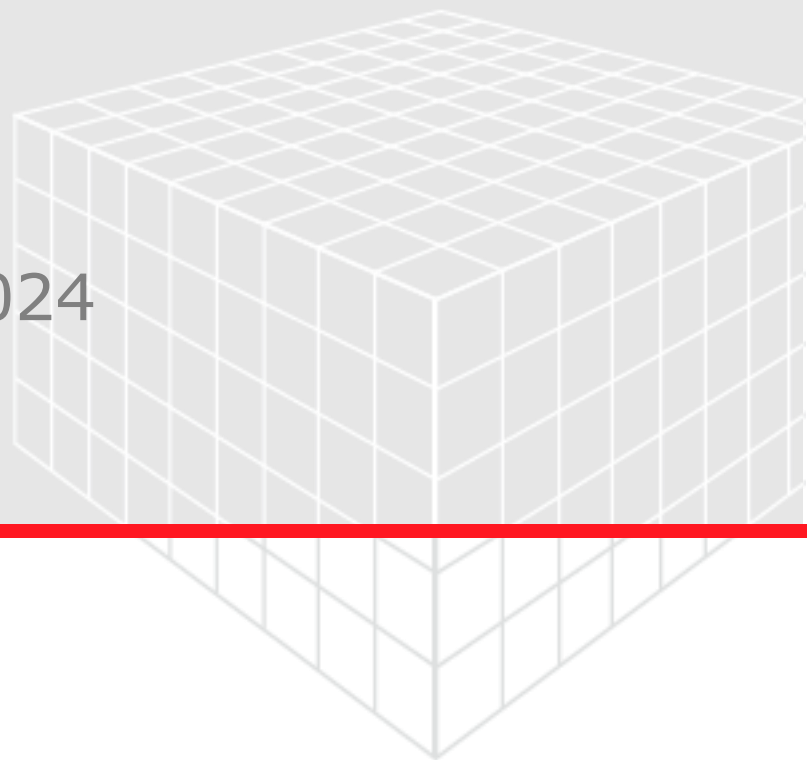


EXPORTGEO-FLUENT

User Guide

GeoDict release 2024

Published: January 19, 2024



GEO DICT

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

© Math2Market GmbH 2024

Citation:

Liping Cheng, Jürgen Becker, Barbara Planas. GeoDict 2024 User Guide. ExportGeo-Fluent handbook. Math2Market GmbH, Germany, doi.org/10.30423/userguide.geodict

All rights reserved. It is not permitted to reproduce the book or parts thereof in any form by photocopy, microfilm or other methods or to transfer it into a language suitable for machines, in particular data processing systems, without the express permission of the publisher. The same applies to the right of public reproduction.

The handbooks in the User Guide series of Math2Market GmbH can be obtained from:

Math2Market GmbH
Richard-Wagner-Strasse 1
67655 Kaiserslautern
Germany

Phone: +49 631 205 605 0
Fax: +49 631 205 605 99
Email: info@math2market.de
Web: www.math2market.de

EXPORTGEO SUBMODULES	1
EXPORTGEOFLUENT	1
EXPORT FLOW SIMULATION (VOXEL)	3
Constituent Materials	3
Solver	5
Grid	7
Example: Export to Fluent for flow computations	8
EXPORT HEAT SIMULATION (VOXEL)	11
Constituent Materials	11
Solver	12
Grid	12
Example: Export to Fluent for heat computations	13
RUN HEAT SIMULATION (VOXEL)	16

EXPORTGEO SUBMODULES

With **ExportGeo**, structures from **GeoDict** can be exported to other formats to make them available in third-party software.

ExportGeo converts voxelized and analytic data material models through three submodules:

- **Export as CAD (**MeshGeo**)** converts voxelized or analytic structures to popular surface triangulation and CAD formats like STL, VRML, or Parasolid.
- **ExportGeo to Fluent (**ExportGeoFluent**)** converts voxelized data to formats for flow and heat computations with **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.
- **Export to Abaqus (**ExportGeoAbaqus**)** converts voxelized data to formats for elasticity computations with **Abaqus**. The .inp files are saved as ASCII files so that, if changes are required, they can be easily opened and edited with a text editor.

Based on the voxel structure, **GeoDict** also generates an unstructured volume mesh (.msh file), as needed for **Fluent™** and **Abaqus**.

EXPORTGEOFLUENT

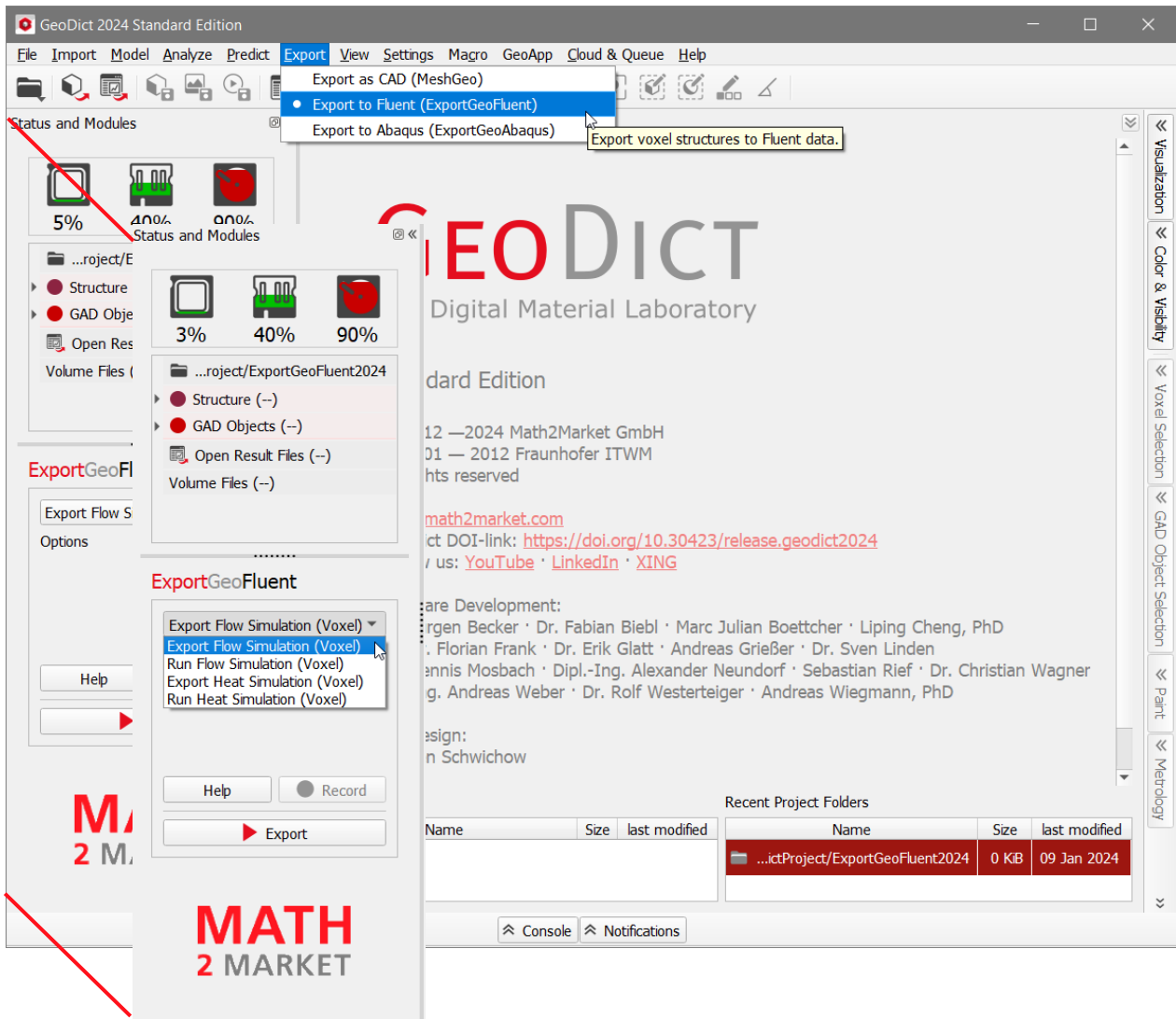
With **ExportGeoFluent**, it is possible to export structures from **GeoDict** and to run fluid flow and heat computations with **Fluent™** on them. The voxel structure in **GeoDict** is exported to a mesh file (*.msh or *.neu), the solver parameters for the **Fluent** simulation are recorded to a journal file (*.jou). Those two files together contain all necessary information for the simulation in **Fluent™**.

