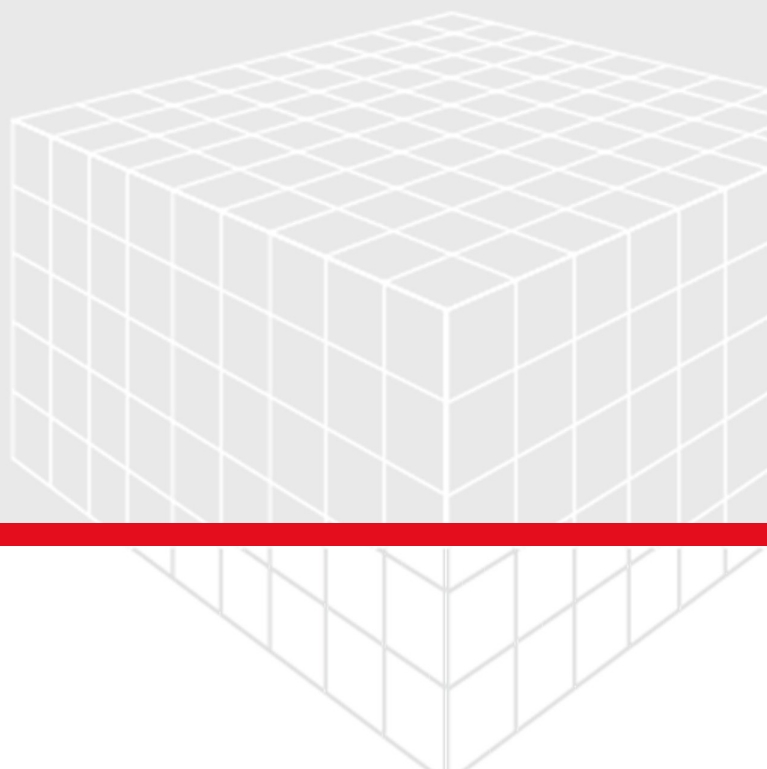


Visualization

User Guide

GEODICT Release 2026

Published: June 2026



GEODICT

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

Visualization

1. Visualization	3
1.1 Visualization Elements	5
1.1.1 View Menu	5
1.1.2 Colors & Visibility	7
1.1.3 View Controls	14
1.1.4 Visualization Area	15
1.1.5 Visualization Tab	19
1.2 Visualization Options	23
1.2.1 Visualization 2D	25
1.2.2 Visualization 3D	42
1.2.3 Legends and Overlays	109
1.2.4 Color Map Dialog	121
1.3 References	131

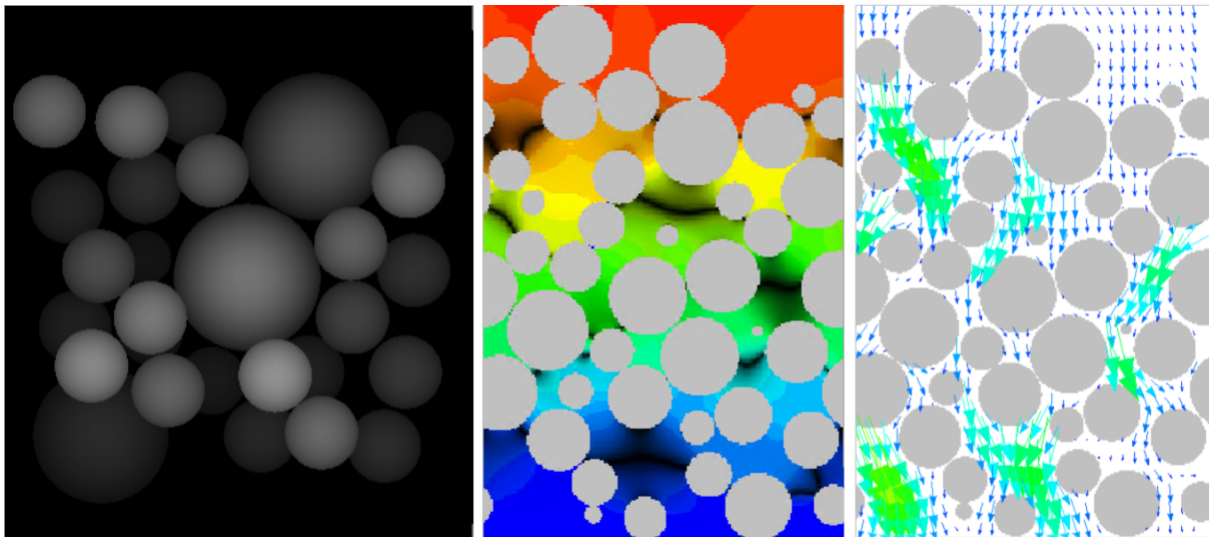
1 Visualization

Visualize your data to show off your materials and simulation results in 2D Cross Section view and 3D view.

For visualization, GeoDict distinguishes two concepts: structure (or geometry-related data) and result fields (or property-related data).

- **Structures** can be created as output data from GeoDict's generation modules (ending in Geo) or imported with GeoDict's import modules (beginning with Import). They are input data for GeoDict's material properties prediction modules (ending in Dict).
- **Result fields** are the output data of GeoDict's material property prediction modules. Using result fields GeoDict provides great visualization options for the simulation results.

Below have a look at some of the various visualization options for 2D Cross Section view:

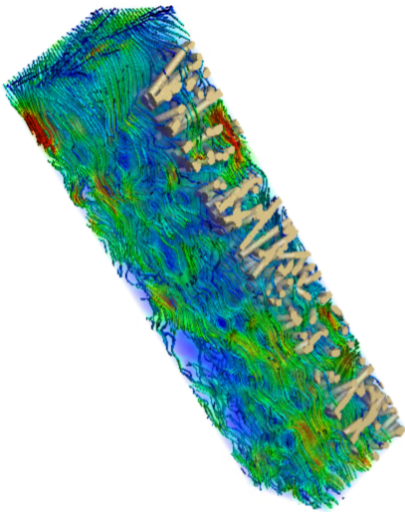


Depth Visualization

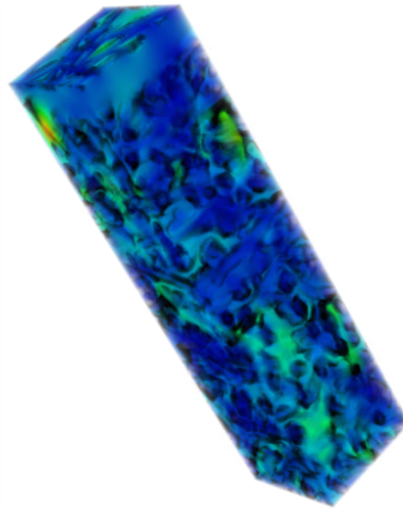
Result Field Visualization:
Volume Field with Lines

Result Field Visualization:
Arrows

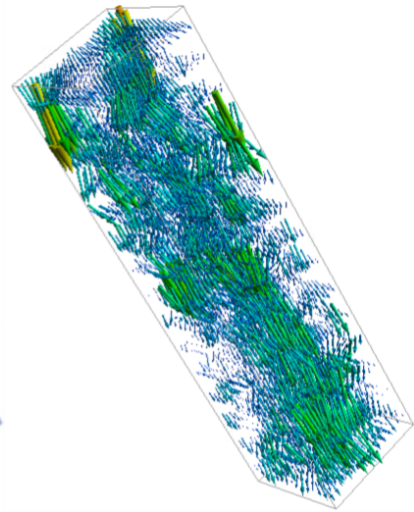
Also in 3D view many options are available, for example:



Visualization of Volume Field and Streamlines



Visualization of Volume Field with GPU Renderer Options



Visualization of Arrows

Learn to visualize your GeoDict results:

- [Visualization Elements](#)⁵
Find out which visualization elements there are in GeoDict.
- [Visualization Options](#)²³
The different visualization options described in detail.
- [References](#)¹³¹
The references used for this topic.



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](#).

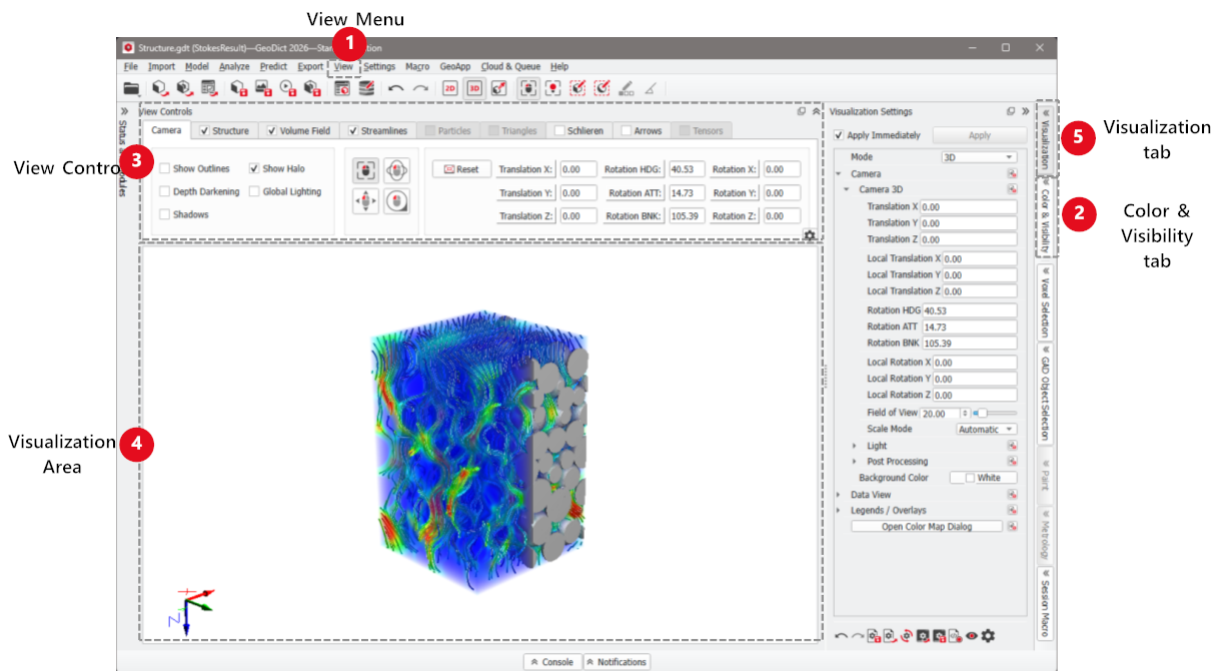
Send your [Support Request](#) to our technical Support.

1.1 Visualization Elements

For in-depth analysis of simulation results, images of geometries and images of properties can be displayed in 2D cross-section (SEM) view and in 3D rendering.

Different features are available for visualization in 2D and in 3D. A great number of attributes and features may be toggled and adjusted to complement and enhance the images' visual information.

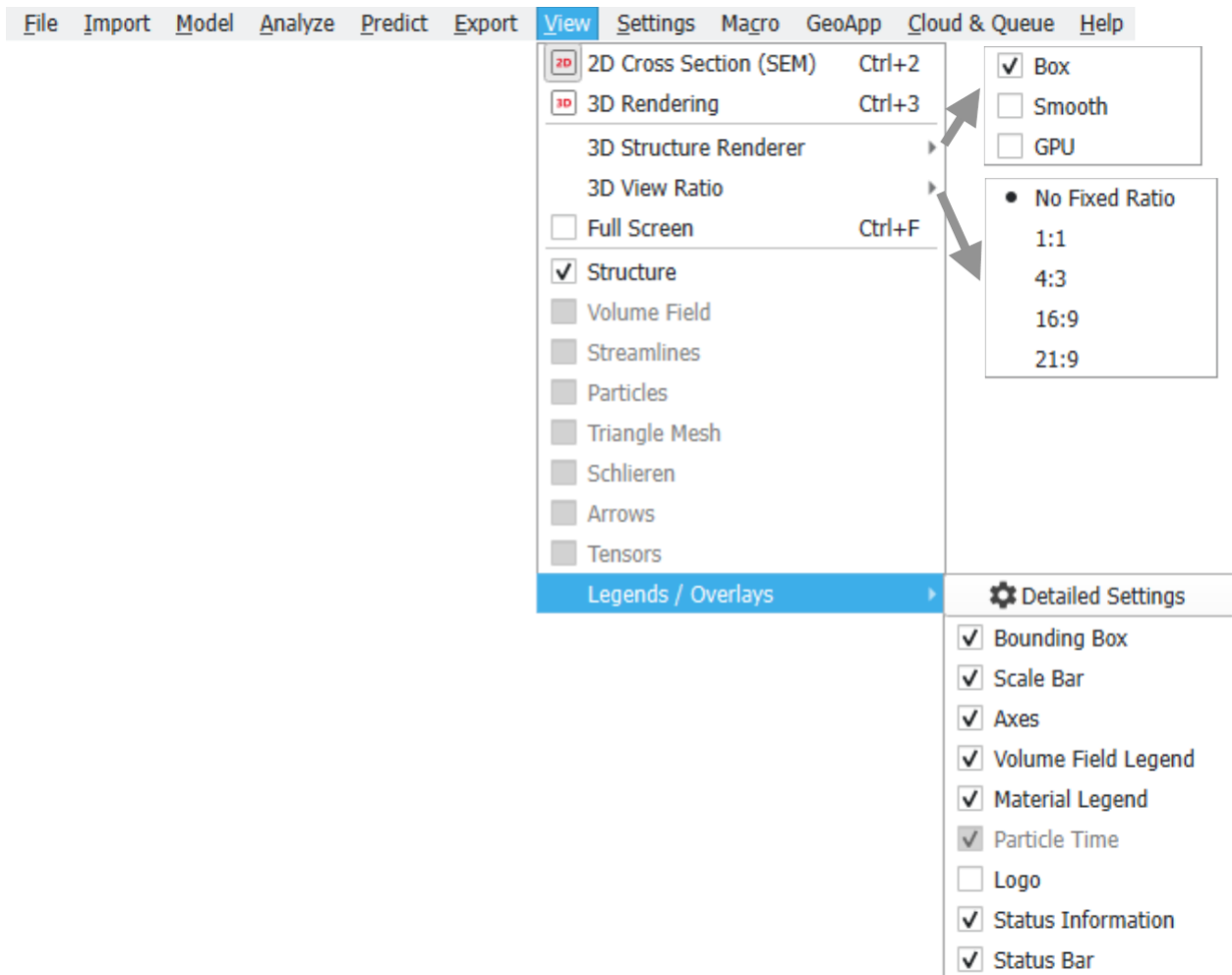
The activation of these GeoDict visualization features is governed directly through five points of access in the **Graphical User Interface** (GUI):



1. [View Menu](#) [5] in the menu bar, which gives access to some general options.
2. [Color & Visibility](#) [7] tab in the GUI Sidebar, which is expandable and may be undocked.
3. [View Controls](#) [14] above the visualization area, grayed out if no structure is loaded in memory.
4. [Visualization Area](#) [15] to view the visualization and move the structure around.
5. [Visualization Tab](#) [19] in the GUI Sidebar, which is expandable and may be undocked.

1.1.1 View Menu

After modeling a structure, either by generating it in GeoDict or after importing 3D μ CT or FIB/SEM images, define how to view your material in the **View** menu by changing the visualization settings.



Set visualization type

Choose the visualization type to be **2D Cross Section (SEM)** or **3D Rendering**, which has additional settings under **3D Structure Renderer** and the **3D View Ratio**.

In 3D Rendering, the **Structure Renderer** can be chosen. The structure can be rendered using the graphics card (**GPU**) or the main processor (**Box** with raycasting or **Smooth** with raycasting). Volume raycasting is an image-based volume rendering technique that computes 2D images from 3D volumetric data sets. Examples to these options can be found [here](#)⁵³.

Selecting **Full Screen** makes the Visualization area occupy the entire screen. Pressing the **Esc** key on the keyboard returns the program to the normal display containing all the components of the GeoDict layout.

Show / hide components

Select which components are displayed in the visualization. The options available depend on the currently loaded file types, e.g., if no Volume Field is loaded, the Volume Field option is grayed-out.

When unchecking one of the components, it disappears from the display. Some components can only be visualized in **3D Rendering** so the effects are only visible when 3D Rendering is chosen.

Among the choices for the visualization are **Structure**, **Volume Field**, **Streamlines**, **Particles**, **Triangle Mesh**, **Schlieren** (Line Integral Convolution), **Arrows**, and **Tensors**.

Legends and Overlays

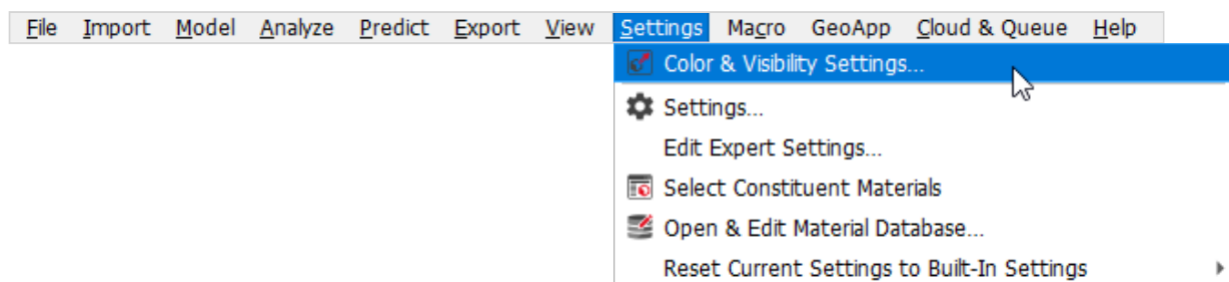
Hover over **Legends / Overlays** to choose which legends or additional information is displayed in the visualization area. Select **Legend / Overlays** → **Detailed Settings** to access the corresponding settings in the **Visualization** tab of the sidebar. All settings are explained [here](#).

1.1.2 Colors & Visibility

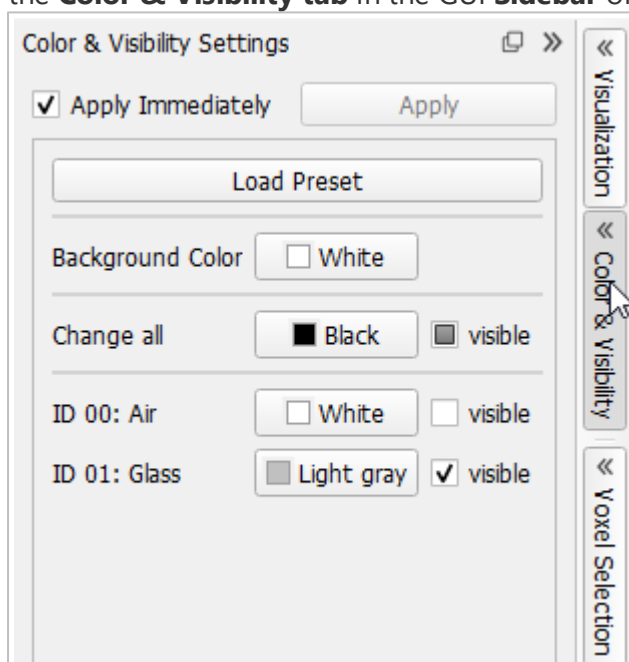
Define the color and visibility of your materials.

The **Color & Visibility Settings** are available from three different places:

- the **Settings** menu in the menu bar.



- the **Color & Visibility tab** in the GUI **Sidebar** on the right side,

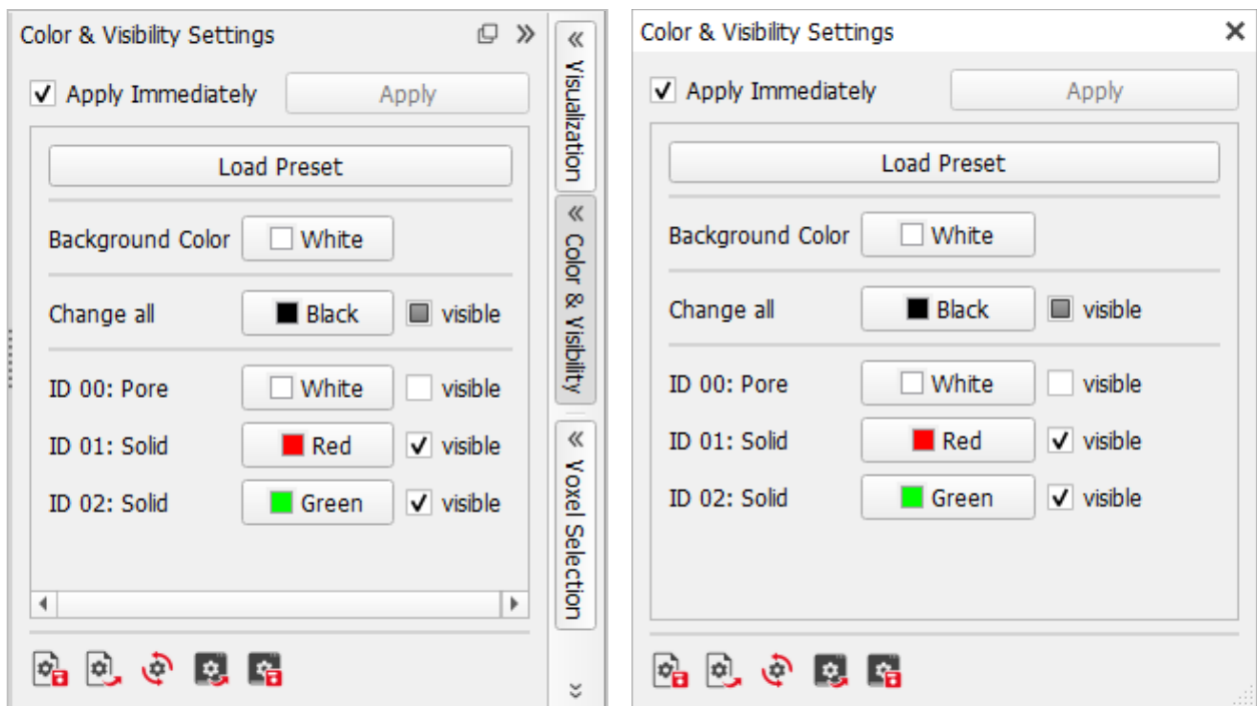


- or clicking the  icon in the toolbar.

In all three cases, the **Color & Visibility Settings** open as a panel that can be also undocked from the GUI by clicking  in the upper right corner.

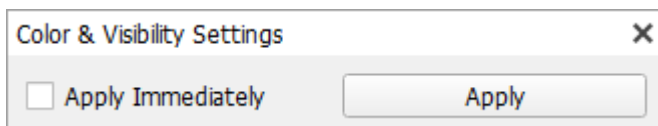
The **Color & Visibility Settings** can be collapsed either by clicking  in the upper right corner or by re-clicking the **Color & Visibility** tab in the GUI **Sidebar**.

The undocked **Color & Visibility** dialog can be closed with  in the upper right corner.



The default color settings, applied to the background and materials during the generation with GeoDict, can be changed through the **Color & Visibility Settings** dialog. While up to 256 material IDs can be present in a structure, only the colors for the IDs contained in the current structure can be edited.

All settings changed in the **Color & Visibility Settings** dialog are applied directly if **Apply Immediately** is checked. When the checkbox is disabled, the **Apply** button next to it is enabled to apply the changes only when desired.



Know how! The icons at the bottom of the dialog can be used to:

-  Saves the settings to a GeoDict settings file (*.gps).
-  Loads the settings from a GeoDict settings file (*.gps) or a GeoDict result file

(*gdr).

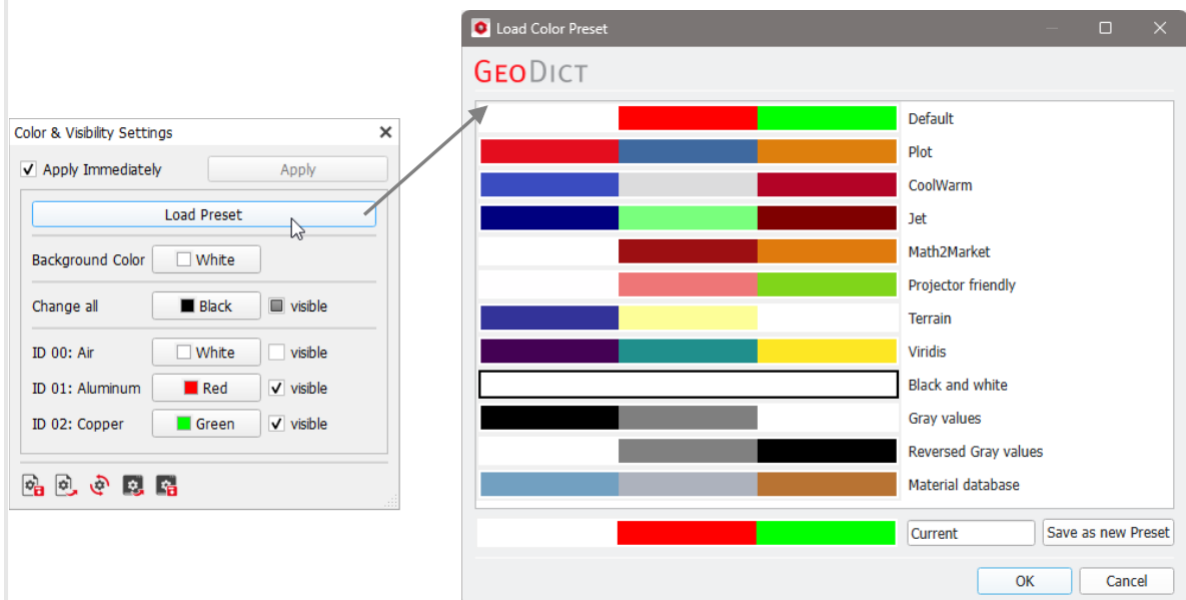
-  Resets all parameter settings to the built-in default settings.
-  Resets all parameter settings to the saved start-up settings.
-  Saves the current parameter settings as start-up settings.

Each option is explained by a tooltip which appears when resting the mouse over the icon.

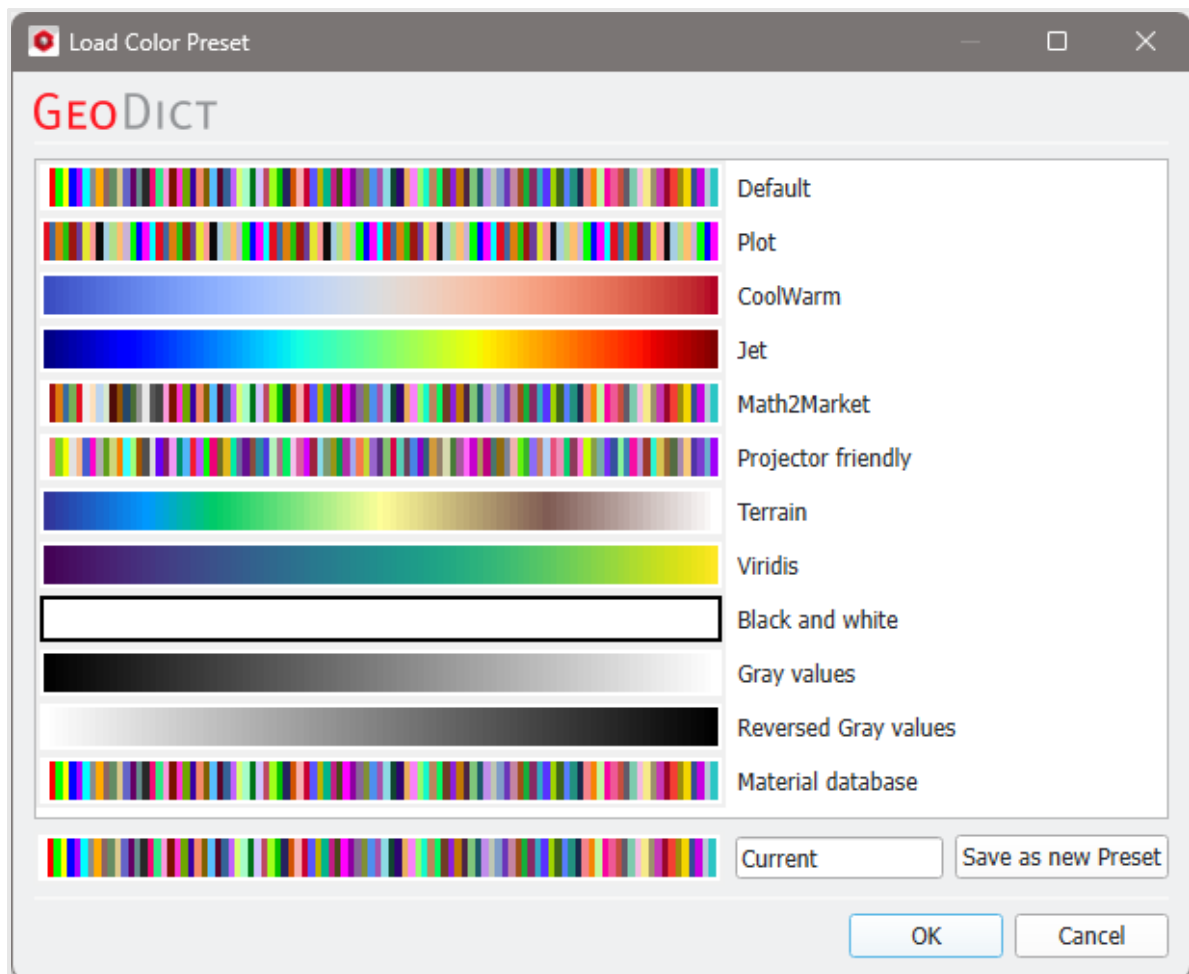
✓ Color Presets

A color preset is a group of colors that are applied together.

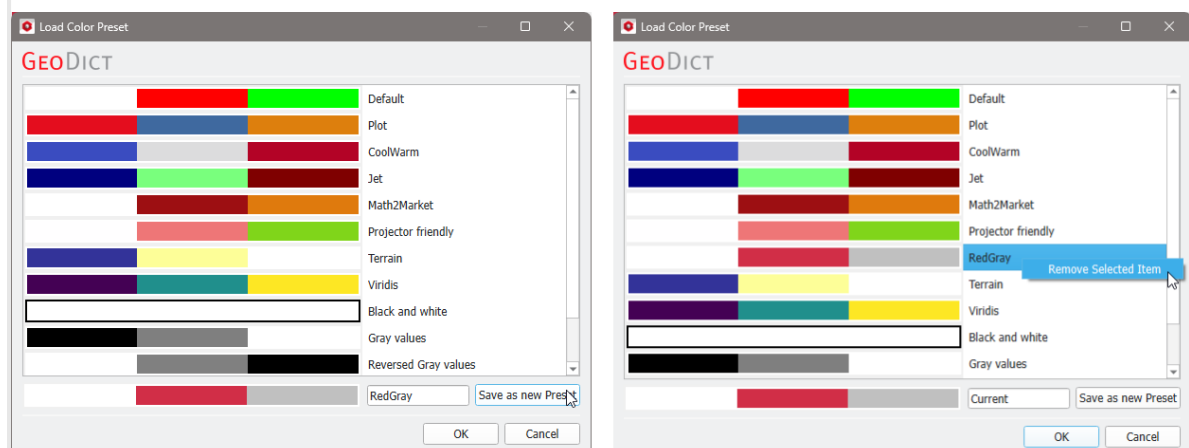
Click **Load Preset** and, in the **Load Color Preset** dialog, select whether to display the generated structure's material using one of the presets.



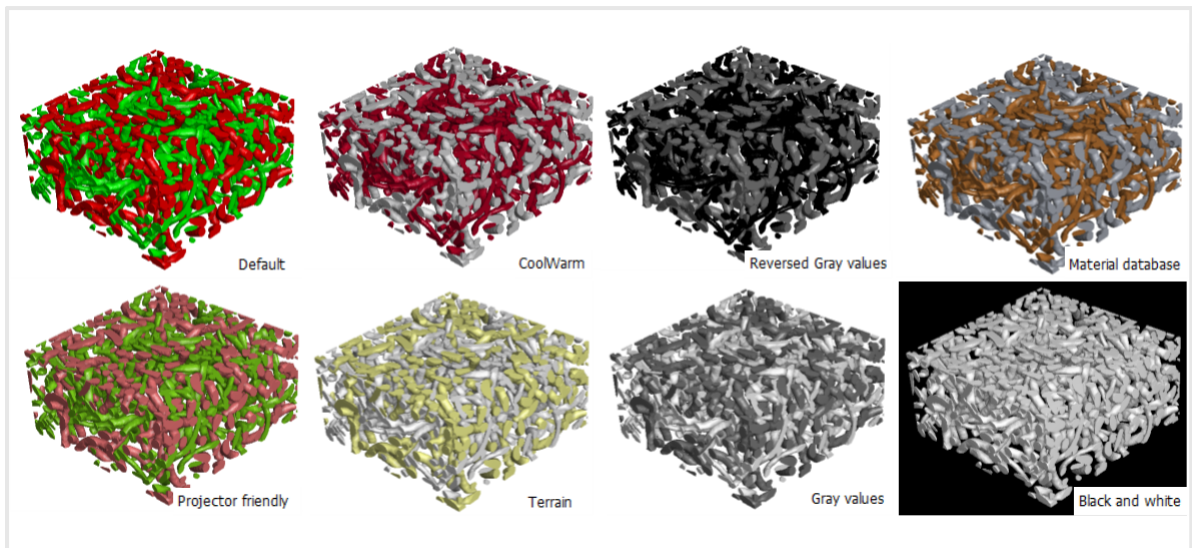
The number of colors shown in the preset previews depends on the number of materials present in the current structure. If for example 100 IDs are present, the dialog looks as follows:



You may select new colors through the buttons in the **Color & Visibility Settings** dialog and save the current group of colors as a preset (**Save as new Preset**), to have it appear in the list of color presets later. The preset can be removed anytime by right-clicking on it and selecting **Remove Selected Item**.

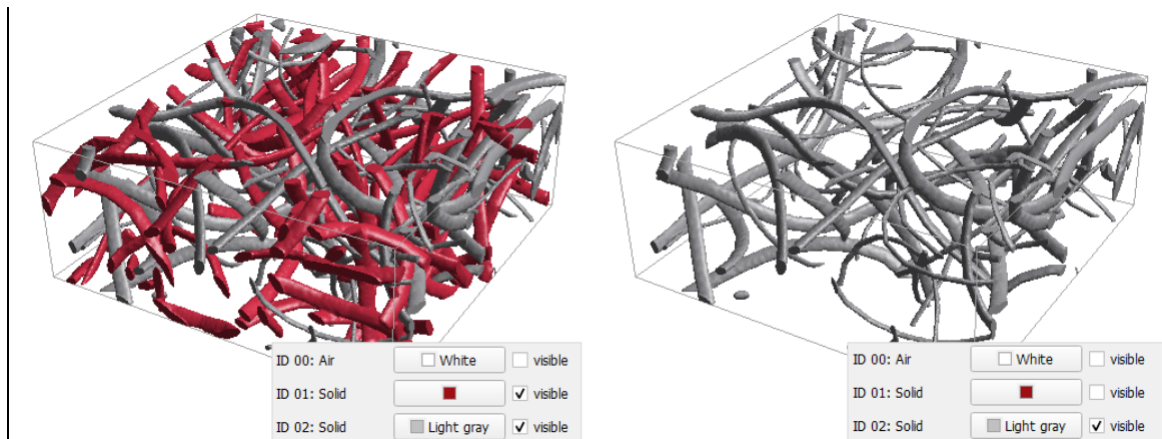


In the following, find some of the color presets applied to our example structure:



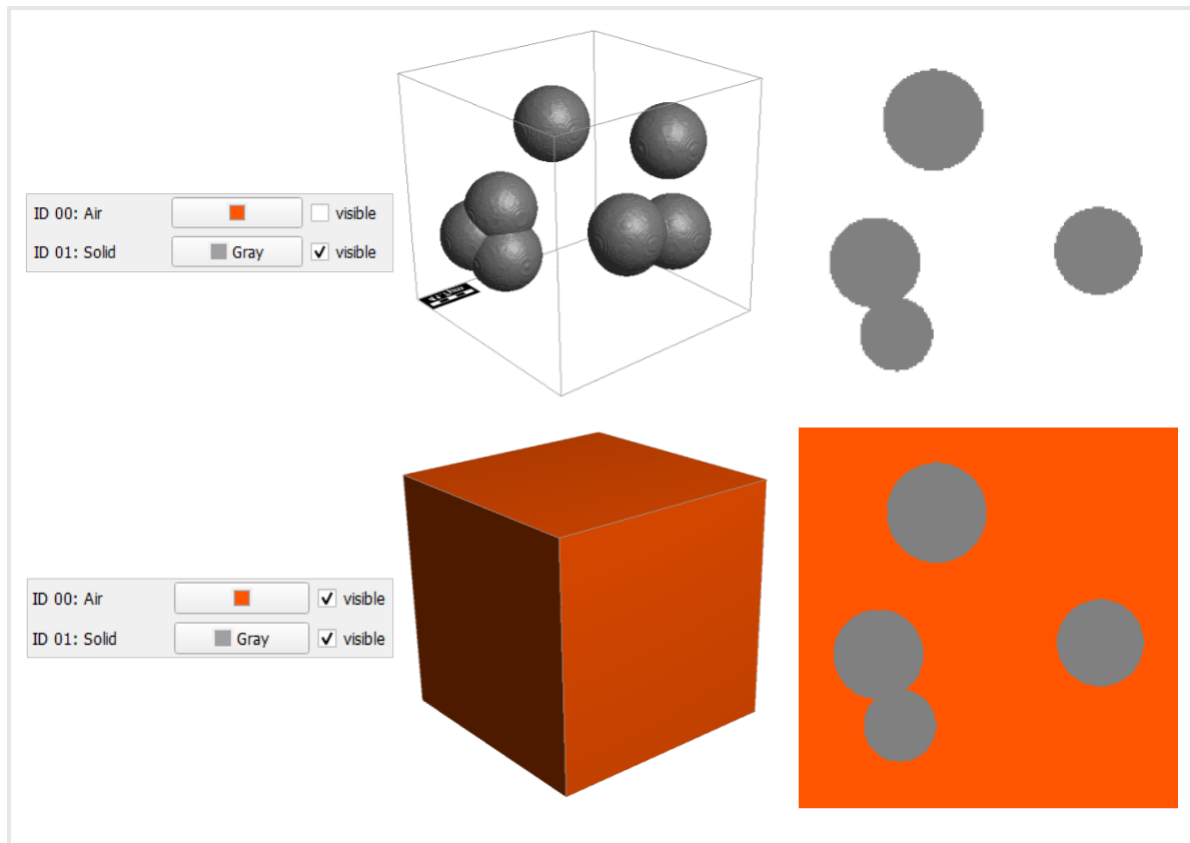
✓ Visibility of Materials

The visibility of the materials in the structure, for example of Material **ID 01**, is controlled by checking or unchecking ☐ **visible**. In the example below, material **ID 01** is still present in the structure, but it is just not visible.



By default, the Material **ID 00** is the background material surrounding the objects, inside the domain indicated by the bounding box. In the default settings, the visibility of Material ID 00 is unchecked, to be able to see the objects in the structure in 3D rendering, and the color is white.

The color and visibility of Material ID 00 can be changed. In this example, this has the effect of showing a block in 3D rendering that corresponds to the domain, in the color selected for Material ID 00 (here, orange).



✓ Change Colors

To change the color selected for a material from the default (e.g., Dark gray) to another color (e.g., yellow), click the button for the color (here: dark gray) and select the yellow color from the table.

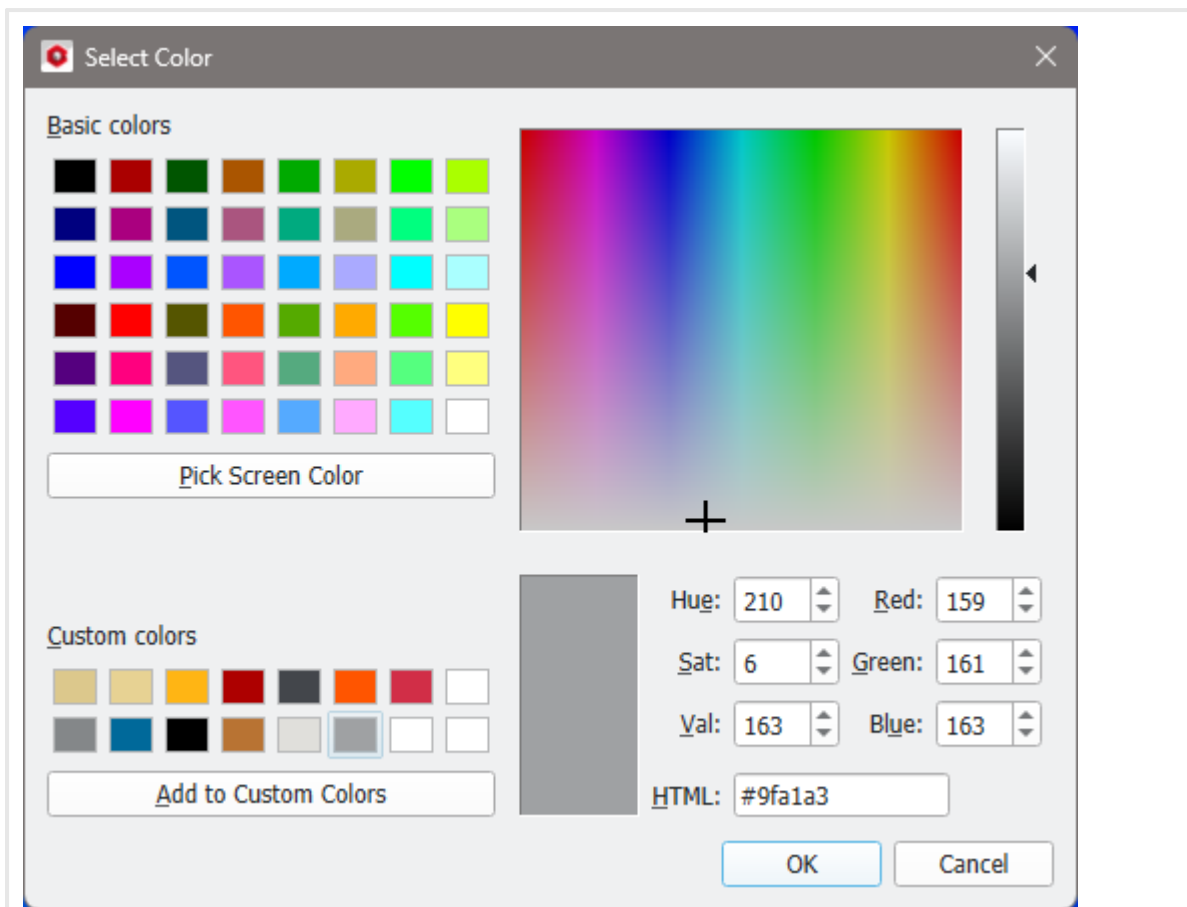


To change the default color to a user-defined color, select the three-dots icon.

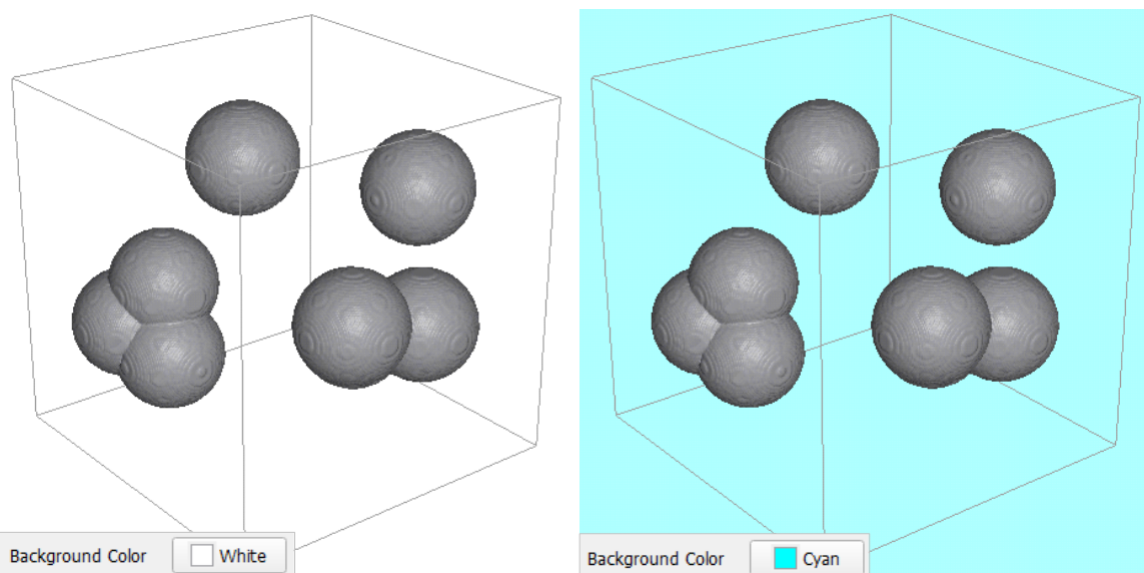


In the opening **Select Color** dialog enter the RGB values that define the color and click **Add to Custom Colors**. Alternatively, the rectangle for the new color can be dragged to occupy the place of the old one in the custom colors.

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

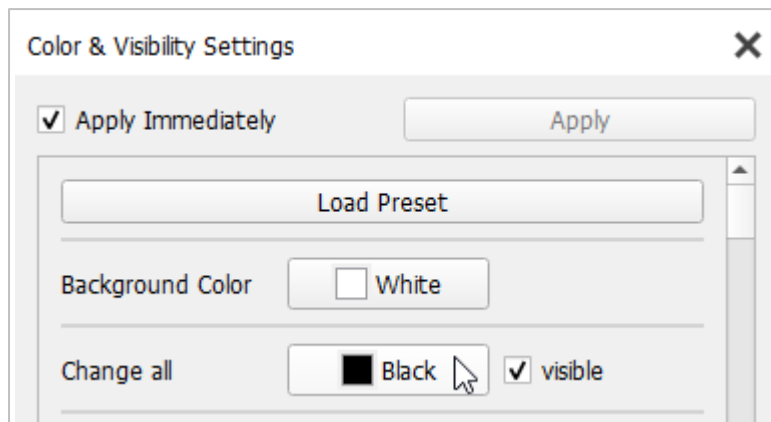


The **Background Color** of the Visualization area, at the top of the Color & Visibility Settings dialog, can be changed in the same way.



The color and the visibility can be changed for all materials at once in the **Change all** line below the **Background Color** line. As soon another color is selected, the color for all material IDs is set to the chosen color. If unchecking (checking) the **visible** checkbox,

all material IDs are set to invisible (visible).



1.1.3 View Controls

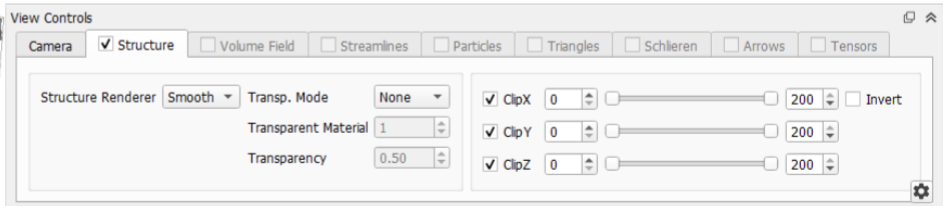
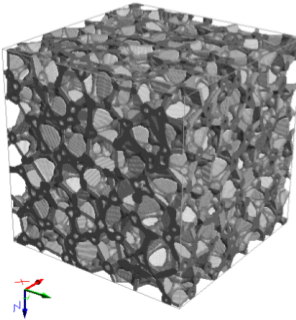
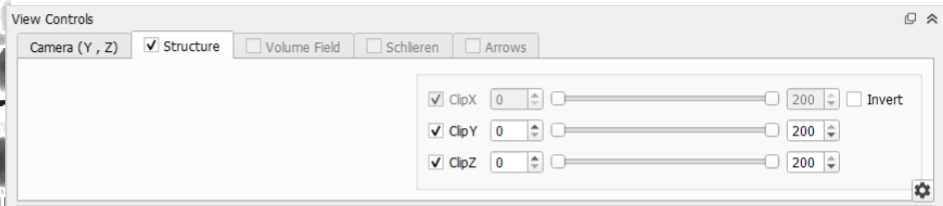
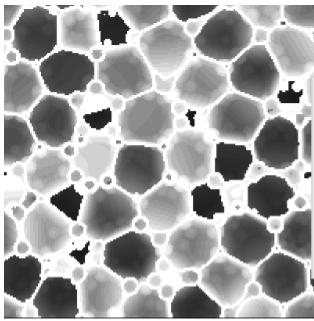
Use the view controls to define the look of your data in the visualization area.

At the program start, the **Visualization area** shows the GeoDict splash screen. When structures or result fields are loaded in GeoDict, they are shown in the **Visualization area** and the **View Controls** panel appears above it.

The **View Controls** panel shows information and options for the visualization of the image displayed in the **Visualization area**. The items shown in the **View Controls** and the image display in the Visualization area are closely related. They are influenced and change accordingly with the parameters selected in the [View](#) ^[5] menu and in the [Visualization](#) ^[19] tab and the [Color & Visibility](#) ^[7] tab of the GUI Sidebar..

The **View Controls** panel is organized into tabs with the controls for structure visualization and result fields computed for it (and accessible through the result file).

Different tabs are available depending on the view and the loaded result fields.



Learn more about setting the options in the [Visualization Options](#) topic to optimize the visualization of images from geometry data and property data.

Collapse the **View Controls** panel by clicking  in the upper right corner.

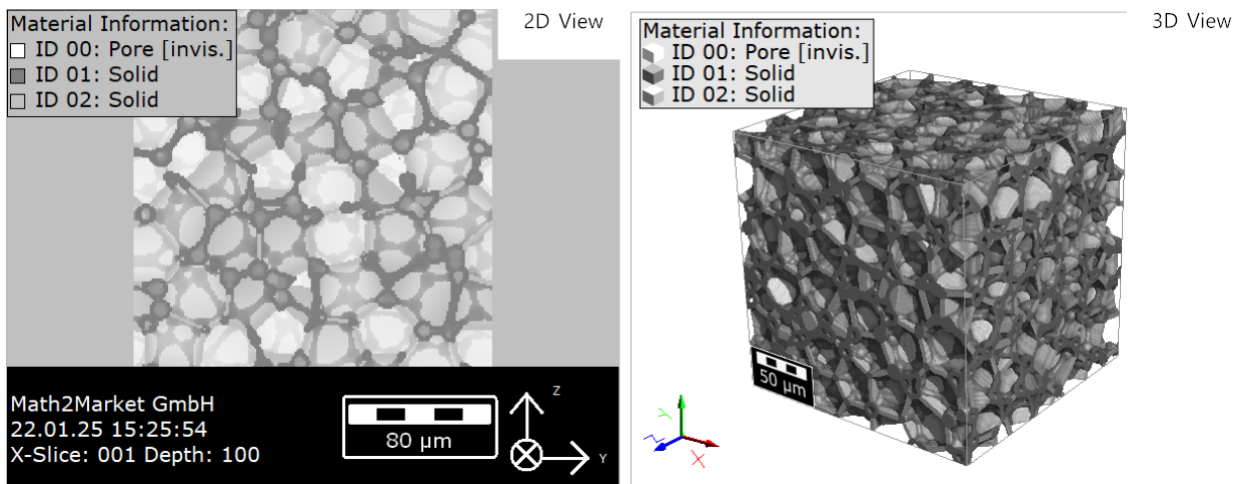
The **View Controls** panel can be undocked by clicking the corresponding icon  in the upper right corner. Close it by clicking  in the upper right corner.



The visualization in 2D or 3D changes the **View Controls**, by activating other tabs to access features for the visualization of geometry data or result data.

1.1.4 Visualization Area

At the program start, the **Visualization area** (also named viewport) shows the GeoDict splash screen. When structures or result fields are loaded in GeoDict, they are shown in the **Visualization area** and the **View Controls** panel appears above it.



In the Visualization area, the **Material Legend** shows information on the colors and materials IDs used for the objects in the structure. GeoDict's material concept and the material database are described in the [Material Database](#) chapter.

The colors for the materials can be changed through **Settings** → **Visibility & Color Settings**, through the icon  in the toolbar, or by unfolding the **Color & Visibility** tab of the GUI **Sidebar**, as described in the [Color & Visibility](#) chapter.

The material IDs can be changed through **Settings** → **Select Constituent Materials...** in the menu bar or the  icon in the toolbar.

2D Cross Section

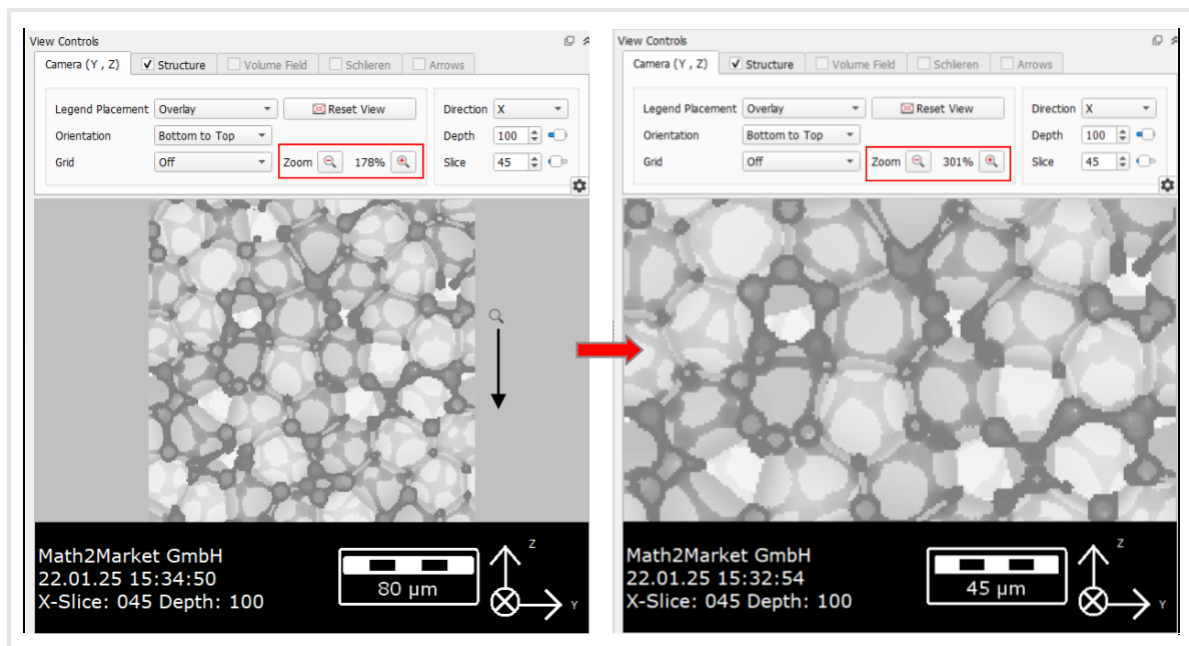
In 2D cross section view, the **Visualization area** shows a slice of the current image of a structure or result field and the positioning of the [Camera](#) can be set in the **View Controls**, with the controllers of the 2D view, under their tabs.

At the bottom of the Visualization area, the **Status Bar** shows information about the visible slice and direction. When switched on, a scale bar shows the physical size of the visible image. The position and visualization of the visualization area elements can be changed under [Legends and Overlays](#).

Additionally, the visualization of the structure or result field can be changed as described in the following.

Zoom

The 2D image itself is displayed in the center of the Visualization area. Zoom either stepwise with the **Zoom** icons in the **Camera** tab or smoothly by keeping the right mouse button pressed and simultaneously moving the mouse forward. By moving the mouse backward, the image zooms out. The zoom value in the **View Controls (Camera tab)** is updated accordingly.



✓ Pan

As long the structure is not zoomed out to be smaller than the domain width, it can be **moved** within the visualization area by pressing and holding the left mouse key and dragging it around.

✓ Reset Camera and Zoom

The zoom and the position in the Visualization area can be reset by clicking  **Reset** in the **Camera** tab.

3D Rendering

In the Visualization area, below the **View Controls**, the **Material Legend** shows information on the colors and materials IDs used for the objects in the structure. The domain axes can be found in the bottom left visualizing how the structure is rotated. If additionally volume fields are loaded, the corresponding legend appears on the right. The positions of the visualization area elements can be changed as described under [Legends and Overlays](#)¹¹⁷.

Additionally, the visualization of the structure or result field can be changed as described in the following.

✓ Rotate

With **3D Rendering** view selected, the structure image can be **rotated** by placing the cursor on the Visualization area, keeping the left mouse button pressed and moving it.

Change the **focus point** for the rotation by double-clicking in the visualization area. Then, the structure can be rotated around this point.

In the **Camera tab** of the **View Controls** the values for **Rotation** (HDG, ATT, and BNK) are updated accordingly.

 Reset	Translation X:	0.00	Rotation HDG:	-43.08	Rotation X:	0.00
	Translation Y:	0.00	Rotation ATT:	1.04	Rotation Y:	0.00
	Translation Z:	0.00	Rotation BNK:	26.67	Rotation Z:	0.00

✓ Zoom

Zoom by keeping the right button pressed and simultaneously moving the mouse forward. By moving the mouse backward, the image zooms out. Zooming happens also by scrolling with the mouse wheel.

In the **Camera tab** of the **View Controls** the value for **Translation Z** is updated accordingly.

 Reset	Translation X:	0.00	Rotation HDG:	0.00	Rotation X:	0.00
	Translation Y:	0.00	Rotation ATT:	0.00	Rotation Y:	0.00
	Translation Z:	6.57	Rotation BNK:	0.00	Rotation Z:	0.00

✓ Pan

Pan the structure around by keeping the middle mouse button pressed and simultaneously moving the mouse in the desired direction.

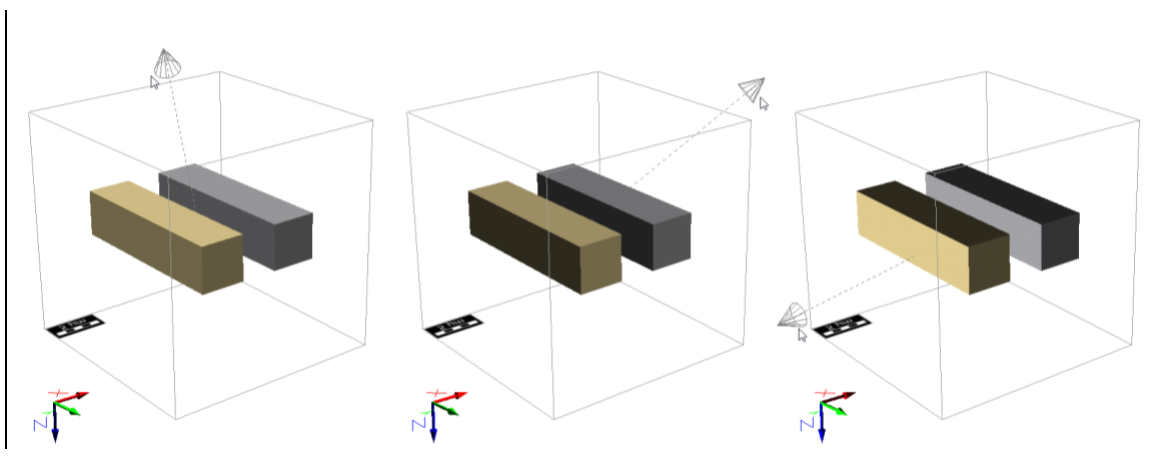
In the **Camera tab** of the **View Controls** the value for **Translation X** and **Y** are updated accordingly.

 Reset	Translation X:	-1.21	Rotation HDG:	0.00	Rotation X:	0.00
	Translation Y:	0.26	Rotation ATT:	0.00	Rotation Y:	0.00
	Translation Z:	0.00	Rotation BNK:	0.00	Rotation Z:	0.00

✓ Rotate Light Source

The source of light falling on the structure can be changed by moving the mouse while keeping the **CTRL** key and the left mouse button pressed.

Also, the light source can be easily changed, by clicking the  icon in the toolbar. Then, a beam spot appears in the Visualization area near the structure model. Move the mouse with the left mouse button pressed to observe the changing position of the light source. The beam spot disappears after clicking the  icon in the toolbar and moving the mouse.



✓ Reset Camera and Light Positions

The camera and light positions can be reset by clicking  in the **Camera** tab.

1.1.5 Visualization Tab

The **Visualization Settings** dialog unfolds when clicking the **Visualization** tab in the **Sidebar** of GeoDict and it covers and combines most parameters, commands, and settings that are located in the menu bar and in the [View Controls](#)^[14], and also some additional parameters.

Thus, it allows to set all values for the visualization of structures and results fields obtained with GeoDict. Some of these parameters are redundant and also displayed in the [View menu](#)^[5] in the menu bar and the [View Controls](#)^[14], but are duplicated in the GUI **Sidebar**. Others are only available in the **Sidebar**.

Collapse the **Visualization Settings** again either by clicking  in the upper right corner or by clicking the **Visualization** tab in the sidebar.

The **Visualization Settings** can be undocked by clicking the corresponding icon  in the upper right corner. Close it by clicking  in the upper right corner.

