Model created in COMSOL Multiphysics 6.4

Terzaghi Compaction

Introduction

Fluids that move through pore spaces in an aquifer or reservoir can shield the porous medium from stress because they bear part of the load from, for instance, overlying rocks, sediments, fluids, and buildings. Withdrawing fluids from the pore space increases the stress the solids bear, sometimes to the degree that the reservoir measurably compacts. The reduction in the pore space loops back and alters the fluid pressures. The feedback brings about more fluid movement, and the cycle continues. Terzaghi Compaction describes a conventional flow model and uses the results in postprocessing to calculate vertical compaction following Terzaghi theory (Ref. 2).

Model Definition

This example analyzes fluid and solid behavior within three sedimentary layers overlying 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.

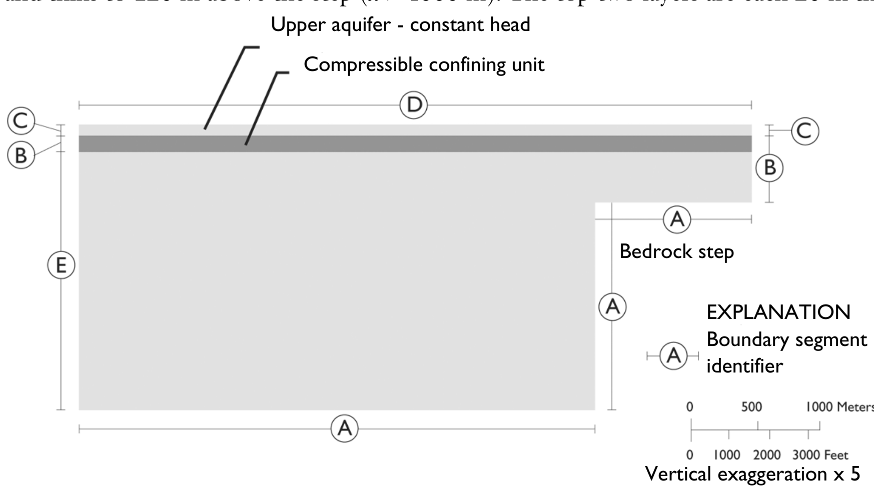

Figure 1: Model geometry showing boundary segments (from Leake and Hsieh, Ref. 1).

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.

This example sets up a traditional flow model and analyzes the vertical displacement during postprocessing. The flow field is fully described using the Darcy velocity in an equation of continuity

$$S_{\mathrm{h}}\frac{\partial H}{\partial t} + \nabla \cdot (-K\nabla H) = 0 \tag{1}$$

where S_{h} is the storage coefficient (m^{-1}), K equals hydraulic conductivity (m/s), and H represents hydraulic head (m). In most conventional flow models, S_{h} represents small changes in fluid volume and pore space. It combines terms describing the fluid's compressibility, the solids' compressibility, and the reservoir's porosity. In the original research (Ref. 1) and in this model, S_{h} is the specific storage of the solid skeleton, S_{sk}.

Instead of solving Darcy's law in the hydraulic head formulation, we solve Equation 1 in the pressure formulation

$$\rho S\frac{\partial p}{\partial t} + \nabla \cdot \rho\left[-\frac{\kappa}{\mu}(\nabla p + \rho g\nabla D)\right] = 0$$

here, the storage coefficient S (1/Pa) is related to the fluid density, acceleration of gravity and the storage coefficient given in Equation 1 by the relation $S = S_{\mathrm{h}}/\rho g$. Also, the hydraulic head is related to the fluid pressure and elevation $H = p/\rho g + D$, and the hydraulic conductivity is related to the permeability and dynamic viscosity of the fluid $K = \kappa\rho g/\mu$.

Because the aquifer is at equilibrium prior to pumping, you set up this model to predict the change in hydraulic head rather than the hydraulic head values themselves. The main advantage of this approach lies in establishing initial and boundary conditions. Here you specify that the hydraulic head along the centerline of the basin decreases linearly by 60 m over ten years, then simply state that the hydraulic head initially is zero and remains so where heads do not change in time.

