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, 3 Rue Jean Monnet, 34830 Clapiers, 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

For any information request, you can contact us.

Web: www.eliis.fr

Europe - Montpellier
Eliis SAS
contact@eliis.fr
+33 (0) 4.67.41.31.16

North America - Houston
Eliis Inc.
contactus@eliis.fr
+1 281 404 1515

Australia - Perth
Eliis Pty Ltd
contactau@eliis.fr
+61 466 303 546

Malaysia – Kuala Lumpur
Eliis Sdn Bhd
contactmy@eliis.fr
+60 162 072 710

Brazil – Rio de Janeiro
Eliis Ltda
contactbr@eliis.fr
+55 (21) 99575-0071

China - Beijing
Geoseismic Signal Corporation
tonyqiu@gssgeo.com
+86 10 8478-3912

Mexico - Merida
Apogee
info@apogeego.com
+52 1.993.113.6525

Senegal - Dakar
WellCome Energy Services
olivier.malinur@eliis.fr
Licensing

PaleoScan™ 2019 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™ 2019. 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 PaleoScan™ projects.

Forward compatibility:
Projects saved in previous versions of the PaleoScan™ can be updated in PaleoScan™ 2019, upon loading of the projects.

Backward compatibility:
Projects created in the PaleoScan™ 2019 can also be opened in previous version (2018 or 2017). 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 the minimum requirements equivalent to the below mentioned items:

- CPU: 6-Core
- RAM: 16 GB
- Operating System: Windows® 7, 8 or 10 (64-bit)
- Graphic card: 512 MB NVIDIA® / ATI® graphic card
- IDE devices: Hard disk with fast rotational speed (> 7200 rpm)
# Table of Contents

PALEOSCAN™ 2019 6

LICENSE MANAGEMENT 7
  License Manager 7

PROJECT MANAGEMENT 7
  Open Project 7

DATA IMPORT/EXPORT 8
  3D Volume SEG-Y Import 8
  GeoCellular Grid Import 8
  Fault Import 10
  SEG-Y Export 10
  GeoCellular Grid Export 10

VIEWERS 12
  2D Viewer 12

GENERAL TOOLS 14
  Polylines 14
  Shortcut 15
  Test Space Disk 15

2D LINES 16
  2D Line Conditioning 16

ATTRIBUTES 18
  Attribute List 18
  Elimination Attributes 18
  Basic Attribute 18
  Frequency Decomposition 19
  Band Pass Attribute 19
  Coloured Inversion 19
  Deterministic Inversion 20
  Synthetic Wavelet Creation 21

MODEL-GRID 22
  2D Model-Grid 22
  3D Geo-Model 22
  Model-Grid Horizon Editing 24
  Horizon Constraint 24

HORIZON / HORIZON STACKS 25
  Horizon Truncation 25
  Output parameters 26
  Horizon Contouring 27
  Isochore Map 27
AUTOMATIC FAULT EXTRACTION (AFE) 28
  Introduction 28
  Workflow Summary 28
  Fault Plane Attribute 29
  Fault Thinning Attribute 31
  Fault Extraction 32

STRUCTURAL INTERPRETATION 35
  Faults Sets in Project Browser 35
  Save Faults in Extraction Area 35
  Remove Hidden Faults 36

WELL/LOG MANAGEMENT 38
  Well QC Table 38
  Composite Log Creation 38

GEOBODY - LAYERS 39
  Geobody Polygons 39
  Geobody Classification 40
  Geobody/Layer Splitting 40

ADVANCED INTERPRETATION 41
  Truncation Option 41
  Strati Viewer 44
  Sequence Layering 47
  Horizon Stack from Sequence 48
  GeoModel from Sequence 51

GEOCELLULAR GRID 53
  GeoCellular Model from Sequence 53
  Properties 57
  Save GeoCellular Model 60
  Property Mapping 61
  GeoCellular Model Extraction 62

PROPERTY MODELLING 64
  Interpolation Method 64

SEISMIC WELL-TIE 65
  Log Display 65

VELOCITY MODELLING 66
  Velocity Type 66

PYTHON 67
  Introduction 67
  Python Documentation 67
  Writing and Executing Python 69
  Custom Attribute with Python 75
  Python Module Finder 77
PaleoScan™ 2019 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™ 2019. A detailed description of each tool can be found in the “New Features Guide” or on the web site (www.eliis.fr).

The 2019 release brings more options to complete the workflow from the exploration to the reservoir characterization, with various new tools: Automatic Fault Extraction, GeoCellular Grid, Stratigraphic Truncation …
License Management

License Manager

While in the License Manager interface, the most potential cause of error linked to server and local licensing (when the licensing tools must be updated) have been made more explicit.

Project Management

Open Project

Project Information details are now available upon Project Opening:
- Added information about the need of update related to the PaleoScan project version.
- Added information when a project is opened by another user.
Data Import/Export

3D Volume SEG-Y Import

The Decimation interface (in the Options tab of the seismic import interface) has evolved. Decimation can be applied during the SEG-Y import. From the Options tab, select the resolution for spatial and vertical directions.

<table>
<thead>
<tr>
<th>Zone of Interest</th>
<th>Spatial Resolution</th>
<th>Check Dimensions</th>
</tr>
</thead>
<tbody>
<tr>
<td>1 line resolution (lnl)</td>
<td>1/3</td>
<td></td>
</tr>
<tr>
<td>1 line resolution (xlnl)</td>
<td>1</td>
<td></td>
</tr>
<tr>
<td>Vertical resolution (zdim)</td>
<td>1/3</td>
<td></td>
</tr>
</tbody>
</table>

Percentage of file to load from the first trace

For instance, if it is set up at 1/3, it means that PaleoScan™ will load one line out of three for each direction (Inline, Xline and Time Slice).

GeoCellular Grid Import

GeoCellular Grid is a new item that can be generated and handled in PaleoScan.

GeoCellular grids of Corner Point type can be imported by selecting the GeoCellular Model option in the Import menu. Import is also available from the context menu (right click on GeoCellular Model in the Project Browser).

The Advanced Interpretation extension is needed and should be activated to access the GeoCellular Model import application.
From the GeoCellular Model import interface, select the file format to import. In PaleoScan™ 2019, only **Eclipse Grid file format (ASCII) GRDECL** is managed. Click on OK to select the input file on the Project Browser.

Once input file has been selected, PaleoScan™ parses it and detects the grid spatial unit (specified after `GRIDUNIT` keyword). An interface pops up. Type the grid name, specify the vertical domain and the vertical unit of the grid. Click on **OK** to launch the import.

