FAQ

Table of Contents

Installation and Settings

Which Java version does Insight use?

When you install DUG Insight, its own version of Java is included. Insight uses this version instead any Java installations on the system. This ensures that you are using a version that is compatible with Insight.

As a result, Insight does not require a separate installation of Java, and does not require updates to the version of Java on the system.

The Java version included with Insight depends on the release:

- **Up to 4.5-902060 (released before March, 2019):**
  - Java 64bit v1.7.0 - “Java SE Version 7”
- **After 4.5-903060 (released after March, 2019):**
  - Java 64bit v1.8.0_201 - “Java SE Version 8”
- **After 4.8-006300 (released after June, 2020):**
  - Java 64bit - "Java SE Version 12.0.2”

If you have questions or problems regarding Java and Insight, please contact our support team at *support@dugeo.com*.

Note: You may be asked to send data of your session from the Logs window for a clearer picture of the problem (see *Viewing and Sending Diagnostic Logs*).

How does Oracle's Java licensing apply to Insight?

Oracle’s Java licensing scheme does not apply to the version of Java included with Insight.

Each Insight installation includes a Java Runtime Environment (JRE). This ensures Insight runs smoothly and the environment is consistent across users and versions.

The included JRE depends on the release:

- **Up to 4.5-902060 (released before March, 2019):**
  - Java 64bit v1.7.0 - “Java SE Version 7”
- **After 4.5-903060 (released after March, 2019):**
  - Java 64bit v1.8.0_201 - “Java SE Version 8”

These releases of Java can be used freely without an Oracle Java licence or subscription. Oracle has stated that after January 2019, Oracle Java SE Version 8 will not receive bug fixes or updates without a subscription.

From Oracle's page, [Java SE General FAQs:]

```
Java SE 8 remains free of charge for general purpose desktop and server
use and is available under the Oracle 11 Binary Code License (BCL) at
https://www.oracle.com/javadownload.
```

Internally, we are successfully running areas of our code on non-Oracle versions of Java. We anticipate moving Insight to one of these versions once we have confidence that it is stable and reliable. Java runtime environments not provided by Oracle are not covered by their licensing, but do receive patches, updates and bug fixes.

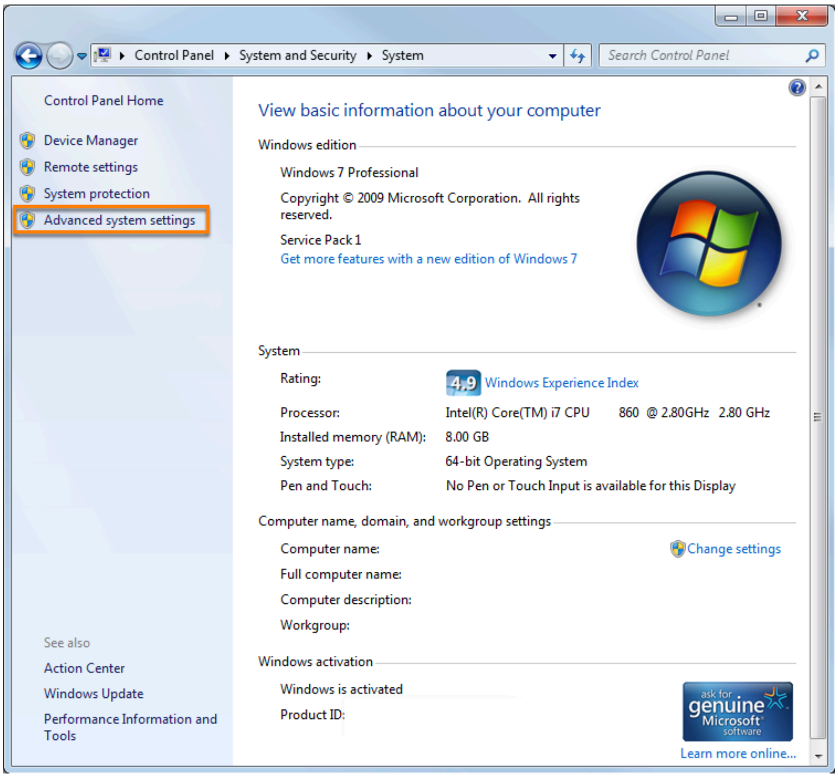
How do I edit the Java's overriding Environment Variable?

Other Java applications installed on your system may have configured Windows to not allow Insight to allocate the memory you set with the **DUG Insight Launcher**. If you are getting out of memory errors and believe the memory allocation to be incorrect, you can confirm the memory allocation *here*.

If this is what you are experiencing, you will need to remove an environment variable called "_JAVA_OPTIONS".

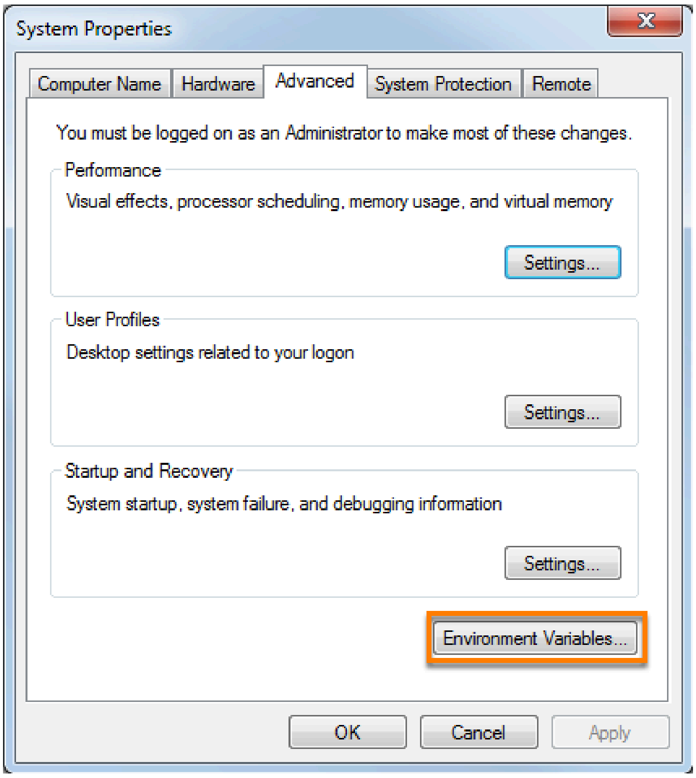
To do this on Windows, follow the steps below.

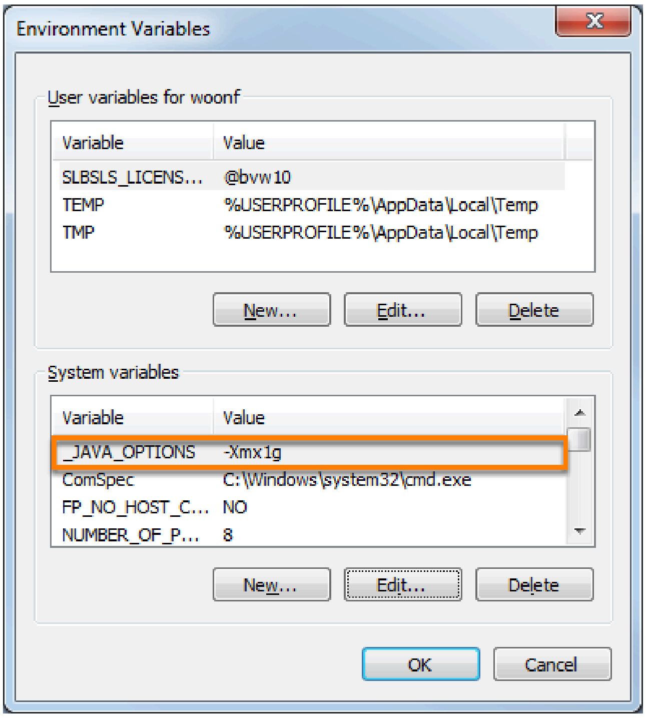
Configure advanced system settings



1. Right Click on **My Computer**
2. Click **Properties**.
3. On the left panel, click **Advanced system settings**. This will open the **Advanced** tab in the **System Properties** window.

Edit environment variable





1. In the **Advanced** tab in the **System Properties** window, click **Environment Variables**. This will open the **Environment Variables** window.
2. In either the **System variables** or the **User variables** section, look for the "_JAVA_OPTIONS" variable. Based on the variable value in the example, Insight will use 1GB RAM and not the memory you have allocated in the **DUG Insight Launcher**.
3. Double-click the variable or select this variable row and click **Edit**.
4. Edit the variable name to something else (effectively removing it by name, but backing it up in case your other Java applications truly require this setting).
5. Test the other application.
6. [Optional] If the other application works, you can now go in and delete the renamed variable.

> ⚠ **Caution:** If you have other Java applications on your system that are using this variable, and given that this setting is there, this is likely to be the case, removing this setting may affect the applications. Hence, we advise to rename the variable and ensure other applications are not affected before deleting.

If you need to merge the variables, contact DUG support at support@dugeo.com.

Insight should now open with the correct memory allocation.

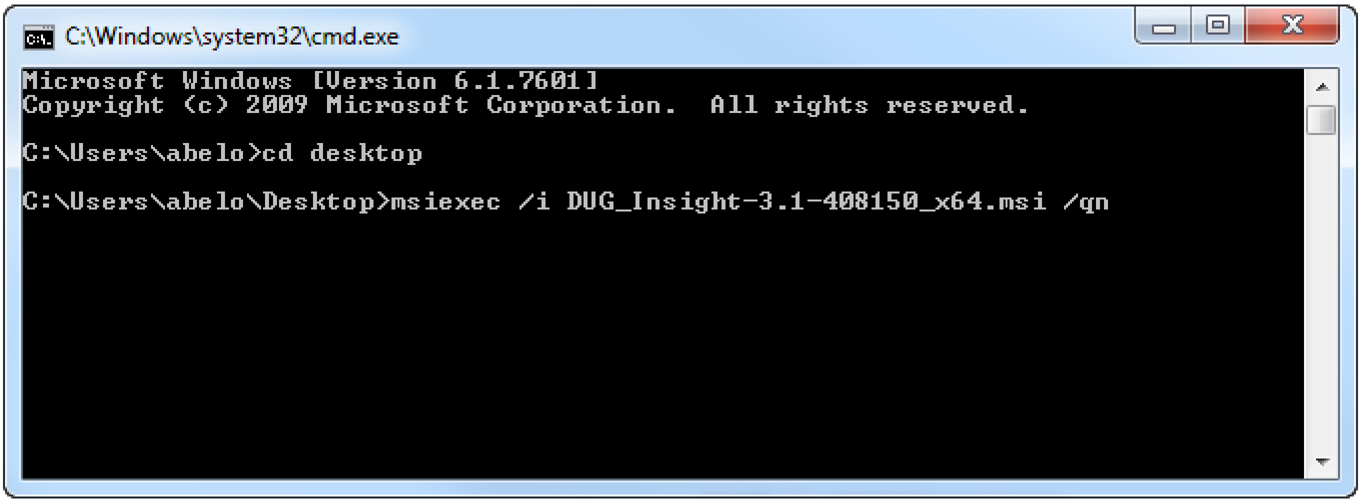
How do I install Insight silently or via SCCM?

The default installation for DUG Insight is an Attended Installation (see *Installing Insight on Windows*).

To perform a silent installation, the installation package needs to run from the command window using the "msiexec /i" command.

If you are using a System Center Configuration Manager (SCCM) to perform the installation, simply run the command lines described below in your SCCM to deploy Insight.

Silent installation per user



1. From the **Start** menu, type "cmd" and open the Windows Command Processor.
2. In the command window, navigate to the folder where the installation file (.msi) resides.
3. To perform a silent installation, type the following:

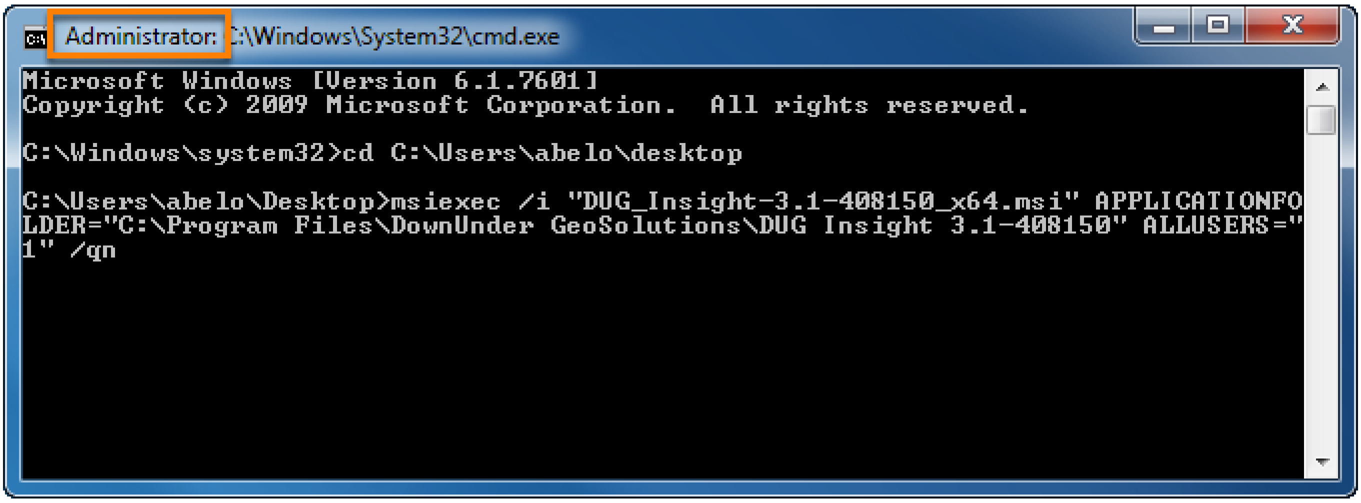
```
msiexec /i DUG_Insight-3.1-408150_x64.msi /qn
```

Note: Change the version number as necessary.

4. Press **Enter**. Insight will then be installed silently for the **current user** only.

Tip: The installer will install Insight in a default system location for user based installations (i.e. *C:\Users\[Username]\AppData\Local\DownUnder Geosolutions*). The location may vary depending on the operating system the machine is running (see *Home directory*).

Silent installation for ALL users



To perform a silent installation for ALL users, administrative privileges are required. Make sure you are logged in to an Admin account or from an admin console.

1. From the **Start** menu, type "cmd".
2. Right click **cmd** and select **Run as administrator** to open the Windows Command Processor as Administrator.
3. In the command window, navigate to the folder where the installation file (.msi) resides.
4. To perform a silent installation for all users, type the following (change the version number and installation folder as required):

```
msiexec /i "DUG_Insight-3.1-408150_x64.msi" APPLICATIONFOLDER="C:\Program Files\
DownUnder GeoSolutions\DUG Insight 3.1-408150" ALLUSERS="1" /qn
```

5. Press **Enter**. Insight will then be installed silently for **ALL** users at the folder specified above.

Silent install Insight with Petrel Link

To include the Petrel Link in the installation, add FEAT_PETRELLINK="1" to the command:

```
msiexec /i "DUG_Insight-3.1-408150_x64.msi" APPLICATIONFOLDER="C:\Program Files\
DownUnder GeoSolutions\DUG Insight 3.1-408150" ALLUSERS="1" FEAT_PETRELLINK="1" /qn
```

Note: This will not fix the problem for an already installed version. Uninstall the current Insight version before executing the command.

Genuine offer of assistance

If the installation is unsuccessful, or if you have any questions, please do not hesitate to contact our support team at support@dugeo.com.

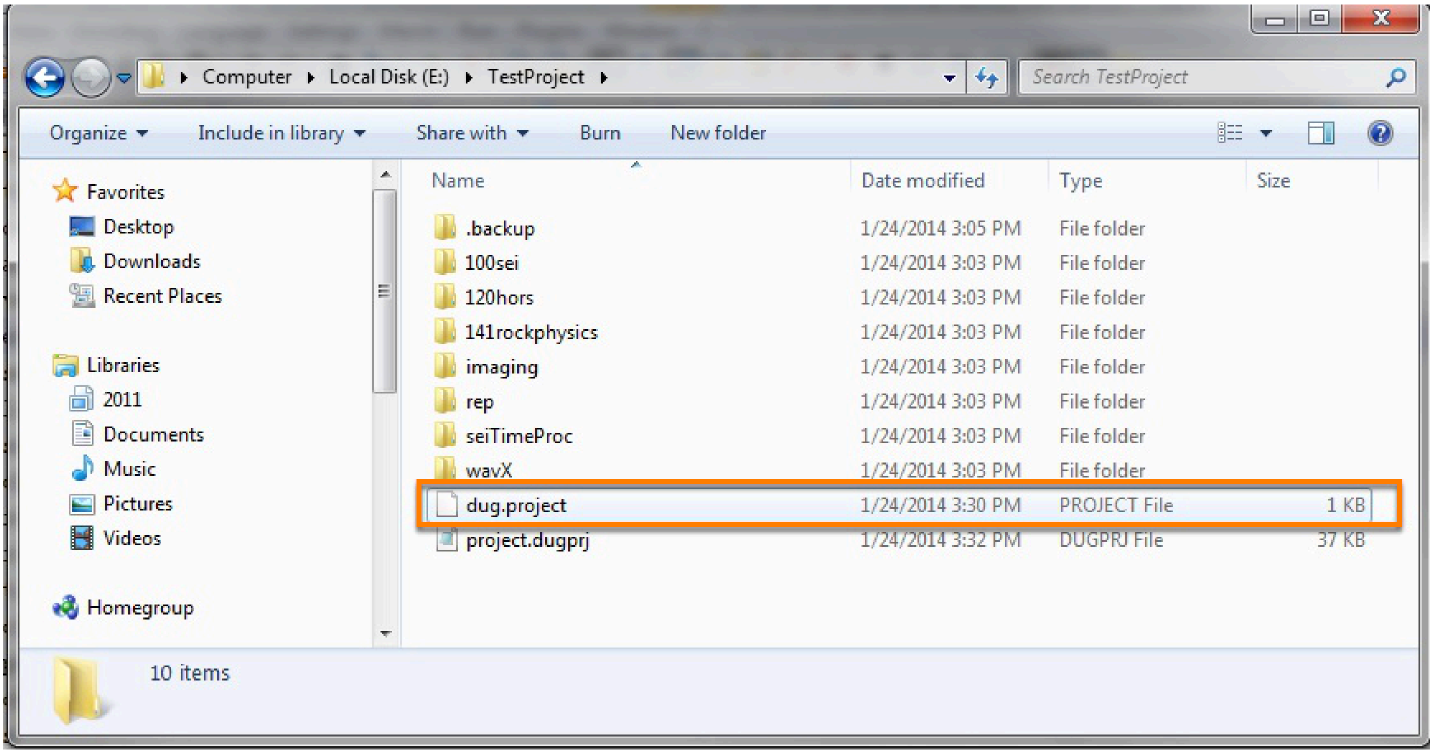
It would be helpful if you could provide logs for the failed installation. To install Insight with logging enabled, see *Installation of Insight failed! What should I do?*

How do I change Insight's default directory?

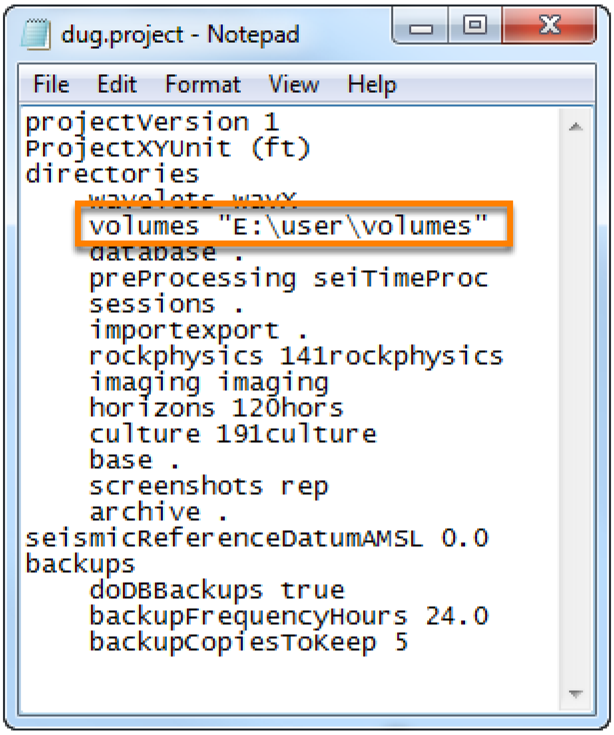
By default, Insight saves its data into the local project folder. If you wish to set a separate default location to save or load your data for convenience, you can change the default by following the steps below.

Note: Please exit Insight prior to making any changes to the project file. Relaunch Insight after saving and closing the project file.

Changing default location



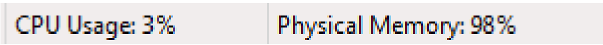
1. Locate and open the project directory in the local disk.
2. Right-click the ***dug.project*** file and open with any text-based editing software.



3. Under **directories**, replace the folder names at the second column with the desired path surrounded by double quotes.
4. When you are done, click on **File** and select **Save**.
5. This will change the import/export/save defaults to the location specified.

The example above changes the default path for seismic volumes. After editing the *dug.project* file, every time the user wants to import or export a volume, Insight will open the desired directory as the default. This just changes the default path; the user can still browse and save the volume at another location.

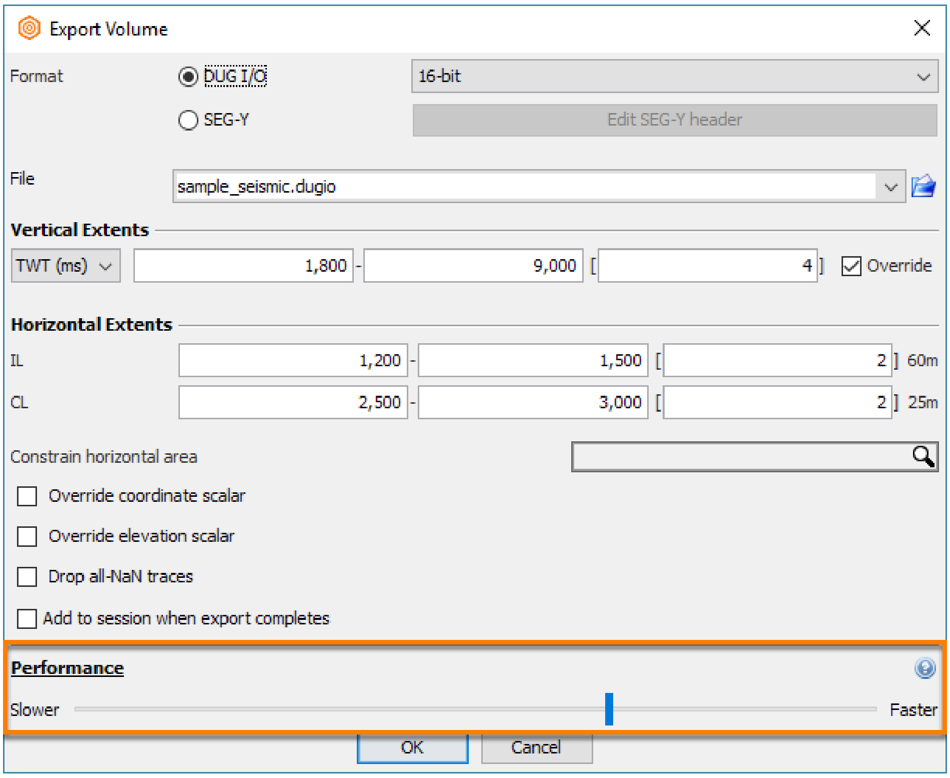
Why is exporting data from Insight light on CPU but heavy on RAM?



Depending on the export, the issue could be *I/O bound*, waiting for the hard disk to spin.

Unless there is anything highly computational in the export, you should expect to see low CPU usage. If you are exporting computationally-intensive data, use the performance slider in the Export Volume window to spread the load over multiple CPU threads.

The export performance slider **controls how many export tasks are created and shared between the CPU cores**. Each task requires memory and in this case, it adds up to using everything that is available.



Pauses occur during export if Insight fills its assigned memory and needs to clean up.

If this is happening regularly, reduce the performance setting to a lower level. Lower values reduce the number of concurrent tasks and the memory requirements.

For processor intensive tasks this will increase the export time.

If possible:

- Increase the memory allocated to Insight in the DUG Launcher (see *Allocating Memory*).
- When running big export tasks, assign 80-90% of the total memory.

This allows more tasks to run concurrently before a memory clean up is required, and may completely bypass the need for a clean-up.

If the issue persists or you would like specific advice, do not hesitate to contact our support team at *support@dugeo.com* for assistance.

For more information, see Exporting a Volume to DUG I/O.

Performance

Why is Insight running unusually slow?

Performance issues can be notoriously difficult to diagnose due to the shear number of factors involved. If you are unhappy with the performance or suspect something is wrong, our advice is to send us diagnostic logs while Insight is slow or unresponsive (see *Viewing and Sending Diagnostic Logs*). We will often get a response to you within minutes.

However, it may be useful to understand how Insight runs on your machine and what might be causing it to perform poorly. This could help you troubleshoot the problem yourself.

How Insight uses memory

When you open a project, Insight only loads the basic details of the sessions and files (e.g. name and type of volume) into memory, not the whole product. The size of the project directory does NOT impact the performance of Insight. Generally, Insight should be as responsive for a 50TB project as it is for a 50MB project.

However, the size of a session – how many products must be loaded, what displays open up, etc – can impact the responsiveness of Insight and the time it takes to open the session. Insight only loads into memory what is required to fulfill the display. Hence, memory usage would always be low until you wanted to display something in Insight's Views (there are several exceptions to this rule, which are *highlighted below*).

When you are displaying data, Insight renders from an in-memory cache, which it loaded from your product (and then, just the parts it needed). So if the cache is much bigger than what it needs to display, it may have other information in memory and flicking between volumes can be very fast (since no more disk IO is required). If, however, your memory allocation is too small (see *Allocating Memory*), it will have to throw away the previous data in order to display the new. This will cause Insight to be slow when rendering new data.

Simply limiting the display of products to a reasonable amount by disabling them in the control panel will allow Insight to run a little faster (see *Activating/Deactivating Items*). Limiting a session to the products required – so that you are not updating a flattener you no longer use, for example – should also improve performance (see *Irrelevant data* below).

Factors affecting performance

The following pointers are the most common reasons why Insight is not performing as well as it should be.

Memory allocation

The amount of memory to allocate to Insight depends on the size and complexity of the project and how much RAM is available on your machine (see *Allocating Memory*).

As a precaution, you should leave approximately 2GB RAM for your operating system. If you run other applications simultaneously, you should also allow for these. If the entire computer begins to feel sluggish, you have probably chosen too large a number. It is generally better to allocate less if you are unsure. For example, on a machine with 16GB of total RAM, you should allocate no more than 14GB for Insight.

Hardware

For recommended hardware requirements, please refer to *System Requirements*.

Network/external hard-drive I/O limitations

Storing volumes, horizons or projects on an external USB or network drive could cause slowness issues if the I/O capabilities are not up to scratch. Generally we recommend 30 MB/s sustained transfer capable connections.

Process volumes

Process volumes are generated on the fly by processes in Insight. Processes can be fed into one-another so that the end volume is the product of several individual processes.

For example, an Incoherence volume might be constructed using: Dip process -> SOF process -> Incoherence process

Viewing the final Incoherence Process is computationally intensive and may be slow to display. In situations such as this, you can to export the final process volume to disk to retain performance (*see Exporting a Volume to DUG I/O*).

Volumes with phase rotation applied

Phase rotation is also a computationally intensive operation. Propagating horizon picks on a phase rotated volume will be far slower than usual because Insight must perform a phase rotation for each trace prior to propagating new picks.

Phase rotated volumes are process volumes and should be exported and saved on disk (*see Exporting a Volume to DUG I/O*).

2D survey tie-points

We have experienced some cases where having a 2D survey comprising very closely spaced tie-points has resulted in performance issues. The reason for overly dense tie-points could de due to poor, erratic navigation data in the original 2D SEG-Y files. You can check the number of tie-points defining each 2D line by exporting the survey file and opening in a text editor such as wordpad. If you suspect this may be the cause, we may need to downsample your survey for you. Or you can regenerate the survey file by re-loading in the SEG-Y files, and at the Survey creation screen, increase the maximum error from its default 10m to 20m.

Horizon used in a process such as flattener

Picking a horizon that is being used in another process such as a flattener can cause slowness because the flattening process updates with changes to the horizon.

Time-depth conversion

Viewing TWT volumes in the TVD domain will place more demand on your machine due to the depth conversion process involved. Furthermore, it is important to ensure that the velocity volume used is not too densely sampled.

The standard practice in DUG's service division is to use a velocity model sampled at around 100m x 100m x 24ms/20m. If your velocity volume is more dense than this, you may wish to consider downsampling using the velocity conversion process.

View extents

Insight only loads data into memory that is required to satisfy the field of view. Zooming in on the map and section views will reduce the field of view and consequently the amount memory and processing time required.

Irrelevant data

As a last resort, any datasets that are not actively being used (such as contours, culture files, and non-essential horizons) can be hidden using the traffic light controls in the control panel. Less information displayed means less demand on your machine, and the faster Insight can perform.

Why is my session taking so long to save?

If Insight is taking longer than several minutes to save a session, there may be something wrong. If you can, *use the DUG Launcher to send logs* and we will investigate and try to understand why.

- *Slow network or busy file server*
- *It is waiting for another copy of Insight in the same project to complete some work*
- *The connection to the server was lost*

There are common reasons Insight can take a long time to save a session:

Slow network or busy file server

Insight relies on the speed of the network and file server. If the file server or network is very busy, it can result in poor Insight performance.

How to identify:

- Many users are affected.
- All projects are affected.
- Other software is affected.
- Talk to other network users. Are they also running slowly?
- Many actions (saving session, creating new items) take longer than expected).

Solutions:

- Send logs.
- Be patient. Wait longer.
- Talk to the system administrator to identify performance problems.

It is waiting for another copy of Insight in the same project to complete some work

It is rare but sometimes another Insight in the same project will take a long time to complete its work. If this happens when it is writing to the database, your copy of Insight must wait for its turn to write the session. This should not take more than a few minutes.

More common when using busy projects with many active users.

How to identify:

- Only one project is affected
- Other users in the same project are affected.

- Talk to other users in the project. Are they having similar problems? Are they also running slowly?
- Many actions (saving session, creating new items) will take longer than expected.

Solutions:

- Send logs.
- Be patient. Wait longer.
- Investigate whether the network or the file server is busy.

The connection to the server was lost

Temporary network errors can cause Windows to lose the connection to the project area. Usually it recovers quickly and is not noticeable. If the error happens when Insight is working, Insight may be unable to use the database without restarting.

Consider this if there are few users of a network project.

How to identify:

- Only one project is affected
- Only one user (or very few) are affected.
- Talk to other users in the project. Are they also running slowly?
- Any actions such as saving sessions, and creating new items will be stuck.

Solutions:

- Send logs.
- Talk to other users in the project. Are they having similar problems?
- Manually close or kill Insight.

If the save takes more than 10 minutes and is not progressing, consider manually closing the Insight process. Insight performs an autosave of a session every 5 minutes (details here), so it is very rare to lose work.

In Linux, why does Insight run out of memory for no good reason?

Insight is a multi-threaded application. Using multiple threads lets a program run more efficiently and also takes advantage of multiple CPUs and CPU cores. When Insight starts, it prepares a pool of "worker threads", ready to perform calculations and collect results.

In the Linux OS, threads and processes are handled very similarly. There is a default setting that limits the total number a user can have at any one time. The default setting is often very low and not appropriate for a single-user workstation.

- *Check your limits*
- *Update the limit*

Check your limits

These commands should all be run from a terminal window.

- To find the maximum number of threads and processes a single user can have at any one time: `ulimit -a`
  - Several pieces of information are returned, look for: `max user processes`
  - For a single-user workstation or virtual machine, this value can be set to a much larger number (e.g. 100000) without causing issues.
- To find the current number of running threads -- this number will vary depending on the number of programs and activity on the machine:
  1. `ps -elfT | wc -l` OR
  2. `cat/proc/loadavg`
     1. Look at the fourth column (it includes a '/' character).
        - This field shows: "currently executing processes" / "total processes"
        - e.g. `0.72 0.58 0.48 2/867 19334` -> 2 executing processes / 867 total
- To find the the total limit across all users for the machine:
  - `sysctl -a | grep kernel.pid_max`

Update the limit

These commands require administrator/root permissions. Be careful when making changes to system files or have your IT administrator do it for you.

- Set the process limit to a new value. The limit will return to the default when the machine is restarted.
  - `ulimit -u 10000`

Updating the limit permanently depends on the version of Linux you are using.

> **Note:** You must completely log out of the account and log back in for these changes to take effect.

If you are using PAM, these settings can be configured in:

- Configuration file: `/etc/security/limits.conf`
- These lines set the limit for all users:

```
*           soft    nproc      16384
*           hard    nproc      16384
```

If you are using a RedHat variant, you can try:

- Configuration file: `/etc/security/limits.d/90-nproc.conf`
- These lines set the limit for all users:

```
*           soft    nproc      16384
*           hard    nproc      16384
```

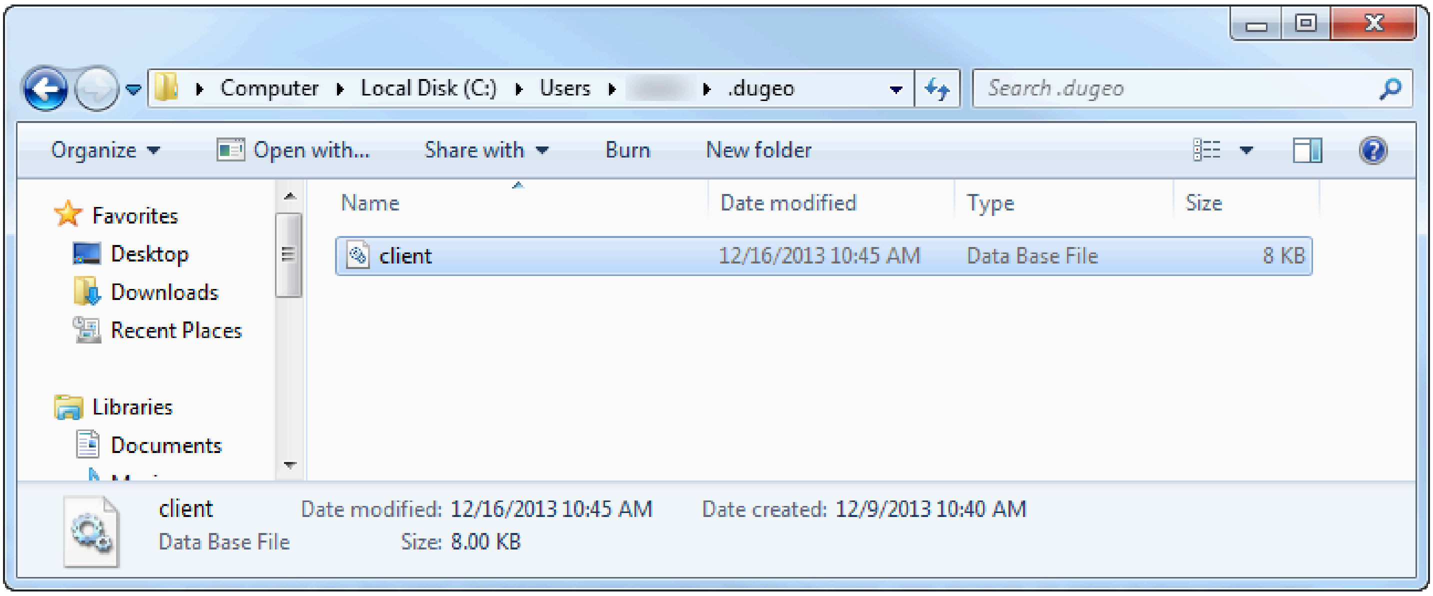
Licensing

How do I activate floating licence for multiple users?

Once you have received your floating licence token from DUG, you can activate the licence for multiple users without having to input the licence token for each user.

After you have activated the licence for one user, the licence details will be stored in a file (*client.db*) in that user's home directory. Follow the steps below to locate and copy the file over to multiple users.

Note: Every instance of Insight consumes a floating licence. Hence, opening multiple projects on a single machine will occupy multiple floating licences.



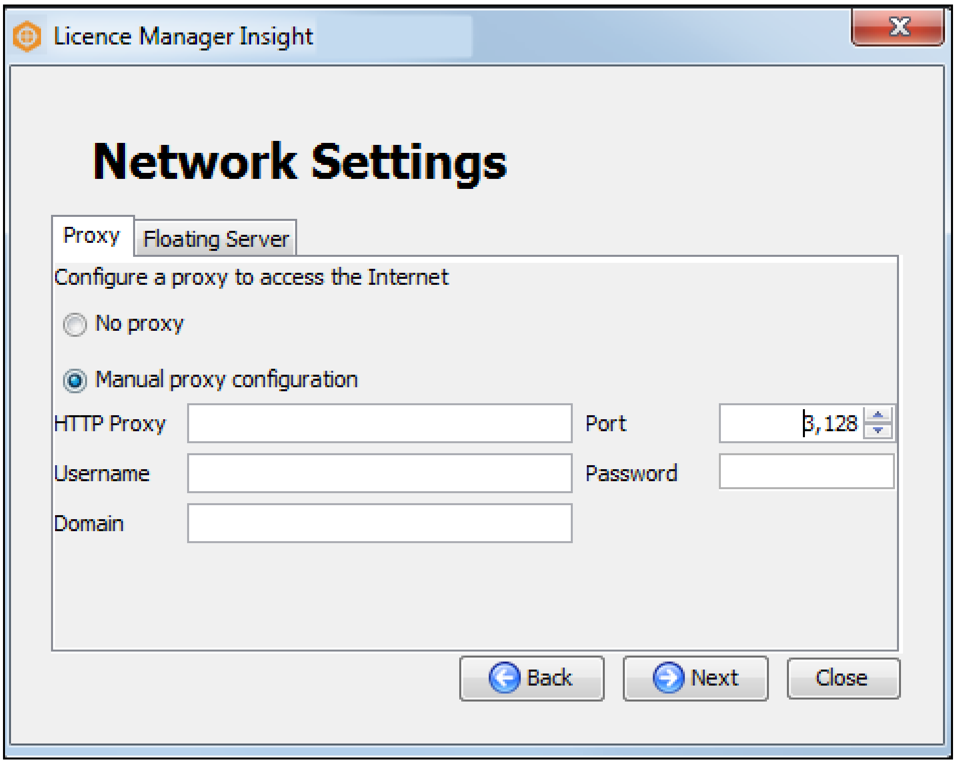
1. Before we begin, make sure you do not have any existing licence token in your workstation. Do a quick search for any *client.db* files and delete them.
2. Activate the licence by following the steps in *Online Activation* or *Email Activation*. This will create a new *client.db* file in your home directory.
3. Search for the file (it is normally located at **C:\Users\[username]\.dugeo\client.db**) and copy (or replace if necessary) the file to each relevant user account on each of their computers.
   Note: The location of this file may vary depending on the operating system the machine is running (*see Home directory*).
4. Restart Insight.

The licence should be activated for all users now.

Why did my online activation fail?

Online activation requires an active internet connection.

If the online activation fails, your computer has either lost its connection to the network, or it requires a proxy to contact our licence servers.



First of all, check that your device is connected to the internet. If it is offline, connect to a network that has an active internet connection.

If you are certain that the device is connected, check your proxy settings by following these steps:

1. At the prompt, or from the **Help** menu, select Licence Manager.
2. From the Licence Manager, click on **Network Settings**. Alternatively, if you have received an error message telling you that the application is locked, select **Configure Network**.
3. In the **Proxy** tab of the **Network Settings** window, select **Manual proxy configuration**. Depending on your system, you may not need to fill all the fields.

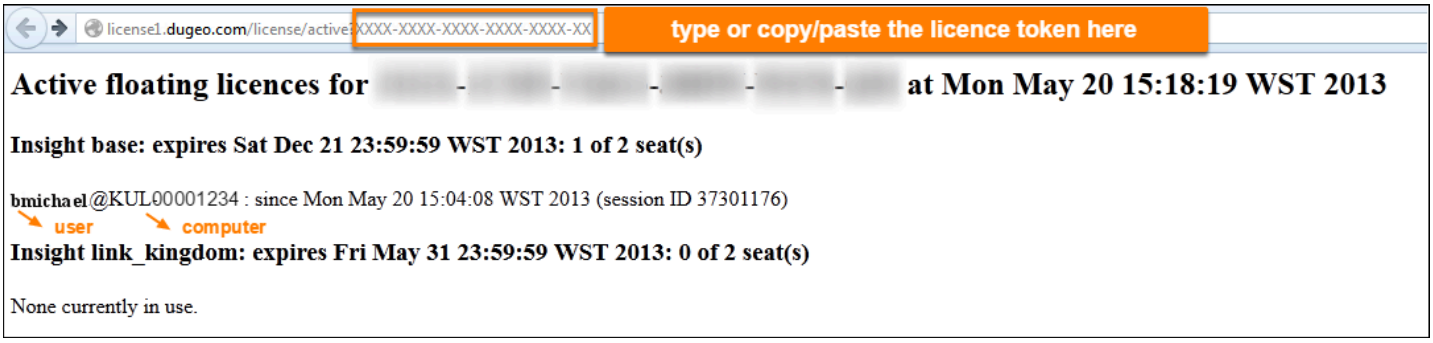
Note: If you are unsure about how to configure your proxy settings, please consult your IT department, or look in the connection settings of a web browser like Internet Explorer or Firefox.

Alternatively, see *Email Activation* to activate your licence offline.

How can I see who is using the floating licences?

To see which users are using which modules on which computers, visit http://license1.dugeo.com/license/active?XXXX-XXXX-XXXX-XXXX-XXXX-XX (replacing the Xs with your DUG licence token).

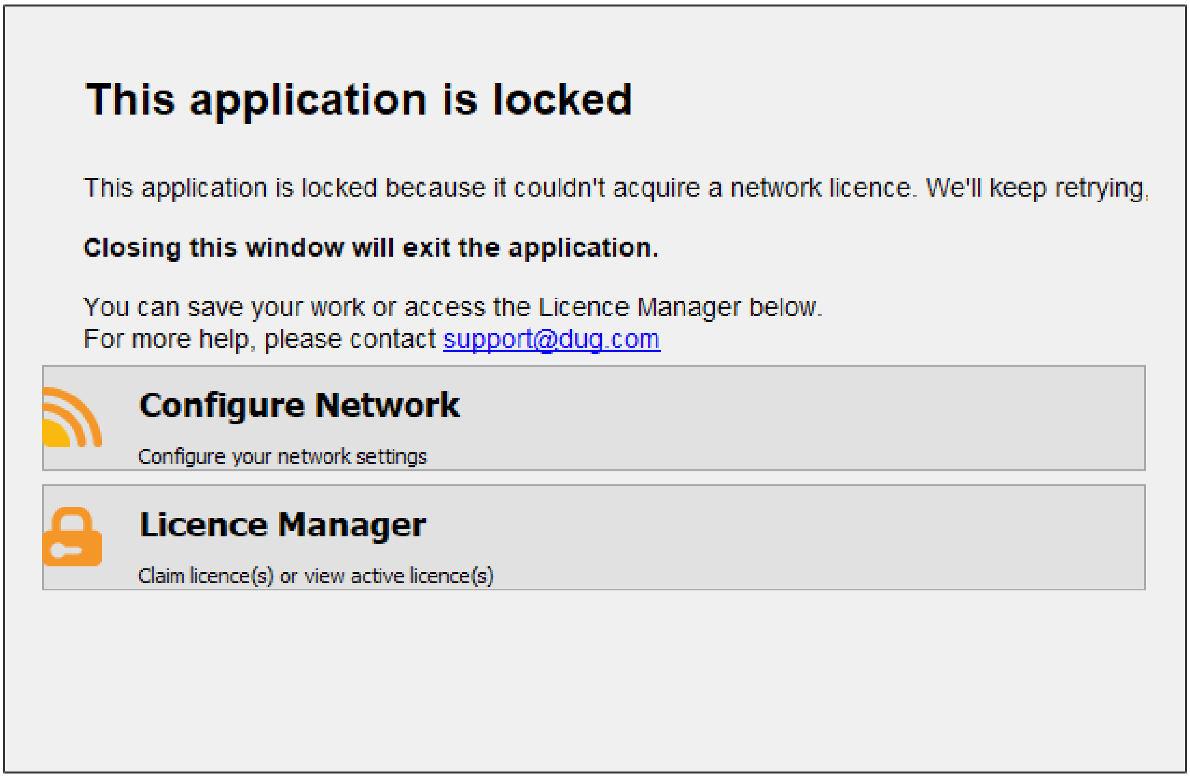
See the following example:



Note: Every instance of Insight consumes a floating licence. Hence, opening multiple projects on a single machine will occupy multiple floating licences.

Why doesn't my floating licence work, even after a successful activation?

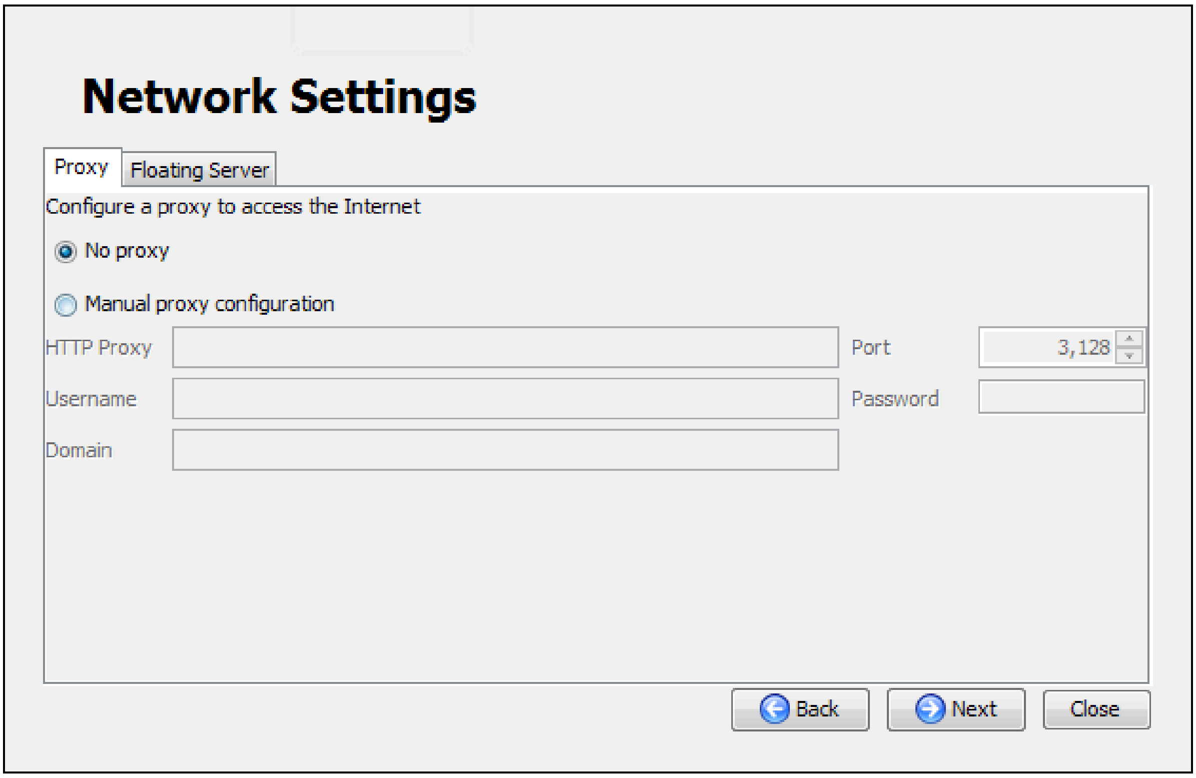
A valid, successfully-activated, floating licence will not work if the software is unable to connect to the licence server, or is configured to use the wrong server.



You may encounter error messages such as this, despite having successfully activated a non-expired floating licence.

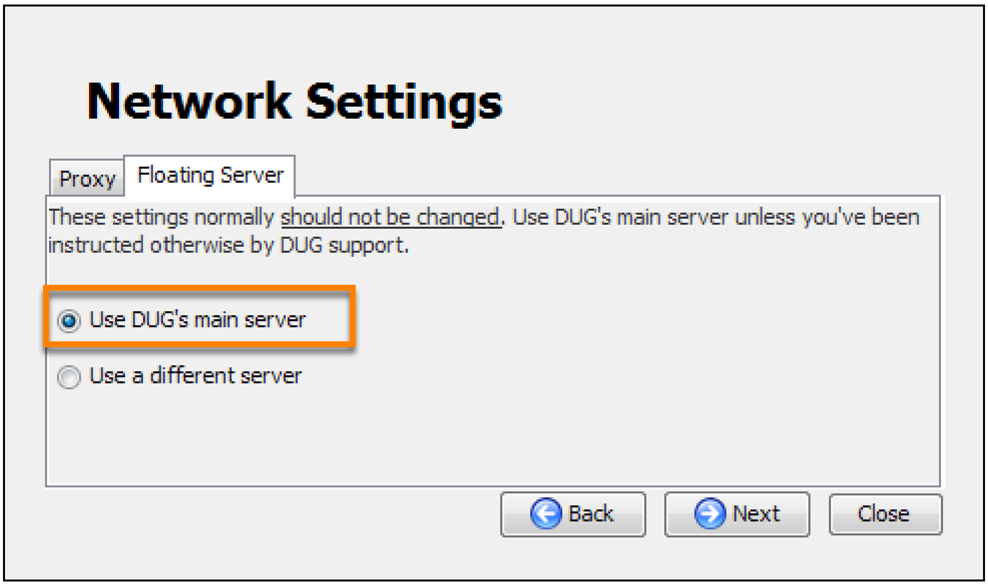
This can be caused by either incorrect proxy settings being entered after activation, or incorrect floating license server details.

Check network settings



Click on **Configure Network** and ensure that the **Proxy settings** are correct. If you do not use a proxy, then ensure that **No Proxy** is selected.

Check floating server



Click on the **Floating Server** tab to check which server you are using.

The default setting is DUG's main server. Unless you or your IT department has made a special arrangement to operate a local licence server, be sure that **Use DUG's main server** is selected.

Otherwise, your licence will activate successfully but you will not be able to actually check out the licence and use Insight.

Sessions and Projects

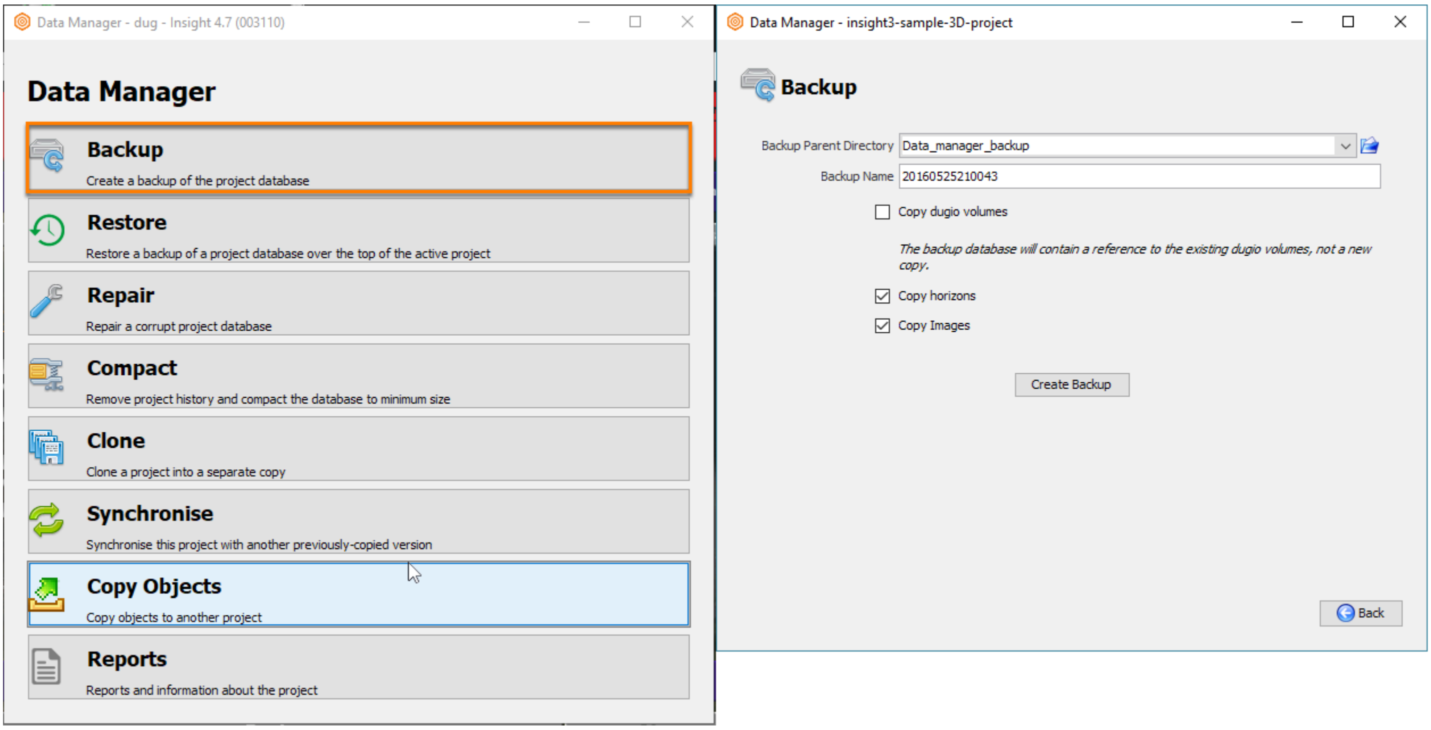
What is the best way to copy and send DUG my project database?

Depending on the situation, Insight support may request your project database for analysis in-house. This is a quick guide on how to copy and send your project database files to us.

Note: We understand that the contents of any data sent to us are confidential and intended for testing and bug fixing purposes only. Under no circumstances will the data be shared or distributed to anyone other than those necessary to solve your problem. You have our promise that we will safeguard your data to the best of our ability. Once we have solved the problem and closed the support ticket, your data will be removed from our system.

- *Copy database via Data Manager*
- *Copy .dugprj file*

Copy database via Data Manager



The safest way to copy a project database is to use the Backup feature in Insight's Data Manager tool (see *Backup a Project Database*).

1. From the DUG Insight Launcher, select the project that you want to copy and the **Data Manager** application, and click **Launch**. The Data Manager window will be displayed.

2. In the Data Manager window, select **Backup**.
3. Choose a **Backup Parent Directory** where you want to keep a copy of the project, or create a new parent directory.
4. Type a name for the Backup folder inside the parent directory (by default, Data Manager sets the name as the current date and time)
5. Deselect **Copy dugio volumes**, **Copy horizons** and **Copy images**. We are only interested in the project database (*.dugprj*) file.
6. Click **Create Backup** to copy your project. A notice will appear to confirm that the copy has been created at the path you specified.

Right click on the backup project directory, select **Send to** and create a compressed (zipped) folder.

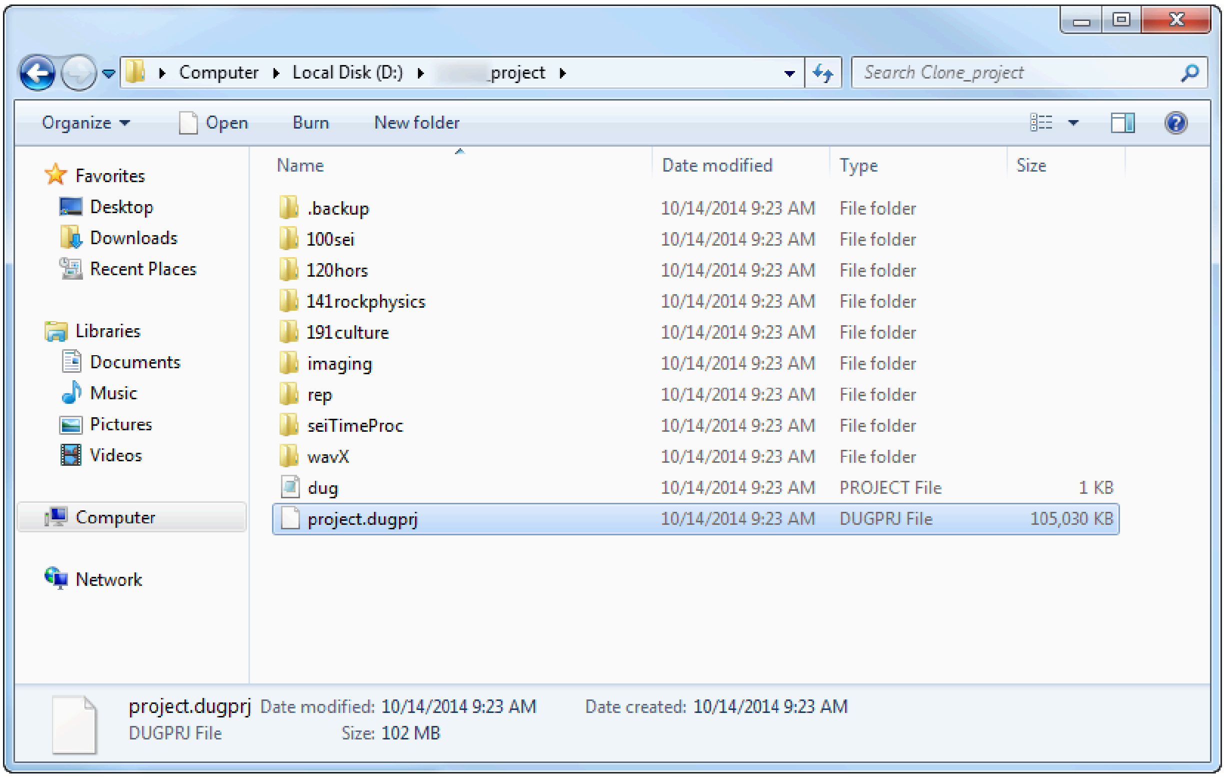
Send this zipped copy to support@dugeo.com or upload it to our FTP site (see *How to upload to our FTP server?* for instructions).

Copy .dugprj file

Alternatively, if you do not have access to the Data Manager tool, you can directly copy the database file from the project directory.

Before copying the project.dugprj file, completely close ALL Insight instances and sessions.
It is important that ALL users are out of the project and the project is completely closed before making a copy.

1. Navigate to the project directory of the database.
2. Right click on the *project.dugprj* file, select **Send to** and create a **Compressed (zipped) folder**.
3. Send this zipped copy to support@dugeo.com or upload it to our FTP site (see *How to upload to our FTP server?* for instructions).

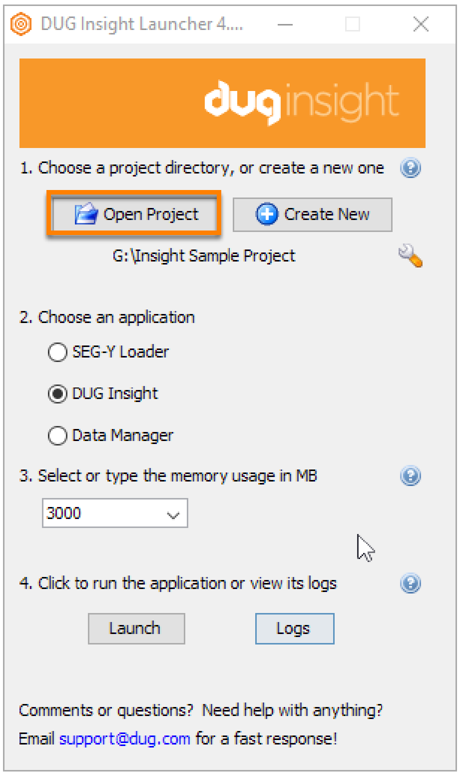

Computer ▸ Local Disk (D:) ▸ project ▸
Search Clone_project
Organize
Open
Burn
New folder
Favorites
Desktop
Downloads
Recent Places
Libraries
Documents
Music
Pictures
Videos
Computer
Network
Name
Date modified
Type
Size
.backup 10/14/2014 9:23 AM File folder
100sei 10/14/2014 9:23 AM File folder
120hors 10/14/2014 9:23 AM File folder
141rockphysics 10/14/2014 9:23 AM File folder
191culture 10/14/2014 9:23 AM File folder
imaging 10/14/2014 9:23 AM File folder
rep 10/14/2014 9:23 AM File folder
seiTimeProc 10/14/2014 9:23 AM File folder
wavX 10/14/2014 9:23 AM File folder
dug 10/14/2014 9:23 AM PROJECT File 1 KB
project.dugprj 10/14/2014 9:23 AM DUGPRJ File 105,030 KB
project.dugprj Date modified: 10/14/2014 9:23 AM Date created: 10/14/2014 9:23 AM
DUGPRJ File Size: 102 MB

How do I load sample data?

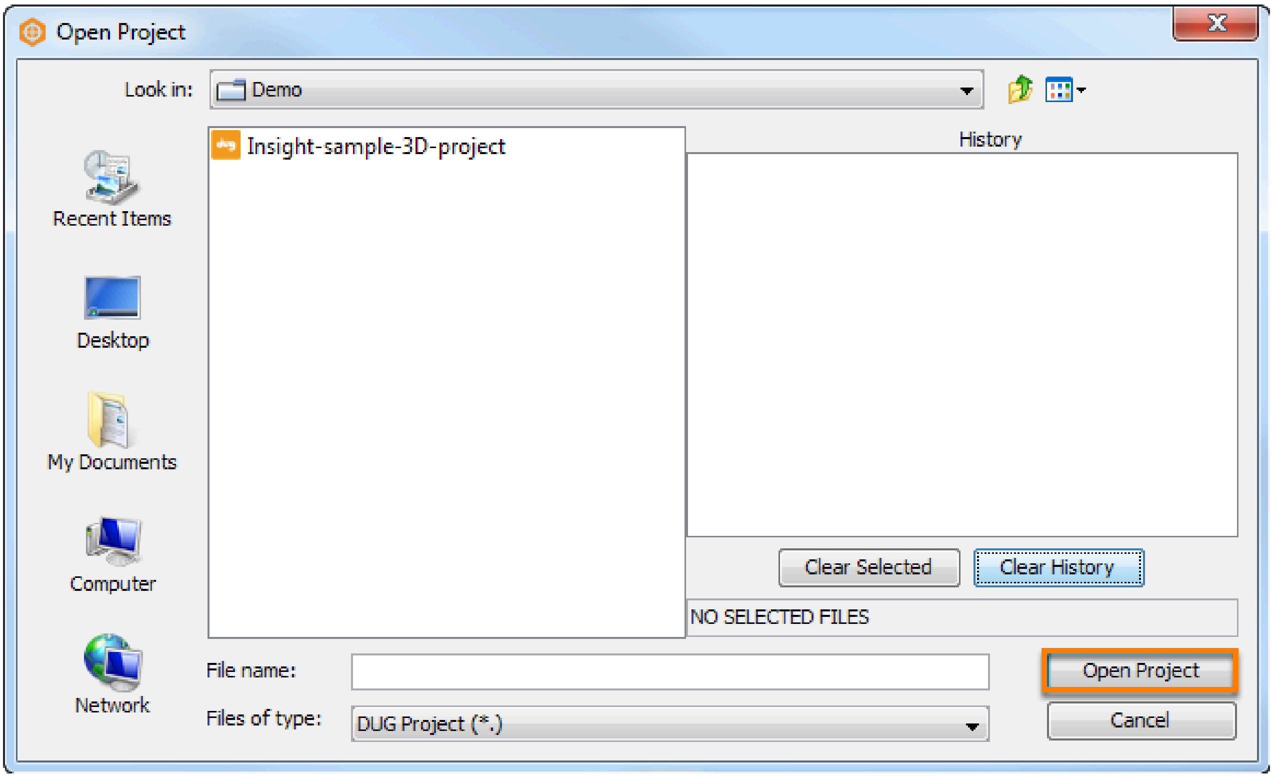
The sample datasets are mini self-contained DUG Insight project directories, each with a pre-configured session. Follow these steps to get started.

Loading sample data

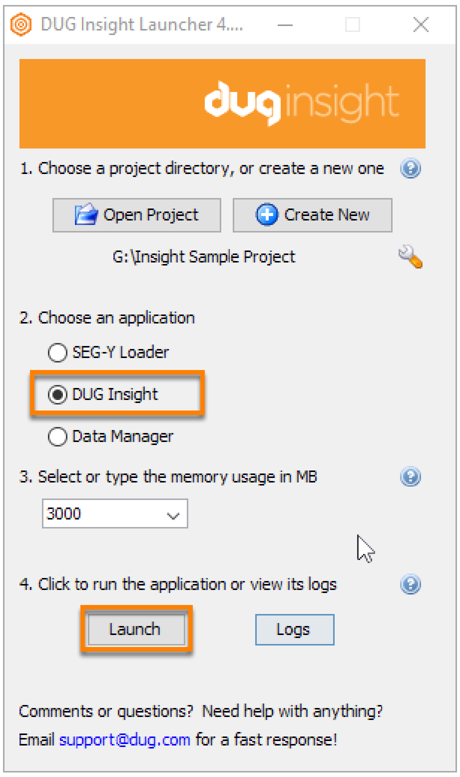
1. Download the sample data from [http://www.dugeo.com/software/resources/sample_data.](http://www.dugeo.com/software/resources/sample_data)
2. Unpacking the ZIP file will create a new directory called "*Insight-sample-2D-project*" or "*Insight-sample-3D-project*".
3. Run the **DUG Insight Launcher** by selecting the Insight icon from the Start menu, Desktop, or (on Mac OS X) Applications directory.



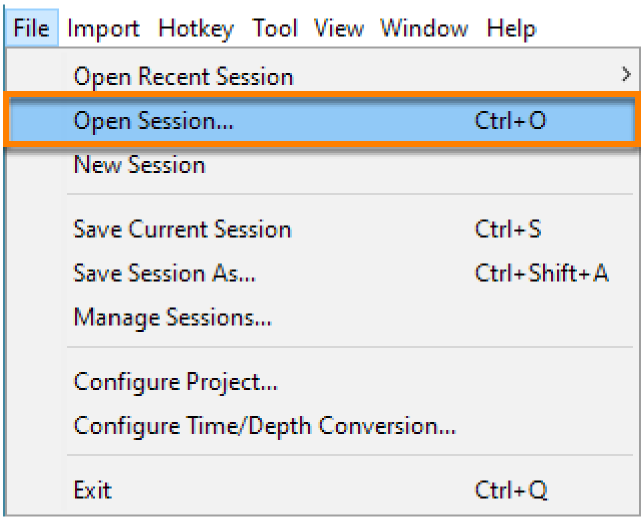
4. Click **Open Project** to browse for the project directory.



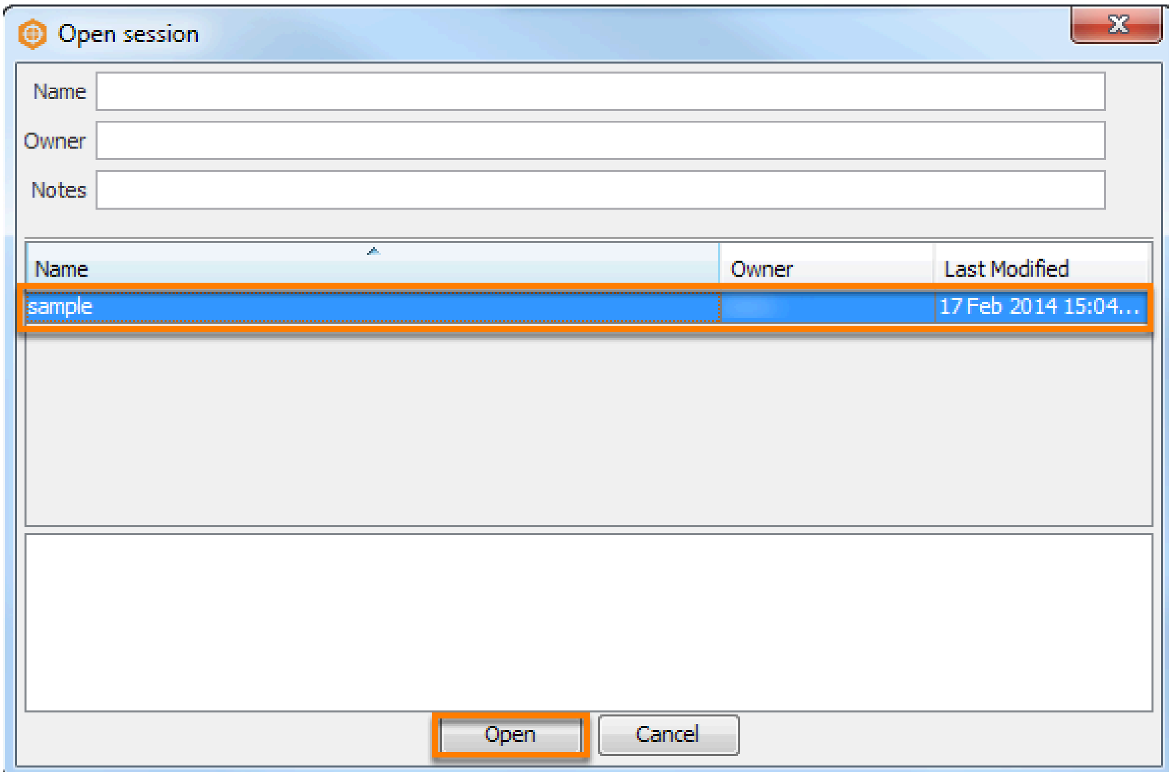
5. Navigate to where you unpacked the ZIP file, and click **Open Project**.



6. Select **DUG Insight** at the Choose an Application section and click **Launch**.



7. Select **Open Session** from the **File** menu.



8. Choose the sample session and click **Open**.

How should I backup my project?

Currently, there are two ways Insight can backup your project:

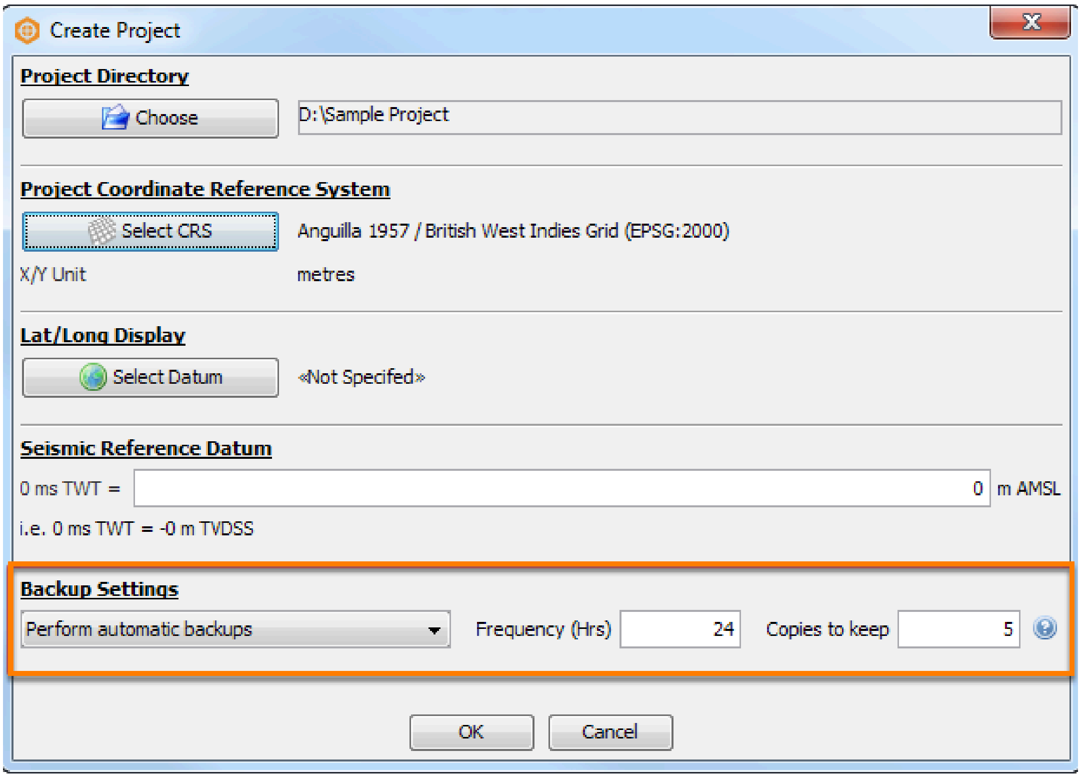
- Insight's default automatic backup, and
- manual backup using Data Manager.

We recommend using both methods to backup your project as they are both unique in the way they perform the backup. The automatic backup creates a copy of the *.dugprj* project database file in a *.backup* folder in your project directory. However, horizons, volumes, TIFFs, etc, are NOT contained within the database. The database file only contains references to the individual objects and not the actual physical copy. To physically backup the project directory, you have to use the Backup feature in Data Manager. Hence, it is recommended that you regularly create full, physical backups of your Insight project as well.

Do NOT copy the project using normal tools (i.e. copy/paste), as Insight might modify the files during copy, or other users might be logged in and modifying the project while the backup is done, resulting in a corrupt copy of the project database. The only safe ways to make backups are the two mentioned in this lesson.

Note: The automatic backups configured in the project settings should not be confused with session autosaves. Autosaves record the *session* (not the project), and occur once every 5 minutes. Only one autosave is available per session as each new autosave overwrites the previous one. As soon as the session is manually saved on closing, the last autosave is discarded (see *Autosaves* in *Project vs Session*). In contrast, the Automatic Backup creates and keeps a number of copies of the *project* database (as defined by the user) at a user-defined frequency.

Automatic backup

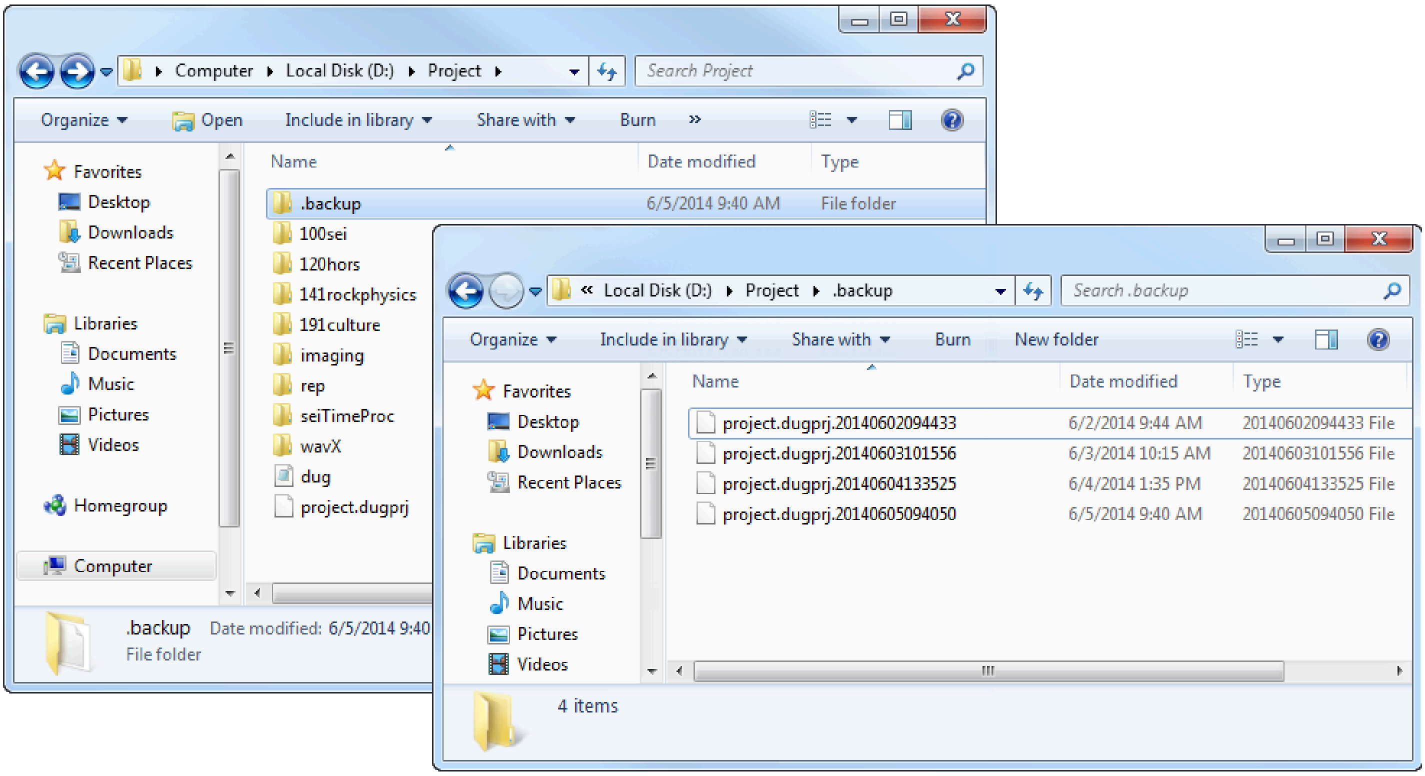


When *creating* or *configuring a project*, you can enable automatic backups by selecting **Perform automatic backups** under the **Backup Settings** section. Specify the number of hours that must pass for Insight to create a new backup and the number of copies you want the system to keep before deleting older copies.

The automatic backup creates a copy of the *.dugprj* project database file in a *.backup* folder in your project directory. Before backing up, Insight will check if there are other users logged in (an "integrity check"), and locks the project from being modified while it backs up the project database to the backup folder. At every start-up of the project, Insight will automatically check if the required hours have passed to create a backup. It it has, then it will create one.

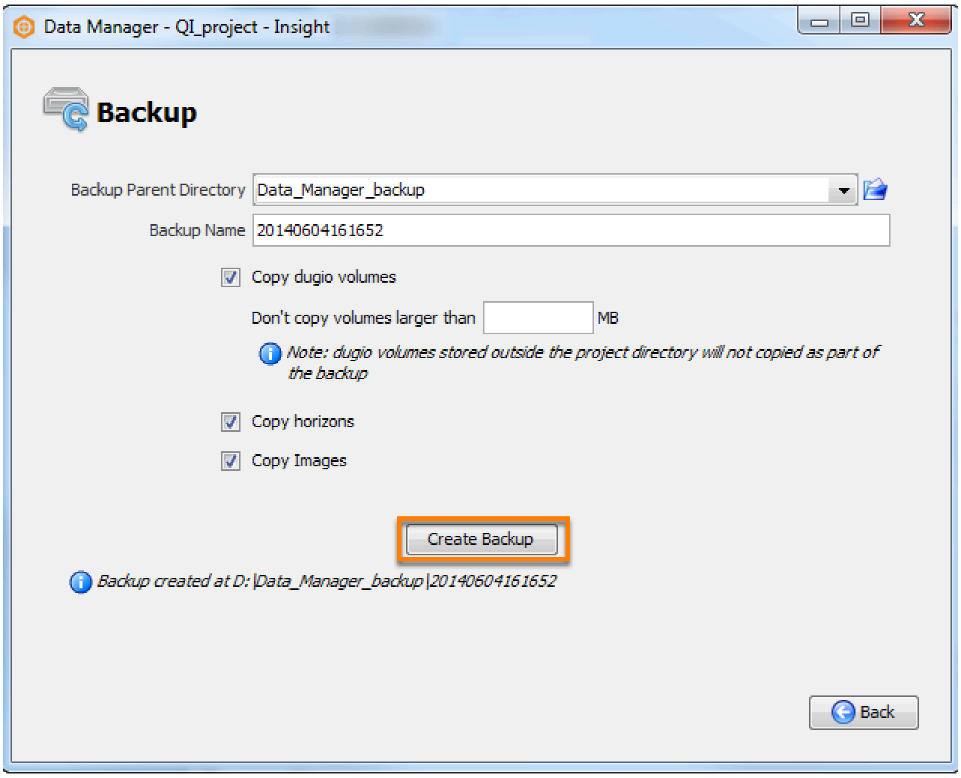
Note: If your database is corrupted, Insight will disable access to the database until the project is fixed. This is to prevent any further file corruption, and prevent Insight from generating backups of the corrupted project. If your database is corrupted, please consult our support team at support@dugeo.com. You might be required to send diagnostic logs of your project session to help us understand why it is happening (see *Viewing and Sending Diagnostic Logs*).

The following image shows an example of the backup folder and backup project database files.



Note: The backup folder might be a "hidden" folder. If you do not see the backup folder in your project directory, you might have to enable viewing of hidden files from the **Folder Options** of your machine's Control Panel (see *Show Hidden Files*).

Data Manager manual backup

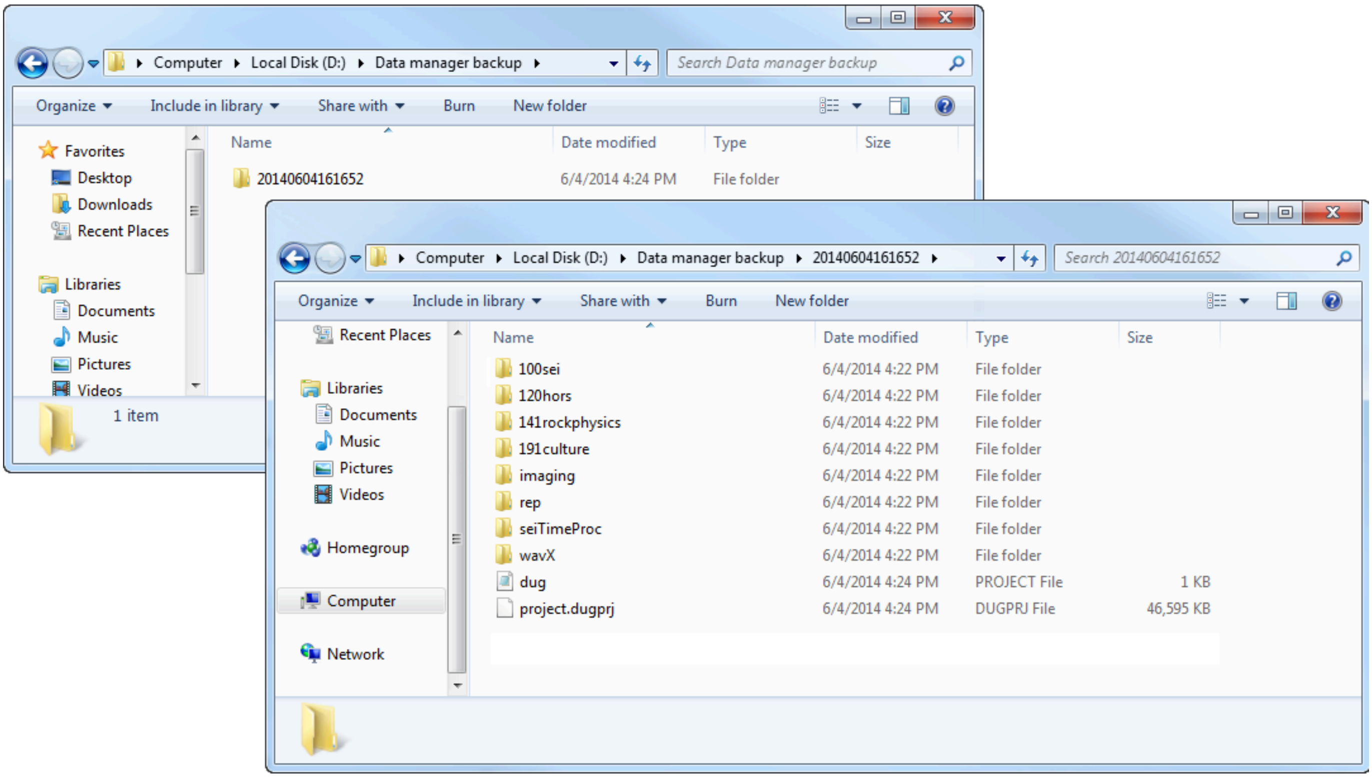


The **Backup** feature in Data Manager creates a physical copy of the project directory into a **Backup Parent Directory** of your choice. By default, Data Manager sets the **Backup Name** as the current date and time for convenience when restoring using the Restore feature (*see* Restore a Project Database).

Select the **Copy dugio volumes/horizons/images** check boxes to backup the physical copies of the objects. As volumes are often large files, you can choose to ignore volumes over a certain size by specifying the maximum size in the **Don't copy volumes larger than X MB** field. If you do not select the check boxes, the backup will only copy references to the objects.

Any volumes external to the project will not be included when you create a backup. To create a copy of the database with external volumes, see *Clone a Project Database*.

Note: Backups of the project database file generated by the automatic backups are NOT included in the Data Manager backup directory.


Computer
Local Disk (D:)
Data manager backup
Search Data manager backup
Organize
Include in library
Share with
Burn
New folder
Favorites
Desktop
Downloads
Recent Places
Libraries
Documents
Music
Pictures
Videos
Name
Date modified
Type
Size
20140604161652
6/4/2014 4:24 PM
File folder
1 item
Computer
Local Disk (D:)
Data manager backup
20140604161652
Search 20140604161652
Organize
Include in library
Share with
Burn
New folder
Recent Places
Libraries
Documents
Music
Pictures
Videos
Homegroup
Computer
Network
Name
Date modified
Type
Size
100sei
6/4/2014 4:22 PM
File folder
120hors
6/4/2014 4:22 PM
File folder
141rockphysics
6/4/2014 4:22 PM
File folder
191culture
6/4/2014 4:22 PM
File folder
imaging
6/4/2014 4:22 PM
File folder
rep
6/4/2014 4:22 PM
File folder
seiTimeProc
6/4/2014 4:22 PM
File folder
wavX
6/4/2014 4:22 PM
File folder
dug
6/4/2014 4:24 PM
PROJECT File
1 KB
project.dugprj
6/4/2014 4:24 PM
DUGPRJ File
46,595 KB

Where is my user home directory?

In a machine with multiple users, user-specific files are stored in a default home directory for each user. For Insight, user-specific files that are saved in user's home directory include the:

- ***global.properties*** file — containing configuration files and preferred settings for HTTP proxy and problem reporter settings (i.e. email, notification enabled, etc)
- ***client.db*** file — containing Insight's licence details for the user (i.e. floating/node-lock, licence number, etc)
- ***DUG_Launcher.properties*** file — containing Insight's launcher settings/history (i.e. recently opened projects and memory allocation).

The location of these files may vary depending on the operating system the machine is running. The table below shows the path of the home directory for different operating systems.

<table>
<tr><th>Operating System</th><th>Path</th><th>Environment Variable</th></tr>
<tr><td>Microsoft Windows NT</td><td>(root)\WINNT\Profiles\(username)</td><td rowspan="3">%UserProfile%</td></tr>
<tr><td>Microsoft Windows 2000, XP and 2003</td><td>(root)\Documents and Settings\(username)</td></tr>
<tr><td>Microsoft Windows Vista, 7 and 8</td><td>(root)\Users\(username)</td></tr>
<tr><td>Unix-Based</td><td>(root)/home/(username)</td><td rowspan="4">$HOME and ~/</td></tr>
<tr><td>Unix-Derived</td><td>/var/users/(username)
/u01/(username)
/usr/(username)
/user/(username)
/users/(username)</td></tr>
<tr><td>SunOS/Solaris</td><td>/export/home/(username)</td></tr>
<tr><td>Linux/BSD (FHS)</td><td>/home/(username)</td></tr>
</table>

AT&T Unix (original version)	(root)/usr/(username)	$HOME
Mac OS X	/Users/(Username)	$HOME and ~/, and path to home folder (in Apple Script)
OpenVMS	(device):[username]	SYS$LOGIN
Android	/data/media/(userid)	

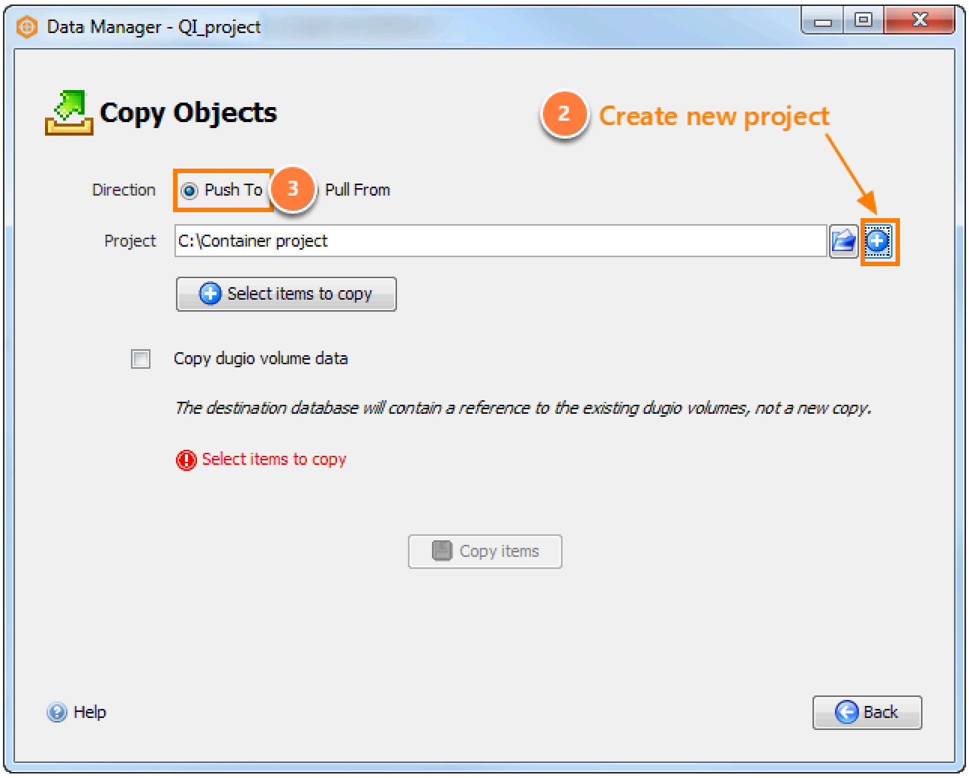
How do I sync my session across offices?

In our experience, many of our users work within teams across multiple offices, and it is vital to be able to sync project data and sessions between offices.

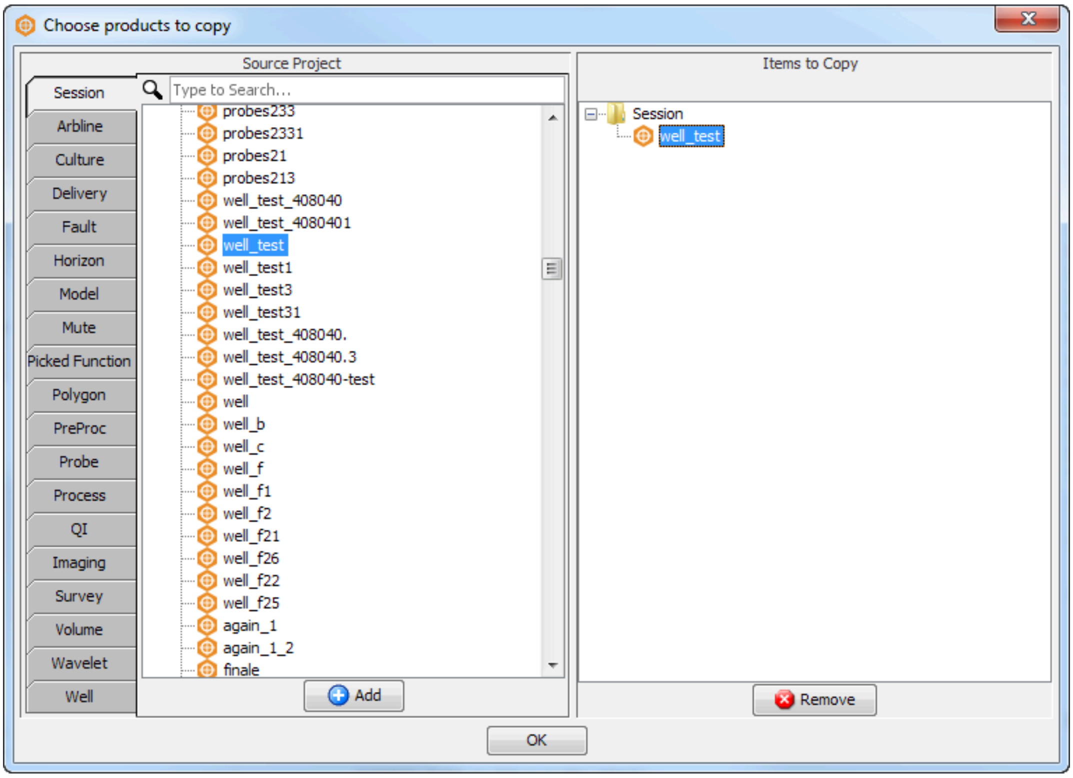
For this purpose, we have created a feature in Data Manager specifically to copy project data and sessions to a new project or an existing project.

For more information, see *Copy Objects to Project*.

Copy Objects



1. Open the **Data Manager** application from the DUG Insight Launcher.
2. In the **Copy Objects** interface, create a new project. The new project will be the container for the items to be transferred. In these instructions, we will call it the "container project".
3. To transfer items to the container, choose **Push To** and click **Select items to copy**. The following window will be displayed.



4. Include your session in the **Items to Copy** section by double clicking, or selecting it and clicking Add. Be careful if you choose to include seismic data! The size of the transfer can get large very quickly.
5. Click **OK**.
6. When you click **Copy Items** and complete the process, all the items in the session will be copied into the container project.

Once the process is complete, the container project can be zipped, archived, or transferred directly using *rsync* (refer to your IT department for assistance).

After transferring the session to the destination, have the destination project open, and use the same Copy Objects function. This time, choose **Pull From** to copy the session out of the container project.

Genuine offer of assistance

If the transfer is unsuccessful, or if you have any questions, please contact our support team at support@dugeo.com.

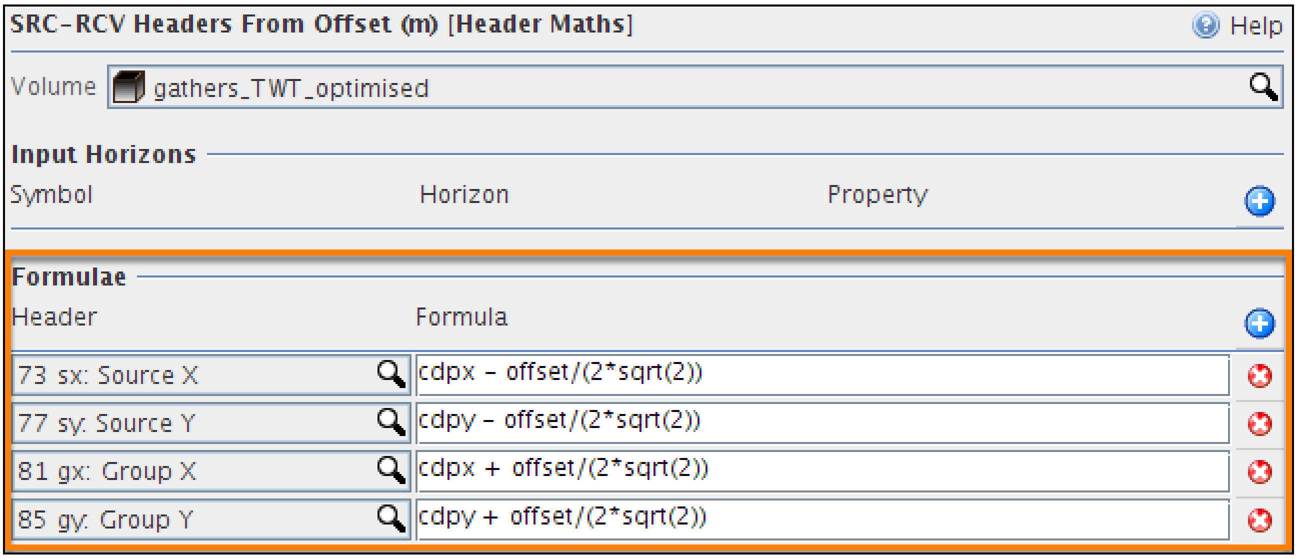
Workflows

How do I create "fake" source and receiver headers from offset values?

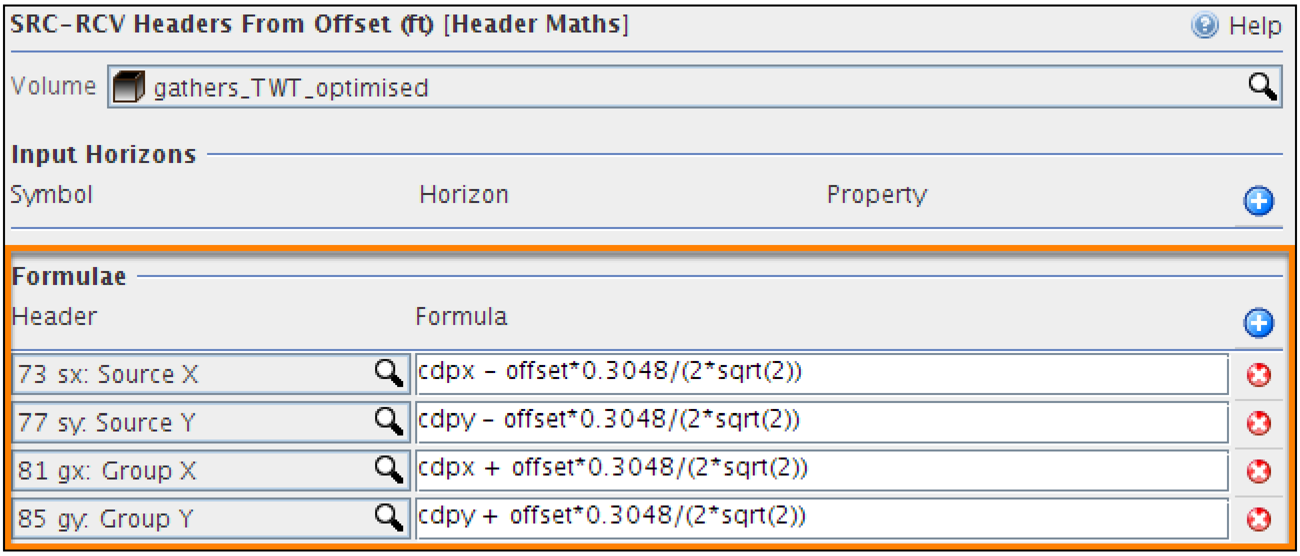
Use the following approach to calculate artificial source and receiver headers that are consistent with the trace offset value.

1. Create a **Header Maths** process.
2. Select the **gathers volume**.
3. Click the **"+"** button in the **Formulae** section to add a formula.

For offset values in the same units as the X/Y coordinates (i.e. metres to metres, feet to feet), use:



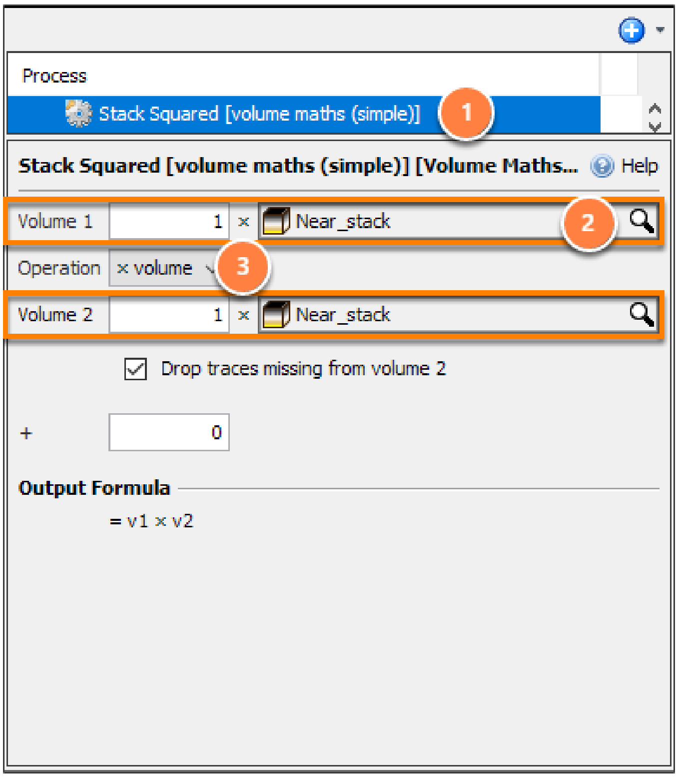
A unit conversion can also be applied. For offset values in feet and X/Y coordinates in metres, use:


SRC-RCV Headers From Offset (ft) [Header Maths]
Help
Volume
gathers_TWT_optimised
Input Horizons
Symbol
Horizon
Property
Formulae
Header
Formula
73 sx: Source X
cdpx - offset*0.3048/(2*sqrt(2))
77 sy: Source Y
cdpy - offset*0.3048/(2*sqrt(2))
81 gx: Group X
cdpx + offset*0.3048/(2*sqrt(2))
85 gy: Group Y
cdpy + offset*0.3048/(2*sqrt(2))

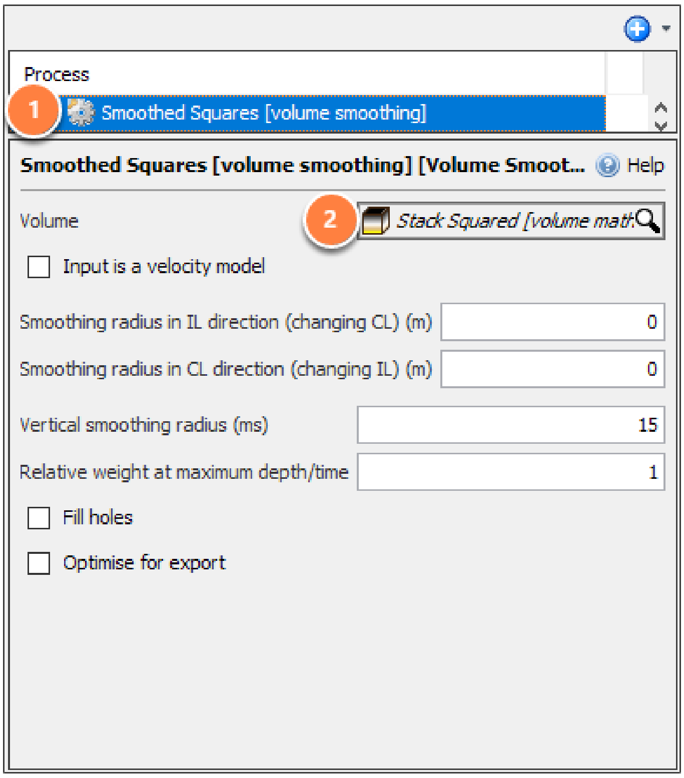
Calculate a Pseudo-Relief Attribute (TECVA)

Generating a pseudo-relief attribute (also known as TECVA) can assist with structural interpretation. It is often easier to correlate events on the enhanced reflectors, simplifying both fault and horizon interpretation.

Reference: Bulhoes, E. M., 2005, Principio da SismoCamada Elemantar e sua aplicacao a Tecnica Volume de Amplitudes (tecVA). Ninth International Congress of the Brazilian Geophysical Society.

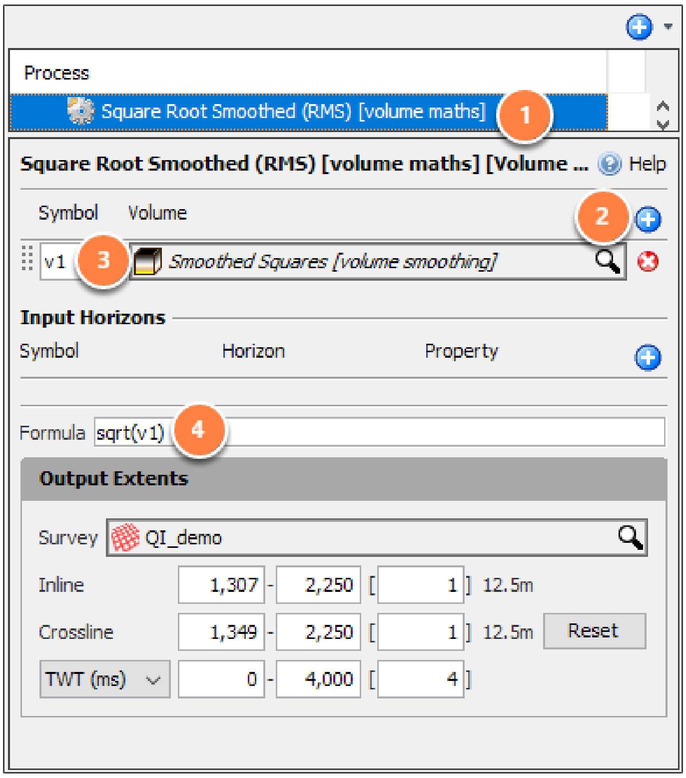


1. Go to the **Process** tab, add a **Volume Maths (Simple)** process (see *Volume Maths Process (Simple)*).
2. **Volume Maths (Simple)** settings:
   - **Volume 1 and Volume 2:** Select the input stack.
   - Use the multiply **operation:** x volume.
3. The result will be a selectable volume.



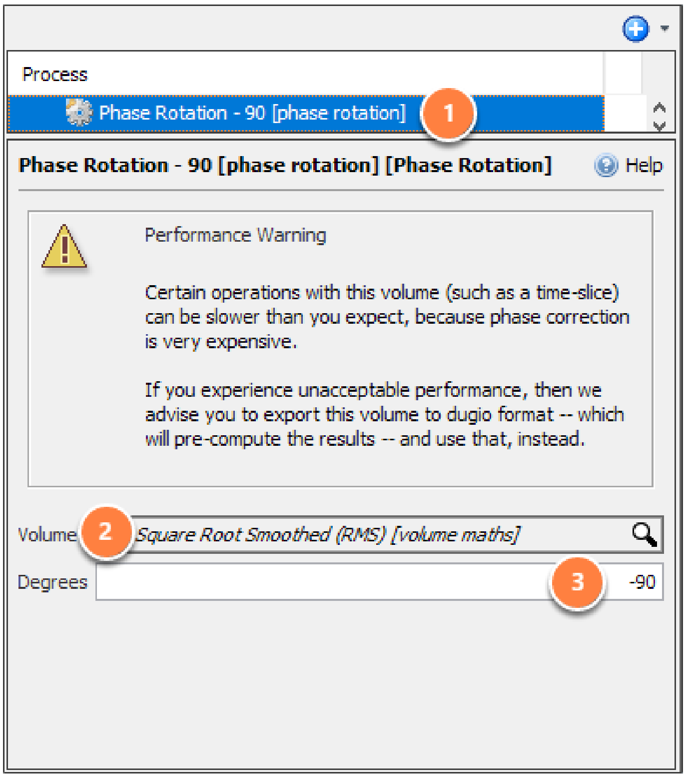
Next, average the squared input stack from the previous process.

1. In the **Process** tab, add a new **Volume Smoothing** process (see *Volume Smoothing*).
2. **Volume:** select the created volume from the previous process.
3. A new volume will be created.



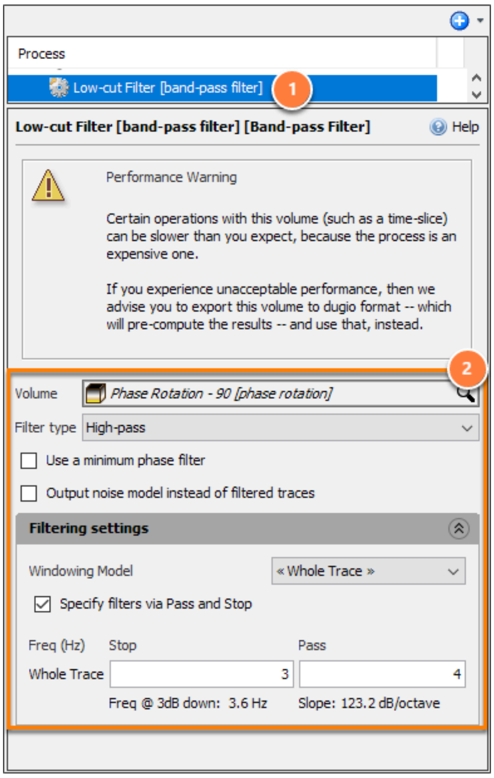
Next, find the square root of the average (from the previous process).

1. Add a **Volume Maths** process from the process tab (see *Volume Maths*).
2. Click the **blue "+"** in the **Volume Maths** panel.
3. **Volume Maths** settings:
   - **Symbol:** v1 - select the volume created from the averaging steps.
   - **Formula:** sqrt (v1)
4. A new volume is created.



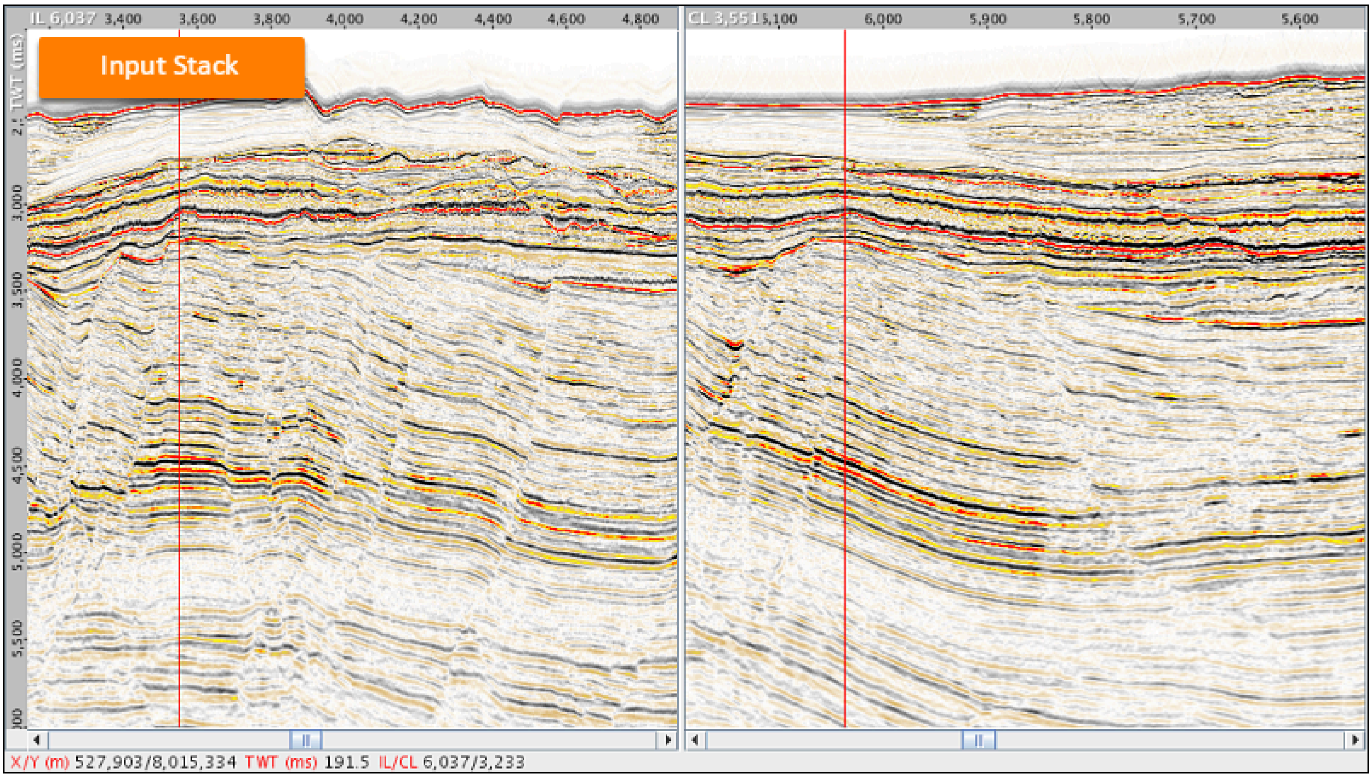
Phase rotate the pseudo-RMS using a process.

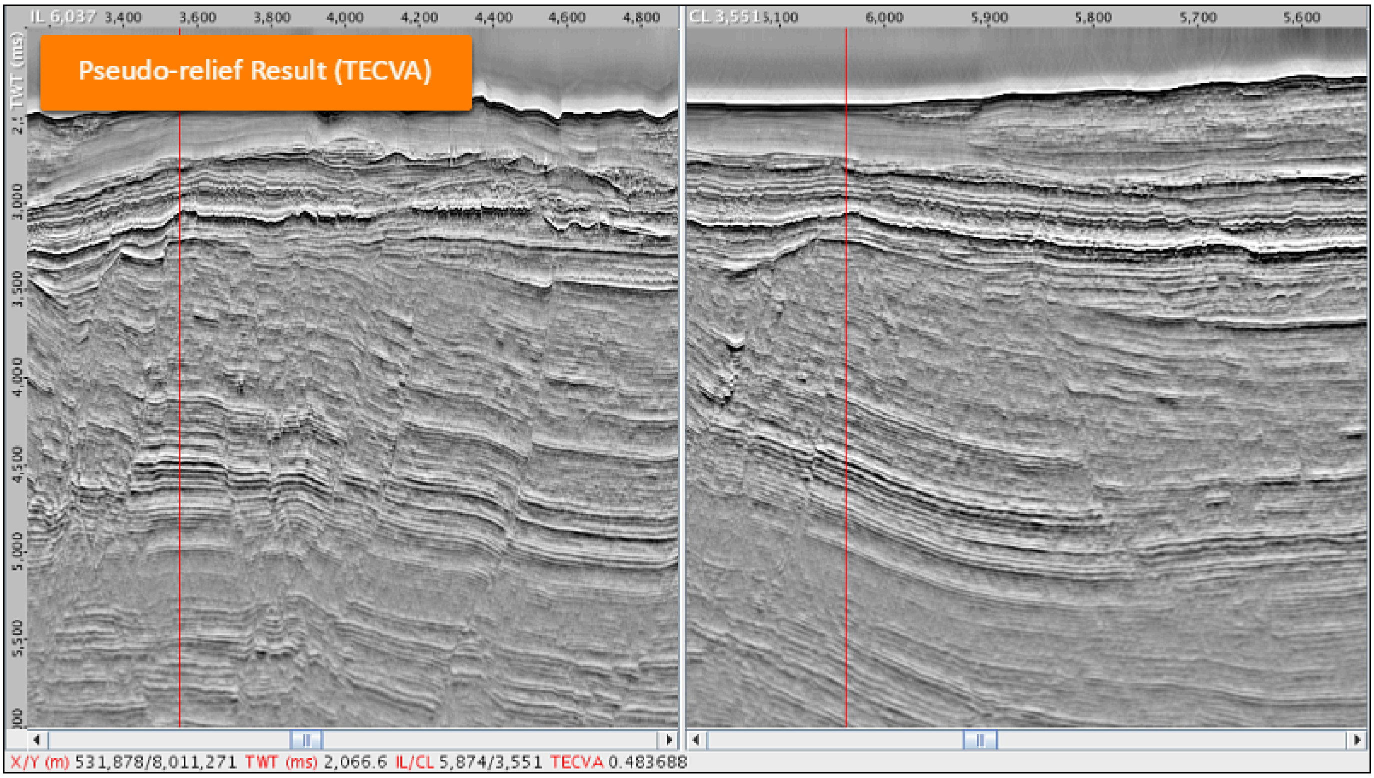
1. Add a **Phase Rotation** process (see *Phase Rotation*)
2. **Phase Rotation** settings:
   - **Volume:** select the square root volume.
   - **Degrees:** -90
3. The new volume is created.

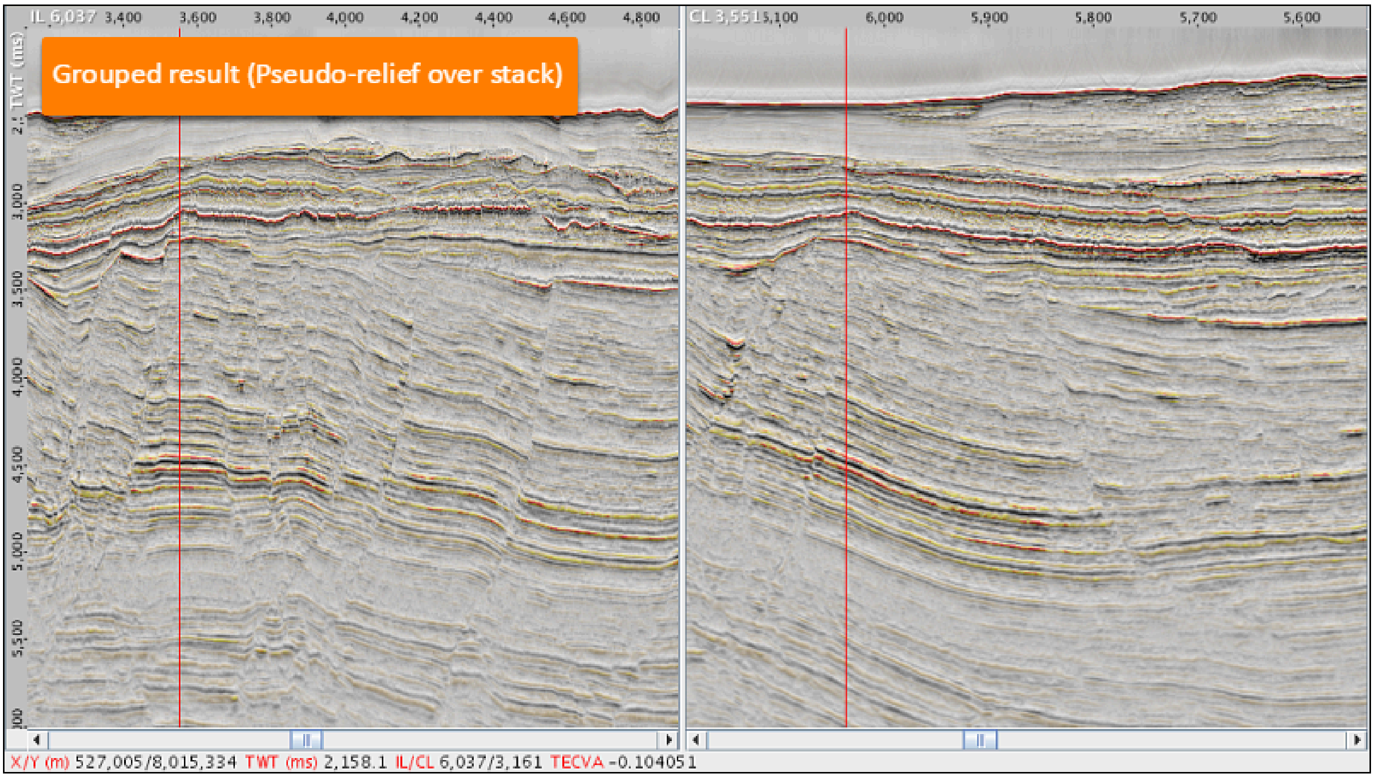


Finally, apply a Low-cut filter to remove low-frequency vertical striping.

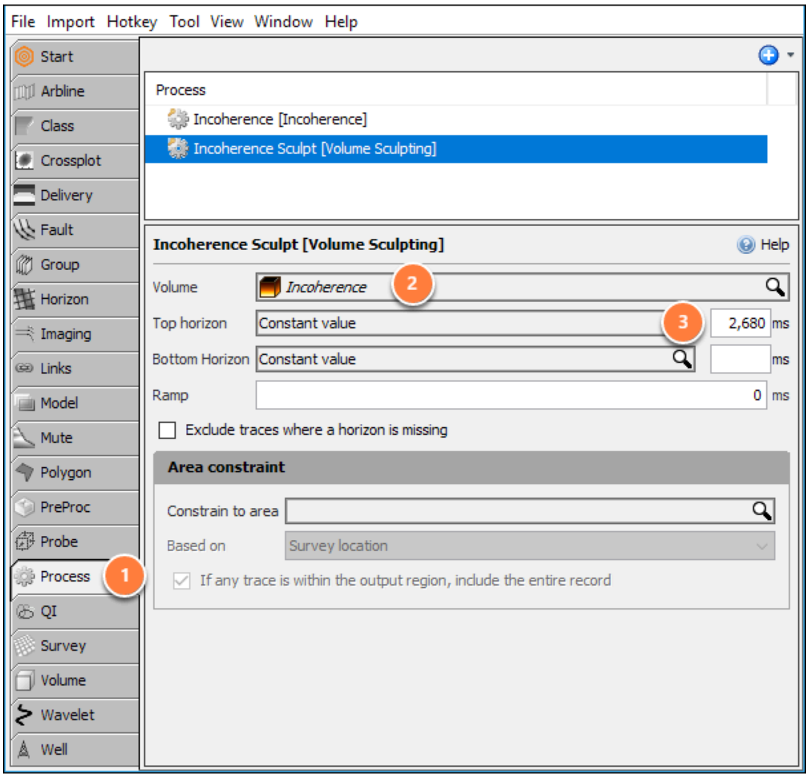
1. In the **Process** tab, add a **Band-pass Filter** process (see *Band-pass Filter process*).
2. **Band-pass Filter** settings:
   - **Volume:** choose the phase-rotation volume
   - **File type:** choose High-pass
   - **Windowing Model:** select <<Whole Trace>>
   - **Stop:** 2Hz
   - **Pass:** 3Hz
3. If the result looks "smeared", try increasing both the **Stop** and **Pass** values by 1 or 2 Hz.


Input Stack
IL 6,037
CL 3,551
TWT (ms)
X/Y (m) 527,903/8,015,334 TWT (ms) 191.5 IL/CL 6,037/3,233

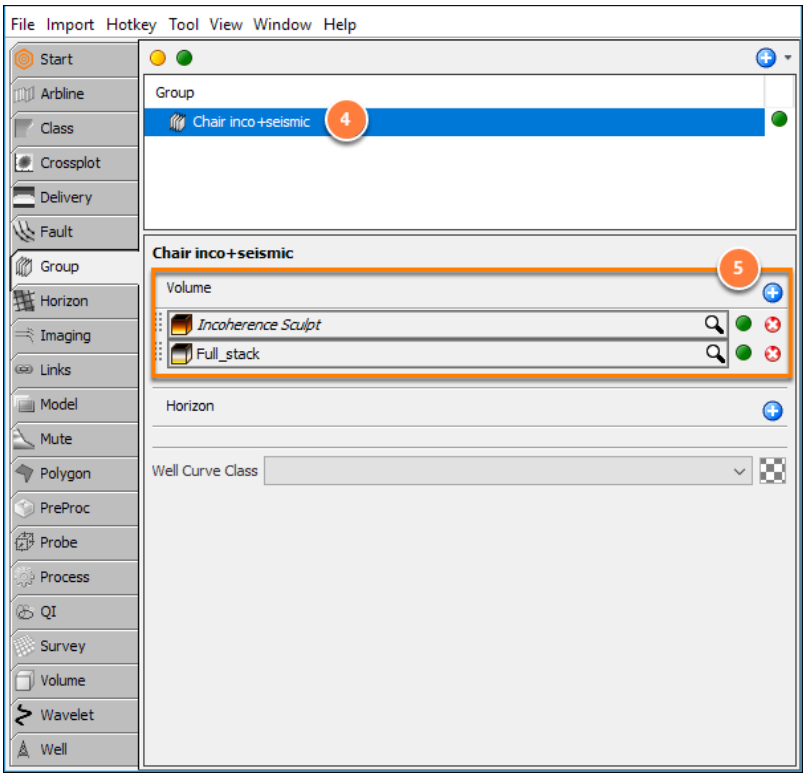

Pseudo-relief Result (TECVA)
IL 6,037
CL 3,551
TWT (ms)
X/Y (m) 531,878/8,011,271 TWT (ms) 2,066.6 IL/CL 5,874/3,551 TECVA 0.483688


Grouped result (Pseudo-relief over stack)
IL 6,037
CL 3,551
TWT (ms)
X/Y (m) 527,005/8,015,334 TWT (ms) 2,158.1 IL/CL 6,037/3,161 TECVA -0.104051

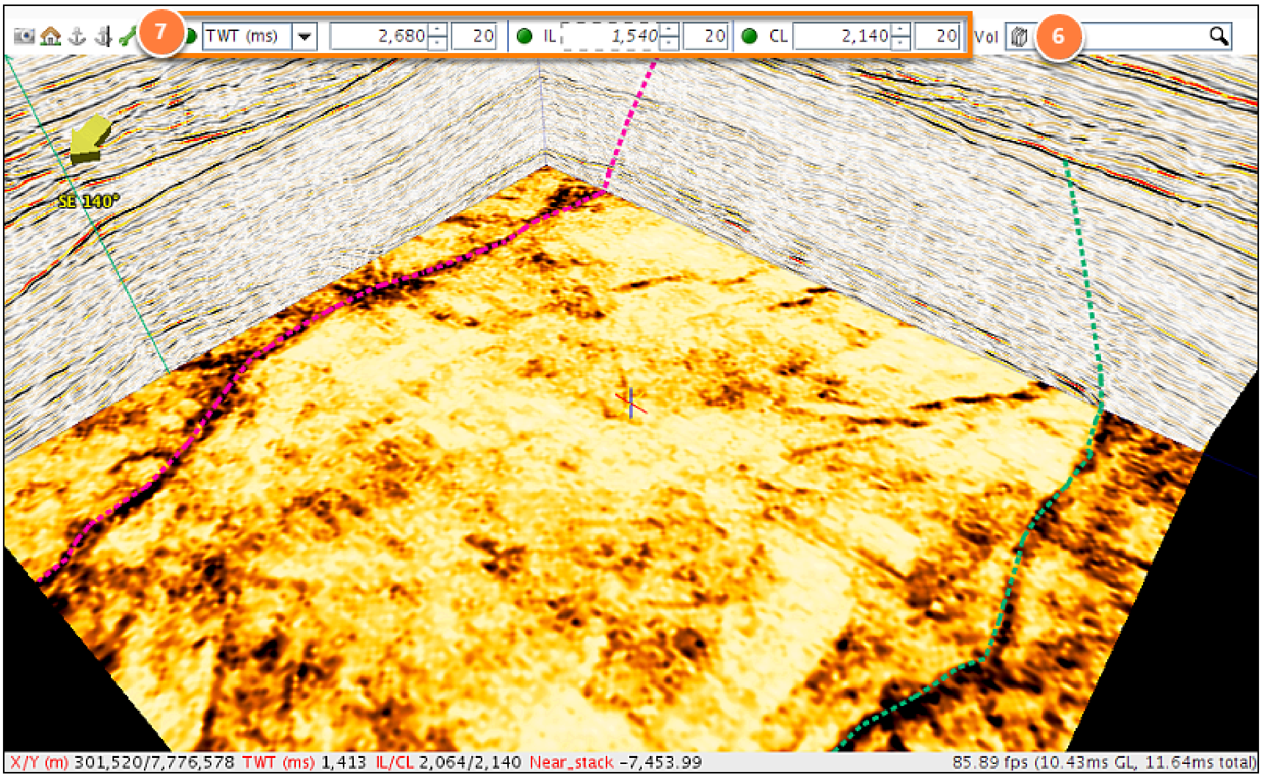
How do I create a "chair" display in the 3D view using two different volumes?



1. Go to the **Process** tab and create a **Volume Sculpt** process (see *Volume Sculpt process)* e.g. "*Incoherence sculpt*".
2. Use the incoherence (or other time-slice attribute) as input for the Volume field.
3. Set the *Top horizon* as **Constant value** of the slice to display, e.g. TWT 2680.



4. Go to the **Group** tab and create a group e.g. "*Chair inco+seismic*".
5. Click the blue "**+**" button and add the volumes for:
   - "*Incoherence sculpt*",
   - Seismic stack (or volume to display vertically).

