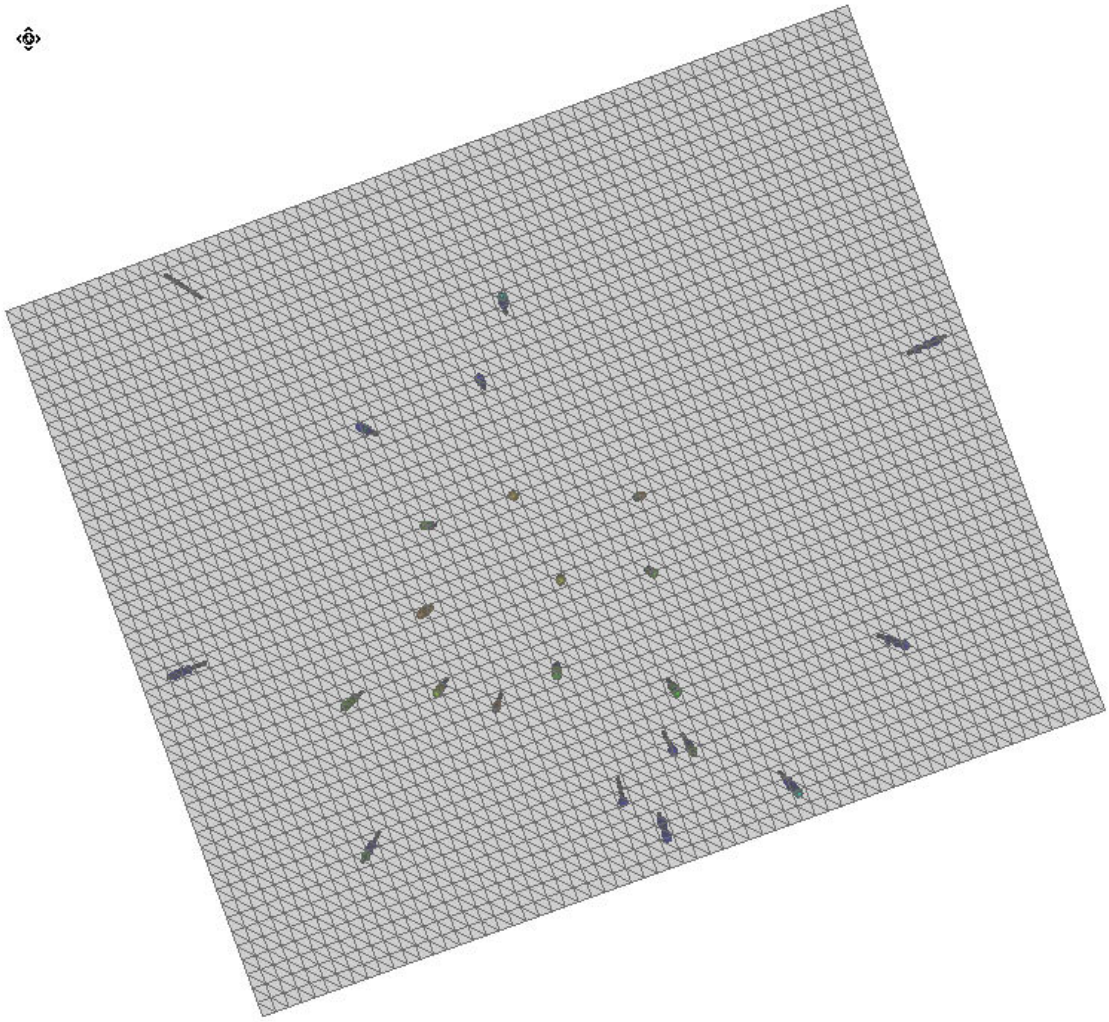
For completeness, let's look at the Rectilinear gridding option. It has an additional parameter which is the grid rotation (from the X axis). Looking at the convex hull grid above, we can see how a grid rotated about 20 degrees (counterclockwise) might better fit our data.





Notice that in the 2D rectilinear grid above, the cells are longer in the X direction than Y because the grid had the same resolutions but different X-Y extents.

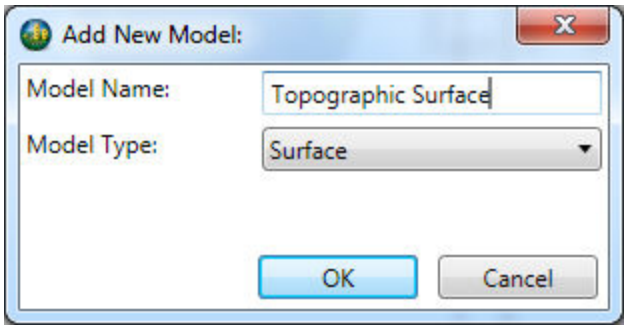
Changing the Resolution X to 65 will give us a more symmetrical grid.

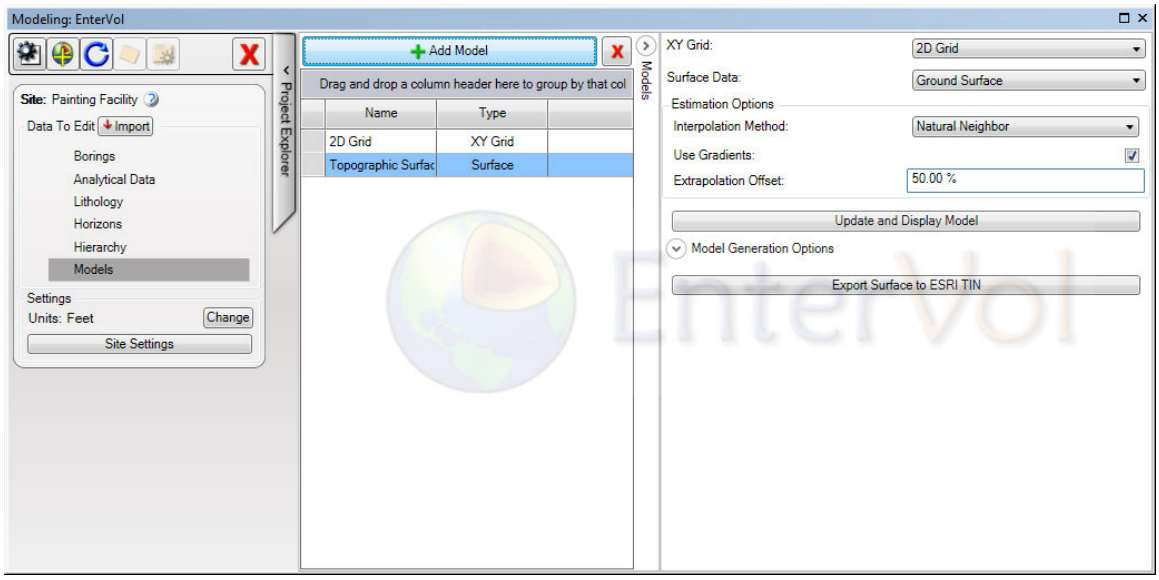


Each time we change the settings our 2D grid is updated (regardless of its name) to reflect the setting chosen. Before moving on, let's go back to *Even Sized Cells Clipped to a Polygon* (Convex Hull) with a 5% (.05) offset.

### 3D Surface Generation

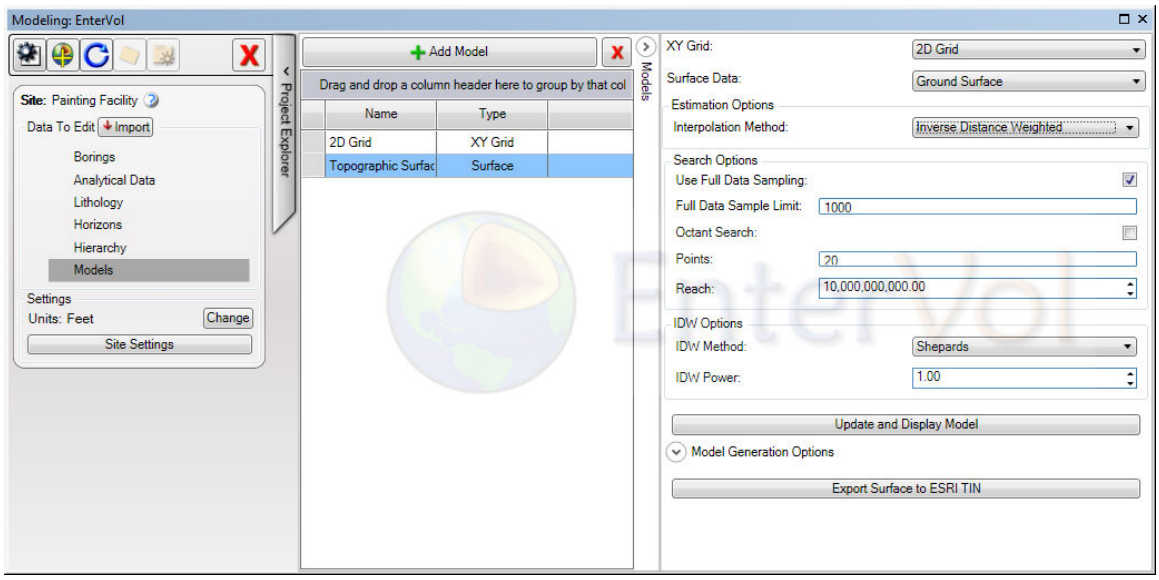
3D Surfaces are created by specifying a previously created 2D Grid and choosing Surface data from one of your Stratigraphy surfaces.



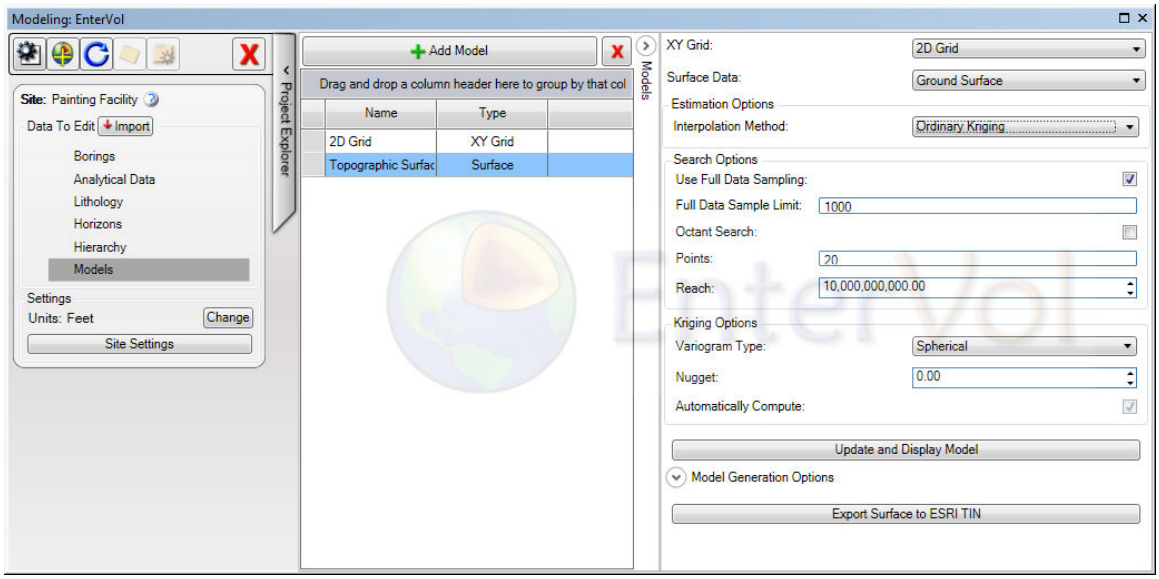




There are three Interpolation Methods. Natural Neighbor is shown above.

Inverse Distance Weighted has different creation options:

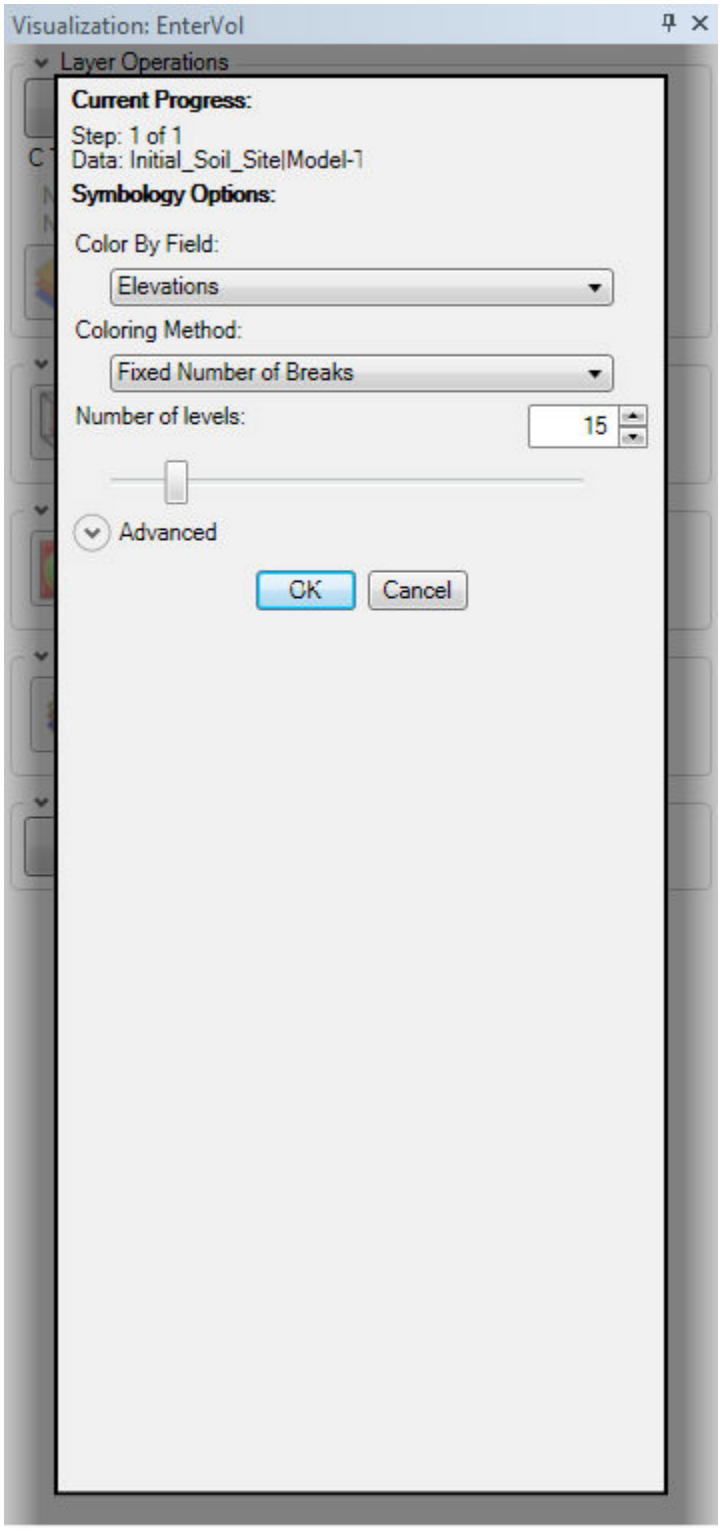



As does Kriging:

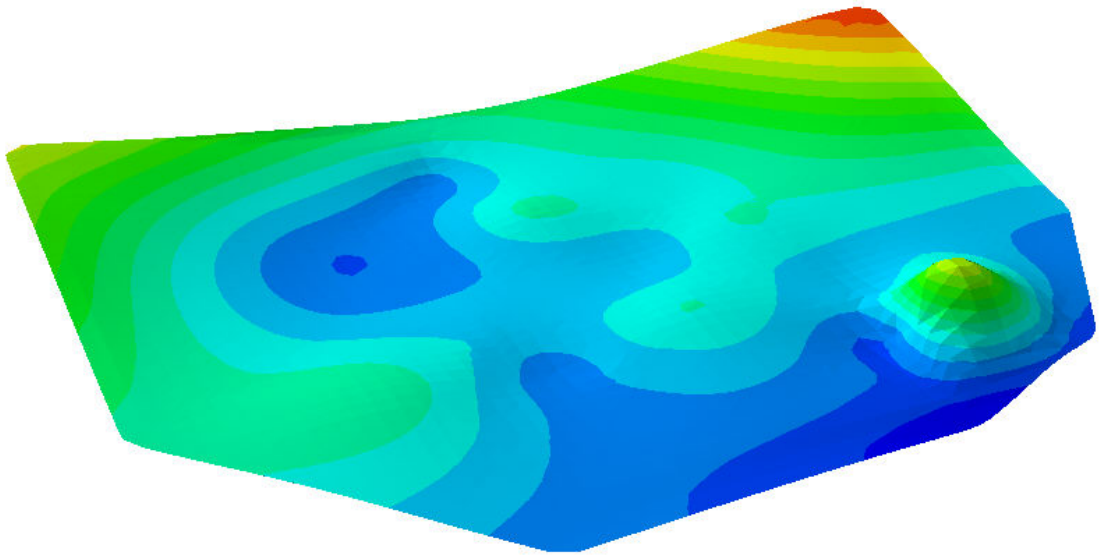



Once you select the surface to be created (from those in Horizons) and the method (let's use Natural Neighbor), you have two options.

- Update and Display Model: This option should be used to add the surface layer to your Table of Contents
- Export Surface to ESRI TIN: This option has some extra steps (since you must specify the folder name for the TIN files), and it also adds the surface layer to your Table of Contents. The big difference is that as a TIN, you can use the surface as elevation data for imagery (aerial photos).

