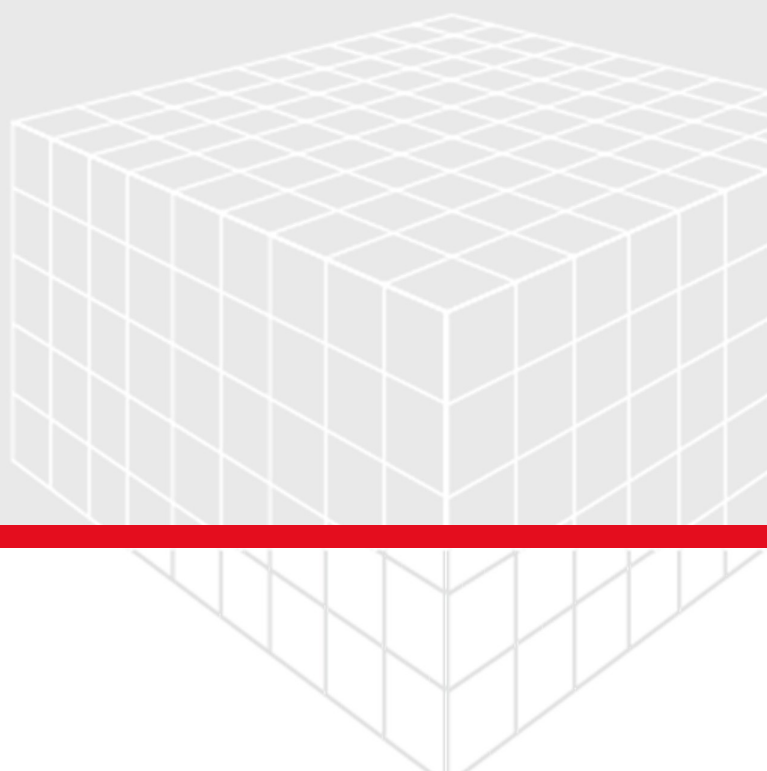


ElastoDict

User Guide

GEODICT Release 2026

Published: June 2026



GEO DICT

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

ElastoDict

1. ElastoDict	4
1.1 Materials in ElastoDict	7
1.2 Theoretical Background: Material Models in ElastoDict	9
1.2.1 Internal Material Laws in FeelMath	9
1.2.2 Setting up the Internal Material Laws	20
1.2.3 Material Fitting for the Internal Material Laws	23
1.2.4 Using UMATs for Custom Material Laws	29
1.3 Deformations	43
1.3.1 Constituent Materials	44
1.3.2 Macroscopic Load Case	46
1.3.3 Geometry Handling	55
1.3.4 Solver	63
1.3.5 Output	68
1.3.6 Result File	70
1.3.7 Restarting a simulation	85
1.3.8 Example: Matrix Damage	86
1.4 Flexural Test	99
1.4.1 Constituent Materials	99
1.4.2 Macroscopic Load Case	102
1.4.3 Solver	105
1.4.4 Output	105
1.4.5 Result File	108
1.5 Effective Stiffness	111
1.5.1 Elasticity Theory	112
1.5.2 Constituent Materials	116
1.5.3 Load Case	119
1.5.4 Solver	122
1.5.5 Output	130
1.5.6 Post-processing	130
1.5.7 Result File	133
1.5.8 Effective Elastic Properties	136
1.5.9 Visualization Example	145
1.5.10 Benchmark Results for Effective Stiffness	155
1.6 Effective Stiffness (AF)	157
1.6.1 Constituent Materials	157
1.6.2 Load Case	160
1.6.3 Post-processing	161
1.6.4 Result File	161
1.7 Mechanics GeoApps	164
1.7.1 Stiffness Estimation	165

1.7.2 Contact Detachment	168
1.8 References	178

1 ElastoDict

The ElastoDict module helps in the development of new materials by providing better understanding of their properties. The key to optimizing your materials are accurate simulations on the 3D micro-structure. Common application areas for ElastoDict are composite materials, ceramics, metals or foams, e.g., but also rock samples.

ElastoDict contains the following sub-modules:

✓ Deformations

ElastoDict-Deformations allows to simulate different types of deformation under user-defined loads. For example, a standard tensile experiment in an arbitrary direction of the 3D micro-structure can be set up. The models of the constituent materials might contain damage, failure, plastic deformation, viscous effects, and many more. For linear elastic materials, new material contacts in the structure can be detected during the simulation. Several nonlinear material models are available, and additionally, UMATs compatible with Abaqus are supported. Results of the simulation are, e.g., a stress-strain curve and local information on the regions where damage sets in and the material ends up failing.

✓ Flexural Test

With the Flexural Test option in ElastoDict, a 3-point bending test is simulated for a complex micro-structure. The result includes detailed strain and stress information, as well as the flexural modulus.

✓ Effective Stiffness

ElastoDict-Effective Stiffness computes the linear elastic properties of complex micro-structures by solving the corresponding partial differential equation on the 3D image or model. For Effective Stiffness, two solvers are available: FeelMath and LIR. The results include the local stresses and strains, the complete stiffness tensor, and the information on the orthotropic, transversal isotropic or isotropic character of the material, indicative of directionally dependent properties. Additionally, many post-processing steps can be carried out on the ElastoDict-Effective Stiffness results.

✓ Effective Stiffness (AF)

ElastoDict-Effective Stiffness (AF) computes analytic approximations and bounds for the linear elastic properties of complex micro-structures. The computation is very fast, as no partial differential equation is solved, and gives a first approximation of the material behavior.

✓ Mechanics GeoApps

This section contains GeoApps which use the functionality of ElastoDict. Currently, two GeoApps are available:

- the Stiffness Estimation GeoApp helps to estimate the stiffness of an unknown material within a complex micro-structure.
- the Contact Detachment GeoApp is used to simulate object separation and detachment.



New in GeoDict 2026

- The [video generator](#)^[80] for ElastoDict-Deformation results now allows to visualize the deformation of the objects in the geometry, if "Enable Object Tracking" is activated for the simulation.

Learn to use ElastoDict

- [Materials in ElastoDict](#)^[7]
General information about materials used for ElastoDict.
- [Theoretical Background: Material Models in ElastoDict](#)^[9]
In this chapter setting up nonlinear and custom material laws for ElastoDict simulations is discussed.
- [Deformations](#)^[43]
The Deformations module allows to simulate different types of deformation under user-defined loads.
- [Flexural Test](#)^[99]
With the Flexural Test option in ElastoDict, a 3-point bending test is simulated for a complex micro-structure.
- [Effective Stiffness](#)^[111]
The module Effective Stiffness computes the linear elastic properties of complex micro-structures by solving the corresponding partial differential equation on the 3D image or model.
- [Effective Stiffness \(AF\)](#)^[157]
The module Effective Stiffness (AF) computes analytic approximations and bounds for the linear elastic properties of complex micro-structures.
- [Mechanics GeoApps](#)^[164]
This section contains GeoApps which use the functionality of ElastoDict
- [References](#)^[178]
Publications cited in this chapter.



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 Materials in ElastoDict

Solid and porous structures can be used as input for computations with ElastoDict. The mechanical properties of all constituent materials must be given. Including the pore/matrix material, GeoDict handles structures with up to 256 different constituent materials.

As a useful convention, for solid materials, the background (Material ID0) contributes to the computation and is referred to as matrix material. For porous media, Material ID0 is used for the pore space (void space).

Solid Materials

In solid structures, all constituent materials have non-zero physical material parameters (e.g., Poisson's ratio and Young's modulus for linear elastic materials; see the [Constituent Materials](#)^[44] section). In structures consisting of multiple solid materials, like, e.g., composite materials, ElastoDict applies perfect contact at the interfaces between different materials. A typical example for a solid structure is a glass fiber reinforced composite.

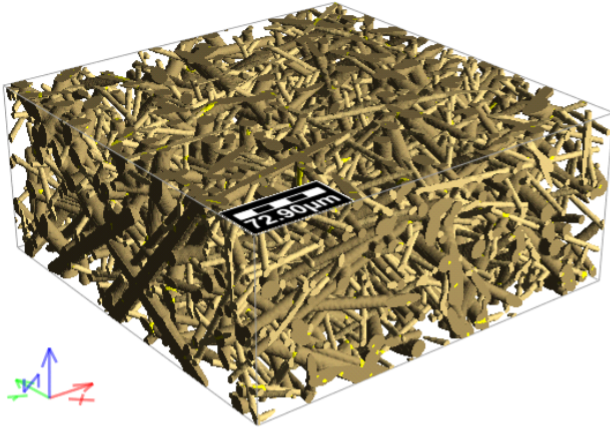
Many different material models (plasticity, visco-plasticity, damage, ...) are available in GeoDict. Additionally, user defined materials can be implemented as an [Abaqus UMAT](#)^[29].

Porous Materials

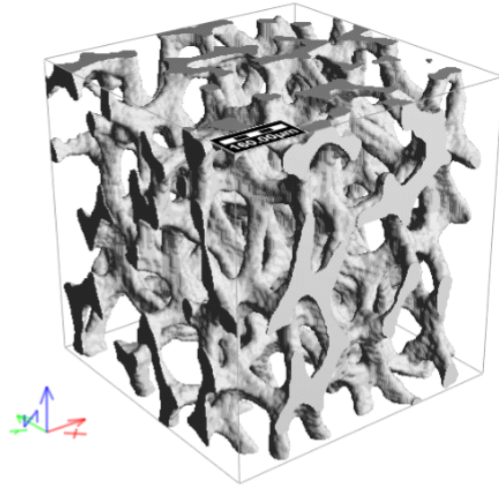
Porous structures in GeoDict are materials where at least one constituent material is selected as Pore (Fluid) material, this means its Young's Modulus and Poisson's ratio are both set to zero. No stresses can occur in the pore space. Examples of porous structures are, e.g., nonwovens and foams.

Some structures whose mechanical properties can be calculated with ElastoDict are:

Isotropic fibrous microstructure model generated using FiberGeo



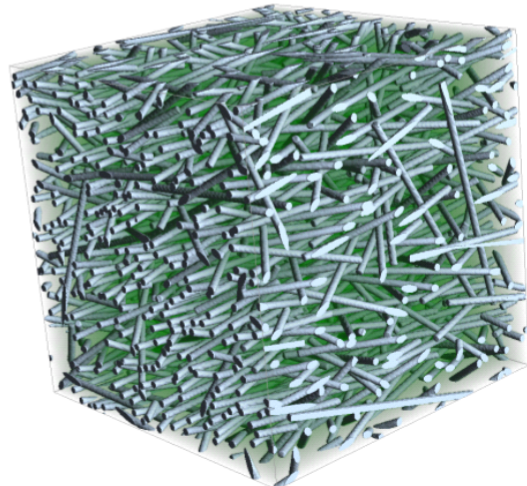
Nickel Foam microstructure model generated using FoamGeo



Berea Sandstone modeled from imported 3D data using ImportGeo-Vol

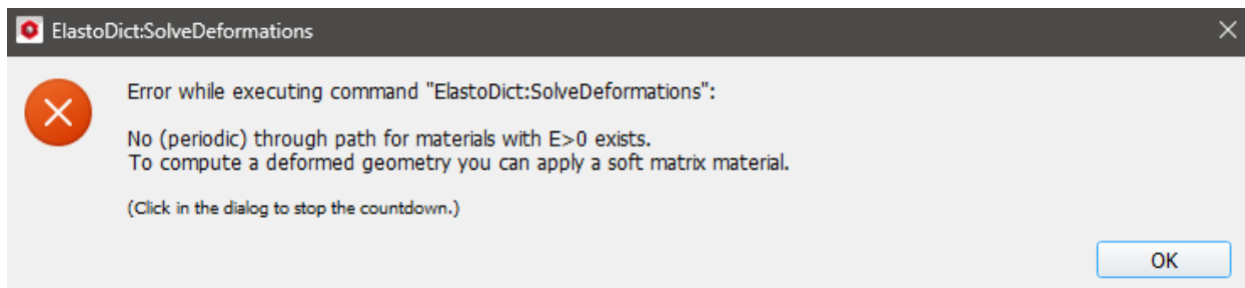


Glass fiber reinforced polymer model generated using FiberGeo



For materials in the pore space, a pore pressure can be set which leads to stresses in the surrounding solid materials.

It is a requirement of all ElastoDict simulations that a continuous load path exists through the structure. This means that the solid materials must form a connected component in the load direction. Otherwise, the error message below is shown, and the simulation cannot be started.



1.2 Theoretical Background: Material Models in ElastoDict

Materials in GeoDict can contain all material properties that are needed for simulations in GeoDict. For setting up ElastoDict simulations correctly, you have to define the mechanical properties of the materials that make up the structure.

In the GeoDict **Material Database** different linear and nonlinear materials are stored to set a the mechanical properties. A selection of standard materials is already predefined, but the true strength of the database is that it allows the user to define additional materials to be used in GeoDict simulations. More information on editing and using the GeoDict Material Database is available in the Material Database User Guide.

In the following sections the usage of nonlinear models and custom material laws are discussed. For how to incorporate predefined materials in ElastoDict, please refer to [Constituent Materials](#)^[44].

In ElastoDict, several types of nonlinear material laws are integrated. The next section explains how these material laws can be set up.

Additionally, it is also possible to implement user-defined material laws through the [UMAT interface](#)^[29].

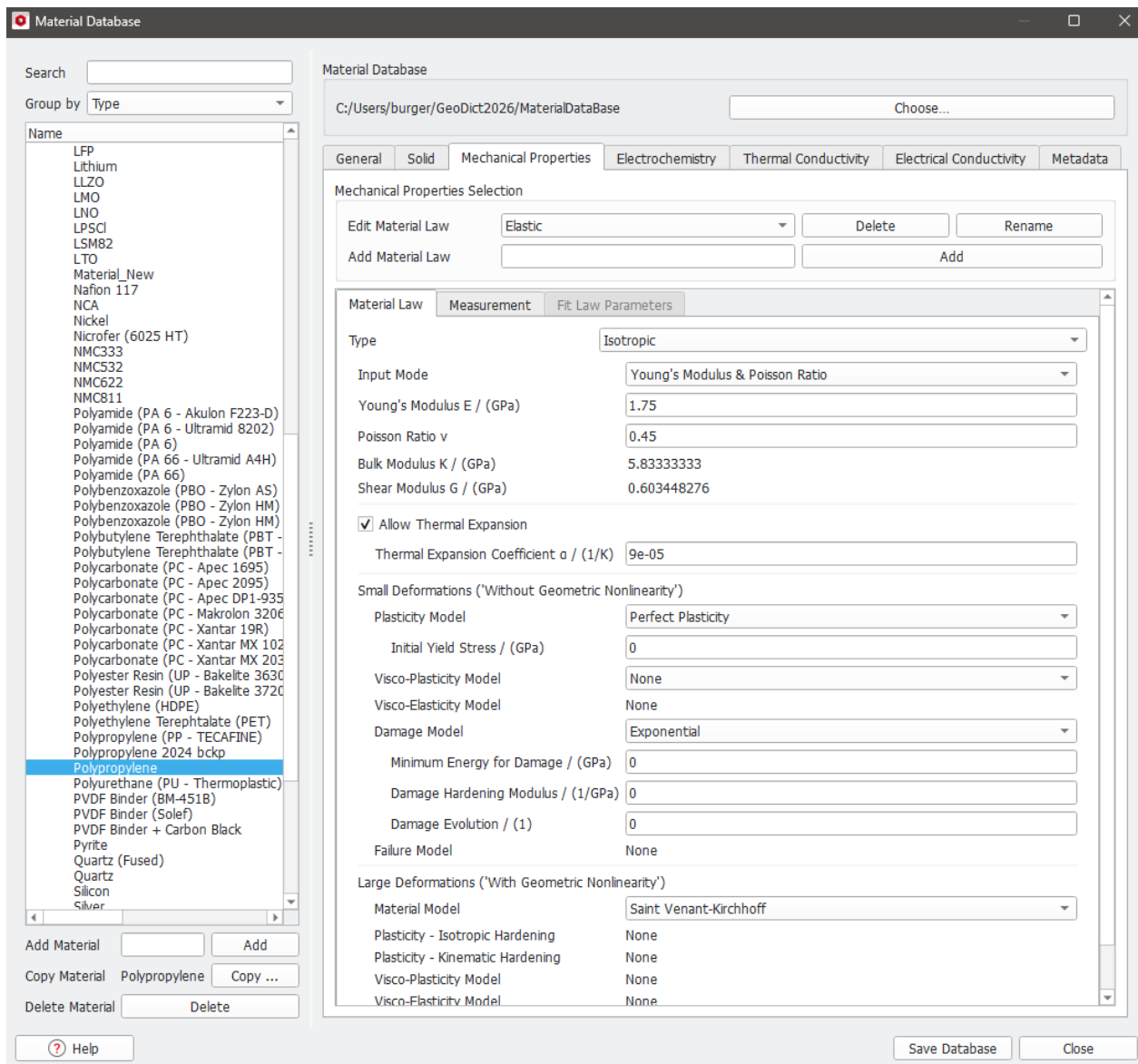
Topics of this section

- [Internal Material Laws in FeelMath](#)^[9]
Description of all material laws available in ElastoDict.
- [Setting up the Internal Material Laws](#)^[20]
Description of ElastoDict's material law setup.
- [Material Fitting for the Internal Material Laws](#)^[23]
Description of the material fitting process in ElastoDict.
- [Using UMATs for Custom Material Laws](#)^[29]
The Usage of UMATs for custom material laws for ElastoDict simulations is presented in this chapter.

1.2.1 Internal Material Laws in FeelMath

Several nonlinear material laws are delivered with GeoDict and may be conveniently set up in the GeoDict Material Database dialog. See the Material Database User Guide for more information.

The available nonlinear material laws are: **Elasticity, Plasticity, Damage, Viscosity** and **Failure**. All these material models are only available for the deformation type [Without Geometric Nonlinearity](#)^[56].



These material laws can be combined in different ways:

- Plasticity, damage, and viscosity can be combined. For these combinations, only the exponential damage model is available.
- The Mazars damage model cannot be combined with any other model.
- The Failure model cannot be combined with any other model.

Topics of this section

- [Linear Elasticity](#)^[11]
Description of how to set up Linear Elasticity in ElastoDict.
- [Plasticity](#)^[12]
Description of plasticity models available in ElastoDict
- [Isotropic and Kinematic Hardening](#)^[13]
Description of Hardening models available in ElastoDict.

Topics of this section

- [Visco-Plasticity](#)¹⁴
Description of Viscosity models available in ElastoDict.
- [Damage](#)¹⁷
Description of Damage models implemented in ElastoDict.
- [Failure](#)¹⁹
Description of Failure model setup.
- [Guidelines for Plasticity, Damage and Failure Models](#)¹⁹
General remarks on how to set up Plasticity, Damage and Failure in ElastoDict.

Linear Elasticity

In GeoDict, the linear elastic material model can be defined by either the parameter pair **Young's Modulus** E and **Poisson Ratio** ν

Material Law	Measurement	Fit Law Parameters
Type	Isotropic	
Input Mode	Young's Modulus & Poisson Ratio	
Young's Modulus E / (GPa)	1.75	
Poisson Ratio ν	0.45	
Bulk Modulus K / (GPa)	5.83333333	
Shear Modulus G / (GPa)	0.603448276	

or the **Bulk Modulus** K and the **Shear Modulus** G . It is redundant for the model which parameter pair is used.

Material Law	Measurement	Fit Law Parameters
Type	Isotropic	
Input Mode	Bulk Modulus & Shear Modulus	
Young's Modulus E / (GPa)	1.75	
Poisson Ratio ν	0.45	
Bulk Modulus K / (GPa)	5.83333333	
Shear Modulus G / (GPa)	0.603448276	

All nonlinear material laws available in GeoDict's Material Database are based on linear isotropic elastic materials, except of the failure model, which is also available for transverse isotropic materials.

Plasticity

Plasticity in materials describes the effect that a deformation remains when the load is removed. In GeoDict, three different plastic material laws are available: **Perfect Plasticity**, **Bilinear Hardening** (Affine Linear Hardening) and **Exponential Hardening**.

For understanding plastic material laws, the two most important parameters are the elastic strain $\varepsilon_{\text{elas}}$ and the plastic strain $\varepsilon_{\text{plas}}$ which add up to the strain ε . The elastic strain is recoverable, whereas the plastic strain remains in the material after unloading. The relation of the elastic strain $\varepsilon_{\text{elas}}$ to the stress σ is described by Hooke's law, and the relation of the plastic strain $\varepsilon_{\text{plas}}$ to the stress σ is described by the chosen plastic material law. Please note that strain values in the formulas below are given in dimensionless form (not in %).

As long as the stress is lower than the yield stress Y of the material, the deformation is purely elastic. After this point, an increase in stress leads to a combination of elastic and plastic strains. The yield stress increases through the plastic deformation, which means that the material is elastic up to this point under a new load. Nevertheless, the course of the plastic material always depends on the initial yield stress.

The three available plastic material laws differ in the relation between the plastic strain and the stress:

Perfect Plasticity

As soon as the initial yield stress Y is reached, the stress remains at that level. From this point, the elastic strain stays constant, and all additional strain is purely plastic.

Bilinear (Affine Linear) Hardening

The stress is a linear function of the plastic strain

$$\sigma(\varepsilon_{\text{plas}}) = Y + H \cdot \varepsilon_{\text{plas}} \quad (1)$$

where H is the **Plastic Strain Hardening Modulus**.

The Plastic Strain Hardening Modulus is not the slope of the stress-strain curve, since it only takes the plastic strain into account. The slope of the stress-strain curve (after the yield stress is reached) is the Tangential Modulus T . The Plastic Strain Hardening Modulus and the Tangential Modulus are related as

$$H = \frac{T}{1 - \frac{T}{E}} \quad (2)$$

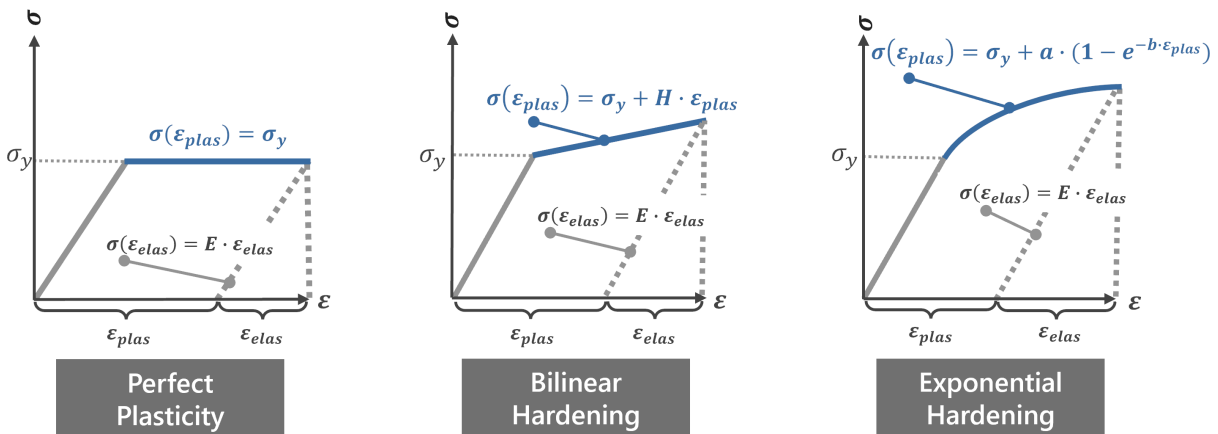
with the young's modulus E . In the material definition in GeoDict, either the Plastic Strain Hardening Modulus or the Tangential Modulus can be set.

Exponential Hardening

The stress is related to the plastic strain via an exponential function

$$\sigma(\varepsilon_{plas}) = Y + a \cdot (1 - e^{-b \cdot \varepsilon_{plas}}) \quad (3)$$

with the **First Hardening Parameter** a (unit: [GPa]) and the dimensionless **Second Hardening Parameter** b .

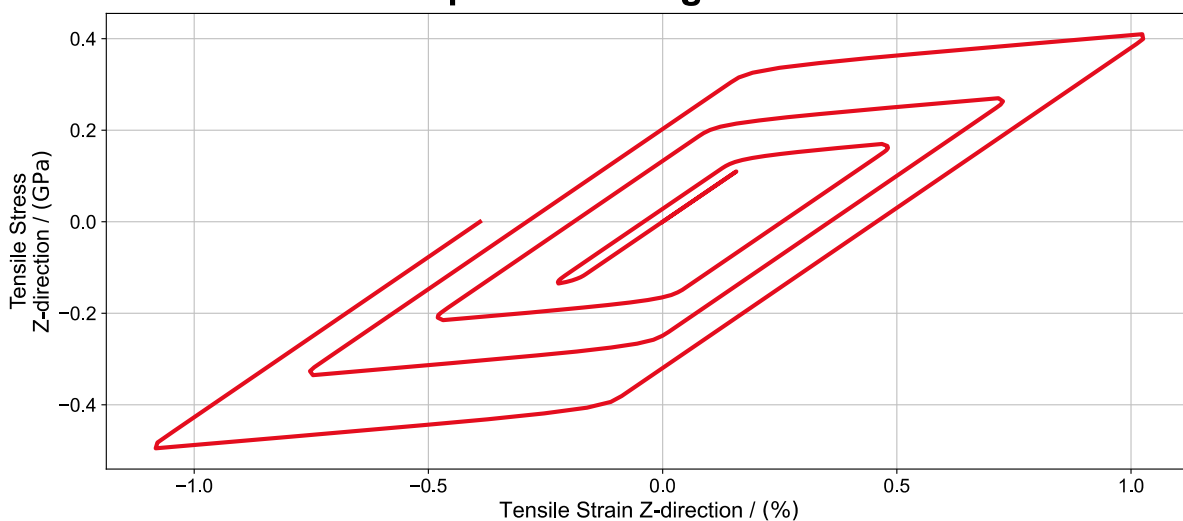


Isotropic and Kinematic Hardening

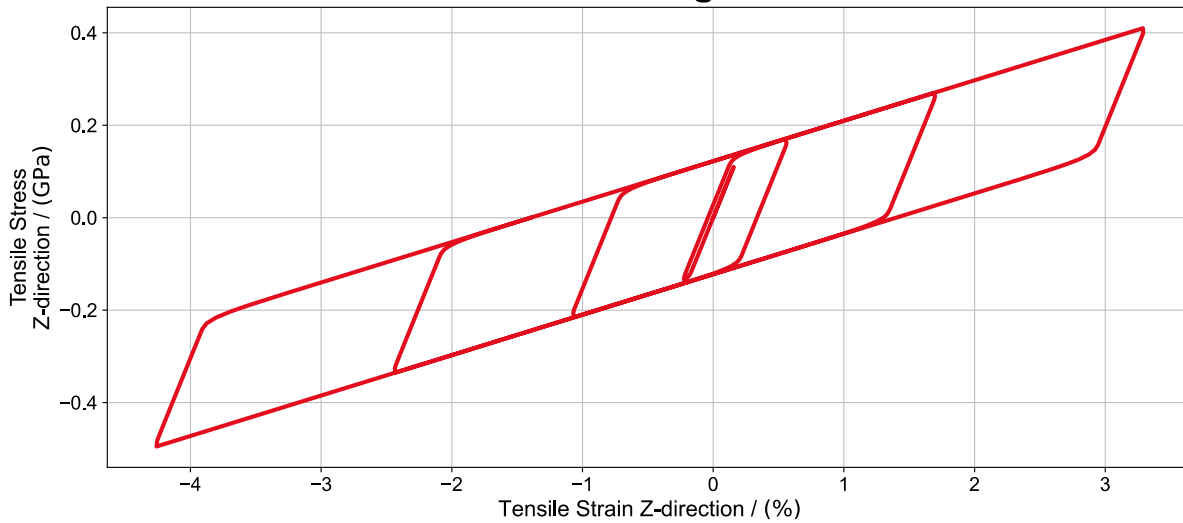
Isotropic and **Kinematic Hardening** describe the effect of the plastic deformation on the yield surface (see [Wikipedia: Yield Surface](https://en.wikipedia.org/wiki/Yield_Surface) for further reference).

In summary, the yield surface grows isotropically with isotropic hardening, whereas it moves with kinematic hardening.

Isotropic Hardening

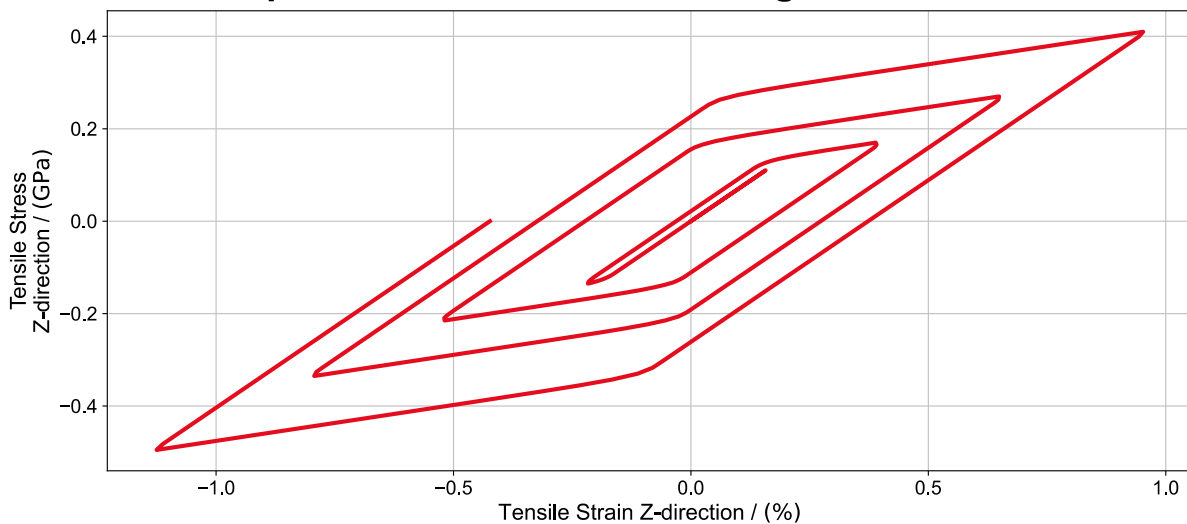


Kinematic Hardening



In real materials, the material behavior is usually a combination of both hardening types.

Isotropic and Kinematic Hardening



How to set up the hardening type for a material is expert knowledge and can only be analyzed by measuring multiple load cycles for a material. Nevertheless, if only tension or compression is applied to a material, the results for both hardening types are the same, and it is sufficient to keep the default **Isotropic Hardening** when setting up a material law.

Visco-Plasticity

The material laws for plasticity can also be combined with viscosity to achieve visco-plastic material behavior. Three models are available: **linear Perzyna**, **nonlinear Perzyna** and **Michel-Suquet**.

The three models differ in their complexity. The linear Perzyna model is the least complex of the three. It has only one additional parameter (the reference strain rate) compared to time-independent plasticity. In general, one should first try if this model already fits the needs.

The other models should only be considered if the real material shows a behavior which is too complex for the linear Perzyna model. Both have the rate sensitivity parameter, which allows to define how a change in strain rate affects the material behavior. The Michel-Suquet model additionally has the drag stress parameter, which further influences the material behavior and allows to fine-tune the model.

In general, setting up a visco-plastic material law is more complex than setting up time-independent materials. Measuring a stress-strain curve is not enough to explain the behavior of a visco-plastic material. In contrast, more complex experiments are necessary, like, e.g., creep tests, relaxation tests or DMA measurements (Dynamic mechanical analysis).

The different models for visco-plasticity are described by their flow rules. A **flow rule** defines how the material deforms plastically under a given stress and provides a mathematical expression for the rate at which plastic deformation accumulates. The central parameters in each model are the reference strain rate $\dot{\epsilon}_0$, which indicates how fast the plastic deformation occurs, and the yield stress σ_y , which is updated by the [hardening](#)^[13] law.

In the following the core formulas of the available plasticity models as well as the corresponding parameters to set up these models in GeoDict are described.

Linear Perzyna Model

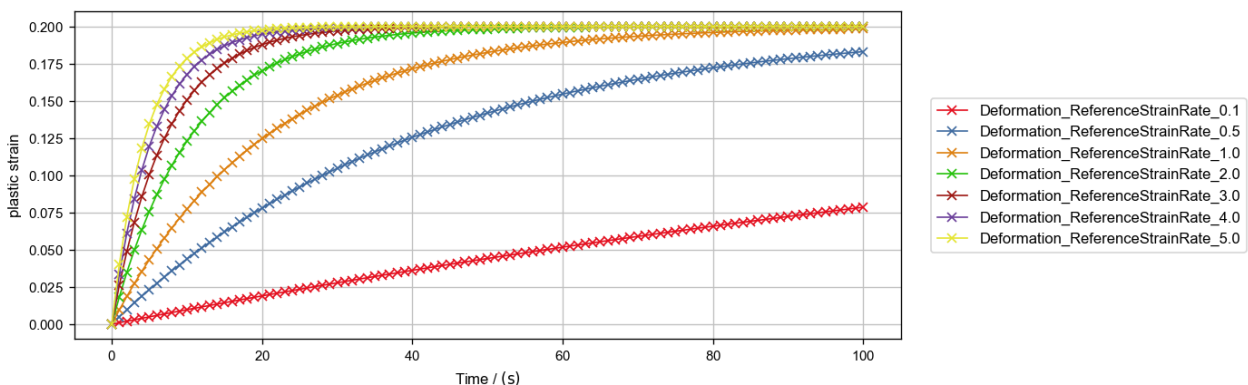
The flow rule describes a linear relationship between the overstress $\sigma - \sigma_y$ and plastic strain rate $\dot{\epsilon}_p$:

$$\dot{\epsilon}_p = \dot{\epsilon}_0 (\sigma - \sigma_y) \text{ for } \sigma > \sigma_y \quad (4)$$

where σ_y is the yield stress and $\dot{\epsilon}_0$ the reference strain rate.

Example: Creep experiment with varying reference strain rates

For this experiment a material was loaded with a constant stress and the plastic strains were monitored for different reference strain rates. In the result plots you can see that with higher reference strain rates the plasticity occurs faster in the material.



Nonlinear Perzyna Model

The flow rule describes a nonlinear relationship between the overstress $\sigma - \sigma_y$ and plastic strain rate $\dot{\epsilon}_p$:

$$\dot{\epsilon}_p = \dot{\epsilon}_0 \left(\frac{\sigma - \sigma_y}{\sigma_y} \right)^n \quad \text{for } \sigma > \sigma_y \quad (5)$$

where σ_y is the yield stress, $\dot{\epsilon}_0$ the reference strain rate and n the strain rate sensitivity exponent. For the nonlinear Perzyna Model, the relationship between overstress $\sigma - \sigma_y$ and plastic strain rate $\dot{\epsilon}_p$ is raised to a power of n , where the strain rate sensitivity exponent n characterizes the sensitivity of $\dot{\epsilon}_p$ to overstress.

Michel-Suquet Model

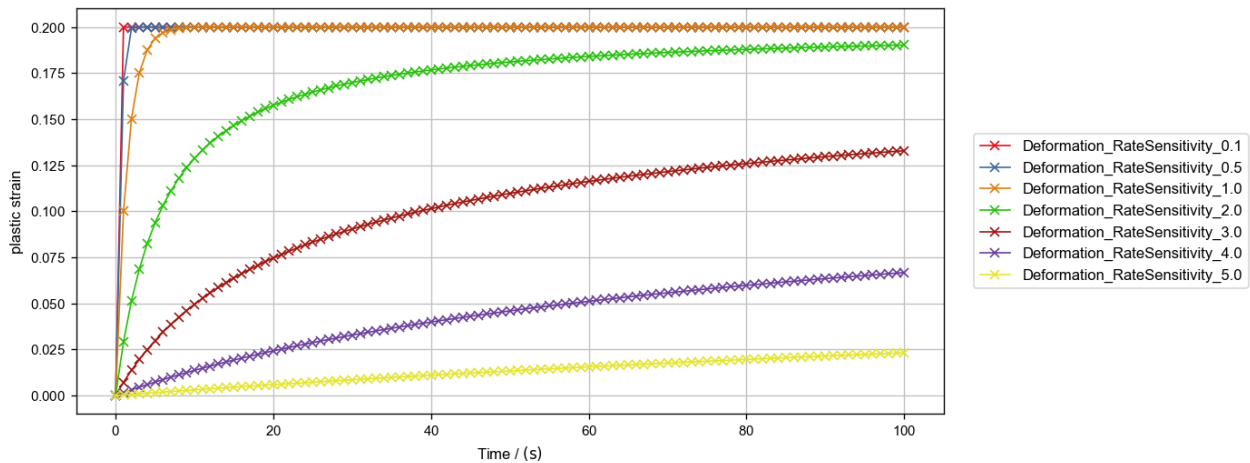
In the Michel-Suquet Model an additional stress threshold, σ_d , is introduced, allowing a more detailed description of the flow rule.

$$\dot{\epsilon}_p = \dot{\epsilon}_0 \left(\frac{\sigma - \sigma_y}{\sigma_d} \right)^n \quad \text{for } \sigma > \sigma_y \quad (6) \text{ Description (fakultativ)}$$

where σ_y is the yield stress, $\dot{\epsilon}_0$ the reference strain rate, n the strain rate sensitivity exponent and σ_d the drag stress, which controls the rate at which plastic strain accumulates in the material.

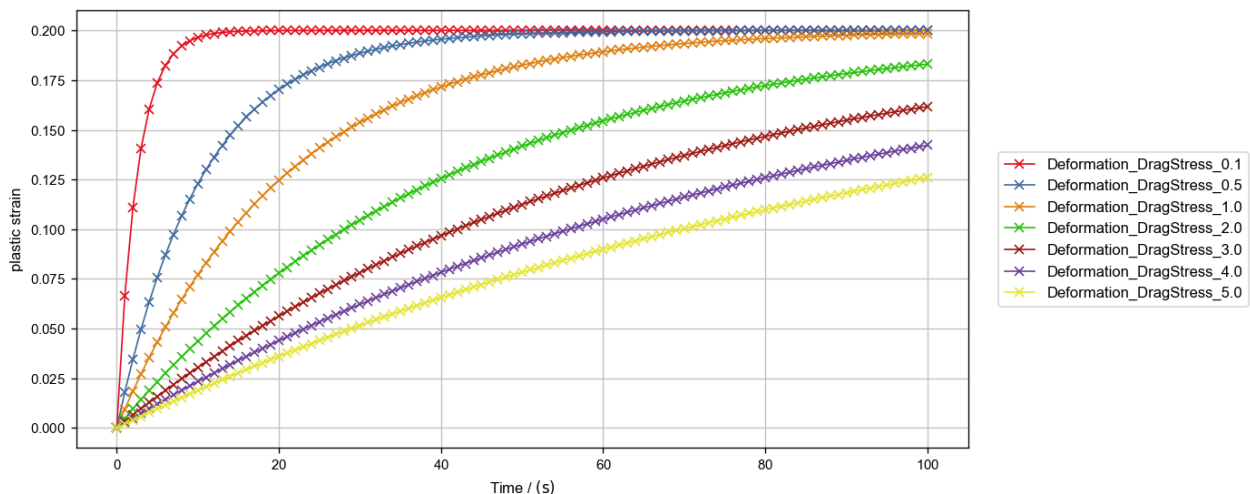
Example: Creep Experiment with varying strain rate sensitivity exponents

For this experiment a material was loaded with a constant stress and the plastic strains were monitored for different **rate sensitivity exponents**. In the plot of plastic strain ϵ_{plas} over time, you can see how the rate sensitivity exponent affects how quickly plastic deformation develops.



Example: Creep Experiment with varying drag stresses

For this experiment a material was loaded with a constant stress and the plastic strains were monitored for different **drag stresses**. In the plot of plastic strain $\varepsilon_{\text{plas}}$ over time, you can see that the higher the drag stress, the slower plasticity occurs in the material.

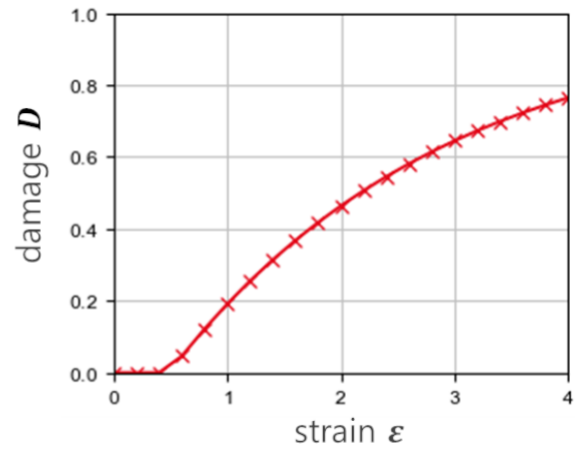
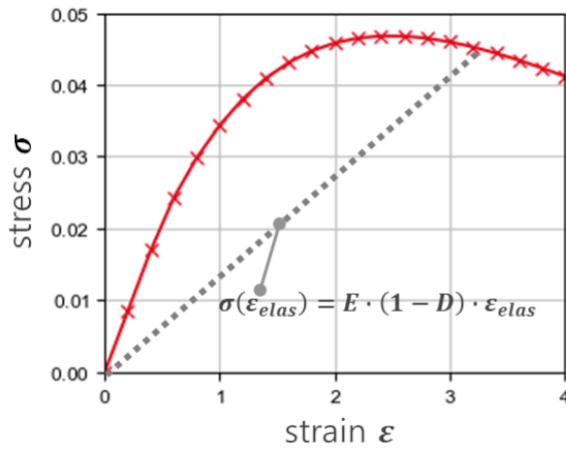


Damage

With the damage models, the stiffness of the material is decreased with increasing material damage:

$$\sigma = (1 - D) \cdot E \cdot \varepsilon \quad (7)$$

with the damage variable D . The evolution of the damage D depends on the selected damage model, and the damage can only increase. The initial value is $D = 0$, and the maximal allowed value for the damage is $D = 0.99$. When unloaded, the material deforms elastically (with the decreased stiffness) back to its initial state.



Two damage models are available in GeoDict: The **Exponential Damage** model and **Mazars' Damage** model (see [Mazars, 1981](#)^[178], [Alfano et al. 2001](#)^[178], and [Michel and Suquet, 2016](#)^[178] for more details). Both models are very similar and differ only in the way they depend on the strain.

Mazars' Damage: The damage depends on the strain, and the degradation begins as soon as the strain reaches the threshold d_0 :

$$D = 1 - \frac{d_0(1 - d_2)}{\varepsilon} - \frac{d_2}{e^{d_1(\varepsilon - d_0)}}, \varepsilon > d_0 \quad (8)$$

with d_0 : minimal strain for damage, d_1 : damage hardening modulus, d_2 : damage evolution.

Exponential Damage: The damage depends on the strain energy

$$U = U_{\text{elas}} + U_{\text{plas}} = \frac{E}{2} \cdot \varepsilon_{\text{elas}}^2 + \int_0^{\varepsilon_{\text{plas}}} (\sigma(\varepsilon_p) - Y) d\varepsilon_p \quad (9)$$

which is the sum of the elastic and plastic strain energies U_{elas} and U_{plas} . If no plastic deformations occur, the strain energy depends only on the elastic strains. The degradation begins as soon as the minimal energy for damage d_0 is reached:

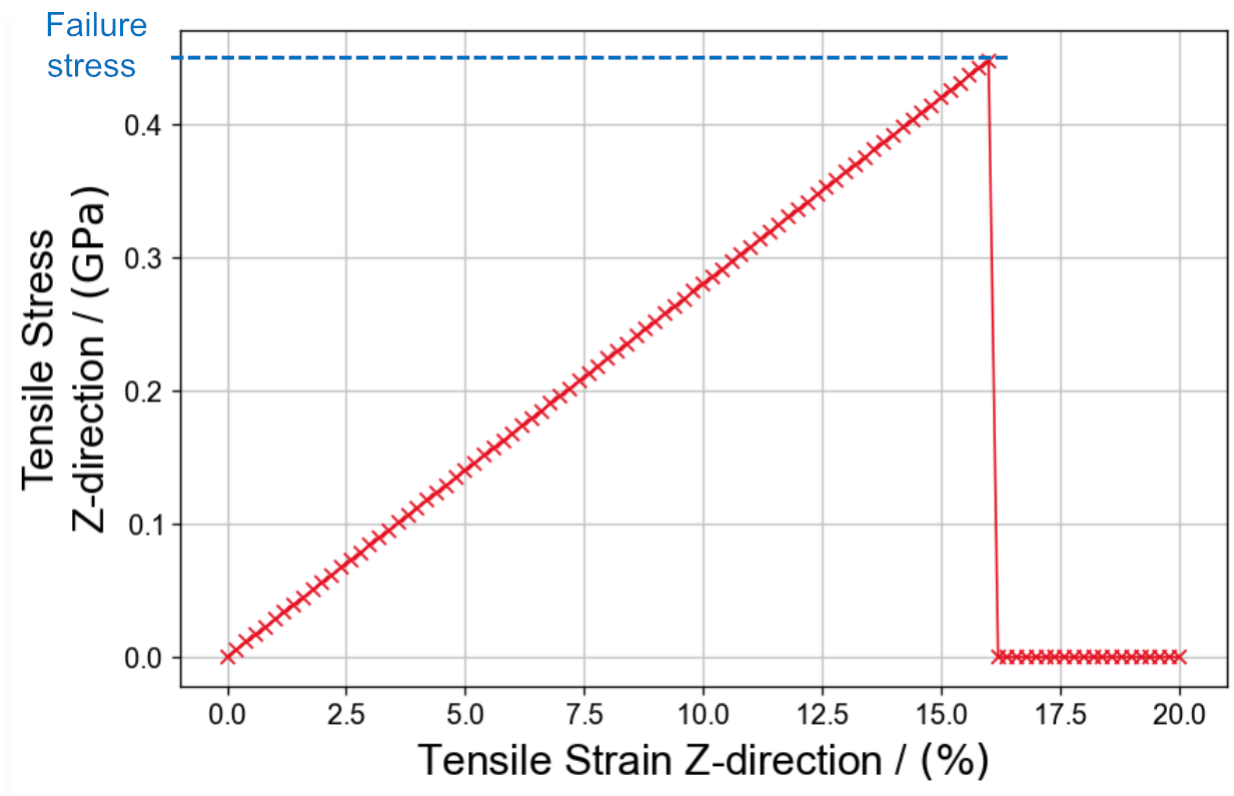
$$D = 1 - \frac{d_0(1 - d_2)}{U} - \frac{d_2}{e^{d_1(U - d_0)}}, U > d_0 \quad (10)$$

with d_0 : minimal strain for damage, d_1 : damage hardening modulus, d_2 : damage evolution.

The damage evolution formula is the same for both models, the significant difference is that the damage is driven by the strain for **Mazars' Damage** and driven by the elastic strain energy in the **Exponential Damage** model. In other words, the damage criterion for Mazars depends linearly on the strains, and a quadratic relation holds for the exponential damage model.

Failure

The **Failure** model in GeoDict is closely related to the damage models, but its approach is simpler. Instead of the continuous damage variable D , the failure model knows only two states: Failure or no failure. The material fails if the Failure Stress is exceeded, and the stiffness is degraded to 1 % of the original stiffness. The stress strain curve of an exemplary tensile experiment for an example material with a **Failure Stress** of 0.45 GPa looks as follows:



As soon as the defined Failure Stress is achieved, the material is considered to have failed and is no longer able to sustain any stress: the Tensile Stress in the strain stress plot is zero from that point on. Please also refer to the [matrix damage](#)^[86] example.

The volume field for the failure parameter contains a 0 for voxels without failure, and a 1 for failed voxels. However, when calculating with downsampling, or when calculating on deformed geometries, there might be also failure values between 0 and 1 resulting from a combination of several voxels.

The **Failure** model is available for isotropic and for transverse isotropic materials. In the isotropic case, the reference stress is the von Mises stress, and in the transverse isotropic case, the stress in the respective directions is checked.

Check the guidelines in [Mazars, 1981](#)^[178], [Alfano et al. 2001](#)^[178], and [Michel and Suquet, 2016](#)^[178] for a comparison of the damage and failure models in GeoDict.

Guidelines for Plasticity, Damage and Failure Models

The convergence for nonlinear materials can be bad if the simulation settings are not set up properly. Therefore, this section contains some best practices for these simulation types.

Load type recommendations



Note! If possible, do not use the **Force Controlled** load type.

With the **Force Controlled** load type, the applied load might be too high for the materials in the structure: For plastic materials and for the damage or failure models, the maximal stress is limited by the parameters of the material model. This can lead to unsolvable equations.

If it is important to run the simulation with a force controlled load (as, e.g., in a cyclic load case, where the stresses should decrease to zero again), we recommend one of the following workarounds

- Do a first simulation with a **Path Controlled** load type, to check out the maximal allowed stresses for the structure.
- Check the materials carefully to estimate the maximum allowed load.
- Run a force controlled simulation without failure/ damage/ plasticity first and use the deformation results as strains for a second, path controlled simulation with failure/ damage/ plasticity.

Nevertheless, even if the maximum allowed load is not reached, the convergence might be slow. For example, when using the exponential hardening model, a load slightly lower than the maximal allowed load might lead to strains near to infinity (which is also not realistic).

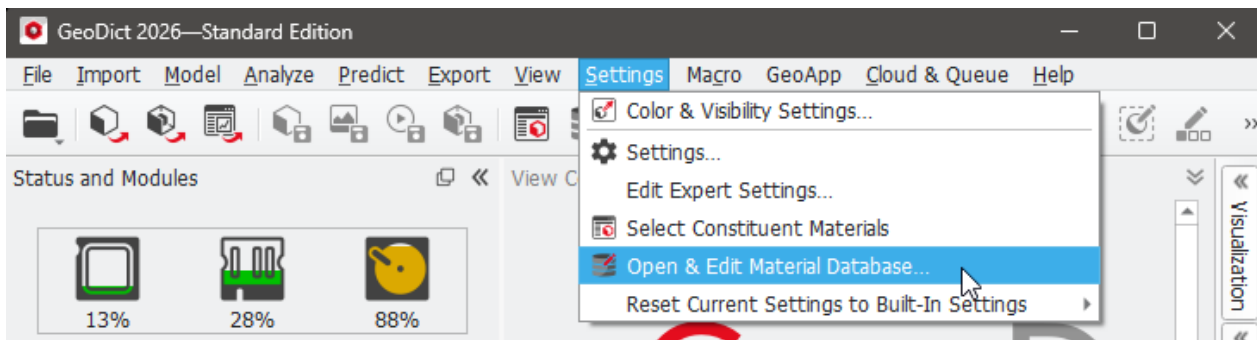
Damage models vs. failure models

The main difference between the damage model and the failure model is that the failure model decreases the stiffness suddenly, which introduces a discontinuity into the stress-strain relationship. The damage models lead to a gradual reduction of the stiffness. Thus, the damage models introduce no discontinuity, this means that the simulation is more stable and usually faster. Nevertheless, this gradual reduction of stiffness can be arbitrarily fast – this means that nearly the same results as with the failure model can be achieved. Additionally, the damage models can be combined with other models like, e.g., plasticity.

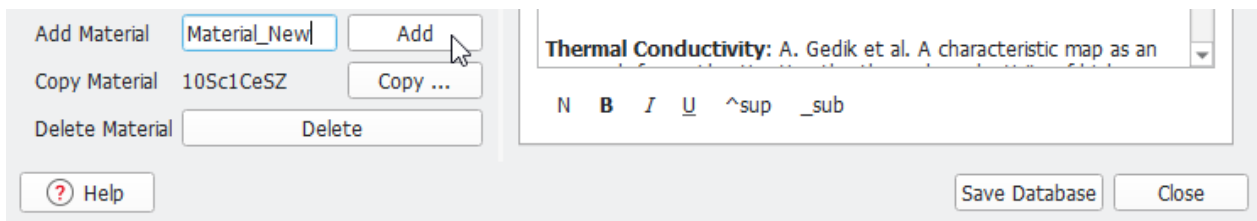
The main advantage of the failure model is that it is easier to set up, since it contains less parameters. Additionally, the damage models are not available for the transverse isotropic case. Altogether, it is strongly recommended to use the damage models instead of the failure model, whenever possible.

1.2.2 Setting up the Internal Material Laws

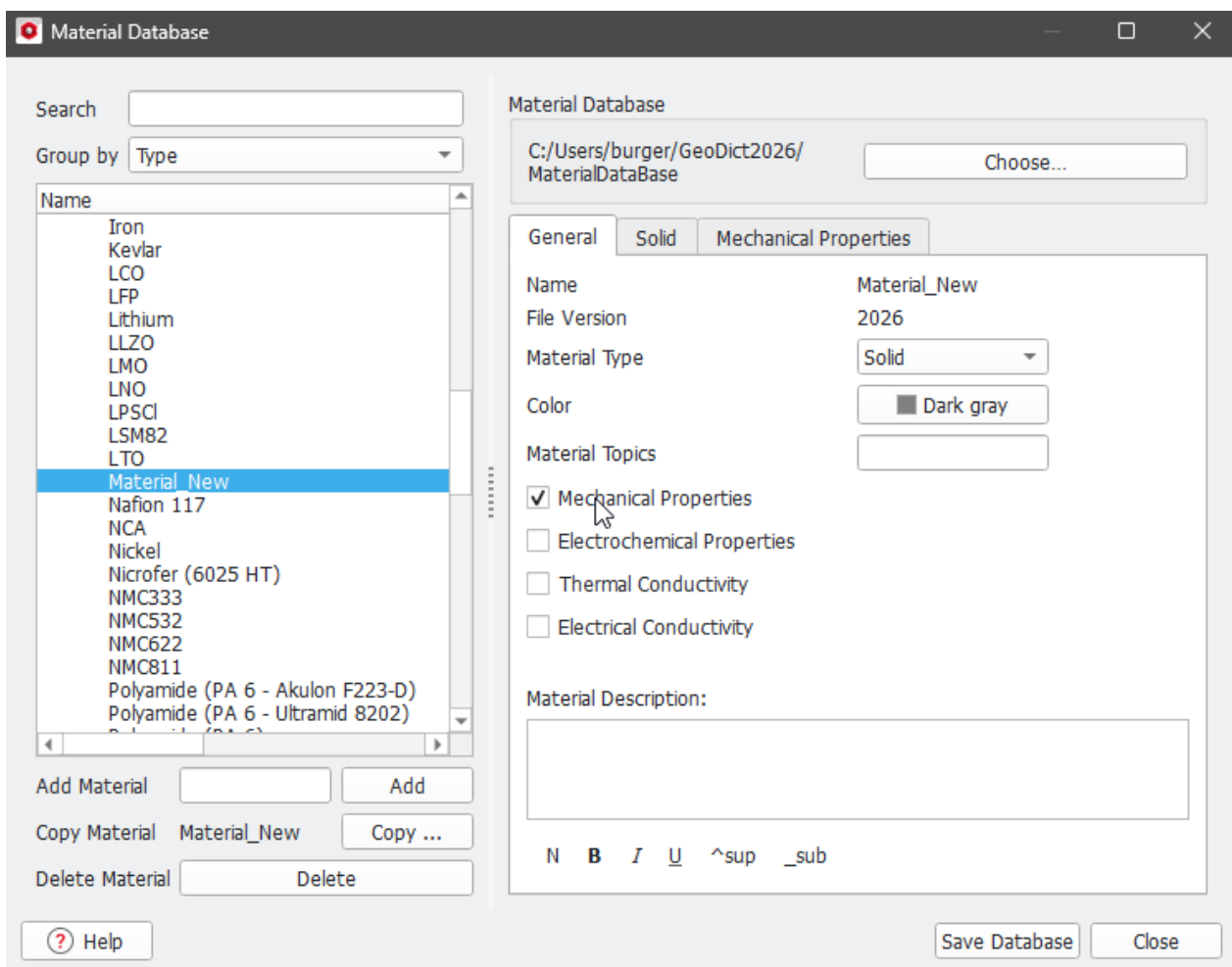
To set up a material law in GeoDict, first open the material data base via **Settings – Open & Edit Material Database ...** in the menu bar:



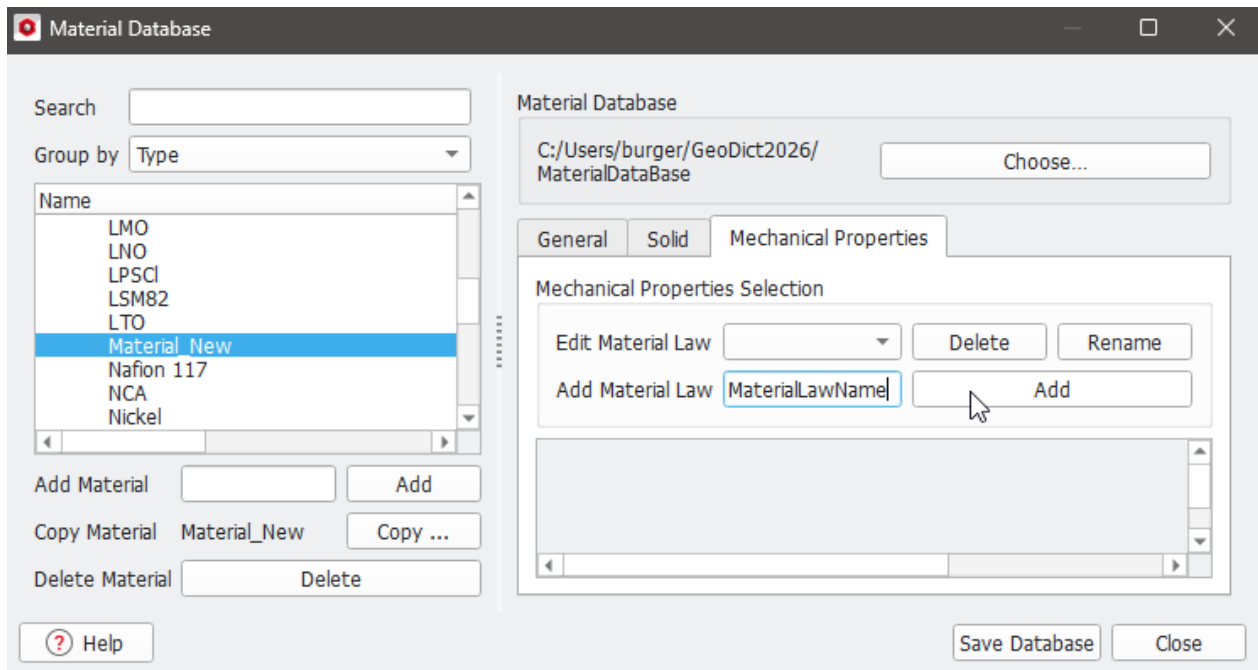
Then, either create a new material by entering a material name and clicking the **Add** button (as shown below), or open one of the materials in the database and add a material law to it.



When creating a new material, make sure that the **Mechanical Properties** are activated under the **General** tab:

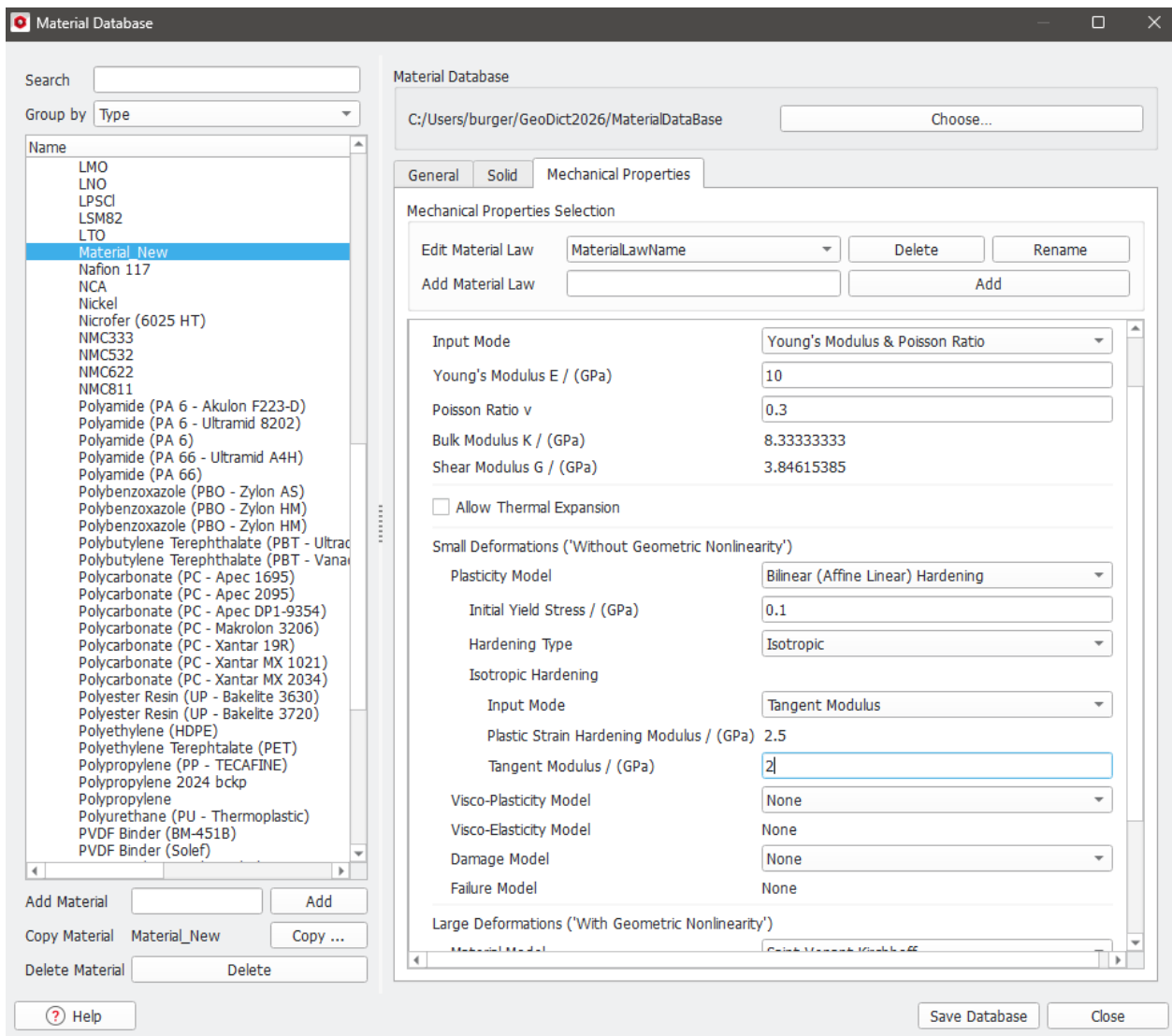


Switch to the **Mechanical Properties** tab. Here, existing material laws can be edited, or new material laws can be added. In the figure below, it is shown how to add a new material law: Enter the material law name in the field **Add Material Law** and click the **Add** button.



Then, the material parameters can be defined after selecting the material law. In the figure below, it is shown how to set up a material with Bilinear Hardening.

Other material laws can be defined by selecting the corresponding fields in the interface. After all material parameters are defined, make sure to save the database with the **Save Database** button in the lower right corner.



1.2.3 Material Fitting for the Internal Material Laws

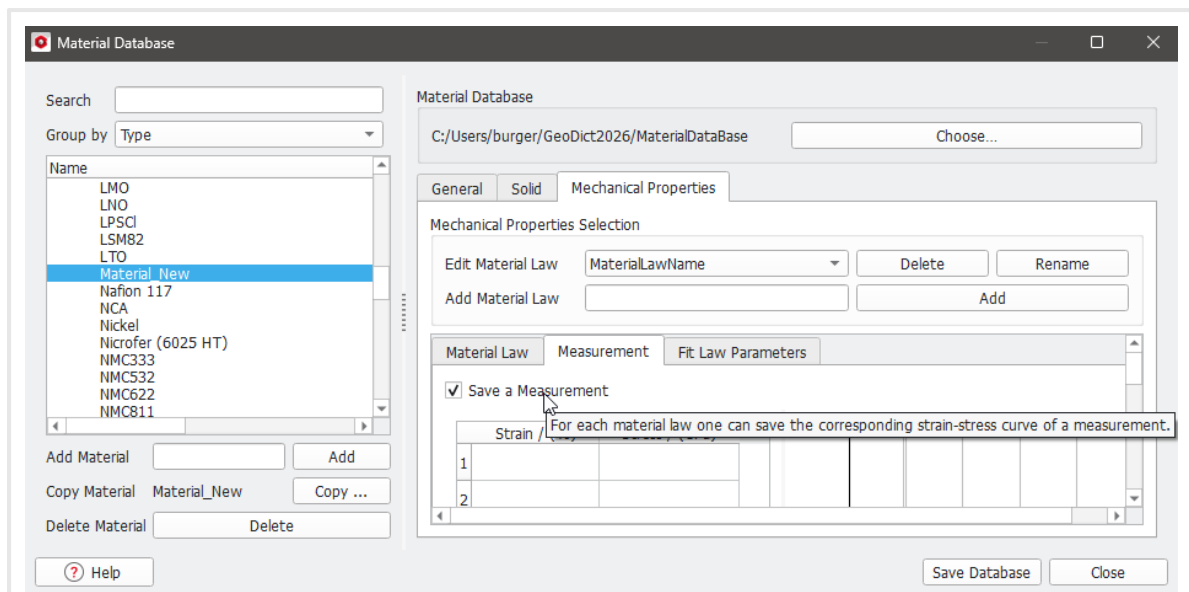
For nonlinear materials, finding the material parameters might be complicated. If measurement data for a given material is available, this data can be used to estimate the material parameters.

Material fitting is available for **Perfect Plasticity**, **Bilinear Hardening**, **Exponential Hardening** and both **Damage** models. For the material fitting, currently no combined material models are available.

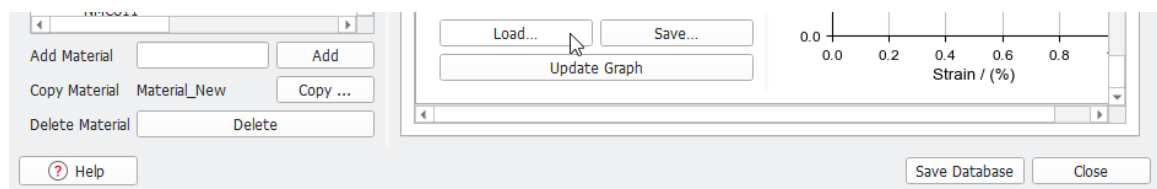
Since GeoDict 2022, the material fitting routines are much faster than in previous versions. Now, the fitting relies on analytic formulas instead of repeated simulations – therefore the fitting finishes in several seconds instead of minutes as before.

✓ Measurement

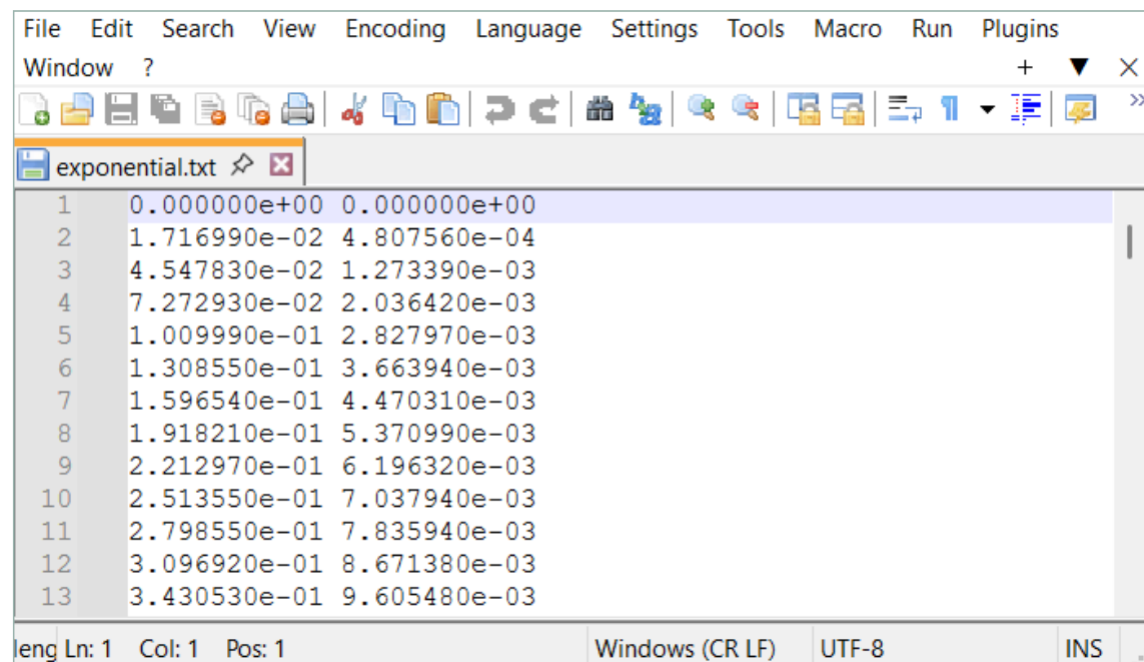
To fit a material law, a measurement must be loaded for the material. For this, check **Save a Measurement** under the **Measurement** tab of the current material law.



Material data can be loaded from a *.txt file with the **Load...** button.

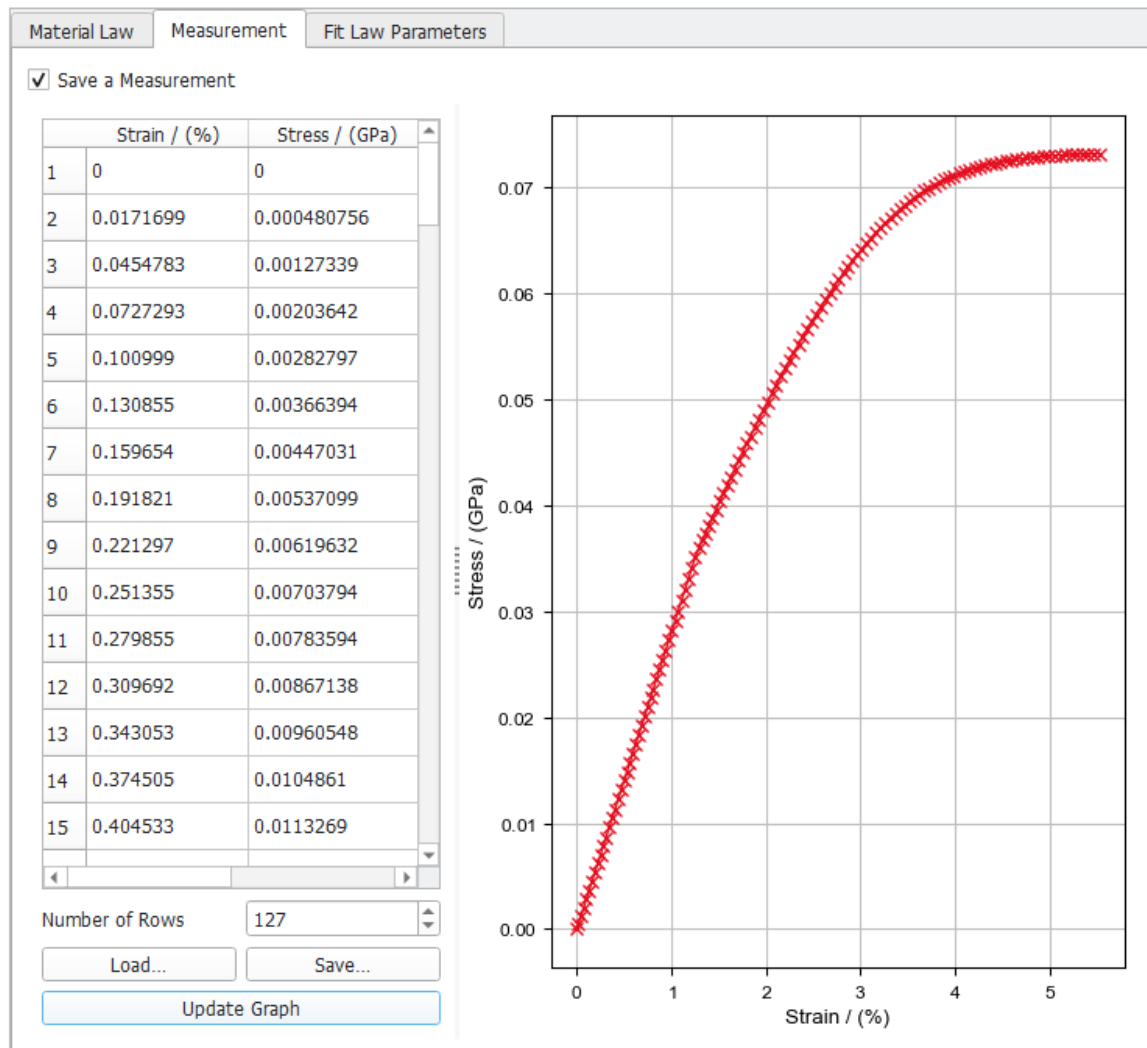


The data must be formatted in two columns, the first column represents the strain values in percent, and the second row contains the corresponding stress values in GPa. Both columns must be separated by spaces.



