Model created in COMSOL Multiphysics 6.4

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.

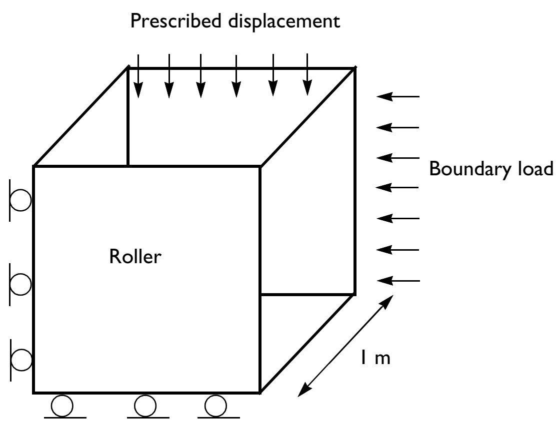

Figure 1: Dimensions, boundary conditions, and loads for the test.

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 nonassociated 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 in 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.

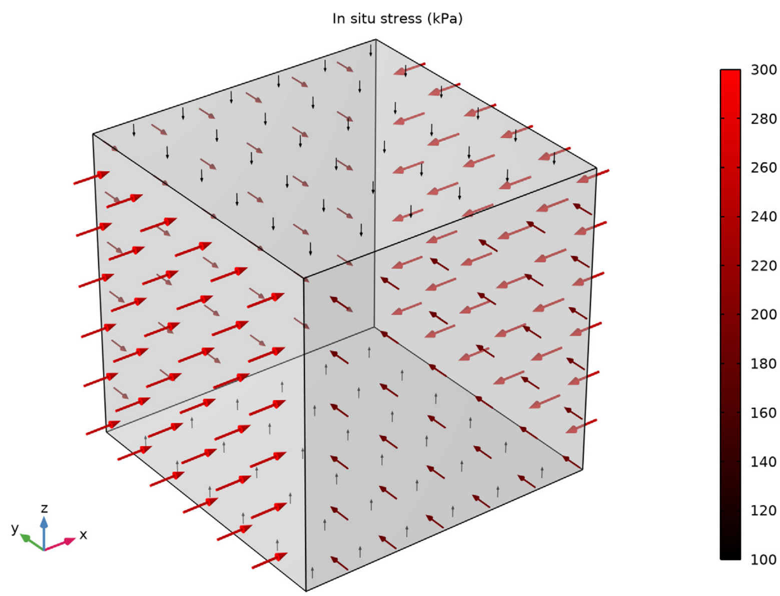

Figure 2: In situ stress applied on the surface of the block.

Results and Discussion

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

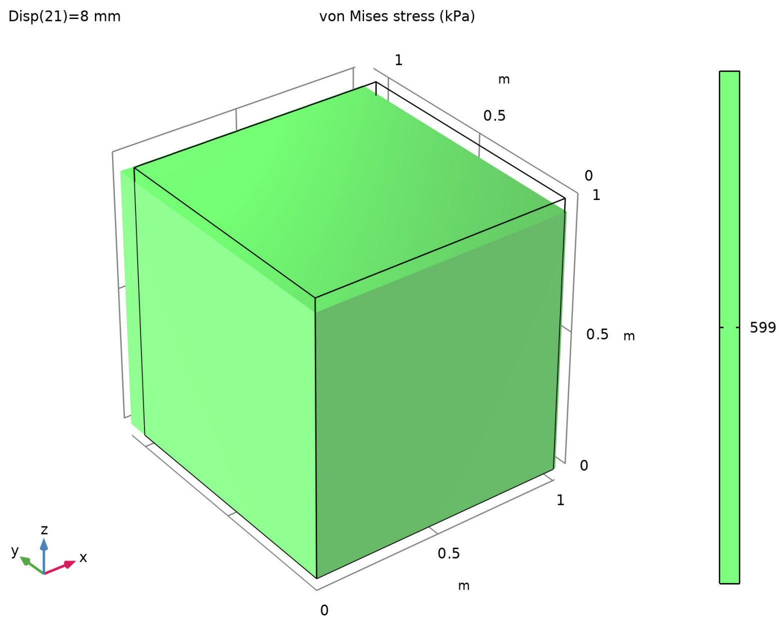

Figure 3: Equivalent stress and deformation in the soil sample after applying 8 mm displacement from the top.

