Created in COMSOL Multiphysics 5.6

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 non excavated 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 excavation side of the retaining wall, equal to (and therefore balancing) the in-situ stresses on the nonexcavated side, for the part of the wall that is below 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..

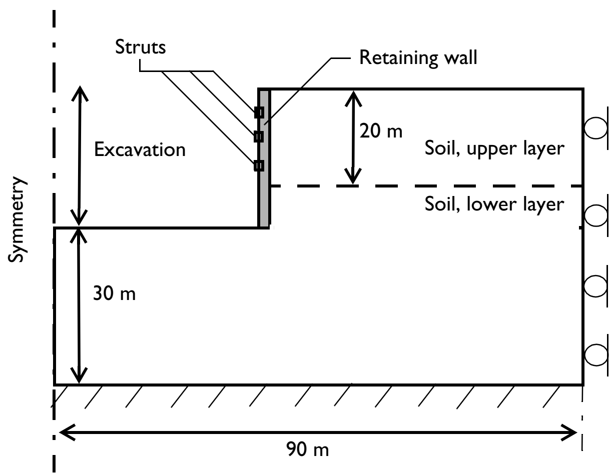


Figure 1: Dimensions and boundary conditions for the deep excavation example.

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 $\rho = 1900\ \mathrm{kg/m^3}$.
- Cohesion $c = 0$ Pa, and angle of internal friction $\phi = 35°$.

SOIL, LOWER LAYER

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

STRUTS

- Young's modulus $E = 200$ GPa, length $l = 30$ m and cross sectional area $A = 15\ \mathrm{cm^2}$.

RETAINING WALL

- Young's modulus $E = 30$ GPa, Poisson's ratio $\nu = 0.15$, and density $\rho = 2400\ \mathrm{kg/m^3}$.

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 stresses 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 stresses account 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 decreased 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 extrusion operators. These operators constrain 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.

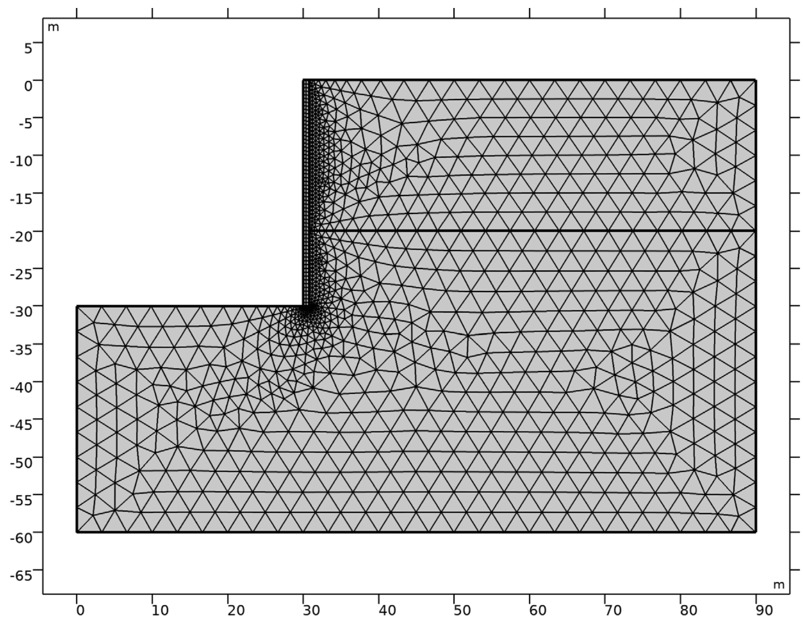


Figure 2: A mapped mesh on the retaining wall domain gives a better convergence.

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.

