Model created in COMSOL Multiphysics 6.4

Linear Biphasic Poroelasticity

Introduction

This example demonstrates how to use the Poroelasticity multiphysics coupling between the Solid Mechanics and Darcy's Law interfaces to model linear biphasic poroviscoelastic behavior of soft biological tissues. Soft biological tissues are often modeled as biphasic materials with incompressible solid and fluid constituents. Note that even if both constituents are modeled as incompressible, the porous medium can still undergo large volume changes as a result of fluid inflow or outflow.

The implementation is verified using two numerical benchmarks from the literature, a torsion test and an indentation test on cylindrical plugs of articular cartilage.

Model Definition

The model consists of two components, one for the first benchmark test which is a torsion test, one for the second, which is an indentation test.

The first benchmark test, the infinitesimal torsion test, is used to determine the intrinsic viscoelastic behavior of the solid matrix under shear. The volume is preserved, so no pore pressure nor fluid flow develops. The design, a cylinder of 3 mm radius and 1.5 mm height, and operating conditions are taken from Ref. 1. Figure 1 shows a part of this cylinder. The arrows mark the tangential prescribed displacement. Due to rotational symmetry the problem can be reduced to modeling only the symmetry plane.

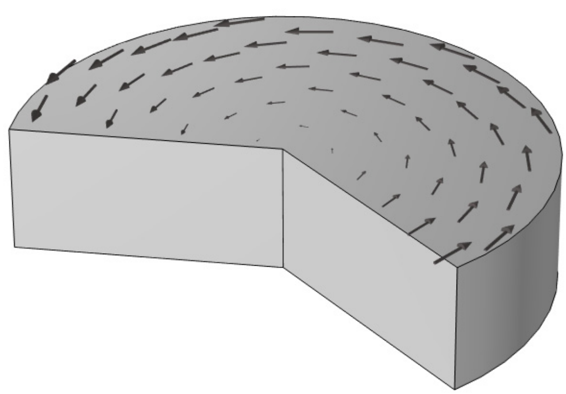

Figure 1: Full geometry and prescribed displacement of the torsion test (model component 1). The simulation is performed on the symmetry plane in 2D.

The second benchmark test is the indentation test, where the pore pressure contributes to the load, and volumetric changes in pore space affect the mass balance equation for the fluid. Again, due to rotational symmetry the simulation can be reduced to modeling only the symmetry plane. Figure 2 shows the model geometry and the prescribed indentation load.

Figure 2: Model geometry and indentation load for the second benchmark test. The simulation in performed on the symmetry plane in 2D.

In both cases, a linear isotropic viscoelastic material is used. The total stress σ in the tissue consists of the effective stress σ_e in the porous matrix and the pore pressure p_f where

$$\sigma = \sigma_e - p_f \mathbf{I} . \quad (1)$$

The effective stress contains both elastic and inelastic contributions; in this example, deviatoric and volumetric viscoelasticity are included based on Ref. 1. The viscoelastic behavior is described with a generalized Maxwell model.

The fluid permeation through the porous matrix is modeled by Darcy's law.

$$\mathbf{u} = -\frac{\kappa}{\mu}\nabla p_f \quad (2)$$

where μ (Pa·s) is the fluid viscosity, κ (m^2) the permeability of the porous substrate, and p_f the pore pressure.

Results and Discussion

Figure 3 and Figure 4 show the results of the infinitesimal torsion test. Figure 3 shows the von Mises stress distribution over the surface of the cylindrical articular cartilage (color table) and the displacement, which is shown by the deformation of the cylinder. Stress and displacement are increasing with increasing distance from the rotation axis.

