Created in COMSOL Multiphysics 6.1

# Slope Stability in an Embankment Dam

## Introduction

Slope stability analysis is an essential technique for predicting the settlement, deformation, and slippage of soil due to various loading and environmental conditions. In an embankment dam, slope stability analysis is important to determine the safety of the dam.

The present model is inspired by an example included in Ref. 1. The pore pressure in the soil is modeled by Darcy's law, while the Mohr–Coulomb criterion is used for the elastoplastic analysis.

The technique used for studying the slope stability is called the strength reduction method, where the Mohr–Coulomb material parameters are functions of the Factor of Safety (FOS). In this method, the FOS is gradually increased, which reduces the material parameters and shear strength of the soil, which eventually becomes unstable. This phenomenon can produce a collapse of the embankment at a certain FOS level for given loading conditions. More details of this technique are given in Ref. 1.

## Model Definition

Figure 1 shows the cross section of the embankment dam. The dimensions $L_1$, $L_2$, and $L_3$ are 24 m, 5 m, and 24 m, respectively, and the height of the embankment $L_4$ is 12 m. The water level is 10 m and the possible seepage height is 4 m. The total width of the embankment is $L_1 + L_2 + L_3$. To avoid boundary effects, a soil domain is added below the embankment (not shown in Figure 1).

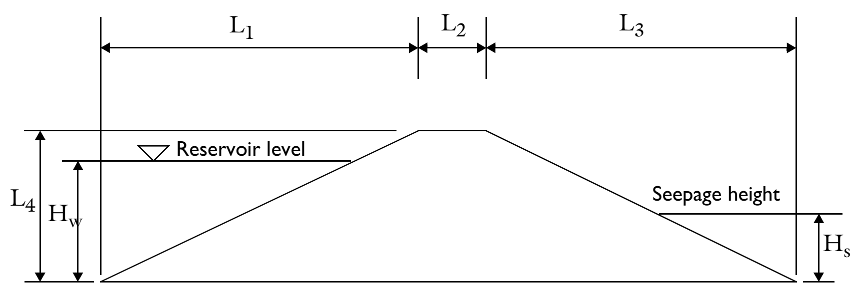


*Figure 1: Illustration of the embankment dam.*

In this example, a plane strain approximation is used to model the embankment dam in 2D. The effects of gravity and hydrostatic pressure are also included. The material properties for the Mohr–Coulomb model are parameterized with respect to a factor of safety parameter, $\mathrm{FOS}$. A parametric study increases the $\mathrm{FOS}$ parameter, thereby reducing the strength of the soil with every parameter step. The model does not converge for FOS values above $1.92$, which indicates a collapse of the slope.

The Mohr–Coulomb yield function and associated plastic potential is

$$F = Q = m\sqrt{J_2} + \alpha I_1 - k \tag{1}$$

with

$$\alpha = \frac{\sin\Phi}{3}, k = C\cos\Phi \tag{2}$$

The parameterized cohesion $C$ and angle of internal friction $\Phi$ are given as

$$C = \frac{c}{\mathrm{FOS}}, \Phi = \mathrm{atan}\left(\frac{\tan\phi_u}{\mathrm{FOS}}\right)(p<0) + \mathrm{atan}\left(\frac{\tan\phi_s}{\mathrm{FOS}}\right)(p\geq 0) \tag{3}$$

where $c$ is the material cohesion, $\phi_{\mathrm{u}}$ and $\phi_s$ are angles of internal friction for unsaturated and saturated soils, and $p$ is the pore pressure given by Darcy's law.

## *Results and Discussion*

The pressure head in the embankment dam is shown in Figure 2; it varies from 0 m to 10 m on the submerged wall, while it is 0 m on the seepage face. Positive pressure head means positive pore pressure, which indicates a saturated soil, while unsaturated soil is represented with zero pressure head. The zero pressure head line in the figure is the location of a phreatic surface that divides saturated from the unsaturated soil.

