PaleoScan™ 2018
New Features

Global Seismic Interpretation Software
Copyright Notice

All rights reserved. No part of this document may be reproduced, stored in a retrieval system, or translated in any form or by any means, electronic or mechanical, including photocopying and recording, without the prior written permission of Eliis SAS, 187 rue Helene Boucher, 34170 Castelnau-le-Lez, FRANCE.

Disclaimer

The use of this product is governed by the PaleoScan™ Software License Agreement. Eliis makes no warranty, expressed, implied, or statutory, with respect to the product described herein and disclaims without limitation any warranty of merchantability or fitness for a particular purpose. Eliis reserves the right to revise the information in this manual at any time without notice.

Contact

Feel free to contact us worldwide for any information request.

Web: www.eliis.fr

Eliis S.A.S
Parc Mermoz
187, rue Hélène Boucher
34170 Castelnau-le-Lez
FRANCE

Phone: +33 (0) 4.67.41.31.16
Email: contact@eliis.fr
Licensing

PaleoScan™ 2018 including all add-on modules can be downloaded from Eliis web site. A personal user account is required. If you do not have a login and password to access Eliis extranet, you can fill up this form.

Eliis provides you a free 30-day temporary license to evaluate PaleoScan™ 2018. The temporary license will give you a full access to the software with all add-on modules.

Project Compatibility

The PaleoScan™ platform remains compatible with all the projects.

Forward compatibility:
Projects saved in previous versions of the PaleoScan™ can be updated in PaleoScan™ 2018, upon loading of the project.

Backward compatibility:
Projects created in the PaleoScan™ 2018 can also be opened in previous version (2017 or 2016). However, some new objects properties could not be read in earlier versions.

Hardware Requirements

PaleoScan™ is a Microsoft Windows® stand-alone software, running on PC equipped with a 64 bit processor with requirements equivalent to following:

- Mother board: Gigabyte Technology Co., Ltd. H55M-UD2H x
- Processor: Intel Core i3 540
- Operating system: XP/Vista and 7 (64 bit)
- Graphics card: ATI Radeon HD 5450 (512 RAM GDDR3/DDR2)
- IDE devices: ST3500418AS CC46 (SATA II, 465.76 Go, tampon: 16 Mo) 500Go 7200T
# Table of Contents

## PALEOSCAN™ 2018

<table>
<thead>
<tr>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>7</td>
</tr>
</tbody>
</table>

## PROJECT MANAGEMENT

<table>
<thead>
<tr>
<th>Assignment of a Coordinate Reference System (CRS)</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>Project Properties</td>
<td>8</td>
</tr>
</tbody>
</table>

## DATA IMPORT/EXPORT

<table>
<thead>
<tr>
<th>3D Volume SEG-Y Import</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>2D Line SEG-Y Import</td>
<td>9</td>
</tr>
<tr>
<td>New Objects Import</td>
<td>10</td>
</tr>
<tr>
<td>Multi-SEG-Y Export</td>
<td>14</td>
</tr>
<tr>
<td>New Objects Export</td>
<td>15</td>
</tr>
</tbody>
</table>

## VOLUME

<table>
<thead>
<tr>
<th>Volume Extraction: Combining Z values and Horizons</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>24</td>
</tr>
</tbody>
</table>

## UNIT/CRS CORRECTION

<table>
<thead>
<tr>
<th>Unit Management of New Objects</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>CRS Management of all Objects</td>
<td>25</td>
</tr>
<tr>
<td>CRS Assignment of Projects</td>
<td>26</td>
</tr>
<tr>
<td>Custom CRS Creation</td>
<td>27</td>
</tr>
<tr>
<td>Objects Conversion to Project CRS</td>
<td>30</td>
</tr>
</tbody>
</table>

## VOLUME EDITING

<table>
<thead>
<tr>
<th>IL, XL, Survey Coordinates Editing</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>Seismic Reference Datum (SRD)</td>
<td>31</td>
</tr>
</tbody>
</table>

## GENERAL TOOLS

<table>
<thead>
<tr>
<th>3D Polylines Picking</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>Multi-Polyline Editing</td>
<td>34</td>
</tr>
<tr>
<td>CRS Engine Disabling</td>
<td>35</td>
</tr>
<tr>
<td>2D Objects Settings</td>
<td>35</td>
</tr>
</tbody>
</table>

## VIEWERS

<table>
<thead>
<tr>
<th>Context Menu Display</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>Real Time Smoothing Property</td>
<td>36</td>
</tr>
<tr>
<td>SRD Value for Object Properties</td>
<td>37</td>
</tr>
<tr>
<td>H/V Ratio</td>
<td>38</td>
</tr>
<tr>
<td>Section</td>
<td>Page</td>
</tr>
<tr>
<td>---------</td>
<td>------</td>
</tr>
<tr>
<td>MODEL-GRID</td>
<td>41</td>
</tr>
<tr>
<td>Model-Grid on 2D Lines</td>
<td>41</td>
</tr>
<tr>
<td>Horizon Interpretation: Tracking Mode (shortcut: B)</td>
<td>45</td>
</tr>
<tr>
<td>Data Mapping on 3D Model-Grid Patches</td>
<td>45</td>
</tr>
<tr>
<td>Model-Grid Mapped Data in Project Data Exchange Tool</td>
<td>47</td>
</tr>
<tr>
<td>New Real-Time Preview Quality: Marked Only</td>
<td>47</td>
</tr>
<tr>
<td>Geo-Model Computation from 2D Model-Grid</td>
<td>48</td>
</tr>
<tr>
<td>Mis-tie Correction</td>
<td>49</td>
</tr>
<tr>
<td>HORIZON STACK / HORIZON</td>
<td>53</td>
</tr>
<tr>
<td>2D Horizon Stack</td>
<td>53</td>
</tr>
<tr>
<td>2D Horizon Stack Interpolation</td>
<td>55</td>
</tr>
<tr>
<td>2D Horizon</td>
<td>57</td>
</tr>
<tr>
<td>Create a New Picked 2D Horizon</td>
<td>58</td>
</tr>
<tr>
<td>Horizon Picking Mode</td>
<td>58</td>
</tr>
<tr>
<td>Redo Option</td>
<td>59</td>
</tr>
<tr>
<td>Horizon Validation</td>
<td>59</td>
</tr>
<tr>
<td>Horizon Interpolation</td>
<td>60</td>
</tr>
<tr>
<td>Isolines Display and Saving on Horizon and Horizon Stack</td>
<td>63</td>
</tr>
<tr>
<td>3D Geo-Model from Horizons: Horizon Stack used as Input</td>
<td>64</td>
</tr>
<tr>
<td>2D Geo-Model from Horizons: Horizons or Horizon Stack used as Input</td>
<td>65</td>
</tr>
<tr>
<td>Gross Rock Volume Table from Horizon</td>
<td>66</td>
</tr>
<tr>
<td>2D LINES</td>
<td>71</td>
</tr>
<tr>
<td>Arbitrary Line along Wells: New Search Optimal Trajectory Option</td>
<td>71</td>
</tr>
<tr>
<td>2D Line Set Creation from Several 2D Lines</td>
<td>71</td>
</tr>
<tr>
<td>2D Line Set Viewer</td>
<td>72</td>
</tr>
<tr>
<td>Navigation</td>
<td>73</td>
</tr>
<tr>
<td>2D Line Set Normalization</td>
<td>76</td>
</tr>
<tr>
<td>2D Line Set Extraction</td>
<td>77</td>
</tr>
<tr>
<td>Extraction Area Saving from a 2D Line Set Saving</td>
<td>79</td>
</tr>
<tr>
<td>MULTI-Z</td>
<td>80</td>
</tr>
<tr>
<td>2D Multi-Z Picking</td>
<td>80</td>
</tr>
<tr>
<td>ADVANCED INTERPRETATION</td>
<td>81</td>
</tr>
<tr>
<td>Stratigraphic Sequence Picking Based on a 2D Model</td>
<td>81</td>
</tr>
<tr>
<td>COLOR BLENDING</td>
<td>83</td>
</tr>
<tr>
<td>Indexed Color Objects Saving</td>
<td>83</td>
</tr>
<tr>
<td>VELOCITY MODELING</td>
<td>85</td>
</tr>
<tr>
<td>Velocity Computation</td>
<td>85</td>
</tr>
</tbody>
</table>
PaleoScan™ 2018 is a new generation of 3D seismic interpretation software, where geoscientists build a geological model while interpreting seismic volumes. With this new release, Eliis continues to innovate in seismic interpretation and brings more tools to interpret larger seismic data sets, with added speed and precision.

This document lists all the new features upgrades and corrections implemented in PaleoScan™ 2018. A detailed description of each tool can be found in the “New Features Guide” or on the web site (www.eliis.fr).

