

Strength Reduction Method for Slope Stability

Introduction

The strength reduction method in combination with finite element analysis is a tool to find the factor of safety (FOS) in geomechanics, particularly for the stability of slopes and embankments. In the strength reduction method, the characteristic material properties are gradually reduced until failure occurs. Although the factor of safety has different definitions in other context, in the context of geotechnical problems it is defined with respect to the strength parameters of the soil, as discussed in [Ref. 1](#).

The strength reduction method is applicable to linear failure criteria, like the Mohr–Coulomb criterion. When the Mohr–Coulomb criterion is used with the strength reduction method, the cohesion, the angle of internal friction, and the dilatation angle are simultaneously reduced until mechanical equilibrium is lost. Decreasing the material parameters results in reduction of the shear strength of the soil, which eventually becomes unstable. This phenomena produces a collapse of the slope for a certain combination of loads, material parameters, and boundary conditions. The ratio between the initial cohesion and the cohesion at failure gives the factor of safety (FOS). More details of the method are given in [Ref. 1](#) and [Ref. 2](#).

The geometry, boundary conditions, loading conditions, and material parameters in this example are the same as discussed in [Ref. 1](#). Quartic order shape functions are used to match the discretization used in [Ref. 1](#).

Model Definition

[Figure 1](#) shows the cross section of the soil embankment. The lengths L_1 and L_2 are 85 m, and 20 m, respectively, and the heights of the embankment H_1 and H_2 are 20 m and 10 m, respectively. The slope angle α varies from 15° to 45° .

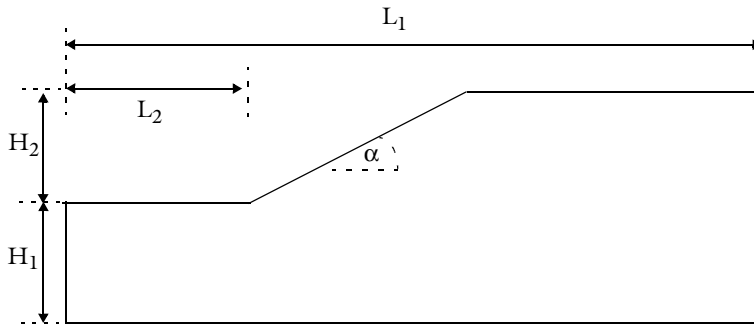


Figure 1: Dimensions of the slope.

The material properties for both associative and nonassociative plasticity are summarized in [Table 1](#), and taken from [Ref. 1](#)

TABLE 1: MATERIAL PROPERTIES.

PROPERTY	MATERIAL SET 1	MATERIAL SET 2
E	20 MPa	20 MPa
ν	0.3	0.3
c	20 kPa	20 kPa
ϕ	25°	25°
ψ	0°	25°
ρ	1940 kg/m ³	1940 kg/m ³

A plane strain approximation is used to model the soil embankment in 2D. The effects of gravity are also included. The material properties for the Mohr–Coulomb model are parameterized with respect to a factor of safety parameter, FOS. A parametric study increases the FOS parameter, thereby reducing the strength of the soil with every parameter step. The actual factor of safety is the value of the FOS parameter at which the model no longer converges, which is an indication of the collapse of the slope.

The Mohr–Coulomb yield function F and plastic potential Q are

$$F = m\sqrt{J_2} + \frac{\sin\Phi}{3}I_1 - C\cos\Phi \quad (1)$$

$$Q = m_q\sqrt{J_2} + \frac{\sin\Psi}{3}I_1 - C\cos\Psi \quad (2)$$

where I_1 is the first stress invariant and J_2 is the second deviatoric stress invariant. The parameterized cohesion C , parameterized angle of internal friction Φ , and parameterized dilatation angle Ψ are given as

$$C = \frac{c}{\text{FOS}}, \Phi = \text{atan}\left(\frac{\tan\phi}{\text{FOS}}\right), \Psi = \text{atan}\left(\frac{\tan\psi}{\text{FOS}}\right) \quad (3)$$

where c is the cohesion, ϕ is the angle of internal friction, and ψ is the dilatation angle. Note that c , ϕ , and ψ are initial, unreduced material parameters. For the associative flow rule, $\phi = \psi$.

For the nonassociative flow rule, ψ is kept constant as long as it is smaller than Φ , see [Ref. 2](#) for details. However, in this example, ψ is zero when using the nonassociative flow rule, so no special treatment is needed, and [Equation 3](#) is applicable to the associative as well as the nonassociative flow rule.

For the nonassociative flow rule, the strength reduction method might trigger numerical instabilities, which in turn can result in a nonunique failure surface and FOS. To avoid potential instabilities and convergence issues, the *Davis procedure B* approach, as suggested in [Ref. 1](#) and [Ref. 2](#), is used. For this approach, the associative flow rule is applied with reduced values of the cohesion and the angle of internal friction to capture the effects of the nonassociative flow rule. The reduced cohesion c' and the reduced angle of internal friction ϕ' are given by

$$c' = \beta c, \phi' = \text{atan}(\beta \tan \phi) \quad (4)$$

where the *reduction factor* β is

$$\beta = \frac{\cos\left(\text{atan}\left(\frac{\tan \phi}{\text{FOS}}\right)\right) \cos\left(\text{atan}\left(\frac{\tan \psi}{\text{FOS}}\right)\right)}{1 - \sin\left(\text{atan}\left(\frac{\tan \phi}{\text{FOS}}\right)\right) \sin\left(\text{atan}\left(\frac{\tan \psi}{\text{FOS}}\right)\right)}$$

Hence, [Equation 3](#) is rewritten in terms of the reduced cohesion c' and the reduced angle of internal friction ϕ' for the associative and nonassociative flow rules, as the reduction factor β is unity for the associative flow rule.