In **GRDECL format**, grid geometry is specified by two keywords: **COORD** and **ZCORN**. **COORD** integrates the coordinates of the top and the bottom of each pillar while **ZCORN** refers the Z coordinate of each corner. The exact coordinates of a single corner is given by intersecting the corresponding pillar and the z-value given in ZCORN. Refer to Eclipse software documentation to get information on the format.

Grid is imported using the global coordinates (it does not read **MAPAXES** keyword) so make sure the import file is not defined in local coordinates.

Grid orientation should be adapted to PaleoScan™ conventional orientation: I index increases toward the East, J index increases toward the North and K index increases downward.

**SPECGRID** (grid dimensions specification), **COORD** and **ZCORN** are Eclipse keywords that are mandatory for a basic grid import in PaleoScan™.

Importing properties for an existing grid is possible by using the context menu (right click on the grid name) from the Project Browser.
### Fault Import

Interpreted faults from another interpretation platform can be imported in PaleoScan™ using different formats. Click on the 3D Fault option in the Import menu.

For all import formats, once the import is done, a message box will inform you that the faults are successfully imported. Faults are stored in the Fault tab of the Project Browser. For all imported faults, the default method of gridding between the created sticks now corresponds to a minimum envelope. The interpolation size of this gridding is automatically adapted to link all sticks of a given fault. The gridding method and relative parameters can later be modified via the Properties panel.

### SEG-Y Export

As for the SEG-Y import, the decimation tool for the export of volumes was updated. The decimation tool is available from the Options tab of the SEG-Y export interface.

### GeoCellular Grid Export

To export a GeoCellular Model, click on the Export button from the general toolbar and select the GeoCellular Model option.

The Advanced Interpretation extension is needed and should be activated to access the GeoCellular Model export application.
GeoCellular Model and GeoCellular Model properties can also be exported using the context menu (right click on the GeoCellular Model’s or the GeoCellular Model Property’s name) from the Project Browser.

GeoCellular grids will only be exported in **Eclipse Grid file format (ASCII) GRDECL**. Select the Eclipse Grid file format (ASCII) GRDECL and then click on **OK**.

If the option **Create one file** is activated, it results in the creation of one single export file. Otherwise, PaleoScan™ will generate 5 or 6 files (MAIN, COORD, ZCORN, ACTNUM, PROPS and the FAULTS files). Define the output unit settings and select the output folder. The coordinates of the grid will be converted and exported in the units defined in the settings. Then click on **OK** to launch the export.
Viewers

2D Viewer

Scale Display

In order to lighten axis information, the display of InLines, XLines and Time-slice are hidden by default in the 2D viewers.

One can choose to display them again, by going in Tools, Settings, 2D Viewers section, All Viewers, XLine/Time Slice/InLine, DisplayMode and switch the default option to All. The choice has to be saved by clicking on Save. The option will be applied upon opening a new 2D viewer.

For a given 2D viewer, the Property can also be temporarily changed from the Properties panel, XLine/Time Slice/InLine section, DisplayMode option.
Proportion display

In the Properties panel, the H/V ratio relative to 2D viewers was improved with decimal values to better manage Depth data.
Polylines

Open Polylines

From the Main toolbar, it is now possible to create open Polylines.

Polylines from 3D Object Intersection

A new option is available to save a Polyline from 3D intersection. Polylines can be created from 2D/3D objects intersecting viewers. Managed 2D/3D objects and viewers are listed in the below table:

<table>
<thead>
<tr>
<th>Object Type</th>
<th>3D Volume viewer (L, R)</th>
<th>3D Volume viewer (2D)</th>
<th>2D Line viewer</th>
<th>2D Horizon viewer</th>
<th>2D Horizon Stack viewer</th>
<th>3D Fault viewer</th>
</tr>
</thead>
<tbody>
<tr>
<td>2D Curve</td>
<td>x</td>
<td>x</td>
<td>x</td>
<td>x</td>
<td>x</td>
<td>x</td>
</tr>
<tr>
<td>2D Volume</td>
<td>x</td>
<td>x</td>
<td>x</td>
<td>x</td>
<td>x</td>
<td>x</td>
</tr>
<tr>
<td>2D Fault</td>
<td>x</td>
<td>x</td>
<td>x</td>
<td>x</td>
<td>x</td>
<td>x</td>
</tr>
<tr>
<td>2D Horizon</td>
<td>x</td>
<td>x</td>
<td>x</td>
<td>x</td>
<td>x</td>
<td>x</td>
</tr>
<tr>
<td>3D Volume</td>
<td>x</td>
<td>x</td>
<td>x</td>
<td>x</td>
<td>x</td>
<td>x</td>
</tr>
<tr>
<td>3D Fault</td>
<td>x</td>
<td>x</td>
<td>x</td>
<td>x</td>
<td>x</td>
<td>x</td>
</tr>
<tr>
<td>3D Horizon</td>
<td>x</td>
<td>x</td>
<td>x</td>
<td>x</td>
<td>x</td>
<td>x</td>
</tr>
<tr>
<td>3D Volume</td>
<td>x</td>
<td>x</td>
<td>x</td>
<td>x</td>
<td>x</td>
<td>x</td>
</tr>
<tr>
<td>3D Fault</td>
<td>x</td>
<td>x</td>
<td>x</td>
<td>x</td>
<td>x</td>
<td>x</td>
</tr>
<tr>
<td>3D Horizon</td>
<td>x</td>
<td>x</td>
<td>x</td>
<td>x</td>
<td>x</td>
<td>x</td>
</tr>
</tbody>
</table>

To create and save the polyline corresponding to the intersection between a 3D fault and a 3D horizon stack:
1. Select the fault within the active Horizon Stack viewer;
2. Click on Save Polyline, then choose the format(s) of interest and define an output name;
3. Once saved within the database, the 3D polyline corresponding to the intersection between the 3D fault and the 3D horizon stack can be displayed in 3D.

NOTE: Several objects with different classes can be selected at the same time. When creating/saving the polylines, individual files with a common prefix name are created.

Shortcut

A new shortcut “Escape” to switch between the two latest mouse modes (e.g. Selection mouse mode, Cross-navigation mouse mode) is now available.

Test Space Disk

A space disk test is now performed on the main PaleoScan™ tools to check the free space before launching computations:

- 2D and 3D Model from Horizons
- All 3D Models computation
- Data Mapping on patches
- Absolute Ages Model
- 3D HS/HS from Strati
- Geomodel from Sequence
- 2D Line Set Creation
- Inversion
- Spectral Blueing
- Colour Inversion
- Convolution
2D Line Conditioning

New tools are now available to crop 2D Lines vertically and spatially. The **2D Lines Conditioning** tool is useful for cropping or splitting 2D Lines. In order to compute a 2D Model-Grid, lines should have less than 30,000 traces for performance reasons.

