



Model created in COMSOL Multiphysics 6.4

# Deep Excavation

## Introduction

---

There are two ways to model an excavation in COMSOL Multiphysics, both of which include a parametric sweep. One option involves removing the soil one step at a time by means of a sweep of the geometry. As the soil is removed, the support it supplies is removed as well, subjecting the retaining wall to soil stresses from the nonexcavated side. The other option is to start with the already excavated geometry, and simulate the soil removal by modifying a boundary load. The boundary load applies a force on the nonexcavated side of the retaining wall, equal to the *in situ* stress on the excavation side, for the part of the wall that is above the virtual excavation depth.

This deep excavation example is inspired by a benchmark exercise specified by a working group of the German Society for Geotechnics (Ref. 1 and Ref. 2). In the example, a 26 meter excavation is modeled by means of a parametric sweep. As the excavation deepens, three struts are activated using a ramp function and boolean expressions. As the excavation reaches their depths, the struts are activated as long as the horizontal wall deflection is greater than what it is allowed to be.

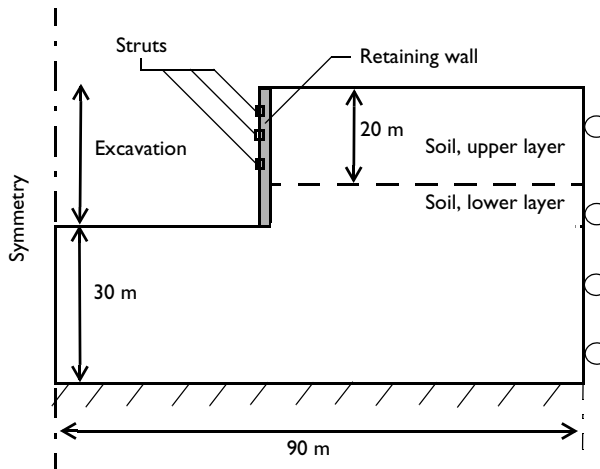


Figure 1: Dimensions and boundary conditions for the deep excavation example.

## Model Definition

---

In this example, a Drucker–Prager criterion is used for studying the soil plasticity, and the retaining wall is made of a linear elastic material. The following material parameters are used:

#### **SOIL, UPPER LAYER**

- Young's modulus  $E = 20$  MPa, Poisson's ratio  $\nu = 0.3$ , and density  $\rho = 1900$  kg/m<sup>3</sup>.
- Cohesion  $c = 0$  Pa, and angle of internal friction  $\phi = 35^\circ$ .

#### **SOIL, LOWER LAYER**

- Young's modulus  $E = 60$  MPa, Poisson's ratio  $\nu = 0.3$ , and density  $\rho = 1900$  kg/m<sup>3</sup>.
- Cohesion  $c = 0$  Pa, and angle of internal friction  $\phi = 35^\circ$ .

#### **STRUTS**

- Young's modulus  $E = 200$  GPa, length  $l = 30$  m and cross sectional area  $A = 15$  cm<sup>2</sup>.

#### **RETAINING WALL**

- Young's modulus  $E = 30$  GPa, Poisson's ratio  $\nu = 0.15$ , and density  $\rho = 2400$  kg/m<sup>3</sup>.

#### **CONSTRAINTS AND LOADS**

- The bottom soil layer is supported by a rigid and perfectly rough base. Therefore, apply a fixed constraint on the lower horizontal boundary.
- Due to symmetry, model only the right half of the domain. Use the symmetry boundary condition at the left vertical boundary.
- Add *in situ* stress with the External Stress feature. Note that the stress caused by gravity is compressive, which in the convention used in the Structural Mechanics Module means negative sign.
- The *in situ* stress accounts for the local stress prior excavating. You can define a different value in the vertical and horizontal directions. The horizontal *in situ* stress,  $X\_stress$ , is also used on the boundary load applied to the retaining wall.
- The boundary load on the retaining wall is gradually added in order to simulate the excavation steps. At the bottom of the excavation it ensures zero initial displacement. This strategy avoids remeshing the excavated volume and thus saves memory.
- Add struts that can be activated and deactivated according to the excavation depth by means of boolean expressions.
- The struts are active after a maximum horizontal deflection is reached. Use a ramp function to restrict the allowed wall deflection  $U\_max$  to 25 mm, and to supply the axial force that the struts apply to the wall.

