Created in COMSOL Multiphysics 6.2

# Parameter Estimation of Elastoplastic Materials

## *Introduction*

This tutorial model demonstrates how to estimate the material parameters of an elastoplastic constitutive model based on cyclic shear data. The mathematical definition of the inverse problem is formulated in Parameter Estimation of Hyperelastic Materials, wherein a similar inverse problem is solved for the case of hyperelasticity.

In this example, we assume that the material is isotropic and that the Young's modulus $\boldsymbol{E}$ and Poisson's ratio $\nu$ are known a priori (for example, extracted directly from the initial part of the stress–strain curve), and we will focus on how to estimate the plastic material behavior. The problem setup is inspired by Ref. 1.

## *Model Definition*

The load case considered is a cyclic shear test, consisting of strain-controlled cyclic loading up to a maximum engineering shear strain $\gamma = 2.5\%$. The stress–strain curve is shown in Figure 1. The elastic material parameters $\boldsymbol{E} = 200$ GPa and $\nu = 0.3$ are considered known.

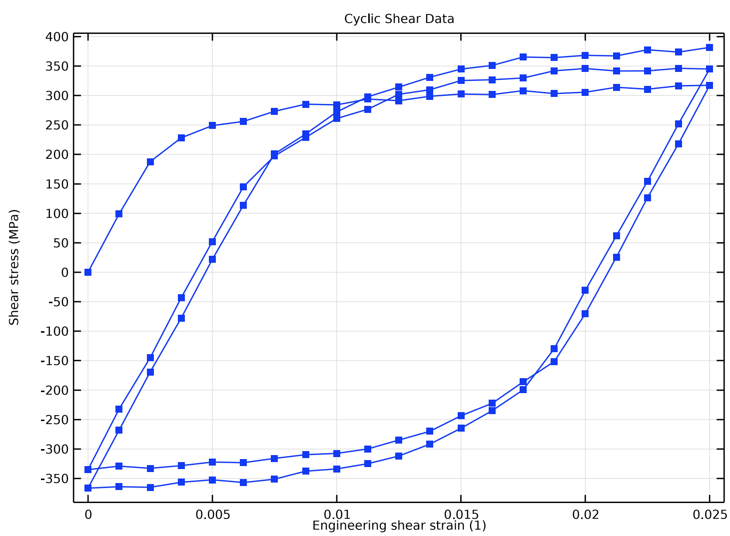



*Figure 1: Cyclic simple shear data.*

Before setting up the model, it is a good idea to conduct a preliminary analysis of the stress–strain curve to choose an appropriate elastoplastic material model. First, note that initial yielding starts at a shear stress $\tau$ of about 200 MPa. We can thus estimate the initial yield stress $\sigma_{y0}$ as

$$\sigma_{y0} = \sqrt{3}\tau \approx 350\,\mathrm{MPa} \tag{1}$$

by using the definition of the von Mises stress $\sigma_{\mathrm{mises}} = \sqrt{\frac{3}{2}(\mathbf{s} \cdot \mathbf{s})}$ ; $\mathbf{s}$ being the deviatoric stress.

Next, we analyze the hardening behavior. Note that the maximum stress in the first loading is about 300 MPa, and the onset of plastic flow during unloading starts around $\tau = -150$ MPa. This indicates a significant Bauschinger effect, which means that the material model needs to include both isotropic and kinematic hardening. Assuming von Mises-associated plasticity, the yield function $F$ can be expressed as

$$F = \sqrt{\frac{3}{2}((\mathbf{s} - \mathbf{s}_\mathrm{b}) \cdot (\mathbf{s} - \mathbf{s}_\mathrm{b}))} - \sigma_\mathrm{y}(\varepsilon_\mathrm{pe}) \tag{2}$$

where $\mathbf{s}_\mathrm{b}$ denotes the back stress, $\sigma_\mathrm{y}$ is the isotropic hardening function, and $\varepsilon_\mathrm{pe}$ is the equivalent plastic strain. For the isotropic hardening part, we select a nonlinear Voce model

$$\sigma_\mathrm{y}(\varepsilon_\mathrm{pe}) = \sigma_{\mathrm{y}0} + \sigma_\mathrm{sat}(1 - \exp(-\beta\varepsilon_\mathrm{pe})) \tag{3}$$

Herein, the two material parameters to estimate are the saturation stress $\sigma_\mathrm{sat}$ and the saturation exponent $\beta$. In particular, this model can capture both cyclic hardening and softening depending on the sign of $\sigma_\mathrm{sat}$.