The same results can be achieved with the **FlowDict** and **ConductoDict** modules directly in **GeoDict** (for further information, refer to the [FlowDict](#) and the [ConductoDict](#) handbooks of this User Guide). 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.

To access **ExportGeoFluent**, select **Export** → **Export to Fluent (ExportGeoFluent)** in the menu bar.

In the **ExportGeoFluent** section, choose an export option in the pull-down menu. With **Export Flow Simulation (Voxel)** and **Export Heat Simulation (Voxel)**, the current voxel structure is exported to **Fluent** and the simulation parameters are recorded as *.jou file. Additionally, with **Run Flow Simulation (Voxel)** and **Run Heat Simulation (Voxel)**, the **Fluent** simulation is directly started after the export.

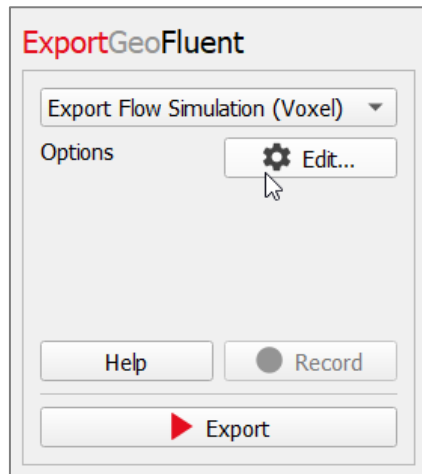
To execute the *.jou file in **Fluent™**, the 3D double-precision solver (fluent 3ddp) must be selected. When exporting under Windows and running simulations under Linux, it might be necessary to convert the *.jou and *.msh files to the Unix format with the dos2unix command.



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 **ExportGeoFluent** section.

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.

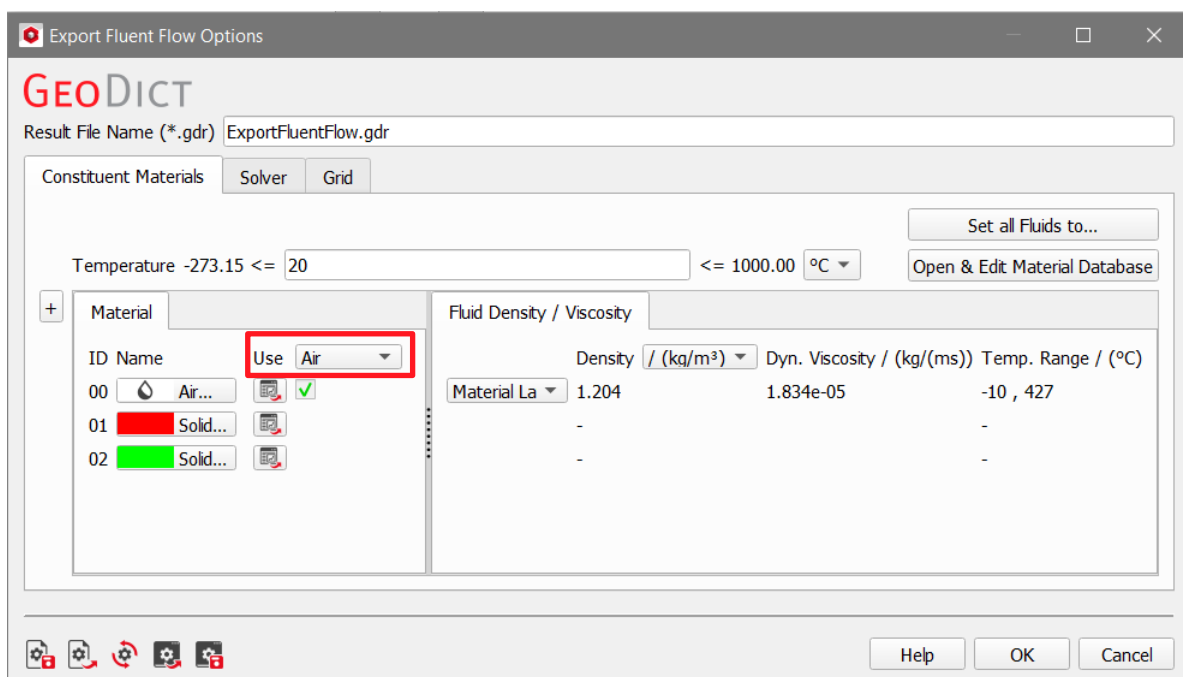


In the **Export Fluent Flow Options** dialog, choose a **Result File Name (*.gdr)** according to the current project. The result file contains all information about the current export. The corresponding result folder contains mesh and journal files for Fluent and the chosen structure.

