Created in COMSOL Multiphysics 5.6

# 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.

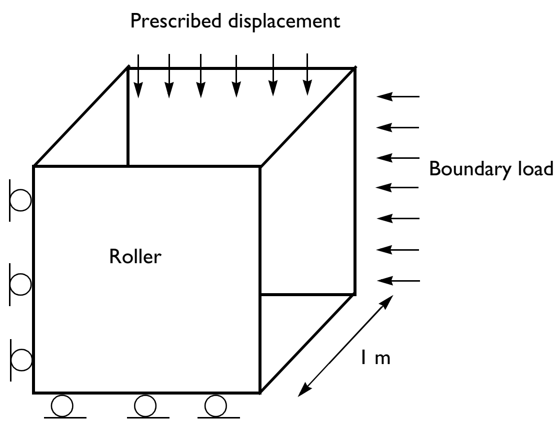


*Figure 1: Dimensions, boundary conditions, and loads for the test.*

### 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 $z = 0$).
- On the other two vertical walls ($x = 1$ m and $y = 1$ m), apply boundary loads of 300 kPa and 200 kPa, as shown on Figure 2.
- The in-situ stress is prescribed via the External Stress node.
- 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.

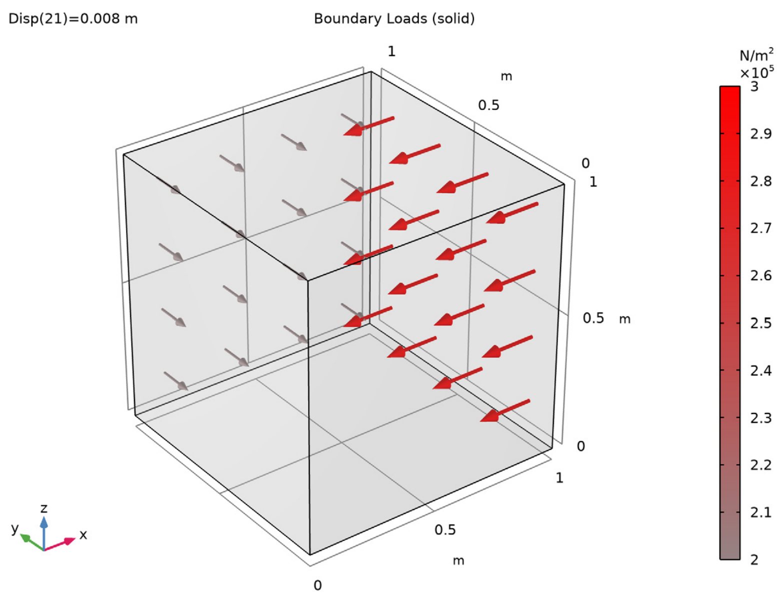


*Figure 2: Boundary loads applied on the block.*

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

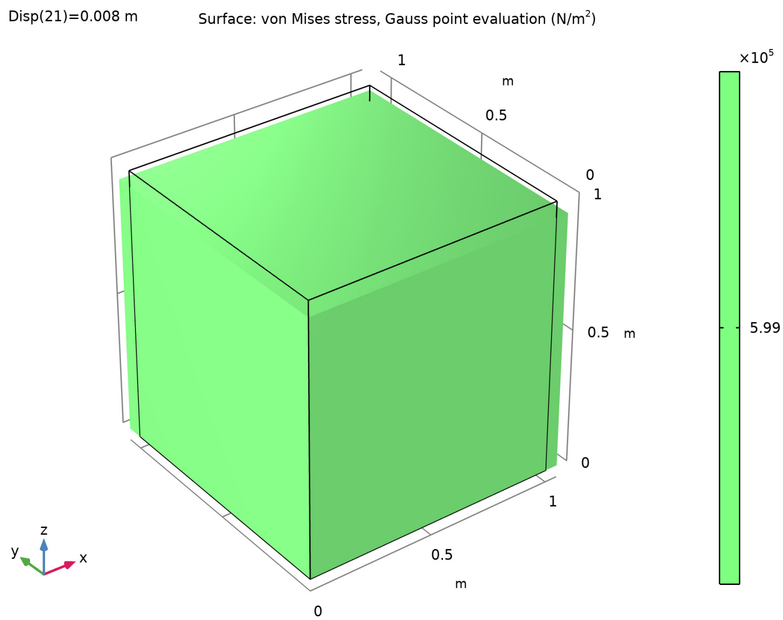


*Figure 3: Equivalent stress and deformation in the soil sample after applying 8 mm displacement from the top.*

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

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

The stresses history together with the analytical value of the stress in the z direction at the onset of yielding is shown in Figure 4. Plastic yielding is reached after a deflection of about 3.5 mm.

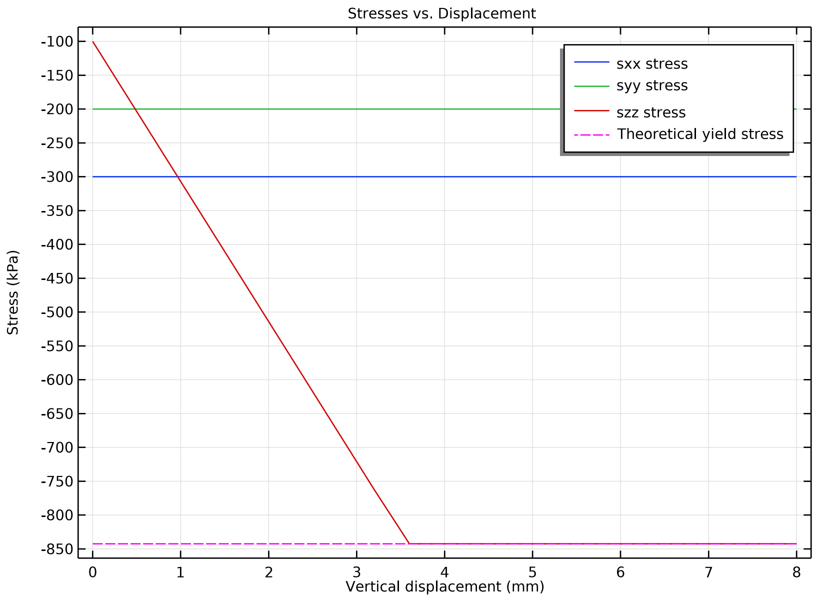


*Figure 4: This plot shows how the soil sample behaves elastically until it reaches the yield surface at the compressive meridian.*

---

**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` | 0 | |
| `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.

## 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 **Yield criterion** 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**.

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

**3** From the list, choose **Diagonal**.

**4** In the $S_{ext}$ 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.

*Boundary Load 1*

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

**2** Select Boundary 6 only.

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

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

| | |
|---|---|
| X_stress | x |
| 0 | y |
| 0 | z |

*Boundary Load 2*

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

**2** Select Boundary 5 only.

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

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

| | |
|---|---|
| 0 | x |
| Y_stress | y |
| 0 | z |

*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 | Basic |
| Poisson's ratio | nu | 0.3 | 1 | Basic |
| 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 | `8[mm]*range(0,0.05,1)` | m |

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

## RESULTS

*Stress (solid)*

The default plot shows the von Mises stress for the final step.

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

7 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.sx - Stress tensor, x 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*1000`.

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

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

9 In the table, enter the following settings:

| Legends |
|---|
| sxx stress |

10 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.sy - Stress tensor, y 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.sz - Stress tensor, z 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**.

The analytical yield level is given by Equation 1.

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

3 In the **Expression** text field, type `(2*solid.cohesion*cos(solid.internalphi)-Y_stress*(1+sin(solid.internalphi)))/(sin(solid.internalphi)-1)`.

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` |

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