Alternatively, data can also be entered by copy-and-paste or manually and save the data as a *.txt file with the **Save...** button.

To plot the data, click the **Update Graph** button.



✓ Fit Law Parameters

To fit the material law, select the **Fit Law Parameters** tab under the current material law. Choose a material law from the pulldown menu:

Material Law

Measurement

Fit Law Parameters

Result File Name (*.gdr)

FitMaterialLaw.gdr

Choose Material Law

Plasticity (Iso. Hardening) - Exponential

Estimated Young's Modulus E / (GPa)

2.79999301

☐ Keep Initial Stiffness

☐ Set Initial Parameter Values

Initial Parameters

Plot Initial Curves

Young's Modulus E / (GPa)

2.79999301

Initial Yield Stress / (GPa)

0.01

First Hardening Parameter / (GPa)

0.02

Second Hardening Parameter

65

Stopping Tolerance

0.0001

☐ Restrict Strain Range

Minimum Strain Value / (%)

0

Maximum Strain Value / (%)

1

Run Parameter Fitting

Linear Elastic

Plasticity (Iso. Hardening) - Constant

Plasticity (Iso. Hardening) - Bilinear

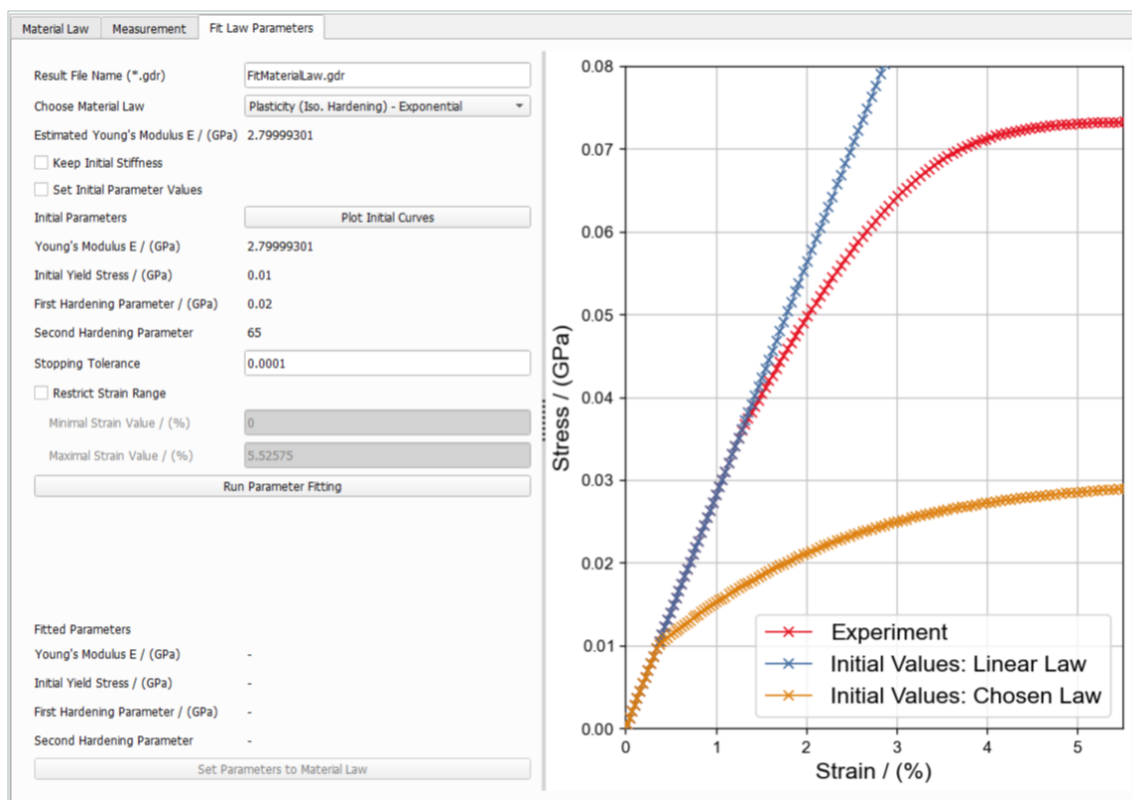
Plasticity (Iso. Hardening) - Exponential

Damage - Exponential

Damage - Mazars

The result is dependent on the start values for the fitting process. Therefore, the initial values can be user defined by checking **Set Initial Parameters** and **Keep Initial Stiffness**.

The quality of the initial values can be checked with **Plot Initial Curves**.



Fitted Parameters

Young's Modulus E / (GPa)

-

Initial Yield Stress / (GPa)

-

First Hardening Parameter / (GPa)

-

Second Hardening Parameter

-

Set Parameters to Material Law

Stress / (GPa)

0.08

0.07

0.06

0.05

0.04

0.03

0.02

0.01

0.00

Strain / (%)

0

1

2

3

4

5

Experiment

Initial Values: Linear Law

Initial Values: Chosen Law

It is recommended to try several combinations of initial values to find good start parameters for the material fitting algorithm. To edit the plot settings, right-click on the plot and choose **Edit Axis Settings**.

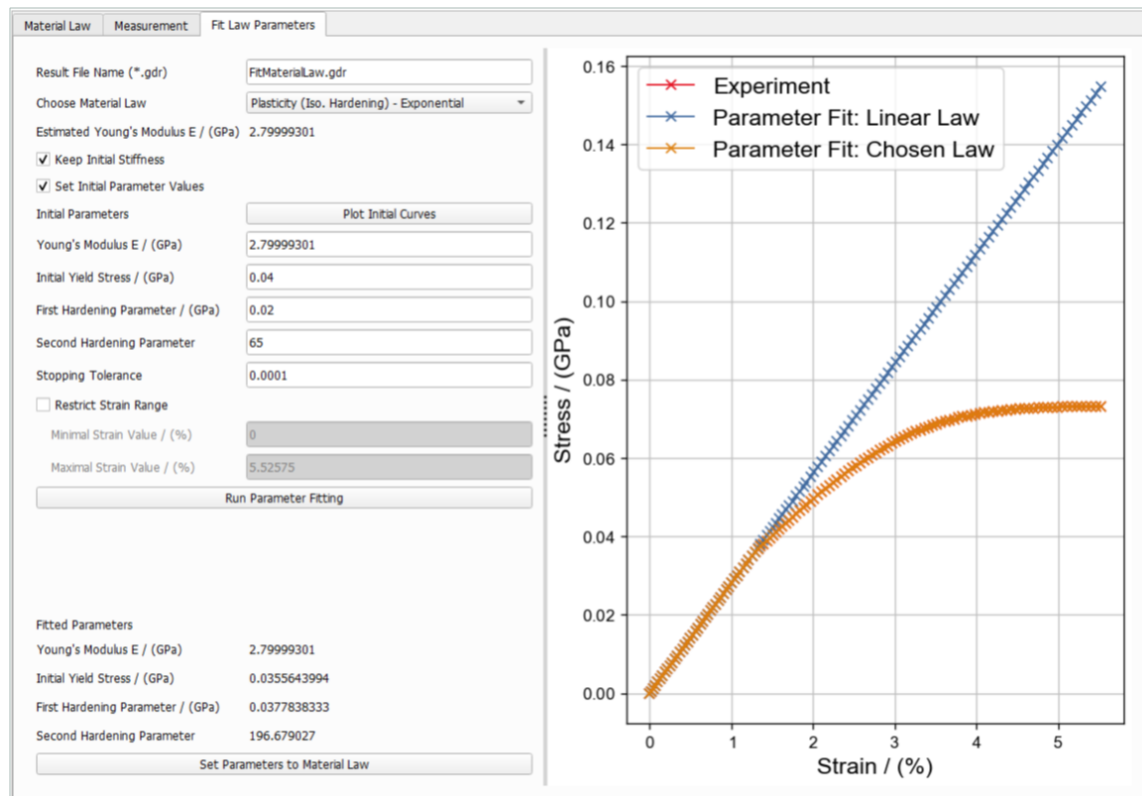
The fitting algorithm stops if the error between the fit and the target (which is the measurement curve defined in the Measurement tab) is smaller than a threshold. This threshold can be defined in the **Stopping Tolerance** field.

If only a part of the measurement data should be used for fitting the material laws, a strain range can be selected with **Restrict Strain Range**. This is especially useful if a part of the data contains effects which are not simulated by the selected material law, e.g., damage effects when fitting a material law for plasticity.

The parameter fitting can be started by clicking **Run Parameter Fitting**. The fitting process will take several seconds, and the results are directly shown in the **Fitted Parameters** section and in the plot.

If the result is not satisfying, the **Initial Parameters** can be adjusted and the parameter fitting process restarted. Another way of improving the result is to reduce the **Stopping Tolerance** and force the fitting algorithm to search longer for a solution.

In the presented example the Initial Yield Stress has to be changed from 0.01 GPa to 0.04 GPa to achieve a good fitting result. In the figure below, you can see that the curve for the exponential plasticity model lays very well over the measurement data after the parameter fit.



With the button **Set Parameters to Material Law**, the fitted parameters are copied to the material law under the **Material Law** tab.

Material Law
Measurement
Fit Law Parameters

Type
Isotropic

Input Mode
Young's Modulus & Poisson Ratio

Young's Modulus E / (GPa)
2.79999301

Poisson Ratio ν
0

Bulk Modulus K / (GPa)
0.933331004

Shear Modulus G / (GPa)
1.39999651

☐ Allow Thermal Expansion

Small Deformations ('Without Geometric Nonlinearity')

Plasticity Model
Exponential Hardening

Initial Yield Stress / (GPa)
0.0355643994

Hardening Type
Isotropic

Isotropic Hardening

First Hardening Parameter / (GPa)
0.0377838333

Second Hardening Parameter
196.679027

Visco-Plasticity Model
None

Visco-Elasticity Model
None

Damage Model
None

Failure Model
None

Finally, save the changes to the material database with the **Save Database** button. Now, the new material law is saved and can be used in ElastoDict.

ElastoDict – Deformations

GEO DICT

Result File Name (*.gdr)DeformationResult.gdr

Constituent Materials
Macroscopic Load Case
Geometry Handling
Solver
Output
Equations & References

Open & Edit Material

+
Material

ID
Name

00
Material_New...

01
Glass...

Mechanical Prop.

Material Law

MaterialLawName

Elastic-Failure (E-Glass)

E / (GPa)
 ν
G / (GPa)
 α / (1/K)
Plasticity
Visco-Pla

Isotropic
2.79999
0
-
-
☒ Exponential
None

Isotropic
72
0.22
-
5e-06
None
None

Now you can select the material law (here MaterialLawName) of the newly created material Material_New. In order to use the nonlinear material model, please make sure to check the according box (here: Exponential under **Plasticity**).

1.2.4 Using UMATs for Custom Material Laws

You can program your own user-defined material laws with the UMAT interface in GeoDict.

UMATs are a concept from the FEA software Abaqus, and the interface is compatible with GeoDict. This means that you can directly use your already existing user materials in GeoDict. To get familiar with the syntax, some UMAT files are delivered with GeoDict. UMATs are Fortran files (*.f or *.f90), but in GeoDict it is also possible to directly use the compiled files (*.so). Nevertheless, *.so files from GeoDict 2020 and before cannot be used in GeoDict 2021 and later, since the required compiler version changed. Additionally, they might be incompatible between different computers. Therefore, it is important to always keep the source files (*.f or *.f90).

However, if a material can be defined by using the [internal material laws](#)^[9], then we recommend using those instead of UMATs, since they are deeper integrated into GeoDict and easier to set up.

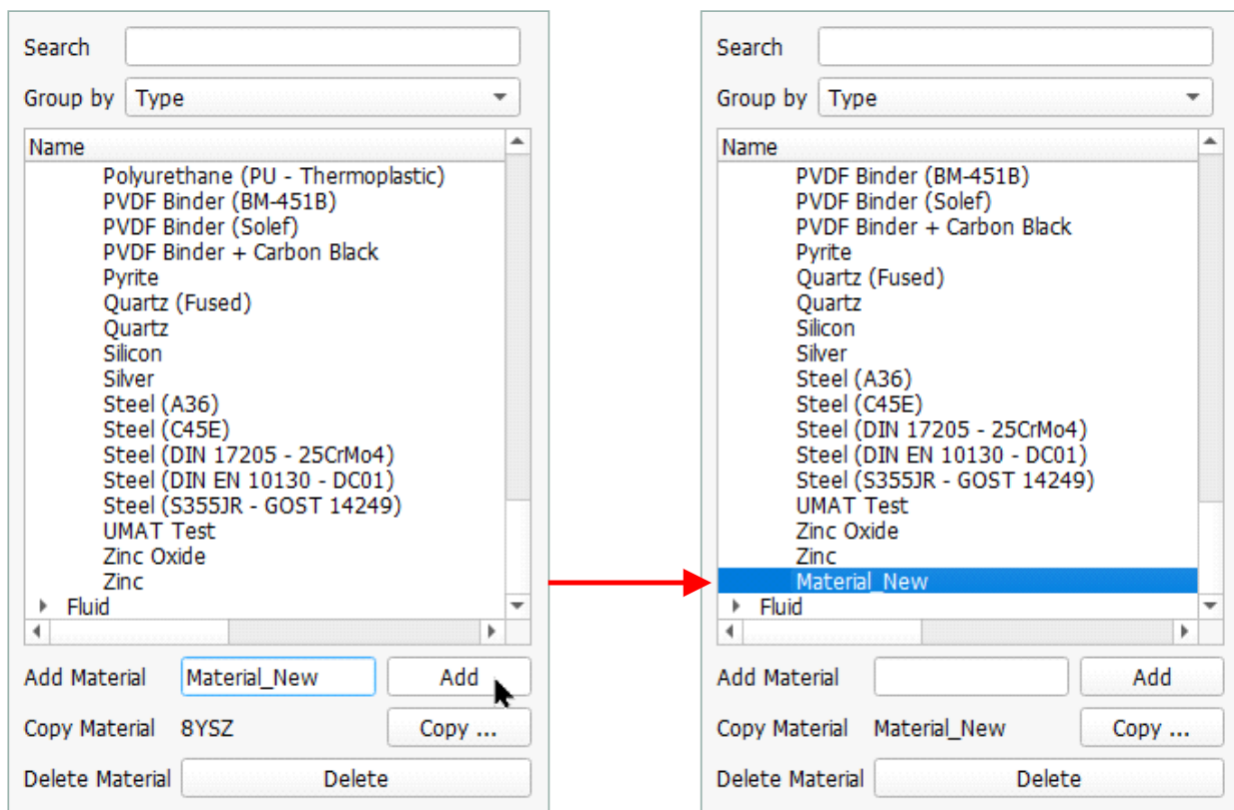
Learn more about using UMATs in the examples:

- [Example: Setting up a Material Law for Damage](#)^[29]
This section describes how to set up a material law using an UMAT.
- [Example: Plastic Deformation](#)^[37]
Another UMAT included with GeoDict is used to simulate plastic deformation.

Example: Setting up a Material Law for Damage

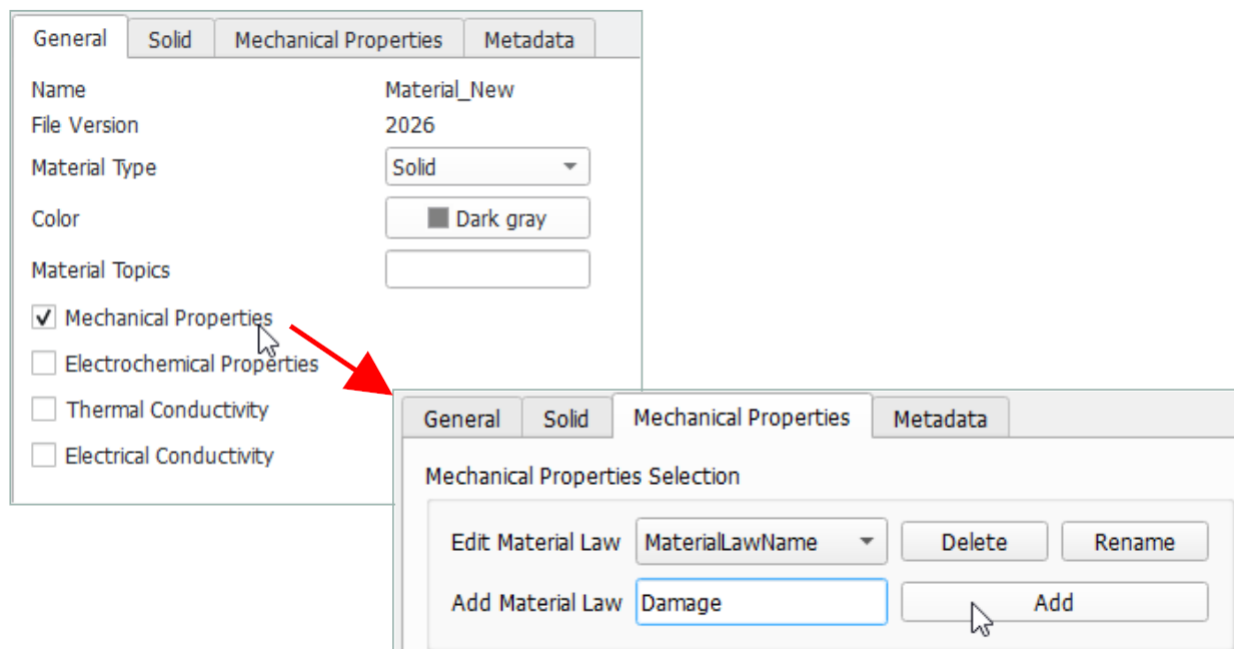
This section describes how to set up a material law using an UMAT.

First, you have to open the Material Database and create a new material, according to the steps described under [Setting up the internal material laws](#)^[20]



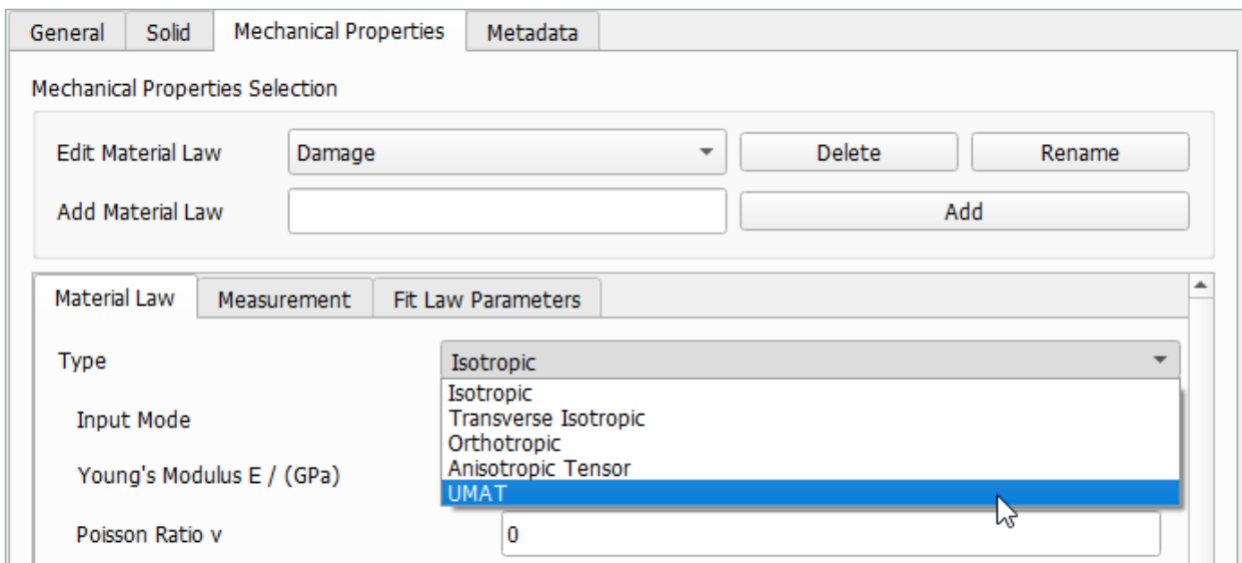
Under the **General** tab, check **Mechanical Properties** for the newly created material.

Under the newly appearing **Mechanical** tab, inside the **Mechanical Properties Selection** panel, enter **Damage** in the **Add Material Law** box, and click **Add**.



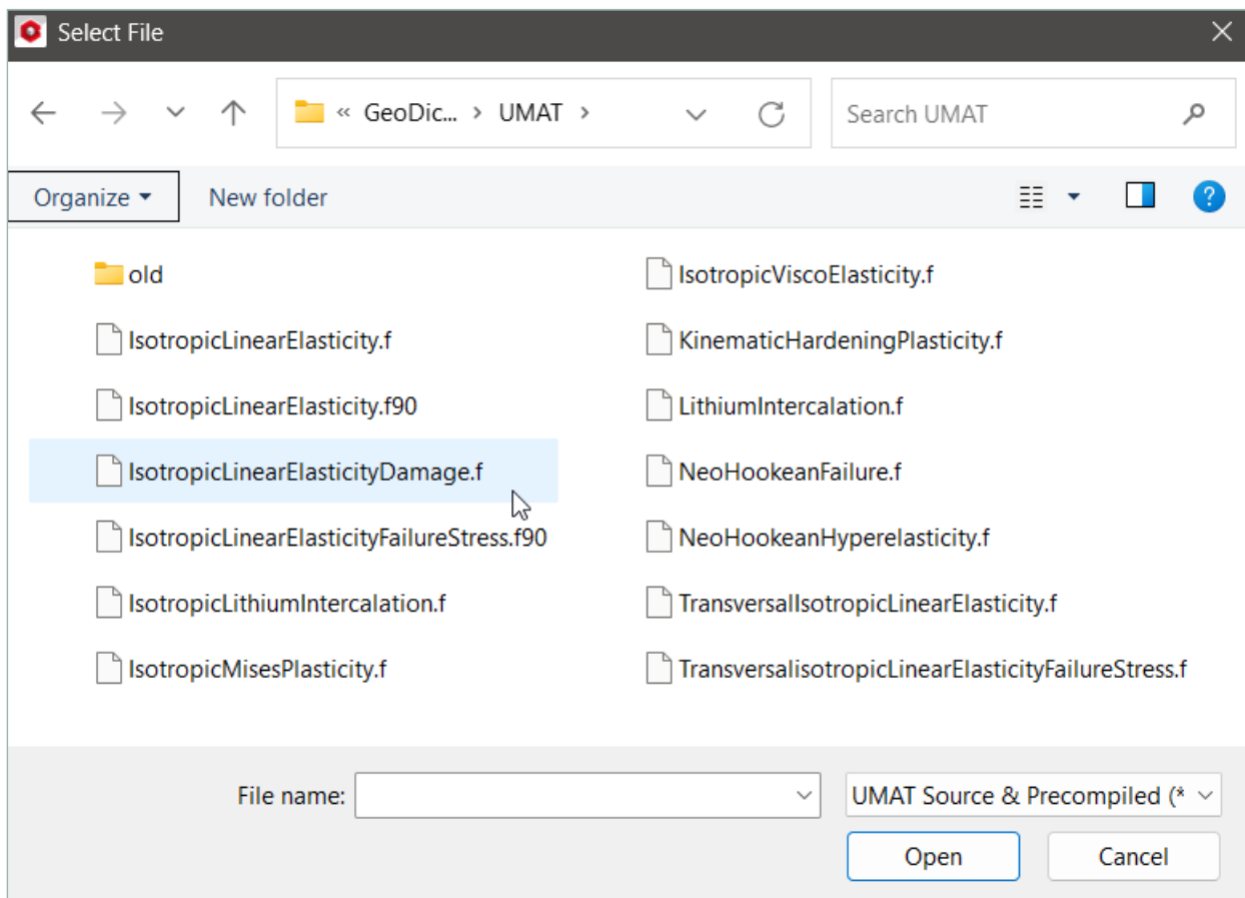
The parameters needed to define the mechanical properties of the new material appear under the panel grouped under the **Law** tab.

From the **Type** pull-down menu, select **UMAT**.



After selecting **UMAT**, a different panel opens below the **Type** pull-down menu, with the **UMAT Parameters**, **Material Parameters**, and **State Variables** tabs.

Under the UMAT Parameters tab, click **Browse** to find the UMATs for GeoDict 2026 in your user folder. Select the file `IsotropicLinearElasticityDamage.f` UMAT and click **Open**.



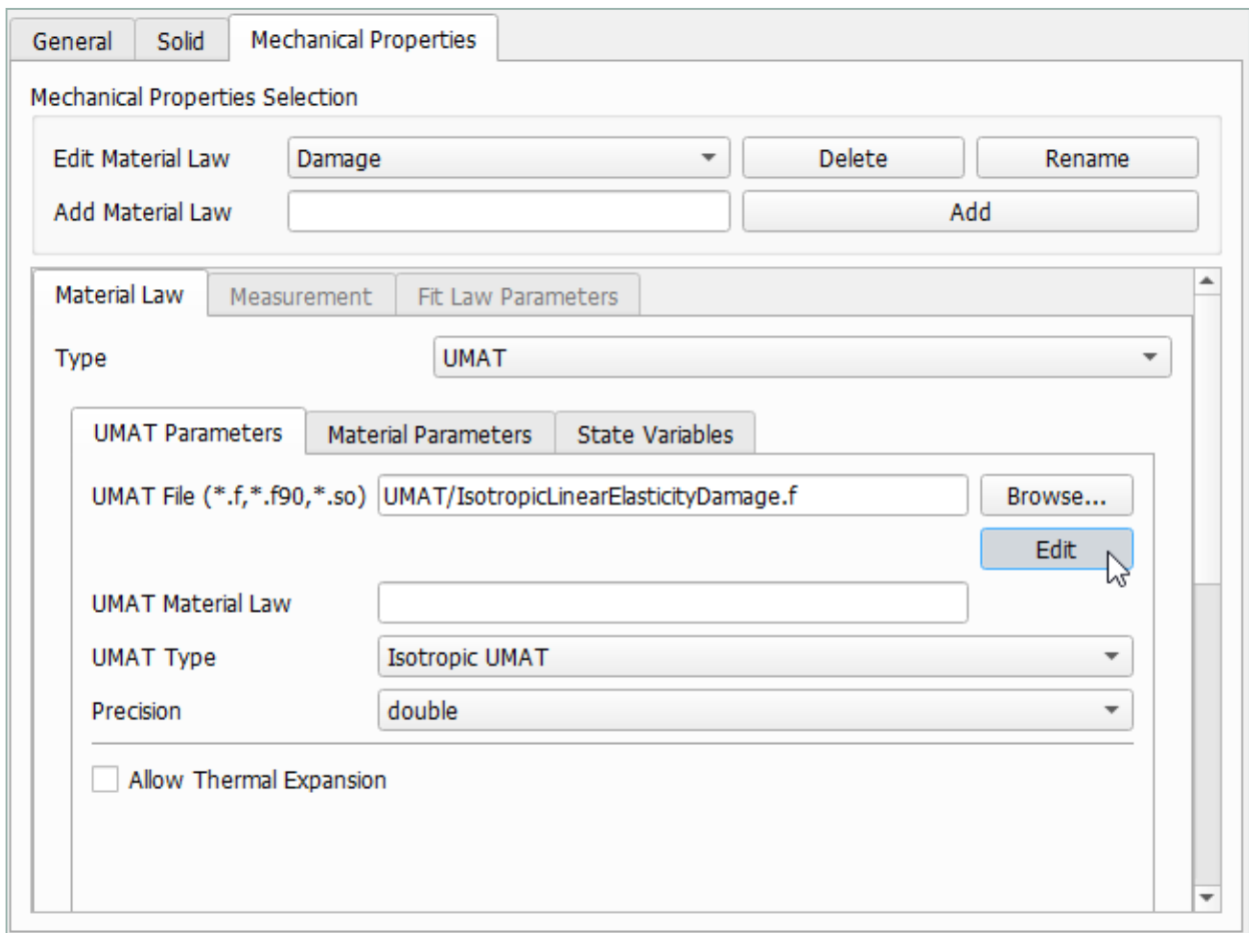
Alternatively, enter a path to the UMAT file relative to the user folder. For this example, this path would be `UMAT/IsotropicLinearElasticityDamage.f`. Relative paths have the advantage

that they work independently from your username. Therefore, material database entries can be shared easily between different PCs, as long as the UMAT file is present in the UMAT folder.

The screenshot shows the 'Mechanical Properties Selection' dialog box in ElastoDict. It has three tabs: 'General', 'Solid', and 'Mechanical Properties', with 'Mechanical Properties' being the active tab. Inside, there's a 'Mechanical Properties Selection' section with 'Edit Material Law' (set to 'Damage'), 'Delete', 'Rename', and 'Add Material Law' (with an 'Add' button). Below this is the 'Material Law' section with 'Measurement' and 'Fit Law Parameters' tabs. The 'Type' is set to 'UMAT'. Under 'UMAT Parameters', there are three sub-tabs: 'UMAT Parameters' (active), 'Material Parameters', and 'State Variables'. The 'UMAT File (*.f, *.f90, *.so)' is set to 'UMAT/IsotropicLinearElasticityDamage.f', with 'Browse...' and 'Edit' buttons. Other fields include 'UMAT Material Law' (empty), 'UMAT Type' (set to 'Isotropic UMAT'), and 'Precision' (set to 'double'). At the bottom, there is a checkbox for 'Allow Thermal Expansion' which is unchecked.

For the material law (defined in the UMAT) to be applied to the **Material_New**, some material parameters must be entered under the **Material Parameters** tab.

These material parameters, that the particular UMAT requires, can be found by opening the UMAT source file, if available (*.f or *.f90). To do so, simply click **Edit** under the **UMAT Parameters** tab and open it with a text editor (for example, with NotePad++).



In the opened file, the text shows that applying the *IsotropicLinearElasticityDamage.f* UMAT to **Material_New** requires assigning the following material parameters or properties (PROPS):

- Young's Modulus (E)
- Poisson's Ratio (NU)
- Damage Hardening Modulus (DAMHARD)
- Initial Threshold for the equivalent strain (DAMAGE0).

```

37      C      -----
38      C      PROPS (1) - E
39      C      PROPS (2) - NU
40      C      PROPS (3) - DAMHARD DAMAGE HARDENING MODULUS
41      C      PROPS (4) - DAMAGE0 INITIAL DAMAGE THRESHOLD
42      C      -----
43      C
44      C      ELASTIC PROPERTIES
45      C
46      C      EMOD=PROPS (1)
47      C      ENU=MIN (PROPS (2) , ENUMAX)
48      C      DAMHARD=PROPS (3)
49      C      DAMAGE0=PROPS (4)
50      C      EBULK3=EMOD/ (ONE-TWO*ENU)
51      C      EG2=EMOD/ (ONE+ENU)
52      C      EG=EG2/TWO
53      C      EG3=THREE*EG
54      C      ELAM= (EBULK3-EG2) /THREE
55      C      FAILSTR=1.0-1.0/WEAKFAC

```

length : 4,640 lines : 164 Ln : 17 Col : 1 Pos : 640 Windows (CR LF) UTF-8 INS

Therefore, in the table under the **Material Parameters** tab, manually enter the needed parameters (Material Parameters), their values for **Material_New** (Value), and a brief parameter explanation (Description). For example, as follows:

General Solid Mechanical Properties Metadata

Mechanical Properties Selection

Edit Material Law: Damage [Delete] [Rename]

Add Material Law: [Add]

Material Law: Measurement Fit Law Parameters

Type: UMAT

UMAT Parameters Material Parameters State Variables

	Material Parameter	Value	Description
1	E	4.23	Young's Modulus
2	nu	0.3	Poisson Ratio
3	DamageHard	20	Damage Hardening ...
4	DamageThreshold	0.1	Initial Threshold for the minimal...

[Save Database] [Cancel]

The damage model defined in the *IsotropicLinearElasticityDamage.f* UMAT depends on the equivalent strain, which is calculated as

$$\epsilon_{eq} = \sqrt{\frac{1}{3} \left(\frac{E}{1-2\nu} - \frac{E}{1+\nu} \right) (\epsilon_{xx} + \epsilon_{yy} + \epsilon_{zz})^2 + \frac{E}{1+\nu} \left[\epsilon_{xx}^2 + \epsilon_{yy}^2 + \epsilon_{zz}^2 + \frac{1}{4} (\epsilon_{xy}^2 + \epsilon_{yz}^2 + \epsilon_{xz}^2) \right]} \quad (11)$$

The degradation begins as soon as the minimal equivalent strain for damage d_0 is reached:

$$D = 1 - e^{d_1(d_0 - \epsilon_{eq})}, \epsilon_{eq} > d_0 \quad (12)$$

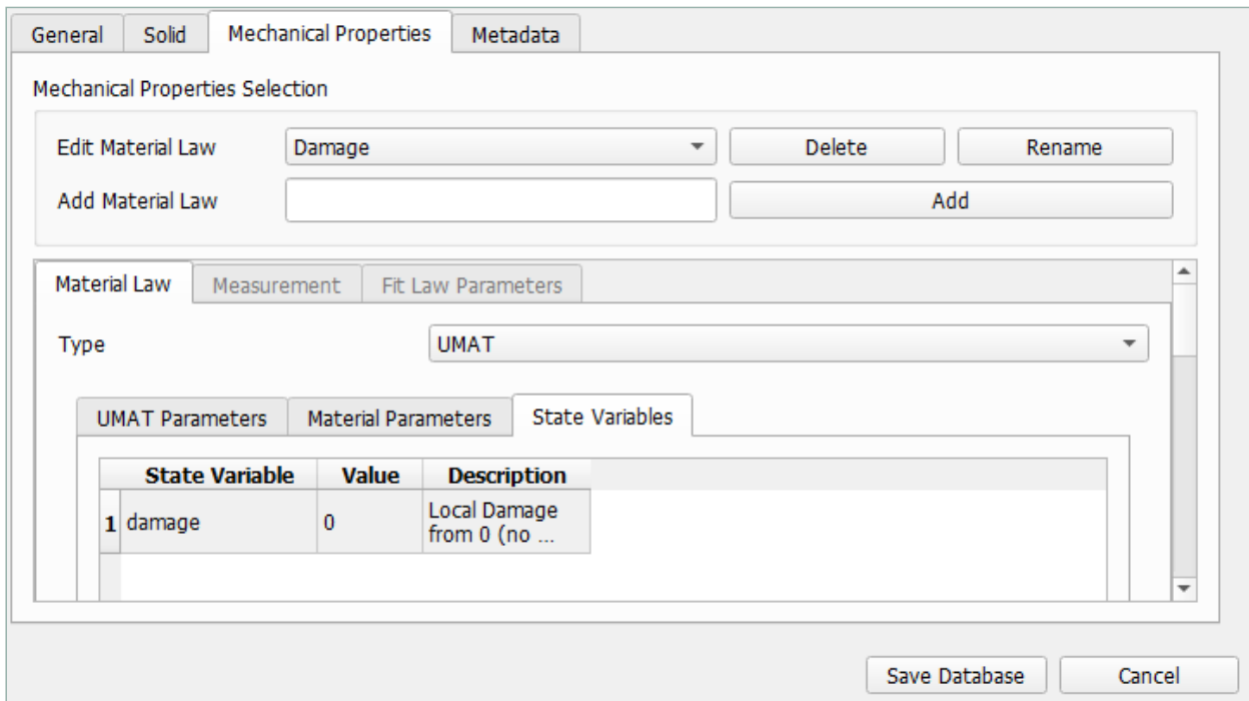
with d_0 : minimal equivalent strain for damage, d_1 : damage hardening modulus.

Next, go to the **State Variables** tab. State variables are internal variables of the UMAT, they store information which is used in the next iteration. When the State Variables are not properly defined, the UMAT does not work as intended, or it doesn't work at all.

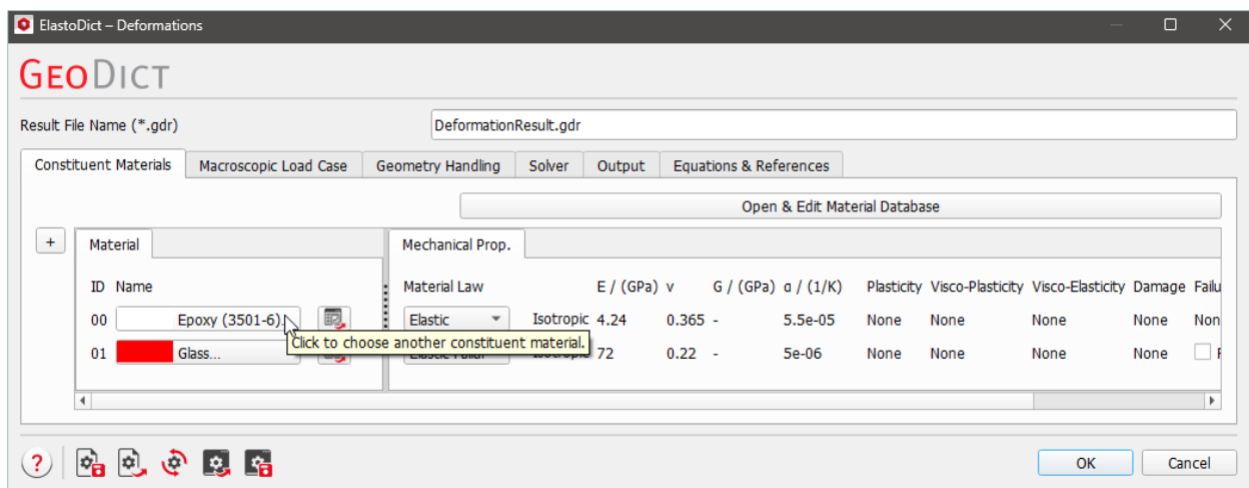
For the UMAT *IsotropicLinearElasticityDamage.f*, there is only one state variable which stores the damage value. It must be defined and initialized with 0, since there is no initial damage in the material. The number of state variables depends on the particular UMAT

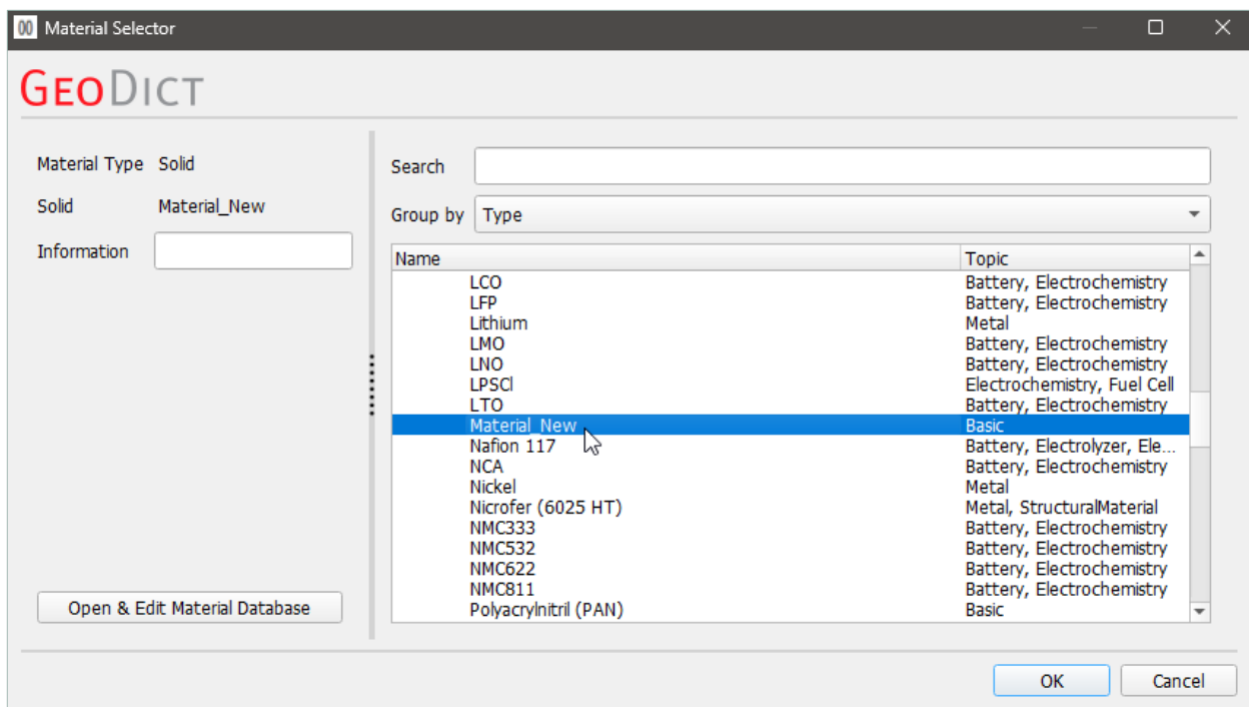
and is not always obvious. If you need more information about UMATs delivered with GeoDict, ask the Math2Market support team for help.

Finally, click **Save Database** at the bottom right of the **Edit Material Data-Base** dialog.

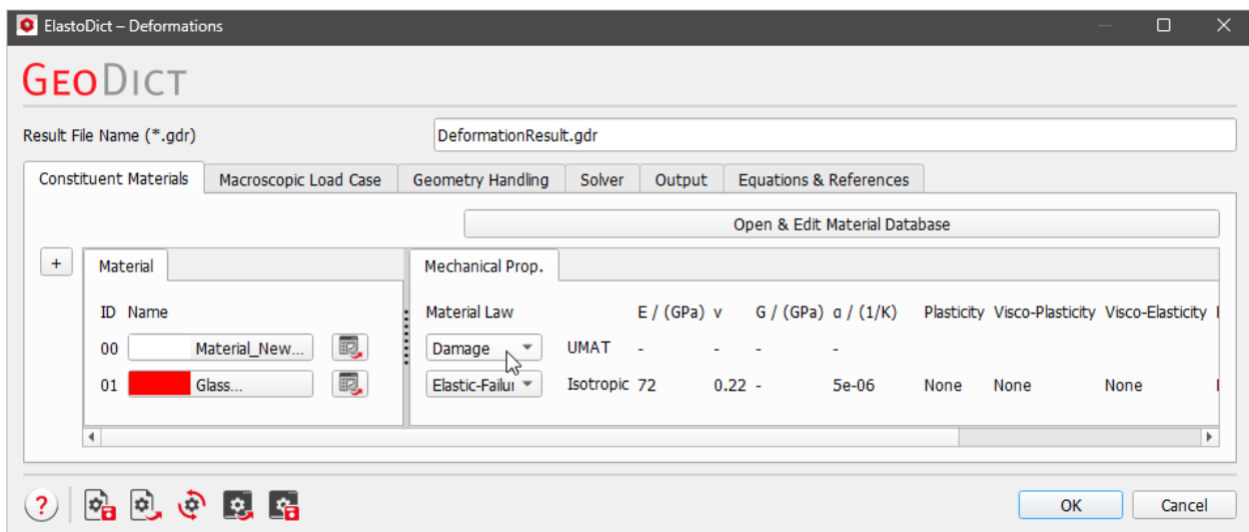


After saving, the newly defined material **Material_New** can be chosen as constituent material when using ElastoDict.





Make sure that the **Damage** material model is selected under Mechanical Properties tab.



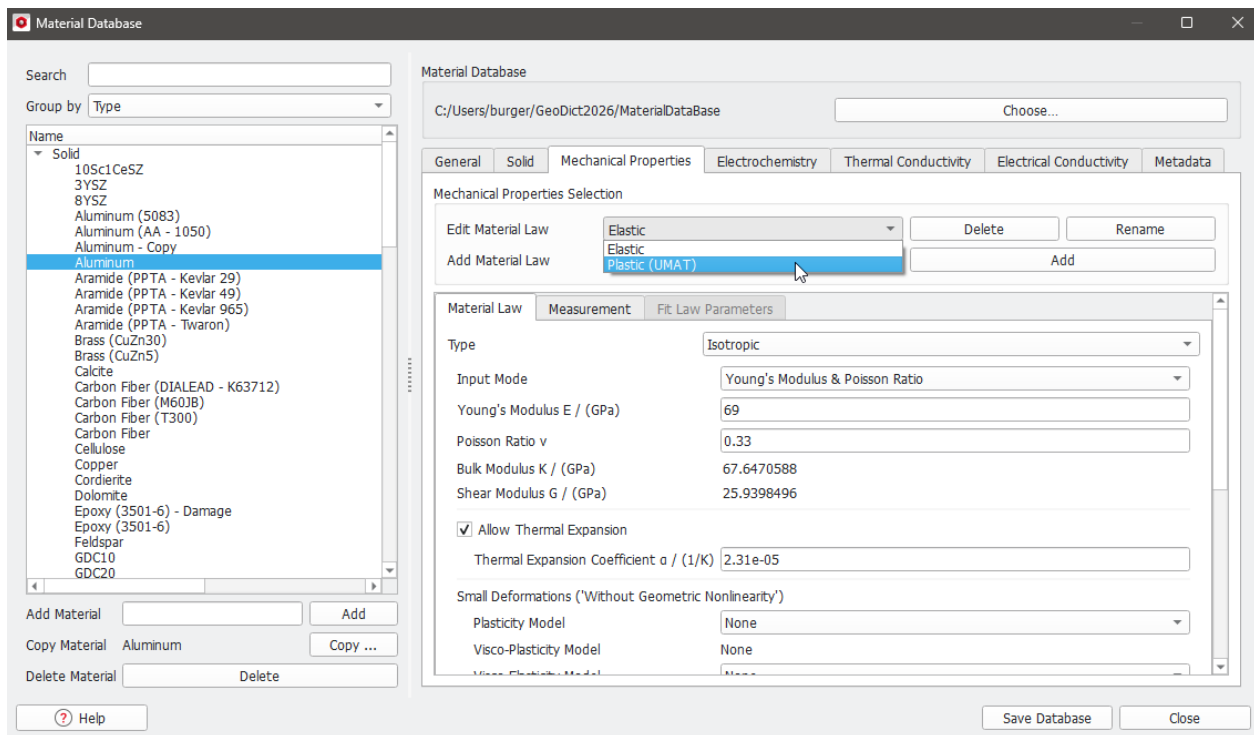
Now, the material law assigned to **Material_New** is applied in the ElastoDict calculations.

Example: Plastic Deformation

This UMAT included with GeoDict is used to simulate plastic deformation.

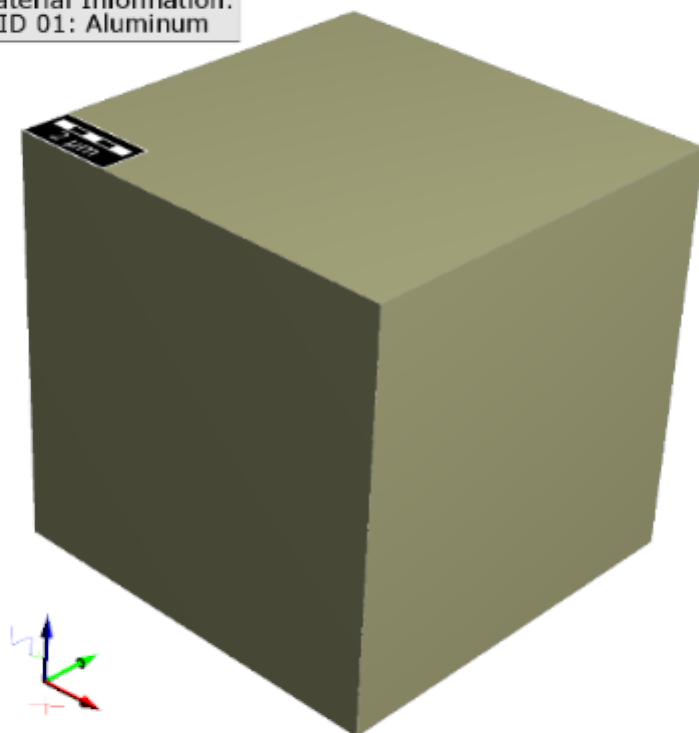
In the GeoDict Material Database, the UMAT IsotropicMisesPlasticity.f is contained in the database entry for **Aluminum** and appropriate parameters are already defined when choosing **Plastic (UMAT)** and **Type - UMAT** under the **Law** tab.

Besides the linear parameters (Young's Modulus E ; Poisson Ratio ν), it includes a piecewise linear hardening curve that was entered beforehand under the **Mechanical Properties – Material Law - Material Parameters** tab.



The plastic deformation of aluminum can be simulated in GeoDict using a solid block (generated through **Model** → **ProcessGeo** → **Empty & Invert Structure** → **Create Empty Structure**, 1x1x1 voxels, and then, **Model** → **ProcessGeo** → **Empty & Invert Structure** → **Invert Structure**).

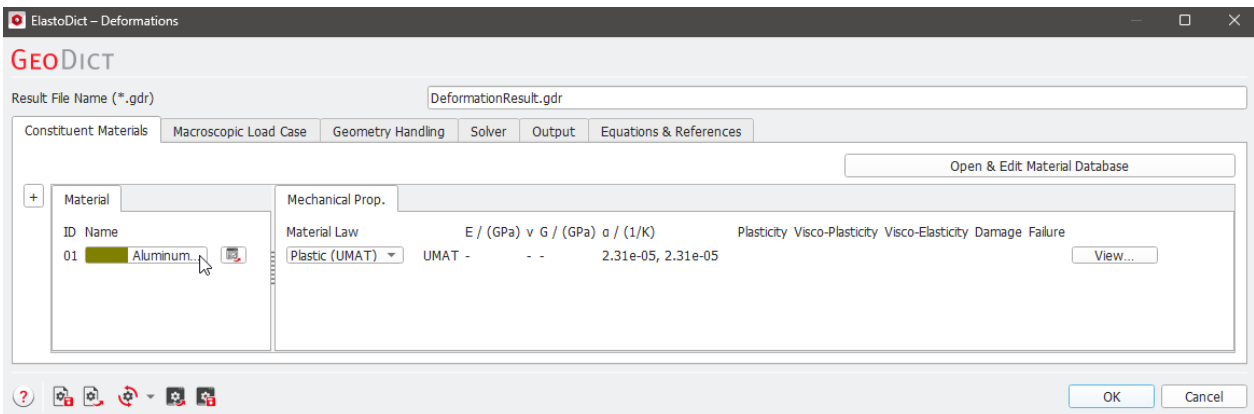
Material Information:
ID 01: Aluminum





Note! Generally, it is recommended to check the material behavior on small structures. If only a single constituent material must be investigated, a structure consisting of a single voxel is already enough.

In the menu bar, select **Settings** → **Select Constituent Materials**. You can choose the material of the voxel by clicking on the button next to the ID number under the **Material** tab (see screenshot below). In the **Material Selector** dialog, select **Aluminum** from the list of available (Solid) constituent materials.



The color of the constituent material (Aluminum) may be changed via **Settings** → **Color & Visibility Settings**.

The plastic deformation can be examined during a cyclic stress experiment which can be set up in ElastoDict. In the menu bar, select **Predict** → **ElastoDict**.

In the ElastoDict section, select **Deformations** and click the **Edit...** button. In the **Deformations** dialog, choose to apply the load in Z-direction in a **Uniaxial Experiment-Tensile**, with **Confined** boundary conditions in tangential direction. Choose the **Load Type** as **Force Controlled**.

To define a cyclic load, use the **Predefined Shape** option to the right of the **Load Table** (screenshot below).

Choose the **Shape** as **Sawtooth** and the settings as shown in the screenshot.

By clicking **Apply**, the settings for the predefined shape are applied and the load table is defined automatically.

ElastoDict – Deformations

GEODICT

Result File Name (*.gdr)

Constituent Materials | **Macroscopic Load Case** | Geometry Handling | Solver | Output | Equations & References

Experiment: Experiment Conditions

Load Type: In Tangential Direction:

Load Case: Direction: X-Direction: Y-Direction:

Plane: Fluid Pressure: Pressure Mode: Fluid Pressure / (GPa):

Angle in Plane / (°):

Load Table | **Load Graph**

	Time / (s)	Stress / (GPa)	Temp. Change / (K)
1	0.5	0.18	0.0
2	1	0.36	0.0
3	1.5	0.54	0.0
4	2	0.72	0.0
5	2.5	0.9	0.0
6	3	1.08	0.0
7	3.5	1.26	0.0

Number of Rows:

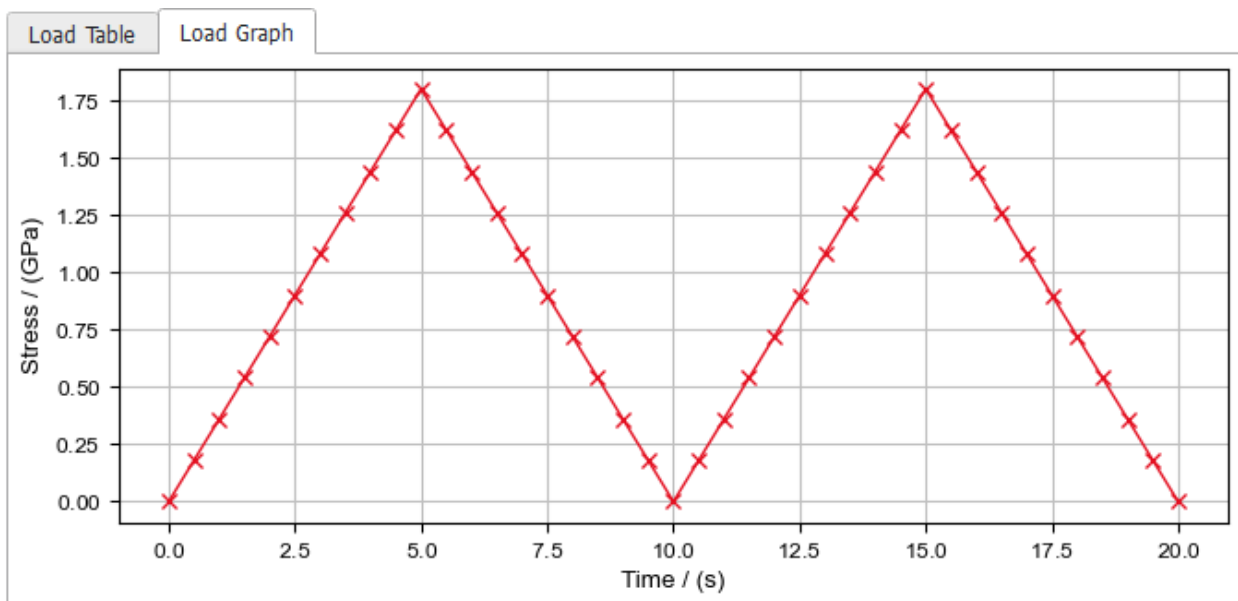
Load... Save...

Predefined Shape: Shape: Repetitions: Amplitudes Max. / (GPa): Amplitudes Min. / (GPa): Ascending Length / (s): Descending Length / (s): Steps per Repetition: Apply

Boundary Conditions: ☒ Periodic ☐ Symmetric ☐ Mixed

OK Cancel

The graph of the load looks as follows:



Keep the default settings under the **Geometry Handling** tab.

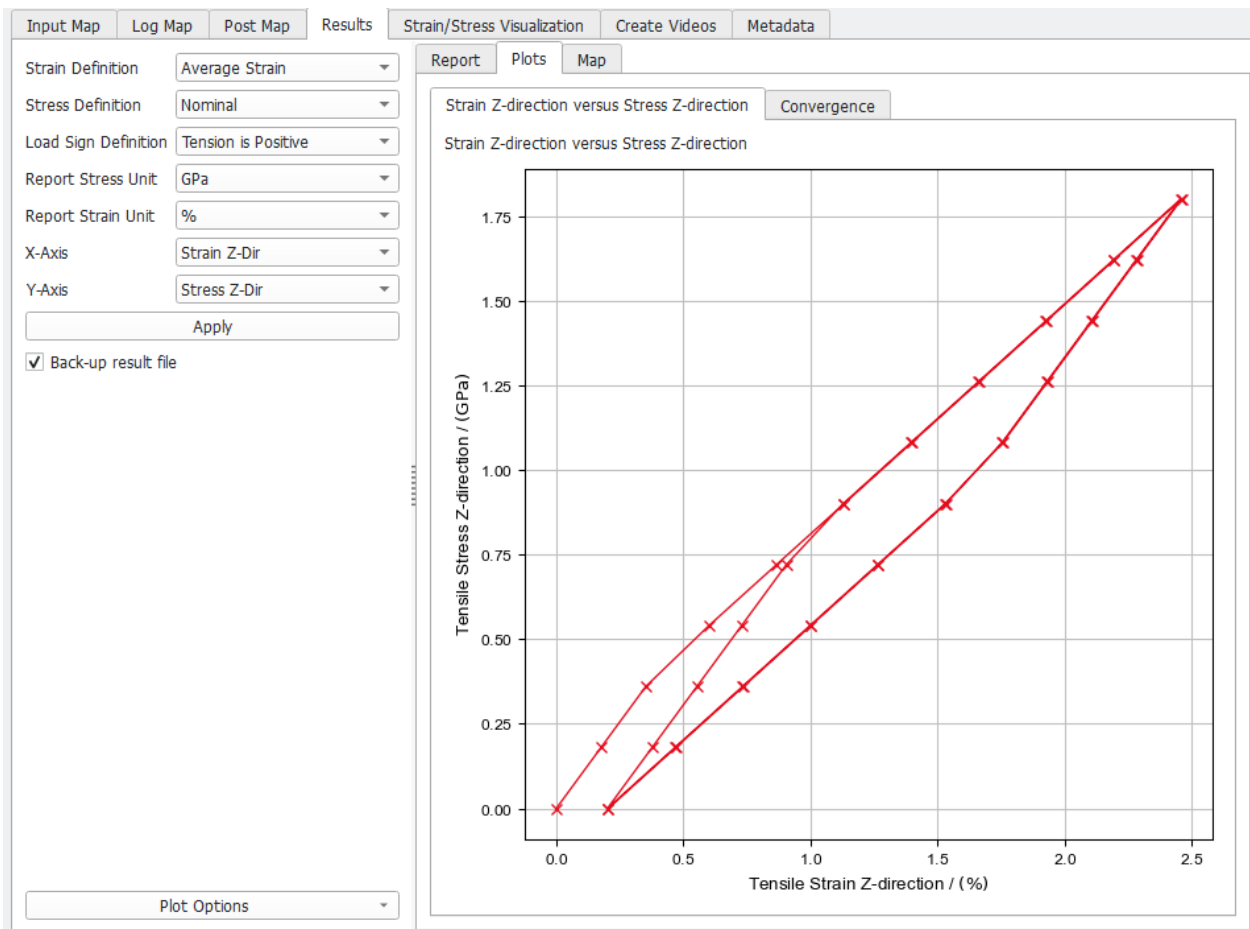
Under the **Solver** tab, keep the default settings as well: Before GeoDict 2022, it was necessary to select the **Memory Efficient (Neumann Series)** when using nonlinear material laws, but this is no longer necessary.

The screenshot shows the 'Solver' tab in the GeoDict software interface. The tabs at the top are 'Constituent Materials', 'Macroscopic Load Case', 'Geometry Handling', 'Solver' (selected), 'Output', and 'Equations & References'. The 'Simulation Stopping Criterion' section includes a 'Tolerance' input field set to '0.0001', and checkboxes for 'Maximum Iterations' (set to 1000) and 'Maximum Run Time / (h)' (set to 240). The 'Method' dropdown is set to 'Intermediate (Memory Efficient Conjugate Gradient)'. The 'Parallelization' section shows '20 Threads (Automatic Maximum)' with an 'Edit...' button. There are checkboxes for 'Use Downsampling' and 'Write Deformation Data to File (slower but less memory)'. The 'Downsampling Factor' is set to 2, and 'Composite Voxels' is set to 'Laminate Theory'. The 'Orientation Mode for Anisotropic Materials' dropdown is set to 'Use Orientation from Analytic Objects (gad)'.

Under the **Output** tab, de-select **Write Deformed Geometry**. In this example, the simulation is done on a structure with only one voxel, and then, the deformed geometry does not provide useful information.

After running the ElastoDict - **Deformations** simulation, the Result Viewer of the result file (.gdr) opens at the **Results** tab. Under the **Results - Plots** subtab, choose to plot the Strain in Z-Direction (%) in the X-Axis against the resulting Stress in Z-Direction (GPa) in the Y-Axis, and click **Apply....**

Observe that the strain does not return to zero after the first load cycle and the plastic deformation remains in the material.



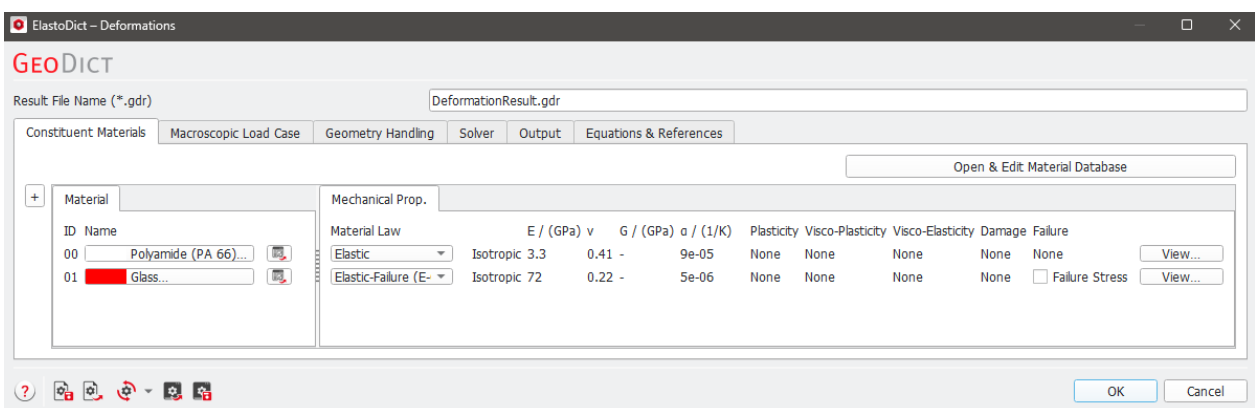
1.3 Deformations

ElastoDict-Deformations allows to simulate different types of deformation under user-defined loads, including damage, failure, plasticity and viscous effects.

For linear elastic materials, new material contacts in the structure can be detected during the simulation. Several nonlinear material models are available, and additionally, UMATs compatible with Abaqus are supported.

Results of the simulation are, e.g., a stress-strain curve and local information on the regions where damage sets in and the material ends up failing.

In the **ElastoDict-Deformations Options** dialog, choose a **Result File Name (*.gdr)** according to your current project.



Deformation Command

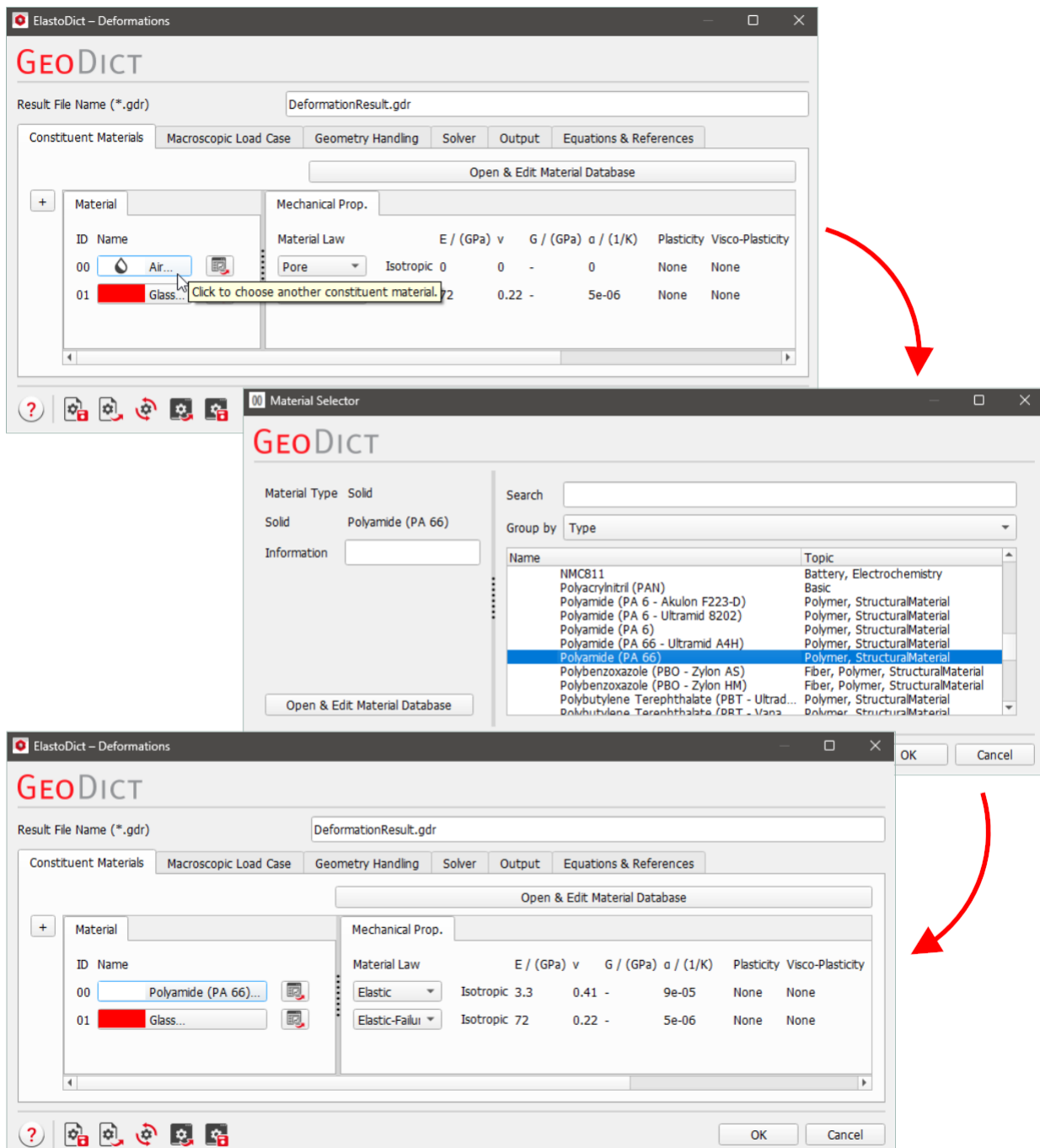
- [Constituent Materials](#)⁴⁴
How to use and setup material laws for ElastoDict - Deformation.
- [Macroscopic Load Case](#)⁴⁶
Description of different load case options, concluded with an example.
- [Geometry Handling](#)⁵⁵
Information about how geometry and its deformation is handled in ElastoDict.
- [Solver](#)⁶³
Description of the solver options.
- [Output](#)⁶⁸
Description of the output options.
- [Result File](#)⁷⁰
Description of the result file generated by an ElastoDict - Deformations simulation step.
- [Restarting a simulation](#)⁸⁵
Information about restart possibilities.
- [Example: Matrix Damage](#)⁸⁶
Matrix Damage Example

1.3.1 Constituent Materials

The materials in the structure model need to be defined under the **Constituent Materials** tab, so that their mechanical parameters, such as the Young's modulus (E) and the Poisson's ratio (ν), are taken into account.

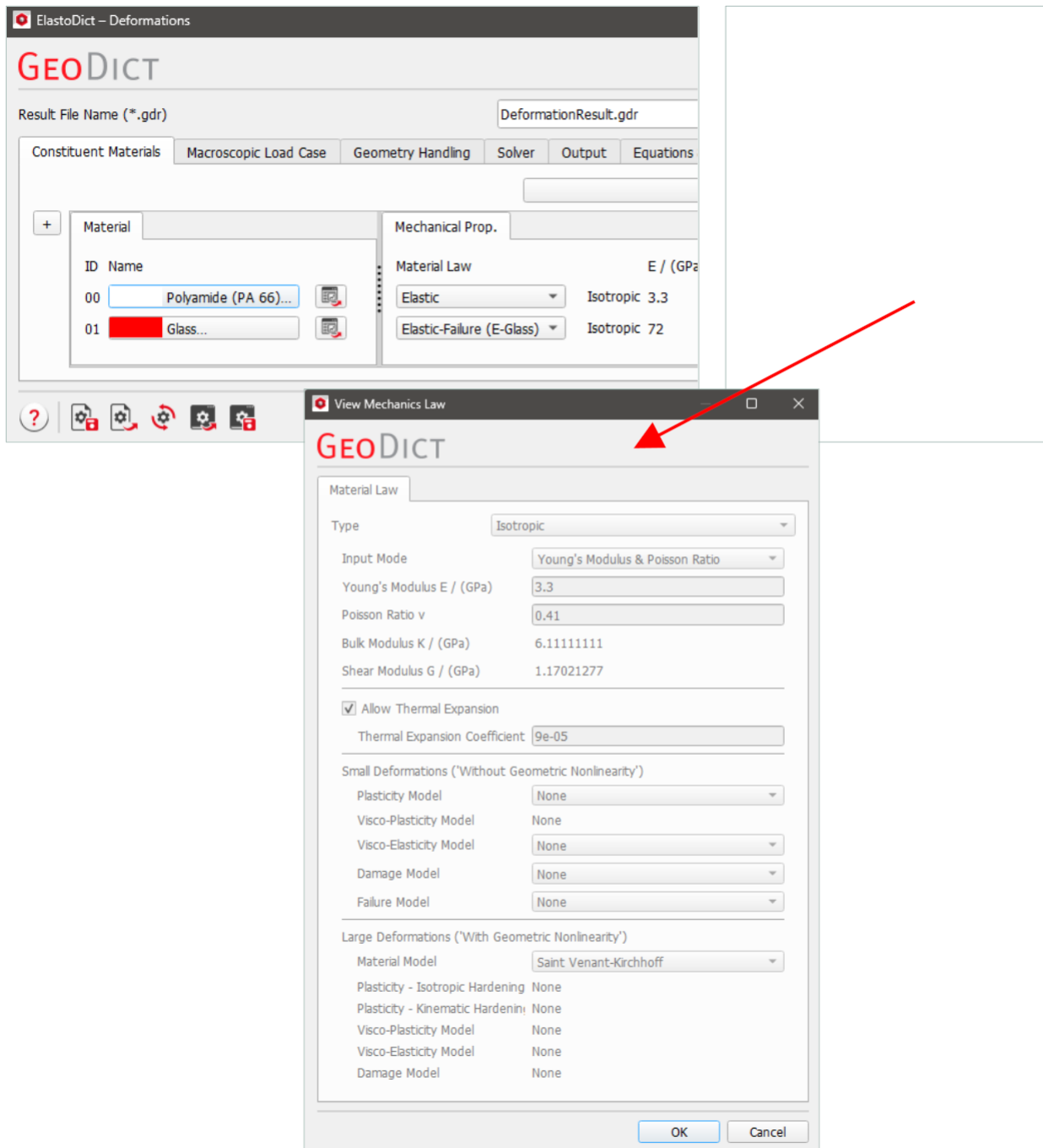
The material for a material ID can be selected by clicking on the corresponding material name. Then, the **Material Selector** dialog is opened.