The elastoplastic analysis does not converge for $\mathrm{FOS}$ values greater than $1.92$; hence, the simulation is performed until its value becomes $1.92$, which is the value at which the slope collapses due to the increase in plastic strains and subsequent reduction in shear strength.

.

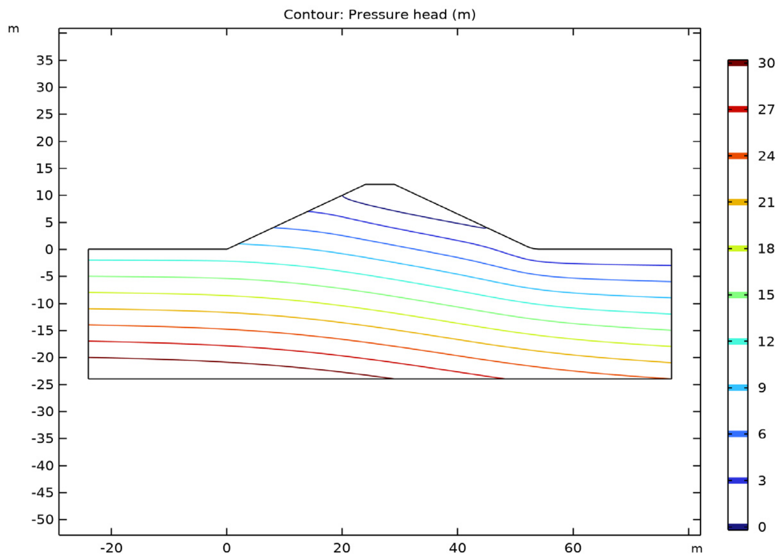


*Figure 2: Pressure head in the embankment dam. The zero pressure head line shows the location of phreatic surface.*

The equivalent plastic strain just before the collapse shows a different pattern, which gives an indication of the failure mechanism (see Figure 3).

The slip surface is shown in Figure 4. The arrows show the direction of displacement for the soil particles. This figure illustrates the phenomenon of soil slippage. The soil near the lower-right corner does not slip because of the fixed constraint on the lower boundary. The slip surface figure matches well qualitatively with the results given in Ref. 1.

A 3D visualization of the displacement field is shown in Figure 5 with the help of an extrusion dataset. A 2D analysis of an embankment dam is an efficient way to predict soil instability for wide dams where a plane strain approximation is reasonable; a 3D visualization is still possible with the help of the postprocessing tools in COMSOL Multiphysics. This simplification avoids solving a larger numerical problem in 3D.

The plot of the maximum displacement versus the FOS is shown in Figure 6. The maximum displacement increases significantly at around FOS = 1.9, which indicates the onset of the collapse of the slope.

