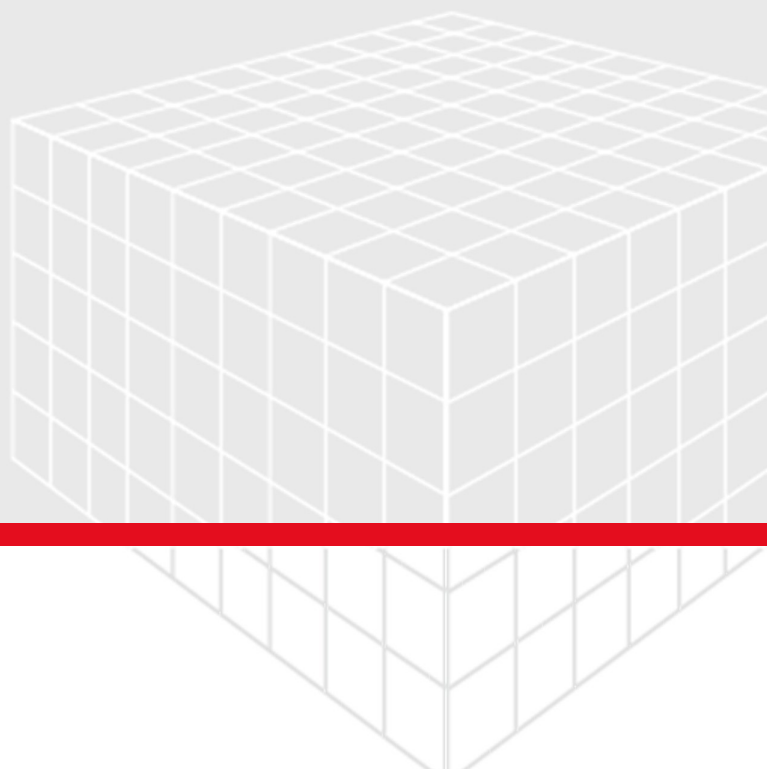


ExportGeo-Abaqus

User Guide

GEODICT Release 2026

Published: June 2026



GEODICT

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

ExportGeo-Abaqus

1. ExportGeo-Abaqus	3
1.1 Export Stiffness Simulation (Voxel)	4
1.1.1 Results	6
1.2 Export Fibers as Beam Elements	10
1.2.1 Results	11

1 ExportGeo-Abaqus

Export to Abaqus (**ExportGeo-Abaqus**) 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 opened and edited with a text editor.

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

Access ExportGeo-Abaqus by selecting **Export** → **ExportGeo-Abaqus** in the menu bar.

Learn to use ExportGeo-Abaqus

- [Export Stiffness Simulation \(Voxel\)](#)

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.

- [Export Fibers as Beam Elements](#)

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



Important! 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 User Guide.

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



Join the conversation

Visit the Community on [GeoDict Forum](#) to be inspired and get answers to top questions.

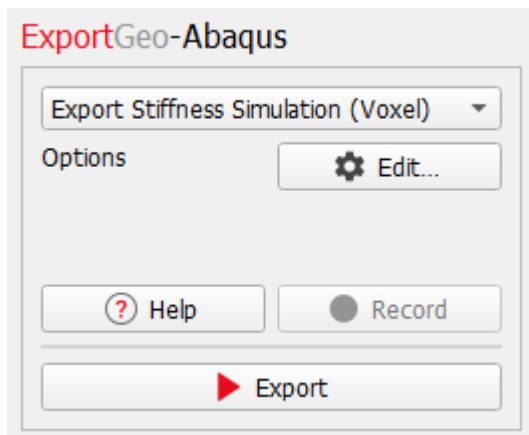
Find illustrative application examples on the [website](#).

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 Stiffness Simulation (Voxel)

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.

