Created in COMSOL Multiphysics 5.3

Biot Poroelasticity

Introduction

Poroelastic models describe the linked interaction between fluids and deformation in porous media. The fluids in a reservoir absorb stress, which registers as fluid pressure or equally hydraulic head. For example, if pumping significantly reduces pore fluid pressures, sediments could shift due to the increased load. Because the reduction in the pore space brings about more fluid movement, the reservoir could compact further. It follows that lateral stretching must compensate for the vertical compaction.

Model Definition

This example analyzes fluid and solid behavior within three sedimentary layers overlying an impermeable bedrock in a basin. The bedrock is faulted, which creates a step near a mountain front. The sediment stack totals 420 m at the centerline of the basin ($x = 0$ m) and thins to 120 m above the step ($x > 4000$ m). The top two layers are each 20 m thick.

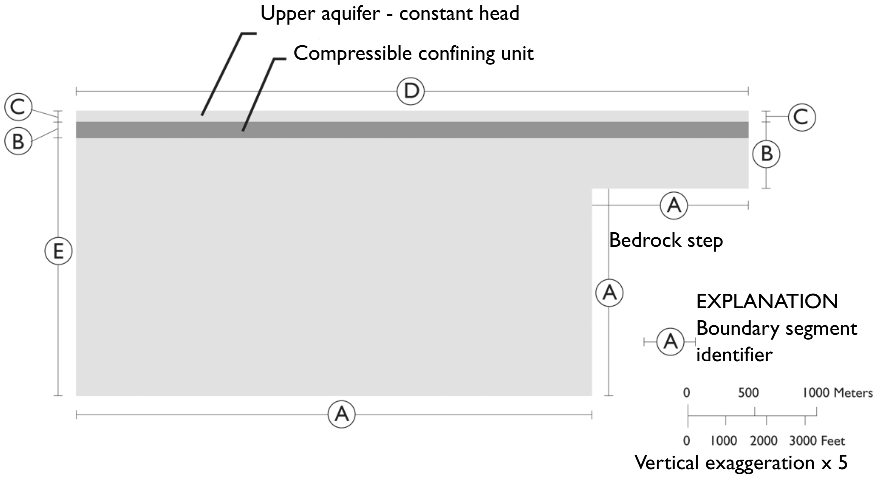

Figure 1: Model geometry showing boundary segments (from Leake and Hsieh, Ref. 1).

Pumping from the lower aquifer reduces hydraulic head down the centerline of the basin by 6 m per year. The head drop moves fluid away from the step. The middle layer is relatively impermeable. The pumping does not diminish the supply of fluids in the unpumped reservoir above it. The flow field is initially at steady state. The period of interest is 10 years.

Governing Equations

This example uses the Poroelasticity multiphysics node, which includes the time rate of change in strain from the Solid Mechanics interface in the Darcy's Law interface and adds the fluid pressure gradient as stress contribution in the Solid Mechanics interface. The following sections describe the built-in equations when using the Poroelastic Material feature as well as the initial and boundary conditions.

FLUID FLOW

Use the Darcy's Law interface to estimate the flow field in the poroelastic model with the pressure head formulation

$$\rho_{\mathrm{f}} S_\alpha \frac{\partial H}{\partial t} + \nabla \cdot \rho_{\mathrm{f}}[-K\nabla H] = -\rho_{\mathrm{f}} \alpha_{\mathrm{B}} \frac{\partial}{\partial t}\varepsilon_{\mathrm{vol}} \qquad (1)$$

where ρ_{f} is the fluid density, H is the pressure head (m), K is the hydraulic conductivity (m/s), $\varepsilon_{\mathrm{vol}}$ is the volumetric strain of the porous matrix, and α_{B} is the Biot-Willis coefficient; in this model the Biot-Willis coefficient equals 1, which is a common value for "soft soils." You can interpret the term on the right-hand side as the time rate of expansion of the porous matrix. The volume fraction available for liquid increases and thereby gives rise to liquid sink, which is why the sign is reversed in the source term. Leake and Hsieh (Ref. 1) defined S_α using coefficients from the solids equation, the Young's modulus, E, and Poisson's ratio, ν. Debate over poroelastic storage coefficients is heated (Ref. 2), and the subscript α here denotes that conventional storage terms might need redefinition for poroelasticity models.