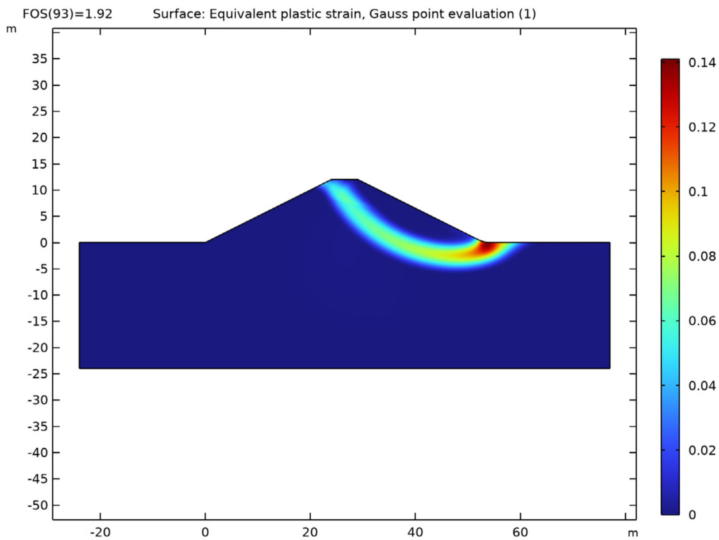


*Figure 3: Equivalent plastic strain just before the collapse of the slope.*

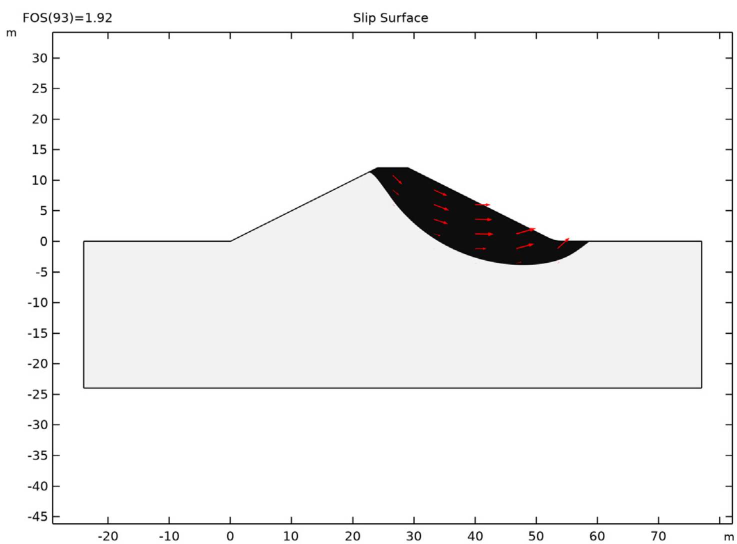


*Figure 4: Slip circle just before the collapse of the slope.*

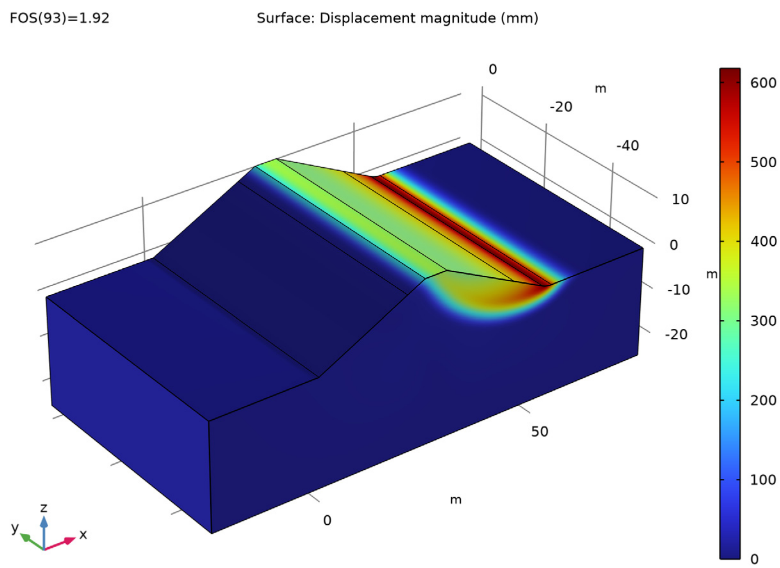


*Figure 5: Displacement magnitude in the embankment dam just before the collapse of the slope.*

