Created in COMSOL Multiphysics 6.2

Parameter Estimation of Hyperelastic Materials

Introduction

Rubber-like components in applications such as tires, seals, insulators, soft sensors, and actuators are often modeled as hyperelastic materials (Ref. 1). In order to accurately predict the behavior of such components using finite element models, the material model chosen needs to be calibrated and validated against experimental data. In contrast to isotropic linear elastic materials, for which the Young's modulus and the Poisson's ratio can be estimated directly from the stress and lateral contraction measured in a uniaxial tension test, the calibration of hyperelastic material models typically requires consideration of multiple load cases over the full range of deformation expected in the final application. This tutorial model demonstrates how to set up this so-called inverse problem in order to estimate the material parameters of a hyperelastic model from experimental data. The data are reproduced from Ref. 2, wherein large deformation uniaxial tension, pure shear, and equibiaxial tension tests were performed on a soft elastomeric material employed in a tactile sensor. The procedure is validated by comparing the material parameters of an Ogden hyperelastic model against the results reported in Ref. 2.

Model Definition

Parameter estimation problems consist of three components: (i) experimental data; (ii) a forward model that represents the physics of the experiments; and (iii) an optimization algorithm that compares the two and updates the model parameters to minimize the difference. This can be formulated mathematically as a nonlinear least-squares minimization problem,

$$\mathbf{q}_{\mathrm{opt}} = \mathrm{argmin}_{\mathbf{q}}\left(\sum_{n=1}^{N} Q_n\right) \tag{1}$$

with

$$Q_n = \frac{1}{2}\sum_{m=1}^{M_n}\left(P_n(\lambda_m, \mathbf{q}) - \hat{P}_{nm}\right)^2 \tag{2}$$

Herein, $\mathbf{q}$ is the vector of material parameters that we want to estimate, N is the number of experiments, M_n is the number of data points per experiment, $\hat{P}_{nm}$ is the mth data point of experiment n, and $P_n(\lambda_m, \mathbf{q})$ denotes the corresponding model prediction given the experimental parameter λ_m.

In this example, we consider $N = 3$ experiments (uniaxial tension, pure shear, and equibiaxial tension), for which the measured quantity P_n is the first Piola–Kirchhoff stress and λ_m is the applied stretch in the loading direction. A schematic of the three experiments is shown in Figure 1, and the data from Ref. 2 are reproduced in Figure 2.

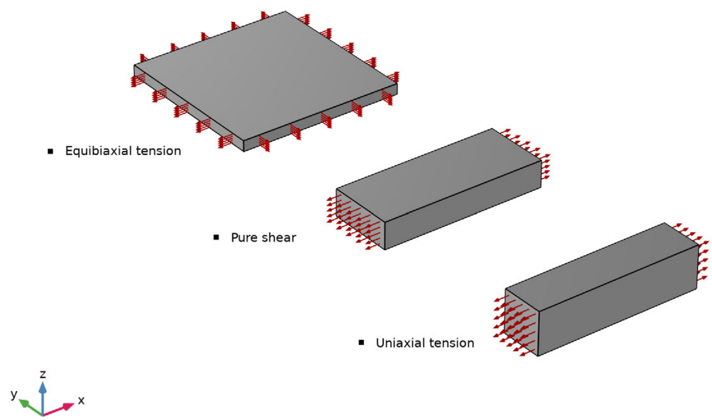


Figure 1: Illustration of the three homogeneous load cases considered.

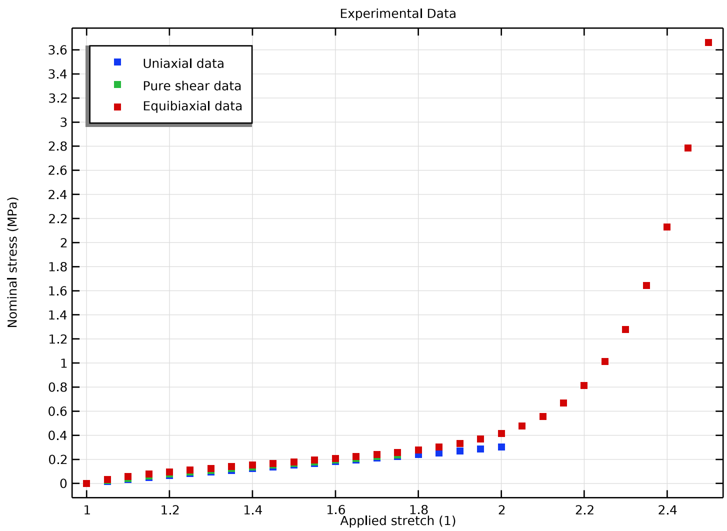


Figure 2: Experimental data from Ref. 2.

