ExportGeo-Abaqus

User Guide

GeoDict release 2022

Published: September 16, 2021

GeoDict

https://doi.org/10.30423/userguide.geodict2022-exportgeoabaqus

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 four submodules:

- ExportGeo–Base converts voxelized data to .raw, .vol, and .png formats. Furthermore, it converts analytic data, particles and streamlines to .gad, and voxelized data to Avizo Binary Files (.am)
- ExportGeo-CAD converts voxelized or analytic structures to popular surface triangulation and CAD formats like STL, VRML, or Parasolid.
- ExportGeo-Fluent converts voxelized data to formats for flow and heat computations with **Fluent™**. The .jou files are saved as ASCII files which can be opened and edited with a text editor.
- ExportGeo-Abaqus convert voxelized data to formats for elasticity computations with **Abaqus**. The .inp files are saved as ASCII files which can be 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™**.

ExportGeo-Abaqus

With ExportGeo-Abaqus, GeoDict structures are exported to Abaqus. Access ExportGeo-Abaqus by selecting **Export → ExportGeo-Abaqus** in the menu bar.

With **Export Stiffness Simulation (Voxel**), the current voxel structure is exported and the parameters for simulations to compute the stiffness tensor are automatically set up for Abaqus. For this, the supported Abaqus versions are Abaqus 6.9 or newer.

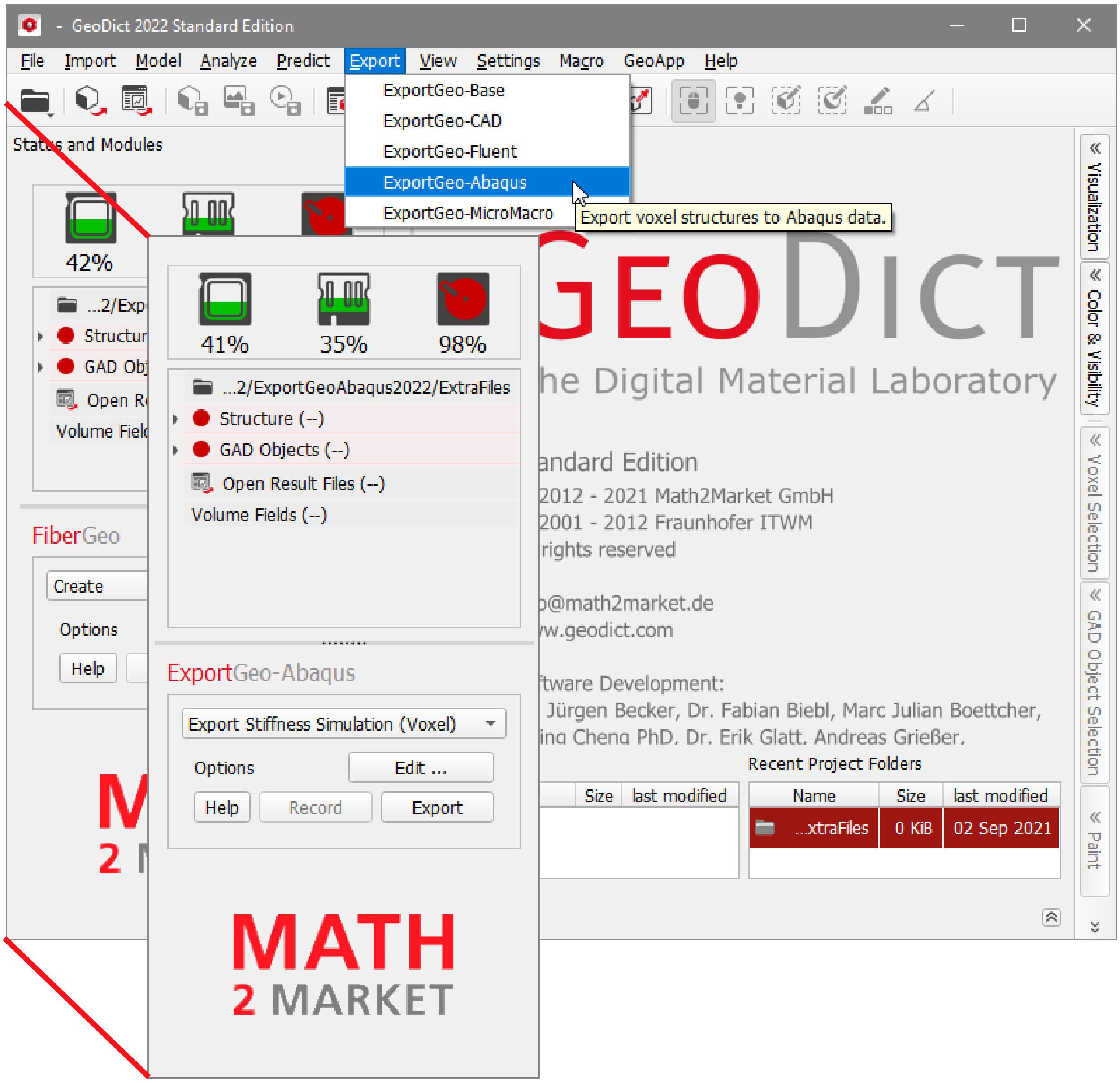
With **Export Stiffness Simulation (Voxel)** and **Run Stiffness Simulation (Voxel)**, the current voxel structure is exported for Abaqus. Additionally, all necessary simulations for computing the effective stiffness tensor are defined. This includes simulations for 6 different load cases.