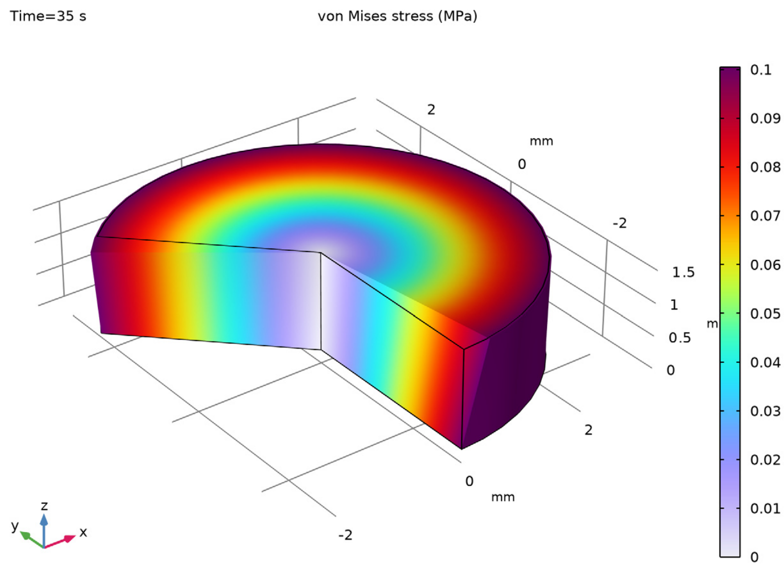

Figure 3: von Mises Stress (MPa) (colored) and tangential displacement (visible as deformation) along the surface of the cylindrical articular cartilage.

The torsional moment as a function of time is shown in Figure 4. The results calculated with COMSOL Multiphysics are shown in green, whereas the results of Ehlers and Markert (Ref. 1) are displayed as blue circles. The progression of the torsional moment indicates the characteristic fast stress relaxation of the organic solid matrix of cartilage tissues.

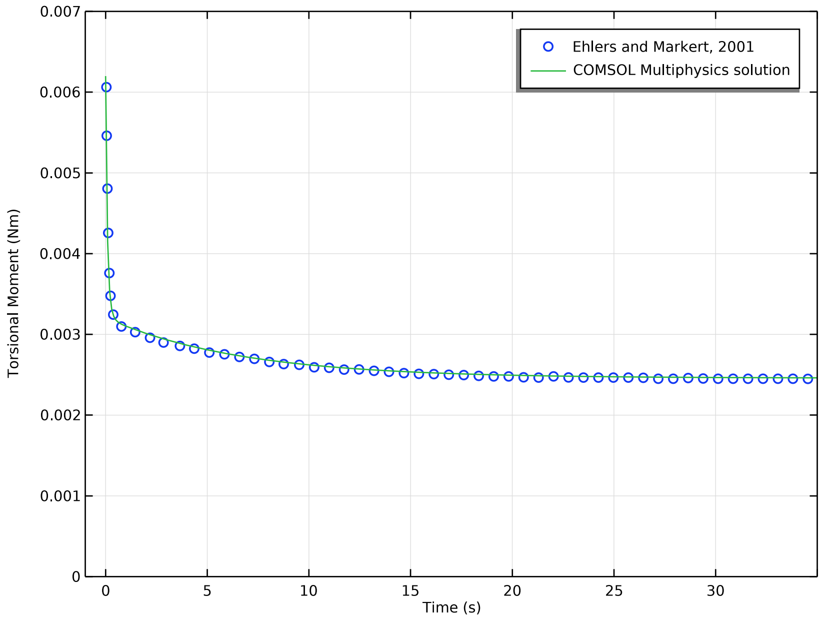

Figure 4: Torsional moment of the infinitesimal torsion test. The blue circles mark the results of Ehlers and Markert (2001), the green curve the results calculated with COMSOL Multiphysics.

[Figure 5](#) and [Figure 6](#) show the results of the second comparison, the indentation test.