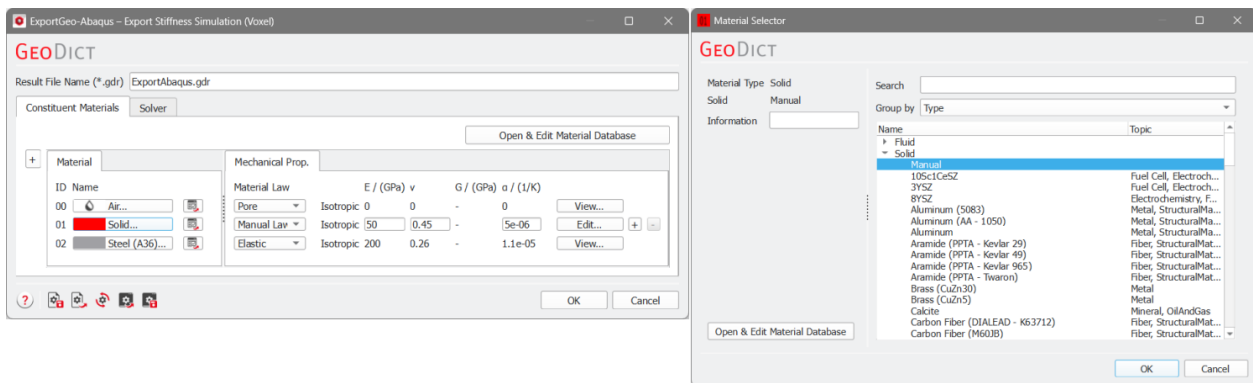
In the **ExportGeo-Abaqus – Export Stiffness Simulation (Voxel)** dialog, choose a **Result File Name (*.gdr)** according to your current project. The result file contains all the information about the current export. The corresponding result folder contains the input files for Abaqus and the chosen 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 clicking on **Open & Edit Material Database**. To select a material from the database, click on the button of the material to open the **Material Selector** and choose the desired material. The mechanical properties of the material are taken from the Material Database and shown in the **Mechanical Prop.** tab.

To define a **Manual** material, choose **Solid** → **Manual** in the **Material Selector**. In the next step, enter the Young's modulus and the Poisson ratio under the **Mechanical Prop.** tab.



Solver

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

✓ Solver Type

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

✓ Accuracy

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

✓ Abaqus Version

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**.

✓ Displacement Ratio

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.

✓ Periodic / Dirichlet

Either **Periodic** or **Dirichlet** boundary conditions can be applied to the load direction. **Dirichlet** boundary conditions correspond to the **Symmetric** boundary condition in **ElastoDict**.

