Hysteresis in Soil Using the Small-Strain Overlay Model

Introduction

The majority of soil types have a high stiffness at small strains in the elastic regime while it decreases monotonically with increasing strain. Soils also displays a hysteresis effect when subjected to cyclic loads. The Small Strain Overlay model presented in Ref. 1 captures the effect of high stiffness at low strain as well as the hysteric behavior under cyclic loading. The formulation allows stiffness degradation with an increase in shear strain, and the full recovery of stiffness at load reversal.

In this example, cyclic tensile and shear tests show the stiffness degradation and the hysteresis effect with the small strain overlay model. The cyclic tensile test mimic the cyclic triaxial loading without any isotropic compression step.

Model Definition

A rectangular soil specimen of 5 cm in width and 10 cm in height is used for both tests. The specimen is represented by a 2D geometry.

SOIL PROPERTIES

Properties for dense sand are presented in Table 1 as taken from Ref. 1.

TABLE 1: MATERIAL PROPERTIES FOR THE SOIL MODEL.

Property	Variable	Value
Poisson's ratio	ν	0.2
Density	ρ	2400 kg/m^3
Initial shear modulus	G_0	185 MPa
Reference shear strain	γ_s	0.0002

CONSTRAINTS AND LOADS

- For the tensile test, the vertical left and bottom boundaries are constrained in the normal direction. On the top boundary, a prescribed displacement is applied in the normal direction.
- For the shear test, the vertical left and bottom boundaries are constrained in the tangential direction. On the vertical right and top boundary, a prescribed displacement is applied in the tangential direction.

Results and Discussion

Note that for consistency with the geomechanics sign convention, compressive stress and strain are plotted along the positive axis in all figures, while tensile stress and strain are plotted along the negative axis.

For the cyclic triaxial test, the axial stress versus axial strain is shown in Figure 1, while the variation of Young's modulus versus axial strain is shown in Figure 2. The small-strain overlay model captures two important aspects of the behavior of soils:

- Monotonic degradation of stiffness with increasing strain.
- Hysteresis in the cyclic loading.

From Figure 1 it is clear that the model satisfies two basic rules of Masing:

- The shear modulus at the start of unloading is equal to the initial modulus of the primary loading curve.
- The shape of the unloading and reloading curve should be the same as the initial loading curve except the scale must be enlarged by the factor of two.

Note that a log scale is used for the axial strain in the Figure 2, which is why the first parametric steps corresponding to zero strain are excluded. The stiffness at the beginning of the unloading curve is the same as the initial stiffness at the beginning of the primary loading.

The hysteresis effect in the cyclic shear test is shown in Figure 3. Figure 4 shows a reduction in the shear stiffness with increasing shear strain. From both figures it can be observed that the stiffness at the start of the unloading cycle is regained and that both Massing rules are satisfied.

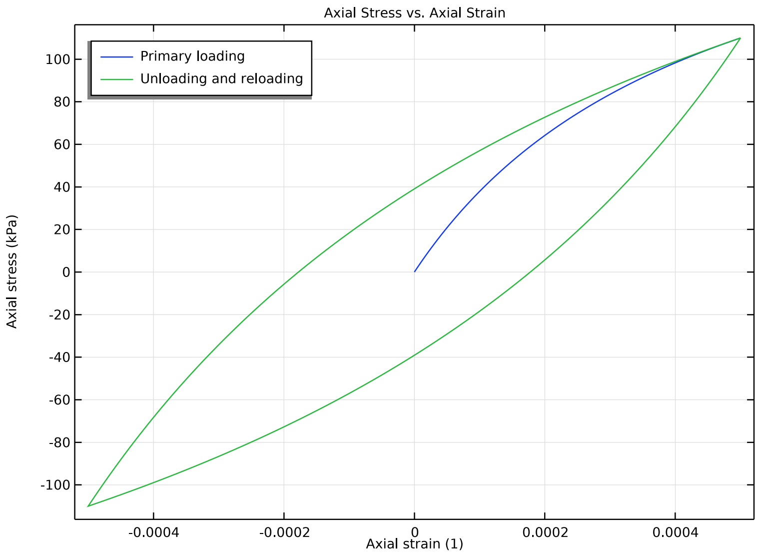

Figure 1: Axial stress versus axial strain for the cyclic tensile test.

