Created in COMSOL Multiphysics 6.0

Flexible and Smooth Strip Footing on a Stratum of Clay

Model Definition

A typical verification example for geotechnical problems is a shallow stratum layer of clay, see Figure 1. In the example, a vertical load is applied on the clay stratum, and the static response as well as the collapse load are of interest. This example is adapted from Ref. 2.

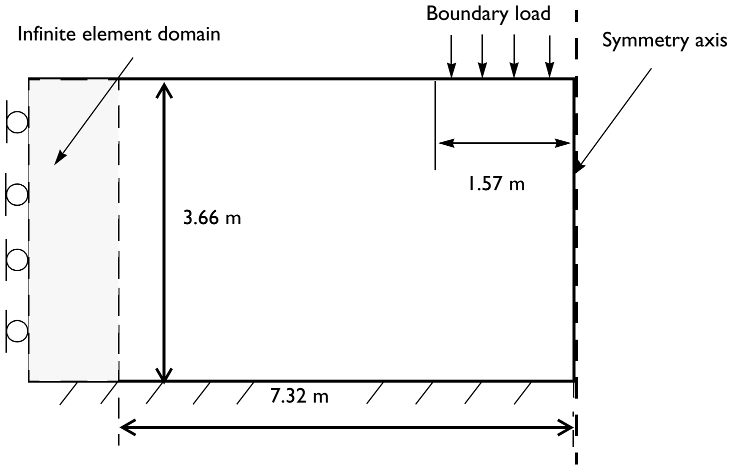

Figure 1: Dimensions, boundary conditions, and pressure load for the stratum of clay.

ANALYSIS TYPE

Yield Surface

Assume plane strain conditions, and model the clay with soil plasticity and the Drucker-Prager criterion.

The *yield surface*, F, for the Drucker-Prager criterion is given by

$$F = \sqrt{J_2} + \alpha I_1 - k = 0$$

where I_1 is the first stress invariant and J_2 is the second deviatoric stress invariant.

The first stress invariant is defined using the trace of Cauchy stress tensor:

$$I_1 = \mathrm{trace}(\sigma)$$

The second stress invariant is defined by

$$I_2 = \frac{1}{2}(I_1^2 - \mathrm{trace}(\sigma^2))$$

The second deviatoric stress invariant can be expressed using the first and the second stress invariants:

$$J_2 = \frac{1}{3}I_1^2 - I_2$$

If two-dimensional plane strain conditions prevail, the Drucker-Prager criterion matches the Mohr-Coulomb criterion. For this case the material parameters α and k are given by the Cohesion c and the Angle of internal friction ϕ (Ref. 1)

$$\alpha = \frac{\tan\phi}{\sqrt{(9+12\tan^2\phi)}}$$

$$k = \frac{3c}{\sqrt{(9+12\tan^2\phi)}}$$

Drucker-Prager criterion is the default choice for the Soil Plasticity feature, and the check box **Match to Mohr-Coulomb criterion** applies the aforementioned matching of the material parameters.

Under Soil Plasticity, it is also possible to use Mohr-Coulomb criterion

$$F = \frac{1}{2}(\sigma_{max} - \sigma_{min}) + \frac{1}{2}(\sigma_{max} + \sigma_{min})\sin\phi - c\cos\phi = 0\,,$$

where σ_{max} and σ_{min} are the biggest and smallest principal stresses. The Mohr-Coulomb criterion defines an irregular hexagon pyramid in the principal stress space. Since this yield function gives rise to singularities in the derivatives of the yield function, the use of a non-associated flow rule with a Drucker-Prager plastic potential is chosen. This is done in the Plastic potential list, with the option **Drucker-Prager matched at compressive meridian**.

Flow Rule

The flow rule defines the relation between the plastic strain increment in a given direction and the current level of stress in the same direction. The relation reads

$$\dot{\varepsilon}_{ij} = \lambda\frac{\partial Q}{\partial\sigma_{ij}}$$

where λ is the plastic multiplier and Q is the plastic potential. If the yield surface, F, and the plastic potential, Q, are identical, that is, if $F = Q$, then it is called an associated flow rule, otherwise it is called a non-associated flow rule.

SOIL PROPERTIES

- Young's modulus, $E = 207$ MPa, and Poisson's ratio $\nu = 0.3$.
- Cohesion c = 69 MPa, and angle of internal friction $\phi = 20$ degrees.

