Model created in COMSOL Multiphysics 6.4

Biot Poroelasticity

This model is licensed under the COMSOL Software License Agreement 6.4.
All trademarks are the property of their respective owners. See www.comsol.com/trademarks.

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.

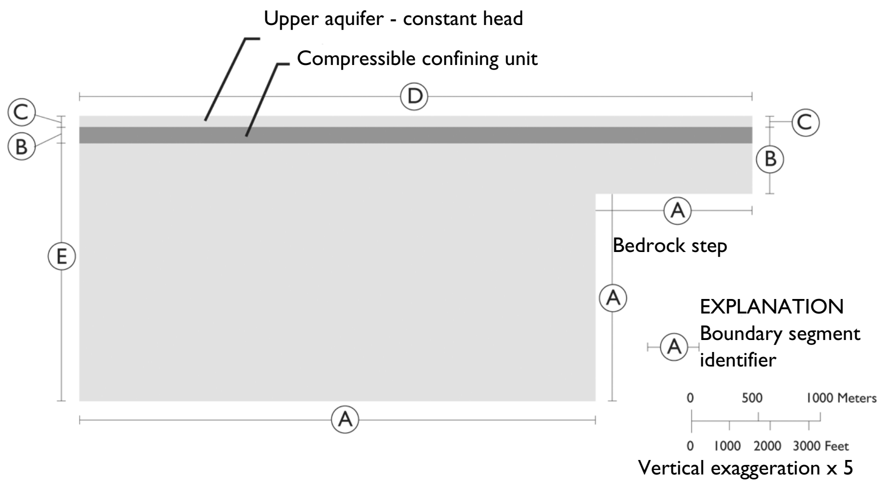

Figure 1: Model geometry showing boundary segments (from Leake and Hsieh, Ref. 1).

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

$$\rho_f S_\alpha \frac{\partial H}{\partial t} + \nabla \cdot \rho_f[-K\nabla H] = -\rho_f \alpha_B \frac{\partial}{\partial t}\varepsilon_{vol} \tag{1}$$

where ρ_f is the fluid density, H is the pressure head (m), K is the hydraulic conductivity (m/s), ε_{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

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

where $\mathbf{n}$ 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 $\mathbf{g}$ is acceleration of gravity. The poroelastic material model uses Equation 2 to describe changes in the stress tensor σ and porous matrix displacement $\mathbf{u}$ 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.

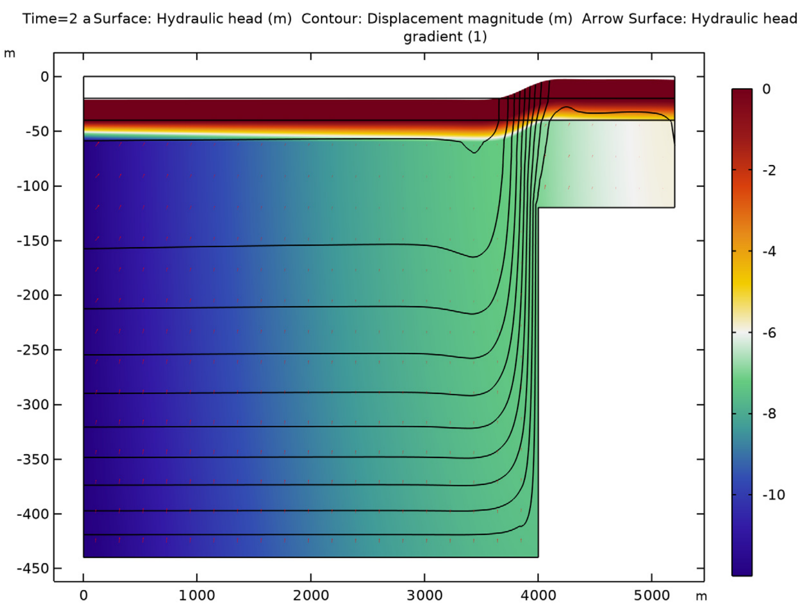

Figure 2: Solution for a poroelasticity analysis of the bedrock step problem of Leake and Hsieh (Ref. 1): Hydraulic head (surface plot), displacement (contours and deformations) at Year 2. The arrows indicate the hydraulic head gradient. The vertical axis is expanded for clarity.