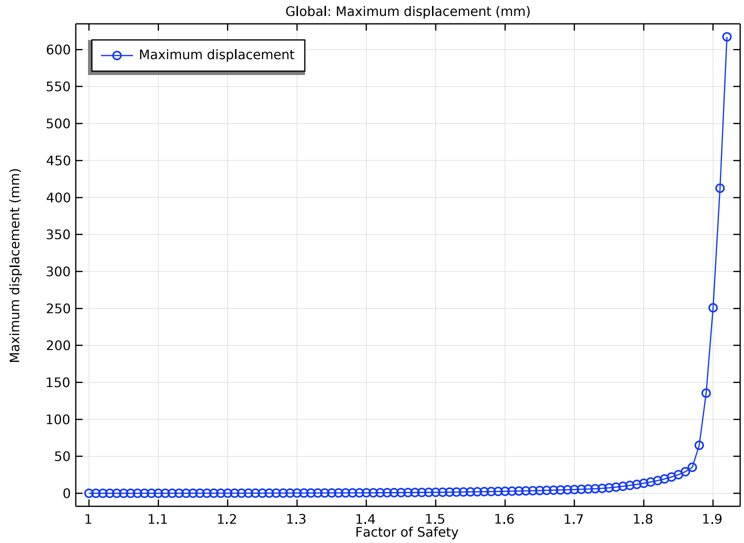


*Figure 6: Maximum displacement versus FOS.*

## *Notes About the COMSOL Implementation*

Three stationary studies are created in order to account for the effects of pore pressure and gravity loads on the stability of the slope. In the first study, only *Darcy's Law* is computed to solve for the pore pressure profile. In a second study, the initial stresses in the embankment are computed given the hydrostatic, pore pressure, and gravity loads. In the third study step, these initial stresses are taken into account by adding an **Initial Stress and Strain** node. Moreover, the Mohr–Coulomb soil plasticity model is used to analyze the elastoplastic failure of the soil under combined hydraulic and gravity loading.

The angle of internal friction is different for saturated and unsaturated soils, hence an expression that depends on the pore pressure is used. No external stresses are applied in regions of unsaturated soil, since pores are considered interconnected and at constant atmospheric pressure.

The studied problem involves localization of deformation into a band, the slip surface. To maintain mesh objectivity of the solution, a length scale is introduced to the material model to limit strain localization to a predefined width. Here the **Implicit gradient** nonlocal plasticity model is used with a length scale $l_{\mathrm{int}} = 0.1$ m. The value $l_{\mathrm{int}}$ is here chosen so that the band of plastic strains is distributed over several mesh elements.

Including plasticity in the material model requires local computations at each Gauss point during the assembly process, which is expensive. By using **Reduced Integration**, the number of Gauss points are reduced by a factor two for the given displacement shape function order and mesh element type. This will speed up the computation significantly.

An additional extrusion dataset is created to generate a 3D plot from the 2D dataset, and the 3D view is adjusted in order to properly visualize it. As stated in Ref. 1, the non-convergence of the simulation is considered as an indicator of slope failure.

## *References*


1. D.V.Griffiths and P.A.Lane, "Slope Stability Analysis by Finite Elements," *Géotechnique*, vol. 49, no. 3, pp. 387–403, 1999.


**Application Library path:** `Geomechanics_Module/Soil/slope_stability`

## *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. In the **Select Physics** tree, select **Structural Mechanics>Solid Mechanics (solid)**.
5. Click **Add**.
6. Click **Study**.
7. In the **Select Study** tree, select **General Studies>Stationary**.
8. Click **Done**.

### GEOMETRY 1

Model parameters and interpolation function data are available in the appended text files.

## GLOBAL DEFINITIONS

*Parameters 1*

1 In the **Model Builder** window, under **Global Definitions** click **Parameters 1**.

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

3 Click **Load from File**.

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

*Interpolation 1 (int1)*

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

2 In the **Settings** window for **Interpolation**, locate the **Definition** section.

3 Click **Load from File**.

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

5 In the **Function name** text field, type `cond`.

6 Locate the **Units** section. In the **Argument** table, enter the following settings:

| Argument | Unit |
|---|---|
| t | m |

7 In the **Function** table, enter the following settings:

| Function | Unit |
|---|---|
| cond | m/s |

## GEOMETRY 1

Construct the 2D geometry using a polygon.

*Polygon 1 (pol1)*

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

2 In the **Settings** window for **Polygon**, locate the **Coordinates** section.

**3** In the table, enter the following settings:

| x (m) | y (m) |
|---|---|
| 0 | 0 |
| L1 | L4 |
| L1+L2 | L4 |
| L1+L2+L3 | 0 |
| L1+L2+L3*2 | 0 |
| L1+L2+L3*2 | -L4*2 |
| -L1 | -L4*2 |
| -L1 | 0 |

Add a fillet to remove the geometric discontinuity.