To model kinematic hardening, an evolution equation for the back stress is needed. The simplest form is a linear relation where the back stress is proportional to the plastic strain tensor,

$$\mathbf{s}_\mathrm{b} = \frac{2}{3}C_\mathrm{k}\varepsilon_\mathrm{pl} \tag{4}$$

Here, $C_\mathrm{k}$ is the kinematic hardening modulus. In COMSOL Multiphysics, the user input controlling the kinematic hardening is the tangent modulus $E_\mathrm{k}$, which is related to $C_\mathrm{k}$ according to

$$\frac{1}{E_\mathrm{k}} = \frac{1}{E} + \frac{1}{C_\mathrm{k}} \tag{5}$$

Since the initial yield stress is given by Equation 1, the inverse problem needs to be solved for three parameters: $\sigma_\mathrm{sat}$, $\beta$, and $C_\mathrm{k}$.

The linear kinematic hardening model will be compared with the Armstrong–Frederick kinematic hardening model, for which the back stress is computed by integrating the evolution equation

$$\dot{\mathbf{s}}_{\mathrm{b}} = \frac{2}{3}C_{\mathrm{k}}\dot{\varepsilon}_{\mathrm{pl}} - \gamma_{\mathrm{k}}\dot{\varepsilon}_{\mathrm{pe}}\mathbf{s}_{\mathrm{b}} \tag{6}$$

The second term, which depends on the dimensionless kinematic hardening parameter $\gamma_{\mathrm{k}}$, models the rate of decrease of the hardening modulus upon accumulation of plastic strain, which leads to a stress–strain response that stabilizes upon repeated cyclic loading.

The material parameters for both models along with an initial guess of their values are provided in Table 1.

TABLE 1: ELASTOPLASTIC MODEL PARAMETERS AND INITIAL VALUES

| **Parameter** | **Name** | **Initial guess, linear kinematic hardening** | **Initial guess, Armstrong–Frederick kinematic hardening** |
|---|---|---|---|
| Saturation stress | sig_sat | `100[MPa]` | `100[MPa]` |
| Saturation exponent | beta | `5.0` | `5.0` |
| Kinematic hardening modulus | C_k | `10[GPa]` | `10[GPa]` |
| Kinematic hardening parameter | gamma_k | – | 100 |

## *Results and Discussion*

The prediction for the linear kinematic hardening model with the initial parameter values in Table 1 is shown in Figure 2. Although this initial prediction is rather poor, in particular with regard to the cyclic hardening, it is suitable as a starting point for the parameter estimation study.

After having estimated the parameters $\sigma_{\mathrm{sat}}$, $\beta$, and $C_{\mathrm{k}}$ of the linear kinematic hardening model, the results with the calibrated material parameters are shown in Figure 3. The final values of the parameters are reported in Table 2.

When switching to the Armstrong–Frederick kinematic hardening model, all four parameters $\sigma_{\mathrm{sat}}$, $\beta$, $C_{\mathrm{k}}$, and $\gamma_{\mathrm{k}}$ are estimated. This leads to a significant improvement in the overall prediction, see Figure 4.