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

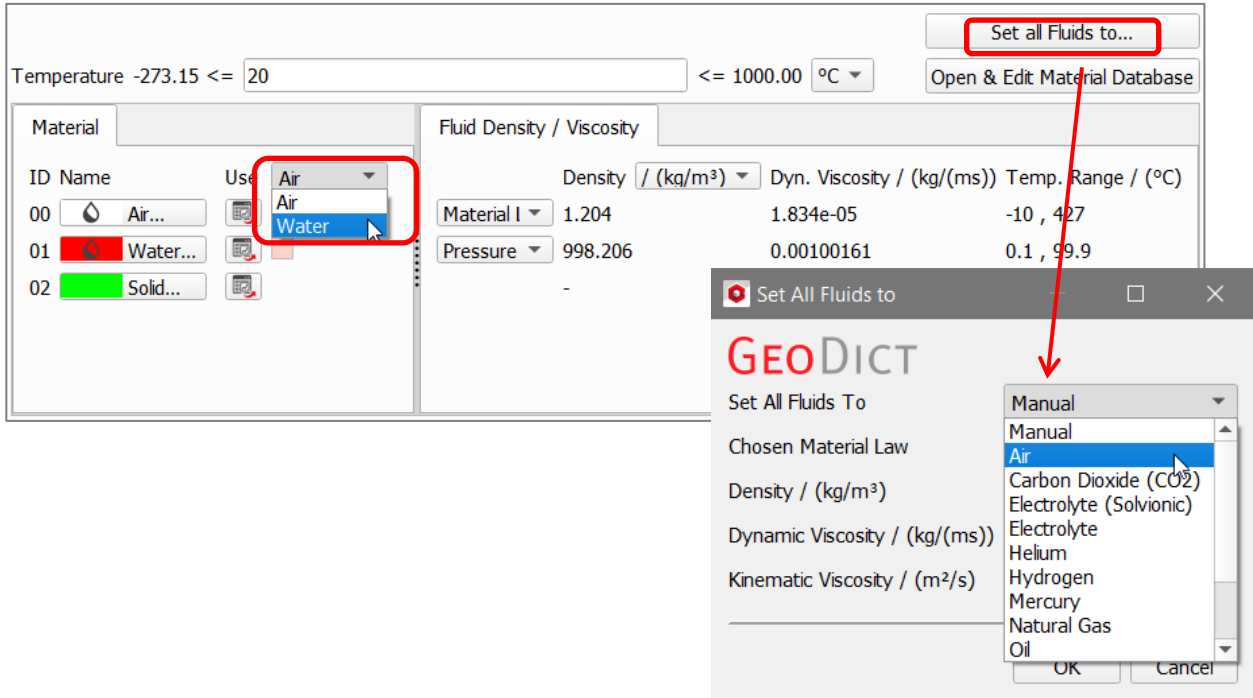
CONSTITUENT MATERIALS

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



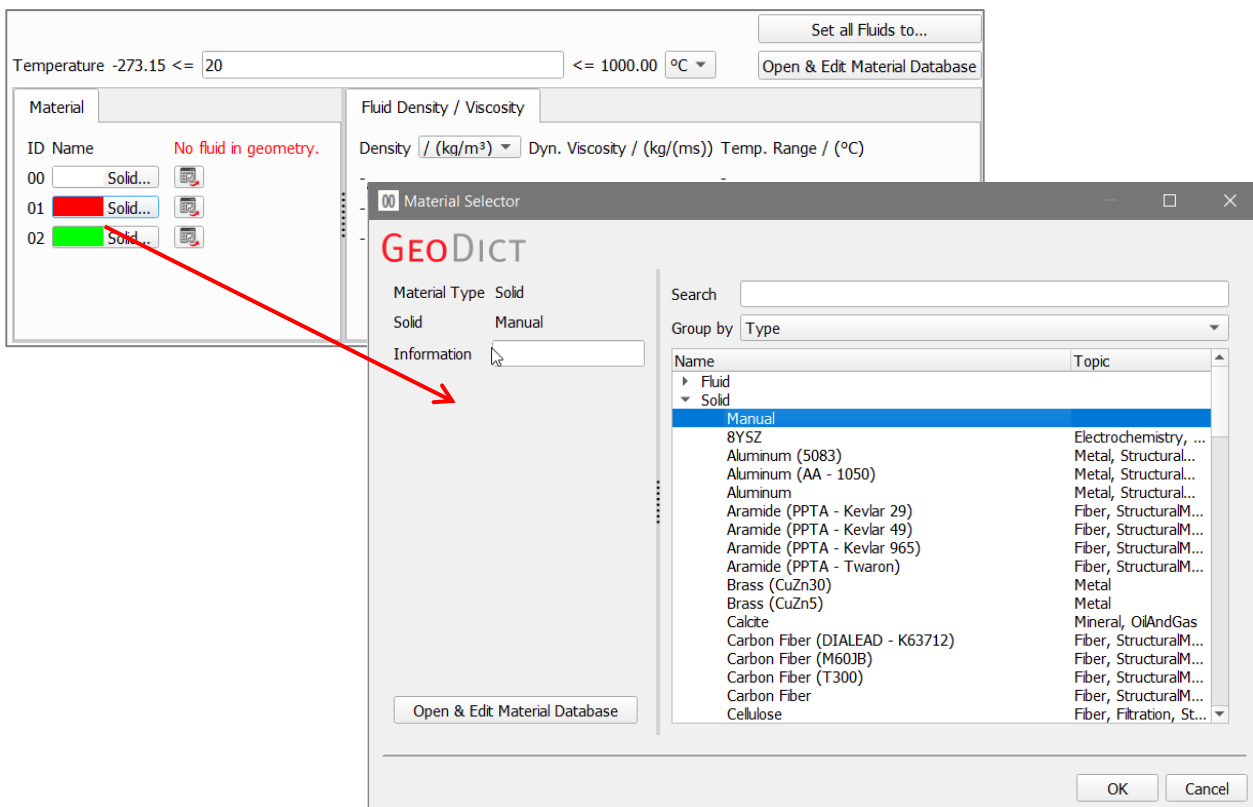
If one fluid is present in the structure, this fluid is chosen automatically. If the structure contains more than one fluid, the fluid used for the flow simulation must be selected through the **Use** pull-down menu. Then, the other fluids in the structure are treated as solid in the simulation.

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



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**.