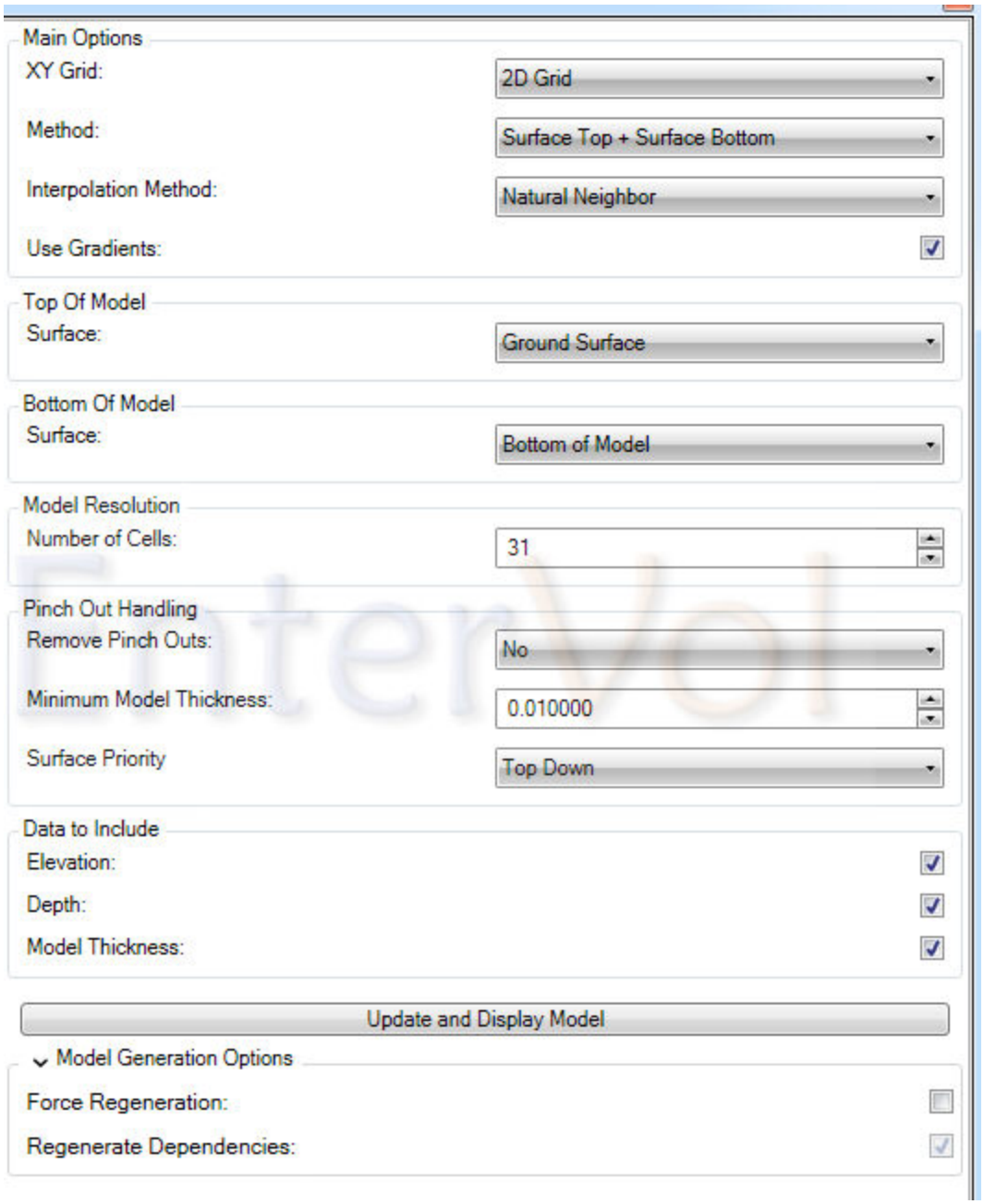



EnterVol for ArcGIS



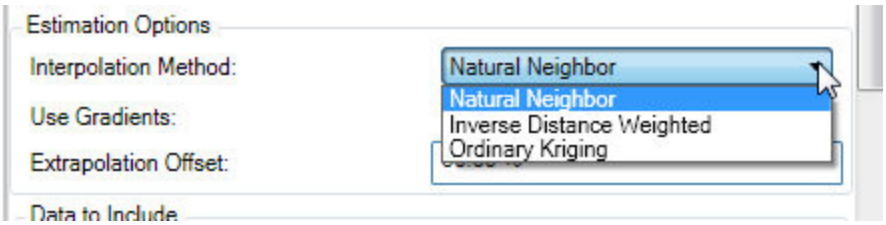
## 3D Grid Generation

3D Grid Generation is one of the processes which has enhanced capabilities when EnterVol Geology is licensed.



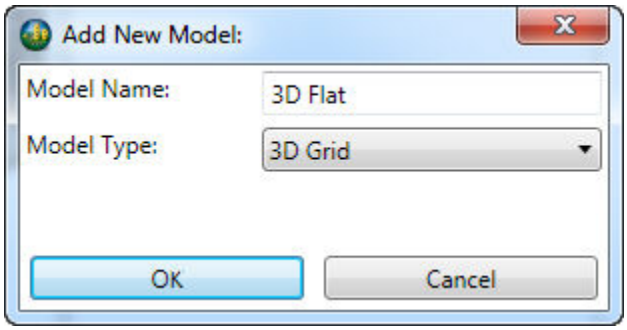

The 3D Grid Generation options are:

- XY Grid: Here you choose from the XY grids that you have created. Your modeling project can have more than one.
- Method:
  - *Flat Top + Flat Bottom* at specified elevations
  - *Surface Top + Flat Bottom*
  - *Flat Top + Surface Bottom*
  - *Constant Thickness*
  - *Surface Top + Surface Bottom (requires EnterVol Geology)*

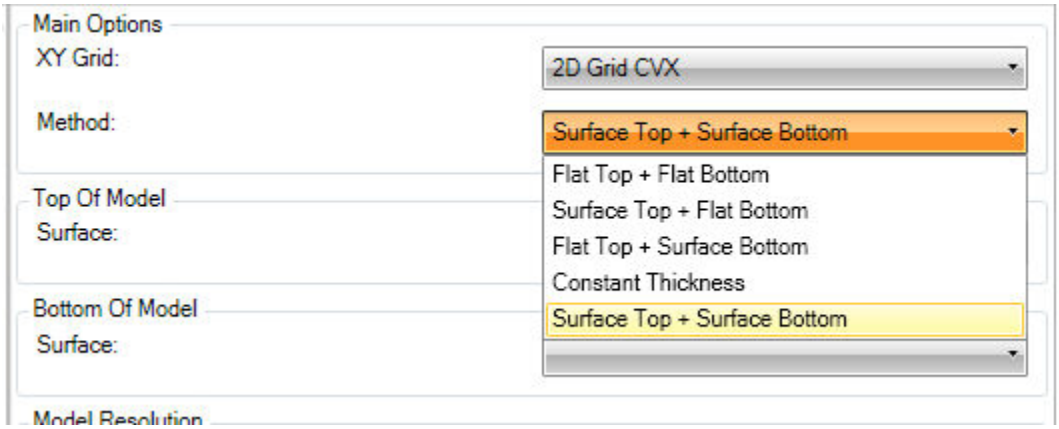



- Interpolation Method: Each method has different options. See the 3D Surface Generation topic for examples.
  - Natural Neighbor is the default and recommended method
  - Inverse Distance Weighted
  - Ordinary Kriging
- Use Gradients: Determines if surface gradients will affect the surface shape. Often makes more realistic surfaces, but occasionally extrapolates in bizarre ways if your data is clustered.
- Top Of Model: Here you select the surface which will define the top of your 3D Grid
- Bottom Of Model: Here you select the surface which will define the bottom of your 3D Grid
- Model Resolution: The number of cells in the vertical direction. If you plan to stratify your model or add analyte data you need a reasonable number of cells (depends on the number of layers and/or amount of data in the z direction).
- Pinch Out Handling
  - Remove Pinch Outs: If layers drop below a threshold thickness they will be removed or will remain as a thin sheet. If your grid will be exported to initialize ground water models you may not want to remove these sections.
  - Minimum Model Thickness: The threshold below which layers will be removed. This will also be the minimum thickness for layers that would otherwise pinch if the above option is set to NO.
  - Surface Priority: If horizons intersect one another, this option determine the hierarchy (precedence) of horizons.
- Data to Include:
  - Elevation (z coordinate)
  - Depth (distance below the ground surface)
  - Model Thickness
- Update and Display Button (creates the model and adds it to your table of contents
- Model Generation Options
  - Force Regeneration: Recomputes all dependent models. In this case the XY grid specified.

Click on the  to create a new model element. Provided that at least one 2D grid has already been created, the option of creating a 3D Grid will be available. Choose a Model Name and hit OK.

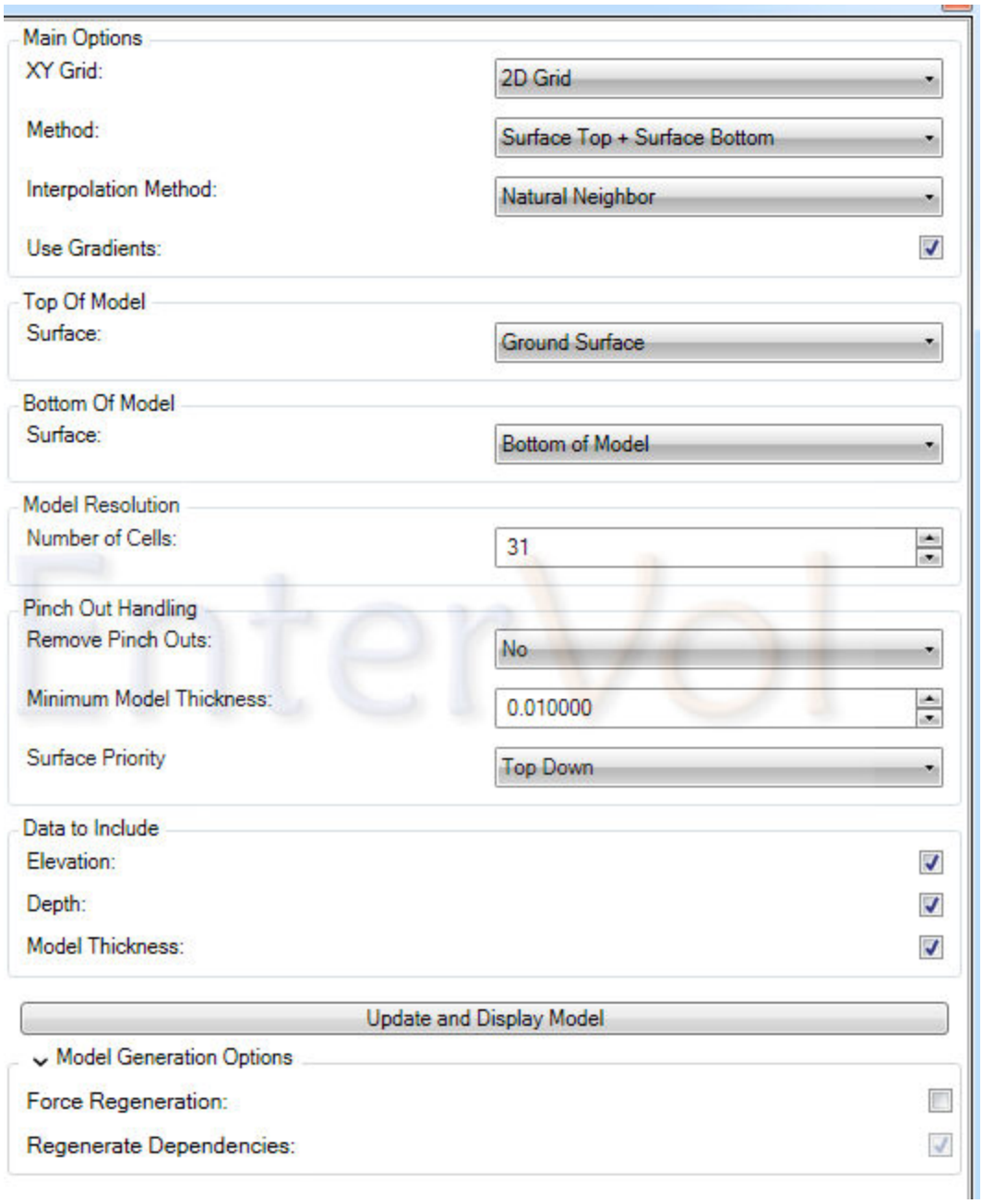



As discussed above, there are 5 basic options for creating the 3D Grid. The simplest is to define an elevation for the top and bottom of the model and the number of cells to span the resulting thickness.



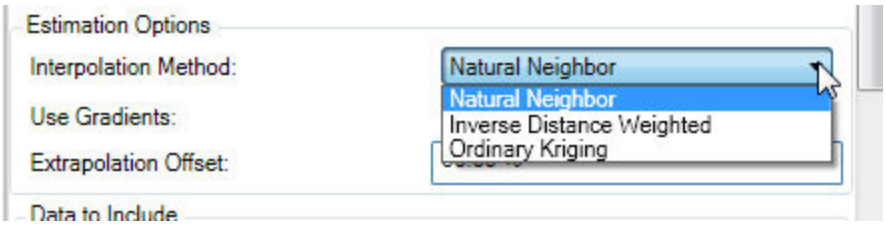

## 3D Grid Generation

3D Grid Generation is one of the processes which has enhanced capabilities when EnterVol Geology is licensed.



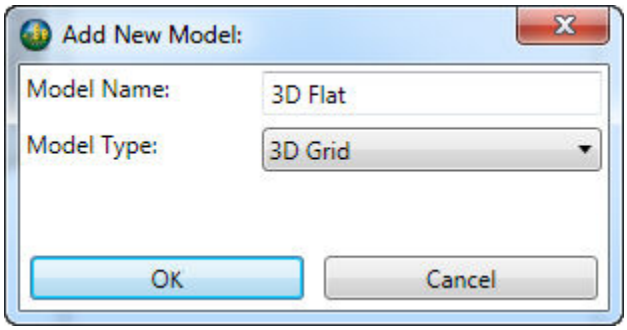

The 3D Grid Generation options are:

- XY Grid: Here you choose from the XY grids that you have created. Your modeling project can have more than one.
- Method:
  - *Flat Top + Flat Bottom* at specified elevations
  - *Surface Top + Flat Bottom*
  - *Flat Top + Surface Bottom*
  - *Constant Thickness*
  - *Surface Top + Surface Bottom (requires EnterVol Geology)*

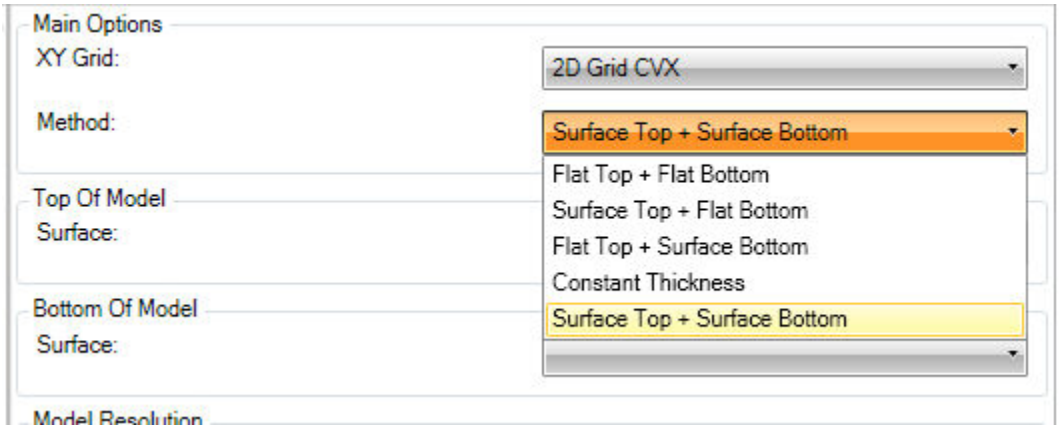



- Interpolation Method: Each method has different options. See the 3D Surface Generation topic for examples.
  - Natural Neighbor is the default and recommended method
  - Inverse Distance Weighted
  - Ordinary Kriging
- Use Gradients: Determines if surface gradients will affect the surface shape. Often makes more realistic surfaces, but occasionally extrapolates in bizarre ways if your data is clustered.
- Top Of Model: Here you select the surface which will define the top of your 3D Grid
- Bottom Of Model: Here you select the surface which will define the bottom of your 3D Grid
- Model Resolution: The number of cells in the vertical direction. If you plan to stratify your model or add analyte data you need a reasonable number of cells (depends on the number of layers and/or amount of data in the z direction).
- Pinch Out Handling
  - Remove Pinch Outs: If layers drop below a threshold thickness they will be removed or will remain as a thin sheet. If your grid will be exported to initialize ground water models you may not want to remove these sections.
  - Minimum Model Thickness: The threshold below which layers will be removed. This will also be the minimum thickness for layers that would otherwise pinch if the above option is set to NO.
  - Surface Priority: If horizons intersect one another, this option determine the hierarchy (precedence) of horizons.
- Data to Include:
  - Elevation (z coordinate)
  - Depth (distance below the ground surface)
  - Model Thickness
- Update and Display Button (creates the model and adds it to your table of contents
- Model Generation Options
  - Force Regeneration: Recomputes all dependent models. In this case the XY grid specified.

Click on the  to create a new model element. Provided that at least one 2D grid has already been created, the option of creating a 3D Grid will be available. Choose a Model Name and hit OK.



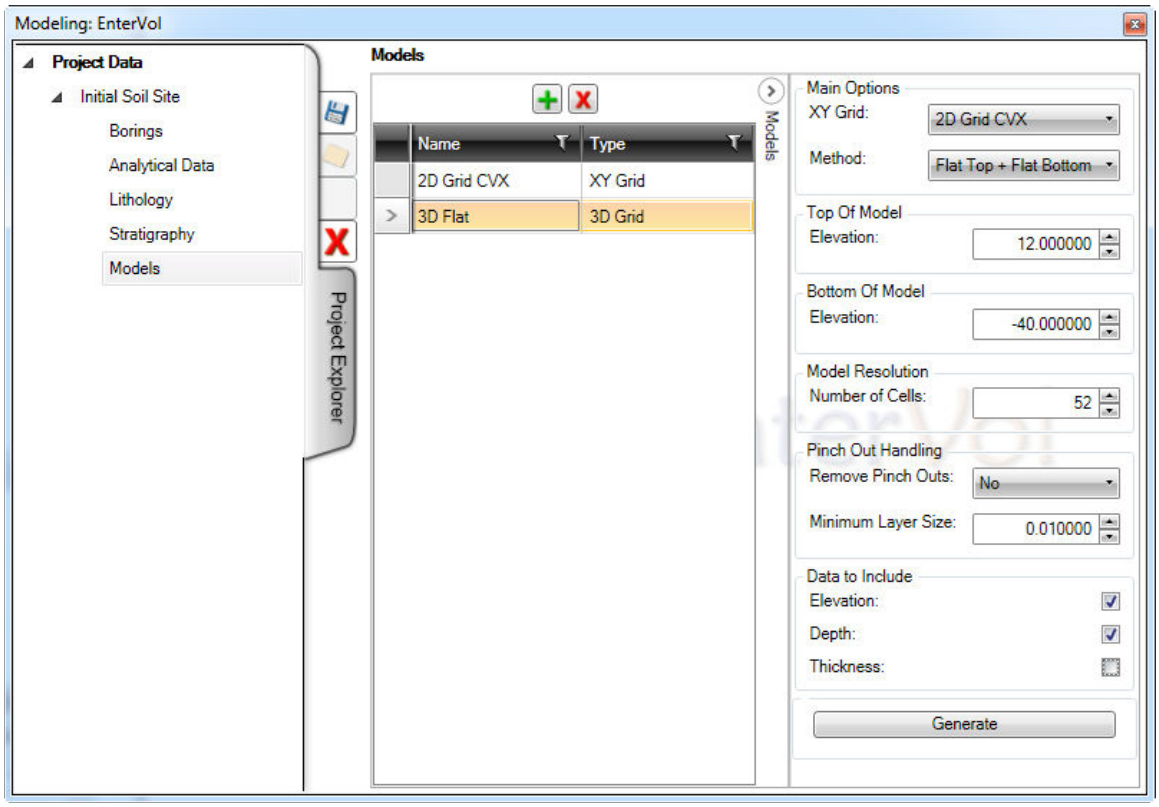

As discussed above, there are 5 basic options for creating the 3D Grid. The simplest is to define an elevation for the top and bottom of the model and the number of cells to span the resulting thickness.




