Module 01: Material Designer Overview

Release 2020 R1

Agenda

- Overview
- Theory
- Basic Workflow in Workbench
- Features
- RVE Types
- Material Assignment
- Mesh
- Analysis Settings
- Solve Process
- Results Review
- Exports
- Example

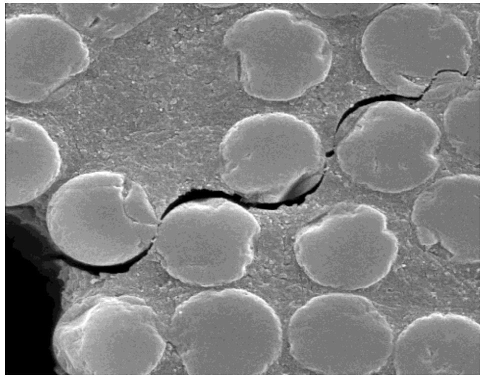
Overview

- Material Designer is an application which facilitates the simulation of models where different length scales are involved:

Rotor Diameter 82m
Hub Height up to 100m

5.000.000 : 1

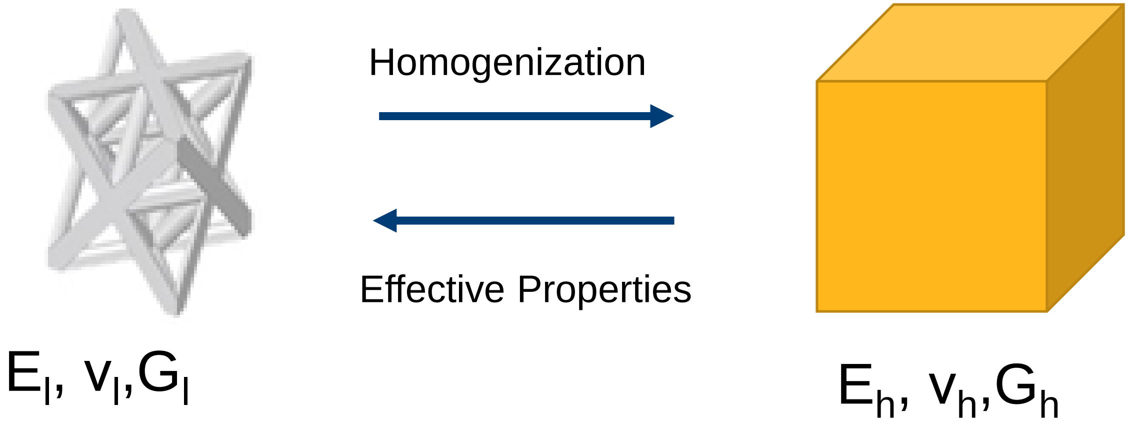
Scale relation geometry to fiber diameter



- It is challenging, with usual finite element approach to simulate the entire windmill, taking into account the microstructure of its materials. The number of elements required would be astronomically large, and computing time really long.

Overview

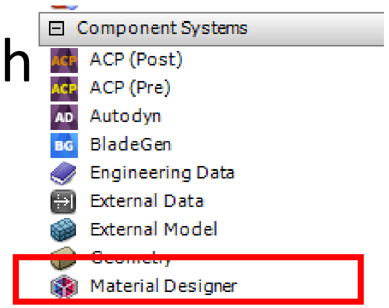
- **Material Designer** is an application that enables you to **model and analyze** microstructures and derive **homogenized material properties** for a Representative Volume Element (RVE).
- A Representative Volume Element (RVE) is the **smallest volume** over which a measurement can be made that will yield a **value representative** of the whole structure.



- **Assumption:** The microscale structures must be significantly smaller than the macroscale.

Overview

- Material Designer can be used for composite simulations as well as for lattice structures simulations (additive manufacturing).
- What are the advantages?
  - Conventional approaches make the numerical simulation **very expensive**.
  - Experimental testing is often used to determine the exact material properties of the composite materials, which can be expensive and time consuming. Knowing the structure of the composite materials and the properties of the base materials is sufficient to use Material Designer to know the global material properties.
  - A **more accurate approach** as it is based on the finite element analysis of a micro-structure.
  - Easily create complex RVEs & test for **parameter** dependence
  - Material Designer computes constant and variable homogenized material properties of lattice structures
- Material Designer is a component system accessible via the ANSYS Workbench project page.
- It uses SpaceClaim interface



Theory

- Materials Designer assumes that the material that is considered has a representative volume element (**RVE**)

- For **periodic materials**, the RVE can easily be identified as one unit cell. In a periodic materials, the unit cell repeats itself in all three coordinate directions.

- For **non-periodic materials**, identifying the size of a RVE is more difficult. One approach consists of choosing an initial RVE and doing the macroscopic calculations with it. Then, increase the size of the RVE and see if the microscopic results have changed significantly. If this is the case, the initial RVE was not large enough. If microscopic results remain fixed, the initial RVE can be used.

Theory

Examples of modeling assumptions made for some RVEs

- Lattice structures:
  - Are made of one isotropic linear-elastic material
  - Are periodic
  - Consist of cylindric trusses

- Unidirectional composites:
  - Consist of an isotropic linear-elastic matrix material and of an isotropic or transversely isotropic linear-elastic material
  - Fibers are infinite, cylindrical and have the same fiber diameter
  - Fibers are perfectly aligned with the X-axis of the RVE
  - Fibers form a regular pattern
  - Bonding between the fibers and matrix material is perfect

Theory

- Computation of Material Properties in the case of Orthotropic Linear-Elastic Material Properties

Constitutive relationship:

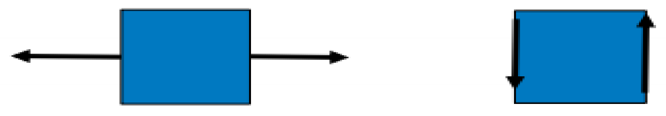
$$\begin{pmatrix}\sigma_x\\ \sigma_y\\ \sigma_z\\ \sigma_{xy}\\ \sigma_{yz}\\ \sigma_{xz}\end{pmatrix}=\begin{pmatrix}D_{11} & D_{12} & D_{13} & & & \\ D_{21} & D_{22} & D_{23} & & & \\ D_{31} & D_{32} & D_{33} & & & \\ & & & D_{44} & & \\ & & & & D_{55} & \\ & & & & & D_{66}\end{pmatrix}\begin{pmatrix}\epsilon_x\\ \epsilon_y\\ \epsilon_z\\ \gamma_{xy}\\ \gamma_{yz}\\ \gamma_{xz}\end{pmatrix}$$

Strain in each direction is fixed to 0.01 and the first column of stiffness matrix is obtained. Representative Volume Element occupies the volume given by the dimensions $[0, L_X] \times [0, L_Y] \times [0, L_Z]$

$$\begin{pmatrix}D_{11}\\ D_{21}\\ D_{31}\\ 0\\ 0\\ 0\end{pmatrix}=\frac{1}{0.01}\begin{pmatrix}\sigma_x\\ \sigma_y\\ \sigma_z\\ \sigma_{xy}\\ \sigma_{yz}\\ \sigma_{xz}\end{pmatrix}$$

+

Load Cases + Periodic/Non-periodic Boundary Conditions & Constraining Rigid Body Motion



+

Forces on surfaces at L_x , L_y , L_z are then used to calculate the macroscopic stresses

