Micromechanical Model of a Particulate Composite

Model created in COMSOL Multiphysics 6.4

Introduction

In this example, a simplified micromechanical model of a particulate composite is analyzed. A repeating unit cell (RUC) 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 material 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 least-squares parameter estimation.

The following considerations are important for the analysis:

- In the particulate composite, only the matrix material is assumed to exhibit viscoelastic behavior. This is a realistic assumption for, for example, polymer matrix composites. Polymer-like resin is typically viscoelastic, 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 viscoelastic parameters from the particulate composite, two different stress responses from a unit cell are required:
  - The shear stress response, which only activates the deviatoric part of the homogeneous viscoelastic model.
  - The normal stress response, which activates both the deviatoric and the volumetric part of the homogeneous viscoelastic model.
- 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 RUC having a spherical particle embedded in the center of the matrix is shown in Figure 1.

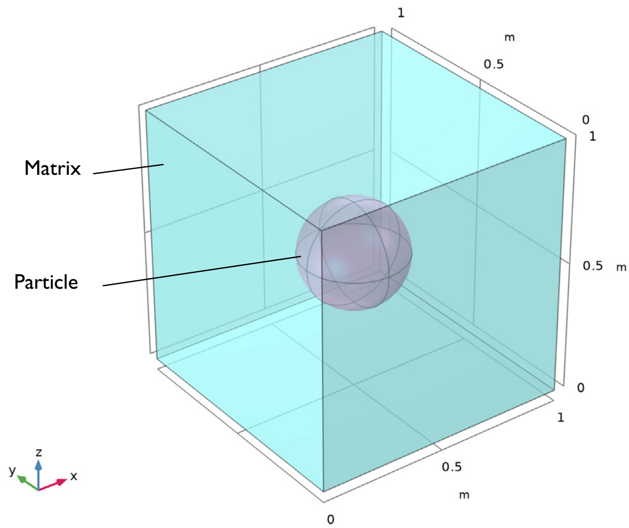

Figure 1: Geometry of the unit cell, consisting of a spherical particle embedded in epoxy resin.

Particle and Matrix Properties

The linear 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 is the relative weight and τ_k is the relaxation time constant of the spring–dashpot pair in branch k. 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 of the viscoelastic matrix 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. As expected, the stresses in the constituents computed in the viscoelastic study are in good agreement with those computed in the elastic study.

The variation in average normal and shear stress with time for the heterogeneous RUC 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 RUC and the equivalent, homogenized material are shown in Figure 7 and Figure 8, respectively. The relative weights for the deviatoric and volumetric parts obtained from the parameter estimation studies 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	0.00957	0.01 s
2	0.04970	0.1 s
3	0.07928	1 s

TABLE 5: HOMOGENIZED VISCOELASTIC VOLUMETRIC PROPERTIES.

Branch	**Relative Weight**	**Relaxation Time Constant**
1	8.941E-4	0.01 s
2	0.00427	0.1 s
3	0.00683	1 s

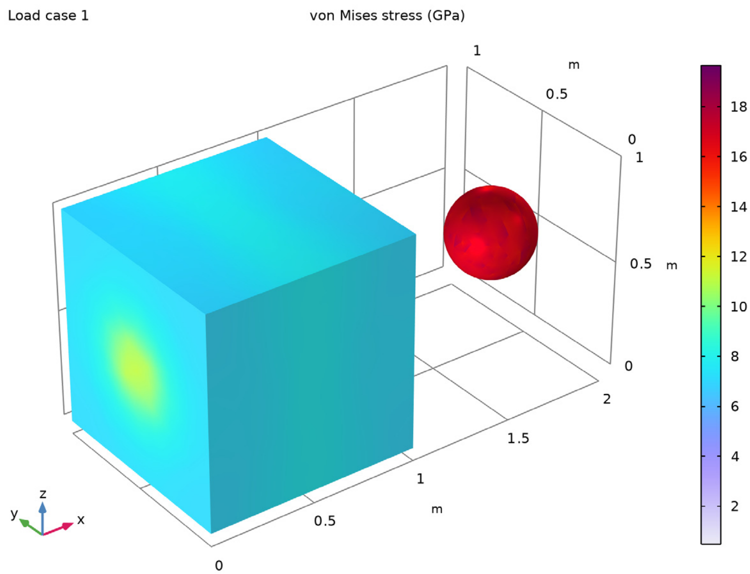

Figure 2: von Mises stress in matrix and particle due to axial loading (elastic conditions).