## Flat Top + Flat Bottom

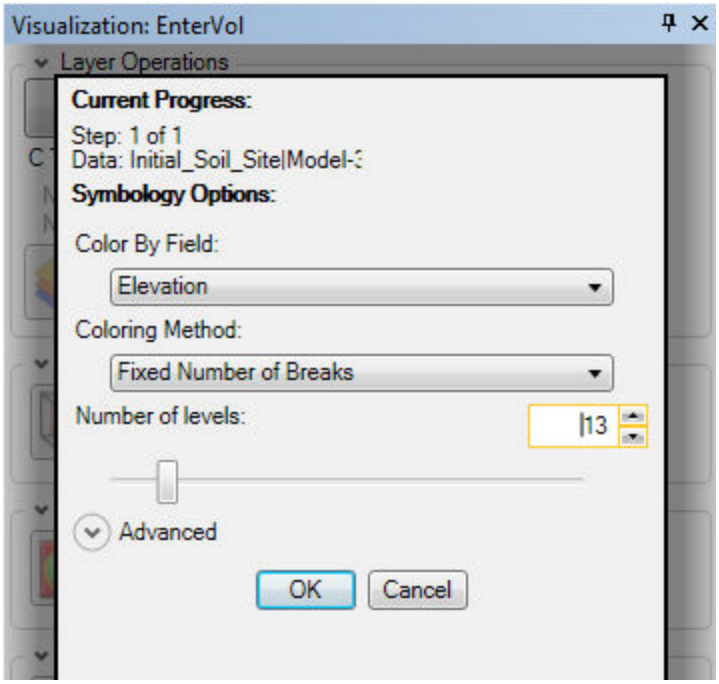
As discussed in the previous topic, there are 5 basic options for creating the 3D Grid. The simplest 3D Grid is defined by an elevation for the top and bottom of the model and the number of cells to span the resulting thickness.

In the example below we create a 3D grid using the 2D grid previously created spanning elevations from -40 to +12. We'll change the default model resolution to be 52 so that each of cells will represent 1.0 units in Z. It is not required to make the cells a particular size.

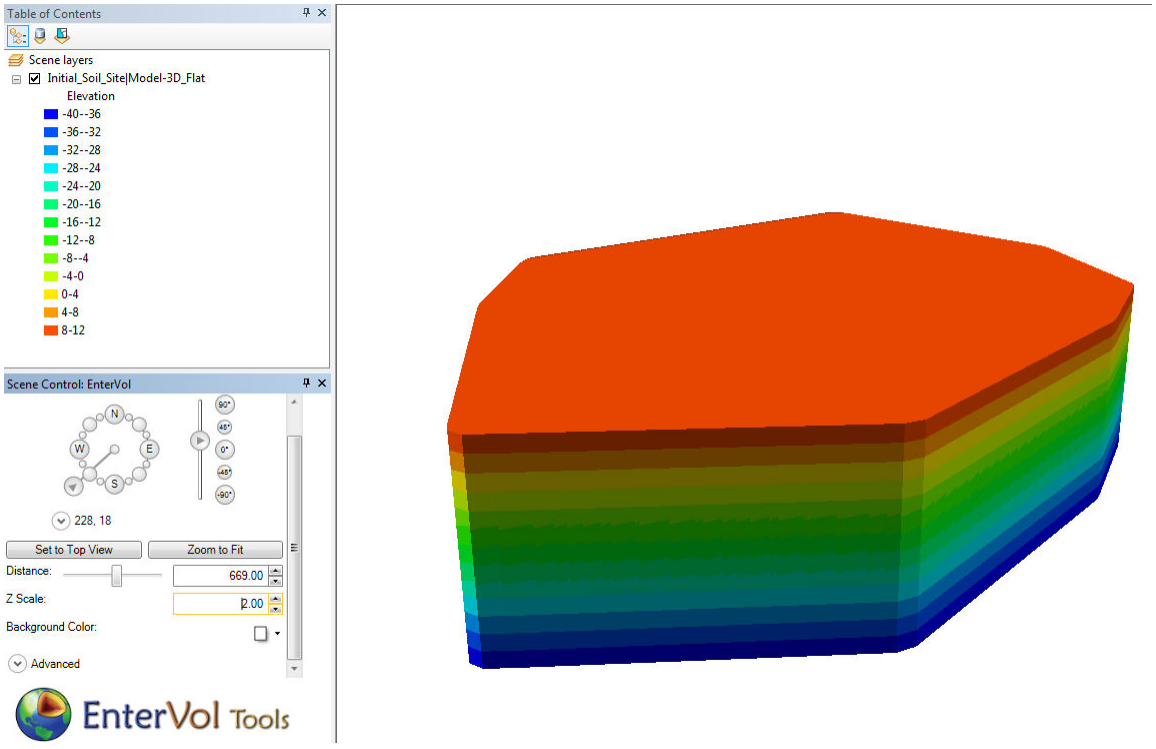



When we click on Generate, the grid will be created and the following window will appear in the Visualization: EnterVol panel.

Since our Z elevations will span 52, let's choose 13 levels for the Symbology so that each break will be 2 meters.



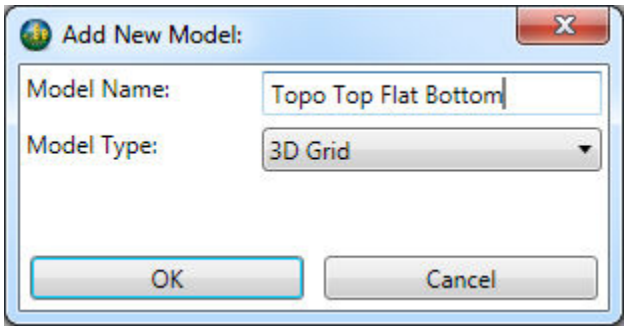

When we select our settings and click OK, the 3D Grid will be added to the project.



## Surface Top + Flat Bottom

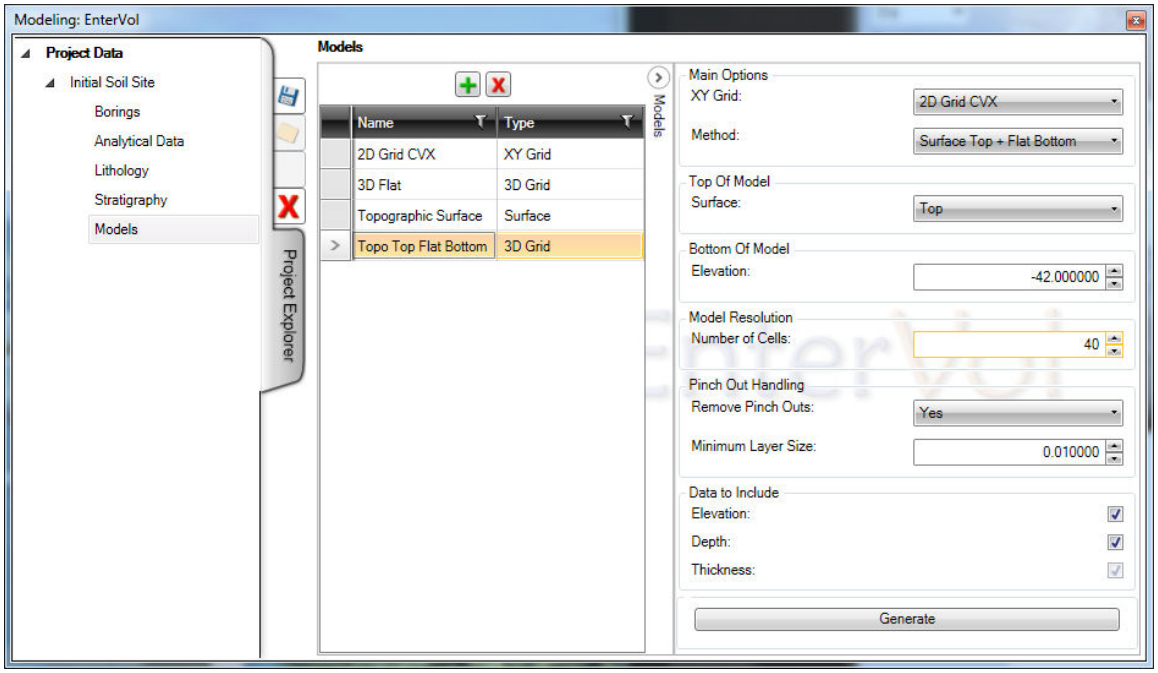
A much more useful 3D Grid can be defined by a surface determined by ground survey data and an elevation for the bottom of the model.

Click on the  to create a new model element with surface topography as our upper bounding surface and a flat bottom at a constant elevation.

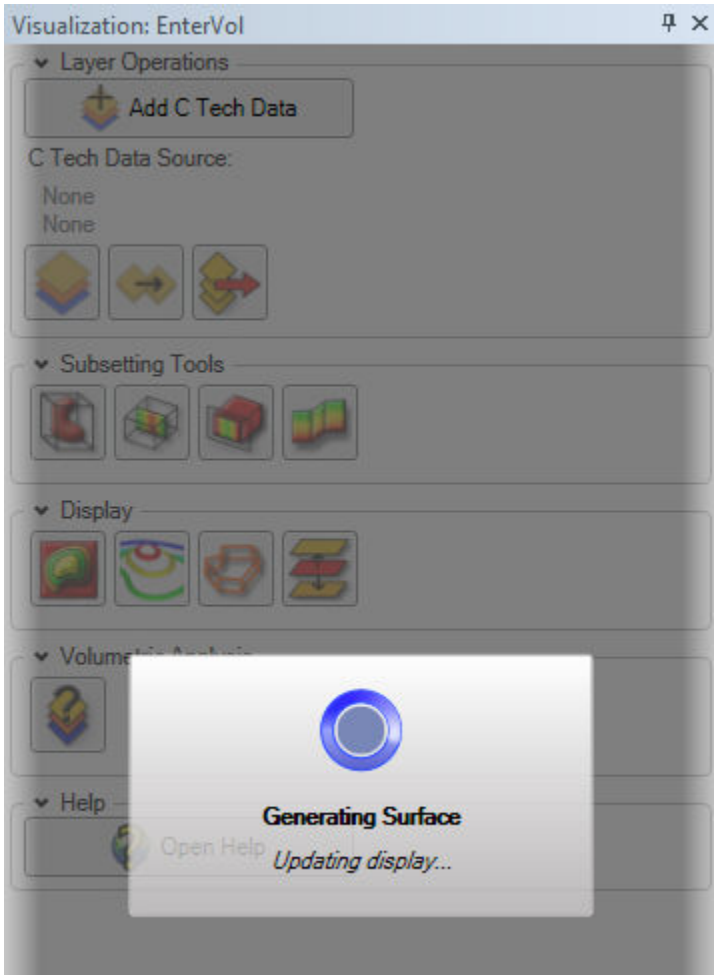



Adjust your settings to match those below. We've chosen the "Top" surface as the Top of Model and an elevation of -42 as the Bottom.

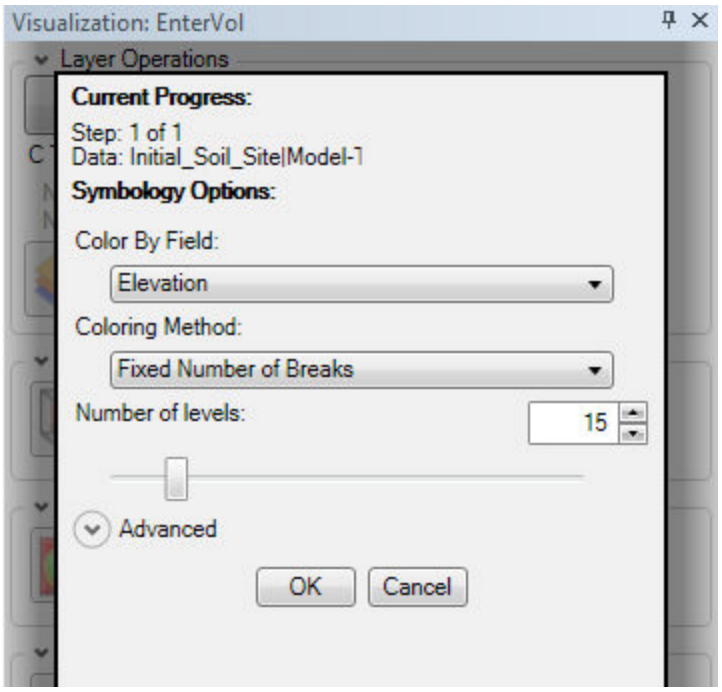
The number of cells (in the Z direction) was increased to 40. All other parameters are defaults.



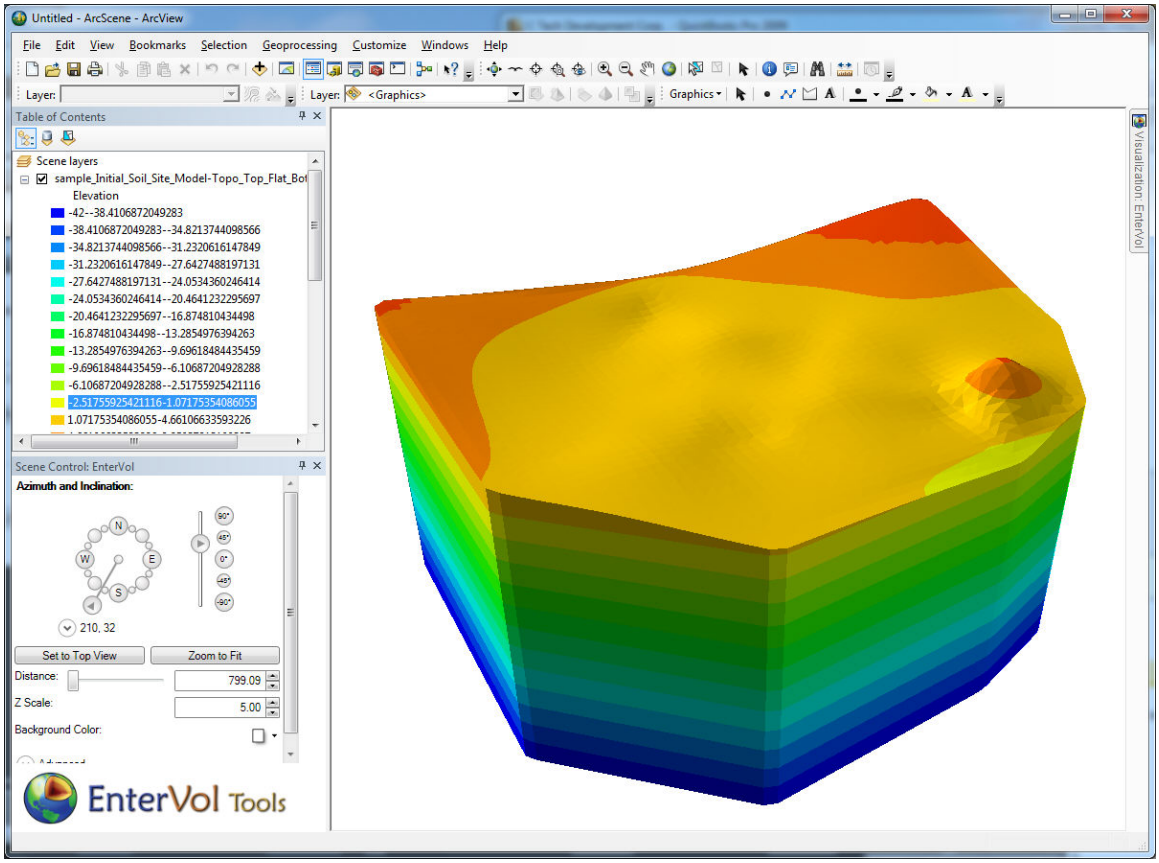
When we click on Generate,




When the grid is created and the following window will appear in the Visualization: EnterVol panel.

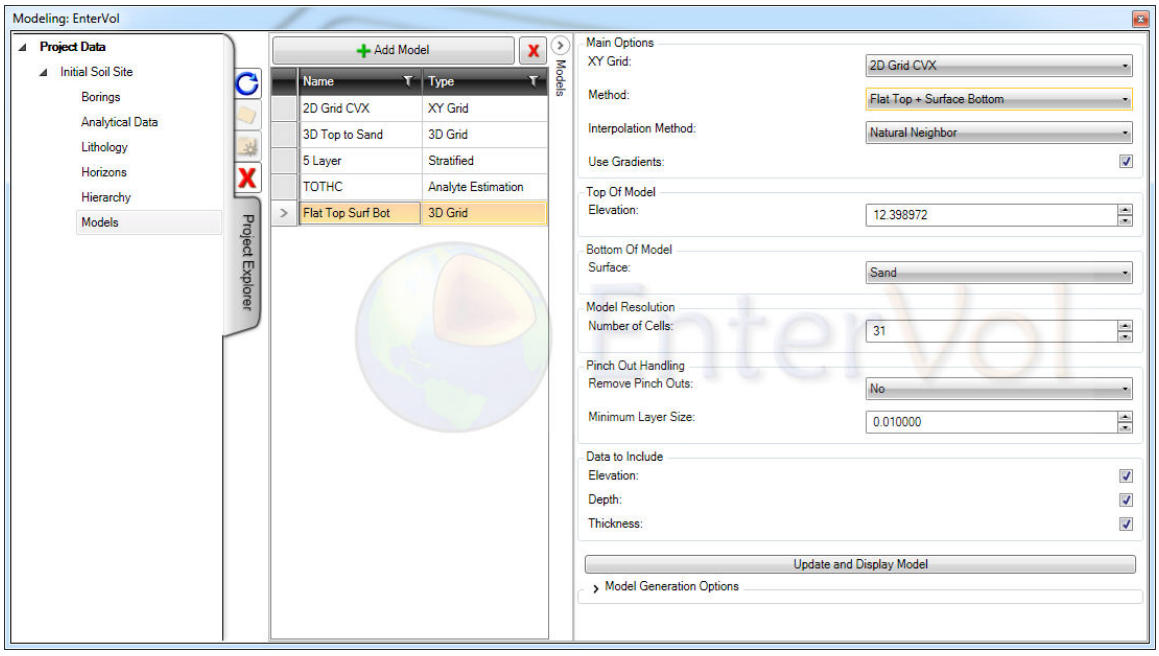



When we select our settings and click OK, the 3D Grid will be added to the project.