The mechanical properties of the structure's constituent materials are taken directly from the GeoDict Material Database. To enter the material parameters for a linear elastic material manually, choose **Manual (Solid)** as material.

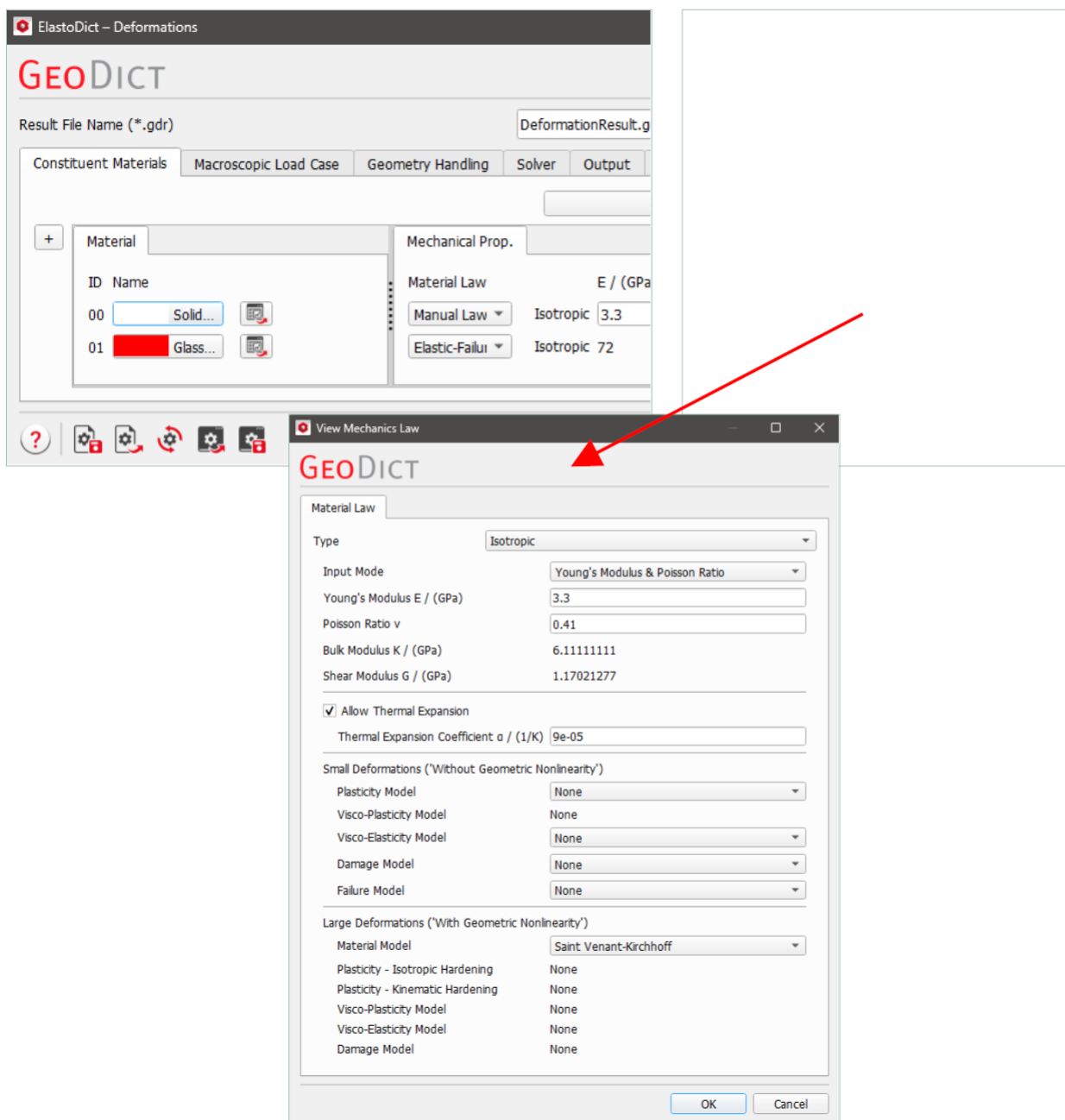


Any values entered manually, e.g., those for a user-defined constituent material, can also be added to the material database through the **Edit Material Database...** button. Nonlinear materials can also be created in the GeoDict **Material Database**.

For materials from the material database, the **View...** button on the right in the **Constituent Materials** tab shows all parameters of the material.



For **Manual** materials, the **View...** button on the right in the **Constituent Materials** tab changes to **Edit...** and allows to edit all parameters of the material.



More information on editing, expanding, and using the GeoDict Material Database is available in the Material Database chapter.

1.3.2 Macroscopic Load Case

The parameters which define the general conditions of the **Macroscopic Load Case** are **Experiment** and **Load Type**. Their choice defines which other parameters are available for the experiment.

ElastoDict – Deformations

GEODict

Result File Name (*.gdr)

Constituent Materials Macroscopic Load Case Geometry Handling Solver Output Equations & References

Experiment Experiment Conditions

Load Type In Tangential Direction

Load Case

Direction X-Direction Free

Plane Y-Direction Free

Angle in Plane / (°)

Fluid Pressure

Pressure Mode

Fluid Pressure / (GPa)

Load Table Load Graph

	Time / (s)	Strain / (%)	Temp. Change / (K)
1	1	2	0
2	2	4	0
3	3	6	0
4	4	8	0
5	5	10	0

Predefined Shape

Shape

Magnitude / (%)

Temperature Change / (K)

Pressure Change / (GPa)

Length / (s)

Number of Steps

Number of Rows

Load... Save... Apply

Boundary Conditions

☒ Periodic ☐ Symmetric ☐ Mixed

OK Cancel

Topics of this section

- [Experiment](#)⁴⁸
Explanation of the Experiment settings under the Macroscopic Load Case tab.
- [Load Type](#)⁵³
Explanation of the Load Type settings under the Macroscopic Load Case tab.
- [Boundary Conditions](#)⁵⁴
Explanation of the Boundary Conditions settings under the Macroscopic Load Case tab.

Experiment

Uniaxial Experiment

In an **Uniaxial Experiment**, the load is applied in one given direction. This direction can be either given by one of the coordinate directions (X, Y, Z), a given angle to one of the coordinate directions in each coordinate plane (e.g., XZ or YZ), or a shear experiment can be defined.

The available load types are **Tensile**, **Compression** or **Shear**. The choice of **Tensile** or **Compression** only affects the sign of the loads, for example negative tension equals compression. The load can be time-dependent, e.g., cyclic.

The screenshot shows the 'ElastoDict – Deformations' window. The 'GEO DICT' logo is at the top left. Below it, the 'Result File Name (*.gdr)' field contains 'DeformationResult.gdr'. The 'Macroscopic Load Case' tab is selected. The 'Experiment' dropdown is set to 'Uniaxial Experiment - Tensile'. The 'Load Type' dropdown is set to 'Path Controlled'. The 'Load Case' section has 'Direction' set to 'Z', 'Plane' set to 'ZX', and 'Angle in Plane / (°)' set to '0'. The 'Experiment Conditions' section has 'In Tangential Direction' set to 'Free', 'X-Direction' set to 'Free', and 'Y-Direction' set to 'Free'. The 'Fluid Pressure' section has 'Pressure Mode' set to 'No Pressure' and 'Fluid Pressure / (GPa)' set to '0'.

✓ Load Case

The choices available for the **Load Case** vary depending on whether **Tensile**, **Compression** or **Shear** is selected. For **Tensile** and **Compression** experiments, a direction, a Plane, and an **Angle in Plane** must be defined. For **Shear** experiments, the **Shear Load** must be selected. If **Without Geometric Nonlinearity** under the **Geometry Handling** tab is chosen, three different shear loads are available: Here, the theory of small deformations applies, therefore the Shear in XY direction corresponds to the shear in YX direction. Thus, three different shear options are available in this case: XY, XZ and YZ.

When the theory of small deformations does not apply (**With Geometric Nonlinearity**), there are three different cases for shear in X and Y directions: Shear in XY direction, shear in YX direction (the deformation gradient is asymmetric in those cases) and symmetric shear in X and Y directions (this is a superposition of the two other cases, the deformation gradient is symmetric in this case). Thus, nine different shear options are available for the geometric nonlinear case.

Refer to the [Geometry Handling](#)⁵⁵ section for more information about the geometry options.

✓ Experiment Conditions

The load conditions in the directions tangential to the load direction can be **Free** (zero mean stress/nominal stress in the geometry), **Confined** (zero mean strain/displacement gradient in the geometry) or **Mixed**. Free means that the structure can expand or contract freely to the load direction, whereas there is no expansion/contraction for confined boundary conditions. With Mixed tangential boundary conditions, two different conditions (**Free** or **Confined**) can be selected for the two tangential directions.

✓ Fluid Pressure

When the structure for the simulation contains pores, a fluid pressure model can be selected. With **No Pressure**, no pressure is applied (this is the default case).

Fluid Pressure

Pressure Mode	No Pressure
Fluid Pressure / (GPa)	0

With **Fixed Pressure**, a fixed pressure can be given. This fixed pressure is applied at the start of the ElastoDict computations.

Fluid Pressure

Pressure Mode	Fixed Pressure
Fluid Pressure / (GPa)	0

With the third option, **Pressure Per Step**, the applied pressure can be defined for each simulation step. These pressure steps must be entered in the **Load Table** (see further explanation below).

Plane
ZX
Angle in Plane / (°)
0

Fluid Pressure
Pressure Mode
Pressure Per Step

Load Table
Load Graph

	Time / (s)	Strain / (%)	Temp. Change / (K)	Fluid Pressure / (GPa)
1	1	2	0	1
2	2	4	0	2
3	3	6	0	3
4	4	8	0	4
5	5	10	0	5

Predefined Shape
Shape
Linear
Magnitude / (%)
10
Temperature Change / (K)
0
Pressure Change / (GPa)
0
Length / (s)
5
Number of Steps
5

Number of Rows
5
Load...
Save...

Apply

The pressure is always applied to all fluids in the structure. This means that it is not only applied to pores in the material, but also to the fluid which surrounds the structure.

✓ Load Definition (Load Table and Predefined Shape)

The load steps to be computed need to be specified in the **Load Table**, where a plot of the load curve can be seen under the **Load Graph** subtab. It is also possible to use a load curve of **Predefined Shape** like **Linear**, **Sawtooth**, or **Sine**, by selecting it from the pull-down menu on the right and clicking **Apply**, to set it in the **Load Table** and visualize it under **Load Graph**.

For the shapes **Linear** and **Sine** there is also the option to define the thermal conditions by setting the **Temperature Change / (K)** for the **Linear** shape and **Temperature Amplitude / (K)** for the **Sine** shape

In case **Pressure Mode** → **Pressure Per Step** is chosen, additionally to the load and temperature curve, the fluid pressure curve can be defined in **Predefined Shape** by setting the **Pressure Change / (GPa)** for the **Linear** shape or **Pressure Amplitude / (GPa)** for the **Sine** shape.

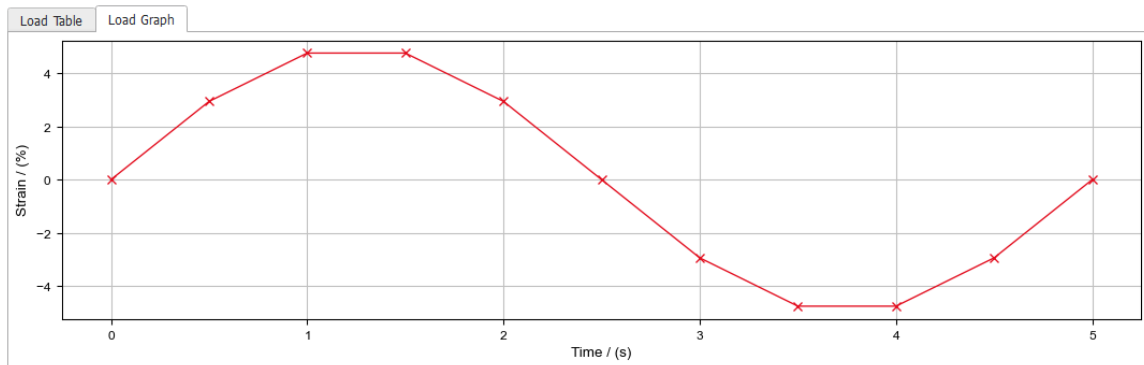
Load Table
Load Graph

	Time / (s)	Strain / (%)	Temp. Change / (K)	Fluid Pressure / (GPa)
1	0.5	2.93893	5.87785	11.7557
2	1	4.75528	9.51057	19.0211
3	1.5	4.75528	9.51057	19.0211
4	2	2.93893	5.87785	11.7557
5	2.5	6.12323e-16	1.22465e-15	2.44929e-15
6	3	-2.93893	-5.87785	-11.7557
7	3.5	-4.75528	-9.51057	-19.0211

Predefined Shape
Shape
Sine
Repetitions
1
Amplitude / (%)
5
Temperature Amplitude / (K)
10
Pressure Amplitude / (GPa)
20
Repetition Length / (s)
5
Steps per Repetition
10

Number of Rows
10
Load...
Save...

Apply



The load table can be loaded from and saved to a *.txt file with the **Load...** and **Save...** buttons.

Alternatively, the data from the load table can be entered from other software like Microsoft Excel[®] via copy and paste.

The time information from the load table is only used if the materials in the structure have a time-dependent behavior. When working with UMATs, this time information is passed to the UMAT.

Complex Load Experiment

The **Complex Load Experiment** allows to define more elaborate load scenarios, e.g., a tensile experiment followed by a shear load or multiple load directions at the same time. The load directions can be defined freely as a combination of normal and shear loads. The complete mean stress (nominal stress) or strain (displacement gradient) tensor for the geometry can be defined.

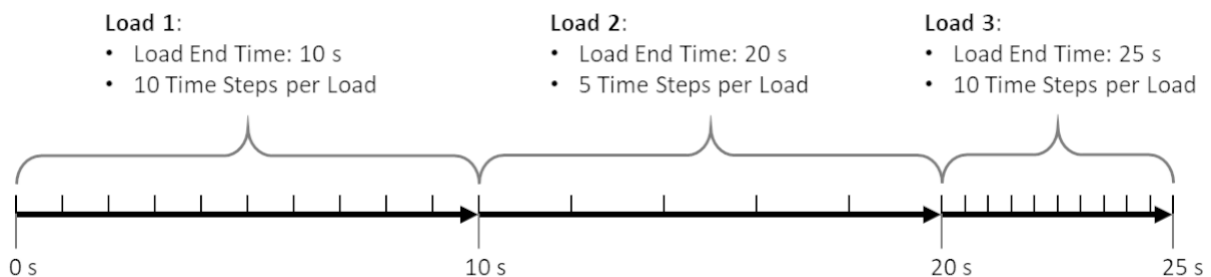
In this way, combinations of multiple loads directions are possible (e.g., biaxial load, triaxial load...). The load directions can be defined freely as a combination of normal and shear loads. In the screenshot below, a tensile strain of 2% in X-direction, a shear load of 1% in XY-direction and a tensile strain of 5% in Z-direction is set.

Constituent Materials	Macroscopic Load Case	Geometry Handling	Solver	Output	Equations & References																
Experiment	Complex Load Experiment																				
Load Type	Path Controlled																				
Time Step Mode	One Step																				
Load Load Scenario User Defined <table border="1"> <thead> <tr> <th></th> <th>X / (%)</th> <th>Y / (%)</th> <th>Z / (%)</th> </tr> </thead> <tbody> <tr> <td>X / (%)</td> <td>2</td> <td>1</td> <td>0</td> </tr> <tr> <td>Y / (%)</td> <td>1</td> <td>0</td> <td>0</td> </tr> <tr> <td>Z / (%)</td> <td>0</td> <td>0</td> <td>5</td> </tr> </tbody> </table> Temperature Change / (K) 0 Fluid Pressure / (GPa) 0							X / (%)	Y / (%)	Z / (%)	X / (%)	2	1	0	Y / (%)	1	0	0	Z / (%)	0	0	5
	X / (%)	Y / (%)	Z / (%)																		
X / (%)	2	1	0																		
Y / (%)	1	0	0																		
Z / (%)	0	0	5																		

Possible options for **Time Step Mode** are **One Step** or **Time Series**.

Whereas **One Step** consists of a single load step, a **Time Series** may consist of several load steps (Length of Load Sequence: 1, 2, 3...), where each load step (Load 1, Load 2, Load 3... tabs) is defined by a **Load End Time**, number of **Time Steps per Load** and a **Load Scenario**. The **Load Scenario** can be **Tensile**, **Compression**, **Shear** or **User Defined**.

The total time of the load is the difference of the current **Load End Time** and the previous **Load End Time**; therefore, the load end time must be strictly increasing. Each load is divided in a given number of **Time Steps per Load**. The concept is illustrated in the following figure:



The load sequences can be combined as desired. You can, e.g., do a compression in X-direction followed by a compression in Y-direction and a shear load.

In the example below, the 3rd load is subdivided into 10 time steps (**Time Steps per Load**) and the load scenario is **Tensile**.

Constituent Materials
Macroscopic Load Case
Geometry Handling
Solver
Output

Experiment
Complex Load Experiment

Load Type
Path Controlled

Time Step Mode
Time Series

Length of Load Sequence
3

Load 1
Load 2
Load 3

Load End Time / (s)
1

Time Steps per Load
10

Load Scenario
Tensile

X-direction / (%)
10

Y-direction / (%)
0

Z-direction / (%)
0

Temperature Change / (K)
0

Fluid Pressure / (GPa)
0

Boundary Conditions

☒ Periodic
☐ Symmetric
☐ Mixed

Load Type

▪ Path Controlled

specify the mean strain in the geometry (for geometric linear experiments) or the mean displacement gradient in the geometry (for geometric nonlinear experiments) in the load direction.

▪ Force Controlled

Specify the mean stress in the geometry (for geometric linear experiments) or the mean nominal stress in the geometry (for geometric nonlinear experiments) in the load direction. It is important to note that the mean stress is computed with respect to the complete geometry. If a structure consists of solids and pore space, this means that the average load on the solids is larger than the given mean stress, since the load in the pore space is zero.

Generally, **Path controlled** should be preferred over **Force Controlled** if possible. One of the challenges with **Force Controlled** is the fact that the Strain-Stress curves for plastic materials are often very flat: This means that a small change in stress can lead to a very large change in strain. On the other hand, a small change in strain leads to an even smaller change in stress. Therefore, the **Path Controlled** simulation is much easier to compute. For simulations with damage, **Force controlled** may be even impossible to solve (because the given stress might be

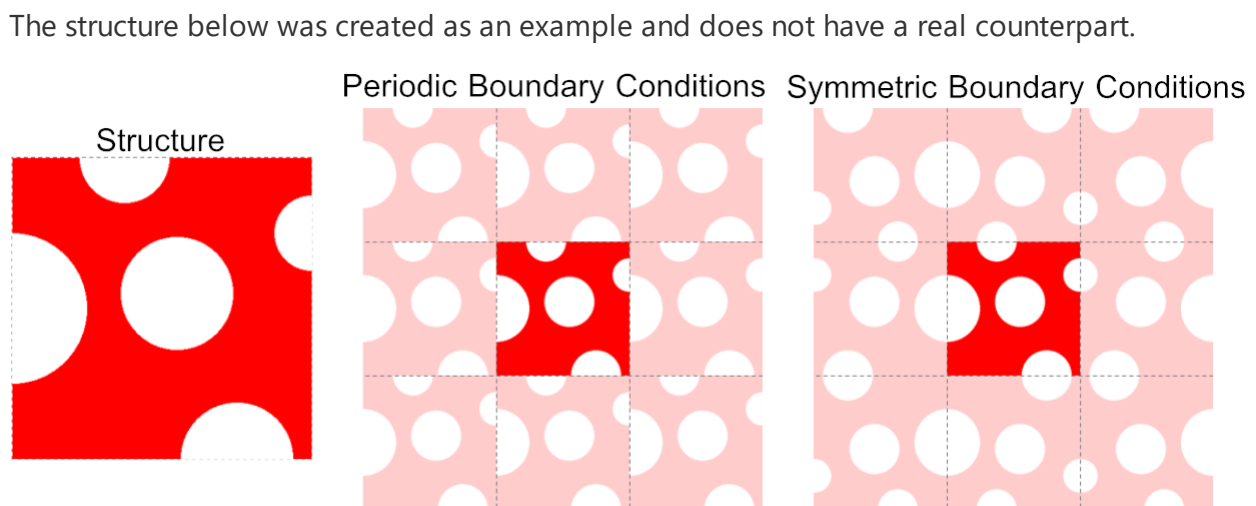
higher than the maximal stress allowed by the material), while **Path Controlled** can always be solved (see also the [load type recommendations](#)^[19]).

Boundary Conditions

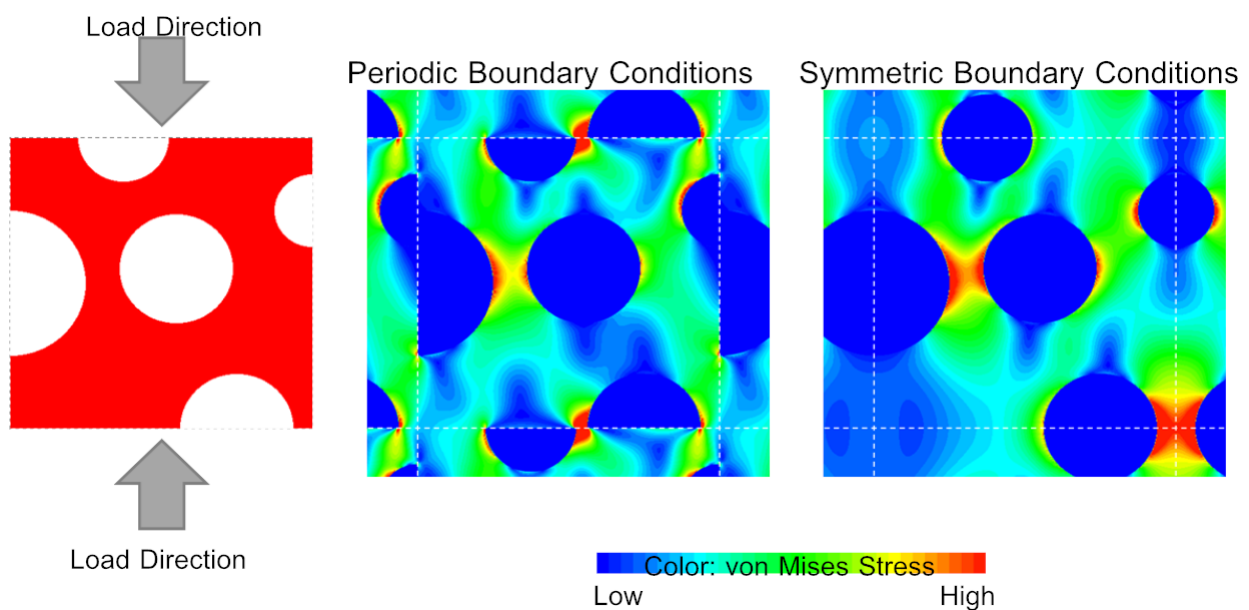
Here, the domain boundary conditions for the strain (and stress respectively) may be set to **Periodic**, **Symmetric**, or **Mixed**. Note that displacements do not need to hold these boundary conditions.

In general, periodic boundary conditions should be used for periodic structures, whereas otherwise symmetric or mixed boundary conditions may be applied.

Mixed boundary conditions mean that symmetric boundary conditions apply in the load direction, whereas periodic boundary conditions apply in the tangential directions. Mixed boundary conditions can only be used for tensile, or compression loads along the coordinate directions. In the figure below, periodic and symmetric boundary conditions are illustrated for a non-periodic structure.



When periodic boundary conditions are applied to non-periodic structures, it might lead to stress peaks at the boundaries as illustrated below for a uniaxial load.

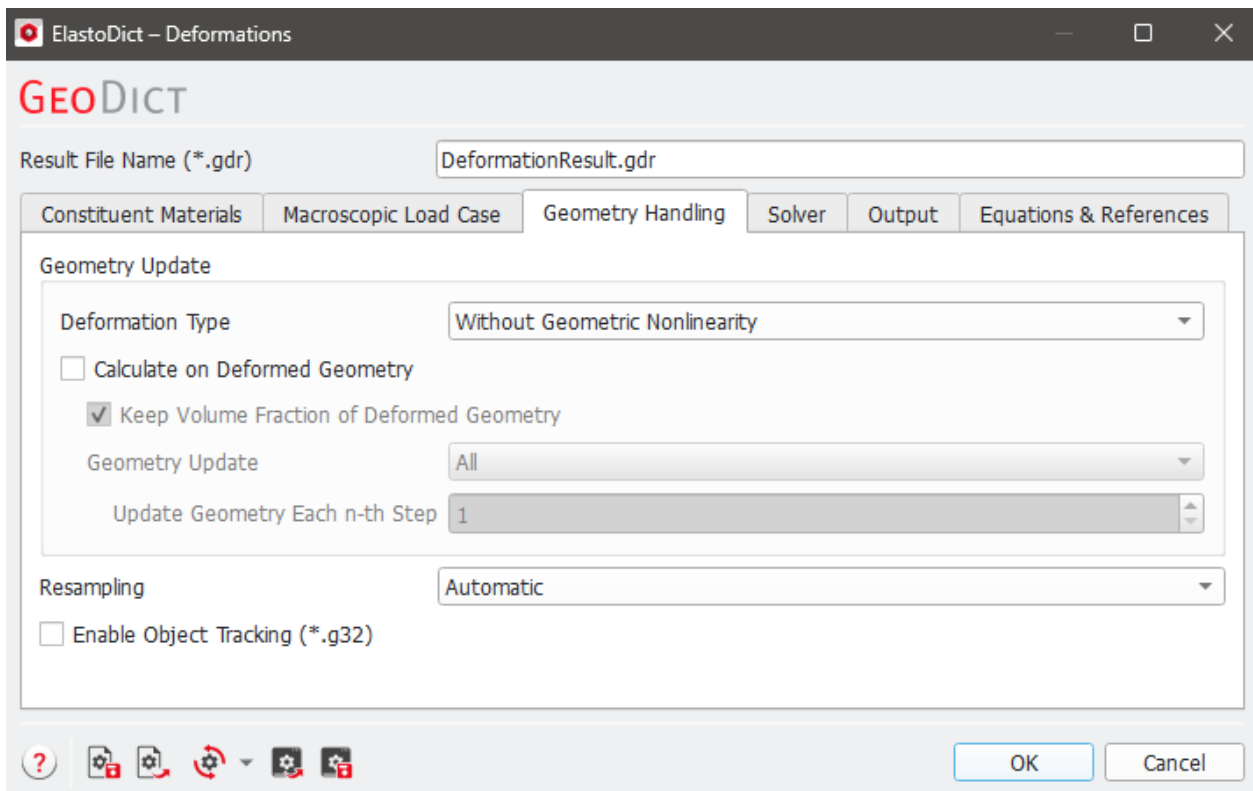


However, simulations with periodic boundary conditions are much faster and need less memory than simulations with symmetric boundary conditions.

In many cases, for example for composite structures with low fiber percentage, the results for periodic boundary conditions are comparable with the results for symmetric boundary conditions even if the analyzed structure is not periodic. Therefore, it is recommended to work with periodic boundary conditions until the best simulation settings are found and only use symmetric boundary conditions in the final simulation.

1.3.3 Geometry Handling

The **Geometry Handling** tab contains all options related to geometry changes during the simulation.



Topics of this section

- [Deformation Type](#)^[56]
Explanation of the Deformation Type settings under the Geometry Handling tab.
- [Calculate on Deformed Geometry](#)^[57]
Explanation of the Calculate on Deformed Geometry setting under the Geometry Handling tab.
- [Resampling](#)^[60]
Explanation of the Resampling setting under the Geometry Handling tab.
- [Enable Object Tracking \(*.g32\)](#)^[61]
Explanation of the Enable Object Tracking setting under the Geometry Handling tab.

Deformation Type

Under **Deformation Type**, the theory which is used for the structural mechanics is selected:

- **Without Geometric Nonlinearity:** This approach is faster and more reliable than the geometric nonlinear case. More material models are available in GeoDict for this case (e.g., Plasticity and Damage models). The boundary conditions for this experiment type are given as mean strains or mean stresses in the analyzed volume.
 - Here, the **infinitesimal strain theory** applies (see more information on [Wikipedia](#)). This is the linearized form of the *finite strain theory* (which applies to **With Geometric Nonlinearity**).

- It is assumed that only small deformations occur, for example it is assumed that no rotations occur.
- This is the simpler theory, but it is much more robust and easier to solve and is therefore preferable in most use cases. We recommend to always try **Without Geometric Nonlinearity** first.
- For applications with larger deformations use **Without Geometric Nonlinearity** together with [Calculate on Deformed Geometry](#) ⁵⁷.
- The internal calculations are based on the engineering strain and stress.
- **With Geometric Nonlinearity:** Should be used when the theory of small deformations does not hold anymore. A nonlinear problem must be solved; therefore, the computation might be considerably slower than for the geometric linear case. The boundary conditions for this experiment type are given as mean displacement gradient or mean nominal stresses in the analyzed volume.
 - Here, the **finite strain theory** applies (see more information on [Wikipedia](#))
 - Here, large deformations are considered, this means that also the effect of rotations is considered correctly.
 - This theory is more complete, but also *much more complicated to solve* (Finite Strain Theory).
 - The internal calculations are based on the true strain and stress. Nevertheless, if the deformations are small, the strains in the nonlinear definitions do not deviate largely from the strains in the linear definition (e.g., for small strains, the Green Lagrange Strain is close to the Strain for **Without Geometric Nonlinearity**).



Note! The difference between the geometric linear and nonlinear case is not special to GeoDict. This concerns all structural mechanics simulations – one always needs to choose between the simpler, but robust theory (**Without Geometric Nonlinearity**) and the more general theory (**With Geometric Nonlinearity**), which is much more complicated to solve and understand, but necessary in some cases. The two options (“With/Without Geometric Nonlinearity”) might have slightly different names in other software, but the theory behind them is the same.

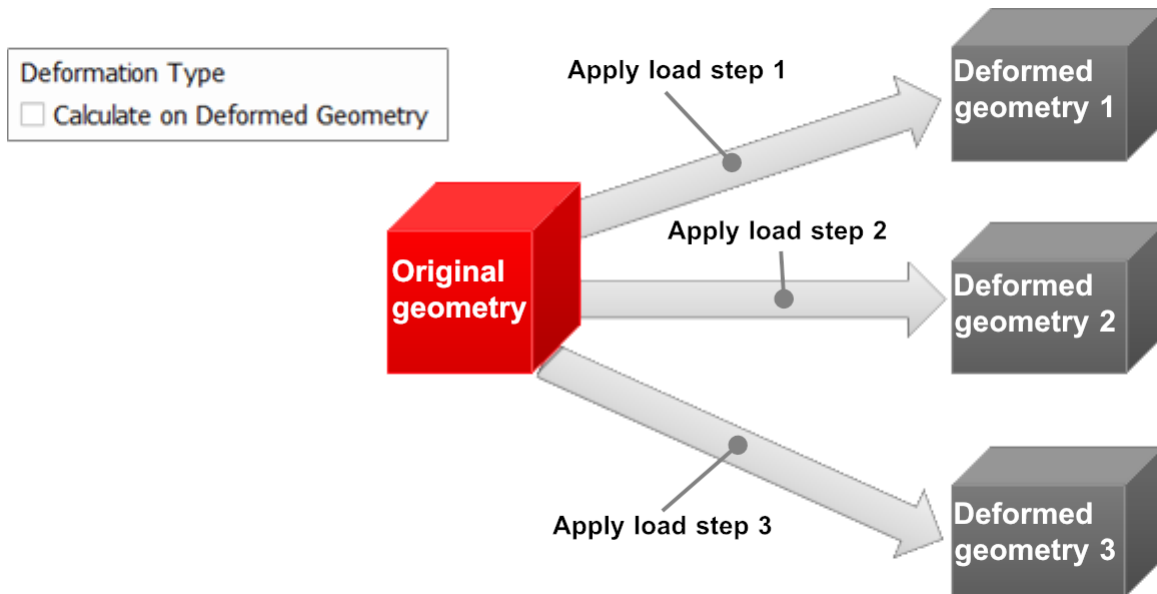
Calculate on Deformed Geometry

With **Calculate on Deformed Geometry** checked, the deformed geometry is re-sampled, and this geometry is used for the calculations in the next step. With this method, it is possible to detect new object contacts during deformation and, e.g., to observe the effect of stiffening during compression. Calculate on Deformed Geometry can only be used without geometric nonlinearity.

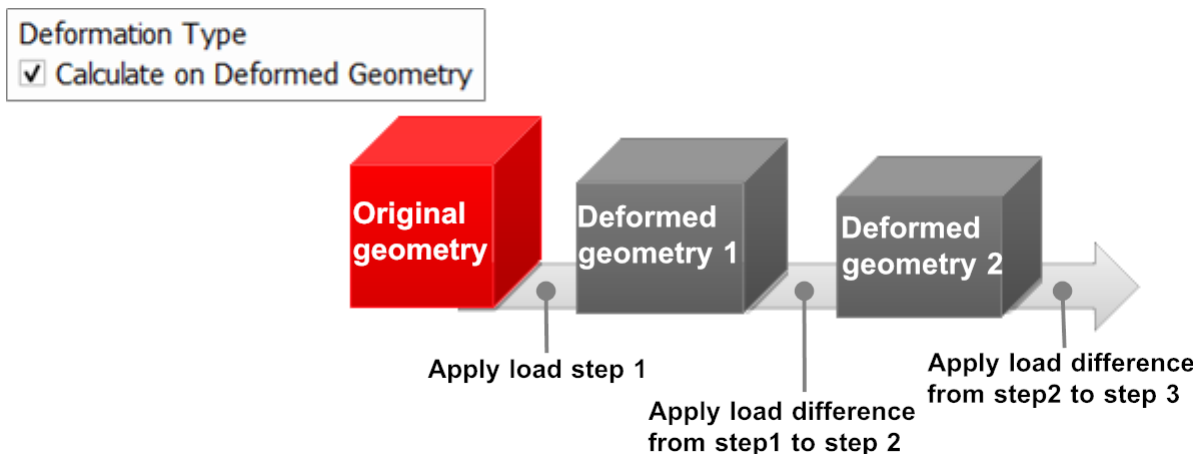
A resampling step is done if the conditions defined under Geometry Update are met. It is important to choose these settings appropriately for the current geometry and deformation

steps: With each resampling step, discretization errors occur, and these errors accumulate if too many resampling steps are done. Therefore, the resampling should only be done after a significant change in the geometry size. This can be achieved by choosing the mode **Size Control** for [Geometry Update](#).^[59]

- **Off:** The deformation is always calculated relatively to the original geometry



- **On:** If the conditions defined under **Geometry Update** are met, the deformed geometry is calculated, and the deformation is calculated relative to the previous geometry. In the figure below, the behavior of the setting **All** for **Geometry Update** is shown.



Keep Volume Fraction of Deformed Geometry

If **Keep Volume Fraction of Deformed Geometry** is checked, the accurate volume fraction information for each material ID after the deformation is stored and updated in memory. Refer to [Resampling](#)^[60] for further information about the volume fractions and how they are further processed to the final result. This needs more memory but leads to a more accurate geometry deformation if many (small steps) are computed. Without this information, the volume

fractions are converted into a structure in each deformation step and the volume fraction gets lost.

Geometry Update

Deformation Type Without Geometric Nonlinearity

☒ Calculate on Deformed Geometry

☐ Keep Volume Fraction of Deformed Geometry

Geometry Update Size Control

Geometry Change / (%) 2

Geometry Update

Under **Geometry Update**, it can be selected when the next simulation step is computed on the newly deformed geometry, or if it is computed based on the existing geometry.

Use the mode **Size Control** for the Geometry Update when using **Calculate on Deformed Geometry**. With this mode, the size of the deformation is checked after each step.

Only if the deformation is larger than the given threshold, **Calculate on Deformed Geometry** is used for this step. Otherwise, the step is done without **Calculate on Deformed Geometry**. We strongly recommend using this mode for simulations of this type.

Geometry Update

Deformation Type Without Geometric Nonlinearity

☒ Calculate on Deformed Geometry

☐ Keep Volume Fraction of Deformed Geometry

Geometry Update All

Update Geometry Each n-th Step

Size Control

- **All** – The geometry is updated in every step

Geometry Update All

Update Geometry Each n-th Step 1

- **Step Control** – Choose how often the geometry is updated with **Update Geometry Each n-th Step**.

Geometry Update Step Control

Update Geometry Each n-th Step 2

- **Size Control** – If the geometry is updated or not depends on the change of the domain size. If the size change for one domain side is larger than the value defined for **Geometry**

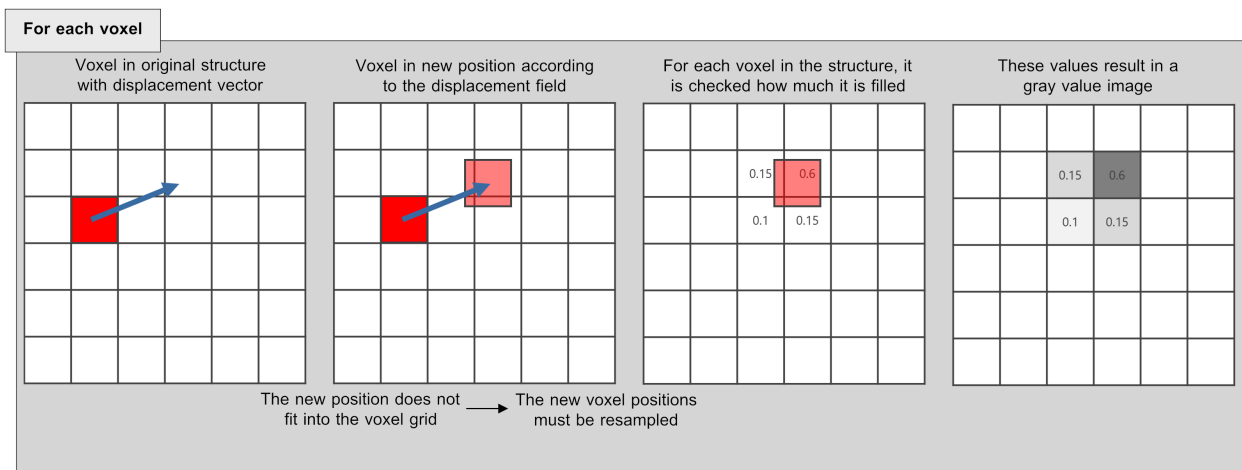
Change / (%), the geometry is updated.

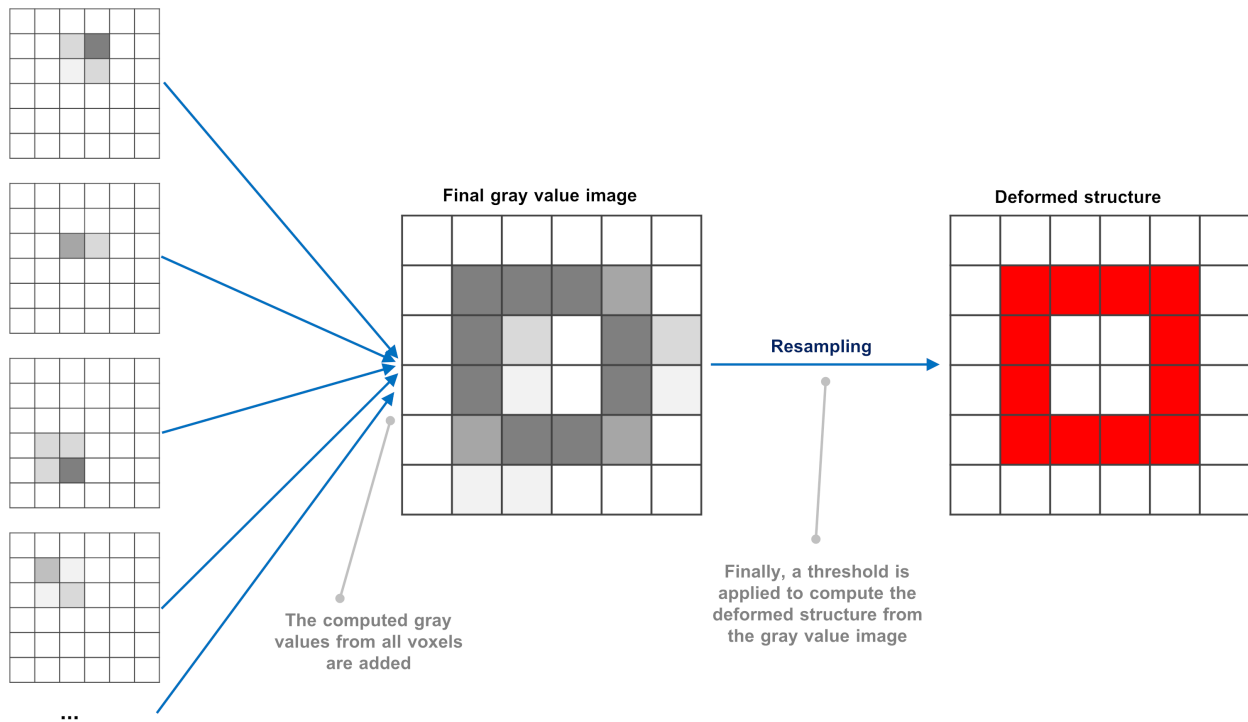
Geometry Update	Size Control
Geometry Change / (%) 2	

Resampling

For **Resampling** you can choose between three options: **Automatic**, **Maintain Volume** or **Maintain Mass**. Deformation in ElastoDict is computed and stored as a displacement field. For each voxel, this displacement field contains the information on how far it is displaced in the current deformation step.

Internally, the new material positions are calculated based on this displacement information. Since the new positions do not necessarily lie on the voxel grid, a gray value image with material **Volume Fractions** is computed for each material ID. These volume fractions contain the information "how much" of a given material is contained in a voxel after the deformation. These volume fractions must be resampled to the voxel geometry. In the figures below, this resampling is shown in a simplified example for one material ID. The deformed geometry is generated by thresholding the resulting gray value image containing the volume fractions.





With **Automatic**, the resampling of the deformed geometry is based on the volumetric strains of the materials. The resampling threshold is chosen in each step such that the calculated volume change is reached. This is the recommended resampling method as it results in most realistic deformations. Especially for applications where the volume of the material changes significantly during the computations, e.g., thermal expansion or expansion due to a negative Poisson's ratio, **Automatic** should be used.

With **Maintain Volume**, the resampling threshold is chosen in such a way that the volume of each solid material in the structure stays roughly the same.

With **Maintain Mass**, the resampling threshold is set to 0.5.

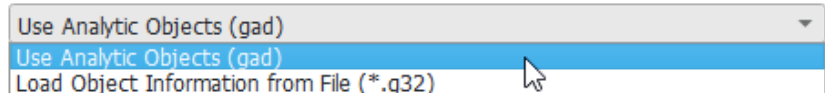
All methods do not apply to the most soft material in the structure (e.g., pores or matrix material in a composite).

Enable Object Tracking (*.g32)

With the object tracking option, a *.g32 file with the object indices for all objects in the structure is created for each deformed geometry. This way, the deformation of each object in the structure can be tracked during the deformation.

☒ Enable Object Tracking (*.g32)

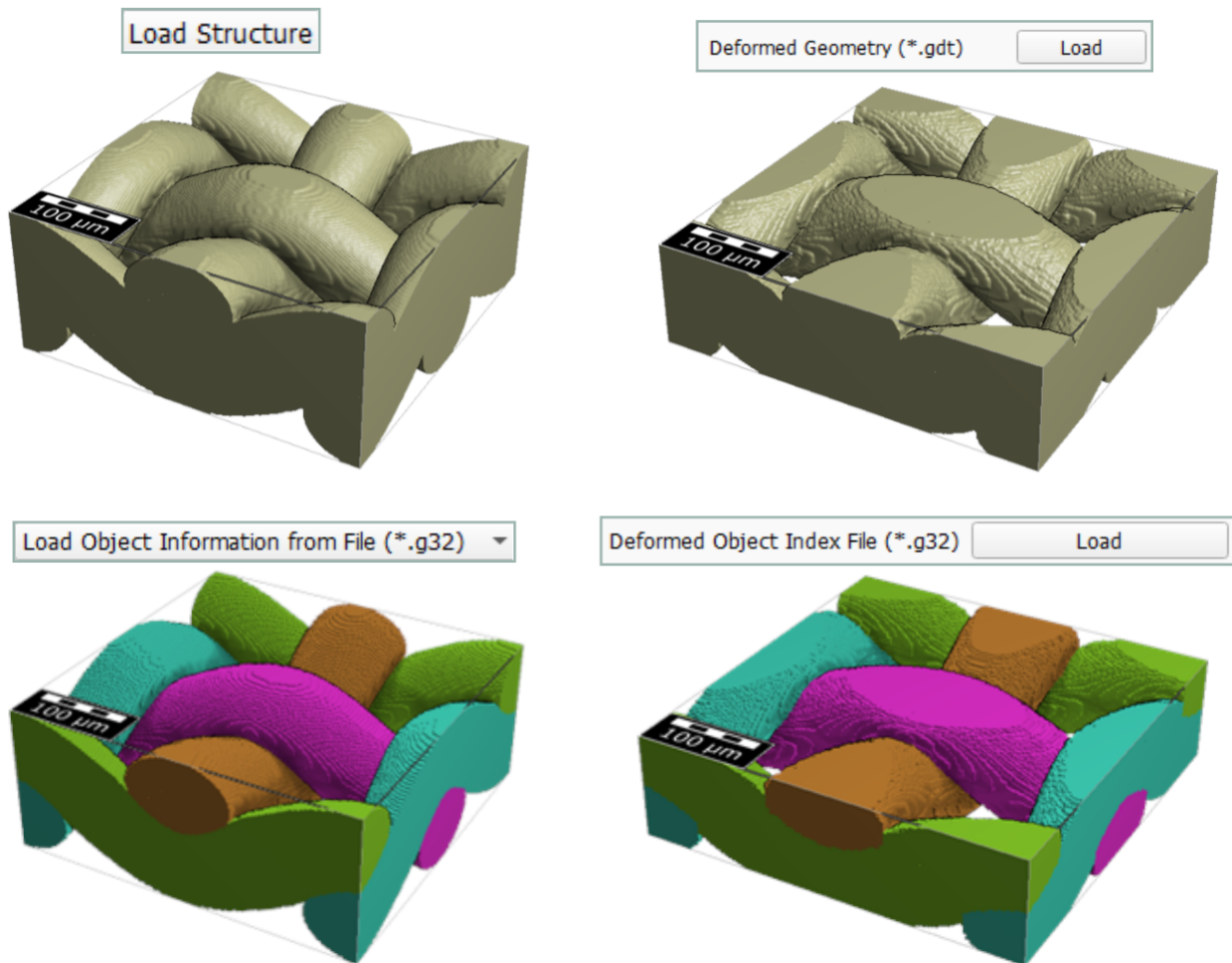
Object Mode



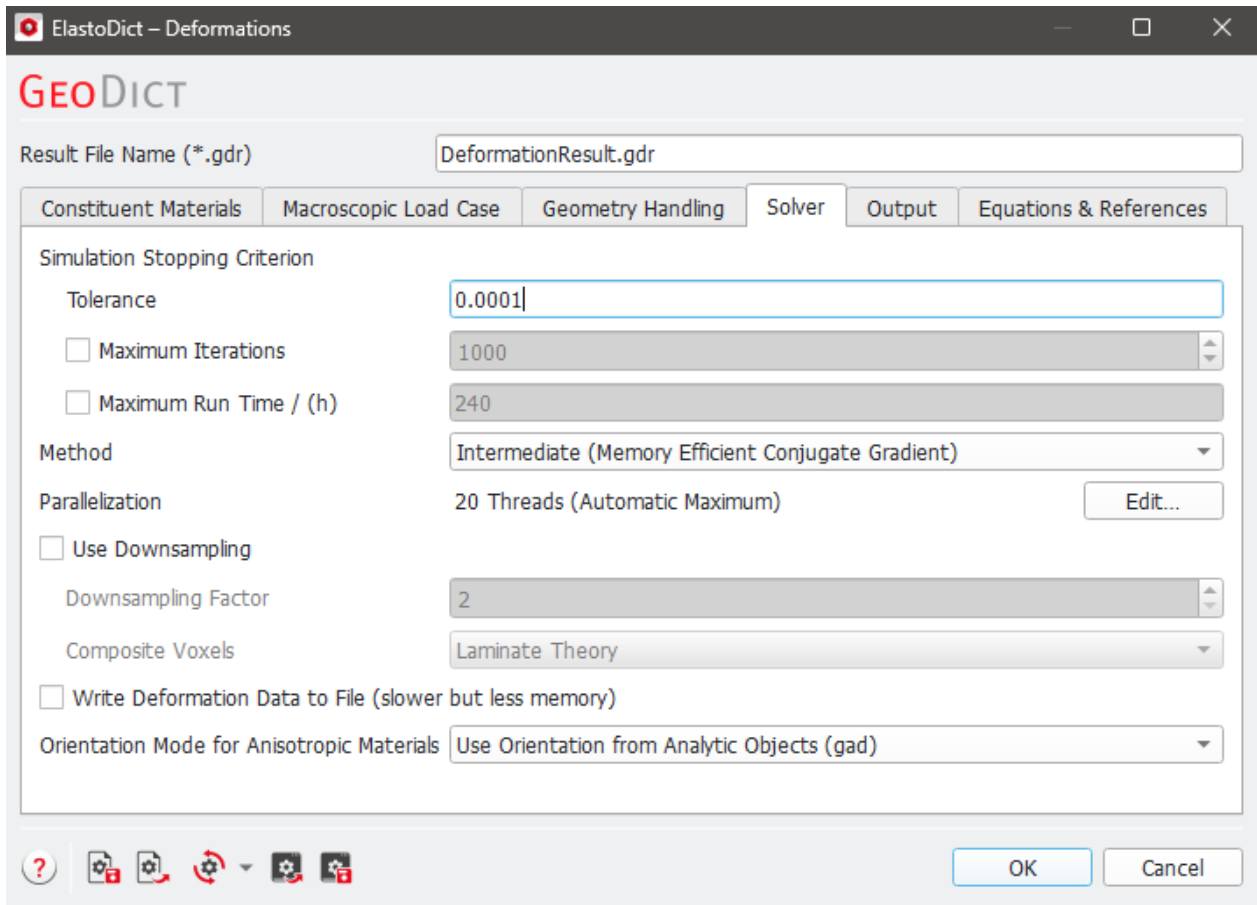
Object Mode

The choice for **Object Mode** depends on the available information: If a structure with analytic object information (e.g., from GeoDict's structure generator modules FiberGeo, GrainGeo, FoamGeo, ...) is used, this information can be used with **Use Analytic Objects (gad)**. In this case, a *.g32 file with object indices is automatically created and saved in the results folder. If no analytic object information is available, **Load Object Information from File (*.g32)** must be selected and a *.g32 file must be loaded. A *.g32 file can be created, e.g., with GrainFind for all grains in the structure.

In the example below, a weave structure from WeaveGeo with the corresponding object indices is shown before and after deformation in ElastoDict. These structures can be loaded directly from the result viewer with the corresponding buttons (shown below).



1.3.4 Solver



Internally ElastoDict uses an iterative solver to solve the equations that describe the mechanical problem at each voxel. The basic idea of an iterative method is:

1. Start with an initial guess for the unknown values.
2. Improve the current values in each iterative step. The speed of the improvement depends on the problem parameters.
3. Repeat the iterative process until one of the stopping criteria occurs.

✓ Simulation Stopping Criterion

The iterative process of the solver is controlled by setting the values for **Tolerance**, **Maximal Iterations**, and **Maximal Run Time (h)**. Only the tolerance must be set, the two other parameters are optional.

For each iteration step, the relative error of the current solution is computed. If this relative error is smaller than the given tolerance, the computation is finished. When there is doubt about the quality of the solution, decrease the tolerance value by a factor of ten for that solver. In general, it is recommended to keep the default setting for tolerance.

When the solver stops because the **Maximal Iterations** or the **Maximal Run Time (h)** are reached, the quality of solution might be doubtful since the requested accuracy is not achieved. Therefore, those two options are disabled by default and should only be selected if there are strong constraints on the allowed runtime.

✓ Method

For the **FeelMath** solver, three different iterative methods are available: **Fast (Conjugate Gradient)**, **Intermediate (Memory Efficient Conjugate Gradient)** and **Memory Efficient (Neumann Series)**.

Fast: here the CG method is used ([Conjugate gradient method - Wikipedia](#)), which is the fastest available approach in ElastoDict, especially for strongly varying material parameters and nonlinear material laws as, e.g., plasticity or damage. The convergence rate is proportional to the square root of the stiffness contrast. However, it needs up to 4 times more memory compared to the **Memory Efficient** method. We recommend using the **Fast** method whenever possible.

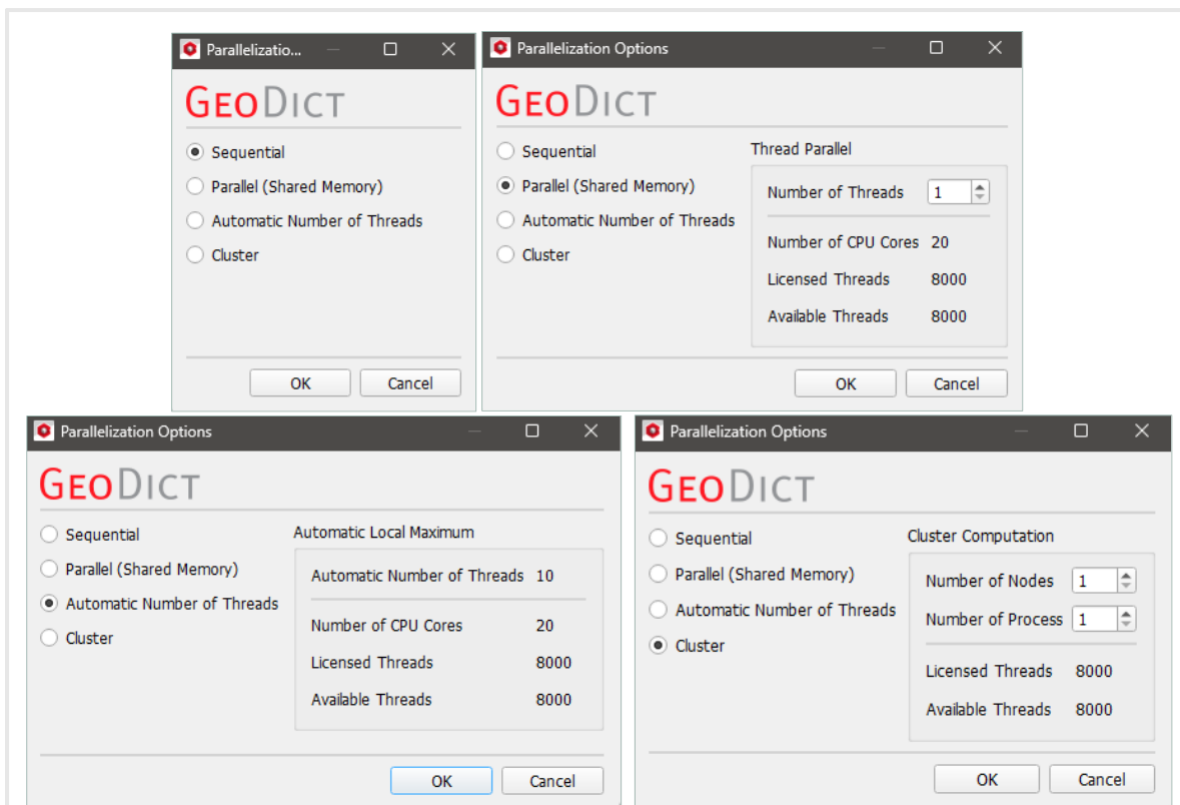
Intermediate: this is a memory efficient CG method, which needs approximately 40 % less memory than the **Fast** method but may be 20-30 % slower.

Memory Efficient: here the Neumann series is used. It is the most memory efficient method, but the convergence rate is proportional to the stiffness contrast in the materials and can therefore be low. This method should only be used for applications with a low stiffness contrast, or if the RAM is not sufficient.

For **Deformations** and **Effective Stiffness**, the **Estimate Memory** button is present in the ElastoDict section. When the structure of interest is shown in the Visualization area, clicking **Estimate Memory** calculates the memory needed for the computations based on the size of the structure and the parameters entered in the solver options. The Estimate Memory button can be used to decide which method is applicable for the current structure on the available computer.

✓ Parallelization

The ElastoDict run can be accelerated by using multiple CPU cores. The **Parallelization Options** dialog opens when clicking the **Edit...** button, to choose between **Sequential**, **Parallel (Shared Memory)**, **Automatic Number of Thread** and **Cluster** as well as **MPI Parallel** for Linux users.



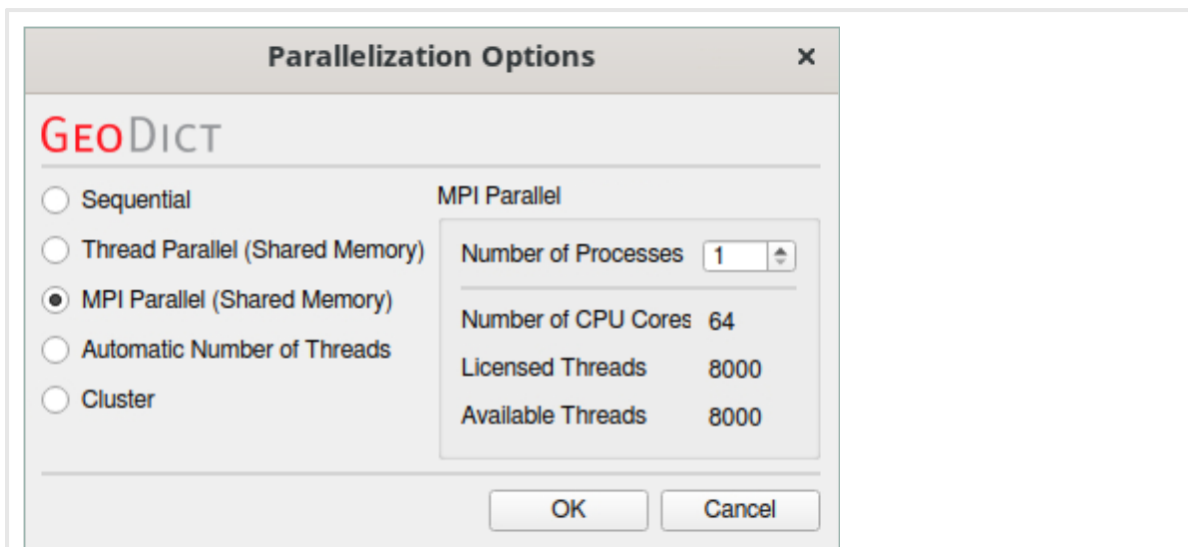
When **Sequential** is selected only one thread is used for the computation.

Selecting **Parallel (Shared Memory)** runs the parallel computation with the specified **Number of Processes** on a shared-memory hardware. Then the maximum number of available processors and the maximum number of licensed parallel processes is shown in the dialog.

If **Automatic Maximum of Threads** is selected, the number of parallel processes is automatically selected for optimal speed, based on the CPU cores and licensed parallel processes. For up to eight Available Threads, all of them will be used. If more than eight threads are available, two cases might occur:

- The number of **Available Threads** is larger (or equal) than the **Number of CPU Cores**:
 - Then the maximum of eight and **Number of CPU Cores** divided by 2 is used.
- The number of **Available Threads** is smaller than the **Number of CPU Cores**:
 - Then the maximum of eight and number of **Available Threads** divided by 2 is used.

For users of **Linux** systems the solver supports the MPI parallelization method. The **Parallelization Options** dialog under a Linux system has the options **MPI Parallel (Shared Memory)** and **Thread Parallel (Shared Memory)** as the parallelization can be done with MPI parallelization or Thread parallelization.



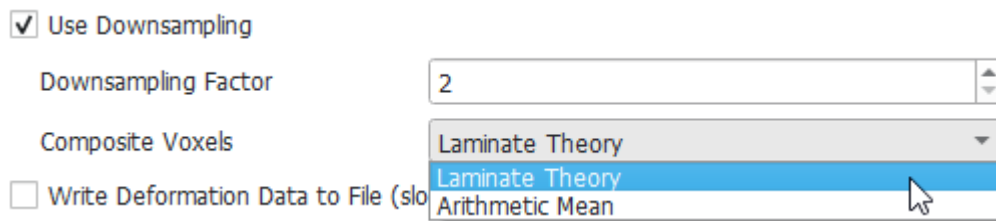
For details on how to set up and run parallel computations, consult the High Performance Computations User Guide.

✓ Use Downsampling

Downsampling reduces the structure size by combining multiple voxels. By default, the **Downsampling Factor** is set to two – this means that 2x2x2 voxels are combined to one voxel. The material model used for each voxel in the downsampled structure is a combination of the material model (e.g., a **Composite Voxel**) of the original voxels. Since the simulated structure size is much smaller than the original structure, **Downsampling** allows to speed up the simulation and to quickly obtain results even on computers with little memory. This option should not be used if highest accuracy is required.

Downsampling Factor

In most cases, a **Downsampling Factor** of 2 is the best choice, but it can also be set to 4 if needed. Then, the computation will be faster, but the results could be more inaccurate.



To be able to use downsampling, the features (e.g., fibers or grains) in each material ID must have a diameter of at least the Downsampling factor in voxels (e.g., 2 voxels for a **Downsampling Factor** of 2). Otherwise, the computation of the material properties might not work properly. Since GeoDict 2021, an error message is shown if all features in the structure are smaller than this diameter, and the simulation cannot be started.

Composite Voxels

Two different methods to compute the **Composite Voxels** during downsampling: **Laminate Theory** (the default), and **Arithmetic Mean**. When using **Laminate Theory**, the composite material properties are computed based on the laminate theory of composite materials. With this option, also direction dependent properties are represented correctly, which is important especially at the boundary between two different materials. With **Arithmetic Mean**, the material properties of the composite voxel are the average of the material properties of the original voxels. This method is less accurate, but the simulation might be more stable, especially if the structure contains many pores (as, e.g., in foams). Nevertheless, it is recommended to keep the default **Laminate Theory** in most use cases for the higher accuracy.

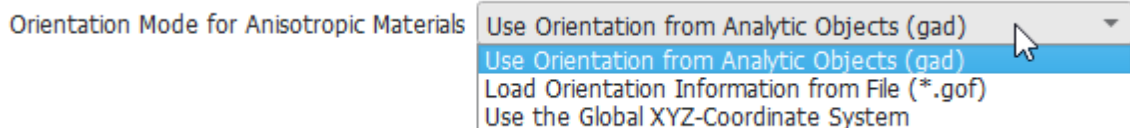
✓ Write Deformation Data to File

For the computation of the deformed structures, intermediate data is computed which describes the displacement for each voxel. Before this computation, it is not clear how large this data will be, since a voxel in the original structure might be partially moved in multiple voxels in the deformed structure. Therefore, this data might not fit in the main memory of the computer. By checking the option **Write Deformation Data to File**, the deformation data will be written to the hard drive .

We recommend keeping this option disabled, and to select it only if the calculation of the deformed geometry fails because the data does not fit into the RAM. Writing and loading data from the hard drive is many times slower than writing and loading data in the main memory (even if the drive is an SSD). This causes a bottleneck, which can slow down the simulation.

✓ Orientation Mode for Anisotropic Materials

ElastoDict can handle non-isotropic constituent materials. For these materials, an orientation has to be additionally specified. If the analyzed structure consists of analytical objects (gad data), the orientation of these objects is used when **Use Orientation from Analytic Objects (gad)** is selected.



Alternatively, with **Load Orientation Information from File (*.gof)**, orientation information can be loaded from a file. Such a file can be generated for example with GrainFind for granular structures.

The third option, **Use the Global XYZ-Coordinate System**, allows to use anisotropic materials even if no orientation information for the structure is available.

1.3.5 Output

The options available in the Output tab control which information is saved to the hard disk during the computation. Choosing only the necessary variables might help to save computation time.

✓ Write All Steps

When **Write all Steps** is chosen, the results fields selected under **Write Volume Field** are written after each computation step. Otherwise, the number of steps after which the results fields are saved can be selected. When computing with many steps, disabling **Write all Steps** saves considerable computation time.

✓ Deformed Geometry

When **Write Deformed Geometry** is chosen, a geometry with the current displacements is resampled and saved after each step. Deformed geometries are written at each step, even if **Write all Steps** is disabled. Writing the deformed geometries is mainly interesting for visualization purposes and to check possible problems in the simulation setup when [Calculate on Deformed Geometry](#)^[57] is chosen.

With **Write Volume Fields on Deformed Geometry**, also the result fields are resampled with the current displacements so that they match the deformed geometries.

With **Write Volume Fractions for Deformed Geometry**, the volume fractions for each material ID in each voxel after the deformation are written to the *.das files. This information is necessary for the computation of the deformed geometries (see [Resampling](#)^[60] for further explanations). In general, it is not necessary to select this option.

The screenshot shows the 'Geometry Handling' tab of a software interface. It contains a 'Geometry Update' section with the following options: 'Deformation Type' set to 'Without Geometric Nonlinearity', a checked checkbox for 'Calculate on Deformed Geometry', an unchecked checkbox for 'Keep Volume Fraction of Deformed Geometry', a 'Geometry Update' dropdown menu set to 'All', and 'Update Geometry Each n-th Step' set to '1'.

Depending on the options chosen in the **Geometry Handling** tab, some **Deformed Geometry** options might be unchangeable. For example, if **Calculate on Deformed Geometry** and **Enable Object Tracking (*.g32)** are selected, **Write Volume Fields for Deformed Geometry** is selected by default and cannot be deselected.

The screenshot shows the 'Output' tab of the software interface. It contains a 'Write Steps' section with a checked checkbox for 'Write All Steps' and a 'Write Result Fields (*.das) for Each n-th Step' dropdown menu set to '2'. Below this is a 'Deformed Geometry' section with three checked checkboxes: 'Write Deformed Geometry', 'Write Volume Fields for Deformed Geometry', and 'Write Volume Fractions for Deformed Geometry'.

✓ Allow Restart for Deformation Simulations

If you want to be able to restart a simulation after an interruption, the complete stress and strain information must be saved for each step. With the option **Allow Restart for Deformation Simulations**, the necessary volume fields are automatically checked under **Write Volume Field** below and therefore saved.

✓ Write Volume Field

Checking or un-checking the boxes for **Displacement**, **Stress**, and **Strain** determines whether these results outputs are available for visualization and post-processing after the run of the ElastoDict-Deformations. Additionally, it is possible to save the deformed geometry and to save the solution fields (for stress and strain) for the deformed geometry. Depending on the structures size, the result files can become large (up to several GB), so it is recommended to deselect all options which are not needed for the current task.

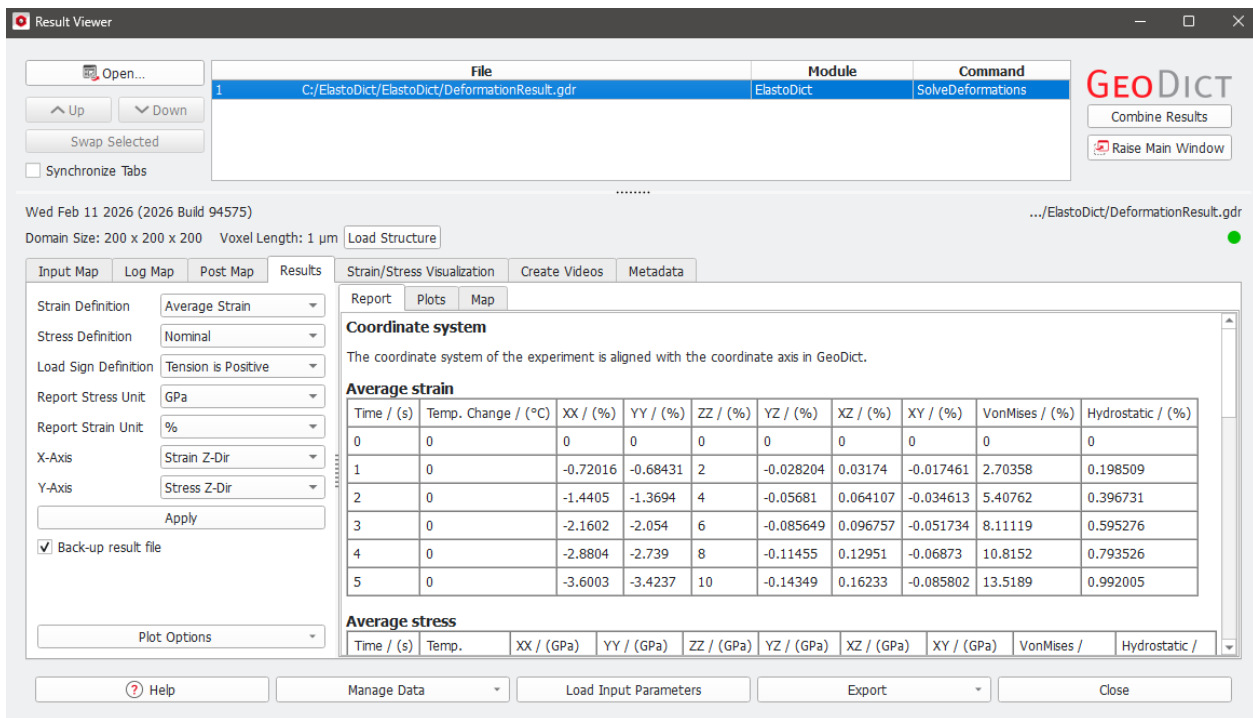
For nonlinear materials, additional state variables might be defined. For example, for plastic deformation, these are the elastic and plastic strains. For materials with damage behavior, this is the damage variable. When **Material State Variables** is checked, these variables are saved as volume fields.