Results and Discussion

The factor of safety (FOS) for different slope angles is shown in [Figure 2](#). The FOS decreases as the slope angle increases, which is expected. The FOS for the same slope inclination with the nonassociative flow rule (material set 1) is always smaller than for the associative flow rule (material set 2), but the influence of the nonassociative flow rule is marginal for the chosen material parameters. The results presented in [Figure 2](#) are in good agreement with the results presented in [Figure 4](#) of [Ref. 1](#).

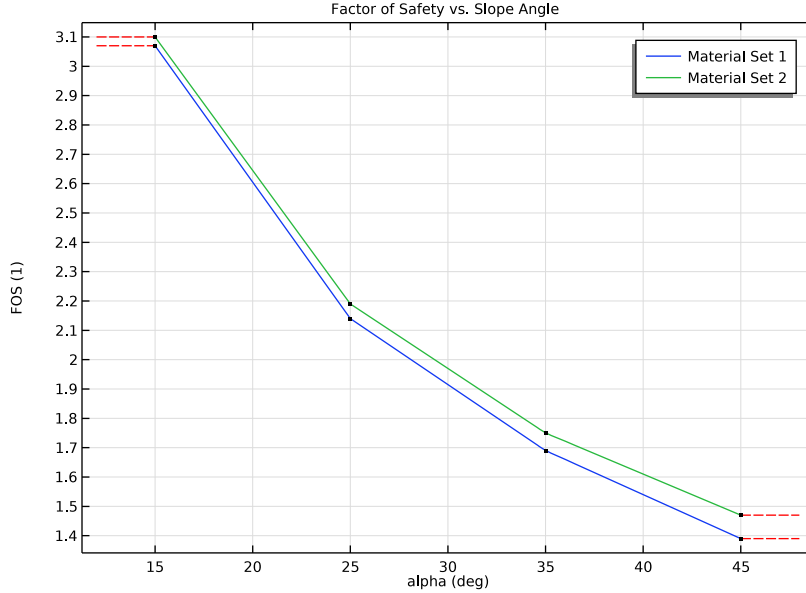


Figure 2: Factor of safety versus slope angle.

The equivalent plastic strain for different slope angles just before collapse is shown in [Figure 3](#) and [Figure 4](#) for the two material sets. The localization of plastic strains in the figures give an indication of the failure surface for different slope angles. It is evident that for lower slope angles, multiple failure surfaces develop in the embankment.

A 3D visualization of the displacement for a slope angle equal to 45° is shown in [Figure 5](#) and [Figure 6](#) for both material sets. The results qualitatively are in good agreement with the results presented in Figure 2 of [Ref. 1](#). The figures also clearly show how the parts of the embankment outside the slip surface starts sliding once the material becomes unstable.

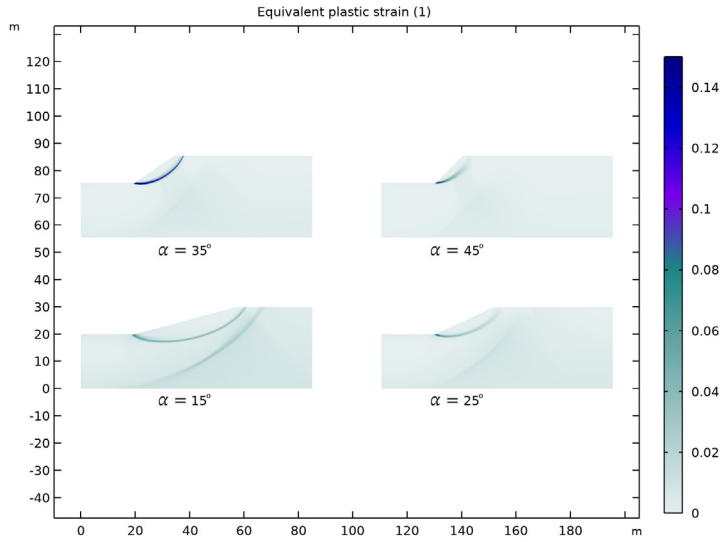


Figure 3: Equivalent plastic strain just before collapse for material set 1.