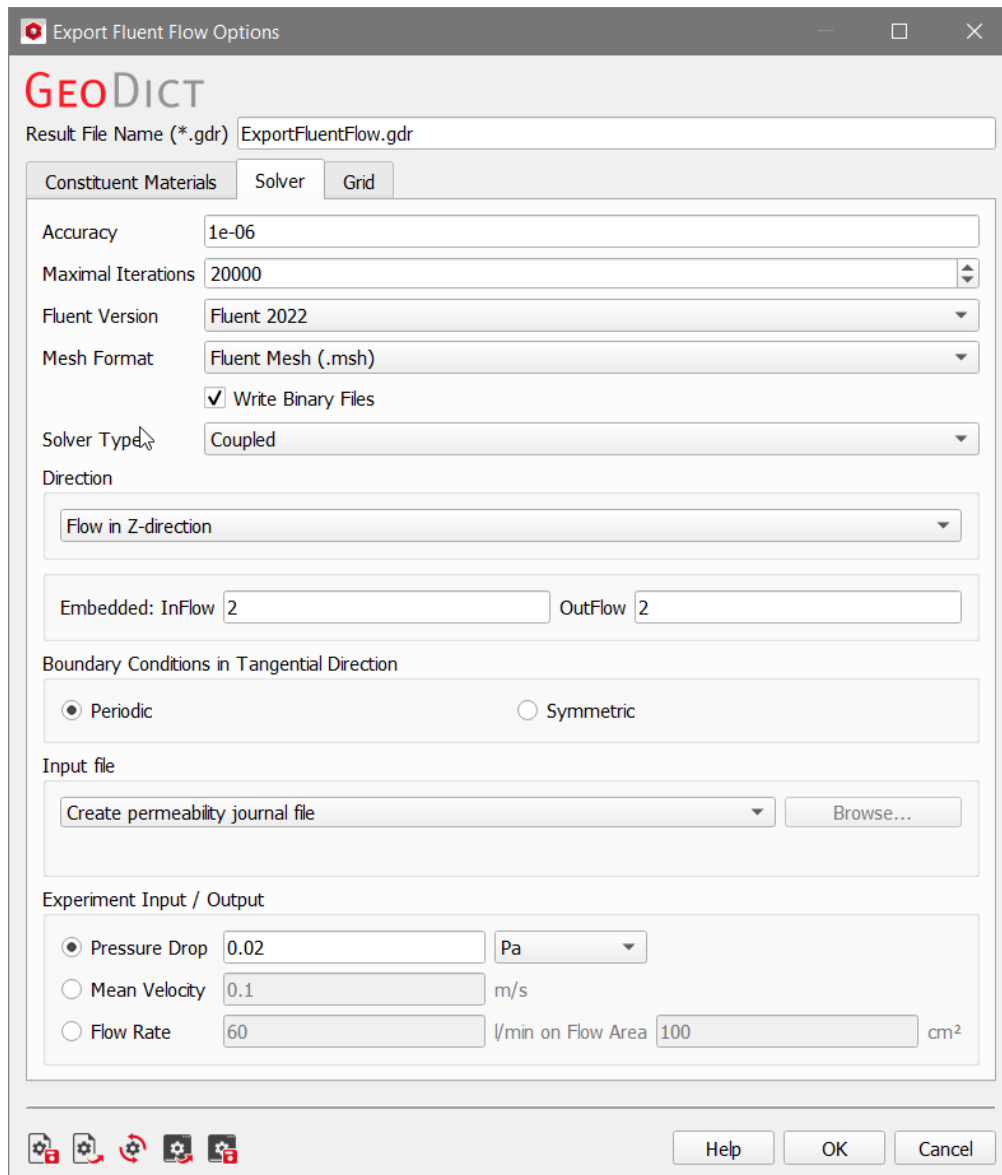
Then, choose a fluid from the material database. To enter the fluid parameters manually, set the material to **Fluid** → **Manual**.



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

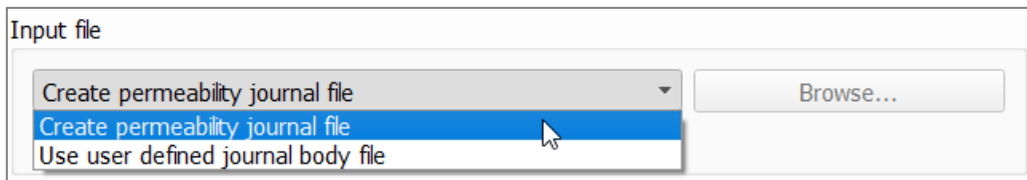
SOLVER

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



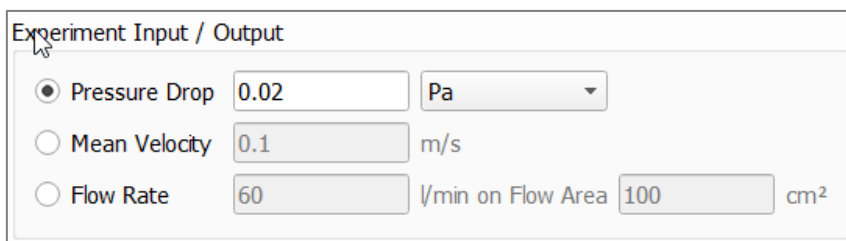
- **Accuracy:** The value in **Accuracy** defines the convergence criterion for the Fluent simulation. Therefore, it controls the precision of the computed solution and should be adapted to the simulation task. When there is doubt about the quality of the solution, increase the accuracy by setting a 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. Note that 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: **Coupled**, **SIMPLE**, **SIMPLEC** and **PISO**.
- **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 Tangential Direction:** Choose the **Boundary Conditions in Tangential Direction**. For the **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\GeoDict2024\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.

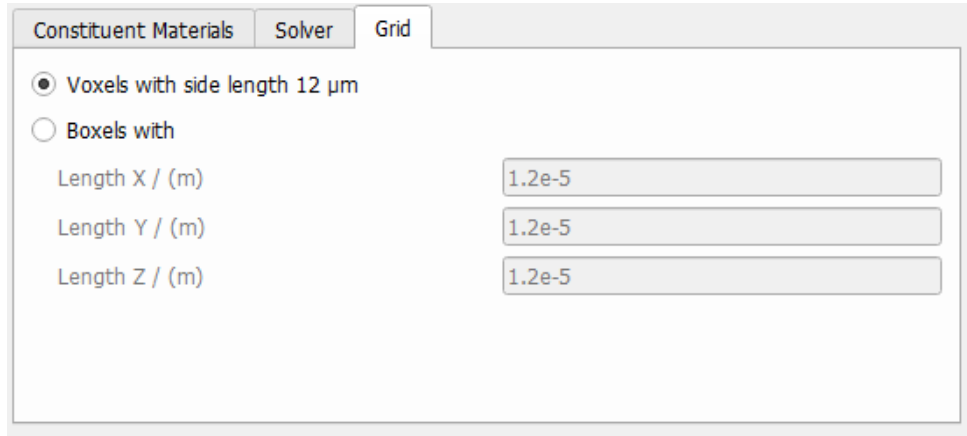


For a given **Pressure Drop** as experiment input, the output is the **Mean Velocity** and vice versa. The **Pressure Drop** is the difference between the inflow and outflow pressures, and the **Mean Velocity** is the average speed of the flow in the flow-direction.

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.