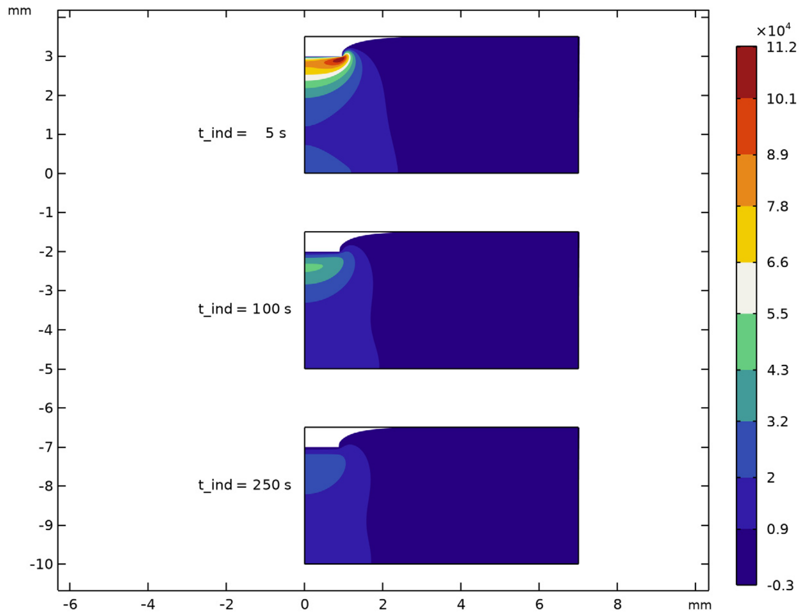

Figure 5: Pore pressure at different loading ramp times.

Figure 5 displays the pore pressure within the model domain for different loading times. The deformation-dependent effect is restricted to the loaded area close to the singularity. In this region, the fluid flow is restrained causing a high local pore pressure. The indenter reaction force is shown in Figure 6. Both figures, especially Figure 6, show that the influence of the viscoelastic properties of the porous matrix decreases by increasing the loading time.

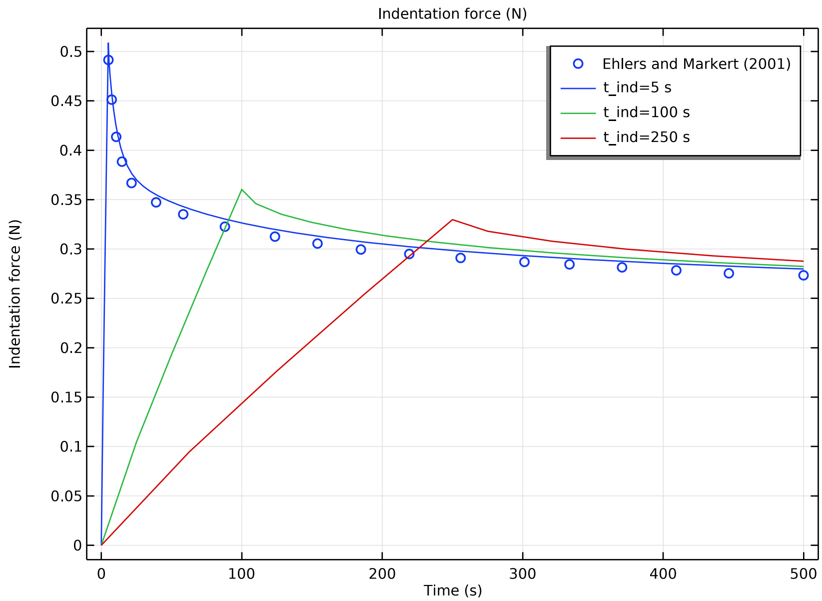

Figure 6: Indentation force as a function of time for different loading times (5 s, 100 s, and 250 s).

Reference

1. W. Ehlers and B. Markert, “A Linear Viscoelastic Biphasic Model for Soft Tissues Based on the Theory of Porous Media,” *J. Biomech. Eng.*, vol. 123, pp. 418–424, 2001.

Application Library path: `Porous_Media_Flow_Module/Verification_Examples/linear_biphasic_poroelasticity`

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 > Solid Mechanics (solid)**.
3 Click **Add**.
4 Click **Study**.
5 In the **Select Study** tree, select **General Studies > Time Dependent**.
6 Click **Done**.

GLOBAL DEFINITIONS

Load the parameters from separate external files for material parameters, torsion test parameters, and indentation test parameters. First, import the material parameters, which were defined according to Ref. 1.

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. Click **Load from File**.
4 Browse to the model's Application Libraries folder and double-click the file `linear_biphasic_poroelasticity_parameters.txt`.

Torsion Parameters

1 In the **Home** toolbar, click **Parameters** and choose **Add > Parameters**.
2 In the **Settings** window for **Parameters**, type `Torsion Parameters` in the **Label** text field.
3 Locate the **Parameters** section. Click **Load from File**.
4 Browse to the model's Application Libraries folder and double-click the file `linear_biphasic_poroelasticity_torsion.txt`.

