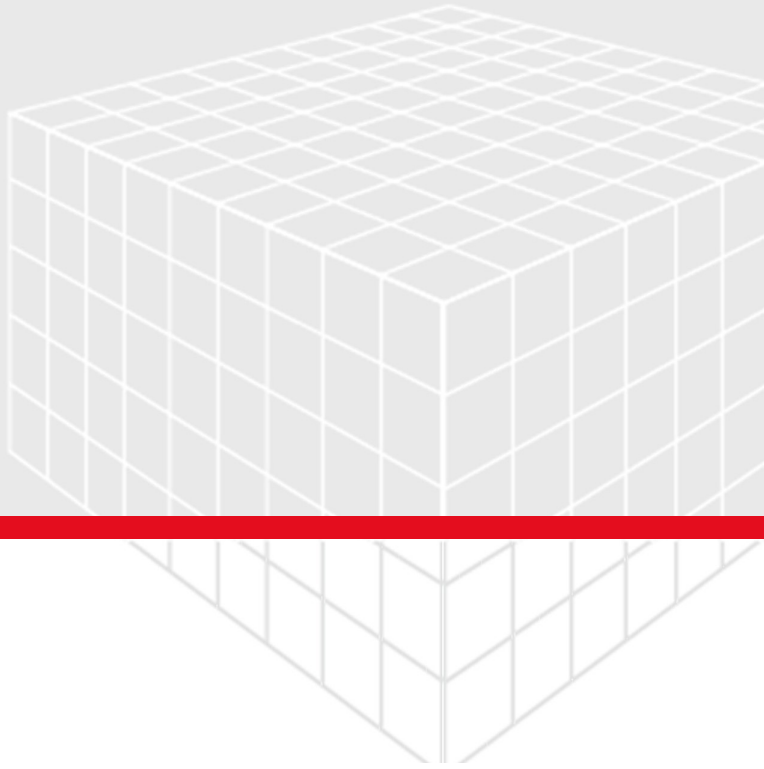
ExportGeo-Fluent

User Guide



GEODICT Release 2026

Published: June 2026

GEODICT

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

ExportGeo-Fluent

1 ExportGeo-Fluent

With ExportGeo-Fluent, it is possible to export structures from GeoDict and to run fluid flow and heat computations with Fluent on them. The voxel structure in GeoDict is exported to a mesh file (*.msh or *.neu), the solver parameters for the Fluent simulation are recorded to a journal file (*.jou). Those two files together contain all necessary information for the simulation in Fluent. The .jou files are saved as ASCII files so that, if changes are required, they can be easily opened and edited with a text editor.

To access ExportGeo-Fluent, select **Export** → **ExportGeo-Fluent** in the menu bar.

Learn to use ExportGeo-Fluent

- Export Flow Simulation (Voxel) 5
  With Export Flow Simulation (Voxel), the current voxel structure is exported to Fluent and the fluid flow simulation parameters are recorded as *.jou file.
- Run Flow Simulation (Voxel) 14
  With Run Flow Simulation (Voxel), the Fluent fluid flow simulation is directly started after the export.
- Export Heat Simulation (Voxel) 17
  With Export Heat Simulation (Voxel), the current voxel structure is exported to Fluent and the heat simulation parameters are recorded as *.jou file.
- Run Heat Simulation (Voxel) 21
  With Run Heat Simulation (Voxel), the Fluent heat simulation is directly started after the export.

Note! To execute the *.jou file in Fluent, the 3D double-precision solver (fluent 3ddp) must be selected. When exporting under Windows and running simulations under Linux, it might be necessary to convert the ***.jou** and ***.msh** files to the Unix format with the dos2unix command.

Important! The same results can be achieved with the FlowDict and ConductoDict modules directly in GeoDict.

In general, this is the recommended way if FlowDict or ConductoDict is in your GeoDict license, since the GeoDict solvers are optimized for voxel structures and therefore faster and more memory efficient for this use case.

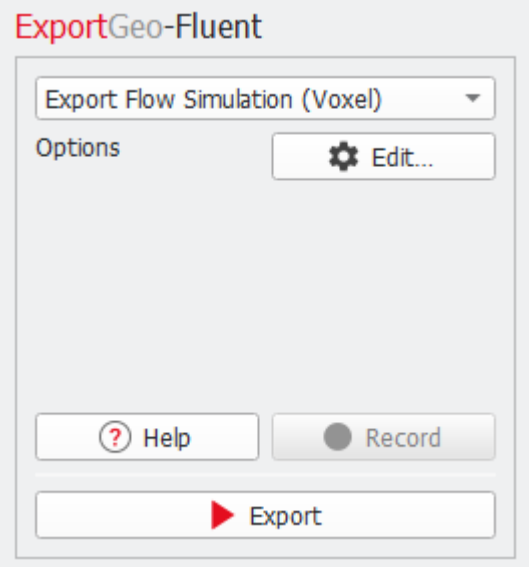
Join the conversation

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 Export Flow Simulation (Voxel)

To generate a journal file and mesh for a flow computation with **Fluent**, select **Export Flow Simulation (Voxel)** from the pull-down menu in the ExportGeo-**Fluent** section.



Important! The same results can be achieved with the FlowDict module directly in GeoDict.

In general, this is the recommended way if FlowDict is in your GeoDict license, since the GeoDict solvers are optimized for voxel structures and therefore faster and more memory efficient for this use case.

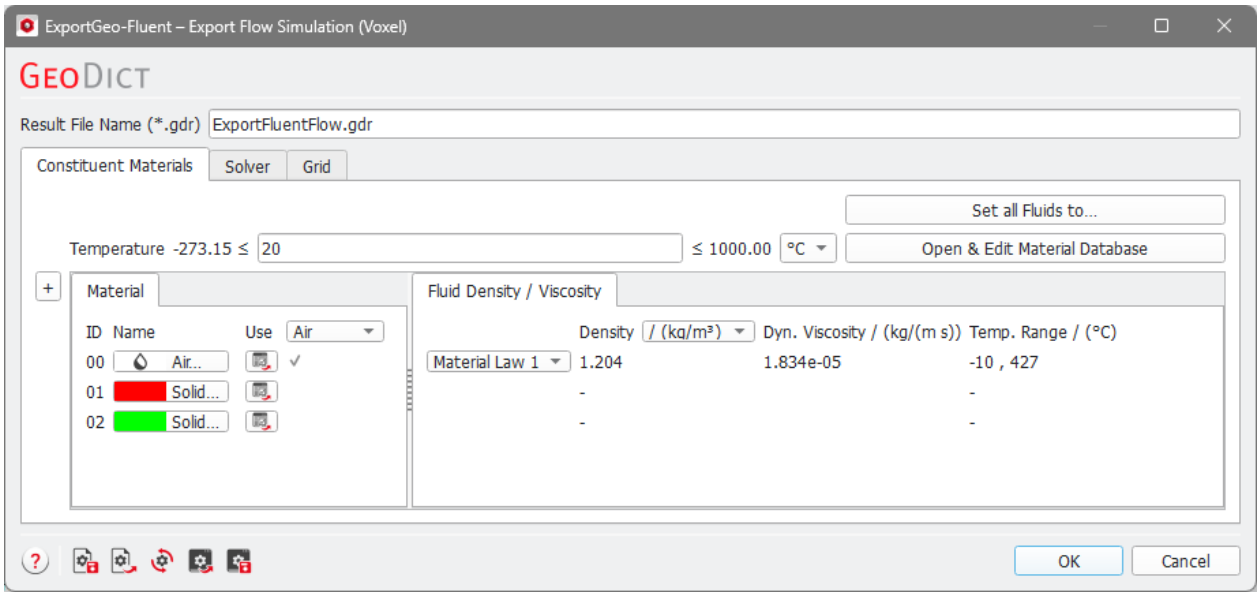
Click the **Options**' **Edit...** button to define the settings for the export to Fluent. After entering all parameters, click **Export** to export the simulation settings for Fluent.

