Model created in COMSOL Multiphysics 6.4

Large-Strain Poroviscoelastic Model of Brain Tissue

Introduction

This model demonstrates how to set up a fully coupled poroviscoelastic model of biological tissue. The model is benchmarked by simulating a cyclic uniaxial tension–compression test on human brain tissue as reported by Greiner and others (Ref. 1).

The brain tissue is modeled as a biphasic material with incompressible solid and fluid constituents based on the theory of poroelasticity. Although the constituents are incompressible, the volume of the tissue can change due to fluid inflow or outflow.

The large-strain viscoelastic model of the solid phase is based on a compressible Ogden strain energy density function. Fluid permeation is modeled by Darcy's law with a deformation-dependent permeability tensor.

Model Definition

A cyclic tension–compression test is performed on a cylindrical sample of 4 mm radius and height. Axial symmetry is taken into account. Three displacement-controlled tension–compression loading cycles are applied to the top boundary, while the lower boundary is fixed. Both the top and bottom of the cylinder are considered impermeable, while free fluid flow is allowed through the radial surfaces.

Under quasistatic conditions, the momentum balance equation reads

$$\nabla \cdot \boldsymbol{P} = 0 \tag{1}$$

where $\boldsymbol{P}$ is the first Piola–Kirchhoff stress tensor of the mixture,

$$\boldsymbol{P} = \boldsymbol{P}_{\mathrm{eq}}^{\mathrm{vol}} + \boldsymbol{P}_{\mathrm{eq}}^{\mathrm{iso}} + \boldsymbol{P}_{\mathrm{neq}}^{\mathrm{iso}} - Jp\boldsymbol{F}^{-\mathrm{T}} \tag{2}$$

which includes volumetric and isochoric equilibrium contributions, isochoric nonequilibrium contributions from viscoelasticity, and a contribution from the pore pressure. The mass-balance equation is given by

$$\begin{aligned} \rho_{\mathrm{f}}\frac{\partial J}{\partial t} + \nabla \cdot (\rho_{\mathrm{f}}\mathbf{v}_{\mathrm{D}}) = 0 \\ \mathbf{v}_{\mathrm{D}} = -\frac{1}{\mu}K(J)\nabla p \end{aligned} \tag{3}$$

where J is the volume ratio, $K(J)$ is the deformation-dependent permeability tensor, p is the pore pressure, ρ_{f} the fluid density, and $\mathbf{v}_{\mathrm{D}}$ is the Darcy velocity.

Results and Discussion

Figure 1 shows the pore pressure and Darcy velocity (arrows) at two different times: at $t = 15$ s, when the sample is compressed (left), and at $t = 45$ s, when the sample is stretched (right).

The pore pressure increases under compression and results in fluid outflow (red arrows) and volume decrease (left). Conversely, the pore pressure is negative under tension, indicating fluid inflow (blue arrows) and volume increase (right).

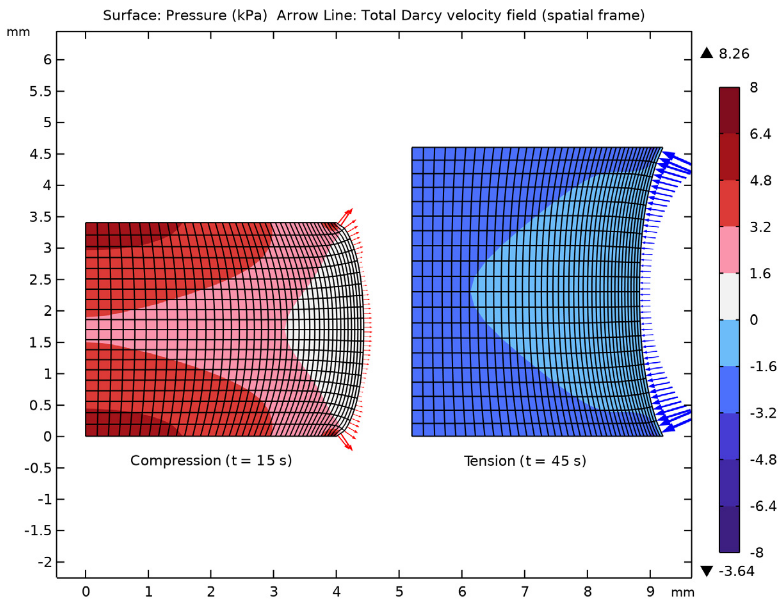

Figure 1: Pore pressure and Darcy velocity under compression (left) and tension (right).