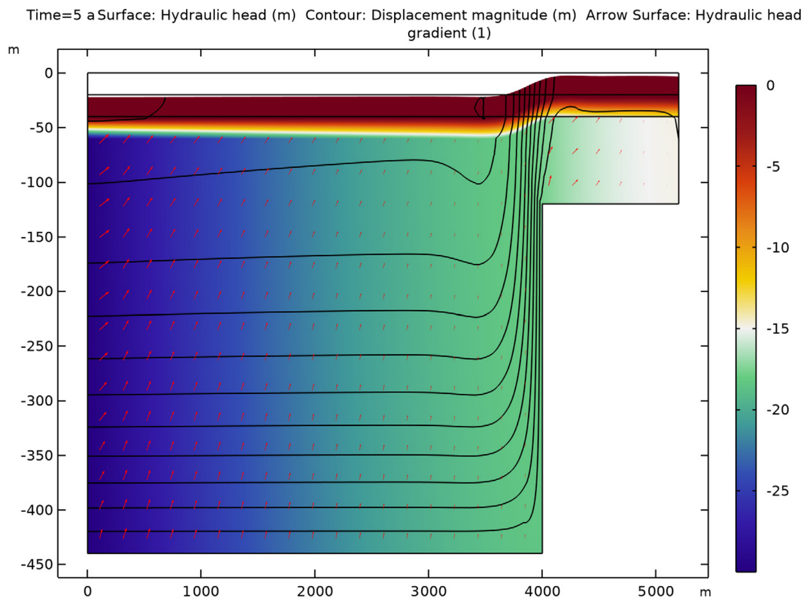

Figure 3: Hydraulic head (surface plot), displacement (contours and deformations), and hydraulic head gradient (arrows) at Year 5. The vertical axis and deformation are exaggerated for clarity.

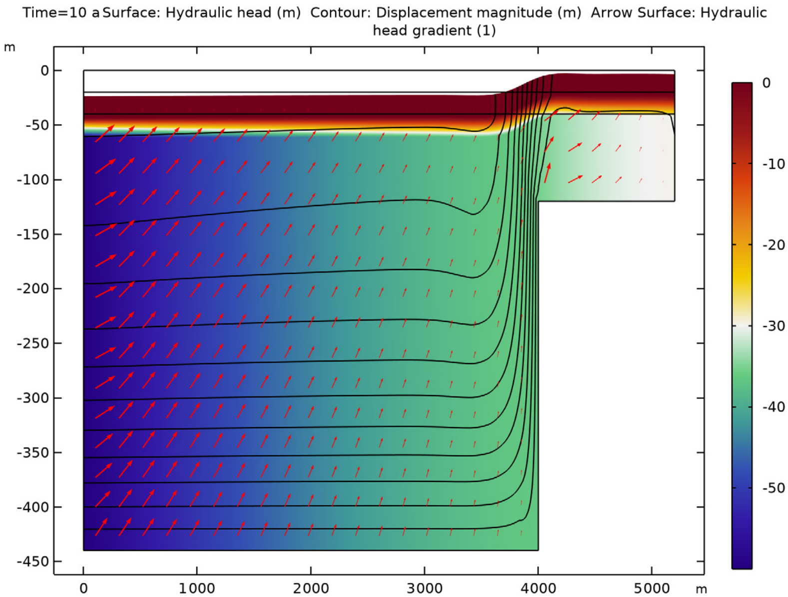

Figure 4: Hydraulic head (surface plot), displacement (contours and deformations), and hydraulic head gradient (arrows) at Year 10. The vertical axis and deformation are exaggerated for clarity.

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).