The **2D Lines Conditioning** window is available from the 2D Line toolbar or by right clicking on lines from Project Browser and select **Condition**...

Drag and drop the lines to condition or click on "Add items..." to select lines. Then click on each line to independently parameterize the vertical cropping or spatial splitting.

**Vertical Cropping**

To apply a vertical cropping, check the **Vertical** option in the **Parameters** section. Then, define the minimum and maximum Z values or click on the mouse mode "Move Sliders" and select the Zmin/Zmax lines to move them.
Spatial Splitting

To apply a spatial splitting, check the **Spatial** option in the **Parameters** section. Then, define the number of splits: this number is limited to 5 sliders and an information of the number of traces of each future 2D line is displayed. Click on the mouse mode **"Move Sliders"** to move the sliders.

Finally, click on **Save**. The split lines will have one trace in common for intersection, especially for 2D lines sets creation.
Attributes

Attribute List

For more convenience, the Attribute List is now sorted by alphabetic order.

Elimination Attributes

Previous “Truncation” and “Truncation 3D” attributes renamed Elimination and Elimination 3D respectively.

Basic Attribute

Polarity

The Polarity highlights the peak reflections sign.
Frequency Decomposition

Morlet wavelet added in the Frequency Decomposition tool.

Band Pass Attribute

Morlet wavelet added in the parameters of Band Pass attribute computation.

Coloured Inversion

New parameters are available from the Low frequency model tab of the Coloured Inversion interface to define the percentage of deviation from the a priori model and to activate the low-pass filtering.
Deterministic Inversion

From a low-frequency a priori model of acoustic impedance, the Deterministic Inversion tool computes an absolute acoustic impedance model. Two methods are available: Model-based and Recursive. Click on the Deterministic Inversion button in the Attribute toolbar:

Theory

The Model-Based method uses an a priori model, low-frequency filtered, and a wavelet to compute a synthetic seismic image. The misfit between the real and the synthetic seismic is then estimated to update the initial input AI model. This process is repeated in order to get a final AI inverted model and its corresponding seismic volume.

The Recursive method integrates the seismic traces and adds the a priori model according to a certain deviation to compute an absolute AI model.

Interface

First, drag and drop the input seismic volume and the method to apply. Then, drag and drop the a priori model. An a priori model can be computed from the Properties Modeling tool. The deviation value corresponds to the maximum of allowed
variation for the model values in order to constrain the output model. A Hann low-frequency filtering can also be applied.

The Model-based method requires more parameters:
- Drag and drop a wavelet, which can be extracted from the volume using the Wavelet Creation tool.
- Select the number of iterations. WARNING: a lot of iterations will generate a time-consuming process.
- Click on Preview to display the result of the Deterministic Inversion. The preview is a blending viewer, thus move the slider to display, more or less, the seismic image and the acoustic impedance a priori model, acoustic impedance preview or synthetic result for model-based method.
- Finally, click on Save to compute the AI output model. For model-based method, a synthetic volume will also be generated.

**Synthetic Wavelet Creation**

The Synthetic wavelet creation tool is now duplicated into the Attribute module.
2D Model-Grid

On top of regular parameter settings (e.g., Patch Size, Polarity), decimation option was added to the 2D Model-Grid computation tool.

3D Geo-Model

A new method to compute 3D Geo-Model based on the Marked Horizons only is now available.

The Marked Only 3D Geo-Model computation only accounts for the marked horizons of the Model-Grid to constrain the creation of the Geo-Model. All other constraints are ignored. Where no interpretation is marked, the geometrical and thickness structure of the model is built from the interpolation of the thickness relationships where the horizons are marked (figure below). This method has the advantage of allowing the interpreter a better control of the Geo-Model geometry in that he can select the most reliable reflectors (corresponding to reliable series of connected patches) to constrain the model. Hence, poor connections (e.g., in a noisy or low amplitude seismic intervals) are not impacting the geometry of the Geo-Model.
To launch the Marked Only 3D Geo-Model computation,

1. From the **Model-Grid** toolbar, click on the **Geo-Model Creation** button and select the **Marked Only 3D Geo-Model** option.

2. In the displayed interface, adapt the Smooth Level (0 corresponding to no smoothing), and toggle off the **Fill Survey** option if you want the Geo-Model inexistent where the survey includes no data.

3. If necessary, drag and drop multi-Z in the Exclusion Zone to avoid creating the Geo-Model at the Multi-Z location.

4. Indicate by toggling the corresponding options if you want to create the associated X-Line and Time-slice volumes and a Horizon Stack whose default boundaries would be the top and bottom of the Geo-Model, then click on Save.

5. Assign a relevant name to the Geo-Model and click **OK** to launch the computation.

6. The created Geo-Model is stored in the Model folder of the Volume tab of the Project Browser.
Model-Grid Horizon Editing

In the Model-Grid toolbar, the former icon labelled Clear was replaced by Horizon Validation.

---

Horizon Constraint

In the Grid Constraints tool, Horizon Constraint interface, the option Respect Locked Horizons now activated by default.
Horizon / Horizon Stacks

Horizon Truncation

The new Horizon Truncation tool is useful to truncate one horizon from another one. Click on the Horizon Truncation button in the Horizon module toolbar.

Inputs

1- Drag and drop a volume to display the background image.
2- Drag and drop the truncated horizon: the horizon which will be truncated (displayed in green).
3- Drag and drop the reference horizon: the horizon which will be used as reference for the truncation (in yellow).
Truncation Options

If the Reference horizon is above the Truncated horizon, the truncation will be a Toplap termination. Else if the Reference is below the Truncated horizon, the truncation will be a Downlap or Onlap termination. If they intersect, the mode can be manually defined.

The Threshold value is the thickness between the two horizons where the truncation is initiated. The Distortion strength defines the inclination of the slope for the truncation.

Output parameters

Select the Fill truncated areas option to populate the truncated horizon with the Z values of the reference horizon. The preview of the truncated horizon is displayed in blue.

Finally, click on OK to display the result in 3D and/or save the truncated horizon.
**Horizon Contouring**

The Horizon Contouring option is henceforth computed on the data mapped on the horizon.

**Isochore Map**

An information was added regarding the expected Isochore units in the Isochore creation wizard.
Automatic Fault Extraction (AFE)

Introduction

Fault interpretation prior to seismic analysis is essential to constraint the horizon interpretation and understand reservoir sealing properties. This process has until now been tedious and time consuming as the interpreter must manually pick fault sticks based on seismic discontinuities. In PaleoScan™, faults are essential to constrain the building of the Model-Grid as they will indicate planes where the automatic propagation of the horizons of the grid should be prevented.