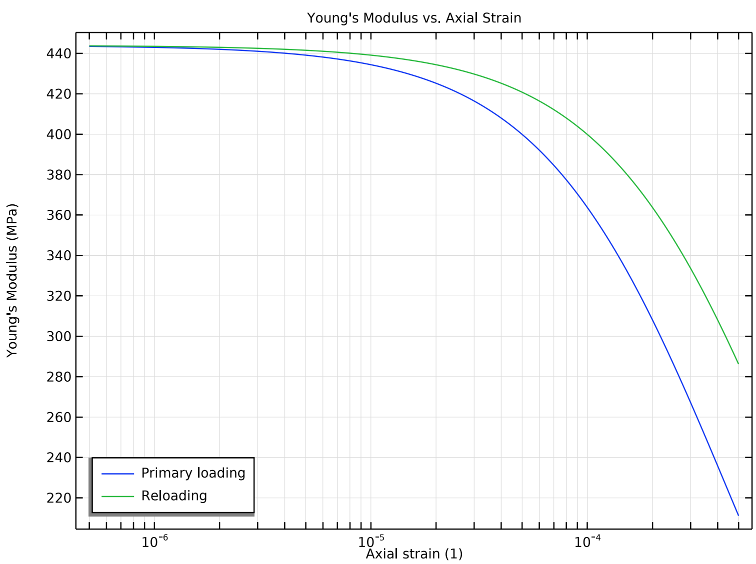

Figure 2: Young's modulus versus axial strain for the cyclic tensile test.

.

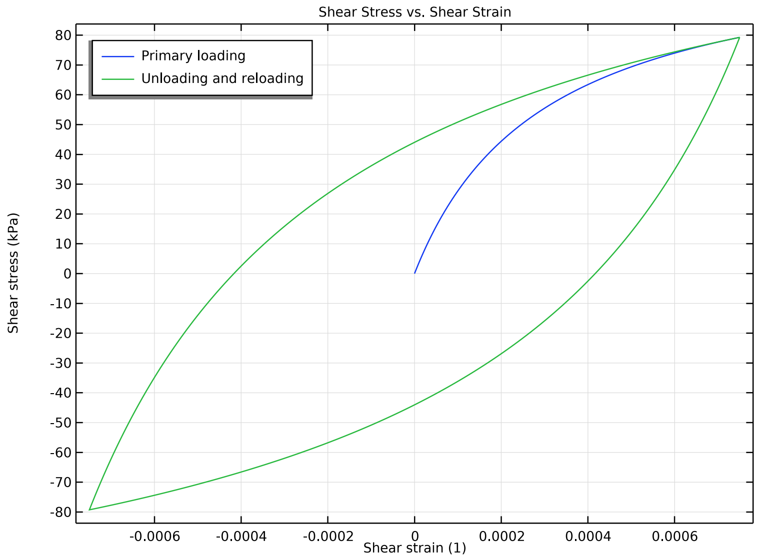

Figure 3: Shear stress versus shear strain in the cyclic shear test.