In the **ExportGeo-Fluent - Export Flow Simulation** dialog, choose a **Result File Name (*.gdr)** according to the current project. The result file contains all information about the current export. The corresponding result folder contains the mesh and journal files for Fluent and the chosen structure saved as GDT file.

The options for ExportGeo-Fluent are organized under the tabs **Constituent Materials**, **Solver**, and **Grid**.

Constituent Materials

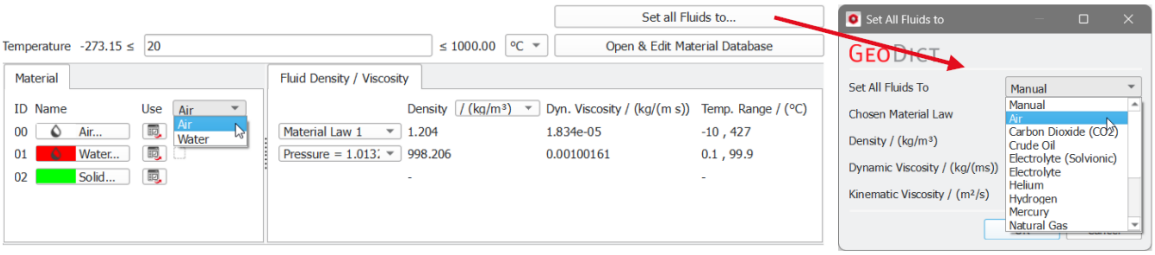
The fluid parameters of all constituent materials in the structure are defined in the corresponding fields under the **Constituent Materials** tab.



The structure already contains a fluid

If one fluid is present in the structure, this fluid is chosen automatically. If the structure contains more than one fluid, the fluid used for the flow simulation must be selected through the **Use** pull-down menu. Then, the other fluids in the structure are treated as solid in the simulation.

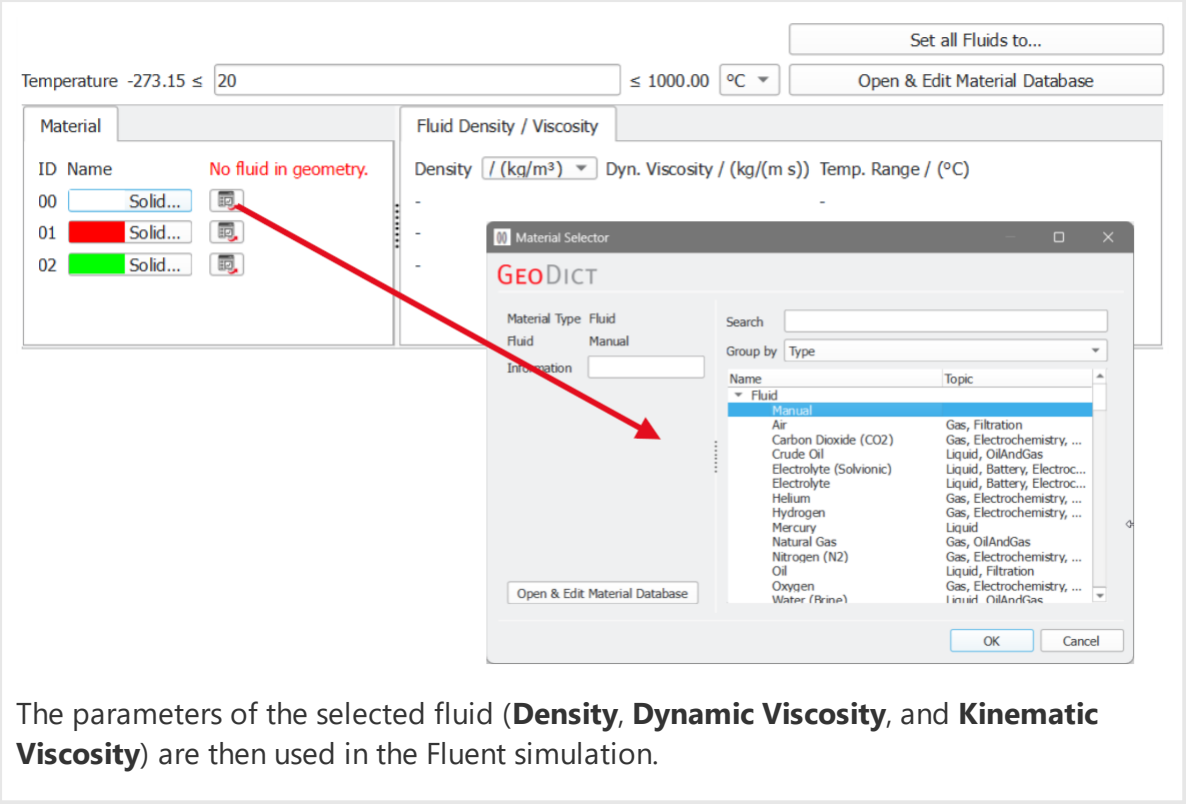
Alternatively, all fluids in the structure can be set to one specific material with **Set all Fluids to...**.



The parameters of the selected fluid (**Density**, **Dynamic Viscosity**, and **Kinematic Viscosity**) are then used in the Fluent simulation.

The structure does not contain a fluid

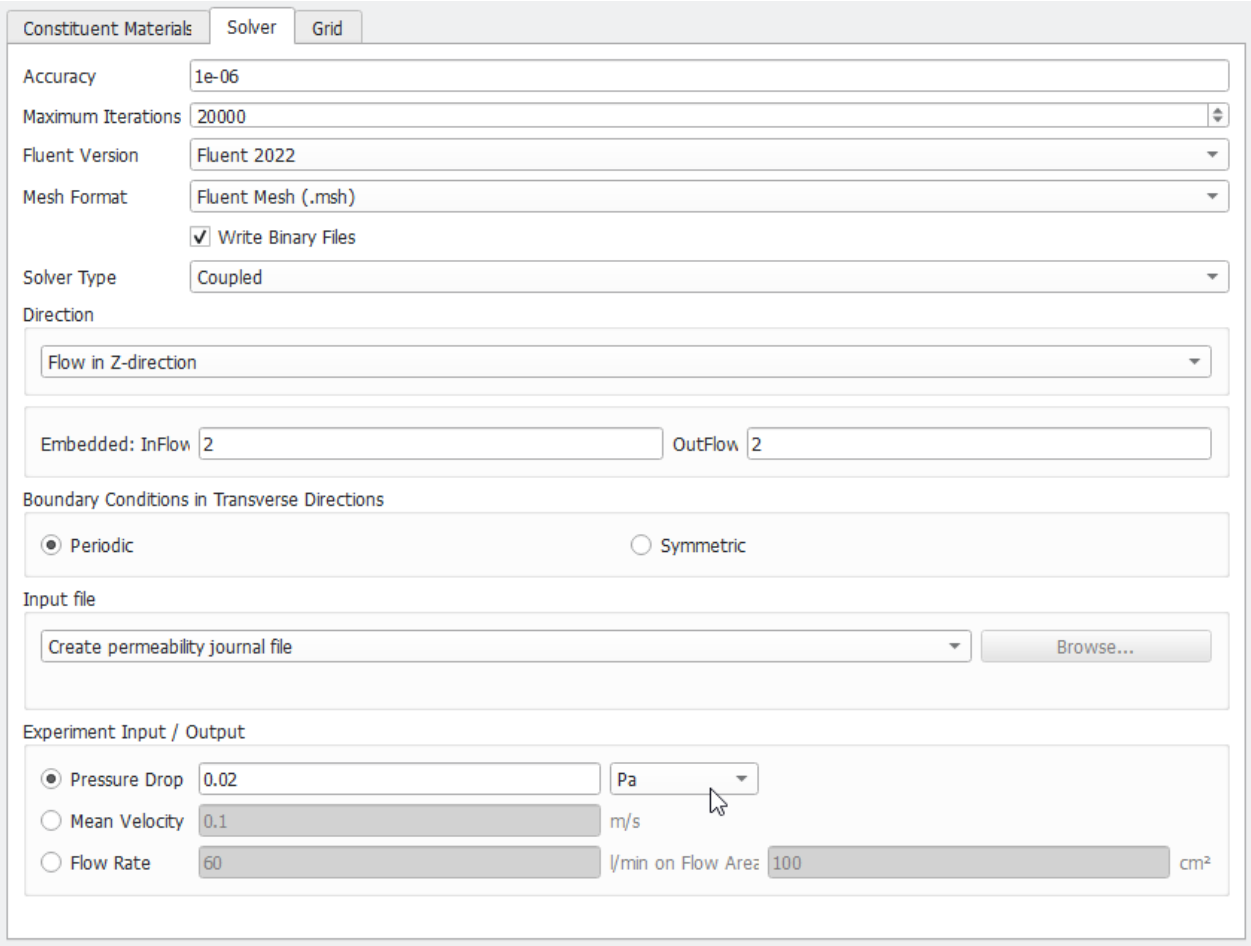
If the structure does not contain a fluid, assign one to a material of a given material ID. For this, click on the material button to open the **Material Selector**. Then, choose a fluid from the material database. To enter the fluid parameters manually, set the material to **Fluid** → **Manual**.