The main differences between the Terzaghi compaction model and this poroelastic analysis lie in material coefficients and sources; the boundary conditions are identical: H is the offset in hydraulic head since an initial steady-state distribution of H_0. This subtle twist simplifies describing the boundary conditions: the value at the outlet boundary becomes the decline in hydraulic head with time, and at the upper aquifer, the hydraulic head is fixed at H_0. All other boundaries have no-flow conditions. The boundary and initial conditions for the Darcy's law analysis are

$$\begin{array}{lll}
\mathbf{n} \cdot K\nabla H = 0 & \partial\Omega \ \text{base} & A \\
\mathbf{n} \cdot K\nabla H = 0 & \partial\Omega \ \text{other} & B \\
H = H_0 & \partial\Omega \ \text{upper edge} & C \\
H = H_0 & \partial\Omega \ \text{surface} & D \\
H = H(t) & \partial\Omega \ \text{outlet} & E
\end{array}$$

where **n** is the normal to the boundary. The letters *A* through *E* come from Leake and Hsieh (Ref. 1), each letter denoting a specific boundary (see Figure 1).

POROUS MATRIX DEFORMATION

The governing equation for the poroelastic material model is

$$-\nabla\cdot\sigma = \rho\mathbf{g} \tag{2}$$

here, σ is the stress tensor, ρ is the total density, and **g** is acceleration of gravity. The poroelastic material model uses Equation 2 to describe changes in the stress tensor σ and porous matrix displacement **u** due to boundary conditions and changes in pore pressure. This focus on changes in displacement is standard poroelasticity and greatly simplifies specifying loads, boundaries, and initial conditions. Equation 2 defines a state of static equilibrium because the changes in the solid equilibrate quickly, unlike vibrations or waves, that is, there are no time-dependent terms. Still the time rate of change in strain $\partial\varepsilon_{\mathrm{vol}}/\partial t$ appears as a coupling term in Darcy's Law because the solids equation becomes quasi-static when solved simultaneously with a time-dependent flow model.

The governing equation for the bedrock step problem describes the deformation state of plane strain, which is the norm for 2D poroelasticity problems (Ref. 1 and Ref. 2), so the strain normal to the *xy*-plane equals zero. COMSOL Multiphysics solves Equation 2 using the Solid Mechanics interface.

For an isotropic porous material under plane strain conditions, this simplifies to

$$\begin{bmatrix}\sigma_{xx}\\ \sigma_{yy}\\ \sigma_{xy}\end{bmatrix} = \frac{E}{(1+\nu)(1-2\nu)}\begin{bmatrix}1-\nu & \nu & 0\\ \nu & 1-\nu & 0\\ 0 & 0 & 1-2\nu\end{bmatrix}\begin{bmatrix}\varepsilon_{xx}\\ \varepsilon_{yy}\\ \varepsilon_{xy}\end{bmatrix} - \begin{bmatrix}\alpha_{\mathrm{B}}p & 0 & 0\\ 0 & \alpha_{\mathrm{B}}p & 0\\ 0 & 0 & \alpha_{\mathrm{B}}p\end{bmatrix} \tag{3}$$

where *E* is Young's modulus (Pa) and ν is Poisson's ratio of the drained porous matrix. The term $\alpha_{\mathrm{B}}p$ (Pa) amounts to the fluid pressure contribution and is often described as the fluid-to-structure coupling expression. This contribution is added by the Poroelasticity multiphysics node.

With small deformations for plane strain analyses, the normal strains ε_{xx}, ε_{yy}, ε_{zz} and shear strains ε_{xy}, ε_{yz}, ε_{xz} relate to the displacements u and v as

$$\varepsilon_{xx} = \frac{\partial u}{\partial x} \qquad \varepsilon_{yy} = \frac{\partial v}{\partial y} \qquad \varepsilon_{xy} = \frac{1}{2}\left(\frac{\partial u}{\partial y} + \frac{\partial v}{\partial x}\right) \qquad \varepsilon_{xy} = \varepsilon_{yx} \qquad \varepsilon_{xz} = \varepsilon_{yz} = \varepsilon_{yz} = 0 \tag{4}$$

Inserting the relationships from Equation 3 and Equation 4 into Equation 2 gives the equation that COMSOL Multiphysics solves.