We describe here a workflow to automatically detect and extract faults from a seismic volume through an optimized processing where the user keeps control of the workflow.

Workflow Summary

The automatic fault extraction workflow is split in three main steps that are presented here and explained in further details in the next sections.

First, the variance is computed from an input volume (commonly the seismic - Fig. 1a) to derive the Fault Plane attribute (Fig. 1b), representing the probability of the vertical component of the deformation. Secondly, the Fault Plane attribute is processed to yield the Fault Thinning attribute (Fig. 1c), representing the skeleton of this deformation. Eventually, fault patches are extracted from the Fault Thinning attribute via a labelling process (Fig. 1d).

In PaleoScan™, the three steps of the workflow are performed with three distinct attributes prefixed “AFE” (standing for “Automatic Fault Extraction) as follows:

1. AFE - Fault Plane, computed from Seismic volume
2. AFE - Fault Thinning, computed from Fault Plane volume
3. AFE - Fault Extraction, computed from Fault Thinning volume.

NOTE: In order to perform the whole workflow in one time, the interface of the attributes used in the first steps of the workflow includes the possibility of creating the subsequent attributes with their respective parameters.
Fault Plane Attribute

Theory

The Fault Plane attribute uses the variance attribute, which is processed to get a volume of deformation probability extrema. The processing method is described as follows:

At each voxel location of the variance volume (Fig. 2a), a scanning disk is rotated in a series of directions to scan the variance (Fig. 2b). The disk size is customizable depending on the dimension of the targeted faults. The orientation of the disk ranges from 0 to 360° in azimuth with a step of 10°, and from a minimum customizable value to 90° in dip with a step of 5°. For each disk orientation, the sum of the variance along the disk is computed, weighted with a Gaussian function (i.e. the summed Variance is more weighted at the centre of the disk, and minimum to its edges).

The outputted Fault Plane attribute (Fig. 2c) consists in a volume of variance sum extrema, where the zones of maxima highlight the location and geometry of the potential faults.
**Interface**

The Fault Plane attribute can be generated from a seismic volume. To compute it, go to the Fault module and select the option **From Seismic** in the pull-down menu of the Fault Auto button:

It opens the Attribute Computation interface in which the attribute **AFE - Fault Plane** is directly selected:

1. Click on the magnifying glass on the right side of the input field to display the possible inputs in the Project Browser. Drag from the Project Browser (preferentially) a seismic volume on which the variance and the Fault Plane must be computed and drop it in the input field.

2. Adapt the attribute parameters:
   - The **Window size** corresponds to the size of the scanning disk (in sample) used to extract the maximum sum of the variance at a given voxel location. Increasing its value allows highlighting synthetically the location of faults of large dimension. Reducing the value may highlight more details in the deformation but increases the probability of highlighting noise.
   - The **Minimum dip (°)** corresponds to the minimum dip that the scanning disk will scan. All orientation combinations involving a dip lower than this value will be ignored, thus reducing the computation time of the attribute. For example, using the default value of 70°, PaleoScan™ will only scan variance sums along disk orientations whose dip is comprised between 70° and 90°.
   - The **Fault Thinning** and **Fault Extraction** parameters are detailed in the next relative sections.

3. Cross-navigate (shortcut [G]) to the desired location on the input volume displayed on the left panel and hit the PREVIEW button to visualize the preview of the Fault Plane around the cursor location in the panel to the right.
4. When satisfied with the Fault Plane parameters, assign an Output name. If necessary, check the corresponding boxes to create the desired dimensions (the Inline creation is mandatory). Click Run to launch the attribute computation.

Fault Thinning Attribute

Theory

The Fault Thinning attribute uses as an input an attribute highlighting the location of the potential faults. Although the input attribute is not imposed, we recommend using the Fault Plane attribute, specifically designed to identify the fault locations. The Fault Thinning aims at isolating the “skeleton” of the vertical component of the deformation. For each pixel of each time-slice of the input volume, a Hessian matrix is used on a sampling window (Fig. 3a) of a few pixels in width to locally find the direction of the highest gradient (most normal to segments of maxima). The value of the considered pixel of the input volume is preserved if the pixel has the highest value along the highest gradient direction in the sampling window (Fig. 3b), else a no data value is assigned to the pixel. The output of this process is the Fault Thinning attribute, representing a pixelized skeleton of the deformation.

Figure 3. Detail of the Fault Thinning attribute processing. (a) The input Fault Plane attribute in Time-slice and the sampling window on which the Hessian matrix H is applied. (b) Fault Thinning output.

Interface

The Fault Thinning attribute can be generated from a Fault Plane volume. To compute it, go to the Fault module and select the option From Fault Plane in the pull-down menu of the Fault Auto button:

It opens the Attribute Computation interface in which the attribute AFE - Fault Thinning is directly selected:

1. Click on the magnifying glass on the right side of the input field to display the possible inputs in the Project Browser. Drag’n drop (preferentially) a Fault Plane volume to the Input field.
2. The **Window size** parameter corresponds to the size of the sampling window of the Hessian matrix (in sample) around the considered time-slice pixel. Increase its value if you are expecting loose fault networks; reduce the value if you are expecting dense fault network.

   The **Fault Extraction** parameters are detailed in the next section.

3. Cross-navigate to the desired location on the Fault Plane volume displayed on the left panel and hit the PREVIEW button to visualize the preview of the Fault Thinning around the cursor location in the panel to the right.

4. Assign an Output name. If necessary, check the corresponding boxes to create the desired dimensions. Click Run to launch the attribute computation.

**Fault Extraction**

**Theory**

The Fault Extraction is performed from the Fault Thinning attribute. During this step, the pixelized skeleton is transformed into discrete elementary patches of a few pixels in dimension (Fig. 4a). The minimisation of a best cost function accounting for the distance and the planarity between the elementary patches is used to label the patches as pertaining to the same future fault or not (Fig. 4b). To avoid the creation of faults corresponding to noise information, a threshold can be applied to prevent the creation of patch wherever the value of the Fault Thinning would be too small. As well, too small groups of patches (same label) can be discarded. The fault extraction is achieved via the creation of fault sticks every 5 lines in the direction most perpendicular to the main trend of the Fault Thinning skeleton (Fig. 4c).
Figure 4. Fault Extraction method. (a) Elementary patch creation (coloured patches pointed by the arrows). (b) Schematic diagram showing the labelling process. (c) Fault Stick creation.