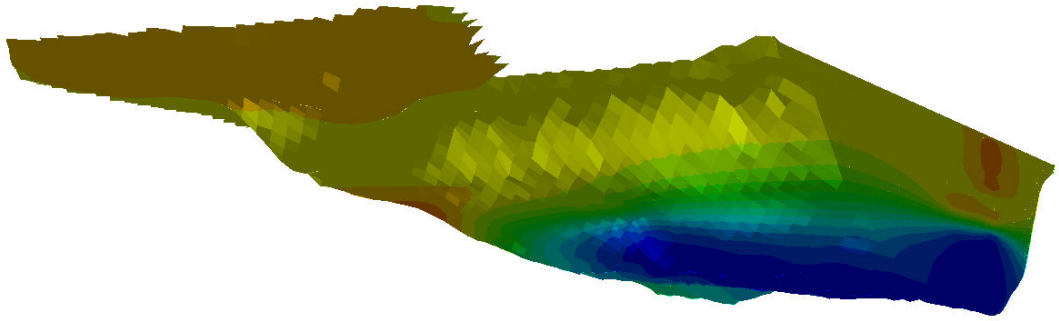


## Flat Top + Surface Bottom

Having a flat top and a variable surface bottom is the correct grid type for most all water bodies (lakes, rivers, reservoirs, etc.).



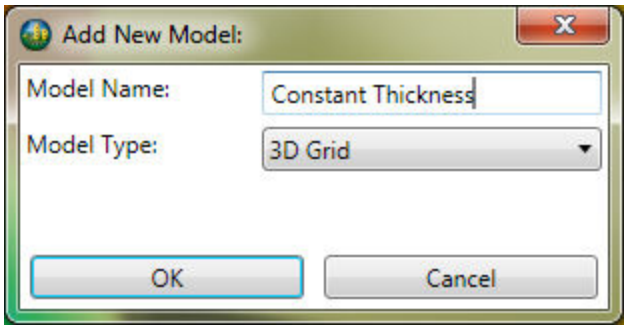

Below is a model of a lake in Saskatchewan, Canada which has been cut through a region with high dissolved oxygen levels.



## Constant Thickness

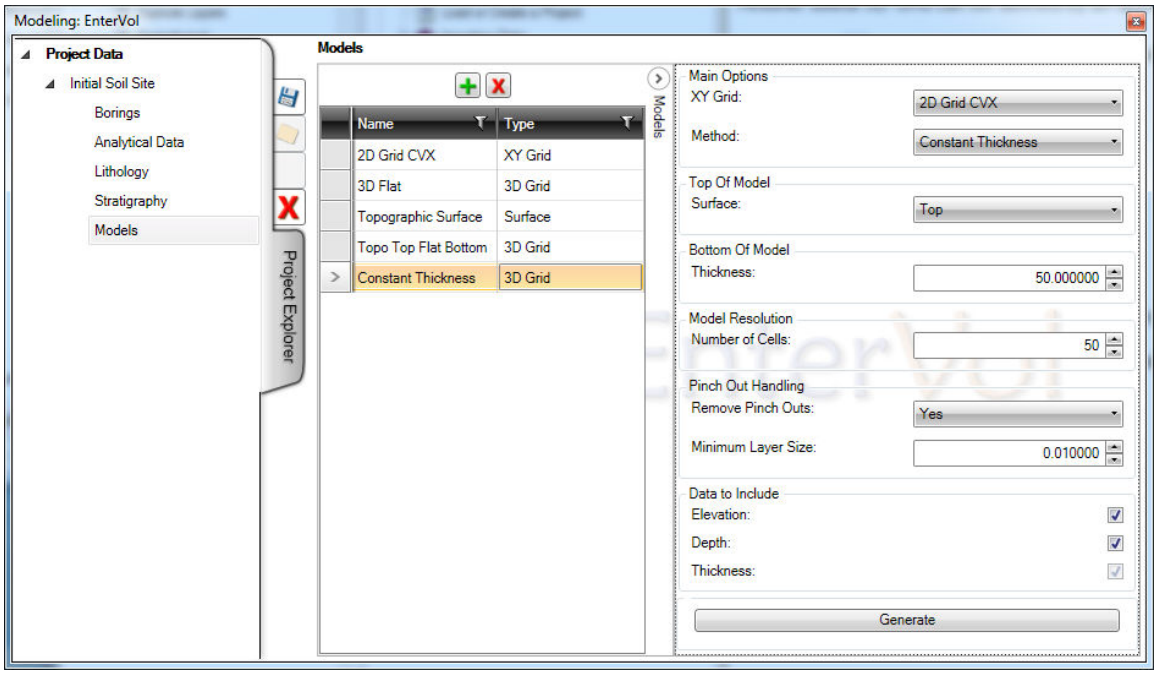
Another useful 3D Grid can be defined by an upper surface and a constant thickness (e.g. constant depth below the upper surface).

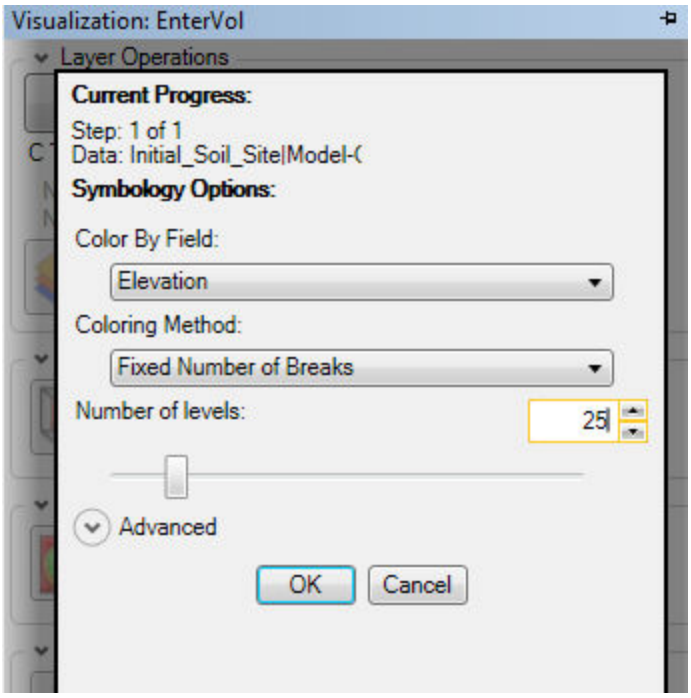
Click on the  to create a new model element with surface topography as our upper bounding surface and a constant depth.



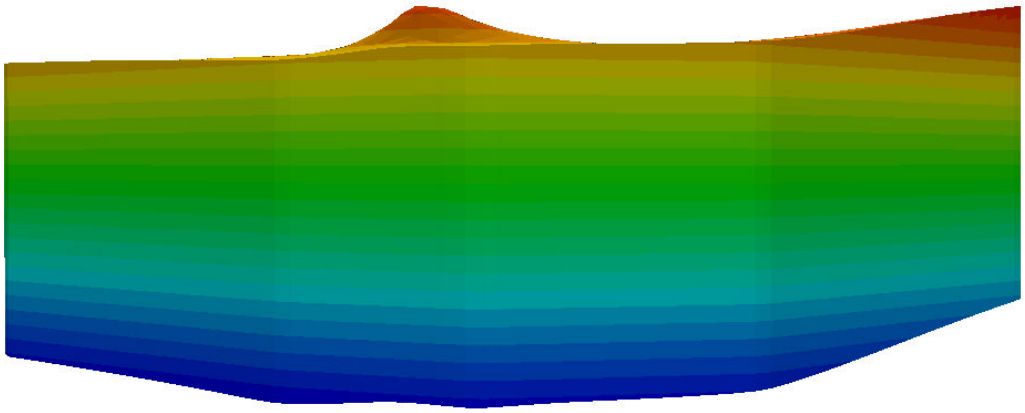

Adjust your settings to match those below. We've chosen the "Top" surface as the Top of Model and a thickness of 50.

The number of cells (in the Z direction) was increased to 50. All other parameters are defaults.






You can see in the view below (Z Scale - 3) that the model is constant thickness.

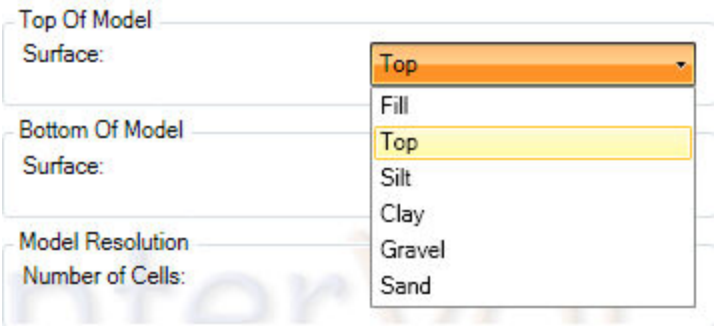


## Surface Top + Surface Bottom

**THIS TOPIC REQUIRES AN EnterVol Geology LICENSE**

EnterVol Geology adds the ability to create 3D grids which are defined by two arbitrary surfaces.

For our example database, we have 6 surfaces. Any of these can be selected as the top of our grid

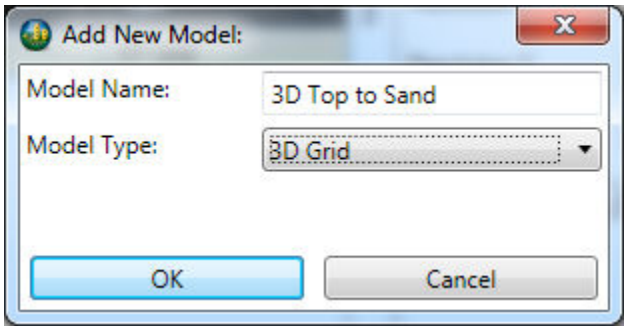


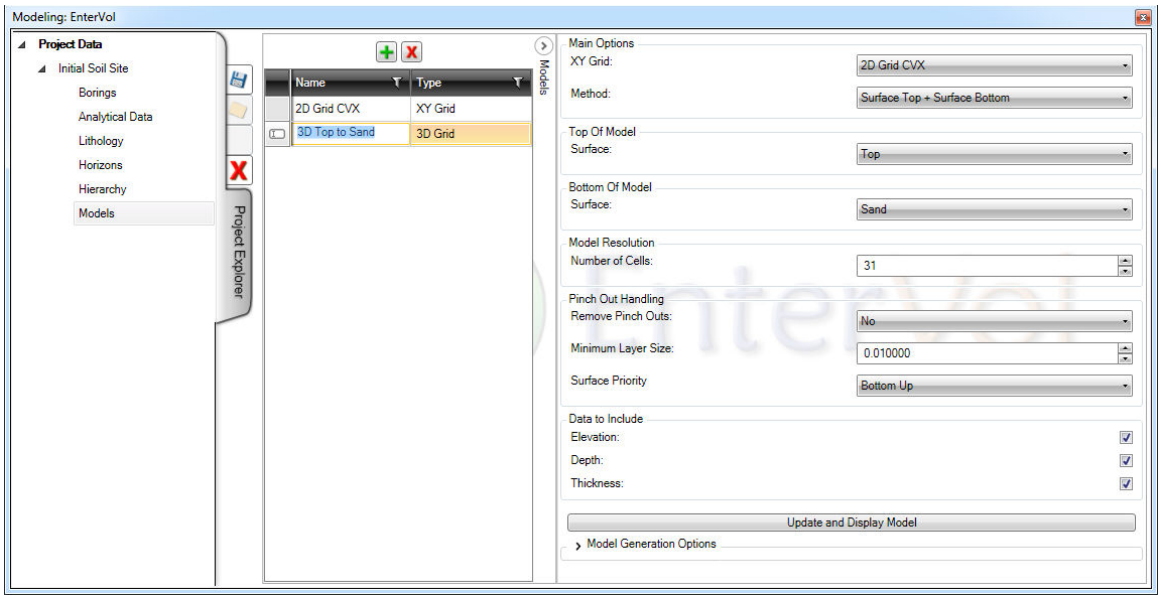

provided we choose a lower surface as the bottom. For these 6 surfaces the hierarchy is:

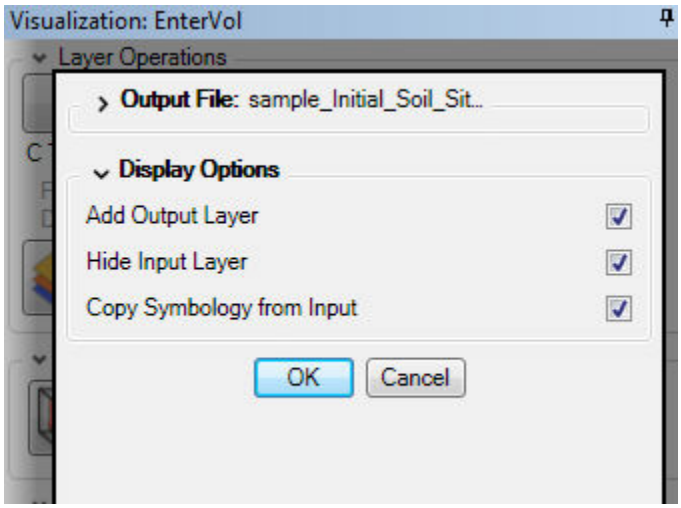
1. Top
2. Fill (bottom)
3. Silt (bottom)
4. Clay (bottom)
5. Gravel (bottom)
6. Sand (bottom)

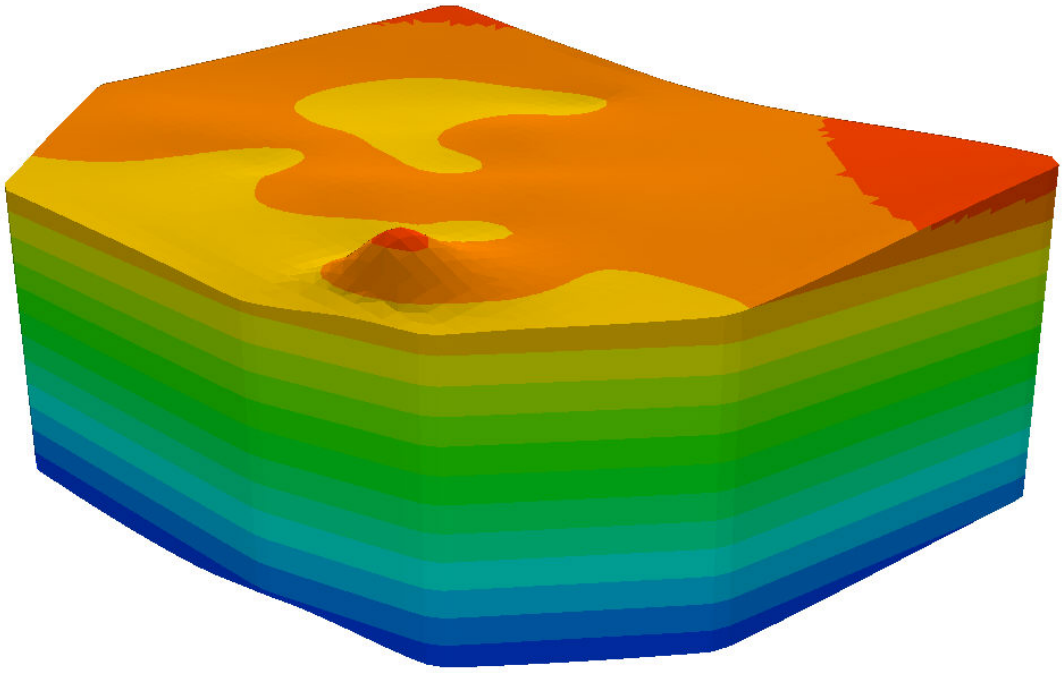
If we chose Top and Fill we would create a model of the Fill layer. If we use Gravel and Sand we'd be building the Sand layer, since the "top of Sand" is the bottom of Gravel.

Let's choose Top and Sand so we create a model that includes all 5 layers of our stratigraphy. We can later use EnterVol Geology to create a Stratified Grid.

