Created in COMSOL Multiphysics 6.1

Block Verification

Introduction

This example shows how to set up a compression test on a prestressed soil sample. Due to a simple stress state, it is possible to determine the vertical yield stress analytically. The soil sample is modeled with soil plasticity and the Mohr–Coulomb criterion.

Model Definition

In this example, we consider a block of soil of 1 m length on each side. The soil is pressed from the sides by boundary loads in the x and y directions, and from the top by a prescribed displacement.

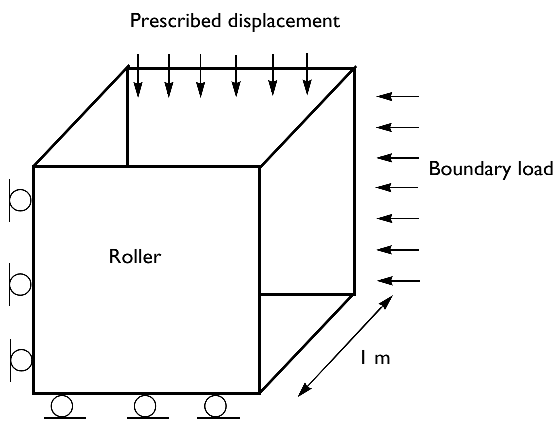

Figure 1: Dimensions, boundary conditions, and loads for the test.

ELASTIC PROPERTIES

The soil properties are taken from standard clay.

- Young's modulus, $E = 207$ MPa, and Poisson's ratio $\nu = 0.3$.

SOIL PLASTICITY

- Cohesion c = 70 kPa, and angle of internal friction $\phi = 30°$.
- Use the Mohr–Coulomb criterion, with non-associated flow rule, and Drucker–Prager matched at compressive meridian as plastic potential.

CONSTRAINTS AND LOADS

- The test is based on uniaxial compression. Fix the normal displacement at the lower, left, and right boundaries with a roller boundary condition (these are the boundaries at $x = 0$, $y = 0$, and $z = 0$).
- The in situ stress is prescribed via the External Stress node, it is 300 kPa, 200 kPa, and 100 kPa in the x, y, and z directions, respectively. The in situ stress components are shown on Figure 2.
- The soil sample is subjected to a loading at the top boundary through a prescribed displacement. By means of a parametric sweep, the displacement is gradually increased up to 8 mm.

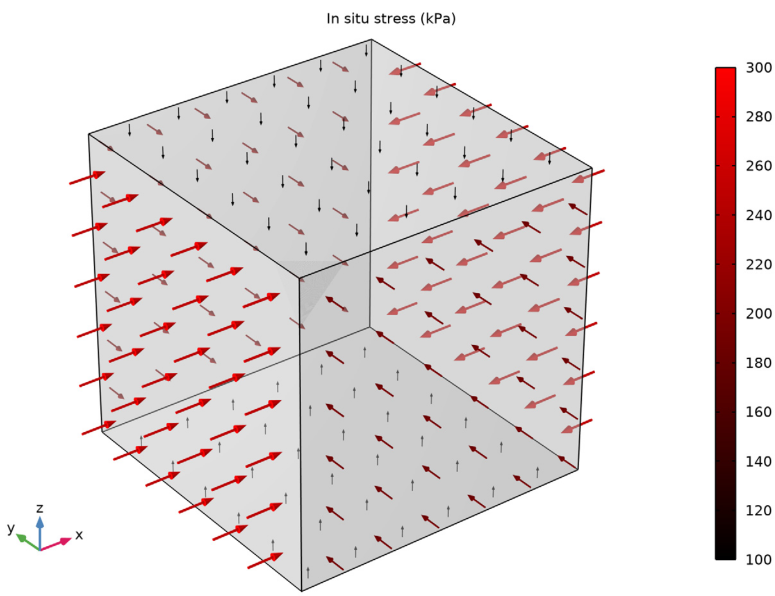

Figure 2: In situ stress applied on the surface of the block.

Results and Discussion

The cube of soil experiences a homogeneous stress state, as shown by distribution of von Mises stress in Figure 3.

