ExportGeo-Abaqus

User Guide

GeoDict release 2024

Published: February 14, 2024

GeoDict

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

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

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

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:

- **Export as CAD (MeshGeo)** converts voxelized or analytic structures to popular surface triangulation and CAD formats like STL, VRML, or Parasolid.
- **Export 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)** convert 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 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**.

ExportGeoAbaqus

With ExportGeoAbaqus, GeoDict structures are exported to Abaqus.

Access ExportGeoAbaqus by selecting **Export → Export to Abaqus (ExportGeoAbaqus)** in the menu bar.

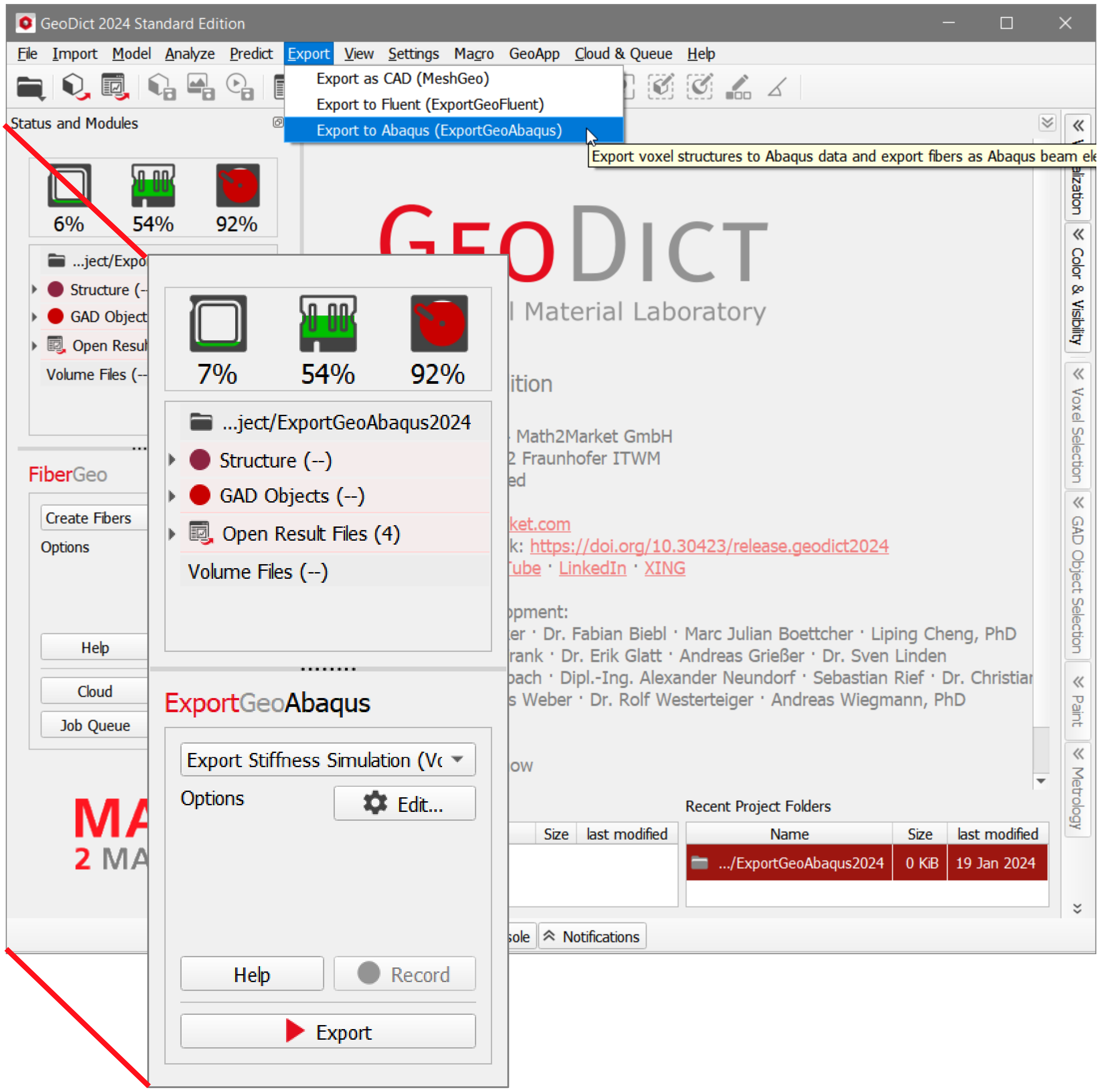
With **Export Stiffness Simulation (Voxel**), the current voxel structure is exported and the parameters for simulations to compute the stiffness tensor are automatically set up for Abaqus. For this, the supported Abaqus versions are Abaqus 6.9 or newer. Additionally, all necessary simulations for computing 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. Find the details about how to set calculations of Effective Stiffness (FeelMath-VOX) in the ElastoDict 2024 handbook of this User Guide.

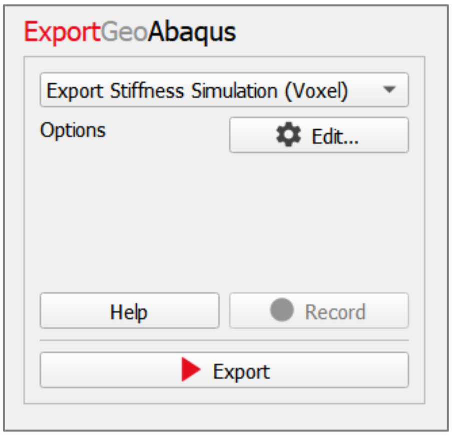
In general, this is the recommended way to proceed if ElastoDict is present in the GeoDict license. Since the FeelMath solver is optimized for voxel structures, the computations are faster and more memory efficient and, therefore, ElastoDict can handle much larger data sets than Abaqus.