CONSTRAINTS AND LOADS

- The clay layer is supported by a rigid and perfectly rough base. Therefore, apply a fixed constraint on the lower horizontal boundary.
- Model only the left half of the domain due to symmetry reasons. Use the symmetry boundary condition at the right vertical boundary.
- The stratum is subjected to a footing that you can consider to be flexible and smooth. The width of the strip footing is 3.14 m, see Figure 2. Gradually increase the footing pressure until the clay layer reaches the collapse load.

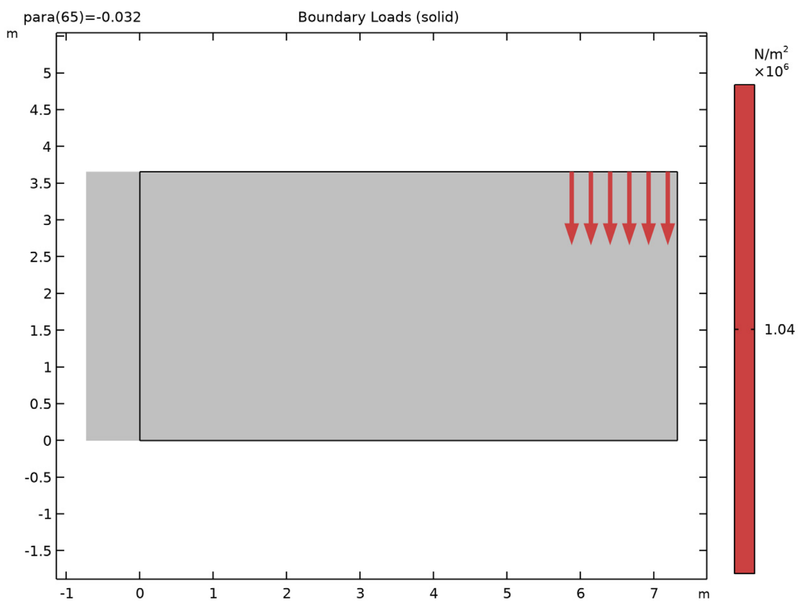

Figure 2: Boundary load applied to model the footing pressure

INFINITE ELEMENT DOMAIN

- In order to mimic an infinite layer of soil, add an Infinite Element Domain. The scaling `1e3*root.mod1.dGeomChar` means that the spatial variables in this domain are scaled thousand times the typical geometry length.

- The left vertical boundary is perfectly smooth and a can be assumed to be of the roller type.

Results and Discussion

You can study the load-displacement curves for both the Mohr-Coulomb and the Drucker-Prager criteria in Figure 3. The figure shows the applied footing pressure versus the centerline displacement (directly beneath the center of the footing) in the y direction. The lines show the load-displacement curves for the Mohr-Coulomb and Drucker-Prager criteria. The curves are identical up to 300 kPa because the whole domain is still within the elastic region. From that point when the pressure increases, the behavior diverges. Both curves reach the collapse load at approximately 1.1 MPa. The development of plastic strains in the soil at different stages of loading is shown in Figure 4.

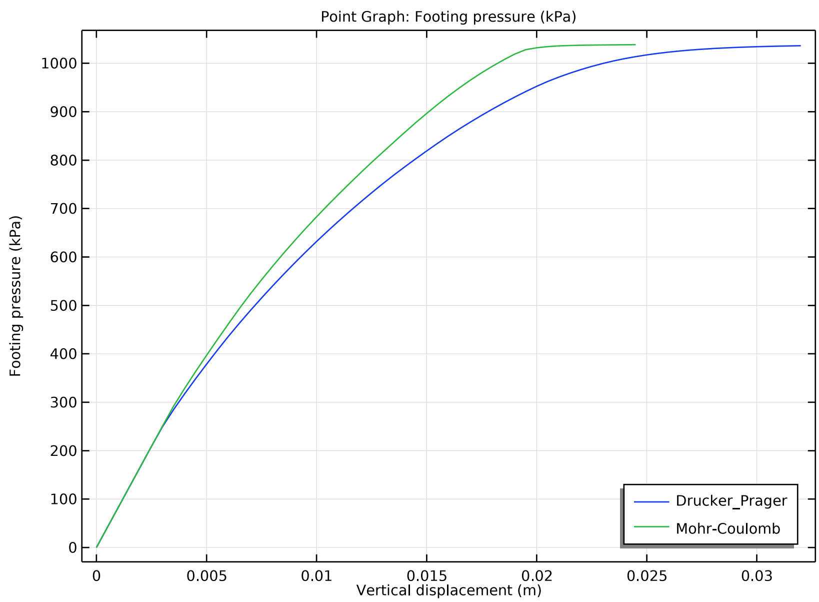

Figure 3: Footing pressure versus vertical displacement for the Mohr-Coulomb and Drucker-Prager material models.