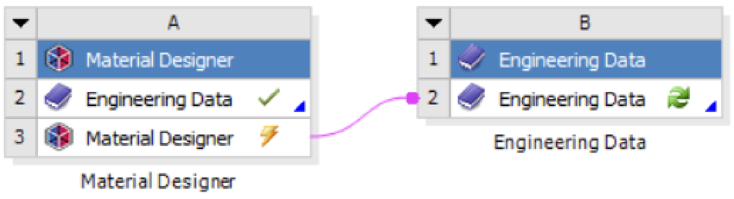
=

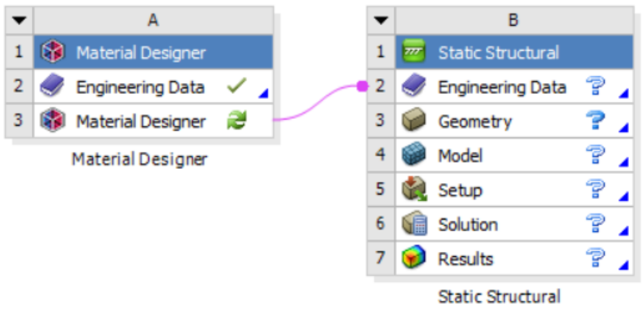
$$[C]=[D]^{-1}$$

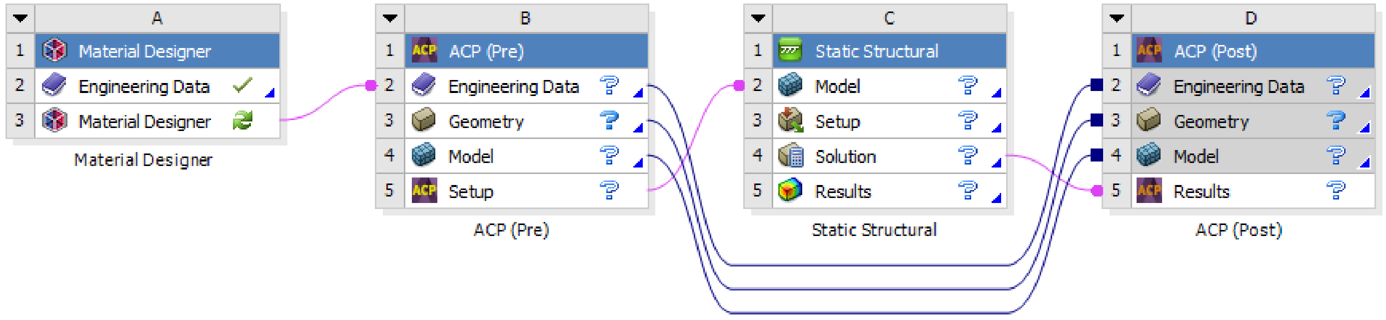
$$[C]=\begin{pmatrix}\frac{1}{E_x} & \frac{-\nu_{yx}}{E_y} & \frac{-\nu_{zx}}{E_z} & & & \\ \frac{-\nu_{xy}}{E_x} & \frac{1}{E_y} & \frac{-\nu_{zy}}{E_z} & & & \\ \frac{-\nu_{xz}}{E_x} & \frac{-\nu_{yz}}{E_y} & \frac{1}{E_z} & & & \\ & & & \frac{1}{G_{xy}} & & \\ & & & & \frac{1}{G_{yz}} & \\ & & & & & \frac{1}{G_{xz}}\end{pmatrix}$$

Basic Workflow in Workbench

- Material Designer component system can be linked to Engineering Data application to use the calculated homogenized material properties in downstream analyses:

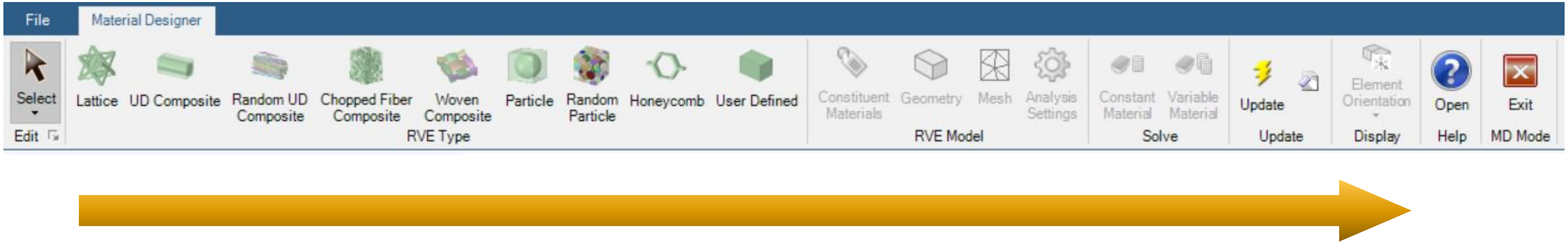






Features

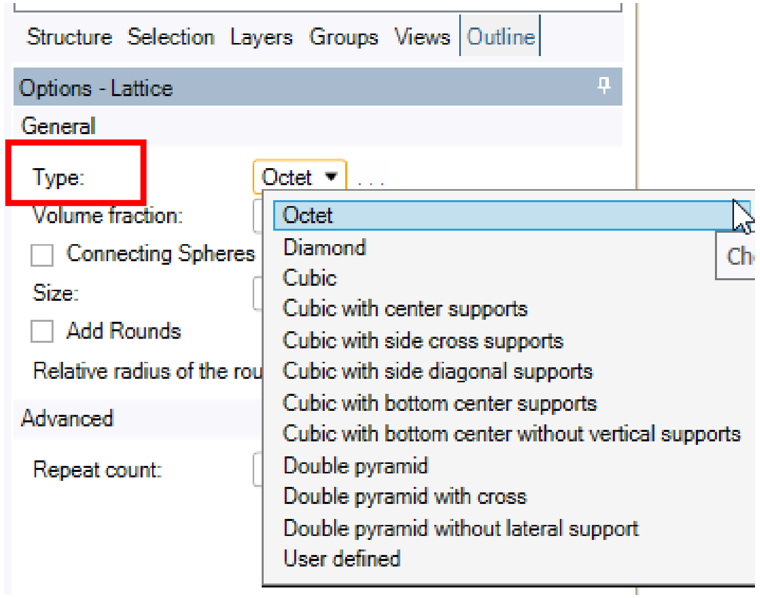
- The workflow inside Material Design consists of the following steps:
  - Create simplified geometry
  - Define material properties
  - Generate mesh
  - Expose RVE to several macro load cases and compute response
  - Extract homogenized material data



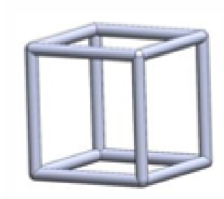
RVE Types

- Start by choosing a RVE type:

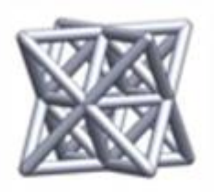
Lattice is a structure made of a repeated lattice pattern. The lattice pattern can be of multiple types. It needs to be chosen in the options:



Some Examples:

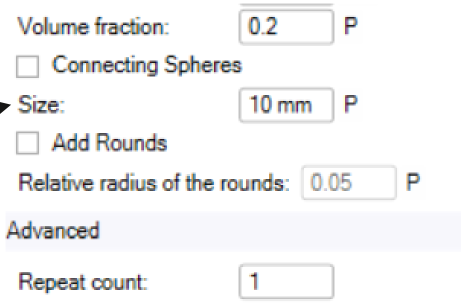

Cubic


Cubic with center supports


Octet

Properties need to be defined:

Control the size of the unit cell

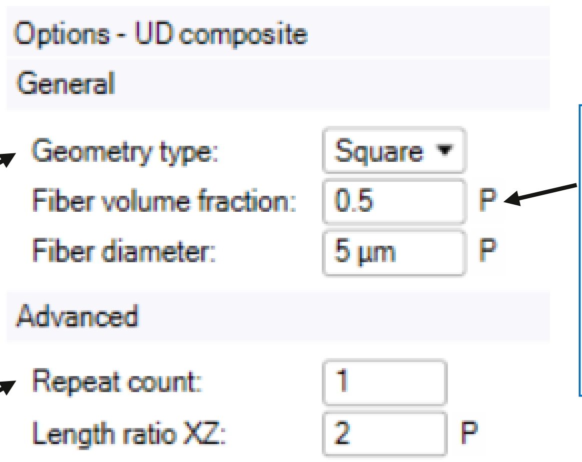


Ansys

RVE Types

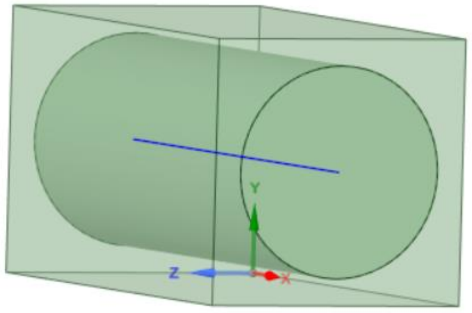
User Defined Composite: it consists of a composite made of fibers oriented in the same direction, surrounded by a matrix material

The options define the type of geometry as well as the fibers dimensions and proportion.