✓ Export VTK File

Besides using GeoDict's own visualization tools, it is possible to export the solution in the commonly used VTK (Visualization Toolkit) format (see: <https://vtk.org/> and, <https://www.paraview.org/>).

1.3.6 Result File

After the solver has finished, the result file (.gdr) is saved in the project folder and the Result Viewer opens automatically.



For detailed information about all result viewer options and functionalities, please refer to the Result Viewer section. In the following, only those results are discussed, which are characteristic for ElastoDict Deformation results.

Deformations Results:

- [Results](#)⁷¹
Description of the Results tab of the result file.
- [Strain/Stress Visualization](#)⁷⁴
Description of the Strain/Stress Visualization tab of the result file.
- [Create Videos](#)⁸⁰
Description of the Create Videos tab of the result file.
- [Metadata](#)⁸³
Description of the Metadata tab of the result file.
- [Result Folder](#)⁸⁴
Description of the result folder.

Results

The **Results** tab gives access to the computed values for the parameters calculated by the selected ElastoDict process. The **Results** tab is divided into three subtabs:

- The **Report** subtab shows the **Average Strain** (displacement gradient when using geometric nonlinearity) and **Average Stress** values in the experiment.

Input Map

Log Map

Post Map

Results

Strain/Stress Visualization

Create Videos

Metadata

Report

Plots

Map

Coordinate system

The coordinate system of the experiment is aligned with the coordinate axis in GeoDict.

Average strain

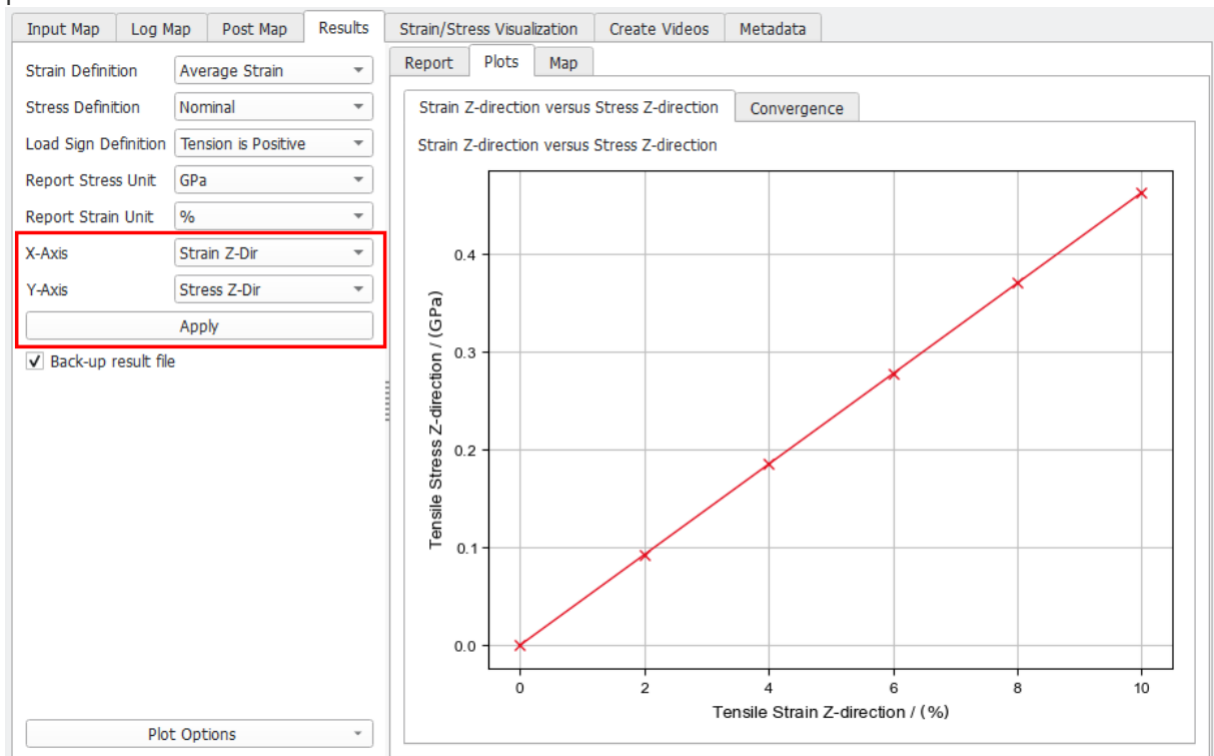
Time / (s)	Temp. Change / (°C)	XX / (%)	YY / (%)	ZZ / (%)	YZ / (%)	XZ / (%)	XY / (%)	VonMises / (%)	Hydrostatic / (%)
0	0	0	0	0	0	0	0	0	0
1	0	-0.72016	-0.68431	2	-0.028204	0.03174	-0.017461	2.70358	0.198509
2	0	-1.4405	-1.3694	4	-0.05681	0.064107	-0.034613	5.40762	0.396731
3	0	-2.1602	-2.054	6	-0.085649	0.096757	-0.051734	8.11119	0.595276
4	0	-2.8804	-2.739	8	-0.11455	0.12951	-0.06873	10.8152	0.793526
5	0	-3.6003	-3.4237	10	-0.14349	0.16233	-0.085802	13.5189	0.992005

Average stress

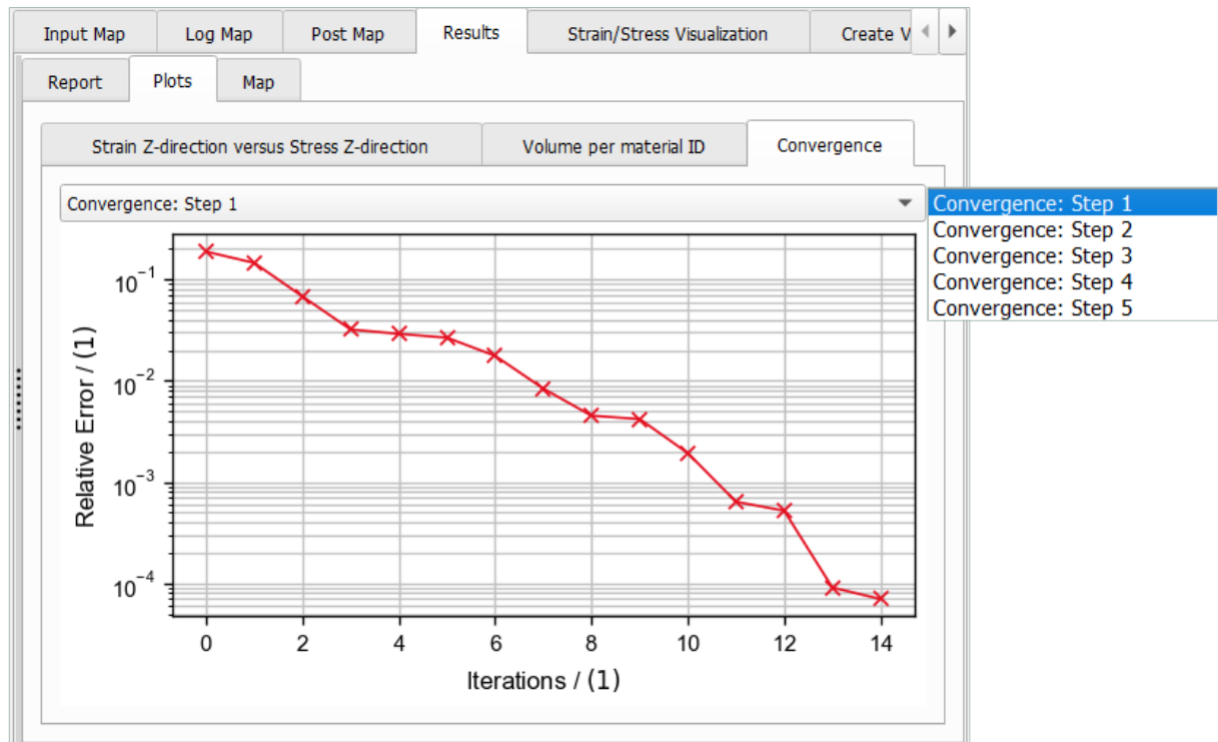
Time / (s)	Temp. Change / (°C)	XX / (GPa)	YY / (GPa)	ZZ / (GPa)	YZ / (GPa)	XZ / (GPa)	XY / (GPa)	VonMises / (GPa)	Hydrostatic / (GPa)
0	0	0	0	0	0	0	0	0	0
1	0	1.7287e-05	3.8263e-05	0.092525	5.8641e-06	-1.0071e-05	-1.0365e-05	0.0924974	0.0308602
2	0	-2.4247e-05	-3.5952e-06	0.18497	1.1866e-05	-1.4721e-05	-1.045e-06	0.184987	0.0616484
3	0	1.1603e-05	2.5796e-05	0.27747	7.9435e-06	-1.2163e-05	-1.0448e-05	0.27745	0.0925019
4	0	-1.606e-05	-6.0597e-06	0.36995	9.0828e-06	-1.3736e-05	-1.2069e-06	0.369957	0.123308
5	0	5.4725e-06	9.5366e-06	0.46244	6.6072e-06	-8.4339e-06	-6.5437e-06	0.462433	0.154152

Initial stiffness: 4.626 GPa

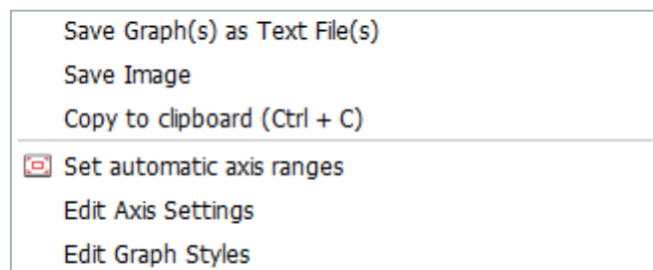
- The **Plots** tab contains three plots:
The first tab shows the **strain-stress diagram** by default. The parameters on the X/Y-Axis and the corresponding units are specified to the left of the plot. Click **Apply** to update the plot.



The second tab shows the **Convergence** for each load step of the simulation by displaying the relative error for each iteration.



By right-clicking on a plot, it is possible to adjust the plot settings, for example, the font sizes, data range etc. Additionally, the plot can be saved as an image and the data can be saved as a *.txt file.

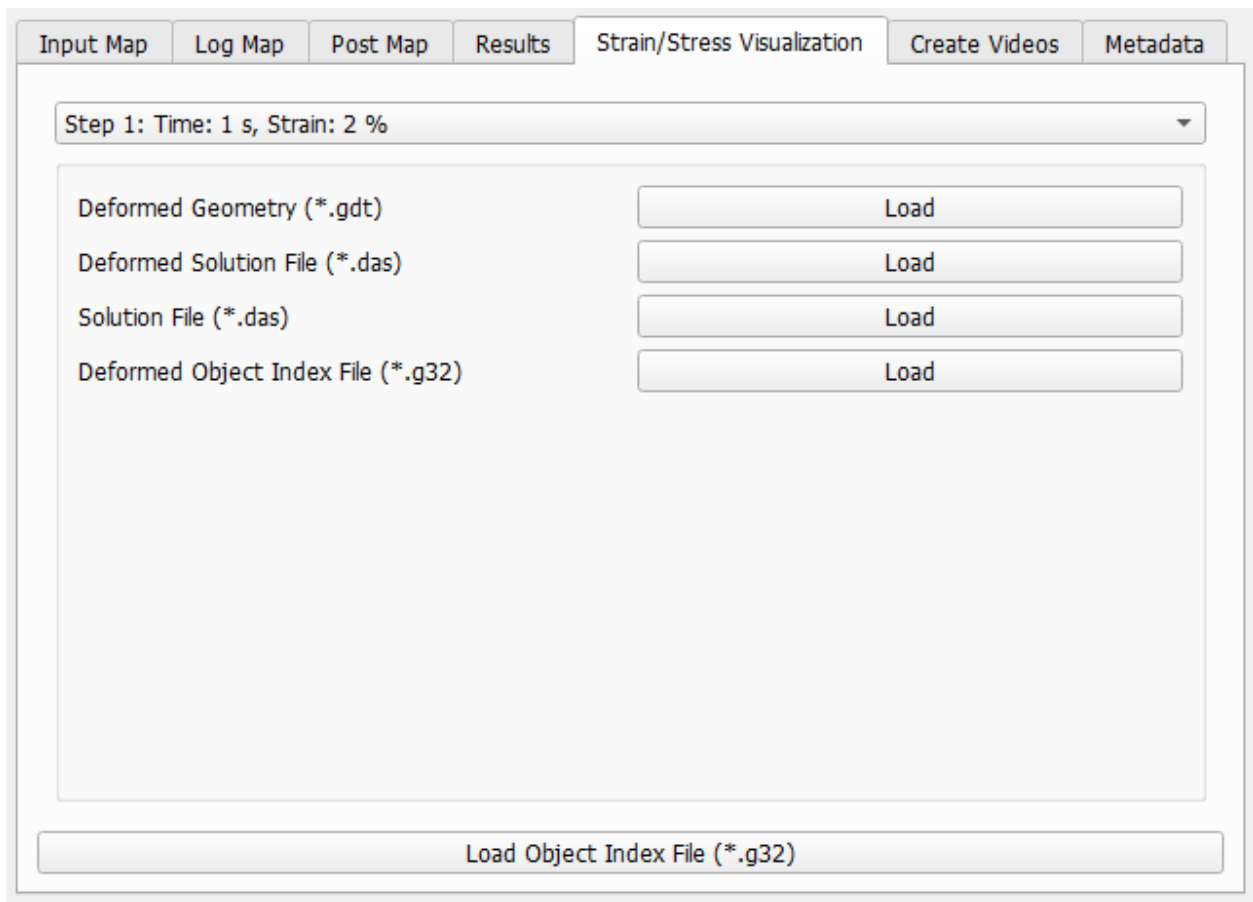


- The **Map** subtab contains all result values like strains and stresses for all load steps or volume fractions for the deformed structure.

Input Map	Log Map	Post Map	Results	Strain/Stress Visualization	Create Videos	Metadata
Report	Plots	Map				
Key	Unit	Value				
StructureFile		Structure.gdt				
Runtime	s	117.392				
PhysicalVisualization						
Information						
ErrorMessage						
InitialStiffness	GPa	4.61168				
LoadCoordinateSystem		1, 0, 0, 0, 1, 0, 0, 0, 1				
Step0						
Time	s	0				
TemperatureChange	K	0				
FluidPressure	GPa	0				
DeformedVolumeFraction	1	0.899740125, 0.100259875, 0, 0, 0, 0, 0, 0, 0, 0, 0, 0, 0, 0, 0, 0,...				
GeometryVolume	m ³	8e-12				
Strain						
Stress						
Step1						
Time	s	1				
TemperatureChange	K	0				
FluidPressure	GPa	0				
DeformedVolumeFraction	1	0.89974, 0.10026, 0, 0, 0, 0, 0, 0, 0, 0, 0, 0, 0, 0, 0, 0, 0, 0,...				
GeometryVolume	m ³	8.04514e-12				
Strain						
Stress						
DeformedGeometry		DeformedGeometry_0.gdt				
DeformedG32		DeformedObjectIndices_0.g32				
ElastoSoluton		StrainStressResult_0.das				
FlastoSolutionDef		StrainStressResult_0_deformed.das				

Strain/Stress Visualization

In the **Strain/Stress Visualization** tab, the computed result fields (*.das) and deformed geometries (*.gdt) of each load step can be loaded into the GeoDict GUI for visualization and postprocessing purpose.



Deformed Geometry

GeoDict stores deformed geometries and volume fields using boxels. While GeoDict generally operates on voxels (cubes), ElastoDict internally uses boxels, which are cuboids with varying side lengths.

ElastoDict saves the boxel structure directly. This is required to support restarting simulations with **On Deformed Geometry**, as the solver relies on the boxel structure to continue computations. Using boxels ensures that the number of elements remains constant throughout the simulation.

The resampling to a voxel structure is done when the structure is loaded in GeoDict. The smallest boxel side length is selected as the new voxel length (instead of the original voxel length as before). With this choice, more detail can be preserved especially for compression simulations and the structure is shown with more details.

It is also possible to load the structure with the original voxel length (as before in GeoDict 2021 and earlier). For this, the structure has to be imported by selecting **File → Import Structure (*.gdt)...** in the main GUI. The voxel length for the structure can be defined in **Voxel Length / (μm)**, below the **Resample to Voxels** checkbox. It is also possible to load the boxel geometry without resampling by unchecking **Resample to Voxels**.

The values for **CropX**, **CropY** and **CropZ** always correspond to the boxel geometry and do not represent the values of the resampled voxel geometry.

GDT2 Import – DeformedGeometry_4.gdt

µm

☐ Import as Analytical Structure

☒ Import as Voxel Structure

NX 200

NY 200

NZ 200

This file contains a boxel geometry.
By default, it will be re-sampled to the chosen voxel length.

Boxel Length X / (µm) 0.964377828

Boxel Length Y / (µm) 0.965049142

Boxel Length Z / (µm) 1.1

☒ Resample to Voxels

Voxel Length / (µm) 0.964377828

CropX 0 200 NX: 200 (193 µm)

CropY 0 200 NY: 200 (193 µm)

CropZ 0 200 NZ: 200 (193 µm)

Cancel Import

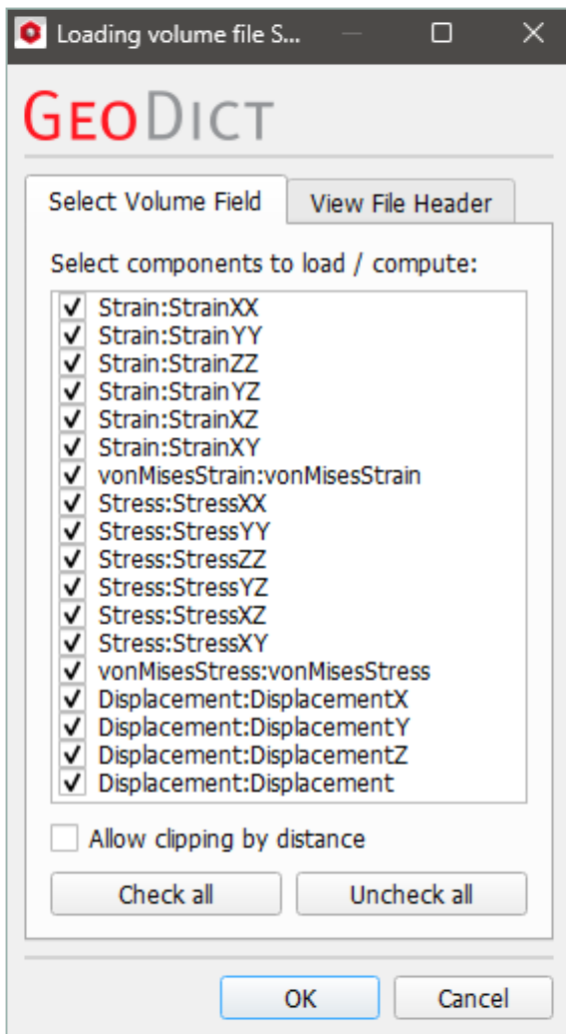
Solution File

Depending on whether the calculation was made on deformed geometry or not (see [Geometry Handling](#)), the corresponding *.das file is available and can be loaded for each calculation step. In the example below, **Calculate on Deformed Geometry** in the **Geometry Handling** tab was unchecked, and thus only the **Solution File (*.das)** can be loaded.

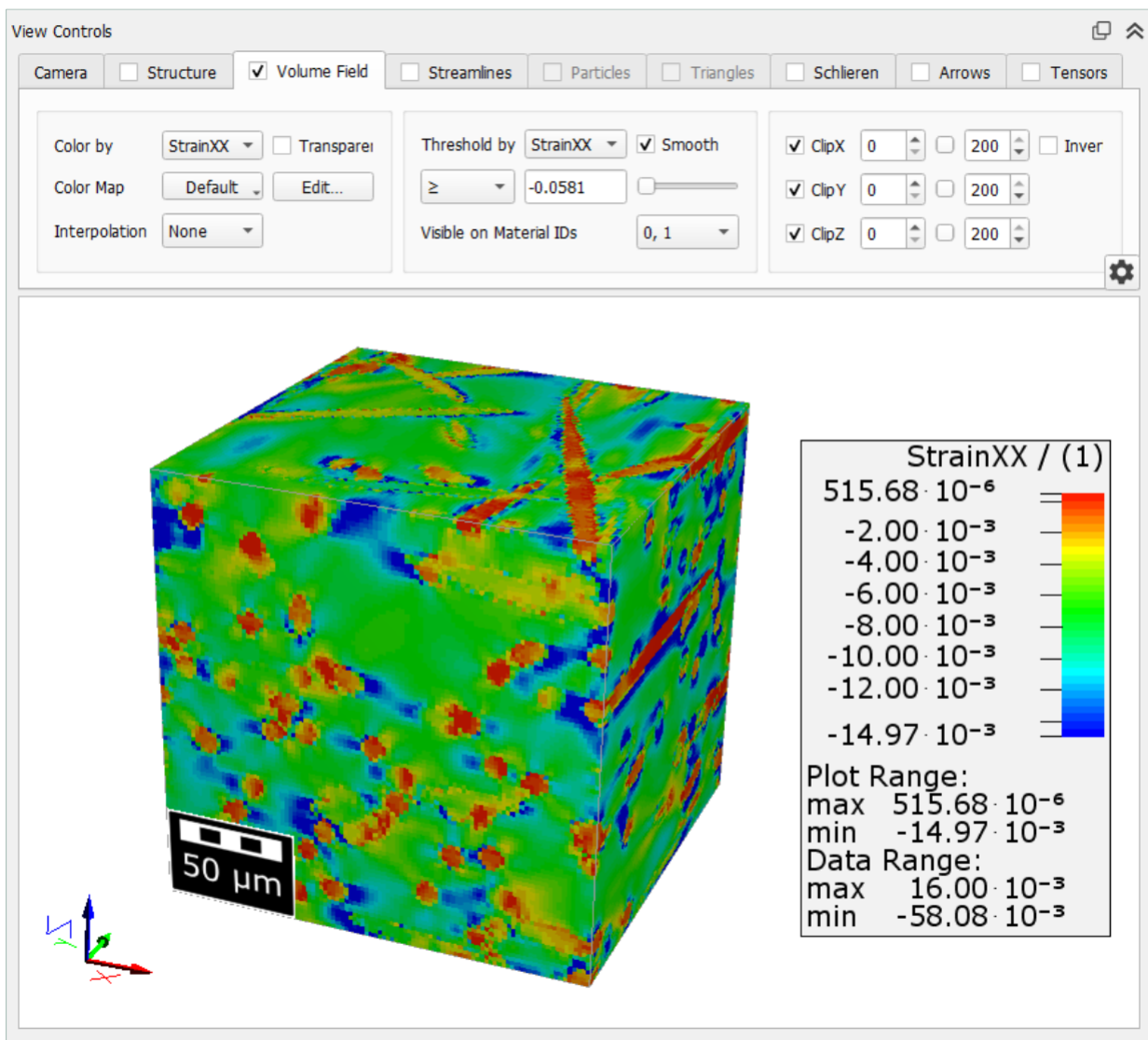
Deformed Solution File (*.das)	unavailable
Solution File (*.das)	Load

By clicking on **Load** (for both solution file options) a selection dialog opens. The result values, which are checked here, are visible in the **View Controls – Volume field** tab in the GeoDict GUI after the loading step is finished.

This is also the list of available result values which can be used if the **Manual** option is chosen for the video generation process, see [Create Videos](#)^[80]. The choice of volume fields has to be made in the [Output](#)^[68] tab of the Large Deformation Options dialog before running the simulation.



After loading all required components, the visualization settings can be set for these values separately.

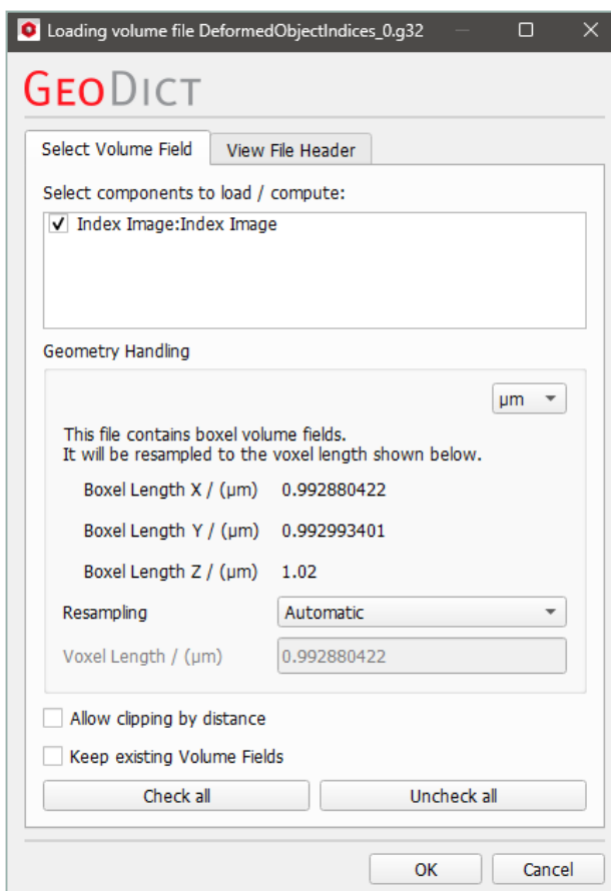


Object Index File

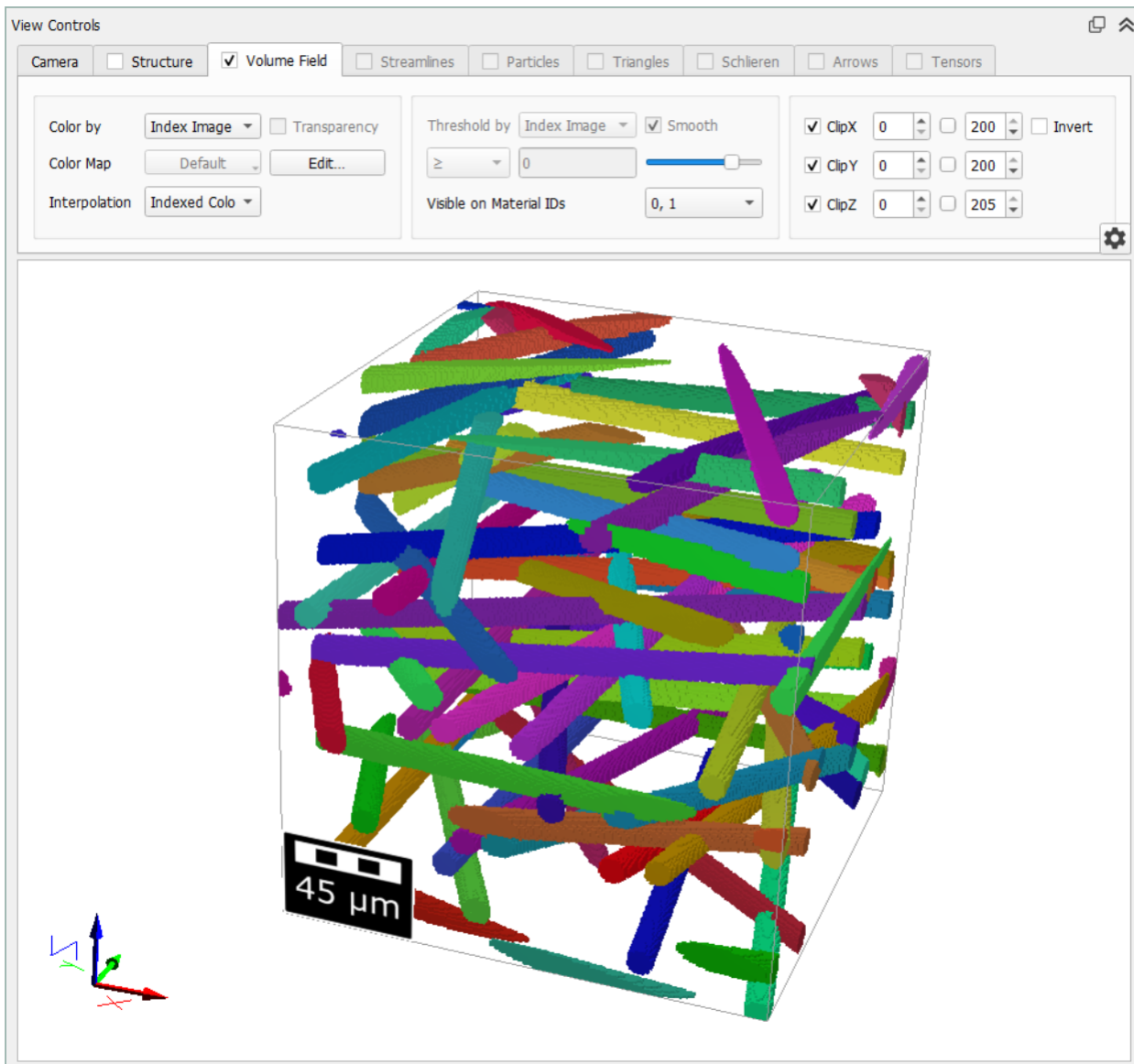
The last two options in the **Strain/Stress Visualization** tab of the Result Viewer are for loading the **Object Index File (*.g32)**. In case the calculations were done on deformed geometry with **Enable Object Tracking (*.g32)** checked (see [Geometry Handling options](#)^[61]), the corresponding index file can be loaded.

By clicking on the **Load Object File Index (*.g32)** button at the bottom of the **Stress/Strain Visualization** tab the volume field with object indices of the undeformed structure is loaded.

The object indices of the deformed structure can be loaded by choosing **Load for Deformed Object Index File (*.g32)**. A dialog will open with a list of indices to choose. Additionally, the information about the [resampling from boxels to a voxel](#)^[75] structure procedure is displayed.

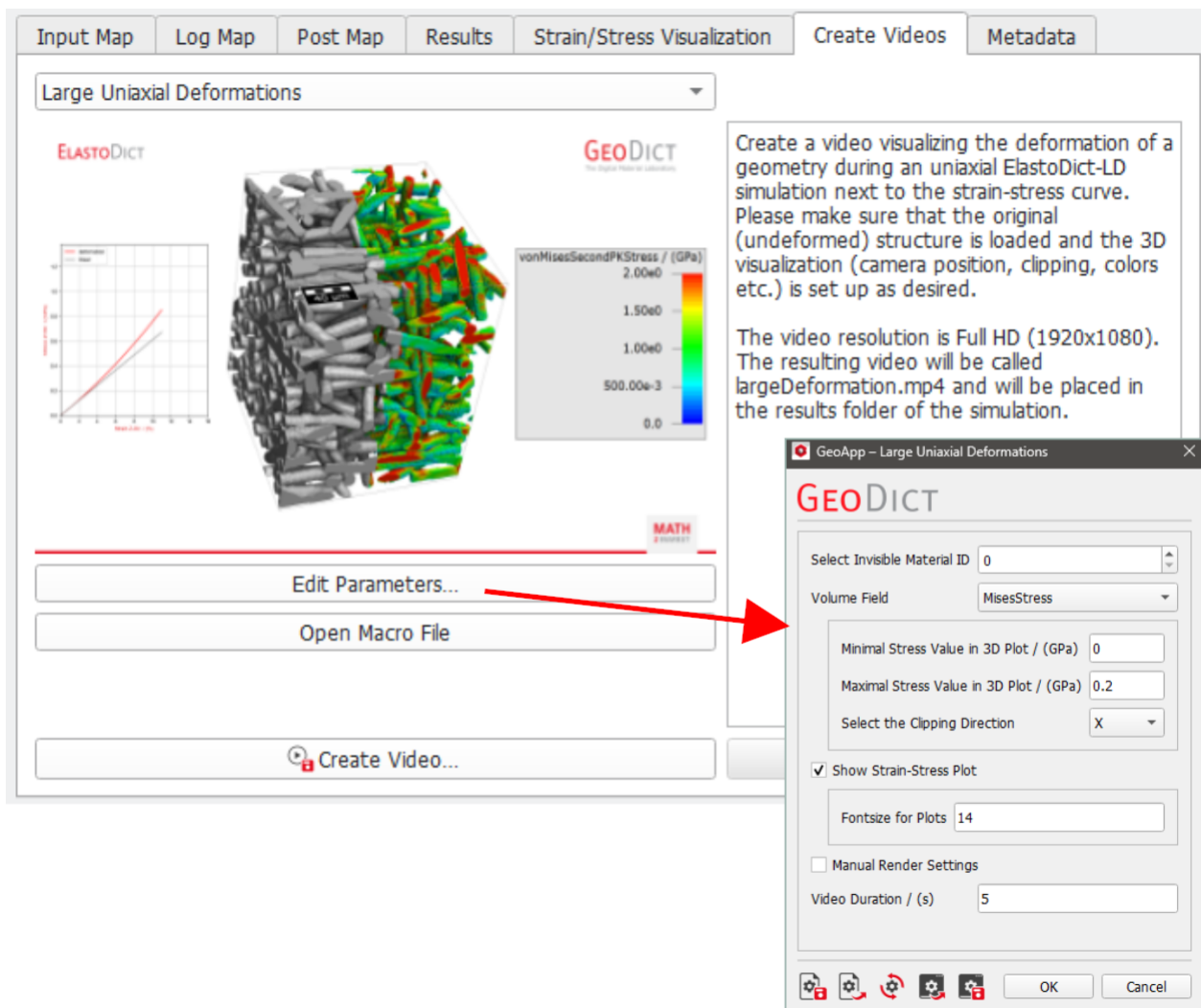


Loading the Object Index File results for both cases in a colored representation of the different objects in the **Volume Field** tab. More information of each object can be viewed in the tooltip dialog.

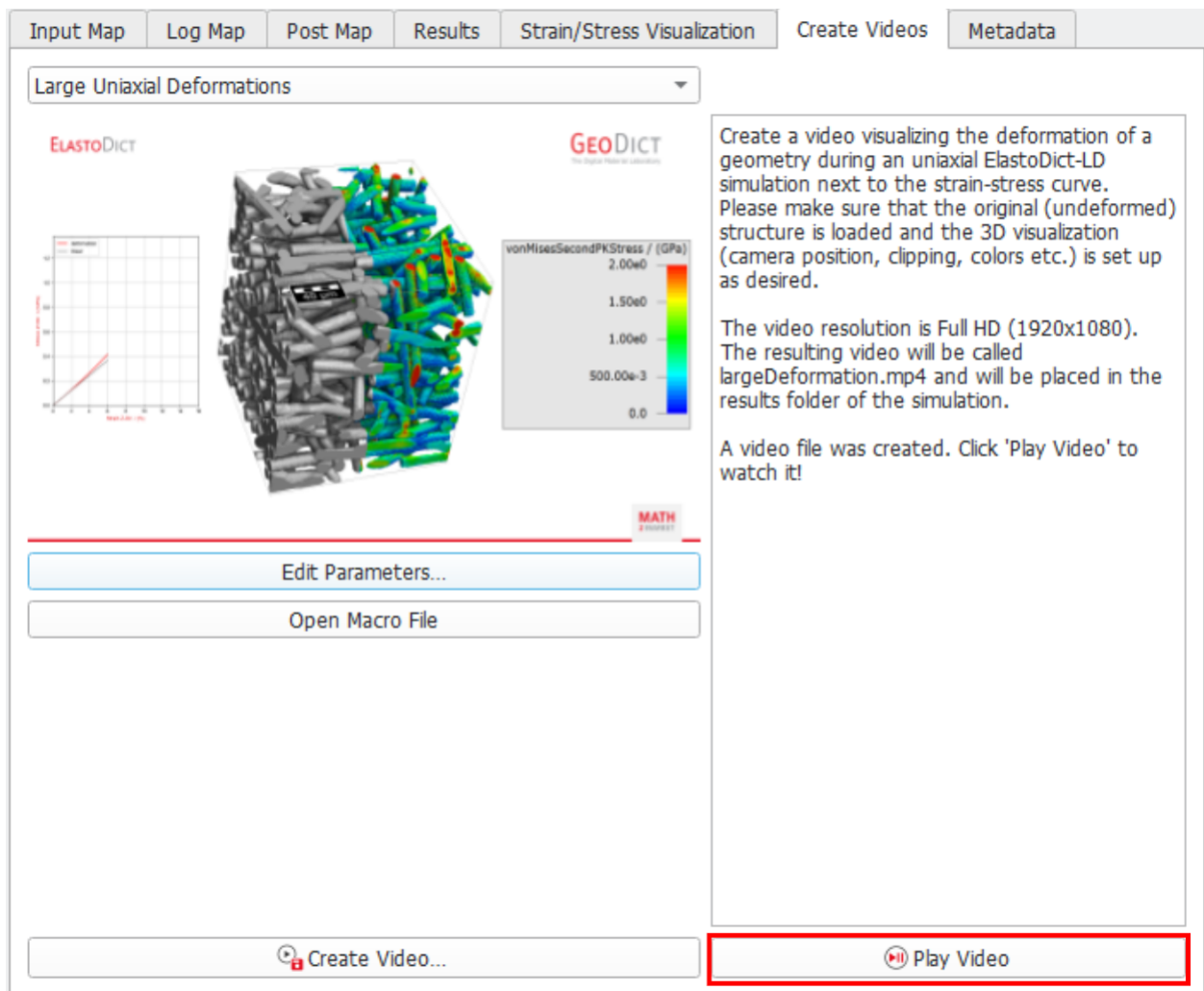


Create Videos

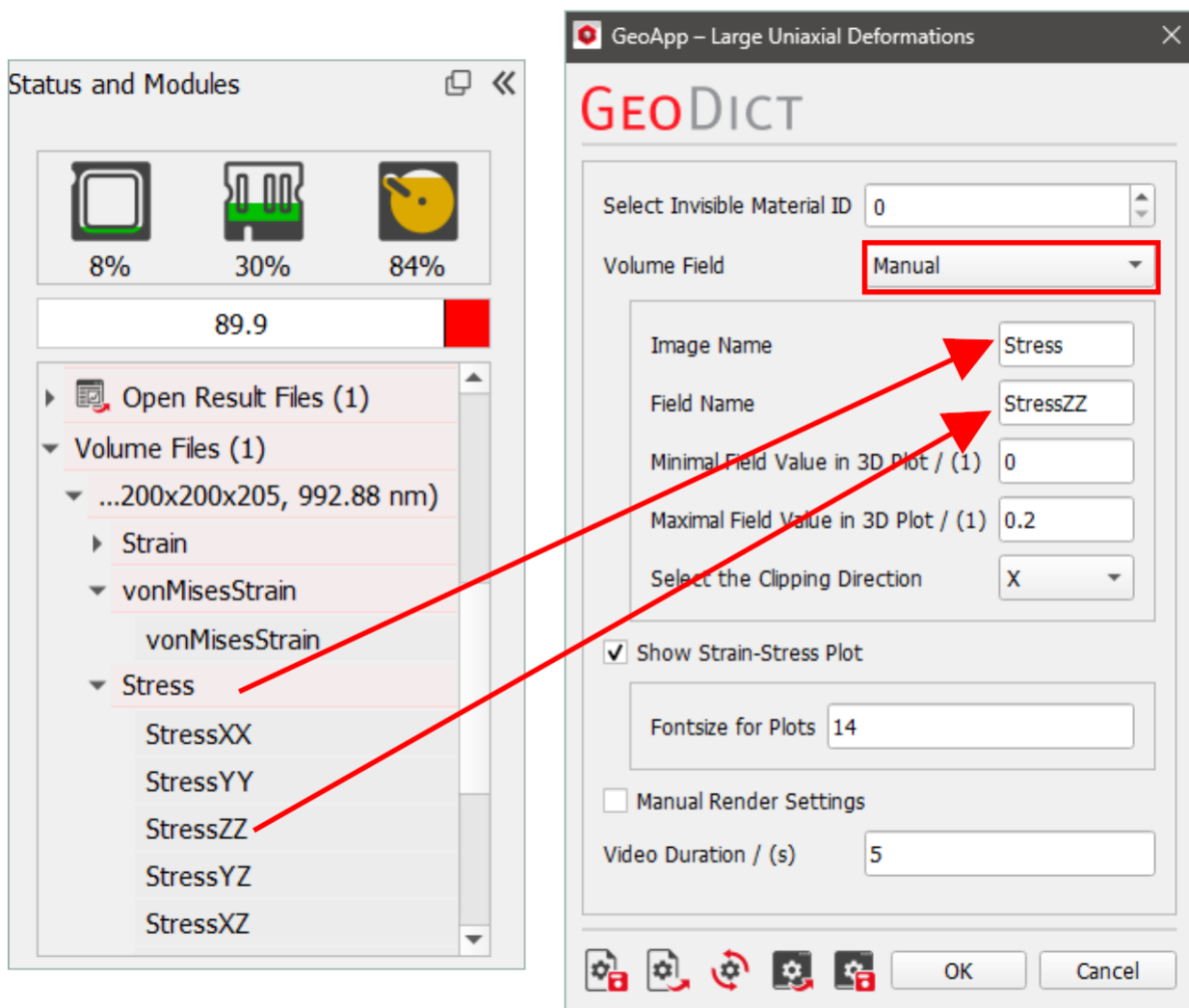
The **Create Videos** tab allows to create a video of the current large deformation simulation directly from the results viewer. Click **Create Video...** to use the default settings or **Edit Parameters...** to set them to your needs. The video creation is performed with an underlying python macro which can be opened and edited with the **Open Macro File** button.



After the parameters are set, click **Create Video...** to create the video. After the generation is finished, the video file is saved in the result folder and can be opened with the **Play Video** button.



Since GeoDict 2024, it is also possible to create videos for other results than for von Mises strain or stress. For this, choose the **Manual** option in the **Edit Parameters** dialog and define the result value you want to record. The correct names for **Image Name** and **Field Name** can be looked up in the **Status and Modules** section of the GeoDict GUI or the [solution field loading](#) dialog.



Note that **Deformed Solution Files** have to be calculated in the simulation, and all volume fields that are to be visualized in the video have to be checked in the [Output](#) tab. Before starting the video generation, the original geometry needs to be loaded.

Metadata

The **Metadata** tab contains a free text panel for additional input. Additionally, it contains the macro parameters when the simulation was started through a macro.

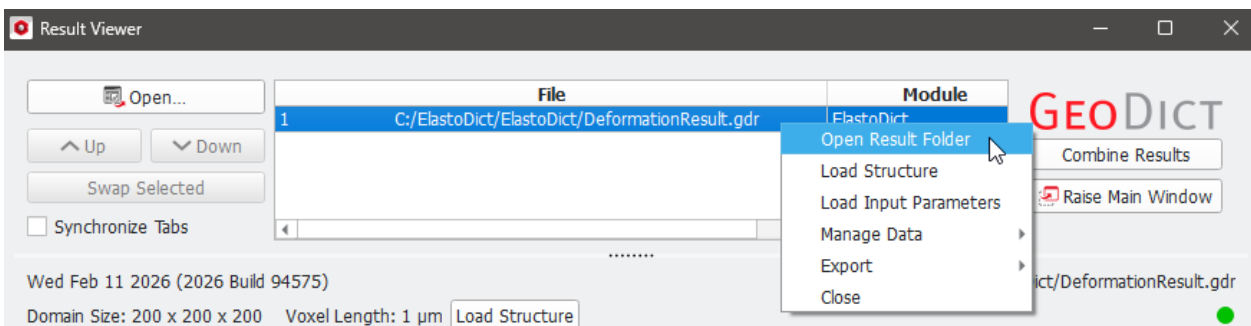
Result file for GeoDict 2026
created at 11:28:36 on 2026Feb11
by Support of Math2Market GmbH.

Parameter Map

Key	Unit	Value
StructureDescription		FeelMathLDResult
StructureFile		Structure.gdt

Result Folder

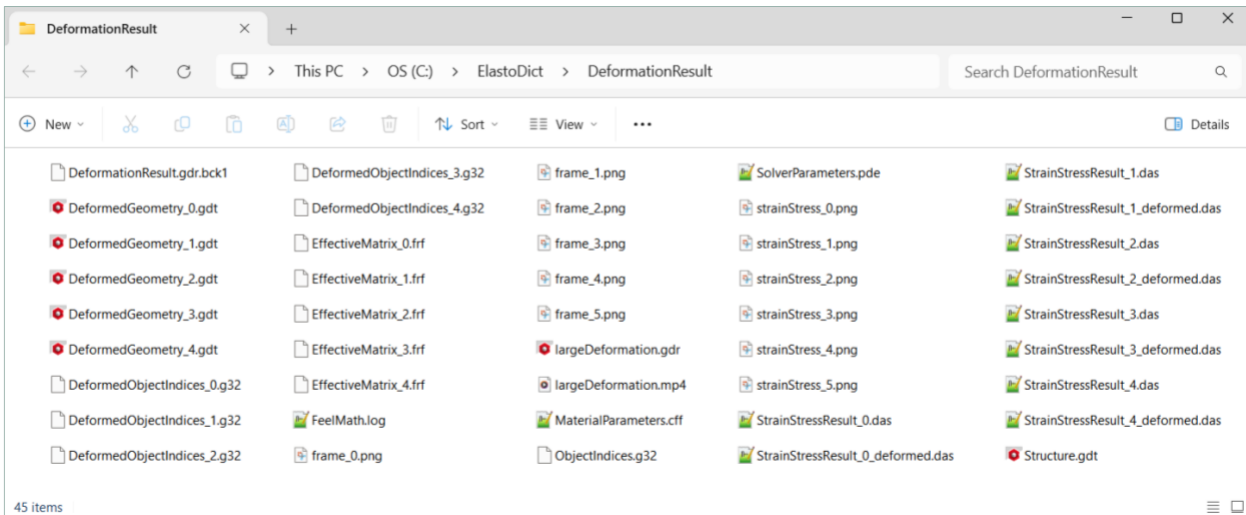
The result folder has the same name as the result file and is saved in the current project folder. It can be opened directly from the Result Viewer by right-clicking on the result file name and selecting **Open Result Folder**.



The result folder contains the following file formats:

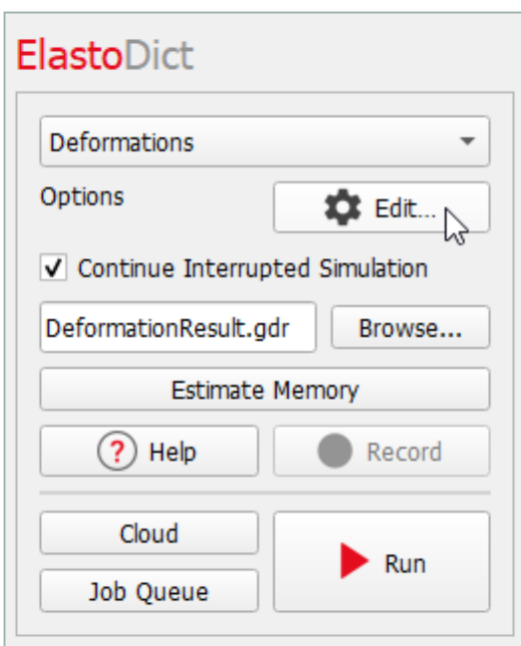
- ***.gdt**: geometry data, e.g., the original structure and files with the deformed geometry (e.g., `DeformedGeometry_1.gdt`)
- ***.frf**: Free Report Form file (e.g., `EffectiveMatrix_2.frf`). It contains the results of each solution step. This information can also be found in the Results tab in the Results viewer.
- ***.cff**: coefficient file format (e.g., `MaterialParameters.cff`). This file contains the material properties of the materials in the structure as input data for the solver.
- ***.das**: e.g., `StrainStressResult_0.das`. These files contain the result fields and can be visualized in GeoDict. They can be loaded directly from the results viewer.
- ***.log**: e.g., `FeelMath.log`. A log file of the current solver run.
- ***.pde**: e.g., `SolverParameters.pde`. This file contains the parameters of the current simulations as input data for the FeelMath solver.

- ***.fmr** for `FeelMathRestart.fmr`: If a simulation is aborted, a restart file might be written which contains information to be used for a restart of the simulation. These files can be quite large, therefore they should only be kept if the simulation will be restarted later.
- ***.gdr.bck[i]**: e.g., `DeformationResult.gdr.bck1`: result file back-ups, which are generated if **Back-up result file** is checked in the **Result Viewer** → **Results** tab.



1.3.7 Restarting a simulation

Deformation simulations can contain many steps, and therefore it may be necessary to interrupt a simulation. Such simulations can be restarted with the option **Continue interrupted simulation**. Click **Browse** to select the corresponding result file and then **Run** to restart the simulation.



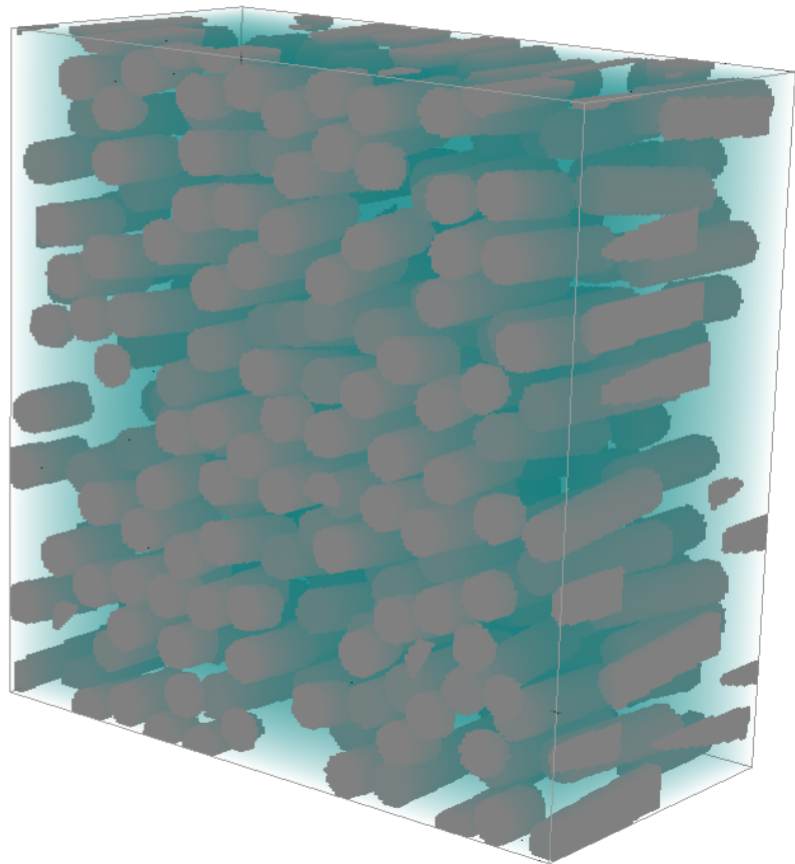
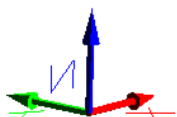
For more information about the requirements for continuing interrupted simulations, see [Allow Restart for Deformation Simulations](#)⁶⁹.

If a simulation cannot be continued, a corresponding error message is shown.

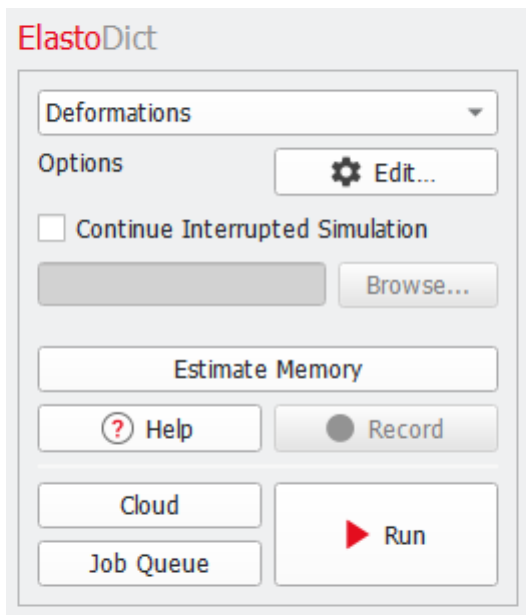
1.3.8 Example: Matrix Damage

ElastoDict-Deformations can be used to determine matrix damage in a composite model. In this example, E-Glass fibers are combined with an epoxy matrix. The structure used for this example is generated with FiberGeo.

Material Information:
ID 00: Epoxy (3501-6)
ID 01: Glass

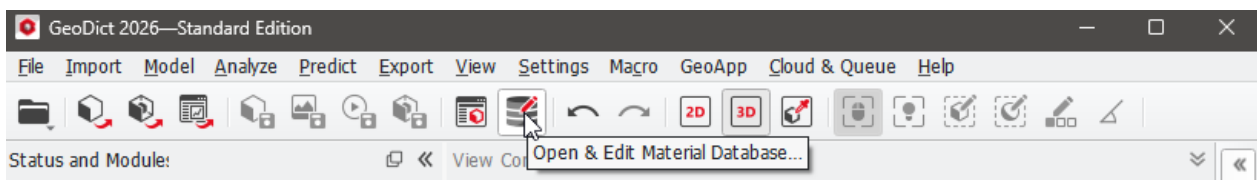


Select **Deformations** from the pull-down menu in the ElastoDict module section.

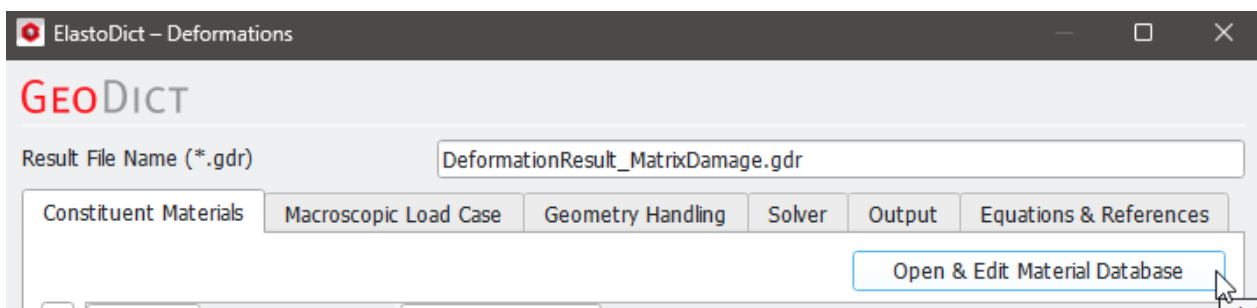


Model damage for the matrix material

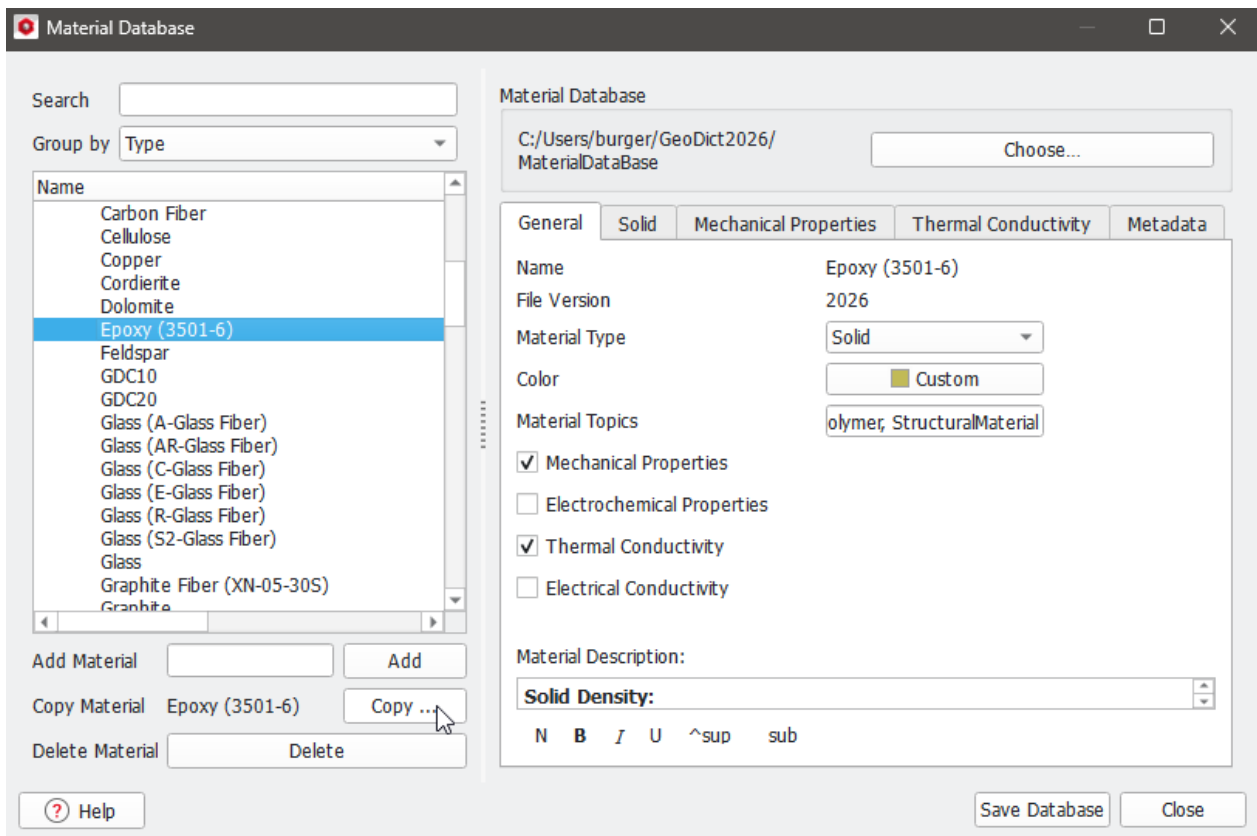
In this example, the matrix material is supposed to take damage under a tensile load. Therefore we want to add a damage model to the material *Epoxy (3501-6)*. You can directly change the predefined material stored in the Material Database or, as is recommended, define a new material by copying the existing one and add the damage model. Therefore, open the Material Database by either clicking on the corresponding button in the main UI



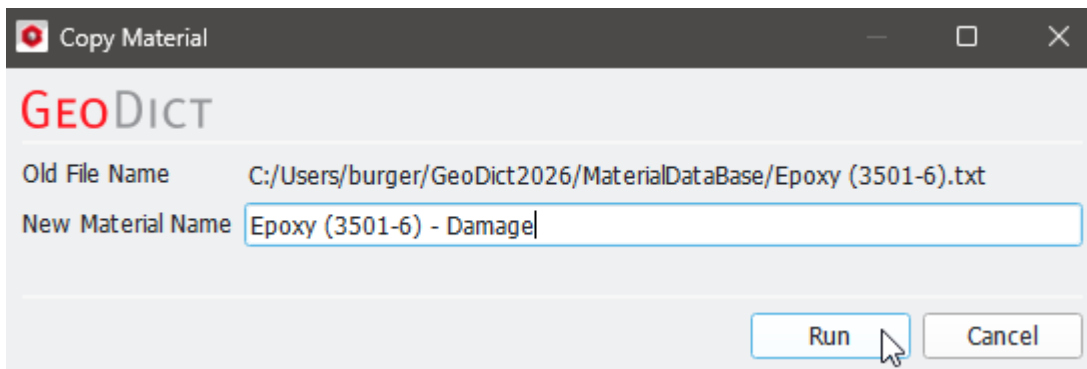
or by clicking on Open & Edit Material Database:



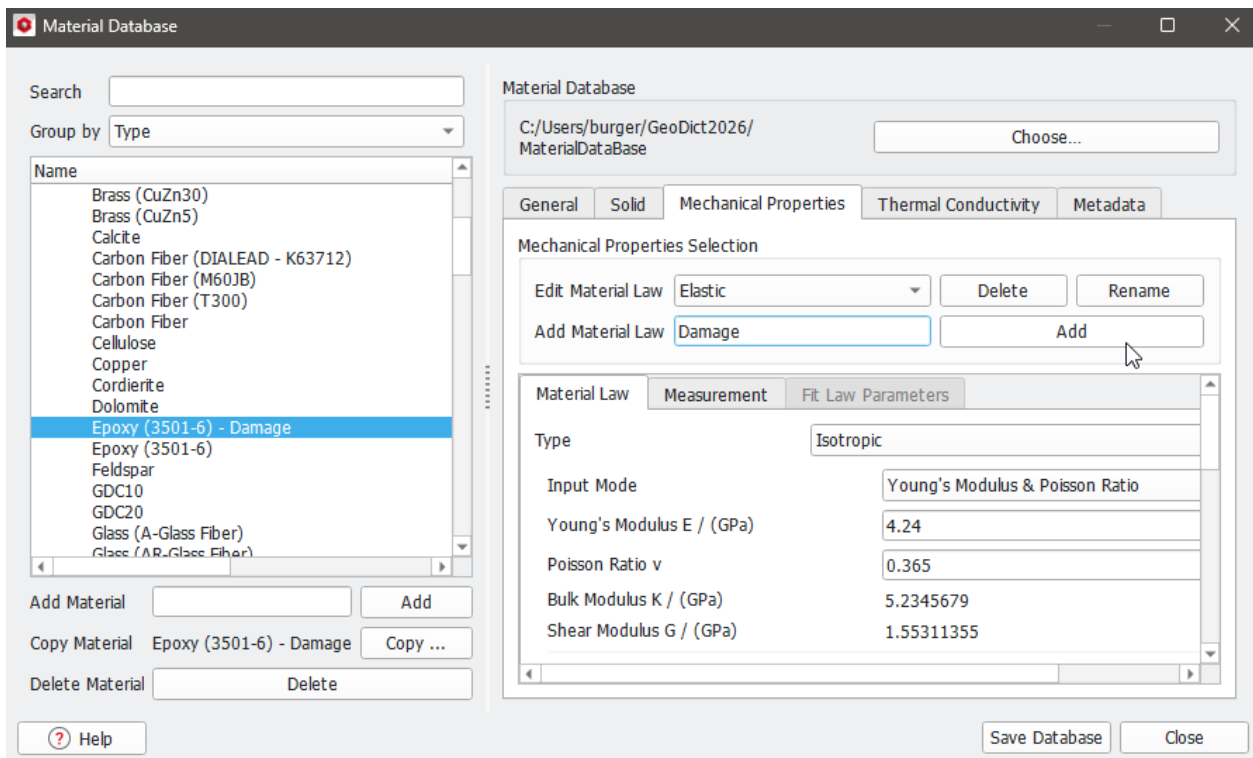
In the Material Database dialog first click *Epoxy (3501-6)* and then on **Copy...**



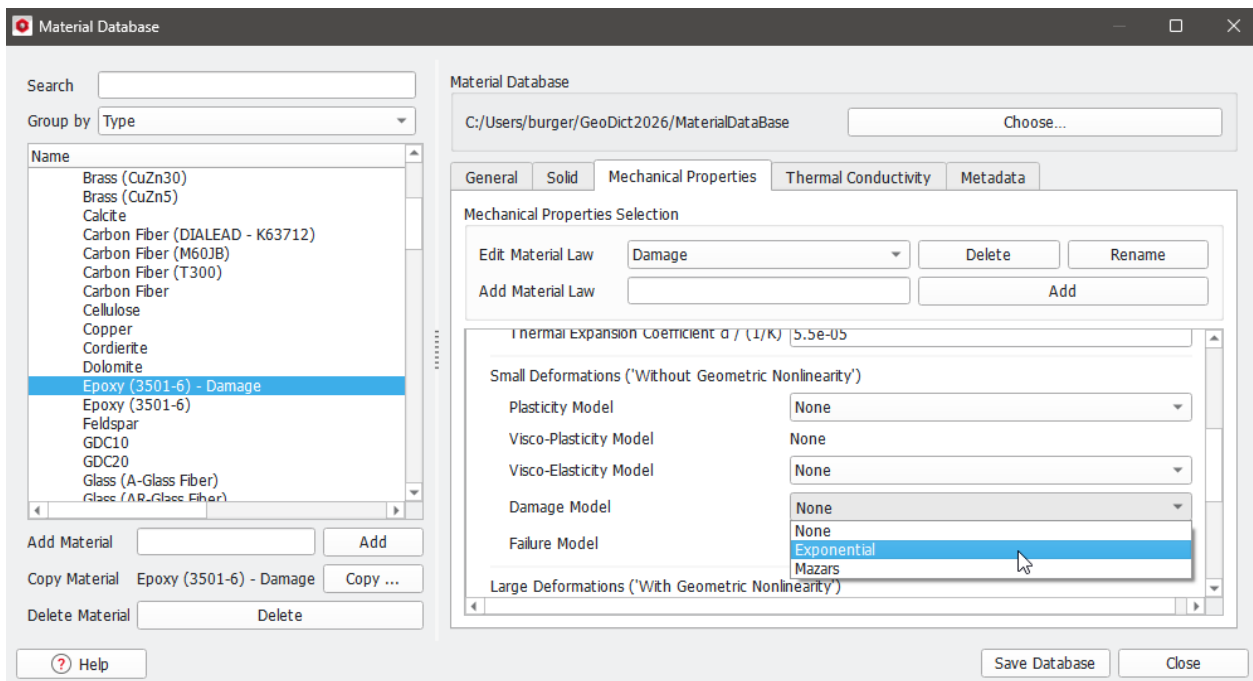
Now, you can choose a name for the copy, e.g., *Epoxy (3501-6) - Damage*, and click on **Run**.



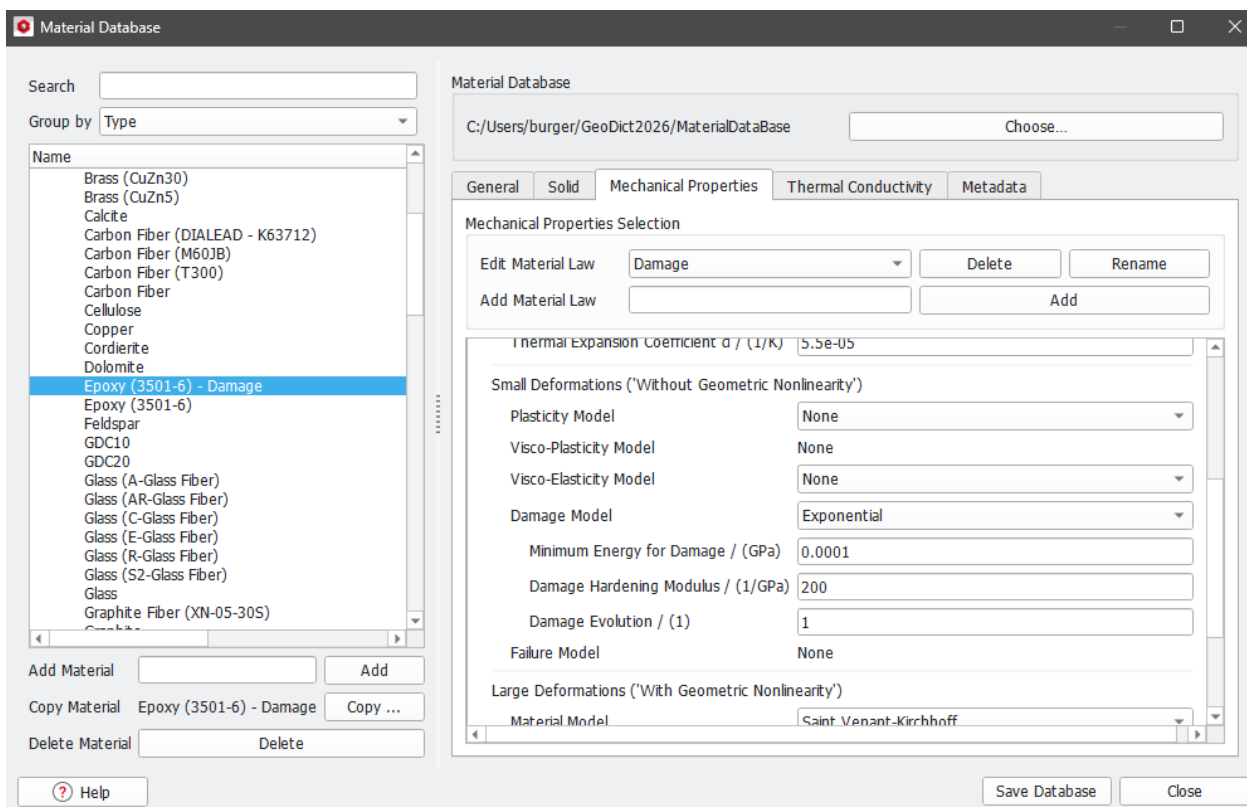
Now, you can choose the new material *Epoxy (3501-6) - Damage* in the Material Database and define damage by **Adding** a **Material Law** under the **Mechanical Properties** tab. Therefore, enter the name of the new law (*Damage* for this example) into the dialog box and click on **Add**.



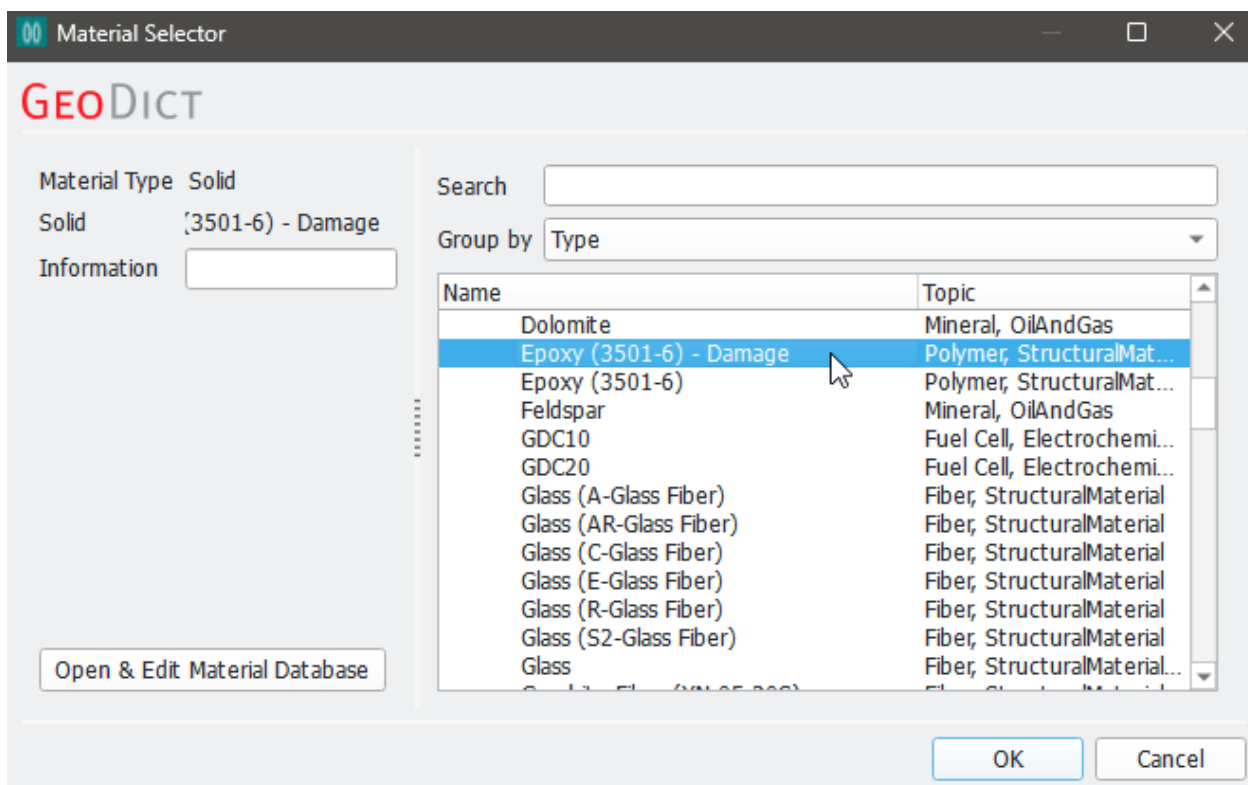
The material law Damage is visible under **Edit Material Law** automatically and you can choose which damage law you want to use by expanding the dropdown menu under **Damage Model**. For this example, we choose the [Exponential damage model](#)¹⁷.