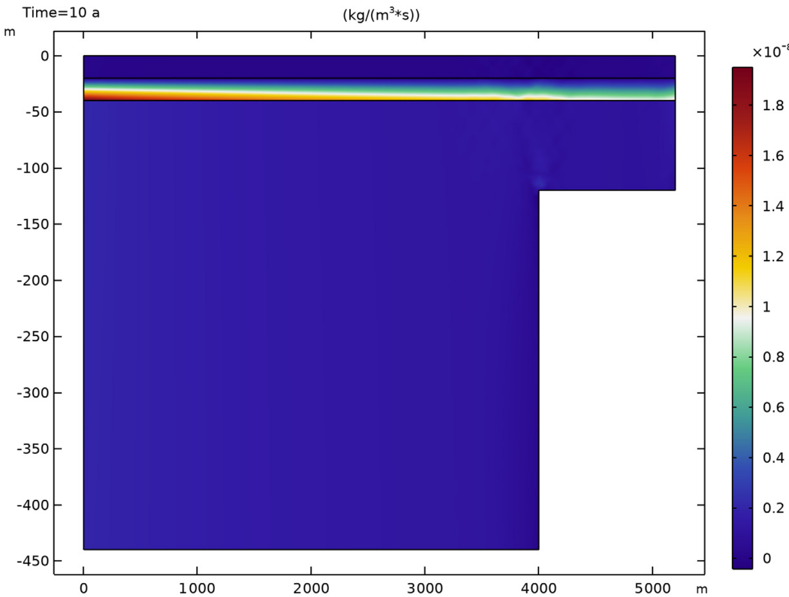

Figure 5: The solid-to-fluid coupling term evaluated at Year 10 with the poroelastic analysis. The squeeze of porous matrix results in an apparent mass source term.

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.

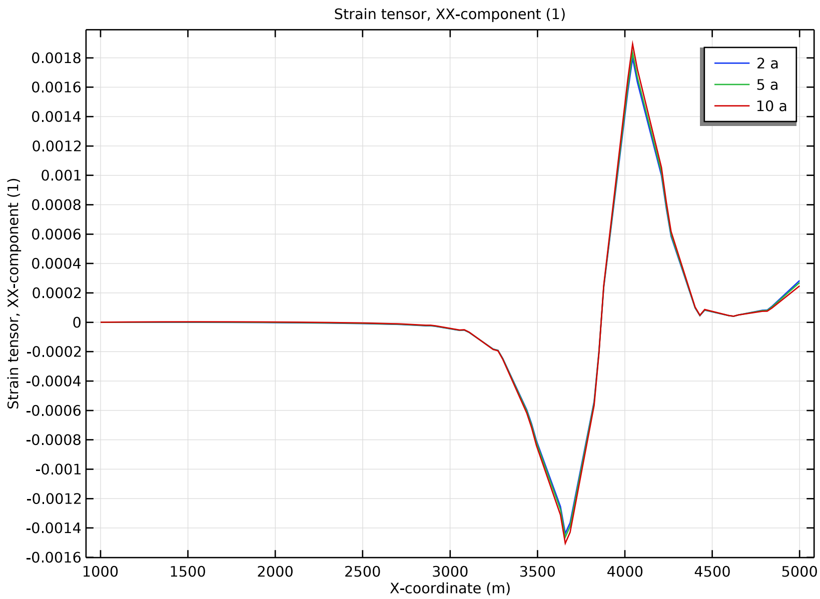

Figure 6: COMSOL Multiphysics estimates of horizontal strain from poroelastic analysis for the bedrock step problem of Leake and Hsieh: Year 2 (blue line), Year 5 (green line), and Year 10 (red line).

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 elastoplastic materials dialogs.

