Created in COMSOL Multiphysics 6.1

Micromechanical Model of a Particulate Composite

Introduction

In this example, a simplified micromechanical model of a particulate composite is analyzed. A representative volume element (RVE) based on a predetermined particle spacing is assumed to represent the microstructure of the composite. The homogenized elastic and viscoelastic properties of the composite material are computed based on the individual properties of the particles and the matrix. Transient analyses of shear and normal loading of the composite microstructure yield the viscoelastic response of the composite, which is used to determine the homogenized viscoelastic parameters using curve fitting optimization.

The following considerations are important for the analysis:

- In the particulate composite, only the matrix material is assumed to show viscoelastic behavior. This is a realistic assumption for, for example, polymer matrix composites. Polymer-like resin shows viscoelastic behavior, whereas the embedded fibers or particles are assumed to be elastic.
- The viscoelastic properties of the matrix in the heterogeneous material and in the equivalent homogenized material are assumed to be represented by a generalized Maxwell model, having a Prony series representation.
- The viscoelastic model of the matrix in the heterogeneous material as well as the model of the equivalent homogeneous material are isotropic.
- The viscoelastic model of the matrix material is deviatoric. In contrast, the viscoelastic model of the homogenized material can be volumetric as well as deviatoric due to the inclusion of the particles. When determining the homogenized viscoelastic parameters through an optimization routine, it is assumed that the relaxation times of the homogenized material are the same as those of the matrix material.
- To determine the homogenized, isotropic, viscoelastic parameters from the particulate composite, two different stress responses from a unit cell RVE are required:
  - The shear stress response, which only activates the deviatoric part of the homogeneous, isotropic, viscoelastic model.
  - The normal stress response, which activates both the deviatoric and the volumetric part of the homogeneous, isotropic, viscoelastic model.
- During step loading in the time-dependent study, both the instantaneous and the long-term response of the heterogeneous RVE are purely elastic. They are therefore excluded

when determining the homogeneous, viscoelastic parameters through the optimization routine.

- In the current example, uniform spacing of particles is assumed in all three directions. However, it is also possible to perform micromechanical analyses for nonuniformly distributed particles.

Model Definition

The composite is assumed to be made of a periodic microstructure identified as a primitive cubic structure. A unit cube RVE having a spherical particle embedded in the center of the matrix is shown in Figure 1.

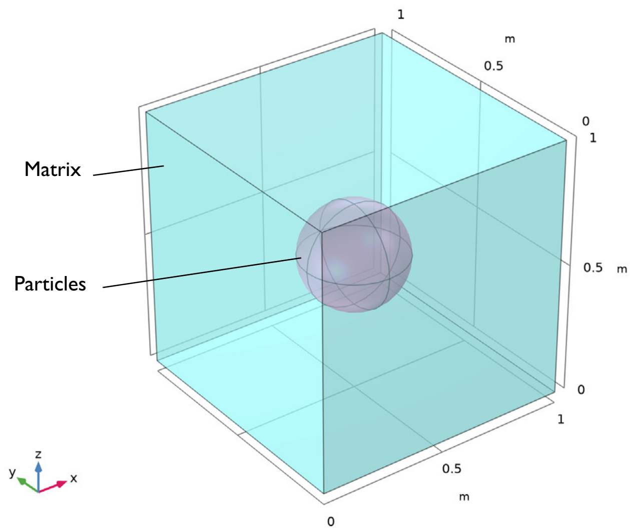

Figure 1: Geometry of the unit cell, consisting of a spherical particle embedded in epoxy resin.

Particle and Matrix Properties

The elastic material properties of particles and matrix are given in Table 1 and Table 2, respectively.

TABLE 1: PARTICLE MATERIAL PROPERTIES.

Material Property	**Value**
E_p	230 GPa
υ_p	0.2

TABLE 2: MATRIX MATERIAL PROPERTIES.

Material Property	**Value**
E_m	10 GPa
υ_m	0.35

The relaxation function $\Gamma(t)$ in the viscoelastic model for the matrix is expressed in terms of the instantaneous shear modulus G_0 and a set of N relative weights g_k and relaxation times τ_k, so that the Prony series is given as

$$\Gamma(t) = G_0\left(g_\infty + \sum_{k=1}^{N} g_k \exp\left(-\frac{t}{\tau_k}\right)\right)$$

where g_k and τ_k are the relative weight and the relaxation time constant of the spring-dashpot pair in branch k, respectively. In this case, the long-term shear modulus G is related to the instantaneous shear modulus G_0 by the weight $g_\infty < 1$

$$G = g_\infty G_0$$

and the shear modulus in each branch k is defined by the weight g_k

$$G_k = g_k G_0$$

It must be assumed that the weights fulfill the constraint

$$g_\infty + \sum_{k=1}^{N} g_k = 1$$

The relative weights and relaxation time constants for the three branches are given in Table 3.

TABLE 3: VISCOELASTIC PROPERTIES OF THE MATRIX.

Branch	**Relative Weight**	**Relaxation Time Constant**
1	0.01	0.01 s
2	0.05	0.1 s
3	0.08	1 s

Results and Discussion

The von Mises stress in the constituents when viscoelasticity is neglected is shown in Figure 2 and Figure 3 for normal and shear loading, respectively. The corresponding results when viscoelasticity in the matrix is included are reported in Figure 4 and Figure 5 for normal and shear loading, respectively. Here, the results are shown at the end of the simulation when the viscous branches are fully relaxed. Note that the stresses in the constituents are in good agreement with those computed in the elastic study, as expected.

The variation in average normal and shear stress with time for the heterogeneous RVE is shown in Figure 6. The initial response is elastic, which is followed by stress relaxation in the viscous branches.

The variations in average shear and normal stresses with time for the heterogeneous RVE and the homogenized, equivalent material are shown in Figure 7 and Figure 8, respectively. The relative weights for the deviatoric and volumetric parts obtained from the optimization routine are given in Table 4 and Table 5, respectively. It can be seen that the deviatoric relative weights for the homogenized material are close to those of the matrix due to the low particle volume fraction, which means that the viscoelastic response of the composite is dominated by the matrix viscoelasticity. Note, however, that the heterogeneity does result in nonzero volumetric relative weights.

TABLE 4: HOMOGENIZED VISCOELASTIC DEVIATORIC PROPERTIES.

Branch	**Relative Weight**	**Relaxation Time Constant**
1	0.00942	0.01 s
2	0.04918	0.1 s
3	0.07934	1 s

TABLE 5: HOMOGENIZED VISCOELASTIC VOLUMETRIC PROPERTIES.

Branch	**Relative Weight**	**Relaxation Time Constant**
1	6.516E-4	0.01 s
2	0.00436	0.1 s
3	0.00681	1 s

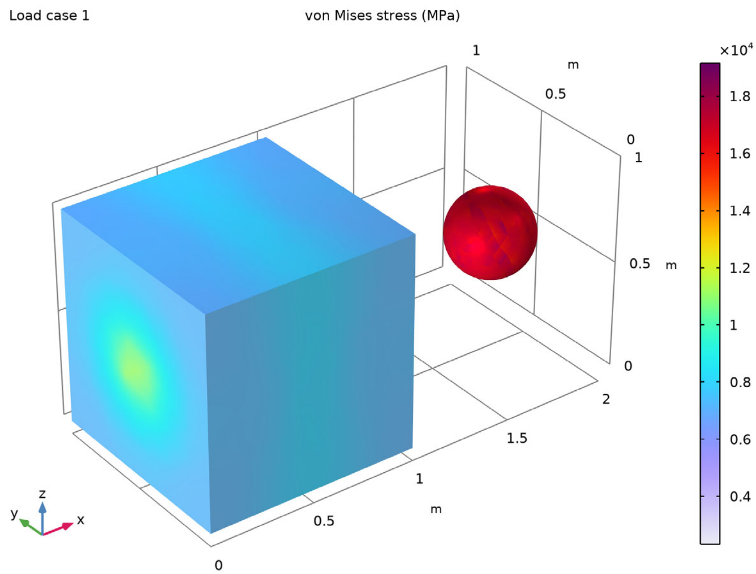

Figure 2: von Mises stress in matrix and particle due to axial loading (elastic conditions).