In Ref. 2, these data were used to fit the parameters of an incompressible second-order Ogden model. This form of the Ogden strain energy density function reads

$$W_s = \sum_{p=1}^{2} \frac{\mu_p}{\alpha_p}(\lambda_1^{\alpha_p} + \lambda_2^{\alpha_p} + \lambda_3^{\alpha_p} - 3) \tag{3}$$

which consists of four unknown material parameters that we will estimate, that is, $\mathbf{q} = (\mu_1, \alpha_1, \mu_2, \alpha_2)$. Note that for the model to yield physically admissible predictions, the parameters need to satisfy the constraints $\mu_p\alpha_p > 0$ for all terms p. The parameters along with an initial guess of their values are provided in Table 1.

TABLE 1: OGDEN MODEL PARAMETERS AND INITIAL VALUES

Parameter	Name	Initial guess
Ogden modulus, branch 1	mu1	200[kPa]
Ogden exponent, branch 1	alpha1	2.0
Ogden modulus, branch 2	mu2	-1[kPa]
Ogden exponent, branch 2	alpha2	-2.0

It is important to note that experimental data from standardized material tests are usually given in terms of stress–strain curves for a *homogeneous state of stress and deformation*.

In this case, the forward model can be set up with a single element, reduced integration, and idealized boundary conditions. This reduces the computational cost significantly compared to solving the full physical problem for every model evaluation within the optimization solver. However, if the assumption of homogeneity does not hold for the experimental data at hand, you may have to perform the inverse analysis by simulating the full geometry and comparing the global force–displacement curve instead of stress–strain data.

Results and Discussion

The model prediction for the initial guess of the parameter values in Table 1 is shown in Figure 3. Note that the uniaxial and pure shear behavior is of the correct order of magnitude, but the equibiaxial response is off by an order of magnitude for large stretches.

After running the parameter estimation study, the results for the calibrated material model are shown in Figure 4. The fit to the experimental data is excellent, and the final material parameters agree with those reported in Ref. 2, see Table 2.

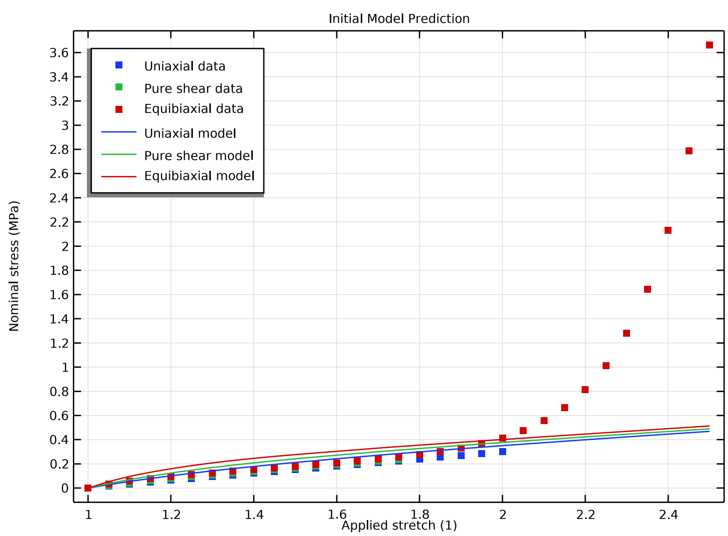


Figure 3: Model prediction with the initial values of the material parameters.