The boundary conditions on the porous matrix are a series of constraints on the displacement that allow for horizontal movement at the surface and throughout the basin. The base of the sediments is fixed, which means you constrain horizontal and vertical displacement to zero. The upper surface is free to vary in the horizontal and the vertical directions. The boundary on left of Figure 1 is a roller condition, so the sediments are free to move in the vertical direction, but there is no horizontal displacement. These conditions result in the following boundary expressions

fixed constraint	$\partial\Omega$	base	A
free	$\partial\Omega$	other	B
free	$\partial\Omega$	upper edge	C
free	$\partial\Omega$	surface	D
roller	$\partial\Omega$	outlet	E

where, once again, the letters A through E come from Leake and Hsieh, and denote the boundaries in Figure 1.

Model Data

The coefficients and parameters for the poroelasticity model are as follows:

VARIABLE	DESCRIPTION	VALUE
g	Gravity	9.82 m/s^2
ρ_f	Fluid density	1000 kg/m^3
ρ_d	Drained density of porous layer	2750 kg/m^3
S_α	Poroelastic storage coefficient, aquifer layers	$1\cdot10^{-6}$ m^{-1}
S_α	Poroelastic storage coefficient, confining layer	$1\cdot10^{-5}$ m^{-1}
K	Hydraulic conductivity, aquifer layers	25 m/day
K	Hydraulic conductivity, confining layer	0.01 m/day
α_B	Biot-Willis coefficient	1
H_0	Initial hydraulic head	0 m
$H(t)$	Declining head boundary	(6 m/year)·t
E	Drained Young's modulus, aquifer layers	$8\cdot10^8$ Pa
E	Drained Young's modulus, confining layer	$8\cdot10^7$ Pa
ν	Drained Poisson's ratio, all regions	0.25

Results and Discussion

Figure 2, Figure 3, and Figure 4 are Year 2, Year 5, and Year 10 snapshots, respectively, from the COMSOL Multiphysics solution to the problem (Ref. 1) of linked fluid flow and solid deformation near a bedrock step in a sedimentary basin. The shading and arrows, respectively, represent the change in hydraulic head and velocities brought about by pumping from the basin interior at $x = 0$ m. The fluid moves from the surface toward the well screens in the lower aquifer, with an abrupt change in direction and velocity near the bedrock step. In this way, the flow solution here is similar to the results in Terzaghi compaction model.

The results for the solids displacement tell another story. In Figure 2, Figure 3, and Figure 4, contours and deformed shapes illustrate the total displacement. The plot sequence illustrates the evolution of lateral deformations that compensate for the changing surface elevation above the bedrock step.

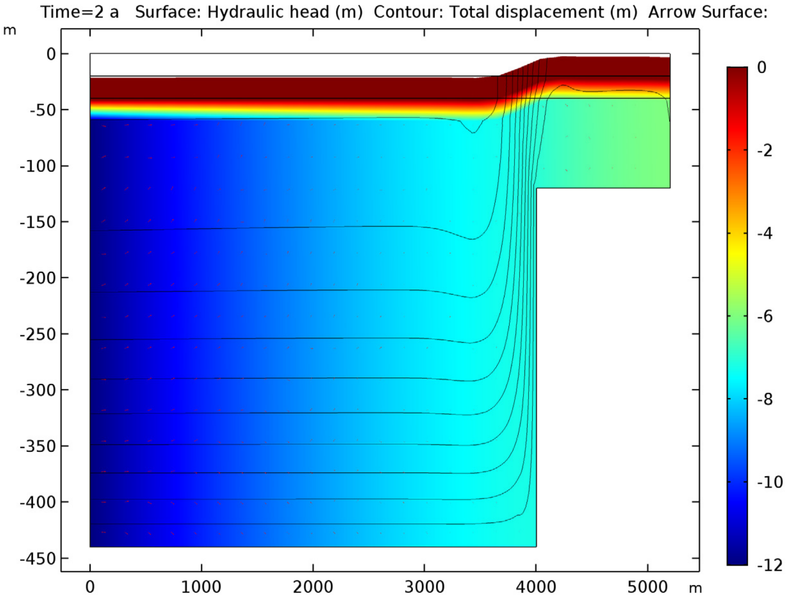

Figure 2: Solution for a poroelasticity analysis of the bedrock step problem of Leake and Hsieh (Ref. 1): Hydraulic head (surface plot), displacement (contours and deformations) at Year 2. The vertical axis is expanded for clarity.