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


GeoDict 2024 Standard Edition
File Import Model Analyze Predict Export View Settings Macro GeoApp Cloud & Queue Help
Export as CAD (MeshGeo)
Export to Fluent (ExportGeoFluent)
Export to Abaqus (ExportGeoAbaqus)
Export voxel structures to Abaqus data and export fibers as Abaqus beam ele
Status and Modules
7%
54%
92%
...ject/ExportGeoAbaqus2024
Structure (--)
GAD Objects (--)
Open Result Files (4)
Volume Files (--)
ExportGeoAbaqus
Export Stiffness Simulation (Vo
Options
Edit...
Help
Record
Export
FiberGeo
Create Fibers
Options
Cloud
Job Queue
Material Laboratory
Math2Market GmbH
Fraunhofer ITWM
https://doi.org/10.30423/release.geodict2024
LinkedIn · XING
Recent Project Folders
Name
Size
last modified
.../ExportGeoAbaqus2024
0 KiB
19 Jan 2024
Notifications

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.

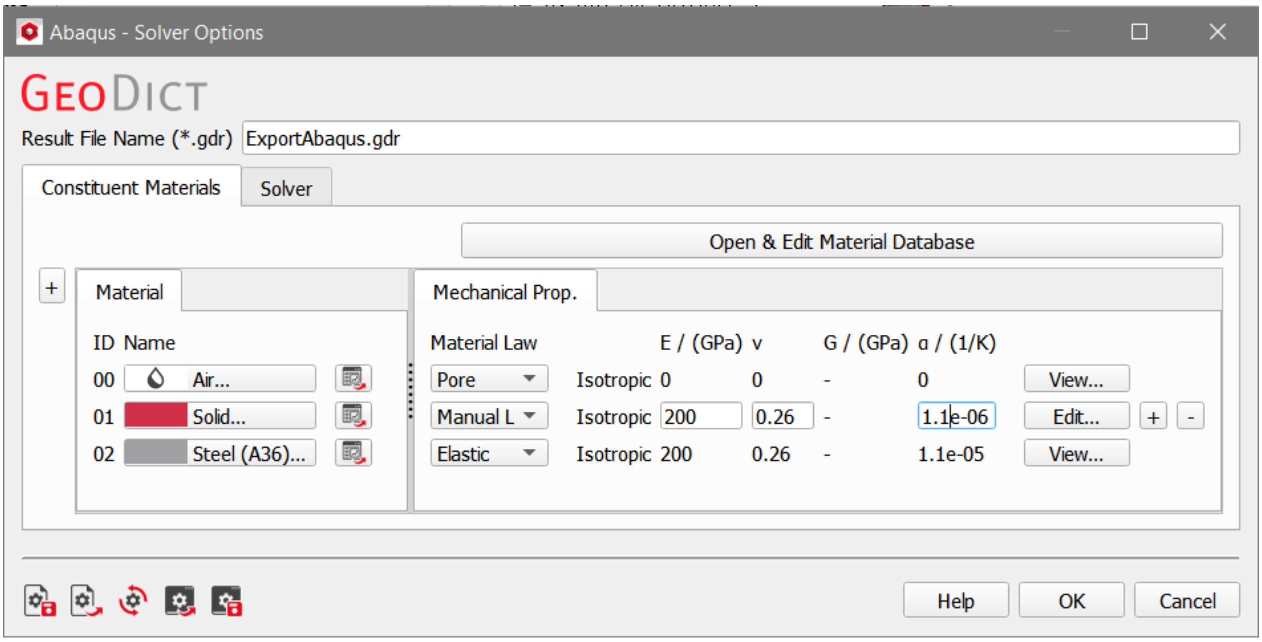


After entering all parameters, click **Export** to export the simulation for Abaqus.

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 the 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.