*Fillet 1 (fil1)*

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

**2** On the object **pol1**, select Point 4 only.

**3** In the **Settings** window for **Fillet**, locate the **Radius** section.

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

**5** Click **Build All Objects**.

Add points at the reservoir level and the possible seepage level to partition the sides of the dam.

*Point 1 (pt1)*

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

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

**3** In the **x** text field, type `Hw*L1/L4`.

**4** In the **y** text field, type `Hw`.

*Point 2 (pt2)*

**1** Right-click **Point 1 (pt1)** and choose **Duplicate**.

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

**3** In the **x** text field, type `L1+L2+L3-Hs*L1/L4`.

**4** In the **y** text field, type `Hs`.

**5** Click **Build Selected**.

## DEFINITIONS

*Variables 1*

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

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

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

4 In the table, enter the following settings:

| Name | Expression | Unit | Description |
|---|---|---|---|
| Saturated | dl.Hp>=0 | | Boolean variable for saturated region |
| Unsaturated | dl.Hp<0 | | Boolean variable for unsaturated region |
| K | cond(dl.Hp) | m/s | Hydraulic conductivity |
| C | c/FOS | Pa | Parameterized cohesion |
| PHI | atan(tan(phi_un)/FOS)*Unsaturated+atan(tan(phi_sat)/FOS)*Saturated | rad | Parameterized friction angle |

*Maximum 1 (maxop1)*

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

2 Select Domain 1 only.

Add two blank materials, one for the soil and one for the water, then rename them accordingly. For the water material, keep the domain selection empty.

## GLOBAL DEFINITIONS

*Soil*

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

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

*Water*

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

2 In the **Settings** window for **Material**, type `Water` 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 In the **Settings** window for **Porous Material**, locate the **Homogenized Material** section.

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

*Fluid 1 (pmat1.fluid1)*

1 Right-click **Porous Material 1 (pmat1)** and choose **Fluid**.

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

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

*Solid 1 (pmat1.solid1)*

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

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

3 In the $\theta_s$ text field, type `1-psi`.

## DARCY'S LAW (DL)

*Porous Matrix 1*

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

## GLOBAL DEFINITIONS

*Soil (mat1)*

1 In the **Model Builder** window, under **Global Definitions>Materials** click **Soil (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 | `rho_soil` | kg/m³ | Basic |

| Property | Variable | Value | Unit | Property group |
|---|---|---|---|---|
| Young's modulus | E | `E_soil` | Pa | Young's modulus and Poisson's ratio |
| Poisson's ratio | nu | `nu_soil` | 1 | Young's modulus and Poisson's ratio |

*Water (mat2)*

1 In the **Model Builder** window, click **Water (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_wat` | kg/m³ | Basic |

### DARCY'S LAW (DL)

Add a pressure head to the submerged parts of the downstream and upstream sides of the dam.

*Pressure Head 1*

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

2 Select Boundaries 1, 3, and 4 only.

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

4 In the $H_{p0}$ text field, type `Hw-y`.

*Pressure Head 2*

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

2 Select Boundaries 8, 9, and 11 only.

*Pressure Head 3*

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

2 Select Boundary 10 only.

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

4 In the $H_{p0}$ text field, type `-y`.

5 In the **Model Builder** window, click **Darcy's Law (dl)**.

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

7 Select the **Include gravity** check box.

### *Gravity 1*

1. In the **Model Builder** window, click **Gravity 1**.
2. In the **Settings** window for **Gravity**, locate the **Gravity** section.
3. From the **Specify** list, choose **Elevation**.

Use a finer mesh in the region where we expect the slip surface to form.

## MESH 1

1. In the **Model Builder** window, under **Component 1 (comp1)** click **Mesh 1**.
2. In the **Settings** window for **Mesh**, locate the **Sequence Type** section.
3. From the list, choose **User-controlled mesh**.