Indentation Parameters

1 In the **Home** toolbar, click **Parameters** and choose **Add > Parameters**.
2 In the **Settings** window for **Parameters**, type `Indentation Parameters` in the **Label** text field.
3 Locate the **Parameters** section. Click **Load from File**.
4 Browse to the model's Application Libraries folder and double-click the file `linear_biphasic_poroelasticity_indentation.txt`.

In the next step, define the materials as global materials so that you can refer to them from different model components. Introduce them as empty material nodes; as soon as the physics has been defined the material node's context menu will show you which properties are needed for the simulation. You can then just fill in the values defined in the **Parameters** list.

Solid Phase

1 In the **Model Builder** window, under **Global Definitions** right-click **Materials** and choose **Blank Material**.

2 In the **Settings** window for **Material**, type `Solid Phase` in the **Label** text field.

Fluid Phase

1 Right-click **Materials** and choose **Blank Material**.

2 In the **Settings** window for **Material**, type `Fluid Phase` in the **Label** text field.

Add a step function to use for applying the instantaneous twist in the torsion test case.

Step 1 (step1)

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

2 In the **Settings** window for **Step**, click to expand the **Smoothing** section.

3 Clear the **Size of transition zone** checkbox.

4 Click to expand the **Plot Parameters** section.

For the indentation test case, add a ramp function to ramp up the indentation displacement.

Ramp 1 (rm1)

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

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

3 Select the **Cutoff** checkbox.

4 Click to expand the **Smoothing** section. Click to expand the **Plot Parameters** section.

Now, set up the torsion load case.

COMPONENT 1: TORSION TEST

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

2 In the **Settings** window for **Component**, type `Component 1: Torsion Test` in the **Label** text field.

GEOMETRY 1

1 In the **Model Builder** window, under **Component 1: Torsion Test (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_t`.

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

5 Click **Build Selected**.

Form Union (fin)

In the **Geometry** toolbar, click **Build All**.

MATERIALS

The volume is preserved in the torsion test case. Therefore, no pore pressure is developed and no fluid flow occurs, so only a solid material definition is needed. Link the local material to the globally defined one.

Material Link 1 (matlnk1)

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

SOLID MECHANICS (SOLID)

1 In the **Settings** window for **Solid Mechanics**, locate the **Axial Symmetry Approximation** section.

2 Select the **Include circumferential displacement** checkbox.

3 Locate the **Structural Transient Behavior** section. From the list, choose **Quasistatic**.

Linear Elastic Material 1

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

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

3 From the **Specify** list, choose **Lamé parameters**.

Viscoelasticity, Deviatoric

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

2 In the **Settings** window for **Viscoelasticity**, type `Viscoelasticity, Deviatoric` in the **Label** text field.

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

Branch	Shear modulus (Pa)	Relaxation time (s)
1	mu1	tau1_dev

4 Click **Add**.

5 In the table, enter the following settings:

Branch	Shear modulus (Pa)	Relaxation time (s)
2	mu2	tau2_dev

6 Click **Add**.

7 In the table, enter the following settings:

Branch	Shear modulus (Pa)	Relaxation time (s)
3	mu3	tau3_dev

Linear Elastic Material 1

In the **Model Builder** window, click **Linear Elastic Material 1**.

Viscoelasticity, Volumetric

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

2 In the **Settings** window for **Viscoelasticity**, type `Viscoelasticity, Volumetric` in the **Label** text field.

3 Locate the **Viscoelasticity Model** section. From the **Viscoelastic strains** list, choose **Volumetric**.

4 In the table, enter the following settings:

Branch	Bulk modulus (Pa)	Relaxation time (s)
1	lambda1+2*mu1/3	tau1_vol

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 phi direction** list, choose **Prescribed**.

5 In the u_{0phi} text field, type `theta*R*step1(t[1/s])`.

GLOBAL DEFINITIONS

Having defined the physics, now fill in the empty expressions in the global **Materials** node.