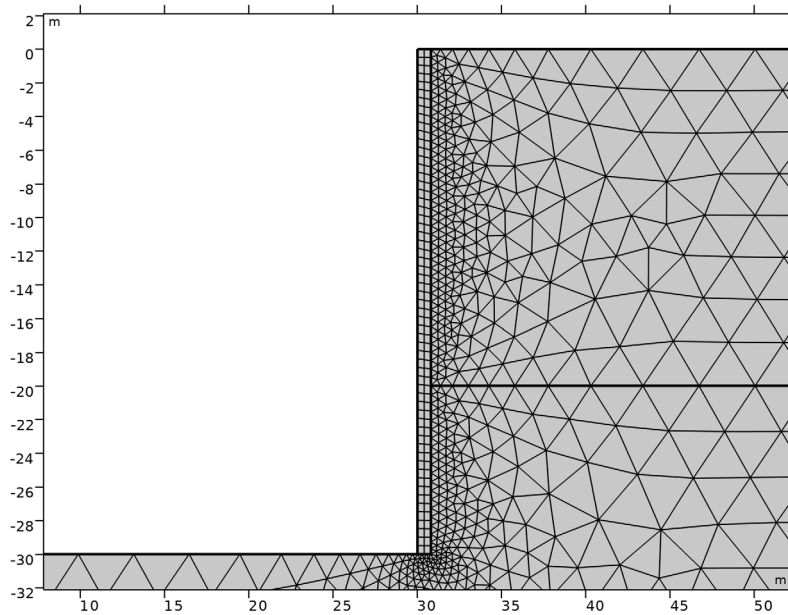
- The boundaries between the retaining wall and the soil are modeled with the Interior Contact feature. This feature constrains the normal displacement between the retaining wall and the soil to stay in contact, while the tangential displacement is unconstrained.
- The axial stiffness of the struts is estimated from their cross sectional area  $A$ , the length  $l$ , and the material Young's modulus  $E$  as

$$S = E \frac{A}{l}$$

## *Results and Discussion*

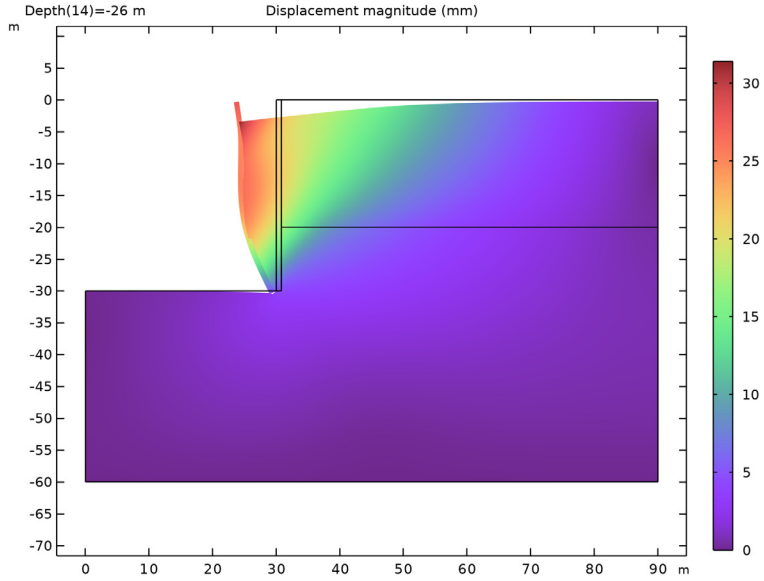
---

In order to improve convergence on the soil-wall boundary, apply a mapped mesh on the retaining wall domain, as shown in [Figure 2](#).