The parameters of the selected fluid (**Density**, **Dynamic Viscosity**, and **Kinematic Viscosity**) are then used in the Fluent simulation.

Solver

All settings for the Fluent solver and the simulation parameters are defined under the **Solver** tab.



Accuracy

The value in **Accuracy** defines the convergence criterion for the Fluent simulation. Therefore, it controls the precision of the computed solution and should be adapted to the simulation task. When there is doubt about the quality of the solution, increase the accuracy by setting a smaller value for that solver.

Maximal Iterations

The number of iterations for the flow solver can be limited with **Maximal Iterations**. When the solver stops because the number of **Maximal Iterations** is reached, the quality of the solution might be doubtful since the requested **Accuracy** is not reached. Therefore, it is recommended to choose a large value here and adapt it only if there are strong constraints on the allowed runtime.

Fluent Version

From the pull-down menu, select your **Fluent Version**. Currently **Fluent 6.2**, **Fluent 6.3**, **Fluent 12**, **Fluent 18** and **Fluent 2022** are supported.

Mesh Format

The desired mesh format (*.msh or *.neu) can be selected from the pull-down menu **Mesh Format**. The check box **Write Binary Files** determines if the created mesh file (*.msh) is stored in binary or ASCII format.

Know how! In binary format file sizes are smaller, whereas ASCII files are human readable.

Solver Type

From the **Solver Type** pull-down menu, select the available implementations of the **Fluent** solver: **SIMPLE**, **SIMPLEC**, **PISO**, and **Coupled**.

Direction

Choose the direction of the flow, available are **Flow in X-**, **Y-**, or **Z-Direction**.

Inflow and Outflow

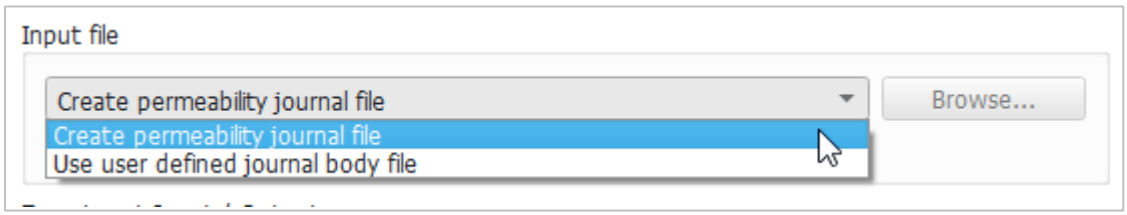
The size of the necessary **Inflow** and **Outflow** region depends on the structure and on the specific simulation task. Choose the number of Voxels that are added at the **Inflow** and at the **Outflow** before export.

Boundary Conditions in Transverse Directions

For **Periodic** boundary conditions, Fluent requires the first and last layer of voxels to be identical on opposite boundaries. This is not necessary for **Symmetric** boundary conditions. Therefore, the exported mesh is slightly larger if **Periodic** is selected.

Input file

In the **Input file** panel, choose between the permeability journal file (**Fluent** computes the permeability in the flow direction) or to use your own journal body file.

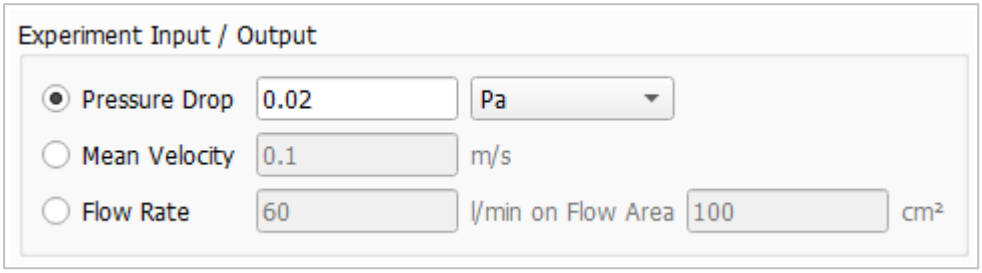


The **Fluent** manuals explain how to create **Fluent** journal body files (*.jbf). Several journal body files (**pressureDrop_X.jbf**, **pressureDrop_Y.jbf**, **pressureDrop_Z.jbf**, and **massFlux.jbf**) are included as examples in the **ExportGeoFluent** folder in the GeoDict

installation directory (e.g., `C:\Program Files\Math2Market GmbH\GeoDict2026\ExportGeoFluent`).

Experiment Input/Output

As Experiment Input, either the **Pressure Drop**, the **Mean Velocity**, or the **Flow Rate** (for a given **Flow Area**) can be defined.

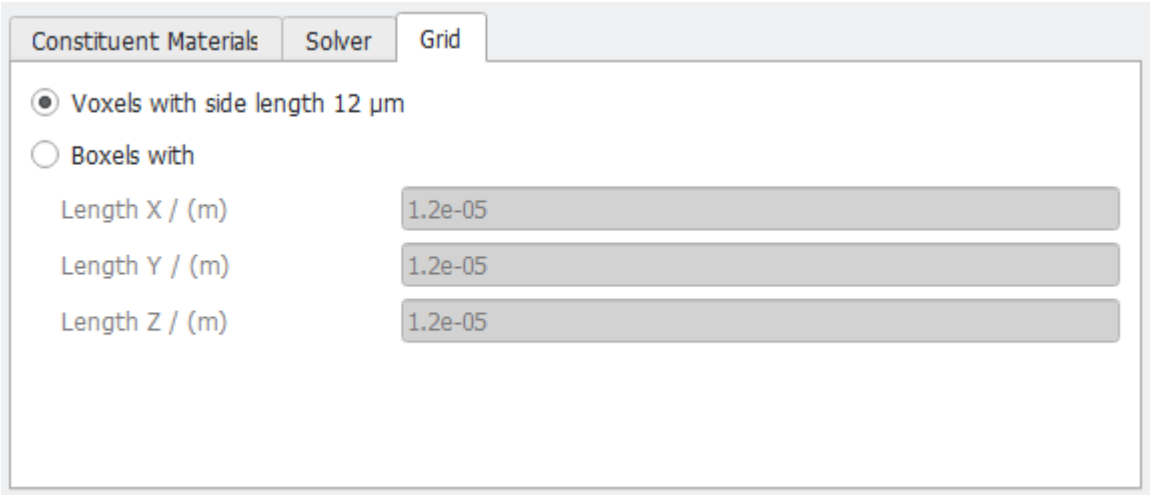


For a given **Pressure Drop** as experiment input, the output is the **Mean Velocity** and vice versa. The **Pressure Drop** is the difference between the inflow and outflow pressures, and the **Mean Velocity** is the average speed of the flow in the flow-direction.

Grid

Structures in GeoDict are always based on voxels. With **Voxels with side length ...**, the voxel structure from GeoDict is exported as-is to Fluent.