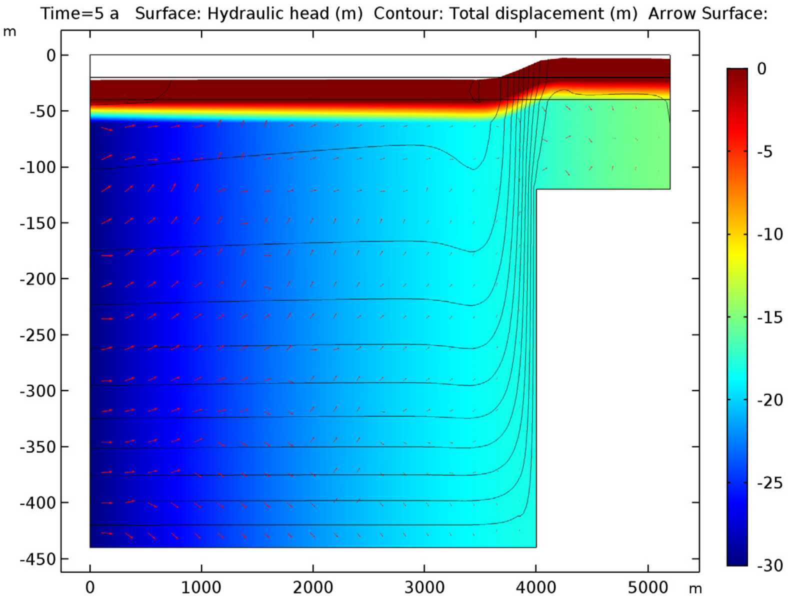

Figure 3: Hydraulic head (surface plot), displacement (contours and deformations) at Year 5. The vertical axis and deformation are exaggerated for clarity.

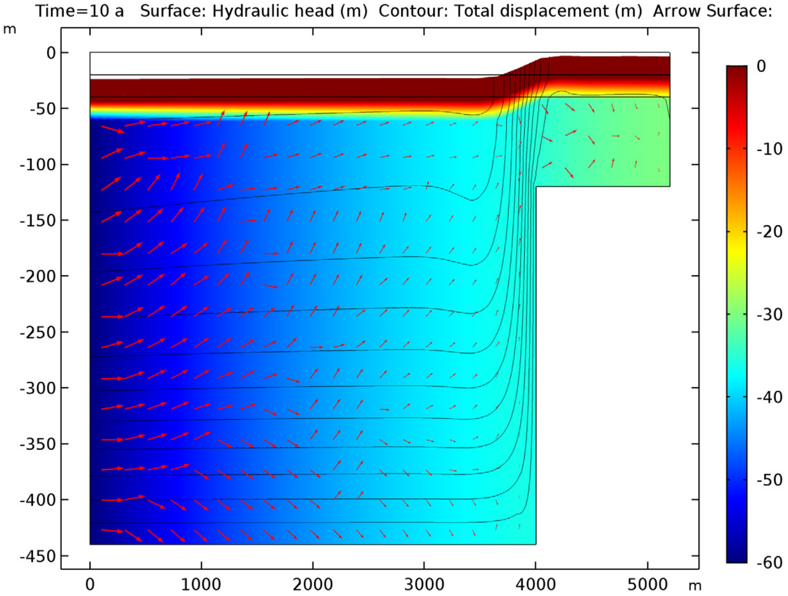

Figure 4: Hydraulic head (surface plot), displacement (contours and deformations) at Year 10. The vertical axis and deformation are exaggerated for clarity.

Figure 5, represents the coupling terms that link the fluid and structure equations. The shading gives the structure-to-fluid link, which is the negative of the time rate of change in strain (see Equation 1).