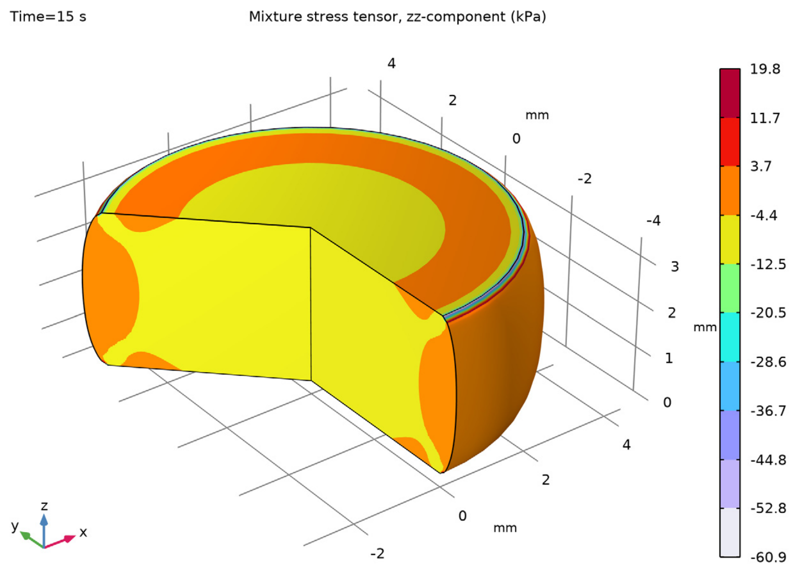

Figure 2: Mixture stress tensor, zz-component, at t = 15 s, when the sample is compressed.

Figure 2 and Figure 3 show the *zz*-component of the stress tensor for the compression case (after 15 s) and the tension case (after 45 s), respectively.

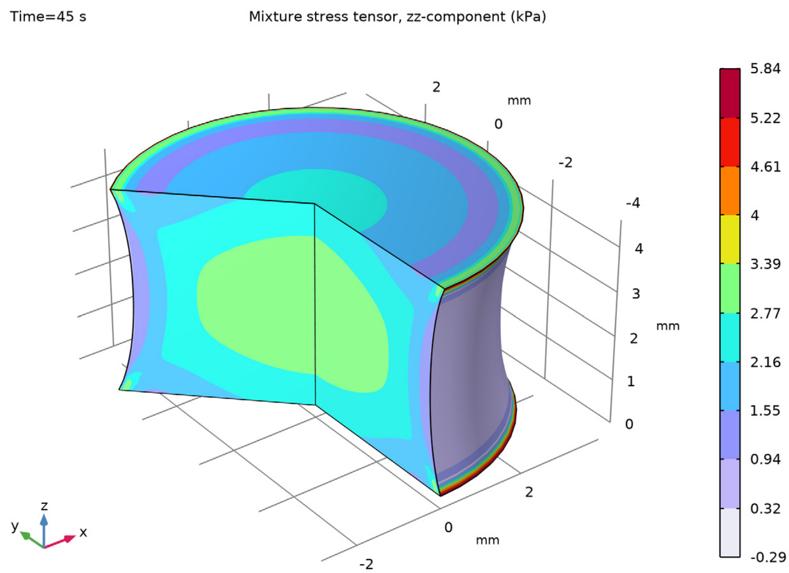

Figure 3: Mixture stress tensor, zz-component, at t = 45 s, when the sample is stretched.

The nominal stress is plotted against the applied stretch in Figure 4. The stress–stretch curve is in good agreement with Figure 3 in Ref. 1.