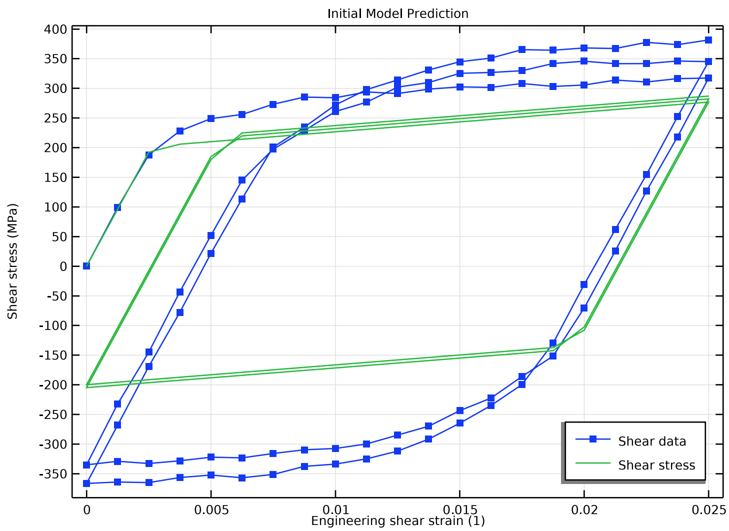



*Figure 2: Model prediction with the initial guess of the material parameters.*

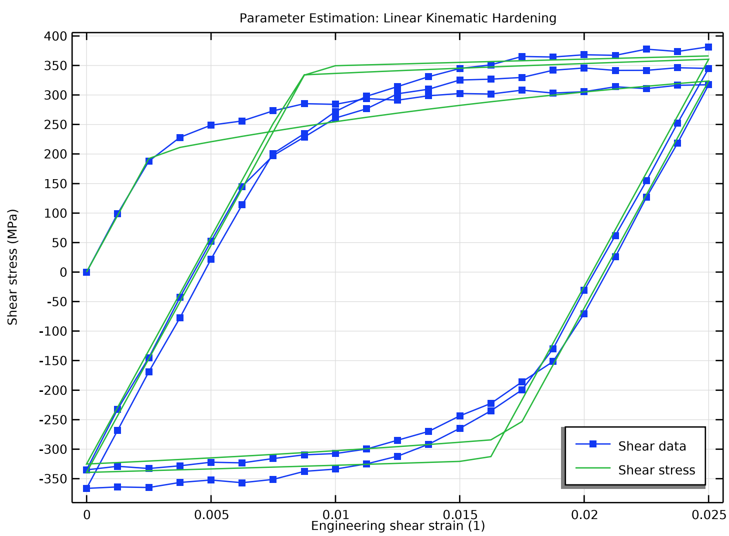



*Figure 3: Model prediction after estimating the parameters of the linear kinematic hardening model.*

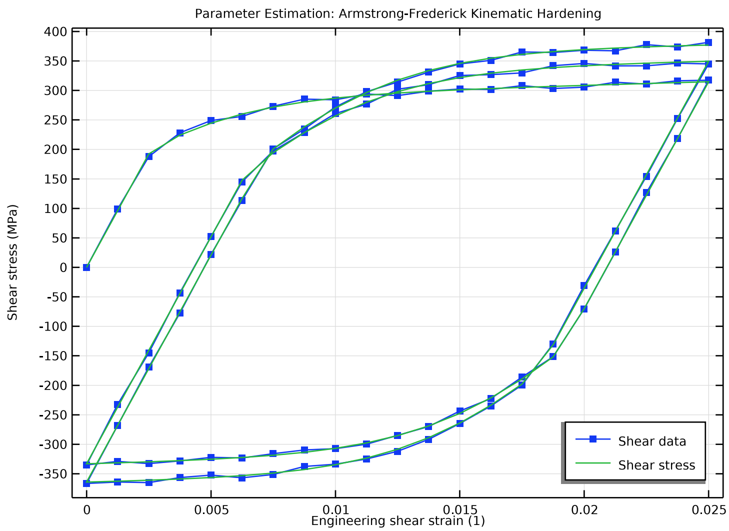



*Figure 4: Model prediction after estimating the parameters of the Armstrong–Frederick kinematic hardening model.*

TABLE 2: MATERIAL PARAMETERS OF THE CALIBRATED ELASTOPLASTIC MODELS.

| **Parameter** | **Name** | **Estimated parameters, linear kinematic hardening** | **Estimated parameters, Armstrong–Frederick hardening** |
|---|---|---|---|
| Saturation stress | sig_sat | 250.42 MPa | 388.84 MPa |
| Saturation exponent | beta | 98.807 | 10.385 |
| Kinematic hardening modulus | C_k | 3.0586 GPa | 75.786 GPa |
| Kinematic hardening parameter | gamma_k | – | 507.24 |

## *Notes About the COMSOL Implementation*

The **Parameter Estimation** functionality is available in COMSOL Multiphysics in the context menu of a **Component** or under **Optimization** in the **Physics** toolbar. To solve an inverse problem, add one **Global Least-Squares Objective** subnode for each experiment together with a study containing a **Parameter Estimation** study step. For most least-squares problems, the **Levenberg–Marquardt** algorithm with a finite difference approximation of the Jacobian is a robust and efficient choice of optimization solver.

## *References*

1. J. Fu, F. Barlat, J.-H. Kim, and F. Pierron, "Identification of nonlinear kinematic hardening constitutive model parameters using the virtual fields method for advanced high strength steels," *Int. J. Solids Struct.*, vol. 102–103, pp. 30–43, 2016.

**Application Library path:** `Nonlinear_Structural_Materials_Module/Plasticity/parameter_estimation_plasticity`

## *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 **Study**.

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

6 Click **Done**.

### ROOT

1 In the **Model Builder** window, click the root node.

2 In the root node's **Settings** window, locate the **Unit System** section.

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

The MPa base unit system is often convenient to use when working with structural mechanics problems.

### RESULTS

Start by importing the cyclic shear data.