The boundary and initial conditions are

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

where $\mathbf{n}$ is the normal to the boundary. The letters A through E, taken from Leake and Hsieh (Ref. 1), denote the boundary (see Figure 1).

Terzaghi theory uses skeletal specific storage or aquifer compressibility to calculate the vertical compaction Δb (m) of the aquifer sediments in a given representative volume as

$$\Delta b = S_{sk} b(-H) \qquad (2)$$

where b is standard notation for the vertical thickness of aquifer sediments (m).

Model Data

The following table gives the data for the Terzaghi compaction model:

TABLE 1: MODEL DATA.

VARIABLE	DESCRIPTION	VALUE
ρ_f	Fluid density	1000 kg/m^3
S_{sk}	Skeletal specific storage, aquifer layers	$1\cdot10^{-5}$ m^{-1}
	Skeletal specific storage, confining layer	$1\cdot10^{-4}$ m^{-1}
K_s	Hydraulic conductivity, aquifer layers	25 m/d
	Hydraulic conductivity, confining layer	0.01 m/d
$H(0)$	Initial hydraulic head	0 m
$H_0(t)$	Declining head boundary	(6 m/year)·t

Results and Discussion

Figure 2 shows a Year-10 snapshot from the COMSOL Multiphysics solution to the Terzaghi compaction example. The results describe conventional Darcy flow toward the centering of a basin, moving away from a bedrock step ($x > 4000$ m). The shading represents the change in hydraulic head brought on by pumping at $x = 0$ m. The streamlines and arrows denote the direction and magnitude of the fluid velocity. The flow goes from vertical near the surface to horizontal at the outlet. Where the sediments thicken at the edge of the step, the hydraulic gradient and the fluid velocities change abruptly.

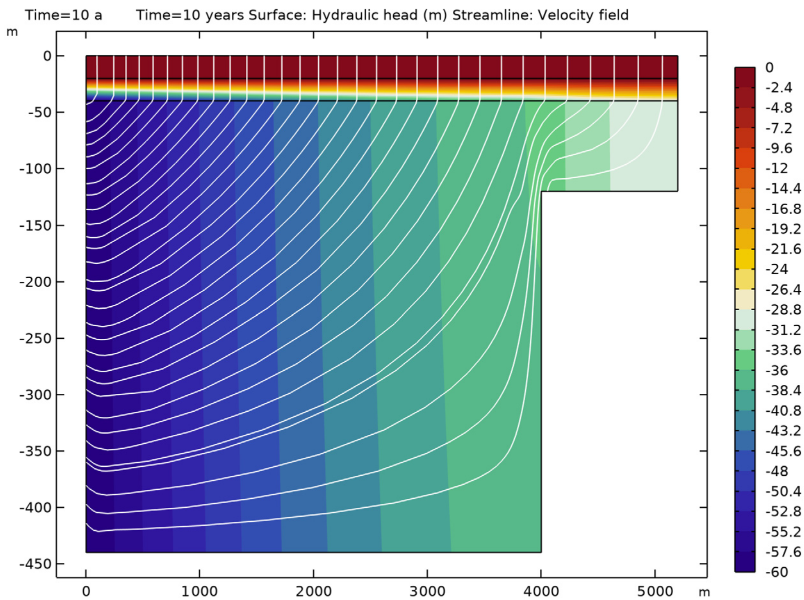

Figure 2: COMSOL Multiphysics solution to a Terzaghi flow problem. The figure shows change in hydraulic head (surface plot) and fluid velocity (streamlines).

The compaction calculated according to Equation 2 is shown as a contour plot in Figure 3.