Abaqus elasticity computations are accurate but require high amounts of memory and are generally highly time-consuming.

The same accurate results can be achieved with the ElastoDict module directly in GeoDict. See to set calculations of Effective Stiffness (FeelMath-VOX) in the ElastoDict 2022 handbook of this User Guide.

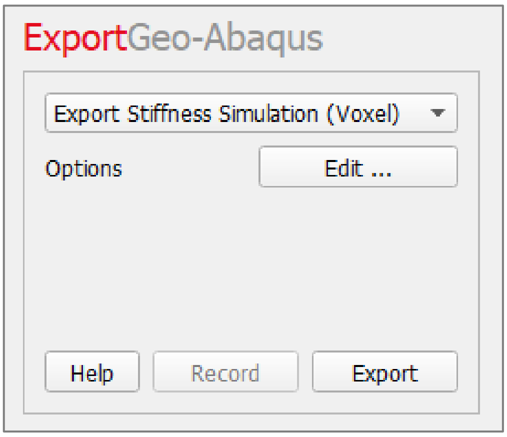
In general, this is the recommended way to proceed if ElastoDict is present in the GeoDict license. Since the FeelMath solver is optimized for voxel structures, the computations are faster and more memory efficient and, therefore, ElastoDict can handle much larger data sets than Abaqus.

With **Export Fibers as Beam elements**, fibers from a GeoDict structure (e.g. created with FiberGeo) are exported as beam elements for Abaqus simulations.



Export Stiffness Simulation (Voxel)

After starting ExportGeo-Abaqus by selecting **Export → ExportGeo-Abaqus** in the menu bar, click the Options **Edit...** button to define the settings for the export to Abaqus.

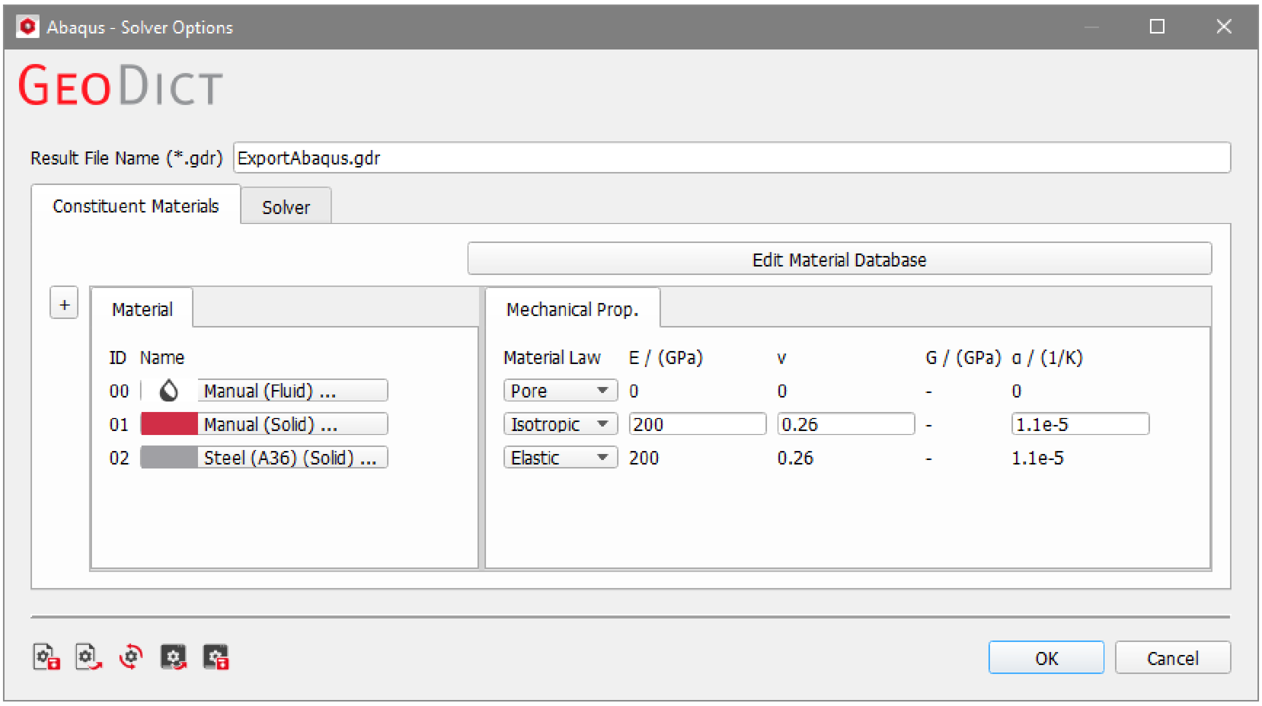


After entering all parameters, click **Export** to export the simulation for Abaqus.

