Model created in COMSOL Multiphysics 6.4

Parameter Estimation of Viscoplastic Polymers

Introduction

The Bergstrom–Boyce viscoplastic model for engineering rubbers and soft biological tissues has been successful in capturing nonequilibrium material behavior such as strain-rate dependence and hysteresis under cyclic loading (Ref. 1). In order to accurately predict the time-dependent behavior of such materials using finite element models, the Bergstrom–Boyce model needs to be calibrated against a rich set of experimental data. In this example, we demonstrate how to set up a parameter estimation problem in which the material parameters of the original Bergstrom–Boyce model are calibrated against cyclic uniaxial tension and compression data at two different strain rates. Further, it is shown how additional experimental data obtained at elevated temperatures can be used to include temperature dependence in the model formulation. For further information on the mathematical formulation of the inverse problem and the Bergstrom–Boyce implementation in COMSOL Multiphysics, please refer to the models Parameter Estimation of Hyperelastic Materials and Chloroprene Rubber Compression Test in the Nonlinear Structural Mechanics Module Application Library, respectively.

Model Definition

The original Bergstrom–Boyce model consists of two networks: one hyperelastic Arruda–Boyce network that determines the equilibrium behavior in parallel with a second Arruda–Boyce network in series with a viscoplastic component that captures the nonequilibrium viscoplastic flow. Since the networks act in parallel, the total deformation gradient $\mathbf{F}$ is decomposed multiplicatively as $\mathbf{F} = \mathbf{F}_{\mathrm{el}}^{\mathrm{eq}} = \mathbf{F}_{\mathrm{el}}^{\mathrm{neq}}\mathbf{F}_{\mathrm{vp}}^{\mathrm{neq}}$, with $\mathbf{F}_{\mathrm{el}}^{\mathrm{eq}}$, $\mathbf{F}_{\mathrm{el}}^{\mathrm{neq}}$, and $\mathbf{F}_{\mathrm{vp}}^{\mathrm{neq}}$ denoting the elastic deformation gradient in the equilibrium network, the elastic deformation gradient in the nonequilibrium network, and the viscoplastic deformation gradient, respectively.

The strain energy density of the nearly incompressible Arruda–Boyce model reads

$$W_{\mathrm{s}}(\mathrm{C}_{\mathrm{el}}) = \mu_0 \sum_{p=1}^{5} \frac{c_p}{N^{p-1}}(\bar{I}_{1,\,\mathrm{el}}^{p} - 3^p) + W_{\mathrm{vol}}(J_{\mathrm{el}}) \qquad (1)$$

Here, μ_0 is the shear modulus of the network; N is the number of chain segments (related to the locking stretch $\lambda_{\mathrm{lock}} = \sqrt{N}$); c_{p} are known constants derived from a five-term approximation of the inverse Langevin function; and C_{el}, $\bar{I}_{\mathrm{el}}$, and J_{el} denote the right elastic Cauchy–Green deformation tensor of the network, its first isochoric invariant, and the elastic volume ratio, respectively. The shear moduli of the two networks are related by the energy factor β_{v}, such that $\mu_0^{\mathrm{neq}} = \beta_{\mathrm{v}}\mu_0^{\mathrm{eq}}$. In the nearly incompressible formulation, the volumetric strain energy density W_{vol} acts as a penalty term on the volume ratio J_{el} given a large bulk modulus K with respect to the shear modulus of the network.

The viscoplastic flow occurs in the direction of the equivalent stress in the nonequilibrium network, and its magnitude depends on the viscoplastic multiplier λ,

$$\lambda = A(\lambda_{\mathrm{vpe}} - 1 + \varepsilon)^c \left(\frac{\sigma_{\mathrm{vm}}^{\mathrm{neq}}}{\sigma_{\mathrm{res}}}\right)^n \tag{2}$$

Herein, A is the viscoplastic rate coefficient (SI unit: 1/s), $\lambda_{\mathrm{vpe}} = \sqrt{I_{1,\mathrm{vp}}/3}$ is the effective chain stretch in the viscoplastic element, $\varepsilon = 0.001$ is a numerical correction factor to avoid division by zero (Ref. 2), $c \in [-1{,}0]$ is a material parameter controlling the strain hardening, the flow resistance σ_{res} is a parameter introduced for dimensional consistency, and n is the stress hardening exponent. In summary, the six unknown material parameters that need to be estimated from experimental data are μ_0^{eq}, N, β_{v}, A, c, and n, see Table 1. It is worth noting that A and σ_{res} are dependent parameters; here, we choose to fix σ_{res} and only include A in the parameter estimation.

TABLE 1: MATERIAL PARAMETERS OF THE BERGSTROM–BOYCE MODEL, TOGETHER WITH INITIAL VALUES FOR PARAMETER ESTIMATION.

Parameter	**Name**	**Initial guess**	**Estimate?**
Shear modulus, equilibrium network	mu0_eq	`1[MPa]`	yes
Number of chain segments	Nsegm	5	yes
Bulk modulus	K	`1[GPa]`	no
Energy factor	beta	1	yes
Viscoplastic rate coefficient	A	`1[s^-1]`	yes
Strain hardening exponent	c	`-0.5`	yes
Stress hardening exponent	n	5	yes
Temperature hardening exponent	m	10	yes
Flow resistance	sig_res	`sqrt(3)[MPa]`	no

In this example, it is assumed that cyclic uniaxial tension and compression data obtained at room temperature ($T = T_{\mathrm{ref}} = 293.15$ K) for two nominal strain rates, 0.001 1/s and 0.1 1/s, are provided. The maximum nominal strains applied in the tension and compression experiments are 0.6 and –0.3, respectively. The stress–strain curves are reported in Figure 1. Note that the data used here were generated using the Bergstrom–Boyce model with parameter values inspired by Ref. 1; we will therefore compare the final calibrated material parameters with those used to generate the data in order to verify the implementation.

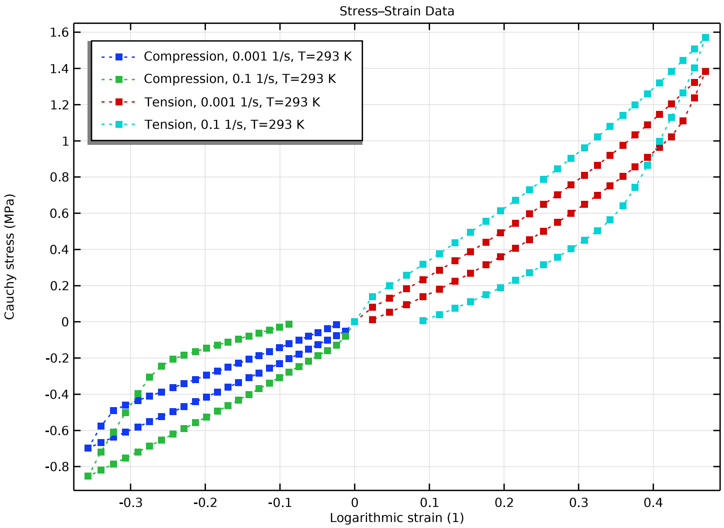


Figure 1: Cyclic uniaxial tension and compression data inspired by Ref. 1.