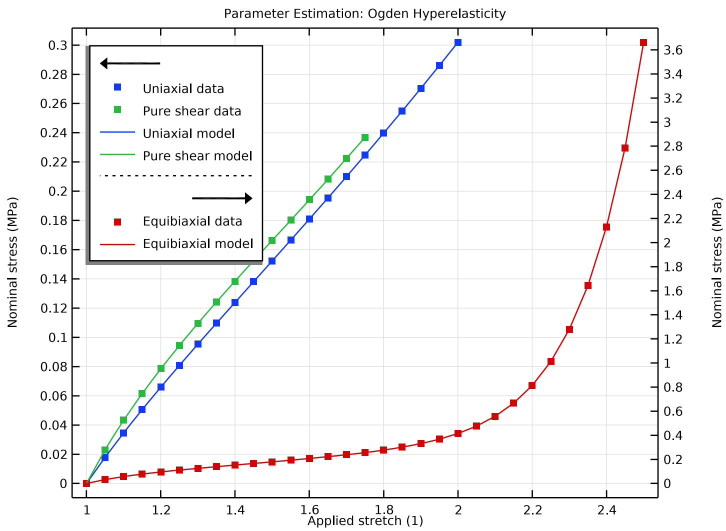


Figure 4: Model prediction with the calibrated material parameters. Note that the equibiaxial curves are displayed on the second y-axis to better visualize all datasets in a single plot.

TABLE 2: MATERIAL PARAMETERS OF THE CALIBRATED OGDEN MODEL.

Parameter	**Name**	**Estimated parameter**	**Reference value (Ref. 2)**
Ogden modulus, branch 1	mu1	`85.117[kPa]`	`85.1168[kPa]`
Ogden exponent, branch 1	alpha1	`2.8991`	`2.8991`
Ogden modulus, branch 2	mu2	`-0.002[kPa]`	`-0.002[kPa]`
Ogden exponent, branch 2	alpha2	`-8.2915`	`-8.2915`

Notes About the COMSOL Implementation

In parameter estimation problems, it is good practice to first set up and test the forward model before solving the inverse problem. When the experimental data consists of multiple load cases with different boundary conditions, it can be more efficient to solve them in parallel than in series. This is demonstrated here by creating three unit cube elements, one for each experiment.

The **Parameter Estimation** functionality is available in COMSOL Multiphysics in the context menu of a **Component** or under **Optimization** in the **Physics** toolbar, wherein each **Global Least-Squares Objective** node adds an objective corresponding to Equation 2 to the model. To solve the inverse problem, these need to be combined with a study containing a **Parameter Estimation** study step. When multiple objectives are selected in the study step, the total objective function that is minimized will be the sum of all objectives selected, see Equation 1. 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. P. Steinmann, M. Hossain, and G. Possart, "Hyperelastic models for rubber-like materials: consistent tangent operators and suitability for Treloar's data," *Arch. Appl. Mech.*, vol. 82, pp. 1183–1217, 2012.

2. C. Sferrazza, A. Wahlsten, C. Trueeb, and R. d'Andrea, "Ground Truth Force Distribution for Learning-Based Tactile Sensing: A Finite Element Approach," *IEEE Access*, vol. 7, pp. 173438–173449, 2019.

Application Library path: `Nonlinear_Structural_Materials_Module/Hyperelasticity/parameter_estimation_hyperelasticity`

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>Stationary**.

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 experimental data files to result tables.

Uniaxial 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 `Uniaxial 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_hyperelasticity_uniaxial.txt`.

Pure Shear Data

1 In the **Results** toolbar, click **Table**.

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

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

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

Equibiaxial Data

1 In the **Results** toolbar, click **Table**.

2 In the **Settings** window for **Table**, type `Equibiaxial Data` in the **Label** text field.

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

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

Experimental Data

1 In the **Results** toolbar, click **1D Plot Group**.

2 In the **Settings** window for **1D Plot Group**, type `Experimental 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 `Applied stretch (1)`.