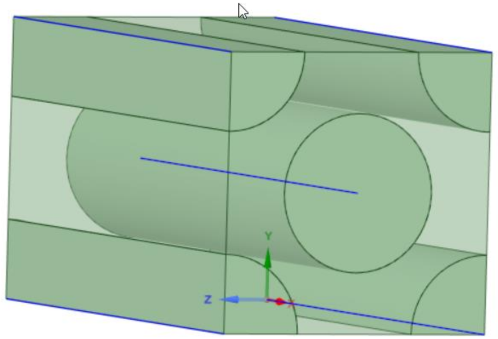


Fraction of space within the RVE occupied by fiber material

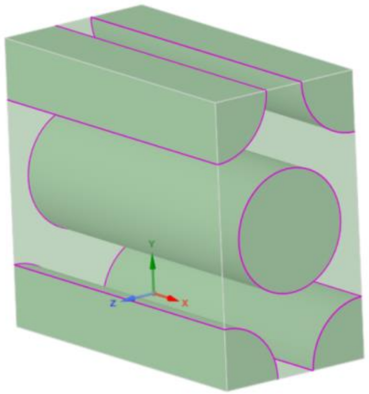
Control the number of repetitions of the pattern



Geometry type = square



Diamond

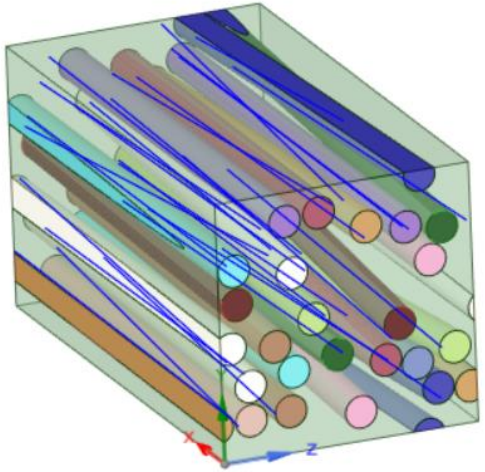


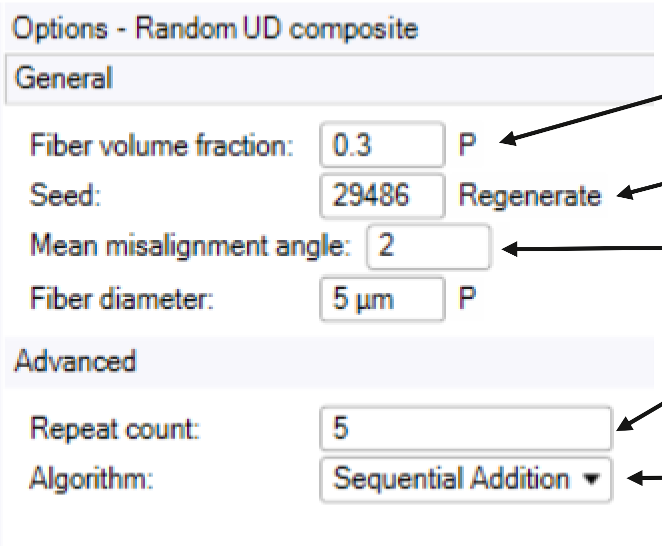
Hexagonal

Ansys

RVE Types

Random User Defined Composite is made of fibers with random variation in the orientation direction, surrounded by a matrix material





Fraction of space within the RVE occupied by fiber material

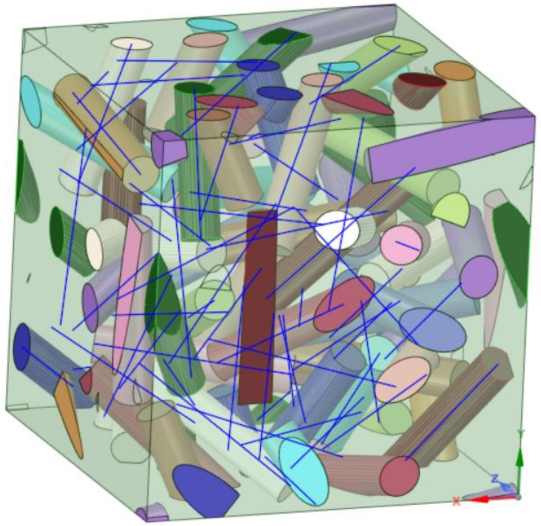
Seed number used to generate the fiber directions (initiate random algorithm)

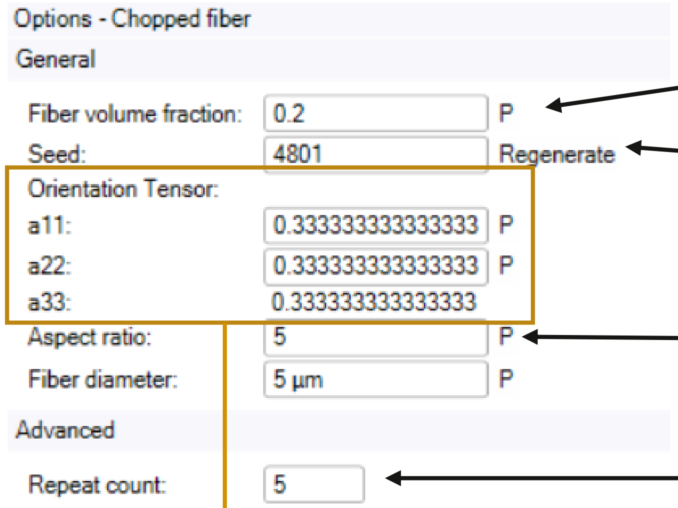
Misalignment in the x direction

Control the number of repetitions of the pattern

Sequential: fibers are added one after each other
OR
Perturbation: start from a regular pattern and perturb the fibers

Chopped Fiber Composite: short fibers arranged randomly in a matrix material.





Fraction of space within the RVE occupied by fiber material

Seed number used to generate the fiber directions (initiate random algorithm)

Ratio of length to the diameter of the fibers

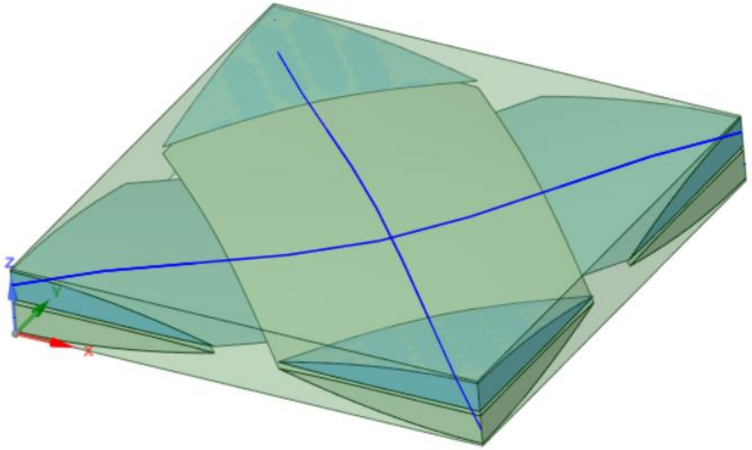
Control the number of repetitions of the pattern