✓ Load Cases

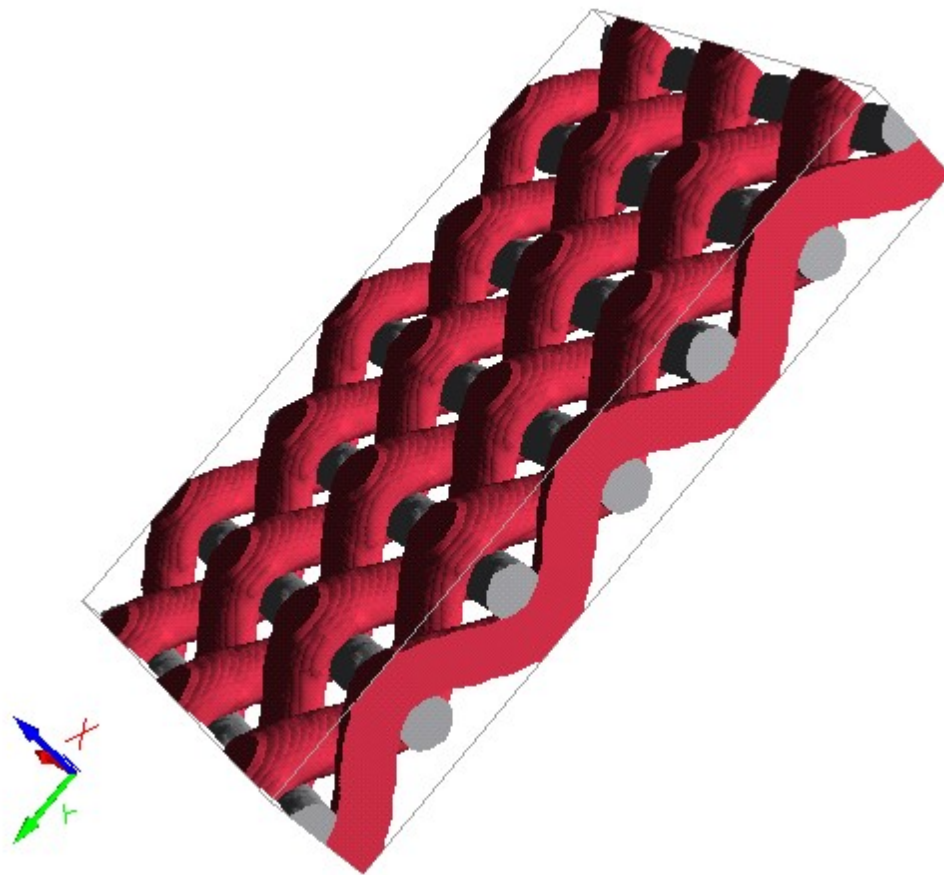
Select the **Load Cases** which are simulated in Abaqus. For every direction, a different experiment is computed:

- **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.