Solid Phase (mat1)

1 In the **Model Builder** window, under **Global Definitions** > **Materials** click **Solid Phase (mat1)**.

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

3 In the table, enter the following settings:

Property	Variable	Value	Unit	Property group
Lamé parameter λ	lambLame	`lambda_eq`	N/m²	Lamé parameters
Lamé parameter μ	muLame	`mu_eq`	N/m²	Lamé parameters
Density	rho	`rho_solid`	kg/m³	Basic

MESH 1

1 In the **Model Builder** window, under **Component 1: Torsion Test (comp1)** click **Mesh 1**.

2 In the **Settings** window for **Mesh**, locate the **Physics-Controlled Mesh** section.

3 From the **Element size** list, choose **Finer**.

4 Click **Build All**.

STUDY 1: TORSION TEST

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

2 In the **Settings** window for **Study**, type `Study 1: Torsion Test` in the **Label** text field.

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

Step 1: Time Dependent

1 In the **Model Builder** window, under **Study 1: Torsion Test** 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, 0.1, 1) range(1, 1, 35)` to resolve the step function properly.

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

RESULT TEMPLATES

To compare the results with Figure 3, add a **Stress, 3D (solid)** plot.

1 In the **Home** toolbar, click **Result Templates** to open the **Result Templates** window.

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

3 In the tree, select **Study 1: Torsion Test/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

Surface 1

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

2 In the **Settings** window for **Surface**, locate the **Expression** section.

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

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

DEFINITIONS

To create Figure 4, define a variable for the torsional moment by integrating the reaction moment `solid.RMz` over the surface boundary. Remember to update the solution so that the newly defined variable is calculated.

Integration 1 (intop1)

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

2 In the **Settings** window for **Integration**, locate the **Source Selection** section.

3 From the **Geometric entity level** list, choose **Boundary**.

4 Select Boundary 3 only.

5 Locate the **Advanced** section. From the **Method** list, choose **Summation over nodes**.

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
`Tm`	`intop1(solid.RMz)`	N·m	`Torsional moment`

STUDY 1: TORSION TEST

In the **Study** toolbar, click **Update Solution**.

RESULTS

Read in the reference data from Ref. 1 for comparison.

Torsion Data, Ehlers and Markert (2001)

1. In the **Results** toolbar, click **Table**.
2. In the **Settings** window for **Table**, type `Torsion Data, Ehlers and Markert (2001)` 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 `linear_biphasic_poroelasticity_torsion_comp.txt`.

TORSION DATA, EHLERS AND MARKERT (2001)

1. Go to the **Torsion Data, Ehlers and Markert (2001)** window.
2. Click the **Table Graph** button in the window toolbar to plot the reference data in a table plot.

RESULTS

Torsional Moment

1. In the **Model Builder** window, under **Results** click **1D Plot Group 2**.
2. In the **Settings** window for **1D Plot Group**, type `Torsional Moment` in the **Label** text field.

Table Graph 1

1. In the **Model Builder** window, click **Table Graph 1**.
2. In the **Settings** window for **Table Graph**, locate the **Coloring and Style** section.
3. Find the **Line style** subsection. From the **Line** list, choose **None**.
4. Find the **Line markers** subsection. From the **Marker** list, choose **Circle**.
5. Click to expand the **Legends** section. Select the **Show legends** checkbox.
6. From the **Legends** list, choose **Manual**.

7 In the table, enter the following settings:

Legends
`Ehlers and Markert, 2001`

Global 1

1 In the **Model Builder** window, right-click **Torsional Moment** and choose **Global**.

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

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

4 Click **Add Expression** in the upper-right corner of the **y-Axis Data** section. From the menu, choose **Component 1: Torsion Test (comp1)** > **Definitions** > **Variables** > **Tm - Torsional moment - N·m**.

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

6 In the table, enter the following settings:

Legends
`COMSOL Multiphysics solution`

Torsional Moment

1 In the **Model Builder** window, click **Torsional Moment**.

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

3 Select the **Manual axis limits** checkbox.

4 In the **x minimum** text field, type `-1`.

5 In the **x maximum** text field, type `35`.

6 In the **y minimum** text field, type `0`.

7 In the **y maximum** text field, type `0.007`.

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

9 Locate the **Plot Settings** section. Select the **x-axis label** checkbox.

10 Select the **y-axis label** checkbox. In the associated text field, type `Torsional Moment (Nm)`.

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

12 In the **x spacing** text field, type `5`.

13 In the **y spacing** text field, type `0.001`.

14 In the **Torsional Moment** toolbar, click **Plot**.

ROOT

Now set up the indentation test. For this purpose, add a second model component.