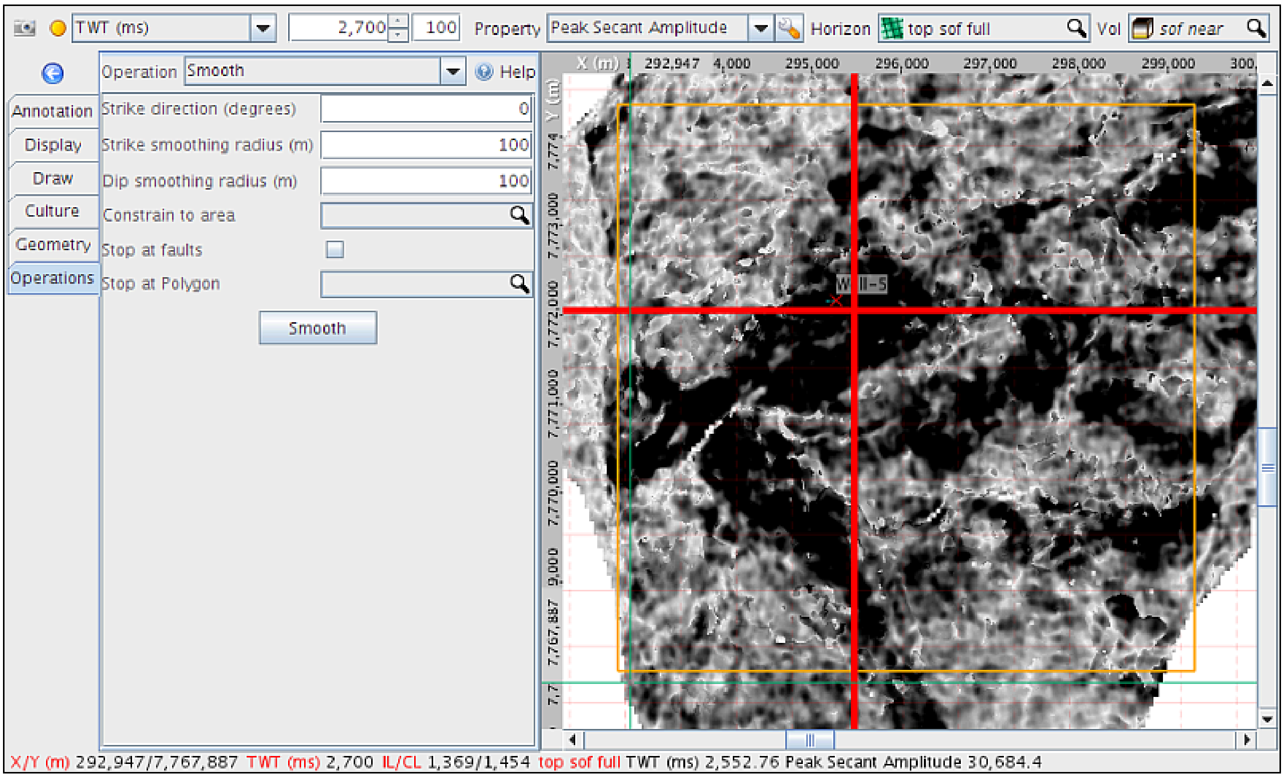


6. In the **3D View**, select the group in the **Vol** search box.

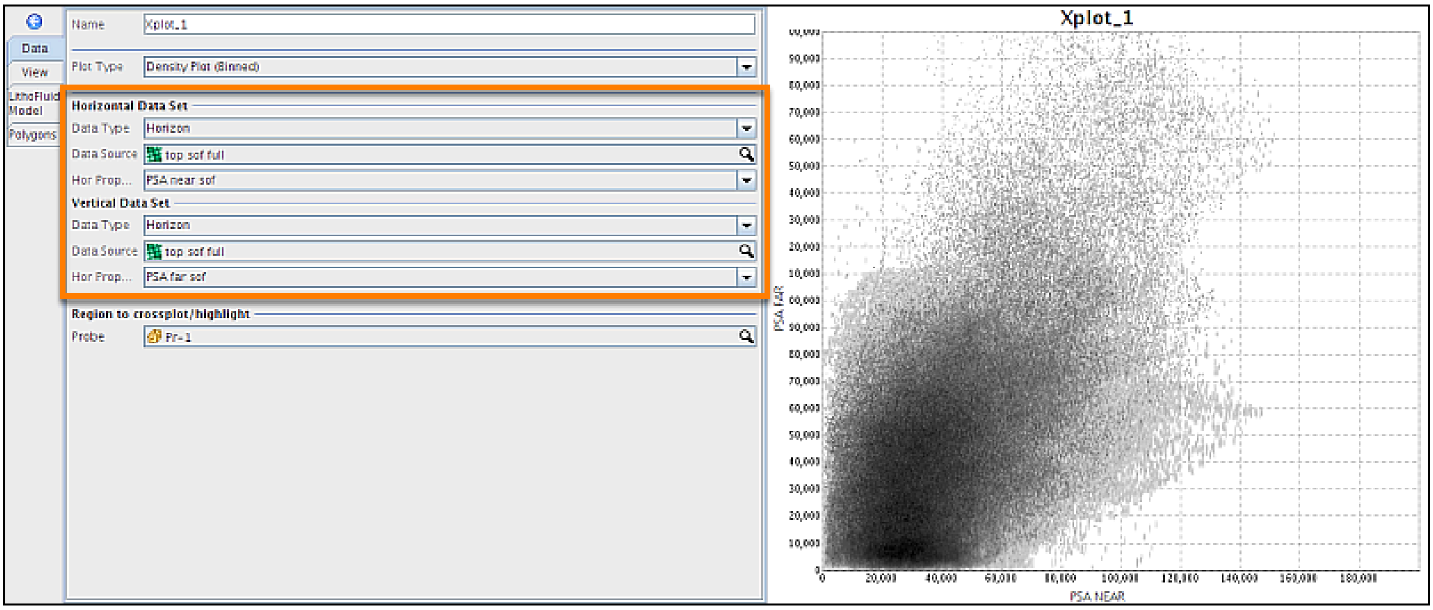
7. Turn on **IL**, **CL** and **TWT** (e.g. at TWT=2680).

How to calibrate interface PDFs to seismic amplitudes using crossplots

The workflow is based around a specific event. Also, instead of determining the reflectivity from the seismic, we will scale the reflectivity PDF model to match the seismic. Some kind of inversion is the best way to calculate reflectivity from seismic.

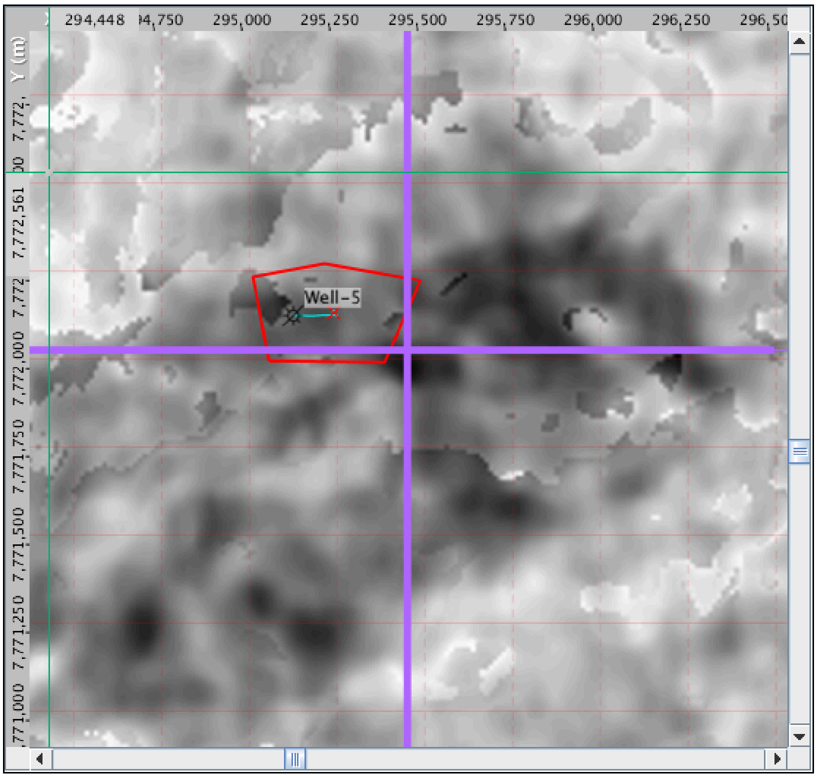


1. Start by picking as much of the event as possible.
2. View the horizon property, "Peak Secant Amplitude" [PSA] to extract amplitudes from the stacks.
   - This property correlates better with reflectivity than "vanilla" peak amplitude.
   - View the result in the map view, right click and "Save horizon as a custom property" and add to the picked event.
   - Do this for near and far stacks.
3. Create a probe that encompasses the area of interest.
4. Create a new crossplot.

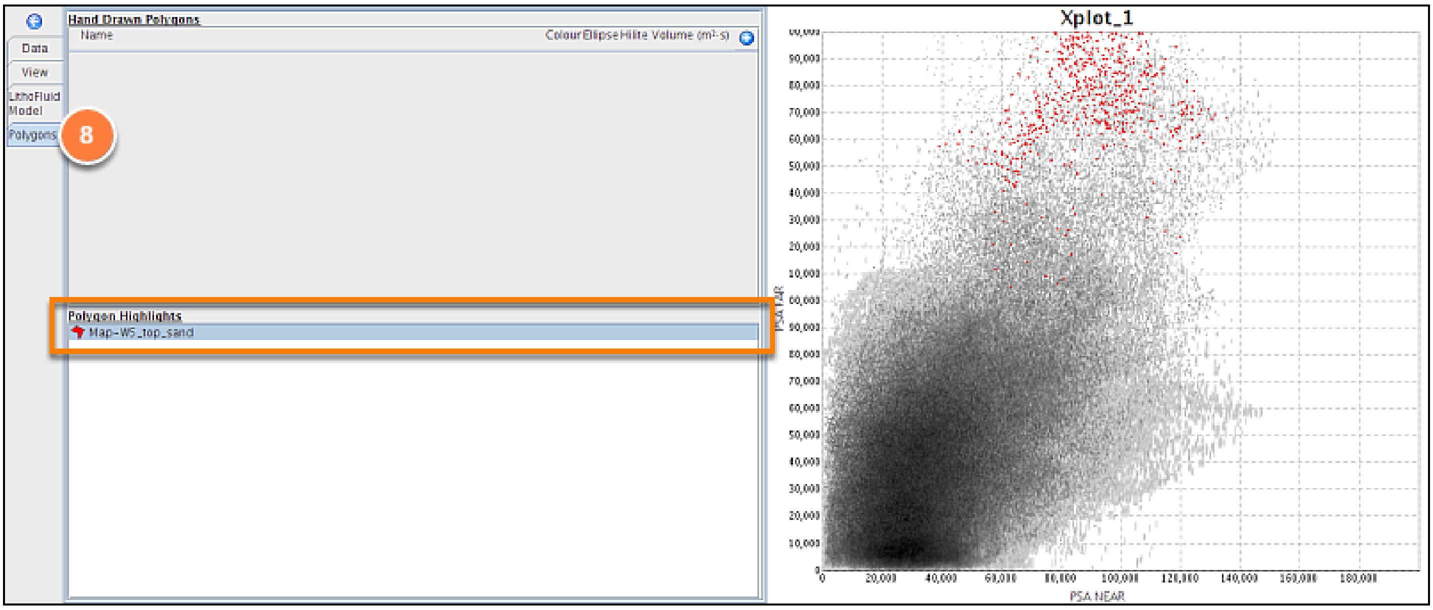


5. Choose the probe area of interest and configure:
   - **Data Type**: Horizon,
   - **Data Source**: your picked event,
   - **Property**: PSA (near or far)
6. In the **LithoFluid Model** tab:
   - Choose the reflectivity (dupdf) model (must be loaded in the QI tab first), and select the near vs far attribute.
   - Select the appropriate depth for the event from the model depths.

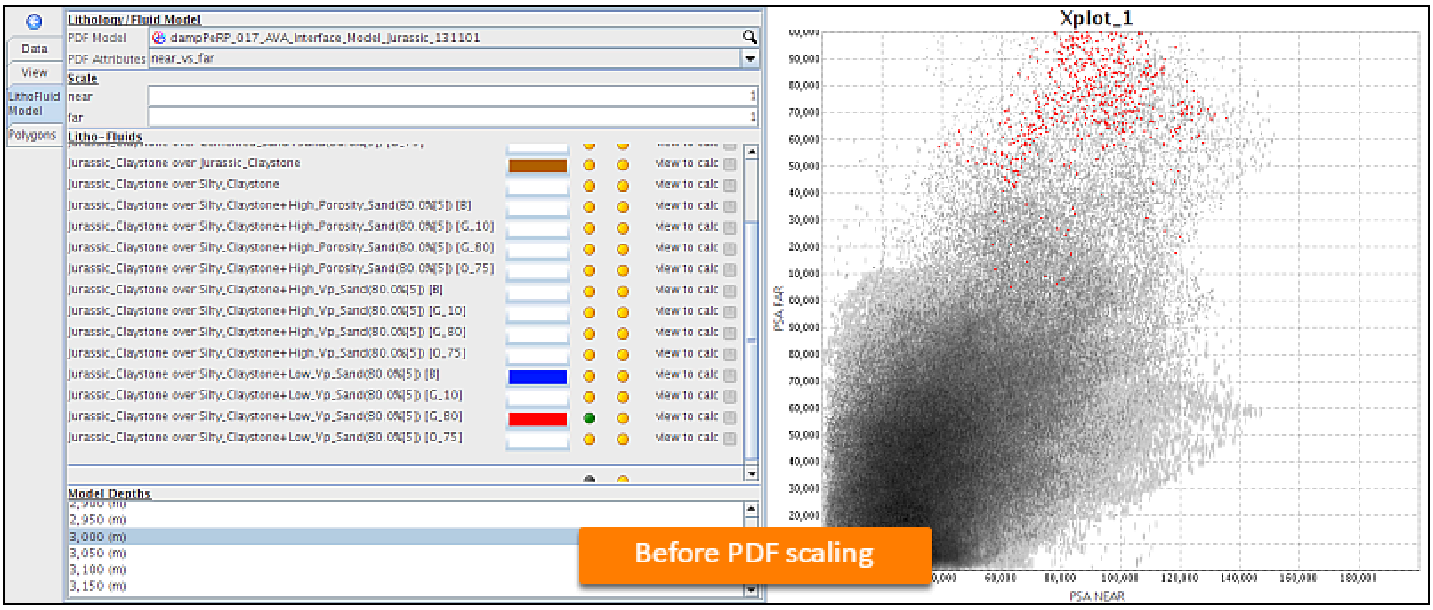
The next step is finding the correct scaling. This is a visual calibration approach, rather than an analytical approach. The robustness of this method depends strongly on the well control.

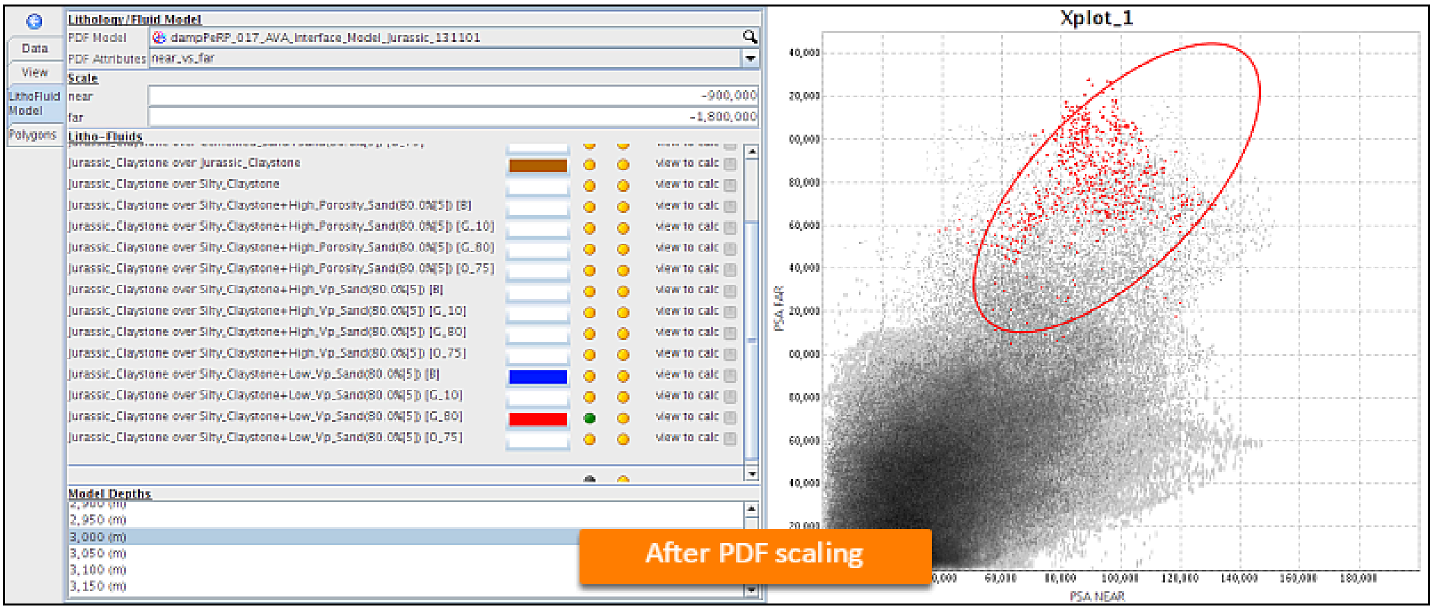


7. On the map, draw a polygon around one of the wells that intersects the event.
   - Name it appropriately. In this case, it is a top reservoir gas event.

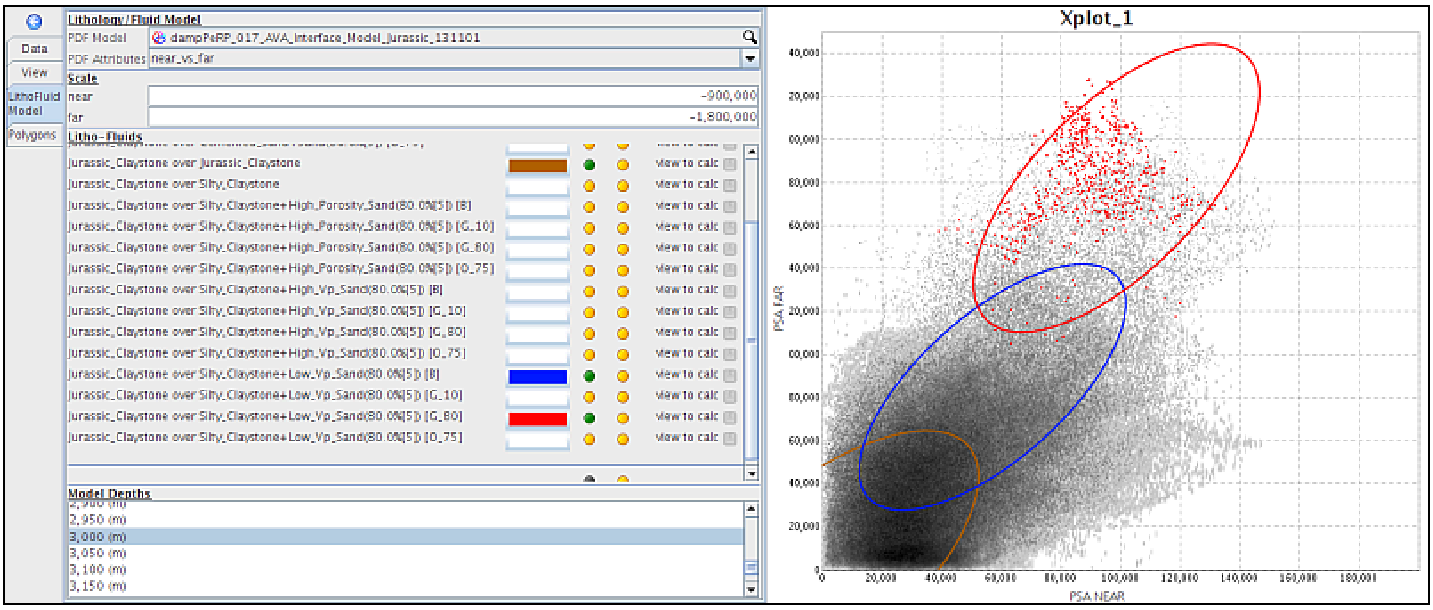


8. In the crossplot, highlight these points
   - In the **Polygons** tab, click the polygon and view the points on the crossplot.
9. You now have a single calibration area for a known lithology interface at the well.
   - Repeat this a few times at other wells, ideally highlighting different lithologies.
10. In the **Lithofluid** model tab, enable the corresponding PDF for the lithology at the well.
11. Adjust the **Scale** values until the PDF ellipse lands on the appropriate crossplot points.

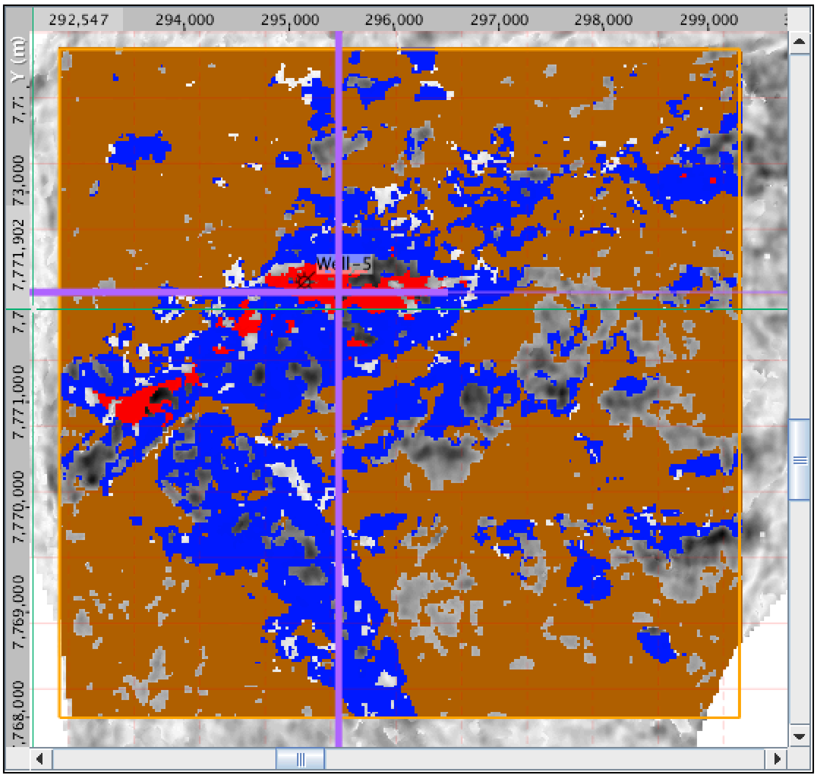




12. Enable additional lithologies and confirm the scaling makes sense.



13. Enable highlighting to review how the scaled ellipses correspond to locations in the map.



How can I created a layered result from many different input volumes merged at horizons?

Use volume maths with a series of nested “if” statements.

Any number of volumes can be merged this way, but it can be difficult to spot problems when using more than five. It is easier to work in separate sets (i.e. shallow set, middle set, and deeper set), then combine the shallow/middle/deep in a final process.

e.g. To merge 14 volumes (v1… v14) at horizons (h1… h14)

- Merge volumes 1-5 in one process > “MergeShallow”
- Merge volumes 6-10 in one process > “MergeMiddle”
- Merge volumes 11-14 in one process > “MergeDeep”
- Review each set of layers before combining.
- Combine “MergeShallow”, “MergeMiddle”, “MergeDeep”

Merge 5 volumes at horizons e.g. "MergeShallow”

- Given horizons:
  - h1, h2, h3, h4, h5
- and volumes:
  - v1, v2, v3, v4, v5

```
if(twt_ms <h1, v1,
        if(twt_ms<h2, v2,
                if(twt_ms<h3, v3,
                        if(twt_ms<h4, v4, v5)
                )
        )
)
```

Repeat for the other sets. Merge these combined models using a similar approach.

Given horizons (from above):

- h5, h10

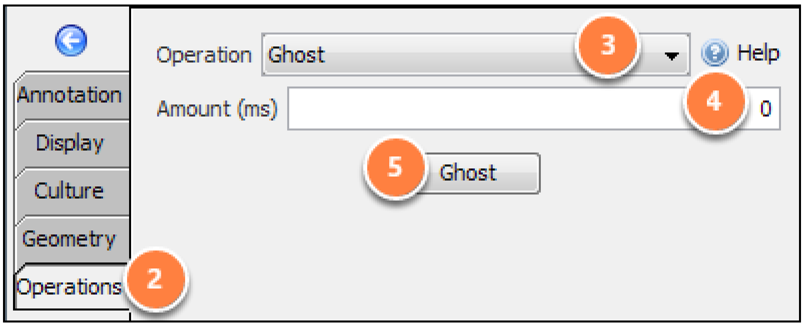
and 3 combined volumes "MergeShallow", "MergeMiddle", "MergeDeep":

- v1, v2, v3

```
if(twt_ms<h5, v1,
        if(twt_ms<h10, v2, v3)
)
```

How to interpolate RMS values on a 2D horizon?

Ghost the 2D horizon

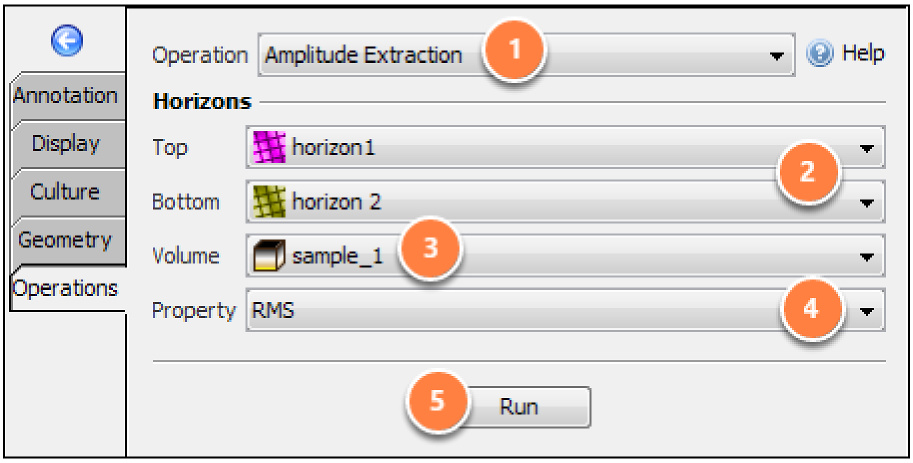


1. In the **Map View**, select to display the 2D horizon which you want to create the ghost horizon.
2. Expand the left panel and open the **Operations** tab.
3. Select **Ghost** at Operation.
4. Ghost the 2D horizon to the top and bottom of the interval that you want RMS over. At **Amount (ms)**, a negative value will place the ghost horizon above the original horizon and a positive value below.
5. Click **Ghost**.

Note: The **Ghost** button will be disabled if the horizon you are using in the operation is pending on another operation such as propagation. You will see a notification of the pending operation that you should complete before continuing with the current operation.

6. When the ghost horizon has been generated, click **Save as New Horizon** to keep the horizon.

Amplitude Extraction to get RMS



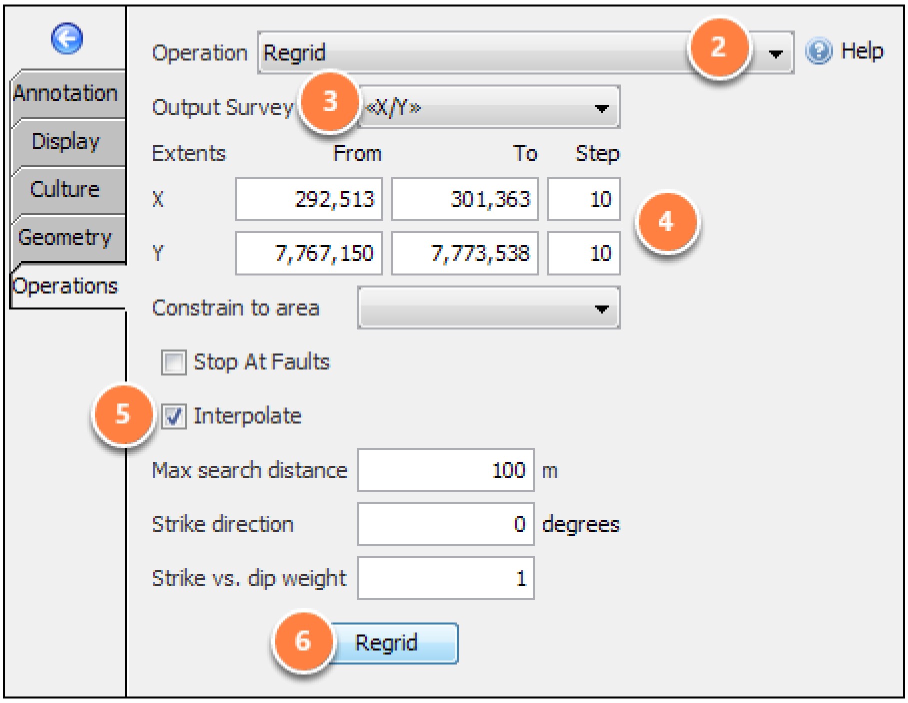
1. From the **Operations** tab in the map view, select **Amplitude Extraction** at Operation.
2. At **Top** and **Bottom**, select the top and bottom horizons (ghosts) between which Insight will extract the amplitude.
3. Select the input volume for the operation at **Volume**.
4. At **Property**, select **RMS** as the extraction method to be used in the operation.
5. Click **Run** to run the operation.

Note: The **Run** button will be disabled if any of the input horizon you are using in the operation is pending on another operation such as propagation. You will see a notification of the pending operation that you should complete before continuing with the current operation.

6. When the operation is completed, select **Create new** to save as a new horizon.

Tip: Click on the **Help** icon to read more information about this operation.

Regrid the extracted RMS



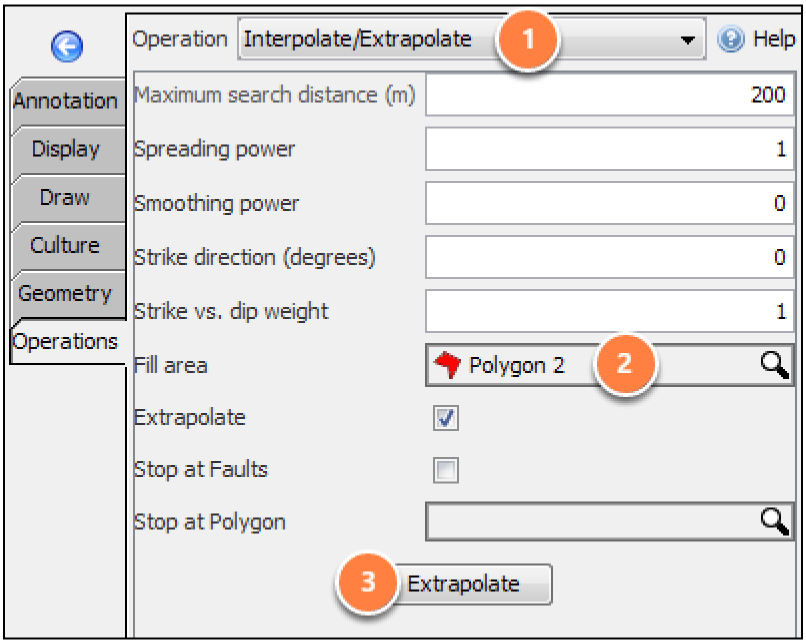
1. In the **Map View**, select the extracted RMS as the Horizon.
2. From the **Operations** tab, select **Regrid** at Operation.
3. At Output Survey, select **<<X/Y>>**.
4. Regrid the extracted RMS with a small enough increment to honour the CMP bins.
5. Clear the **Interpolate** check box, or use a small search distance.
6. Click **Regrid**.

Note: The **Regrid** button will be disabled if the horizon you are using in the operation is pending on another operation such as propagation. You will see a notification of the pending operation that you should complete before continuing with the current operation.

7. Once the horizon has been processed, select **Save as New Horizon**.

Note: After regridding, the interpretation may appear very small. Zoom in to confirm regridding occurred.

Interpolate/extrapolate



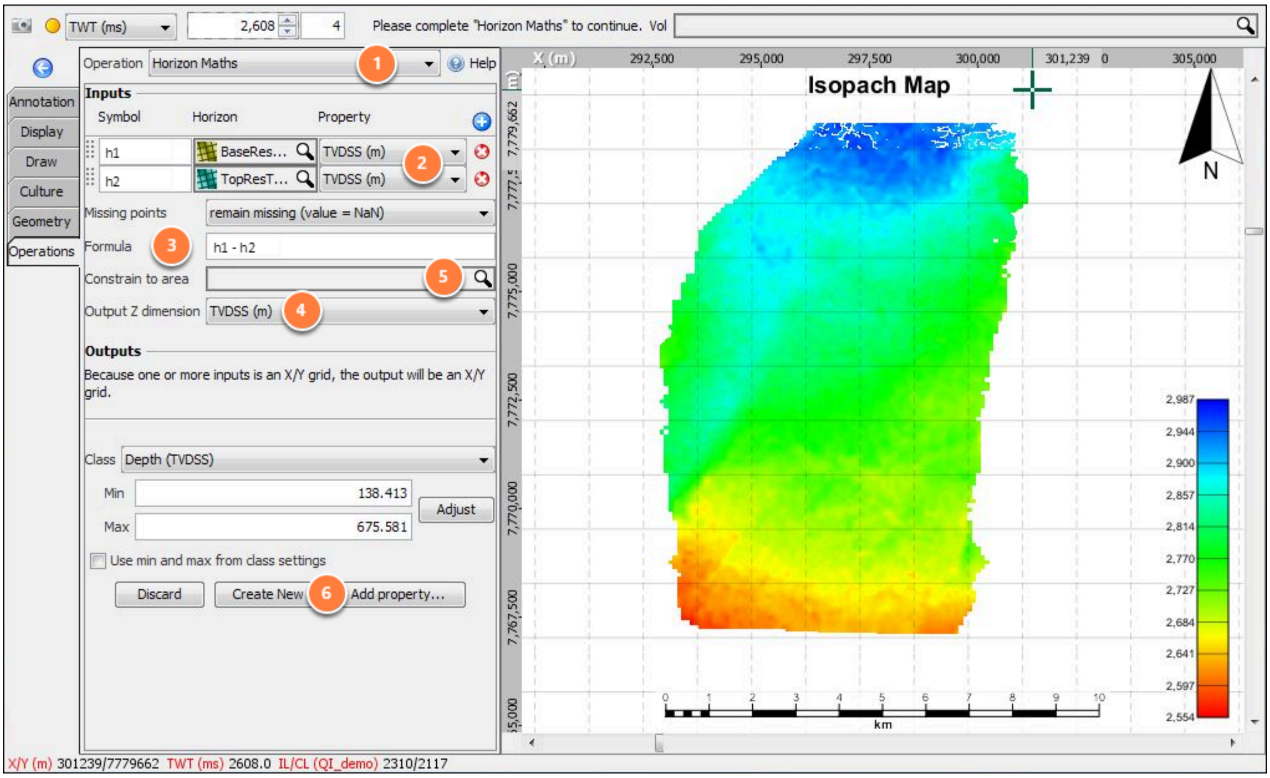
1. From the **Operations** tab in the map view, select **Interpolate/Extrapolate** at Operation.
2. Select the **Extrapolate** checkbox.
3. At **Fill area**, select a polygon to constrain the area to fill with the selected polygon.
4. Click **Extrapolate**.

> **Note:** The **Extrapolate** button will be disabled if the horizon you are using in the operation is pending on another operation such as propagation. You will see a notification of the pending operation that you should complete before continuing with the current operation.

5. Select **Replace this horizon** or **Save as New Horizon** to keep this horizon.
6. This should generate an interpolated RMS map.

How do I make an isochore or isochron map in Insight?

A simple horizon maths operation can be used to create an isochore or isochron map in Insight. An isochore map is the true vertical thickness between two horizons; an isochron map is the vertical time between two horizons. Both maps are created in a similar way. See *Isopachs, Isochores, and Isochrons* for more details and to create isopach (stratigraphic thickness) maps.



1. Open **Map View** and begin a **Horizon Maths operation** (see *Horizon Maths*).
2. At the Inputs section, input horizon 'h1' and 'h2'.
   a. For an **isochore map**, set the property as **TVDSS (m)**.
   b. For an **isochron map**, set the property as **TWT (ms)**.
3. At **Formula**, insert the formula: **h1 - h2**. Make sure you are subtracting the shallower horizon from the deeper one.
4. Select the output vertical dimension.
   a. For an **isochore** map, set the output property as **TVDSS (m)**.
   b. For an **isochron** map, set the output property as **TWT (ms)**.

> **Note:** If you do not want the newly created horizon to show in section view, set the vertical dimension to <none>. Without the vertical dimension, you will only be able to view it in *Map View*.

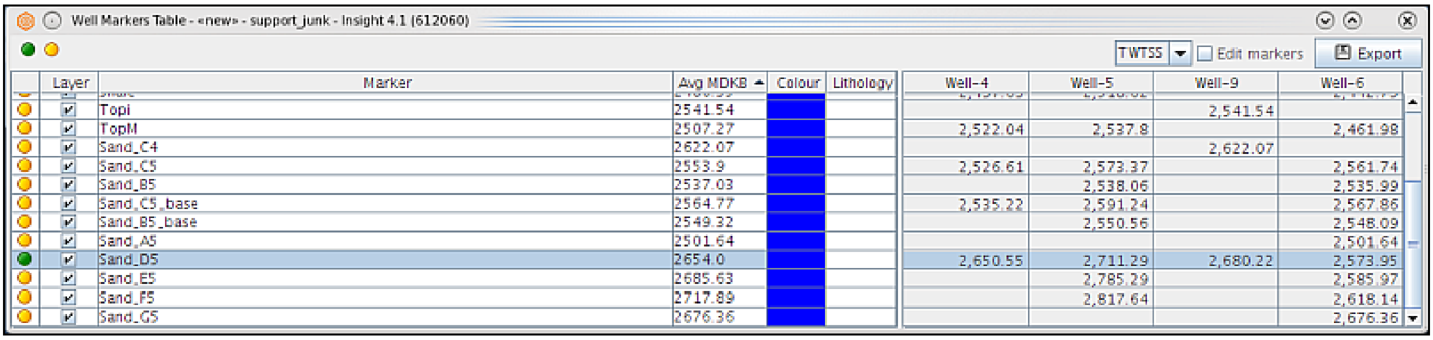
5. You can optionally constrain your isochore/isochron map by selecting a polygon for the '**Area**'.
6. The output is a horizon property of vertical thickness (isochore) or time (isochron) between the input horizons. You can save this as a *custom property* to an **existing horizon** or **create a new horizon**.

How can I update a horizon to tie markers at wells?

There are two approaches to this:

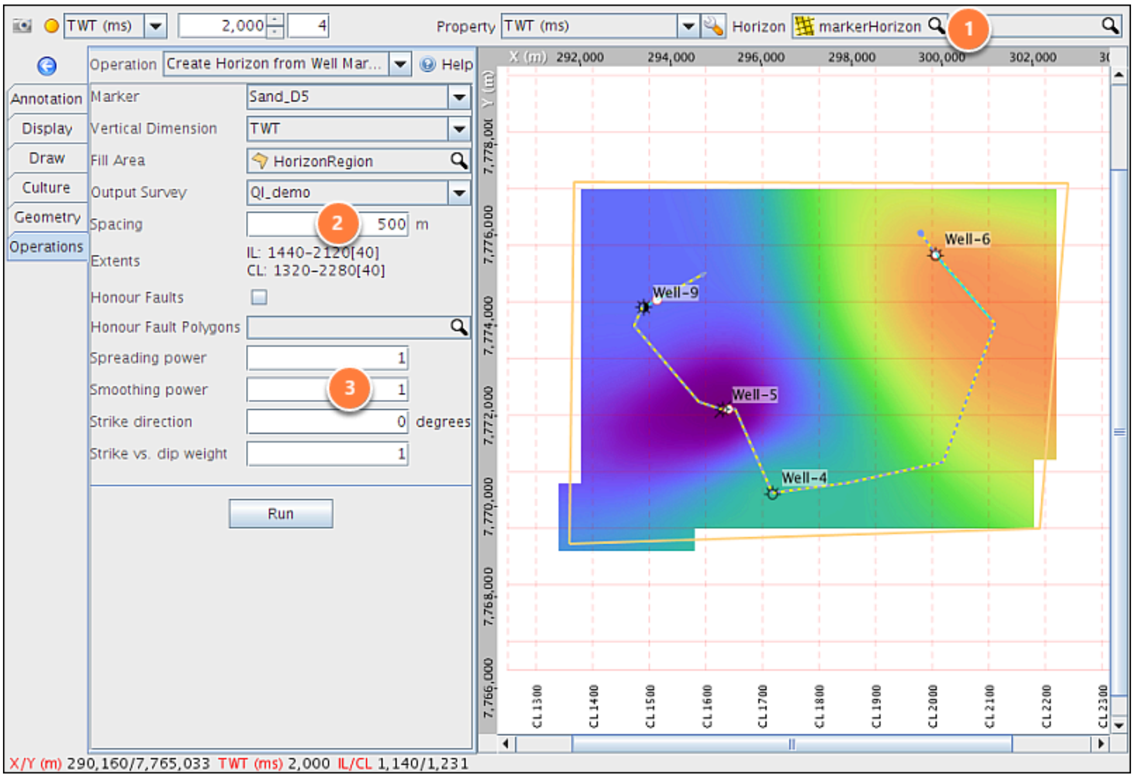
1. [Shift the entire horizon by some constant]()
   - Find an average shift to apply to the entire horizon
2. [Shift the horizon by a spatially varying value]()
   - Find a local shift to apply in regions around the wells

Constant Shift



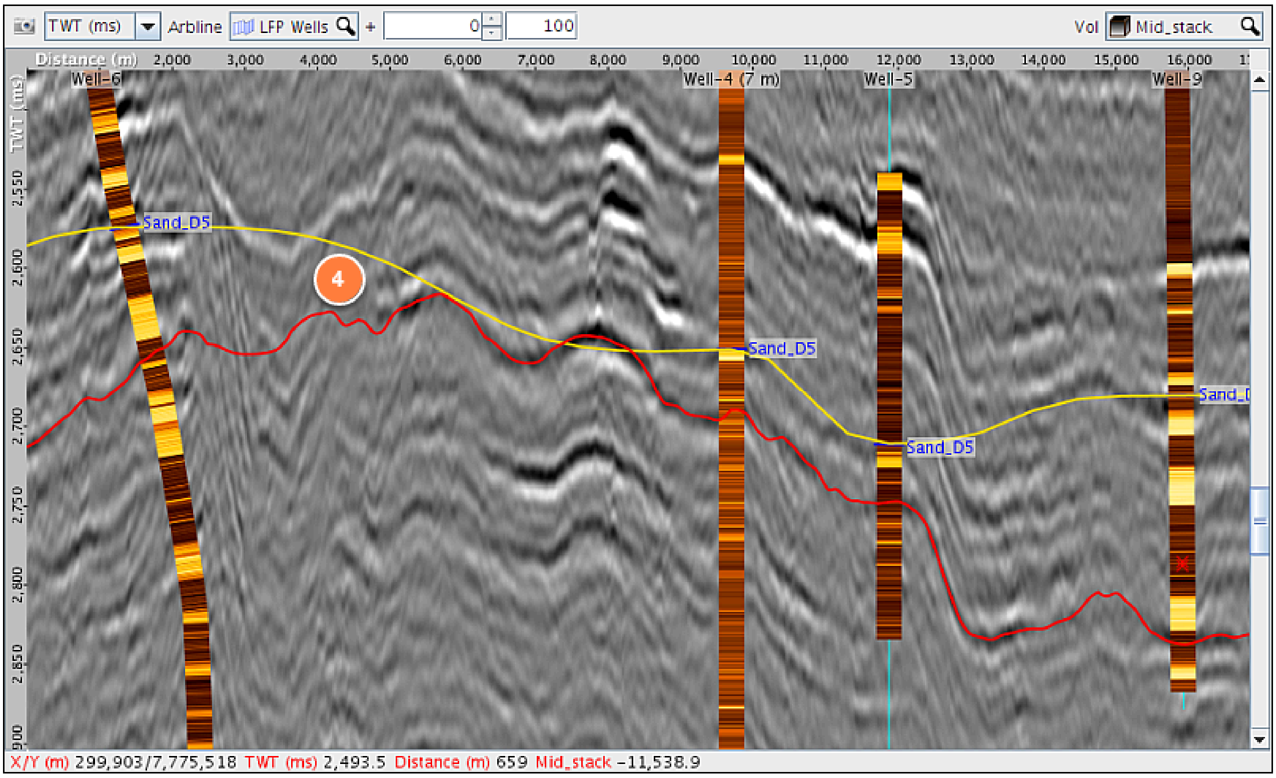
1. Calculate the difference between this value and the average horizon depth, then use the **Horizon Operation / Ghost** to create a shifted version. Read about the [Ghost (horizon)]() operation.

Spatially Varying Shift

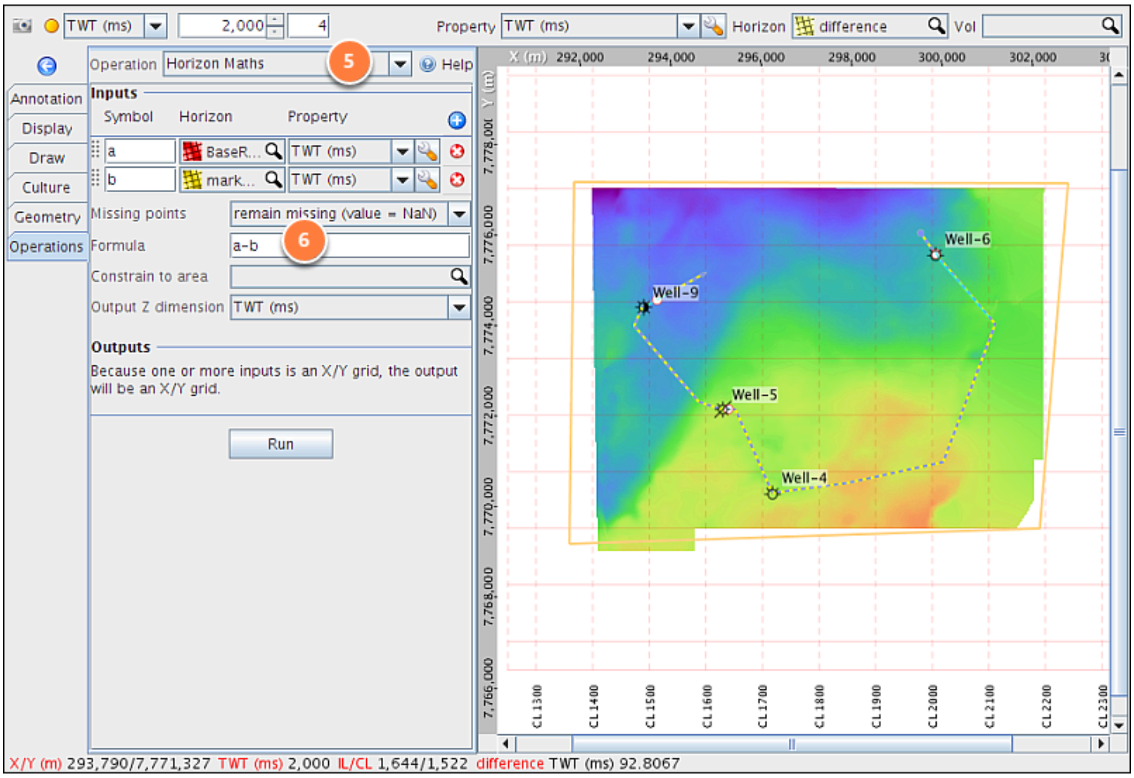


The other approach, which works nicely if the horizon is picked consistently, is to vary the shift spatially depending on the difference at each well. Use the following workflow:

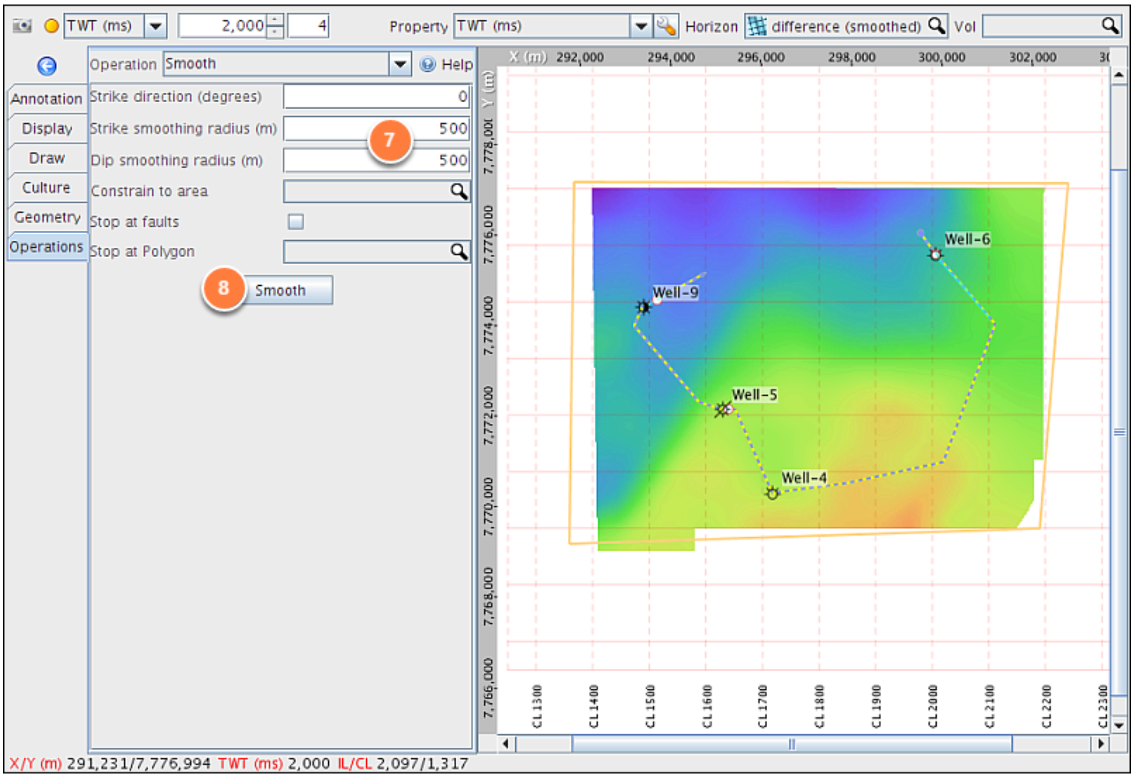
1. Create a horizon from the well markers. Give it a name e.g. "markerHorizon".
2. Use a **sparse incremen**t e.g. 500m.
3. Set the **smoothing power** to 1 or 2.



4. Comparing original horizon to marker horizon.

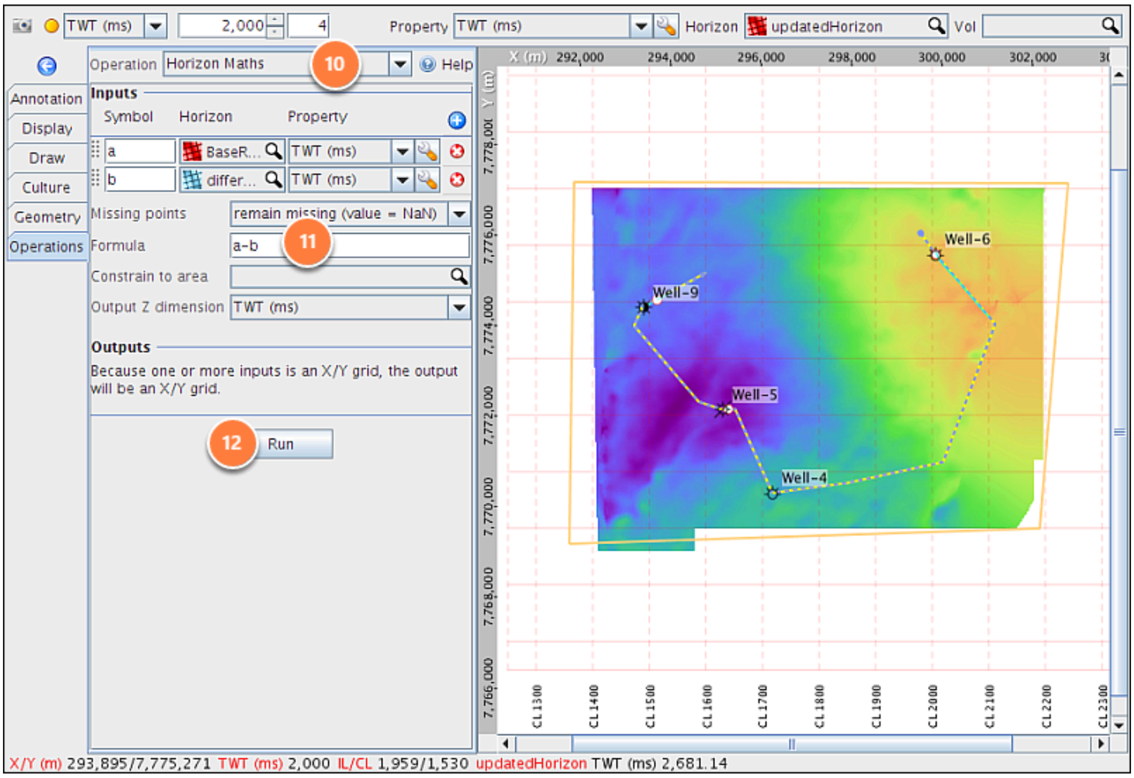


5. Calculate the difference between the horizon and the marker horizon "difference". Read more about Horizon Maths operation.
6. Use the **Horizon Maths** operation and use the formula: **a - b** (where **a** is your horizon and **b** is the markerHorizon).

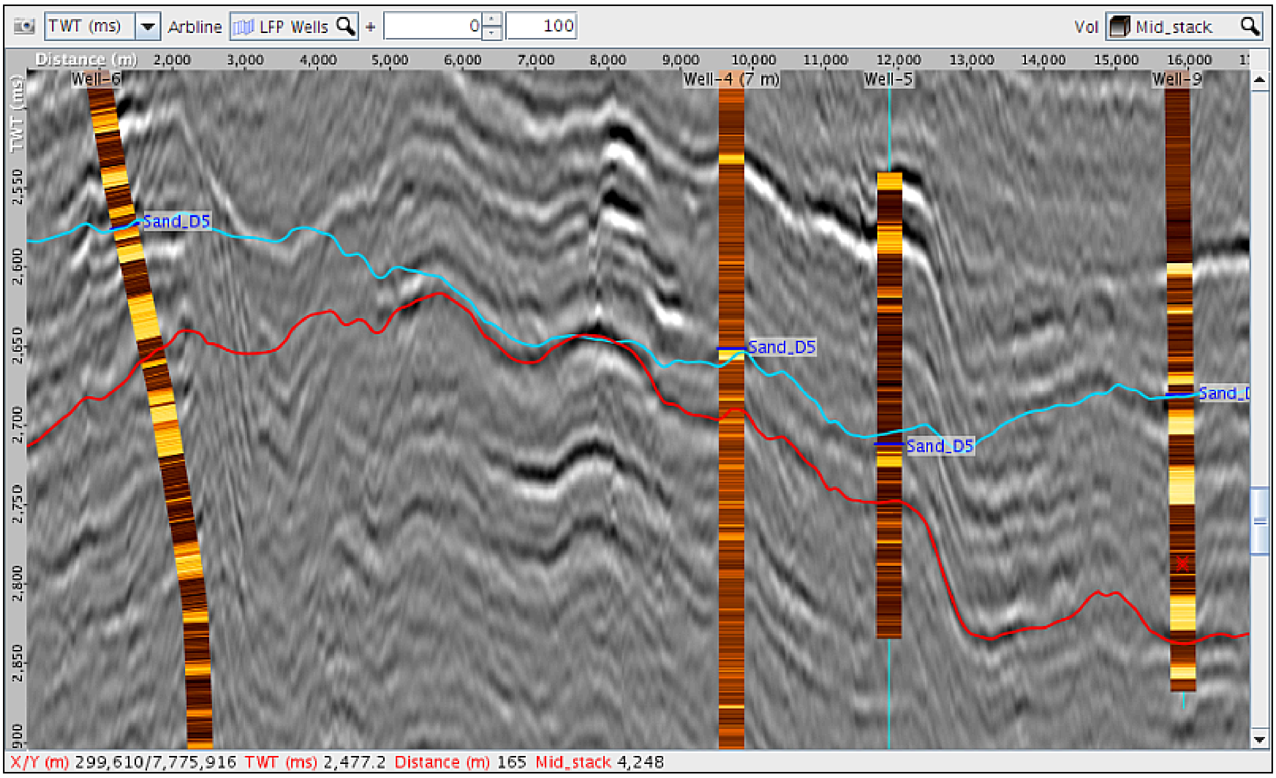


7. Use the Smooth operation on the "difference" horizon.
   - Use a **large radius** e.g. 500m or larger.
   - Larger smoothing values take longer, but preserve the character of the original horizon better.

8. Click the **Smooth** button and **save** the result e.g "smoothedDifference".



9. Add the smoothed result to the original horizon.
10. Select the **Horizon Maths** operation again.
11. Use the formula: **a - b** (with a being your horizon and b being the just created smoothedDifference horizon.)
12. Click **Run** and **save** the new horizon.



13. The resulting horizon should closely tie the markers at each of the wells. Note the is greater distortion of the original horizon when the correction values are large.

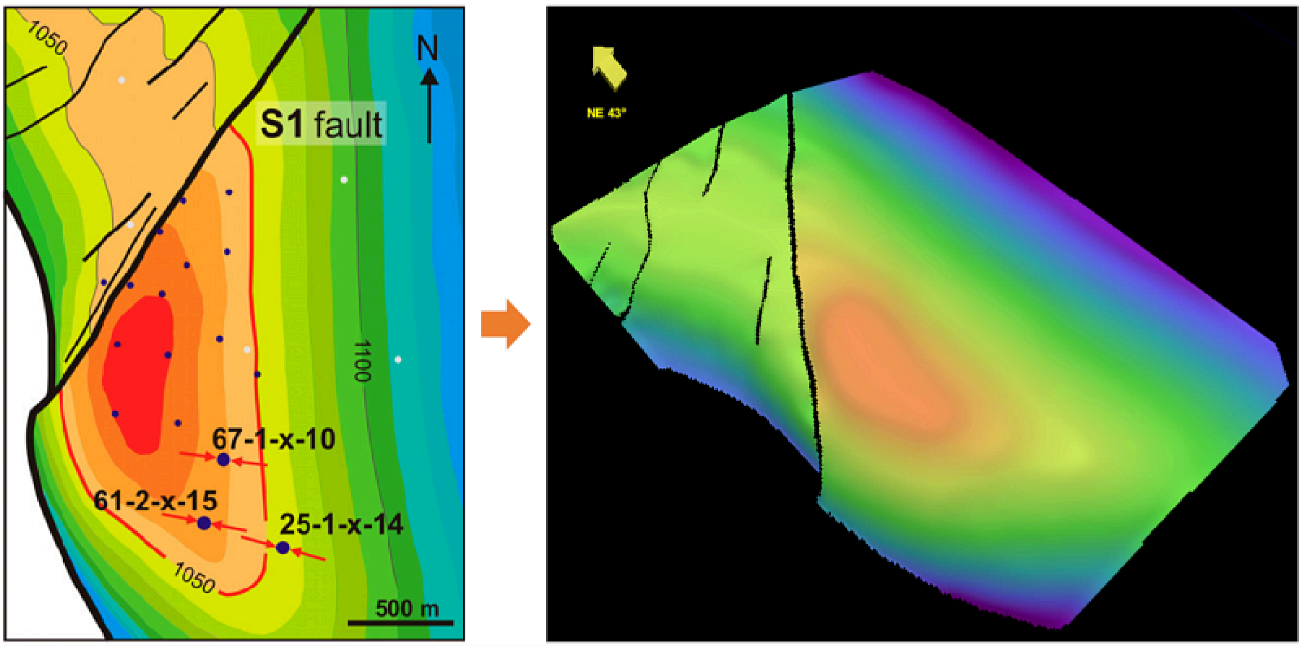
Transforming images into 3D horizons in Insight

Integrating all available data is crucial for proper subsurface evaluation. Sometimes available data may include structure contour maps taken from literature or other sources. This workflow demonstrates how to trace contours using faults sticks to create a file of XYZ points that can be imported into Insight as a horizon.

In this example, an image of Top Tensleep TWT structure from Chiaramonte et al. (2011) is transformed into a horizon in Insight.

- *Step 1: Map lines on contour map using fault sticks*
- *Step 2: Create polygons to constrain the horizon*
- *Step 3: Export fault sticks into a single file and modify*
- *Step 4: Import .dat as a horizon*
- *Step 5: Regrid and interpolate*
- *Step 6: QC and smooth*

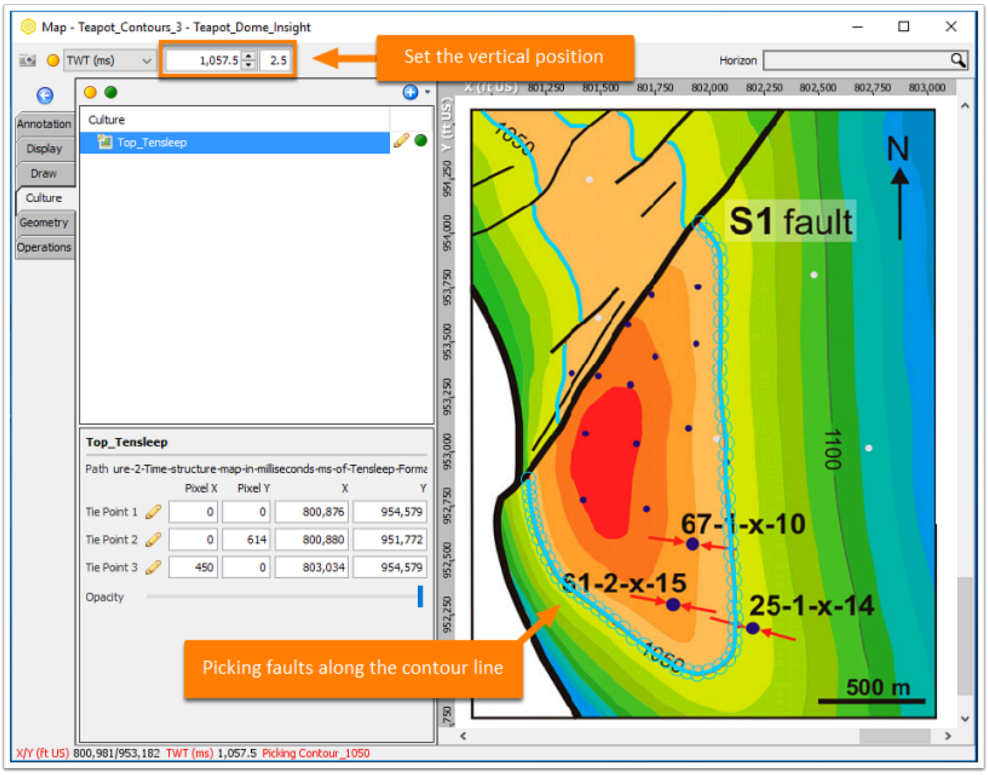
Workflow



Chiaramonte, L., Zoback, M., Friedmann, J., Stamp, V., and Zahm, C., 2011, Fracture characterization and fluid flow simulation with geomechanical constraints for a CO2EOR and sequestration project Teapot Dome oil field, Wyoming, USA: Energy Procedia, v. 4, p. 39733980.

Step 1: Map lines on contour map using fault sticks

1. Import the image via **Import > Culture** (see *Importing Culture Files*).

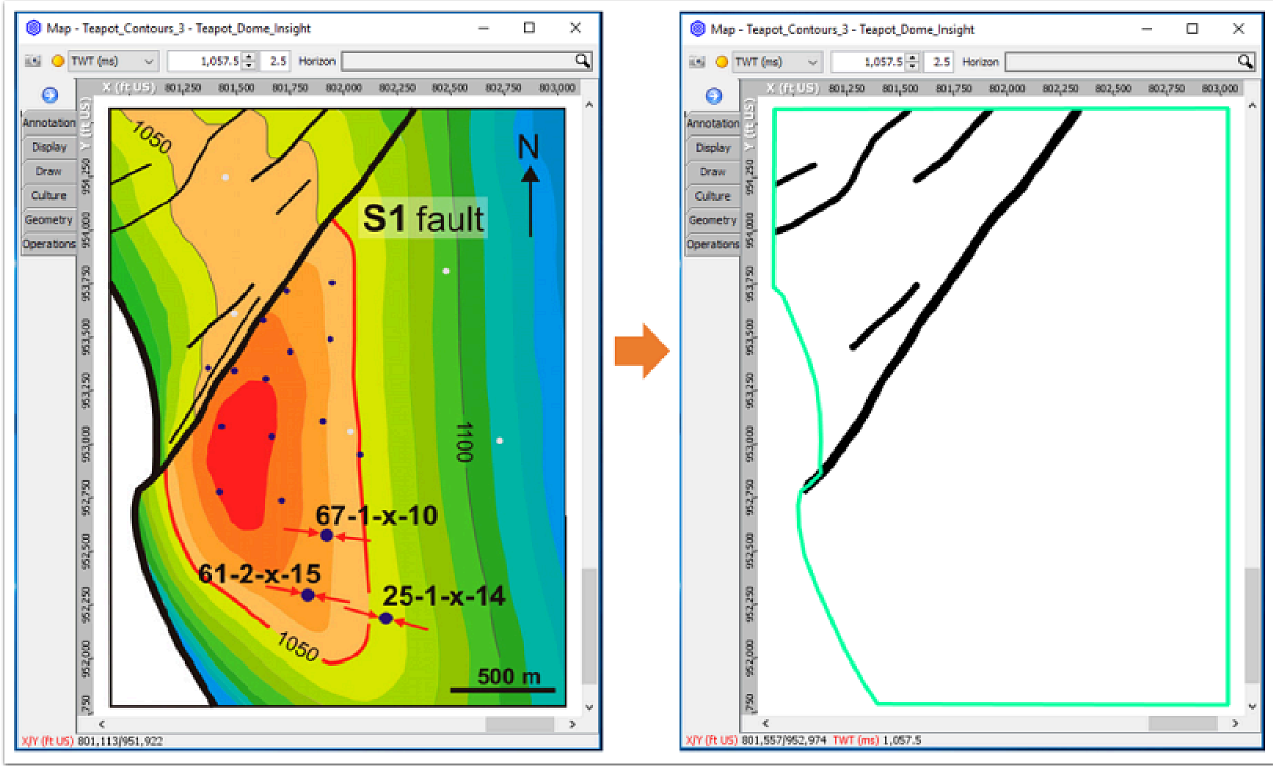


2. Create a new fault for each contour line (see *Picking Faults*).
   - To set the vertical position that the fault stick is mapped on, adjust the time or depth value in the top left corner of the map view

Important: Pick faults at a high resolution! The goal is to create a horizon based on an input file containing XYZ points. Each individual point will be used to create a horizon once the file is imported back into Insight as a horizon. In this example, a point is added about every 50 m along a contour line.

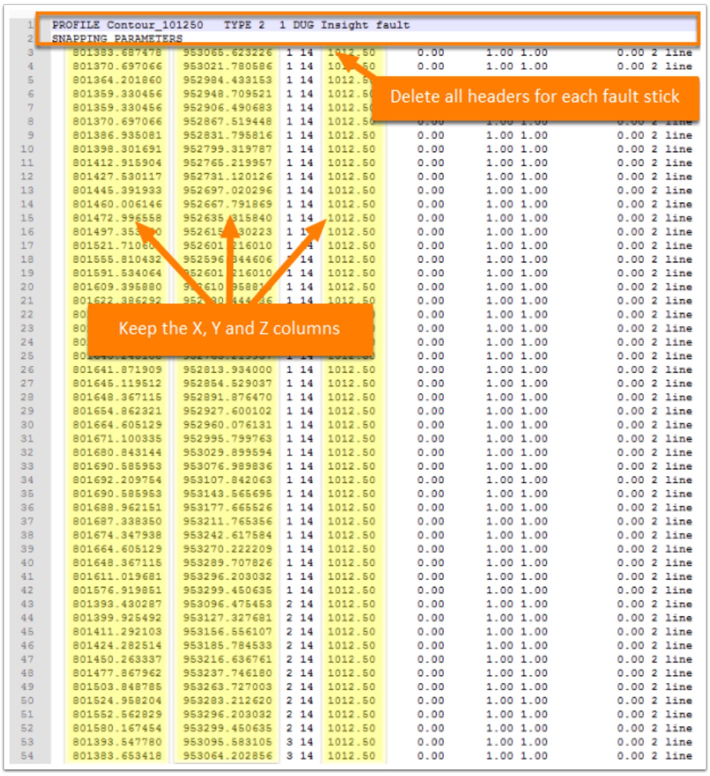
Note: The flat appearing areas above contour 1010 and 1050 in the example likely have some curvature that is below the resolution of the contours. To make the horizon more geologically realistic, add a fault stick at the crest with a TWT value that is less than the contour value just below. This will add points higher in the structure so the horizon does not appear totally flat in these areas.

Step 2: Create polygons to constrain the horizon



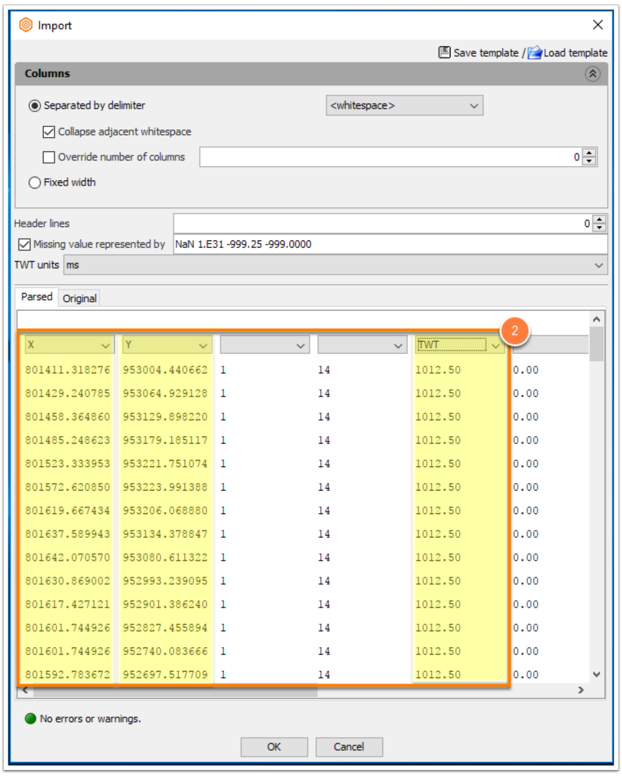
3. With the culture file displayed, add polygons around the horizon boundaries and faults.
   - This will constrain the area that is interpolated when using the regrid operation after importing the file as a horizon.

Step 3: Export fault sticks into a single file and modify



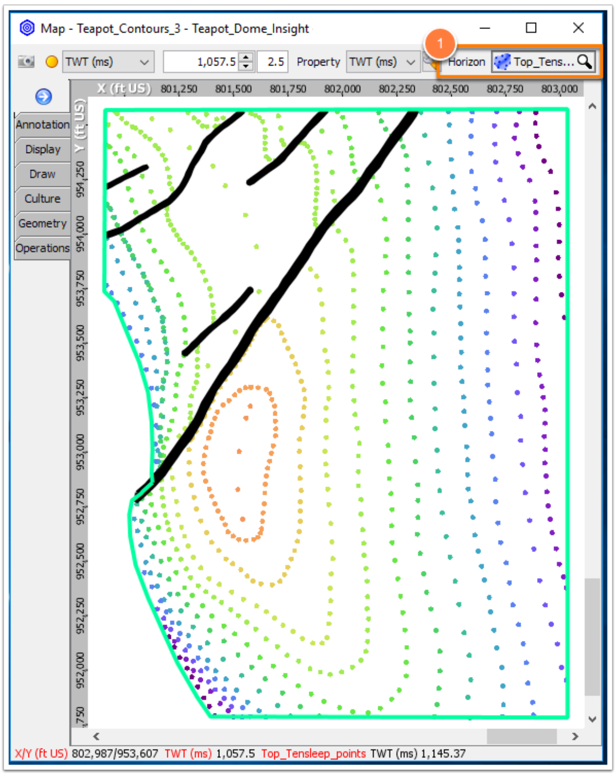
1. Highlight each fault sticks of the mapped contour in the **Fault** panel.
2. **Right-click > Export (**see *Exporting Faults*).
3. Open the **.dat** file in a text editor (e.g. Notepad)
   - The exported file will contain information that isn't needed when reimporting the file as a horizon.
   - Delete all headers for each fault stick
   - Once the headers are deleted, save the file.

Step 4: Import .dat as a horizon

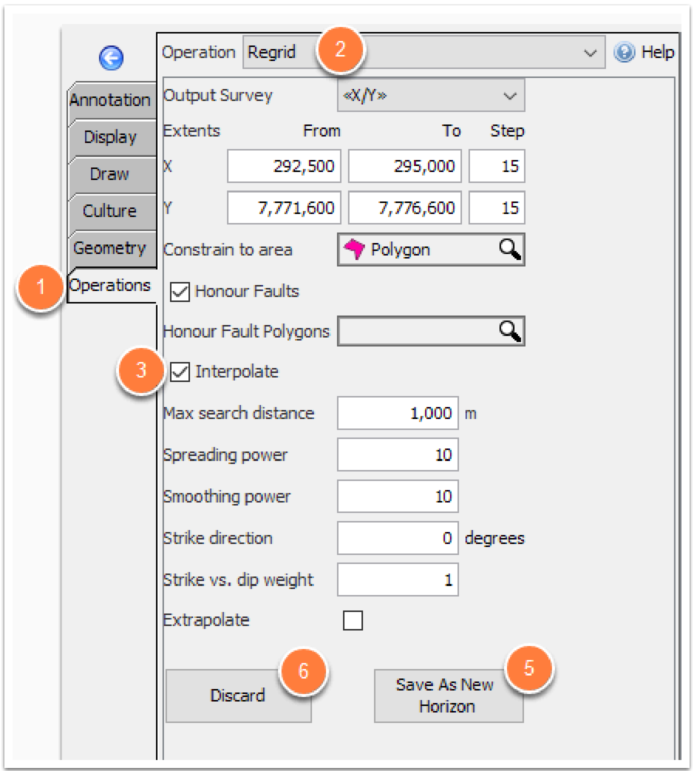


1. Import modified file as a horizon via **Import > Horizons**. (see *Importing Horizons*).
2. Assign the X, Y and Z values to the correct column.
3. Click **OK** to continue importing.

Step 5: Regrid and interpolate



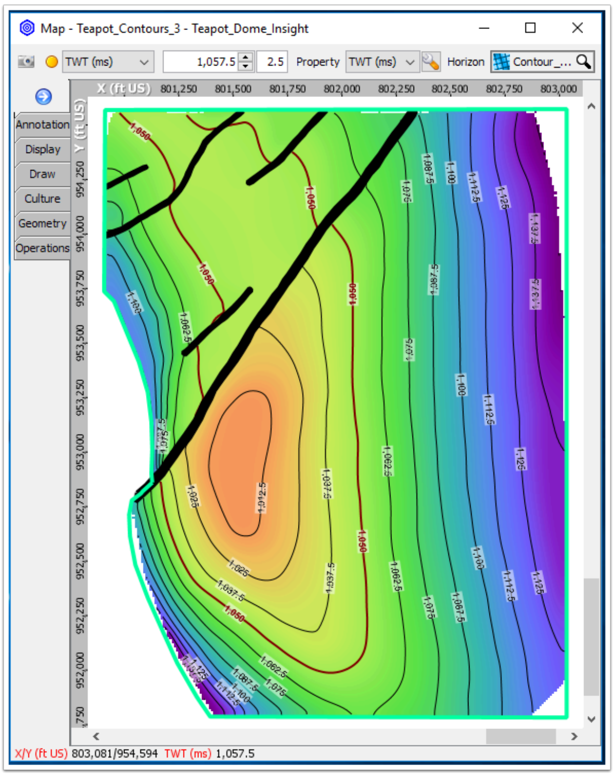
1. Go to **Map View** and select the horizon.
2. The horizon will display as points on an irregular grid.



To create an interpolated horizon, you'll need to regrid the horizon and interpolate.

3. In the **Map View**, click the **Operations** tab and select the **Regrid** operation to put horizon onto a regular grid.
4. Tick the **Interpolate** box and adjust interpolate parameters accordingly. Also consider to:
    - adjust the steps,
    - use polygon to constrain area,
    - and fault polygons to stop interpolation at faults
5. View the output and click **Save As a New horizon** to save the result.
6. Click **Discard** if you would like to make adjustments to the regrid parameter.

Step 6: QC and smooth



1. Once you have regridded and interpolated the horizon, QC the result in **Map View**.
2. Turn on contours for easy comparison to the original contour map that the horizon is based on (see *Contouring in Map View*).
3. Use **Operations > Smooth** as necessary to create a smoother image (see *Horizon Smoothing*).

Some useful tips:

- In areas that appear flat, may want to add control points slightly lower (if a trough) or higher (if a dome) so that horizon isn’t totally flat. This should make for a smoother, more realistic horizon and may help avoid odd contours lines.
- Only draw polygons in areas with faults that have visible offset. Areas in my example that had faults with no apparent offset caused gridding problems
- Add control points (using fault sticks) in areas with large gaps to avoid the gaps from appearing when interpolating.
- Use small tile size when smoothing, then smooth over multiple iterations.

Calculating Angle of Incidence

Angle of incidence can be useful in analysis and in calculations. While angle-mutes, raytraced using a velocity model, are the most accurate means of displaying the intercept angle, they can be awkward to use analytically. The two methods shown here calculate an angle of incidence for use in other calculations.

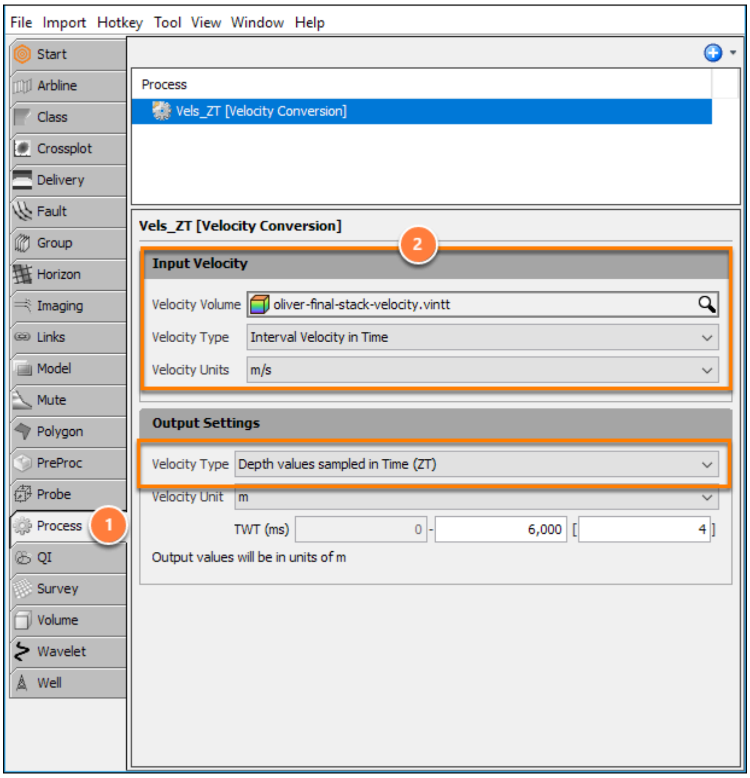
- *Approximate method (Straight-ray)*
- *Accurate method (NMO deviation)*

Approximate Method

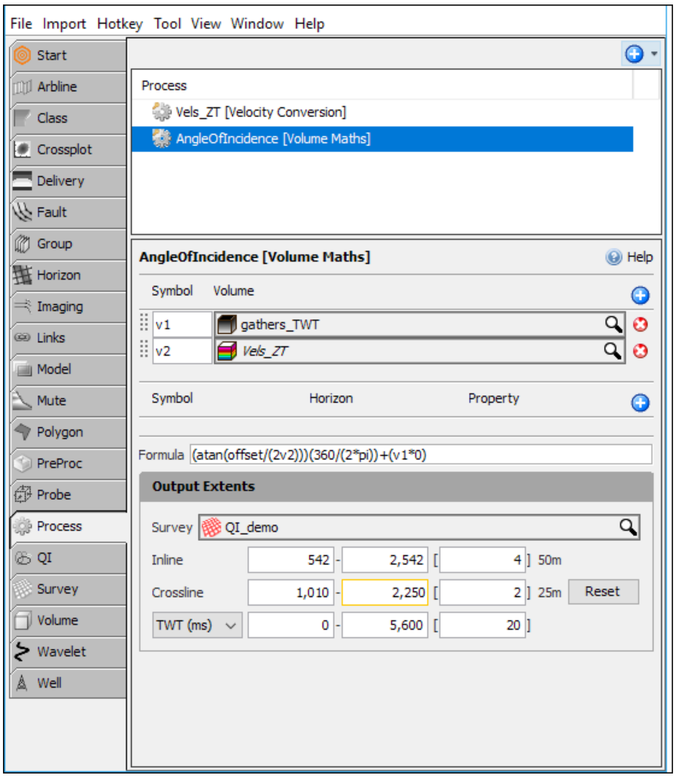
This method finds the angle of incidence assuming a straight-ray cast from the offset to the calculated point. This flow creates gathers of incidence angle, from which you can extract amplitudes from as you would any other volume.

It requires:

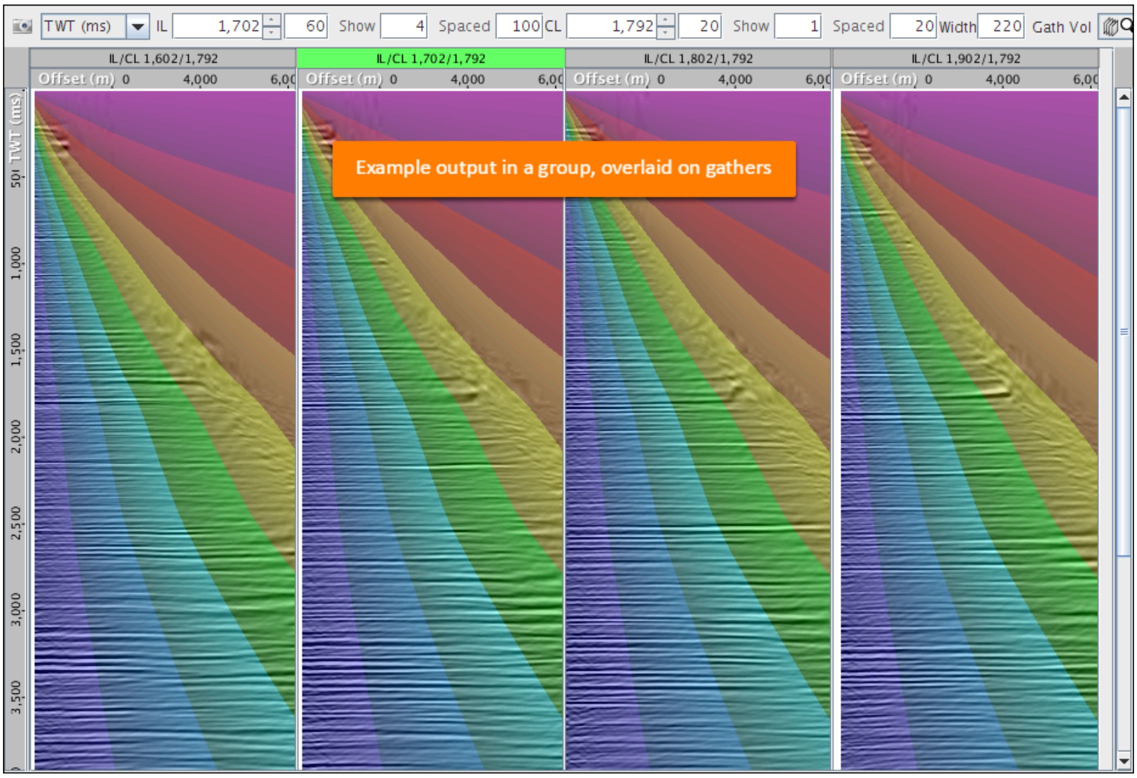
- Velocity Model
- Gathers (to define the offset extents)



1. To Calculate depth values sampled in time from the velocities, click **Process tab >> add New Process >> Velocity Conversion**.
2. Add a name for this Velocity Conversion process and set the values below:
   - **Input**: Velocity Model
   - **Output type**: Depth values sampled in time (ZT)



1. Calculate the angle of incidence at each offset. Go to **Process tab >> add New Process >> Volume Maths**
2. Add a name for this Volume Maths process and set the values below:
   - **Input**: Gathers [v1]
   - **Input**: Velocity-DepthInTime ("Vels_ZT") [v2]
   - **Formula**: (atan (offset / (2 * v2))) (360 / (2 * pi)) + (v1 * 0)
3. Set the class of the "**AngleOfIncidence**" volume to a new class "**Angle**" and choose a useful colourbar (e.g. Banded-Decile-Light) with the range 0 to 100. Read *[Classes and Colourbars Overview](#)*.



Accurate Method

This method, derived from the NMO equation, finds a more accurate angle of incidence.

Given:

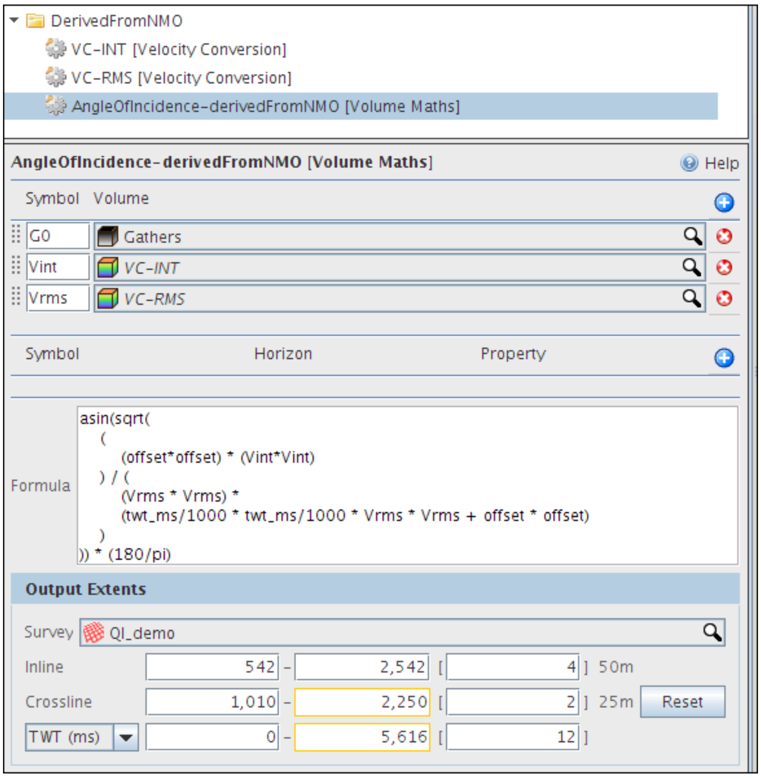
- **G0** : A volume of gathers, to define the offset extents (metres)
- **Vint** : The velocity model, converted to interval velocity (m/s)
  - Use a Velocity Conversion process if necessary.
- **Vrms** : The velocity model, converted to RMS velocity (m/s)
  - Use a Velocity Conversion process if necessary.

Create a volume maths process, using the above volumes as inputs:

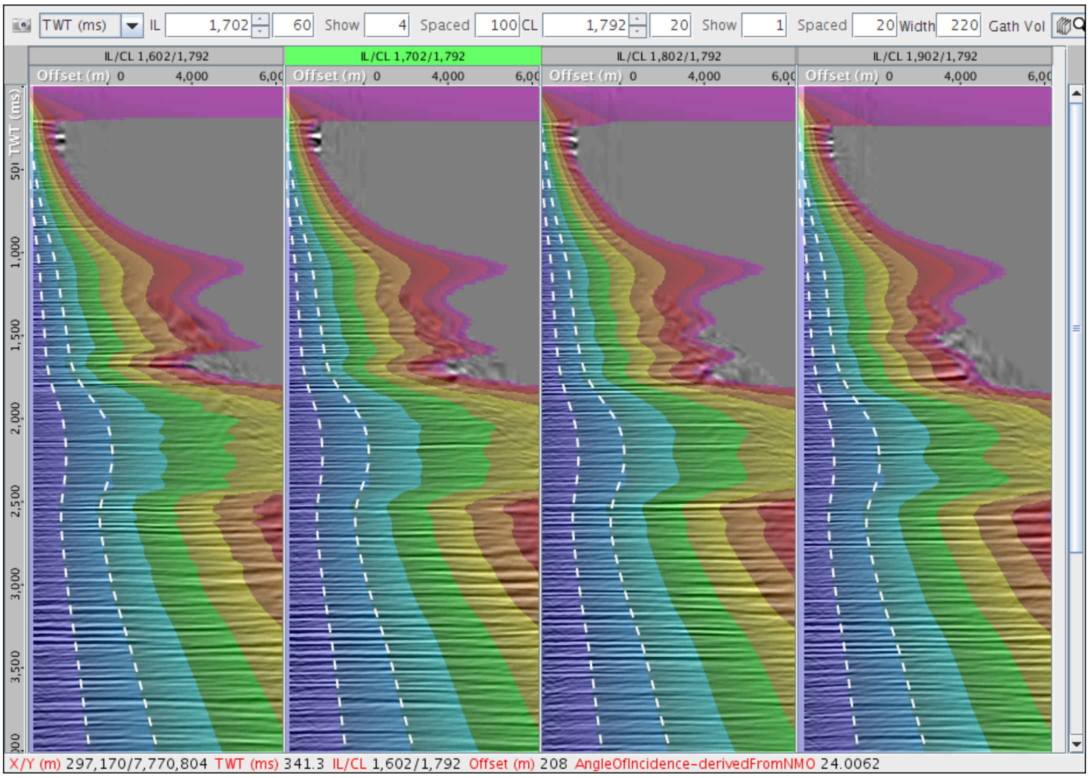
- **Formula**:

```
asin(sqrt(
    (
        (offset*offset) * (Vint*Vint)
    ) / (
        (Vrms * Vrms) *
        (twt_ms/1000 * twt_ms/1000 * Vrms * Vrms + offset * offset)
    )
)) * (180/pi)
```

The process looks like:



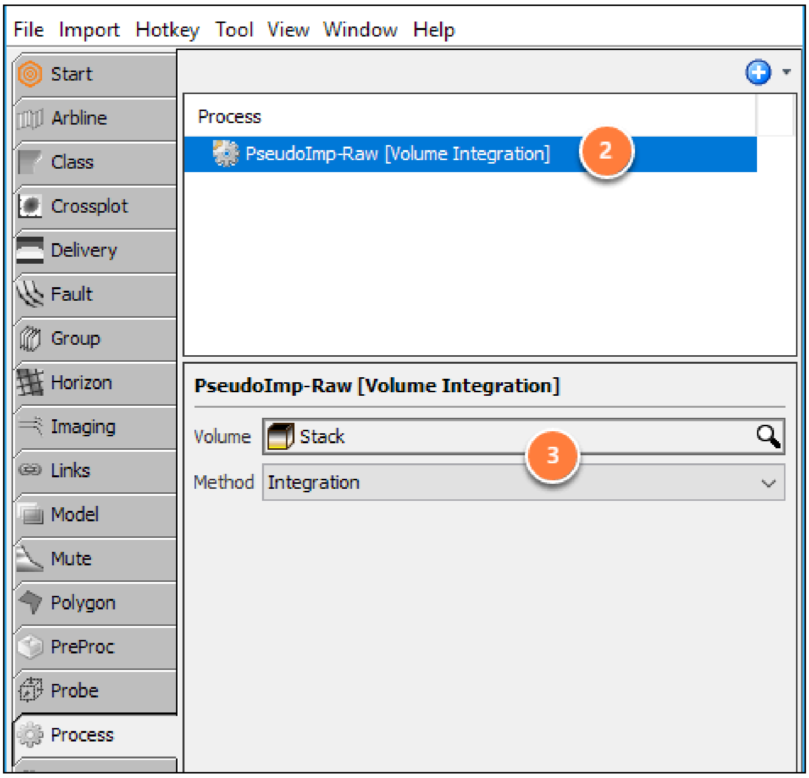
And the result:



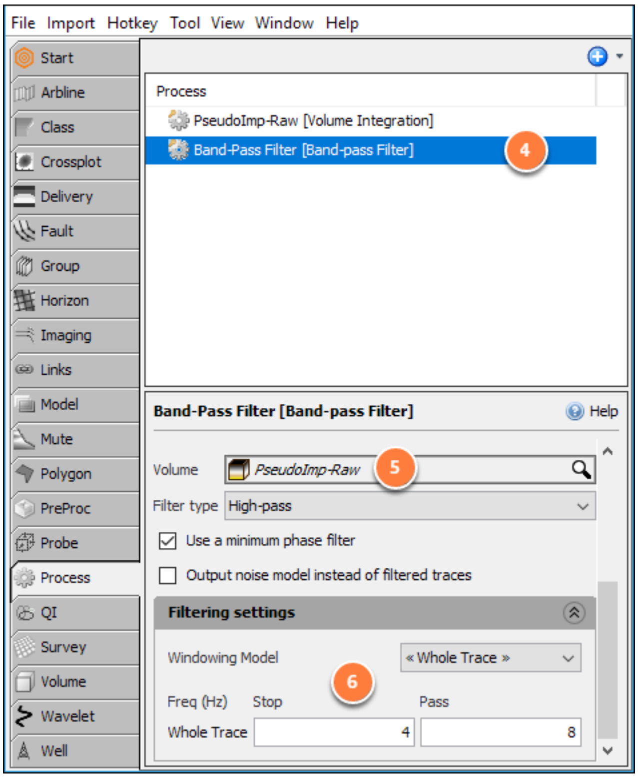
In this example, two mutes at 10 and 20 degrees are displayed over a group containing the angle of incidence (accurate method, in colour) over offset gathers (in grey). the mutes correspond to changes in angle using a banded colour bar and appropriate ranges.

How can I calculate pseudo-impedance from seismic data?

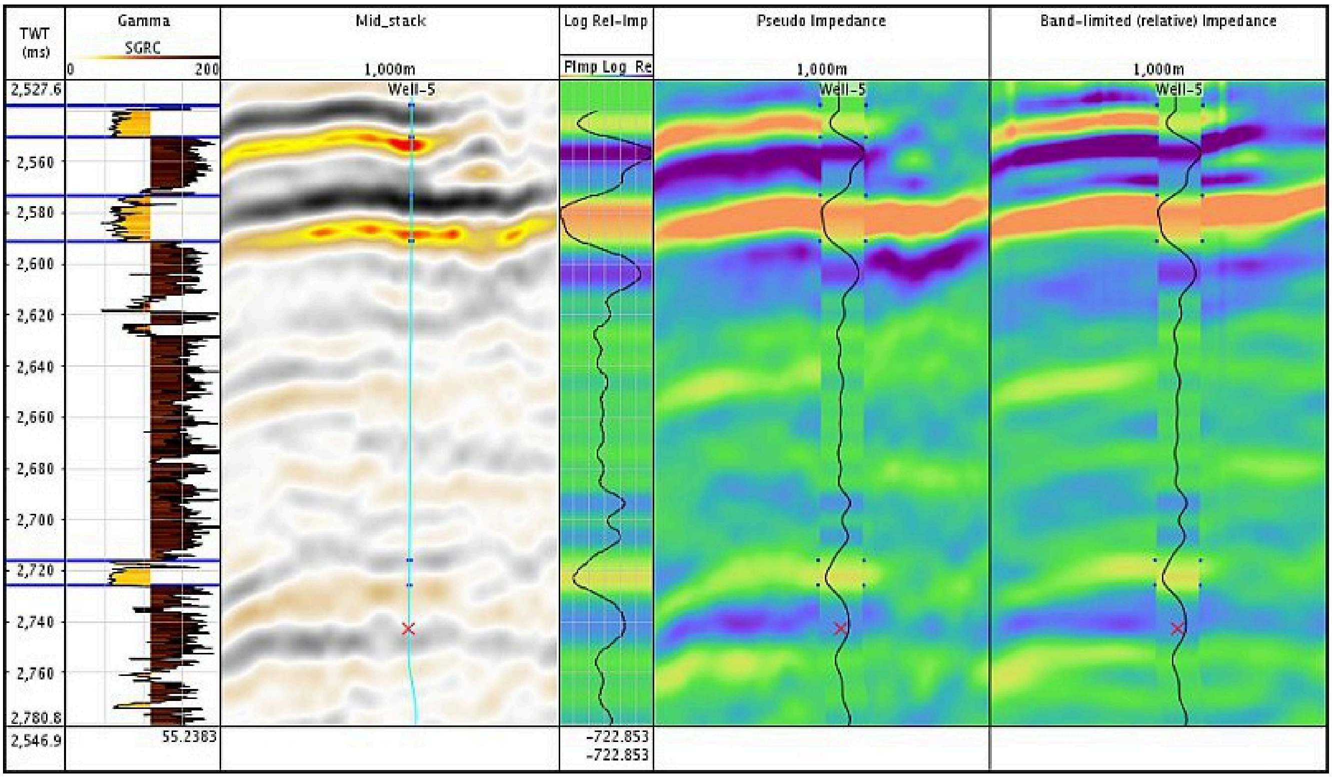
You can calculate pseudo-Impedance which can be used as a quick-look approximation of actual impedance.



1. Load a seismic stack from the **Volume** tab e.g. "Stack".
2. Calculate pseudo-impendance by using the **Volume Integration** process. Go to the **Process** tab and add a new **Volume Integration** process e.g. "PseudoImp-Raw".
3. Select the earlier loaded stack as **Volume** and set the method as **Integration**.



4. Apply a bandpass filter to remove low-frequency bias. in the **Process** tab, add a **Band-Pass Filter** process.
5. Select the volume created from the **Volume Integration** process e.g. "PseudoImp-Raw".
6. Select "**High-pass**" as the type and the frequency range of 4 to 8 (adjust according to seismic bandwidth).



An example comparing the log relative-impedance to pseudo-impedance and a band-limited impedance (from 3 stacks).

Generating synthetic stacks from Intercept and Gradient volumes

An overview of the workflow:

1. Calculate reference angle stacks from the gathers (see *Angle Stacks*).
2. Calculate intercept and gradient from the gathers (see *Intercept/Gradient*).
3. Use Shuey's approximation to calculate the synthetic volume (see *Volume Maths*).
4. (optional) Band-limit the synthetic to match the reference stack (see *Band-pass Filter*).
5. Calculate the residual using Volume Maths.

Setup

In this example, we prepared the following data:

For Angle Stacks

- Load gathers
- Load velocity model

For Spectral Analysis

- Draw a map polygon
- Create a window model to evaluate spectra at the target

For I/G calculation (from gathers)

- Define angle mutes
- Load gathers
- Load velocity model

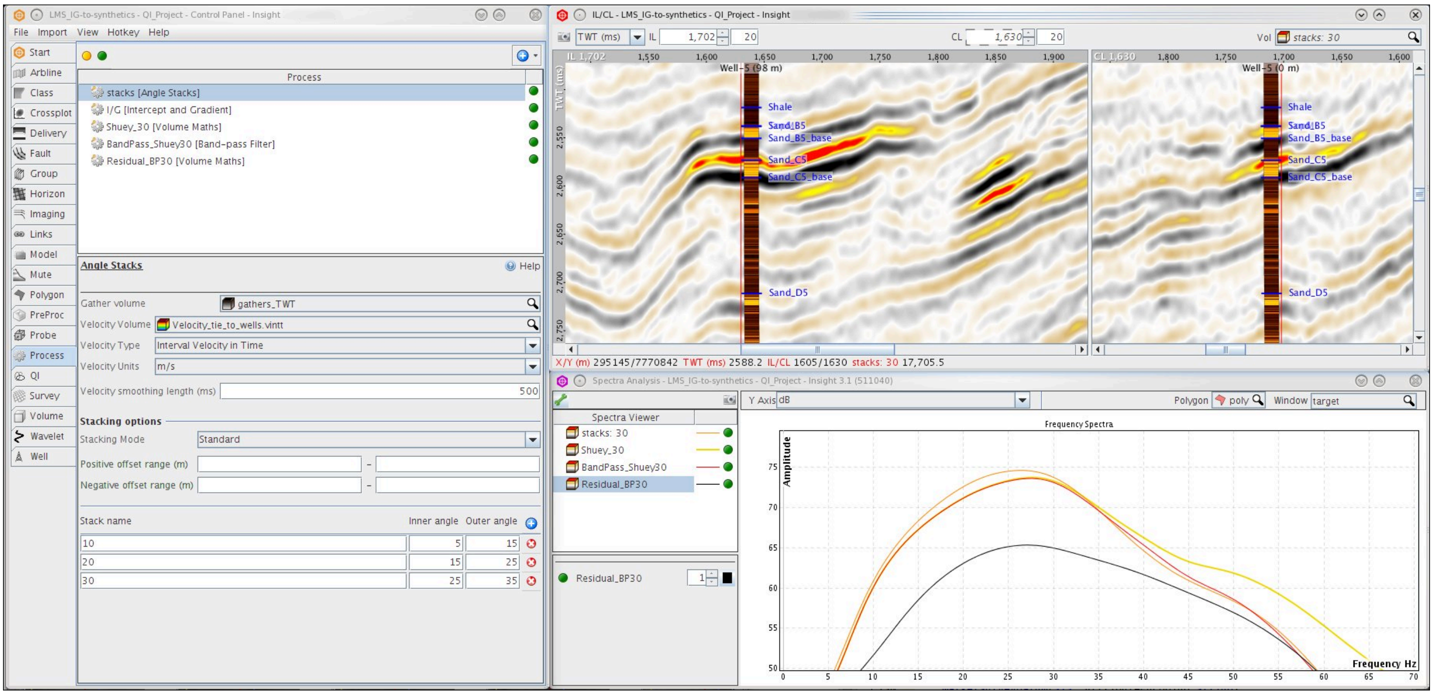
Use this workflow. Repeat steps 3, 4 and 5 for additional angles.

The spectral analysis window is included as a reference.

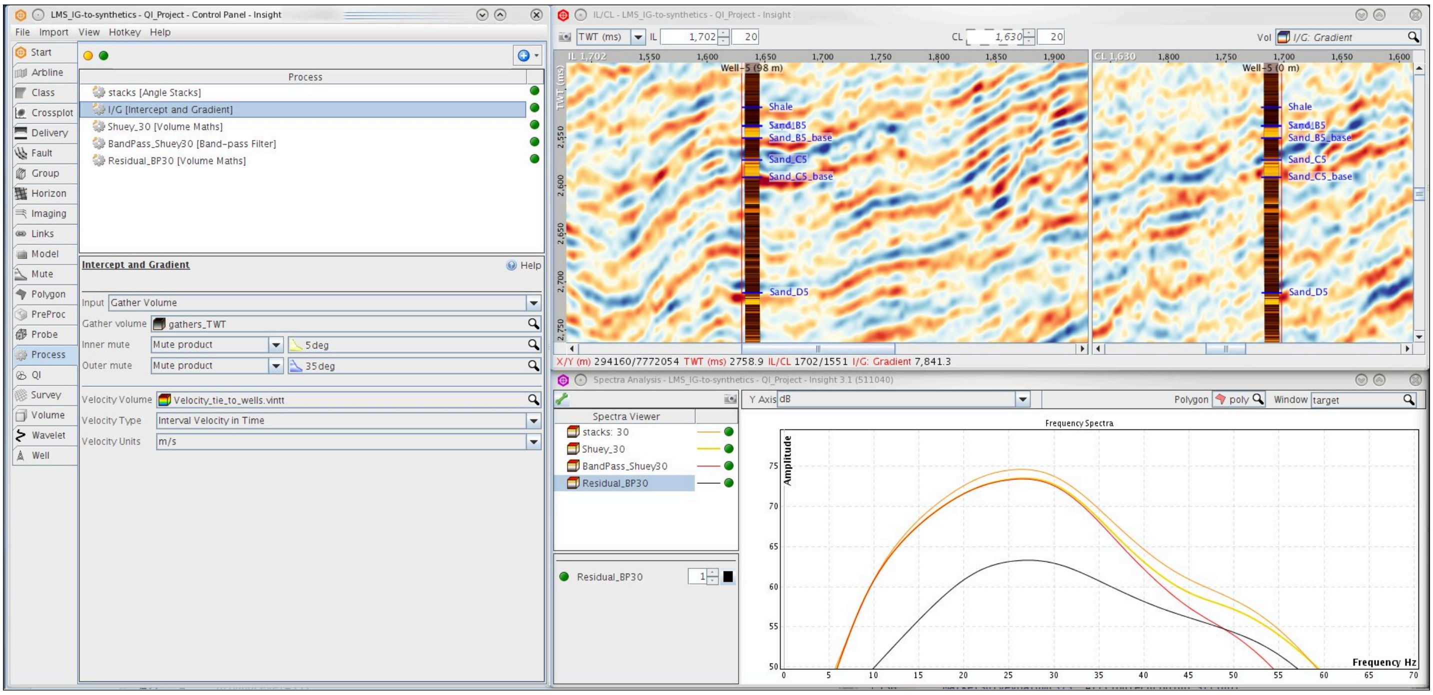
- Yellow = Stack
- Orange = Synthetic
- Red = Filtered Synthetic
- Black = Residual

Workflow

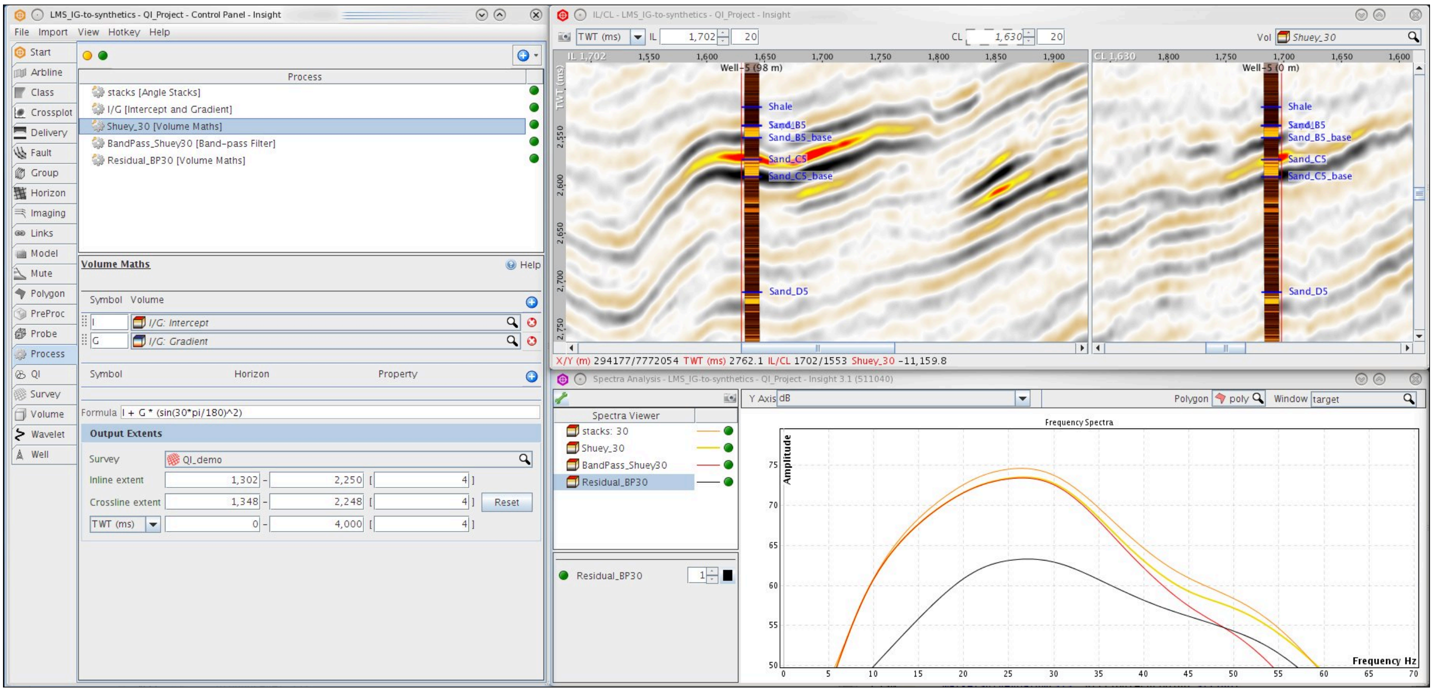
1. Create angle stacks from the gathers. We will be using the average of the min and max stacking angle for Shuey's equation.



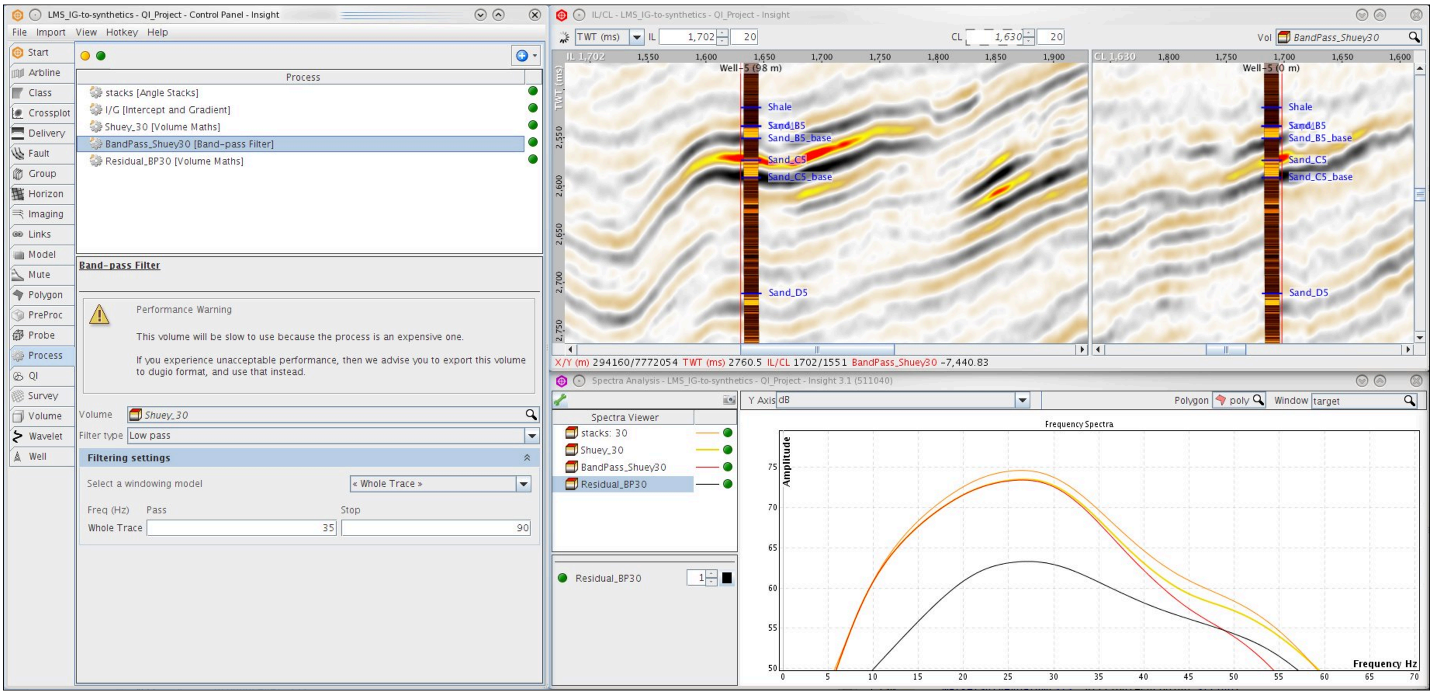
2. Calculate Intercept and Gradient from the gathers. Use angle mutes to avoid including near-offset noise and NMO stretch.



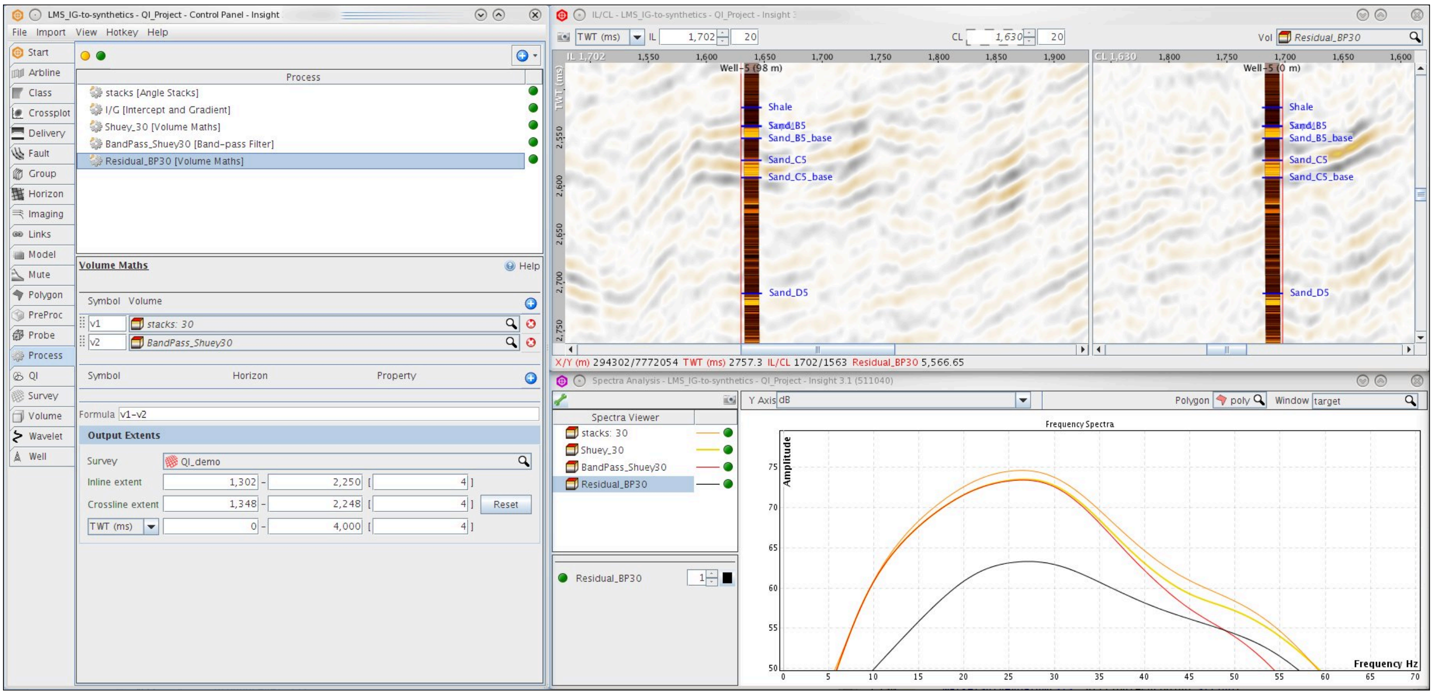
3. Use Volume Maths and Shuey's equation to calculate the amplitude using the Intercept and Gradient results for a specific angle. Note that the trigonometric functions expect radians as input (e.g. for 30 degrees).
   - Formula: I + G * (sin(30*pi/180)^2)



4. Depending on your data, the synthetic result may contain more high-frequencies than your angle stack, increasing the energy in the residual. By applying a bandpass filter, we can match the synthetic spectrum to the stack spectrum.



5. Finally, calculate the residual between the stack and the synthetic using volume maths.

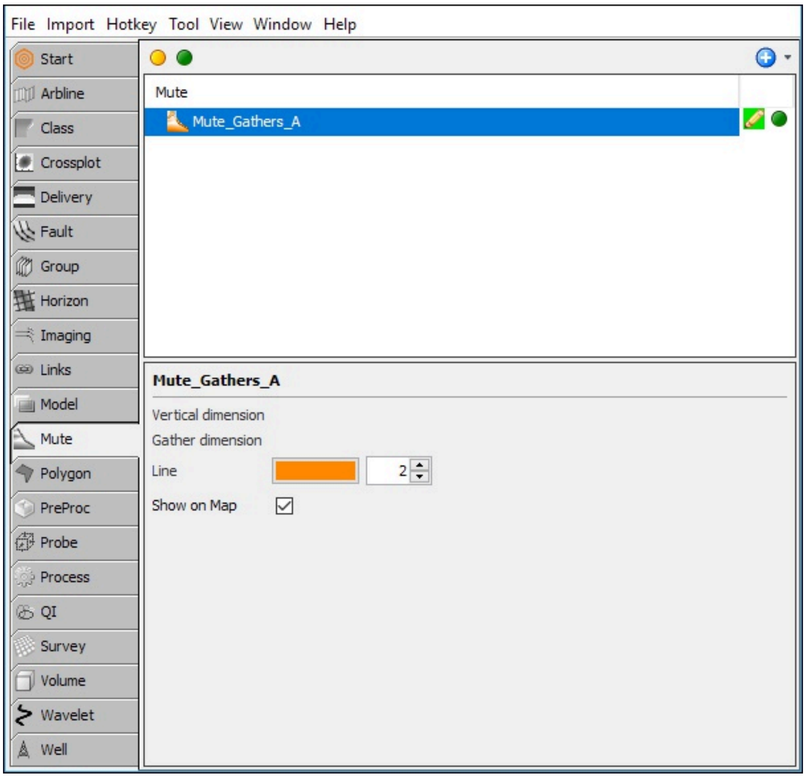

LMS_IG-to-synthetics - QI_Project - Control Panel - Insight
Process
stacks [Angle Stacks]
I/G [Intercept and Gradient]
Shuey_30 [Volume Maths]
BandPass_Shuey30 [Band-pass Filter]
Residual_BP30 [Volume Maths]
Volume Maths
Formula v1-v2
Output Extents
Survey QI_demo
Inline extent
Crossline extent
Shale
Sand_B5
Sand_B5_base
Sand_C5
Sand_C5_base
Sand_D5
Spectra Viewer
stacks: 30
Shuey_30
BandPass_Shuey30
Residual_BP30
Frequency Spectra
Amplitude
Frequency Hz

How to apply complex hand-drawn mutes to gathers and combine 2D results?

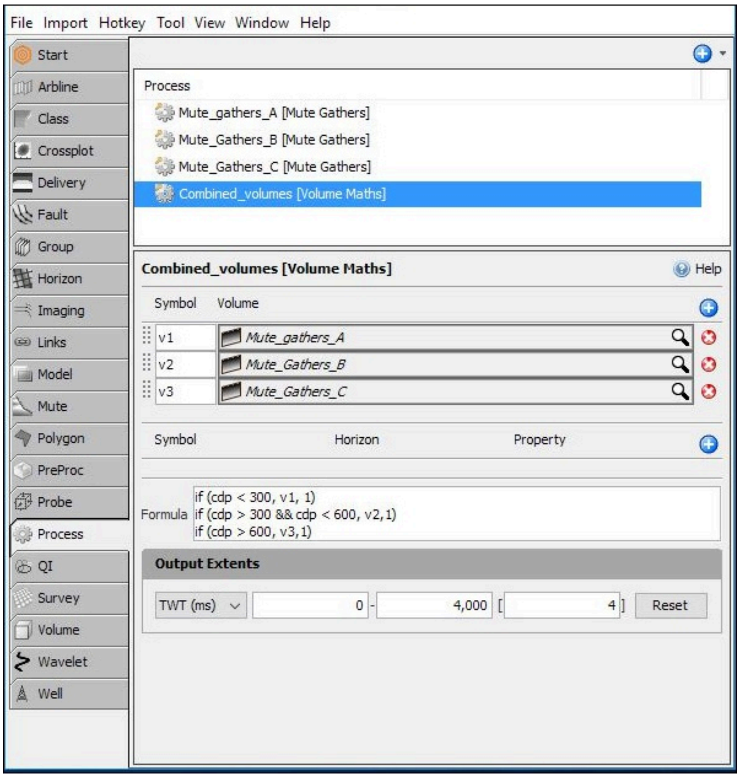
When working with complex hand-drawn mutes, the number of control points can quickly grow to unmanageable numbers. If you have access to the *Image Gather Processing module*, then consider using a polygon mute instead. This workflow is useful for anyone without access to the polygon mute process.

Instead of creating a single polygon mute for the entire line, create multiple hand drawn mutes, applying each mute separately, and combine these muted volumes (using volume maths) so that each mute is only applied over a certain CMP range.

Because it works with CMP values, this workflow is limited to a single line at a time.



1. Go to the **Mute** tab and add a **New Hand Drawn Mute** (see *Manual Picking (Mute)*).
2. Pick mutes for **first N traces** until an unworkable amount of control points appear.
3. Next, go to the **Process** tab and add a **Mute Gathers** process (see *Mute Gathers*).
4. Apply **Mute Gathers process** and select the previously picked mute to create *Muted Volume A*.
5. Return to the **Mute** tab and add another **New Hand Drawn Mute.**
6. Pick mutes for **second N traces** until the mute is unworkable.
7. Apply a new **Mute Gather process** to create *Muted Volume B*.
8. Repeat these steps until mute has been picked over the line.



1. Go to the **Process** tab and add a **Volume Maths** process (see *Volume Maths*)
2. Use **Volume Maths process** to combine the *Muted volumes*.
3. Click the blue "**+**" button and add the mute gathers created earlier.
4. In the formula box, apply this equation:

```
if (cdp <= 300, v1,
        if (cdp <= 600, v2, v3, 1)
)
```

This approach can be adapted for more sections by extending the formula, e.g. for 5 sections.

- v1 to CMP300
- v2 to CMP600
- v3 to CMP900
- v4 to CMP1200
- v5 rest of line

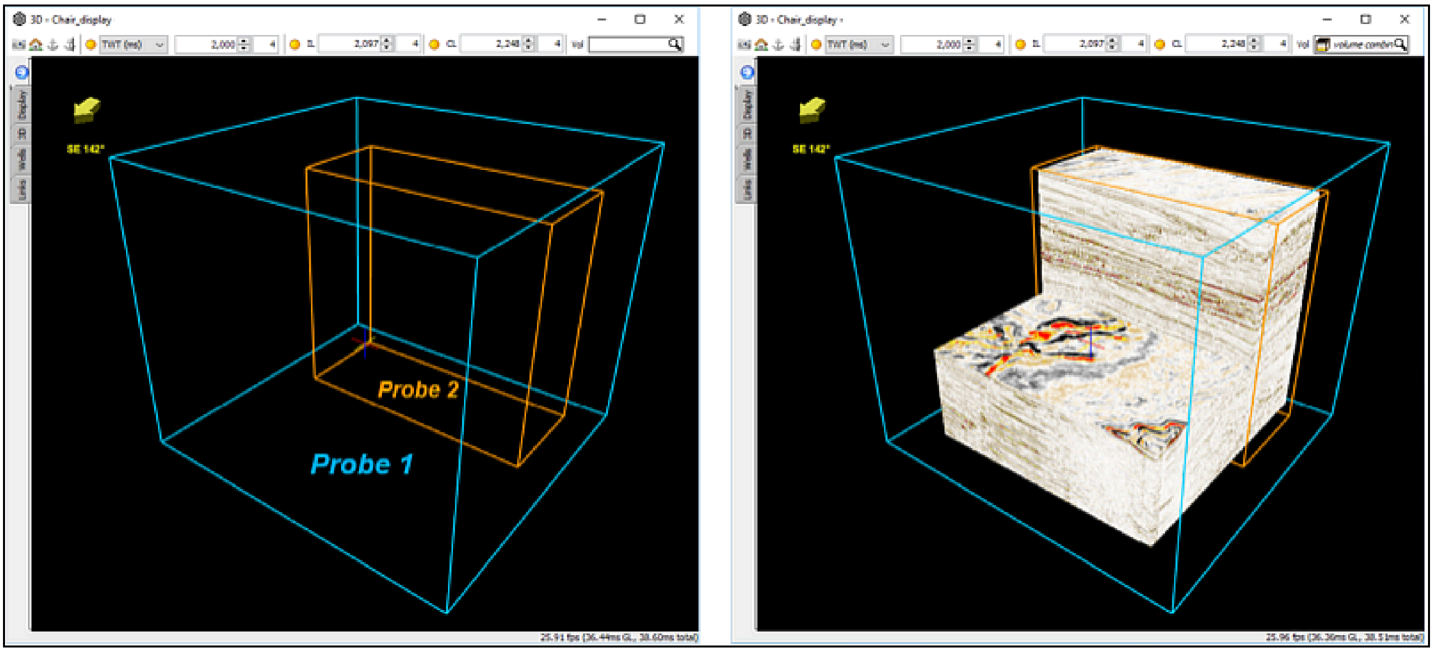
```
if (cdp <= 300, v1,
        if (cdp <= 600, v2,
                if (cdp <= 900, v3,
                        if (cdp <= 1200, v4,  v5)
                )
        )
)
```

How do I create a 3D Chair?

Chair cut seismic displays allow for concurrent viewing of multiple inlines, crosslines, and time/depth slices. This display can be created by combining sculpted volumes and can be updated on the fly by modifying process inputs.

Note: The **Explorationist** module is required for the *Volume Sculpting* process and the *Volume visualisation* feature (see *Explorationist module*).

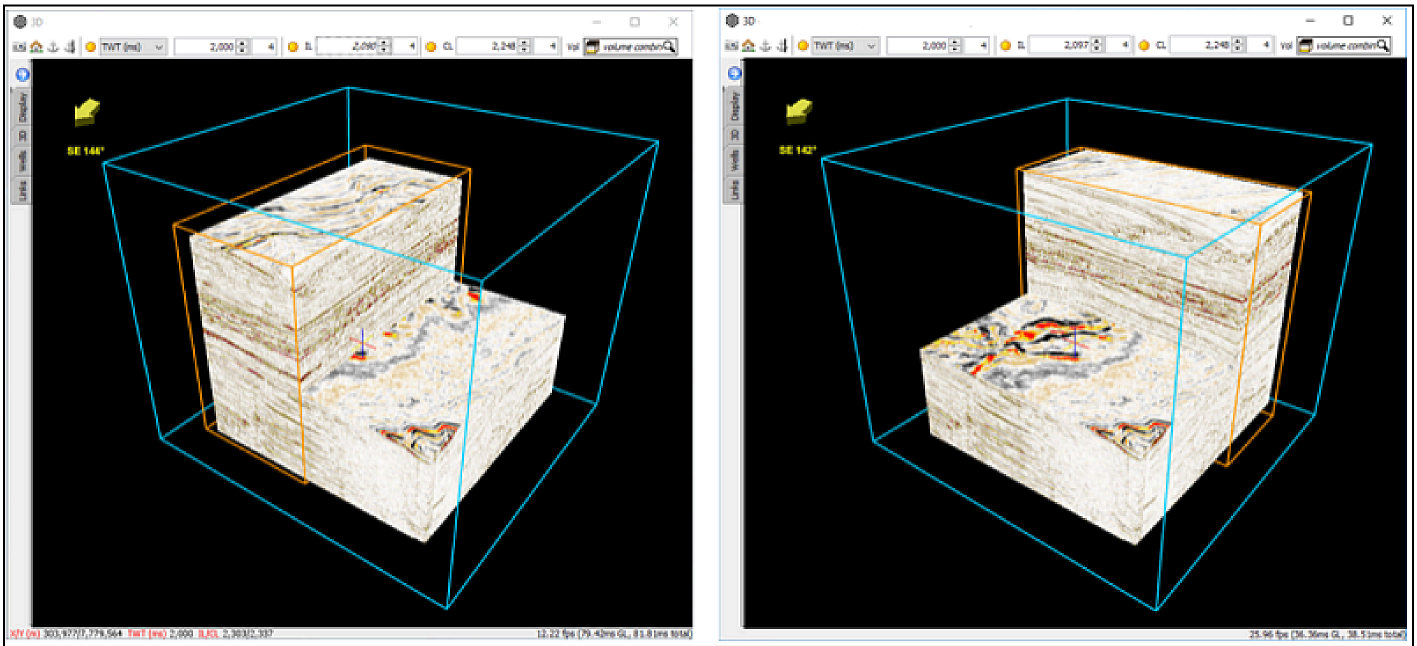
Step 1: Create Two Probes



The first probe will be used to cover the full extent of the volume which will allow the entire 3D 'cube' to be displayed in 3D view. To display the data using the probe, select the probe in the **3D view > 3D tab** and check the **Use Probe** box.

1. **Probe 1** - used to set the display extents of the 3D view area.
   - Probe 1 can be as large or a small as you would like. This probe will define the maximum display extents.
   - In **3D view**, open the **3D tab**, check the **Use Probe** box, and select **Probe 1**.
2. **Probe 2** - This probe will be used to adjust the horizontal extents of the 'back' of the chair.

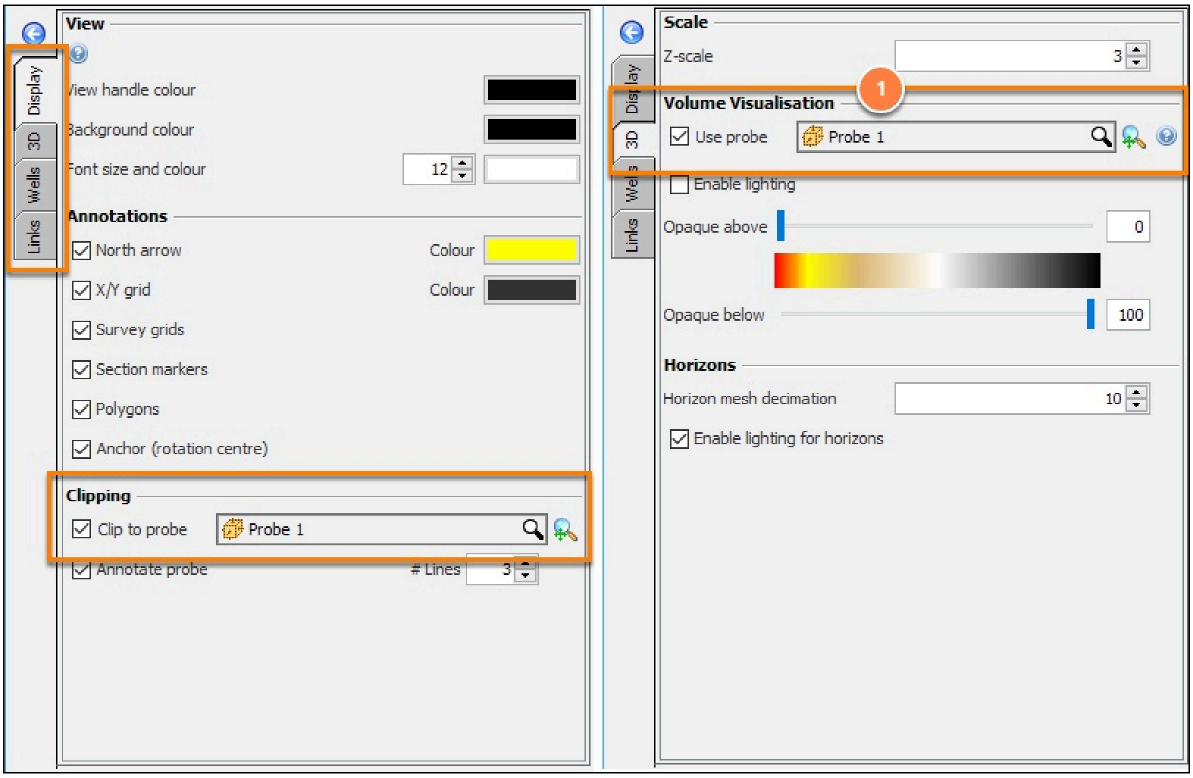
Step 2: Use Three Processes



The display extents of the volume can be modified interactively by adjusting the dimensions of the probes in 3D view or the process parameters for the sculpted volumes.

