COMSOL MULTIPHYSICS®

Flexible and Smooth Strip Footing on Stratum of Clay

SOLVED WITH COMSOL MULTIPHYSICS 3.5a

© COPYRIGHT 2008. All right reserved. No part of this documentation may be photocopied or reproduced in any form without prior written consent from COMSOL AB. COMSOL, COMSOL Multiphysics, COMSOL Reaction Engineering Lab, and FEMLAB are registered trademarks of COMSOL AB. Other product or brand names are trademarks or registered trademarks of their respective holders.

COMSOL

Flexible and Smooth Strip Footing on Stratum of Clay

Model Definition

A typical verification model for geotechnical problems is a shallow stratum layer of clay, depicted in Figure 1 below. The model applies a vertical load on the clay stratum, and then studies the static response and the collapse load.

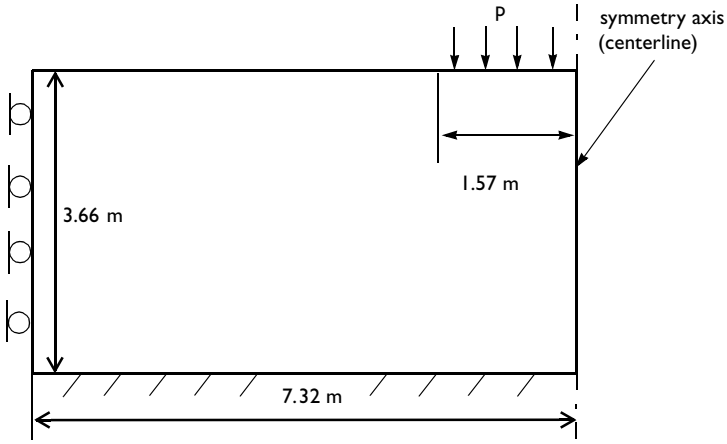


Figure 1: Dimensions, boundary conditions, and pressure load for the stratum of clay.

ANALYSIS TYPE

Yield Surface

Model the clay as a linear elastic-perfectly plastic material, and assume plane-strain conditions in this analysis.

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

$$F = \alpha I_1 + \sqrt{J_2} = K$$

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 normal stress components:

$$I_1 = \sigma_x + \sigma_y + \sigma_z$$

The second stress invariant is defined by

$$I_2 = \sigma_x\sigma_y + \sigma_y\sigma_z + \sigma_z\sigma_x - \tau_{xy}^2$$

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 becomes identical to the Mohr-Coulomb criterion if the material parameters α and K are given by (see Ref. 1)

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

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

The model employs the Mohr-Coulumb yield condition with an associated flow rule and a nonassociated flow rule.

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

$$d\varepsilon_{ij} = d\lambda\frac{\partial Q}{\partial\sigma_{ij}}$$

where $d\lambda$ 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 the rule is called an associated flow rule, otherwise a nonassociated flow rule.

The associated flow unfortunately does not always describe the real physical process in a sufficiently realistic manner, especially if the material is pressure sensitive (soil, for example). Usually, the associated flow rule overestimates the volumetric plastic strains.

The nonassociated flow rule case uses a von Mises type of function for the plastic potential in this model. This function does not include volumetric plastic strains. The plastic potential can thus be written as

$$Q = \sqrt{J_2}$$

MATERIAL 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, fix the lower horizontal boundary.
- The left vertical boundary is perfectly smooth and a roller boundary condition is suitable to use at this boundary.
- Model only the left half of the domain due to symmetry reasons. Thus, use 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. Gradually increase the footing pressure until the clay layer reaches the collapse load.

This example is derived from *Nonlinear Analysis in Soil Mechanics* by W.F. Chen and E. Mizuno (Ref. 1).

Results and Discussion

You can study in Figure 2 the load-displacement curves for both the associated case and the nonassociated case. The figure shows the applied footing pressure versus the centerline displacement (directly beneath the footing's center) in the y direction. The dashed line shows the load-displacement curve for the nonassociated flow rule case and the solid line shows the load-displacement curve for associated flow rule case.