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

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

Uniaxial Data

1 Right-click **Experimental Data** and choose **Table Graph**.

2 In the **Settings** window for **Table Graph**, type `Uniaxial Data` in the **Label** text field.

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

4 From the **Color** list, choose **Cycle (reset)**.

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

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

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

8 In the table, enter the following settings:

Legends
`Uniaxial data`

9 Right-click **Uniaxial Data** and choose **Duplicate**.

Pure Shear Data

1 In the **Model Builder** window, under **Results>Experimental Data** click **Uniaxial Data 1**.

2 In the **Settings** window for **Table Graph**, type `Pure Shear Data` in the **Label** text field.

3 Locate the **Data** section. From the **Table** list, choose **Pure Shear Data**.

4 Locate the **Coloring and Style** section. From the **Color** list, choose **Cycle**.

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

Legends
`Pure shear data`

6 Right-click **Pure Shear Data** and choose **Duplicate**.

Equibiaxial Data

1 In the **Model Builder** window, under **Results>Experimental Data** click **Pure Shear Data 1**.

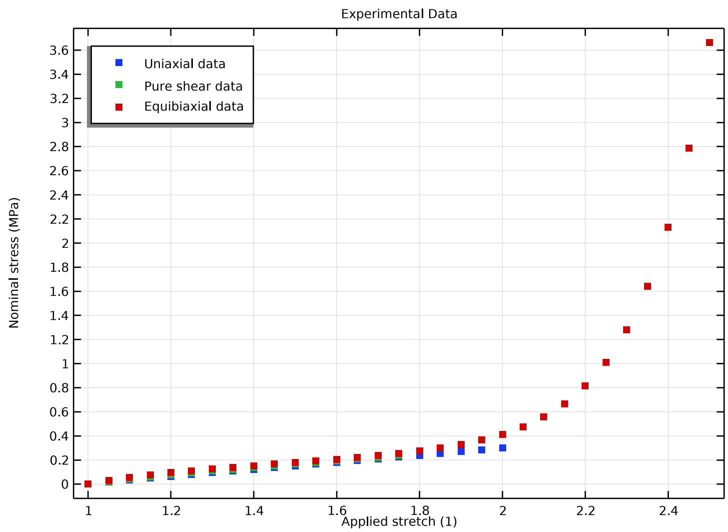
2 In the **Settings** window for **Table Graph**, type `Equibiaxial Data` in the **Label** text field.

3 Locate the **Data** section. From the **Table** list, choose **Equibiaxial Data**.

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

Legends
`Equibiaxial data`

5 In the **Experimental Data** toolbar, click **Plot**.



GLOBAL DEFINITIONS

Continue with defining the material parameters and setting up the forward problem.

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
`mu1`	`200[kPa]`	0.2 MPa	`Ogden modulus, branch 1`
`alpha1`	`2.0`	2	`Ogden exponent, branch 1`
`mu2`	`-1.0[kPa]`	-0.001 MPa	`Ogden modulus, branch 2`
`alpha2`	`-2.0`	-2	`Ogden exponent, branch 2`
`stretch`	`1`	1	`Applied stretch`

GEOMETRY 1

Create three unit cubes, one for each load case.

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Geometry 1**.
2 In the **Settings** window for **Geometry**, locate the **Units** section.
3 From the **Length unit** list, choose **mm**.

Block 1 (blk1)

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

Array 1 (arr1)

1 In the **Geometry** toolbar, click **Transforms** and choose **Array**.
2 Select the object **blk1** only.
3 In the **Settings** window for **Array**, locate the **Size** section.
4 In the **y size** text field, type 3.
5 Locate the **Displacement** section. In the **y** text field, type 3.
6 In the **Geometry** toolbar, click **Build All**.
7 Click the **Zoom Extents** button in the **Graphics** toolbar.

SOLID MECHANICS (SOLID)

Set up the three load cases in the **Solid Mechanics** interface. Since they result in homogeneous stresses and deformations, use linear shape functions with reduced integration to reduce the computation cost.

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

Hyperelastic Material 1

1 In the **Physics** toolbar, click **Domains** and choose **Hyperelastic Material**.
2 Click in the **Graphics** window and then press Ctrl+A to select all domains.
3 In the **Settings** window for **Hyperelastic Material**, locate the **Hyperelastic Material** section.
4 From the **Material model** list, choose **Ogden**.
5 From the **Compressibility** list, choose **Incompressible**.