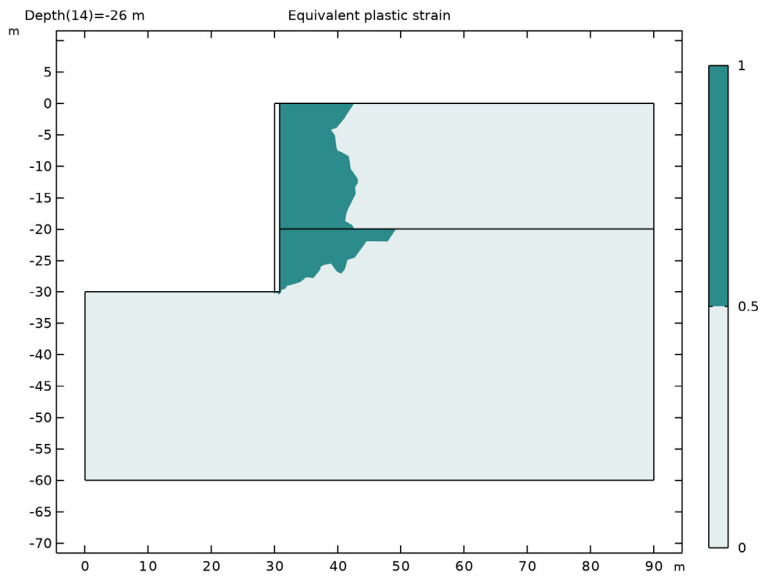


*Figure 2: A mapped mesh on the retaining wall domain gives a better convergence.*

[Figure 3](#) shows the soil deformation and the wall deflection after excavating 26 meters. [Figure 4](#) shows that the different properties of the two soil layers have an impact on the plastic deformation.



*Figure 3: Deformation of the soil and the retaining wall after excavating 26 meters of soil.*



*Figure 4: Plastic deformation after excavating 26 meters of soil.*

The struts — placed 4.8 m, 9.3 m, and 14.35 m below the initial surface (Ref. 1) — help to increase the overall stiffness of the retaining wall. In Figure 5, a maximum allowed displacement of 2.5 cm constrains the horizontal displacement of the retaining wall.

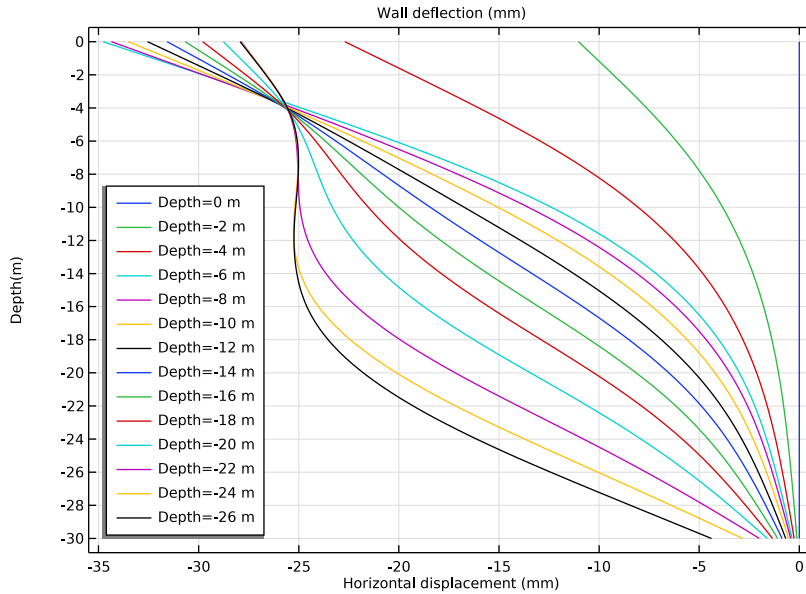


Figure 5: Wall deflection as a function of depth for different excavation steps (color lines).

As the excavation progresses, it is possible to observe the settlement of the unexcavated region. As expected, the settlement increases as the excavation progresses, and decreases with the distance to the retaining wall, see Figure 6.