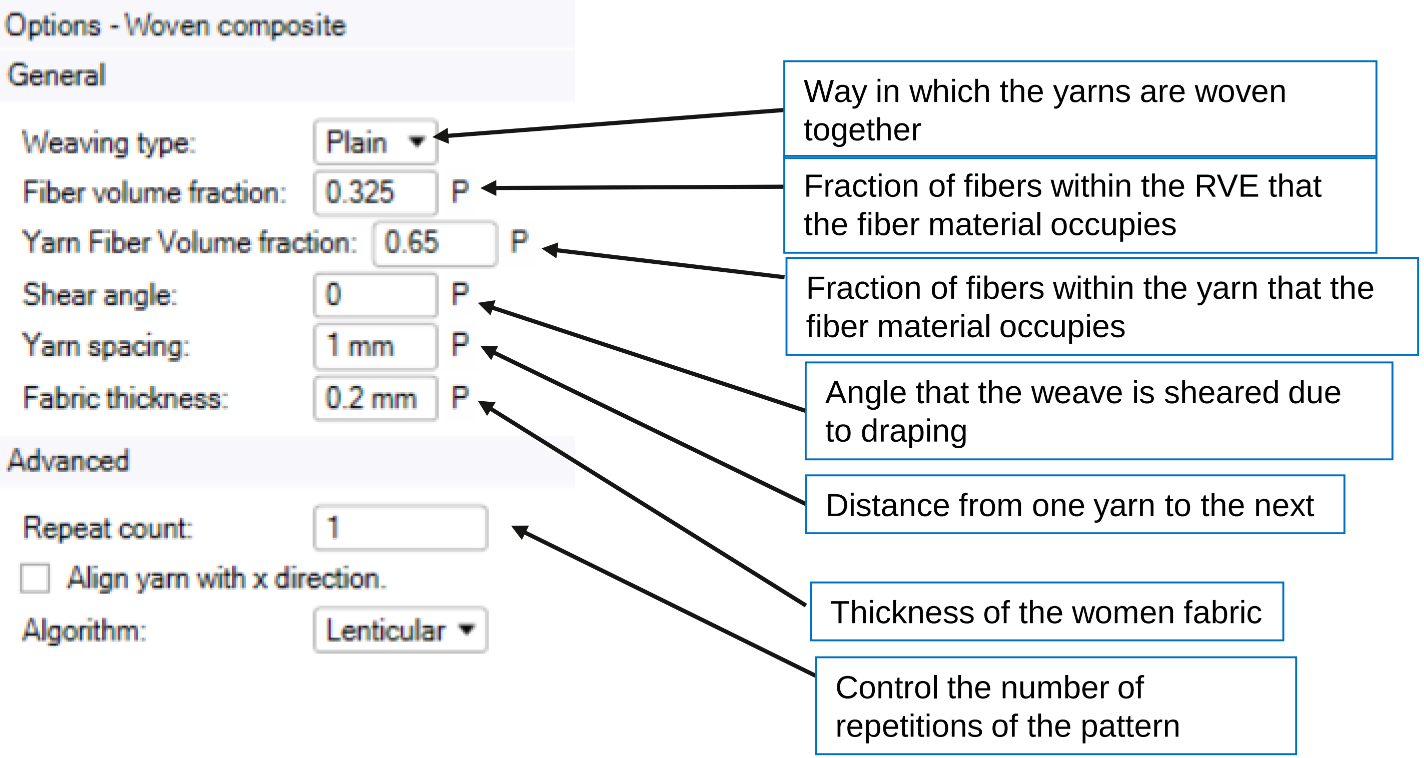
Orientation tensor: allows to specify how the fibers are aligned to the global direction.
This is a symmetric tensor with the sum of its diagonal terms equals 1.
a11 = 1; a22 and a33 are 0 gives: $A = \begin{pmatrix} 1 & 0 & 0 \\ 0 & 0 & 0 \\ 0 & 0 & 0 \end{pmatrix}$ all the fibers are in this case, aligned in X direction

$A = \begin{pmatrix} 0.2 & 0 & 0 \\ 0 & 0.8 & 0 \\ 0 & 0 & 0 \end{pmatrix}$ means that the fibers are oriented parallel to the XY plane

RVE Types

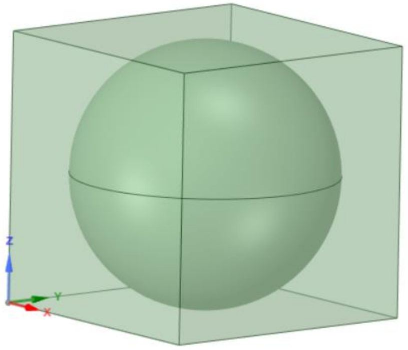
Woven Composite: material that have been woven together in two directions, surrounded by a matrix material. For the simulation, the fibers and the surrounding matrix material are grouped into a yarn material. The simulation is performed at the yarn level.



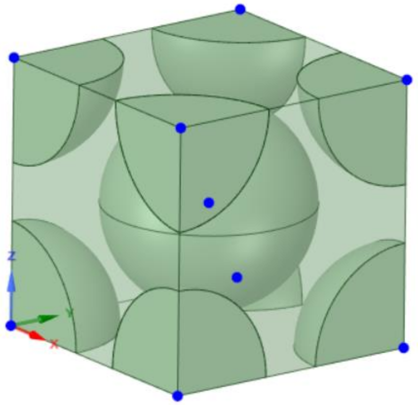


RVE Types

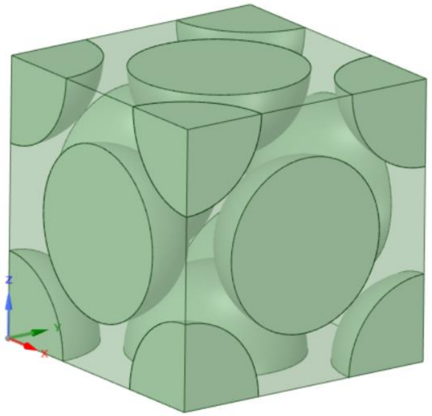
Particle: composites consist of spherical particles regularly arranged in a matrix material



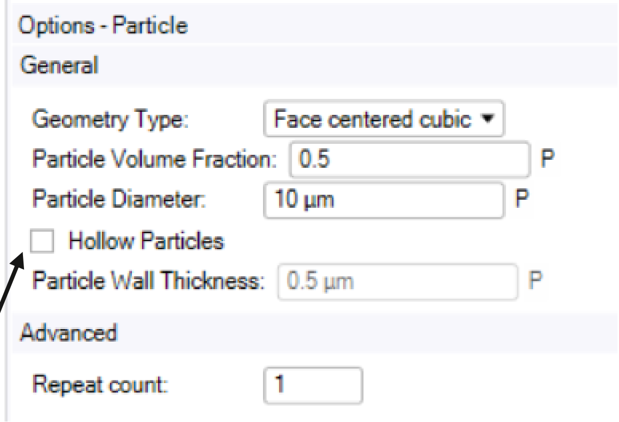
Simple cubic



Body centered cubic



Face centered cubic

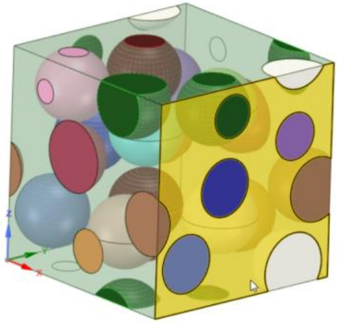


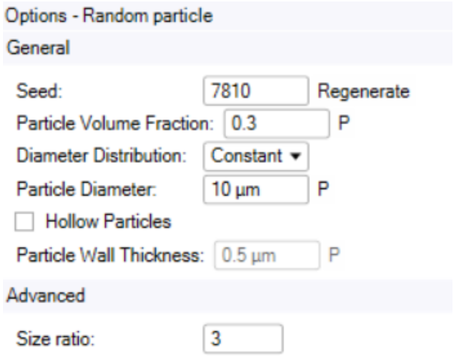
Hollow particles can de defined

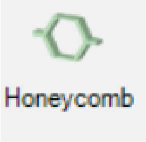
Ansys

RVE Types

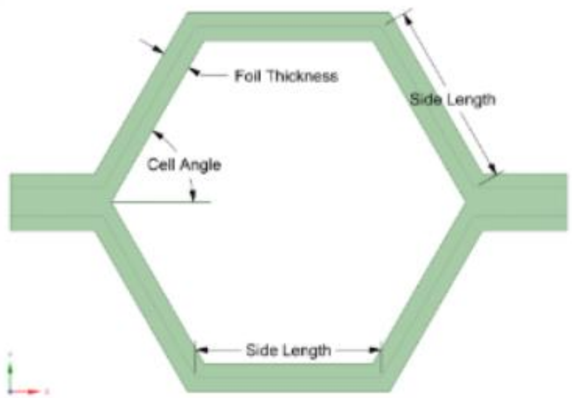
Random Particle: composites are made of spherical particles arranged randomly in a matrix material. Both size and position of the particles can be randomly defined.