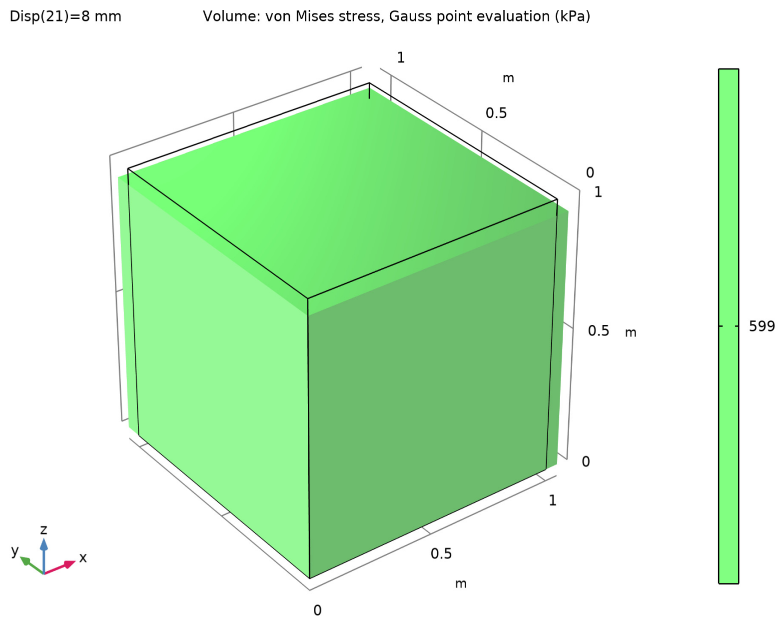

Figure 3: Equivalent stress and deformation in the soil sample after applying 8 mm displacement from the top.

The stress increases with the compression of the block, as shown on Figure 4.

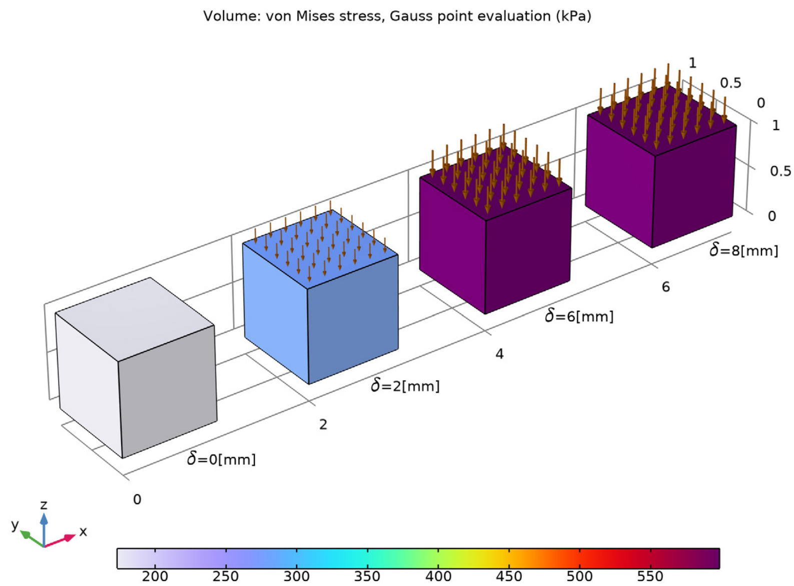

Figure 4: Equivalent stress with compression displacement of 0 mm, 2 mm, 6 mm and 8 mm

From the Mohr circle, the Mohr–Coulomb criterion can be written in terms of the biggest and smallest principal stress:

$$\frac{1}{2}(\sigma_1 - \sigma_3) + \frac{1}{2}(\sigma_1 + \sigma_3)\sin\phi - c\cos\phi = 0$$

Since stress in the y direction is the largest principal stress and the stress in the z direction is the smallest principal stress at the onset of yielding, its analytical value can be obtained. Manipulation of the above formula gives

$$\sigma_{ZZ} = \frac{2c\cos\phi - \sigma_{YY}(1 + \sin\phi)}{(\sin\phi - 1)} \tag{1}$$

The stresses history together with the analytical value of the stress in the z direction at the onset of yielding is shown in Figure 5. Plastic yielding is reached after a deflection of about 3.5 mm.