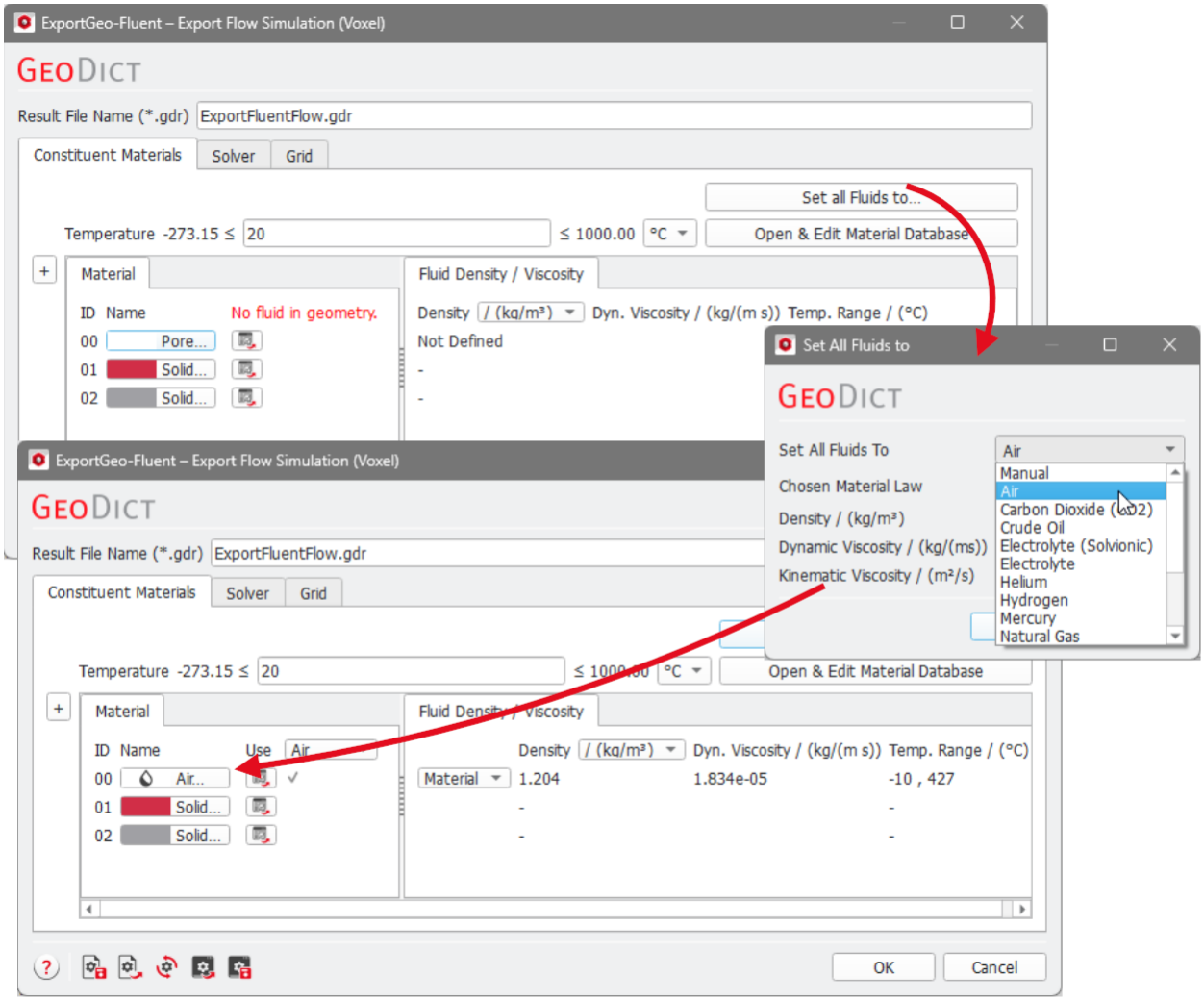
If the simulation in Fluent should be done on a boxel grid, the size of these boxels can be entered with **Boxels with...**. With this option, every voxel in GeoDict is mapped on a boxel with the given size.



1.1.1 Results

In this example, a weave structure with two materials is exported to Fluent to run flow computations.

Under the **Constituent Materials** tab, click **Set all Fluids to...** and select **Air**. Then, the density and viscosity for air from the GeoDict material database are used.

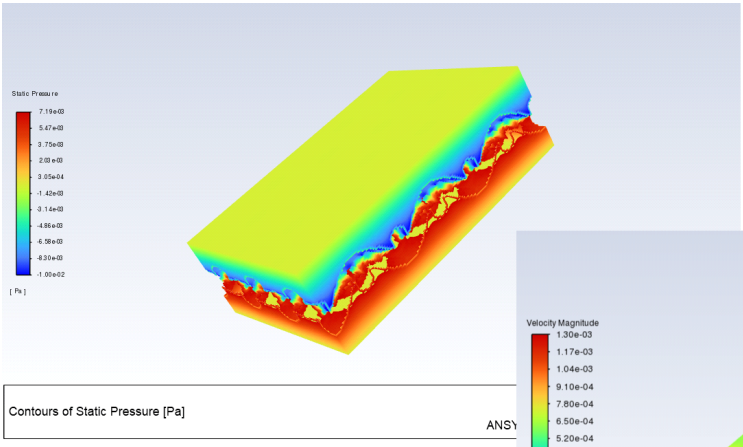


In the **Solver** tab, keep the default values, as well as under the **Grid** tab (**Voxels with side length...**). With this option, the voxel structure from GeoDict is used as it is.

Click **OK** to close the dialog and return to the **ExportGeo-Fluent** section. Click **Export** to export the structure and simulation settings for Fluent.

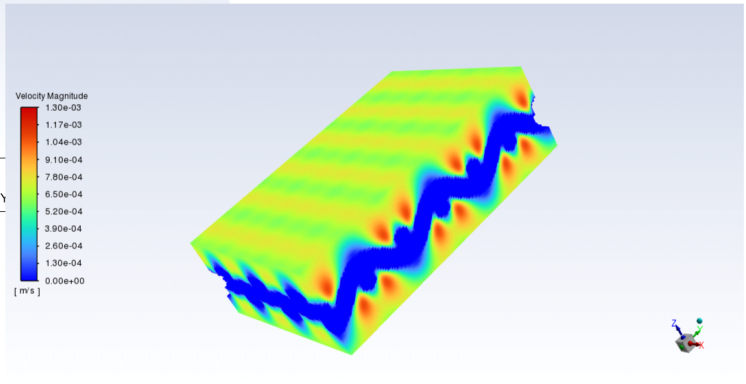
The result file (*.gdr) is automatically opened in the **Result Viewer** and contains all information about the current export. The corresponding result folder contains the mesh and journal files for Fluent.

To run the flow computation, start **Fluent** and open the journal file by selecting **File** → **Read** → **Journal...**. After the simulation, the computation results can be visualized in **Fluent**. In the figure below, contour plots of the pressure and mean velocity in the structure are shown.



Contours of Static Pressure [Pa]

ANSY



Contours of Velocity Magnitude [m/s]

Sep 02, 2022
ANSYS Fluent 2022 R1 (3d, dp, pbns, sstkw)

1.2 Run Flow Simulation (Voxel)

Run Flow Simulation (Voxel) contains the same options as **Export Flow Simulation (Voxel)**, but the simulation in Fluent is directly started after clicking on **Run** and the results are shown in GeoDict after the simulation is finished. A .gdr file is created and velocity and pressure fields can be loaded and visualized in GeoDict.

Important! The same results can be achieved with the FlowDict module directly in GeoDict.

In general, this is the recommended way if FlowDict is in your GeoDict license, since the GeoDict solvers are optimized for voxel structures and therefore faster and more memory efficient for this use case.

The available options are explained under Export Flow Simulation (Voxel).