ADD COMPONENT

In the **Model Builder** window, right-click the root node and choose **Add Component** > **2D Axisymmetric**.

COMPONENT 2: INDENTATION TEST

In the **Settings** window for **Component**, type `Component 2: Indentation Test` in the **Label** text field.

ADD PHYSICS

1 In the **Home** toolbar, click **Add Physics** to open the **Add Physics** window.

2 Go to the **Add Physics** window.

3 In the tree, select **Structural Mechanics** > **Poroelasticity** > **Poroelasticity, Solid**.

4 Click the **Add to Component 2: Indentation Test** button in the window toolbar.

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

GEOMETRY 2

1 In the **Settings** window for **Geometry**, locate the **Units** section.

2 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_ind`.

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

5 Click **Build Selected**.

Rectangle 2 (r2)

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 `2*r_ind`.

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

5 Locate the **Position** section. In the **z** text field, type `height_ind-r_ind`.

6 Click **Build Selected**.

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

Partition Edges 1 (pare1)

1 In the **Geometry** toolbar, click **Booleans and Partitions** and choose **Partition Edges**.

2 On the object **r2**, select Boundary 3 only.

3 In the **Settings** window for **Partition Edges**, click **Build Selected**.

Form Union (fin)

1 In the **Model Builder** window, click **Form Union (fin)**.

2 In the **Settings** window for **Form Union/Assembly**, click **Build Selected**.

Mesh Control Domains 1 (mcd1)

1 In the **Geometry** toolbar, click **Virtual Operations** and choose **Mesh Control Domains**.

2 On the object **fin**, select Domain 2 only.

3 In the **Settings** window for **Mesh Control Domains**, click **Build Selected**.

4 In the **Geometry** toolbar, click **Build All**.

MATERIALS

Porous Material 1 (pmat1)

1 In the **Model Builder** window, under **Component 2: Indentation Test (comp2)** 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**.

Solid 1 (pmat1.solid1)

In the **Model Builder** window, right-click **Porous Material 1 (pmat1)** and choose **Solid**.

The **Linear Elastic Material** properties in **Component 2** are the same as in **Component 1**, so you can just copy the **Viscoelasticity** subnodes from there.

SOLID MECHANICS (SOLID)

Viscoelasticity, Deviatoric, Viscoelasticity, Volumetric

Right-click and choose **Copy**.

SOLID MECHANICS 2 (SOLID2)

Linear Elastic Material 1

1 In the **Model Builder** window, under **Component 2: Indentation Test (comp2)** > **Solid Mechanics 2 (solid2)** click **Linear Elastic Material 1**.

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

3 From the **Specify** list, choose **Lamé parameters**.

4 Right-click **Component 2: Indentation Test (comp2)** > **Solid Mechanics 2 (solid2)** > **Linear Elastic Material 1** and choose **Paste Multiple Items**.

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 z direction** list, choose **Prescribed**.

5 In the u_{0z} text field, type `-disp*rm1(t/t_ind)`.

DARCY'S LAW (DL)

Pressure 1

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

2 Select Boundaries 3–5 only.

MULTIPHYSICS

Poroelasticity 1 (poro1)

1 In the **Model Builder** window, under **Component 2: Indentation Test (comp2)** > **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

Now check the material properties and enter additional material properties where needed.

Fluid 1 (pmat1.fluid1)

1 In the **Model Builder** window, under **Component 2: Indentation Test (comp2)** > **Materials** > **Porous Material 1 (pmat1)** click **Fluid 1 (pmat1.fluid1)**.

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

3 From the **Material** list, choose **Fluid Phase (mat2)**.

GLOBAL DEFINITIONS

Fluid Phase (mat2)

1 In the **Model Builder** window, under **Global Definitions** > **Materials** click **Fluid Phase (mat2)**.

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

3 In the table, enter the following settings:

Property	Variable	Value	Unit	Property group
Density	rho	`rho_fluid`	kg/m³	Basic
Dynamic viscosity	mu	`mu_fluid`	Pa·s	Basic

MATERIALS

Solid 1 (pmat1.solid1)

1 In the **Model Builder** window, under **Component 2: Indentation Test (comp2)** > **Materials** > **Porous Material 1 (pmat1)** click **Solid 1 (pmat1.solid1)**.