✓ Doubling of Visualization Options

In the description of the [visualization options](#)^[23], many options are shown in both, the **View Controls** panel and the **Visualization** tab of the sidebar. For example:

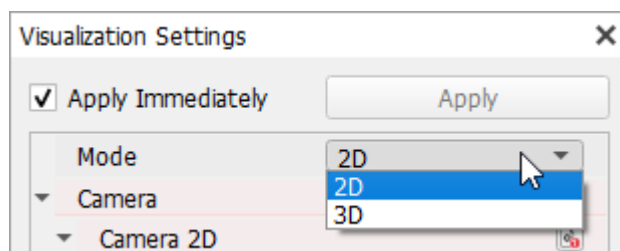


General Options

Initially, only visualization settings related to the current structure are shown in the dialog.

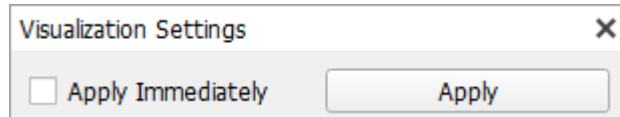
Clicking the gear wheel icon  in the bottom right of the **View Controls** opens the **Visualization Settings** dialog automatically to the corresponding section.

In the example above, the structure is shown in 2D cross section view in the Visualization area and, when opening the **Visualization Settings** dialog, the **Mode** is automatically selected as **2D** and all parameters concerning the display are listed in the panel. This feature simplifies changing the values of these parameters, as it is not necessary to search for them in the menu bar or in the **View Controls**. The selection of the **Mode** to be **2D** or **3D** affects the choices of settings for these two ways of rendering.



Tooltips guide you through the parameters of the **Visualization Settings** dialog, when resting the cursor on the respective parameters.

All settings changed in the **Visualization Settings** dialog are applied directly if **Apply Immediately** is checked. When the checkbox is disabled, the **Apply** button becomes clickable to apply the changes only when the user decides to do it.

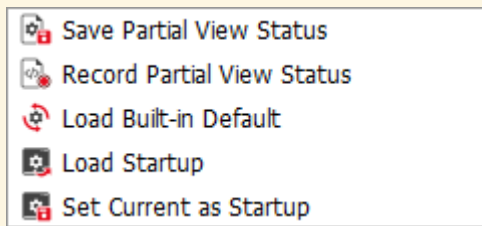


Know how! On the bottom of the **Visualization Settings** dialog, a group of icons controls the following in regard to visualization. Resting the mouse pointer over an icon prompts a tooltip showing the icon function to appear. For example, through them, it is easy to save *all* the current view settings and reload them:

	Undos the last change in the view settings.
	Redos the last undo.
	Saves the current view settings to a GeoPy (*.py) file.
	Loads view settings from a GeoPy (*.py) file.
	Resets all view settings to the built-in default settings.
	Resets all view settings to the saved start-up settings.
	Saves the current view settings as start-up settings.
	Records the current view settings in the session macro. If a macro is recording, the current view settings are also recorded to the macro.
	Synchronize the visualization settings of two GeoDict instances on the same Computer to compare structures and volume fields.
	Shows all visualization settings. If not selected, only the view settings affecting the current visualization are shown.



Know how! Additionally, many view settings can be saved and loaded partially by clicking on the corresponding icon  next to the setting parameter.



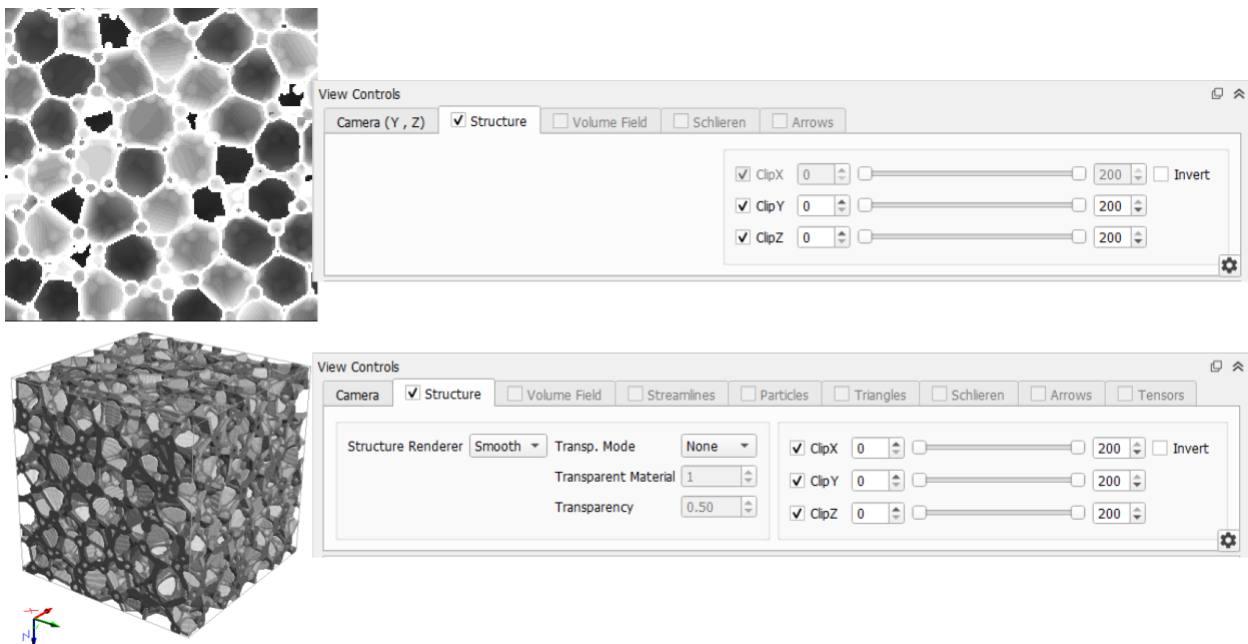
- **Save Partial View Status** saves all settings corresponding to the selected section in a GeoPy macro (*.py) and can be applied whenever needed by selecting Macro Execute Macro / Script from the menu bar. To learn more about how to record, execute and edit macros refer to the GeoPy scripting chapter.
- **Record Partial View Status** records the settings corresponding to the selected section to the macro currently recording. When no macro is recorded, nothing happens when selecting this option.
- **Load Built-in Default** loads the built-in default values only for the selected section.
- **Load Startup** loads previously saved custom start-up settings for the selected section. If no custom start-up settings were saved, the built-in default values are loaded.
- Use **Set Current as Startup** to set the current settings as start-up settings for the selected section.

1.2 Visualization Options

When the image of a structure or a result (volume) field is displayed in the Visualization area, many preferences and options can be toggled and adjusted in the **View Controls**, under the **View menu**, and in the **Visualization tab** in the GUI **Sidebar** to complement and enhance the images' visual information. The access to several of the visualization options is redundant, meaning that they can be controlled from these three locations, for the user's convenience.

The default display, after starting GeoDict, is 2D Cross Section (SEM) view. The 2D view can be obtained at any time by clicking the 2D icon in the toolbar or selecting the 2D mode in the **Visualization Settings** dialog.

Different tabs appear in the [View Controls](#)^[14] depending on the current view mode (2D Cross Section SEM or 3D Rendering). These tabs are grayed out or selectable depending on the data associated with the image. The content in these tabs is reflected in the [Visualization Settings](#)^[19] dialog to be found in the sidebar.

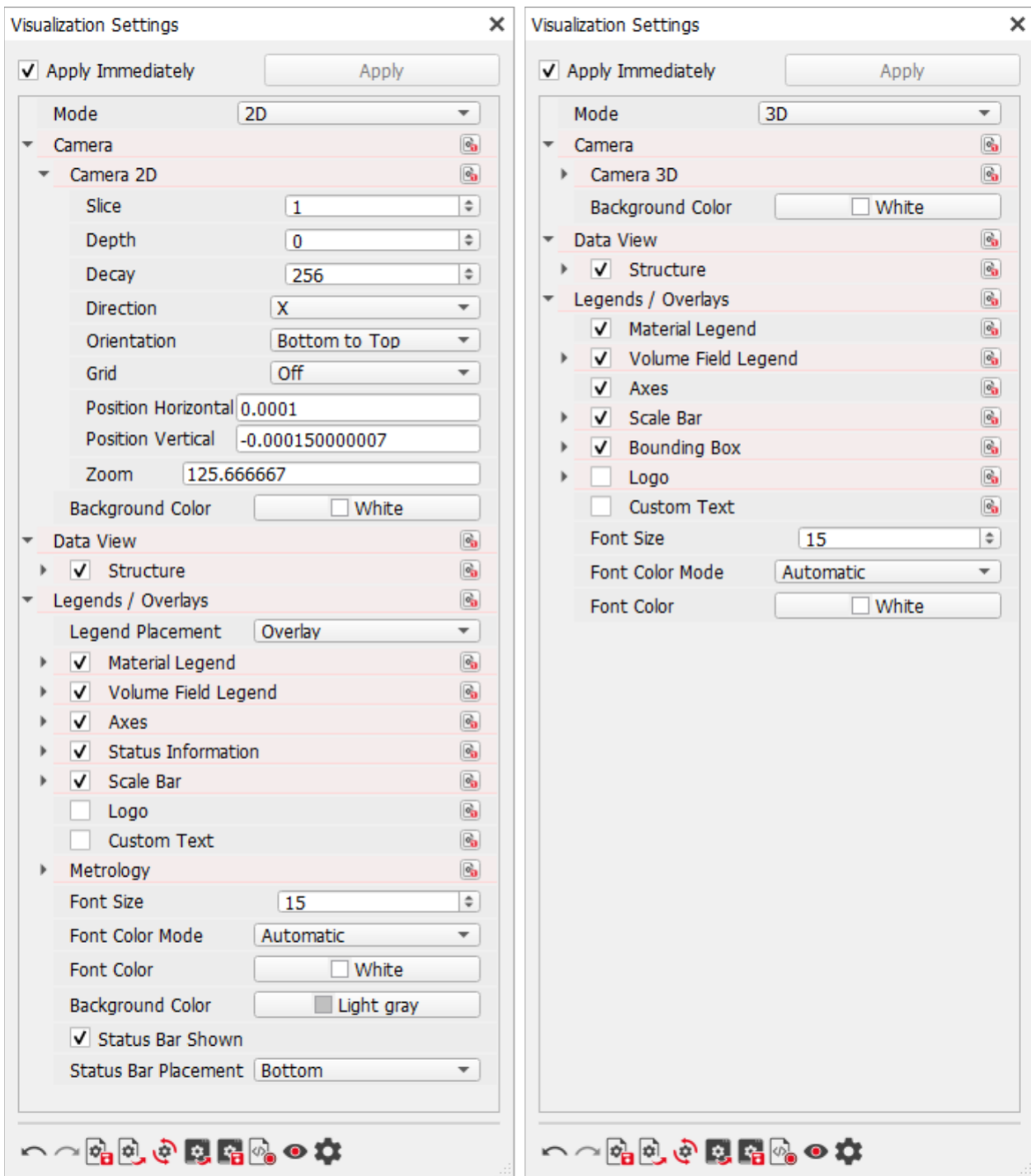


The next graphical examples (below) explain how to set visualization options through the tabs in the **View Controls** and in the trees of the **Visualization Settings** dialog.

All parameters under the [View](#)^[5] menu of the menu bar are integrated in the **Visualization Settings** dialog.

Simulations run on a microstructure with the property predictor modules of GeoDict produce result files (.gdr). The results in the files are numerical in nature and can be directly analyzed and plotted. For many of the predictor modules, a portion of the results are saved into files for visualization in 2D view and 3D rendering as property images. For more information on result files (*.gdr), refer to the [Result Viewer](#) chapter.

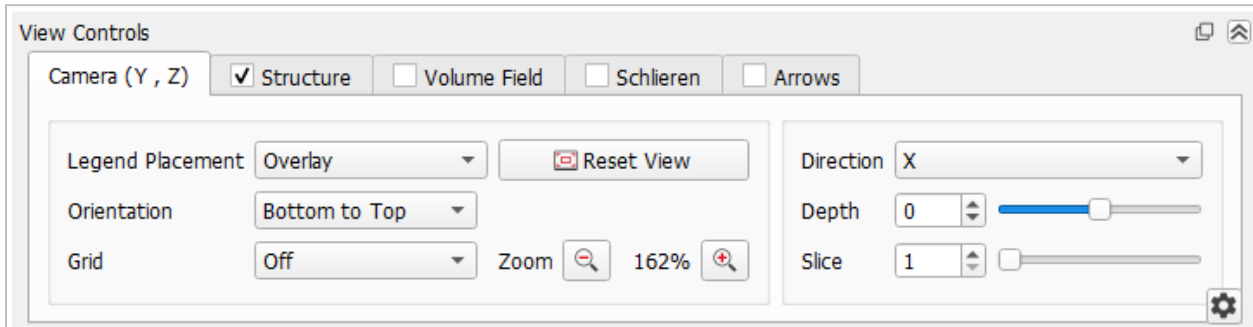
The visualization parameters are explained for [2D cross section](#)^[25] and [3D rendering](#)^[42].



Note! To show the visualization options we use the geometry data of a fiber structure generated with FiberGeo, a grain structure generated with GrainGeo or the volume field from representative result files obtained from thermal conductivity computations with ConductoDict, from flow simulations with FlowDict, from a simulation of particle filtration with FilterDict, and from a simulation of a flexural test with ElastoDict.

1.2.1 Visualization 2D

A structure appears in the Visualization area after loading a file (*.gdt, *.gad). Then, select **View** → **2D Cross Section (SEM)** in the menu bar or click  in the toolbar. Five tabs are present to control the 2D Cross Section View settings: **Camera**, **Structure**, **Volume Field**, **Schlieren** and **Arrows**.



The grayed-out tabs correspond to the visualization of property data from result files. They become available when corresponding result files are loaded.

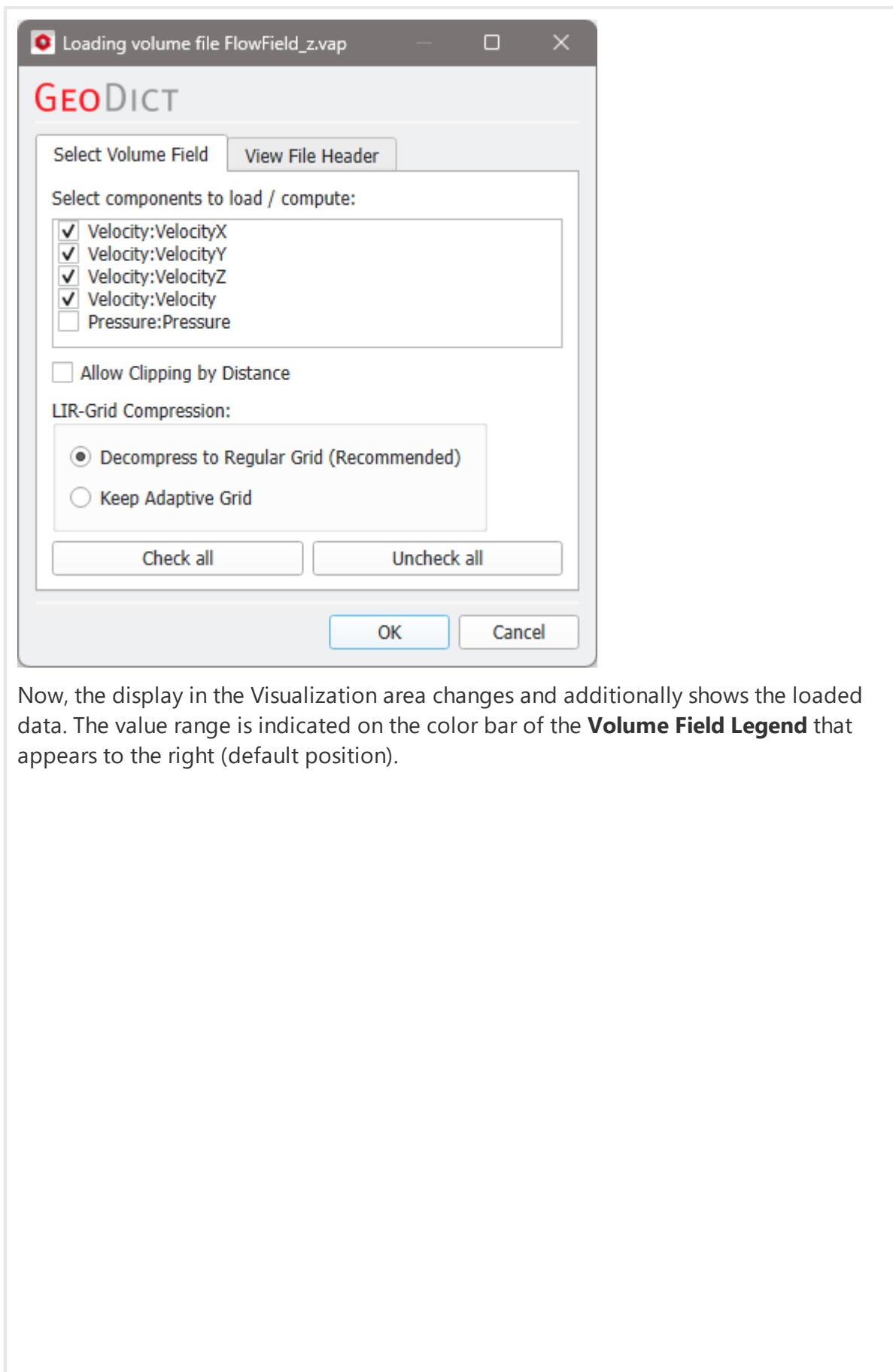
✓ Load result fields to GeoDict

Result fields contain property data from simulations. Many of the analysis and all of the simulation and prediction modules write 3D result fields. For the 2D view, three different kinds of result fields can be visualized: [Volume Field](#)³⁶, [Schlieren](#)⁴¹ and [Arrows](#)⁴¹.

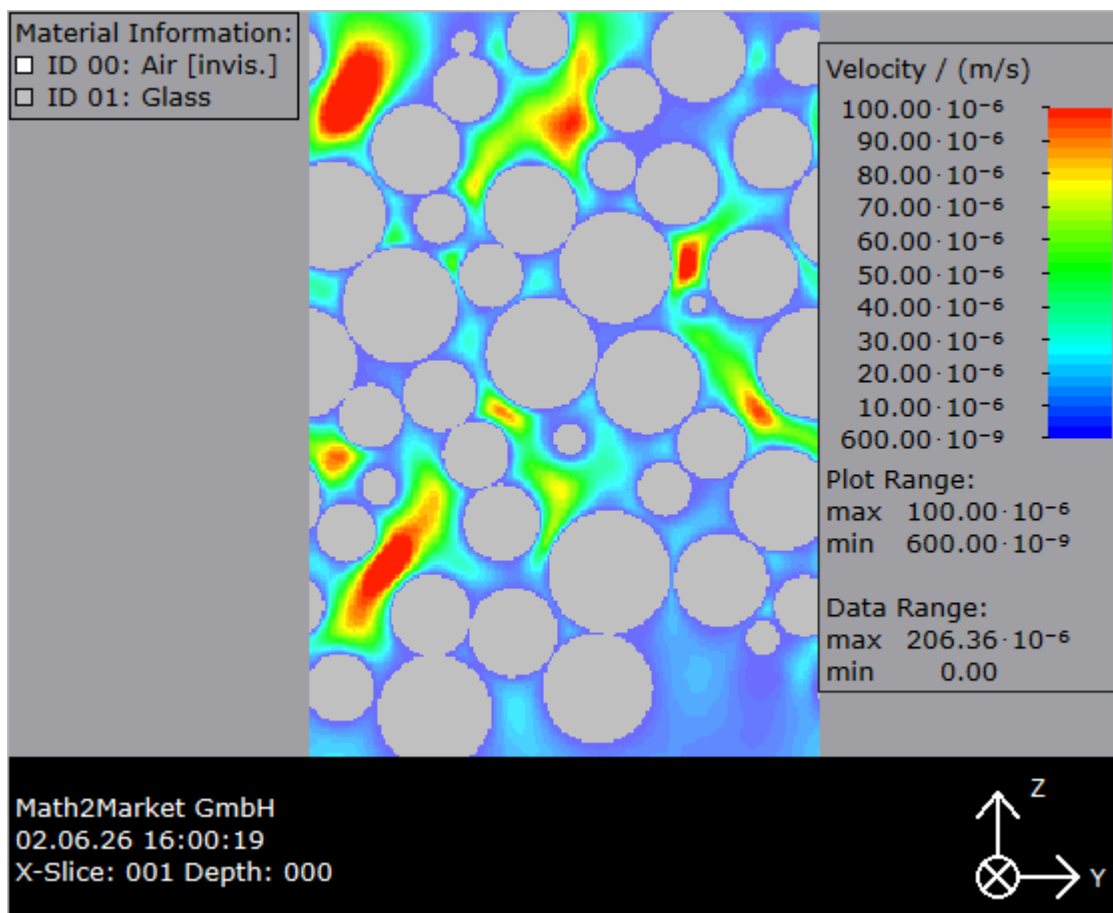
There are two ways to open a result field. The first, and most common, way is loading them from the Result Viewer, as described in the Result Viewer User Guide. Another way is via the File menu (File - Load Volume File).

When loading a volume file a dialog shows up, where you can choose the components whose results you want to visualize.

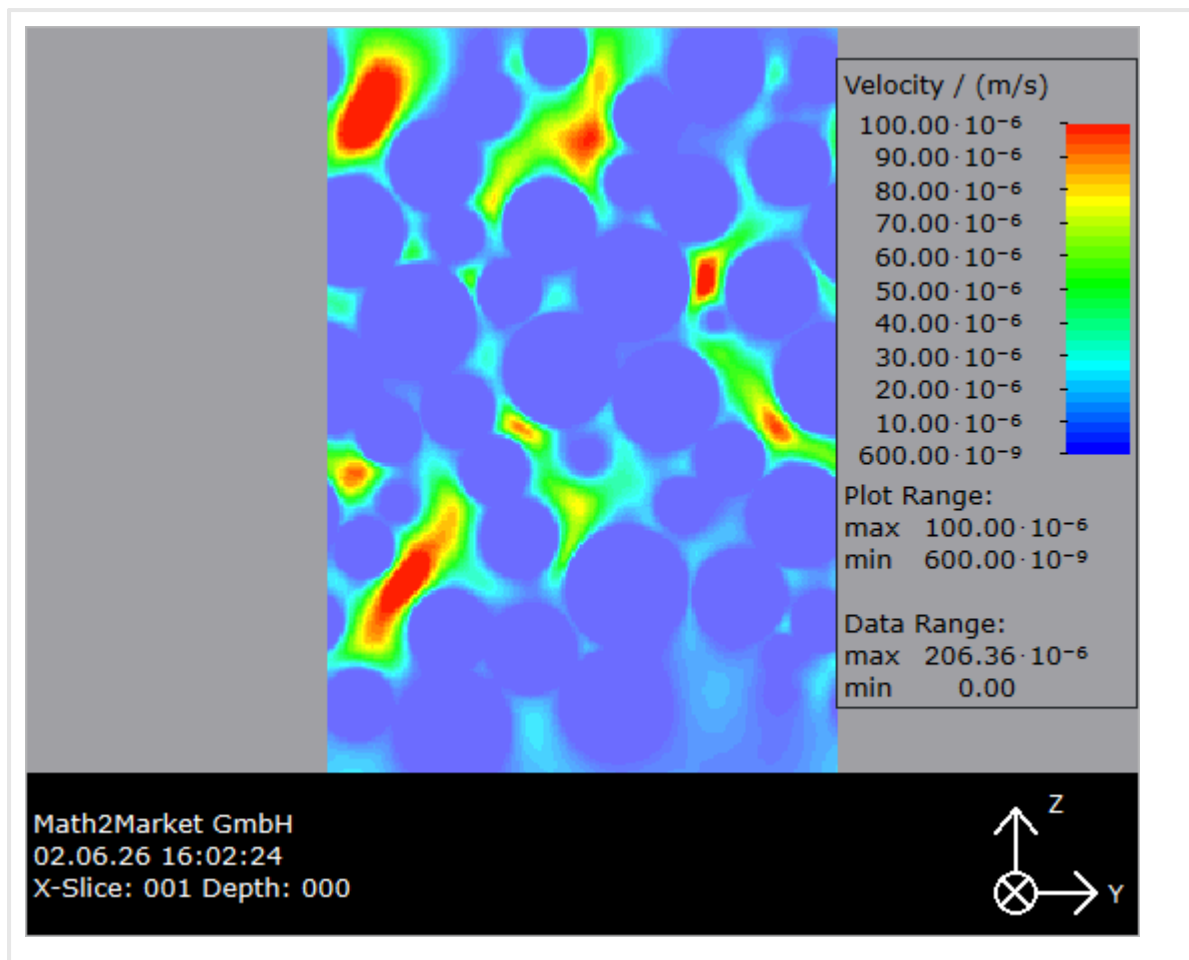
As part of the visualization of the flow field, several GeoDict modules, e.g., FilterDict or FlowDict, can also display the streamlines left by the fluid when flowing through the structure, an arrow field for the flow, and the schlieren produced when the fluid flows through the structure (see [Schlieren imaging](#) for more info). For visualizing streamlines, schlieren, or arrows a complete vector field needs to be loaded, i.e., all components of one image, for example all Velocity components for a flow simulation.



Now, the display in the Visualization area changes and additionally shows the loaded data. The value range is indicated on the color bar of the **Volume Field Legend** that appears to the right (default position).



It may be useful to switch off the [structure](#)^[33] and the display of the [Material Legend](#)^[110] (**View** → **Legends / Overlays** → **Material Legend** in the menu bar or in the **GUI Sidebar**).



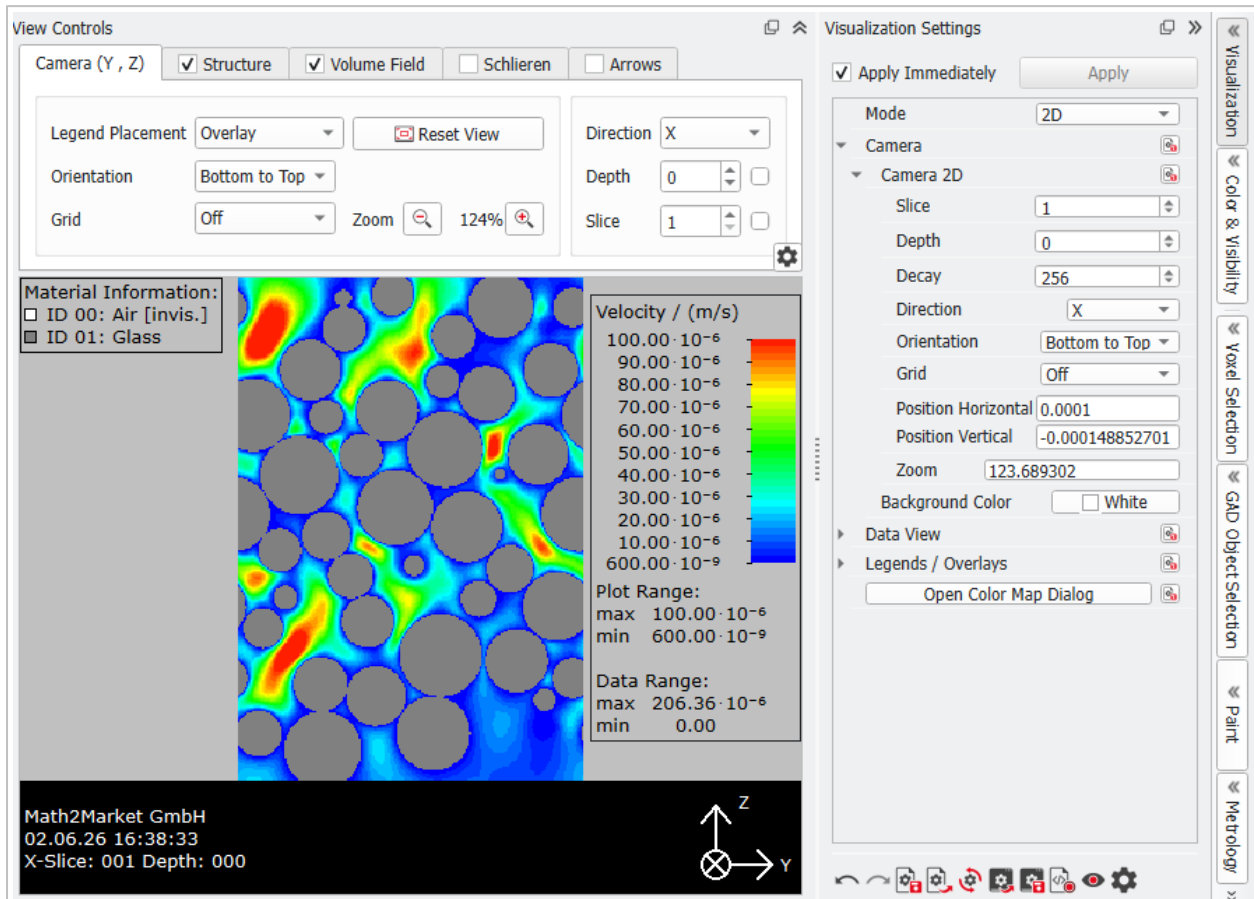
The following tabs are available in 2D view:

- [Camera](#)²⁸
Set the camera settings, as slice and depth.
- [Structure](#)³³
The Structure tab is always selectable and checked by default when a structure is loaded.
- [Volume Field](#)³⁶
Visualize many kinds of result data in 2D, for example velocity, ion concentration, diffusion flux and many more.
- [Schlieren](#)⁴¹
Visualize fluid flow fields through line integral convolution.
- [Arrows](#)⁴¹
Visualize flow via arrows.

Camera

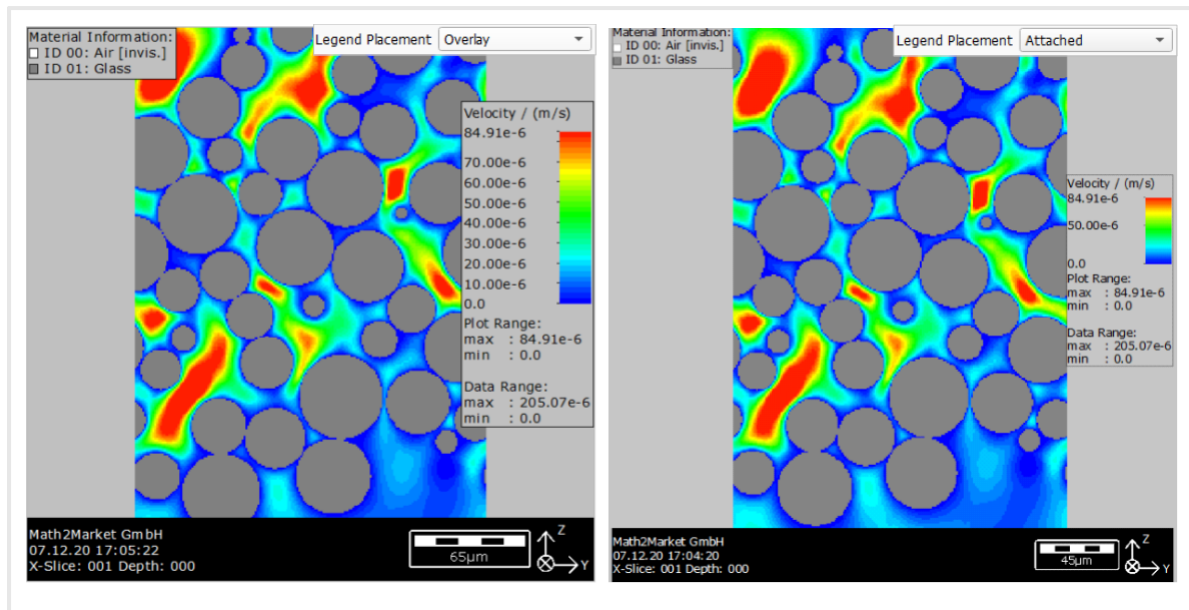
Under the **Camera** tab of the **View Controls**, you can choose which part of the structure is shown in the Visualization area. Here, with the initial default of **Direction X**, the tab is **Camera (Y, Z)**.

Some additional options and most of the options from the **View Controls** are also found in the **Visualization Settings** dialog. Expand the **Visualization Settings** dialog either by clicking the gear wheel icon in the bottom right of the **Camera** tab or by clicking the **Visualization** tab in the sidebar and unfolding the **Camera** → **Camera 2D** section.



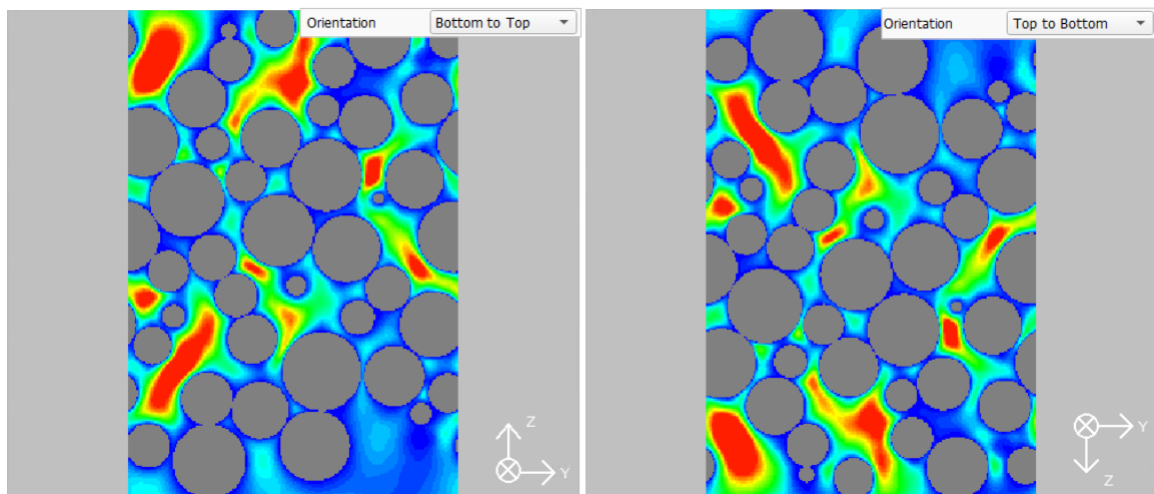
✓ Legend Placement

Two visualization options are available for the **Legend Placement**. If **Overlay** is selected, the legends are displayed above the 2D image, while for **Attached** the legends are placed around the image.



Orientation

Change the image's **Orientation** by switching between **Bottom to Top** and **Top to Bottom**.

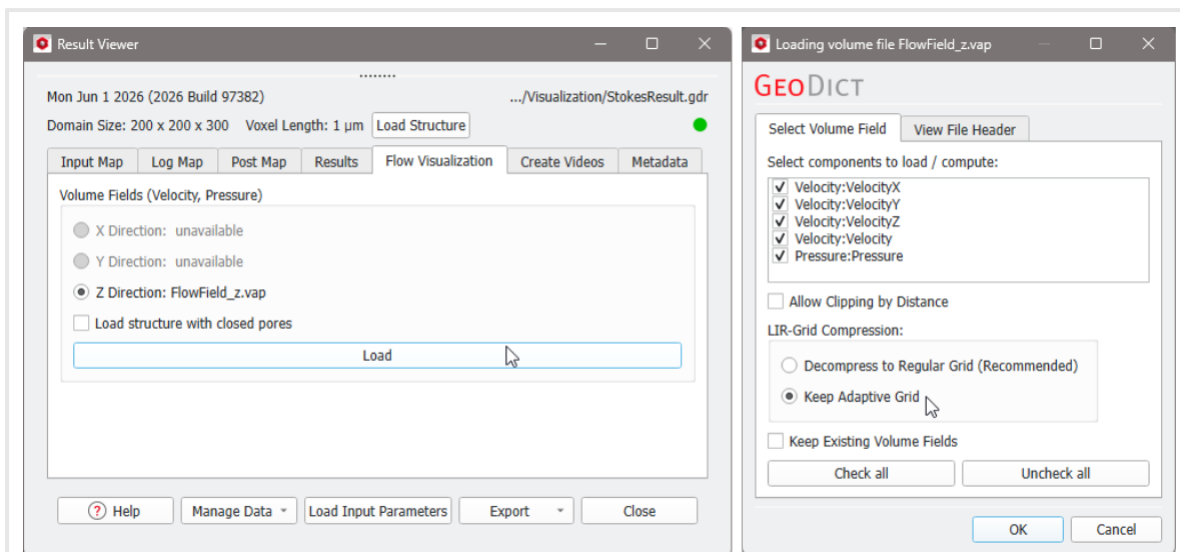


Grid

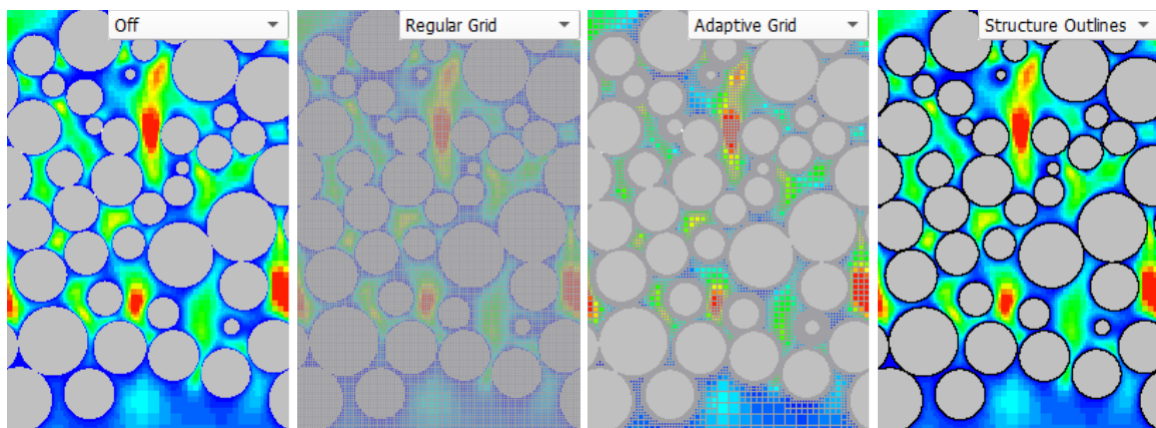
By default, the **Grid** is set **Off**. Enable the grid by selecting one of the other three options: **Regular Grid**, **Adaptive Grid** and **Structure Outlines**.

If a compressed flow field computed with the LIR solver is loaded, the **Adaptive Grid** is visualized when you check **Keep Adaptive Grid** instead of the default **Decompress to Regular Grid** while loading the volume field. (A compressed field is saved when checking **Write Compressed Volume Fields** under the Solver tab of the **LIR Solver Options** dialog).

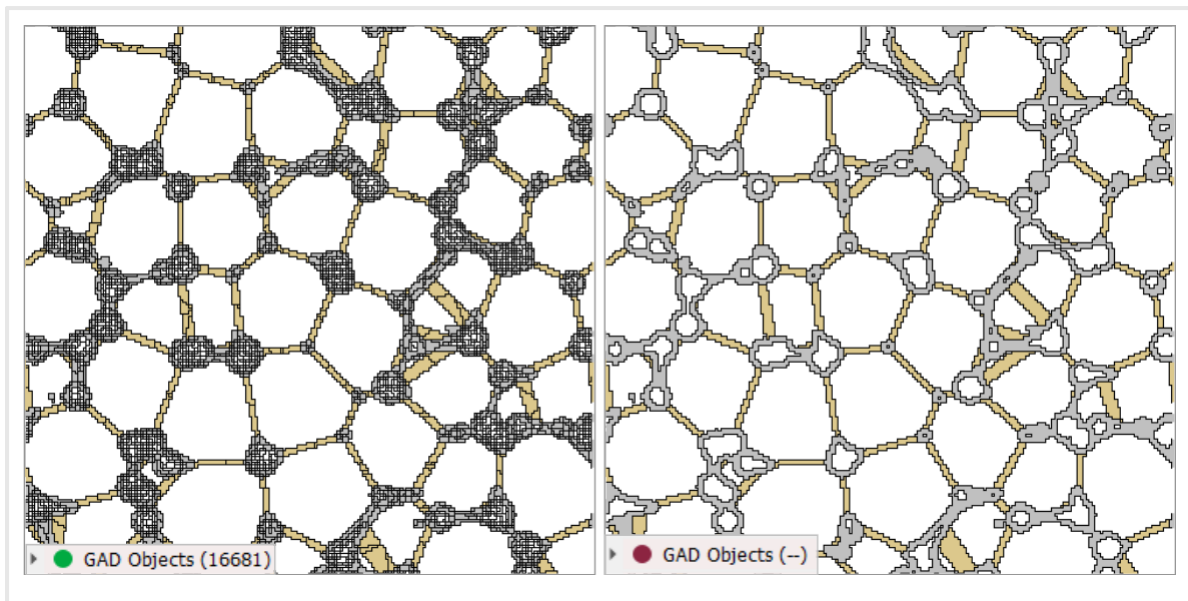
Checking **Keep Adaptive Grid** has the effect of disabling some visualization features.



Select **Adaptive Grid**, as the **Grid** to be shown, under the **Camera** tab in the **View Controls**. Observe the small grid cells displayed near the pore-solid interface and the large grid cells in the center of big pores. Select **Structure Outlines** to display the outlines of the structure.



If GAD-objects are loaded, **Structure Outlines** are computed considering the GAD-objects. Thus, the outlines of each GAD-object are shown. In the example below, the Grid is set to Structure Outlines. On the left, the GAD-Objects are loaded with the foam structure and on the right the GAD-objects were discarded, leaving only the voxel structure in the visualization area.



✓ Zoom and Reset View

Use the buttons to zoom in and out of your loaded structure / result field. Instead you can also use the mouse as described [here](#)¹⁶. In the **Visualization Settings** dialog, control the **Zoom** by entering values directly.

Click **Reset View** to reset zoom and panning.

✓ Direction, Depth and Slice

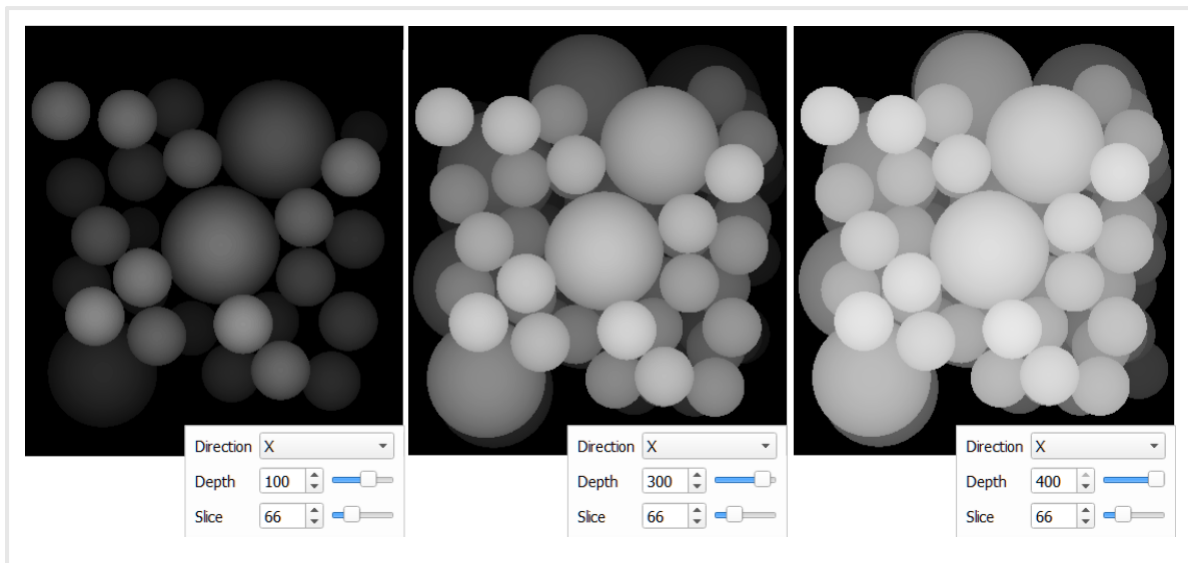
Direction X, Y, or Z, selectable through the pull/down menu, are the directions to which the cutting plane is perpendicular.

For example, choosing **X-Direction** slices the structure along the X-axis and shows images in the YZ-plane. The **Slice** slider controls the display of the cross-section in the selected cutting plane. The slice can be changed by moving the slider, by entering the slice number, or through the up and down arrows.

The **Depth** slider controls the slice view profundity when visualizing in 2D cross section mode. This allows to visualize images very similar to those produced by **Scanning Electron Microscopy (SEM)**.

The maximal **Depth** value is given (and can be changed) by the **Decay** value (**Visualization tab** of the sidebar: **Camera** → **Camera 2D** → **Decay**)

The results of changing the depth are most noticeable in black and white view (**Color & Visibility tab** from the sidebar → **Load Preset** button → **Black and white**).



✓ Position Horizontal and Vertical

When panning the structure in the visualization area with the left mouse key, the values for **Position Horizontal** and **Position Vertical** change accordingly. The structure can only be panned, if zoomed in.

Position Horizontal	0
Position Vertical	0

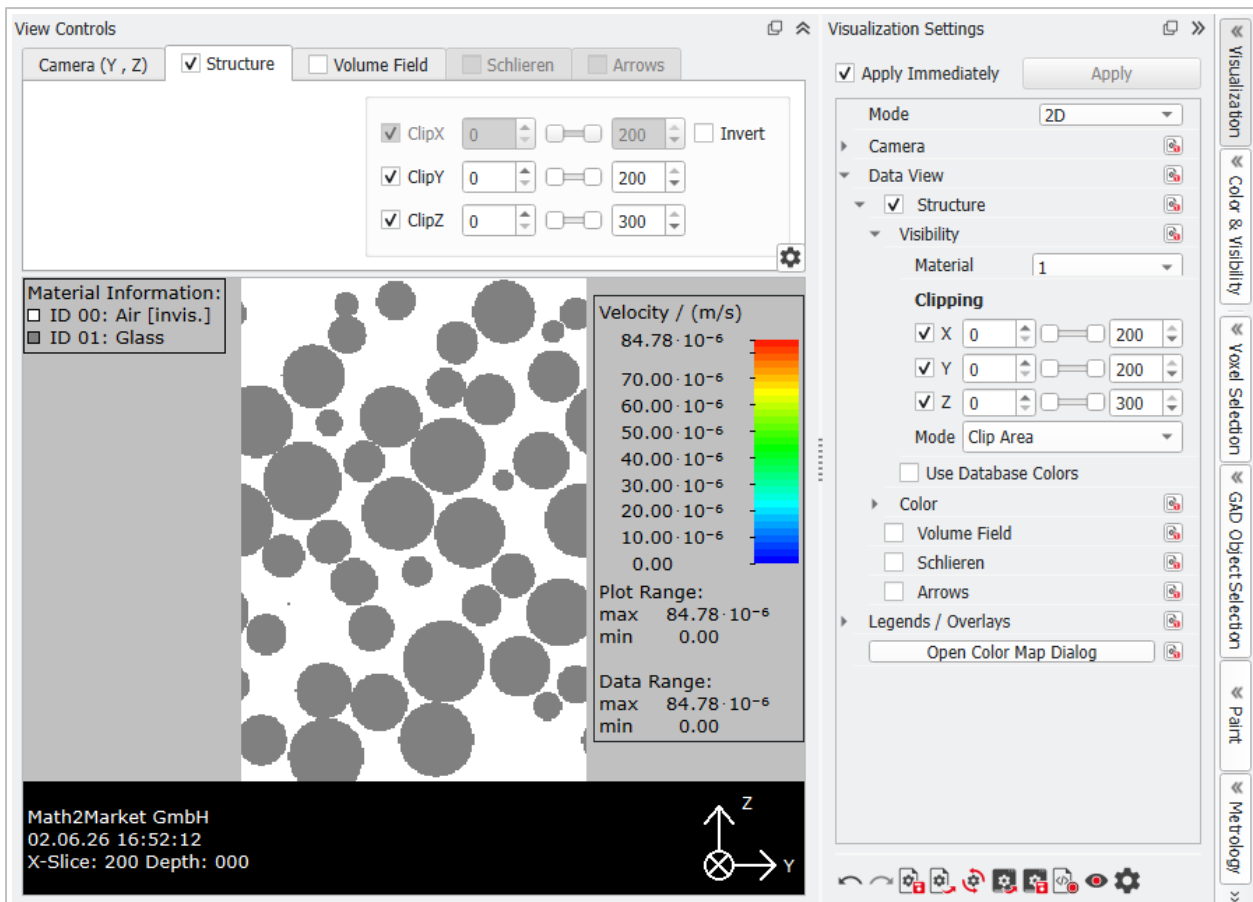
Click **Reset View** to reset zoom and panning.

✓ Background Color

Change the **Background Color** for the structure by clicking on the color button.

Structure

The parameters that control the visualization of the structure in 2D are under the **Structure** tab of the **View Controls** and, with a few more choices, in the **GUI Sidebar** under the **Data View ► Structure** tree as well. The **Structure** tab is selectable when a structure is loaded and it is checked by default.



By unchecking the **Structure** tab, the structure disappears from the Visualization area because the display is disabled. Only the **Material Legend**, a white rectangle showing the domain, and the **Status Bar** remain visible.

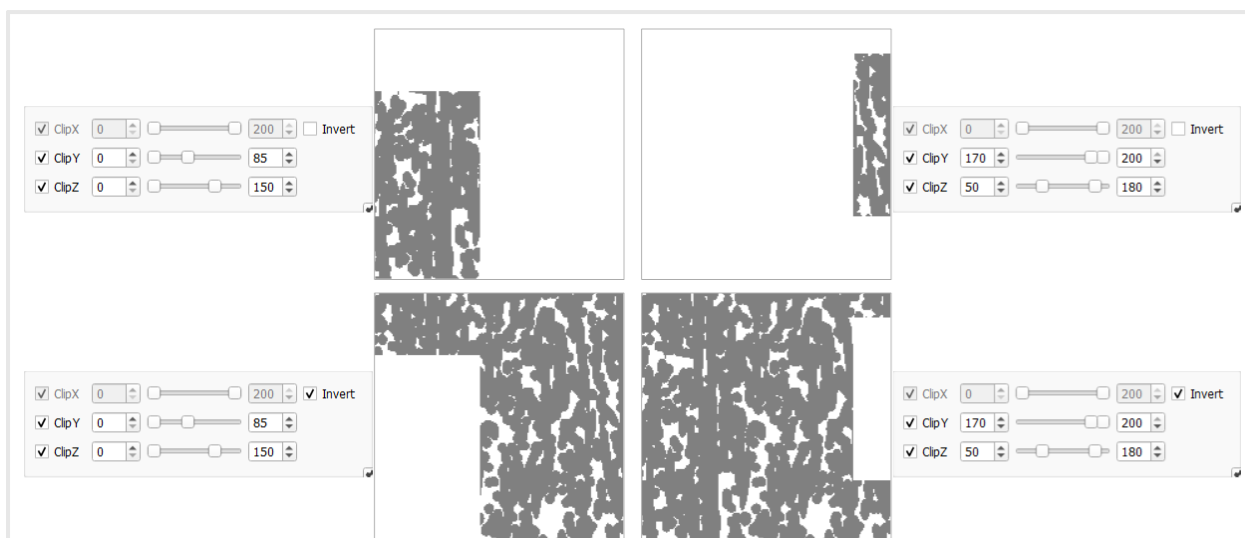
The same effect is achieved by unchecking ☐ **Structure** through the **View** menu or from the **Visualization** tab of the GUI **Sidebar**.

✓ Clipping

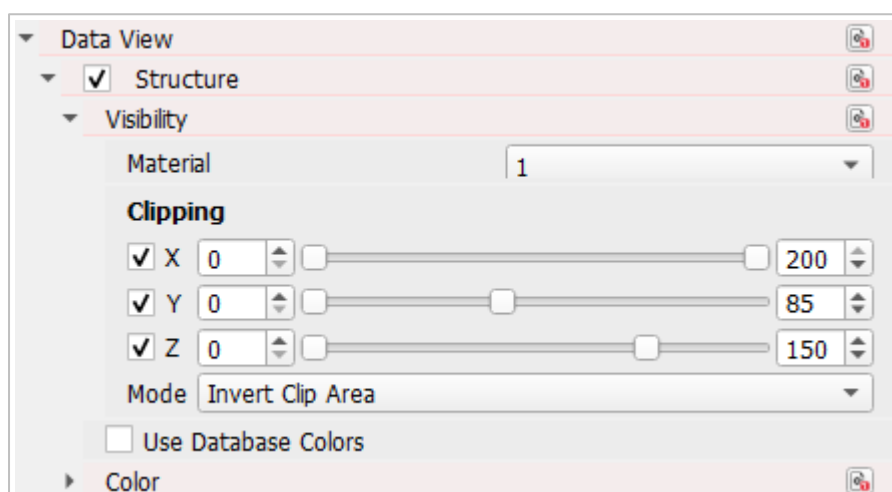
In the **Structure** tab, **Clipping** of the values **ClipX**, **ClipY**, and **ClipZ** allows cutting the displayed geometry in two of the three spatial directions with the sliders or by direct input. The clipping values corresponding to the selected view direction are not available, and thus grayed out.

Additionally, after defining a clipping area with the sliders or direct input, the user can limit this as the area of interest by checking **Invert** in the **View Controls**. Only the area defined by the clipping values remains visible in the Visualization area.

In the following example, **X** was selected for **Direction** in the **Camera** tab. Thus, the clipping is done in Y- and Z-direction.



Find the clipping options in the **Visualization Settings** dialog under **Visibility**.

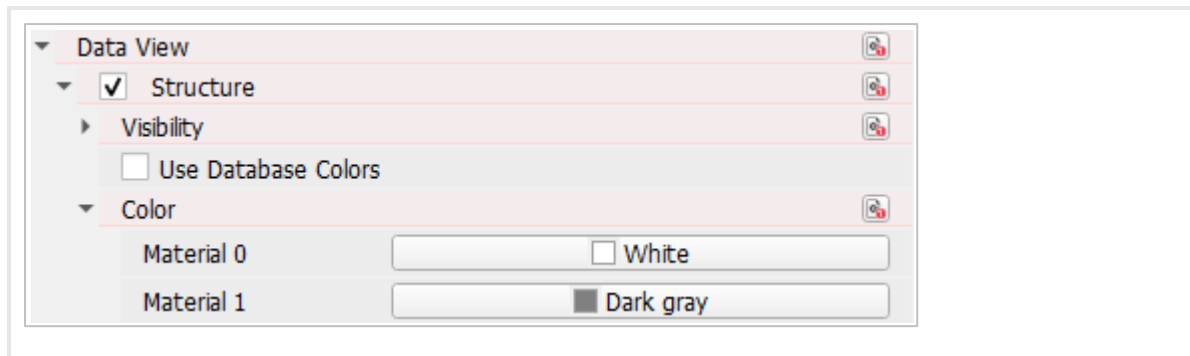


There, define which materials should be visible from the **Material** pull-down menu. This is the same option as the visible checkboxes in the [Color & Visibility](#)¹¹ tab.

✓Color

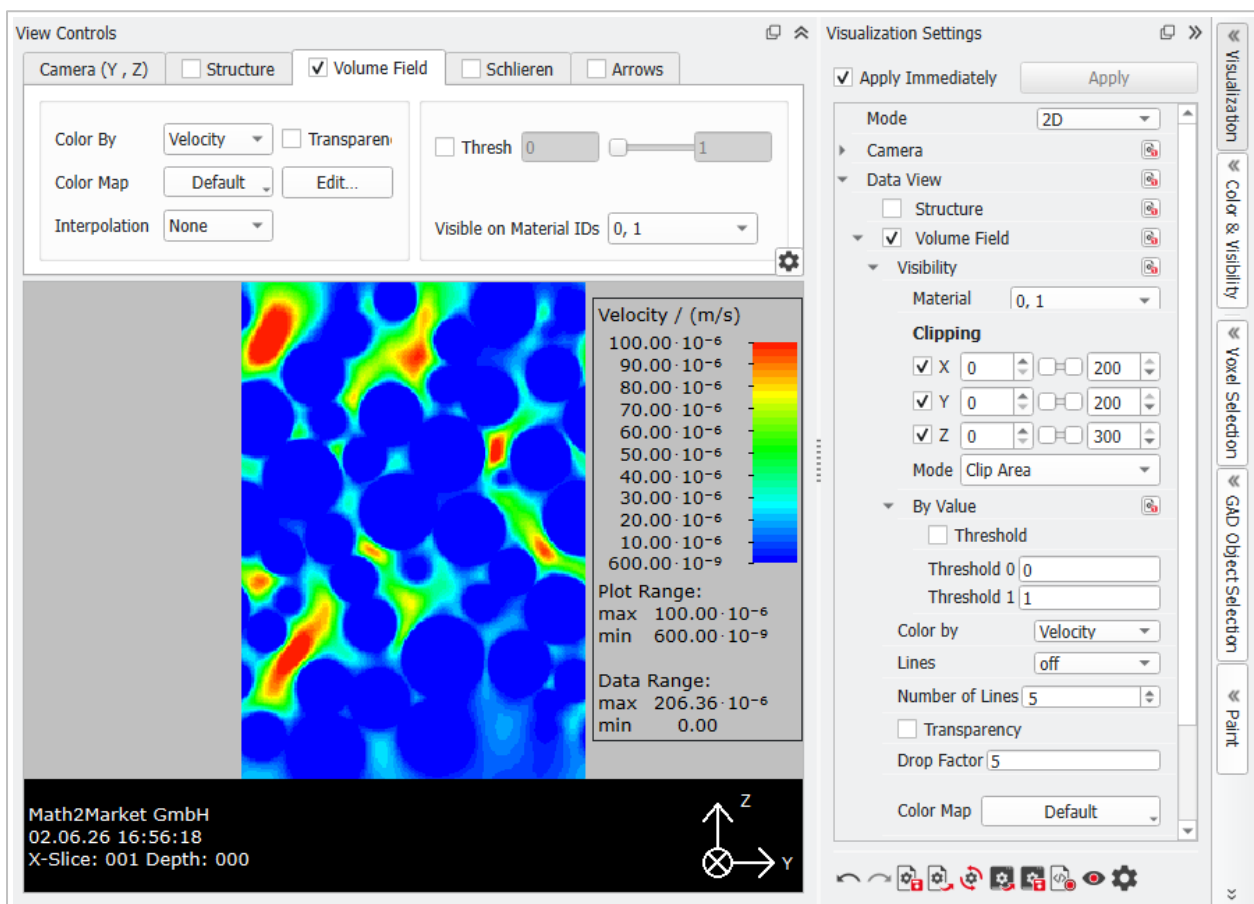
The **Structure** section in the **Visualization Settings** dialog allows to set the **Visibility** and the **Color** of the materials. They may also be defined in the [Color & Visibility](#)¹² tab.

Select **Use Database Colors** to visualize the materials in the colors saved in the GeoDict material database. The colors are also automatically changed according to the materials, if they are reassigned or a new structure is loaded or generated. For materials set to manual, the colors are not changed. To learn more about the database and how to set material colors, refer to the [Material Database](#) chapter.



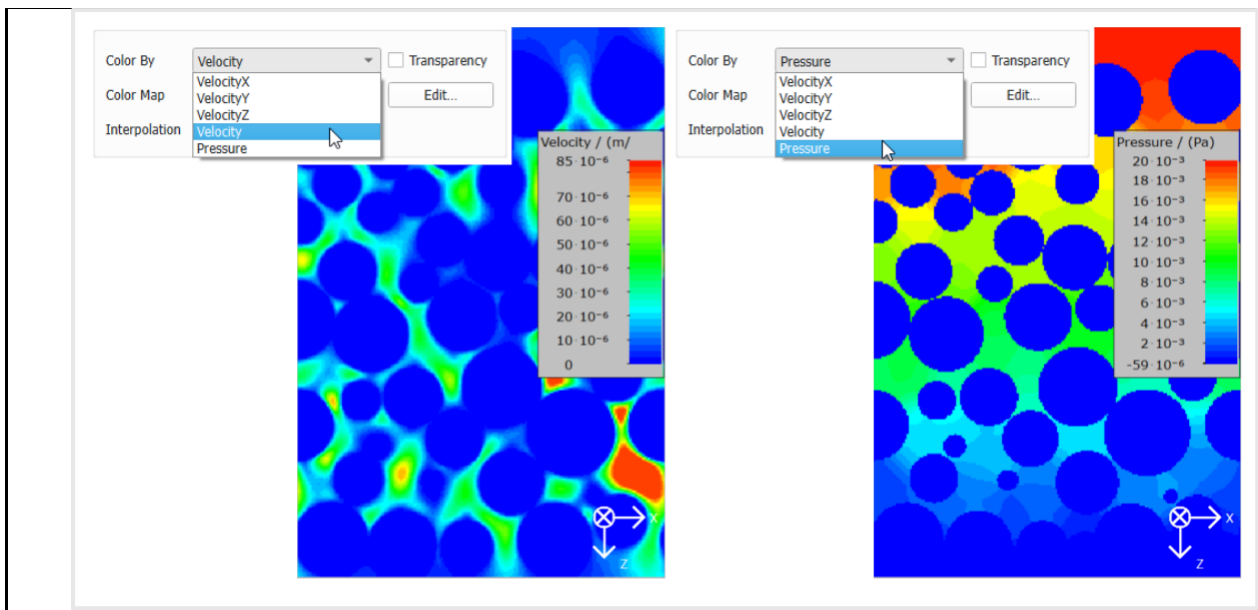
Volume Field

The parameters that control the visualization of the volume field in 2D are under the **Volume Field** tab of the **View Controls** and, with a few more choices, in the **GUI Sidebar** under the **Data View ► Volume Field** tree as well.