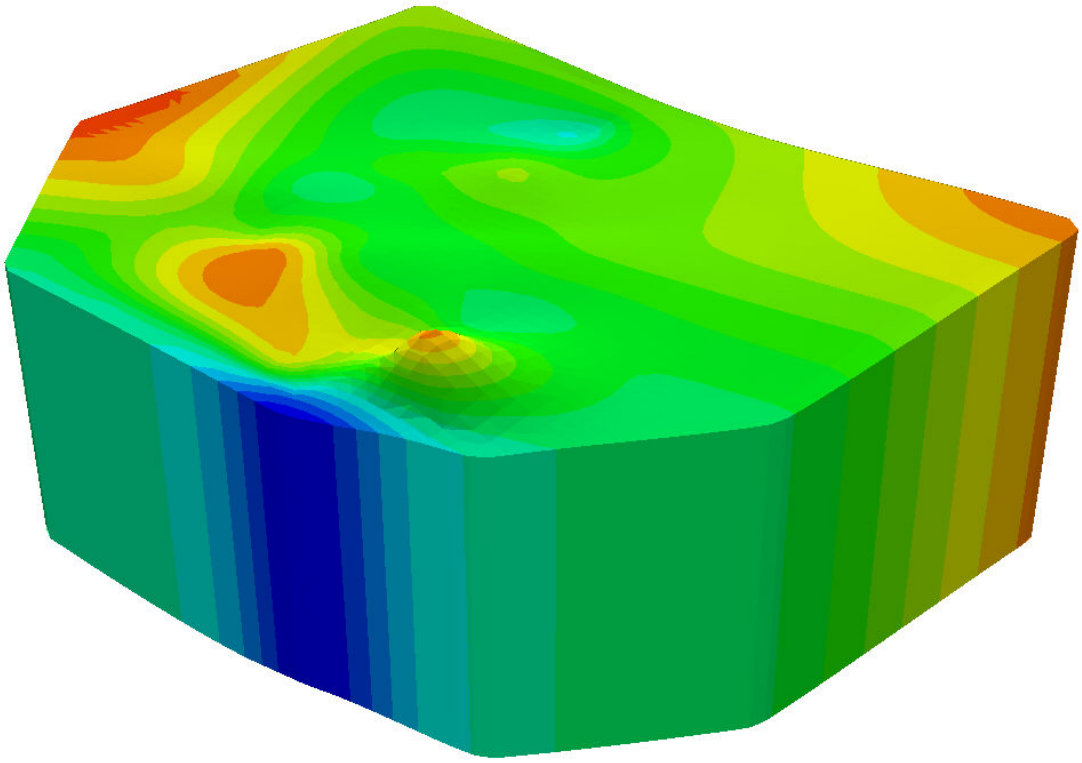



Modeling: EnterVol
Project Data
Initial Soil Site
Borings
Analytical Data
Lithology
Horizons
Hierarchy
Models
Project Explorer
Name
Type
2D Grid CVX
XY Grid
3D Top to Sand
3D Grid
Models
Main Options
XY Grid:
2D Grid CVX
Method:
Surface Top + Surface Bottom
Top Of Model
Surface:
Top
Bottom Of Model
Surface:
Sand
Model Resolution
Number of Cells:
31
Pinch Out Handling
Remove Pinch Outs:
No
Minimum Layer Size:
0.010000
Surface Priority
Bottom Up
Data to Include
Elevation:
Depth:
Thickness:
Update and Display Model
Model Generation Options


Visualization: EnterVol
Layer Operations
Output File: sample_Initial_Soil_Sit...
Display Options
Add Output Layer
Hide Input Layer
Copy Symbology from Input
OK
Cancel



One of the best ways to appreciate that this grid has variable surfaces on top and bottom is to compare the elevation contours above to the contours of Thickness shown below. To create this, choose the model above and Set Symbology

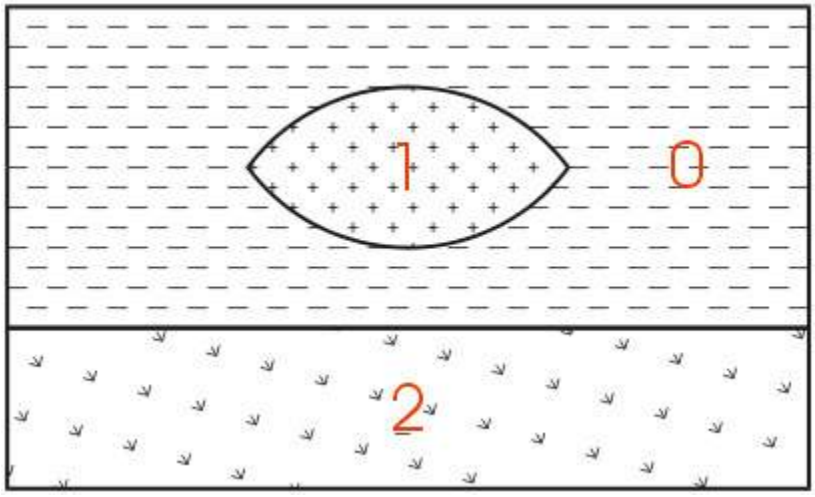


## Geology: Stratified Models

**THIS TOPIC REQUIRES AN EnterVol Geology LICENSE**

EnterVol allows you to use stratigraphic data to further refine and subset your 3D grids. Stratigraphic data represents surfaces which form a Geologic Hierarchy.

Simply stated, hierarchy requires that all geologic horizons throughout the domain be ordered from top to bottom and that a consistent hierarchy be used for all borings. At first, it may not seem possible for a uniform layer hierarchy to be applicable for all borings. Layers often pinch out or exist as localized lenses. Also layers may be continuous in one portion of the domain, but may be split by another layer in other portions of the domain. However, all of these scenarios and many others can be easily modeled using a hierarchical approach.



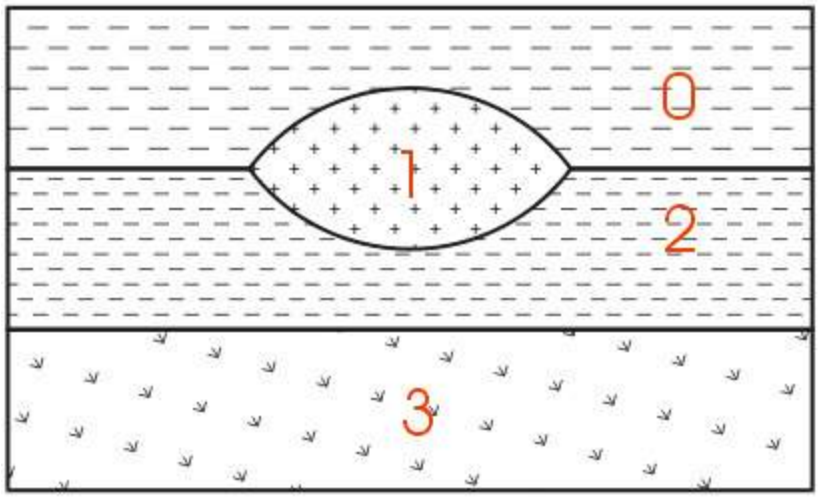

The easiest way to describe geologic hierarchy is with an example. Consider the example above of a clay lens in sand with gravel below.

Imagine borings on the left and right sides of the domain and one in the center. Those outside the center would not detect the clay lens. On the sides, it appears that there are only two layers in the hierarchy, but in the middle there are three materials and four layers.

EnterVol's hierarchical geologic modeling approach accommodates the clay lens by treating every layer as a sedimentary layer. Because we can accommodate "pinching out" layers (making the thickness of layers ZERO) we are able to produce most geologic structures with this approach.

Geologic layer hierarchy requires that we treat this domain as 4 geologic layers. These layers would be

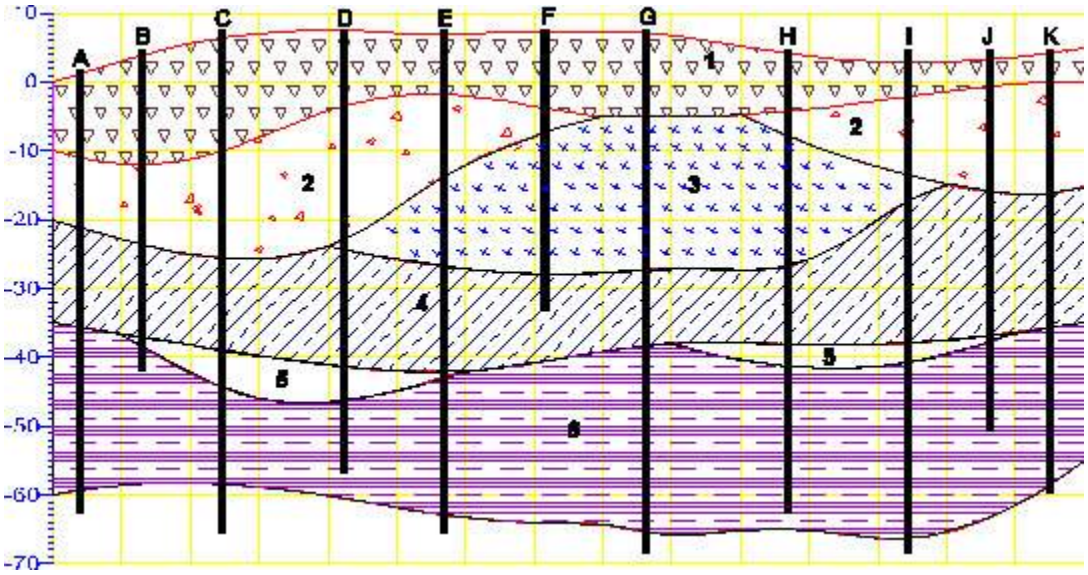
1. Upper Sand (0)
2. Clay (1)
3. Lower Sand (2)
4. Gravel (3).




If desired, both Upper and Lower Sand can have identical symbology in the final output.

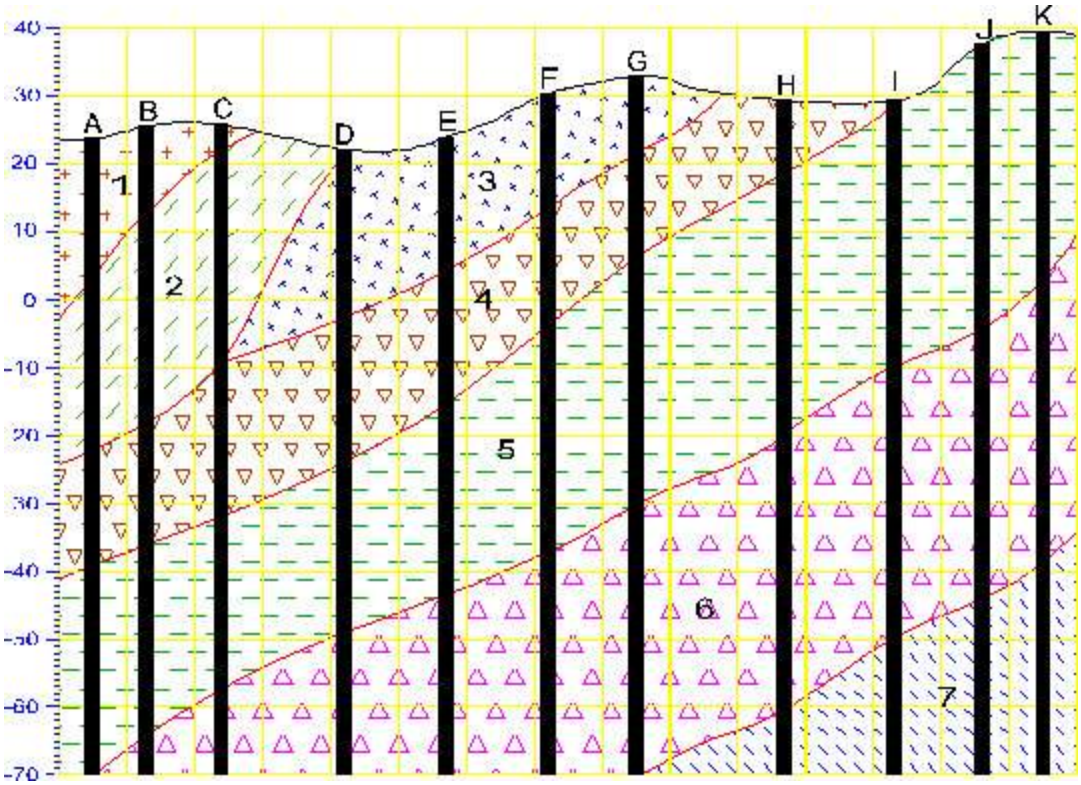
For those sites that can be described using the above method, it remains the best approach for building a 3D geologic model. Each layer has smooth boundaries and the layers (by nature of hierarchy) can be exploded apart to reveal the individual layer surface features. In the above example, the numbers represent the layer numbers for this site (even though the layers labeled 0 and 2 are both sand). Two examples of much more complex sites that are best described by this original approach are shown below.

Stratified Model Example: Sedimentary Layers and Lenses




Stratified Model Example: Outcrop of Dipping Strata

EnterVol is not limited to sedimentary layers or lenses. The figure below shows a cross-section through an outcrop of dipping geologic strata. EnterVol can easily model the layers truncating on the top ground surface.

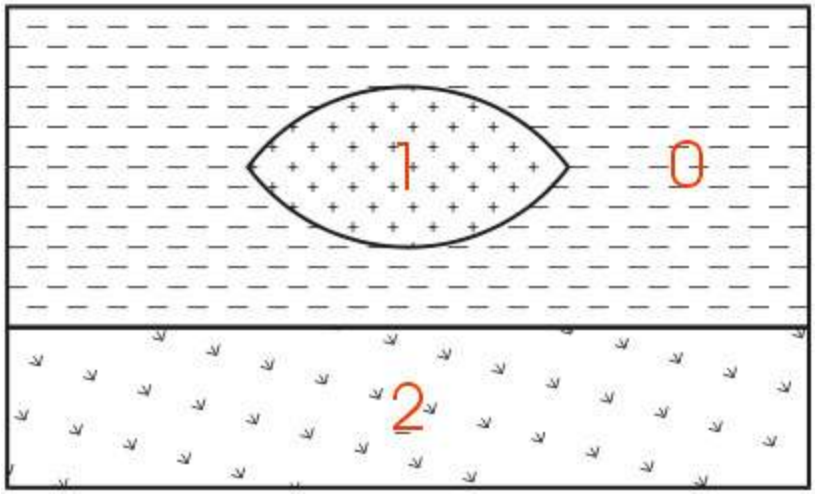



## Geology: Stratified Models

**THIS TOPIC REQUIRES AN EnterVol Geology LICENSE**

EnterVol allows you to use stratigraphic data to further refine and subset your 3D grids. Stratigraphic data represents surfaces which form a Geologic Hierarchy.

Simply stated, hierarchy requires that all geologic horizons throughout the domain be ordered from top to bottom and that a consistent hierarchy be used for all borings. At first, it may not seem possible for a uniform layer hierarchy to be applicable for all borings. Layers often pinch out or exist as localized lenses. Also layers may be continuous in one portion of the domain, but may be split by another layer in other portions of the domain. However, all of these scenarios and many others can be easily modeled using a hierarchical approach.



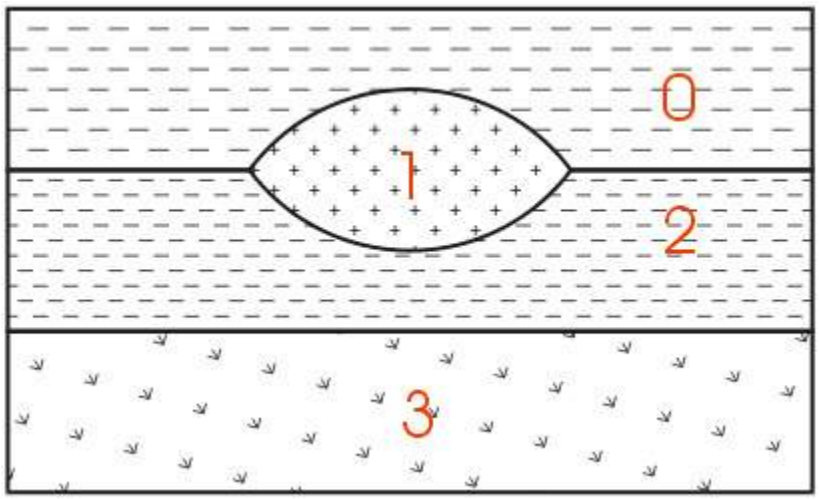

The easiest way to describe geologic hierarchy is with an example. Consider the example above of a clay lens in sand with gravel below.

Imagine borings on the left and right sides of the domain and one in the center. Those outside the center would not detect the clay lens. On the sides, it appears that there are only two layers in the hierarchy, but in the middle there are three materials and four layers.

EnterVol's hierarchical geologic modeling approach accommodates the clay lens by treating every layer as a sedimentary layer. Because we can accommodate "pinching out" layers (making the thickness of layers ZERO) we are able to produce most geologic structures with this approach.

Geologic layer hierarchy requires that we treat this domain as 4 geologic layers. These layers would be

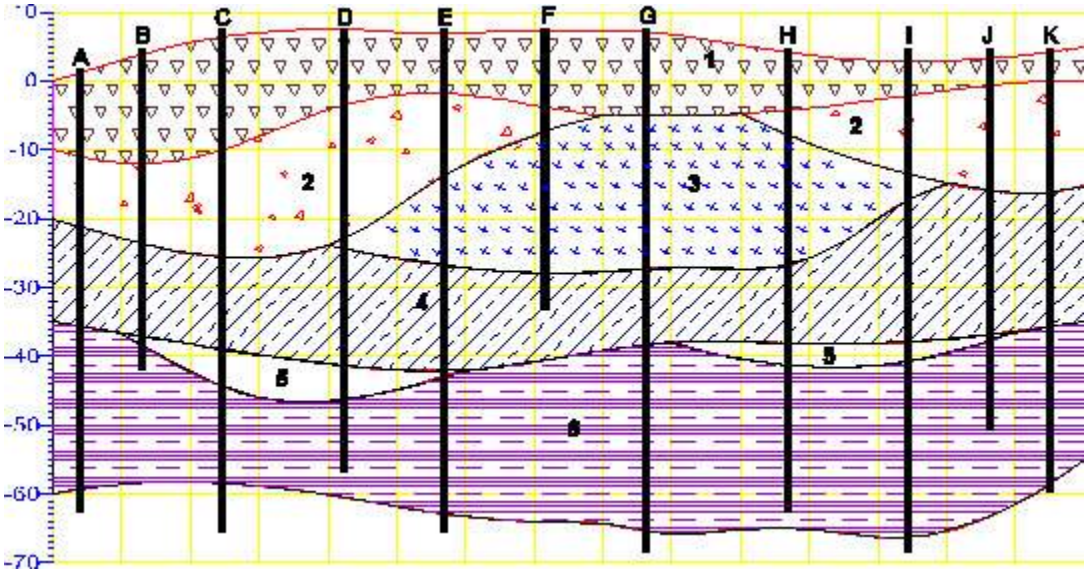
1. Upper Sand (0)
2. Clay (1)
3. Lower Sand (2)
4. Gravel (3).



If desired, both Upper and Lower Sand can have identical symbology in the final output.