Figure 4: Evolution of the equivalent plastic strain on the clay layer during the parametric loading.

Notes About the COMSOL Implementation

Both the Mohr-Coulomb or Drucker-Prager criterion are predefined in the Soil Plasticity subfeature. The default Mohr-Coulomb criterion uses a nonassociated flow rule, with a plastic potential implemented as **Drucker-Prager matched at compressive meridian**.

A suitable modeling technique in a case where the relation between the applied load and the displacement is highly nonlinear, is to use an algebraic equation that controls the applied pressure so that the model reaches the desired displacement increments. This is implemented using a **Global Equation**, and the parametric solver incrementally increases the displacement up to the desired vertical displacement.

References

1. W.F. Chen and E. Mizuno, *Nonlinear Analysis in Soil Mechanics*, Elsevier, 1990.

2. A. Mar. *How To Undertake Finite Element Based Geotechnical Analysis*, NAFEMS, 2002.

Application Library path: `Geomechanics_Module/Soil/flexible_footing`

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. Click **Add**.
4. Click **Study**.
5. In the **Select Study** tree, select **General Studies>Stationary**.
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 `7.32`.

4 In the **Height** text field, type `3.66`.

5 Click **Build Selected**.

Add a rectangle to model the infinite element domain.

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 `7.32*0.1`.

4 In the **Height** text field, type `3.66`.

5 Locate the **Position** section. In the **x** text field, type `-7.32*0.1`.

6 Click **Build Selected**.

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 `7.32-1.57`.

4 In the **y** text field, type `3.66`.

5 Click **Build Selected**.

Form Union (fin)

1 In the **Model Builder** window, click **Form Union (fin)**.

2 In the **Settings** window for **Form Union/Assembly**, click **Build Selected**.

DEFINITIONS

Infinite Element Domain 1 (ie1)

1 In the **Definitions** toolbar, click **Infinite Element Domain**.

The infinite element domain is scaled by a factor of 1000.

2 Select Domain 1 only.

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 In the table, enter the following settings:

Name	**Expression**	**Value**	**Description**
para	0	0	Prescribed displacement

DEFINITIONS

Use a nonlocal integration coupling to evaluate the displacement in the center of the applied pressure (Point 7).

Integration 1 (intop1)

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

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

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

4 Select Point 7 only.

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 Select the **Match to Mohr-Coulomb criterion** check box.

Linear Elastic Material 1

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

Soil Plasticity 2

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**.

Fixed Constraint 1

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

2 Select Boundaries 2 and 5 only.

Symmetry 1

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

2 Select Boundary 8 only.

Roller 1

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

2 Select Boundary 1 only.

Boundary Load 1

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

2 Select Boundary 7 only.

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

4 Specify the $\mathbf{F}_A$ vector as

0	x
footing_pressure	y

5 Click the **Show More Options** button in the **Model Builder** toolbar.

6 In the **Show More Options** dialog box, in the tree, select the check box for the node **Physics>Equation-Based Contributions**.

7 Click **OK**.

Global Equations 1

1 In the **Physics** toolbar, click **Global** and choose **Global Equations**.

2 In the **Settings** window for **Global Equations**, locate the **Global Equations** section.

3 In the table, enter the following settings:

Name	f(u,ut,utt,t) (1)	Initial value (u_0) (1)	Initial value (u_t0) (1/s)	Description
footing_pressure	intop1(v)-para	0	0	

4 Locate the **Units** section. Click **Select Dependent Variable Quantity**.

5 In the **Physical Quantity** dialog box, type `pressure` in the text field.

6 Click **Filter**.

7 In the tree, select **General>Pressure (Pa)**.

8 Click **OK**.

9 In the **Settings** window for **Global Equations**, locate the **Units** section.

10 Click **Select Source Term Quantity**.

11 In the **Physical Quantity** dialog box, type `displacement` in the text field.

12 Click **Filter**.

13 In the tree, select **General>Displacement (m)**.

14 Click **OK**.

MATERIALS

Material 1 (mat1)

1. In the **Model Builder** window, under **Component 1 (comp1)** right-click **Materials** and choose **Blank Material**.
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
Young's modulus	E	207e6	Pa	Young's modulus and Poisson's ratio
Poisson's ratio	nu	0.3	1	Young's modulus and Poisson's ratio
Cohesion	cohesion	69e3	Pa	Mohr-Coulomb
Angle of internal friction	internalphi	20[deg]	rad	Mohr-Coulomb

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 **Finer**.