The 2018 release brings more options to complete the workflow from the exploration to the reservoir characterization, with various new tools: watertight model, spectral decomposition, cross plots (2 and 3 channels), velocity modeling...
Assignment of a Coordinate Reference System (CRS)

A CRS can be applied while creating a new project thanks to the Project CRS option.

Project Properties

The project properties window shows the **Project Name** and the **Project Directory**. It also allows to update the project CRS if needed. Click on the "*Project Properties...*" option in the "File" menu.
3D Volume SEG-Y Import

**Multi-Import**
Several 3D volume SEG-Y files can be selected and loaded at once if all the files have the same encoded bytes. In this case, the parameters and options will be applied on all imported data.

![Multi-Import Image]

**CRS Option**
To define the adequate CRS, click on the Globe icon and choose it within the CRS list.

![CRS Option Image]

**SRD Option**
If a seismic reference datum (SRD) has to be assigned to the seismic volume, it can be defined within the SRD tab. Warning: this is a reverse elevation value.
Example: The SRD is positioned at -50 meters TVDSS, the user should enter +50.

The option *Apply vertical shift* is available if the imported volume is detected in depth.
- By applying the vertical shift on depth volume, the Z coordinates of the volume will be changed.
- If the vertical shift is not applied while importing the data, the SRD value will only be assigned to the volume. This SRD value can be used later in the workflow during the depth conversion process.
2D Line SEG-Y Import

Multi-import
Several 2D line SEG-Y files can be selected and loaded at once if all the files have the same encoded bytes. In this case, the parameters and options will be applied on all imported data.

CRS Option
To define the adequate CRS to the imported 2D line(s), click on the Globe icon and choose it within the CRS list.

SRD Option
If a seismic reference datum (SRD) has to be assigned to the imported 2D lines, it can be defined within the SRD tab. Warning: this is a reverse elevation value. Example: The SRD is positioned at -50 meters TVDSS, the user should enter +50.
The option Apply vertical shift is available if the imported 2D line is detected in depth. 
- By applying the vertical shift on depth data, the Z coordinates of the line will be changed.
- If the vertical shift is not applied while importing the data, the SRD value will only be assigned to the line. This SRD value can be used later in the workflow during the depth conversion process.

New Objects Import

Well Heads (Shapefile)
Select the item Well Heads in the list of data to import, and then select the file(s). Press Import to open the Shapefile Well Import interface.

Choose a column to be the Name Column. If no name column exists in the shapefile, create a new column with the Create New Column button. Select the UTM units and press Import to import the well heads. Wells will be created in the database according to the selection. In case of import issues, problems are displayed in the Problems window.
2D Horizons
Select the **2D Horizon** Import tool from the general toolbar.

The horizon import manages the following formats:
- ASCII (X, Y, Z)
- Charisma
- GeoFrame (IESX)
- OpenWorks 2003
- OpenWorks 5000
- Kingdom
- Gocad

Click on the "Browse Folder" button to select the file(s) to import. Then, click on **Next**. Each file is scanned to detect X, Y and Z columns positions. *PaleoScan™* can automatically import the horizon(s). Otherwise, user has to manually define the columns position and manage the horizon import from the Custom interface.

Check all the parameters and select the seismic volume associated to the same UTM zone. First the coordinate system, then both spatial and vertical unit. If the horizon to import has data, check the box Map data, if needed specify a “no value” data, check the position of the columns. Define a reference volume and click on **OK** to launch the importation.
Once the data are imported, the horizons are opened in 3D and automatically saved and stored in the database, under the **Horizon Tab**.

Note that several horizon files with the same format can be imported at the same time. Use the list to check the parameters if needed.

### 3D Faults in Gocad format

Interpreted faults from another interpretation platform can be imported in PaleoScan™ using different formats: Custom fault sticks, Charisma; IESX and Gocad. Fault files imported at the same time must have the same format.

### CRS Import as .prj files

CRS objects can be imported in .prj file by selecting the **CRS** option in the Import menu.

Browse the CRS file in your folder and click on **OK**. The parameters will be displayed, click **OK**. The CRS will be stored in the **Other** tab of the Project Browser, under the CRS folder. It can then be used for CRS assignment to any object of the project or used for the project CRS.
Multi-SEG-Y Export

3D Volume SEG-Y

Several volumes can be exported at the same time thanks to a batch process. Drop Volumes or click on Add items button to add volume(s) to the export process. Check the parameters in the SEG-Y export dialog box, Units, CRS and enter the output file name for each volume. All the exported volumes will share the same byte id assignment. Click on OK to launch the export.

The byte numbers of the File and the Traces Header are proposed by default. However, if some SEG-Y volumes already exist, parameters can be loaded in the Parameters pull down menu.

Real time volumes and Flattened Volumes can also be exported in the same way.

2D Line SEG-Y

Several lines can be exported at the same time thanks to a batch process.
Drop lines or click on Add items button to add 2D Lines to the export process. Check the parameters in the SEG-Y export dialog box, Units, CRS and enter the output file name for each line. All 2D lines will share the same byte id assignment. Click on OK to launch the export.

New Objects Export

**2D Horizons**

To export 2D horizons, click on the Export button from the general toolbar and select the Horizon option.

Horizons can also be exported by using the context menu (right click on the horizon(s) name) from the Project Browser and from the context menu of a horizon viewer (right click on the top bar of the viewer):
1. Select the export format.
2. Select the horizon(s) to export in the list,
3. Spatial and vertical units can be defined from the Unit tab,
4. Then define the export name and click on OK to launch the export.

Note: For the Custom format, the X, Y and Z columns have to be defined when the UTM system is selected. By selecting the Survey system, the IL, XL and Z columns have to be defined as well as a reference volume which can be a survey object.
There are different ways to export a 2D horizon from a 2D Horizon Stack. The option is available from:
- the general toolbar,
- the context menu of the horizon stack in the Project Browser
- the horizon stack viewer.

Select the horizon stack in the list and choose the horizons to export and select the format and the parameters associated.

Different formats of export can be used such as Charisma, Geoframe, OpenWorks, Kingdom, Gocad and Custom (X, Y, Z). Except for the last one which is detailed later, the following window opens:
Data Import/Export

- **Charisma Format**

- **Geoframe (IESX) Format**

- **OpenWorks (Default: SingleFormat)**
- **OpenWorks (MltImp/MltExp)**

    ![OpenWorks (MltImp/MltExp) Screen Shot]

- **Kingdom Format**

    ![Kingdom Format Screen Shot]

- **Custom Format (X, Y, Z)**

    If the Custom Format (X, Y and Z) is selected to export the horizon(s), a specific interface is opened to adjust the export parameters.

    1. Select the **horizon(s)** to export in the File(s) Selection part,
    2. Define the Coordinates System; if the **Survey system** is selected, define the reference volume used for the survey; if it is **UTM coordinates**, the spatial unit will have to be defined. Click on the Units tab to choose the export units.
    3. Check the positions of the columns.
    4. Select the output folder and press **OK** to launch the export.
2D Faults
To export 2D faults, click on the Export button from the general toolbar and select the 2D Fault option.

Faults can also be exported by using the context menu (right click on the fault(s) name) from the Project Browser:

Different formats of export can be used for 2D faults such as Charisma, IESX and Fault Sticks XYZ.
1. Select the export format.
2. Select the Fault Set(s) to export in the list, the reference volume and the output folder.
3. Then click on OK to launch the export.

By clicking on the **Units** tab, you can define the exported spatial and vertical units.

### 3D Polylines

The 3D polyline can be exported in ASCII format. To do so, do a right click on one of the 3D polyline stored in the database under the **Polyline** tab and select the **Export** option. 3D Polylines can also be exported via the general toolbar, by selecting the **Polyline > 3D polyline** option in the export option.

You must choose the object within the **File(s) selection tab** and then precise the **X, Y and Z columns** and the **units** and export by clicking **OK**.
2D Multi-Z
The 2D multi-Z objects are exported in Gocad format.

Click on the Export button from the general toolbar and select the Multi-Z > 2D multi-Z option.
2D multi-Z can also be exported by using the context menu (right click on the object name) from the Project Browser:

1. Select the format
2. Select the 2D multi-Z, units and output folder.
3. Click OK to export
- **Irap (Picked points) Format**

  ![Image of Irap (Picked points) Format](image1)

- **Irap (polygons) Format**

  ![Image of Irap (polygons) Format](image2)

- **CPS (polygons) Format**

  ![Image of CPS (polygons) Format](image3)
Volume Extraction: Combining Z values and Horizons

Extraction Area and vertical boundaries using horizons can be combined:
Select an extraction area or smaller cube in the pull down menu and select the Horizon option for the vertical boundaries.
Z values and Horizon can be mixed while defining vertical limits.
Unit/CRS Correction

Unit Management of New Objects

The Unit/CRS Correction tool can be used to make unit correction due to wrong units defined while importing the data. Data generated from an input with wrong units will have wrong units too.