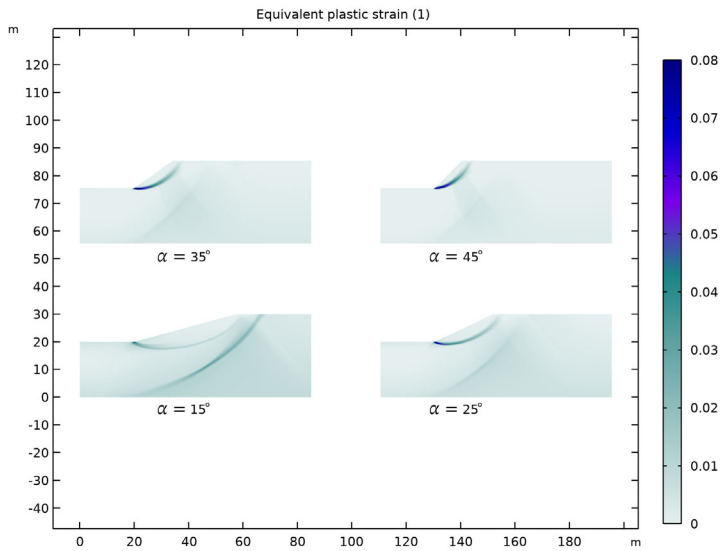


Figure 4: Equivalent plastic strain just before collapse for material set 2.

Material Parameter Set 1, $\alpha=45$ FOS(40)=1.39 Surface: Displacement magnitude (m)

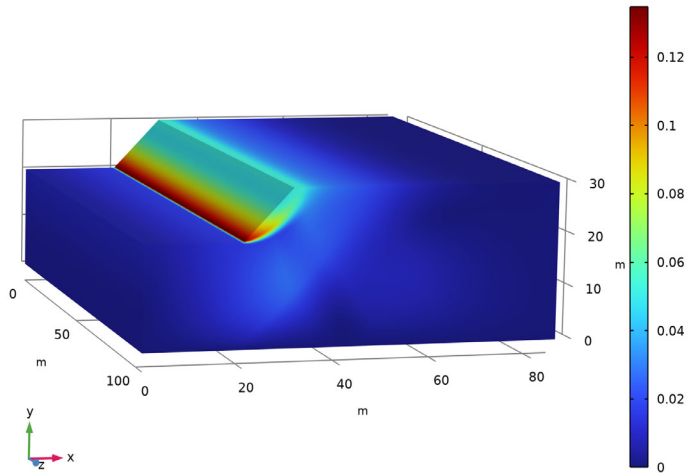


Figure 5: Displacement magnitude of the soil embankment just before slope collapse for material set 1.

Material Parameter Set 2, $\alpha=45$ FOS(48)=1.47 Surface: Displacement magnitude (m)

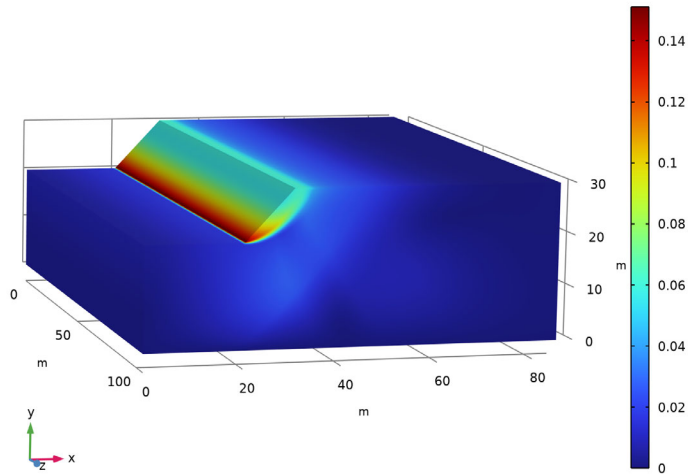


Figure 6: Displacement magnitude of the soil embankment just before slope collapse for material set 2.

Notes About the COMSOL Implementation

A **Mesh Control Domain** is added to the **Geometry** node, where a denser mesh is prescribed in the region where soil slippage is expected. The mesh control domain is removed from the geometry sequence once the mesh is generated, so this virtual domain is not visible in the physics selections.

Two stationary study steps are added. The first study step computes the in-situ stresses due to gravity. The Mohr-Coulomb criterion is added to the second study study to compute the elastoplastic failure due to the combined effect of gravity and the reduction in strength, where the initial stresses generated in the first step are incorporated in the analysis with the help of an **Initial Stress and Strain** node.

Two outer parametric sweeps are added to change the slope angle and the set of material parameters.

Because the model stores a NaN solution corresponding to the nonconverged values of the FOS, the generation of default plots can be problematic, as default plots are set to portrait the last parameter value. Also, finding the last converged FOS parameter value manually for each value of the slope angle and material parameter set can be cumbersome. To avoid these issues, use a model method to generate a 1D plot for the factor of safety versus slope angle for each material parameter set. The model method also generates a von Mises stress plot, a 3D displacement plot, as well as plastic strain plots for each slope angle for the two material sets.

The model method is written assuming that a certain dataset tag is available and that the intended plots, functions, and datasets are not already created. Although fail-safe usage conditions and error messages are added, the model method could fail if the default material, geometry, study, or result parameters or nodes change. The model method then has to be modified according to the current model setup.