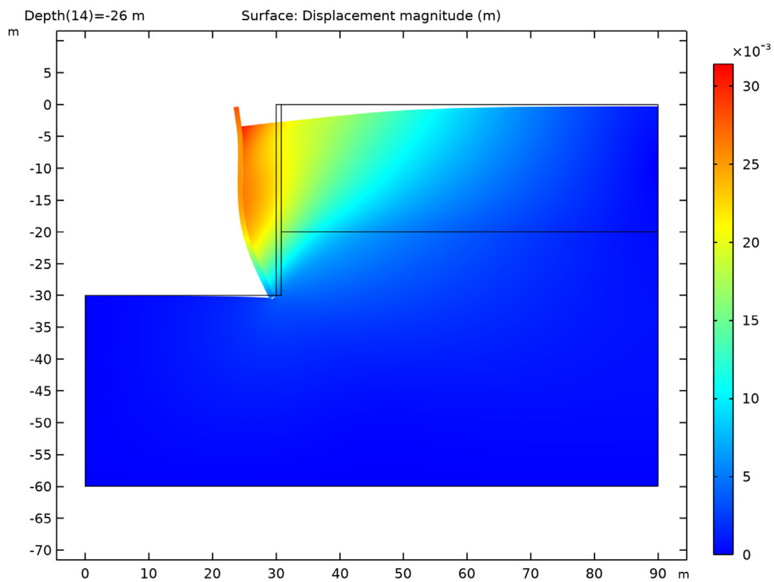


Figure 3: Deformation of the soil and the retaining wall after excavating 26 meters of soil.

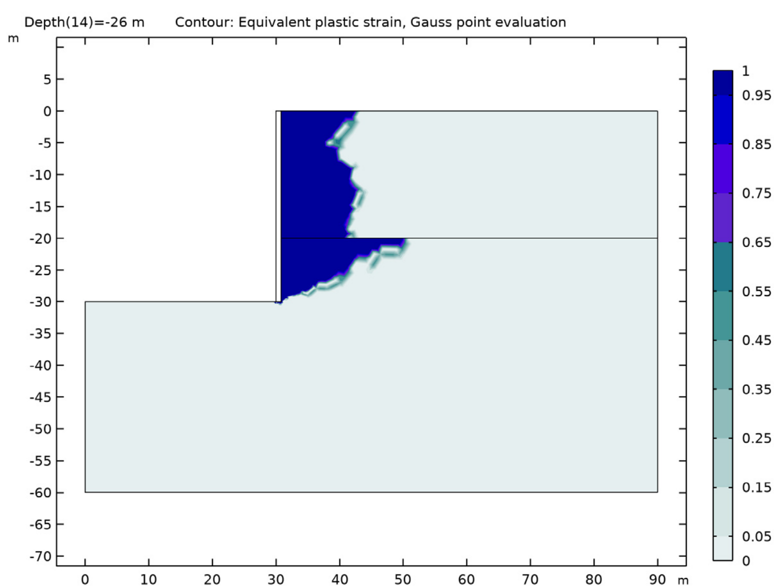


Figure 4: Plastic deformation after excavating 26 meters of soil.

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.

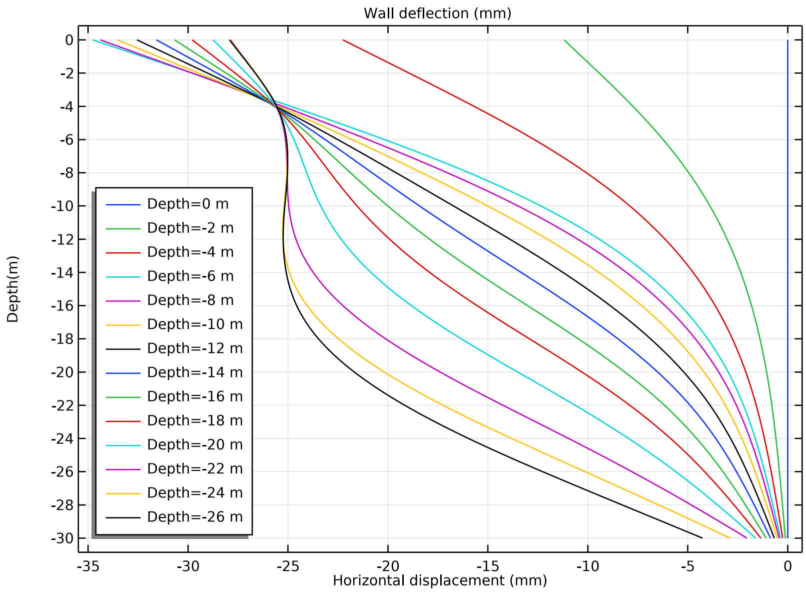


Figure 5: Wall deflection as a function of depth for different excavation steps (color lines).