Color By

Choose the component to be rendered through the **Color By** pull down menu.

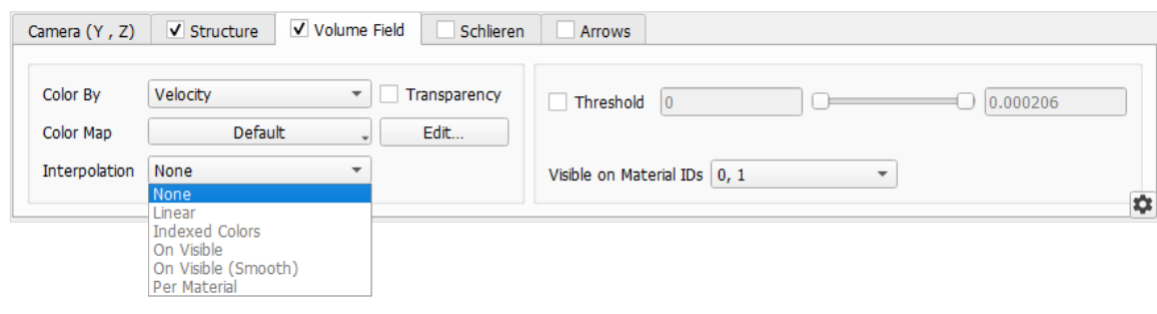


✓ Color Map and Transparency

Define the color range and which values should be transparent. For a more detailed explanation see [here](#)^[121].

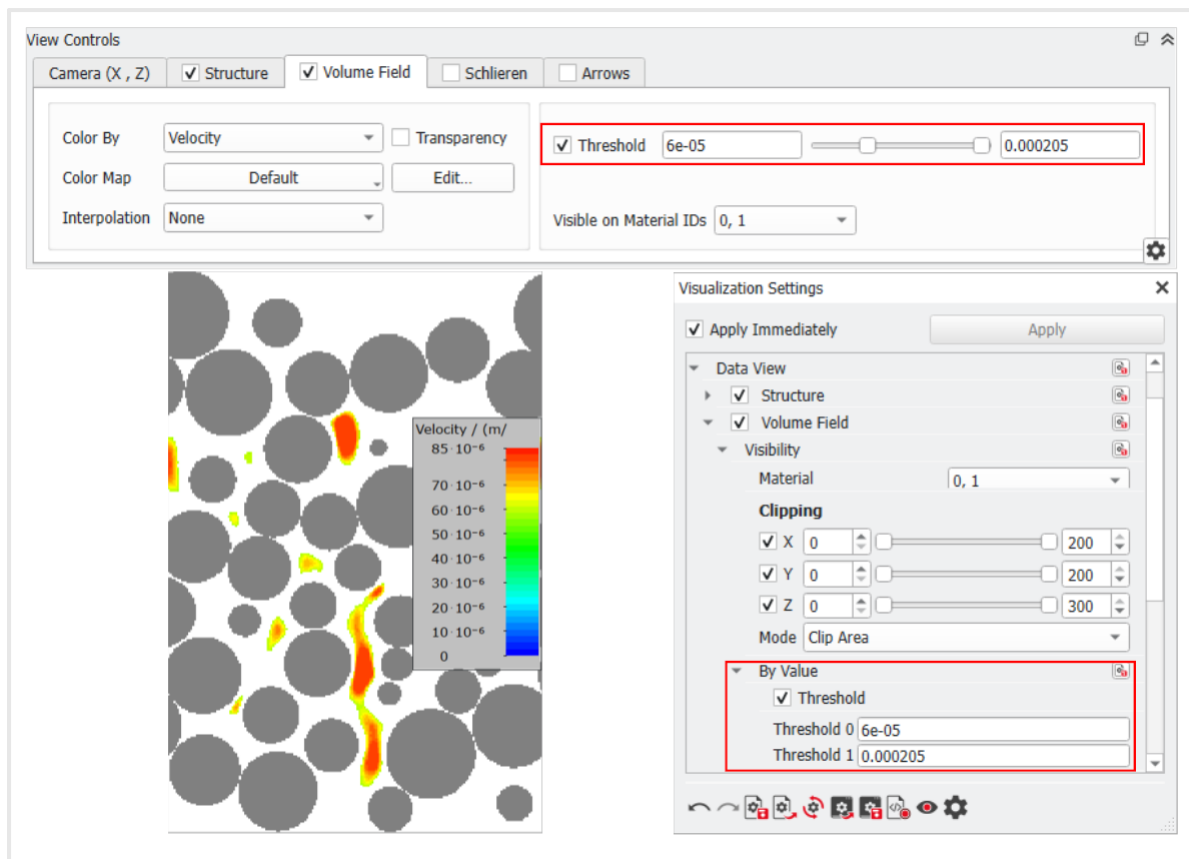
✓ Interpolation

In **2D mode** the volume field colors cannot be interpolated. Thus, only **None** is selectable from the pull-down menu for **Interpolation**. The different [interpolation options](#)^[60] are described for the 3D mode.



✓ Threshold

Checking **Threshold** allows a clipping that is not related to the spatial coordinates but to the visualized data range. The areas with values between the respective threshold values appear white/transparent. This might be used, for example, to visualize only the flow velocity values between $0.00006 \text{ (} 6 \times 10^{-5} \text{) m/s}$ and the maximum of $0.0002 \text{ (} 2.05 \times 10^{-4} \text{) m/s}$. In the **Visualization Settings** dialog, find the option as **Visibility** → **By Value** → **Threshold**.

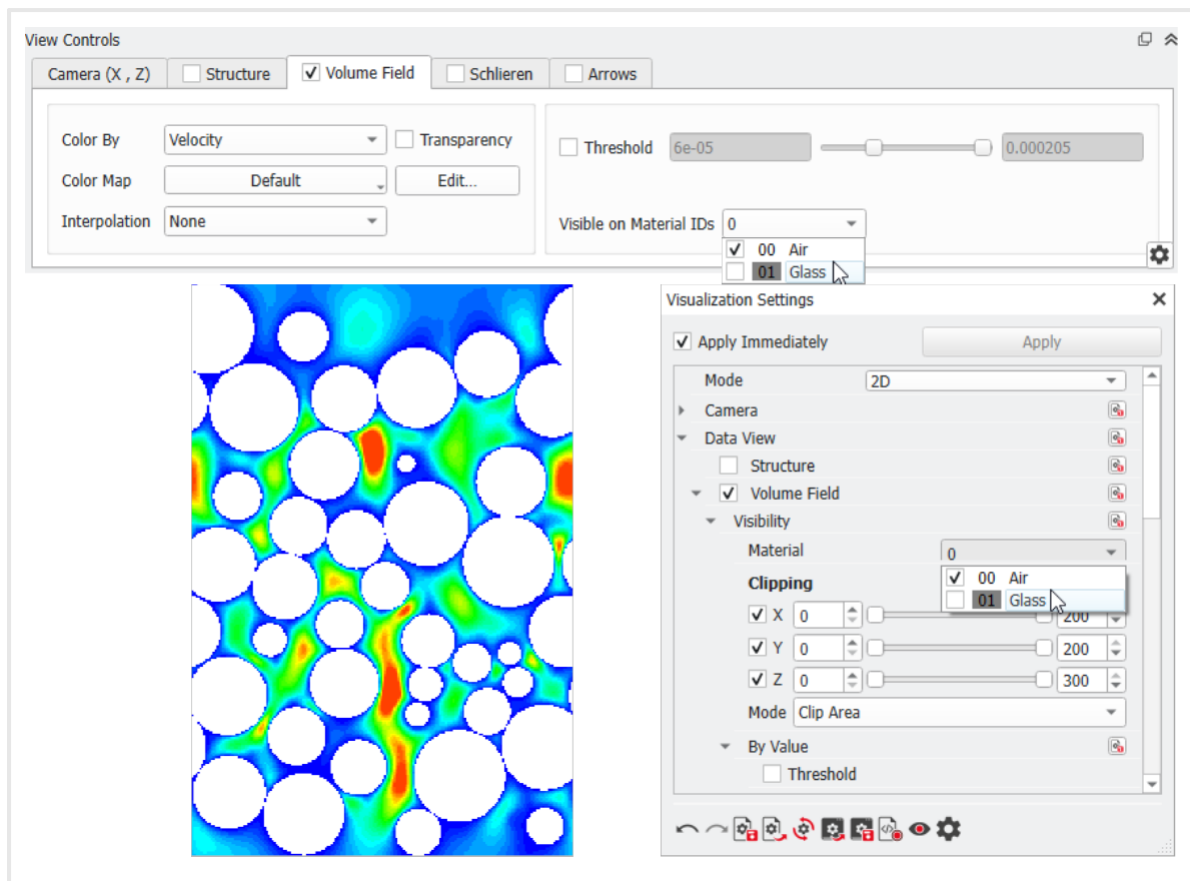


✓ Visible on Materials IDs

In the **View Controls** or the **Visualization Settings** dialog, choose the Material IDs in which the volume field is visualized. This helps in better observing a value of a computed component of the volume field. To do so, select these materials from the **Material ID's** pull-down menu.

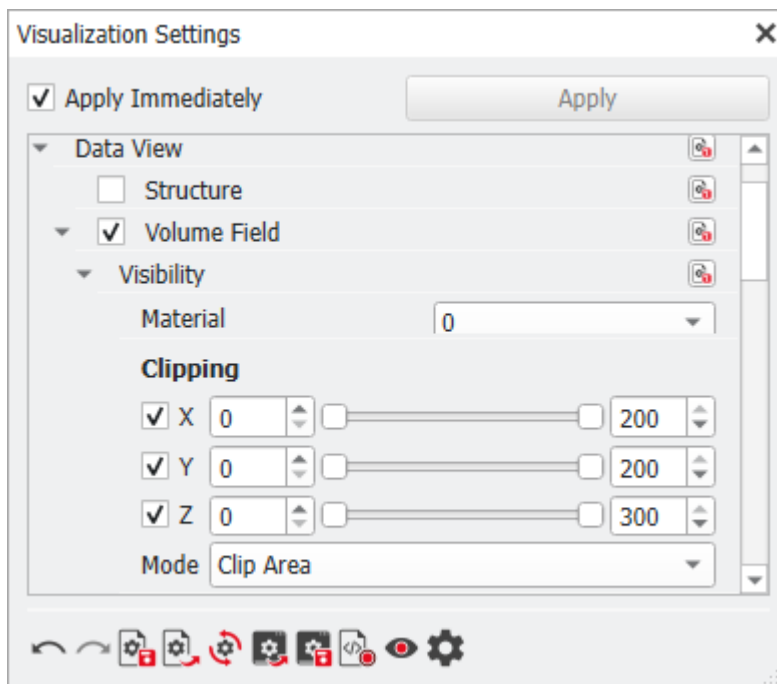
By default, all Material IDs present in the structure are selected.

In the case shown here, the flow only happens in the pore space (here, Material ID 00), since the fluid in the structure does not flow through the solids (here, Material ID 01). The flow velocity is displayed in the space occupied by Material ID 00. The flow velocity is not displayed when Material ID 00 is disabled.



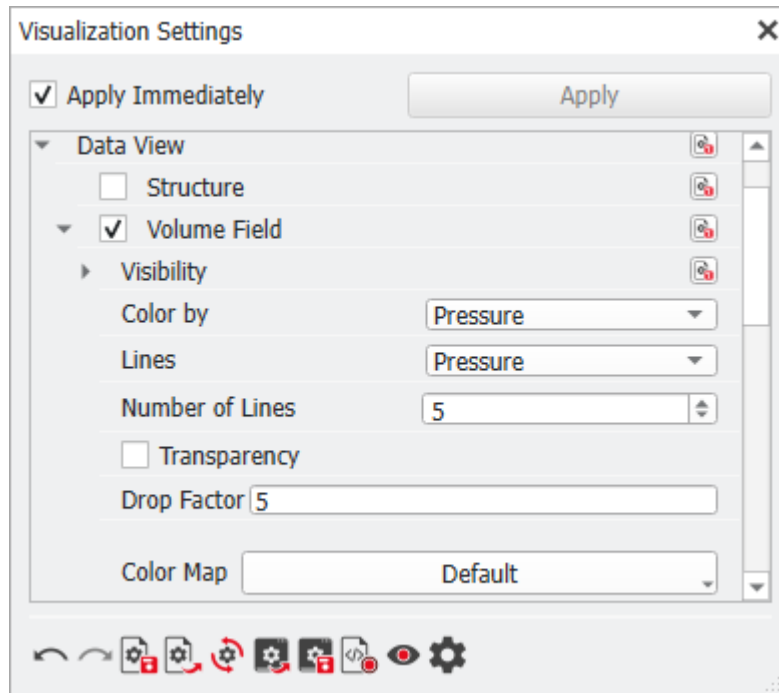
✓ Clipping

In the **Visualization Settings** dialog under **Visibility** you can also clip the volume field as shown for the [structure](#)³⁴.



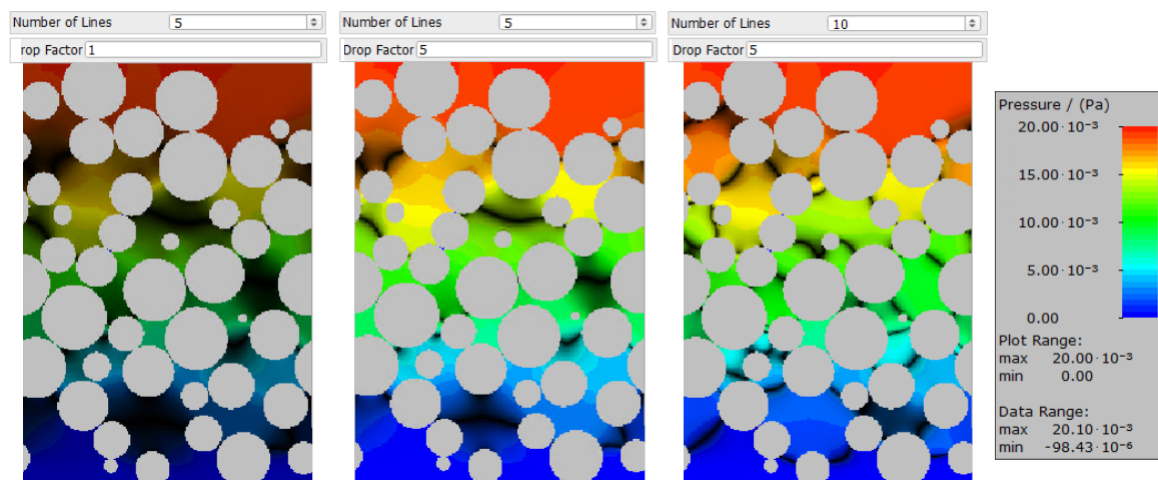
✓ Lines

Points of equal value of the selected local field can be visualized as cross-sections of isolines (appearing as black lines) when changing the default selection **Lines**→ **off** in the **Visualization Settings** dialog (**Data View** ► **Volume Field** ► **Lines**).



Choose the result field component for the lines. It is also possible to visualize for example the velocity colors, while visualizing the Pressure lines. This gives the possibility to look for correlations.

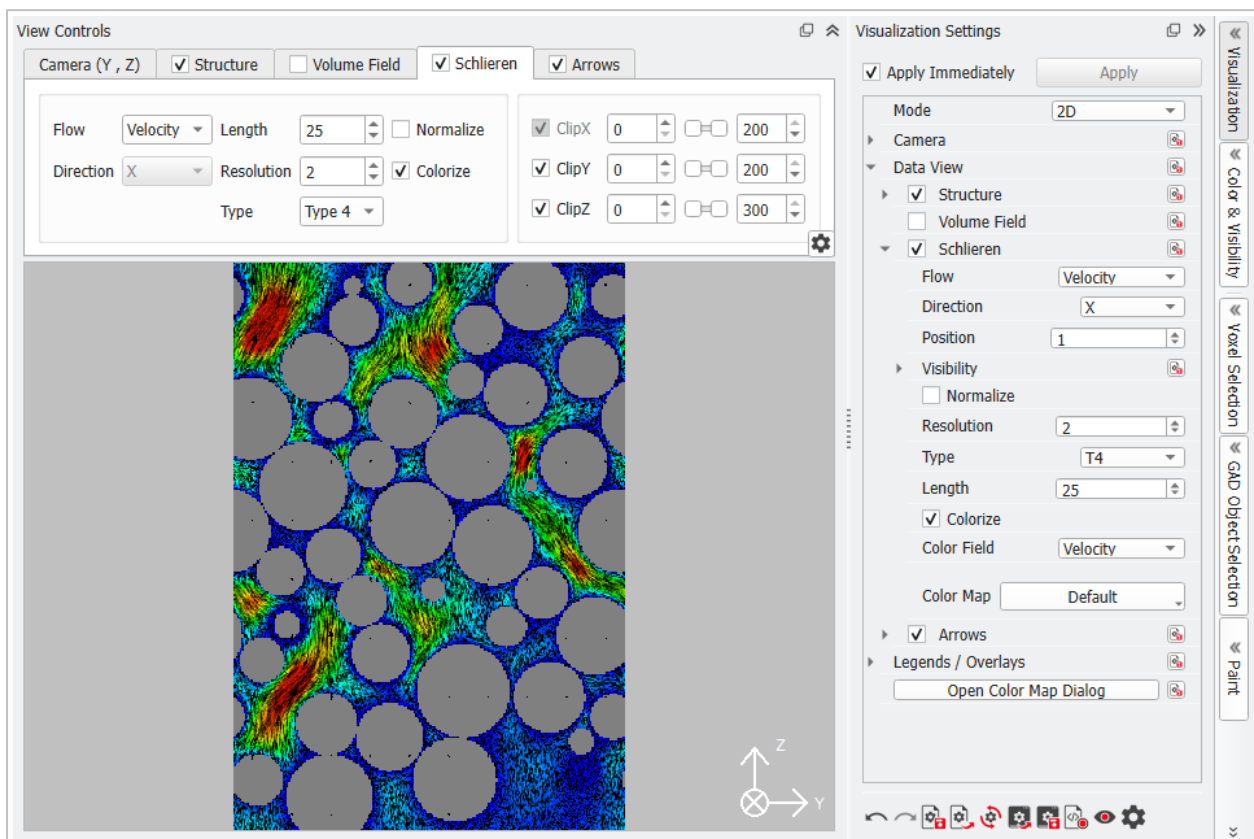
Enter the desired **Number of Lines** and modify the appearance of the lines through the value **Drop Factor**. Smaller values for **Drop Factor** result in larger line shadows, while higher values lead to smaller shadows.



Schlieren

Schlieren (Line Integral Convolution (LIC)) is another option to visualize a fluid flow field computed in FlowDict, AddiDict or FilterDict. LICs are visually displayed as distorted striations or streaks in 2D sections that represent fluid motion through the 3D flow field, in any of the three directions.

The parameters that control the visualization of the Schlieren in 2D are under the **Schlieren** tab of the **View Controls** and, with a few more choices, in the **GUI Sidebar** under the **Data View ► Schlieren** tree as well.



The **Flow** pull-down menu shows one choice for each loaded [vector field](#)^[44]. For a flow field for example, only the field Velocity is available. The Schlieren are computed from the selected field.

The **Direction** is only available for 3D view. In 2D the Schlieren are always visualized for the current view and slice.

Define the **Length**, **Resolution** and **Type** of the Schlieren and decide if they should be **Normalized** and / or **Colorized** as described for the [3D view](#)^[94].

The **Clipping** works similar as for the [structure](#)^[34].

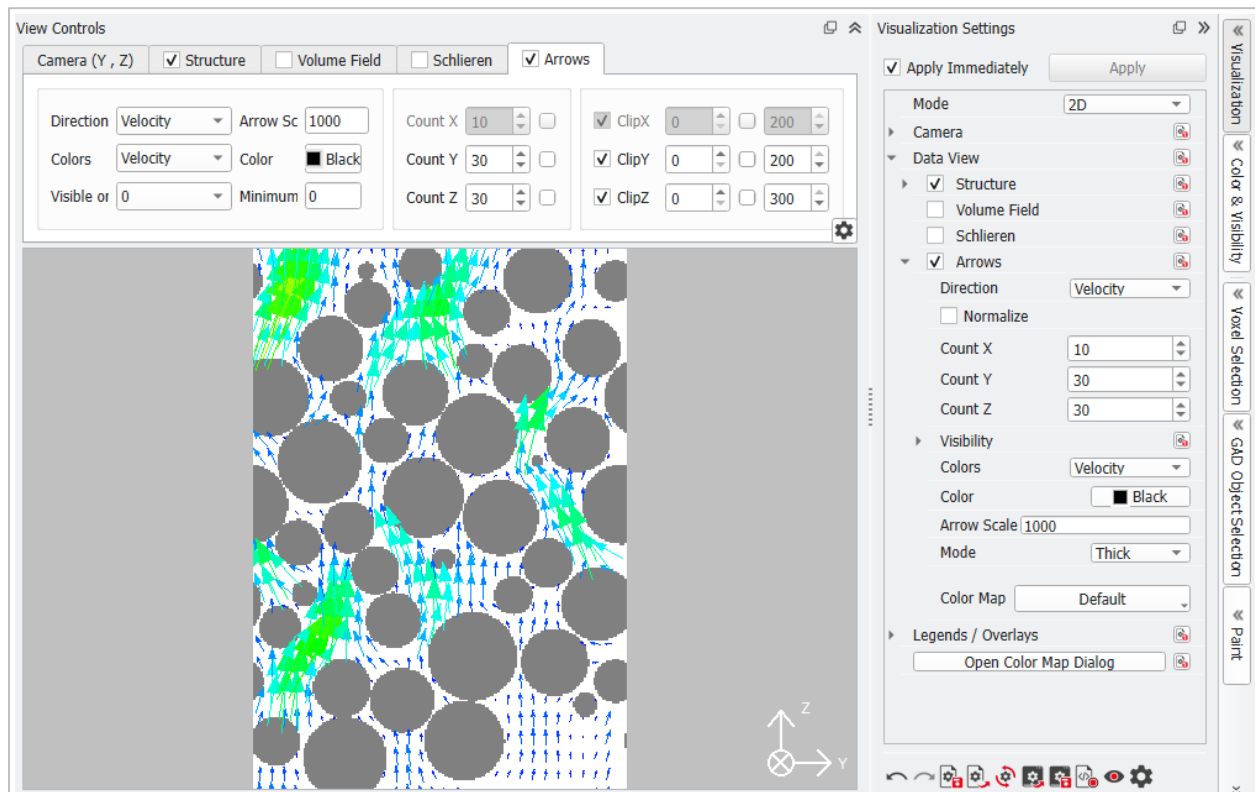
Arrows

Some volume fields as for example a flow field can be represented by an arrow field. Here, the properties of the fluid flow field are imaged at all positions of a regular grid that do not

belong to the solid. At each of these positions, the local orientation of the flow field is indicated by a vector and its length and color are coded for the fluid velocity.

Small vectors indicate a slow fluid velocity and long vectors indicate a high fluid velocity. Vectors are located with their middle in the center of the voxel for which they represent the flow velocity.

To see the arrows, check only **View** → ☒ **Arrows** in the menu bar, the **Arrows** tab in the **View Controls**, or **Data View** ► **Arrows** in the **Visualization Settings** dialog.

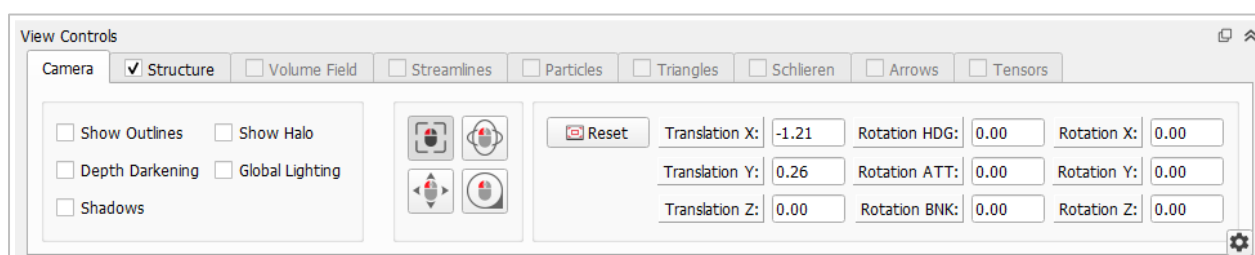


The options are the same as for the [3D visualization](#)⁹⁹.

1.2.2 Visualization 3D

A structure appears in the Visualization area after loading a file (*.gdt, *.gad). Then, select **View** → **3D Rendering** in the menu bar or click **3D** in the toolbar.

In the **View Controls**, above the structure displayed in 3D, up to nine tabs are selectable to control the 3D view settings: **Camera**, **Structure**, **Volume Field**, **Streamlines**, **Particles**, **Triangles**, **Schlieren**, **Arrows** and **Tensors**.

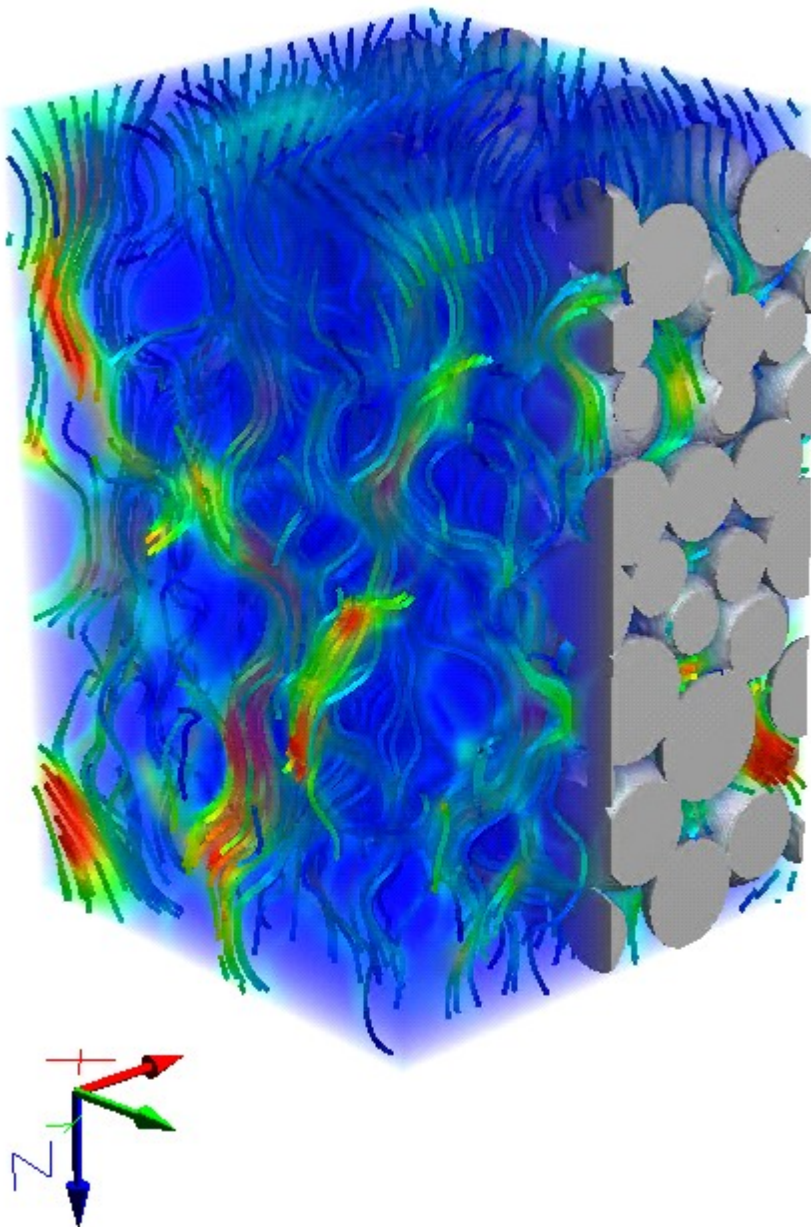


The grayed-out tabs correspond to the visualization of property data from result files. They become available when corresponding result files are loaded. Many of the analysis and all of the simulation and prediction modules write 3D result fields.

The following tabs are available in 3D view:

- [Camera](#)  46
Set the camera settings, as camera position and lighting.
- [Structure](#)  52
The Structure tab is always selectable and checked by default when a structure is loaded.
- [Volume Field](#)  58
Visualize many kinds of result data in 3D, for example velocity, ion concentration, diffusion flux and many more.
- [Streamlines](#)  68
Visualize the flow through your structure with streamlines or even animated streamlets.
- [Particles](#)  78
Visualize and animate the particles moving through your structure .
- [Triangles](#)  90
Visualize surface meshes in stl format in GeoDict.
- [Schlieren](#)  94
Visualize fluid flow fields through line integral convolution.
- [Arrows](#)  99
Visualize flow via arrows.
- [Tensors](#)  105
After performing a flexural test, you can visualize the tensor field.

For beautiful visualizations, also more fields can be visualized at once, by enabling the checkboxes of the corresponding tabs in the **View Controls**, or in the **Data View** section of the **Visualization Settings**, or in the **View** menu. To obtain a nice visualization with more than one of the data elements, for example by [clipping them in parts](#)  56, or if the volume field is turned on, enable **Transparency** in the **Volume Field** tab and adjust the [transparency settings](#)  121 in the [Color Map](#)  121 dialog. In the image below, the structure is clipped halfway in X-direction, the volume field is set to transparent, and the streamlines are enabled.



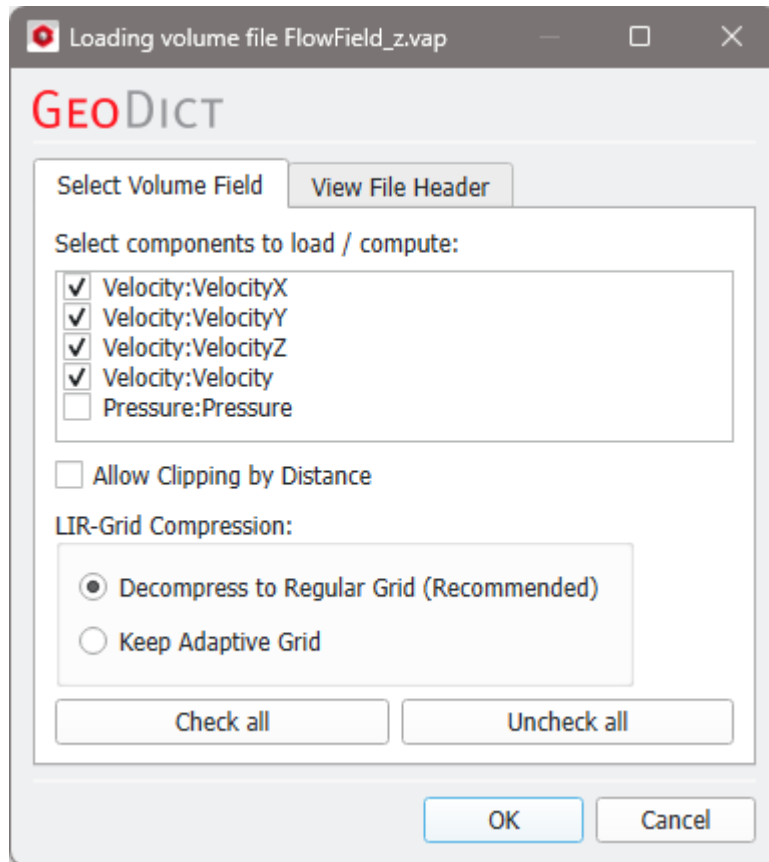
✓ Open result fields

There are two ways to open a result field. The first, and most common, way is loading them from the Result Viewer, as described in the Result Viewer User Guide. Another way is via the File menu (File - Load Volume File).

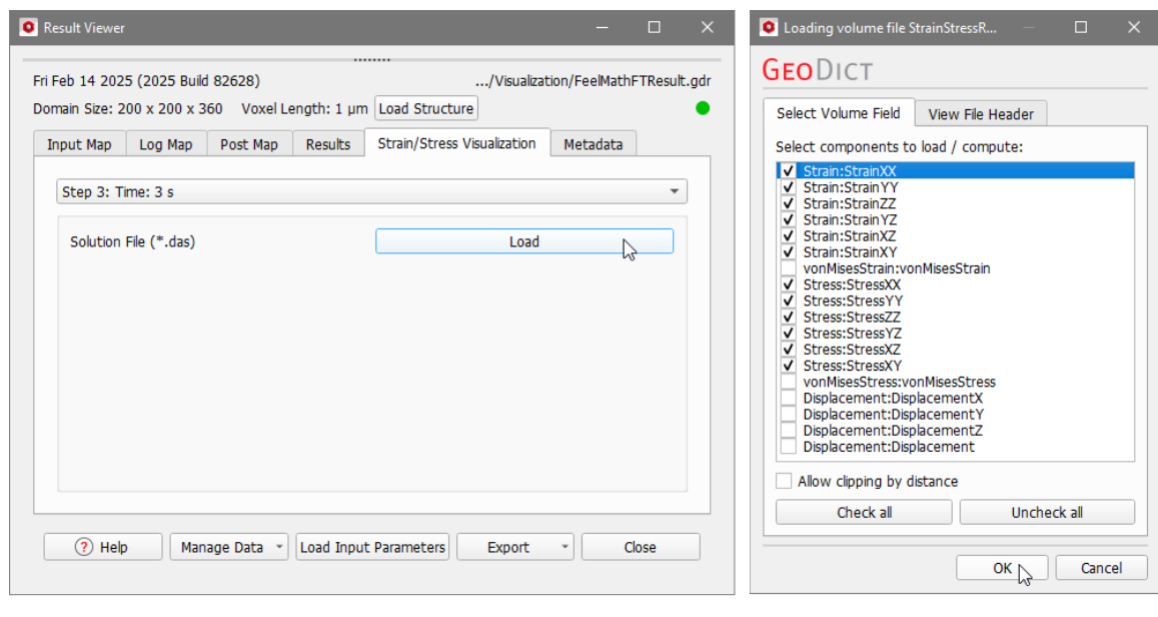
When loading a volume file a dialog shows up, where you can choose the components whose results you want to visualize.

As part of the visualization of the flow field, several GeoDict modules, e.g., FilterDict or FlowDict, can also display the streamlines left by the fluid when flowing through the structure, an arrow field for the flow, and the schlieren produced when the fluid flows through the structure (see [Schlieren imaging](#) for more info). For visualizing streamlines, schlieren, or arrows a complete vector field needs to be loaded, i.e., all components of

one image, for example all Velocity components for a flow simulation.



Simulating a flexural test using the ElastoDict module, it is possible to visualize the tensors. For this also a complete vector field must be loaded, e.g., all strain or all stress components.

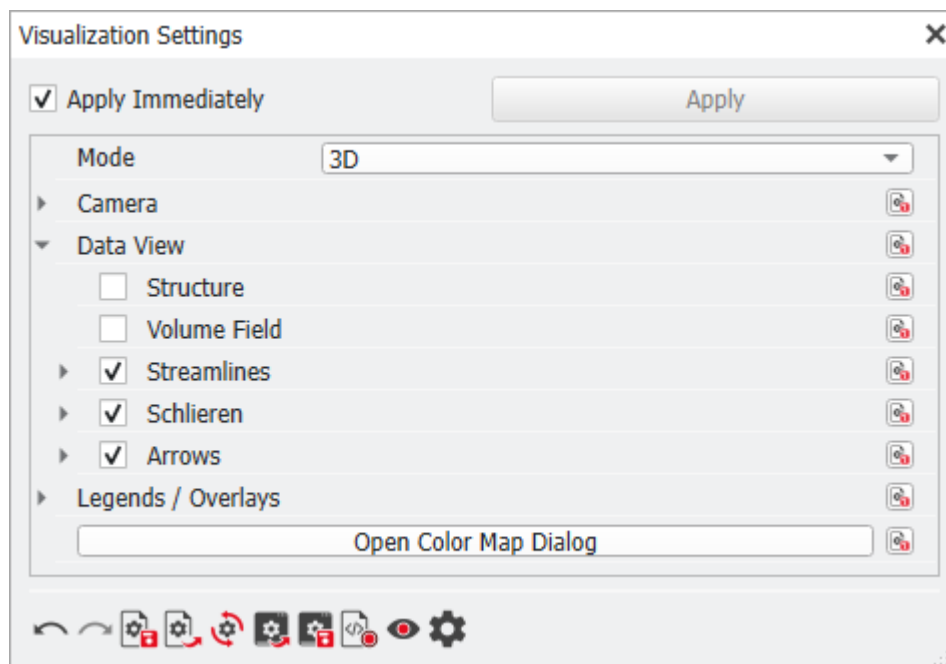


✓ Choose which components to visualize

The initial default visualization after loading can be optimized by modifying the parameters under the corresponding tabs in the **View Controls**, above the Visualization area.

Check these tabs to make their options selectable. At the same time, to only visualize the selected data components, turn off the visualization of the other components by unchecking the corresponding tabs. If needed, also switch off the display of the [structure](#) ⁵² model.

This can also be done through the [View](#) ⁵ menu or through the **Visualization Settings** dialog, as follows:

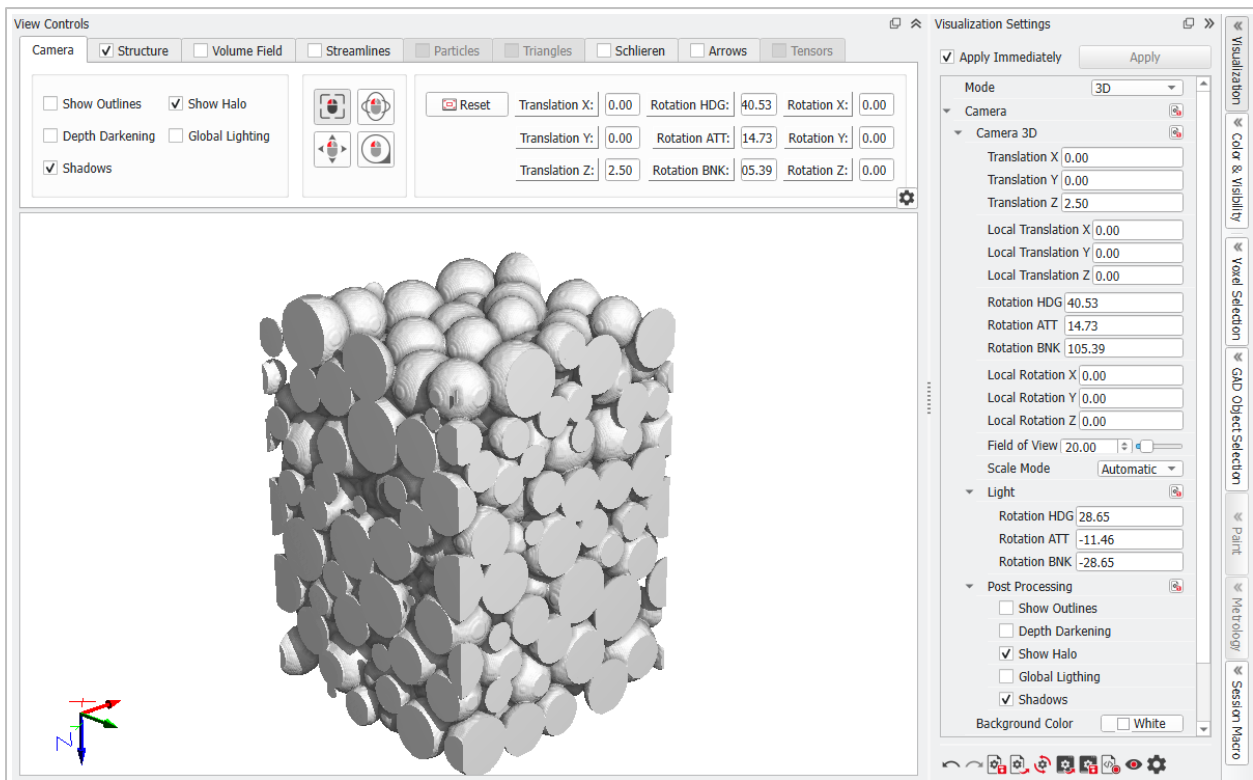


As soon as a component is checked, it is shown in the Visualization area, and the parameters needed to set their visualization can be modified.

Camera

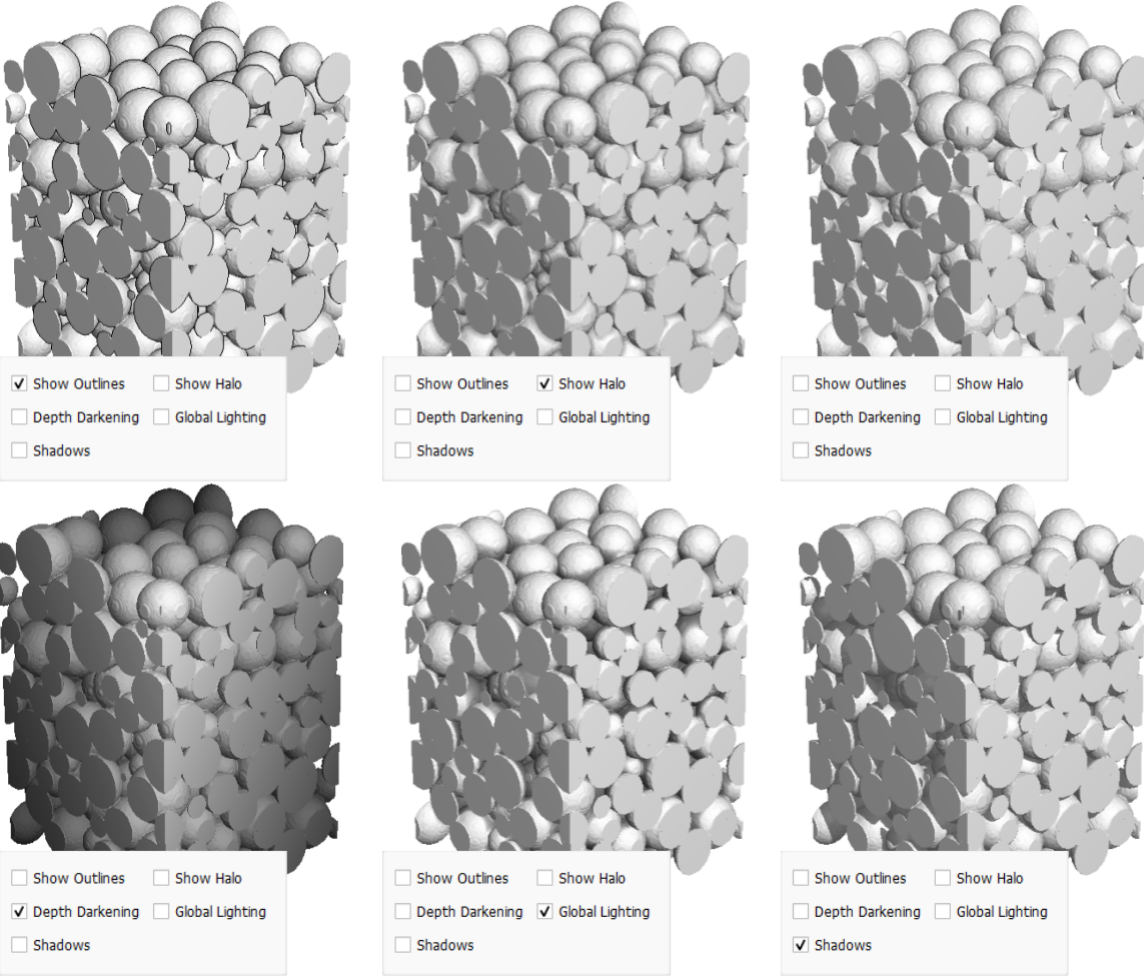
Set the camera settings, as camera position and lighting.

The **Camera** tab controls the light behavior, the camera angle from which the structure is viewed and the position of the structure in the Visualization area. Some additional options and most of the options from the **View Controls** can also be found in the **Visualization Settings** dialog. Expand the **Visualization Settings** dialog either by clicking the gear wheel icon in the bottom right of the Camera tab or by clicking **Visualization tab** in the sidebar and unfolding the **Camera** → **Camera 3D** section.



✓ Check Boxes for Visual Perception

The check boxes improve the visual perception of the structure, as follows:



They can also be found on the right in the **Visualization** tab of the sidebar under **Post Processing**.

✓ Mouse Modes

In the Visualization area, with **3D Rendering** view the mouse can be used to move the structure. Zooming always works while scrolling the mouse wheel. The function of the mouse buttons can be changed, by choosing one of the icons:

	Default mode. Rotate the image by keeping the left mouse button pressed and moving it. Zoom by keeping the right button pressed and simultaneously moving the mouse forward. By moving the mouse backward, the image zooms out. Use the middle mouse button to pan and move the structure.
	Rotate the image by keeping one of the mouse buttons pressed and moving it.
	Pan the structure by keeping one of the mouse buttons pressed and moving it.



Zoom in and out by keeping one of the mouse buttons pressed and moving it up and down.

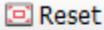
✓ Translation and Rotation

The controls **Translation X**, **Translation Y**, and **Translation Z** determine the position of the structure in the Visualization area. By changing these values in one axis direction, the structure is translated in this direction by the specified amount. When panning the structure with the middle mouse button or using the panning icon , the values in the translation parameter boxes change simultaneously.

The controls **Rotation HDG**, **Rotation ATT**, and **Rotation BNK** define the rotation of the structure in the Visualization area. To reproduce a specific view of a 3D structure, pre-recorded values may be manually entered. The acronyms stand for Heading, Altitude, and Banking and correspond to terminology for aircraft control simulation. When rotating the structure with the left mouse key the values are updated automatically.

Additionally, using the controls **Rotation X**, **Rotation Y**, and **Rotation Z** the structure can be turned around the specified axis. In the **Visualization Settings** dialog they are called **Local Rotation**.

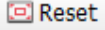
Clicking the **Reset View** button, the 3D image in the **View Controls** returns to the 0.00 position for all translation controls and rotation controls.

	Translation X:	0.00	Rotation HDG:	40.53	Rotation X:	0.00
	Translation Y:	0.00	Rotation ATT:	14.73	Rotation Y:	0.00
	Translation Z:	0.00	Rotation BNK:	105.39	Rotation Z:	0.00

Additional to the **Translation** parameters also **Local Translation** can be determined for the three directions from the **Visualization Settings** dialog. The local translation depends on the structure's rotation, since it always takes place in direction of the geometric axes, displayed in the bottom left of the visualization area.

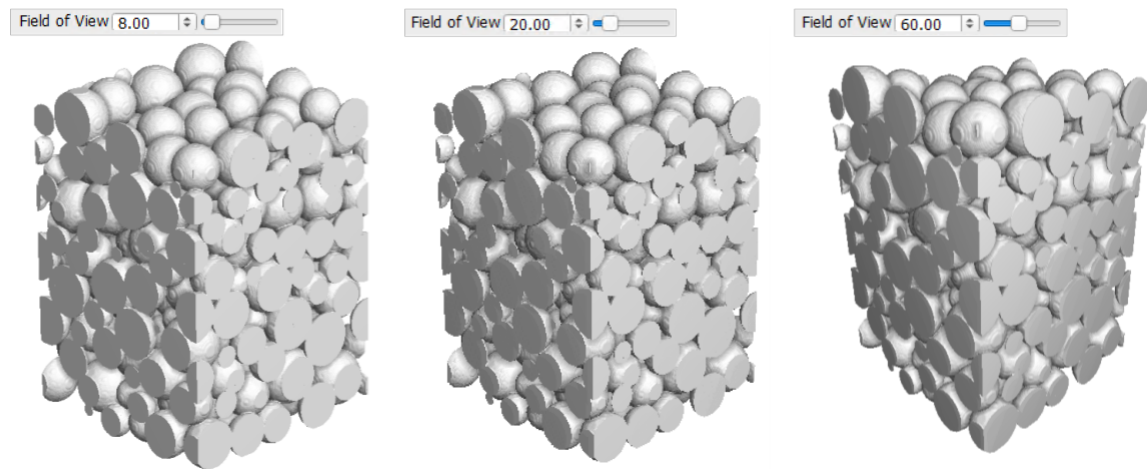


Know how! It is also possible to change the translation and rotation values by clicking on the respective button and dragging the mouse left to decrease the value or right to increase the value.

	Translation X:	0.00	Rotation HDG:	0.00	Rotation X:	0.00
	Translation Y:	0.33	Rotation ATT:	0.00	Rotation Y:	0.00
	Translation Z:	0.00	Rotation BNK:	0.00	Rotation Z:	0.00

✓ Field of View

The **Field of View** defines the viewing arc in degrees.



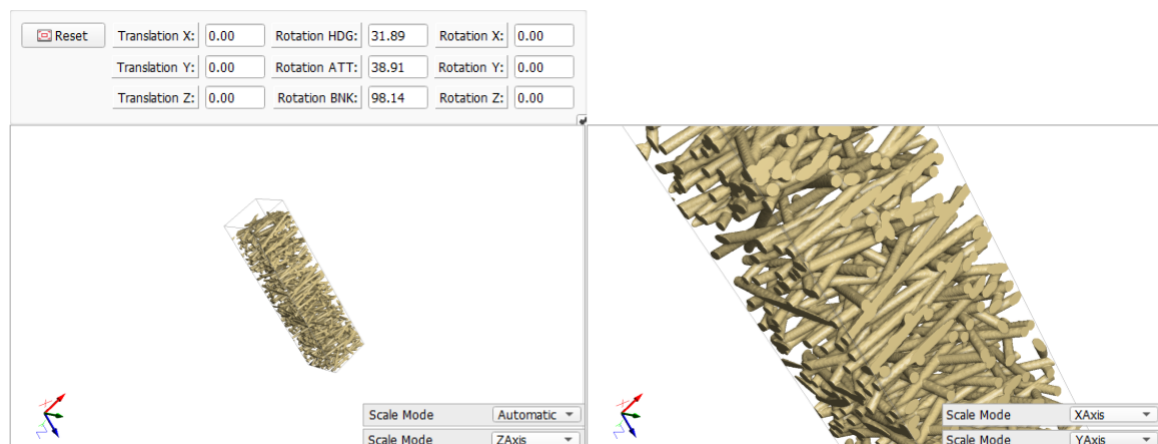
✓ Scale Mode

The **Scale Mode** is another option for zooming. Higher values zoom out and smaller values zoom in.

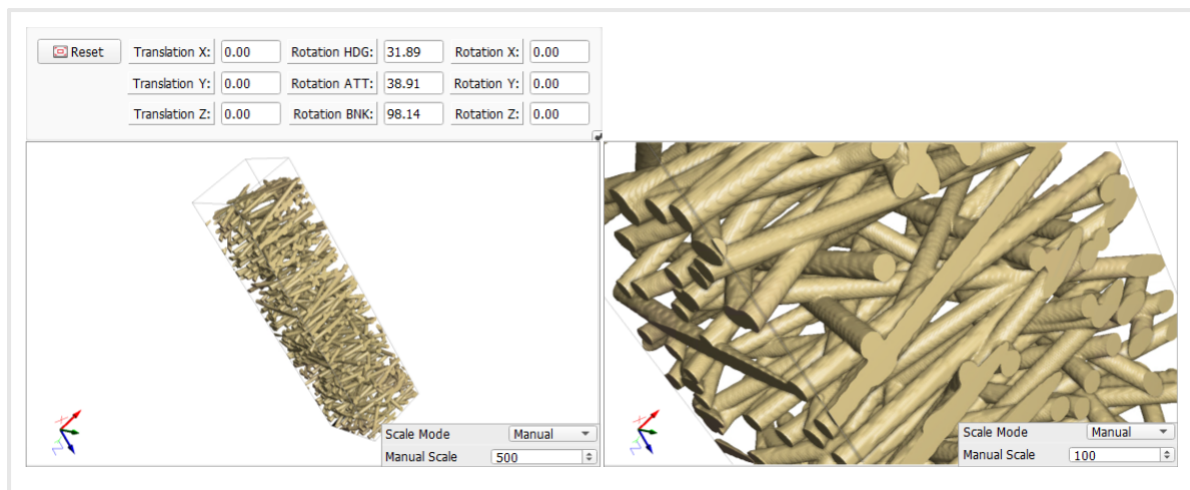
If setting the **Scale Mode** to **Automatic** it chooses the axis with the highest number of voxels as a scale value. This way, always the complete structure is visible, when having all [translation](#)⁴⁹ values set to 0.

Instead, you can choose **X-**, **Y-** or **Z-Axis** from the pull-down menu. Choosing shorter domain directions zooms in. Select **Manual** to enter a **Manual Scale**.

In the example below, the structure has the same number of voxels in X- and Y-direction (200) but has a lot more voxels in Z-direction (810). Thus, setting the **Scale Mode** to **Automatic** or **Z-Axis** results in the same zoom, where the structure is completely visible, whereas **X-** and **Y-Axis** result in the same zoomed in visualization.

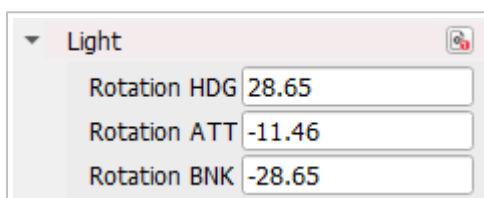


In the next example, a manual scale is used.



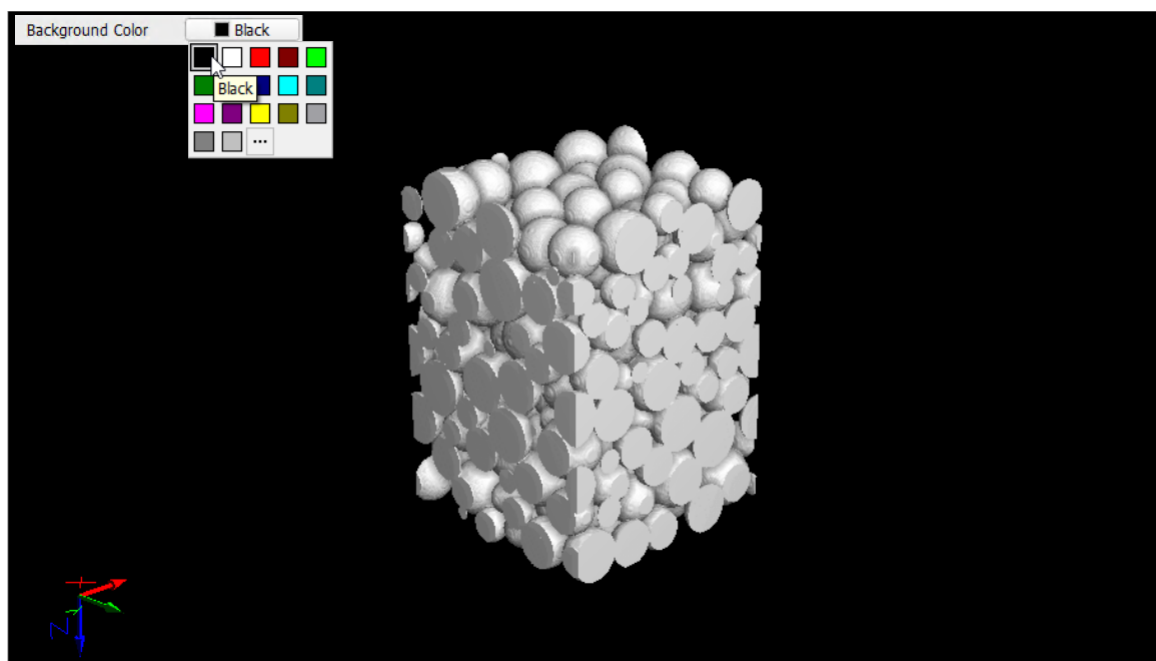
✓Light

The **Light** rotation parameters control the position of the light source. The values also change automatically when changing the [light position](#)¹⁸.



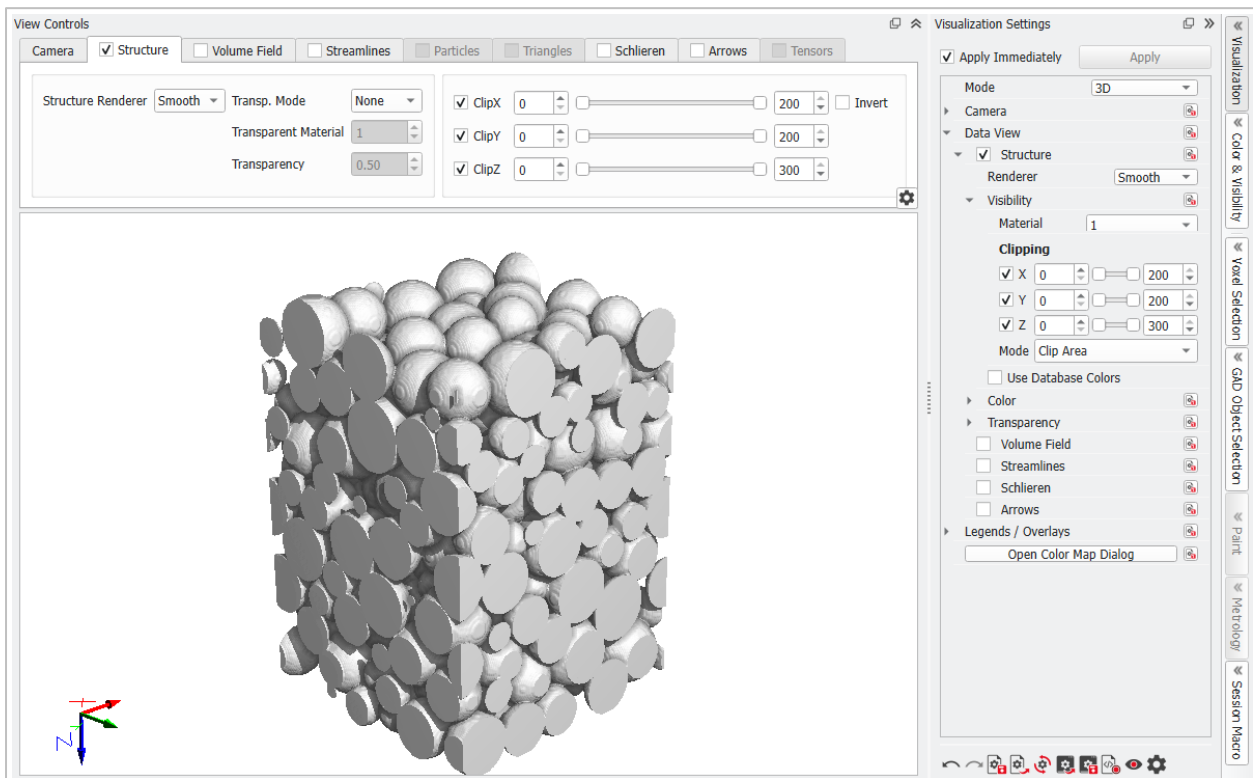
✓Background Color

Define the color of the background in the visualization area.



Structure

The **Structure** tab controls the visibility of the structure. Some additional options and all the options from the **View Controls** can also be found in the **Visualization Settings** dialog. Expand the **Visualization Settings** dialog either by clicking the gear wheel icon in the bottom right of the Structure tab or by opening the **Visualization tab** in the sidebar and unfolding the **Data View → Structure** section.

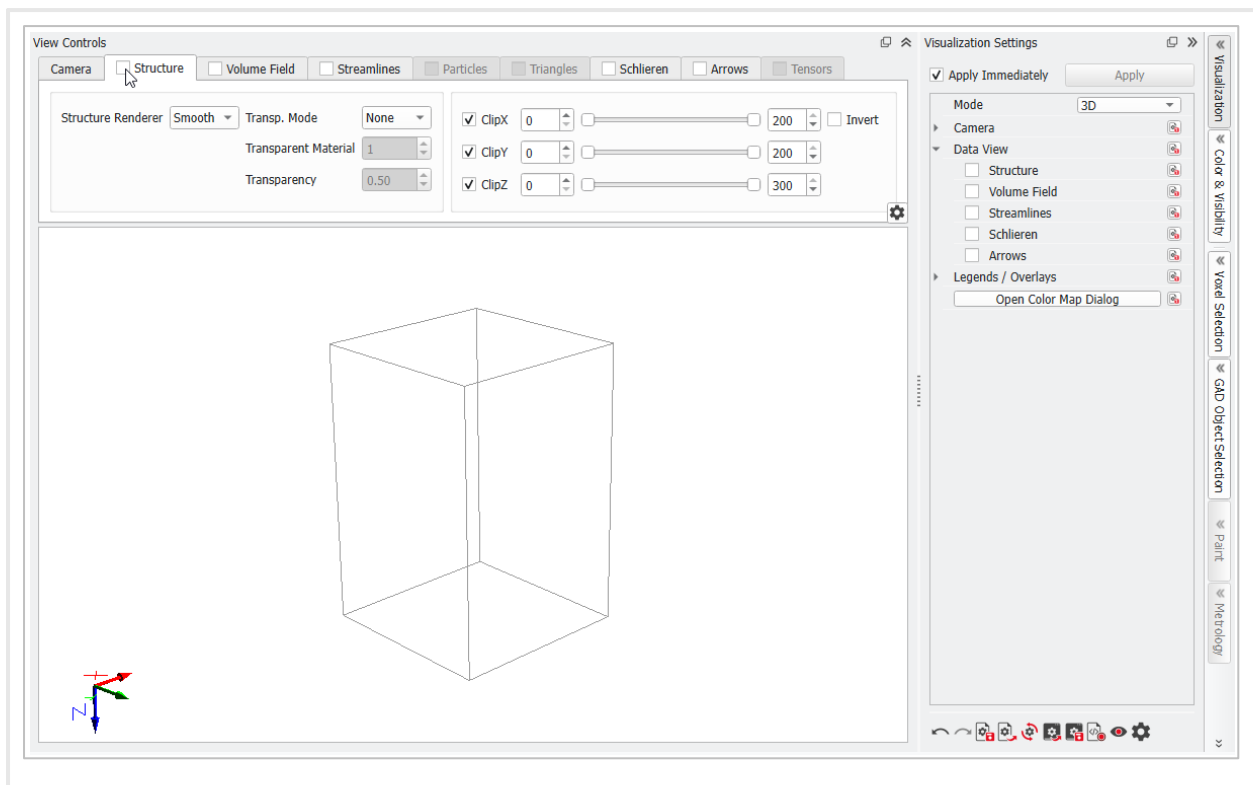


✓ Show Structure

The **Structure** tab is always selectable (when a structure is loaded) and it is checked by default.