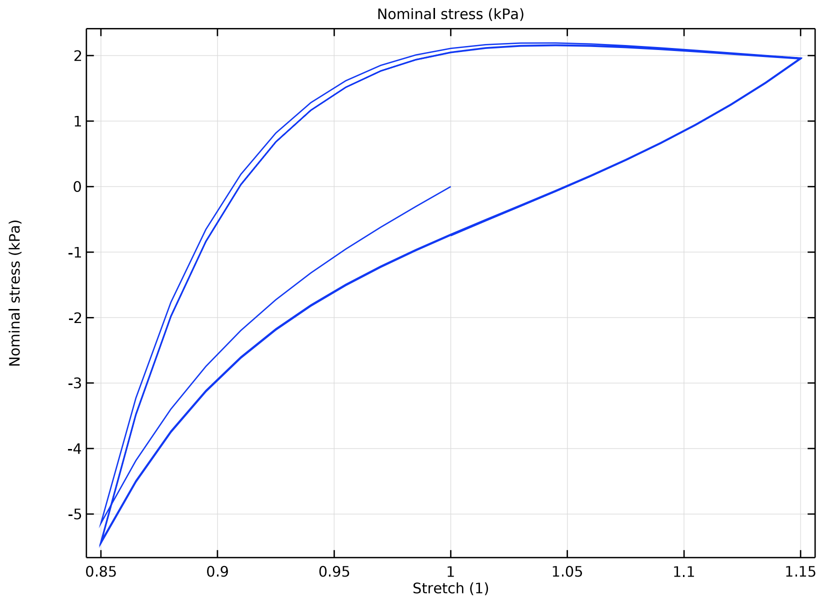

Figure 4: Stress–stretch curve for the uniaxial tension–compression test.

Notes About the COMSOL Implementation

The geometry and displacement field are interpolated using quadratic serendipity shape functions, combined with linear shape functions for the pore pressure p. The Ogden strain energy density is used in its compressible, uncoupled form.

The pore pressure acts as a load contribution in the momentum balance, and the volumetric changes in pore space contribute to the mass balance of fluid. These multiphysics effects are automatically included with the **Poroelasticity** node.

Reference

1. A. Greiner, N. Reiter, F. Paulsen, G.A. Holzapfel, P. Steinmann, E. Comellas, and S. Budday, “Poro-Viscoelastic Effects During Biomechanical Testing of Human Brain Tissue,” *Front. Mech. Eng.*, vol. 7, 708350, 2021.

Application Library path: `Porous_Media_Flow_Module/Poromechanics/brain_poroviscoelasticity`

Modeling Instructions

From the **File** menu, choose **New**.

NEW

In the **New** window, click **Model Wizard**.

MODEL WIZARD

1 In the **Model Wizard** window, click **2D Axisymmetric**.

2 In the **Select Physics** tree, select **Structural Mechanics** > **Poroelasticity** > **Poroelasticity, 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** to set up the model according to Ref. 1.

GLOBAL DEFINITIONS

Next, define the model parameters. You can load them from an external file.

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 Click **Load from File**.

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

Cyclic Loading

Define a **Waveform** function for applying the cyclic load.

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

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

3 Locate the **Parameters** section. From the **Type** list, choose **Triangle**.

4 Clear the **Smoothing** checkbox.

5 In the T text field, type `4*T`.

6 In the A text field, type `-umax`.

GEOMETRY 1

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**.

Rectangle 1 (r1)

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

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

3 In the **Width** text field, type `radius`.

4 In the **Height** text field, type `height`.

5 Click **Build All Objects**.

6 Click the **Zoom Extents** button in the **Graphics** toolbar.

DEFINITIONS

Define variables for the volumetric strain energy density and the deformation-dependent permeability tensor.

Permeability Variables

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

2 Right-click **Definitions** and choose **Variables**.

3 In the **Settings** window for **Variables**, type `Permeability Variables` in the **Label** text field.

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

Name	Expression	Unit	Description
kdef	k0*((solid.J-phiS)/(1-phiS))^n	mm²	Deformation-dependent intrinsic permeability
KRR	solid.J*kdef*solid.Cil11	mm²	Permeability tensor, RR-component
KRZ	solid.J*kdef*solid.Cil13	mm²	Permeability tensor, RZ-component
KZZ	solid.J*kdef*solid.Cil33	mm²	Permeability tensor, ZZ-component

COMPONENT 1 (COMP1)

Now, set up the physics. Use quadratic serendipity shape functions for the geometry and the displacement field, combined with linear shape functions for the pore pressure.

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

2 In the **Settings** window for **Component**, locate the **Curved Mesh Elements** section.

3 From the **Geometry shape function** list, choose **Quadratic serendipity**.

SOLID MECHANICS (SOLID)

Hyperelastic Material 1

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

2 Select Domain 1 only.

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 **Compressible, uncoupled**.

6 From the **Volumetric strain energy** list, choose **User defined**.

7 In the W_{svol} text field, type `lame1*(1-phiS)^2*((solid.Jel-1)/(1-phiS)-log((solid.Jel-phiS)/(1-phiS)))`.

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

p	Shear modulus (MPa)	Alpha parameter (1)
1	mu1inf	alpha1

Viscoelasticity 1

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

2 In the **Settings** window for **Viscoelasticity**, locate the **Viscoelasticity Model** section.

3 In the table, enter the following settings:

Branch	Energy factor (1)	Relaxation time (s)
1	beta	tau

Fixed Constraint 1

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

2 Select Boundary 2 only.

Prescribed Displacement 1

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

2 Select Boundary 3 only.

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

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

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

6 In the u_{0z} text field, type `wv1(t)`.

DARCY'S LAW (DL)

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Darcy's Law (dl)**.