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

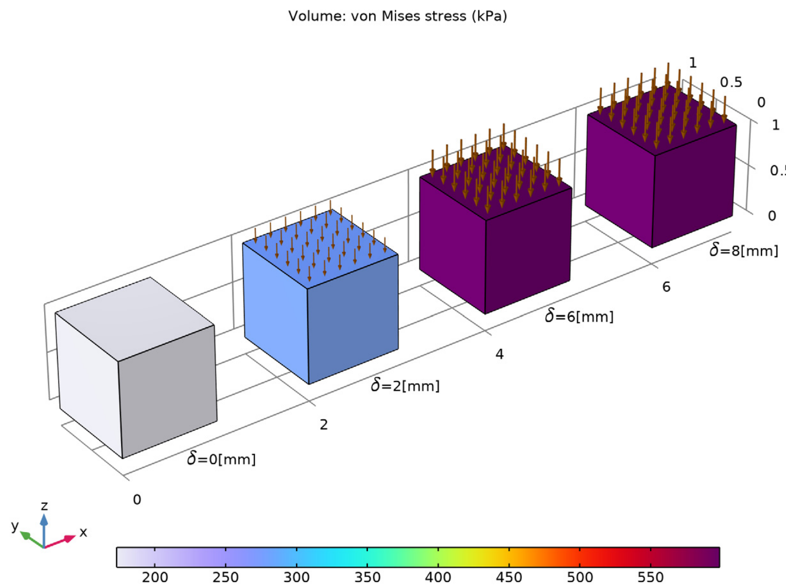

Figure 4: Equivalent stress with compression displacement of 0 mm, 2 mm, 6 mm, and 8 mm.

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.

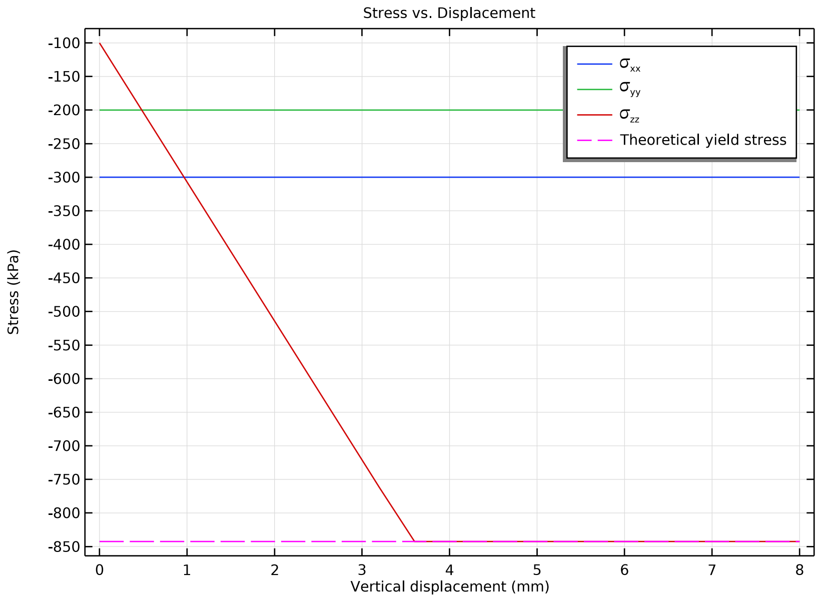

Figure 5: This plot shows how the soil sample behaves elastically until it reaches the yield surface at the compressive meridian.

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	-300[kPa]	-3E5 Pa	In situ stress, xx-component
Y_stress	-200[kPa]	-2E5 Pa	In situ stress, yy-component
Z_stress	-100[kPa]	-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/m²	Axial force
szz_th	(2*solid.cohesion* cos(solid.phis)- Y_stress*(1+ sin(solid.phis)))/ (sin(solid.phis)-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 Model** section.
3 From the F_f list, choose **Mohr–Coulomb**.

   The **Mohr–Coulomb** criterion is used to define yield surface.

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 Specify the σ_{ins} matrix as

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 From the **Displacement in z direction** list, choose **Prescribed**.