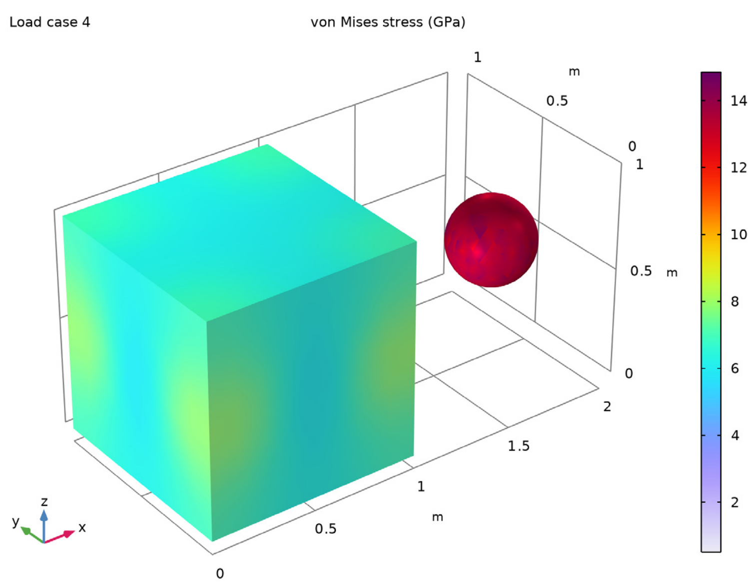

Figure 3: von Mises stress in matrix and particle due to shear loading (elastic conditions).

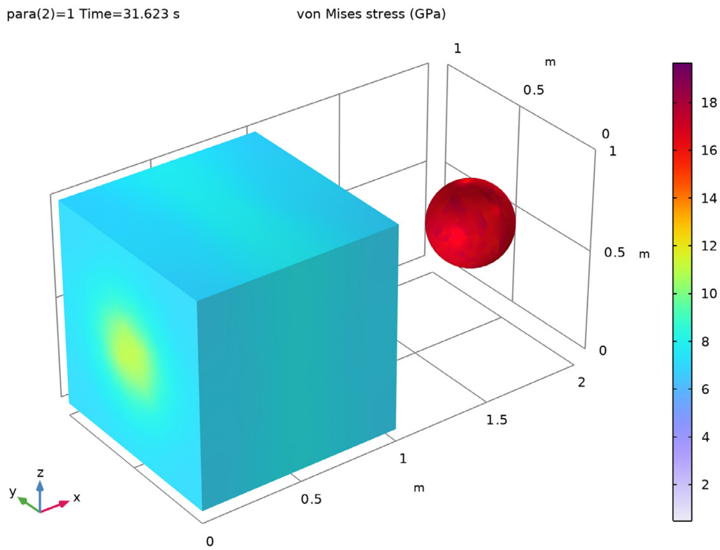

Figure 4: von Mises stress in particle and matrix under axial loading (viscoelasticity in matrix).

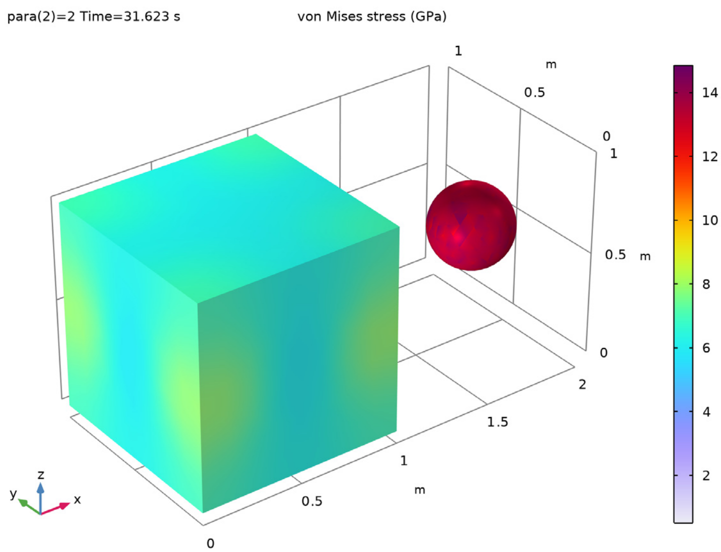

Figure 5: von Mises stress in particle and matrix under shear loading (viscoelasticity in matrix).