By unchecking the **Structure** tab, the structure disappears from the Visualization area because the display is disabled. Only the **Material Legend**, the **Bounding Box** showing the domain, and the **Axes** remain visible.

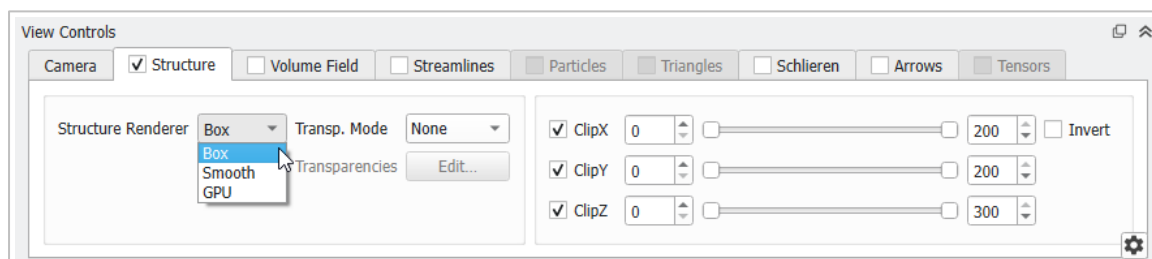
The same effect is achieved by unchecking ☐ **Structure** through the [View](#)  menu or the **Visualization** tab.



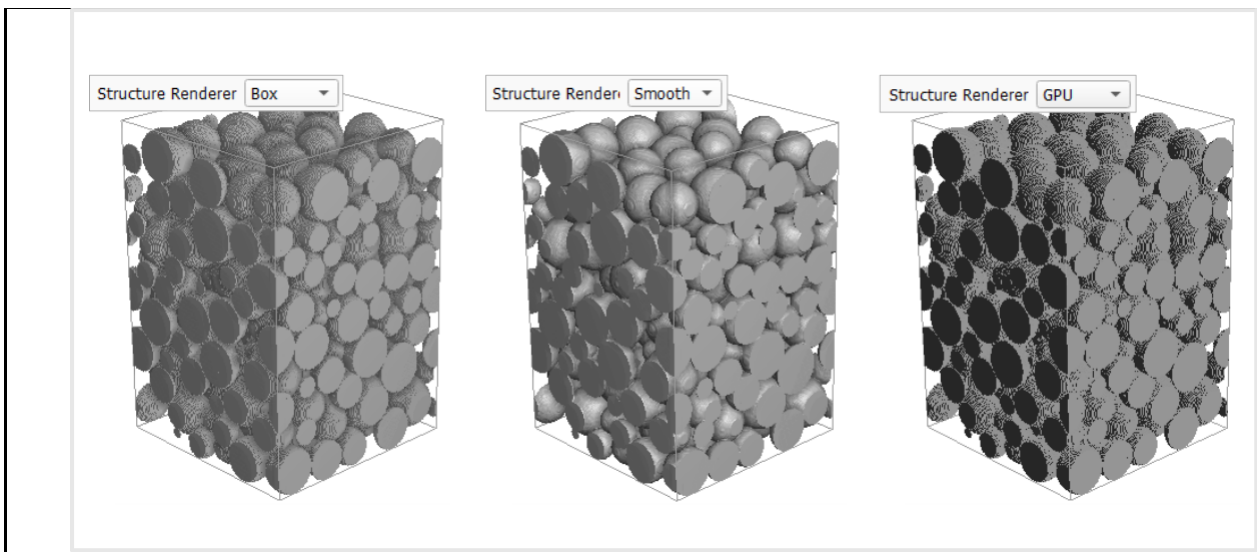
✓ Structure Renderer

In the **Structure** tab, it is possible to switch between the three 3D **Structure Renderers**:

- **Box**: show the voxel structure rendered on the CPU.
- **Smooth**: smooth the voxel surfaces.
- **GPU**: show the voxel structure rendered on the GPU.

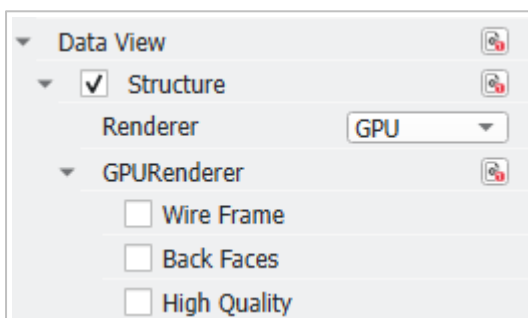


Alternatively, the user can switch between the three **Structure Renderers** through the checkboxes in the [View](#) menu (**View** → **3D Structure Renderer**) or the **Renderer** pull-down menu in the **Visualization Settings** dialog.



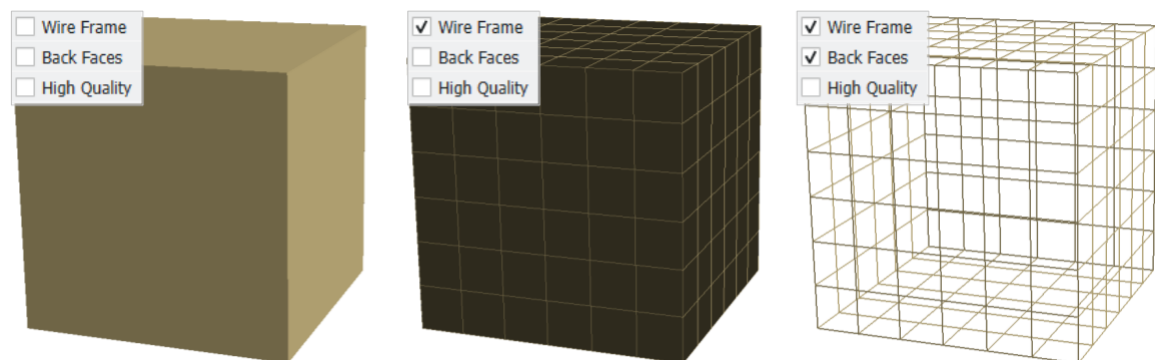
✓ GPU Renderer Options

If **GPU** is selected for **Structure Renderer**, the transparency functions disappear in the Structure tab, but instead in the **Visualization Settings** the **GPU Renderer** section becomes available. Three options can be chosen.



- The structure can be visualized as a **Wire Frame**.
- If **Wire Frame** is selected, visualize the **Back Faces** of the structure while the structure itself becomes invisible. You can then see the complete wire frame of the structure.
- Checking **High Quality** increases the quality in trade-off to performance.

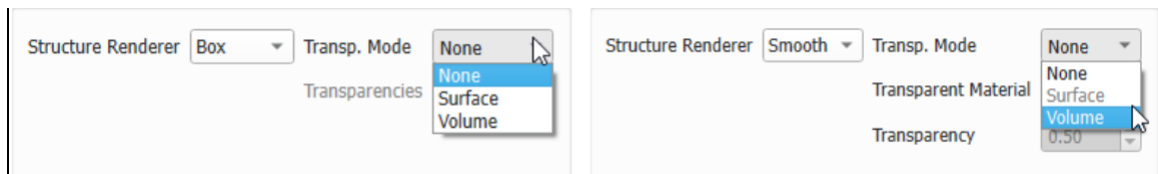
In the example below, the structure only consists of one cube.



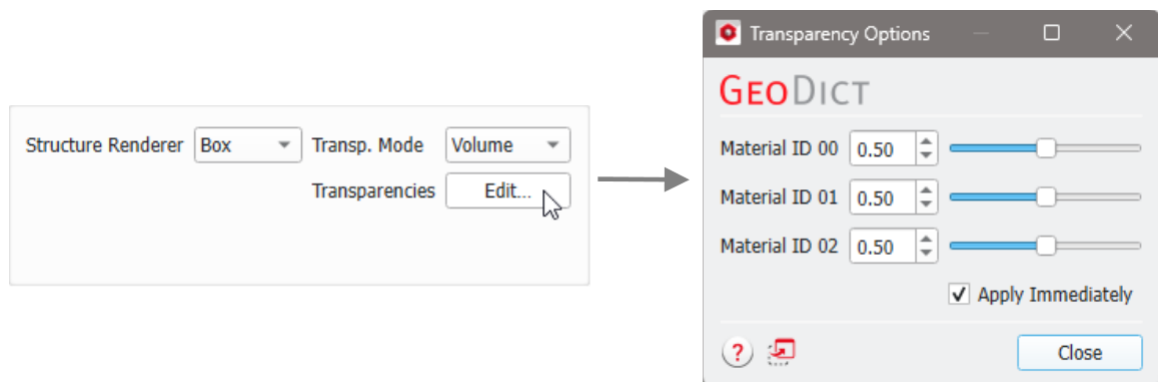
✓ Transparency

While using the **Box** renderer, transparency can be applied to the displayed structure by selecting **Surface** or **Volume** as the **Transparency Mode**, instead of the default **None**.

For the **Smooth** Renderer, transparency is applied after selecting **Volume** (Surface is not selectable)



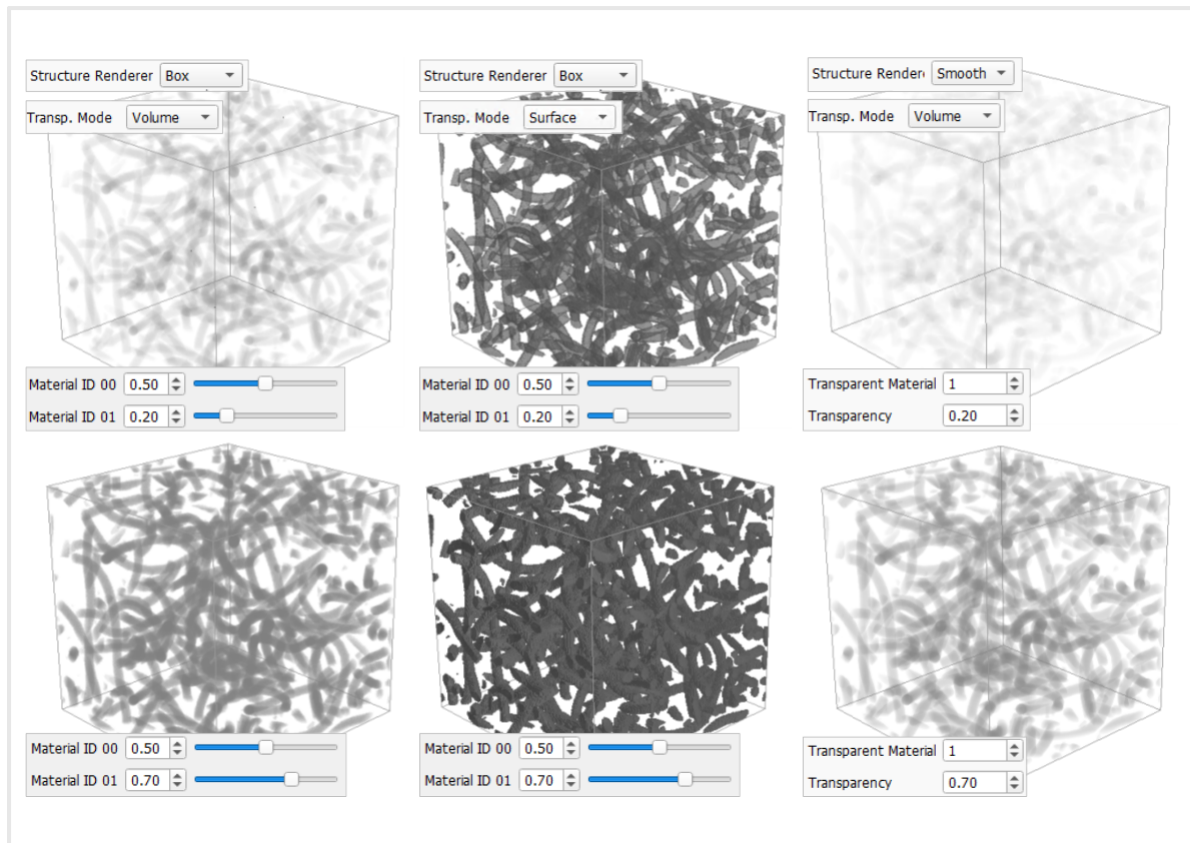
For the box renderer, clicking the **Transparencies' Edit...** button opens the **Transparency Options** dialog. Here, the user can define the transparency level for all materials in the structure. When unchecking **Apply Immediately**, confirm the transparency options by clicking **Apply**. Otherwise, they are applied automatically. Close the dialog by clicking **Close**. Additionally, the transparency options are also accessible through the **Visualization Settings** dialog.



The **Transparency** for the **Smooth** renderer is similar as for the **Box** renderer, but there is an essential difference: Transparency can be only applied to a *single material* in the displayed structure.

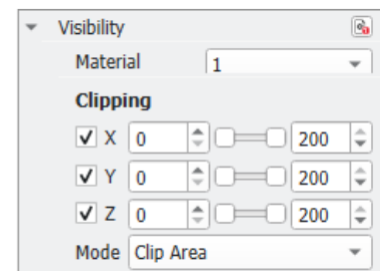
After selecting **Volume** as the **Transparency Mode**, select the material to which transparency is applied (**Transparent Material**) and the level of transparency. The smaller the value, the more transparency is applied, as shown in the example.

In this example, Material ID 00 is set to invisible, but the transparency of Material ID 01 (solid) can be defined with values between 0 and 1. The smaller the value, the more transparency is applied.

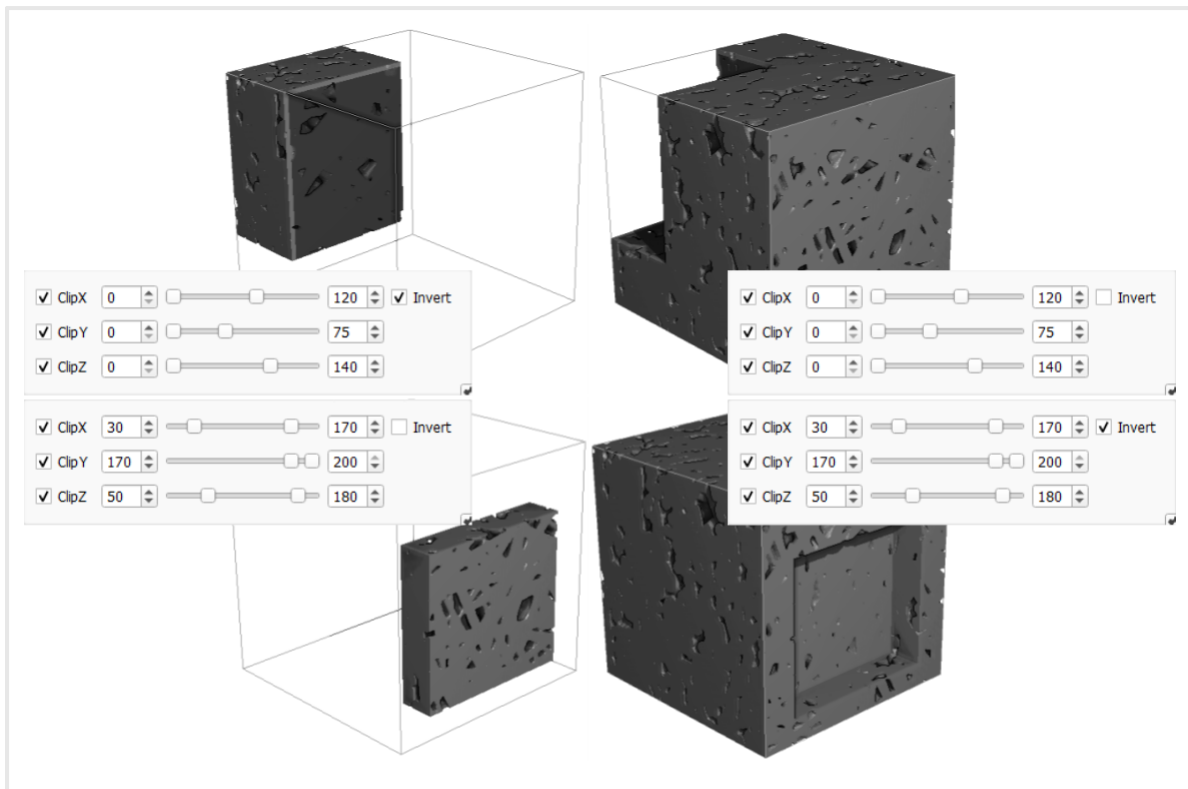


✓ Structure Clipping

On the right side of the **Structure** tab and in the **GUI Sidebar**, **Clipping** of the values **ClipX**, **ClipY**, and **ClipZ** (or **X**, **Y**, and **Z**) allows cutting the displayed geometry in all three spatial directions with the sliders or by direct input.

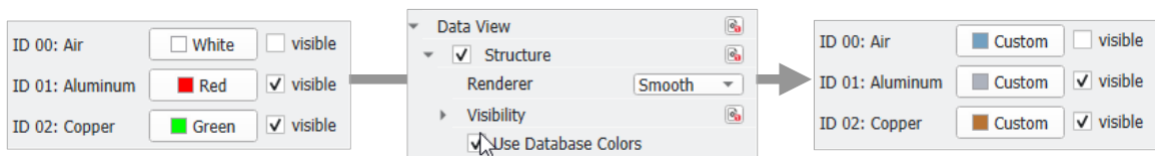


Additionally, after defining a clipping area with the sliders or direct input, you can limit this as the area of interest by checking **Invert** in the **View Controls** or selecting **Invert Clip Area** for **Clipping Mode** in the **Visualization Settings** dialog. Only the area defined by the clipping values remains visible in the Visualization area.

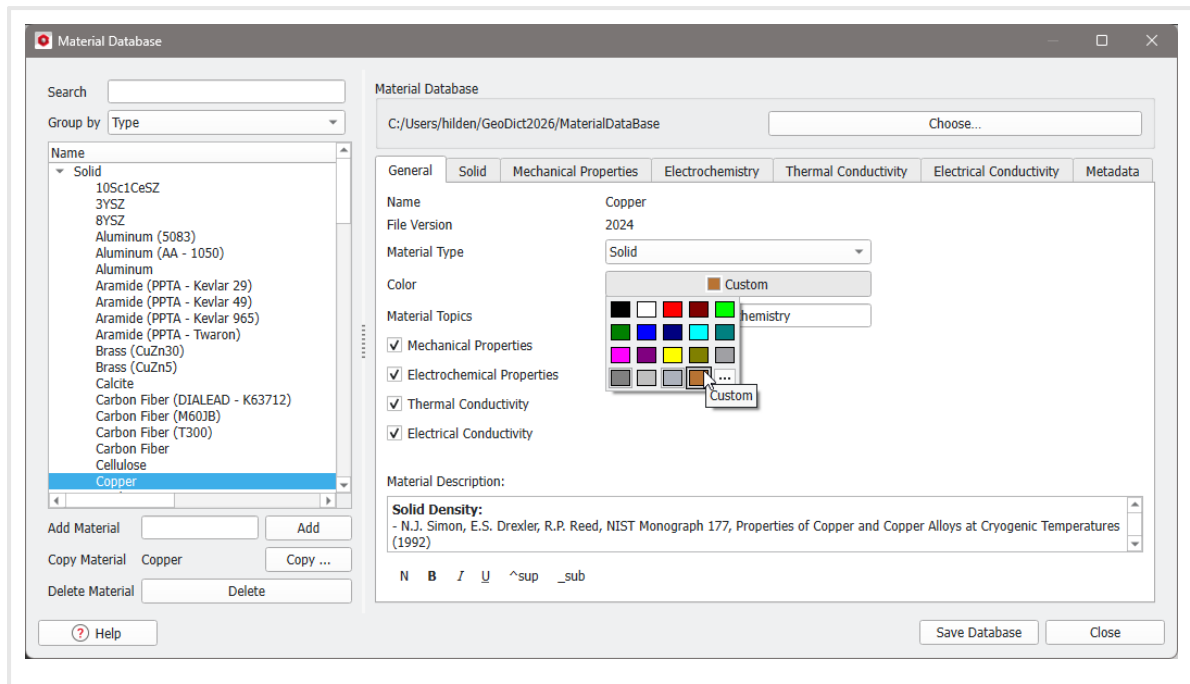


✓ Use Database Colors

Enabling the checkbox **Use Database Colors** in the **Structure** section of the **Visualization Settings** dialog sets the colors of the constituent materials according to their defined color from the GeoDict Material Database. If the material is set to **Manual**, the color is not changed. Changing the color afterwards disables the checkbox.



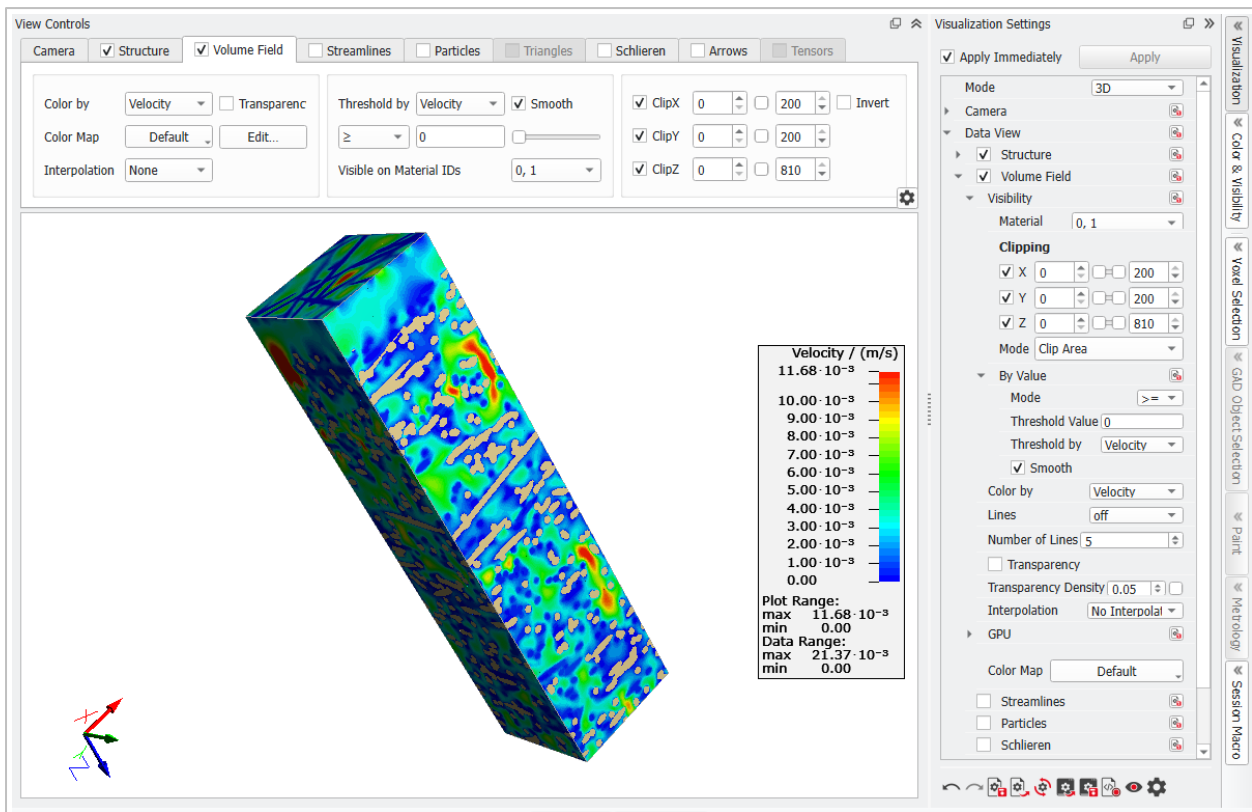
The material color can be defined in the GeoDict Material Database accessible by selecting **Settings - Open & Edit Material Database** from the menu bar or by clicking the corresponding icon  in the toolbar. Select the material to change on the left, choose the desired color under the **General** tab on the right and click **Save Database**.



Volume Field

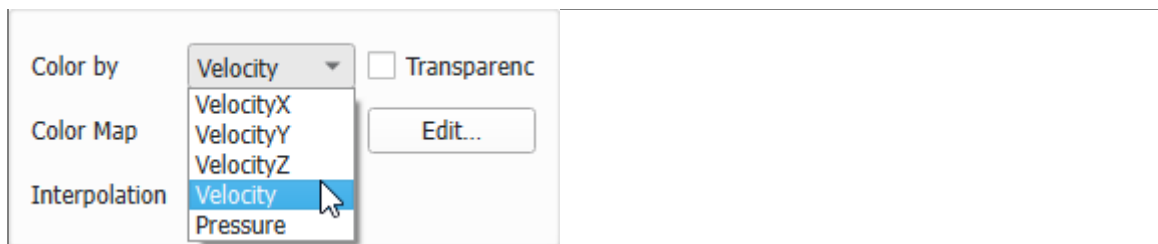
Visualize many kinds of result data in 3D, for example velocity, ion concentration, diffusion flux and many more.

The **Volume Field** tab controls the visibility of the volume field. Some additional options and all the options from the **View Controls** can also be found in the **Visualization Settings** dialog. Expand the **Visualization Settings** dialog either by clicking the gear wheel icon in the bottom right of the Volume Field tab or by opening the **Visualization tab** in the sidebar and unfolding the **Data View → Volume Field** section.



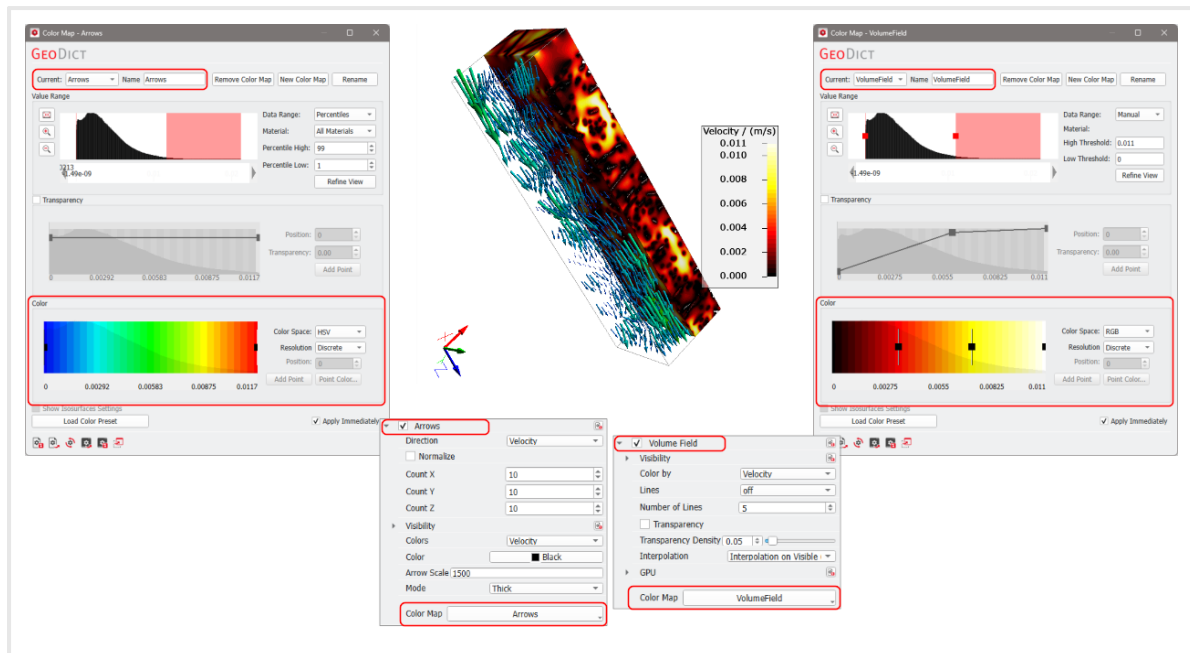
✓ Color by

The data of a particular flow field component can be chosen for display from the **Color By** pull-down menu (here it is Velocity).



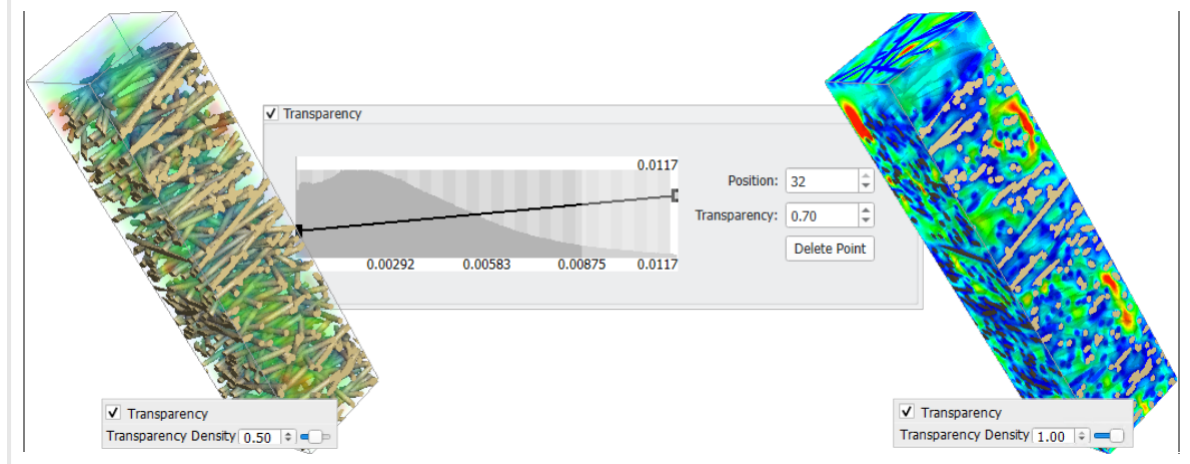
✓ Color Map

Define different color maps in the [Color Map](#)^[121] dialog. From the pull-down menu for **Color Map** in the **Visualization** tab on the right you can then select between your [custom color maps](#)^[122]. You can define and select a different color map for each of the components. For example, visualize arrows with the jet [color preset](#)^[125] and the volume field with the hot preset:



✓ Transparency and Transparency Density

Transparency can be enabled or disabled in the **Color Map** dialog, from the **View Controls** or the **Visualization Settings** dialog. In the **Color Map** dialog the transparency can individually be set as explained [here](#)¹²⁴. In the **Visualization Settings** dialog, also the overall transparency **Density** can be changed.



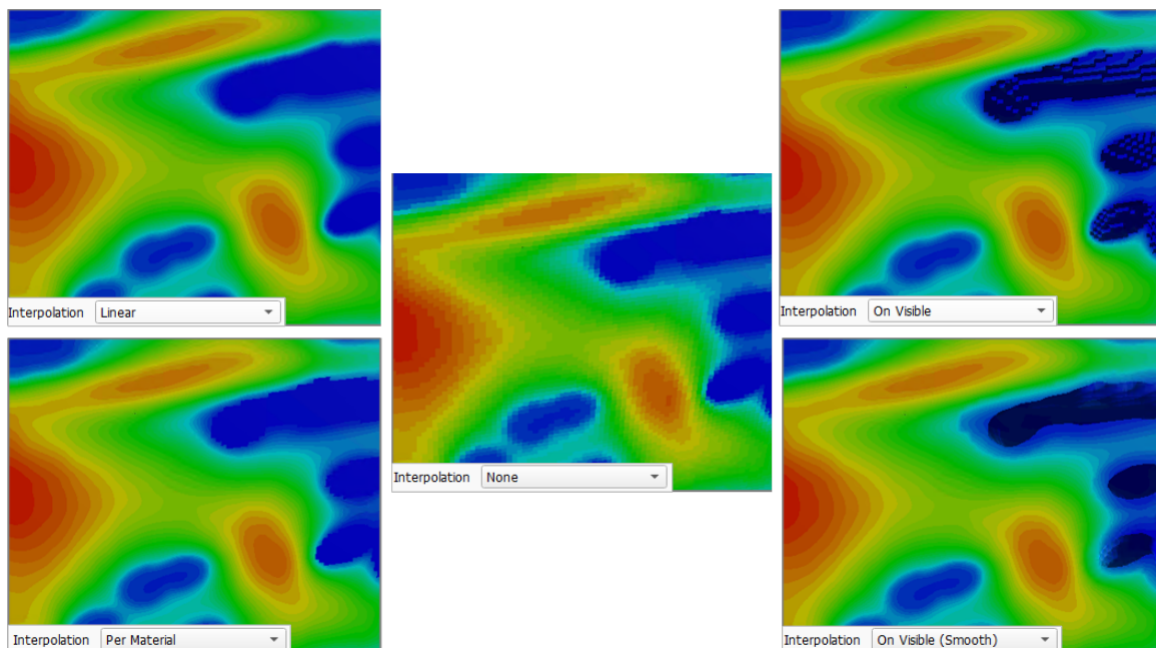
✓ Interpolation

In the **View Controls** and in the **Visualization Settings** dialog, several options are available for **Interpolation**:

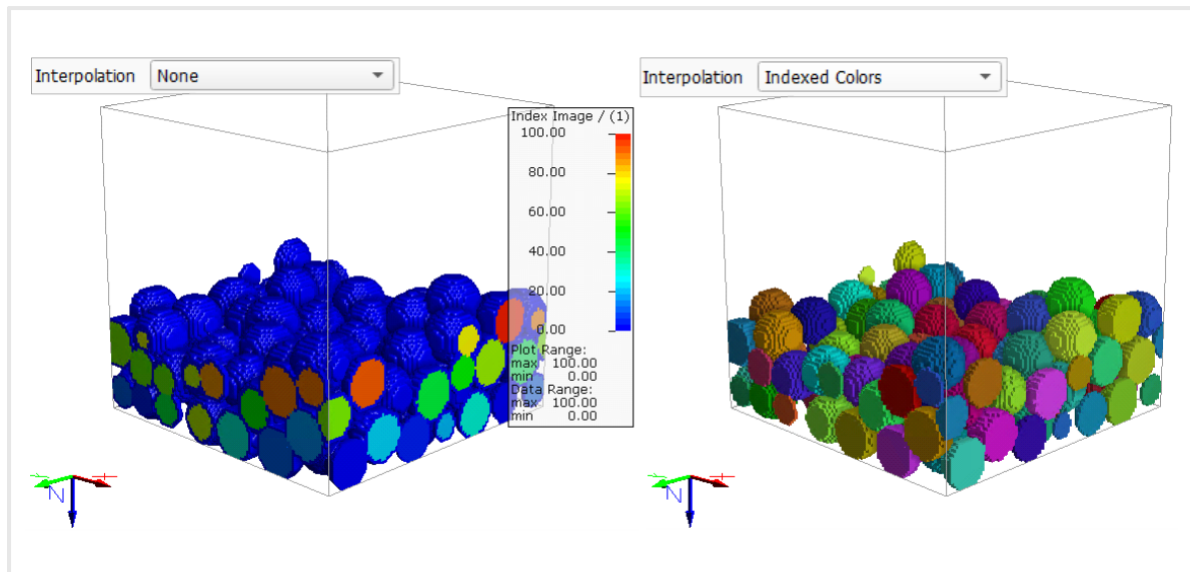
- **None:** For most volume fields, by default, the colors of the volume field are not interpolated and just the raw voxel values are shown.
- **Linear:** Select to interpolate the colors linearly between surrounding voxel centers.

- **Indexed Colors:** Only available for integer fields, as for example *.g32 files. For integer fields, usually this is the default setting. If selected, the different index values have distinguishable colors.
- **On Visible:** Choose to interpolate the colors linearly between surrounding voxels, while not interpolating into invisible areas. Thus, color fringes are removed. In the example below, material ID 01 is selected as invisible. Thus, the flow field is not visualized in the glass fibers. How to choose on which material IDs the volume field is visible is described [here](#).
- **On Visible (Smooth):** Interpolation works similar to interpolation **On Visible**, but generates a smooth surface.
- **Per Material:** Select to linearly interpolate only within the materials to preserve sharp edges.

Below the different interpolation modes are shown for the flow field from the FilterDict simulation:

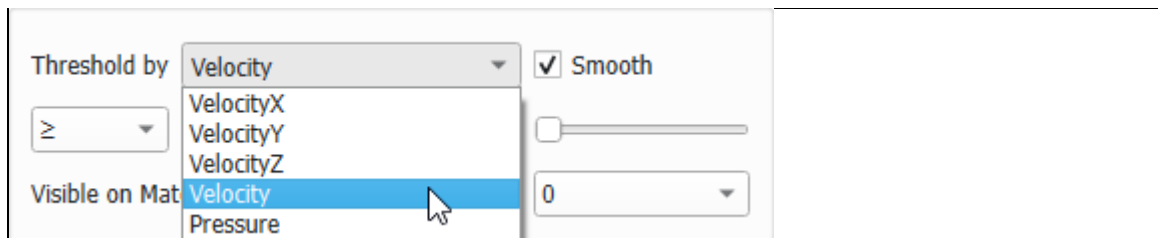


Below find an example for interpolation on an index field:

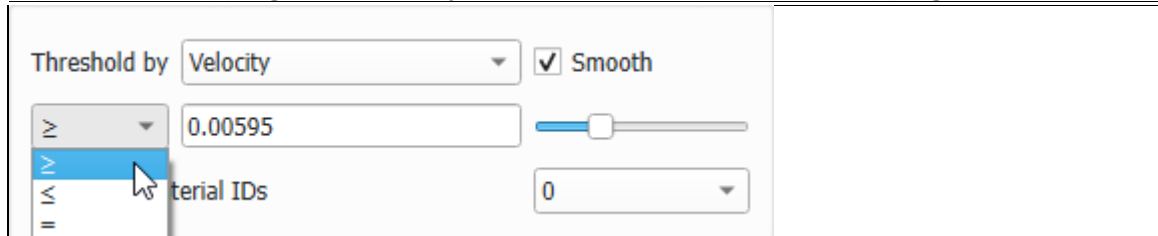


✓ Threshold

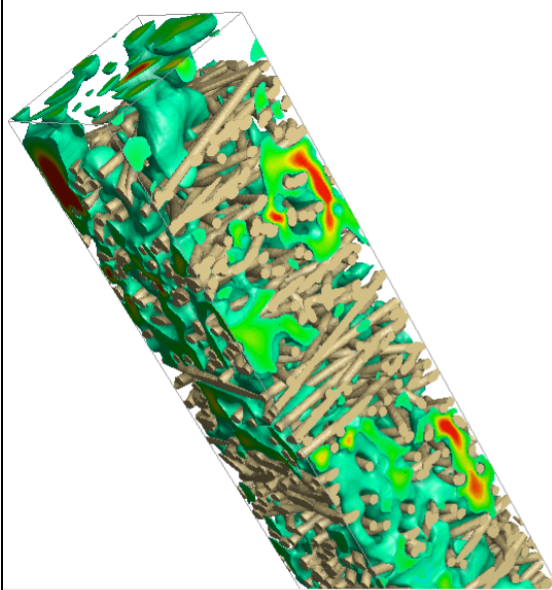
The **Threshold By** pull-down menu in the center panel of the **View Controls** and, the **Volume Field ► Visibility ► By Value** option in the **Visualization Settings** dialog, allows a clipping that is not related to the spatial coordinates but to the data range. The **Threshold By** menu has the same options as **Color** (here VelocityX, VelocityY, VelocityZ, Velocity, and Pressure). You can clip the current fields by values from another field, for example to view the pressure in certain velocities.



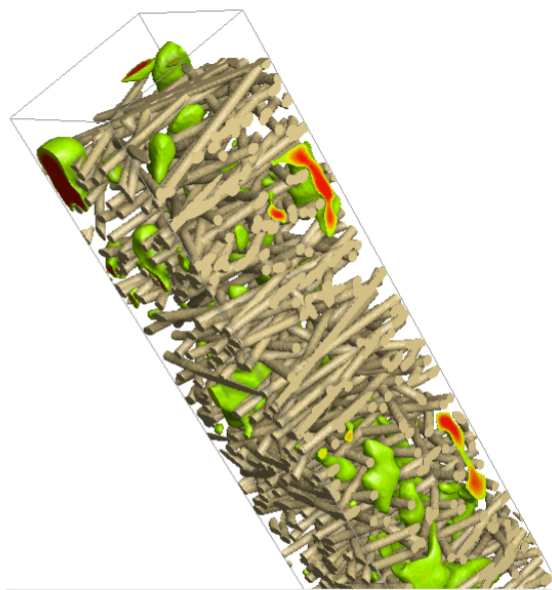
Enter the value defining the clipping range below or move the slider. The pull-down menu on the left determines the clipping direction. By default, the threshold mode (or **By Value ► Mode**) displays values greater than or equal to the clipping value, but the user may choose, from the pull-down menu, to display only values smaller than or equal to the clipping value or only values that are equal to the clipping value.



For example, only flow velocities above $0.004 \text{ (} 4 \cdot 10^{-3} \text{) m/s}$ or then above $0.008 \text{ (} 8 \cdot 10^{-3} \text{) m/s}$ are visualized.

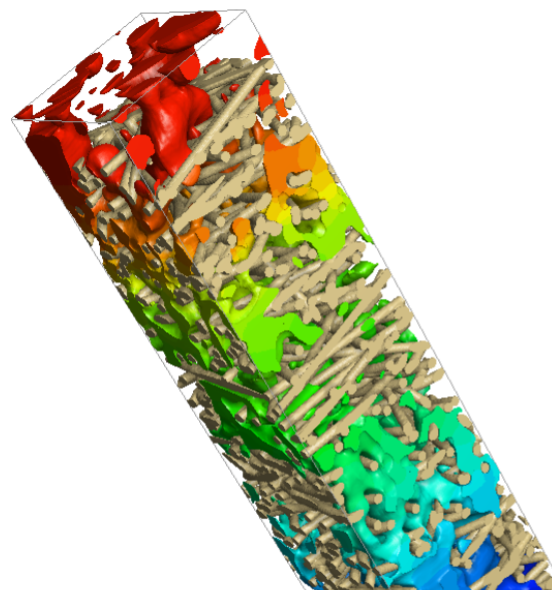


Threshold by Velocity ☒ Smooth
 $\geq$ 0.004
 Visible on Material IDs 0, 1

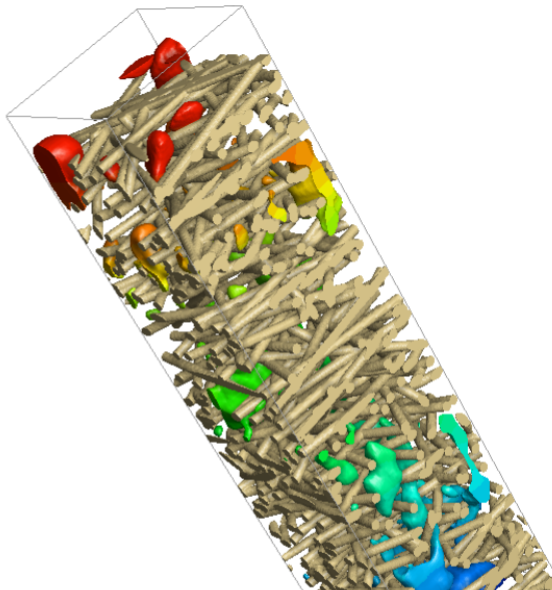


Threshold by Velocity ☒ Smooth
 $\geq$ 0.008
 Visible on Material IDs 0, 1

Observe how the same clipping is done while the pressure field is visualized.

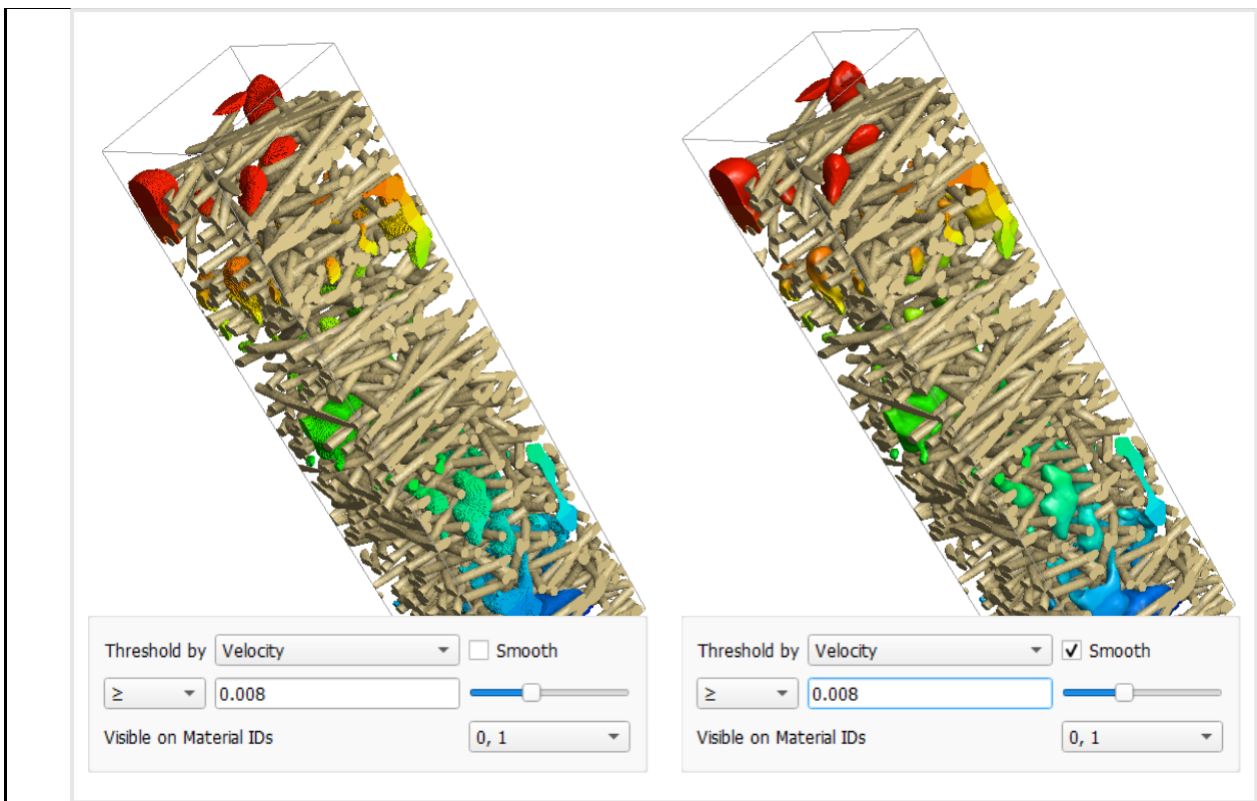


Threshold by Velocity ☒ Smooth
 $\geq$ 0.004
 Visible on Material IDs 0, 1



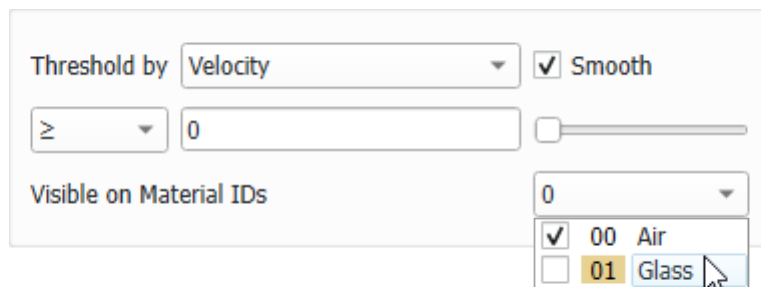
Threshold by Velocity ☒ Smooth
 $\geq$ 0.008
 Visible on Material IDs 0, 1

Checking **Smooth** smooths the voxel surface of the volume field.



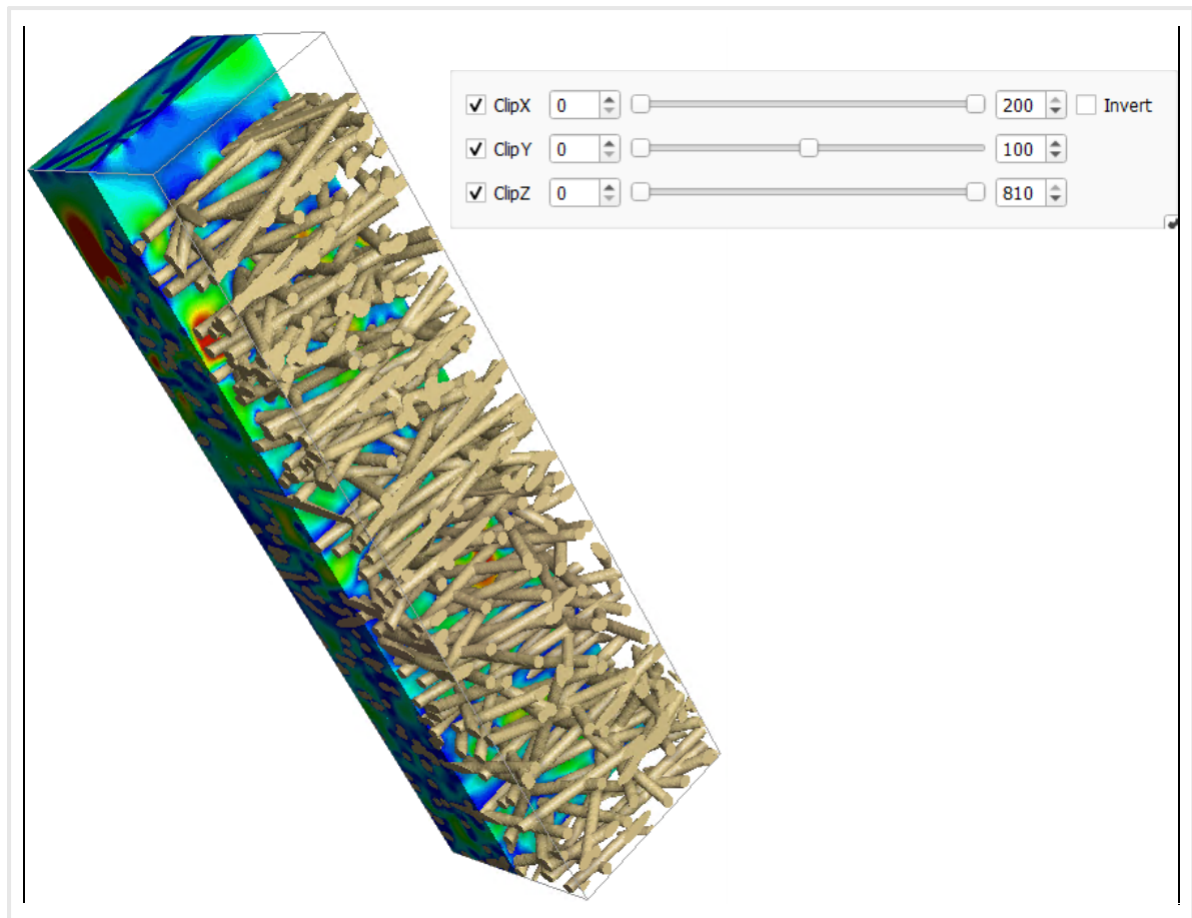
✓ Visible on Material IDs

From the pull-down menu **Visible on Material IDs** select in which material IDs the volume field is visualized. If for example, a flow field is loaded, no flow happens in the solid materials, and thus they can be unchecked.



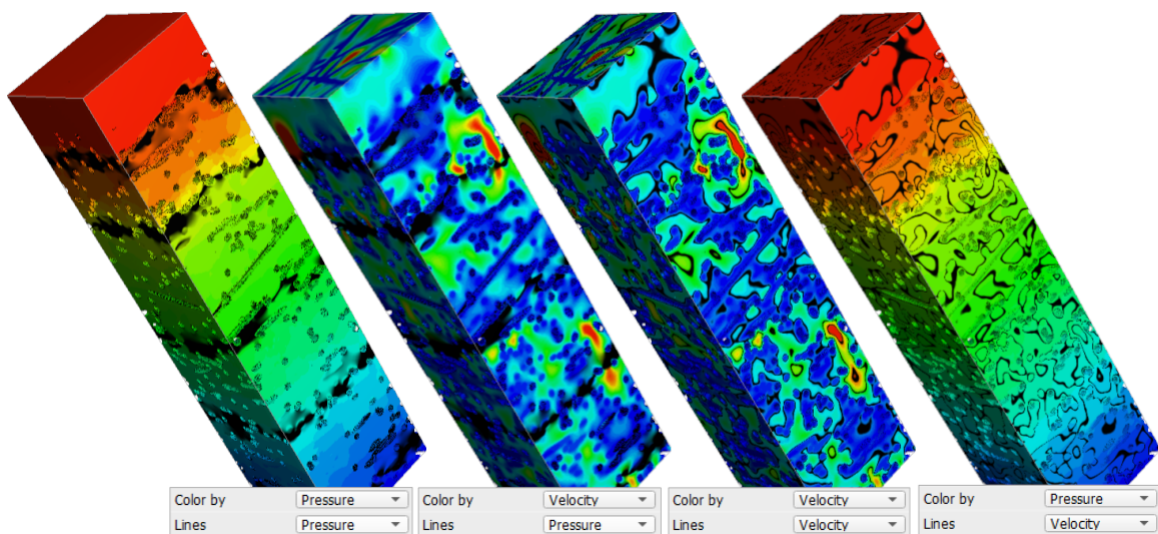
✓ Clipping

The displayed data can be spatially clipped in all three directions (**ClipX**, **ClipY**, **ClipZ**) with the sliders or by direct input. This facilitates the simultaneous visualization of, e.g., structure model and flow field. They work as described for the [structure clipping](#)⁵⁶.



Lines

The settings for the display of flow field isosurface **Lines** through the **Visualization Settings** dialog (from the sidebar) are explained [for the 2D visualization](#)⁴⁰ and are only visualized, if transparency is disabled.

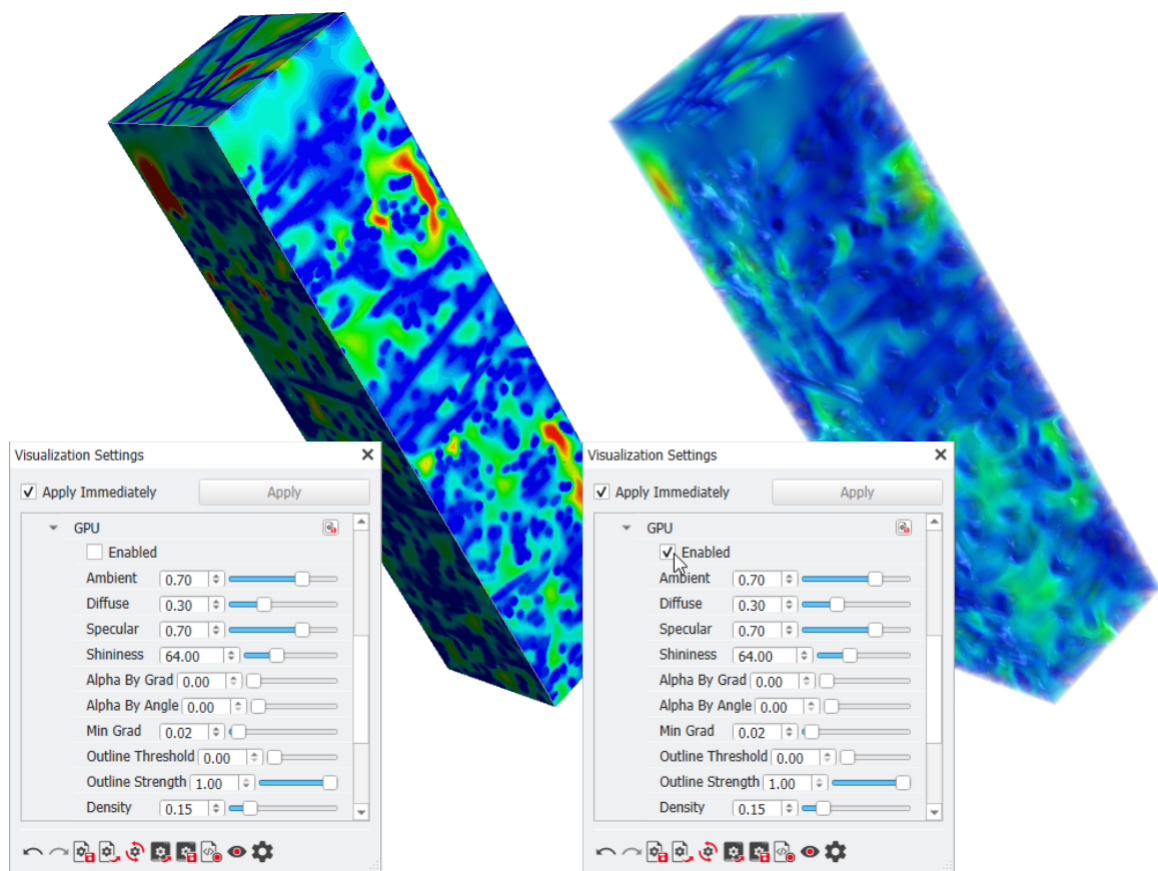




Important! This feature is independent from the Isosurfaces Settings in the [Color Map](#) 128 dialog which are only displayed when transparency is enabled.

GPU Visualization

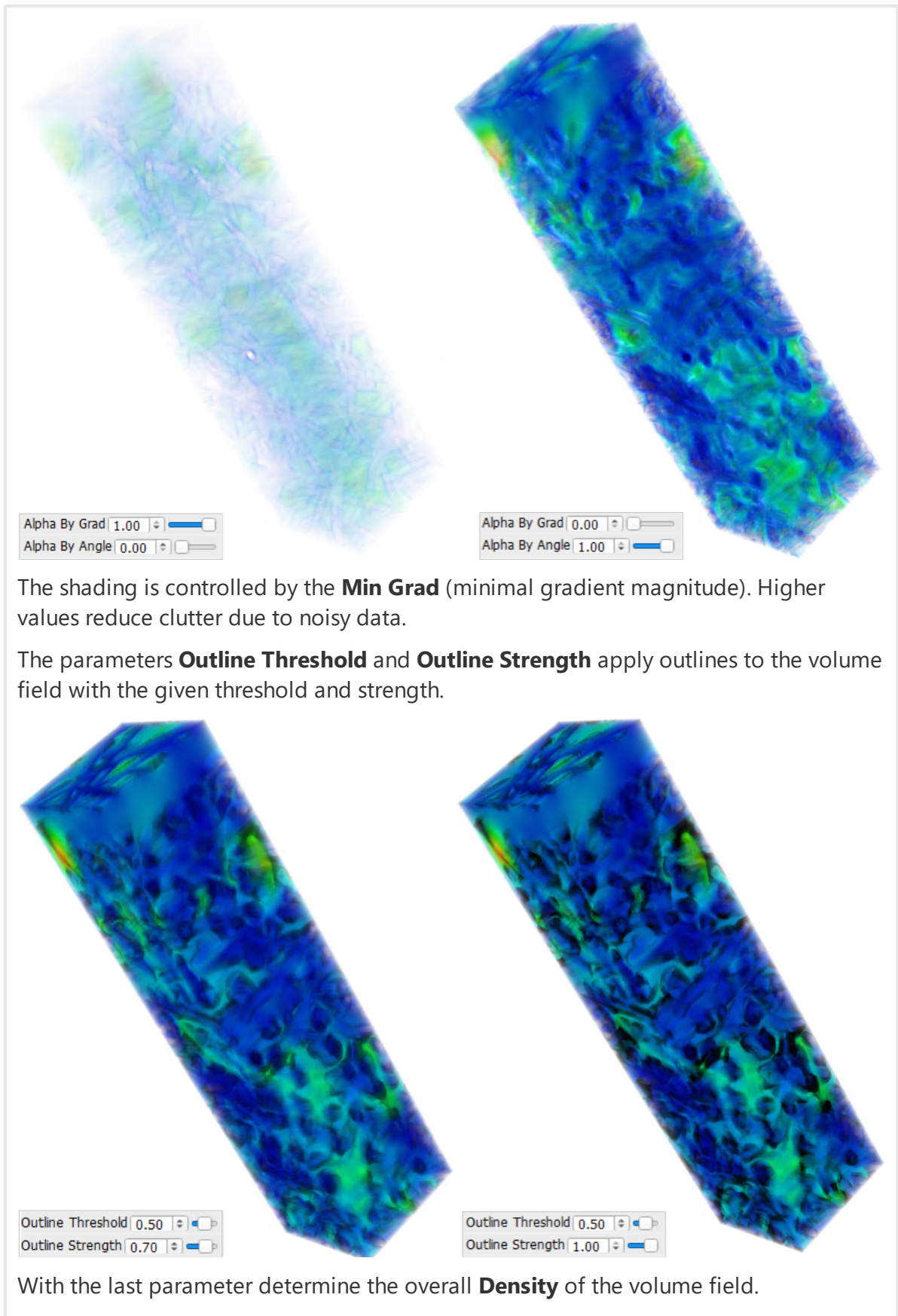
The **GPU** section, in the **Visualization Settings** dialog, offers additional visualization options using the graphics card instead of the main processor, when checking **Enabled**. The data needs to fit in the GPU memory to use this feature.