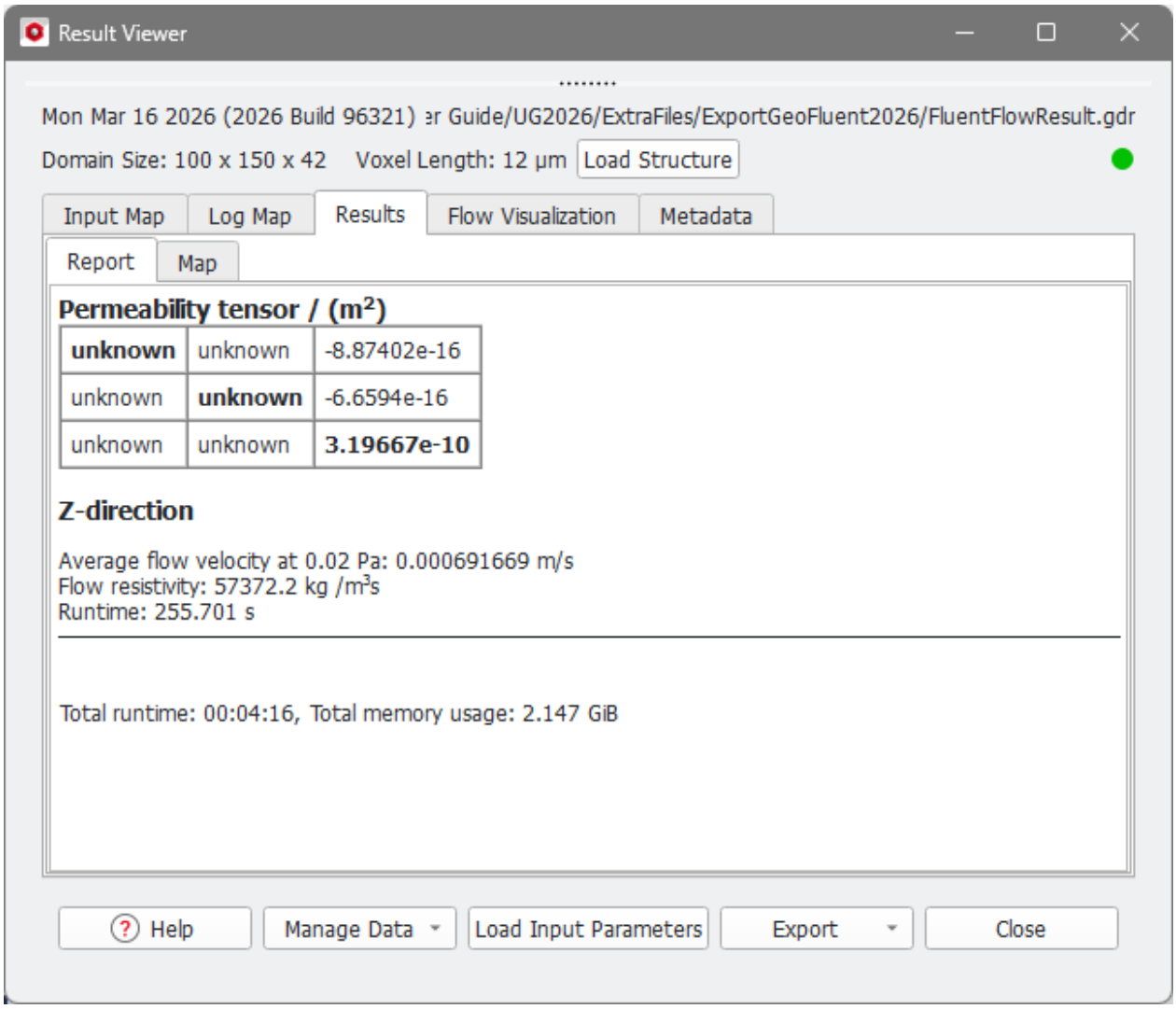
To make **Run Flow Simulation (Voxel)** work in ExportGeo-Fluent, **Ansys Fluent** must be installed on the same computer. Moreover, GeoDict must know the installation path of **Ansys Fluent**. To achieve that, open **GD2026_externals.py** found in `C:\Program Files\Math2Market GmbH\GeoDict 2026\` in Windows or in the installation folder in Linux.

```
416     mpi_args = []
417
418     matlab = 'matlab' + suffix
419
420     cadExec=r"C:\Program Files\CADlook\CADlook64_V14.0\CADlook64.exe"
421     for version in cadlookVersions:
422         if os.path.exists(version):
423             cadExec = version
424     if manual_cadlook_path != "" and os.path.exists(manual_cadlook_path):
425         cadExec = manual_cadlook_path
426
427     fluentExec=r"C:\Program Files\ANSYS Inc\v252\fluent\ntbin\win64\fluent.exe"
428
429     # ====== setup internal executables =======
430
```

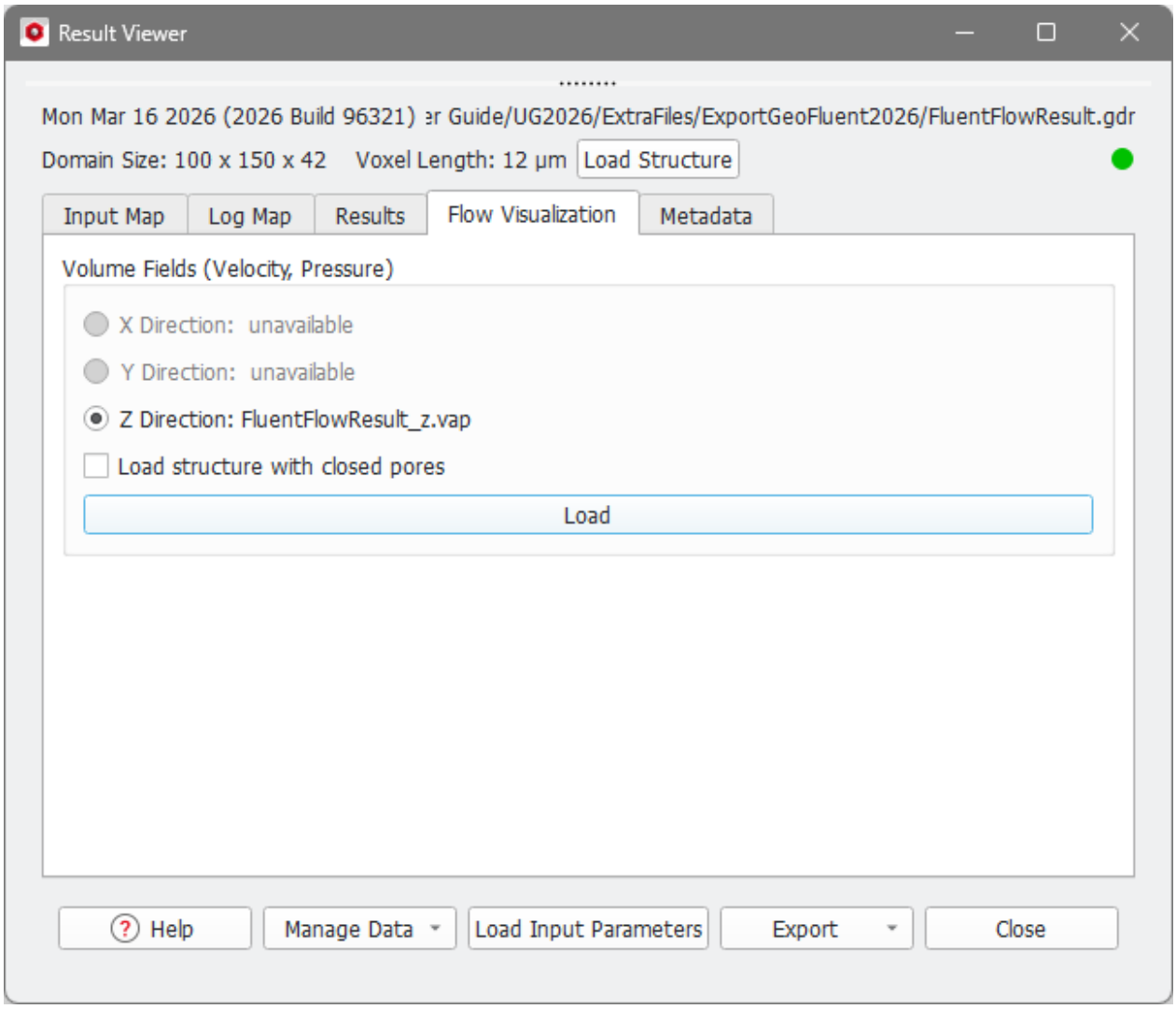
Find the line specifying the path of “fluentExec” and replace the path with your installation path of **Ansys Fluent** on your computer.