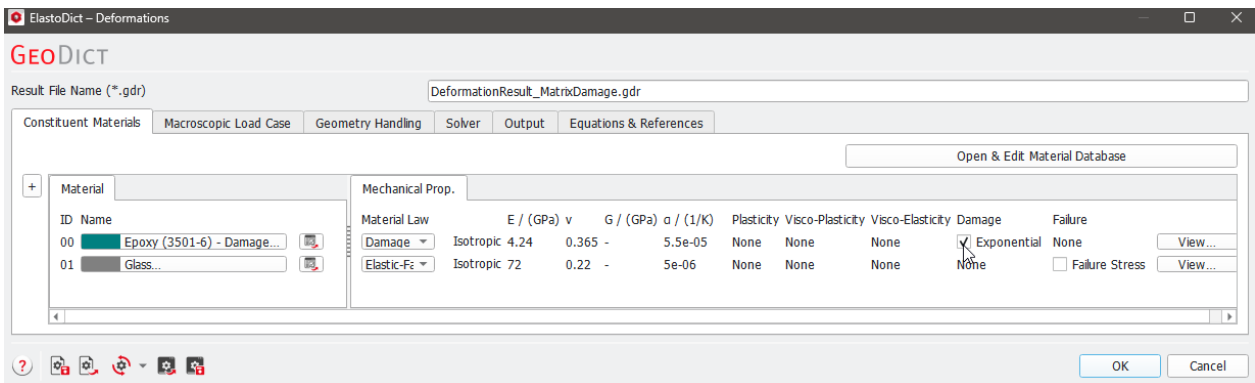
After choosing the damage model, the material parameters have to be defined. For this example the exponential damage is defined by the following parameters:



After all material changes are done, click on **Save Database** to save these changes to your local Material Database and **Close** to close the editor. Now, you can choose the newly defined material under the **Constituent Materials** tab in the **Deformations Edit** dialog.



Under the **Mechanical Properties** tab choose Damage for **Material Law** and activate the Exponential Damage by checking the Exponential checkbox.



For the glass fibers choose the *Glass* with the *Elastic-Failure (E-Glass)* material model. The **Failure Stress** option for *Glass* can additionally be enabled, although it is not necessary for this example.

Set up all other parameters

Under the **Macroscopic Load Case** tab set the experiment to **Uniaxial Experiment - Tensile**. A maximal strain of 1% is applied in 20 steps, therefore

- choose Predefined Shape,
- set Magnitude to 1%,
- set the Length to 20s,
- and the Number of Steps to 20.
- Click apply to set the load in the Load Table.

For damage simulations, it is useful to choose small steps for the load (here, 0.05% strain per step).

ElastoDict – Deformations

GEODICT

Result File Name (*.gdr)

Constituent Materials Macroscopic Load Case **Geometry Handling** Solver Output Equations & References

Experiment Experiment Conditions

Load Type In Tangential Direction

Load Case

Direction X-Direction

Plane Y-Direction

Angle in Plane / (°)

Fluid Pressure

Pressure Mode

Fluid Pressure / (GPa)

Load Table Load Graph

Time / (s)	Strain / (%)	Temp. Change / (K)
15 15	0.75	0
16 16	0.8	0
17 17	0.85	0
18 18	0.9	0
19 19	0.95	0
20 20	1	0

Number of Rows

Load... Save...

Predefined Shape

Shape

Magnitude / (%)

Temperature Change / (K)

Pressure Change / (GPa)

Length / (s)

Number of Steps

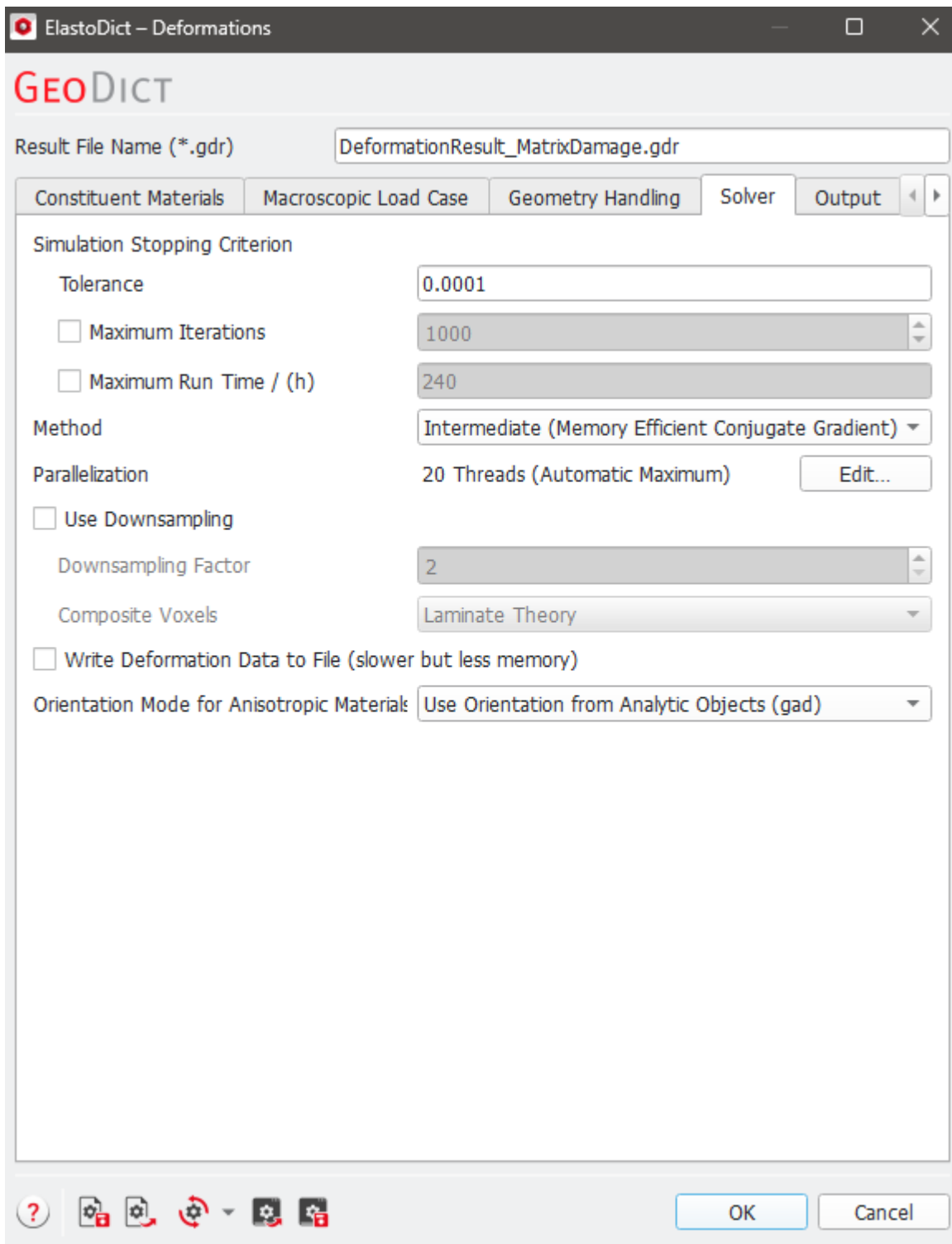
Apply

Boundary Conditions

☒ Periodic ☐ Symmetric ☐ Mixed

OK Cancel

Under the **Geometry Handling** and the **Solver** tabs, keep the default settings. In particular, keep the **Tolerance** at the default of 0.0001 since a high accuracy is important, especially for simulations with damage.



Under the **Output** tab, uncheck **Write Deformed Geometry** since generating the deformed geometries takes additional computation time and the deformed geometries are not relevant in most damage simulations (since the strain at the point of failure is often very low).

Analogously, writing volume fields might take a significant amount of disk space and time, depending on the size of the structure. Some disk-space and computational time is saved by unchecking stress, strain, and displacement fields which do not need to be visualized.

Therefore, keep only the volume fields **Stress - Von Mises** and **Strain - Von Mises** and the **Material State Variables** (containing the damage variable in this example).

ElastoDict – Deformations

GEODICT

Result File Name (*.gdr)

Constituent Materials Macroscopic Load Case Geometry Handling Solver **Output**

Write Steps

☒ Write All Steps

Write Result Fields (*.das) for Each n-th Step

Deformed Geometry

☐ Write Deformed Geometry

☐ Write Volume Fields for Deformed Geometry

☐ Write Volume Fractions for Deformed Geometry

☐ Allow Restart for Deformation Simulations

Write Volume Field

Displacement ☐ X ☐ Y ☐ Z

Stress ☐ XX ☐ YY ☐ ZZ ☐ YZ ☐ XZ ☐ XY ☒ Von Mises

Strain ☐ XX ☐ YY ☐ ZZ ☐ YZ ☐ XZ ☐ XY ☒ Von Mises

☒ Material State Variables

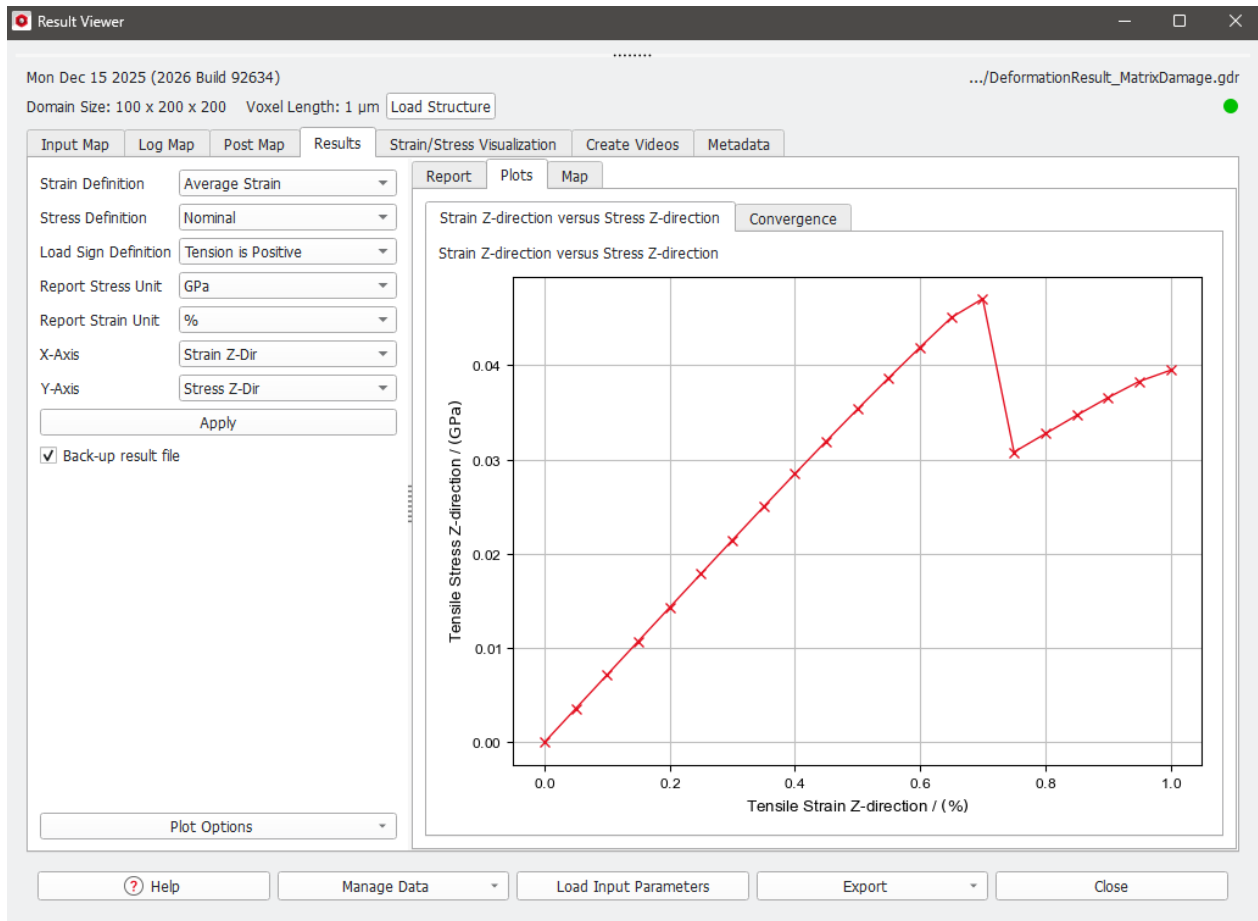
☐ Export VTK File (*.vti)

Click **OK** to close the **Options** dialog and start the simulation by clicking **Run** in the GeoDict GUI main screen.

Results

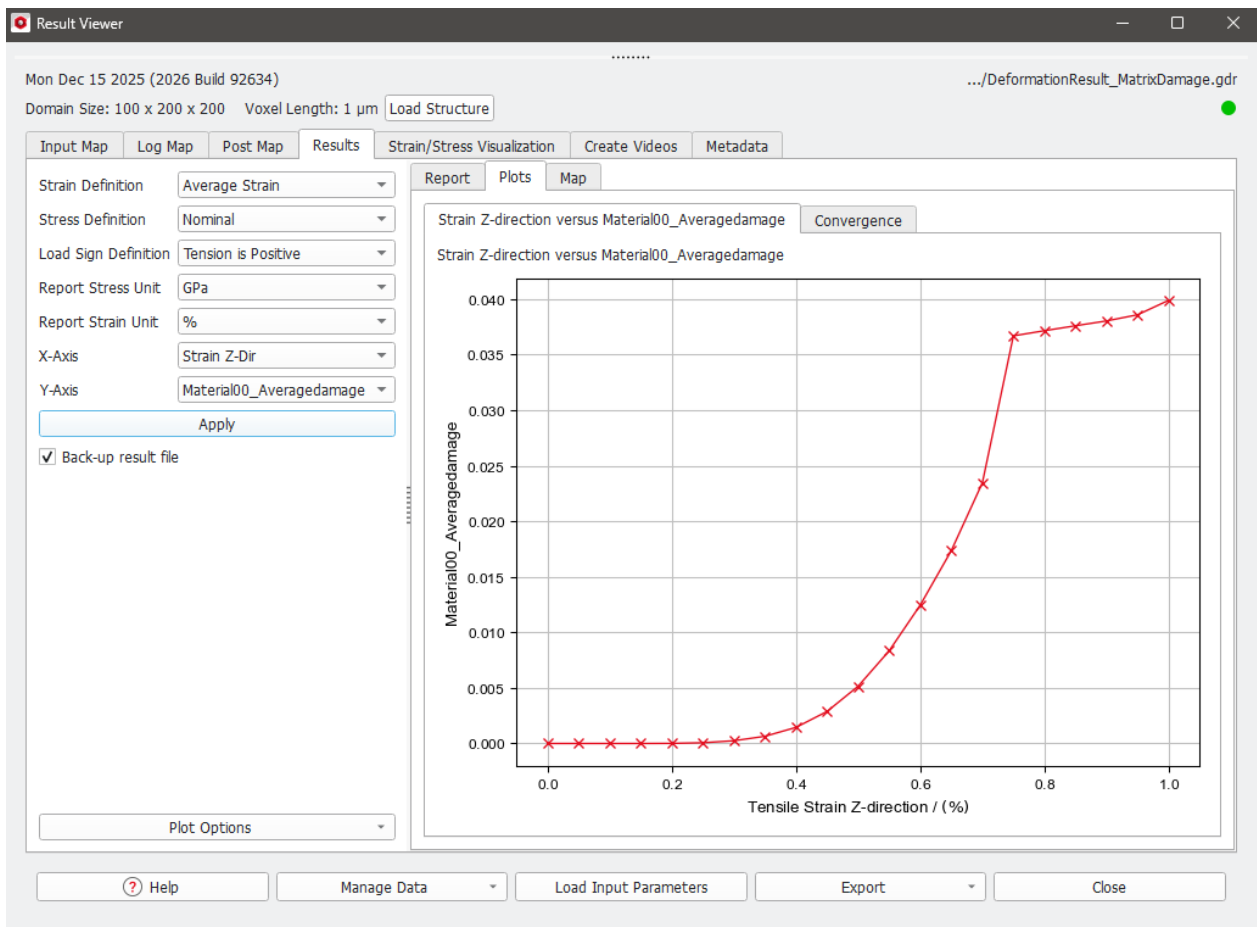
After the simulation is finished, the result file is opened automatically in the Result Viewer at the **Results - Report** subtab.

Under the **Results - Plots** subtab, the results of the deformation during the time series experiment are plotted as **Stress-Strain** chart (X-Axis: Strain Z-Dir; Y-Axis: Stress Z-Dir). Increasing damage leads to decreasing stiffness on the geometry.



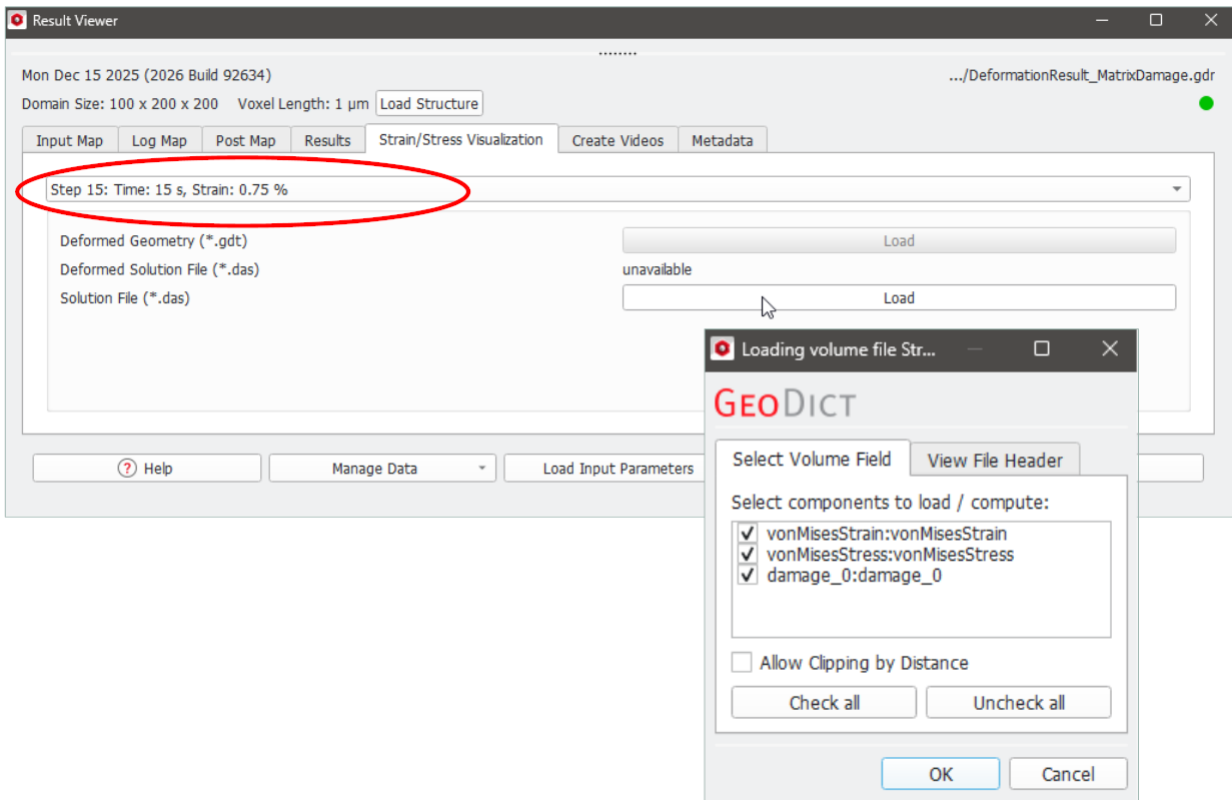
Choose **Material00_Damage** for the Y-Axis and click **Apply...** to plot the average matrix damage in the the matrix material (0 is no damage, 1 is maximal damage).

As shown in the plot below, the average damage rises steadily until the point of failure, which is indicated by the drop in the stress-strain curve at a strain of 0.75% in the upper image. After failure the damage continues to increase slightly.



Under the **Strain/Stress Visualization** tab volume fields for specific **Time Steps** can be loaded. Therefore, selected the desired time step from the pull-down menu (here, the point of failure at 0.75% Strain and 15s is selected) and load the Solution File by clicking **Load**.

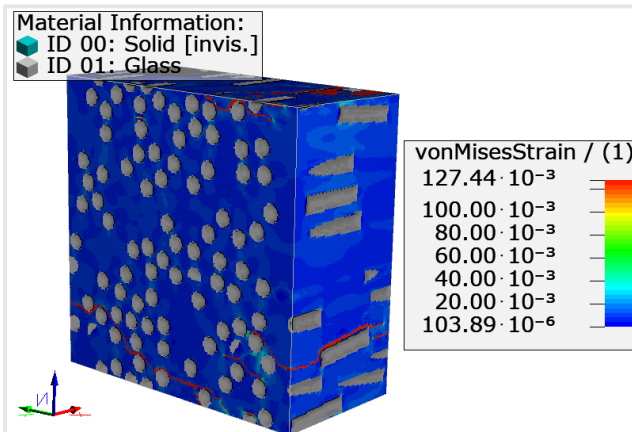
After clicking on Load a new dialog opens with all available volume fields listed. Check the Volume Fields to be loaded and visualized. In this example *von Mises Strain*, *von Mises Stress* and *Damage* are available. The numbers 0 in *Damage_0* stand for the Material IDs to which the volume field belongs.



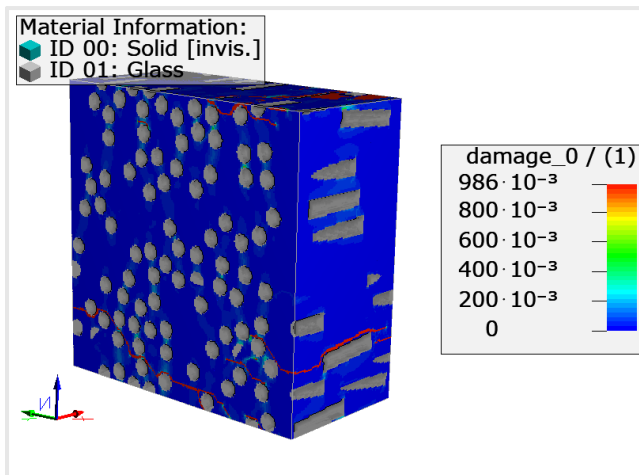
Click **OK** to load the components.

In 2D and 3D Rendering, a color bar appears to the right (default position) during the visualization of the result file, indicating the value range of the selected component.

Further visualization parameters can be set under the **Volume Field** tab, in the Visualization panel above the Visualization Area. For more information, please refer to chapter Visualization.

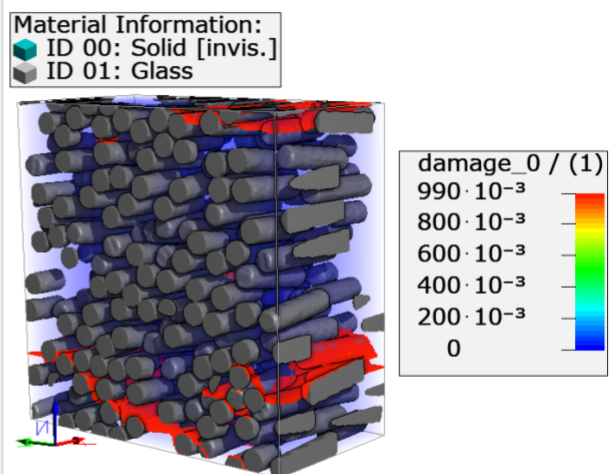
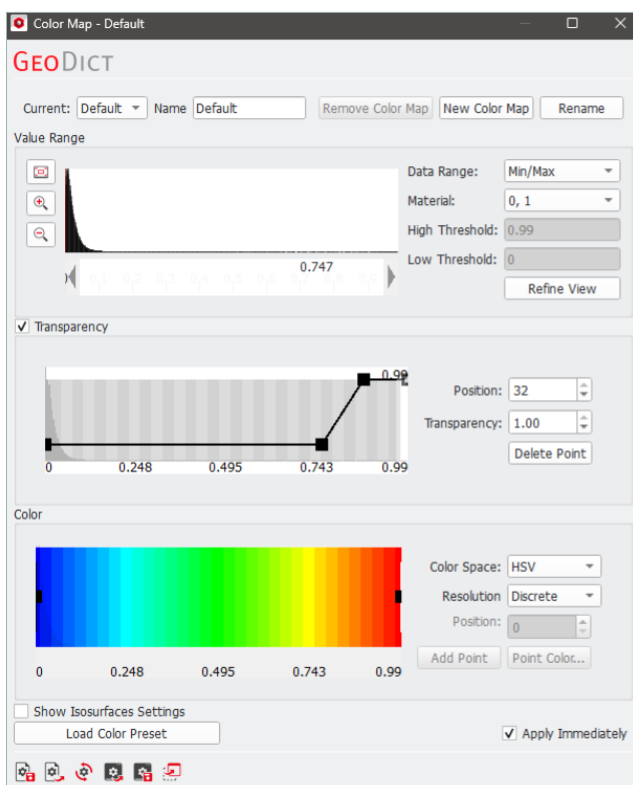


The highest strain can be observed at cracks. There, the stiffness of the matrix material is lowered due to the damage, which leads to high strain values and low stress values.



When damage is visualized, it is recommended to set the Data Range in the Color Map dialog to Min/Max or manually to a minimal value of 0.0 and a maximal value of 1.0 (see also screenshot on the next page). Then, the fully damaged areas are clearly visible.

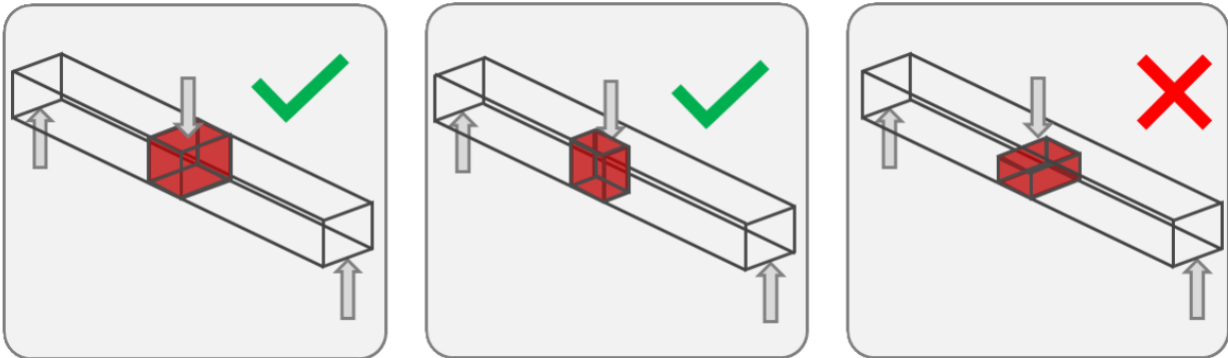
The crack in the material can be best visualized with the **Transparency** option for the volume field. For this, set the transparency as shown in the screenshot below: Choose low values (e.g., 0.2) for low damage values (in this example, for damage values lower than 0.75), and high values (1, which means no transparency) for high damage values.



1.4 Flexural Test

With the **Flexural Test** option in ElastoDict, the reaction of a material to bending loads can be analyzed. The simulation corresponds to a three-point bending flexural test according to ISO 14125 [12] provides the flexural modulus as well as flexural stress and strain as results.

As in most applications in GeoDict, only a small domain of the structure is needed for the simulation. However, it is necessary that the structure in GeoDict contains the complete height of the real structure.



Topics of this section

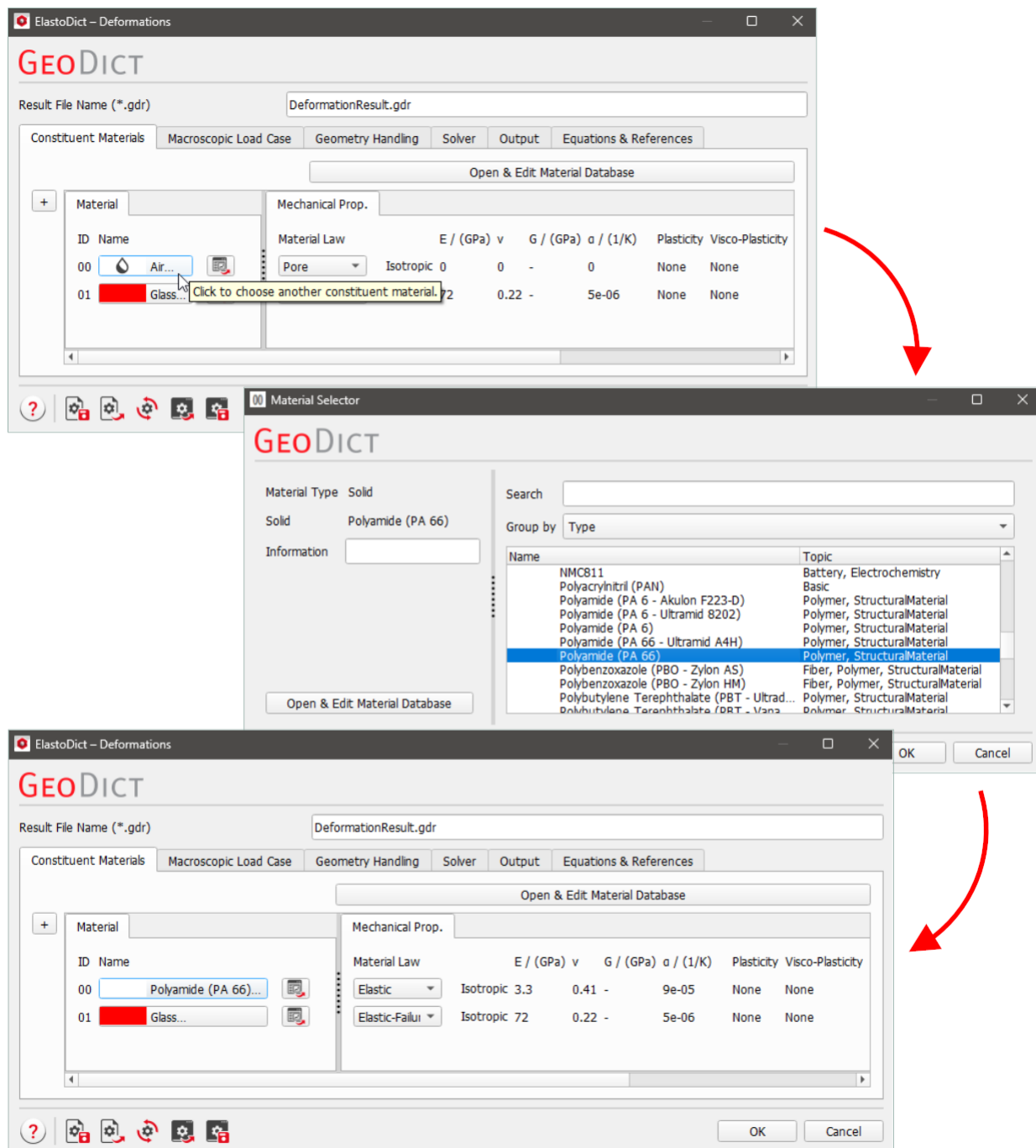
- [Constituent Materials](#)^[99]
Description of how to use and setup material laws for ElastoDict - Flexural Test.
- [Macroscopic Load Case](#)^[102]
Description of different load case options.
- [Solver](#)^[105]
Description of the solver options.
- [Output](#)^[105]
Description of the output options.
- [Result File](#)^[108]
Description of the Result File generated by an ElastoDict - Flexural Test simulation step.

1.4.1 Constituent Materials

The materials in the structure model need to be defined under the **Constituent Materials** tab, so that their mechanical parameters, such as the Young's modulus (E) and the Poisson's ratio (ν), are taken into account.

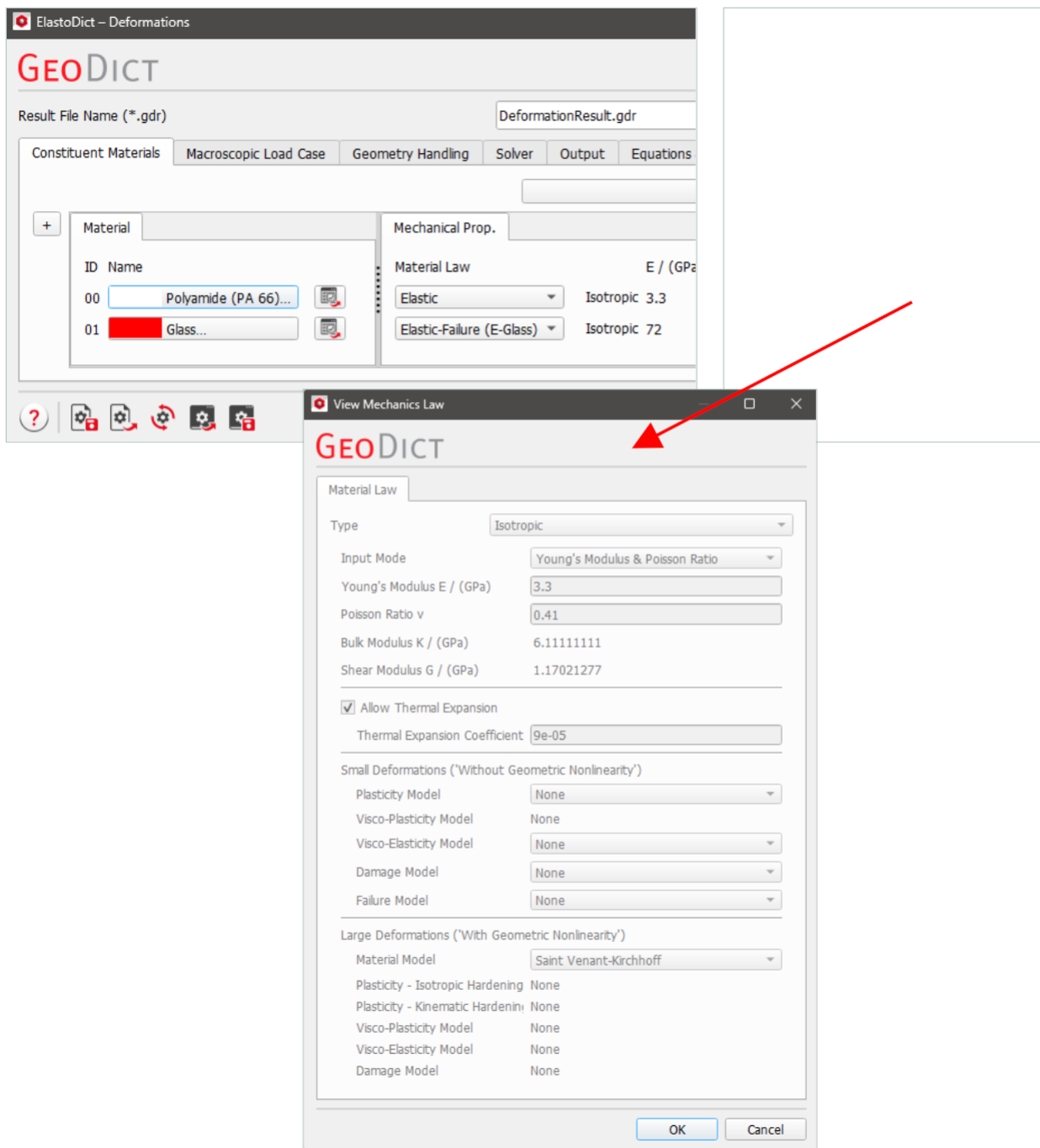
The material for a material ID can be selected by clicking on the corresponding material name. Then, the **Material Selector** dialog is opened.

The mechanical properties of the structure's constituent materials are taken directly from the GeoDict Material Database. To enter the material parameters for a linear elastic material manually, choose **Manual (Solid)** as material.

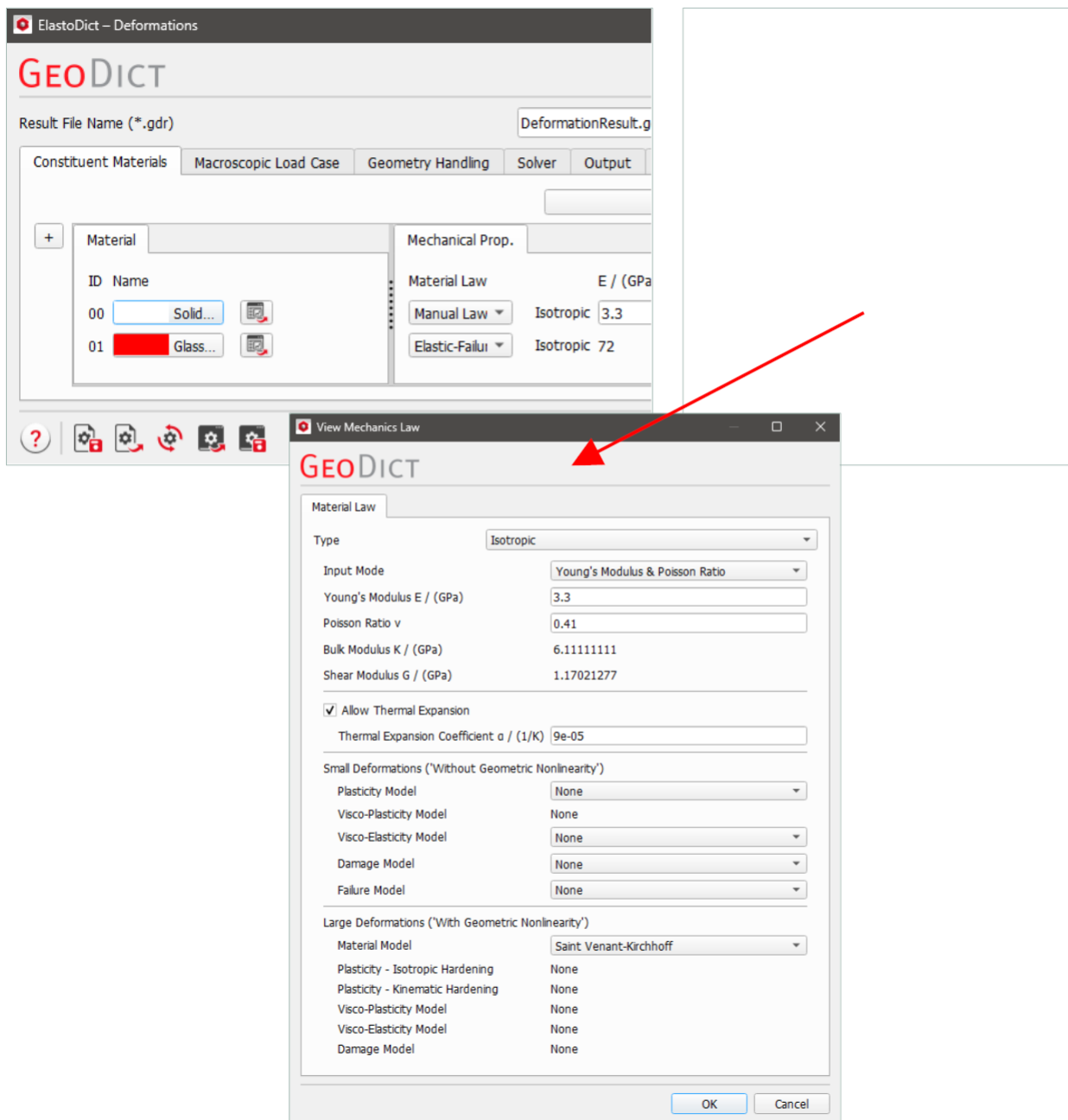


Any values entered manually, e.g., those for a user-defined constituent material, can also be added to the material database through the **Edit Material Database...** button. Nonlinear materials can also be created in the GeoDict **Material Database**.

For materials from the material database, the **View...** button on the right in the **Constituent Materials** tab shows all parameters of the material.



For **Manual** materials, the **View...** button on the right in the **Constituent Materials** tab changes to **Edit...** and allows to edit all parameters of the material.



More information on editing, expanding, and using the GeoDict Material Database is available in the Material Database chapter.

1.4.2 Macroscopic Load Case

Under the **Macroscopic Load Case** tab, the experiment settings for the flexural test are defined.

ElastoDict – Flexural Test

GEODICT

Result File Name (*.gdr) mm ▾

Constituent Materials Macroscopic Load Case Solver Output Equations & References

Experiment and Sample Size

Load Type Flexural Deflection ▾

Beam Direction X ▾

Support Span / (mm)

Beam Width / (mm)

Beam Height / (mm)

Time / (s)	Flexural Deflection / (mm)
1 1	1
2 2	2
3 3	3

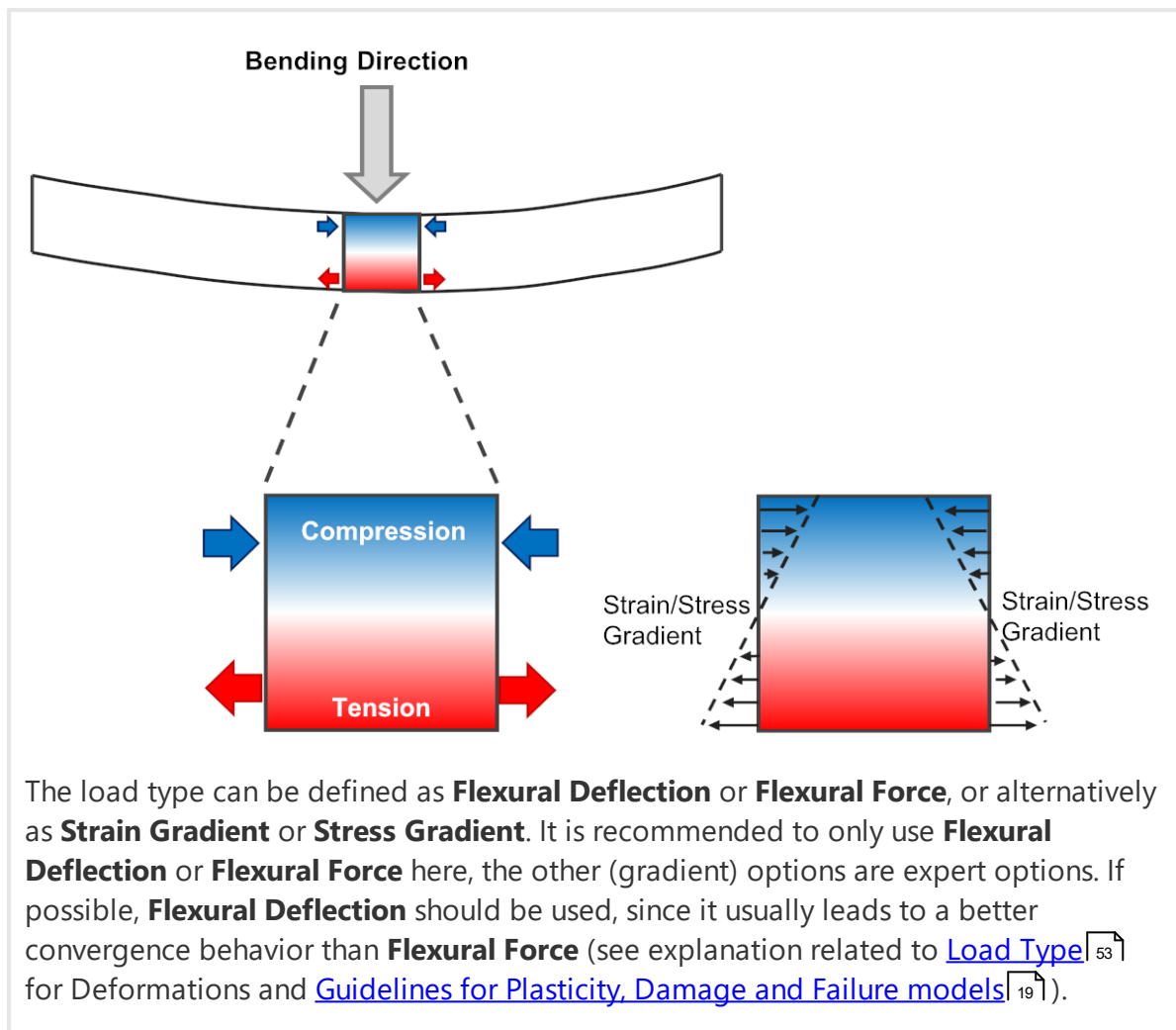
Number of Rows

Boundary Conditions

☒ Periodic
 ☐ Symmetric
 ☐ Mixed

✓ Load Type

The bending of the beam leads to compression in the upper part of the beam, and to tension in the lower part. This can also be understood as a **Strain Gradient** and **Stress Gradient** (see also the figure below). In GeoDict, the flexural test is computed based on these gradients. If **Flexural Deflection** or **Flexural Force** are defined, these gradients are automatically computed by GeoDict.

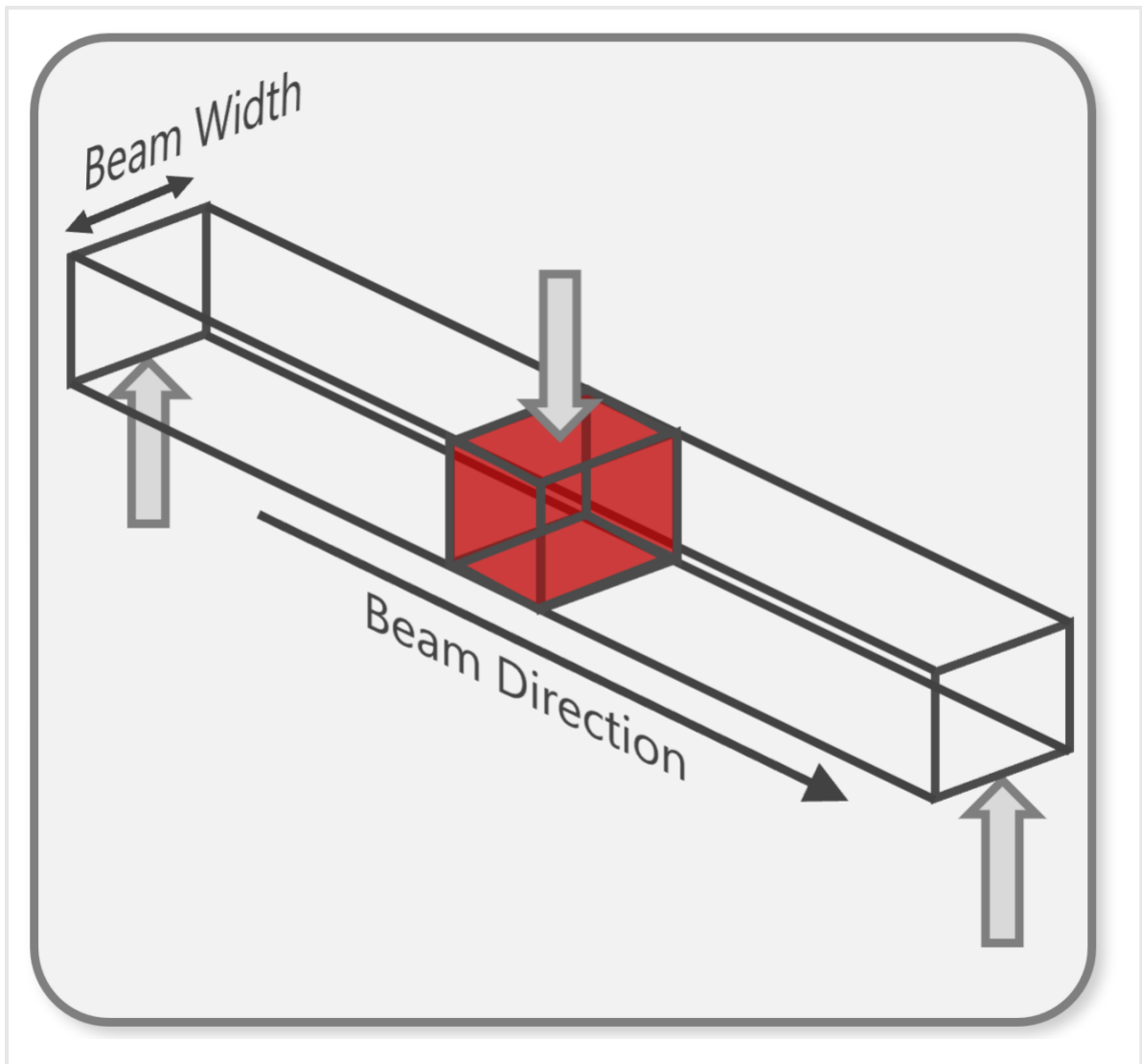


✓ Support Span

The support span is the distance between the two supporting pins, as indicated in the picture under **Experiment and Sample Size**. It must be entered in the user interface, since the structure in GeoDict is usually a small cutout of the full sample.

✓ Beam Width and Beam Height

The **Beam Width** is perpendicular to the beam and bending directions. It influences the ratio between load and deflection (since the beam can carry higher loads if it is wider), but it has no influence on the effective material parameters like the flexural modulus. The value of the **Beam Height** cannot be changed, since height of the beam is the height of the structure in GeoDict.



✓ Load Table

In the **Load Table**, the load steps and the corresponding times can be defined. For computing the flexural modulus, one step is sufficient. For more complex, non-linear materials, like, e.g., plasticity and damage, more load steps are necessary to analyze their effects

1.4.3 Solver

The **Solver** tab is identical to the one for ElastoDict-[Deformations](#) ⁶³.

1.4.4 Output

The options available in the Output tab control which information is saved to the hard disk during the computation. Choosing only the necessary variables might help to save computation time.

ElastoDict – Deformations

GEODICT

Result File Name (*.gdr)

Constituent Materials Macroscopic Load Case Geometry Handling Solver **Output** Equations & References

Write Steps

☒ Write All Steps

Write Result Fields (*.das) for Each n-th Step

Deformed Geometry

☒ Write Deformed Geometry

☒ Write Volume Fields for Deformed Geometry

☒ Write Volume Fractions for Deformed Geometry

☐ Allow Restart for Deformation Simulations

Write Volume Field

Displacement ☒ X ☒ Y ☒ Z

Stress ☒ XX ☒ YY ☒ ZZ ☒ YZ ☒ XZ ☒ XY ☒ Von Mises

Strain ☒ XX ☒ YY ☒ ZZ ☒ YZ ☒ XZ ☒ XY ☒ Von Mises

☒ Material State Variables

☐ Export VTK File (*.vti)

✓ Write All Steps

When **Write all Steps** is chosen, the results fields selected under **Write Volume Field** are written after each computation step. Otherwise, the number of steps after which the results fields are saved can be selected. When computing with many steps, disabling **Write all Steps** saves considerable computation time.

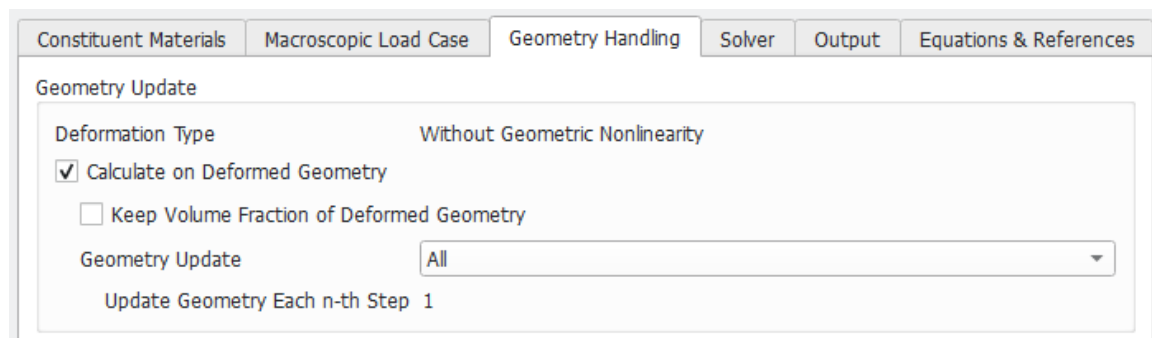
✓ Deformed Geometry

When **Write Deformed Geometry** is chosen, a geometry with the current displacements is resampled and saved after each step. Deformed geometries are written at each step, even if **Write all Steps** is disabled. Writing the deformed

geometries is mainly interesting for visualization purposes and to check possible problems in the simulation setup when [Calculate on Deformed Geometry](#)^[57] is chosen.

With **Write Volume Fields on Deformed Geometry**, also the result fields are resampled with the current displacements so that they match the deformed geometries.

With **Write Volume Fractions for Deformed Geometry**, the volume fractions for each material ID in each voxel after the deformation are written to the *.das files. This information is necessary for the computation of the deformed geometries (see [Resampling](#)^[60] for further explanations). In general, it is not necessary to select this option.



Constituent Materials Macroscopic Load Case **Geometry Handling** Solver Output Equations & References

Geometry Update

Deformation Type Without Geometric Nonlinearity

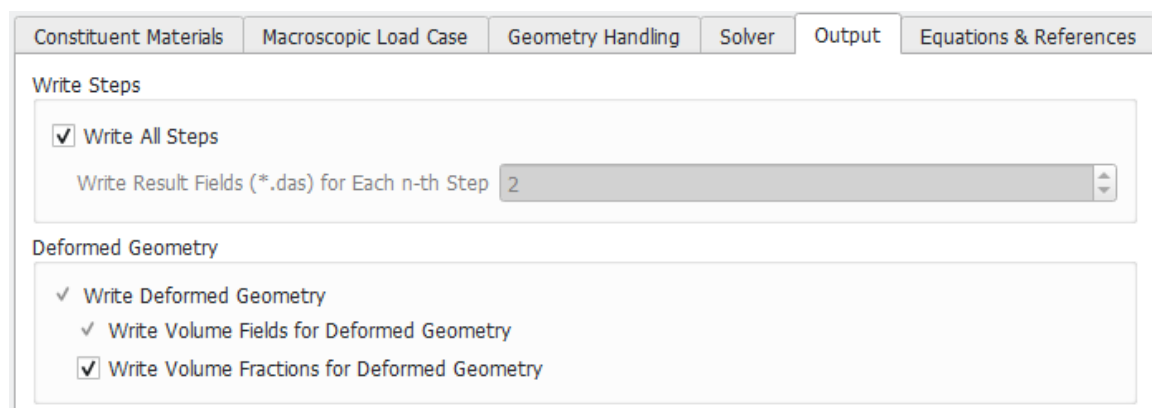
☒ Calculate on Deformed Geometry

☐ Keep Volume Fraction of Deformed Geometry

Geometry Update All

Update Geometry Each n-th Step 1

Depending on the options chosen in the **Geometry Handling** tab, some **Deformed Geometry** options might be unchangeable. For example, if **Calculate on Deformed Geometry** and **Enable Object Tracking (*.g32)** are selected, **Write Volume Fields for Deformed Geometry** is selected by default and cannot be deselected.



Constituent Materials Macroscopic Load Case **Geometry Handling** Solver Output Equations & References

Write Steps

☒ Write All Steps

Write Result Fields (*.das) for Each n-th Step 2

Deformed Geometry

☒ Write Deformed Geometry

☒ Write Volume Fields for Deformed Geometry

☒ Write Volume Fractions for Deformed Geometry

✓ Allow Restart for Deformation Simulations

If you want to be able to restart a simulation after an interruption, the complete stress and strain information must be saved for each step. With the option **Allow Restart for Deformation Simulations**, the necessary volume fields are automatically checked under **Write Volume Field** below and therefore saved.

✓ Write Volume Field

Checking or un-checking the boxes for **Displacement**, **Stress**, and **Strain** determines whether these results outputs are available for visualization and post-processing after the run of the ElastoDict-Deformations. Additionally, it is possible to save the deformed geometry and to save the solution fields (for stress and strain) for the deformed geometry. Depending on the structures size, the result files can become large (up to several GB), so it is recommended to deselect all options which are not needed for the current task.

For nonlinear materials, additional state variables might be defined. For example, for plastic deformation, these are the elastic and plastic strains. For materials with damage behavior, this is the damage variable. When **Material State Variables** is checked, these variables are saved as volume fields.

✓ Export VTK File

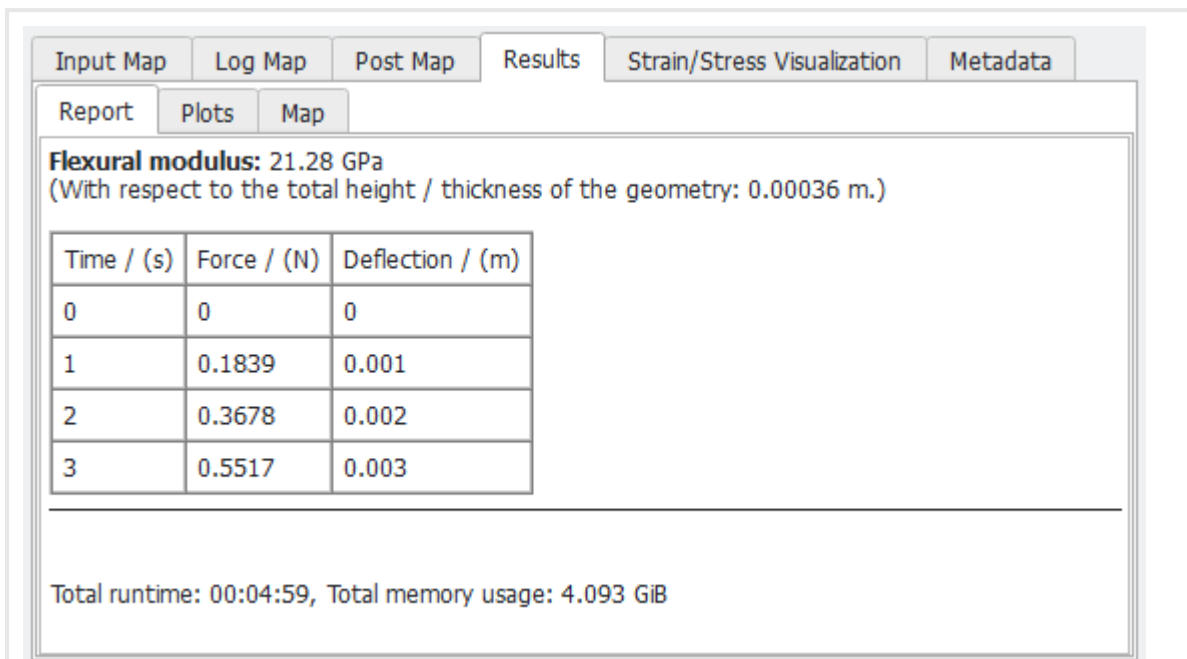
Besides using GeoDict's own visualization tools, it is possible to export the solution in the commonly used VTK (Visualization Toolkit) format (see: <https://vtk.org/> and, <https://www.paraview.org/>).

1.4.5 Result File

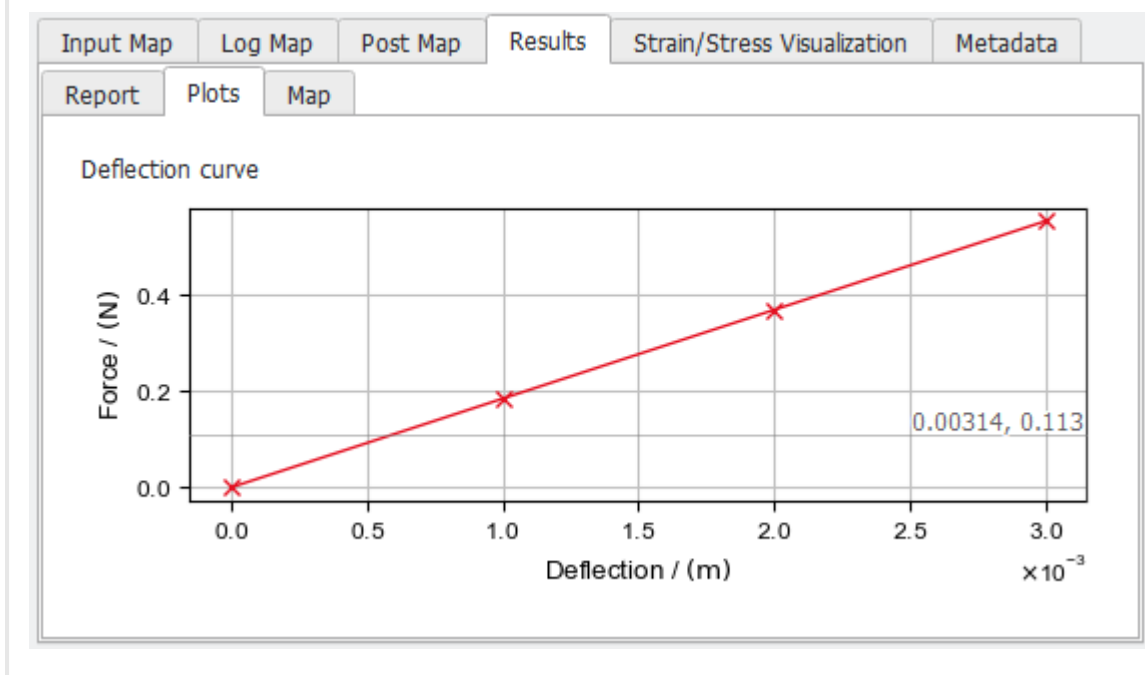
The handling of the result file works analogous as for ElastoDict-[Deformations](#)⁷⁰. Only the features specific for the flexural test are explained here in detail.

✓ Results

The **Results - Report** tab contains a table with the forces and corresponding deflections in the experiment as well as the **flexural modulus**, which is computed based on the first step of the simulation.



Under the **Plots** subtab the forces and corresponding deflections in the experiment are plotted.



✓ Strain/Stress Visualization

In the **Strain/Stress Visualization** tab, the computed result fields (*.das) and deformed geometries (*.gdt) can be loaded into GeoDict for visualization.

In the example below, a flexural test is done on a beam with a material structure as shown on the left. On the right, the distribution of the strain in X-direction is shown, and

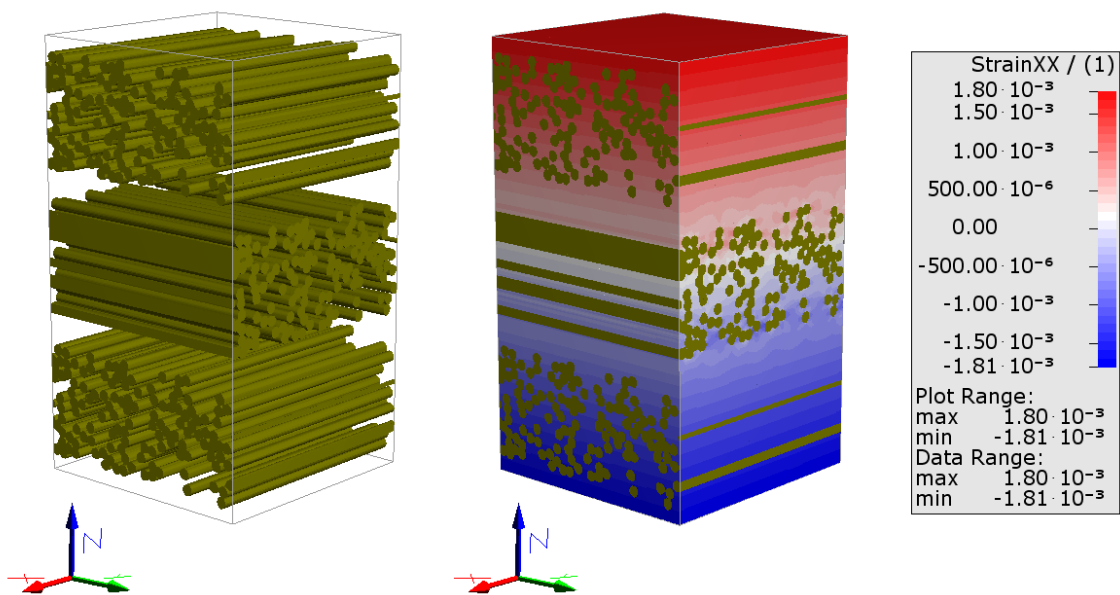
it is visible that the compressed on the top (negative strain, colored in blue) and stretched on the bottom (positive strain, colored in red).

Make sure, that all displacements, strains and stresses of interest are chosen in the [Output](#) ⁶⁸ tab to be written in the result files.

The simulation is based on the strain or stress gradients in the beam in the load direction. Therefore, it is not possible to see the bending in the results. This corresponds to the idea of analyzing a very small part of the beam.

Material Information:
ID 00: Epoxy (3501-6) [invis.]
ID 01: Glass

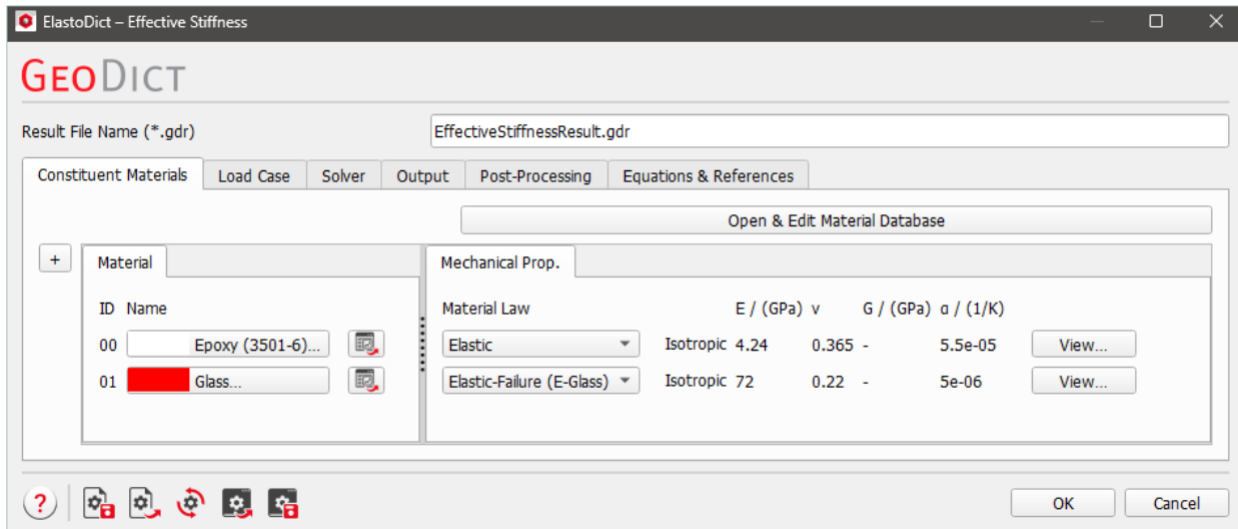
Material Information:
ID 00: Epoxy (3501-6) [invis.]
ID 01: Glass



1.5 Effective Stiffness

With ElastoDict-Effective stiffness, the effective elastic properties of a given structure can be computed. Effective elastic properties are the mechanical properties of a homogeneous material which shows the same elastic behavior as the given non-homogenous material.

In the **Effective Stiffness Options** dialog, choose the **Result File Name (*.gdr)** as you wish.



If a GeoDict results file (*.gdr) with the given name already exists in the project folder, a warning message is shown at the start of the creation process. You can either decide to back up the old file, to overwrite it or cancel and choose a new file name. If no action is taken, the default option back-up is automatically chosen after a waiting time.

Topics of this section

- [Elasticity Theory](#)^[112]
Theoretical background.
- [Constituent Materials](#)^[116]
Description of how to use and setup material laws for ElastoDict - Effective Stiffness.
- [Load Case](#)^[119]
Description of different load case options.
- [Solver](#)^[122]
Description of the solver options for the FeelMath solver as well as the LIR solver.
- [Output](#)^[130]
Description of the output options.
- [Post-processing](#)^[130]
Description of the postprocessing options.
- [Result File](#)^[133]
Description of the Result File generated by an ElastoDict - Effective Stiffness simulation step.
- [Effective Elastic Properties](#)^[136]
Description of the effective elastic properties by means of an example.

Topics of this section

- [Visualization Example](#)^[145]
Description of different Visualization settings.
- [Benchmark Results for Effective Stiffness](#)^[155]
Effective Stiffness runtime benchmark.

1.5.1 Elasticity Theory

A material is said to be elastic if it deforms under external forces (stress), but returns to its original shape when the stress is removed. For small deformations, the stress is roughly proportional to the strain in many solids. The constant of proportionality between stress and strain is given by the **Young's modulus (E)** which is a measure of stiffness. This linear relationship between stress and strain is called Hooke's law and is the basis for the theory of linear elasticity.

The general relationship (generalized Hooke's law) between multi-axial stress and strain is described by using 2nd order tensors for stress and strain and the 4th order elasticity tensor for their relation.

In solid mechanics, the **Young's modulus E** can be experimentally determined from the slope of a stress-strain curve created during tensile tests conducted on a test specimen of the material. Most metals and ceramics are isotropic. i.e., their mechanical properties are the same in all directions.

If, e.g., metals are treated in a special way for example by deep drawing, they become anisotropic, so that the **Young's modulus** depends on the direction from which the force is applied. Some materials, which are composites of two or more constituents, such as wood or reinforced concrete, are strongly anisotropic materials and display widely different mechanical properties when load is applied in different directions. For example, they exhibit a higher Young's modulus (stiffness) when loaded parallel to the fibers. This is true under the assumption that the fibers have a higher **Young's modulus** than the surrounding matrix material.

Based on the input parameters, GeoDict solves six load cases or experiments (three compressions in x, y and z, and three shearing experiments) which are sequentially computed by the ElastoDict solver to predict the entries of the 6x6 effective elasticity tensor. Each of these six simulations is done by assigning load case-specific displacements on the boundaries of the structure and calculating the corresponding stresses. By averaging the stresses over the structure, the **Hooke's Law** in the general anisotropic case is obtained,

$$\sigma_{ij} = \sum_{r,s=1}^3 c_{ijrs} \epsilon_{rs}, i, j \in \{1,2,3\} \quad (13)$$

where c is a symmetric fourth order tensor, called also elasticity tensor. Usually, the elasticity tensor and the stresses and strains are written in Voigt notation (see Wikipedia). This way, the

elasticity tensor is reduced to a 6x6 matrix, and the stresses and strains are written as 6x1 vectors. This leads to a more compact and readable notation.