Interface

The Fault Extraction is performed from a Fault Thinning volume. To undertake a fault extraction, go to the Fault module and select the option From Fault Thinning in the pull-down menu of the Fault Auto button:

From the Attribute Computation interface, the attribute AFE - Fault Extraction is directly selected:

1. Click on the magnifying glass on the right side of the input field to display the possible inputs in the Project Browser. Drag’n drop a Fault Thinning volume to the Input field.

2. Adapt the attribute parameters:
   - The Minimum window size corresponds to the minimum number of voxels (too small groups of elementary patches) that should be accounted for the fault
creation. Increase its value if you want to preserve only the biggest faults; reduce the value if you want to include the smallest faults.
- The **Minimum thinning value** is the threshold value below which the Fault Thinning information is ignored for the elementary patch creation and therefore for fault extraction.
- The **Fault smoothing** allows modifying the smoothness of the created faults. The smoothing is minimum with a value of 1 and increases with increasing value.

3. Cross-navigate to the desired location on the Fault Thinning volume displayed on the left panel and hit the PREVIEW button to visualize the preview of the extracted faults. As the previewed faults are temporarily created in 3D around the cursor location, the fault intersection is displayed on both the left and right panels.

4. Assign an Output name. Click Run to launch the fault extraction.

The created faults are stored in the Project Browser, Fault tab, under the form of fault sets including up to 2000 faults per set. Depending on the number of detected faults, an unlimited number of fault set can be created.
Faults Sets in Project Browser

The display of fault sets in the Project Browser has been improved: they are now displayed as a tree with the names of the faults when they were specified.

Save Faults in Extraction Area

It is possible to save faults located in a specific area as a new fault set. The faults and an extraction area must be opened in the 3D Viewer. The extraction area must be selected:

Then click on **Save Faults in Extraction Area** from the **Fault** toolbar. In the saving dialog box, give a name to the new fault set and click on OK:
All the faults intersecting the extraction area are saved:

Faults that intersect the AOI have been saved in the new fault set.

Greyed faults don’t intersect the AOI, they haven’t been saved.

**Remove Hidden Faults**

A new option to remove the hidden faults from 3D was added to the Fault module toolbar. This option was created to help managing the potentially dense faultsets created with the AFE workflow.

Hidden faults can be removed from the 3D Viewer. A fault is considered as hidden if it is unchecked in the 3D Objects List or if it is filtered by the Dip Azimuth tool.
In order to remove the hidden faults from the 3D Viewer, click on Remove Hidden Faults from 3D from the Fault toolbar:

A message indicates how many faults will be removed. Click Yes and the faults will be removed from the 3D Viewer:
Well QC Table

The Markers QC Table can now be exported thanks to Export Well QC button. The table is exported with the .txt format.

Composite Log Creation

New output logs were added to the Composite Logs Creation tool (Well/Log module):
- Young modulus,
- Poisson’s ratio.

Young Modulus: The Young modulus, $E$ in GPa, is a measure of elasticity of the medium.

$$ E = \rho V_s^2 \times (3 \rho V_p - 4 \rho V_s) / (\rho V_p - \rho V_s) $$

Poisson Ratio: The Poisson ratio, $\nu$, represents expansion of the medium in directions perpendicular to the direction of compression.

$$ \nu = \frac{1}{2} \times ((\rho V_p)^2 - 2(\rho V_s)^2) / (\rho V_p - \rho V_s) $$
Geobody Polygons

On the “Build Geobody from Horizon Stack” interface (Geobody Manual module), a new option is available to save geobody polygons at the Geobody creation.

Note that, in 2D viewers, you can also save the polylines defined by the intersection between a Geobody and the Horizon Stack where they come from by selecting the displayed polyline (double-click) and clicking on the Save Polyline tool from the main toolbar.
Geobody Classification

There are now two kinds of surface information in the Geobody Classification tool:
- Total Surface
- Projected Surface.

Geobody/Layer Splitting

The Geobody Splitting tool has been improved with the possibility to specify the lowest and highest threshold values.
Truncation Option

New Truncation options were implemented to manage stratigraphic terminations into PaleoScan™.
The truncations are preserved to generate Strati outputs:
- Horizons from Strati,
- Horizon Stack from Strati,
- Geomodel From Strati,
- Layers from Strati,
- Geobodies from Strati,
- Isochore from Strati.
The truncation is managed in Wheeler display.

Stratigraphic Sequence Boundaries are based on the RGT Model. Therefore, they are originally continuous over the survey and never intersect. Truncation tool allows defining a stratigraphic truncation between two Sequence Boundaries.

Truncation in Sequence Stratigraphy tool bar

Truncation Management actions are present from the Sequence Stratigraphy tool bar.

Undo and Redo can be executed from the tool bar or using the shortcuts CTRL + Z (undo), CTRL + Y (redo).

Truncation Mode

Creating a Stratigraphic Truncation requires to activate the Truncation Mode from the Strati Viewer.

The Truncation Mode allows visualizing the effects of the created truncations.
Truncation List

To open the list of all generated truncations, click on **Edit** button on the truncation panel of the Strati Viewer.

One truncation is defined between two sequence boundaries by the following parameters:
- **The type:** top if the lower boundary is truncated by the top boundary, bottom conversely,
- **The truncated boundary** and the **reference boundary,**
- **The thickness threshold:** thickness of the layer from which the truncated boundary will be modified,
- **The distortion strength:** similar to a slope.

Create Truncation

To activate truncation mouse mode, click on “**TRUNC**” icon under the Sequence Stratigraphy toolbar.

A specific double arrows cursor appears (in green). Hold left click and navigate through the sequence. The cursor fits the layer it relies on and shows its thickness. This cursor defines the thickness threshold parameter of the truncation. From this thickness threshold, the truncated boundary will be modified to intersect the reference boundary.

Select the desired thickness threshold and click on **Truncation Top** (or **Truncation Bottom**).
It automatically proposes a solution for the truncation.

Then, parameters can be edited from the Truncation List.

**Truncation Parameters Effects**
Multi-Truncations

Select several Sequence Boundaries and click on **Multi-truncation Top** or **Multi-truncation Bottom**.

If Multi-truncation Top is selected, the uppermost Sequence Boundary will be identified as the reference boundary. The lower boundaries will be truncated on this reference boundary. Conversely, if Multi-truncation Bottom is selected, the lowermost Sequence Boundary will be identified as the reference boundary and the upper boundaries will truncate on it.

Then, define thickness threshold parameter for each truncation and click on **Ok**. It requires having an idea of the thicknesses of the layers. PaleoScan™ applies the truncations if truncation Mode is activated. Truncation parameters can still be modified from the Truncation List.