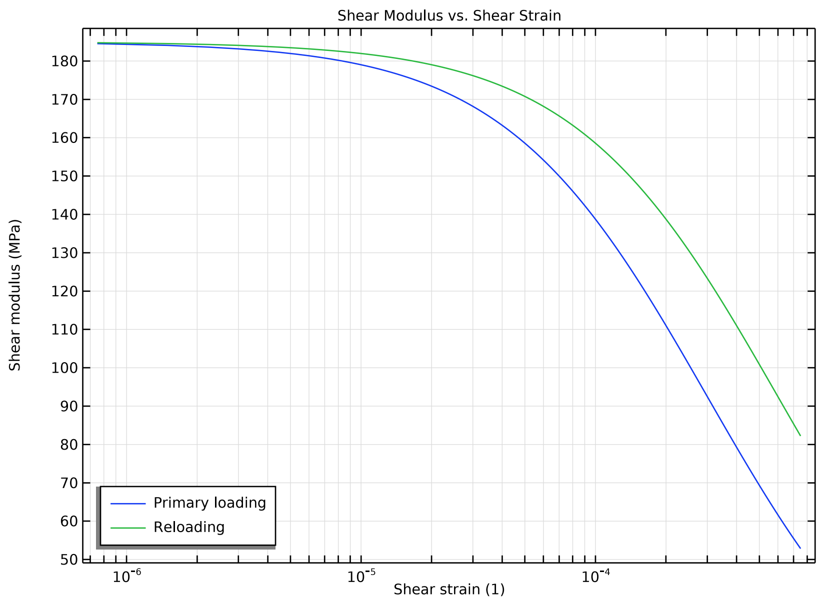

Figure 4: Shear modulus versus shear strain in the cyclic shear test.

Notes About the COMSOL Implementation

For cyclic loading, the load reversal points in the small-strain overlay model are found automatically based on the increments of the deviatoric strain tensor. However, in some load cases, the automatic algorithm is not effective. In such scenarios the user can set **Load reversal points** to **User defined**. With this option, a set of load reversal points can be specified in terms of the parameter for a parametric study or the time variable for a transient study.

Reference

1. T. Benz, P. A. Vermeer, and R. Schwab, "A small-strain overlay model," *Int. J. Numer. Anal. Methods Geomech.*, vol. 33, pp. 25–44, 2009.

Application Library path: `Geomechanics_Module/Verification_Examples/hysteresis_small_strain_overlay`

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 **2D**.

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

3 Click **Add**.

4 Click **Study**.

5 In the **Select Study** tree, select **General Studies>Stationary**.

6 Click **Done**.

GLOBAL DEFINITIONS

Parameters 1

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

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

3 In the table, enter the following settings:

Name	Expression	Value	Description
para	0	0	Parameter
Nu	0.2	0.2	Poisson's ratio
G_0	185[MPa]	1.85E8 Pa	Initial shear modulus
gamma_a	2e-4	2E-4	Threshold shear strain
a	0.385	0.385	Material parameter
gamma_ref	gamma_a/a	5.1948E-4	Reference shear strain

DEFINITIONS

Interpolation 1 (int1)

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

2 In the **Settings** window for **Interpolation**, locate the **Definition** section.

3 In the **Function name** text field, type `appliedDisp`.

4 In the table, enter the following settings:

t	f(t)
0	0
1	5e-3
2	0
3	-5e-3
4	0
5	5e-3

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

Function	Unit
appliedDisp	cm

6 In the **Argument** table, enter the following settings:

Argument	Unit
t	1

GEOMETRY 1

Rectangle 1 (r1)

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

2 In the **Settings** window for **Rectangle**, locate the **Size and Shape** section.

3 In the **Width** text field, type `5[cm]`.

4 In the **Height** text field, type `10[cm]`.

5 Click **Build Selected**.

SOLID MECHANICS (SOLID)

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Solid Mechanics (solid)**.