Because of these symmetry properties, it is convenient to use the so-called “reduced suffix notation” (e.g., Chadwick et. al.) with the following index assignment:

Tensor notation index ij	11	22	33	23, 32	31, 13	12, 21
denoted also by	xx	yy	zz	yz	zx	xy
Voigt notation index I	1	2	3	4	5	6

Then, taking into account also the symmetry of the elasticity tensor,

$$c_{ijkl} = c_{jikl} = c_{ijlk} = c_{klij}, \quad i, j, k, l, \in \{1, 2, 3\} \quad (14)$$

The stress tensor σ_{ij} can be written as σ_I in Voigt notation:

$$\sigma = \begin{pmatrix} \sigma_{11} \\ \sigma_{22} \\ \sigma_{33} \\ \sigma_{23} \\ \sigma_{31} \\ \sigma_{12} \end{pmatrix} \quad (15)$$

The strain tensor ε_{ij} can be written as ε_I in Voigt notation:

$$\varepsilon = \begin{pmatrix} \varepsilon_{11} \\ \varepsilon_{22} \\ \varepsilon_{33} \\ \varepsilon_{23} \\ \varepsilon_{31} \\ \varepsilon_{12} \end{pmatrix} \quad (16)$$

With the Voigt notation, the symmetric fourth order elasticity tensor c_{ijkl} can be reduced to a symmetric matrix C_{IJ} with the following entries (coefficients):

$$\begin{pmatrix} C_{11} = c_{1111} = c_{xxxx} & C_{12} = c_{1122} = c_{xxyy} & C_{13} = c_{1133} = c_{xxzz} & C_{14} = c_{1123} = c_{xxyz} & C_{15} = c_{1131} = c_{xxzx} & C_{16} = c_{1112} = c_{xxxy} \\ & C_{22} = c_{2222} = c_{yyyy} & C_{23} = c_{2233} = c_{yyzz} & C_{24} = c_{2223} = c_{yyyz} & C_{25} = c_{2231} = c_{yyzx} & C_{26} = c_{2212} = c_{yyxy} \\ & & C_{33} = c_{3333} = c_{zzzz} & C_{34} = c_{3323} = c_{zzyz} & C_{35} = c_{3331} = c_{zzzx} & C_{36} = c_{3312} = c_{zzxy} \\ & & & C_{44} = c_{2323} = c_{yzyz} & C_{45} = c_{2331} = c_{yzzx} & C_{46} = c_{2312} = c_{yzxy} \\ & & & & C_{55} = c_{3131} = c_{zxzx} & C_{56} = c_{3112} = c_{zxxy} \\ & & & & & C_{66} = c_{1212} = c_{xyxy} \end{pmatrix} \quad (17)$$

Therefore, the generalized Hooke's law can be written as a matrix-vector product:

$$\sigma_I = \sum_{J=1}^6 C_{IJ} \varepsilon_J, \quad I = 1, \dots, 6 \quad (18)$$

$$\begin{pmatrix} \sigma_{xx} \\ \sigma_{yy} \\ \sigma_{zz} \\ \sigma_{yz} \\ \sigma_{xz} \\ \sigma_{xy} \end{pmatrix} = \begin{pmatrix} C_{11} & C_{12} & C_{13} & C_{14} & C_{15} & C_{16} \\ C_{12} & C_{22} & C_{23} & C_{24} & C_{25} & C_{26} \\ C_{13} & C_{23} & C_{33} & C_{34} & C_{35} & C_{36} \\ C_{14} & C_{24} & C_{34} & C_{44} & C_{45} & C_{46} \\ C_{15} & C_{25} & C_{35} & C_{45} & C_{55} & C_{56} \\ C_{16} & C_{26} & C_{36} & C_{46} & C_{56} & C_{66} \end{pmatrix} \begin{pmatrix} \varepsilon_{xx} \\ \varepsilon_{yy} \\ \varepsilon_{zz} \\ 2\varepsilon_{yz} \\ 2\varepsilon_{xz} \\ 2\varepsilon_{xy} \end{pmatrix}$$

with σ : stress tensor, C : elasticity or stiffness tensor, and ε : deformation tensor.

This elasticity or stiffness tensor describes the most general stress-strain relations for a linear elastic anisotropic solid (e.g., triclinic solid which has no material symmetry, [Nayfeh, A.H., 1995^{\[178\]}](#)). According to Nayfeh, the constitutive relations for the different symmetry classes can be listed as follows:

Symmetry class	Number of parameters to define parameters	Elasticity tensor
Anisotropic (triclinic or general)	21	$C_{anisotropic} = \begin{pmatrix} C_{11} & C_{12} & C_{13} & C_{14} & C_{15} & C_{16} \\ C_{12} & C_{22} & C_{23} & C_{24} & C_{25} & C_{26} \\ C_{13} & C_{23} & C_{33} & C_{34} & C_{35} & C_{36} \\ C_{14} & C_{24} & C_{34} & C_{44} & C_{45} & C_{46} \\ C_{15} & C_{25} & C_{35} & C_{45} & C_{55} & C_{56} \\ C_{16} & C_{26} & C_{36} & C_{46} & C_{56} & C_{66} \end{pmatrix}$
Monoclinic	13	$C_{monoclinic} = \begin{pmatrix} C_{11} & C_{12} & C_{13} & 0 & 0 & C_{16} \\ C_{12} & C_{22} & C_{23} & 0 & 0 & C_{26} \\ C_{13} & C_{23} & C_{33} & 0 & 0 & C_{36} \\ 0 & 0 & 0 & C_{44} & C_{45} & 0 \\ 0 & 0 & 0 & C_{45} & C_{55} & 0 \\ C_{16} & C_{26} & C_{36} & 0 & 0 & C_{66} \end{pmatrix}$

Symmetry class	Number of parameters to define parameters	Elasticity tensor
Orthotropic	9	$C_{orthotropic} = \begin{pmatrix} C_{11} & C_{12} & C_{13} & 0 & 0 & 0 \\ C_{12} & C_{22} & C_{23} & 0 & 0 & 0 \\ C_{13} & C_{23} & C_{33} & 0 & 0 & 0 \\ 0 & 0 & 0 & C_{44} & 0 & 0 \\ 0 & 0 & 0 & 0 & C_{55} & 0 \\ 0 & 0 & 0 & 0 & 0 & C_{66} \end{pmatrix}$
Transversely Isotropic	5	$C_{transv.isotropic} = \begin{pmatrix} C_{11} & C_{12} & C_{13} & 0 & 0 & 0 \\ C_{12} & C_{22} & C_{23} & 0 & 0 & 0 \\ C_{13} & C_{23} & C_{33} & 0 & 0 & 0 \\ 0 & 0 & 0 & C_{44} & 0 & 0 \\ 0 & 0 & 0 & 0 & C_{44} & 0 \\ 0 & 0 & 0 & 0 & 0 & \frac{C_{11} - C_{12}}{2} \end{pmatrix}$
Cubic	3	$C_{cubic} = \begin{pmatrix} C_{11} & C_{12} & C_{12} & 0 & 0 & 0 \\ C_{12} & C_{11} & C_{12} & 0 & 0 & 0 \\ C_{12} & C_{12} & C_{11} & 0 & 0 & 0 \\ 0 & 0 & 0 & C_{66} & 0 & 0 \\ 0 & 0 & 0 & 0 & C_{66} & 0 \\ 0 & 0 & 0 & 0 & 0 & C_{66} \end{pmatrix}$
Isotropic	2	$C_{isotropic} = \begin{pmatrix} C_{11} & C_{12} & C_{12} & 0 & 0 & 0 \\ C_{12} & C_{11} & C_{12} & 0 & 0 & 0 \\ C_{12} & C_{12} & C_{11} & 0 & 0 & 0 \\ 0 & 0 & 0 & \frac{C_{11} - C_{12}}{2} & 0 & 0 \\ 0 & 0 & 0 & 0 & \frac{C_{11} - C_{12}}{2} & 0 \\ 0 & 0 & 0 & 0 & 0 & \frac{C_{11} - C_{12}}{2} \end{pmatrix}$

The 6x6 matrix C_{IJ} for the homogeneous material "nearest" to the inhomogeneous original material is shown in the *.gdr result file after the solver has finished the stiffness estimation computations.

When using ElastoDict with linear laws for the constituent materials, it is important to keep in mind that the elasticity solver is working in the range of linear elasticity. To compute large deformations, plastic yield, damage effects, possible crack initiation, crack growth, and other nonlinear effects one has to use nonlinear material laws, which describe the effects under consideration, for the constituent materials. For such cases, the Hooke's law does not hold.

Nonlinear materials, e.g., for plastic deformation and damage laws are described in section [Theoretical Background: Material Models in ElastoDict](#)^[9]. Additionally, nonlinear material laws can be defined by an [Abaqus UMAT](#)^[29]. With the GeoDict installation several examples for UMATs are available.

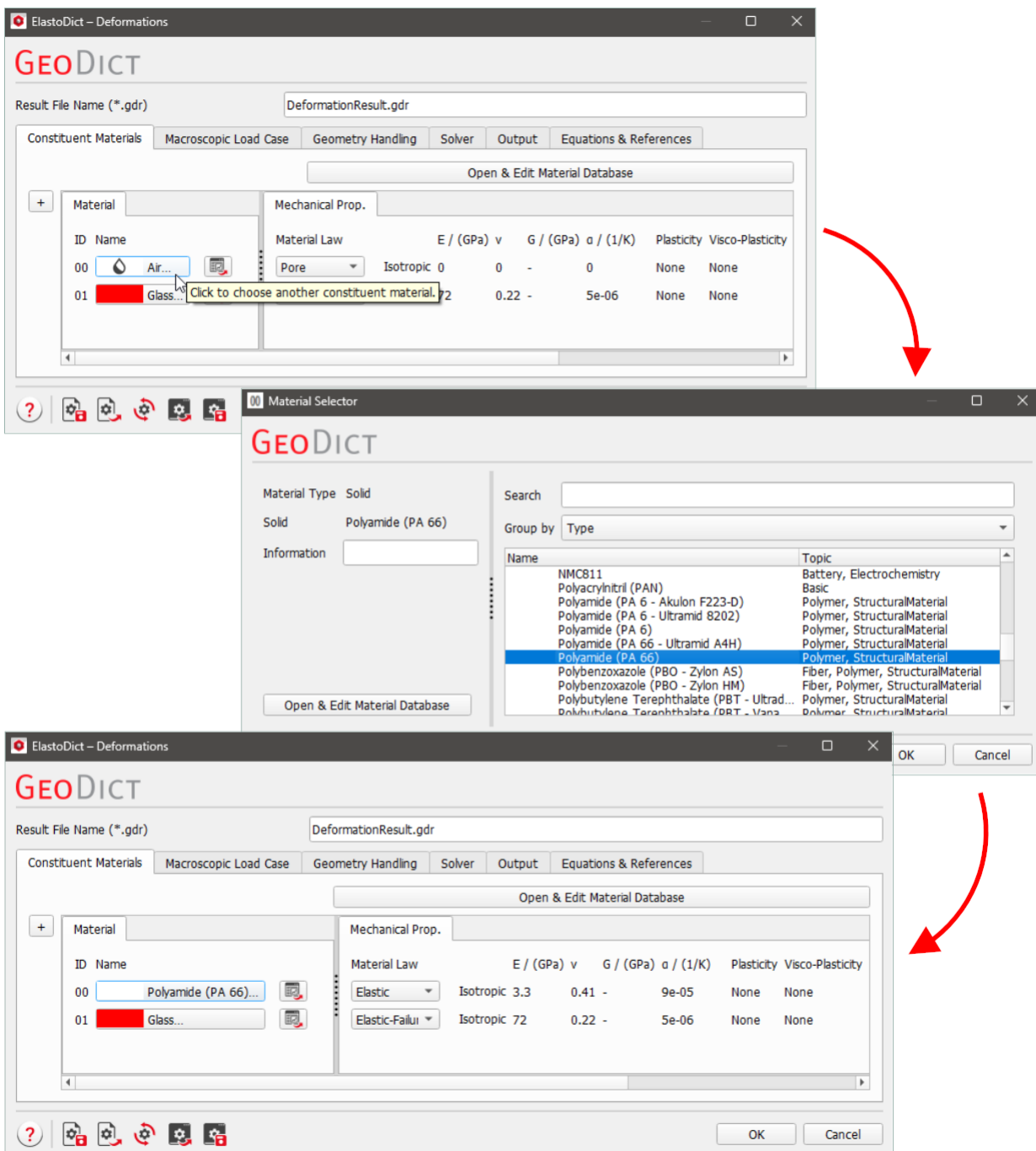
1.5.2 Constituent Materials

In ElastoDict-Effective Stiffness, only linear elastic materials can be used.

The materials in the structure model need to be defined under the **Constituent Materials** tab, so that their mechanical parameters, such as the Young's modulus (E) and the Poisson's ratio (ν), are taken into account.

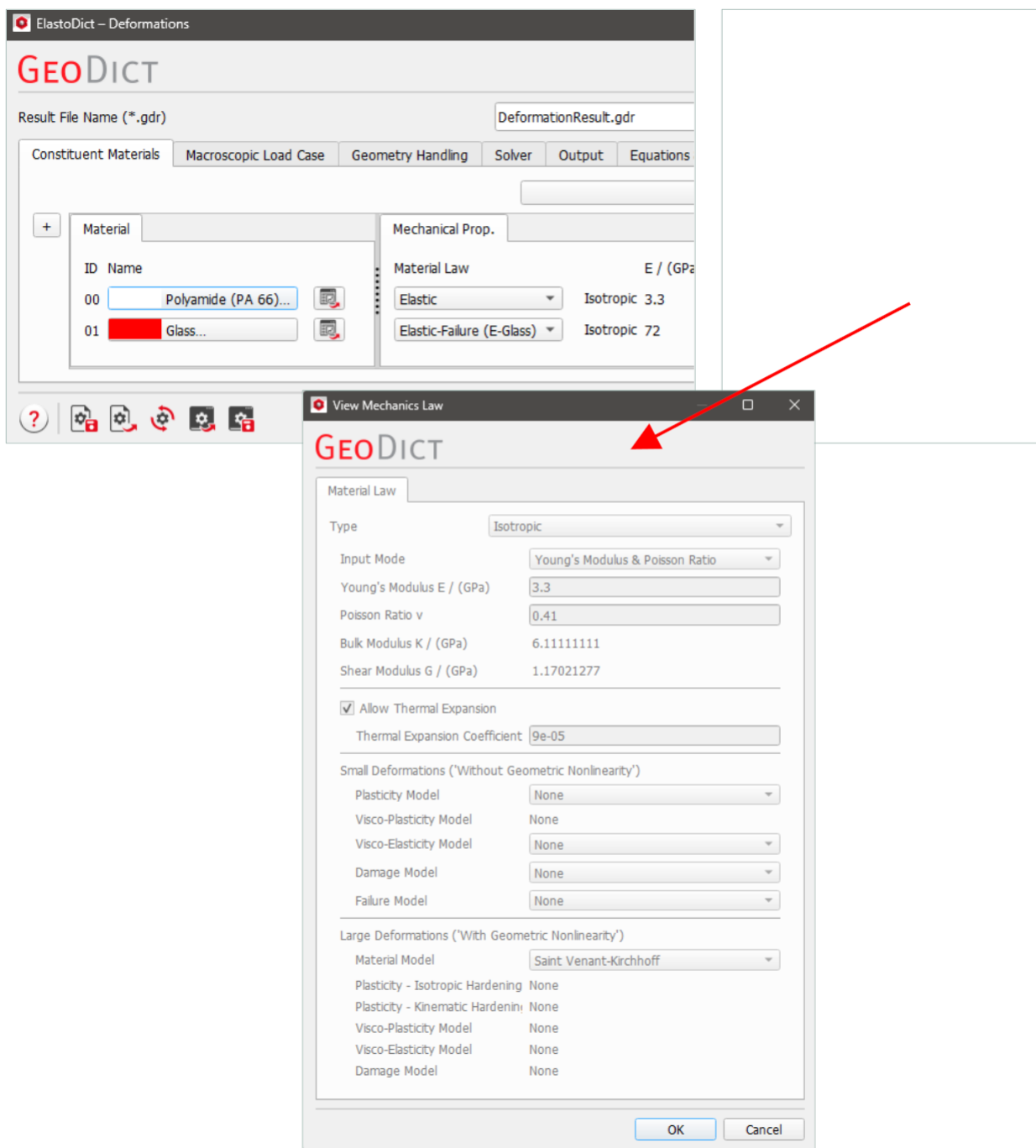
The material for a material ID can be selected by clicking on the corresponding material name. Then, the **Material Selector** dialog is opened.

The mechanical properties of the structure's constituent materials are taken directly from the GeoDict Material Database. To enter the material parameters for a linear elastic material manually, choose **Manual (Solid)** as material.

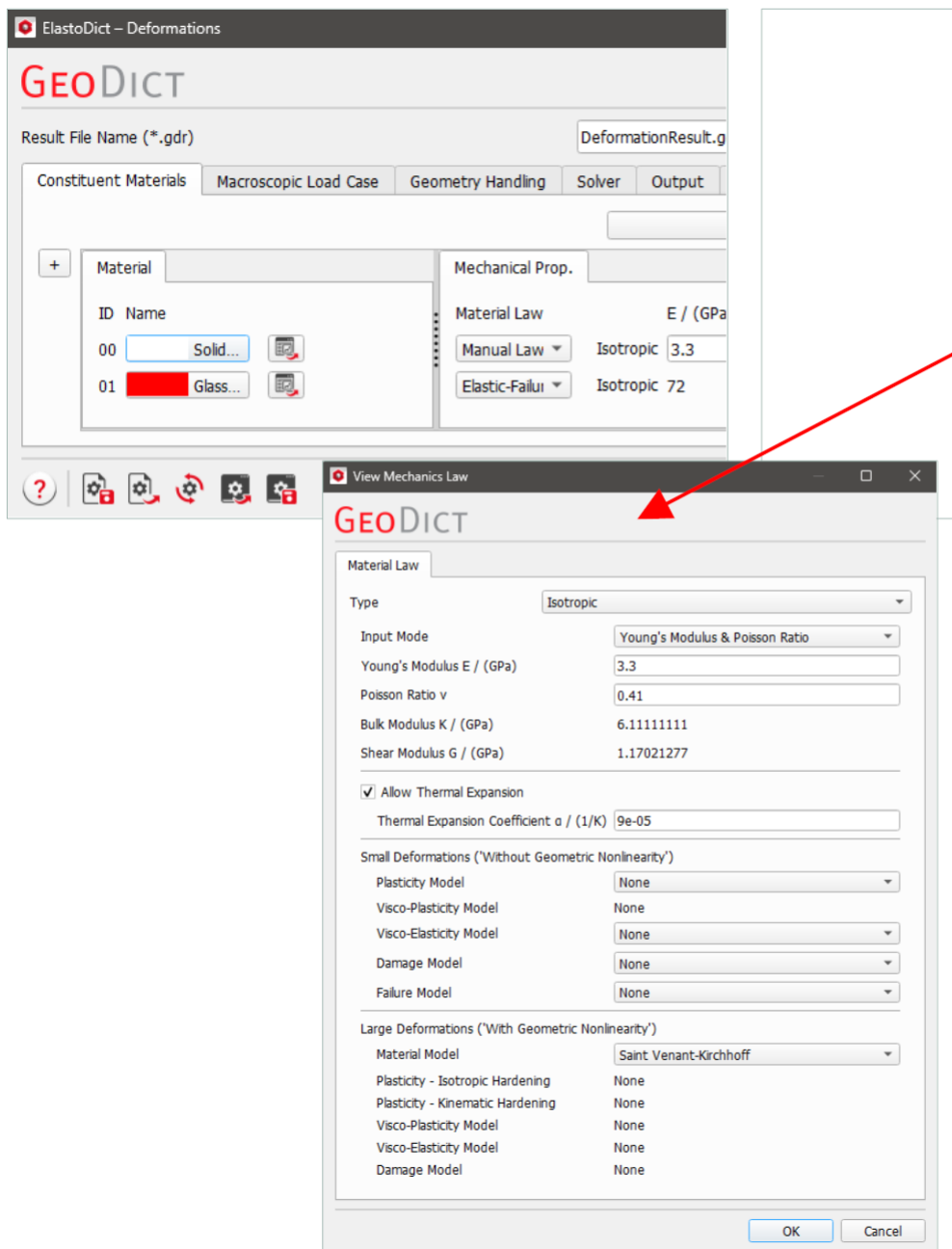


Any values entered manually, e.g., those for a user-defined constituent material, can also be added to the material database through the **Edit Material Database...** button. Nonlinear materials can also be created in the GeoDict **Material Database**.

For materials from the material database, the **View...** button on the right in the **Constituent Materials** tab shows all parameters of the material.



For **Manual** materials, the **View...** button on the right in the **Constituent Materials** tab changes to **Edit...** and allows to edit all parameters of the material.



More information on editing, expanding, and using the GeoDict Material Database is available in the Material Database chapter.

1.5.3 Load Case

In the **Load Case** tab, the loads applied to the geometry are defined.

✓ Stiffness Mode

Either the **Stiffness Tensor** for the structure, or the effective stiffness for a selected direction can be computed (**Stiffness X-Direction**, ...).

The stiffness tensor contains more information and allows to understand the properties of the structure more thoroughly. Nevertheless, if only the information for one direction is needed, the option to compute only a selected direction is much faster.

✓ Load Type

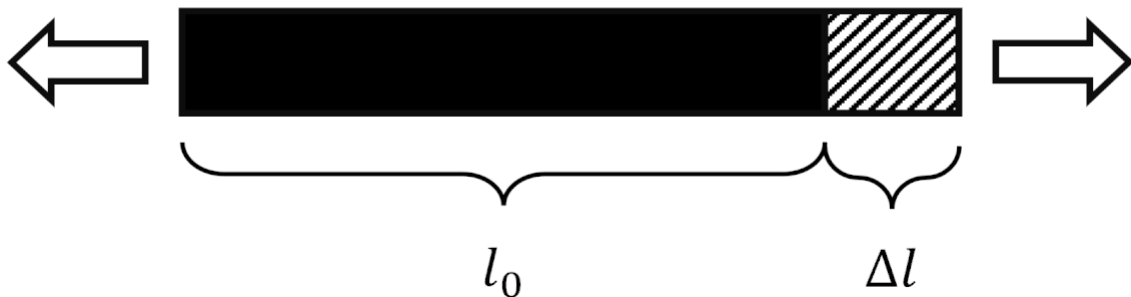
Choose whether the load applied to the material should be **Strain** or **Stress** from the pull-down menu. Choosing **Strain** is favorable for the solver and is often faster, but the choice has no effect on the quality of the solution.

Constituent Materials	Load Case	Solver	Output	Post-Processing	Equations & References
Stiffness Mode	Stiffness Tensor				
Load Type	Strain Strain Stress				
Strain Increment / (%)					

✓ Strain/Stress Increment

Depending on the selected **Load Type**, a **Strain** or **Stress Increment** must be set. Generally, it is recommended to keep the default settings. Since the equation solved in ElastoDict-Effective Stiffness is linear, scaling the input increment (**Strain** respectively **Stress**) by a factor also scales the output (**Stress** respectively **Strain**) by the same factor. In conclusion, changing the input increment does not change the computed stiffness tensors.

$$\text{Strain Increment } \varepsilon = \frac{\Delta l}{l_0} \times 100 \%$$



✓ Load Case

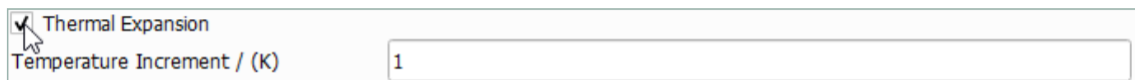
Choose the **Load Directions** in which the effective stiffness should be calculated. The complete stiffness tensor can only be computed if all six load case directions are simulated (See [Effective Elastic Properties](#)^[136] for an exemplary analysis of ElastoDict-Effective Stiffness results). Otherwise, only part of the stiffness tensor is calculated. If only parts of the stiffness tensor are computed, the results might be inaccurate, especially if the structure is highly anisotropic. Therefore, we recommend using the option to compute the stiffness in selected directions (see [Stiffness Tensor](#)^[120]) instead of computing only parts of the stiffness tensor.

For the strain equivalence principle (see the **Solver** tab), each load case adds one column to the stiffness tensor. For the energy equivalence principle, only the entries of

the stiffness tensor can be calculated, for which both the load case belonging to the column-number and the load case belonging to the row-number, have been solved.

✓ Thermal Expansion

When thermal expansion is chosen, the tensor with the thermal expansion coefficients is computed for the current structure. Thermal expansion is a separate load case independent from the strain or stress loads. The temperature change is then specified as **Temperature Increment [K]**.



Thermal Expansion
Temperature Increment / (K) 1

✓ Boundary Conditions

Here, the domain boundary conditions can be set. In general, periodic boundary conditions should be used for periodic structures, whereas otherwise symmetric boundary conditions should be applied. For further explanations, see the corresponding paragraph in the ElastoDict-[Deformations](#)^[54] section.

For the **FeelMath** solver (explained under **Solver** below), periodic boundary conditions are usually much faster and need less memory than symmetric boundary conditions. In many cases, for example for composite structures with low fiber percentage, the results for periodic boundary conditions are comparable with the results for symmetric boundary conditions even if the analyzed structure is not periodic.

The **LIR** solver is less affected by the choice of the boundary conditions. Usually, it is slightly faster with symmetric boundary conditions than with periodic boundary conditions.

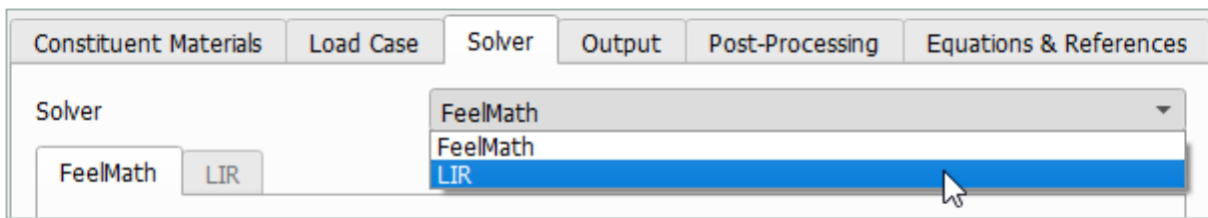
If the **Stiffness Mode** is set to **Stiffness Tensor**, only **Periodic** or **Symmetric** boundary conditions are available. For **Stiffness in X,Y or Z-Direction**, also **Mixed** boundary conditions can be set. Mixed boundary conditions are symmetric in the load direction and periodic in the tangential directions. They are usually faster to calculate than symmetric boundary conditions.

1.5.4 Solver

Two solvers are available for the **Effective Stiffness** calculation: [FeelMath](#)^[123] and [LIR](#)^[125].

The FeelMath solver is more versatile, has more features and is currently recommended for most use cases.

The LIR solver has a low memory consumption on highly porous structures, and it is fast with symmetric boundary conditions.



Internally, both solvers solve several equations for strain and stress at each voxel by using iterative solvers. The basic idea of an iterative method is to:

1. Start with some initial guess for the unknown values.
2. Improve the current values in each iterative step. The improvement can be fast or extremely slow depending on the parameters.
3. Repeat the iterative process until the stopping criteria is fulfilled.

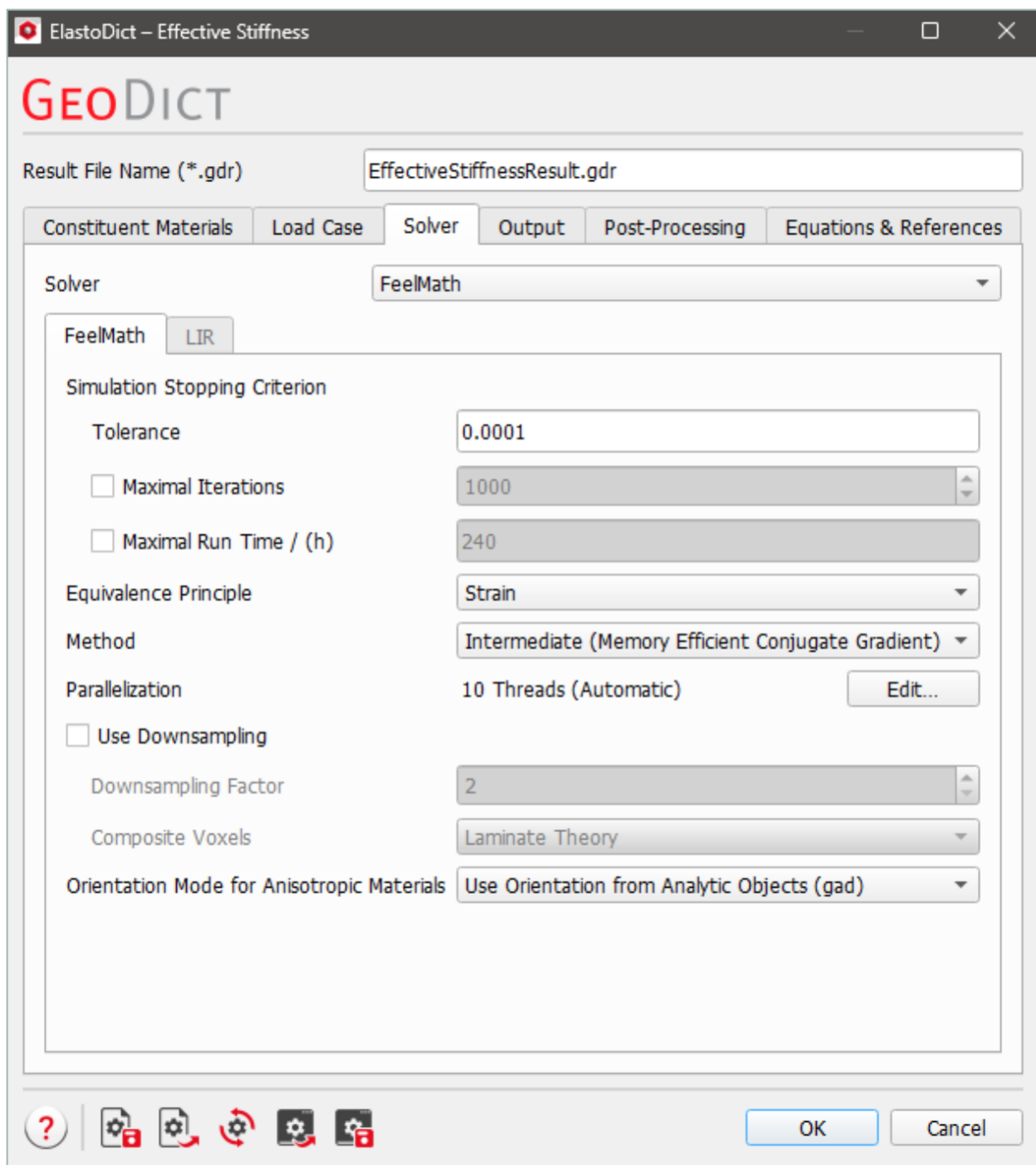
The parameters for FeelMath and LIR differ and are explained in the following two sections.

Available Solvers

- [FeelMath](#)¹²³
Description of the FeelMath solver.
- [LIR](#)¹²⁵
Description of the LIR solver.

FeelMath

The ElastoDict-Effective Stiffness FeelMath solver options are mostly the same as for the ElastoDict-[Deformations](#)⁶³. Here, the Equivalence Principle can be chosen additionally, which is not available for ElastoDict-Deformations.



Equivalence Principle

The **Equivalence principle** defines the equations which are solved for the computation of the effective properties of the structure. In principle, the effective properties are the properties of a homogeneous material with the same properties as the given structure. There are two approaches to compute these properties:

- Based on **Strain Equivalence**: the stress caused by a given strain is the same for the homogeneous and non-homogeneous material.

- Based on **Energy and Strain Equivalence**: the elastic energy stored in the material caused by a given strain is the same in the homogeneous and non-homogeneous material. The strain equivalence is calculated additionally to the energy equivalence if this option is chosen.

In ElastoDict-Effective Stiffness, either **Strain Equivalence** or **Energy and Strain Equivalence** can be chosen. By computing the latter, also the deviation in the results of both methods is evaluated in the result file and provides a measure for the quality of the solution. Depending on the structure, both methods might lead to slightly different results.

LIR

The solver parameters for the LIR solver that differ from those of the FeelMath solver are explained below.

Solver: LIR

FeelMath | LIR

☒ Error Bound Error Bound 0.01

☐ Maximal Iterations 100000

☐ Maximal Run Time / (h) 240

Parallelization 10 Threads (Automatic) Edit...

▼ Advanced Options

☒ Write Compressed Volume Fields

Optimization Options

☒ Use Multigrid Method

Use Krylov Subspace Method Enabled

Relaxation 0.5

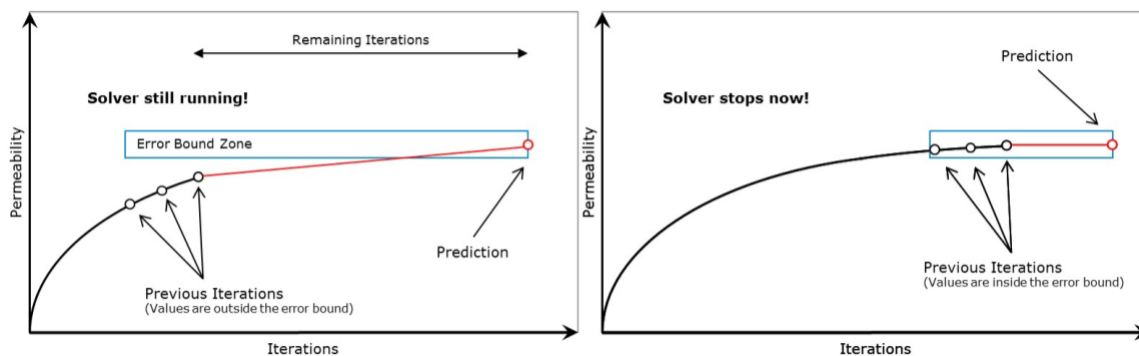
Optimize for Speed

Grid Type LIR-Tree

✓ Error Bound

The stopping criterion **Error Bound** uses the result of previous iterations, and predicts the final solution based on linear and quadratic extrapolation. The solver stops if the relative difference between computed and predicted solution is smaller than the specified error bound. The stopping criterion recognizes oscillations in the convergence behavior and prevents premature stopping at local minima or maxima. A damped convergence curve is fit through the oscillating curve and the solver stops then

regarding the damped convergence curve.

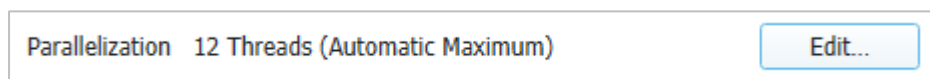


If the Krylov method (under LIR - Advanced Options) is activated, the definition of **Error Bound** is somewhat different. Here, no prediction is made, instead the continuity between neighboring cells and the conservation of mass is checked. The maximal value is normalized by the mean flow and if this value is smaller than the **Error Bound** the simulation stops.

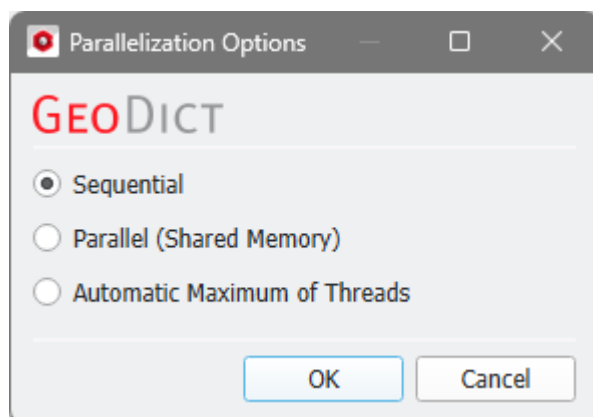
✓ Parallelization

Control how many threads are used for the computation. Parallelization is possible if your license and hardware allow it.

The **Parallelization Options** dialog opens when clicking the **Edit** button and you can choose between **Sequential**, **Parallel (Shared Memory)**, or **Automatic Maximum of Threads**.

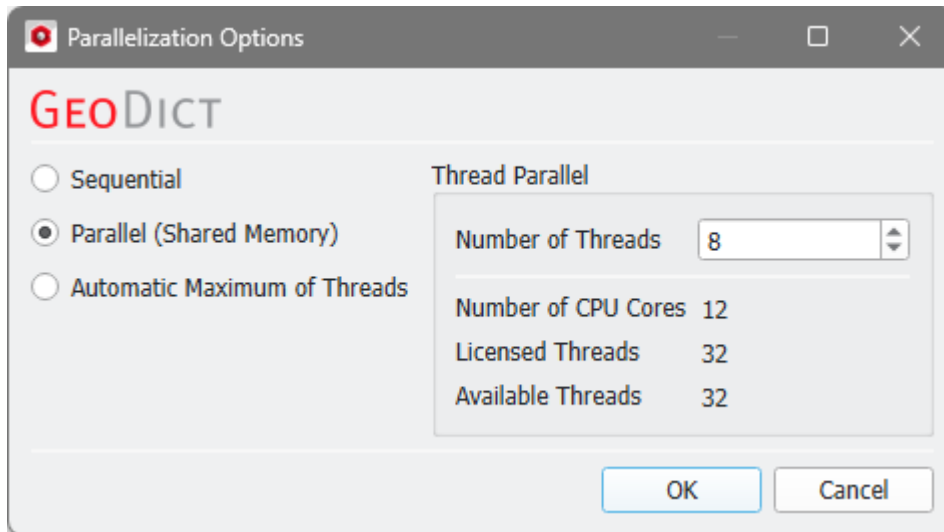


Selecting **Sequential** will not apply parallelization and only one thread is used for the computation.

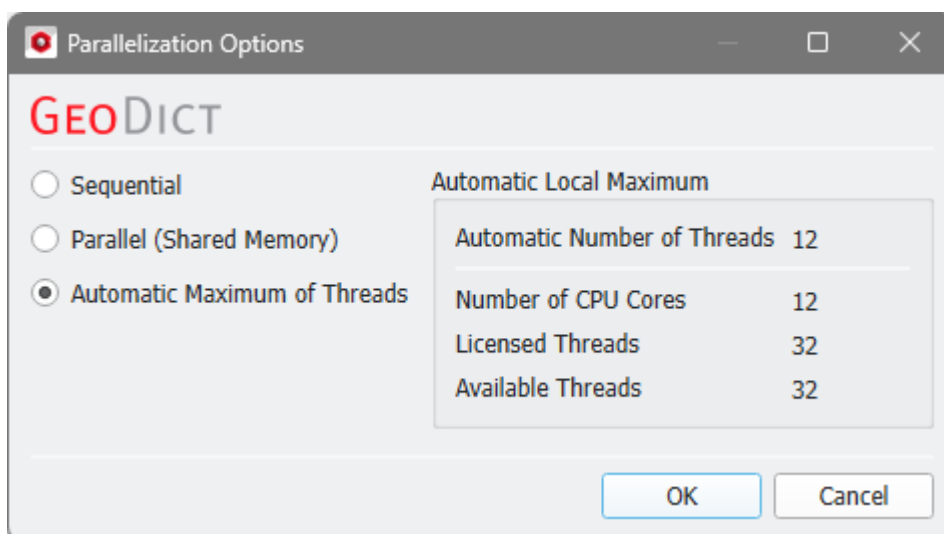


When **Parallel (Shared Memory)** is selected, the **Number of Threads** can be entered. Below, the **Number of CPU Cores** that the current machine has, the maximum number of **Licensed Threads** and the number of those licensed threads that are available (**Available Threads**) are shown in the dialog. Of course, the maximal number of

parallel processes you can use, is the smallest of those three numbers.



If **Automatic Maximum of Threads** is selected, the number of parallel processes is automatically selected for optimal speed, based on the CPU cores and licensed parallel processes.



The **Automatic Local Maximum** of processes is automatically selected, which is the minimum of **Number of CPU Cores**, **Licensed Threads**, and **Available Threads**.

✓ Write Compressed Volume Files

The LIR solver uses a very memory efficient adaptive grid structure for the simulations.

If the option **Write Compressed Volume Fields** is checked, then the adaptive grid is used as compression method for writing out *.das files. This option allows to save 80-90% space on hard drive. The runtime for writing *.das files is also reduced significantly. But the runtime for loading and uncompressing of compressed *.das is increased by the amount of runtime that was saved for writing out compressed *.das files.

☒ Write Compressed Volume Fields

If the option **Write Compressed Volume Fields** is not checked, then a usual regular grid is used for writing out *.das files.

✓ Optimization Options

Use Multigrid Method

The **Multigrid Method** (see, e.g., [Wesseling, 2004](#)) was introduced to speed-up the computation and reduce the runtime significantly. The main idea of **Multigrid** is the usage of multiple coarser adaptive grids to speed up convergence behavior but requires only little more memory.

☒ Use Multigrid Method

The method is available to solve the Stokes and Stokes–Brinkman equations as well as for solving mechanics, diffusion, thermal, and electrical conduction and is enabled by default.

Use Krylov Subspace Method

Another speed-up option to accelerate the convergence behavior of the LIR solver is called **Krylov Subspace Method**.

The runtime of the LIR solver depends on many different properties of the structure and the simulation parameters. The BiCGStab algorithm is used, which can reduce the runtime for challenging simulation very drastically.



Note! Using the BiCGStab method approximately doubles the amount of RAM needed for the computation.

Unfortunately, the Krylov method is not always faster than a simulation without the Krylov method and therefore we introduced an **Automatic** mode which uses some heuristics to choose the most efficient method based on structure, material parameters, and boundary conditions automatically. Of course, it is possible to explicitly enable (**Enabled**) or disable (**Disabled**) the method.

☐ Use Krylov Subspace Method

Automatic

Automatic

Enabled

Disabled



Note! In case that the Krylov subspace method (BiCGStab) is used, the **Relaxation** may also be adapted automatically.

Relaxation

Depending on the material parameters and geometry of the structure, the underlying mathematical problem can vary in complexity, thus influencing the behavior of the solver. The more complex the problem is, the more stable the solver settings should be.

Relaxation	1
------------	--------------------------------

With the **Relaxation** number, the solver is adjusted from **Stable** (which results in higher number of iterations, slower time stepping, and longer solver run times), to **Fast**, which makes the solver run less iterations but implies the risk that the solver does not converge. The **Relaxation** is a parameter of the [SOR method](#) and must be between 0 and 2 to ensure convergence. For relaxation values smaller than one (<1.0), the simulation is more stable. For relaxation values larger than one (>1.0), the simulation converges faster.

Optimize for

The **LIR** solver can **Optimize for Speed** or **Memory**.

Optimize for	Speed Memory Speed
--------------	---

- If **Speed** is chosen, the solver constructs additional optimization structures. The runtime is decreased by up to 30% but requires up to 50% more memory compared to the other option.
- If **Memory** is chosen, the runtime is increased by up to 40% but the solver requires up to 50% less memory.

✓ Grid Type

The **Grid Type** decides what kind of tree structure is used for the simulation.

Grid Type	LIR-Tree LIR-Tree Regular Grid
-----------	---

The default option is **LIR-Tree** and should always be used. The solver uses an adaptive grid structure called LIR-tree and needs up to 10 times less runtime and memory compared to the **Regular Grid** option.

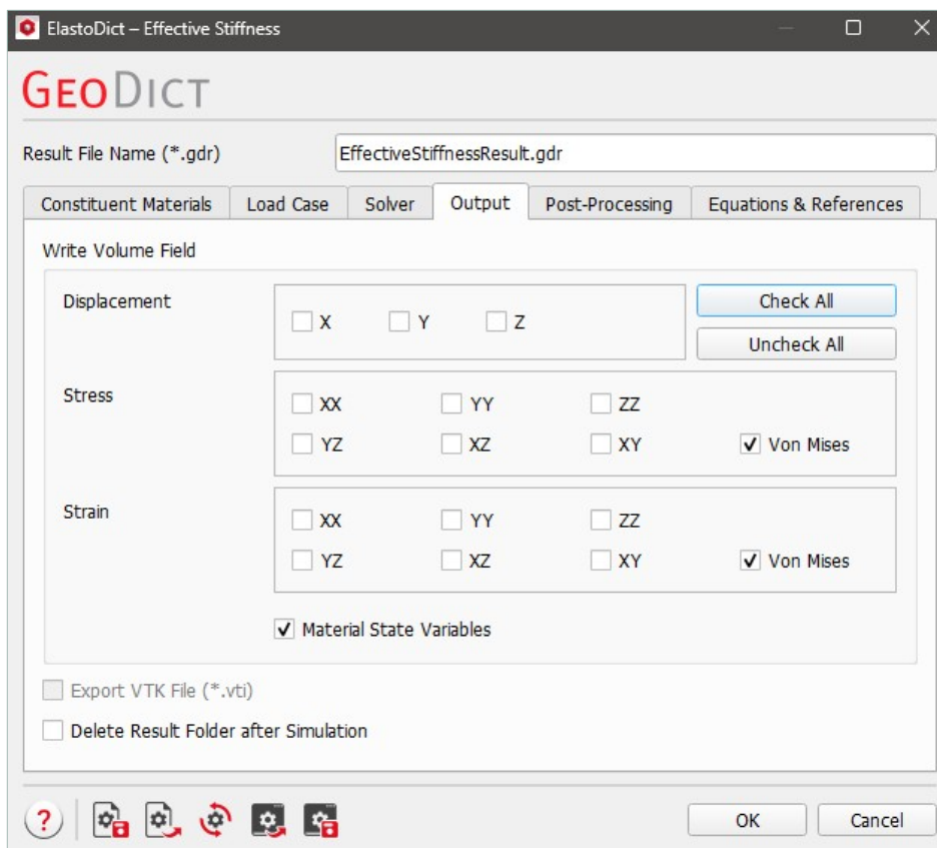
The solver can analyze the result field during the computation and improves the adaptive grid in places where more accuracy is needed. The LIR solver splits cells where a high gradient occurs.

1.5.5 Output

The ElastoDict-Effective Stiffness **Output** options are mostly the same as for ElastoDict-[Deformations](#) ⁶⁸.

The option to **Delete Result Folder after Simulation** is only available for Effective Stiffness. When this option is chosen, only the .gdr file is kept after the run and the corresponding solver files are deleted (log files, solver input files, result files, ...). This option is useful when only the effective properties are of interest and no result fields are needed.

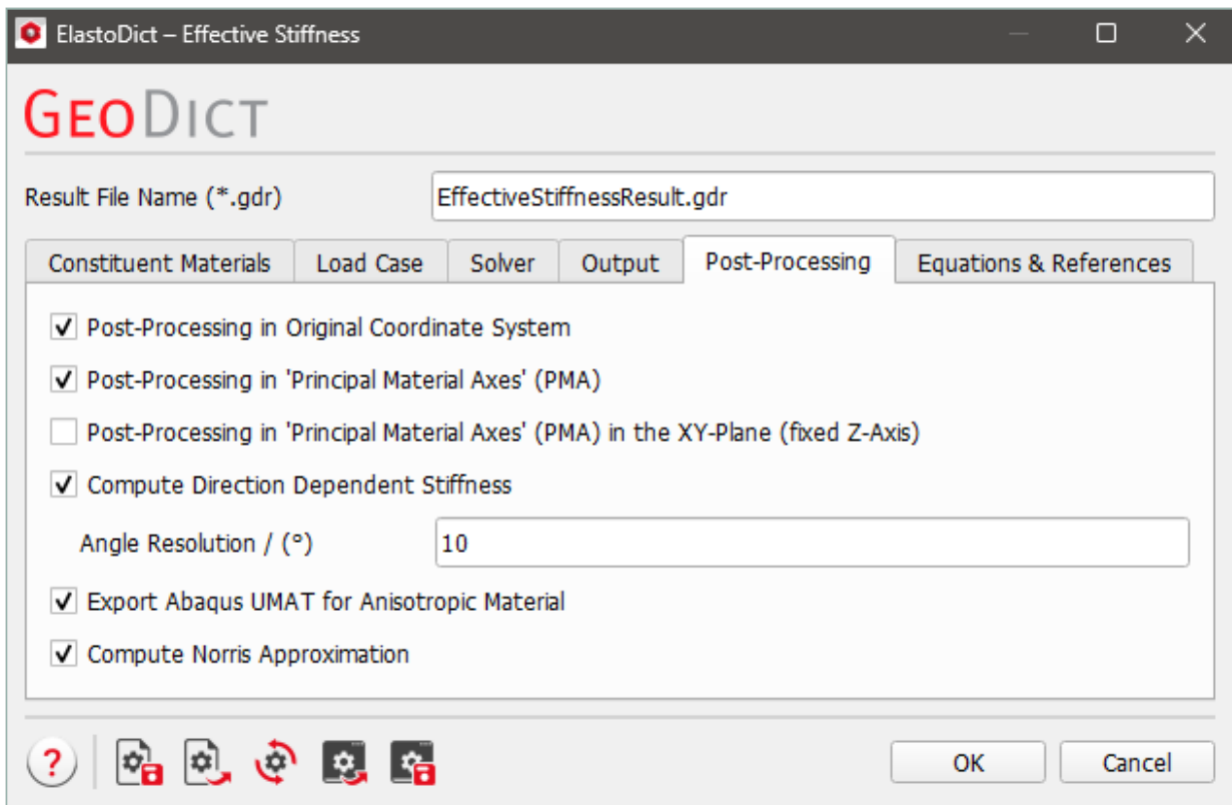
The option **Export VTK File** is not available for the LIR solver.



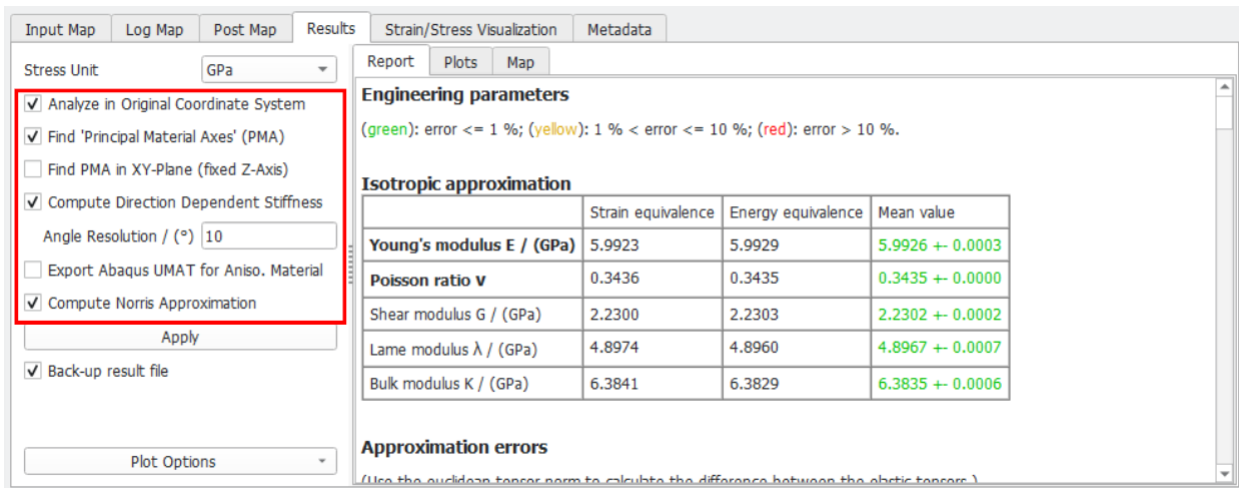
1.5.6 Post-processing

The options available under **Post-Processing** are optional and they can also be set later in the Result Viewer of the result file (*.gdr) after the computations.

The options in the ElastoDict-Effective Stiffness settings are shown here.



How they look in the Result Viewer is shown below. **Post-Processing in Original Coordinate System** is always available and selected by default. All other post-processing options are only available if all load directions (XX, YY, ZZ, YZ, XZ, and XY) have been previously checked under the **Load Case** tab.



✓ Stress Unit

When post-processing the plots in the Result Viewer, the **Stress Unit** for the results can be selected. This choice has an effect on the values under the Report subtab and on the Plots subtab. The stress unit in the Results – Map subtab is always GPa.

✓ Post-Processing in Original Coordinate System

Check **Post-Processing in Original Coordinate System** to compute the engineering parameters and elasticity-tensor approximations in the original coordinate system (X, Y, Z).

✓ Post-Processing in Principal Material Axes (PMA)

Check **Post-Processing in Principal Material Axes (PMA)** to find the PMA of the material and compute the engineering parameters and elasticity-tensor approximations in the PMA system. This is only possible if all six load cases (under **Load Case** tab) are calculated.

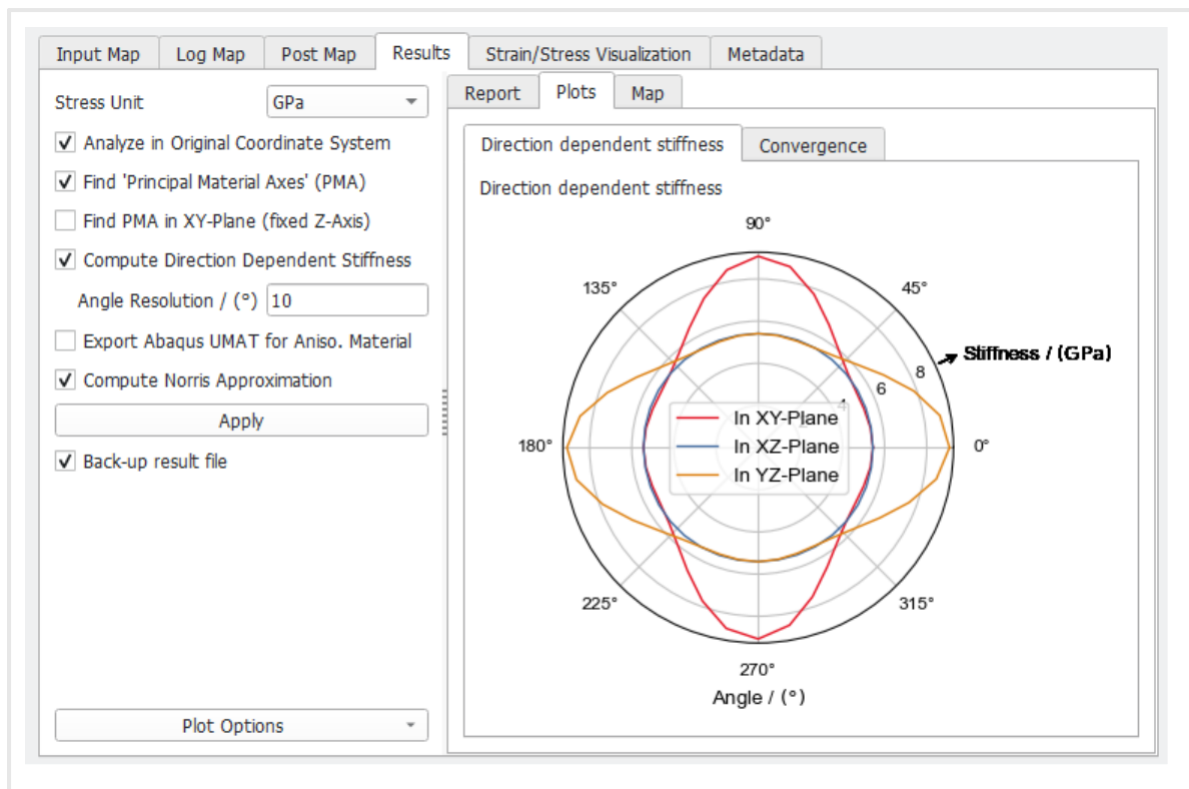
✓ Post-Processing in Principal Material Axes (PMA) in the XY-Plane (fixed Z-Axis)

Check **Post-Processing in Principal Material Axes (PMA) in the XY-plane** to find the PMA of the material in the XY-plane and to compute the engineering parameters and elasticity-tensor approximations in this PMA system. This is only possible if all six load cases (under **Load Case** tab) are calculated.

✓ Compute Direction-Dependent Stiffness

Check **Compute Direction-Dependent Stiffness** to calculate the material stiffness in the XY, XZ, and YZ plane, in the original coordinate system. Additionally, the material stiffness can be analyzed for arbitrary angles. With the **Angle Resolution (°)** setting the angle steps for this calculation is defined.

The result file obtained after computations includes a table showing the calculated Young's Modulus **E** in the three planes according to the entered angle resolution. Additionally, a plot for the direction dependent stiffness can be found in the results file under the **Plots** tab.



✓ Export Abaqus UMAT for Anisotropic Material

When Export Abaqus UMAT for Anisotropic Material is checked, an UMAT with the computed anisotropic material properties is exported and can be found in the result folder. This UMAT can directly be used in Abaqus, or it can also be used as material in GeoDict.

✓ Compute Norris Approximation

Check **Compute Norris Approximation** to compute the cubic elasticity tensor approximation following [Norris, 2006](#)^[178]. See also the section about the [Effective Elastic Properties](#)^[136] for a more details discussion on the results. This is only possible if all six load cases (under **Load Case** tab) are calculated.

1.5.7 Result File

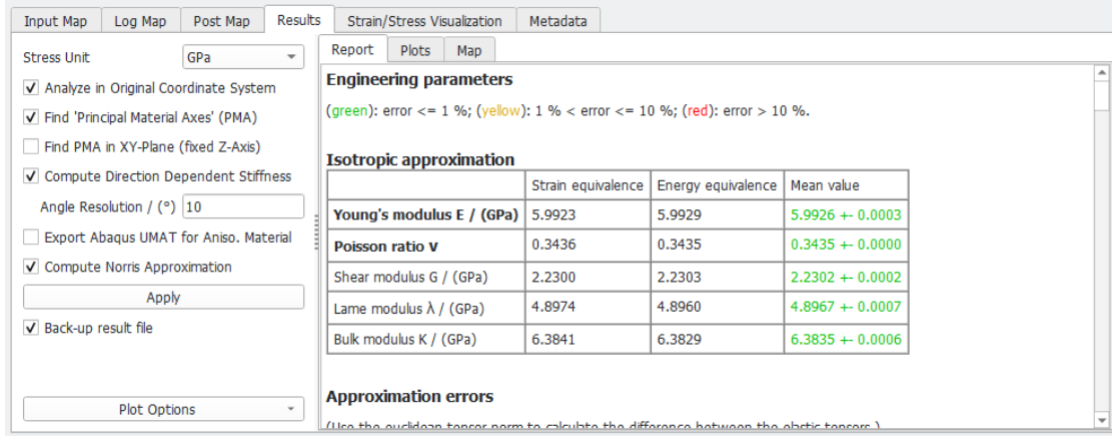
The handling of the result file works analogous as for ElastoDict-[Deformations](#)^[70], therefore only the features specific for ElastoDict-Effective Stiffness are explained here in detail.

Clicking **Load Structure** in the result file loads the original structure to the visualization area.

✓ Results

The **Results** tab gives access to the computed values for the parameters calculated by the selected ElastoDict process. The Results tab contains two or three subtabs:

- The **Report** subtab shows the values of the computed mechanical properties. These values can be exported in various ways by selecting an option from the **Export** pull-down menu at the bottom of the Result Viewer. For detailed information about all result viewer options and functionalities, please refer to the Result Viewer section.



Input Map Log Map Post Map Results Strain/Stress Visualization Metadata

Stress Unit: GPa

☒ Analyze in Original Coordinate System

☒ Find 'Principal Material Axes' (PMA)

☐ Find PMA in XY-Plane (fixed Z-Axis)

☒ Compute Direction Dependent Stiffness

Angle Resolution / (°): 10

☐ Export Abaqus UMAT for Aniso. Material

☒ Compute Norris Approximation

Apply

☒ Back-up result file

Plot Options

Report Plots Map

Engineering parameters

(green): error <= 1 %; (yellow): 1 % < error <= 10 %; (red): error > 10 %.

Isotropic approximation

	Strain equivalence	Energy equivalence	Mean value
Young's modulus E / (GPa)	5.9923	5.9929	5.9926 ± 0.0003
Poisson ratio ν	0.3436	0.3435	0.3435 ± 0.0000
Shear modulus G / (GPa)	2.2300	2.2303	2.2302 ± 0.0002
Lamé modulus λ / (GPa)	4.8974	4.8960	4.8967 ± 0.0007
Bulk modulus K / (GPa)	6.3841	6.3829	6.3835 ± 0.0006

Approximation errors

(Use the euclidean tensor norm to calculate the difference between the elastic tensors.)

For ElastoDict-Effective Stiffness (AF) and ElastoDict-Effective Stiffness, if all 6 load cases have been computed, an analysis of the effective stiffness tensors can be carried out on the tables shown in this view. This analysis allows to judge, whether the material exhibits some degree of symmetry, such as transverse isotropy.

- In the ElastoDict-Effective Stiffness result file, the **Plots** subtab shows a plot of the direction dependent stiffness in the X-, Y- and Z-planes. Right-clicking on the plot view allows generating an image or exporting the values as .txt file or save the image as .png file.
- The **Map** subtab shows a structured list of all result values.

If only the stiffness in one direction is computed instead of the complete stiffness tensor (by selecting, e.g., **Stiffness in X-Direction** as **Stiffness Mode** under the **Load Case** tab), the results report is much simpler. It contains only the Young's modulus in the selected direction, and the average strain and the average stress in the structure (see example below).

Input Map

Log Map

Post Map

Results

Strain/Stress Visualization

Metadata

Stress Unit

GPa

☒ Analyze in Original Coordinate System
 ☒ Find 'Principal Material Axes' (PMA)
 ☐ Find PMA in XY-Plane (fixed Z-Axis)
 ☒ Compute Direction Dependent Stiffness

Angle Resolution / (°) 10

☒ Export Abaqus UMAT for Aniso. Material
 ☒ Compute Norris Approximation

Apply

☒ Back-up result file

Plot Options

Report

Plots

Map

Stiffness in X-Direction

Young's modulus E / (GPa)	5.9930
Average stress / (GPa)	0.0002997
Average strain / (%)	0.005

Total runtime: 00:00:07, Total memory usage: 2.690 GiB

Strain/Stress Visualization

In the **Visualization** tab, the computed solution volume fields (*.das) and visualizations of the principal material axes (PMA) can be loaded.

Input Map

Log Map

Post Map

Results

Strain/Stress Visualization

Metadata

Volume Fields (Displacement, Strain, Stress)

☐ XX Load Case: StrainStressResult_xx.das
 ☐ YY Load Case: StrainStressResult_yy.das
 ☒ ZZ Load Case: StrainStressResult_zz.das
 ☐ YZ Load Case: StrainStressResult_yz.das
 ☐ XZ Load Case: StrainStressResult_xz.das
 ☐ XY Load Case: StrainStressResult_xy.das
 ☒ TE Load Case: unavailable

Load

Principal Material Axes - Strain Equivalence

Principal Material Axes - Energy Equivalence

Principal Material Axes in XY-Plane (fixed Z-Axis) - Strain Equivalence

Principal Material Axes in XY-Plane (fixed Z-Axis) - Energy Equivalence

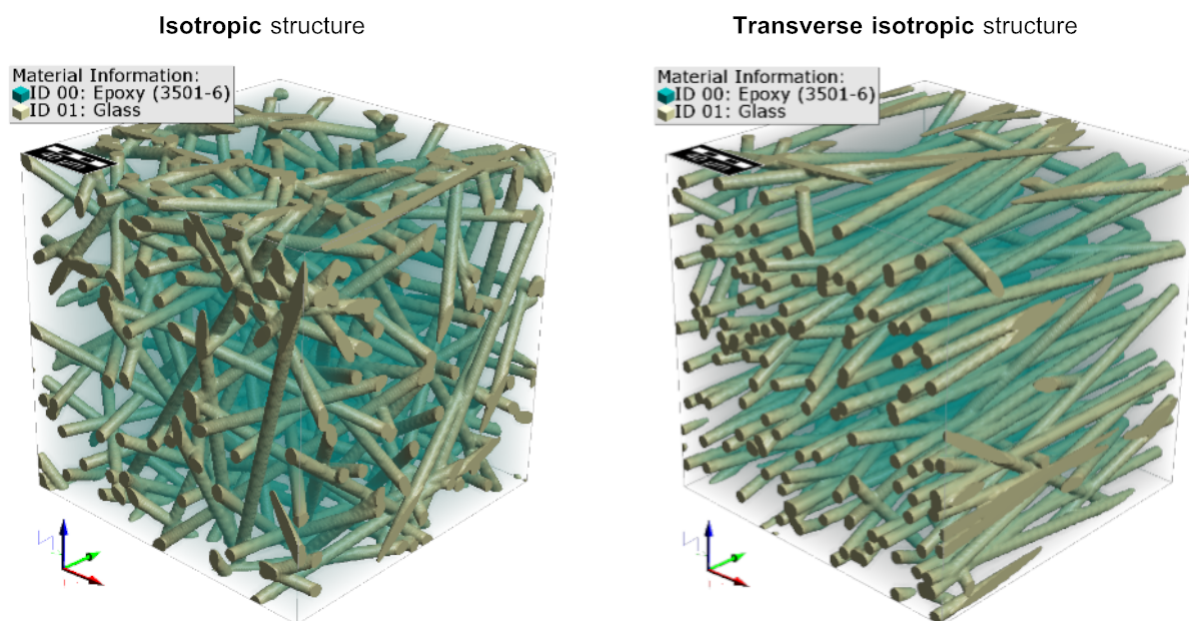
Refer also to [Examples and Visualization](#)^[145] for more possibilities of visualizing the results from ElastoDict-Effective Stiffness.

1.5.8 Effective Elastic Properties

To demonstrate and discuss different ElastoDict's effective elastic properties, ElastoDict-**Effective Stiffness** computation results for two different fiber-reinforced composite structures are presented. The structures were generated with the FiberGeo module with [isotropic](#) and [transverse isotropic](#) fiber orientations.

Both composite structures contain the same type of fiber (curved elliptical, diameter 6 μm), at the same solid volume percentage, but differ in their fiber orientation. The isotropic structure has been generated with isotropic fiber orientation, while transverse isotropic structure has been generated with *Anisotropy 1* = 1 and *Anisotropy 2* = 10.

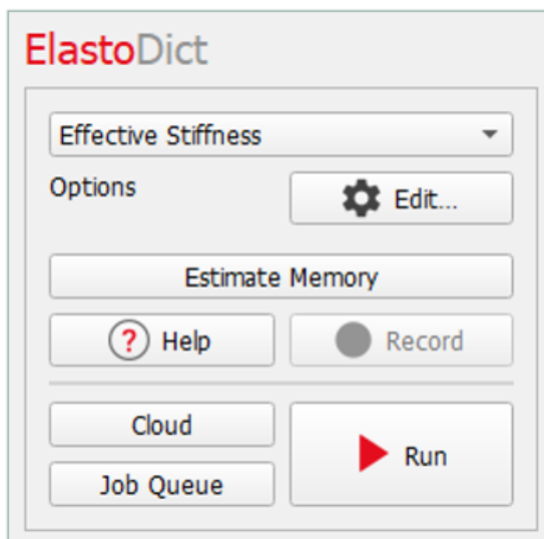
For more information on generating isotropic and anisotropic structures, see the FiberGeo User Guide.



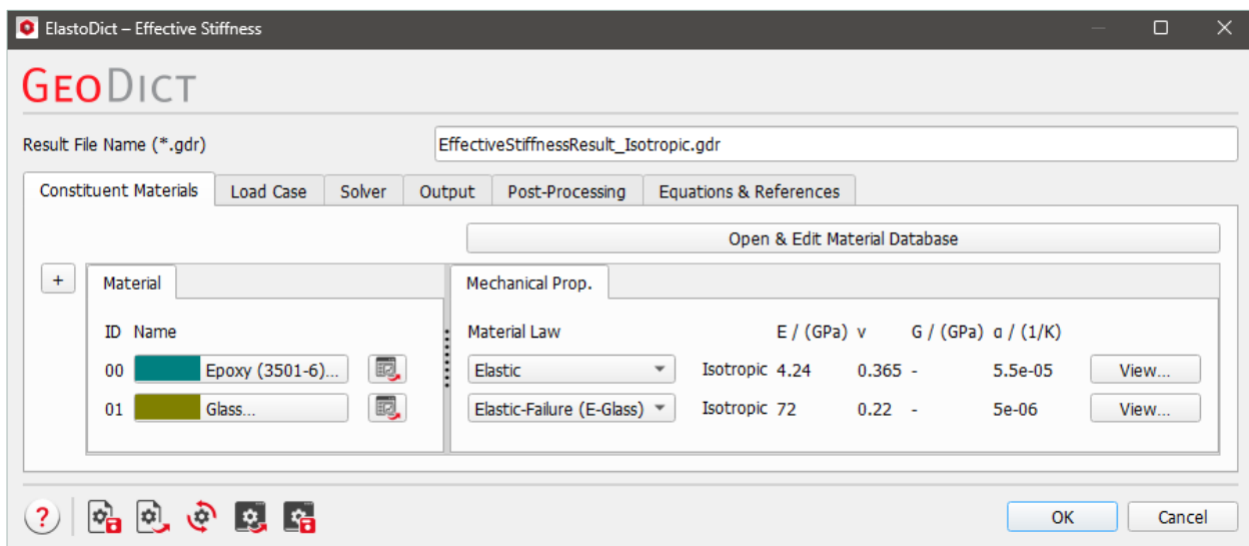
Here, already knowing that the fibers in the structure are isotropic or anisotropic helps explaining how to interpret the results obtained with ElastoDict. However, the true power of the ElastoDict analysis lies in cases where the orientation of the media is not known in advance, such as imported segmented 3D-images obtained from CT (Computed Tomography) scans.

The two composite structures generated with FiberGeo can be loaded for further simulations by selecting **File** → **Open Structure (*.gdt, *.gad)...** in the menu bar.

To start ElastoDict, select **Predict** → **ElastoDict** in the menu bar. For each of the two runs, select **Effective Stiffness** from the pull-down menu.



In the **Effective Stiffness Options** dialog which opens after clicking the **Edit...** button, the values under the tabs are adjusted as seen below. For example, under the Constituent Materials tab, the values are adjusted to the mechanical properties values of **Epoxy 3501-6 – Elastic** and **Glass – Elastic-Failure (E-Glass)**.