Abaqus – Solver Options

In the **Abaqus – Solver 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 input files for Abaqus and the chosen structure.

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

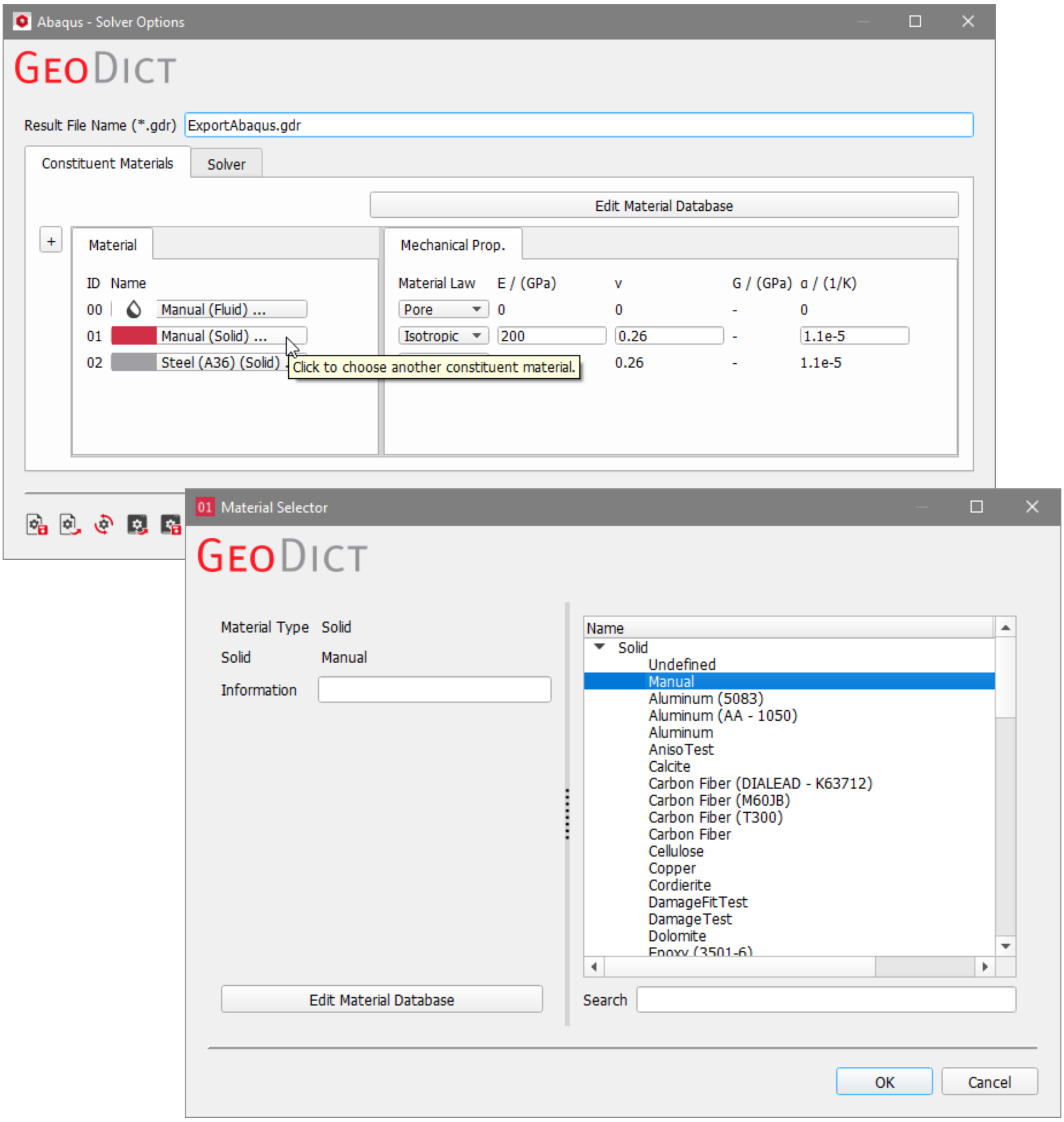


Constituent Materials

The mechanical properties of all materials in the structure are defined in the corresponding fields under the **Constituent Materials** tab. Since the goal is the computation of the effective elastic properties of the structure, only linear elastic materials are supported here. Materials can either be chosen from the GeoDict Material Database, or manual materials can be directly defined in the dialog.