In a second step, suppose that we in addition to the data in Figure 1 also have performed tension experiments at different temperatures. The temperature-dependent tension data is shown in Figure 2. Adding a temperature dependence to the Bergstrom–Boyce model of elastomeric materials can be done by accounting for (i) the entropic stiffening of the polymer network; (ii) the increase in viscoplastic flow at higher temperatures; and (iii) thermal expansion. Here, the latter is assumed negligible. The entropic stiffening is modeled by noting that, based on the statistical theory of rubber elasticity, the shear modulus is proportional to the absolute temperature, so that we can write

$$\mu_0(T) = \mu_0(T_{\mathrm{ref}})\left(1+\left(\frac{T-T_{\mathrm{ref}}}{T_{\mathrm{ref}}}\right)\right) \tag{3}$$

Thermal effects can be included in the expression for the viscoplastic multiplier by the temperature-dependent function $g(T)$, so that Equation 2 becomes

$$\lambda = A(\lambda_{\mathrm{vpe}} - 1 + \varepsilon)^c \left(\frac{\sigma_{\mathrm{vm}}^{\mathrm{neq}}}{\sigma_{\mathrm{res}}}\right)^n g(T) \tag{4}$$

In this example, a power-law formulation will be used,

$$g(T) = \left(\frac{T}{T_{\mathrm{ref}}}\right)^m \tag{5}$$

with the temperature exponent m that needs to be estimated from experimental data.

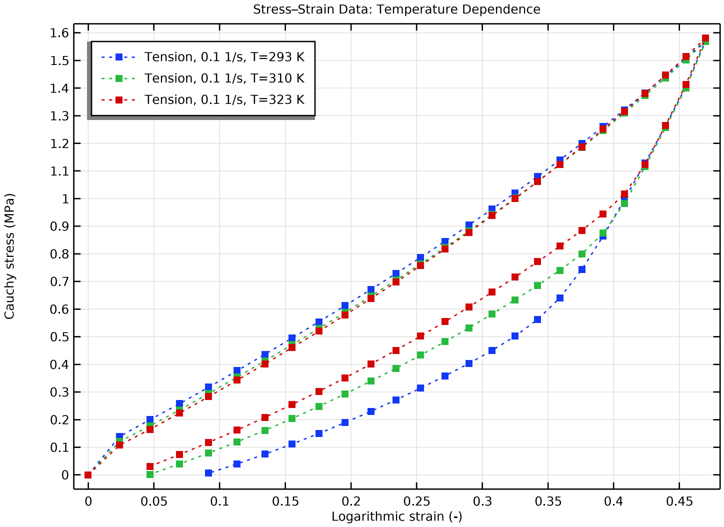


Figure 2: Cyclic tension experiments at three different temperatures.

Results and Discussion

The model prediction for the initial guess of the parameter values in Table 1 is shown in Figure 3. 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 well with those used to generate the data, see Table 2. The same holds for the temperature-dependent cyclic tension data, which is shown together with the final model prediction in Figure 5. Note that, for the values of the material parameters considered here, the viscoplastic flow shows the strongest temperature dependence.

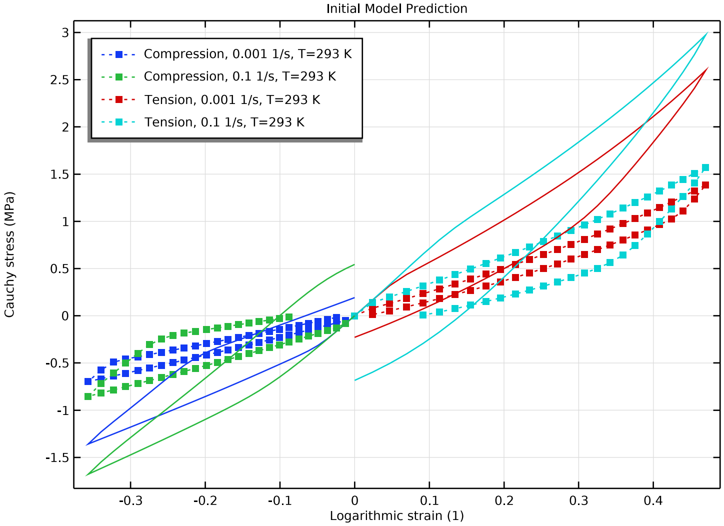


Figure 3: Model prediction with the initial values of the material parameters.

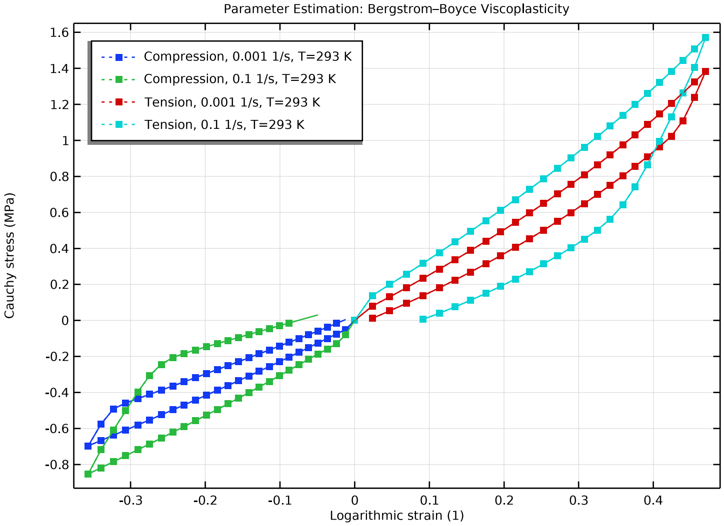


Figure 4: Model prediction with the calibrated material parameters.

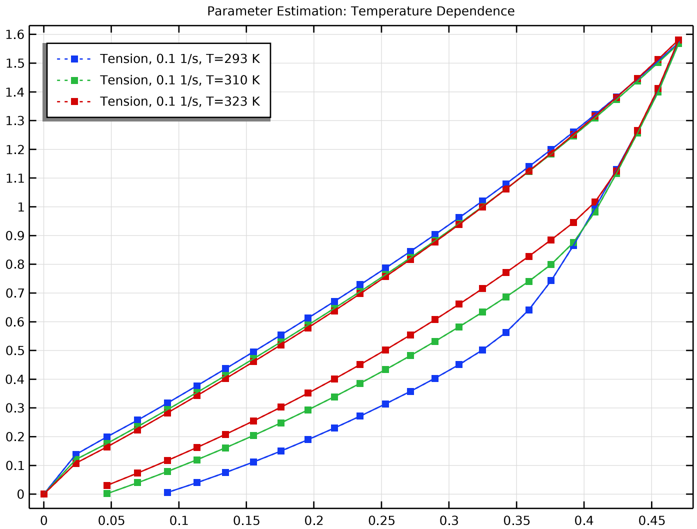


Figure 5: Model prediction of the temperature-dependent cyclic tension data.

TABLE 2: MATERIAL PARAMETERS OF THE CALIBRATED BERGSTROM–BOYCE MODEL.