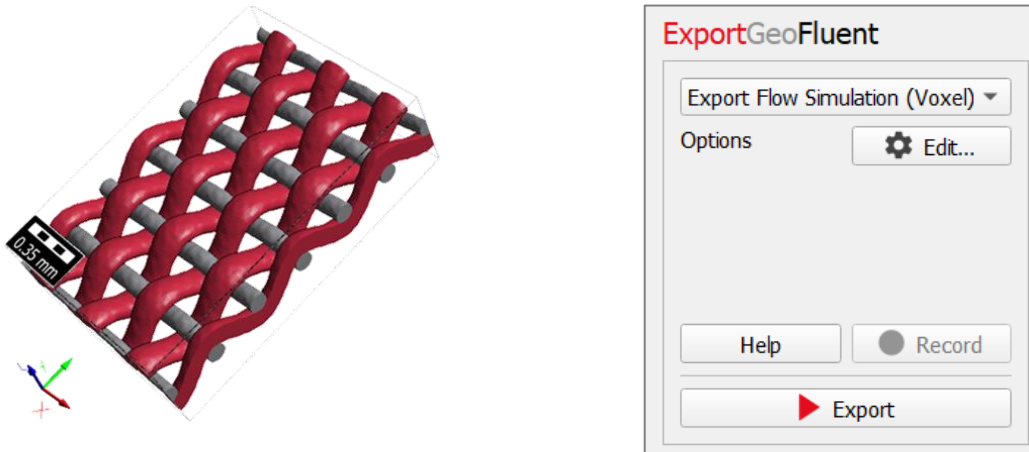
If the simulation in Fluent should be done on a boxel grid, the size of these boxels can be entered with **Boxels with** With this option, every voxel in **GeoDict** is mapped on a boxel with the given size.



EXAMPLE: EXPORT TO FLUENT FOR FLOW COMPUTATIONS

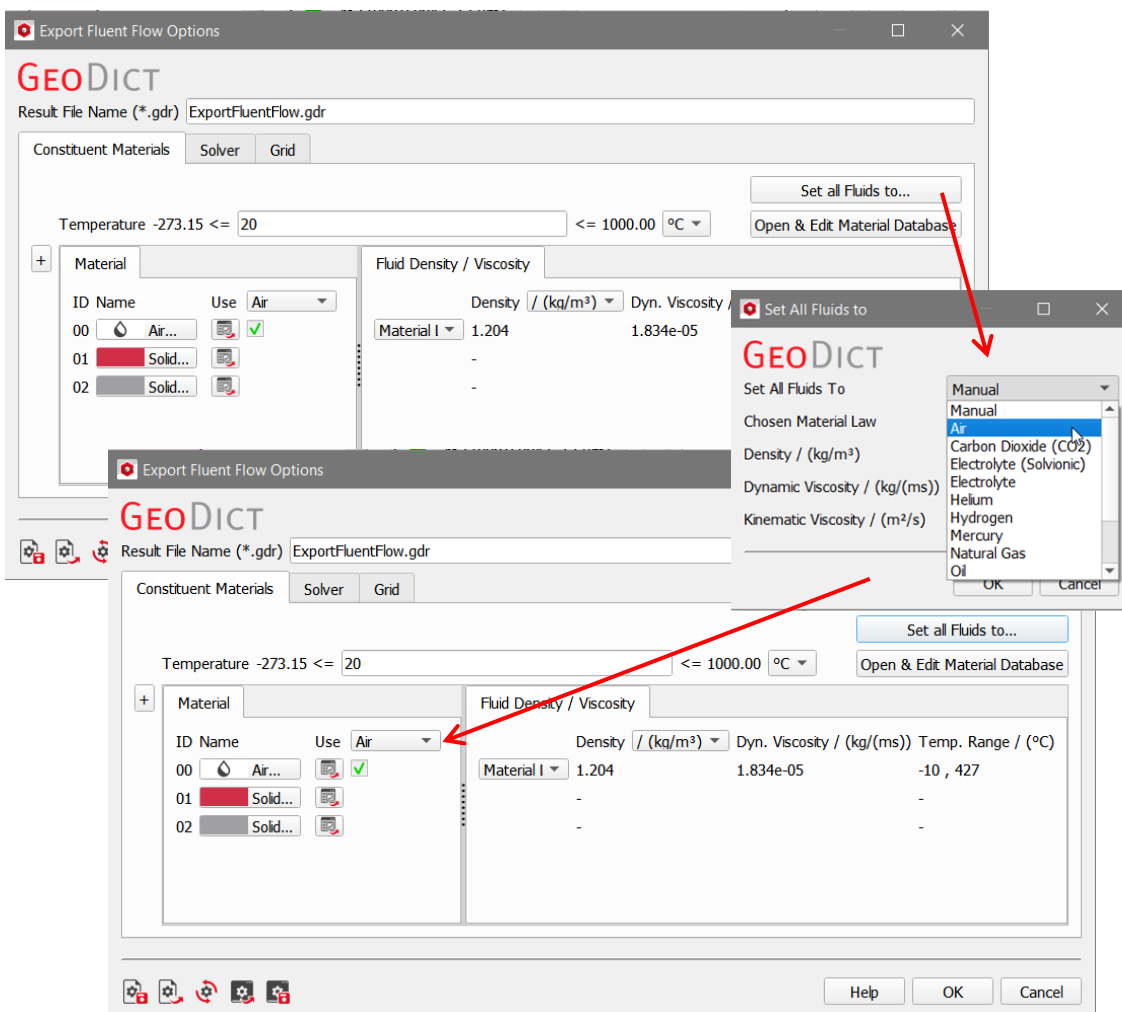
Start **ExportGeoFluent** by selecting **Export** → **Export to Fluent (ExportGeoFluent)** in the menu bar. Select **Export Flow Simulation (Voxel)** from the pull-down menu in the **ExportGeo-Fluent** section.

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



Click the **Options' Edit...** button to open the **Export Fluent Flow Options** dialog. Then, enter the parameters for the Fluent solver.

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:

The screenshot shows the Solver tab of a software interface. The parameters are as follows:

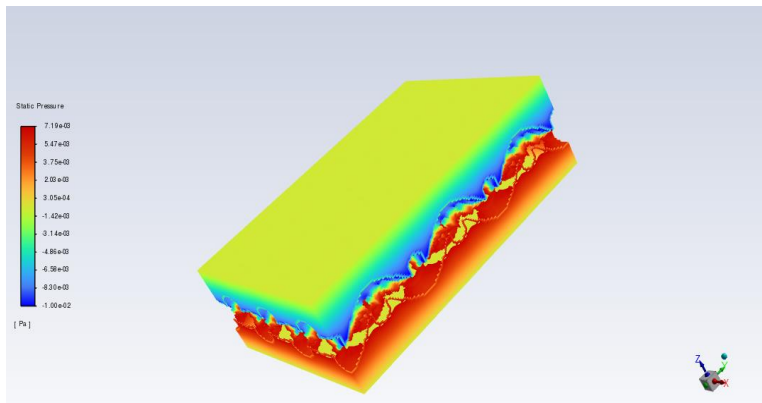
- Accuracy: 1e-06
- Maximal Iterations: 20000
- Fluent Version: Fluent 2022
- Mesh Format: Fluent Mesh (.msh)
- Write Binary Files
- Solver Type: Coupled
- Direction: Flow in Z-direction
- Embedded: InFlow 2, OutFlow 2
- Boundary Conditions in Tangential Direction:
 - Periodic
 - Symmetric
- Input file: Create permeability journal file (with a Browse... button)
- Experiment Input / Output:
 - Pressure Drop: 0.02 Pa
 - Mean Velocity: 0.1 m/s
 - Flow Rate: 60 l/min on Flow Area 100 cm²

Also keep the default value (**Voxels with side length ...**) under the **Grid** tab. With this option, the voxel structure from **GeoDict** is used as it is. Click **OK** to close the **Export Fluent Flow Options** dialog and return to the **ExportGeo-Fluent** section. Click **Export** to export the structure and simulation settings for Fluent.