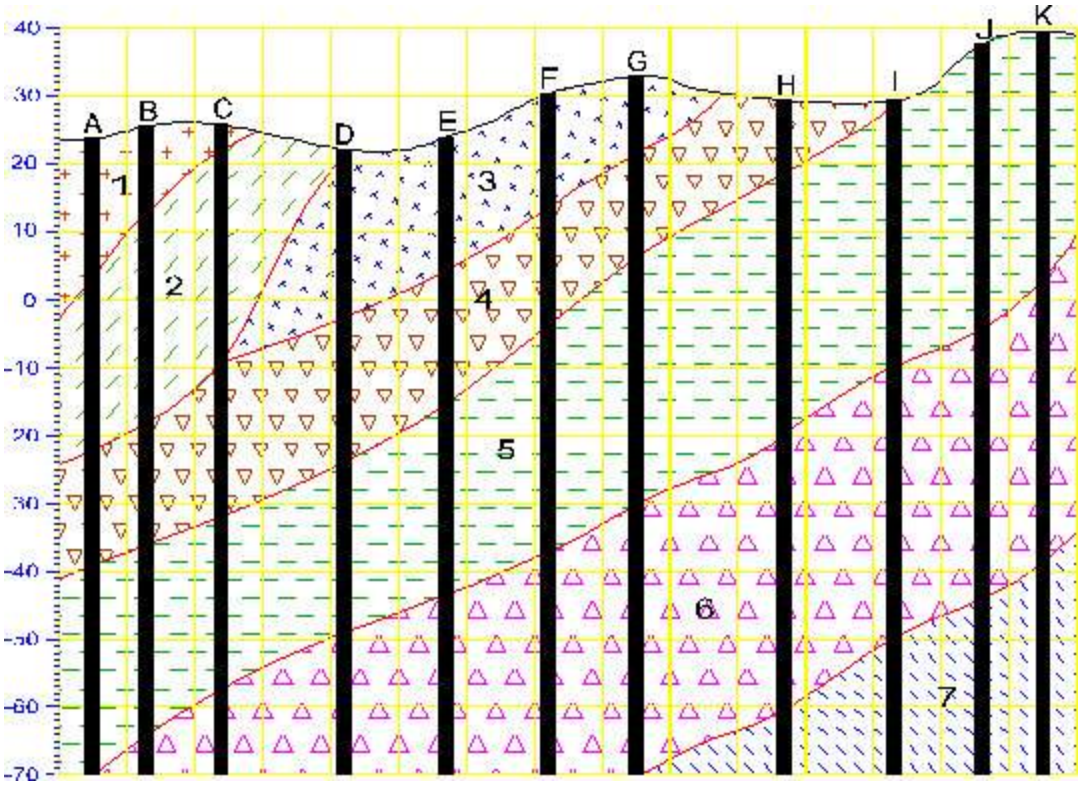
For those sites that can be described using the above method, it remains the best approach for building a 3D geologic model. Each layer has smooth boundaries and the layers (by nature of hierarchy) can be exploded apart to reveal the individual layer surface features. In the above example, the numbers represent the layer numbers for this site (even though the layers labeled 0 and 2 are both sand). Two examples of much more complex sites that are best described by this original approach are shown below.

Stratified Model Example: Sedimentary Layers and Lenses



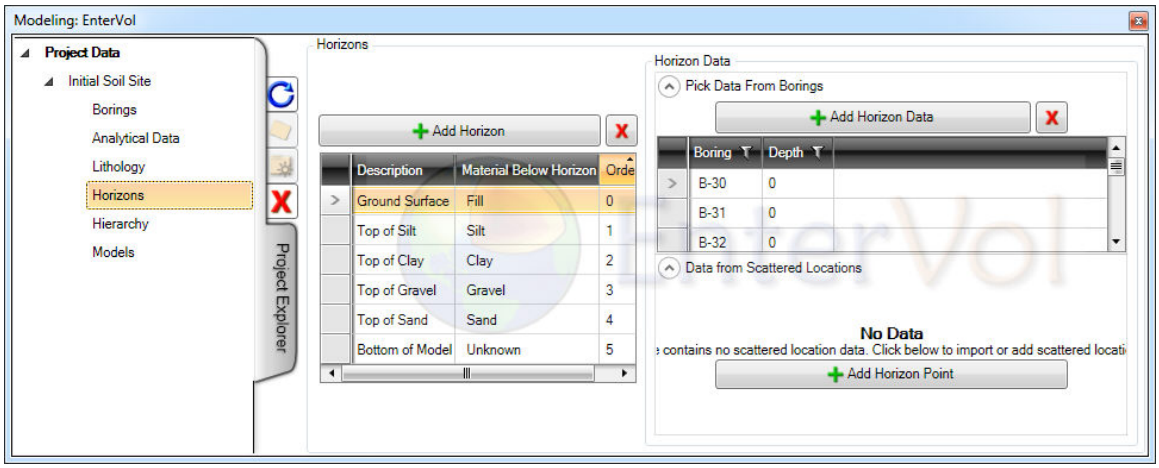

Stratified Model Example: Outcrop of Dipping Strata

EnterVol is not limited to sedimentary layers or lenses. The figure below shows a cross-section through an outcrop of dipping geologic strata. EnterVol can easily model the layers truncating on the top ground surface.




## Horizons

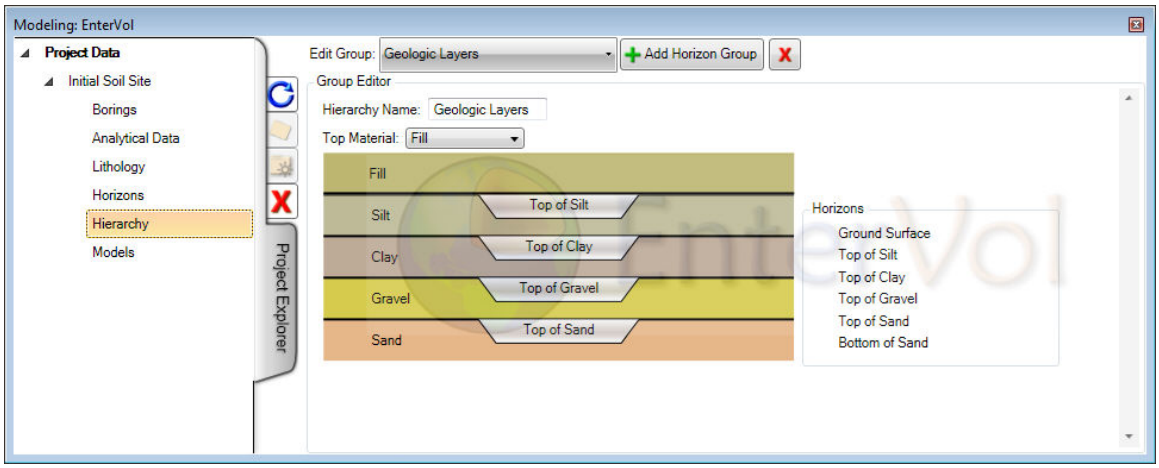
Selecting Horizons under your Site in the Modeling Window will bring up the following:




## Hierarchy

Though it may seem that the settings in Hierarchy are redundant with those in Horizons, this window provides some important capabilities. The horizon ordering in Hierarchy are those that are actually used when making a Stratigraphic model.

The *Hierarchy* section lists the stratigraphic layers in order and all horizons used to define stratigraphy. By selecting the horizons in the display below, you can move the order of horizons or eliminate layers in the hierarchy. Removing a horizon eliminates the layer below it.




Removing the horizon "Top of Clay" above, would eliminate clay from the hierarchy resulting in a Silt layer that extends from *Top of Silt* down to the *Top of Gravel.*

## Geology: Build Stratified Grid

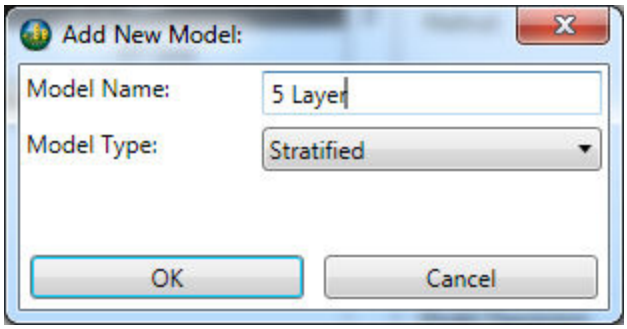
Once you have your data that describes stratigraphy in your site, the grid can be refined in amazing ways.

What can we do?

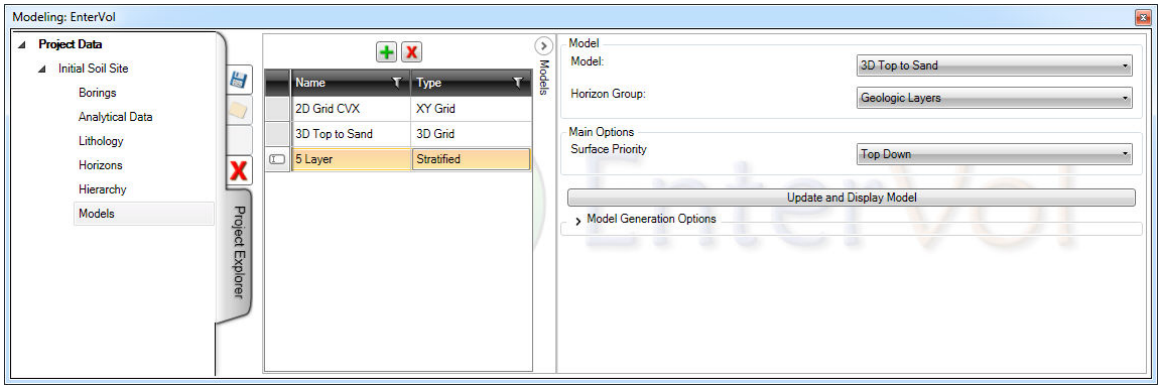
- Refine our 3D grid into stratigraphic layers.
- Segregate the saturated (below water table) and unsaturated (above water table) zones
- Segregate the stratigraphic layers by whether they are saturated or unsaturated
- Remove excavation pits and/or compute excavation volumes and masses

We can now take the 3D Grid which was defined from our Top surface down to the bottom of Sand and refine it to include all 5 of our stratigraphic layers.

Create a new Model:

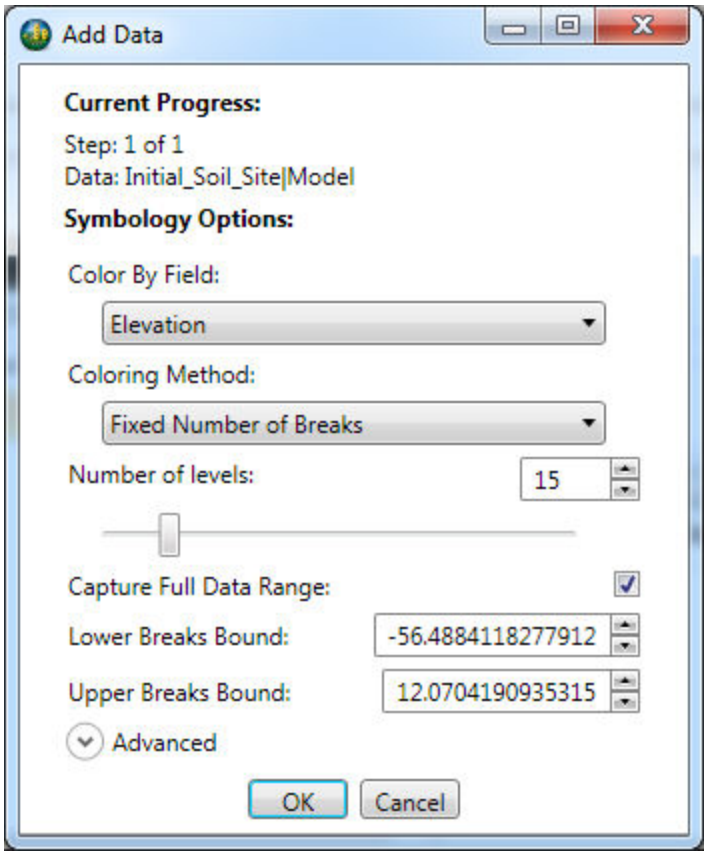



The window will now show:

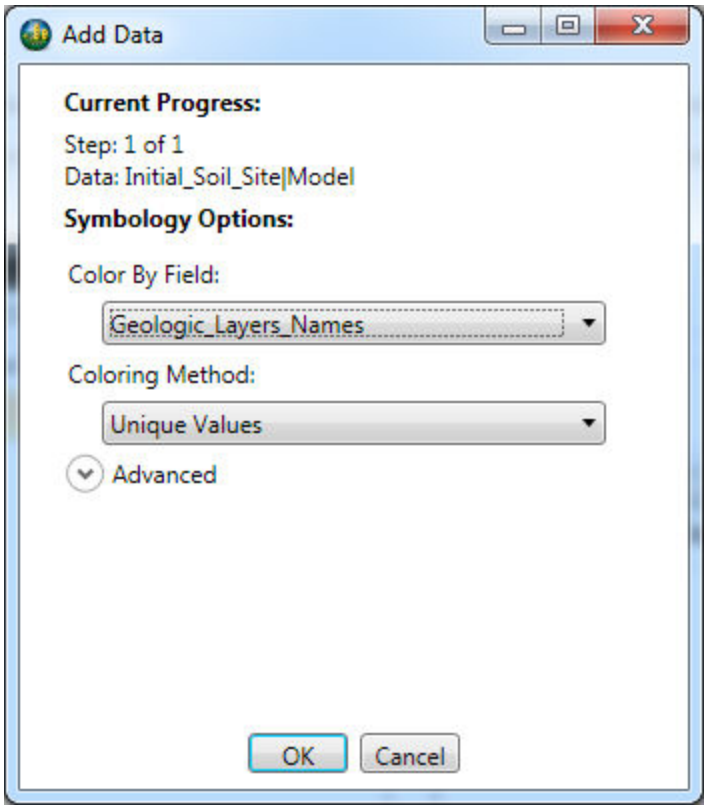


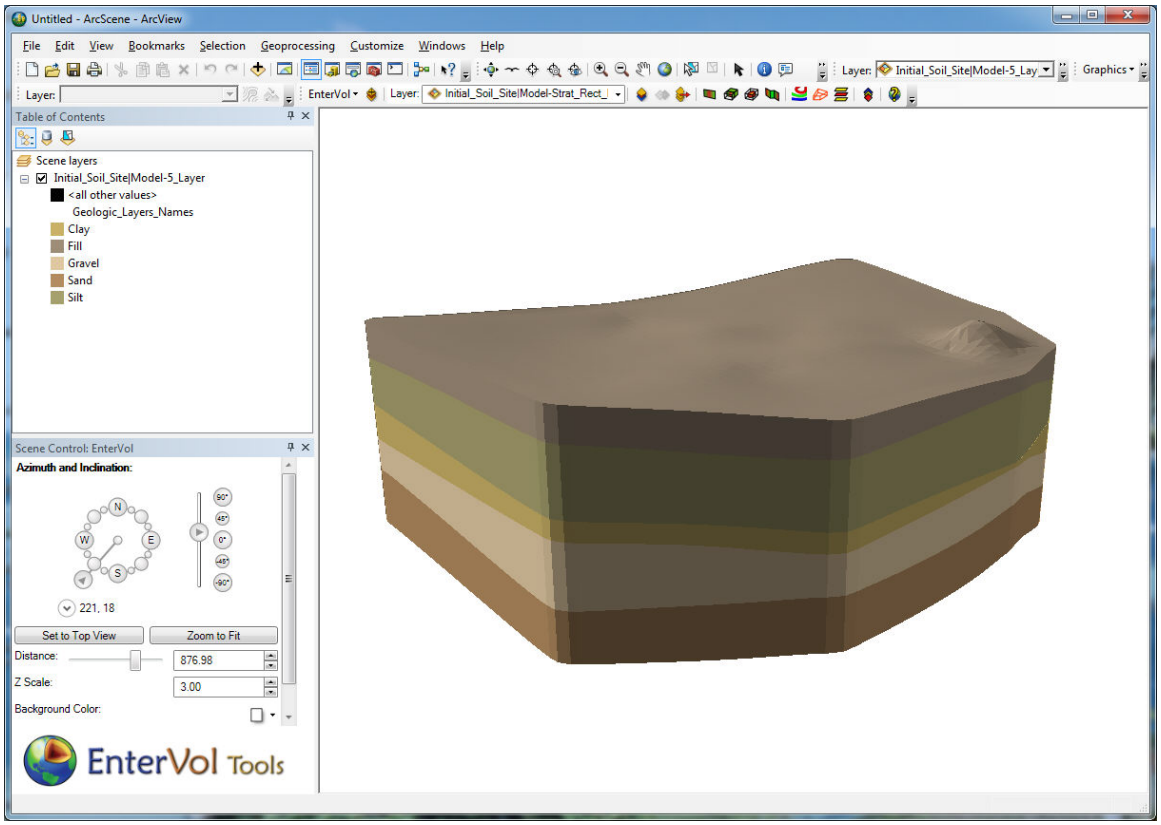

Click on *Update and Display Model*

The window below will appear to specify symbology:




We'll change the "Color By Field" to be our Layer names


Add Data
Current Progress:
Step: 1 of 1
Data: Initial_Soil_Site|Model
Symbology Options:
Color By Field:
Geologic_Layers_Names
Coloring Method:
Unique Values
Advanced
OK
Cancel

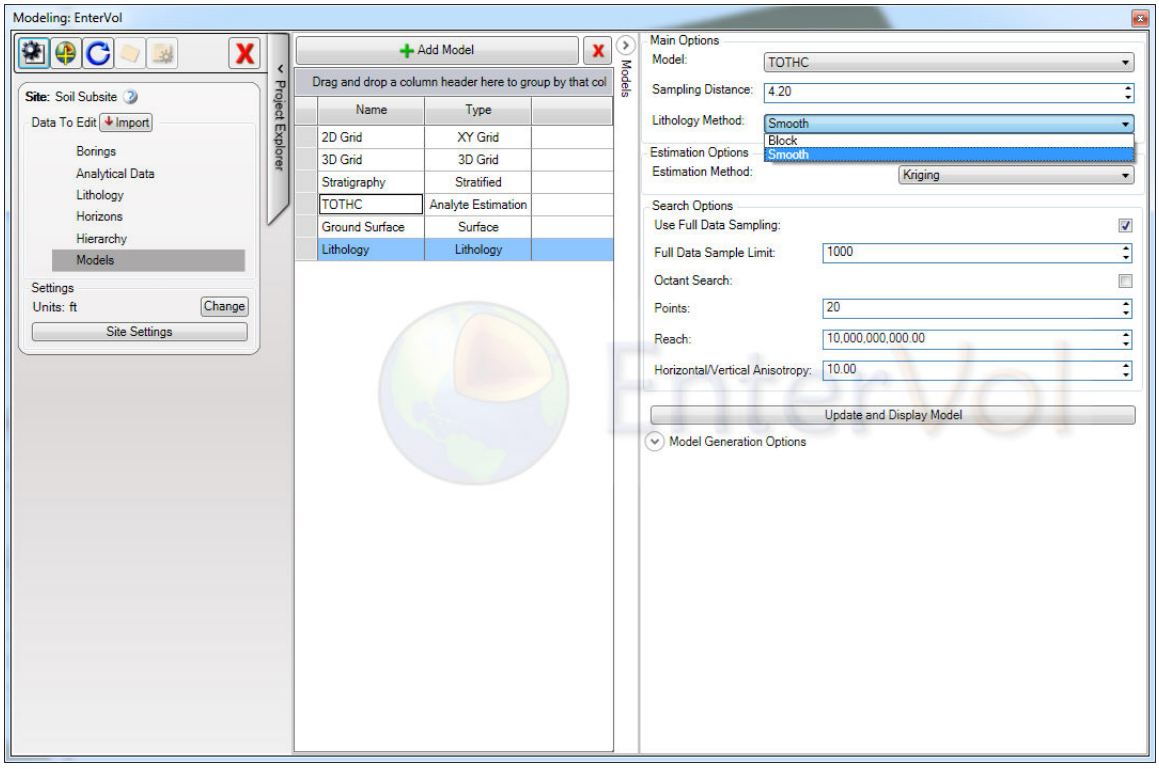

Untitled - ArcScene - ArcView
File Edit View Bookmarks Selection Geoprocessing Customize Windows Help
Table of Contents
Scene layers
Initial_Soil_Site|Model-5_Layer
<all other values>
Geologic_Layers_Names
Clay
Fill
Gravel
Sand
Silt
Scene Control: EnterVol
Azimuth and Inclination:
Set to Top View
Zoom to Fit
Distance:
876.98
Z Scale:
3.00
Background Color:
EnterVol Tools

## Geology: Lithology

**THIS TOPIC WILL REQUIRES AN EnterVol Geology LICENSE**

This geologic modeling approach uses lithology data directly to compute the probability for each material throughout the grid. This is a completely different paradigm than a stratified model which is created using geologic horizons.