Control the amount of **Ambient** light, **Diffuse** scattered light and **Specular** mirror-like highlights, while determining the size of highlights with **Shininess**. These parameters correspond the [Phong Reflection Model](#), often used in 3D computer graphics. The light intensity controlled by them is independent of the light direction. Higher values for **Shininess** result in smaller highlights.

Alpha By Grad sets a focus to the edge regions. For high values only the edges are visible.

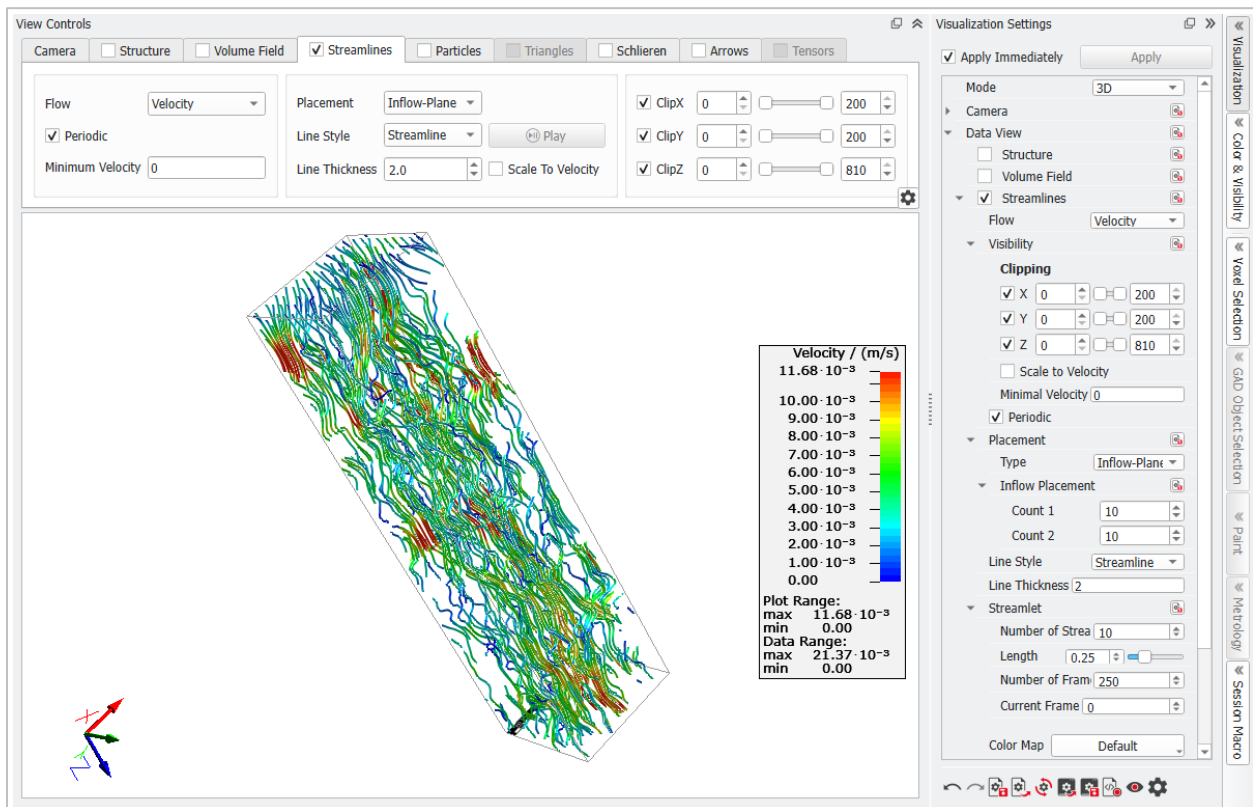
Alpha By Angle determines the visibility for the current camera angle. Higher values result in a more transparent volume field.



Streamlines

Visualize the flow through your structure with streamlines or even animated streamlets.

The **Streamlines** tab controls the visibility of the streamlines or streamlets. Some additional options and all the options from the **View Controls** can also be found in the **Visualization Settings** dialog. Expand the **Visualization Settings** dialog either by clicking the gear wheel icon in the bottom right of the Streamlines tab or by opening the **Visualization tab** in the sidebar and unfolding the **Data View → Streamlines** section.

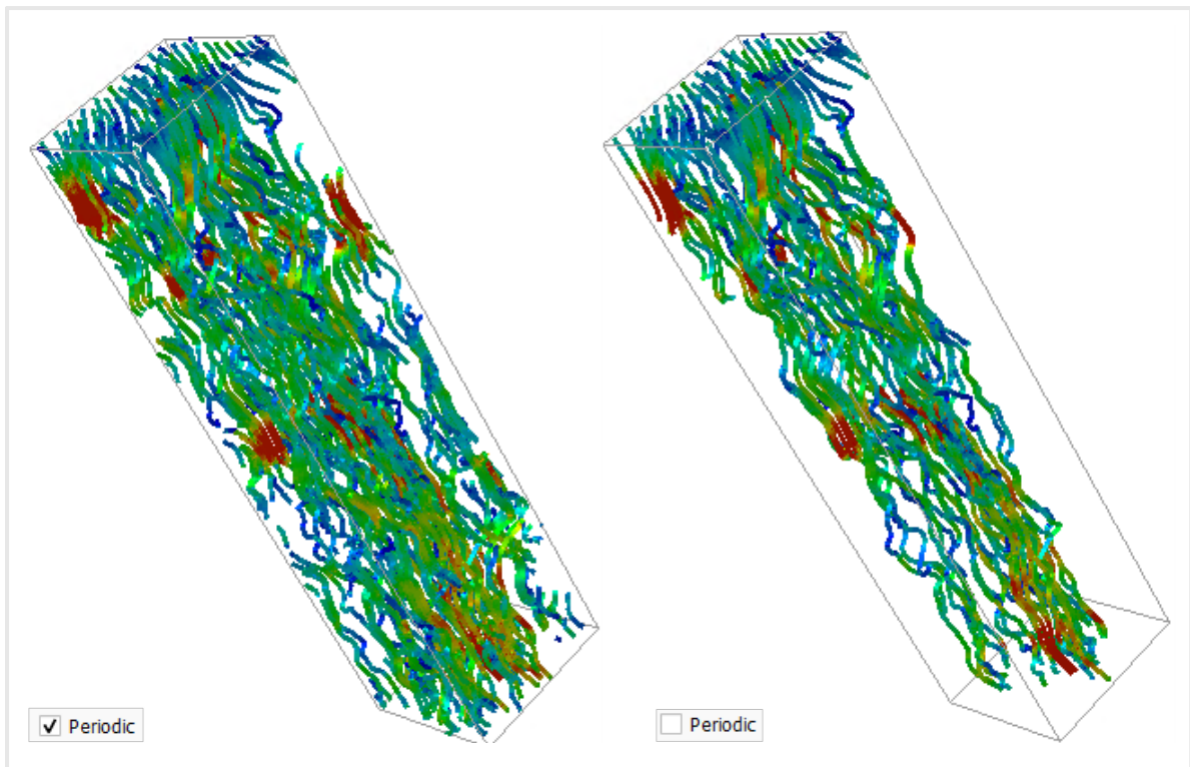


✓ Flow

The **Flow** pull-down menu shows one choice for each loaded [vector field](#)⁴⁴. For a flow field for example, only the field Velocity is available. The color coding of the streamlines is based on the chosen field.

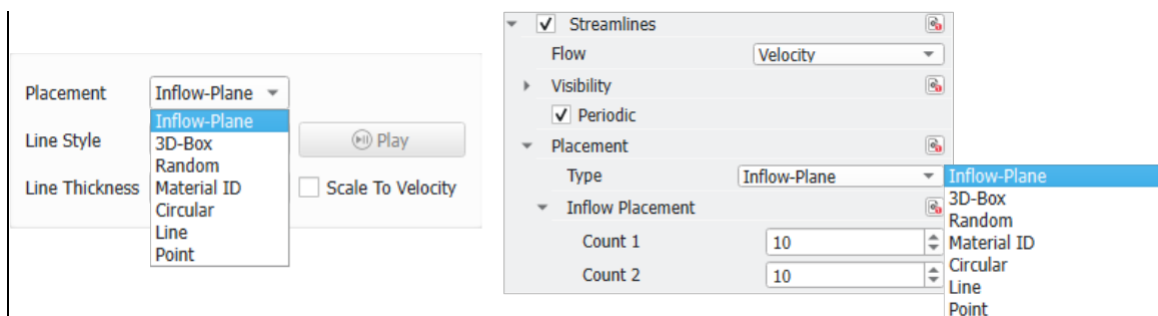
✓ Periodic

Checking **Periodic** enables periodic boundaries for the display of the streamlines, i.e., the streamlines exiting the tangential domain boundaries are reentering the domain periodically.



✓ Placement

The choices for **Placement** define from where the streamlines, streamlets, or streamtubes are calculated.



Select the streamline starting positions from the pull-down menu for **Placement** (or **Placement ► Type**). Choosing any of them gives options for their display, as seen above for the **Inflow-Plane Placement**.

Either the starting points (or seeds) for the calculation are placed

- at the **Inflow-Plane**,
- in a grid in the 3D-domain (**3D-Box**),
- randomly distributed in the entire domain (**Random**),
- in one of the materials (**Material ID**),
- inside of a circle (**Circular**),
- along a line (**Line**),

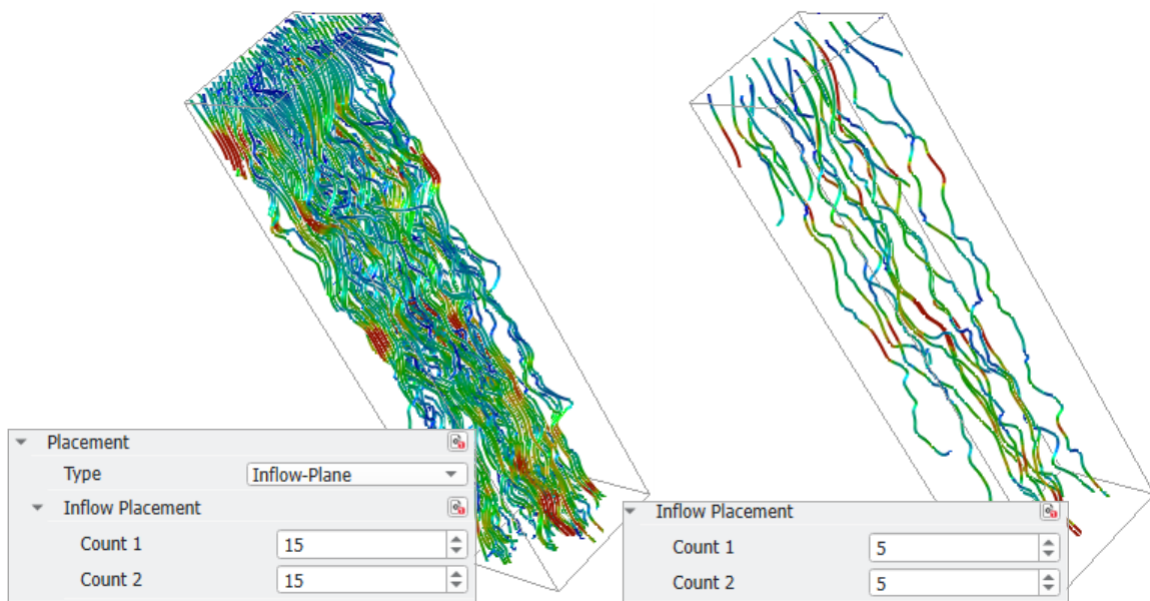
- or at a given point for a single streamline (**Point**).



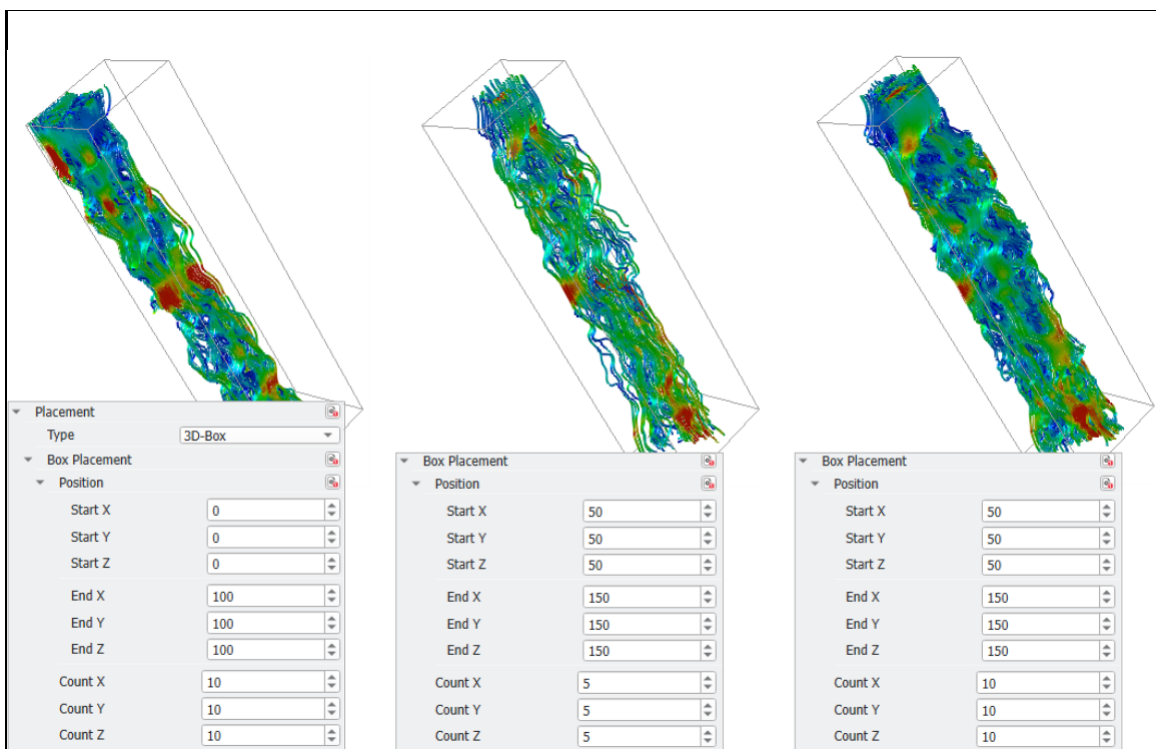
Know how! The placement only defines the starting positions. However, streamlines are not shown in materials where they have a length of 0. Considering a flow field, for example, if a streamline starts in, or quickly runs into a solid voxel the streamline is invisible.

When there is flow through the entire domain, entering on one side, leaving on the opposite side, the **Inflow-Plane** option is preferable. For a flow, which starts somewhere in the domain, such as Lithium-flow in a battery simulation, either **3D-Box** or **Random** are appropriate.

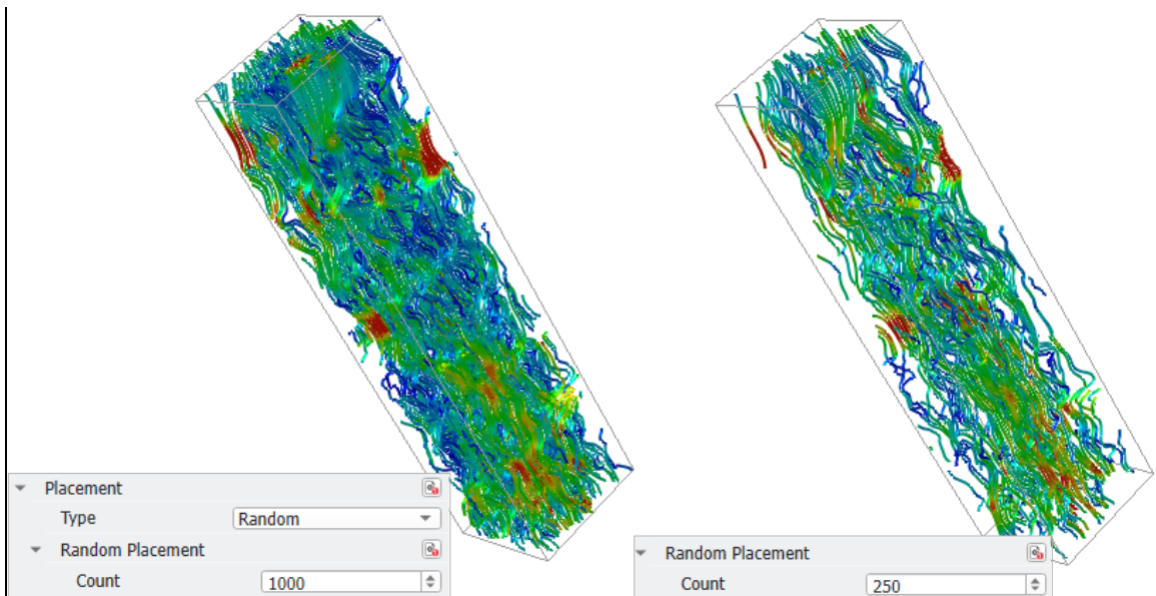
Selecting **Inflow-Plane** distributes the starting positions on the flow field inflow plane. In flow fields, this plane is determined by the flow direction. If for example, the flow is computed in Z-direction, the inflow plane is the XY-plane. Then, **Count 1** and **Count 2** define a grid in the inflow plane. Increasing the number, increases the number of streamlines in the respective direction.



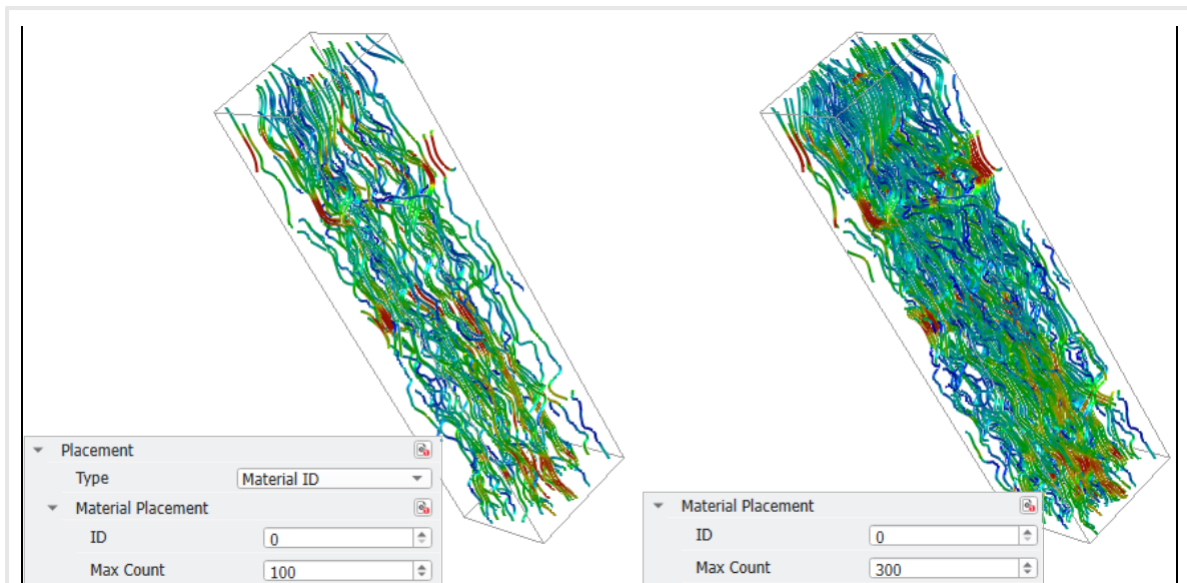
Selecting **3D-Box** for **Placement Type** defines a three-dimensional box with corners (**Start 0**, **Start 1**, **Start 2**) and (**End 0**, **End 1**, **End 2**). Inside this box **Count X** x **Count Y** x **Count Z** streamlines are placed regularly.



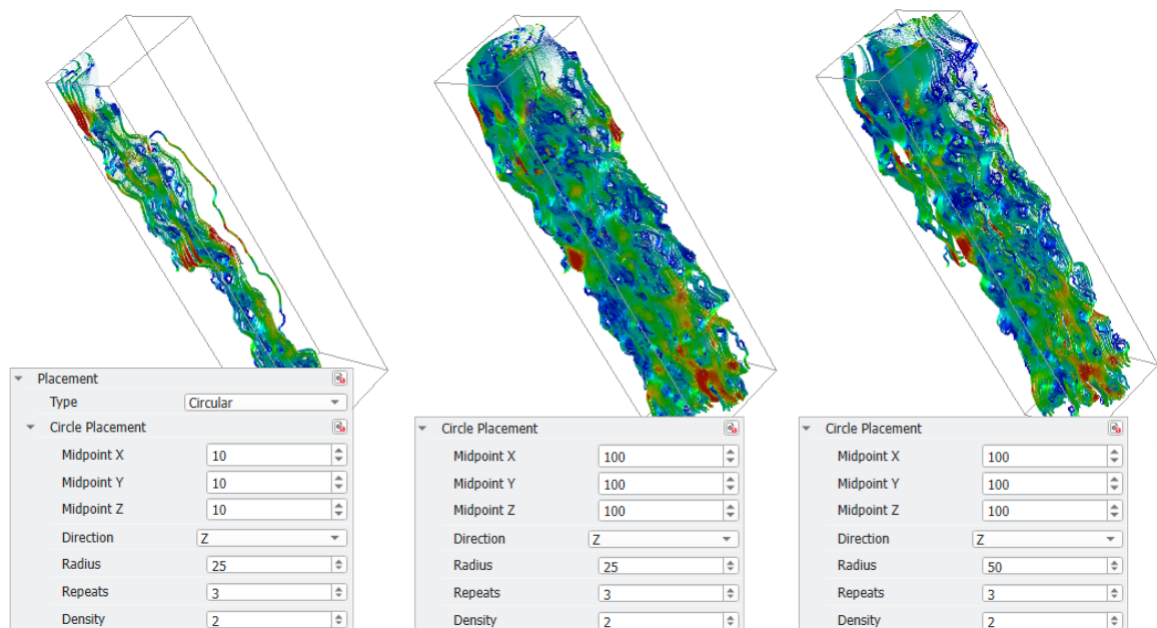
To distribute the starting positions randomly, select **Random**. For **Random Placement** choose a value for **Count**, controlling the number of starting positions for the streamlines.



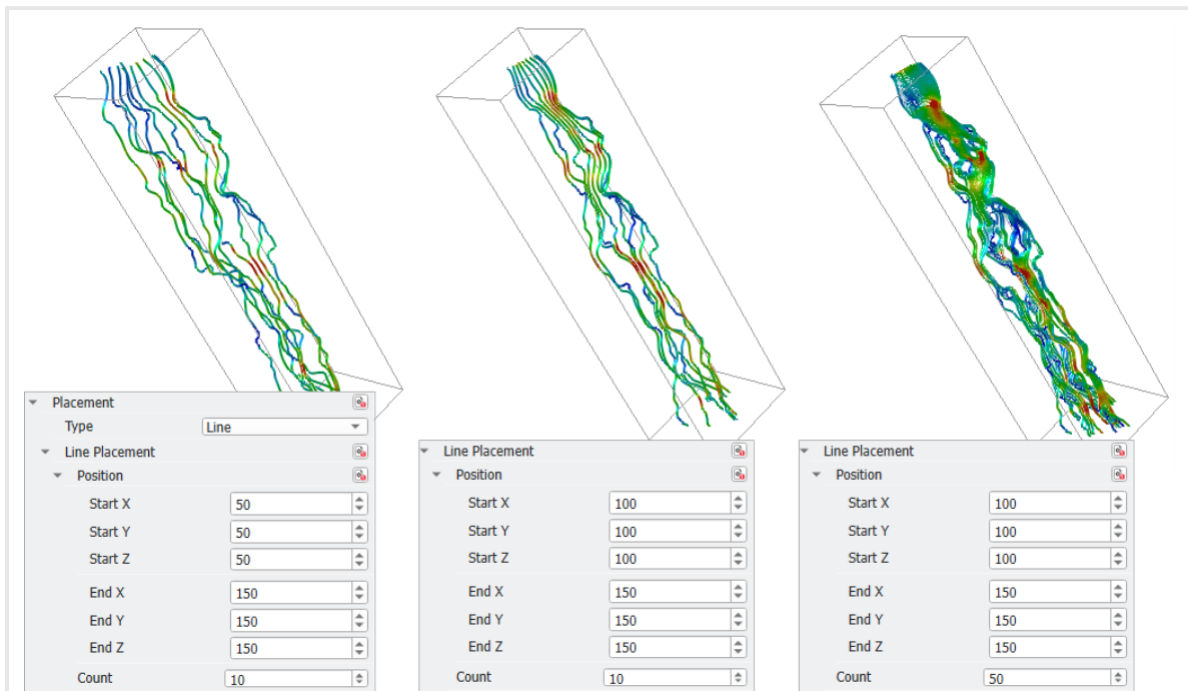
Choosing **Material ID** places seeding positions only inside voxels of the selected material ID. **Max Count** defines the number of starting positions for streamlines. The seeding points are all placed at voxel center positions. If there are less voxels of the material ID present, than given for **Max Count**, less streamlines are visualized.



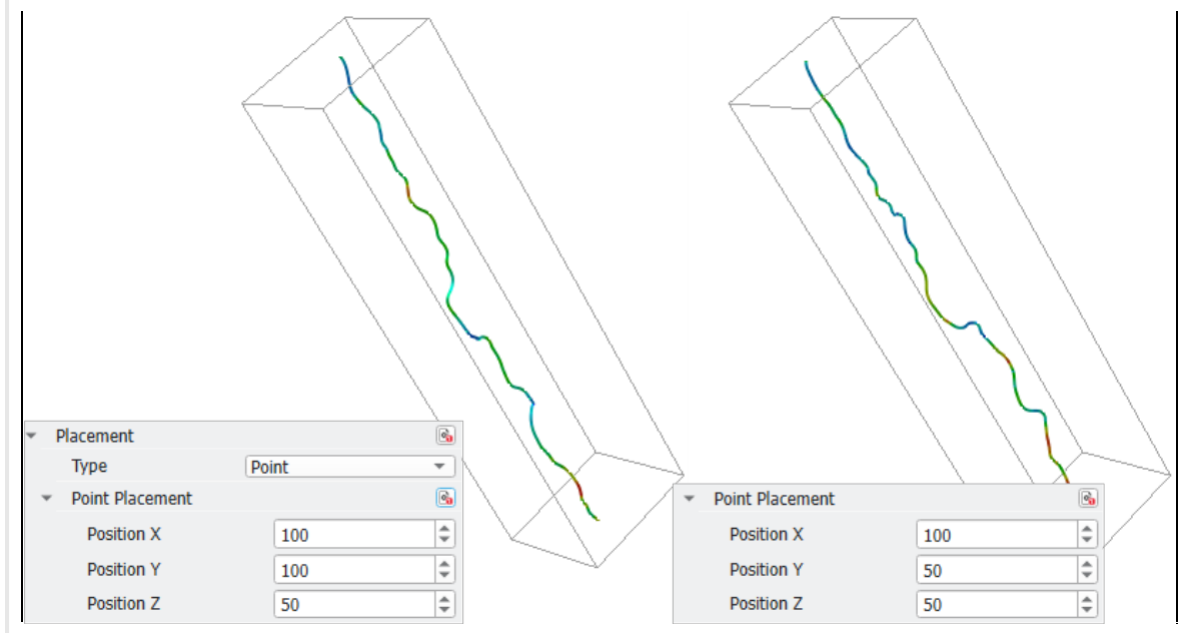
Select **Circular** to place the streamline starting positions on circles around the defined **Midpoint**. The **Direction** then defines an axis through the **Midpoint** on which the circles are placed. Define the **Radius** for all these circles. The number of circles is controlled with the value entered for **Repeat**. The distance between the streamline starting positions on the circles is defined by **Density**.



The placement type **Line** places the starting positions on a line from (**Start 0, Start 1, Start 2**) to (**End 0, End 1, End 2**). The value for **Count** controls the number of streamline starting positions.

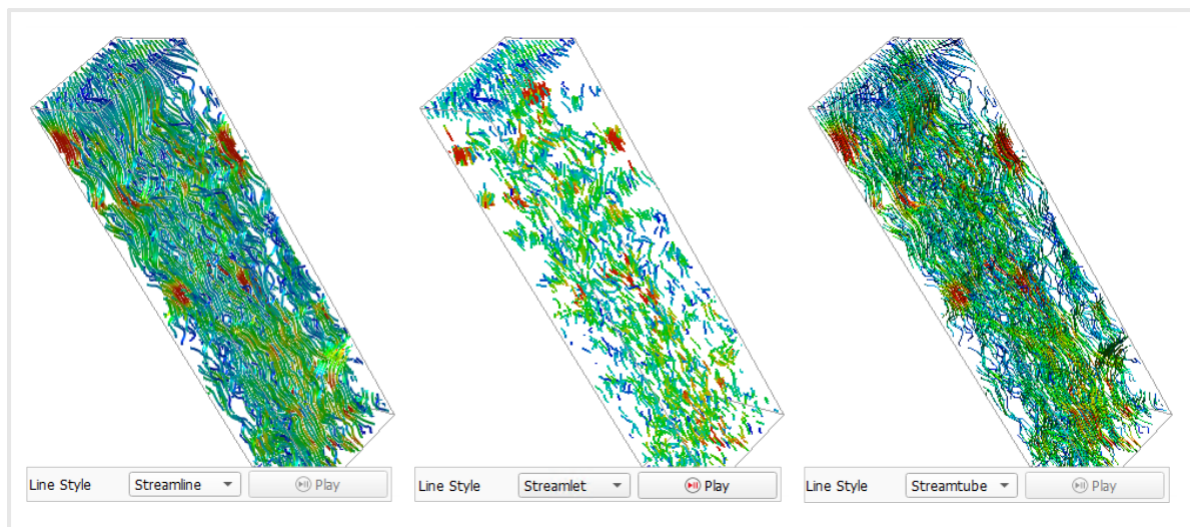


Select the placement type **Point** to place a single streamline at the starting position defined by **Position 0**, **Position 1**, and **Position 2**.



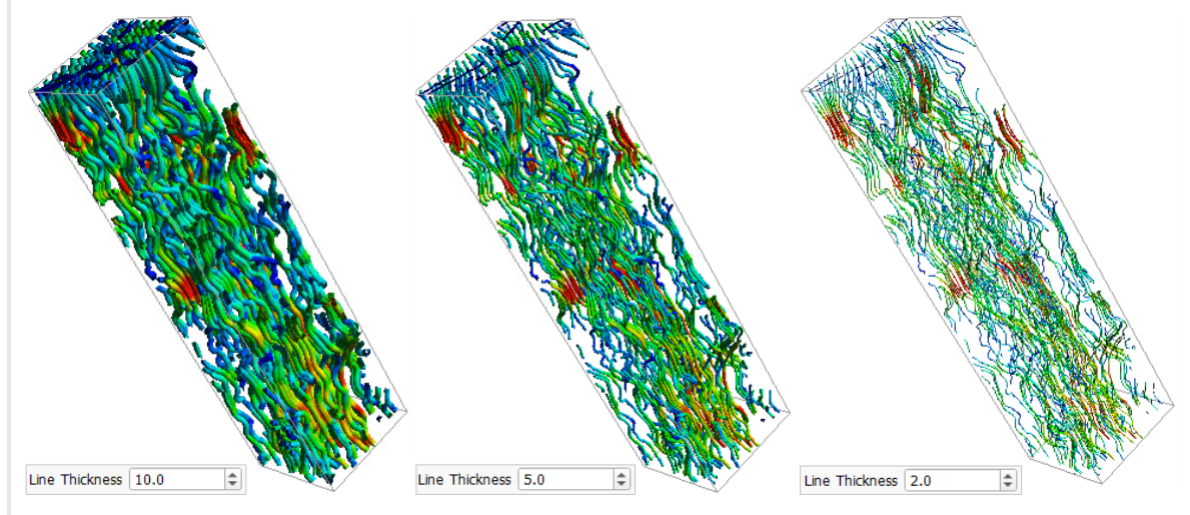
✓ Line Style

The choices in **Line Style** are **Streamline**, **Streamlet**, and **Streamtube**. If Streamlet is selected, the Play button becomes available. How to animate streamlets is explained [here](#) ⁷⁴.



✓ Line Thickness

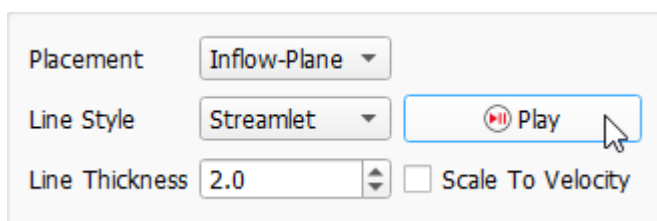
The thickness for each line style is controlled through **Line Thickness**, here shown for streamtubes.



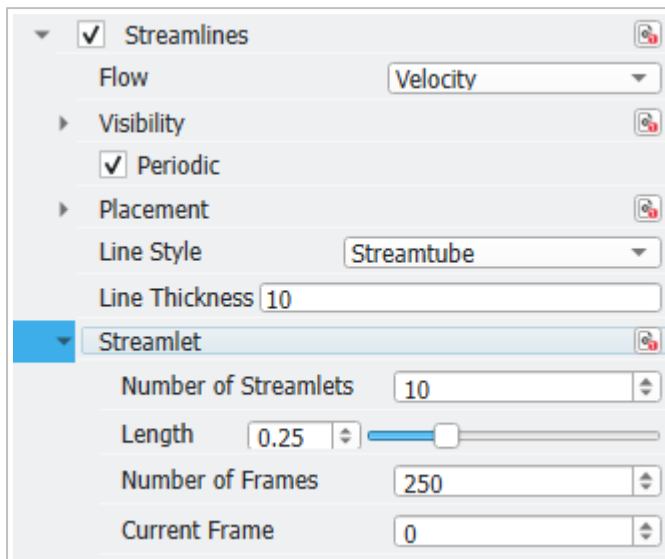
✓ Streamlets Animation

Choosing **Streamlet** activates the visualization of the streamlines as streamlets, a visualization in which a streamline is fragmented into segments.

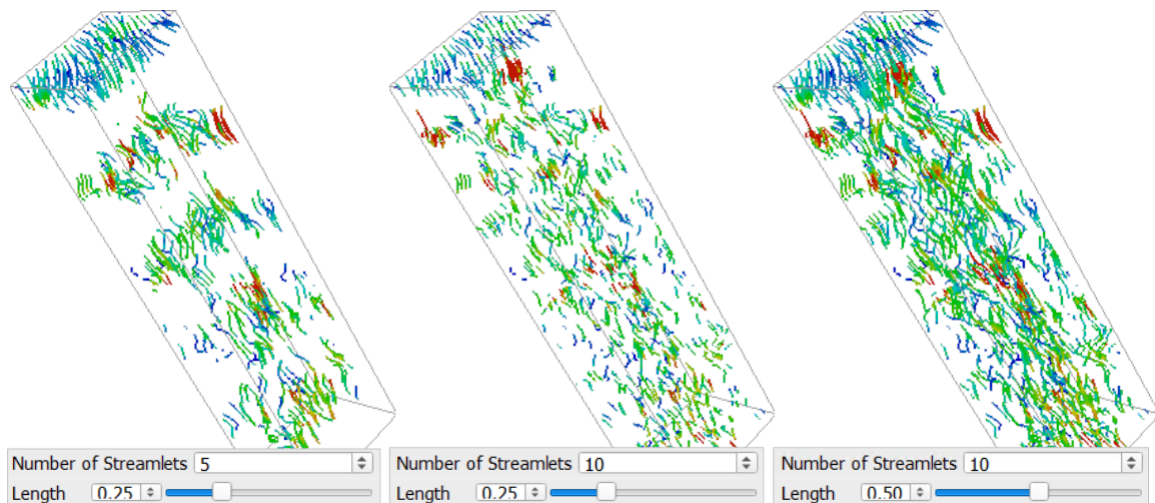
The movement of these segments along the streamline is visualized and can be animated by clicking **Play**. The animation is repeated in an infinite loop until **Play** is clicked again.



The visualization and animation of streamlets is controlled by four options in the **Visualization Settings** dialog in the **Streamlet** section:

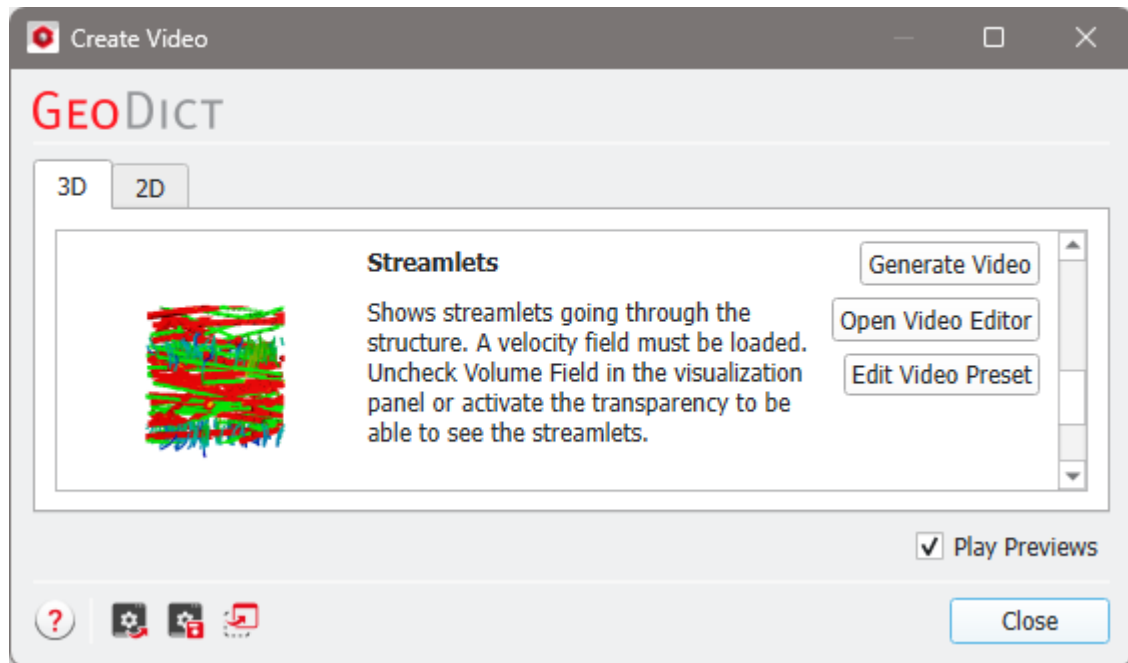


- **Number of Streamlets** defines the number of segments that represent a streamline. The larger the number, the easier it becomes to identify the complete streamlines.
- The streamlet **Length** determines the fraction of each streamline that is displayed. In combination with the number of Streamlets per Streamline, it defines the length of the streamlets. The larger the number, the lower the distance between the streamlets. For a length of 1, the streamlets connect to complete streamlines.
- The **Number of Frames** controls the speed of the animation by specifying the number of frames that are to be visualized when clicking **Play**. Small numbers produce fast and short animations. Large numbers produce slow animations, with more frames.
- The **Current Frame** is updated automatically, when stopping the animation. While no animation is played, any frame between 0 and the entered **Number of Frames** can be visualized.



When **Streamlets** is chosen, [Scale to Velocity](#)⁷⁶ additionally adjusts the movement of the streamlets in relation to the maximum velocity of the flow field. As a result, the streamlets move with individual velocities and not with a uniform speed, as they would do without a scaled velocity.

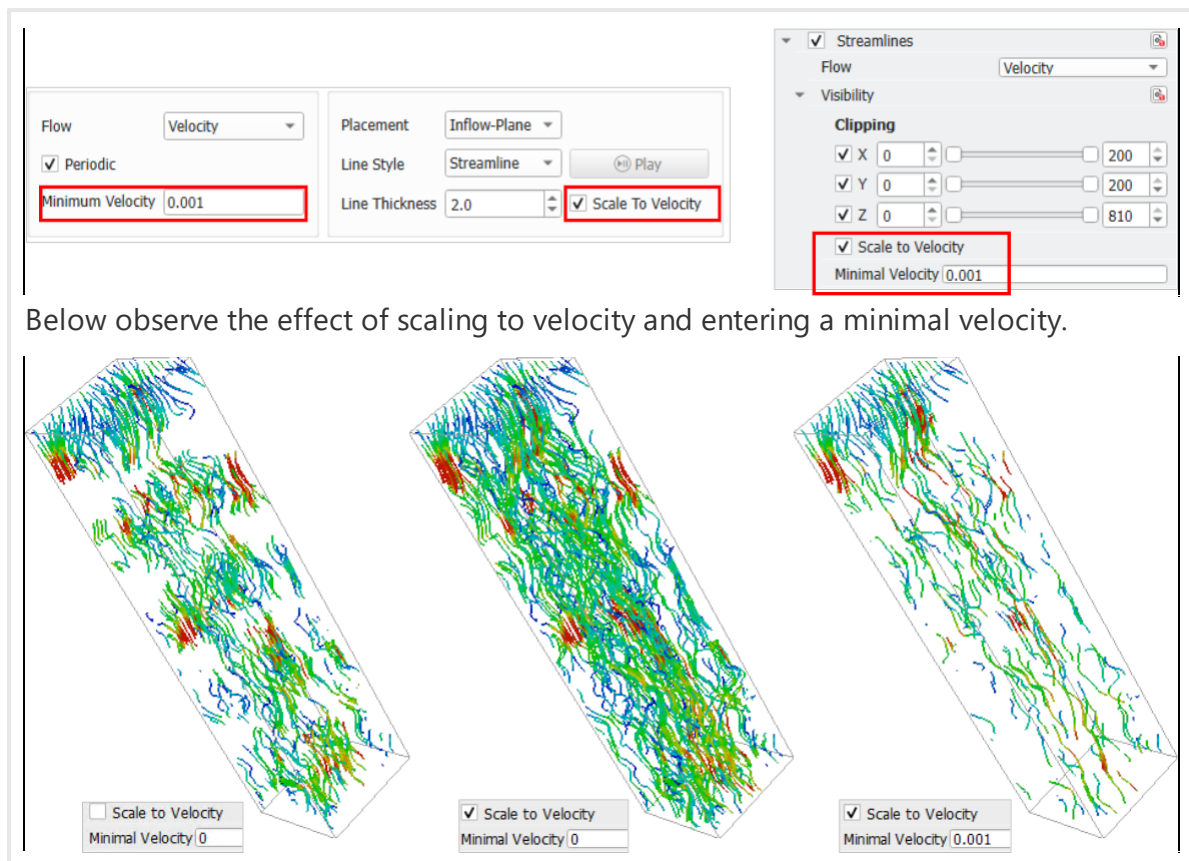
Generate a video using the **Create Video** dialog (**File - Save Video as**). Select the **Streamlets** preset. You can either just click **Generate Video** or change some video settings by clicking **Open Video Editor**. How to create videos in GeoDict is explained in detail in the Create Videos chapter.



✓ Scale to Velocity and Minimal Velocity

The setting **Scale to Velocity** affects the visualization of streamlines and streamlets and is coupled with **Minimal Velocity**. This is a clipping tool based on the velocity of the fluid. In this case, streamlines and streamlets with a velocity lower than the specified **Minimal Velocity** are not shown in the visualization. In the **Visualization Settings** dialog the option is found under **Data View ► Streamlines ► Visibility**.

When **Streamlets** is chosen, **Scale to Velocity** additionally scales the streamlet lengths to their velocity, i.e., for higher velocities, the streamlets are longer. Also, it adjusts the movement of the streamlets in relation to the maximum velocity of the flow field, when [animating](#)⁷⁴ the streamlets.



Below observe the effect of scaling to velocity and entering a minimal velocity.

✓ Clipping

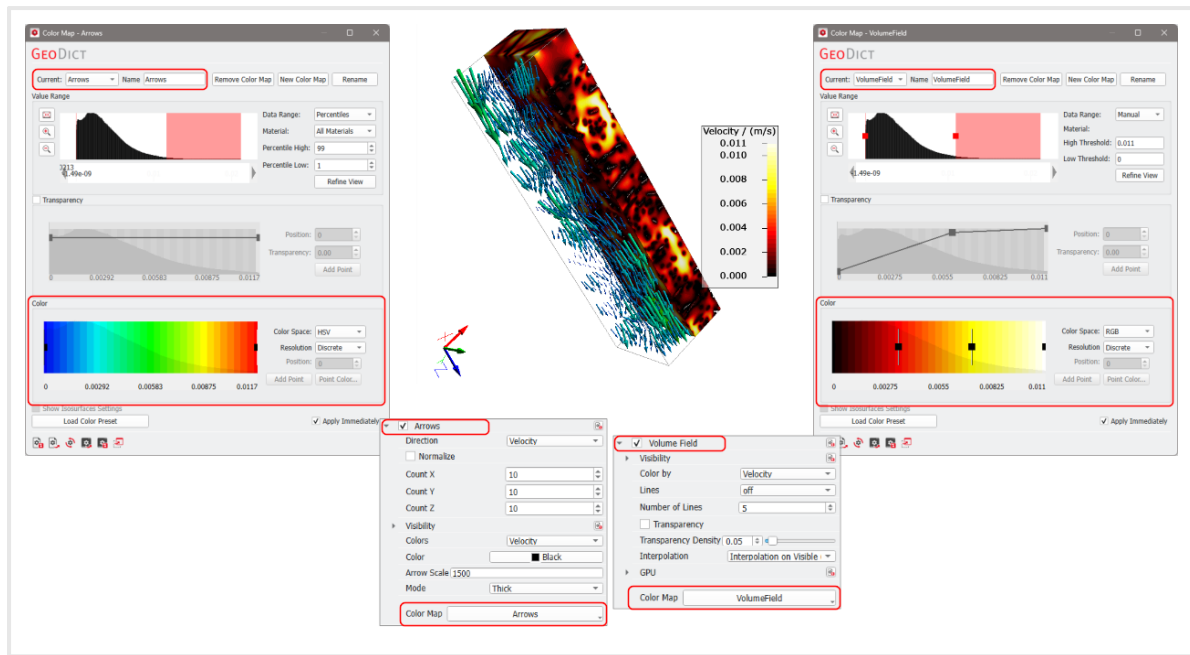
The streamlines, streamlets, and streamtubes cannot only be clipped according to their velocity but also conforming to their spatial position, in a similar way as explained for the [structure clipping](#)^[56].

It is possible to define the range of voxels in each spatial direction that are to be displayed by moving the slider or by direct input of the value.



✓ Color Map

Define different color maps in the [Color Map](#)^[121] dialog. From the pull-down menu for **Color Map** in the **Visualization** tab on the right you can then select between your [custom color maps](#)^[122]. You can define and select a different color map for each of the components. For example, visualize arrows with the jet [color preset](#)^[125] and the volume field with the hot preset:

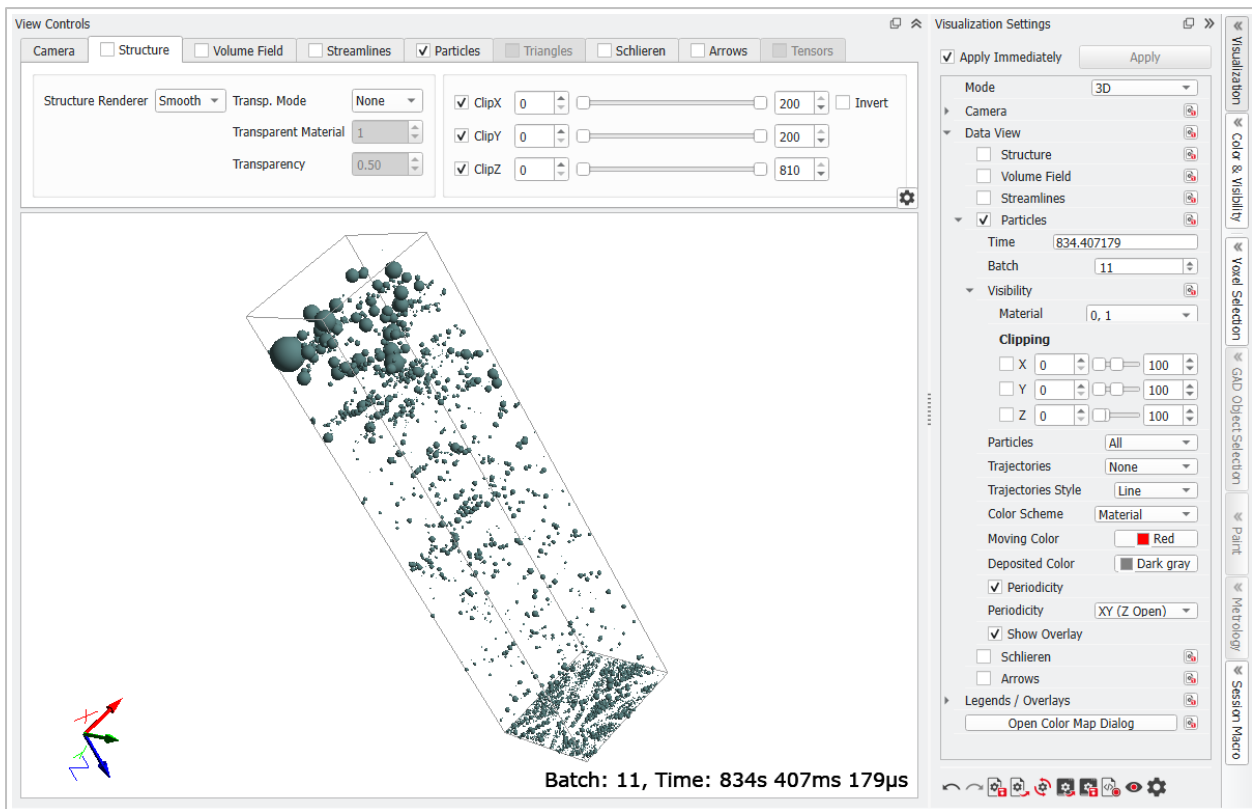


Particles

Visualize and animate the particles moving through your structure.

The **Particles** visualization tab is automatically enabled when trajectories or final particle positions are loaded. You can visualize the movement of a particle when you have loaded its complete trajectory. If you have loaded only the final position of a particle, the movement is not visualized, but it will appear at its final position.

The most important settings for the particle visualization can be accessed through the **Particles** tab. Additional options are found in the **Visualization Settings** dialog. Expand the **Visualization Settings** dialog either by clicking the gear wheel icon in the bottom right of the Particles tab or by opening the **Visualization tab** in the sidebar and unfolding the **Data View** → **Particles** section.



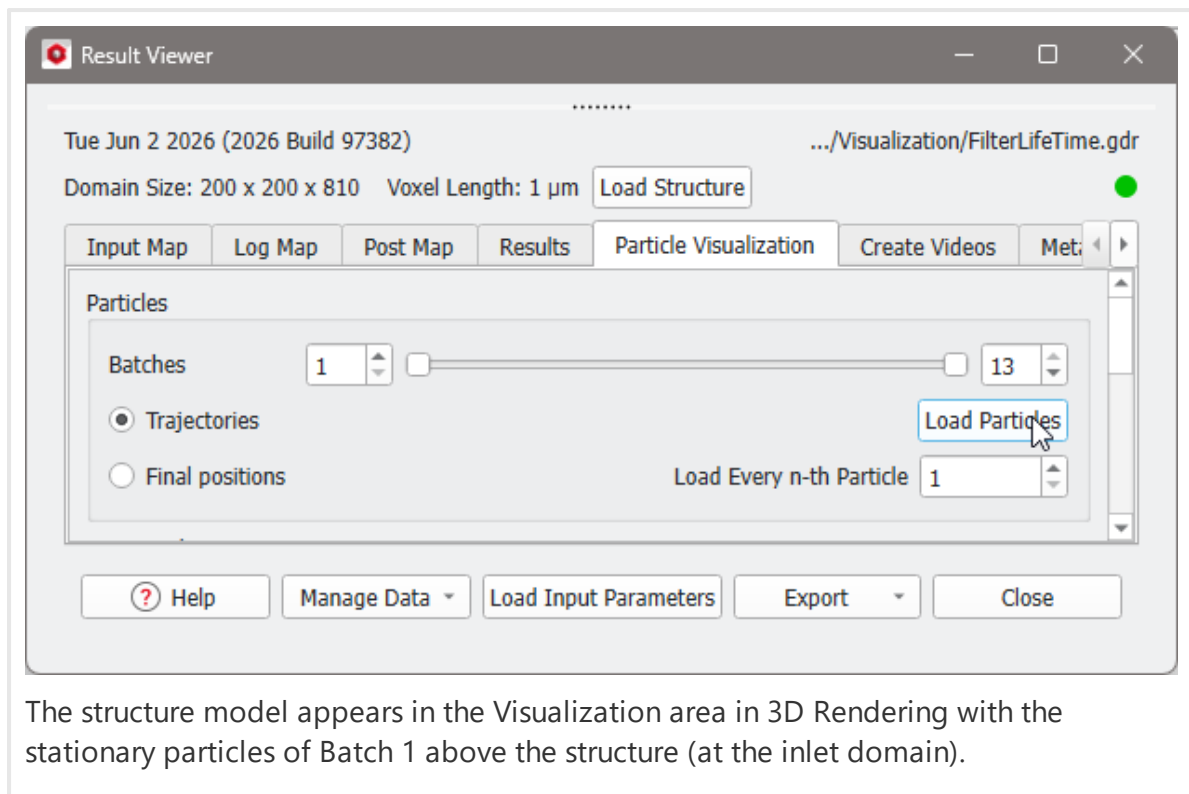
✓ Load Particles

Click the **Particle Visualization** tab. Buttons in three panels give access to the visualization of particle trajectories, the visualization of the particle positions with time, the visualization of the deposited dust with time and the visualization of the fluid flow field.

In the **Particles** panel, move the slider to select the **Batches** of particles to be visualized. For example, keep the choice to visualize all batches that ran in this filtration simulation, which is batches 1 to 13 in the example.

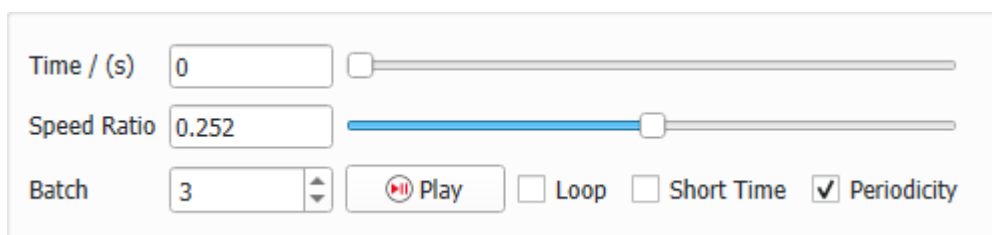
Select **Trajectories** to animate the particle motion from inflow to outflow or deposition place. If a large number of particles was simulated, loading all the trajectories at once is memory consuming and may not be possible. Loading only the **Final positions** does not allow to animate the particle motion, but is less memory consuming. Memory consumption may also be reduced by visualizing only a fraction of the simulated particles per batch by loading only **Every n-th Particle**.

Then, click **Load Particles**.

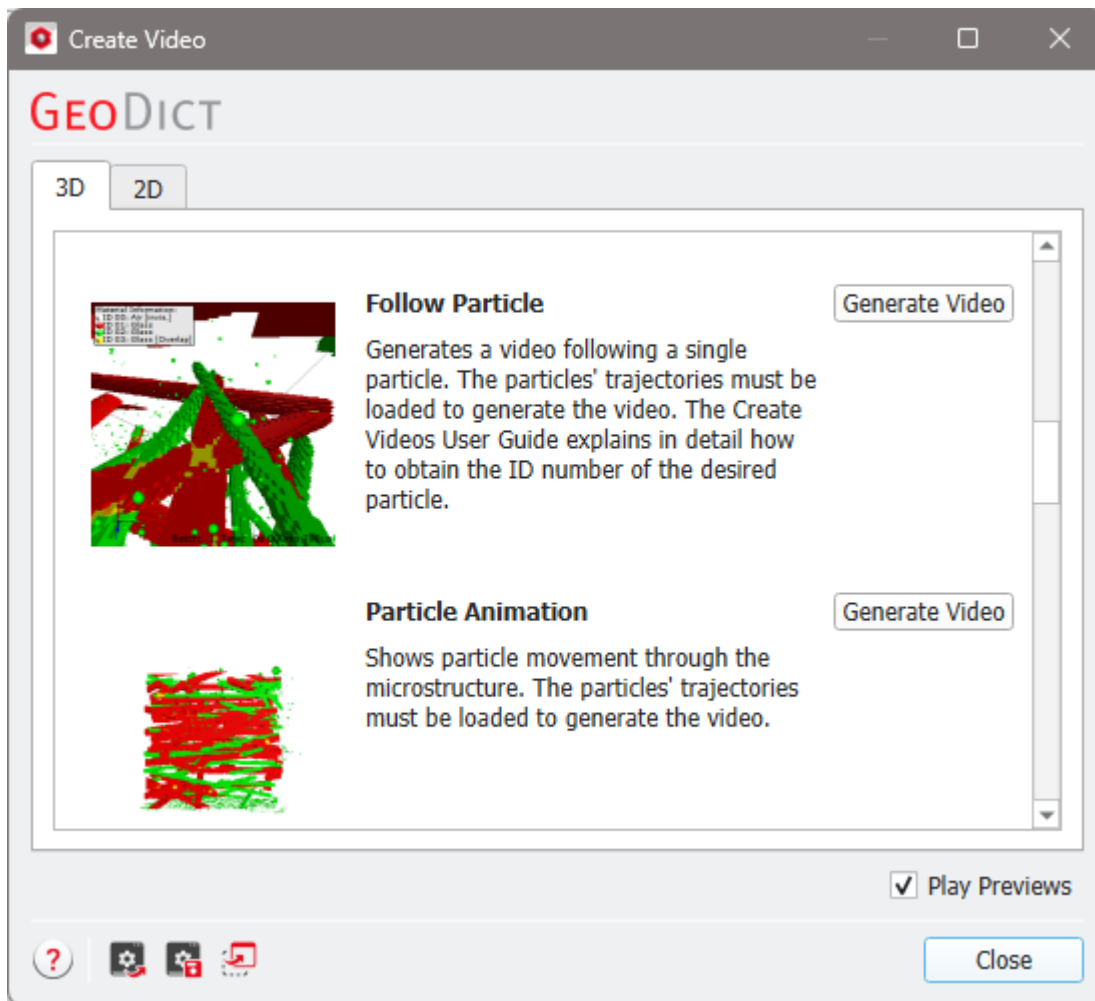


Animation Controls

In the **View Controls**, the sliders for **Time** and **Speed Ratio** control the visualization of particle movement.



You can also generate videos using the **Create Video** dialog (**File - Save Video as**). Select the **Follow Particle** preset or the **Particle Animation** preset and click **Generate Video**. How to create videos in GeoDict is explained in detail in the Create Videos chapter.



✓Time

Moving the slider or directly entering a **Time** allows displaying the particles at a particular point in time.

✓Speed Ratio

The **Speed Ratio** controls the visualization speed by entering values directly or by moving the slider. Low values of **Speed Ratio** result in the particle movement process being shown in slow motion, whereas high values lead to faster motion.

✓Batch

The animation of the particles, in which the particles move through the filter structure, starts by clicking **Play**. The **Batch** number changes from 1 to the last batch, as the batches of particles appear in the inflow region above the filter media model and pass through it. The bigger particles are caught and held by the filter material whereas the smaller ones pass through to reach the outflow region.

Below, two images of batches 6 and 12 are shown. The color of the particles hovering over the structure has been changed from the default green to red. The deposited particles are shown in dark gray.



Batch: 6, Time: 386s 332ms 011µs



Batch: 12, Time: 929s 356ms 033µs

✓ Loop

The **Time** slider does not return to zero when the animation has finished and must be set to the initial value to restart the animation. However, checking **Loop** makes the animation play endlessly.

✓ Short Time

Checking **Short Time** adapts the visualization **Time** to the period when most of particles move. This option cuts off the final part of a simulation where only a few particles move at very low velocities and virtually no changes are visible to the observer.

✓ Periodicity

Checking **Periodicity** activates periodic boundary conditions for the visualization of the simulation results. This is only recommended if the simulation was also done with periodic boundary conditions. In the **Visualization Settings** dialog, select where to apply periodicity from the pull-down menu for **Periodicity**. If for example, **XY (Z Open)** is selected, the periodicity is only applied in X- and Y- direction, while the Z- direction is open for particle movement.

Particles Shown

✓ Material IDs

If the particles move through different materials, choose the **Material IDs** where the particles should be visible.