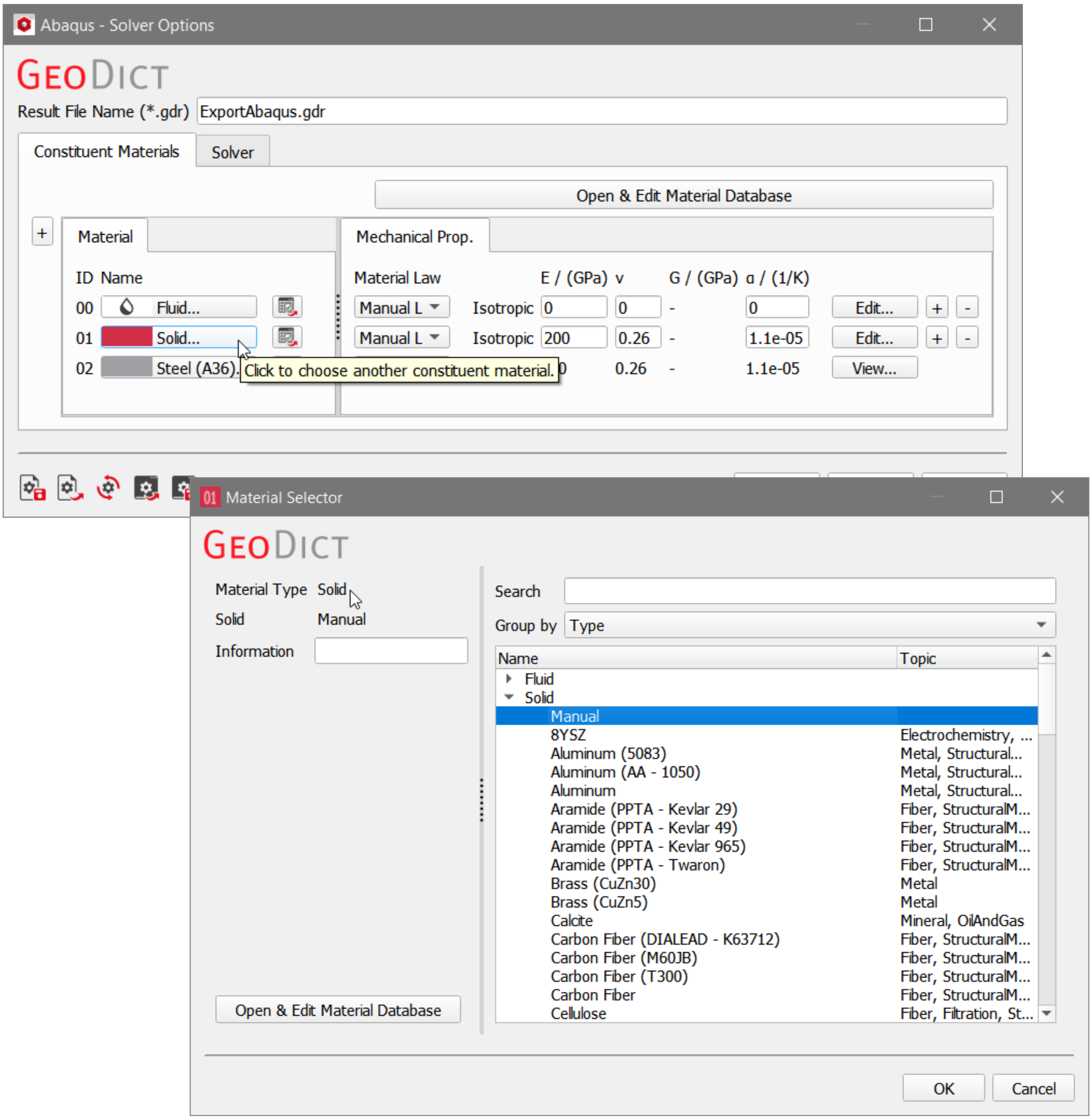


CONSTITUENT MATERIALS

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.

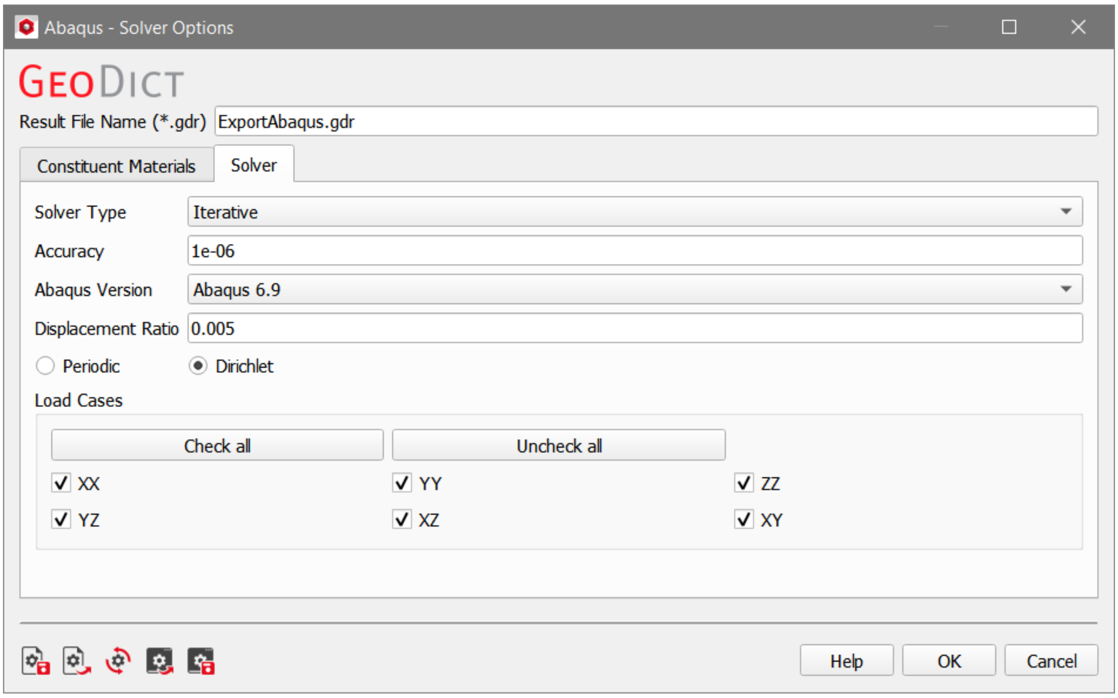
The GeoDict Material Database can be edited after selecting **Settings → Open & Edit Material Data Base…** in the menu bar. More information is available from the Material Database handbook of this User Guide.

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.

ExportGeoAbaqus 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.

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:

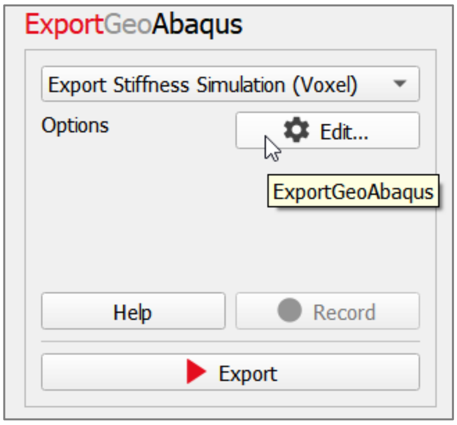
- **XX**, **YY**, **ZZ**: tension in X-, Y- and Z-directions
- **YZ**, **XZ**, **XY**: shear in YZ-, XZ and XY-planes.