Parameter	**Name**	**Estimated value**	**Reference value**
Shear modulus, equilibrium network	mu0_eq	`0.60013[MPa]`	`0.6[MPa]`
Number of chain segments	Nsegm	`8.0152`	`8.0`
Energy factor	beta	`2.4981`	`2.5`
Viscoplastic rate coefficient	A	`5.0502[s^-1]`	`5.0[s^-1]`
Strain hardening exponent	c	`-0.99809`	`-1.0`
Stress hardening exponent	n	`4.0004`	`4.0`
Temperature hardening exponent	m	`24.971`	`25.0`

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. This is particularly important when the model is highly nonlinear, for which a robust and efficient solver is required. When the experimental data consists of multiple load cases with different boundary conditions, it can be more efficient to solve the load cases in parallel than in series. This is demonstrated here by creating two unit cube elements, one for the compression test and one for the tension test.

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 is used to link an experimental data file to the corresponding model variables. To solve the inverse problem, the model needs 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.

For most least-squares problems for nonlinear material models, the **Levenberg–Marquardt** algorithm with a finite difference approximation of the Jacobian is a robust and efficient choice of optimization solver. By default, the **Levenberg–Marquardt** solver is set to terminate if either the increment of the (scaled) parameters or the maximum angle between the error vector and the Jacobian is smaller than a given optimality tolerance (default `1e-3`). In the settings of the **Optimization Solver**, you can optionally include an additional termination criterion based on the relative change of the objective function by selecting the **Terminate also for defect reduction** checkbox, which can be useful if the solver reaches a relatively flat local minimum in parameter space where improvements in the objective function are small. The default termination criteria are normally more robust, however, and these will be used here. To monitor the progress of the optimization visually, we will set up a plot while solving that compares the current model prediction with the experimental data.

References

1. J. S. Bergström and M. C. Boyce, “Constitutive modeling of the large strain time-dependent behavior of elastomers,” *J. Mech. Phys. Solids*, vol. 46, pp. 931–954, 1998.

2. J. S. Bergström and M. C. Boyce, “Constitutive modelling of the time-dependent and cyclic loading of elastomers and application to soft biological tissues,” *Mech. Mater.*, vol. 33, pp. 523–530, 2001.

Application Library path: `Nonlinear_Structural_Materials_Module/Viscoplasticity/parameter_estimation_polymer_viscoplasticity`

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 four data files for the experimental data at 293 K.

Compression, 0.001 1/s, T=293 K

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 `Compression, 0.001 1/s, T=293 K` 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_polymer_viscoplasticity_compression_1e-3_T293K.txt`.

Compression, 0.1 1/s, T=293 K

1 Right-click **Compression, 0.001 1/s, T=293 K** and choose **Duplicate**.
2 In the **Settings** window for **Table**, type `Compression, 0.1 1/s, T=293 K` 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_polymer_viscoplasticity_compression_1e-1_T293K.txt`.

Tension, 0.001 1/s, T=293 K

1. Right-click **Compression, 0.1 1/s, T=293 K** and choose **Duplicate**.
2. In the **Settings** window for **Table**, type `Tension, 0.001 1/s, T=293 K` 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_polymer_viscoplasticity_tension_1e-3_T293K.txt`.

Tension, 0.1 1/s, T=293 K

1. Right-click **Tension, 0.001 1/s, T=293 K** and choose **Duplicate**.
2. In the **Settings** window for **Table**, type `Tension, 0.1 1/s, T=293 K` 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_polymer_viscoplasticity_tension_1e-1_T293K.txt`.

For future use, import also the tension data at 310 K and 323 K.

Tension, 0.1 1/s, T=310 K

1. Right-click **Tension, 0.1 1/s, T=293 K** and choose **Duplicate**.
2. In the **Settings** window for **Table**, type `Tension, 0.1 1/s, T=310 K` 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_polymer_viscoplasticity_tension_1e-1_T310K.txt`.

Tension, 0.1 1/s, T=323 K

1. Right-click **Tension, 0.1 1/s, T=310 K** and choose **Duplicate**.
2. In the **Settings** window for **Table**, type `Tension, 0.1 1/s, T=323 K` 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_polymer_viscoplasticity_tension_1e-1_T323K.txt`.

Stress–Strain Data

Plot the tension and compression data at 293 K.

1. In the **Results** toolbar, click ∿ **1D Plot Group**.
2. In the **Settings** window for **1D Plot Group**, type `Stress-Strain 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** checkbox. In the associated text field, type `Logarithmic strain (1)`.
6. Select the **y-axis label** checkbox. In the associated text field, type `Cauchy stress (MPa)`.
7. Locate the **Legend** section. From the **Position** list, choose **Upper left**.

Compression, 0.001 1/s, T=293 K

1. Right-click **Stress–Strain Data** and choose **Table Graph**.
2. In the **Settings** window for **Table Graph**, type `Compression, 0.001 1/s, T=293 K` in the **Label** text field.
3. Locate the **Data** section. From the **x-axis data** list, choose **Logarithmic strain (-)**.
4. From the **Plot columns** list, choose **Manual**.
5. In the **Columns** list box, select **Cauchy stress (MPa)**.
6. Locate the **Coloring and Style** section. Find the **Line style** subsection. From the **Line** list, choose **Dotted**.
7. Find the **Line markers** subsection. From the **Marker** list, choose **Point**.
8. Click to expand the **Legends** section. Select the **Show legends** checkbox.
9. Find the **Include** subsection. Clear the **Headers** checkbox.
10. Select the **Label** checkbox.

Compression, 0.1 1/s, T=293 K

1. Right-click **Compression, 0.001 1/s, T=293 K** and choose **Duplicate**.
2. In the **Settings** window for **Table Graph**, type `Compression, 0.1 1/s, T=293 K` in the **Label** text field.
3. Locate the **Data** section. From the **Table** list, choose **Compression, 0.1 1/s, T=293 K**.

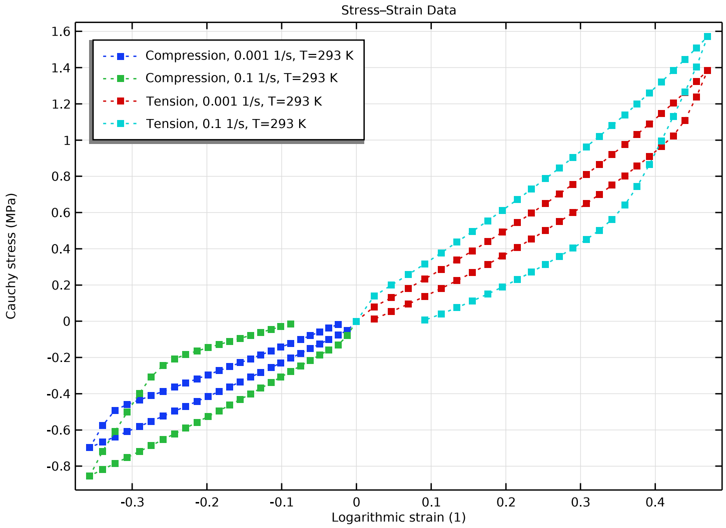
Tension, 0.001 1/s, T=293 K

1. Right-click **Compression, 0.1 1/s, T=293 K** and choose **Duplicate**.

2. In the **Settings** window for **Table Graph**, type `Tension, 0.001 1/s, T=293 K` in the **Label** text field.
3. Locate the **Data** section. From the **Table** list, choose **Tension, 0.001 1/s, T=293 K**.