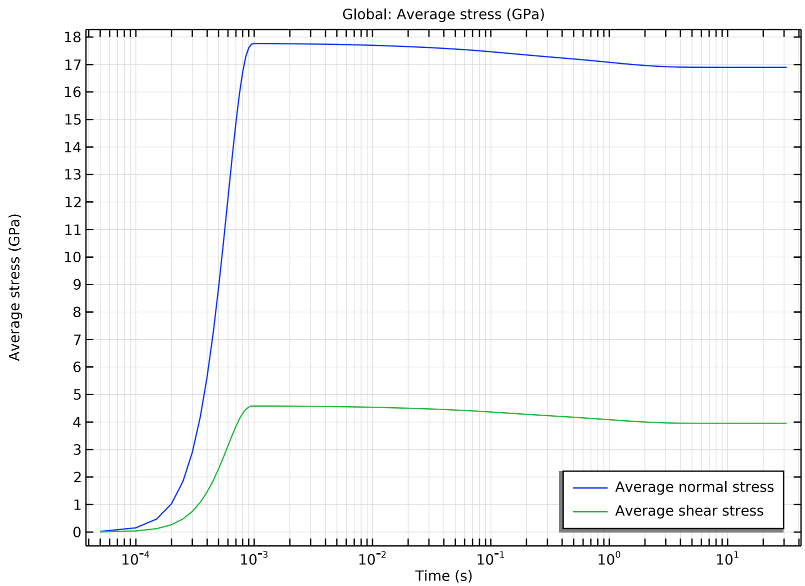

Figure 6: Average viscoelastic normal and shear stress in the heterogeneous RUC.

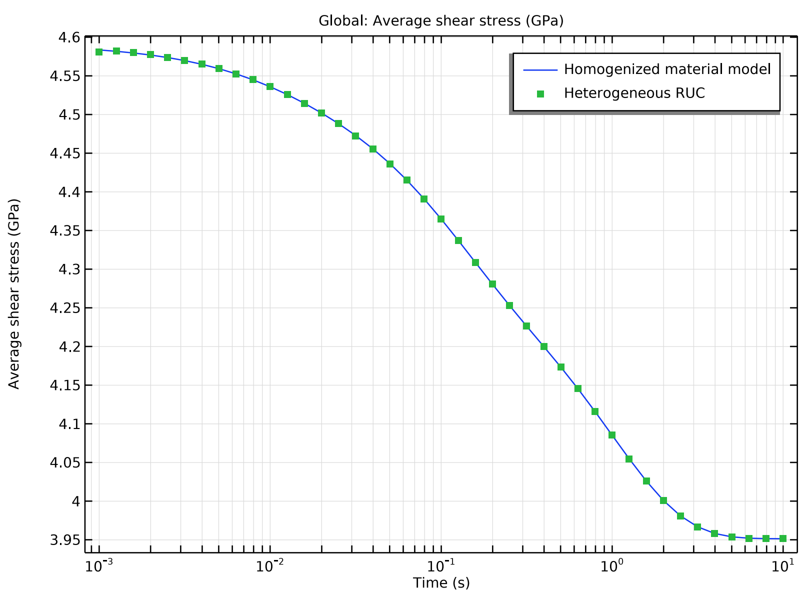

Figure 7: Average viscoelastic shear stress for the composite and the homogenized material.

.