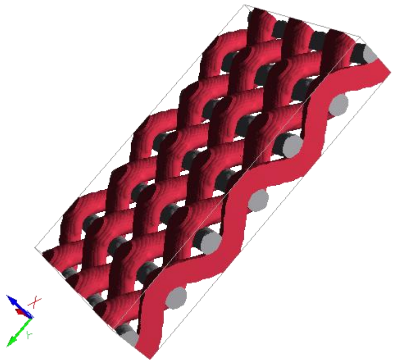
By default, all six load directions are selected since they are necessary for computing the stiffness tensor. Therefore, it is recommended to keep the default setting. In the tangential directions, a zero-strain boundary condition is applied.

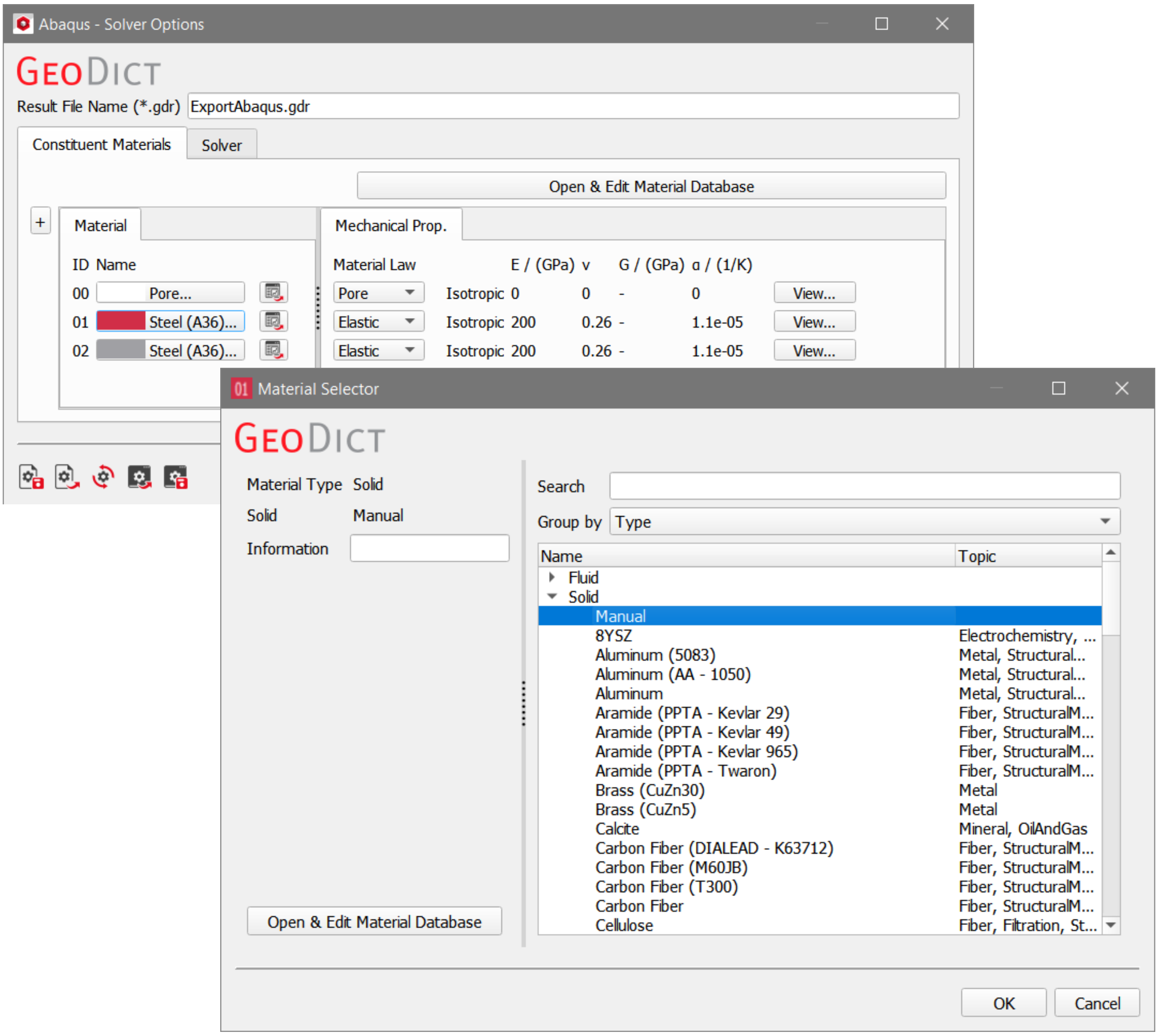
Example: Export to Abaqus for elasticity computations

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

Start ExportGeoAbaqus by selecting **Export → Export to Abaqus (ExportGeoAbaqus)** 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.