Tension, 0.1 1/s, T=293 K

1. Right-click **Tension, 0.001 1/s, T=293 K** and choose **Duplicate**.
2. In the **Settings** window for **Table Graph**, type `Tension, 0.1 1/s, T=293 K` in the **Label** text field.
3. Locate the **Data** section. From the **Table** list, choose **Tension, 0.1 1/s, T=293 K**.
4. In the **Stress–Strain Data** toolbar, click **Plot**.



GLOBAL DEFINITIONS

Now, set up the forward model. Start by defining the material parameters.

Material Parameters

1. In the **Model Builder** window, under **Global Definitions** click **Parameters 1**.
2. In the **Settings** window for **Parameters**, type `Material Parameters` in the **Label** text field.

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

Name	**Expression**	**Value**	**Description**
`mu0_eq`	`1[MPa]`	1 MPa	`Shear modulus, equilibrium network`
`Nsegm`	`5`	5	`Number of chain segments`
`K`	`1[GPa]`	1000 MPa	`Bulk modulus`
`beta`	`1`	1	`Energy factor`
`A`	`1[s^-1]`	1 1/s	`Viscoplastic rate coefficient`
`c`	`-0.5`	-0.5	`Strain hardening exponent`
`n`	`5`	5	`Stress hardening exponent`
`m`	`10`	10	`Temperature hardening exponent`
`sig_res`	`sqrt(3)[MPa]`	1.7321 MPa	`Flow resistance`

Load Parameters

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

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

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

Name	**Expression**	**Value**	**Description**
`strain_rate`	`1e-3[s^-1]`	0.001 1/s	`Nominal strain rate`
`t_end`	`1/strain_rate`	1000 s	`Loading duration`
`emax_ten`	`0.6`	0.6	`Nominal strain, tension load case`
`emax_comp`	`-0.3`	-0.3	`Nominal strain, compression load case`
`Tref`	`293.15[K]`	293.15 K	`Reference temperature`
`T`	`293.15[K]`	293.15 K	`Temperature`

Load Cycle

Add a **Triangle** function for prescribing the cyclic loading. The function will be given in terms of dimensionless units so that it can be used for both the tension and compression load case, independent of the strain rate.

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

2 In the **Settings** window for **Triangle**, type `Load Cycle` in the **Label** text field.

3. Locate the **Parameters** section. In the **Lower limit** text field, type `0`.
4. In the **Upper limit** text field, type `2`.
5. Click to expand the **Smoothing** section. Clear the **Size of transition zone** checkbox.

GEOMETRY 1

Create two unit cubes, one for the compression and one for the tension 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 `2`.
5. Locate the **Displacement** section. In the **y** text field, type `2`.
6. In the **Home** toolbar, click **Build All**.
7. Click the **Go to Default View** button in the **Graphics** toolbar.

MATERIALS

Both the **Arruda–Boyce** and **Bergstrom–Boyce Viscoplasticity** material property groups need to be added for defining all material parameters in the Bergstrom–Boyce constitutive model.

Bergstrom–Boyce 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 `Bergstrom-Boyce Material` in the **Label** text field.
3. Click to expand the **Material Properties** section. In the **Material properties** tree, select **Solid Mechanics** > **Hyperelastic Material** > **Arruda–Boyce**.
4. Click **Add to Material**.

5. In the **Material properties** tree, select **Solid Mechanics** > **Viscoplastic Material** > **Bergstrom–Boyce Viscoplasticity**.
6. Click + **Add to Material**.
7. In the **Model Builder** window, expand the **Component 1 (comp1)** > **Materials** > **Bergstrom–Boyce Material (mat1)** node, then click **Arruda–Boyce (ArrudaBoyce)**.
8. In the **Settings** window for **Arruda–Boyce**, locate the **Output Properties** section.
9. In the table, enter the following settings:

Property	**Variable**	**Expression**	**Unit**	**Size**
Number of segments	Nseg	`Nsegm`	1	1x1
Macroscopic shear modulus	mu0	`mu0_eq*(1+(T-Tref)/Tref)`	MPa	1x1

10. In the **Model Builder** window, under **Component 1 (comp1)** > **Materials** > **Bergstrom–Boyce Material (mat1)** click **Bergstrom–Boyce viscoplasticity (BergstromBoyce)**.
11. In the **Settings** window for **Bergstrom–Boyce Viscoplasticity**, locate the **Output Properties** section.
12. In the table, enter the following settings:

Property	**Variable**	**Expression**	**Unit**	**Size**
Viscoplastic rate coefficient	A_BB	`A`	1/s	1x1
Flow resistance	sigRes_BB	`sig_res`	MPa	1x1
Stress exponent	n_BB	`n`	1	1x1
Strain exponent	c_BB	`c`	1	1x1
Cutoff stress	sigmaco_BB	`0[MPa]`	MPa	1x1

SOLID MECHANICS (SOLID)

Continue to set up the Bergstrom–Boyce material model and the uniaxial tension and compression load cases under the **Solid Mechanics** interface. Because the states of stress and strain will be homogeneous, we can use linear shape functions and reduced integration to reduce the computational cost of the forward model.

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 both domains.
3. In the **Settings** window for **Hyperelastic Material**, locate the **Hyperelastic Material** section.
4. From the **Material model** list, choose **Arruda–Boyce**.
5. From the **Volumetric strain energy** list, choose **Miehe**.
6. In the κ text field, type `K`.
7. Locate the **Quadrature Settings** section. Select the **Reduced integration** checkbox.

Polymer Viscoplasticity 1

1. In the **Physics** toolbar, click **Attributes** and choose **Polymer Viscoplasticity**.
2. In the **Settings** window for **Polymer Viscoplasticity**, locate the **Viscoplasticity Model** section.
3. Find the **Hyperelastic element** subsection. In the β_v text field, type `beta`.
4. Locate the **Thermal Effects** section. From the $g(T)$ list, choose **Power law**.
5. In the T_{ref} text field, type `Tref`.
6. In the m text field, type `m`.
7. Locate the **Time Stepping** section. From the **Method** list, choose **Domain ODEs**.

   When the number of degrees of freedom is small, formulating the viscoplastic rate equations in terms of domain ODEs yields better performance than solving them locally.
8. Locate the **Model Input** section. From the T list, choose **User defined**. In the associated text field, type `T`.

Roller 1

Add ideal boundary conditions for the two load cases.

1. In the **Physics** toolbar, click **Boundaries** and choose **Roller**.
2. Select Boundaries 1–3 and 6–8 only.

Uniaxial Compression

1. In the **Physics** toolbar, click **Boundaries** and choose **Prescribed Displacement**.
2. In the **Settings** window for **Prescribed Displacement**, type `Uniaxial Compression` in the **Label** text field.