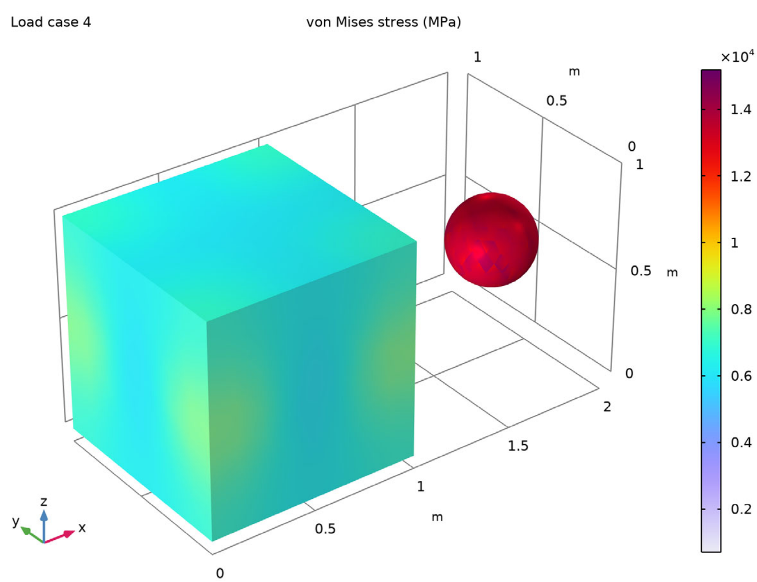

Figure 3: von Mises stress in matrix and particle due to shear loading (elastic conditions).

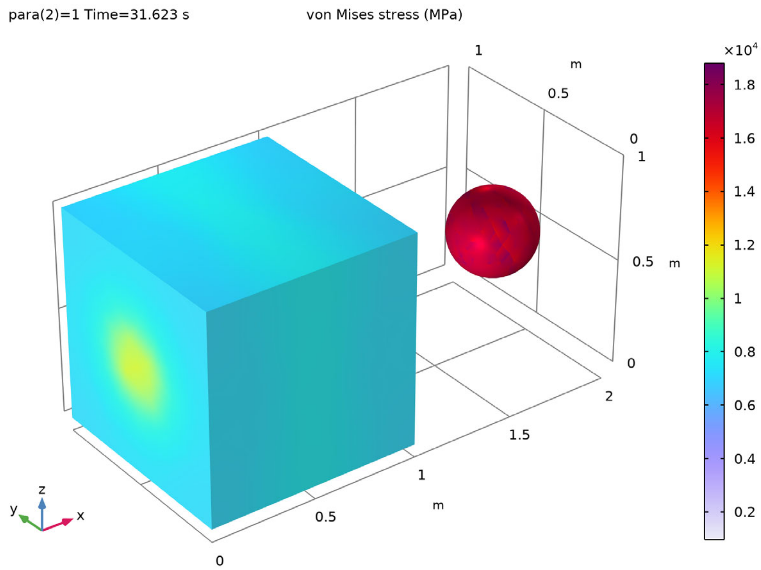

Figure 4: von Mises stress in particle and matrix under axial loading (viscoelasticity in matrix).

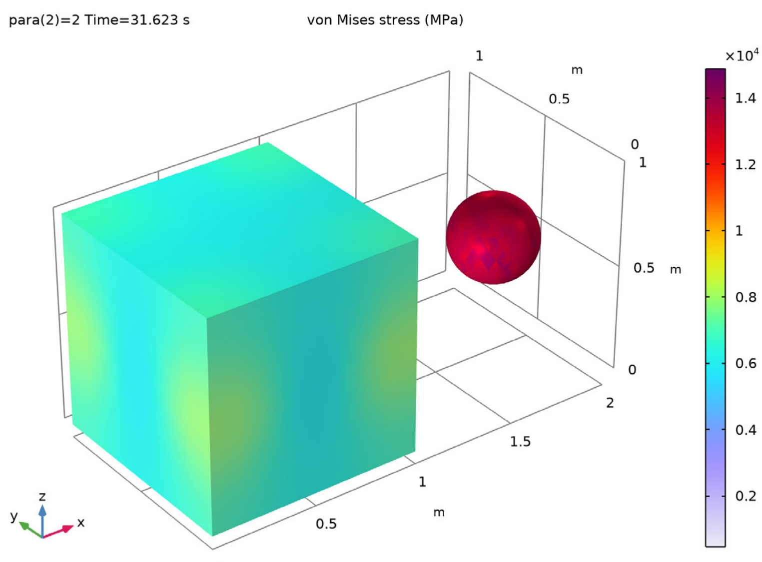

Figure 5: von Mises stress in particle and matrix under shear loading (viscoelasticity in matrix).

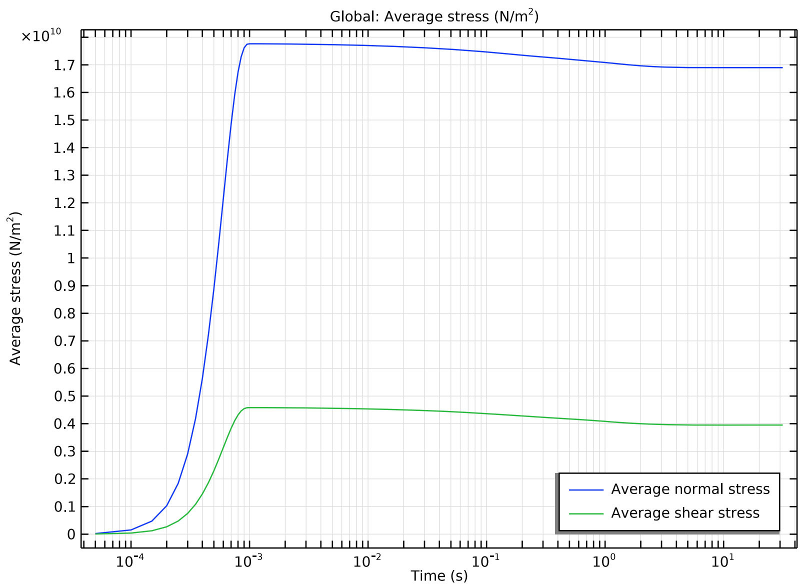

Figure 6: Average viscoelastic normal and shear stress in the heterogeneous RVE.

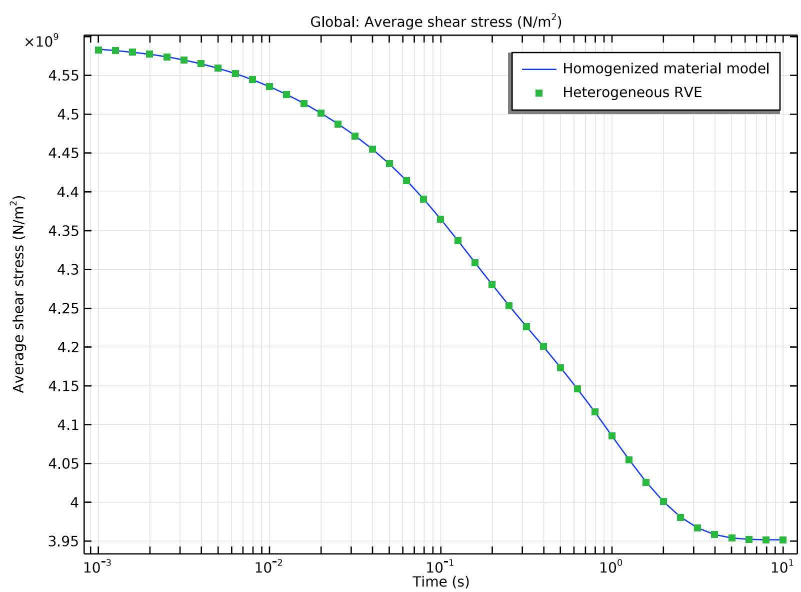

Figure 7: Average viscoelastic shear stress for the composite and the homogenized material.

.