### *Size*

1. In the **Model Builder** window, under **Component 1 (comp1)>Mesh 1** click **Size**.
2. In the **Settings** window for **Size**, locate the **Element Size** section.
3. From the **Predefined** list, choose **Fine**.

### *Size Expression 1*

1. In the **Model Builder** window, right-click **Mesh 1** and choose **Size Expression**.
2. Drag and drop **Size Expression 1** below **Size**.
3. In the **Settings** window for **Size Expression**, locate the **Element Size Expression** section.
4. In the **Size expression** text field, type `if(Y>-L4&&X>L1/2&&X<(L1+L2+L3*1.5),0.75, 10)`.
5. In the **Number of cells per dimension** text field, type `50`.
6. Click **Build All**.

## STUDY 1-DARCY'S LAW

Disable the default plots for this study.

1. In the **Model Builder** window, click **Study 1**.
2. In the **Settings** window for **Study**, type `Study 1-Darcy's Law` in the **Label** text field.
3. Locate the **Study Settings** section. Clear the **Generate default plots** check box.

Disable the **Solid Mechanics** physics from the study to solve only for the pore pressure.

### *Step 1: Stationary*

1. In the **Model Builder** window, under **Study 1-Darcy's Law** click **Step 1: Stationary**.
2. In the **Settings** window for **Stationary**, locate the **Physics and Variables Selection** section.

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

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

Add a contour plot of the pressure head.

## RESULTS

*Pressure Head*

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

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

*Contour 1*

**1** Right-click **Pressure Head** 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)>Darcy's Law>Velocity and pressure>dl.Hp - Pressure head - m**.

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

**4** In the **Levels** text field, type `range(0,3,30)`.

**5** Locate the **Coloring and Style** section. From the **Contour type** list, choose **Tube**.

**6** Select the **Radius scale factor** check box. In the associated text field, type `0.1`.

**7** In the **Pressure Head** toolbar, click **Plot**.

## SOLID MECHANICS (SOLID)

Add the water pressure as a boundary load on the downstream side of the dam.

**1** In the **Model Builder** window, under **Component 1 (comp1)** click **Solid Mechanics (solid)**.

*Boundary Load 1*

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

**2** Select Boundaries 3 and 4 only.

**3** In the **Settings** window for **Boundary Load**, locate the **Force** section.

**4** From the **Load type** list, choose **Pressure**.

**5** In the $p$ text field, type `p`.

*Fixed Constraint 1*

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

**2** Select Boundary 2 only.

*Roller 1*

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

2 Select Boundaries 1 and 10 only.

*Gravity 1*

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

*Linear Elastic Material 1*

Use reduced integration to speed up the simulation.

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

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

3 Select the **Reduced integration** check box.

*External Stress 1*

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

2 In the **Settings** window for **External Stress**, locate the **External Stress** section.

3 From the **Stress input** list, choose **Pore pressure**.

4 In the $p_A$ text field, type `p*Saturated`.

5 From the $\alpha_B$ list, choose **User defined**. In the associated text field, type `1`.

6 In the $p_{ref}$ text field, type `0`.

Add another study to compute the in situ stresses generated by gravity and pore pressure. Disable the default plots for this study.

## 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>Stationary**.

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

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

## STUDY 2 - SOLID MECHANICS (IN SITU STRESS INITIALIZATION)

1 In the **Model Builder** window, click **Study 2**.

2 In the **Settings** window for **Study**, type `Study 2 - Solid Mechanics (In Situ Stress Initialization)` in the **Label** text field.

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

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

### SOLID MECHANICS (SOLID)

*Linear Elastic Material 1*

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

*Soil Plasticity 1*

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

2 In the **Settings** window for **Soil Plasticity**, locate the **Soil Plasticity** section.

3 From the **Material model** list, choose **Mohr-Coulomb**.

4 From the **Plastic potential** list, choose **Associated**.

Use a nonlocal plasticity model to improve the solution during strain localization.