The model uses the SI unit system. The load-displacement curves in Ref. 1 are however given in psi (pounds per square inch) and inches. To facilitate comparisons, the load-displacement plots use these units.

The curves are identical up to 40 psi because the whole domain is still within the elastic region. When the pressure increases the behavior diverges. The curve for the associated flow rule bends at approximately 150 psi and reaches the collapse load at approximately 160 psi. The curve for the nonassociated case deviates gradually from

the associated case and reaches a collapse load at approximately 146 psi. The collapse load deviates only by approximately 3% from the solution given in Ref. 1.

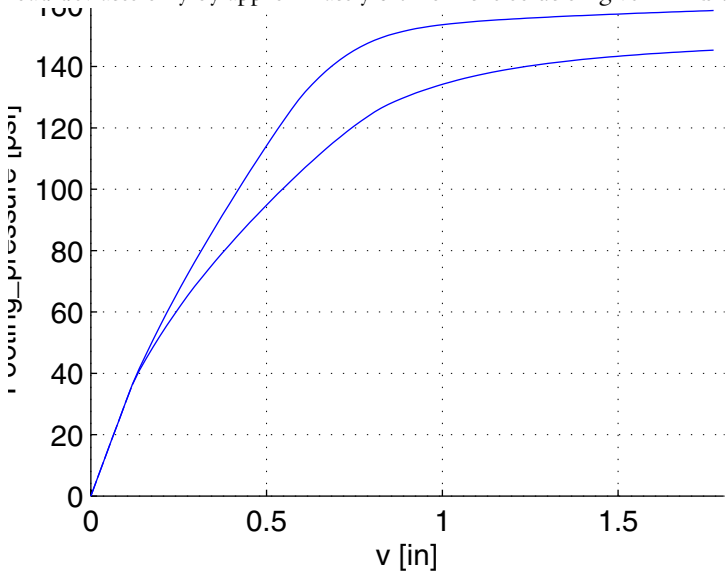


Figure 2: Footing pressure vs. vertical displacement for the nonassociated case (upper line) and the associated case (lower line).

Modeling in COMSOL Multiphysics

Implement the Mohr-Coulomb elastic-perfectly plastic material model with a user-defined yield function and a perfectly plastic hardening function. The yield function contains the first stress invariant, I_1, and the second deviatoric stress invariant, J_2, as well as the material property constants α and K. You define the stress variables and the material properties as scalar expression variables and constants.

Model the associated flow rule case in the graphical user interface. For the nonassociated flow rule case, on the other hand, you need to model on the command line. The workflow for the nonassociated flow rule case:

1. Export the FEM structure for the associated case from the graphical user interface to the command line.
2. Edit the plastic potential definition.
3. Solve and postprocess the model on the command line.

A suitable modeling technique in a case like this, 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. The ODE functionality in COMSOL Multiphysics provides the means for entering the algebraic equation, and the parametric solver steps up the desired vertical displacement.

Convergence problems can occur in elasto-plastic problems like this. If this happens, try scaling the solution components manually.

Reference

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

Model Library path: `Structural_Mechanics_Module/Nonlinear_Material_Models/flexible_footing`

Modeling Using the Graphical User Interface

1. In the **Model Navigator** select **2D** from the **Space dimension** list on the **New** page.
2. Select **Structural Mechanics Module>Plane Strain>Static analysis elasto-plastic material**.
3. From the **Element** list select **Lagrange - Linear**.
4. Click **OK**.