1. Volume sculpting 1
   - Use Probe 2 as input
   - Adjusting Probe 2 will change the horizontal extents of this sculpted volume
   - Set ‘Top Horizon’ to Constant value and enter in a value to set the vertical extent of the sculpted volume
2. Volume sculpting 2
   - Used to adjust 'seat' of the chair, no probe is needed as input
   - Instead of using a probe, define top and bottom vertical extents by entering constant values in process parameters
3. Volume combine
   - Add in Volume sculpting 1 and 2, and the output survey
   - This combines the two sculpted volumes into one volume
   - Adjusting the parameters in the input sculpted volumes will allow you to customize the display extents

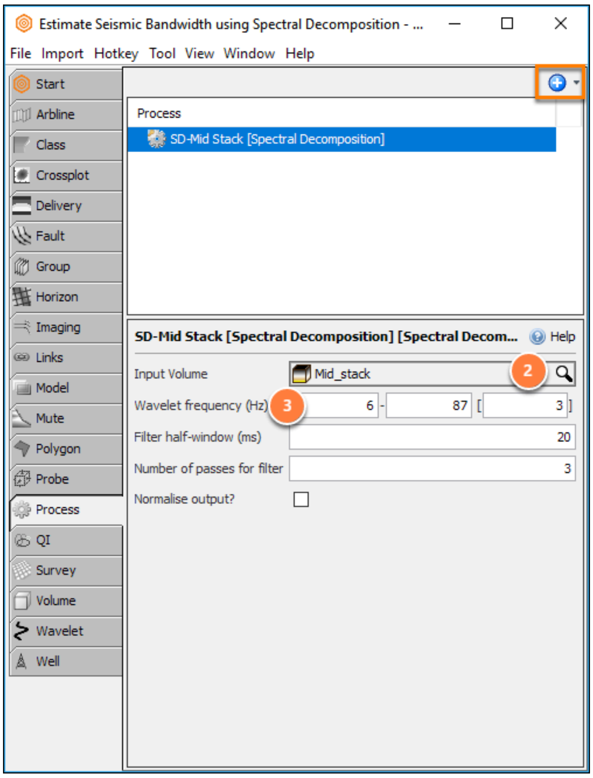
Step 3: Configure 3D view



1. In the **3D tab**, make sure the **Use Probe** box is checked and select **Probe 1** (see *Volume Visualisation*).
2. Select the **Volume Combine** volume from the volume drop down menu.
   - To define the horizontal extents of the back of the chair, click the pencil icon for **Probe 2** and click-drag the probe to adjust the extents. Click the pencil icon again to leave editing mode and the volume extents will update in **3D view**.
   - To define the vertical extents, modify the **Top Horizon** value in the **Volume sculpting 1** process.
   - To define the vertical extent of the seat of the chair, open the **Volume sculpting 2** process, and enter a value for the top horizon.
   - If you'd like to limit the horizontal extents of the seat, this can be done by modifying the extents of **Probe 1**.
   - If you'd like to hide the probes, this can be done by disabling the display in the **Control Panel > Probe tab**.

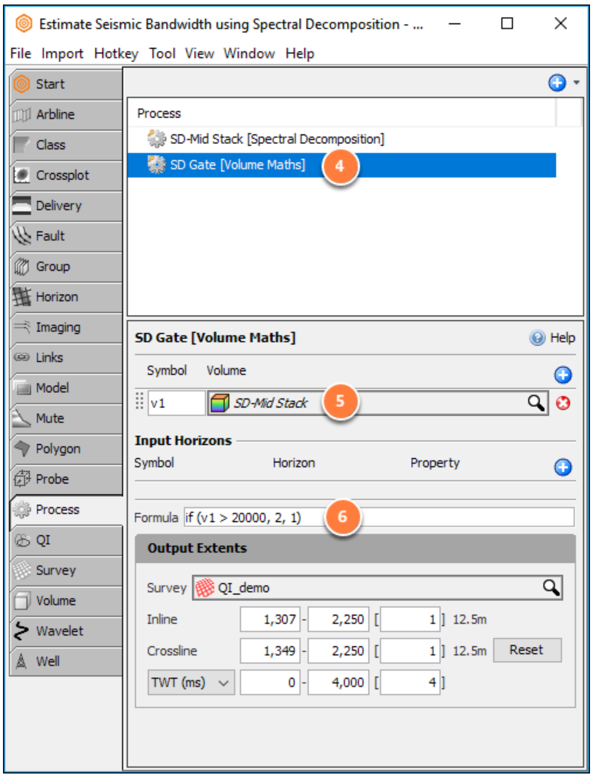
How to estimate seismic bandwidth using spectral decomposition?

Workflow



Calculate the **Spectral Decomposition** result:

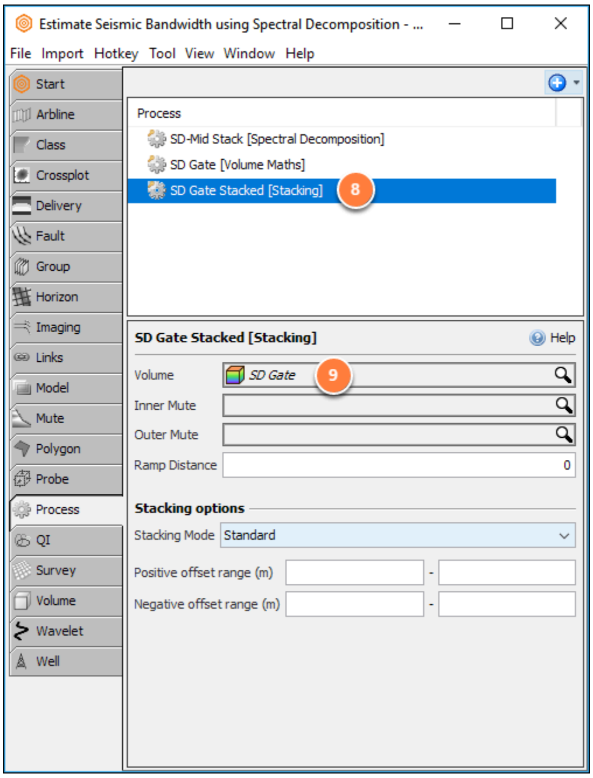
1. Click the **Process** tab, click the blue "+" button and add a **Spectral Decomposition** process.
2. Select an **Input Volume**.
3. Pay attention to the **Spectral Decomposition** frequency interval, it's needed for the last step.



Apply a gate / threshold function to specify the minimum amplitude to qualify for inclusion.

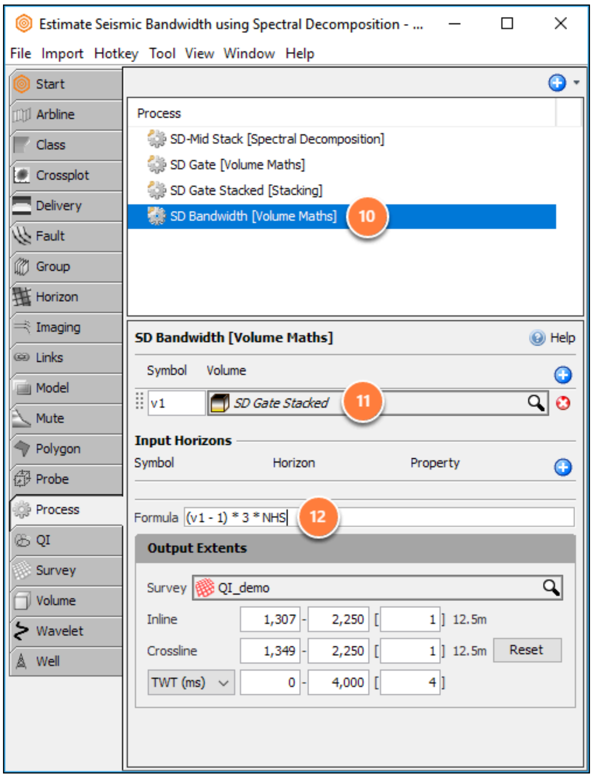
4. Click the blue "**+**" button and add a **Volume Maths** process.
5. In the details panel, add a symbol and select the result from the previous process.
6. Use formula: **if (v1 > 20000, 2, 1)**
7. Tune this threshold to adjust your results. This example uses 20,000, but it will vary depending on your spectral decomposition amplitudes.

Note: The gate uses values of 1 & 2 to avoid creating "missing values" in the stacking.



Stack the gated result (in effect counting the number of frequencies above the threshold).

8. Click the blue "**+**" button and add a **Stacking** process.
9. In the details panel, select the result from the previous **Volume Maths** process in the volume search box.

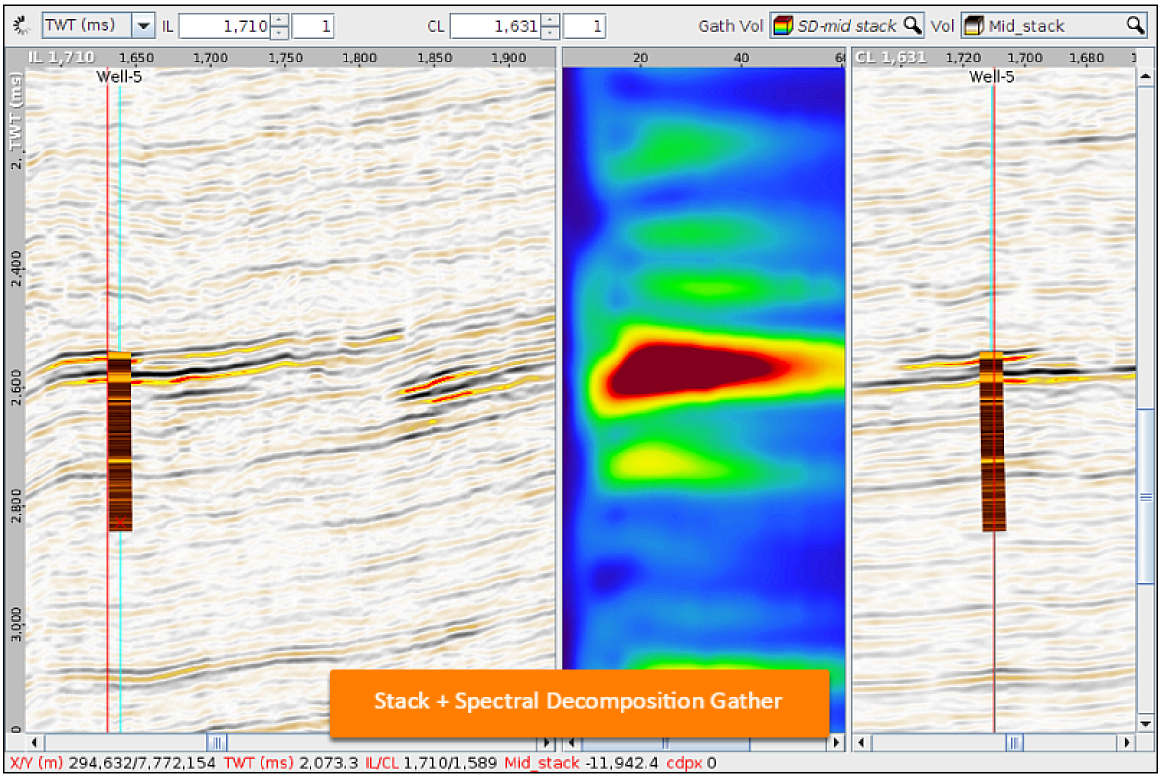


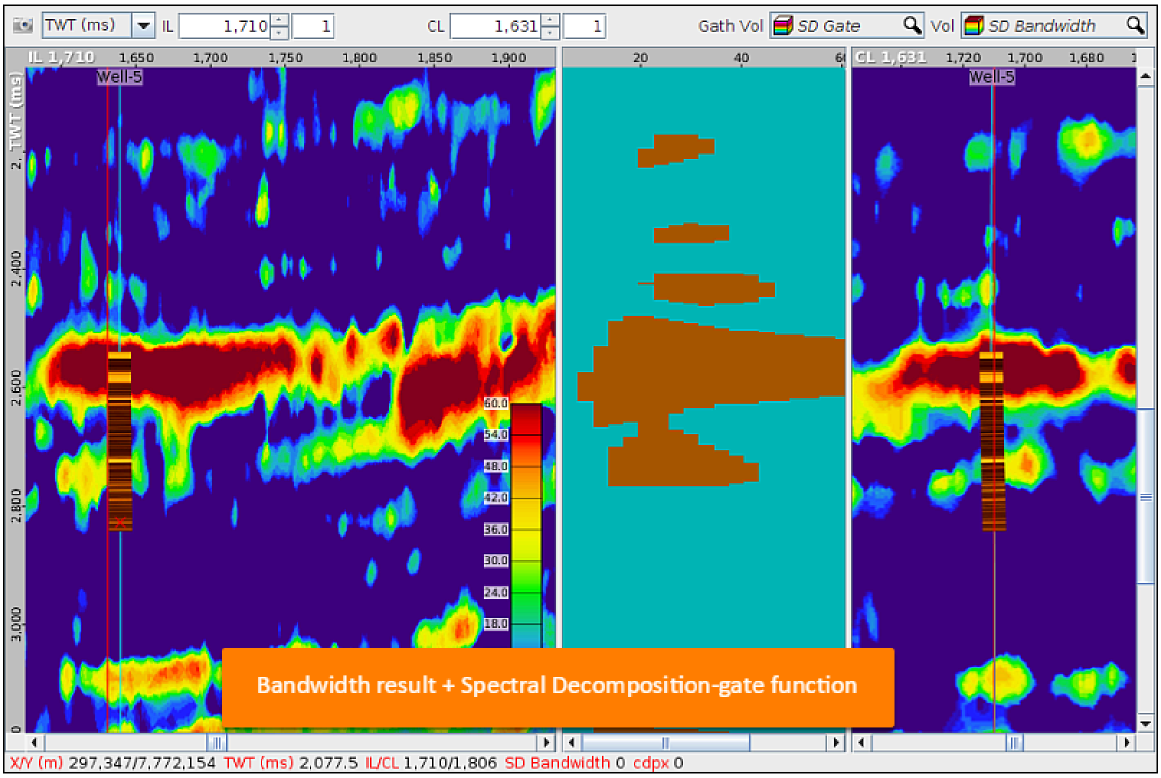
Rescale the stack by the number of stacked traces (NHS trace header) and the frequency interval from step 1. This gives the bandwidth estimate.

10. Click the blue "**+**" button and add a **Volume Maths** process.
11. In the details panel, add a symbol and select the result from the **Stacking** process.
12. Use formula: **(v1 - 1) * 3 * NHS**

Note: A constant 1 value is subtracted from the stack to account for the 1-2 range of the gate. For this example, the value 3 is the frequency interval from step 1.

Results




TWT (ms)
IL 1,710 1
CL 1,631 1
Gath Vol SD Gate
Vol SD Bandwidth
Well-5
Bandwidth result + Spectral Decomposition-gate function
X/Y (m) 297,347/7,772,154 TWT (ms) 2,077.5 IL/CL 1,710/1,806 SD Bandwidth 0 cdpx 0

How do I reverse the CDP to X/Y relationship for a seismic line?

This procedure will permanently reverse the traces along a 2D line. Be sure you know what you are doing!

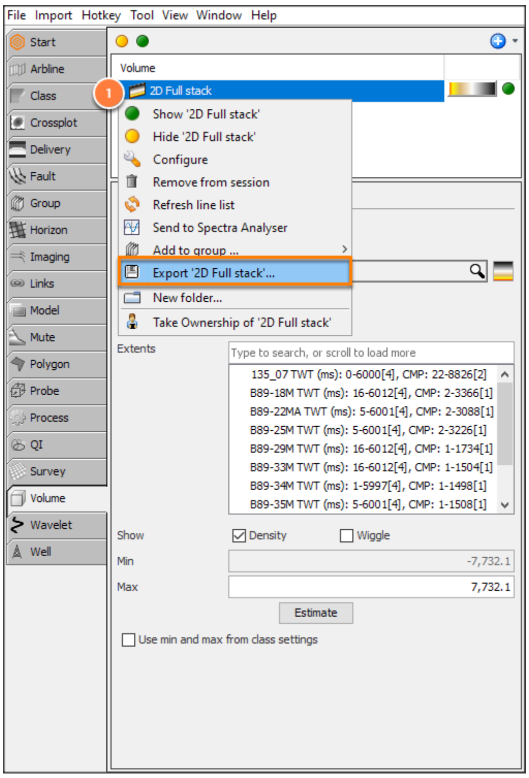
Each trace on the line is moved to a corresponding CMP, measured from the end of the line. e.g.

- The first trace of the line is swapped with the last trace.
- The fifth trace of the line becomes the fifth from the end of the line.

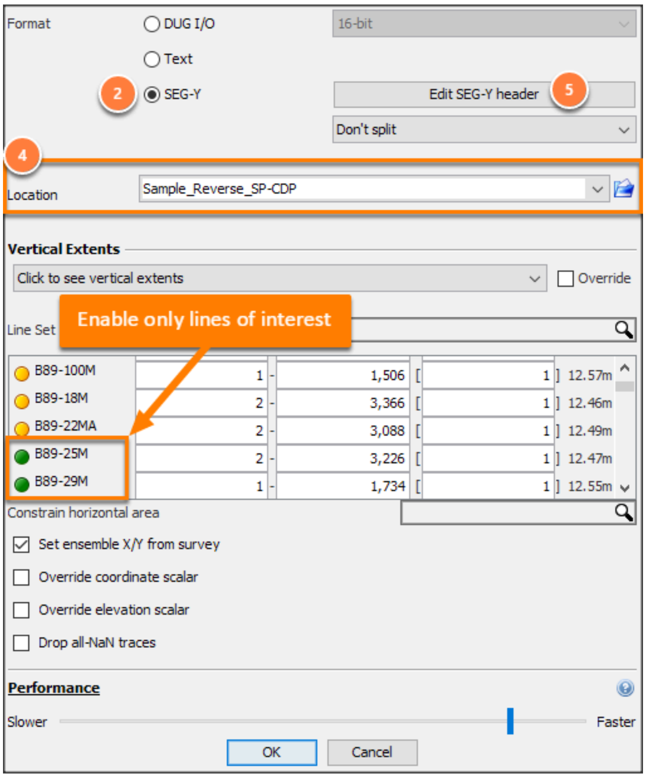
A copy of the line is exported to SEG-Y, making changes to the CMP header as it is exported. The seismic line is then reimported, using the replaced CMP values for each trace. The 2D survey geometry is not changed, so the new CMP values result in the trace order being reversed.

Normally, Insight does not allow changing headers that determine the position of a trace. For 3D data, these are the IL/CL headers. For 2D data, this is the CMP value.

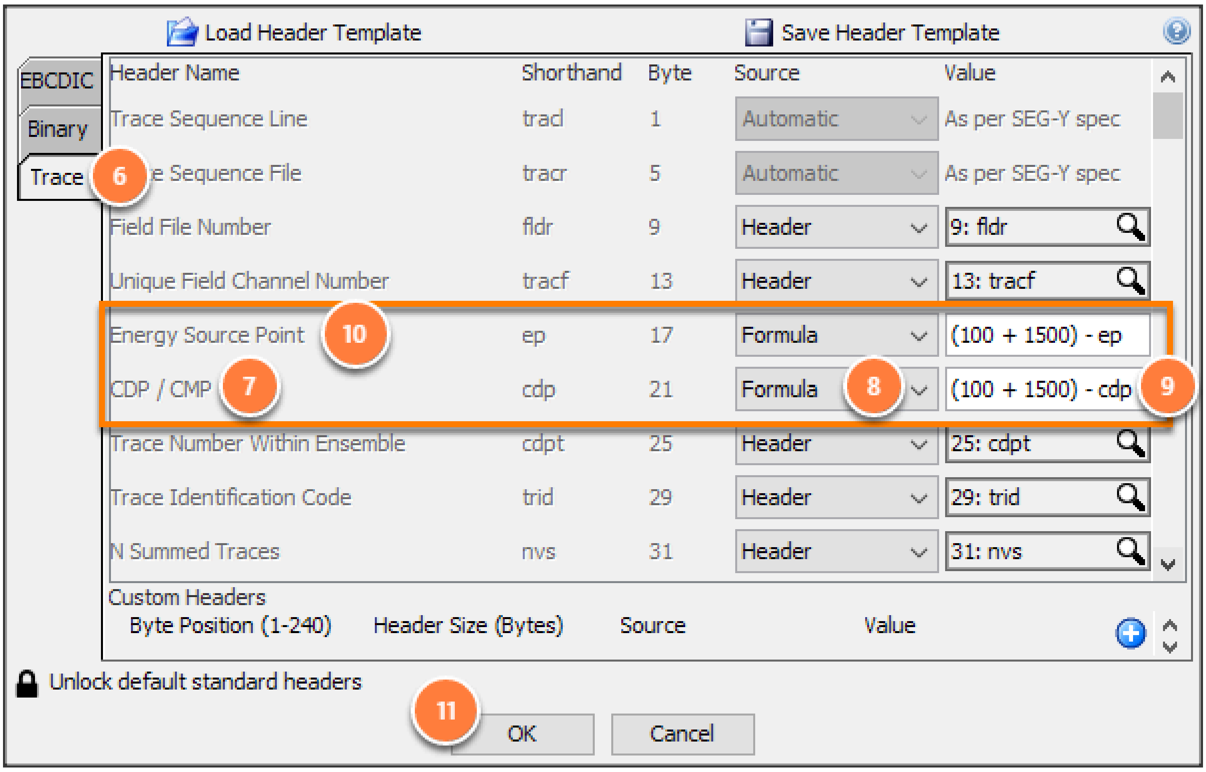
When exporting to SEG-Y, Insight quietly looks the other way and does not apply the same restriction.



1. Right click on the **2D** volume, select **Export '*2D volume name*'...**

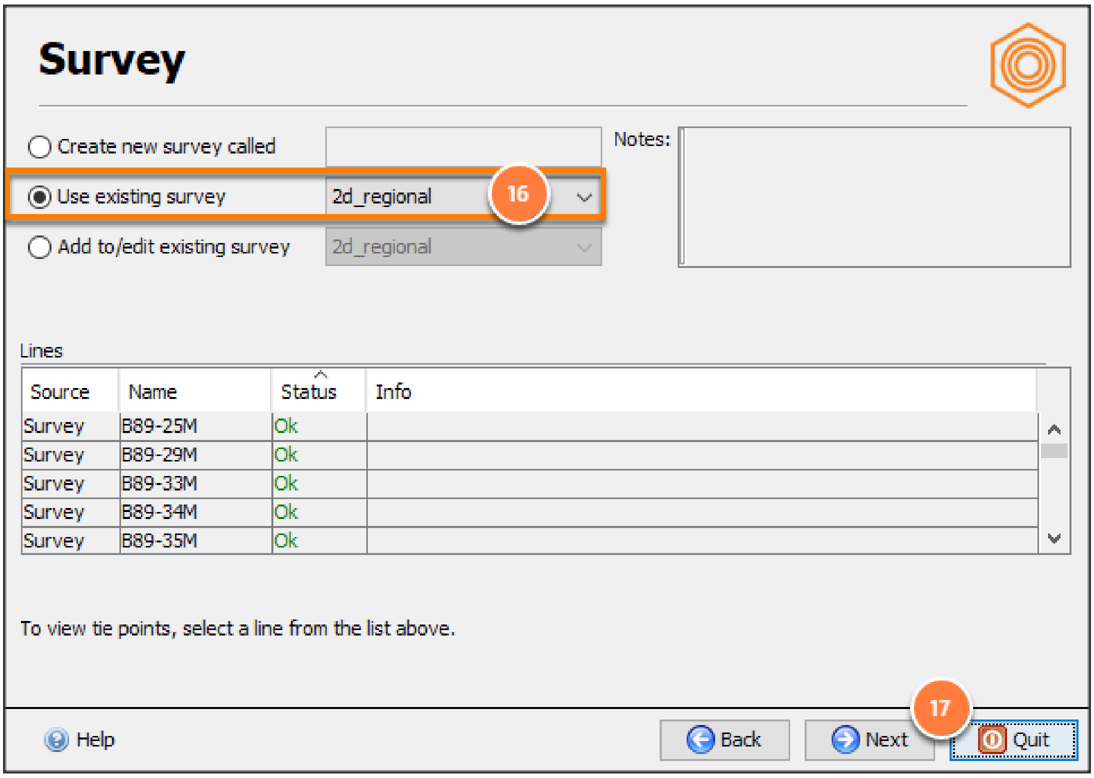


2. In the **Export Volume window**, select **SEG-Y** as the **Format** (see *[Export a 2D volume to SEG-Y](#)*).
3. Disable all lines, then enable the lines that need to be reversed.
4. Give the file a name at **Location**.
5. Click **Edit SEG-Y header**.

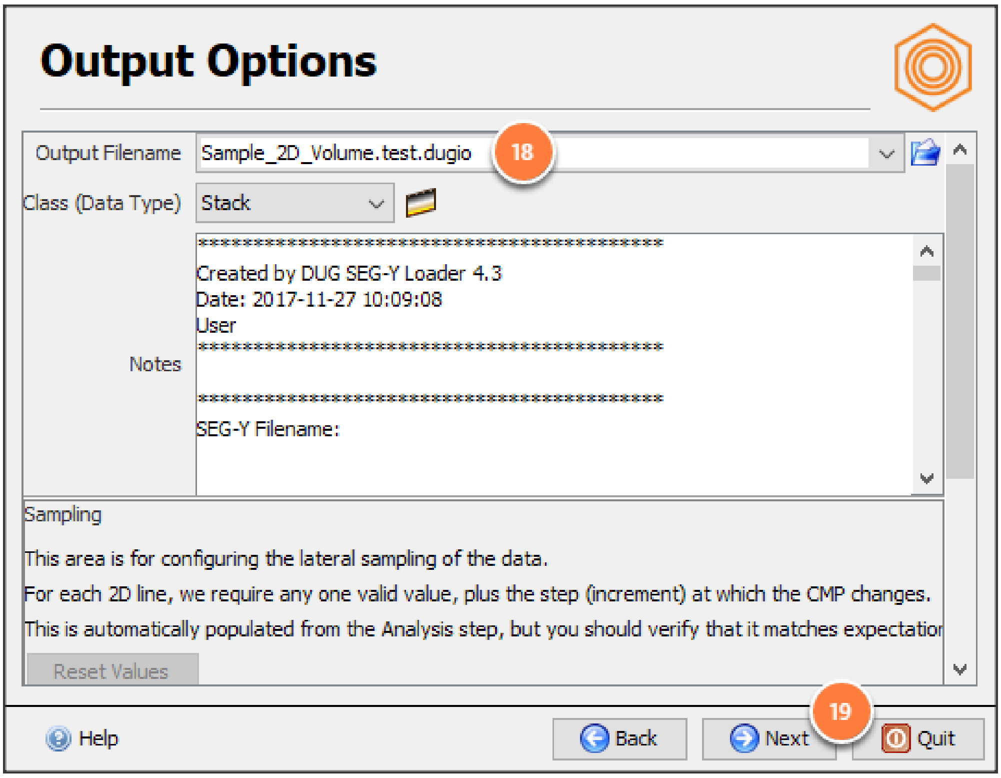


6. In the **SEG-Y Header Editor** window, click the **Trace** tab.
7. Find **CDP / CMP** under **Header Name (byte 21)**
8. Click the Source drop-down, and select "**Formula**".
9. Enter the following formula for "**Value**":
   - **(first CDP + last CDP) - cdp**
   - E.g. for a line with CDP ranging from 100 to 1500, use: **(100 + 1500) - cdp**
10. To also update the shotpoint, at "Energy Source Point" (byte 17), use the formula:
    - **(first SP + last SP) - ep**
11. Click **OK** to save.
12. Click **OK** again to start exporting.
13. Once the export is complete, save the session and exit **Insight**.

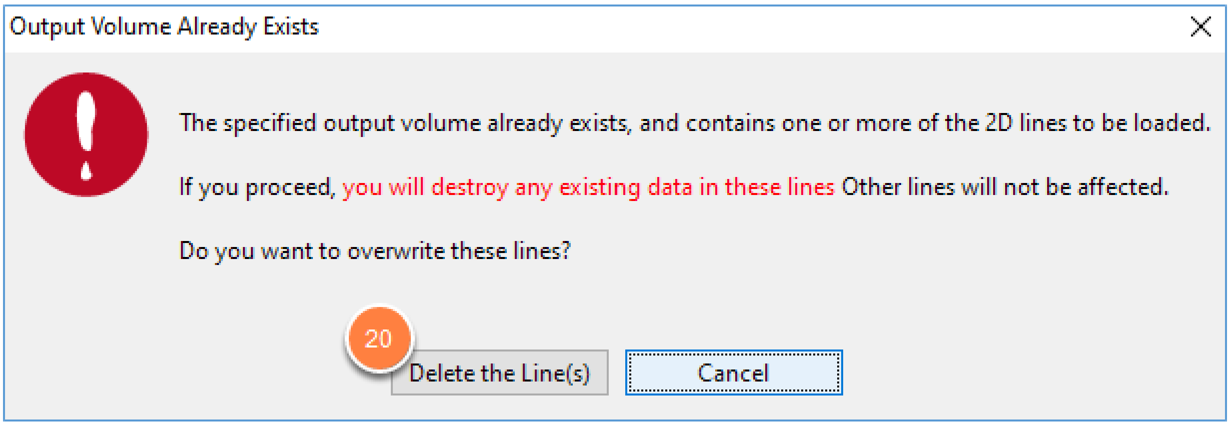
Tip: To QC the result, export the original survey and volume as backup.



14. Use **SEG-Y Loader** to import the **SEGY** file created in step 1 (see *Launching SEG-Y Loader*).
15. Select the exported lines (no need to rename).
16. In the **Survey** page, select **Use existing survey**.
17. Click **Next**.



18. In the **Output Options** page, choose the existing .dugio file containing your 2D data.
19. Click **Next**.



20. A warning will appear letting you know that you will be overwriting existing data. Click **Delete the Line(s)** to continue.
    - Only line being loaded will be overwritten.
    - Data on other lines will not be affected.
21. Launch **Insight** again and load the session to check the new lines.
22. Import the original survey and volume from the earlier backup to QC the result.

SEG-Y

Why am I having issues loading SEG-Y?

We've updated this information. You can get the latest details *here*!

SEG-Y data cannot be loaded directly into DUG Insight. It must first be converted into DUG's optimised dugio format (*see* SEG-Y Loader).

Getting SEG-Y files into interpretation software can be a complex and confusing task. The amount of time wasted with persistent errors is frustrating, not to mention the potential repercussions of having incorrectly loaded data. Most of these errors and misunderstanding revolve around the issue of Header Mapping – an essential part of the SEG-Y loading process.

A good investment is to learn how the SEG-Y header information is stored, and understand the necessary steps involved in any SEG-Y loading process.

This document is aimed at the non-data specialists, and designed to help users understand the complexities of SEG-Y files, and facilitate the task of loading these into Insight. It describes a SEG-Y file in layman's terms, with particular attention paid to the use of SEG-Y header information, during the SEG-Y loading process.

SEG-Y Overview

SEG-Y is an internationally recognised file format developed by the Society of Exploration Geophysicists (SEG) for storing geophysical data, particularly in the Seismic Industry.

Originally released in 1975, SEG-Y rev 0, experienced widespread usage, which brought about many proprietary variations. These variations have resulted in many inconsistencies throughout the industry, particularly in the way header information (such as positioning data) is stored.

The most recent revision of the SEG-Y format was published in 2002, named the rev 1 specification (http://www.seg.org/documents/10161/77915/seg_y_rev1.pdf).

One of the major changes introduced by SEG-Y rev 1, was standardising the way header information is stored for improved consistency, and also to cater for modern acquisition and processing techniques.

Despite improved standardisation and consistencies, SEG-Y data is still notoriously "non-standard", particularly with older vintages, which consequently gives rise to difficulties loading data.

So what is in SEG-Y?

A SEG-Y file comprises data traces (which contain the actual seismic records) as well as three types of header information: a Text Header, a Binary Header, and individual Trace Headers.

Text Header

The Text Header (aka **EBCIDIC** Header), provides a human-readable description of the data in the SEG-Y file (a bit like a readme file), and usually (though not always), contains useful information on how the SEG-Y file should be loaded.

Binary Header

The Binary Header contains information (values) that apply to the whole SEG-Y file, such as the sampling interval and the total number of samples in the traces. **The Binary Header** appears like a table, comprising rows of information, divided into three columns. Each row pertains to a specific piece of information, and the three columns are: 1: Byte code, 2: a human-readable descriptor, 3. the value. See example table below:

Byte Code	***Descriptor***	***Value***
hns	*Samples per trace*	*1250*
Hdt	*Sample Interval (µs)*	*4000*

Table 1: Example Binary Header Format

Trace Headers

The data Trace Headers contain information (values), pertinent to each and every data trace (seismic record). These values may change on a trace-by-trace basis, such as the CMP number, and the geographic X and Y coordinates of each data trace.

The information (values) contained within each of the trace headers are stored in different "byte locations".

A byte can be thought of as a repository in the file, where non-seismic information (values) can be stored. These bytes are numbered from 1 to 240. Often, the information to be stored is too large to be stored on a single byte alone and consequently 2 or even 4 bytes are commonly used to store a particular piece of information.

Note: In computer coding lingo, 2 byte and 4 byte locations are also known as "int16" and "int32" respectively. These distinctions are relevant in the latter sections of this guide.

The SEG-Y rev 1 standard defines specific bytes for the storing of specific information (values), needed for the processing and interpretation of SEG-Y files. The table below describes some of the more commonly used bytes and the specific information they contain, as per the SEG-Y rev 1 standard:

Trace Header Byte Location	Description of Value
1-4	Trace Number
17-20	Energy Source Point Number
21-24	Ensemble (CMP) Number
37-40	Offset
71-72	Scalar to be applied to Coordinates
73-76	Source X Coordinate
77-80	Source Y Coordinate
109-110	Record Delay
181-184	Ensemble (CMP) X Coordinate
185-188	Ensemble (CMP) Y Coordinate
189-192	Inline Number
193-196	Crossline Number
197-200	Shot Point Number

Table 2: SEG-Y Rev 1 Standard Trace Header byte locations for commonly used values in the processing/interpretation of SEG-Y data.

Data Traces

The actual seismic data in a SEG Y file is made up of a series of individual seismic records, known as the data traces. Depending on the type of data (2D, 3D, gathers etc), these data traces will be organised in a particular fashion in order to be loaded and displayed correctly in the interpretation software.

How does Insight know?

As part of the SEG-Y Loading process, Insight needs to know which bytes contain the required information such as CMP Number, CMP X Coordinate and CMP Y Coordinate etc. This process is known as “mapping the headers”, and is done in Stage 3 of the loading process: “Specify the byte locations of headers” screen, Figure 1 below.

Notes:

- When specifying byte locations in Insight SEG-Y Loader, only the first of the 2 or 4 bytes (required to store a particular piece of information) is required to be stated e.g. for X Coordinate, we would simply specify byte 181, not 181 to 184.
- The byte length (181-184) is specified by selecting int32 from the adjacent drop down menu (see **Note** in *So what is in SEG-Y?* above).

By default, Insight automatically looks in the relevant standard byte locations (Table 2 above). If the SEG-Y file being loaded does adhere to the SEG-Y standard, then the user simply has to check the values obtained are as expected, and then move on to the next stage.

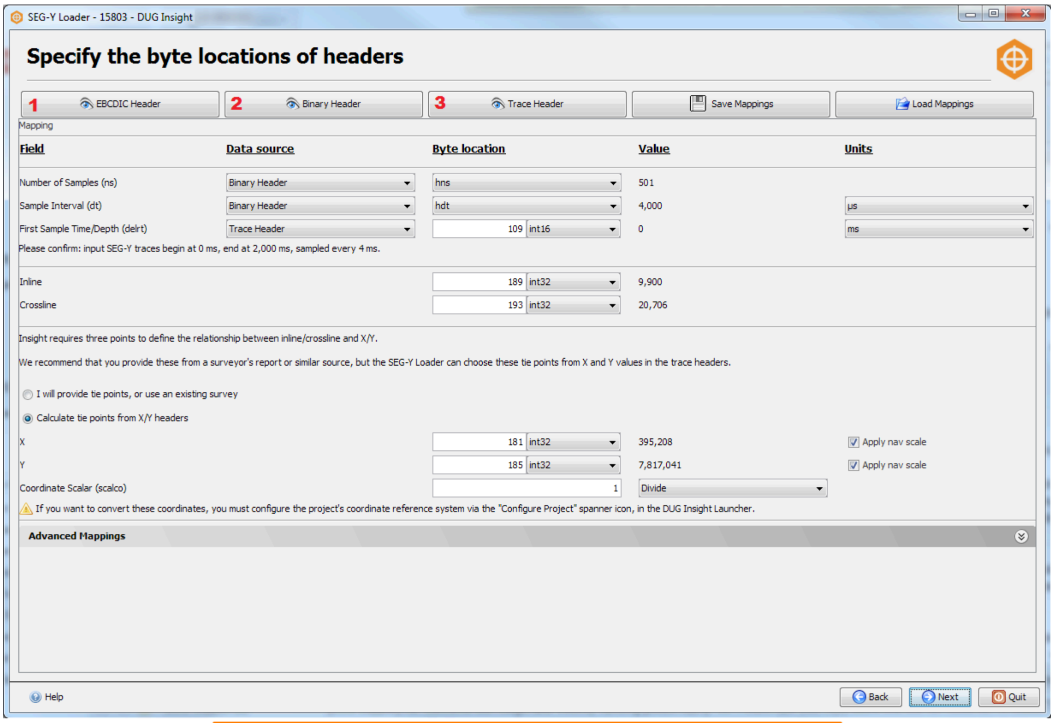


Figure 1: Header Mapping, showing the default byte locations for 3D Seismic SEG-Y

But here is the catch

Unfortunately, due to the variety of acquisition systems and processing standards employed by various companies around the world, these SEG-Y standard byte locations are often not adhered to, and the user consequently has to map these byte location manually.

Some examples of commonly “misplaced” information are:

- CMP X and Y coordinates stored in place of the Source X and Y coordinates in byte’s 73 and 77 respectively.
- Ensemble (CMP) Number stored on byte 25, not byte 21
- Inline Number and Crossline Number on byte 9 and 13, not 189 and 193.

The possibilities are endless.

How does the user know?

In the **likely** event that the SEG-Y file does not adhere to the SEG-Y rev 1 standard, the user is required to manually tell Insight which bytes contain the required information.

Assuming the user has received no specific loading instructions, the first place to look is in the Text (EBCIDIC) Header.

The purpose of the Text (EBCIDIC) Header is to provide a description of the SEG-Y file (Figure 1, button 1), and should (but often does not) include a list of which Trace Header byte locations contain all the important information.

In the example Text Header shown in Figure 2 below, we see that the Text Header states the X and Y coordinates are contained within bytes 73 and 77 respectively; and the inline and crossline numbers are in bytes 9 and 13 respectively. The user can then specify these byte locations, and move on with the process.

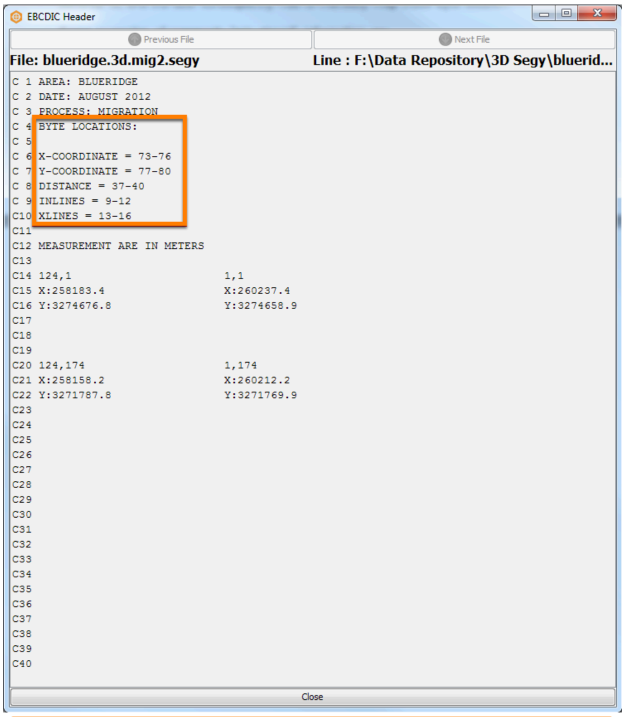


Figure 2: Example Text Header, describing which Trace Header byte locations contain the required information

Sometimes however, the Text Header provides no information at all, or even worse: misleading incorrect information. In this case, the user will then have to manually inspect the **Trace Headers** themselves (Figure 1, button **3**). The process of manually inspecting the trace headers basically involves looking for values that fit the description of the value you are looking for. For example, X and Y coordinates usually have a recognisable six and seven digit number, respectively (for UTM coordinates). In the case of coordinates, the user must also be wary of any coordinate scalar value.

When you view the trace headers in the **Trace Headers** window, the byte locations are labeled according to the SEG-Y REV-1 specification (blue box in Figure 3 below). Only the values are read from the SEG-Y file (green box in Figure 3 below).

In the example shown in Figure 3 below, we see seven and eight digit numbers in bytes 73 and 77 respectively, which would otherwise fit our expected, X and Y coordinates. There is also a coordinate scalar of 1/10 (divide by 10), to be applied to these values. This is indicated by the "-10" on byte 71 (Coordinate scalar) on the int16 (2 byte) column.

Note: From Table 2 above, Coordinate scalar is a 2 byte header, and 2 byte is also known as int16, hence the coordinate scalar appears on the int16 column.

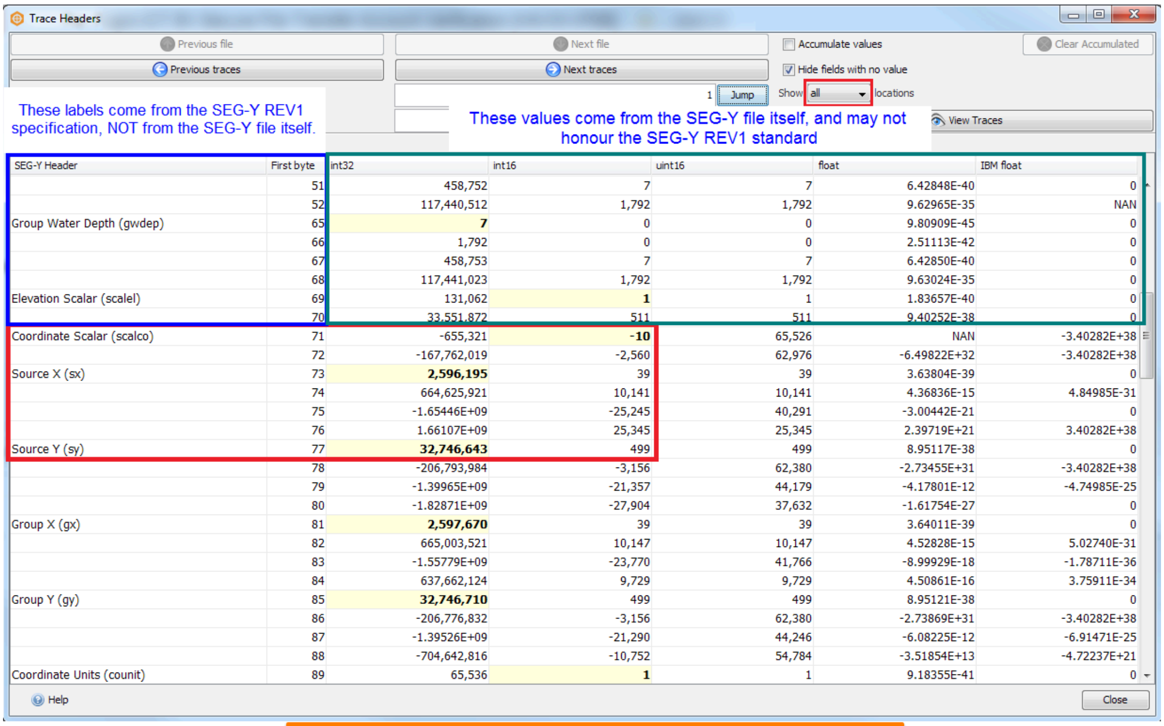


Figure 3: Manual inspection of All Trace Header values

The trace headers window will, by default, show only the "likely" byte locations. The term "likely" here refers to only those byte locations (SEG-Y rev1 standard) relevant to loading of that particular SEG-Y, e.g. Inline and Crossline (bytes 189 and 193) are not shown when loading 2D data. Therefore, may need to select to show "all" locations in order to show the bytes you need to inspect.

Depending on the type of data being loaded the following Table 3 below, presents a summary of the essential information required for data loading into Insight, and its usual (SEG-Y standard) byte location.

Required Information	***Usual Location***
Sample Interval (dt)	*Binary Header – Line 6 (hdt)*
Number of Samples (ns)	*Binary Header – Line 8 (hns)*
First Sample Time/Depth / Record Delay	*Trace Header Byte 109*
Coordinate Scalar	*Trace Header Byte 71 (int16)*
Ensemble X Coordinate	*Trace Header Byte 181 (int32)*
Ensemble Y Coordinate	*Trace Header Byte 185 (int32)*
Common Midpoint (2D Only)	*Trace Header Byte 21 (int32)*
Inline Number (3D Only)	*Trace Header Byte 189 (int32)*
Crossline Number (3D Only)	*Trace Header Byte 193 (int32)*
Offset (Gathers only)	*Trace Header Byte 37 (int32)*

Table 3: Essential information required for data loading in Insight

Some **useful tools** available to you in the Trace Headers window:

- View information for multiple traces at the same time, by increasing the value in the **Number of Traces** box. The numbers displayed, will now be the range of values, across that range of traces, and any periodic increment will be indicated in square brackets
- **View the Data Traces** – This brings up a separate window displaying the seismic records for those traces being viewed (Figure 4). This can be particularly useful, for example, to work out the number of traces per Inline/Crossline/Gather, if it is unknown.

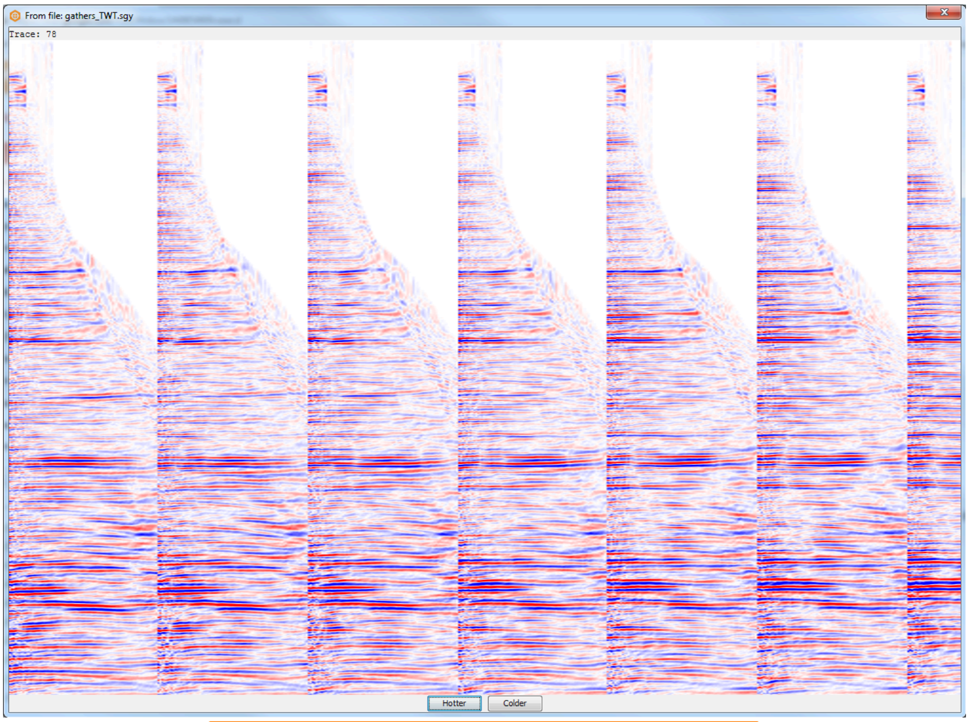


Figure 4: View Trace's window, trace number for cursor position displayed top left.

For further details of the controls and use of the Trace Headers window, click on the Help icon to refer to the specific documentation on this.

Once you are confident you have specified the correct byte locations for all the required information, the user can now progress to the next stage of the SEG-Y loading process.

Which SEG-Y formats are supported by the SEG-Y Loader?

SEG-Y is the only supported format for transferring 2D and 3D trace data into Insight.

The DUG SEG-Y Loader supports both SEG-Y Revision 0 (1975) and Revision 1 (2002).

The trace data format must be specified in the Data Sample Format Code. The codes are:

1 = 4-byte IBM floating-point

2 = 4-byte, twos complement integer

3 = 2-byte, twos complement integer

4 = 4-byte fixed-point with gain (NOT SUPPORTED)

5 = 4-byte IEEE floating-point

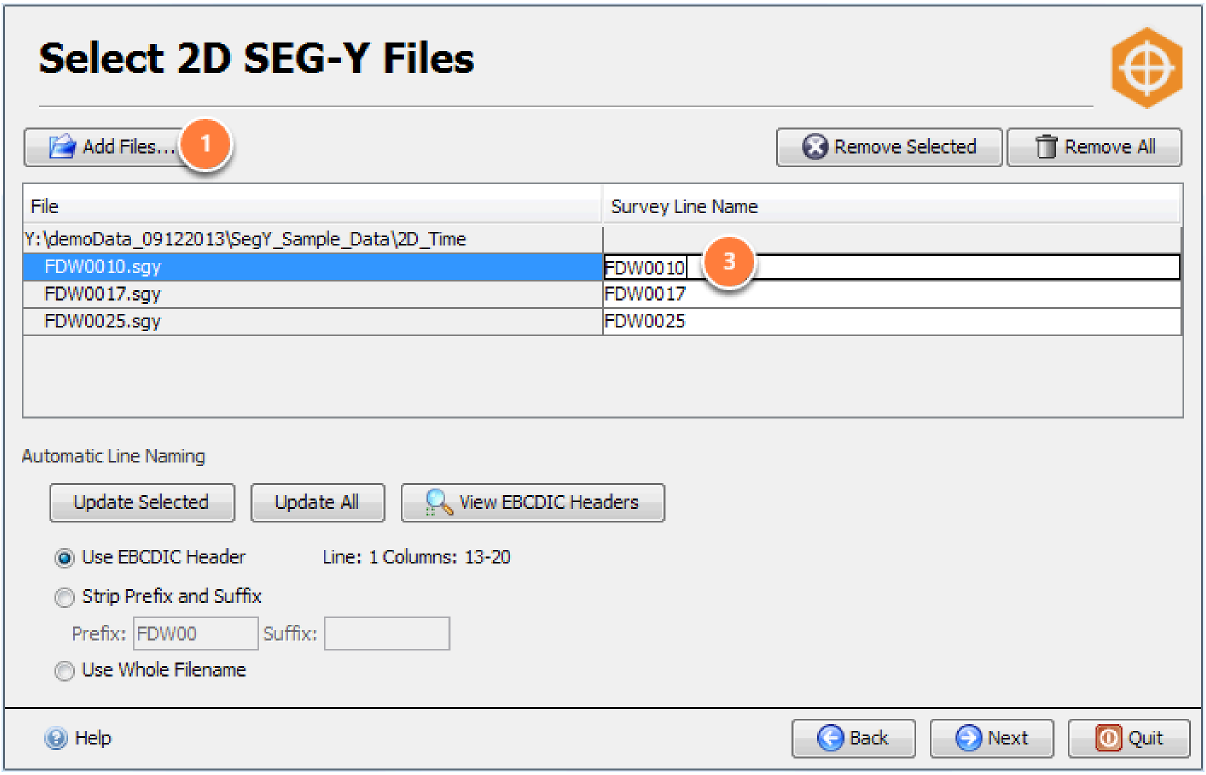
6 = Not currently used

7 = Not currently used

8 = 1-byte, twos complement integer

Insight does not support SEG-Y in 4-byte fixed-point with gain format.

Renaming lines in SEG-Y Loader



1. Click on **Add Files** to select the SEG-Y files to include in this import session.
2. By default, Insight will name the lines according to the file name minus ".segy".
3. Edit a survey line name by clicking on the line name. Do not include the data type in the line name.
4. Make sure that line names and files are correct.

How do I load multiple stacks or volumes from SEG-Y at the same time?

If you are trying to get to your data in a hurry (and you know your disks and network can handle it), you CAN load multiple volumes at the same time!

There are a few things you need to be aware of to make things easier.

2D Data

2D SEG-Y data is usually stored in one-line-per-file. An Insight 2D volume includes all the lines from the same seismic acquisition for a single type of data. For example, one Insight volume contains full stack data stored on several 2D lines, while a second volume contains velocity model data for the same 2D lines.

When you load 2D lines, you should include all the SEG-Y files for the same acquisition and type of data, at the same time.

Note: 2D volumes are flexible and allow you to revisit a volume to load data after the volume is first written.

3D Data

3D SEG-Y data can be extremely large, sometimes spanning several tapes or disks. In this case, large 3D surveys will be divided into several files.

When you load 3D data, you should include all the SEG-Y files for the same acquisition and type of data at the same time.

Note: 3D volumes will NOT let you append or overwrite data after the volume has been completed!

Surveys

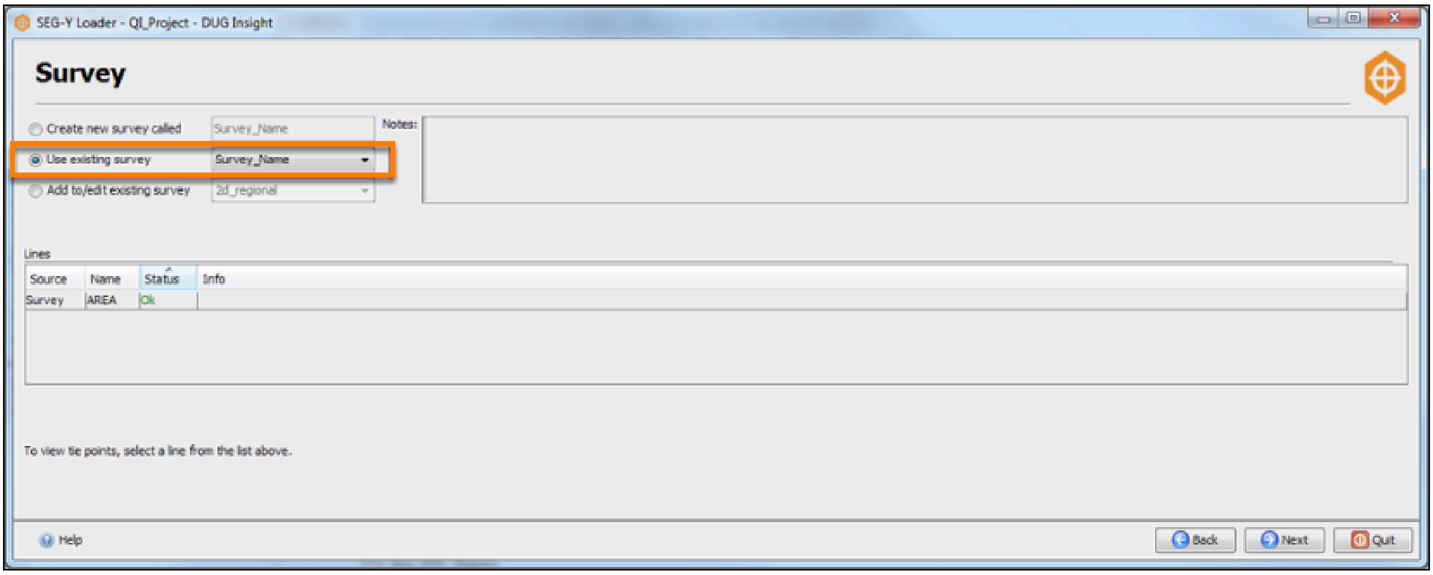
A survey refers to the navigational positioning of a dataset. Insight uses the survey to position data in the correct location.

In DUG Insight, a 3D survey defines the relationship between Inline/Crossline and X/Y. A 2D survey defines the relationship between CMP (or shot-point) and X/Y.

Tie points define the relationship between the survey coordinates and X/Y. Different combinations of tie points can give the same mathematical survey relationship. In these cases, Insight treats them as being the same survey. It will not let you load multiple separate surveys that are equivalent.

Example: Load full-stack and near-stack volumes for the same survey

1. Start SEG-Y Loader and configure it to load the full-stack data (see *SEG-Y Loader Overview*).
2. Create the survey if it does not already exist (see *Defining a 3D Survey*, *Defining a 2D survey* and *Loading External Navigation Files*).
3. Select the output configuration details and start writing (see *2D* and *3D Output and Writing Options*).
4. Once the full-stack volume has started writing to the *.dugio* volume file, return to the DUG Launcher and start a new instance of the SEG-Y Loader.
5. Configure the second instance to load the near-stack data.
6. Select the existing survey created when configuring the SEG-Y loader for the full-stack. Use the option **Use Existing Survey**.



7. Select the output configuration details and start writing.
8. The two .dugio volumes will be generated simultaneously.

9. You can start more instances of the SEG-Y loader for additional volumes, but be aware that the number of instances is limited by the capacity of your machine and the network.

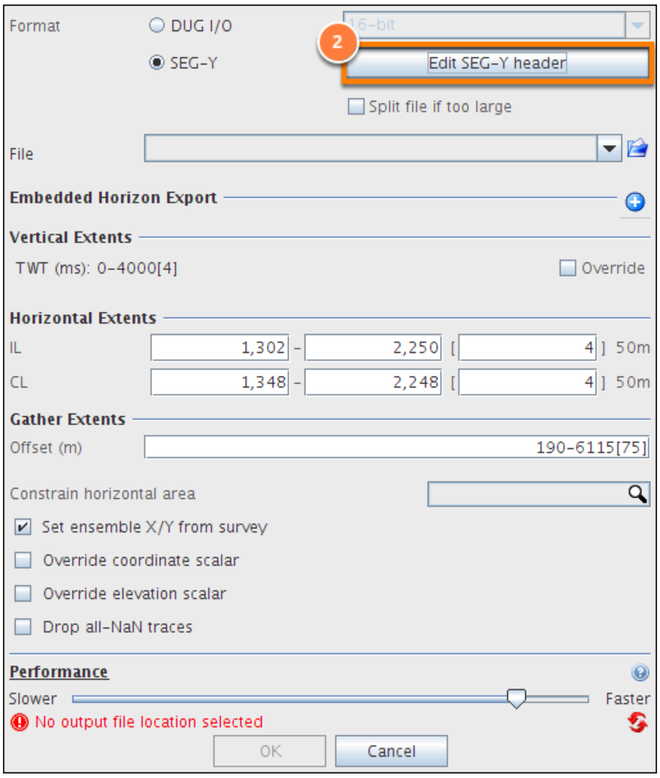
To view the volumes that you have loaded, see *How do I view multiple volumes at the same time?*

For more information on configuring the SEG-Y Loader, see *Importing 3D SEG-Y Data* and *Importing 2D SEG-Y Data.*

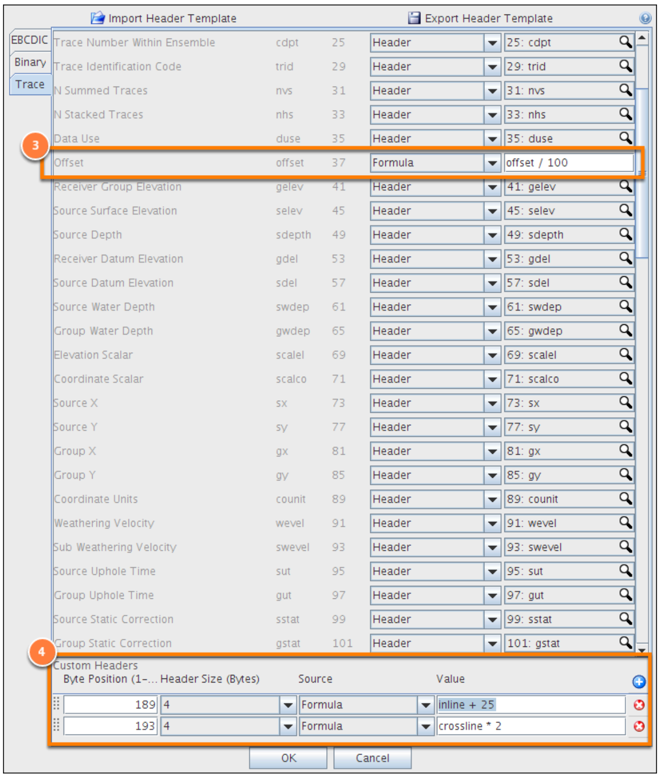
How do I update restricted, locked, protected trace headers like offset, inline and cmp?

Insight is careful to protect the headers that determine the location of a trace. This can be frustrating when trying to correct or reposition data that was loaded incorrectly.

One solution is to export the data to SEG-Y and update the trace headers as they are exported. Once exported, reimport the SEG-Y to a new volume with the updated headers.



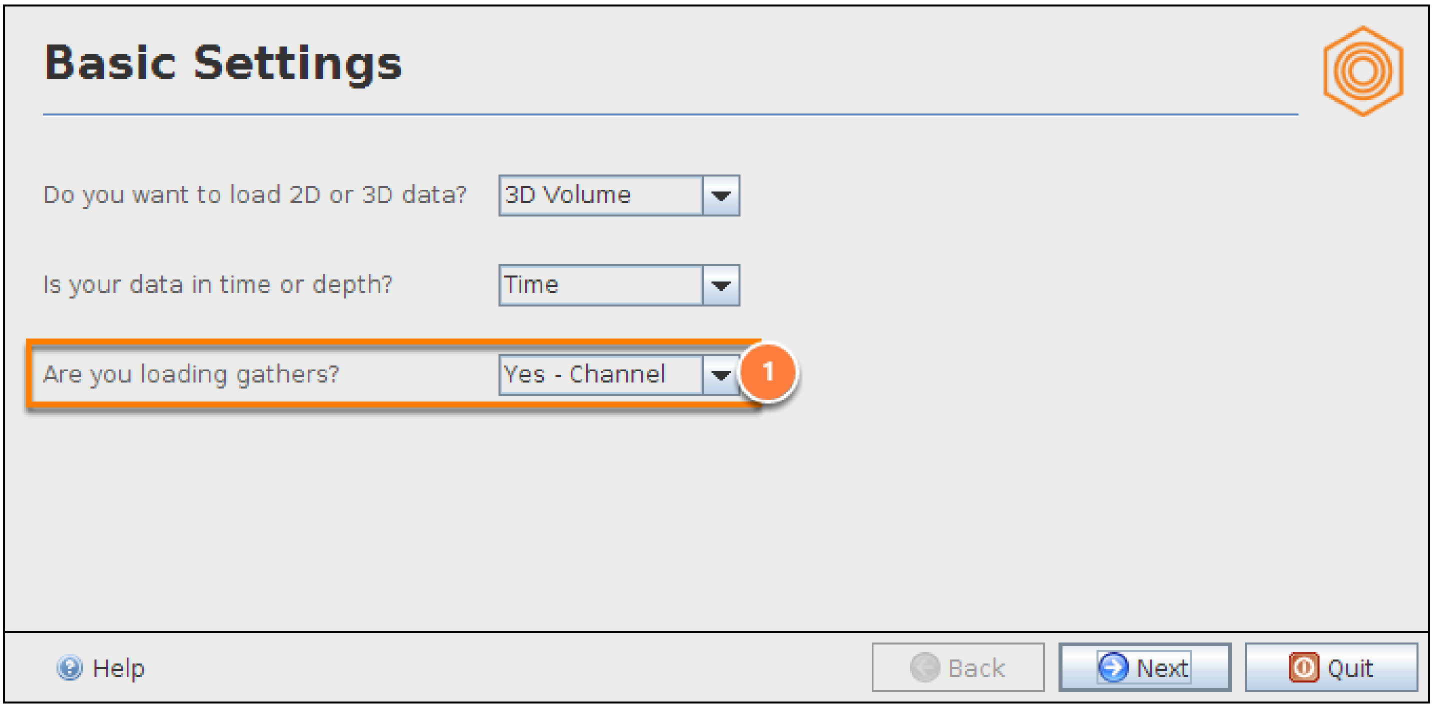
1. In the **Volume** tab, right click the required volume and click **Export...**.
2. Select **SEG-Y** as the **Format** and click the **Edit SEG-Y header** button.



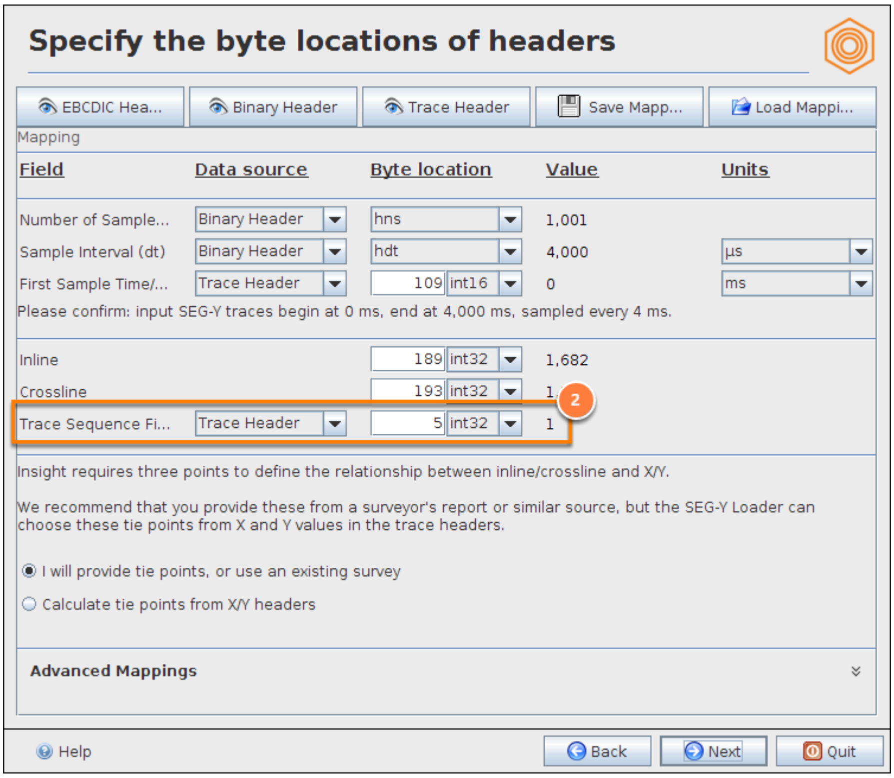
3. In this example, the offset header value in the SEG-Y file will be calculated using the formula:
   - offset / 100

4. Arbitrary custom headers (deviating from the SEG-Y standard) can also be specified as shown. In this case, the inline values will be shifted by 25 and the crossline header values will be doubled.

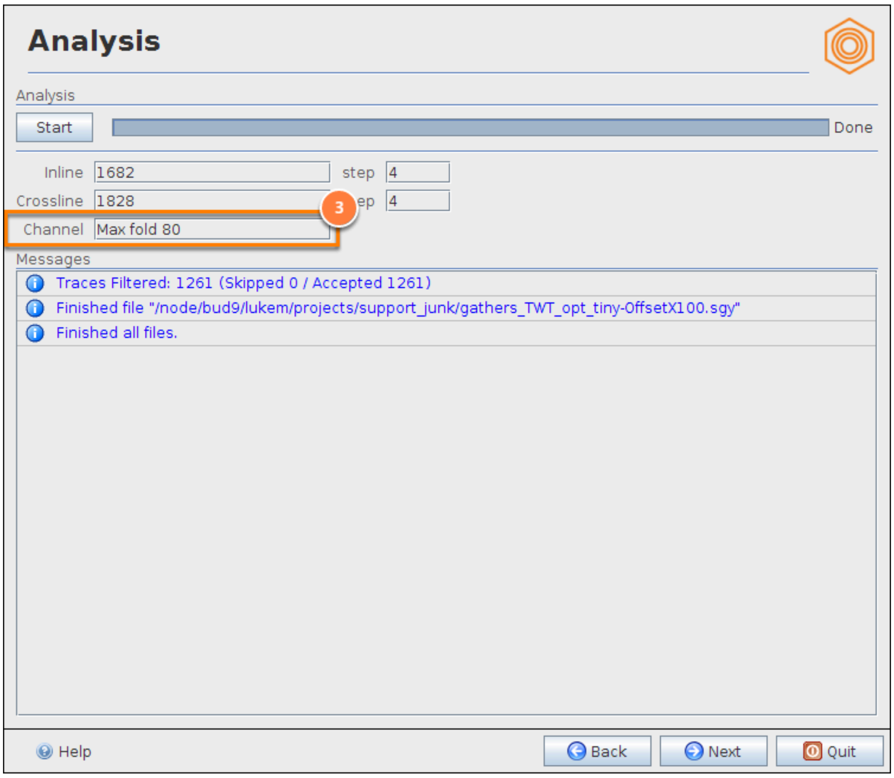
SEG-Y Loading Gathers with Offsets in Centimetres



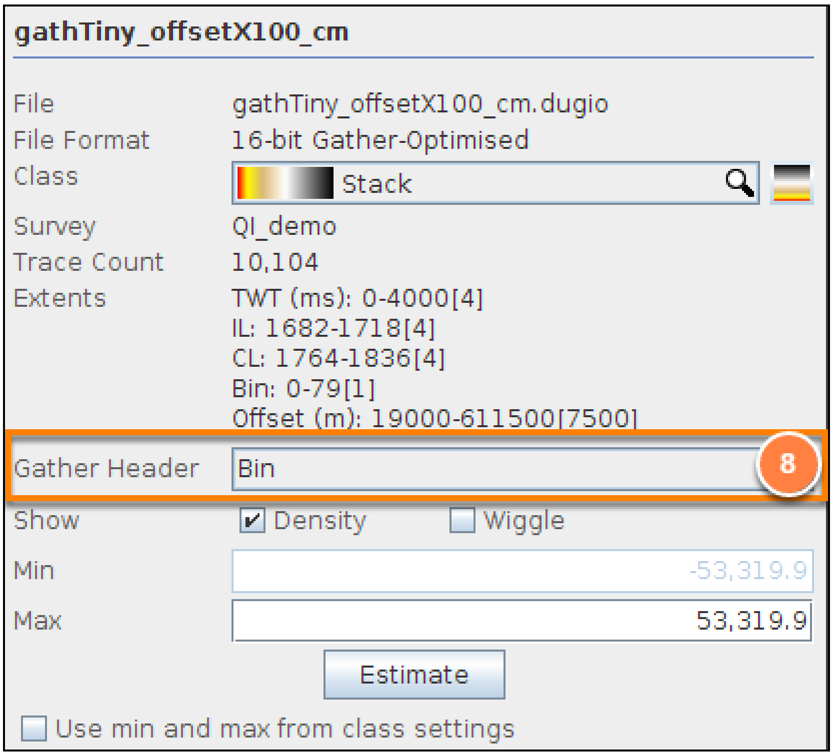
1. Run **SEGY-Y Loader** and load the SEG-Y as channel gathers.



2. Specify one of the headers as channel.
   - This example used the header at byte 5 (tracr)

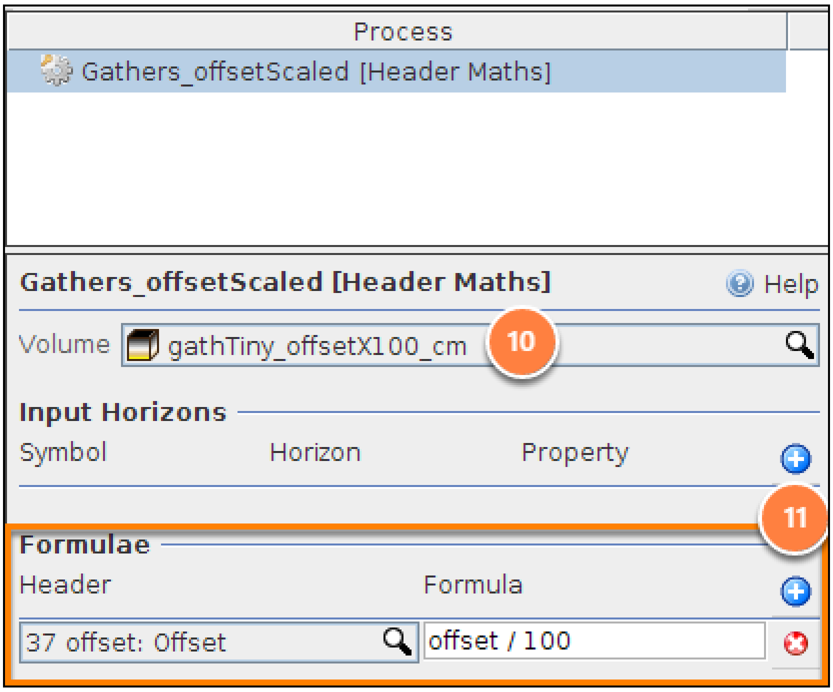


3. Check the **bin size**, make sure it is correct (e.g. "Max fold 80").
4. Write the result to a .dugio volume.

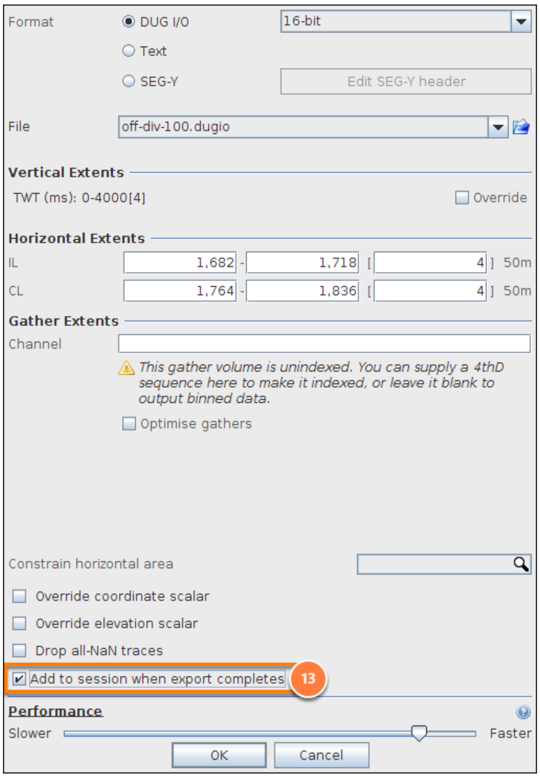


5. Start **Insight**.
6. *Add the volume* to the session.
7. Select the volume in the **Control Panel**.

8. Set the **Gather Header** as **Bin**.

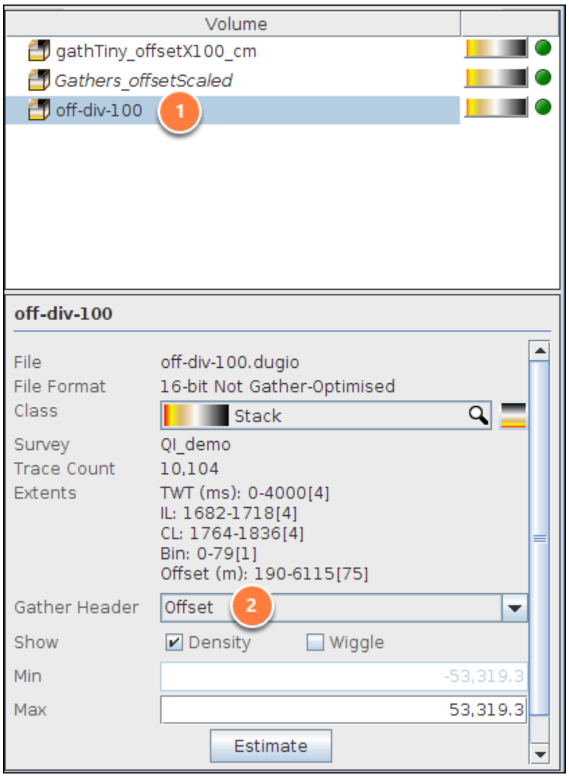


9. Create a [*Header Math process*](): "Gathers_offsetScaled"
10. Input **Volume**: select the added **Gathers**.
11. In **Formulae**, click the **blue "+" icon** to add a row:
    - **Header:** 37 offset
    - **Formula:** offset/100



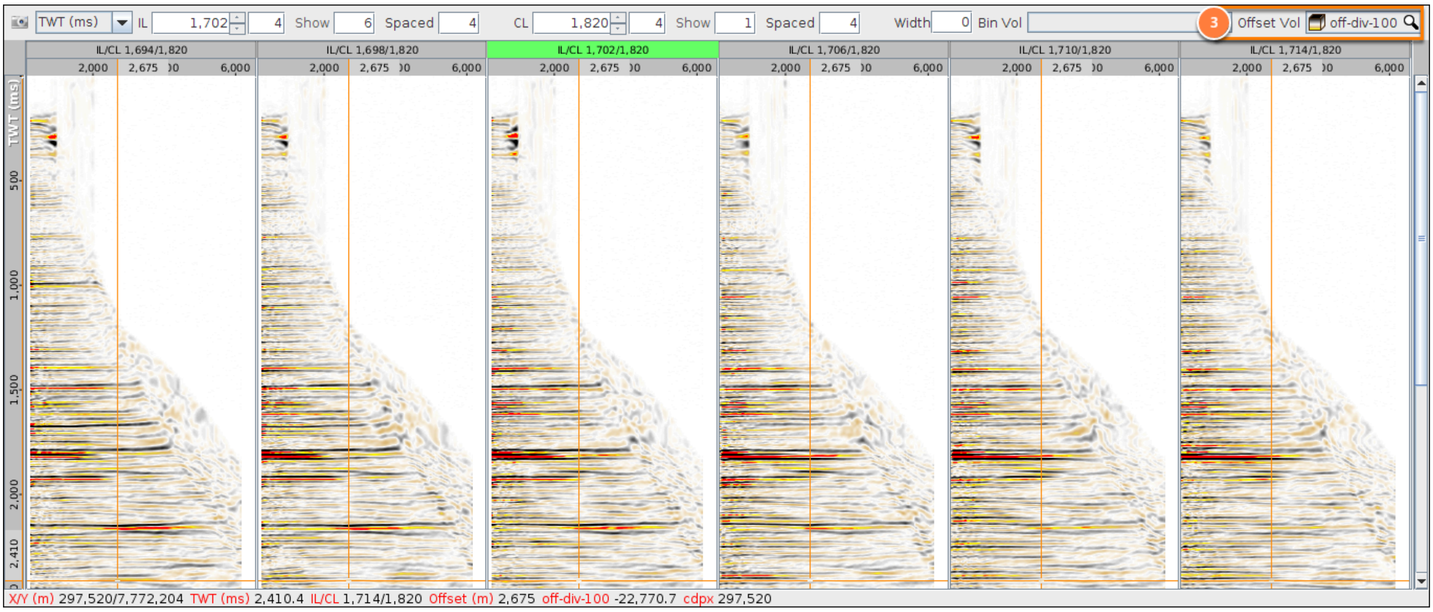
12. In the **Volume** tab, select the **Header Maths** result volume: "Gathers_offsetScaled"

13. *Export the volume* "Gathers_offsetScaled" as DUGIO.
    - Tick [**X**] the **Add to session when complete** option at the bottom.



After the export completes, the new volume will be added to the session. If it isn't, the "Add to session" option was not checked. Use the Import / Volume menu item in the Control Panel to add the exported dugio volume to the session.

1. Select the new **DUGIO** volume.
2. For **Gather Header**, choose **Offset**.
    - The offset range should now be correct.



3. *View the gathers*
    - **Offset Vol**: Select the new volume.
4. If the original unscaled gathers are in the session, the Offset axis will be huge (i.e. 100 times larger than necessary). To correct the display, remove the original gathers from the session or zoom in to the left edge of the offset gathers.

Load SEG-Y containing traces with varying start times (DELRT)

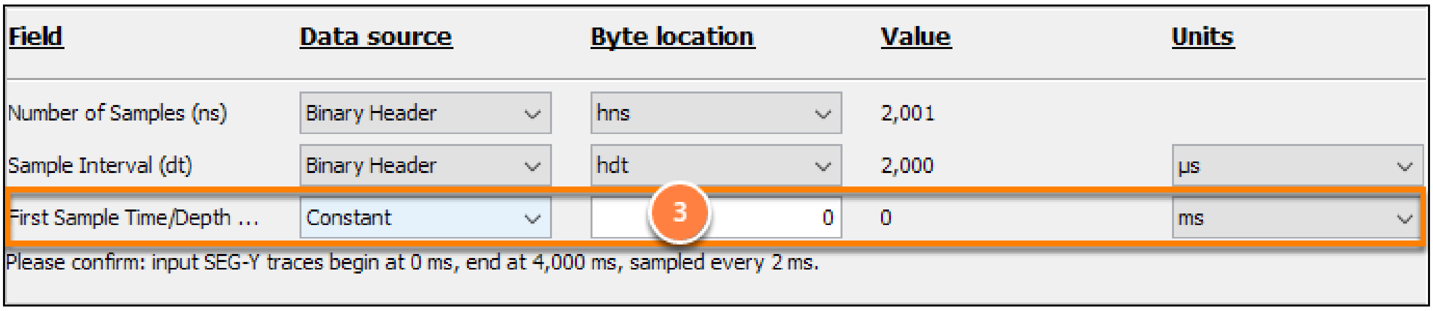
To load SEG-Y data containing traces with varying start times:

1. During loading
   - Load the data with a constant start time
   - Store the variable start time in a trace header
2. After loading, apply a trace shift process to reposition the traces

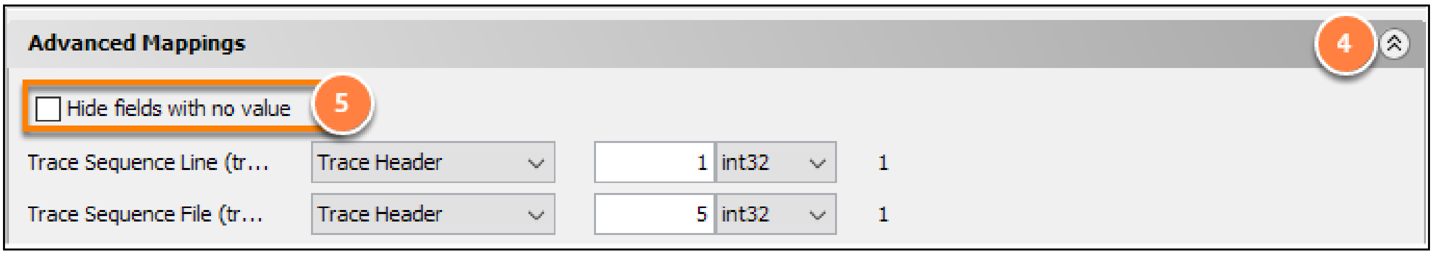
Workflow

In SEG-Y Loader

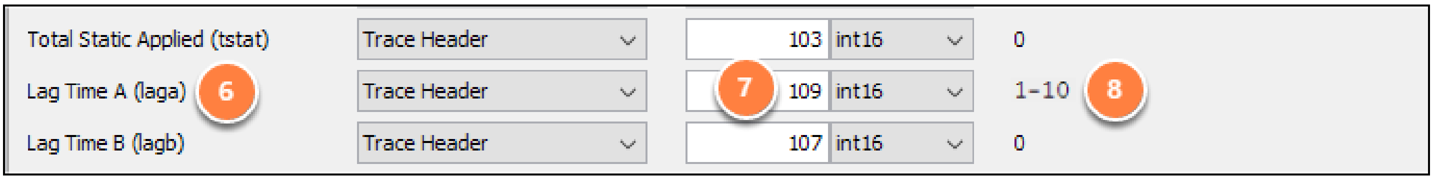
1. Start *SEGY Loader*.
2. Configure and select files as you would normally.



3. In the **"Specify byte locations of Headers"** window:
   - At **"First Sample Time/Depth"** choose **Constant**, with a value of 0.

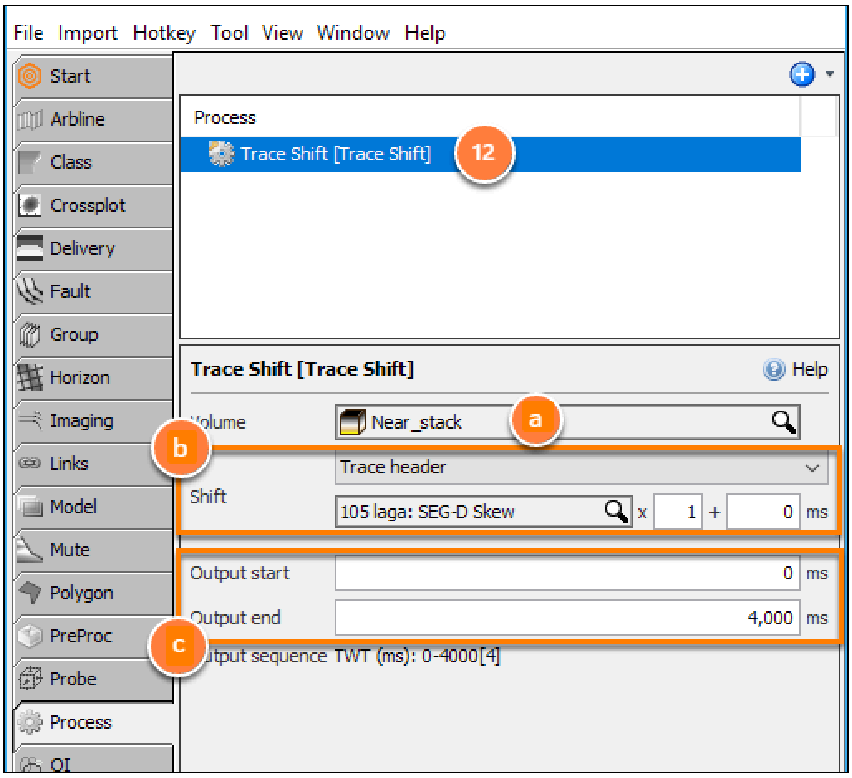


4. Expand the **Advanced Mappings** panel.
5. Uncheck the **"Hide fields with no value"** box.



6. Choose an unused header, e.g. "laga"
7. Enter the byte location for the shift data, usually the DELRT header: 109
8. Confirm that the value range is sensible.
9. Complete the SEG-Y load and output the dugio volume.

In Insight



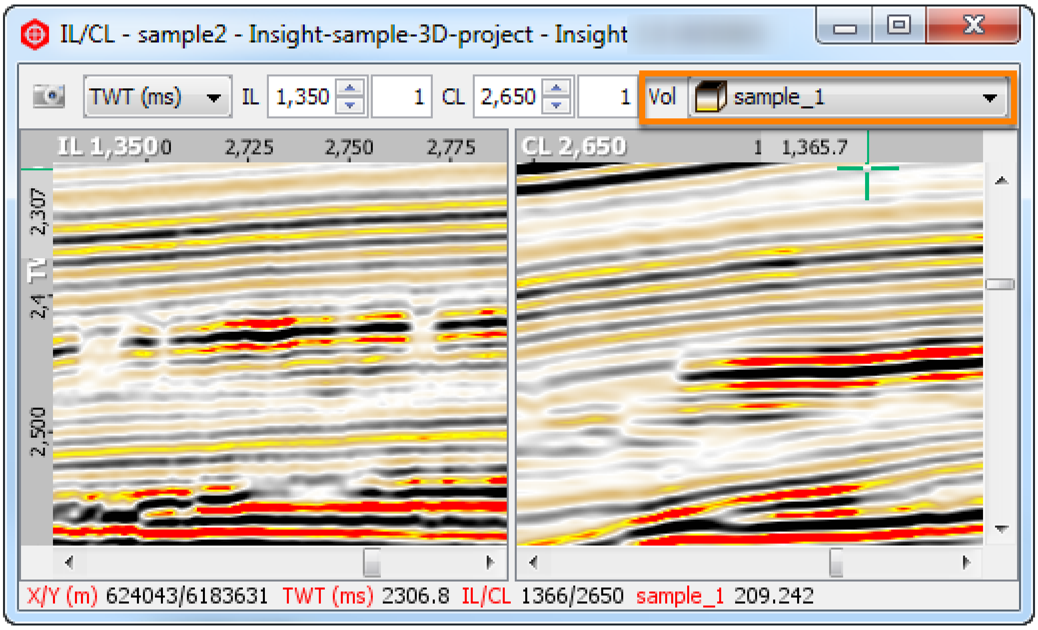
10. Open the project
11. Add the new volume to the session.
12. Create a Trace Shift process
    a. **Volume:** Select the loaded, unshifted volume
    b. **Shift:** Choose "Trace Header" and select the advanced mappings header containing the start time, e.g. laga.
    c. Update the output start and end values to include the full length of the shifted traces.

Views and Display

My data loaded successfully. Why can't I see anything?

After loading your data, first make sure to open an appropriate view.

IL/CL view

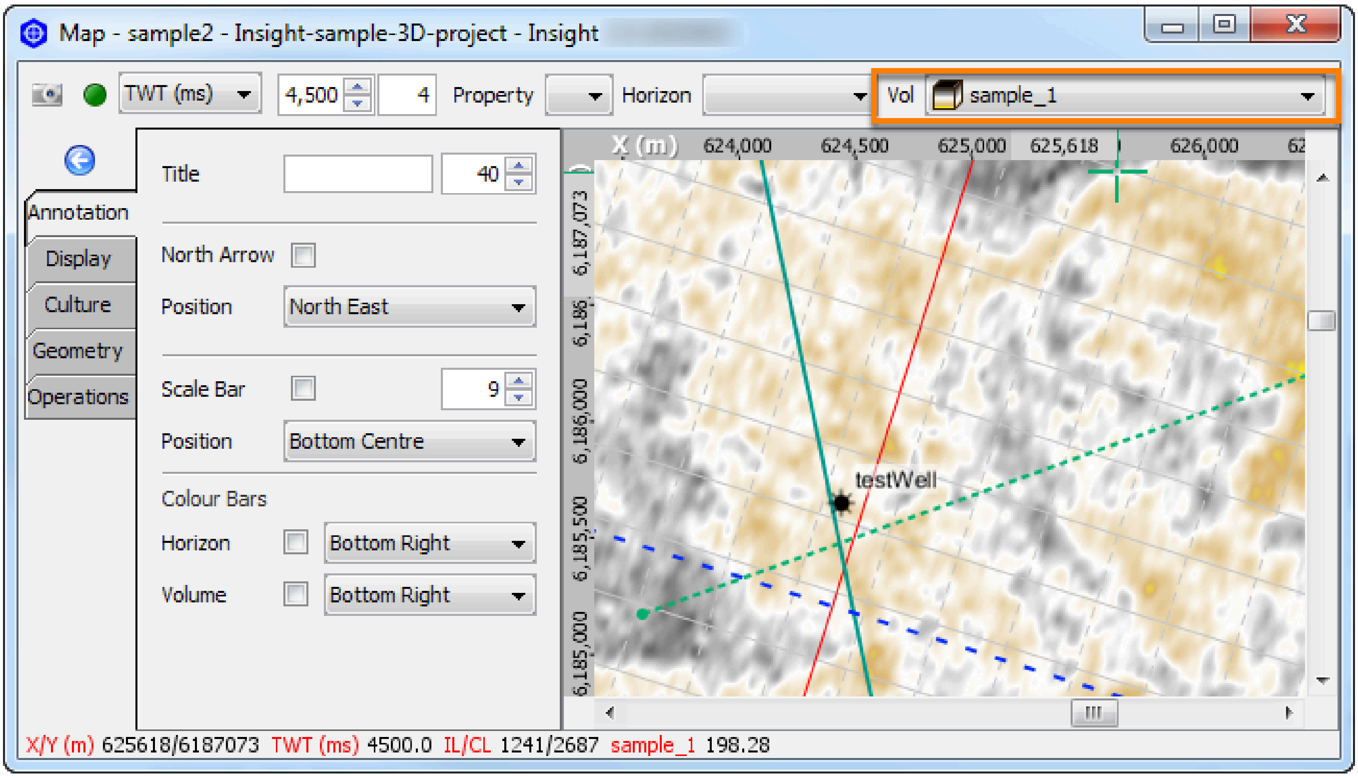


From the Control Panel, open the **View** menu and select **IL/CL View** for inline and crossline cross-section views of data volumes.

In the upper right-hand corner of the window there is a volume selector drop-box. Make sure to choose the volume that you wish to view, and that you are viewing an inline or crossline that contains data. If your volume is not appearing in the drop-down list, check that the circle beside the volume in the **Volume** tab is **green**. If the circle is **amber**, click on it to turn it to **green** to enable the volume.

Tip: You can click on the volume in the **Volume** tab to display the Details Panel. In the Details Panel, check that the IL and CL in the **IL/CL View** are within the IL and CL extents of the volume.

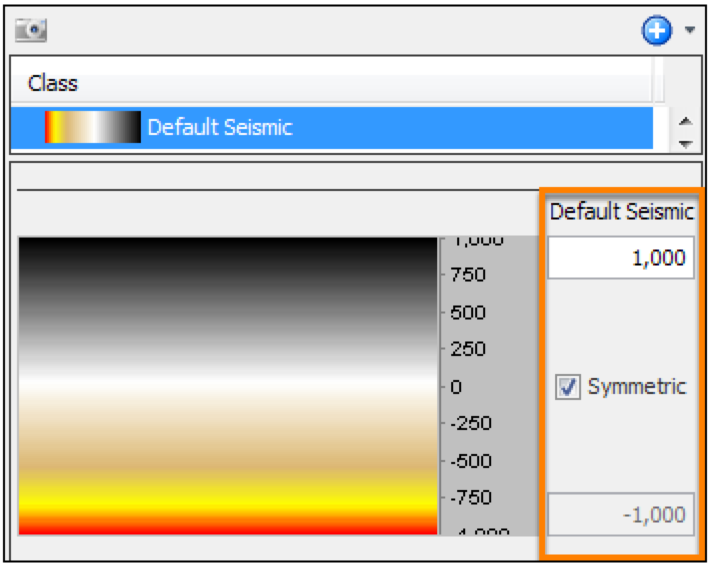
Map view



If you wish to see time or depth slices, select **Map View** from the **View** menu.

In the upper right-hand corner of the window there is a volume selector drop-box. Make sure to choose the volume that you wish to view. If the volume is not appearing in the drop-down list, check that the circle beside the volume in the **Volume** tab is **green**. If the circle is **amber**, click on it to turn it to **green** to enable the volume.

Class settings



If you have a view window open and you are still unable to see your data, you may need to adjust your class settings. In the **Volume** tab, open the Details Panel for your volume and look at which class it is using (the default class is called "Default Seismic"). In the **Class** tab, open the Details Panel for the class. Examine the maximum and minimum clipping values for your colourbar and make sure they are correct for your data. If the values are far too large, your data will appear as a single colour, often white.

For more information about Class Settings, see *Classes and Colourbars*.

Why is the 3D view not opening / giving errors / crashing?

Help, something is wrong with the 3D View!

If you are having a problem with the Insight 3D view, such as:

- a message warning of insufficient resources appears when opening the 3D view
- the software freezing or crashing when opening or using the 3D View, or
- OpenGL errors at any time,

then you have come to the right place.

Executive Summary

99.9% of the time, problems with the 3D view are caused by buggy, out-of-date video drivers, and not Insight itself.

Almost all modern hardware is capable of running Insight's 3D view when paired with modern drivers.

If you have an NVIDIA or ATI/AMD graphics card made after 2007, then upgrading to the latest drivers (available from http://nvidia.com or http://ati.com) is almost guaranteed to resolve your problem.

Read on for more details.

Basic Requirements

The Insight 3D view requires a video system that supports at least OpenGL 2.0. As OpenGL 2.0 was released in 2004 (the latest version is 4.2), every vendor on Earth claims to support it.

Intel

Unfortunately, not all such claims are equal. If you have an Intel display device, its OpenGL 2.0 support is probably not good enough to run Insight's 3D view.

Many modern laptops include two display devices: a low-power, low-capability Intel chip, and a high-capability NVIDIA or ATI/AMD. If yours is like this, it should automatically switch to the high-performance chip when Insight launches, but may not. Check the Control Panel (Windows) or the Energy Saver panel (Mac OS) for this setting.

NVIDIA / ATI / AMD

If you have any display device made since 2007 by NVIDIA or ATI/AMD, then it will almost certainly work when paired with modern device drivers.

3D Display Drivers

Because video drivers run at the very lowest level of the system, bugs in video drivers can cause virtually infinite problems, even those that seem unrelated to 3D.

As Insight is modern, demanding software, it is very good at exposing problems in video drivers. 99.9% of all problems that users experience with the Insight 3D view are not the fault of Insight, but rather the video drivers. Upgrading to the latest drivers resolves the issue in all cases.

Because new drivers are released so often, IT departments rarely keep them up-to-date unless there is a specific problem. It is common to find new computers with drivers that are 2 to 3 years old!

Please install the latest version of the drivers for your device from either http://nvidia.com or http://ati.com.

> **Note:** In general, the latest versions of the drivers from Microsoft or your system vendor are not good enough. Those versions often lag months or years behind those released by NVIDIA or ATI/AMD. You need to visit the NVIDIA or ATI/AMD site to get the actual latest versions.

Remote Desktop

Some remote-desktop/terminal-server products (such as Citrix, VNC, etc) provide proper support for 3D graphics, and some do not. Sometimes it is a matter of configuration.

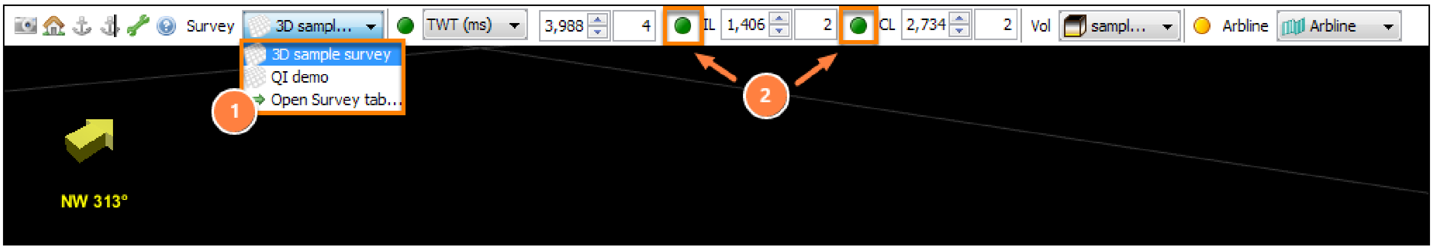
We will certainly do our best to help, but issues with those products will likely have to be referred to the vendor.

Still Having Trouble?

We want to help! Please contact support for immediate assistance, and let us know what you have already tried.

Why are there extra sections/lines showing up in 3D View, and how do I remove them?

If you are dealing with multiple surveys and notice some unwanted seismic sections or lines in your 3D View, it is likely that they belong to other surveys in your session.



Each survey has its own independently-controllable inline (IL) and crossline (CL) slice in the 3D View. To turn them off:

1. Select the survey in the drop-down box at the top left.
2. Click on the **green** circle beside the **IL** and **CL** to switch off the IL and CL slice for that particular survey (**green** is on, **amber** is off).
3. Repeat this step for *every* survey you wish to disable in the 3D View.

Note: Disabling the surveys in the Control Panel does NOT remove the section/line from the 3D View, it simply stops displaying the survey grid lines.

"Zoom to fit" makes everything tiny and the zoom detail and scaling level is limited

When you open 3D or any section views, if you notice that the data is zoomed very far out, or the horizontal and vertical scales are unusually large, there might be something in your session that was imported with the wrong units or on the wrong survey.

To figure out what is causing the display to be skewed, the **Product Extents Table** will allow you to check all the extents for all the products in your session.

Product Extents Table

To open the **Products Extents Table**, click on the **View** menu from the Control Panel and select **Product Extents Table**.

Product Extents Table - AO - crossplots - QI_project - Insight

Name	Survey	X		Y		IL		CL		TWT		TVDSS		Offset	
		min	max	min	max	min	max	min	max	min	max	min	max	min	max
QI demo	QI demo	290,395	304,146	7,766,303	7,780,155	1,242	2,350	1,250	2,350						
South Australia 2D	South Australia 2D	574,721	851,734	6,040,737	6,406,567										
VpVs	QI demo	291,626	302,901	7,767,135	7,778,910	1,309	2,250	1,349	2,250			3,000	4,500		
P_Impedance	QI demo	291,626	302,901	7,767,135	7,778,910	1,309	2,250	1,349	2,250			3,000	4,500		
gath	QI demo	295,995	298,595	7,771,979	7,774,579	1,698	1,902	1,700	1,904	0	4,000			190	6,115
2D_W00FDW	South Australia 2D									0	9,000				
acmeQI_001_vel...	QI demo	290,270	304,270	7,766,179	7,780,179	1,242	2,342	1,250	2,350	0	5,600				
gen1	QI demo	291,626	302,901	7,767,123	7,778,910	1,308	2,250	1,349	2,250	1,148.56	1,807.95				
gen2	QI demo	291,626	302,901	7,767,123	7,778,910	1,308	2,250	1,349	2,250	1,752.8	2,573.01				
fault		297,270	297,270	7,775,221	7,776,940					1,469.47	1,874.88				
well1		292,637	293,400	7,774,055	7,774,411					2,421.75	3,003.55	-22.4	5,769.87		
well6		295,145	295,271	7,772,151	7,772,167					2,449.75	2,847.25	-22	5,970.1		
probe		294,270	297,758	7,770,879	7,773,417	1,608	1,811	1,560	1,839			3,164	3,325		
Polygon		295,633	296,908	7,772,992	7,772,992	1,777	1,777	1,669	1,771			935	6,129		

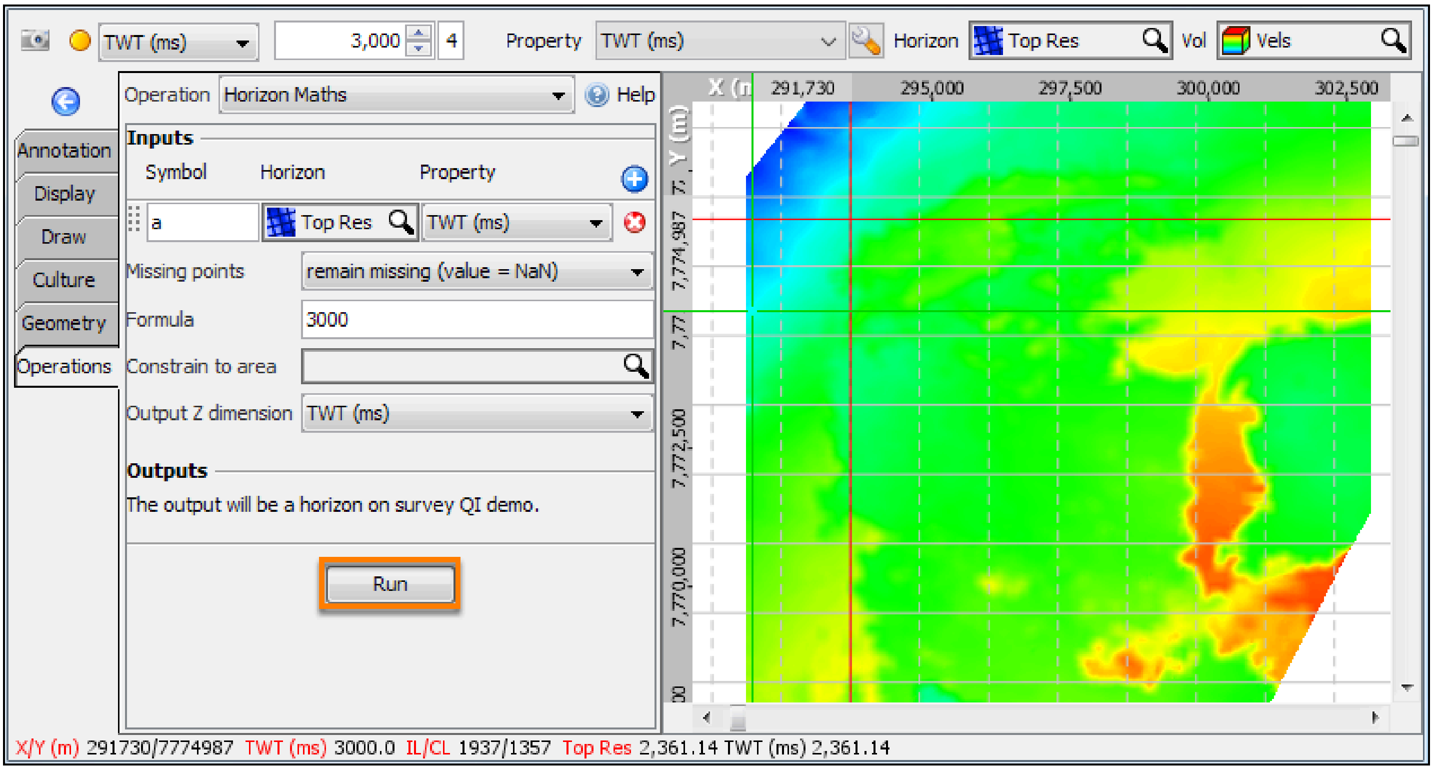
From the example above, you can see that the X and Y extents for the 2D survey is the reason for the skewed display. Remove, or re-import the anomaly with the correct units, to fix the display.

Tip: Click on any table header to sort the list according to that column in alphabetical order (for text columns) or ascending/descending order (for numerical columns).

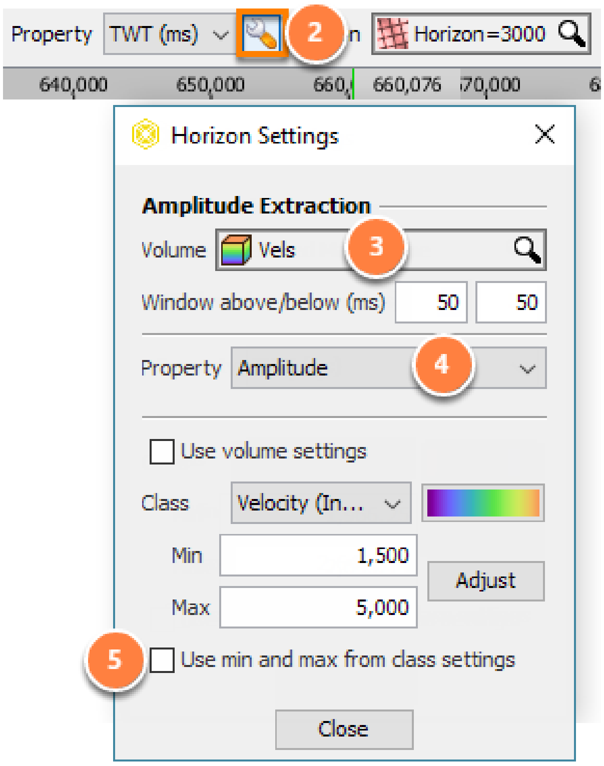
Contours Based on Amplitude Extraction

In addition to displaying contours on the map, you can use this feature to combine the display of contours with amplitude extraction. The following shows an example of a workflow to display contours based on a velocity volume at the time slice of 3,000 ms.

Display contours based on amplitude extraction



Part 1: Create a flat horizon at 3,000 ms time slice (see *How do I create a flat horizon?*)



Part 2: Select velocity volume for horizon

1. In the Map View, select "Horizon=3000" in the navigation bar.
2. Click on the spanner icon beside Property. The Horizon Settings window will appear.
3. In the Details Panel of the 'Horizon=3000' horizon, select the velocity volume at **Volume**. The contours will be generated from the selected velocity volume.
   Note: To show amplitudes from a different volume, select another volume at **Volume**.
4. Select **Amplitude** at **Property**.
5. Clear the **Use min and max from class settings** check box and then click **Adjust** to adjust the minimum and maximum clip of the class settings. This will make the best use of the colour bar in displaying the horizon.

Part 3: Display contours for "Horizon=3000" based on amplitude property

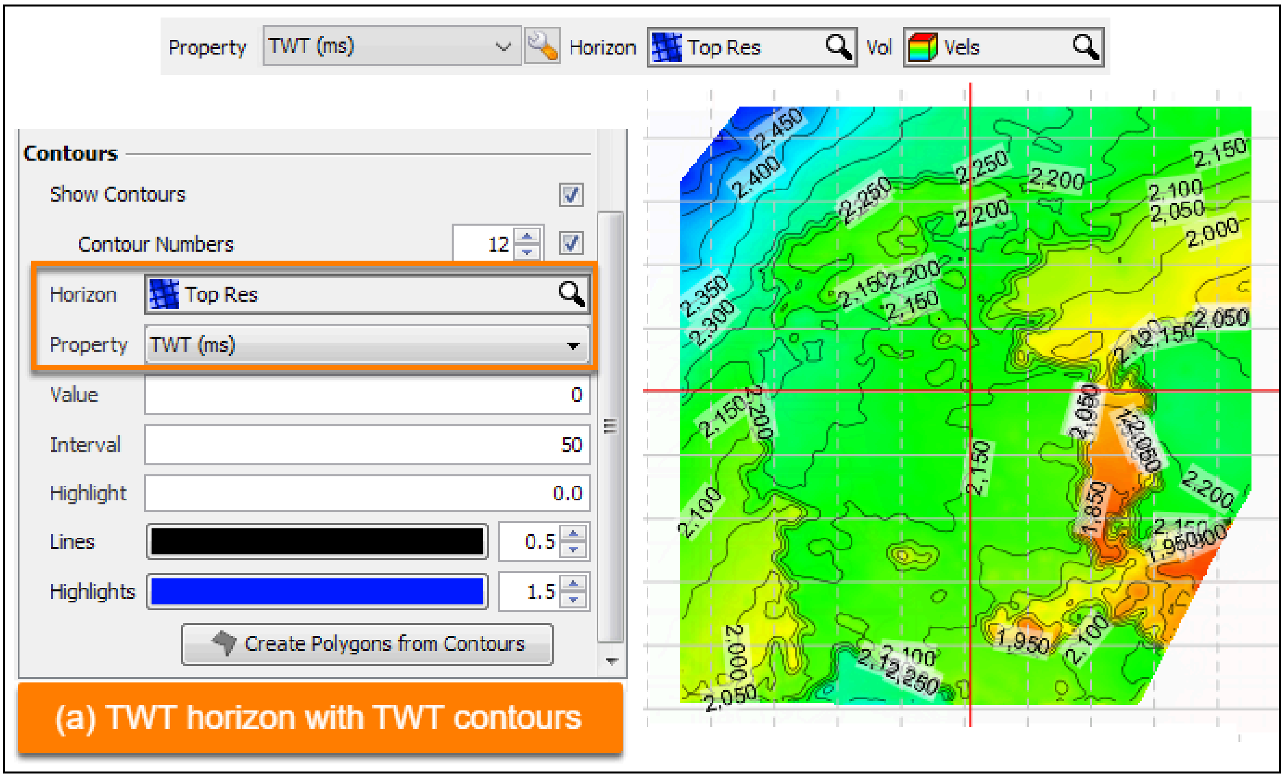
1. At the navigation bar in the **Map View**, select 'Horizon=3000' at **Horizon**.
2. Expand the left panel and open the **Display** tab.
3. In the **Contours** section, select the **Show Contours** check box to display the contours on the map.
4. Select the **Contour Numbers** check box to display the contour values on the map. Type the font size for the contour values.
5. At **Horizon**, select "Horizon=3000".
6. At **Property**, select **Amplitude**.

See the following examples:

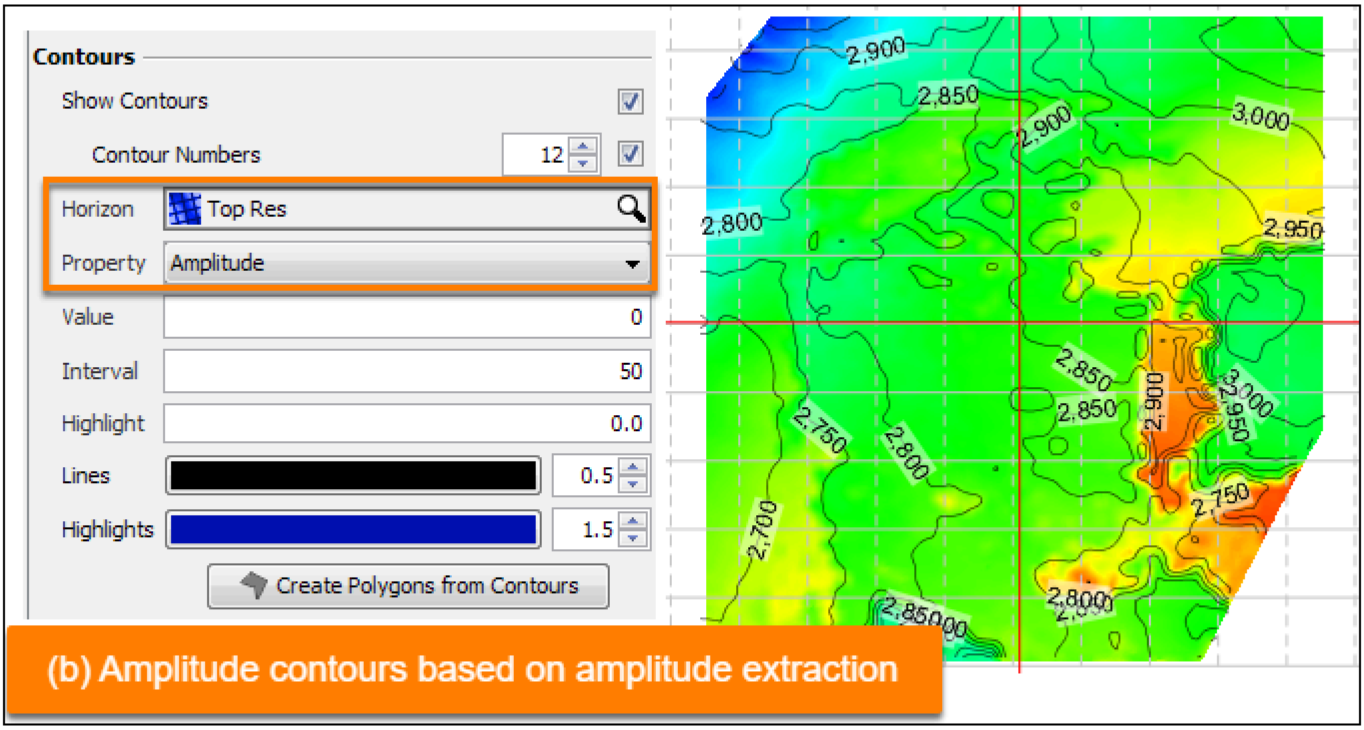
a) Display time contours for a TWT horizon by selecting the TWT

property for the "Top Res" horizon.

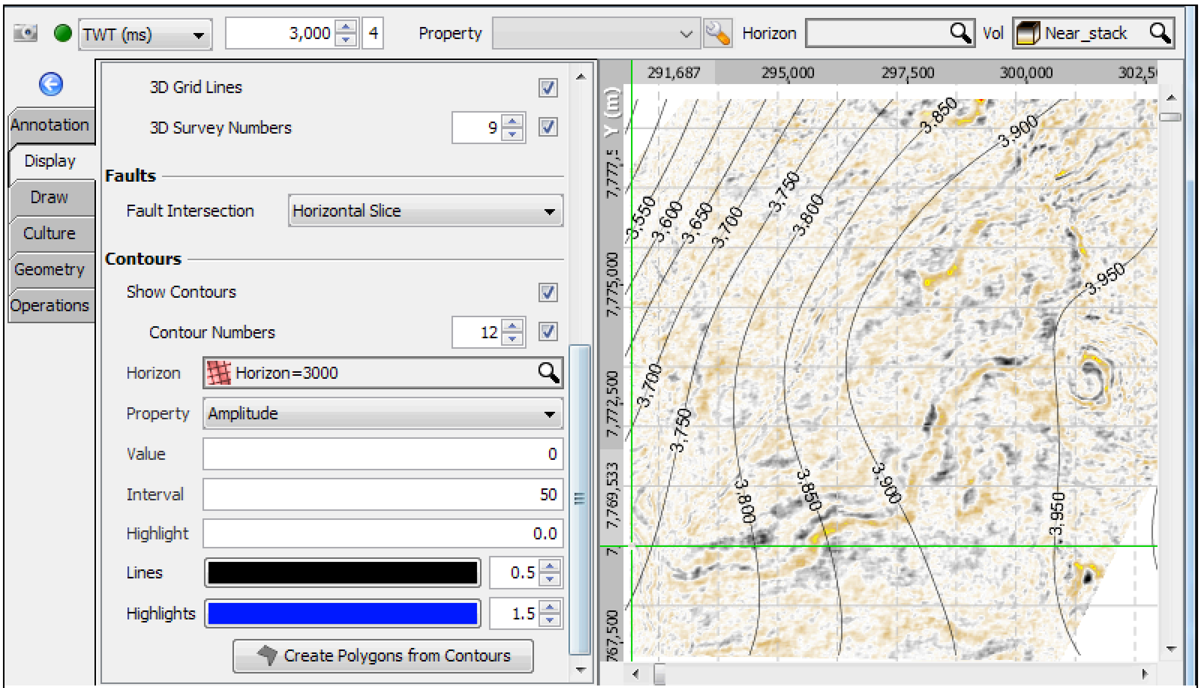
Note: If a velocity model has been specified for the Time-Depth parameters (see *Converting Time-Depth*), depth contours could be displayed instead.



b) Display contours for extracted amplitudes on a TWT horizon by selecting the Amplitude property for the "Top Res" horizon.



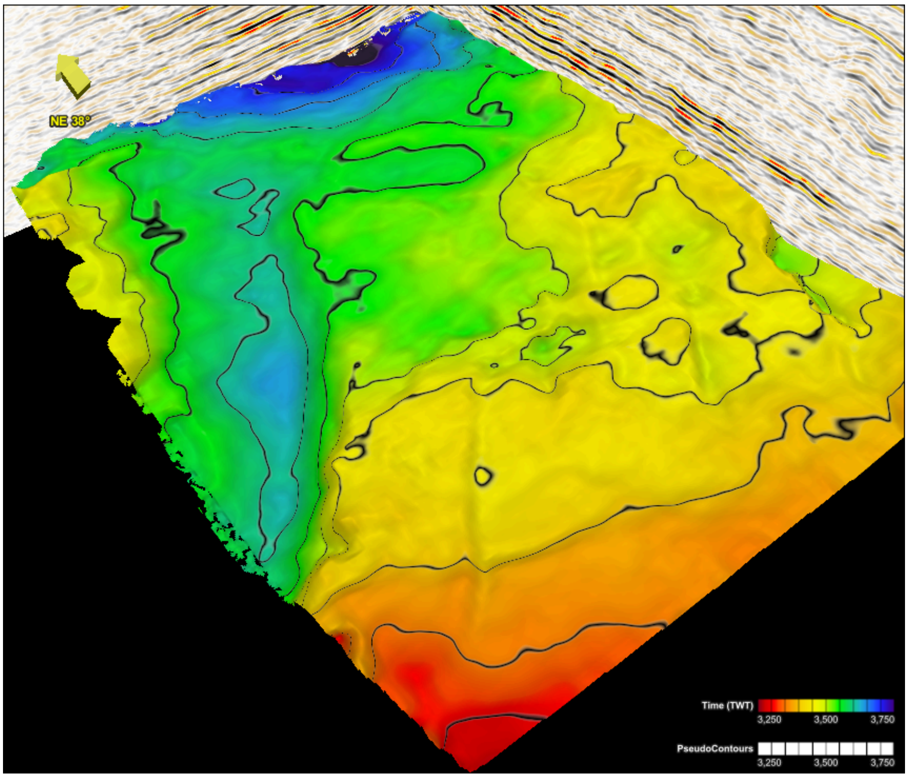
c) Display contours for extracted amplitudes at a constant time from one volume, while showing the corresponding slice from a stack by using this workflow.



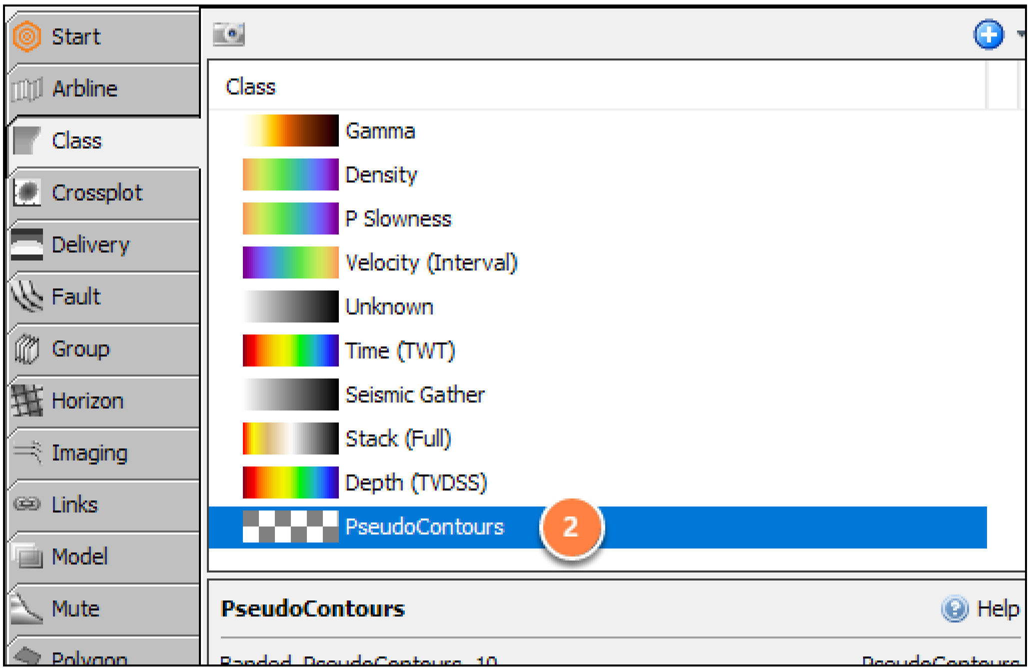
(c) Stack slice with amplitude contours

How do I display a horizon with its contours in 3D View?

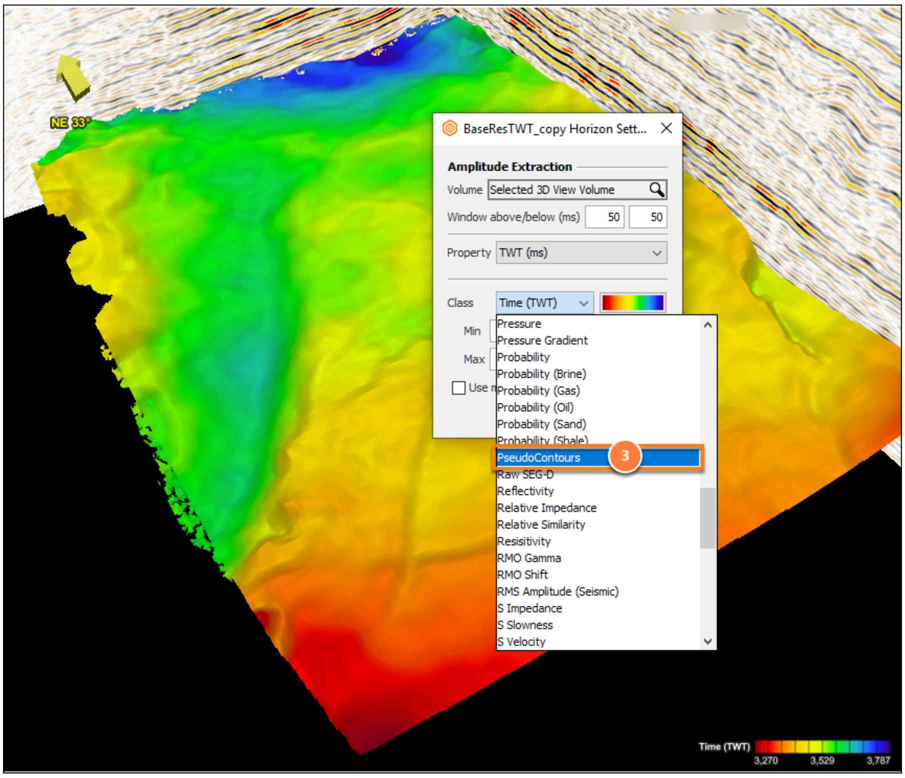
Contours are generated in the Map View (see *Contouring in Map View*). We can simulate contours in 3D View by creating pseudo-contours.



1. Make a copy of the horizon you wish to contour (see *Copying a Horizon*).
2. Create a new Class (see *Creating a Class*) and import the "banded_PseudoContours_10.ducolbar" coloubar from DUG-Insight’s default list of colourbars (see *Importing a Colourbar*).

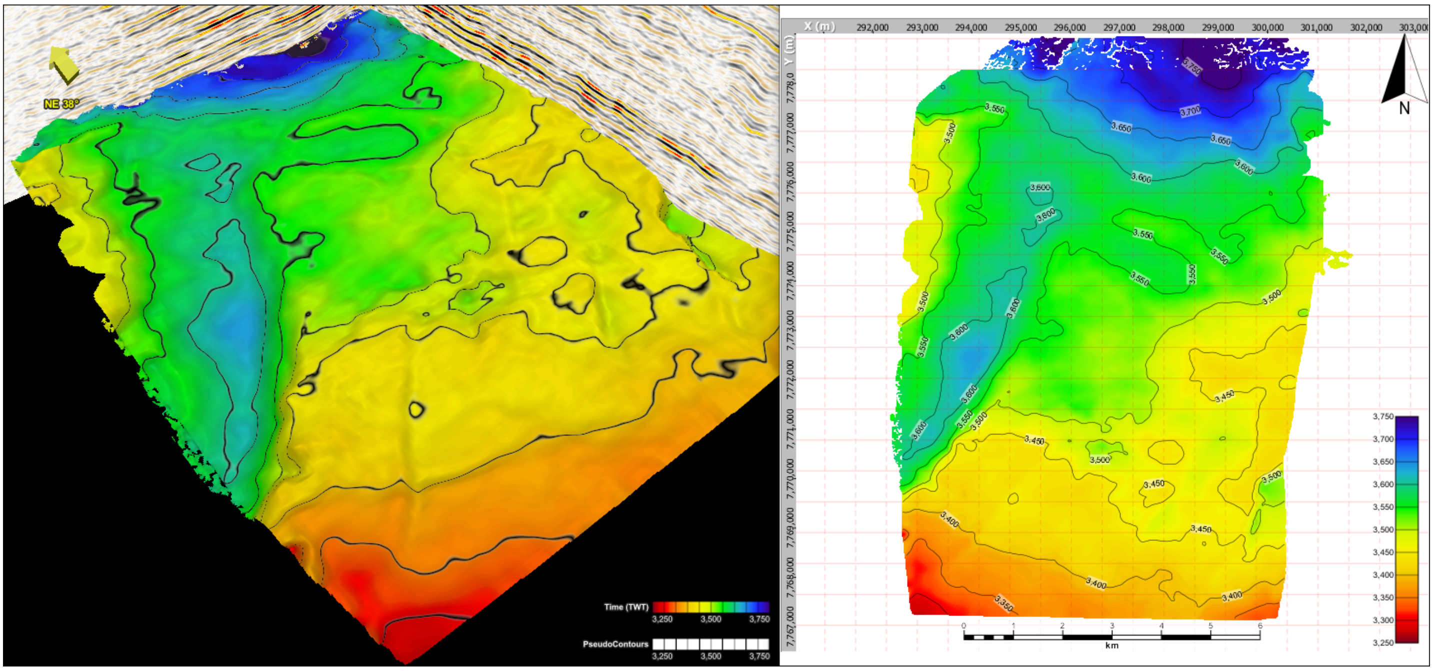


3. Change the class of the **Horizon Copy** to the new class (PseudoContours).



4. Turn on both horizons as surfaces in the **3D View**.
5. The copied horizon's banded colourbar is largely transparent, you can still see the original horizon, but with the discrete black lines overlaid.
6. Manually adjust the min/max cut-off values of the copy so that the black lines fall on exact values. The colourbar will be divided into 10 bands between the min and max values.

7. For example, in the following image, the difference between the min and max values is 500, so every 50 will be black lines (interval = 50).



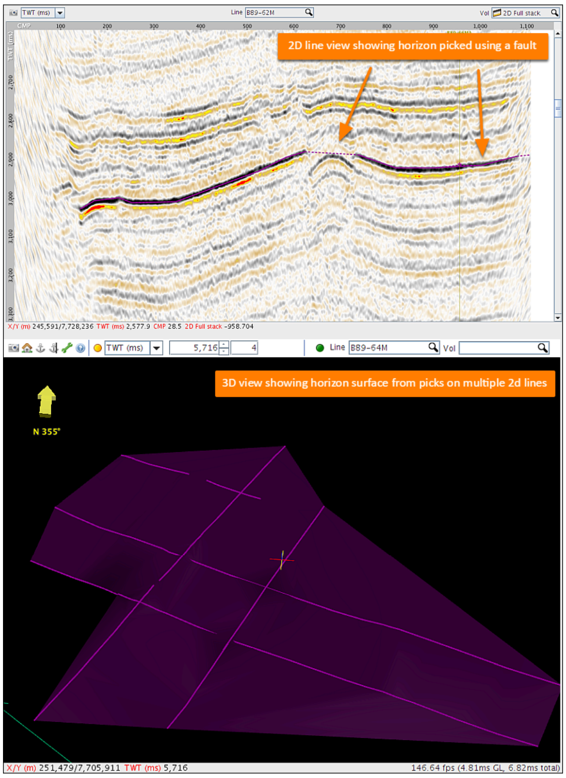
How do I pick contours in Insight?

You can't pick contours directly, but there is a workaround that you can use. If you pick the horizon as a fault, you can use the map view to create the contour lines as fault picks which you can manually pick.