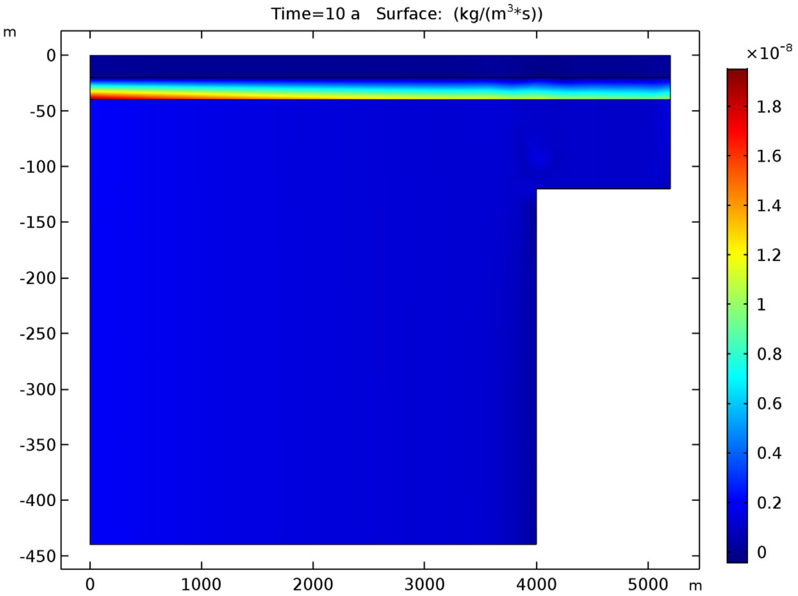

Figure 5: The solid-to-fluid coupling term evaluated at Year 10 with the poroelastic analysis. The squeeze of porous matrix results in an apparent mass source term.

The Terzaghi and Biot solutions differ most when it comes to predicting the horizontal strain at the edge of the bedrock step. The Biot poroelasticity analysis predicts horizontal strain; the Terzaghi compaction analysis does not. The horizontal strains at the ground surface from the Biot poroelasticity approach appear in Figure 6. It depicts negative strain or compaction immediately on the basin side of the step; the positive strains correspond to tension or lengthening on the mountain side.

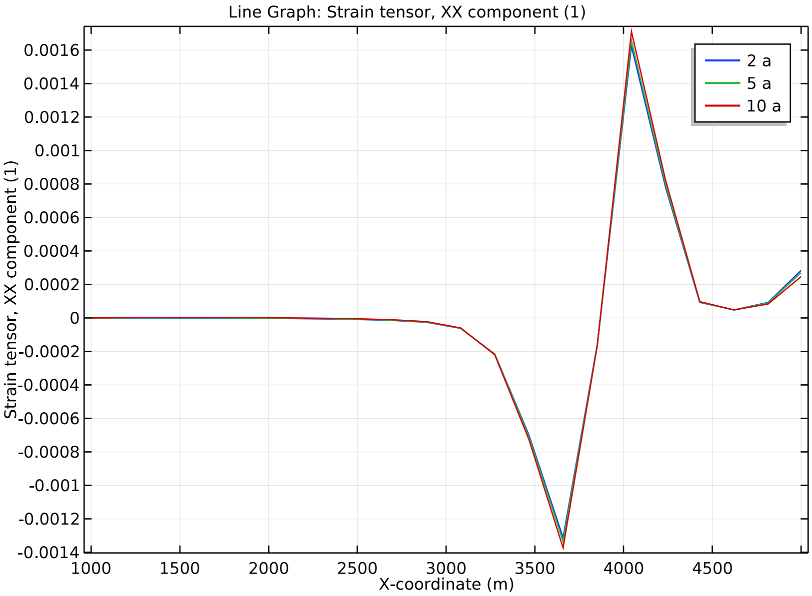

Figure 6: COMSOL Multiphysics estimates of horizontal strain from poroelastic analysis for the bedrock step problem of Leake and Hsieh: Year 2 (blue, solid line), Year 5 (green, dotted line), and Year 10 (red, dashed line).

Failure criteria or expressions defining a critical threshold for stress, strain, or displacements facilitate evaluating whether the strain differential at the bedrock steps is big enough to produce fissures. Figure 7 plots von Mises stresses (surface plot), fluid velocities (streamlines), and total displacement (deformation). The von Mises stresses are postprocessing variables defined by COMSOL Multiphysics in all structural-deformation analyses. The von Mises stresses are variables in many failure expressions and you can also use them as yield functions in the elasto-plastic materials dialog boxes.