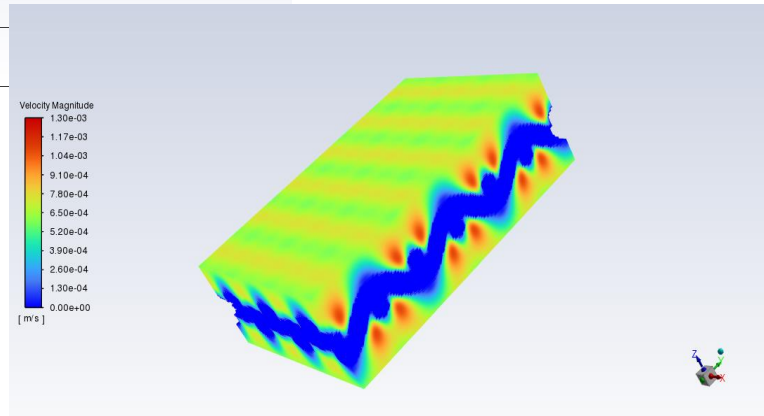
The result file (*.gdr) is automatically opened in the **Result Viewer** and contains all information about the current export. The corresponding result folder contains the mesh and journal files for Fluent.

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]

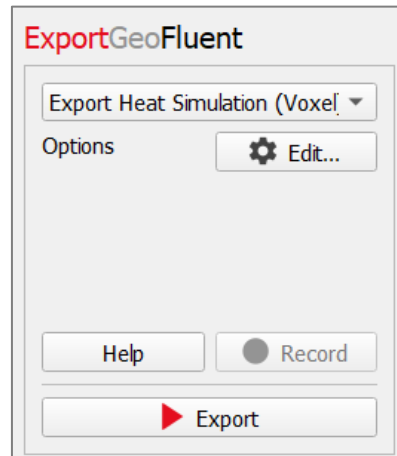


Contours of Velocity Magnitude [m/s]

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

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 **ExportGeoFluent** section.

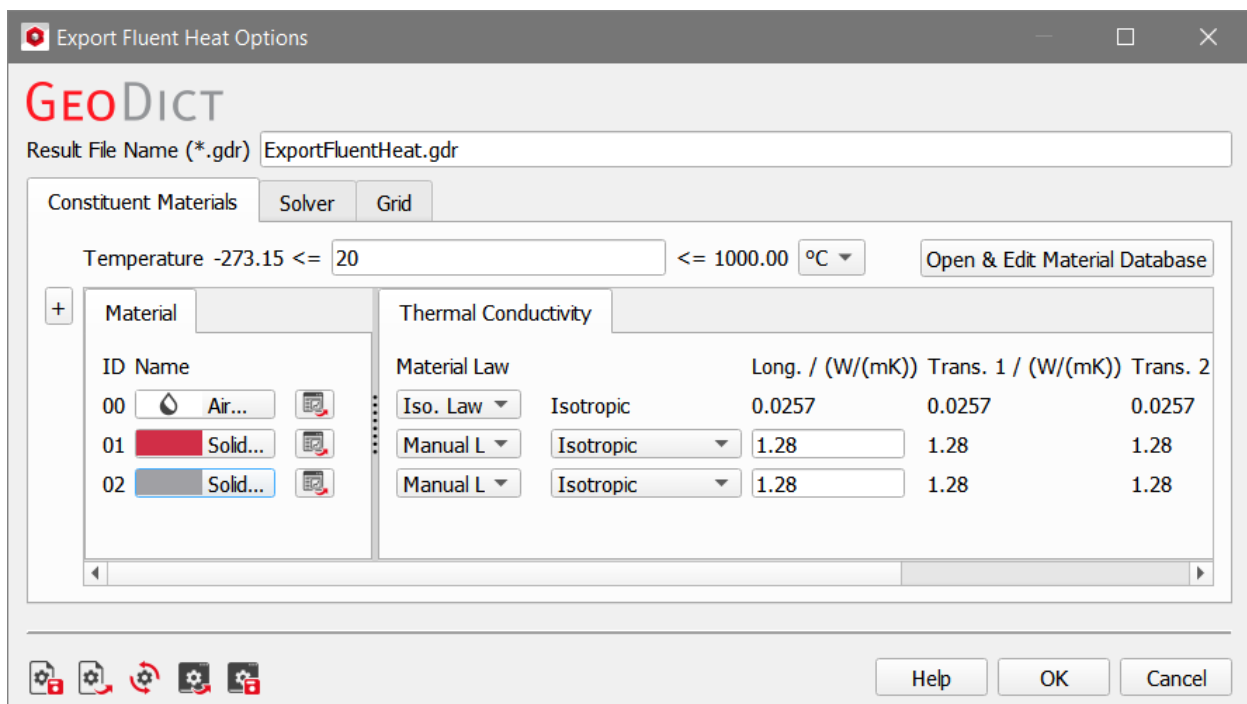


In the **Export Fluent Heat Options** dialog, choose a **Result File Name (*.gdr)** according to your current project. The result file contains all information about the current export. The corresponding result folder contains the mesh and journal files for Fluent and the chosen structure.

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

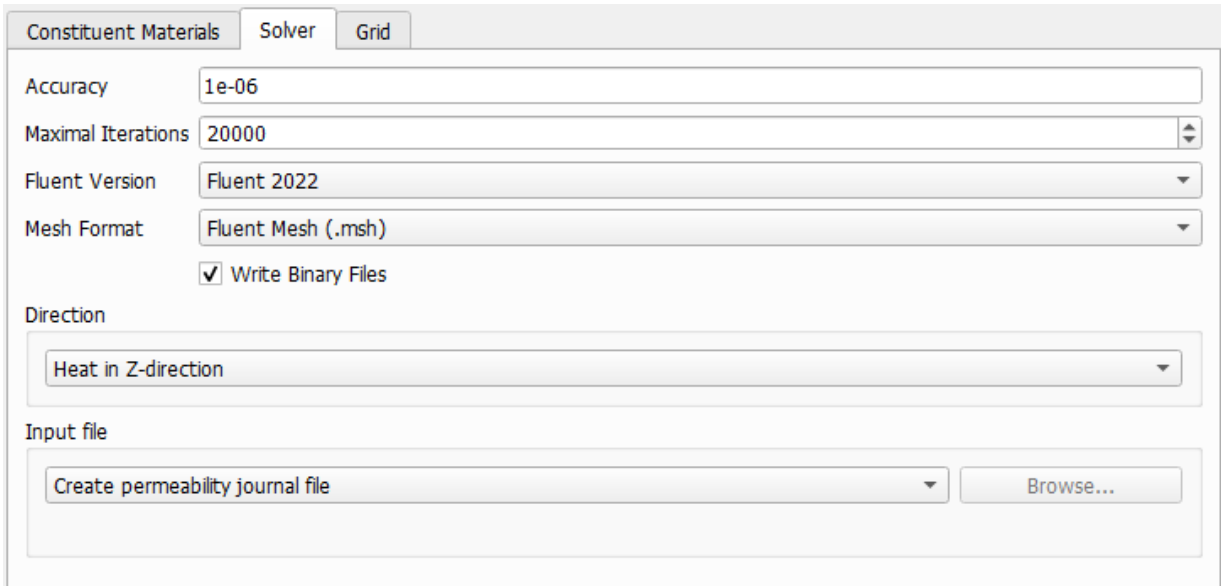
CONSTITUENT MATERIALS

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 already explained for [Export Flow Simulation \(Voxel\)](#) on pages 5ff. In contrast to flow simulations, Fluent V. 6.2 is not supported for heat simulations.