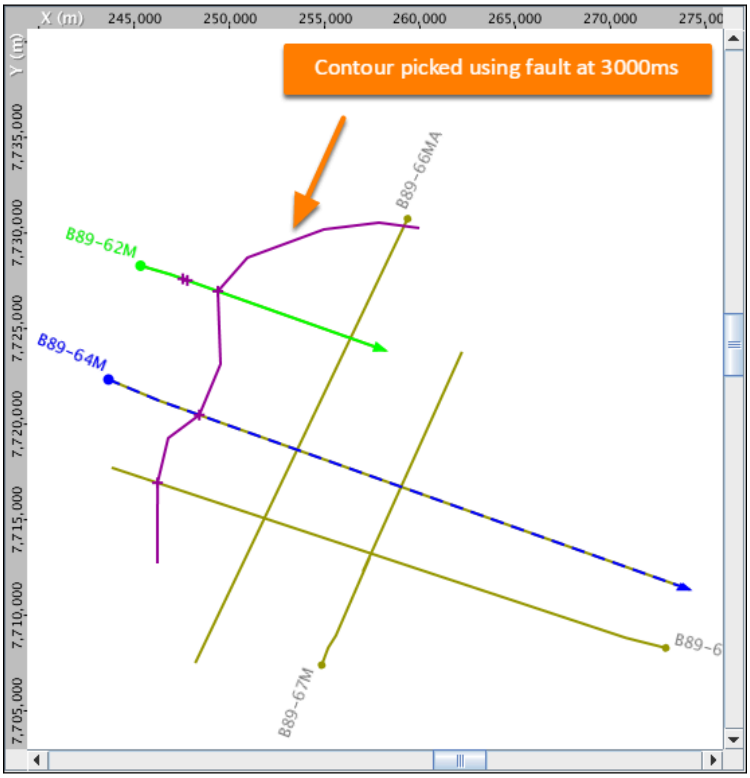
Depending on the picks, the fault surface may not display sensibly, but the sticks will contain all the necessary points to create a good horizon.

To do it, export the fault to a file -- make a small change, and import the result as a horizon. Regrid to X/Y and extrapolate to complete the surface.

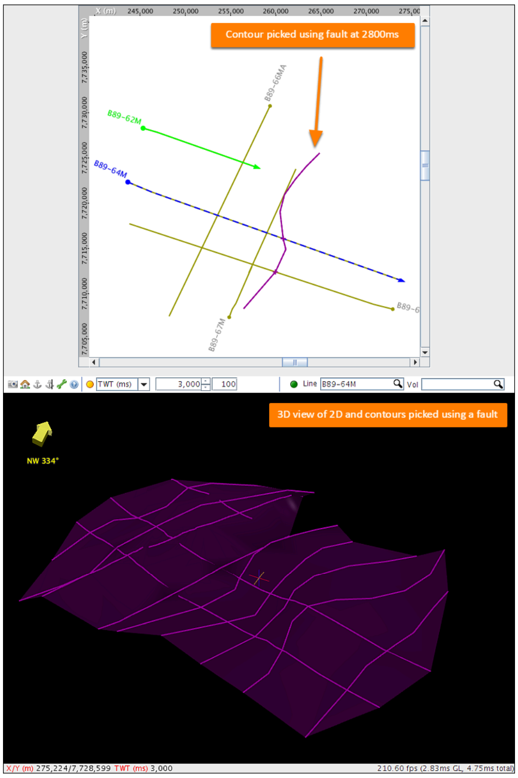
1. From the **Fault** tab, click the **"+"** button and create a **New Fault**. Read more about *Faults*.
2. Type a name for the fault (e.g. "horizonY-as-fault").
3. In section (i.e. 2D Line View or Arbline view), create fault sticks for the horizon on the available lines.



4. In **Map view**, select the contour TWT and create a new fault stick.
5. Repeat at different TWT values for additional contours.



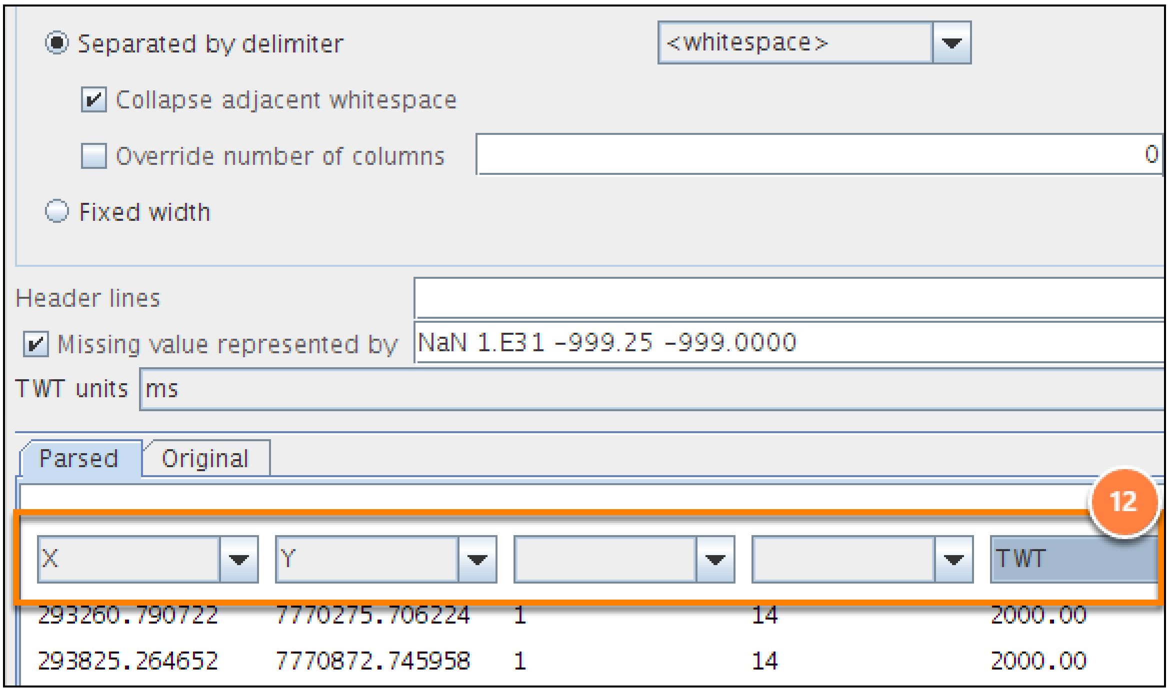
6. Repeat at different TWT values for additional contours.



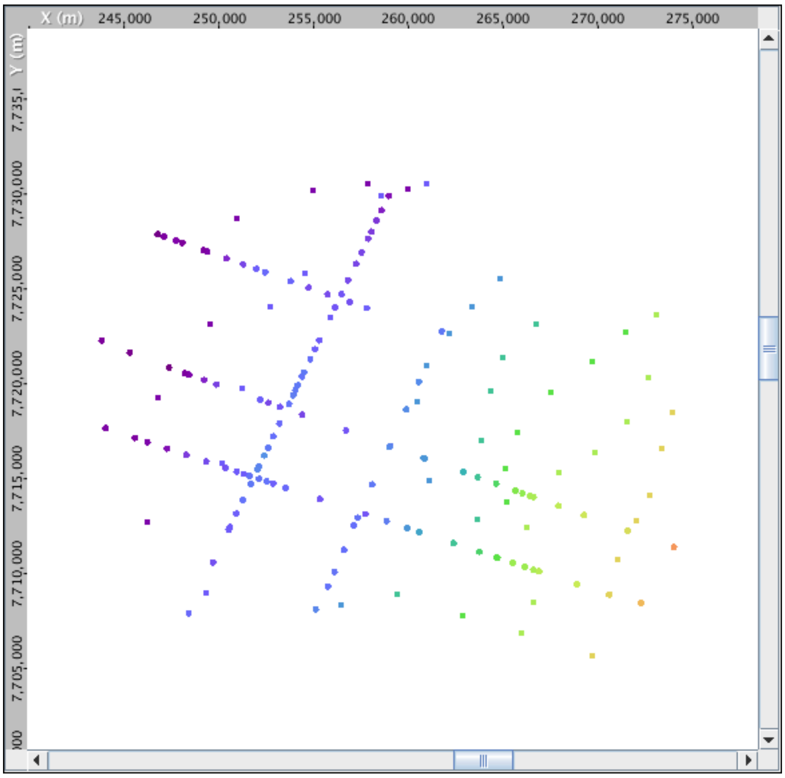
7. Repeat until the level of detail is satisfactory.
8. When finished picking, go to the **Fault** tab, right-click the fault and export it to a file (e.g. "horizonY-as-fault.dat").
9. Open the **.dat** file with any text editor and remove these lines:

```
PROFILE horizonY-as-fault                    TYPE 2   1 DUG Insight fault
SNAPPING PARAMETERS
...
EOD horizonY-as-fault
```

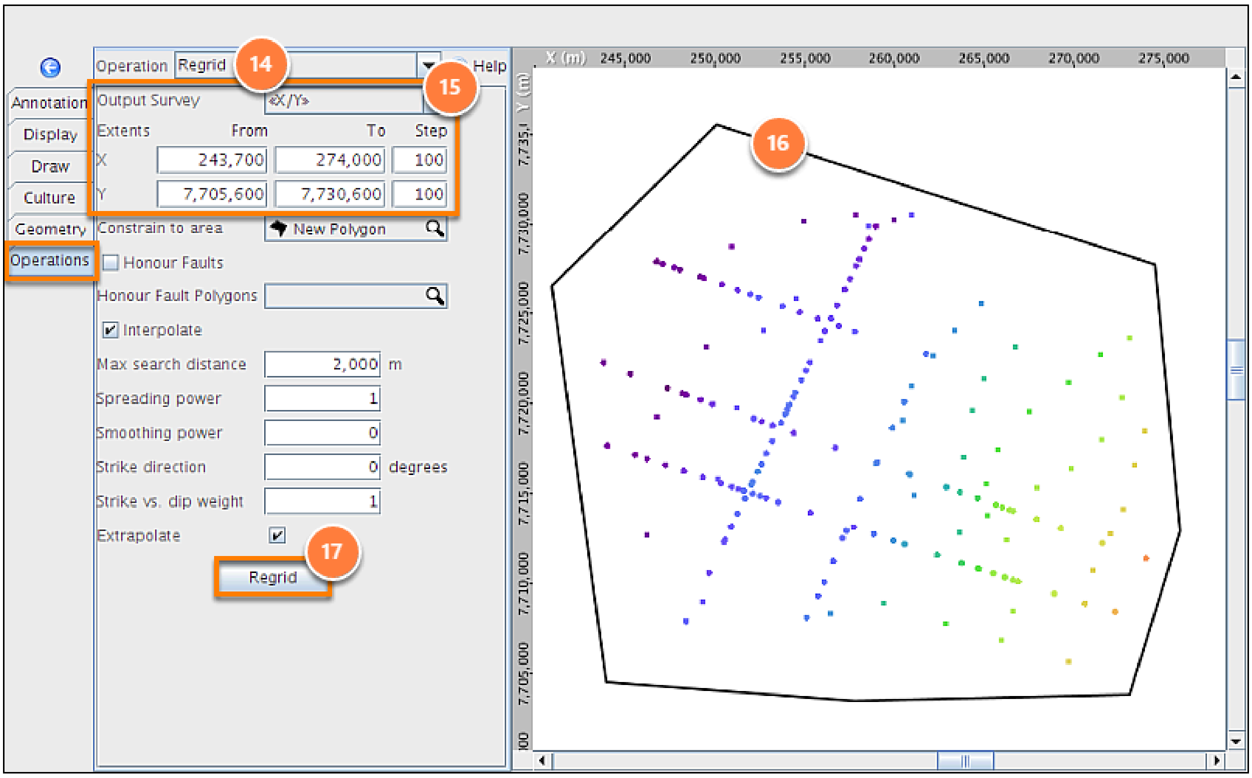
10. Save the file as "horizonY-as-fault.txt" (note the .txt extension)
11. From the **Control Panel** drop-down menu, select **Import >> Horizon** and choose the edited **.txt** file.



12. In the **Import** screen, set the column types as **X, Y, (blank), (blank), TWT** and click **OK**.

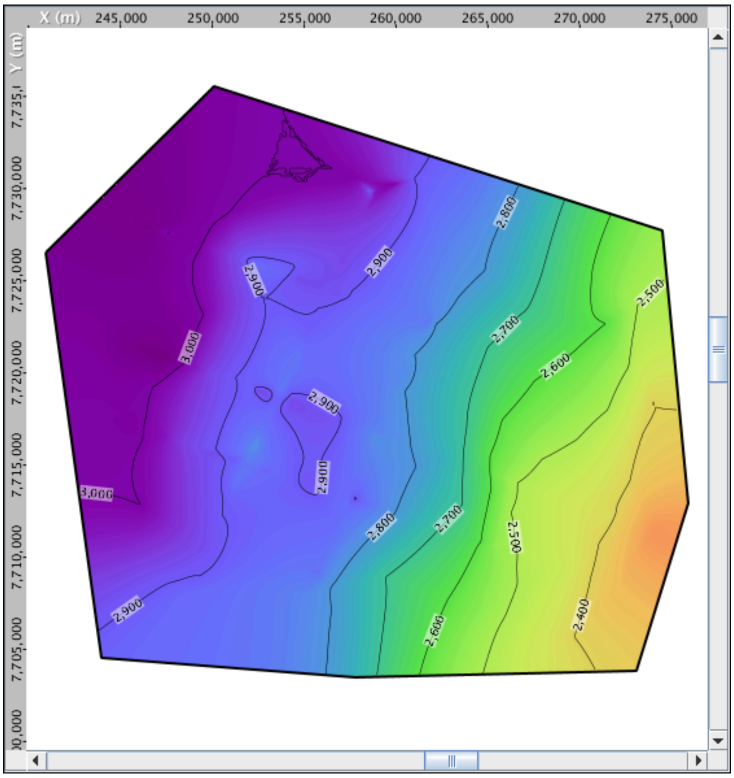


13. Go to **Map View** and check the result.

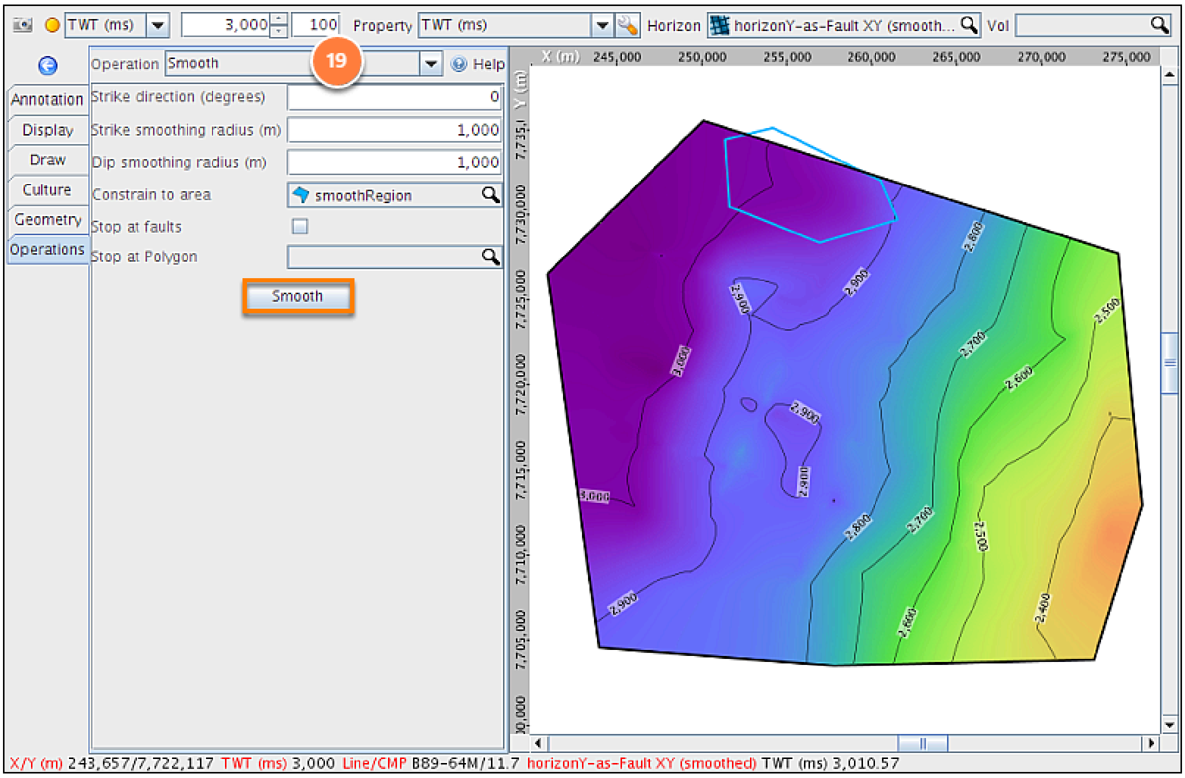


14. Next step is to regrid to X/Y. Select the **Operations** tab in the **Map View** and choose **Regrid** from the **Operation** drop-down box (if not yet selected, choose the imported horizon from the **Horizon** drop-down box).

15. Select **X/Y** as the **Output Survey**. For regional horizons, try an increment of **100m x 100m** or larger. Set these for both X and Y extents in the **Step** textbox.
16. Draw a polygon around the imported horizon to Regrid. Right click on the **Map View** and select **Create a New Polygon**. Read more on *Polygons*.
17. Select this polygon in the **Contrain** to area box and click **Regrid**.



18. Insight will interpolate/extrapolate the horizon within the polygon. Review the result once completed.

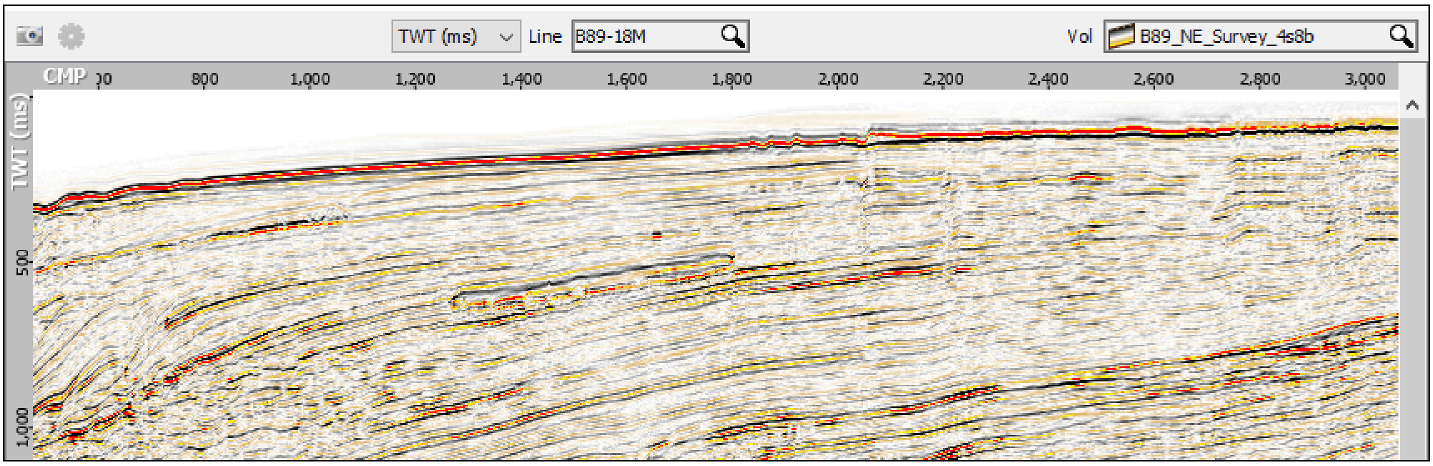


19. Apply some smoothing if the contours are irregular via the **Operations** drop-down option. Read more about *Smoothing (Horizon)*.

Dashed Line Selection Outlines in 2D Views

In the 2D Line View line selection box, lines are organized by survey name, which is left-justified. Line names are indented below the survey name, and they are sorted in alphanumerical order. It is possible to load duplicate 2D line names to a project associated with different surveys, if this happens you can use the left-justified survey name at the top of the line list to differentiate the line names.

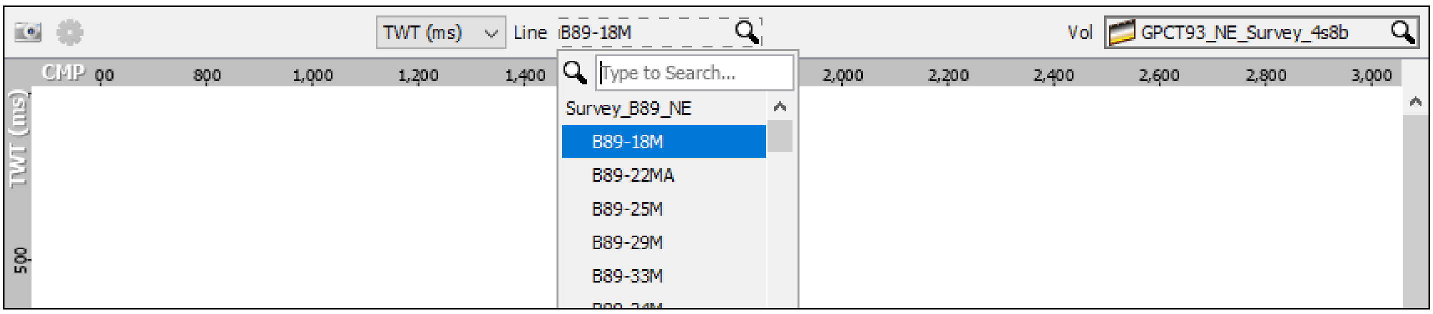
As well as the trace data displaying in the view, the Line selection box will be drawn with a solid grey colour when you select a survey line name with traces present in the active displayed volume in the display.



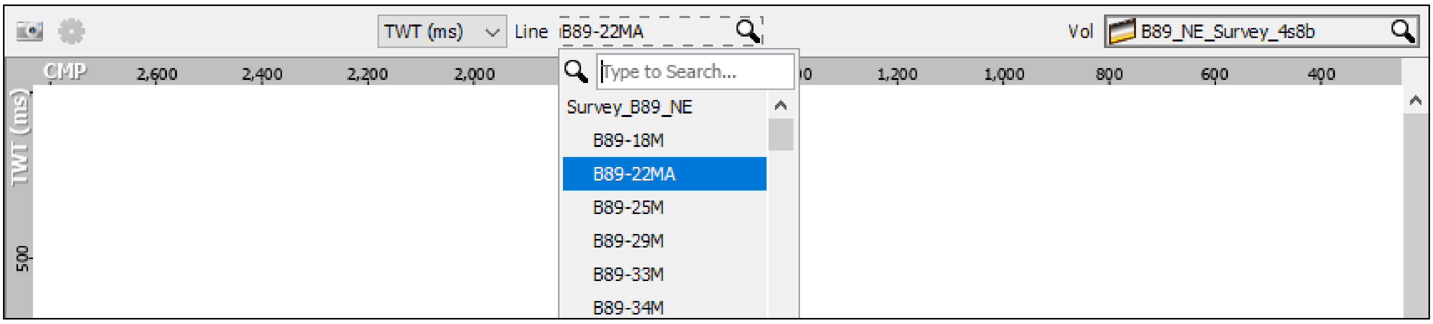
Here the line B89-18M has associated trace data in volume B89_NE_Survey_4s8b. Trace data appears in the view and the Line selection box draws with a solid grey outline indicating this is so.

The line selection box can also be displayed with a dashed red or dashed grey outline to indicate a mismatch between the selected line and the displayed volume.

If you select a line name which does not have traces in the active displayed volume, no trace data will be displayed in the view and you will see a dashed grey outline around the line selection box indicating that the volume data does not exist for that line selection.



Here Survey_B89_NE is not associated with volume GPCT93_NE_Survey_4s8b and so Line name B89-18M does not exist in volume GPCT93_NE_Survey_4s8b and there is no associated trace data. No data is displayed, and the Line selection box is drawn with a dashed grey outline. The volume must be changed to match the survey selection for traces to be displayed and the line selection box outline to become a solid grey line again.

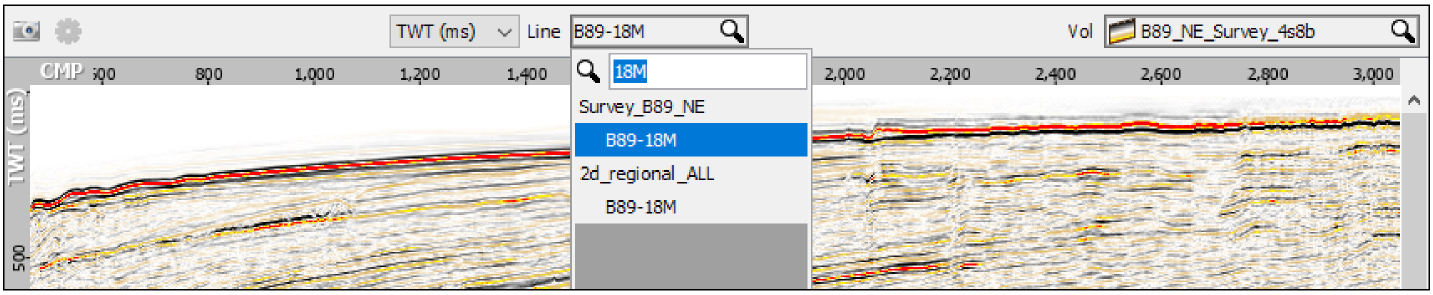


Here, volume B89_NE_Survey_4s8b is associated to Survey_B89_NE, but the Line name B89-22MA does not have any traces loaded to volume B89_NE_Survey_4s8b and so no data is displayed. If the SEGY data can be located and loaded for line B89-22MA, the volume can be displayed and the line selection box outline to become a solid grey line again.

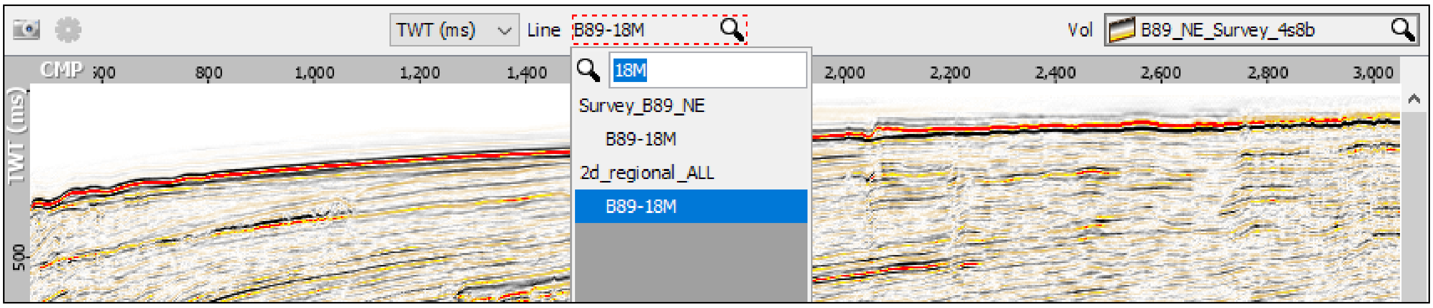
The dashed red box outlining a line selection means that you've selected a line name which exists in the displayed volume, but you've selected it from a different survey to the one associated to the volume. This happens when the same 2D line has been loaded to multiple surveys creating duplicate line names.

In this example two surveys are available, Survey_B89_NE and 2d_regional_ALL. The line name B89-18M is duplicated and exists in both surveys. Using the search filter in the line name box it is possible to see the line listed in both surveys.

The active displayed volume in the 2D line view is B89_NE_Survey_4s8b and is associated to Survey_B89_NE, not 2d_regional_ALL.



Selecting line B89-18M from Survey_B89_NE displays the trace data as expected and draws the line selection box with a solid grey colour indicating a survey - volume match.



Selecting line B89-18M from 2d_regional_ALL displays the trace data but draws the line selection box with a dashed red colour indicating that the line selected is part of a different survey from the display volume. Trace data is still displayed because the line name and navigation information is duplicated and available in the active displayed volume.

Interpreting on a volume when the line selection outline is dashed red will store horizon picks with another survey to the one intended and can cause confusion and problems in the future. Avoid interpreting horizons when the line selection box is red unless you have good reason to do so.

To make sure you're displaying the correct data, always check you've selected your line from the survey associated with the displayed volume.

Find Out More!

- *Surveys vs. Volumes*
- *Survey Overview*

Surveys

Why does SEG-Y Loader tell me I have a duplicate survey even though the coordinates are not the same?

In general, a survey simply defines the relationship between Inline/Crossline and Eastings/Northings.

Any given survey can be defined by an infinite number of orthogonal tie point sets, so long as they define the same relationship in the conversion from IL/CL to X/Y.

The following example shows two tie point sets that has different coordinates but describe exactly the same relationship between IL/CL and X/Y - and therefore are duplicate surveys:

Tie Points 1

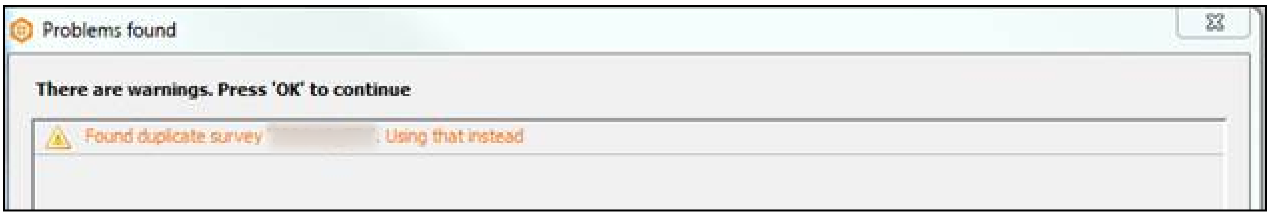
Inline	Crossline	Easting	Northing
1000	1000	100,000	1,000,000
1000	1001	100,000	1,000,010
1001	1000	100,010	1,000,000

Tie Points 2

Inline	Crossline	Easting	Northing
1001	1001	100,010	1,000,010
1001	1002	100,010	1,000,020
1002	1001	100,020	1,000,010

Inline and Crossline spacing equals 10 metres.

This is why you receive the "duplicate survey" warning (image below). Even though the tie points sets are different, they define the same relationship as the existing survey, and hence it is labeled as a duplicate.

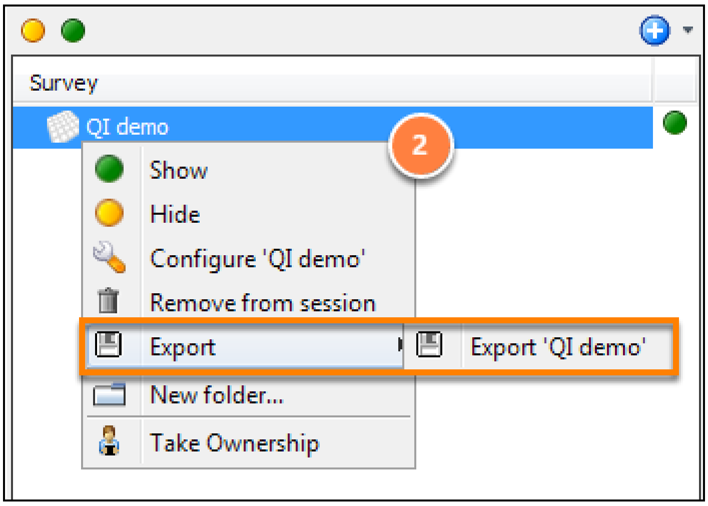


How do I change survey CRS to project CRS?

WARNING: Changing the coordinates for a survey will reposition all data associated with that survey. For example, altering a 3D survey will change the X/Y coordinates of seismic and any horizons picked on the survey.

This workflow assumes that a survey has been loaded without applying a CRS conversion to the survey. The solution exports the uncorrected survey, then redefines the current survey using the file with corrections applied..

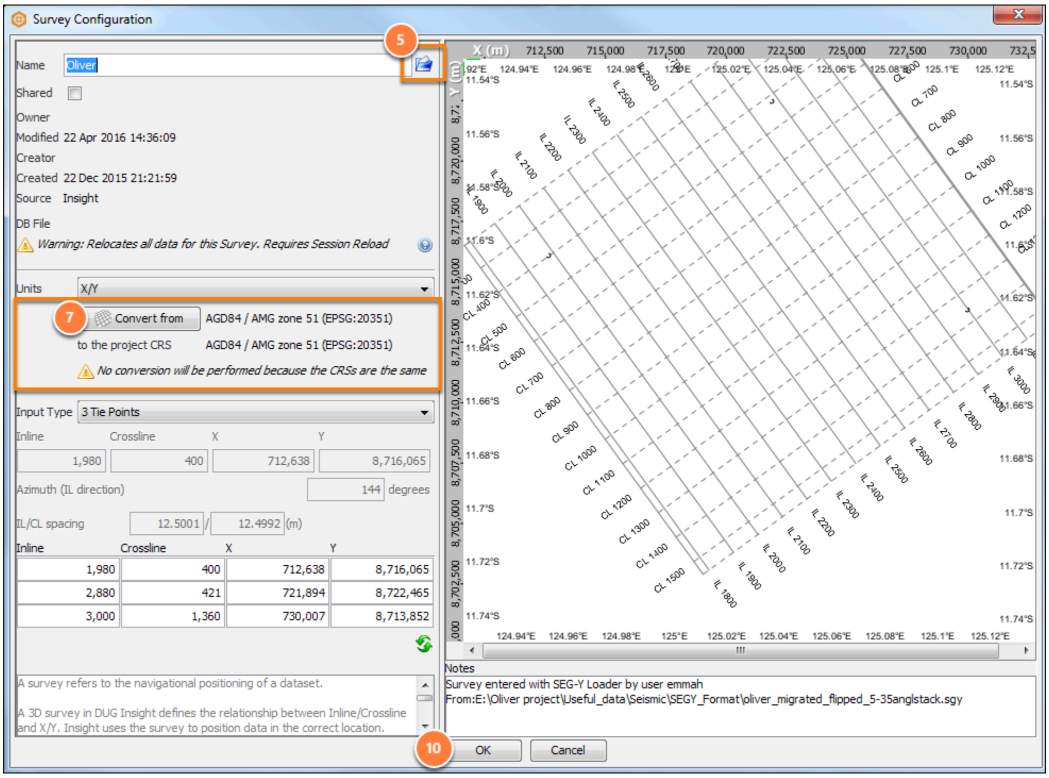
Export the survey (without corrections applied)



1. Go to **Control Panel > Survey** tab.
2. Right click on the survey and select **Export**.
3. Enter a filename and location for the exported survey.

Configure the current survey

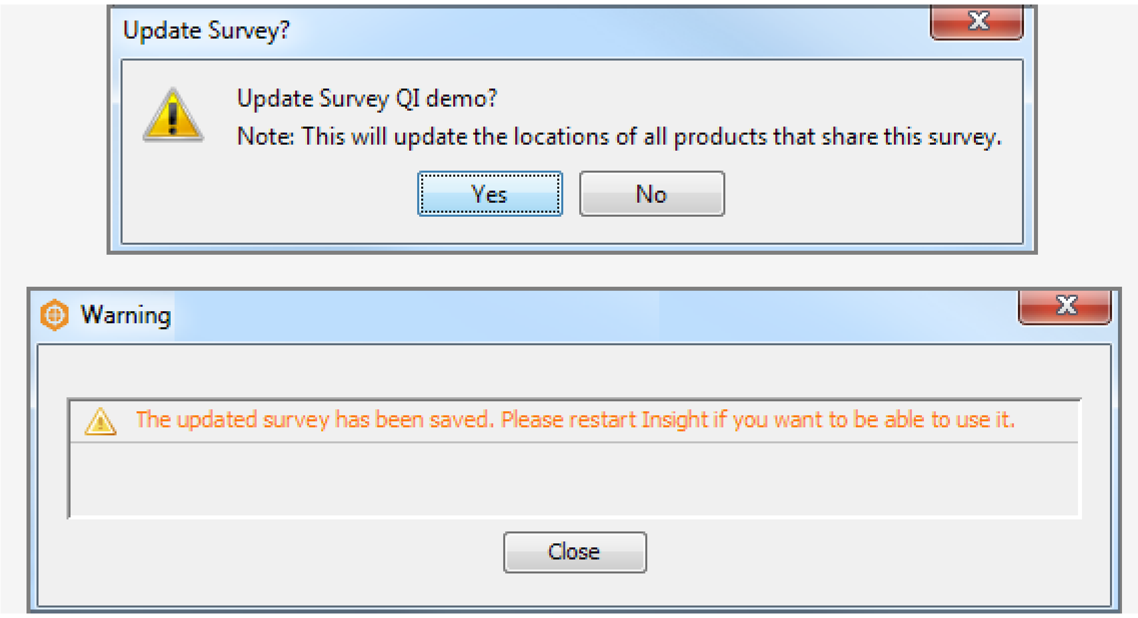
4. In the **Survey** tab, double click on the survey.



5. In the **Survey Configuration window**, click the **Select File icon**.
6. Select the exported 3D survey file and click **Open**.

Specify the original CRS for the survey

7. Choose the CRS used in the survey by clicking **Convert from**.
8. Insight will convert the survey coordinates to the project CRS.
9. Volumes and horizons will be repositioned accordingly.
10. Click **OK**.

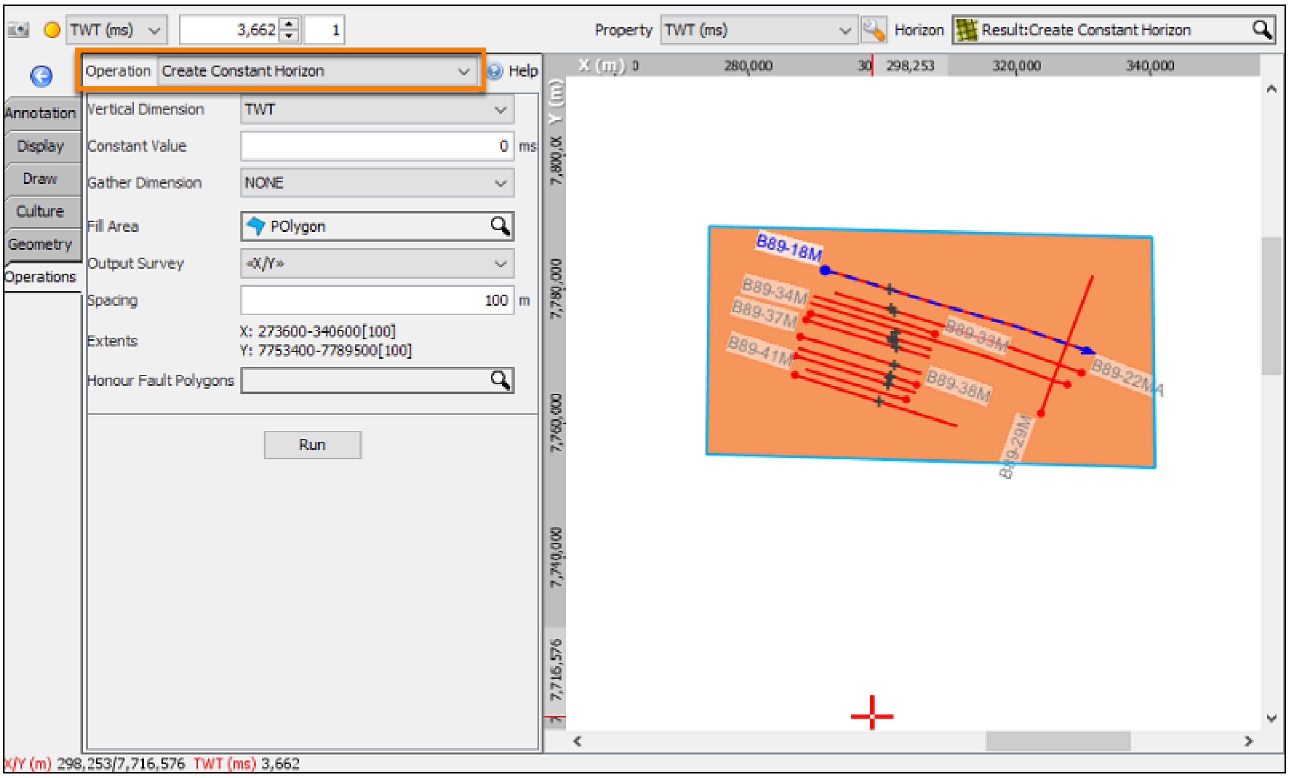


10. The **Survey Updated window** will appear to ask you to reload the session.
11. Click **Save** and **Reload** Session.

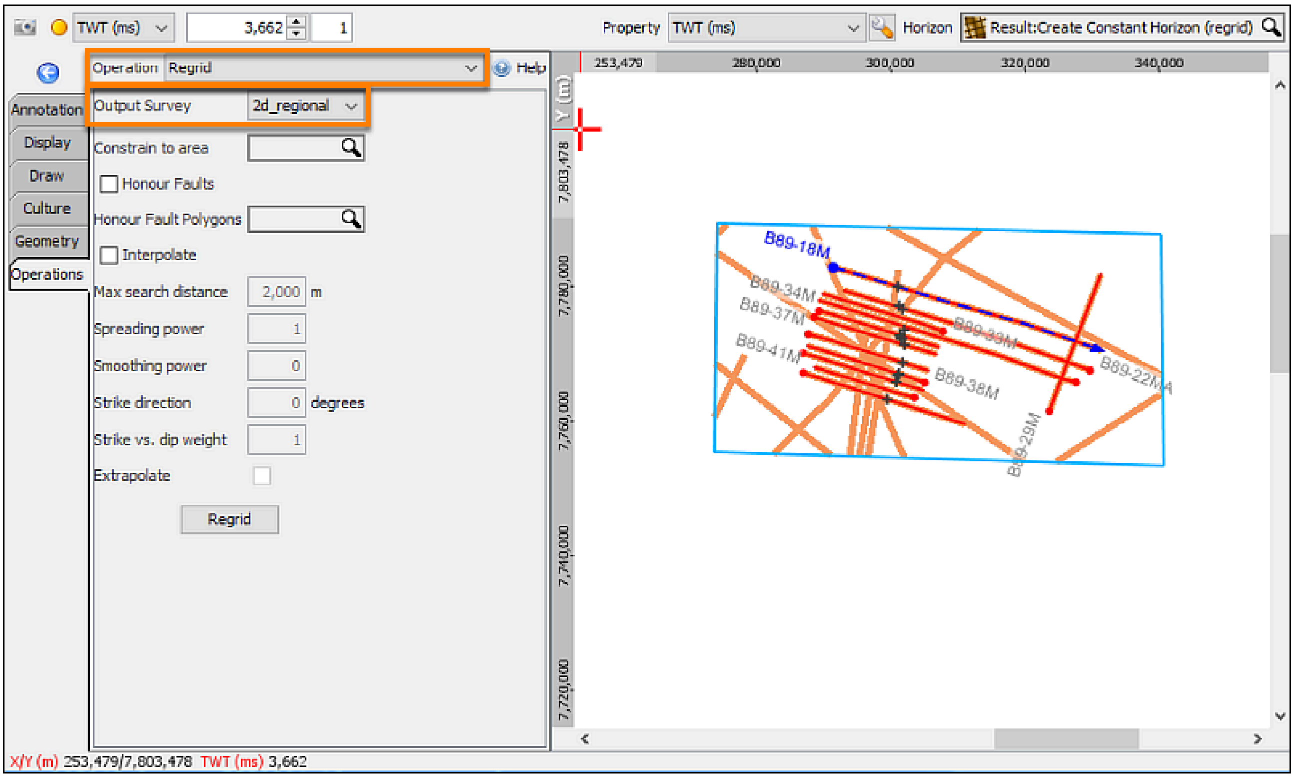
The current session must be reloaded for the changes to take effect. All items using the old survey will be repositioned to the new coordinates once the session is saved and reloaded.

Export 2D survey as simple XY TWT file

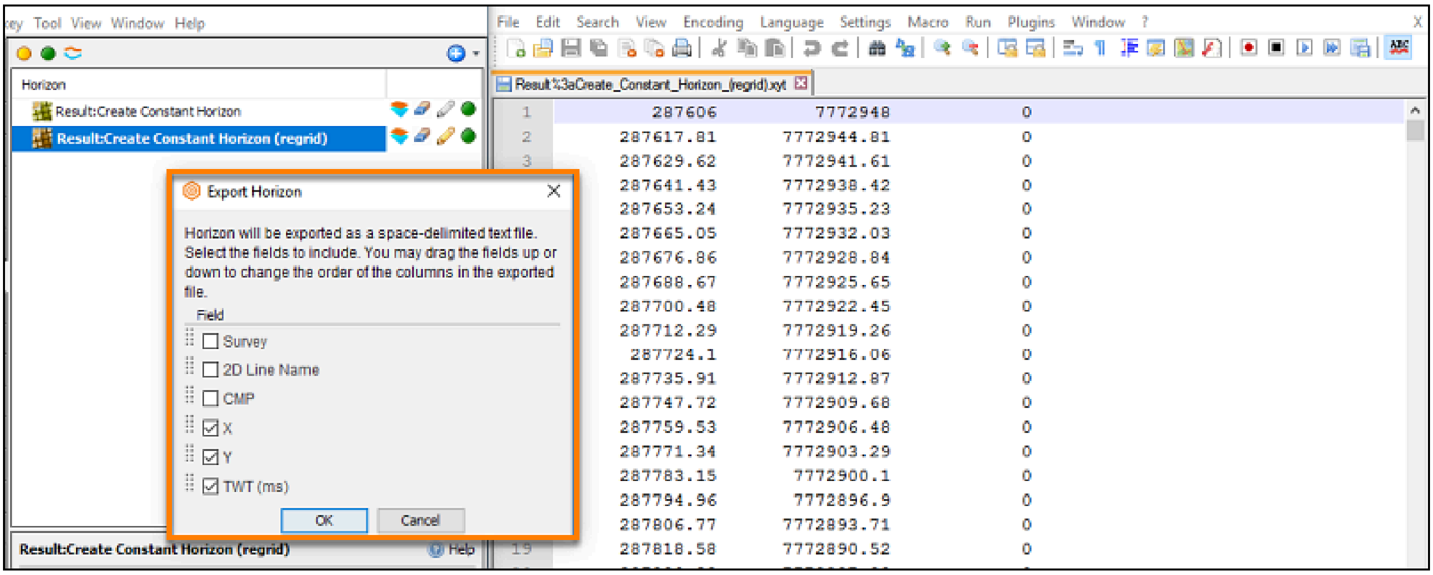
ARCGIS (and other softwares) import a simple XY TWT file. We can use the workflow below in Insight to create a horizon for XY values from the 2D surveys, export this to an XYT file for import into ARCGIS.



1. Create a constant horizon covering all the 2D surveys you wish to transfer. (A 2D line set can be used to group the 2D lines).



2. Regrid this horizon to the 2D survey using the Regrid tool. Create a new horizon.



3. Export this new horizon, select custom export and XY TWT. If needed, CMP and Line name can also be exported.
4. A simple XYT file will be exported. Import into ARCGIS.

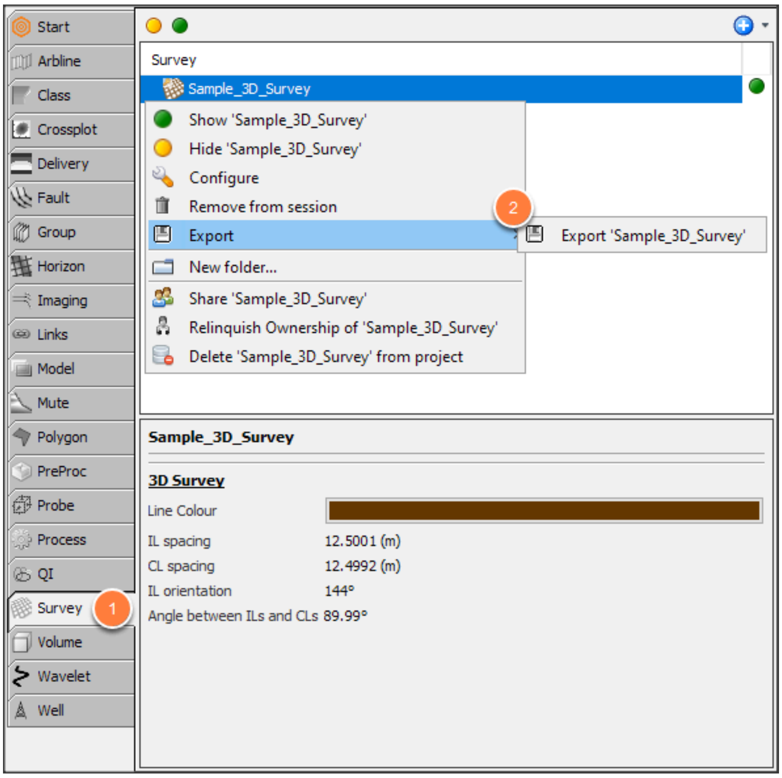
How do I set boundaries for my 3D survey?

Insight automatically extends 3D surveys to include associated data, horizons and volumes. Remember to check the *Product Extents Table* for any objects that might cause the 3D survey to expand.

In some cases, the survey extends further than is required, making the **Map View** harder to use.

Use this workflow to reduce the displayed survey extents in the **Map View**.

Note: The survey boundaries are for display only. They update to contain the coordinates of anything assigned to the survey. Updating them will not affect the positioning of anything in the project.



Export the 3D survey

1. **Control Panel > Survey tab**, right-click the desired survey and choose **Export** (see *Exporting a Survey*).

2. Save to file, e.g. '*3d_survey.survey*'

Example format of .survey file below, *more examples available here*.

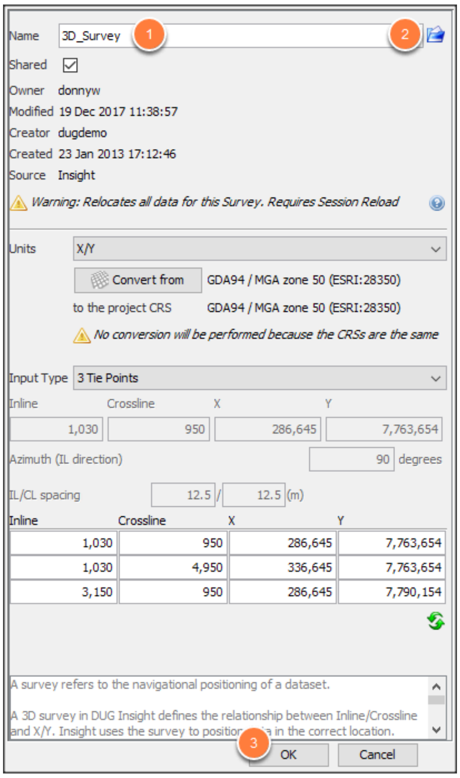
```
survey_type=3D
survey_name=Sample_3D_Survey

# Survey entered with SEG-Y Loader by user Sample
# From:E:\Oliver project\Useful_data\Seismic\SEGY_Format\oliver_migrated_flipped_5-35anglstack.sgy

# survey master grid. inline and crossline extents. Note these can't have increments.
survey=il:1470-3510+cl:-80-1840

# tie points. inline/crossline to X/Y
survey_pt1_IL_CL_X_Y=1980.000000/400.000000,712638.000000/8716065.000000
survey_pt2_IL_CL_X_Y=2880.000000/421.000000,721894.000000/8722465.000000
survey_pt3_IL_CL_X_Y=3000.000000/1360.000000,730007.000000/8713852.000000
```

Survey files are easy to update. Use any text editor to open the .survey file and update the min and max values for **IL** and **CL**.

1. Search for line: **survey=**
2. Update **il: ####-####**
   - Enter the desired min and max inline
3. Update **cl: ####-####**
   - Enter the desired min and max crossline
4. Save the file.

We're ready to update the survey with the new extents.

Redefine the 3D survey in Insight



1. Open the **Survey** tab in the **Control Panel**.
2. Double-click the survey to adjust.
   - Alternatively, right-click the survey and choose **Configure 'Survey'**.
3. Type a new name in **Name** field.
4. Click the **Select File** icon to the right of **Name** field.
5. Select the 3D survey file (e.g. '*3d_survey.survey*') that you updated and click **Open**.
6. Click **OK** to replace the survey.

Volumes and Data

Can I store a volume outside the project directory?

Sure you can!

We understand that volumes are huge files that can take up a lot of space in your machine. Hence, we have made sure that Insight projects can be stored independently from the dugio volumes without any problems.

By default, Insight uses the 100sei directory as the default location for storing and retrieving volumes. This is to help you keep things organised, especially in a multi-user environment. However, if you find yourself juggling data to manage disk space, you can move the .dugio volume to a different location (secondary disk, network, external USB storage, etc.) after the SEG-Y loading process is completed. Provided the volume is not in use (i.e. Insight is not open), .dugio volumes can be moved around and copied safely. The operating system will treat them as folders.

Tip: To change the default directory to another location, see *How do I change Insight's default directory?* in the FAQ section.

If Insight cannot locate a volume when opening a session (or adding a volume from the project), it will prompt you with a dialog to locate and reconnect it. This might happen because a network share is unavailable, an external drive is disconnected, or the volume was moved (see *Missing files when loading volume. Where did my volume go?*).

When importing from SEG-Y, make sure you output the volume to a fast and reliable location with enough space. The new volume will automatically be added to the project. SEG-Y loading is an I/O intensive task, so performance is governed by the read and write speed.

Gather Types

There are three types of gathers:

1. **Type 1** (Regular offset) — Gathers that have regular offsets for every CMP (**Type 1** = same, same).
2. **Type 2** (Irregular offset) — Gathers that have irregular offsets, but the same irregularity for every CMP (**Type 2** = different, same).
3. **Type 3** (Irregular offset, and location-dependent) — Gathers that have irregular offsets, and the offset irregularity is different for every CMP (**Type 3** = different, different).

Examples

Type 1 example:

CMP1: Trace 1 offset 75m, Trace 2 offset 150, Trace 3 offset 225m, etc (regular 75m increment)

Type 2 example:

CMP1: Trace 1 offset 75m, Trace 2 offset 155m, Trace 3 offset 240m, etc (no regular increment)

CMP2: Trace 1 offset 75m, Trace 2 offset 155m, Trace 3 offset 240m, etc (but the irregularity is the same for every CMP)

Type 3 example

CMP1: Trace 1 offset 75m, Trace 2 offset 155m, Trace 3 offset 240m, etc (no regular increment)

CMP2: Trace 1 offset 80m, Trace 2 offset 165m, Trace 3 offset 245m, etc (and irregularity between CMPs).

General Discussion

Most gathers are typically Type 1.

Depending on the acquisition geometry and processing, some gather datasets may be Type 2 and, even less commonly, Type 3. The most common use for Type 2 and Type 3 is when the gather dimension is not offset (i.e. frequency).

Type 2 gathers have the following properties:

- low fold data
- acquired in feet, offsets converted to metres (rounding errors give increments of 24, 25, 24, 25, 25, 24)
- spectral decomposition frequency (i.e. many lower frequencies, but more sparse in higher frequencies).

Type 3 gathers are mostly used for shot gathers, channel gathers and other similar raw data. It treats each gather like a bucket of traces and has to sort through them when it gets there.

Insight's Automatic Volume Class Assignment

When importing a volume, Insight will try to guess a volume's class using keywords found in the volume's name.

These keywords are not case sensitive, and are checked in the order shown below. The *first matched* keyword will be used to determine the volume's class.

Velocity volume check

If the volume filename ends with any of the following AND either *.su* or *.dugio*, the specified class will be used.

Ends with	Uses Class
vintt	Interval Velocity in Time
vintz	Interval Velocity in Depth
vrmst	RMS Velocity
vavgt	Average Velocity in Time
vavgtz	Average Velocity converted to Depth
zt	Depth values sampled in Time (ZT)
tz	Time values sampled in Depth (TZ)

Keywords class check

Otherwise, the presence of the first of any of the following keywords will cause the specified class to be selected.

Keyword	Class
ufar	Stack (Ultra-Far)
far	Stack (Far)

<table>
<tr><td>mid</td><td>Stack (Mid)</td></tr>
<tr><td>near</td><td>Stack (Near)</td></tr>
<tr><td>stack</td><td>Stack</td></tr>
<tr><td>gather</td><td>Seismic Gather</td></tr>
<tr><td>impedance AND s-</td><td rowspan="2">S Impedance</td></tr>
<tr><td>simp</td></tr>
<tr><td>impedance AND p-</td><td rowspan="2">P Impedance</td></tr>
<tr><td>pimp</td></tr>
<tr><td>rho</td><td rowspan="2">Density</td></tr>
<tr><td>density</td></tr>
<tr><td>rho AND lambda</td><td>Lambda-Rho</td></tr>
<tr><td>rho AND mu</td><td>Mu-Rho</td></tr>
<tr><td>prob AND oil</td><td>Probability (Oil)</td></tr>
<tr><td>prob AND brine</td><td>Probability (Brine)</td></tr>
<tr><td>prob AND gas</td><td>Probability (Gas)</td></tr>
<tr><td>poisson</td><td>Poisson's Ratio</td></tr>
<tr><td>vp AND vs</td><td>Vp/Vs</td></tr>
<tr><td>vs</td><td>S Velocity</td></tr>
<tr><td>vp</td><td>P Velocity</td></tr>
</table>

How do I load a velocity ASCII cube into Insight?

Unfortunately, velocity ASCII files cannot be loaded into Insight. However, if you send it to us, we are happy to convert it to *.dugio* for you to import into your Insight project.

Note: We understand that the contents of any data sent to us are confidential and intended for testing and bug fixing purposes only. Under no circumstances will the data be shared or distributed to anyone other than those necessary to solve your problem. You have our promise that we will safeguard your data to the best of our ability. Once we have solved the problem and closed the support ticket, your data will be removed from our system.

Send us your velocity

Kindly send the file and the survey onto which the velocity is to be imported (see *Exporting a Survey*) to support@dugeo.com or upload it to our FTP site (see *How to upload to our FTP server?* for instructions).

After receiving the data, it will take us a day or two to prepare the results.

How do I crop a seismic volume using a polygon?

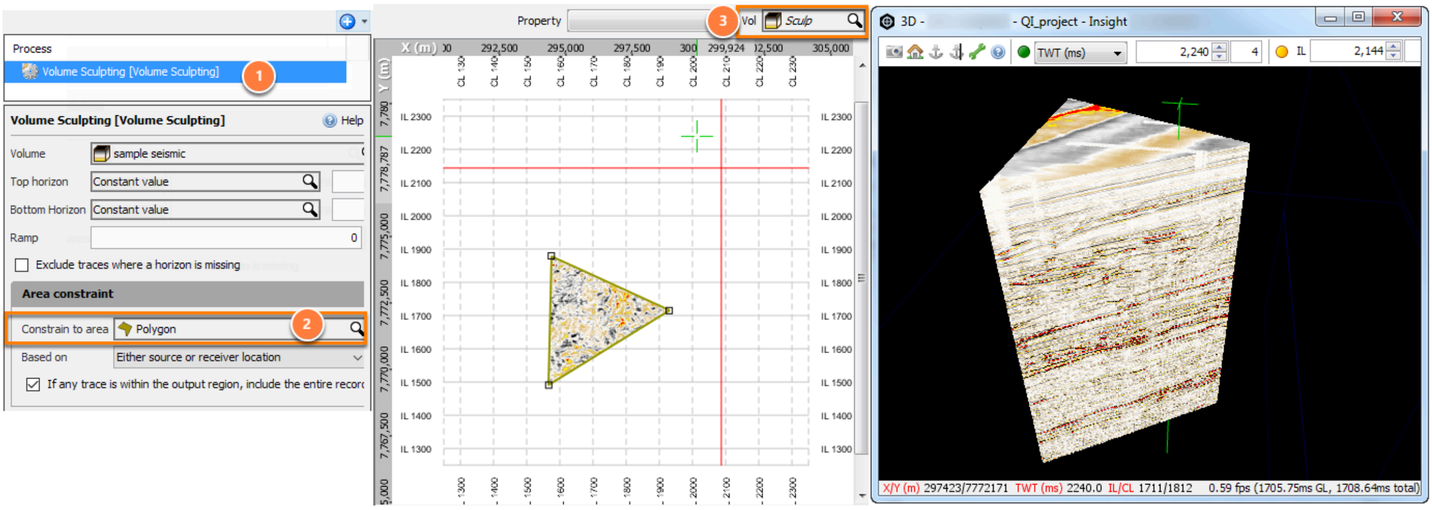
Volume Sculpting and Volume Maths can be used to remove the area *inside* or *outside* a polygon.

A probe can also be used, but is limited to the shape of a cube.

Remove seismic outside polygon

1. In the Process tab, create a **Volume Sculpting** process (see *Volume Sculpting*).
2. Choose the volume and select a polygon to constrain the data inside the polygon. You can also constrain the vertical extents of the cropped seismic by selecting a constant time/depth or horizon.
3. In the views, select the process volume to display.

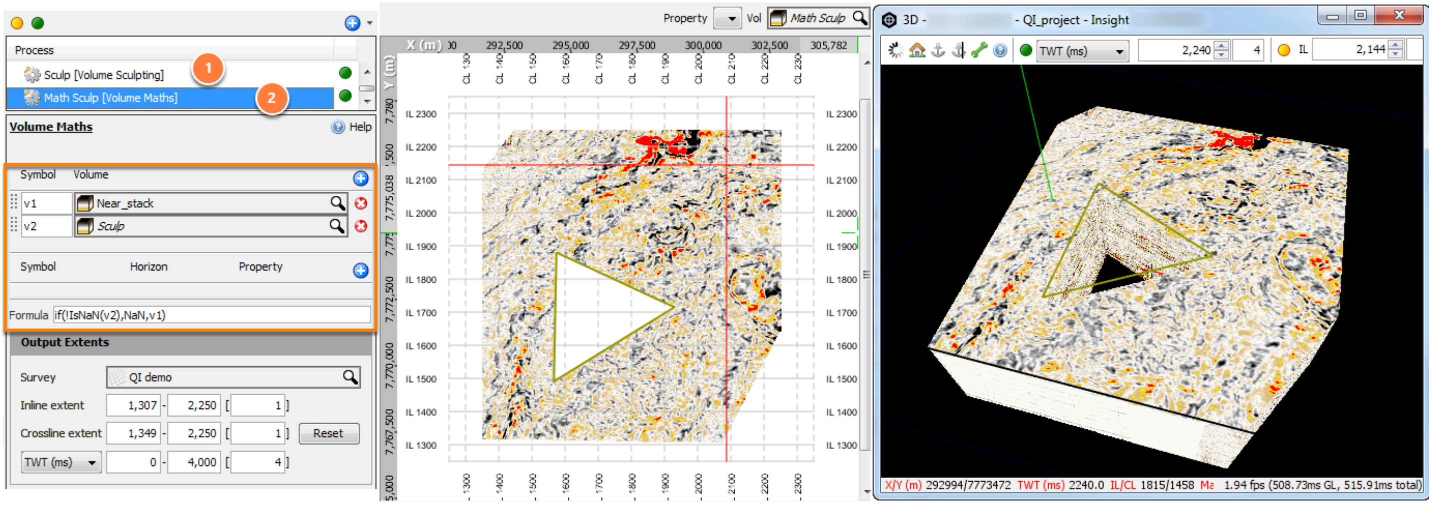
This will create a process volume within the constrains of the polygon. Process volumes are updated on-the-fly and may be computationally intensive. Export this process volume to disk to improve performance (see *Exporting a Volume*).



Remove seismic inside polygon

1. In the Process tab, create a **Volume Sculpting** process (see *Volume Sculpting*). Choose the volume and select a polygon.
2. Create a **Volume Maths** process.
   - Select the original volume as v1 and sculpted volume as v2.
   - At Formula, type the following formula: **if(!IsNaN(v2),NaN,v1)**

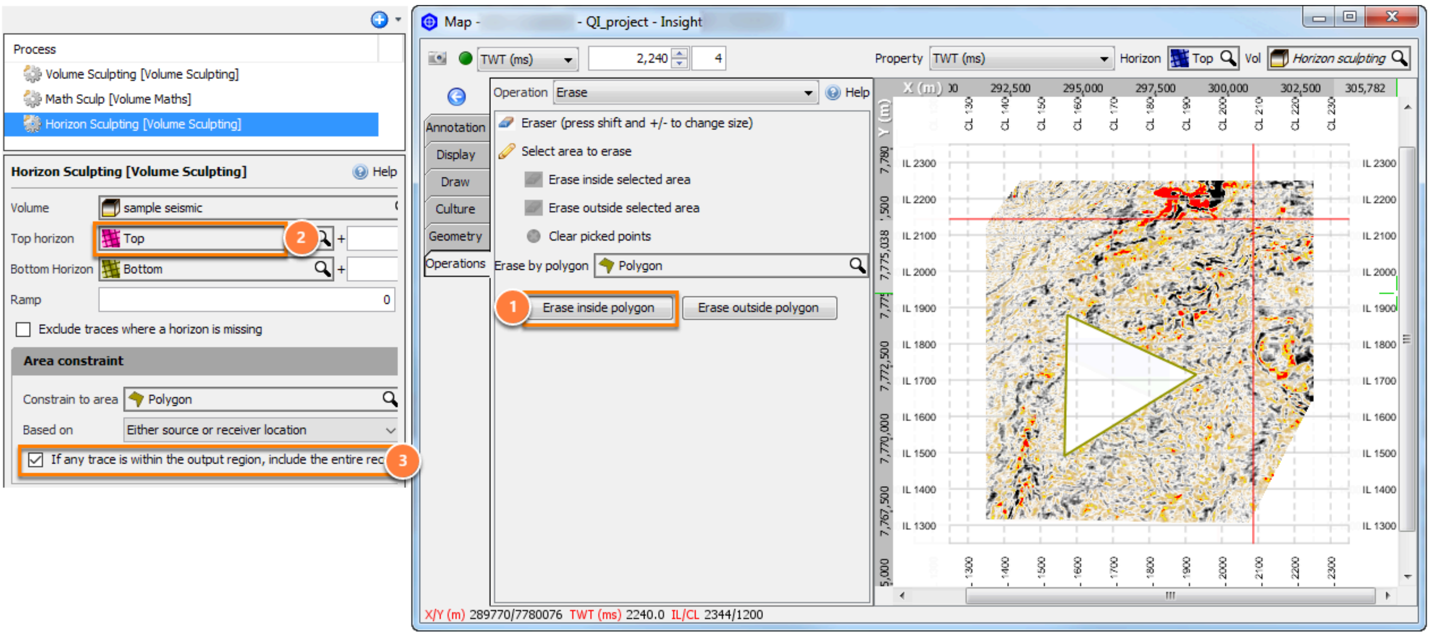
This will create a process volume that omits the area inside the polygon. Process volumes may be computationally intensive. Export this process volume to disk to improve performance (see *Exporting a Volume*).



Alternatively...

Another way to remove the volume inside a polygon is to create a horizon with missing values inside the polygon (see *Horizon Erase*), and then select to **Exclude traces where a horizon is missing** during Volume Sculpting.

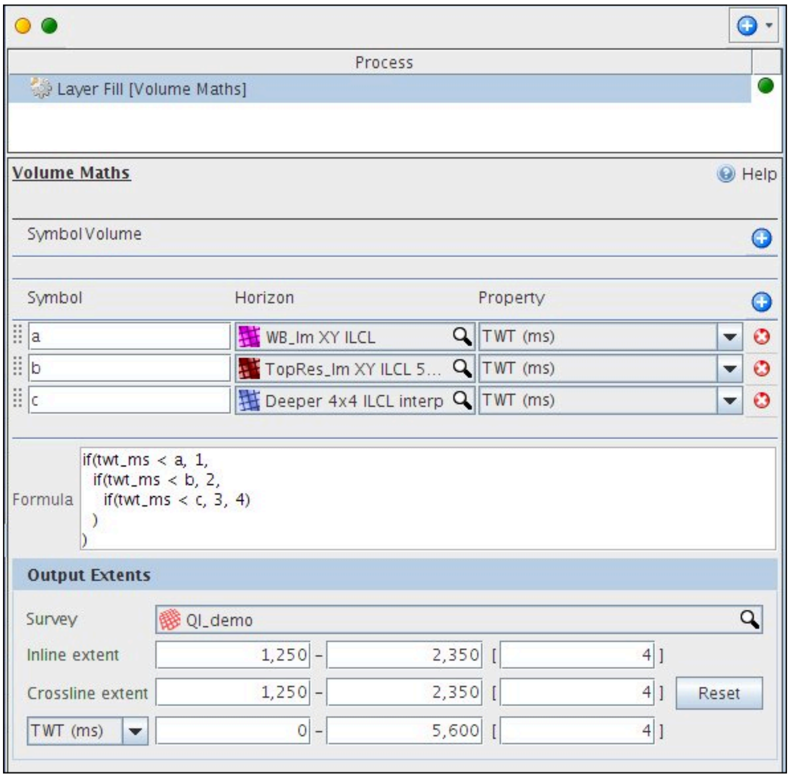
If either (you only technically need one) of the horizons used in Volume Sculpting have null values, the nulls will carry through into the process volume.



How do I colour a volume differently between horizons?

This workflow allows for flexibility in colouring the different stratigraphy in your volume.

Use Volume Maths process to fill values between horizons



1. Create a **Volume Maths** process. In this example, we name it "Layer Fill".
2. Add each of the horizons to define the layers. It will be easier if the horizons are correctly sequenced.
3. Construct an *if* statement using the following structure:
   - if TWT above A, use value 1, otherwise
   - if TWT above B, use value 2, otherwise
   - if TWT above C, use value 3, otherwise use value 4

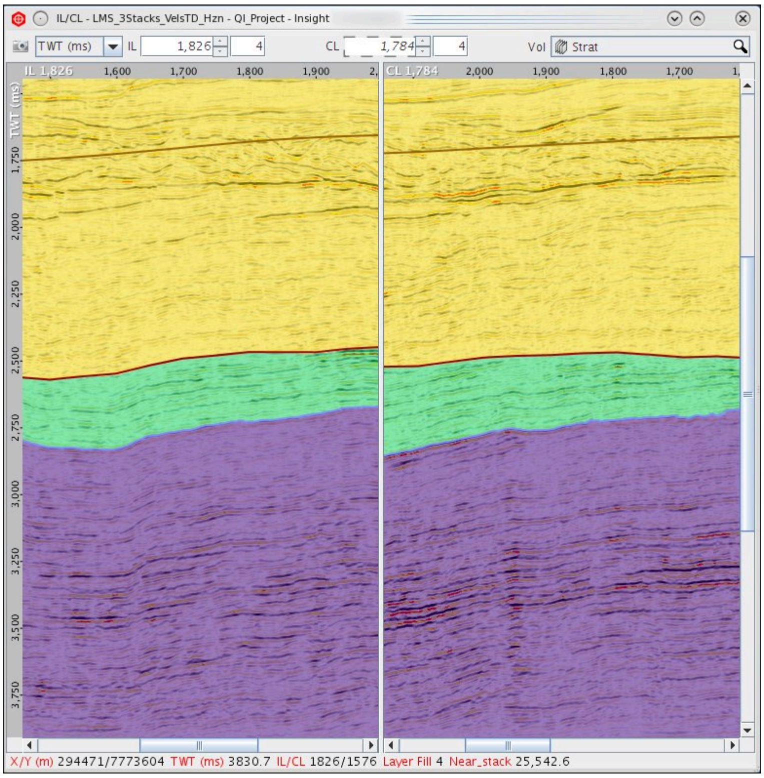
   The actual formula looks like this:

```
if (twt_ms < a, 1,
  if (twt_ms < b, 2,
    if (twt_ms < c, 3, 4)
  )
```

```
)
```

4. Finally, choose a nice colourbar and set the range from 0 or 1 to your maximum value.
5. If you would like to overlay the results, create a group containing the "Layer Fill" virtual volume, put that layer at the top of the group, and adjust its transparency in the class settings.

Here are the results achieved using a quick test:



Why does the result of my Volume Combine process look different to my group of the same volumes?

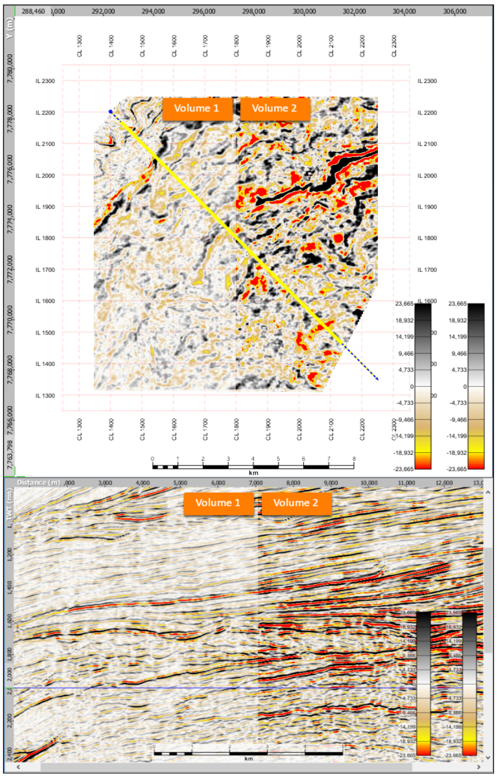
The group will use the settings of the individual volumes to adjust the displayed amplitudes. In a group, the volumes might have different amplitude ranges, but still display similarly.

Volume combine doesn't "see" the display settings, only the actual trace values. After combining, the resulting volume will contain the unchanged amplitudes from the original volumes.

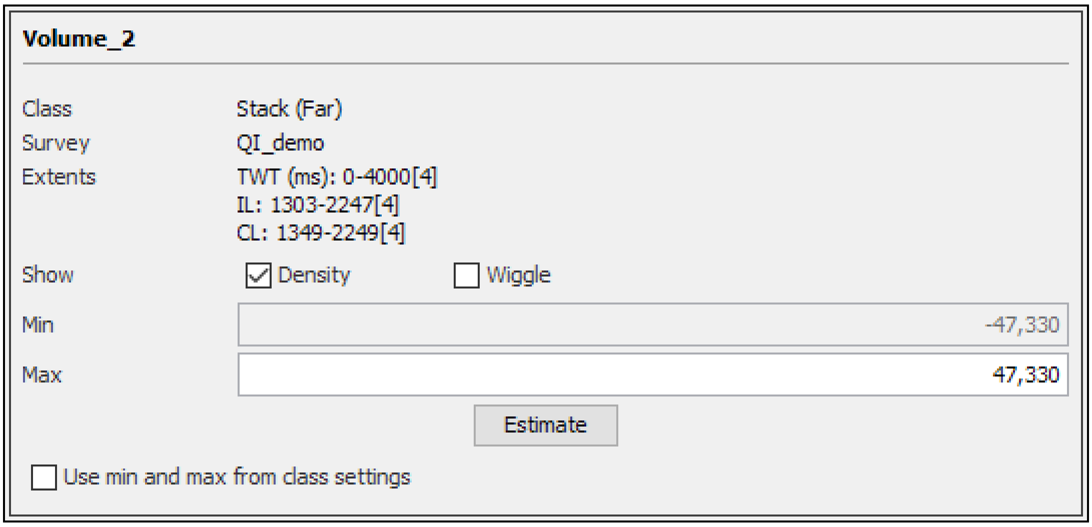
The solution is to apply a scale to the volumes before combining. There are two easy ways to do this:

1. Use a *Volume Maths (Simple)* process to apply a scale, and combine the resulting volume; or
2. Use *manual mistie correction* to adjust the scale of the volume as it is read.

Adjusting amplitude scale of two volume groups

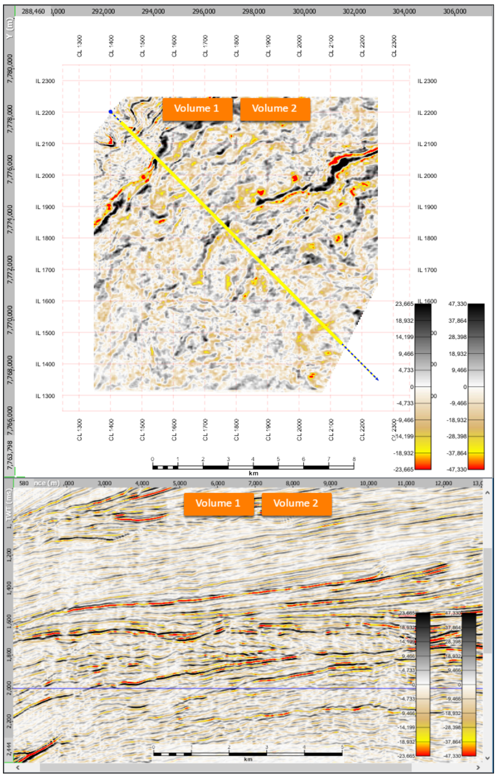


A timeslice view of both volume 1 and volume 2 grouped together. However, volume 2 has twice the amplitude range of volume 1, making the amplitudes display more intensity.



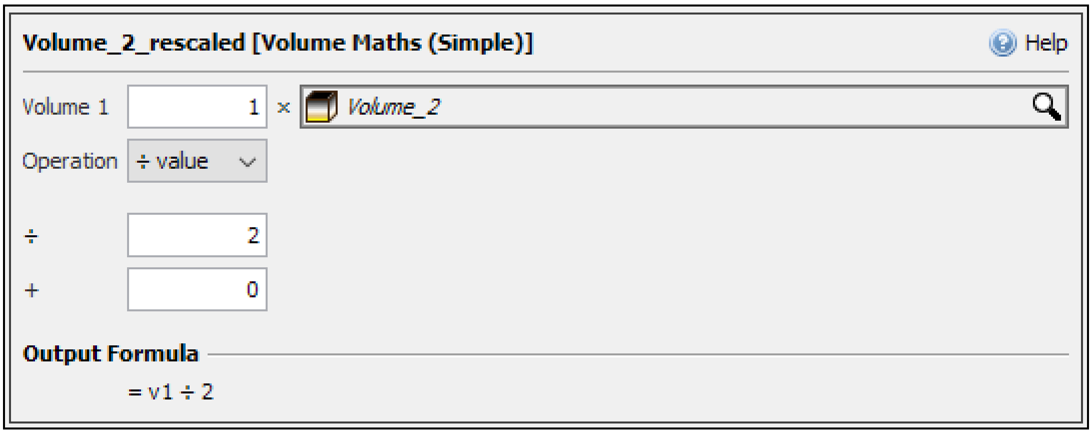
To adjust the colorbar range so that they have the same scale relative to amplitude values for each volume:

1. Uncheck the ‘Use min and max from class settings’ box
2. Adjust the the Min/Max values for volume 2 to correlate with those in volume 1.
3. The colourbar range for each volume is scaled to amplitude values (of one volume) so that amplitudes is shown with the same intensity across both volumes.



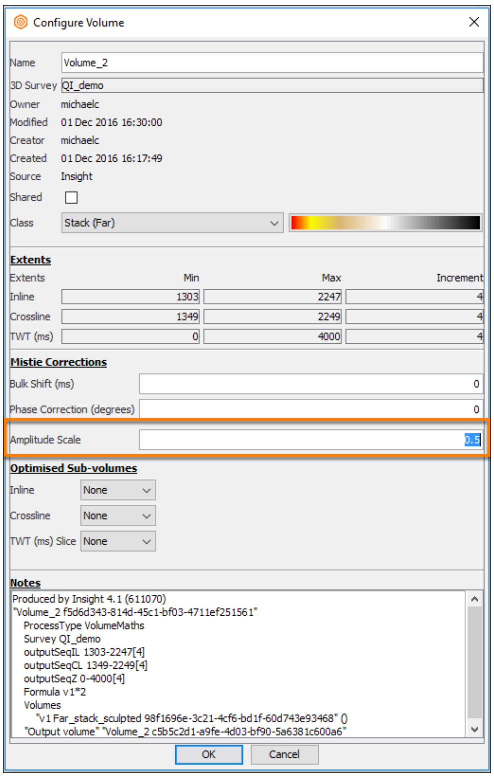
Create a combined volume with the same amplitude value

range



1. Go to **Process** tab and add a **Volume Maths (Simple)** process.
2. We will use **Volume Maths (Simple)** to rescale volume 2 to volume 1. Since volume 2 has twice the amplitude range of volume 1, dividing the values by two will properly scale volume 2.
3. Still in the **Process** tab, add a **Volume Combine** process.
4. Using the **Volume Combine** process, select volume 1 and the rescaled version of volume 2 (created from **Volume Maths (Simple)** process earlier) as the **Input Volumes**. This will create a new volume combining volume 1 and volume 2, with both having the same amplitude scaling.

Manually adjust the amplitude scale.



The amplitude scale can also be adjusted manually from the **Configure Volume** window.

- To scale the amplitude of Volume 2 to Volume 1, enter ‘0.5’ as the input for ‘Amplitude Scale’.

What should I know about Insight's 16-bit data compression?

Being relatively new to Insight processing I would like to better understand how intermediate data is stored in 16-bit. I haven't found any documentation so have the following questions:

Q: What processing should be applied to the original 32-bit floating point input data before it can be safely stored in 16-bit (integer?) format (e.g. any amplitude balancing or scaling required)?

A: Your data should be despiked. 16 bit data results in a fixed-size compression ratio of about 1.5:1 (i.e. an output file that's about 60% the size of the uncompressed input) with a loss of precision of approximately 1 part in 32,000, provided that the data has no spikes. Hard zeroes (0.0) are preserved exactly.

Q: Does the sample format of the internal processing depend on the storage format of the input data or is it always done at 32 or 16-bit?

A: Internal processing is always done at full precision

Q: Are there any issues associated with the subsequent return of data from 16 to 32-bit when the final data are written out to SEGY format?

A: Exporting back to 32 bit will retain the precision you had at 16 bits.

Q: If a large spike is detected on a 16-bit stored dataset I assume that this could impact the accuracy of the nearby amplitudes and subsequently the correct approach to fixing this would be to recreate the dataset without a spike rather than just fix the spike on the input data?

A: The volume is packed in blocks of 32 samples. For each block of 32 samples, the 16-bit range is scaled to match the range of those samples.

A spike of extreme amplitude blows out the range of the block, ruining the storage of the other samples. This is why we recommend using 16 bit data after the input has been processed to remove spikes.

Approaches to despiking are data dependent. We recommend consulting an experienced seismic processor for guidance.

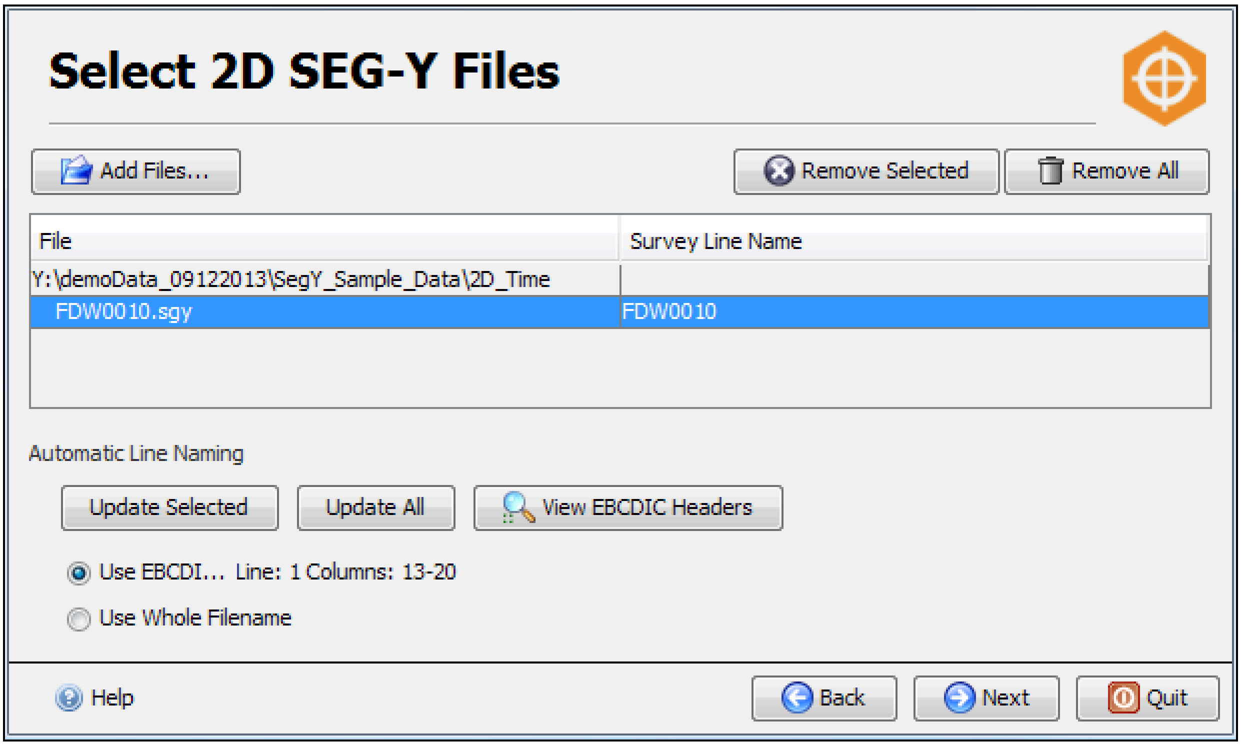
How to add/replace 2D lines to an existing dugio

After loading a volume from SEG-Y, a 2D line may need to be added or replaced in the volume (e.g. because of an error in the SEG-Y file).

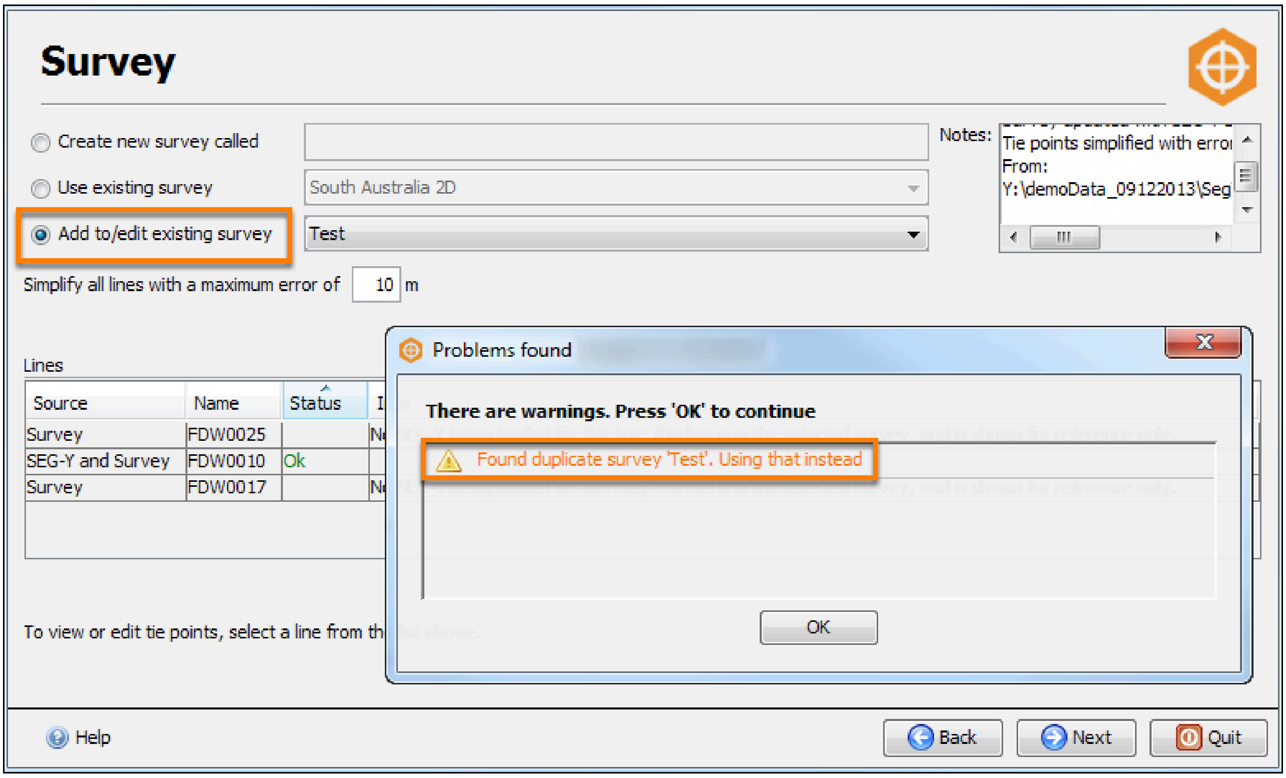
This can be done in the SEG-Y Loader by specifying an existing 2D dugio volume as the output file.

Note: We recommend making a backup of your existing dugio volume and survey before proceeding with this workflow.

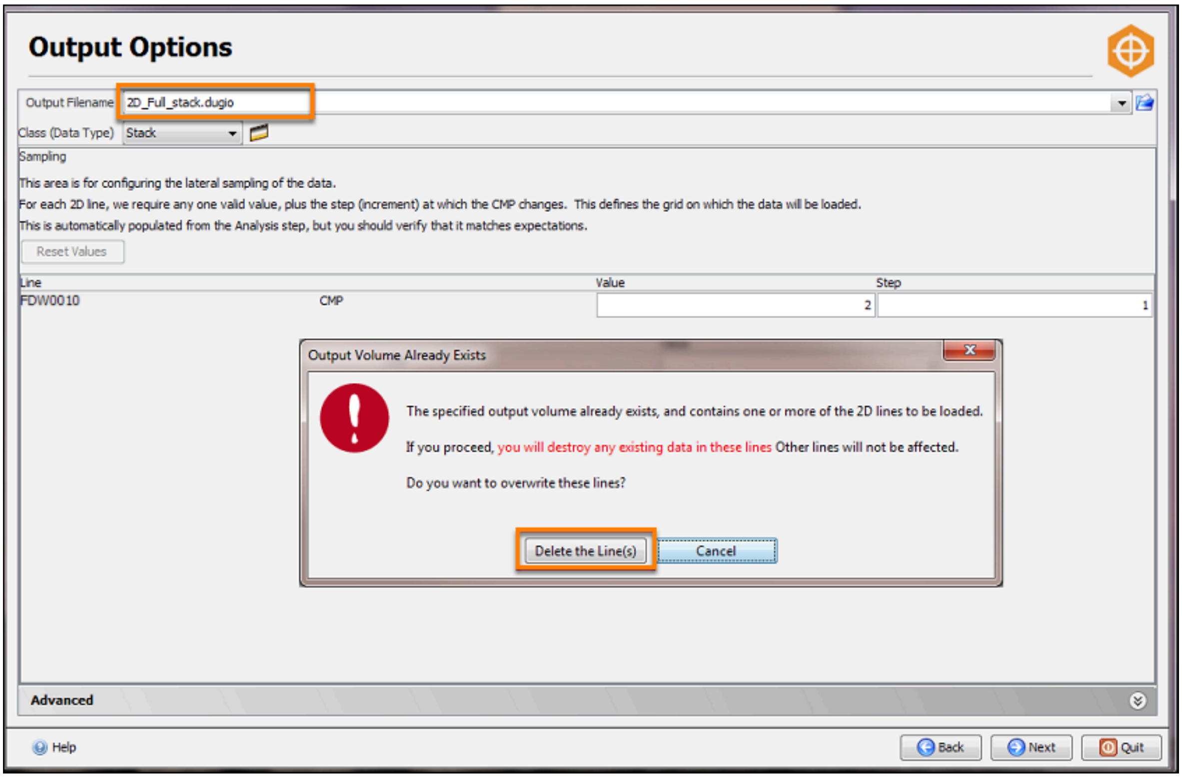
Add/replace 2D line



1. Follow the steps in *Importing 2D SEG-Y Data*.
2. In the SEG-Y Loader, select your lines and make sure the byte locations and configurations are correct.



3. In the case of adding a new line to an existing 2D dugio, select **Add to/edit existing survey**.
4. If the line had previously been loaded, it will already be defined in the survey - the loader should recognise it and force you to use the existing survey.



5. In the last panel (**Output Options**), select your existing *.dugio* file in the **Output Filename** text field.

6. Make sure that the volume you are adding the lines to is not loaded in your Insight session at the same time. Insight will need to delete the existing line(s), and will not be able to if the volume is currently loaded.
7. Click on **Delete the line(s)** when prompted.
8. Insight will load the data and overwrite the existing one.
9. You may reload the volume to your session to check the changes.

Genuine offer of assistance

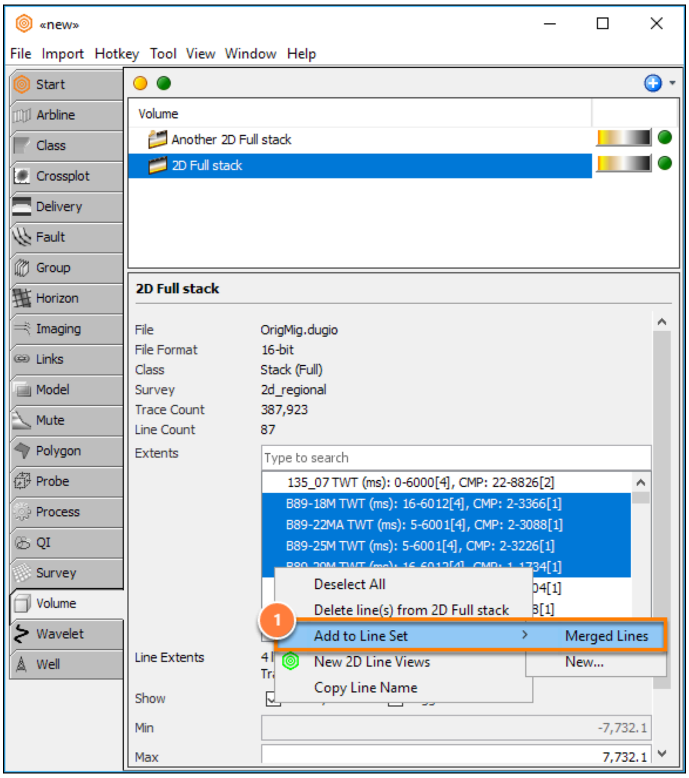
DUG understands that importing SEG-Y data can be a confusing task.

Whether you are using a trial version or a fully-paid licence, we will help you to import your data to Insight so that you will get the maximum out of your data.

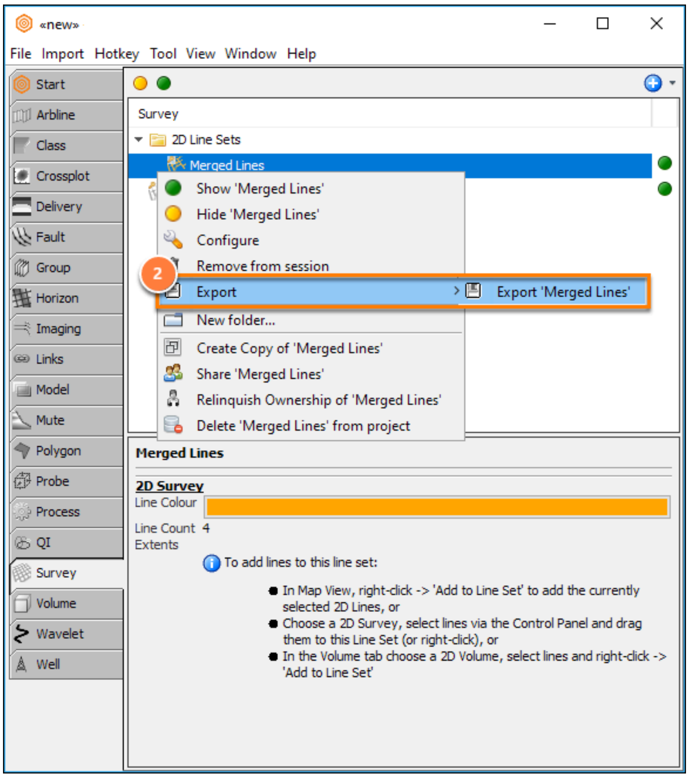
For more information, contact support at support@dugeo.com.

How do I merge lines from multiple 2D volumes and surveys into a single volume and survey?

This workflow can be used to efficiently consolidate 2D data contained in multiple 2D volumes spread over several surveys into a single 2D volume and survey.

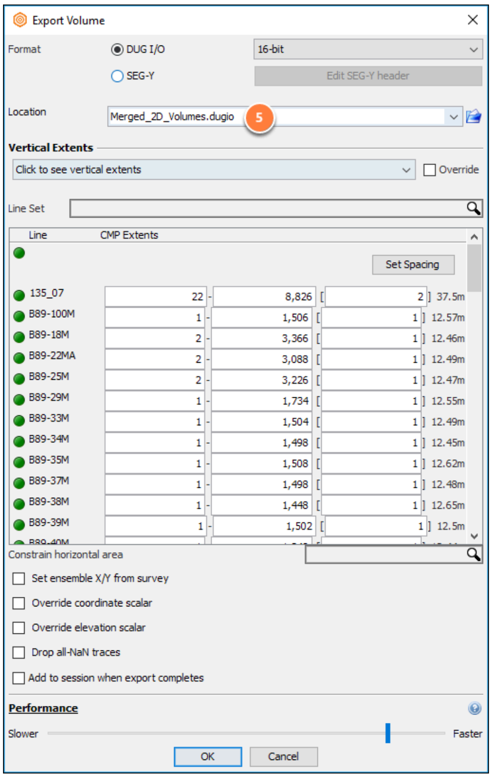


1. Create a line-set containing lines from both 2d surveys e.g. "Merged Lines" (see *Managing 2D line set*)
   - In the Volume tab, highlight each line name found in the **Extents** section, then right-click and select **Add to Line Set > New...**
   - Alternatively, select the lines in **Map** view by pressing Ctrl and clicking the lines. To add lines, right-click and select **Add lines to line set > New...**.



2. Export the line set e.g. Merged Lines (see [*Exporting 2D line sets*](#))
   - Go to the **Survey** tab, right-click on the line set, and select **Export > Export "line set**.

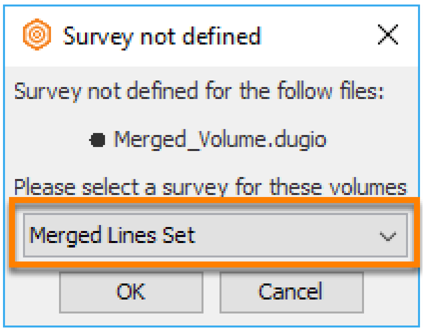
3. Import the line set as a 2D survey via **Control Panel > Import > Surveys...**



4. Export both 2D volumes to the same .dugio file (see [*Exporting to DUG I/O*](#)).

5. Give the first volume a name, select the location to export to, and click **OK**.
   - This new volume will contain the merged data.
6. To add lines to the merged volume from the second volume, use the same file name and location when exporting.

Note: Exporting lines from the second volume to the merged volume will not overwrite data already contained in the merged volume, unless the lines being export are already in the merged volume.



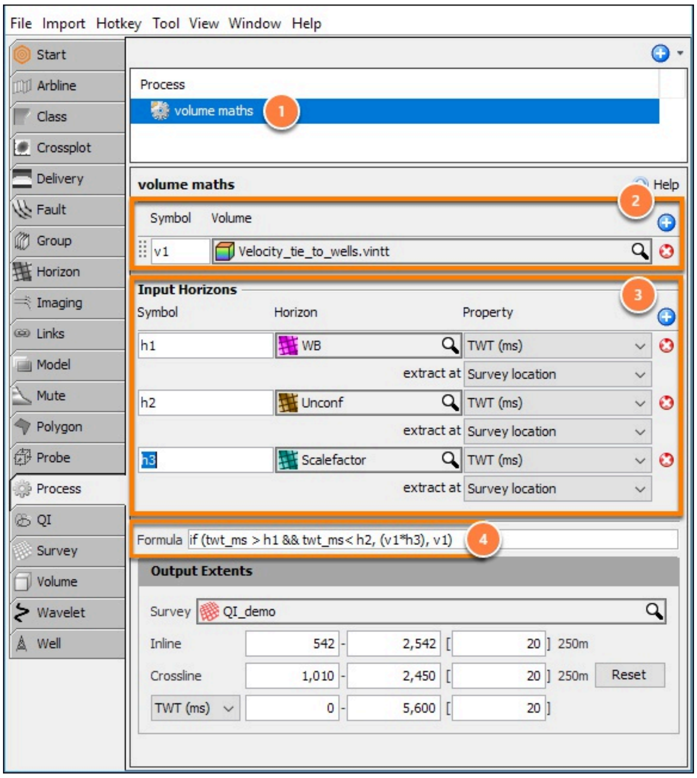
7. Import the merged volumes via **Control Panel > Import > Volume**.
8. When prompted to select a volume by the **Survey not defined** window, select the merged line survey imported in Step 2.

How do I gain a volume between two horizons, using a spatially varying multiplier?

In this workflow, we apply a gain to the values of a volume between two horizons. The amount of gain applied is defined by a third horizon.

It requires the following items loaded in the session:

- The volume to scale
- An upper bounding horizon
- A lower bounding horizon
- A horizon containing scalar (gain) values



1. Create a *Volume Maths* process.
2. Click the **Volume (+)** and select the volume to scale. Use the default variable name, "**v1**".
   - This example uses a volume called "*Velocity_tie_to_wells*".
3. Click the **Horizons (+)** three times to and add three horizons:
   - **h1**: choose the upper-bound horizon.
   - **h2**: choose the lower-bound horizon.
   - **h3**: choose the is the horizon (grid) based scale factor.

4. Use the formula:

```
if (twt_ms > h1 && twt_ms< h2, (v1*h3), v1)
```

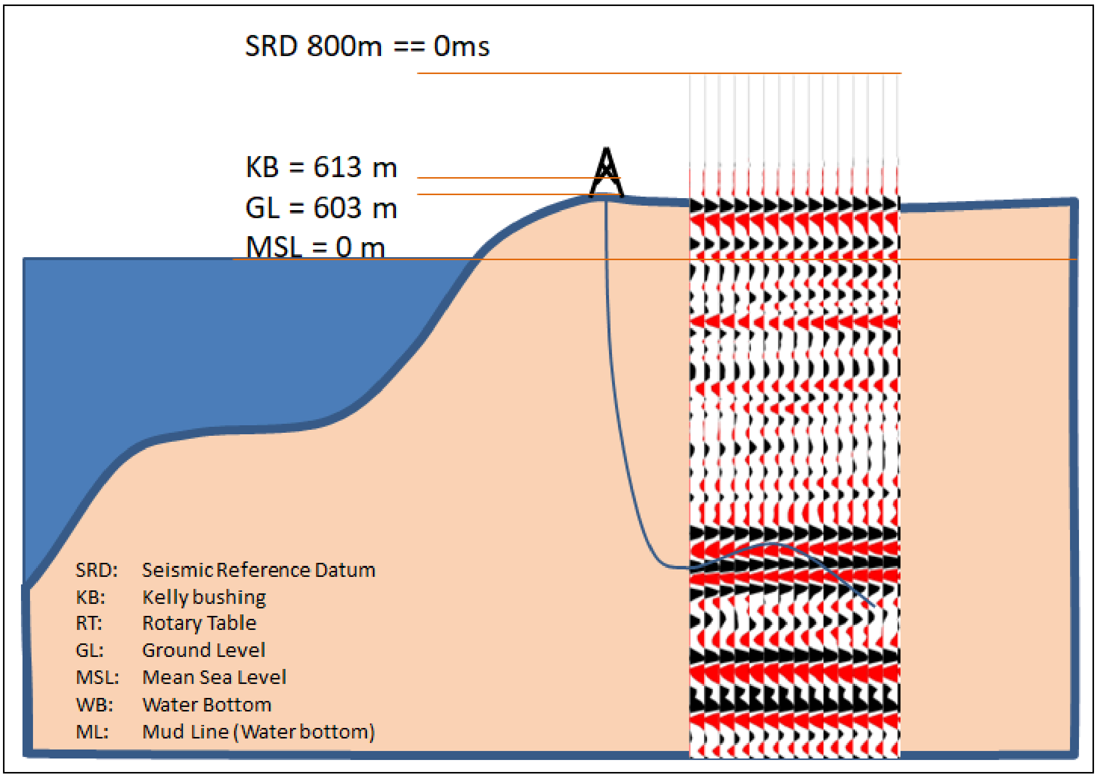
5. The result of this process is a new volume, located in the **Volume** tab. **Note**: Use the **CTRL-P** hotkey to quickly switch between a process and its result volume.
6. Go to **View > New IL/CL View** or **3D View** to QC the result.

Seismic Reference Datums

Marine seismic is usually referenced to Mean Sea Level (MSL) which is a global datum for measuring height. In a marine project, zero elevation (the mean sea level) will have a time of 0 ms.

Because elevation on land is variable, land seismic is often referenced to a Seismic Reference Datum (SRD). The SRD is a plane that sits above the highest elevation in the data. In a land project with a seismic reference datum set, the SRD will have a time of 0 ms. All time data, e.g. seismic and velocities, will start at this datum.

The seismic reference datum will be above the elevation of most of the land surface. For seismic, values in the is gap will be empty. In velocity models, a replacement velocity fills the data gap between the surface and the datum. It's an important value to note when manipulating volumes with different datums.



In a project, all wells and traces are referenced to MSL enabling objects to be compared to each other.

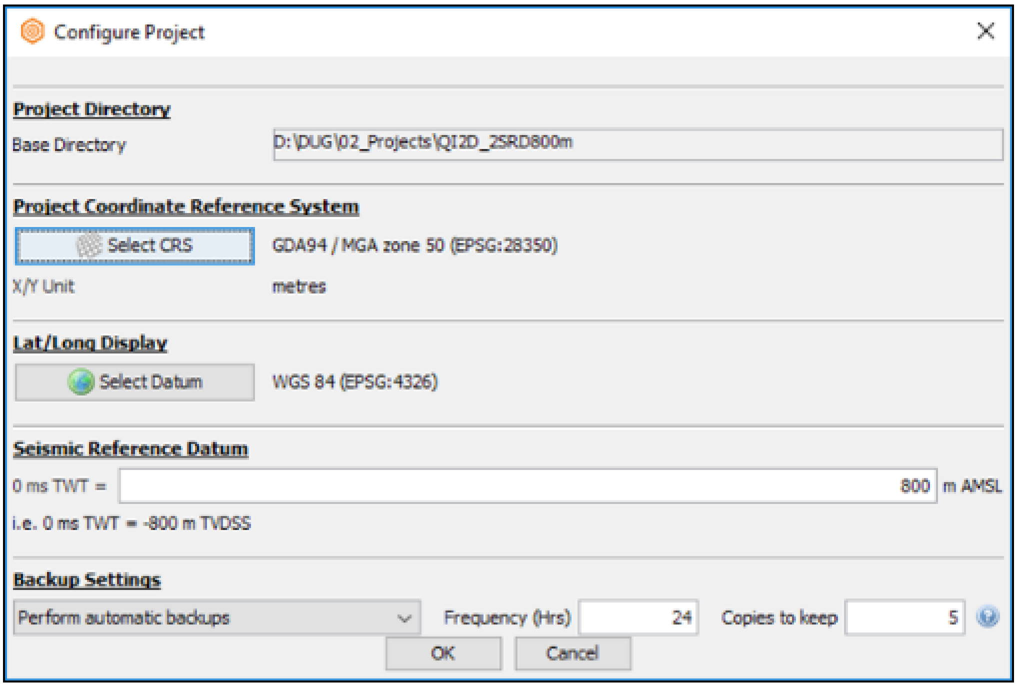
- Seismic is Measured from MSL or SRD
- Well data is most commonly referenced to KB, DF or RT.
- Checkshots are most commonly referenced to SRD, MSL and sometimes KB
- GL or ML(WB) can also be used but are not as common.

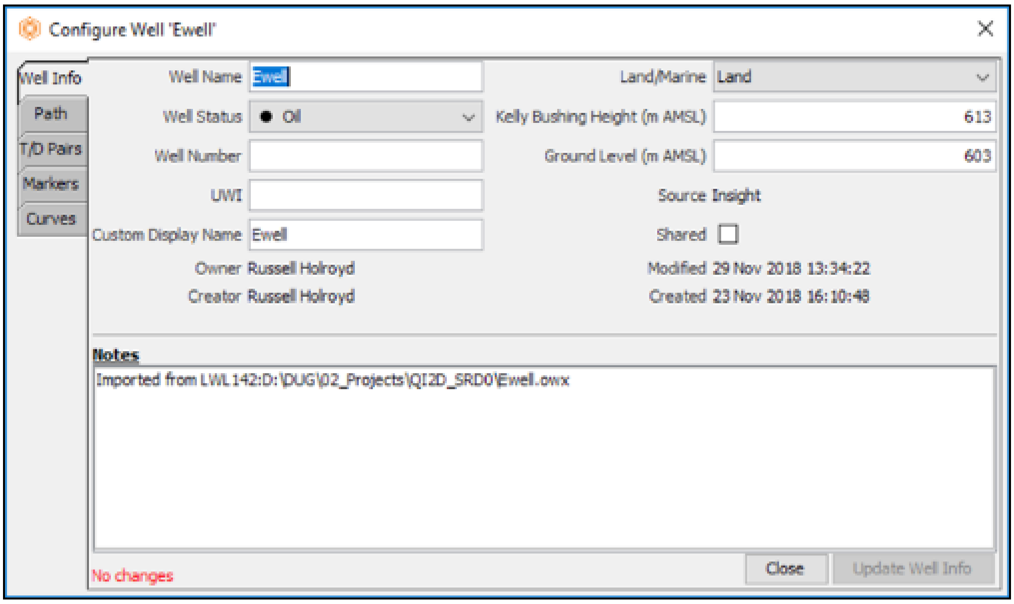
In Insight, the default seismic reference datum is 0m. With marine data you do not need to change this value. There are two ways to configure a project with land data referencing a non zero SRD.

Method 1 - Set a seismic reference datum for the project

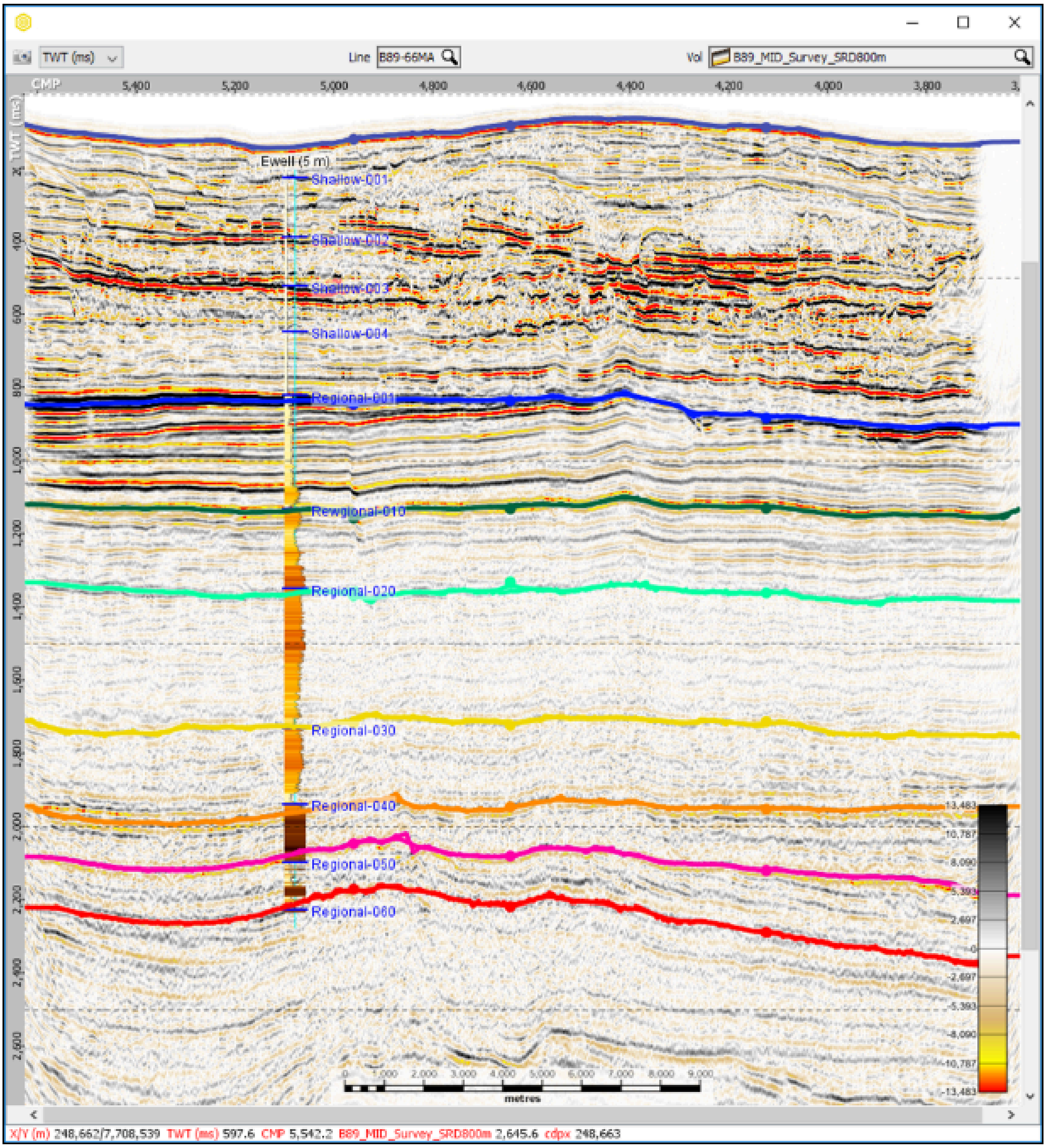
In **Configure project**, set the SRD to match the seismic survey SRD.

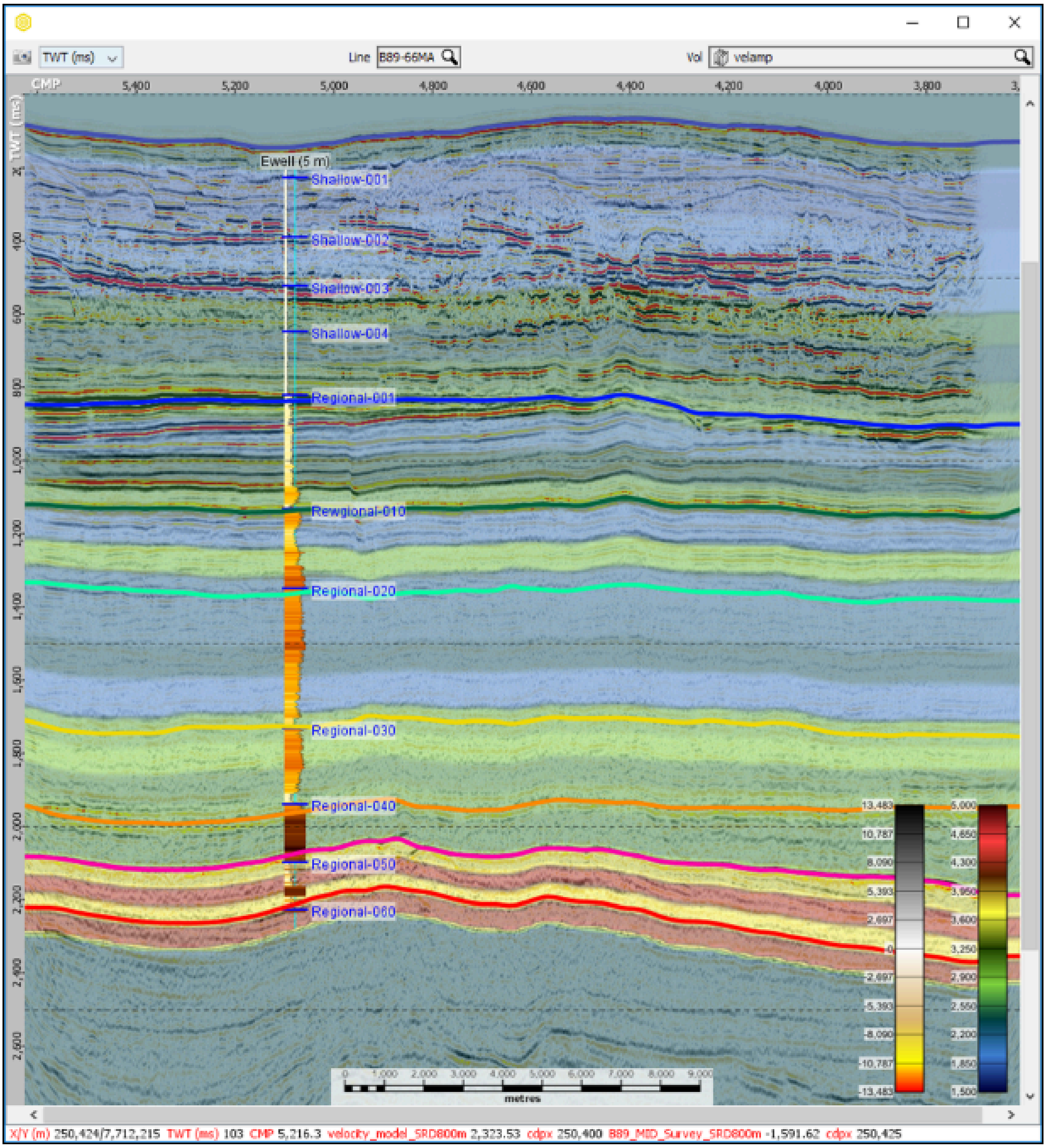
In this example 800 m = 0 ms. The replacement velocity is 2000 m/s.





The well has its own datum set at 613 m above MSL.

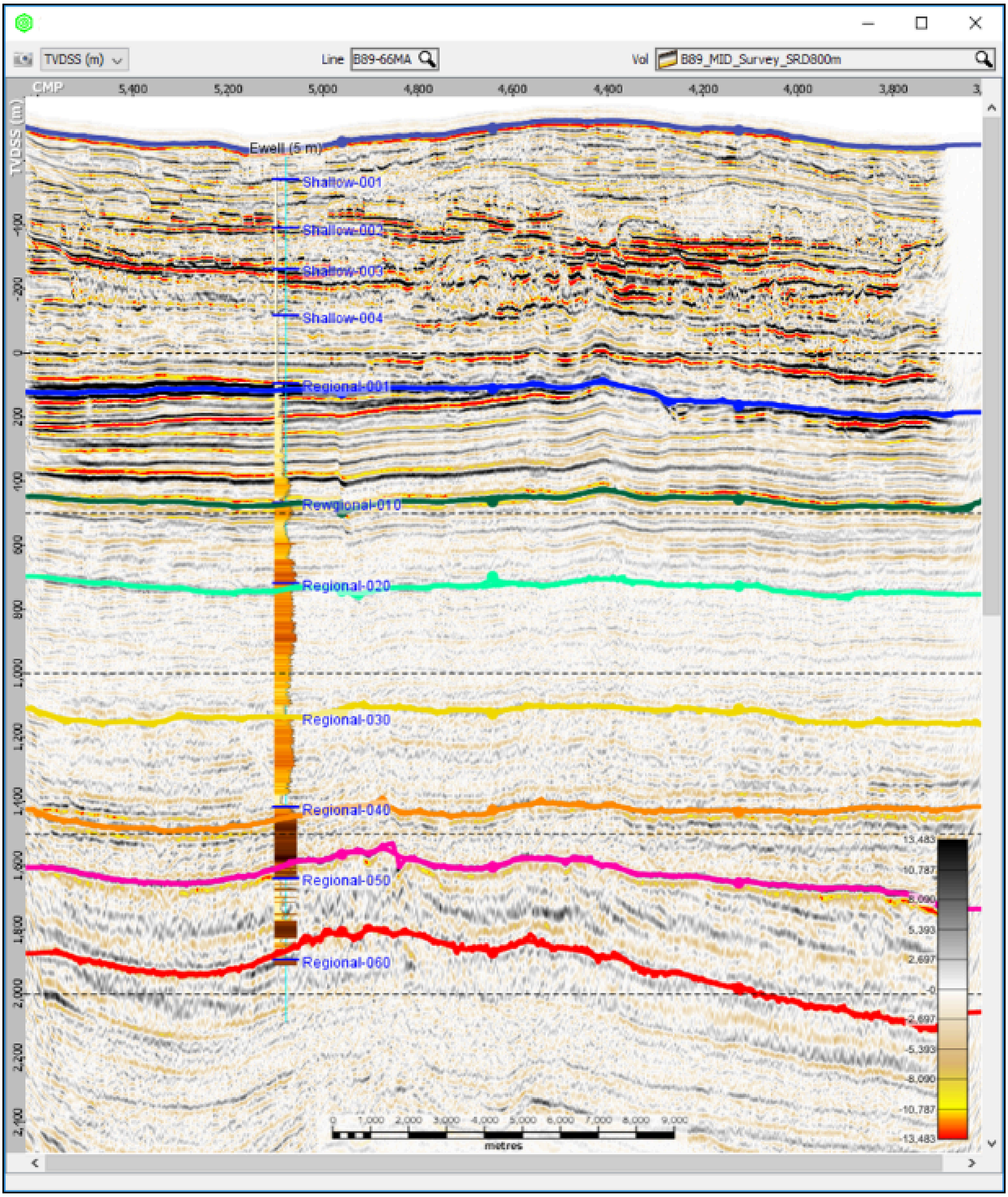

TWT (ms)
Line B89-66MA
Vol B89_MID_Survey_SRD800m
CMP
5,400
5,200
5,000
4,800
4,600
4,400
4,200
4,000
3,800
Ewell (5 m)
Shallow-001
Shallow-002
Shallow-003
Shallow-004
Regional-001
Rewgional-010
Regional-020
Regional-030
Regional-040
Regional-050
Regional-060
13,483
10,787
8,090
5,393
2,697
0
-2,697
-5,393
-8,090
-10,787
-13,483
0
1,000
2,000
3,000
4,000
5,000
6,000
7,000
8,000
9,000
metres
X/Y (m) 248,662/7,708,539 TWT (ms) 597.6 CMP 5,542.2 B89_MID_Survey_SRD800m 2,645.6 cdpx 248,663



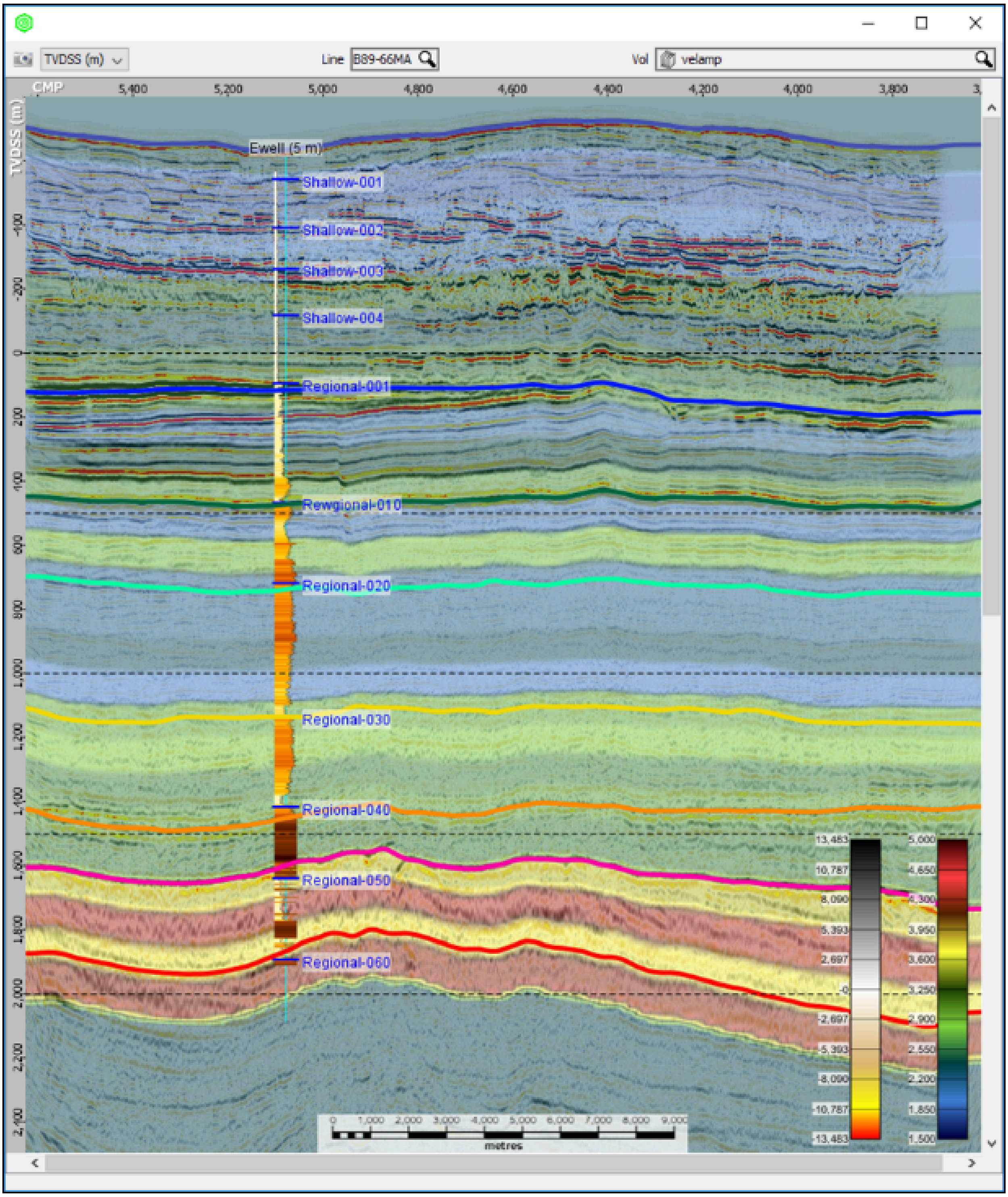
With the seismic reference datum configured as 800m:

In TWT

- The line at time zero is the SRD, 800 m above MSL
- Unconverted TWT traces begin at time zero, all the samples are after time zero.
- The entire well converted to TWT is visible.
- The wells and seismic tie.
- Unconverted TWT-Interval Velocity data (inset image) starts from time zero.



TVDSS (m)
Line B89-66MA
Vol B89_MID_Survey_SRD800m
CMP
Ewell (5 m)
Shallow-001
Shallow-002
Shallow-003
Shallow-004
Regional-001
Rewgional-010
Regional-020
Regional-030
Regional-040
Regional-050
Regional-060
metres



In TVDSS

- The line at depth zero is the MSL.
- TWT traces converted to TVDSS start at the SRD, 800 m above MSL.
- The unconverted depth well is fully visible.
- The wells and seismic tie.
- The TWT-Interval Velocity data (inset image) converted to TVDSS starts from time zero, 800 m above MSL

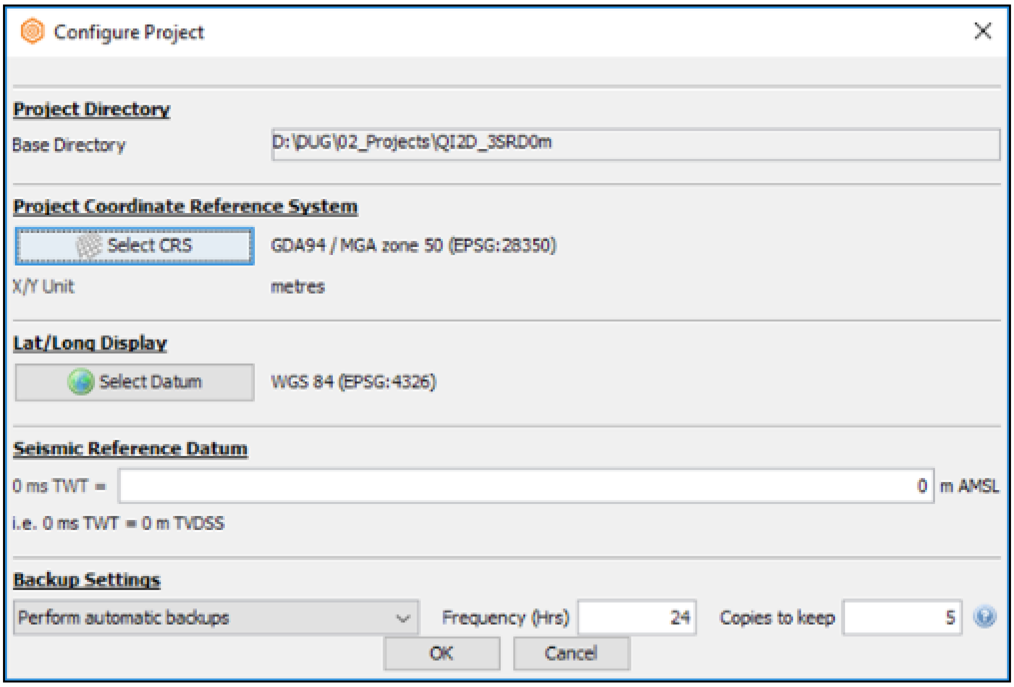
This is ideal. All the data is displayed in both TWT and TVDSS, and the velocity model extends to the seismic reference datum. Time-depth conversion using the model covers the entire vertical domain.

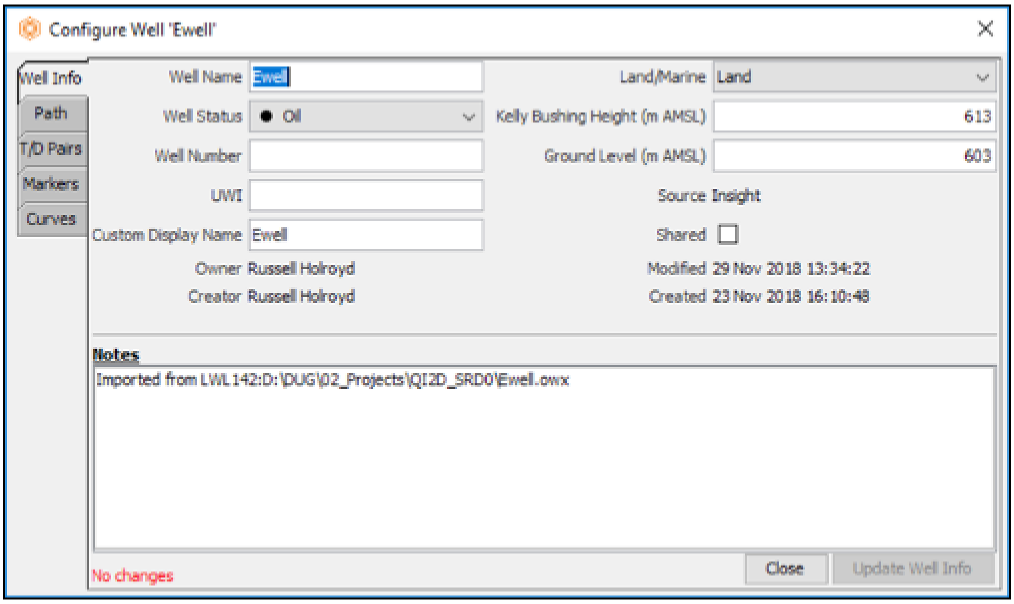
Method 2 - Use Mean Sea Level (0m) as the seismic reference datum

The trace data was originally processed using a SRD of 800m. The time traces have been shifted using replacement velocity of 2000 m/s from the original SRD of 800 m to a new 0 m MSL datum.