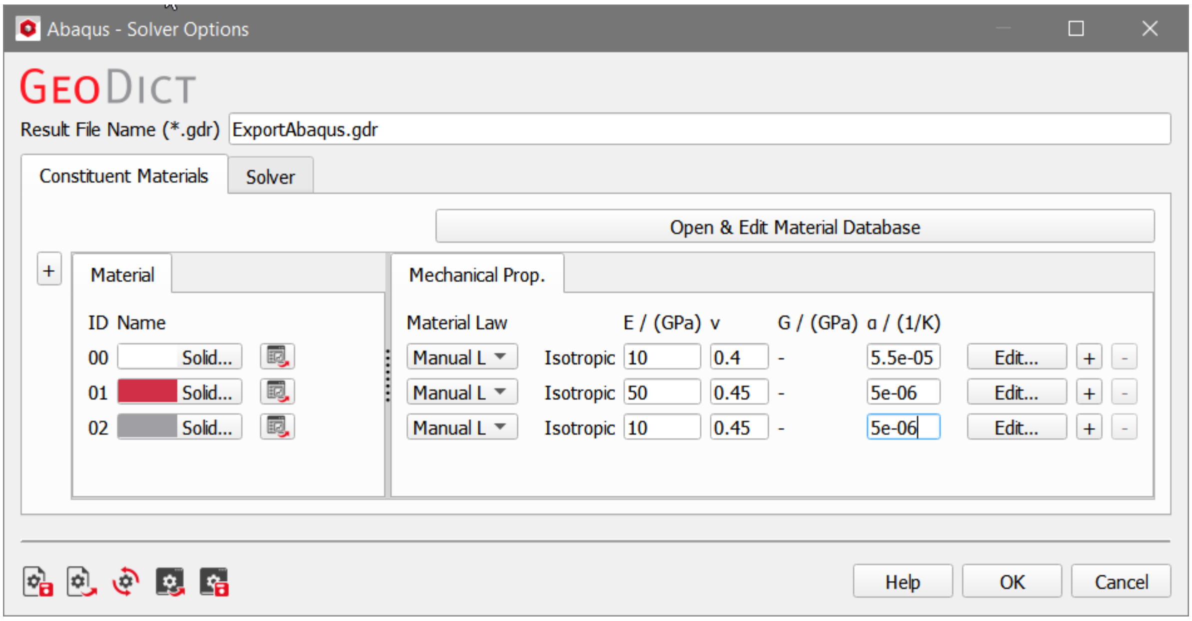






Under the **Constituent Materials** tab, set the constituent materials for Material ID 00, Material ID 01, and Material ID 02 to **Solid** → **Manual** through the Material Selector.

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

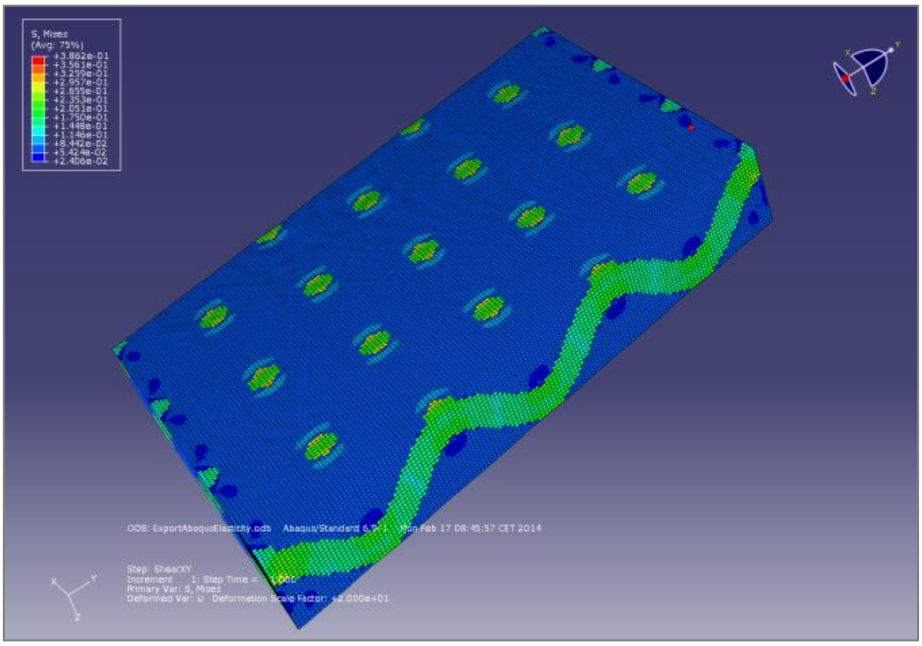


Under the **Solver** tab, use the default values for the **Iterative** solver and click **OK** to close the **Abaqus - Solver Options** dialog and return to the ExportGeo**Abaqus** section.

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

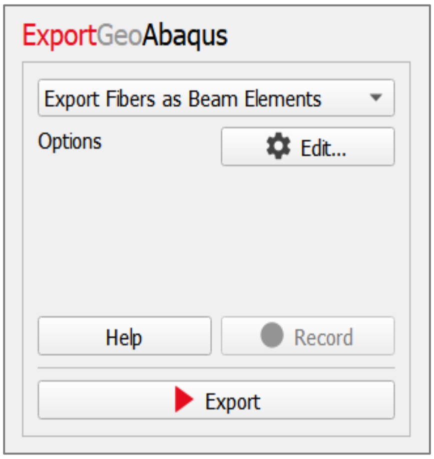
Besides the structure file, two more files are generated in the result folder: the **Abaqus** input file **Abaqus.inp** and the mesh file **Abaqus.msh**. Both can be opened and edited using a text editor.