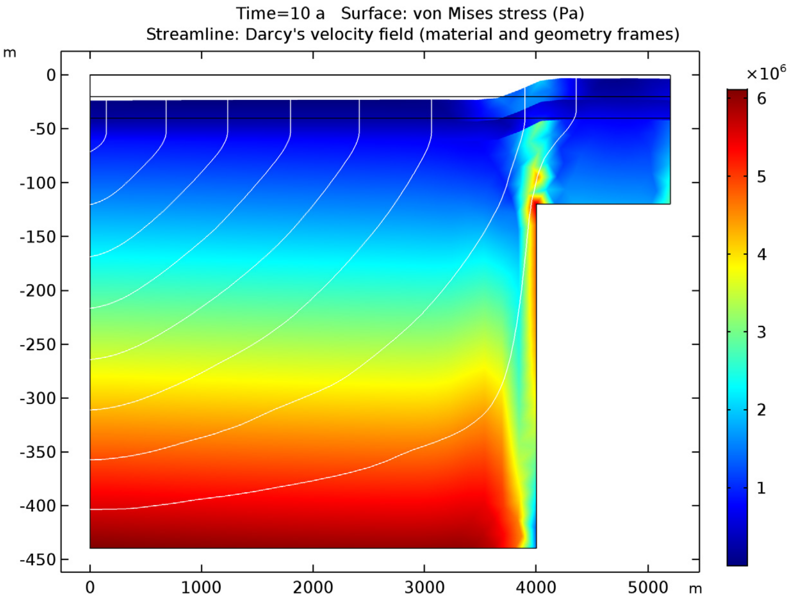

*Figure 7: COMSOL Multiphysics estimates of von Mises stresses (surface plot), fluid velocities (streamlines), and displacement (deformation) at Year 10. These results correspond to the poroelastic analysis from Leake and Hsieh (*Ref. 1*). The vertical axis and deformation are exaggerated for clarity.*

References

1. S.A. Leake and P.A. Hsieh, *Simulation of Deformation of Sediments from Decline of Ground-Water Levels in an Aquifer Underlain by a Bedrock Step*, U.S. Geological Survey Open File Report 97-47, 1997.

2. H.F. Wang, *Theory of Linear Poroelasticity with Application to Geomechanics and Hydrogeology*, Princeton Univ. Press, 2000.

Application Library path: `Subsurface_Flow_Module/Flow_and_Solid_Deformation/biot_poroelasticity`

Modeling Instructions

ROOT

1. From the **File** menu, choose **Open**.
2. Browse to the model's Application Libraries folder and double-click the file `terzaghi_compaction.mph`.

RESULTS

Contour 1

Add a **Solid Mechanics** interface and a **Poroelasticity** multiphysics node.

ADD PHYSICS

1. On 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>Solid Mechanics (solid)**.
4. Click **Add to Component** in the window toolbar.

SOLID MECHANICS (SOLID)

1. On the **Home** toolbar, click **Add Physics** to close the **Add Physics** window.
2. On the **Home** toolbar, click **Add Multiphysics**.

ADD MULTIPHYSICS

1. Go to the **Add Multiphysics** window.
2. In the tree, select **Structural Mechanics>Poroelasticity**.
3. Click **Add to Component** in the window toolbar.

MULTIPHYSICS

1. On the **Home** toolbar, click **Add Multiphysics**.

   The insertion of the multiphysics node automatically inserts a poroelastic storage material model in Darcy's Law and sets the **Discretization** to 'linear'.

DARCY'S LAW (DL)

On the **Physics** toolbar, click **Solid Mechanics (solid)** and choose **Darcy's Law (dl)**.

Poroelastic Storage 1

1. In the **Model Builder** window, expand the **Darcy's Law (dl)** node, then click **Poroelastic Storage 1**.

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

3 From the **Fluid material** list, choose **Fluid (mat1)**.

4 Locate the **Matrix Properties** section. From the **Permeability model** list, choose **Hydraulic conductivity**.

5 In the K text field, type `K_s`.

SOLID MECHANICS (SOLID)

On the **Physics** toolbar, click **Darcy's Law (dl)** and choose **Solid Mechanics (solid)**.

Gravity 1

1 On the **Physics** toolbar, click **Domains** and choose **Gravity**.

2 In the **Settings** window for **Gravity**, locate the **Domain Selection** section.

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

Fixed Constraint 1

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

2 Select Boundaries 2, 8, and 9 only.

Roller 1

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

2 Select Boundaries 1, 3, and 5 only.

MULTIPHYSICS

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 α_B list, choose **User defined**. In the associated text field, type `1`.

Fill the required fields in the material properties tables.

MATERIALS

Fluid (mat1)

1 In the **Model Builder** window, under **Component 1 (comp1)>Materials** click **Fluid (mat1)**.

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

3 In the table, enter the following settings:

Property	**Name**	**Value**	**Unit**	**Property group**
Compressibility of fluid	chif	`4e-10`	1/Pa	Basic
Density	rho	`1000`	kg/m³	Basic

Porous 1 (mat2)

1 In the **Model Builder** window, under **Component 1 (comp1)>Materials** click **Porous 1 (mat2)**.

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

3 In the table, enter the following settings:

Property	**Name**	**Value**	**Unit**	**Property group**
Young's modulus	E	`800[MPa]`	Pa	Basic
Poisson's ratio	nu	`0.25`	1	Basic
Density	rho	`2750`	kg/m³	Basic
Porosity	epsilon	`0.25`	1	Basic