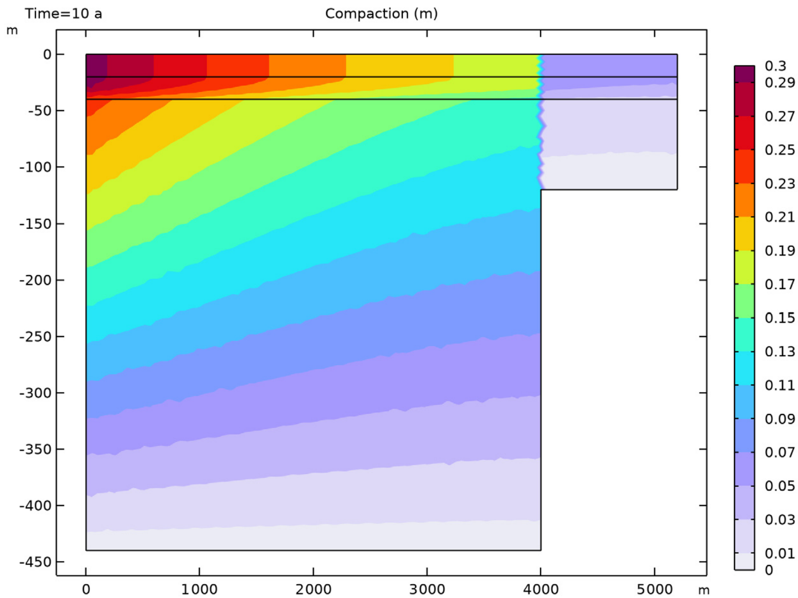

Figure 3: Contour plot of the compaction after 10 years.

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. K. Terzaghi, *Theoretical Soil Mechanics*, John Wiley & Sons, p. 510, 1943.

Application Library path: `Subsurface_Flow_Module/Poromechanics/terzaghi_compaction`

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**.
2 In the **Select Physics** tree, select **Fluid Flow** > **Porous Media and Subsurface Flow** > **Darcy's Law (dl)**.
3 Click **Add**.
4 Click **Study**.
5 In the **Select Study** tree, select **General Studies** > **Time Dependent**.
6 Click **Done**.

GEOMETRY 1

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 5200.
4 In the **Height** text field, type 440.
5 Locate the **Position** section. In the **y** text field, type -440.
6 Click to expand the **Layers** section. Select the **Layers on top** checkbox.
7 Clear the **Layers on bottom** checkbox.
8 In the table, enter the following settings:

Layer name	Thickness (m)
Layer 1	20
Layer 2	20

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 1200.
4 In the **Height** text field, type 320.
5 Locate the **Position** section. In the **x** text field, type 4000.
6 In the **y** text field, type -440.

Difference 1 (dif1)

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

2 Select the object **r1** only, to add it to the **Objects to add** list.

3 In the **Settings** window for **Difference**, locate the **Difference** section.

4 Click to select the **Activate Selection** toggle button for **Objects to subtract**.

5 Select the object **r2** only.

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

DEFINITIONS

Because the model geometry is so elongated, you get a better view of it by turning off the default setting that preserves the aspect ratio.

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

Axis

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

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

3 From the **View scale** list, choose **Automatic**.

4 Click **Update**.

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

Now, define variables for the skeletal specific storage and saturated hydraulic conductivity. Using local variables allows you to define variables on different domains with different expressions but identical names.

Variables 1

1 In the **Definitions** toolbar, click **Local Variables**.

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

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

4 Select Domains 1 and 3 only.

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

Name	Expression	Unit	Description
S_sk	1e-5[m^-1]	1/m	Skeletal specific storage
K_s	25[m/day]	m/s	Saturated hydraulic conductivity

Variables 2

1 In the **Definitions** toolbar, click **Local Variables**.
2 In the **Settings** window for **Variables**, locate the **Geometric Entity Selection** section.
3 From the **Geometric entity level** list, choose **Domain**.
4 Select Domain 2 only.
5 Locate the **Variables** section. In the table, enter the following settings:

Name	Expression	Unit	Description
S_sk	1e-4[m^-1]	1/m	Skeletal specific storage
K_s	0.01[m/day]	m/s	Saturated hydraulic conductivity

GLOBAL DEFINITIONS

Fluid

1 In the **Model Builder** window, under **Global Definitions** right-click **Materials** and choose **Blank Material**.
2 In the **Settings** window for **Material**, type `Fluid` in the **Label** text field.

MATERIALS

Porous Material 1 (pmat1)

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Materials** and choose **More Materials** > **Porous Material**.
2 Select Domains 1 and 3 only.
3 In the **Settings** window for **Porous Material**, locate the **Porosity** section.
4 In the ε_p text field, type `0.25`.
5 Locate the **Phase-Specific Properties** section. Click **Add Required Phase Nodes**.