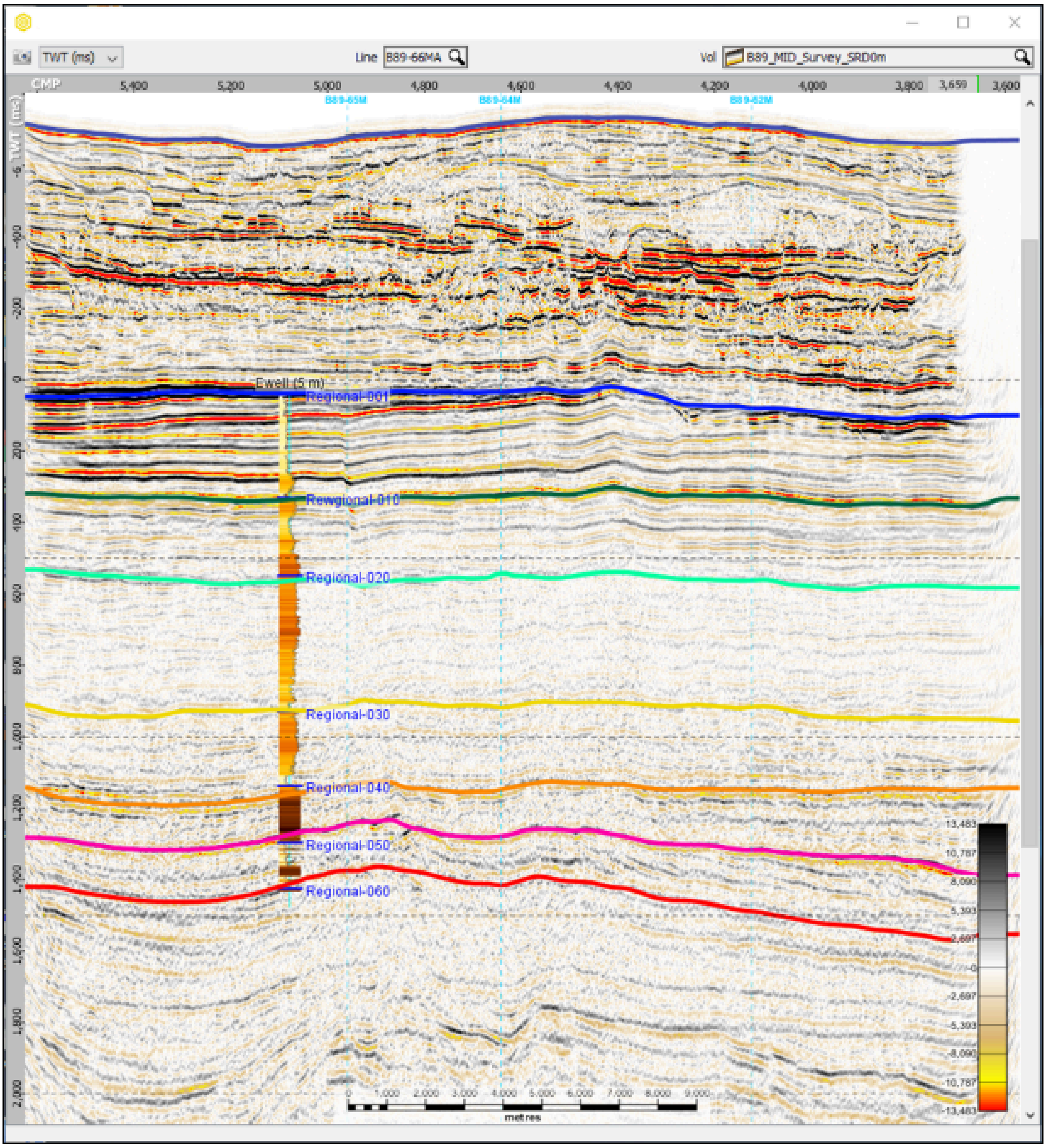
In **Configure project**, set the SRD to 0m (MSL).

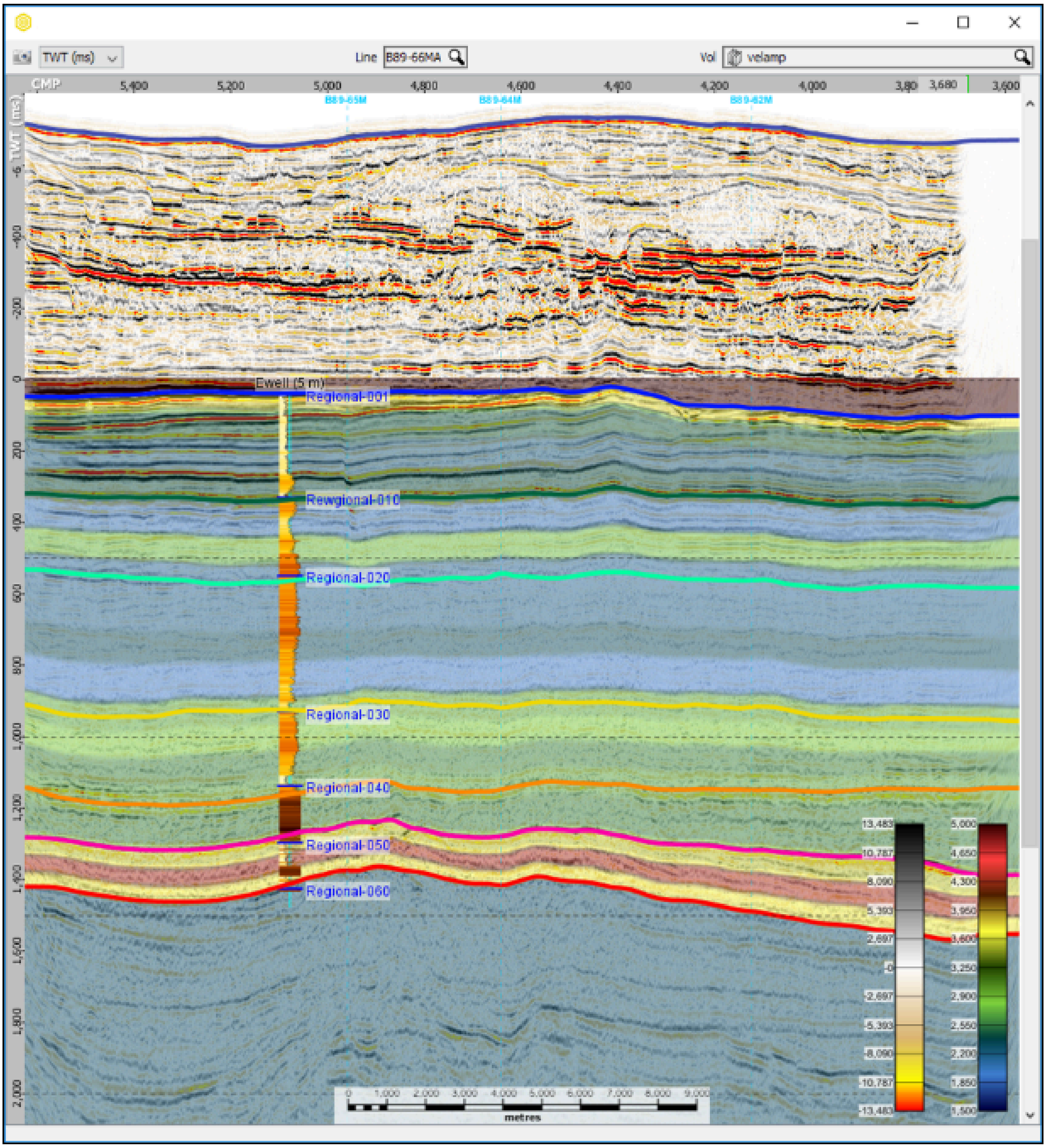
In this example: 0 m = 0 ms.





The well has its own datum set at 613 m above MSL.

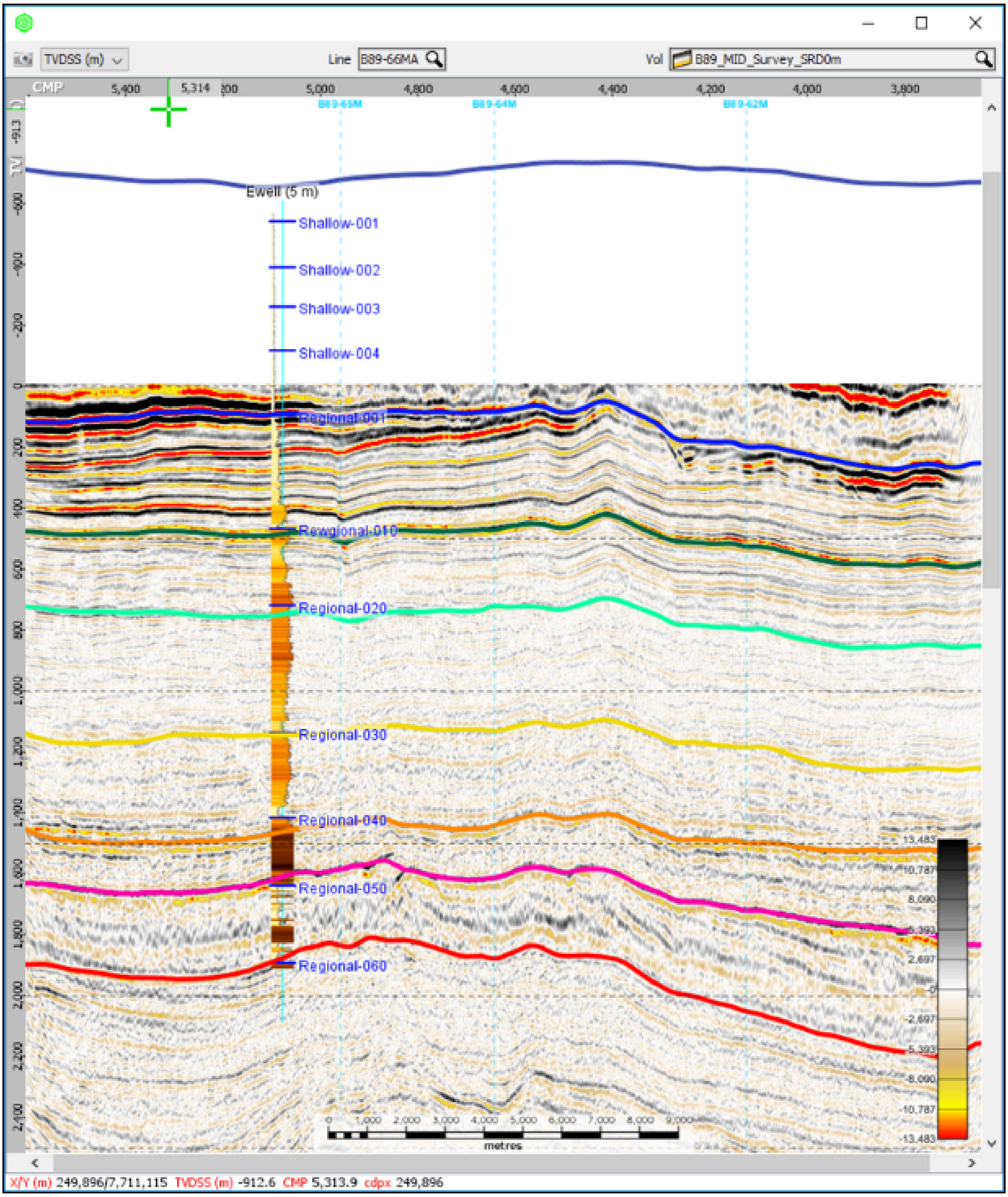

TWT (ms)
Line B89-66MA
Vol B89_MID_Survey_SRD0m
CMP
5,400
5,200
5,000
4,800
4,600
4,400
4,200
4,000
3,800
3,659
3,600
B89-65M
B89-64M
B89-62M
Ewell (5 m)
Regional-001
Rewgional-010
Regional-020
Regional-030
Regional-040
Regional-050
Regional-060
0
200
400
600
800
1,000
1,200
1,400
1,600
1,800
2,000
13,483
10,787
8,090
5,393
2,697
0
-2,697
-5,393
-8,090
-10,787
-13,483
0
1,000
2,000
3,000
4,000
5,000
6,000
7,000
8,000
9,000
metres



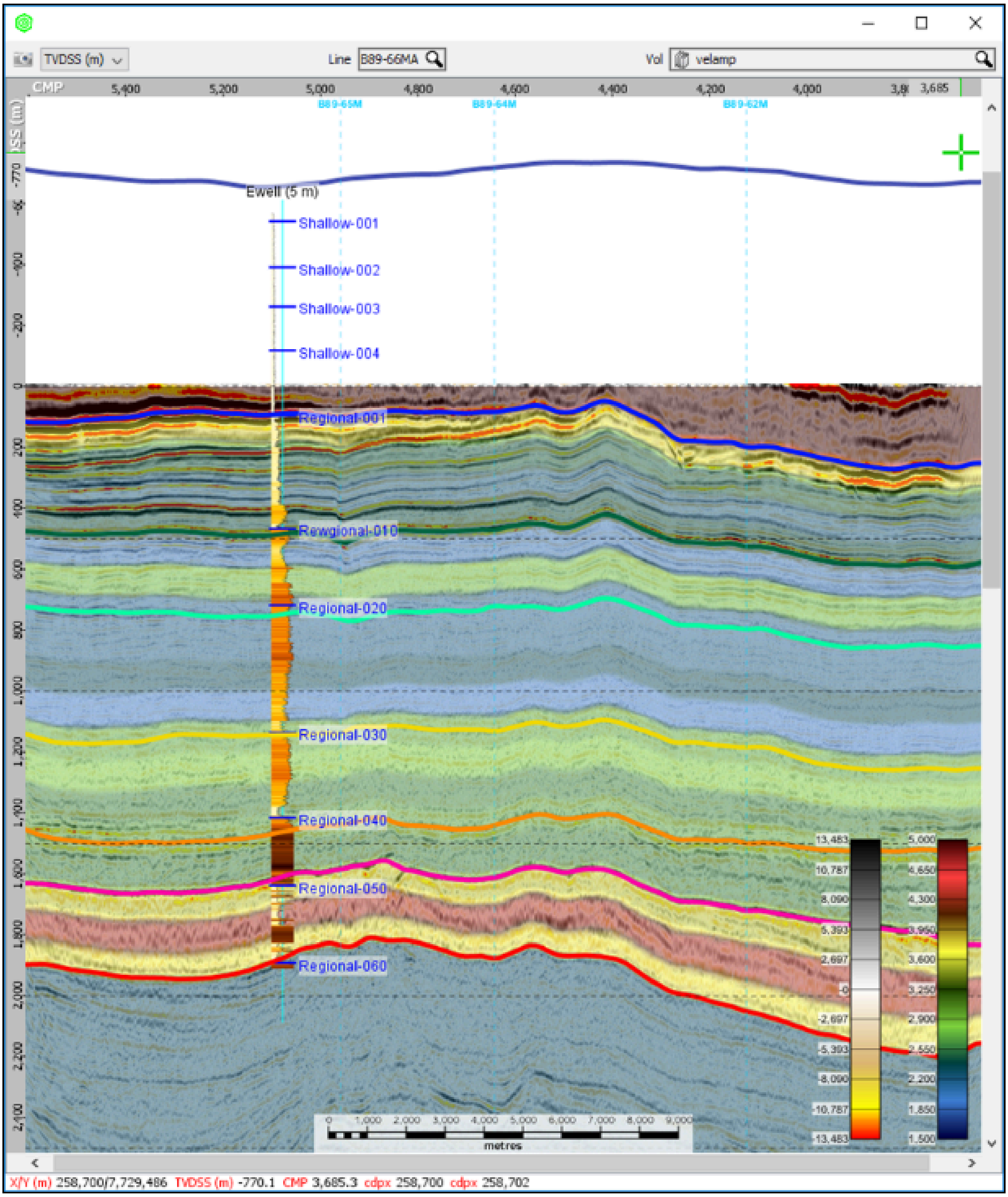
With the seismic reference datum set to MSL (0 m):

In TWT

- The line at time zero is the MSL
- Unconverted TWT Traces begin at time -800 ms, the time shift calculated from the original SRD using a replacement velocity of 2000 m/s.
- The entire TWT trace is visible above and below time zero (MSL).
- The well converted to TWT is only visible below 0 ms. “Negative time” values are not displayed.
- Wells and seismic tie for the visible portion.
- Unconverted TWT-Interval Velocity data starts at time zero. “Negative time” values are not displayed.



TVDSS (m)
Line B89-66MA
Vol B89_MID_Survey_SRD0m
CMP
5,400
5,314
5,000
4,800
4,600
4,400
4,200
4,000
3,800
B89-65M
B89-64M
B89-62M
Ewell (5 m)
Shallow-001
Shallow-002
Shallow-003
Shallow-004
Regional-001
Rewgional-010
Regional-020
Regional-030
Regional-040
Regional-050
Regional-060
13,483
10,787
8,090
5,393
2,697
0
-2,697
-5,393
-8,090
-10,787
-13,483
metres
X/Y (m) 249,896/7,711,115 TVDSS (m) -912.6 CMP 5,313.9 cdpx 249,896



In TVDSS

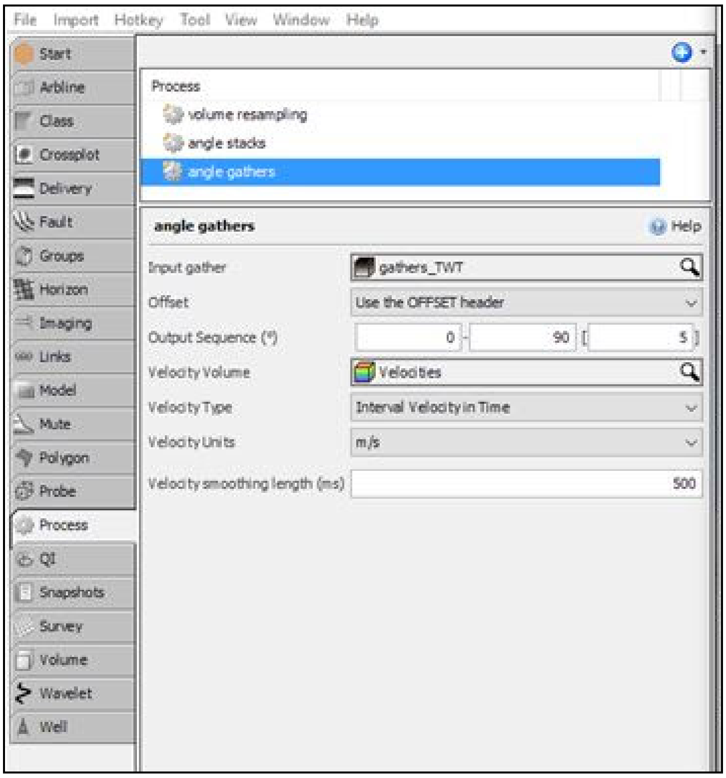
- The line at depth zero is the MSL.
- The TWT traces converted to TVDSS are only visible below 0 m. “Negative Depth” values are not displayed.
- The entire unconverted Depth well is visible
- The wells and seismic tie for the visible portion
- The TWT Interval Velocity data (inset image) converted to TVDSS is only visible below 0 m. “Negative Depth” values are not displayed.

This method truncates well data at MSL when displayed in TWT, and seismic time data at MSL when displayed in TVDSS. The velocity model is modelled from below MSL, with only the constant replacement velocity available to use above MSL.

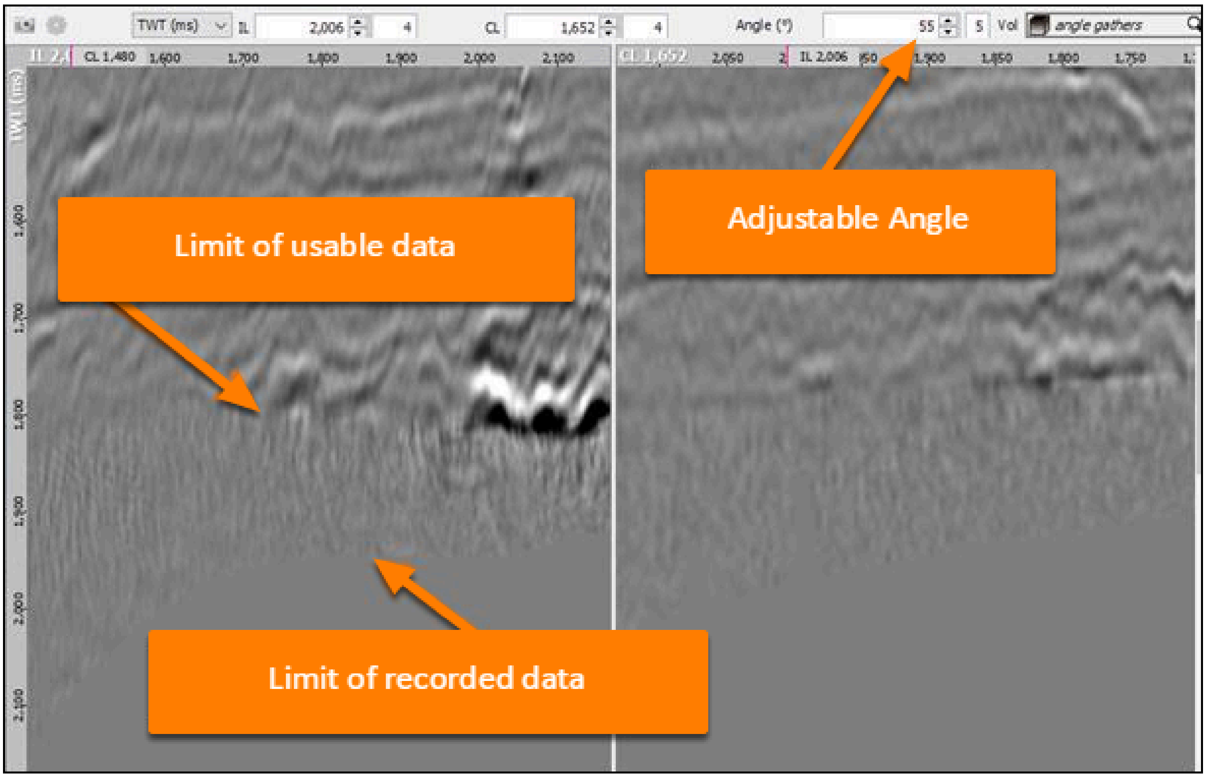
For a single seismic reference datum this isn't the best method, but this method becomes useful when using multiple surveys with multiple seismic reference datums. After defining a common seismic reference datum, each survey is shifted to align with each other.

A quick way to determine the angle to stack your gather data

Looking for the optimal angle to stack your offset gathers? This simple approach visually inspects gathers using our angle gathers process. Quickly distinguish between possible noise and real data.



1. In the **Control Panel**, select the **Process** tab.
2. Click the **blue (+) icon** and create a new *Angle gathers process*.
   - **Input gather**: Select the gather volume.
   - **Offset:** Use the OFFSET header or Calculate offset from SX/SY/GX/GY.
   - **Output Sequence:** Specify the first angle, last angle and increment of the angle gathers to create.



3. View the output in the *IL/CL view*, using the angle selector to scroll through the angles.

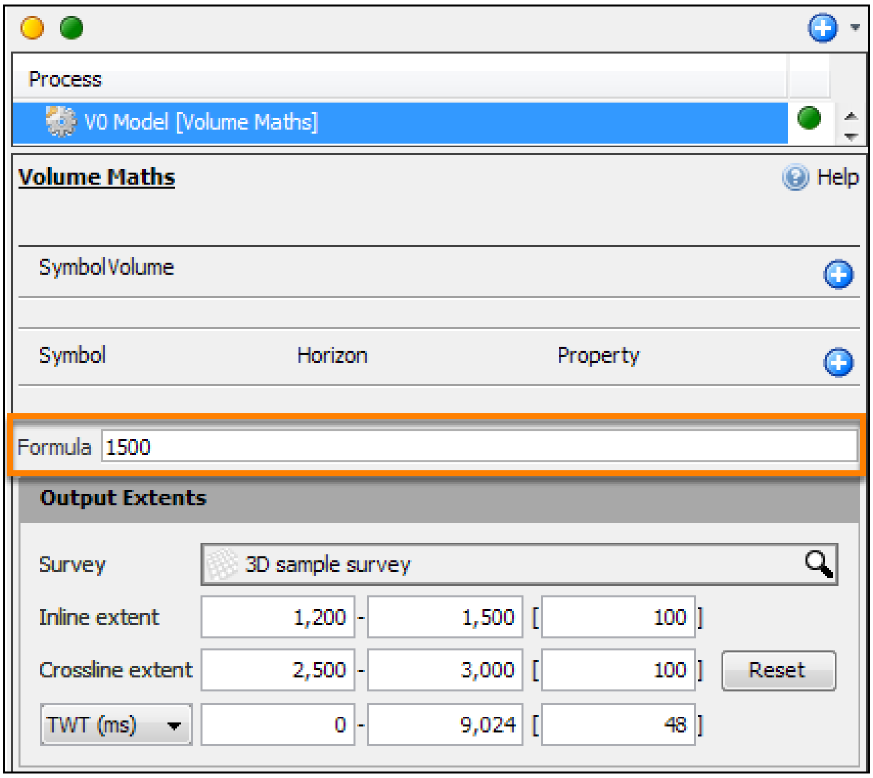
Velocity

What are some good ways to create synthetic velocity models?

The following describes the workflow to create a Vo+kz velocity model with varying gradients between horizons for the purpose of time/depth conversion.

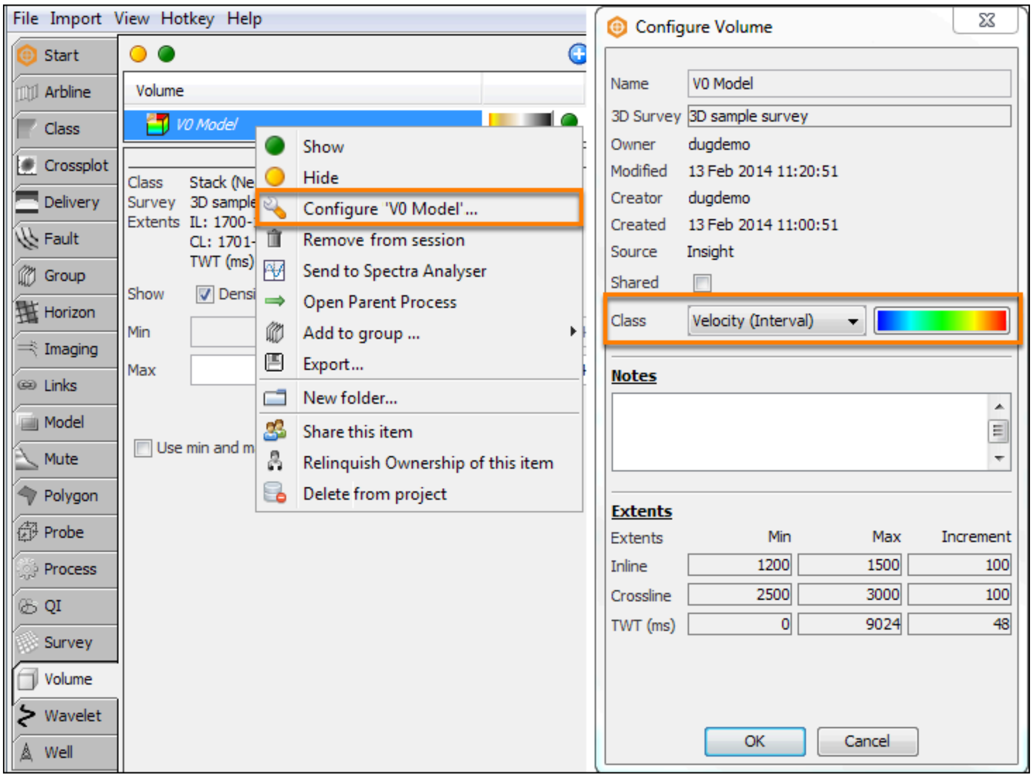
- *A simple V0 model*
- *A simple V0 + k(z) model*
- *A layered V0 model*.

Create a simple V0 model



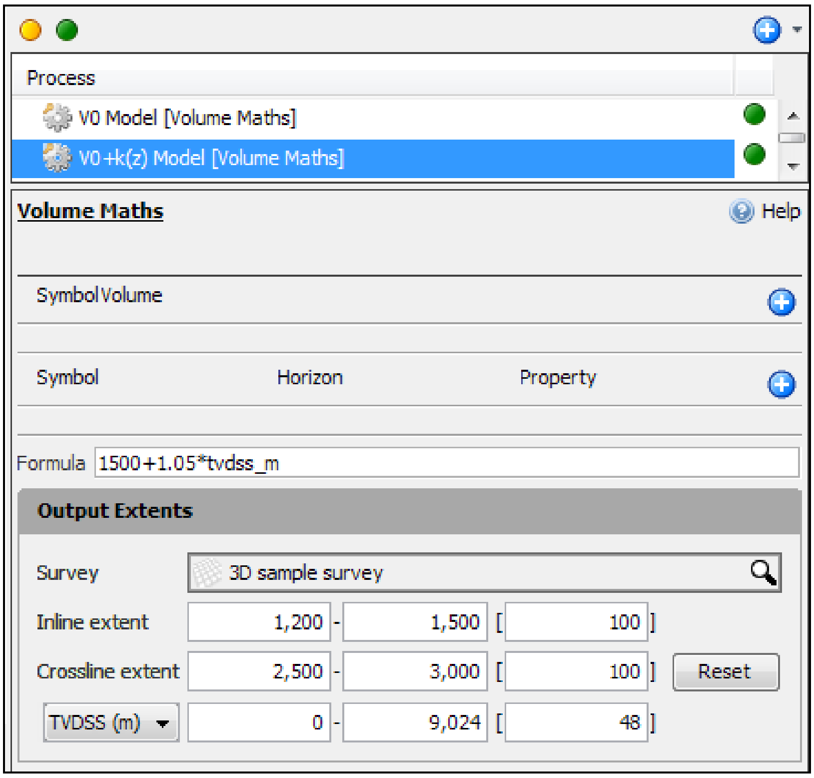
1. In the **Process** tab, click on the Add icon and select **New Process**.
2. Create a **Volume Maths** process and type a **Name** for the process. Click **OK**.
3. In the Details Panel, at **Formula**, type the formula **1500** to create a velocity model at a constant value of 1500.
4. Under the **Output Extents** section, select a 3D survey.
5. Enter the appropriate **Inline** and **Crossline extents** for the model.
6. Models can be created in **TWT** or **TVDSS**, depending on the domain selected in the output extents.
7. The velocity model will appear in the **Volume** tab.

Assign a velocity class to the model

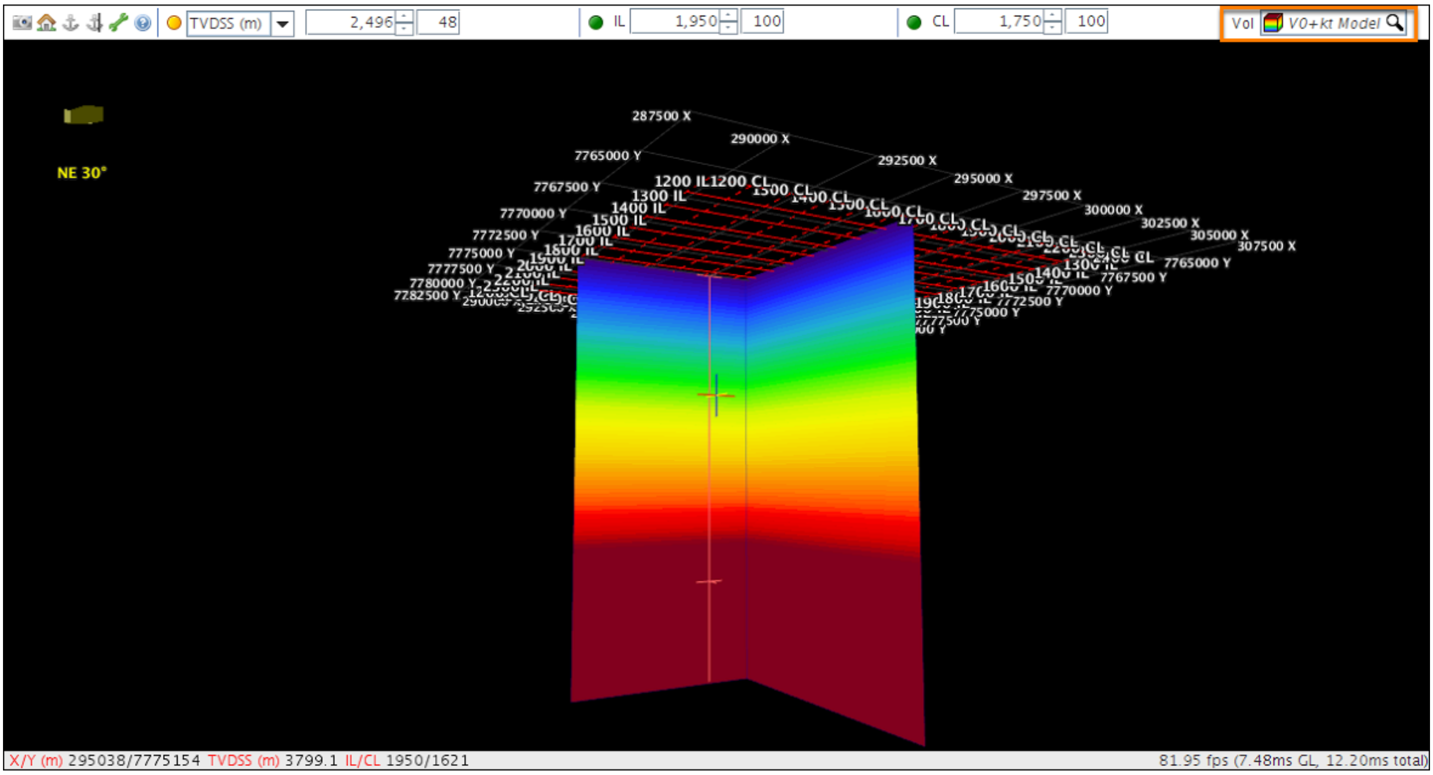


1. In the **Control Panel**, open the **Volume** tab.
2. Double click the velocity volume or right-click and select **Configure 'Volume'**.
3. At **Class**, select **Velocity (Interval)** from the drop-down list.
4. Click **OK**.

Create a simple V0 + k(z) model

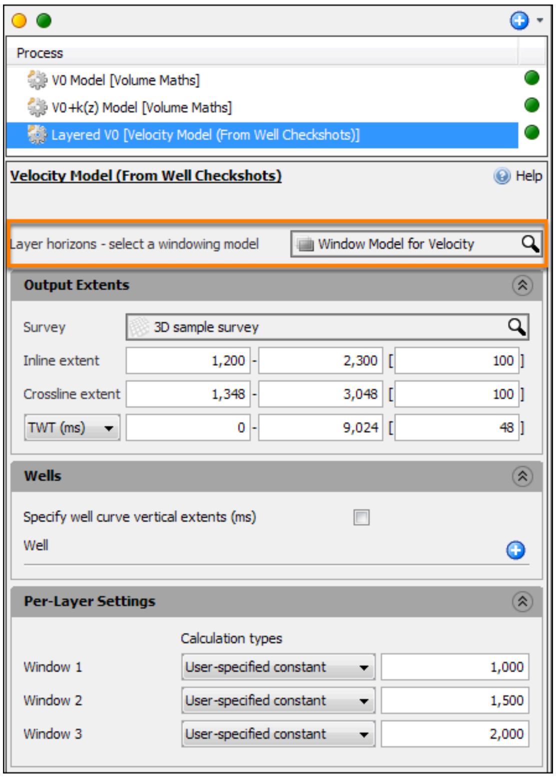


1. In the **Process** tab, click on the Add icon and select **New Process**.
2. Create a **Volume Maths** process and type a **Name** for the process. Click **OK**.
3. In the Details Panel, at **Formula**, type the formula **1500+1.05*tvdss_m**.
4. Under the **Output Extents** section, select a 3D survey.
5. Enter the appropriate **Inline** and **Crossline extents** for the model, and select TVDSS as the output domain.
6. The velocity model will appear in the **Volume** tab.
7. In the **Volume** tab, double click the velocity volume or right-click and select **Configure 'Volume'**.
8. Change the **Class** of the volume to **Velocity (Interval)** from the drop-down list.

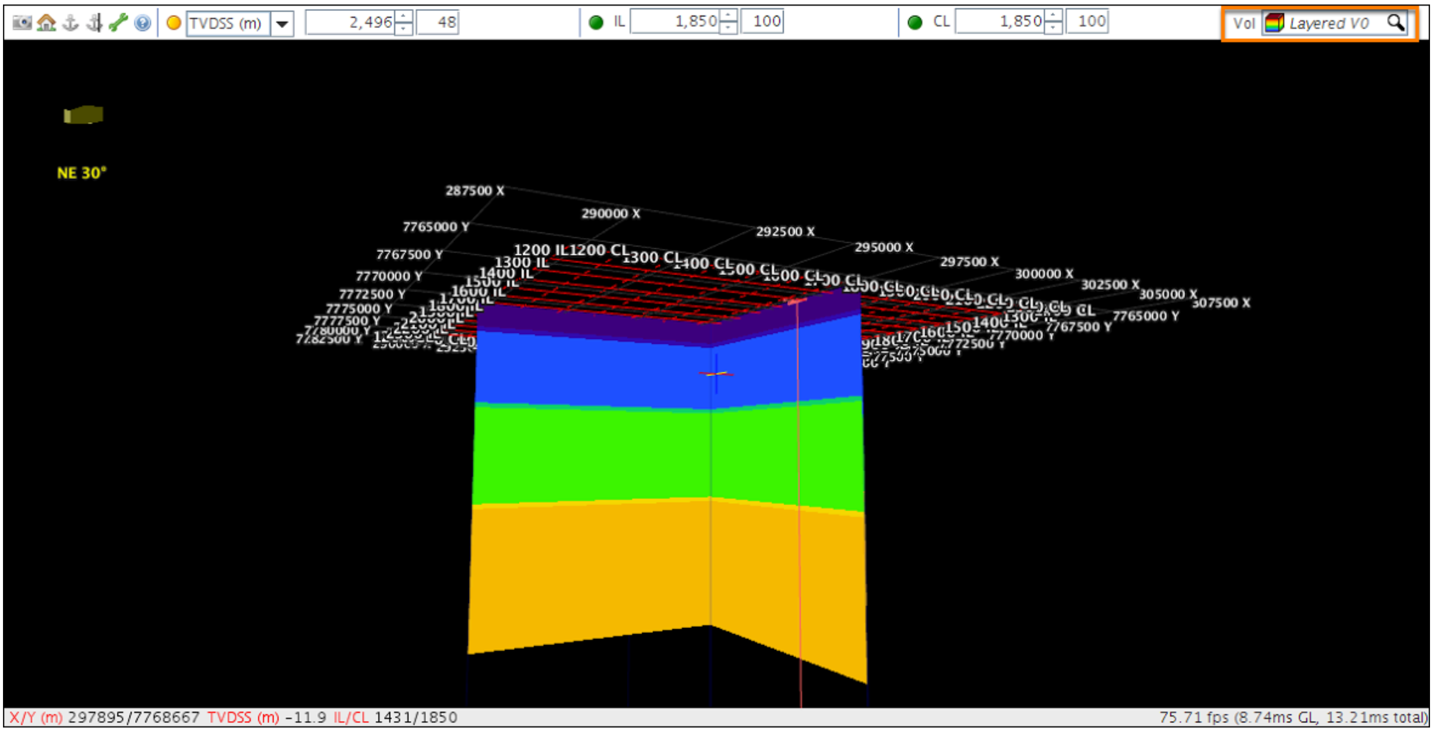


Create a layered V0 model

1. Create a Window Model using constants or horizons (see *Creating a Window Model*).
   Note: If you are creating a velocity model in depth, you must create a window model in depth.
2. In the **Process** tab, click on the Add icon and select **New Process**.
3. Create a ***Velocity Model (From Well Checkshots)*** process and type a **Name** for the process. Click **OK**.



4. Select the **Windowing Model** that you have created.
5. Under the **Output Extents** section, select a 3D survey.
6. Enter the appropriate **Inline** and **Crossline extents** for the model.
7. Under the **Per-Layer Settings** section, change the **Calculation types** to **User-specified constant** and enter an appropriate value in the adjacent box.



How do I create a velocity model from time-depth (T-Z) pairs?

There are a few approaches to creating a velocity model from T-Z pairs to be used as a velocity volume in time-depth conversion.

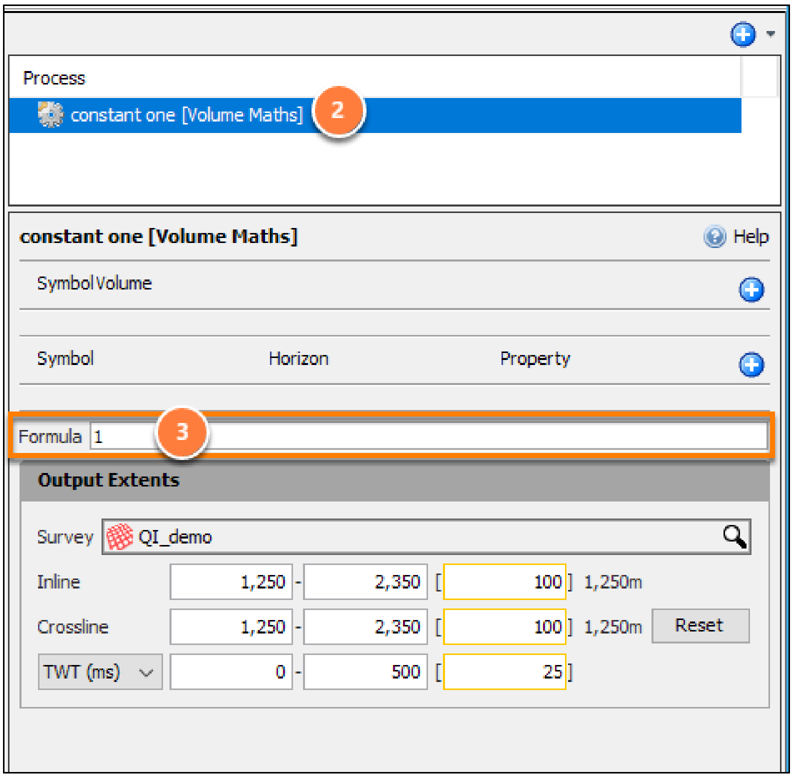
Depending on your data, see the following example workflows:

- *Workflow for 3D data*
- *Workflow for 2D data*.

For 3D data

For 3D data, we recommend creating a constant volume to define the model size, then scale it using a gain function defined by the T-Z pairs.

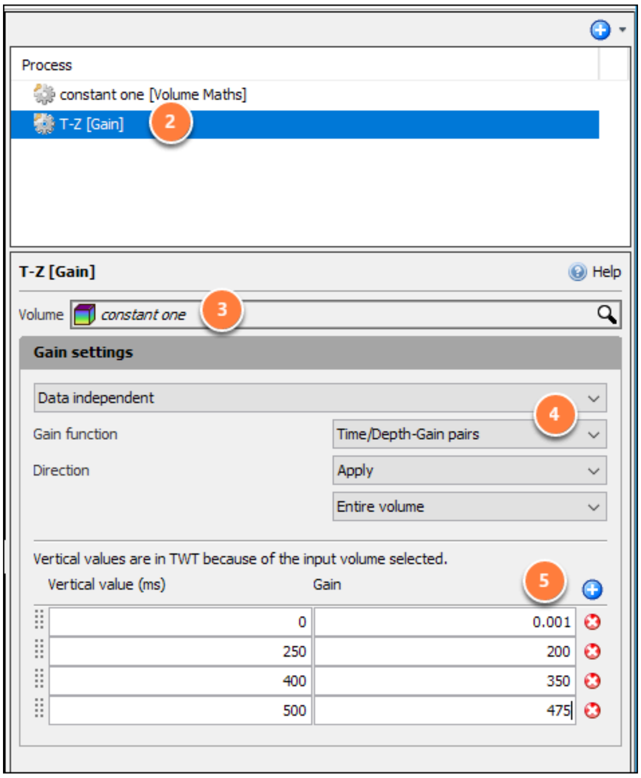
Create a constant volume to define the model size



1. In the **Process** tab, click the Add icon and select **New Process**.

2. Select **Volume Maths** from the list and type a name for the process (see *Volume Maths*). In this example, let's call it *CONSTANT ONE*.
3. Input a constant (in this case: 1) as your **Formula** and configure the **Output Extents**.

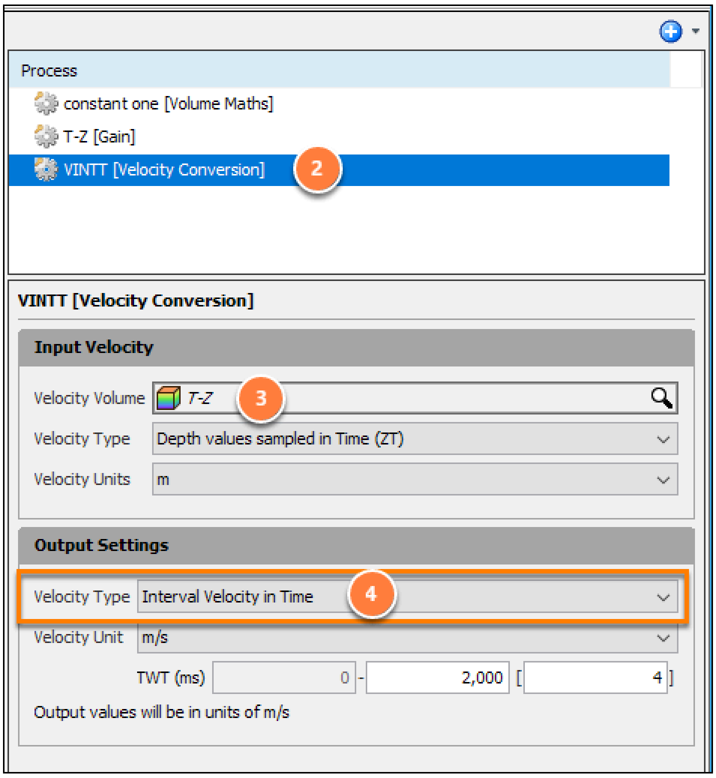
Create a gain function of the T-Z pairs



1. In the **Process** tab, create a **New Process** and select **Gain** from the list (see *Gain*).
2. Give the process a name and click OK. In this example, let's call it *T-Z*.
3. In the Details Panel, select the constant volume that you have created.
4. Choose **Data independent** and **Time/Depth-Gain pairs** as your gain function.
5. Click the Add icon at **Vertical values** to apply gains to the volume. The new process volume will be created in the **Volume** tab.
6. Set this volume's class by right-clicking (or double-click) it in the **Volume** tab and select **Configure**.
7. In the **Class** dropdown box, select **Velocity (TVDSS values sampled in TWT)** and click **OK**. Read more about *Changing a Volume's Class*.

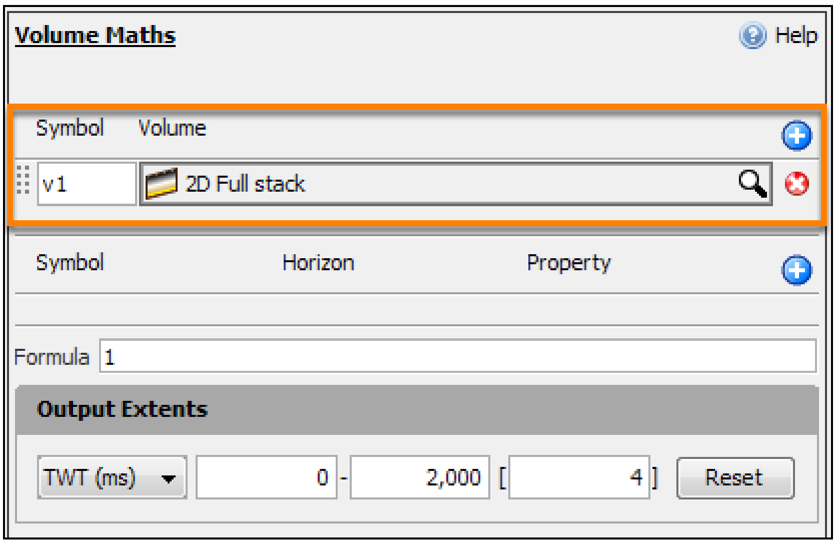
Note: At least 2 time/depth pairs must be provided.

Convert the T-Z result to interval velocity (VINTT)



1. In the **Process** tab, create a **New Process** and select **Velocity Conversion** from the list (see *Velocity Conversion*).
2. Give the process a name and click OK. In this example, let's call it *VINTT*.
3. Select the gain function (*T-Z*) as your **Velocity Volume**.
4. In the **Output Settings**, choose to convert the T-Z result to **Interval Velocity**.

For 2D data



If you are working with 2D data instead, select your 2D volume as an input when creating the constant volume. Insight will detect a 2D volume and change the Output Extents section accordingly.

The rest of the workflow remains the same as if you were using 3D data (refer to the workflow above).

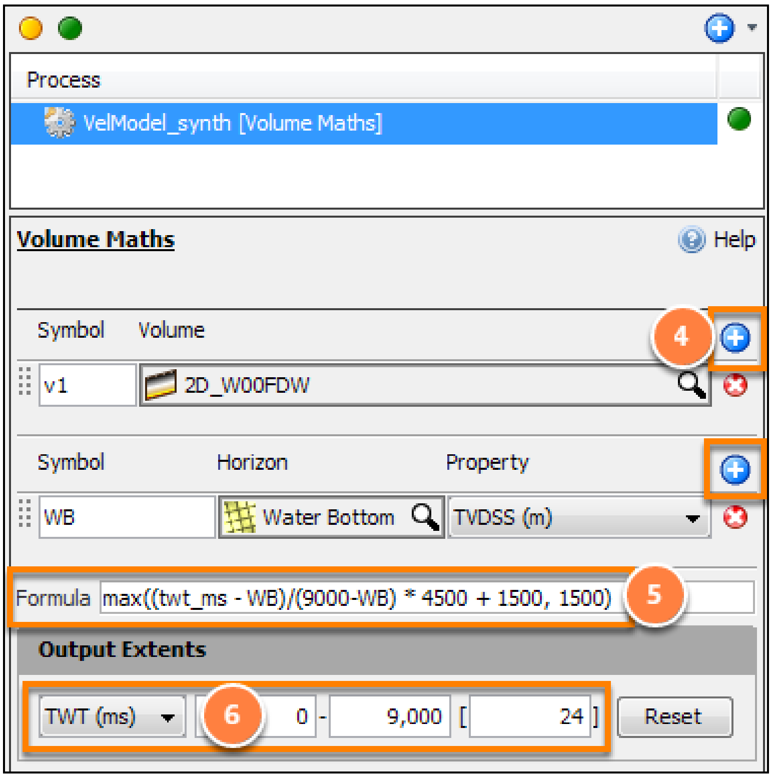
For more information on velocity models for 2D data, see *How can I create a simple velocity model for 2D data?*

How can I create a simple velocity model for 2D data?

A velocity model can be constructed using Volume Maths (see *Volume Maths*). While the equation may be a little daunting, the result is very flexible. For 2D data, we will use stack data to define the output lines.

Create a Volume Maths process

1. In the **Process** tab, click on the Add icon and select **New Process**.
2. Create a **Volume Maths** process and type a **Name** for the process.
3. Click **OK**.



4. In the Details Panel, click the Add icon to add your 2D volume and horizon.
5. Input the following formula:

```
max((twt_ms - WB)/(9000-WB) * 4500 + 1500, 1500)
```

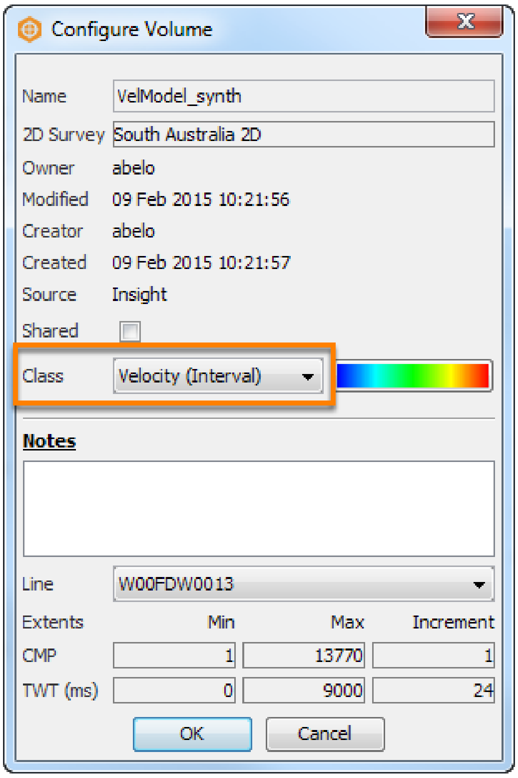
(See below for a brief explanation of the equation.)

6. Set the output extents as follows:

- TWT (ms), 0 - 9000 [24]

7. The process should now yield a resulting volume in the **Volume** tab.

Set the class of the result volume to Velocity (Interval)



1. Right-click on the **Volume Maths** process and select **Open Child Volume**, or find the resulting volume directly from the **Volume** tab (the process volume should be in *Italics*).
2. Double click on the volume, or right click and select **Configure Volume**.
3. In the **Configure Volume** window, set the class of the resulting volume to **Velocity (Interval)**.

A brief explanation of the equation

```
max((twt_ms - WB)/(9000-WB) * 4500 + 1500, 1500)
```

1. max(result, 1500)
   - The minimum velocity is 1500; only output the result if it is larger than 1500.
2. (twt_ms - WB) / (9000-WB)
   - The current time sample (twt_ms), as a fraction (0.0 to 1.0) between the water bottom and 9000ms.
   - To scale between two horizons, add another horizon (symbol 'h2') and replace '9000' with 'h2'.

3. (fraction * 4500) + 1500
   - Calculate a velocity between 1500 and 6000, scaled by the fraction.
   - The fraction is scaled by 4500.
     - 4500 = 6000 - 1500
     - (Scaling) = (the model maximum velocity) - (the model minimum velocity)

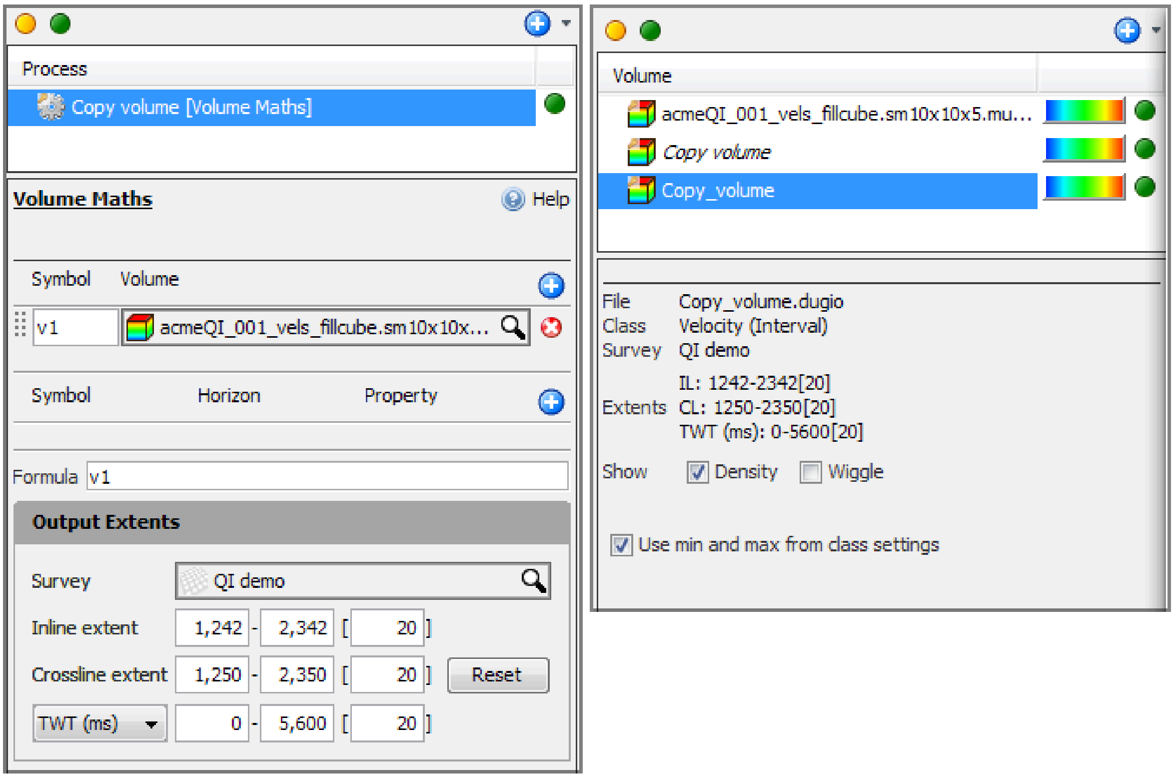
Genuine offer of assistance

If you have any questions regarding the workflow, or if you need a variation of this process for a 3D model, don't hesitate to contact our support team at support@dugeo.com.

How do I create a velocity gradient volume?

Follow the steps below to display the vertical gradient of a velocity volume at any point. Using the velocity gradient, large changes in the velocity can be easily seen.

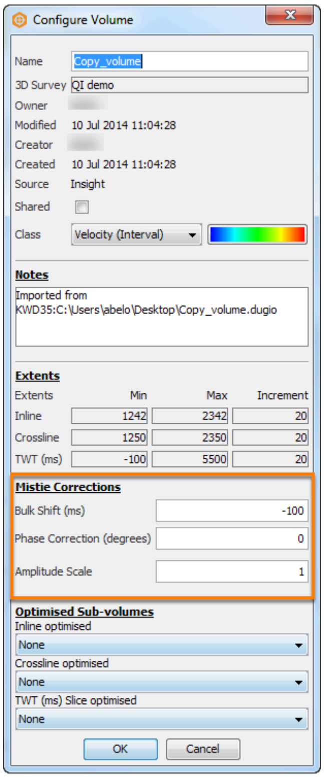
1. Copy volume



To be able to display the velocity gradient, a duplicate of the velocity volume is required.

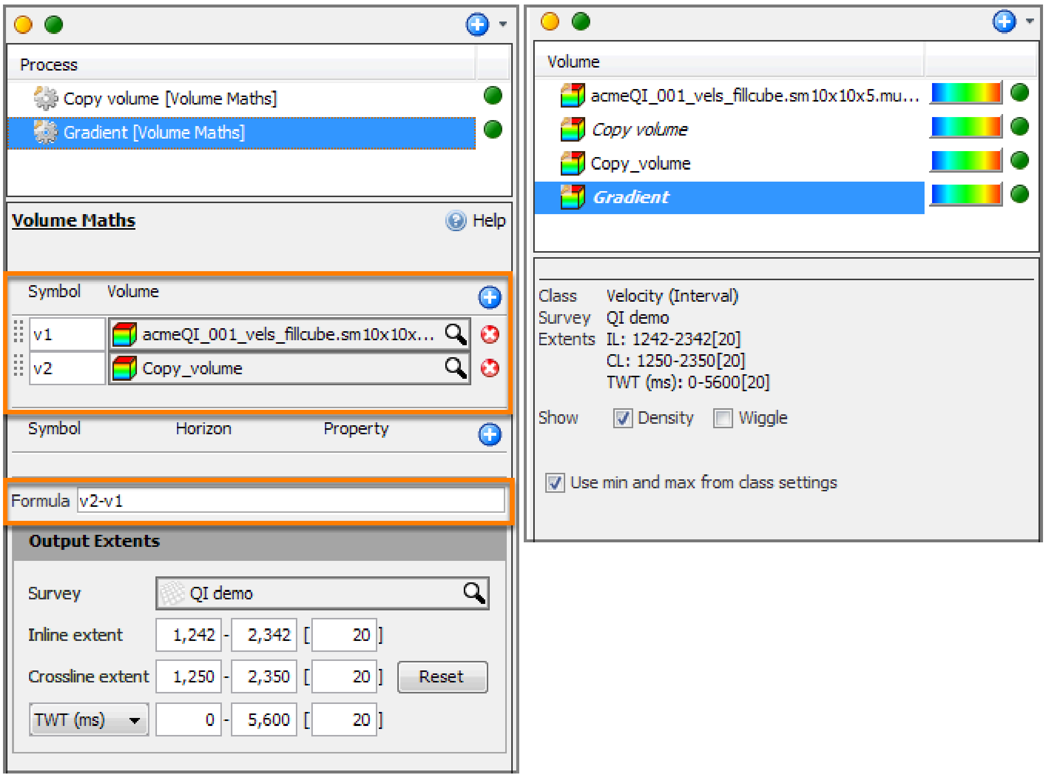
Follow the steps in *Creating a Copy of a Volume* to create a copy, export and configure the velocity volume.

2. Set bulk shift



After creating a copy of the velocity volume, shift the duplicate volume by the desired value, e.g. -100 ms (*see* Manual Mistie Corrections).

3. Volume math



Create another Volume Maths process, and select the original and the duplicate volume. At **Formula**, type

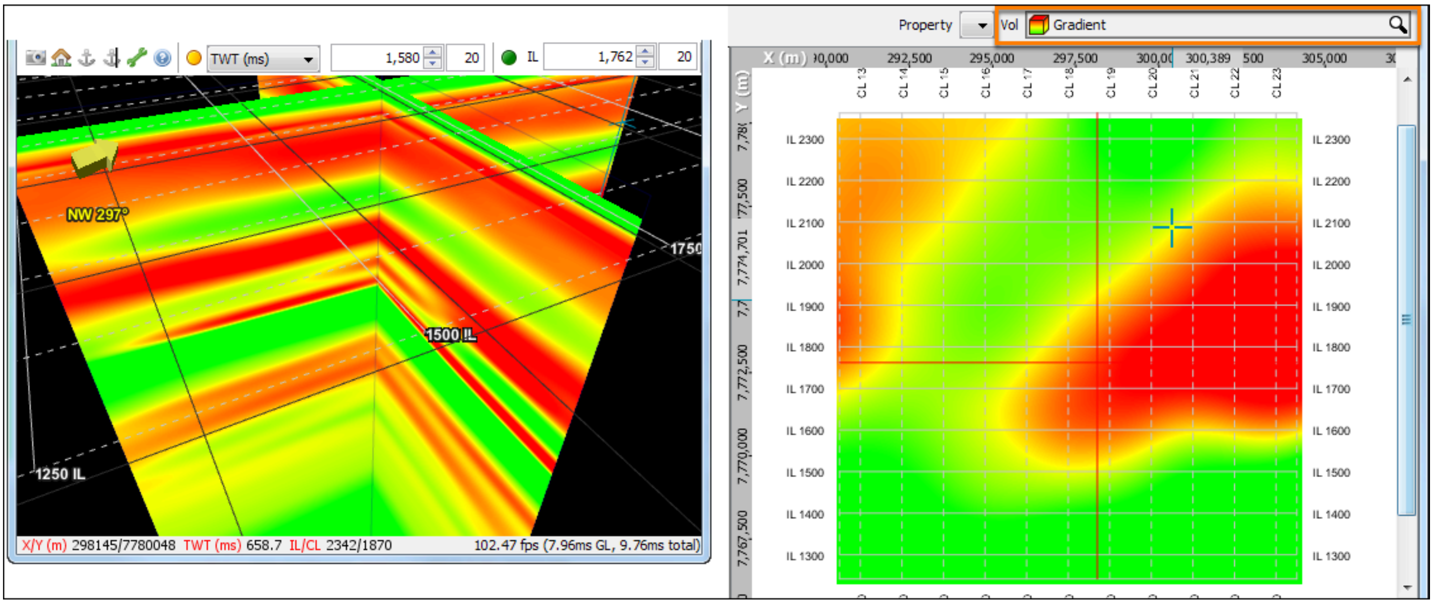
```
v2-v1
```

where:

- v2 = copied and shifted velocity
- v1 = original velocity volume.

As a result of this process, a new volume is available in the **Volume** tab based on the Formula.

4. Viewing the gradient



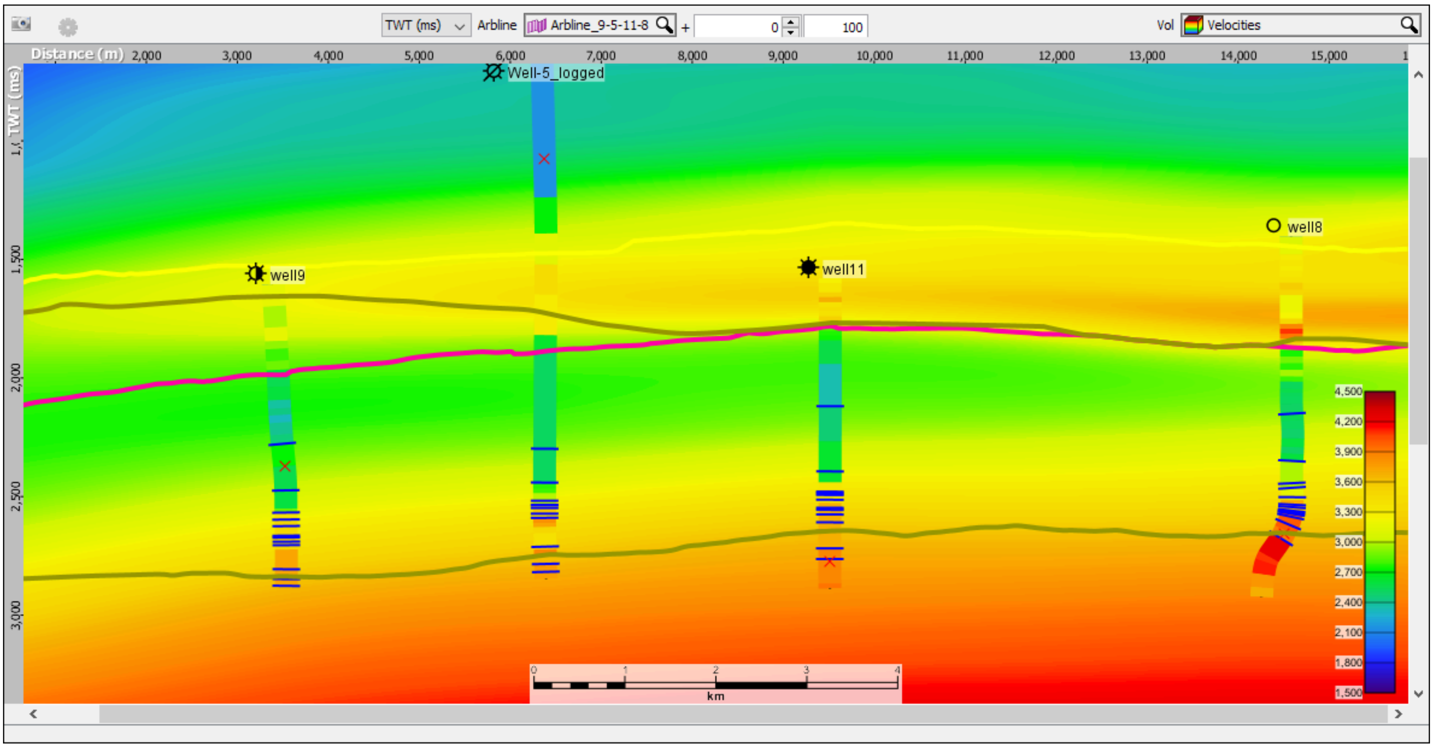
At the navigation bar, select the **Gradient** volume.

The gradient can be displayed in 3D View, Map View, and any section views. Press **F5** or **F6** to make the colourbar hotter or colder respectively.

Simple velocity tie (to wells)

Creating velocity volumes is easy in Insight - just take a look at some of our *suggested workflows*.

Matching a velocity to tie the well location can be more troublesome. A simple velocity tie can be achieved by calculating the vertical interval velocity in time (V0) from any "isotropic interval velocity model" in time, possibly from depth migration or velocity picking, and an interval delta model in time. Insight packages this workflow up into easy-to-use processes available in the *Explorationist* and *Image Gather Processing* modules.

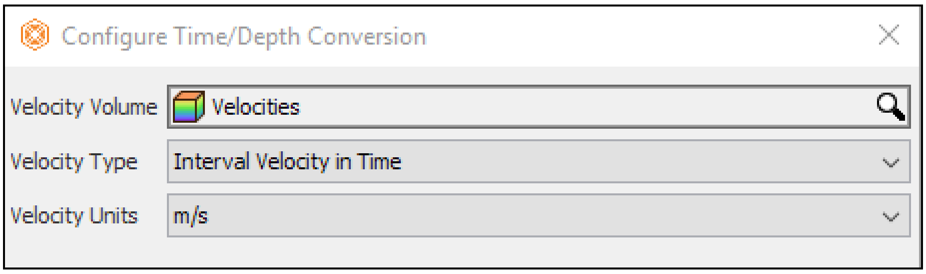


The workflow can be split into 3 stages:

1. Use the *Synthetics* tool to create delta curves for each well that you want to influence the model.
2. Create an interval delta model in time.
3. Calculate V0 from delta and isotropic depth mig velocities.

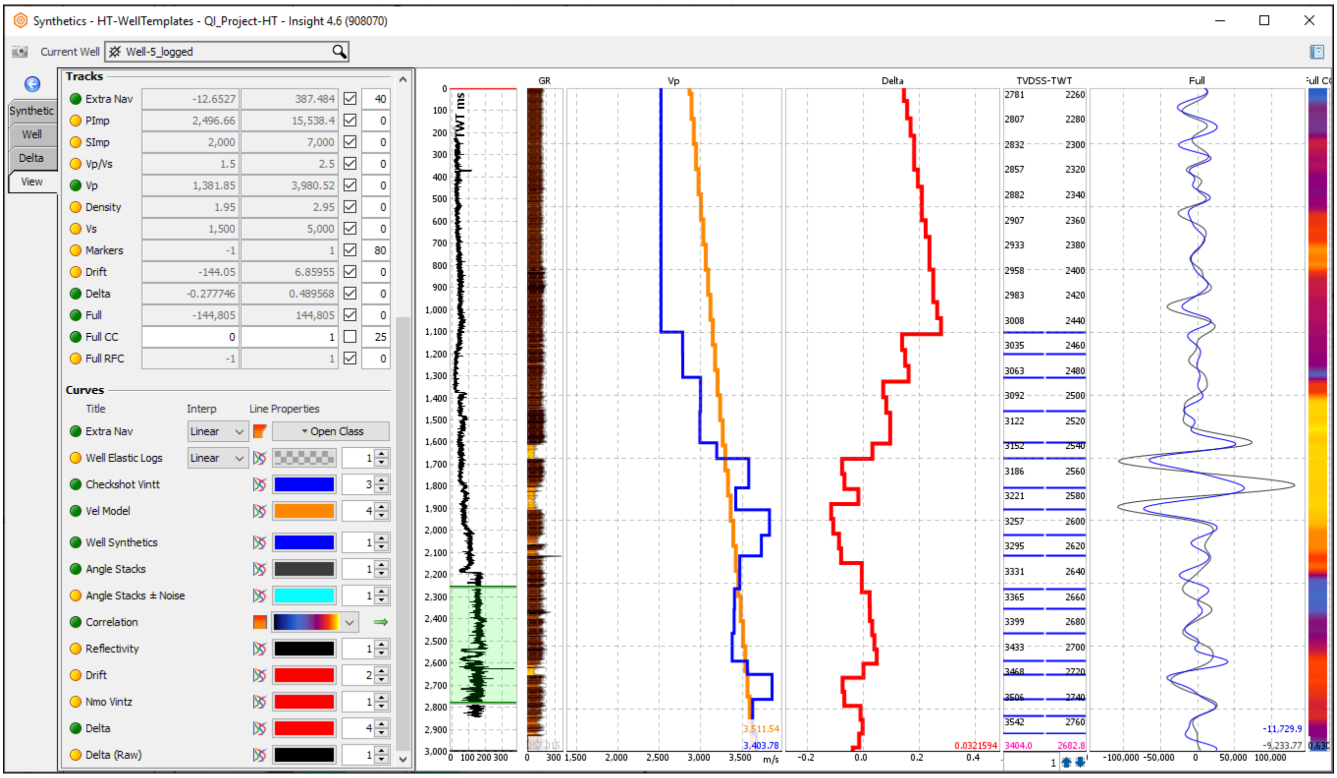
1. Synthetics tool delta curves

Generate the delta curve for the wells in DUG-Insight’s Synthetics tool. The delta curves will then be used to build 3D delta models (see *Delta Model*).

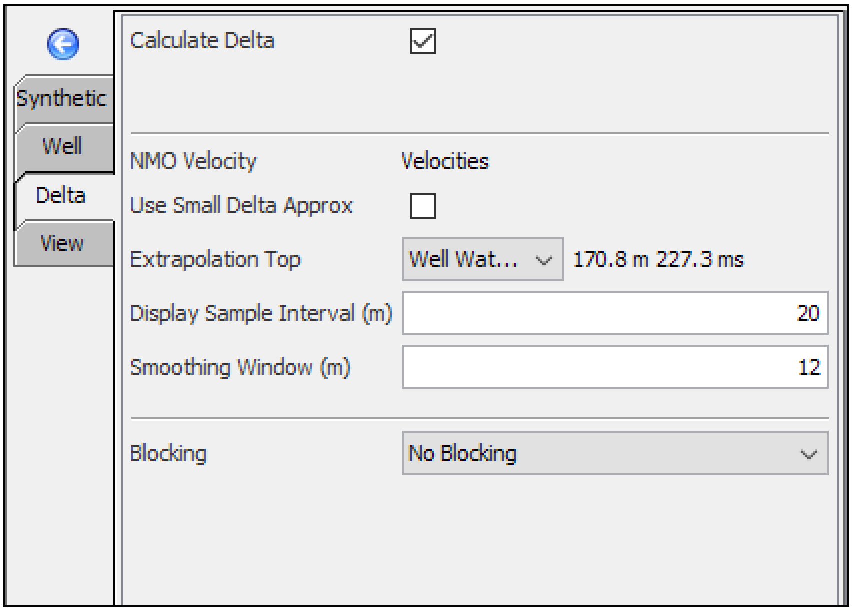


1. As a prerequisite, you need to load the (NMO) interval velocity volume and select it for time/ depth conversion as described in *Converting Time-Depth*.

Note: The delta curve is calculated at the wells using the well's checkshot Vint and an initial (NMO) isotropic Vint field selected for Time-Depth Conversion (See *Generate Delta Curve*).



1. In the **Synthetics** window, open the **View** tab.
2. Under the **Tracks** section, click the amber circle at **Delta** to display the Delta track in Synthetics.



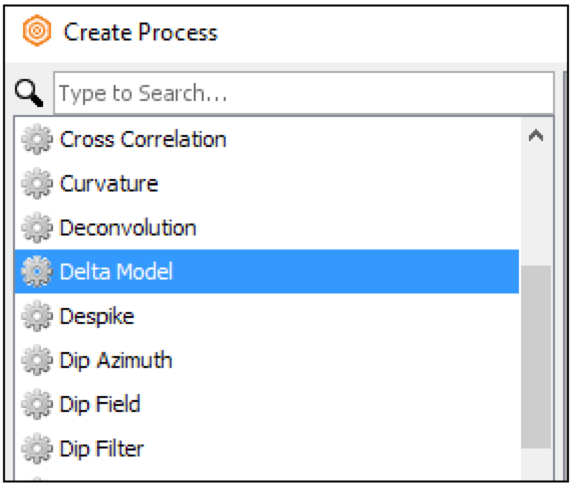
3. Open the **Delta** tab in the **Synthetics** window.
4. Select the Calculate Delta check box. This is to create the delta curve.
   - **Note:** Delta values are required to be in the range of -0.25 to 0.35.
5. At **NMO Velocity**, the velocity volume selected for time/depth conversion is shown.
6. At **Extrapolation Top**, select **Well Water Bottom** or **Horizon** (and select the water bottom horizon). This parameter defines the depth above which the delta value will be extrapolated.
7. Type the display sample interval for the delta curve at **Display Sample Interval (m)**. By default, this is set to "**20**".
8. At Smoothing Window (m), type the smoothing value for the delta curve. By default, this is set to "**12**".
9. Select the blocking option to be used for the delta curve after smoothing at **Blocking**.
   - **No Blocking:** Display the smoothed delta based on the display sample interval.
   - **Regular Blocking:** Use the regular blocking option and type the block size in meters for the delta curve (this is equivalent to making a coarse sample interval).
   - **Horizon Blocking:** Use the horizon blocking option and select the horizon(s) used in the blocking. To add the horizons, click the Add Row icon.
10. QC the delta curve generated. Valid delta curve results must be in the range from **-0.25** to **0.35**.

2. Create an interval delta model in time.

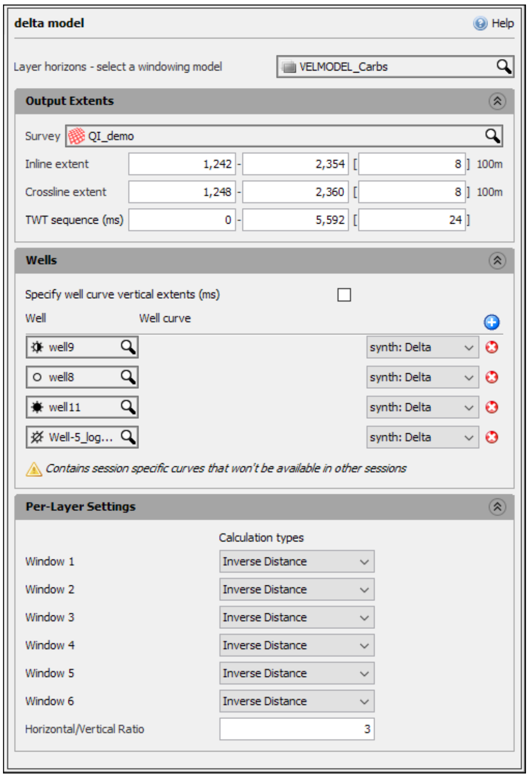
The Delta Model process builds a delta model from well curves. It requires a windowing model (see *Creating a Windowing Model*) and at least one well with a delta curve.

Window model layers are automatically divided by ghost horizons using the conformability specified in the windowing model. Delta values are selected for the model at the intersection between each well and the ghost layer. These values are combined according to the specified per-layer settings.

Note: Create the Window model in Time as Delta model process creates an interval delta model in time.

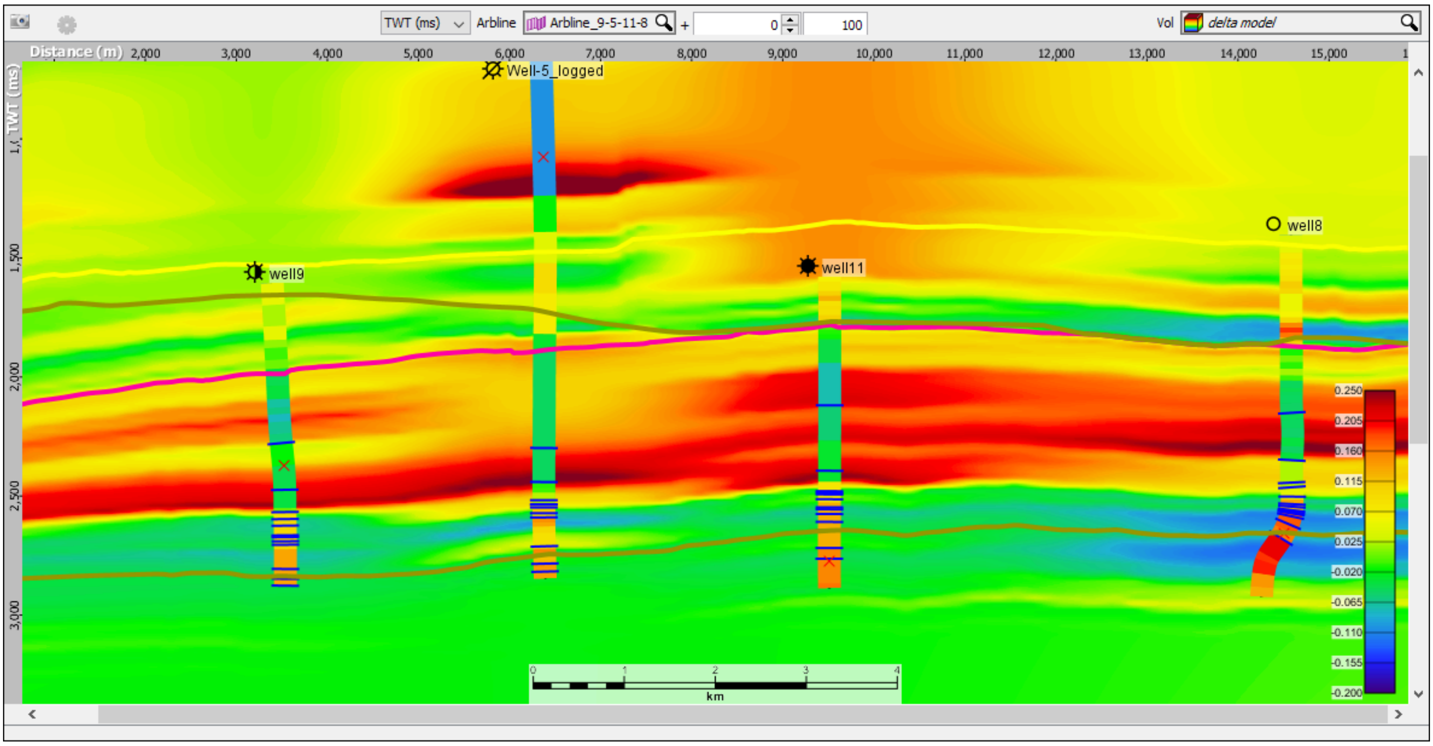


1. In the **Control Panel**, open the **Process** tab.
2. At the tab header, click the **blue "+" icon** and select **New Process**.
3. Double-click **Delta Model**.
4. Type a name for the process and click **OK.**



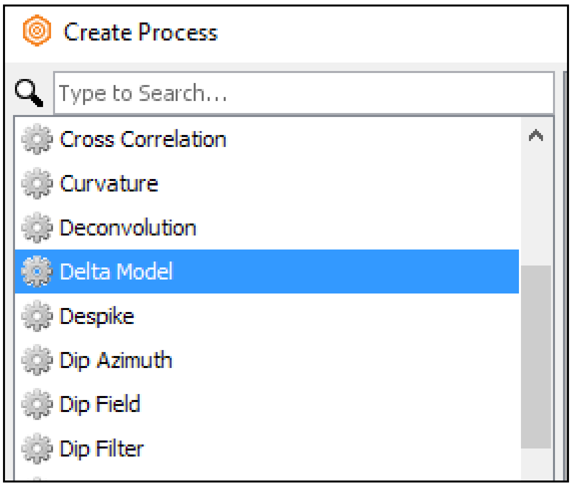
5. Select the windowing model to be used in the process at Layer horizons - select a windowing model (see *Creating a Windowing Model*).

6. In the **Output Extents** section, select a survey to be used in the delta modeling process at **Survey**.
7. Type the range of Inline extent, Crossline extent and time (TWT) extent for the delta model and the respective sample intervals.
8. In the **Wells** section, click the **Add Row (blue "+") icon** to add the wells used in the delta modeling.
   Note: Specific vertical extents can be specified at each well by selecting Specify well curve vertical extents (ms) option.
9. Select the delta curve for each well to be used in the delta modeling.
10. In the Per-Layer Settings section, select the calculation type used to distribute delta in each layer in the windowing model. Available options include:
    - **Inverse Distance**
    - **Average of Well Values**
    - **User-specified constant**
11. As a result of this process, a new delta volume is added in the Volume tab. You can further smooth the delta model using the *Volume Smoothing* process.

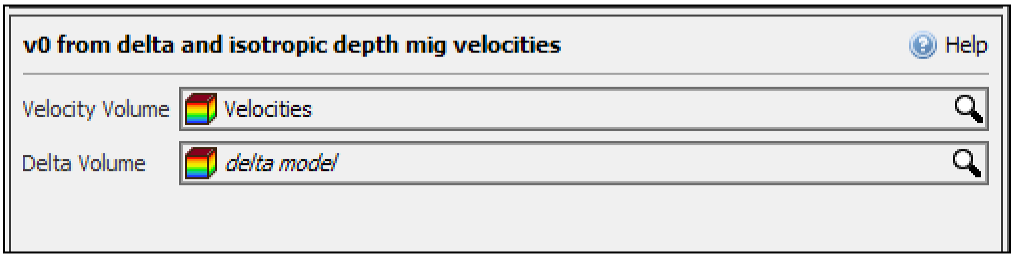


3. Calculate V0 from delta and isotropic depth mig velocities.

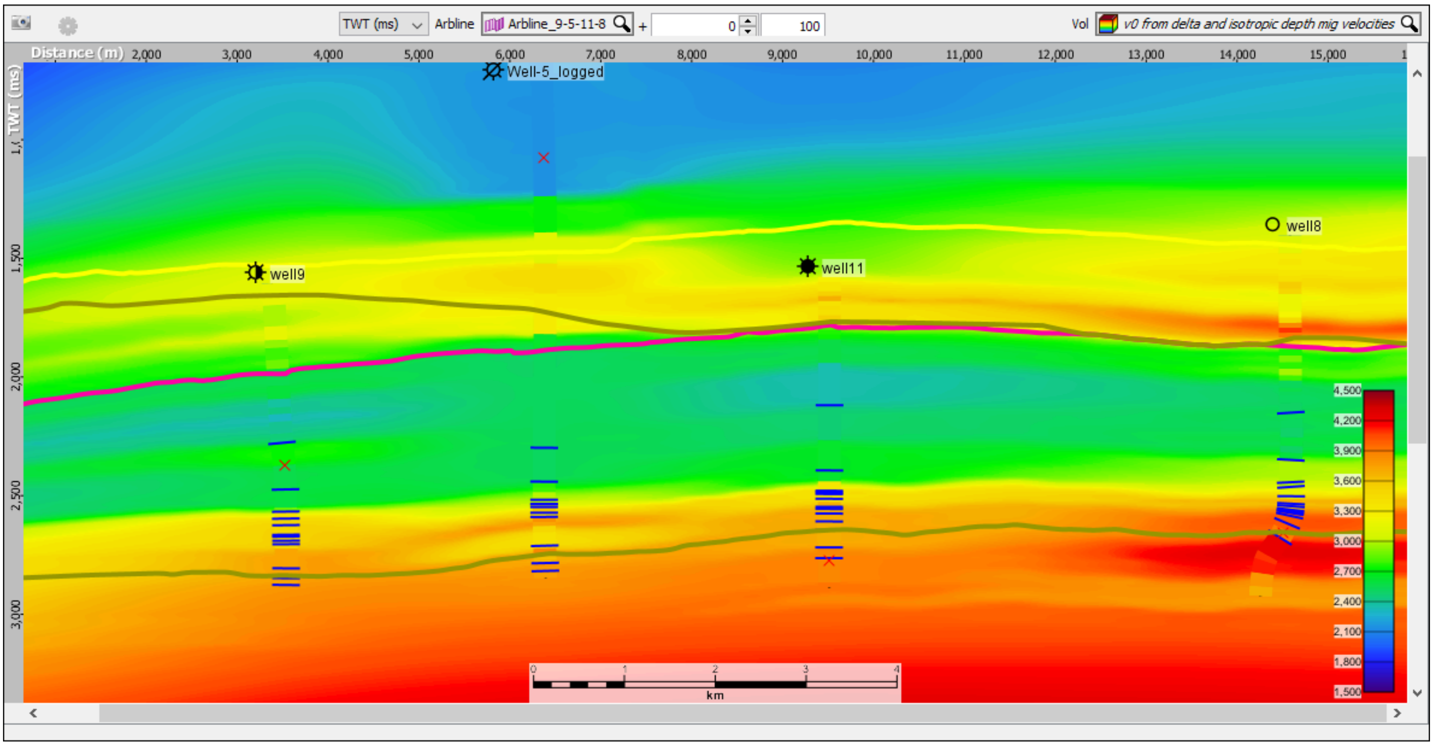
Calculates the vertical interval velocity in time (V0) from an isotropic interval velocity model in time, and an interval delta model in time.



1. In the **Control Panel**, open the **Process** tab.
2. At the tab header, click the blue "+" icon and **select New Process**.
3. Double-click **V0 from delta and isotropic depth mig velocities**.
4. Type a name for the process and click **OK**.



5. Select the **nmo interval velocity volume (in time)** used in delta calculation.
6. Select the **delta model (time)**.
   - Volume is calculated using formula **Vo/Vnmo = 1/sqrt(1+2*delta)**
   - See paper Tsvankin, I., 2012, Seismic signatures and analysis of reflection data in anisotropic media: Society of Exploration Geophysicists, third edition.
7. As a result of this process, a new volume that ties the well locations is added in the **Volume** tab. You can further smooth the delta model using the *Volume Smoothing* process.

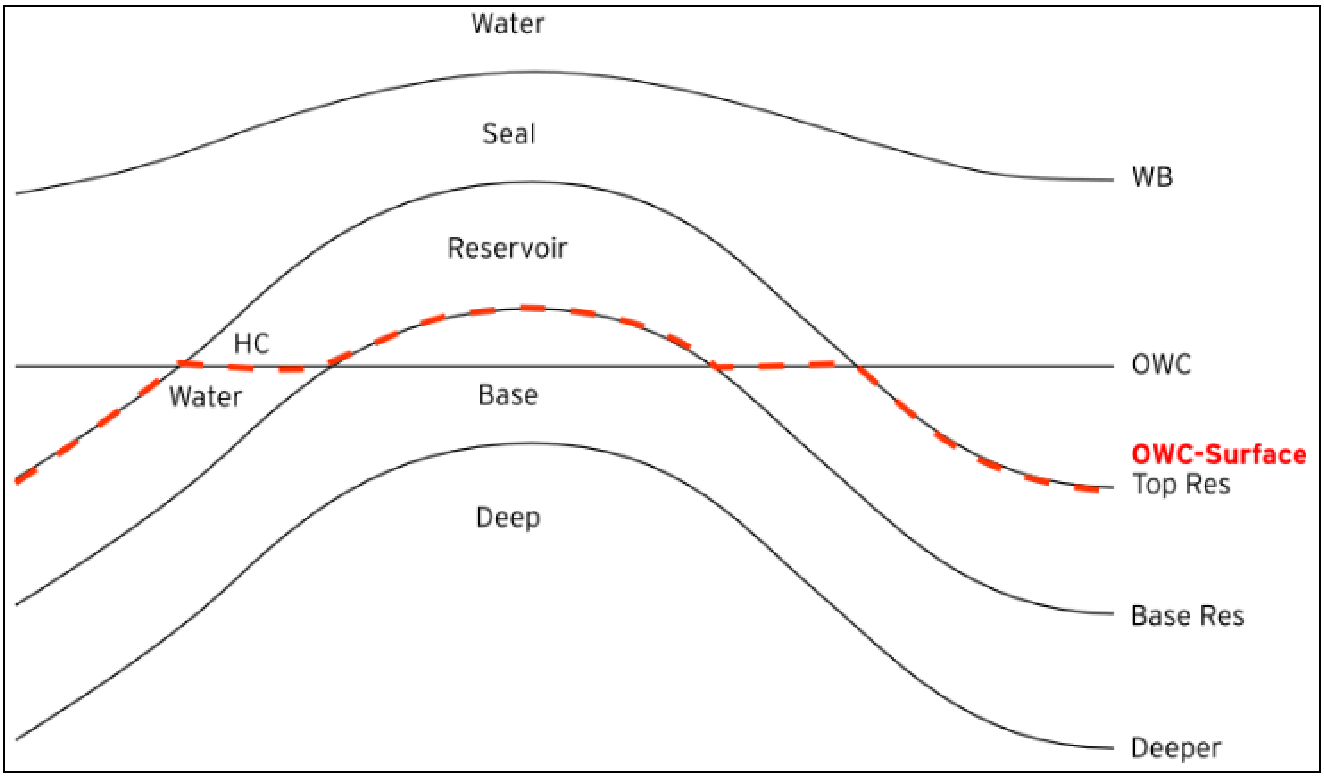

TWT (ms)
Arbline
Arbline_9-5-11-8
100
Vol
v0 from delta and isotropic depth mig velocities
Distance (m)
TWT (ms)
Well-5_logged
well9
well11
well8
km

Horizons

How do I create a contact horizon for use in layered models?

To construct the model including a contact, you will need to construct a version of the contact horizon that is bound by the neighbouring horizons.

Refer to this diagram:



1. In Map View, open the **Operations** tab and select **Horizon Math**.
2. Add the two bounding horizons ("top" - Top Res, "base" - Base Res).
3. Set the property of each to **TVD** (assumes you have a velocity model configured for Time-Depth conversion).
4. Use this formula to create a constrained constant surface (e.g. at 2500):
   if (top > 2500, top, if (base < 2500, base, 2500))
5. Set the output Z dimension to **TVD**.
6. Click **Run**.

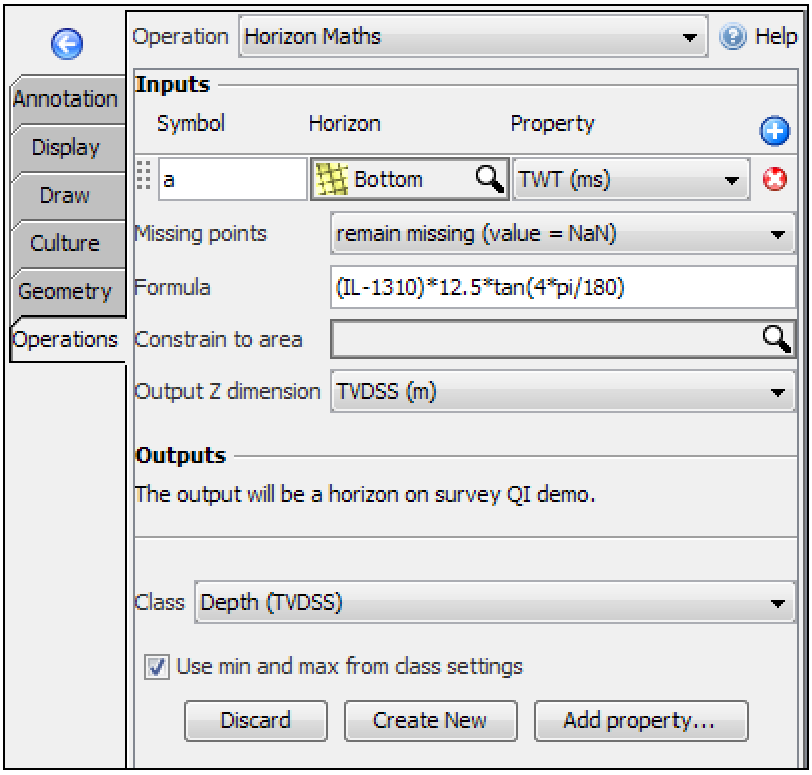
Rename the result to something useful, e.g. constrainedOWC.

Replace the constant value in your model with this constrained horizon.

How do I perform an amplitude extraction at different angles through the volumes?

This can be done by performing amplitude extractions on horizons that dip at specific angles.

Creating the initial angle horizon

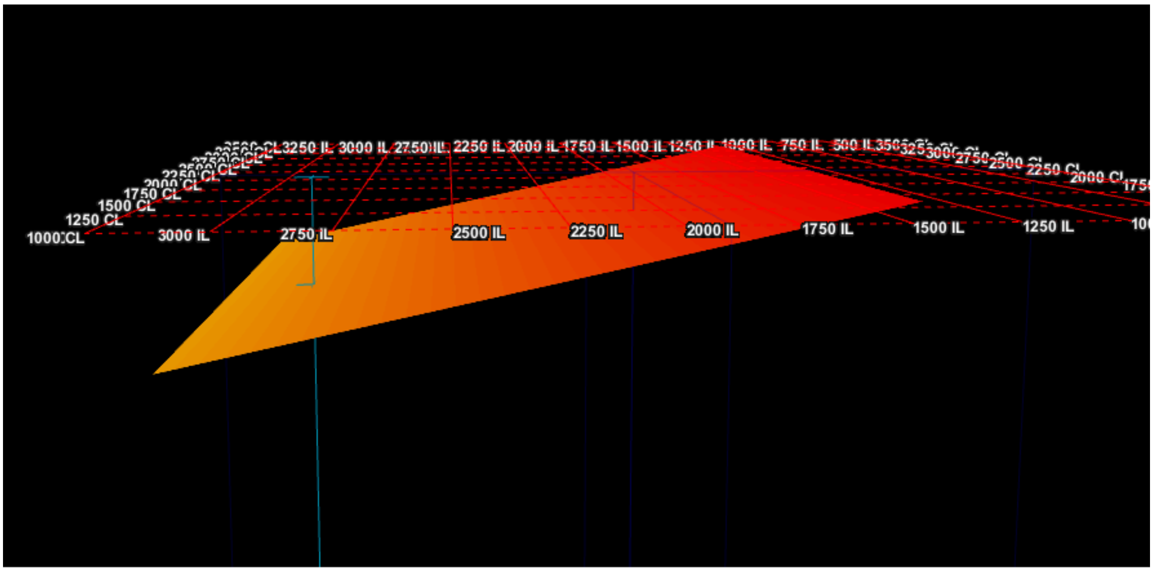


1. From the View menu in the Control Panel, open the Map View.
2. Select an existing horizon that covers the area (the waterbottom might be a good choice).
3. Expand the Operations tab and select **Horizon Maths**.
4. Use the base formula:

```
IL - <minimum inline>*<inline spacing>*tan(<oblique angle>*pi/180)
```

- This formula calculates a new horizon with depth values at a specific angle dipping in the crossline direction.
- You can replace the IL with CL to set the dip direction to the inline direction.

5. Define the output as **TVDSS (m)** since the survey spacing is measured in meters.
6. Click **Run** to begin the operation.

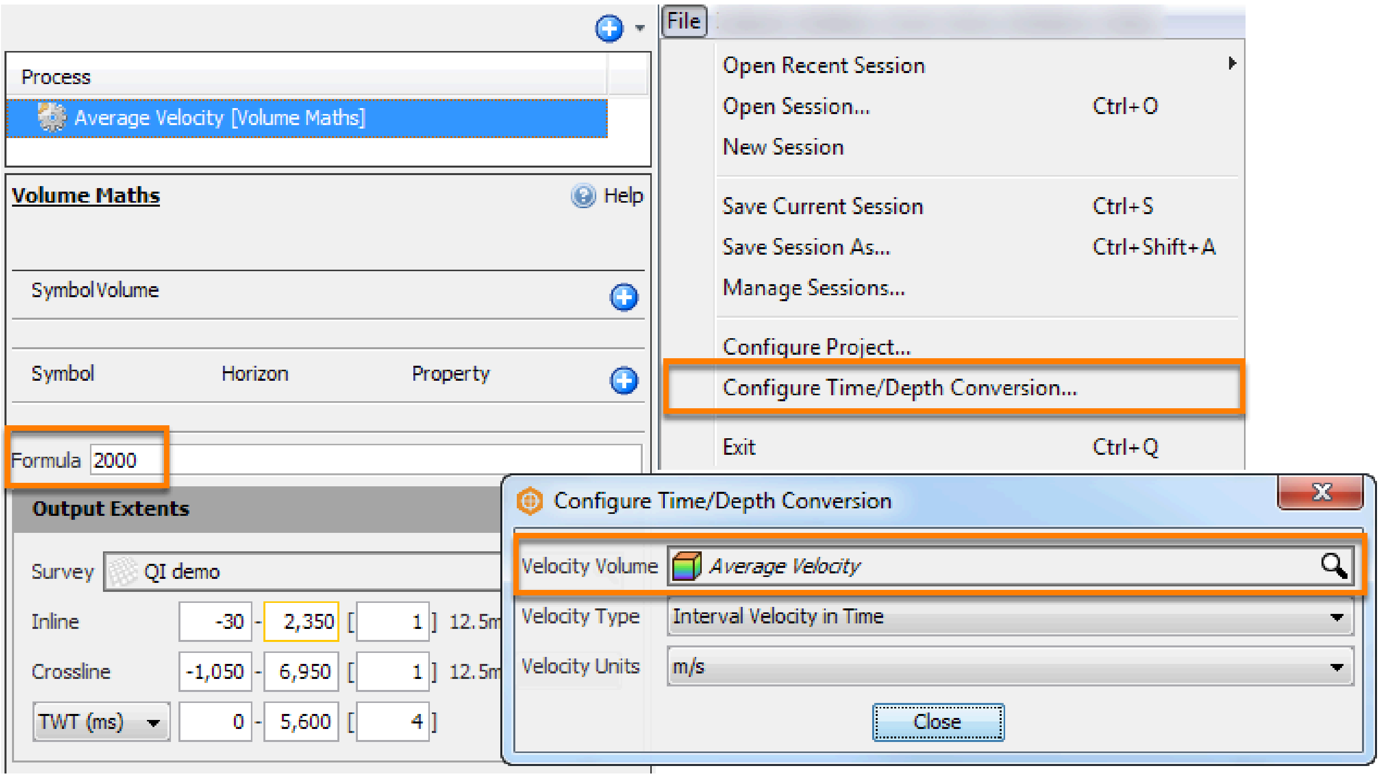


Example horizon created with the following parameters:
Angle = 4 degrees dipping in the crossline direction
Inline spacing = 12.5 meters
Minimum inline = 1310

Formula = (IL-1310)*12.5*tan(4*pi/180)

Once you have this base angle horizon, you can use the Ghost operation in Map View to move the horizon up or down by a constant amount (see *Ghost*). This will allow you to cut the volumes at different depths/times.

Converting time-depth



If you need to convert the horizon to time so you can extract amplitudes from time volumes, a constant velocity volume that will allow a 1:1 conversion between the TWT and TVDSS for the horizon can easily be created.

1. From the Control Panel, open the **Process** tab.
2. Click the Add icon and select **New Process**.
3. Select **Volume Maths** and give the process an appropriate name.
4. Set the **Formula** to a constant 2000. This will create a new volume under the Volume tab.
5. Change the class of the volume to **Average Velocity**.
6. From the **File** menu in the Control Panel, select **Configure Time/Depth Conversion**.
7. Choose the new volume as the **Velocity Volume**.

Now you have dipping horizons on which you can view amplitudes/values from your volumes. Remember you can just change the horizon property to Amplitude in the Map View to do the on-the-fly amplitude extraction.

Why is the amplitude value as shown by the cursor different than the horizon extracted value?

The amplitude value displayed by the location of the cursor in section views, and the amplitude value extracted by horizon amplitude extraction, are fundamentally different.

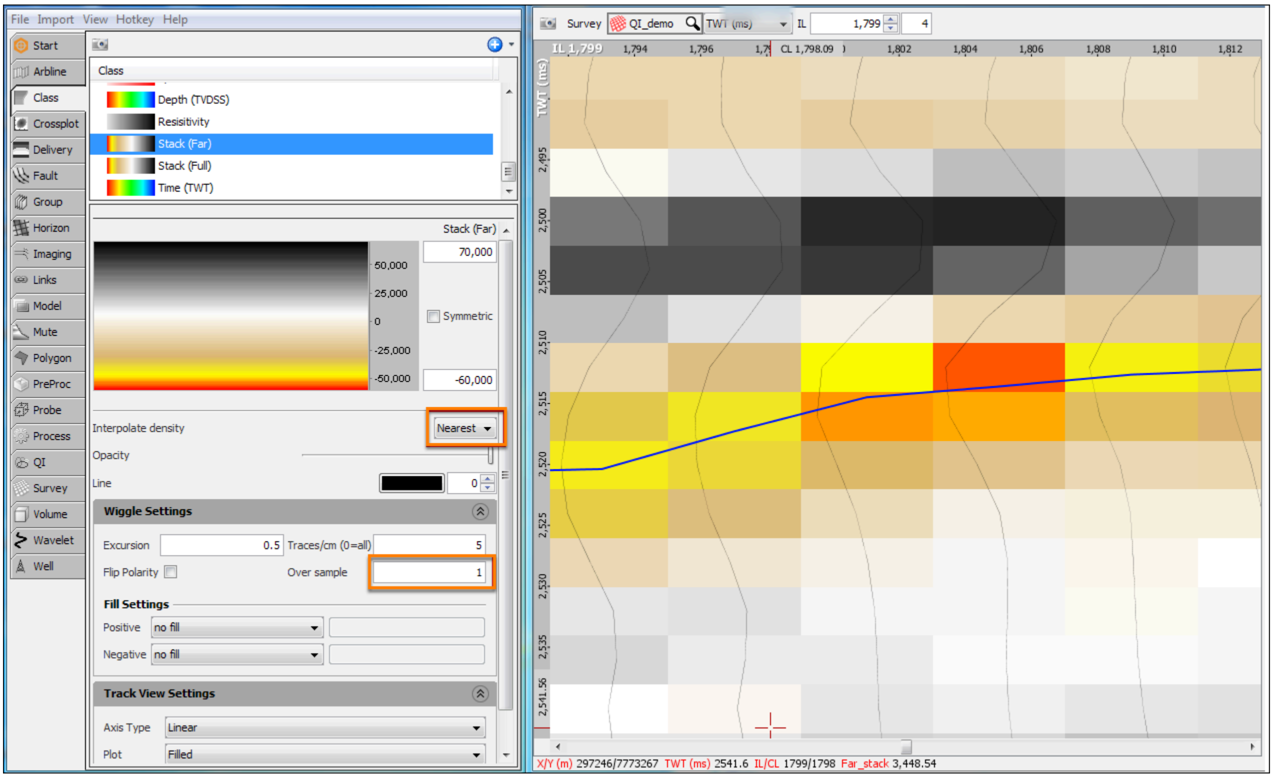
- The cursor value displays the amplitude based on the nearest true sample value.
- The horizon amplitude extraction extracts the amplitude to the nearest 1000nth of a milli-second, based on the interpolation function.

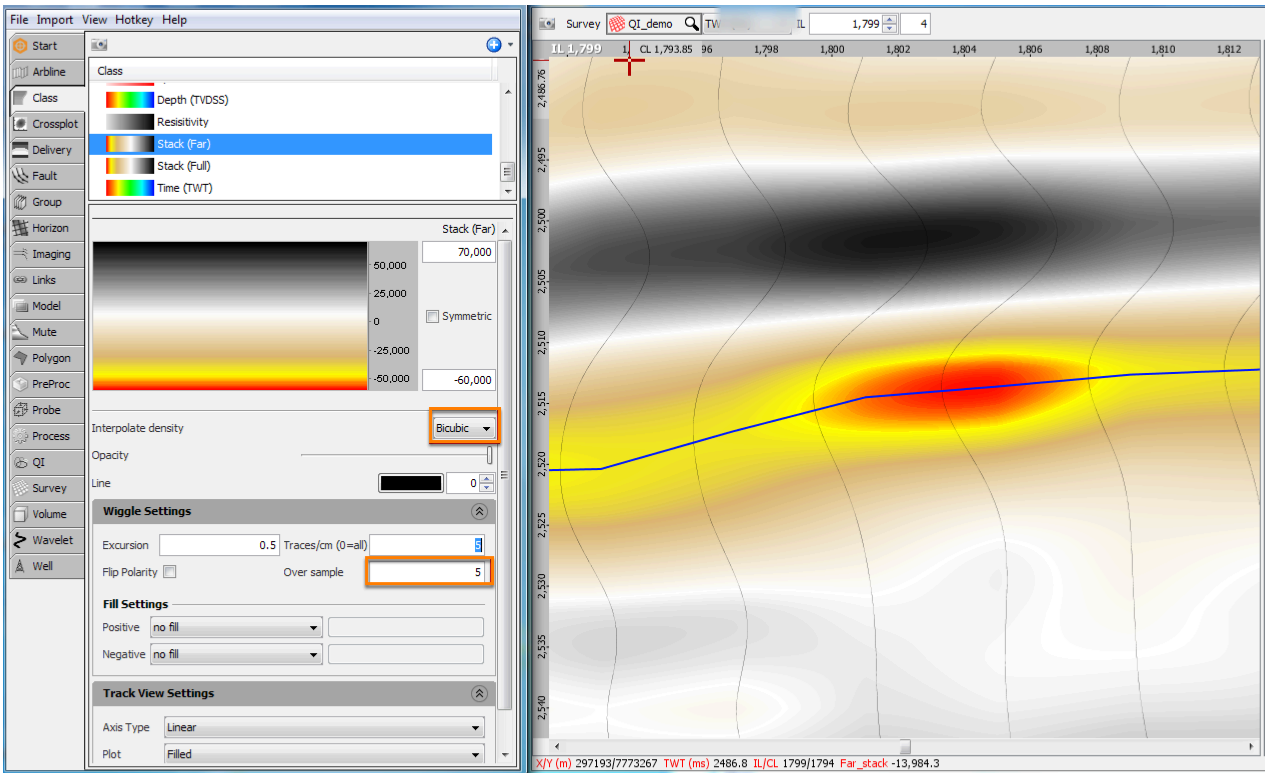
True amplitude sample value vs Interpolated function value

The difference between the true amplitude sample value and interpolated function value can be demonstrated by switching the display interpolation method in **Class** settings. The three following options are available:

- **Nearest** — Displays true amplitude sample values only. Quickest to render but provides the lowest quality of result.
- **Bilinear** — A simple linear interpolation to render the colour information and is slightly slower than the nearest neighbour method.
- **Bicubic** — Uses bicubic splines to interpolate your data. This provides the best output at the expense of performance.

The following images demonstrate the effect of switching between **Nearest** and **Bicubic** interpolation methods. The images also include the wiggle traces, with no oversample (in Nearest), and oversampled (in Bicubic) for clearer understanding. **Over sample** controls the smoothness of the wiggles. When set to **1**, you will essentially see the raw trace. When set to **5**, you will see something that resembles interpolation function.





In both the images, a horizon is displayed, which was picked with **Snap** enabled. The snap function snaps the horizon to the highest amplitude value based on the interpolation function. The horizon extraction extracts the same value based on the interpolation function. Hence, the extracted value will almost certainly always be different to the nearest true sample value.

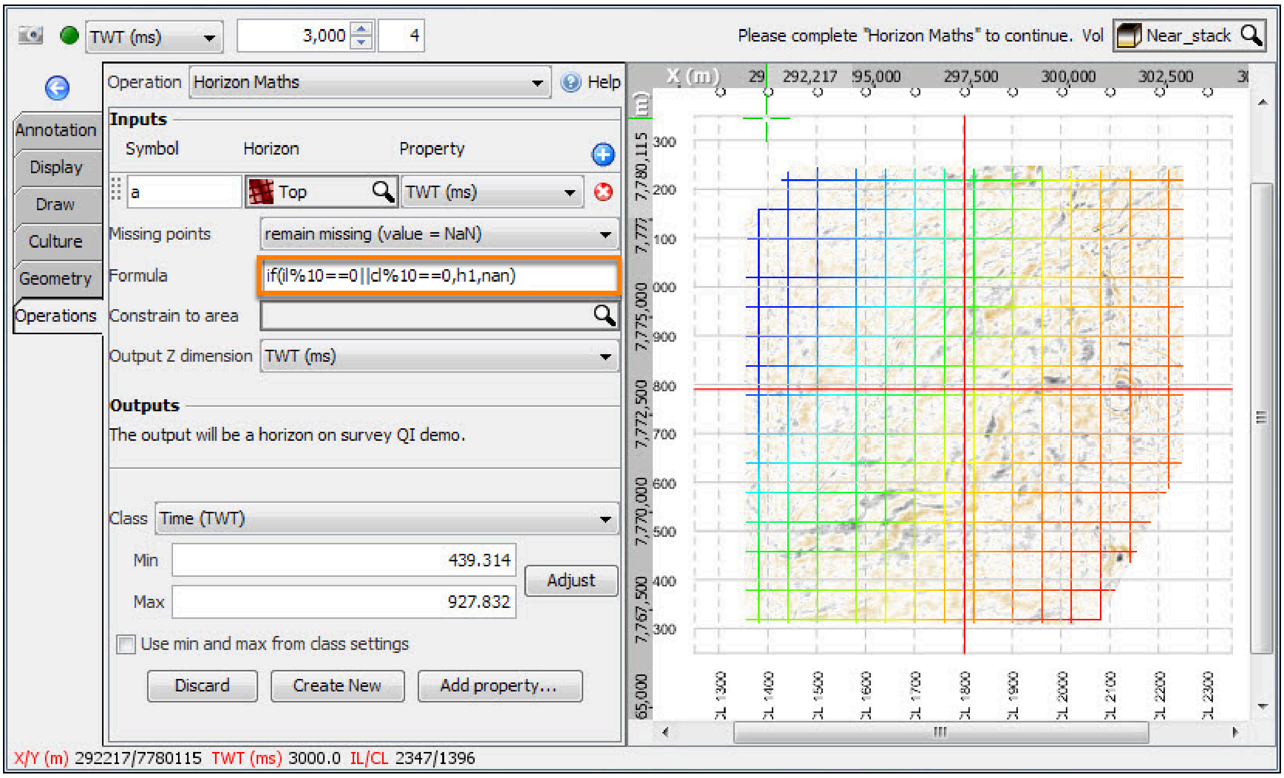
It is important to note however that, even with the cursor value based on the interpolation function, unless you can place the mouse cursor on the EXACT same time as the horizon, you will still NEVER see the exact same value.

How do I decimate an interpolated surface back to its original picked lines?

If you picked your original surface on a regular grid, you can extract the original picks as a lattice horizon using *Horizon Maths*.

Workflow

1. In the Map View, select **Horizon Maths** and input your IL/CL horizon. Call it 'h1'.
2. Define the formula: ***if(il % 10 == 0 || cl % 10 == 0, h1, nan)***. This formula translates as: Keep inlines and crosslines that, when divided by ten, have a remainder of zero.



Note: The modulus operation (%) used in this formula is a fast way to find the remainder after dividing by some value. In cases where the horizon locations are not evenly divisible by the lattice increment (i.e. there is some offset), no picks will be returned.

For example:

- If inline values are on the sequence (105, 115, 125...), (il % 10) will never be zero.
- If crossline values fall on odd values (101, 103, 105...), (cl % 2) will never be zero.
- In these cases, add or subtract a constant so the result is evenly divisible by your lattice increment.

Also:

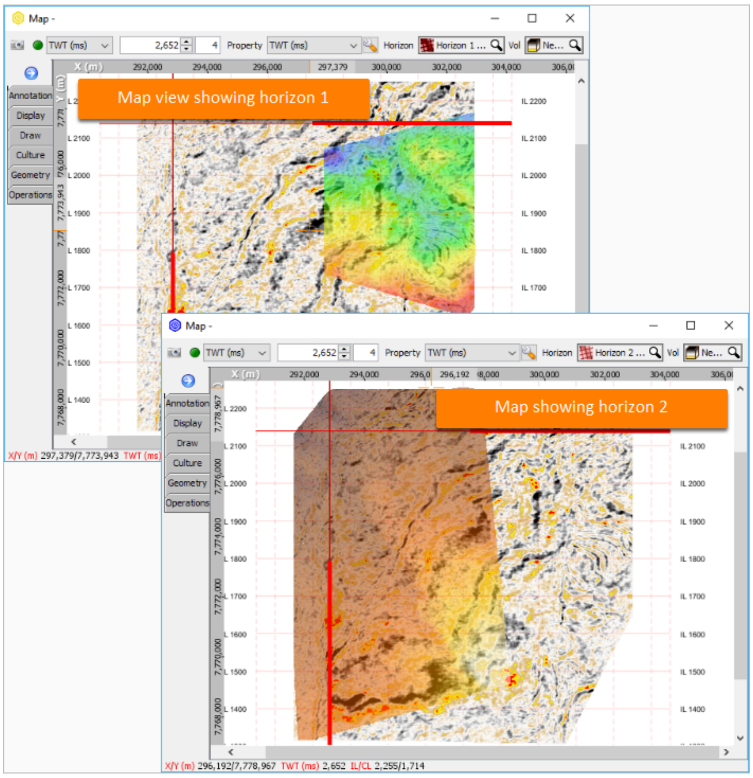
- If inline values are on the sequence (105, 115, 125...), ((il + 5) % 10) will work.
- If crossline values fall on odd values (101, 103, 105...), ((cl - 1) % 2) will work.

How to display data from two or more horizons simultaneously

To view multiple horizons, or multiple attributes of one horizon, there are a few options, depending on what you would like to achieve:

- *Use multiple map view*
- *Merge using horizon maths*
- *Using 3D view*

Use multiple map view

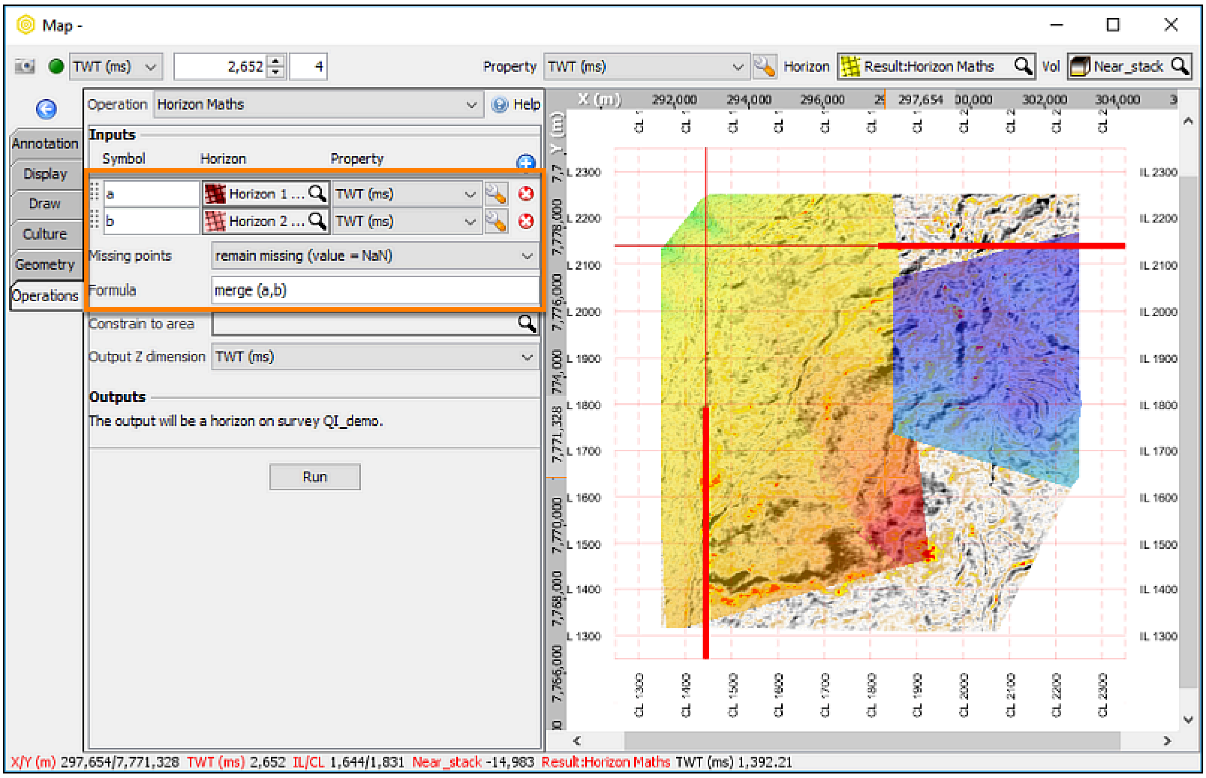


Open a second Map view to display each horizon (see *Volumes and horizons in Map View*).

There are two approaches:

- From the **Control Panel**, **View** menu, choose **New Map View**, OR
- Right-click on an existing map view and choose *"Duplicate this window"*

Merge using horizon maths

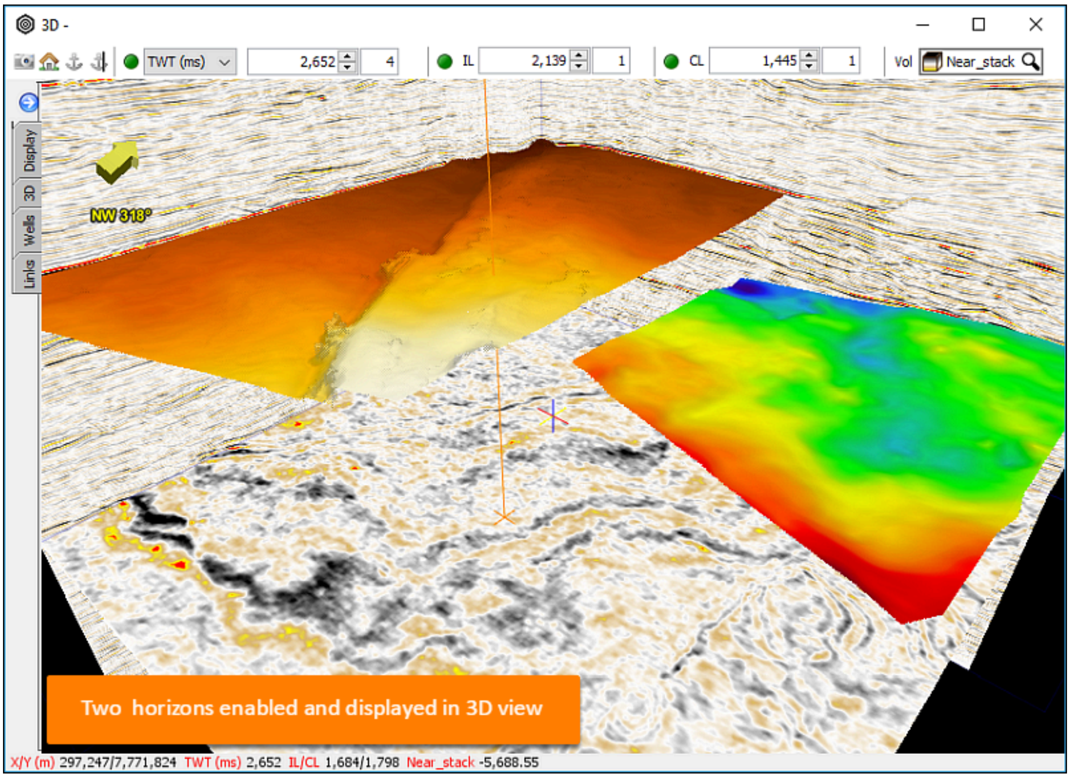


To combine two (or more) pieces of a horizon to a single surface:

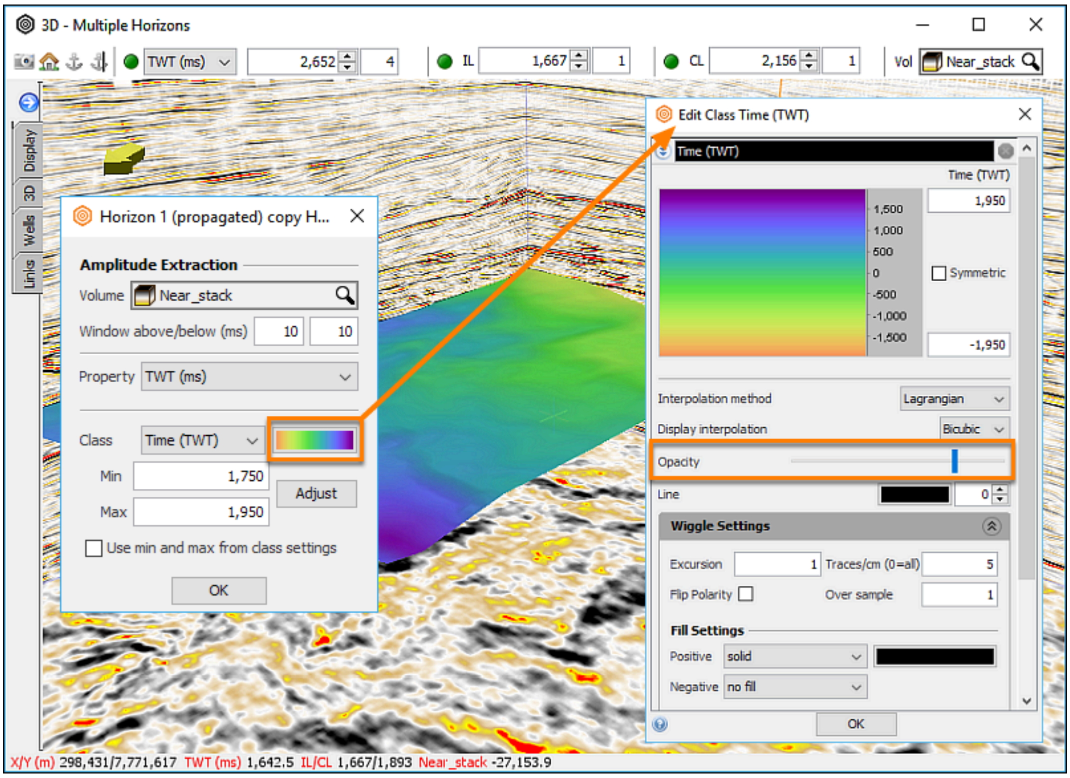
- Use **Operations > Horizon Maths**
- Add each of the horizons to merge
- Use the **Formula**: merge (a, b)

For more on horizon maths see *Common horizon tasks in horizon maths*.

Using 3D view



- Open the 3D view
- In the **Control Panel > Horizon** tab, toggle the display of each horizon using the '*traffic light*' icon
- Enable the option in the **Control Panel** to display the horizon as a 3D surface, not the intersection (see *Horizon Overview*).



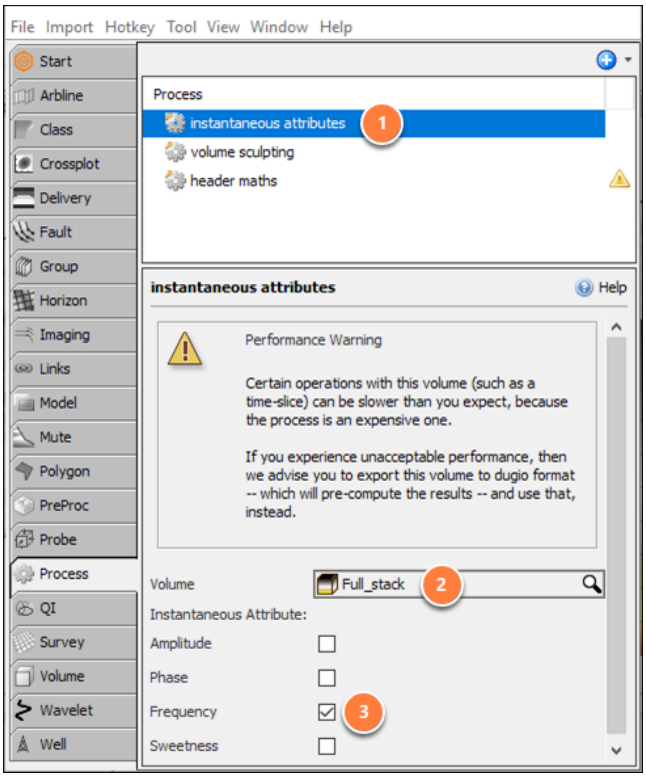
In the **3D view**, different attributes can be displayed for horizon. To display multiple attributes for the same horizon, create a duplicate and display a different attribute for each.

1. Duplicate your horizon (see *Copying a Horizon*).
2. In the **Control Panel**, display the original and hide the copy.
3. Right-click on the original horizon in the **3D view** and choose "*Settings for Horizon*".
    - Select the property to display.
    - Click the class and set the opacity.
4. In the **Control Panel**, hide the original and show the copy.
5. Right-click on the duplicate horizon in the **3D view** and choose "*Settings for Horizon*".
    - Select the property to display.
6. In the **Control Panel**, show the original horizon again.
7. Both horizon attributes are now displayed together.

For more on viewing horizon attributes in **3D view**, see *Horizon options in 3D view*.

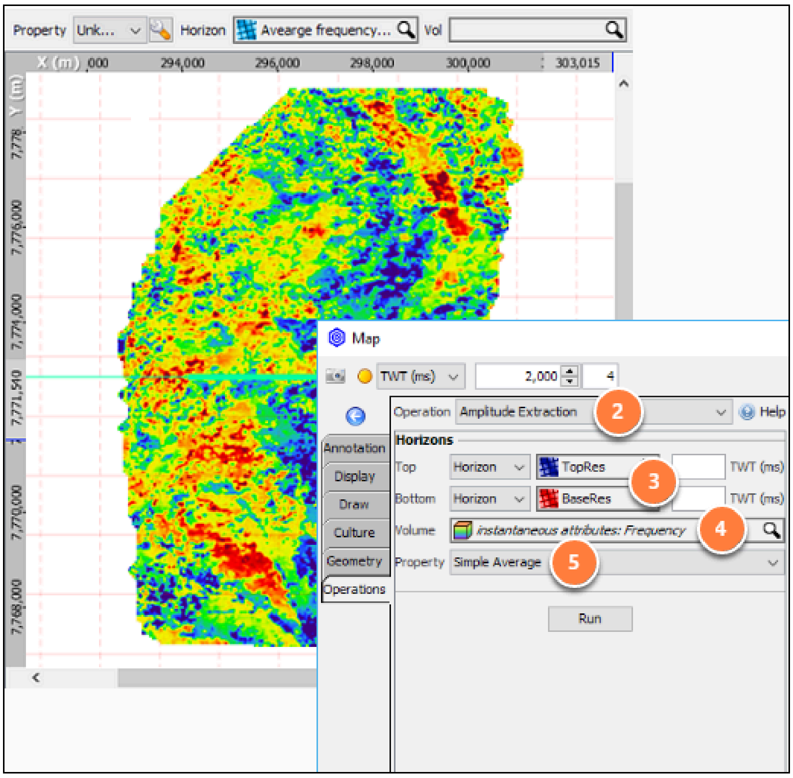
How do I extract the number of events between 2 time horizons?

Use this workflow to calculate a rough estimate for event counts, based on average frequency and time thickness of the target zone. This example calculates the seismic event counts from TopRes to BaseRes horizons for a 3D seismic volume in time.