This tool manages units and CRS for the following data including new types in bold:

- Attribute volumes,
- 3D Models,
- **2D Models**,  
- **Extraction Areas**,  
- 3D Model-Grids,  
- **2D Model-Grids**,  
- 3D Horizon Stacks,  
- **2D Horizon Stacks**,  
- 3D Horizons,  
- **2D Horizons**,  
- Geobodies,  
- Layers,
- **Cultures**,  
- **Polylines**,  
- **3D Polylines**,  
- **3D Multi-Z**,  
- **2D Multi-Z**,  
- **3D Faults**,  
- **2D Faults**,  
- **2D Lines**,  
- **Faults polygons**,  
- Watertight Models,
- Watertight Horizons,  
- Wells.

To access the Unit/CRS Correction Tool, click on the Unit/CRS button from the general toolbar:

![Unit/CRS Editor...]

CRS Management of all Objects

All objects in PaleoScan™ can have a CRS. This CRS can be defined upon import, or from the Unit/CRS tool. As for the project, assigning a CRS to an object is not definitive. The object CRS can be changed at any time from the Unit/CRS tool.

The Unit/CRS tool can be used to change the CRS of PaleoScan™ objects. By changing the CRS of an object, the original data isn’t altered as no conversion is applied.

**NOTE:** THIS IS NOT A CRS CONVERSION TOOL BUT A CRS CORRECTOR TOOL.

To access the Unit/CRS tool, click on the Unit/CRS button from the general toolbar:

![Unit/CRS Editor...]

All data are listed in a table. For each data, the CRS is displayed in the last column. If the object doesn’t have any CRS, Undefined is displayed.

To change the CRS of an object, double click on the CRS to edit it:
When editing a CRS:

- Click on the CRS selection button to select the proper CRS within an interactive list.
- Click on the red cross to delete the current object CRS. The CRS will be Undefined.

CRS Selection

The CRS selection window shows all available CRS in PaleoScan™. It contains a predefined list of common CRS, and custom CRS of the current project.

Click on an item in the list to select it and display more information about it in the right panel. The relevance area of the selected CRS is also displayed in the world map, if available.

The list can be filtered in two ways:

- Using the text filter on the top of the window. It will filter the name, authority and authority code columns.
- By double clicking inside the world map. It will filter by CRS relevance area. Be careful using this filter, it may remove from the list some CRS that don’t have any relevance area defined like custom CRS of your current project. Double click at the same location to remove this filter.

If the desired CRS is not inside this list, it can be created by clicking on the top right button.

CRS Assignment of Projects

A project CRS must be defined to take advantage of the CRS management in PaleoScan™. If your project has no CRS, the data will be displayed as it is, without any conversion. The project CRS assignment is not definitive and can be changed at any time.

For informative purpose, the project CRS is displayed in the PaleoScan™ window title:
The project CRS can easily be changed at any time from the Project Properties available from the File menu:

When changing the project CRS, all windows and 3D objects inside PaleoScan™ will be closed.

Custom CRS Creation

Custom CRS can be created for your PaleoScan™ project. The CRS creation interface can be reached from the CRS selection window or from the Tools menu:

Then, fill the CRS creation form:

1. The first three parameters (name, authority and authority code) are informative parameters.
2. The CRS definition can be filled in 3 ways:
   - Using the **proj4** format, which is the native CRS format used by PaleoScan™. It is the recommended way to define a custom CRS. An exhaustive list of proj4 definition parameters can be found here: https://proj4.org/usage/projections.html
   - Using the **WKT** (Well Known Text) format. The WKT definition will be converted in proj4 format by PaleoScan™. If a proj4 definition is available, prefer use it instead of WKT.
   - Using a dedicated form. This method is not exhaustive, but it can be used for simple CRS definitions.

Once the form is filled, click on **OK**. Choose the output name and click on **OK**.
Once created, the new CRS will be available in any CRS selection window and in the Project Browser, under the Other tab:

In project browser

AND

In CRS selection window
Opening a custom CRS from the Project Browser will display a window with all the available information about that CRS:

**Restrictions**

PaleoScan™ has two restrictions regarding the CRS definition:

1. **The CRS must be in meter.** The CRS engine of PaleoScan™ can perform conversion from one CRS in meter to another. If the object has an original CRS in feet, it still can be used in PaleoScan™ because the Unit engine of PaleoScan™ is complementary. Just set the spatial unit of the object in feet so that PaleoScan™ engine can convert the object from feet to meter before performing the CRS conversion into project CRS.

2. **The CRS projection can’t be geographic (Lat/Long coordinates),** because such coordinate system is not handle by PaleoScan™.

If the CRS definition does not match the PaleoScan™ requirements, a message will explain you what is wrong with the definition.

**Datum shift grids**

Thanks to the proj4 engine used by PaleoScan™, while creating a new CRS, a shift grid file can be specified to handle conversion between datums. New grids can be added in the PaleoScan™ installation folder, under the \**resources\**\**grid** directory.

For more information about grids, read the official proj4 engine documentation: https://proj4.org/resource_files.html#transformation-grids.
Objects Conversion to Project CRS

Every object in PaleoScan™ can be converted into project CRS. Right click on an object in the Project Browser and click on **Convert into project CRS**. Choose the output name and click **OK**. A new object will be created with its coordinates converted into project CRS.

Unlike the Unit/CRS tool, this action performs a CRS CONVERSION and will alter the object in the PaleoScan™ database. You must keep in mind that coordinates of the resulting object may be definitely distorted. However, this action always creates a new object to preserve your original data.
Volume Editing

IL, XL, Survey Coordinates Editing

Volume coordinates can be changed for correction. Modifications will be done from key dot coordinates while choosing either UTM or Survey management. The number of traces never changes when using this tool. Thus, IL and/or XL values modification (Survey management) for each key dot may change the resolution of the volume (step).

Seismic Reference Datum (SRD)

3D Volumes

A Seismic Reference Datum (SRD) representing the depth shift from Mean Sea Level to the seismic starting point (reverse elevation) can be assigned and/or applied to a volume either while importing the SEG-Y file or in the Coordinates Editor.

Since it represents a reverse elevation, the SRD value has to be positive in PaleoScan™. Example: The SRD of the volume is defined at -50 meters TVDSS, the user should enter +50m.

- In Time domain, the SRD value will be informative and can be used later in the workflow during the depth conversion process.

- In Depth domain, the vertical shift can be applied by ticking the option. By applying the shift related to the SRD, the Z coordinates of the volume will be changed.

When the SRD is defined while importing the volume, the SRD value is displayed under the viewer windows. Otherwise, the value is 0.

To define or modify the SRD value, click on the SRD button available in the bottom part of the Coordinates Editor.
2D Lines

A Seismic Reference Datum (SRD) representing the depth shift from Mean Sea Level to the seismic starting point (reverse elevation) can be assigned and/or applied to a 2D Line either while importing the SEG-Y file or in the 2D Line SRD Editor.

Since it represents a reverse elevation, the SRD value has to be positive in PaleoScan™.

- In Time domain, the SRD value will be informative and can be used later in the workflow during the depth conversion process.

- In Depth domain, the vertical shift can be applied by ticking the option. By applying the shift related to the SRD, the Z coordinates of the 2D Line will be changed.

When the SRD is defined while importing the 2D Line, the SRD value is displayed under the viewer windows. Otherwise, the value is 0.

To define or modify the SRD value, use the right click on a line in the Project Browser, and select the SRD Editor option.
3D Polyline Picking

3D Polylines can be created in PaleoScan™ by picking a polyline and saving it as 3D Polyline.

1. Open a Time Slice view, a horizon stack viewer, a horizon viewer or the survey viewer.
2. Pick the polyline by using the Polyline mouse mode available from the main toolbar or from the Geobody Manual toolbar (shortcut for Polyline: Shift+R).

3. Select the polyline by using the Selection mouse mode (shortcut F) and save it (Ctrl+S).

4. The polyline can be saved as a polyline, a 3D polyline or a culture. It will be stored under the Polyline tab of the Project Browser, in the Culture folder if it has been saved as a culture, in the Polyline folder if it has been saved as a polyline, or in the 3D Polyline folder if it has been saved as a 3D polyline.
Note: A culture data can be opened in 3D in order to be displayed in every single time slice, it will be displayed at Z=0 in the 3D viewer. A polyline can only be displayed in 2D, on the frame on which it has been created. A 3D polyline can be displayed in 2D, on the frame on which it has been created, and in 3D.

Arbitrary lines can be created from polylines (not from 3D polylines or cultures).

Multi-Polyline Editing

A polyline can be edited by doing a right click on the polyline and selecting the option **Edit**. Change the Value straight in the table and click on **OK** to update the polyline.