2 In the **Settings** window for **Darcy's Law**, click to expand the **Discretization** section.

3 From the **Pressure** list, choose **Linear**.

Fluid 1

1 In the **Model Builder** window, under **Component 1 (comp1)** > **Darcy's Law (dl)** > **Porous Medium 1** click **Fluid 1**.

2 In the **Settings** window for **Fluid**, locate the **Fluid Properties** section.

3 From the **Fluid type** list, choose **Incompressible**.

Porous Matrix 1

1 In the **Model Builder** window, click **Porous Matrix 1**.

2 In the **Settings** window for **Porous Matrix**, locate the **Matrix Properties** section.

3 From the κ list, choose **User defined**. From the list, choose **Symmetric**.

4 Specify the κ matrix as

KRR	KRZ
KR Z	KZZ

Pressure 1

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

2 Select Boundary 4 only.

MULTIPHYSICS

Poroelasticity 1 (poro1)

1 In the **Model Builder** window, under **Component 1 (comp1)** > **Multiphysics** click **Poroelasticity 1 (poro1)**.

2 In the **Settings** window for **Poroelasticity**, locate the **Poroelastic Coupling Properties** section.

3 From the **Poroelasticity model** list, choose **Biphasic**.

4 From the p_{ref} list, choose **Reference pressure level (dl)**.

MATERIALS

Having defined the physics, add a **Porous Material** and the required material properties.

Porous Material 1 (pmat1)

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

2 In the **Settings** window for **Porous Material**, locate the **Phase-Specific Properties** section.

3 Click **Add Required Phase Nodes**.

Fluid 1 (pmat1.fluid1)

1 In the **Model Builder** window, click **Fluid 1 (pmat1.fluid1)**.

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

3 In the table, enter the following settings:

Property	Variable	Value	Unit	Property group
Density	rho	rhoF	t/mm³	Basic
Dynamic viscosity	mu	muF	t/(mm·s)	Basic

Solid 1 (pmat1.solid1)

1 In the **Model Builder** window, right-click **Porous Material 1 (pmat1)** and choose **Solid**.
2 In the **Settings** window for **Solid**, locate the **Solid Properties** section.
3 In the θ_s text field, type `phiS`.

Porous Material 1 (pmat1)

1 In the **Model Builder** window, click **Porous Material 1 (pmat1)**.
2 In the **Settings** window for **Porous Material**, locate the **Homogenized Properties** section.
3 In the table, enter the following settings:

Property	Variable	Value	Unit	Property group
Density	rho	`1[g/cm^3]`	t/mm³	Basic
Bulk modulus	K	`lame1+mu1inf*alpha1/3`	N/m²	Bulk modulus and shear modulus

MESH 1

Mapped 1

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

Distribution 1

1 Right-click **Mapped 1** and choose **Distribution**.
2 Select Boundaries 2 and 3 only.
3 In the **Settings** window for **Distribution**, locate the **Distribution** section.
4 From the **Distribution type** list, choose **Predefined**.
5 In the **Number of elements** text field, type `40`.
6 In the **Element ratio** text field, type `4`.
7 From the **Growth rate** list, choose **Exponential** to refine the mesh toward the lateral surface of the cylinder, which alternately serves as inflow or outflow boundary.

Distribution 2

1 In the **Model Builder** window, right-click **Mapped 1** and choose **Distribution**.
2 Select Boundaries 1 and 4 only.
3 In the **Settings** window for **Distribution**, locate the **Distribution** section.
4 In the **Number of elements** text field, type `20`.
5 Click **Build All**.

STUDY 1

Step 1: Time Dependent

1 In the **Model Builder** window, under **Study 1** 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 `T*range(0,0.1,12)`.

Before solving the model, modify the default solver settings for faster convergence.

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

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

6 Clear the **Generate default plots** checkbox.

Solution 1 (sol1)

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

2 In the **Model Builder** window, expand the **Solution 1 (sol1)** node, then click **Time-Dependent Solver 1**.

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

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

5 Find the **Algebraic variable settings** subsection. From the **Error estimation** list, choose **Exclude algebraic**.

6 In the **Model Builder** window, expand the **Study 1** > **Solver Configurations** > **Solution 1 (sol1)** > **Time-Dependent Solver 1** node, then click **Direct**.