3 Select Boundary 11 only.

4 Locate the **Prescribed Displacement** section. From the **Displacement in x direction** list, choose **Prescribed**.

5 In the u_{0x} text field, type `emax_comp*tri1(t/t_end)*1[mm]`.

Uniaxial Tension

1 Right-click **Uniaxial Compression** and choose **Duplicate**.

2 In the **Settings** window for **Prescribed Displacement**, type `Uniaxial Tension` in the **Label** text field.

3 Select Boundary 12 only.

4 Locate the **Prescribed Displacement** section. In the u_{0x} text field, type `emax_ten*tri1(t/t_end)*1[mm]`.

MESH 1

Mesh each domain with a single hexahedral element.

Mapped 1

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

2 Select Boundaries 1 and 6 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

Before setting up the solver, add global variables for the volume-averaged Cauchy stress and logarithmic strain in the two load cases. These will be used for the comparison with experimental data.

Average 1 (aveop1)

1. In the **Definitions** toolbar, click **Nonlocal Couplings** and choose **Average**.
2. Select Domain 1 only.

Average 2 (aveop2)

1. Right-click **Average 1 (aveop1)** and choose **Duplicate**.
2. Select Domain 2 only.

Global Stress and Strain Variables

1. In the **Model Builder** window, right-click **Definitions** and choose **Variables**.
2. In the **Settings** window for **Variables**, type `Global Stress and Strain Variables` in the **Label** text field.
3. Locate the **Variables** section. In the table, enter the following settings:

Name	**Expression**	**Unit**	**Description**
`sig_comp`	`aveop1(solid.sxx)`	MPa	`Cauchy stress, compression`
`elog_comp`	`aveop1(solid.elogxx)`		`Logarithmic strain, compression`
`sig_ten`	`aveop2(solid.sxx)`	MPa	`Cauchy stress, tension`
`elog_ten`	`aveop2(solid.elogxx)`		`Logarithmic strain, tension`

FORWARD PROBLEM

Set up a first study to verify that the forward model is defined correctly. A parametric sweep is used to compute the response for the two strain rates.

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** checkbox.

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
strain_rate (Nominal strain rate)	`0.001 0.1`	`1/s`

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(0,0.02,1)*2*t_end`.

Solution 1 (sol1)

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

Add manual scalings to the dependent variables to improve the convergence rate.

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 **Viscoplastic Strain Tensor, Local Coordinate System (comp1.solid.hmm1.pvp1.evp)**.

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

5 In the **Scale** text field, type `1`.

6 In the **Model Builder** window, under **Forward Problem** > **Solver Configurations** > **Solution 1 (sol1)** > **Dependent Variables 1** click **Equivalent Viscoplastic Strain (comp1.solid.hmm1.pvp1.evpe)**.

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

8 In the **Scale** text field, type `1`.

9 In the **Model Builder** window, under **Forward Problem** > **Solver Configurations** > **Solution 1 (sol1)** > **Dependent Variables 1** click **Auxiliary Pressure (comp1.solid.hmm1.pw)**.

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

11 In the **Scale** text field, type `1[MPa]`.

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

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

14 In the **Scale** text field, type `emax_ten`.

15 In the **Model Builder** window, under **Forward Problem** > **Solver Configurations** > **Solution 1 (sol1)** click **Time-Dependent Solver 1**.

16. In the **Settings** window for **Time-Dependent Solver**, click to expand the **Time Stepping** section.
17. From the **Steps taken by solver** list, choose **Strict**.

    When solving inelastic problems, strict time stepping is often required to avoid interpolation errors.
18. Find the **Algebraic variable settings** subsection. From the **Consistent initialization** list, choose **Off**.

    The boundary and initial conditions in the model are consistent, so **Consistent initialization** can be disabled.
19. In the **Model Builder** window, under **Forward Problem** > **Solver Configurations** > **Solution 1 (sol1)** > **Time-Dependent Solver 1** click **Fully Coupled 1**.
20. In the **Settings** window for **Fully Coupled**, click to expand the **Method and Termination** section.
21. From the **Nonlinear method** list, choose **Constant (Newton)**.

    The constant Newton solver is a fast and robust choice for nonlinear problems.
22. From the **Jacobian update** list, choose **Once per time step**.
23. In the **Maximum number of iterations** text field, type `25`.
24. In the **Study** toolbar, click = **Compute**.

RESULTS

Compare the initial model prediction with the experimental data.

Initial Model Prediction

1. In the **Model Builder** window, right-click **Stress–Strain Data** and choose **Duplicate**.
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/ Parametric Solutions 1 (sol2)**.

Initial Prediction, Compression

1. Right-click **Initial Model Prediction** and choose **Global**.
2. In the **Settings** window for **Global**, type `Initial Prediction, Compression` in the **Label** text field.

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

Expression	**Unit**	**Description**
`sig_comp`	`MPa`	`Cauchy stress, compression`

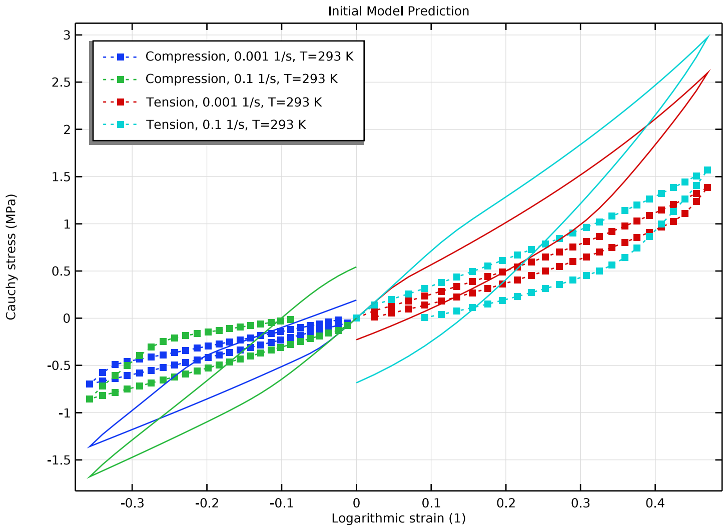
4. Locate the **x-Axis Data** section. From the **Parameter** list, choose **Expression**.
5. In the **Expression** text field, type `elog_comp`.
6. Click to expand the **Coloring and Style** section. From the **Color** list, choose **Cycle (reset)**.
7. Click to expand the **Legends** section. Clear the **Show legends** checkbox.

Initial Prediction, Tension

1. Right-click **Initial Prediction, Compression** and choose **Duplicate**.
2. In the **Settings** window for **Global**, type `Initial Prediction, Tension` in the **Label** text field.
3. Locate the **y-Axis Data** section. In the table, enter the following settings:

Expression	**Unit**	**Description**
`sig_ten`	`MPa`	`Cauchy stress, tension`

4. Locate the **x-Axis Data** section. In the **Expression** text field, type `elog_ten`.
5. Locate the **Coloring and Style** section. From the **Color** list, choose **Cycle**.
6. In the **Initial Model Prediction** toolbar, click **Plot**.



COMPONENT 1 (COMP1)

Now, move on to the inverse problem. For this, we need to define four **Least-Squares Objectives**, one for each experiment at 293 K.

Compression, 0.001 1/s, T=293 K

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

2 In the **Settings** window for **Least-Squares Objective**, type `Compression, 0.001 1/s, T=293 K` in the **Label** text field.