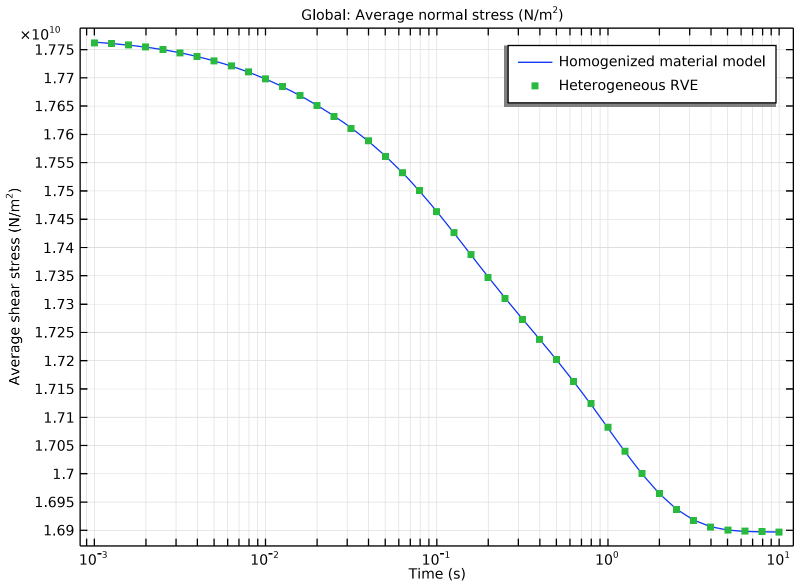

Figure 8: Average viscoelastic normal stress for the heterogeneous RVE and the equivalent homogenized material.

Notes About the COMSOL Implementation

- The micromechanical analysis of particles in a bulk matrix can be performed using the **Cell Periodicity** node available in the **Solid Mechanics** interface. Using this functionality, the elasticity matrix of the homogenized material can be computed for given particle and matrix properties.
- The **Cell Periodicity** node has three action buttons in the toolbar of the section called **Periodicity Type**: **Create Load Groups and Study**, **Create Material by Value**, and **Create Material by Reference**. The action button **Create Load Groups and Study** generates load groups and a stationary study with load cases. The action button **Create Material by Value** generates a **Global Material** with homogenized material properties, with material properties as numbers. The action button **Create Material by Reference** generates a **Global Material** with homogenized material properties, with material properties as variables. The action buttons are active depending on the choices in the **Periodicity Type** and **Calculate Average Properties** lists.
- The viscoelastic model of the matrix can be modeled using the **Generalized Maxwell** material model available in the **Viscoelasticity** feature.

Application Library path: `Structural_Mechanics_Module/Material_Models/micromechanical_model_of_a_particulate_composite`

Modeling Instructions

From the **File** menu, choose **New**.

NEW

In the **New** window, click **Model Wizard**.

MODEL WIZARD

1 In the **Model Wizard** window, click **3D**.

2 In the **Select Physics** tree, select **Structural Mechanics>Solid Mechanics (solid)**.

3 Click **Add**.

4 Click **Done**.

GLOBAL DEFINITIONS

Geometric Properties

1 In the **Model Builder** window, under **Global Definitions** click **Parameters 1**.

2 In the **Settings** window for **Parameters**, type `Geometric Properties` in the **Label** text field.

3 Locate the **Parameters** section. In the table, enter the following settings:

Name	Expression	Value	Description
para	0	0	Parameter
L	1[m]	1 m	Unit cell length
dp	0.4[m]	0.4 m	Particle diameter

Material Properties

1 In the **Home** toolbar, click **Parameters** and choose **Add>Parameters**.

2 In the **Settings** window for **Parameters**, type `Material Properties` in the **Label** text field.

3 Locate the **Parameters** section. Click **Load from File**.

4 Browse to the model's Application Libraries folder and double-click the file `micromechanical_model_of_a_particulate_composite_material_properties.txt`.

5 In the **Model Builder** window, right-click **Global Definitions** and choose **Geometry Parts>Part Libraries**.

PART LIBRARIES

1 In the **Part Libraries** window, select **COMSOL Multiphysics>Representative Volume Elements>3D>particulate_primitive_cubic** in the tree.

2 Right-click **Global Definitions** and choose **Add to Model**.

3 In the **Select Part Variant** dialog box, select **Specify particle diameter** in the **Select part variant** list.

4 Click **OK**.

Create one RVE geometry for the heterogeneous material and one for the homogenized material.

GEOMETRY 1

Heterogeneous RVE

1 In the **Geometry** toolbar, click **Parts** and choose **Particulate Composite, Primitive Cubic**.

2 In the **Settings** window for **Part Instance**, type `Heterogeneous RVE` in the **Label** text field.

To define the RVE geometry, enter the geometric properties in the input parameters of the part.

3 Locate the **Input Parameters** section. In the table, enter the following settings:

Name	Expression	Value	Description
dp	dp	0.4 m	Particle diameter
wm	L	1 m	Width of RVE
dm	L	1 m	Depth of RVE
hm	L	1 m	Height of RVE

Homogeneous RVE

1 In the **Geometry** toolbar, click **Block**.

2 In the **Settings** window for **Block**, type `Homogeneous RVE` in the **Label** text field.

3 Locate the **Size and Shape** section. In the **Width** text field, type `L`.
4 In the **Depth** text field, type `L`.
5 In the **Height** text field, type `L`.
6 Locate the **Position** section. In the **x** text field, type `2*L`.
7 Click **Build Selected**.

MATERIALS

Matrix

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Materials** and choose **Blank Material**.
2 In the **Settings** window for **Material**, type `Matrix` in the **Label** text field.
3 Locate the **Geometric Entity Selection** section. From the **Selection** list, choose **Matrix (Heterogeneous RVE)**.
4 Locate the **Material Contents** section. In the table, enter the following settings:

Property	Variable	Value	Unit	Property group
Young's modulus	E	E_m	Pa	Young's modulus and Poisson's ratio
Poisson's ratio	nu	nu_m	1	Young's modulus and Poisson's ratio

Particulates

1 Right-click **Materials** and choose **Blank Material**.
2 In the **Settings** window for **Material**, type `Particulates` in the **Label** text field.
3 Locate the **Geometric Entity Selection** section. From the **Selection** list, choose **Particle (Heterogeneous RVE)**.
4 Locate the **Material Contents** section. In the table, enter the following settings:

Property	Variable	Value	Unit	Property group
Young's modulus	E	E_p	Pa	Young's modulus and Poisson's ratio
Poisson's ratio	nu	nu_p	1	Young's modulus and Poisson's ratio

First, set up the **Solid Mechanics** interface to compute the homogenized elastic properties.

Set the structural transient behavior to quasistatic as the inertial response is of no interest.

SOLID MECHANICS: HETEROGENEOUS RVE

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Solid Mechanics (solid)**.
2 In the **Settings** window for **Solid Mechanics**, type `Solid Mechanics: Heterogeneous RVE` in the **Label** text field.
3 Locate the **Domain Selection** section. From the **Selection** list, choose **All (Heterogeneous RVE)**.
4 Locate the **Structural Transient Behavior** section. From the list, choose **Quasistatic**.

Linear Elastic Material 1

Use reduced integration to speed up the simulation.

1 In the **Model Builder** window, under **Component 1 (comp1)> Solid Mechanics: Heterogeneous RVE (solid)** click **Linear Elastic Material 1**.
2 In the **Settings** window for **Linear Elastic Material**, locate the **Quadrature Settings** section.
3 Select the **Reduced integration** check box.

Cell Periodicity for Elastic Properties

1 In the **Physics** toolbar, click **Domains** and choose **Cell Periodicity**.
2 In the **Settings** window for **Cell Periodicity**, type `Cell Periodicity for Elastic Properties` in the **Label** text field.
3 Locate the **Periodicity Type** section. From the list, choose **Average strain**.
4 From the **Calculate average properties** list, choose **Elasticity matrix, Standard (XX, YY, ZZ, XY, YZ, XZ)**.

Boundary Pair 1

1 In the **Physics** toolbar, click **Attributes** and choose **Boundary Pair**.
2 In the **Settings** window for **Boundary Pair**, locate the **Boundary Selection** section.
3 Click **Clear Selection**.
4 From the **Selection** list, choose **Pair 1 (Heterogeneous RVE)**.

Boundary Pair 2

1 Right-click **Boundary Pair 1** and choose **Duplicate**.
2 In the **Settings** window for **Boundary Pair**, locate the **Boundary Selection** section.
3 Click **Clear Selection**.
4 From the **Selection** list, choose **Pair 2 (Heterogeneous RVE)**.

Boundary Pair 3

1 Right-click **Boundary Pair 2** and choose **Duplicate**.

2 In the **Settings** window for **Boundary Pair**, locate the **Boundary Selection** section.

3 Click **Clear Selection**.

4 From the **Selection** list, choose **Pair 3 (Heterogeneous RVE)**.

Cell Periodicity for Elastic Properties

With the **Average strain** option in the **Cell Periodicity** feature, appropriate load groups, a study, and a material with computed elastic properties can be generated automatically.