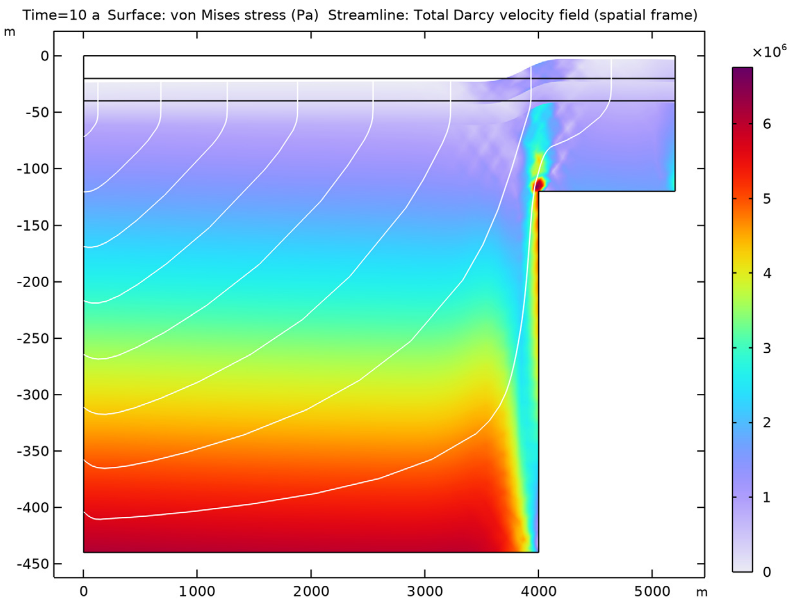

*Figure 7: COMSOL Multiphysics estimates of von Mises stresses (surface plot), fluid velocities (streamlines), and displacement (deformation) at Year 10. These results correspond to the poroelastic analysis from Leake and Hsieh (*Ref. 1*). The vertical axis and deformation are exaggerated for clarity.*

Notes About the COMSOL Implementation

During the simulation an equilibrium between the gravity forces and the poroelastic forces is reached after about 0.2 years. After that, the simulation results change further only due to the changing hydraulic head at the left-hand side boundary. To reproduce this behavior correctly, it is important to start with a small time step in the beginning to reproduce the development of an equilibrium state correctly, whereas later on, a bigger time step is sufficient and should be chosen to save computational time.

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/Poromechanics/biot_poroelasticity`

Modeling Instructions

ROOT

Start by opening the model containing the solution for the compaction according to Terzaghi theory and use it as entry point for the poroelasticity study according to Biot's theory.

APPLICATION LIBRARIES

1 From the **File** menu, choose **Application Libraries**.

2 In the **Application Libraries** window, select **Subsurface Flow Module** > **Poromechanics** > **terzaghi_compaction** in the tree.

3 Click **Open**.

ADD PHYSICS

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

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** > **Solid Mechanics (solid)**.

4 Find the **Physics interfaces in study** subsection. In the table, clear the **Solve** checkbox for **Study 1**.

5 Click the **Add to Component 1** button in the window toolbar.

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

ADD MULTIPHYSICS

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

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

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

4 Find the **Multiphysics couplings in study** subsection. In the table, clear the **Solve** checkbox for **Study 1**.

5 Click the **Add to Component** button in the window toolbar.

6 In the **Physics** toolbar, click **Add Multiphysics** to close the **Add Multiphysics** window.

SOLID MECHANICS (SOLID)

Gravity 1

In the **Physics** toolbar, click **Global** and choose **Gravity**.

Fixed Constraint 1

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

2 Select Boundaries 2, 8, and 9 only.

Roller 1

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

2 Select Boundaries 1, 3, and 5 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 α_B list, choose **User defined**. 1 is the default value for the Biot–Willis coefficient.

MATERIALS

Fill the required fields in the material properties tables.

Porous Material 1 (pmat1)

1 In the **Model Builder** window, expand the **Component 1 (comp1)** > **Materials** node, then 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	`2750`	kg/m³	Basic
Young's modulus	E	`800[MPa]`	Pa	Young's modulus and Poisson's ratio
Poisson's ratio	nu	`0.25`	1	Young's modulus and Poisson's ratio

Porous Material 2 (pmat2)

1 In the **Model Builder** window, click **Porous Material 2 (pmat2)**.

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	`2750`	kg/m³	Basic
Young's modulus	E	`80[MPa]`	Pa	Young's modulus and Poisson's ratio
Poisson's ratio	nu	`0.25`	1	Young's modulus and Poisson's ratio

DEFINITIONS (COMP1)

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

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(solid.evol,TIME)*dl.rho`	kg/(m³·s)

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.

STUDY 2

Step 1: Time Dependent

Now set up the fully coupled study. As described in Notes About the COMSOL Implementation in the beginning, the time step has to be small until an equilibrium between gravity forces and poroelastic forces is reached. Studies have shown that this is the case after 0.2 years. After that, a bigger time step is sufficient.