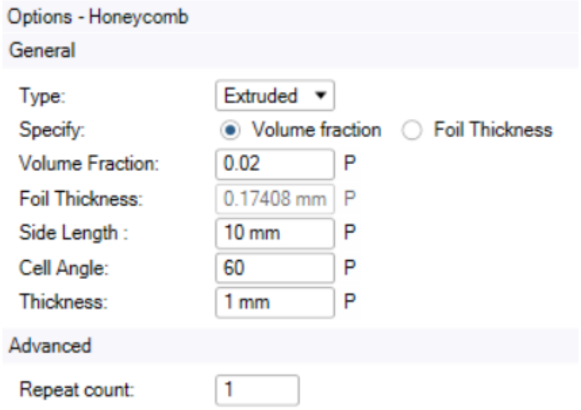






Honeycomb: the structure consists of cells with hexagonal shape repeated in two directions



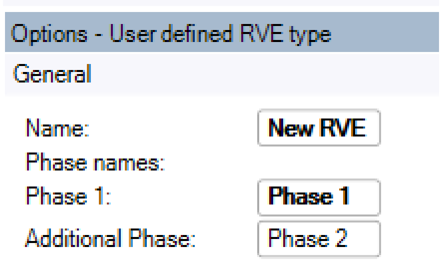


Ansys

RVE Types

User Defined RVE: consists of one or more phases. Each phase consists of one or more solid bodies.

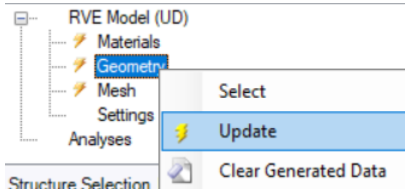
- You need to define phase names



- Import or create the RVE geometry into SpaceClaim and assign the phases to the bodies of the geometry

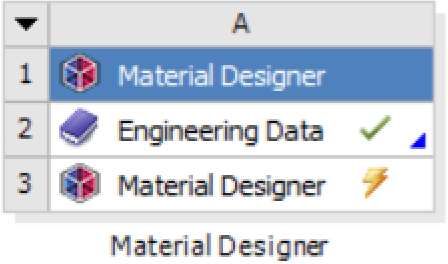
Note for all RVE types: Once you have select a RVE type and want to change it, you can use the Change Button and select a new RVE type

Right Click on Geometry > Update to see the resulting RVE geometry



Material Assignment

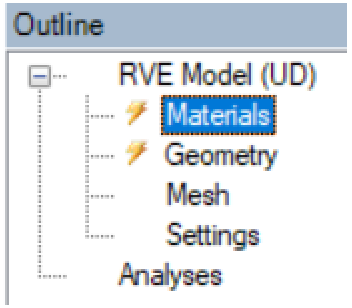
- Composite materials are made of one or more materials. To assign these materials in Material Designer, you should first define the materials in the Engineering Data application

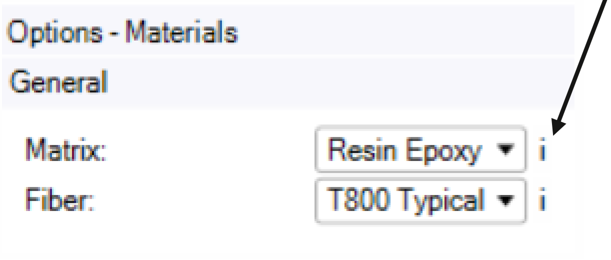


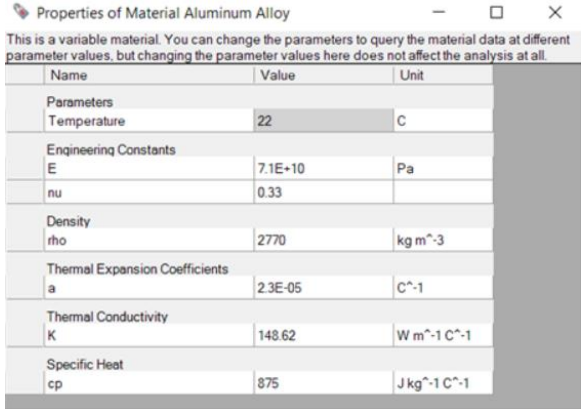
- Then, define materials for the fibers and the matrix within Material Designer interface:

From the tool bar or from the outline tree

Press the i to display the material properties

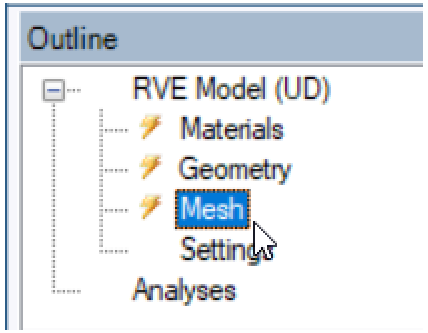


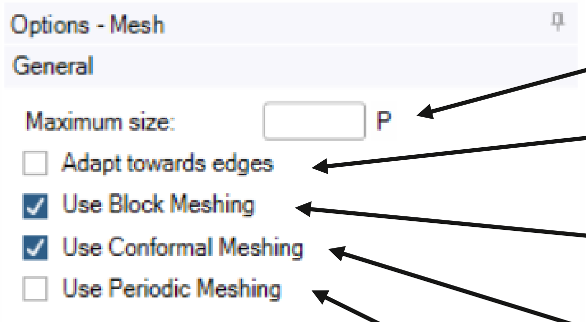




Mesh

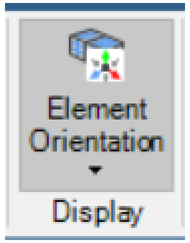
- The RVE needs to be meshed for finite element analysis

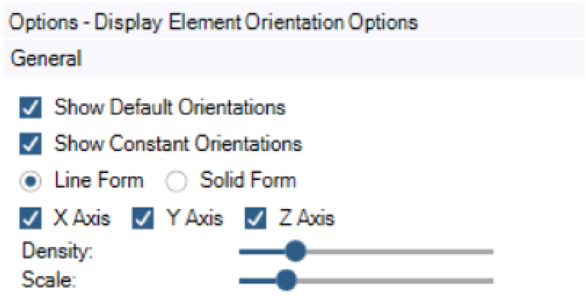


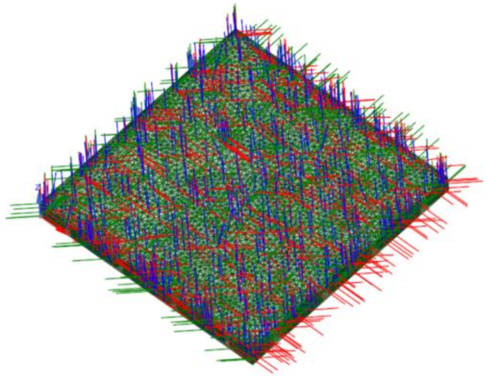


- Maximum element size
- Refine the mesh toward the inner edges
- Uses a block decomposition to generate block structured mesh
- Conformal mesh; implies nodes at the interfaces are shared
- Generate periodic meshes (meshes of opposite faces of the RVE must be the same)