Constituent Materials	Load Case	Solver	Output	Post-Processing	Equations & References
Stiffness Mode		Stiffness Tensor			
Load Type		Strain			
Strain Increment / (%)		0.005			
Load Case					
☒ XX		☒ YY		☒ ZZ	
☒ YZ		☒ XZ		☒ XY	
☐ Thermal Expansion					
Boundary Conditions					
☒ Periodic		☐ Symmetric		☐ Mixed	

Constituent Materials	Load Case	Solver	Output	Post-Processing	Equations & References
Solver		FeelMath			
FeelMath		LIR			
Simulation Stopping Criterion					
Tolerance		0.0001			
☐ Maximal Iterations		1000			
☐ Maximal Run Time / (h)		240			
Equivalence Principle		Strain			
Method		Intermediate (Memory Efficient Conjugate Gradient)			
Parallelization		10 Threads (Automatic)		Edit...	
☐ Use Downsampling					
Downsampling Factor		2			
Composite Voxels		Laminate Theory			
Orientation Mode for Anisotropic Materials		Use Orientation from Analytic Objects (gad)			

Constituent Materials
Load Case
Solver
Output
Post-Processing
Equations & References

Write Volume Field

Displacement

☒ X
☒ Y
☒ Z

Check All

Uncheck All

Stress

☒ XX
☒ YY
☒ ZZ

☒ YZ
☒ XZ
☒ XY
☒ Von Mises

Strain

☒ XX
☒ YY
☒ ZZ

☒ YZ
☒ XZ
☒ XY
☒ Von Mises

☒ Material State Variables

☐ Export VTK File (*.vti)

☐ Delete Result Folder after Simulation

Constituent Materials
Load Case
Solver
Output
Post-Processing
Equations & References

☒ Post-Processing in Original Coordinate System

☒ Post-Processing in 'Principal Material Axes' (PMA)

☐ Post-Processing in 'Principal Material Axes' (PMA) in the XY-Plane (fixed Z-Axis)

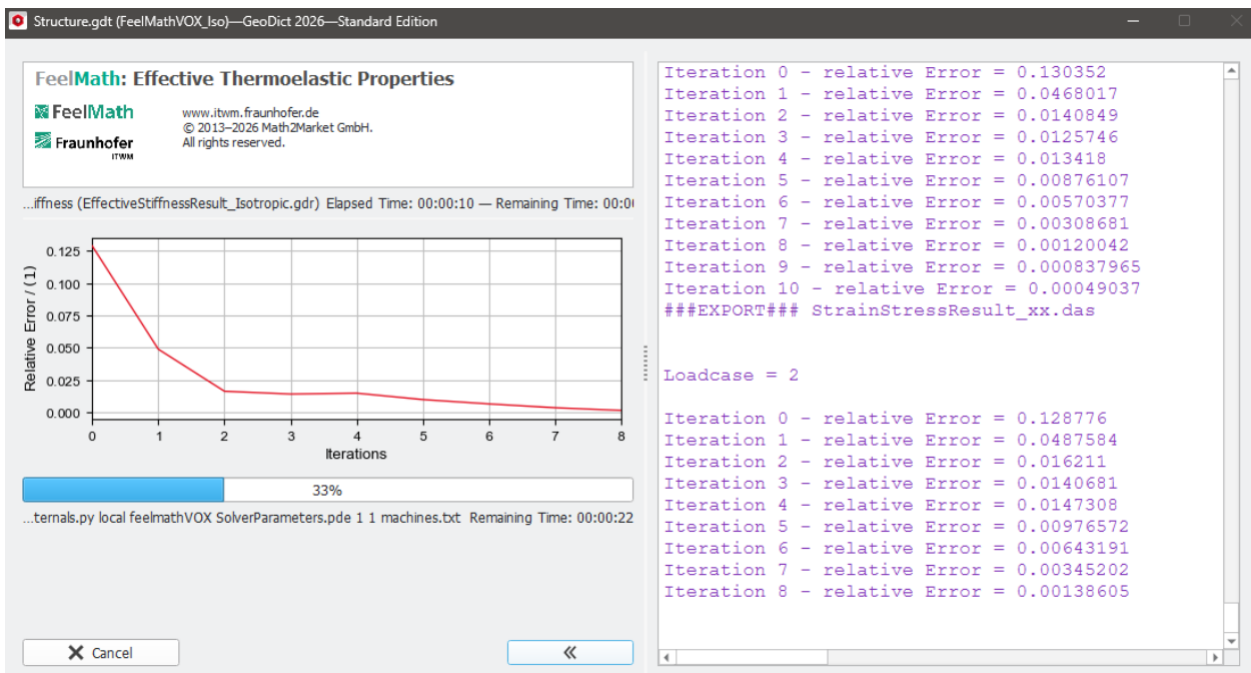
☒ Compute Direction Dependent Stiffness

Angle Resolution / (°)

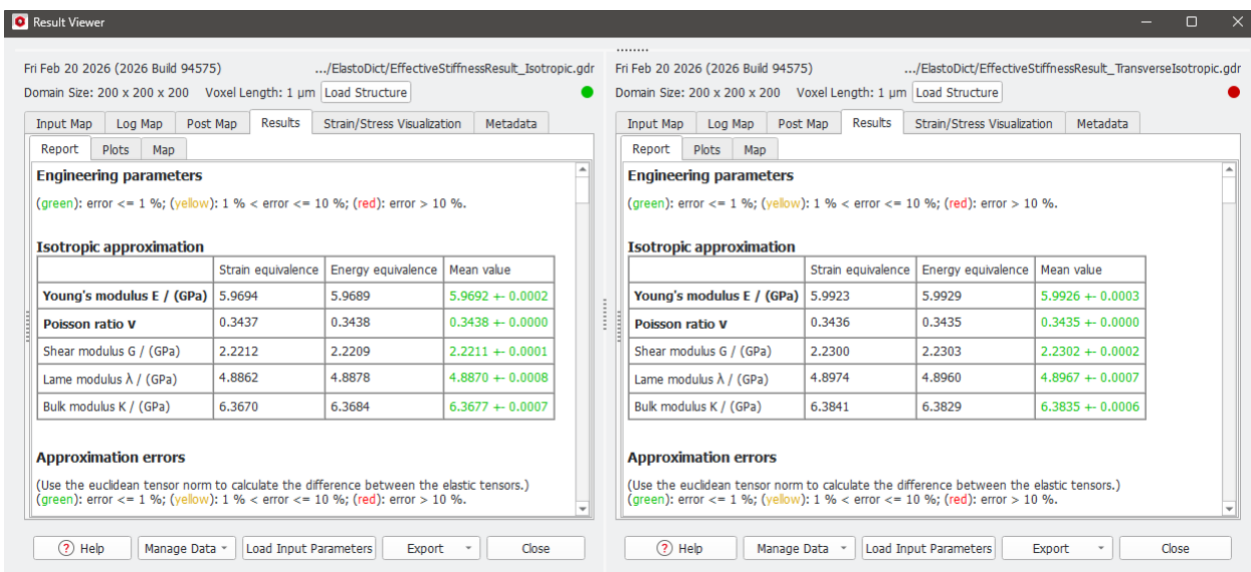
☒ Export Abaqus UMAT for Anisotropic Material

☐ Compute Norris Approximation

Click **OK** to close the solver options dialog and return to the ElastoDict section. After clicking **Run**, the iteration steps made during the calculation can be followed in the console.



At the end of each of the two computations, the Result Viewer of the result files open.



Under the **Results - Report** subtab, several outputs and approximations of the result are provided. These approximations correspond to the two different symmetry classes presented in this example.

Post-Processing Options

- [Anisotropic Elasticity Tensor](#)^[141]
Comparison of the Anisotropic Elasticity Tensor for both composite structure types.
- [Isotropic approximation for Strain Equivalence \(GPa\)](#)^[142]
Comparison of the full 6x6 elasticity tensor and the matrix for the isotropic symmetry class.
- [Estimated Principal Material Axes \(PMA\)](#)^[143]
Using Principal Material Axes to find the composite structure's intrinsic symmetries.

Post-Processing Options

- [Anisotropic Elasticity Tensor in PMA Coordinates](#)¹⁴⁴

Using the anisotropic elasticity tensor in PMA coordinates to get further information of the composite structure's symmetry.

Anisotropic Elasticity Tensor

The **Stiffness Formulation for Strain Equivalence (GPa)** is the first tensor displayed for the **Anisotropic Elasticity tensor**. It is the direct result of the numerical computation and the starting point for all other approximations. This symmetric tensor contains 21 distinct parameters and describes the relationship between the stress and the strain most accurately, see also Wikipedia on [Elasticity Tensor](#).

For the isotropic and the transversely isotropic composite structures being compared here, this tensor looks as follows:

Isotropic Structure

Anisotropic elasticity tensor

Stiffness formulation for Strain equivalence / (GPa)

9.2954	4.8687	4.8439	-0.020077	0.064913	0.006581
4.8689	9.4487	4.8986	-0.036519	0.01093	0.0042821
4.844	4.8985	9.3368	-0.05976	0.079339	0.019859
-0.020742	-0.035824	-0.059043	2.2378	0.011265	0.005995
0.065072	0.0099453	0.079532	0.011263	2.1749	-0.032335
0.0070061	0.0047815	0.019023	0.0059983	-0.032335	2.2032

Transversely Isotropic Structure

Anisotropic elasticity tensor

Stiffness formulation for Strain equivalence / (GPa)

8.4675	4.6582	4.6857	0.0038078	-0.0037467	0.017167
4.6596	12.42	4.7088	0.074576	0.0057563	0.065597
4.6857	4.7072	8.4655	0.023208	0.0054864	0.0049319
0.0044764	0.075028	0.022637	2.1052	0.006555	-0.011456
-0.0026726	0.0052163	0.0064562	0.006398	1.8998	0.0057872
0.016584	0.066091	0.0055608	-0.011441	0.0059422	2.0447

The values obtained for the coefficients defining the relationship between stress and strain can be compared to the matrix for the general elasticity tensor.

$$C_{general} = \begin{pmatrix} C_{11} & C_{12} & C_{13} & C_{14} & C_{15} & C_{16} \\ C_{12} & C_{22} & C_{23} & C_{24} & C_{25} & C_{26} \\ C_{13} & C_{23} & C_{33} & C_{34} & C_{35} & C_{36} \\ C_{14} & C_{24} & C_{34} & C_{44} & C_{45} & C_{46} \\ C_{15} & C_{25} & C_{35} & C_{45} & C_{55} & C_{56} \\ C_{16} & C_{26} & C_{36} & C_{46} & C_{56} & C_{66} \end{pmatrix} \quad (19)$$

The other approximations of this general elasticity tensor are shown when scrolling down in the result file. A large difference between the full tensor and an approximation could mean that:

- The representative volume element is too small, and a larger structure should be used for computations.
- The chosen approximation principally does not hold. For example, the assumption of isotropy for a given structure may be wrong if the stiffness-tensor entries of its isotropic model deviate markedly from the corresponding entries of the true anisotropic tensor.

Isotropic approximation for Strain Equivalence (GPa)

Isotropy means independence of direction. An isotropic material can be described with two parameters, the Young's modulus E , and the Poisson's ratio ν , which are the same in all directions. In the isotropic case there are only two independent elastic coefficients: E, ν or alternatively, the Lamé parameters λ, μ .

For this approximation, the full 6x6 elasticity tensor is matched against the matrix for the isotropic symmetry class (see the section about [Elasticity theory](#)^[112]). From an isotropic stiffness tensor, the parameters E, ν or λ, μ can be calculated as described below.

$$C_{isotropic} = \begin{pmatrix} C_{11} & C_{12} & C_{12} & 0 & 0 & 0 \\ C_{12} & C_{11} & C_{12} & 0 & 0 & 0 \\ C_{12} & C_{12} & C_{11} & 0 & 0 & 0 \\ 0 & 0 & 0 & \frac{C_{11} - C_{12}}{2} & 0 & 0 \\ 0 & 0 & 0 & 0 & \frac{C_{11} - C_{12}}{2} & 0 \\ 0 & 0 & 0 & 0 & 0 & \frac{C_{11} - C_{12}}{2} \end{pmatrix} \quad (20)$$

The Lamé parameters μ (Shear Modulus) and λ (Lamé Modulus) are found by calculating:

$$\mu = \frac{C_{11} - C_{12}}{2} \text{ and } \lambda = C_{12} \quad (21)$$

From the values of μ and λ , the estimates for the effective **Young's modulus** (E) and effective **Poisson's ratio** (ν) are computed as:

$$E = \frac{\mu(2\mu + 3\lambda)}{\mu + \lambda} \quad (22)$$

and

$$\nu = \frac{\lambda}{2(\mu + \lambda)} \quad (23)$$

Other engineering constants can also be obtained by transformation formulas (given, e.g., by the [Wikipedia page on Hooke's Law](#)).

For the isotropic and transversely isotropic structures, the Isotropic Approximation for Strain Equivalence look as follows:

Isotropic Structure

Isotropic approximation for Strain equivalence / (GPa)

9.3286	4.8862	4.8862	0	0	0
4.8862	9.3286	4.8862	0	0	0
4.8862	4.8862	9.3286	0	0	0
0	0	0	2.2212	0	0
0	0	0	0	2.2212	0
0	0	0	0	0	2.2212

Transversely Isotropic Structure

Isotropic approximation for Strain equivalence / (GPa)

9.3574	4.8974	4.8974	0	0	0
4.8974	9.3574	4.8974	0	0	0
4.8974	4.8974	9.3574	0	0	0
0	0	0	2.23	0	0
0	0	0	0	2.23	0
0	0	0	0	0	2.23

Comparing the full anisotropic stiffness matrix with the isotropic approximation, the known structure characteristics are confirmed: the approximated isotropic stiffness values fit well to the corresponding values in the full anisotropic stiffness tensor. But for the transversely isotropic case there is a discrepancy between the approximated isotropic stiffness tensor and the full anisotropic one:

Anisotropic elasticity tensor

Stiffness formulation for Strain equivalence / (GPa)

8.4675	4.6582	4.6857	0.0038078	-0.0037467	0.017167
4.6596	12.42	4.7088	0.074576	0.0057563	0.065597
4.6857	4.7072	8.4655	0.023208	0.0054864	0.0049319
0.0044764	0.075028	0.022637	2.1052	0.006555	-0.011456
-0.0026726	0.0052163	0.0064562	0.006398	1.8998	0.0057872
0.016584	0.066091	0.0055608	-0.011441	0.0059422	2.0447

Isotropic approximation for Strain equivalence / (GPa)

9.3574	4.8974	4.8974	0	0	0
4.8974	9.3574	4.8974	0	0	0
4.8974	4.8974	9.3574	0	0	0
0	0	0	2.23	0	0
0	0	0	0	2.23	0
0	0	0	0	0	2.23

Estimated Principal Material Axes (PMA)

For non-isotropic materials, an appropriate coordinate system is given by the so-called principal material axes. While the Cartesian coordinates are the direction of computation (and experiment, e.g., taking CT and applying loads), the structure material has its own symmetries, e.g., the plane perpendicular to the mean direction of the fibers. These intrinsic symmetries may not be aligned with the Cartesian coordinates, but they define another orthonormal coordinate system. The estimation of principal axes is a means to find these intrinsic symmetries.

The principal material axes are fitted as in [Rutka et al., 2006](#)^[179] who followed [Browaeys and Chevrot, 2004](#)^[178]. They are shown in the results file and are used for approximations of the elasticity tensors.

Isotropic Structure

Estimated principal material axes (PMA)

0.659384	0.232756	0.714866
0.568851	-0.776165	-0.27198
0.49155	0.585998	-0.6442

Transversely Isotropic Structure

Estimated principal material axes (PMA)

0.998779	-0.0443871	-0.0217342
-0.0227877	-0.024598	-0.999438
0.0438287	0.998712	-0.0255266

Here, once again, if the orientation of a structure is not known, the PMAs might give a hint: the PMA of the isotropic structure are more uniformly distributed than the transversely isotropic

structure. The letter has one dominant PMA (value) for each coordinate of the original coordinate system (e.g., the first column: 0.99 for the x-axis and -0.03 and 0.02 for y-and z-axis) whereas the discrepancy in magnitude between the values for the isotropic structure are not high (e.g., first column: 0.64, 0.48, and 0.58).

If we denote by U the matrix with the new coordinate vectors:

$$U = \begin{pmatrix} u_1 & v_1 & w_1 \\ u_2 & v_2 & w_2 \\ u_3 & v_3 & w_3 \end{pmatrix} \quad (24)$$

Then, the coordinates of the elasticity tensor c_{ijkl} in the new coordinate system are

$$c'_{\gamma\delta kl} = U_{\alpha\gamma} U_{\beta\delta} U_{jk} U_{jl} c_{\alpha\beta ij} \quad (25)$$

See the section about the [elasticity theory](#)^[112] for the relationship between the elasticity tensor as 4th order tensor and the matrix notation.

The elasticity tensor for different assumptions like anisotropy or orthotropy, e.g., are displayed behind the estimated PMAs, as described in the following section.

Anisotropic Elasticity Tensor in PMA Coordinates

The second matrix displayed for the **Stiffness Formulation for Strain Equivalence** shows the Anisotropic Elasticity Tensor (GPa) in PMA Coordinates. In particular, the difference from this tensor can be used to estimate the quality of the Orthotropic Approximation, Transverse Isotropic Approximation, Cubic Approximation (all in PMA coordinates), and Isotropic Approximation shown also in the result file.

The Anisotropic Elasticity Tensor in PMA Coordinates looks similar to the **Anisotropic Elasticity Tensor**, but it has been transformed to another coordinate system. The off-diagonal coefficients are smaller, so that setting them to zero would lead to smaller errors. For our examples, we obtain:

Isotropic Structure

Anisotropic elasticity tensor / (GPa) in PMA coordinates

9.3214	4.9295	4.9003	0.0079969	-0.0058504	0.0065828
4.9293	9.4801	4.8847	-0.018564	-0.010204	-0.0098945
4.9004	4.8852	9.0725	0.010499	0.015244	0.0037883
0.0078268	-0.018464	0.010633	2.236	0.0028486	-0.0086857
-0.0059422	-0.011018	0.015693	0.0035113	2.2204	0.0076584
0.0064374	-0.010723	0.0041985	-0.0093033	0.0076534	2.263

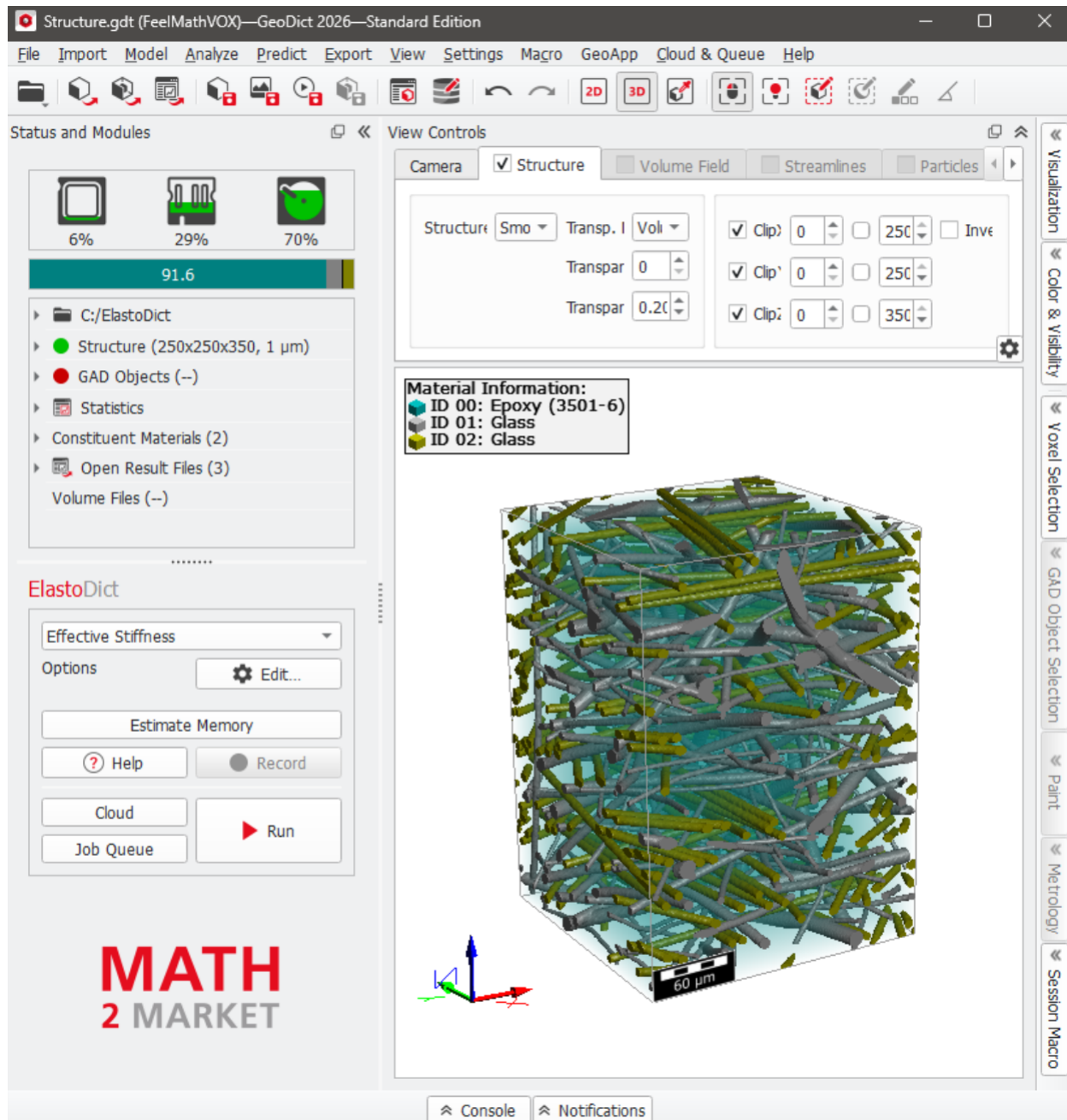
Transversely Isotropic Structure

Anisotropic elasticity tensor / (GPa) in PMA coordinates

8.4657	4.6861	4.6584	-0.0044165	-0.011838	-0.0035319
4.6858	8.4629	4.7063	-0.010852	-0.0057902	0.0037868
4.6598	4.7078	12.425	0.015381	0.014832	0.0070146
-0.0051046	-0.010228	0.015002	2.1057	-0.0096092	-0.0026607
-0.011294	-0.006371	0.014365	-0.0095958	2.0434	-0.0021071
-0.0024581	0.0047533	0.0064355	-0.0026054	-0.0019326	1.9001

1.5.9 Visualization Example

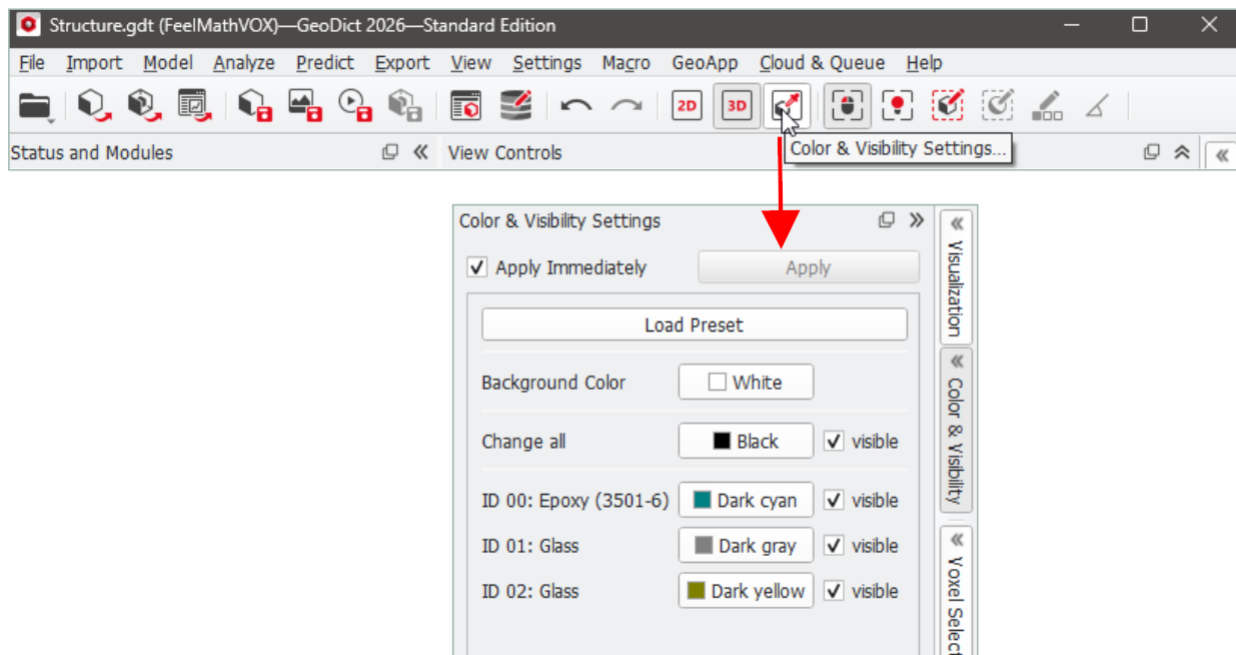
The general visualization options for ElastoDict-Effective Stiffness are presented here. These options apply for arbitrary ElastoDict-Effective Stiffness runs. The simulation settings are not relevant here.



Render the structure in 3D (by clicking ) and go to the **Structure** tab in the visualization panel (above the Visualization area). Set the **Structure Renderer** to **Smooth**. Then, select **Transp. Mode** to **Volume**, set the **Transparent Material ID** to 00, and set the **Transparency** value to 0.10.

Usually, the material with ID 00 is set to invisible, so the transparency setting has no effect.

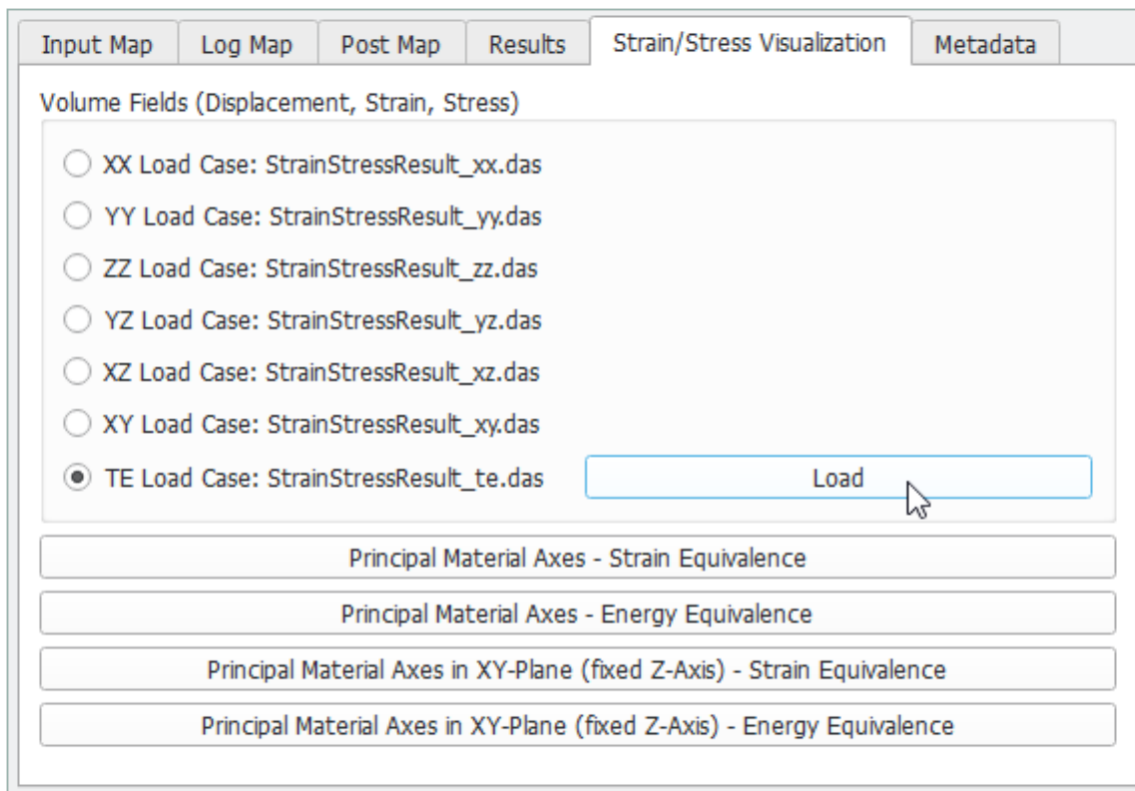
Click  to open the **Color & Visibility Settings** dialog and check **visible** to enable visibility for the material ID 00.



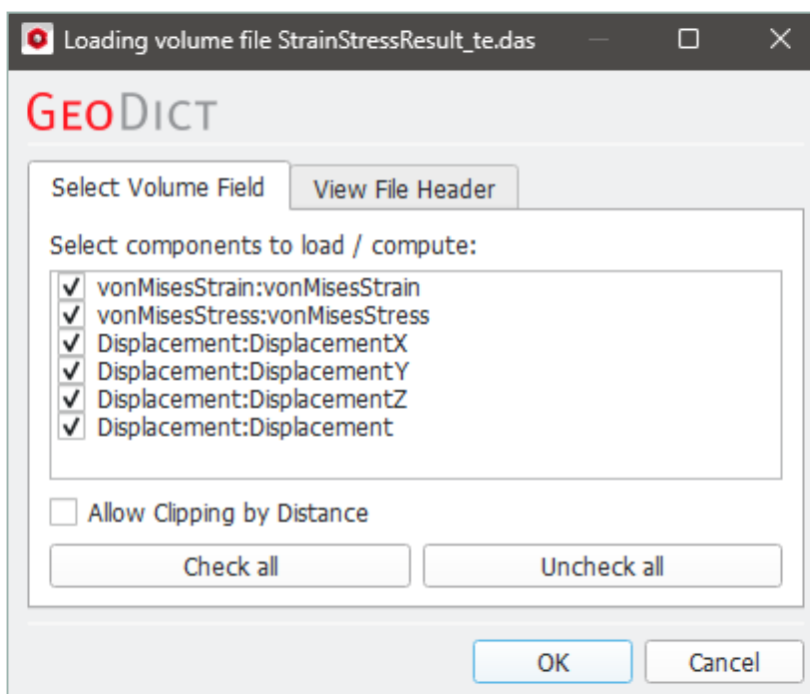
You can find more on the Visualization options in the Visualization chapter.

Visualization of displacement and stresses

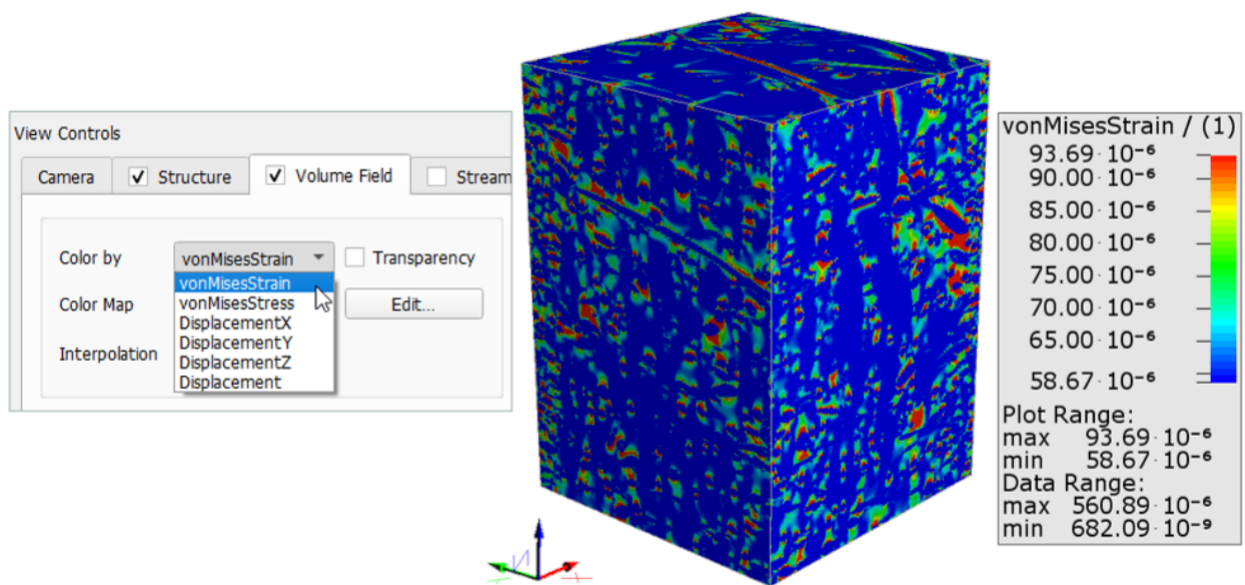
After all selected load cases have been computed, result fields can be loaded by choosing the corresponding *.das file and clicking **Load Results** in the Result Viewer (*.gdr).



In the next dialog, choose the components which should be visualized: Strain and Stress for all calculated load cases, like, e.g., von Mises Strain and Stress, DisplacementX, DisplacementY, DisplacementZ, and Displacement. Check the load cases of interest and click **OK**.



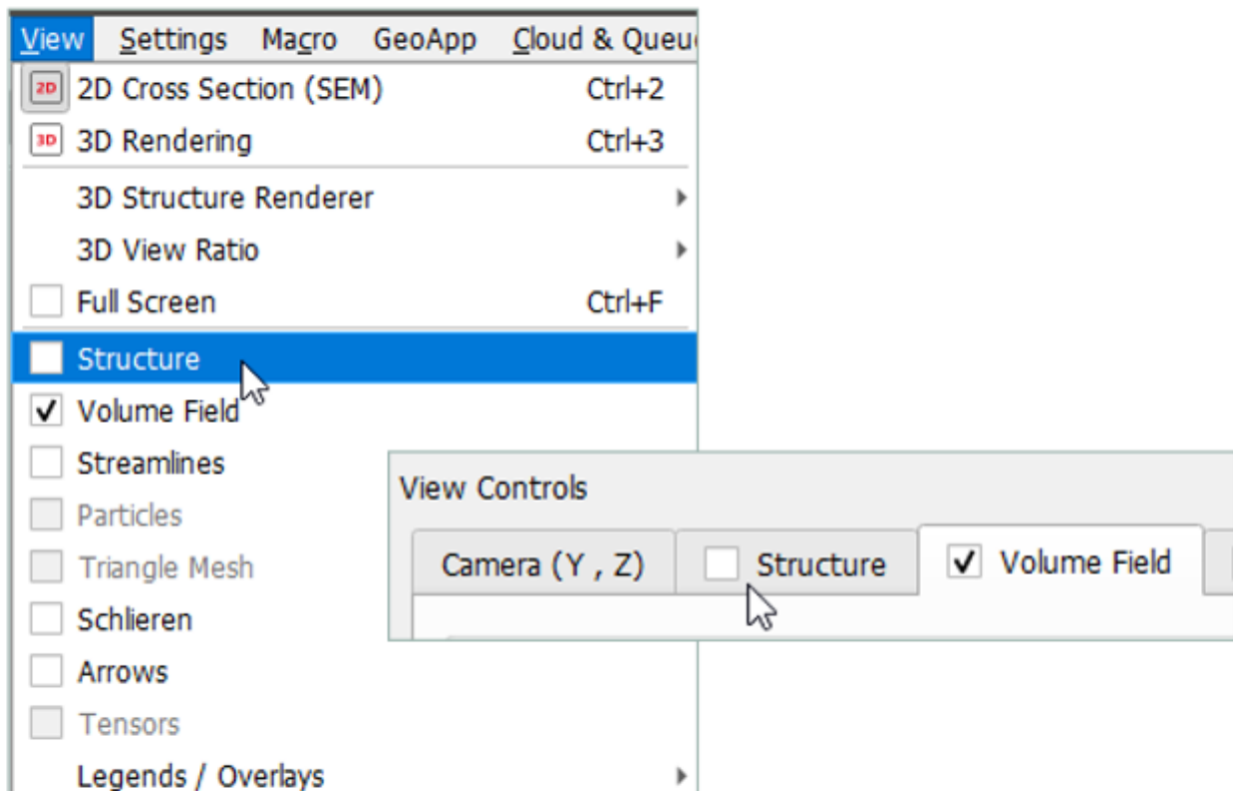
Choose the displayed data through the **Color by** pull-down menu, under the **Volume Field** tab, in the Visualization panel above the Visualization Area.



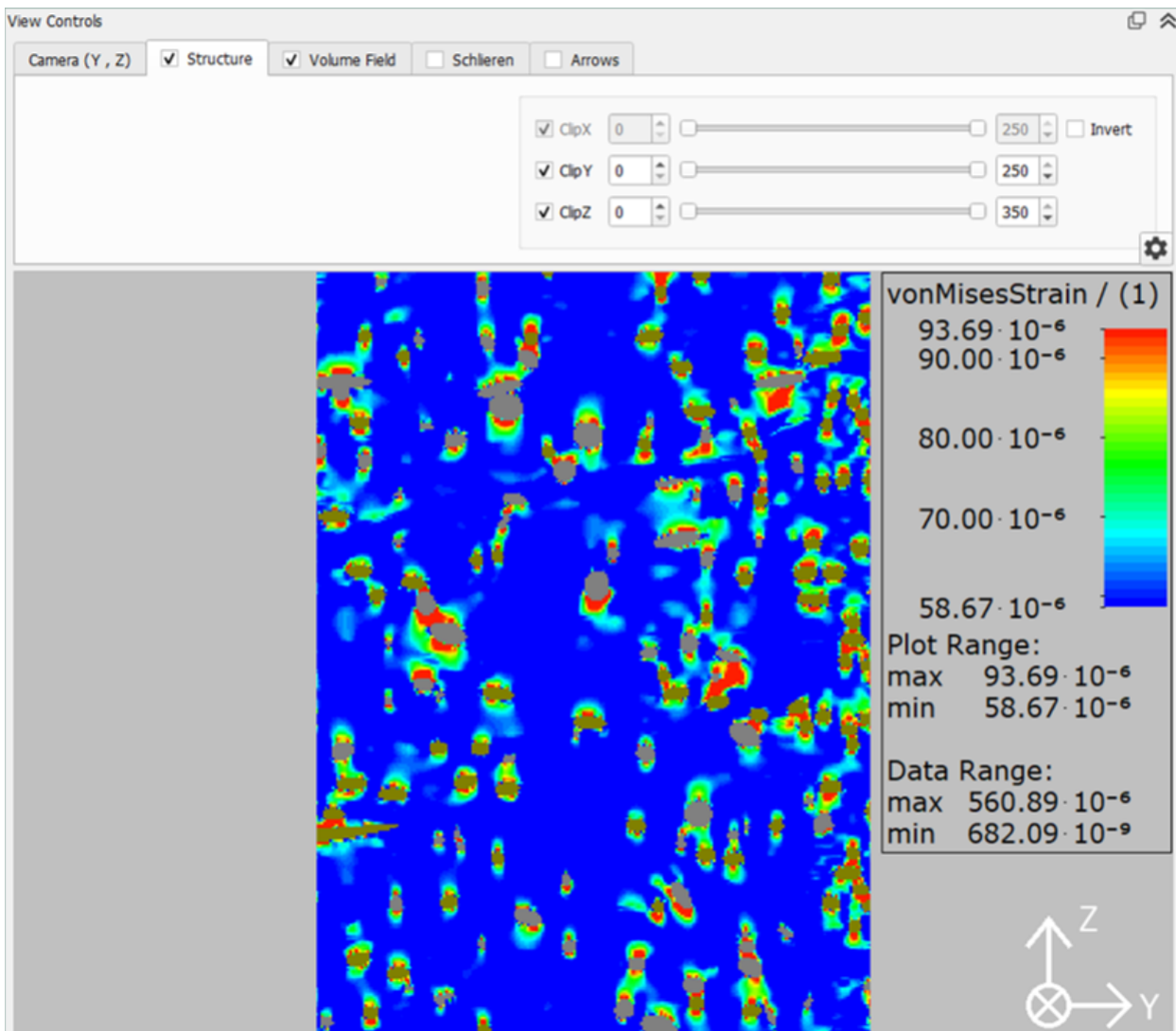
Tools are available for the visualization in 2D Cross Section (SEM) and in 3D Rendering.

For 2D visualization, select **View** → **2D Cross Section (SEM)** in the Menu bar.

The structure in the model can be set to invisible to observe only the results (Volume Field). For this, select **View** and un-check **Structure** in the menu bar or directly in the **View Controls** tab.

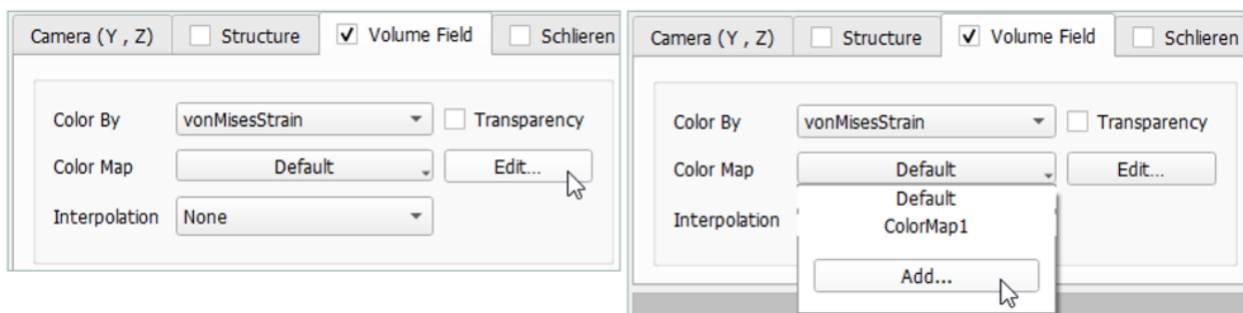


The control of 2D visualization parameters is also done through the Visualization panel above the Visualization Area.



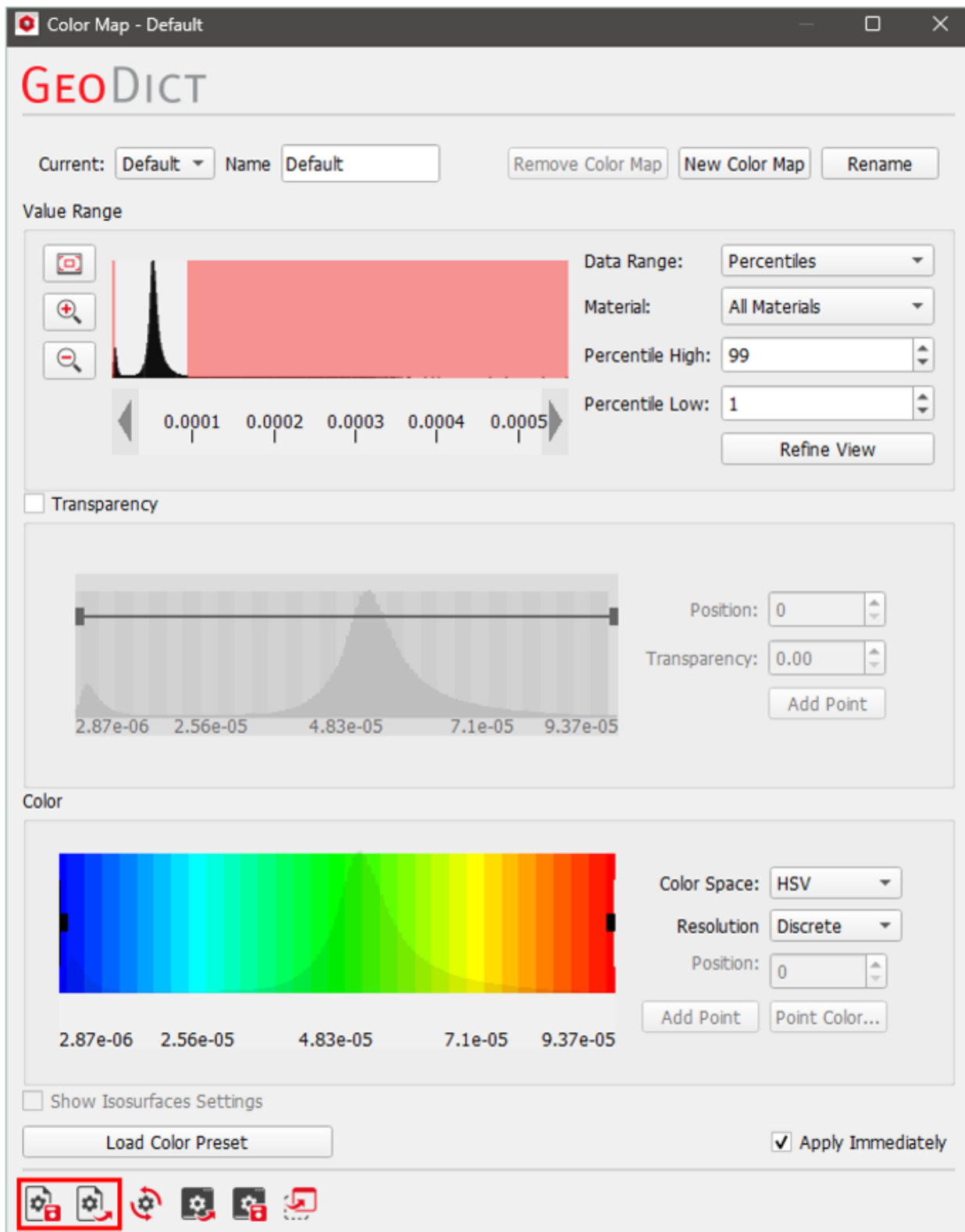
In 2D and 3D Rendering, a color bar appears to the right (if the default position is not changed) during the visualization of the result file, indicating the gradation of the currently selected property.

Click the **Edit...** button to edit the Default Color Map or add a new Colormap by choosing **Add...** in the dropdown menu (and then click the **Edit...** button). Alternatively, new colormaps can also be defined in the **Color Map** dialog.

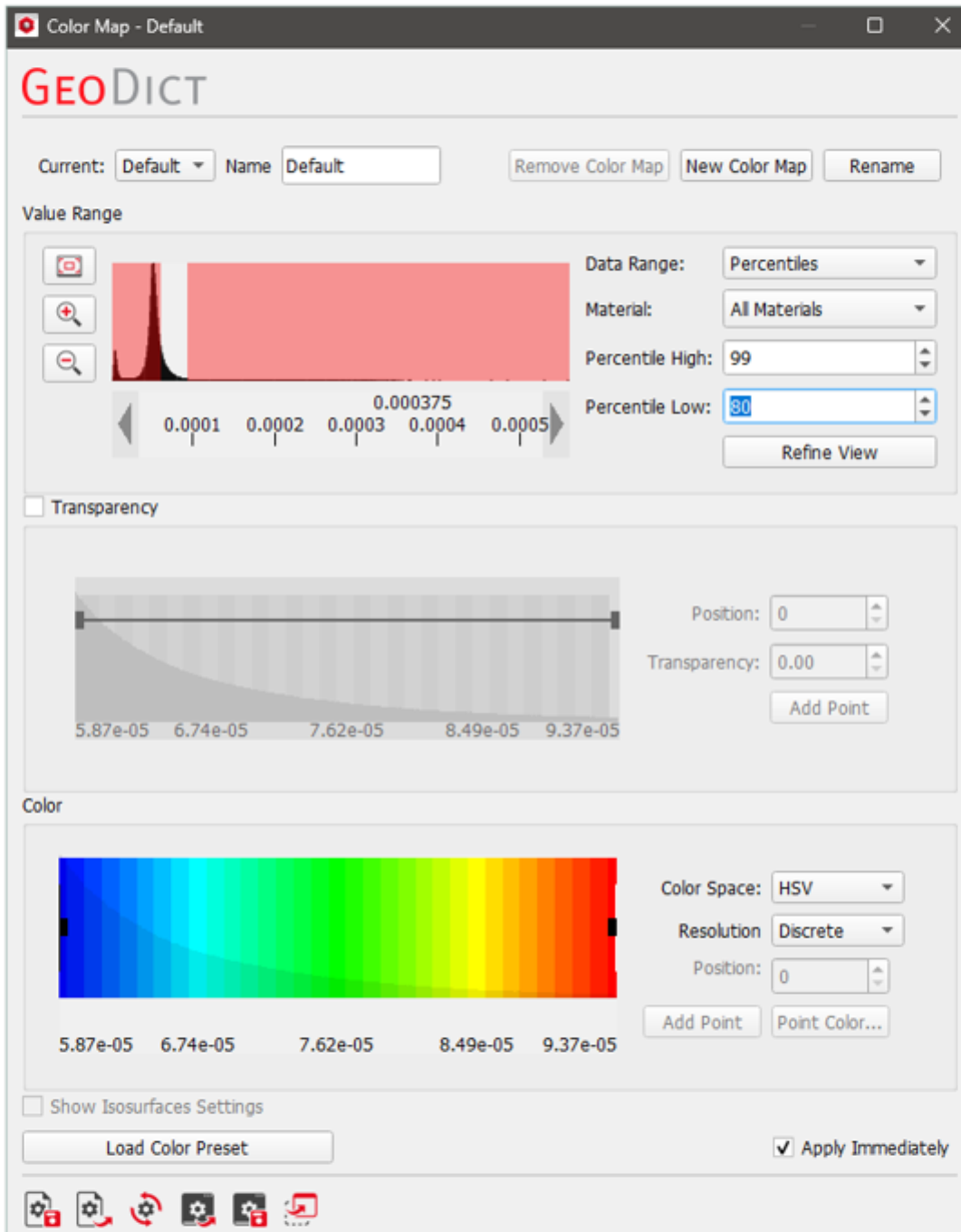


In the **Color Map** dialog, define the upper and lower limits of the color scale, add transparency to the visualized result field, and assign a particular **Color Space** (RGB, HSV, LAB).

The values entered in the **Color Map** dialog can be saved into a *.gps (GeoDict Project Settings) file by clicking on the first icon in the lower left corner. By clicking on the second icon settings can be loaded again from a *.gps file.

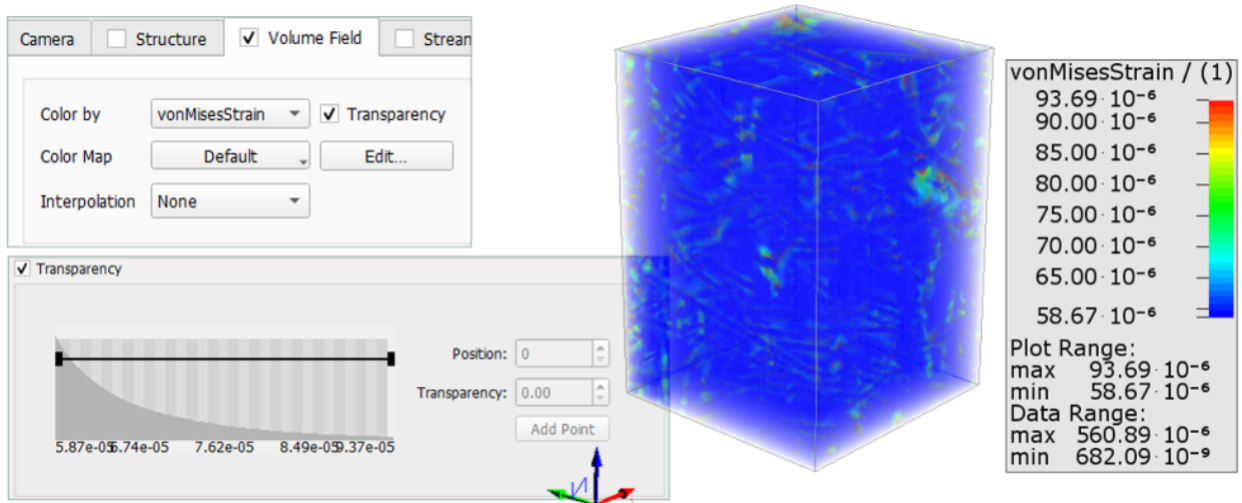


As you can observe in the histogram in the **Value Range** panel, the values concentrate in an area below a value 5.0×10^{-5} . To highlight the areas with high strain, we increase the **Percentile Low** to 80.

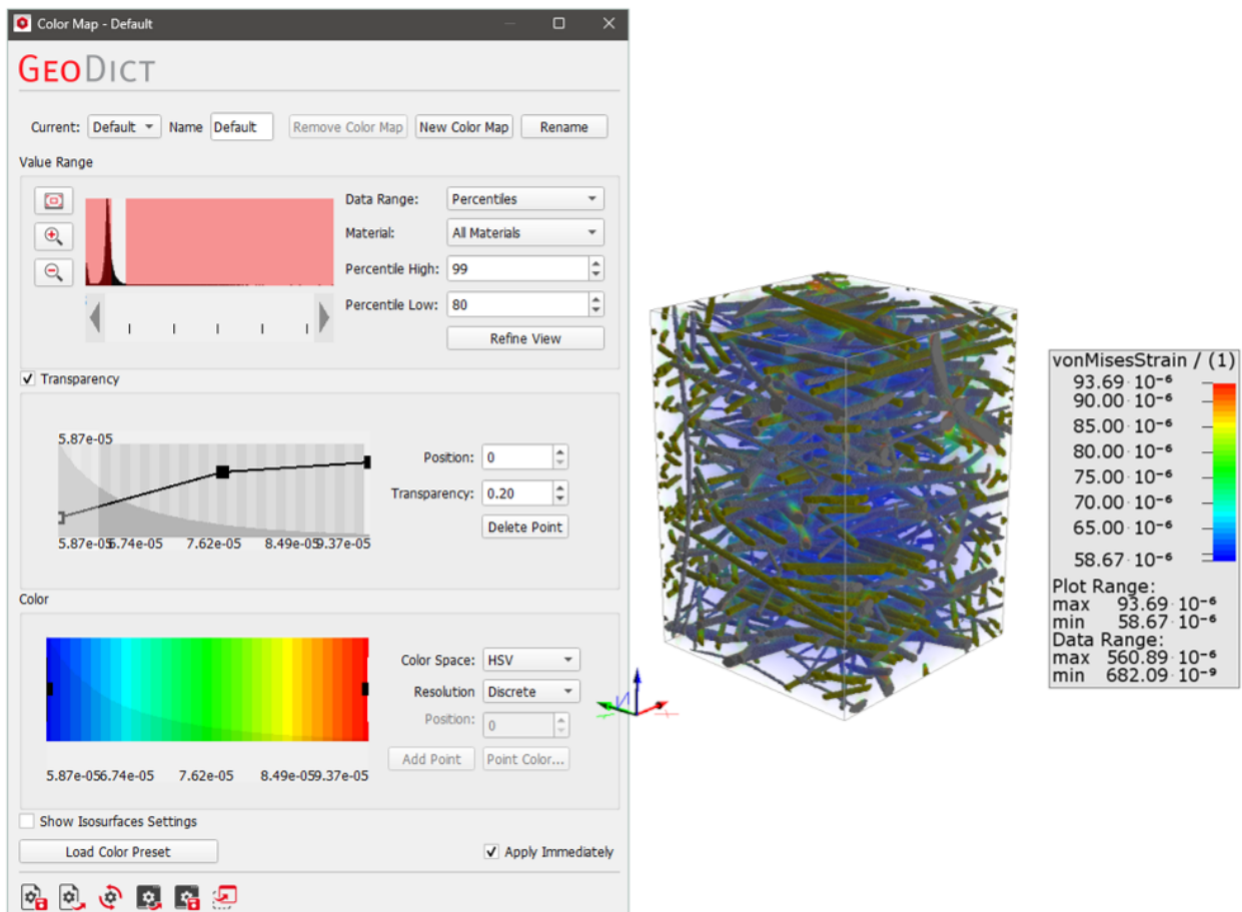


Note how the color of the result field in the Visualization area and the histogram change accordingly.

To emphasize the areas where the strain is elevated, you can activate **Transparency** under the Volume Field tab or directly in the **Color Map** dialog. Make sure that **No Interpolation** is selected in the **Interpolation** dropdown menu and then modify the transparency curve in the Color Map dialog, as indicated in the image on the right.



In the figure below, the transparency for low values is increased to see through these areas, and upper values with a high transparency are highlighted.



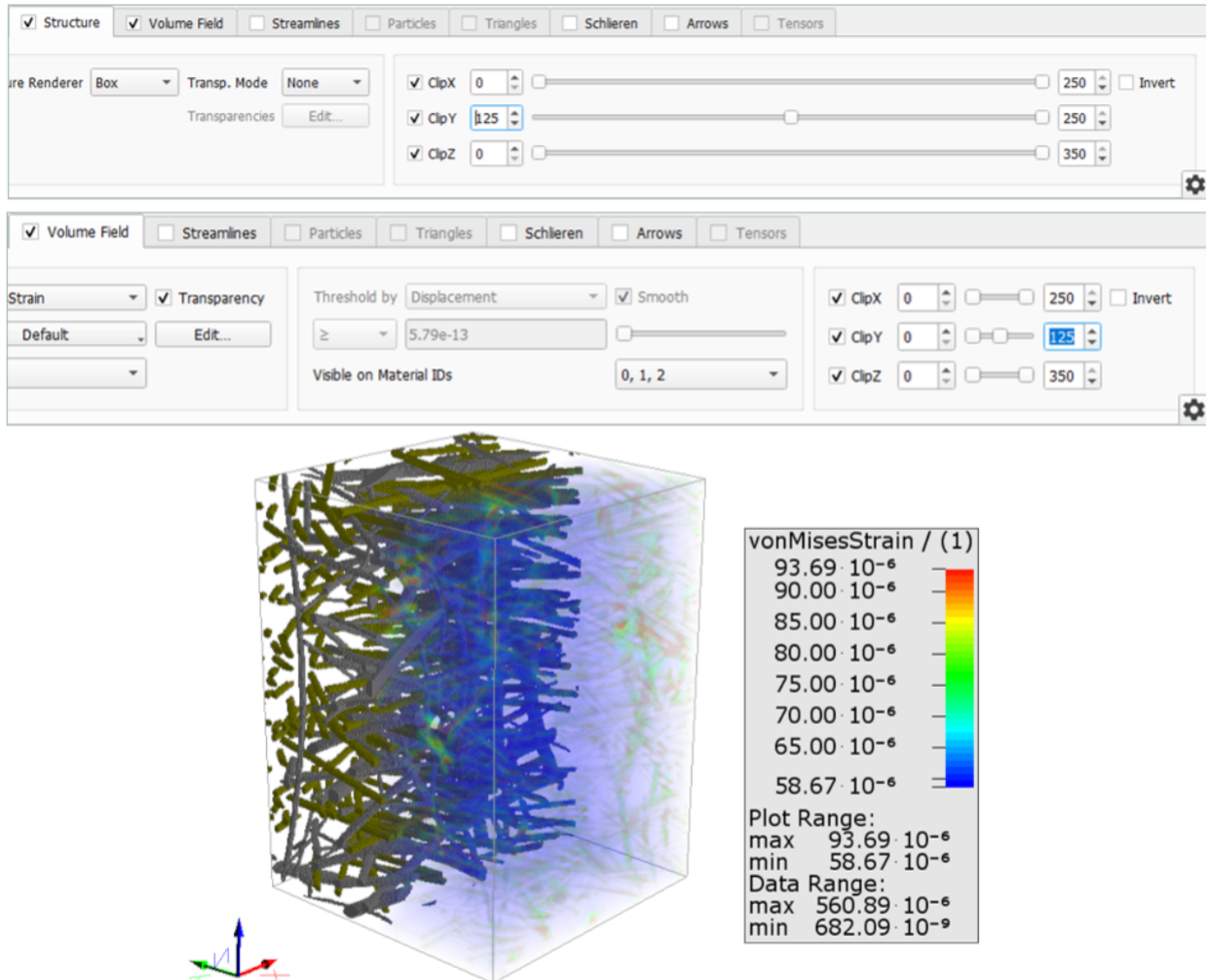
Topics of this section

- [Clipping Structure and Results](#)¹⁵³
Description of Clipping settings.
- [Visualization of Principal Material Axes](#)¹⁵³
Description of how to visualize PMA.

Clipping Structure and Results

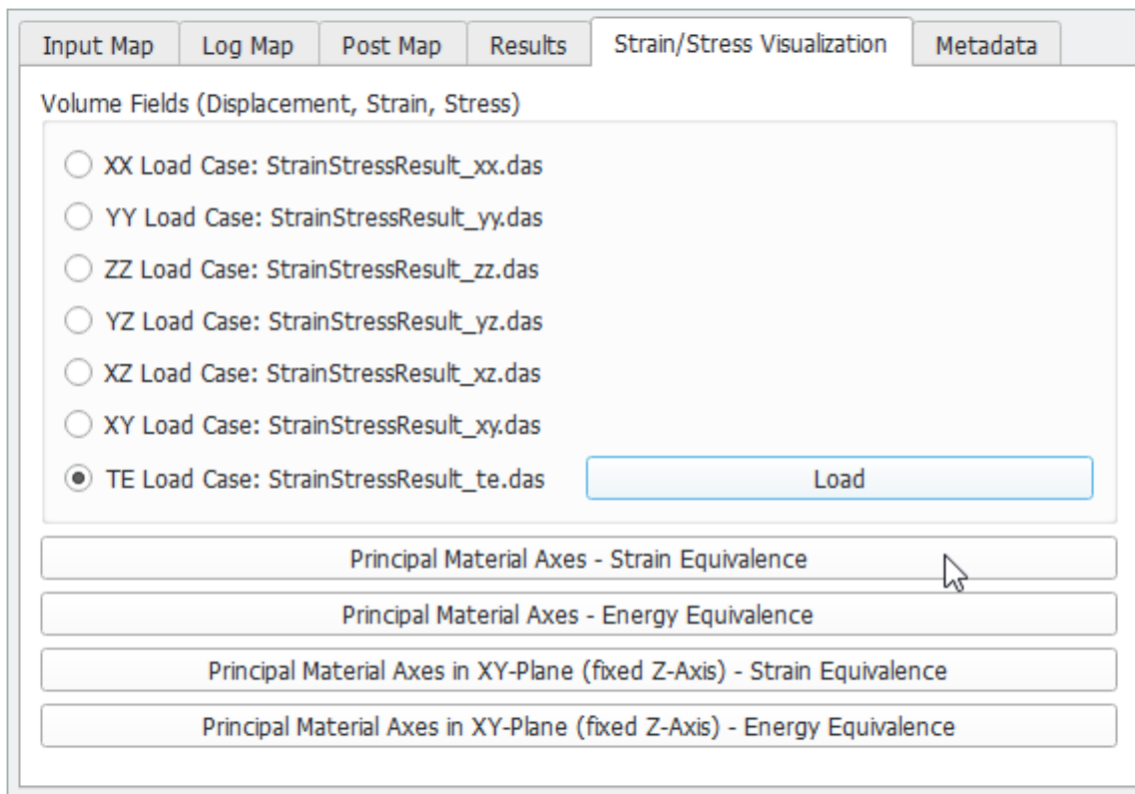
The displayed results can also be clipped spatially in all three directions (**ClipX**, **ClipY**, **ClipZ**) with the sliders or by direct input. This enables the simultaneous visualization of the structure and the corresponding results.

Make sure that the structure is set to visible (**View** → checked **Structure**).

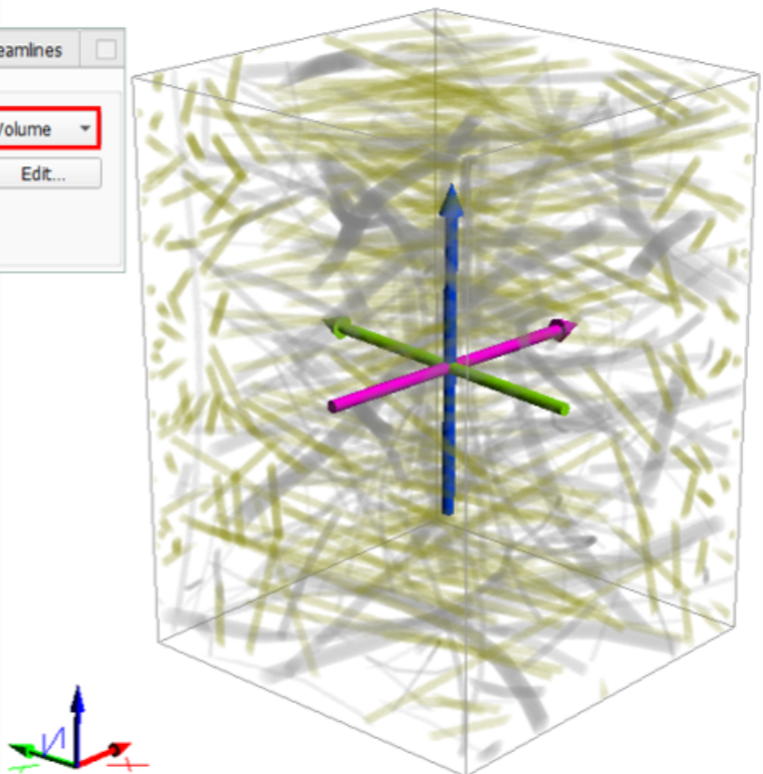
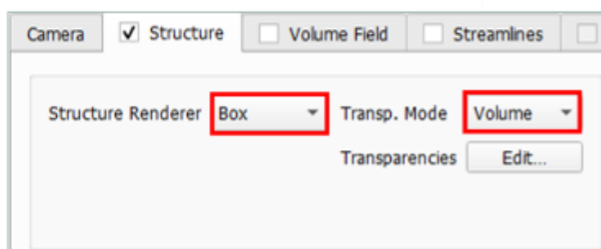


Visualization of Principal Material Axes

When all load cases (XX, YY, ZZ, XY, XZ, YZ) have been computed, the principal material axis can be visualized by clicking **Principal Material Axes – Strain Equivalence** or **Principal Material Axes - Energy Equivalence** under the Strain/Stress Visualization tab of the Result Viewer.

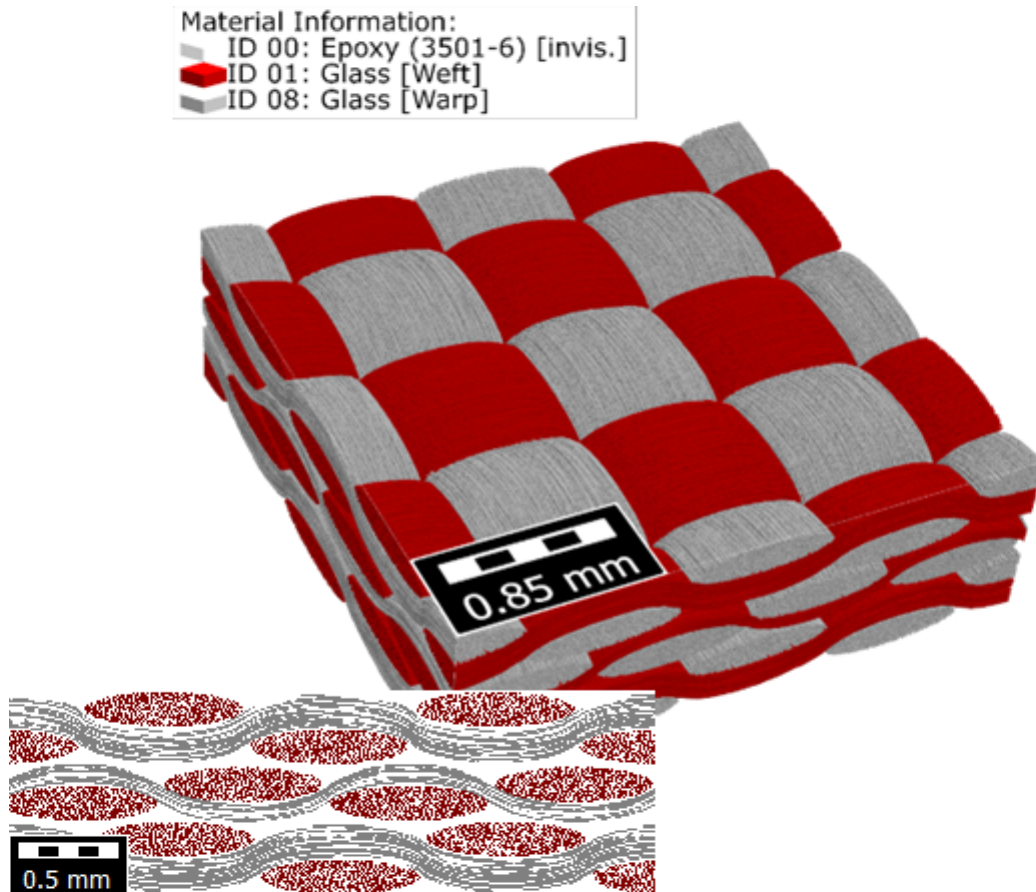


The axes are best to observe when the structure is transparent. For this, switch off the visualization of the results (**View** → uncheck **Volume Fields**), choose the Structure Renderer to **Box** and set the **Transp. Mode** to **Volume** (both options under the **Structure** tab in the Visualization panel).



1.5.10 Benchmark Results for Effective Stiffness

For the weave structure shown here, the effective stiffness is computed with ElastoDict-Effective Stiffness and the FeelMath solver. All fibers are fully resolved in the structure of 2,590 x 2,590 x 821 voxels (i.e., in total ~5.5 billion voxels).



For strain load type and periodic boundary conditions, all six load cases are computed using the conjugate gradient method.

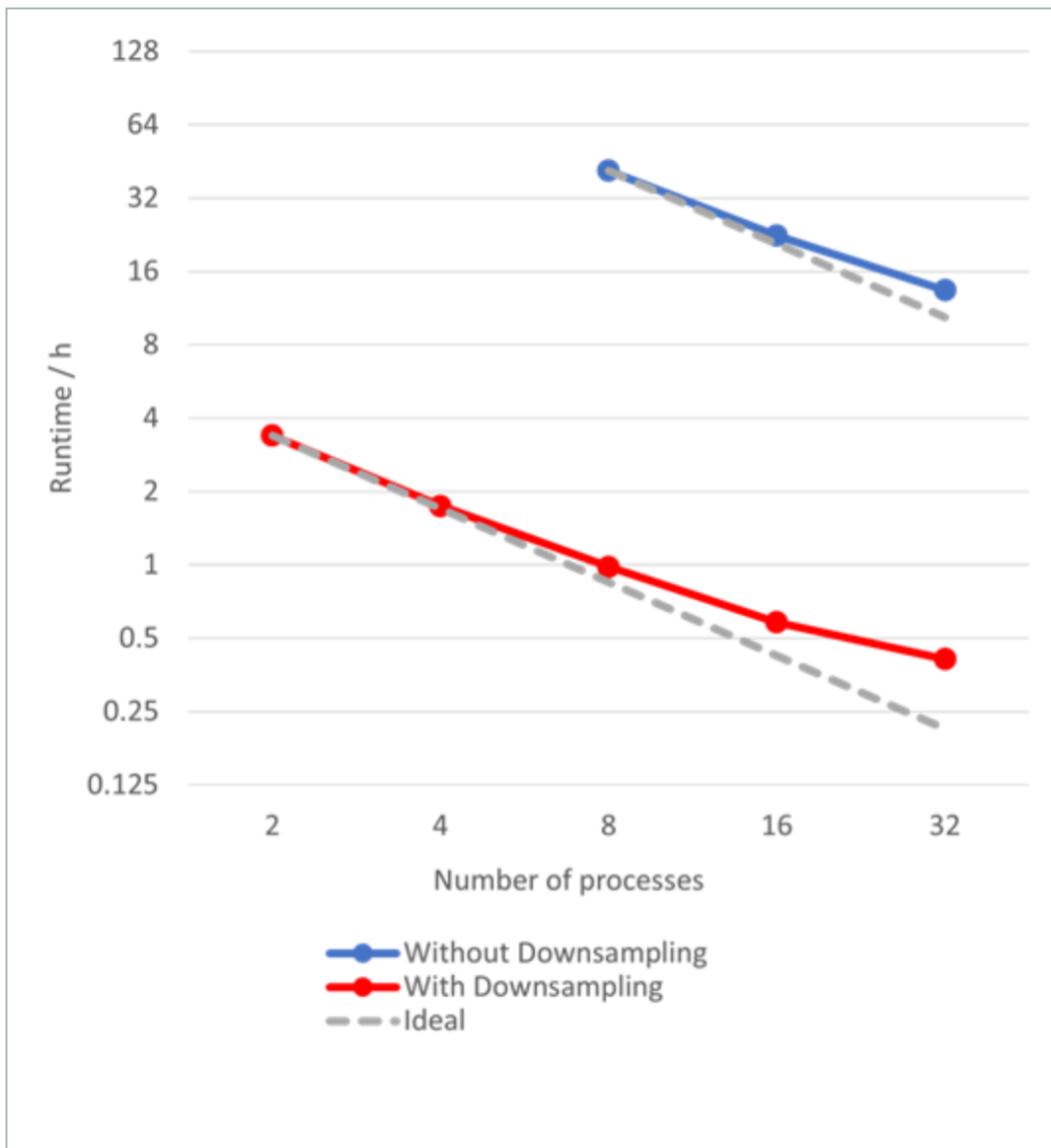
The computation was run on our server, with 2 x Intel E5-2697A v4 processors with 16 cores each, running with a maximum of 3.60 GHz, and 1024 GB RAM.