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

In the **Column Settings** section, we need to provide definitions for each column in the data file. Here, the first column contains the time, the second contains the logarithmic strain, and the third contains Cauchy stress data. Since the tests are displacement controlled, we will only use the stress data for the evaluation of the objective function and assume that the strain is measured with negligible error. The `Logarithmic strain` column is therefore ignored.

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

Columns	Type	Settings
Logarithmic strain (-)	Ignored column	

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

In the **Model expression** field, enter the name of the global variable we defined earlier for the Cauchy stress in the compression load case.

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

7 In the **Column name** text field, type `comp_slow`.

The **Variable name** is used to refer to variables for the data and the model expression in postprocessing.

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

In the **Experimental Conditions** section, specify the value of the model parameters that are constant for this particular dataset, but varies between datasets. In this example, we want to add the parameters `strain_rate` and `T` and set them to the values corresponding to the experiment. Since the temperature in these four experiments is constant and equal to the value given under **Global Definitions**, we do not need to specify it explicitly here.

9 Locate the **Experimental Conditions** section. Click + **Add**.

10 In the table, enter the following settings:

Name	Expression
strain_rate (Nominal strain rate)	`0.001[s^-1]`

Compression, 0.1 1/s, T=293 K

Continue in a similar fashion with the remaining three datasets.

1. Right-click **Compression, 0.001 1/s, T=293 K** and choose **Duplicate**.
2. In the **Settings** window for **Least-Squares Objective**, type `Compression, 0.1 1/s, T=293 K` in the **Label** text field.
3. Locate the **Experimental Data** section. From the **Result table** list, choose **Compression, 0.1 1/s, T=293 K**.
4. Locate the **Data Column Settings** section. In the table, click to select the cell at row number 3 and column number 1.
5. In the **Column name** text field, type `comp_fast`.
6. Locate the **Experimental Conditions** section. In the table, enter the following settings:

Name	Expression
strain_rate (Nominal strain rate)	`0.1[s^-1]`

Tension, 0.001 1/s, T=293 K

1. Right-click **Compression, 0.1 1/s, T=293 K** and choose **Duplicate**.
2. In the **Settings** window for **Least-Squares Objective**, type `Tension, 0.001 1/s, T=293 K` in the **Label** text field.
3. Locate the **Experimental Data** section. From the **Result table** list, choose **Tension, 0.001 1/s, T=293 K**.
4. Locate the **Data Column Settings** section. In the table, click to select the cell at row number 3 and column number 1.
5. In the **Model expression** text field, type `comp1.sig_ten`.
6. In the **Column name** text field, type `ten_slow`.
7. Locate the **Experimental Conditions** section. In the table, enter the following settings:

Name	Expression
strain_rate (Nominal strain rate)	`0.001[s^-1]`

Tension, 0.1 1/s, T=293 K

1. Right-click **Tension, 0.001 1/s, T=293 K** and choose **Duplicate**.

2. In the **Settings** window for **Least-Squares Objective**, type `Tension, 0.1 1/s, T=293 K` in the **Label** text field.
3. Locate the **Experimental Data** section. From the **Result table** list, choose **Tension, 0.1 1/s, T=293 K**.
4. Locate the **Data Column Settings** section. In the table, click to select the cell at row number 3 and column number 1.
5. In the **Column name** text field, type `ten_fast`.
6. Locate the **Experimental Conditions** section. In the table, enter the following settings:

Name	Expression
strain_rate (Nominal strain rate)	`0.1[s^-1]`

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.

PARAMETER ESTIMATION

1. In the **Settings** window for **Study**, type `Parameter Estimation` 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** checkboxes for **Compression, 0.001 1/s, T=293 K**, **Compression, 0.1 1/s, T=293 K**, **Tension, 0.001 1/s, T=293 K**, and **Tension, 0.1 1/s, T=293 K**.
5. Locate the **Estimated Parameters** section. Click + **Add** six times.

6 In the table, enter the following settings:

Parameter	Initial value	Scale	Lower bound	Upper bound	Unit
mu0_eq (Shear modulus, equilibrium network)	`1[MPa]`	1	0		MPa
Nsegm (Number of chain segments)	5	1	2		
beta (Energy factor)	1	1	0		
A (Viscoplastic rate coefficient)	`1[s^-1]`	1	0		1/s
c (Strain hardening exponent)	`-0.5`	1	`-1`	0	
n (Stress hardening exponent)	5	1	`1.01`	20	

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**.

Solution 5 (sol5)

Generate the solver sequence and edit the default solver settings as we did for the first study.

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 **Parameter Estimation** > **Solver Configurations** > **Solution 5 (sol5)** > **Dependent Variables 1** node, then click **Viscoplastic Strain Tensor, Local Coordinate System (comp1.solid.hmm1.pvp1.evp)**.

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

5 In the **Scale** text field, type `1`.

6 In the **Model Builder** window, under **Parameter Estimation** > **Solver Configurations** > **Solution 5 (sol5)** > **Dependent Variables 1** click **Equivalent Viscoplastic Strain (comp1.solid.hmm1.pvp1.evpe)**.

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

8 In the **Scale** text field, type `1`.

9 In the **Model Builder** window, under **Parameter Estimation** > **Solver Configurations** > **Solution 5 (sol5)** > **Dependent Variables 1** click **Auxiliary Pressure (comp1.solid.hmm1.pw)**.

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

11 In the **Scale** text field, type `1`.

12 In the **Model Builder** window, under **Parameter Estimation** > **Solver Configurations** > **Solution 5 (sol5)** > **Dependent Variables 1** click **Displacement Field (comp1.u)**.

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

14 In the **Scale** text field, type `emax_ten`.

15 In the **Model Builder** window, expand the **Parameter Estimation** > **Solver Configurations** > **Solution 5 (sol5)** > **Optimization Solver 1** node, then click **Time-Dependent Solver 1**.

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

17 From the **Steps taken by solver** list, choose **Strict**.

18 Find the **Algebraic variable settings** subsection. From the **Consistent initialization** list, choose **Off**.

19 In the **Model Builder** window, expand the **Parameter Estimation** > **Solver Configurations** > **Solution 5 (sol5)** > **Optimization Solver 1** > **Time-Dependent Solver 1** node, then click **Fully Coupled 1**.

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

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

22 From the **Jacobian update** list, choose **Once per time step**.

23 In the **Maximum number of iterations** text field, type `25`.

RESULTS

Before solving the study, create a plot comparing the model prediction with the experimental data to monitor the progress of the optimization solver visually. Start by copying the plot group **Stress–Strain Data** that already contains all experimental data curves.

Parameter Estimation: Bergstrom–Boyce Viscoplasticity

1 In the **Model Builder** window, right-click **Stress–Strain Data** and choose **Duplicate**.

2 In the **Settings** window for **1D Plot Group**, type `Parameter Estimation: Bergstrom-Boyce Viscoplasticity` in the **Label** text field.

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

Model Prediction, Compression 0.001 1/s

1 Right-click **Parameter Estimation: Bergstrom–Boyce Viscoplasticity** and choose **Global**.

2 In the **Settings** window for **Global**, type `Model Prediction, Compression 0.001 1/s` in the **Label** text field.

Each **Least-Squares Objective** feature defines a variable `lso.variable_name.model`, `lso` being the tag of the feature and `variable_name` the string entered in the **Variable name** field of the **Value** column, that contains the corresponding model expression.