1. Select sequence boundaries

2. Action Strati Multi-truncation Top

3. Edit Thickness Threshold values

4. Adjust Truncation Parameters

**Strati Viewer**

The Strati Viewer now includes a horizon map viewer that displays:

- a **horizon map** in case of selected Boundary,
- a **thickness map** in case of selected Layer,
- if a **truncation** is selected from the Truncation List, a **thickness map** and two associated polylines corresponding to the truncation parameters.

The Horizon Map Viewer helps for visualizing in a Map View the effects of a truncation.
To display the Horizon Map Viewer, click on “Horizontal Map” on the right side of Strati Viewer.

Select a boundary: the z value of the boundary is mapped.

Select a layer: the thickness of the selected layer is mapped.

Select a truncation from Truncation List: the thickness between the two affected boundaries is mapped and two polylines are displayed. The blue polyline is the Thickness Threshold polyline. It shows the points from which the truncated boundary is modified by
the truncation. The red polyline is the Thickness zero polyline. It shows the exact places where the two boundaries truncate.

Increase truncation distortion, the blue polyline is not modified, and the red polyline gets closer to the blue one. It materializes a steeper truncation.

Decrease thickness threshold, the thickness from which the modification occurs appears lower.
Sequence Layering

It is now possible via the Strati Viewer to manage sub-layering for each layer of a PaleoScan™ sequence. This sub-layering option allows defining a sub-layering scheme. The sub-layering scheme will be used in other PaleoScan™ tools such as GeoCellular Model computation.

To generate sub-layers, click on **Edit** button under the Layering panel.

The list associates for each layer of the sequence:
- its name,
- its lithology pattern,
- its layering method: iso-proportional, parallel to top, parallel to bottom,
- its number of sub-layers.
- the maximum sub-layer thickness, obtained by dividing the maximum thickness of the layer by the number of sub-layers.

Adding sub-layers in the table will generate internal layers in the sequence layers, with different patterns:
- **Iso Proportional** between the top and the bottom of the layer,
- **Parallel** to the **Top** of the layer,
- **Parallel** to the **Bottom** of the layer.
The sub-layering scheme for a layer can also be edited from the Layer properties.

### Horizon Stack from Sequence

The **Advanced Interpretation** add-on module allows the creation of Horizon Stack with specific horizons from the created sequence.

First, a stratigraphic sequence must be created. Secondly, the user must click on the **New Horizon Stack from Sequence** button, under the Sequence Stratigraphy tab.

Depending on if the Truncation mode is activated or not, different types of interface will appear.

#### Truncation Mode Disabled

The Horizon Stack creation window appears with several options:

1. The Background Attribute must be dragged and dropped from the Project Browser in the corresponding field,

2. The number of horizons which will compose the Horizon Stack(s) must be defined via three possible options:
   a. By default, the **None** option is selected. So, there will be as many horizons as sequences boundaries.
b. By toggling on the **Between top/bottom** option, the user can add some horizons (the maximum number of additional horizons is 1000), distributed proportionally between the first and the last boundary.

c. By choosing the **By layer** option, the user can choose how many additional horizons will be created in each layer of the sequence.

The **Number of horizons** line computes the total number of horizons that will be created in the Horizon stack.

Toggle on the Horizon preview option in order to have a better visualization of the position of the future horizons and be sure not to miss any important event.
3- Click on the blue arrow on the right to switch panel. Select the attribute to map on each horizon. Note that several attributes can be computed meaning a Horizon Stack will be generated for each attribute.

4- Define the attribute parameters to set up such as the volume input, the sample value, the vertical window size (described the chapter: Horizon - Attributes Mapping into the User Guide 2019 document),

5- Give an output name to the Horizon Stack(s) before clicking on Run.

**Truncation Mode Enabled**

In the case where the Truncation Mode is activated, a specific interface similar to the Horizon Stack Duplication interface is displayed.

In this case, the generated Horizon Stack will only comprise horizons corresponding to the layer boundaries of the current sequence.

The Fill truncated areas option allows, if toggled off, to avoid the creation of the horizon where truncated. If toggled on, the truncated part of the horizon will be populated with the Z values of the reference horizon.
We address the reader to section *Horizon Stack - Duplicate Horizon Stack with New Data* to complete the Horizon Stack creation through this interface.

## GeoModel from Sequence

It is now possible to generate a Geo-Model form stratigraphic sequences based on picked sequences and model types.

To create a Geomodel from a Sequence, click on “Edit/Create a Geo-Model from Sequence” in Sequence Stratigraphy tool bar.

An interface pops up. Define the interpolation patterns for each layer and type the model name.

Interpolation patterns (except RGT Model type) can also be edited from the properties of a layer or from sub-layering table.
Click on **Ok** to launch the computation. The Geomodel from sequence is saved in the project, under the Volume\Model tab.

The result is a Geomodel with specific interpolation patterns between each sequence boundary limit.
GeoCellular Grid

From PaleoScan 2019, the creation and exportation of GeoCellular Grid based on the geometry obtained in the Geo-Model.

**GeoCellular Model from Sequence**

To compute a GeoCellular Model, a **Stratigraphic Sequence** must be picked. The sub-layering patterns and the stratigraphic truncations of the stratigraphic sequence defines the geometry of the GeoCellular Model. They also stand for vertical dimensions of the grid.

NOTE: The computation is available from the 3D Strati Viewer only. It is not available for strati objects based on 2D RGT models.

**Grid Structure**

The GeoCellular Model is a stratigraphic grid of corner point type. The grid is 2D regular on the XY plane and the pillars are vertical. It is conforming to the stratigraphic sequence it is created from. The faults are integrated as stair-stepped faults.

The stratigraphic sequence boundaries are derived from the RGT model. Therefore, the RGT model drives the geometry of the GeoCellular Model. For each layer in the stratigraphic sequence, it is possible to define a sub-layering scheme: a number of sub-layers and a sub-layering pattern. Hence, it defines the vertical resolution of the grid.

**Grid Layering**

The layering of the GeoCellular Model is defined from the stratigraphic sequence within the strati viewer. To define the vertical layering of a stratigraphic sequence, click on **Edit Layering** from the top overlay of the Strati Viewer.
The displayed table contains information on each Stratigraphic layer (1. layer name and 2. lithology) as well as the sub-layering scheme. The sub-layering scheme is made up of:

3. The layering pattern: iso proportional, parallel to top, parallel to bottom. In the latter case the maximum vertical thickness is replicated.
4. The number of sub-layers.
5. The maximum vertical cell size which is obtained by dividing the maximum thickness of the layer by the number of sub-layers. For parallel to top and parallel to bottom patterns the maximum vertical cell size is always replicated.
The sub-layering scheme for one stratigraphic layer can also be assigned from the Properties window. First select the object by double clicking on it in the stratigraphic viewer and modify the parameters as desired.