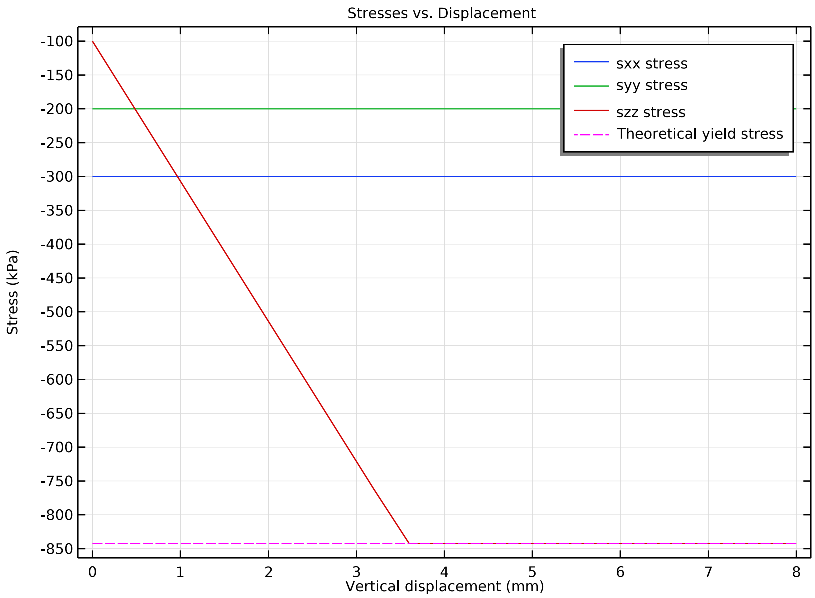

Figure 5: This plot shows how the soil sample behaves elastically until it reaches the yield surface at the compressive meridian.

Application Library path: `Geomechanics_Module/Verification_Examples/block_verification`

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  **3D**.

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**.

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
Disp	0[mm]	0 m	Displacement parameter
X_stress	-3e5[Pa]	-3E5 Pa	In situ stress, xx-component
Y_stress	-2e5[Pa]	-2E5 Pa	In situ stress, yy-component
Z_stress	-1e5[Pa]	-1E5 Pa	In situ stress, zz-component

In situ stresses are set with negative sign to fit the structural mechanics convention which assumes negative stresses in compression, and positive in tension.

GEOMETRY 1

Block 1 (blk1)

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

2 Right-click **Geometry 1** and choose **Block**.

3 In the **Settings** window for **Block**, click **Build All Objects**.

The geometry consists of a simple unit block.

DEFINITIONS

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 **Boundary**.

4 Select Boundary 4 only.

Variables 1

1 In the **Model Builder** window, right-click **Definitions** and choose **Variables**.

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

3 In the table, enter the following settings:

Name	Expression	Unit	Description
Forcez	intop1(solid.sz)/ intop1(1)-Z_stress	N/m^2	Axial force
szz_th	(2*solid.cohesion* cos(solid.internalphi)- Y_stress*(1+ sin(solid.internalphi)))/ (sin(solid.internalphi)- 1)		Theoretical yield stress

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**.

The **Mohr-Coulomb** criterion is used to define yield surface. Use the non-associated Drucker-Prager plastic potential matched to the Mohr-Coulomb model at the compressive meridian.

Linear Elastic Material 1

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

External Stress 1

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

Choose the **In situ** option in order to apply the in situ stresses.

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

3 From the **Stress input** list, choose **In situ stress**.

4 From the list, choose **Symmetric**.

5 In the σ_{ins} table, enter the following settings:

X_stress	0	0
0	Y_stress	0
0	0	Z_stress

Roller 1

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

2 Select Boundaries 1–3 only.

Prescribed Displacement 1

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

2 Select Boundary 4 only.

3 In the **Settings** window for **Prescribed Displacement**, locate the **Prescribed Displacement** section.

4 Select the **Prescribed in z direction** check box.

5 In the u_{0z} text field, type `-Disp`.

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'smodulus	E	207e6	Pa	Young'smodulus and Poisson's ratio
Poisson's ratio	nu	0.3	1	Young'smodulus and Poisson's ratio
Density	rho	2000	kg/m³	Basic
Cohesion	cohesion	70e3	Pa	Mohr-Coulomb
Angle of internal friction	internalphi	30[deg]	rad	Mohr-Coulomb

MESH 1

Mapped 1

1 In the **Mesh** toolbar, click **Boundary** and choose **Mapped**.

2 Select Boundary 4 only.

Swept 1

In the **Mesh** toolbar, click **Swept**.