✓ Particles and Trajectories

The **Particles** and **Trajectories** pull-down menus can be set to:

- **None** to show no particles (or no trajectories).
- **Filtered** to show only the filtered particles (or their corresponding trajectories).
- **Not Filtered** to only show the particles that are not filtered (or their corresponding trajectories).
- **Time Out** to display only particles (or their corresponding trajectories) whose movement could not be simulated completely within the time defined for each batch. For example, in some cases, a few particles can have very small velocities in the inflow region compared to most of particles and therefore their behavior cannot be simulated completely within the defined batch time.
- **All** to show all particles.
- **Custom**. This option is only available in the Particles pull-down menu, but also applies for trajectories. If selected, click **Edit Custom** to define the particle visualization as described [here](#) ⁸⁵.

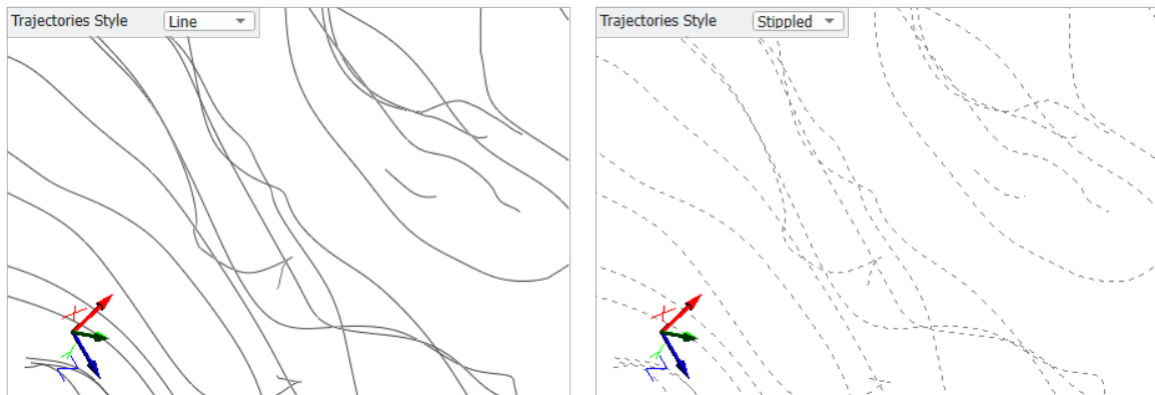
✓ Clipping

Clip the particle visualization area by enabling a clip direction, and moving the corresponding sliders or entering clipping values, working the same way as for the [Structure](#) ⁵⁶.

Additional Parameters in Visualization Settings

✓ Trajectories Style

If trajectories are [shown](#)⁸³, you can change their style between **Stripped** and **Line**.

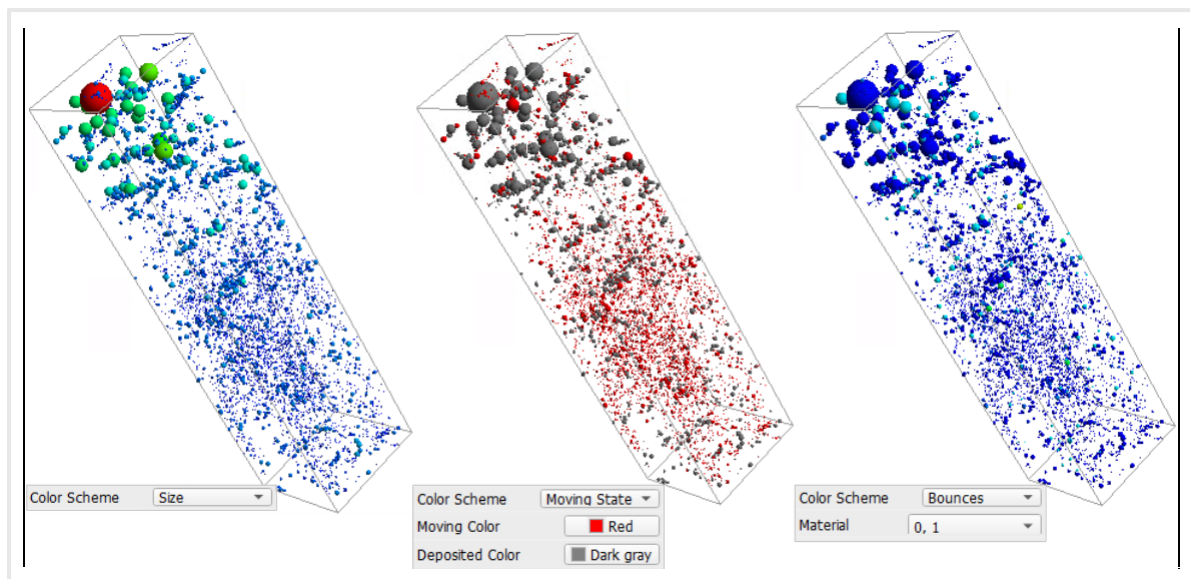


✓ Color Scheme

In the [Visualization Settings](#)¹⁹ dialog (**Data View ► Particles**), control the **Color Scheme** by selecting between the following options from the pull-down menu:

- **Moving State:** If selected the default green color of the non-filtered particles can be changed with the parameter **Moving Color**. The color of the filtered particles can also be changed from the default dark gray with **Deposited Color**.
- **Material:** The colors are selected automatically depending on the material.
- **Size:** The colors are selected automatically depending on the particle size.
- **Batch ID:** The colors are selected automatically depending on the batch ID.
- **Residence Time:** The colors are selected according to the time spent in the chosen **Materials**.
- **Bounces:** The colors are selected depending on the number of bounces from the selected **Materials**.
- **Entries:** The colors are selected depending on how often the particles enter the chosen **Materials**.
- **Trapped:** The colors are selected depending on the material they are trapped in.

Sometimes it is useful to switch off the visualization of the structure by deselecting **Data View ► Structure**.



✓ Show Overlay

At the bottom-right of the visualization area, observe the **Batch** number and the point in **Time** of the filtration process. Determine if they should be shown or not by enabling / disabling the checkbox **Show Overlay** in the **Particles** section of the **Visualization Settings** dialog. The font size and color can be changed as described in the [Legends / Overlays](#)¹¹⁷ section.

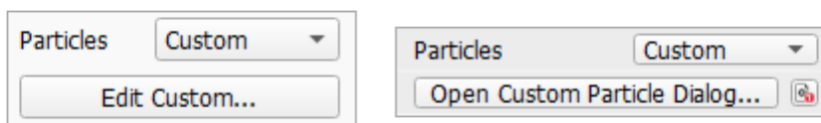
Batch: 3, Time: 102s 143ms 512μs

Custom Visualization of Particles

The display of the particles moving through the structure and their trajectories can be customized.

When choosing **Custom** for the visualization of **Particles**, the **Edit Custom...** button overlays and replaces the standard **Trajectories** pull-down menu described above. Now, the visualization of trajectories is controlled through the **Visualization - Trajectories** option of the **Custom Particle Selection** dialog.

Settings for a user-defined, customized visualization can be defined by clicking the **Edit Custom...** button. Alternatively, the same dialog can be accessed by clicking **Open Custom Particle Dialog** in the Visualization Settings panel.



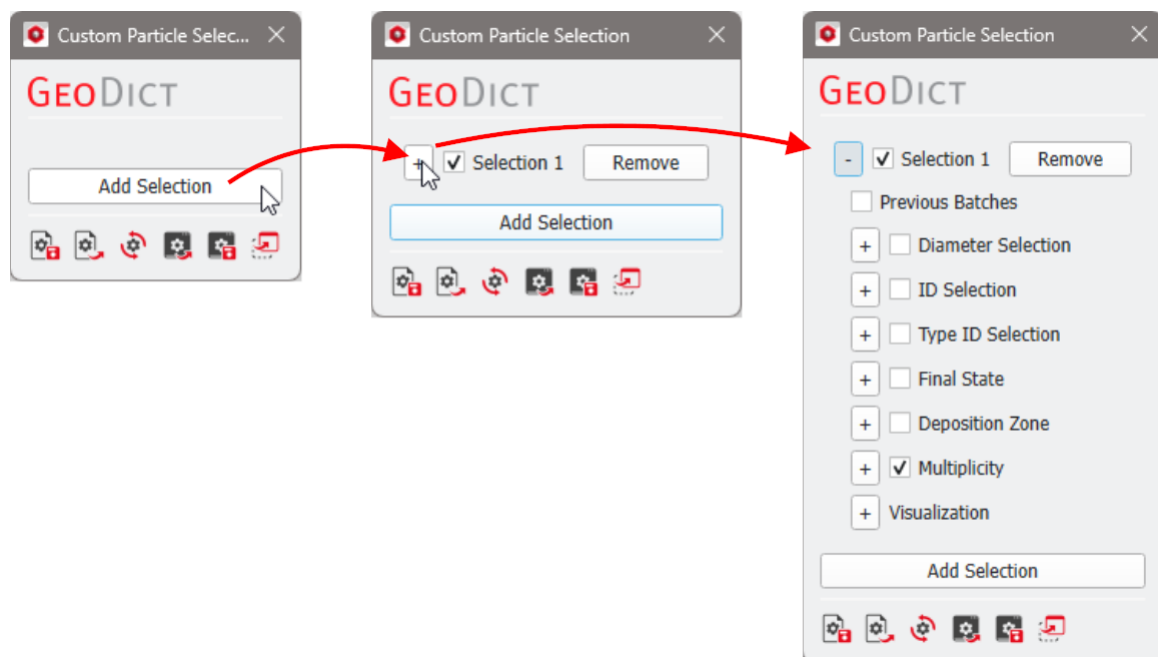
The **Custom Particle Selection** dialog opens. The dialog may remain open to observe the changes that the selections have on the visualization of particles moving through the structure.

The custom particle options can be changed while the particle animation is playing, so that the effect of the selection can be observed immediately.

✓ Add Selection

Clicking **Add Selection** generates **Selection 1**. The selection can be disabled by unchecking the corresponding checkbox or deleted by clicking **Remove**.

Clicking  at the left of **Selection 1** opens the options: **Previous Batches**, **Diameter Selection**, **ID Selection**, **Type ID Selection**, **Final State**, **Deposition Zone**, **Multiplicity**, and **Visualization**.



Parameters for all these options are accessed by clicking on their  symbol.

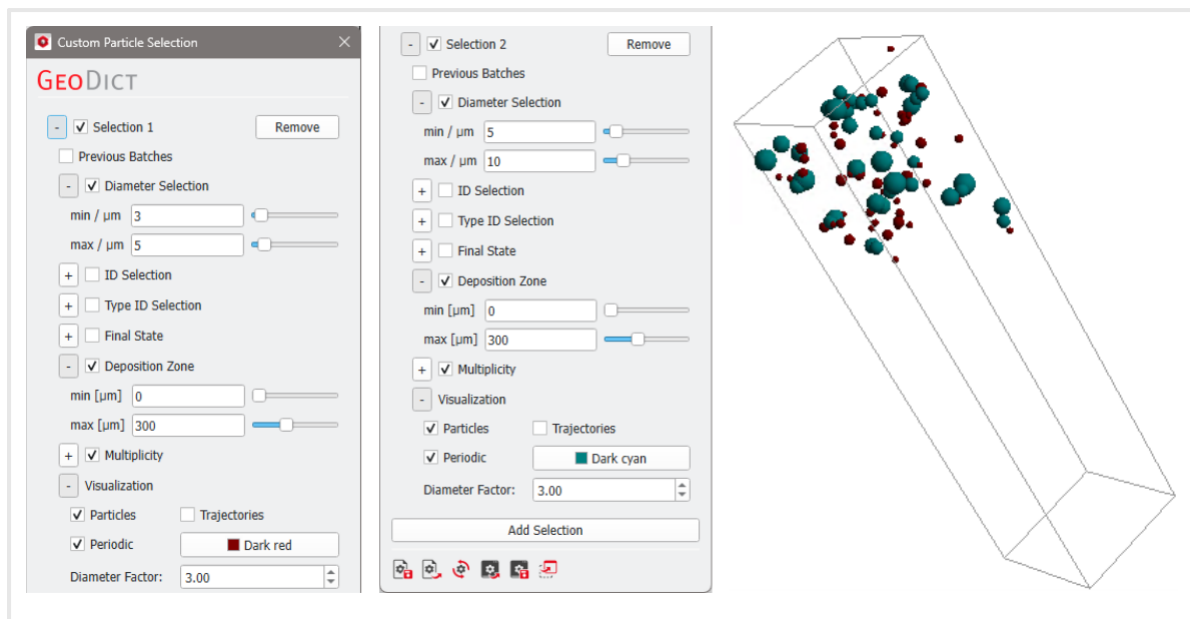
Clicking **Add Selection** again makes a new selection appear (e.g., Selection 2), to combine different visualization selection models. It is useful to observe, for example, particles with two different diameter ranges (through **Diameter Selection**).

As an example, here, we show the settings to visualize particles with a diameter from 3 μm to 5 μm in , and particles with a diameter of 5 μm to 10 μm in .

Particles with a diameter larger than 10 μm and smaller than 3 μm are not shown at all.

Here, only particles with a **Deposition Zone** between 0 μm and 300 μm will be visualized.

The particles are scaled with **Diameter Factor** 3 to see the particles better.



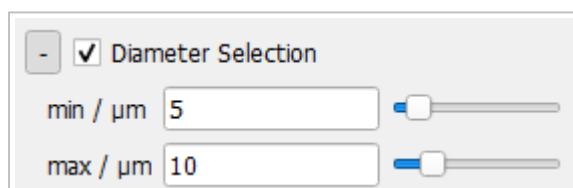
✓ Previous Batches

If **Previous Batches** is unchecked, the selection applies only to the current batch.

When **Previous Batches** is checked, the selection only applies to the particles deposited in previous batches. The selection does not apply for the current batch (the particles currently in motion in the visualization).

✓ Diameter Selection

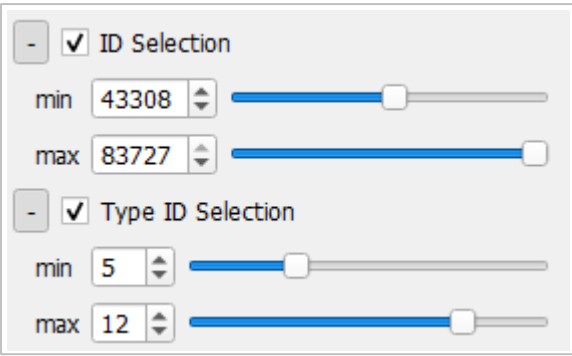
Checking **Diameter Selection** limits the visualized particles to those with a particle diameter in the range specified by min and max. This option only has an effect if the particles exhibit a particle size distribution.



✓ ID Selection and Type ID Selection

Every particle has a specific **ID** and a specific **Type ID**. Every particle is counted and has its own specific **ID**. Furthermore, every diameter has its **Type ID**, which means that if you have 30 different diameters, there are 30 different Type IDs.

This option limits the visualized particles to those with a particle ID / Type ID in the range specified by min and max.



☒ ID Selection

min 43308

max 83727

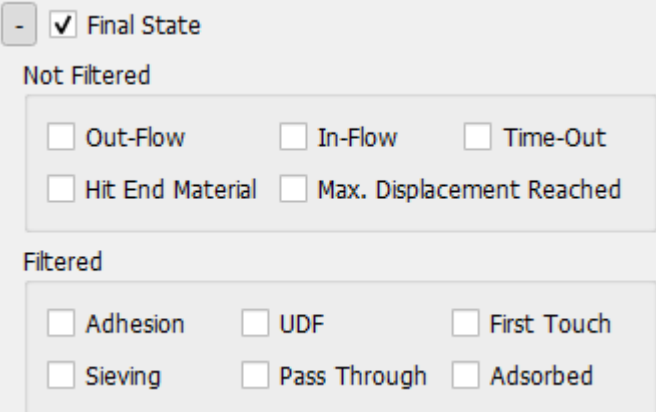
☒ Type ID Selection

min 5

max 12

✓ Final State

Final State enables the visualization of particles that obtained a particular state at the end of the simulation.



☒ Final State

Not Filtered

☐ Out-Flow ☐ In-Flow ☐ Time-Out

☐ Hit End Material ☐ Max. Displacement Reached

Filtered

☐ Adhesion ☐ UDF ☐ First Touch

☐ Sieving ☐ Pass Through ☐ Adsorbed

In the **Not Filtered** panel, particles that did not get attached to the filter/porous medium are selected as follows:

- **Out-Flow** denotes particles that left the computational domain via the fluid outflow region.
- **In-Flow** designates particles that left the computational domain via the fluid inflow region.
- **Time-Out** identifies all particles which are still moving at the end of the time interval that corresponds to this batch.
- **Hit End Material** only shows particles which hit the end material. This only applies if the **Particle End Position** in the simulation is set to material ID and not to the outflow plane.
- The **Max. Displacement reached** state can only be achieved in AddiDict.

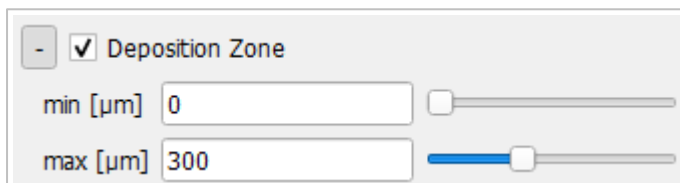
The **Filtered** panel contains the options for particles that became attached to the filter/porous medium:

- **Adhesion** selects all particles that are filtered by the adhesion (Hamaker) model.

- **UDF** selects all particles that are filtered by a model from a user defined function, which sets the `trapCode` to 33 (see Collision UDF). UDFs are only available in FilterDict.
- **First Touch** selects particles that are filtered by the caught on first touch model.
- **Sieving** selects particles that are filtered by the sieving model.
- **Pass Through** selects particles that are filtered by the pass-through model.
- **Adsorbed** selects particles that were adsorbed in a porous medium. The adsorption model is only available in AddiDict.

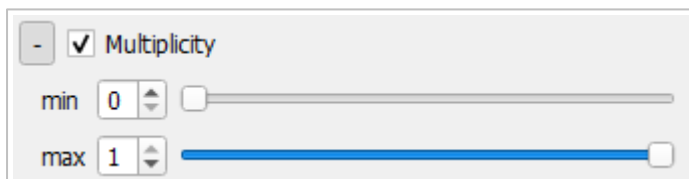
✓ Deposition Zone

The **Deposition Zone** defines a location in the computational domain where the particles must be situated at the end of the simulation to be eligible for visualization. This zone is delimited by the min / max coordinates of the Cartesian direction in which fluid flow occurs, i.e., Z-axis.



✓ Multiplicity

Particles can be selected depending on their multiplicity. Particle Multiplicity is a simulation option for unresolved particles which helps reducing simulation time for very small particles. For detailed information see the [FilterDict](#) chapter.



✓ Visualization

Checking **Visualization** allows more adjustments for the visualization of simulation results.

- Visualization

☒ Particles

☐ Trajectories

☒ Periodic

Dark cyan

Diameter Factor:

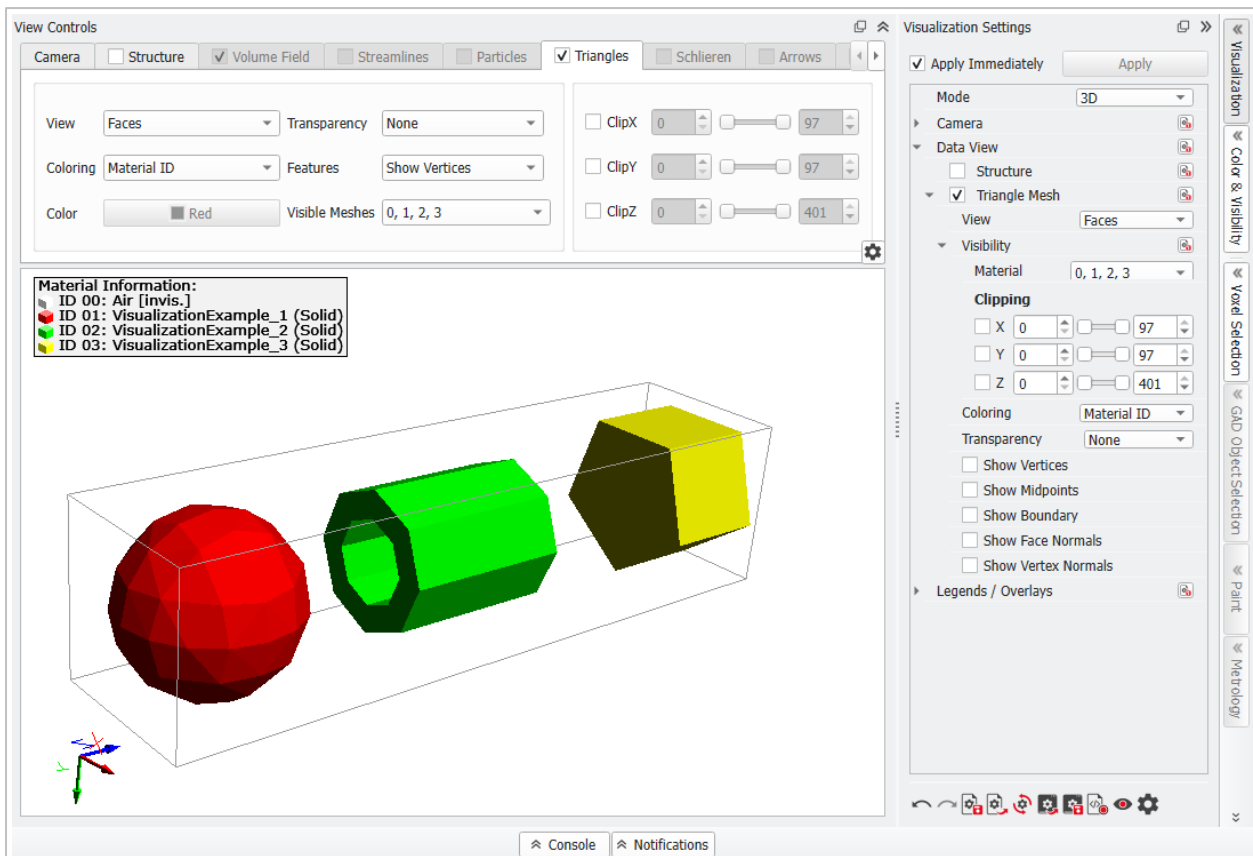
3.00

- Checking **Particles** displays all particles that conform to the options previously set in the dialog.
- Checking **Trajectories** displays the trajectories which correspond to the selected particles. With this option, it is (for example) possible to show only the trajectories which correspond to a certain particle type.
- Checking **Periodic** enables periodic boundaries for the particle movements display. Particles which leave the domain at one side re-enter at the opposite side.
- The default **color** green for the particles can be changed to another color through the button (for example,  Dark cyan).
- The **Diameter Factor** scales the particles sizes relative to their original size for visualization purposes and enables the display of particles that are nominally below the visibility of the visualization settings. The diameter of all particles moving in the simulation *artificially* grows by this factor when visualizing them.

Triangles

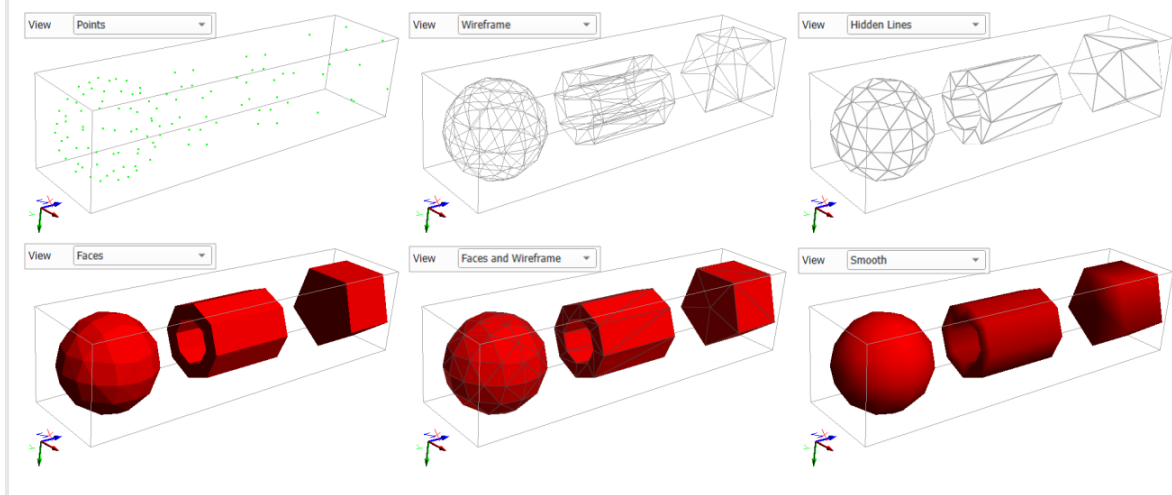
For structures with surface triangulation meshes in stl format, e.g., a cylindrical pleat generated with PleatGeo or structures imported with ImportGeo-CAD the **Triangles** tab becomes available. Modifying the parameters allows to visualize the triangles in different ways.

When visualizing triangulated surfaces, and to observe the difference between the **View Mode** and **Coloring** choices, the visualization of the voxelized structure must be switched off by unchecking the [Structure](#)  tab.



✓ View

Select between different **View** modes. If setting the view to none, no triangles are shown at all.



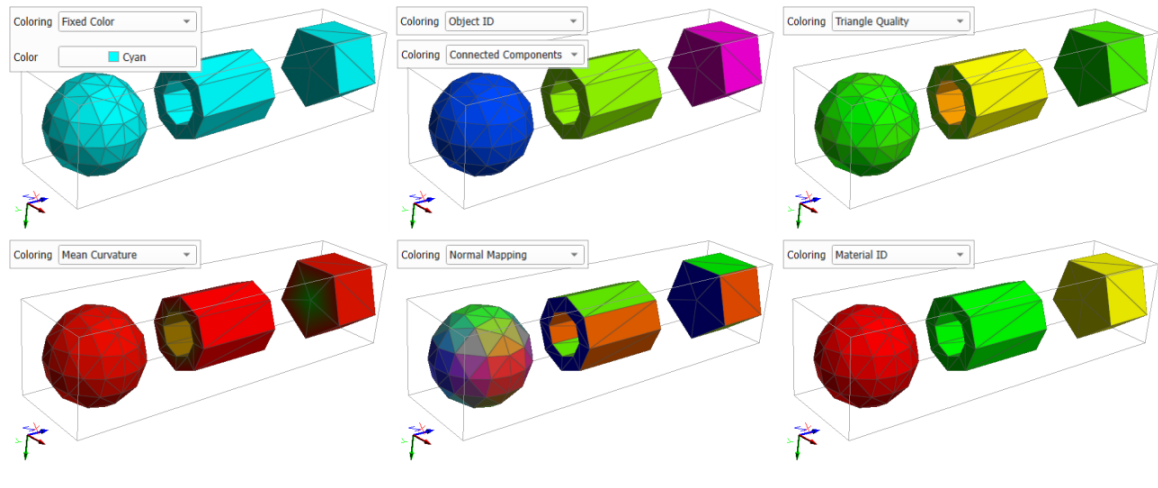
✓ Coloring

Select between different **Coloring** schemes.

For the first coloring scheme, fixed color, you can select a desired color for the triangle faces by clicking on the **Color** button below.

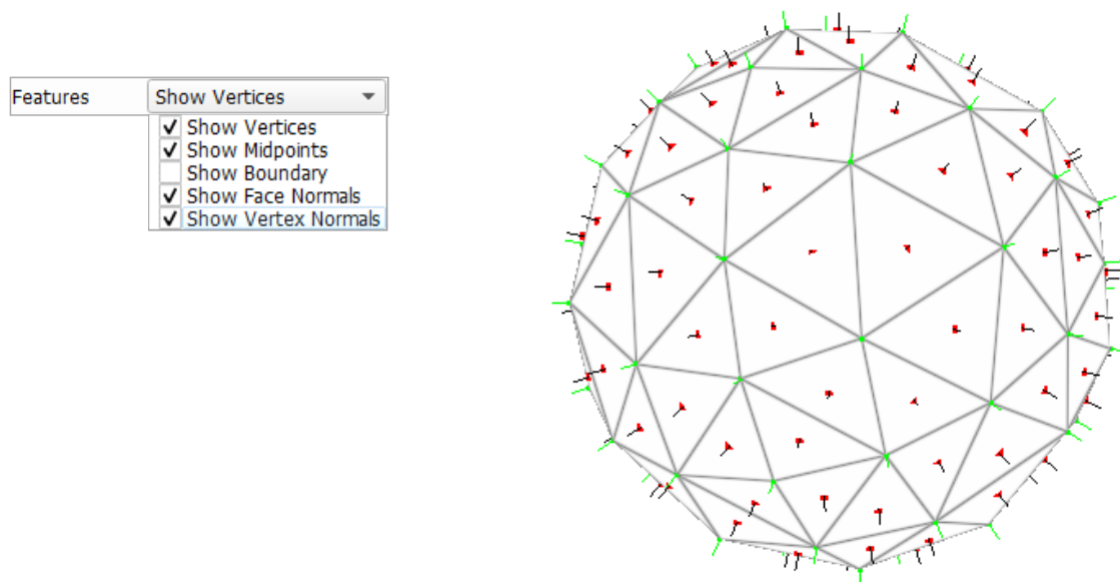
In this example each object ID is a single component. Therefore, in this case the coloring schemes **Connected Components** and **Object ID** are identical.

The coloring scheme **Material ID** depends on the colors chosen in the [Color & Visibility](#) Settings for the structure.



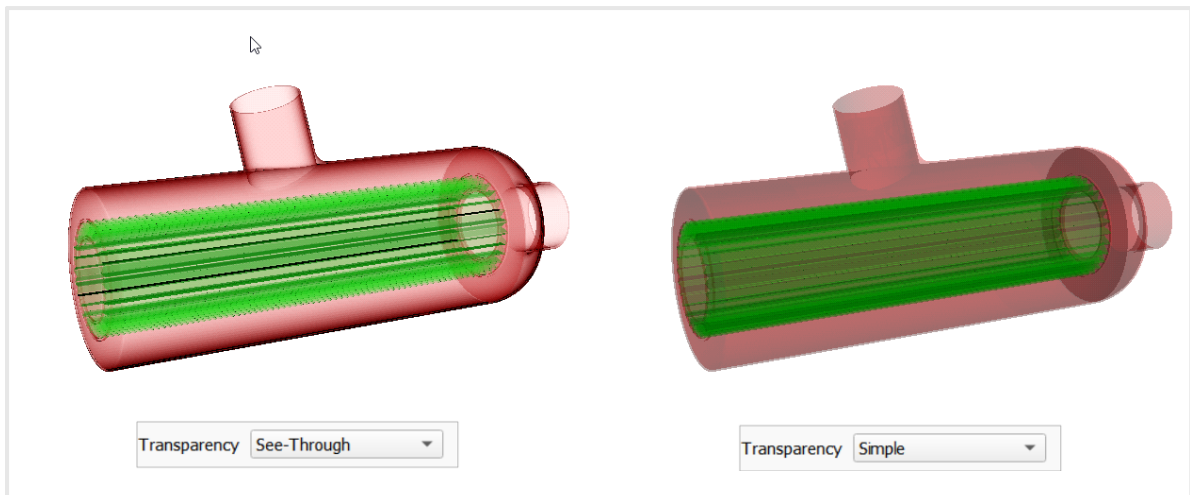
✓ Features

The **Features** pull-down menu allows to visualize vertices, midpoints, and face and vertex normals.



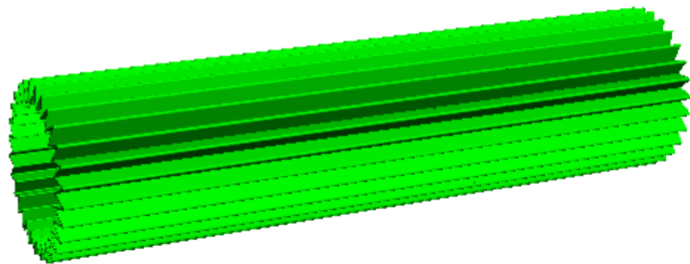
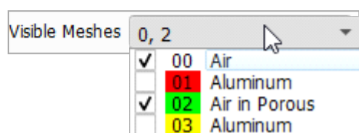
✓ Transparency

Two different **Transparency** modes are available, which allow to visualize the inner parts of the structure:



✓ Visible Meshes

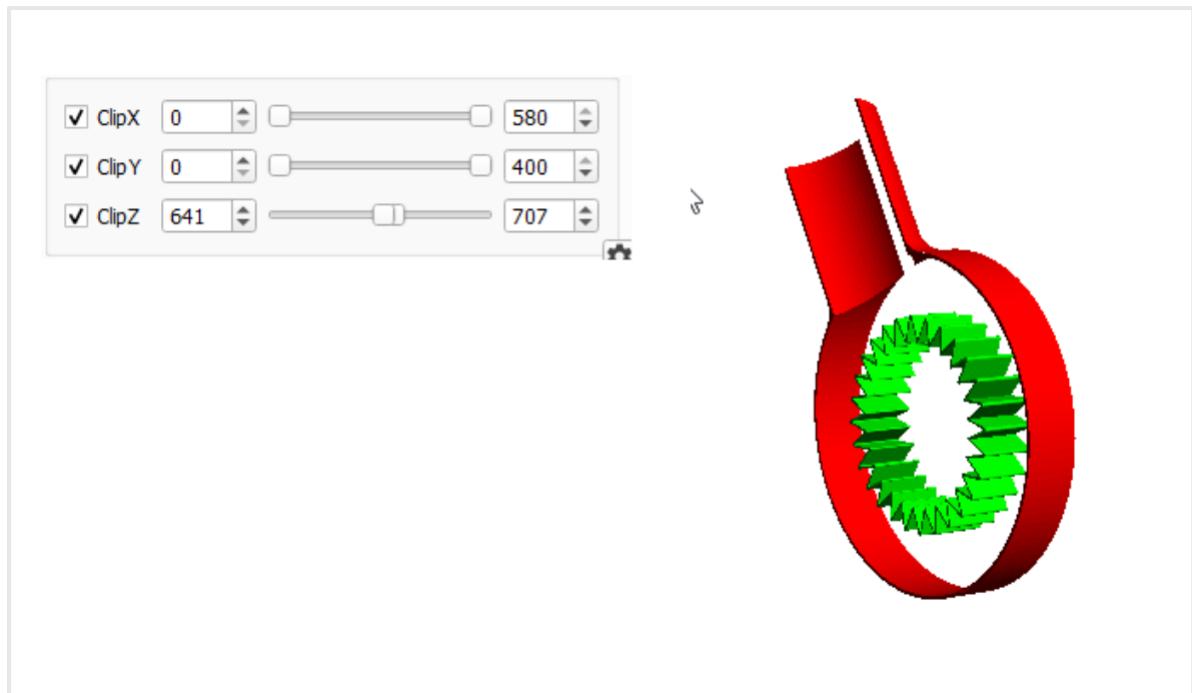
With the **Visible Meshes** pull-down menu, the visualization of every imported mesh can individually be switched on and off.



✓ Clipping

The triangles can be clipped conforming to their spatial position, in a similar way as explained for the [structure clipping](#) ⁵⁶.

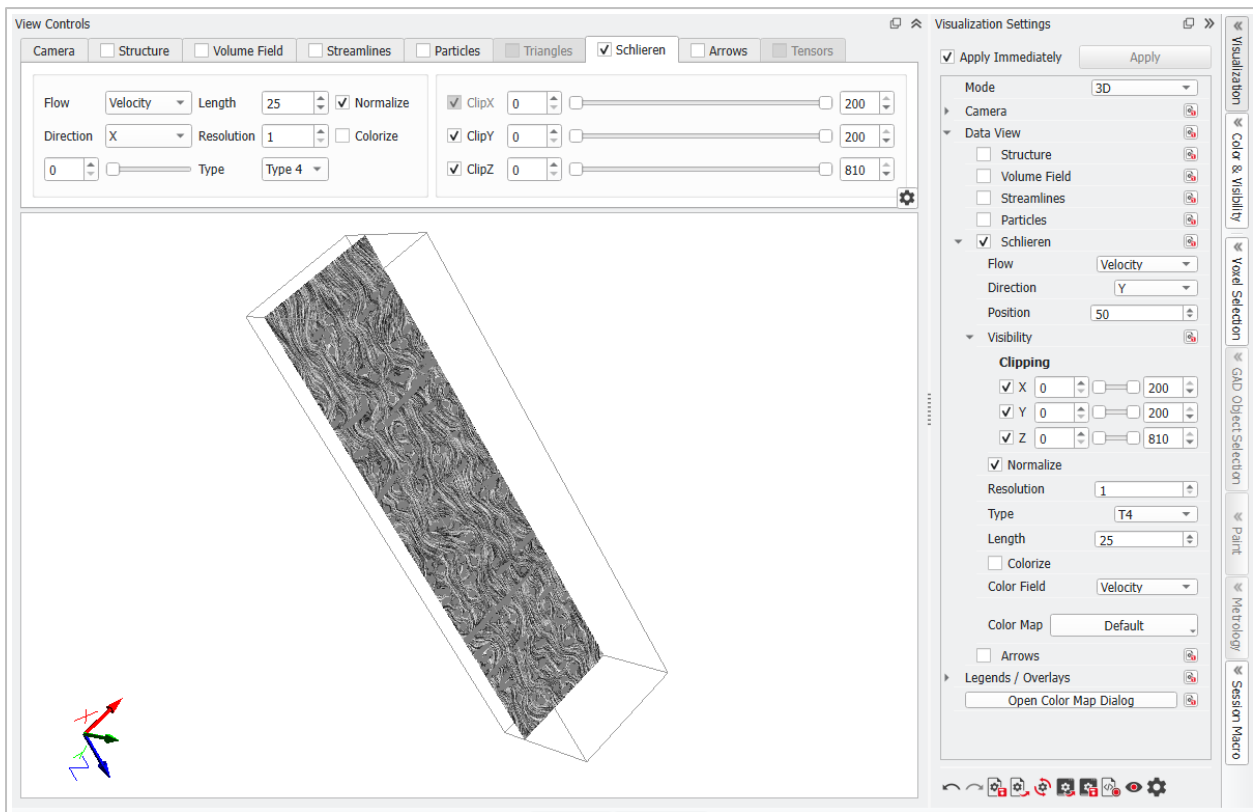
It is possible to define the range of voxels in each spatial direction that are to be displayed by moving the slider or by direct input of the value.



Schlieren

Schlieren (Line Integral Convolution (LIC)) is another possibility to visualize the fluid flow field provided in FlowDict, AddiDict and FilterDict. LICs are visually displayed as distorted striations or streaks in 2D sections that represent fluid motion through the 3D flow field, in any of the three directions.

The **Schlieren** tab controls the visibility of the schlieren. Some additional options and all the options from the **View Controls** can also be found in the **Visualization Settings** dialog. Expand the **Visualization Settings** dialog either by clicking the gear wheel icon in the bottom right of the Schlieren tab or by opening the **Visualization tab** in the sidebar and unfolding the **Data View** → **Schlieren** section.

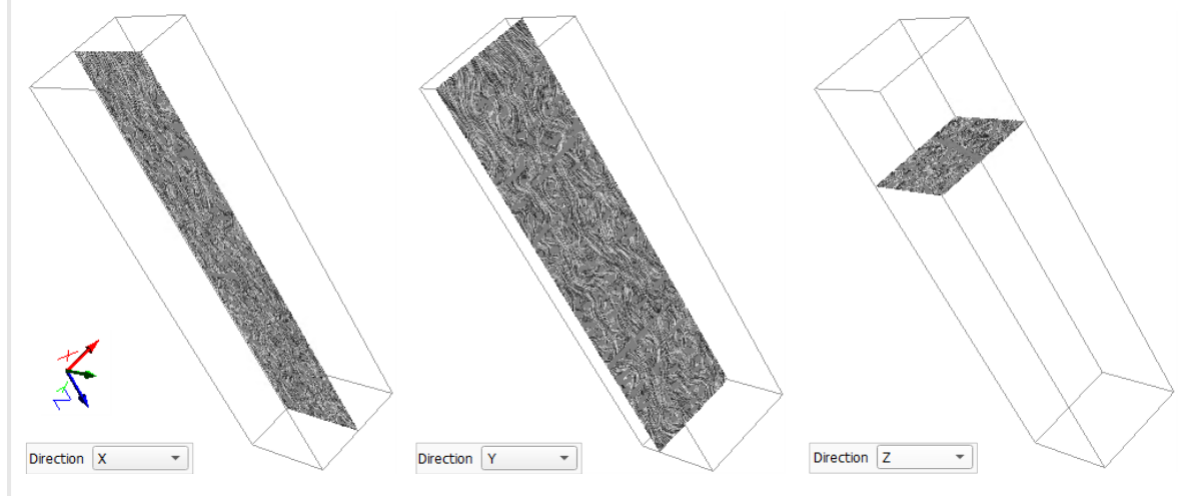


✓ Flow

The **Flow** pull-down menu shows one choice for each loaded [vector field](#)⁴⁴. For a flow field for example, only the field Velocity is available. The Schlieren are computed from the selected field.

✓ Direction

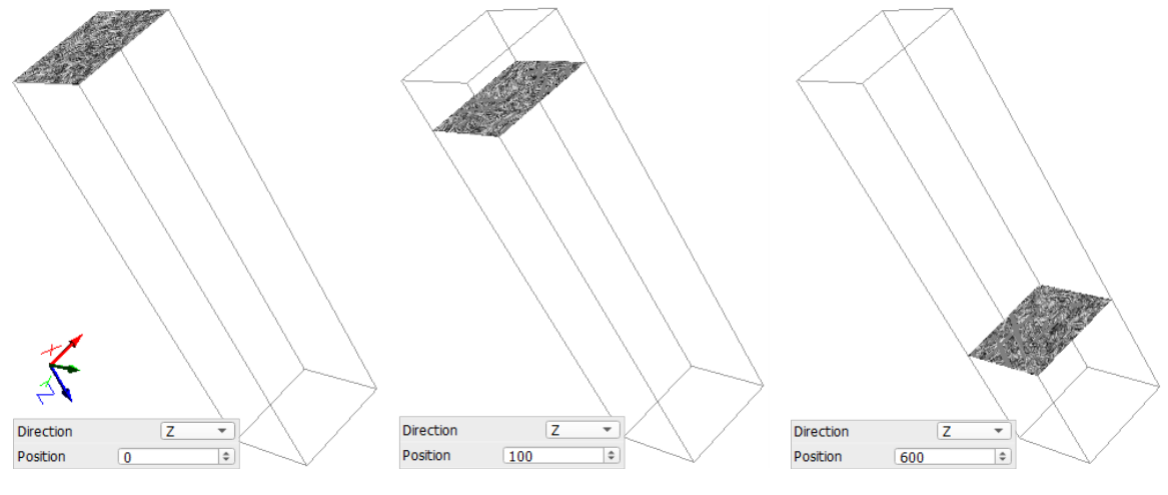
Direction defines the location of the schlieren 2D section in the three directions.



✓ Position

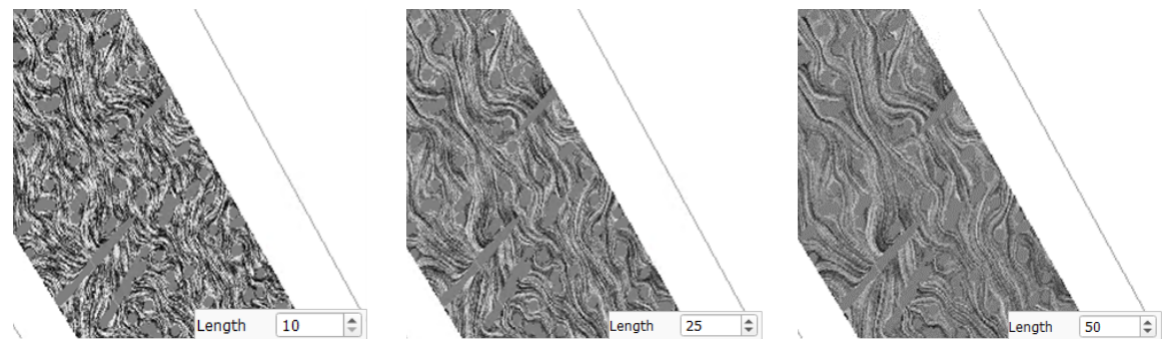
The **Position** of the 2D schlieren section can be changed either by moving the slider below **Direction** in the Schlieren tab or by entering different values in the **Visualization Settings** dialog (**Data View - Schlieren - Position**)

For example, to examine the schlieren along the Z-direction, change the **Position** value as indicated:



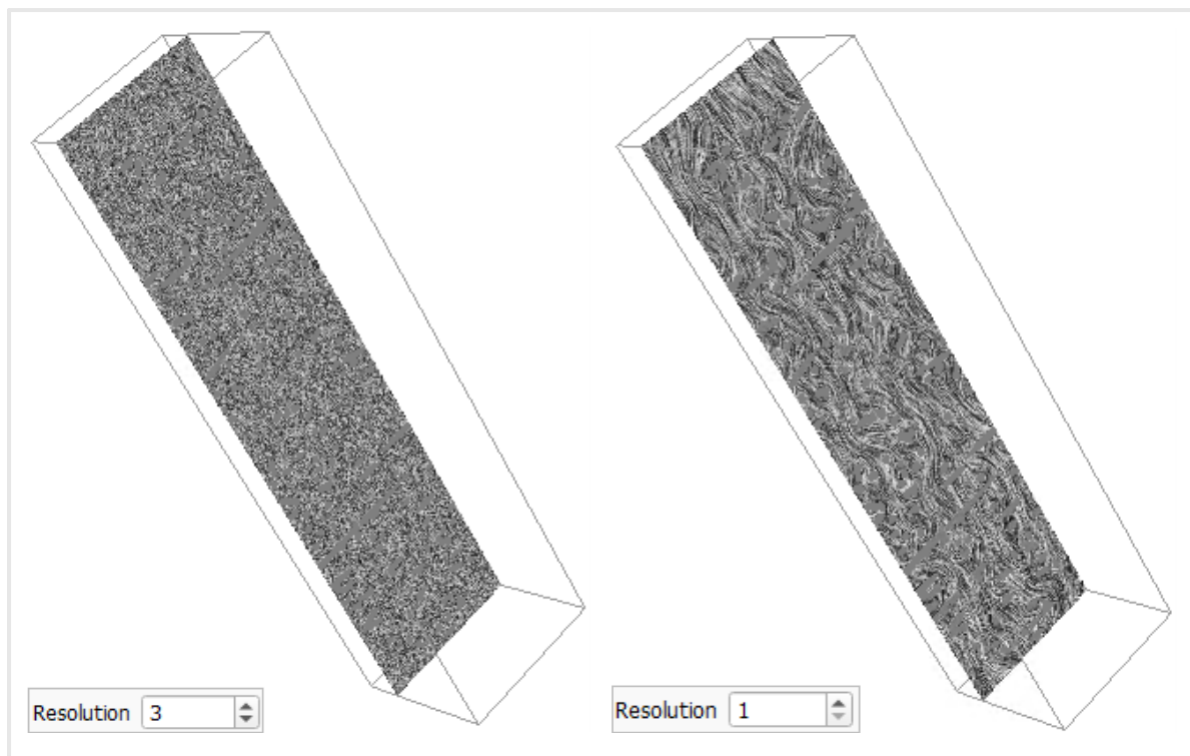
✓ Length

Length defines the elongation of the individual LIC lines.



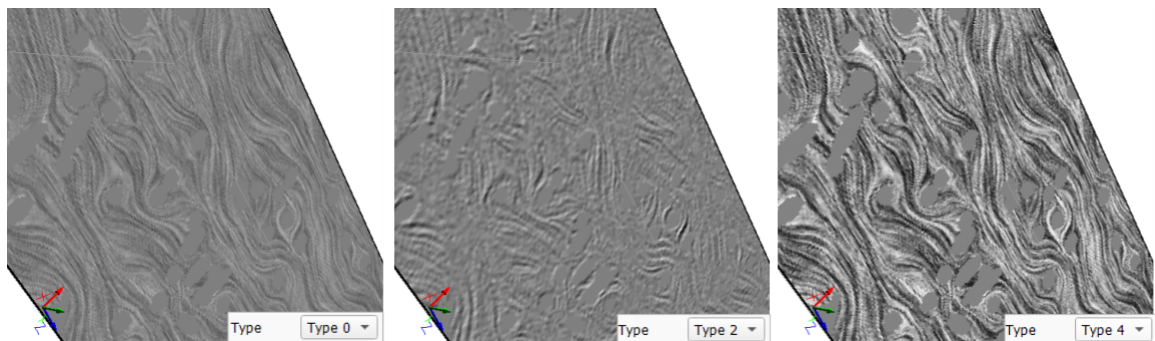
✓ Resolution

Resolution specifies the number of pixels in one spatial direction that represent the related voxel.



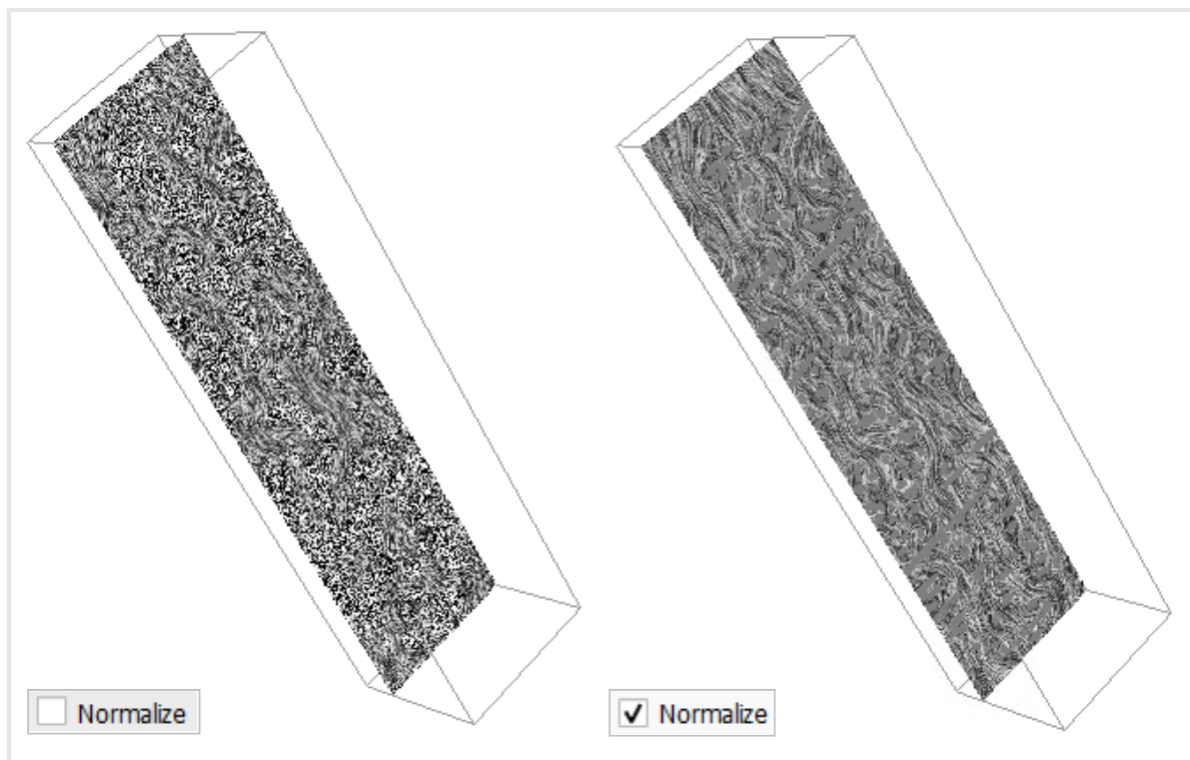
✓Type

Type allows predefined post-processing of the schlieren by choosing from the pull-down menu. **Type 0** displays the original LIC and **Type 1** to **Type 4** each applies a different sharpening filter to the original LIC.



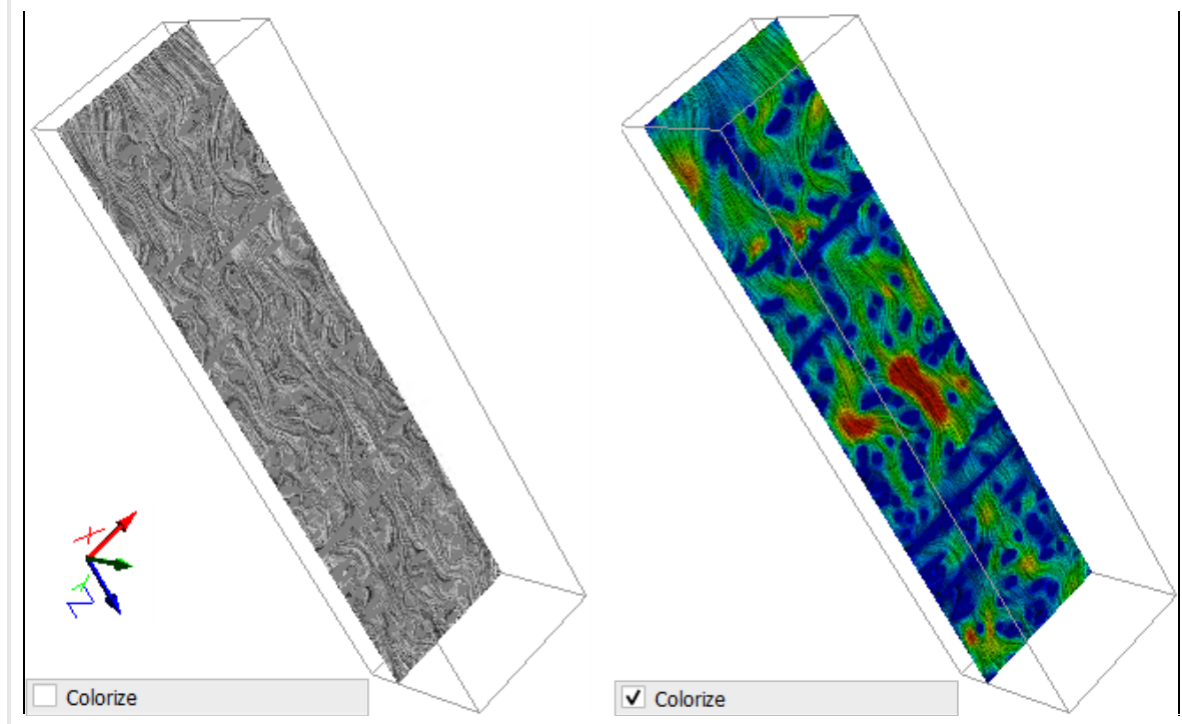
✓Normalize

Checking **Normalize** activates the normalization of the visualization in relation to the maximum fluid velocity.



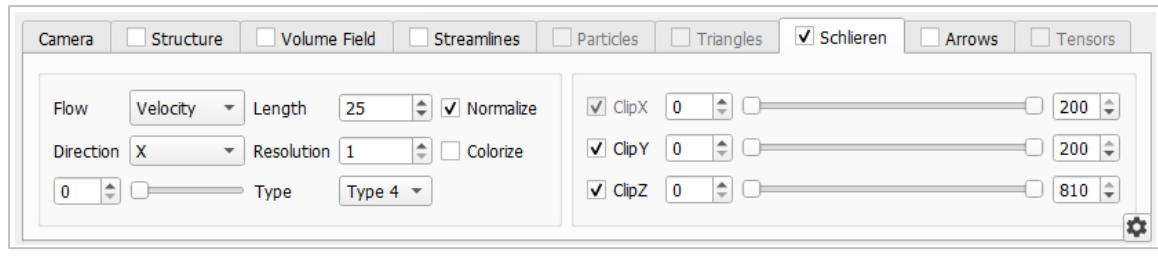
✓ Colorize and Color Field

Colorize defines whether the schlieren are displayed in gray scales or in color. The color then can be selected from the flow fields, available in the pull-down menu for **Color Field** (**Visualization Settings** dialog: **Data View - Schlieren - Color Field**).



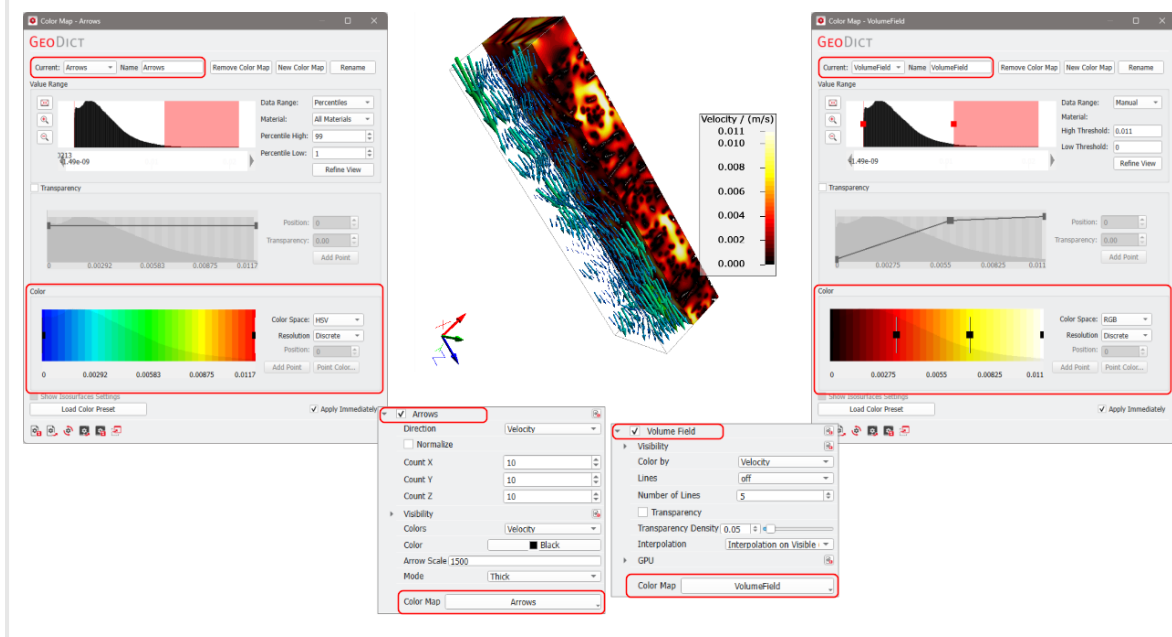
✓ Clipping

Clipping works similar as for the [structure](#)^[56] and the other fields, but only two directions are available for clipping, depending on the chosen direction.



✓ Color Map

Define different color maps in the [Color Map](#)^[121] dialog. From the pull-down menu for **Color Map** in the **Visualization** tab on the right you can then select between your [custom color maps](#)^[122]. You can define and select a different color map for each of the components. For example, visualize arrows with the jet [color preset](#)^[125] and the volume field with the hot preset:



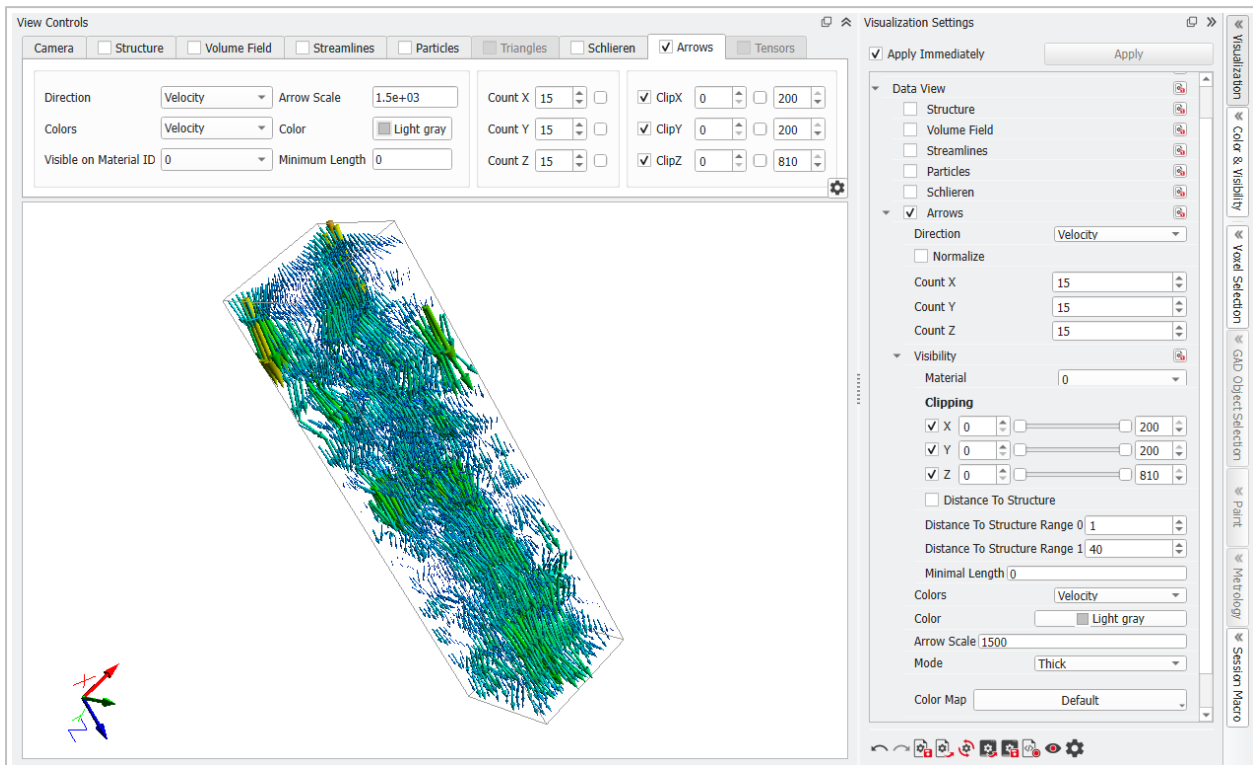
Arrows

One flow field visualization option is representing the volume field by an arrow field.