In the case of a multi-polyline, i.e. a polyline composed of several disconnected lines, a red line with NaN values (-9999.00) as X, Y and Z coordinates will appear in the table to separator one line from the other.
CRS Engine Disabling

Project units and CRS are directly managed from the viewer properties. Default spatial and vertical units can be defined. Regarding the vertical unit setting, units have to be defined for Time and Depth vertical domains.

Under CRS settings, an “ignore CRS” option can be activated (Yes or No). If Yes is selected, no CRS conversion will be performed, the objects CRS specified in the Unit/CRS Tool will therefore not be taken into account.

Viewer unit settings have an effect on:
- **Viewers**: the units of axes will depend on unit settings;
- **Coordinates**: X, Y and Z displayed units;
- **Horizontal Scale** in viewer;
- **Tools** involving units: such as Horizon Shifting, Survey Editor, etc...
- **Distance Measure** tool: Vertical and spatial measurement depend on units from settings.

2D Objects Settings

For each 2D object (2D Horizon, 2D Multi-Z or 2D Fault), several display parameters can be defined. These display parameters are applied at the creation of these objects.
Viewers

Context Menu Display

The context menu from the viewer is specific to each kind of viewer. All the options proposed in these menus refer to the specific modules.

There are different ways to display the context menu of a window:
(1) Click on the Right Mouse Click on the top bar of the window,
(2) Use the shortcut Double Right Mouse Click inside the viewer,
(3) Use the shortcut Alt + Right Mouse Click inside the viewer.

Context menus are available from all viewers such as:
- Seismic (volume) viewer,
- Model-Grid viewer,
- Horizon viewer,
- Horizon Stack viewer,
- Fault viewer,
- Arbitrary Line viewer,
- Strati viewer,
- Log viewer,
- Colour Blending viewer,
- Etc...
Real Time Smoothing Property

Smoothing options are available in Real Time from the Properties of Volume, 2D Line, Horizon Stack and Horizon viewers.
Several smoothing method are available:
- Gaussian
- Median
- Structure Oriented
SRD Value for Object Properties

The parameters of the displayed object are displayed in the **Object** tab of the **Properties** panel. The main parameters are:

- The file size (in Gb);
- The number of Inlines, Crosslines and Z samples;
• The spatial and vertical units;
• The attributed CRS;
• The object resolution; if a **SRD (Seismic Reference Datum)** has been defined for the selected volume, its value will be listed here;
• The object dimensions.

**H/V Ratio**

**2D Viewers**

By default, the volume display is composed of square pixels (identical horizontal and vertical sizes). The H/V option allows the user to adjust the vertical exaggeration factor. For a Time seismic volume with a spatial resolution of 25 m x 25 m and a vertical resolution of 4 ms, the H/V parameter will be 6.25 (25/4).

In case of depth volume, the display still follows this rule. Thanks to the H/V parameter, the H/V ration can be re-adjusted (set to 1) in order to get the real display.

The **H/V ratio** of the current viewer can be adjusted in the **Properties** window, under the **Display** tab.
3D Viewer

To vertically stretch the 3D view, adjust the H/V value accordingly. In this viewer, the H/V ration has to be negative.
Model-Grid on 2D Lines

2D Line Set Creation

Model-Grids can only be generated on 2D line sets, the first step is therefore to generate a 2D line set. To do so, under the 2D Line module, click on New 2D Line Set.

Drag and drop the 2D lines that will make up the 2D line set. The intersections between the lines are shown on the Horizon Viewer. It is possible to navigate on the map using the cross-navigation mouse mode (G), the displayed line will be updated according to the navigation.

The Values normalization is activated by default. It will normalize the amplitudes of the selected 2D lines so that they all have the same range of amplitudes. This option can be deactivated by unticking the box.

2D Model-Grid Creation

To create a 2D Model-Grid, select the Model-Grid module and click on the New 2D Model-Grid button.

The Model-Grid creation on 2D lines interface enables testing the behaviour of the horizon propagation depending on the selected parameters (Patch size, Polarity, Correlation Threshold...), such as for 3D Model-Grids.
Select the 2D line set in the Input Line Set list, the Model-Grid will be generated on this line set.

**Interactive Navigation between Lines**

Once a new 2D Model-Grid is computed, a new interpretation process can be started from the Auto Interp, where patches have been automatically linked.

To open the Auto Interp, right click on it in the Project Browser and select the Open option from the context menu or double click on the Autointerp file. An Interp viewer with three seismic sections and a horizon viewer opens.

When Cross navigating (G shortcut) between the lines on the horizon viewer, two lines are highlighted. Two Interp viewers (a) and (b) are automatically updated and display the two highlighted lines. The last interp viewer (c) displays the intersection between the two previous lines.

![Interp Viewers Screenshot](image-url)

(a) Interp Viewer: line 1  
(b) Interp Viewer: line 2  
(c) Interp Viewer: intersection  
(d) Horizon Viewer: Line 1 and 2 are highlighted
Such as for 3D, the interpretation refinement task will consist in editing and changing the links between horizon patches (by merging and splitting) in order to improve the Model-Grid.

**Add 2D Lines**

2D lines can be added to an existing 2D Model-Grid. To do so, right click on an existing 2D interpretation and select the **Add 2D Lines** option from the pull-down menu.

![Add 2D Lines](image)

The option is also available from the Model-Grid toolbar:

![Model-Grid toolbar](image)

The **Model-Grid 2D add line(s)** window pops up, select the interpretation from the list and click on the blue arrow on the right hand side of the window. The chosen 2D lines to be added can be dragged and dropped in the **Input 2D Lines** window. The **Values Normalization** is also available from that interface. Click on Save and give a name to the new Model-Grid. A new Model-Grid will be generated and will include the added 2D lines.
Disconnect all Links between Lines

In the case of a 2D Model-Grid interpretation, an option to disconnect all the links between the 2D lines composing the 2D line set is available from the Model-Grid toolbar: Disconnect All Links between Lines. The links between patches will be preserved for each specific line but not between the lines. This option allows the user to start over the grid refinement between lines if too many incorrect links have been defined.

Disconnected line

Disconnect One Line

In the case of a 2D Model-Grid interpretation, an option to disconnect one line from the others is available from the Model-Grid toolbar: Disconnect One Line. A window pops up where the line to be disconnected can be selected. The links of the selected line will be preserved but the links with the other lines composing the line set will be removed.
Horizon Interpretation: Tracking Mode (shortcut: B)

The propagation can also be done using the **Tracking Mode (B keyboard)**, from the Model-Grid toolbar. The horizon patch at the cursor position will be automatically highlighted both on the seismic section and on the horizon viewer. Do a left mouse click on the selected patch and then continue linking to other patches using the same mouse button.

The **Tracking Mode (B)** allows visualizing each individual patch that can be linked to the activated horizon, which makes the grid refinement easier.

**Note:** In a 2D Model-Grid, the tracking mode is available only on one side of the intersection viewer (on the line on which the yellow seed is placed).

Data Mapping on 3D Model-Grid Patches

Data from an existing volume can be mapped on the patches of a 3D Model-Grid. Select the **Data Mapping on Patches** option from the Model-Grid toolbar.

Select the volume that you want to map on the Model-Grid patches, click on Next, give a name to the output and click on Run.
The mapped attribute will be displayed automatically on the Horizon Viewer displaying the active Model-Grid horizon. The mapped data can be changed from the Properties of the Horizon Viewer, under the Data Mapping Option. All the data that have previously been mapped on the Model-Grid patches will be accessible from that window.

To remove data mapped on patches, select the option **Delete Mapped Data on Patches** from the Model-Grid toolbar. Select the data to delete (the size of the data to be removed is indicated at the bottom left of the window) and click on Remove. This option is useful to free up some disk space.
Model-Grid Mapped Data in Project Data Exchange Tool

Data mapped on patches can be exchanged from one project to another with the Project Data Exchange tool.

New Real-Time Preview Quality: Marked Only

In the interpretation window properties of a 3D Model-Grid, a fourth preview quality can be displayed.