*Cyclic Shear Data*

1 In the **Model Builder** window, expand the **Results** node.

2 Right-click **Results>Tables** and choose **Table**.

3 In the **Settings** window for **Table**, type `Cyclic Shear Data` in the **Label** text field.

4 Locate the **Data** section. Click **Import**.

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

### CYCLIC SHEAR DATA

1 Go to the **Cyclic Shear Data** window.

2 Click **Table Graph** in the window toolbar.

### RESULTS

*Table Graph 1*

1 In the **Model Builder** window, under **Results>1D Plot Group 1** click **Table Graph 1**.

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

3 From the **x-axis data** list, choose **Engineering shear strain (-)**.

4 From the **Plot columns** list, choose **Manual**.

5 In the **Columns** list, select **Shear stress (MPa)**.

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

*Cyclic Shear Data*

1 In the **Model Builder** window, under **Results** click **1D Plot Group 1**.

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

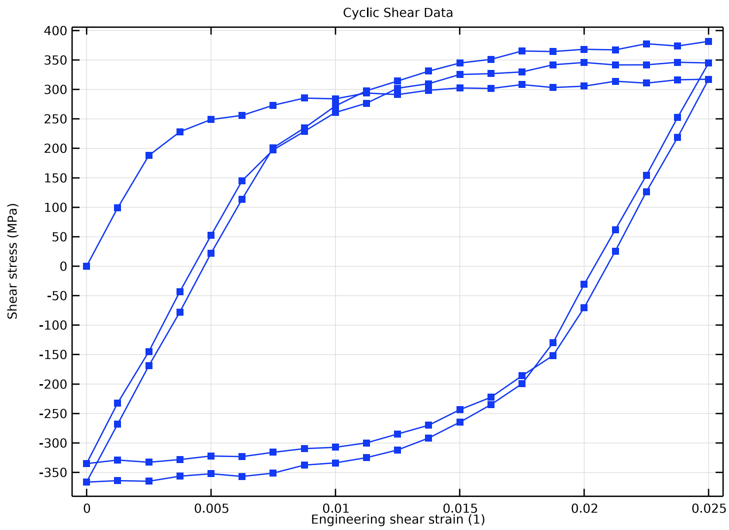
3 Click to expand the **Title** section. From the **Title type** list, choose **Label**.

4 Locate the **Plot Settings** section.

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

6 Select the **y-axis label** check box. In the associated text field, type `Shear stress (MPa)`.

**7** In the **Cyclic Shear Data** toolbar, click **Plot**.




## GLOBAL DEFINITIONS

Now, set up the forward model. Start by creating an interpolation function for the applied shear strain as a function of time.

### *Shear Strain*

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

**2** In the **Settings** window for **Interpolation**, type `Shear Strain` in the **Label** text field.

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

**4** Find the **Functions** subsection. In the table, enter the following settings:

| Function name | Position in file |
|---|---|
| `shear_strain` | 1 |

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

| Function | Unit |
|---|---|
| shear_strain | 1 |

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

| Argument | Unit |
|---|---|
| Column 1 | s |

**7** Click **Plot**.

*Parameters 1*

**1** In the **Model Builder** window, 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 |
|---|---|---|---|
| L | 1[mm] | 1 mm | Unit length |
| rho0 | 8000[kg/m^3] | 8E-9 t/mm³ | Density |
| E0 | 200[GPa] | 2E5 MPa | Young's modulus |
| nu0 | 0.3 | 0.3 | Poisson's ratio |
| sig_y0 | 350[MPa] | 350 MPa | Initial yield stress |
| sig_sat | 100[MPa] | 100 MPa | Saturation stress |
| beta | 5 | 5 | Saturation exponent |
| C_k | 10[GPa] | 10000 MPa | Kinematic hardening modulus |
| E_k | 1/(1/E0+1/C_k) | 9523.8 MPa | Kinematic tangent modulus |
| gamma_k | 100 | 100 | Kinematic hardening parameter |

## GEOMETRY 1

*Block 1 (blk1)*

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

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

**3** In the **Width** text field, type L.

**4** In the **Depth** text field, type L.

**5** In the **Height** text field, type L.

## MATERIALS

### *Material 1 (mat1)*

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

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

3 In the table, enter the following settings:

| Property | Variable | Value | Unit | Property group |
|---|---|---|---|---|
| Young's modulus | E | `E0` | Pa | Young's modulus and Poisson's ratio |
| Poisson's ratio | nu | `nu0` | 1 | Young's modulus and Poisson's ratio |
| Density | rho | `rho0` | t/mm³ | Basic |

4 Click to expand the **Material Properties** section. In the **Material properties** tree, select **Solid Mechanics>Elastoplastic Material>Elastoplastic Material Model**.