1.2.1 Results

The result of **Run Flow Simulation (Voxel)** provides a .gdr file, just like the other GeoDict commands. The permeability tensor is reported, as well as the other information of the simulation.



A visualization of the flow field is also possible by loading the .vap file under the **Flow Visualization** tab.


Result Viewer
Mon Mar 16 2026 (2026 Build 96321) ar Guide/UG2026/ExtraFiles/ExportGeoFluent2026/FluentFlowResult.gdr
Domain Size: 100 x 150 x 42
Voxel Length: 12 µm
Load Structure
Input Map
Log Map
Results
Flow Visualization
Metadata
Volume Fields (Velocity, Pressure)
X Direction: unavailable
Y Direction: unavailable
Z Direction: FluentFlowResult_z.vap
Load structure with closed pores
Load
Help
Manage Data
Load Input Parameters
Export
Close

1.3 Export Heat Simulation (Voxel)

To generate a journal file and a mesh file for a heat computation with **Fluent**, select **Export Heat Simulation (Voxel)** from the pull-down menu in the ExportGeo-Fluent section.

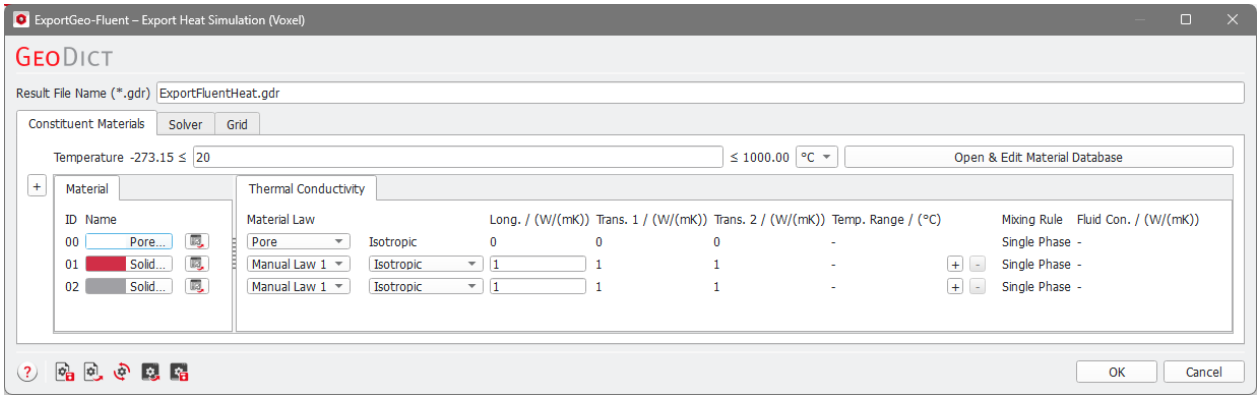
Important! The same results can be achieved with the ConductoDict module directly in GeoDict.

In general, this is the recommended way if ConductoDict is in your GeoDict license, since the GeoDict solvers are optimized for voxel structures and therefore faster and more memory efficient for this use case.

In the **ExportGeo-Fluent – Export Heat Simulation (Voxel)** dialog, choose a **Result File Name (*.gdr)** according to your current project. The result file contains all information about the current export. The corresponding result folder contains the mesh and journal files for Fluent and the chosen structure as GDT file. The options for the **Fluent** solver are organized under the tabs **Constituent Materials**, **Solver**, and **Grid**.

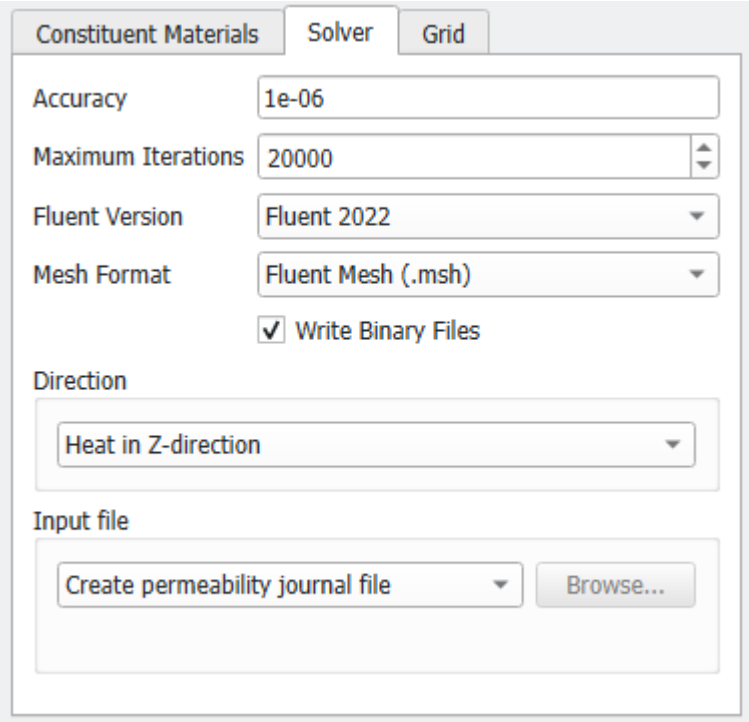
Constituent Materials

The thermal conductivities of all materials present in the structure are defined in the corresponding fields under the **Constituent Materials** tab.



Solver

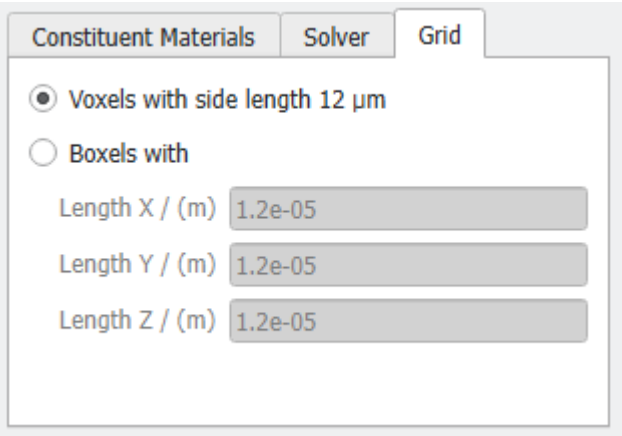
All settings for the Fluent solver and the simulation parameters are defined under the **Solver** tab, and are already explained for Export Flow Simulation (Voxel) 7. In contrast to flow simulations, Fluent V. 6.2 is not supported for heat simulations.