1. **Low**: Fast computation involving only the current frame (figure 1);
2. **Medium** (default mode): As fast as the low preview quality but takes a “link probability” into account (figure 2);
3. **High**: The preview is done around the cross-navigation cursor (need to place it inside the Interp Viewer), it takes into account the three main Geo-Model computation parameters (link probability, smoothing and interpolation). The preview is similar to the final 3D computation (figure 3);
4. **Marked Only**: The preview uses only the marked horizons; geological ages are interpolated iso-proportionally between the horizons, using the patches as a guideline (figure 4).
Geo-Model Computation from 2D Model-Grid

The 2D Geo-Model is computed from an opened 2D Model-Grid interpretation, through the following workflow:

1. Click on the Compute Geo-Model button of the Model-Grid toolbar, or right click on the top bar of one of the Interp Viewers to open its context menu;

2. From the context menu, click on Compute Geo-Model. Several parameters need to be set up in the Geo-Model Creation interface;

3. The Geo-Model can be generated either on all the lines composing the line set or just on one specific line. To select a single line, toggle on the Single Line option and select the desired line from the drop-down menu;

4.a - 2D Multi-Z can be included by dropping them from the Project Browser into the Exclusion Zone(s) area. No Relative Geological Time data will be computed within these areas;

4.b - Remove one or all the Exclusion Zone(s) previously added;

5. Different saving options can be toggled on:
   - Model with Corrected Mis-ties: A Geo-Model corrected from mis-ties will be generated on the 2D line set/single line;
- **Seismic with Corrected Mis-ties**: A seismic corrected from mis-ties will be generated on the 2D line set/single line;
- **Z Variant Shift**: A 2D line set/single line of vertical shift will be generated. The vertical shift corresponds to the mis-tie correction applied to each point of the 2D line set/single line;
- **Z Original Position**: The Z value before mis-tie correction is generated for the 2D line set/single line.

6- Click on the **Save** button;

7- The default name can be changed;

8- Click on **OK** to launch the computation. The 2D Geo-Model is then automatically stored within the **2D Line** tab of the Project Browser (under 2D Model if it has been computed on a single line, under 2D Line Set Model if it has been computed on all the lines).

**Mis-tie Correction**

When we analyse the traces coming from both cross sections along the intersection, the traces won’t be necessarily identical and their interpretation may not vertically match (see Trace 1 and Trace 2).

In the mis-tie process, only marked horizons are considered as “time lines”.

In our example illustrating 2 cross sections intersecting, 4 horizons have been marked. PaleoScan™ analyses the vertical difference of each marked horizons along the intersection and computes their associated delta ($\Delta_{1,2,3,4}$).
When applying a mis-tie, all the 2D lines will move up or down in relation to each other in contrary of using a master 2D Line.

In order to minimize the vertical displacement (delta), half a delta movement will be applied on both sides of the intersection at the marked horizon locations.

Hence, for each trace, a Variant Shift function is calculated using half the delta as input and interpolated based on the Spline interpolation method.

Finally, the Variant Shift function is continuously applied all along both traces in order to shift every single point. The traces are then re-sampled to the seismic resolution.

This workflow is applied to all the traces along the lines in order to readjust all the 2D lines in relation to each other.
In case of multi-intersections along a single line, a horizontal Variate Shift function is calculated for each constant $Z$, based on the previous method and applied on each traces of the line to laterally spread the shift.

Horizontal variante shift function calculated for the cross section 1 at Depth $Z_1$

The workflow described above has to be applied looking at the Cross 2D Line viewer (bottom right corner of the 2D Model-Grid viewer).
To sum up, the first step is to mark (M) some key horizons to fix the mis-tie and then to click on the **Shift** button, located at the bottom right corner of the viewer. PaleoScan will automatically calculate the Variant Shift function and correct the mis-tie. A yellow triangle will appear within the viewer meaning that the mis-tie correction is on. This process is done almost instantaneously.
**2D Horizon Stack**

2D Horizon Stacks can be generated from 2D Model composed of one or more 2D lines. As in the case of the 3D Horizon Stack, the horizons generation is based on the relative ages of the input 2D Model. The 2D Horizon Stack workflow does not include data mapping since the horizons are only composed of lines.

Use the **New 2D Horizon Stack** button in the **Horizon Stack** module to generate it.

1. Select the 2D Model to be used as input; 2D Model computed on only one line can be selected as well as 2D Line Set Model computed on several 2D lines,
2. Select the background 2D line to better define the top and bottom boundaries – in case of 2D Line Set Model, the 2D Line available for the background display is filtered according to the displayed line of the preview,
3. Define the top and the bottom boundaries of the 2D Horizon Stack, using the sliders. These boundaries are displayed in the viewer (Top of Stack: green line; Bottom of Stack: blue line). For more precision, use the green and blue arrows.
4. Enter the number of horizons which will be generated within the interval.
5. Toggle on “Horizons preview” to have the position of the future horizons. It is possible to change the color of the preview by clicking on the colored square.
6. Click on **Run** to launch the computation.

After the computation, the 2D Horizon Stack will automatically be saved and stored in the 2D Horizon Stack data base, under the Horizon tab.
A survey is associated to the 2D Horizon Stack in order to define:
- its spatial boundaries,
- its orientation,
- its X, Y and Z sampling.
All the survey properties are available by selecting the green survey from the 2D Horizon Stack viewer. This way, the UTM and Z sampling can be modified from the properties as well as the survey orientation from the map view.

This survey can be used in the 2D interpretation workflow, especially in the 3D Geo-Model computation from horizons tool where a survey is needed to compute the model.

The 2D Horizon Stack survey can be saved from the context menu of the 2D Horizon Stack viewer by selecting the option **Save Extraction Area**...
2D Horizon Stack Interpolation

A 2D Horizon Stack is composed of a set of 2D Horizons generated from a 2D line set Model. The 2D horizon Stack Interpolation tool allows converting a 2D Horizon Stack to a 3D Horizon Stack in order to get 3D Horizons. Many options can be handled in the interpolation process to control the output Horizon Stack (Interpolation methods – Polyline constraint – Survey – Data for mapping).

The 2D Horizon Stack Interpolation tool is available from:

a/ Horizon Stack module tool bar (works from both 2D & 3D visualisation);
b/ Project Browser context menu, as Interpolate 2D Horizon Stack option (no visualisation);
c/ Horizon Stack Viewer context menu, as 2D Horizon Stack Interpolation option (exclusively works from 2D visualisation).
The resulting Horizon Stack will be considered as **3D Horizon Stack** and saved in the Project Browser under the 3D Horizon Stack folder of the Horizon tab.

**Interpolation Methods**

The 2D Horizon Stack Interpolation can be performed using three different methods: Diffusion – inverse Distance – Kriging (see User Guide 2018, chapter Horizon – Horizon Interpolation).

**Polyline Constraint**

The 2D Horizon Stack Interpolation can be limited inside a polyline (see User Guide 2018, chapter Horizon – Horizon Interpolation).

**Survey**

A survey is needed in the interpolation process to define the boundaries of the interpolated Horizon Stack as well as its orientation.

- By default, the survey associated to the 2D Horizon Stack is used in this process and saved as Extraction Area at the end of the computation. To avoid this saving, toggle off the Save Survey option.

- Another Survey available in the database can be applied to the resulting interpolated Horizon Stack. Drop an Extraction Area in the corresponding field to apply a new survey to the interpolated Horizon Stack.
Input Data
While interpolating a 2D Horizon Stack, data from volume or from 2D Line(s) can be mapped on the new interpolated 3D Horizon Stack. The data will also be interpolated according to the chosen interpolation methods and parameters.

2D Horizon
2D Horizons can be created from 2D Model-Grid horizon saving, or from 2D Horizon Stack extraction. This new item is stored under the folder 2D Horizon in the Project Browser. An option available from the context menu of 2D Horizons in the Project Browser allows converting 2D Horizon in 3D horizons.
Create a New Picked 2D Horizon

2D horizons can be manually picked on 2D Lines by clicking on the New Horizon button in the Horizon module.

2D horizon picking respects the same workflow and polarity behaviors than 3D horizon picking.

Horizon Picking Mode

Two new picking modes (on volumes and 2D lines) are available in the 2018 PaleoScan release. For both modes, a real time preview is displayed while moving the mouse on the volume or the 2D line. The different mouse mode can be selected in the Horizon toolbar.

Best Correlation Tracking

This mode is a 2D horizon tracking which starts from the picked point and search the best correlated neighbour. The tracking stops if the correlation is lower than the user-defined correlation threshold or if there is no polarity corresponding to the horizon polarity in the search window. This mode is suitable for strong reflectors but does not pass the biggest faults. The user can manually define the correlation threshold to adjust the behaviour of the tracking.
Optimum Path Tracking

This mode is a 2D horizon tracking which find the best path regarding the trace correlation between two input seeds. It can be suitable to pass faults but can lead to wrong results if the tracked horizon is close to a strong reflector. In such case the shortest path may follow the wrong horizon.

Redo Option

In the context of a horizon picking edition, a redo mode is now available. This function can be used by pressing Ctrl+Y keys or by opening the undo/redo options in the horizon toolbar and selecting the required picking step.

Horizon Validation

Once the horizon picking is finished, the picked horizon has to be validated in order to stop the picking and save the picked horizon. Click on the Validation button from the Horizon module.