5 Click to expand the **Nonlocal Plasticity Model** section. From the list, choose **Implicit gradient**.

6 In the $l_{\mathrm{int}}$ text field, type `0.1`.

*Linear Elastic Material 1*

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

*Initial Stress and Strain 1*

1 In the **Physics** toolbar, click **Attributes** and choose **Initial Stress and Strain**.

Add the stresses computed in the second study step as initial stresses. You can access these stresses by using the `withsol` operator as follows:

2 In the **Settings** window for **Initial Stress and Strain**, locate the **Initial Stress and Strain** section.

3 In the $\mathrm{S}_0$ table, enter the following settings:

| | | |
|---|---|---|
| `withsol('sol2', solid.sx)` | `withsol('sol2', solid.sxy)` | `withsol('sol2', solid.sxz)` |
| withsol('sol2',solid.sxy) | `withsol('sol2', solid.sy)` | `withsol('sol2', solid.syz)` |
| withsol('sol2',solid.sxz) | withsol('sol2',solid.syz) | `withsol('sol2', solid.sz)` |

### GLOBAL DEFINITIONS

*Soil (mat1)*

Add material properties to the **Mohr-Coulomb** model. These are functions of the factor of safety from the parameter list.

1 In the **Model Builder** window, under **Global Definitions>Materials** click **Soil (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 |
|---|---|---|---|---|
| Cohesion | cohesion | `comp1.C` | Pa | Mohr-Coulomb |
| Angle of internal friction | internalphi | `comp1.PHI` | rad | Mohr-Coulomb |

Disable the **Soil Plasticity** and the **Initial Stress and Strain** nodes from the second study step. Add a third study for the elastoplastic analysis.

## STUDY 2 - SOLID MECHANICS (IN SITU STRESS INITIALIZATION)

### *Step 1: Stationary*

1 In the **Model Builder** window, under **Study 2 - Solid Mechanics (In Situ Stress Initialization)** click **Step 1: Stationary**.

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

3 Select the **Modify model configuration for study step** check box.

4 In the tree, select **Component 1 (comp1)>Darcy's Law (dl)**.

5 Click **Disable in Solvers**.

6 In the tree, select **Component 1 (comp1)>Solid Mechanics (solid)> Linear Elastic Material 1>Soil Plasticity 1** and **Component 1 (comp1)> Solid Mechanics (solid)>Linear Elastic Material 1>Initial Stress and Strain 1**.

7 Click **Disable**.

8 Click to expand the **Values of Dependent Variables** section. Find the **Values of variables not solved for** subsection. From the **Settings** list, choose **User controlled**.

9 From the **Method** list, choose **Solution**.

10 From the **Study** list, choose **Study 1-Darcy's Law, Stationary**.

## 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>Stationary**.

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

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

## STUDY 3 - SOLID MECHANICS (FACTOR OF SAFETY)

Disable the default plots for this study.

1 In the **Model Builder** window, click **Study 3**.

2 In the **Settings** window for **Study**, type `Study 3 - Solid Mechanics (Factor of Safety)` in the **Label** text field.

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

### *Step 1: Stationary*

Disable the **Darcy's Law** physics in this study.

1 In the **Model Builder** window, under **Study 3 - Solid Mechanics (Factor of Safety)** click **Step 1: Stationary**.

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

3 In the table, clear the **Solve for** check box for **Darcy's Law (dl)**.

   The pore pressure variable is not solved for in this study step; instead, its value is taken from the first solution. Create an auxiliary sweep over the parameter `FOS`.

4 Locate the **Values of Dependent Variables** section. Find the **Values of variables not solved for** subsection. From the **Settings** list, choose **User controlled**.