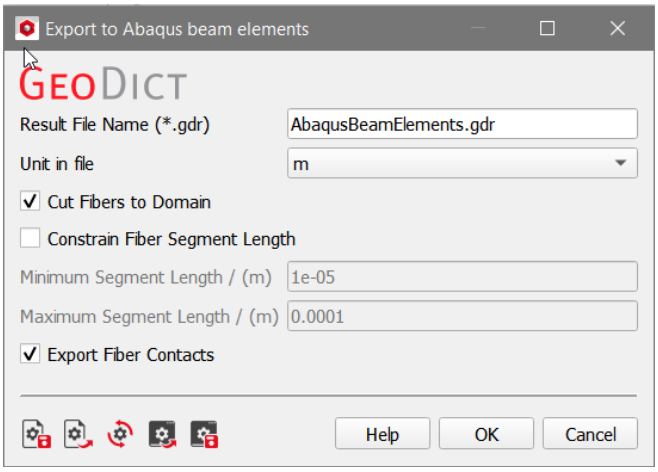
To run the elasticity computation in **Abaqus**, run the input file (Abaqus.inp) in **Abaqus**.



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.



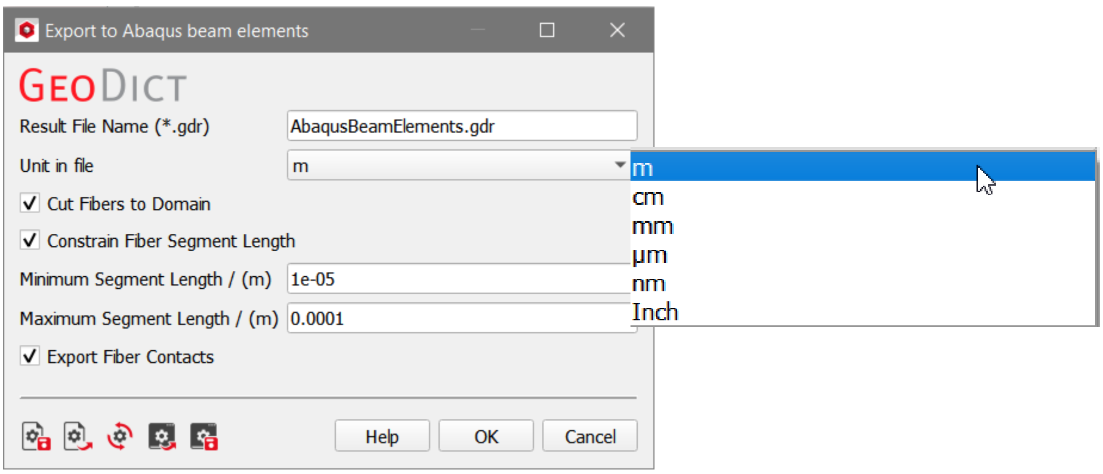


In the **Export to Abaqus Beam Elements** dialog, choose a **Result File Name (*.gdr)** according to the current project. The result file contains all information about the current export. The corresponding result folder contains the input files for Abaqus and the chosen structure.

To export fibers to Abaqus beam elements, the setting **Cut Fibers to Domain** is to limit the fibers to the domain.

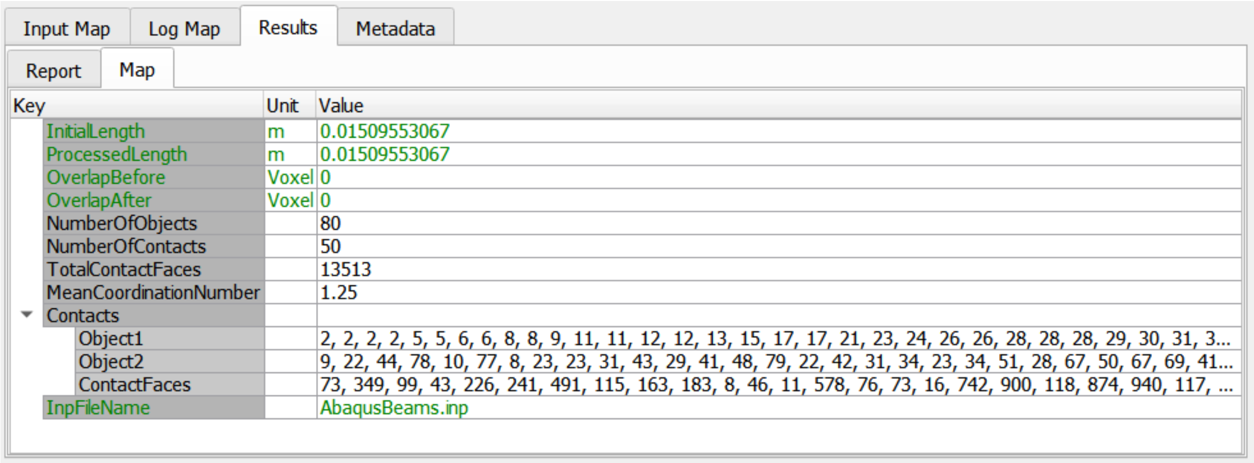
In GeoDict, fiber objects can be larger than the domain. This has no effect on GeoDict simulations, since all GeoDict simulations are limited to the domain. However, fibers which leave the domain may lead to unexpected results in other simulation software, such as Abaqus. Therefore, it is recommended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

Unit in file makes it possible to convert the object coordinates into the chosen unit when writing them into the .inp file. The available length units are **m**, **cm**, **mm**, **µm**, **nm**, and **inch**.