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

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

3 Click **Range**.

4 In the **Range** dialog, type `0.01` in the **Step** text field.

5 In the **Stop** text field, type `0.2`.

6 Click **Replace**.

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

8 Click **Range**.

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

10 In the **Start** text field, type `1`.

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

12 Click **Add**.

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

RESULTS

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

Hydraulic Head, Poroelasticity

1 In the **Results** toolbar, click **2D Plot Group**.

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

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

Surface 1

1 Right-click **Hydraulic Head, Poroelasticity** 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 **Component 1 (comp1)** > **Darcy's Law** > **Velocity and pressure** > **dl.H - Hydraulic head - m**.

Contour 1

1 In the **Model Builder** window, 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 **Component 1 (comp1)** > **Solid Mechanics** > **Displacement** > **solid.disp - Displacement magnitude - m**.

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** checkbox.

8 Click to expand the **Quality** section. From the **Evaluation settings** list, choose **Manual**.

9 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 Select the **Description** checkbox. In the associated text field, type `Hydraulic head gradient (1)`.

6 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, right-click **Surface 1** 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 `10`.

4 Right-click **Deformation 1** and choose **Copy**.

Deformation 1

In the **Model Builder** window, right-click **Contour 1** and choose **Paste Deformation**.

Arrow Surface 1

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

2 Select the **Scale factor** checkbox.

3 In the **Hydraulic Head, Poroelasticity** toolbar, click **Plot**. In the associated text field, type `2e4`.

Deformation 1

Right-click **Arrow Surface 1** and choose **Paste Deformation**.

Hydraulic Head, Poroelasticity

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

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

3 In the **Hydraulic Head, Poroelasticity** toolbar, click **Plot**.

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

Compare the resulting plot with that in Figure 2.

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

6 In the **Hydraulic Head, Poroelasticity** toolbar, click **Plot**.

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

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

8 In the **Hydraulic Head, Poroelasticity** toolbar, click **Plot**.

Compare with Figure 4.

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

von Mises Stress

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.

Surface 1

1 In the **Model Builder** window, expand the **von Mises Stress** node, then click **Surface 1**.

2 In the **Settings** window for **Surface**, click to expand the **Quality** section.

3 From the **Smoothing threshold** list, choose **None**.

Deformation

1 In the **Model Builder** window, expand the **Surface 1** node, then click **Deformation**.

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

3 Select the **Scale factor** checkbox. In the associated text field, type `10`.

Streamline 1

1 In the **Model Builder** window, 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 **Starting-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. Find the **Point style** subsection. From the **Color** list, choose **White**.

8 Click to expand the **Quality** section. From the **Evaluation settings** list, choose **Manual**.

9 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

In the **Model Builder** window, right-click **Streamline 1** and choose **Paste Deformation**.

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

Solid-to-Fluid Coupling Term

1 In the **Results** toolbar, click **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 **Dataset** 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 **Component 1 (comp1)** > **Definitions** > **Variables** > **Q_biot - kg/(m^3·s)**.

3 In 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 In the **Results** toolbar, click **Cut Line 2D**.
2 In the **Settings** window for **Cut Line 2D**, locate the **Data** section.
3 From the **Dataset** 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`.

Horizontal Strain

1 In 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 **Dataset** 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 **Component 1 (comp1)** > **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 **Component 1 (comp1)** > **Geometry** > **Coordinate (material and geometry frames)** > **X - X-coordinate**.
5 Click to expand the **Legends** section. Select the **Show legends** checkbox.
6 In the **Horizontal Strain** toolbar, click **Plot**.

Hydraulic Head, Poroelasticity

To produce a plot array of the hydraulic head at different times, follow the steps below.

Hydraulic Head, Poroelasticity, Array

1 In the **Model Builder** window, right-click **Hydraulic Head, Poroelasticity** and choose **Duplicate**.
2 In the **Model Builder** window, click **Hydraulic Head, Poroelasticity 1**.
3 In the **Settings** window for **2D Plot Group**, type `Hydraulic Head, Poroelasticity, Array` in the **Label** text field.

4 Click to expand the **Plot Array** section. From the **Array type** list, choose **Square**.

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

6 In the **Row displacement** text field, type -500.

7 In the **Column displacement** text field, type 6000.

Solution Array 1

1 In the **Model Builder** window, right-click **Surface 1** 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 (a)** list, choose **2**, **5**, **8**, and **10**.