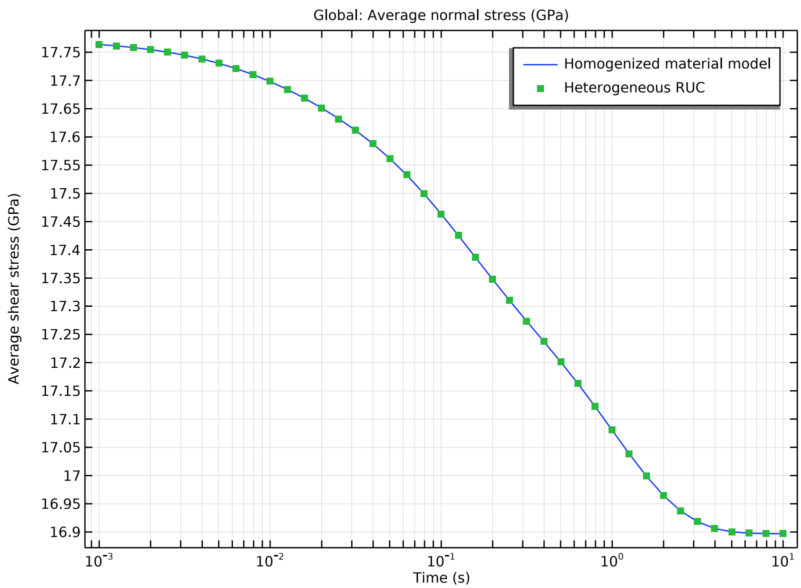

Figure 8: Average viscoelastic normal stress for the heterogeneous RUC and the equivalent homogenized material.

Notes About the COMSOL Implementation

- Micromechanical analyses RUCs 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** subnode.

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** > **Unit Cells and RVEs** > **Particulate Composites** > **particulate_primitive_cubic** in the tree.

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

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

4 Click **OK**.

Create one repeating unit cell (RUC) geometry for the heterogeneous material and one for the homogenized material.

GEOMETRY 1

Heterogeneous RUC

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

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

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	Cell width
dm	L	1 m	Cell depth
hm	L	1 m	Cell height

Homogeneous RUC

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

2 In the **Settings** window for **Block**, type `Homogeneous RUC` 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 RUC)**.

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 RUC)**.

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 RUC

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 RUC` in the **Label** text field.

3 Locate the **Domain Selection** section. From the **Selection** list, choose **All (Heterogeneous RUC)**.

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 RUC (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** checkbox.

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 Settings** section. From the **Boundary conditions** list, choose **Average strain**.

4 Locate the **Effective Properties** section. Select the **Compute elasticity matrix, standard notation** checkbox.

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 RUC)**.

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 RUC)**.

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 RUC)**.

Cell Periodicity for Elastic Properties

With the **Average strain** option in the **Cell Periodicity** node, 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 **Automated Model Setup** in the upper-right corner of the **Periodicity Settings** section. From the menu, choose **Create Load Groups and Study**.

MESH 1

Free Triangular 1

1 In the **Mesh** toolbar, click **More Generators** 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** checkbox.

7 Select the **Minimum element size** checkbox.

8 Select the **Maximum element growth rate** checkbox.

9 Select the **Curvature factor** checkbox.

10 Select the **Resolution of narrow regions** checkbox.

11 Click **Build Selected**.

Free Triangular 2

1 In the **Mesh** toolbar, click **More Generators** 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** checkbox. In the associated text field, type `0.07`.

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

9 Select the **Maximum element growth rate** checkbox.

10 Select the **Curvature factor** checkbox.

11 Select the **Resolution of narrow regions** checkbox.

12 Click **Build Selected**.

Identical Mesh 1

1 In the **Mesh** toolbar, click **More Attributes** and choose **Identical Mesh**.

2 In the **Settings** window for **Identical Mesh**, locate the **First Entity Group** section.

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

4 Locate the **Second Entity Group** section. From the **Selection** list, choose **Pair 1, Destination (Heterogeneous RUC)**.

Identical Mesh 2

1 In the **Mesh** toolbar, click **More Attributes** and choose **Identical Mesh**.

2 In the **Settings** window for **Identical Mesh**, locate the **First Entity Group** section.

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

4 Locate the **Second Entity Group** section. From the **Selection** list, choose **Pair 2, Destination (Heterogeneous RUC)**.

Identical Mesh 3

1 In the **Mesh** toolbar, click **More Attributes** and choose **Identical Mesh**.

2 In the **Settings** window for **Identical Mesh**, locate the **First Entity Group** section.

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

4 Locate the **Second Entity Group** section. From the **Selection** list, choose **Pair 3, Destination (Heterogeneous RUC)**.