Options and Settings

1. From the **Options** menu select **Constants**. Enter constant names, expressions, units and descriptions (optional) according to the following table:

NAME	EXPRESSION	DESCRIPTION
E	2.07e8[Pa]	Young's modulus
nu	0.3	Poisson's ratio
phi	20/180*pi	Angle of internal friction
c	69e3[Pa]	Cohesion

NAME	EXPRESSION	DESCRIPTION
alpha	tan(phi)/sqrt(9+12*(tan(phi))^2)	Drucker-Prager material constant
K	(3*c)/sqrt(9+12*(tan(phi))^2)	Drucker-Prager material constant

2 Click **OK**.

3 From the **Options** menu select **Expressions>Scalar Expressions**. Enter expression names, expressions, and descriptions (optional) according to the following table:

NAME	EXPRESSION	DESCRIPTION
I1	sx_smpn+sy_smpn+sz_smpn	First stress invariant
I2	sx_smpn*sy_smpn+ sy_smpn*sz_smpn+ sz_smpn*sx_smpn-sxy_smpn^2	Second stress invariant
J2	1/3*I1^2-I2	Second deviatoric stress invariant
F	(alpha*I1+sqrt(J2))	Yield surface
Footing_pressure	-Pressure/1[psi]	Pressure in psi

Note that the unit for the variable `Footing_pressure` has red colored brackets because an ODE variable (`Pressure`) is used in the expression for this variable. Dividing the `Pressure` variable with `1[psi]` extracts the value of this variable measured in the unit psi.

4 Click **OK** to close the dialog box.

GEOMETRY MODELING

1 Shift-click the **Rectangle/Square** button on the Draw toolbar. In the **Width** edit field type `7.32` and in the **Height** edit field type `3.66`. Click **OK**.

2 Shift-click the **Point** button. In the **x** edit field type `5.75` and in the **y** edit field type `3.66`, then click **OK**.

3 Click the **Zoom Extents** button on the Main toolbar.

PHYSICS SETTINGS

Boundary Settings

1 From the **Physics** menu, select **Boundary Settings**.

2 Specify boundary settings according to the following table; when done, click **OK**.

BOUNDARY	PAGE	CONSTRAINT CONDITION	QUANTITY	VALUE
1	Constraint	Roller		
2	Constraint	Fixed		
5	Constraint	Symmetry plane		
4	Load		Fy	`Pressure`

Subdomain Settings

1 From the **Physics** menu, select **Subdomain Settings**.

2 Select Subdomain 1, then verify and specify data on the **Material** page according to the following table:

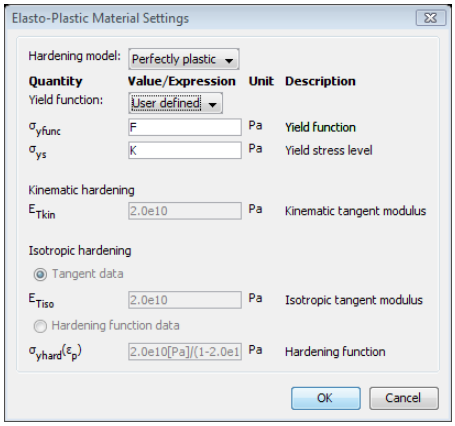
SETTINGS	VALUE
Material model	Elasto-plastic
E	E
ν	nu

3 Click the **Elasto-Plastic Material Settings** button to open the **Elasto-Plastic Material Settings** dialog box.

4 From the **Hardening model** list select **Perfectly plastic**.

5 From the **Yield function** list select **User defined**.

6 In the σ_{yfunc} edit field type F and in the σ_{ys} edit field type K.



7 Click **OK**.

8 Click **OK** to close the **Subdomain Settings** dialog box.

Integration Coupling Variable

1 From the **Options** menu, select **Integration Coupling Variables>Point Variables**.

2 Select Point 5.

3 Type `vc` in the **Name** edit field and `v` in the **Expression** edit field.

4 Click **OK** to close the **Point Integration Variables** dialog box.

By defining the above point integration coupling variable *vc* the deformation in *y* direction at Point 5 is made available for the next step where an algebraic equation is created.