- Once a mesh is generated, you can review the Element Orientation



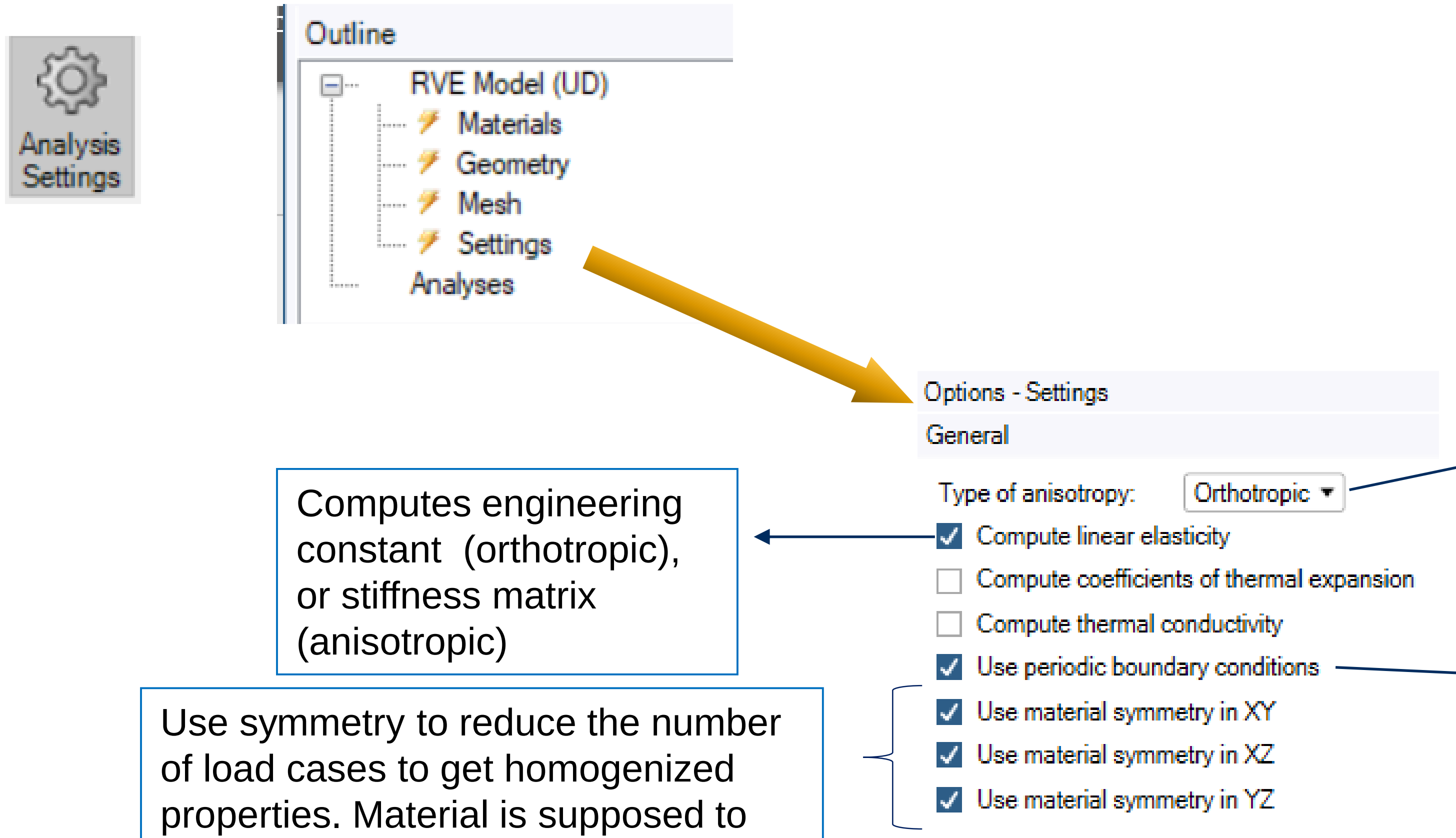




Ansys

Analysis Settings

- Analysis settings define the properties of the finite element analysis that will be realized on the RVE.



Orthotropic or Anisotropic

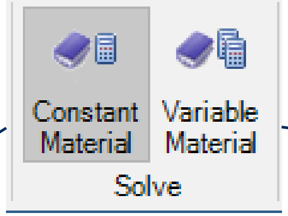
Computes engineering constant (orthotropic), or stiffness matrix (anisotropic)

Apply periodic boundary conditions to the finite element analysis

Use symmetry to reduce the number of load cases to get homogenized properties. Material is supposed to behave the same in the listed directions.

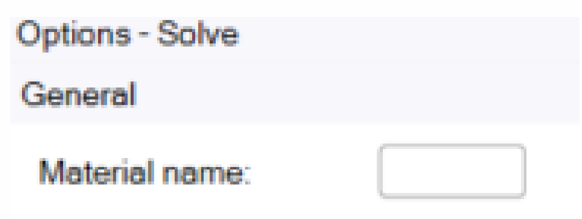
Solve Process

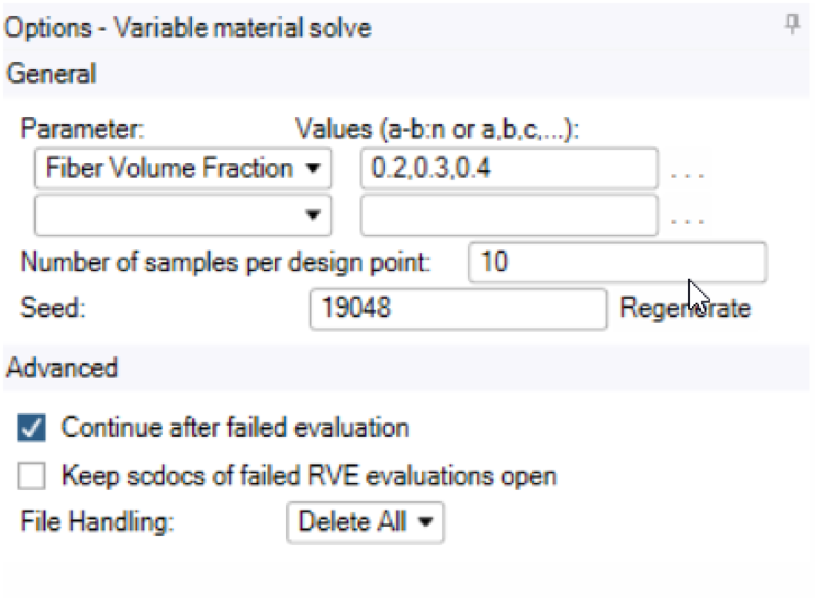
- Solve Process: Constant Material or Variable Material