Here, the properties of the fluid flow field are imaged at all positions of a regular grid that do not belong to the solid. At each of these positions, the local orientation of the flow field is indicated by a vector and its length and color are coded for the fluid velocity. Small vectors indicate a slow fluid velocity and long vectors indicate a high fluid velocity. Vectors are located with their middle in the center of the voxel for which they represent the flow velocity.

The **Arrows** tab controls the visibility of the arrows. Some additional options and all the options from the **View Controls** can also be found in the **Visualization Settings** dialog.

Expand the **Visualization Settings** dialog either by clicking the gear wheel icon in the bottom right of the Arrows tab or by opening the **Visualization** tab in the sidebar and unfolding the **Data View** → **Arrows** section.

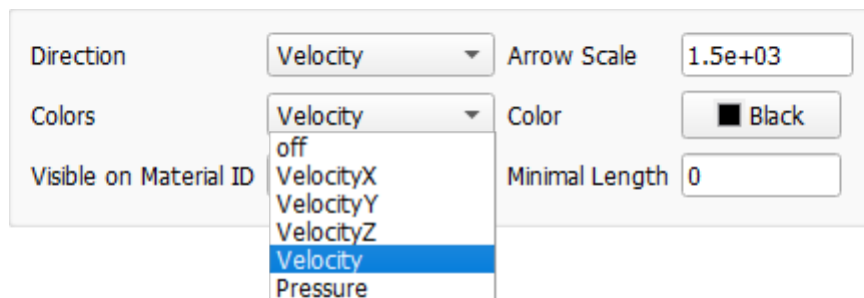


✓ Direction

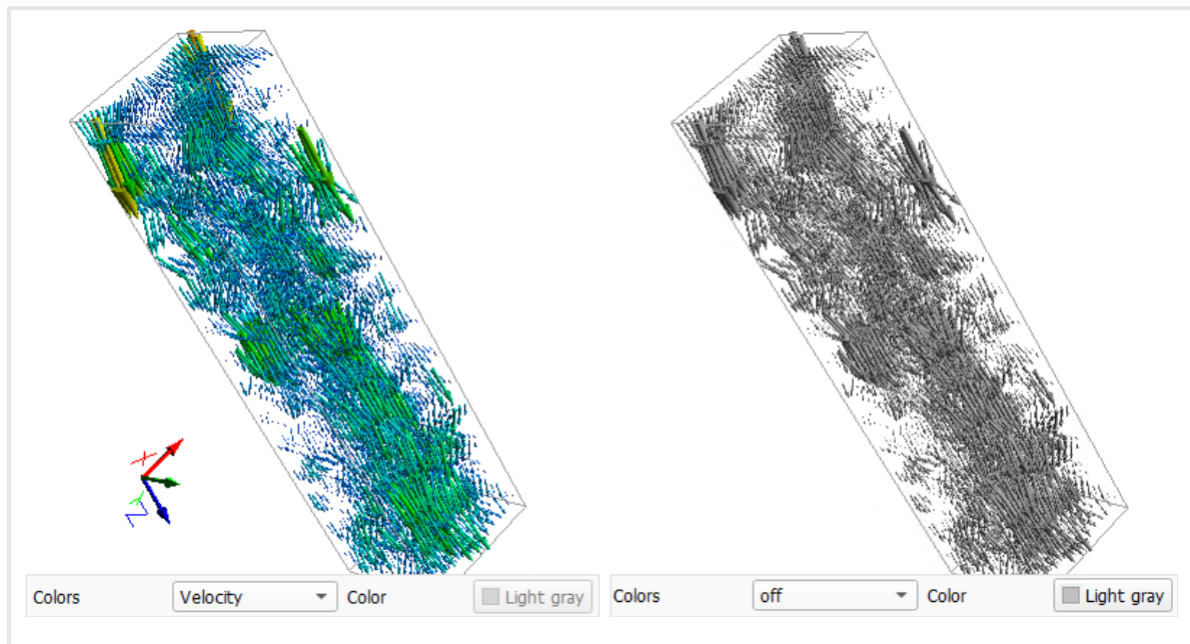
The **Direction** pull-down menu shows one choice for each loaded [vector field](#)⁴⁴. For a flow field for example, only the field Velocity is available. The **Direction** field is computed from the selected field.

✓ Colors

The **Colors** pull-down menu applies these properties to the vectors resulting from the simulation. **Colors** are determined by the property value at the corresponding position. **Colors** can be assigned to all loaded [components](#)⁴⁴.



If **Colors** is turned **off**, a custom **Color** can be applied to all arrows by clicking on the color button.



✓ Visible on Material IDs

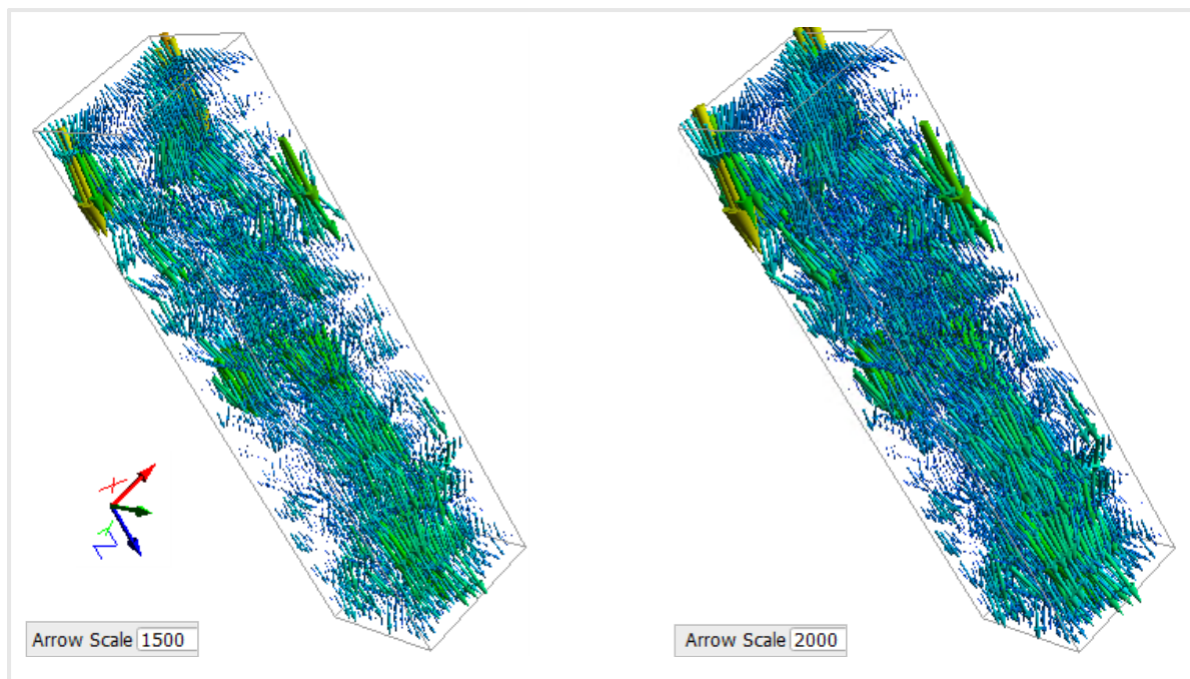
Visible on Material IDs defines the material IDs in which the arrow field is shown. This allows to visualize the flow also in porous media where fluid flow is computed according to the Stokes-Brinkman equations (porous voxels / solid voxels / pores). Several material IDs can be selected at the same time.

Direction	Velocity	Arrow Scale	1.5e+03
Colors	Velocity	Color	Black
Visible on Material ID	0, 1	Minimal Length	0

☒	00	Air
☒	01	Glass

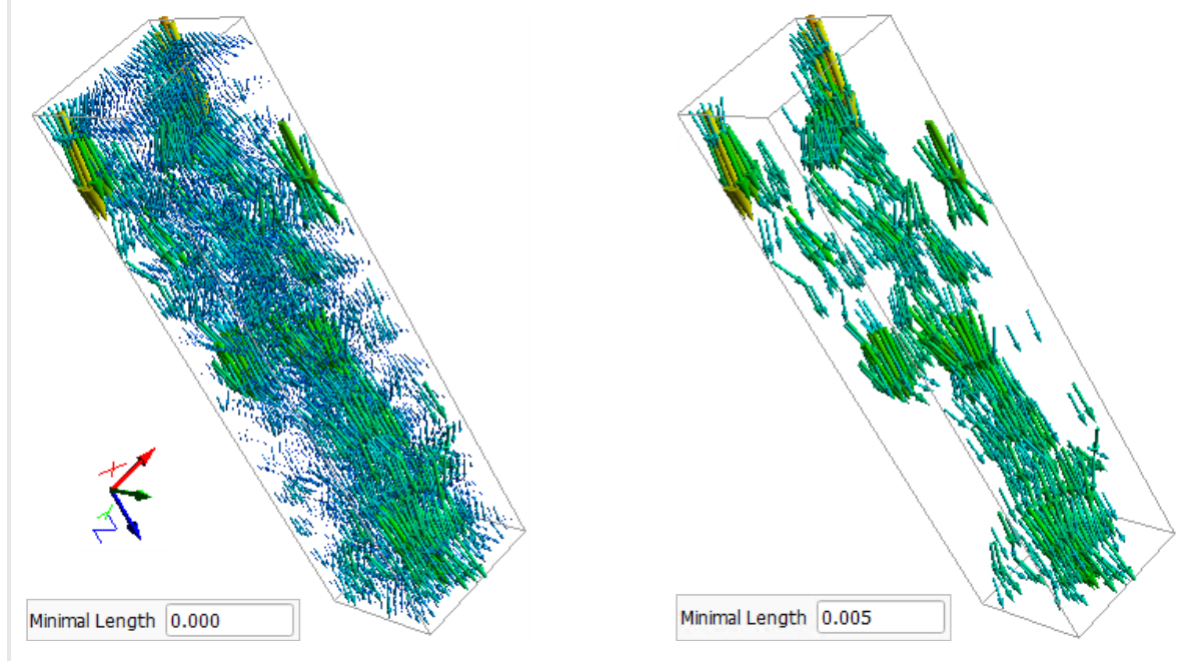
✓ Arrow Scale

The **Arrow Scale** scales the size of the arrows as a percentage of the maximum dimension (here, velocity).



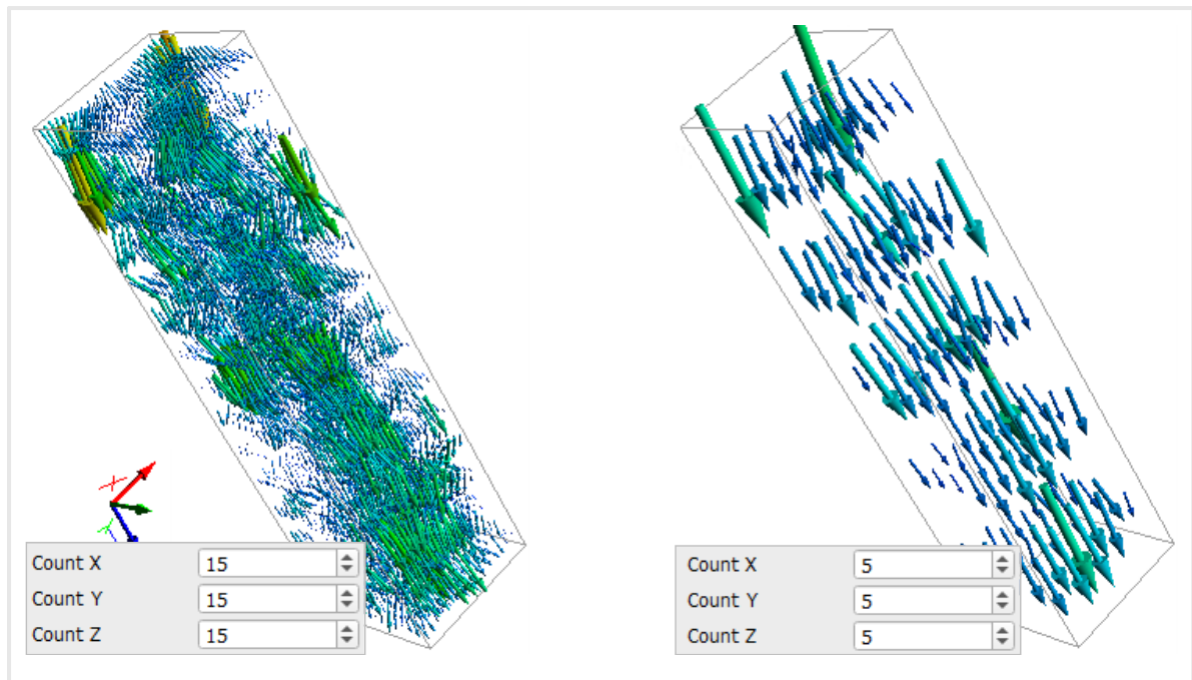
✓ Minimal Length

Minimal Length sets the minimum length used for the slowest arrow, before scaling. The values are limited to those between the maximum and the minimum in the **Plot Range** (here, between 0.00 and 0.012 m/s). This parameter is available in the **View Controls** and in the **Visualization Settings** dialog (under **Visibility**).



✓ Count

Count X, **Count Y**, and **Count Z** define the number of arrows for each axis.

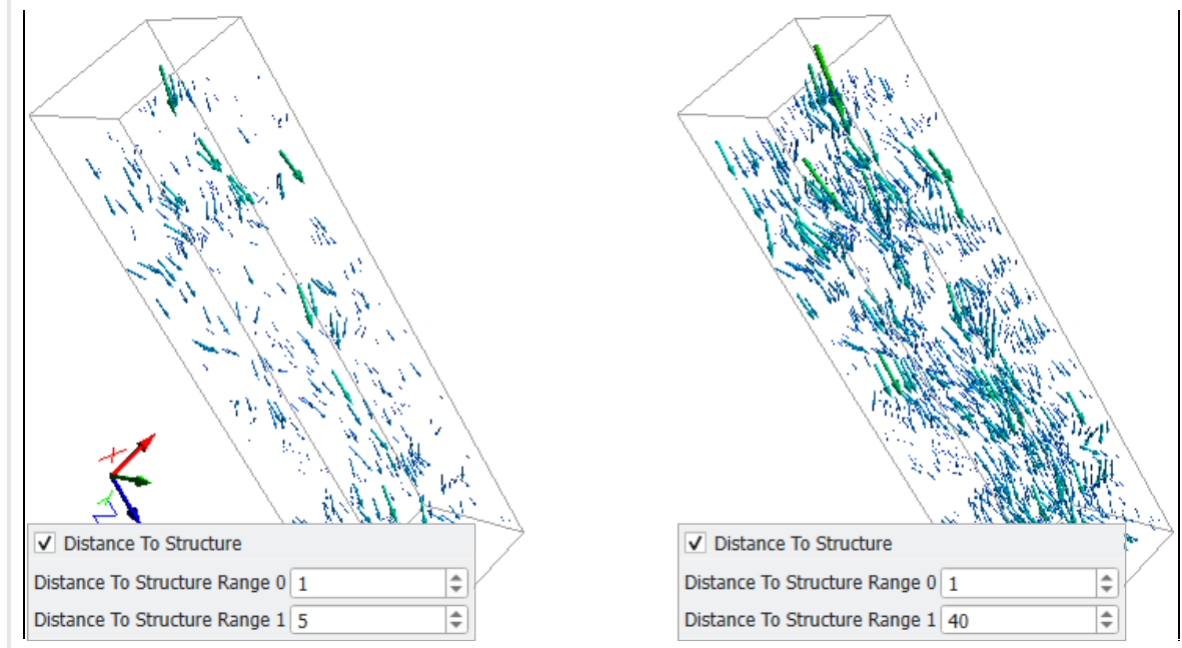


✓ Clipping

Clipping works in the same way as seen for the [structure](#) ⁵⁶.

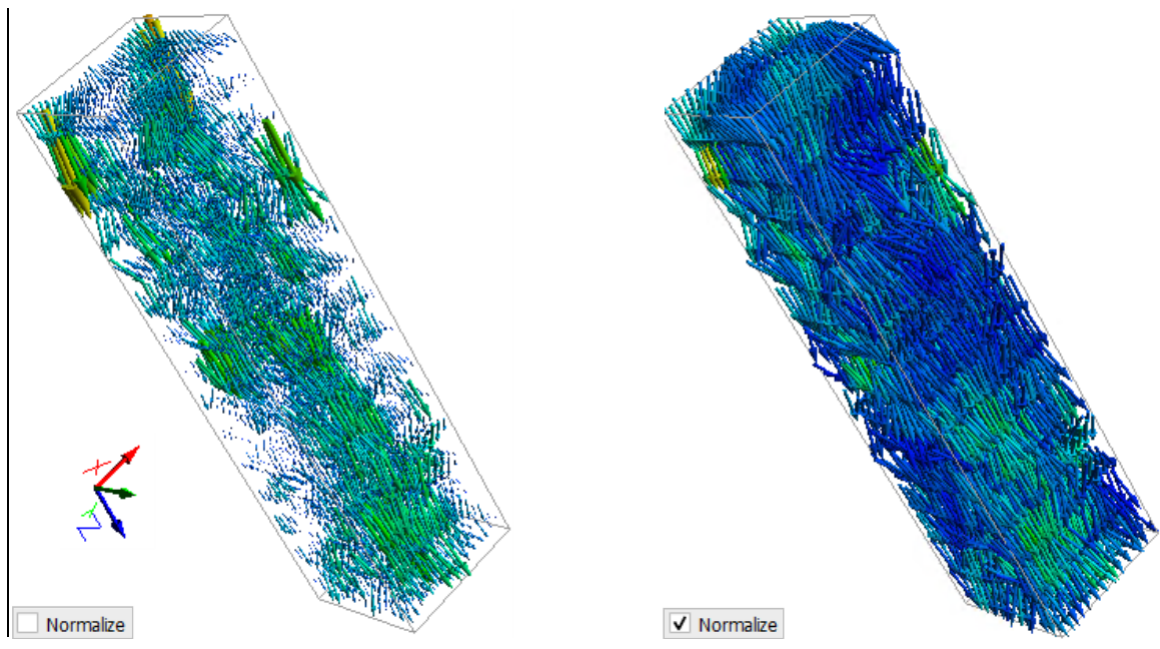
✓ Distance to Structure

In the **Visualization Settings** under **Visibility** find also the option to set a **Distance To Structure**. This defines a space around the solid structure model that is to remain void of arrows. If enabled, **Distance To Structure Range 0** and **Distance To Structure Range 1** give the range of distance where the vectors are shown (in voxels).



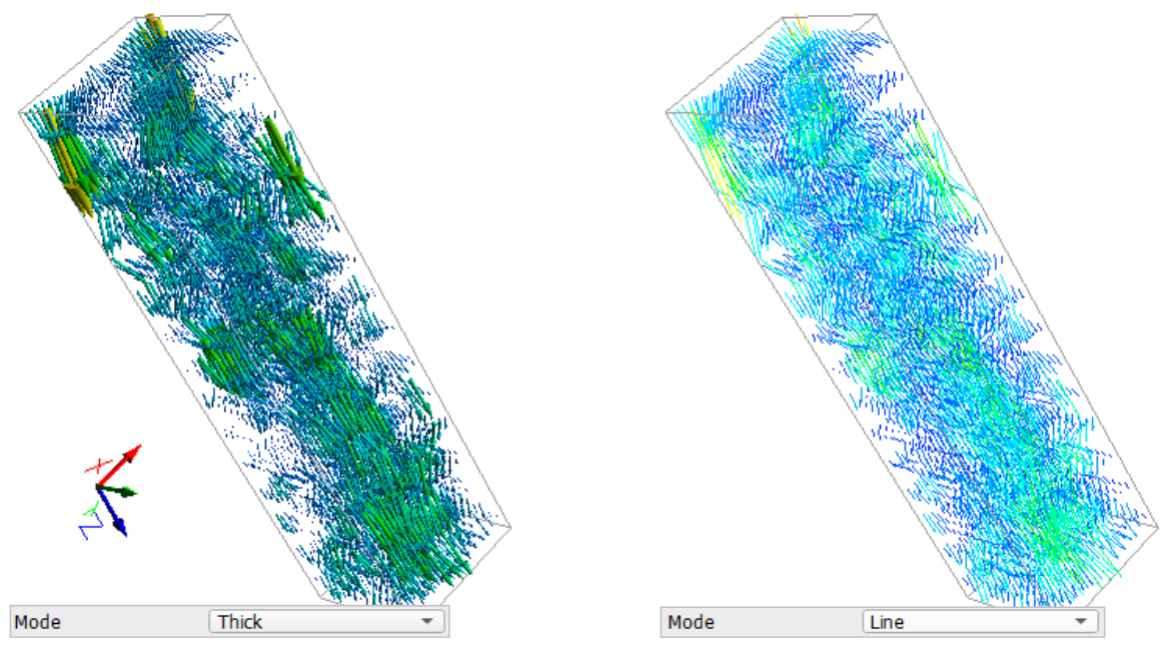
✓ Normalize

Checking **Normalize** visualizes all arrows in the same size. This option is only available from the **Visualization Settings** dialog.



✓ Mode

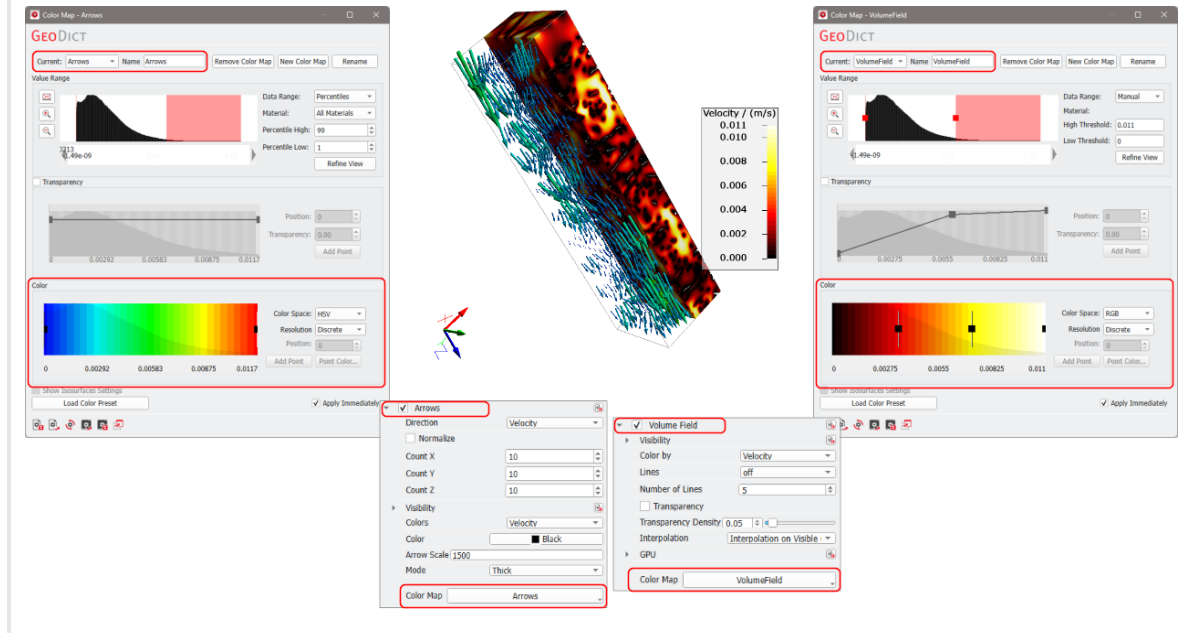
Mode changes the display of the arrows to appear as a **Line** or **Thick**. This option is only available from the **Visualization Settings** dialog.



✓ Color Map

Define different color maps in the [Color Map](#)¹²¹ dialog. From the pull-down menu for **Color Map** in the **Visualization** tab on the right you can then select between your

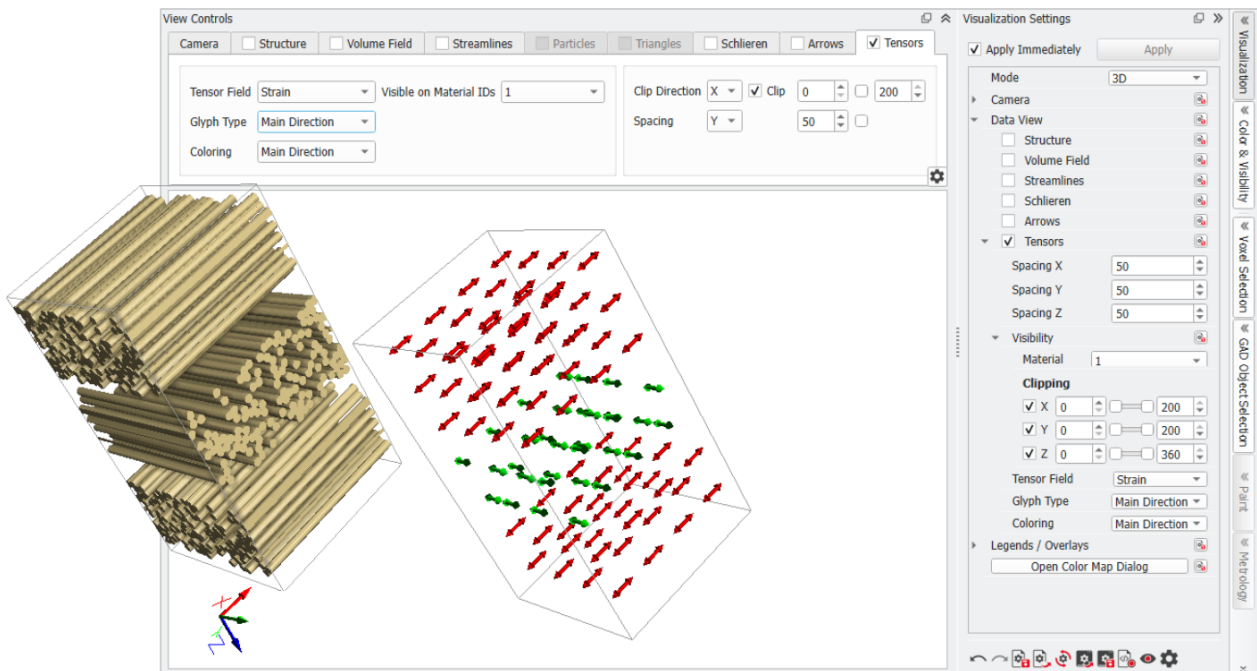
[custom color maps](#)^[122]. You can define and select a different color map for each of the components. For example, visualize arrows with the jet [color preset](#)^[125] and the volume field with the hot preset:



Tensors

After performing a simulation which produced stress or strain volume fields, you can visualize the tensor field. The simulations in ElastoDict and BatteryDict-Degradation produce these fields when selected under **Output Options**.

The **Tensors** tab controls the visibility of the tensors. Some additional options and all the options from the **View Controls** can also be found in the **Visualization Settings** dialog. Expand the **Visualization Settings** dialog either by clicking the gear wheel icon in the bottom right of the Tensors tab or by opening the **Visualization tab** in the sidebar and unfolding the **Data View → Tensors** section.

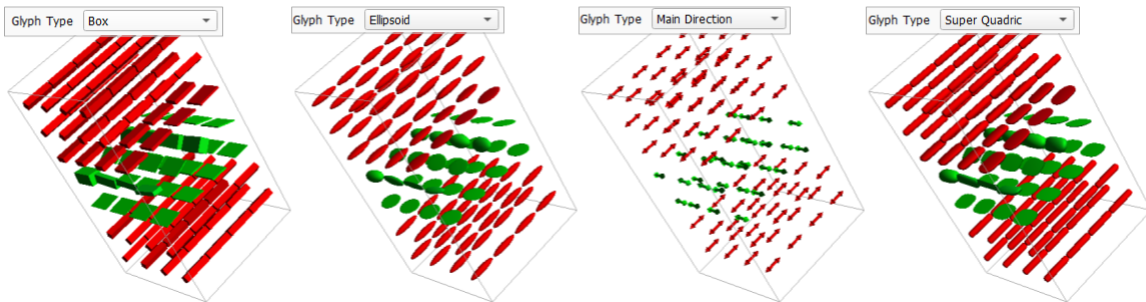


✓ Tensor Field

Select, if the **Tensor Field** should be displayed for **Strain** or **Stress**.

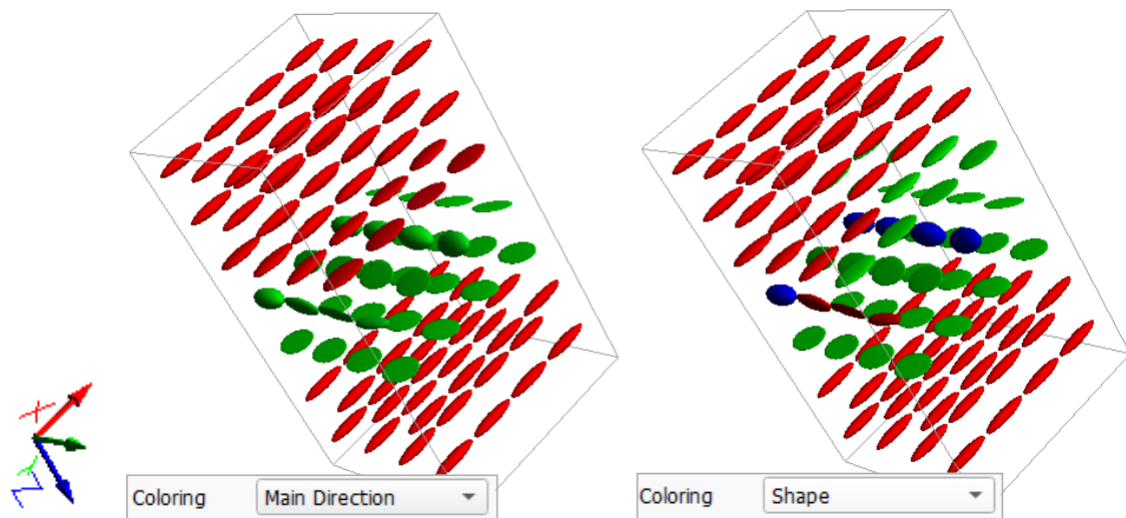
✓ Glyph Type

The **Glyph Type** can be set to **Box**, **Ellipsoid**, **Main Direction** or **Super Quadric**.



✓ Coloring

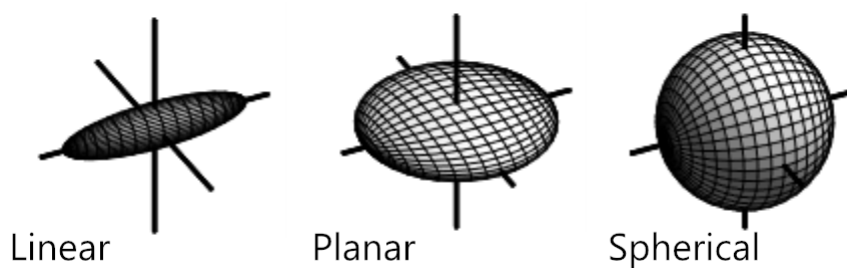
The **Coloring** can either be set to **Main Direction** or **Shape**.



For **Main Direction**, the main eigenvector is interpreted as RGB, i.e., red, green, and blue. An eigenvector for strain or stress is the direction (x,y,z) of the largest principal strain or the largest principal stress, respectively. If the vector is pointing in X-direction, the color is red, if it is pointing in Y-direction, the color is green and for the Z-direction it is blue. For everything in between, the colors are interpolated.

For **Shape**, the color depicts the shape of the tensor, i.e., the color depends on whether the tensor is mainly linear, planar, or spherical. If the shape is mostly linear, the color is red, if it is mostly planar, the color is green, and if it is mostly spherical, the color is blue.

The figure below (see also [Kindlmann, 2004](#)^[131]) shows how the certainties (c) for the tensor shape are computed based on the non-negative eigenvalues $\lambda_1 \geq \lambda_2 \geq \lambda_3$ of the tensor. For strain and stress, the three eigenvalues are the three principal strains or stresses.



$$c_l = \frac{\lambda_1 - \lambda_2}{\lambda_1 + \lambda_2 + \lambda_3}$$

(1) Linear - red

$$c_p = \frac{2(\lambda_2 - \lambda_3)}{\lambda_1 + \lambda_2 + \lambda_3}$$

(2) Planar - green

$$c_s = \frac{3\lambda_3}{\lambda_1 + \lambda_2 + \lambda_3}$$

(3) Spherical
- blue

✓ Visible on Material ID

From the pull-down menu **Visible on Material IDs** select in which material IDs the volume field is visualized.

Tensor Field: Strain
Glyph Type: Main Direction
Coloring: Shape

Visible on Material IDs: 1

- ☐ 00 Epoxy ...501-6)
- ☒ 01 Glass

✓ Clipping

Clip the particle visualization area by selecting a **Clip Direction**, enabling **Clip** and moving the corresponding sliders or entering clipping values.

Clip Direction: X
Spacing: X
Y
Z

☒ Clip

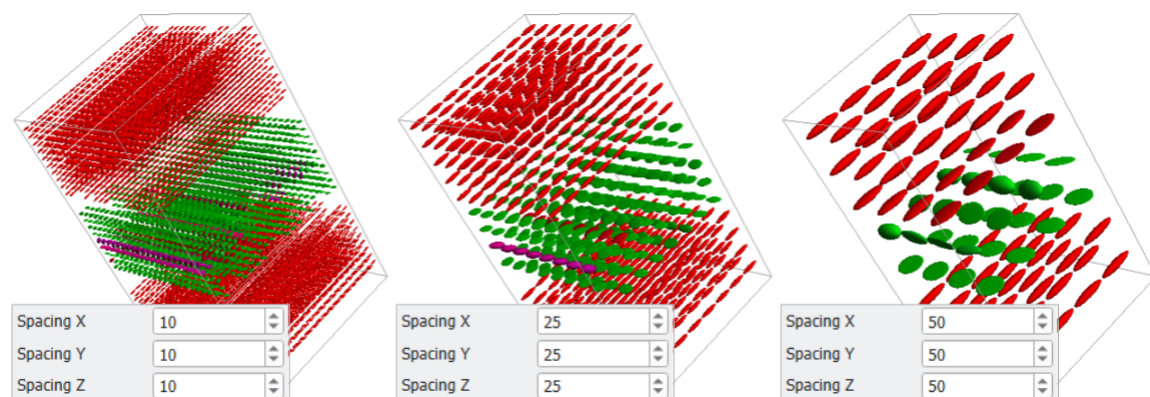
65 200

5

Otherwise, in the **Visualization Settings** dialog, use the option **Data View ► Tensors ► Visibility ► Clipping**, working the same way as described for the [structure](#)⁵⁶.

✓ Spacing

Control the **Spacing** between the tensors in the three directions. Smaller spacing leads to smaller tensors, while larger values for spacing results in larger tensors.



Otherwise, in the **Visualization Settings** dialog, use the option **Data View ► Tensors ► Spacing**.

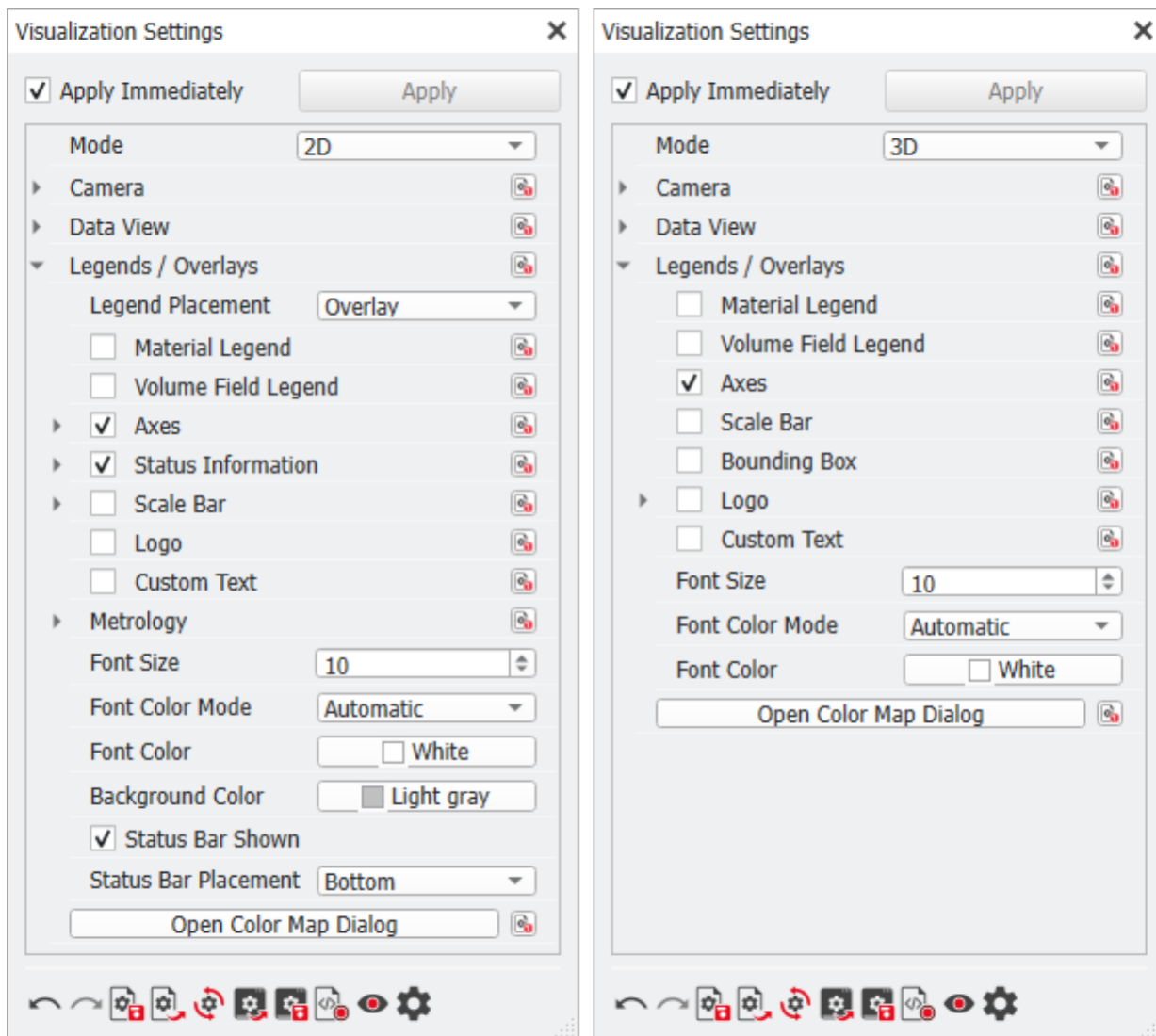
1.2.3 Legends and Overlays

If a structure or a volume field are loaded to GeoDict, the settings of all legends and overlays in the **Visualization area** can be changed. For this, open the **Visualization Settings** dialog by clicking the **Visualization tab** in the GUI **Sidebar** and unfold the section **Legends / Overlays**.

Unchecking the box for a legend or overlay option disables the corresponding legend or overlay.

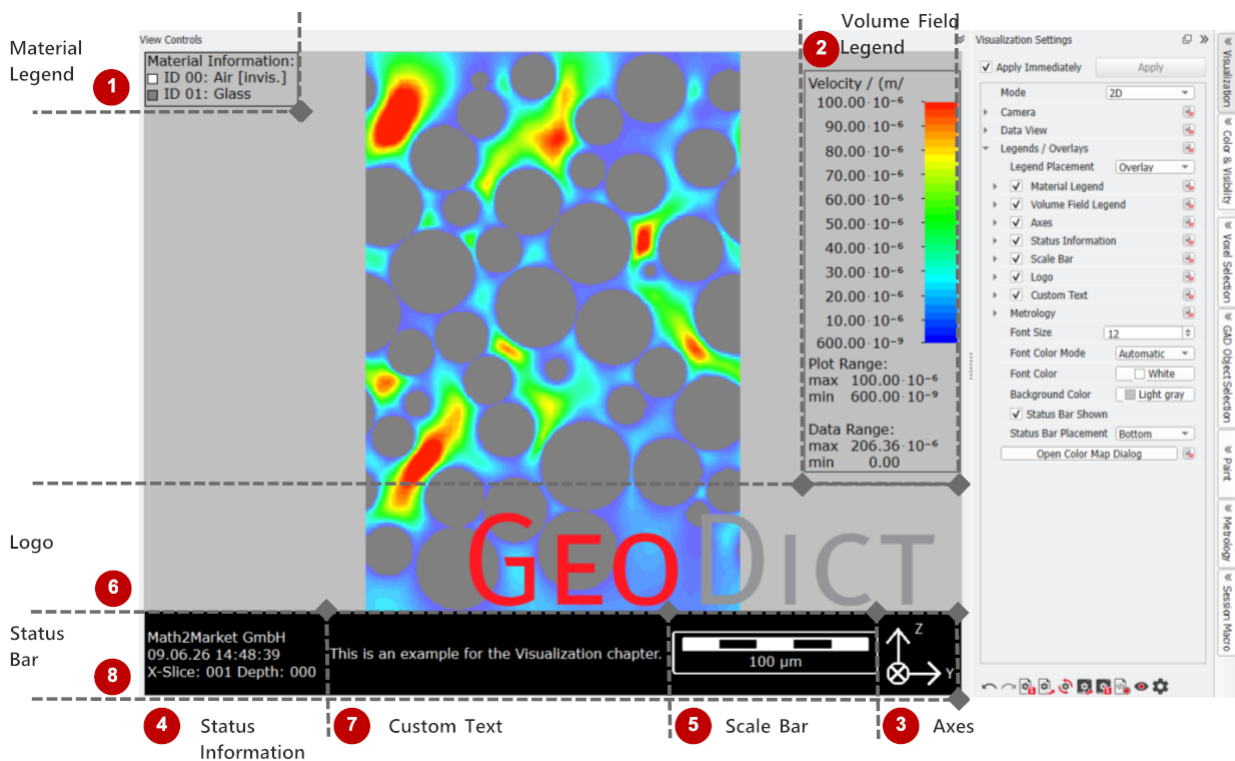
Different options are available for [2D Mode](#)^[109] and [3D Mode](#)^[117].

Tooltips guide you through the parameters of the **Legends / Overlays** section.



Legends 2D

Define the visualization of legends and overlays for the 2D view.



For all legends and overlays unfolding the corresponding sections gives access to more changeable settings.

✓ Legend Placement

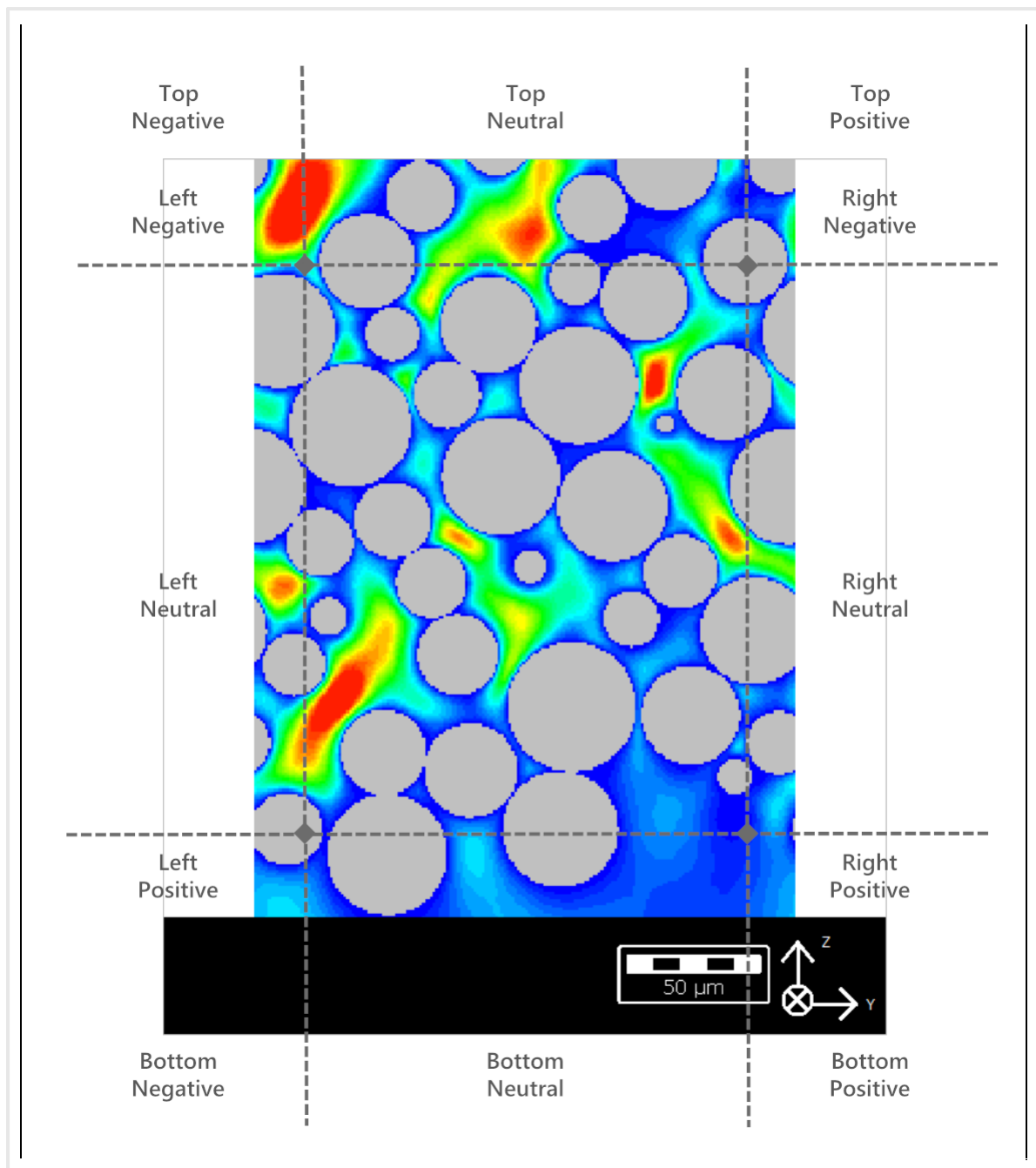
Decide, if the **Legend Placement** should be **Overlay** or **Attached**. This option is also available in the [Camera](#) ²⁹ tab of the **View Controls**.

✓ Material Legend

The **Material Legend** shows which materials are contained in the loaded structure.

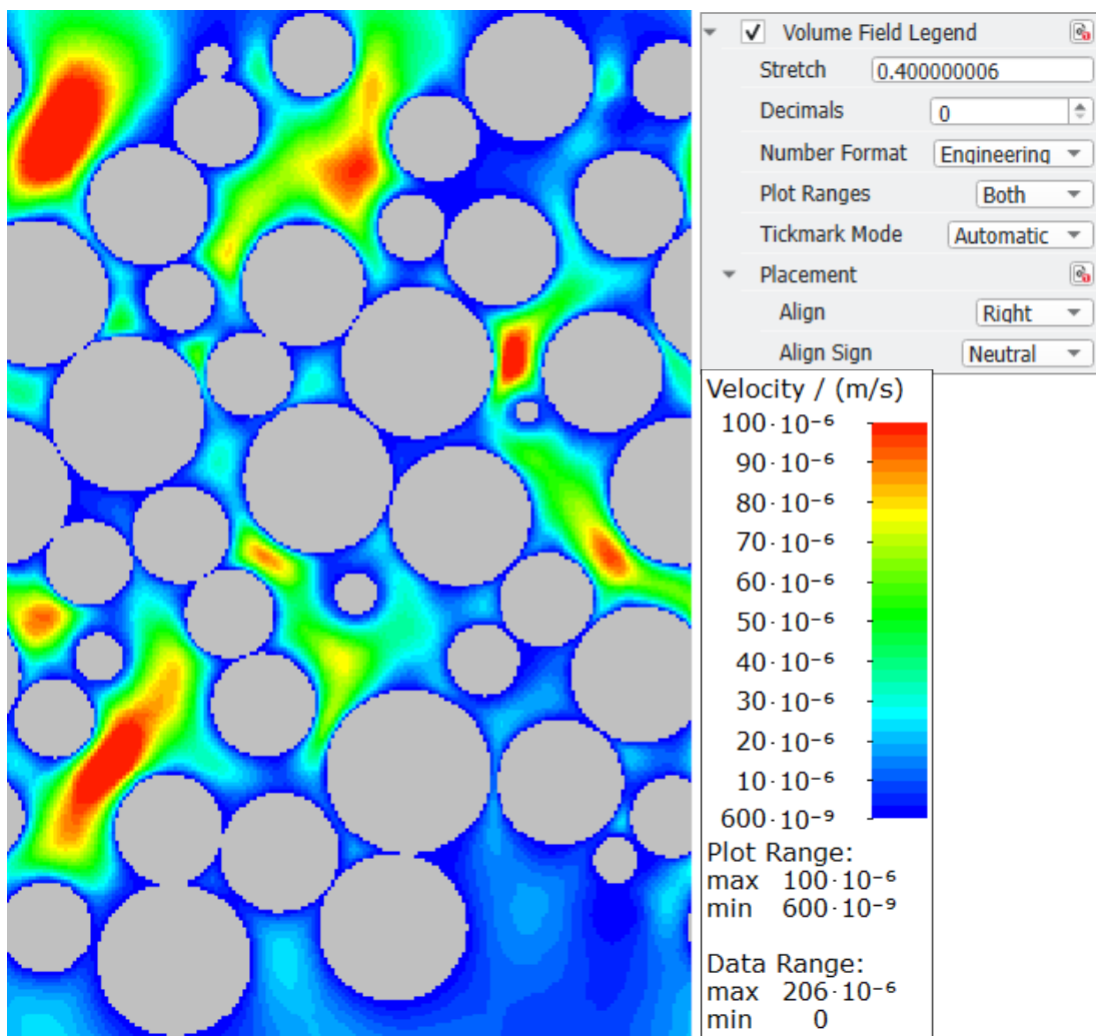


Control the legend's **Placement** with the parameters **Align** and **Align Sign**. **Align** controls, if the legend is placed at the **Left**, **Right**, **Top** or **Bottom** area of the visualization area and **Align Sign** controls where exactly in this area the legend is placed.



Volume Field Legend

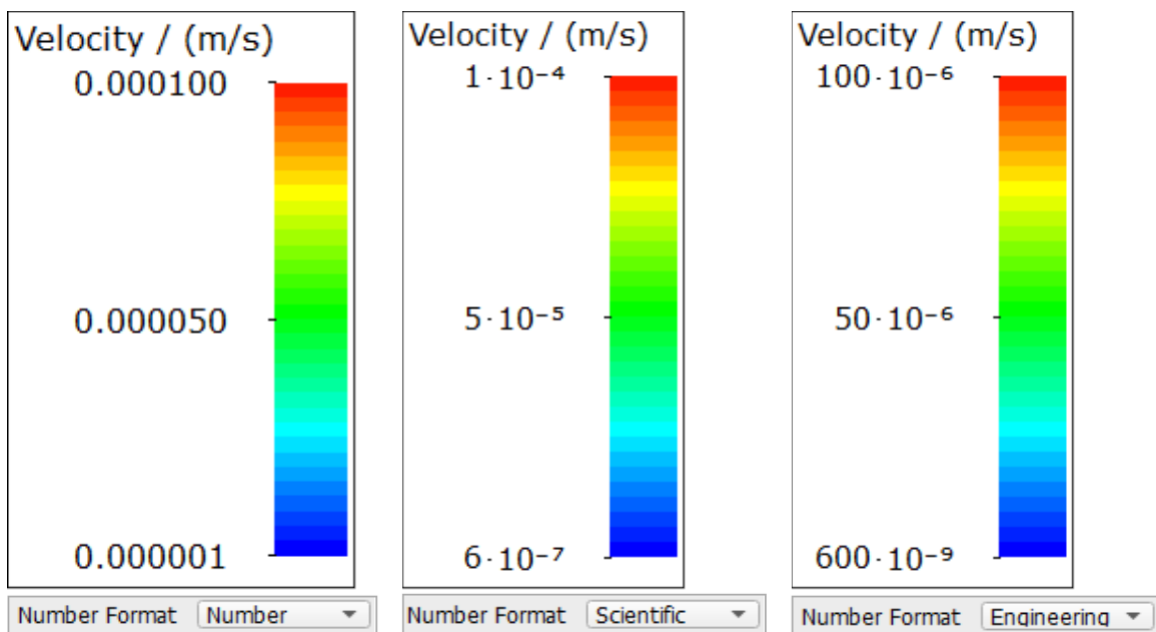
The following explains the functionality of the **Volume Field Legend** parameters.



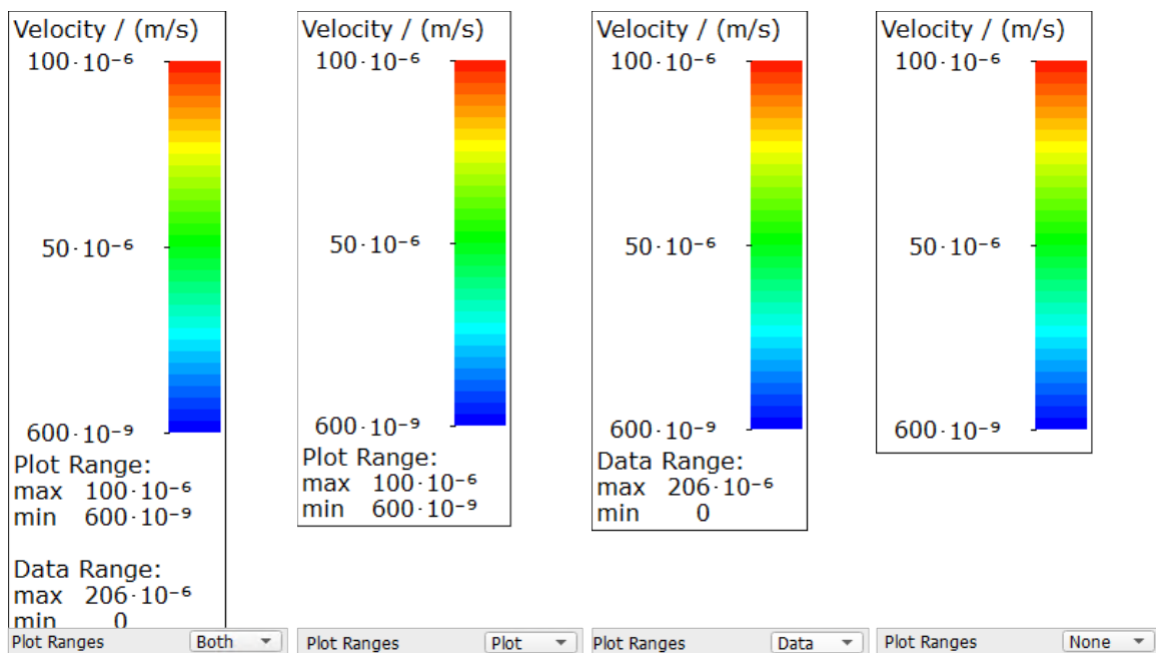
Control the size of the legend by changing the value for **Stretch**. Enter values between 0 and 1. This parameter depends on the total GeoDict GUI size. Thus, a **Stretch** of 1 would be the as wide or high (depending on the legend placement) as the GeoDict window. If the **Stretch** is chosen too high for the visualization area, the legend is not displayed anymore. Thus, it is recommended to choose values between 0.2 and 0.6.

The value in **Decimals** determines the number of digits shown after the decimal point.

Number Format controls the numbers' display in the legend. Choose between **Number**, **Scientific**, and **Engineering**. In the example for the choice of **Number** additionally the number of decimals was increased from 0 to 6, because otherwise all values would have been 0.



The choice of **Plot Ranges** controls how to display the spread of the data in the result legend. Decide to display **Both**, only the **Plot** or the **Data** range, **None** of the two ranges, or set it to **Auto** (Automatic).



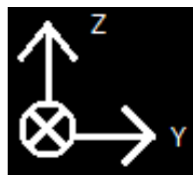
The **Tickmark Mode** controls whether the selection of the number of tick marks in the legend should be **Manual** or **Automatic**. If set to **Manual** enter a desired **Number of Tickmarks**. As the tickmark numbers must be “nice numbers”, the tickmark number then is set to a number near the entered value, still resulting in nice numbers.

The **Placement** works as described for the [Material Legend](#) ¹¹⁰.



Axes

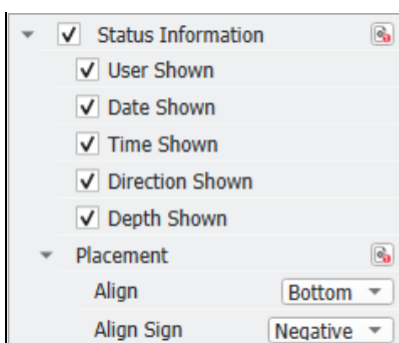
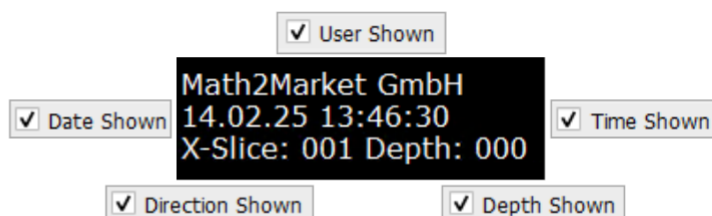
The **Placement** of the **Axes** works as described for the [Material Legend](#)¹¹⁰.

✓ Status Information

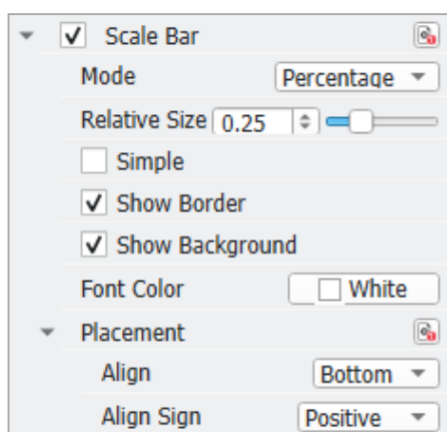
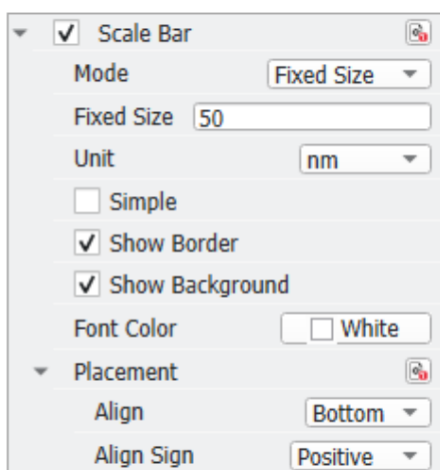
Decide which **Status Information** should be shown.

The **Placement** for this legend works as described for the [Material Legend](#)¹¹⁰.

✓ Scale Bar

Select the **Mode** to be **Percentage** or **Fixed Size**. If **Percentage** is selected, enter a **Relative Size** or change the value with the slider. This value then determines the size proportional to the length of the structure. If **Fixed Size** is selected, choose a **Unit** and enter a **Fixed Size**.

Enable **Simple** to display the scale bar with simple lines.

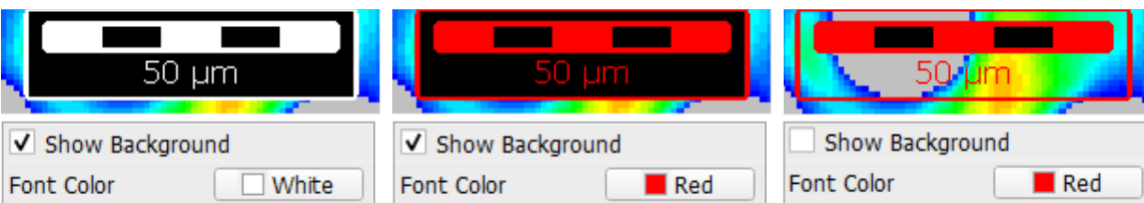


Select **Show Border** to display the rectangle around the scale bar.



Unchecking **Background** turns the black background of the scale bar invisible.

Also, the **Font Color** can be changed as desired.



The **Placement** for this legend works as described for the [Material Legend](#)¹¹⁰.