Free Triangular 1

1. In the **Model Builder** window, click **Free Triangular 1**.
2. In the **Settings** window for **Free Triangular**, locate the **Domain Selection** section.
3. From the **Geometric entity level** list, choose **Domain**.
4. Select Domain 2 only.

Mapped 1

1 In the **Mesh** toolbar, click **Mapped**.

Use a mapped mesh in the infinite element domain to improve convergence.

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

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

4 Select Domain 1 only.

5 Click **Build All**.

The first study is parametric and solves the model assuming a Drucker-Prager criterion. The parameter represents the vertical displacement in the center of the applied pressure (Point 7). It runs from 0 to -32 mm with a step size of 0.5 mm.

DRUCKER-PRAGER CRITERION

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

2 In the **Settings** window for **Study**, type `Drucker-Prager Criterion` in the **Label** text field.

Step 1: Stationary

1 In the **Model Builder** window, under **Drucker-Prager Criterion** 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 2**.

5 Click **Disable**.

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
para (Prescribed displacement)	`range(0,-5e-4,-32e-3)`	

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 **Drucker-Prager Criterion>Solver Configurations>Solution 1 (sol1)>Stationary Solver 1** node, then click **Parametric 1**.

4 In the **Settings** window for **Parametric**, click to expand the **Continuation** section.

To improve convergence, set the predictor of the parametric solver to constant.

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

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

ROOT

The second study is also parametric and solves the model assuming a Mohr-Coulomb criterion. Again, the parameter represents the vertical displacement in the center of the applied pressure (Point 7), this time running from 0 to -24.5 mm with a step size of 0.5 mm.

ADD STUDY

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

MOHR-COULOMB CRITERION

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

2 In the **Settings** window for **Study**, type `Mohr-Coulomb Criterion` in the **Label** text field.

Step 1: Stationary

1 In the **Model Builder** window, under **Mohr-Coulomb Criterion** 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**.

5 Click **Disable**.

6 Locate 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
para (Prescribed displacement)	`range(0,-5e-4,-24.5e-3)`	

Solution 2 (sol2)

1 In the **Study** toolbar, click **Show Default Solver**.

To improve convergence, change the relative tolerance from 1e-3 to 1e-4 and set the predictor of the continuation solver to constant.

2 In the **Model Builder** window, expand the **Solution 2 (sol2)** node, then click **Stationary Solver 1**.

3 In the **Settings** window for **Stationary Solver**, locate the **General** section.

4 In the **Relative tolerance** text field, type `1e-4`.

5 In the **Model Builder** window, expand the **Mohr-Coulomb Criterion>Solver Configurations>Solution 2 (sol2)>Stationary Solver 1** node, then click **Parametric 1**.

6 In the **Settings** window for **Parametric**, locate the **Continuation** section.

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

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

RESULTS

Remove the infinite element domain from the dataset for plotting.

Drucker-Prager Criterion/Solution 1 (sol1)

In the **Model Builder** window, expand the **Results>Datasets** node, then click **Drucker-Prager Criterion/Solution 1 (sol1)**.

Selection

1 In the **Results** toolbar, click **Attributes** and choose **Selection**.

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

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

4 Select Domain 2 only.

Mohr-Coulomb Criterion/Solution 2 (sol2)

In the **Model Builder** window, under **Results>Datasets** click **Mohr-Coulomb Criterion/Solution 2 (sol2)**.

Selection

1 In the **Results** toolbar, click **Attributes** and choose **Selection**.

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

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

4 Select Domain 2 only.

Use mirror dataset to visualize the von Mises stress in the soil sample.

Mirror 2D 1

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

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

3 In row **Point 1**, set **X** to `7.32`.

4 In row **Point 2**, set **X** to `7.32`.

Mirror 2D 2

1 Right-click **Mirror 2D 1** and choose **Duplicate**.

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

3 From the **Dataset** list, choose **Mohr-Coulomb Criterion/Solution 2 (sol2)**.

Stress, Drucker-Prager Criterion

1 In the **Model Builder** window, under **Results** click **Stress (solid)**.

2 In the **Settings** window for **2D Plot Group**, type `Stress, Drucker-Prager Criterion` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Mirror 2D 1**.

4 Locate the **Plot Settings** section. Clear the **Plot dataset edges** check box.

Surface 1

1 In the **Model Builder** window, expand the **Stress, Drucker-Prager Criterion** node, then click **Surface 1**.

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

3 From the **Unit** list, choose **kPa**.

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** check box.

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

Arrow Line 1

1 In the **Model Builder** window, right-click **Stress, Drucker-Prager Criterion** and choose **Arrow Line**.

2 In the **Settings** window for **Arrow Line**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1 (comp1)> Solid Mechanics>Load>solid.F_Ax,solid.F_Ay - Load (spatial frame)**.