5 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 RUC)**.

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. Click the **Custom** button.

5 Locate the **Element Size Parameters** section. Select the **Maximum element size** checkbox.

6 Select the **Minimum element size** checkbox.

7 Select the **Maximum element growth rate** checkbox.

8 Select the **Curvature factor** checkbox. In the associated text field, type `0.4`.

9 Select the **Resolution of narrow regions** checkbox.

10 Click **Build All**.

Mapped 1

1 In the **Mesh** toolbar, click **More Generators** 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 RUC)

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 RUC)` in the **Label** text field.

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

Set default units for result presentation.

RESULTS

Preferred Units 1

1 In the **Results** toolbar, click **Configurations** and choose **Preferred Units**.

2 In the **Settings** window for **Preferred Units**, locate the **Units** section.

3 Click **Add Physical Quantity**.

4 In the **Physical Quantity** dialog, select **Solid Mechanics** > **Stress tensor (N/m^2)** in the tree.

5 Click **OK**.

6 In the **Settings** window for **Preferred Units**, locate the **Units** section.

7 In the table, enter the following settings:

Quantity	Unit	Preferred unit
Stress tensor	N/m^2	GPa

8 Click **Apply**.

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 RUC)**.

5 Select the **Apply to dataset edges** checkbox.

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

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

8 Locate the **Plot Settings** section. Clear the **Plot dataset edges** checkbox.

9 Click to expand the **Plot Array** section. From the **Array type** list, choose **Linear**.

Selection 1

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

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

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

4 From the **Selection** list, choose **Matrix (Heterogeneous RUC)**.

Deformation

In the **Model Builder** window, 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** checkbox.

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 RUC)**.

SOLID MECHANICS: HETEROGENEOUS RUC (SOLID)

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

Cell Periodicity for Elastic Properties

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

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

GLOBAL DEFINITIONS

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

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 RUC

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 RUC` 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 RUC (SOLID)

Linear Elastic Material 1

In the **Model Builder** window, under **Component 1 (comp1)** > **Solid Mechanics: Heterogeneous RUC (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 RUC)**.
4. Locate the **Viscoelasticity Model** section. Click + **Add** twice.

5 In the table, enter the following settings:

Branch	**Shear modulus (Pa)**	**Relaxation time (s)**
1	G_m0*g1	Tau1
2	G_m0*g2	Tau2
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 **Effective Properties** section. Clear the **Compute elasticity matrix, standard notation** checkbox.

4 Locate the **Periodicity Settings** section. Specify the ε_{avg} matrix as

(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 the **Add Study** button in the window toolbar.

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

TRANSIENT STUDY FOR VISCOELASTIC RESPONSE (HETEROGENEOUS RUC)

In the **Settings** window for **Study**, type `Transient Study for Viscoelastic Response (Heterogeneous RUC)` 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** checkbox.

5 In the tree, select **Component 1 (comp1)** > **Solid Mechanics: Heterogeneous RUC (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** checkbox. In the associated text field, type `5e-7`.

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

RESULTS

Stress, Viscoelastic Response

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

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

2 Locate the **Selection** section. From the **Geometric entity level** list, choose **Domain**.

3 From the **Selection** list, choose **All (Heterogeneous RUC)**.

4 Select the **Apply to dataset edges** checkbox.

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

6 In the **Title** text area, type `von Mises stress (GPa)`.

7 Locate the **Plot Settings** section. Clear the **Plot dataset edges** checkbox.

8 Locate the **Plot Array** section. From the **Array type** list, choose **Linear**.

Selection 1

1 In the **Model Builder** window, expand the **Stress, Viscoelastic Response** node.

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

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

4 From the **Selection** list, choose **Matrix (Heterogeneous RUC)**.

Deformation

In the **Model Builder** window, 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** checkbox.

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 RUC)**.

Average Normal and Shear Stresses

1 In the **Results** toolbar, click **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 RUC)/ Parametric Solutions 1 (sol2)**.