5 Click **Add to Material**.

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

| Property | Variable | Value | Unit | Property group |
|---|---|---|---|---|
| Initial yield stress | sigmags | `sig_y0` | MPa | Elastoplastic material model |
| Kinematic tangent modulus | Ek | `E_k` | MPa | Elastoplastic material model |

7 Locate the **Material Properties** section. In the **Material properties** tree, select **Solid Mechanics>Elastoplastic Material>Voce**.

8 Click **Add to Material**.

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

| Property | Variable | Value | Unit | Property group |
|---|---|---|---|---|
| Saturation flow stress | sigma_voc | `sig_sat` | MPa | Voce |
| Saturation exponent | beta_voc | `beta` | 1 | Voce |

10 Locate the **Material Properties** section. In the **Material properties** tree, select **Solid Mechanics>Elastoplastic Material>Armstrong-Frederick**.

11 Click **Add to Material**.

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

| Property | Variable | Value | Unit | Property group |
|---|---|---|---|---|
| Kinematic hardening modulus | Ck | `C_k` | MPa | Armstrong-Frederick |
| Kinematic hardening parameter | gammak | `gamma_k` | 1 | Armstrong-Frederick |

### SOLID MECHANICS (SOLID)

When solving inverse problems, the forward model will be solved multiple times for each iteration of the optimization solver. If the material tests are designed such that the distributions of stress and strain are homogeneous, it is computationally advantageous to set up a single element model of the experiment by using idealized boundary conditions, linear shape functions, and reduced integration.

1. In the **Model Builder** window, expand the **Material 1 (mat1)** node, then click **Component 1 (comp1)>Solid Mechanics (solid)**.
2. In the **Settings** window for **Solid Mechanics**, locate the **Structural Transient Behavior** section.
3. From the list, choose **Quasistatic**.
4. Click to expand the **Discretization** section. From the **Displacement field** list, choose **Linear**.

#### *Linear Elastic Material 1*

1. In the **Model Builder** window, under **Component 1 (comp1)>Solid Mechanics (solid)** click **Linear Elastic Material 1**.
2. In the **Settings** window for **Linear Elastic Material**, locate the **Geometric Nonlinearity** section.
3. From the **Formulation** list, choose **Geometrically linear**.
4. Locate the **Quadrature Settings** section. Select the **Reduced integration** check box.

#### *Linear Kinematic Hardening*

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

   First, we define an elastoplastic model with Voce isotropic hardening and linear kinematic hardening.
2. In the **Settings** window for **Plasticity**, type `Linear Kinematic Hardening` in the **Label** text field.

**3** Locate the **Plasticity Model** section. Find the **Isotropic hardening model** subsection. From the list, choose **Voce**.

**4** Find the **Kinematic hardening model** subsection. From the list, choose **Linear**.

*Fixed Constraint 1*

Define the idealized boundary conditions for simple shear in the xz-plane. Since we use linear shape functions, we do not need to apply any constraints to the x boundaries for them to remain flat during deformation.

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

**2** Select Boundary 3 only.

*Roller 1*

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

**2** Select Boundaries 2 and 5 only.

*Prescribed Displacement 1*

**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 x direction** list, choose **Prescribed**.

**5** In the $u_{0x}$ text field, type `shear_strain(t)*L`.

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

**7** From the **Displacement in z direction** list, choose **Prescribed**.

## DEFINITIONS

Add a variable for the volume-averaged shear stress. This global variable will be used later on in the **Global Least-Squares Objective**.

*Average 1 (aveop1)*

**1** In the **Definitions** toolbar, click **Nonlocal Couplings** and choose **Average**.

**2** Select Domain 1 only.

*Variables 1*

**1** In the **Model Builder** window, right-click **Definitions** and choose **Variables**.

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

**3** In the table, enter the following settings:

| Name | Expression | Unit | Description |
|---|---|---|---|
| tau | aveop1(solid.sxz) | MPa | Shear stress |

## MESH 1

Mesh the unit cube with a single hexahedral element.

### *Mapped 1*

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

**2** Select Boundary 1 only.

### *Distribution 1*

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

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

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

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

### *Swept 1*

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

### *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** In the **Model Builder** window, right-click **Mesh 1** and choose **Build All**.

## FORWARD PROBLEM

Solve the forward model once to check that everything is set up correctly.

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

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

**3** Clear the **Generate default plots** check box.

**4** In the **Label** text field, type `Forward Problem`.

### *Step 1: Time Dependent*

**1** In the **Model Builder** window, under **Forward Problem** 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(0,1,100)`.

### *Solution 1 (sol1)*

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

2 In the **Model Builder** window, expand the **Solution 1 (sol1)** node.

3 In the **Model Builder** window, under **Forward Problem>Solver Configurations>Solution 1 (sol1)** 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 Find the **Algebraic variable settings** subsection. From the **Consistent initialization** list, choose **Off**.