References

1. H.F.Schweiger “Strength reduction technique with finite element method for slopes without stabilization measures,” *Benchmark, International Magazine for Engineers, Designers and Analysts from NAFEMS*, pp. 51–58, 2020.
2. S. Oberhollenzer, F. Tschuchnigg, and H.F.Schweiger “Finite element analysis of slope stability problems using non-associated plasticity,” *Journal of Rock Mechanics and Geotechnical Engineering*, vol. 10, pp. 1091–1101, 2018.

Application Library path: Geomechanics_Module/Verification_Examples/
strength_reduction_method

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 **Structural Mechanics>Solid Mechanics (solid)**.
- 3 Right-click and choose **Add Physics**.
- 4 Click  **Study**.
- 5 In the **Select Study** tree, select **General Studies>Stationary**.
- 6 Click  **Done**.

Parameters for the model geometry, material and solver are available in the appended text files.

GLOBAL DEFINITIONS

Geometry and Solver Parameters

- 1 In the **Model Builder** window, under **Global Definitions** click **Parameters 1**.
- 2 In the **Settings** window for **Parameters**, type Geometry and Solver Parameters in the **Label** text field.
- 3 Locate the **Parameters** section. Click  **Load from File**.
- 4 Browse to the model's Application Libraries folder and double-click the file `strength_reduction_method_parameters.txt`.

Material Parameters

- 1 In the **Home** toolbar, click  **Parameters** and choose **Add>Parameters**.
- 2 In the **Settings** window for **Parameters**, type Material Parameters in the **Label** text field.

- 3 Locate the **Parameters** section. Click  **Load from File**.
- 4 Browse to the model's Application Libraries folder and double-click the file `strength_reduction_method_material_parameters1.txt`.
- 5 In the **Home** toolbar, click  **Parameter Case**.
- 6 In the **Settings** window for **Case**, type `Material Parameter Set 1` in the **Label** text field.
- 7 In the **Home** toolbar, click  **Parameter Case**.
- 8 In the **Settings** window for **Case**, type `Material Parameter Set 2` in the **Label** text field.
- 9 Locate the **Parameters** section. Click  **Load from File**.
- 10 Browse to the model's Application Libraries folder and double-click the file `strength_reduction_method_material_parameters2.txt`.

DEFINITIONS

Define the parameterized cohesion and angle of internal friction for Mohr-Coulomb criterion.

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
beta_f	$\cos(\text{atan}(\tan(\phi)/FOS)) * \cos(\text{atan}(\tan(\psi)/FOS)) / (1 - \sin(\text{atan}(\tan(\phi)/FOS)) * \sin(\text{atan}(\tan(\psi)/FOS)))$		Reduction factor
c_r	$\text{beta_f} * c$	Pa	Reduced cohesion
phi_r	$\text{atan}(\text{beta_f} * \tan(\phi))$	rad	Reduced friction angle
c_p	c_r / FOS	Pa	Parameterized cohesion
phi_p	$\text{atan}(\tan(\phi_r) / FOS)$	rad	Parameterized friction angle

GEOMETRY I

Polygon 1 (pol1)

- 1 In the **Geometry** toolbar, click  **Polygon**.
- 2 In the **Settings** window for **Polygon**, locate the **Coordinates** section.
- 3 From the **Data source** list, choose **Vectors**.
- 4 In the **x** text field, type $0, L1, L1, L2+H2/\tan(\alpha), L2, 0$.
- 5 In the **y** text field, type $0, 0, H1+H2, H1+H2, H1, H1$.
- 6 Click  **Build Selected**.

Polygon 2 (pol2)

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

Split the geometry with a polygon to add a **Mesh Control Domain**.

- 1 In the **Settings** window for **Polygon**, locate the **Object Type** section.
- 2 From the **Type** list, choose **Open curve**.
- 3 Locate the **Coordinates** section. From the **Data source** list, choose **Vectors**.
- 4 In the **x** text field, type $0.8*L2, 0.8*L2, 1.3*L2+H2/\tan(\alpha), 1.3*L2+H2/\tan(\alpha)$.
- 5 In the **y** text field, type $H1, H1/2, H1/2, H1+H2$.
- 6 Click  **Build Selected**.

Mesh Control Domains 1 (mcd1)

- 1 In the **Geometry** toolbar, click  **Virtual Operations** and choose **Mesh Control Domains**.
- 2 On the object **fin**, select Domain 2 only.
- 3 In the **Settings** window for **Mesh Control Domains**, click  **Build Selected**.

Change the discretization to **Quartic Lagrange**.

SOLID MECHANICS (SOLID)

- 1 In the **Model Builder** window, under **Component 1 (comp1)** click **Solid Mechanics (solid)**.
- 2 In the **Settings** window for **Solid Mechanics**, click to expand the **Discretization** section.
- 3 From the **Displacement field** list, choose **Quartic Lagrange**.

Linear Elastic Material 1

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

Soil Plasticity I

- 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 **Yield criterion** list, choose **Mohr-Coulomb**.
- 4 From the **Plastic potential** list, choose **Associated**.