4 Locate the **Plot Settings** section.

5 Select the **y-axis label** checkbox. In the associated text field, type `Average stress (GPa)`.

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

7 In the **Title** text area, type `Global: Average stress (GPa)`.

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

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 RUC)/ 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	GPa	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	GPa	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 RUC)/ 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`	`GPa`	`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 RUC)/ 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	GPa	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 RUC

In the **Settings** window for **Group**, type `Heterogeneous RUC` 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

Name	Expression	Value	Description
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 the **Add to Component 1** button in the window toolbar.

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

SOLID MECHANICS: HOMOGENEOUS RUC

1 In the **Settings** window for **Solid Mechanics**, type `Solid Mechanics: Homogeneous RUC` 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 RUC (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 Click **Add** twice.

4 In the table, enter the following settings:

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

Branch	Shear modulus (Pa)	Relaxation time (s)
2	G_H0*gg2	Tau2
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 Settings** section. From the **Boundary conditions** list, choose **Average strain**.

4 Specify the ε_{avg} matrix as

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.

COMPONENT 1 (COMP1)

Add a **Parameter Estimation** node with a **Least-Squares Objective** to set up the inverse problem for the deviatoric parameters.

Shear Stress Response

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

2 In the **Settings** window for **Least-Squares Objective**, type `Shear Stress Response` in the **Label** text field.

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

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

5 Locate the **Data Column Settings** section. In the **Model expression** text field, type `comp1.solid2.cp1.savgXY`.

6 In the **Column name** text field, type `shear_stress`.

7 In the **Unit** text field, type `GPa`.

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 the **Add Study** button in the window toolbar.

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

DEVIATORIC PRONY SERIES PARAMETER ESTIMATION (HOMOGENEOUS RUC)

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

2 Locate the **Study Settings** section. Clear the **Generate default plots** checkbox.

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 **Selected Least-Squares objectives**.

4 Locate the **Objective Function** section. In the table, select the **Active** checkbox for **Shear Stress Response**.

5 Locate the **Estimated Parameters** section. Click **Add** three times.

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

Parameter	Initial value	Scale	Lower bound	Upper bound	Unit
gg1 (Deviatoric Prony series parameter of homogenized material, branch 1)	g1	0.1	0	1	
gg2 (Deviatoric Prony series parameter of homogenized material, branch 2)	g2	0.1	0	1	
gg3 (Deviatoric Prony series parameter of homogenized material, branch 3)	g3	0.1	0	1	

7 Locate the **Parameter Estimation Method** section. From the **Method** list, choose **Levenberg–Marquardt**.

8 From the **Least-squares time/parameter list method** list, choose **Use only least-squares data points**.

Step 1: Time Dependent

Disable the Solid Mechanics: Heterogeneous RUC interface.

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

2 In the **Settings** window for **Time Dependent**, locate the **Physics and Variables Selection** section.

3 Select the **Modify model configuration for study step** checkbox.

4 In the tree, select **Component 1 (comp1)** > **Solid Mechanics: Heterogeneous RUC (solid)**.

5 Click **Disable in Model**.

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 RUC)** > **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 From the **Steps taken by solver** list, choose **Strict**.

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

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

RESULTS

Average Shear Stress

1 In the **Results** toolbar, click **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 RUC)/Solution 5 (sol5)**.

4 Locate the **Plot Settings** section.

5 Select the **y-axis label** checkbox. In the associated text field, type `Average shear stress (GPa)`.

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

7 In the **Title** text area, type `Global: Average shear stress (GPa)`.

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

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	GPa	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

Table Graph 1

1 In the **Model Builder** window, right-click **Average Shear Stress** and choose **Table Graph**.

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

3 From the **Source** list, choose **Evaluation group**.

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

5 Locate the **Coloring and Style** section. Find the **Line style** subsection. From the **Line** list, choose **None**.

6 Find the **Line markers** subsection. From the **Marker** list, choose **Point**.

7 Click to expand the **Legends** section. Select the **Show legends** checkbox.

8 From the **Legends** list, choose **Manual**.

9 In the table, enter the following settings:

Legends
Heterogeneous RUC

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

SOLID MECHANICS: HOMOGENEOUS RUC (SOLID2)

Duplicate the **Viscoelasticity** and **Cell Periodicity** nodes 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** node.

Viscoelasticity 2

1 In the **Model Builder** window, under **Component 1 (comp1)** > **Solid Mechanics: Homogeneous RUC (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 Click **Add** twice.