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.

.

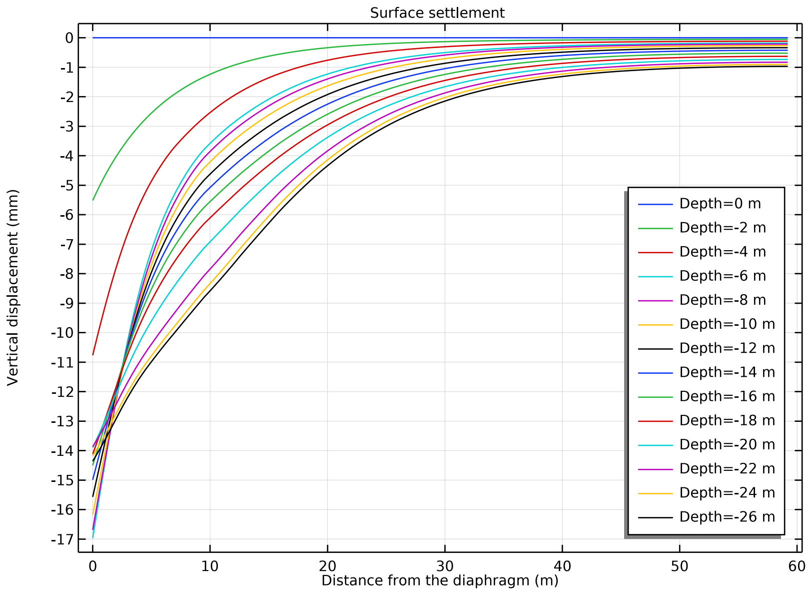


Figure 6: Surface settlement as a function of the distance from the wall. The different color lines show the settlement for different excavation stages (0 to 26 m).

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 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
X_stress	-24e3[Pa]	-24000 Pa	In-situ stress, xx component
Y_stress	-35e3[Pa]	-35000 Pa	In-situ stress, yy component
Z_stress	-24e3[Pa]	-24000 Pa	In-situ stress, zz component
E_struts	2e5[MPa]	2E11 Pa	Young's modulus of the struts
A_struts	15[cm^2]	0.0015 m²	Cross section area of the struts
l_struts	30[m]	30 m	Length of struts
S_struts	E_struts*A_struts/l_struts	1E7 N/m	Stiffness of the struts
U_max	-25[mm]	-0.025 m	Allowed wall deflection
Stage_1	-4.8[m]	-4.8 m	First excavation step, first strut
Stage_2	-9.3[m]	-9.3 m	Second excavation step, second strut
Stage_3	-14.35[m]	-14.35 m	Third excavation step, third strut
Depth	0[m]	0 m	Excavation depth (parameter)

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 1 (rm1)

1 In the **Home** toolbar, click f(X) **Functions** and choose **Global>Ramp**.

2 In the **Settings** window for **Ramp**, locate the **Parameters** section.

3 In the **Location** text field, type `-U_max`.

4 In the **Slope** text field, type `S_struts/U_max`.

5 Click to expand the **Smoothing** section. Select the **Size of transition zone at start** check box.

6 In the associated text field, type `0.0001`.

GEOMETRY 1

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 **Model Builder** window, expand the **Geometry 1** node, then click **Rectangle 1 (r1)**.
3 In the **Settings** window for **Rectangle**, locate the **Size and Shape** section.
4 In the **Width** text field, type `90`.
5 In the **Height** text field, type `60`.
6 Locate the **Position** section. In the **y** text field, type `-60`.
7 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 `90`.
4 In the **Height** text field, type `20`.
5 Locate the **Position** section. In the **y** text field, type `-20`.
6 Click **Build Selected**.

Union 1 (uni1)

1 In the **Geometry** toolbar, click **Booleans and Partitions** and choose **Union**.
2 Click in the **Graphics** window and then press Ctrl+A to select both objects.
3 In the **Settings** window for **Union**, 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 `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 4 (r4)

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 **uni1** only.

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

4 Find the **Objects to subtract** subsection. Select the **Activate Selection** toggle button.

5 Select the objects **r3** and **r4** only.

6 Select the **Keep input objects** check box.

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 **Clear Selection**.