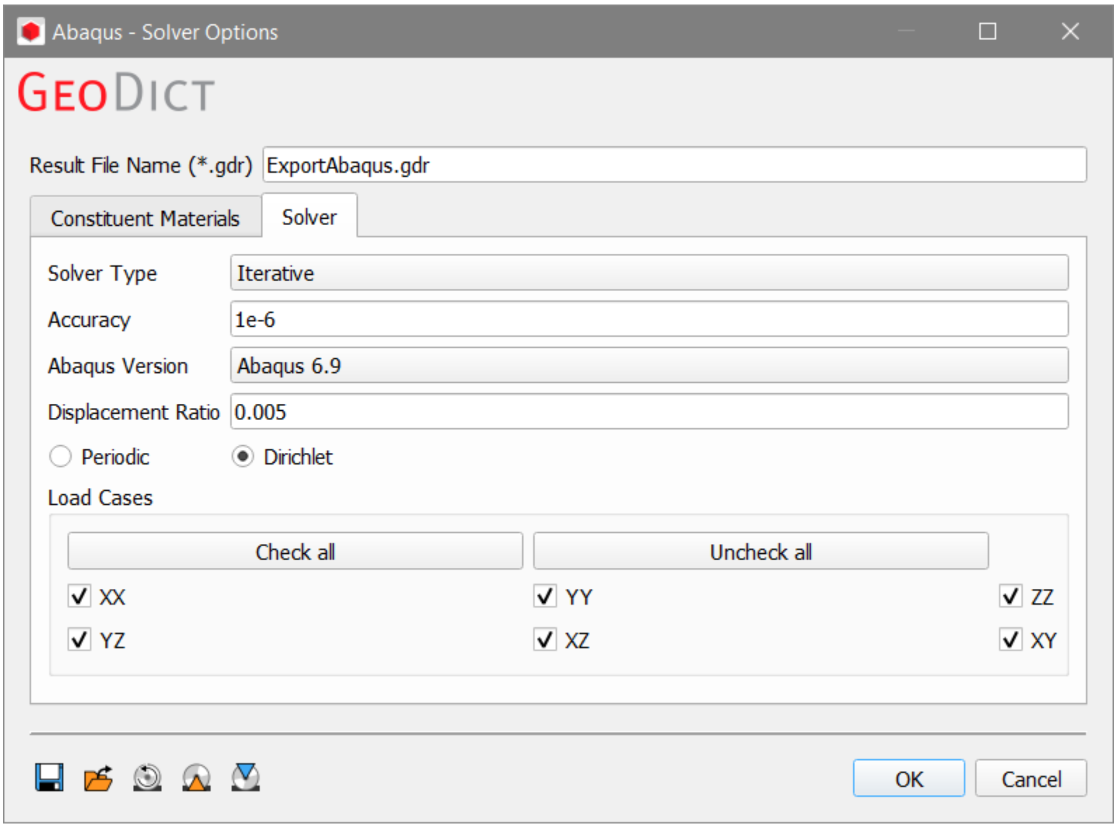
The GeoDict Material Database can be edited after selecting **Settings → Edit Material Data Base…** in the menu bar. More information is available from the Material Database 2022 handbook of this User Guide.

To define a **Manual** material, click on the button of the material to open the **Material Selector** and choose **Solid → Manual**. In the next step, enter the Young's modulus and the Poisson ratio under the **Constituent Materials** tab.



Solver

The settings for the Abaqus simulation are defined under the **Solver** tab.



Select the **Solver Type** for the simulation in Abaqus with the corresponding dropdown menu. The available options are **Direct** or **Iterative**.

If the **Iterative** solver is selected, choose the corresponding solver **Accuracy** here.

ExportGeo-Abaqus supports Abaqus version 6.9 and higher. If you use an Abaqus version between 6.9 and 6.11, select **Abaqus 6.9**. If you use Abaqus version 6.11 or newer, choose **Abaqus 6.11**.

The stiffness is calculated based on tensile experiments in six different load directions. The **Displacement Ratio** is the applied strain in these experiments. The default value of 0.005 corresponds to 0.5 % strain.

In **ExportGeo-CAD**, either **Periodic** or **Dirichlet** boundary conditions can be applied to the load direction. **Dirichlet** boundary conditions correspond to the **Symmetric** boundary condition in **ElastoDict**.

Select the **Load Cases** which are simulated in Abaqus. For every direction, a different experiment is computed:

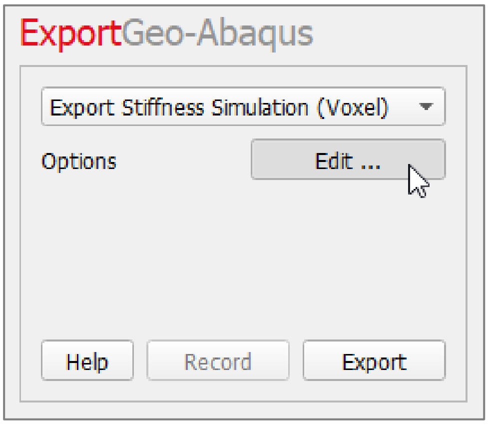
- **XX**, **YY**, **ZZ**: tension in X-, Y- and Z-directions
- **YZ**, **XZ**, **XY**: shear in YZ-, XZ and XY-planes.

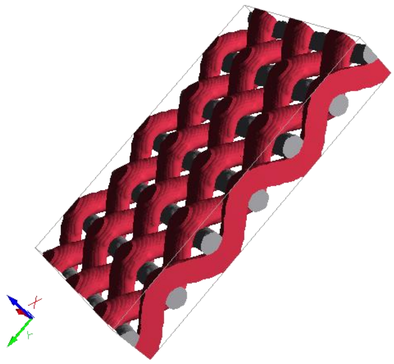
By default, all six load directions are selected since they are necessary for computing the stiffness tensor. Therefore, it is recommended to keep the default setting. In the tangential directions, a zero-strain boundary condition is applied.