1.1.1 Results

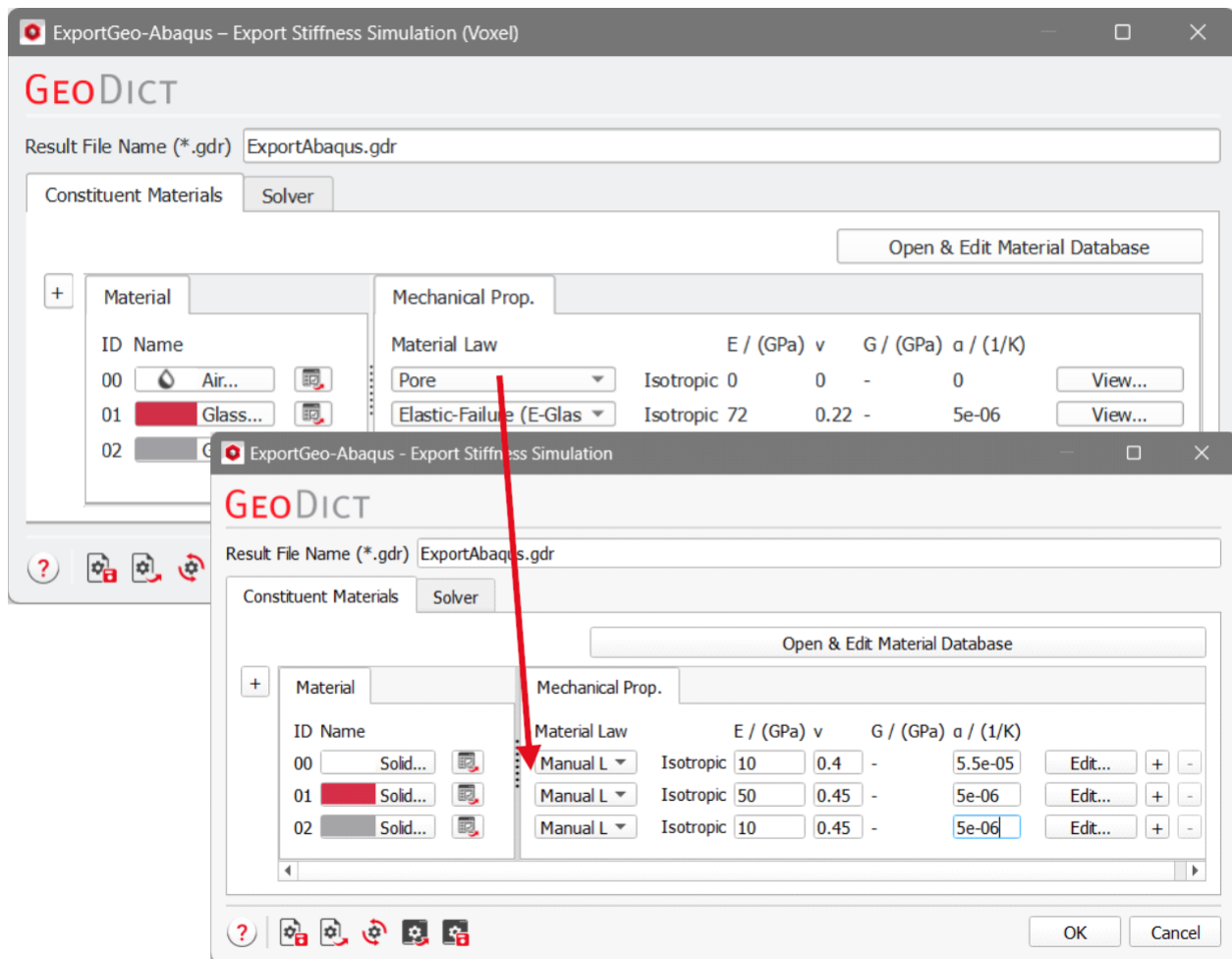
As an example we show how a weave structure with two materials is exported to Abaqus to run elasticity computations.