Porous Material 2 (pmat2)

1 In the **Model Builder** window, right-click **Porous Material 1 (pmat1)** and choose **Duplicate**.
2 In the **Settings** window for **Porous Material**, locate the **Geometric Entity Selection** section.
3 Click **Clear Selection**.
4 Select Domain 2 only.
5 Locate the **Porosity** section. In the ε_p text field, type `0.025`.

DARCY'S LAW (DL)

Define all physical effects. After that, the material properties can be completed.

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

2 In the **Settings** window for **Darcy's Law**, locate the **Gravity Effects** section.

3 Select the **Include gravity** checkbox.

Gravity 1

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

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

3 From the **Specify** list, choose **Elevation**.

Porous Medium 1

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

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

3 From the **Storage model** list, choose **User defined**. In the S_p text field, type `S_sk/dl.rho/g_const`.

We will enter a constant fluid density, therefore to catch compressibility of the fluid, choose "compressible, linearized" and enter a fluid compressibility in the **Materials** settings.

Fluid 1

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

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

3 From the **Fluid type** list, choose **Compressible, linearized**.

Porous Matrix 1

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

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

3 From the **Permeability model** list, choose **Hydraulic conductivity**.

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

GLOBAL DEFINITIONS

Fluid (mat1)

1 In the **Model Builder** window, under **Global Definitions** > **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	Variable	Value	Unit	Property group
Density	rho	1000	kg/m³	Basic
Compressibility of fluid	chif	4e-10	1/Pa	Basic

Now, set up the boundary conditions.

DARCY'S LAW (DL)

Hydraulic Head 1

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

2 Select Boundary 1 only.

3 In the **Settings** window for **Hydraulic Head**, locate the **Hydraulic Head** section.

4 In the H_0 text field, type `-6[m/year]*t`.

Hydraulic Head 2

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

2 Select Boundaries 5, 7, and 12 only.

To calculate the compaction according to Equation 2 first, define a projection operator to integrate along the y-axis. Add a new variable which uses the projection operator to perform a convolution integral using the `dest` operator.

DEFINITIONS (COMP1)

General Projection 1 (genproj1)

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

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

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

Variables 3

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
b	`genproj1(-dl.H*(y<=dest(y))*S_sk)`	m	Compaction

To get a good resolution for the compaction in the *y* direction, use a finer mesh size and scale the geometry for the mesh algorithm.

MESH 1

Free Triangular 1

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

2 In the **Settings** window for **Free Triangular**, click to expand the **Scale Geometry** section.

3 In the **y-direction scale** text field, type `10`.

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**.

4 Click **Build All**.

STUDY 1

Step 1: Time Dependent

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

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

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

4 In the **Output times** text field, type `range(0,1,10)`.

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

RESULTS

Hydraulic Head

Follow these steps to reproduce the plot shown in Figure 2:

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

2 Click to expand the **Title** section. From the **Title type** list, choose **Manual**.

3 In the **Title** text area, type `Time=10 years Surface: Hydraulic head (m) Streamline: Velocity field`.

Surface

1 In the **Model Builder** window, expand the **Hydraulic Head** 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 **Component 1 (comp1)** > **Darcy's Law** > **Velocity and pressure** > **dl.H - Hydraulic head - m**.

Hydraulic Head

In the **Hydraulic Head** toolbar, click **Streamline**.

Streamline 1

1 Select Boundary 7 only.

2 In the **Settings** window for **Streamline**, locate the **Streamline Positioning** section.

3 From the **Point distribution** list, choose **Magnitude controlled**.

4 In the **Maximum relative distance** text field, type `0.05`.

5 Locate the **Coloring and Style** section. Find the **Point style** subsection. From the **Color** list, choose **White**.

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

Plot the compaction as shown in Figure 3.

Compaction

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

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

Contour 1

1 Right-click **Compaction** 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)** > **Definitions** > **Variables** > **b - Compaction - m**.

3 Locate the **Levels** section. In the **Total levels** text field, type `15`.

4 Locate the **Coloring and Style** section. From the **Contour type** list, choose **Filled**.

5 From the **Color table** list, choose **Prism**.

6 In the **Compaction** toolbar, click **Plot**.