5 Select the objects **r3** and **uni1** 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**.

Union 2 (uni2)

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

2 Select the objects **ls1**, **ls2**, **ls3**, and **r4** only.

3 In the **Settings** window for **Union**, click **Build Selected**.

Form Union (fin)

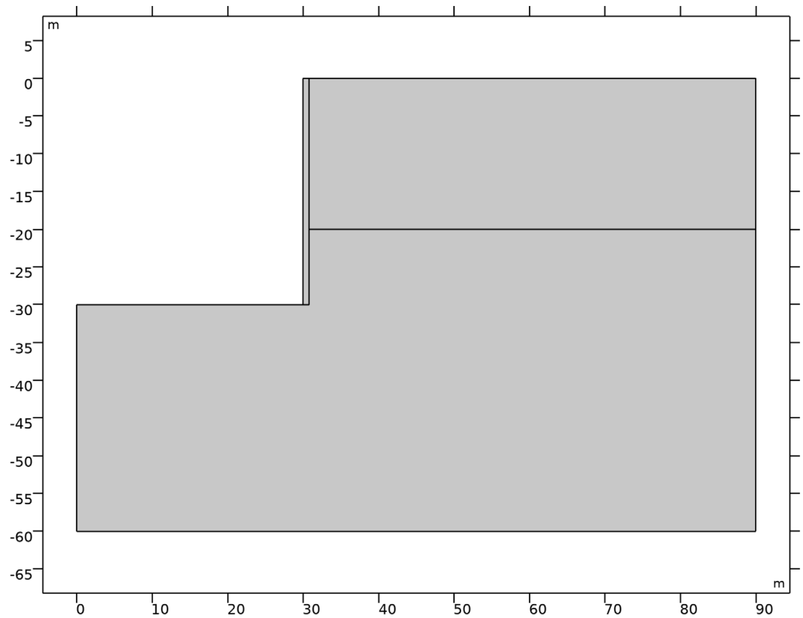
1 In the **Model Builder** window, under **Component 1 (comp1)>Geometry 1** click **Form Union (fin)**.

2 In the **Settings** window for **Form Union/Assembly**, locate the **Form Union/Assembly** section.

3 From the **Action** list, choose **Form an assembly**.

4 Clear the **Create pairs** check box.

5 Click **Build Selected**.



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 Boundary 20 only.

Wall soil

1 In the **Definitions** toolbar, click **Explicit**.

2 In the **Settings** window for **Explicit**, type `Wall soil` in the **Label** text field.

3 Locate the **Input Entities** section. From the **Geometric entity level** list, choose **Boundary**.

4 Select Boundaries 5 and 6 only.

General Extrusion 1 (genext1)

1. In the **Definitions** toolbar, click **Nonlocal Couplings** and choose **General Extrusion**.
2. In the **Settings** window for **General Extrusion**, locate the **Source Selection** section.
3. From the **Geometric entity level** list, choose **Boundary**.
4. Click **Paste Selection**.
5. In the **Paste Selection** dialog box, type `12` in the **Selection** text field.
6. Click **OK**.
7. In the **Settings** window for **General Extrusion**, locate the **Source** section.
8. From the **Source frame** list, choose **Material (X, Y, Z)**.
9. Locate the **Destination Map** section. In the **X-expression** text field, type `X`.
10. In the **Y-expression** text field, type `Y`.

General Extrusion 2 (genext2)

1. In the **Definitions** toolbar, click **Nonlocal Couplings** and choose **General Extrusion**.
2. In the **Settings** window for **General Extrusion**, locate the **Source Selection** section.
3. From the **Geometric entity level** list, choose **Boundary**.
4. From the **Selection** list, choose **Wall diaphragm**.
5. Locate the **Source** section. From the **Source frame** list, choose **Material (X, Y, Z)**.
6. Locate the **Destination Map** section. In the **X-expression** text field, type `X`.
7. In the **Y-expression** text field, type `Y`.

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. Select the **Match to Mohr-Coulomb criterion** check box.
4. Select Domains 1 and 2 only.

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

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 9 and 10 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 y direction** check box.
5. In the u_{0y} text field, type `genext1(v)`.