6 Click **Add**.

7 In the **Ogden parameters** table, enter the following settings:

p	Shear modulus (MPa)	Alpha parameter (1)
1	mu1	alpha1
2	mu2	alpha2

8 Locate the **Quadrature Settings** section. Select the **Reduced integration** check box.

Roller 1

Add symmetry boundary conditions with the **Roller** feature to suppress rigid body motions.

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

2 Select Boundaries 1–3, 6–8, and 11–13 only.

Prescribed Displacement 1

The displacement in the x direction is identical for all load cases.

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

2 Select Boundaries 16–18 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 `stretch-1`.

Prescribed Displacement 2

Add a lateral constraint for the pure shear load case.

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

2 Select Boundary 10 only.

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

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

Prescribed Displacement 3

Finally, add another **Prescribed Displacement** node to prescribe the y displacement in the equibiaxial load case.

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

2 Select Boundary 15 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 `stretch-1`.

MESH 1

Mesh each object with a single hexahedral element.

Mapped 1

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

2 Select Boundaries 1, 6, and 11 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**.

DEFINITIONS

Define global variables for the nominal stress in the three load cases. These can be expressed as volume averages over each domain.

Average 1 (aveop1)

1 In the **Model Builder** window, expand the **Component 1 (comp1)>Definitions** node.

2 Right-click **Definitions** and choose **Nonlocal Couplings>Average**.

3 Select Domain 1 only.

4 In the **Settings** window for **Average**, locate the **Advanced** section.

5 From the **Frame** list, choose **Material (X, Y, Z)**.

6 Right-click **Average 1 (aveop1)** and choose **Duplicate**.

Average 2 (aveop2)

1 In the **Model Builder** window, click **Average 2 (aveop2)**.

2 Select Domain 2 only.

3 Right-click **Average 2 (aveop2)** and choose **Duplicate**.

Average 3 (aveop3)

1 In the **Model Builder** window, click **Average 3 (aveop3)**.

2 Select Domain 3 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
P_ua	aveop1(solid.PxX)	MPa	Uniaxial
P_ps	aveop2(solid.PxX)	MPa	Pure shear
P_eb	aveop3(solid.PxX)	MPa	Equibiaxial

FORWARD PROBLEM

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

2 In the **Settings** window for **Study**, type `Forward Problem` 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 **Forward Problem** click **Step 1: Stationary**.

2 In the **Settings** window for **Stationary**, click to expand 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
stretch (Applied stretch)	range(1.0, 0.05, 2.5)	

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, expand the **Forward Problem>Solver Configurations>Solution 1 (sol1)>Dependent Variables 1** node, then click **Auxiliary pressure (comp1.solid.hmm1.pw)**.

4 In the **Settings** window for **Field**, locate the **Scaling** section.

5 From the **Method** list, choose **Manual**.

6 In the **Scale** text field, type `10[MPa]`.

7 In the **Model Builder** window, under **Forward Problem>Solver Configurations>Solution 1 (sol1)>Dependent Variables 1** click **Displacement field (comp1.u)**.

8 In the **Settings** window for **Field**, locate the **Scaling** section.

9 From the **Method** list, choose **Manual**.

10 In the **Model Builder** window, expand the **Forward Problem>Solver Configurations>Solution 1 (sol1)>Stationary Solver 1** node, then click **Fully Coupled 1**.

11 In the **Settings** window for **Fully Coupled**, click to expand the **Method and Termination** section.

12 From the **Nonlinear method** list, choose **Constant (Newton)**.

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

RESULTS

Compare the model prediction for the initial guess of the material parameters with the experimental stress–stretch curves.

Experimental Data

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

Initial Model Prediction

1 In the **Model Builder** window, under **Results** click **Experimental 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)**.