With Lithology modeling, the distances to surrounding borings and the nearest materials will determine the material assignments. In contrast, Stratigraphic models assign materials based on geologic horizon surfaces. When materials should not be correlated by continuous surfaces, Lithology modeling is best.




In the Lithology modeling window above you can see the options:

**Model** determines which grid or existing model will be used as the foundation for adding Lithology data.

**Sampling Distance** is a physical distance in your site units (e.g. meters or feet) that is used to convert the intervals of lithology data into discrete points for the material assignment process. Smaller distances would generally increase accuracy, but at the expense of compute time and memory

usage. This distance should generally never be set smaller than the vertical size of your model cells.

The **Lithology Method** selector includes two options:

- **Block** is similar to EVS-PRO's indicator_geology module. The material having the highest probability (for an individual cell) is assigned to the cell. All of the individual material probabilities are also provided as additional cell data components. This will allow you to identify regions where the material assignment is somewhat ambiguous. The resulting model has a BLOCKY appearance because lithology is not refined to a sub-cell level.
- **Smooth** uses algorithms unique to C Tech and EnterVol. The resulting model has smooth boundaries between materials because lithology is refined to a sub-cell level. This refinement takes more time and results in increased effective resolution and greater memory requirements. The number of materials represented in your lithology has a profound affect on the model creation time and memory requirements.

The **Estimation Method** selector includes three options:

- Nearest Neighbor is a Quick Method which assigns the geologic material to the nearest geologic material (in anisotropic space) to the cells (Block method) or nodes (Smooth Method) in your grid. Smooth still refines your grid further, but not as accurately as the other two methods below which compute more accurate probabilities.
- Inverse Distance Weighted IDW provides a more rigorous (than Nearest Neighbor) probabilistic approach to lithology modeling because it computes the probabilities for each material in your lithology.
- Kriging provides the most rigorous probabilistic approach to lithology modeling because it computes the probabilities for each material in your lithology.

The **Use Full Data Sampling** toggle causes all of your data to be used for the estimation process instead of just the N nearest points. This is subject to the **Full Data Sample Limit** parameter.

The **Full Data Sample Limit** parameter is the upper limit for using all samples. Since the compute time and memory tends towards the square of the total number of points used for estimation, this parameter prevents accidentally using an option which might cause your memory or compute time to be huge. The default of 1000 is a reasonable upper limit, but values of 3000 to 5000 are occasionally justified.

**Points** is the maximum number of points used for estimation, if the **Use Full Data Sampling** toggle is off. The default value for points is 20, which generally provides reasonably smooth modeled parameter distributions. The effects of decreasing and increasing the values for reach and points on the

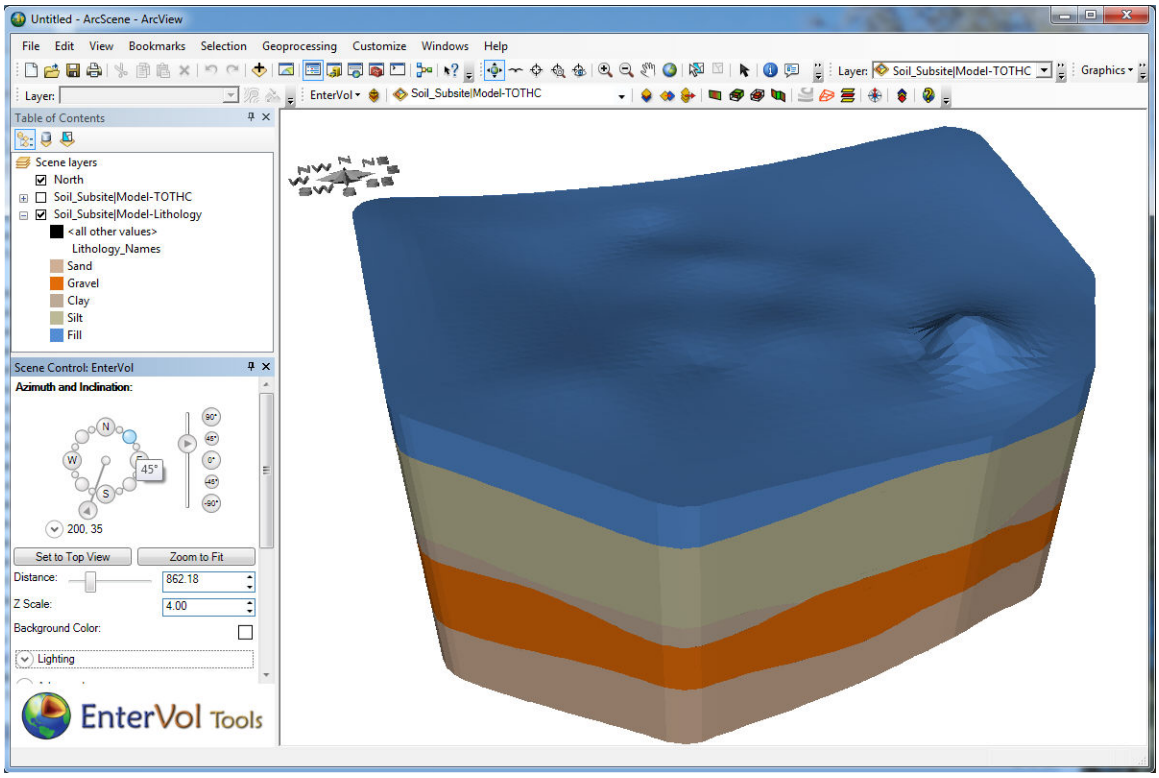
model output are somewhat similar, but for different reasons. If the data have a fairly even spatial distribution throughout the domain, then increasing these values will generally include more of the input data points that will be used to krige the value for a given model node, and thus will result in smoother modeled data distributions. Decreasing the values of reach and points (in an evenly distributed data set) results in fewer input data points being used to calculate the parameter estimates at a given model node, and result in modeled distributions with greater variations across smaller areas.

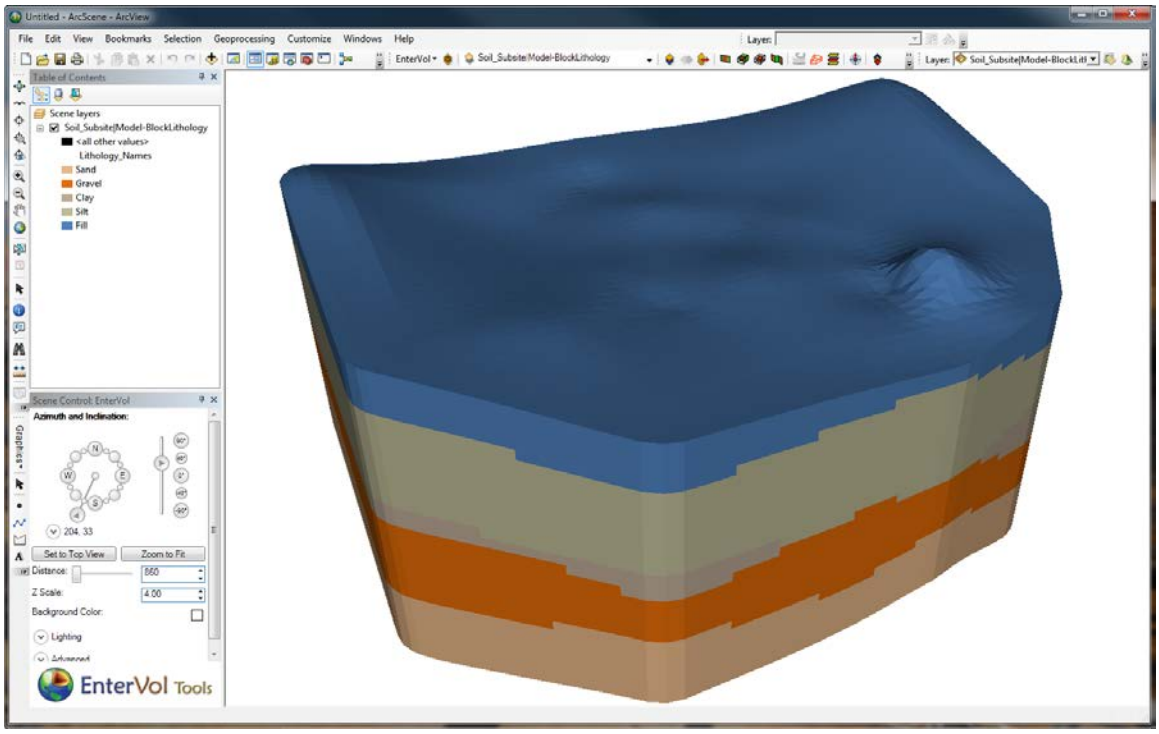
**Reach** is the maximum distance (in user units) from any given model node that the estimation method will look for data points to be included in the estimation of the model parameter at that node. The default value of reach is 10 billion, which results in Points being the limiting factor.

The user should consider both the spatial distribution and the range of values in the input data set when deciding upon values for the reach and points parameters. If the specified reach is too small to allow the kriging module to locate at least one point within the search area, then no kriging can be done and the module will terminate with an error message.

The ***Horiz./Vert. Anisotropy Ratio*** basically tells the kriging algorithm what multiplication factor should be used to apply biased weighting on data points in horizontal and vertical directions away from a given model node. The default value is 10, which allows data points in a horizontal direction away from a model node to influence the kriged value at that node 10 times more than data points an equal distance away in a vertical direction. However, the user can specify any positive number with a magnitude up to 100,000.

Below is an example of both SMOOTH and BLOCK Lithology models.

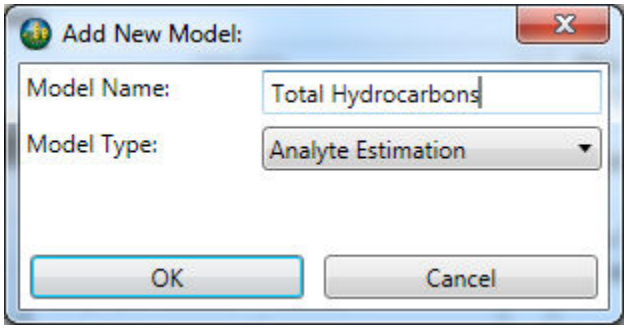



## GeoStats: Analytical Data Estimation

**THIS TOPIC REQUIRES AN EnterVol GeoStats LICENSE**

Provided that at least one 3D Grid has been created, Click on the  to map (estimate) analytical data onto a 3D volumetric grid.




The **Estimation Method** selector includes three options:

- Nearest Neighbor is a Quick Method which assigns the data based on the nearest analyte sample (in anisotropic space) nodes in your grid.
- Inverse Distance Weighted IDW provides a more rigorous (than Nearest Neighbor) probabilistic approach to analyte modeling because it computes the probabilities for each material in your lithology.
- Kriging provides the most rigorous probabilistic approach to analyte modeling because it computes the probabilities using expert system computed weighting factors

The **Use Full Data Sampling** toggle causes all of your data to be used for the estimation process instead of just the N nearest points. This is subject to the **Full Data Sample Limit** parameter.

The **Full Data Sample Limit** parameter is the upper limit for using all samples. Since the compute time and memory tends towards the square of the total number of points used for estimation, this parameter prevents accidentally using an option which might cause your memory or compute time to be huge. The default of 1000 is a reasonable upper limit, but values of 3000 to 5000 are occasionally justified.

The **Octant** search toggle

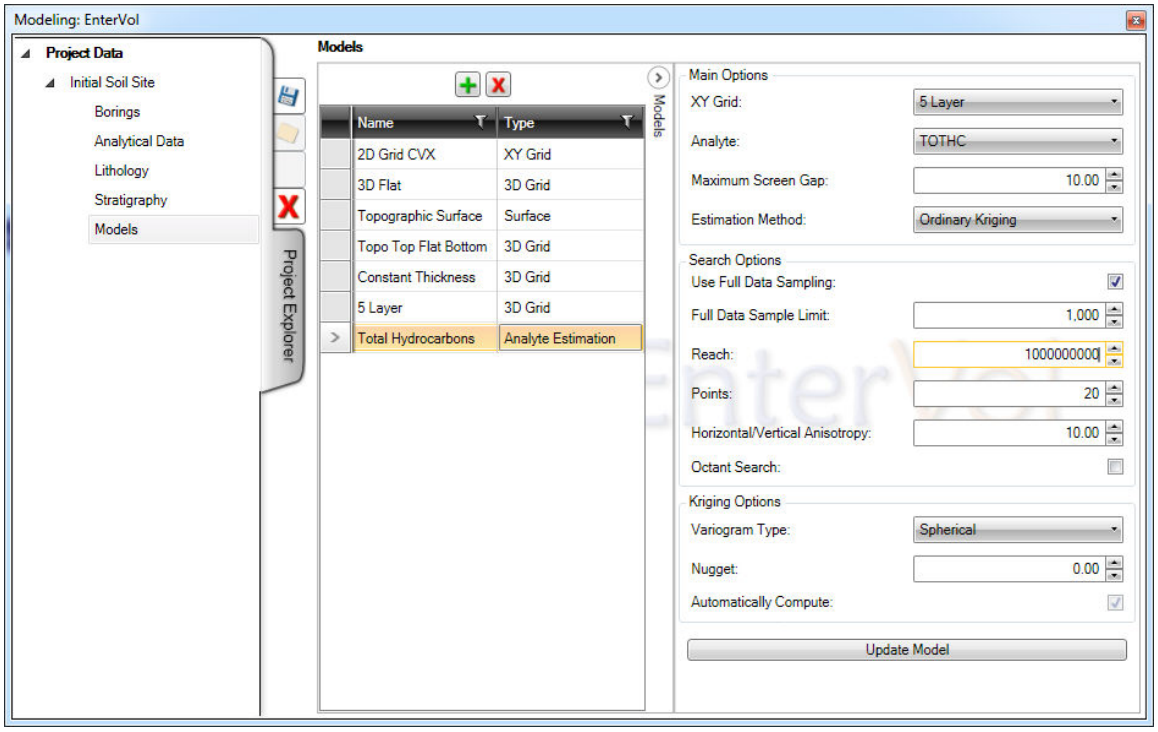
**Points** is the maximum number of points used for estimation, if the **Use Full Data Sampling** toggle is off. The default value for points is 20, which generally provides reasonably smooth modeled parameter distributions. The effects of decreasing and increasing the values for reach and points on the model output are somewhat similar, but for different reasons. If the data have a fairly even spatial distribution throughout the domain, then increasing these values will generally include more of the input data points that will be used to krige the value for a given model node, and thus will result in smoother modeled data distributions. Decreasing the values of reach and points (in an evenly distributed data set) results in fewer input data points being used to calculate the parameter estimates at a given model node, and result in modeled distributions with greater variations across smaller areas.

**Reach** is the maximum distance (in user units) from any given model node that the estimation method will look for data points to be included in the estimation of the model parameter at that node. The default value of reach is 10 billion, which results in Points being the limiting factor.

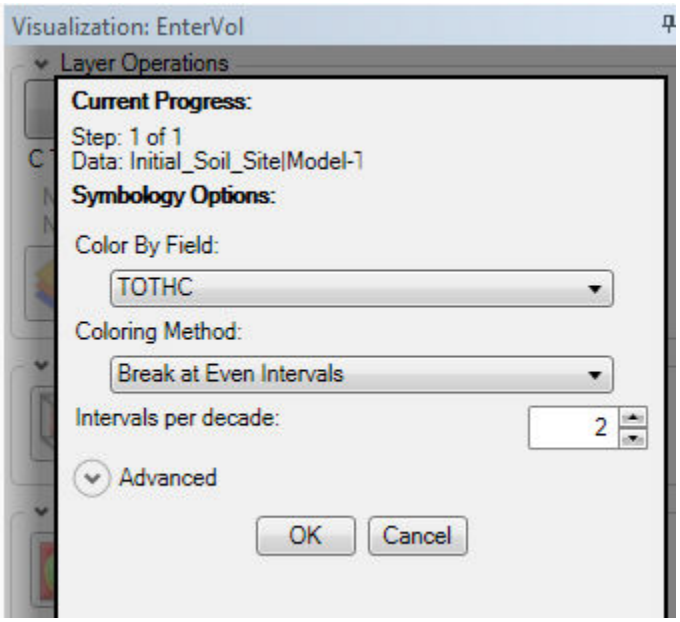
The user should consider both the spatial distribution and the range of values in the input data set when deciding upon values for the reach and points parameters. If the specified reach is too small to allow the kriging module to locate at least one point within the search area, then no kriging can be done and the module will terminate with an error message.

The ***Horiz./Vert. Anisotropy Ratio*** parameter allows the user to consider the effects of anisotropy in the conductivity of the soil layers. In most cases, geologic materials are deposited with platy clay minerals oriented horizontally, and thus the flow of water or contaminants in both the saturated and unsaturated zones can be slower in the vertical direction than in the horizontal direction. Also, ore deposition can occur along horizontal or vertical fault or fracture systems. Chemical constituents being transported with flowing fluids may therefore show a larger degree of spreading in one or the other direction. The Horiz./Vert. Anisotropy Ratio basically tells the kriging algorithm what multiplication factor should be used to apply biased weighting on data points in horizontal and vertical directions away from a given model node. The default value is 10, which allows data points in a horizontal direction away from a model node to influence the kriged value at that node 10 times more than data points an equal distance away in a vertical direction. However, the user can specify any positive number with a magnitude up to 100,000.