A horizon saving is proposed while validating the picked horizon. After clicking on Yes, the picked horizon will be saved and stored in the Horizon tab of the Project Browser. In case of 3D horizon picking, it will be stored in the Picked Horizon folder under the 3D Horizon item; In case of 2D horizon picking, it will be stored at the root of the 2D Horizon folder.

Once validated, the picked horizon turns in a random color signifying that the horizon is not under editing anymore.
Horizon Interpolation

Re-Apply

Once an interpolation or propagation has been performed, the parameters are preserved and it is possible to directly apply the same interpolation by pressing the bottom propagation icon in the viewer.

Inverse Distance Interpolation Method

The inverse distance interpolation is a little slower than diffusion but offers more geologically consistent results. This interpolation method is based on the formula:

\[ z(x) = \frac{\sum_{i=0}^{N} w_i(x)z_i}{\sum_{i=0}^{N} w_i(x)} \]

with \[ w_i(x) = \frac{1}{d(x,x_i)^p} \]

With \( d(x,x_i) \) the distance between the computed point \( x \) and the input point \( x_i \) and \( p \) the power. The Dist max parameter defines the maximum distance where an input point is taken into account.
Kriging Interpolation Method
Kriging is a geostatistical interpolation method which minimize the variance of the result regarding the selected parameters. This method is adapted to interpolation with a few points but not to a too high density of picking or time map. The parameters consist in defining a variogram model for the interpolation with a variogram type, a range and if the interpolation is anisotropic, a minor range and a direction angle.

Three variogram types are available which are spherical, gaussian and exponential.
Data Interpolation

It is now possible to choose whether interpolating the Z-Value or the Data mapped on the horizon. In case of data interpolation, only the data map will be interpolated and the Interpolation Inputs option is disabled. Note that the data will be interpolated only at locations where a time map already exists.

Area Restriction

The Polyline Constraint option offers to constrain the interpolation inside a polyline. If the polyline is stored in the Project Browser, check the From Database option and drop the required polyline. Otherwise, check the From Viewer option and create a polyline in the viewer with the corresponding mouse modes and select it.
Isolines Display and Saving on Horizon and Horizon Stack

Display Isoline
An isoline can be visualized in a Horizon viewer or Horizon Stack viewer. To display it, select the viewer and open the Isoline menu. Check the Visible option to display the isoline. If the On Goto Point option is checked, cross navigate through the horizon to find the value of the isoline. Otherwise, fill the Iso Value option to choose the required value. If Display Information is checked, the isoline value is displayed in the viewer. The style of the line can be updated in the Edge menu.

By default, the isoline is created following the data image. You can visualize the z-value isoline by selecting Z-Value in the Data Type option.
The **Smooth Level** and **Precision** parameters define the accuracy of the isoline. Small values may slow down the process on big horizons.

**Save Isoline**

The isoline can be saved as a **Polyline**, **3D Polyline** or **Culture data**. To do so, select the isoline by double clicking on it in the horizon viewer. The press the Save button or press **Ctrl+S**.

**3D Geo-Model from Horizons: Horizon Stack used as Input**

The 3D Geo-Model with External Interpretation (iso-proportional method of interpolation between horizons) available from the Horizon module now accepts Horizon Stacks as inputs.
2D Geo-Model from Horizons: Horizons or Horizon Stack used as Input

The 2D Geo-Model with External Interpretation is a different way to generate a 2D Geo-Model in PaleoScan™. It represents an iso-proportional method of interpolation between 2D horizons coming from external interpretation (decorrelated from the Model-Grid). This tool is available from the Horizon module.

1. Drop a 2D Line as input.
2. Drop all the 2D horizons to be used for the iso-proportional 2D Geo-Model computation.
3. Launch the preview by click on the Preview button and adjust the vertical interpolation size (expressed in smp) if needed. Then, click on Save to generate the iso-proportional 2D Geo-Model which will be saved and stored under the 2D Model folder of the 2D Line tab.
Gross Rock Volume Table from Horizon

Gross Rock Volume Computation

When a horizon viewer is opened, it is possible to create a Gross Rock Volume Table based on the displayed horizon. The first step consists in defining a base z-value as reference; the second step consists in defining an area of interest to focus a specific target. These two steps can easily be performed in the horizon viewer but the viewer must be in z-value display mode.

Once the viewer has been displayed in z-value display mode, use the Color Bar widget and press the Select Right or Select Left button to create an area of interest. Use Select Right for a syncline shape and Select Left for an anticline shape. Then move the limit to the required reference z-value.

One or several blue areas will be displayed in the viewer. Focus the targets by selecting the required area(s) of interest with the Polyline Creation mouse mode.
Once the z-value and the areas of interest has been defined, the computation can start. Press the Gross Rock Volume button in the Horizon module.

The following dialog box allows to change the reference z-value (by default the value is the limit of the blue area defined previously but it can be manually adjusted to be more accurate) and the step of the gross rock volume computation. Pressing yes without selecting a base horizon will create a gross rock volume table for the horizon displayed in the viewer only.

It is possible to adjust the display of the table and the graph with several options. Change the surface and volume units to update it in the table. In case of a time horizon it is possible to define the time/depth ratio which impact the volumes by updating the 1 ms (in meter) option. It is possible to choose whether to display the surface graph or the volume graph.
A base horizon can be added to the computation in order to get a volume difference in a specific layer. To do so, open the gross rock volume dialog and select the base horizon. In this case, extra columns appear corresponding to surface and volume for the base horizon alone and the difference between the main horizon and the base horizon.

The computation is performed with the different inputs with the following rules:
As previously, you can play with the display with the different parameters and viewers to get a suitable display.

**Edit Gross Rock Volume Table**

It is possible to manually update the automatically computed z values. To do so, double click in the required cell in the z value column and enter the new value. The surface and volume will be updated accordingly.
Export Gross Rock Volume Table

The gross rock volume table can be exported as semicolon separated value file (.csv). Select the file path and press **Export**.
2D Lines

Arbitrary Line along Wells: New Search Optimal Trajectory Option

In the process of creating an Arbitrary Line along wells, the interface henceforth offers the possibility to automatically sort the wells in order to get the most optimized trajectory. To do so, press the Search optimum trajectory button.

2D Line Set Creation from Several 2D Lines

To create a 2D line set, open the New 2D Line Set interface from the 2D Line module. A 2D line set can be created from 2D lines and/or from a pre-existing 2D line set. Hence a single 2D line can easily be added to a 2D line set by creating a new 2D line set containing the new 2D line and the initial 2D line set.
2D Lines

From 2D Lines

1/ Drag and drop 2D lines to be added to the 2D line set.
2/ Toggle on **Values normalization** if the 2D line set values must be normalized (normalization is recommended before any Model-Grid interpretation). The normalization can also be performed after the 2D line set creation.
3/ Navigate through the viewers and check the consistency of the 2D lines inputs.
4/ Press **Save** to store the 2D line set in the Project Browser under the **2D Line Set** folder in the **2D Line** tab.

From 2D Line Sets

A 2D line set can also be created from one or several other 2D line set(s). To do so, drag and drop 2D line set(s) instead of 2D lines in the interface. The initial 2D line set(s) is split into their different former 2D lines. In the interface, the icons for the 2D lines coming from 2D line sets are different from the normal 2D lines. The next steps are similar to creation From 2D Lines.

2D Line Set Viewer

2D line set viewers can be opened from the **2D Line Set** or **2D Line Set Model** tabs of the **2D Line** module by double clicking on a 2D line set or by pressing open in the context menu.
Navigation

The 2D line set viewer is composed of 4 viewers:
- two 2D line viewers,
- a map view
- a cross 2D line viewer.

The map view permits to select the two 2D lines to be displayed and the cross viewer display the intersection of the two 2D lines if they are intersecting. To select the displayed 2D lines, cross navigate through the map view and the closest lines inside the yellow circle will replace the current ones.

The directions of the two displayed 2D lines are indicated by random letters.
For each sub viewer, a context menu is available by right clicking the corresponding title bar.
Maximize/Restore Viewer

It is possible to **Maximize** and **Restore** original size for each viewer of the interface. Press the **Maximize/Restore** button on the top right corner of the required viewer.

![Maximize/Restore Viewer](image)

Synchronize Display

It is possible to synchronize or desynchronize the **Color bar** and the **Real Time Attributes** of the three 2D line viewers by checking/unchecking the **Synchronize Palette** option in the 2D line set viewer properties. Open the properties of the 2D line set viewer by selecting one of the 2D line viewers in the 2D line set viewer.

![Synchronize Display](image)

Windows List Management

In the **Windows List** manager, each viewer can be checked/unchecked separately for the **Show Intersection** and **3D Synchronisation** modes.

![Windows List Management](image)
2D Line Set Normalization

To make the 2D Model-Grid interpretation possible, the 2D line set must be normalized. If it has not been normalized at the creation, it is possible to normalized it from several ways.