Grid

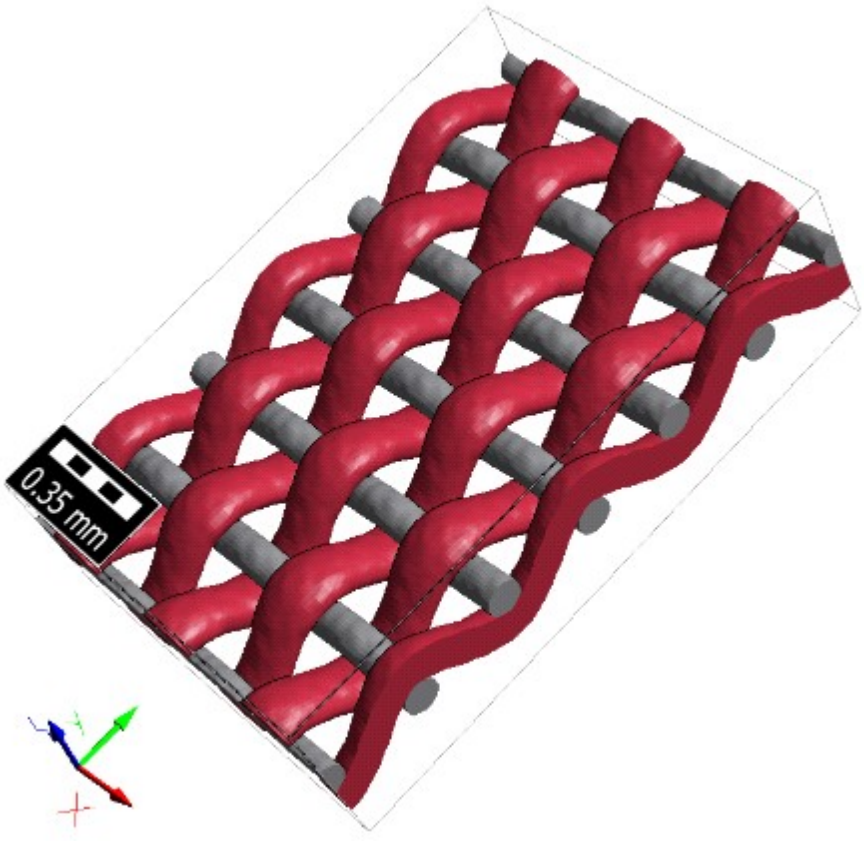
Structures in GeoDict are always based on voxels. With **Voxels with side length ...**, the voxel structure from GeoDict is exported as-is to Fluent.

If the simulation in Fluent should be done on a boxel grid, the size of these boxels can be entered with **Boxels with...**. With this option, every voxel in GeoDict is mapped on a boxel with the given size.

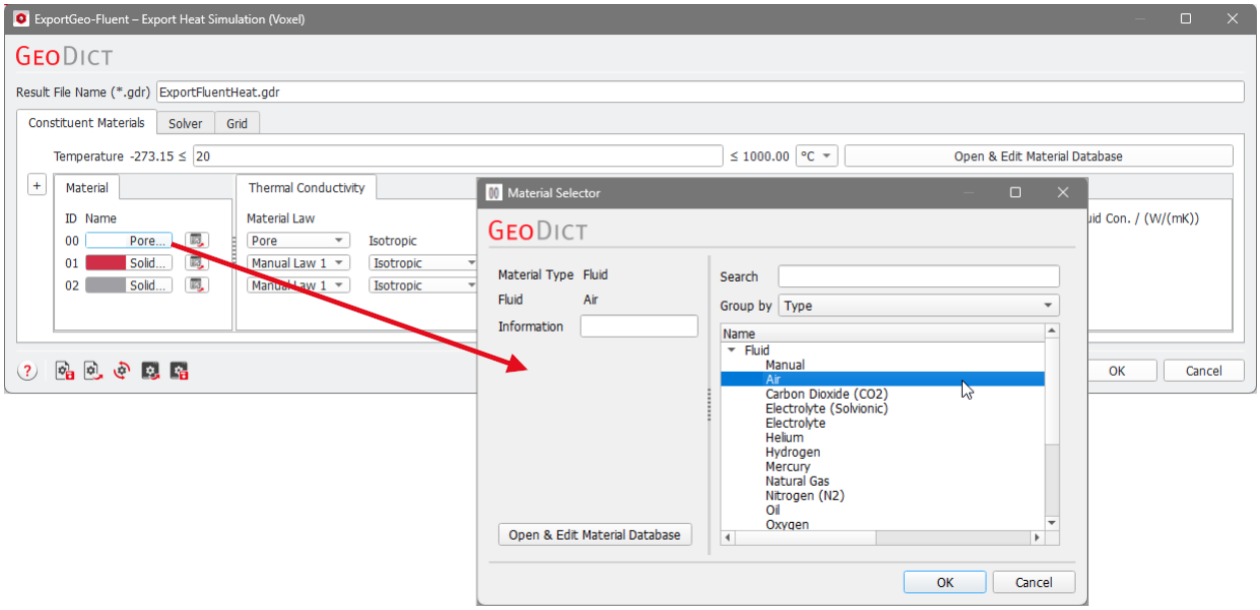


1.3.1 Results

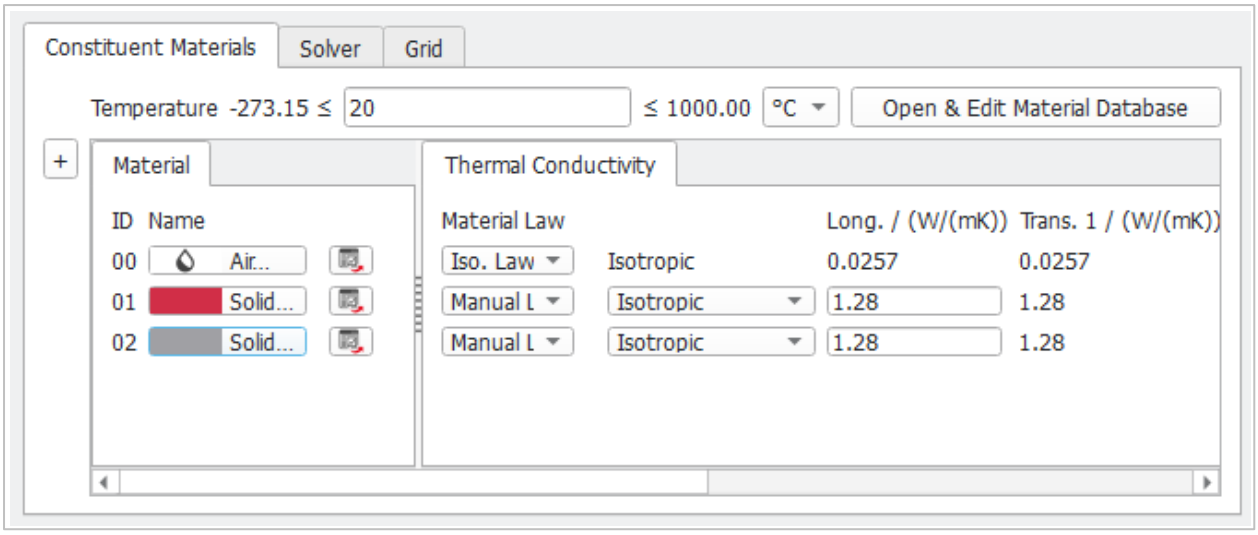
In this example, a weave structure with two materials is exported to Fluent to run heat computations.



Under the **Constituent Materials** tab, set the Material ID of the material occupying the pore space by clicking the **Pore...** button and selecting **Pore (Fluid)** → **Air** from the list in the Material Selector.



Then, the conductivity for air from the GeoDict material database is used. Enter the conductivity for Material ID 01 and Material ID 02 manually as shown below.



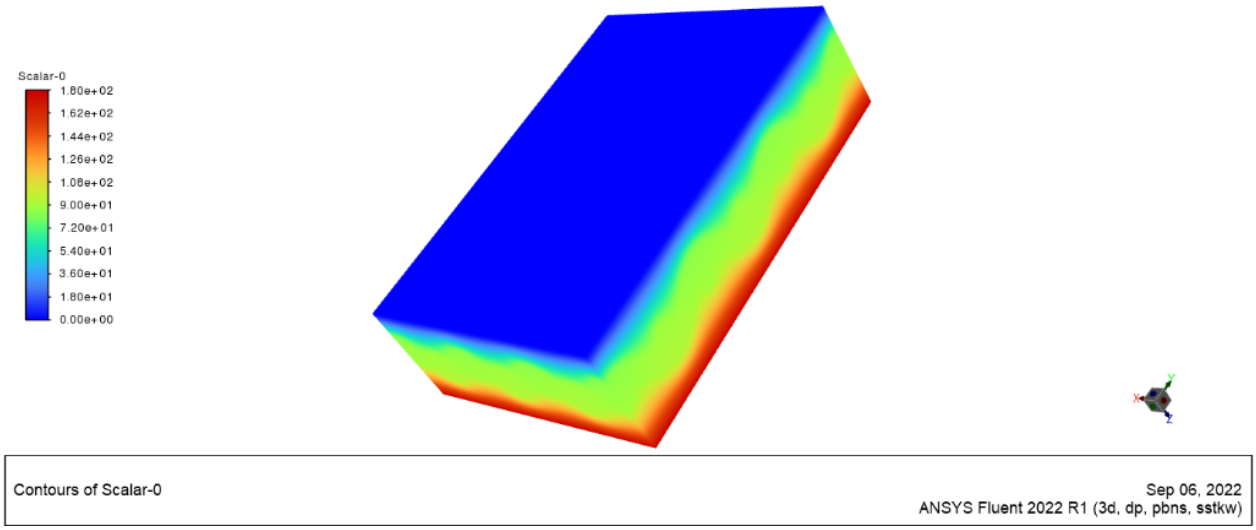
In the **Solver** tab, keep the default values, as well as under the **Grid** tab (**Voxels with side length...**). With this option, the voxel structure from GeoDict is used as it is.