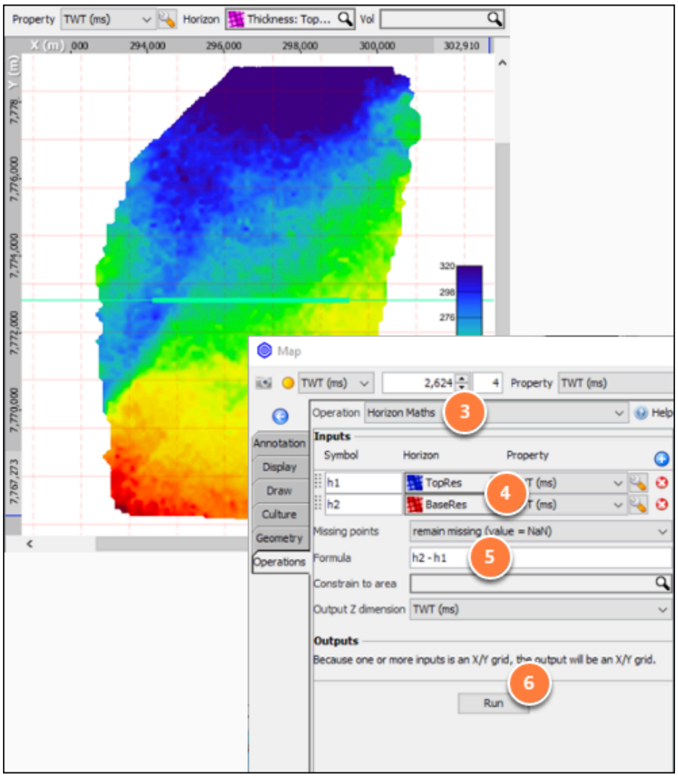
Step1: Create an instantaneous frequency for stack:

1. Go to **Control panel > Process** and add an **Instantaneous Attributes** process
2. Select the volume.
3. Select the **Frequency** checkbox.



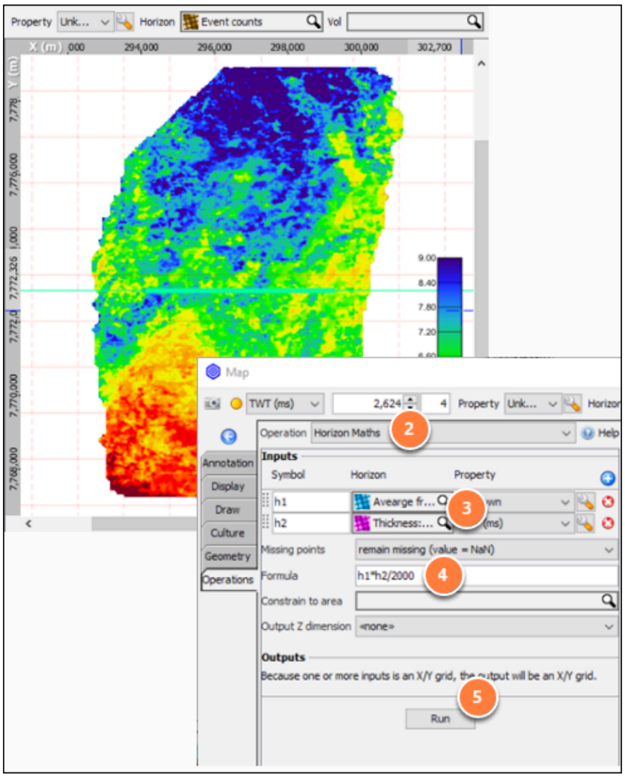
Step 2: Calculate average of instantaneous frequency (Ave_IF) from the top to bottom horizon:

1. Go to **Control panel > Map view**.
2. Click the **Operations** tab and select **Amplitude Extraction** from the **Operation** dropdown menu.
3. Under **Horizons**, use this configuration:
   - **Top**: pick **Horizon** from the dropdown and select the top horizon e.g. TopRes.
   - **Bottom**: pick **Horizon** from the dropdown and select the bottom horizon e.g. BaseRes.
4. Use the created volume from the **Instantaneous Attributes** process e.g. instantaneous attributes: frequency.
5. Use **Simple Average** as the property.
6. Click **Run**.



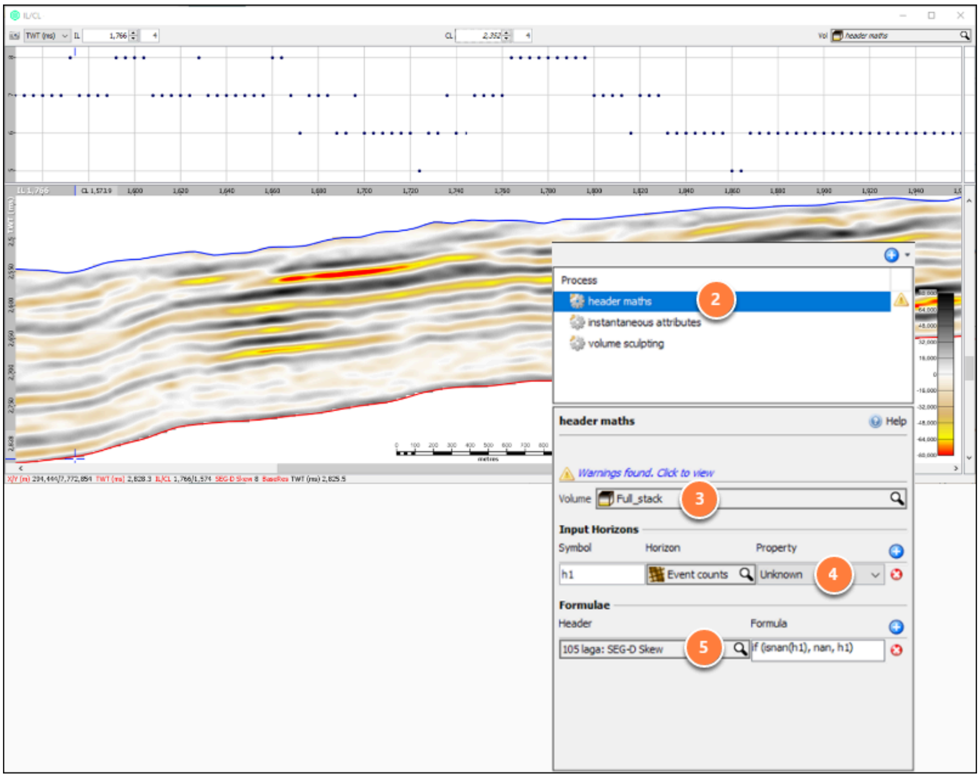
Step 3: Calculate time thickness from the top to bottom horizon:

1. Calculate time thickness from the top to bottom horizon:
2. Go to **Control panel > Map view**.
3. Click the **Operations** tab and select **Horizon Maths** from the **Operation** dropdown menu.
4. Add **2** symbols by clicking the blue "**+**" button and set:
   - **h1**: top horizon e.g. TopRes
   - **h2**: bottom horizon e.g. Base Res
5. Use formula: **h2 - h1**.
6. Click **Run**.



Step 4: Calculate Number of events from top to bottom horizon:

1. Go to **Control panel > Map view**.
2. Click the **Operations** tab and select **Horizon Maths** from the **Operation** dropdown menu.
3. Add **2** symbols by clicking the blue "**+**" button and set:
    - **h1**: the calculated average of instantaneous e.g. Ave_IF
    - **h2**: the calculated time thickness e.g. Thickness_time
4. Use formula: **Ave_IF x (Thickness_time / 2000)** or **h1 * (h2 / 2000)**.
5. Click **Run**.



The results can be QC with the Event counts graphed on top of the seismic section by writing the results from step 4 to one of the unused header in the seismic:

1. Go to **Control panel > Process**.
2. Click the blue "**+**" button and add a **Header Maths** process.
3. Select the seismic in the **Volume** search box.
4. Add a new symbol by clicking the blue "**+**" button.
   - Select the "***Event counts***" horizon as h1.
5. Add a new header formulae by clicking the blue "+" button:
   - Select header: **105 laga: SEG-D Skew**.
   - Use formula: **if (isnan(h1), nan, h1)**.

Note: To QC this event, use the IL/CL view and enable header graphs.

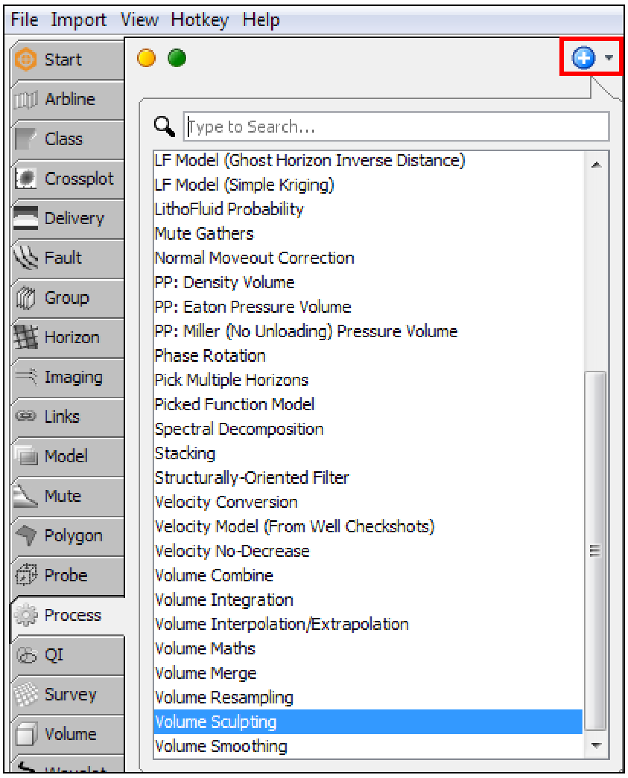
How do I perform an amplitude extraction between two horizons?

An amplitude extraction between two horizons can be done from the Map View (see *Horizon Operations in Map View - Amplitude Extraction*).

Alternatively, you can also perform an amplitude extraction between two horizons by using a sculpted volume. Follow the steps below for a different approach to amplitude extraction between two horizons.

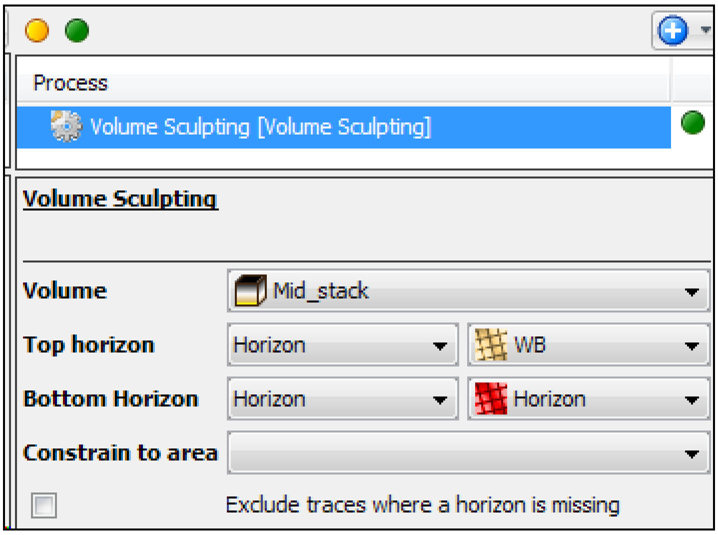
Note: You should first familiarise yourself with how to perform an extraction along a single horizon (see the "*Mapping and Attributes*" tutorial video, or the *Single Horizon Amplitude Extraction* section in the User Manual).

Create a volume sculpting process



1. From the Control Panel, open the **Process** tab.
2. In the **Process** tab, click on the Add icon (+).
3. Scroll down and double click on **Volume Sculpting**.
4. Type a name for the process and click **OK**.

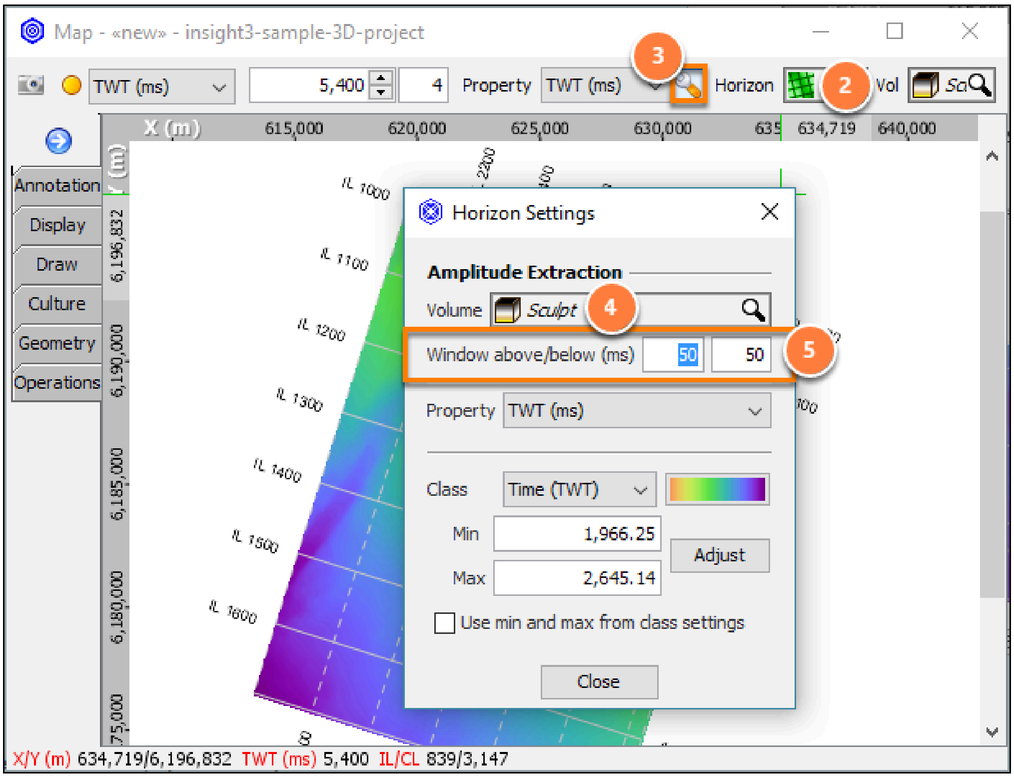
Sculpt volume between horizons



5. In the Details Panel, select the input volume to sculpt at **Volume**.
6. At **Top** and **Bottom Horizon**, choose the top and bottom horizons that you want to extract between (and your data volume).
7. At **Constrain to area**, select a polygon/probe to constrain the sculpting to the area covered by the polygon/probe (*see* *Creating a Polygon in Map View*, *Creating a Probe*).
8. Select the **Exclude traces where a horizon is missing** check box to remove traces where one or both horizons are not present. By default, areas of the volume not covered by the horizon(s) are unmodified.

As a result of this process, a sculpted volume is available in the Volume tab.

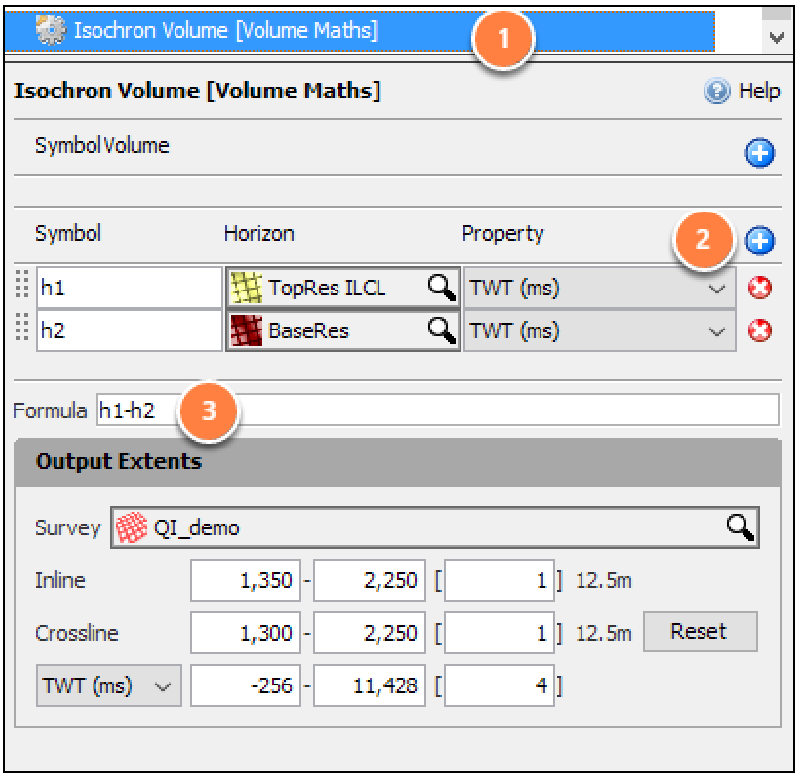
Set amplitude extraction window



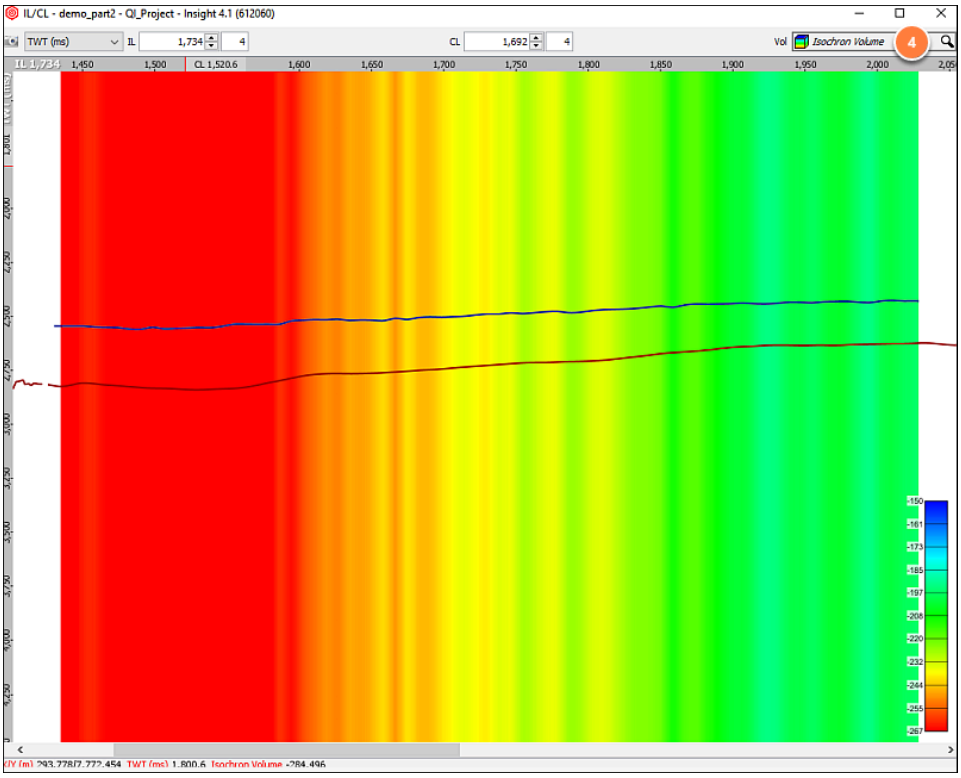
1. From the Control Panel, open the Map View.
2. At **Horizon**, select either the top or bottom horizon.
3. Click on the spanner icon beside the horizon property.
4. Choose the sculpted volume from the previous step as the **Volume** to use.
5. If you chose the top horizon, set your Below amplitude extraction window large enough to cover the entire sculpted volume (or the Above window if you chose the Below horizon).

How do I dynamically update an Isochron Map?

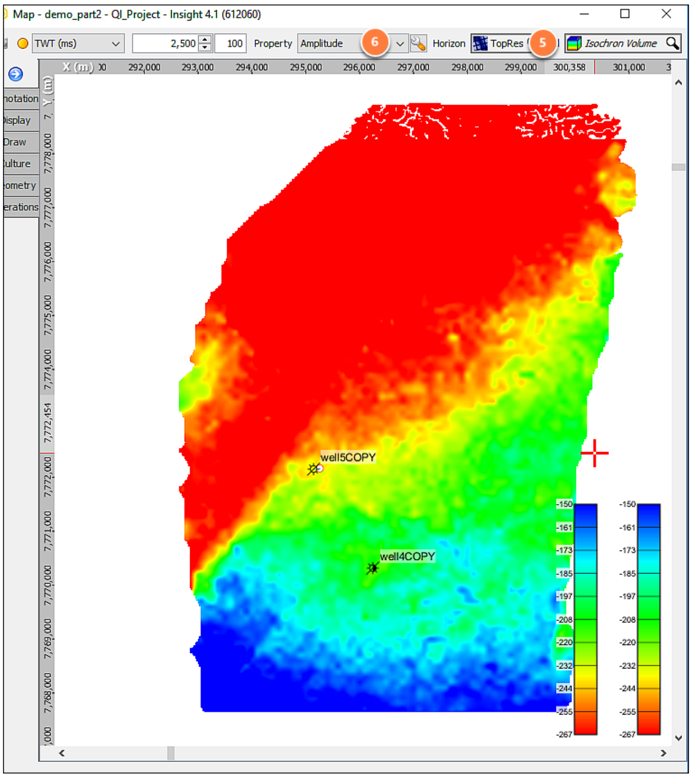
Use Volume Maths to create a volume based on the Isochron values and then extract them onto the horizon. As you make edits to the horizon map, Insight will dynamically update the volume.



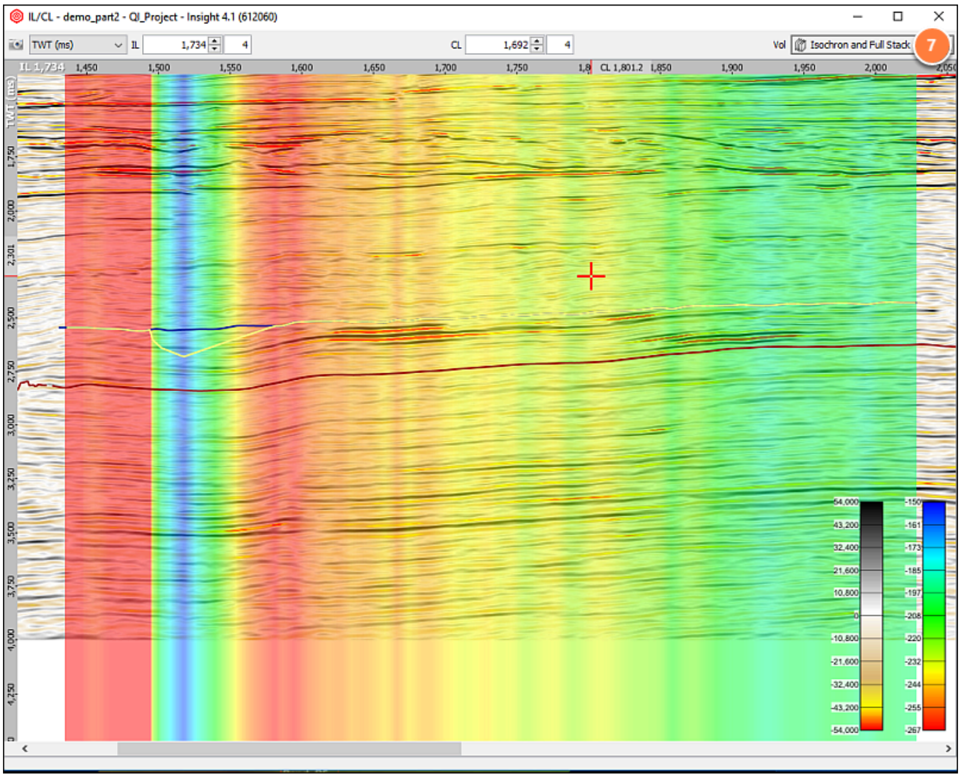
1. Go to the **Process** tab and add a **Volume Maths** process.
2. Add two horizons with the **"+"** in the **Volume Maths panel** and select horizons for **h1** and **h2** respectively e.g. Top and Base horizons for Isochron.
3. Use the formula **h1-h2** in the **Formula** text box. This will create an Isochron volume.



4. Go to the **View** menu and display the volume in **IL/CL View**, with horizons overlaid.

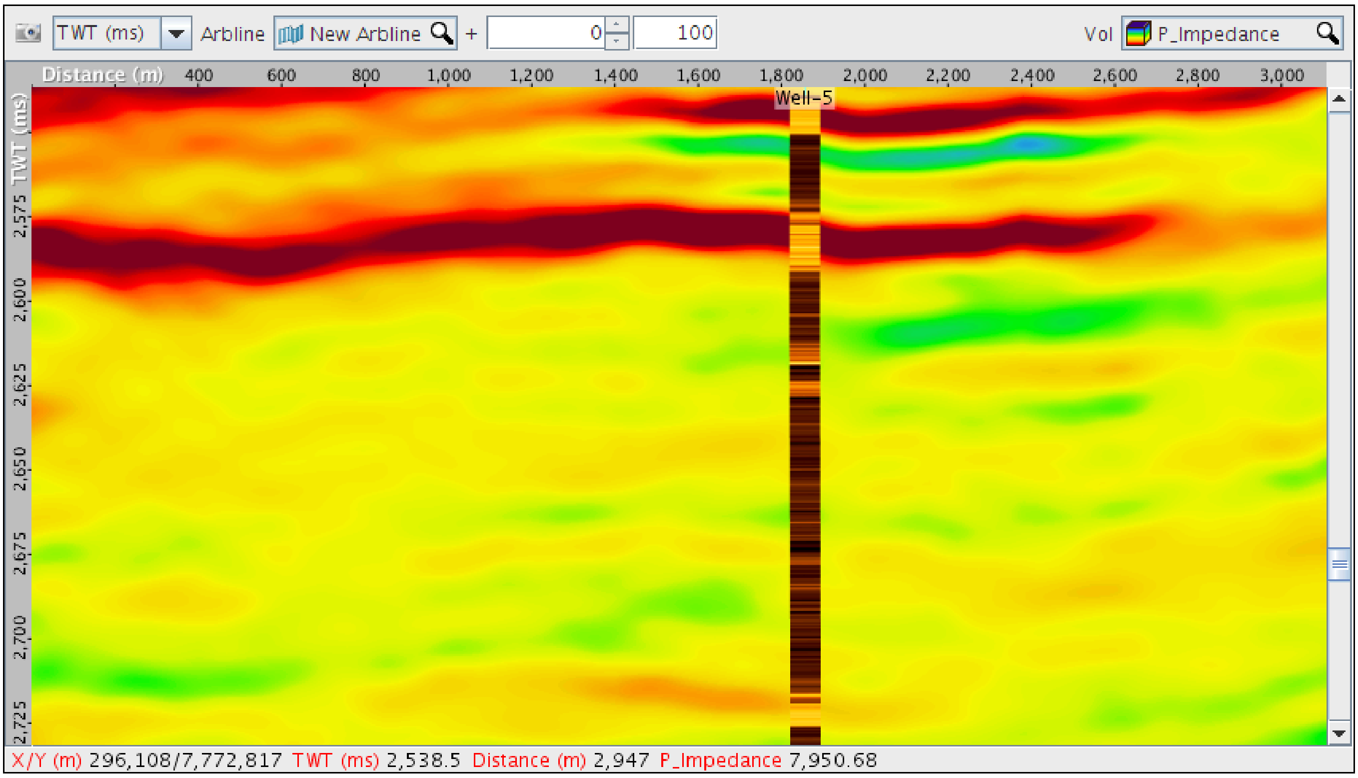


5. Display a horizon you wish to drape the isochron onto in **Map View**.
6. Click ther **Operations** tab and perform amplitude extraction (learn about *Amplitude Extraction*).

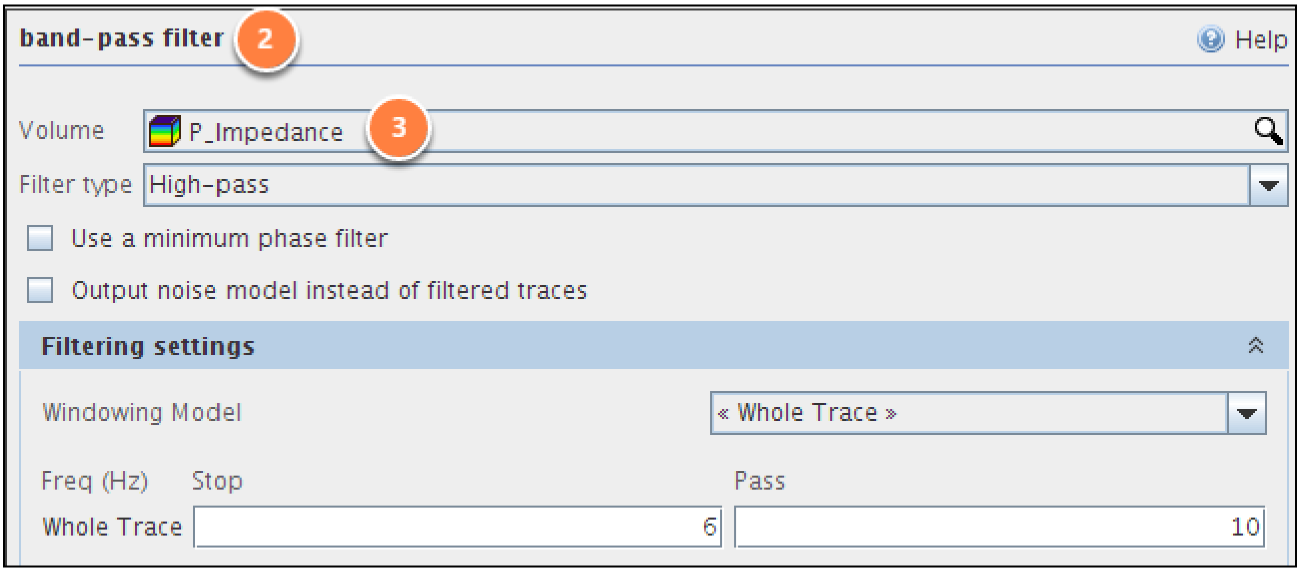


7. Create a **Group** with Isochron volume and Full Stack and set the Isochron volume opacity (in class) to mid (read about *Creating a Group* and *Class Opacity*).
8. Edit anomalous values directly on the input horizons and the **Volume** and extraction will dynamically update.

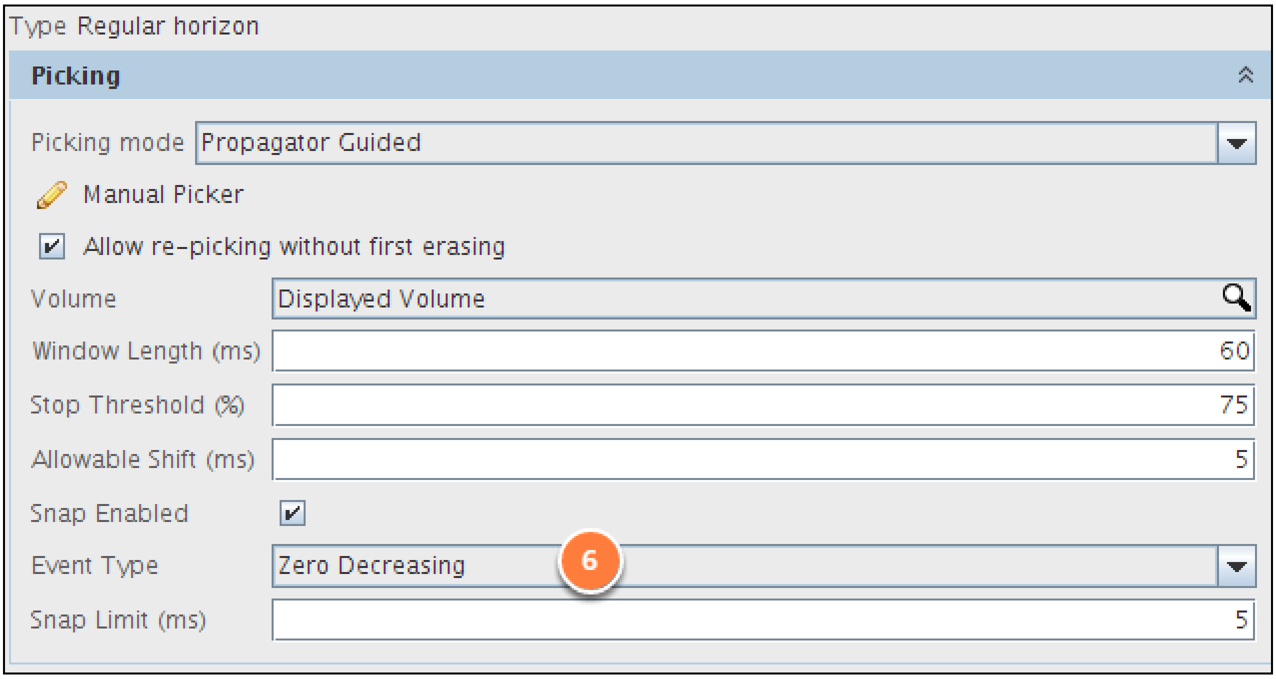
How do I pick events on absolute impedance and other non-zero mean data?



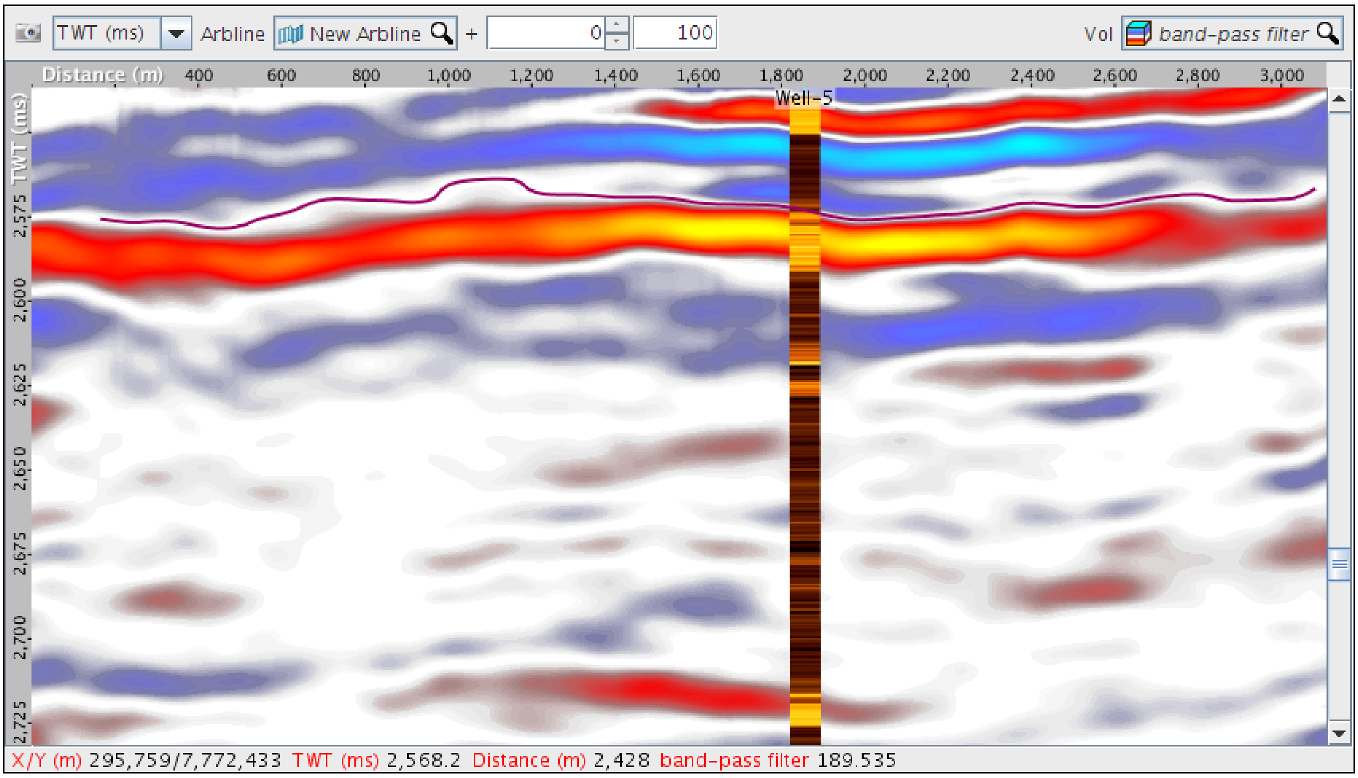
1. From the Volume tab, load impedance or velocity volume (continuous, non-zero mean data).



2. Go to the Process tab and apply a Bandpass Process to remove the low-frequency signal.
3. Select the impedance or velocity volume in the Volume search box.
4. Choose High-pass Filter type.



5. Go to the Horizon tab and pick a new horizon.
6. In the horizon picking settings, choose the appropriate event type. In this example, Zero Decreasing was used to identify this top-sand.



7. Arbline showing Horizon Picked on Band-limited P-Impedance.

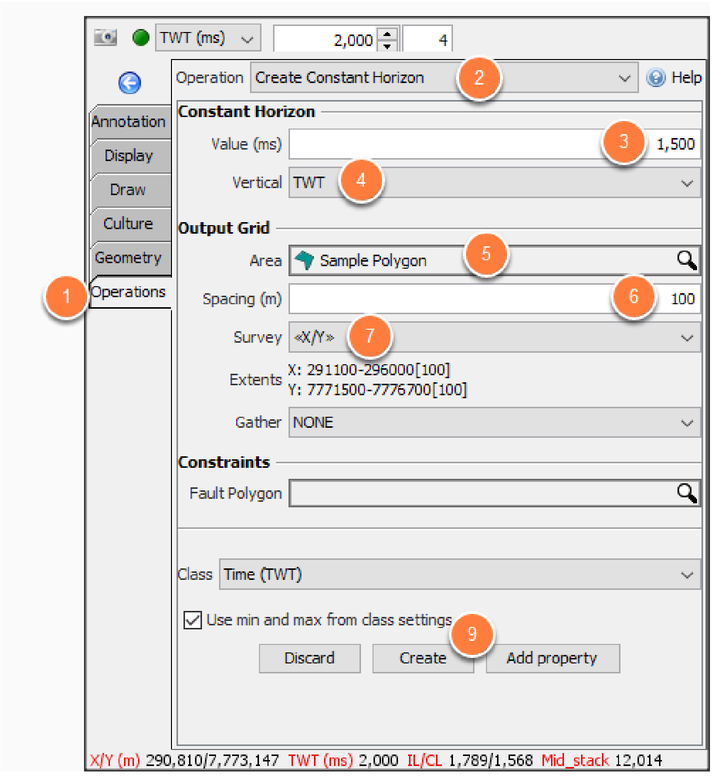
How do I export a time-slice as an X, Y, Amplitude map?

To export a time-slice as an X, Y, Amplitude map:

1. [Create a constant time or depth horizon](#)
2. [Configure the amplitude extraction](#)
3. [Capture the result as a custom property](#)
4. [Custom horizon export as X/Y/Amplitude](#)

Workflow

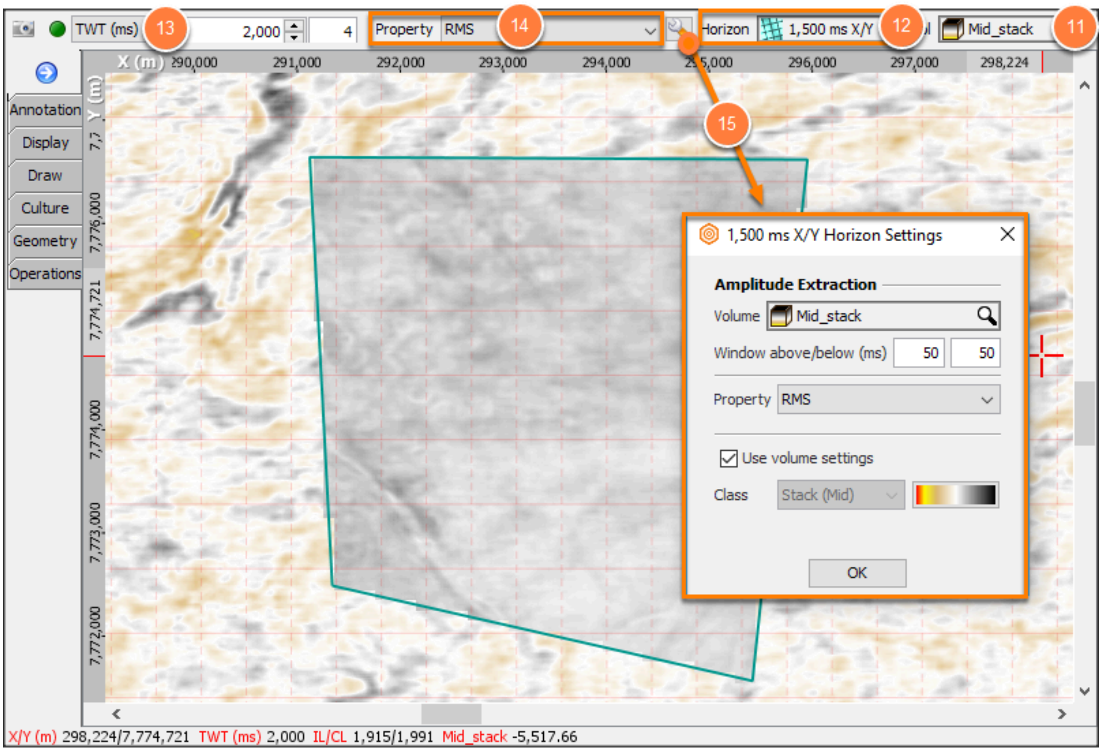
Create a constant time or depth horizon



1. Go to the **Map View > Operations** tab.
2. Select **Create Constant Horizon** in the **Operations** dropdown
3. Enter a **Value** for the horizon.
4. Select the **Vertical** dimension to use.
5. At **Area**, select a polygon or probe.
   - To create a probe or a polygon, see [Creating a Polygon in Map View](#) or [Creating a Probe](#).

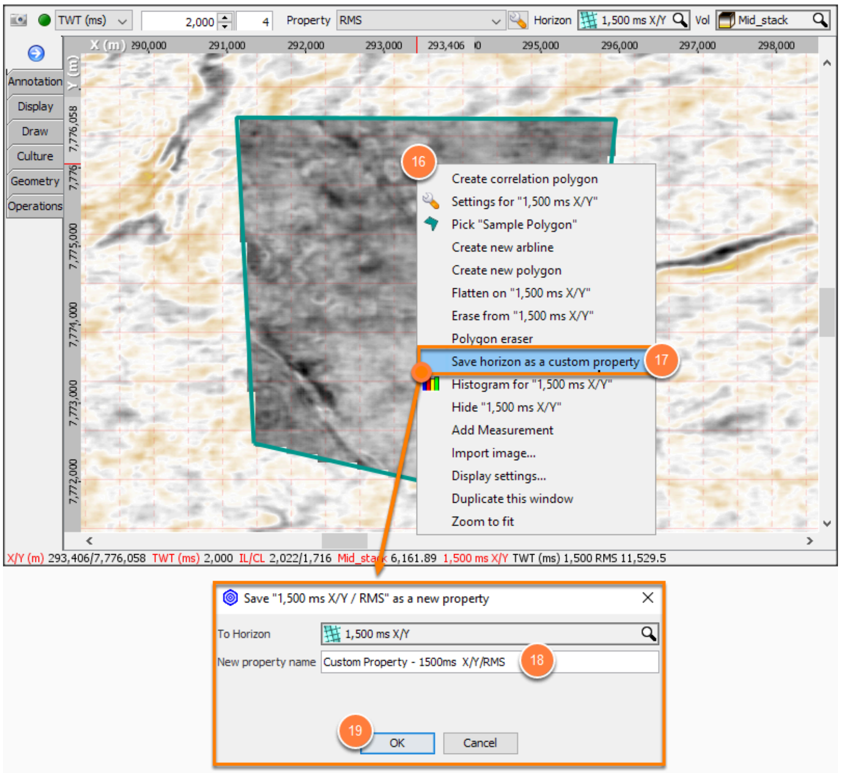
6. Enter the **Spacing (m)** (grid cell size, or increment) for the horizon.
7. Select the **Survey** to use.
   - Choose **<<X/Y>>** for a regularly spaced grid without a survey.
8. Click **Calculate** to start the operation.
9. Click **Create** to create a new horizon containing the extracted values.
10. By default, the new horizon will be named: “***constant_value survey/gathers***” e.g. ***1,500 ms X/Y***.

Configure the amplitude extraction



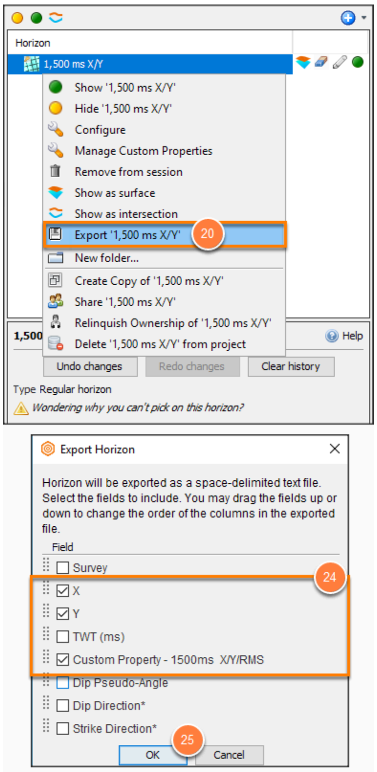
11. Select the **volume** to extract from.
12. Select the **constant horizon**.
13. (*Optional*)Turn off the time slice.
14. Select the **horizon property** to extract (e.g. Amplitude).
15. Click the **tool button** to adjust extraction properties (for windowed properties such as RMS).

Capture the result as a custom property



16. Right click on the displayed amplitude map
17. Click **Save horizon as a custom property**.
18. Enter a name for the property.
19. Click **OK**. The property will be added to that selected horizon.

Custom horizon export as X/Y/Amplitude



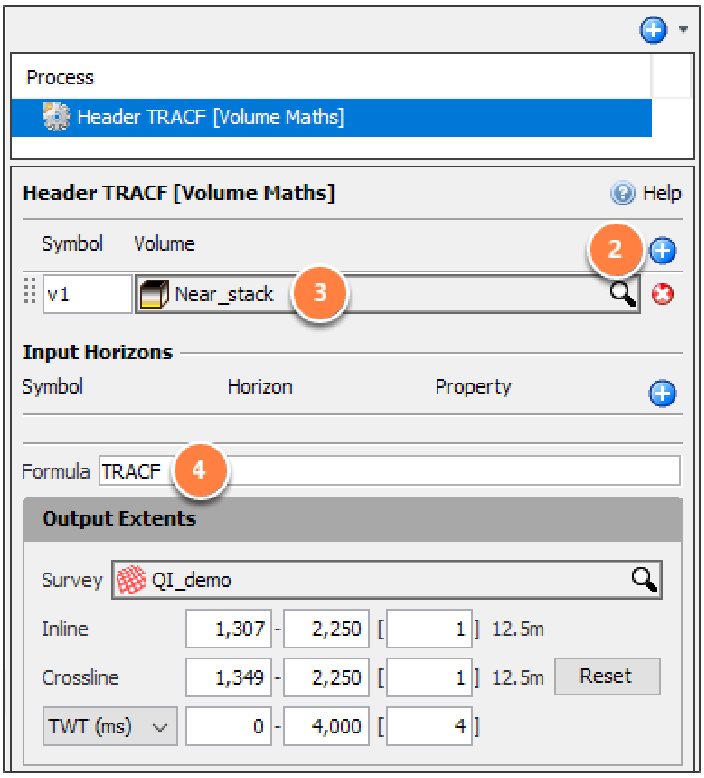
20. Go to **Control Panel > Horizon** tab.
21. Right-click on the constant horizon and choose **Export '*horizon_name*'**.
22. In the **Export window**, choose **Export customised text file**.
23. Click **OK**.
24. In the **Export Horizon** window, choose columns **X**, **Y** and the **custom property**.
25. Click **OK** and save the file in a desired location and name.

How to create a horizon from trace headers

To create a horizon from trace headers, use a combination of *Volume Maths* and the *Amplitude Extraction* horizon operation.

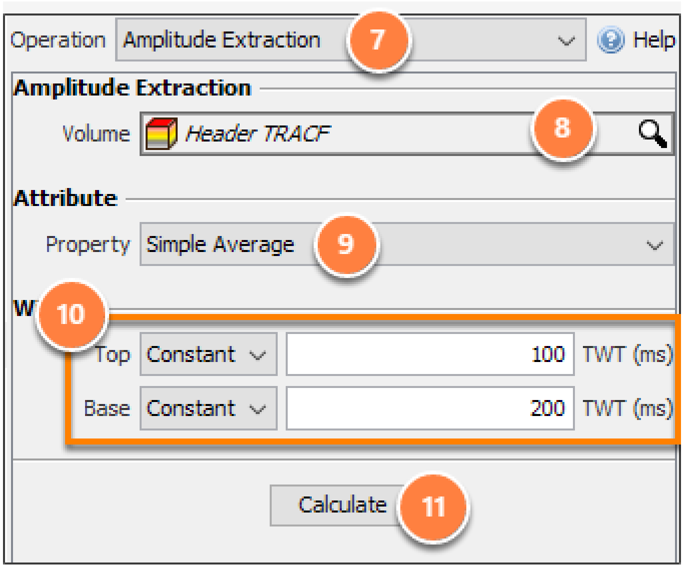
Extract the header values to a volume

1. In the **Control Panel / Process** tab, add a new **Volume Maths** process. Name the process, e.g. "*Header TRACF*", and Click **OK**.



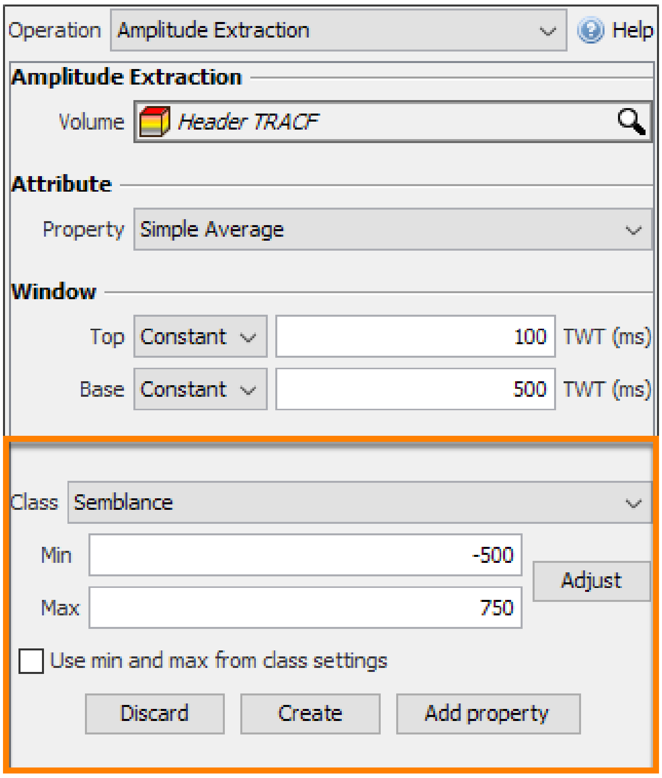
2. In the **Symbol** & **Volume** section, click the blue "+" icon.
3. Choose the **volume** containing the header values to extract.
4. In the **formula**, enter the header name, e.g. TRACF or SWDEP
   - See this list of available *trace headers*.
5. The result volume containing the header value will be in the **Volume** tab.

Create a horizon from the volume



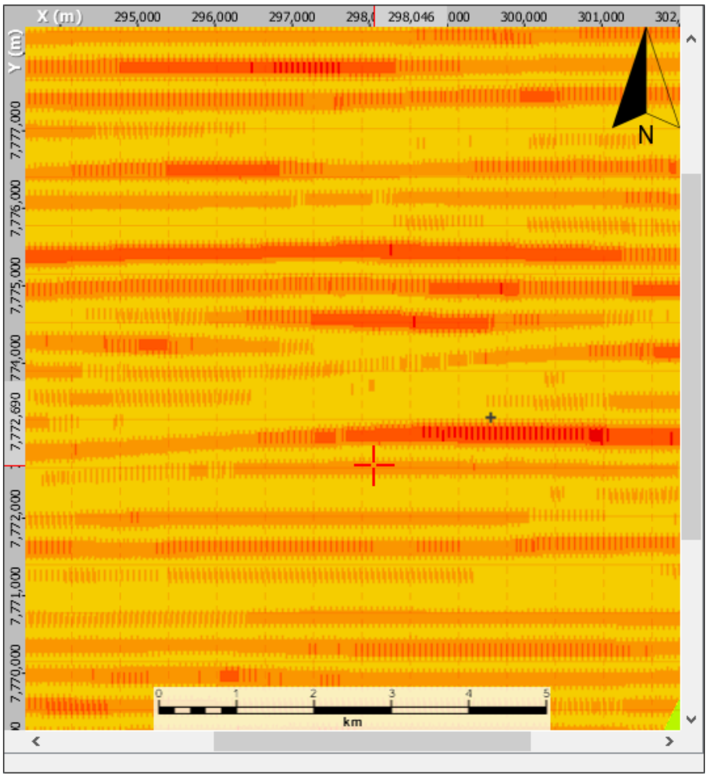
6. In a **Map View**, open the **Operations** tab.
7. Choose **Operation / Amplitude Extraction**.
8. For **Volume**, choose the volume math result, e.g. “*Header TRACF*”
9. For **Property**, choose **Simple Average**.
10. Enter any two different values for **Top** and **Bottom**.
11. Click **Calculate**.

Completing the operation



After running the operation:

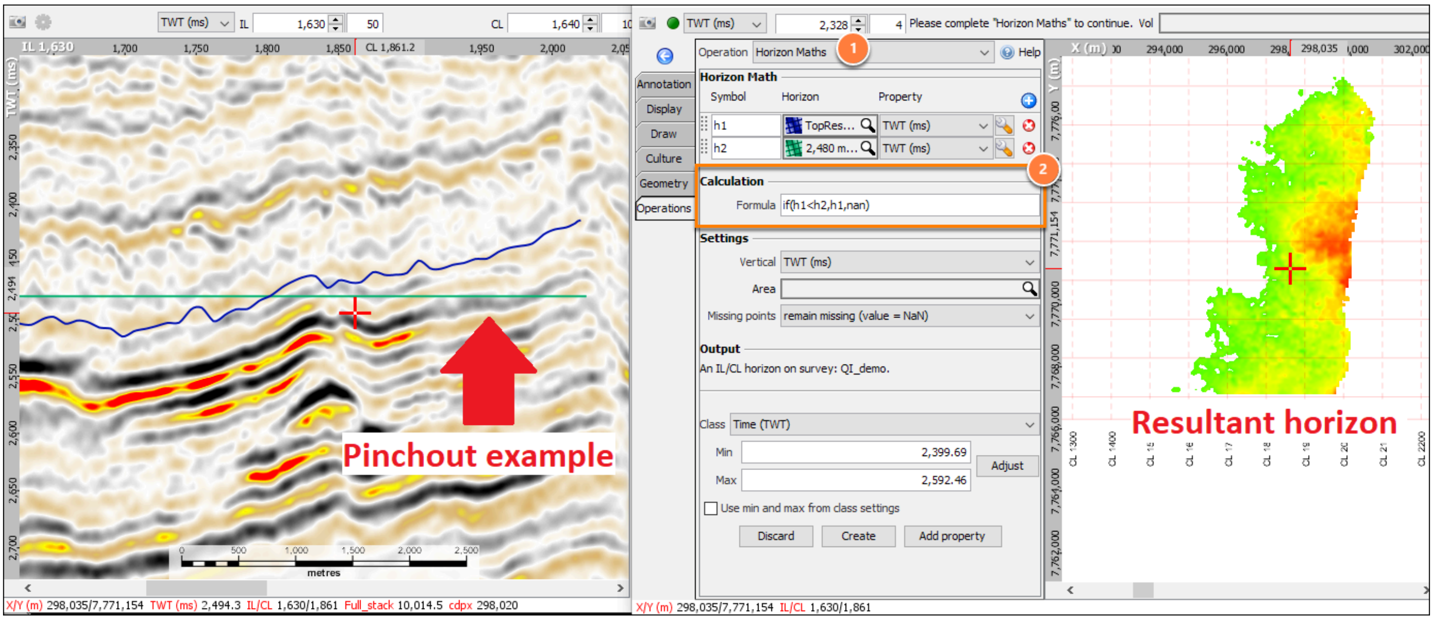
- At Class: Choose the [*Display Class*]() for the results
  - Use **Unknown** for raw values unless a better choice is available.
- Review the results in the display by:
  - Check: **Use min and max from class settings**
  - Click **Adjust** to set the **Min** and **Max** from the extracted amplitude.
- Click **Create** to save the values as a new horizon


X (m)
295,000
296,000
297,000
298,046
300,000
301,000
Y (m)
7,777,000
7,776,000
7,775,000
7,774,000
7,772,690
7,772,000
7,771,000
7,770,000
N
km

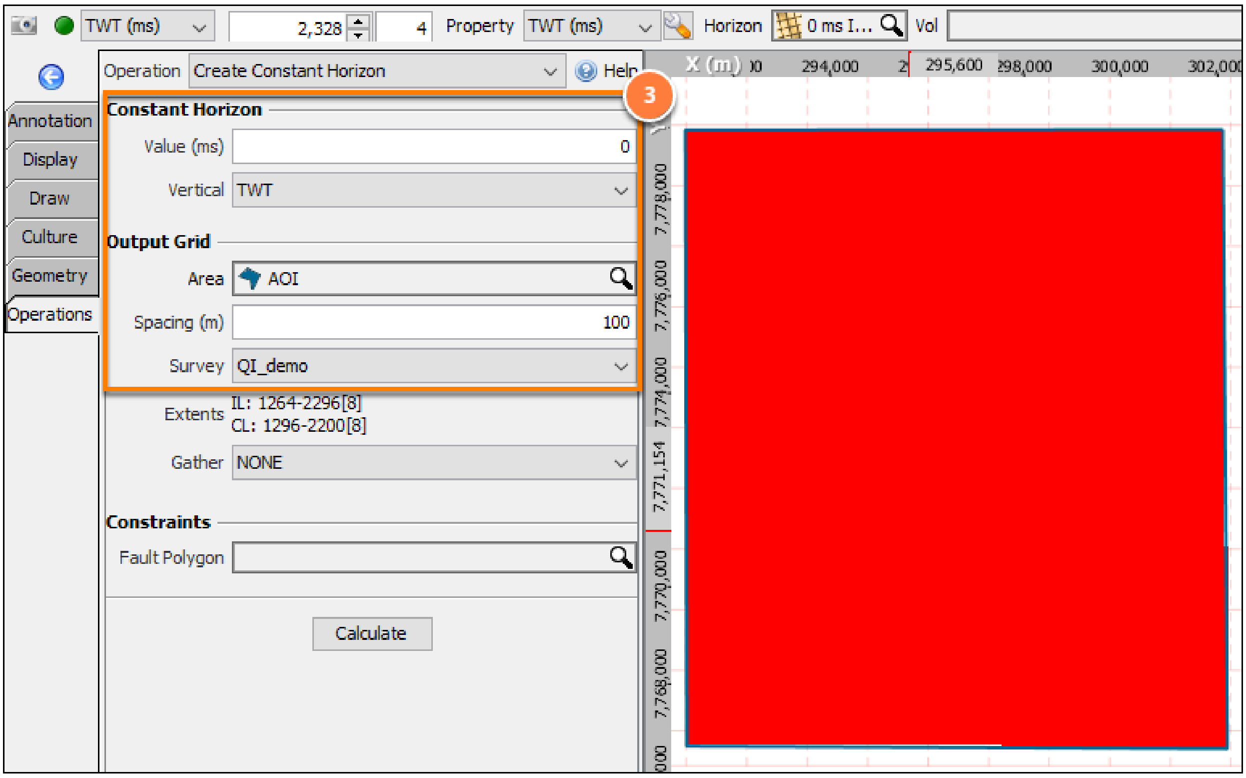
How to mark a pinchout/intersection of two surfaces using a polygon?

Investigating stratigraphic traps or a stratigraphic surface which has been interrupted by a fault? How do we map the resultant pinchout/intersection between these two horizons using DUG-Insight? The workflow below discusses how to create a detailed polygon outlining the pinchout/intersection.

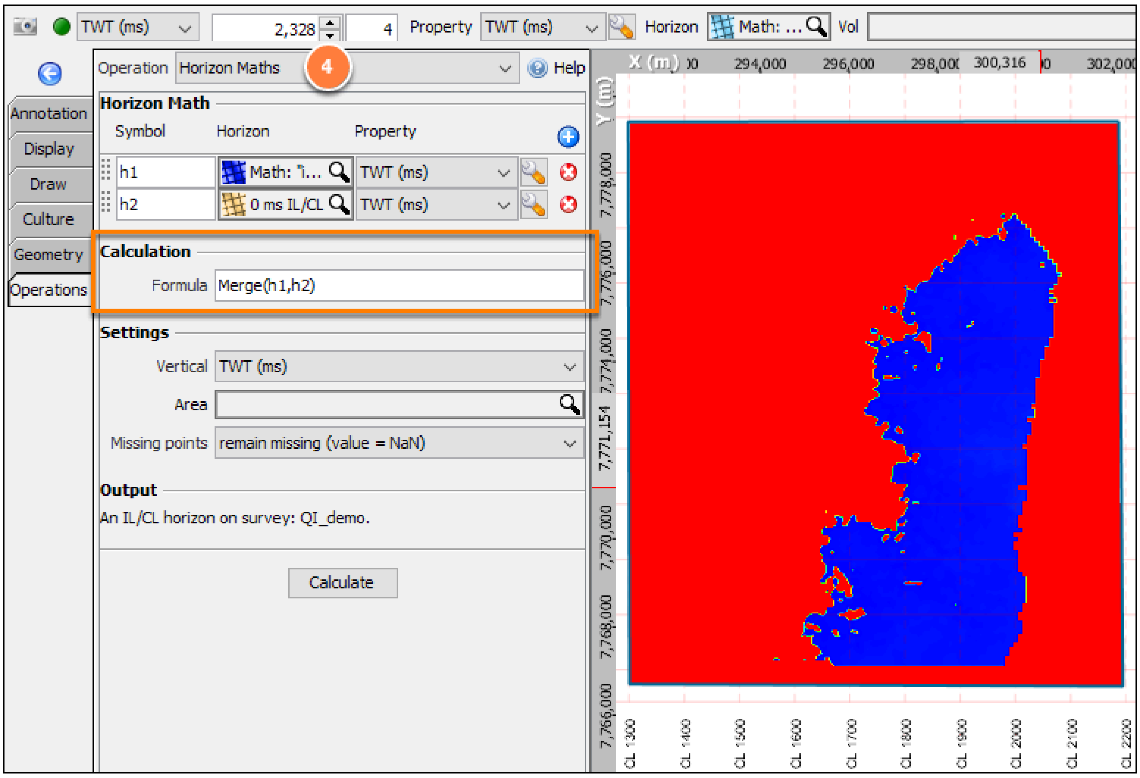
Workflow



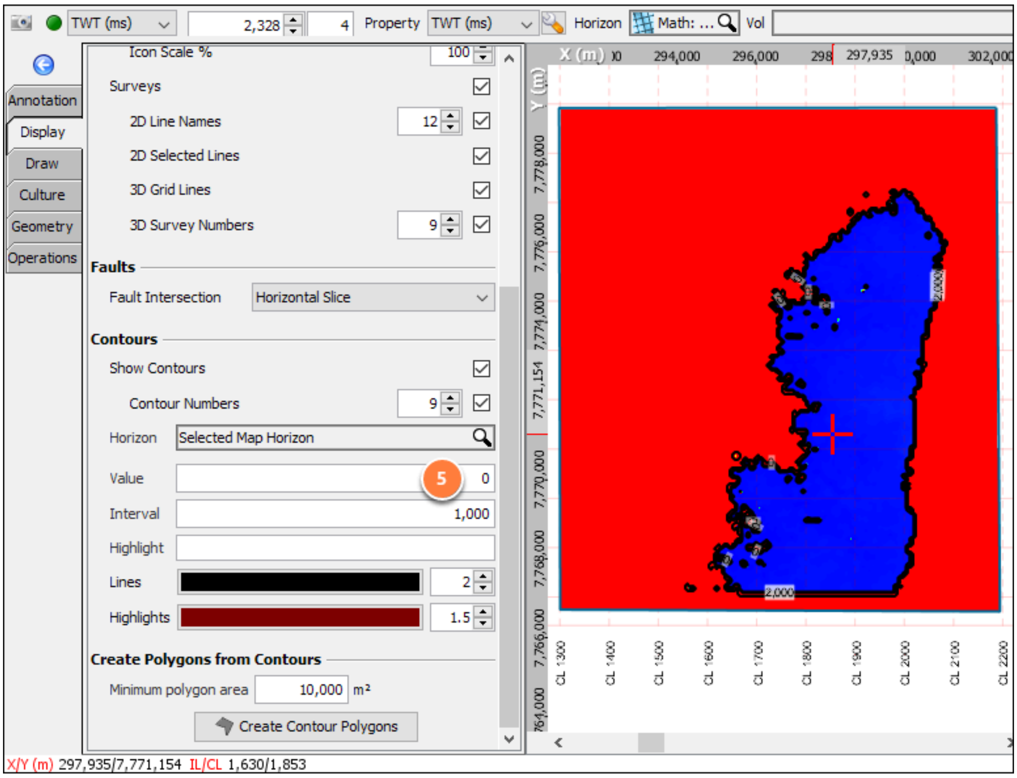
1. Pick the **top (h1)** and **base (h2)** horizons of your pinchout structure
2. Create a pinchout horizon using *Horizon maths*
   - **Inputs:** h1 = top horizon, h2 = base horizon
   - **Formula:** if (h1<h2, h1, nan)
   - (If top horizon is greater than base horizon use top horizon. If not use nan as value.)



3. Create a constant horizon using the *[Constant horizon]*
    - **Value:** 0 ms
    - **Vertical:** TWT
    - **Output grid:** polygon which covers entire horizon area
    - **Spacing:** 100
    - **Survey:** Same 3D survey as original horizons



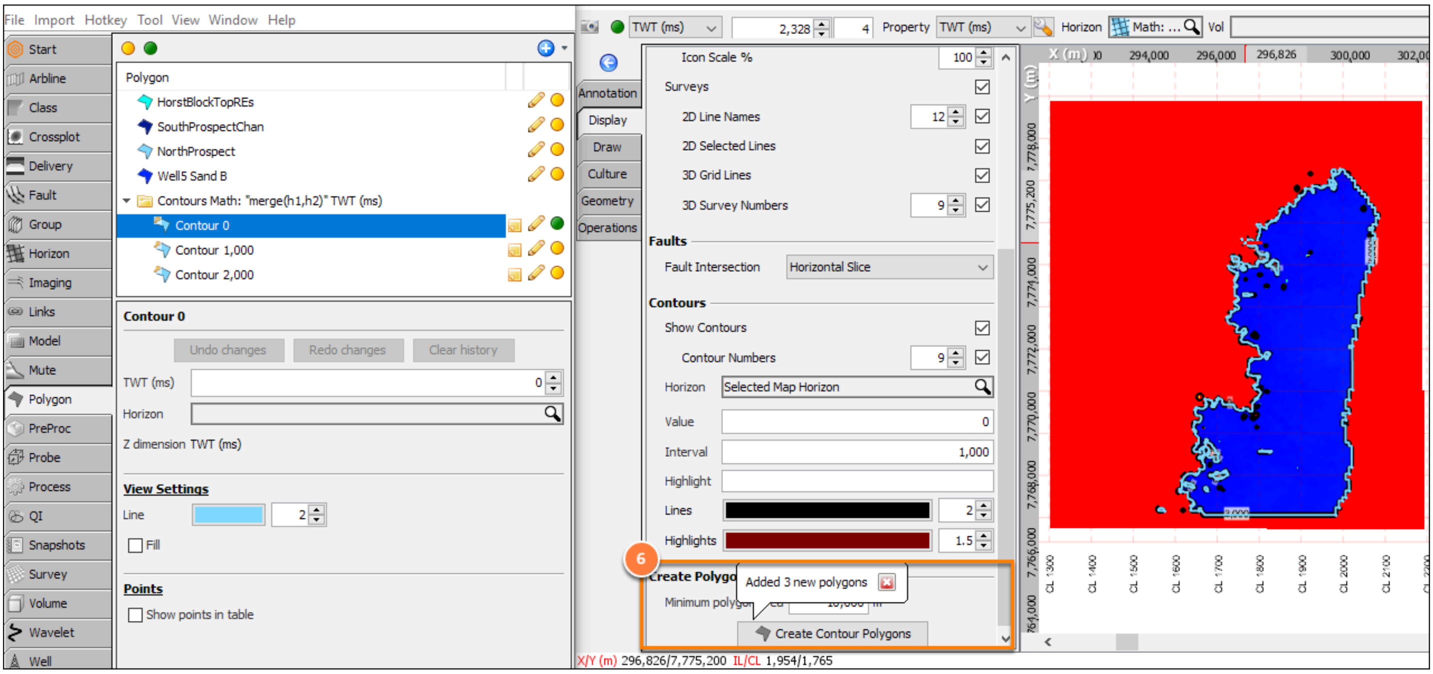
4. Merge the pinchout horizon created in step 2 and constant horizon created in step 3 using [*Horizon Maths*]
   - **Inputs:** h1 = pinchout horizon, h2 = constant horizon
   - **Formula:** merge (h1, h2)
   - (Merge pinchout horizon and constant horizon, give priority to pinchout horizon)



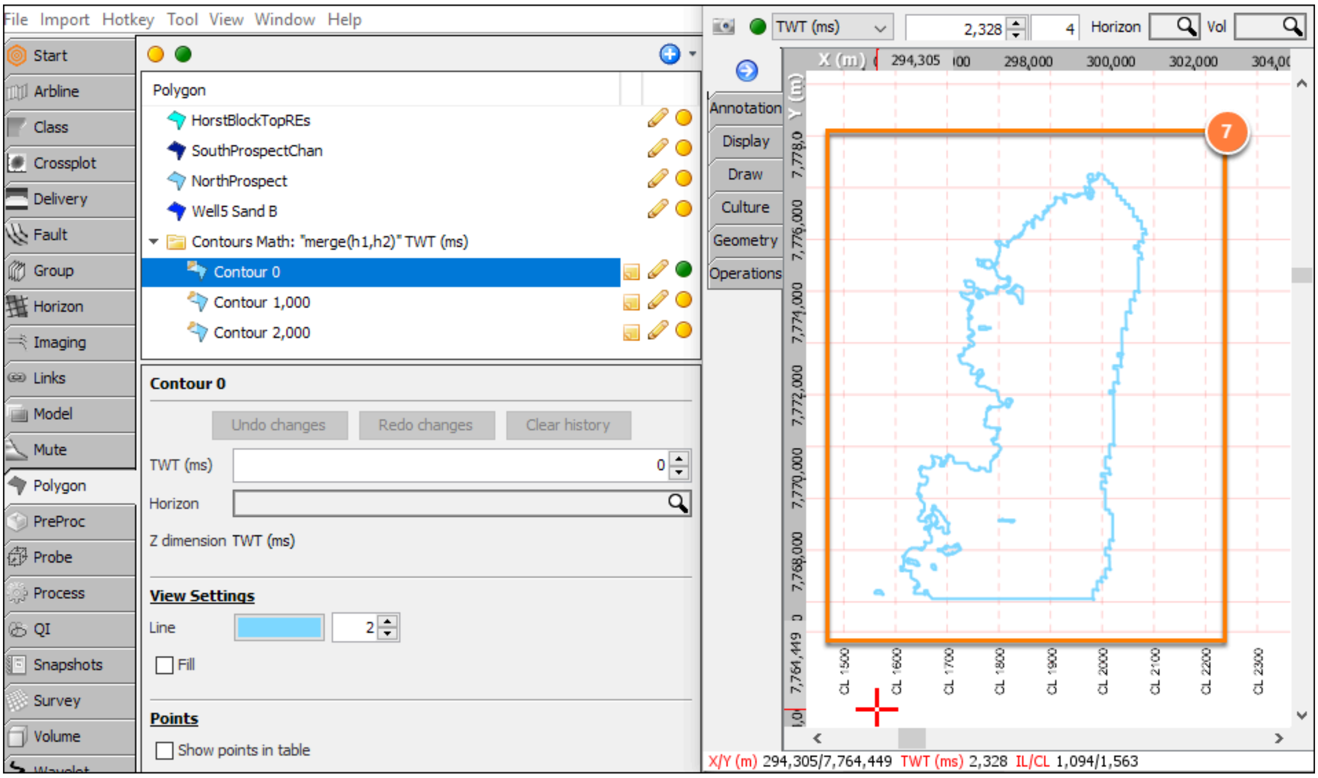
5. Display in Map View.

We now have the intersection represented clearly on the merged horizon. Create a polygon along this boundary using create polygon from contours tool.

6. Open **Display** tab and select contours
7. Display 0 *contour* in Map View



8. Create polygon from **contour 0** using *Create contour Polygons*



9. Final polygon showing pinchout intersection.

Faults

How do I import or export faults?

Insight can import and export faults in three ways: using either the GeoQuest IESX format, VoxelGeo XYZ or using DUG's own (.*dufault*) format.

Importing fault sticks from a GeoQuest IESX file

```
PROFILE test            TYPE 2  1 DUG Insight fault
SNAPPING PARAMETERS
   626281.000000  6178923.000000  1 14  2604.58      0.00      1.00 1.00          0.00 2 line
   626269.250000  6178885.500000  1 14  2706.64      0.00      1.00 1.00          0.00 2 line
   626210.460887  6178697.875172  1 14  2833.34      0.00      1.00 1.00          0.00 2 line
   626110.375000  6179213.000000  2 14  2602.02      0.00      1.00 1.00          0.00 2 line
   626051.687500  6179027.000000  2 14  2860.21      0.00      1.00 1.00          0.00 2 line
   626009.880042  6178894.498409  2 14  3027.05      0.00      1.00 1.00          0.00 2 line
   625971.242065  6178772.041944  2 14  3113.45      0.00      1.00 1.00          0.00 2 line
   625918.750000  6179436.500000  3 14  2590.09      0.00      1.00 1.00          0.00 2 line
   625858.500000  6179245.500000  3 14  2987.83      0.00      1.00 1.00          0.00 2 line
   625792.725892  6179036.987889  3 14  3194.59      0.00      1.00 1.00          0.00 2 line
   625732.796727  6178847.004976  3 14  3373.37      0.00      1.00 1.00          0.00 2 line
   625721.687500  6179643.000000  4 14  2575.68      0.00      1.00 1.00          0.00 2 line
   625643.375000  6179394.500000  4 14  3031.01      0.00      1.00 1.00          0.00 2 line
   625573.404587  6179172.470980  4 14  3340.07      0.00      1.00 1.00          0.00 2 line
   625501.865309  6178945.463661  4 14  3515.30      0.00      1.00 1.00          0.00 2 line
```

This standard format can be produced by almost every software package. A single file can contain an arbitrary number of faults, each of which will be imported as a separate fault in Insight. Unassigned fault sticks are imported in the same way.

Insight imports only fault sticks, from which it will build its own tesselated surface and fault polygons. Insight does not yet import fault polygons or surfaces from other software.

The example above shows a GeoQuest IESX data file.

Importing fault sticks from a DUG fault file

The DUG fault file is a simple text format that may be more convenient for scripting.

It consists of a single X/Y/Z point per line, separated by whitespace, with a blank line to indicate a new fault stick. The filename generally has the extension ".*dufault*"

This is an example of a three-stick .*dufault* file:

```
626281 6178923 2604
```

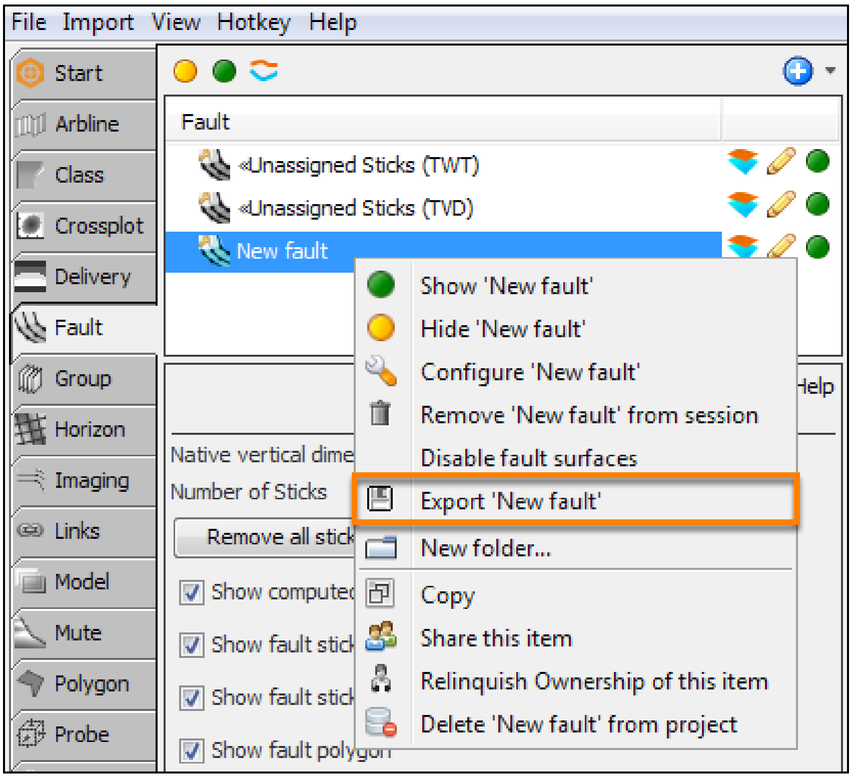
```
626269 6178885 2706
626210 6178697 2833
626110 6179213 2602
626051 6179027 2860
626009 6178894 3027
625971 6178772 3113
625918 6179436 2590
625858 6179245 2987
625792 6179036 3194
625732 6178847 3373
```

Importing fault sticks from a VoxelGeo XYZ fault file

```
VoxelGeo XYZ data file

Name:  CS40_CS60_40,30,50_002_N-E_EM_21X_Yellow
Type:  Fault
World Coordinates
Units:  Meters  Meters  Meters

Surface Polyline Group
Color: (0.988235 0.988235 0)
          x                y              depth     nline
        685452.4         4040129          2400    1
        685487.7         4040167          2400    1
        685573.2         4040259          2400    1
        685578.1         4040264          2400    1
        685664.2         4040357          2400    1
        685671.7         4040366          2400    1
        685718.1         4040416          2400    1
        684945.2         4039562          2440    2
        684960.4         4039578          2440    2
        685040.9         4039665          2440    2
        685054.7         4039680          2440    2
        685135.3         4039767          2440    2
        685211.6         4039849          2440    2
        685231.6         4039871          2440    2
        685313.9         4039959          2440    2
        685337.2         4039984          2440    2
        685413.3         4040066          2440    2
        685485.7         4040145          2440    2
        685554.9         4040219          2440    2
        685587.4         4040254          2440    2
        685661.6         4040334          2440    2
        685694.2         4040370          2440    2
        685801.4         4040485          2440    2
        685895.1         4040586          2440    2
        684908.2         4039502          2480    3
        684974.2         4039573          2480    3
        685031.8         4039635          2480    3
        685078.1         4039685          2480    3
        685138.7         4039750          2480    3
        685196.1         4039812          2480    3
        685245.9         4039866          2480    3
        685317.9         4039944          2480    3
        685370.5         4040000          2480    3
        685428.6         4040063          2480    3
        685483.8         4040122          2480    3
        685536.6         4040179          2480    3
        685596.8         4040244          2480    3
        685659.1         4040311          2480    3
        685716.7         4040374          2480    3
        685808.1         4040472          2480    3
        685920.2         4040593          2480    3
        685944.3         4040619          2480    3
        686063.2         4040747          2480    3
        686064.4         4040748          2480    3
        686072.1         4040757          2480    3
```

The example above shows a VoxelGeo XYZ data file.

Exporting fault sticks as a GeoQuest IESX file



This standard format can be read by almost every software package. By default, Insight exports the fault sticks to the GeoQuest IESX format (.*dat* file extension). If you have defined a velocity volume for time/depth conversion (see *Converting Time-Depth*), you can export the fault based on the time/depth domain.

1. In the **Fault** tab, right click on the fault(s) you wish to export.
2. Click **Export 'Fault'**.
3. In the **Export Fault** window, select the time (TWT) or depth (TVDSS) domain to export your fault.
4. Click **OK**.
5. Browse to the location you want to export the fault(s) and click **Save**.

Exporting fault surfaces as a GoCAD file

When exporting a single fault, Insight will also export the tesselated fault surface in a separate GoCAD TSurf file alongside the GeoQuest file containing the sticks.

Wells

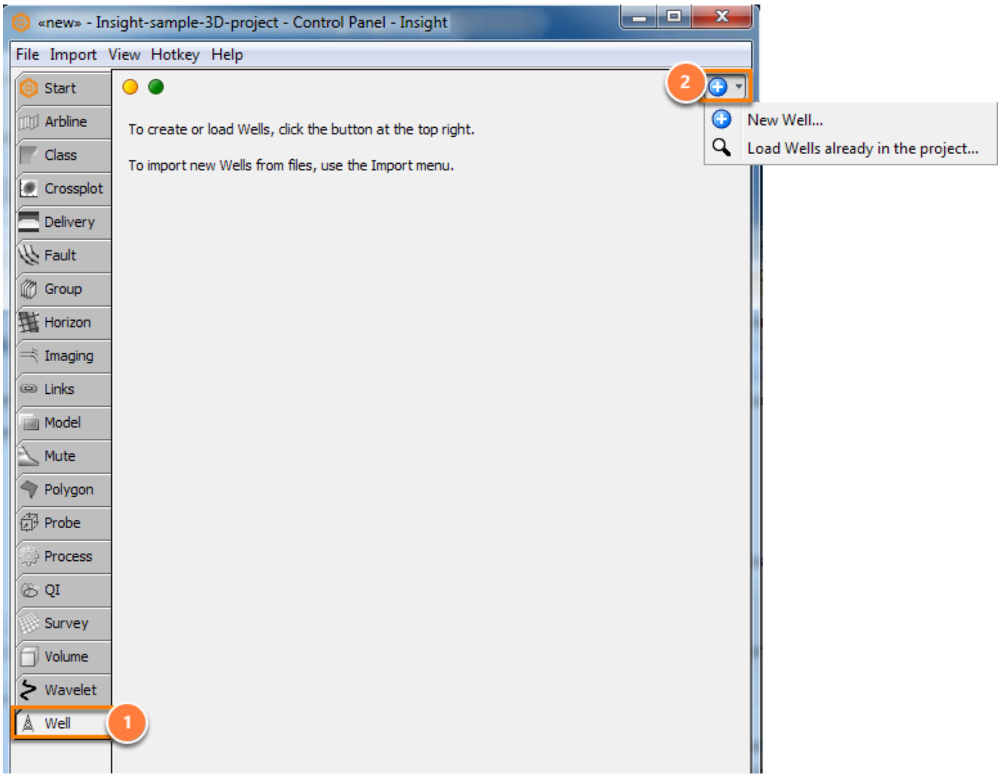
How do I load well data?

DUG Insight reads well data from text files. Curve and log data should be stored in LAS format, with a single LAS file per well.

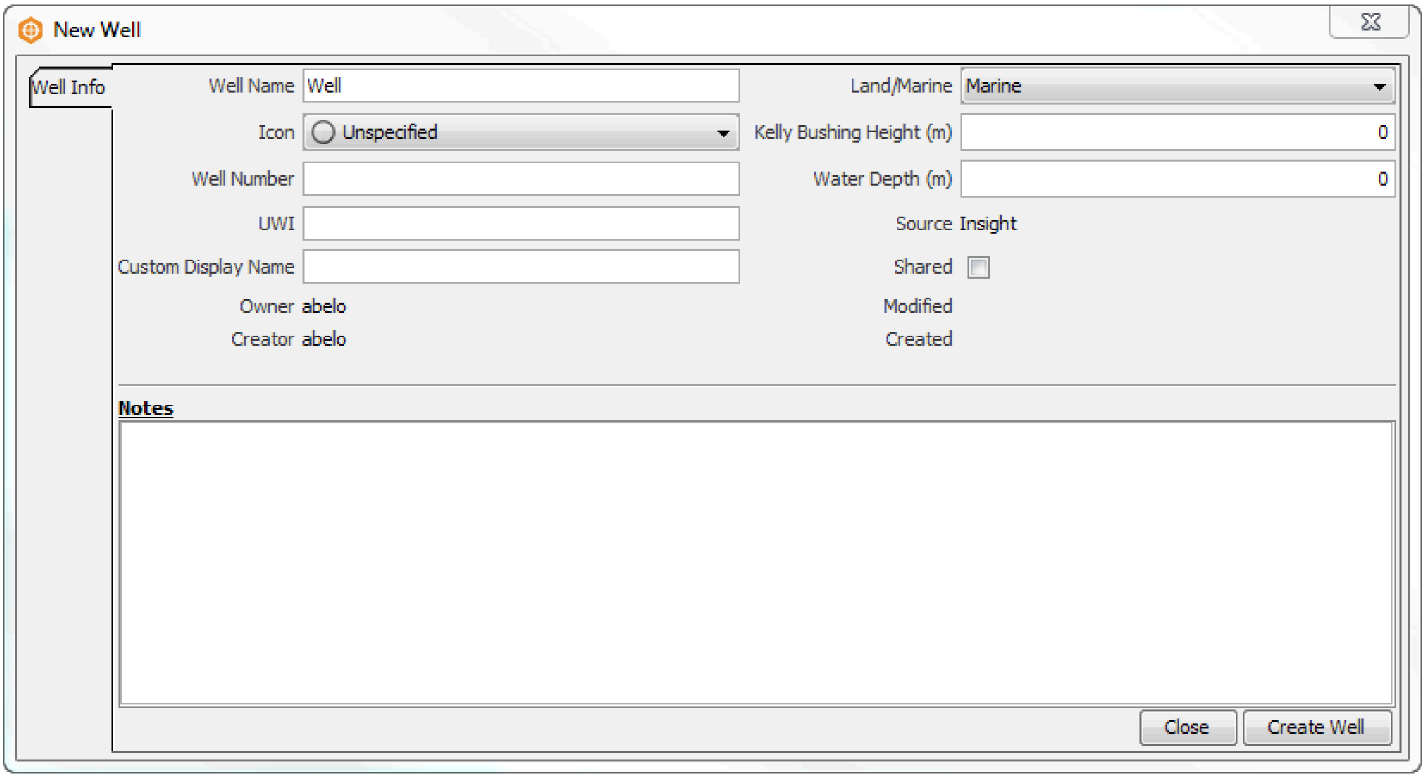
The first time you import a well for use in DUG Insight, you will use the Well Import dialog to create a duwell file. The duwell file describes the well, and contains pointers to the LAS, checkshot, deviation, and markers files.

You can also bulk import all well data to create a new well.

Create well file



1. From the Control Panel, open the **Well** tab.
2. Click on the Add icon and select **New Well**.
3. The New Well window will be displayed.



4. Fill in all of the relevant information for your well and click **Create Well**.
5. Once you have a duwell file, you can load it by clicking on the **Add** icon and select **Load Wells already in the project**.

Checkshot (time-depth) Surveys

Time-to-depth information is required for DUG Insight to display well data in time. Checkshot data points require two values:

- the measured depth at the well, and
- its corresponding two-way-time

The depth column label should indicate the depth datum. Expected depth labels are:

- MDKB: Measured Depth from Kelly Bushing / Round Table
- MDSS: Measured Depth Sub Sea
- MD: Measured Depth (assumed to be MDKB)

The time column label should be "TWT". No other time formats are accepted.

Sample checkshot survey:

```
Time-depth survey for Well-1
2
MDKB    TWT
1210.00 1170.07
1710.00 1500.07
```

```
2210.00 1800.04
2700.00 2150.06
3200.00 2520.03
3730.00 2810.07
4000.00 2960.74
```

Deviation Surveys

Deviation surveys are required for DUG Insight to accurately display deviated wells. Deviation surveys require four values for each point:

- the measured depth
- the true vertical depth
- the UTM X coordinate at this depth
- the UTM Y coordinate at this depth

The measured depth column label should indicate the datum. Expected measured depth labels are:

- MDKB: Measured Depth from Kelly Bushing / Round Table
- MDSS: Measured Depth Sub Sea
- MD: Measured Depth (assumed to be MDKB)

The true vertical depth column label should indicate the datum. Expected true vertical depth labels are:

- TVDKB: True Vertical Depth from Kelly Bushing / Round Table
- TVDSS: True Vertical Depth Sub Sea
- TVDBML: True Vertical Depth Below Mud Line
- TVD: True Vertical Depth (assumed to be TVDKB)

The UTM X and Y columns should be labelled "UTMX" and "UTMY" or "X" and "Y", respectively.

Sample deviation survey:

```
Deviation survey for Well-1
4
MDKB       TVDSS      X           Y
0.00       -20.06     39844.56    24589.34
1000.00    1020.02    39844.47    24588.95
2000.00    1990.05    39845.82    24590.47
3000.00    2980.00    39895.21    24524.15
4000.00    3410.03    40453.486   23864.922
5000.00    3910.05    41009.257   23219.492
6000.00    4890.06    41036.517   23192.534
```

Well Markers

Markers are text labels applied to points along the well path. Well marker files require two values:

- the marker name (the text placed beside the point at the well)
- the marker depth

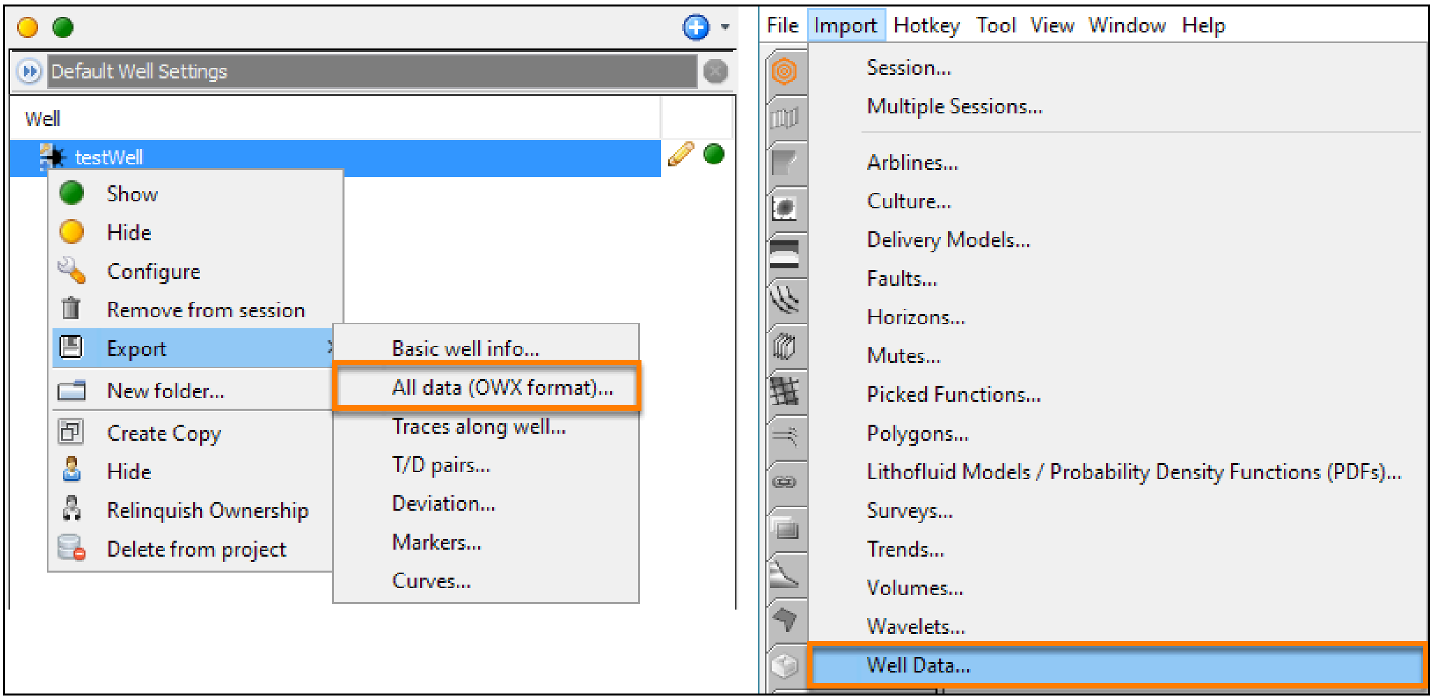
The marker name column should be a short description of the point. Spaces and commas are not allowed in the name. We recommend the use of underscore '_' or hyphens '-' to replace spaces. The marker name column should be labelled "Name".

The marker depth column should indicate the datum, as above.

Sample marker file:

```
Markers for Well-1
2
NAME          MDKB
WaterBottom   185.0
Cretaceous    2700.00
Jurassic      3200.00
Triassic      3730.00
Permian       4000.00
```

Bulk import well data



Well data that was exported to OWX and ASC format can also be imported as a new well.

1. To bulk export well data, right click on the well in the Control Panel and select Export > **All data (OWX format)**.
2. To import this file, open the Import menu and select **Well Data**.

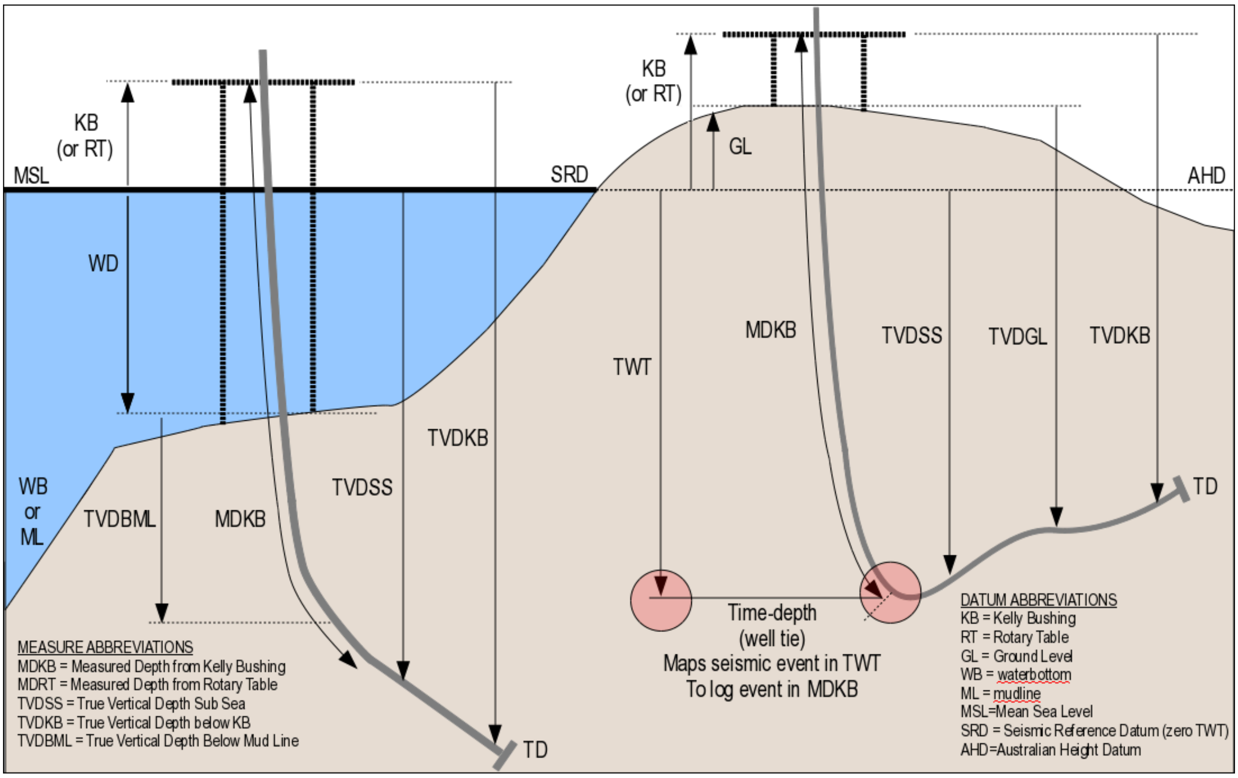
Creating checkshots from a velocity model

This FAQ is no longer relevant. DUG Insight (4.0+) includes a more convenient approach. See this page for details:

- http://help.dugeo.com/m/Insight4-0/l/542676

Height datums and abbreviations

The diagram below depicts the measurement of each height datum together with its abbreviation.



TVDSS is True Vertical Depth SS. This is measured from mean sea level (MSL). It does not reference KB.

Kelly Bushing (KB) is the height of the Derek (drill rig) measured from the MSL. The interpreter should manually enter the KB height under the well info tab (see *Creating a New Well*).

The KB information is also stored when MDKB or TVDKB is entered in at a well (usually in the well Path/Deviation). In these instances, the KB will be checked against the KB in the well info to ensure they match. If they do not, the user will be given a warning.

Why can't I see my well in section views?

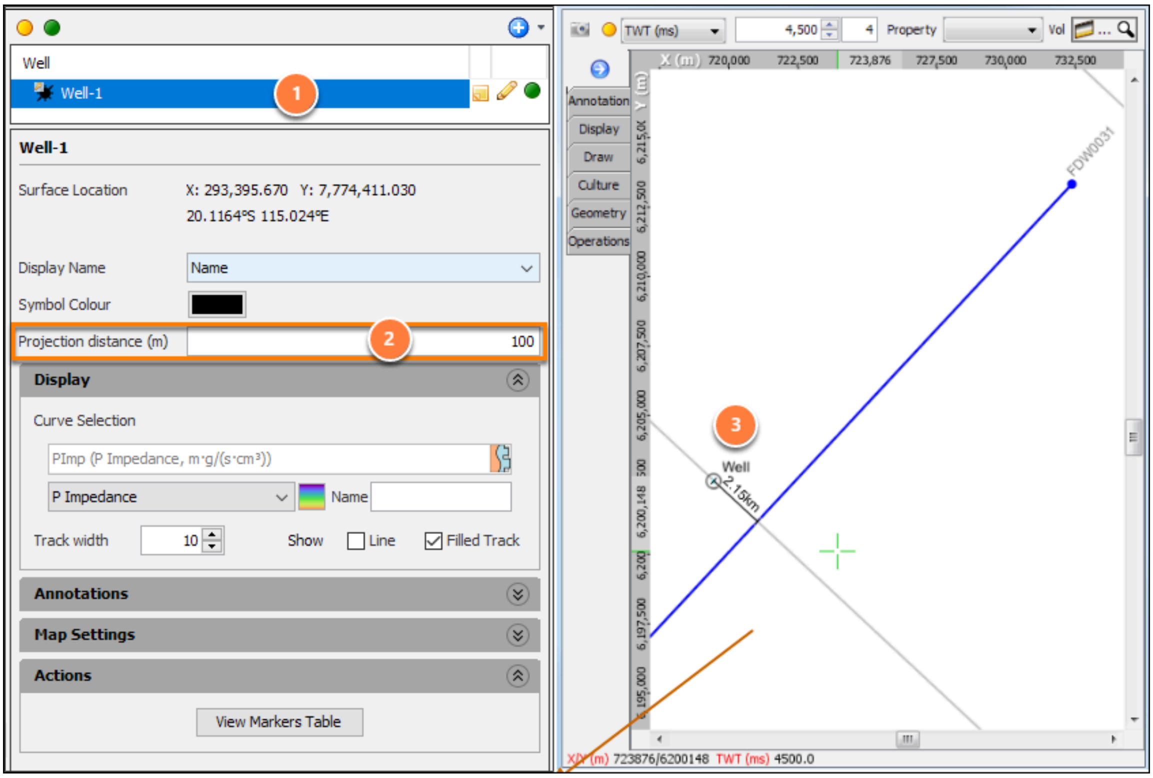
Do you have a well that is near an inline or crossline but is not showing up on the seismic in section views?

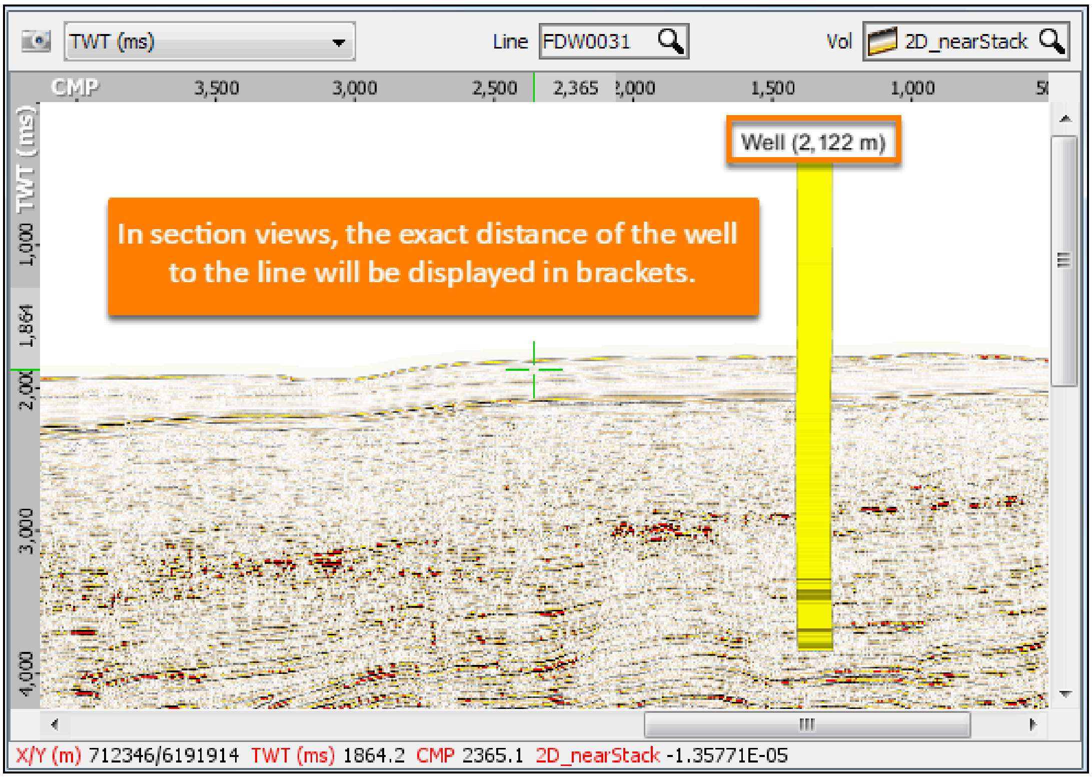
If the well does not go through the line exactly, you may need to increase the well projection distance in order to display it.

Increase well projection distance

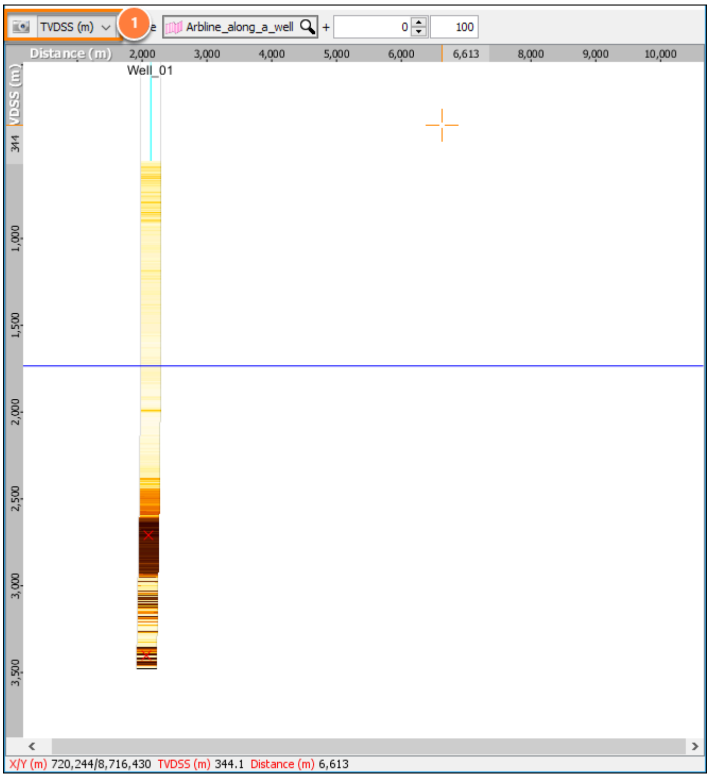
The well Projection Distance controls how far away a well can be displayed from a section view. Choosing larger values includes the well on more distant IL/CL or arb lines. Smaller values will only include the well on very close IL/CL or arb lines.

1. Select the well in the **Well** tab of the **Control Panel** (see Defining Well Settings).
2. Increase the **Projection Distance (m)** until the displayed section is within this distance from the well path.
3. One way to figure out how much distance is required is to right click in **Map View** and select **Add Measurement**. Draw a straight line from the well to the line to calculate its length (see *Measuring in Map View*).



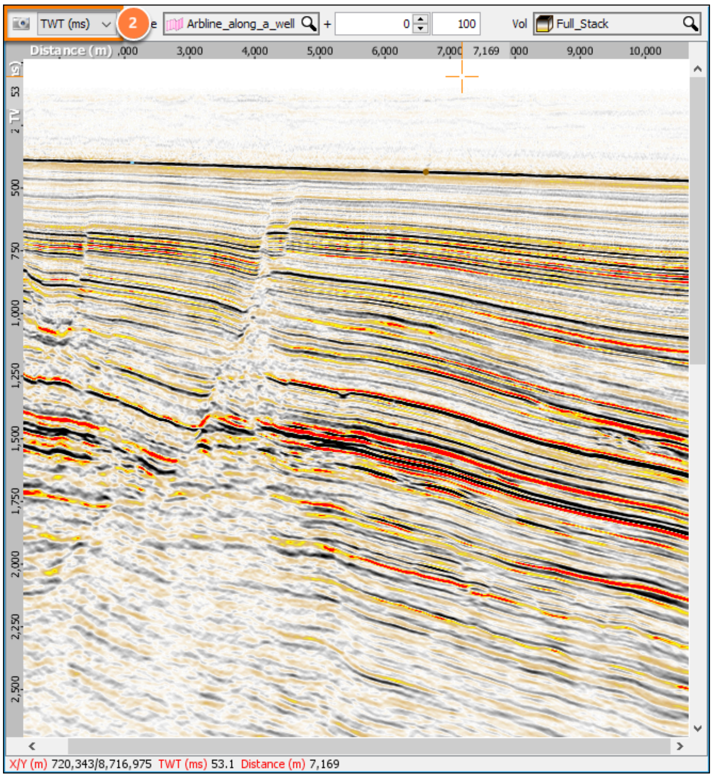


My projection distance is okay, what else can I try?



Are you viewing the wells in TWT or TVD?

1. Try switching the section view to TVD. Wells can almost always be displayed in depth! Displaying in depth requires only a well path, or surface location.



2. To display in TWT, wells require time-depth conversion, provided by the checkshots or time-depth pairs configured for the well. If there are no time-depth pairs, the well will not display in TWT sections.

Insight's Automatic Well Curve Class Assignment

When importing a well, Insight will assign the class of the well curve as follows.

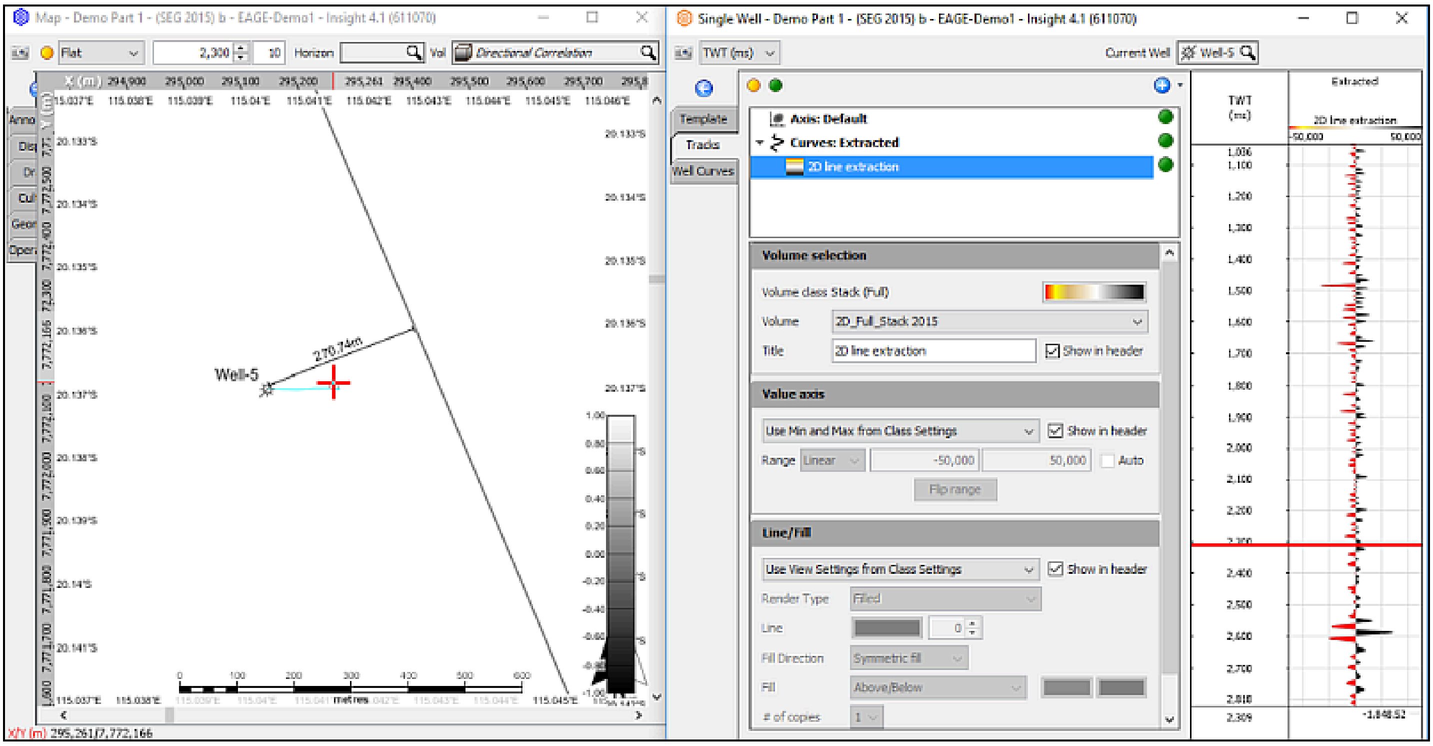
Well Curve Class	Keyword
DENSITY	rho, RHO, RHOB, ZDEN, RHOZ, FDC, DENSITY, DEN, DENC
DEPTH	depth, DEPTH, DEPT, MD
TVD	TVD
S SLOWNESS	DTS, SDT, DTTS, DTSM, DT_SH, DT2, DTS -RC, BATS, ACS
P SLOWNESS	DT, DTCO, DT4P, SLOW, SONIC, DTM, DT4, BTCS, BATC, NCTL_DT, NCTL_VEL, VELMODEL_DT, OSCTL_DT, AC
PVELOCITY	Vp, VP
SVELOCITY	Vs, VS
GAMMA	GR, GAMMA, GMR, GRM, SGR, CGR
RESISTIVITY	resistivity, LLD, LLS, MSFL, LLG, RES, ILD, ILS, RT, RXO, SFL, ML, NCTL_RES, OSCTL_RES, RMIC
RESISTIVITY DEEP	resistivity deep, LLD, LLS, MSFL, LLG, RES, ILD, ILS, RT, RXO, SFL, ML, RDEP
RESISTIVITY MID	resistivity mid, LLD, LLS, MSFL, LLG, RES, ILD, ILS, RT, RXO, SFL, ML, RMED
RESISTIVITY SHALLOW	resistivity shallow, LLD, LLS, MSFL, LLG, RES, ILD, ILS, RT, RXO, SFL, ML

RESISTIVITY XSHALLOW	resistivity xshallow, LLD, LLS, MSFL, LLG, RES, ILD, ILS, RT, RXO, SFL, ML
VOLUME SHALE	Vsh, VSH, VCL
WATER SATURATION	Sw, SW, SXO, XSXO
WATER SATURATION (EFFECTIVE)	Sw, SW, SXO, XSXO
WATER SATURATION (TOTAL)	Sw, SW, SXO, XSXO
WATER SATURATION (INVADED)	Sw, SW, SXO, XSXO
PIMPEDANCE	PI, PIMP, IMP, AI
SIMPEDANCE	SI, SIMP
POROSITY	phi, PHI, POR
POROSITY (EFFECTIVE)	PHIE, PORE
POROSITY (TOTAL)	PHIT, PORT
BULK MODULUS	K
SHEAR MODULUS	mu
TIME	time, TIME
REFLECTIVITY	reflectivity, REFL
ANGLE	angle, ANGLE
SEISMIC	seismic
RATIO	ratio, RUGOSITY
PHOTOELECTRIC EFFECT	PEF, PE
SPONTANEOUS POTENTIAL	SP
CALIPER	CALI, CAL

NEUTRON POROSITY	NPHI, SNP, CNL, NEUT, NEU
DENSITY CORRECTION	DRHO
TEMPERATURE	TEMP
VPONVS	VPONVS, VP_VS, VPVS
MU_RHO	MRHO, MURHO, MU_RHO
LAMBDA_RHO	LRHO, LAMBDARHO, LAMBDA_RHO
PRESSURE	PRES, PP_DT_Eaton, PP_RES_Eaton, PP_VEL_Eaton, PP_P10_DT_Eaton, PP_P10_RES_Eaton, PP_P10_VEL_Eaton, PP_P90_DT_Eaton, PP_P90_RES_Eaton, PP_P90_VEL_Eaton
PRESSURE_GRADIENT	PP, PPG_DT_Eaton, PPG_RES_Eaton, PPG_VEL_Eaton, PPG_P10_DT_Eaton, PPG_P10_RES_Eaton, PPG_P10_VEL_Eaton, PPG_P90_DT_Eaton, PPG_P90_RES_Eaton, PPG_P90_VEL_Eaton
PORE_PRESSURE	PP_Interp
FRACTURE_PRESSURE	FP_MK
OVERBURDEN_PRESSURE	OBP
PORE_PRESSURE_GRADIENT	PPG_Interp
FRACTURE_PRESSURE_GRADIENT	FG_MK
OVERBURDEN_PRESSURE_GRADIENT	OBG

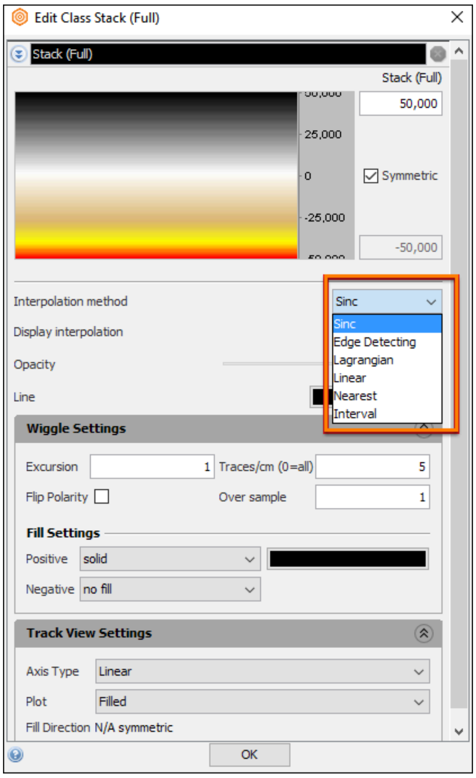
Where does Single Well View extract 2D value from?

In Single Well View, a seismic trace can be extracted along the well path from an input volume. When using a 2D volume, which the well does not directly intersect, Insight will still extract a curve. What is it extracting here?



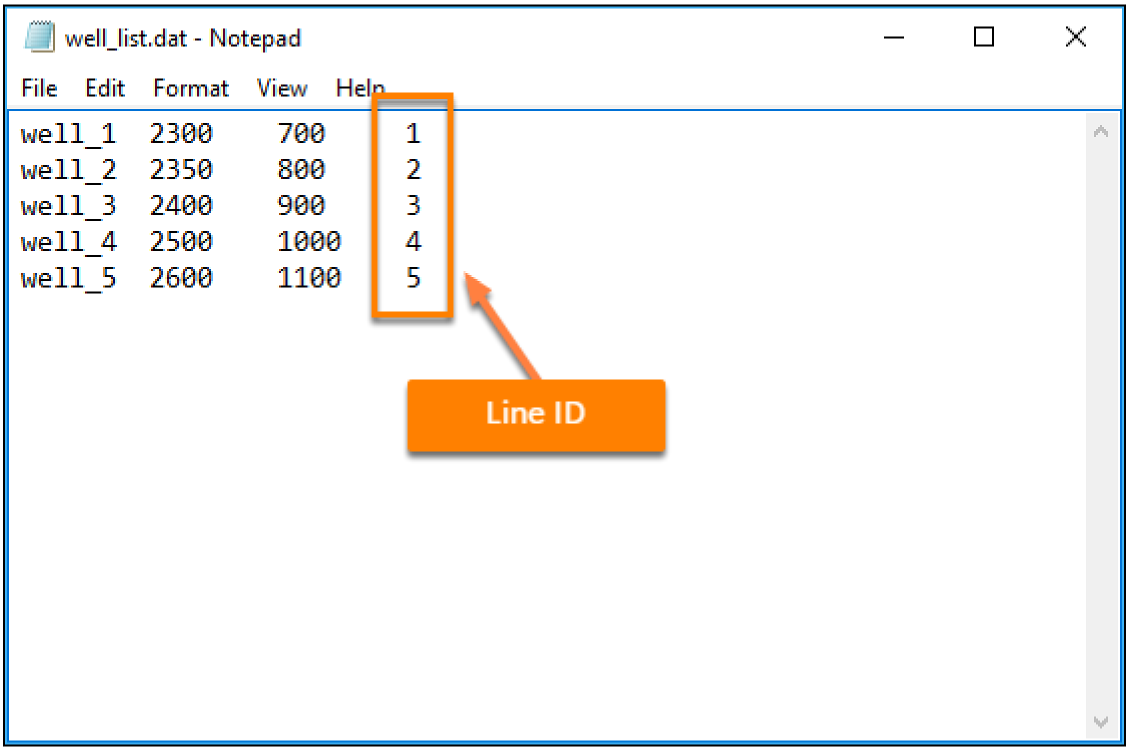
For a 2D volume, it uses the "nearest line" based on the well's surface location. This may be different to the nearest line selected for display in a seismic track, which is based on the weighted average distance from the full well path. The well path is projected onto the line using flux projection using the same approach as displaying the well on a 2D or arbline section.

Four-point lagrangian interpolation is used for horizontal interpolation, while vertical interpolation is determined by the volume's class configuration, see picture below.

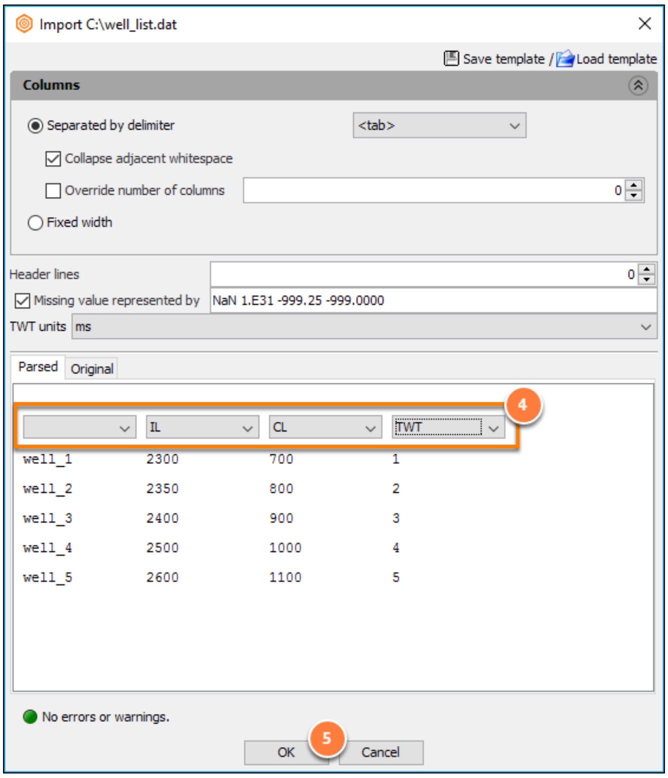


The curve is produced in MDKB by sampling at one metre intervals along the well path, defined by points in X/Y/(TWT or TVD depending on the volume's domain). The well path is interpolated linearly between those points.

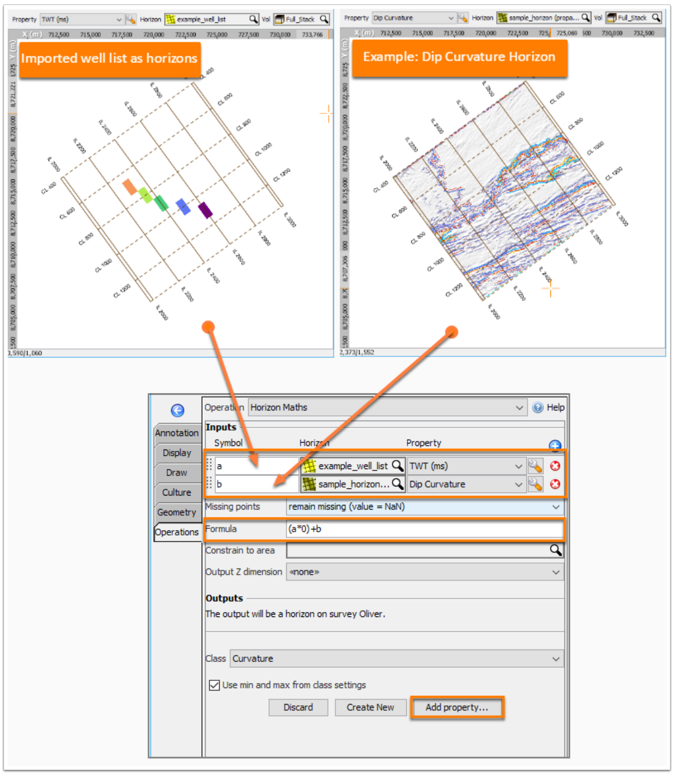
How to export map values at specific IL/CL locations?



1. Open a text editor and create a column of line ID besides the well inline/xline locations.
2. Save this file as a .dat file.
3. Open Insight and import them as horizons (see *Using the Horizon Text Importer*).



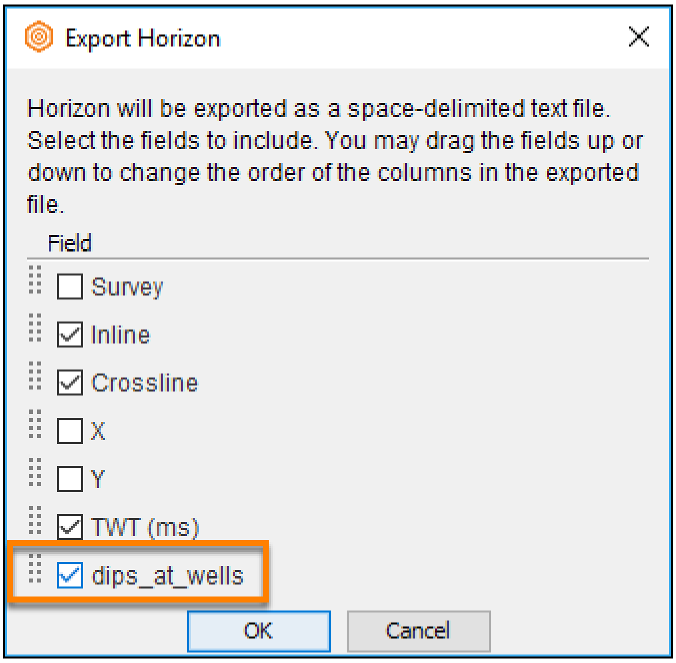
4. Set the IL, CL and TWT values in their respective columns.
5. Click **OK** to continue importing.
6. Go to **Control Panel > Map View**.



7. Go to the Operations tab and select Horizon Maths operation.
8. Click the blue "+" button to add two symbols:

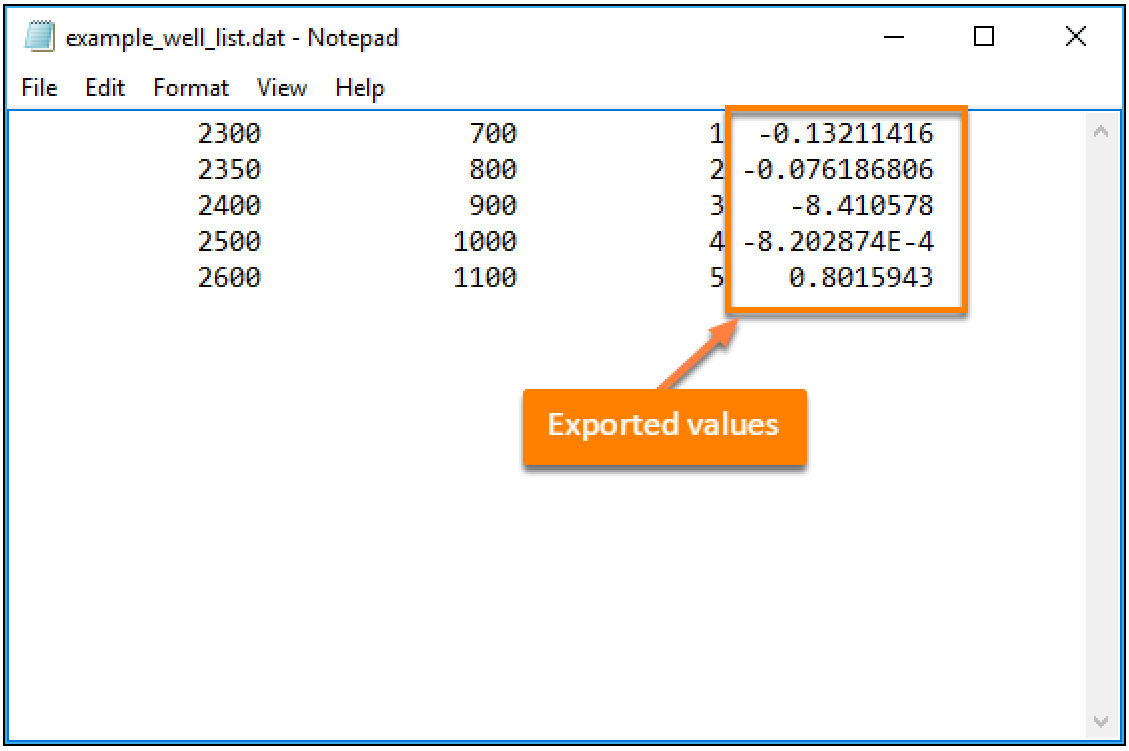
- a: imported well list
- b: horizon (e.g. dip curvature horizon)

9. In the formula field, use: **(a * 0) + b**.
10. Click **Run**.
11. Click the **Add property** button and type a property name.
12. Click **OK**.



13. Right click on the imported horizon and select Export "*horizon_name*" (see [Exporting Horizon]).
14. Select **Export customised text file** and click **OK**.
15. Click the checkbox of the newly added properties from the previous horizon maths result and click **OK**.
16. Type the name of the file and click **Save**.

The saved file will contain the property values of the wells.


example_well_list.dat - Notepad
File Edit Format View Help
2300 700 1 -0.13211416
2350 800 2 -0.076186806
2400 900 3 -8.410578
2500 1000 4 -8.202874E-4
2600 1100 5 0.8015943
Exported values

Calculating Well Marker-Horizon Mistie

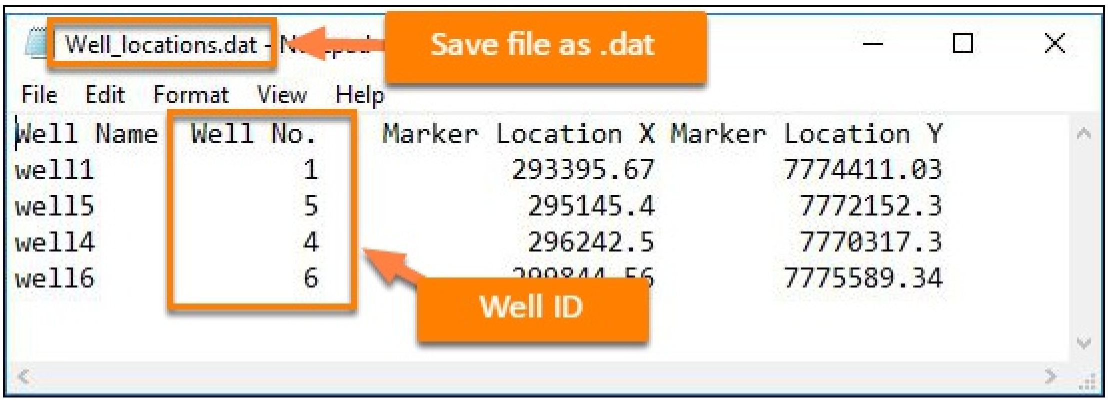
Use this workflow to determine the well marker and horizon intersection depths at a specified well marker location and use these to either generate a table of residuals, or calculate a residual map.

- *Calculate the Residual Mistie*
- *Result 1: Export Residual Mistie*
- *Result 2: Map Residual Mistie*

Build a map of marker locations

The workflow requires a table of XY locations for each marker location. The table will be used create a well marker location horizon and to sample the structural horizon data at that location for comparison. If the wells are vertical, all the marker locations will be the same and so only one generic table is required.