From Toolbar

The 2D line set normalization is available from the 2D Line toolbar by pressing the 2D Line Set Normalization button.

From Project Browser

It is also available from the Project Browser by right clicking a 2D line set and selecting the Normalize option.
From 2D Line Set Viewer

It is also available from a 2D line set viewer by opening the context menu of the parent viewer and pressing **2D Line Set Normalization**.

Once the Finish button has been pressed, the normalized 2D line set is stored in the Project Browser under the **2D Line Set** folder in the **2D Line** tab.

2D Line Set Extraction

2D lines can be extracted from a 2D line set and saved as single 2D lines in the Project Browser or just displayed in a 2D line viewer. This action can be launched from several ways.

**From Toolbar**

Press the **Extract 2D Line(s) from Set** under the 2D Line module.

1/ Drag and drop the required 2D line set from the Project Browser.
2/ Select the 2D line(s) to extract.
3/ Select whether to **Display**, **Save** or **Save and Display** the selected 2D line(s).
4/ If **Save** or **Save and Display** are selected, type a prefix to store the 2D line(s).
From Project Browser

The same interface can be opened from the Project Browser by right clicking on a 2D line set and pressing **Extract 2D Lines**.

From 2D Line Set Viewer

The same interface can be opened from a 2D line set viewer in the context menu of the parent viewer by pressing **Extract 2D Line(s) from Set**.
Extraction Area Saving from a 2D Line Set Saving

An Extraction Area can be created from a 2D line set, useful afterwards in the 2D Interpretation workflow at the 3D domain conversion from 2D Model.

Create the Extraction Area, press the **Save Extraction Area from 2D object** button in the **2D Line** module.

1/ Select the required 2D line set.
2/ Press OK.
3/ The survey can be resized in the map view of the 2D line set viewer to obtain a suitable result.

4/ Press the **Save** button or press **Ctrl+S** to save the survey.
5/ Select an output name and store the survey in the **Volume** tab of the Project Browser.
2D Multi-Z Picking

In order to insert no data areas inside the 2D Geomodel (to stop the horizon propagation against the body flanks), the Multi-Z picking on 2D Lines has been implemented. All required tools regarding the Multi-Z Picking are available from the Multi-Z Picking module.

The method for 2D Multi-Z creation, edition and saving is the same as for 3D Multi-Z picking. The resulting object will be saved in the 2D Multi-Z folder of the Geobody tab.
Stratigraphic Sequence Picking Based on a 2D Model

The 2D line Stratigraphic viewer can be used with 2D Models, to create sequence using 2D interpretation.

To open a 2D Stratigraphic viewer, click on the second icon called **New 2D Line Viewer...**

A new window appears in which a Geomodel has to be chosen (2D Model or 3D Model).

Once the Geomodel has been selected, click on **Ok**:
- For 2D Model: Drag and drop any 2D Line attributes.
- For 3D Model: Choose a 2D line to display the strati viewer. Click **OK**, then drag and drop any attribute.
Stratigraphic objects can then be created by using the **Stratigraphic Sequence Picking Mouse Mode** [J], similarly to Sequence picking on 3D Models.

The **Stratigraphic Sequence Picking Mouse Mode** is also automatically activated when a **New Sequence** is generated.
Indexed Color Objects Saving

Volume, horizon, horizon stack or 2D line opened in Color Blending Viewer can be saved as an indexed color object, regardless the selected color method.

The indexed color horizon, horizon stack or 2D line is saved with integer from 0 to 255 and -9999 as no value and can be exported to other application.

Indexed color volumes will be saved in 8 bit, stored with integer from 0 to 255. It allows visualizing the blending method without having to use the 3 channel inputs. The saved volume is so lighter than the original one but less precise and can be exported to other application as seg-y file.

The saving option related to indexed color object is available from the context menu of any color blending viewer.

To save the indexed color object,
1. Select a color bar; by default a 256 colors color bar is selected,
2. Click on Next,
3. Define the Top and Bottom of the indexed color Horizon Stack and click OK.

At the opening of an indexed color object, some display adjustments are required to display the best colors associated to the object:
1. From the Properties of the object, switch the Interpolation from Bilinear to None,
2. In the Color Bar, change the color range to -0.5 to 254.5.
RGB Color Blending view
Indexed Color view
Velocity Computation

Average, RMS and Dix velocities can be computed from input velocity volumes. First, choose a velocity type.

Then, select the input velocity volume and click on **Next**. Finally, enter an output name and click on **Finish** to compute the velocity volume.

**Average Velocity**

The average velocity $V_{avg,n}$ is the mean of velocities $v$ of upper layers from the input volume from $Z = 0$ to the depth $Z_n$:

$$V_{avg,n} = \frac{\sum_{i=0}^{n} v_i}{Z_n}$$
**RMS Velocity**

The Root Mean Square velocity $V_{rms}$ is computed from interval velocities values $v$ and the time intervals $τ$:

$$V_{rms,n} = \sqrt{\frac{\sum_{i=0}^{n} v_i^2 \tau_i}{\sum_{i=0}^{n} \tau_i}}$$

**Dix Velocity**

The velocities $V_{Dix}$ computed from the Dix formula are estimations of interval velocities based on RMS velocities $V_{rms}$:

$$V_{Dix,n} = \sqrt{\frac{v_{rms,n}^2 t_n - v_{rms,n-1}^2 t_{n-1}}{t_n - t_{n-1}}}$$
Cross Plots

Open Cross Plot from Logs

Cross plots can also be created using logs data. To create a new cross plot from logs interface, open the Cross Plot menu and press the cross plot from logs button.

Interface

The interface is composed of four main parts.

1. The input panel offers the possibility to choose the input properties, select the required log sets, define areas of interest and perform data selection.
2. The output panel allows to create logs containing the class value as integer value in order to obtain facies logs.
3. The log viewer offers a real time preview of the input data and the class logs corresponding to the selected classes in the cross plot viewer.
4. The cross plot viewer offers a view of the data in a X/Y/Colour domain.
Workflow

Input Panel
Several options are available to define the inputs used for the cross plotting and display them.

1. The first step consists in choosing the properties used for the cross plotting. A third property can be selected by checking the Color range check box. In such case, the third property will be displayed as a colour range in the cross plot viewer. It is also possible to apply a usual function to the input properties. The different functions available are log10(x), ln(x), exp(x), 1/x, sqrt(x), cos(x), sin(x) and tan(x).

2. The Display template offers the possibility to use a well template in order to have a similar display for every log of the same type in the log viewer.

3. This drop zone allows to choose the log sets used for the cross plotting. The cross plotting is multi well, which means that several log sets can be used in the same cross plot. If one of the selected sets does not contain one or several of the input properties, it will be ignored during the cross plotting and not displayed in the log viewer.
4. The Area of interest offers the possibility to select markers as depth limits for the cross plotting.
5. Pressing the REFRESH button will update the cross plot points according to the selected log sets and properties.
6. The selection tools allow to highlights certain parts of the cross plot by selecting areas in the log viewer.

Area of Interest
When selecting markers in the area of interest panel, the markers are displayed in the log viewer and used for the computation of the cross plot points. Three methods of bounding are available: between markers, above marker and below marker.

Selection Tools
The selection tools offer the possibility to select one or several parts of the logs in the log viewer. By toggling on Show the points of the corresponding area(s) are highlighted in red in the cross plot viewer. While Show is enabled, the highlighting is updated in real time when the selection is enlarged or erased. While Hide is on, the selection is disabled in the cross plot viewer.
1. Selection mouse mode which allows to select areas in the log viewer. (Pressing Shift using this mouse mode will enable the erase mouse mode).
2. Erase mouse mode which allows to erase parts of the selections.
3. Pressing the remove all button will clear all selections in the log viewer.

Class Selection and Retro-mapping
In the same way as volume cross plot viewers, classes can be created in the cross plot from logs. The retro-mapping is done in a new log named CLASSES in the log viewer. This log is updated in real time when the classes are modified.

It is possible to assign a lithologic pattern to every class in order to get a geological display. To update the lithologic pattern, open the dialog window by double clicking in the Style column corresponding to the required class.

Outputs
The class logs can be stored in the log database as logs of integers numbers corresponding to the class rank in the class table. The output logs are saved as property logs in the corresponding log sets with the selected name.
1. Select the area of interest where the outputs will be created. If no area of interest is selected, the class log is computed from the top to the bottom of the well.

2. Type the name of the log where you want to store the class logs.

3. Press save to store the computed class logs in the well database.

However, the lithologic style of the class logs can only be displayed in the cross plot from log interface. It is not possible to display it with the style in a classic log viewer.