Under the **Constituent Materials** tab, set the constituent materials for Material ID00, Material ID01, and Material ID02 to **Solid** → **Manual** through the Material Selector.

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

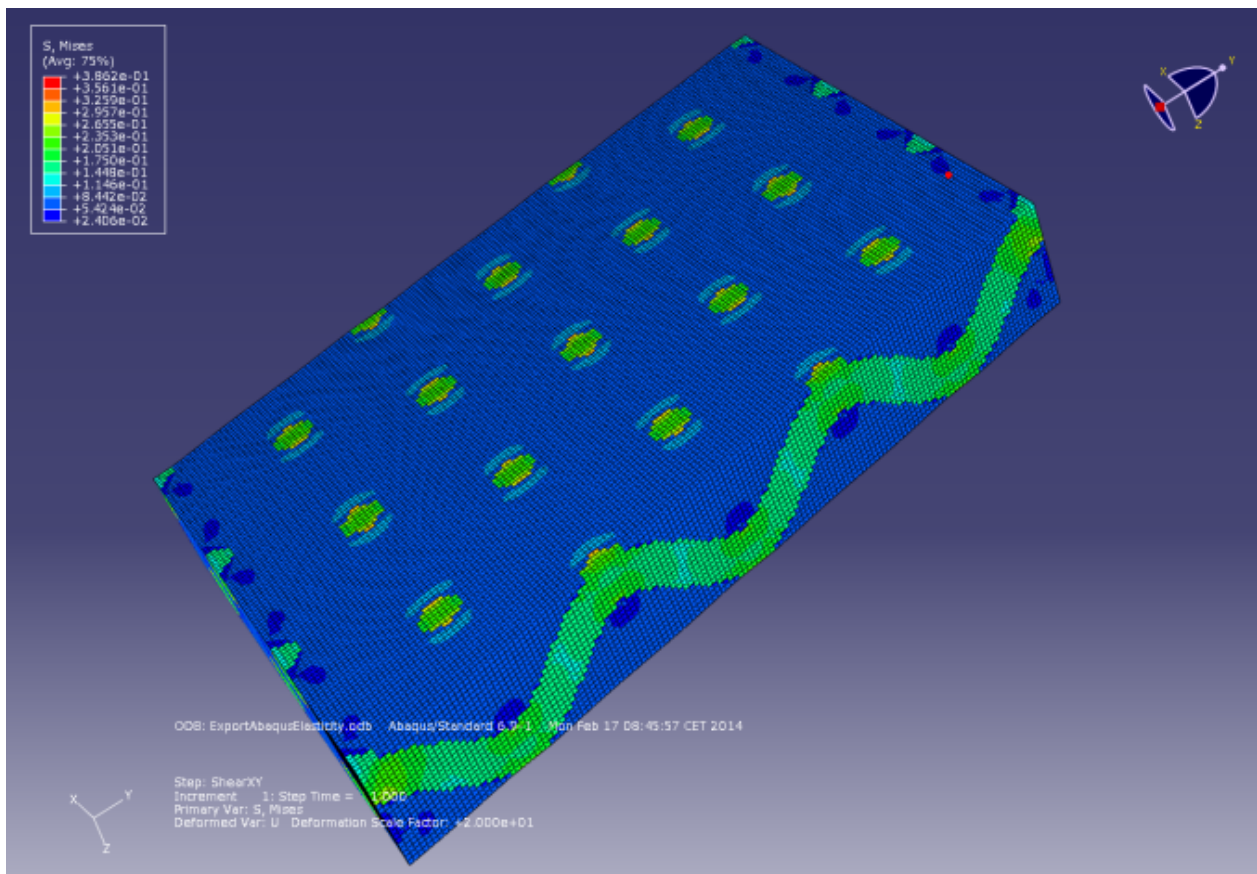
ExportGeo-Abaqus: Export Stiffness Simulation (Voxel)