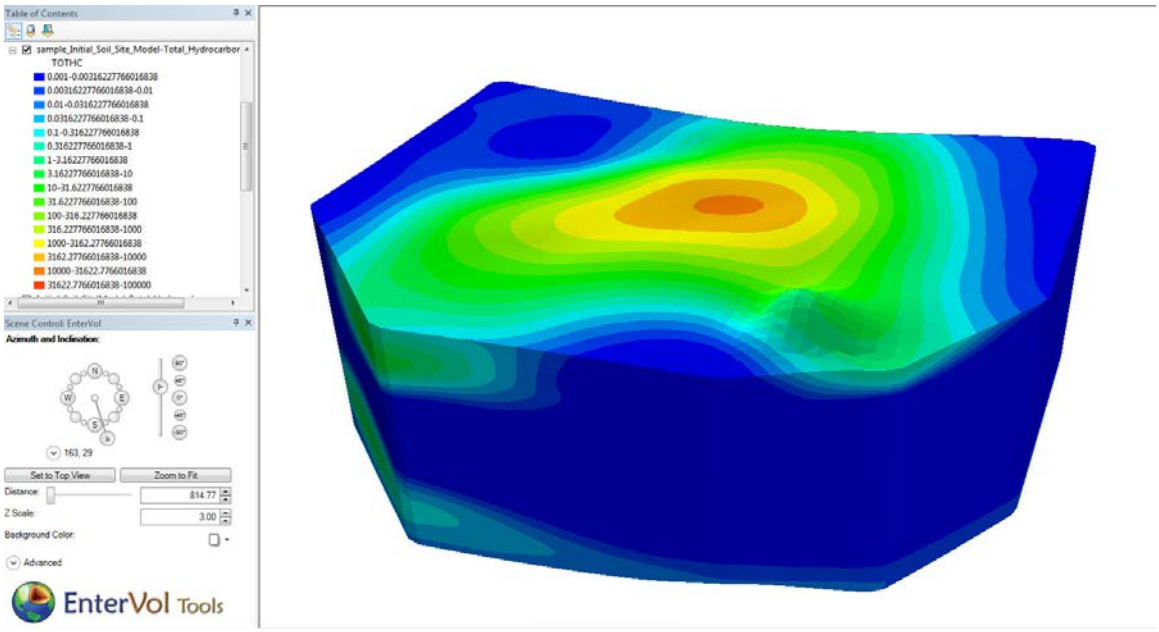
We'll choose to use all of the sample points in our kriging. With the setting shown below.



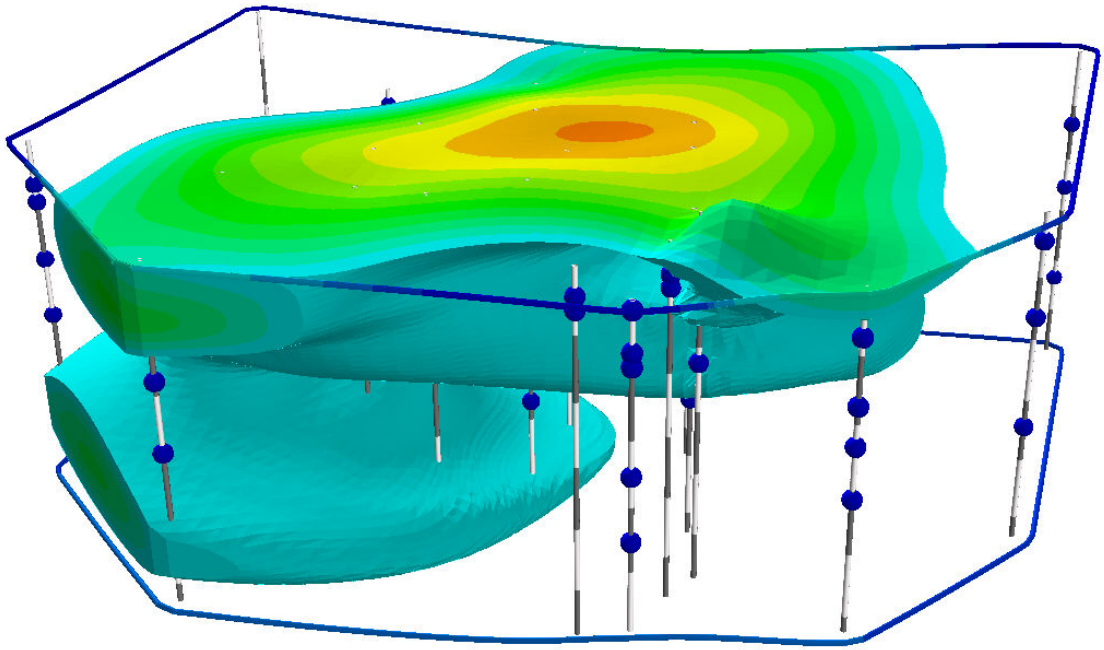

After Updating (creating) the model, we'll choose 2 intervals per decade for our Log Processed data.




The resulting model is:




Adding the borings, TOTHC sample points and a plume at 0.1 mg/kg yields

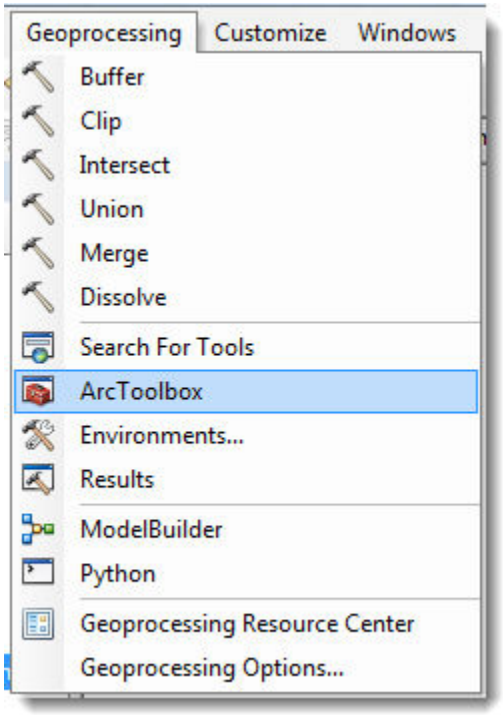


# Geoprocessing Tools

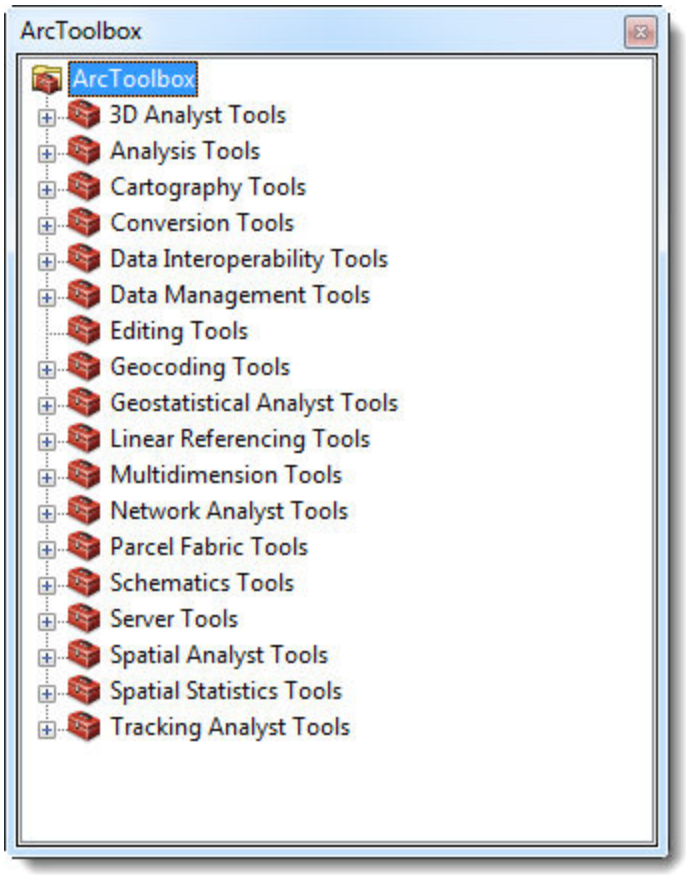
## Geoprocessing Tools

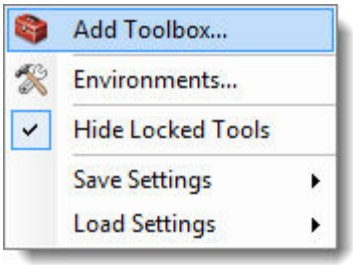
EnterVol's Geoprocessing Tools menus provide an advanced method to perform the subsetting, display and scene manipulations.

First you must add these tools to ArcToolbox. The example below adds the toolbox to ArcScene. The procedure also applies to ArcMap.



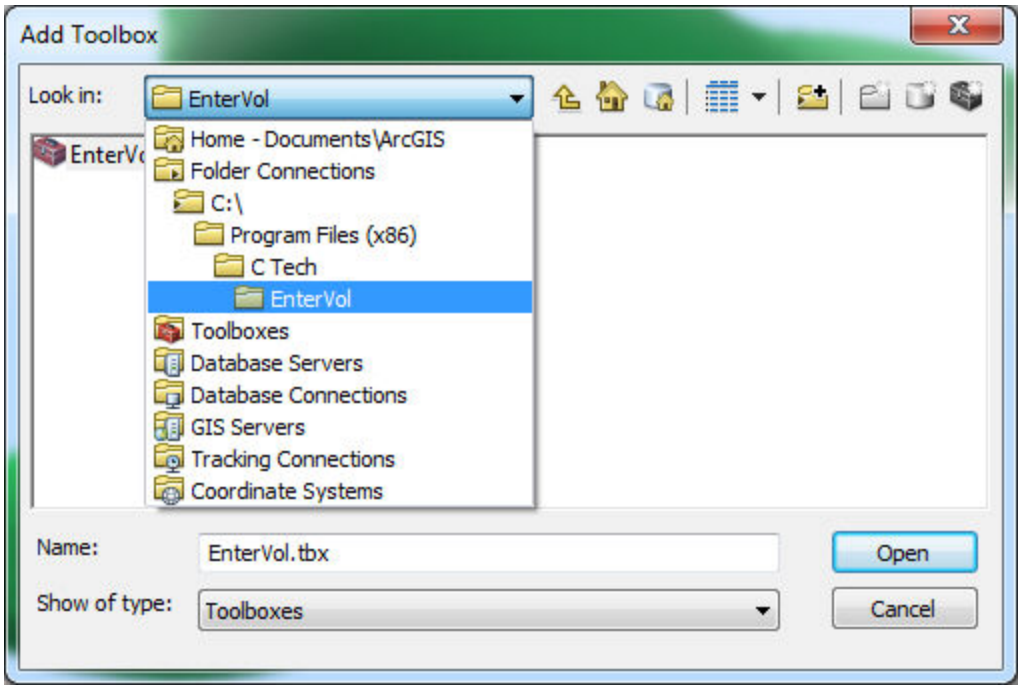

The ArcToolbox window appears:



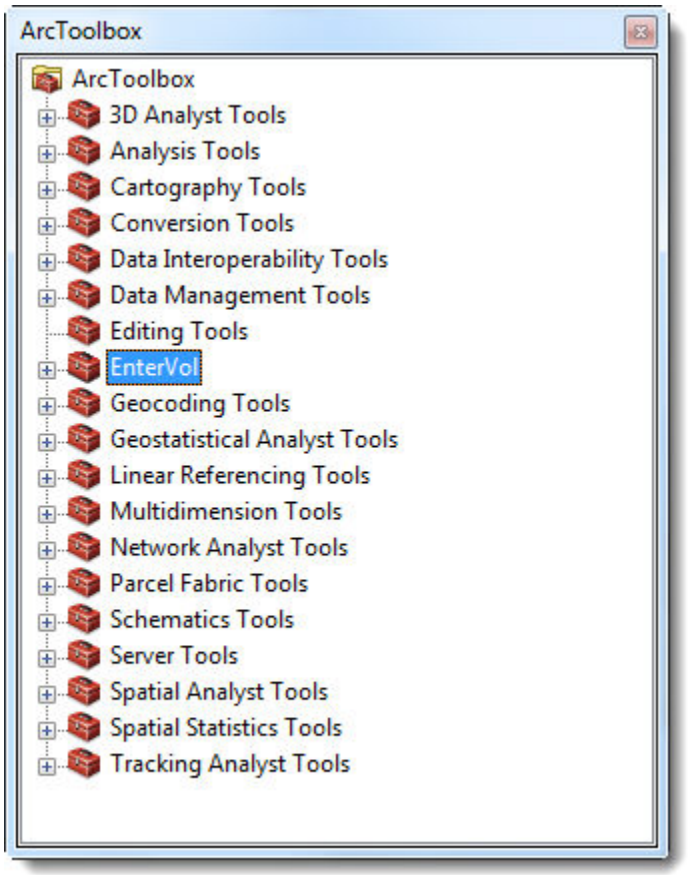




Right click in the window and select Add Toolbox:

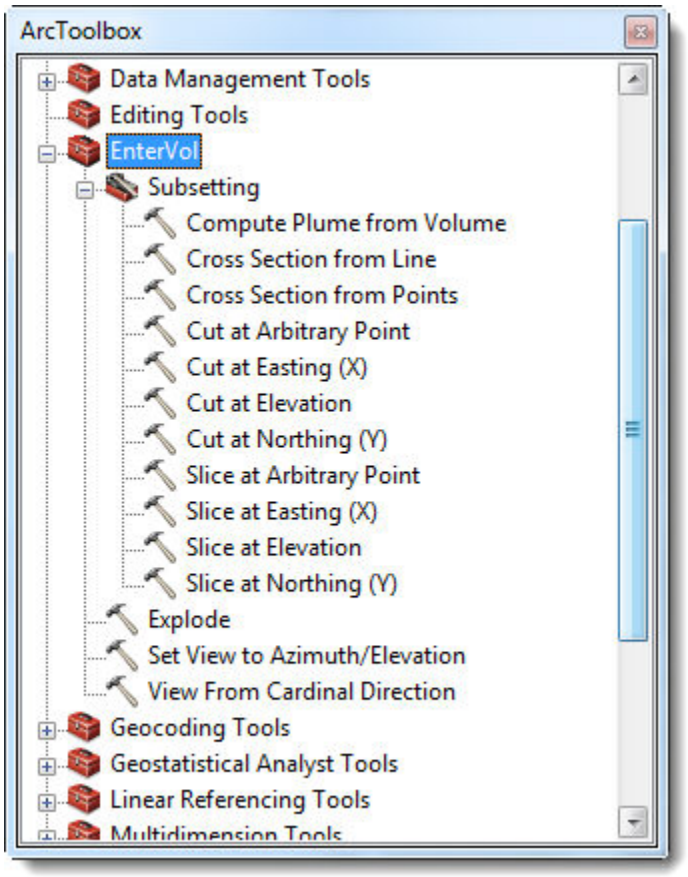
Browse to the EnterVol install folder and choose EnterVol.tbx (do not double click or it will go into it)




This will add EnterVol



and when expanded you will see the full list




## Subsetting

### Compute Plume from Volume

Input parameters are:

- Input Volume
- Data to Subset
- Subset Level
- Cut Above (on) or Below (off) Level: Toggle
- Output FeatureClass

### Cross Section from Line

Input parameters are:

- Input Volume
- Lines
- Attempt to vix T-splits at joints (optional): Toggle
- Straighten to 2D: Toggle
- Offset Between Straightened Cross Sections (optional)
- Output FeatureClass

### Cross Section from Points

Input parameters are:

- Input Volume
- Points
- Attempt to vix T-splits at joints (optional): Toggle
- Straighten to 2D: Toggle
- Output FeatureClass

### Cut at Arbitrary Point

Input parameters are:

- Input Volume
- Coordinate X
- Coordinate Y
- Coordinate Z
- Dip Value
- Dip Direction Value
- Reverse Cut: Toggle
- Output FeatureClass

### Cut at Easting (X)

Input parameters are:

- Input Volume
- Coordinate X
- Reverse Cut: Toggle
- Output FeatureClass

### Cut at Elevation

Input parameters are:

- Input Volume
- Coordinate Z
- Reverse Cut: Toggle
- Output FeatureClass

### Cut at Northing (Y)

Input parameters are:

- Input Volume
- Coordinate Y
- Reverse Cut: Toggle
- Output FeatureClass

### Slice at Arbitrary Point

Input parameters are:

- Input Volume
- Coordinate X
- Coordinate Y
- Coordinate Z
- Dip Value
- Dip Direction Value
- Output FeatureClass

### Slice at Easting (X)

Input parameters are:

- Input Volume
- Coordinate X
- Output FeatureClass

### Slice at Elevation

Input parameters are:

- Input Volume
- Coordinate Z
- Output FeatureClass

### Slice at Northing (Y)

Input parameters are:

- Input Volume
- Coordinate Y
- Output FeatureClass

## Explode tool

Input parameters are:

- Input Volume
- Method
- Distance to Explode
- Output FeatureClass

## Set View to Azimuth/Inclination

This tool can only be used in ArcMap (not ArcScene) only because it rotates 3D layers making them appear more three dimensional in ArcMap maps.

It sets rotations for an individual layer (EFF) for pseudo 3D display in ArcMap. Once this tool is applied, the layer is not in its true position in your map and therefore this should be used only for annotation purposes.

Input parameters are:

- Input Volume
- Azimuth
- Inclination
- Output FeatureClass

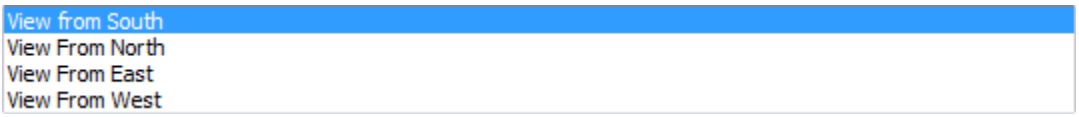
## Set View from Cardinal Direction

This tool can only be used in ArcMap (not ArcScene) only because it rotates 3D layers making them appear more three dimensional in ArcMap maps.

It sets rotations for an individual layer (EFF) for pseudo 3D display in ArcMap. Once this tool is applied, the layer is not in its true position in your map and therefore this should be used only for annotation purposes.

Input parameters are:

- Input Volume
- Direction




- Output FeatureClass