Linear Elastic Material I

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

Initial Stress and Strain I

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

Add two study steps in order to account for the in situ stresses due to gravity. Add the in situ stresses computed in the first study step as initial stresses for the second study step. You can access these stresses 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 S_0 table, enter the following settings:

<code>withsol('sol2', solid.sx)</code>	<code>withsol('sol2', solid.sxy)</code>	<code>withsol('sol2', solid.sxz)</code>
<code>withsol('sol2',solid.sxy)</code>	<code>withsol('sol2', solid.sy)</code>	<code>withsol('sol2', solid.syz)</code>
<code>withsol('sol2',solid.sxz)</code>	<code>withsol('sol2',solid.syz)</code>	<code>withsol('sol2', solid.sz)</code>

Gravity I

- 1 In 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**.

Roller I

- 1 In the **Physics** toolbar, click  **Boundaries** and choose **Roller**.
- 2 Select Boundaries 1 and 6 only.

Fixed Constraint I

- 1 In the **Physics** toolbar, click  **Boundaries** and choose **Fixed Constraint**.
- 2 Select Boundary 2 only.

MATERIALS

Soil Material

- 1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Materials** and choose **Blank Material**.
- 2 In the **Settings** window for **Material**, type Soil Material in the **Label** text field.
- 3 Locate the **Material Contents** section. In the table, enter the following settings:

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	I	Young's modulus and Poisson's ratio
Density	rho	rho_soil	kg/m ³	Basic
Cohesion	cohesion	c_p	Pa	Mohr-Coulomb
Angle of internal friction	internalphi	phi_p	rad	Mohr-Coulomb

MESH 1

Free Triangular 1

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

Size

Use an extremely fine mesh in the mesh control domain.

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

Size 1

- 1 In the **Model Builder** window, right-click **Free Triangular 1** and choose **Size**.
- 2 In the **Settings** window for **Size**, locate the **Geometric Entity Selection** section.

- 3 From the **Geometric entity level** list, choose **Domain**.
- 4 Select Domain 2 only.
- 5 Locate the **Element Size** section. From the **Predefined** list, choose **Extremely fine**.
- 6 In the **Model Builder** window, right-click **Mesh 1** and choose **Build All**.

The material strengths are parameterized with the help of the FOS parameter. Add an auxiliary sweep for FOS in the second study step. For certain values of FOS the solution will not converge and the default plots, which are set to the last FOS value, will give an error. Disable the default plots for this study in order to avoid the error message.

Add two **Parametric Sweep** nodes to change the slope angle and material parameters.

STUDY 1

- 1 In the **Model Builder** window, click **Study 1**.
- 2 In the **Settings** window for **Study**, locate the **Study Settings** section.
- 3 Clear the **Generate default plots** check box.

Step 1: Stationary

- 1 In the **Model Builder** window, under **Study 1** 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)>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**.
- 5 Right-click and choose **Disable**.

Stationary 2

- 1 In the **Study** toolbar, click  **Study Steps** and choose **Stationary>Stationary**.
- 2 In the **Settings** window for **Stationary**, click to expand the **Values of Dependent Variables** section.
- 3 Find the **Initial values of variables solved for** subsection. From the **Settings** list, choose **User controlled**.
- 4 From the **Method** list, choose **Solution**.
- 5 From the **Study** list, choose **Study 1, Stationary**.
- 6 Click to expand the **Study Extensions** section. Select the **Auxiliary sweep** check box.
- 7 Click  **Add**.

8 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
FOS (Factor of safety)	range(stepi,steps,stepf)	

Parametric Sweep

- 1 In the **Study** toolbar, click  **Parametric Sweep**.
- 2 In the **Settings** window for **Parametric Sweep**, locate the **Study Settings** section.
- 3 Click  **Add**.
- 4 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
alpha (Slope angle)	range(stepi_a,steps_a,stepf_a)	deg

Parametric Sweep 2

- 1 In the **Study** toolbar, click  **Parametric Sweep**.
- 2 In the **Settings** window for **Parametric Sweep**, locate the **Study Settings** section.
- 3 From the **Sweep type** list, choose **Parameter switch**.
- 4 Click  **Add**.
- 5 In the table, enter the following settings:

Switch	Cases	Case numbers
Material Parameters	All	range(1,1,2)

Solution 1 (sol1)

- 1 In the **Study** toolbar, click  **Show Default Solver**.
- 2 In the **Model Builder** window, expand the **Solution 1 (sol1)** node.
- 3 In the **Model Builder** window, expand the **Study 1>Solver Configurations>Solution 1 (sol1)>Stationary Solver 2** node, then click **Parametric 1**.
- 4 In the **Settings** window for **Parametric**, locate the **General** section.
- 5 From the **On error** list, choose **Store empty solution**.
- 6 Click to expand the **Continuation** section. Select the **Tuning of step size** check box.
- 7 In the **Initial step size** text field, type steps/10.
- 8 In the **Minimum step size** text field, type steps/10.
- 9 In the **Maximum step size** text field, type steps.

10 From the **Predictor** list, choose **Constant**.

Use a double dogleg solver in order to improve the convergence.

11 In the **Model Builder** window, under **Study 1>Solver Configurations>Solution 1 (sol1)>Stationary Solver 2** click **Fully Coupled 1**.

12 In the **Settings** window for **Fully Coupled**, click to expand the **Method and Termination** section.

13 From the **Nonlinear method** list, choose **Double dogleg**.

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

Create model methods to generate the required plots. Note that the method editor is only available in the Windows® version of the COMSOL Desktop.