Initial Model Prediction

1 Right-click **Initial Model Prediction** and choose **Global**.

2 In the **Settings** window for **Global**, type `Initial Model Prediction` in the **Label** text field.

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

Expression	Unit	Description
P_ua	MPa	Uniaxial
P_ps	MPa	Pure shear
P_eb	MPa	Equibiaxial

4 Click to expand the **Coloring and Style** section. From the **Color** list, choose **Cycle (reset)**.

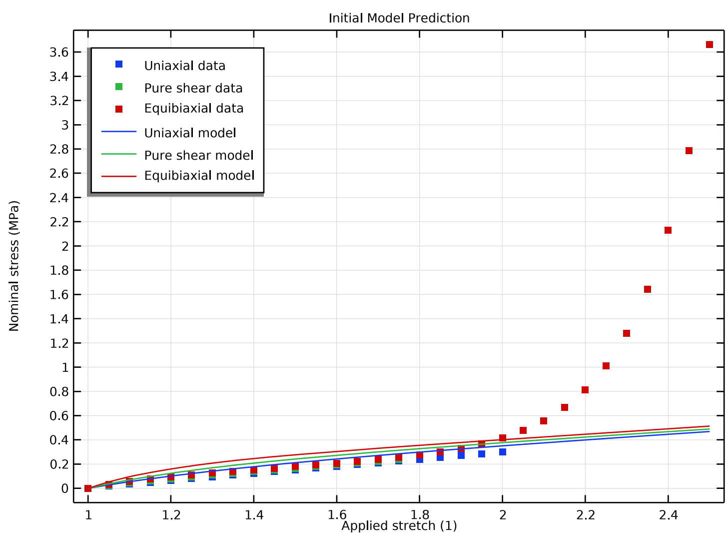
5 Click to expand the **Legends** section. Select the **Show legends** check box.

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

7 In the table, enter the following settings:

Legends
Uniaxial model
Pure shear model
Equibiaxial model

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



The initial model parameters yield a prediction of the uniaxial and pure shear data that is off, but of the correct order of magnitude. However, they fail to capture the strain-stiffening behavior in equibiaxial tension.

COMPONENT 1 (COMP1)

Now, we will set up the **Parameter Estimation** problem. Three **Global Least-Squares Objective** nodes will be added, one for each load case.

Uniaxial Tension Test

1 In the **Physics** toolbar, click **Optimization** and choose **Parameter Estimation**.
2 In the **Settings** window for **Global Least-Squares Objective**, type `Uniaxial Tension Test` in the **Label** text field.
3 Locate the **Experimental Data** section. From the **Data source** list, choose **Result table**.

   The first data column contains the applied stretch, which is the parameter for which the solution needs to be computed.
4 Locate the **Data Column Settings** section. In the table, enter the following settings:

Columns	Type	Settings
Applied stretch (-)	Parameter	Name=stretch

5 From the **Name** list, choose **stretch (Applied stretch)**.
6 In the **Unit** text field, type `1`.

   The second data column contains the nominal stress measured. These are the values that will be used to evaluate the objective in Equation 2.
7 In the table, click to select the cell at row number 2 and column number 1.

   The **Model expression** field is used to set the global variable that should be compared with the experimental data. Use the volume-averaged stress variables that we defined when setting up the forward model.
8 In the **Model expression** text field, type `comp1.P_ua`.
9 In the **Variable name** text field, type `UA`.
10 In the **Unit** text field, type `MPa`.

Pure Shear Test

Proceed in a similar fashion for the two remaining objectives.

1 In the **Parameter Estimation** toolbar, click **Global Least-Squares Objective**.
2 In the **Settings** window for **Global Least-Squares Objective**, type `Pure Shear Test` 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 **Pure Shear Data**.

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

Columns	Type	Settings
Applied stretch (-)	Parameter	Name=stretch

6 From the **Name** list, choose **stretch (Applied stretch)**.

7 In the **Unit** text field, type `1`.

8 In the table, click to select the cell at row number 2 and column number 1.

9 In the **Model expression** text field, type `comp1.P_ps`.

10 In the **Variable name** text field, type `PS`.

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

Equibiaxial Tension Test

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

2 In the **Settings** window for **Global Least-Squares Objective**, type `Equibiaxial Tension Test` 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 **Equibiaxial Data**.

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

Columns	Type	Settings
Applied stretch (-)	Parameter	Name=stretch

6 From the **Name** list, choose **stretch (Applied stretch)**.

7 In the **Unit** text field, type `1`.

8 In the table, click to select the cell at row number 2 and column number 1.

9 In the **Model expression** text field, type `comp1.P_eb`.

10 In the **Variable name** text field, type `EB`.

11 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>Stationary**.