**Save Cross Plot from Logs**

The cross plot from logs can be saved under the Project Browser. The saving preserves the input properties, the selected log sets, the classes, the display template and the regressions.
Automatic Classification

The **Classif auto** tab offers the possibility to automatically create classes using 2D machine learning methods: Self-Organizing Map (SOM) and K-Mean.

**Self-organizing map**: This method is an iterative process which start from a regular grid in the data space. The number or classes created after the iteration is defined with the parameter *Size X* multiplied by *Size Y*.

**K-Mean**: This method starts from a number of nodes defined by the parameter *Nb nodes*. An iterative process classifies the data space and creates a number of classes equal to the number of nodes.

Regressions

The **Regression** tab offers the possibility to create regression curve according to usual formulas. The different available formulas are:

- **Linear**
  \[ y = a \times x + b \]
- Logarithmic: \( y = a \log(x) + b \)
- Exponential: \( y = a \exp(bx) \)
- Power: \( y = a x^b \)
- Polynomial: \( y = \sum a_n x^n \)

Pressing the Create button will generate the curve which best fit the cross-plot data according to the selected curve type. The curve is then displayed in the viewer and added to the regression table.
1. Check to hide/show the curve in the viewer.
2. Double click to change curve color.
3. Double click to rename the curve.
4. Double click to update the curve parameters.
5. Press to remove the selected curve(s).
6. Press to remove all the curves.
7. Press to save the curve as a formula in the project (the formulas can then be opened in the calculator).
OpenWorks® Data Link

Introduction

The OpenWorks® data link is an essential tool for Decision Space® users willing to maximize benefits from the PaleoScan™ interpretation. The data link provides a rapid exchange of data such as volumes, faults, horizons, wells and surface picks (markers).

It is available from PaleoScan™ and it enables seamless data exchange between OpenWorks® and PaleoScan™ and vice versa, saving considerable time in data manipulation. Data are fetched from one application and sent to the other on-demand, with the OpenWorks® data link, showing the two project databases in their data tree.

This OpenWorks® and PaleoScan™ data exchange tool is developed and distributed by Eliis.

Requirements

<table>
<thead>
<tr>
<th>PaleoScan™ - OpenWorks® data link</th>
</tr>
</thead>
<tbody>
<tr>
<td>Required PaleoScan™ versions</td>
</tr>
<tr>
<td>PaleoScan™ 2018.1 (64-bit) and above</td>
</tr>
</tbody>
</table>

This plug-in is launched from PaleoScan™ and communicates with OpenWorks® database. The two applications do not necessarily need to be launched on the same computer.

Getting Started

Access OpenWorks® Data Link

The OpenWorks® data link is available from PaleoScan™. First of all, to use the data link, the OpenWorks® license needs to be enabled from the Extensions tab in PaleoScan™.
The data link can then be opened from the File menu by clicking on the **OpenWorks® Data Link** option or a specific **OpenWorks® Icon** is dedicated to it in the main toolbar.

A new interface will open in which the user will be able to select a PaleoScan™ project and an OpenWorks® database to exchange data.

Note that all or several objects of the same nature can be selected in once by using the windows shortcut **CTRL+A** or **CTRL+click** respectively.

Projects Management

Once the OpenWorks® data link is opened, a PaleoScan™ project and an OpenWorks® database need to be opened to exchange data.

**PaleoScan™ Projects**

A PaleoScan™ project can be chosen by clicking on the **magnifying glass**, on the right hand part of the interface.
The list of the PaleoScan™ projects available refers to the list of projects available in PaleoScan™ by clicking on the **File - Open** tab. Hence, this list can be managed from the PaleoScan™ application itself.

Once the list of projects has been managed in PaleoScan™, click on the **pull down** menu to have access to a specific project.

**OpenWorks® Projects**

The list of the OpenWorks® district available in the pull down menu represents all the districts readable on the network. Once the district has been selected, all the projects within this district are accessible.
PaleoScan™ to OpenWorks®

Open the OpenWorks® data link and select a PaleoScan™ project to visualize its database. Several objects of the same nature can be exchanged at once or separately. Note that if an object with a standing name already exists in the OpenWorks® database, the data link will return an error message. Prior importing any objects, make sure the name of the transferred files is free.

3D Volumes
Select the 3D volume(s) from the PaleoScan™ database and click on lower green arrow to transfer the volume(s) from PaleoScan™ to OpenWorks®.

Since the Relative Geological Time model (RGT) is a 3D volumes, RGT volumes as well as seismic and attributes volumes can be transferred at once.

Faults
To have access to the Fault database, click on the pull down menu and select the Fault tab.
Select the **fault set** from the PaleoScan™ database and click on the **lower green arrow** to transfer the fault set.

At this point, a **survey** needs to be assigned to the fault set. If a 3D volume is already available in the OpenWorks® project, it can be dragged and dropped in the dedicated dropping box.

Note that several fault sets can be exchanged at once.
In such a case, one survey per fault set can be assigned by dragging and dropping every single survey in each dropping box.

A single survey can also be assigned to the entire fault sets. If there is no 3D volume existing in the OpenWorks® database, a survey can be picked from the PaleoScan™ database by changing the survey source from OpenWorks® to PaleoScan™.
Once the survey has been chosen, click on **OK** to launch the transfer. Note that all the faults contained in the fault set(s) will be imported as independent faults in the OpenWorks® database.

**Horizons**

To have access to the Horizon database, click on the pull down menu and select the **Horizon** tab.

Select the horizon(s) from the PaleoScan™ database and click on the lower green arrow to transfer the horizon(s) into OpenWorks®.
At this point, a survey needs to be assigned to every single horizon. If a 3D volume is already loaded in the OpenWorks® project, it can be dragged and dropped in the dedicated dropping area called “Apply to all”. This way, the survey will be assigned to the entire set of horizons.

If there is no 3D volume in the OpenWorks® database, a survey can be selected from the PaleoScan™ database by changing the survey source from OpenWorks® to PaleoScan™.

Once the survey has been chosen, click on OK to launch the transfer.
Wells
To have access to the Well database, click on the pull down menu and select the Well tab.

Select the well(s) from the PaleoScan™ database and click on the lower green arrow to transfer the well(s) from PaleoScan™ to OpenWorks®.

A well list needs to be assigned to every single well. If a well list is already available in the OpenWorks® project, you can drag and drop it into the dropping area. If there are no well lists available, a new one can be created straight from the interface by clicking on the button “Create new Well list”. A name has to be given to the well list, as well as an Interpreter. The created well list can be assigned to all the new wells by toggling on the dedicated option called “Assign to all new wells”. Once it is done, click on Create to generate the well list.
Several well lists can be created and assigned to specific wells. When transferring the wells, all the data associated to the well are transferred at once (trajectory, deviation, time-to-depth relationship, markers and checkshots).
If several time-to-depth relationships are associated to a well, all of them will be transferred and the previously assigned time-to-depth relationship will be maintained.

**Surface Picks (Markers)**

This tab, available from the OpenWorks® project side of the interface (left hand side), is useful to see if all markers are well imported in the OpenWorks® database. To have access to the marker database, click on the **pull down menu** and select the **surface picks** tab.

This tab is useful to check the surface picks import done while transferring wells. If all the markers are present in the surface pick tab, it means that the transfer was successful. PaleoScan™ will preserve the color of each marker.
OpenWorks® to PaleoScan™

Open the OpenWorks® data link and select an OpenWorks® project to visualize its content. Several objects of the same nature can be exchanged at once or separately. Note that if an object with a standing name already exists in the PaleoScan™ database, the data link will return a “file conflict” and the application will ask you to either enter a new file name or to overwrite or skip.

3D Volumes

Select the 3D volume(s) from the OpenWorks® database and click on upper green arrow to transfer the volume(s) from OpenWorks® to PaleoScan™.

Faults

To have access to the Fault database, click on the pull down menu and select the Fault tab.
Then, select the fault(s) from the OpenWorks® database and click on upper green arrow to transfer the fault(s) from OpenWorks® to PaleoScan™.

Horizons
To have access to the Horizon database, click on the pull down menu and select the horizons tab.
Then, select the **horizons(s)** from the OpenWorks® database and click on **upper green arrow** to transfer the horizon(s) from OpenWorks® to PaleoScan™.

**Wells and Surface Picks (Markers)**

In order not to corrupt the well database, we strongly recommend you to close all the PaleoScan™ applications pointing at the used project. To have access to the Well and Marker databases, click on the **pull down menu** and select the **"wells"** tab.

Then, select the **well(s)** from the OpenWorks® database and click on **upper green arrow** to transfer the well(s) from OpenWorks® to PaleoScan™.
When transferring the wells, all the data associated to each well are transferred at once (trajectory, deviation, time-to-depth relationship, surface picks and checkshots). If several time-to-depth relationships are associated to a well, all of them will be transferred and the previously assigned time-to-depth relationship will be maintained.