1 In the **Model Builder** window, click **Cell Periodicity for Elastic Properties**.

2 In the **Settings** window for **Cell Periodicity**, click **Study and Material Generation** in the upper-right corner of the **Periodicity Type** section. From the menu, choose **Create Load Groups and Study**.

MESH 1

Free Triangular 1

1 In the **Mesh** toolbar, click **Boundary** and choose **Free Triangular**.

2 Select Boundaries 1–3 only.

Size 1

1 Right-click **Free Triangular 1** and choose **Size**.

2 In the **Settings** window for **Size**, locate the **Geometric Entity Selection** section.

3 From the **Geometric entity level** list, choose **Entire geometry**.

4 Locate the **Element Size** section. From the **Predefined** list, choose **Finer**.

5 Click the **Custom** button.

6 Locate the **Element Size Parameters** section. Select the **Maximum element size** check box.

7 Select the **Minimum element size** check box.

8 Select the **Maximum element growth rate** check box.

9 Select the **Curvature factor** check box.

10 Select the **Resolution of narrow regions** check box.

11 Click **Build Selected**.

Free Triangular 2

1 In the **Mesh** toolbar, click **Boundary** and choose **Free Triangular**.

2 Select Boundaries 6–13 only.

Size 1

1 Right-click **Free Triangular 2** and choose **Size**.

2 In the **Settings** window for **Size**, locate the **Geometric Entity Selection** section.

3 From the **Geometric entity level** list, choose **Entire geometry**.

4 Locate the **Element Size** section. From the **Predefined** list, choose **Fine**.

5 Click the **Custom** button.

6 Locate the **Element Size Parameters** section.

7 Select the **Maximum element size** check box. In the associated text field, type `0.07`.

8 Select the **Minimum element size** check box. In the associated text field, type `0.05`.

9 Select the **Maximum element growth rate** check box.

10 Select the **Curvature factor** check box.

11 Select the **Resolution of narrow regions** check box.

12 Click **Build Selected**.

Copy Face 1

1 In the **Mesh** toolbar, click **Copy** and choose **Copy Face**.

2 In the **Settings** window for **Copy Face**, locate the **Source Boundaries** section.

3 From the **Selection** list, choose **Pair 1, Source (Heterogeneous RVE)**.

4 Locate the **Destination Boundaries** section. Click to select the **Activate Selection** toggle button.

5 From the **Selection** list, choose **Pair 1, Destination (Heterogeneous RVE)**.

6 Click **Build Selected**.

Copy Face 2

1 In the **Mesh** toolbar, click **Copy** and choose **Copy Face**.

2 In the **Settings** window for **Copy Face**, locate the **Source Boundaries** section.

3 From the **Selection** list, choose **Pair 2, Source (Heterogeneous RVE)**.

4 Locate the **Destination Boundaries** section. Click to select the **Activate Selection** toggle button.

5 From the **Selection** list, choose **Pair 2, Destination (Heterogeneous RVE)**.

6 Click **Build Selected**.

Copy Face 3

1 In the **Mesh** toolbar, click **Copy** and choose **Copy Face**.

2 In the **Settings** window for **Copy Face**, locate the **Source Boundaries** section.

3 From the **Selection** list, choose **Pair 3, Source (Heterogeneous RVE)**.

4 Locate the **Destination Boundaries** section. Click to select the **Activate Selection** toggle button.

5 From the **Selection** list, choose **Pair 3, Destination (Heterogeneous RVE)**.

6 Click **Build Selected**.

Free Tetrahedral 1

1 In the **Mesh** toolbar, click **Free Tetrahedral**.

2 In the **Settings** window for **Free Tetrahedral**, locate the **Domain Selection** section.

3 From the **Geometric entity level** list, choose **Domain**.

4 From the **Selection** list, choose **All (Heterogeneous RVE)**.

Size 1

1 Right-click **Free Tetrahedral 1** and choose **Size**.

2 In the **Settings** window for **Size**, locate the **Geometric Entity Selection** section.

3 From the **Geometric entity level** list, choose **Entire geometry**.

4 Locate the **Element Size** section. From the **Predefined** list, choose **Normal**.

5 Click the **Custom** button.

6 Locate the **Element Size Parameters** section. Select the **Maximum element size** check box.

7 Select the **Minimum element size** check box.

8 Select the **Maximum element growth rate** check box.

9 Select the **Curvature factor** check box. In the associated text field, type `0.4`.

10 Select the **Resolution of narrow regions** check box.

11 Click **Build All**.

Mapped 1

1 In the **Mesh** toolbar, click **Boundary** and choose **Mapped**.

2 Select Boundary 15 only.

Distribution 1

1 Right-click **Mapped 1** and choose **Distribution**.

2 Select Edges 28 and 30 only.

3 In the **Settings** window for **Distribution**, locate the **Distribution** section.

4 In the **Number of elements** text field, type `1`.

Swept 1

1 In the **Mesh** toolbar, click **Swept**.

2 In the **Settings** window for **Swept**, locate the **Domain Selection** section.

3 From the **Geometric entity level** list, choose **Domain**.

4 Select Domain 3 only.

Distribution 1

1 Right-click **Swept 1** and choose **Distribution**.

2 In the **Settings** window for **Distribution**, locate the **Distribution** section.

3 In the **Number of elements** text field, type `1`.

4 Click **Build All**.

CELL PERIODICITY STUDY FOR ELASTIC PROPERTIES (HETEROGENEOUS RVE)

1 In the **Model Builder** window, click **Cell Periodicity Study**.

2 In the **Settings** window for **Study**, type `Cell Periodicity Study for Elastic Properties (Heterogeneous RVE)` in the **Label** text field.

3 In the **Home** toolbar, click **Compute**.

RESULTS

Stress, Elastic Response

1 In the **Model Builder** window, under **Results** click **Stress (solid)**.

2 In the **Settings** window for **3D Plot Group**, type `Stress, Elastic Response` in the **Label** text field.

3 Click to expand the **Selection** section. From the **Geometric entity level** list, choose **Domain**.

4 From the **Selection** list, choose **All (Heterogeneous RVE)**.

5 Select the **Apply to dataset edges** check box.

6 Click to expand the **Title** section. From the **Title type** list, choose **Manual**.

7 In the **Title** text area, type `von Mises stress (MPa)` .

8 Locate the **Plot Settings** section. Clear the **Plot dataset edges** check box.

9 Click to expand the **Plot Array** section. Select the **Enable** check box.

Volume 1

1 In the **Model Builder** window, expand the **Stress, Elastic Response** node, then click **Volume 1**.

2 In the **Settings** window for **Volume**, locate the **Expression** section.

3 From the **Unit** list, choose **MPa**.

Selection 1

1 Right-click **Volume 1** and choose **Selection**.

2 In the **Settings** window for **Selection**, locate the **Selection** section.

3 From the **Selection** list, choose **Matrix (Heterogeneous RVE)**.

Deformation

In the **Model Builder** window, under **Results>Stress, Elastic Response>Volume 1** right-click **Deformation** and choose **Delete**.

Volume 2

1 In the **Model Builder** window, under **Results>Stress, Elastic Response** right-click **Volume 1** and choose **Duplicate**.

2 In the **Settings** window for **Volume**, click to expand the **Inherit Style** section.

3 From the **Plot** list, choose **Volume 1**.

4 Click to expand the **Plot Array** section. Clear the **Apply to dataset edges** check box.

Selection 1

1 In the **Model Builder** window, expand the **Volume 2** node, then click **Selection 1**.

2 In the **Settings** window for **Selection**, locate the **Selection** section.

3 Click **Clear Selection**.

4 From the **Selection** list, choose **Particle (Heterogeneous RVE)**.

Stress, Elastic Response

Click the **Go to Default View** button in the **Graphics** toolbar.

Before you set up the physics to analyze the viscoelastic response, create a homogenized material from the **Cell Periodicity** feature.

The homogenized material can be created by using either of the two action buttons in the **Periodicity type** section, **Create Material by Reference** or **Create Material by Value**. Choose the second action button in order to generate a material with numbers.