7 In the **Settings** window for **Direct**, locate the **General** section.

8 From the **Solver** list, choose **PARDISO**.

9 In the **Model Builder** window, under **Study 1** > **Solver Configurations** > **Solution 1 (sol1)** > **Time-Dependent Solver 1** click **Fully Coupled 1**.

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

11 From the **Jacobian update** list, choose **On every iteration**.

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

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

RESULTS

Plot the pore pressure distribution and the Darcy velocity during both compression and tension. Here, it is suitable to use kPa as preferred unit for both stresses and pressures.

Preferred Units 1

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

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

3 Click **Add Physical Quantity**.

4 In the **Physical Quantity** dialog, type `stress` in the text field.

5 In the tree, select **Solid Mechanics > Stress tensor (MPa)**.

6 Click **OK**.

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

8 In the table, enter the following settings:

Quantity	Unit	Preferred unit
Stress tensor	MPa	kPa

9 Select the **Apply conversions to expressions with the same dimensions** checkbox.

RESULT TEMPLATES

1 In the **Home** toolbar, click **Windows** and choose **Result Templates**.

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

3 In the tree, select **Study 1/Solution 1 (sol1) > Darcy's Law > Pressure (dl)**.

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

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

RESULTS

Pressure (dl)

1 In the **Settings** window for **2D Plot Group**, locate the **Plot Settings** section.

2 Clear the **Plot dataset edges** checkbox.

3 Locate the **Color Legend** section. Select the **Show maximum and minimum values** checkbox.

Pore Pressure

1 In the **Model Builder** window, expand the **Pressure (dl)** node, then click **Surface**.

2 In the **Settings** window for **Surface**, type `Pore Pressure` in the **Label** text field.

3 Click to expand the **Range** section. Select the **Manual color range** checkbox.

4 In the **Minimum** text field, type `-8`.

5 In the **Maximum** text field, type `8`.

6 Locate the **Coloring and Style** section. From the **Color table** list, choose **Wave**.

7 In the **Number of bands** text field, type `10`.

Deformation 1

1 Right-click **Pore Pressure** and choose **Deformation**.

2 In the **Settings** window for **Deformation**, locate the **Scale** section.

3 Select the **Scale factor** checkbox. In the associated text field, type `1`.

Pore Pressure

Add a **Solution Array** node to plot the pore pressure at two output times side by side in the Graphics window.

Solution Array 1

1 Right-click **Pore Pressure** and choose **Solution Array**.

2 In the **Settings** window for **Solution Array**, locate the **Data** section.

3 From the **Time selection** list, choose **From list**.

4 In the **Times (s)** list, choose **15** and **45**.

Darcy Velocity (compression)

1 In the **Model Builder** window, right-click **Pressure (dl)** and choose **Arrow Line**.

2 In the **Settings** window for **Arrow Line**, type `Darcy Velocity (compression)` in the **Label** text field.

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

4 From the **Time (s)** list, choose **15**.

5 Locate the **Expression** section. In the **R-component** text field, type `dl.u`.

6 In the **Z-component** text field, type `dl.w`.

7 Locate the **Arrow Positioning** section. In the **Number of arrows** text field, type `50`.

8 Locate the **Coloring and Style** section. From the **Arrow length** list, choose **Logarithmic**.

9 Select the **Scale factor** checkbox. In the associated text field, type `7e2`.

10 Click to expand the **Inherit Style** section. From the **Plot** list, choose **Pore Pressure**.

11 Clear the **Arrow scale factor** checkbox.

12 Clear the **Color** checkbox.

13 Clear the **Color and data range** checkbox.

14 Click to expand the **Plot Array** section. Select the **Manual indexing** checkbox.

Selection 1

1 Right-click **Darcy Velocity (compression)** and choose **Selection**.

2 Select Boundary 4 only.

Deformation 1

In the **Model Builder** window, right-click **Darcy Velocity (compression)** and choose **Deformation**.