✓ Logo

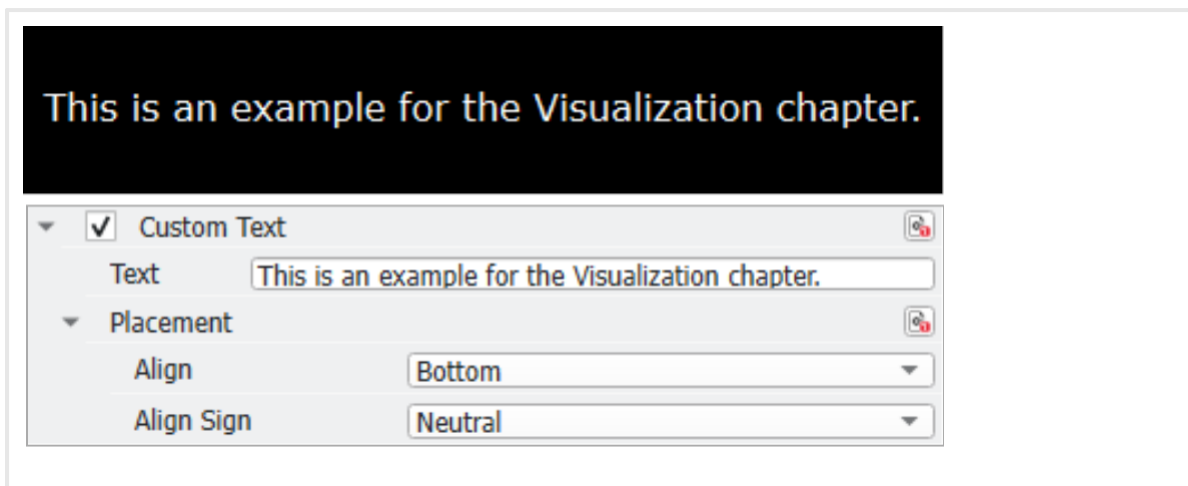
The Placement of the **Logo** works as described for the [Material Legend](#)¹¹⁰.



✓ Custom Text

Enter a custom **Text** to be shown in the visualization area.

The **Placement** for this overlay works as described for the [Material Legend](#)¹¹⁰.



✓ Metrology

For **Metrology**, select the **Line Thickness** and the **Line Color**.

The Metrology feature is described along with the other sidebar tabs.

✓ Font Size, Font Color, and Background Color

Change the **Font Size** for all legends by entering the desired value. Also, the **Font Color Mode** can be changed from the default **Automatic** to **Manual**. If set to **Manual**, a custom **Font Color** can be chosen. The automatic mode only affects the material legend and the volume field legend. The status bar contents stay white.

Change the **Background Color** of the **Visualization area**, if needed.

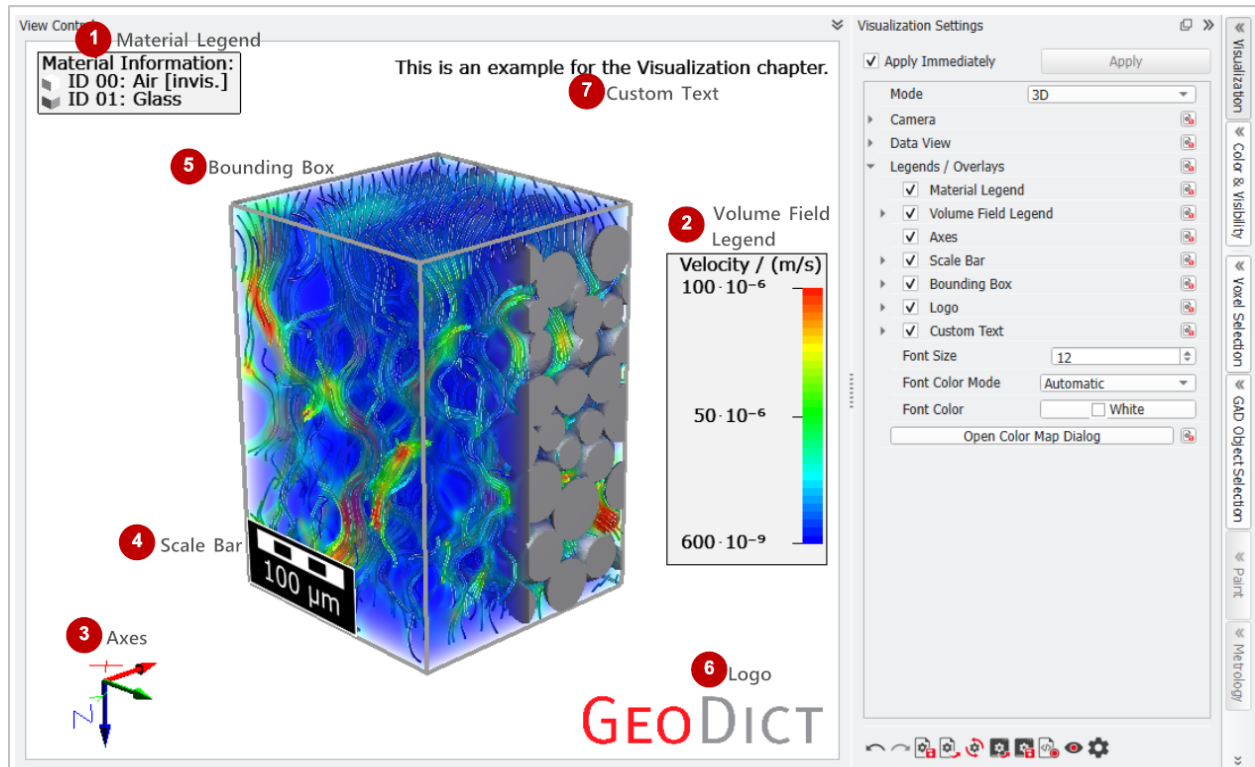
✓ Status Bar

Decide whether the **Status Bar** should be **Shown** and select the **Status Bar Placement** to be at the **Bottom** or **Top** of the visualization area.

Legends 3D

Define the visualization of legends and overlays for the 3D view. For **Material Legend** and **Axes** only select if they are shown or not and change the font size and font color as desired.

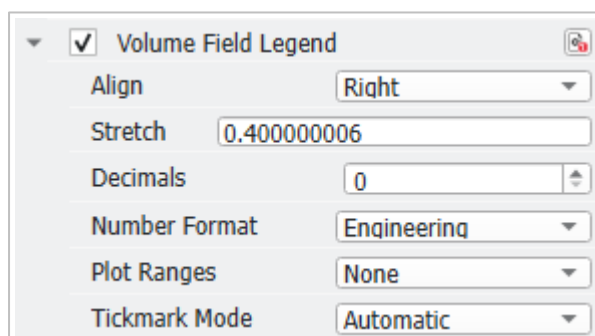
For all other legends and overlays, unfolding the corresponding sections gives access to more changeable settings.



✓ Volume Field Legend

For the **Volume Field Legend** options refer to the [2D options](#)¹¹¹.

Only the legend placement differs, as for the 3D mode, only the **Align** menu is available with the options **Left**, **Right**, **Top**, and **Bottom**. The volume field legend will always be placed in the center of the selected area.



While it is displayed vertically for **Left** and **Right**, it is displayed horizontally for **Top** and **Bottom**.

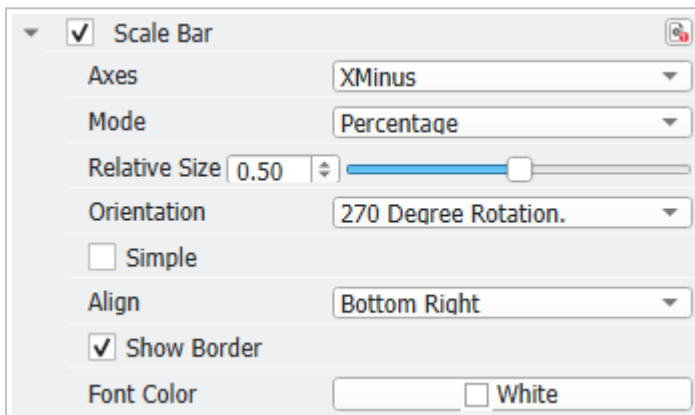
When selecting a position interfering with the **Material Legend**, the material legend is moved to another position automatically.

✓ Scale Bar

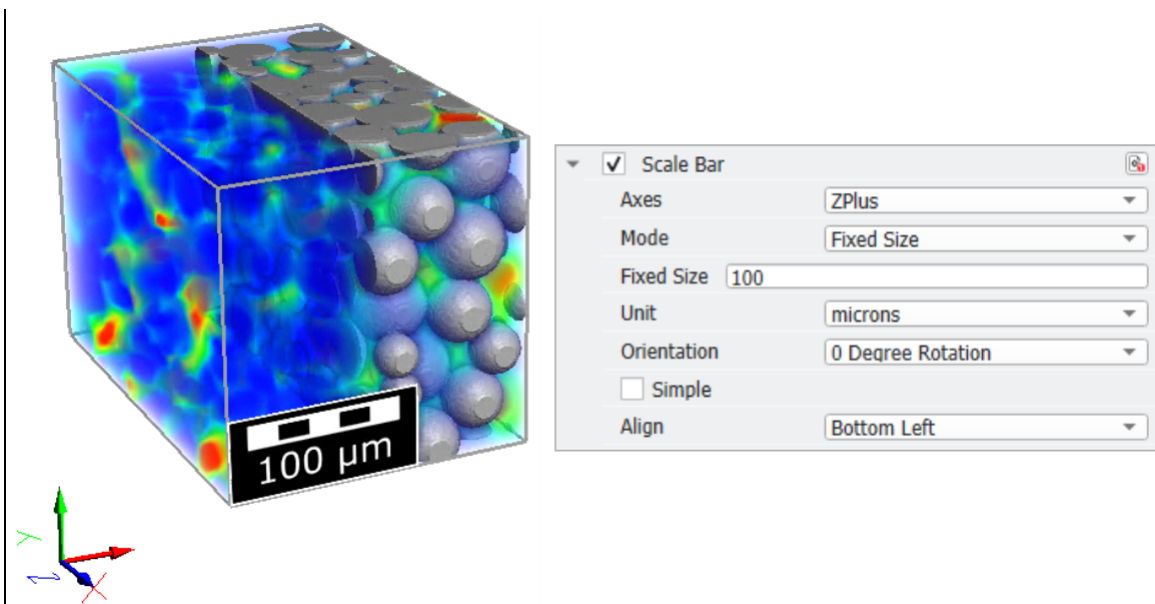
In 3D mode, the **Scale Bar** can be placed on the structure according to the three **Axes** (**X**, **Y** and **Z**). **Minus** then places the scale bar on the origin side of the specified axis and **Plus** on the opposite side.

Rotate the scale bar as desired, by changing the **Orientation**.

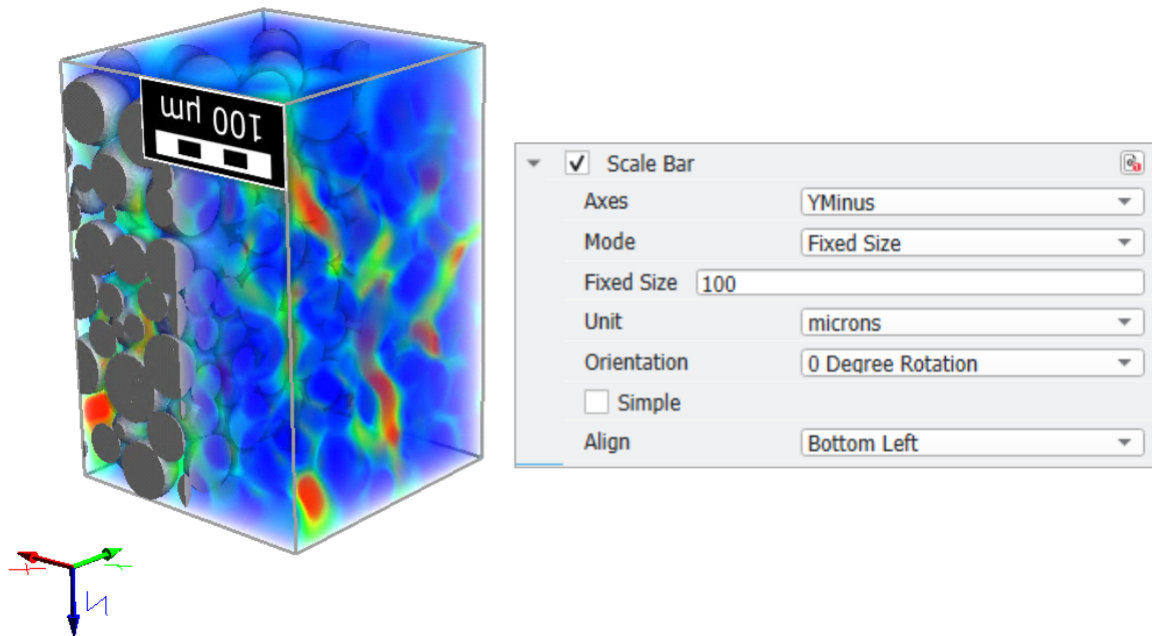
The scale bar can be **Aligned** in the **Bottom Left**, **Bottom Right**, **Top Left** or **Top Right** of the specified structure side.



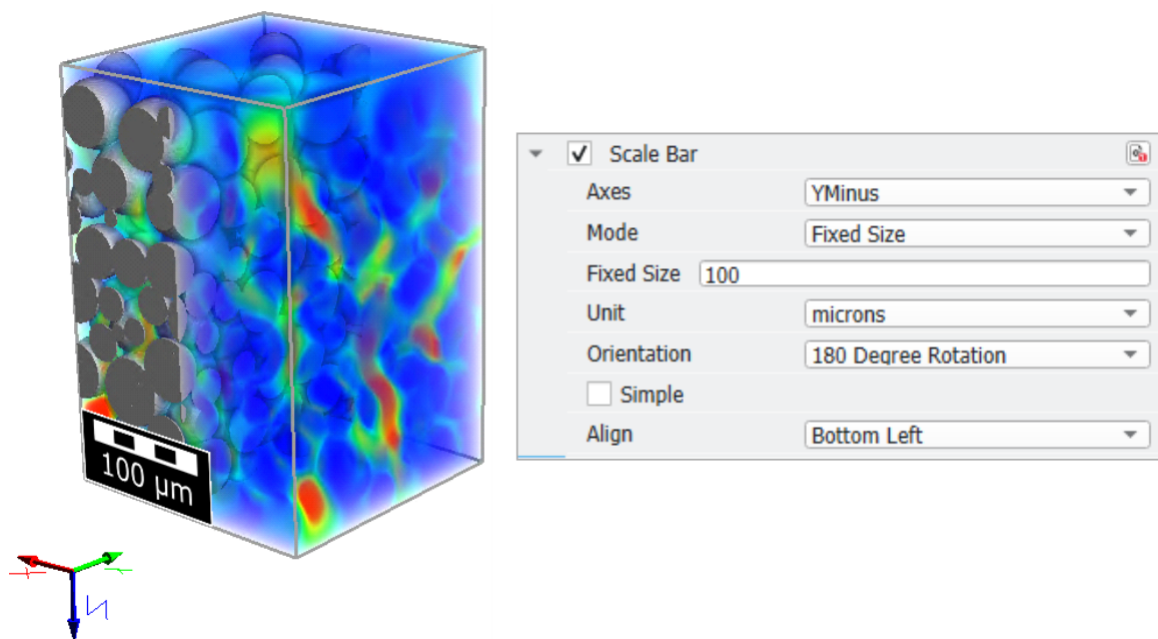
In the following example, the scale bar position is set to default, i.e., the **Plus** side of the **Z**-axis at the **Bottom Left**, with a **0 Degree Rotation**.



Other positions may be better if the structure is rotated differently. In the following figure, the **Minus** side of the **Y**-axis is turned to the front. Thus, selecting **YMinus** for **Axes** lets appear the scale bar in the front again. But without changing the **Orientation**, it is displayed upside down in the upper right corner of the YMinus side.



Changing the **Orientation** to **180 Degree Rotation** turns the scalebar about 180 degrees. Additionally, selecting **Top Right** moves the scale bar to the top right from the structures perspective, in this case resulting in a bottom left position from the viewer's perspective.

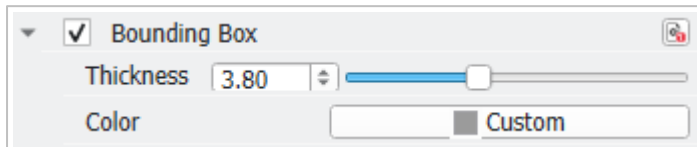


Setting the **Axes** to **Auto** always places the legend on the front of the structure in the correct orientation.

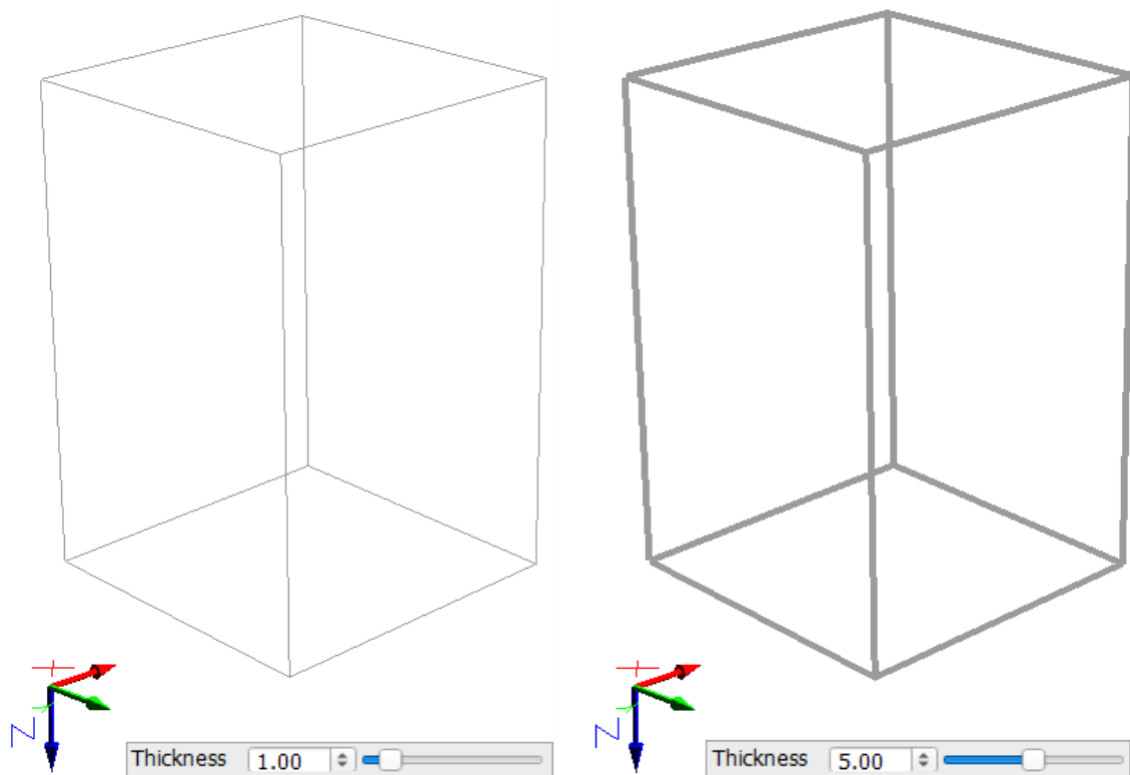
The other available options for **Scale Bar** are described [for the 2D mode](#)¹¹⁴.

✓ Bounding Box

If **Bounding Box** is enabled, the structure's domain border indicators are displayed.

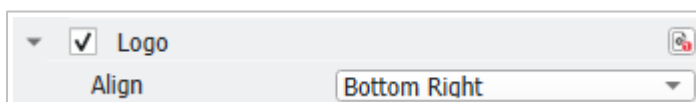


Change **Thickness** and **Color** as desired.



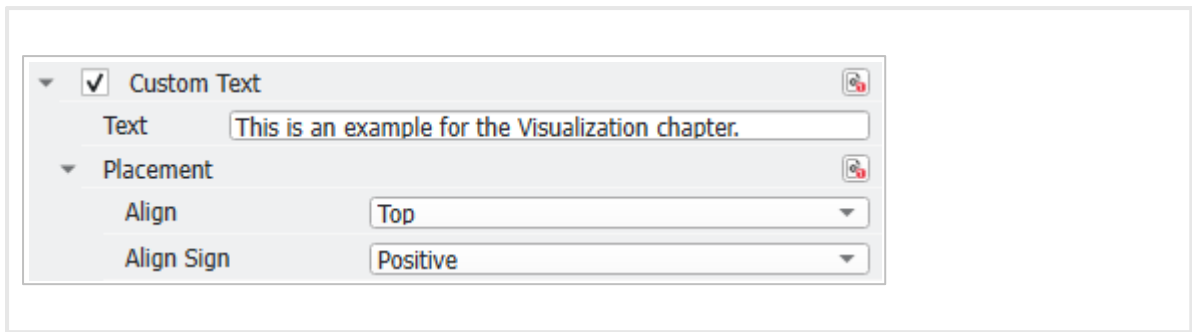
✓ Logo

The GeoDict **Logo** can be **Aligned** in the **Bottom Left**, **Bottom Right**, **Top Left** or **Top Right** corners of the visualization area.



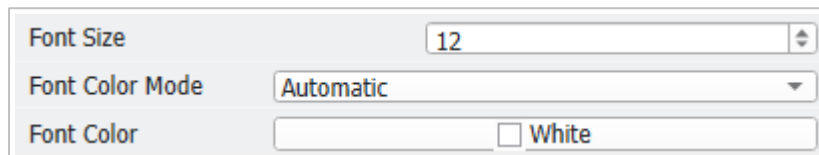
✓ Custom Text

The **Custom Text** parameters are the same as for the [2D mode](#)¹¹⁵.



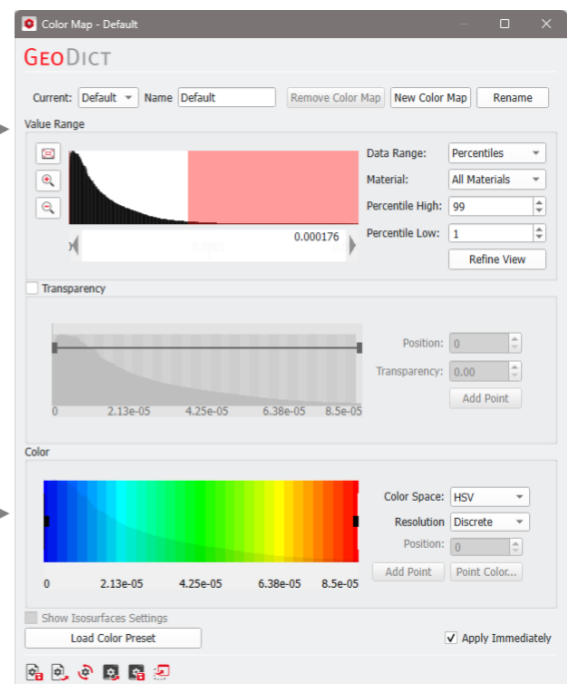
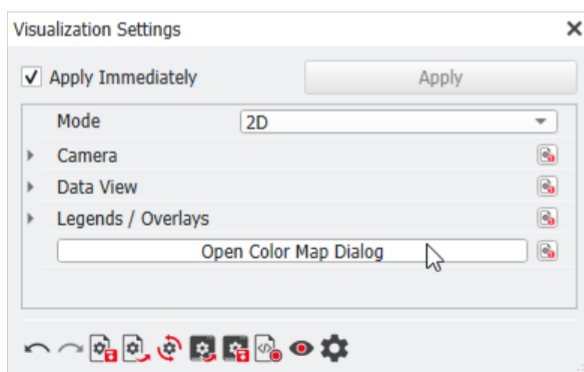
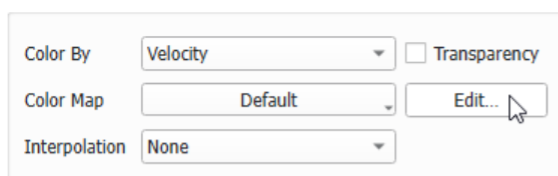
✓ Font Size and Font Color

Change the **Font Size** for all legends by entering a value. Also, the **Font Color Mode** can be changed from the default **Automatic** to **Manual**. If set to **Manual**, a custom **Font Color** can be chosen.



1.2.4 Color Map Dialog

Clicking the **Edit...** button for **Color Map...** in the **Volume Field** tab, or **Open Color Map Dialog** in the **Visualization Settings** dialog, opens the **Color Map** dialog to fine-tune the visualization of results. Different color maps can be chosen for volume field, arrows, schlieren, and streamlines.

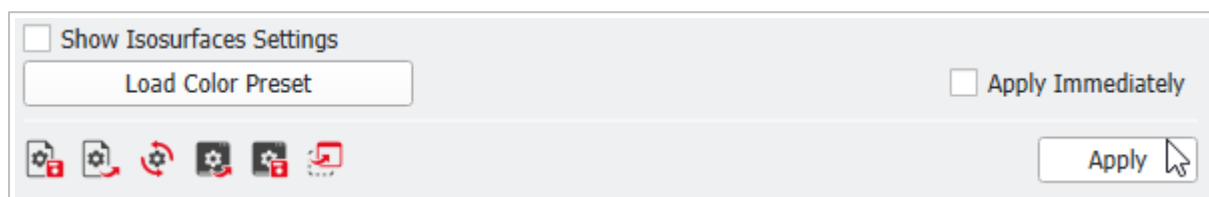




Know how! Save all color map parameters for later use in other GeoDict sessions, in a *.gps file or as start-up settings with the corresponding buttons in the bottom of the **Color Map** dialog. Resting the mouse pointer over an icon prompts a Tooltip showing the icon's function.



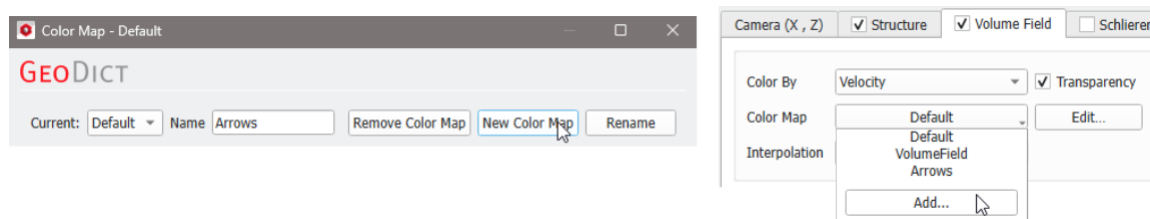
If **Apply Immediately** at the bottom right of the dialog is checked, these choices are instantly used on the display in the Visualization area, while keeping the **Color Map** dialog open to make other changes. Otherwise, click **Apply** to change the visualization settings.



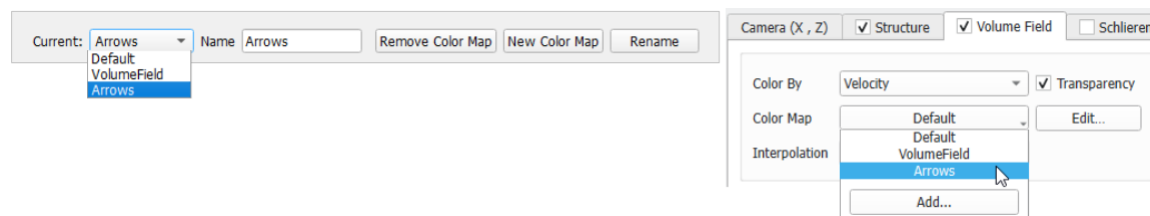
Close the dialog when the visualization fits your needs by clicking in the upper right corner.

Create Custom Color Maps

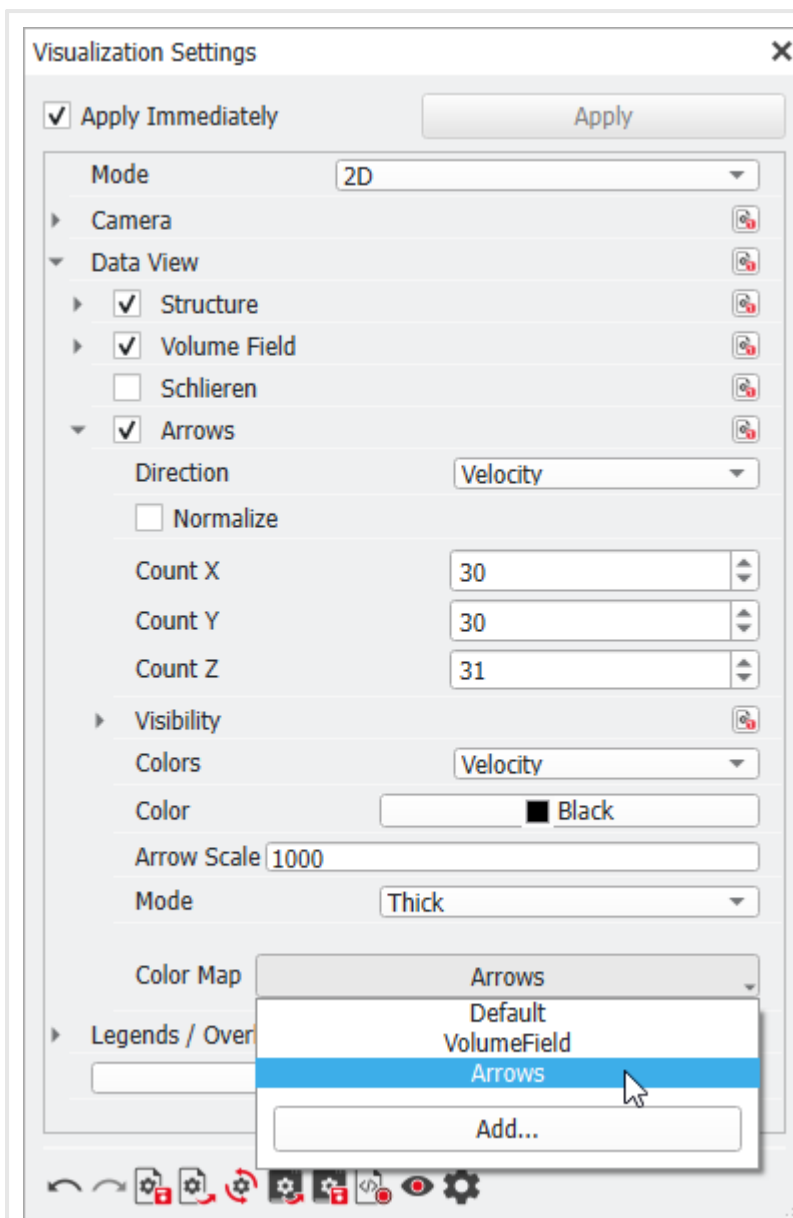
Create different color maps in the current session for the different result fields. To do so, in the **Color Map** dialog enter a **Name** and click **New Color Map** in the **Color Map** dialog or, in the **View Controls**, unfold the pull-down menu for **Color Map** and click **Add**.



Edit the settings as desired, and switch between the current color maps by selecting the needed color map from the pull-down menu in the **Color Map** dialog, in the **View Controls** or through the **Visualization Settings** dialog of the GUI **Sidebar**.



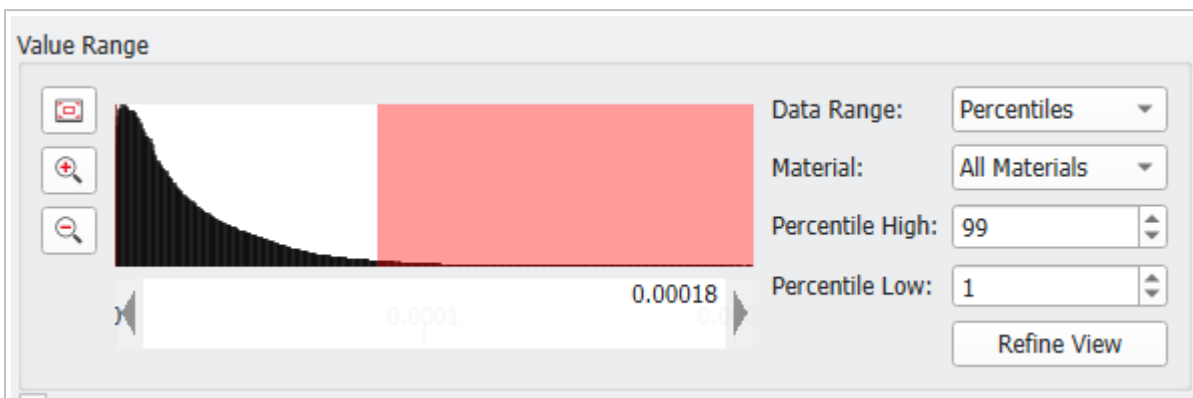
Having different color maps is very useful, if different result fields should be displayed in different color styles. In the **Visualization Settings** dialog, for each of the result fields under **Data View**, find the option to select a **Color Map** for the corresponding result field; here shown for the **Arrows**.



The **Color Map** dialog also offers the possibility to rename or remove color maps. Simply select the respective color map and click **Remove Color Map** or enter a new name and click **Rename**.

✓ Value Range

Set the value range for the volume field visualization. Define lower and upper value limits and how the data range should be specified.

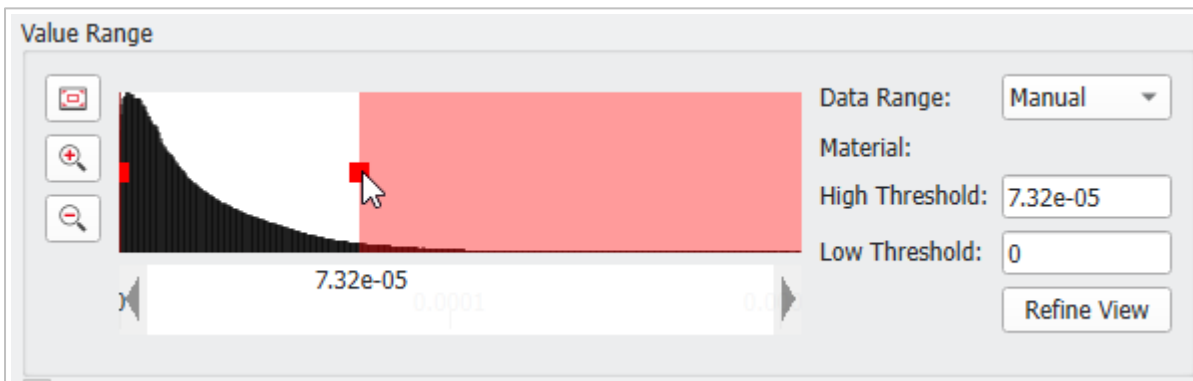


A color map is applied to the given **Data Range**. The data range can be selected in the following ways:

- **Manual** – allows to enter minimum and maximum values directly or move the square handles in the histogram to define upper and lower limits for the value range.
- **Min/Max** – the minimum and maximum values from the histogram are used.
- **Extended** – the minimum and maximum values are chosen to be an easy-to-read number rounded to the extended range.
- **Percentiles** - minimum and maximum values are computed from the field's histogram.

If **Percentiles**, **Min/Max** or **Extended** is selected, the **Material** can be selected. Then, only the values within the chosen material(s) are considered for the histogram.

If **Manual** is selected for **Data Range**, the thresholds can also be set with the red squares in the histogram.



The histogram is displayed on the left and can be zoomed in and out with the **Zoom** icons.

The view can be reset again with the corresponding icon . When zooming in deeply into the histogram and then moving left or right, the resolution is not refined automatically, as it can take some time. Therefore, then click **Refine View**.

 **Transparency**

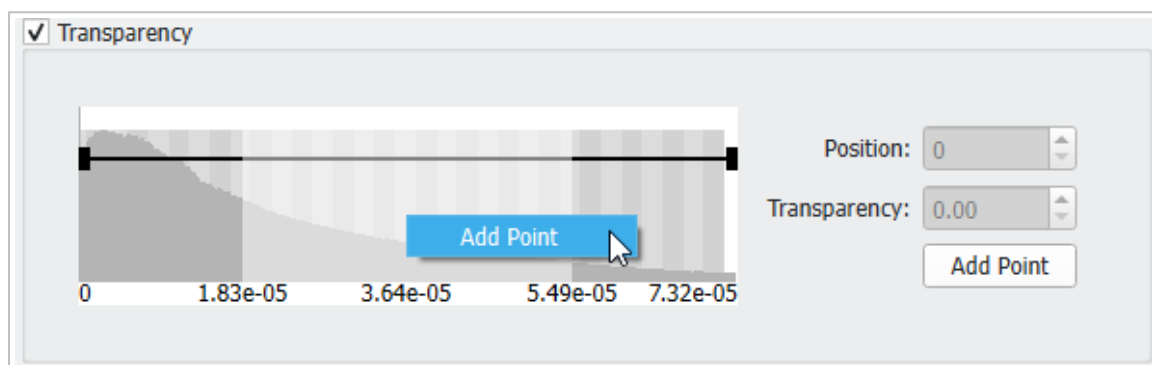
For many simulation results, it is interesting to “see-through” the volume field and to emphasize certain areas. The transparency option only has an effect on volume fields. Arrows, streamlines and schlieren cannot be visualized transparent.

For example, consider the result of a FlowDict simulation where we want to emphasize areas of fast flow.

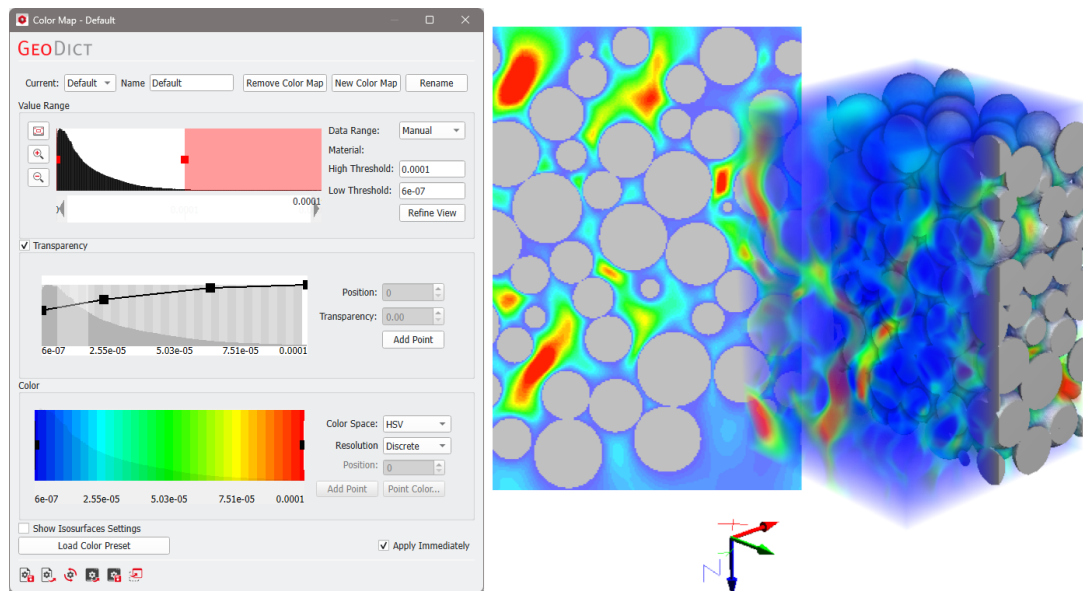
Click **Transparency** in the **Color Map** dialog, and change the shape of the transparency curve by adding and moving new anchor points.

To add new points either left-click in the plot and click **Add Point** on the right or right-click in the plot and click **Add Point** there.

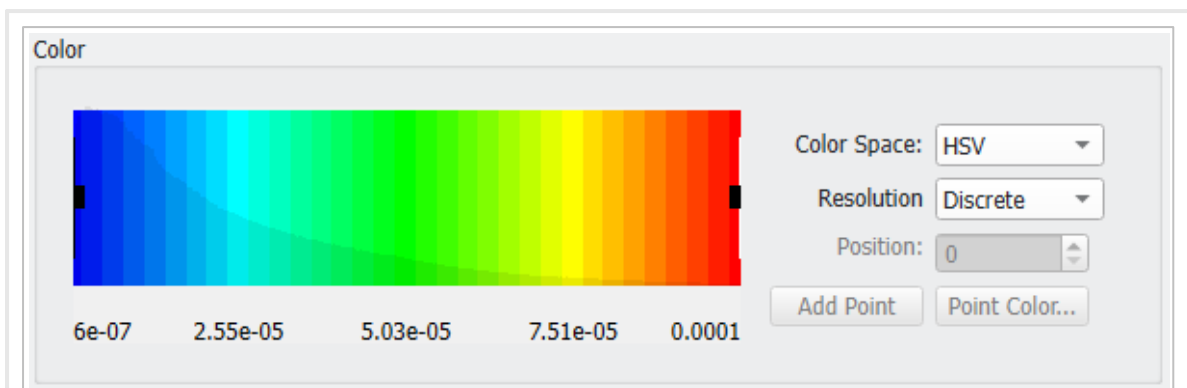
Move the points either by drag and drop or by entering the desired values for **Position** and **Opacity** for the selected point.



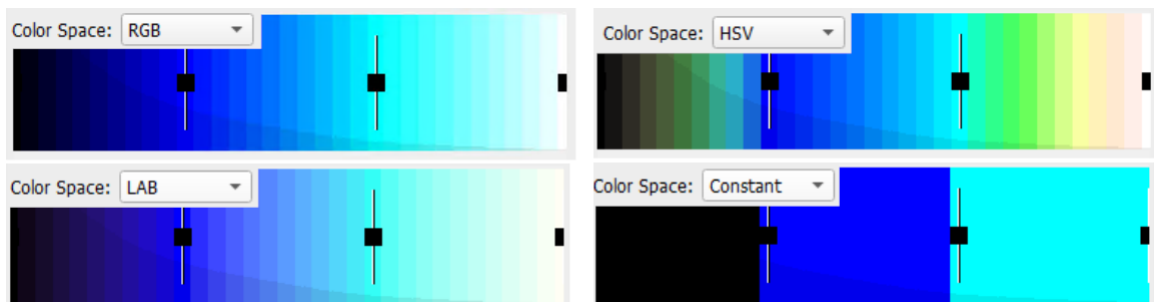
Observe the transparency gradient in the flow field with the following settings:



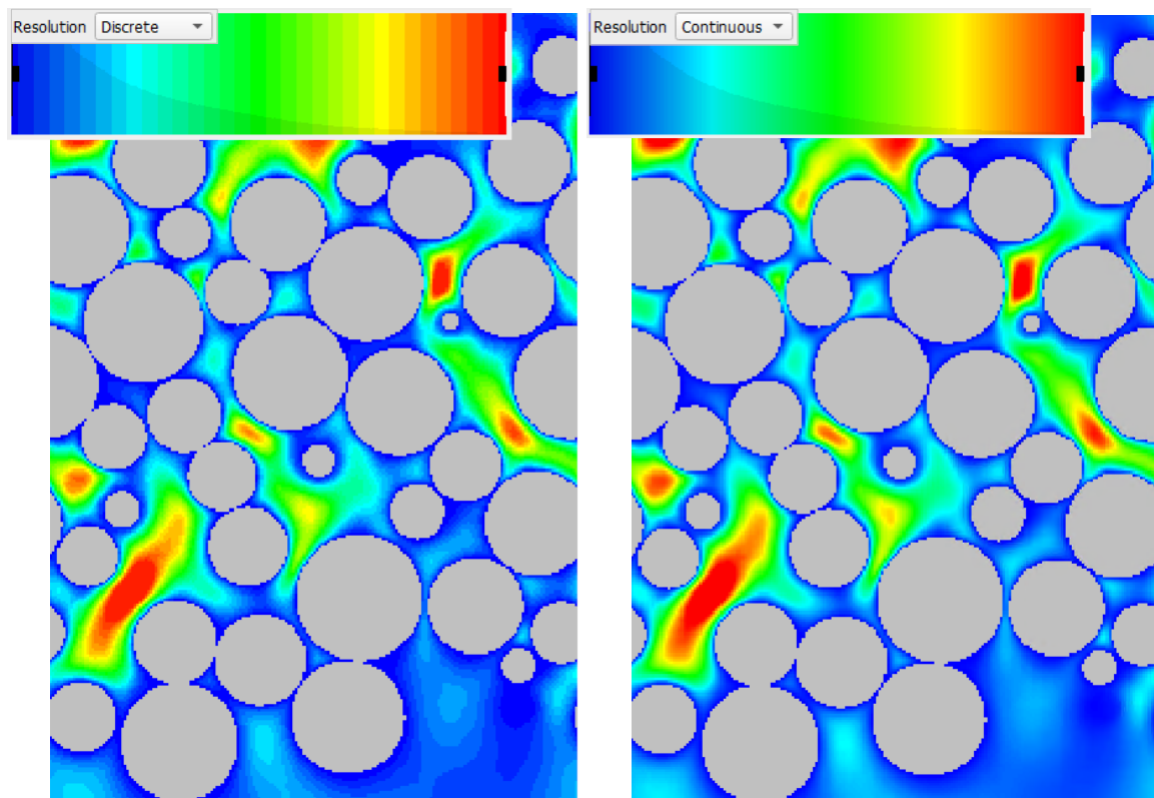
In the **Color** panel, control the color gradient.



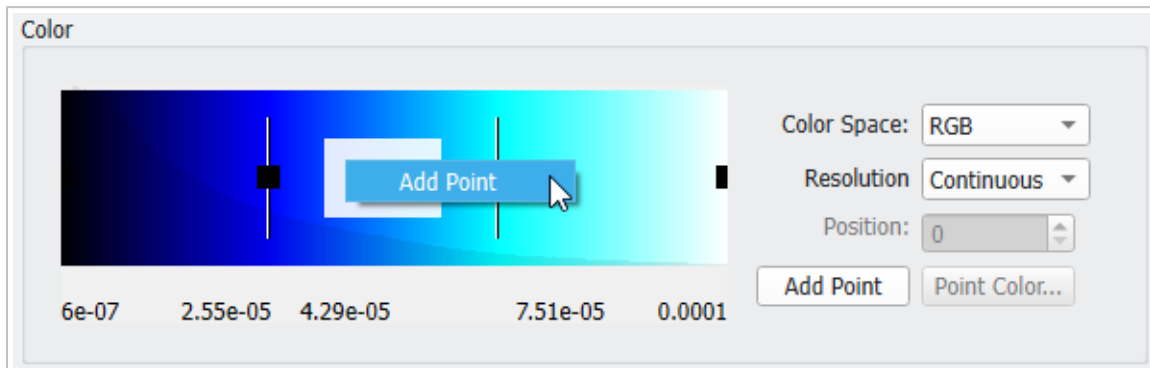
Change the complete **Color Space**, by selecting a different option from the pull-down menu. The color space defines if and how the colors should be interpolated between two defined points, here the four options **RGB**, **HSV**, **LAB** and **Constant** are shown for the color preset **Cold**.



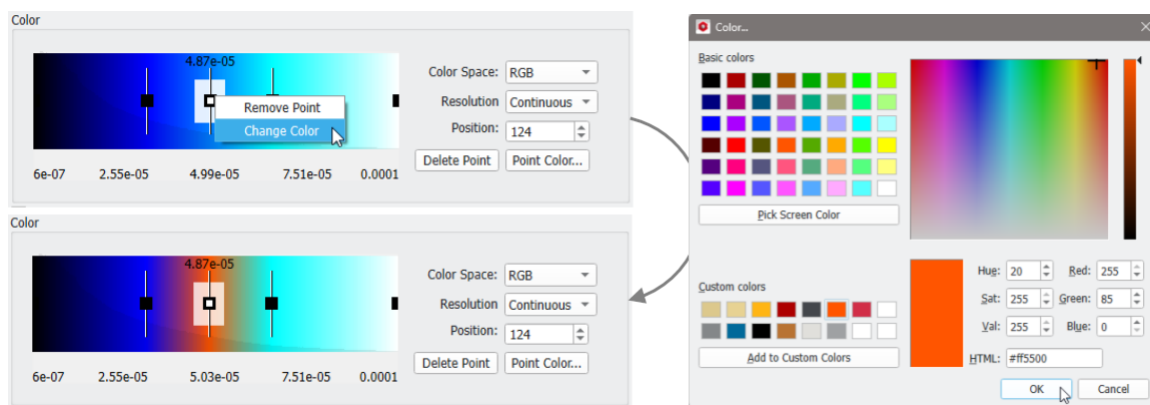
The **Resolution** can be set to **Discrete** or **Continuous**.



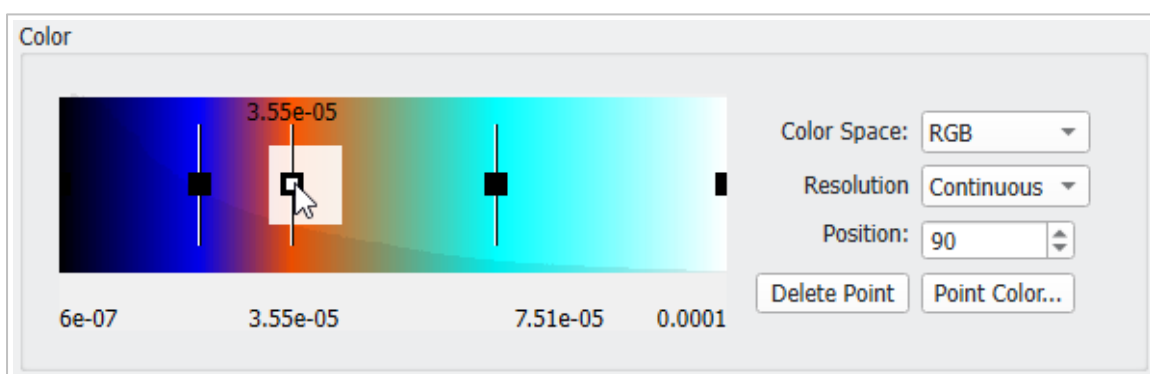
To create your own volume field colors, in the graph points can be added by left- or right-click and selecting **Add Point**.



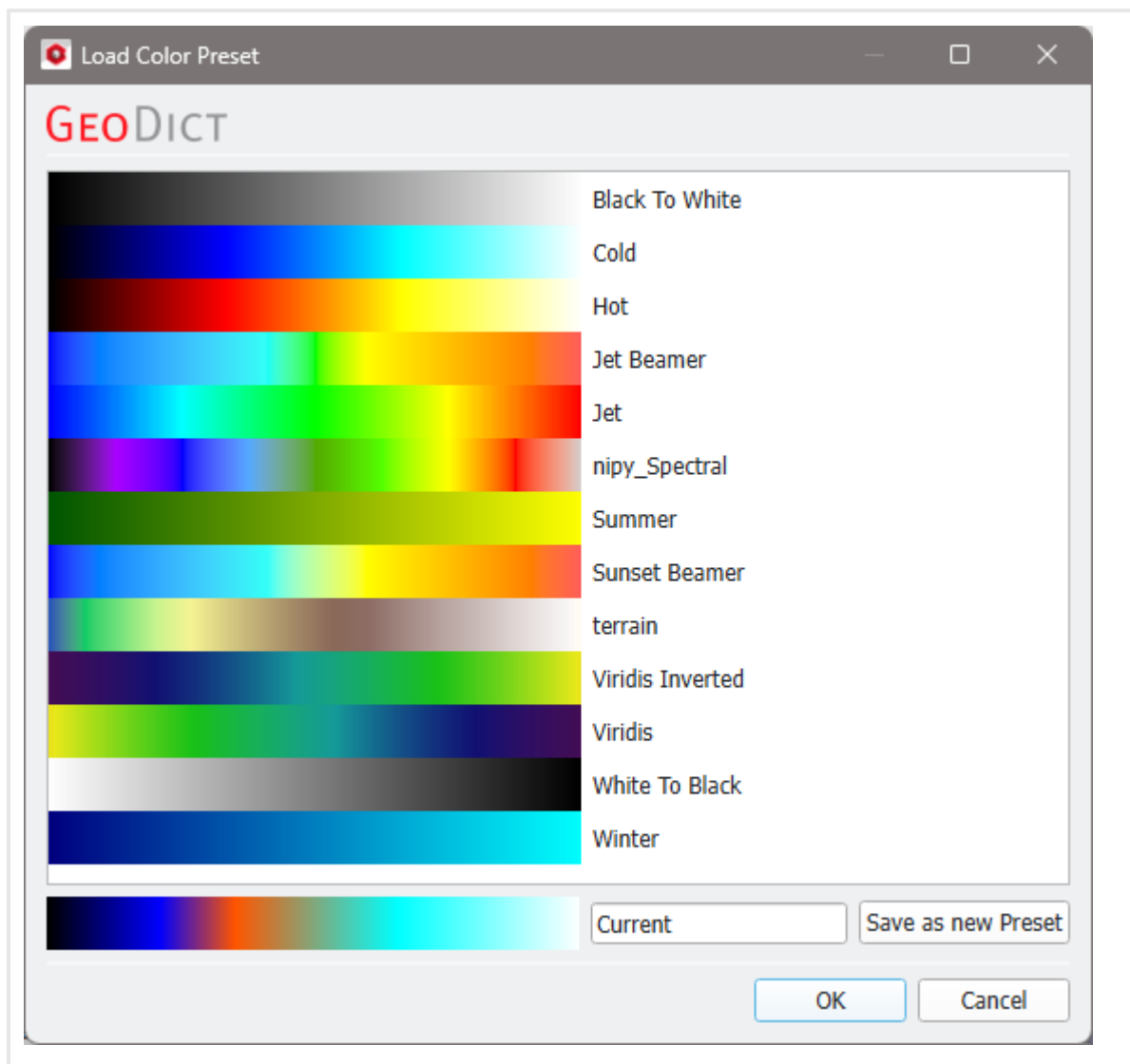
The **Point Color** of any point can be changed by selecting the respective point and either right-click and select **Change Color** or on the right click **Point Color...**



The **Position** can be changed either by dragging the points left and right or by entering a position on the right.



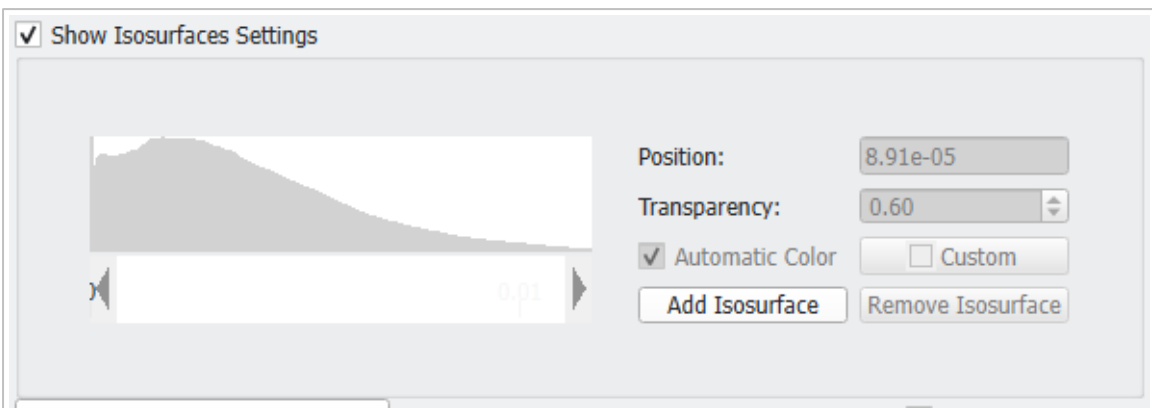
Select between many presets by clicking **Load Color Preset** at the bottom of the Color Map dialog or save your own presets similar as shown for the [Color & Visibility](#) options.



✓ Show Isosurfaces Settings

In 3D rendering also the **Isosurfaces** in the **Color Map** dialog can be displayed, if **Transparency** is checked. This setting has only an effect on volume fields.

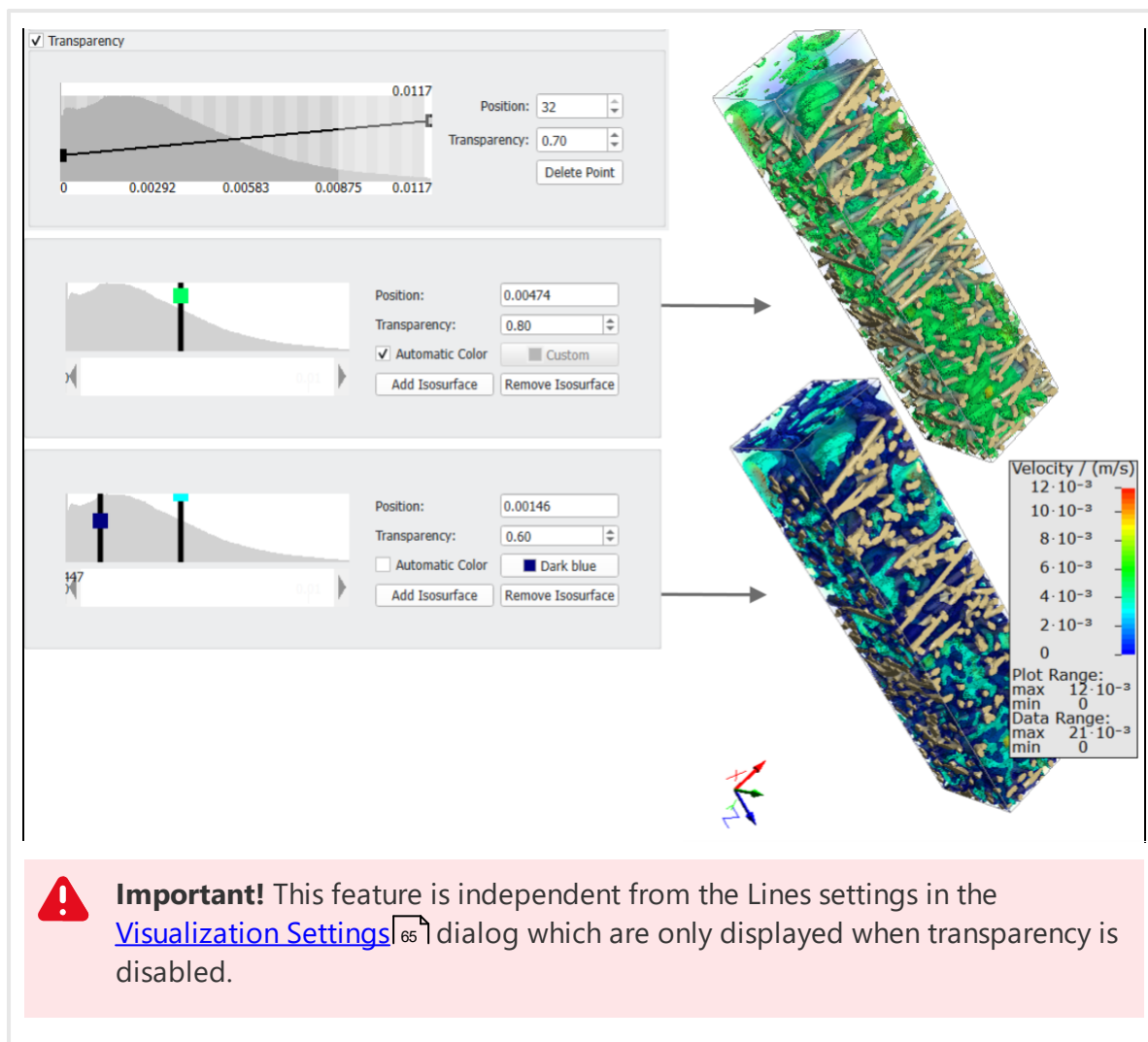
Enable **Show Isosurfaces Settings** and emphasize selected values in the volume field, by changing their transparency.



Important! Note, that disabling the checkbox does only hide the isosurfaces settings in the Color Map dialog and NOT the isosurfaces in the visualization area.

Click **Add Isosurface** to add a point in the graph. Move the point as desired or enter a **Position**. Set the **Transparency** and the color for the selected isosurface.

In the example, the **Structure** tab in the **View Controls** is disabled and the transparency settings are as shown above. An isosurface was added for a velocity of 0.0051 m/s and the transparency was set to 0.8. As **Automatic Color** is checked, the color of the value is not changed, but due to the higher transparency value, the green isosurface is emphasized.



The screenshot displays the 'Transparency' settings panel in a software interface. The panel is titled 'Transparency' and contains three sections for configuring different visualization elements. Each section includes a graph showing a profile, a 'Position' input field, a 'Transparency' input field, and buttons for 'Add Isosurface' and 'Remove Isosurface'. The first section has a 'Delete Point' button. The second section has an 'Automatic Color' checkbox and a 'Custom' button. The third section has an 'Automatic Color' checkbox and a 'Dark blue' color selection button. To the right of the settings panel, two 3D models are shown. The top model is a transparent rectangular block with a green and brown internal structure. The bottom model is a transparent rectangular block with a blue and brown internal structure. A color scale legend for 'Velocity / (m/s)' is located to the right of the 3D models, ranging from 0 to $12 \cdot 10^{-3}$. The legend also includes 'Plot Range' and 'Data Range' information.

Transparency Settings:

- Section 1:** Position: 32, Transparency: 0.70, Delete Point button.
- Section 2:** Position: 0.00474, Transparency: 0.80, Automatic Color checked, Custom button, Add Isosurface, Remove Isosurface buttons.
- Section 3:** Position: 0.00146, Transparency: 0.60, Automatic Color unchecked, Dark blue button, Add Isosurface, Remove Isosurface buttons.

Velocity / (m/s) Legend:

Velocity / (m/s)
$12 \cdot 10^{-3}$
$10 \cdot 10^{-3}$
$8 \cdot 10^{-3}$
$6 \cdot 10^{-3}$
$4 \cdot 10^{-3}$
$2 \cdot 10^{-3}$
0

Plot Range:

Plot Range
max $12 \cdot 10^{-3}$
min 0

Data Range:

Data Range
max $21 \cdot 10^{-3}$
min 0

Important! This feature is independent from the Lines settings in the [Visualization Settings](#) dialog which are only displayed when transparency is disabled.

1.3 References

G. Kindlmann, *Superquadric Tensor Glyphs*, Eurographics / IEEE VGTC Symposium on Visualization (2004), <http://dx.doi.org/10.2312/VisSym/VisSym04/147-154>

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



Math2Market GmbH

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