SOLID MECHANICS: HETEROGENEOUS RVE (SOLID)

Cell Periodicity for Elastic Properties

1 In the **Model Builder** window, under **Component 1 (comp1)>Solid Mechanics: Heterogeneous RVE (solid)** click **Cell Periodicity for Elastic Properties**.

2 In the **Settings** window for **Cell Periodicity**, click **Study and Material Generation** in the upper-right corner of the **Periodicity Type** section. From the menu, choose **Create Material by Value** to generate a global material node with the computed elastic properties.

Set up the physics interface to analyze the viscoelastic response of the composite.

GLOBAL DEFINITIONS

Step 1 (step1)

1 In the **Home** toolbar, click f(X) **Functions** and choose **Global>Step**.

2 In the **Settings** window for **Step**, type `strainFunction` in the **Function name** text field.

3 Locate the **Parameters** section. In the **Location** text field, type `5e-4[s]`.

4 Click to expand the **Smoothing** section. In the **Size of transition zone** text field, type `1e-3[s]`.

DEFINITIONS

Variables: Heterogeneous RVE

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Definitions** and choose **Variables**.

2 In the **Settings** window for **Variables**, type `Variables: Heterogeneous RVE` in the **Label** text field.

3 Locate the **Variables** section. In the table, enter the following settings:

Name	Expression	Unit	Description
G_m	E_m/(2*(1+nu_m))	Pa	Shear modulus of matrix
sum_g	g1+g2+g3		Sum of weights
G_m0	G_m/(1-sum_g)	Pa	Instantaneous shear modulus of matrix

SOLID MECHANICS: HETEROGENEOUS RVE (SOLID)

Linear Elastic Material 1

In the **Model Builder** window, under **Component 1 (comp1)> Solid Mechanics: Heterogeneous RVE (solid)** click **Linear Elastic Material 1**.

Viscoelasticity 1

1 In the **Physics** toolbar, click **Attributes** and choose **Viscoelasticity**.

2 In the **Settings** window for **Viscoelasticity**, locate the **Domain Selection** section.

3 From the **Selection** list, choose **Matrix (Heterogeneous RVE)**.

4 Locate the **Viscoelasticity Model** section. In the table, enter the following settings:

Branch	Shear modulus (Pa)	Relaxation time (s)
1	G_m0*g1	Tau1

5 Click Add.

6 In the table, enter the following settings:

Branch	Shear modulus (Pa)	Relaxation time (s)
2	G_m0*g2	Tau2

7 Click Add.

8 In the table, enter the following settings:

Branch	Shear modulus (Pa)	Relaxation time (s)
3	G_m0*g3	Tau3

Cell Periodicity for Viscoelastic Properties

1 In the **Model Builder** window, right-click **Cell Periodicity for Elastic Properties** and choose **Duplicate**.

2 In the **Settings** window for **Cell Periodicity**, type `Cell Periodicity for Viscoelastic Properties` in the **Label** text field.

3 Locate the **Periodicity Type** section. From the **Calculate average properties** list, choose **None**.

4 In the ε_{avg} table, enter the following settings:

(para==1)*strainFunction(t)	(para==2)*0.5*strainFunction(t)	0
(para==2)*0.5*strainFunction(t)	0	0
0	0	0

ADD STUDY

1 In the **Home** toolbar, click **Add Study** to open the **Add Study** window.

2 Go to the **Add Study** window.

3 Find the **Studies** subsection. In the **Select Study** tree, select **General Studies> Time Dependent**.

4 Click **Add Study** in the window toolbar.

5 In the **Home** toolbar, click **Add Study** to close the **Add Study** window.

TRANSIENT STUDY FOR VISCOELASTIC RESPONSE (HETEROGENEOUS RVE)

1 In the **Model Builder** window, click **Study 1**.

2 In the **Settings** window for **Study**, type `Transient Study for Viscoelastic Response (Heterogeneous RVE)` in the **Label** text field.

Parametric Sweep

1 In the **Study** toolbar, click **Parametric Sweep**.

2 In the **Settings** window for **Parametric Sweep**, locate the **Study Settings** section.

3 Click **Add**.

4 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
para (Parameter)	`range(1,1,2)`	

Step 1: Time Dependent

1 In the **Model Builder** window, click **Step 1: Time Dependent**.

2 In the **Settings** window for **Time Dependent**, locate the **Study Settings** section.

3 In the **Output times** text field, type `range(0,0.5e-4,9.5e-4) 10^{range(-3,0.1,1.5)}`.

4 Locate the **Physics and Variables Selection** section. Select the **Modify model configuration for study step** check box.

5 In the tree, select **Component 1 (comp1)>Solid Mechanics: Heterogeneous RVE (solid)>Cell Periodicity for Elastic Properties**.

6 Right-click and choose **Disable**.

Customize the solver settings by choosing a smaller initial time step for better convergence.

Solution 1 (sol1)

1 In the **Study** toolbar, click **Show Default Solver**.

2 In the **Model Builder** window, expand the **Solution 1 (sol1)** node, then click **Time-Dependent Solver 1**.

3 In the **Settings** window for **Time-Dependent Solver**, click to expand the **Time Stepping** section.

4 Select the **Initial step** check box. In the associated text field, type `5e-7`.

5 In the **Study** toolbar, click **= Compute**.

Visualize the stress in the composite when matrix viscoelasticity is activated.

RESULTS

Stress, Viscoelastic Response

1 In the **Model Builder** window, under **Results** click **Stress (solid)**.
2 In the **Settings** window for **3D Plot Group**, type `Stress, Viscoelastic Response` in the **Label** text field.
3 Click to expand the **Selection** section. From the **Geometric entity level** list, choose **Domain**.
4 From the **Selection** list, choose **All (Heterogeneous RVE)**.
5 Select the **Apply to dataset edges** check box.
6 Locate the **Title** section. From the **Title type** list, choose **Manual**.
7 In the **Title** text area, type `von Mises stress (MPa)` .
8 Locate the **Plot Settings** section. Clear the **Plot dataset edges** check box.
9 Locate the **Plot Array** section. Select the **Enable** check box.

Volume 1

1 In the **Model Builder** window, expand the **Stress, Viscoelastic Response** node, then click **Volume 1**.
2 In the **Settings** window for **Volume**, locate the **Expression** section.
3 From the **Unit** list, choose **MPa**.

Selection 1

1 Right-click **Volume 1** and choose **Selection**.
2 In the **Settings** window for **Selection**, locate the **Selection** section.
3 From the **Selection** list, choose **Matrix (Heterogeneous RVE)**.

Deformation

In the **Model Builder** window, under **Results>Stress, Viscoelastic Response>Volume 1** right-click **Deformation** and choose **Delete**.

Volume 2

1 In the **Model Builder** window, under **Results>Stress, Viscoelastic Response** right-click **Volume 1** and choose **Duplicate**.
2 In the **Settings** window for **Volume**, locate the **Inherit Style** section.

3 From the **Plot** list, choose **Volume 1**.

4 Locate the **Plot Array** section. Clear the **Apply to dataset edges** check box.

Selection 1

1 In the **Model Builder** window, expand the **Volume 2** node, then click **Selection 1**.

2 In the **Settings** window for **Selection**, locate the **Selection** section.

3 Click **Clear Selection**.

4 From the **Selection** list, choose **Particle (Heterogeneous RVE)**.

Stress, Viscoelastic Response

Click the **Go to Default View** button in the **Graphics** toolbar.

Average Normal and Shear Stresses

1 In the **Home** toolbar, click **Add Plot Group** and choose **1D Plot Group**.

2 In the **Settings** window for **1D Plot Group**, type `Average Normal and Shear Stresses` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Transient Study for Viscoelastic Response (Heterogeneous RVE)/ Parametric Solutions 1 (sol2)**.

4 Locate the **Plot Settings** section.

5 Select the **y-axis label** check box. In the associated text field, type `Average stress (N/m<sup>2</sup>)`.

6 Click to expand the **Title** section. From the **Title type** list, choose **Manual**.

7 In the **Title** text area, type `Global: Average stress (N/m<sup>2</sup>)` .

8 Locate the **Axis** section. Select the **x-axis log scale** check box.

9 Locate the **Legend** section. From the **Position** list, choose **Lower right**.

Global 1

1 Right-click **Average Normal and Shear Stresses** and choose **Global**.

2 In the **Settings** window for **Global**, locate the **Data** section.

3 From the **Dataset** list, choose **Transient Study for Viscoelastic Response (Heterogeneous RVE)/ Parametric Solutions 1 (sol2)**.

