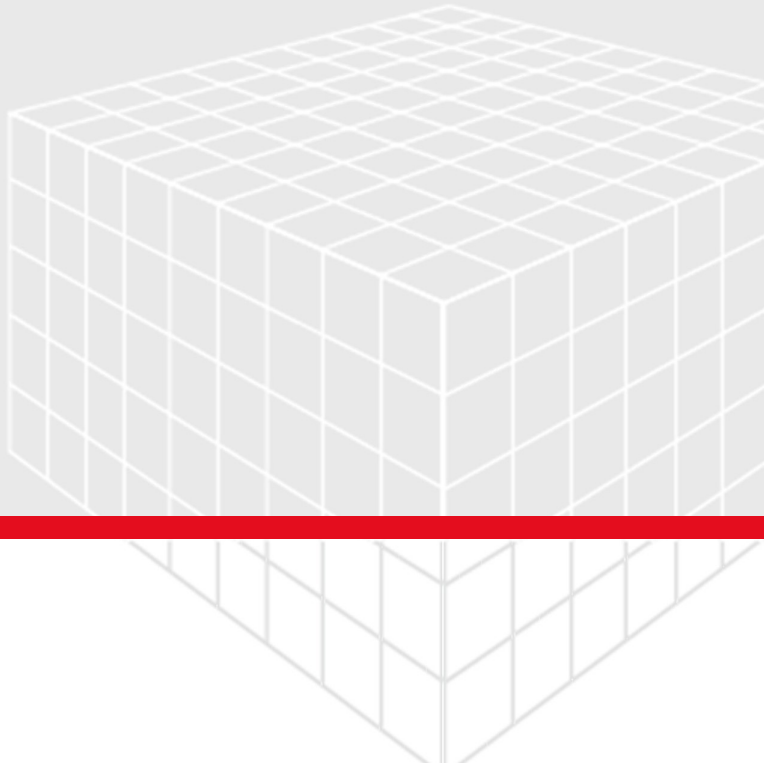
Result Viewer

User Guide



GEODICT Release 2026

Published: June 2026

GeoDict

https://doi.org/10.30423/userguide.geodict

Result Viewer

1 Result Viewer

Nearly all commands in GeoDict – for example, structure generation, simulation or analysis - produce a so-called **result file** with the file ending ***.gdr**. It contains information about the run and its results, such as input parameters, runtime, and outputs of the specified command.

The result file is saved in the chosen project folder. In most cases, a folder with the same name is also created which contains additional files associated with the result file (e.g., volume fields). It will additionally contain the ***.gdt** file of the structure on which the simulation was done or which was created during the computation.

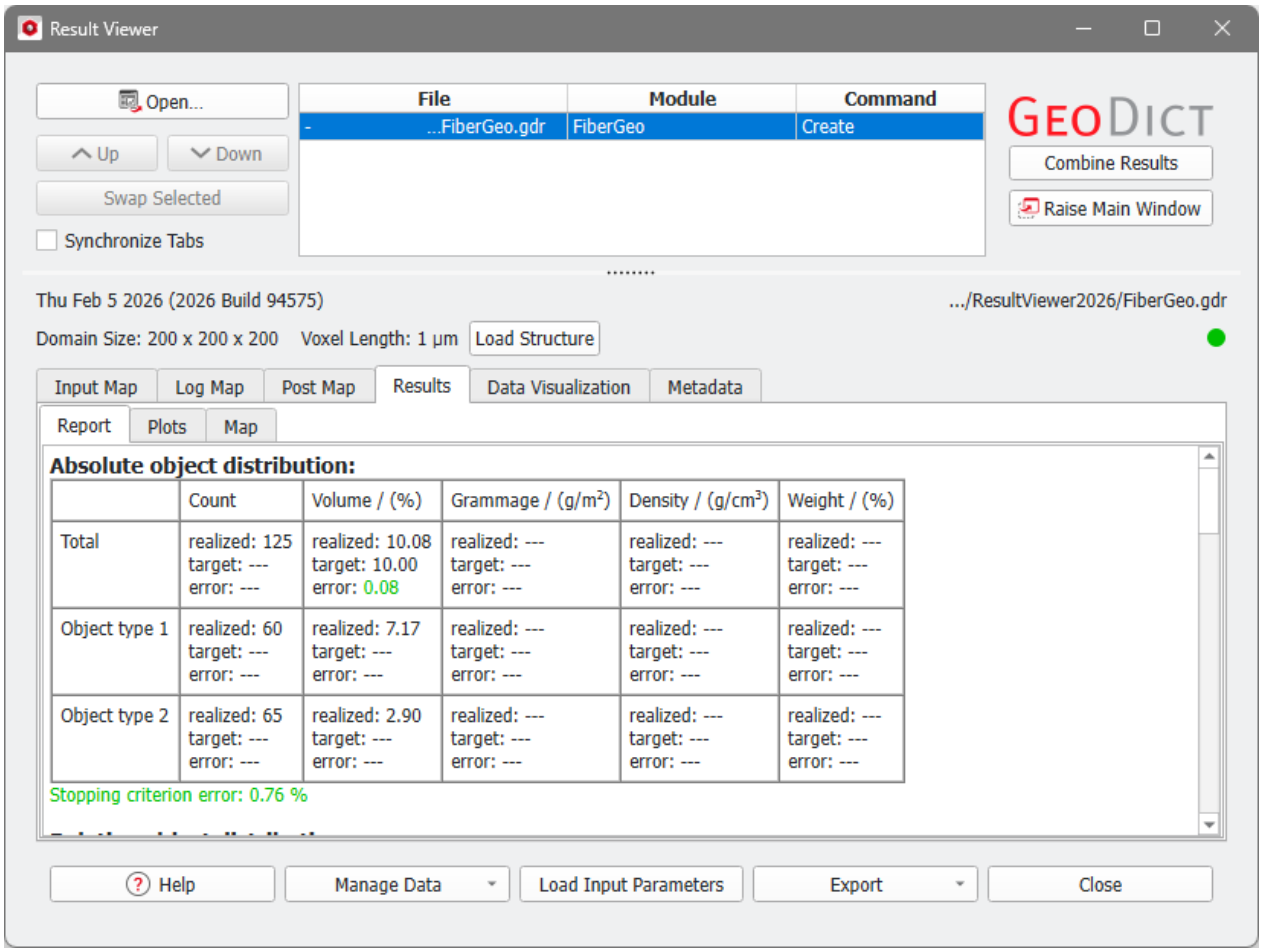
After execution of a GeoDict command from the User Interface (GUI), the corresponding result file is automatically opened in the GeoDict **Result Viewer**. From the Result Viewer you can not only view the results in text form, but you can load the structure and the produced additional result files (such as volume fields, particle trajectories, grain fits, ...).

Know how! If result files (*.gdr) are copied to another directory, always keep in mind that the corresponding folder should also be copied there so you can still load the additional files.

The Result Viewer can open multiple result files at the same time. They can be viewed one by one or next to each other for comparison.

Open any *.gdr file can be done through:

- the menu bar (File → Open Results (*.gdr)...)
- the toolbar: 
- the **Open** button in the top left of the Result Viewer (if already opened)
- dragging-and-dropping the *.gdr file on the GeoDict GUI or on the Result Viewer window
- the GeoDictResults:LoadGdrFile command in a Python macro



New in GeoDict 2026

- For plots with many graphs, the visibility settings have been moved to the Edit Graph Styles 31 dialog.

Learn how to view GeoDict results in the Result Viewer

- Data Format 6
  The format, in which the data is saved, is explained here.
- Structure of the Result Viewer 8
  Get an overview of the different areas in the Result Viewer.
- Plots in Result Files 23
  View and modify the plots shown in the Result Viewer.
- Combine Results 39
  Combine multiple result files (*.gdr) to one result file for comparison.

Join the conversation

Visit the Community on GeoDict Forum to be inspired and get answers to top questions.
Find illustrative application examples on the website.
Our tutorials and seminar videos give you a first hands-on experience. They are collected in the Learning Center.
Send your Support Request to our technical Support.

1.1 Data Format

GeoDict result files are saved in YAML-format and contain many different **Key : Value Maps**, where parameters are hierarchically arranged.

These **Maps** consist of a describing **Key**word and the corresponding **Value**, where the value can have a corresponding **Unit** or not:

Key	Unit	Value
Pressure	Pa	123

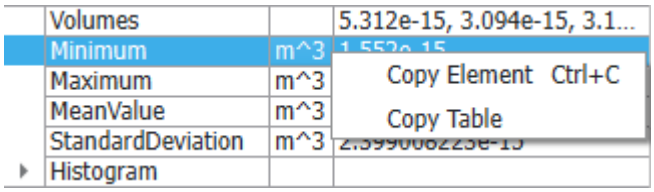
The **Value** in such a combination can be a **Key : Value Map** again**,** which is then called a **Sub-Map**. This is often used in GeoDict's result files to clarify associated parameters, for example:

Key	Unit	Value
Batch1		
Pressure	Pa	123
Batch 2		
Pressure	Pa	456

In GeoDict's Result Viewer, those maps are shown in well-arranged tables, where sub-maps can be unfolded by clicking on the rotating triangle:

Key	Unit	Value
▸ CoreResults		
▾ DerivedResults		
▸ ShapeAnalysis		
▾ Volume		
Volumes		5.312e-15, 3.094...
Minimum	m^3	1.552e-15
Maximum	m^3	1.8546e-14
MeanValue	m^3	3.455328571e-15
StandardDeviation	m^3	2.399008223e-15
▸ Histogram		
▸ EquivalentDiameter		
▸ InnerDiameter		
▸ Sphericity		

Clicking on an entry here will highlight the line for better readability. In the right-click content menu, the options to copy only the entry on which the right-click was performed (**Ctrl + C**) or the whole table are available.



The *.gdr files contain all results in ASCII text. Therefore, it is possible to open them simply with a text editor, e.g., if GeoDict is not available. All data inside a result file is arranged in a **Key : Value Map** with several **Sub-Maps**, to keep the file in an organized shape.

The raw text of **Key : Value Maps** is written in **YAML** code similar to **Python dictionaries** and looks as follows:

```
"InputMap" : {
  "ResultFileName"         : "IdentifyGrains.gdr",
  "ConstituentMaterials" : {
    "Temperature"          : [293.15, "K"],
    "Material00" : {
      "Type"                 : "Fluid"
      "Name"                 : "Air",
   }
 }
```

Maps are always surrounded by curly braces { }. If a key describes a **Sub-Map** the sub-map again is embraced by { } symbols, denoting start and end of a sub-map or sub dictionary. A key-value pair is always separated by a colon. The entries in a dictionary are separated by comma.

Some text editors can understand this syntax and will highlight the text accordingly.

To activate YAML syntax highlighting for a *.gdr in Notepad++, simply click on **Language** → **YAML** after the file is loaded:

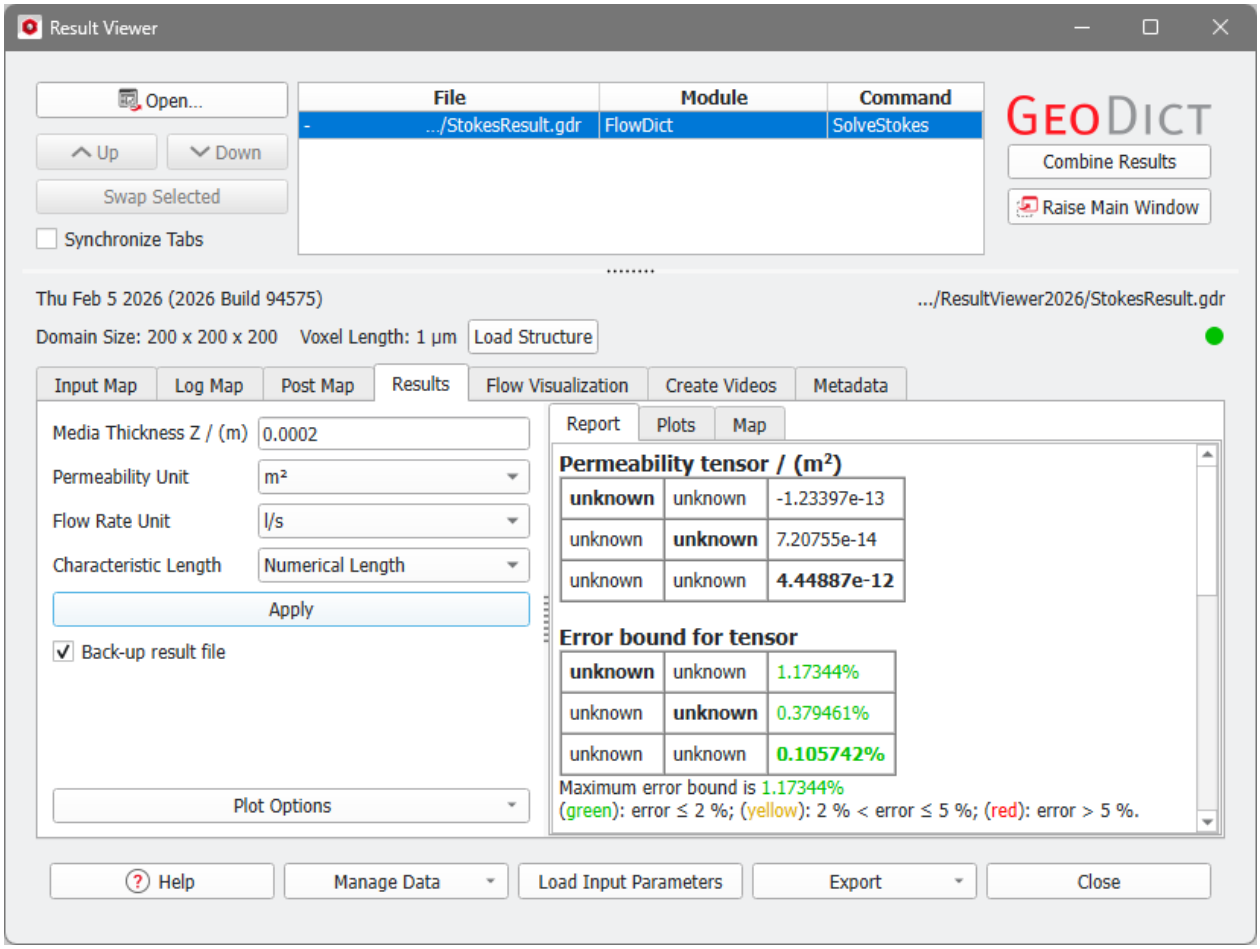
```
"InputMap" : {
  "ResultFileName"                    : "IdentifyGrains.gdr",
  "ConstituentMaterials" : {
    "Temperature"                     : [293.15, "K"],
    "Material00" : {
      "Type"                          : "Fluid",
      "Name"                          : "Air",
```

In Notepad++, it is then also possible to collapse Sub-Maps for clarity by clicking on the [-] minus control at the left border of the page.

Such **Key : Value Maps** are also to be found in Python macros and look the same.

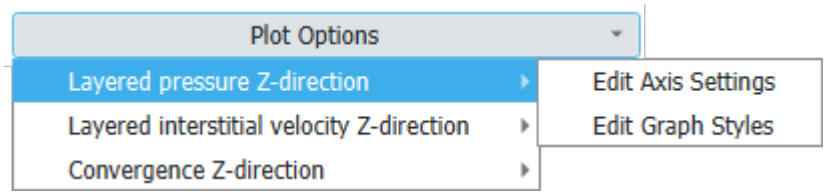
1.2 Structure of the Result Viewer

The Result Viewer consists of the **Header Section** 9 on top, the **Result Section** 10 underneath, and the **Bottom Section** 18 below. The name of the displayed result file and the path to it are highlighted in blue in the Header Section. The **Header Section** of the Result Viewer can be collapsed (and expanded) by pulling up (or down) the dotted line between the Header Section and the Result Section. For some commands, the Results tab in the Result Section contains a **post-processing panel** 13 on the left, which can also be collapsed or expanded in the same way.



At the bottom of the post-processing panel find the **Plot Options** menu. For each plot in the **Plots** tab the **Edit Axis Settings** and the **Edit Graph Styles** options are available.

They can also be accessed via a right-click in the corresponding plot. For an explanation see Edit Axis Settings and Graph Styles 27.

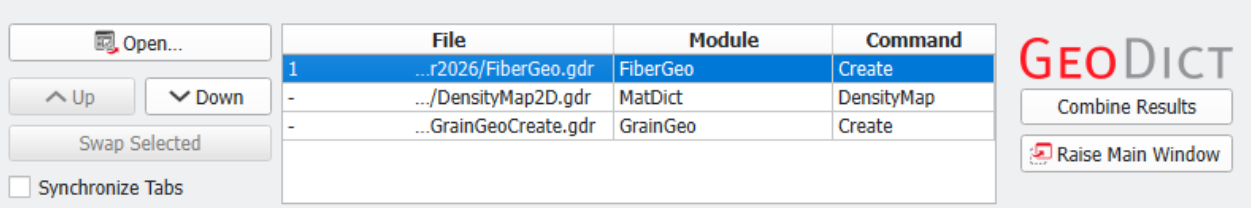


Topics of this section

- Header Section 9
  The Header Section shows information on the currently opened result files. You can sort them and open new result files. Additionally, open several result files at once or combine them to compare the results.
- Result Section 10
  All data from the result file is shown here. You can inspect the results, do post-processing steps and save plots as images.
- Bottom Section 18
  With the buttons at the bottom you can manage and export the data from the result file in other file formats. Furthermore, you can load the input parameters into GeoDict to repeat the same simulation.

1.2.1 Header Section

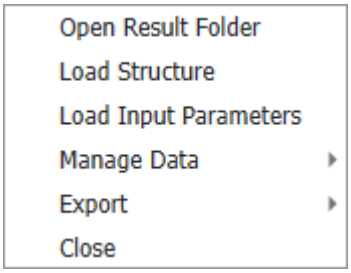
In the Header Section of the Result Viewer, the table in the middle contains a row for each currently opened result file. The columns show the full **File** path, the name of the **Module**, and the name of the **Command** that created the file. Only the content of the file(s) highlighted in blue is visible below in the **Result Section**.



You can change the column width of the table by clicking and holding on the line between the column headers and then pulling to the left or to the right.

Clicking on a column header sorts the files in alphabetical order (either by file name, module name or command name), clicking again inverts the order to anti-alphabetical.

Right-clicking in a row opens a context menu for the result file. Select **Open Result Folder** to open the result folder corresponding to the result file in the file explorer. With **Load Structure**, which is also available as button in the Result Section 10, the structure belonging to the result file is opened in the current GeoDict instance. The options **Load Input Parameters, Manage Data, Export,** and **Close** are also available in the Bottom Section 18 and explained there.



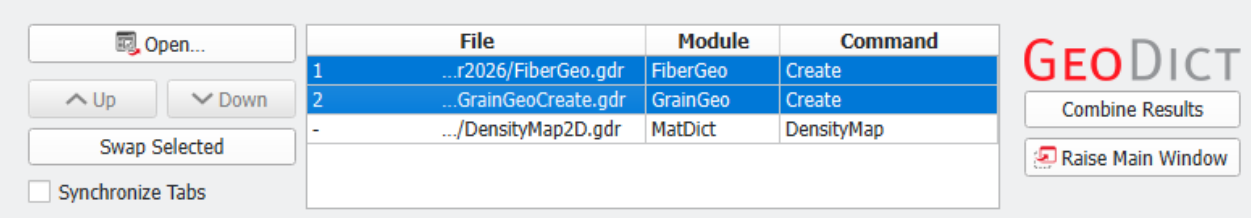
Know how! The options from the context menu are always accessible for loaded result files! To use the same options in the Result Section 10 or the Bottom Section 18, the result file must be selected and shown in the Result Section.

The buttons on the left side can be used to **Open** a new *.gdr result file directly from the Result Viewer and to move opened *.gdr files **Up** and **Down** in the table shown in the middle. Use the options **Swap Selected** and **Synchronize Tabs** when multiple result files are shown (see below).

On the right side, click to **Combine Results** 39 into one *.gdr file. Additional (not already loaded) gdr files can be added in the opening dialog. A click on **Raise Main Window** Raise Main Window brings up the main screen of the GeoDict GUI in front of the Result Viewer.

Show multiple result files simultaneously

Multiple result files can be selected by using **Ctrl+click** (selects only clicked files) or **Shift+click** (selects all files in between) on the list of entries. The selected files are highlighted in blue and shown next to each other in the Result Section. The number on the left in the table rows indicates the position of the files content in the Result Section. Additionally, in all opened **Map** tabs, the differences or similarities between the results are highlighted 55 and are easier to spot.



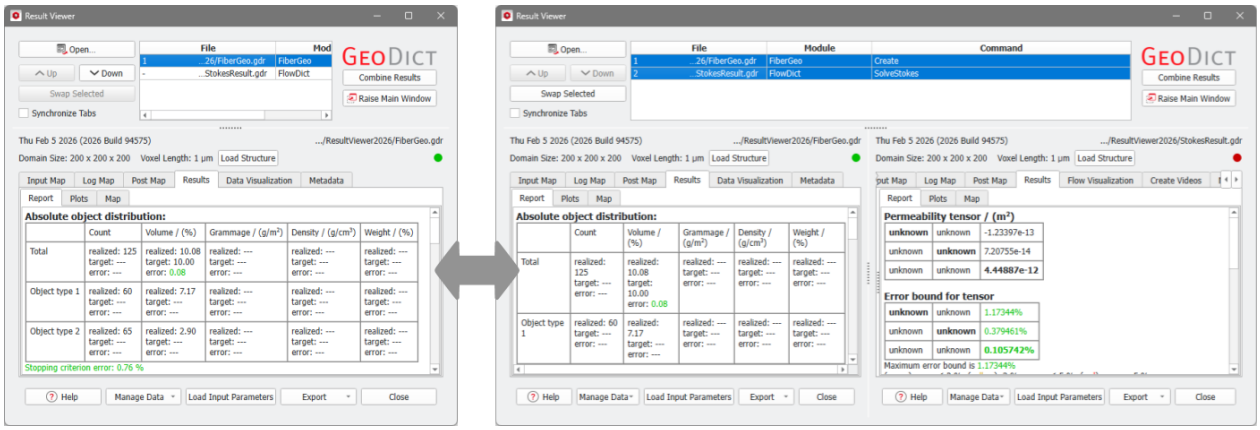
If exactly two result files are selected at once click **Swap Selected** result files to interchange their positions in the list of files. This will also swap the order in which the Result Sections are displayed.

Check **Synchronize Tabs** if multiple result files are highlighted. Then, clicking a tab in one of the result files will display this tab in all highlighted result files simultaneously. This facilitates the navigation through multiple opened result files.

If you click **Combine Results**, only the selected files are combined into one gdr file.