Under the **Solver** tab, use the default values for the **Iterative** solver. Click **OK** to close the **ExportGeo-Abaqus – Export Stiffness Simulation (Voxel)** dialog and 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.

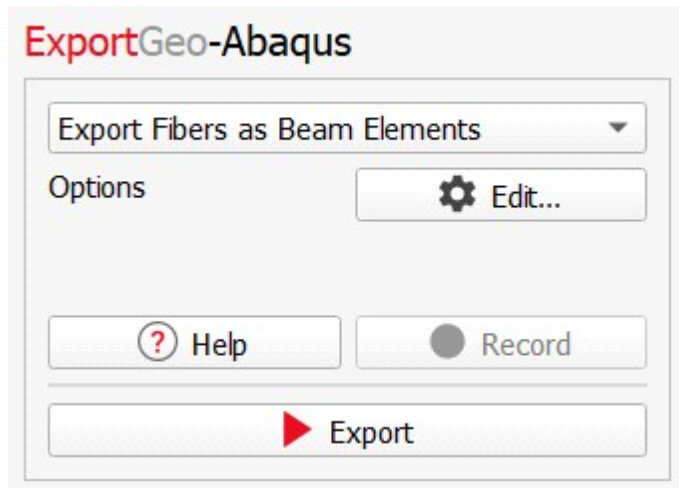
To run the elasticity computation in **Abaqus**, run the input file (Abaqus.inp) in **Abaqus**.