4 From the **Parameter selection (para)** list, choose **First**.

5 Locate the **y-Axis Data** section. In the table, enter the following settings:

Expression	**Unit**	**Description**
solid.cp2.savgXX	N/m^2	Average stress, XX direction

6 Click to expand the **Legends** section. From the **Legends** list, choose **Manual**.

7 In the table, enter the following settings:

Legends
Average normal stress

Global 2

1 Right-click **Global 1** and choose **Duplicate**.

2 In the **Settings** window for **Global**, locate the **Data** section.

3 From the **Parameter selection (para)** list, choose **Last**.

4 Locate the **y-Axis Data** section. In the table, enter the following settings:

Expression	**Unit**	**Description**
solid.cp2.savgXY	N/m^2	Average stress, XY direction

5 Locate the **Legends** section. In the table, enter the following settings:

Legends
Average shear stress

6 In the **Average Normal and Shear Stresses** toolbar, click **Plot**.

Evaluation Group: Normal Stress Response

1 In the **Results** toolbar, click **Evaluation Group**.

2 In the **Settings** window for **Evaluation Group**, type `Evaluation Group: Normal Stress Response` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Transient Study for Viscoelastic Response (Heterogeneous RVE)/ Parametric Solutions 1 (sol2)**.

4 From the **Parameter selection (para)** list, choose **First**.

5 From the **Time selection** list, choose **Manual**.

6 In the **Time indices (1-66)** text field, type `range(21,1,61)`.

Global Evaluation 1

1 Right-click **Evaluation Group: Normal Stress Response** and choose **Global Evaluation**.

2 In the **Settings** window for **Global Evaluation**, locate the **Expressions** section.

3 In the table, enter the following settings:

Expression	Unit	Description
t	s	Time
solid.cp2.savgXX	N/m^2	Average stress, XX component

Evaluation Group: Normal Stress Response

1 In the **Model Builder** window, click **Evaluation Group: Normal Stress Response**.

2 In the **Settings** window for **Evaluation Group**, click to expand the **Format** section.

3 From the **Include parameters** list, choose **Off**.

4 In the **Evaluation Group: Normal Stress Response** toolbar, click **Evaluate**.

Evaluation Group: Shear Stress Response

1 In the **Results** toolbar, click **Evaluation Group**.

2 In the **Settings** window for **Evaluation Group**, type `Evaluation Group: Shear Stress Response` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Transient Study for Viscoelastic Response (Heterogeneous RVE)/ Parametric Solutions 1 (sol2)**.

4 From the **Parameter selection (para)** list, choose **Last**.

5 From the **Time selection** list, choose **Manual**.

6 In the **Time indices (1-66)** text field, type `range(21,1,61)`.

Global Evaluation 1

1 Right-click **Evaluation Group: Shear Stress Response** and choose **Global Evaluation**.

2 In the **Settings** window for **Global Evaluation**, locate the **Expressions** section.

3 In the table, enter the following settings:

Expression	Unit	Description
t	s	Time
solid.cp2.savgXY	N/m^2	Average stress, XY component

Evaluation Group: Shear Stress Response

1 In the **Model Builder** window, click **Evaluation Group: Shear Stress Response**.

2 In the **Settings** window for **Evaluation Group**, locate the **Format** section.

3 From the **Include parameters** list, choose **Off**.

4 In the **Evaluation Group: Shear Stress Response** toolbar, click **Evaluate**.

Average Normal and Shear Stresses, Evaluation Group: Normal Stress Response, Evaluation Group: Shear Stress Response, Stress, Elastic Response, Stress, Viscoelastic Response

1 In the **Model Builder** window, under **Results**, Ctrl-click to select **Stress, Elastic Response**, **Stress, Viscoelastic Response**, **Average Normal and Shear Stresses**, **Evaluation Group: Normal Stress Response**, and **Evaluation Group: Shear Stress Response**.

2 Right-click and choose **Group**.

Heterogeneous RVE

In the **Settings** window for **Group**, type `Heterogeneous RVE` in the **Label** text field.

GLOBAL DEFINITIONS

Optimization parameters

1 In the **Home** toolbar, click **Parameters** and choose **Add>Parameters**.

2 In the **Settings** window for **Parameters**, type `Optimization parameters` in the **Label** text field.

3 Locate the **Parameters** section. In the table, enter the following settings:

Name	Expression	Value	Description
gg1	0	0	Deviatoric Prony series parameter of homogenized material, branch 1
gg2	0	0	Deviatoric Prony series parameter of homogenized material, branch 2
gg3	0	0	Deviatoric Prony series parameter of homogenized material, branch 3
kg1	0	0	Volumetric Prony series parameter of homogenized material, branch 1
kg2	0	0	Volumetric Prony series parameter of homogenized material, branch 2
kg3	0	0	Volumetric Prony series parameter of homogenized material, branch 3

DEFINITIONS

Variables: Homogenized material

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Definitions** and choose **Variables**.

2 In the **Settings** window for **Variables**, type `Variables: Homogenized material` in the **Label** text field.

3 Locate the **Variables** section. In the table, enter the following settings:

Name	Expression	Unit	Description
G_H	solid2.D66		Shear modulus of homogenized material
K_H	solid2.D11-4*G_H/3		Bulk modulus of homogenized material
sum_gH	gg1+gg2+gg3		Sum of weights
sum_kH	kg1+kg2+kg3		Sum of weights
G_H0	G_H/(1-sum_gH)		Instantaneous shear modulus of homogenized material
K_H0	K_H/(1-sum_kH)		Instantaneous bulk modulus of homogenized material

MATERIALS

Homogeneous Material

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Materials** and choose **More Materials>Material Link**.

2 In the **Settings** window for **Material Link**, type `Homogeneous Material` in the **Label** text field.

3 Select Domain 3 only.

ADD PHYSICS

1 In the **Home** toolbar, click **Add Physics** to open the **Add Physics** window.

2 Go to the **Add Physics** window.

3 In the tree, select **Structural Mechanics>Solid Mechanics (solid)**.

4 Click **Add to Component 1** in the window toolbar.

5 In the **Home** toolbar, click **Add Physics** to close the **Add Physics** window.

SOLID MECHANICS: HOMOGENEOUS RVE

1 In the **Settings** window for **Solid Mechanics**, type `Solid Mechanics: Homogeneous RVE` in the **Label** text field.

2 Select Domain 3 only.

3 Locate the **Structural Transient Behavior** section. From the list, choose **Quasistatic**.

4 Click to expand the **Discretization** section. From the **Displacement field** list, choose **Linear**.

Two separate studies are required to compute the homogenized viscoelastic parameters. First, apply a unit (engineering) shear strain in order to find the homogenized deviatoric Prony series parameters.

Linear Elastic Material 1

1 In the **Model Builder** window, under **Component 1 (comp1)> Solid Mechanics: Homogeneous RVE (solid2)** click **Linear Elastic Material 1**.

2 In the **Settings** window for **Linear Elastic Material**, locate the **Linear Elastic Material** section.

3 From the **Material symmetry** list, choose **Anisotropic**.

Viscoelasticity 1

1 In the **Physics** toolbar, click **Attributes** and choose **Viscoelasticity**.

2 In the **Settings** window for **Viscoelasticity**, locate the **Viscoelasticity Model** section.

3 In the table, enter the following settings:

Branch	Shear modulus (Pa)	Relaxation time (s)
1	`G_H0*gg1`	`Tau1`

4 Click **Add**.

5 In the table, enter the following settings:

Branch	Shear modulus (Pa)	Relaxation time (s)
2	`G_H0*gg2`	`Tau2`

6 Click **Add**.

7 In the table, enter the following settings:

Branch	Shear modulus (Pa)	Relaxation time (s)
3	`G_H0*gg3`	`Tau3`

Cell Periodicity: Shear Strain Loading

1 In the **Physics** toolbar, click **Domains** and choose **Cell Periodicity**.

2 In the **Settings** window for **Cell Periodicity**, type `Cell Periodicity: Shear Strain Loading` in the **Label** text field.