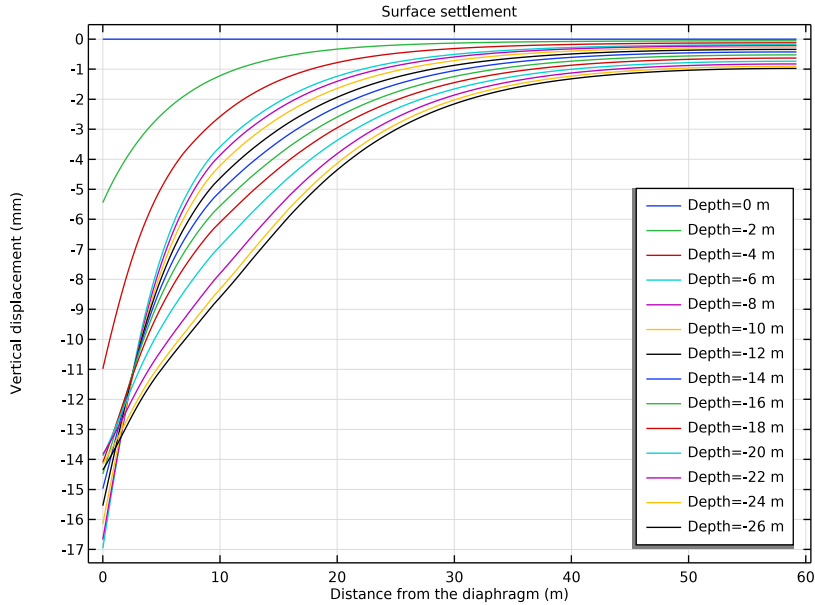


Figure 6: Surface settlement as a function of the distance from the wall. The different color lines show the settlement for different excavation stages (0 to 26 m).

## References

1. H.F. Schweiger, *Benchmarking in Geotechnics 1*. Technical Report CGG IR006 2002, Institute for Soil Mechanics and Foundation Engineering, Graz University of Technology, Austria.
2. D. Potts and L. Zdravkovic, *Finite Element Analysis in Geotechnical Engineering*, Thomas Telford Publishing, 2001.

---

**Application Library path:** Geomechanics\_Module/Soil/deep\_excavation

---



## 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**.


## GLOBAL DEFINITIONS

### *Parameters I*

- 1 In the **Model Builder** window, under **Global Definitions** click **Parameters I**.
- 2 In the **Settings** window for **Parameters**, locate the **Parameters** section.
- 3 Click  **Load from File**.
- 4 Browse to the model's Application Libraries folder and double-click the file `deep_excavation_parameters.txt`.

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

### *Ramp: Strut Force*

- 1 In the **Home** toolbar, click  **Functions** and choose **Global > Ramp**.
- 2 In the **Settings** window for **Ramp**, type Ramp: Strut Force in the **Label** text field.
- 3 In the **Function name** text field, type `strut_force`.
- 4 Locate the **Parameters** section. In the **Location** text field, type `U_max`.
- 5 In the **Slope** text field, type `S_struts`.
- 6 In the **Baseline** text field, type `0[N]`.
- 7 Click to expand the **Smoothing** section.
- 8 Select the **Size of transition zone at start** checkbox. In the associated text field, type `0.1 [mm]`.


## GEOMETRY I

Create the geometry.



To simplify this step, you can insert a prepared geometry sequence. In the **Geometry** toolbar, click **Insert Sequence**. Browse to the model's Application Libraries folder and double-click the file `deep_excavation.mph`. Click **Build All** in the **Geometry** toolbar. Then, continue with the instruction after the geometry plot below.

Otherwise, proceed with the following instructions to create the geometry from scratch:



#### *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 90.
- 4 In the **Height** text field, type 60.
- 5 Locate the **Position** section. In the **y** text field, type -60.
- 6 Click to expand the **Layers** section. In the table, enter the following settings:


Layer name	Thickness (m)
Layer 1	20

- 7 Clear the **Layers on bottom** checkbox.
- 8 Select the **Layers on top** checkbox.
- 9 Click  **Build Selected**.
- 10 Click  **Build Selected**.

#### *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 30.
- 4 In the **Height** text field, type 30.
- 5 Locate the **Position** section. In the **y** text field, type -30.
- 6 Click  **Build Selected**.

#### *Rectangle 3 (r3)*

- 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 0.8.
- 4 In the **Height** text field, type 30.
- 5 Locate the **Position** section. In the **x** text field, type 30.