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

## RESULTS

Compare the initial model prediction with the shear data.

### *Cyclic Shear Data*

In the **Model Builder** window, under **Results** right-click **Cyclic Shear Data** and choose **Duplicate**.

### *Initial Model Prediction*

1 In the **Model Builder** window, under **Results** click **Cyclic Shear Data 1**.

2 In the **Settings** window for **1D Plot Group**, type `Initial Model Prediction` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Forward Problem/Solution 1 (sol1)**.

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

### *Table Graph 1*

1 In the **Model Builder** window, expand the **Initial Model Prediction** node, then click **Table Graph 1**.

2 In the **Settings** window for **Table Graph**, click to expand the **Legends** section.

3 Select the **Show legends** check box.

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

5 In the table, enter the following settings:

| Legends |
|---|
| `Shear data` |

*Global 1*

1 In the **Model Builder** window, right-click **Initial Model Prediction** and choose **Global**.

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

3 In the table, enter the following settings:

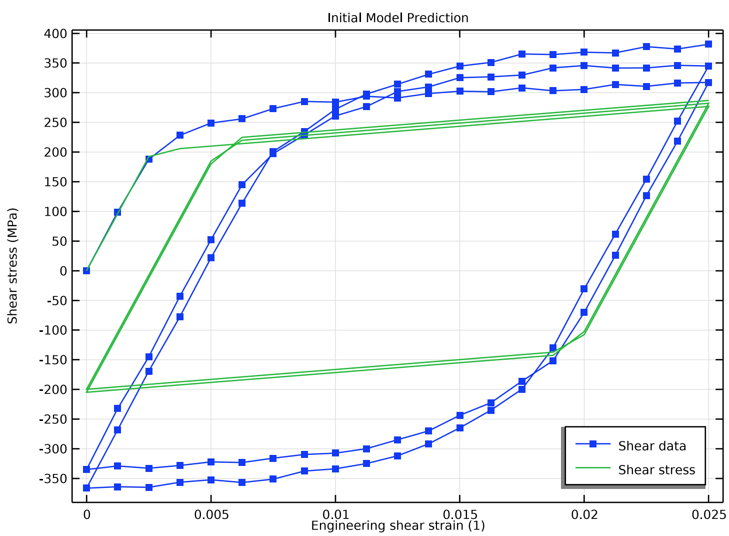
| Expression | Unit | Description |
|---|---|---|
| `tau` | `MPa` | `Shear stress` |

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

5 In the **Expression** text field, type `shear_strain(t)`.

6 Click to expand the **Legends** section. Find the **Include** subsection. Clear the **Solution** check box.

7 In the **Initial Model Prediction** toolbar, click **Plot**.




## COMPONENT 1 (COMP1)

Now, set up the inverse problem to improve the model parameters.

*Global Least-Squares Objective 1*

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

2 In the **Settings** window for **Global Least-Squares Objective**, locate the **Experimental Data** section.

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

The second column containing the shear strain data can be ignored in the optimization problem, since we use these data to prescribe the boundary conditions.

4 Locate the **Data Column Settings** section. In the table, enter the following settings:

| Columns | Type | Settings |
|---|---|---|
| Engineering shear strain (-) | Ignored column | |

The third column contains the shear stress data, which will be used in the evaluation of the objective function. In the **Model expression** field, enter the expression for the global variable of the shear stress τ.

5 In the table, click to select the cell at row number 3 and column number 1.

6 In the **Model expression** text field, type `comp1.tau`.

7 In the **Variable name** text field, type `shear_stress`.

8 In the **Unit** text field, type `MPa`.

### 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.

### PARAMETER ESTIMATION: LINEAR KINEMATIC HARDENING

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

2 In the **Settings** window for **Study**, type `Parameter Estimation: Linear Kinematic Hardening` in the **Label** text field.

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

#### *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 **All Least-Squares objectives**.

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

**5** In the table, enter the following settings:

| **Parameter name** | **Initial value** | **Scale** | **Lower bound** | **Upper bound** |
|---|---|---|---|---|
| sig_sat (Saturation stress) | `100[MPa]` | `100[MPa]` | | |
| beta (Saturation exponent) | `5` | `5` | | |
| C_k (Kinematic hardening modulus) | `10[GPa]` | `10[GPa]` | | |

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

**7** Find the **Solver settings** subsection. From the **Least-squares time/parameter method** list, choose **Use only least-squares data points**.