2 In the **Settings** window for **Solid Mechanics**, click to expand the **Discretization** section.

3 From the **Displacement field** list, choose **Linear**.

Nonlinear Elastic Material 1

1 In the **Physics** toolbar, click **Domains** and choose **Nonlinear Elastic Material**.

2 Select Domain 1 only.

3 In the **Settings** window for **Nonlinear Elastic Material**, locate the **Nonlinear Elastic Material** section.

4 From the **Material model** list, choose **Small strain overlay**.

Roller 1

1 In the **Physics** toolbar, click **Boundaries** and choose **Roller**.

2 Select Boundaries 1 and 2 only.

Prescribed Displacement 1

1 In the **Physics** toolbar, click **Boundaries** and choose **Prescribed Displacement**.

2 Select Boundary 3 only.

3 In the **Settings** window for **Prescribed Displacement**, locate the **Prescribed Displacement** section.

4 From the **Displacement in y direction** list, choose **Prescribed**.

5 In the u_{0y} text field, type `-appliedDisp(para)`.

Prescribed Displacement 2

1 In the **Physics** toolbar, click **Boundaries** and choose **Prescribed Displacement**.

2 Select Boundary 4 only.

3 In the **Settings** window for **Prescribed Displacement**, locate the **Prescribed Displacement** section.

4 From the **Displacement in y direction** list, choose **Prescribed**.

5 In the u_{0y} text field, type `appliedDisp(para)`.

6 Right-click **Prescribed Displacement 2** and choose **Duplicate**.

Prescribed Displacement 3

1 In the **Model Builder** window, click **Prescribed Displacement 3**.

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

3 Click **Clear Selection**.

4 Select Boundary 1 only.

5 Locate the **Prescribed Displacement** section. In the u_{0y} text field, type `0`.

Prescribed Displacement 4

1. In the **Physics** toolbar, click **Boundaries** and choose **Prescribed Displacement**.
2. Select Boundary 3 only.
3. In the **Settings** window for **Prescribed Displacement**, locate the **Prescribed Displacement** section.
4. From the **Displacement in x direction** list, choose **Prescribed**.
5. In the u_{0x} text field, type `appliedDisp(para)`.
6. Right-click **Prescribed Displacement 4** and choose **Duplicate**.

Prescribed Displacement 5

1. In the **Model Builder** window, click **Prescribed Displacement 5**.
2. In the **Settings** window for **Prescribed Displacement**, locate the **Boundary Selection** section.
3. Click **Clear Selection**.
4. Select Boundary 2 only.
5. Locate the **Prescribed Displacement** section. In the u_{0x} text field, type `0`.

Prescribed Displacement 1, Roller 1

1. In the **Model Builder** window, under **Component 1 (comp1)>Solid Mechanics (solid)**, Ctrl-click to select **Roller 1** and **Prescribed Displacement 1**.
2. Right-click and choose **Group**.

Cyclic Triaxial Loading

In the **Settings** window for **Group**, type `Cyclic Triaxial Loading` in the **Label** text field.

Prescribed Displacement 2, Prescribed Displacement 3, Prescribed Displacement 4, Prescribed Displacement 5

1. In the **Model Builder** window, under **Component 1 (comp1)>Solid Mechanics (solid)**, Ctrl-click to select **Prescribed Displacement 2**, **Prescribed Displacement 3**, **Prescribed Displacement 4**, and **Prescribed Displacement 5**.
2. Right-click and choose **Group**.

Cyclic Shear Loading

In the **Settings** window for **Group**, type `Cyclic Shear Loading` in the **Label** text field.

MATERIALS

Soil Material

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 `Soil Material` in the **Label** text field.

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

Property	Variable	Value	Unit	Property group
Reference shear strain	gammaRef	`gamma_ref`	1	Nonlinear elastic material
Poisson's ratio	nu	`Nu`	1	Young's modulus and Poisson's ratio
Initial shear modulus	G0	`G_0`	N/m²	Nonlinear elastic material
Density	rho	2400	kg/m³	Basic

One mesh element is sufficient for this analysis.

MESH 1

Mapped 1

In the **Mesh** toolbar, click **Mapped**.

Distribution 1

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

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

3 From the **Selection** list, choose **All boundaries**.

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

5 Click **Build Selected**.

STUDY: CYCLIC TRIAXIAL LOADING

Disable the default plots for this study.

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

2 In the **Settings** window for **Study**, type `Study: Cyclic Triaxial Loading` in the **Label** text field.

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

Step 1: Stationary

1 In the **Model Builder** window, under **Study: Cyclic Triaxial Loading** click **Step 1: Stationary**.

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

3 Select the **Modify model configuration for study step** check box.

4 In the tree, select **Component 1 (comp1)>Solid Mechanics (solid)>Cyclic Shear Loading**.

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

6 Click to expand the **Study Extensions** section. Select the **Auxiliary sweep** check box.

7 Click **Add**.

8 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
para (Parameter)	`range(0,0.001,5)`	

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

Add a second study for the cyclic shear loading.

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>Stationary**.

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

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

STUDY: CYCLIC SHEAR LOADING

Disable the default plots for this study.