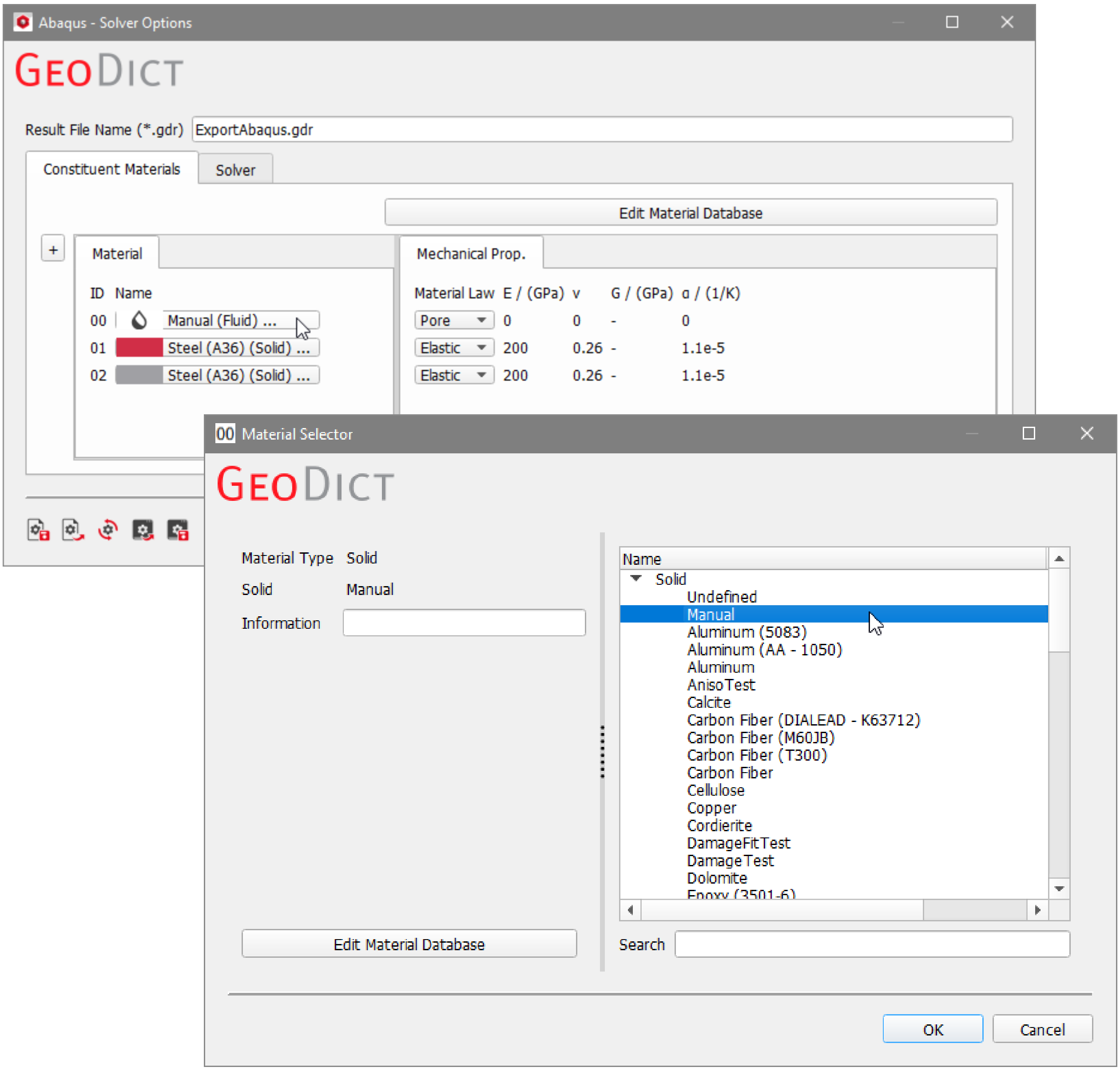
Example: Export to Abaqus for elasticity computations

The data of a weave structure with two materials is exported to Abaqus to run elasticity computations.

Start ExportGeo-Abaqus by selecting **Export** → **ExportGeo-Abaqus** in the menu bar and click the **Options**' **Edit...** button to open the **Abaqus - Solver Options** dialog and enter the settings to be used by the **Abaqus** solver.