The screenshot shows the 'Solver' tab in a software interface. It contains the following settings:

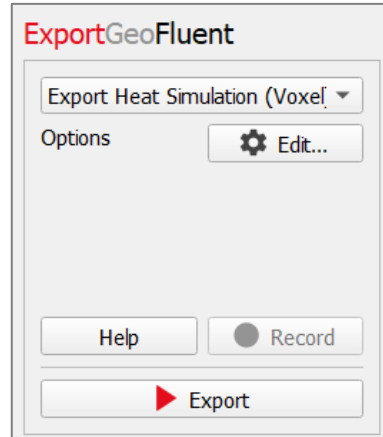
- Accuracy: 1e-06
- Maximal Iterations: 20000
- Fluent Version: Fluent 2022
- Mesh Format: Fluent Mesh (.msh)
- Write Binary Files
- Direction: Heat in Z-direction
- Input file: Create permeability journal file (with a 'Browse...' button)

GRID

The options under the **Grid** tab are the same as for [Export Flow Simulation \(Voxel\)](#) (see page 7).

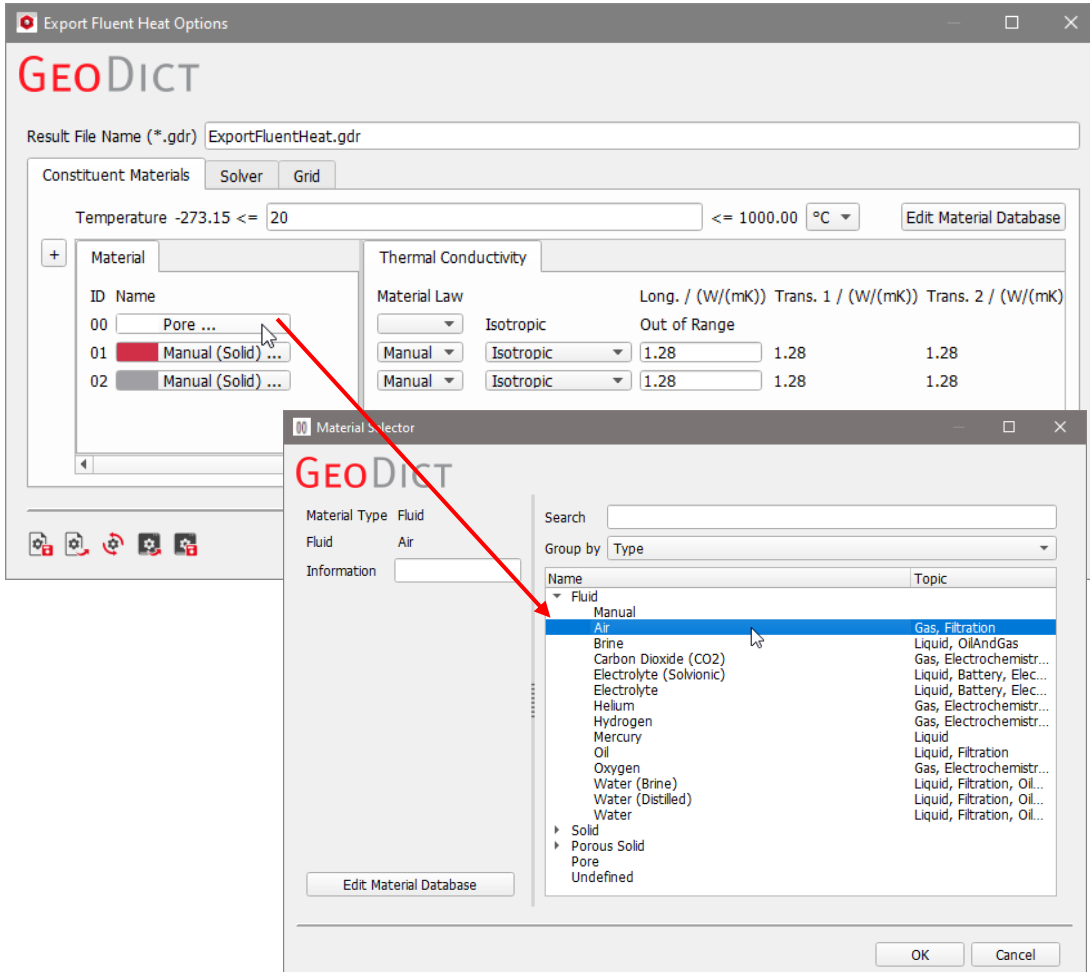
EXAMPLE: EXPORT TO FLUENT FOR HEAT COMPUTATIONS

Start **ExportGeo-Fluent** by selecting **Export** → **ExportGeoFluent** in the menu bar. Select **Export Heat Simulation (Voxel)** from the pull-down menu in the **ExportGeoFluent** section. In this example, a weave structure with two materials is exported to Fluent to run heat computations.

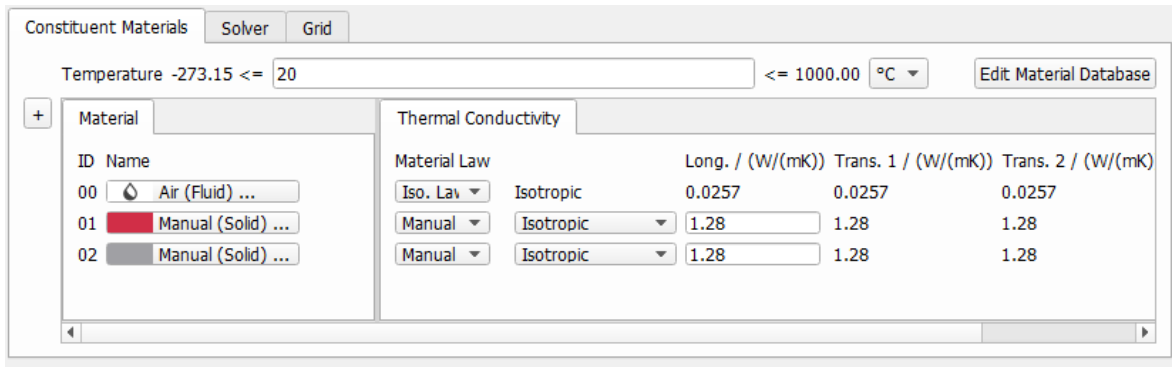


Click the **Solver Options' Edit...** button to open the **Fluent Solver Options** dialog. Then, enter the parameters for the Fluent solver.