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

2 In the **Settings** window for **Study**, type `Study: Cyclic Shear Loading` in the **Label** text field.

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

1 In the **Model Builder** window, under **Study: Cyclic Shear Loading** click **Step 1: Stationary**.

2 In the **Settings** window for **Stationary**, locate the **Study Extensions** section.

3 Select the **Auxiliary sweep** check box.

4 Click **Add**.

5 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
para (Parameter)	`range(0,0.001,5)`	

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

RESULTS

Axial Stress vs. Axial Strain

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

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

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

4 In the **Title** text area, type `Axial Stress vs. Axial Strain`.

5 Locate the **Plot Settings** section.

6 Select the **x-axis label** check box. In the associated text field, type `Axial strain (1)`.

7 Select the **y-axis label** check box. In the associated text field, type `Axial stress (kPa)`.

8 Locate the **Legend** section. From the **Position** list, choose **Upper left**.

Point Graph 1

1 Right-click **Axial Stress vs. Axial Strain** and choose **Point Graph**.

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

3 From the **Dataset** list, choose **Study: Cyclic Triaxial Loading/Solution 1 (sol1)**.

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

5 In the **Parameter indices (1-5001)** text field, type `range(1,1,1001)`.

6 Select Point 4 only.

7 Locate the **y-Axis Data** section. In the **Expression** text field, type `-solid.SYY`.

8 From the **Unit** list, choose **kPa**.

9 Locate the **x-Axis Data** section. From the **Parameter** list, choose **Expression**.

10 In the **Expression** text field, type `-solid.eYY`.

11 Click to expand the **Legends** section. Select the **Show legends** check box.

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

13 In the table, enter the following settings:

Legends
`Primary loading`

14 Right-click **Point Graph 1** and choose **Duplicate**.

Point Graph 2

1. In the **Model Builder** window, click **Point Graph 2**.
2. In the **Settings** window for **Point Graph**, locate the **Data** section.
3. In the **Parameter indices (1-5001)** text field, type `range(1001,1,5001)`.
4. Locate the **Legends** section. In the table, enter the following settings:

Legends
`Unloading and reloading`

Axial Stress vs. Axial Strain

1. In the **Model Builder** window, click **Axial Stress vs. Axial Strain**.
2. In the **Axial Stress vs. Axial Strain** toolbar, click **Plot**.
3. Right-click **Axial Stress vs. Axial Strain** and choose **Duplicate**.

Young's Modulus vs. Axial Strain

1. In the **Model Builder** window, under **Results** click **Axial Stress vs. Axial Strain 1**.
2. In the **Settings** window for **1D Plot Group**, type `Young's Modulus vs. Axial Strain` in the **Label** text field.
3. Locate the **Title** section. In the **Title** text area, type `Young's Modulus vs. Axial Strain`.
4. Locate the **Plot Settings** section. In the **y-axis label** text field, type `Young's Modulus (MPa)`.

Point Graph 1

1. In the **Model Builder** window, expand the **Young's Modulus vs. Axial Strain** node, then click **Point Graph 1**.
2. In the **Settings** window for **Point Graph**, locate the **y-Axis Data** section.
3. In the **Expression** text field, type `solid.E`.
4. From the **Unit** list, choose **MPa**.
5. Locate the **x-Axis Data** section. In the **Expression** text field, type `abs(solid.eYY)`.

Point Graph 2

1. In the **Model Builder** window, click **Point Graph 2**.
2. In the **Settings** window for **Point Graph**, locate the **Data** section.
3. In the **Parameter indices (1-5001)** text field, type `range(3002,1,4001)`.
4. Locate the **y-Axis Data** section. In the **Expression** text field, type `solid.E`.
5. From the **Unit** list, choose **MPa**.
6. Locate the **x-Axis Data** section. In the **Expression** text field, type `abs(solid.eYY-withsol('sol1',solid.eYY,setval(para,3)))`.
7. Locate the **Legends** section. In the table, enter the following settings:

Legends
`Reloading`

Young's Modulus vs. Axial Strain

1. In the **Model Builder** window, click **Young's Modulus vs. Axial Strain**.
2. In the **Settings** window for **1D Plot Group**, locate the **Axis** section.
3. Select the **x-axis log scale** check box.
4. Locate the **Legend** section. From the **Position** list, choose **Lower left**.
5. In the **Young's Modulus vs. Axial Strain** toolbar, click **Plot**.

Axial Stress vs. Axial Strain

In the **Model Builder** window, right-click **Axial Stress vs. Axial Strain** and choose **Duplicate**.