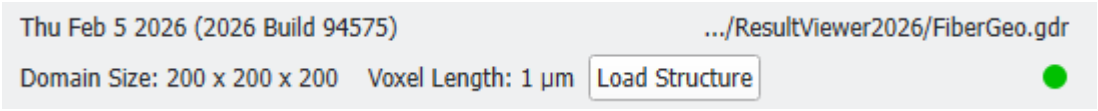
1.2.2 Result Section

In the **Result Section** the content of the selected result file (highlighted in the Header Section 9 table) is shown. If multiple results are selected and highlighted, the Result Sections of each result file appear side by side. The Result Section consists of two parts, the **Information header** and the **Tab section.**



Information header

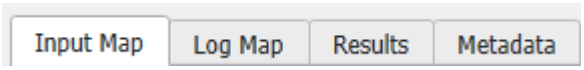
The **Information header** shows the date of creation and the version of GeoDict that was used to create this result file.



The **Domain Size** and **Voxel Length** are also shown here. If the structure for which the result file was created is available, the **Load Structure** button appears to load the structure in the visualization area. On the right, the path to the location of the result file is shown. The dot below turns green when the corresponding structure is currently loaded in the Visualization area of the GeoDict GUI, otherwise, the dot is red.

Tab section

Each step of the commands to obtain a result file is mirrored by tabs in the Result Viewer. The tabs are arranged in an organized way from left to right, starting with the inputs and ending with the quantitative results and visualization.

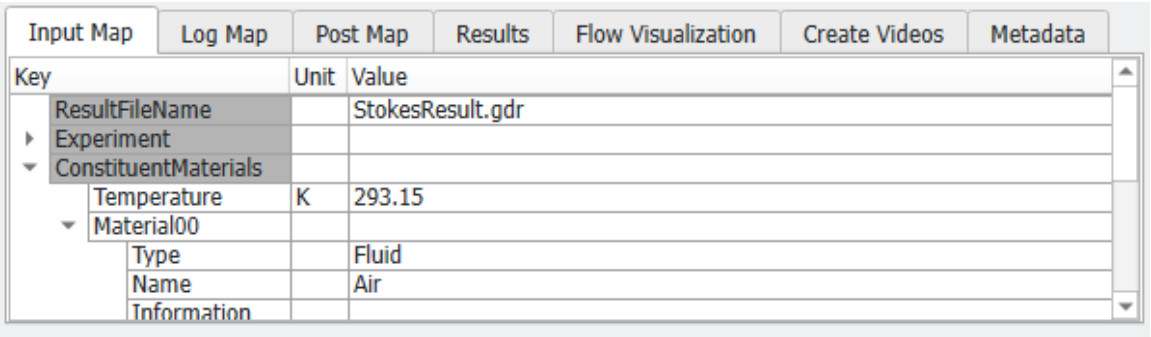


The tabs **Input Map**, **Log Map**, **Results**, and **Metadata** are always available for a result file. Additionally, the tabs **Post Map**, **Visualization**, and **Create Videos** might appear, depending on the command from the which the result file originates. For Combined Result Files 39 the tabs **Metadata**, **Data Visualization**, and **Create Videos** are not available. Instead, there is the **Compare Maps** 55 tab, where the single result files can be compared.

Some tabs show simple **Key : Value Map** tables while other tabs are filled with plots, report texts, or even more interactive forms of result management.

Input Map

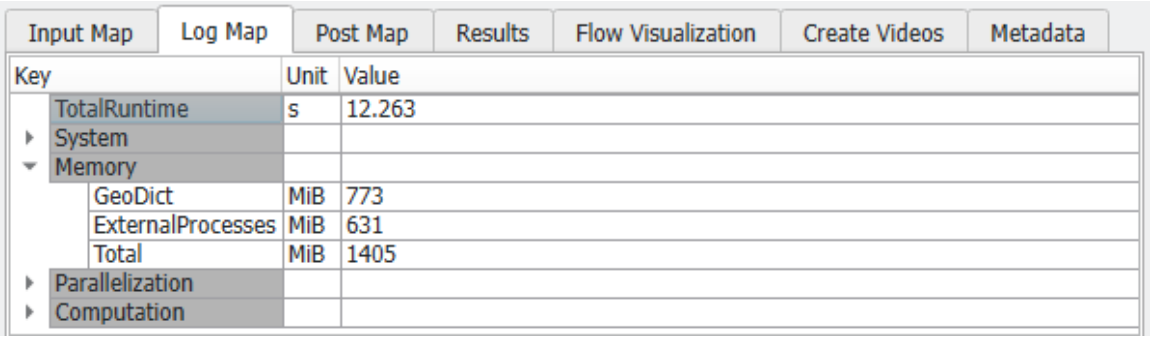
The **Input Map** tab shows an overview of all input data, parameters and their values, entered in the command options as Key : Value Map. If the command uses **Constituent Materials**, all material parameters are listed in the input map as well.



Know how! Use the Load Input Parameters 21 button in the Bottom Section to load all the parameters in the corresponding module GUI. You can check the **Module** and **Command** in the table of the Header Section 9 of the Result Viewer.

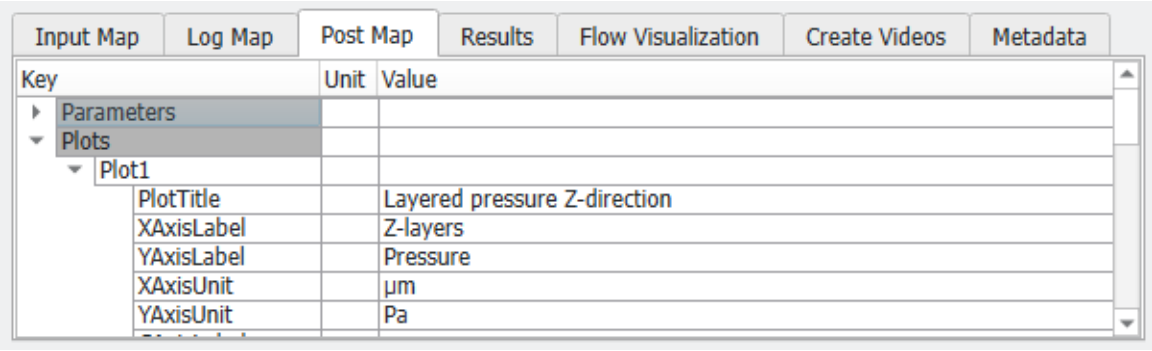
Log Map

The **Log Map** tab shows information of runtime, system and memory consumption during the command execution as Key : Value Map.



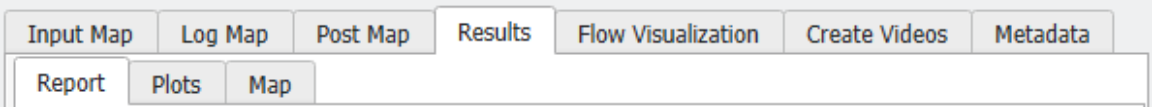
Post Map

The information for all the plots contained under the **Results** tab is saved under the **Post Map** (postprocessing map) tab as Key : Value Map.

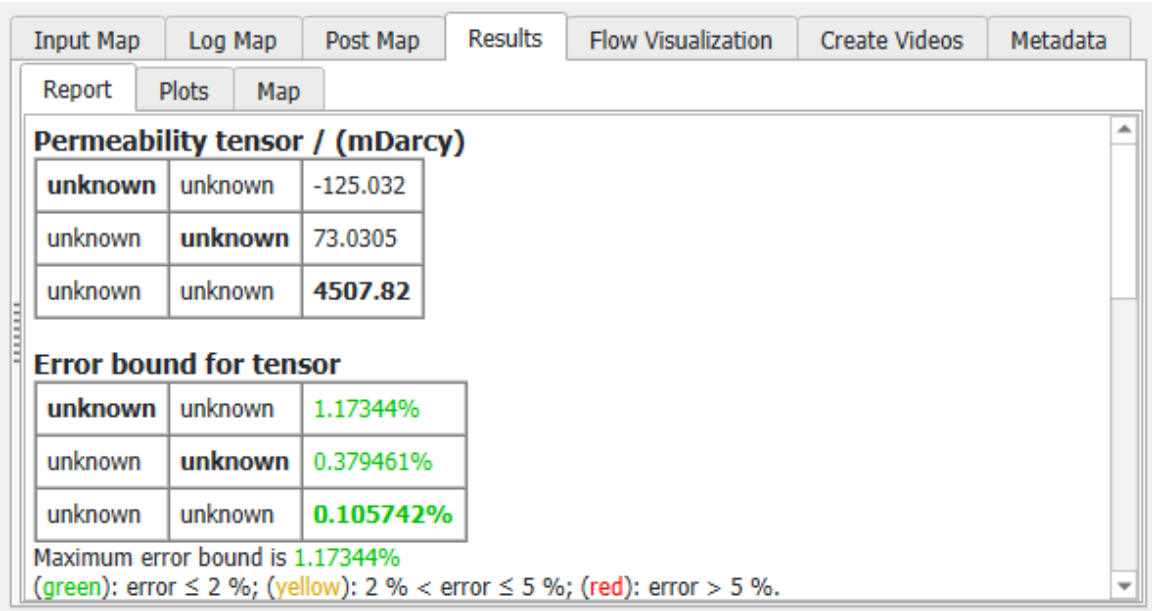


Results

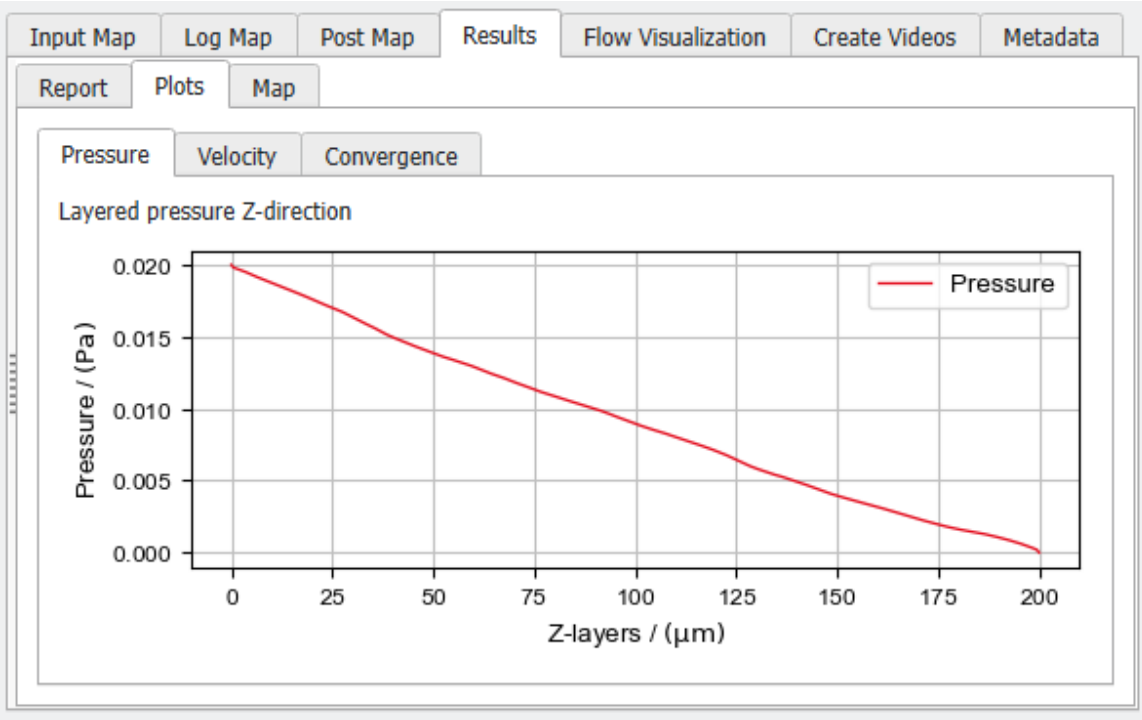
The **Results tab** contains the **Report**, **Plots** (only if plots are available), and **Map** subtabs.



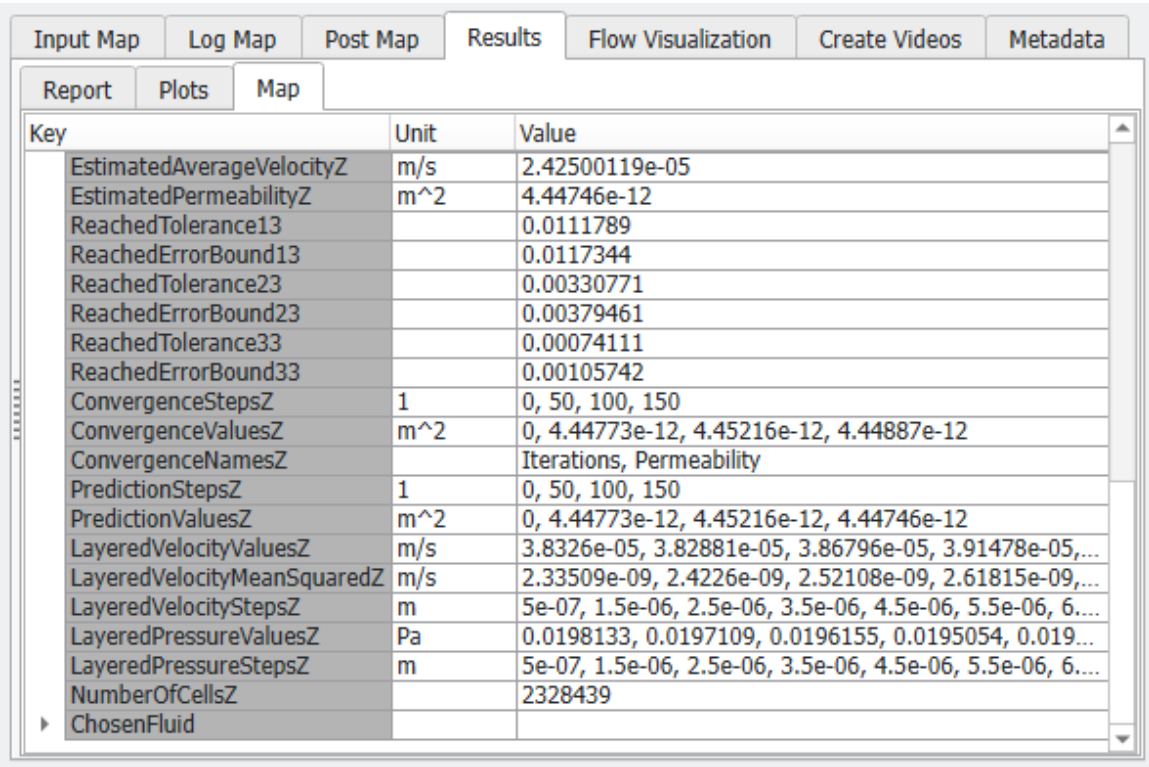
The **Report** subtab displays the report page, which is individually generated for each command. The report is a summary of all the important values to be found in a result file. Result tables or tensor data are shown here in a clear way.



The **Plots** subtab shows different plots to graphically display the output data given under the Report or the Map subtab. This subtab is not available for all commands, because not all result files contain plottable data. More information on plots is given under Plots in Result Files.

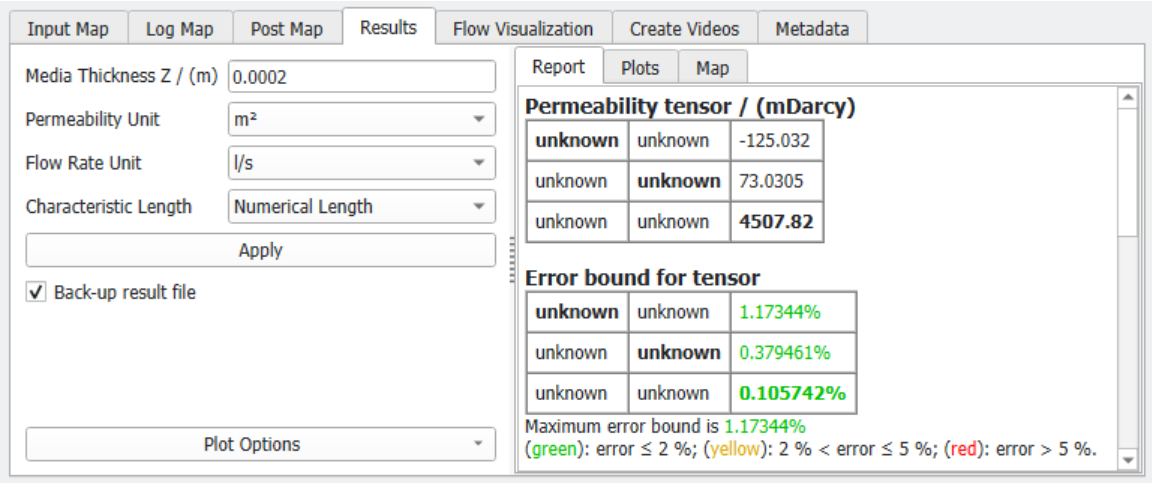


The **Map** subtab gives access to the computed values for the corresponding command as Key : Value Map. The data contained in the Report subtab is also available in the Map subtab. Additional details are found here, e.g., solution file names and information about the iteration process (number of iterations, reached stopping criteria, etc.).



Key	Unit	Value
EstimatedAverageVelocityZ	m/s	2.42500119e-05
EstimatedPermeabilityZ	m^2	4.44746e-12
ReachedTolerance13		0.0111789
ReachedErrorBound13		0.0117344
ReachedTolerance23		0.00330771
ReachedErrorBound23		0.00379461
ReachedTolerance33		0.00074111
ReachedErrorBound33		0.00105742
ConvergenceStepsZ	1	0, 50, 100, 150
ConvergenceValuesZ	m^2	0, 4.44773e-12, 4.45216e-12, 4.44887e-12
ConvergenceNamesZ		Iterations, Permeability
PredictionStepsZ	1	0, 50, 100, 150
PredictionValuesZ	m^2	0, 4.44773e-12, 4.45216e-12, 4.44746e-12
LayeredVelocityValuesZ	m/s	3.8326e-05, 3.82881e-05, 3.86796e-05, 3.91478e-05,...
LayeredVelocityMeanSquaredZ	m/s	2.33509e-09, 2.4226e-09, 2.52108e-09, 2.61815e-09,...
LayeredVelocityStepsZ	m	5e-07, 1.5e-06, 2.5e-06, 3.5e-06, 4.5e-06, 5.5e-06, 6....
LayeredPressureValuesZ	Pa	0.0198133, 0.0197109, 0.0196155, 0.0195054, 0.019...
LayeredPressureStepsZ	m	5e-07, 1.5e-06, 2.5e-06, 3.5e-06, 4.5e-06, 5.5e-06, 6....
NumberOfCellsZ		2328439
ChosenFluid		

Some GeoDict modules and commands have the possibility to run post-processing on how to display result data. If this is the case, a **Post-processing panel** with the available settings is displayed on the left, under the Results tab. The post-processing panel can be collapsed and expanded by pulling the dotted line to the left or right. After setting the parameters, click **Apply** to see the changes.



The check-box **Back-up result file** is activated by default. Each time the **Apply** button in the post-processing section is clicked, the version of the result file before the post-processing step is saved in the result folder. It has the same name as the original result file and a numbered suffix .bck#, where # indicates the version of the file (starting with 1). The back-up file can still be opened in a text editor (e.g., Notepad++). Simply delete the suffix to have a result file that can be read by the GeoDict Result Viewer. In this way, intermediate post processing results can be saved, but additional hard disk space is needed.

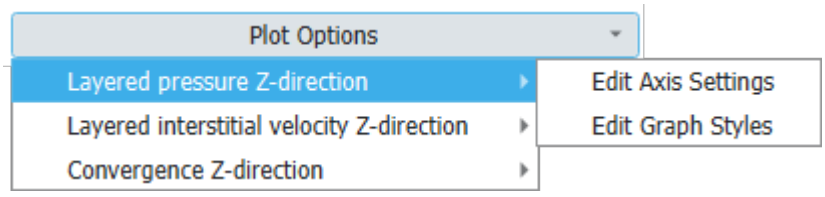
Know how! The backed-up files have the same size as the original result file. Thus, for large result files, we recommend to deactivate the **Back-up result file** option.

The post processing steps are also stored in the Session Macro and can be used to generate a new macro file. Details are explained in the GeoPy Scripting User Guide.

Note! The User Guides of the corresponding GeoDict modules explain in more detail the post-processing settings available for commands that are specific to those modules.

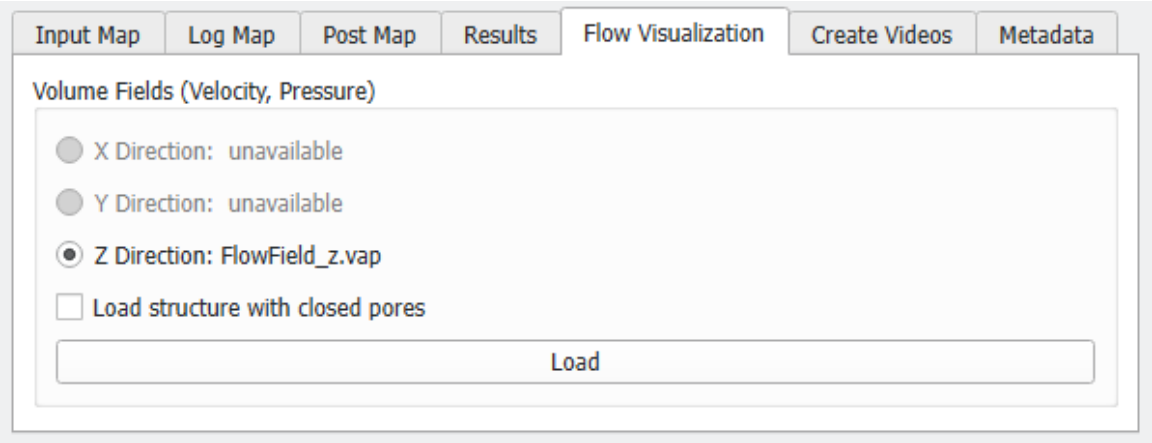
At the bottom of the post-processing panel find the **Plot Options** menu. For each plot in the **Plots** tab the **Edit Axis Settings** and the **Edit Graph Styles** options are available.

They can also be accessed via a right-click in the corresponding plot. For an explanation see Edit Axis Settings and Graph Styles 27.



Visualization

The **Visualization** tab appears only for commands that write additional files for visualization. The **Visualization** tab often takes the name **Data Visualization**, but the name may be adapted to the GeoDict module that produced the result file (e.g., for FlowDict results it is named Flow Visualization).



Load additional result files saved in the result folder from the **Visualization** tab, e.g., volumetric result fields, particle trajectories, or triangulation meshes.