Size

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

2 In the **Settings** window for **Size**, locate the **Element Size** section.

3 From the **Predefined** list, choose **Extra coarse**.

4 Click **Build All**.

STUDY 1

Step 1: Stationary

Set up an auxiliary continuation sweep for the `Disp` parameter.

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

2 In the **Settings** window for **Stationary**, click to expand the **Study Extensions** section.

3 Select the **Auxiliary sweep** check box.

4 Click **Add**.

5 In the table, enter the following settings:

Parameter name	**Parameter value list**	**Parameter unit**
Disp (Displacement parameter)	`8*range(0,0.05,1)`	`mm`

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

RESULTS

Volume 1

1 In the **Model Builder** window, expand the **Stress (solid)** node, then click **Volume 1**.

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

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

4 In the **Stress (solid)** toolbar, click **Plot**.

Stress (solid)

The default plot shows uniform stress (Figure 3). Modify it to show the von Mises stress at different stages of uniaxial compression, (Figure 4).

1 In the **Model Builder** window, click **Stress (solid)**.
2 In the **Settings** window for **3D Plot Group**, click to expand the **Title** section.
3 From the **Title type** list, choose **Custom**.
4 Find the **Solution** subsection. Clear the **Solution** check box.
5 Locate the **Color Legend** section. From the **Position** list, choose **Bottom**.
6 Click to expand the **Plot Array** section. Enable array to plot the von Mises stress at different stages of uniaxial compression.
7 Select the **Enable** check box.
8 In the **Relative padding** text field, type 1.

Volume 1

1 In the **Model Builder** window, click **Volume 1**.
2 In the **Settings** window for **Volume**, locate the **Data** section.
3 From the **Dataset** list, choose **Study 1/Solution 1 (sol1)**.
4 From the **Parameter value (Disp (mm))** list, choose **0**.
5 Click to expand the **Plot Array** section. Select the **Manual indexing** check box.

Deformation

1 In the **Model Builder** window, expand the **Volume 1** node.
2 Right-click **Results>Stress (solid)>Volume 1>Deformation** and choose **Delete**.

Volume 1

Duplicate the **Volume** node three times to plot the von Mises stress at different stages of uniaxial compression.

Volume 2

1 In the **Model Builder** window, under **Results>Stress (solid)** right-click **Volume 1** and choose **Duplicate**.
2 In the **Settings** window for **Volume**, locate the **Data** section.
3 From the **Parameter value (Disp (mm))** list, choose **2**.
4 Click to expand the **Inherit Style** section. Click to expand the **Title** section. From the **Title type** list, choose **None**.
5 Locate the **Inherit Style** section. From the **Plot** list, choose **Volume 1**.

6. Locate the **Plot Array** section. In the **Index** text field, type `1`.

Volume 3

1. Right-click **Volume 2** and choose **Duplicate**.
2. In the **Settings** window for **Volume**, locate the **Data** section.
3. From the **Parameter value (Disp (mm))** list, choose **6**.
4. Locate the **Plot Array** section. In the **Index** text field, type `2`.

Volume 4

1. Right-click **Volume 3** and choose **Duplicate**.
2. In the **Settings** window for **Volume**, locate the **Data** section.
3. From the **Parameter value (Disp (mm))** list, choose **8**.
4. Locate the **Plot Array** section. In the **Index** text field, type `3`.
5. In the **Stress (solid)** toolbar, click **Plot**.