Darcy Velocity (tension)

1 Right-click **Darcy Velocity (compression)** and choose **Duplicate**.

2 In the **Settings** window for **Arrow Line**, type `Darcy Velocity (tension)` in the **Label** text field.

3 Locate the **Data** section. From the **Time (s)** list, choose **45**.

4 Click to expand the **Title** section. From the **Title type** list, choose **None**.

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

6 Locate the **Plot Array** section. In the **Index** text field, type `1`.

7 Locate the **Coloring and Style** section. From the **Arrow base** list, choose **Head**.

Mesh 1

1 In the **Model Builder** window, right-click **Pressure (dl)** and choose **Mesh**.

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

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

4 From the **Time (s)** list, choose **15**.

5 Click to expand the **Title** section. From the **Title type** list, choose **None**.

6 Locate the **Coloring and Style** section. From the **Element color** list, choose **None**.

7 Click to expand the **Inherit Style** section. From the **Plot** list, choose **Pore Pressure**.

8 Click to expand the **Plot Array** section. Select the **Manual indexing** checkbox.

Deformation 1

Right-click **Mesh 1** and choose **Deformation**.

Mesh 2

1 In the **Model Builder** window, under **Results** > **Pressure (dl)** right-click **Mesh 1** and choose **Duplicate**.

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

3 From the **Time (s)** list, choose **45**.

4 Locate the **Plot Array** section. In the **Index** text field, type `1`.

Pressure (dl)

1 In the **Model Builder** window, click **Pressure (dl)**.

2 In the **Settings** window for **2D Plot Group**, click to expand the **Title** section.

3 Find the **Solution** subsection. Clear the **Solution** checkbox.

Table Annotation 1

1 In the **Pressure (dl)** toolbar, click **More Plots** and choose **Table Annotation**.

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

3 From the **Source** list, choose **Local table**.

4 In the table, enter the following settings:

x-coordinate	y-coordinate	Annotation
2	-0.4	Compression (t = 15 s)
7	-0.4	Tension (t = 45 s)

5 Locate the **Coloring and Style** section. From the **Anchor point** list, choose **Center**.

6 Clear the **Show point** checkbox.

7 In the **Pressure (dl)** toolbar, click **Plot**.

8 Click the **Zoom Extents** button in the **Graphics** toolbar.

RESULT TEMPLATES

Next, create a 3D plot of the stress in the sample by modifying a plot from the **Result Templates** menu.

1 In the **Home** toolbar, click **Windows** and choose **Result Templates**.

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

3 In the tree, select **Study 1/Solution 1 (sol1)** > **Solid Mechanics** > **Stress, 3D (solid)**.

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

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

RESULTS

Stress, 3D (solid)

1 In the **Model Builder** window, click **Stress, 3D (solid)**.

2 In the **Settings** window for **3D Plot Group**, locate the **Data** section.

3 From the **Time (s)** list, choose **15**.

Surface 1

1 In the **Model Builder** window, expand the **Results** > **Stress, 3D (solid)** > **Surface 1** node, then click **Surface 1**.

2 In the **Settings** window for **Surface**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1 (comp1)** > **Solid Mechanics** > **Stress** > **Mixture stress tensor (spatial frame) - MPa** > **poro1.sGpzz - Mixture stress tensor, zz-component**.

3 Locate the **Coloring and Style** section. From the **Color table type** list, choose **Discrete**.

4 In the **Stress, 3D (solid)** toolbar, click **Plot**.

5 Click the **Zoom Extents** button in the **Graphics** toolbar.

Stress, 3D (solid)

You have now plotted the *zz*-component of the mixture stress tensor during compression. For the tension case, choose another output time.

1 In the **Model Builder** window, click **Stress, 3D (solid)**.

2 In the **Settings** window for **3D Plot Group**, locate the **Data** section.

3 From the **Time (s)** list, choose **45**.

4 In the **Stress, 3D (solid)** toolbar, click **Plot**.

5 Click the **Zoom Extents** button in the **Graphics** toolbar.

Stress-Stretch Curve

Finally, plot the nominal stress-stretch curve as is typically measured in an experiment.

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

2 In the **Settings** window for **1D Plot Group**, type `Stress-Stretch Curve` in the **Label** text field.

3 Locate the **Grid** section. Select the **Manual spacing** checkbox.

4 In the **x spacing** text field, type `0.05`.

5 Locate the **Legend** section. Clear the **Show legends** checkbox.

Global 1

1 Right-click **Stress-Stretch Curve** and choose **Global**.

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

3 In the table, enter the following settings:

Expression	Unit	Description
`solid.disp1.RFsumz/(pi*radius^2)`	`kPa`	`Nominal stress`

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

5 In the **Expression** text field, type `1+wv1(t)/height`.

6 Select the **Description** checkbox. In the associated text field, type `Stretch`.

7 In the **Stress-Stretch Curve** toolbar, click **Plot**.