ROOT

1 From the **File** menu, choose **Preferences**.

2 In the **Preferences** dialog box, click **OK**.

APPLICATION BUILDER

In the **Home** toolbar, click  **Application Builder**.

METHODS

plotUtil

1 In the **Home** toolbar, click  **More Libraries** and choose **Utility Class**.

2 Right-click **util1** and choose **Rename**.

3 In the **Rename Utility Class** dialog box, type **plotUtil** in the **New name** text field.

4 Click **OK**.

5 Right-click **plotUtil** and choose **Edit**.

6 Copy the following code into the **plotUtil** window:

```
//This utility creates requested plots

//Method to create plot group
public static ResultFeature createPlot(String dset, String plottype, String tag,
String label, String title, String view, boolean showlegends,boolean plotArray,
double[][] index, int k) {
    int lastloopn = (int) Math.round(model.param().evaluate("nstep_a")-1);
    ResultFeature plot = model.result().create(tag, plottype);
    model.result(tag).label(label+Integer.toString(k+1));
    with(model.result(tag));
    set("data", dset);
    try {
```

```

        setIndex("looplevel", k+1, 2);
        setIndex("looplevel", index[k][lastloopn], 0);
    }
    catch (Exception e) {
        error("The required looplevel is not available in the dataset, check the model
or model method.");
    }
    if (!title.equals("default")) {
        set("titletype", "manual");
        set("title", title);
        set("paramindicator", "");
    }
    if (!showlegends)
        set("showlegends", false);
    if (!view.equals("default"))
        set("view", view);
    if (plotArray) {
        set("plotarrayenable", true);
        set("arrayshape", "square");
    }
    set("edges", false);
    endwith();
    return plot;
}
//Method to create surface plot
public static ResultFeature createSubPlot(ResultFeature pg, String dset, String
plottype, String tag, String expr, boolean isPlasticityPlot, boolean
isStressPlot, double[][] index, int k, int i) {
    ResultFeature pgsub = pg.create(tag, plottype);
    with(pgsub);
    set("expr", expr);
    if (isPlasticityPlot)
        set("colortable", "AuroraAustralisDark");
    if (!dset.endsWith("fromParent")) {
        set("data", dset);
        try {
            setIndex("looplevel", k+1, 2);
            setIndex("looplevel", i+1, 1);
            setIndex("looplevel", index[k][i], 0);
        }
        catch (Exception e) {
            error("The required looplevel is not available in the dataset, check the model
or model method.");
        }
    }
    endwith();
    return pgsub;
}

```

NEW METHOD

- 1 In the **Application Builder** window, right-click **Methods** and choose **New Method**.
- 2 In the **New Method** dialog box, type generatePlots in the **Name** text field.
- 3 Click **OK**.

generatePlots

1 In the **Application Builder** window, under **Methods** click **generatePlots**.

2 Copy the following code into the **generatePlots** window:

```
String dset = "dset3"; //Required parametric dataset tag

if (!contains(model.result().dataset().tags(), dset)) //Throw an exception if
accessing dataset fails
    error("The solution with the required dataset tag is not available, check the
model or model method.");

int nstep = (int) Math.round(model.param().evaluate("nstep"));
int nstepa = (int) Math.round(model.param().evaluate("nstep_a"));
int nmat = (int) Math.round(model.param().evaluate("nmat"));
int stepsa = (int) Math.round(model.param().evaluate("steps_a")*180/
model.param().evaluate("pi"));
double[][] FOS = new double[nmat][nstepa];
double[][] alpha = new double[nmat][nstepa];
double[][] index = new double[nmat][nstepa];

boolean isEvaluationGrpExists =
contains(model.result().evaluationGroup().tags(), "eg1");
boolean isExtrusionDsetExists = contains(model.result().dataset().tags(),
"extr1");
boolean isViewExists = contains(model.view().tags(), "view3");
boolean isInterpolationFunExists = false;
boolean isNodeGrpExists = false;
boolean isPlotGrpExists = false;
for (int k = 0; k < nmat; k++) {
    String int1 = "int"+Integer.toString(k+1);
    if (contains(model.func().tags(), int1)) {
        isInterpolationFunExists = true;
        break;
    }
}
for (int k = 0; k < nmat; k++) {
    String grp = "grp"+Integer.toString(k+1);
    if (contains(model.nodeGroup().tags(), grp)) {
        isNodeGrpExists = true;
        break;
    }
}

for (int k = 0; k < 3*nmat+1; k++) {
    String pg = "pg"+Integer.toString(k+1);
    if (contains(model.result().tags(), pg)) {
        isPlotGrpExists = true;
        break;
    }
}
if ((isEvaluationGrpExists || isExtrusionDsetExists || isInterpolationFunExists
|| isPlotGrpExists || isNodeGrpExists || isViewExists)) {
    /** Opens a confirm dialog to check whether the user would like to
```

```

* delete the existing model nodes and regenerate again.*/

String answer = confirm("Certain nodes already exist, do you want to delete
them and regenerate again?", "Delete and regenerate", "Yes", "No");

if (answer.equals("Yes")) {
if (isEvaluationGrpExists)
model.result().evaluationGroup().remove("eg1");
if (isExtrusionDsetExists)
model.result().dataset().remove("extr1");
if (isInterpolationFunExists) {
for (int k = 0; k < nmat; k++) {
String int1 = "int"+Integer.toString(k+1);
if (contains(model.func().tags(), int1)) {
model.func().remove(int1);
}
}
}
if (isPlotGrpExists) {
for (int k = 0; k < 3*nmat+1; k++) {
String pg = "pg"+Integer.toString(k+1);
if (contains(model.result().tags(), pg)) {
model.result().remove(pg);
}
}
}
if (isNodeGrpExists) {
for (int k = 0; k < nmat; k++) {
String grp = "grp"+Integer.toString(k+1);
if (contains(model.nodeGroup().tags(), grp)) {
model.nodeGroup().remove(grp);
}
}
}
if (isViewExists)
model.view().remove("view3");
}
else {
if (isEvaluationGrpExists)
error("The evaluation group already exists, check the model or model method");
if (isExtrusionDsetExists)
error("The extrusion dataset already exists, check the model or model method.");
if (isInterpolationFunExists)
error("The interpolation function already exists, check the model or model
method.");
if (isPlotGrpExists)
error("The plot already exists, check the model or model method.");
if (isNodeGrpExists)
error("The node group already exists, check the model or model method.");
if (isViewExists)
error("The view already exists, check the model or model method.");
}
}

model.result().evaluationGroup().create("eg1", "EvaluationGroup");