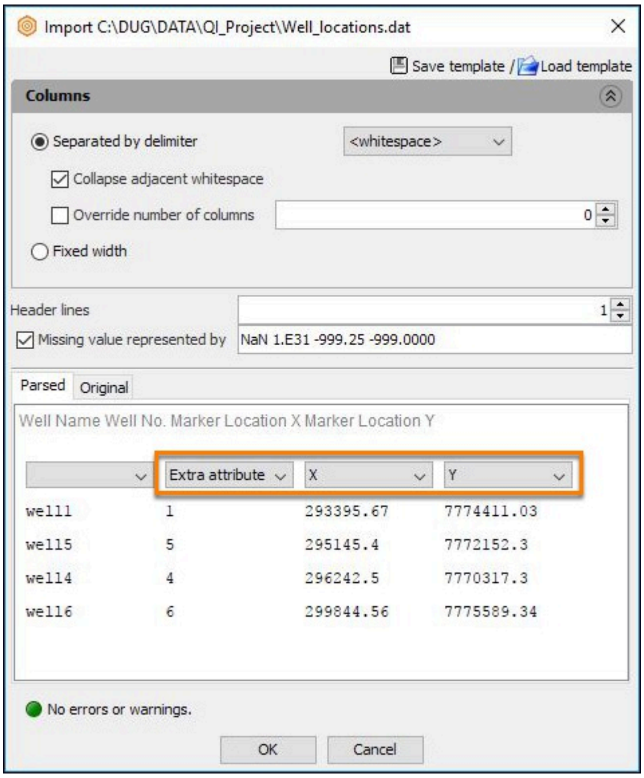
In additional to a well name, the table should also include a reference number for each row, it must be numeric, and should be a well number or code. Exported results will refer to this ID.



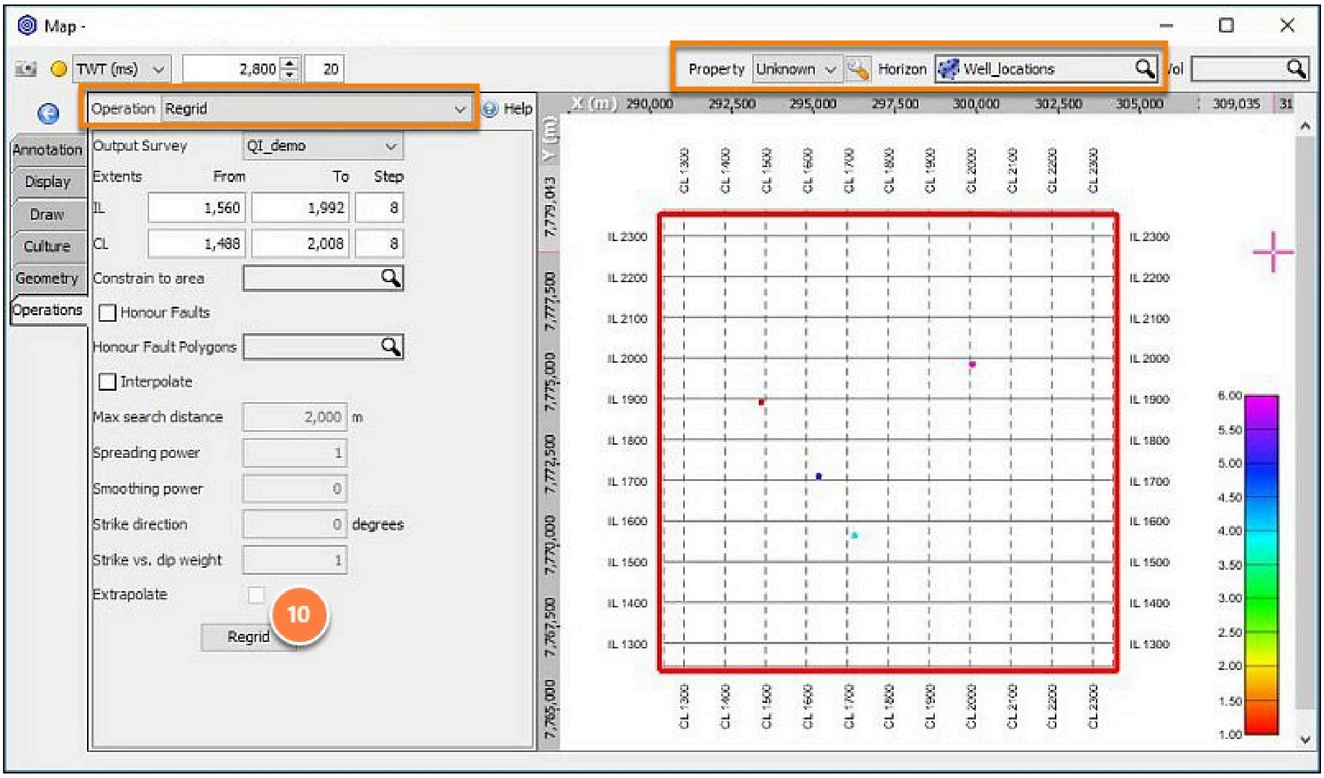
1. Open a text editor and create a column of well ID besides the well marker X/Y locations.

Note: For wells without a marker, either exclude them from the table or set the Well No. to a missing value such as -999.99.

2. Save this file as a .dat file.
3. Open Insight and import them as horizons (see *Using the horizon text importer*).



4. In the **Import** window, set the X,Y values in their respective columns.
5. Set the well ID as “Extra Attribute”
6. Click **OK** to continue importing.
7. Go to **Control Panel > Map View**.



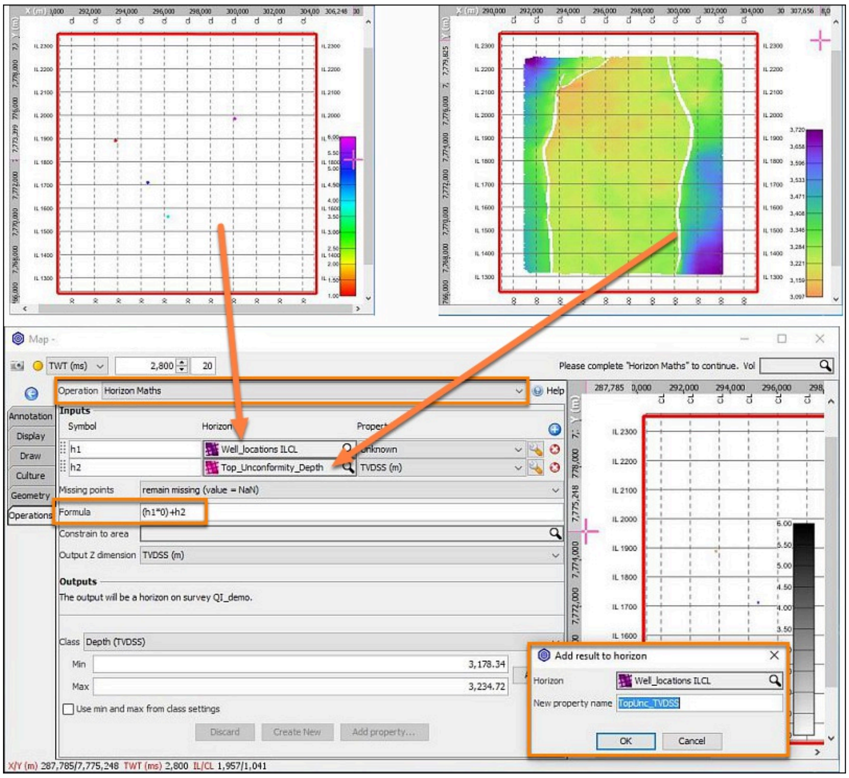
8. Display the imported horizon in the map view

9. Click the **Operations** tab and select the **Regrid** Operation to regrid the imported XY well locations to the 3D Survey (ILCL) (see *Regrid Horizon*)
10. Click **Regrid**.
11. Save as a new horizon (ILCL).

Extract the horizon depths at the markers

Use the marker location map to filter the horizon depths, i.e. keep only the depth values at these locations.

Use the marker location map to sample and store the structural horizon at the well marker locations.

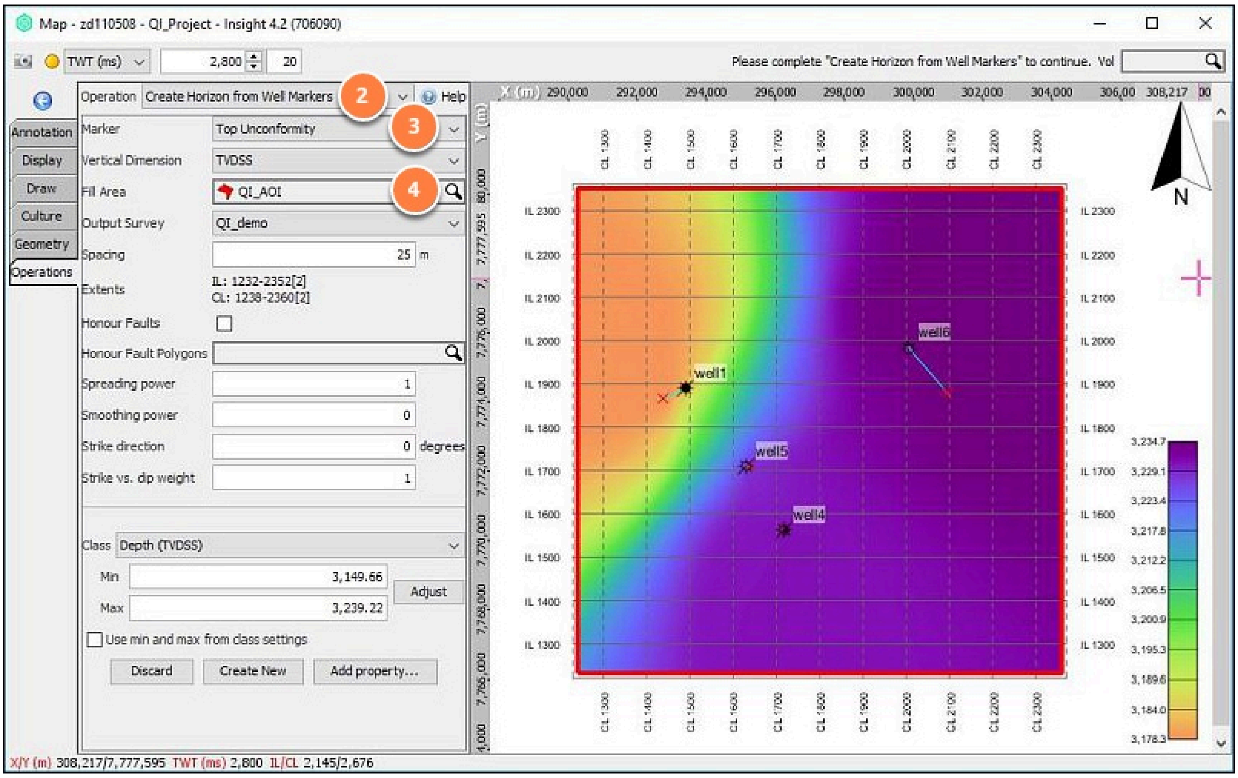


1. Go to the **Operations** tab and select **Horizon Maths** operation.
2. Click the blue "**+**" button and add two symbols:
   - **h1**: Imported well list
   - **h2**: Depth converted horizon
3. In the formula field, use: **(h1*0)+h2**.
   - **Note:** This formula discards the "Well ID" value by multiplying by zero. Because "h1" only has values at marker locations, the "missing points remain missing". Only the structural horizon points coincident with the Wellid locations will be included.
4. Click **Run**.
5. Click the **Add property** button, select the well marker location horizon and type a property name to identify the structural horizon as a source. (e.g. Horizonname_TVDSS)
6. Click **OK**.

Create a Current Marker Value Horizon

> **Note:** The initial table (Step 1) can optionally include a column of marker Z values to avoid this calculation. To be of use in the workflow, the table values would need to be stored as TVD. If the marker depth of interest in the well was then adjusted the table would need to be updated accordingly and reloaded.

By calculating a marker map, using the steps below, these factors are automatically taken care of, the latest marker is always used with the correct TVD.

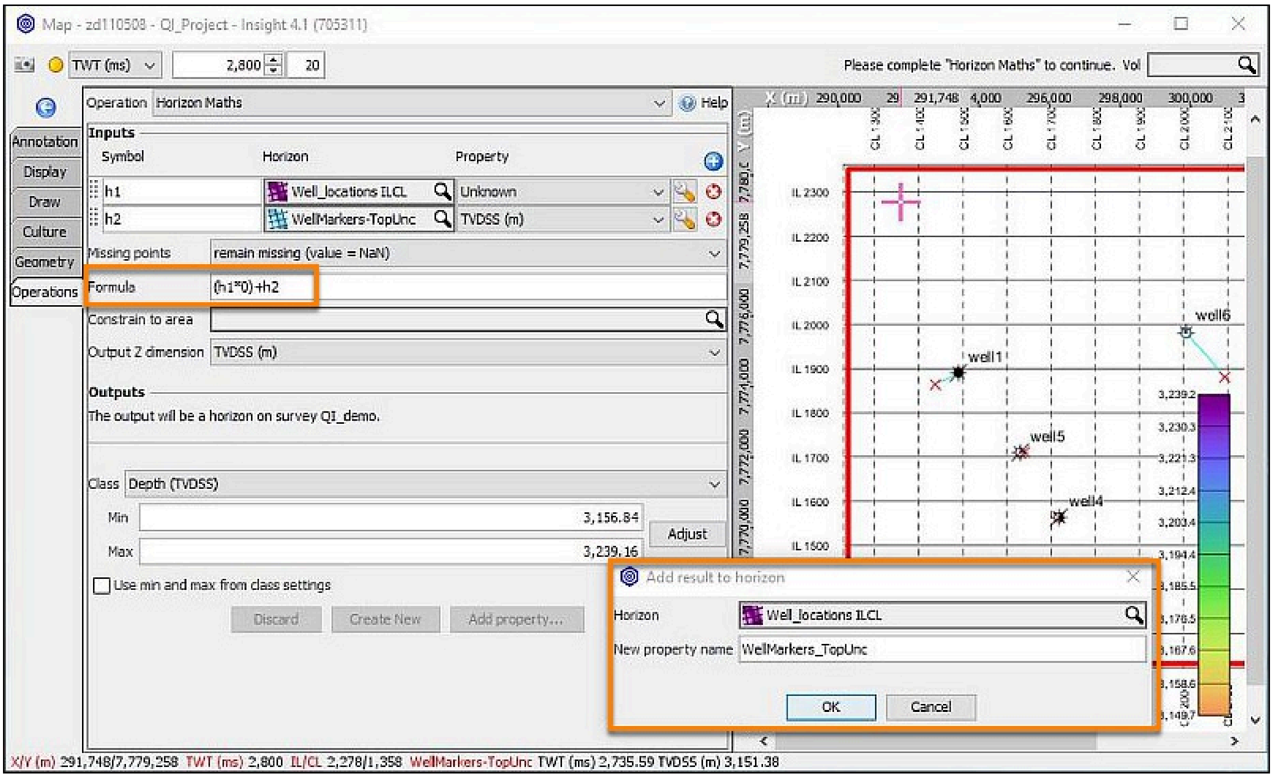


1. Enable the wells in the **Control Panel > Wells** tab.
2. Click the **Operations** tab and select **Create Horizon from Well Markers** operation.
3. Select the **Marker** name.
4. Select a polygon that includes the wells of interest.
5. Click **Run**.
6. Click the **Create New** button, rename the horizon to reflect the markers as the source. (e.g. WellMarkers_markername)

Extract the Marker depths at the Marker Locations

This samples the current marker horizon at the defined locations for comparison with the structural horizon.

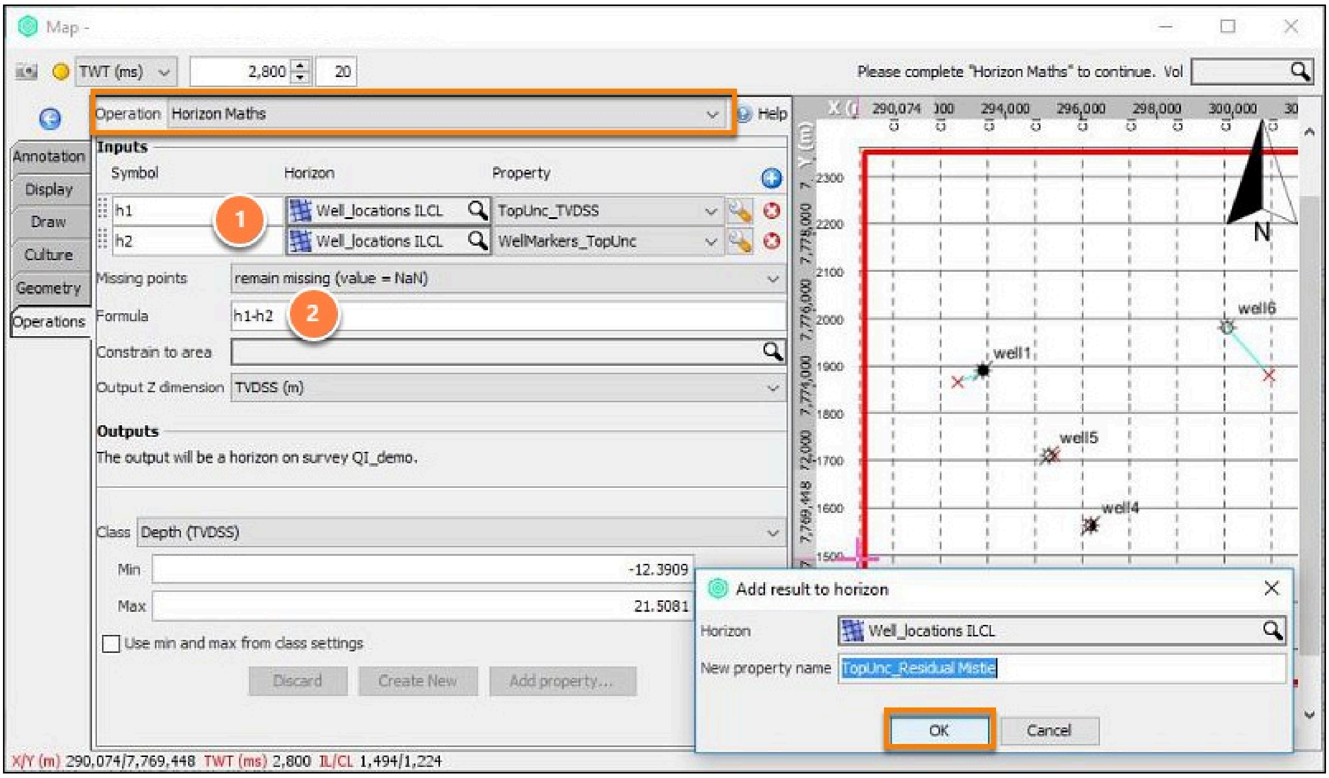
Note: This calculation can be skipped if the table in Step 1 includes a valid TVD marker depth.



1. Go to the **Operations** tab and select **Horizon Maths** operation.
2. Click the blue "**+**" button to add two symbols:
   - **h1**: Imported well list
   - **h2**: WellMarkers horizon
3. In the formula field, use: **(h1 * 0)+h2**
4. Click **Run**.
5. Click the **Add property** button, select the well marker location horizon and type a property name to identify the markers as a source. (e.g. WellMarkers_markername)
6. Click **OK**.

Calculate the Residual Mistie

This is the main (and final) calculation that determines the difference between the well marker TVD and the structural horizon at the same location.

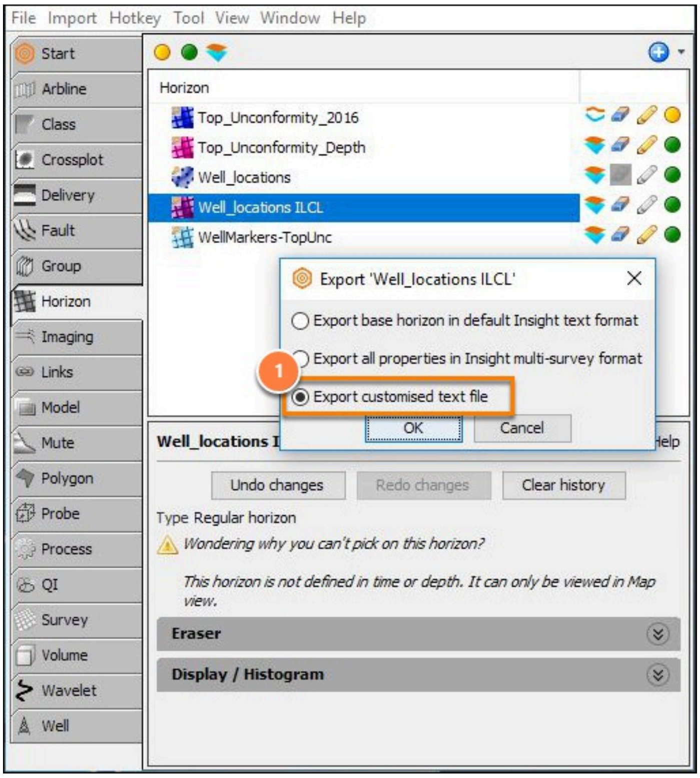


1. Go to the **Operations** tab and select **Horizon Maths** operation.
2. Click the blue "**+**" button and add two symbols:
   - **h1**: Depth horizon sampled at well list
   - **h2**: WellMarkers horizon sampled at well list
3. In the formula field, use: **(h1 - h2)**
4. Click **Run**.
5. Click the **Add property** button, select the Well marker location horizon and type a property name to identify the residual . (e.g. Markername_ResidualMistie)
6. Click **OK**.

Depending on your ultimate objective, you can now either look at the residual differences as a table, or turn the differences into a horizon for surface correction

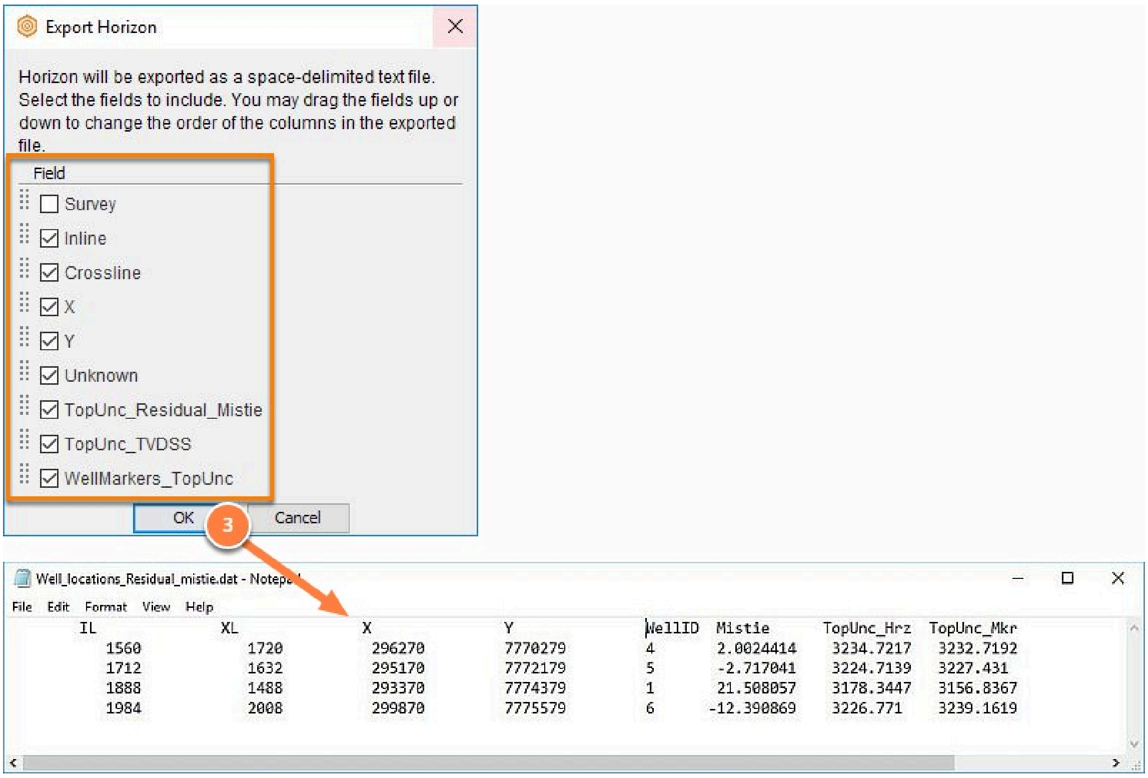
Results 1: Export Residual Mistie

This optional section explains how to view the results as a table of values.



1. Export the horizon as a customise text file from the control panel (see *export customised text file*).
2. Select all the attributes of interest.
3. Click **OK** to export these attributes.

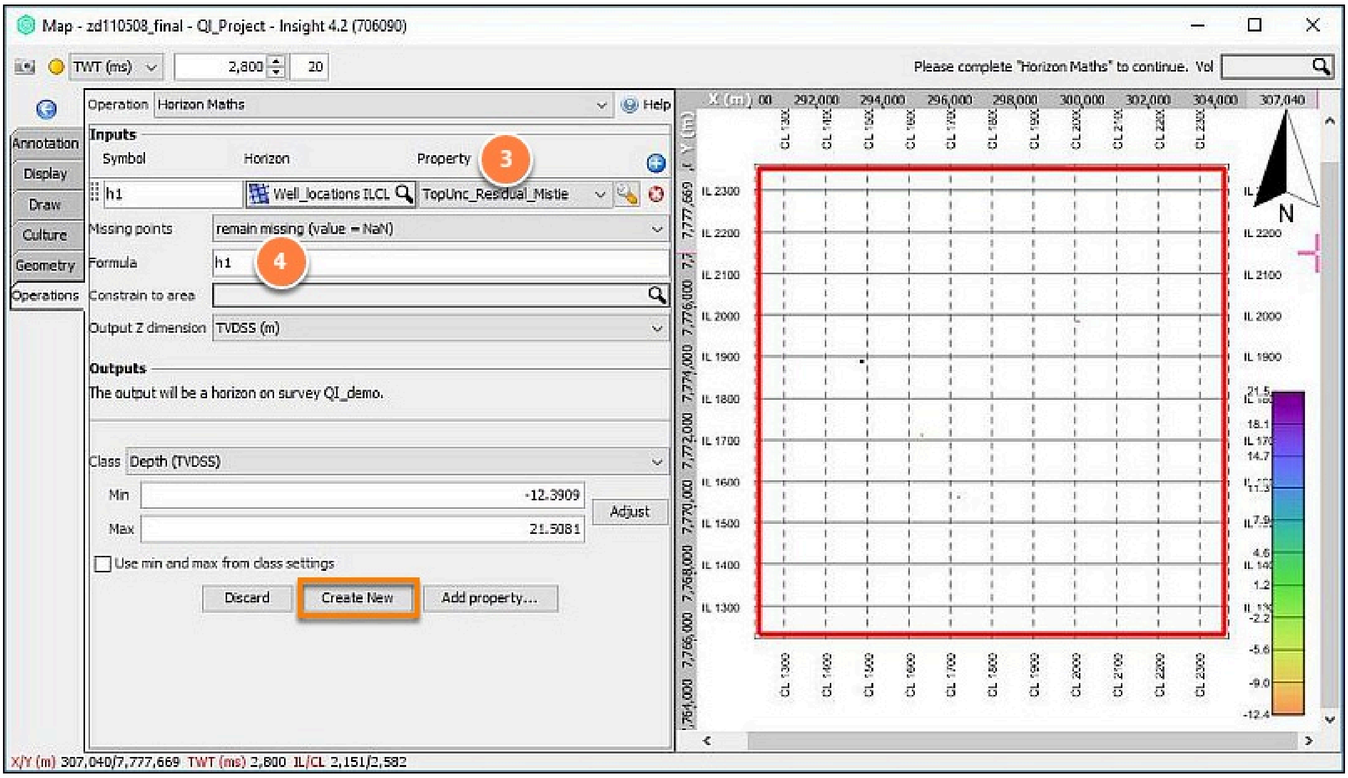
Note: The well name was dropped during the regrid operations, which is why the Well ID (number) column was introduced in order to identify the final results.



Results 2: Map Residual Mistie

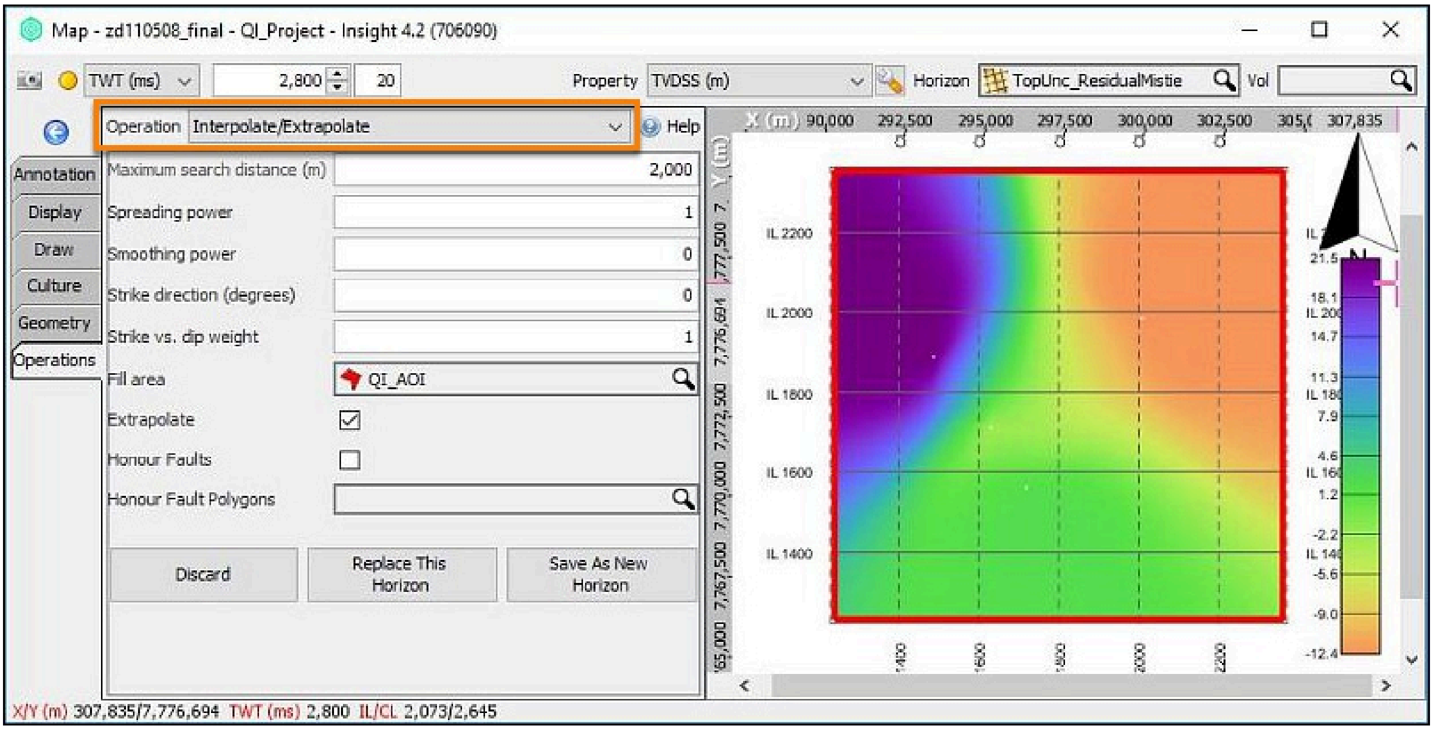
This optional section explains how to create a residual horizon for surface correction.

Horizon regrid and interpolate / extrapolate operate on the primary stored value of a horizon, not the additional (residual) attributes that have just been calculated. Create a separate horizon distinct from the custom attribute horizon to ensure that the correct attribute is being used in the interpolation process.



Use Horizon Math:

1. Go to **Map View > Operations > Horizon Math**.
2. Click the blue (+) and add your well marker location horizon with calculated residual as an attribute.
3. Choose the Property to use (e.g. Markername_Residual Mistie).
4. Use formula: h1 (or whatever symbol you've chosen).
5. Check the Class is set to Depth (TVDSS)
6. Click Run.
7. Click Create New to save as a new horizon. Provide a suitable name for easy identification (e.g. Markername_Residual Mistie).



Interpolate the residual data

1. Select the new TVDSS horizon of residual mistie values.
2. Select Operations > Interpolate/Extrapolate.
3. Click Interpolate.
4. Save as new horizon.

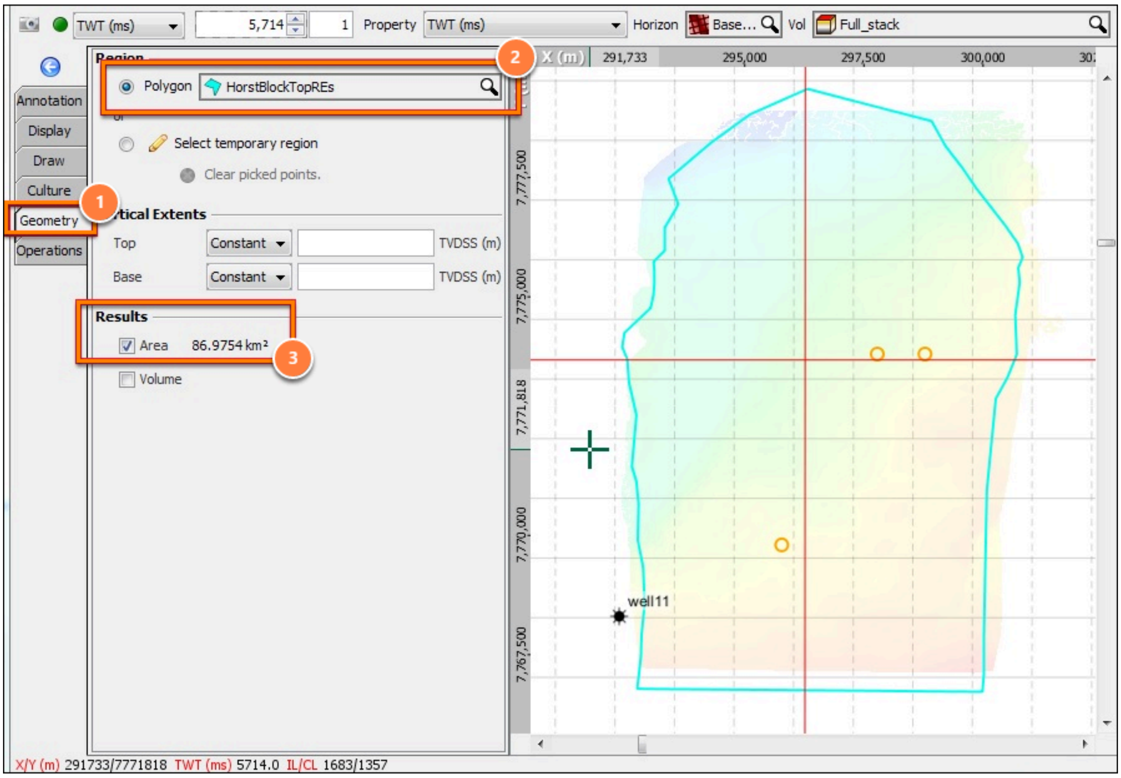
Note: This horizon can then be added to the original structural horizon to create an accurate well tie.

Polygons

How do I find out the area and volume of a polygon?

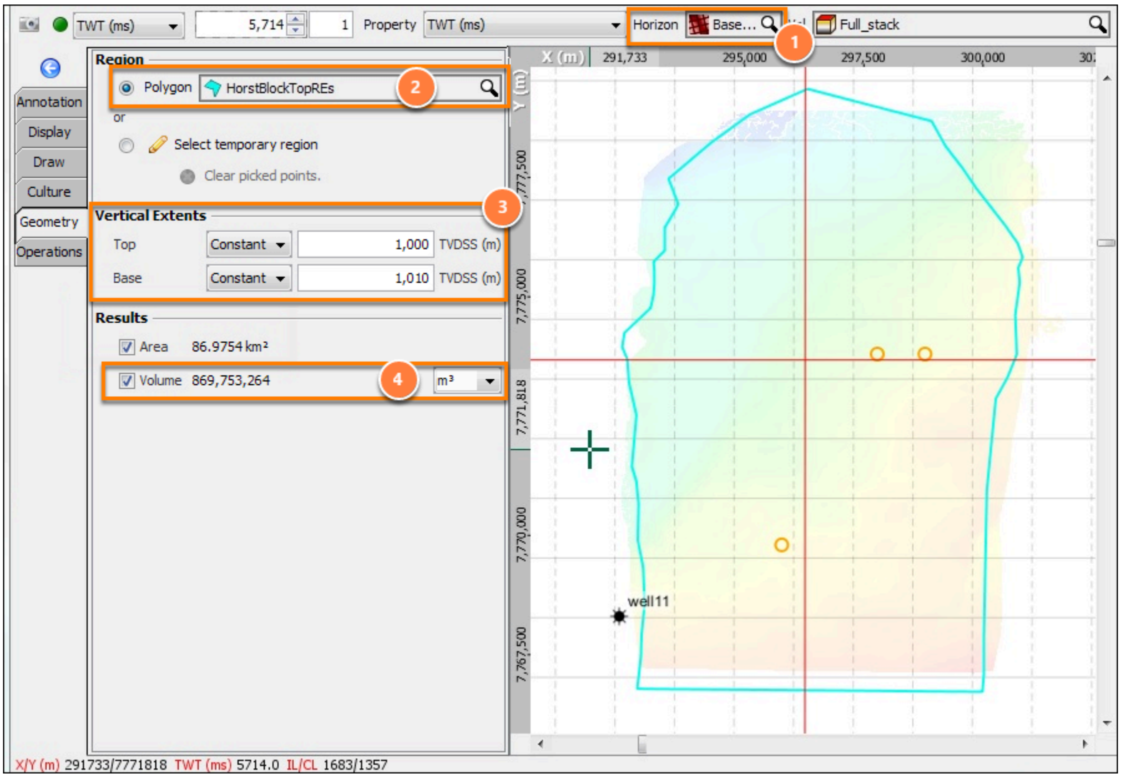
You can obtain the area and volume of a created polygon from the Map View (*see* *[Measuring in the Map View](#)*).

Area of a polygon



1. In the Map View, open the **Geometry** tab.
2. Under **Region**, select the polygon you wish to use.
3. In the Results section, select the Area check box to calculate the area in the polygon.

Volume in a polygon

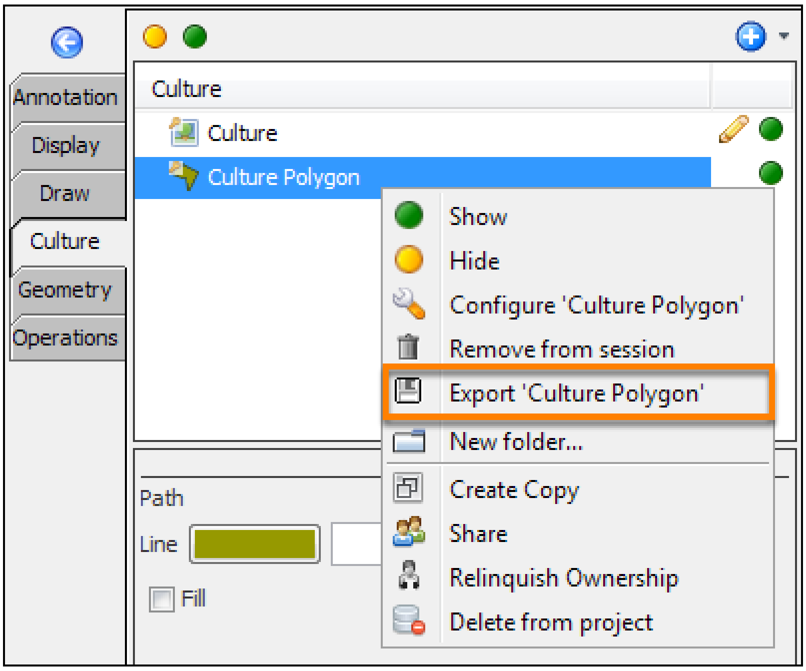


1. To find the volume within a polygon, select a **Horizon** in the navigation bar of the Map View.
2. Open the **Geometry** tab and select the polygon you wish to use.
3. In the Vertical Extents section, define the Top and Base boundaries using a constant or a horizon, or both.
4. Select the **Volume** check box in the Results section to calculate the volume.

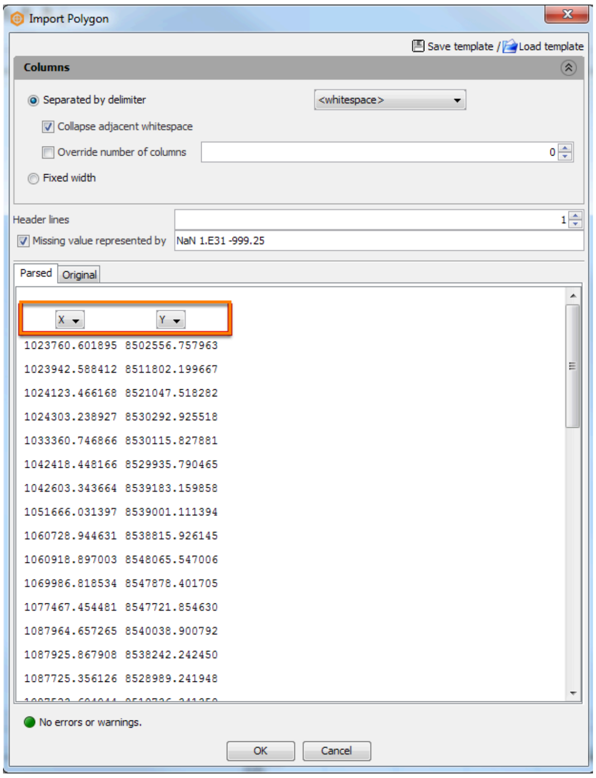
How do I convert culture polygons into regular polygons?

These instructions are for importing a single polygon into Insight. In order to import multiple polygons, see *How do I convert a .duculture file containing many shapes to a .dupoly file*.

Note: Simply right-click on a culture product and select **Export shapes to new map polygon** (see *Working with Culture*).



1. From the **Map View> Culture** tab, right-click on the culture file and select **Export 'Culture'**.
2. Export the file as an ASCII format *.duculture* file.
3. In the Insight Control Panel, expand the **Import** menu and select **Polygon**.
4. At **Files of type**, select **All Files**.
5. Select the exported *.duculture* file and click **Open**.

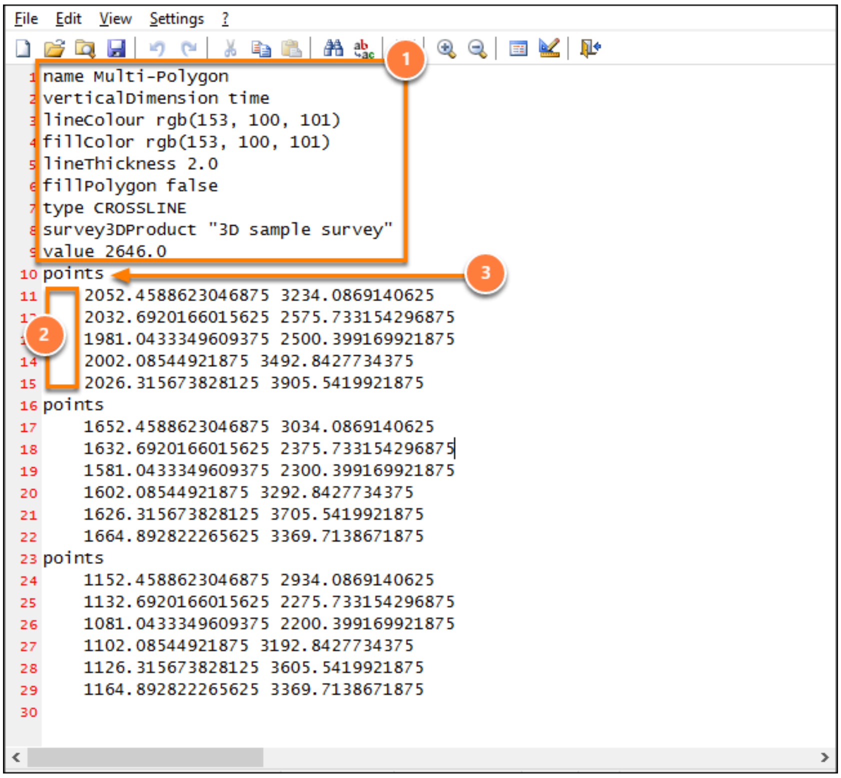


6. In the **Import Polygon** window, define the X and Y columns.
7. If there are no errors, click **OK**.

You should now have your culture as editable polygons.

How do I convert a .duculture file containing many shapes to a .dupoly file?

To convert a culture polygon into regular polygon, see *[How do I convert culture polygons into regular polygons](#)*.



1. Insert Header information at the top of the file e.g. copy and paste from another file.
2. Insert a space or indentation before every row of X/Y coordinates.
3. Insert the word "points" between every set of X/Y coordinates, dividing each shape.
4. Remove all other blank rows in the file.
5. Save this file as .dupoly.

Crossplotting and Spectra

What is the best way to crossplot the points in a series of channel structures?

The primary method of selecting points for crossplotting is by using polygons. This approach is favoured because of its speed and efficiency.

1. To best utilise this approach with channel structures, draw an arbline down the center (or most significant section) of your channel (see *Creating an Arbline*).
2. In **Arbline View**, select the new line and draw a polygon highlighting the channel structure (see *Picking Polygon in Arbline View*).
3. Create additional arblines and polygons for each of your channel structures.
   Note: The crossplots will permit overlapping polygons, allowing you to have polygons such as "Channel 1", "Channel 1 High Porosity", etc.
4. In the crossplot window, select and deselect the polygons to highlight the points inside each channel.

Crossplot points in specific channel

To crossplot all the points in a specific channel, you will need to use a different approach. By using a **Volume Sculpting** process on the input volumes, you can constrain the points available for the crossplot (see *Volume Sculpting*).

1. Draw a polygon that closely follows the edge of the channel structure in Map View.
2. Create a **Volume Sculpting** process: "P-Impedance - Channel 1"
   - Volume: P-Impedance
   - Top horizon: Channel top event
   - Base horizon: Channel base event (Or channel top + estimated channel thickness)
   - Constrain to Area: Channel edge polygon
3. Use the result of this process as one of the inputs to the crossplot.

Multiple channels

For the more complex, multiple channel case, repeat the procedure for each channel, resulting in one impedance volume for each channel, e.g. "P-Imp Channel 1", "P-Imp Channel 2". Then try the following:

1. Configure the crossplot to use the original (unsculpted) P-Impedance and Vp/Vs.
2. Select a probe that includes all the channels.
3. Select the crossplot **Plot Type** as **Point by Point**.
4. For the **Colour Data Set**, select the output from a volume maths process defined as:
   - **Process:** Volume Maths, "Channel Identifier"

- **Symbol:** "C1" Volume: "P-Imp Channel 1"
- **Symbol:** "C2" Volume: "P-Imp Channel 2"
- **Formula:** if(!isNaN(C1),1,if(!isNaN(C2),2,0))

This example can be extended to any number of channels. The performance will be slower than the polygon/section approaches described earlier.

For more information, see the following pages from the online manual:

- *Creating a Crossplot*
- *Highlighting Volume Points in Crossplot*.

If you are having problems or would like to ask us a question, do not hesitate to contact our Support team at support@dugeo.com.

Processes

Stacking process across NaNs

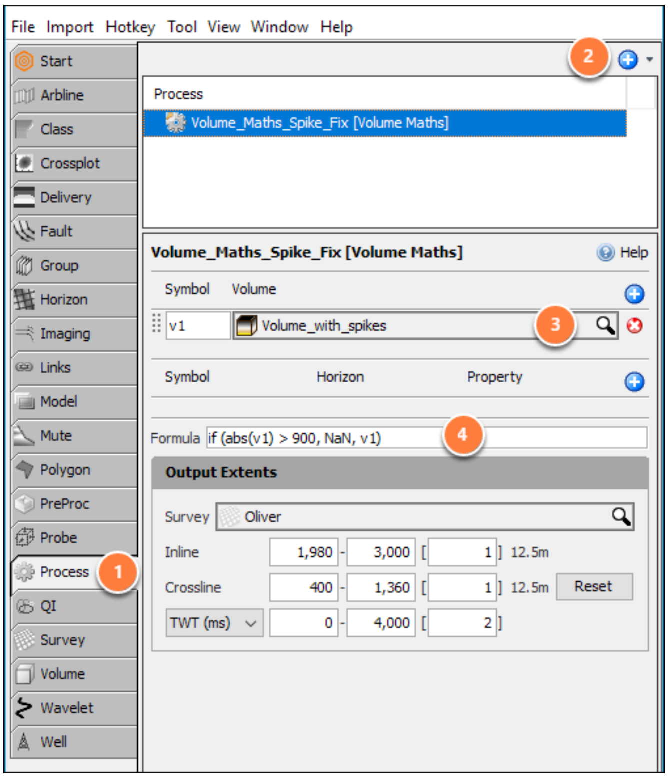
Regardless of whether you use a mute, stacking should exclude both NaNs and zeros.

As a general rule, NaNs are always excluded. Zeros are excluded if they are at the far offset edge of the data (i.e. there are no non-zero, non-NaN samples between this one and the furthest offset).

Note: The steps on how to perform the stacking process can be obtained from the *Stacking* section.

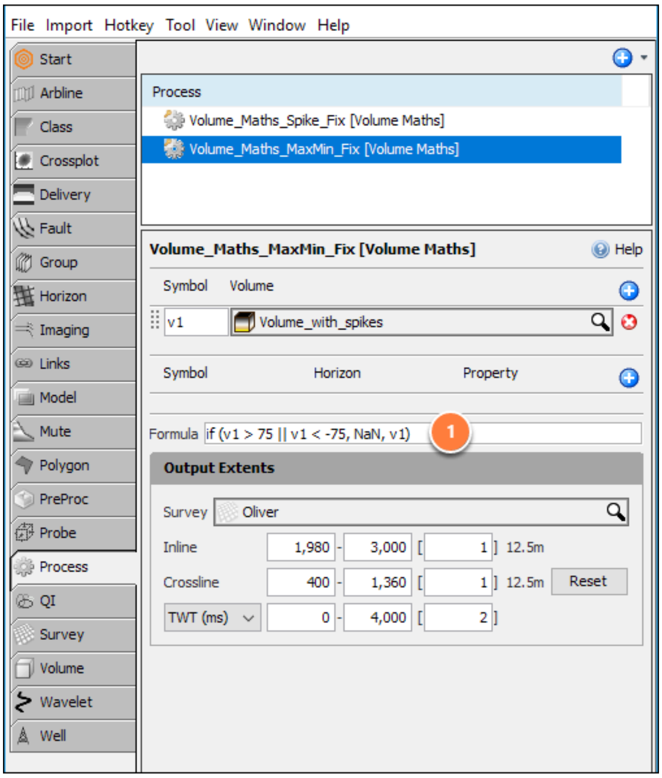
How do I filter large amplitudes, extreme values and spikes from my data?

Volume Maths is a very usefull process in Insight that allows manipulation of seismic data (to a certain degree). In this instance, we will use Volume Maths to identify large amplitudes, extreme values and/or spikes and replaced it with NaN values. We can then interpolate these NaN values using the NaN Removal process to repair the volume.



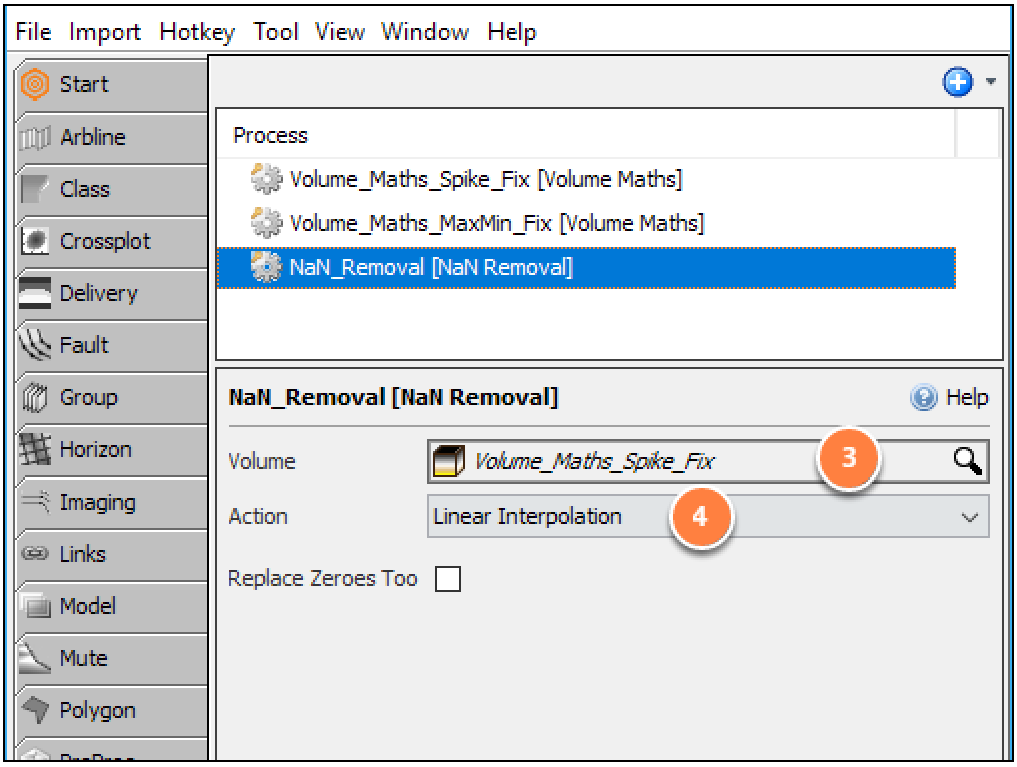
For this situation, we use the logic that **if values in volume v1 are greater than 900 (or a more suitable extreme number for the volume) replace it with NaN, if not use v1 (input values)**.

1. Open the **Process tab** and click the **"+"** button to add a **New Process**. Search and select **Volume Maths**.
2. Type in a name for this **Volume Maths** process.
3. In the **Volume Maths** panel, click the **"+"** button and select the abnormal volume.
4. In the **Formula text box**, enter the formula: **if (abs(v1) > 900, NaN, v1)**.



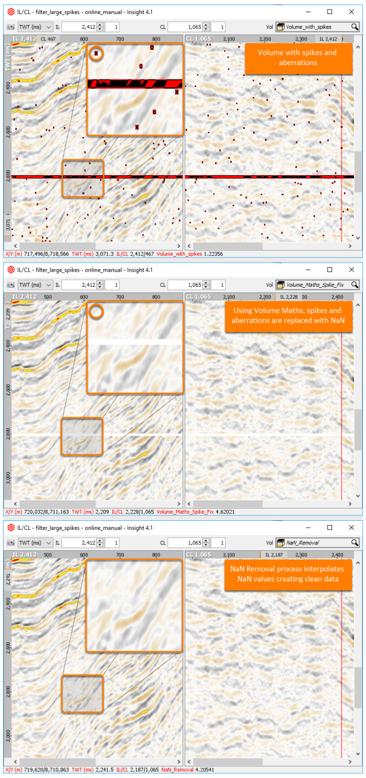
To crop on specific high min and max values, use the logic that **if input volume v1 is greater than 75 (or a suitable max value) OR if v1 is less than -75 (or a suitable min value), replace with it NaN, othewise use v1 (original value)**.

1. Add formula **if (v1 > 75 || v1 < -75, NaN, v1)** in the **Formula** text box of the **Volume Maths** panel.



Interpolate NaN values using the NaN Removal process.

1. In the **Process** tab, click the **"+"** button and add a new process. Search and select **NaN Removal**.
2. Type in a name for this NaN Removal process.
3. Under the **NaN Removal** panel, select the volume created from the above **Volume Maths** process in the **Volume** text box.
4. Select **Linear Interpolation** from the **Action** drop box. Read more about *NaN Removal process*.
5. A volume is created and is available in the Volume tab.



When should I choose a Dip filter over an F-K filter?

Both processes remove linear noise but there are some circumstances in which one works better than the other:

- F-K can filter on certain frequencies. This helps when noise occurs within a specific frequency range.
- F-K tends to work better when spatial resolution is poor.
- Dip filter works better if there is aliasing in the F-K domain.

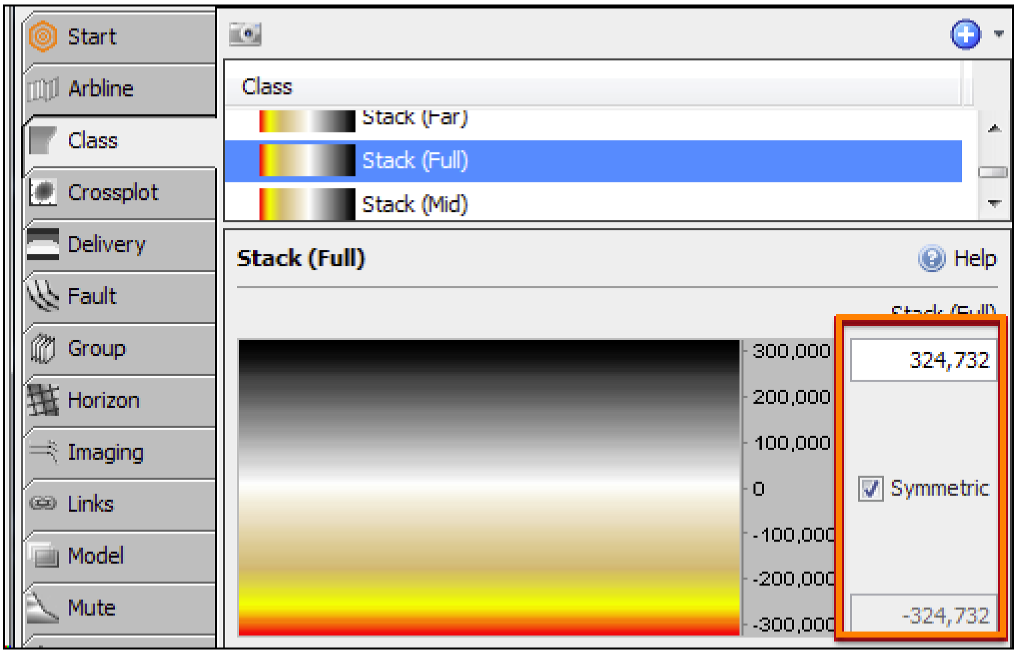
Read more aboout *F-K Transform* and *Dip Filter*.

Synthetics

How do I scale synthetic trace relative to seismic?

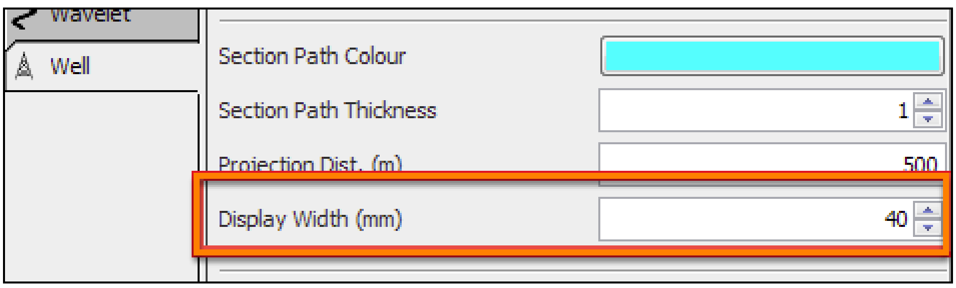
In addition to the wavelet scalar, the display of the synthetic trace in section view is determined by two main factors:

1 - The min/max values of the Class colourbar



Note: The synthetic trace automatically inherits the same Class as the volume you are comparing it to.

2 - The display width of the well track

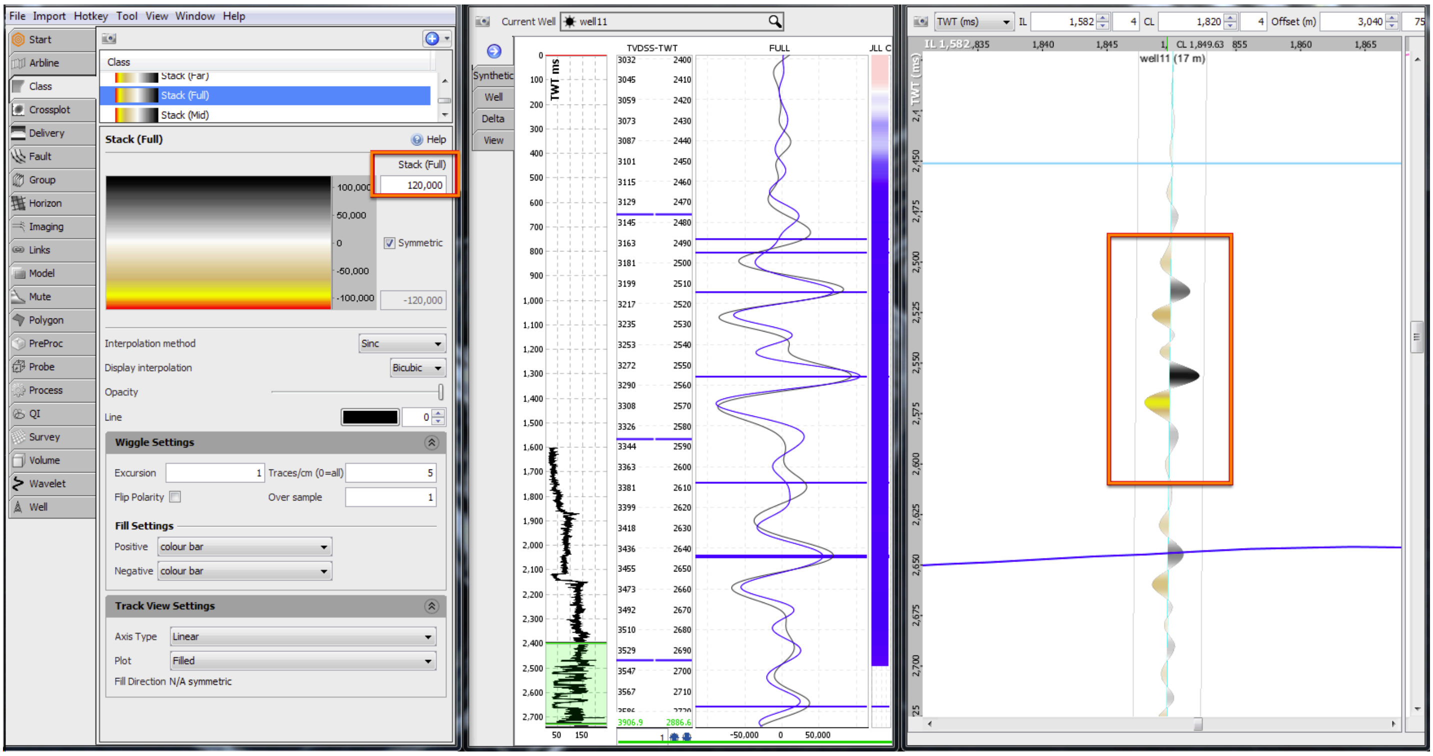


Note: The edges of the well track represent the min/max values of the colourbar (i.e. if your synthetic trace has amplitudes -100 to +100 and you also set the min/max colourbar values as -100 to +100, then the maximum peaks and troughs should just hit the edges of the well track).

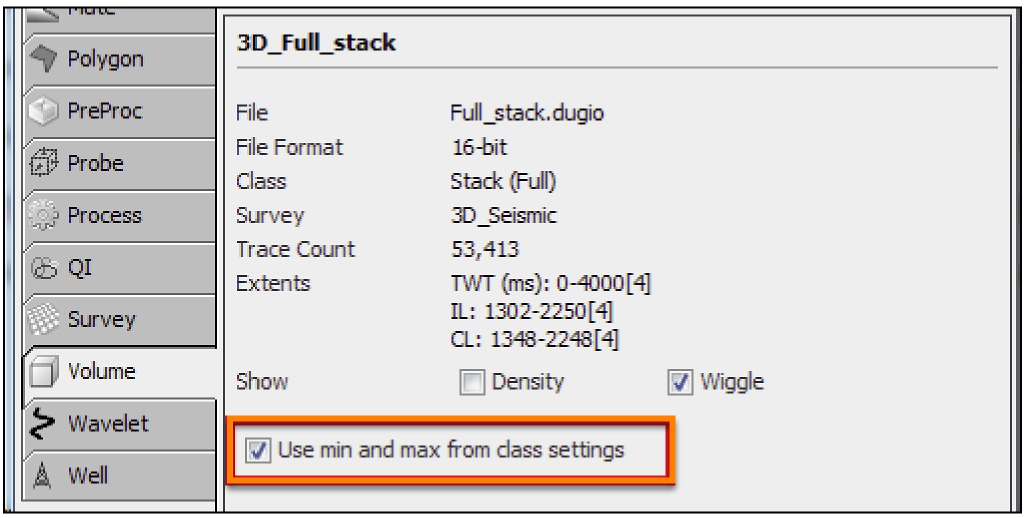
Workflow

Please follow these four steps for scaling of your synthetic trace:

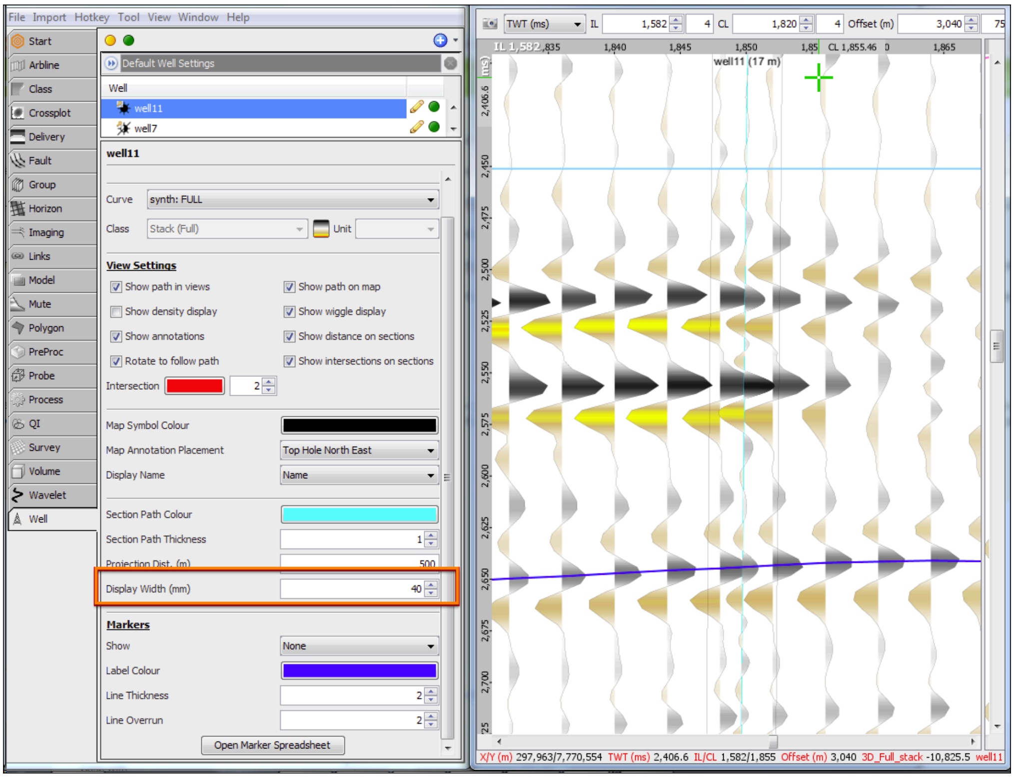
1. Set the wavelet scalar such that it matches the seismic trace in the synthetic view.
2. Set the Class min/max values such that the peaks/troughs of the synthetic trace are just inside the edges of the well track in section view:



3. Set the volume to **Use min and max from class setting**:



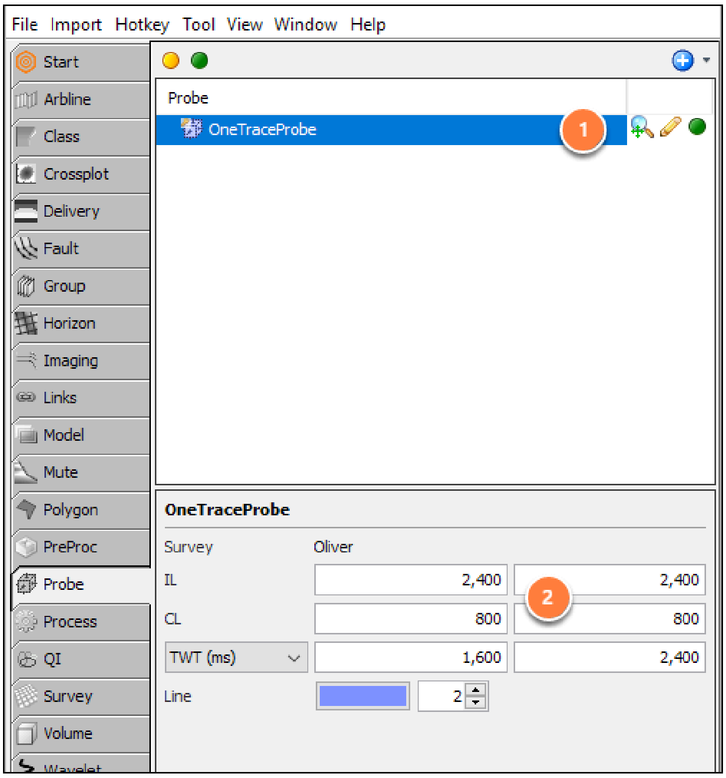
4. You may now wish to adjust the **Display Width** of the well track depending on the zoom level in the section view:



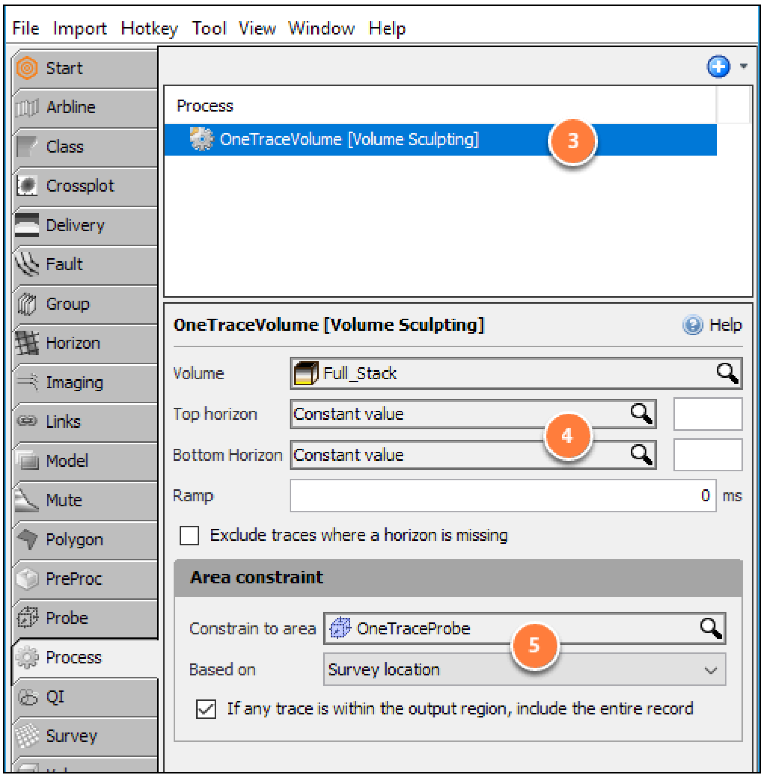
The display of the seismic traces is zoom dependent. The display of the well logs is zoom agnostic. Therefore there will likely always be some necessary tweaking of the well track display width. You may also find it useful to deselect **Use min/max from class** for the volume and adjust the min/max display of the volume (using **F5/F6**) independently to the synthetic trace.

How do I find the RMS amplitude for a single trace over a given time window? How do I do this with a synthetic?

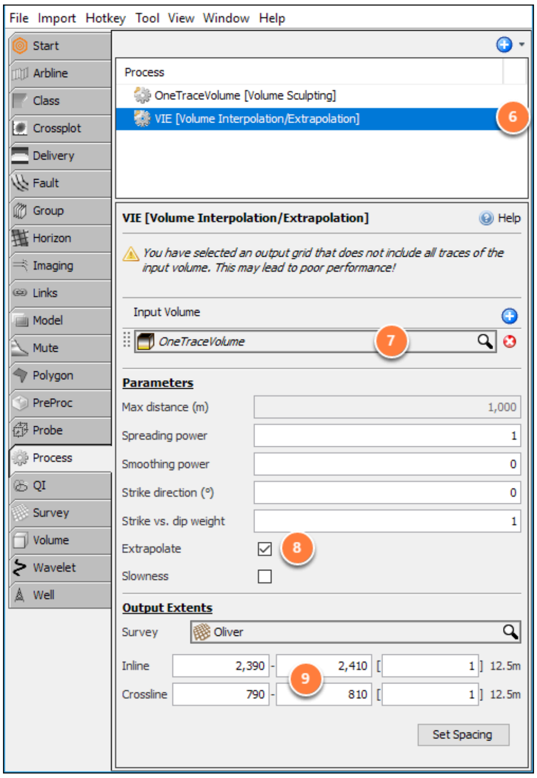
For a single trace from a volume



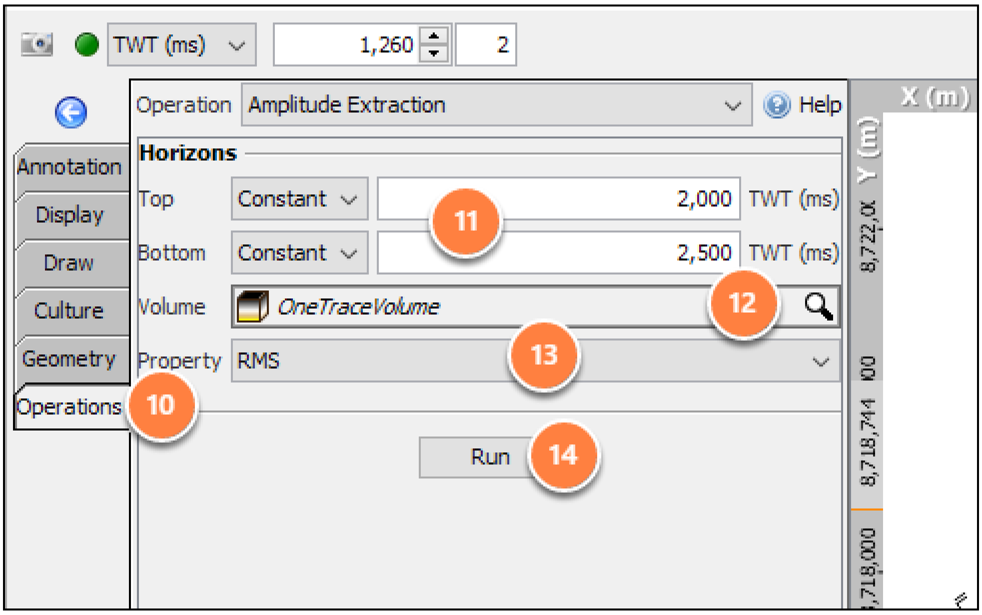
1. Go to the **Process** tab and create a new **Probe**. Type a name for this probe e.g. "*OneTraceProbe*".
2. Set the **IL/CL** size manually in the details panel. Use the same value for min and max for each extent.



3. While still in the **Process** tab, create a **Volume Sculpting process** and type a name for it e.g. "*OneTraceVolume*".
4. Select your volume in the **Input Volume** text box.
5. Select the earlier created probe (e.g. "*OneTraceProbe*") for the **Constrain to area** option and choose **Based on** *Survey location*.



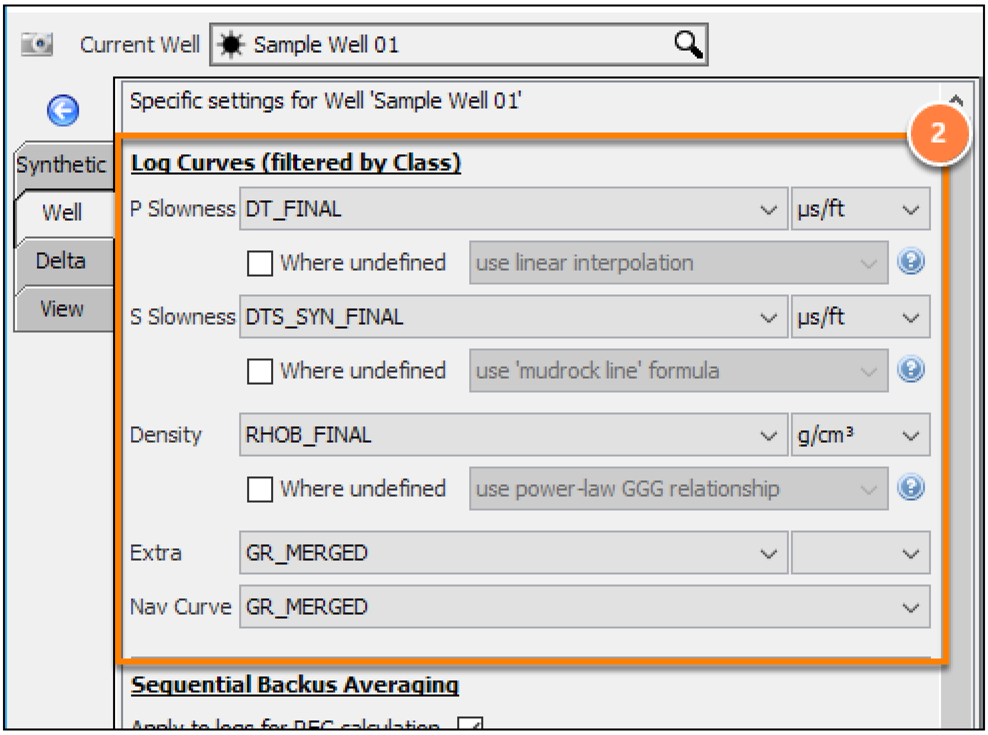
6. Next, create a **Volume Interpolation/Extrapolation process**.
7. Select the **Input Volume** as the one created from the **Volume Sculpting process** e.g. "*OneTraceVolume*".
8. Tick the **Extrapolate** checkbox.
9. For **Extents**, select a small arbitrary area around the probe, approximately 20 x 20 traces.



10. In the **Map View**, click the **Operations** tab on the right and select **Amplitude Extraction** in the **Operations** dropdrop box.
11. Set the interval to analyse by entering a top and base (constant) time, or by choosing horizons.
12. Choose your created volume from the Volume Sculpting process e.g. "*OneTraceVolume*" for the **Volume** option.
13. Select Property as "**RMS**".
14. Click the **Run** button.
15. Set the class to "**RMS**" and click the "**Create New**" button.

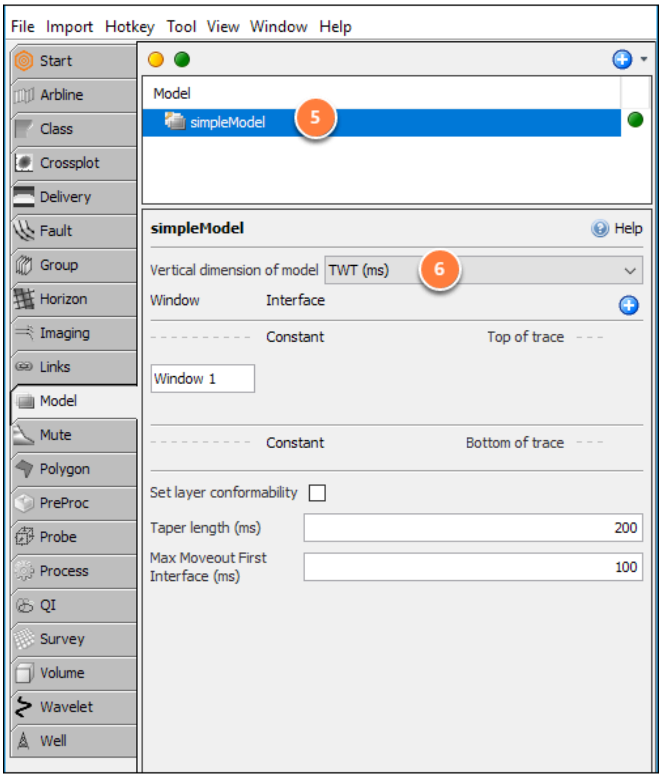
The map view will return a single result for the extrapolated area containing the RMS (or other property) for the single trace.

For a synthetic trace from a well

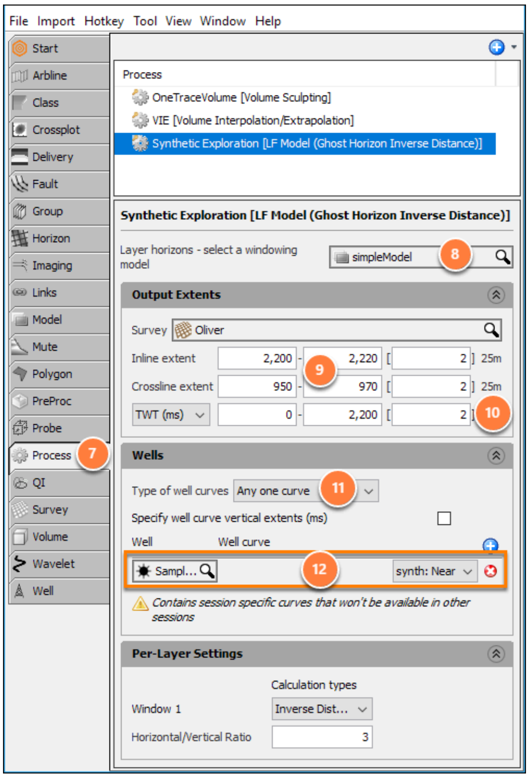


To generate the volume for the synthetic trace:

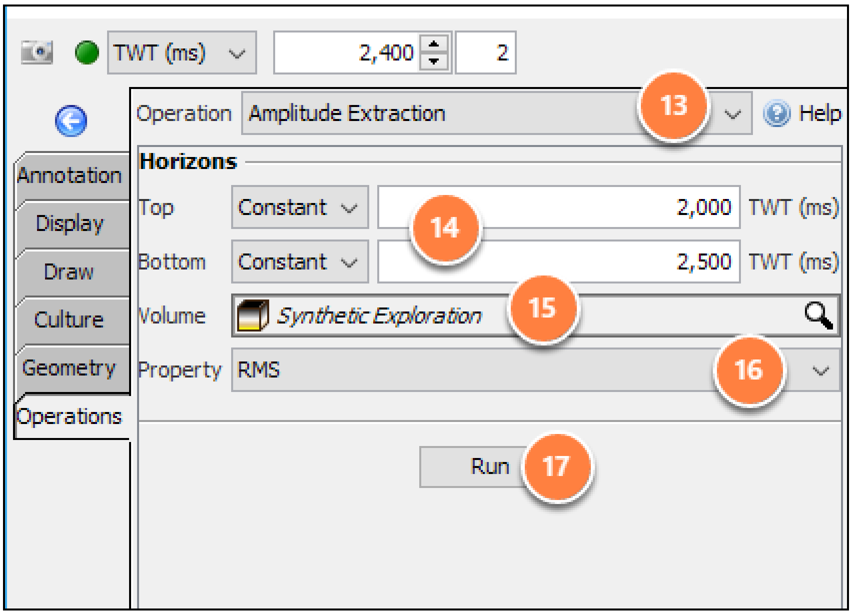
1. Go to **Tool > Synthetics** to configure the well in the **Synthetics** window.
2. Click the **Well** tab and select the input well curves for the synthetic: **P Slowness**, **S Slowness** and **Density**.
3. Select the **Angle**, **Wavelet**, and **Stack** parameters in the Synthetics tab.
4. Confirm that a synthetic is generated!



5. Go to the **Model** tab and create a new arbitrary window **Model**. Type a name for the model e.g. "*simpleModel*".
6. Select **TWT** domain for the **Vertical dimension of model** setting. It does not require any windows or layers.



7. Go to the **Process** tab and create an **LF Model (Ghost Horizon...)** process. Name the process e.g. "*Synthetic Extrapolation*".
8. Select the earlier created **Model** as the **Layer horizons** e.g. "*simpleModel*".
9. In **Output Extents**, pick a small arbitrary area around the well, approximately 20x20 traces.
10. Set the **TWT** sampling to 2 or 4ms (match the rate of the seismic).
11. Select **Type of well curves** as "*Any one curve*".
12. Select the well and the "synth: (near)" synthetic trace - *If the trace isn't listed, confirm that the synthetics are configured correctly (from step 1)*.

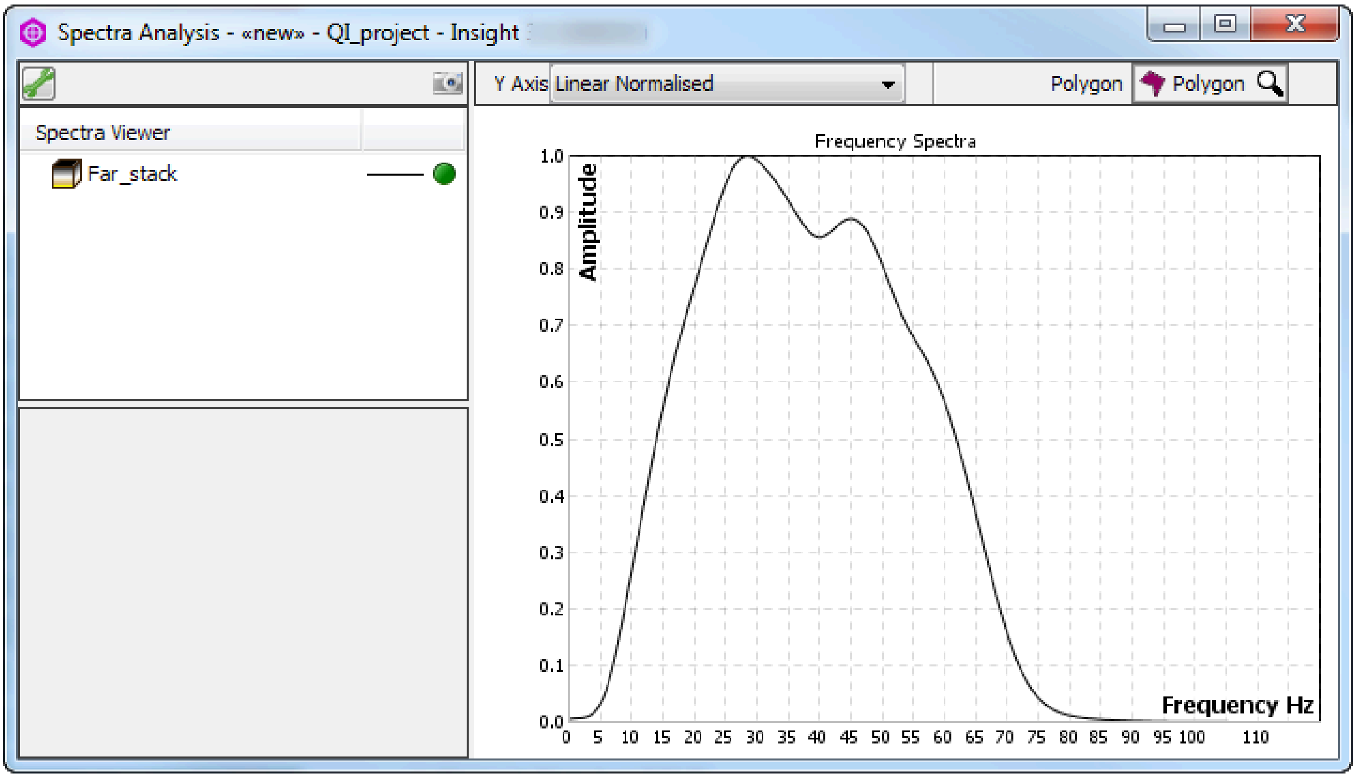


13. In the **Map View**, click the **Operations** tab on the right and select **Amplitude Extraction** in the **Operations** dropdrop box.
14. Set the interval to analyse by entering a top and base (constant) time, or by choosing horizons.
15. Choose your created volume from the **LF Model** process e.g. "*Synthetic Extrapolation*" for the **Volume** option.
16. Select Property as "**RMS**".
17. Click the **Run** button.
18. Set the class to "**RMS**" and click the "**Create New**" button.

Spectra Analysis

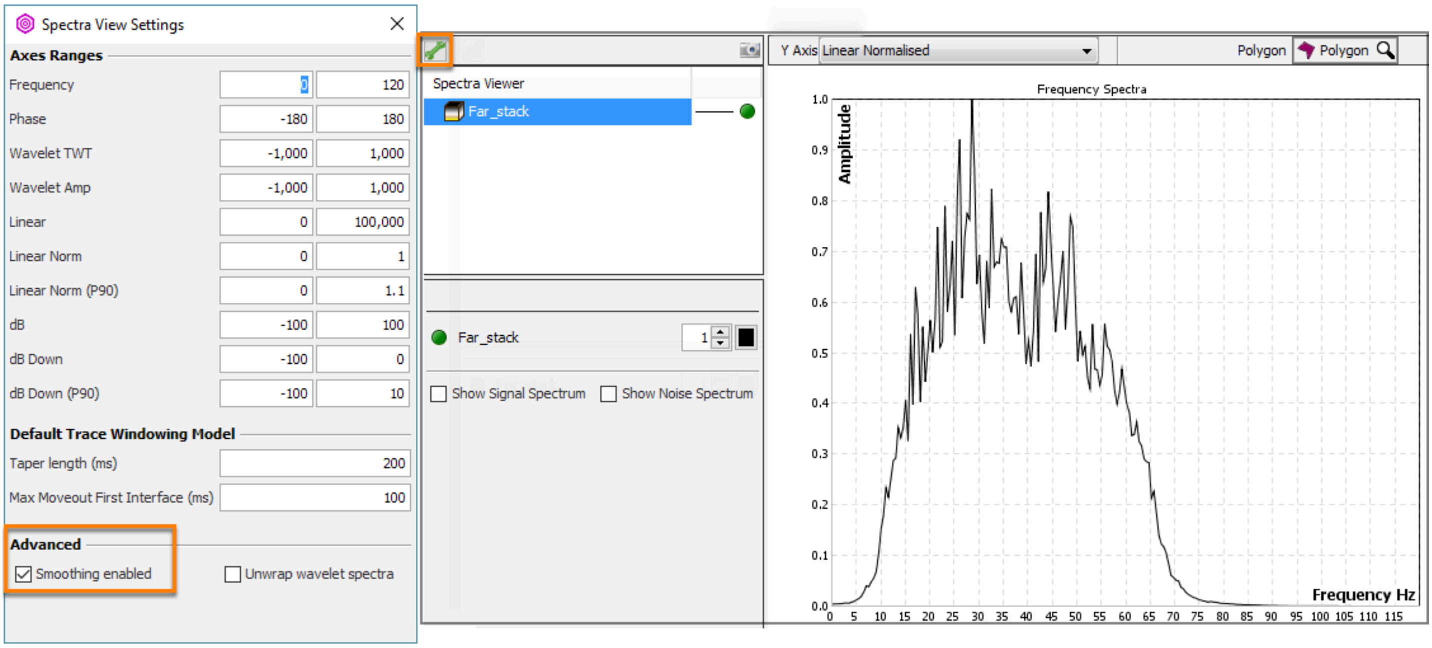
Why does the amplitude spectrum not show deeper peaks and troughs as expected?

When trying to display the seismic amplitude spectra using Spectra Analysis, you may notice that the resulting wavelet does not resemble a typical spectrum.



This is because the frequency spectrum generated is, by default, smoothed to give you a better picture. This setting can be changed by clicking the spanner icon and disabling the **Smoothing enabled** check box (see also *Configuring View Settings*).

This should reveal the deeper peaks and troughs that is expected (see image below).


Spectra View Settings
Axes Ranges
Frequency 0 120
Phase -180 180
Wavelet TWT -1,000 1,000
Wavelet Amp -1,000 1,000
Linear 0 100,000
Linear Norm 0 1
Linear Norm (P90) 0 1.1
dB -100 100
dB Down -100 0
dB Down (P90) -100 10
Default Trace Windowing Model
Taper length (ms) 200
Max Moveout First Interface (ms) 100
Advanced
Smoothing enabled
Unwrap wavelet spectra
Spectra Viewer
Far_stack
Far_stack
Show Signal Spectrum
Show Noise Spectrum
Y Axis Linear Normalised
Polygon
Polygon
Frequency Spectra
Amplitude
Frequency Hz

Kingdom

Why does my Kingdom survey appear with a different spacing in Insight?

The survey "bin spacing" displayed by Kingdom is based on the survey's increment, whereas the spacing displayed by Insight is based on a "single" IL/CL.

Note: In Insight, a 3D survey has no increment or extents — it just represents the geometry. The increment or extents are defined in the volume (in Kingdom, this is called "data type"). This means that many volumes with different extents can easily be loaded into Insight without having to define multiple surveys.

For example, take the following 3D survey as defined in Kingdom:

```
                    Min Value     Max Value     Increment
Inline:             1001          2001          10
Crossline:          3001          4001          5
Bin spacing (m):    Inline:  200      Crossline:  100
```

When loaded into Insight, it will appear as:

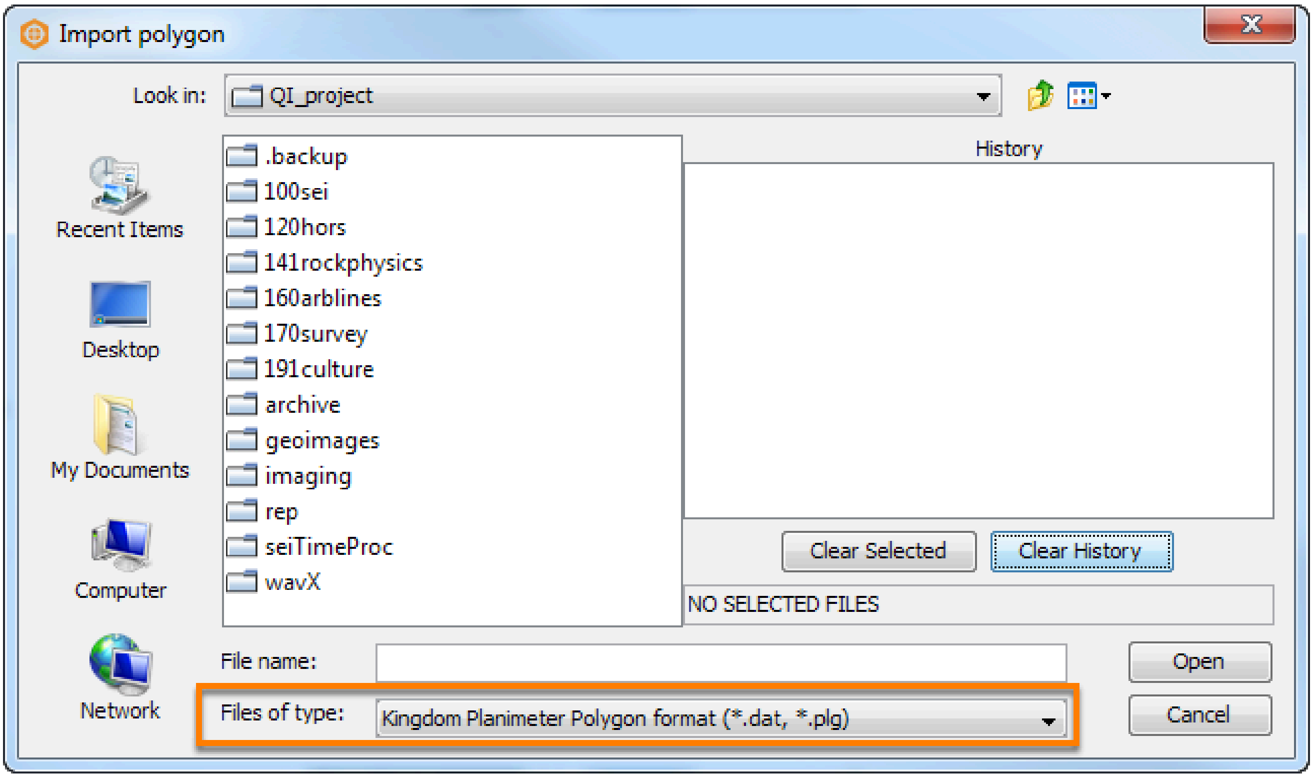
```
 IL spacing:   20 (m)
 CL spacing:   20 (m)
```

How do I import a Kingdom polygon to use in Insight?

Kingdom polygons are not stored in the Kingdom database and thus will not appear in the Manage Remote Resources window when using the Kingdom Reader.

To import a Kingdom polygon, open the **Import** menu in the Control Panel, select **Polygons** and change the file type to the **Kingdom Planimeter Polygon format (.plg)**.

Import polygon



1. From the Control Panel, click on the **Import** menu and select **Polygons**.
2. In the **Import Polygon** window, change the **Files of type** to Kingdom Planimeter Polygon format (*.dat, *.plg).
3. Browse to the polygon that was exported from Kingdom and click **Open**.

Exporting a Kingdom SQL Database to Access

If you have a Kingdom project stored as a Microsoft SQL database, you could convert it to an Access database to avoid all the requirements of an SQL database, including being able to access your data on Linux/Mac. Follow the steps below to export an SQL database to Access by using either Kingdom's Database Admin Tool or Microsoft SQL Server's Import and Export Wizard. Attached is an empty Access database file that you would need if you are using the second method.

empty.mdb

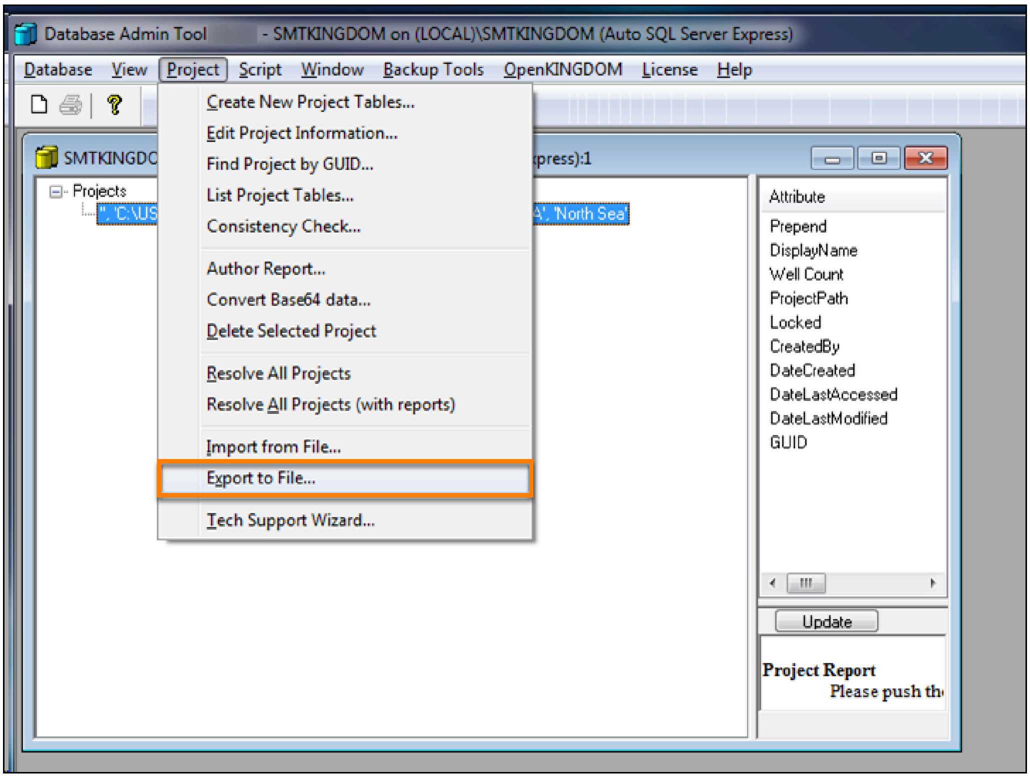
Export using Kingdom's DB Admin Tool

If you have Kingdom installed on your computer, you can access the Database Admin Tool that comes with Kingdom to create a database dump file and a dummy Access project, and do a database dump from SQL to Access.

Notes:

- If your SQL database (*mdf* and *ldf* files) is larger than 2GB, it will be too large to dump into an Access database.
- To work with an Access database, a 32-bit Kingdom DB Admin Tool is required. If you have a 64-bit Kingdom, you will need access to a 32-bit Kingdom.

Open Kingdom's DB Admin Tool



1. At the **Start** menu, click on **All Programs**.
2. Open the **Database Admin Tool** under the Kingdom Software.
3. From the **Database** menu, click **Connect** to connect to the project database.
4. Select **DB type (Auto/Manual SQL)** and select the Kingdom project's .tks file.
5. The database should appear in the main window. Highlight the database.
6. At the **Project** menu, select **Export to file**. This will create a database dump file (*export.dmp*). This file will have all the information from your SQL database file.
7. Then, create a dummy Access project with Kingdom, this will create an empty Access database (.mdb).

Import from file

To import the database dump file into the dummy Access project:

1. From the **Database** menu, click **Connect** to connect to the Access database.
2. Select **Access** and navigate to the Access database file (.mdb). Select the project's .tks file.
3. The database should appear in the main window. Highlight the database.
4. At the **Project** menu, select **Import from file**.
5. Import the database dump file (*export.dmp)* into the new Access database. You will now have all the information from the SQL database in an Access database.

Proceed to *Insight and Kingdom Overview* to begin linking your Access database with Insight.

Export using SQL Server Import/Export Wizard

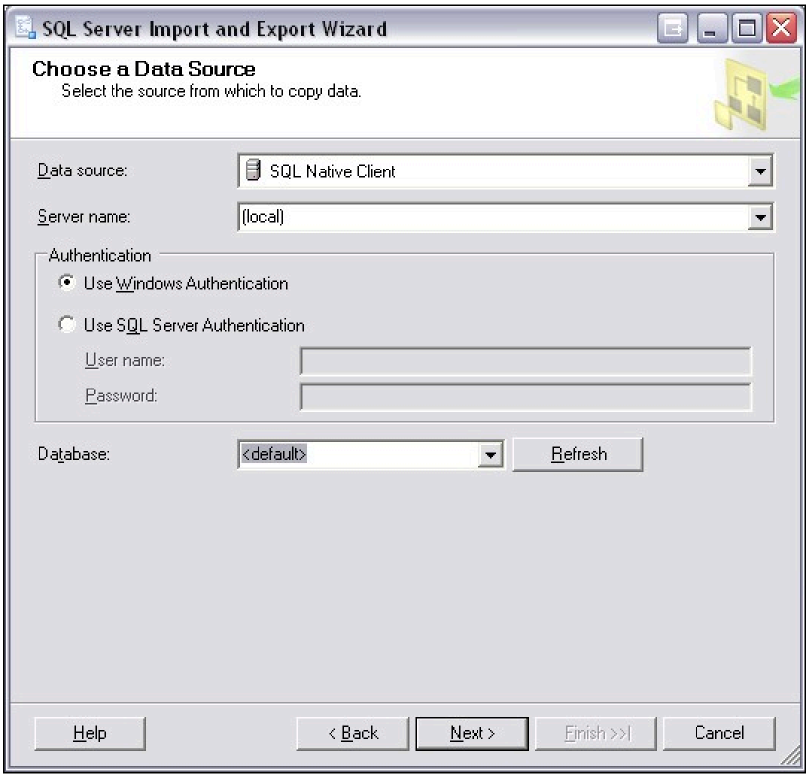
If you do not have Kingdom, you can also convert an SQL database to Access by using the **SQL Server Import and Export Wizard**. To begin:

1. Download the empty Access database file attached at the top of this article (*empty.mdb*) and copy it to your Kingdom project directory.
2. Rename the Access database *<project_name>70sav.mdb* to adhere to Kingdom's naming convention for inactive Access databases. This will help remove any confusion about which database Kingdom will use (the SQL database will remain as the project's active database) as Kingdom does not allow users to select when opening a project.

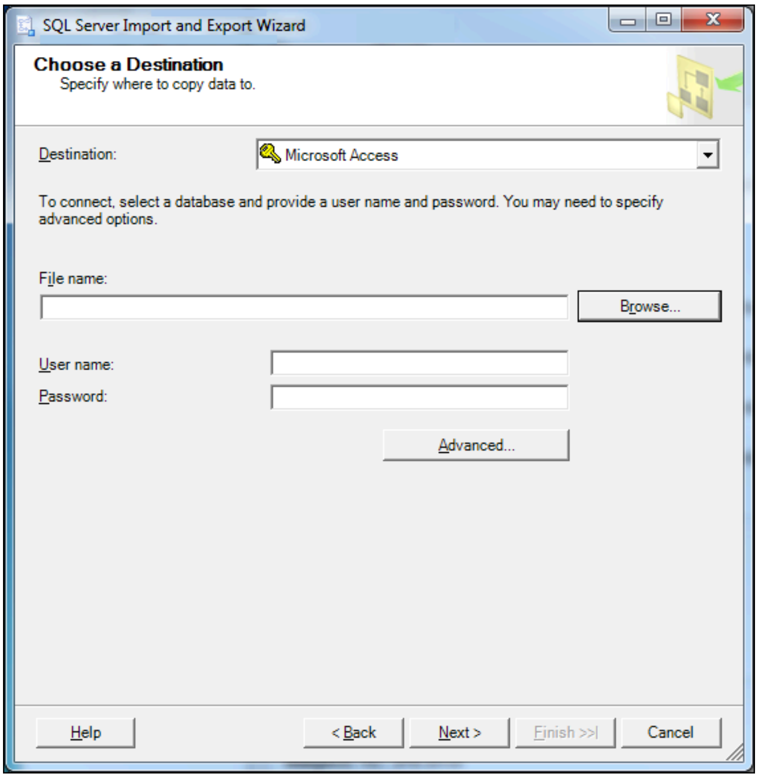
Notes:

- To work with an Access database, a 32-bit SQL Server Import and Export Wizard is required. To install the 32-bit version, select either Client Tools or SQL Server Data Tools (SSDT) during setup.
- Also, the SQL Server Import and Export Wizard is only available in SQL Server 2008 and newer versions.

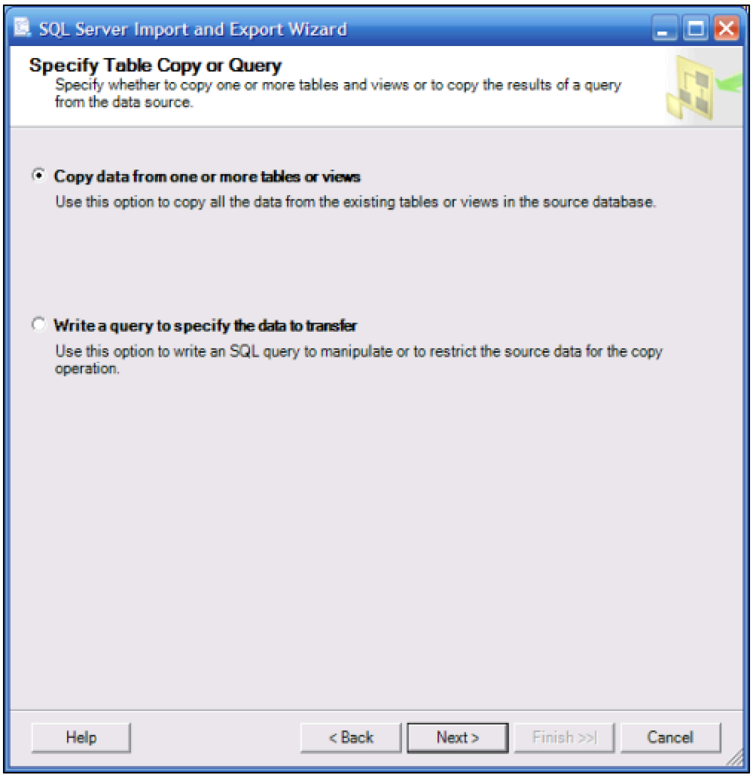
Open SQL Server Import and Export Wizard



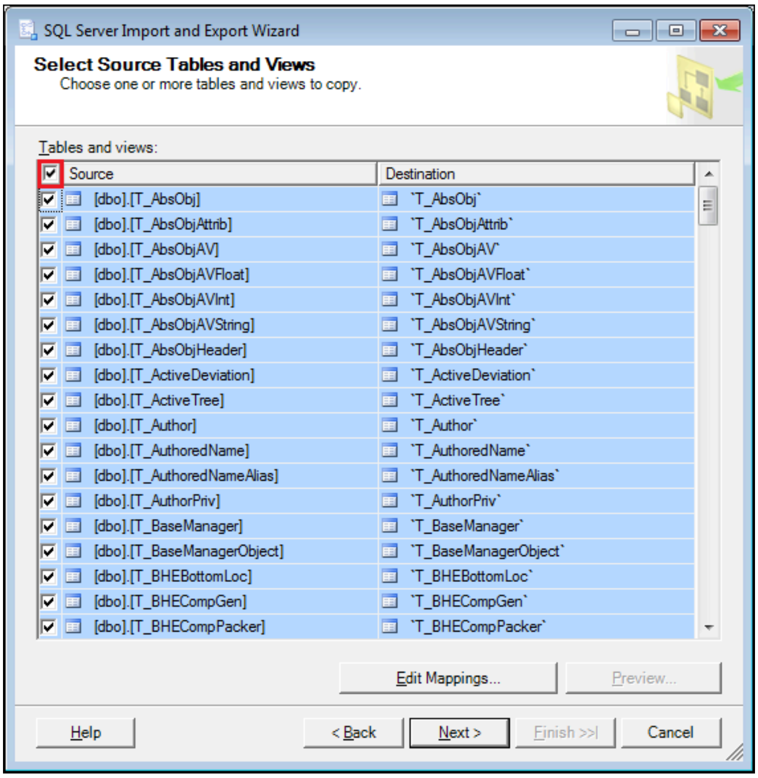
1. Click the **Start** button to access the Start menu.
2. At the Start menu, click on **All Programs** and expand the **Microsoft SQL Server 20XX** folder. Click on **Import and Export Data** to open the Import/Export wizard.
   Alternatively, you can access the wizard from SQL Server Management Studio by right-clicking on a database, selecting **Tasks** and click **Import/Export Data**.
3. Click **Next** to proceed.
4. In the **Choose a Data Source** page, choose **SQL Server Native Client** as your Data Source and complete the following:
   - **Server name** — Type the name of the server or choose from the drop-down list.
   - **Use Windows/SQL Server Authentication** — Choose whether the wizard should use Windows or SQL Server Authentication to log in to the database. If you choose SQL Server Authentication, input a **User name** and **Password**.
   - **Database** — Select from the list the database you wish to export.
   - **Refresh —** Click to restore the list of available databases.
5. Click **Next**.



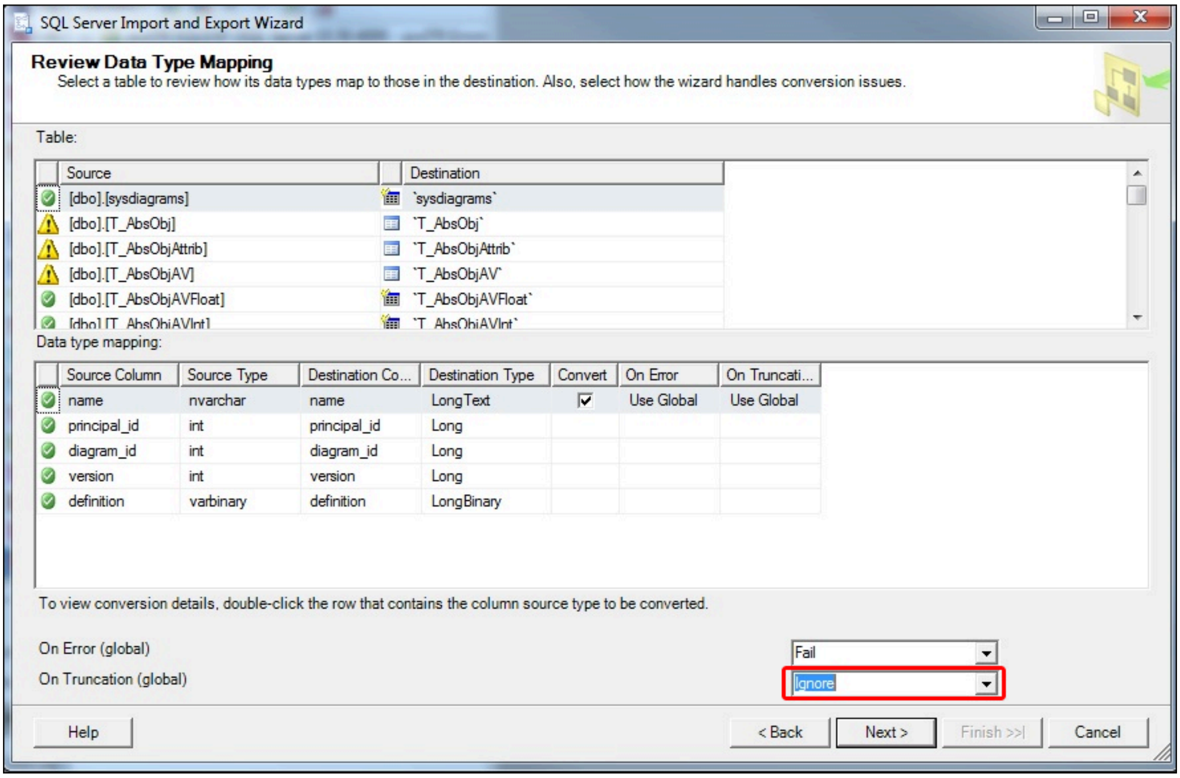
6. In the **Choose a Destination** page, select **Microsoft Access** as your data destination and complete the following:
   - **File name** — Browse and select the empty Access database file (*empty.mdb*) given in this article.
   - **User name** & **Password** — Leave these blank. The empty database does not require any kind of authentication.
7. Click **Next** to proceed.



8. Select **Copy data from one or more tables or views** and click **Next**.



9. In the **Select Source Tables and Views** window, select all tables by clicking on the top left check box.
10. Click **Next** to review data type mapping.



11. At the **Review Data Type Mapping** page, select **Ignore** from the drop-down list at **On Truncation (global)** to allow the wizard to ignore any truncation and continue the process. By default, this is set to **Fail**.
12. Click **Next** to proceed with the rest of the settings.
13. Once the export is successful, you will have all the information from the SQL database in the Access database.
14. Save the package and set your preferred protection level.

This process only creates an Access database for use with Insight. It does NOT convert the Kingdom project itself to Access. Any changes made in the Kingdom project after this will render the Access database out-of-date and the user must then repeat this process.

Note: The SQL Server Import and Export Wizard tool is NOT part of Insight. As such, please make sure to check that your data is unaffected after completing this process.

Proceed to *Insight and Kingdom* to begin linking your Access database with Insight.

How to get interpretation into Kingdom

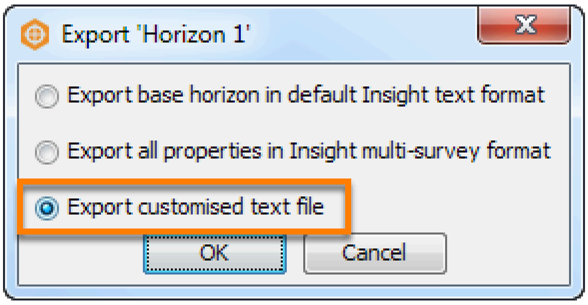
This is a quick reference guide of the formats best suited to transfer data *into* Kingdom.

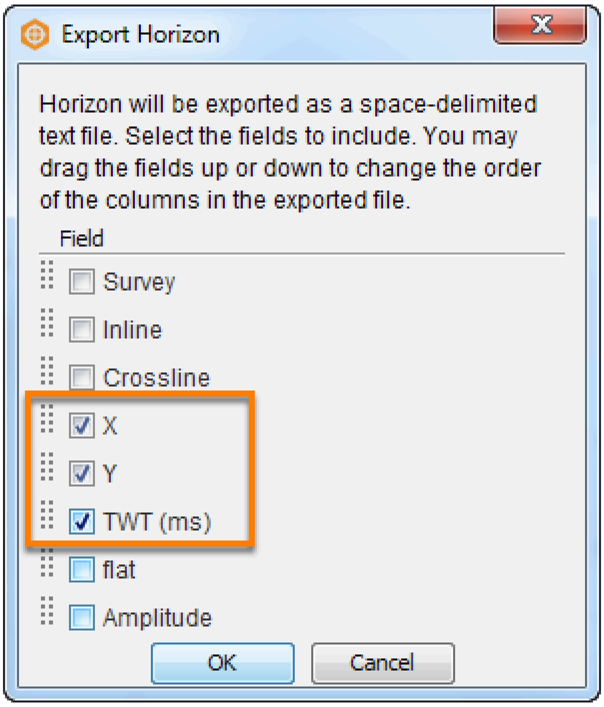
To transfer data *from* a Kingdom database, see *Insight and Kingdom Overview*.

Note: This guide is given in “good faith”, information here is based on experience from our users. You may still need to edit your files between software.

Horizons into Kingdom

Option 1

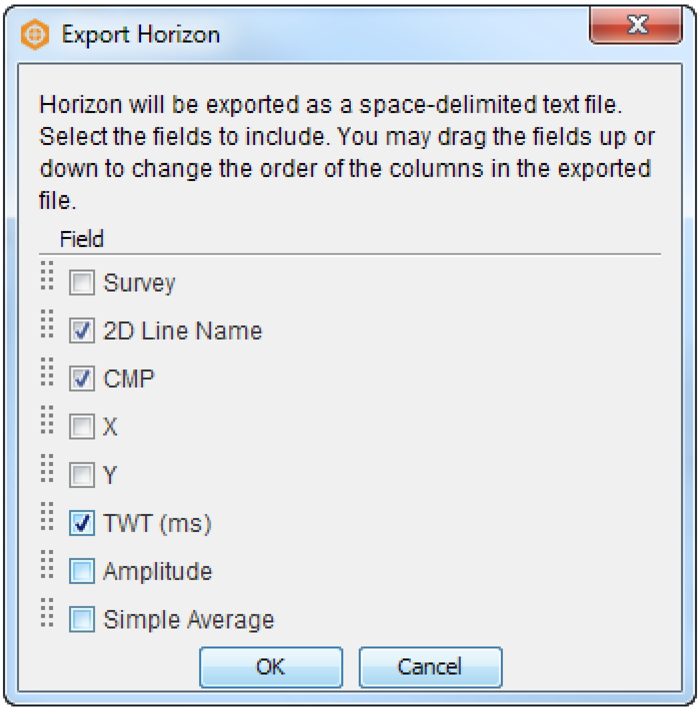




In general, we recommend exporting horizons from Insight as X/Y/TWT grids and importing them into Kingdom as a grid. You can then use Kingdom's **Grid to Horizon** feature to convert these grids to editable horizons.

To export horizons for import into Kingdom, select **Export customised text file** when importing from Insight and export **X, Y, TWT/TVD**.

Option 2

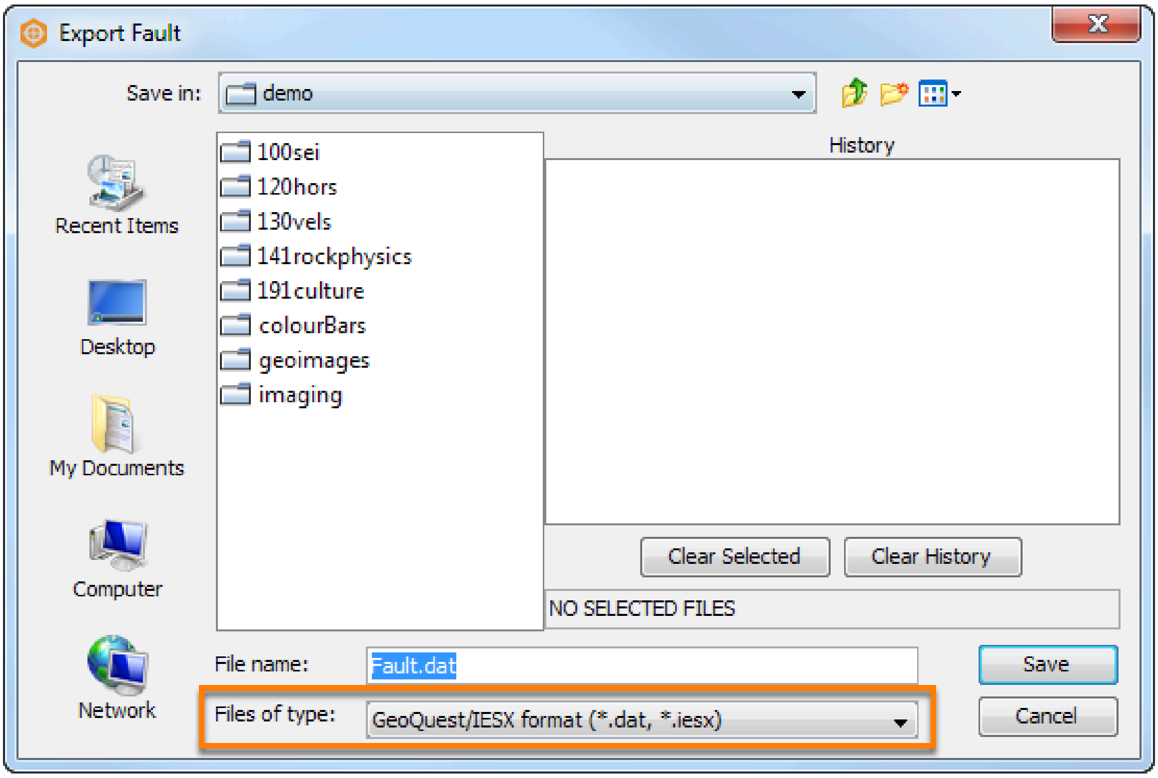


There is a second option, which is technically more difficult, but potentially more reliable:

- The reason for the enhanced difficulty is due to the fact that Insight handles 2D data in relation to CMPs, where as Kingdom handles 2D data in relation to SPs.
- Export your Insight horizons as a customised text file with the following format: 2D Line Name, CMP, TWT.
- If you are sure of the SP–CMP relationship, you could then edit this file (in MS Excel or likewise), changing the CMPs to equivalent SPs, and then import into Kingdom directly as a horizon using the format 2D Line name, Trace (SP), TWT.
- The horizon interpretation data should then be located directly on to the corresponding 2D Survey in Kingdom (assuming all line names are the same).

Faults into Kingdom

Fault surfaces are exported from Insight in Geoquest/IESX format.



At the **Import fault surface** dialog in Kingdom, specify these as **Assigned Fault Lines in a Named Fault** and choose the input format **GeoQuest IESX v.11**, and then assign them to a survey.

Unassigned fault sticks are also exported from Insight in Geoquest/IESX format and can subsequently be imported to Kingdom in a similar way. At the **Import fault surface** dialog in Kingdom, specify that these are **Unassigned Fault Lines** and choose the input format **GeoQuest IESX v.10.x**, and assign them to a particular survey.

Note: Insight operates in XY space. This can cause problems importing 2D faults as Kingdom requires 2D survey data that is not available in Insight. The best way to get around this is to create a dummy 3D survey in Kingdom and import the faults as Geoquest IESX V10.

SQL Server versions and backwards compatibility

Microsoft SQL Server can attach databases from older versions, but not newer versions. For example, SQL Server 2008 can attach a database that was detached from the 2005 version, but not one detached from the 2012 version.

However, when attaching a database from an older version, the database is upgraded to the current server version. That is, after a 2005 database is attached to SQL Server 2008, it can no longer be attached to SQL Server 2005!

Since some Kingdom releases include older SQL Server versions, it is possible that Kingdom may not be able to use the database after attaching it to a newer version. Therefore, if there is any chance the project is going to be used in Kingdom in the future, it is recommended to create a backup of the project database before attaching it for use with Insight.

To backup the project database:

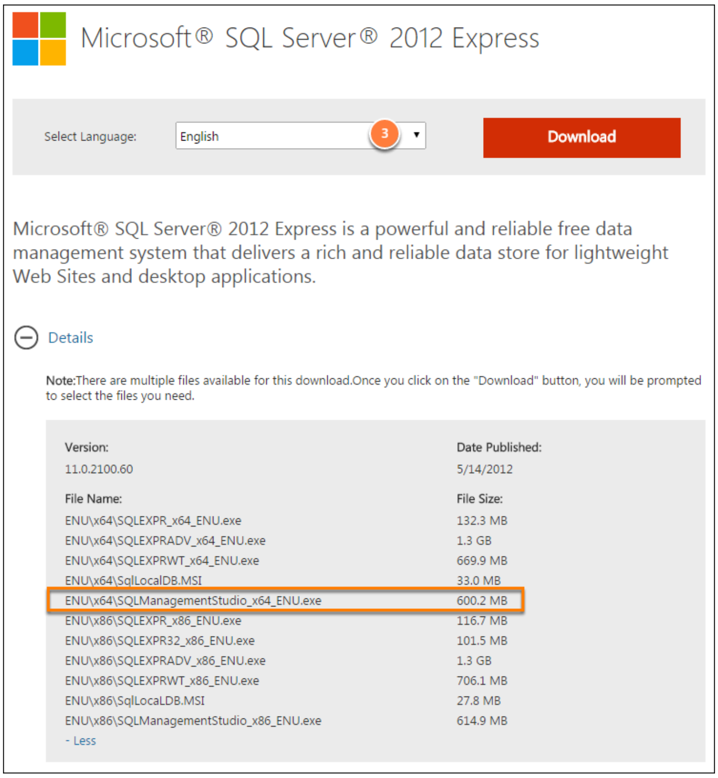
1. Find the .MDF and .LDF file (usually in the "Project Database" folder within the project)
2. Make a copy of the files
3. Rename the files to .MDF.BAK and .LDF.BAK

Installing MS SQL Management Studio

For troubleshooting purposes, Kingdom projects in Auto SQL Server Express (SSE) mode can be manually attached to the SQL Server Management Studio. Follow the steps below to download and install Microsoft SQL Server Management Studio.

Upon successful installation, see *Manually Attaching Kingdom Database from an Auto SQL Server Express (SSE) Project* to attach the Auto SSE Kingdom database manually.

Download and install

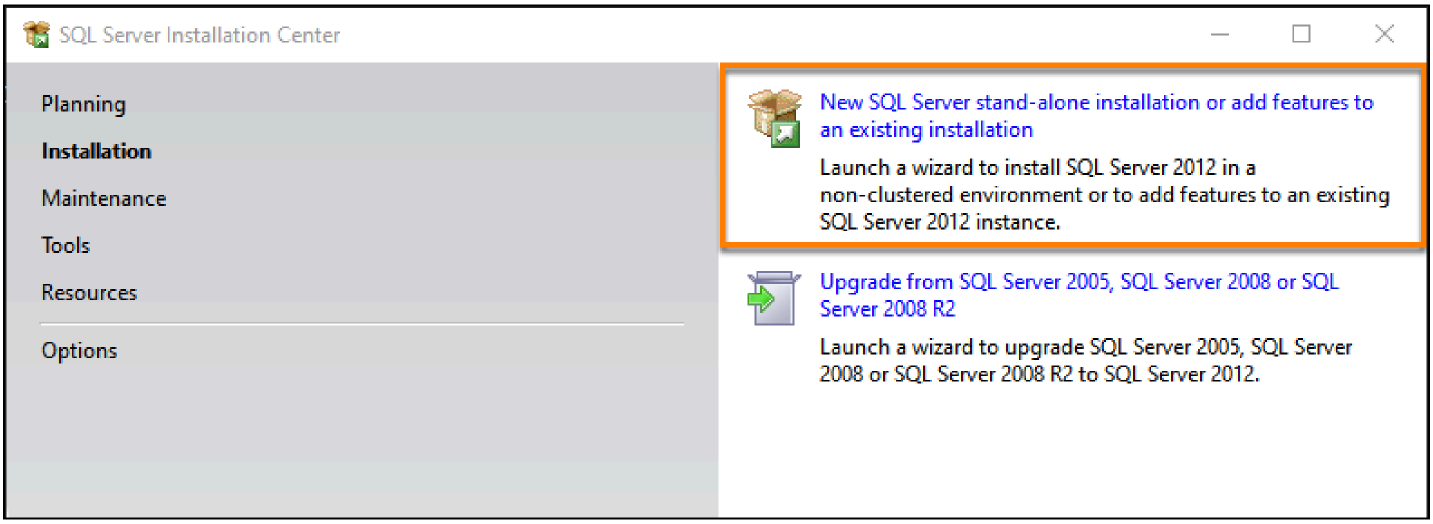


1. Open your preferred browser.
2. Download Microsoft SQL Server Management Studio Express from http://www.microsoft.com/en-us/download/details.aspx?id=29062.
3. Choose your language and click **Download**.
4. Select the **ENU\x64\SQLManagementStudio_x64_ENU.exe** check box to only download the Management Studio for 64-bit operating systems.
   - For 32-bit operating systems, select **ENU\x86\SQLManagementStudio_x86_ENU.exe**.

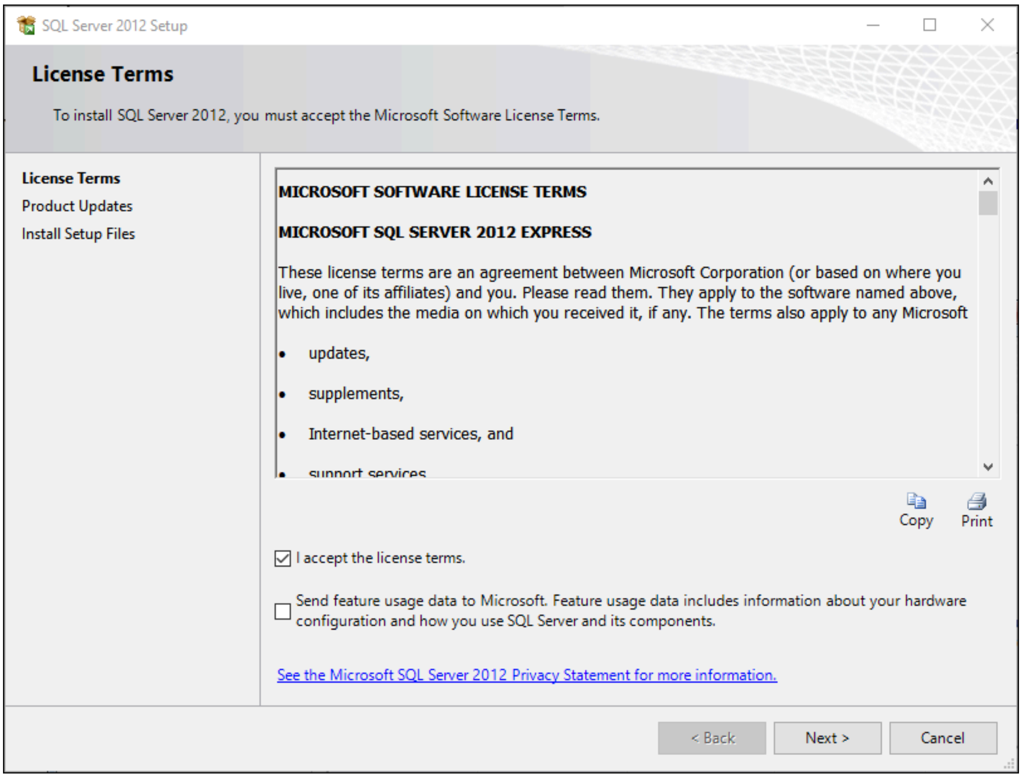
Note: If you do not have SQL Server 2012 installed on your machine, select **ENU\x64\SQLEXPRWT_x64_ENU.exe** (or **ENU\x86\SQLEXPRWT_x86_ENU.exe** for 32-bit) to download and install both SQL Server 2012 and Management Studio in a bundle.