Global Equations

1 From the **Physics** menu, select **Global Equations**.

2 Enter `Pressure` in the **Name** edit field and `vc-para` in the **Equation** edit field.

3 Click **OK**.

MESH GENERATION

1 From the **Mesh** menu, choose **Free Mesh Parameters**.

2 From the **Predefined mesh sizes** list, select **Finer**.

3 Click the **Remesh** button and then click **OK**.

COMPUTING THE SOLUTION

1 Click the **Solver Parameters** button on the Main toolbar.

2 In the **Parameter name** edit field type the name `para`.

3 In the **Parameter values** edit field type `range(0,-0.0005,-0.045)`. Click **OK**.

4 Click the **Solve** button on the Main toolbar.

POSTPROCESSING AND VISUALIZATION

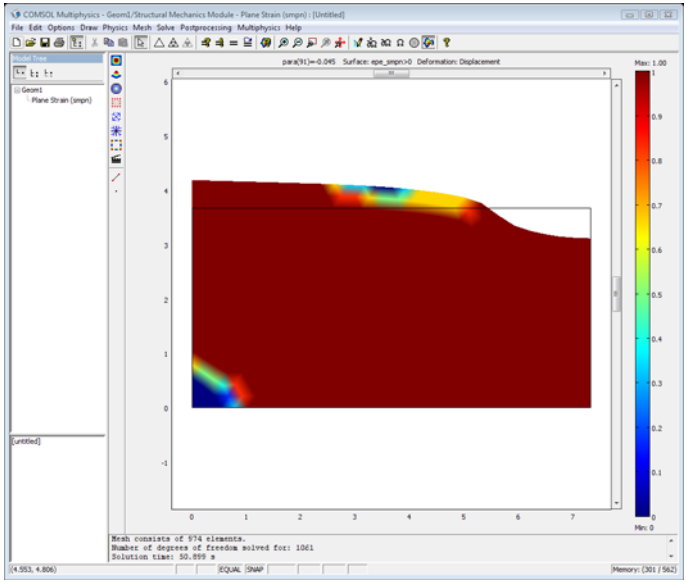
1 Click the **Plot Parameters** button on the Main toolbar.

2 Select the **Surface** and **Deformed shape** check boxes in the **Plot type** area on the **General** page.

3 Click the **Surface** tab.

4 In the **Expression** edit field type `epe_smpn>0` (`epe_smpn` is the variable name for the is the effective plastic strain).

5 Click **OK**.



6 From the **Postprocessing** menu, select **Domain Plot Parameters**.

7 Click the **Point** tab.

8 Select Point 5.

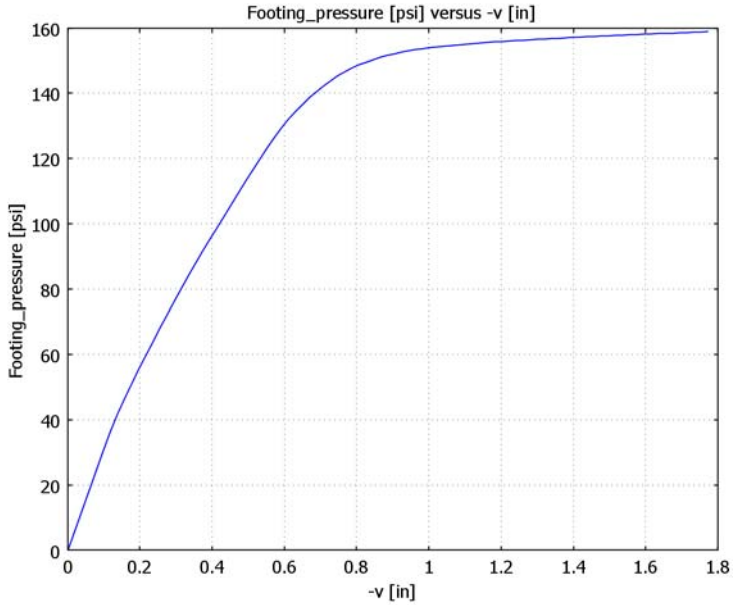
9 In the **Expression** edit field, type `Footing_pressure`.