5 From the **Method** list, choose **Solution**.

6 From the **Study** list, choose **Study 1-Darcy's Law, Stationary**.

7 Click to expand the **Study Extensions** section. Select the **Auxiliary sweep** check box.

8 Click **Add**.

9 In the table, enter the following settings:

| Parameter name | Parameter value list | Parameter unit |
|---|---|---|
| FOS (Factor of Safety) | `range(1,0.01,1.92)` | |

10 In the **Home** toolbar, click **Compute**.

## RESULTS

*Slip Surface*

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

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

3 Locate the **Data** section. From the **Dataset** list, choose **Study 3 - Solid Mechanics (Factor of Safety)/Solution 3 (sol3)**.

4 Click to expand the **Title** section. From the **Title type** list, choose **Label**.

To show the slip surface, customize the settings for the Contour plot.

*Contour 1*

1 Right-click **Slip Surface** 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 `0 0.1 Inf`.

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

6 Click **Change Color Table**.

7 In the **Color Table** dialog box, select **Linear>GrayPrint** in the tree.

8 Click **OK**.

9 In the **Settings** window for **Contour**, locate the **Coloring and Style** section.

10 From the **Color table transformation** list, choose **Reverse**.

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

*Arrow Surface 1*

1 In the **Model Builder** window, right-click **Slip Surface** and choose **Arrow Surface**.

2 In the **Settings** window for **Arrow Surface**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1 (comp1)>Solid Mechanics>Displacement>u,v - Displacement field**.

3 In the **Slip Surface** toolbar, click **Plot**.

*Equivalent Plastic Strain*

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

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

**3** Locate the **Data** section. From the **Dataset** list, choose **Study 3 - Solid Mechanics (Factor of Safety)/Solution 3 (sol3)**.

*Surface 1*

**1** Right-click **Equivalent Plastic Strain** 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)>Solid Mechanics>Strain (Gauss points)>solid.epeGp - Equivalent plastic strain, Gauss point evaluation**.

**3** In the **Equivalent Plastic Strain** toolbar, click **Plot**.

Set up a 1D plot in order to visualize the maximum displacement in the domain.

*Factor of Safety*

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

**2** In the **Settings** window for **1D Plot Group**, type `Factor of Safety` in the **Label** text field.

**3** Locate the **Data** section. From the **Dataset** list, choose **Study 3 - Solid Mechanics (Factor of Safety)/Solution 3 (sol3)**.

*Global 1*

**1** Right-click **Factor of Safety** and choose **Global**.

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

**3** In the table, enter the following settings:

| Expression | Unit | Description |
|---|---|---|
| maxop1(solid.disp) | mm | Maximum displacement |

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

**5** Click **Replace Expression** in the upper-right corner of the **x-Axis Data** section. From the menu, choose **Global definitions>Parameters>FOS - Factor of Safety**.

**6** Click to expand the **Coloring and Style** section. Find the **Line markers** subsection. From the **Marker** list, choose **Circle**.

*Factor of Safety*

**1** In the **Model Builder** window, click **Factor of Safety**.

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

**3** From the **Position** list, choose **Upper left**.

**4** In the **Factor of Safety** toolbar, click **Plot**.

Create an Extrusion dataset to use for visualizing the displacement field in 3D.

*Extrusion 2D 1*

1 In the **Results** toolbar, click **More Datasets** and choose **Extrusion 2D**.

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

3 From the **Dataset** list, choose **Study 3 - Solid Mechanics (Factor of Safety)/ Solution 3 (sol3)**.

4 Locate the **Extrusion** section. In the **z maximum** text field, type `L1+L2+L3`.

5 In the **z variable** text field, type `Z`.

6 Find the **Embedding** subsection. From the **Map plane to** list, choose **xz-plane**.

*Displacement*

1 In the **Results** toolbar, click **3D Plot Group**.

2 In the **Settings** window for **3D Plot Group**, type `Displacement` in the **Label** text field.

*Surface 1*

1 Right-click **Displacement** 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)>Solid Mechanics> Displacement>solid.disp - Displacement magnitude - m**.

3 Locate the **Expression** section. From the **Unit** list, choose **mm**.

4 In the **Displacement** toolbar, click **Plot**.

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