The **Load** button is grayed out if a file is **unavailable**, e.g., because calculating this result and saving it was turned off in the command's Output Options when setting the simulation. This is also the case if the file was deleted from the project folder.

As default, suitable view settings are automatically applied when a volume file is loaded via the **Visualization** tab. For example, if a **.g32* index file from **GrainFind** → **IdentifyGrains** is loaded, the visualization automatically switches to show only the index image and you are asked if the structure visualization should be set to invisible.

Note! The User Guides of the corresponding GeoDict modules explain in more detail whether additional files might be outputted for a command and are available for visualization.

Create Videos

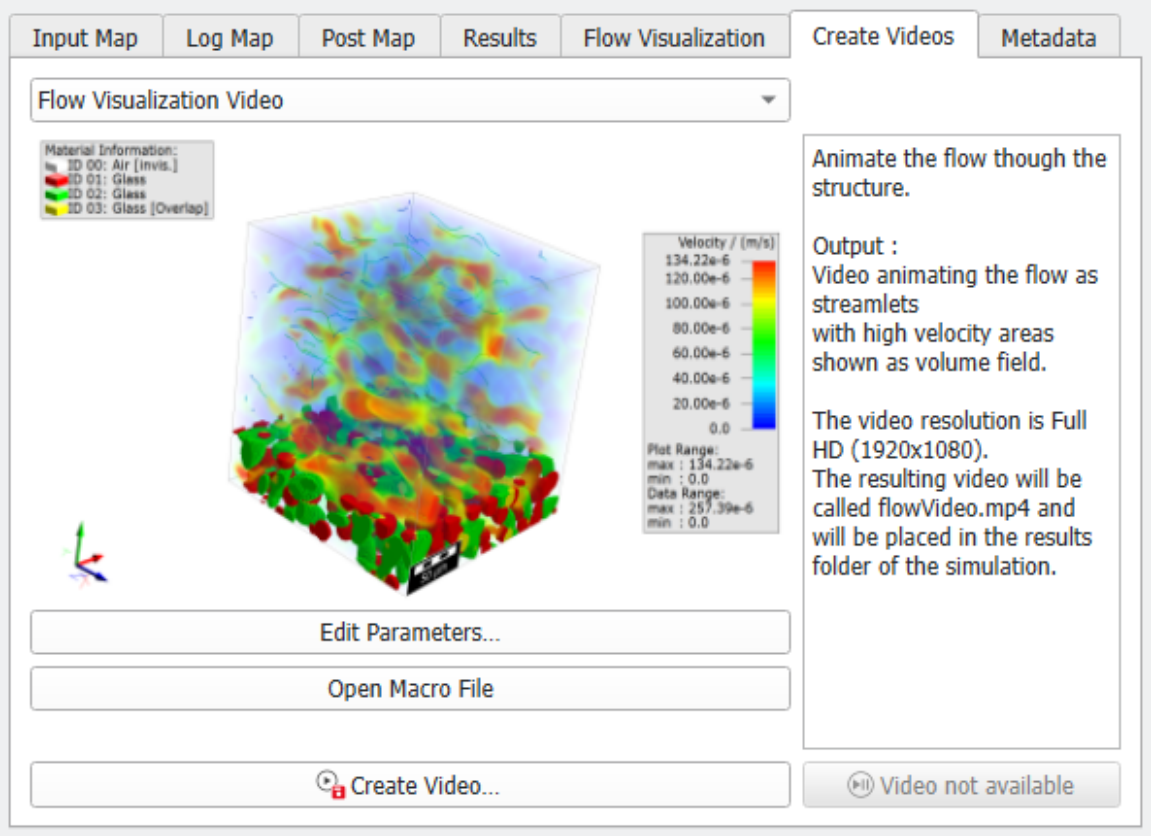
Some modules and their commands come with the capability to generate videos automatically from the results. In this case the **Create Videos** tab is displayed in the

Result Viewer. The process of generating the video is automated through a Python macro that loads the result data into GeoDict, sets up the needed view options for optimal visualization, and then outputs a video.

Creating a video is especially handy if there are time sequences in the result data. For example, some of the modules and the commands to create result videos are:

- GrainFind:IdentifyGrains
- PoroDict:IdentifyPores
- FlowDict:SolveStokes
- SatuDict:CapillaryPressure
- FilterDict:MediaLifeTime
- ElastoDict:SolveDeformations
- BatteryDict:ChargeBattery
- BatteryDict:ChargeElectrode

For some commands, there are multiple types of videos available that can be chosen from the drop-down menu directly at the top of the tab.



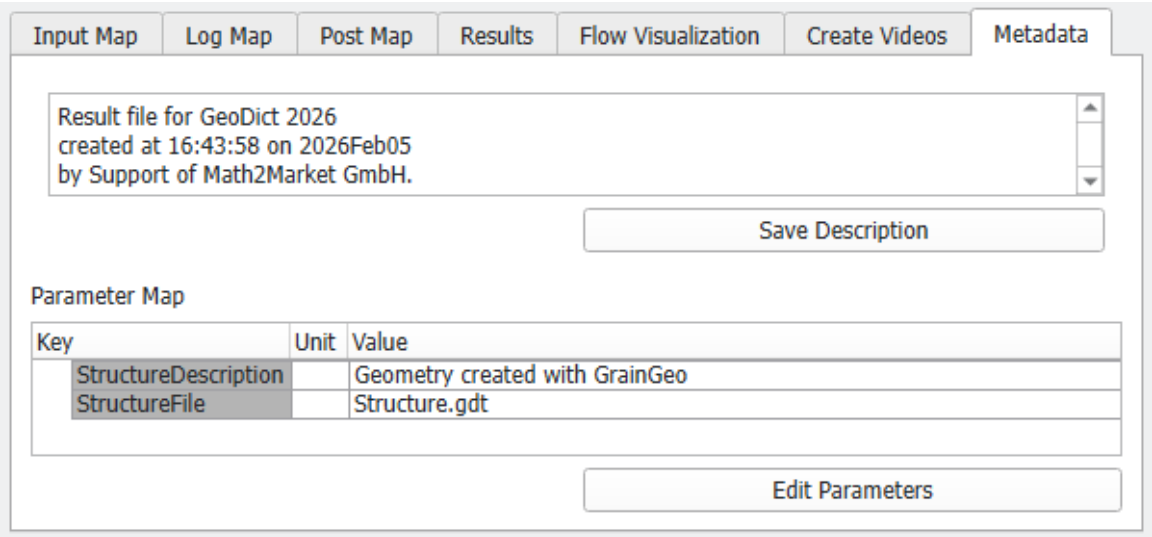
Below the drop-down menu, a panel includes a **preview** of the video and **instructions**, on the right. By clicking the buttons under the preview, it is possible to **Edit Parameters** for the creation of the video, to open the macro file controlling it (**Open**

Macro File) in a text editor for modifications, and finally, to create the video (**Create Video...**). When the video was created, the greyed-out button **Video not available** changes to **Play Video**.

Click on it to open and watch the video in an external video player.

Metadata

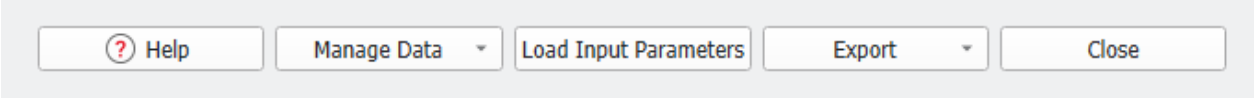
Under the **Metadata** tab, create your own descriptions, notes, or remarks about the results and save them by clicking **Save Description**.



Under **Parameter Map**, GeoDict also shows the values of parameters that might not be saved in other maps, such as the values for **Expert Settings** or input values from a **Python Macro**. These can be modified by clicking **Edit Parameters**.

1.2.3 Bottom Section

The five buttons found in the **Bottom Section** start actions concerning only the data for the result file shown in the **Result Section** above. If three or more result files are selected and shown in the Result Section, the Bottom Section is not displayed anymore. But, as explained in the Header Section 9 topic, the options are available in the right-click context menu.

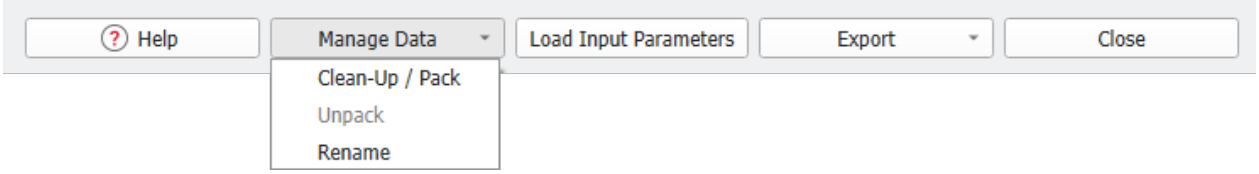


Help

The Help button leads to the Result Viewer section 3 of this user guide.

Manage Data

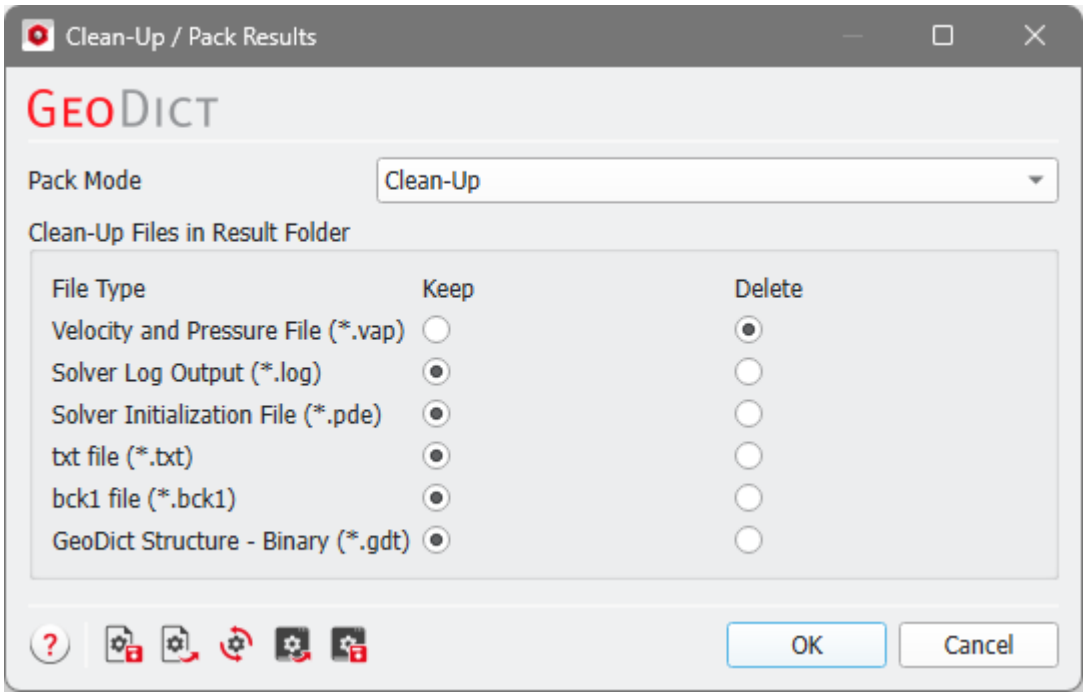
Click **Manage Data** and select **Clean-Up / Pack** to delete or compress additional files that are saved in the result folder of the current result file. Compressed files can be easily **Unpacked** through the result viewer. You can also **Rename** a result file and its result folder at once.



Know how! You do not have to open the result folder in the file explorer to rename or delete simulation result data, everything can be done from the result viewer!

Clean-Up

When the Pack Mode **Clean-Up** is selected, you must decide for every file type in the result folder if it should be kept or deleted. The chosen action will be applied to all files of this type.



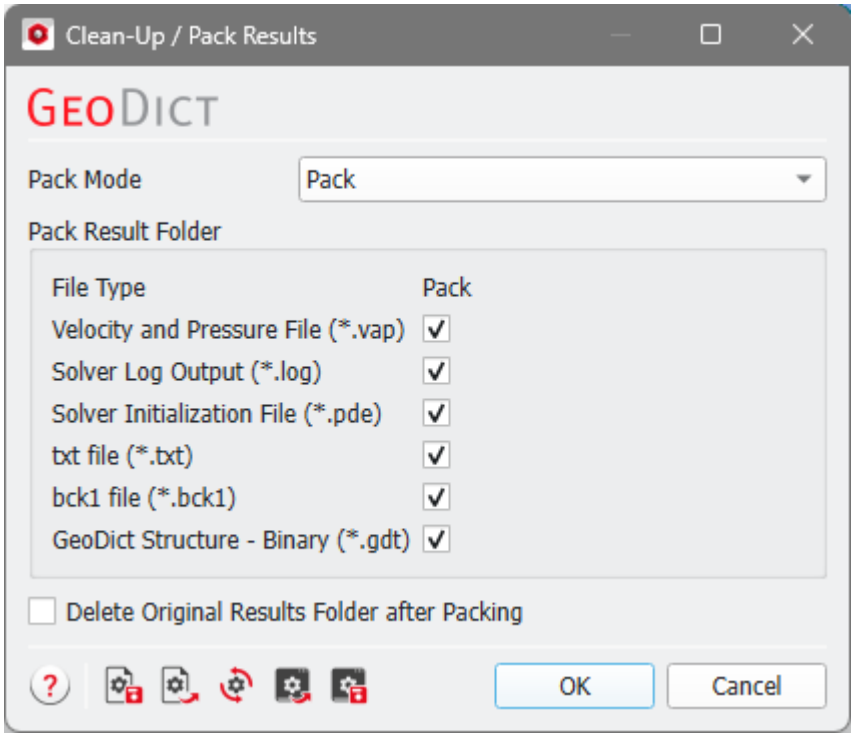
Note! Fast cleaning-up of the result folder is useful if, for example, a FilterDict simulation with many batches was done and you want to keep the particle trajectories but not the Velocity and Pressure files (*.vap) of each batch. Then, you can easily delete all *.vap files from the result folder and all its sub-folders.

Pack & Unpack

When choosing **Pack**, select all the files which should be contained in the compressed ZIP-folder. The original files are still present in the results folder. Check **Delete Original Results Folder after Packing** to delete the original result folder.

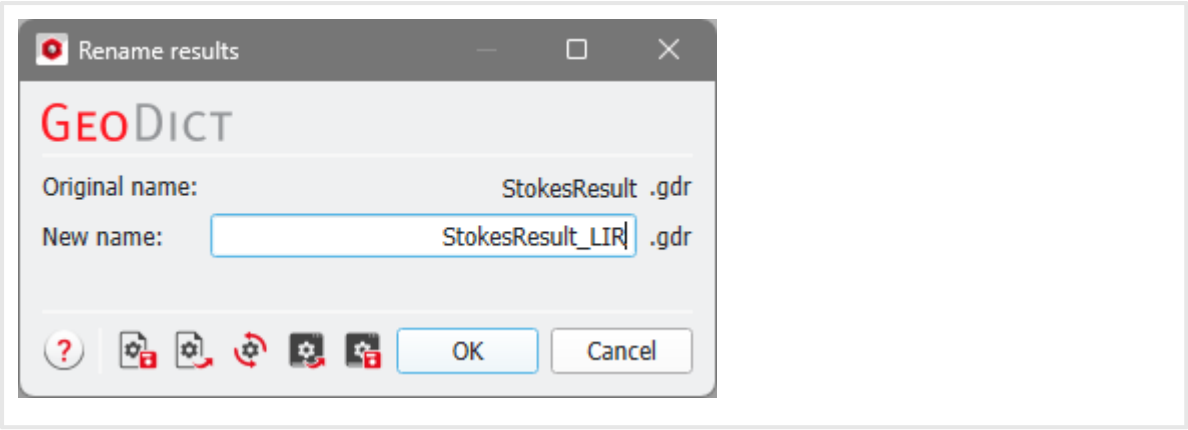
Important! Delete Original Results Folder after Packing will delete the whole results folder including the files which were not added to the compressed ZIP-folder!

When using this option (or after deleting the original result folder manually), the ZIP-folder can later be decompressed with the **Unpack** option.



Rename

Select **Rename** to change the name of the result file to a new one. Additionally, the name of the result folder associated with the result file is also renamed accordingly. Thus, additional files belonging to this result file are still connected to the renamed file and can later be opened from the Result Viewer.



Load Input Parameters

Clicking the **Load Input Parameters** button loads all the settings displayed under the Input Map 11 tab into the corresponding command options dialog of the corresponding GeoDict module. This can be verified as follows:

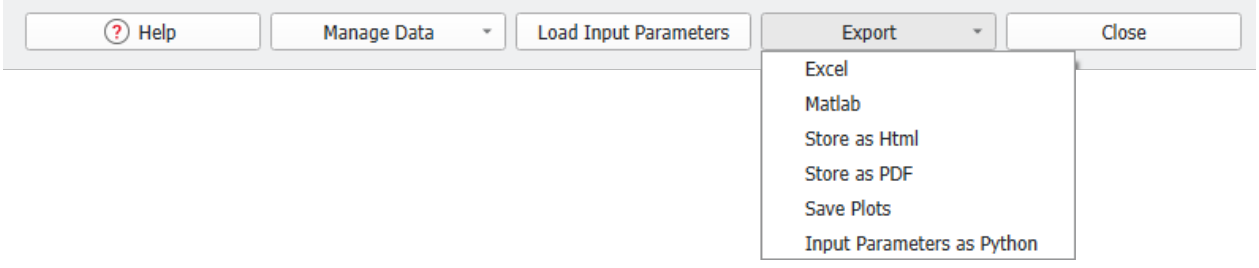
1. Check the **Module** and **Command** that appear in the table of the Header Section 9 of the Result Viewer.
2. Navigate to the GeoDict GUI, and select this GeoDict **Module**.
3. Open the options dialog for the **Command** and observe that the default (or other) settings have been replaced by those from the input map.

Taking the settings directly from the input map to run the corresponding command again on the same structure produces exactly the same results. This can be very useful to run the same digital simulation experiment on a variety of structure models. Of course some settings can also be changed to adjust the command accordingly.

Know how! Use this option if you directly want to re-run the (adapted) simulation or view the simulation settings in the GeoDict GUI.

Export

Click the **Export** button to export values from the result file in different formats.



With **Excel** the result file is exported as an MS Excel *.xlsx file using Python. The installation of MS Excel on the computer is not needed, but if it is installed, the exported file opens automatically. This makes the export of results to Excel on computers with Linux Operating Systems possible.

The **Matlab** export command starts MATLAB and automatically loads the *.gdr result file there.

Of course, this only works if MATLAB is installed on the computer.

Selecting **Store as Html** exports only the result report tab to a *.html file, **Store as PDF** saves it into a PDF file.

Save Plots can be used to save all plots contained in the result file as images (*.png or *.svg). The size and resolution in DPI can be specified and apply for all plots. Thus, they all have the same size and resolution. The image files are saved into the result folder belonging to the result file. If you want to set individual sizes for different plots use the Save Image option in the plot context menu.

Select **Input Parameters as Python** to save the input parameters shown in the Input Map as a GeoPy (.py) script. Running the script sets the input parameters and executes the according command.

Know how! Use this option if you want to re-run the (adapted) simulation again at a later time.

Close

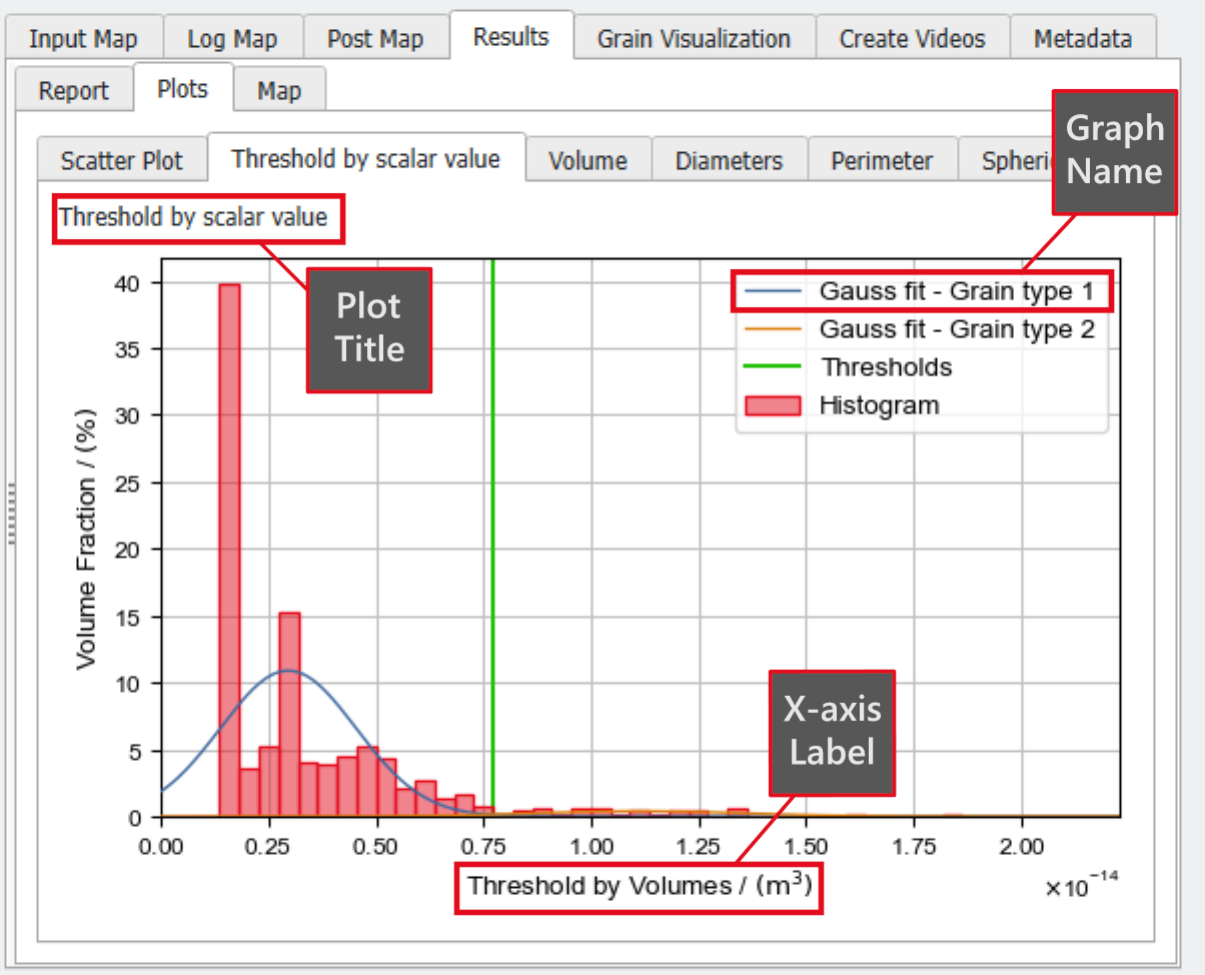
Clicking the **Close** button closes the **currently shown** result file and the next result file from the list, if multiple files are opened, is shown in the Result Viewer. Otherwise the **Result Viewer** is closed. Closing the **Result Viewer** by clicking the close button () at the top right of the Result Viewer, closes **all** result files at once.