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

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

PARAMETER ESTIMATION

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

2 In the **Settings** window for **Study**, type `Parameter Estimation` 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
mu1 (Ogden modulus, branch 1)	`200[kPa]`	`200[kPa]`	0	
alpha1 (Ogden exponent, branch 1)	`2.0`	1	0	
mu2 (Ogden modulus, branch 2)	`-1.0[kPa]`	`1[kPa]`		0
alpha2 (Ogden exponent, branch 2)	`-2.0`	1		0

Note that the inequality constraint on the Ogden material parameters can be enforced by setting a lower/upper bound of 0 on each μ,α-pair. With the settings entered here, we will force the first pair of parameters to be positive and the second pair to be negative.

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

Step 1: Stationary

Because the forward problem is nonlinear, it is important to activate the continuation solver for the `stretch` parameter by adding an **Auxiliary sweep**. The range given will be merged with the experimental data points, so it is sufficient to specify only the start and end points here.

1 In the **Model Builder** window, 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
stretch (Applied stretch)	`1.0 2.5`	

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>Solver Configurations>Solution 2 (sol2)>Dependent Variables 1** node, then click **Auxiliary pressure (comp1.solid.hmm1.pw)**.

4 In the **Settings** window for **Field**, locate the **Scaling** section.

5 From the **Method** list, choose **Manual**.

6 In the **Scale** text field, type `10[MPa]`.

7 In the **Model Builder** window, under **Parameter Estimation>Solver Configurations>Solution 2 (sol2)>Dependent Variables 1** click **Displacement field (comp1.u)**.

8 In the **Settings** window for **Field**, locate the **Scaling** section.

9 From the **Method** list, choose **Manual**.

10 In the **Model Builder** window, expand the **Parameter Estimation>Solver Configurations>Solution 2 (sol2)>Optimization Solver 1>Stationary 1** node, then click **Fully Coupled 1**.

11 In the **Settings** window for **Fully Coupled**, locate the **Method and Termination** section.

12 From the **Nonlinear method** list, choose **Constant (Newton)**.

RESULTS

Before computing the study, it is often useful to set up a plot that monitors the optimization by comparing the model expression with the experimental data. This type of plot is useful to get visual feedback of the quality of the fit, which can be helpful to detect if the forward model or the solver settings need to be improved.

Parameter Estimation: Ogden Hyperelasticity

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

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

3 Locate the **Data** section. From the **Dataset** list, choose **Parameter Estimation/ Solution 2 (sol2)**.
4 Locate the **Title** section. From the **Title type** list, choose **Label**.
5 Locate the **Plot Settings** section.
6 Select the **x-axis label** check box. In the associated text field, type `Applied stretch (1)`.
7 Select the **y-axis label** check box. In the associated text field, type `Nominal stress (MPa)`.

   Add a second y-axis to be able to visualize all datasets in a single plot. The second axis will be used to plot the much stiffer equibiaxial response.

8 Select the **Two y-axes** check box.
9 Select the **Secondary y-axis label** check box. In the associated text field, type `Nominal stress (MPa)`.
10 Locate the **Legend** section. From the **Position** list, choose **Upper left**.

Initial Model Prediction

The **Table Graphs** displaying the experimental data can be copied from the plot group created to show the initial model prediction.

In the **Model Builder** window, expand the **Results>Initial Model Prediction** node.

Equibiaxial Data, Pure Shear Data, Uniaxial Data

1 In the **Model Builder** window, under **Results>Initial Model Prediction**, Ctrl-click to select **Uniaxial Data**, **Pure Shear Data**, and **Equibiaxial Data**.
2 Right-click and choose **Copy**.

Parameter Estimation: Ogden Hyperelasticity

In the **Model Builder** window, under **Results** right-click **Parameter Estimation: Ogden Hyperelasticity** and choose **Paste Multiple Items**.

Equibiaxial Data

1 In the **Model Builder** window, click **Equibiaxial Data**.
2 In the **Settings** window for **Table Graph**, locate the **y-Axis** section.
3 Select the **Plot on secondary y-axis** check box.
4 In the **Parameter Estimation: Ogden Hyperelasticity** toolbar, click **Plot**.