3 Click to expand the **Title** section. From the **Title type** list, choose **None**.

4 Locate the **Coloring and Style** section. From the **Arrow base** list, choose **Head**.

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

6 In the associated text field, type `5E-7`.

7 Click to expand the **Inherit Style** section. From the **Plot** list, choose **Surface 1**.

Deformation 1

Right-click **Arrow Line 1** and choose **Deformation**.

Stress, Drucker-Prager Criterion

In the **Stress, Drucker-Prager Criterion** toolbar, click **Plot**.

Applied Loads, Drucker-Prager Criterion

1 In the **Model Builder** window, under **Results** click **Applied Loads (solid)**.

2 In the **Settings** window for **Group**, type `Applied Loads, Drucker-Prager Criterion` in the **Label** text field.

Stress, Mohr-Coulomb Criterion

1 In the **Model Builder** window, expand the **Applied Loads, Drucker-Prager Criterion** node, then click **Results>Stress (solid) 1**.

2 In the **Settings** window for **2D Plot Group**, type `Stress, Mohr-Coulomb Criterion` in the **Label** text field.

Plastic Region, Mohr-Coulomb Criterion

1 In the **Model Builder** window, under **Results** click **Equivalent Plastic Strain (solid) 1**.

2 In the **Settings** window for **2D Plot Group**, type `Plastic Region, Mohr-Coulomb Criterion` in the **Label** text field.

Contour 1

1 In the **Model Builder** window, expand the **Plastic Region, Mohr-Coulomb Criterion** node, then click **Contour 1**.

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

3 In the **Expression** text field, type `solid.epeGp>0`.

Deformation 1

1 Right-click **Contour 1** and choose **Deformation**.

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

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

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

5 In the **Plastic Region, Mohr-Coulomb Criterion** toolbar, click **Plot**.

Applied Loads, Mohr-Coulomb Criterion

1 In the **Model Builder** window, under **Results** click **Applied Loads (solid) 1**.

2 In the **Settings** window for **Group**, type `Applied Loads, Mohr-Coulomb Criterion` in the **Label** text field.

Plastic Region, Drucker-Prager Criterion

1 In the **Model Builder** window, under **Results** click **Equivalent Plastic Strain (solid)**.

2 In the **Settings** window for **2D Plot Group**, type `Plastic Region, Drucker-Prager Criterion` in the **Label** text field.

Contour 1

1 In the **Model Builder** window, expand the **Plastic Region, Drucker-Prager Criterion** node, then click **Contour 1**.

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

3 In the **Expression** text field, type `solid.epeGp>0`.

Deformation 1

1 Right-click **Contour 1** and choose **Deformation**.

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

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

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

5 In the **Plastic Region, Drucker-Prager Criterion** toolbar, click **Plot**.

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

Footing Pressure vs. Displacement

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

2 Drag and drop below **Plastic Region, Mohr-Coulomb Criterion**.

3 In the **Settings** window for **1D Plot Group**, type `Footing Pressure vs. Displacement` in the **Label** text field.

4 Locate the **Legend** section. From the **Position** list, choose **Lower right**.

Point Graph 1

1 Right-click **Footing Pressure vs. Displacement** and choose **Point Graph**.

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

3 From the **Dataset** list, choose **Drucker-Prager Criterion/Solution 1 (sol1)**.

4 Select Point 7 only.

5 Locate the **y-Axis Data** section. In the **Expression** text field, type `abs(footing_pressure)`.

6 From the **Unit** list, choose **kPa**.

7 Select the **Description** check box.

8 In the associated text field, type `Footing pressure`.

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

10 In the **Expression** text field, type `abs(v)`.

11 Select the **Description** check box.

12 In the associated text field, type `Vertical displacement`.

13 Click to expand the **Legends** section. Select the **Show legends** check box.

14 From the **Legends** list, choose **Manual**.

15 In the table, enter the following settings:

Legends
`Drucker_Prager`

Point Graph 2

1 Right-click **Point Graph 1** and choose **Duplicate**.

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

3 From the **Dataset** list, choose **Mohr-Coulomb Criterion/Solution 2 (sol2)**.

4 Click to expand the **Title** section. From the **Title type** list, choose **None**.

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

Legends
`Mohr-Coulomb`

Footing Pressure vs. Displacement

1 In the **Model Builder** window, click **Footing Pressure vs. Displacement**.

2 In the **Footing Pressure vs. Displacement** toolbar, click **Plot**.