Arrow Surface 1

1. In the **Model Builder** window, right-click **Stress (solid)** and choose **Arrow Surface**.
2. In the **Settings** window for **Arrow Surface**, locate the **Data** section.
3. From the **Dataset** list, choose **Study 1/Solution 1 (sol1)**.
4. From the **Parameter value (Disp (mm))** list, choose **0**.
5. Locate the **Expression** section. In the **X-component** text field, type `0`.
6. In the **Y-component** text field, type `0`.
7. In the **Z-component** text field, type `Forcez`.
8. Click to expand the **Title** section. From the **Title type** list, choose **None**.
9. Locate the **Arrow Positioning** section. In the **Number of arrows** text field, type `40`.
10. Locate the **Coloring and Style** section. From the **Arrow base** list, choose **Head**.
11. From the **Color** list, choose **Custom**.
12. On Windows, click the colored bar underneath, or — if you are running the cross-platform desktop — the **Color** button.
13. Click **Define custom colors**.
14. Set the RGB values to 128, 64, and 0, respectively.
15. Click **Add to custom colors**.
16. Click **Show color palette only** or **OK** on the cross-platform desktop.
17. Click to expand the **Plot Array** section. Select the **Manual indexing** check box.

Selection 1

1 Right-click **Arrow Surface 1** and choose **Selection**.

2 Select Boundary 4 only.

Duplicate the **Arrow Surface** node three times to plot the axial force at different stages of uniaxial compression.

Arrow Surface 2

1 In the **Model Builder** window, under **Results>Stress (solid)** right-click **Arrow Surface 1** and choose **Duplicate**.

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

3 From the **Parameter value (Disp (mm))** list, choose **2**.

4 Locate the **Plot Array** section. In the **Index** text field, type 1.

5 Locate the **Coloring and Style** section.

6 Select the **Scale factor** check box. In the associated text field, type 4E-7.

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

8 Clear the **Arrow scale factor** check box.

Arrow Surface 3

1 Right-click **Arrow Surface 2** and choose **Duplicate**.

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

3 From the **Parameter value (Disp (mm))** list, choose **6**.

4 Locate the **Plot Array** section. In the **Index** text field, type 2.

5 Locate the **Inherit Style** section. From the **Plot** list, choose **Arrow Surface 2**.

Arrow Surface 4

1 Right-click **Arrow Surface 3** and choose **Duplicate**.

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

3 From the **Parameter value (Disp (mm))** list, choose **8**.

4 Locate the **Plot Array** section. In the **Index** text field, type 3.

5 In the **Stress (solid)** toolbar, click **Plot**.

Stress (solid)

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

Table Annotation 1

1 In the **Stress (solid)** toolbar, click **More Plots** and choose **Table Annotation**.

2 In the **Settings** window for **Table Annotation**, locate the **Data** section.

3 From the **Source** list, choose **Local table**.

4 In the table, enter the following settings:

x-coordinate	y-coordinate	z-coordinate	Annotation
0.5	-0.1	0	δ=0[mm]
2.5	-0.1	0	δ=2[mm]
4.5	-0.1	0	δ=6[mm]
6.5	-0.1	0	δ=8[mm]

5 Select the **LaTeX markup** check box.

6 Locate the **Coloring and Style** section. Clear the **Show point** check box.

Stress (solid)

1 Click the **Show Grid** button in the **Graphics** toolbar.

2 In the **Model Builder** window, click **Stress (solid)**.

3 In the **Settings** window for **3D Plot Group**, locate the **Plot Settings** section.

4 From the **View** list, choose **New view**.

5 In the **Stress (solid)** toolbar, click **Plot**.

Add a 1D plot to show the evolution of the stress-tensor components versus the displacement at the top surface.

Stresses vs. Displacement

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

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

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

4 Locate the **Plot Settings** section.

5 Select the **x-axis label** check box. In the associated text field, type `Vertical displacement (mm)`.

6 Select the **y-axis label** check box. In the associated text field, type `Stress (kPa)`.

Point Graph 1

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

2 Select Point 1 only.

3 In the **Settings** window for **Point Graph**, click **Replace Expression** in the upper-right corner of the **y-Axis Data** section. From the menu, choose **Component 1 (comp1)>Solid Mechanics>Stress>Stress tensor (spatial frame) - N/m^2>solid.sxx - Stress tensor, xx-component**.