To show the possible benefit of using downsampling, runtime and memory requirements are compared with and without downsampling, for a different number of processes, using GeoDict 2025 (r81163).

Runtime results are shown in the following figure. The computation, which needs 40h with 8 processes, can be run in 25 min with 32 processes and downsampling. Even without increasing the number of processes, the runtime is reduced by 97%.

The memory requirement is reduced by 97% as well. The computation that needed 1TB of RAM without downsampling requires only 28 GB with downsampling and thus can be run on a standard laptop or desktop computer.

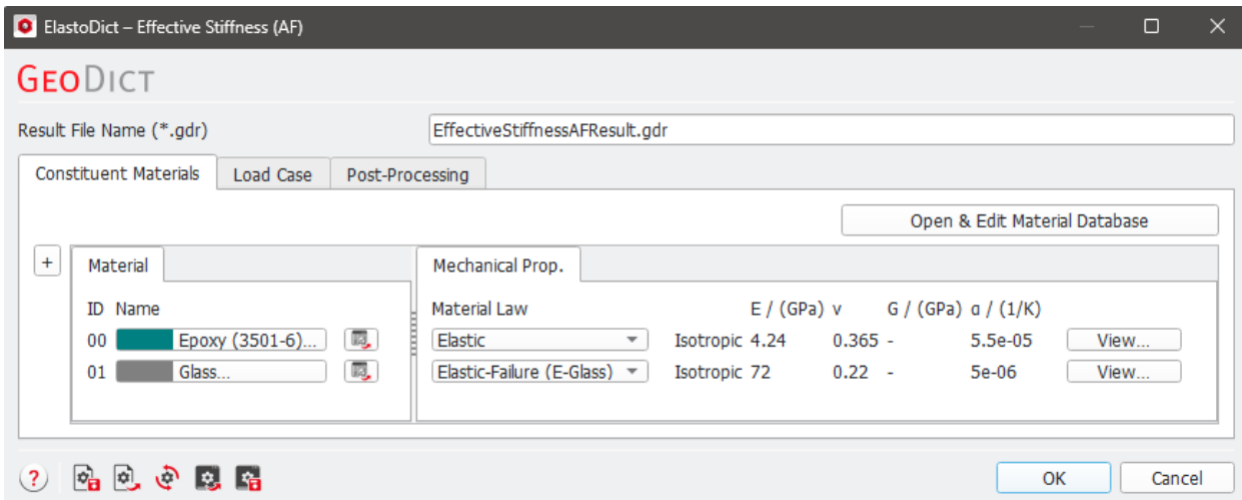
The ideal speedup, i.e., getting half the runtime for twice the number of processes is also shown in the figure for the computation with downsampling. Computations with FeelMath have a very good speedup compared to the ideal line.



1.6 Effective Stiffness (AF)

With ElastoDict-Effective Stiffness (AF), the effective elastic properties of a given structure can be computed using an analytical approach.

The FeelMath-AF solver provides a fast analytic approximation of the effective elastic properties. It computes the **Voigt** (upper bound), the **Reuss** (lower bound) and the **Mori-Tanaka** approximations. AF stands for analytic functions, since here the equations are solved by an analytic approximation instead of an iterative numerical solver.



ElastoDict-Effective Stiffness (AF) only works for linear elastic materials.

Topics of this section

- [Constituent Materials](#)¹⁵⁷
Description of how to use and setup material laws for ElastoDict - Effective Stiffness (AF).
- [Load Case](#)¹⁶⁰
Description of different load case options.
- [Post-processing](#)¹⁶¹
Description of the postprocessing options.
- [Result File](#)¹⁶¹
Description of the Result File generated by an ElastoDict - Effective Stiffness (AF) simulation step.

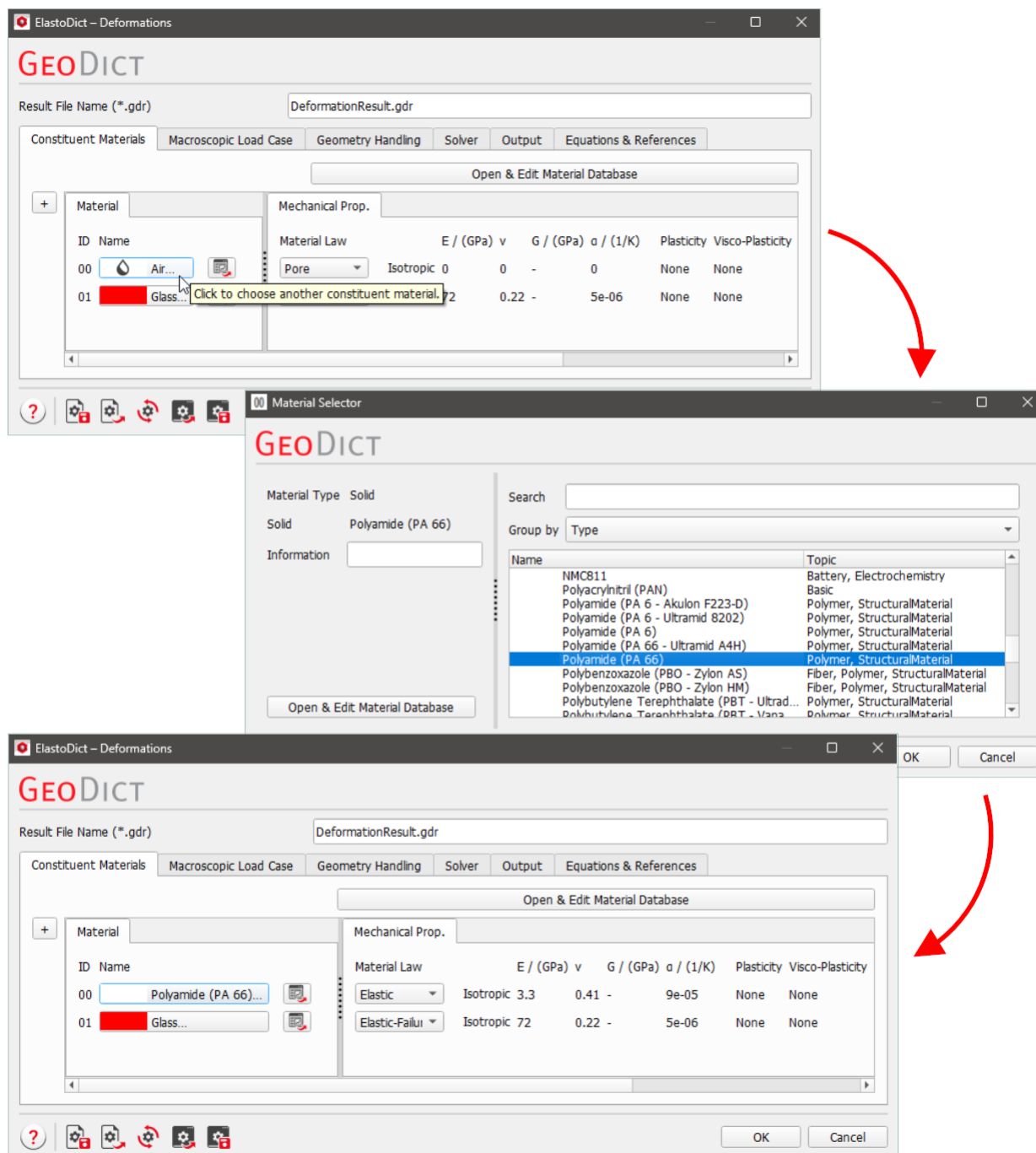
1.6.1 Constituent Materials

ElastoDict-**Effective Stiffness (AF)** only works for linear materials. Choosing nonlinear materials results in an error message when starting the solver.

The materials in the structure model need to be defined under the **Constituent Materials** tab, so that their mechanical parameters, such as the Young's modulus (E) and the Poisson's ratio (ν), are taken into account.

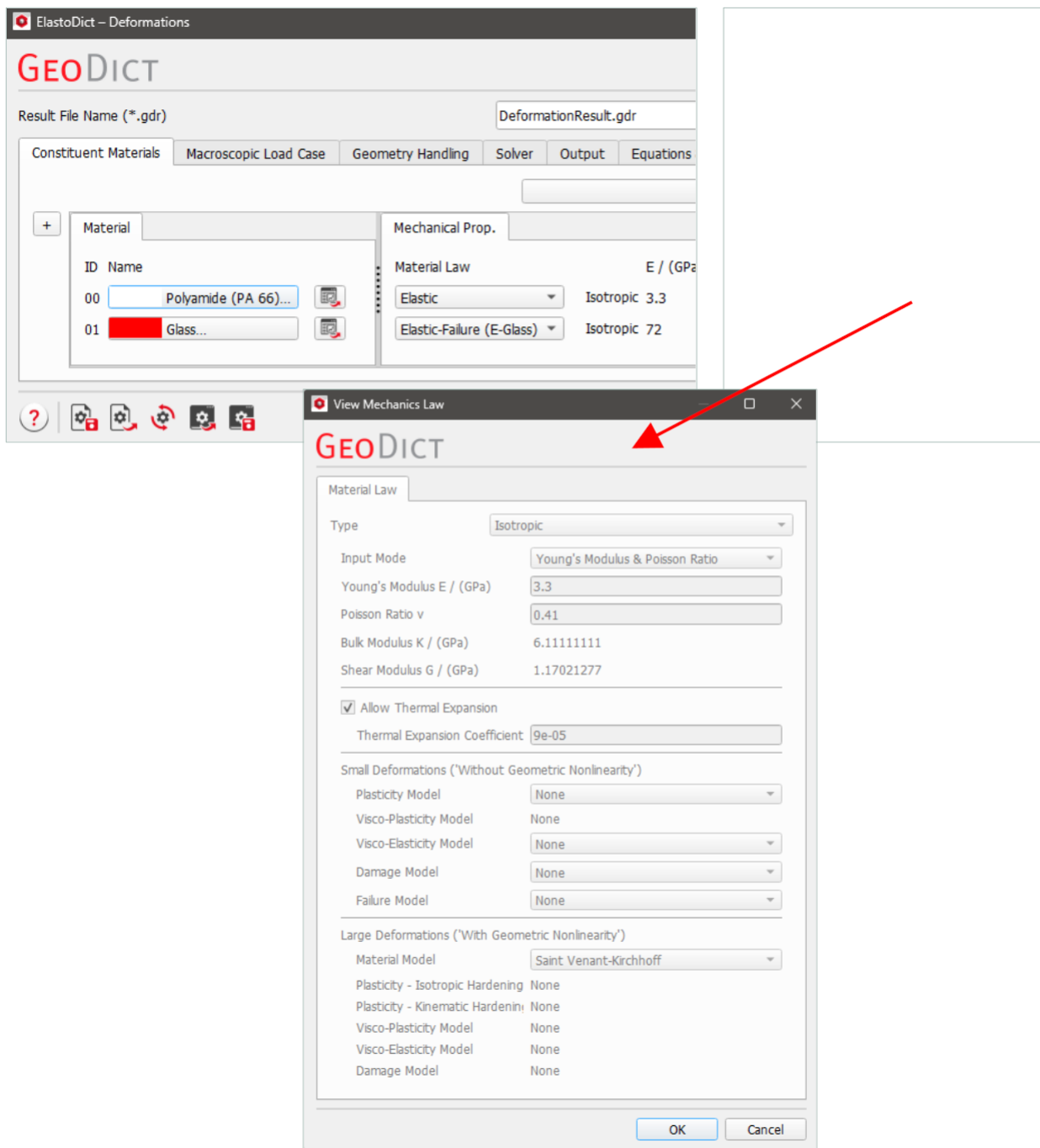
The material for a material ID can be selected by clicking on the corresponding material name. Then, the **Material Selector** dialog is opened.

The mechanical properties of the structure's constituent materials are taken directly from the GeoDict Material Database. To enter the material parameters for a linear elastic material manually, choose **Manual (Solid)** as material.

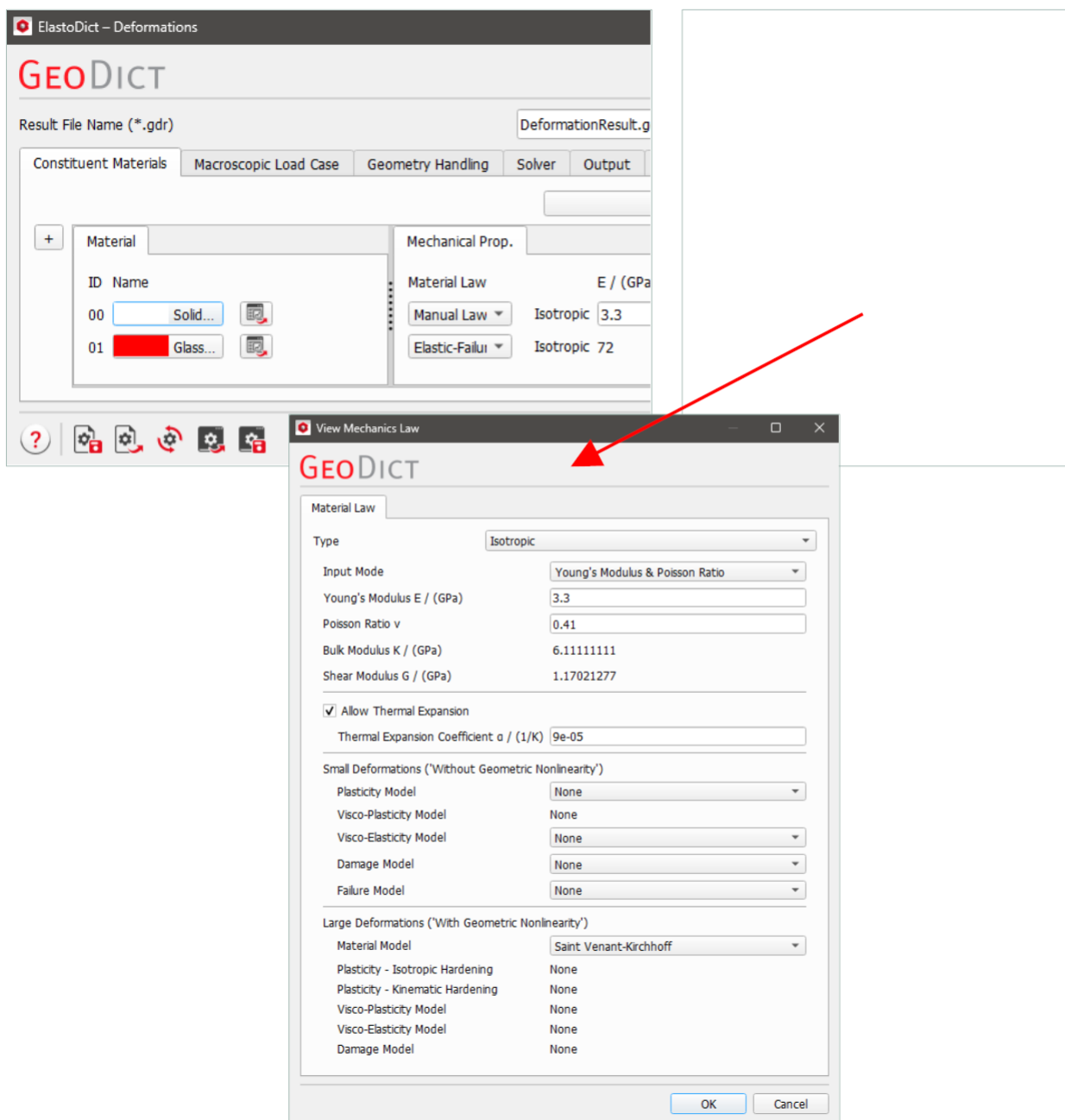


Any values entered manually, e.g., those for a user-defined constituent material, can also be added to the material database through the **Edit Material Database...** button. Nonlinear materials can also be created in the GeoDict **Material Database**.

For materials from the material database, the **View...** button on the right in the **Constituent Materials** tab shows all parameters of the material.



For **Manual** materials, the **View...** button on the right in the **Constituent Materials** tab changes to **Edit...** and allows to edit all parameters of the material.

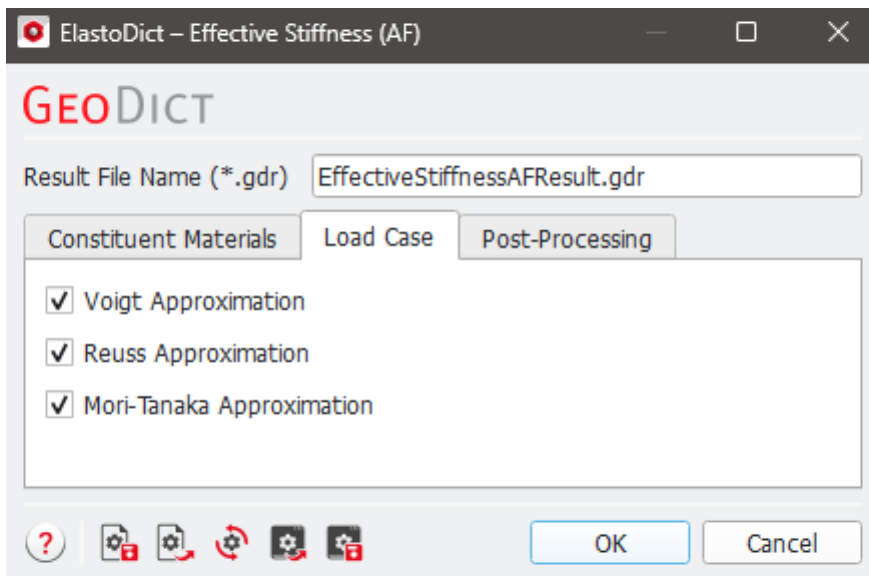


More information on editing, expanding, and using the GeoDict Material Database is available in the Material Database chapter.

1.6.2 Load Case

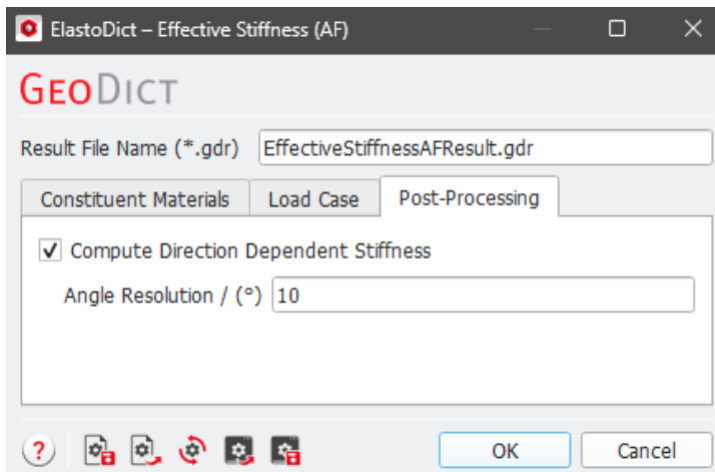
The **Voigt** and **Reuss** approximations can be interpreted as the ratio of average stress and average strain within the microstructure. The stress and strain are generally unknown and expected to be non-uniform.

The upper bound (**Voigt**) assumes that the strain is uniform everywhere. The lower bound (**Reuss**) assumes that the stress is uniform everywhere. The **Mori-Tanaka** approximation works for Ellipsoids and circular fibers and needs the input of analytic data (*.gad files).



1.6.3 Post-processing

Analogous to ElastoDict-Effective Stiffness, the post-processing options can also be chosen after the computation in the Result Viewer. For ElastoDict-Effective Stiffness (AF), only the option **Compute Direction Dependent Stiffness**, and the option to set a corresponding **Angle Resolution** is available. Refer to the corresponding ElastoDict-[Effective Stiffness](#)¹³² section for further details.



1.6.4 Result File

The handling of the result file is analogous to that of the ElastoDict-[Deformations](#)⁷⁰. Therefore, only the features specific for ElastoDict-Effective Stiffness (AF) are explained here in detail.

The **Results** tab gives access to the computed values for the parameters calculated by the selected ElastoDict process. The **Results** tab is divided into three subtabs:

- The **Report** subtab shows the values of the computed mechanical properties. These values can be exported in various ways by selecting an option from the **Export** pull-down menu at the bottom of the Result Viewer.

An analysis of the effective stiffness tensors can be carried out on the tables shown. This analysis allows to judge, whether the material exhibits some degree of symmetry, such as transverse isotropy.

The screenshot shows the 'Report' subtab of the ElastoDict software. The left sidebar contains settings for the analysis, including 'Stress Unit' set to GPa, 'Direction Dependent Stiffness' checked, 'Angle Resolution' set to 10 degrees, and 'Back-up result file' checked. The main area displays the following data:

Engineering parameters

Isotropic approximation

	Voigt	Reuss	Mori-Tanaka
Young's modulus E / (GPa)	11.2375	4.6806	5.9578
Poisson ratio ν	0.2918	0.3641	0.3445
Shear modulus G / (GPa)	4.3494	1.7157	2.2156
Lame modulus λ / (GPa)	6.0983	4.5947	4.9081
Bulk modulus K / (GPa)	8.9978	5.7385	6.3852

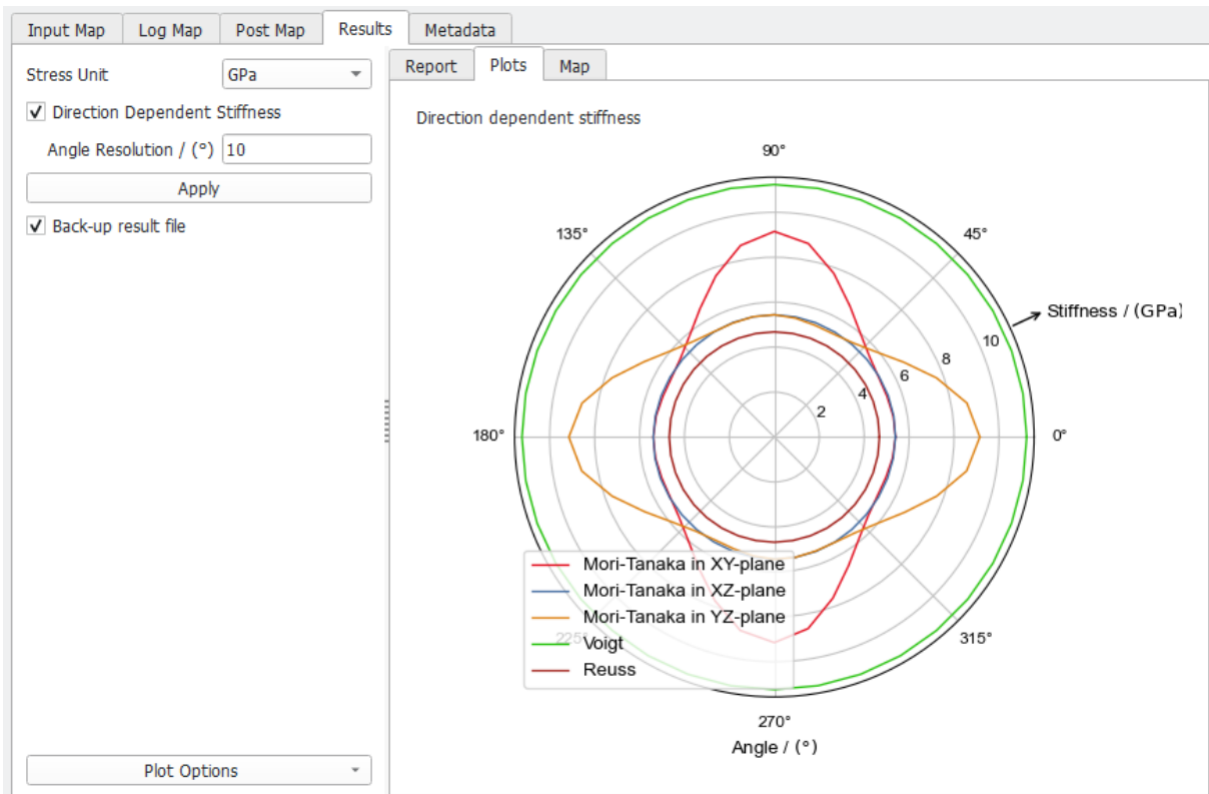
Direction dependent stiffness for Voigt

Stiffness in the planes of the original coordinate system

Angle / (Deg)	Young's modulus E in XY-Plane / (GPa)	Young's modulus E in XZ-Plane / (GPa)	Young's modulus E in YZ-Plane / (GPa)
0	11.2375	11.2375	11.2375
10	11.2375	11.2375	11.2375
20	11.2375	11.2375	11.2375
30	11.2375	11.2375	11.2375
40	11.2375	11.2375	11.2375
50	11.2375	11.2375	11.2375

- The **Plots** subtab shows a plot of the direction dependent Mori-Tanaka stiffness in the X-, Y- and Z-planes as well as the **Voigt** (upper bound) and the **Reuss** (lower bound) approximations. Right-clicking on the plot view allows generating an image or exporting the values as .txt

file.



- The **Map** subtab shows a structured list of all result values.

Input Map

Log Map

Post Map

Results

Metadata

Stress Unit

GPa

☒ Direction Dependent Stiffness

Angle Resolution / (°) 10

Apply

☒ Back-up result file

Plot Options

Report

Plots

Map

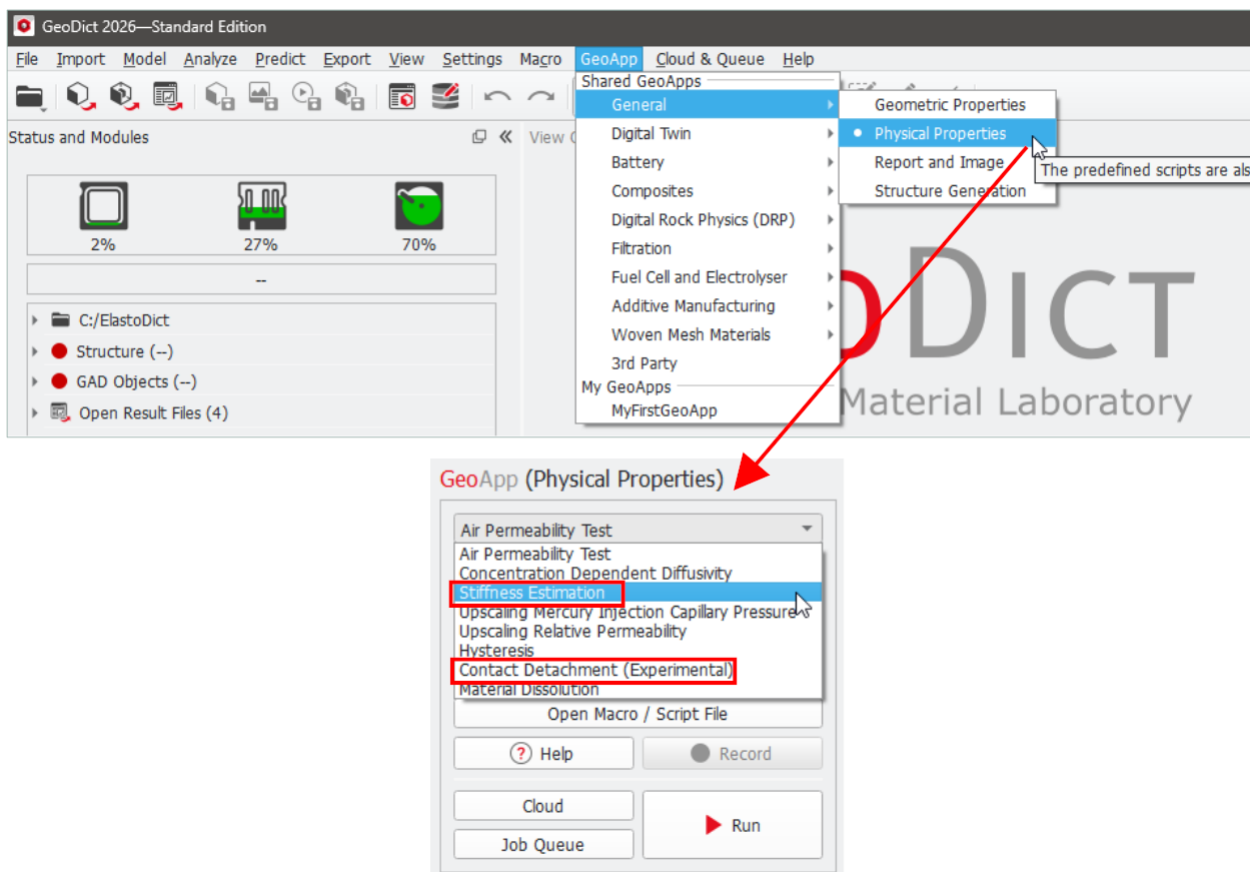
Key	Unit	Value
StructureFile		Structure.gdt
Runtime	s	0.407
PhysicalVisualization		
FileSuffixDAS		
ShowFieldInMaterialID		true, true, true, true, true, true, true, tr...
Information		
Voigt		
Stiffness	GPa	14.797, 6.09825, 6.09825, 0, 0, 0, 6.09...
Compliance	1/GPa	0.088988, -0.0259711, -0.0259711, 0, 0...
Isotropic		
OriginalSystem		
PMA		
DirectionStiffness		
Reuss		
Stiffness	GPa	8.02606, 4.59466, 4.59466, 0, 0, 0, 4.5...
Compliance	1/GPa	0.213647, -0.0777797, -0.0777797, 0, 0...
Isotropic		
OriginalSystem		
PMA		
DirectionStiffness		
MoriTanaka		
Stiffness	GPa	8.4026, 4.68959, 4.65842, -0.00857623,...
Compliance	1/GPa	0.185481, -0.0394193, -0.0802073, -0.0...
Isotropic		
OriginalSystem		
PMA		
DirectionStiffness		

1.7 Mechanics GeoApps

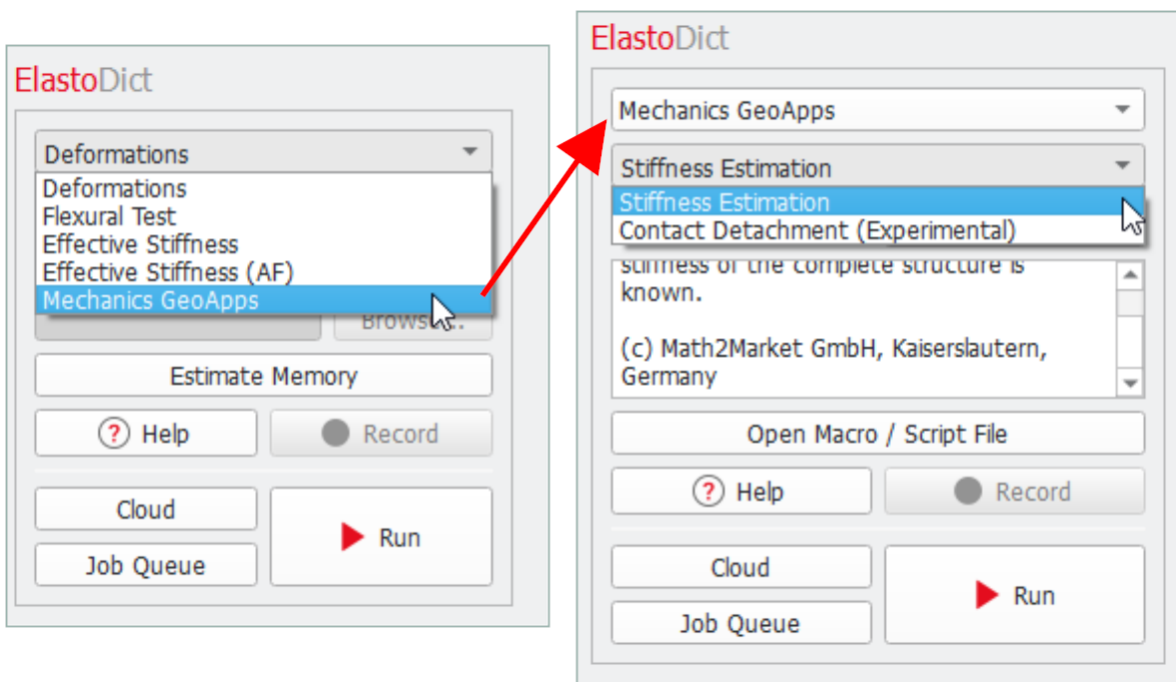
The GeoApp dropdown menu in GeoDict gives access to several pre-defined macros for workflows that are often requested.

There are two possibilities to access the GeoApps for ElastoDict.

One way is to choose **GeoApp** → **Modules** → **ElastoDict** in the GeoDict main GUI. With this option, the ElastoDict section in the main GUI changes to **GeoApp (ElastoDict)**. You can choose one of the two available GeoApps from the dropdown menu.



Another possibility is to choose a **Mechanics GeoApps** directly as one of the options in the ElastoDict section of the main GeoDict GUI.



When **Mechanics GeoApps** is chosen in the module section, two representative calculations are available – **Contact Detachment** and **Stiffness Estimation**.

Choosing one of them allows the user to start a simulation much easier since only specific parameters need to be set. By clicking on **Edit...** a parameter dialog opens, which contains all relevant parameters and solver settings for each simulation in a simplified and straightforward way.

For more information on GeoApps in general, refer to the [GeoApps User Guide](#).

Topics of this section

- [Stiffness Estimation](#)¹⁶⁵
With this GeoApp, the stiffness of constituent material can be estimated, if the effective stiffness of the complete structure is known.
- [Contact Detachment](#)¹⁶⁸
With this GeoApp, it is possible to simulate detachment of contacts between objects in ElastoDict-Deformations simulations.

1.7.1 Stiffness Estimation

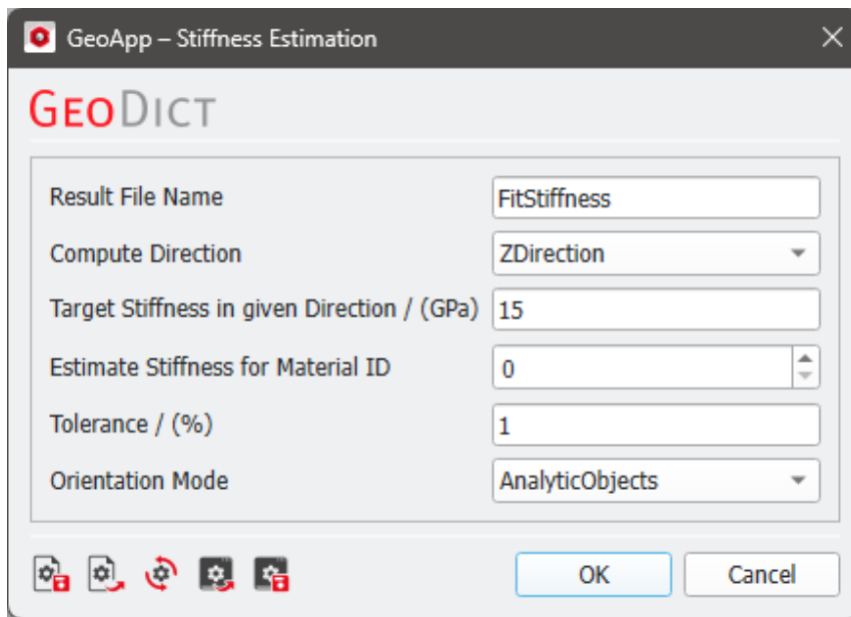
The **Stiffness Estimation** GeoApp estimates the stiffness for one constituent material, if the stiffness of the complete structure is known.



Modules needed to run this GeoApp:

ElastoDict-FeelMath

Click **Edit...** to open the **Stiffness Estimation Parameters** dialog .



The result file name can be entered in **Result File Name (*.gdr)**.

The **Compute Direction** (X, Y, or Z) for the stiffness and the **Orientation Mode** are defined in their drop-down menus. Refer to the [ElastoDict Deformations solver settings](#)^[67] for more details. In this example, the stiffness in Z-Direction of material with ID 01 is estimated.

The material stiffness of the structure in the given direction is set in **Target Stiffness in given Direction / (GPa)**.

Enter the ID number of the material for which the stiffness should be estimated in **Estimate Stiffness for Material ID**.

The stopping criteria is defined in **Tolerance / (%)**. If the relative error for the estimated structure stiffness is smaller than the specified tolerance, the estimation stops.

After all parameters are set, click **OK** to close the **Stiffness Estimation** parameters dialog and click **Run** to start the estimation.

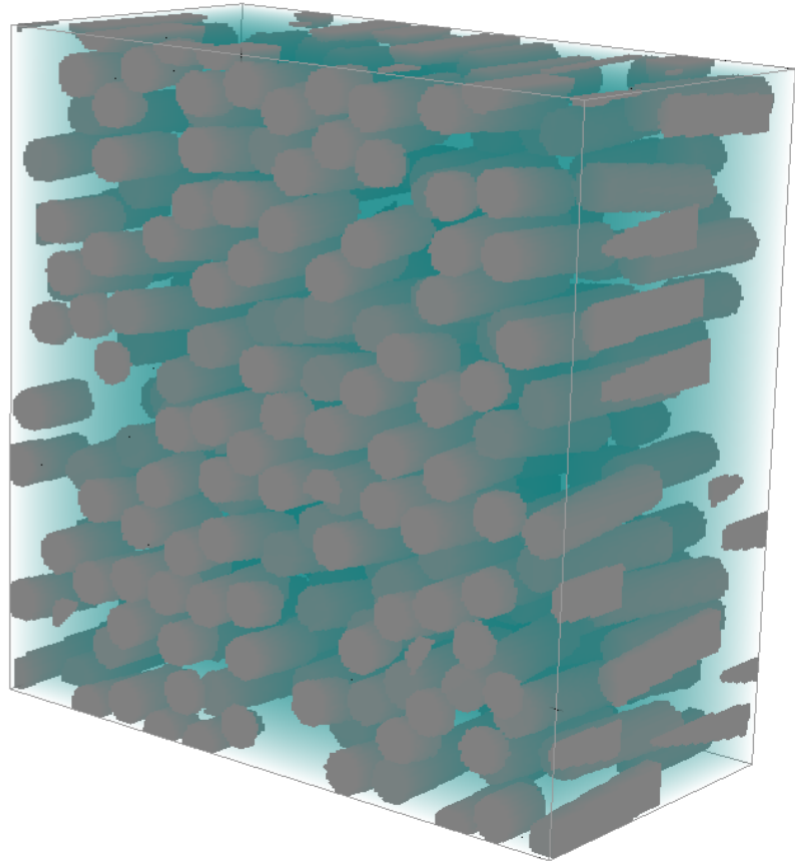
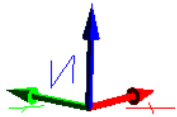
Results

The estimation result is documented among others in the **Results** tab of the **Result Viewer**. The table shown in the **Report** tab summarizes the GeoApp results. In addition to the user-defined target stiffness of the structure, the stiffness reached during the fitting process is shown.

Input Map	Log Map	Post Map	Results	Data Visualization	Metadata
Report	Map				
Estimate stiffness for Material00					
Parameter	Value	Unit			
Target Stiffness	7.2	GPa			
Reached Stiffness	7.159	GPa			
Stiffness Deviation	0.5675	%			
Estimated Stiffness for Material00	4.429	GPa			
Number of iterations	2				

In this example, the stiffness of the epoxy matrix (material ID00) of a composite structure is estimated. The table displayed in the **Results - Report** subtab shows that the estimated overall structure stiffness (**Reached Stiffness**) is 7.159 GPa with a defined **Target Stiffness** of 7.2 GPa. The **Estimated Stiffness for Material00**, which is the stiffness for the epoxy matrix, is 4.429 GPa. The fitting process in this example took 3 iterations.

Material Information:
ID 00: Epoxy (3501-6)
ID 01: Glass



1.7.2 Contact Detachment

The **Contact Detachment** GeoApp allows using the work in progress version of the contact detachment feature. With this feature, it is possible to simulate detaching of contacts between objects in [ElastoDict - Deformations](#)⁴³ simulations.



Modules needed to run this GeoApp:

ElastoDict-FeelMath

Click **Edit...** to open the **Contact Detachment** parameters dialog.

GeoApp – Contact Detachment

GEODICT

Experiment Type: Uniaxial

Result File Name (*.gdr): DetachContacts

Uniaxial Experiment: Tension

Tensile Strain / (%): 55

Tensile Steps: 55

Load Direction: Z

☒ Keep Gray Value Image

Resampling: Automatic

☒ Allow Contact Detachment

☒ Detach Initial Contacts

☒ Calculate Surface Normals

Contact Penalty Factor: 0.2

Shear Contact Penalty Factor: 0.02

Object Mode: UseGAD

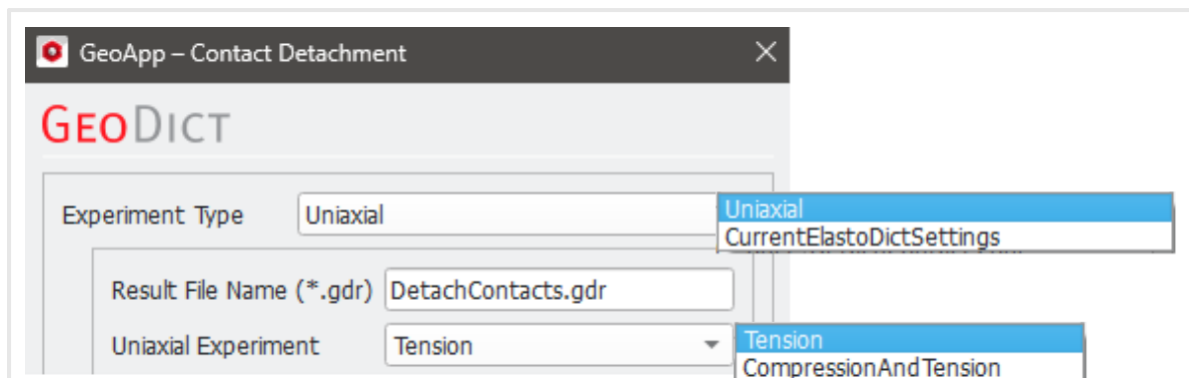
☐ Adaptive Time Steps

☒ Analyze Geometry

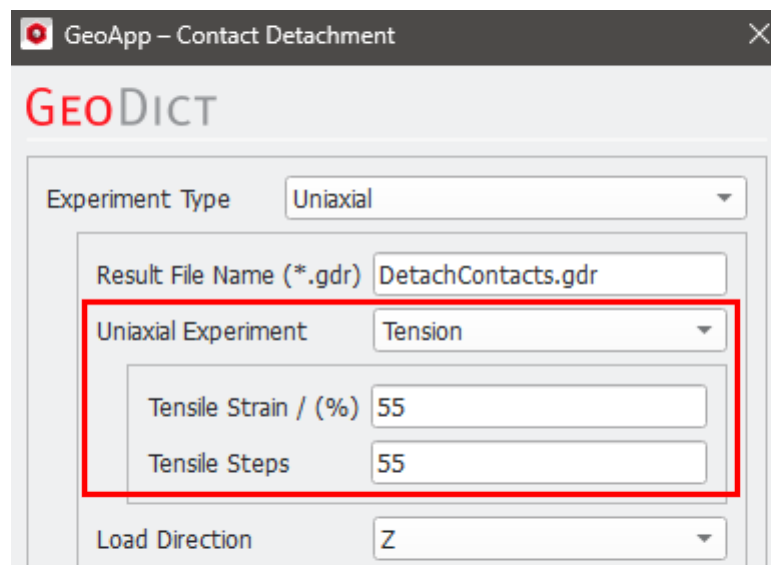
OK Cancel

✓ Experiment Type

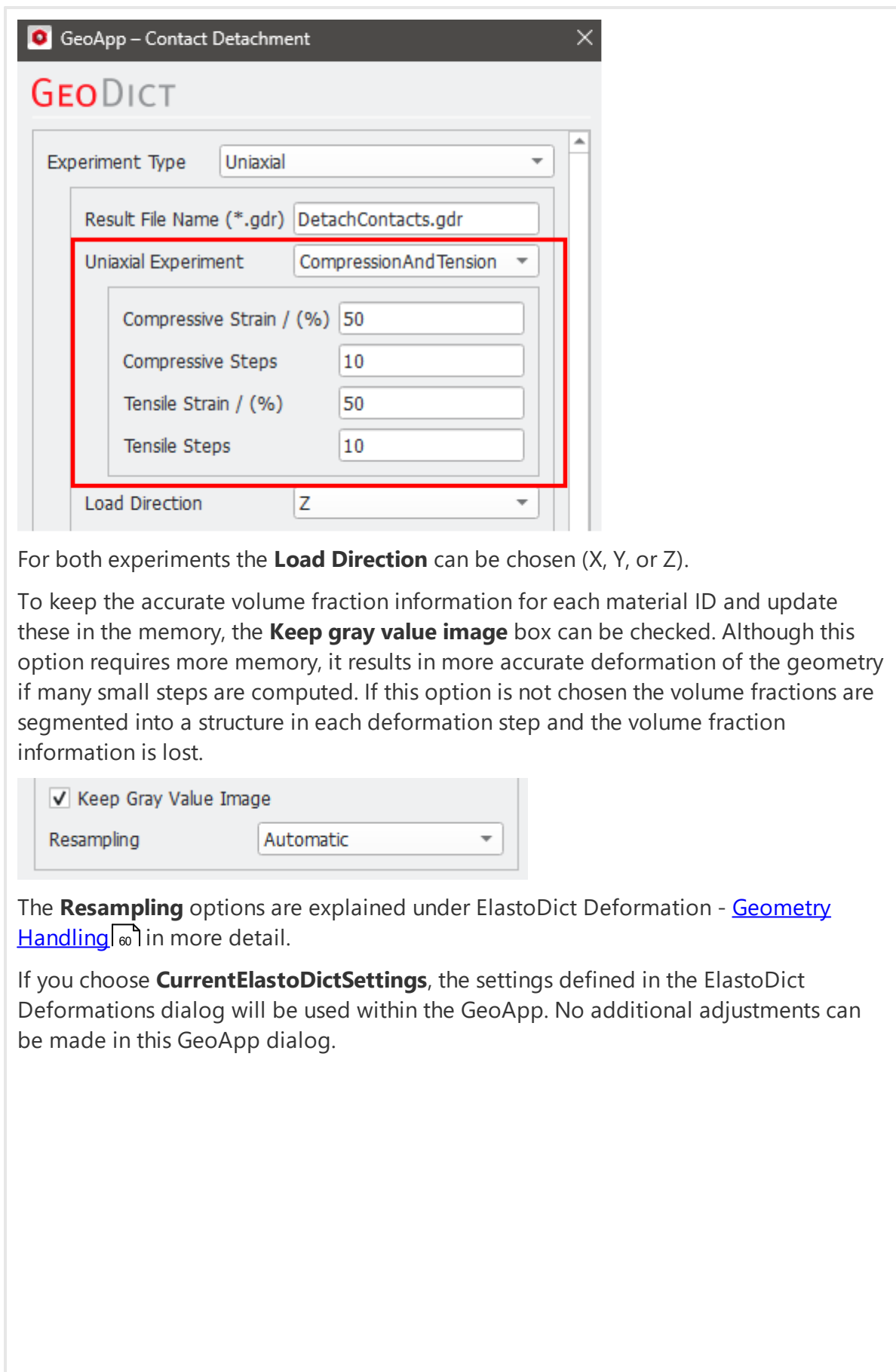
In the GeoApp it is either possible to directly set a uniaxial tensile experiment, a combined compressive/tensile experiment, or to use the current settings of ElastoDict Deformations.

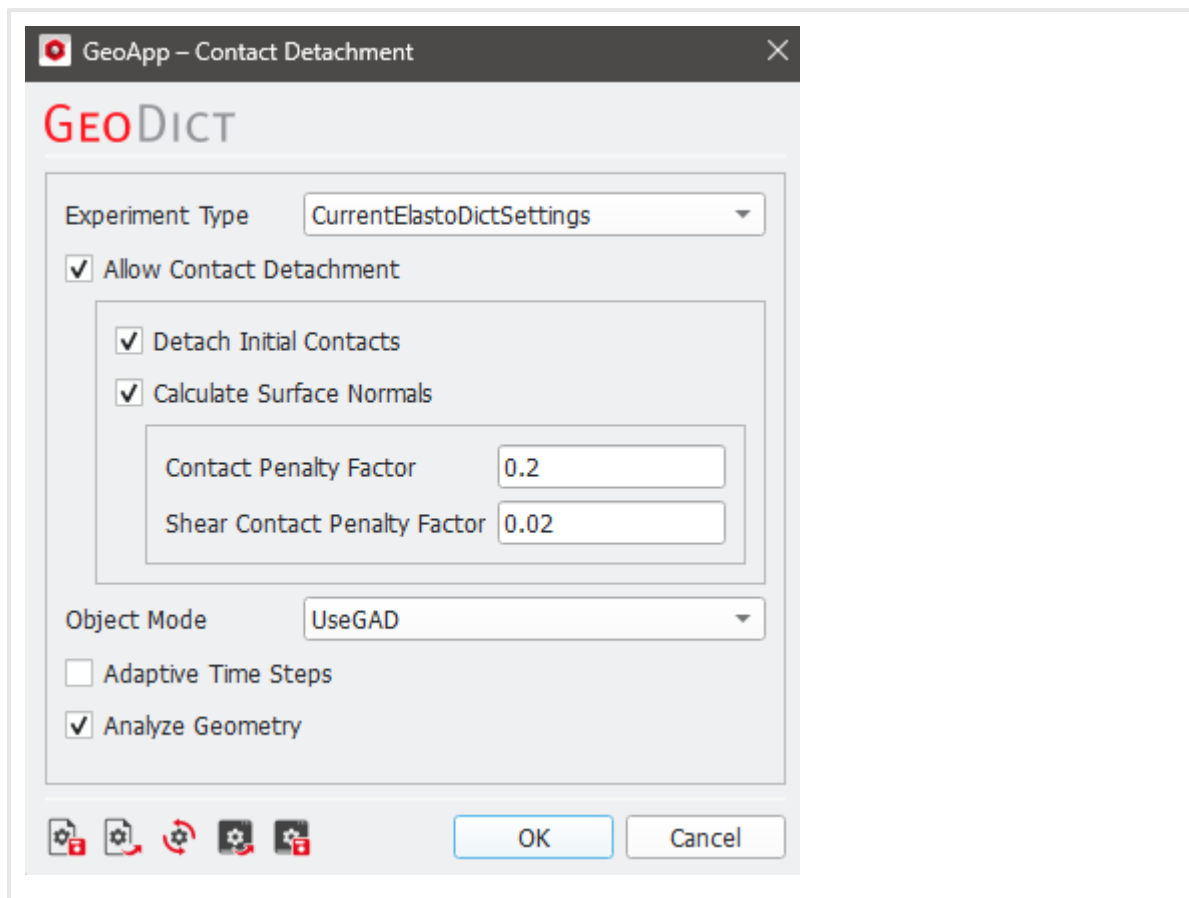


The tensile experiment is chosen by selecting **Tension** in the dropdown menu. The experiment is defined by the maximal tensile strain (in percentage with respect to the original geometry) and the number of steps to apply this strain.



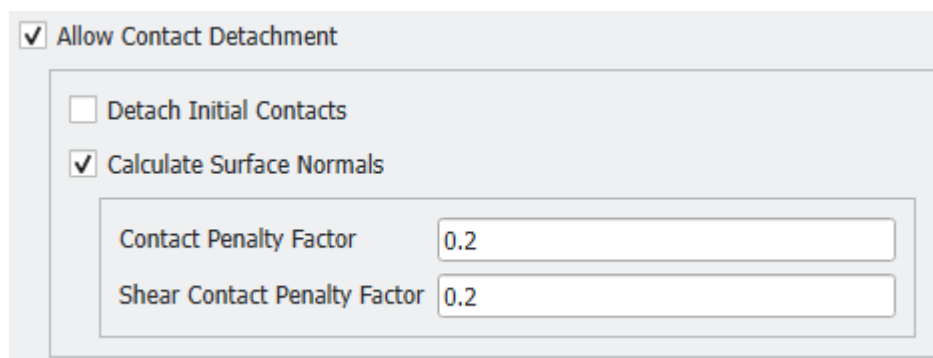
For the **Compression and Tension** experiment, a compression is applied to the geometry, followed by a tension. Again, both deformations are defined by the maximal strain (in percentage with respect to the original geometry) and the number of steps for each deformation.





✓ Allow Contact Detachment

If the option **Allow Contact Detachment** is checked, a detachment between different objects during the simulation run is possible.



Detachment is detected as soon as the stress changes from compression to tension in the contact area.

By choosing **Detach Initial Contacts**, a detachment is allowed for all objects, especially also for those, which are attached to each other initially. Otherwise, only objects can be detached, which were detached initially but get attached to each other during the simulation.

To **Calculate Surface Normals** between objects check the corresponding option. Only

the tensile part of the surface normal is used for the detachment algorithm.

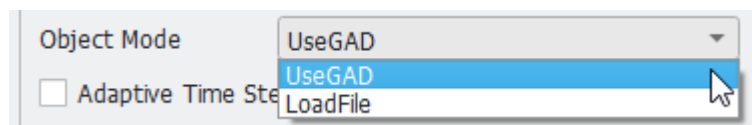
With the **Contact Penalty Factor** the detachment for contacts under normal load can be controlled. Choose a value between 0.0 (no resistance against detaching) and 1.0 (perfect contact). The higher the penalty factor, the later the detachment. Very small values can lead to convergence problems. For most cases the default value of 0.2 works well.

When **Calculate Surface Normals** is checked you can also define the **Shear Contact Penalty Factor**. With this parameter the detachment under shear load can be controlled. Choose a value between 0.0 (no resistance against detaching) and 1.0 (perfect contact). The higher the penalty factor, the later the detachment. Very small values can lead to convergence problems. For most cases the default value of 0.2 works well.

✓ Object Mode

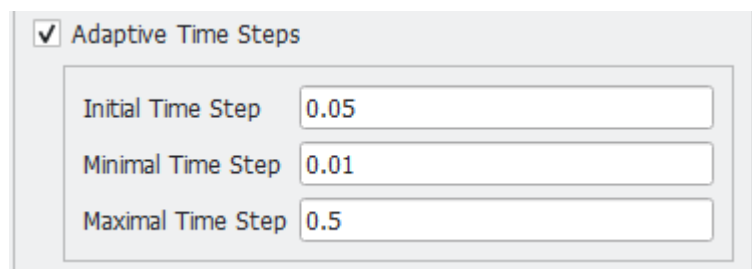
This simulation requires information about the objects in the structure. This information can be provided in two ways:

1. As GAD objects (select **Object Mode - UseGAD**)
2. Via a G32 file (select **Object Mode - LoadFile**)



✓ Adaptive Time Steps

Since GeoDict 2026 it is possible to run the simulation with **Adaptive Time Steps**.



Define the **Initial Time Step**, with which the solver starts the simulation. The number of iterations from the first step is kept as reference value. For all subsequent simulation steps the number of iterations is compared to this reference number of iterations: if it is much smaller, the time step size is increased. If it is much larger, the time step is discarded and restarted with a smaller time step size. As soon as the minimal time step size is reached, the solver finishes the step and proceeds with the next step.

The time step size is always kept within the predefined bounds: it will never exceed the predefined **Maximal Time Step** and never fall below the **Minimal Time Step**.

✓ Analyze Geometry

When the **Analyze Geometry** box is checked, GeoDict verifies the existence of a solid load path before beginning the simulation. This ensures that the geometry is properly connected as is typically intended.

☒ Analyze Geometry

In special cases, you can disable this check by unchecking **Analyze Geometry**. You may want to do this if you are using the same structure again or if you have already performed this check and know that a path exists.

Results

Click **OK** to close the **Contact Detachment Parameters** dialog. Then click **Run** to start the **Contact Detachment** simulation. The results are stored in the result file previously entered under the **Result File Name (*.gdr)** field of the **Contact Detachment Parameters** dialog.

After a successful run, the **Result Viewer** opens automatically showing the **Results - Report** subtab.

Mon Mar 2 2026 (2026 Build 94575) .../ElastoDict/DetachContacts.gdr

Domain Size: 64 x 64 x 64 Voxel Length: 250 nm Load Structure

Input Map Log Map Post Map Results Strain/Stress Visualization Create Videos Metadata

Report Plots Map

Coordinate system
The coordinate system of the experiment is aligned with the coordinate axis in GeoDict.

Average strain

Time / (s)	Temp. Change / (°C)	XX / (%)	YY / (%)	ZZ / (%)	YZ / (%)	XZ / (%)	XY / (%)	VonMises / (%)	Hydrostatic / (%)
0	0	0	0	0	0	0	0	0	0
1	0	-0.080948	-0.080948	1	-0.0044244	-0.0044244	0.0012284	1.081	0.279368
2	0	-0.15771	-0.15771	2	-0.010582	-0.010582	0.0024286	2.15787	0.561525
3	0	-0.2301	-0.2301	3	-0.021218	-0.021218	0.0033938	3.23053	0.846597
4	0	-0.29912	-0.29912	4	-0.029019	-0.029019	0.004353	4.29971	1.13392
5	0	-0.36364	-0.36364	5	-0.037652	-0.037652	0.0056163	5.36445	1.42424
6	0	-0.42196	-0.42196	6	-0.052677	-0.052677	0.0067691	6.42326	1.7187
7	0	-0.47675	-0.47675	7	-0.062675	-0.062675	0.0074965	7.47833	2.0155
8	0	-0.52649	-0.52649	8	-0.0762	-0.0762	0.0078526	8.52855	2.31567

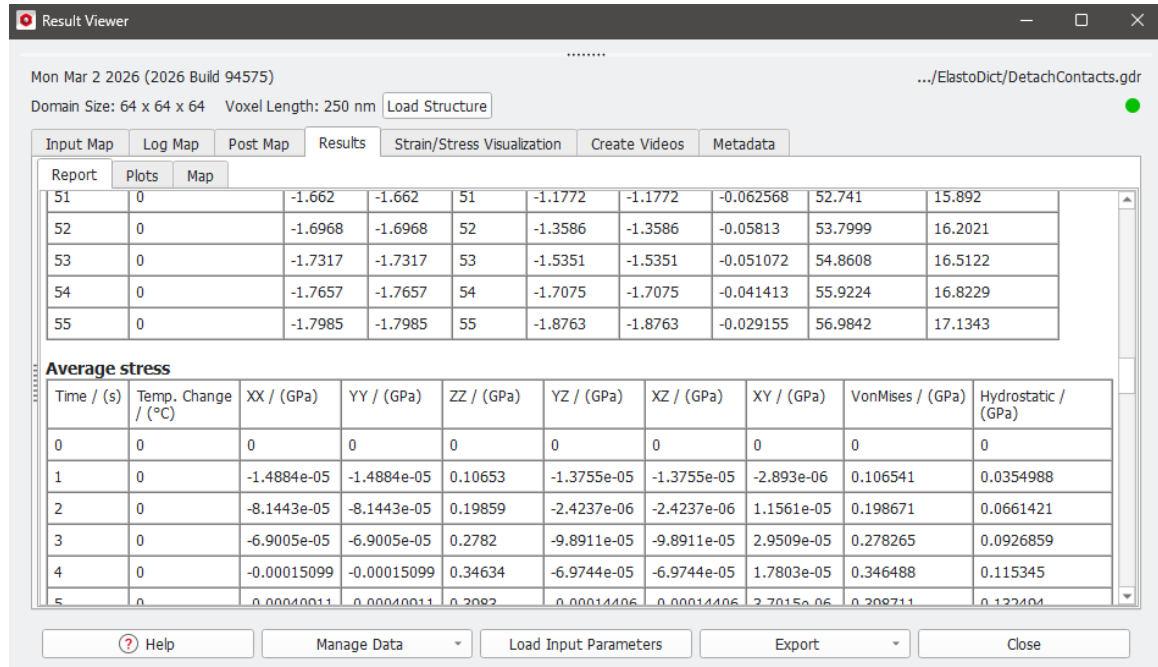
? Help Manage Data Load Input Parameters Export Close

In this example, a **Tension** experiment is simulated in which two objects are pulled in the Z-direction, resulting in object detachment.

✓ Report

The strain that the objects undergo is documented in the first table of the Report tab in

the Result Viewer. Since the **Uniaxial Experiment - Tensile** with a maximum strain of 55% was defined in the **Macroscopic Load** tab, the average strain values for the ZZ column start at 0% and end at 55% after 55 steps.



Mon Mar 2 2026 (2026 Build 94575) .../ElastoDict/DetachContacts.gdr

Domain Size: 64 x 64 x 64 Voxel Length: 250 nm Load Structure

Input Map Log Map Post Map Results Strain/Stress Visualization Create Videos Metadata

Report Plots Map

51	0	-1.662	-1.662	51	-1.1772	-1.1772	-0.062568	52.741	15.892
52	0	-1.6968	-1.6968	52	-1.3586	-1.3586	-0.05813	53.7999	16.2021
53	0	-1.7317	-1.7317	53	-1.5351	-1.5351	-0.051072	54.8608	16.5122
54	0	-1.7657	-1.7657	54	-1.7075	-1.7075	-0.041413	55.9224	16.8229
55	0	-1.7985	-1.7985	55	-1.8763	-1.8763	-0.029155	56.9842	17.1343

Average stress

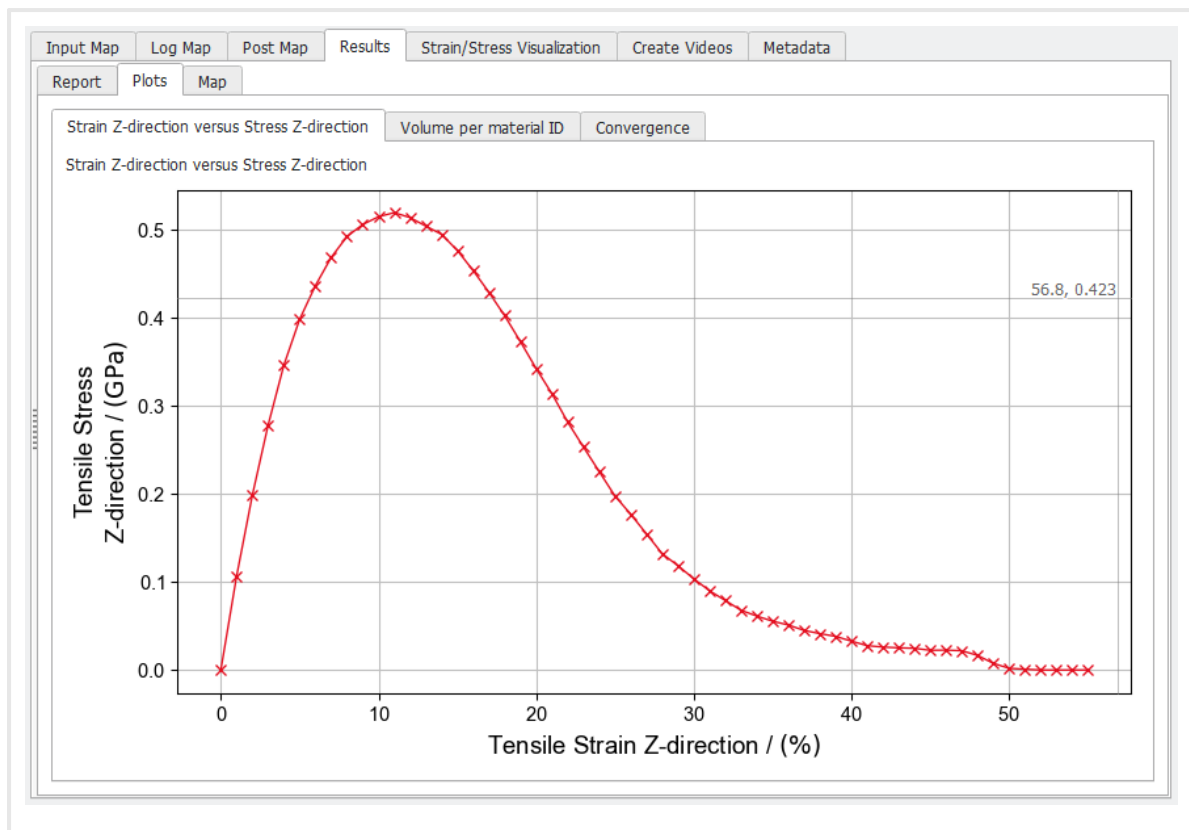
Time / (s)	Temp. Change / (°C)	XX / (GPa)	YY / (GPa)	ZZ / (GPa)	YZ / (GPa)	XZ / (GPa)	XY / (GPa)	VonMises / (GPa)	Hydrostatic / (GPa)
0	0	0	0	0	0	0	0	0	0
1	0	-1.4884e-05	-1.4884e-05	0.10653	-1.3755e-05	-1.3755e-05	-2.893e-06	0.106541	0.0354988
2	0	-8.1443e-05	-8.1443e-05	0.19859	-2.4237e-06	-2.4237e-06	1.1561e-05	0.198671	0.0661421
3	0	-6.9005e-05	-6.9005e-05	0.2782	-9.8911e-05	-9.8911e-05	2.9509e-05	0.278265	0.0926859
4	0	-0.00015099	-0.00015099	0.34634	-6.9744e-05	-6.9744e-05	1.7803e-05	0.346488	0.115345
5	0	0.00040011	0.00040011	0.2082	0.00014406	0.00014406	2.7015e-06	0.208711	0.123404

Help Manage Data Load Input Parameters Export Close

In the table below, the **Average stress** is displayed.

✓ Plots

The figure in the **Plots** subtab of the **Results** tab plots the tensile stress over tensile strain, which is a graphical representation of the table content in the [Report](#) tab. Tensile stress reaches its maximum at 11% strain and becomes zero at approximately 51% strain. At this point, the two objects begin to separate.



✓ Strain / Stress Visualization

In the **Strain/Stress Visualization** tab, the different result files can be loaded according to the description for ElastoDict [Deformations](#)^[74].



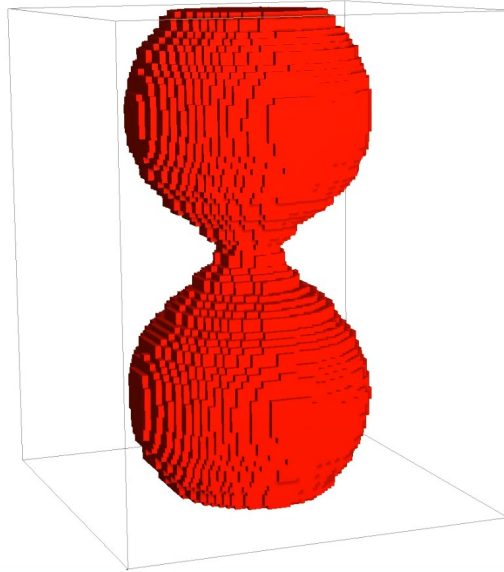
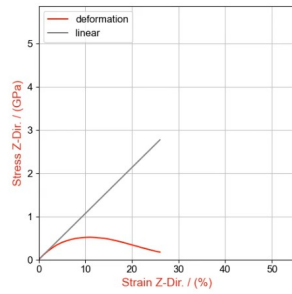
Know how! For this type of experiment, **Box Structure Rendering** might be a better choice since it allows for more precise observation of detachment, as seen in the pictures below.

✓ Create Videos

The detachment process can be visualized using the options found under the [Create Videos](#)^[80] tab.

ELASTODICT

GEO-DICT
The Digital Material Laboratory



MATH
2 MARKET

1.8 References

- G. Alfano, F. De Angelis and L. Rosati, *General solution procedures in elasto/viscoplasticity*, Computer Methods in Applied Mechanics and Engineering, **190(39)** (2001).
- J.Th. Browaeys, S. Chevrot, *Decomposition of the elastic tensor and geophysical applications*, Geophysical Journal International 159(2) (2004), <https://doi.org/10.1111/j.1365-246X.2004.02415.x>.
- P. Chadwick, M Vianello and S.C. Cowin, *A new proof that the number of linear elastic symmetries is eight*, Journal of Mechanics and Physics of Solids, 49 (2001)
- H. Grimm-Strele, M. Kabel, *Runtime optimization of a memory efficient CG solver for FFT-based homogenization: implementation details and scaling results for linear elasticity*, Computational Mechanics, Volume 64, Volume 5 (2019), <https://doi.org/10.1007/s00466-019-01713-3>
- H. Grimm-Strele, M. Kabel, *Fast Fourier transform based homogenization with mixed uniform boundary conditions*, International Journal for Numerical Methods in Engineering (2021), <https://doi.org/10.1002/nme.6830>
- ISO 14125:1998, *Fibre-reinforced plastic composites - Determination of flexural properties*, International Standard.
- M. Kabel, D. Merkert, M. Schneider, *Use of composite voxels in FFT-based homogenization*, Computer Methods in Applied Mechanics and Engineering, Volume 294 (2015), <http://dx.doi.org/10.1016/j.cma.2015.06.003>
- M. Kabel, S. Fliegner, M. Schneider, *Mixed boundary conditions for FFT-based homogenization at finite strains*, Computational Mechanics, Volume 57, Number 2 (2016), <http://dx.doi.org/10.1007/s00466-015-1227-1>
- M. Kabel, A. Fink, M. Schneider, *The composite voxel technique for inelastic problems*. Computer Methods in Applied Mechanics and Engineering, Volume 322 (2017), <https://doi.org/10.1016/j.cma.2017.04.025>
- M. Kabel, *Mixed strain/stress gradient loadings for FFT-based computational homogenization methods*, Computational Mechanics 70. (2022), <https://doi.org/10.1007/s00466-022-02168-9>
- J. Mazars, *Mechanical damage and fracture of concrete structures*, Advances in fracture research. (1981) 4
- J-C. Michel and P. Suquet, *A model-reduction approach in micromechanics of materials preserving the variational structure of constitutive relations*. Journal of the Mechanics and Physics of Solids, 90 (2016)
- H. Moulinec, P.Suquet, *A numerical method for computing the overall response of nonlinear composites with complex microstructure*, Computer Methods in Applied Mechanics and Engineering, Volume 157, Issues 1-2 (1998), [https://doi.org/10.1016/S0045-7825\(97\)00218-1](https://doi.org/10.1016/S0045-7825(97)00218-1)
- A.H. Nayfeh, *Wave propagation in layered anisotropic media with applications to composites*, Elsevier Science, North-Holland series in Applied Mathematics and Mechanics, Vol. 39 (1995)
- A. Norris, *Elastic Moduli Approximation*, J. Acoust. Soc. Am. Vol. 119, No 4, April 2006

V. Rutka, A. Wiegmann, *Explicit jump immersed interface method for virtual material design of the effective elastic moduli of composite materials*, Numerical Algorithms 43 (2006),
<https://doi.org/10.1007/s11075-007-9063-9>

V. Rutka, A. Wiegmann, H. Andrä, *EJIM for calculation of effective elastic moduli in 3D linear elasticity*, Berichte des Fraunhofer ITWM, Nr. 93 (2006)

M. Schneider, F. Ospald, M. Kabel, *Computational homogenization of elasticity on a staggered grid*. Int. Journal for Numerical Methods in Engineering, Volume 105, Issue 9 (2016),
<https://doi.org/10.1002/nme.5008>

- M -

matrix damage 19

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



Math2Market GmbH

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