Spatial Resolution and other Options

Once a stratigraphic sequence has been picked, click on the first icon called GeoCellular Model from Stratigraphic under the GeoCellular Model module.

A second interface pops up in which the interpreter must select:
- the input area of interest (AOI). The AOI can also be defined spatially by moving the white dots on the edge of the volume. The AOI cannot be defined vertically.
- the fault set which faults are to staircase
- the smooth factor: geometrical smooth to create more synthetic models from the RGT model
- the spatial resolution of the grid (the vertical resolution is defined in the layering scheme in the stratigraphic sequence). The cell spatial size can be edited as well as the equivalent bin size.
- the activation of extrapolated cells in ACTNUM. Indeed, where the RGT model contains no values, PaleoScan™ creates extra data (by minimization of the thickness variation). The extrapolated geometries can be identified in ACTNUM by inactivating the corresponding cells.
NOTE: The grid spatial resolution is defined as a spatial undersampling of the RGT model. Hence, the X and Y axis of the grid will be parallel to the Xlines and Inlines of the RGT model respectively. To define the grid spatial orientation, the RGT model must be re-oriented and the Stratigraphic Sequence must be based on this RGT model.

**GeoCellular Model Computation**

Click on **PREVIEW** to launch a glimpse of the calculation inside the AOI. The grid geometry is displayed.

By clicking on the **TEST** button, PaleoScan™ will test the feasibility of the computation and estimate the grid resolution. If the volume is too big, PaleoScan™ won’t be able to generate the GeoCellular Model. Otherwise, the computation is possible, and the grid resolution is displayed.

Finally, click on **OK** to launch the computation. Once it is done, PaleoScan™ opens a save dialog. Enter the model name and click on **OK** to save it. Then the 3D Viewer pops up with the GeoCellular Model displayed inside. From now on, the Strati Viewer and the sequence can be closed. Since the GeoCellular Model is a 3D object, it is visible in the other viewers as an intersection.
Staircased faults are displayed in the 3D Viewer as GeoCellular Fault objects. They are initially named like the input fault set, but they can be renamed afterwards from their Properties panel.

GeoCellular Model computation automatically saves the object in the project database. In the project browser, the GeoCellular Model is inserted as a folder. The main folder has the GeoCellular Model’s name. The sub-elements are the GeoCellular model properties.

GeoCellular Model computation generates three initial properties:
1. **LAYERS** materialize the k indexing cells or layers.
2. **ACTNUM** materializes the activated cells. If the option Activate Extrapolated Cells was ticked, they will all appear as active.
3. **ZONES** materialize the initial stratigraphic layers picked in the stratigraphic sequence.

**Properties**

**IJK Navigation**

GeoCellular Model visualization can be restricted to a selected IJK index range. The range is defined in the Properties window.
IKJ navigation can also be done directly from the 3D viewer. First select the grid by double clicking on it. According to the selected facet, the navigation through the grid is performed as below:

- **Top facet**: from top to bottom
- **Bottom facet**: from bottom to top
- **East facet**: from East to West
- **West facet**: from West to East
- **South facet**: from South to North
- **North facet**: from North to South

Then use the mouse wheel to navigate along one direction.

**Property Range**

GeoCellular Model visualization can be restricted to a selected range in the displayed property values. From the Properties window, select the **minimum** and **maximum** property values to display the cells holding a property value in this interval.
Show Grid

Click on **Show Grid** option from the **Properties** window to draw the grid lines.
Check Conformity

This option tests whether two neighboring cell facets are collocated. If they don’t, which is the case when there is a fault throw, the two facets are displayed. This option implies a longer mesh computation time and can slow down PaleoScan™.

Show Inactive Cells

This option allows to hide or show the inactive cells from the ACTNUM property.

![Image of active and inactive cells with property range table]

NOTE: in the GeoCellular Model computation, grid geometry is extrapolated where the reference RGT model has no values. By default, the extrapolated cells are tagged inactive in ACTNUM.

Save GeoCellular Model

To save the GeoCellular Model, select the object (click on it with the selection mode) and click on Save GeoCellular Model:
Choose an output name and click on OK. The Model is stored under the GeoCellular tab of the project browser.

Property Mapping

3D volumes of any kind can be mapped on a GeoCellular Grid. They can be well properties created from the Properties Modeling tool of PaleoScan™ (Property Modeling license is required) but also seismic attributes.

First, select the reference GeoCellular Model by double clicking on it in the 3D viewer. Then click on Data Mapping on GeoCellular Grid in the GeoCellular Model toolbar:

A second interface that lists the attribute volumes of the project presents itself. By selecting a volume in the list, the intersection between the GeoCellular Grid and a section of the volume is displayed.

Click on Finish. A save dialog pops up. Enter the name of the grid property which will be associated to the GeoCellular Model and click on OK to launch the computation.

Once finished, the resulting property is visible in the 3D viewer and the property appears in the project browser. A specific icon is given to this new Grid Property.
NOTE: Upscaling is done using the average method.

**GeoCellular Model Extraction**

If a GeoCellular Model is too big, one can proceed its extraction and create a smaller model.

First, select the original GeoCellular Model by double clicking on it. Then click on **Extract a portion of a GeoCellular Model** in the GeoCellular model toolbar:

A window presents itself, showing the IJK extraction range and the resulting grid dimensions. Select the desired IJK extraction range and click on **OK**. A saving interface pops up. Type the name of the extracted GeoCellular Model and click on **OK** to launch the computation:
Once the computation is finished, the original GeoCellular Model is removed from the 3D Viewer and replaced by the extracted one. The extracted GeoCellular Model is saved in the project and all the original properties are extracted either.
Property Modelling

Interpolation Method

In the Wells Property Modelling tool interface, for each interpolation method, an option is available to prevent propagation in the selected layer. If No Propagation is applied on a layer, there will be no preview on horizons in that layer.
Seismic Well-Tie

Log Display

In the Seismic Well-Tie tool interface, in the Display tab, an option is available to use log Templates to customize the log display in the tool.
Velocity Modelling

**Velocity Type**

In the Edit/Save Velocity Model tool interface (while having a Sequence open), a new Velocity Type is available: “Linear from Bottom.”
Python

Python now allows users to add up their own home-made wizards, attributes, interface customization in PaleoScan™. Note that any PaleoScan modification made via the Python plugin will not be taken into account for support maintenance.

Introduction

Starting from PaleoScan™ 2019, Python code can be executed directly from the software. PaleoScan™ ships with Python 3.6.