Shear Stress vs. Shear Strain

1. In the **Model Builder** window, under **Results** click **Axial Stress vs. Axial Strain 1**.
2. In the **Settings** window for **1D Plot Group**, type `Shear Stress vs. Shear Strain` in the **Label** text field.
3. Locate the **Data** section. From the **Dataset** list, choose **Study: Cyclic Shear Loading/Solution 2 (sol2)**.
4. Locate the **Title** section. In the **Title** text area, type `Shear Stress vs. Shear Strain`.
5. Locate the **Plot Settings** section. In the **x-axis label** text field, type `Shear strain (1)`.
6. In the **y-axis label** text field, type `Shear stress (kPa)`.

Point Graph 1

1. In the **Model Builder** window, expand the **Shear Stress vs. Shear Strain** node, then click **Point Graph 1**.
2. In the **Settings** window for **Point Graph**, locate the **Data** section.

3 From the **Dataset** list, choose **Study: Cyclic Shear Loading/Solution 2 (sol2)**.

4 Locate the **y-Axis Data** section. In the **Expression** text field, type `solid.SXY`.

5 Locate the **x-Axis Data** section. In the **Expression** text field, type `solid.eXY`.

Point Graph 2

1 In the **Model Builder** window, click **Point Graph 2**.

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

3 From the **Dataset** list, choose **Study: Cyclic Shear Loading/Solution 2 (sol2)**.

4 Locate the **y-Axis Data** section. In the **Expression** text field, type `solid.SXY`.

5 Locate the **x-Axis Data** section. In the **Expression** text field, type `solid.eXY`.

6 In the **Shear Stress vs. Shear Strain** toolbar, click **Plot**.

Young's Modulus vs. Axial Strain

In the **Model Builder** window, under **Results** right-click **Young's Modulus vs. Axial Strain** and choose **Duplicate**.

Shear Modulus vs. Shear Strain

1 In the **Model Builder** window, under **Results** click **Young's Modulus vs. Axial Strain 1**.

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

3 Locate the **Data** section. From the **Dataset** list, choose **Study: Cyclic Shear Loading/Solution 2 (sol2)**.

4 Locate the **Title** section. In the **Title** text area, type `Shear Modulus vs. Shear Strain`.

5 Locate the **Plot Settings** section. In the **x-axis label** text field, type `Shear strain (1)`.

6 In the **y-axis label** text field, type `Shear modulus (MPa)`.

Point Graph 1

1 In the **Model Builder** window, expand the **Shear Modulus vs. Shear Strain** node, then click **Point Graph 1**.

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

3 From the **Dataset** list, choose **Study: Cyclic Shear Loading/Solution 2 (sol2)**.

4 Locate the **y-Axis Data** section. In the **Expression** text field, type `solid.G`.

5 Locate the **x-Axis Data** section. In the **Expression** text field, type `abs(solid.eXY)`.

Point Graph 2

1 In the **Model Builder** window, click **Point Graph 2**.

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

3 From the **Dataset** list, choose **Study: Cyclic Shear Loading/Solution 2 (sol2)**.

4 Locate the **y-Axis Data** section. In the **Expression** text field, type `solid.G`.

5 Locate the **x-Axis Data** section. In the **Expression** text field, type `abs(solid.eXY-withsol('sol2',solid.eXY,setval(para,3)))`.

6 In the **Shear Modulus vs. Shear Strain** toolbar, click **Plot**.

Axial Stress vs. Axial Strain, Young’s Modulus vs. Axial Strain

1 In the **Model Builder** window, under **Results**, Ctrl-click to select **Axial Stress vs. Axial Strain** and **Young’s Modulus vs. Axial Strain**.

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

Triaxial Loading

In the **Settings** window for **Group**, type `Triaxial Loading` in the **Label** text field.

Shear Modulus vs. Shear Strain, Shear Stress vs. Shear Strain

1 In the **Model Builder** window, under **Results**, Ctrl-click to select **Shear Stress vs. Shear Strain** and **Shear Modulus vs. Shear Strain**.

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

Shear Loading

In the **Settings** window for **Group**, type `Shear Loading` in the **Label** text field.