6 In the **y** text field, type -30.

7 Click  **Build Selected**.

#### *Difference 1 (dif1)*

1 In the **Geometry** toolbar, click  **Booleans and Partitions** and choose **Difference**.

2 Select the object **r1** only.

3 In the **Settings** window for **Difference**, locate the **Difference** section.

4 Click to select the  **Activate Selection** toggle button for **Objects to subtract**.

5 Select the objects **r2** and **r3** only.

6 Select the **Keep objects to subtract** checkbox.

7 Click  **Build Selected**.

#### *Delete Entities 1 (del1)*

1 In the **Model Builder** window, right-click **Geometry 1** and choose **Delete Entities**.

2 In the **Settings** window for **Delete Entities**, locate the **Entities or Objects to Delete** section.

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

4 Click the  **Clear Selection** button for **Selection**.

5 Select the object **r2** only.

6 Click  **Build Selected**.

#### *Line Segment 1 (ls1)*

1 In the **Geometry** toolbar, click  **More Primitives** and choose **Line Segment**.

2 In the **Settings** window for **Line Segment**, locate the **Starting Point** section.

3 From the **Specify** list, choose **Coordinates**.

4 Locate the **Endpoint** section. From the **Specify** list, choose **Coordinates**.

5 Locate the **Starting Point** section. In the **x** text field, type 30.

6 Locate the **Endpoint** section. In the **x** text field, type 30.

7 Locate the **Starting Point** section. In the **y** text field, type Stage\_1+1.


8 Locate the **Endpoint** section. In the **y** text field, type Stage\_1.

9 Click  **Build Selected**.



#### *Line Segment 2 (ls2)*

1 In the **Geometry** toolbar, click  **More Primitives** and choose **Line Segment**.


2 In the **Settings** window for **Line Segment**, locate the **Starting Point** section.

- 3 From the **Specify** list, choose **Coordinates**.
- 4 Locate the **Endpoint** section. From the **Specify** list, choose **Coordinates**.
- 5 Locate the **Starting Point** section. In the **x** text field, type 30.
- 6 Locate the **Endpoint** section. In the **x** text field, type 30.
- 7 Locate the **Starting Point** section. In the **y** text field, type Stage\_2+1.
- 8 Locate the **Endpoint** section. In the **y** text field, type Stage\_2.
- 9 Click  **Build Selected**.

#### *Line Segment 3 (ls3)*


- 1 In the **Geometry** toolbar, click  **More Primitives** and choose **Line Segment**.
- 2 In the **Settings** window for **Line Segment**, locate the **Starting Point** section.
- 3 From the **Specify** list, choose **Coordinates**.
- 4 Locate the **Endpoint** section. From the **Specify** list, choose **Coordinates**.
- 5 Locate the **Starting Point** section. In the **x** text field, type 30.
- 6 Locate the **Endpoint** section. In the **x** text field, type 30.
- 7 Locate the **Starting Point** section. In the **y** text field, type Stage\_3+1.
- 8 Locate the **Endpoint** section. In the **y** text field, type Stage\_3.
- 9 Click  **Build Selected**.

#### *Form Union (fin)*

In the **Geometry** toolbar, click  **Build All**.

## **DEFINITIONS**

#### *Wall Diaphragm*

- 1 In the **Definitions** toolbar, click  **Explicit**.
- 2 In the **Settings** window for **Explicit**, type Wall Diaphragm in the **Label** text field.
- 3 Locate the **Input Entities** section. From the **Geometric entity level** list, choose **Boundary**.
- 4 Select Boundaries 13 and 14 only.

#### *Soil Domain*


- 1 In the **Definitions** toolbar, click  **Explicit**.
- 2 Select Domains 1 and 3 only.
- 3 In the **Settings** window for **Explicit**, type Soil Domain in the **Label** text field.

## 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 **Match to Mohr–Coulomb** list, choose **Inscribe (plane strain)**.
- 4 Locate the **Domain Selection** section. From the **Selection** list, choose **Soil Domain**.

### *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 **Domain Selection** section.
- 3 From the **Selection** list, choose **Soil Domain**.
- 4 Locate the **External Stress** section. From the **Stress input** list, choose **In situ stress**.
- 5 From the list, choose **Diagonal**.
- 6 Specify the  $\sigma_{\text{ins}}$  matrix as

X_stress	0	0
0	Y_stress	0
0	0	Z_stress


### *Symmetry 1*

- 1 In the **Physics** toolbar, click  **Boundaries** and choose **Symmetry**.
- 2 Select Boundary 1 only.


### *Fixed Constraint 1*

- 1 In the **Physics** toolbar, click  **Boundaries** and choose **Fixed Constraint**.
- 2 Select Boundary 2 only.


### *Roller 1*

- 1 In the **Physics** toolbar, click  **Boundaries** and choose **Roller**.
- 2 Select Boundaries 17 and 18 only.


*Interior Contact 1*

- 1 In the **Physics** toolbar, click  **Boundaries** and choose **Interior Contact**.
- 2 Select Boundaries 5, 13, and 14 only.

*Adhesion 1*


- 1 In the **Physics** toolbar, click  **Attributes** and choose **Adhesion**.
- 2 In the **Settings** window for **Adhesion**, locate the **Adhesive Activation** section.
- 3 From the **Activation criterion** list, choose **Always active**.
- 4 Locate the **Adhesive Stiffness** section. In the  $n_\tau$  text field, type 0.