5 Locate the **Plot Array** section. From the **Array shape** list, choose **Square**.

Arrow Surface 1

1 In the **Model Builder** window, under **Results** > **Hydraulic Head, Poroelasticity, Array** click **Arrow Surface 1**.

2 In the **Settings** window for **Arrow Surface**, click to expand the **Plot Array** section.

3 Select the **Manual indexing** checkbox.

Solution Array 1

1 Right-click **Arrow Surface 1** 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 (a)** list, choose **2**, **5**, **8**, and **10**.

5 Locate the **Plot Array** section. From the **Array shape** list, choose **Square**.

Contour 1

1 In the **Model Builder** window, under **Results** > **Hydraulic Head, Poroelasticity, Array** click **Contour 1**.

2 In the **Settings** window for **Contour**, click to expand the **Plot Array** section.

3 Select the **Manual indexing** checkbox.

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

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

Contour 2

1 Right-click **Results** > **Hydraulic Head, Poroelasticity, Array** > **Contour 1** and choose **Duplicate**.

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

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

4 Locate the **Plot Array** section. In the **Column index** text field, type 1.

Contour 3

Right-click **Contour 2** and choose **Duplicate**.

Contour 3

1 In the **Model Builder** window, expand the **Results** > **Hydraulic Head, Poroelasticity, Array** > **Contour 1** node, then click **Results** > **Hydraulic Head, Poroelasticity, Array** > **Contour 3**.

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

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

4 Locate the **Plot Array** section. In the **Row index** text field, type 1.

5 In the **Column index** text field, type 0.

Contour 4

1 Right-click **Results** > **Hydraulic Head, Poroelasticity, Array** > **Contour 3** and choose **Duplicate**.

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

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

4 Locate the **Plot Array** section. In the **Column index** text field, type 1.

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

Hydraulic Head, Poroelasticity, Array

In the **Model Builder** window, click **Hydraulic Head, Poroelasticity, Array**.

Table Annotation 1

1 In the **Hydraulic Head, Poroelasticity, Array** 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
0.2e4	-450	time = 2 a
0.85e4	-450	time = 5 a

x-coordinate	y-coordinate	Annotation
`0.2e4`	`-950`	`time = 8 a`
`0.85e4`	`-950`	`time = 10 a`

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

6 Clear the **Show point** checkbox.

7 In the **Hydraulic Head, Poroelasticity, Array** toolbar, click **Plot**.

Hydraulic Head, Poroelasticity, Array

1 In the **Model Builder** window, click **Hydraulic Head, Poroelasticity, Array**.

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

3 From the **Title type** list, choose **Manual**.

4 In the **Title** text area, type `Surface: Hydraulic head (m) Contour: Displacement magnitude (m) Arrow Surface: Hydraulic head gradient (1)`.

5 Clear the **Parameter indicator** text field.

6 In the **Hydraulic Head, Poroelasticity, Array** toolbar, click **Plot**.

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

Group the plots to keep the **Result** node clear.

Horizontal Strain, Hydraulic Head, Poroelasticity, Hydraulic Head, Poroelasticity, Array, Pressure (dl), Solid-to-Fluid Coupling Term, Velocity (dl) 1, von Mises Stress

1 In the **Model Builder** window, under **Results**, Ctrl-click to select **Pressure (dl)**, **Velocity (dl) 1**, **von Mises Stress**, **Hydraulic Head, Poroelasticity**, **Solid-to-Fluid Coupling Term**, **Horizontal Strain**, and **Hydraulic Head, Poroelasticity, Array**.

2 Right-click and choose **Group**.

Biot

In the **Settings** window for **Group**, type `Biot` in the **Label** text field.

Compaction, Hydraulic Head, Velocity (dl)

1 In the **Model Builder** window, under **Results**, Ctrl-click to select **Hydraulic Head**, **Velocity (dl)**, and **Compaction**.

2 Right-click and choose **Group**.

Terzaghi

In the **Settings** window for **Group**, type `Terzaghi` in the **Label** text field.