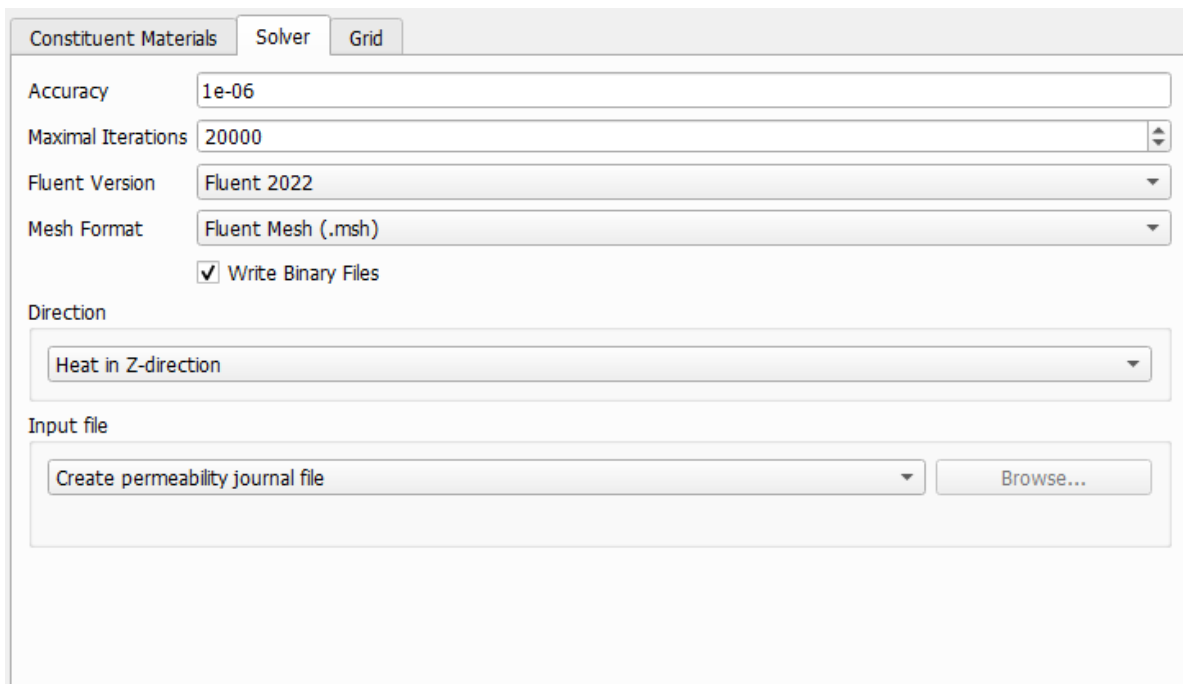
Under the **Constituent Materials** tab, set the Material ID of the material occupying the pore space by clicking the **Pore...** button and selecting **Pore (Fluid)** → **Air** from the list in the Material Selector. Then, the conductivity for air from the **GeoDict** material database is used.



Enter the conductivity for Material ID 01 and Material ID 02 manually as shown below.



In the **Solver** tab, keep the default values:

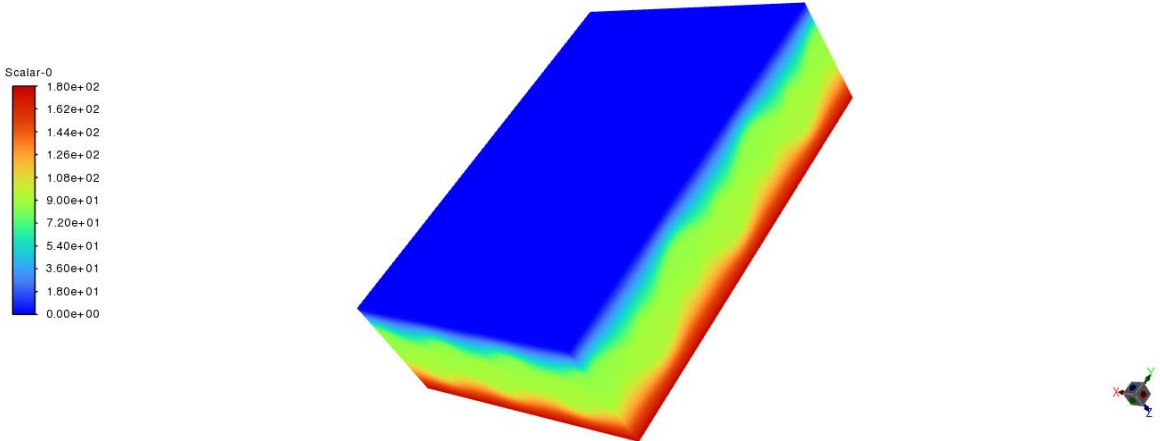


Also keep the default value (**Voxels with side length ...**) under the **Grid** tab. With this option, the voxel structure from **GeoDict** is used as it is. Click **OK** to close the **Export Fluent Heat Options** dialog and return to the **ExportGeoFluent** section. Click **Export** to export the 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....**

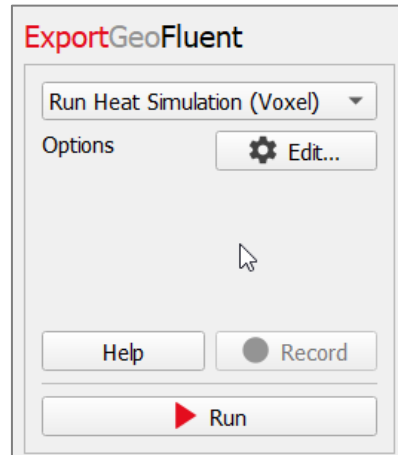
After the simulation, the computation results can be visualized in **Fluent**. For example, the temperature distribution in the structure is shown in the figure below.



Contours of Scalar-0 Sep 06, 2022
ANSYS Fluent 2022 R1 (3d, dp, pbns, sstk)

RUN HEAT SIMULATION (VOXEL)

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



The available options are explained under [Export heat simulation \(voxel\)](#), on pages [11ff.](#)

The same as **Run Flow Simulation (Voxel)**, **Run Heat Simulation (Voxel)** also requires that Fluent is installed on the same machine, and GeoDict should be able to find the Fluent executable that is described in the Section [How to make GeoDict find the Fluent executable](#) on page [Error! Bookmark not defined.](#)

Technical
documentation:

Liping Cheng
Jürgen Becker
Barbara Planas

MATH
2 MARKET

Math2Market GmbH

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