1.3 Plots in Result Files

Many result files have a **Plots** subtab under the **Results** tab of the **Result Viewer**. In the **Plots** subtab, the values are displayed using different types of plots. All data for each plot is saved in the **Post Map** of the result file. If any changes are made in the plot settings or if a post-processing step is done, the result file is updated and saved. Thus, plot appearance stays the same even after closing the **Result Viewer**.

This updating and saving of plot settings can also be recorded as a macro and always appears in the Session Macro. This means that everything done while post-processing to the plots can also be scripted/automated using Python macros.

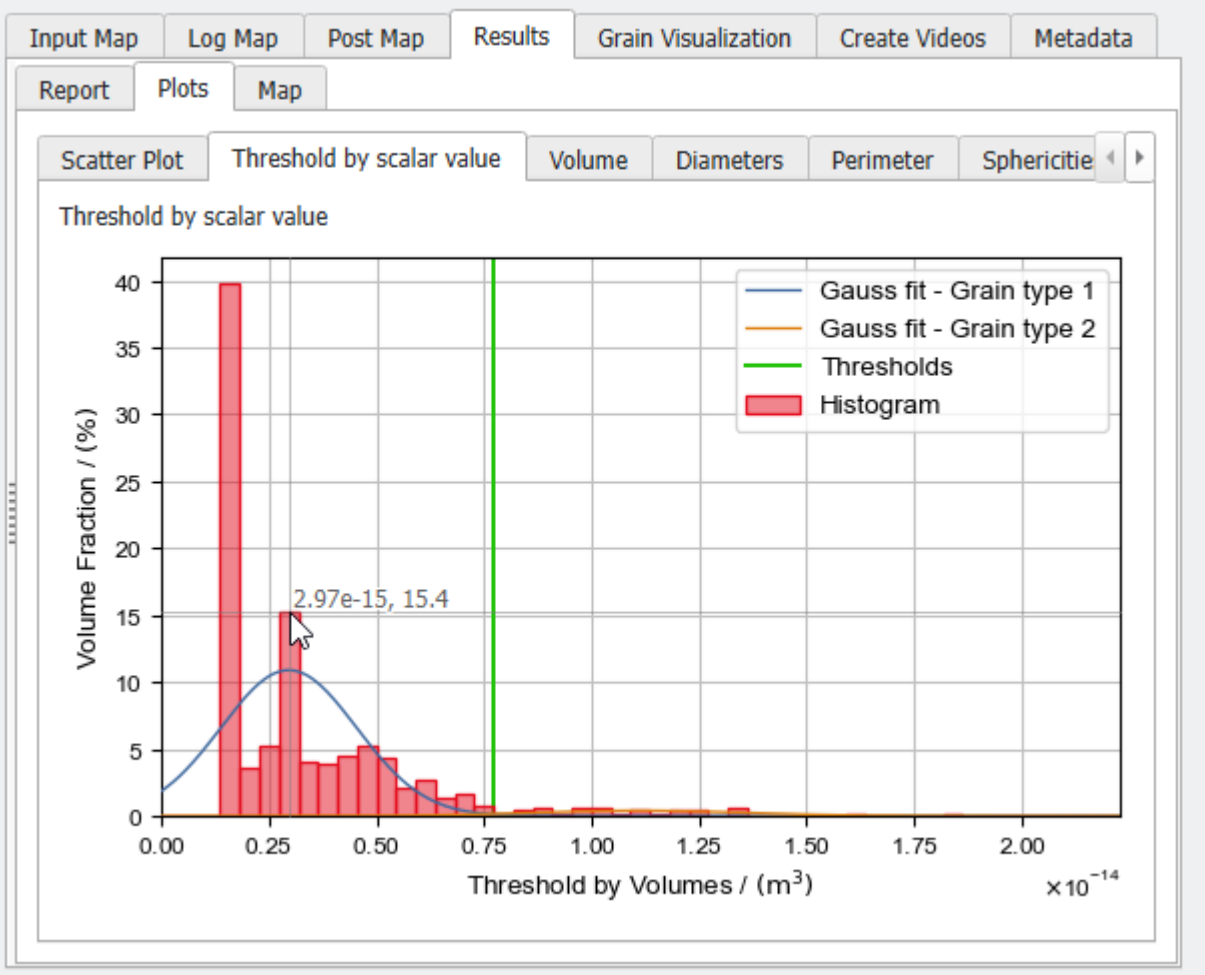
Each plot has a **Plot Title**, which is shown above the plot. A plot contains at least one graph: a set of data points, which has a unique marker and color, e.g., a blue line or a set of bars. All graphs in a plot share the same value range in X and Y axis directions and have unique **Graph Names** that are shown in the legend. Both the X-and Y-axis have a **Label** showing the quantity that is plotted and the corresponding unit.



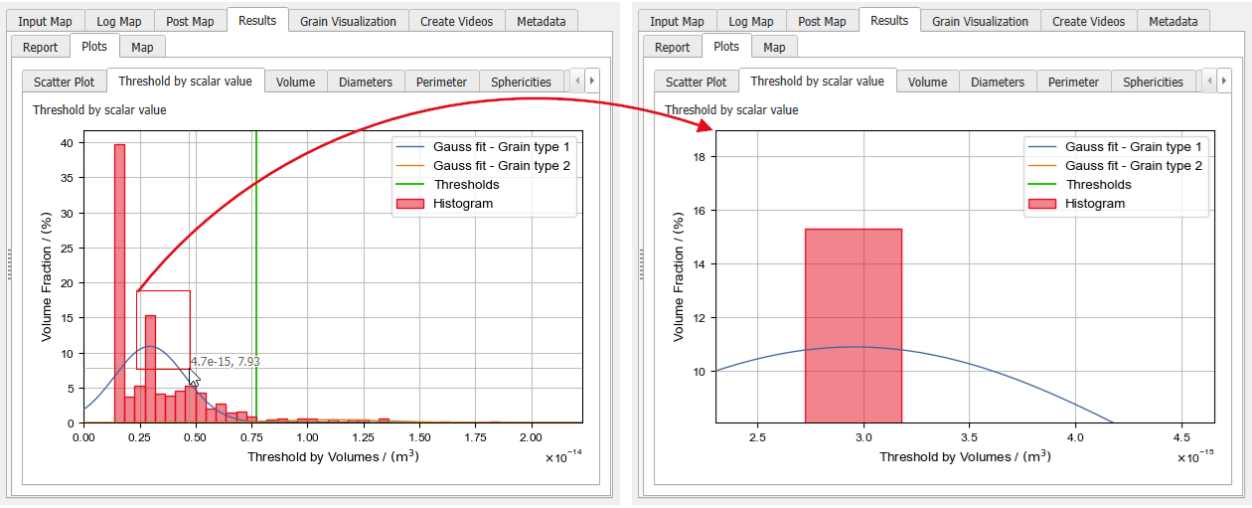
Each plot can have multiple graphs and each result can have multiple plots. Thus, some plots are grouped in other subtabs, for higher simplicity. If there are more than two plots in a subtab, the active plot can be chosen from a drop-down menu. This is often the case for time sequences or similar types of data.

If there are exactly two plots in a group, both of them are displayed. The two plots are separated by a horizontal slider, such that the size can be adjusted.

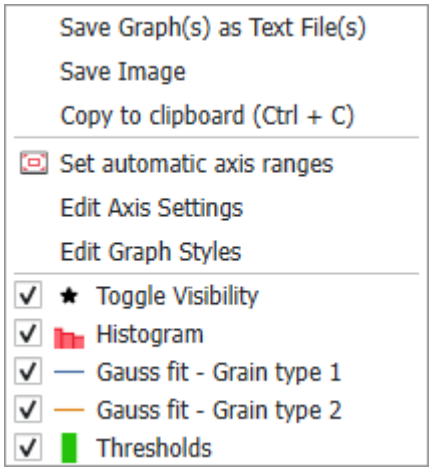
Hovering with the cursor over the plot will show indicator lines and the values of the axes at the point of the cursor. The first value shown belongs to the X-axis, the second one shows the value of the Y-axis.



Click and hold with the left mouse button and draw a rectangle to zoom into the plot at this position.



Right-clicking inside the plot area opens the **Plot Context Menu**. Here, the plot can be exported in different ways, and the plot settings can be changed, especially the zoom can be reset with **Set automatic axis ranges**.

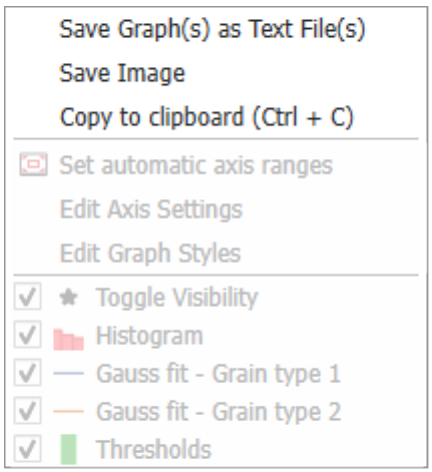


The Plot Context Menu:

- Save Options 25
  Save the graph(s) as text files and save the whole plot as image.
- Edit Axis Settings and Graph Styles 27
  Modify how the data is plotted.
- Graph List 36
  Quickly define the visibility of the graphs in the plot.

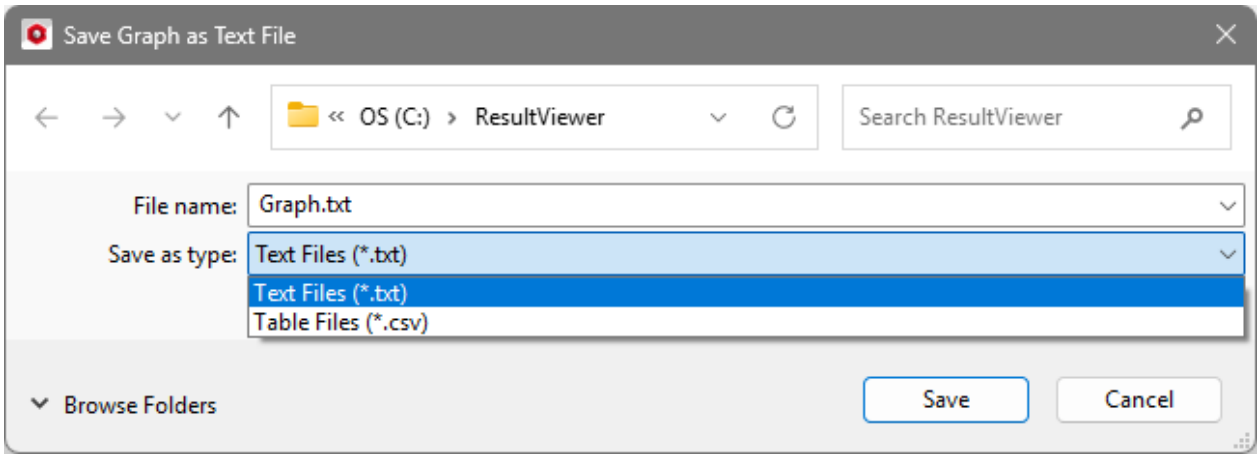
1.3.1 Save Options

The first section of the **Plot Context Menu** offers the possibility to save the plot data in different formats.



Save Graph(s) as Text File(s)

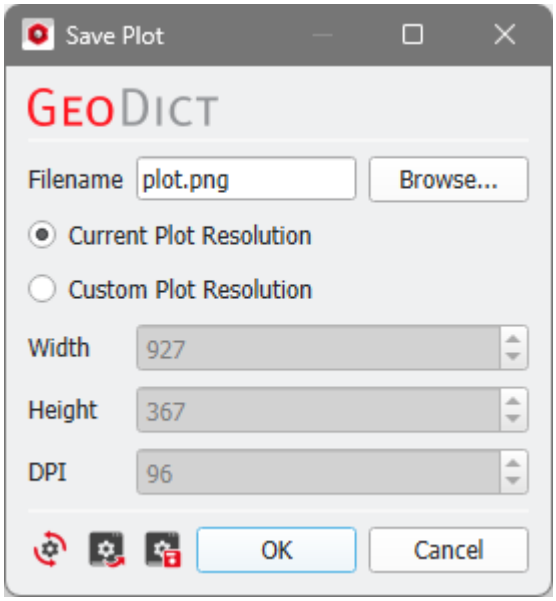
In the opening **Save Graph as Text File** dialog choose to write all graph points as *.txt or *.csv files. Each graph of the plot is saved in an individual text file.



Save Image

All plots can be saved as pixel (*.png) or vector (*.svg) image by clicking on **Save Image**. In the **Save Plot** dialog, choose the filename and the resolution of the saved image.

Via the **Browse...** button, choose the image format (*.png/*.svg) or simply type in the desired filename with suffix.



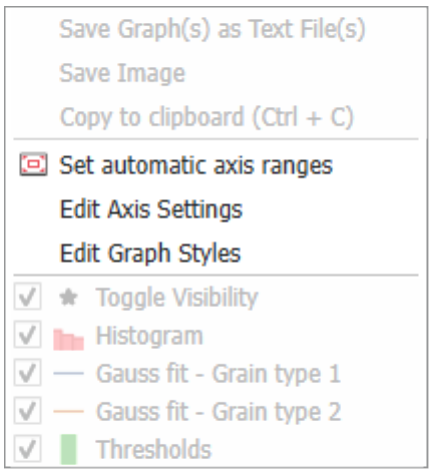
The **Current Plot Resolution** depends on the current size of the Result Viewer window and the values for **Width**, **Height**, and **DPI** (dots per inches) are entered automatically. With the **Custom Plot Resolution** option, these values can be freely chosen. Define a higher DPI value to increase font size and line thickness for better visibility, e.g., when using the plot for presentation slides.

Copy to clipboard (Ctrl + C)

Select **Copy to clipboard (Ctrl + C)** to copy the plot as image to the clipboard (or press Ctrl + C) and then paste it (Ctrl + V) where needed.

1.3.2 Edit Axis Settings and Graph Styles

With the options in the second section of the **Plot Context Menu** you can edit the way the data is shown in the plots. The settings for the axes and the graphs are done separately.

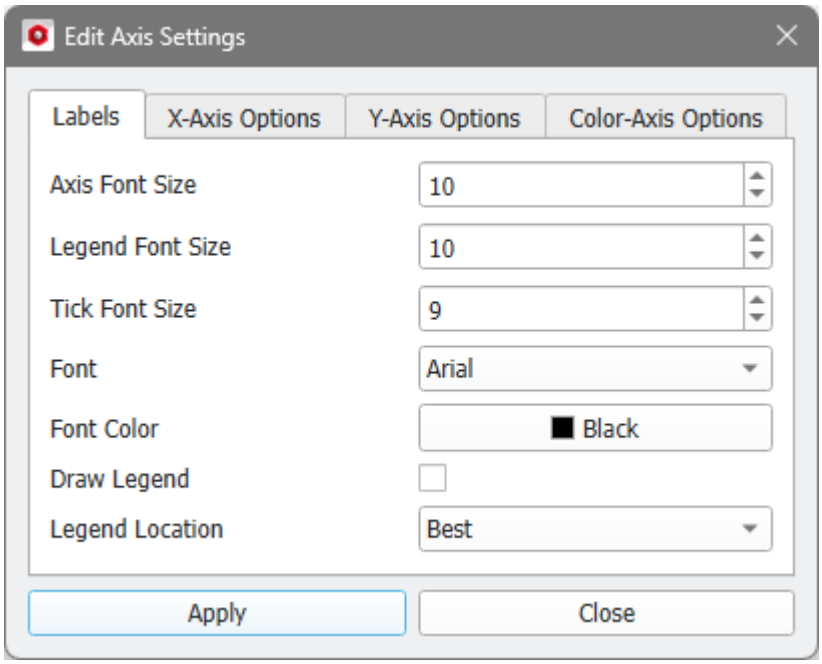


Set automatic axis ranges

This will reset the axis settings to be automatic. If you zoomed into the plot, this will reset the zoom to the initial value. This is the same as checking the **Scale Axis automatically** box in both the X-Axis Options tab and the Y-Axis Options tab of the **Edit Axis Settings** dialog.

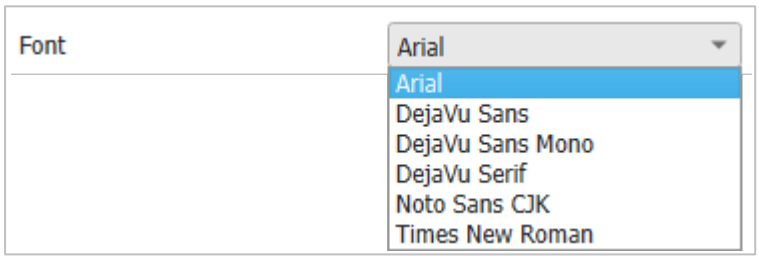
Edit Axis Settings

Selecting **Edit Axis Settings** opens a dialog to specify the settings for labels and axes. It is organized in up to four sub-tabs **Labels**, **X-Axis Options**, **Y-Axis Options**, and **Color-Axis Options**. Clicking the **Apply** button updates the plot and saves the new settings via the **PlotSettings Command**. Thus, the changes are also stored in the Session Macro. The new settings are automatically saved to the result file, such that the plot appearance is the same when opening the result file again.



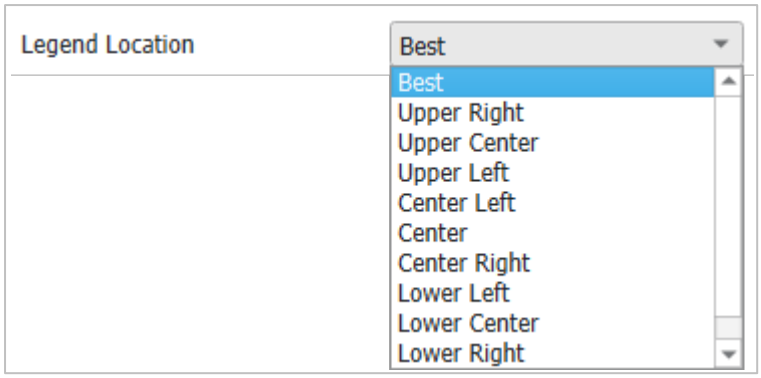
Labels

In the **Labels** tab, the **Axis Font Size**, the **Legend Font Size**, and the **Tick Font Size** can be changed. The chosen **Font** will be the same for the axis labels, the ticks and the legend. For the axes labels and the ticks, the **Font Color** can be chosen (default is black). The settings apply for both the x- and y-axis.



Note! The default font (Arial) cannot handle Japanese characters. Select the font **Noto Sans CJK** to display them properly.

Check **Draw Legend** if you want to display the legend and select the **Legend Location** as desired.



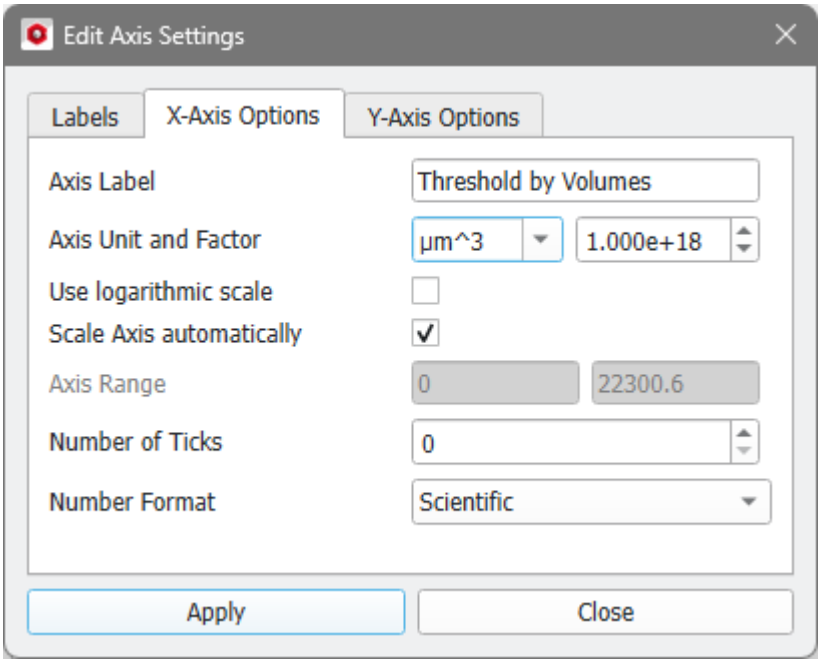
X-Axis and Y-Axis Options

The two tabs **X-Axis Options** and **Y-Axis Options** have the same options, but each axis has its own set of options.

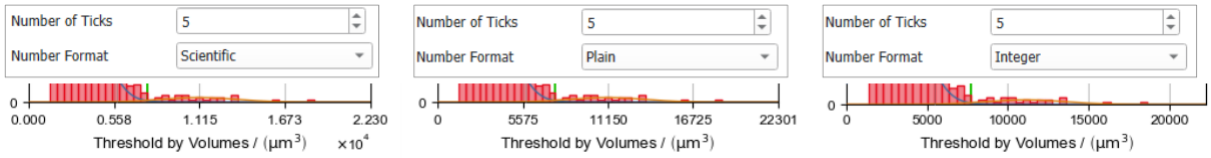
The text shown below the axis is changed in the **Axis Label** box.

The **Axis Unit** and thus the numbers shown at the ticks can be manipulated. Choose the new unit from the drop-down menu and the **Factor**, with which the tick numbers have to be multiplied, is automatically adjusted. But you can also enter custom units and have to set the factor correctly.

It is also possible to **Use logarithmic scale** for every axis by checking it. By default, each axis is scaled automatically, such that all values are plotted. Uncheck **Scale Axis automatically** to enter a custom **Axis Range** (minimum and maximum values) in which the data is plotted.