3 Locate the **Periodicity Type** section. From the list, choose **Average strain**.

4 In the ε_{avg} table, enter the following settings:

0	0.5*strainFunction(t)	0
0.5*strainFunction(t)	0	0
0	0	0

Boundary Pair 1

1 In the **Physics** toolbar, click **Attributes** and choose **Boundary Pair**.

2 In the **Settings** window for **Boundary Pair**, locate the **Boundary Selection** section.

3 Click **Clear Selection**.

4 Select Boundaries 15 and 20 only.

Boundary Pair 2

1 Right-click **Boundary Pair 1** and choose **Duplicate**.

2 In the **Settings** window for **Boundary Pair**, locate the **Boundary Selection** section.

3 Click **Clear Selection**.

4 Select Boundaries 16 and 19 only.

Boundary Pair 3

1 Right-click **Boundary Pair 2** and choose **Duplicate**.

2 In the **Settings** window for **Boundary Pair**, locate the **Boundary Selection** section.

3 Click **Clear Selection**.

4 Select Boundaries 17 and 18 only.

ADD STUDY

1 In the **Home** toolbar, click **Add Study** to open the **Add Study** window.

2 Go to the **Add Study** window.

3 Find the **Studies** subsection. In the **Select Study** tree, select **General Studies> Time Dependent**.

4 Find the **Physics interfaces in study** subsection. In the table, clear the **Solve** check box for **Solid Mechanics: Heterogeneous RVE (solid)**.

5 Click **Add Study** in the window toolbar.

6 In the **Home** toolbar, click **Add Study** to close the **Add Study** window.

DEVIATORIC PRONY SERIES PARAMETER ESTIMATION (HOMOGENEOUS RVE)

1 In the **Model Builder** window, click **Study 2**.

2 In the **Settings** window for **Study**, type `Deviatoric Prony Series Parameter Estimation (Homogeneous RVE)` in the **Label** text field.

3 Locate the **Study Settings** section. Clear the **Generate default plots** check box.

To get the homogenized viscoelastic parameters, the initial and final elastic responses of the heterogeneous RVE can be neglected. This means that the interesting time range is from 0.001 s to 10 s.

Step 1: Time Dependent

1 In the **Model Builder** window, under **Deviatoric Prony Series Parameter Estimation (Homogeneous RVE)** click **Step 1: Time Dependent**.

2 In the **Settings** window for **Time Dependent**, locate the **Study Settings** section.

3 In the **Output times** text field, type `10^{range(-3,0.1,1)}`.

Parameter Estimation

1 In the **Study** toolbar, click **Optimization** and choose **Parameter Estimation**.

2 In the **Settings** window for **Parameter Estimation**, locate the **Experimental Data** section.

3 From the **Data source** list, choose **Result table**.

4 From the **Result table** list, choose **Evaluation Group: Shear Stress Response**.

5 Locate the **Column Settings** section. In the table, click to select the cell at row number 2 and column number 1.

6 In the **Model expression** text field, type `comp1.solid2.cp1.savgXY`.

7 In the **Unit** text field, type `N/m^2`.

8 Locate the **Parameters** section. Click **Add** three times.

9 Row by row, select the parameter name in the first column, then set the corresponding initial value, scale, and bounds as follows:

Parameter name	**Initial value**	**Scale**	**Lower bound**	**Upper bound**
gg1 (Deviatoric Prony series parameter of homogenized material, branch 1)	g1	1	0	1
gg2 (Deviatoric Prony series parameter of homogenized material, branch 2)	g2	1	0	1
gg3 (Deviatoric Prony series parameter of homogenized material, branch 3)	g3	1	0	1

10 Locate the **Parameter Estimation Method** section. From the **Method** list, choose **Levenberg-Marquardt**.

Solution 5 (sol5)

1 In the **Study** toolbar, click **Show Default Solver**.

2 In the **Model Builder** window, expand the **Solution 5 (sol5)** node.

3 In the **Model Builder** window, expand the **Deviatoric Prony Series Parameter Estimation (Homogeneous RVE)>Solver Configurations> Solution 5 (sol5)>Optimization Solver 1** node, then click **Time-Dependent Solver 1**.

4 In the **Settings** window for **Time-Dependent Solver**, click to expand the **Time Stepping** section.

5 Select the **Initial step** check box. In the associated text field, type `5e-5`.

6 In the **Study** toolbar, click **Compute**.

RESULTS

Average Shear Stress

1 In the **Home** toolbar, click **Add Plot Group** and choose **1D Plot Group**.

2 In the **Settings** window for **1D Plot Group**, type `Average Shear Stress` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Deviatoric Prony Series Parameter Estimation (Homogeneous RVE)/Solution 5 (sol5)**.

4 Locate the **Plot Settings** section.

5 Select the **y-axis label** check box. In the associated text field, type `Average shear stress (N/m<sup>2</sup>)`.

6 Locate the **Title** section. From the **Title type** list, choose **Manual**.

7 In the **Title** text area, type `Global: Average shear stress (N/m<sup>2</sup>)` .

8 Locate the **Axis** section. Select the **x-axis log scale** check box.

Global 1

1 Right-click **Average Shear Stress** and choose **Global**.

2 In the **Settings** window for **Global**, locate the **y-Axis Data** section.

3 In the table, enter the following settings:

Expression	Unit	Description
solid2.cp1.savgXY	N/m^2	Average stress, XY direction

4 Locate the **Legends** section. From the **Legends** list, choose **Manual**.

5 In the table, enter the following settings:

Legends
Homogenized material model

Global 2

1 Right-click **Global 1** and choose **Duplicate**.

2 In the **Settings** window for **Global**, locate the **Data** section.

3 From the **Dataset** list, choose **Transient Study for Viscoelastic Response (Heterogeneous RVE)/ Parametric Solutions 1 (sol2)**.

4 From the **Parameter selection (para)** list, choose **Last**.

5 From the **Time selection** list, choose **Interpolated**.

6 In the **Times (s)** text field, type `10^{range(-3,0.1,1)}`.

7 Locate the **y-Axis Data** section. In the table, enter the following settings:

Expression	Unit	Description
solid.cp2.savgXY	N/m^2	Average stress, XY direction

8 Click to expand the **Coloring and Style** section. Find the **Line markers** subsection. From the **Marker** list, choose **Point**.

9 Find the **Line style** subsection. From the **Line** list, choose **None**.

10 Locate the **Legends** section. In the table, enter the following settings:

Legends
Heterogeneous RVE

11 In the **Average Shear Stress** toolbar, click **Plot**.

Duplicate the **Viscoelasticity** and **Cell Periodicity** features to set up a normal strain load case in order to compute the homogenized volumetric Prony series parameters. Use the homogenized deviatoric Prony series parameters obtained in the previous optimization study in the new **Viscoelasticity** feature.

SOLID MECHANICS: HOMOGENEOUS RVE (SOLID2)

Viscoelasticity 2

1 In the **Model Builder** window, under **Component 1 (comp1)>Solid Mechanics: Homogeneous RVE (solid2)>Linear Elastic Material 1** right-click **Viscoelasticity 1** and choose **Duplicate**.

2 In the **Settings** window for **Viscoelasticity**, locate the **Viscoelasticity Model** section.

3 From the **Viscoelastic strains** list, choose **Volumetric and deviatoric**.

4 In the table, enter the following settings:

Branch	Bulk modulus (Pa)	Shear modulus (Pa)	Relaxation time (s)
1	K_H0*kg1	G_H0* withsol('sol5', gg1)	Tau1

5 Click **Add**.

6 In the table, enter the following settings:

Branch	Bulk modulus (Pa)	Shear modulus (Pa)	Relaxation time (s)
2	K_H0*kg2	G_H0* withsol('sol5', gg2)	Tau2

7 Click **Add**.

8 In the table, enter the following settings:

Branch	Bulk modulus (Pa)	Shear modulus (Pa)	Relaxation time (s)
3	K_H0*kg3	G_H0* withsol('sol5', gg3)	Tau3

Cell Periodicity: Normal Strain Loading

1 In the **Model Builder** window, right-click **Cell Periodicity: Shear Strain Loading** and choose **Duplicate**.

Apply a unit step in the normal strain.

2 In the **Settings** window for **Cell Periodicity**, type `Cell Periodicity: Normal Strain Loading` in the **Label** text field.