10 In the **x-axis data** frame, select **Expression**.

11 Click the **Expression** button and enter `-v` in the **Expression** edit field.

12 From the **Unit** list select **in**, then click **OK**.

13 Click **OK**.



While the graphical user interface supports the associated flow rule, you need to define, solve, and postprocess the nonassociated flow rule model on the command line. You need MATLAB for this part of the simulation. The following instructions show how to change the plastic potential to a von Mises function, solve the problem, and plot the footing pressure as function of the displacement in the same figure as for the associated case. The instructions assume that you are running COMSOL Multiphysics with MATLAB.

1 From the **File** menu, select **Export>FEM Structure as 'fem'** to export the model to the MATLAB workspace.

2 First recreate the previous plot in a MATLAB figure by entering the following commands at the command prompt:

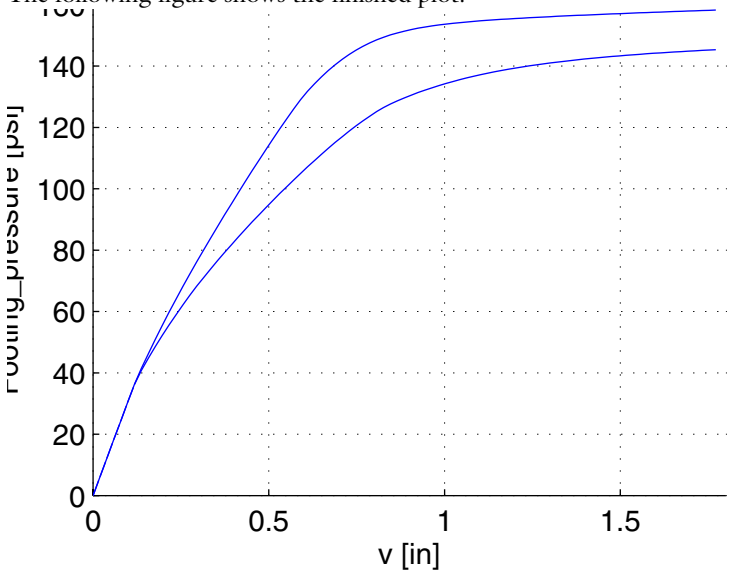
```
figure
postcrossplot(fem,0,[5], ...
              'pointdata','Footing_pressure', ...
              'pointxdata',{'-v','unit','in'}, ...
              'refine','auto');
```

3 Enter the following script at the command line prompt or run the M-file `flexible_footing_nonassoc`, which is available in your MATLAB path.

```
G = fem.elemmph(3);
G{1}.geomdim{1}{3}.G{1}{1} = 'd(mises_smpn,sx_smpn)';
G{1}.geomdim{1}{3}.G{1}{2} = 'd(mises_smpn,sy_smpn)';
G{1}.geomdim{1}{3}.G{1}{3} = 'd(mises_smpn,sz_smpn)';
G{1}.geomdim{1}{3}.G{1}{4} = '0.5*d(mises_smpn,sxy_smpn)';
fem.elemmph(3)=G;

% Extend mesh
fem.xmesh=meshextend(fem);

% Solve problem
fem.sol=femnlin(fem, ...
'u',0, ...
    'solcomp',{'u','v','Pressure'}, ...
    'outcomp',{'epx_smpn','u','epe_smpn','epz_smpn','v',...
    'epxy_smpn','epy_smpn','Pressure'}, ...
    'pname','para', ...
    'pinitstep',1e-4, ...
    'pminstep',1e-6, ...
    'pmaxstep',0.01,...
    'plist',range(0,-0.0005,-0.045), ...
    'uscale',{'u','0.01','v','0.01'}, ...
    'hnlin','on');

% Plot in cross-section or along domain
hold on
postcrossplot(fem,0,[5], ...
    'pointdata','Footing_pressure', ...
    'pointxdata',{'-v','unit','in'}, ...
    'title','Footing_pressure [psi] versus displacement [in]', ...
    'axislabel',{'v [in]','Footing_pressure [psi]'}, ...
    'refine','auto','linstyle','--');
```

The following figure shows the finished plot.