5 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

Branch	Bulk modulus (Pa)	Shear modulus (Pa)	Relaxation time (s)
2	`K_H0*kg2`	`G_H0* withsol('sol5', gg2)`	`Tau2`
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 Settings** section. Specify the ε_{avg} matrix as

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

PARAMETER ESTIMATION

Add a **Least-Squares Objective** node for the normal stress load case.

Normal Stress Response

1 In the **Parameter Estimation** toolbar, click **Least-Squares Objective**.

2 In the **Settings** window for **Least-Squares Objective**, type `Normal Stress Response` in the **Label** text field.

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

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

5 Locate the **Data Column Settings** section. In the **Model expression** text field, type `comp1.solid2.cp2.savgXX`.

6 In the **Column name** text field, type `normal_stress`.

7 In the **Unit** text field, type `GPa`.

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 the **Add Study** button in the window toolbar.

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

VOLUMETRIC PRONY SERIES PARAMETER ESTIMATION (HOMOGENEOUS RUC)

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

2 Locate the **Study Settings** section. Clear the **Generate default plots** checkbox.

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 **Selected Least-Squares objectives**.

4 Locate the **Objective Function** section. In the table, enter the following settings:

Objective functions from components	Active
Shear Stress Response	
Normal Stress Response	√

5 Locate the **Estimated Parameters** section. Click **Add** three times.

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

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

7 Locate the **Parameter Estimation Method** section. From the **Method** list, choose **Levenberg–Marquardt**.

8 From the **Least-squares time/parameter list method** list, choose **Use only least-squares data points**.

Step 1: Time Dependent

Make sure to disable the Solid Mechanics: Heterogeneous RUC interface as well as the **Viscoelasticity** and **Cell Periodicity** nodes that you set up for the shear strain load case, since they only apply for the homogenized deviatoric response.

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

2 In the **Settings** window for **Time Dependent**, locate the **Physics and Variables Selection** section.

3 Select the **Modify model configuration for study step** checkbox.

4 In the tree, select **Component 1 (comp1)** > **Solid Mechanics: Heterogeneous RUC (solid)**.

5 Right-click and choose **Disable in Model**.

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

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

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 RUC)** > **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 From the **Steps taken by solver** list, choose **Strict**.

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

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

RESULTS

Average Normal Stress

1 In the **Results** toolbar, click **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 RUC)/Solution 6 (sol6)**.

4 Locate the **Plot Settings** section.

5 Select the **y-axis label** checkbox. In the associated text field, type `Average shear stress (GPa)`.

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

7 In the **Title** text area, type `Global: Average normal stress (GPa)`.

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

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`	`GPa`	`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`

Table Graph 1

1 In the **Model Builder** window, right-click **Average Normal Stress** and choose **Table Graph**.

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

3 From the **Source** list, choose **Evaluation group**.

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

5 Locate the **Coloring and Style** section. Find the **Line style** subsection. From the **Line** list, choose **None**.

6 Find the **Line markers** subsection. From the **Marker** list, choose **Point**.

7 Locate the **Legends** section. Select the **Show legends** checkbox.

8 From the **Legends** list, choose **Manual**.

9 In the table, enter the following settings:

Legends
`Heterogeneous RUC`

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

RESULT TEMPLATES

Collect the calibrated Prony series parameters of the homogenized material in an **Evaluation Group**.

1 In the **Results** toolbar, click **Result Templates** to open the **Result Templates** window.

2 Go to the **Result Templates** window.

3 In the tree, select **Deviatoric Prony Series Parameter Estimation (Homogeneous RUC)/ Solution 5 (sol5)** > **Solid Mechanics: Homogeneous RUC** > **Estimated Parameters (std2)**.

4 Click the **Add Result Template** button in the window toolbar.

5 In the **Results** toolbar, click **Result Templates** to close the **Result Templates** window.

RESULTS

Homogenized Prony Series Parameters

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

Global Evaluation 2

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

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

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

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

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`

6 In the **Homogenized Prony Series Parameters** toolbar, click **Evaluate**.

Average Normal Stress, Average Shear Stress, Homogenized Prony Series Parameters

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

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

Homogeneous RUC

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