3 Locate the **Periodicity Type** section. In the ε_{avg} table, enter the following settings:

`strainFunction(t)`	`0`	`0`
`0`	`0`	`0`
`0`	`0`	`0`

ADD STUDY

1 In the **Home** toolbar, click **Add Study** to open the **Add Study** window.

2 Go to the **Add Study** window.

3 Find the **Studies** subsection. In the **Select Study** tree, select **General Studies> Time Dependent**.

4 Find the **Physics interfaces in study** subsection. In the table, clear the **Solve** check box for **Solid Mechanics: Heterogeneous RVE (solid)**.

5 Click **Add Study** in the window toolbar.

6 In the **Home** toolbar, click **Add Study** to close the **Add Study** window.

VOLUMETRIC PRONY SERIES PARAMETER ESTIMATION (HOMOGENEOUS RVE)

1 In the **Model Builder** window, click **Study 3**.

2 In the **Settings** window for **Study**, type `Volumetric Prony Series Parameter Estimation (Homogeneous RVE)` in the **Label** text field.

3 Locate the **Study Settings** section. Clear the **Generate default plots** check box.

To get the homogenized viscoelastic parameters, the initial and final elastic responses of the heterogeneous RVE can be neglected. This means that the interesting time range is from 0.001 s to 10 s.

Step 1: Time Dependent

1 In the **Model Builder** window, under **Volumetric Prony Series Parameter Estimation (Homogeneous RVE)** click **Step 1: Time Dependent**.

2 In the **Settings** window for **Time Dependent**, locate the **Study Settings** section.

3 In the **Output times** text field, type `10^{range(-3,0.1,1)}`.

4 Locate the **Physics and Variables Selection** section. Select the **Modify model configuration for study step** check box.

5 In the tree, select **Component 1 (comp1)>Solid Mechanics: Homogeneous RVE (solid2)>Linear Elastic Material 1>Viscoelasticity 1** and **Component 1 (comp1)>Solid Mechanics: Homogeneous RVE (solid2)>Cell Periodicity: Shear Strain Loading**.

6 Right-click and choose **Disable**.

Parameter Estimation

1 In the **Study** toolbar, click **Optimization** and choose **Parameter Estimation**.

2 In the **Settings** window for **Parameter Estimation**, locate the **Experimental Data** section.

3 From the **Data source** list, choose **Result table**.

4 From the **Result table** list, choose **Evaluation Group: Normal Stress Response**.

5 Locate the **Column Settings** section. In the table, click to select the cell at row number 2 and column number 1.

6 In the **Model expression** text field, type `comp1.solid2.cp2.savgXX`.

7 In the **Unit** text field, type `N/m^2`.

8 Locate the **Parameters** section. Click + **Add** three times.

9 Row by row, select the parameter name in the first column, then set the corresponding initial value, scale, and bounds as follows:

Parameter name	**Initial value**	**Scale**	**Lower bound**	**Upper bound**
kg1 (Volumetric Prony series parameter of homogenized material, branch 1)	0.001	1	0	1
kg2 (Volumetric Prony series parameter of homogenized material, branch 2)	0.001	1	0	1
kg3 (Volumetric Prony series parameter of homogenized material, branch 3)	0.001	1	0	1

10 Locate the **Parameter Estimation Method** section. From the **Method** list, choose **Levenberg-Marquardt**.

Solution 6 (sol6)

1 In the **Study** toolbar, click **Show Default Solver**.

2 In the **Model Builder** window, expand the **Solution 6 (sol6)** node.

3 In the **Model Builder** window, expand the **Volumetric Prony Series Parameter Estimation (Homogeneous RVE)>Solver Configurations>Solution 6 (sol6)>Optimization Solver 1** node, then click **Time-Dependent Solver 1**.

4 In the **Settings** window for **Time-Dependent Solver**, locate the **Time Stepping** section.

5 Select the **Initial step** check box. In the associated text field, type `5e-5`.

6 In the **Study** toolbar, click **Compute**.

RESULTS

Average Normal Stress

1 In the **Home** toolbar, click **Add Plot Group** and choose **1D Plot Group**.

2 In the **Settings** window for **1D Plot Group**, type `Average Normal Stress` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Volumetric Prony Series Parameter Estimation (Homogeneous RVE)/Solution 6 (sol6)**.

4 Locate the **Plot Settings** section.

5 Select the **y-axis label** check box. In the associated text field, type `Average shear stress (N/m<sup>2</sup>)`.

6 Locate the **Title** section. From the **Title type** list, choose **Manual**.

7 In the **Title** text area, type `Global: Average normal stress (N/m<sup>2</sup>)` .

8 Locate the **Axis** section. Select the **x-axis log scale** check box.

Global 1

1 Right-click **Average Normal Stress** and choose **Global**.

2 In the **Settings** window for **Global**, locate the **y-Axis Data** section.

3 In the table, enter the following settings:

Expression	Unit	Description
solid2.cp2.savgXX	N/m^2	Average stress, XX direction

4 Locate the **Legends** section. From the **Legends** list, choose **Manual**.

5 In the table, enter the following settings:

Legends
`Homogenized material model`

Global 2

1 Right-click **Global 1** and choose **Duplicate**.

2 In the **Settings** window for **Global**, locate the **Data** section.

3 From the **Dataset** list, choose **Transient Study for Viscoelastic Response (Heterogeneous RVE)/ Parametric Solutions 1 (sol2)**.

4 From the **Parameter selection (para)** list, choose **First**.

5 From the **Time selection** list, choose **Interpolated**.

6 In the **Times (s)** text field, type `10^{range(-3,0.1,1)}`.

7 Locate the **y-Axis Data** section. In the table, enter the following settings:

Expression	**Unit**	**Description**
`solid.cp2.savgXX`	`N/m^2`	`Average stress, XX direction`

8 Locate the **Coloring and Style** section. Find the **Line markers** subsection. From the **Marker** list, choose **Point**.

9 Find the **Line style** subsection. From the **Line** list, choose **None**.

10 Locate the **Legends** section. In the table, enter the following settings:

Legends
`Heterogeneous RVE`

11 In the **Average Normal Stress** toolbar, click **Plot**.

Evaluation Group: Homogenized Prony Series Parameters

1 In the **Results** toolbar, click **Evaluation Group**.

2 In the **Settings** window for **Evaluation Group**, type `Evaluation Group: Homogenized Prony Series Parameters` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Deviatoric Prony Series Parameter Estimation (Homogeneous RVE)/Solution 5 (sol5)**.

4 From the **Time selection** list, choose **First**.

Global Evaluation 1

1 Right-click **Evaluation Group: Homogenized Prony Series Parameters** and choose **Global Evaluation**.

2 In the **Settings** window for **Global Evaluation**, locate the **Expressions** section.

3 In the table, enter the following settings:

Expression	Unit	Description
gg1		Deviatoric Prony series parameter of homogenized material, branch 1
gg2		Deviatoric Prony series parameter of homogenized material, branch 2
gg3		Deviatoric Prony series parameter of homogenized material, branch 3

Global Evaluation 2

1 Right-click **Global Evaluation 1** and choose **Duplicate**.

2 In the **Settings** window for **Global Evaluation**, locate the **Data** section.

3 From the **Dataset** list, choose **Volumetric Prony Series Parameter Estimation (Homogeneous RVE)/Solution 6 (sol6)**.

4 From the **Time selection** list, choose **First**.

5 Locate the **Expressions** section. In the table, enter the following settings:

Expression	Unit	Description
kg1		Volumetric Prony series parameter of homogenized material, branch 1
kg2		Volumetric Prony series parameter of homogenized material, branch 2
kg3		Volumetric Prony series parameter of homogenized material, branch 3

Evaluation Group: Homogenized Prony Series Parameters

1 In the **Model Builder** window, click **Evaluation Group: Homogenized Prony Series Parameters**.

2 In the **Settings** window for **Evaluation Group**, locate the **Transformation** section.

3 Select the **Transpose** check box.

4 Locate the **Format** section. From the **Include parameters** list, choose **Off**.

5 In the **Evaluation Group: Homogenized Prony Series Parameters** toolbar, click **Evaluate**.

Average Normal Stress, Average Shear Stress, Evaluation Group: Homogenized Prony Series Parameters

1 In the **Model Builder** window, under **Results**, Ctrl-click to select **Average Shear Stress**, **Average Normal Stress**, and **Evaluation Group: Homogenized Prony Series Parameters**.
2 Right-click and choose **Group**.

Homogeneous RVE

In the **Settings** window for **Group**, type `Homogeneous RVE` in the **Label** text field.