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

3 In the θ_s text field, type `phi_solid_ref`.

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 Material** section.

3 From the **Material** list, choose **Solid Phase (mat1)**.

4 Click **Go to Material**.

GLOBAL DEFINITIONS

Solid Phase (mat1)

1 In the **Model Builder** window, under **Global Definitions** > **Materials** click **Solid Phase (mat1)**.

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

3 In the table, enter the following settings:

Property	Variable	Value	Unit	Property group
Permeability	kappa_iso ; kappaii = kappa_iso, kappaij = 0	kappa0	m^2	Basic

DEFINITIONS (COMP2)

Add some necessary variable definitions to calculate the indentation force.

Integration 2 (intop2)

1. In the **Model Builder** window, expand the **Component 2: Indentation Test (comp2)** > **Definitions** node.
2. Right-click **Component 2: Indentation Test (comp2)** > **Definitions** and choose **Nonlocal Couplings** > **Integration**.
3. In the **Settings** window for **Integration**, locate the **Source Selection** section.
4. From the **Geometric entity level** list, choose **Boundary**.
5. Select Boundary 3 only.
6. Locate the **Advanced** section. From the **Method** list, choose **Summation over nodes**.

Variables 2

1. 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
RF	-intop2(solid2.RFz)	N	Indentation force

MESH 2

Free Triangular 1

In the **Mesh** toolbar, click **Free Triangular**.

Size

1. In the **Model Builder** window, click **Size**.
2. In the **Settings** window for **Size**, locate the **Element Size** section.
3. From the **Predefined** list, choose **Finer**.

Size 1

1 In the **Model Builder** window, right-click **Free Triangular 1** and choose **Size**.

2 In the **Settings** window for **Size**, locate the **Geometric Entity Selection** section.

3 From the **Geometric entity level** list, choose **Domain**.

4 Select Domain 2 only.

5 Locate the **Element Size** section. Click the **Custom** button.

6 Locate the **Element Size Parameters** section.

7 Select the **Maximum element size** checkbox. In the associated text field, type `0.05[mm]`.

8 Click **Build Selected**.

STUDY 1: TORSION TEST

Now that you have added a second component to the model, you have to restrict **Study 1** to **Component 1** in case you want to run it again. Add a second study to solve for the indentation test in **Component 2**.

Step 1: Time Dependent

1 In the **Model Builder** window, under **Study 1: Torsion Test** click **Step 1: Time Dependent**.

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

3 In the **Solve for** column of the table, clear the checkbox for **Component 2: Indentation Test (comp2)**.

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 Find the **Physics interfaces in study** subsection. In the table, clear the **Solve** checkbox for **Solid Mechanics (solid)**.

5 Click the **Add Study** button in the window toolbar.

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

STUDY 2: INDENTATION TEST

1 In the **Settings** window for **Study**, type `Study 2: Indentation Test` in the **Label** text field.

2 Locate the **Study Settings** section. Clear the **Generate default plots** checkbox.

1 In the **Model Builder** window, under **Study 2: Indentation Test** 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, 0.25*t_ind, t_ind) 10^{range(log10(1.1*t_ind[1/s]),1/15,log10(t_end[1/s]))} t_end`.

Add a parametric study to vary the time interval for the loading ramp function.

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
t_ind (Loading ramp)	5 100 250	s

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

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

RESULTS

To plot Figure 5, follow the steps below.

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 2: Indentation Test/Parametric Solutions 1 (5) (sol3)** > **Darcy's Law** > **Pressure (dl)**.

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

Pore Pressure (dl), Indentation Test

1 In the **Settings** window for **2D Plot Group**, type `Pore Pressure (dl), Indentation Test` in the **Label** text field.

2 Click to collapse the **Data** section. Click to expand the **Data** section. From the **Parameter value (t_ind (s))** list, choose **5**.

3 From the **Time (s)** list, choose **5**.

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

Surface

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

2 In the **Settings** window for **Surface**, locate the **Coloring and Style** section.

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

Deformation 1

Right-click **Surface** and choose **Deformation**.

Surface 2

Right-click **Surface** and choose **Duplicate**.