*Boundary Load 1*

- 1 In the **Physics** toolbar, click  **Boundaries** and choose **Boundary Load**.
- 2 In the **Settings** window for **Boundary Load**, locate the **Boundary Selection** section.
- 3 From the **Selection** list, choose **Wall Diaphragm**.
- 4 Locate the **Force** section. Specify the  $\mathbf{f}_A$  vector as


X_stress*(y>Depth)	x
0	y

*Boundary Load: Strut 1*

- 1 In the **Physics** toolbar, click  **Boundaries** and choose **Boundary Load**.
- 2 In the **Settings** window for **Boundary Load**, type Boundary Load: Strut 1 in the **Label** text field.
- 3 Select Boundary 10 only.
- 4 Locate the **Force** section. From the **Load type** list, choose **Total force**.
- 5 Specify the  $\mathbf{F}_{tot}$  vector as

strut_force(abs(u))*(Depth<Stage_1)	x
0	y


*Boundary Load: Strut 2*

- 1 In the **Physics** toolbar, click  **Boundaries** and choose **Boundary Load**.
- 2 In the **Settings** window for **Boundary Load**, type Boundary Load: Strut 2 in the **Label** text field.
- 3 Select Boundary 8 only.
- 4 Locate the **Force** section. From the **Load type** list, choose **Total force**.

5 Specify the  $\mathbf{F}_{\text{tot}}$  vector as

strut_force(abs(u))*(Depth<Stage_2)	x
0	y


*Boundary Load: Strut 3*

- 1 In the **Physics** toolbar, click  **Boundaries** and choose **Boundary Load**.
- 2 In the **Settings** window for **Boundary Load**, type Boundary Load: Strut 3 in the **Label** text field.
- 3 Select Boundary 6 only.
- 4 Locate the **Force** section. From the **Load type** list, choose **Total force**.
- 5 Specify the  $\mathbf{F}_{\text{tot}}$  vector as

strut_force(abs(u))*(Depth<Stage_3)	x
0	y

## MATERIALS

*Soil, Upper Layer*

- 1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Materials** and choose **Blank Material**.
- 2 In the **Settings** window for **Material**, type Soil, Upper Layer in the **Label** text field.
- 3 Locate the **Geometric Entity Selection** section. From the **Selection** list, choose **Manual**.
- 4 Click  **Clear Selection**.
- 5 Select Domain 3 only.
- 6 Locate the **Material Contents** section. In the table, enter the following settings:

Property	Variable	Value	Unit	Property group
Young's modulus	E	20 [MPa]	Pa	Young's modulus and Poisson's ratio
Poisson's ratio	nu	0.3	l	Young's modulus and Poisson's ratio
Density	rho	1900	kg/m <sup>3</sup>	Basic
Initial cohesion	cohesion0	0	Pa	Soil material
Friction angle	phis	35 [deg]	rad	Soil material

### Soil, Lower Layer

- 1 Right-click **Materials** and choose **Blank Material**.
- 2 In the **Settings** window for **Material**, type Soil, Lower Layer in the **Label** text field.
- 3 Select Domain 1 only.
- 4 Locate the **Material Contents** section. In the table, enter the following settings:

Property	Variable	Value	Unit	Property group
Young's modulus	E	60 [MPa]	Pa	Young's modulus and Poisson's ratio
Poisson's ratio	nu	0.3	l	Young's modulus and Poisson's ratio
Density	rho	1900	kg/m <sup>3</sup>	Basic
Initial cohesion	cohesion0	0	Pa	Soil material
Friction angle	phis	35 [deg]	rad	Soil material


### Retaining Wall

- 1 Right-click **Materials** and choose **Blank Material**.
- 2 In the **Settings** window for **Material**, type Retaining Wall in the **Label** text field.
- 3 Select Domain 2 only.
- 4 Locate the **Material Contents** section. In the table, enter the following settings:

Property	Variable	Value	Unit	Property group
Young's modulus	E	30 [GPa]	Pa	Young's modulus and Poisson's ratio
Poisson's ratio	nu	0.15	l	Young's modulus and Poisson's ratio
Density	rho	2400	kg/m <sup>3</sup>	Basic

## MESH 1

### Free Triangular 1

- 1 In the **Mesh** toolbar, click  **Free Triangular**.
- 2 In the **Settings** window for **Free Triangular**, locate the **Domain Selection** section.
- 3 From the **Geometric entity level** list, choose **Domain**.
- 4 From the **Selection** list, choose **Soil Domain**.