1.2 Export Fibers as Beam Elements

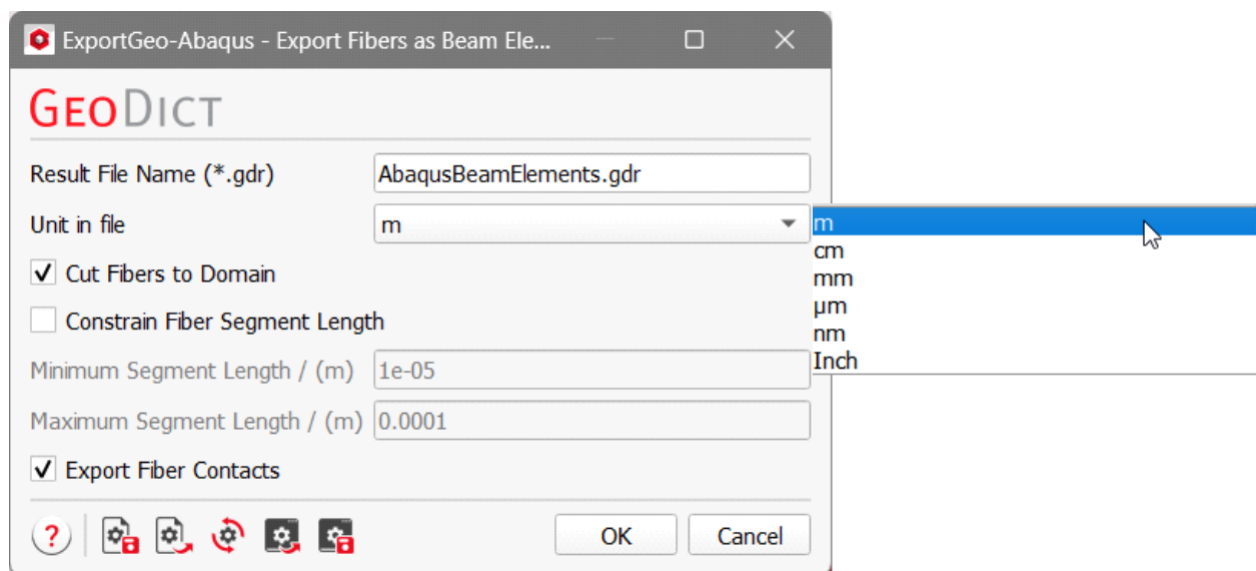
With **Export Fibers as Beam Elements**, fiber objects from GeoDict can be exported to Abaqus.

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 **ExportGeo-Abaqus – Export Fibers as 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.



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**.

To export fibers to Abaqus beam elements, the setting **Cut Fibers to Domain** is to limit the fibers to the domain. In GeoDict, analytic fiber objects (GAD 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 (see also the GadGeo command Cut Fibers to Domain).

☒ **Constrain Fiber Segment Length**

Minimum Segment Length / (m)

1e-05

Maximum Segment Length / (m)

0.0001

If it is necessary to constrain the maximum and minimum of the fiber segment length, check **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.

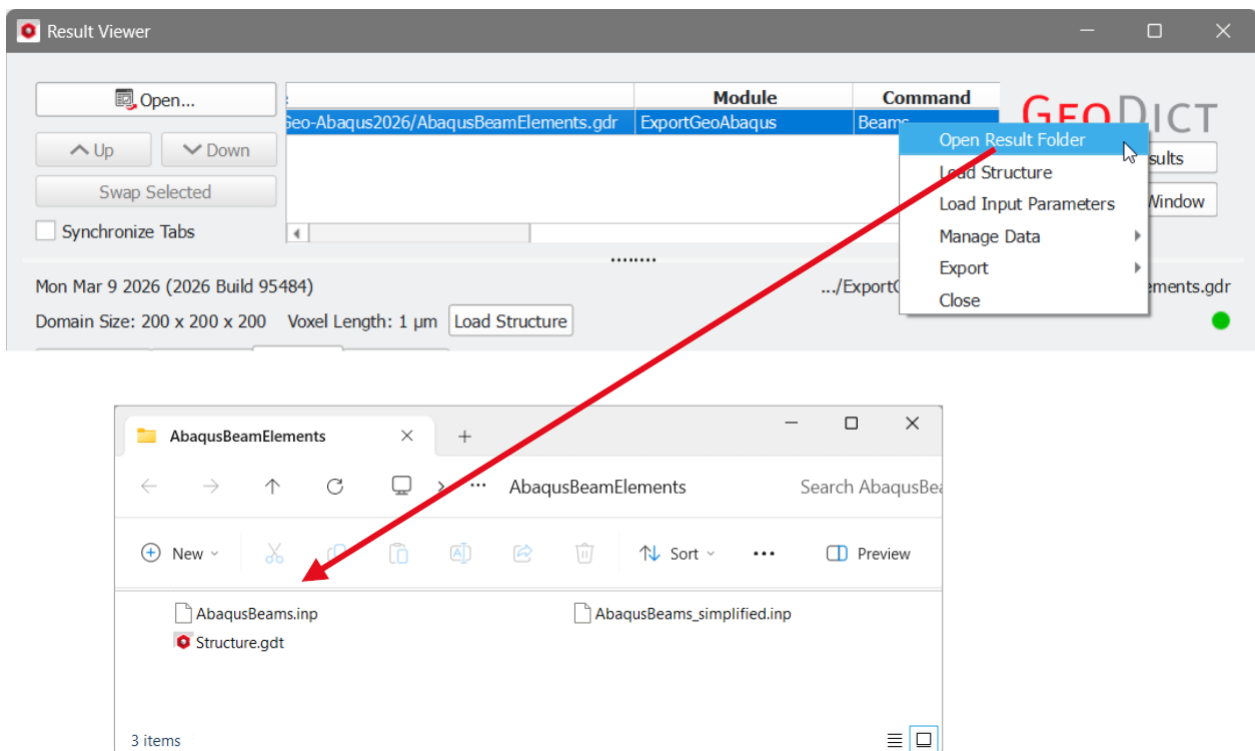
When **Export Fiber Contacts** is checked, the fiber contact information is included in the result .gdr file and is found in the **Result Map**.