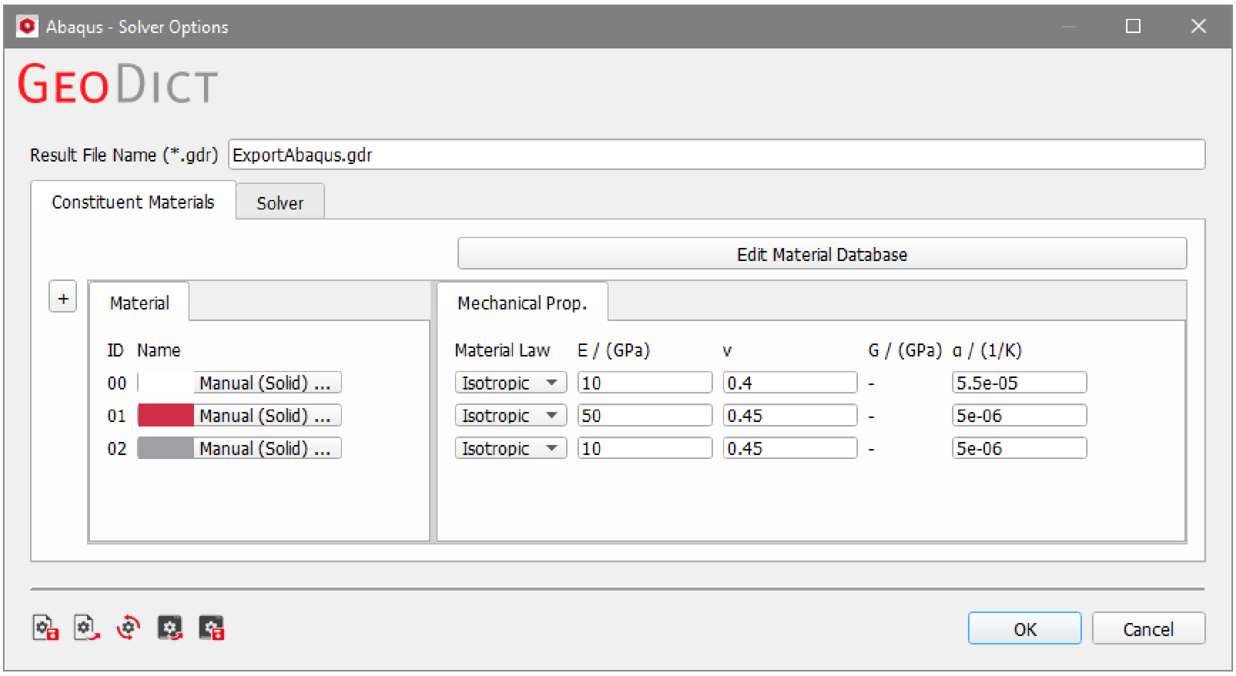






Under the **Constituent Materials** tab, set the constituent materials for Material ID 00, Material ID 01, and Material ID 02 to **Solid** → **Manual** through the Material Selector.

Then, enter the shown values of Young's modulus (E) and Poisson ratio (υ) for the materials with the IDs 00, 01, and 02.

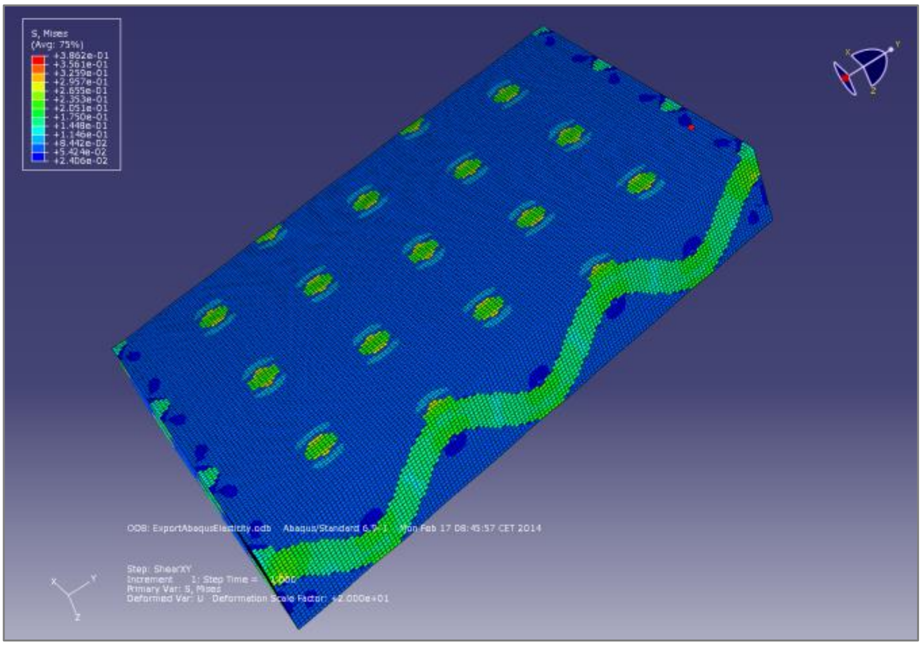


Under the **Solver** tab, use the default values for the **Iterative** solver and click **OK** to close the **Abaqus - Solver Options** dialog and return to the **ExportGeo-Abaqus** section.

Click **Export** to export the stiffness experiment for Abaqus.

Two files are generated in the selected location: the **Abaqus** input file **ExportAbaqus.inp** and the mesh file **ExportAbaqus.msh** with the structure. Both can be opened and edited using a text editor.