4 Locate the **y-Axis Data** section. From the **Unit** list, choose **kPa**.

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

6 In the **Expression** text field, type `Disp`.

7 From the **Unit** list, choose **mm**.

8 Click to expand the **Coloring and Style** section. Click to expand the **Legends** section. Select the **Show legends** check box.

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

10 In the table, enter the following settings:

Legends
sxx stress

11 In the **Stresses vs. Displacement** toolbar, click **Plot**.

Point Graph 2

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

2 In the **Settings** window for **Point Graph**, click **Replace Expression** in the upper-right corner of the **y-Axis Data** section. From the menu, choose **Component 1 (comp1)>Solid Mechanics>Stress>Stress tensor (spatial frame) - N/m^2>solid.syy - Stress tensor, yy-component**.

3 In the **Stresses vs. Displacement** toolbar, click **Plot**.

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

Legends
syy stress

Point Graph 3

1 In the **Model Builder** window, under **Results>Stresses vs. Displacement** right-click **Point Graph 1** and choose **Duplicate**.

2 In the **Settings** window for **Point Graph**, click **Replace Expression** in the upper-right corner of the **y-Axis Data** section. From the menu, choose **Component 1 (comp1)>Solid Mechanics>Stress>Stress tensor (spatial frame) - N/m^2>solid.szz - Stress tensor, zz-component**.

3 In the **Stresses vs. Displacement** toolbar, click **Plot**.

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

Legends
`szz stress`

Point Graph 4

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

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

3 In the **Expression** text field, type `szz_th`.

4 Locate the **Coloring and Style** section. Find the **Line style** subsection. From the **Line** list, choose **Dashed**.

5 From the **Color** list, choose **Magenta**.

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

Legends
`Theoretical yield stress`

Stresses vs. Displacement

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

2 In the **Stresses vs. Displacement** toolbar, click **Plot**.

Finally, plot the in situ stress on the boundaries to reproduce Figure 2.

In Situ Stress

1 In the **Home** toolbar, click **Add Plot Group** and choose **3D Plot Group**.

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

3 Locate the **Title** section. From the **Title type** list, choose **Manual**.

4 In the **Title** text area, type `In situ stress (kPa)`.

5 Clear the **Parameter indicator** text field.

Volume 1

1 In the **In Situ Stress** toolbar, click **Volume**.

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

3 In the **Expression** text field, type `1`.

4 Locate the **Coloring and Style** section. From the **Coloring** list, choose **Uniform**.

5 From the **Color** list, choose **Gray**.

Transparency 1

1 In the **In Situ Stress** toolbar, click **Transparency**.
2 In the **Settings** window for **Transparency**, locate the **Transparency** section.
3 Set the **Transparency** value to **0.7**.

In Situ Stress

In the **Model Builder** window, under **Results** click **In Situ Stress**.

Arrow Surface 1

1 In the **In Situ Stress** toolbar, click **Arrow Surface**.
2 In the **Settings** window for **Arrow Surface**, locate the **Expression** section.
3 In the **X-component** text field, type `solid.SinsXX*solid.nX+solid.SinsXY*solid.nY+solid.SinsXZ*solid.nZ`.
4 In the **Y-component** text field, type `solid.SinsXY*solid.nX+solid.SinsYY*solid.nY+solid.SinsYZ*solid.nZ`.
5 In the **Z-component** text field, type `solid.SinsXZ*solid.nX+solid.SinsXZ*solid.nY+solid.SinsZZ*solid.nZ`.
6 Locate the **Coloring and Style** section. From the **Arrow base** list, choose **Head**.

Color Expression 1

1 In the **In Situ Stress** toolbar, click **Color Expression**.
2 In the **Settings** window for **Color Expression**, locate the **Expression** section.
3 From the **Color data** list, choose **Arrow length**.
4 From the **Unit** list, choose **kPa**.
5 Locate the **Coloring and Style** section. From the **Coloring** list, choose **Gradient**.
6 From the **Top color** list, choose **Red**.
7 In the **In Situ Stress** toolbar, click **Plot**.