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

Expression	**Unit**	**Description**
`lso1.comp_slow.model`	`MPa`	`Least-squares model value`

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

5 In the **Expression** text field, type `elog_comp`.

6 Locate the **Coloring and Style** section. From the **Color** list, choose **Cycle (reset)**.

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

Model Prediction, Compression 0.1 1/s

1 Right-click **Model Prediction, Compression 0.001 1/s** and choose **Duplicate**.

2 In the **Settings** window for **Global**, type `Model Prediction, Compression 0.1 1/s` in the **Label** text field.

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

Expression	**Unit**	**Description**
`lso2.comp_fast.model`	`MPa`	`Least-squares model value`

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

Model Prediction, Tension 0.001 1/s

1 Right-click **Model Prediction, Compression 0.1 1/s** and choose **Duplicate**.

2 In the **Settings** window for **Global**, type `Model Prediction, Tension 0.001 1/s` in the **Label** text field.

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

Expression	**Unit**	**Description**
`lso3.ten_slow.model`	`MPa`	`Least-squares model value`

4 Locate the **x-Axis Data** section. In the **Expression** text field, type `elog_ten`.

Model Prediction, Tension 0.1 1/s

1. Right-click **Model Prediction, Tension 0.001 1/s** and choose **Duplicate**.
2. In the **Settings** window for **Global**, type `Model Prediction, Tension 0.1 1/s` in the **Label** text field.
3. Locate the **y-Axis Data** section. In the table, enter the following settings:

Expression	**Unit**	**Description**
`lso4.ten_fast.model`	`MPa`	`Least-squares model value`

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** section.
3. Select the **Plot** checkbox.
4. In the table, enter the following settings:

Plot group	**Plot window**
Parameter Estimation: Bergstrom-Boyce Viscoplasticity	Graphics

5. Select the **Show individual objective values** checkbox.
6. Select the **Table graph** checkbox.
7. In the **Study** toolbar, click = **Compute**.

SOLID MECHANICS (SOLID)

The calibrated material parameters of the Bergstrom–Boyce model can now be copied from the **Objective Probe Table**, or accessed within the model using the `withsol` operator. We will use the latter to create a new **Hyperelastic Material** with the calibrated material properties. This is needed for the next step where we will estimate the temperature dependence.

Hyperelastic Material 2

1. In the **Model Builder** window, under **Component 1 (comp1)** > **Solid Mechanics (solid)** right-click **Hyperelastic Material 1** and choose **Duplicate**.
2. In the **Settings** window for **Hyperelastic Material**, locate the **Hyperelastic Material** section.
3. From the μ_0 list, choose **User defined**. In the associated text field, type `withsol('sol5', mu0_eq)*(1+(T-Tref)/Tref)`.

4. From the N list, choose **User defined**. In the associated text field, type `withsol('sol5', Nsegm)`.

Polymer Viscoplasticity 1

1. In the **Model Builder** window, expand the **Hyperelastic Material 2** node, then click **Polymer Viscoplasticity 1**.
2. In the **Settings** window for **Polymer Viscoplasticity**, locate the **Viscoplasticity Model** section.
3. Find the **Hyperelastic element** subsection. In the β_v text field, type `withsol('sol5', beta)`.
4. Find the **Inelastic element** subsection. From the A list, choose **User defined**. In the associated text field, type `withsol('sol5', A)`.
5. From the σ_{res} list, choose **User defined**. In the associated text field, type `sig_res`.
6. From the n list, choose **User defined**. In the associated text field, type `withsol('sol5', n)`.
7. From the σ_{co} list, choose **User defined**. Find the **Isotropic hardening model** subsection. From the c list, choose **User defined**. In the associated text field, type `withsol('sol5', c)`.

RESULTS

Plot the temperature-dependent tension data.

Stress–Strain Data: Temperature Dependence

1. In the **Results** toolbar, click **1D Plot Group**.
2. In the **Settings** window for **1D Plot Group**, type `Stress-Strain Data: Temperature Dependence` in the **Label** text field.
3. Locate the **Data** section. From the **Dataset** list, choose **None**.
4. Locate the **Title** section. From the **Title type** list, choose **Label**.
5. Locate the **Legend** section. From the **Position** list, choose **Upper left**.

Tension, 0.1 1/s, T=293 K

1. Right-click **Stress–Strain Data: Temperature Dependence** and choose **Table Graph**.
2. In the **Settings** window for **Table Graph**, locate the **Data** section.
3. From the **Table** list, choose **Tension, 0.1 1/s, T=293 K**.
4. From the **x-axis data** list, choose **Logarithmic strain (-)**.
5. From the **Plot columns** list, choose **Manual**.

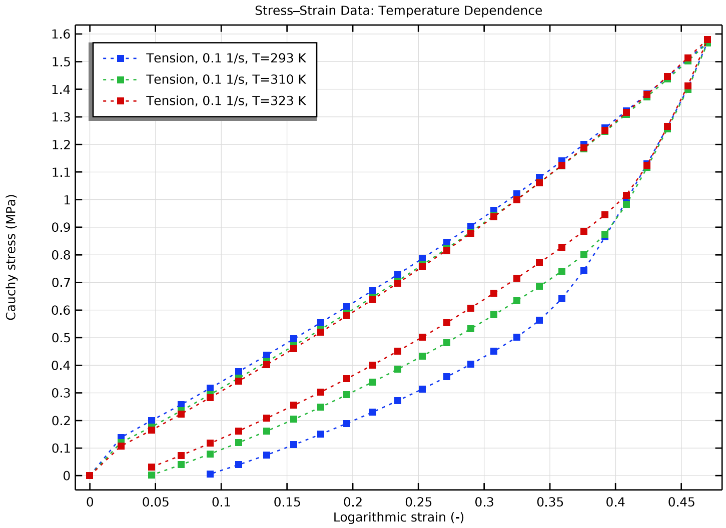
6. In the **Columns** list box, select **Cauchy stress (MPa)**.
7. Locate the **Coloring and Style** section. Find the **Line style** subsection. From the **Line** list, choose **Dotted**.
8. Find the **Line markers** subsection. From the **Marker** list, choose **Point**.
9. Locate the **Legends** section. Select the **Show legends** checkbox.
10. Find the **Include** subsection. Select the **Label** checkbox.
11. Clear the **Headers** checkbox.
12. In the **Label** text field, type `Tension, 0.1 1/s, T=293 K`.

Tension, 0.1 1/s, T=310 K

1. Right-click **Tension, 0.1 1/s, T=293 K** and choose **Duplicate**.
2. In the **Settings** window for **Table Graph**, type `Tension, 0.1 1/s, T=310 K` in the **Label** text field.
3. Locate the **Data** section. From the **Table** list, choose **Tension, 0.1 1/s, T=310 K**.

Tension, 0.1 1/s, T=323 K

1. Right-click **Tension, 0.1 1/s, T=310 K** and choose **Duplicate**.
2. In the **Settings** window for **Table Graph**, type `Tension, 0.1 1/s, T=323 K` in the **Label** text field.
3. Locate the **Data** section. From the **Table** list, choose **Tension, 0.1 1/s, T=323 K**.
4. In the **Stress–Strain Data: Temperature Dependence** toolbar, click **Plot**.