Click **OK** to close the dialog and return to the ExportGeo-Fluent section. Click **Export** to export the structure and simulation settings for Fluent.

The result file (*.gdr) is automatically opened in the **Result Viewer**. It contains all information about the current export. The corresponding result folder contains the mesh and journal files for Fluent.

To run the flow computation, start **Fluent** and open the journal file by selecting **File** → **Read** → **Journal...**.

After the simulation, the computation results can be visualized in **Fluent**. For example, the temperature distribution in the structure is shown in the figure below.



1.4 Run Heat Simulation (Voxel)

Run Heat Simulation (Voxel) contains the same options as **Export Heat Simulation (Voxel)** 17, but the Fluent simulation is directly started after clicking **Run**.

> **Important!** The same results can be achieved with the ConductoDict module directly in GeoDict.
>
> In general, this is the recommended way if ConductoDict is in your GeoDict license, since the GeoDict solvers are optimized for voxel structures and therefore faster and more memory efficient for this use case.

The available options are explained under Export heat simulation (voxel) 17.

To make **Run Heat Simulation (Voxel)** work in ExportGeo-Fluent, **Ansys Fluent** must be installed on the same computer. Moreover, GeoDict must know the installation path of **Ansys Fluent**. To achieve that, open **GD2026_externals.py** found in `C:\Program Files\Math2Market GmbH\GeoDict 2026\` in Windows or in the installation folder in Linux.

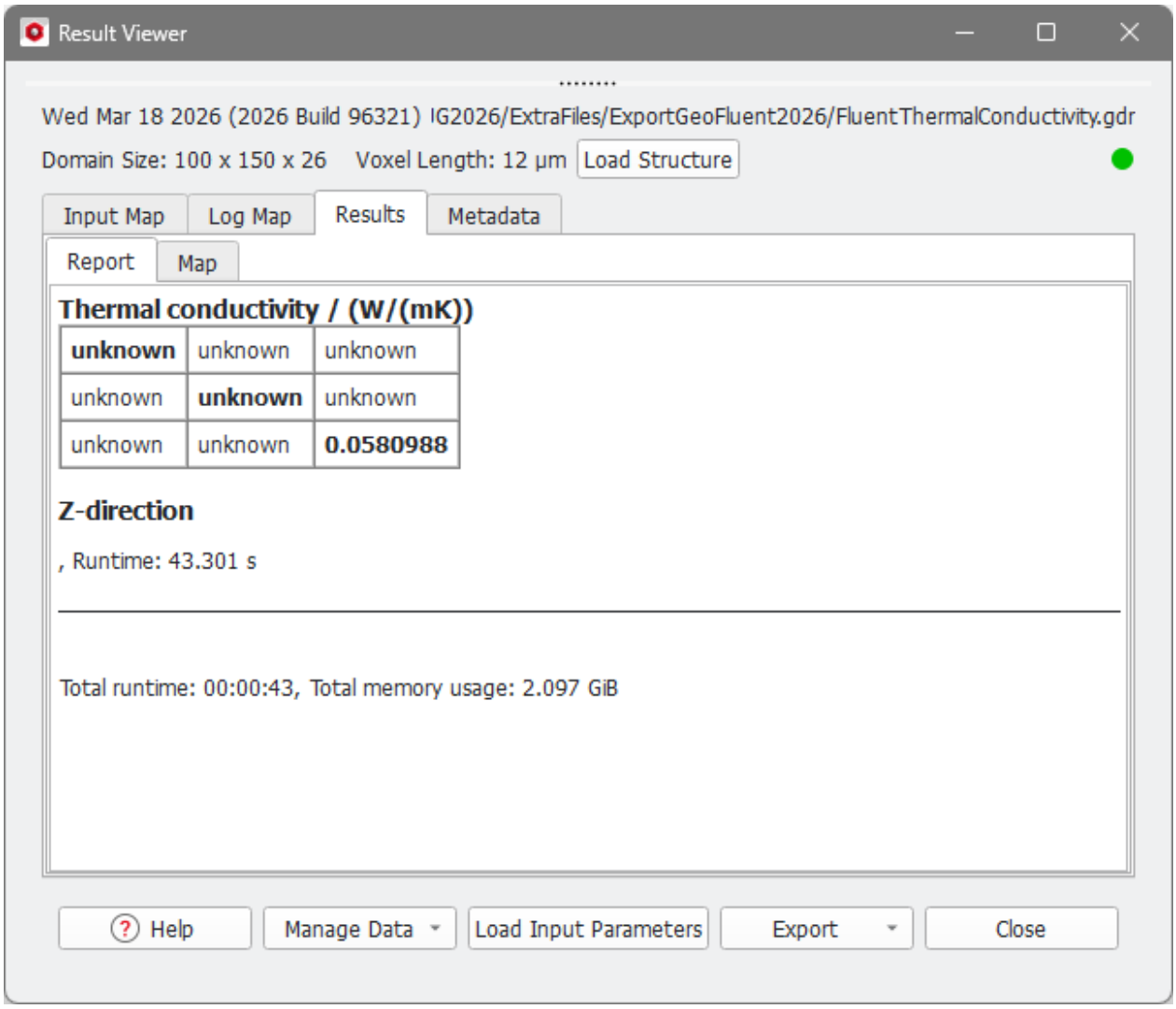
```
416     mpi_args = []
417
418     matlab = 'matlab' + suffix
419
420     cadExec=r"C:\Program Files\CADlook\CADlook64_V14.0\CADlook64.exe"
421     for version in cadlookVersions:
422         if os.path.exists(version):
423             cadExec = version
424     if manual_cadlook_path != "" and os.path.exists(manual_cadlook_path):
425         cadExec = manual_cadlook_path
426
427     fluentExec=r"C:\Program Files\ANSYS Inc\v252\fluent\ntbin\win64\fluent.exe"
428
429     # ====== setup internal executables =======
430
```

Find the line specifying the path of "fluentExec" and replace the path with your installation path of **Ansys Fluent** on your computer.

Additionally, for the heat case, a User Defined Function (UDF) is provided to calculate the temperature gradient. This UDF must be compiled in Fluent when executing the journal file exported from **Export Heat Simulation (Voxel)** or when running **Run Heat Simulation (Voxel)**. Therefore, your computer should have a compiler installed.

1.4.1 Results

After the computation is finished, a .gdr file is generated and reports the thermal conductivity.


Result Viewer
Wed Mar 18 2026 (2026 Build 96321) IG2026/ExtraFiles/ExportGeoFluent2026/FluentThermalConductivity.gdr
Domain Size: 100 x 150 x 26 Voxel Length: 12 µm
Load Structure
Input Map
Log Map
Results
Metadata
Report
Map
Thermal conductivity / (W/(mK))
unknown unknown unknown
unknown unknown unknown
unknown unknown 0.0580988
Z-direction
, Runtime: 43.301 s
Total runtime: 00:00:43, Total memory usage: 2.097 GiB
Help
Manage Data
Load Input Parameters
Export
Close

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

MATH 2 MARKET

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