To use Python features of PaleoScan™, a Python license is required. To activate the Python module, check the Python feature in the Extensions menu:

All Python actions are available from the Python toolbar:

Python Documentation

PaleoScan™ ships with a technical documentation in the form of an HTML website. This documentation provides a lot of examples and describes all the classes and functions exported by PaleoScan™. In order to access this documentation, ensure to let the Documentation option checked while installing the setup:
Then, click on the **Open Python Documentation** of the Python toolbar. It will open the home page of the HTML documentation in the default browser:

![Open Python Documentation](image)

**Search Documentation**

Enter a word in the **Search** field located in the top left corner of any page of the Python documentation to find more information about this term. In the example below, we search the documentation of the `vec3` class:
The Python editor also offers the ability to search a term.

## Writing and Executing Python

### Python Editor

PaleoScan™ provides a basic editor which should be enough to write little scripts. Feel free to use an external editor, especially if you write plugins. To open a Python editor in PaleoScan™, click on the **Python Editor** button in the Python toolbar:

![Python Editor button](image)

The context menu of this editor can be opened by right clicking on the title bar or in the editor:
Search in the Editor

Use **Search** to find a specific word in the editor. It opens an input box on the top right corner of the editor. Enter a term to search in this box. Press Escape to close the search box.

Right click on a term and click on **"Search for X"** to open the search box filled with that term:
Search Documentation from the Editor

To search a Python term directly in the HTML documentation, right click on the term you want to search and click on "Search PaleoScan™ Python Documentation for X":

![Image of search functionality]

Search Results

From here you can search these documents. Enter your search words into the box below and click "search". Note that the search function will automatically search for all of the words. Pages containing fewer words won’t appear in the result list.

Search

Execute

To execute the Python code of an editor, either right click inside the viewer and click on Execute OR click on the Execute button of the Python toolbar OR press F8. PaleoScan™ will execute the code of the focused Python Editor.

![Image of execute functionality]
Save

To save the Python code of an editor, either right click inside the viewer and click on **Save** OR click on the **Save** button of the PaleoScan™ toolbar OR press **Ctrl+S**.

If a new script is saved, PaleoScan™ will save it in the Python Script section of the project browser with the given name:

**Python Script**

Python scripts are stored in the **Python Script** folder of a project, in the **Other** section. Unlike Python plugins, they are specific to a project like other PaleoScan™ objects:
To execute a Python script from the project browser, right click on the script in the project browser and click on **Execute Script**:

![execute_script]

**Python Plugin**

PaleoScan™ provides a plugin system. Plugins allow users to write custom functionalities inside PaleoScan™ which can be dynamically activated and deactivated and can be shared easily. A plugin is a folder containing several files. PaleoScan™ provides import>> and export tools for plugins, so that plugins can be transferred from one PaleoScan™ installation to another as ZIP archives.

Plugins are related to a PaleoScan™ installation. Thus, they are available for all users using the same PaleoScan™ installation.

For technical information about PaleoScan™ Python plugins, refer to the Python Documentation, in the **Plugins** section:

![plugins]

**Plugin Location**

By default, plugins are stored in `%USERPROFILE%/.paleoscan/plugins`. New plugins can be manually copied inside this folder, they will appear in the PaleoScan™ plugin list. The plugin location can be customized in the Python section of PaleoScan™ settings.
Plugin List

To open the Python plugin list, click on the List Python Plugins button in the Python toolbar:

![List Python Plugins button](image)

The Python Plugins window lists all available Python plugins and allows several actions:

- **Create new plugin**
- **Refresh list**
- **Search in plugin list**
- **List of plugins**
- **Close plugin list**

### Actions specific to a plugin:
- Load/unload plugin
- Load plugin when Python is enabled
- Open plugin’s main script
- Delete plugin

Create Plugin

To create a new Python plugin for PaleoScan™, open the Python Plugins window and click on New Plugin:

![New Plugin button](image)

Two fields are mandatory:

- **Name**: The name of the plugin, useful to identify it in the plugin list.
- **Script**: The main script of the plugin. This script will be loaded by PaleoScan™ when using the load/unload button of the Python Plugins window.

Once the form is filled, click on Create and the Python Plugin window comes back to the plugin list. The new plugin should be available in the list:
Custom Attribute with Python

PaleoScan™ manage a dynamic registry of attributes. An attribute is algorithm applied on a specific input data (Volume or 2D Line), that produce an output data. PaleoScan™ already offers a lot of attributes, but you can create and register your own attributes with the Python module of PaleoScan™.

Attribute Tester

PaleoScan™ provides a sandbox to test Python plugins defining custom attributes. Click on the Python Attribute Tester button in the Python toolbar:

To test an attribute, follow these steps:
1. Select the Python Plugin defining the attribute to test.
2. Select the Attribute to test. A Python plugin can register as many attributes as it needs.
3. Select an Input to test the attribute.
4. Click on Preview to apply the attribute on the given input.
Plugin does not contain any Attribute
If a plugin does not register any attribute, the Attribute Tester cannot fill the attributes list. Thus, no attribute can be tested from this plugin:

Plugin already loaded in PaleoScan™
If a plugin is already loaded in PaleoScan™, the Attribute Tested won’t be able to load this plugin another time and it will raise an error message:
In that case, open the Plugin and unload the plugin.

**Attribute is not register in the plugin**

The Attribute Tester can detect attributes that are defined but not registered. In such case, it will display a message when the plugin is loaded:

![Attribute is not register in the plugin](image)

**Reload a Plugin from the Attribute Tester**

If a file changed in a plugin loaded in the Attribute Tester, a message appears at the bottom of the window to inform the user to reload the plugin. Click on the **Reload Plugin** button to reload the plugin in the Attribute Tester:

![Reload a Plugin from the Attribute Tester](image)

**Python Module Finder**

PaleoScan™ 2019 ships with **Pip 18.1** for package management.

![Python Module Finder](image)

Click on the **Show Python Modules** button in the Python toolbar to open the **Python Module Finder**.

---

PaleoScan™ is a registered trademark of Ellis S.A.S All Rights reserved

Page 77 of 78
The **Python Module Finder** also presents itself automatically if a Python script using a missing package is executed.

### Installing Packages from the Network

To install a new package from the network:

1. Enter the **package name**.
2. **Check the box** next to the package.
3. Click on **Apply Changes**.

### Installing Package from a Local Archive

From the Python Module Finder, you can also install Python packages from archive files. If your package install fails, please ensure that the archive file is valid for the version of Python that is embedded in PaleoScan™, and that your package is compatible with Windows. Please also ensure that you are using a 64-bit version of the package.

To install a package from an archive:

1. Click on the related tab.
2. Click on **Add files** and select the archive.
3. Click on **Install selected archives**.