```

```

with(model.result().evaluationGroup("eg1"));
set("data", dset);
setIndex("looplevelinput", "manual", 2);
setIndex("looplevelinput", "manual", 1);
setIndex("looplevelinput", "manual", 0);
set("includeparameters", false);
endwith();
model.result().evaluationGroup("eg1").create("pev1", "EvalPoint");
model.result().evaluationGroup("eg1").feature("pev1").selection().set(2);
with(model.result().evaluationGroup("eg1").feature("pev1"));
setIndex("expr", "alpha", 0);
setIndex("unit", "deg", 0);
setIndex("expr", "FOS", 1);
setIndex("unit", 1, 1);
setIndex("expr", "u", 2);
endwith();

for (int k = 0; k < nmat; k++) {
String int1 = "int"+Integer.toString(k+1);
model.func().create(int1, "Interpolation");

model.func(int1).label("FOS_MatSet"+Integer.toString(k+1));
for (int j = 0; j < nstepa; j++) {
for (int i = 0; i < nstep; i++) {
with(model.result().evaluationGroup("eg1"));
setIndex("looplevel", new int[]{k+1}, 2);
setIndex("looplevel", new int[]{j+1}, 1);
setIndex("looplevel", new int[]{i+1}, 0);
model.result().evaluationGroup("eg1").run();
double[][] FOS1 = model.result().evaluationGroup("eg1").getReal();
double u = FOS1[0][2];
if (Double.isNaN(u))
break;
FOS[k][j] = FOS1[0][1];
alpha[k][j] = FOS1[0][0];
index[k][j] = i+1;
endwith();
}
model.func(int1).setIndex("table", FOS[k][j], j, 1);
model.func(int1).setIndex("table", alpha[k][j], j, 0);
}
model.func(int1).set("argunit", "deg");
model.func(int1).set("fununit", "1");

if (k == 0) {
model.func(int1).createPlot("pg1");
model.result("pg1").label("Factor of Safety vs. Slope Angle");
model.result("pg1").set("titletype", "label");
model.result("pg1").set("xlabelactive", true);
model.result("pg1").set("xlabel", "alpha (deg)");
model.result("pg1").set("ylabelactive", true);
model.result("pg1").set("ylabel", "FOS (1)");
model.result("pg1").feature("plot1").set("legend", true);
model.result("pg1").feature("plot1").set("legendmethod", "manual");
model.result("pg1").feature("plot1").setIndex("legends", "Material Set "+

```

```

Integer.toString(k+1), 0);
}
else {
String plot1 = "plot"+toString(k);
String plot2 = "plot"+toString(k+1);
String dataset = model.result("pg1").getString("data");
model.result().dataset(dataset).set("function", "all");
model.result("pg1").feature().duplicate(plot2, plot1);
model.result("pg1").feature(plot2).set("expr", "int"+Integer.toString(k+1)+
"(t[1/m][deg])");
model.result("pg1").feature(plot2).setIndex("legends", "Material Set "+
Integer.toString(k+1), 0);
}
}
model.result().evaluationGroup().remove("eg1");
model.result("pg1").run();

//Create an extrusion dataset for 3D plots
model.result().dataset().create("extr1", "Extrude2D");
with(model.result().dataset("extr1"));
set("data", dset);
set("zvar", "Z");
set("zmax", "L1+L2");
endwith();

//Create a camera view
model.view().create("view3", 3);
model.view("view3").camera().set("position", new int[]{-164, 120, 740});
model.view("view3").camera().set("up", new String[]{"0.02", "0.97", "-0.14"});
model.view("view3").camera().set("target", new int[]{42, 14, 52});
model.view("view3").camera().setIndex("viewoffset", -0.1, 0);
model.view("view3").camera().set("viewoffset", new double[]{-0.1, -0.016});
model.view("view3").camera().set("zoomanglefull", 13);
model.view("view3").set("locked", true);

for (int k = 0; k < nmat; k++) {
int n = 3*k+2;
String pg2 = "pg"+toString(n);
String pg3 = "pg"+toString(n+1);
String pg4 = "pg"+toString(n+2);

//von Mises Stress Plot
ResultFeature pgs = plotUtil.createPlot(dset, "PlotGroup2D", pg2, "von Mises
Stress ", "default", "default", true, index, k);
plotUtil.createSubPlot(pgs, "fromParent", "Surface", "surf1", "solid.mises",
false, true, false, index, k, 0);
pgs.run();

//Equivalent Plastic Strain Plot
ResultFeature pge = plotUtil.createPlot(dset, "PlotGroup2D", pg3, "Equivalent
Plastic Strain ", "Equivalent plastic strain (1)", "default", true, true, index,
k);
ResultFeature tlan = pge.create("tlan1", "TableAnnotation");
tlan.set("source", "localtable");