Uniaxial Model

1 In the **Model Builder** window, right-click **Parameter Estimation: Ogden Hyperelasticity** and choose **Global**.

2 In the **Settings** window for **Global**, type `Uniaxial Model` in the **Label** text field.

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

Expression	Unit	Description
`glso1.UA.model`	`MPa`	`Least-squares model value`

4 Click to expand the **Coloring and Style** section. From the **Color** list, choose **Cycle (reset)**.

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

6 In the table, enter the following settings:

Legends
`Uniaxial model`

7 Right-click **Uniaxial Model** and choose **Duplicate**.

Pure Shear Model

1 In the **Model Builder** window, under **Results>Parameter Estimation: Ogden Hyperelasticity** click **Uniaxial Model 1**.

2 In the **Settings** window for **Global**, type `Pure Shear Model` in the **Label** text field.

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

Expression	Unit	Description
`glso2.PS.model`	`MPa`	`Least-squares model value`

4 Click to expand the **Coloring and Style** section. From the **Color** list, choose **Cycle**.

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

Legends
`Pure shear model`

6 Right-click **Pure Shear Model** and choose **Duplicate**.

Equibiaxial Model

1 In the **Model Builder** window, under **Results>Parameter Estimation: Ogden Hyperelasticity** click **Pure Shear Model 1**.

2 In the **Settings** window for **Global**, type `Equibiaxial Model` in the **Label** text field.

3 Locate the **y-Axis** section. Select the **Plot on secondary y-axis** check box.

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

Expression	Unit	Description
`glso3.EB.model`	`MPa`	`Least-squares model value`

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

Legends
`Equibiaxial model`

Uniaxial Model

A **Filter** subnode can be added to hide the part of the model prediction that extends beyond the data range of the experiments.

Filter 1

1. In the **Model Builder** window, right-click **Uniaxial Model** and choose **Filter**.
2. In the **Settings** window for **Filter**, locate the **Point Selection** section.
3. In the **Logical expression for inclusion** text field, type `stretch<=2`.

Filter 1

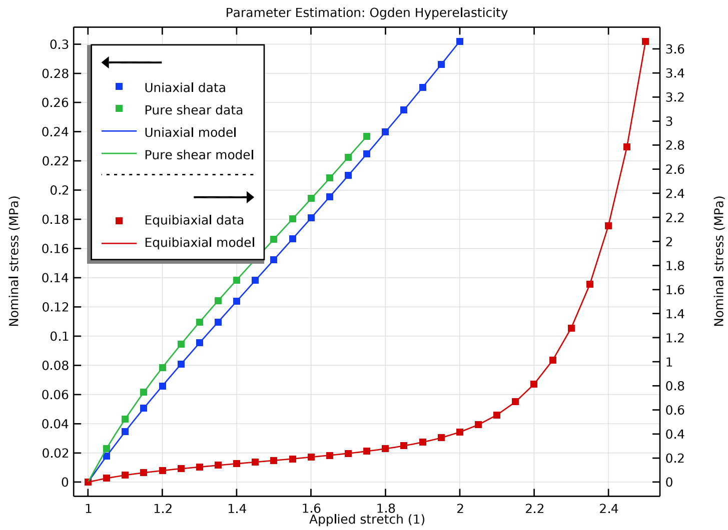
1. In the **Model Builder** window, right-click **Pure Shear Model** and choose **Filter**.
2. In the **Settings** window for **Filter**, locate the **Point Selection** section.
3. In the **Logical expression for inclusion** text field, type `stretch<=1.75`.

PARAMETER ESTIMATION

Parameter Estimation

1. In the **Model Builder** window, under **Parameter Estimation** 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: Ogden Hyperelasticity**.
5. Select the **Show individual objective values** check box.
6. Select the **Table graph** check box.
7. Select the **Compute confidence interval** check box.

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



RESULTS

Finally, collect the estimated material parameters in an **Evaluation Group**. This is available 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/Solution 2 (sol2)>Solid Mechanics>Estimated Parameters (std2)**.

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

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

RESULTS

Estimated Parameters

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

The material model is now calibrated and the final values can be used when simulating the behavior of a real-life component.