Create variable material that depends on one or more parameters

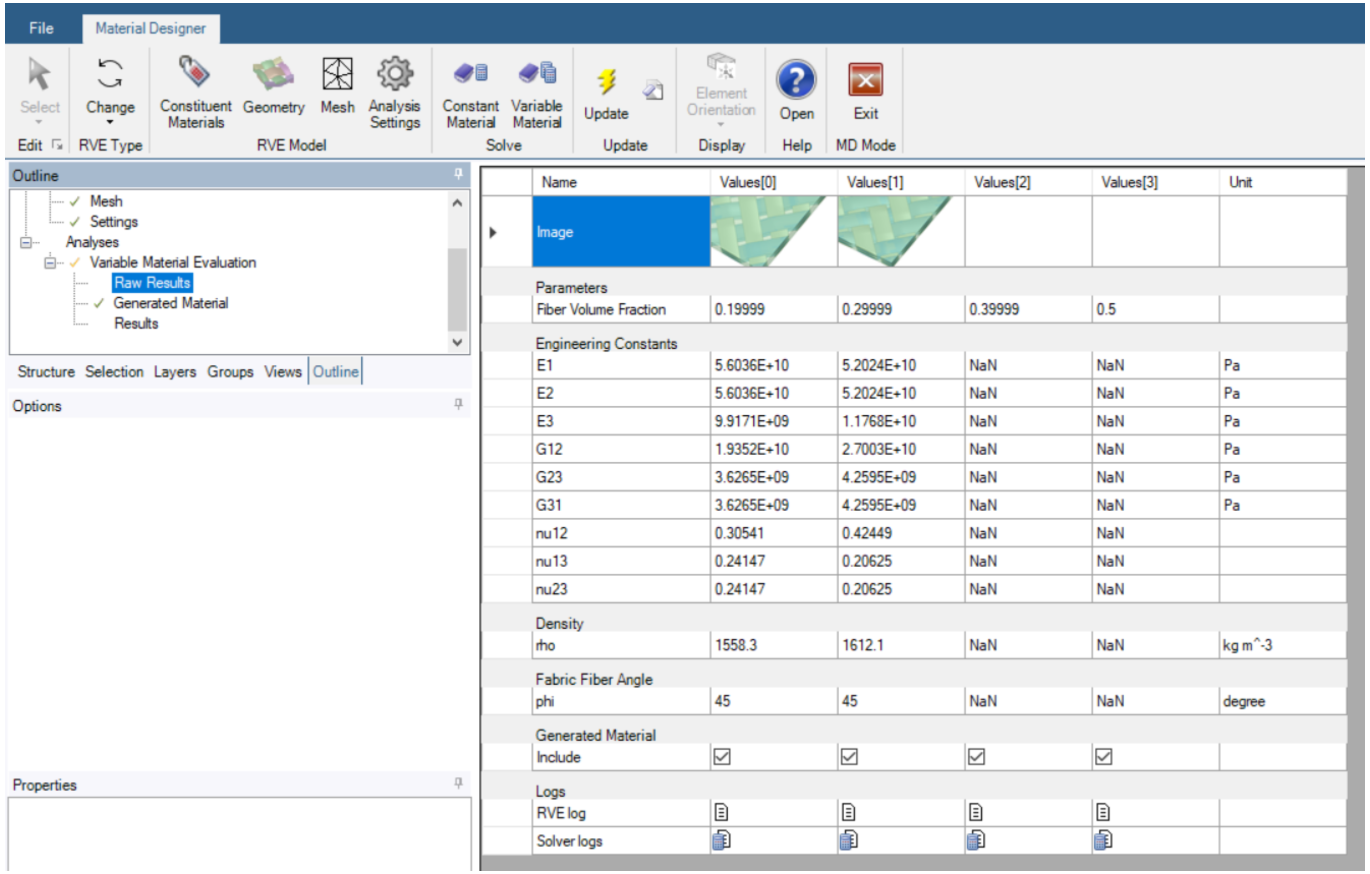
Define Material name





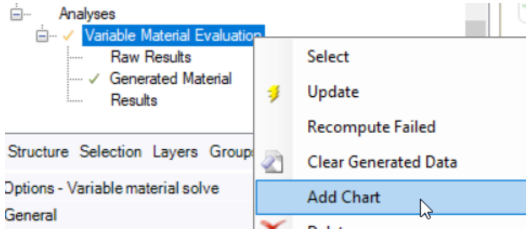
Results Review

- **Update** the analysis and **display property** results. Example for a variable material:

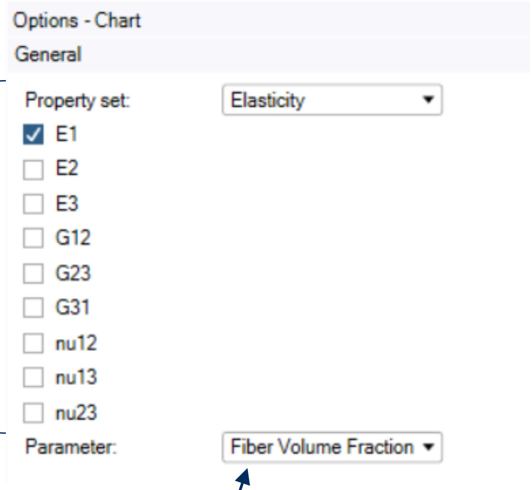


Results Review

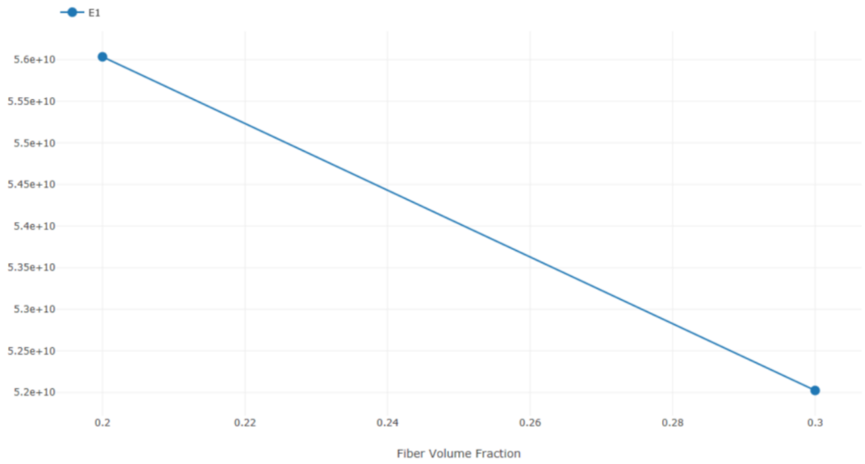
- You can review results under chart form for **variable material evaluation**: Right click on the variable material evaluation row from the outline > Add Chart:



Select the properties you want to display in the chart

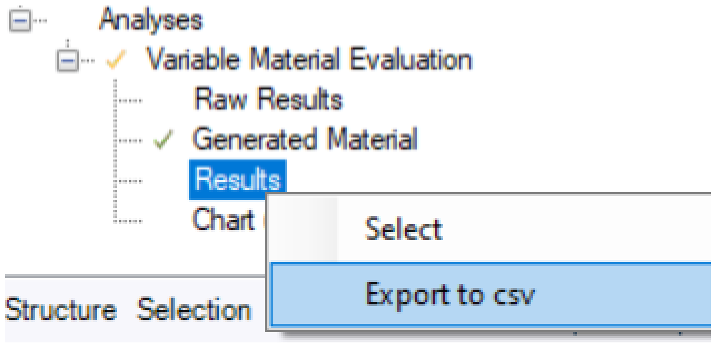


Select the variable parameter

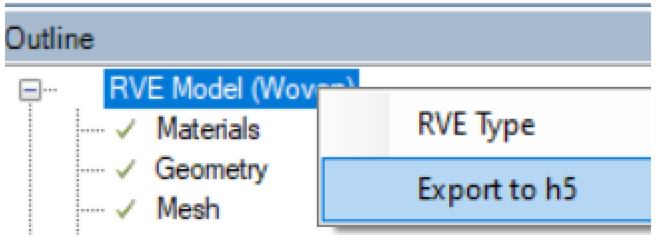


Exports

- Results can be exported as csv format: right click on Results > Export



- RVE model can be exported to hdl file format

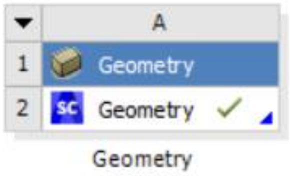


Example

- The aim of this example is to show the procedure to calculate homogenized material data for a user defined RVE.

Use the stator_prox_zone.scdoc geometry file provided with this lecture.

- Open a new Workbench project
- Add a Geometry component system and import the geometry file (Open SpaceClaim to have a quick look at the RVE geometry)
- Add a Material Designer component system to the Workbench project



Example

- Define materials to be used in Material Designer: open Engineering Data application from cell B2
- Add two new materials with the following isotropic elasticity properties:

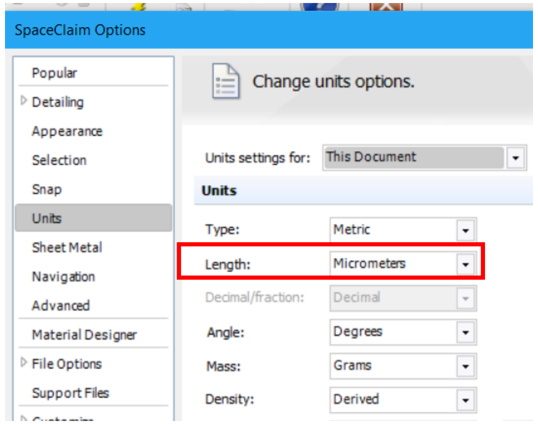
Epoxy:	
Young's Modulus	3 GPa
Poisson's Ratio	0.37
Steel:	
Young's Modulus	207 GPa
Poisson's Ratio	0.25

- Close Engineering Data application

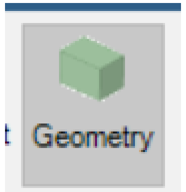
Example

- Launch Material Designer (double click on Cell B3)
- Select User Defined RVE Type from the ribbon bar

- Check the units in Material Designer:
  - File > SpaceClaim options > Units > Select micrometers