Input Map Log Map Results Metadata			
Report Map			
Key	Unit	Value	
InitialLength	m	0.01509553067	
ProcessedLength	m	0.01509553067	
OverlapBefore	Voxel	0	
OverlapAfter	Voxel	0	
NumberOfObjects		80	
NumberOfContacts		50	
TotalContactFaces		13513	
MeanCoordinationNumber		1.25	
▼ Contacts			
Object1		2, 2, 2, 2, 5, 5, 6, 6, 8, 8, 9, 11, 11, 12, 12, 13, 15, 17, 17, 21, 23, 24, 26, 26, 28, 28, 28, 29, 30, 31, 3...	
Object2		9, 22, 44, 78, 10, 77, 8, 23, 23, 31, 43, 29, 41, 48, 79, 22, 42, 31, 34, 23, 34, 51, 28, 67, 50, 67, 69, 41...	
ContactFaces		73, 349, 99, 43, 226, 241, 491, 115, 163, 183, 8, 46, 11, 578, 76, 73, 16, 742, 900, 118, 874, 940, 117, ...	
InpFileName		AbaqusBeams.inp	

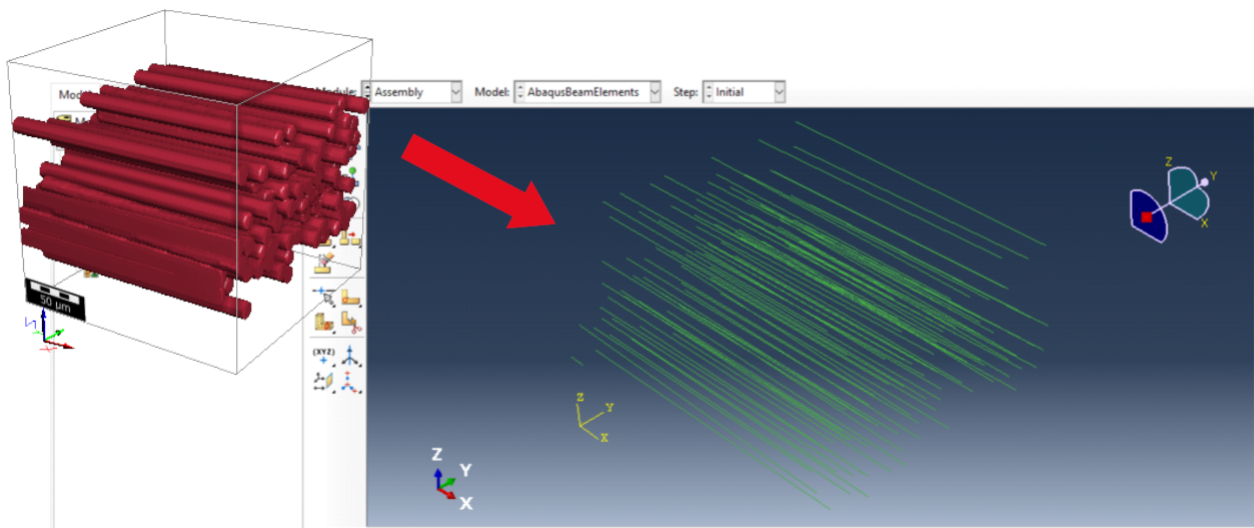
1.2.1 Results

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.

ExportGeo-Abaqus: Export Fibers as Beam Elements



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	AbaqusBeams_simplified.inp
1338 1330, 1.149696e-04, 1.931483e-04, 1.471163e-04	1338 1330, 1.149696e-04, 1.931483e-04, 1.471163e-04
1339 1331, 1.271168e-04, 1.931483e-04, 1.471163e-04	1339 1331, 1.271168e-04, 1.931483e-04, 1.471163e-04
1340 1332, 1.392640e-04, 1.931483e-04, 1.471163e-04	1340 1332, 1.392640e-04, 1.931483e-04, 1.471163e-04
1341 1333, 1.514112e-04, 1.931483e-04, 1.471163e-04	1341 1333, 1.514112e-04, 1.931483e-04, 1.471163e-04
1342 1334, 1.635584e-04, 1.931483e-04, 1.471163e-04	1342 1334, 1.635584e-04, 1.931483e-04, 1.471163e-04
1343 1335, 1.757056e-04, 1.931483e-04, 1.471163e-04	1343 1335, 1.757056e-04, 1.931483e-04, 1.471163e-04
1344 1336, 1.878528e-04, 1.931483e-04, 1.471163e-04	1344 1336, 1.878528e-04, 1.931483e-04, 1.471163e-04
1345 1337, 2.000000e-04, 1.931483e-04, 1.471163e-04	1345 1337, 2.000000e-04, 1.931483e-04, 1.471163e-04
- 1346 ** fiber1	+ 1346 ** fibertype1_elements
- 1347 *ELEMENT, TYPE=B31, ELSET=fiber1_elements	+ 1347 *ELEMENT, TYPE=B31, ELSET=fibertype1_elements
1348 1, 1, 2	1348 1, 1, 2
1349 2, 2, 3	1349 2, 2, 3
1350 3, 3, 4	1350 3, 3, 4
1351 4, 4, 5	1351 4, 4, 5
1352 5, 5, 6	1352 5, 5, 6
1353 6, 6, 7	1353 6, 6, 7
1354 7, 7, 8	1354 7, 7, 8
1355 8, 8, 9	1355 8, 8, 9
1356 9, 9, 10	1356 9, 9, 10
1357 10, 10, 11	1357 10, 10, 11
1358 11, 11, 12	1358 11, 11, 12
1359 12, 12, 13	1359 12, 12, 13
1360 13, 13, 14	1360 13, 13, 14
1361 14, 14, 15	1361 14, 14, 15
1362 15, 15, 16	1362 15, 15, 16
1363 16, 16, 17	1363 16, 16, 17
- 1364 *BEAM SECTION, SECTION=CIRC, ELSET=fiber1_elements, MATERIAL=Solid	
- 1365 Ge=06	
- 1366 ** END fiber1	
- 1367	
- 1368 ** fiber2	
- 1369 *ELEMENT, TYPE=B31, ELSET=fiber2_elements	
1370 17, 18, 19	1364 17, 18, 19
1371 18, 19, 20	1365 18, 19, 20
1372 19, 20, 21	1366 19, 20, 21
1373 20, 21, 22	1367 20, 21, 22

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



Math2Market GmbH

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