```

```

tlan.set("latexmarkup", true);
tlan.set("showpoint", false);
double jj = 0.0;
for (int i = 0; i < nstepa; i++) {
    int D = 15+i*stepsa;
    double aposX = 25;
    double aposY = (int) jj*55;
    if (i%2 == 1) { //Odd index
        aposX = 125;
    }
    String lable = "\\[[][\\alpha \\;=\\;"+D+"^\\circ\\]";
    String surf = "surf"+toString(i+1);
    ResultFeature pges = plotUtil.createSubPlot(pge, dset, "Surface", surf,
"solid.epe", true, false, index, k, i);
    if (i > 0) {
        String surfprev = "surf"+Integer.toString(i);
        pges.set("inheritplot", surfprev);
    }
    tlan.setIndex("localtablematrix", aposX, i, 0);
    tlan.setIndex("localtablematrix", aposY, i, 1);
    tlan.setIndex("localtablematrix", lable, i, 2);
    jj += 0.5;
}
pge.run();

//3D Displacement Plot
ResultFeature pgd = plotUtil.createPlot("extr1", "PlotGroup3D", pg4,
"Displacement ", "default", "view3", true,false, index, k);
plotUtil.createSubPlot(pgd, "fromParent", "Surface", "surf1", "solid.disp",
false, false, index, k, 0);
pgd.run();

//Node group
String grp = "grp"+Integer.toString(k+1);
model.nodeGroup().create(grp, "Results");
model.nodeGroup(grp).set("type", "plotgroup");
model.nodeGroup(grp).add("plotgroup", pg2);
model.nodeGroup(grp).add("plotgroup", pg3);
model.nodeGroup(grp).add("plotgroup", pg4);
model.nodeGroup(grp).label("Results: Material Set "+Integer.toString(k+1));
model.nodeGroup(grp).placeAfter("plotgroup", "pg1");

//Rearrange the groups
if (k == nmat-1) {
    if (contains(model.nodeGroup().tags(), "grp2"))
        model.nodeGroup().move("grp2", 3);
}
}

```

METHODS

In the **Home** toolbar, click  **Model Builder** to switch to the main desktop.

Generate plots using the model methods.

MODEL BUILDER

- 1 In the **Home** toolbar, click  **Model Builder**.
- 2 Click  **Method Call** and choose **generatePlots**.

GLOBAL DEFINITIONS

Generate Plots

- 1 In the **Model Builder** window, under **Global Definitions** click **GeneratePlots 1**.
- 2 In the **Settings** window for **Method Call**, type **Generate Plots** in the **Label** text field.
- 3 In the **Tag** text field, type **GeneratePlots**.
- 4 Click  **Run**.

RESULTS

Go to the generated plots in order to visualize the failure in the embankment with the two different material parameter sets.

Factor of Safety vs. Slope Angle

- 1 In the **Model Builder** window, expand the **Results** node, then click **Factor of Safety vs. Slope Angle**.
- 2 In the **Factor of Safety vs. Slope Angle** toolbar, click  **Plot**.

Results: Material Set 1

In the **Model Builder** window, expand the **Results>Results: Material Set 1** node.

Surface 1

- 1 In the **Model Builder** window, expand the **Results>Results: Material Set 1>Equivalent Plastic Strain 1** node, then click **Surface 1**.
- 2 In the **Settings** window for **Surface**, click to expand the **Range** section.
- 3 Select the **Manual color range** check box.
- 4 In the **Maximum** text field, type **0.15**.
- 5 In the **Equivalent Plastic Strain 1** toolbar, click  **Plot**.

Results: Material Set 2

In the **Model Builder** window, expand the **Results>Results: Material Set 2** node.

Surface 1

- 1** In the **Model Builder** window, expand the **Results>Results: Material Set 2>Equivalent Plastic Strain 2** node, then click **Surface 1**.
- 2** In the **Settings** window for **Surface**, locate the **Range** section.
- 3** Select the **Manual color range** check box.
- 4** In the **Maximum** text field, type 0.08.
- 5** In the **Equivalent Plastic Strain 2** toolbar, click  **Plot**.