*Solution 2 (sol2)*

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

**2** In the **Model Builder** window, expand the **Solution 2 (sol2)** node.

**3** In the **Model Builder** window, expand the **Parameter Estimation: Linear Kinematic Hardening>Solver Configurations> Solution 2 (sol2)>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** Find the **Algebraic variable settings** subsection. From the **Consistent initialization** list, choose **Off**.

## RESULTS

Before computing the study, set up a plot that can be used while solving to monitor the comparison between the data and the model prediction.

*Initial Model Prediction*

In the **Model Builder** window, under **Results** right-click **Initial Model Prediction** and choose **Duplicate**.

*Parameter Estimation: Linear Kinematic Hardening*

**1** In the **Model Builder** window, under **Results** click **Initial Model Prediction 1**.

**2** In the **Settings** window for **1D Plot Group**, type `Parameter Estimation: Linear Kinematic Hardening` in the **Label** text field.

**3** Locate the **Data** section. From the **Dataset** list, choose **Parameter Estimation: Linear Kinematic Hardening/Solution 2 (sol2)**.

## PARAMETER ESTIMATION: LINEAR KINEMATIC HARDENING

### *Parameter Estimation*

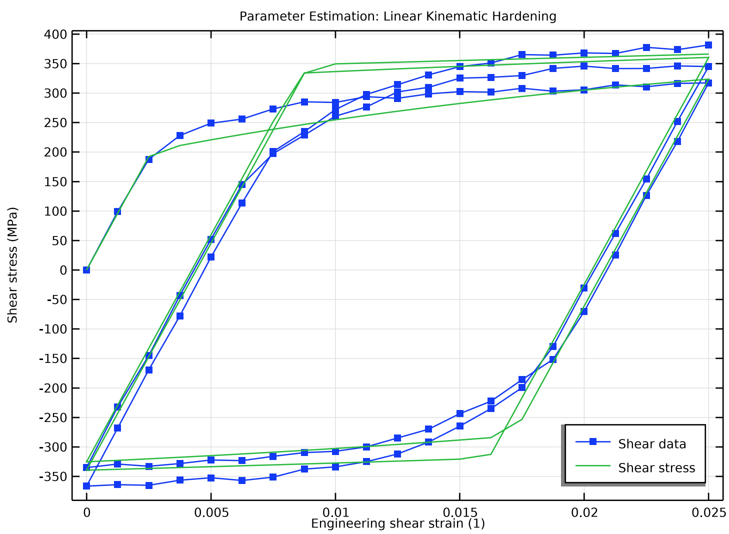
**1** In the **Model Builder** window, under **Parameter Estimation: Linear Kinematic Hardening** click **Parameter Estimation**.

**2** In the **Settings** window for **Parameter Estimation**, click to expand the **Output While Solving** section.

**3** Select the **Plot** check box.

**4** From the **Plot group** list, choose **Parameter Estimation: Linear Kinematic Hardening**.

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




The model is clearly improved, but the shape and cyclic evolution of the hardening is not well captured. Next, see if the prediction can be improved with a nonlinear kinematic hardening model.

## SOLID MECHANICS (SOLID)

### *Linear Kinematic Hardening*

In the **Model Builder** window, under **Component 1 (comp1)>Solid Mechanics (solid)>Linear Elastic Material 1** right-click **Linear Kinematic Hardening** and choose **Duplicate**.

*Armstrong-Frederick Kinematic Hardening*

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

2 In the **Settings** window for **Plasticity**, type `Armstrong-Frederick Kinematic Hardening` in the **Label** text field.

3 Locate the **Plasticity Model** section. Find the **Kinematic hardening model** subsection. From the list, choose **Armstrong-Frederick**.

## ADD STUDY

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

## PARAMETER ESTIMATION: ARMSTRONG-FREDERICK KINEMATIC HARDENING

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

2 In the **Settings** window for **Study**, type `Parameter Estimation: Armstrong-Frederick Kinematic Hardening` in the **Label** text field.

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

*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 **All Least-Squares objectives**.

4 Locate the **Estimated Parameters** section. Click **Add** four times.

5 In the table, enter the following settings:

| Parameter name | Initial value | Scale | Lower bound | Upper bound |
|---|---|---|---|---|
| sig_sat (Saturation stress) | 100[MPa] | 100[MPa] | | |
| beta (Saturation exponent) | 5 | 10 | | |

| Parameter name | Initial value | Scale | Lower bound | Upper bound |
|---|---|---|---|---|
| C_k (Kinematic hardening modulus) | 10[GPa] | 10[GPa] | | |
| gamma_k (Kinematic hardening parameter) | 100 | 100 | | |

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

7 Find the **Solver settings** subsection. From the **Least-squares time/parameter method** list, choose **Use only least-squares data points**.