5. After the download is completed, run the .exe file as **Administrator**.
   - If there is no option to run it as Admin, open your **Start** menu, type "cmd" in the Search bar, right-click and run **cmd** as administrator.
   - At the command prompt, enter the file path of the .msi file (i.e. C:\Users\Guest\Desktop\SQLManagementStudio_x64_ENU.exe) and press **Enter**.
   - Click **Run**.

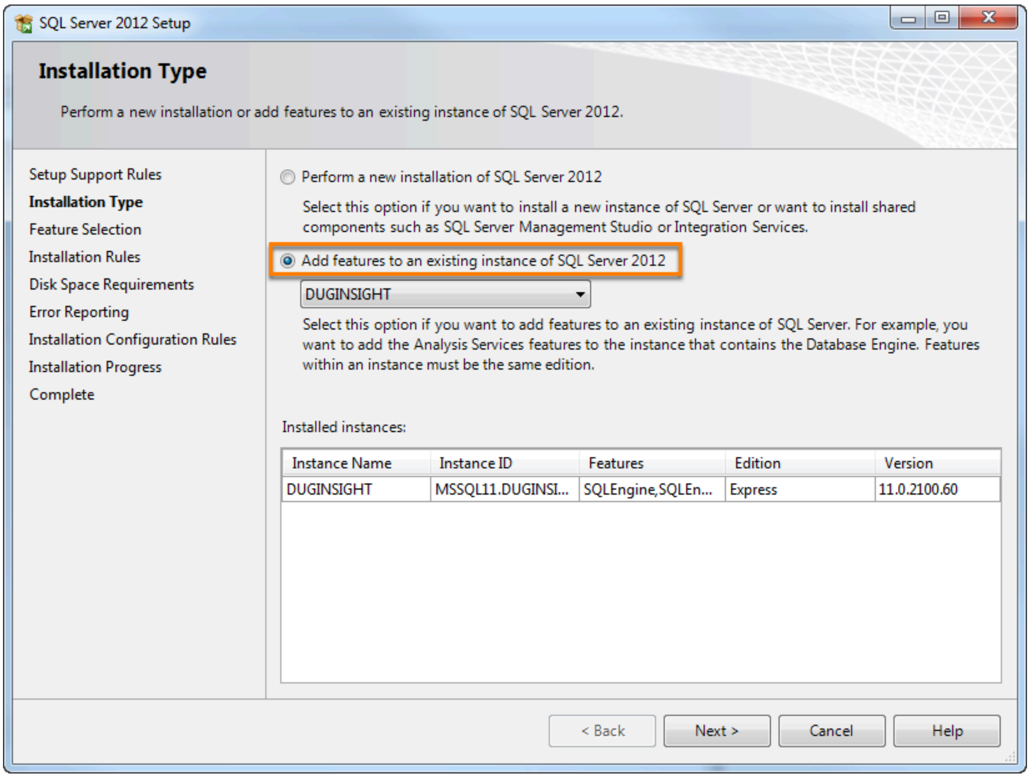
Note: If you are unsure about how to use the command prompt, please consult your IT Department for help.



6. In the Installation Center, click on **New SQL Server stand-alone installation** to add features to an existing SQL Server 2012 instance.

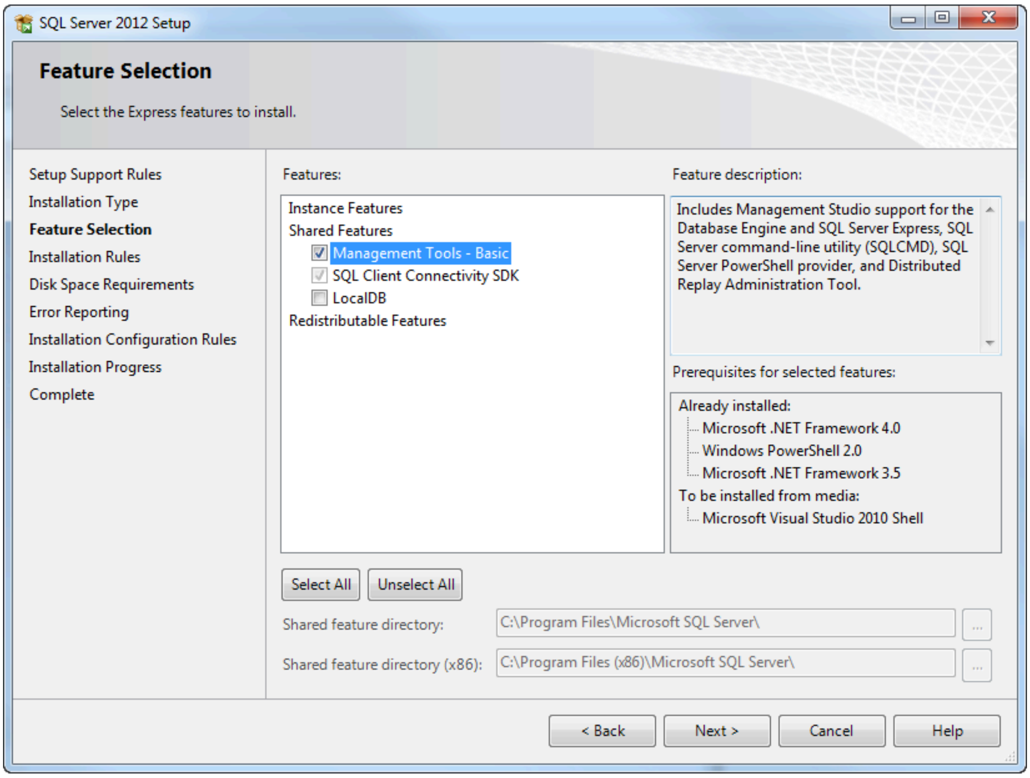


7. Read and accept the **License Agreement**.
8. Click **Next**.
9. At the **Product Updates** window, click **Next** to install the setup files.

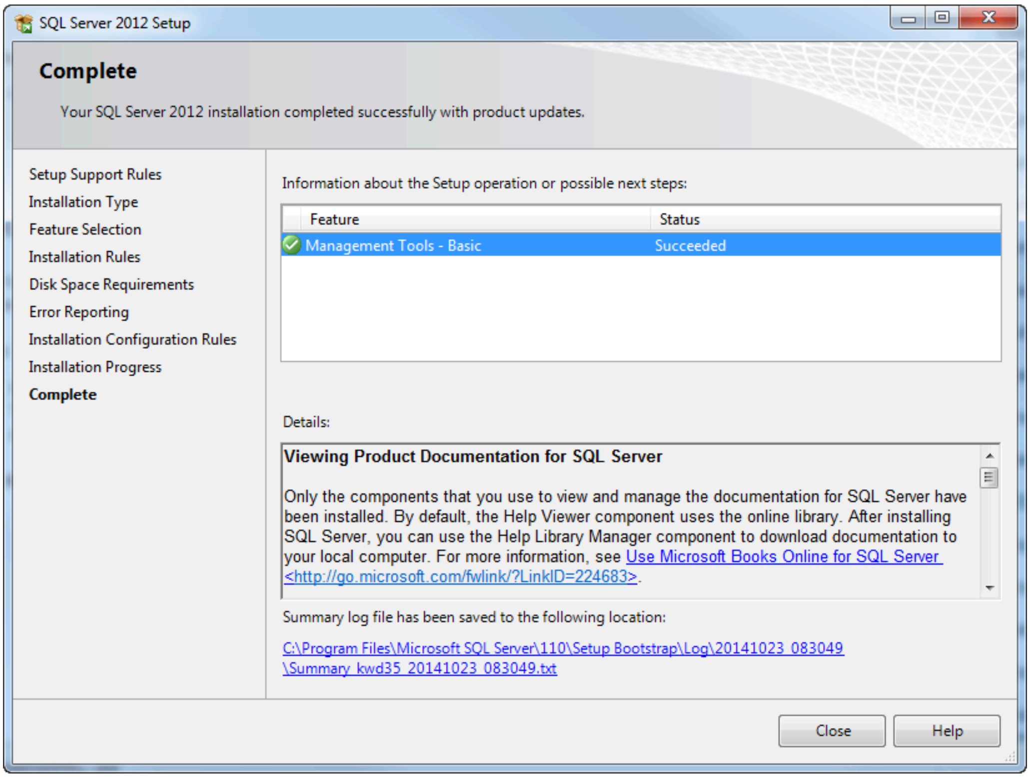


8. After the setup files finish installing, select to **Add features to an existing instance** at the **Installation Type** window.

9. Click **Next** to proceed to Feature Selection.



10. To install the Management Studio, select the **Management Tools - Basic** check box.
11. When you are ready, click **Next**.
12. At the **Error Reporting** window, skip the option to send information to Microsoft (optional), and click **Next** to begin the installation.



13. When the installation is completed, click **Close**.
14. Right-click and run Microsoft SQL Management Studio Express as Administrator from the **Start** menu.
15. Choose the Server and Authentication that was specified in *Downloading and Installing SQL Server*.

Proceed to *Manually Attaching Kingdom Database from an Auto SQL Server Express (SSE) Project* to attach the Auto SSE Kingdom database manually.

Note: If you are having trouble connecting to the instance, it could be because of restricted permissions of the database files or a folder leading to the database files.

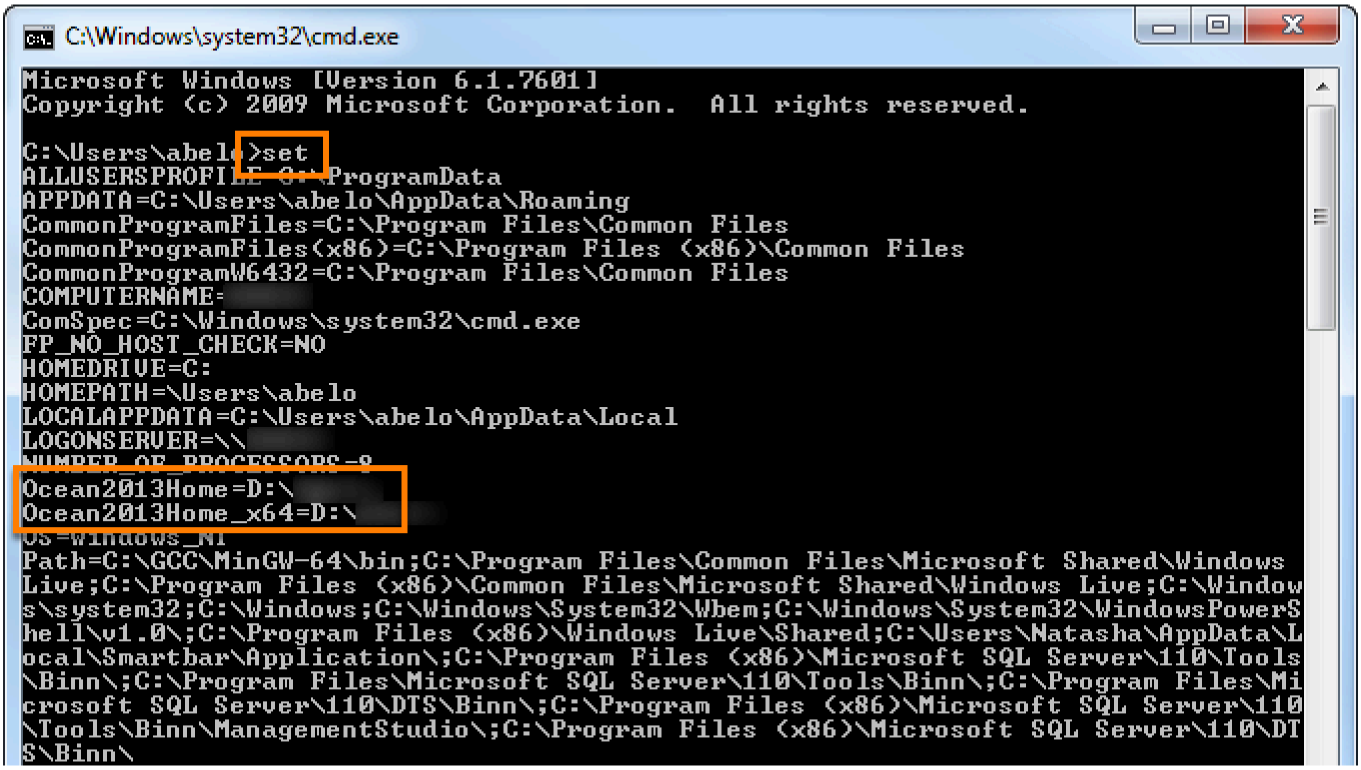
Feel free to contact us at support@dugeo.com if you have any queries.

Petrel

How Insight detects the Petrel installation

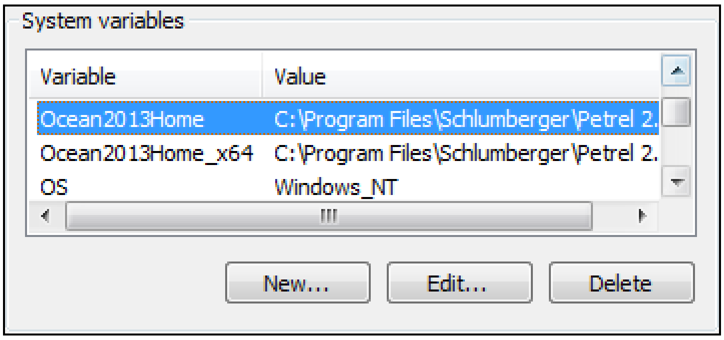
When you run the installer, the Setup Wizard will automatically check if Petrel is installed in your machine. The way it does that is by running a "set" command and searching for the **Ocean201XHome** environment variable.

If you have multiple versions of Petrel installed on one machine, Insight will install the plugin to all detected Petrel versions.



If the installer detects that variable, it will give you the option to install the **Petrel Link** during the setup (*see* *[Downloading and Installing Insight on Windows](#)*).

Petrel environment variable



If Petrel is installed in your machine, but the system is unable to detect the **Ocean201XHome** line, the system variable name or path might be different. To check and/or change the variable:

1. From the **Start** menu, open the Control Panel.
2. Click on **System and Security** and then click on **System**.
3. On the left sidebar, select **Advanced system settings**.
4. In the **System Properties** window, click on **Environment Variables**.
5. Look for the variable that is registered when Petrel was installed.
6. Click **Edit** to change the path stored in a variable:
   - **Update** the path to match the Petrel installation directory.
   - Click **New** to add a new variable (and path), if your current Petrel version is not listed.
7. Run the installation again.

Note: If there are multiple versions of Petrel installed in the machine, Insight will find all versions of the **Ocean201XHome** environment variable to install the Petrel Link.

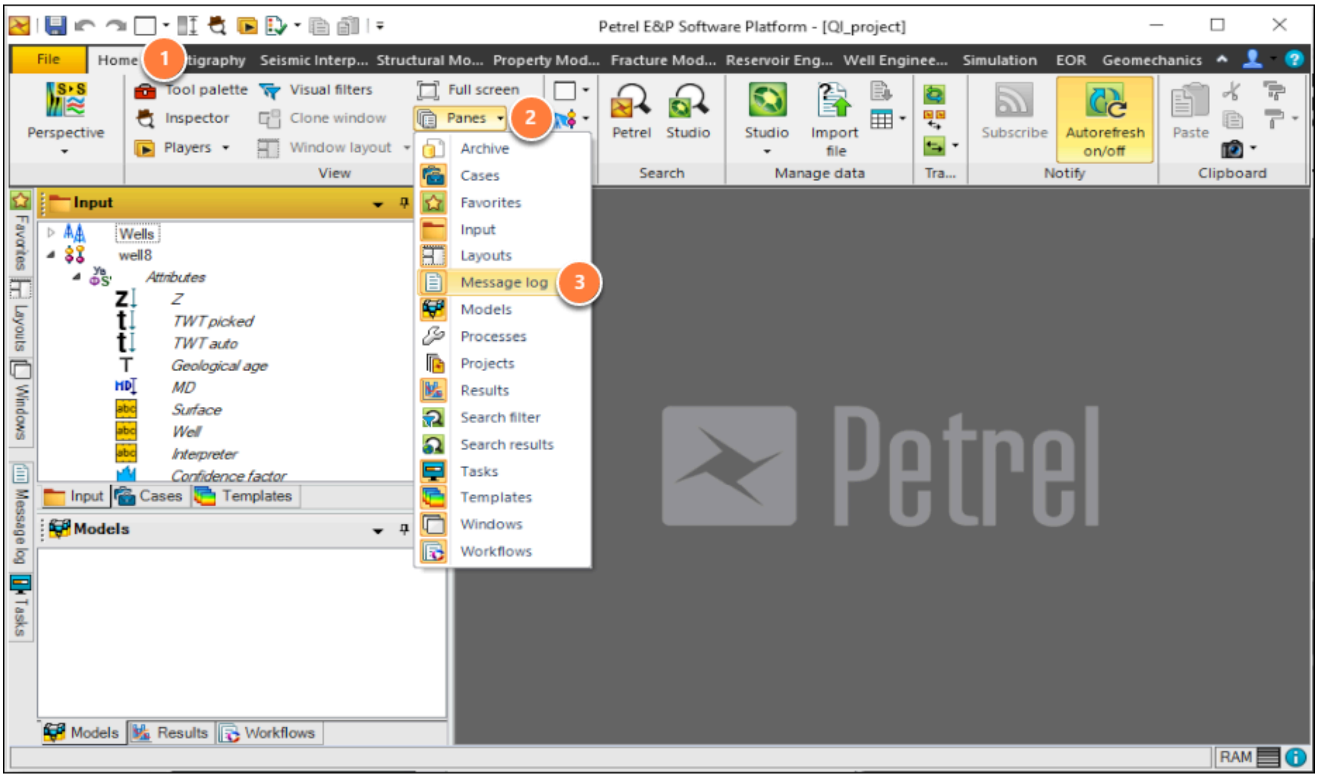
Genuine offer of assistance

If you are still unable to install or connect using the Petrel Link, please do not hesitate to contact our support team at support@dugeo.com.

How do I send the Petrel message log?

If you are experiencing problems working in Petrel with Insight, sending us your Petrel message log could help give us a better idea as to the nature of the problem.

Follow the steps below to enable Petrel's message log, and copy the text inside.



1. In **Petrel**, go to the **Home** tab.
2. Click **Panes**.
3. Click on **Message Log** and make sure that it is toggled on.
4. Locate the **Message Log** window.
5. Click on the **Copy all text to the clipboard** icon (second icon from the left at the top of the **Message Log** window).
6. Open your email and compose a new email. Address the email to [*support@dugeo.com*](mailto:support@dugeo.com).
7. In the main body, right-click and select **Paste**, or press **Ctrl + V** on your keyboard to paste the text.
8. If you have a support number, please indicate it in the subject. Write a brief description of the issue.

The more we know about the problem, the easier it is for us to help you!

How do I export polygons from Petrel to import into Insight?

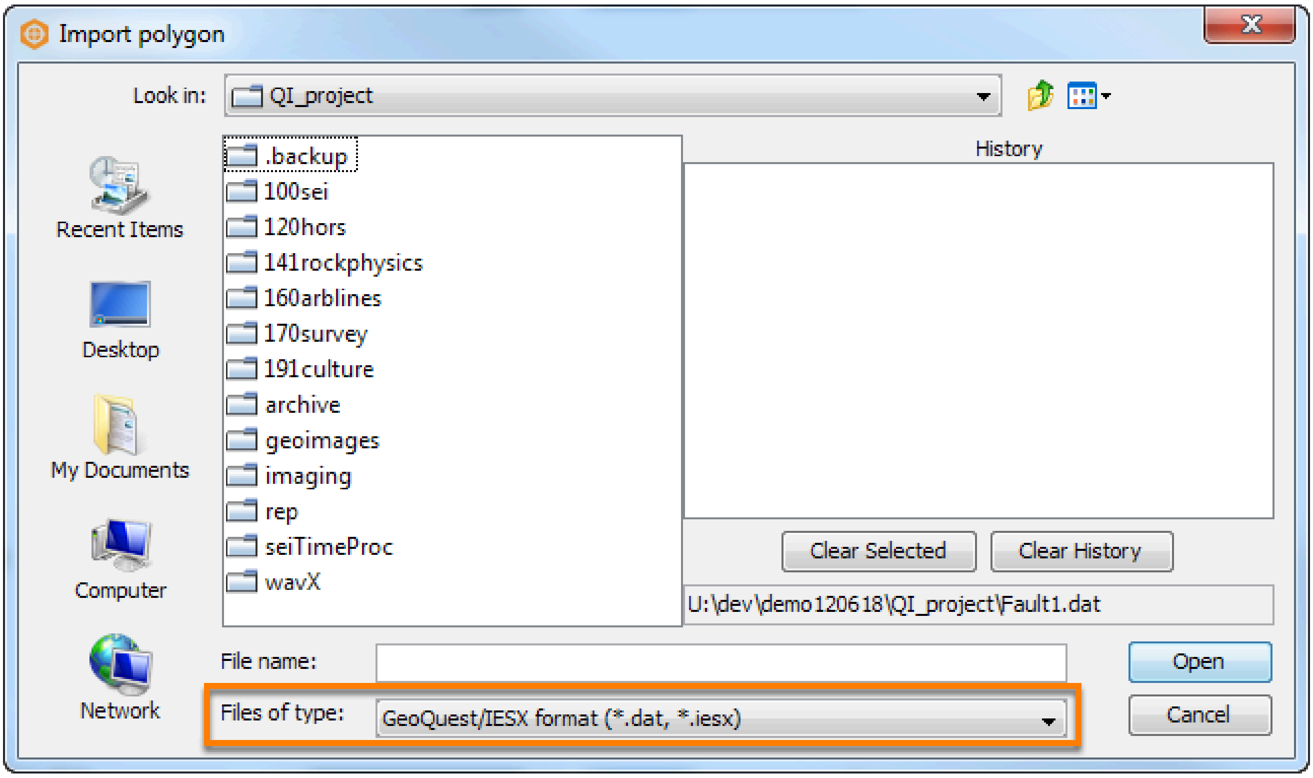
DUG Insight can load IESX format polygon files that have been exported from Petrel.

Export Polygon

Right click on the polygon in the Petrel navigation tree and select **Export**.

Polygons should be exported as **IESX fault polygons (ASCII)** with a .dat or .iesx extension.

Import Polygon

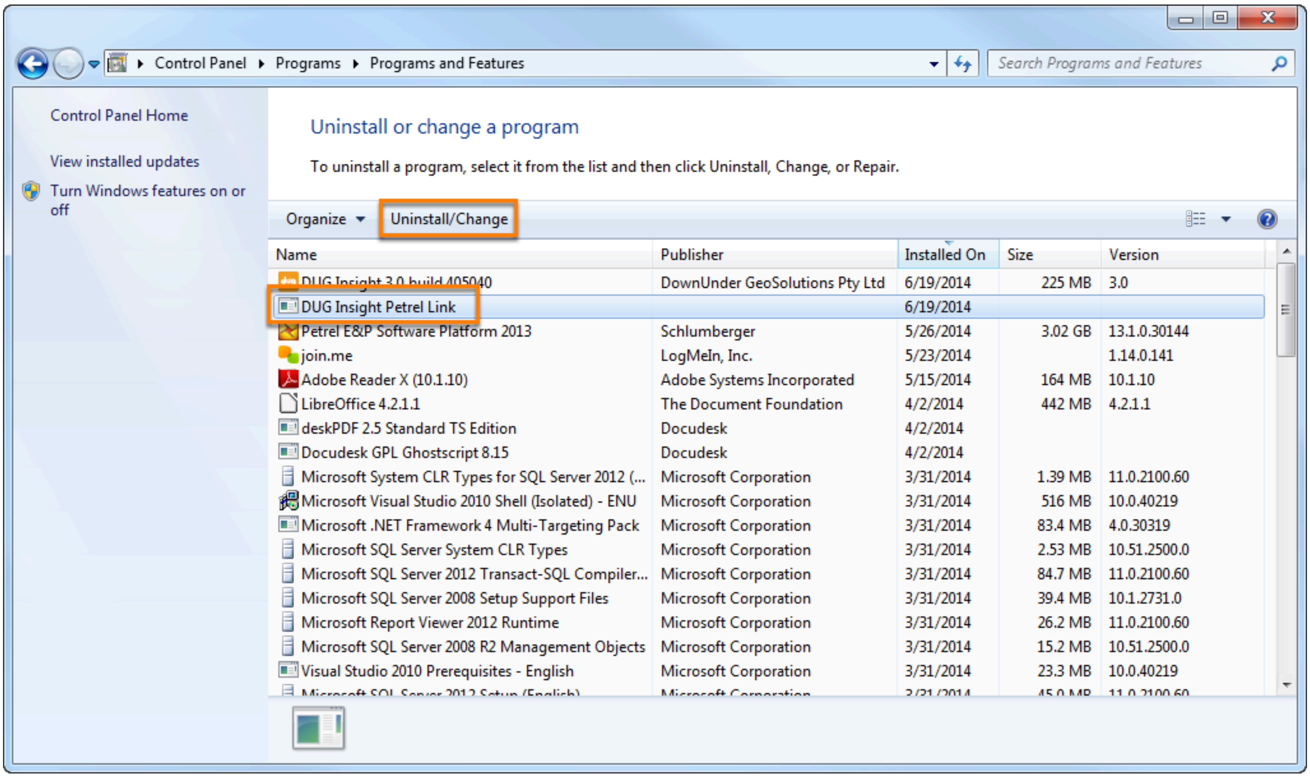


1. From the Control Panel, click on the **Import** menu and select **Polygons**.
2. In the **Import Polygon** window, change the **Files of type** to GeoQuest IESX format.
3. Browse to the polygon that was exported from Petrel and click **Open**.

How do I manually remove the Petrel Link?

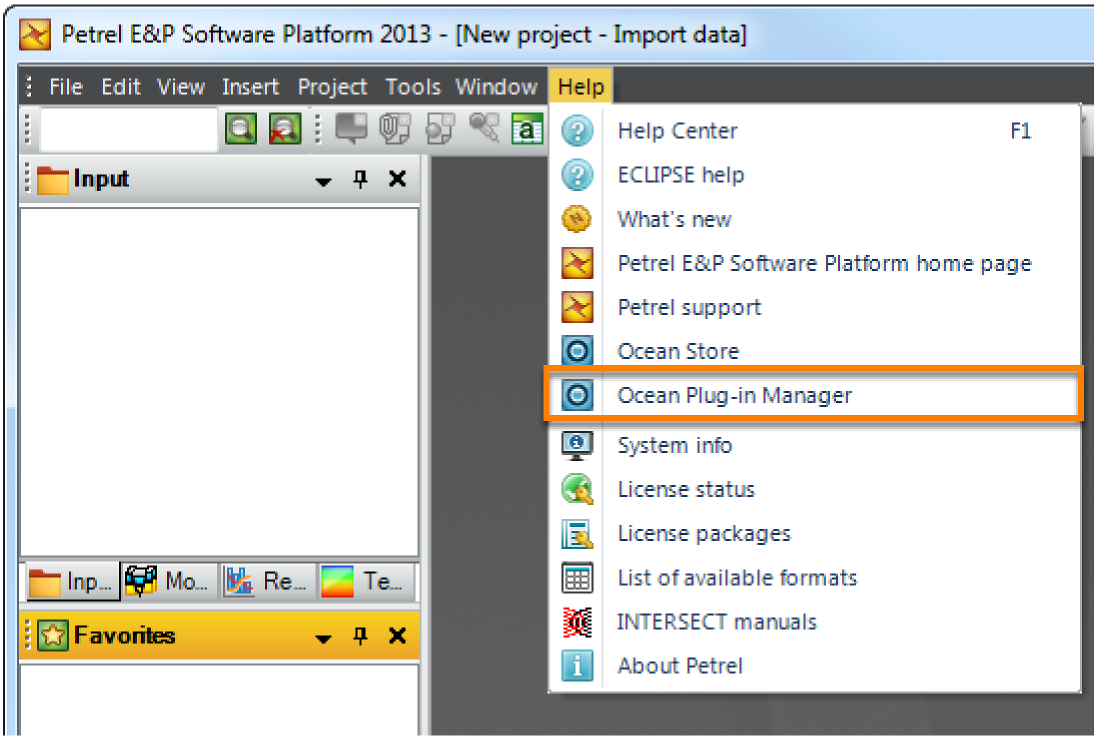
Newer versions of Insight are bundled with a compatible version of the Petrel Link during installation. If you already have the Petrel Link installed for an older version, that version will no longer work and needs to be manually removed before you install the new version. You can uninstall older versions of the Petrel Link manually from your computer's Control Panel or via Petrel's **Ocean Plug-in Manager**.

Uninstall via Control Panel

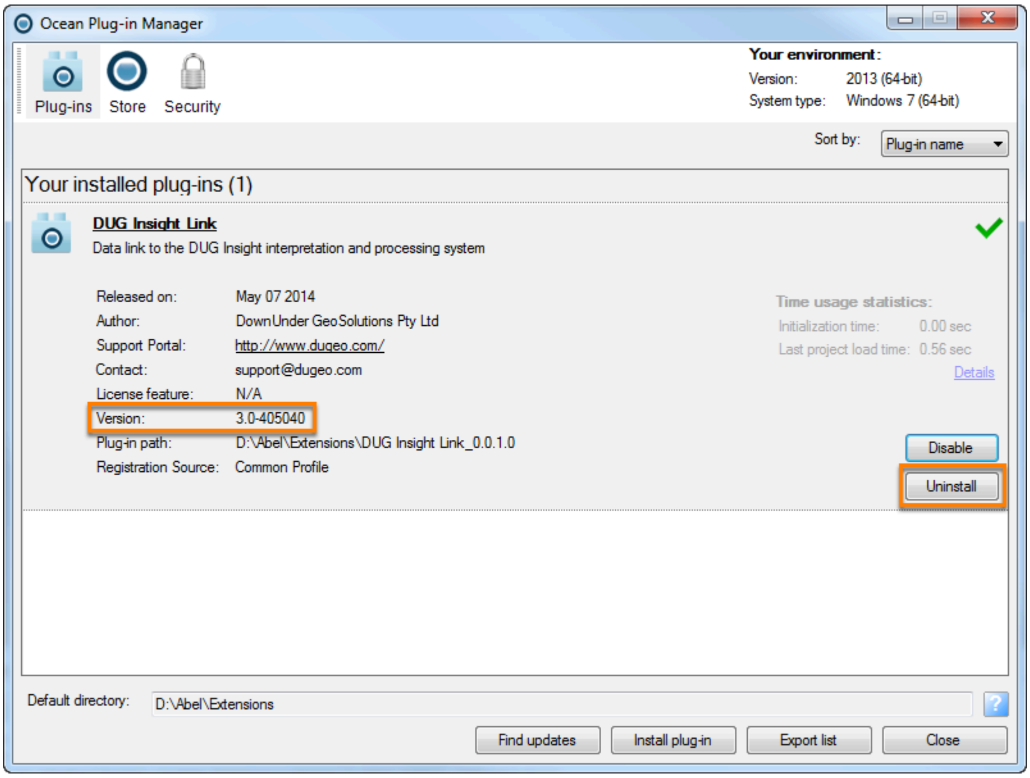


1. From your desktop, open the Start Menu and select **Control Panel**.
2. In the Control Panel, click **Uninstall a program** under Programs.
3. Locate the ***DUG Insight Petrel Link*** and double click to begin uninstalling.
4. Alternatively, select the link and click the **Uninstall/Change** button above the list of programs.

Uninstall via Ocean Plug-in Manager



1. In Petrel, open the **Help** menu.
2. Click on **Ocean Plug-in Manager**.

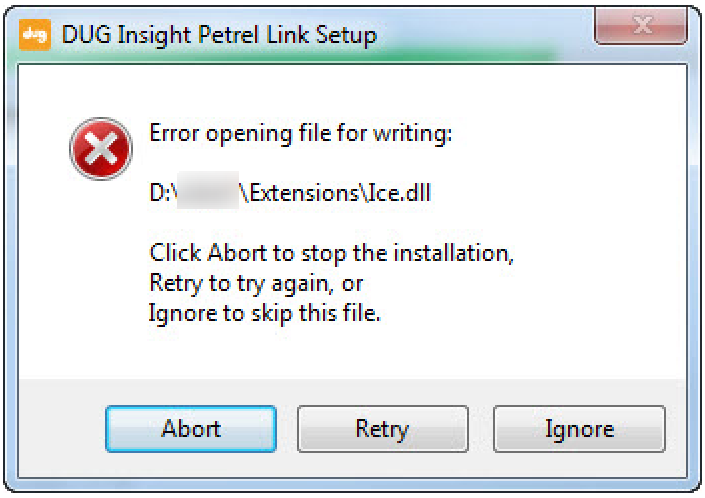


3. In **Ocean Plug-in Manager**, you should see the DUG Insight Link in the list of installed plug-ins.
4. Click on **DUG Insight Link** to expand the plug-in.
5. Check the version you would like to remove and click **Uninstall**.

"Error opening file for writing" when installing Petrel Link

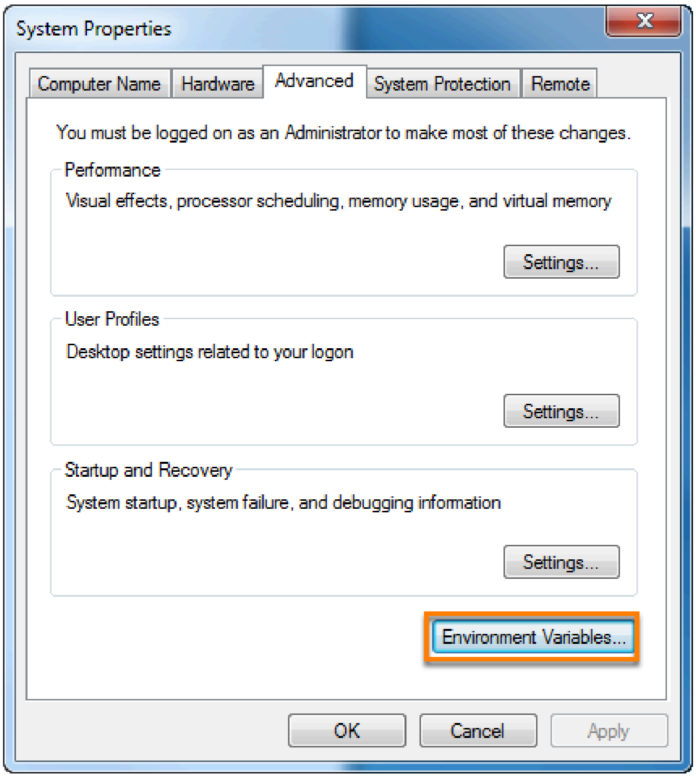
When you install Petrel for the first time, your machine will register *Ocean2013Home* at the installed location. Any plugins (including the Petrel Link) that you attempt to install will automatically look for that path.

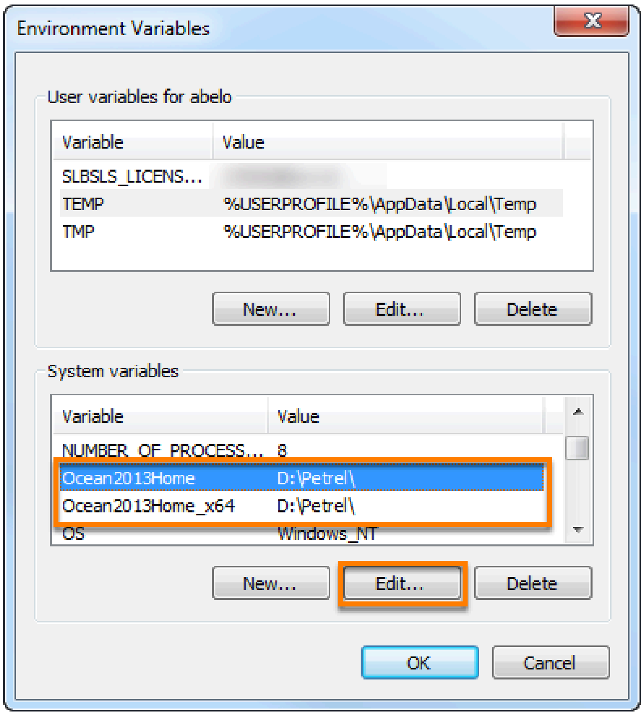
Error message



The error message above will appear when the system is unable to find the registered path to write the required files to. This can occur if you have copied/moved the installed folder to another location or if you have re-installed Petrel at a different path.

Changing the environment variable





To changed the registered path:

1. From the **Start** menu, open the **Control Panel**.
2. Click on **System and Security** and then click on **System**.
3. On the left sidebar, select **Advanced system settings**.
4. In the **System Properties** window, click on **Environment Variables**.
5. Look for the *Ocean2013Home* system variables and select **Edit**.
6. Type the path where Petrel is now located and click **OK**.
7. Run the installation again.

How do I send horizon attributes / amplitude extractions to Petrel?

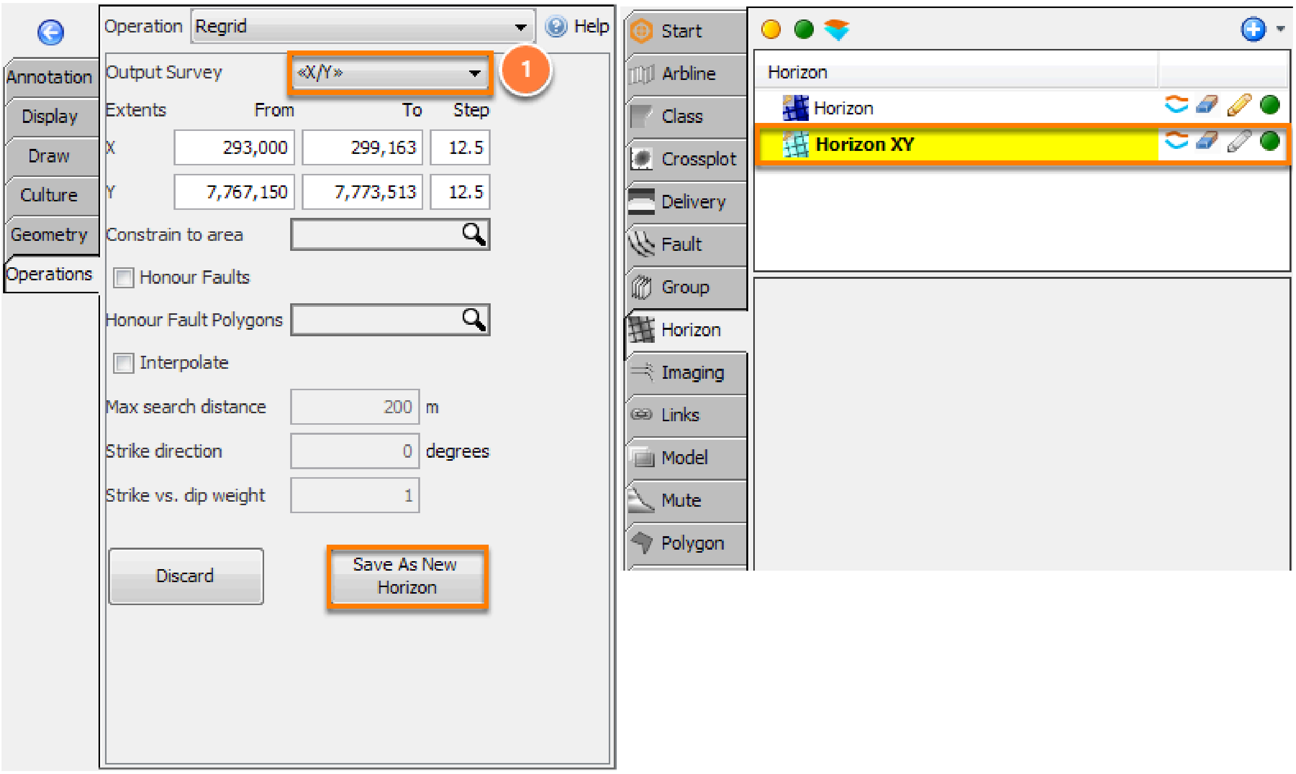
Horizon attributes can be sent to Petrel directly using the Petrel Link (see *Insight and Petrel Overview*) by following the steps below.

When a horizon attribute such as an extracted amplitude is generated, the resulting horizon or horizon property is not defined in time or depth (see *Horizon Amplitude Extraction* and *Single-Horizon Amplitude Extraction*). It can only be viewed in Map View. However, the Petrel Link is only able to transfer the elevation aspect of a horizon (i.e. TWT or TVDSS property). It is important to recognise the distinction between horizons (in IL/CL) and surfaces (in X/Y) in Petrel.

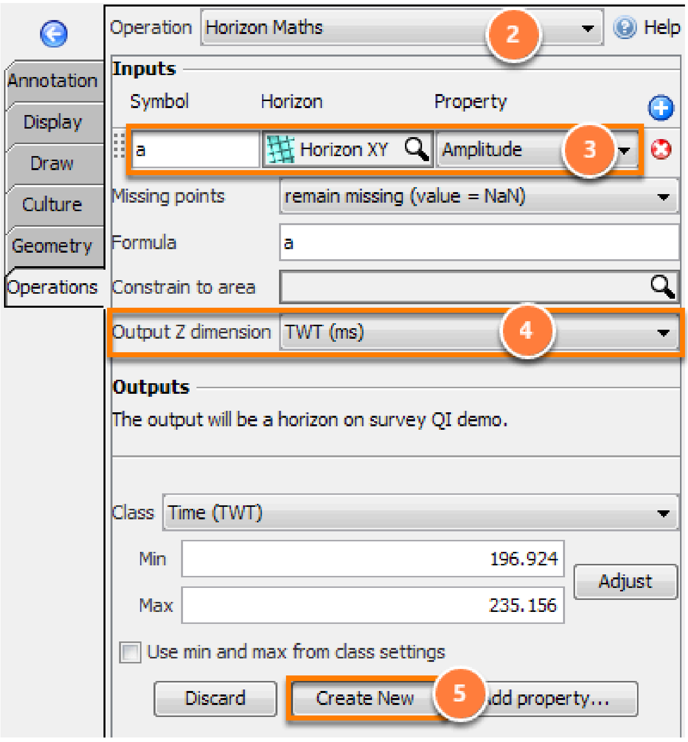
In order to send horizon attributes (in this example, the extracted amplitude) to Petrel, you can assign the amplitude values as a TWT property, and then transfer it to Petrel as a regular surface.

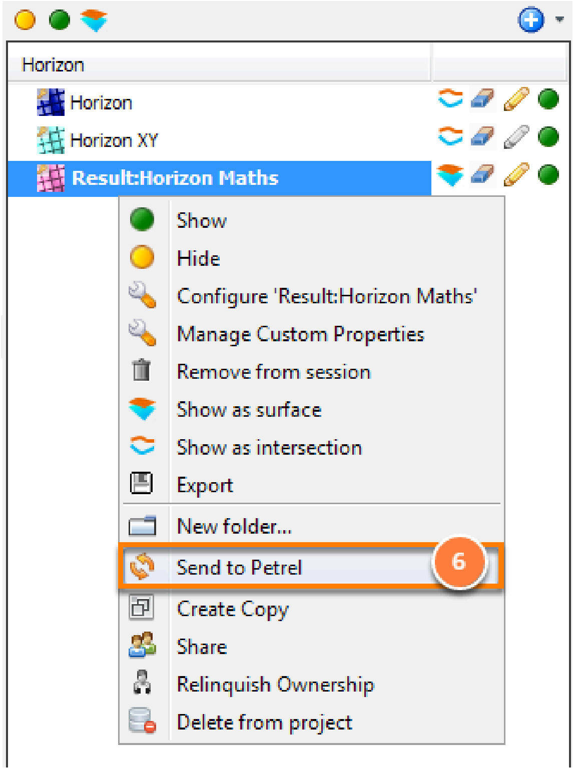
Workflow

1. First, regrid the horizon to X/Y and choose to **Save As New Horizon** (see *Regrid*). A new horizon will be created in the **Horizon** tab with the suffix 'XY' (i.e. *Horizon XY*).



2. Then, use the **Horizon Maths** operation to assign a Z dimension to the amplitude values of the new horizon (see *Horizon Maths*).
3. Select the new XY horizon and input property as **Amplitude**.
4. Assign the Output Z dimension as **TWT**.
5. Click **Run** and select to **Create New** horizon. A new horizon will be created in the Horizon tab (*Result:Horizon Maths*).





6. Right click on the resulting horizon and select **Send to Petrel**.

Once transferred to Petrel, you can change its template to suit amplitude values. It would then be possible to drape this regular (amplitude) surface over the equivalent TWT horizon in Petrel.

Why is the Petrel Link showing me a different project after connecting?

Question: I have the Petrel Link installed and I managed to connect to Petrel from Insight, but the project that appears is not the one that I have opened. Why is this happening and how do I connect to the correct project?

Answer: The Petrel Link will automatically detect and connect to an open Petrel project. However, it can be confused if multiple Petrel projects are open at the same time. In this case, it is best to only have one Petrel project open at a time.

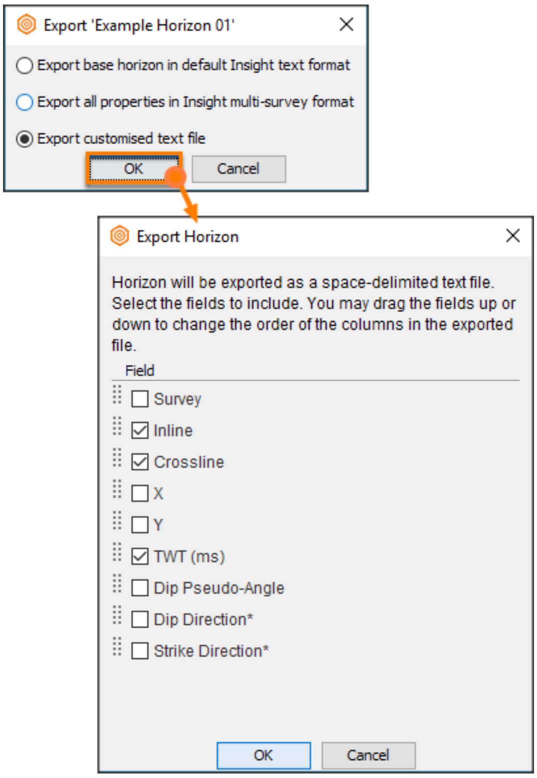
How to get interpretation into Petrel

This is a quick reference guide of the formats best suited to transfer data into Petrel.

Insight also supports connections to Petrel using the Petrel Link. This is a separate module that enables users to establish a bi-directional link to view, edit and transfer objects between Insight and Petrel. For more information, see *Insight and Petrel Overview*. If you are interested in attaining this module, contact our sales team at sales@dugeo.com.

Tip: You can also send interpretations to Petrel by using Insight's context menu, see *Send an Item to Petrel*.

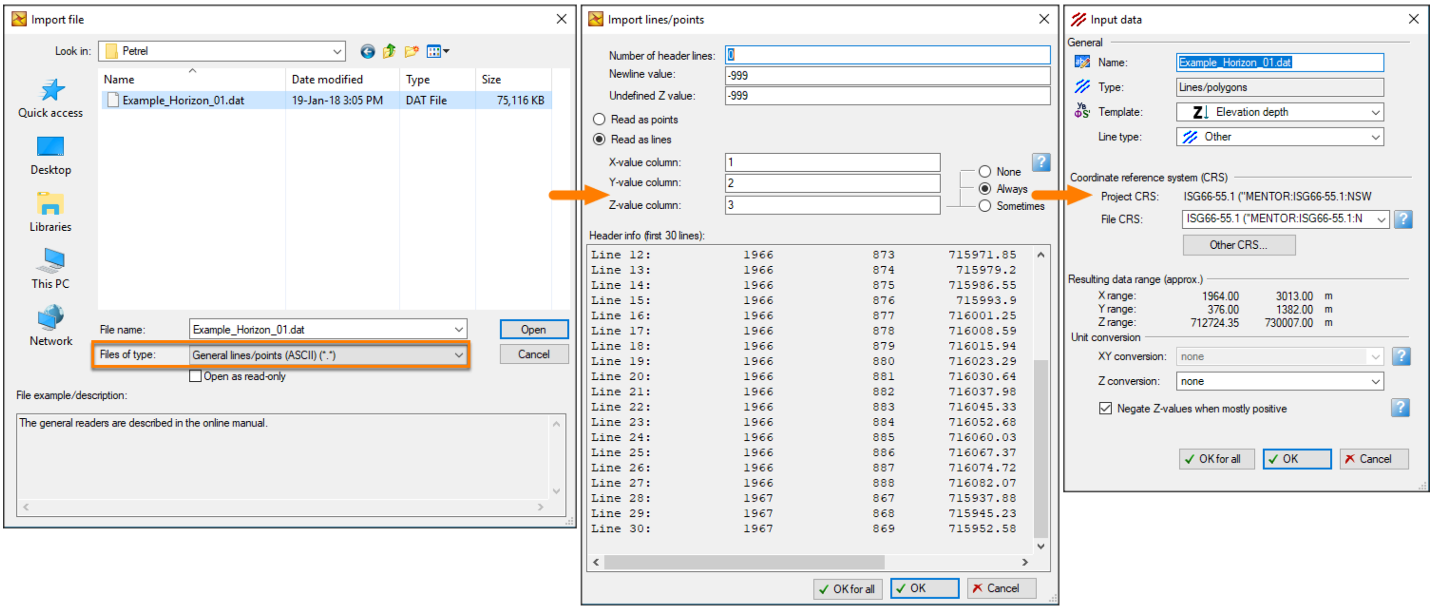
Horizons into Petrel



Horizons in Insight can be exported as a simple customised text file with columns (X/Y/IL/CL/Z or X/Y/IL/CL/Attribute) that can then be modified for import into Petrel.

1. In **Control Panel**, **Horizon** tab, right click horizon
2. Choose **Export>>Export customized text file**.
3. Select relevant data columns.

For more information, see Horizon Export Formats.

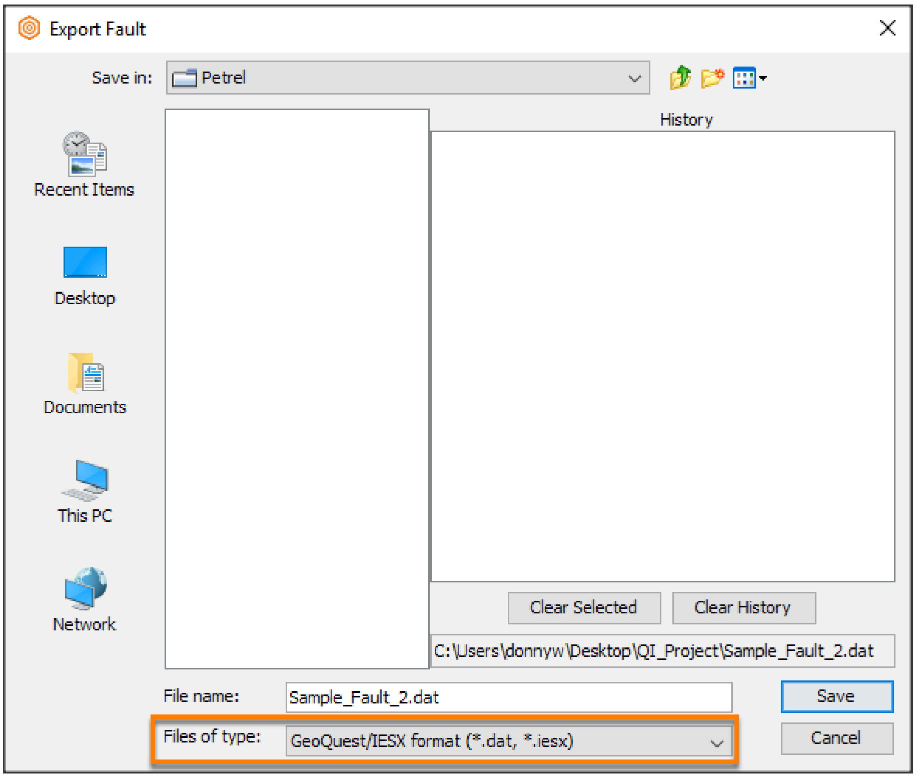


In Petrel, select **General lines/points (ASCII)** format when importing a horizon.

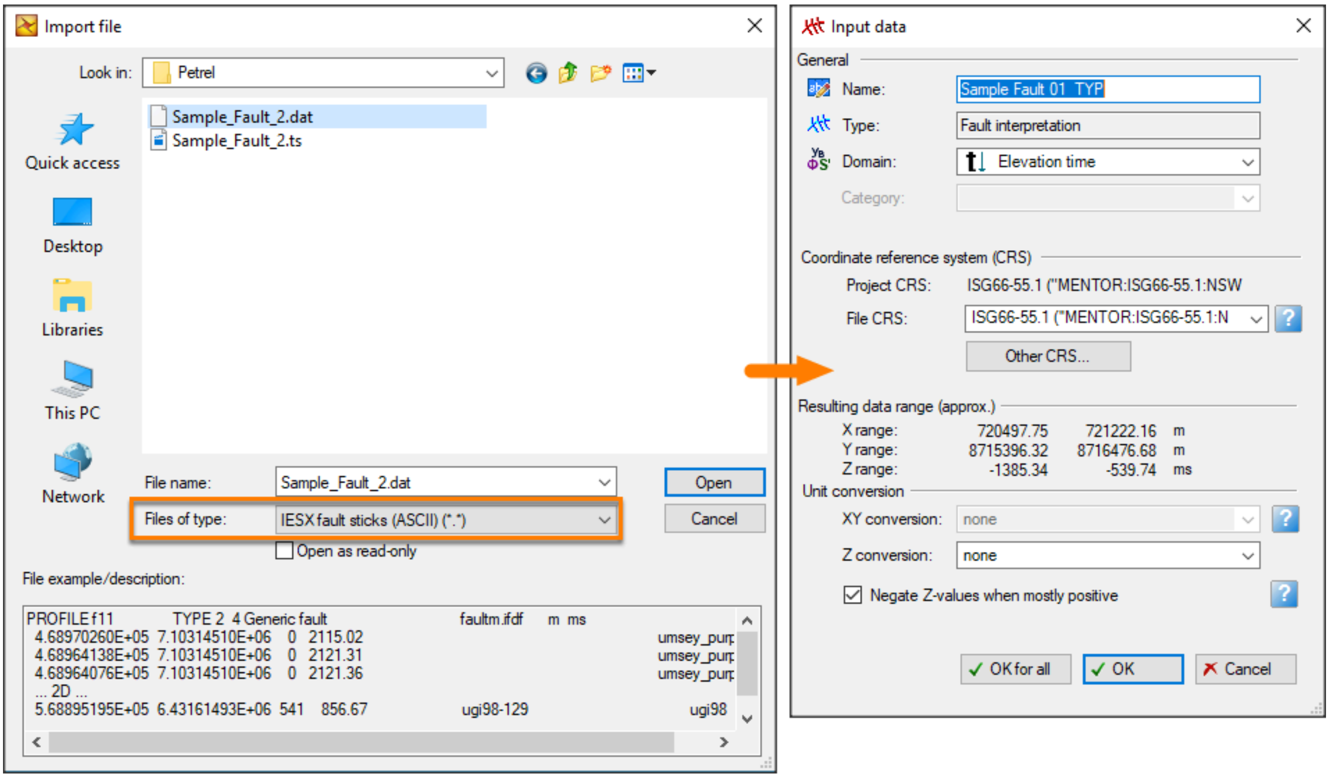
Make sure your columns are mapped properly, and to select the correct Z data type and Coordinate Reference System (CRS).

See also *How do I send horizon attributes / amplitude extractions to Petrel?*

Faults into Petrel



Fault surfaces are exported from Insight in **Geoquest/IESX format**.



1. In Petrel, select the **IESX fault sticks (ASCII)** format when importing a file.
2. If you do not have a CRS defined, you will be prompted to choose a CRS or continue spatially unaware.
3. Make sure to select the correct Domain and CRS in the Input Data window.

Polygons into Petrel

```
PROFILE HorstBlockTopREs TYPE 3 Exported from DUG Insight  m  ms
  298915.1742 7778687.8660    0           2700.000000      HorstBlockTopREs            unspecified
  299213.0037 7778154.9081    0           2700.000000      HorstBlockTopREs            unspecified
  299542.1835 7777731.6769    0           2700.000000      HorstBlockTopREs            unspecified
  299855.6882 7777277.0951    0           2700.000000      HorstBlockTopREs            unspecified
  300467.0222 7776493.3335    0           2700.000000      HorstBlockTopREs            unspecified
  300717.8259 7776132.8032    0           2700.000000      HorstBlockTopREs            unspecified
  300827.5526 7775834.9738    0           2700.000000      HorstBlockTopREs            unspecified
  300733.5012 7775615.5206    0           2700.000000      HorstBlockTopREs            unspecified
  300764.8516 7775302.0159    0           2700.000000      HorstBlockTopREs            unspecified
  300670.8002 7774627.9809    0           2700.000000      HorstBlockTopREs            unspecified
  300702.1507 7773797.1936    0           2700.000000      HorstBlockTopREs            unspecified
  300498.3727 7773295.5862    0           2700.000000      HorstBlockTopREs            unspecified
  300263.2442 7772872.3549    0           2700.000000      HorstBlockTopREs            unspecified
  300075.1414 7770959.9766    0           2700.000000      HorstBlockTopREs            unspecified
  300012.4405 7767871.9559    0           2700.000000      HorstBlockTopREs            unspecified
  300012.4405 7767103.8695    0           2700.000000      HorstBlockTopREs            unspecified
  299981.0900 7766727.6640    0           2700.000000      HorstBlockTopREs            unspecified
  292739.1328 7766790.3649    0           2700.000000      HorstBlockTopREs            unspecified
  292817.5090 7767464.3999    0           2700.000000      HorstBlockTopREs            unspecified
  292833.1842 7767918.9816    0           2700.000000      HorstBlockTopREs            unspecified
  292895.8851 7768561.6661    0           2700.000000      HorstBlockTopREs            unspecified
  292864.5347 7769345.4277    0           2700.000000      HorstBlockTopREs            unspecified
  292754.8080 7769894.0609    0           2700.000000      HorstBlockTopREs            unspecified
  292770.4833 7770662.1472    0           2700.000000      HorstBlockTopREs            unspecified
  292723.4576 7771132.4042    0           2700.000000      HorstBlockTopREs            unspecified
  292629.4062 7771445.9088    0           2700.000000      HorstBlockTopREs            unspecified
  292723.4576 7772527.4998    0           2700.000000      HorstBlockTopREs            unspecified
  292566.7053 7773217.2100    0           2700.000000      HorstBlockTopREs            unspecified
  292535.3548 7773687.4670    0           2700.000000      HorstBlockTopREs            unspecified
  292425.6282 7773953.9460    0           2700.000000      HorstBlockTopREs            unspecified
  292472.6539 7774251.7754    0           2700.000000      HorstBlockTopREs            unspecified
  293021.2870 7774769.0580    0           2700.000000      HorstBlockTopREs            unspecified
  293083.9879 7775051.2122    0           2700.000000      HorstBlockTopREs            unspecified
```

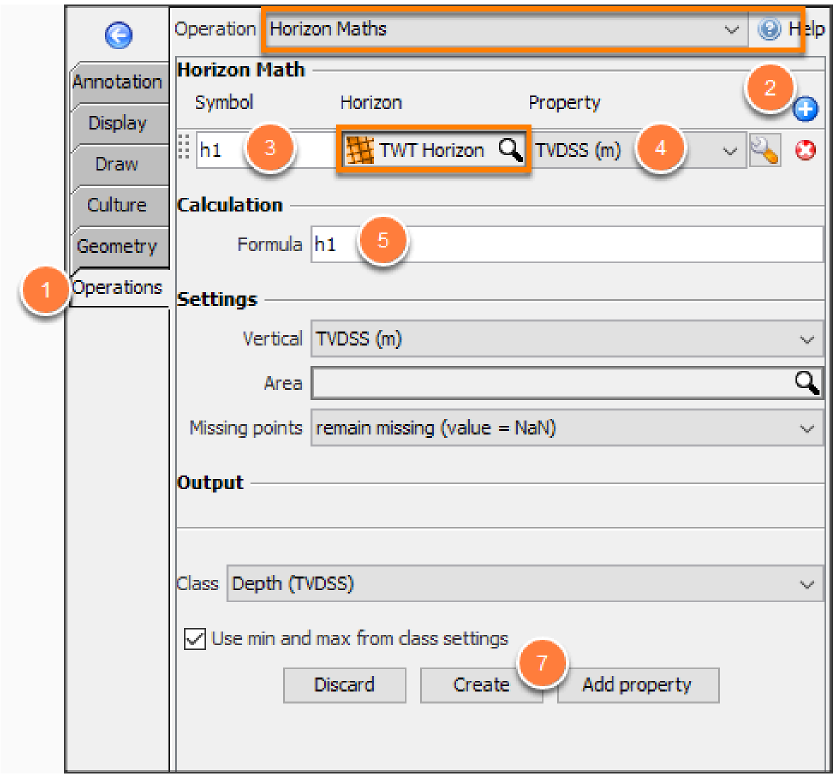
1. In DUG-Insight - Export polygons in **IESX/Geoquest format**
2. Importing into Petrel - select **IESX Fault Polygon** and choose **Generic boundary polygon**

How do I transfer an on-the-fly horizon to Petrel?

To transfer an on-the-fly horizon (e.g. TWT in depth, extracted amplitudes) to Petrel, first output the values to a new horizon.

This example shows how to transfer TVD values from on-the-fly depth-conversion to Petrel.

Note: Ensure a velocity model is configured for depth conversion (see *How it Works: Time-Depth Conversion*). The Petrel link must be connected to transfer horizons and/or faults (see *Insight and Petrel Overview)*.

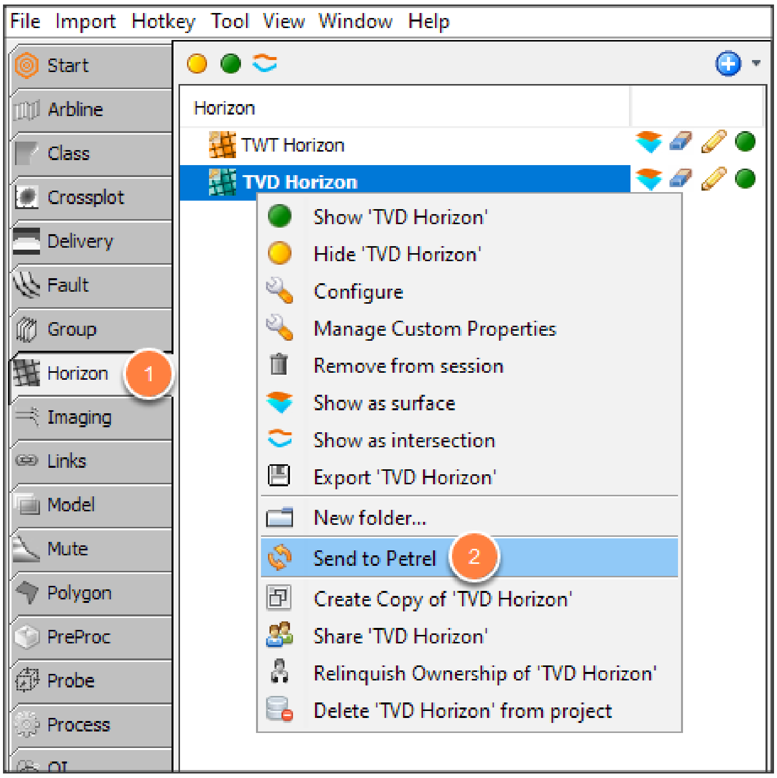


Use horizon maths to output the depth values to a new horizon.

1. In the **Map View > Operations tab**, choose *Horizon Maths*.
2. Add a horizon row.
3. Use the default symbol: '**h1**'.
4. Select the **TVDSS** property.
5. Use the formula: **h1**
6. Click **Calculate**.
7. Click **Create**
8. Rename the new horizon, e.g. “TVD Result”

Push the new horizon to Petrel using the link.

Send to Petrel



1. In the **Horizon** tab, right-click the horizon: “TVD Result”.
2. Select **Send to Petrel**.

How do I transfer depth-converted faults to Petrel?

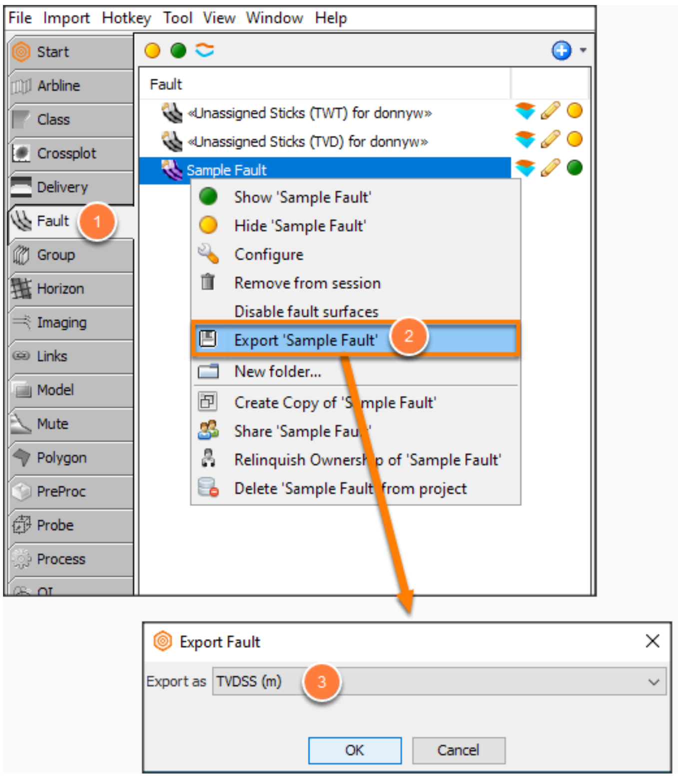
Depth-converted faults can be transferred to Petrel, e.g. **TWT** faults as **TVD**, or **TVD faults** as **TWT**.

The process:

1. Export the faults in the converted domain
2. Import the faults back to Insight
3. Transfer the imported faults to Petrel

Note: Ensure a velocity model is configured for depth conversion (see *How it Works: Time-Depth Conversion*). The Petrel link must be connected so that horizons and/or faults can be transferred (see *Insight and Petrel Overview*).

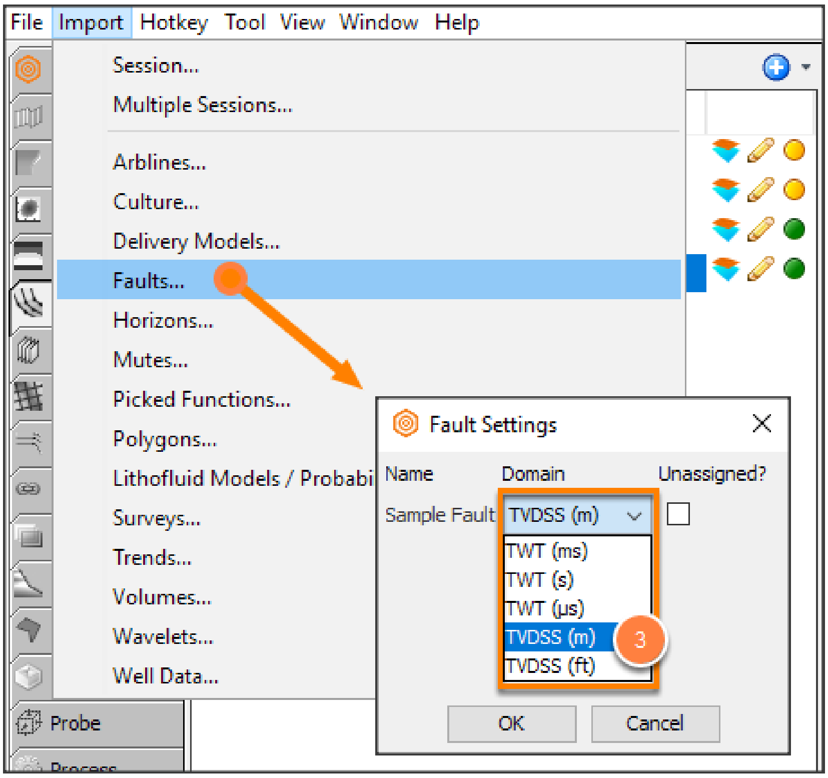
Exporting Faults to depth



1. Go to the **Fault** tab and right-click the fault.
2. Select **Export 'Fault'**.
3. In the **Export as** dropdown box, select **TVDSS (m)**.

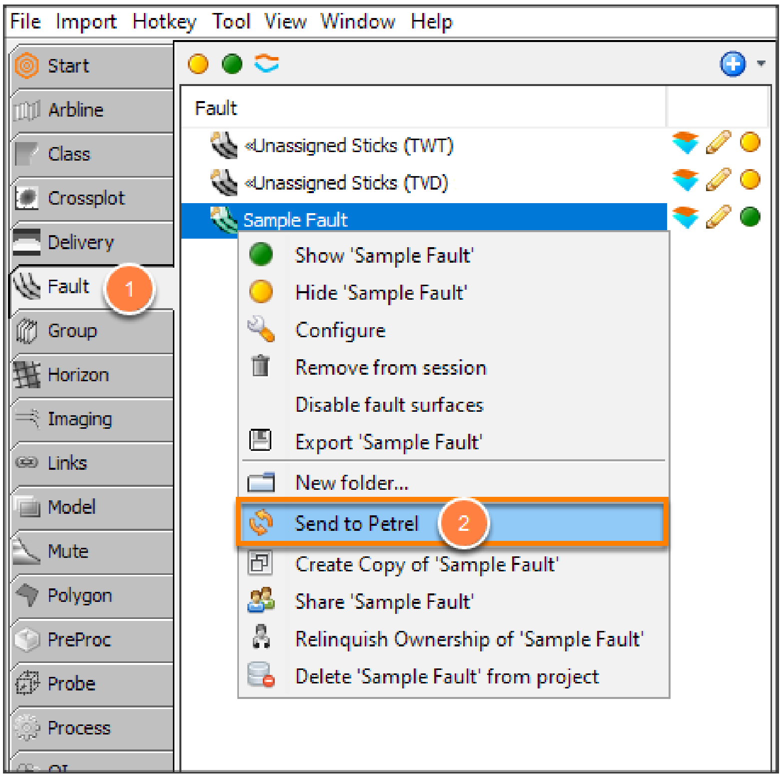
4. Click **OK.**

Importing Converted Faults



1. In the **Import** menu and select **Faults....**
2. Select the exported fault file from the previous step
3. Select the correct domain from the previous step, e.g. **TVDSS (m)**
4. Click **OK**.

Send to Petrel



1. Go to the **Fault** tab and right-click on the imported fault.
2. Select **Send to Petrel**.

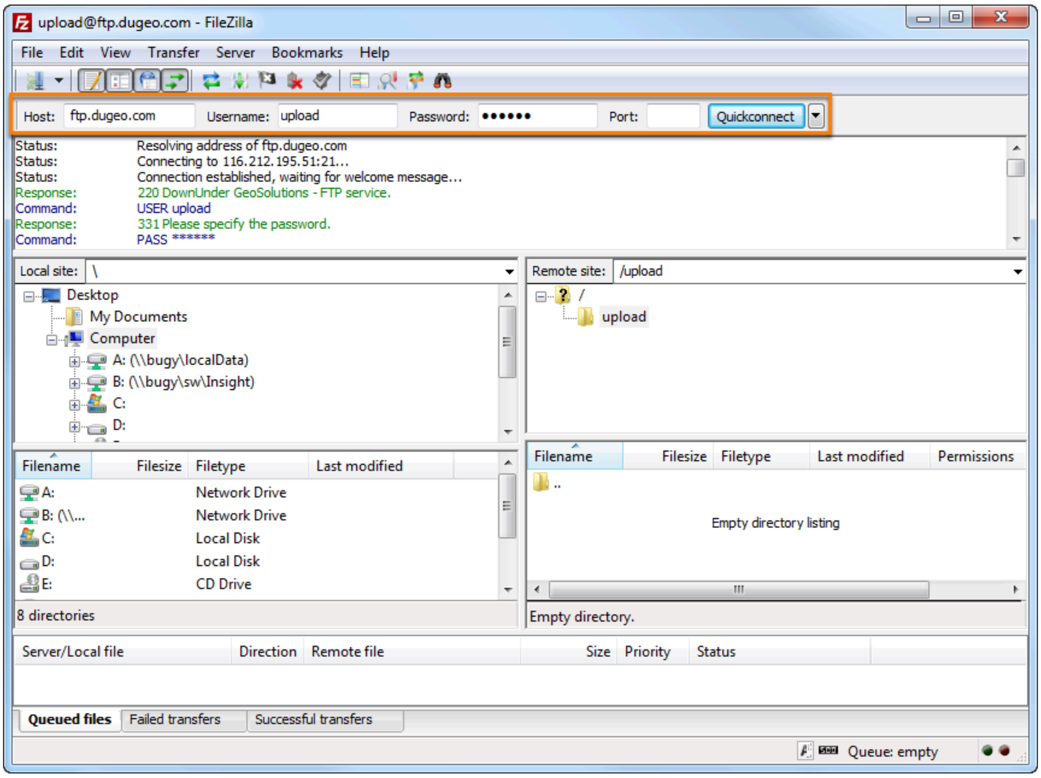
Miscellaneous

How to upload to our FTP server?

If you are having problems converting SEG-Y data into dugio format, or if we require more information from you regarding your database, it might be better for you to send us your data so that we can get a better look at what is causing the error. If your data is too big to send via email, you can upload it to our FTP either by using an FTP client (i.e. FileZilla) or by using Internet Explorer. Follow the steps below for both methods.

Note: We understand that the contents of any data sent to us are confidential and intended for testing and bug fixing purposes only. Under no circumstances will the data be shared or distributed to anyone other than those necessary to solve your problem. You have our promise that we will safeguard your data to the best of our ability. Once we have solved the problem and closed the support ticket, your data will be removed from our system.

Uploading via FTP client



If you have an FTP client like FileZilla, transferring files is a simple three-step process.

1. Open FileZilla from your desktop or Start menu.
2. Type in the following at the top and click **Quickconnect**.
   - **Host:** ftp.dugeo.com

- **Username:** upload
- **Password:** upload

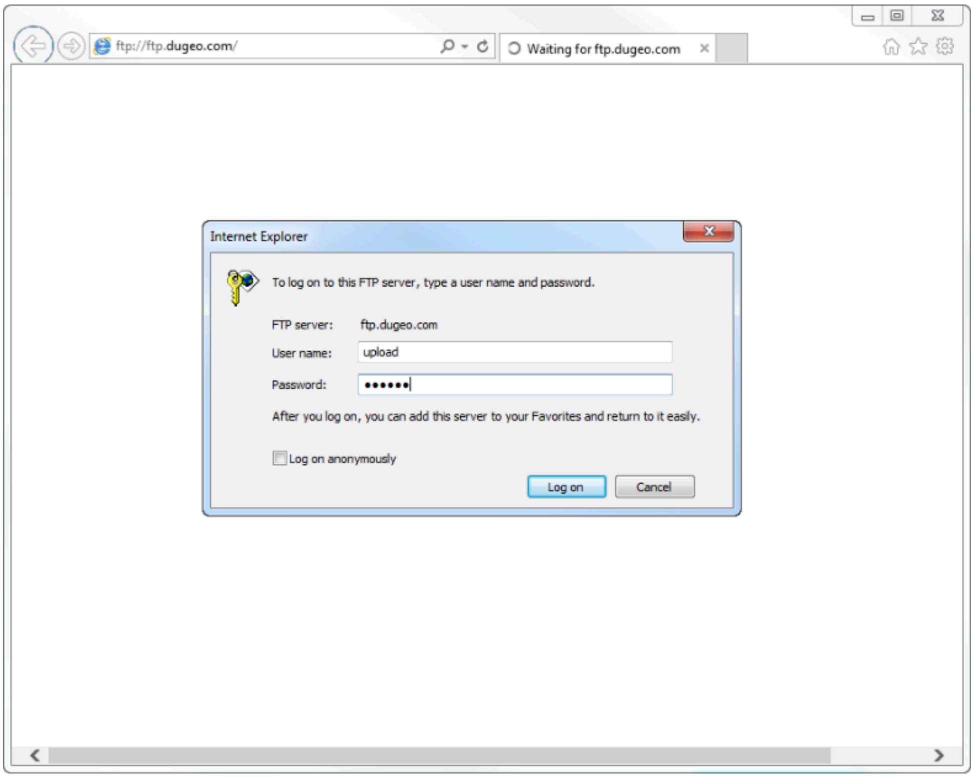
3. Drag and drop the relevant files into the upload folder.

Send us an email at support@dugeo.com to let us know that you have successfully uploaded the file(s) into the folder, and include the name of the file(s) in the email.

Note: The folder will still appear empty after transferring the files into the folder. You can verify that the transfer was successful by looking at the **Successful transfers** tab at the bottom of the window.

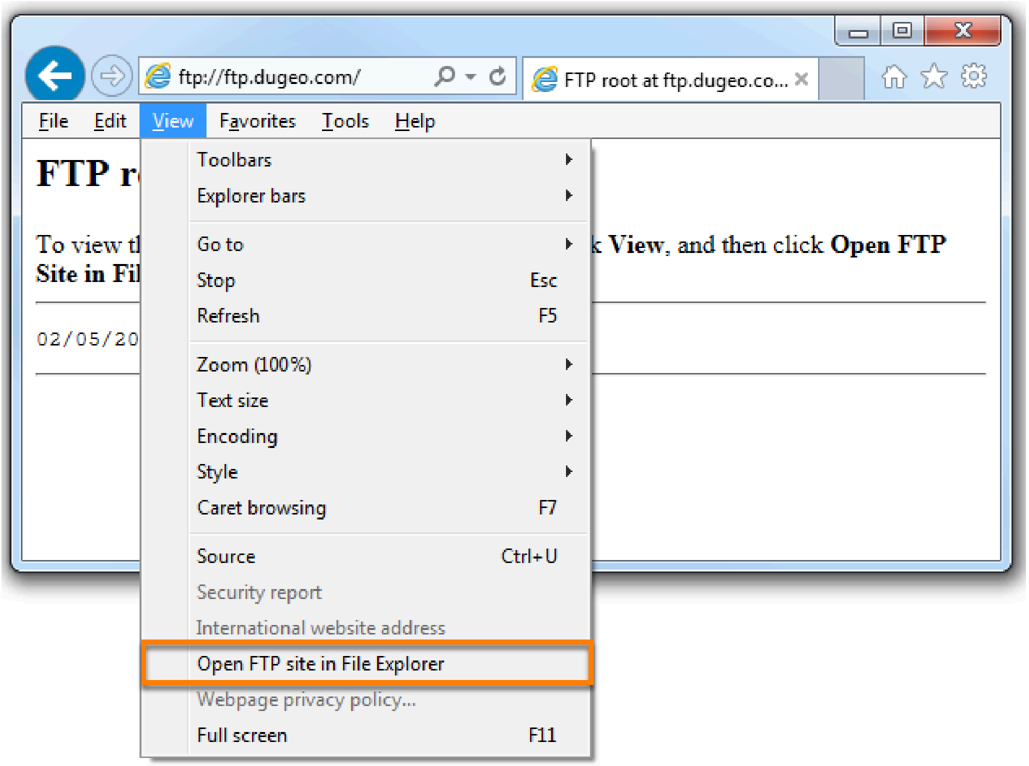
After the problem has been identified, we will revert back to you and send you the usable data. See *How to download, unzip and load volumes and surveys into Insight?* to download and load the files into your Insight project.

Uploading via Internet Explorer

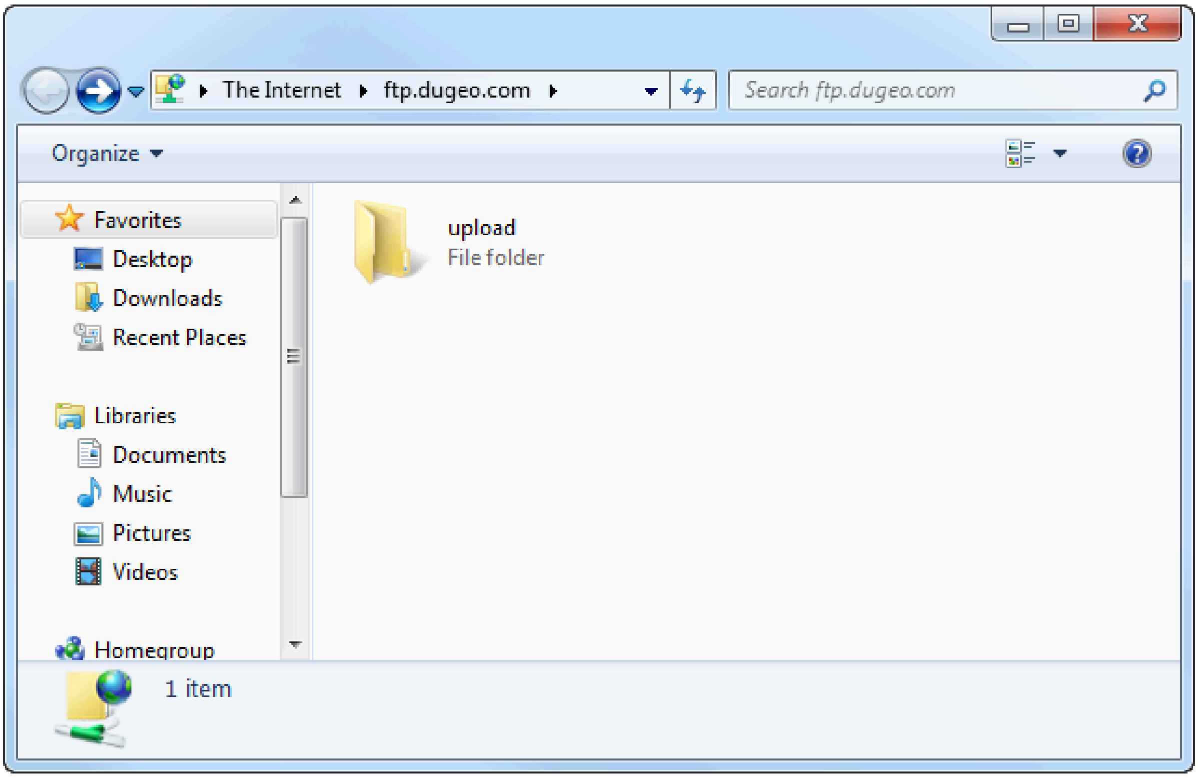


Alternatively, you can also access our FTP using Internet Explorer. This method does not require you to have an FTP client installed.

1. From your desktop or Start menu, open Internet Explorer.
2. In the address bar, type ftp://ftp.dugeo.com and press **Enter** (or click on the link).
3. Enter "upload" as the username and password, and click **Log on**.



4. The FTP root page should appear with instructions to press **Alt**, click **View** and select **Open FTP Site in File Explorer**.
5. The File Explorer window will appear and you are required to log in again.
6. Type in the same username and password as before and click **Log on**.



7. Open the **upload** folder

8. You can now drag and drop your files into the upload folder.
9. Send us an email at support@dugeo.com to let us know that you have successfully uploaded the file(s) into the folder, and include the name of the file(s) in the email.

After the problem has been identified, we will revert back to you and send you the usable data. See *How to download, unzip and load volumes and surveys into Insight?* to download and load the files into your Insight project.

How to download, unzip and load volumes and surveys into Insight?

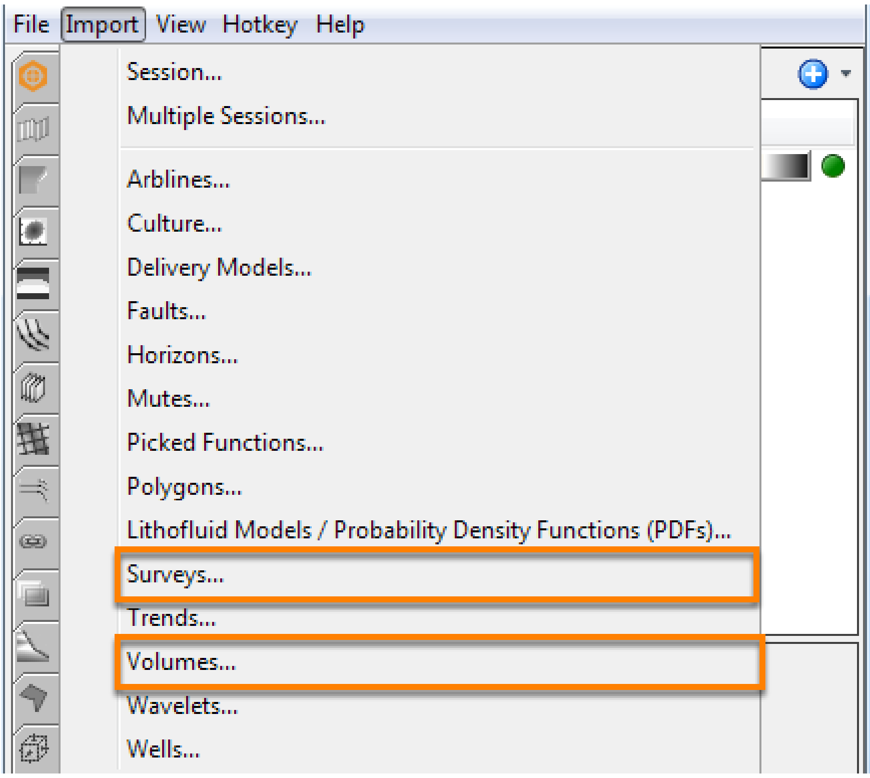
If you have sent us your data for testing and bug fixing (*see How to upload to our FTP server?*), we will revert to you once we have identified and solved the problem. A download link to the .zip file containing the usable data will be sent to you. Follow the steps below to download, unzip and load the data in Insight.

Note: We understand that the contents of any data sent to us are confidential and intended for testing and bug fixing purposes only. Under no circumstances will the data be shared or distributed to anyone other than those necessary to solve your problem. You have our promise that we will safeguard your data to the best of our ability. Once we have solved the problem and closed the support ticket, your data will be removed from our system.

Download and extract

1. Download the .zip file as provided by DUG's support team.
2. When the download is completed, locate the .zip file from the download folder.
3. Double click and extract the contents using WinZip or any similar software.
4. If you have been sent a volume, the *.zip* file should contain a folder with the *.dugio* extension. Copy or extract the folder into your project directory's 100sei folder.
5. If you have been sent a survey, the *.zip* file should contain a file with the *.survey* extension. Copy or extract the file into your project directory folder.
   Note: You can choose a different location for the survey file. However, we generally recommend placing it in the project directory for convenience.

Loading in Insight



Import survey:

1. From Insight's Control Panel, click on the **Import** menu and select **Surveys**.
2. Navigate to the location you copied the survey file and click **Open** to import.
3. The survey will now be loaded in Insight.

Note: You must add a survey before adding any volumes.

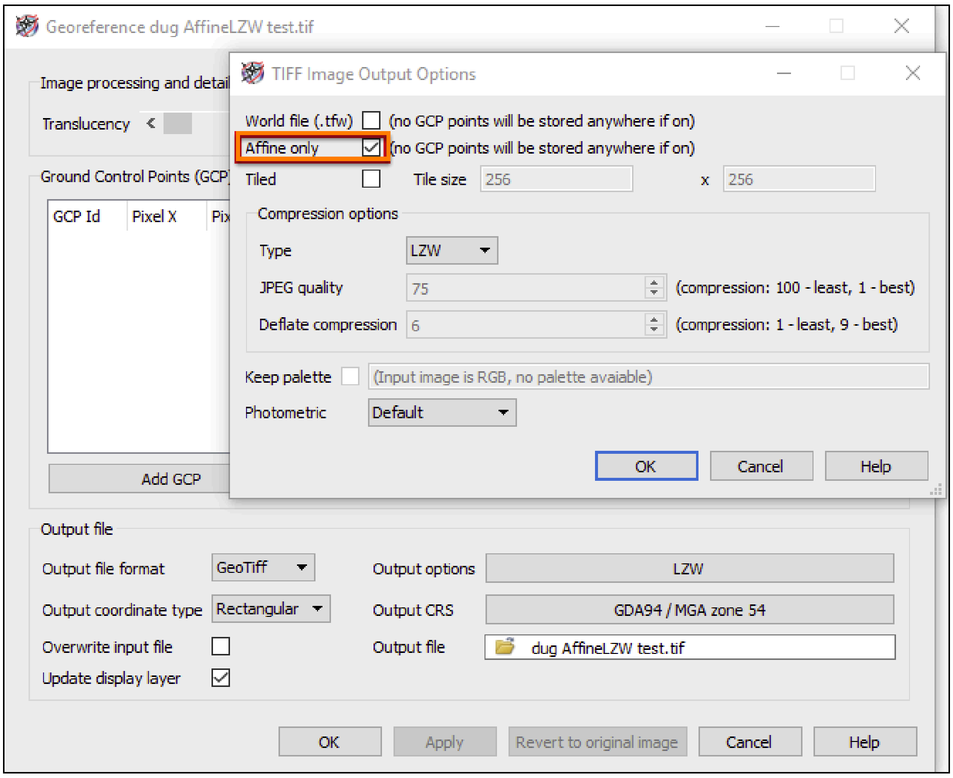
Import volume:

1. To load the volume, click on the **Import** menu and select **Volumes**.
2. Navigate to the .dugio volume you downloaded and click **Open** to import.
3. The volume will now be loaded in Insight. Click on the **View** menu and select any section view to display the volume on the survey.

Exporting GeoTiff files from Petrosys into Insight

GeoTiff files can be exported directly from Petrosys to be loaded into Insight (see *Load Culture Files* and *Import Culture Files*).

When exporting the GeoTiff file from Petrosys, make sure to select the **Affine only** option in the **TIFF Image Output Options** window.



This will allow the embedded positional information (coordinates) of the GeoTiff file to be exported from Petrosys and loaded into Insight alongside the GeoTiff file.

How do I get my data stored in a third-party database into DUG Insight?

There is limited support in DUG Insight for loading directly from certain databases. Please contact us at *support@dugeo.com* to find out about your specific situation.

Generally, the *Petrel Link* and *Kingdom Link* can be used to transfer data into an Insight project. These are available via additional modules. Petrosys projects can use the Petrosys Link, *please contact Petrosys for more information on this feature*. The Petrosys link is NOT supported via the DUG support team.

Volume data can be exported as SEG-Y files (see *Exporting a Volume to SEG-Y*); horizons or faults as GeoQuest CardImage 7 files; well data can be bulk exported as OWX files and then loaded into DUG Insight. A full list of formats Insight supports can be found *here*.

Where are the old Insight user manuals?

Should you require any assistance with an older version of Insight, you can download the PDF of the manuals by clicking on the links below.

- Insight 2.9 — http://downloads.dugeo.com/insight/DUG+Insight+2.9+User+Manual.pdf
- Insight 3.0 — http://downloads.dugeo.com/insight/DUG_Insight_3.0_User_Manual.pdf

Where can I find the manual for older versions of the software (e.g. Insight 3.1)?

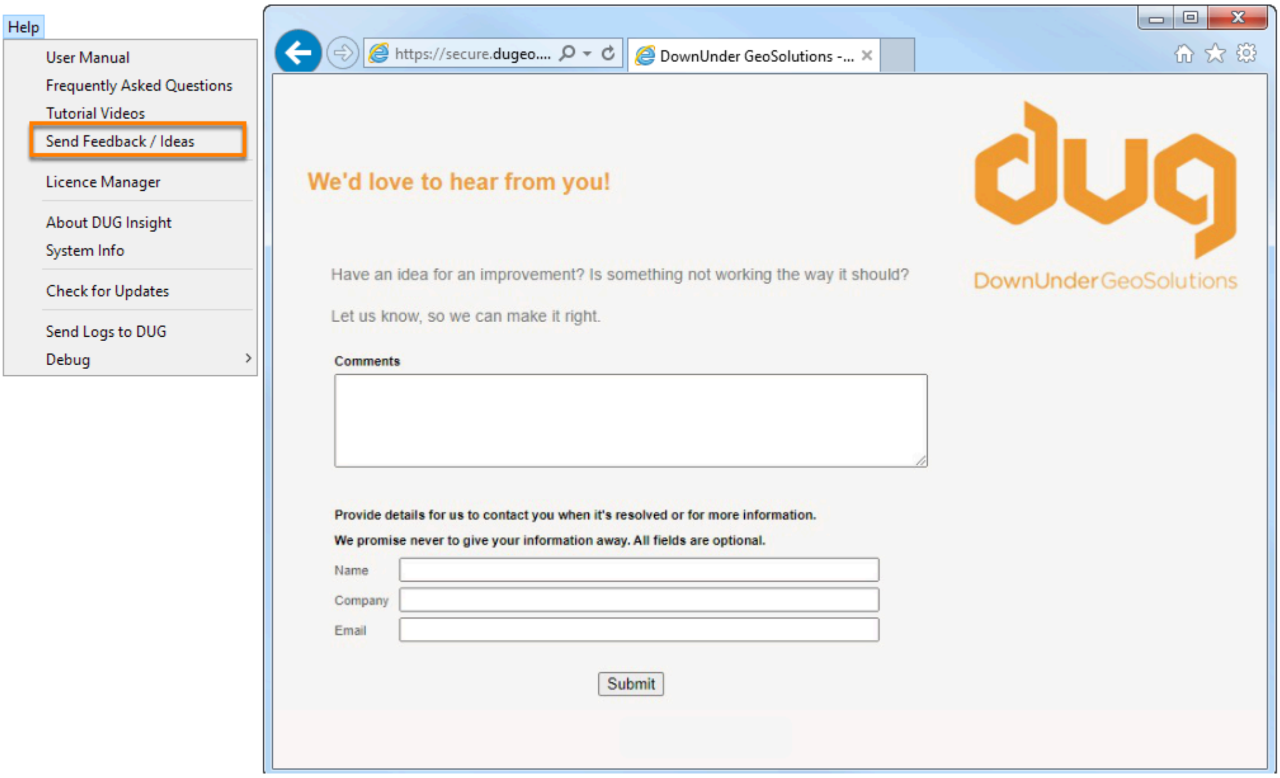
Older versions of the manual are available in PDF format for you to download and read (or print!). Please click on the link below for the correct version of the manual:

- DUG Insight 3.1
- DUG Insight 3.0
- DUG Insight 2.9

I have a great idea for Insight! How do I send DUG my suggestions?

If you have an awesome idea that would revolutionize Insight, or know of a feature that would drastically improve Insight's functionality, we would like to hear from you!

Please, tell us all about it!



1. In Insight's **Control Panel**, go to the **Help** menu.
2. Click **Send Feedback/Ideas**. This will open the online feedback form in your default browser.
3. In the online feedback form, type your suggestion or queries in the **Comments** box.
4. Type your **Name, Company** and **Email** for DUG to contact you. These are optional fields.
5. Click **Submit**.

Insight Error Messages

Display driver has stopped responding. What should I do?

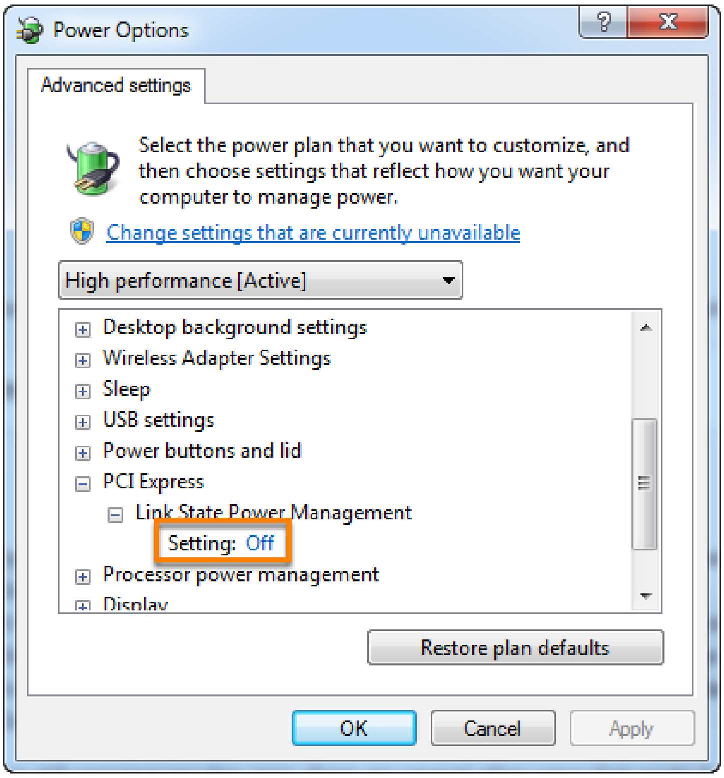
This is a common error message that occurs with any NVIDIA card on any Windows operating system. When the computer detects that your graphics card is not responding, the screen goes black for a few seconds followed by the pop-up box below, and the software utilising the graphics card would stopped working.

Error message

The cause of this error is not an obvious one. It could be a corrupted or outdated graphics card, power supply, ram or overheating. If this error persists, follow the steps below to troubleshoot.

Note: If, at any time, you are not familiar or are uncomfortable performing any of the solutions below, please consult your IT department.

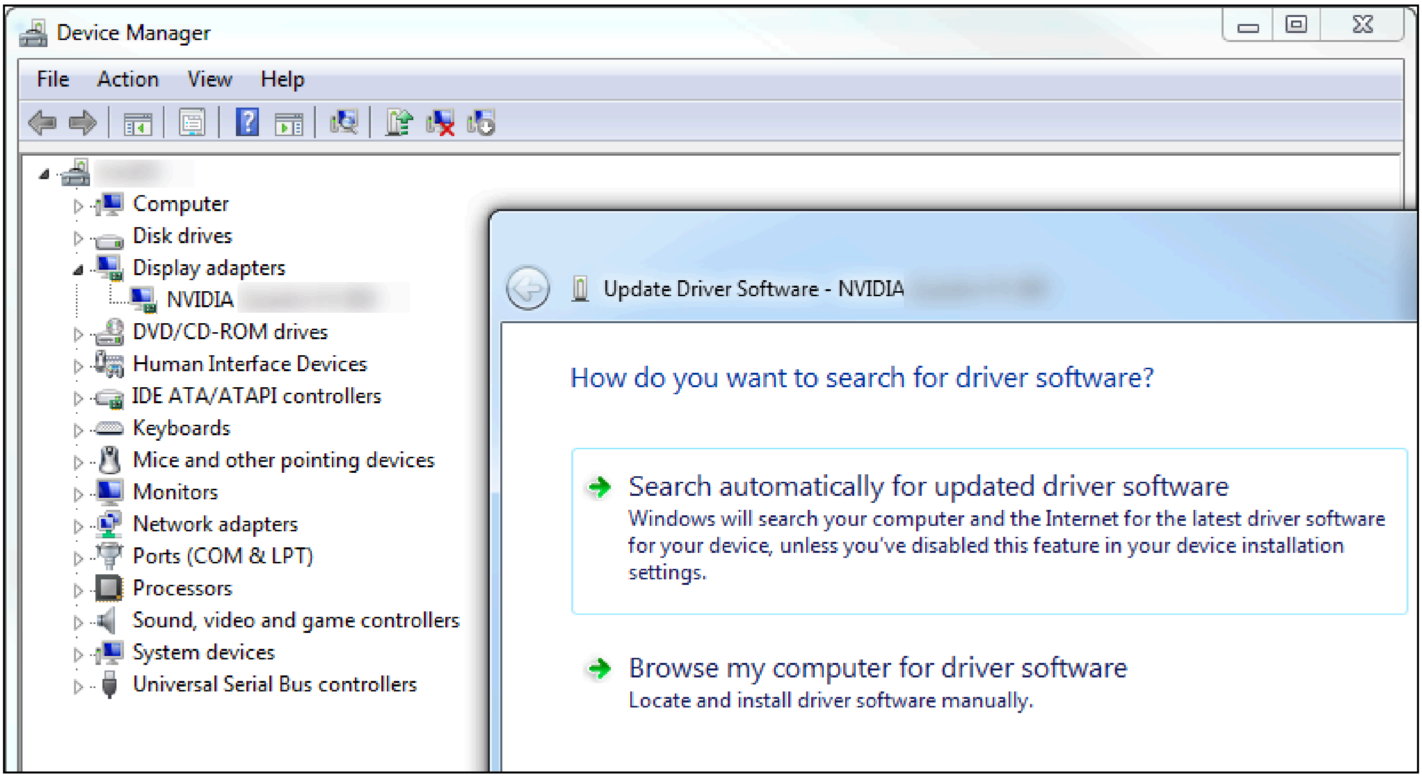
Turning off power management



1. Open your Control Panel from the Start menu and click on **Hardware and Sound**.
2. Click on **Power Options**.
3. At **Preferred plans**, click on **High performance** to choose it as your preferred plan.
4. Click on **Change plan settings** for High performance and select **Change advanced power settings** to display the advanced settings.
5. Scroll down and expand **PCI Express.** Turn off the **Link State Power Management** setting.
6. Click **Apply** to save the settings and click **Ok** to close the window.

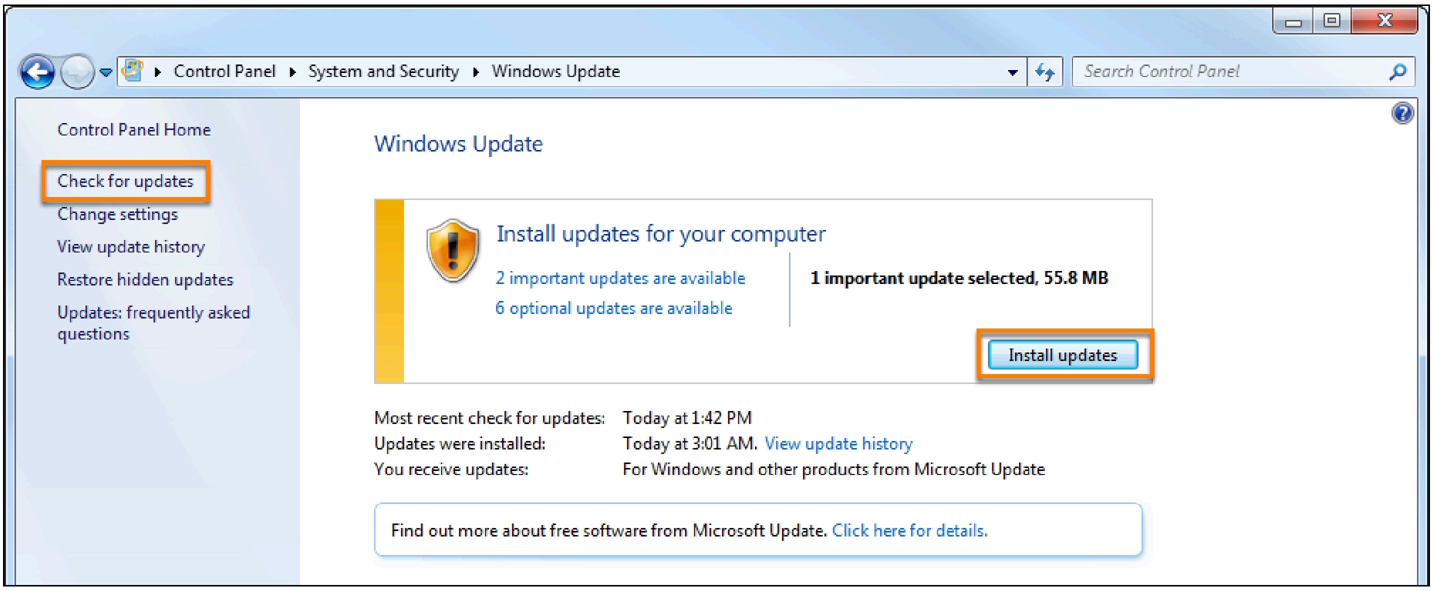
If you are still experiencing this error, there are several other ways that might fix the problem.

Update display driver



1. Open your Control Panel from the Start menu and click on **Hardware and Sound**.
2. Under **Devices and Printers**, click on **Device Manager**.
3. Expand the **Display adapters**. Right-click on the graphics card and select **Update Driver Software**.
4. Select **Search automatically for updated driver software**.
5. Update to the latest driver software.

Update Windows



1. Open your Control Panel from the Start menu and click on **System and Security**.
2. At **Windows Update**, click **Check for updates**.
3. If there are updates to be installed, click **Install updates** to begin installing.

RAM and overheating

If the error does not go away, it could be a faulty RAM or your display drivers are overheating.

- To check your RAM, simply remove all the RAM sticks except one and see if it fixes the problem. Repeat this until you have found the faulty RAM stick. If the error happens no matter which RAM stick you put in, that means the RAM is not the problem here.
- To check the temperature of your display driver, you can download any PC hardware monitoring program. If you find that the temperature increases rapidly when using a graphic intensive software, you can either try cleaning the driver's cooling fan or installing new fans.

Note: If you are not familiar or are uncomfortable performing any of the solutions above, please consult your IT department.

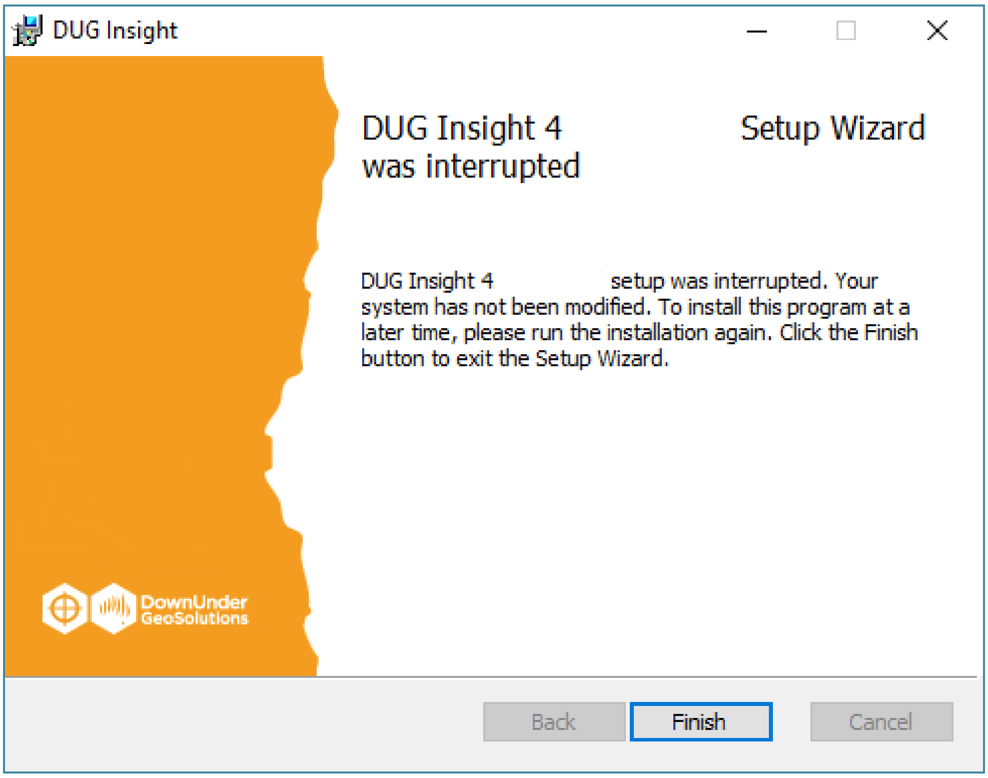
Genuine offer of assistance

If the problem persists, or if you have any questions, feel free to contact our support team at support@dugeo.com.

Installation of Insight failed! What should I do?

If the installation of Insight is unsuccessful, the error message below will be displayed.

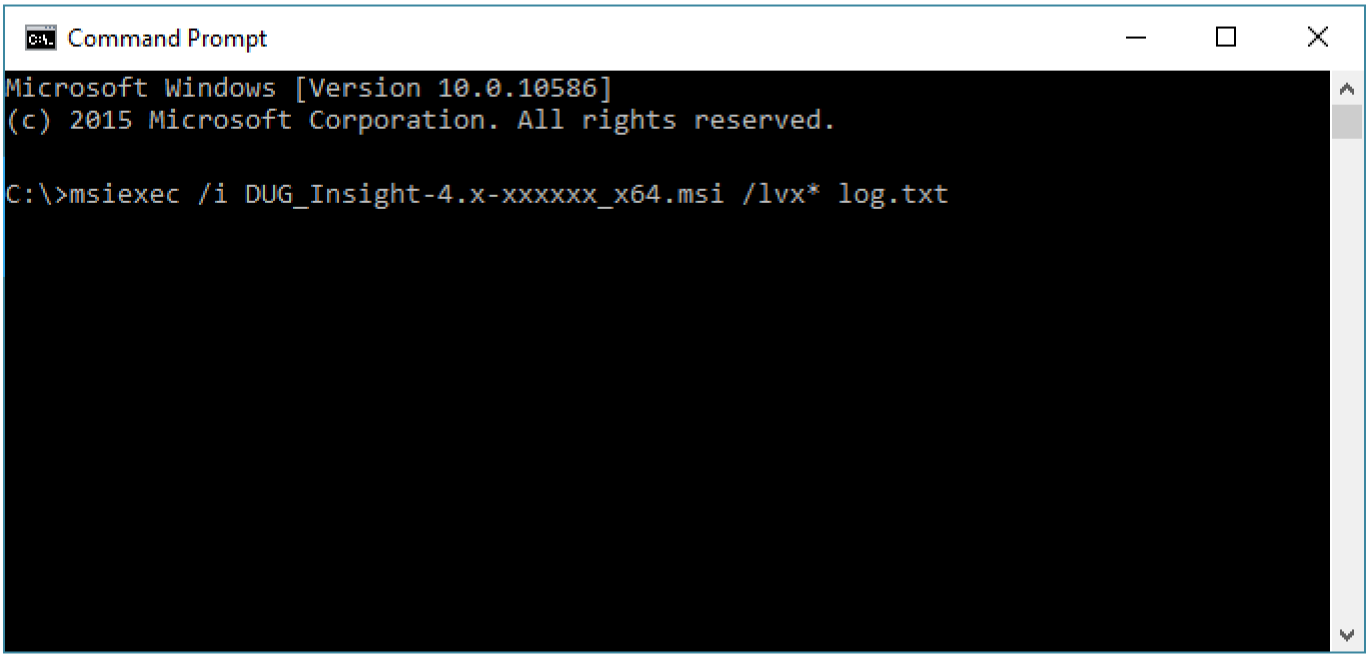
Error message



This error could be due to a number of reasons.

It would be a great help in diagnosing the problem if you could run the installer with extra logging enabled, as described below. This will create a log text file that will tell us exactly what was going on during the installation.

Installing with logging options

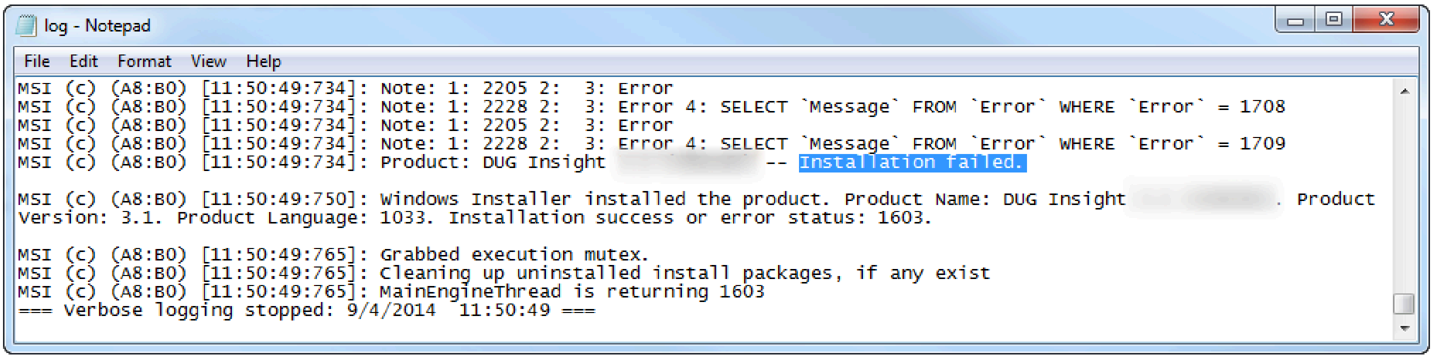


1. From the **Start** menu, type "cmd" and open the Windows Command Processor.
2. In the command window, navigate to the folder where the installation file (.msi) resides.
3. To run the installer with logging options, type the following:

```
msiexec /i DUG_Insight-4.x-xxxxxx_x64.msi /lvx* log.txt
```

Note: Change the version number as necessary.

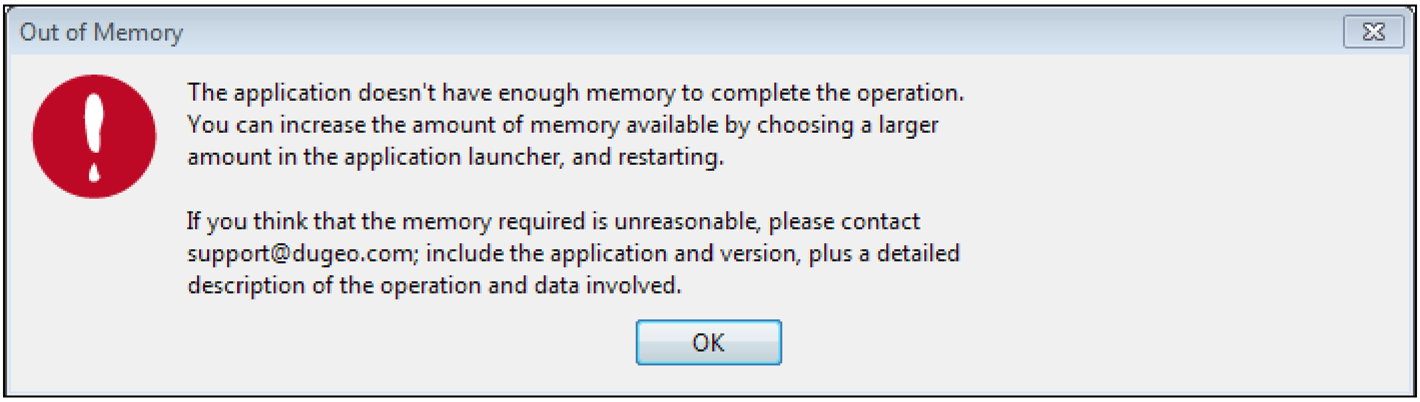
4. Press **Enter** and run the installation again. The log file will document all activity by the installer, including the moment it failed and the error code.



5. Send us a support ticket by emailing support@dugeo.com describing how and when the problem occurred, your system specifications, and attaching the log file.

Out of memory error

Error message



What it means

This error means that the memory you have initially assigned to Insight before opening a project is insufficient.

See *Allocating Memory* for more details on allocating memory for Insight.

Genuine offer of assistance

If the problem persists, or if you have any questions, please contact our support team at support@dugeo.com.

You may be asked to send data of your session from the Logs window for a clearer picture of what happened when the error occurred (see *Viewing and Sending Diagnostic Logs*).

Error launching Insight on 32-bit Windows machine

Error messages

If you are running on a 32-bit Windows machine, you might encounter the following errors when launching Insight:

- Error occurred during initialization of VM.
- Could not reserve enough space for object heap.
- Could not create the Java virtual machine.

What it means

On a 32-bit Windows operating system, you cannot reliably allocate more than approximately 700MB of memory for a single application in the DUG Insight Launcher. This is a fundamental design defect of 32-bit Windows. To solve this problem, reduce the memory allocation or upgrade to a 64-bit Windows operating system.

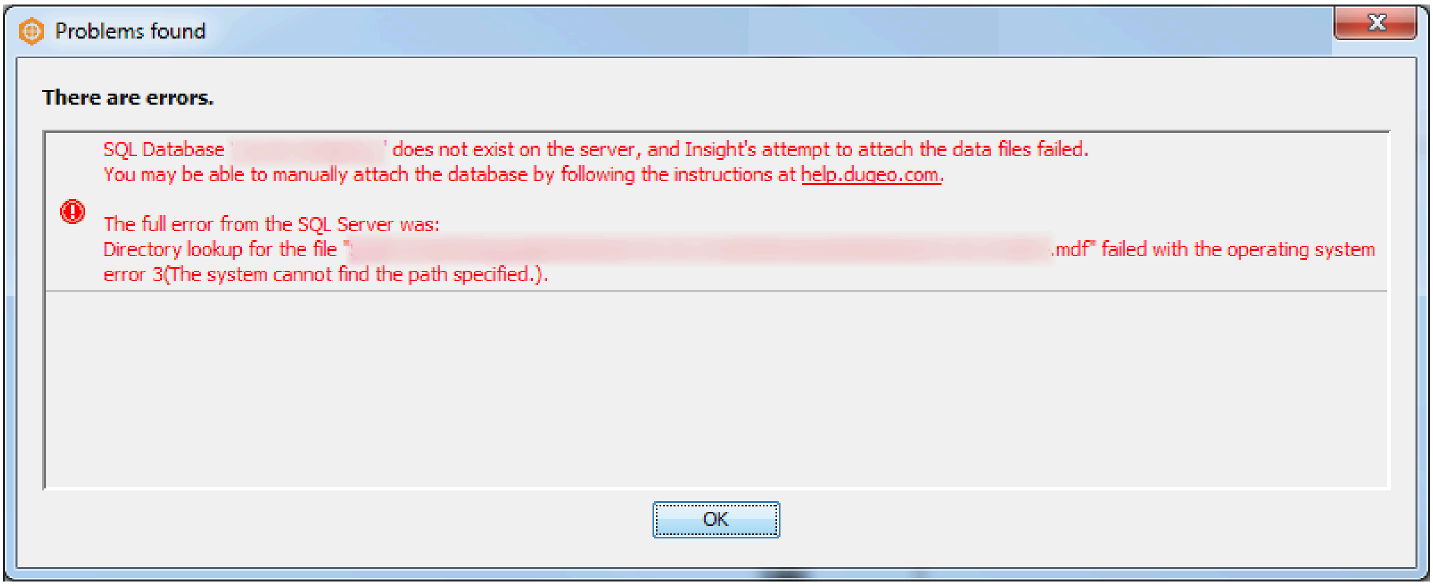
Genuine offer of assistance

If the problem persists, or if you have any questions, please contact our support team at support@dugeo.com.

You may be asked to send data of your session from the Logs window for a clearer picture of what happened when the error occurred (see *Viewing and Sending Diagnostic Logs*).

SQL database does not exist error

Microsoft SQL Kingdom projects in Auto SQL Server Express (SSE) mode need to be attached before you can link the projects in Insight (see *Downloading and Installing SQL Server*). Kingdom does this automatically, thus a project can be left open in Kingdom and then connected to Insight, or manually attached (see *Manually Attaching Kingdom SQL Database*).



This error is coming from the SQL server used to access data from your Kingdom project. The error says the server is unable to access the Kingdom database because the path to the database file (.mdf) on the specified drive is no longer valid. There are two possible reasons for this:

- If this network drive is no longer available, or if the location of the kingdom project has changed, you would get this error message. You may need to check with your IT support for help with this.
- If you did not manually attach the Kingdom database, the project must be open in Kingdom when you connect to the database from Insight. This error will appear if you try to connect to the database without Kingdom running. To connect without Kingdom running, the project has to be manually attached (see *Manually Attaching Kingdom SQL Database*).

Note: If you no longer need to access data from the kingdom database you can disconnect the link and the error will no longer appear.

Genuine offer of assistance

If the problem persists, or if you have any questions, please contact our support team at support@dugeo.com.

You may be asked to send data of your session from the Logs window for a clearer picture of what happened when the error occurred (see *Viewing and Sending Diagnostic Logs*).

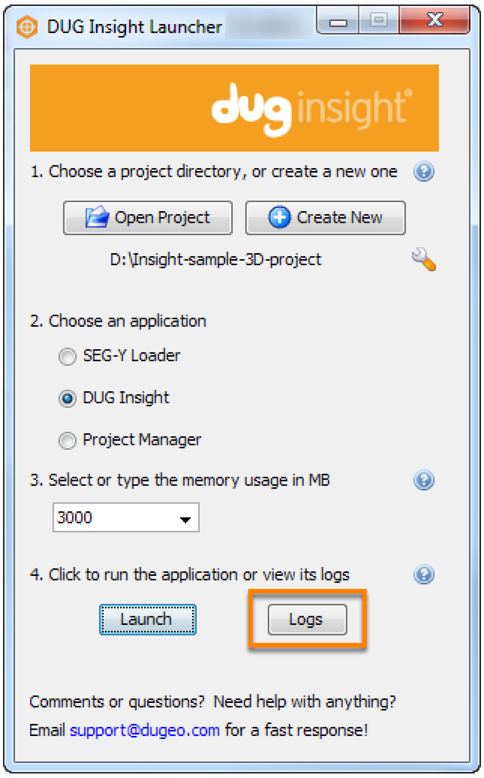
There is a problem. How do I provide information on my issue?

On occasion, DUG Insight may experience an issue which will require more information for us to provide a fix for you. Here is how to get more diagnostic information to us so we can resolve the issue quickly.

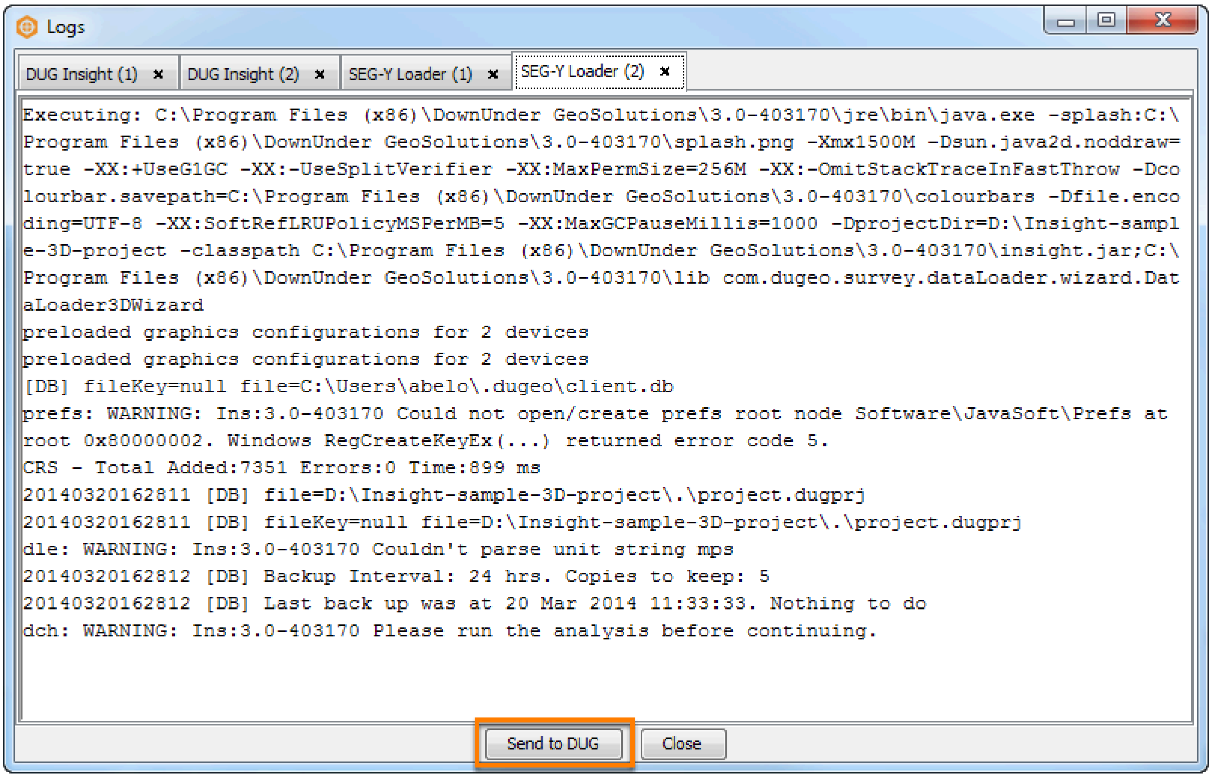
Note: For a fatal error, see *Help, Insight just crashed! What should I do now?*

Repeat or perform the action that causes the issue to occur

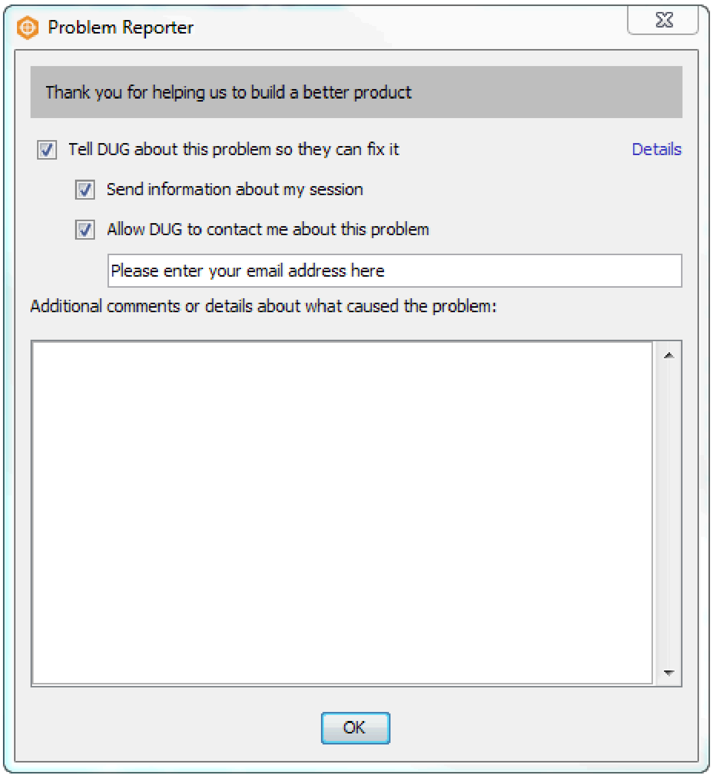
This is important! The issue must be reproduced, and the logs captured at the time of the issue occurring.



1. In the **DUG Insight Launcher**, click the **Logs** button.
2. The Logs window will be displayed



3. There is a tab for every time that Insight or the SEG-Y Loader has been launched. Select the relevant tab that you are currently experiencing a problem with.
4. Click **Send to DUG** to send the log files for diagnostics.



5. Enter your email address and any other relevant information in to the comments box so we can provide a response back to you. This information, along with the contents of the log, will be sent to the support team for analysis.
6. If you want a response from the support team, make sure to provide an email address.
7. Click **OK** to send the logs.

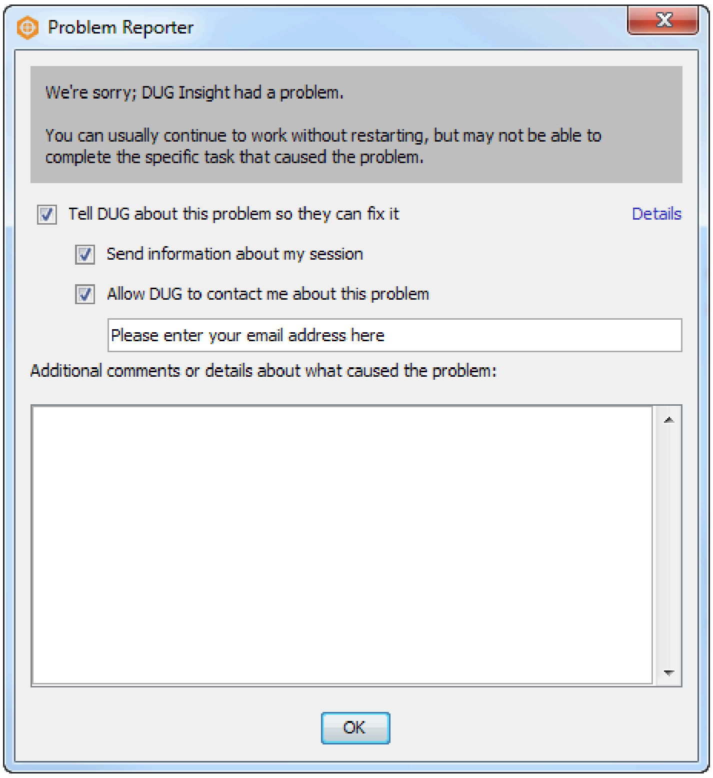
Note: It is important that you send the logs to DUG while you are having that problem. Sending it later will not be useful to provide information related to the problem.

For more information, see Viewing and Sending Diagnostics Logs.

Help, Insight just crashed! What should I do now?

When Insight encounters a fatal error, the Problem Reporter window will appear. It allows you to easily send details about the error to us.

Note: For a non-fatal error, see *Viewing and Sending Diagnostic Logs*.



1. Enter your email address and any other relevant information in to the comments box so we can provide a response back to you. This information, along with the contents of the log, will be sent to the support team for analysis.
2. If you want a response from the support team, make sure to provide an email address.
3. Click **OK** to send the logs.

Note: It is important that you send the logs to DUG while you are having that problem. Sending it later will not be useful to provide information related to the problem.