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'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
Density	rho	2000	kg/m³	Basic
Initial cohesion	cohesion0	`70e3`	Pa	Soil material
Friction angle	phis	`30[deg]`	rad	Soil material

MESH 1

Mapped 1

1 In the **Mesh** toolbar, click **More Generators** 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** checkbox.

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 **Study** toolbar, click **Compute**.

Set default units for result presentation.

RESULTS

Preferred Units 1

1 In the **Results** toolbar, click **Configurations** and choose **Preferred Units**.

2 In the **Settings** window for **Preferred Units**, locate the **Units** section.

3 Click **Add Physical Quantity**.

4 In the **Physical Quantity** dialog, select **Solid Mechanics** > **Stress tensor (N/m^2)** in the tree.

5 Click **OK**.

6 In the **Settings** window for **Preferred Units**, locate the **Units** section.

7 In the table, enter the following settings:

Quantity	Unit	Preferred unit
Stress tensor	N/m^2	kPa

8 Select the **Apply conversions to expressions with the same dimensions** checkbox.

9 Click **Apply**.

Volume 1

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

2 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** checkbox.

5 Locate the **Color Legend** section. From the **Position** list, choose **Bottom**.

Deformation

1 In the **Model Builder** window, expand the **Volume 1** node.

2 Right-click **Deformation** and choose **Delete**.

Solution Array 1

1 Right-click **Volume 1** and choose **Solution Array**.

2 In the **Settings** window for **Solution Array**, locate the **Data** section.

3 From the **Parameter selection (Disp)** list, choose **From list**.

4 In the **Parameter values (Disp (mm))** list, choose **0**, **2**, **6**, and **8**.

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 **Expression** section.

3 In the **X-component** text field, type `0`.

4 In the **Y-component** text field, type `0`.

5 In the **Z-component** text field, type `Forcez`.

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

7 Locate the **Arrow Positioning** section. In the **Number of arrows** text field, type 40.

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

9 From the **Color** list, choose **Custom**.

10 On Windows, click the colored bar underneath, or — if you are running the cross-platform desktop — the **Color** button.

11 Click **Define custom colors**.

12 Set the RGB values to 128, 64, and 0, respectively.

13 Click **Add to custom colors**.

14 Click **Show color palette only** or **OK** on the cross-platform desktop.

15 Select the **Scale factor** checkbox. In the associated text field, type 4E-4.

16 Click to expand the **Plot Array** section. Select the **Manual indexing** checkbox.

Selection 1

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

2 Select Boundary 4 only.

Solution Array 1

1 In the **Model Builder** window, right-click **Arrow Surface 1** and choose **Solution Array**.

2 In the **Settings** window for **Solution Array**, locate the **Data** section.

3 From the **Parameter selection (Disp)** list, choose **From list**.

4 In the **Parameter values (Disp (mm))** list, choose **0**, **2**, **6**, and **8**.

Stress (solid)

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

2 In the **Settings** window for **3D Plot Group**, click to expand the **Plot Array** section.

3 In the **Relative padding** text field, type 1.

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** checkbox.

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

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.

Stress vs. Displacement

1 In the **Results** toolbar, click **1D Plot Group**.

2 In the **Settings** window for **1D Plot Group**, type `Stress 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** checkbox. In the associated text field, type `Vertical displacement (mm)`.

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

Point Graph 1

1 Right-click **Stress 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/m²** > **solid.sGpxx - Stress tensor, xx-component**.

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

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

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

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

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

9 In the table, enter the following settings:

Legends
\sigma_{xx}

10 In the **Stress 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.sGpyy - Stress tensor, yy-component**.

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

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

Legends
\sigma_{yy}

Point Graph 3

1 In the **Model Builder** window, under **Results** > **Stress 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.sGpzz - Stress tensor, zz-component**.

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

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

Legends
\sigma_{zz}

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`

Stress vs. Displacement

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

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

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

In Situ Stress

1 In the **Results** toolbar, click **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 Find the **Transparency** subsection. 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 Locate the **Coloring and Style** section. From the **Coloring** list, choose **Gradient**.
5 From the **Top color** list, choose **Red**.
6 In the **In Situ Stress** toolbar, click **Plot**.