Porous 2 (mat3)

1 In the **Model Builder** window, under **Component 1 (comp1)>Materials** click **Porous 2 (mat3)**.

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

3 In the table, enter the following settings:

Property	**Name**	**Value**	**Unit**	**Property group**
Young's modulus	E	`80[MPa]`	Pa	Basic
Poisson's ratio	nu	`0.25`	1	Basic
Density	rho	`2750`	kg/m³	Basic
Porosity	epsilon	`0.025`	1	Basic

Create a variable to calculate the solid-to-fluid coupling term.

DEFINITIONS

Variables 3

1 In the **Model Builder** window, expand the **Component 1 (comp1)>Definitions** node, then click **Variables 3**.

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

3 In the table, enter the following settings:

Name	**Expression**	**Unit**
`Q_biot`	`-poro1.alphaB*d(poro1.evol,TIME)*dl.rho`	kg/(m³·s)

STUDY 1

Disable the newly created nodes in the existing study.

Step 1: Time Dependent

1 In the **Model Builder** window, expand the **Study 1** node, then click **Step 1: Time Dependent**.

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

3 In the **Physics interface** table, clear the **Solve for** check box for **Solid Mechanics (solid)**.

4 In the **Multiphysics** table, clear the **Solve for** check box for **Poroelasticity 1 (poro1)**.

5 Select the **Modify physics tree and variables for study step** check box.

6 In the **Physics and variables selection** tree, select **Component 1 (comp1)>Darcy's Law (dl)> Poroelastic Storage 1**.

7 Click **Disable**.

ADD STUDY

1 On 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 **Preset Studies> Time Dependent**.

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

STUDY 2

Step 1: Time Dependent

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

2 In the **Model Builder** window, under **Study 2** click **Step 1: Time Dependent**.

3 In the **Settings** window for **Time Dependent**, locate the **Study Settings** section.

4 From the **Time unit** list, choose **a**.

5 Click **Range**.

6 In the **Range** dialog box, type `1` in the **Step** text field.

7 In the **Stop** text field, type `10`.

8 Click **Replace**.

9 On the **Home** toolbar, click **Compute**.

RESULTS

Pressure (dl)

Follow the steps below to reproduce the plots shown in Figure 2 through Figure 4.

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

2 In the **Settings** window for **2D Plot Group**, type `Hydraulic Head, Poroelasticity` in the **Label** text field.

Surface

1 In the **Model Builder** window, expand the **Results>Hydraulic Head, Poroelasticity** node, then click **Surface**.

2 In the **Settings** window for **Surface**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Model>Component 1>Darcy's Law>dl.H - Hydraulic head**.

Contour 1

1 In the **Model Builder** window, under **Results** right-click **Hydraulic Head, Poroelasticity** and choose **Contour**.

2 In the **Settings** window for **Contour**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Model>Component 1>Solid Mechanics> Displacement>solid.disp - Total displacement**.

3 Locate the **Levels** section. From the **Entry method** list, choose **Levels**.

4 In the **Levels** text field, type `range(0,0.2,2.3)`.

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

6 From the **Color** list, choose **Black**.

7 Clear the **Color legend** check box.

8 Click to expand the **Quality** section. From the **Resolution** list, choose **Extra fine**.

Arrow Surface 1

1 Right-click **Hydraulic Head, Poroelasticity** and choose **Arrow Surface**.

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

3 In the **X component** text field, type `d(dl.H,X)`.

4 In the **Y component** text field, type `d(dl.H,Y)`.

5 Locate the **Arrow Positioning** section. Find the **X grid points** subsection. In the **Points** text field, type `25`.

Deformation 1

1 In the **Model Builder** window, under **Results>Hydraulic Head, Poroelasticity** right-click **Surface** and choose **Deformation**.

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

3 Select the **Scale factor** check box.

4 In the associated text field, type `10`.

5 Right-click **Results>Hydraulic Head, Poroelasticity>Surface>Deformation 1** and choose **Copy**.

Arrow Surface 1

1 In the **Model Builder** window, under **Results>Hydraulic Head, Poroelasticity** right-click **Contour 1** and choose **Paste Deformation**.