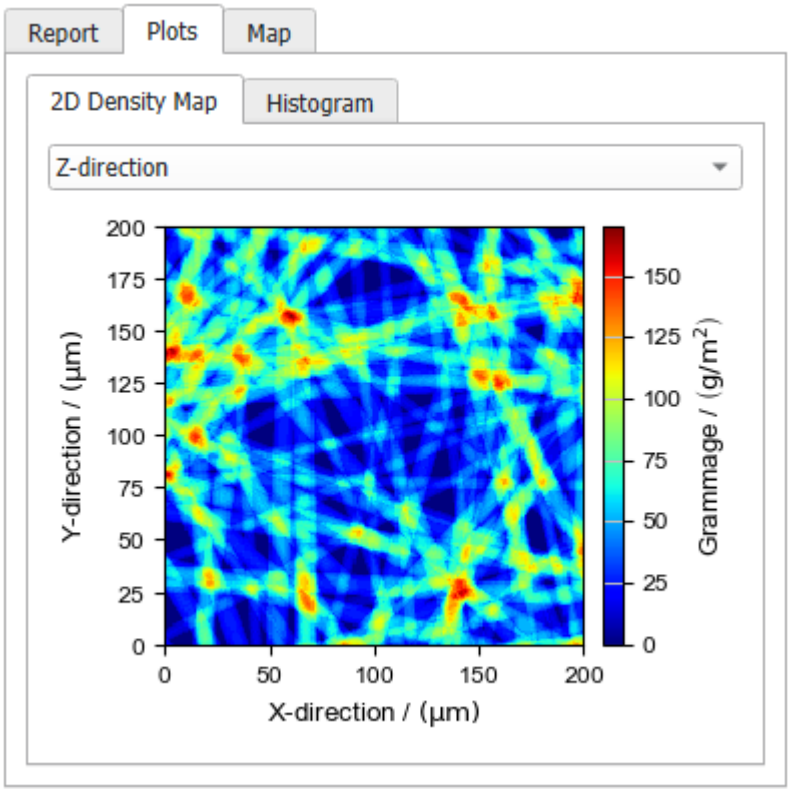
If **Number of Ticks** is set to zero, an appropriate number of ticks is automatically chosen, otherwise the ticks are evenly distributed between the minimal and maximal data values. Choose the **Number Format** to be **Scientific**, **Plain**, or **Integer**.



Color-Axis Options

The **Color-Axis Options** tab is only available for plots of special commands, e.g., for the 2D Density Map command in MatDict (shown here) or results from GrainFind and FiberFind.

In these plots, two-dimensional data is visualized using a color map.

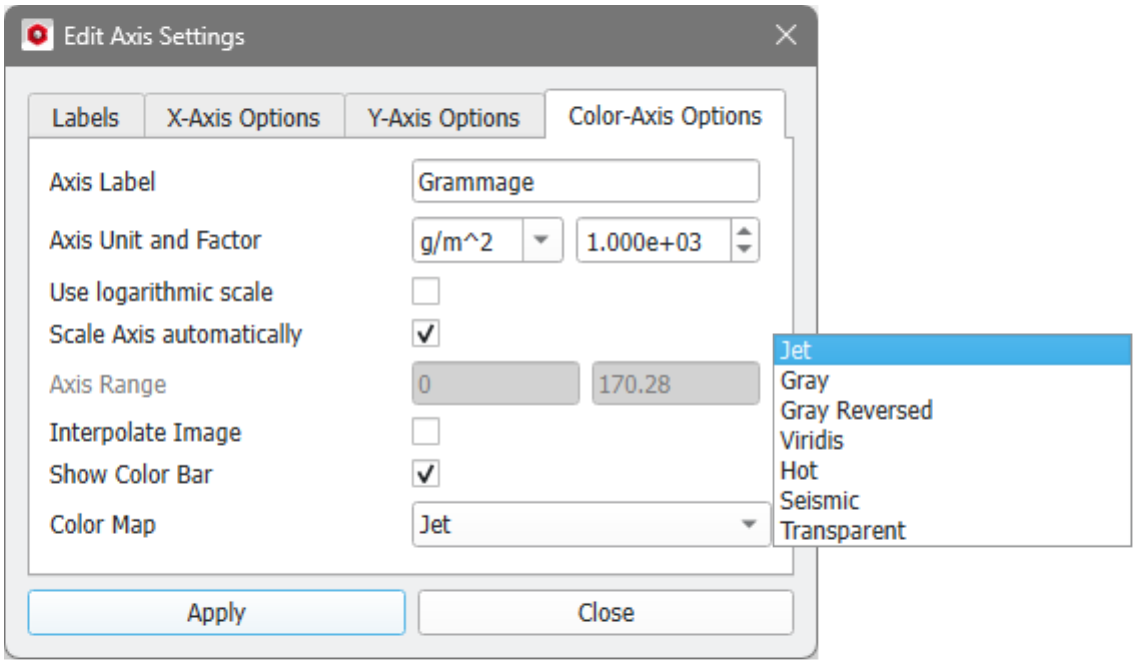


The settings for the color axis are similar to those for the X- and Y-axis. The options **Axis Label**, **Axis Unit and Factor**, **Use logarithmic scale**, **Scale Axis automatically**, and **Axis Range** are the same as described before.

Check **Interpolate Image** to get a smooth visualization.

The visibility of the color map shown on the right can be turned on and off with **Show Color Bar**.

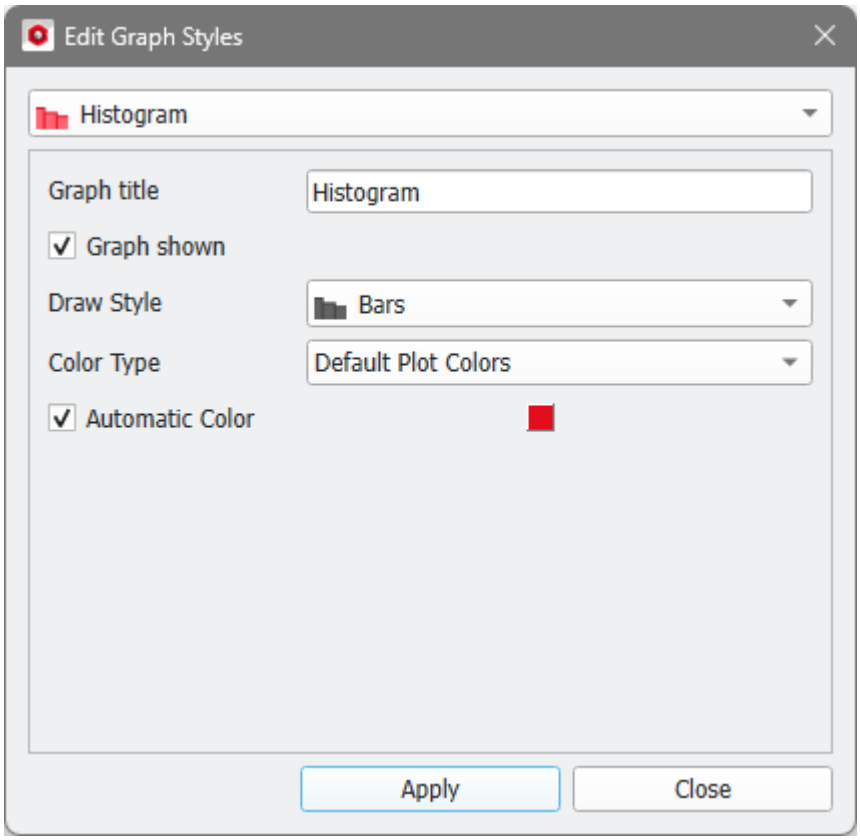
From the **Color Map** drop-down menu select a suitable color map. Find examples for the color maps in the Visualization chapter



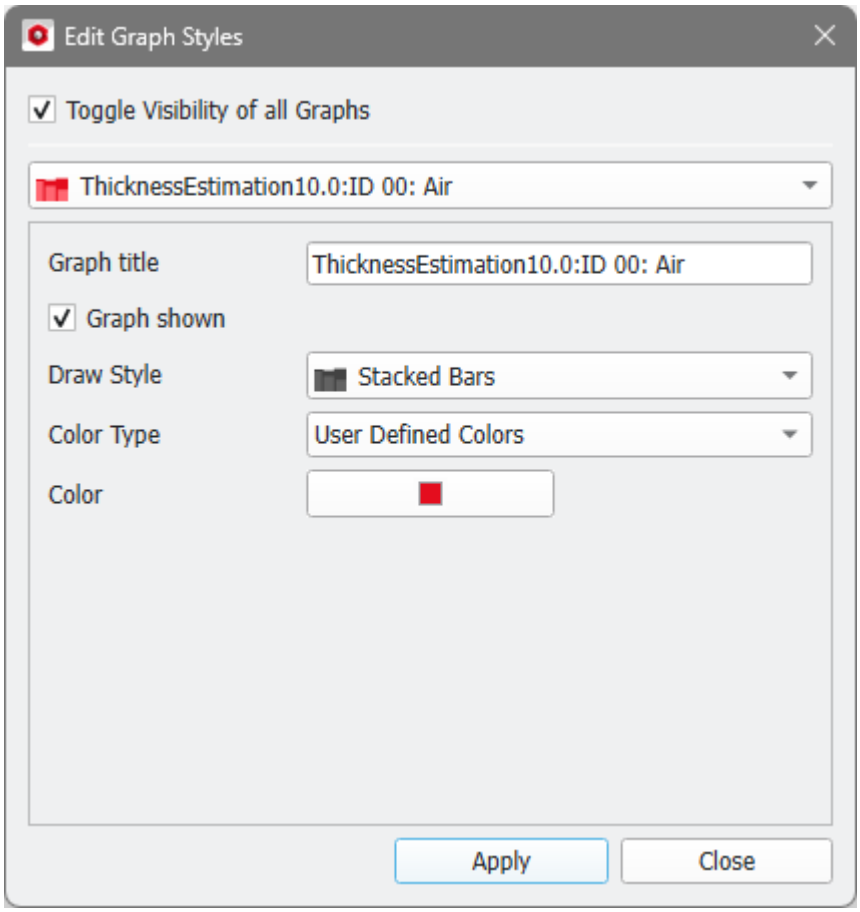
Note! If the Color Map **Transparent** is chosen, the color defined under **Edit Graph Styles** 31 will be used to display the data.

Edit Graph Styles

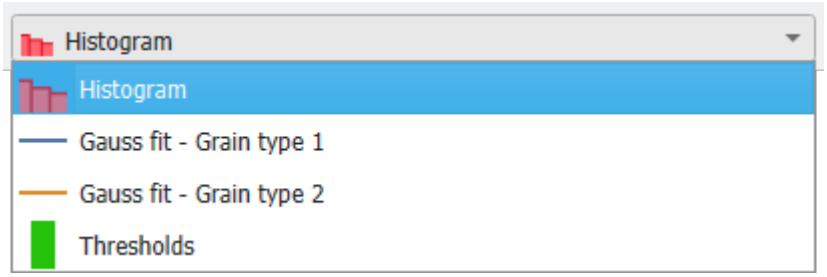
Clicking **Edit Graph Styles** lets you choose how each of the contained graphs is displayed. Changes made here are saved to the result file and recorded via the **PlotStyle Command**.



If a plot contains more than 16 graphs, the **Edit Graph Styles** dialog contains the option to **Toggle Visibility of all Graphs** at the top. For plots with less graphs, this option is available in the **Plot Context Menu** (see also the topic Graph List).



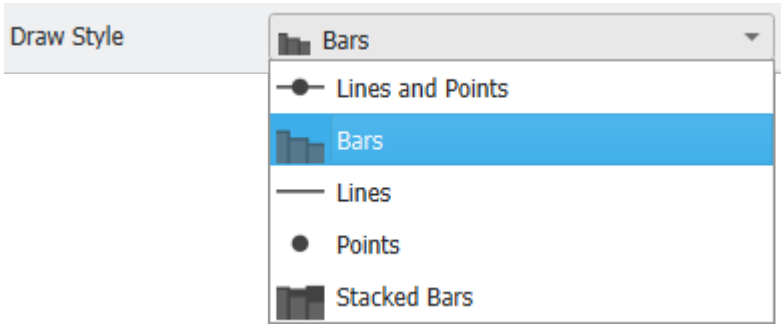
From the pull-down menu select the graph you want to edit.



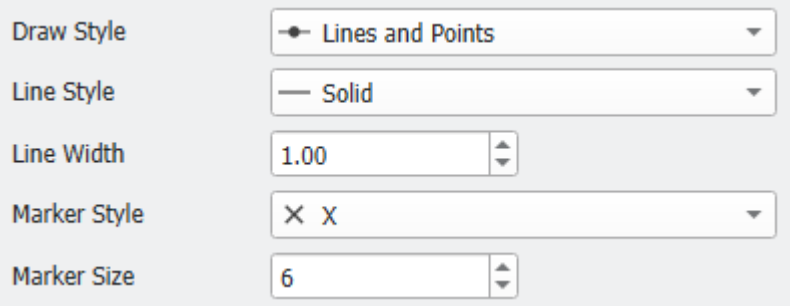
Change the **Graph title** to give the graph a new name, which is also displayed in the plot legend. Below, check **Graph Shown** if it should be displayed in the plot or uncheck to hide this graph.

Style Options

Choose a **Draw Style**, which can be **Lines and Points**, **Bars**, **Lines**, **Points**, and **Stacked Bars**.

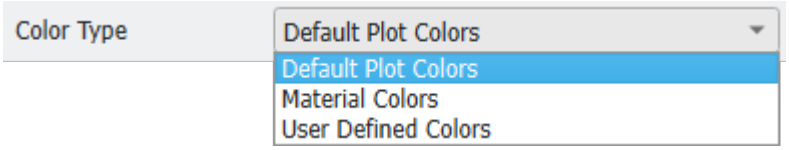


If a graph contains Lines and/or Points, the **Line Style** (Solid, Dashed, Dotted or Dashdot) and **Line Width** and/or **Marker Style** (different shapes) and **Marker Size** can be edited.



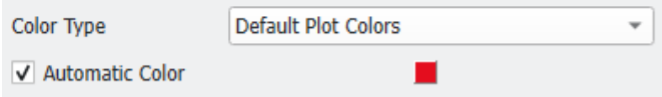
Color Options

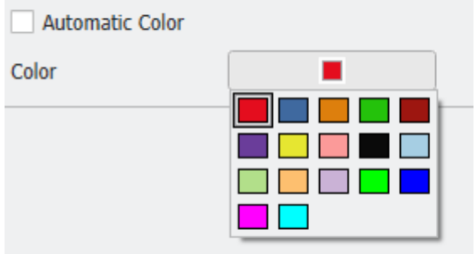
The color of the graph can be edited by first choosing a **Color Type.**



Default Plot Colors

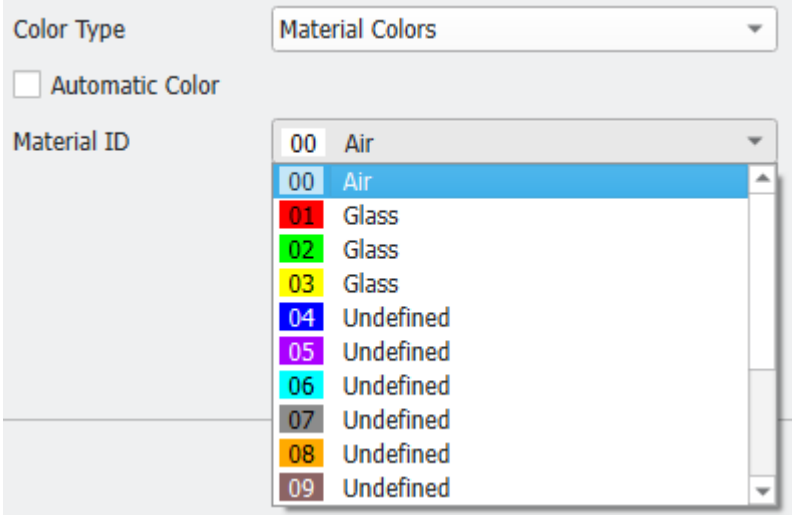
By default, an **Automatic Color** is selected, which can be changed when unchecking the box. Then, a small selection of default colors is available.





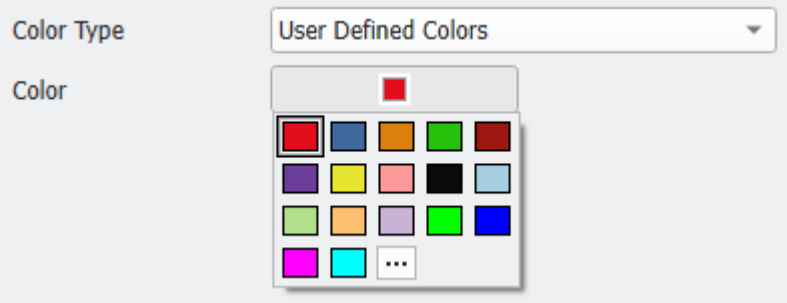
Material Colors

For **Material Colors** the same color map as for the Material IDs is used. If an Automatic Color is assigned, the first plot has the color of Material ID 00, the second plot has the color of Material ID 01, and so on.

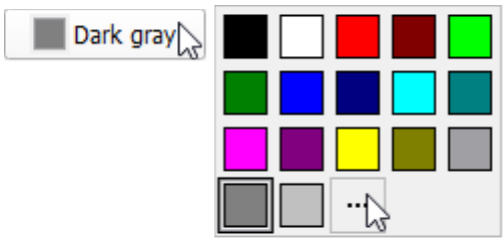


User Defined Colors

For **User Defined Colors**, there is no Automatic Color.

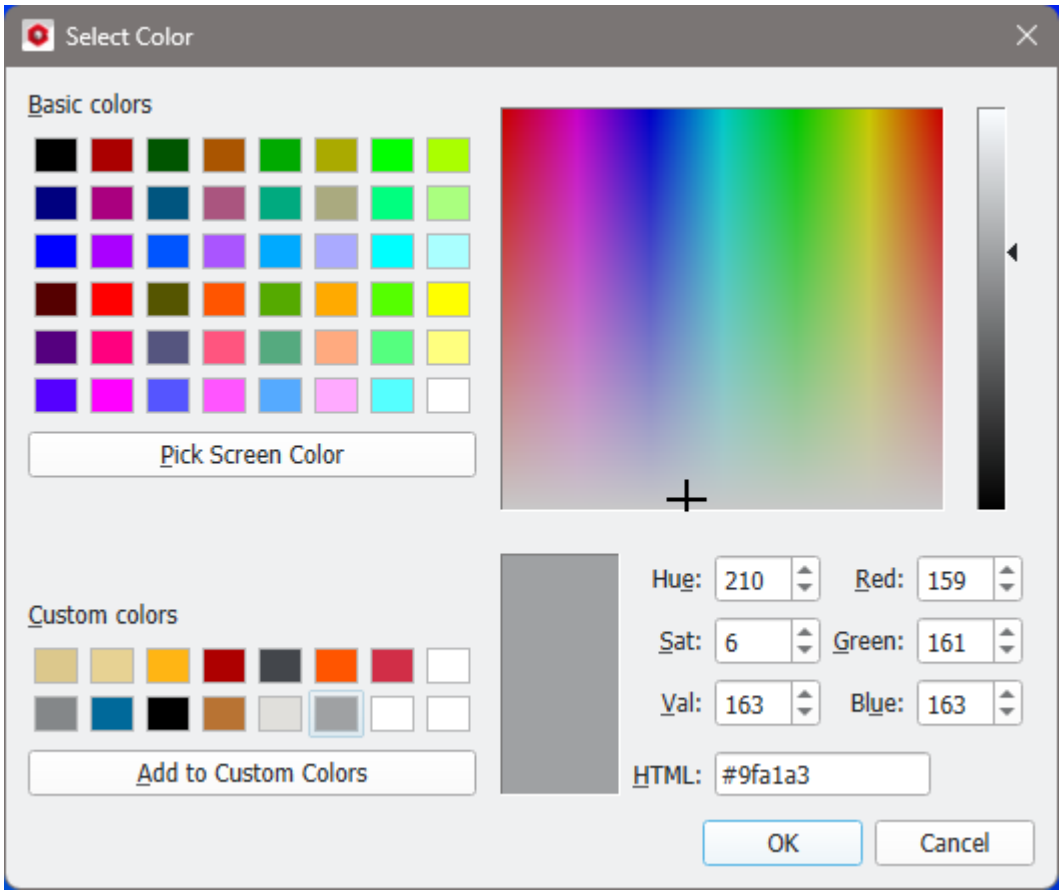


To change the default color to a user-defined color, select the three-dots icon.

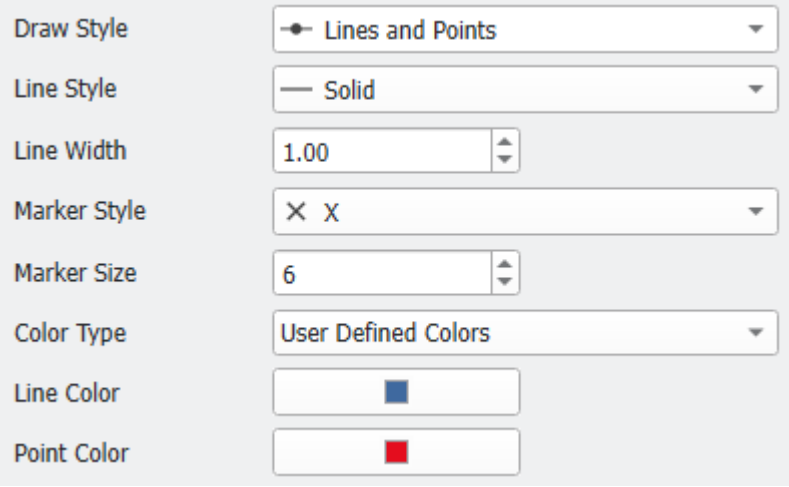


In the opening **Select Color** dialog enter the RGB values that define the color and click **Add to Custom Colors**. Alternatively, the rectangle for the new color can be dragged to occupy the place of the old one in the custom colors.

The new color setting is applied to the material after clicking **OK.**

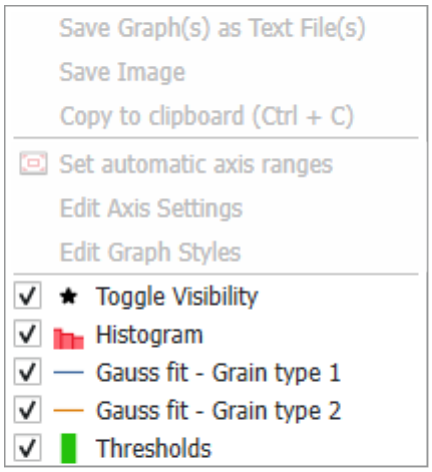


For the Draw Style **Lines and Points** it is possible to select different user defined colors for the lines and the points, respectively.