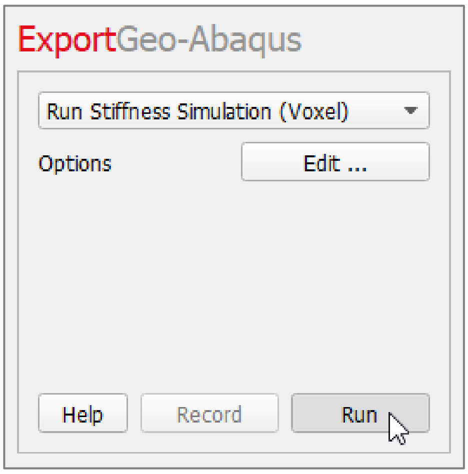
To run the elasticity computation in **Abaqus**, run the input file (ExportAbaqus.inp) in **Abaqus**.



Run Stiffness Simulation (Voxel)

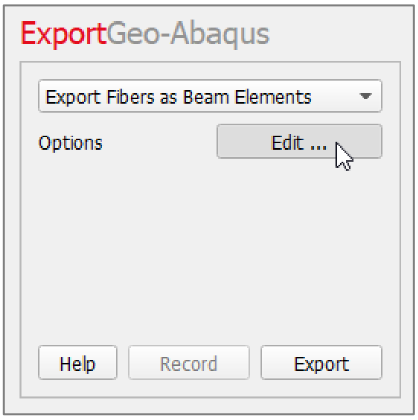
The option **Run Stiffness Simulation (Voxel)** is only available under Linux. It contains the same options as **Export Stiffness Simulation (Voxel)**, but it starts the simulation in Abaqus directly after clicking **Run**.

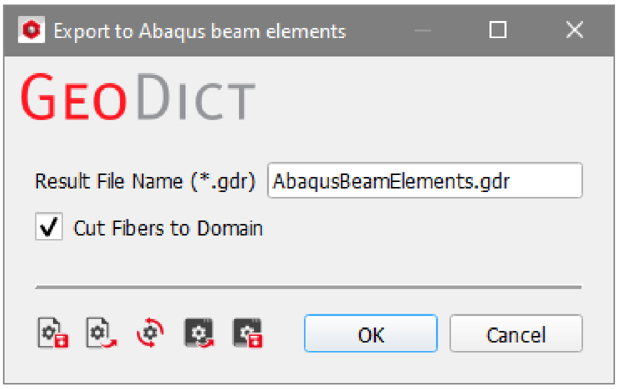
The available options are explained under Export Stiffness Simulation (Voxel) in pages 3ff.



Export Fibers as Beam Elements

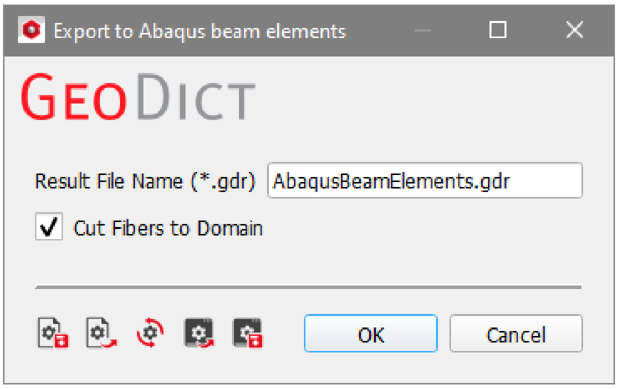
With **Export Fibers as Beam Elements**, fiber objects from GeoDict can be exported to Abaqus. Click the Options **Edit...** button to define the settings for the export to Abaqus. After entering all settings, click **Export** to export the simulation for Abaqus.





In the **Export to Abaqus Beam Elements** 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 input files for Abaqus and the chosen structure.

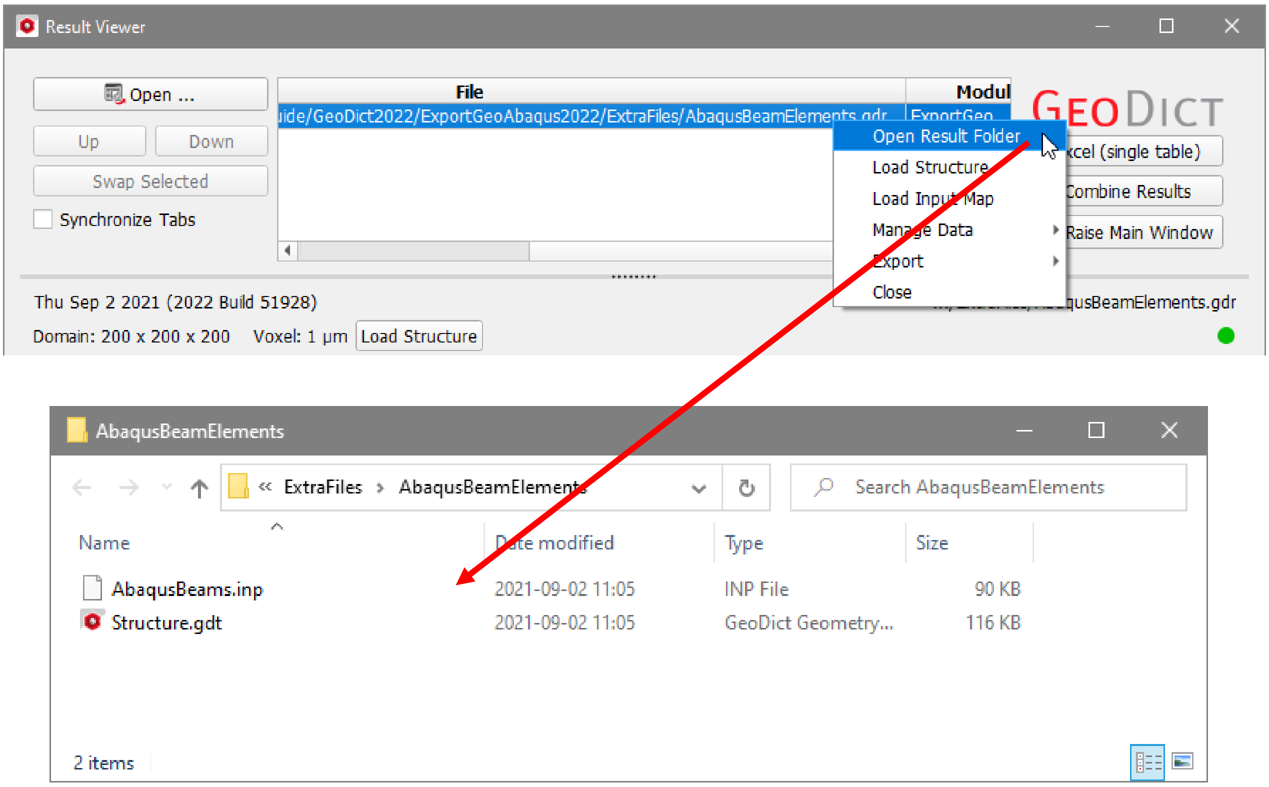
To export fibers to Abaqus beam elements, only the setting **Cut Fibers to Domain** is available. Checking this option limits the fibers to the domain.