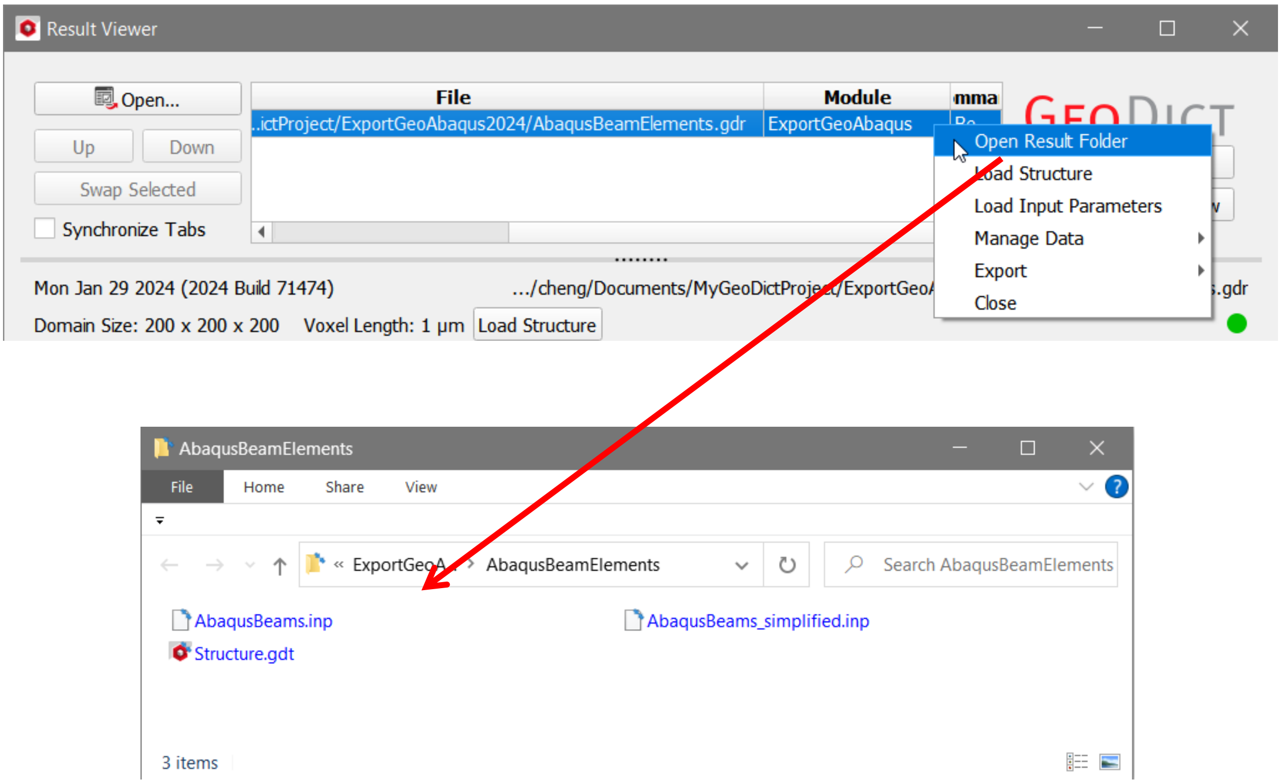
When it is necessary to constrain the maximum and minimum of the fiber segment length, check the option **Constrain Fiber Segment Length**, then the settings of **Minimum Segment Length** and **Maximum Segment Length** become editable. The fiber segment length will be constrained between the two values.

When **Export Fiber Contacts** is checked, the information of fiber contacts is included in the result .gdr file and is found in the ResultMap.