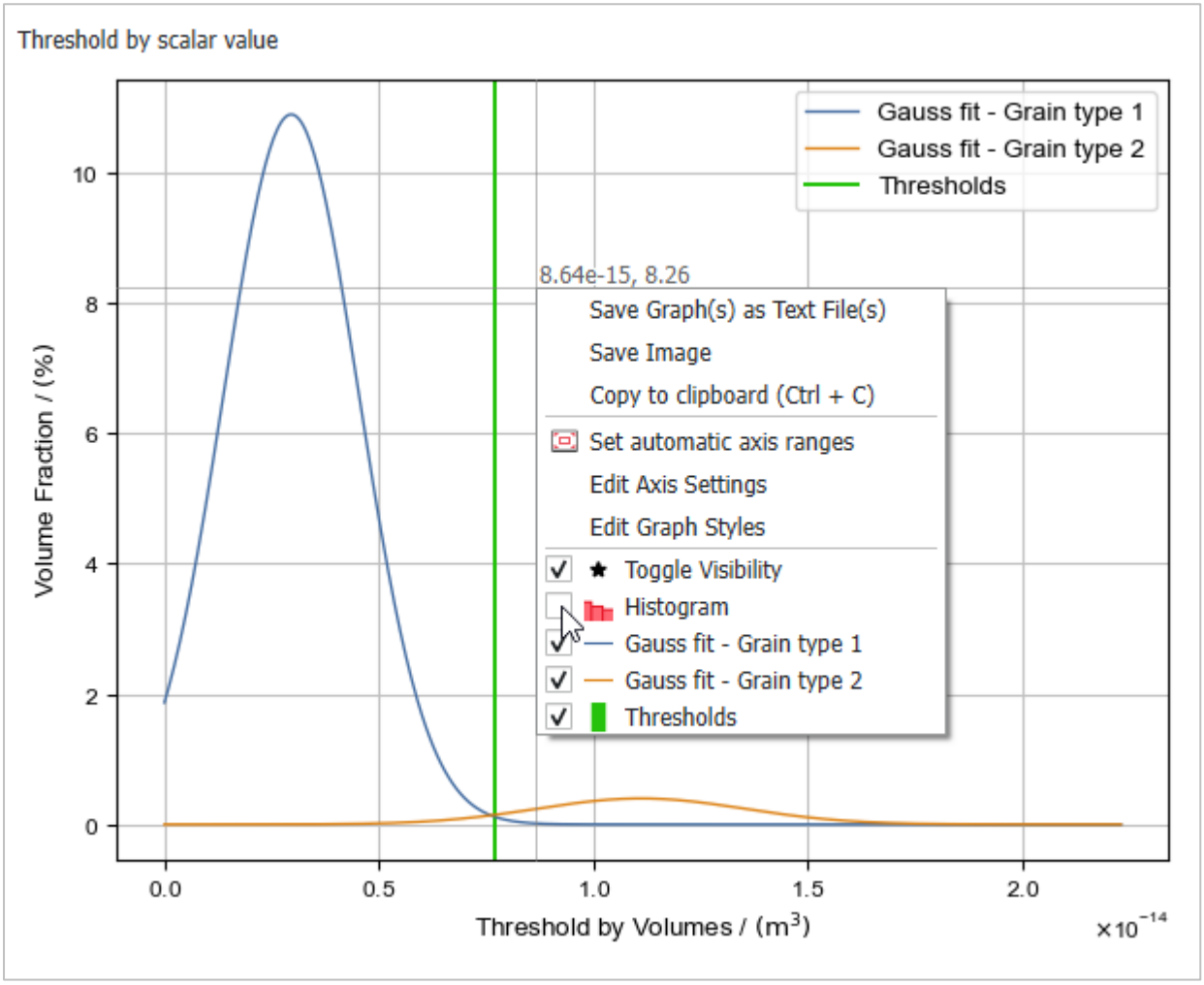
1.3.3 Graph List

Choose the graphs you want to be displayed in the third section of the **Plot Context Menu**.

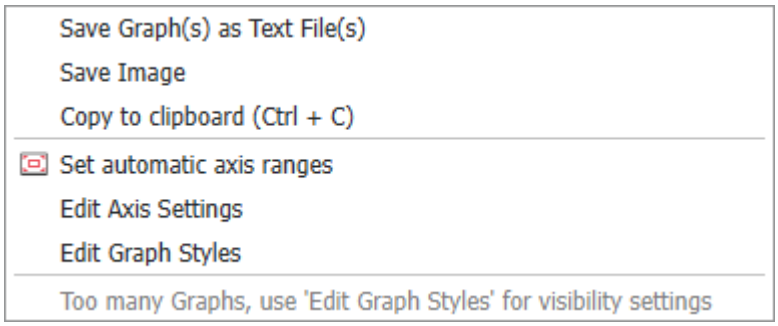


Clicking **Toggle Visibility** deactivates or activates the visibility of all the graphs in the plot at once.

Unchecking one of the entries deactivates the visibility of this particular graph in the plot visualization. It is also removed from the legend.



If the plot contains many graphs (16 or more), the visibility settings cannot be done in the Plot Context Menu anymore. Instead, they you have to use the Edit Graph Styles 31 dialog.


Save Graph(s) as Text File(s)
Save Image
Copy to clipboard (Ctrl + C)
Set automatic axis ranges
Edit Axis Settings
Edit Graph Styles
Too many Graphs, use 'Edit Graph Styles' for visibility settings

1.4 Combine Results

What are combined results files?

Combining results can be very useful to identify parameter correlations, especially when using GeoDict parameter macros. The chosen plots from the different result files are combined into one for easy comparison and custom plots can be generated from the map parameters.

From a set of result files, it allows to condense the important values in a well-arranged plot or table. In this way, result values that are not changed by differing input parameters are separated from result values of importance. This procedure makes it easy to qualitatively identify parameter correlation.

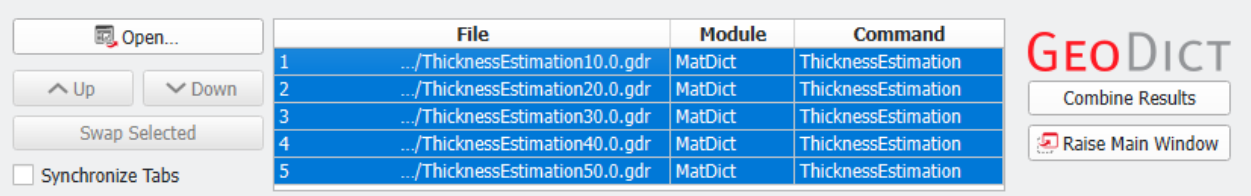
Just some examples:

- A generated structure with varying Solid Volume Percentage (SVP) for which you want to know the corresponding percolation path diameter.
- A flow simulation with varying fluid viscosity, for which you want to know the corresponding pressure drop.
- An imported 3D gray value image with varying binarization threshold, for which you want to know the corresponding relative diffusivity.

Topics of this section

1.4.1 Creating combined result files

Several selected result files can be combined into one single file by clicking **Combine Results** located at the top right of the Result Viewer's Header Section 9.

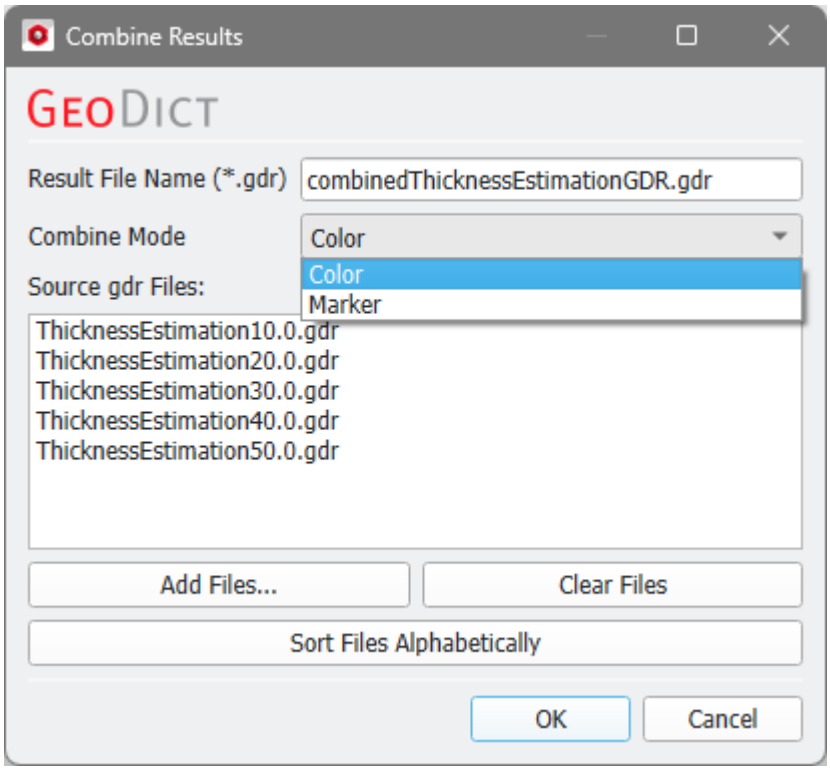


In the example, fiber structures with varying solid volume percentage (10% - 50% SVP) were generated with a python macro and the largest Percolation Path and a Thickness Estimation were computed for each structure.

You can download the ParameterStudy.py file used in this chapter for inspection and recreation in a zipped folder.

Download ParameterStudy.zip

Next, the **Combine Results** dialog opens to define combine settings and input files. In the **Result File Name** box at the top enter a name for the combined result file.

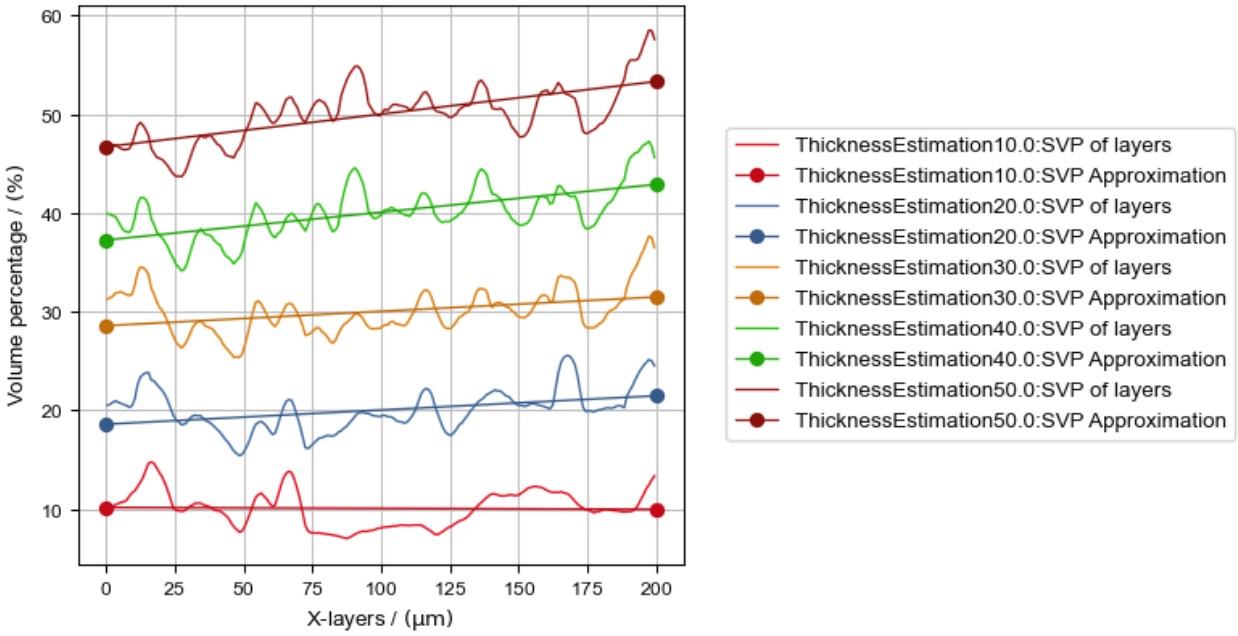


The list of **Source gdr Files** already contains all files that have been highlighted in the Header Section box of the Result Viewer. Click **Add Files...** to add one or multiple files to the selection. Clicking **Clear Files** empties the list completely and clicking **Sort Files Alphabetically** sorts the list in alphabetical order. The files can also be sorted manually by drag and drop in the list.

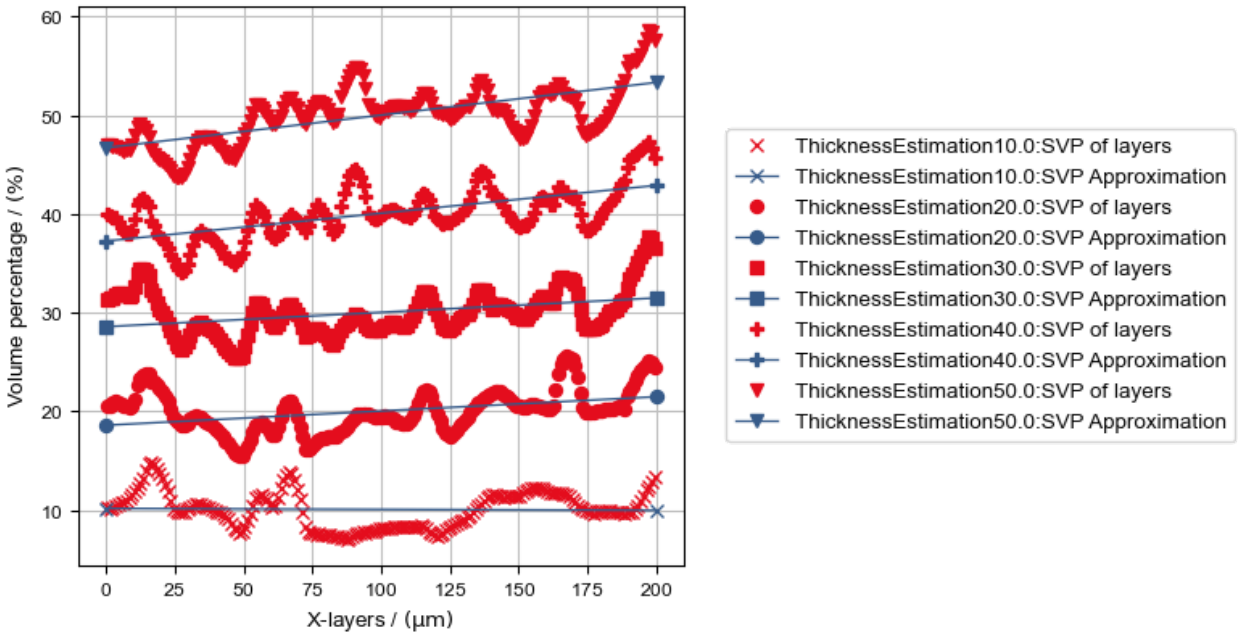
Know how! The order of files in this list mirrors the ordering of files (and plots) in the combined result file.

The two **Combine Modes**, which are **Color** and **Marker**, change the appearance of automatically combined plots.

With the **Color** mode, plot data originating from one result file share the plot color, while with the **Marker** mode, they share the plot markers.



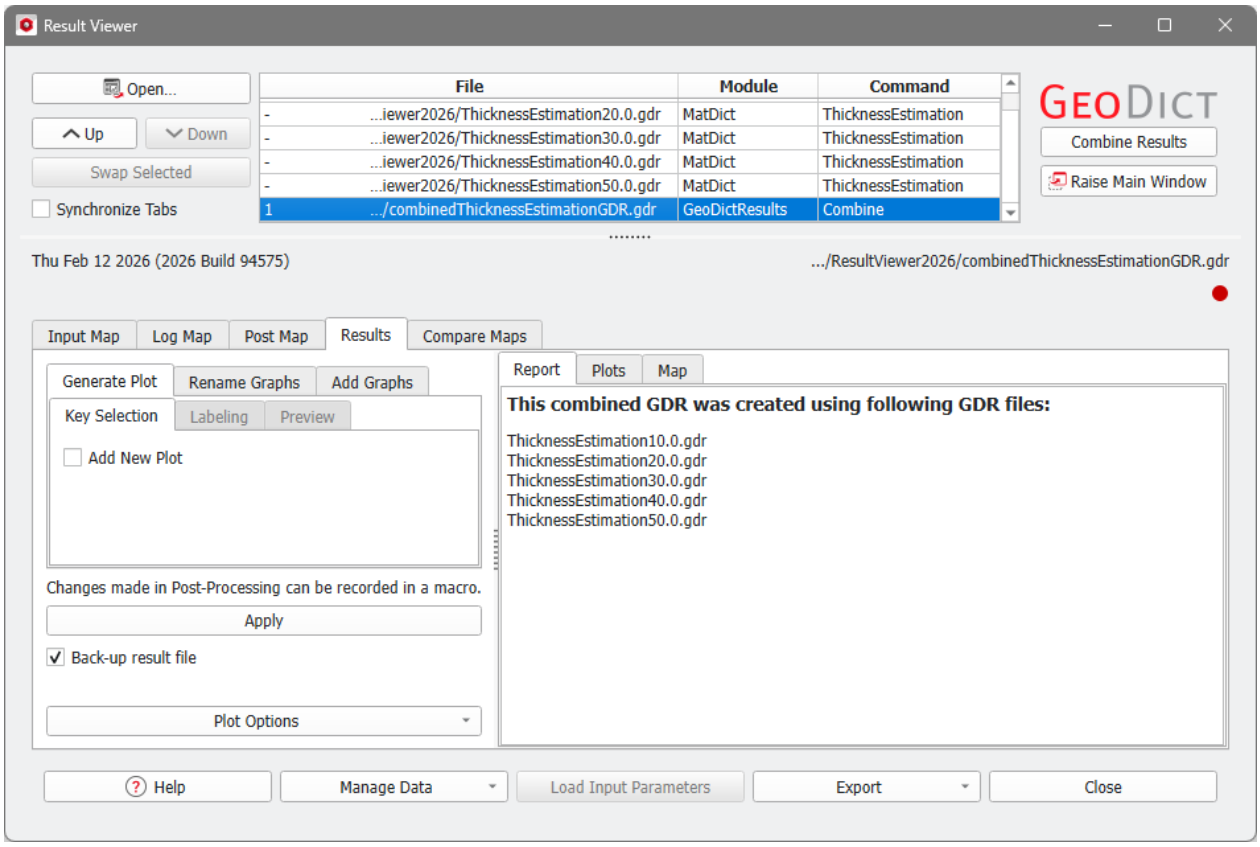
Color Mode



Marker Mode

Clicking **OK** starts the combining process. It combines all values into one file and enables post-processing on the combined results. If possible, plots from the original result files are merged into one plot.

The Result Viewer of the combined result file opens automatically and, like any other result file, it can also be opened at any time by selecting **File** → **Open Results (*.gdr)...** in the menu bar.

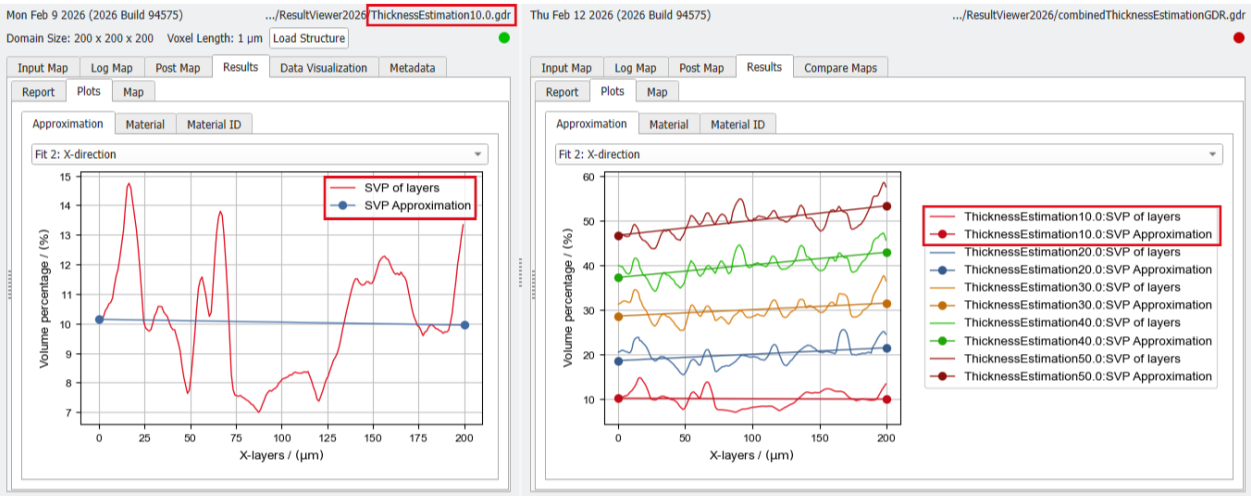


1.4.2 Automatically generated Plots

When creating combined result files, the plots of the individual result files are also included in the **Plots** tab of the combined result. If a plot is contained in more than one of the original result files, the data is merged into one plot. Depending on the **Combine Mode** 39 chosen during creation of the combined result file, the graphs of the merged plot have different colors or different markers.

The graph names shown in the legend fit to the result file names from which they originate. They are of the form **`name:suffix`**, where name is the origin result file name. If it is not located in the current project folder, the relative path to it is shown. The suffix is the graph title in the original result file.

In the example below, one of the original result file names was **ThicknessEstimation10.0** and one of the graphs had the name **SVP of layers**. Thus, the new graph name in the combined result file is **ThicknessEstimation10.0:SVP of layers**.

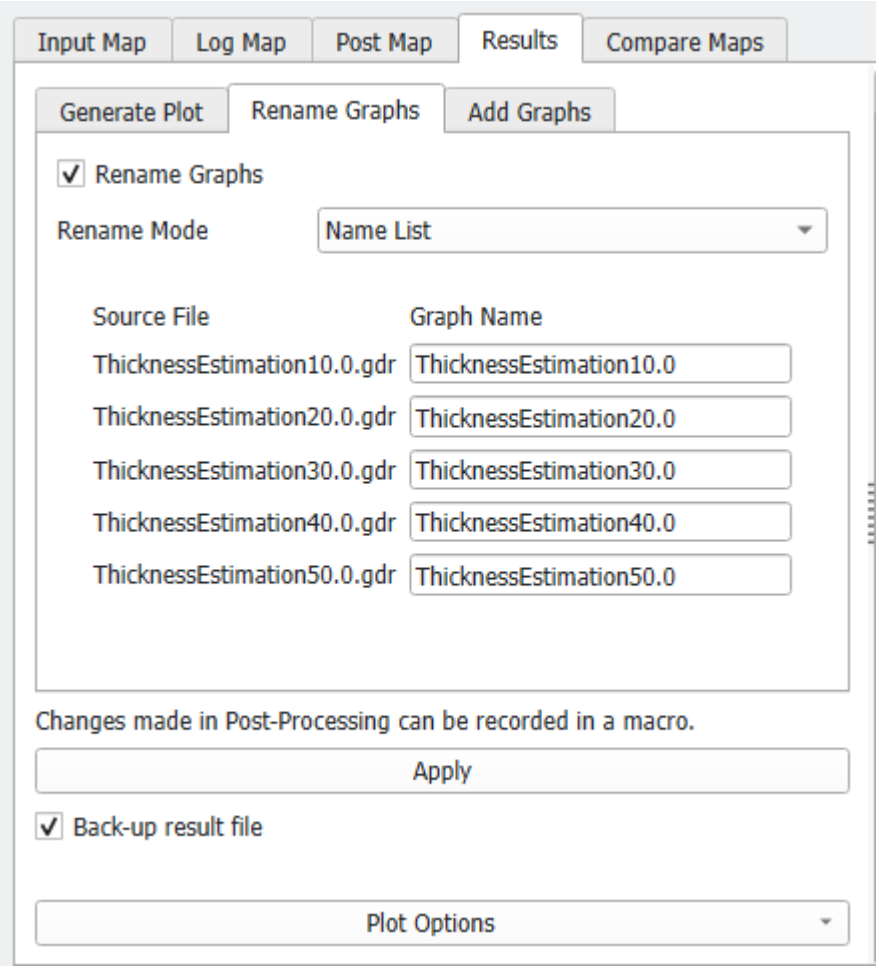


Rename graphs of automatically generated plots

Under the **Rename Graphs** subtab of the post processing panel, the labels of automatically created combined plots can be changed. Check the **Rename Graphs** box and select a **Rename Mode** for the graph name. The suffix will not be changed in any rename mode. Clicking the **Apply...** button renames all **automatically generated plots** on the result file according to the set renaming rules. **Custom generated plots** 46 are **not** renamed in any way.