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

3 Select the **Scale factor** check box.

4 On the **Hydraulic Head, Poroelasticity** toolbar, click **Plot**.

5 In the associated text field, type `2e4`.

Hydraulic Head, Poroelasticity

1 Right-click **Results>Hydraulic Head, Poroelasticity>Arrow Surface 1** and choose **Paste Deformation**.

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

3 From the **Time (a)** list, choose **2**.

4 On the **Hydraulic Head, Poroelasticity** toolbar, click **Plot**.

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

Compare the resulting plot with that in Figure 2.

6 From the **Time (a)** list, choose **5**.

7 On the **Hydraulic Head, Poroelasticity** toolbar, click **Plot**.

The plot in the Graphics window should now look like that in Figure 3.

8 From the **Time (a)** list, choose **10**.

9 On the **Hydraulic Head, Poroelasticity** toolbar, click **Plot**.

Compare with Figure 4.

Next, modify the second default plot to reproduce Figure 7.

Stress (solid)

1. In the **Model Builder** window, under **Results** click **Stress (solid)**.
2. In the **Settings** window for **2D Plot Group**, type `Von Mises Stress` in the **Label** text field.

Von Mises Stress

In the **Model Builder** window, expand the **Results>Von Mises Stress** node.

Deformation

1. In the **Model Builder** window, expand the **Results>Von Mises Stress>Surface 1** node, then click **Deformation**.
2. In the **Settings** window for **Deformation**, locate the **Scale** section.
3. Select the **Scale factor** check box.
4. In the associated text field, type `10`.

Streamline 1

1. In the **Model Builder** window, under **Results** right-click **Von Mises Stress** and choose **Streamline**.
2. In the **Settings** window for **Streamline**, locate the **Streamline Positioning** section.
3. From the **Positioning** list, choose **Start point controlled**.
4. From the **Entry method** list, choose **Coordinates**.
5. In the **X** text field, type `0`.
6. In the **Y** text field, type `range(-450,50,-50)`.
7. Locate the **Coloring and Style** section. From the **Color** list, choose **White**.
8. Click to expand the **Quality** section. From the **Smoothing** list, choose **Everywhere**.

Deformation

In the **Model Builder** window, under **Results>Von Mises Stress>Surface 1** right-click **Deformation** and choose **Copy**.

Deformation 1

In the **Model Builder** window, under **Results>Von Mises Stress** right-click **Streamline 1** and choose **Paste Deformation**.

Next, reproduce the plot in Figure 5 showing the solid-to-fluid coupling term.

2D Plot Group 5

1. On the **Home** toolbar, click **Add Plot Group** and choose **2D Plot Group**.

2 In the **Settings** window for **2D Plot Group**, type `Solid-to-Fluid Coupling Term` in the **Label** text field.

3 Locate the **Data** section. From the **Data set** list, choose **Study 2/Solution 2 (sol2)**.

Surface 1

1 Right-click **Solid-to-Fluid Coupling Term** and choose **Surface**.

2 In the **Settings** window for **Surface**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Model>Component 1>Definitions> Variables>Q_biot**.

3 On the **Solid-to-Fluid Coupling Term** toolbar, click **Plot**.

To set up the plot in Figure 6 proceed as follows:

Cut Line 2D 1

1 On the **Results** toolbar, click **Cut Line 2D**.

2 In the **Settings** window for **Cut Line 2D**, locate the **Data** section.

3 From the **Data set** list, choose **Study 2/Solution 2 (sol2)**.

4 Locate the **Line Data** section. In row **Point 1**, set **X** to `1000`.

5 In row **Point 2**, set **X** to `5000`.

1D Plot Group 6

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

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

3 Locate the **Data** section. From the **Data set** list, choose **Cut Line 2D 1**.

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

5 In the **Times (a)** list, choose **2**, **5**, and **10**.

Line Graph 1

1 Right-click **Horizontal Strain** and choose **Line Graph**.

2 In the **Settings** window for **Line Graph**, click **Replace Expression** in the upper-right corner of the **y-axis data** section. From the menu, choose **Model>Component 1>Solid Mechanics> Strain>Strain tensor (material and geometry frames)>solid.eXX - Strain tensor, XX component**.

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

4. Click **Replace Expression** in the upper-right corner of the **x-axis data** section. From the menu, choose **Model>Component 1>Geometry>**
   Coordinate (material and geometry frames)>X - X-coordinate.
5. Click to expand the **Legends** section. Select the **Show legends** check box.
6. On the **Horizontal Strain** toolbar, click **Plot**.