### *Distribution 1*

- 1 Right-click **Free Triangular 1** and choose **Distribution**.
- 2 Select Boundary 13 only.
- 3 In the **Settings** window for **Distribution**, locate the **Distribution** section.
- 4 In the **Number of elements** text field, type 20.

### *Distribution 2*

- 1 In the **Model Builder** window, right-click **Free Triangular 1** and choose **Distribution**.
- 2 Select Boundary 14 only.
- 3 In the **Settings** window for **Distribution**, locate the **Distribution** section.
- 4 In the **Number of elements** text field, type 40.



### *Distribution 3*

- 1 Right-click **Free Triangular 1** and choose **Distribution**.
- 2 Select Boundary 5 only.
- 3 In the **Settings** window for **Distribution**, locate the **Distribution** section.
- 4 In the **Number of elements** text field, type 3.

### *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**.

### *Mapped 1*

- 1 In the **Mesh** toolbar, click  **Mapped**.  
Use a mapped mesh inside the wall.
- 2 In the **Settings** window for **Mapped**, click  **Build All**.  
The mesh should look like the one in [Figure 2](#).

## **STUDY 1**

### *Step 1: Stationary*

Set up an auxiliary continuation sweep for the Depth 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
Depth (Excavation depth (parameter))	range (0, -2, -26)	m

#### *Solution I (sol1)*

- 1 In the **Study** toolbar, click  **Show Default Solver**.
- 2 In the **Model Builder** window, expand the **Solution I (sol1)** node.
- 3 In the **Model Builder** window, expand the **Study I > Solver Configurations > Solution I (sol1) > Stationary Solver I** node, then click **Parametric I**.
- 4 In the **Settings** window for **Parametric**, click to expand the **Continuation** section.  
Restrict the step size to 0.5 to better capture the development of the plastic zone.
- 5 Select the **Tuning of step size** checkbox.
- 6 In the **Initial step size** text field, type 0.5.
- 7 In the **Maximum step size** text field, type 0.5.

#### *Solution I (sol1)*


- 1 In the **Model Builder** window, expand the **Study I > Solver Configurations > Solution I (sol1) > Dependent Variables I** node, then click **Displacement Field (compl.u)**.
- 2 In the **Settings** window for **Field**, locate the **Scaling** section.
- 3 In the **Scale** text field, type  $1e-2$ .
- 4 In the **Model Builder** window, under **Study I > Solver Configurations > Solution I (sol1) > Stationary Solver I** click **Fully Coupled I**.
- 5 In the **Settings** window for **Fully Coupled**, click to expand the **Method and Termination** section.
- 6 From the **Nonlinear method** list, choose **Automatic (Newton)**.
- 7 In the **Study** toolbar, click  **Compute**.

Set default units for result presentation.

## RESULTS

#### *Preferred Units I*

- 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 **General > Displacement (m)** 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
Displacement	m	mm



- 8 Click  **Apply**.

#### *Displacement*



The default plot shows the von Mises stress for the last value of the continuation parameter. Modify and rename this plot group to display the displacement instead.

- 1 In the **Model Builder** window, under **Results** click **Stress (solid)**.
- 2 In the **Settings** window for **2D Plot Group**, type Displacement in the **Label** text field.

#### *Surface 1*

- 1 In the **Model Builder** window, expand the **Displacement** node, then click **Surface 1**.
- 2 In the **Settings** window for **Surface**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1 (comp1) > Solid Mechanics > Displacement > solid.disp - Displacement magnitude - m**.
- 3 Locate the **Coloring and Style** section. From the **Color table** list, choose **SpectrumLight**.
- 4 In the **Displacement** toolbar, click  **Plot**.
- 5 Click the  **Zoom Extents** button in the **Graphics** toolbar.

#### **RESULT TEMPLATES**


- 1 In the **Results** toolbar, click  **Result Templates** to open the **Result Templates** window.
- 2 Go to the **Result Templates** window.
- 3 In the tree, select **Study 1/Solution 1 (sol1) > Solid Mechanics > Equivalent Plastic Strain (solid)**.
- 4 Click the **Add Result Template** button in the window toolbar.
- 5 In the **Results** toolbar, click  **Result Templates** to close the **Result Templates** window.