*Solution 3 (sol3)*

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

2 In the **Model Builder** window, expand the **Solution 3 (sol3)** node.

3 In the **Model Builder** window, expand the **Parameter Estimation: Armstrong-Frederick Kinematic Hardening>Solver Configurations>Solution 3 (sol3)>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 Find the **Algebraic variable settings** subsection. From the **Consistent initialization** list, choose **Off**.

### RESULTS

Create a plot while solving also for this study.

*Parameter Estimation: Linear Kinematic Hardening*

In the **Model Builder** window, under **Results** right-click **Parameter Estimation: Linear Kinematic Hardening** and choose **Duplicate**.

*Parameter Estimation: Armstrong-Frederick Kinematic Hardening*

1 In the **Model Builder** window, under **Results** click **Parameter Estimation: Linear Kinematic Hardening 1**.

2 In the **Settings** window for **1D Plot Group**, type `Parameter Estimation: Armstrong-Frederick Kinematic Hardening` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Parameter Estimation: Armstrong-Frederick Kinematic Hardening/Solution 3 (sol3)**.

## PARAMETER ESTIMATION: ARMSTRONG-FREDERICK KINEMATIC HARDENING

*Parameter Estimation*

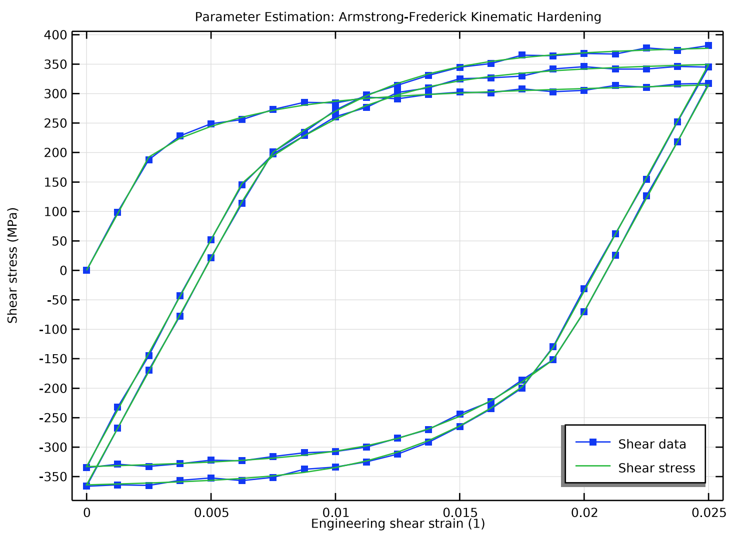
1 In the **Model Builder** window, under **Parameter Estimation: Armstrong-Frederick Kinematic Hardening** click **Parameter Estimation**.

2 In the **Settings** window for **Parameter Estimation**, locate the **Output While Solving** section.

3 Select the **Plot** check box.

4 From the **Plot group** list, choose **Parameter Estimation: Armstrong-Frederick Kinematic Hardening**.

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




The final prediction is now in excellent agreement with the data.

## RESULTS

Collect the final values of the material parameters by creating **Evaluation Groups** from the **Add Predefined Plot** menu.

## ADD PREDEFINED PLOT

1 In the **Home** toolbar, click **Windows** and choose **Add Predefined Plot**.

2 Go to the **Add Predefined Plot** window.

3 In the tree, select **Parameter Estimation: Linear Kinematic Hardening/Solution 2 (sol2)>Solid Mechanics>Estimated Parameters (std2)**.

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

### RESULTS

*Estimated Parameters: Linear Kinematic Hardening*

In the **Settings** window for **Evaluation Group**, type `Estimated Parameters: Linear Kinematic Hardening` in the **Label** text field.

### ADD PREDEFINED PLOT

1 Go to the **Add Predefined Plot** window.

2 In the tree, select **Parameter Estimation: Armstrong-Frederick Kinematic Hardening/Solution 3 (sol3)>Solid Mechanics>Estimated Parameters (std3)**.

3 Click **Add Plot** in the window toolbar.

4 In the **Home** toolbar, click **Windows** and choose **Add Predefined Plot**.

### RESULTS

*Estimated Parameters: Armstrong-Frederick Kinematic Hardening*

1 In the **Model Builder** window, under **Results** click **Estimated Parameters (std3) 1**.

2 In the **Settings** window for **Evaluation Group**, type `Estimated Parameters: Armstrong-Frederick Kinematic Hardening` in the **Label** text field.

The values of the material parameters can now be copied to a new **Material** node, which can be used for further analysis.