Pore Pressure (dl), Indentation Test

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

2 From the **Array type** list, choose **Linear**.

3 From the **Array axis** list, choose **y**.

4 From the **Displacement** list, choose **Absolute**.

5 In the **Cell displacement** text field, type `-5`.

Surface 2

1 In the **Model Builder** window, expand the **Results** > **Pore Pressure (dl), Indentation Test** > **Surface 2** node, then click **Surface 2**.

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

3 From the **Dataset** list, choose **Study 2: Indentation Test/Parametric Solutions 1 (5) (sol3)**.

4 From the **Parameter value (t_ind (s))** list, choose **100**.

5 From the **Time (s)** list, choose **100**.

6 Click to expand the **Inherit Style** section. From the **Plot** list, choose **Surface**.

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

Surface 3

1 Right-click **Surface 2** and choose **Duplicate**.

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

3 From the **Parameter value (t_ind (s))** list, choose **250**.

4 In the **Pore Pressure (dl), Indentation Test** toolbar, click **Plot**.

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

Pore Pressure (dl), Indentation Test

In the **Model Builder** window, click **Pore Pressure (dl), Indentation Test**.

Table Annotation 1

1 In the **Pore Pressure (dl), Indentation Test** 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
-3	11	t_ind = 5 s
-3	6.5	t_ind = 100 s
-3	2	t_ind = 250 s

5 Locate the **Coloring and Style** section. Clear the **Show point** checkbox.

6 From the **Anchor point** list, choose **Middle left**.

7 Click to expand the **Plot Array** section. Select the **Belongs to array** checkbox.

8 Select the **Manual indexing** checkbox.

9 In the **Index** text field, type `2`.

10 In the **Pore Pressure (dl), Indentation Test** toolbar, click **Plot**.

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

Indentation Data, Ehlers and Markert (2001)

Read in the reference data for the indentation test from Ref. 1 to compare the indentation forces.

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

2 In the **Settings** window for **Table**, type `Indentation Data, Ehlers and Markert (2001)` 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 `linear_biphasic_poroelasticity_indentation_comp.txt`.

INDENTATION DATA, EHLERS AND MARKERT (2001)

1 Go to the **Indentation Data, Ehlers and Markert (2001)** window.

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

RESULTS

Indentation Force

1 In the **Model Builder** window, under **Results** click **1D Plot Group 4**.

2 In the **Settings** window for **1D Plot Group**, type `Indentation Force` in the **Label** text field.

Table Graph 1

1 In the **Model Builder** window, click **Table Graph 1**.

2 In the **Settings** window for **Table Graph**, locate the **Coloring and Style** section.

3 Find the **Line style** subsection. From the **Line** list, choose **None**.

4 Find the **Line markers** subsection. From the **Marker** list, choose **Circle**.

5 Locate the **Legends** section. From the **Legends** list, choose **Manual**.

6 In the table, enter the following settings:

Legends
`Ehlers and Markert (2001)`

Global 1

1 In the **Model Builder** window, right-click **Indentation Force** and choose **Global**.

2 In the **Settings** window for **Global**, click **Add Expression** in the upper-right corner of the **y-Axis Data** section. From the menu, choose **Component 2: Indentation Test (comp2)** > **Definitions** > **Variables** > **comp2.RF - Indentation force - N**.

3 Locate the **Data** section. From the **Dataset** list, choose **Study 2: Indentation Test/ Parametric Solutions 1 (4) (sol3)**.

4 Locate the **x-Axis Data** section. From the **Axis source data** list, choose **Inner solutions**.

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

6 In the **Indentation Force** toolbar, click **Plot**.

Table Graph 1

1 In the **Model Builder** window, click **Table Graph 1**.

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

3 Select the **Show legends** checkbox.

Indentation Force

1 In the **Model Builder** window, click **Indentation Force**.

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

3 Select the **y-axis label** checkbox.

4 Select the **x-axis label** checkbox.

5 In the **y-axis label** text field, type `Indentation force (N)`.

Global 1

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

2 In the **Settings** window for **Global**, locate the **Legends** section.

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

4 In the table, enter the following settings:

Legends
t_ind=5 s
t_ind=100 s
t_ind=250 s

5 In the **Indentation Force** toolbar, click **Plot** and compare with Figure 6.