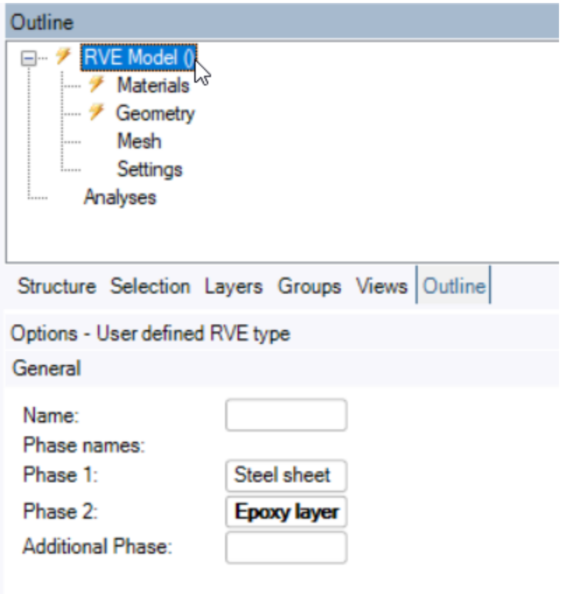
- Return to SpaceClaim interface and select Geometry from the ribbon bar



- Copy and Paste (use Ctrl + C / Ctrl + V) the 3 bodies from the SpaceClaim interface containing RVE geometry to Material Designer interface

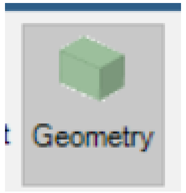
Example

- Highlight RVE Model in the tree outline
- Define two phases: steel sheet and epoxy layer



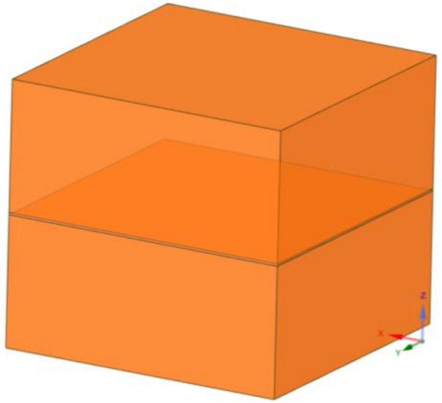
- Select Materials from the outline tree and define the materials as follow:

- Select Geometry in the ribbon bar

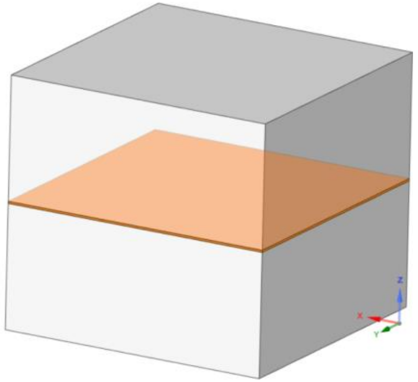


Example

- Click on Steel sheet and select the upper and lower bodies of the matrix (use Ctrl for multiple selection)

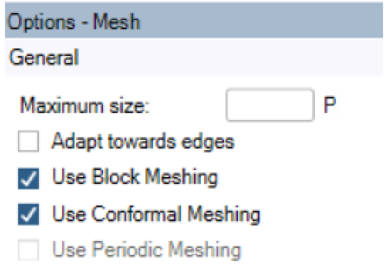


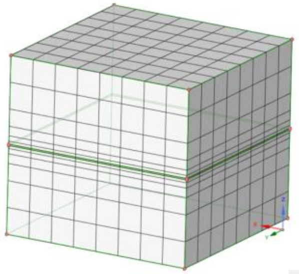
- Press  when done
- Repeat the steps for epoxy layer. Select the thin layer body

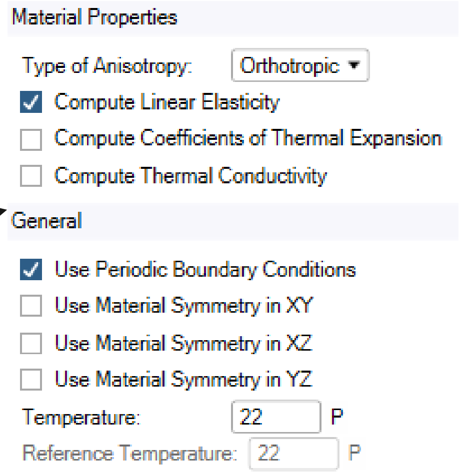


Example

- Click on Mesh button from the ribbon bar 

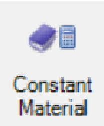
- Select the options as follow: 

- Generate mesh to have an overview  
- Click on the Analysis Settings button in the ribbon bar 

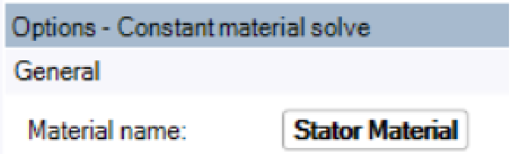
- Choose the options as on the image 

Example

- Select Constant Material in the ribbon bar

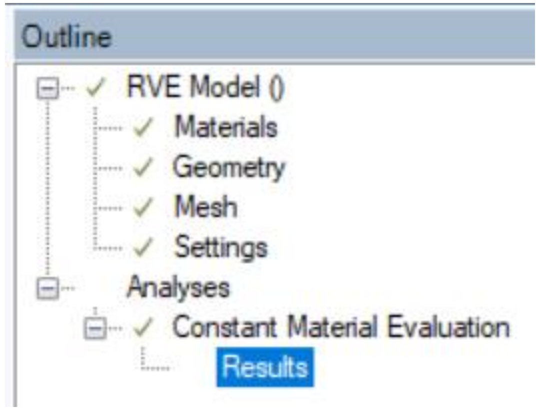


- Set the material name:



- Click complete to start the calculations

- Review the results

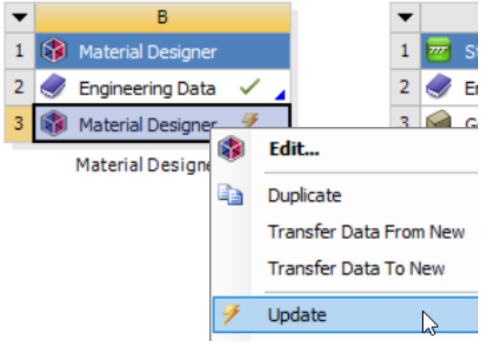


Name	Value	Unit	P
Engineering Constants			
E1	2.0531E+05	MPa	☐
E2	2.0531E+05	MPa	☐
E3	1.5694E+05	MPa	☐
G12	82125	MPa	☐
G23	51215	MPa	☐
G31	51215	MPa	☐
nu12	0.25002		☐
nu13	0.25157		☐
nu23	0.25157		☐

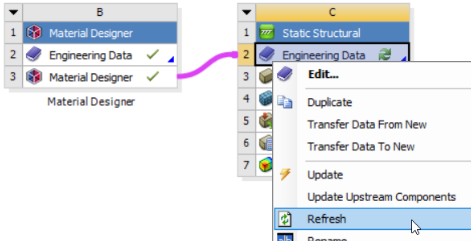
Ansys

Example

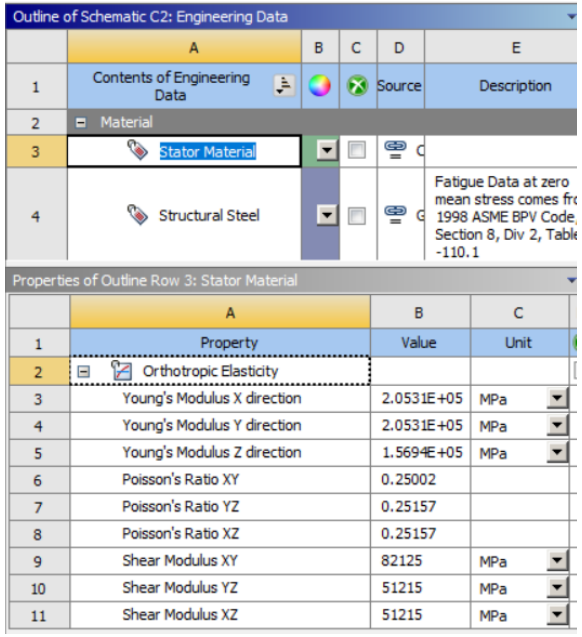
- To use the calculated properties in a downstream Static Structural analysis, return to the Workbench project page
- Add a Static Structural analysis system
- Update Material Designer cell B2



- Link Material Designer cell B2 to Engineering data cell C2 of the Static Structural analysis system, Refresh



- Double click on Engineering Data cell C2 and check the new created material



End of presentation