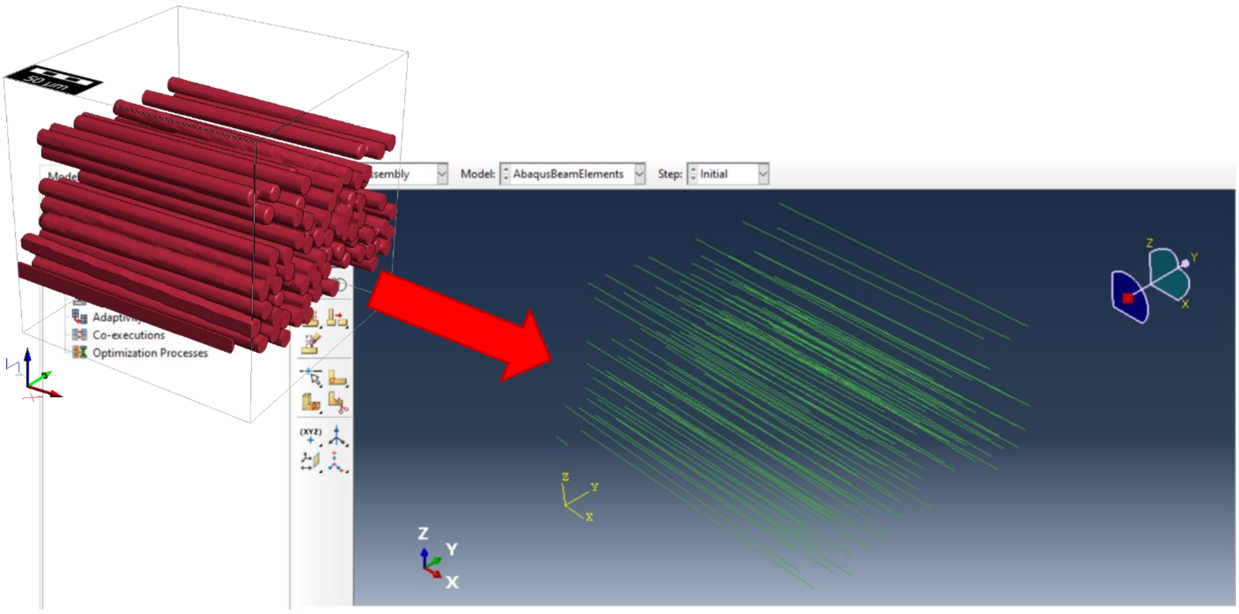
In GeoDict, fiber objects can be larger than the domain. This has no effect on GeoDict simulations, since all GeoDict simulations are limited to the domain. However, fibers which leave the domain may lead to unexpected results in other simulation software, such as Abaqus. Therefore, it is recommended to keep **Cut Fibers to Domain** enabled.

Results

After exporting the fibers, the result file (*.gdr) is automatically opened in the result viewer. The *.inp file for Abaqus is saved in the results folder with the same name as the result file. The results folder can be accessed by right-clicking on the name of the result file and selecting **Open Result Folder** on the results file name.



The *.inp file for Abaqus (with the same name as the result file) can then be imported into Abaqus for further processing, as seen below in the example for a simple fiber structure.



https://doi.org/10.30423/userguide.geodict2022-exportgeoabaqus

Technical documentation:

Sebastian Rief

Jürgen Becker

Barbara Planas

Math2Market GmbH

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