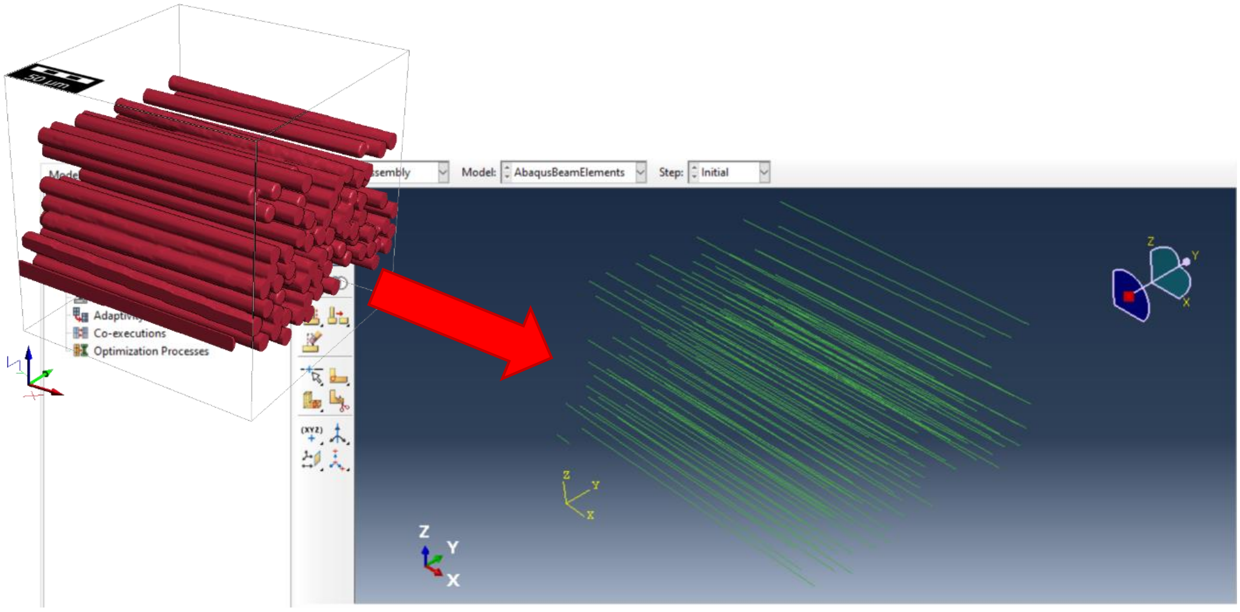


Results

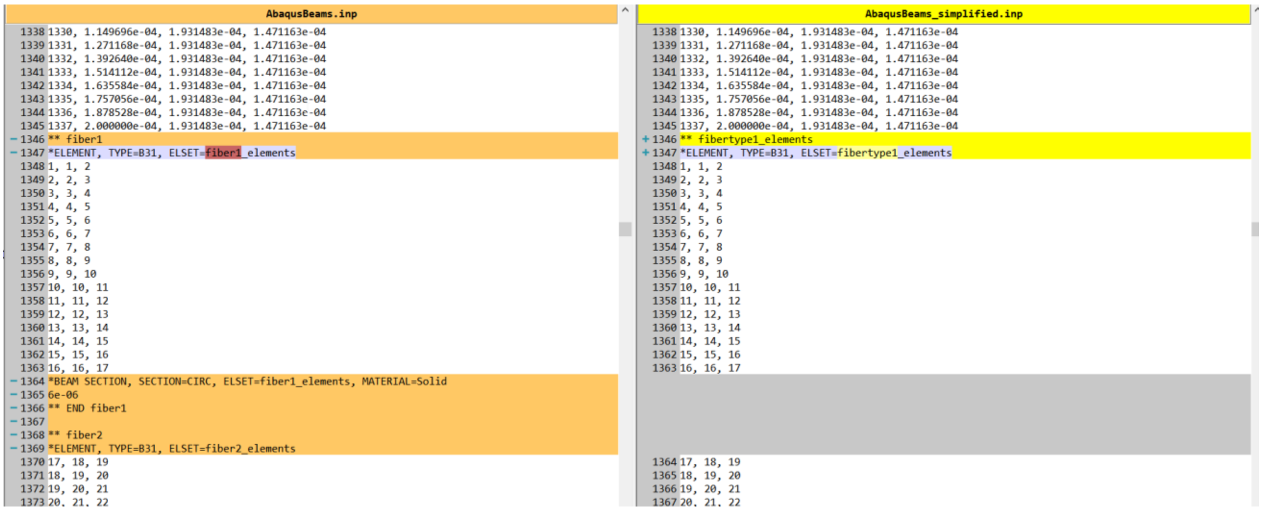
After exporting the fibers, the result file (*.gdr) is automatically opened in the result viewer. The *.inp file for Abaqus is saved in the results folder with the same name as the result file. The results folder can be accessed by right-clicking on the name of the result file and selecting **Open Result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



Both *.inp files, **AbaqusBeams.inp** and **AbaqusBeams_simplified.inp**, can be imported into Abaqus for further processing, as seen below in the example for a simple fiber structure.



The first .inp file has a cross section definition for each fiber. The simplified one shares the definition for all fibers that have the same cross section, which can make the import in Abaqus more robust. See below for the difference between the two .inp files.



AbaqusBeams.inp

```
1330, 1.149696e-04, 1.931483e-04, 1.471163e-04
1331, 1.271168e-04, 1.931483e-04, 1.471163e-04
1332, 1.392640e-04, 1.931483e-04, 1.471163e-04
1333, 1.514112e-04, 1.931483e-04, 1.471163e-04
1334, 1.635584e-04, 1.931483e-04, 1.471163e-04
1335, 1.757056e-04, 1.931483e-04, 1.471163e-04
1336, 1.878528e-04, 1.931483e-04, 1.471163e-04
1337, 2.000000e-04, 1.931483e-04, 1.471163e-04
** fiber1
*ELEMENT, TYPE=B31, ELSET=fiber1_elements
1, 1, 2
2, 2, 3
3, 3, 4
4, 4, 5
5, 5, 6
6, 6, 7
7, 7, 8
8, 8, 9
9, 9, 10
10, 10, 11
11, 11, 12
12, 12, 13
13, 13, 14
14, 14, 15
15, 15, 16
16, 16, 17
*BEAM SECTION, SECTION=CIRC, ELSET=fiber1_elements, MATERIAL=Solid
6e-06
** END fiber1

** fiber2
*ELEMENT, TYPE=B31, ELSET=fiber2_elements
17, 18, 19
18, 19, 20
19, 20, 21
20, 21, 22
```

AbaqusBeams_simplified.inp

```
1330, 1.149696e-04, 1.931483e-04, 1.471163e-04
1331, 1.271168e-04, 1.931483e-04, 1.471163e-04
1332, 1.392640e-04, 1.931483e-04, 1.471163e-04
1333, 1.514112e-04, 1.931483e-04, 1.471163e-04
1334, 1.635584e-04, 1.931483e-04, 1.471163e-04
1335, 1.757056e-04, 1.931483e-04, 1.471163e-04
1336, 1.878528e-04, 1.931483e-04, 1.471163e-04
1337, 2.000000e-04, 1.931483e-04, 1.471163e-04
** fibertype1_elements
*ELEMENT, TYPE=B31, ELSET=fibertype1_elements
1, 1, 2
2, 2, 3
3, 3, 4
4, 4, 5
5, 5, 6
6, 6, 7
7, 7, 8
8, 8, 9
9, 9, 10
10, 10, 11
11, 11, 12
12, 12, 13
13, 13, 14
14, 14, 15
15, 15, 16
16, 16, 17
17, 18, 19
18, 19, 20
19, 20, 21
20, 21, 22
```

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

Technical documentation:

Liping Cheng

Jürgen Becker

Barbara Planas

Math2Market GmbH

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