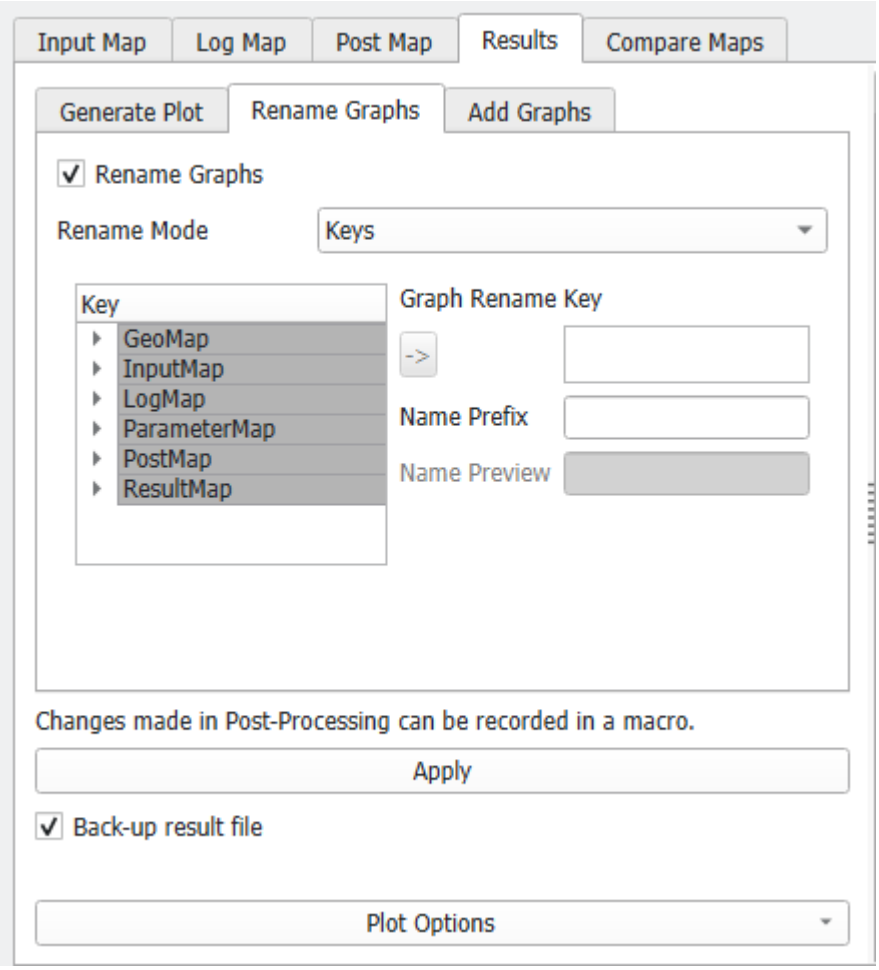
Name List

Choose **Name List** to define the name of the graphs for each original result file separately.



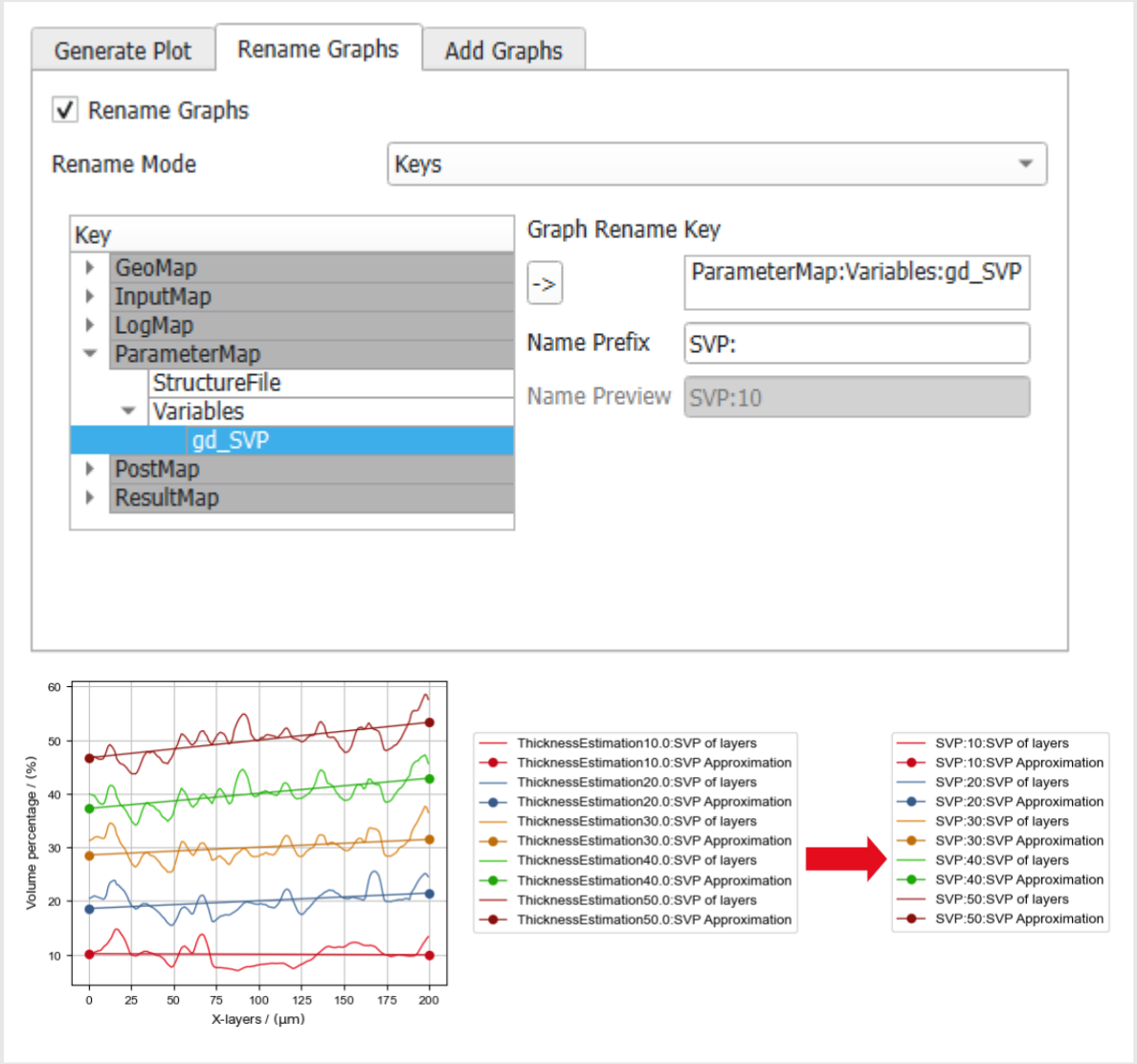
Keys

With the rename mode **Keys** the graph name is generated using a key from the maps occurring in all underlying result files. Select a key from the left and click on the -> icon to select it. The value of the key will then be taken as new name. You can additionally add a **Name Prefix** to the selected key value if it is necessary. The field **Name Preview** shows the final name according to the settings you made.



Example

As an example the graph names of the combined result file shown at the top are changed using the rename mode **Keys**. The new name will be the value of the key **gd_SVP** with the name prefix **SVP:**



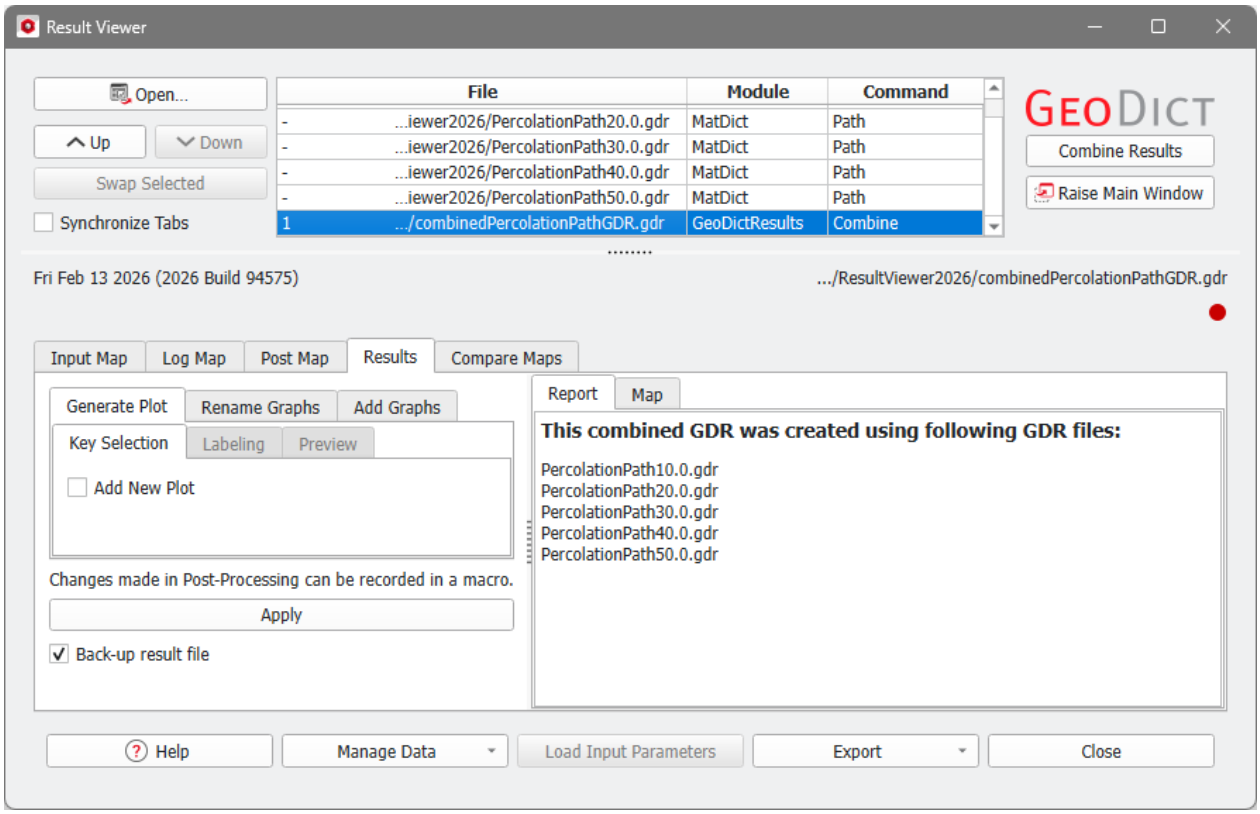
1.4.3 Custom generated Plots

After a combined result file has been generated, it opens automatically and it is possible to run post-processing to generate and edit custom plots from the result values.

In the example, fiber structures with varying solid volume percentage (10% - 50% SVP) were generated with a python macro and the largest Percolation Path and a Thickness Estimation were computed for each structure.

You can download the ParameterStudy.py file used in this chapter for inspection and recreation in a zipped folder.

Download ParameterStudy.zip



To create a custom plot select the **Generate Plot** tab in the post-processing panel on the left. In the **Key Selection** subtab, check **Add New Plot** to start customizing the plot.

Select the Keys for plotting

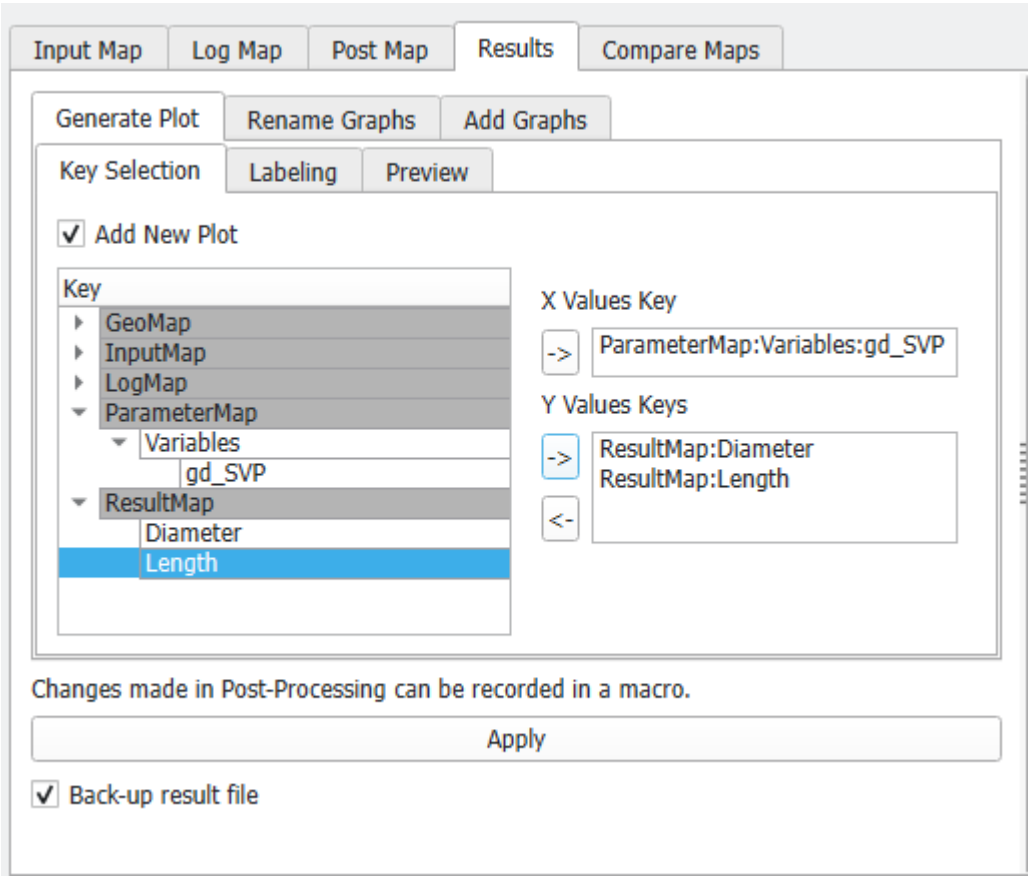
In the **Key Selection** subtab, a list (**Key**) contains the keys that can be used to generate the custom plot. These keys have values that differ amongst the original result files. They are also shown in **orange** or **red** color in the Compare Maps 55 tab.

Keys can be used to identify the data on the X-axis or the Y-axis. For the X-Axis only one key can be chosen, for the Y-axis multiple keys are allowed which means multiple graphs are contained in the final plot.

Keys are added in the following way:

- Click on one of the keys from the **Key** list to highlight it.
- Then, click on the -> icon in either the **X Values Key** or **Y Values Keys** section to add it to the corresponding axis.

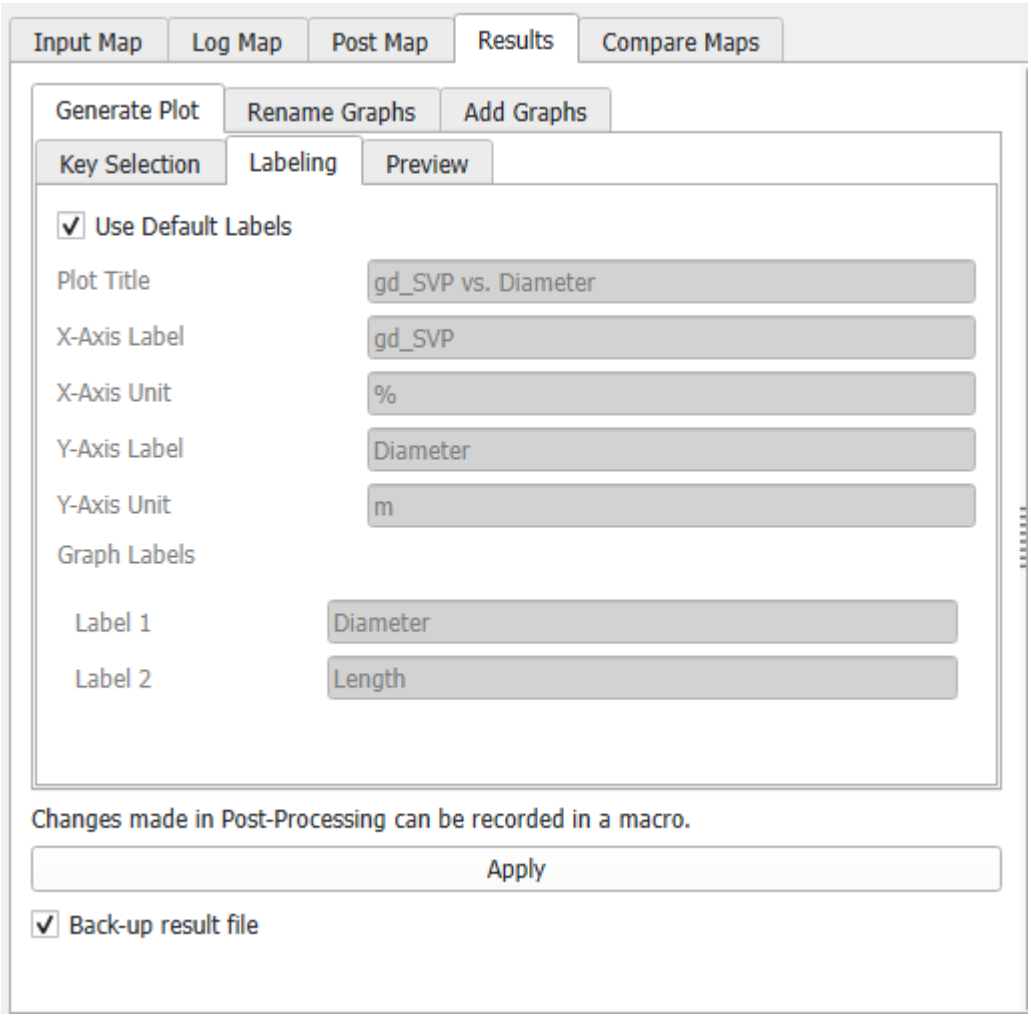
The X Values Key entry is overwritten when clicking the -> icon again. Single keys can be removed from the Y Values Keys again by selecting them and clicking on the <- icon.



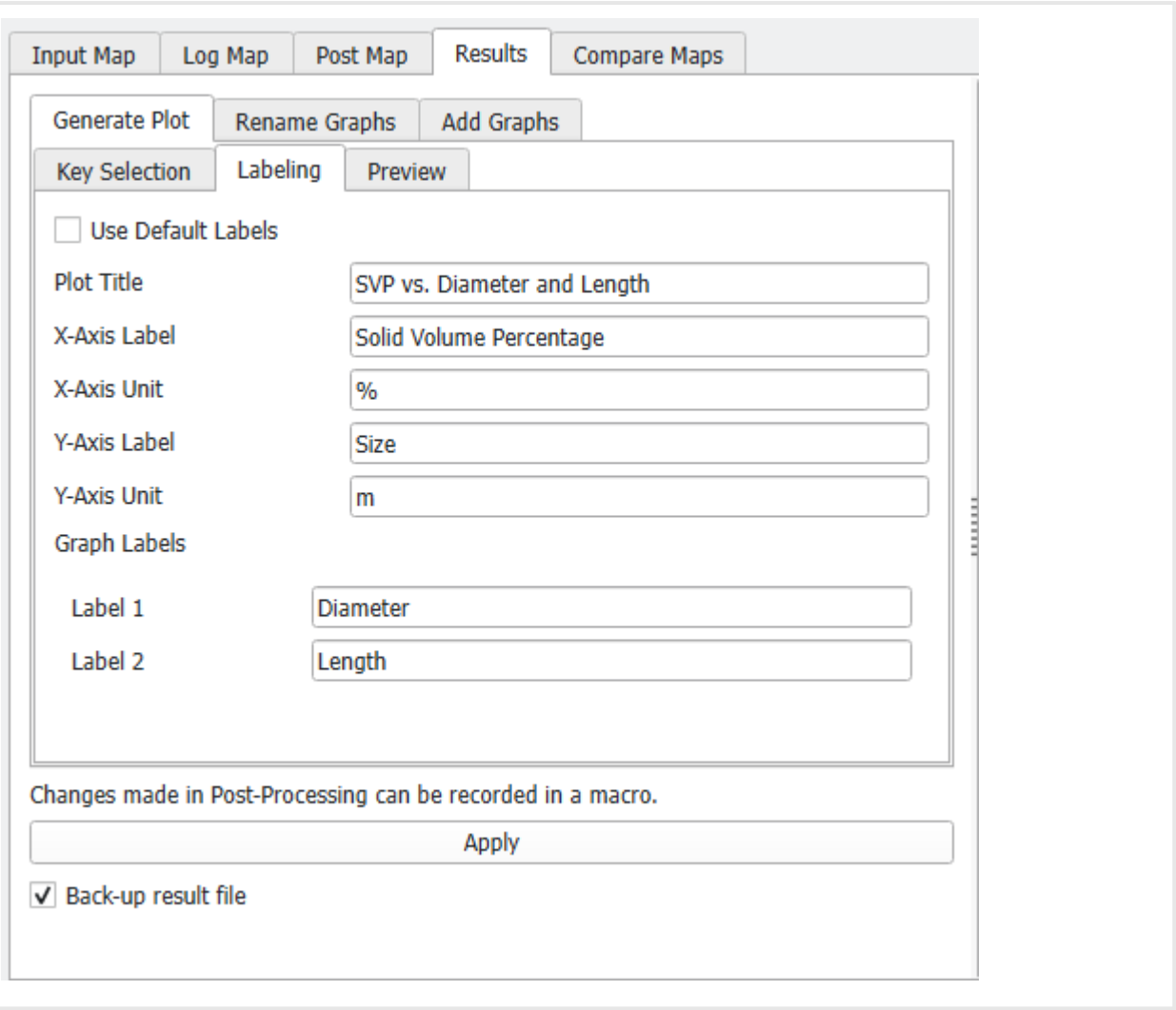
Know how! Variables that were used in a macro file can also be used as keys in the combined gdr result file. They are found under **ParameterMap - Variables**. In this example, the *.gdr result files for the Percolation Path were created by a GeoPy script with the variable gd_SVP, containing the Solid Volume Percentage (10%, 20%, 30%, 40%, and 50%).