PARAMETER ESTIMATION

Add new **Least-Squares Objectives** and a **Parameter Estimation** study to fit the temperature dependence.

Tension, 0.1 1/s, T=310 K

1 In the **Model Builder** window, right-click **Tension, 0.1 1/s, T=293 K** and choose **Duplicate**.

2 In the **Settings** window for **Least-Squares Objective**, type `Tension, 0.1 1/s, T=310 K` in the **Label** text field.

3 Locate the **Experimental Data** section. From the **Result table** list, choose **Tension, 0.1 1/s, T=310 K**.

4 Locate the **Data Column Settings** section. In the table, click to select the cell at row number 3 and column number 1.

5 In the **Column name** text field, type `ten_fast_310K`.

6 Locate the **Experimental Conditions** section. Click + **Add**.

7 In the table, enter the following settings:

Name	Expression
T (Temperature)	`310.15[K]`

Tension, 0.1 1/s, T=323 K

1 Right-click **Tension, 0.1 1/s, T=310 K** and choose **Duplicate**.

2 In the **Settings** window for **Least-Squares Objective**, type `Tension, 0.1 1/s, T=323 K` in the **Label** text field.

3 Locate the **Experimental Data** section. From the **Result table** list, choose **Tension, 0.1 1/s, T=323 K**.

4 Locate the **Data Column Settings** section. In the table, click to select the cell at row number 3 and column number 1.

5 In the **Column name** text field, type `ten_fast_323K`.

6 Locate the **Experimental Conditions** section. In the table, enter the following settings:

Name	Expression
T (Temperature)	`323.15[K]`

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.

PARAMETER ESTIMATION: TEMPERATURE DEPENDENCE

1 In the **Settings** window for **Study**, type `Parameter Estimation: Temperature Dependence` 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**.

The only remaining unknown material parameter is the temperature hardening exponent `m`. To estimate its value, it is sufficient to use the last three objectives.

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
Compression, 0.001 1/s, T=293 K	
Compression, 0.1 1/s, T=293 K	
Tension, 0.001 1/s, T=293 K	
Tension, 0.1 1/s, T=293 K	√
Tension, 0.1 1/s, T=310 K	√
Tension, 0.1 1/s, T=323 K	√

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

6 In the table, enter the following settings:

Parameter	Initial value	Scale	Lower bound	Upper bound	Unit
m (Temperature hardening exponent)	10	10			

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

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 (solid), Controls spatial frame** > **Hyperelastic Material 1**.

5 Click ⊘ **Disable**.

Solution 6 (sol6)

Generate the solver sequence and edit the default solver settings as we did for the first study, but modify the finite difference interval for the Jacobian computation in the **Optimization Solver** to `1e-3`. Using a larger finite difference step can be necessary if the sensitivity to the estimated parameters is low. Refer to the previous study for the exact step-by-step instructions.

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

2 Click = **Compute**.

RESULTS

Plot the model prediction of the temperature-dependent data.

Parameter Estimation: Temperature Dependence

1 In the **Model Builder** window, right-click **Stress–Strain Data: Temperature Dependence** and choose **Duplicate**.

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

3 Locate the **Data** section. From the **Dataset** list, choose **Parameter Estimation: Temperature Dependence/Solution 6 (sol6)**.

Model Prediction, Tension 0.1 1/s, T=293 K

1 Right-click **Parameter Estimation: Temperature Dependence** and choose **Global**.

2 In the **Settings** window for **Global**, type `Model Prediction, Tension 0.1 1/s, T=293 K` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Parameter Estimation: Temperature Dependence/Solution 6 (sol6)**.

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

5 In the **Parameter values (T)** list box, select **293.15**.

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

Expression	**Unit**	**Description**
`lso4.ten_fast.model`	`MPa`	`Least-squares model value`

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

8 In the **Expression** text field, type `elog_ten`.

9 Locate the **Coloring and Style** section. From the **Color** list, choose **Cycle (reset)**.

10 Locate the **Legends** section. Clear the **Show legends** checkbox.

Model Prediction, Tension 0.1 1/s, T=310 K

1 Right-click **Model Prediction, Tension 0.1 1/s, T=293 K** and choose **Duplicate**.

2 In the **Settings** window for **Global**, type `Model Prediction, Tension 0.1 1/s, T=310 K` in the **Label** text field.

3 Locate the **Data** section. In the **Parameter values (T)** list box, select **310.15**.

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

Expression	**Unit**	**Description**
`lso5.ten_fast_310K.model`	`MPa`	`Least-squares model value`

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

Model Prediction, Tension 0.1 1/s, T=323 K

1 Right-click **Model Prediction, Tension 0.1 1/s, T=310 K** and choose **Duplicate**.

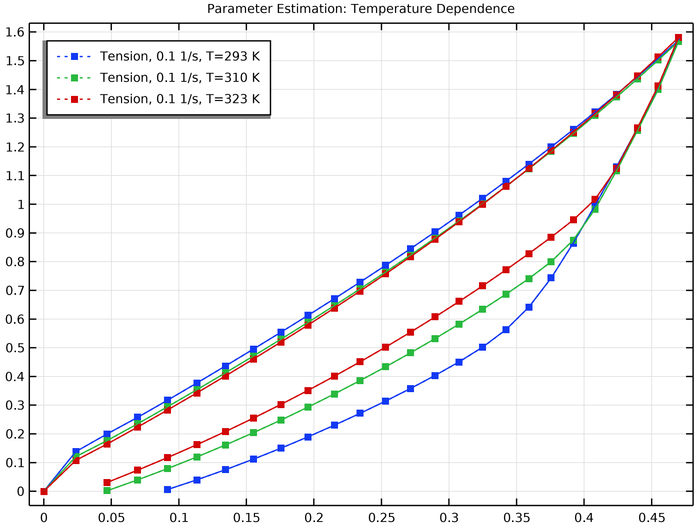
2 In the **Settings** window for **Global**, type `Model Prediction, Tension 0.1 1/s, T=323 K` in the **Label** text field.

3 Locate the **Data** section. In the **Parameter values (T)** list box, select **323.15**.

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

Expression	**Unit**	**Description**
`lso6.ten_fast_323K.model`	`MPa`	`Least-squares model value`

5 In the **Parameter Estimation: Temperature Dependence** toolbar, click **Plot**.



RESULT TEMPLATES

Finish up by collecting the complete set of calibrated material parameters 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 **Parameter Estimation/Solution 5 (sol5)** > **Solid Mechanics** > **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

Calibrated Material Parameters

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

Global Evaluation 2

1. Right-click **Calibrated Material Parameters** and choose **Global Evaluation**.
2. In the **Settings** window for **Global Evaluation**, locate the **Data** section.

3 From the **Dataset** list, choose **Parameter Estimation: Temperature Dependence/ Solution 6 (sol6)**.

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

5 From the **Parameter selection (strain_rate)** list, choose **Last**.

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

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

Expression	**Unit**	**Description**
`m`		`Temperature hardening exponent`

8 In the **Calibrated Material Parameters** toolbar, click = **Evaluate**.