## RESULTS


### *Plastic Region*

In the **Settings** window for **2D Plot Group**, type Plastic Region in the **Label** text field.


### *Surface 1*

- 1 In the **Model Builder** window, expand the **Plastic Region** node, then click **Surface 1**.
- 2 In the **Settings** window for **Surface**, locate the **Expression** section.
- 3 In the **Expression** text field, type `solid.epεp>0`.
- 4 Locate the **Coloring and Style** section. In the **Number of bands** text field, type 2.
- 5 Click to expand the **Quality** section. From the **Resolution** list, choose **Custom**.
- 6 In the **Plastic Region** toolbar, click  **Plot**.


### *Wall Deflection*

- 1 In the **Results** toolbar, click  **ID Plot Group**.
- 2 In the **Settings** window for **ID Plot Group**, type Wall Deflection in the **Label** text field.
- 3 Click to expand the **Title** section. From the **Title type** list, choose **Manual**.
- 4 In the **Title** text area, type Wall deflection (mm).
- 5 Locate the **Plot Settings** section.
- 6 Select the **x-axis label** checkbox. In the associated text field, type Horizontal displacement (mm).
- 7 Select the **y-axis label** checkbox. In the associated text field, type Depth(m).
- 8 Locate the **Legend** section. From the **Position** list, choose **Lower left**.


### *Line Graph 1*

- 1 Right-click **Wall Deflection** and choose **Line Graph**.
- 2 In the **Settings** window for **Line Graph**, locate the **Selection** section.
- 3 From the **Selection** list, choose **Wall Diaphragm**.
- 4 Locate the **y-Axis Data** section. In the **Expression** text field, type `y`.
- 5 From the **Unit** list, choose **m**.
- 6 Locate the **x-Axis Data** section. From the **Parameter** list, choose **Expression**.
- 7 In the **Expression** text field, type `u`.
- 8 Click to expand the **Legends** section. Select the **Show legends** checkbox.
- 9 Find the **Prefix and suffix** subsection. In the **Prefix** text field, type `Depth=`.
- 10 In the **Wall Deflection** toolbar, click  **Plot**.

### *Surface Settlement*


- 1 In the **Results** toolbar, click  **ID Plot Group**.
- 2 In the **Settings** window for **ID Plot Group**, type Surface Settlement in the **Label** text field.
- 3 Locate the **Title** section. From the **Title type** list, choose **Manual**.
- 4 In the **Title** text area, type Surface settlement.
- 5 Locate the **Plot Settings** section.
- 6 Select the **x-axis label** checkbox. In the associated text field, type Distance from the diaphragm (m).
- 7 Select the **y-axis label** checkbox. In the associated text field, type Vertical displacement (mm).
- 8 Locate the **Legend** section. From the **Position** list, choose **Lower right**.

### *Line Graph 1*


- 1 Right-click **Surface Settlement** and choose **Line Graph**.
- 2 Select Boundary 16 only.
- 3 In the **Settings** window for **Line Graph**, locate the **y-Axis Data** section.
- 4 In the **Expression** text field, type  $v$ .
- 5 Locate the **Legends** section. Select the **Show legends** checkbox.
- 6 Find the **Prefix and suffix** subsection. In the **Prefix** text field, type Depth=.
- 7 In the **Surface Settlement** toolbar, click  **Plot**.

Create an extruded plot of displacement.


### *Extrusion 2D 1*

- 1 In the **Results** toolbar, click  **More Datasets** and choose **Extrusion 2D**.
- 2 In the **Settings** window for **Extrusion 2D**, locate the **Extrusion** section.
- 3 In the **z maximum** text field, type 80.


### *Displacement 3D*

- 1 In the **Results** toolbar, click  **3D Plot Group**.
- 2 In the **Settings** window for **3D Plot Group**, type Displacement 3D in the **Label** text field.

### *Surface 1*

- 1 In the **Displacement 3D** toolbar, click  **Surface**.
- 2 In the **Settings** window for **Surface**, locate the **Coloring and Style** section.
- 3 From the **Color table** list, choose **SpectrumLight**.

### Deformation I

In the **Displacement 3D** toolbar, click  **Deformation**.

### Displacement 3D

Use the mouse to replicate the view of the following image.

I In the **Model Builder** window, under **Results** click **Displacement 3D**.

Depth(14)=-26 m

Displacement magnitude (mm)