Define the Labels of the plot

In the **Generate Plot - Labeling** subtab, the **Default Labels** for the plot are shown. For the axis labels and units, the key name and unit from the result file maps are taken. For the label and unit of the y-axis, the first of the selected keys to be plotted is used. The **Graph Labels** will be shown in the plot legend and correspond to the key names of the **Y Value Keys**.

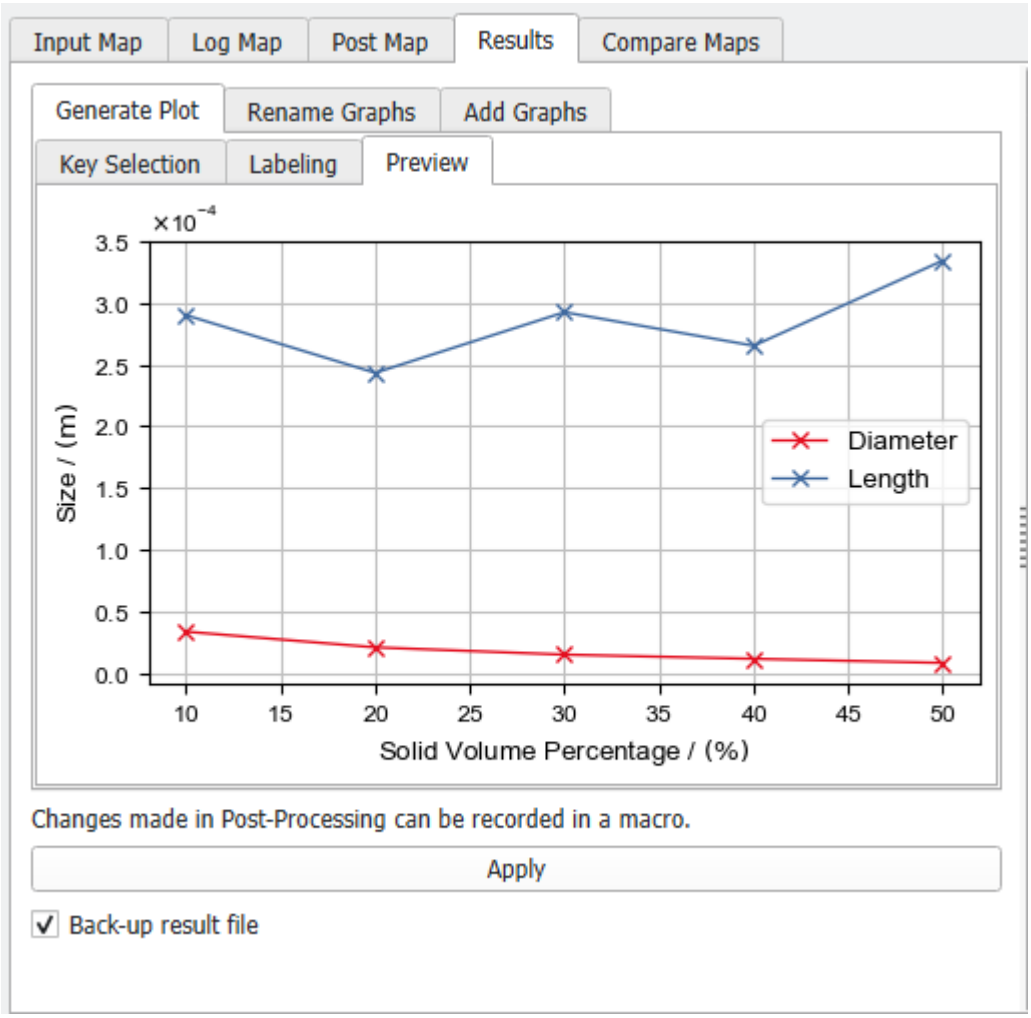


Uncheck **Use Default Labels** to define custom plot labels for axis labels and units and the graph labels.

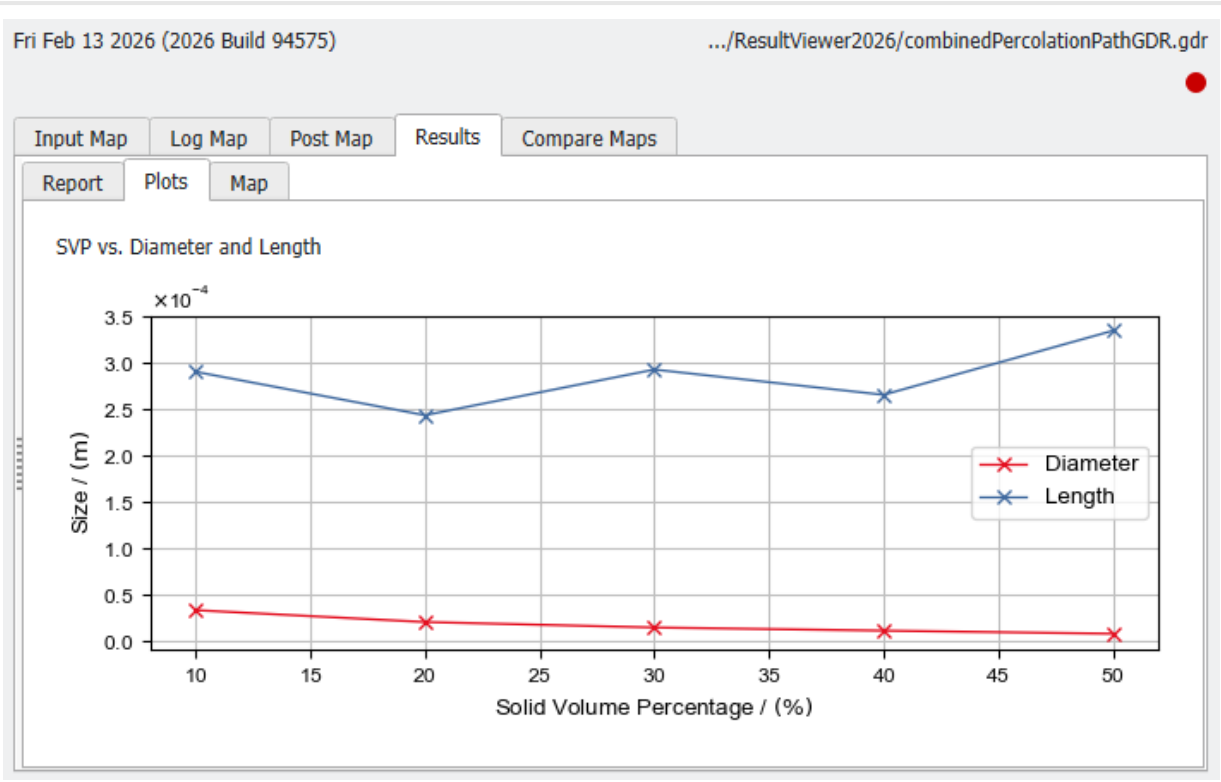


Preview and generation of the plot

Without clicking **Apply...** yet, go to the **Generate Plot - Preview** subtab, where you can control the appearance of your custom plot. Changing settings in the other tabs updates the preview plot immediately.



When clicking **Apply...**, the plot is permanently saved to the combined result file (here combinedPercolationPathGDR.gdr) and appears under the **Results - Plots** subtab in the Result Viewer afterwards.

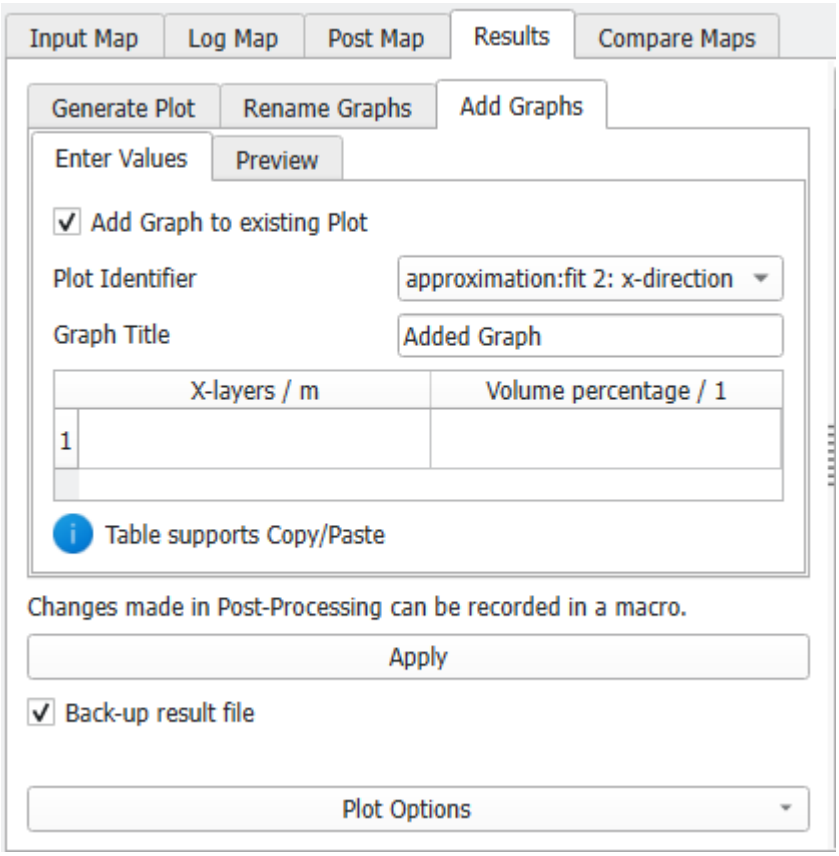


The generation of custom plots is saved in the GeoDict Session Macro and can be recorded as a python macro.

1.4.4 Adding Graphs

In the **Add Graphs** tab of the post processing panel new graphs can be edited to any plot - automatically generated or custom generated plot - in the combined result file. Check the box to activate the **Add Graph to existing Plot** options.

Select the **Plot Identifier** from the drop-down menu which defines the plot to which the graph is added. The name entered in **Graph Title** will be displayed in the plot legend.



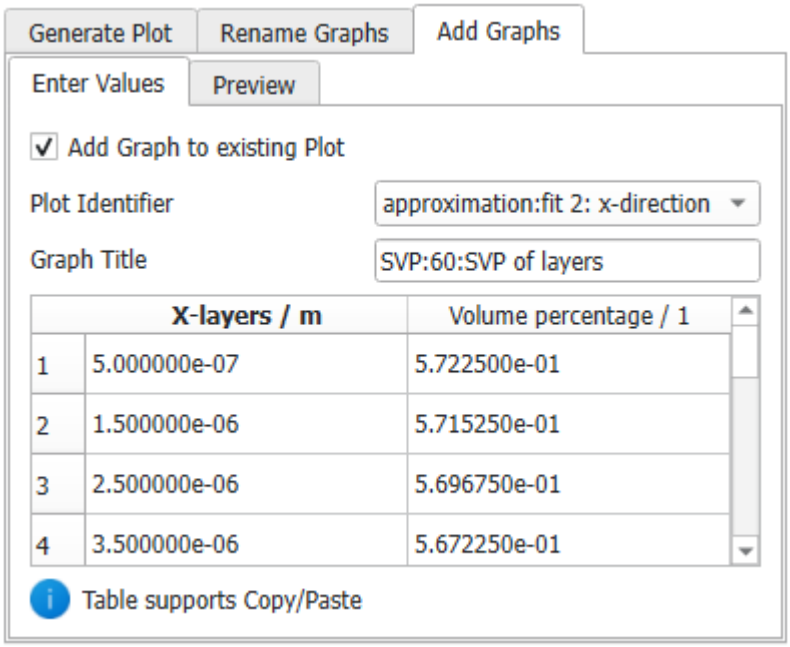
In the following table the new values to be plotted are entered. The columns of the table show the Axis Label and Axis Unit for the X-Axis in the left column and for the Y-Axis in the right column, respectively. This table supports copy and paste, e.g., if the new data is stored in another file.

In the **Preview** subtab, check if all values are correct, then click **Apply...** to save the new graph to the selected plot.

Example

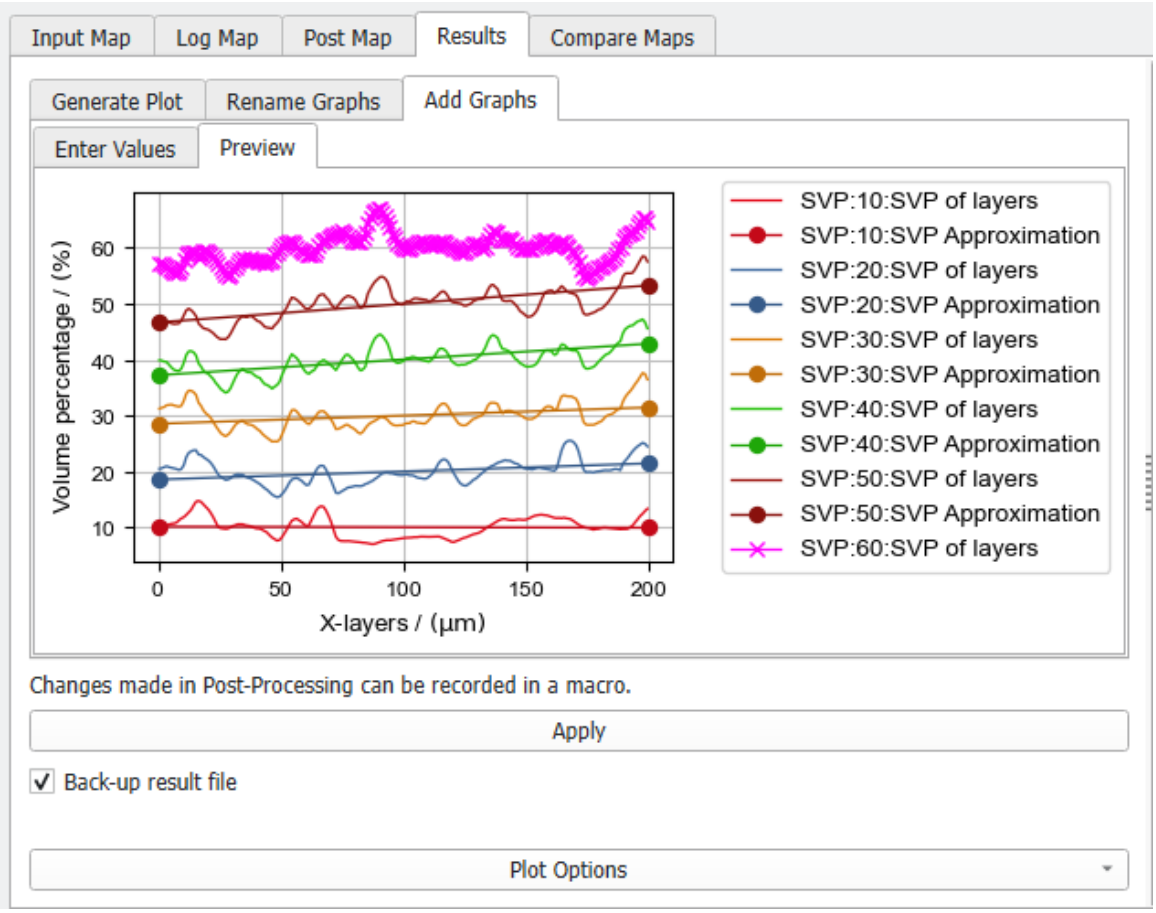
For the example in Automatically generated Plots 42, we calculated the Thickness Estimation again for a structure with 60% solid volume percentage. We now include this graph into the graph from the combined result file. First, in the result file of the thickness estimation with 60% SVP, right click on the corresponding plot (Fit 2: X-direction) and choose **Save Graph(s) as Text File(s)** 25. Change the data type to **Table Files (*.csv)** and choose a meaningful name. The value pairs plotted on the X- and Y-axis are saved for each graph in a separate file. The files get a numbered suffix in the order of the legend entries.

Open the csv files and copy the data. Then it can be pasted into the table in the **Add Graphs** – **Enter Values** subtab. Adapt the Graph title to the graph titles of the plot to which the graph is added.

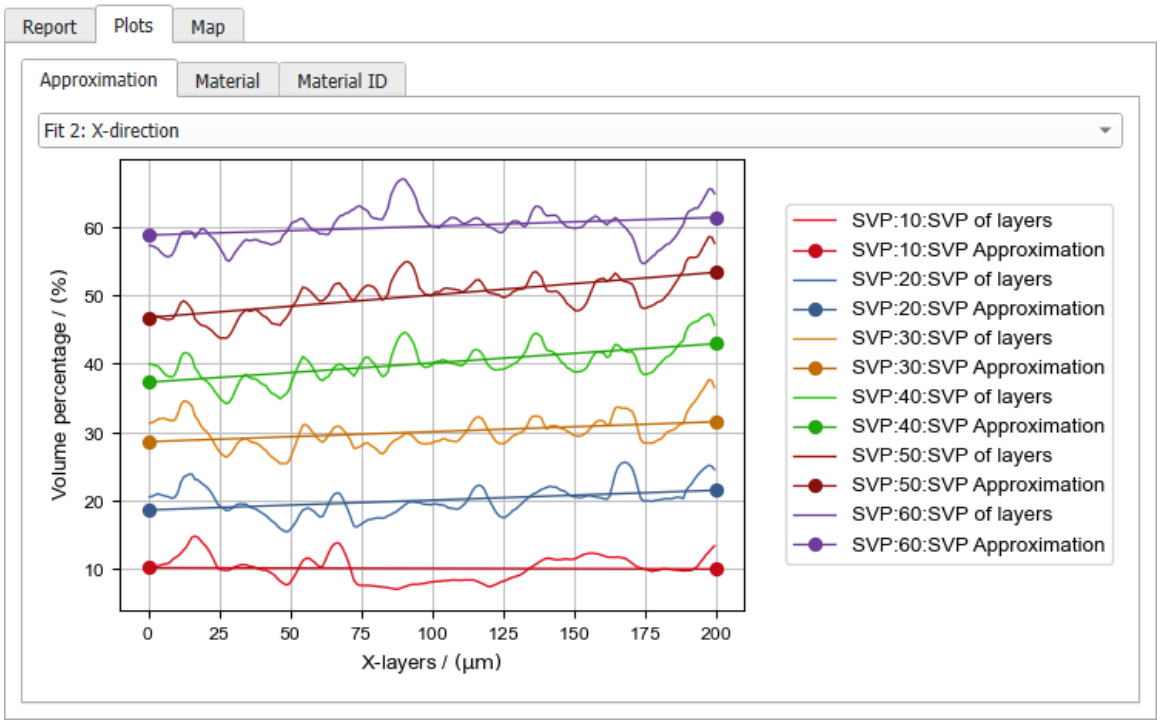


In the **Preview** subtab, check if all values are correct, then click **Apply...** to add the first graph (here SVP of layers).

In the same way, add the SVP approximation graph.

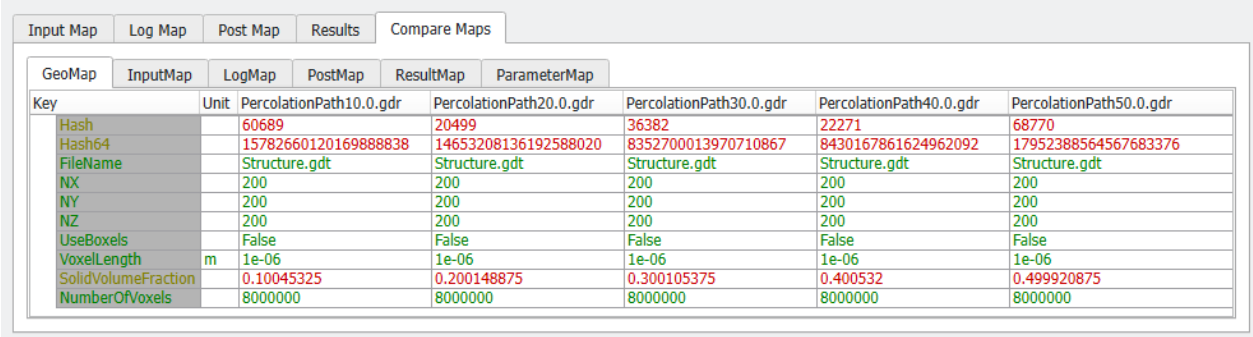


The colors and styles can be changed later under **Edit Graph Styles** 31.



1.4.5 Compare Maps

Combined result files can be compared by clicking the **Compare Maps** tab. Entries in various colors are seen in the tabs **GeoMap**, **InputMap**, **LogMap**, **PostMap**, **ResultMap**, and **ParameterMap** under the **Compare Maps** tab.



Key	Unit	PercolationPath10.0.gdr	PercolationPath20.0.gdr	PercolationPath30.0.gdr	PercolationPath40.0.gdr	PercolationPath50.0.gdr
Hash		60689	20499	36382	22271	68770
Hash64		15782660120169888838	14653208136192588020	8352700013970710867	8430167861624962092	17952388564567683376
FileName		Structure.gdt	Structure.gdt	Structure.gdt	Structure.gdt	Structure.gdt
NX		200	200	200	200	200
NY		200	200	200	200	200
NZ		200	200	200	200	200
UseBoxels		False	False	False	False	False
VoxelLength	m	1e-06	1e-06	1e-06	1e-06	1e-06
SolidVolumeFraction		0.10045325	0.200148875	0.300105375	0.400532	0.499920875
NumberOfVoxels		8000000	8000000	8000000	8000000	8000000

Differences and similarities among the result files are quickly spotted through the colors of these entries. The same color scheme applies for the maps of two or more result files if they are selected simultaneously 10 and shown side by side.

Colors for Keys

- The key is **green** if it is the same in all result files.
- The key is **orange** if it is contained in all result files, but the values differ. This holds also if there is a difference in a sub key.

- The key is **black** if it is not contained in all result files.

Colors for Values and Units

Colors for values and units are similar to colors of keys.

- The value is **green** if it is the same in all result files.
- The value is **red** if it differs in at least one of the result files.
- The value is **black** if it does not appear in all result files

- N -

name:suffix 42

Citation: Math2Market GmbH, 2026: GeoDict 2026 User Guide, Result Viewer, Math2Market GmbH, Germany, doi.org/10.30423/userguide.geodict

MATH 2 MARKET

Math2Market GmbH
Richard-Wagner-Str. 1, 67655 Kaiserslautern / Germany
www.math2market.com