Prescribed Displacement 2

1. In the **Physics** toolbar, click **Boundaries** and choose **Prescribed Displacement**.
2. In the **Settings** window for **Prescribed Displacement**, locate the **Boundary Selection** section.
3. From the **Selection** list, choose **Wall soil**.
4. Locate the **Prescribed Displacement** section. Select the **Prescribed in x direction** check box.

5 In the u_{0x} text field, type `genext2(u)`.

Boundary Load 1

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

2 Select Boundaries 3, 8, and 19 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

Boundary Load 2

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

2 Select Boundaries 11 and 13–18 only.

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

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

-X_stress*(y<Depth)	x
0	y

Strut_1

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

2 In the **Settings** window for **Boundary Load**, type `Strut_1` in the **Label** text field.

3 Select Boundary 17 only.

4 Locate the **Force** section. From the **Load type** list, choose **Total force**.

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

-rm1(-u[1/m])*(Depth<Stage_1)	x
0	y

Strut_2

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

2 In the **Settings** window for **Boundary Load**, type `Strut_2` in the **Label** text field.

3 Select Boundary 15 only.

4 Locate the **Force** section. From the **Load type** list, choose **Total force**.

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

`-rm1(-u[1/m])*(Depth<Stage_2)`	x
`0`	y

Strut_3

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

2 In the **Settings** window for **Boundary Load**, type `Strut_3` in the **Label** text field.

3 Select Boundary 13 only.

4 Locate the **Force** section. From the **Load type** list, choose **Total force**.

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

`-rm1(-u[1/m])*(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 2 only.

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

Property	Variable	Value	Unit	Property group
Young'smodulus	E	`20e6`	Pa	Basic
Poisson's ratio	nu	`0.3`	1	Basic
Density	rho	`1900`	kg/m³	Basic
Cohesion	cohesion	`0`	Pa	Mohr-Coulomb
Angle of internal friction	internalphi	`35[deg]`	rad	Mohr-Coulomb

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'smodulus	E	60e6	Pa	Basic
Poisson's ratio	nu	0.3	1	Basic
Density	rho	1900	kg/m³	Basic
Cohesion	cohesion	0	Pa	Mohr-Coulomb
Angle of internal friction	internalphi	35[deg]	rad	Mohr-Coulomb

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 3 only.

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

Property	Variable	Value	Unit	Property group
Young's modulus	E	30e9	Pa	Basic
Poisson's ratio	nu	0.15	1	Basic
Density	rho	2400	kg/m³	Basic

MESH 1

Mapped 1

1 In the **Mesh** toolbar, click **Mapped**.

Use a mapped mesh inside the wall.

2 In the **Settings** window for **Mapped**, locate the **Domain Selection** section.

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

4 Select Domain 3 only.

Distribution 1

1 Right-click **Mapped 1** and choose **Distribution**.

2 In the **Settings** window for **Distribution**, locate the **Boundary Selection** section.

3 From the **Selection** list, choose **Wall diaphragm**.

4 Locate the **Distribution** section. In the **Number of elements** text field, type 60.

Distribution 2

1 In the **Model Builder** window, right-click **Mapped 1** and choose **Distribution**.

2 In the **Settings** window for **Distribution**, locate the **Boundary Selection** section.

3 Click **Paste Selection**.

4 In the **Paste Selection** dialog box, type 12 in the **Selection** text field.

5 Click **OK**.

6 In the **Settings** window for **Distribution**, locate the **Distribution** section.

7 In the **Number of elements** text field, type 2.

Free Triangular 1

In the **Mesh** toolbar, click **Free Triangular**.

Size 1

1 Right-click **Free Triangular 1** and choose **Size**.

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

3 From the **Predefined** list, choose **Finer**.

Distribution 1

1 In the **Model Builder** window, right-click **Free Triangular 1** and choose **Distribution**.

2 Select Boundary 6 only.

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

4 In the **Number of elements** text field, type 40.

Distribution 2

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 20.

Distribution 3

1 Right-click **Free Triangular 1** and choose **Distribution**.

2 Select Boundary 4 only.

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

4 In the **Number of elements** text field, type 3.

5 Click **Build All**.

6 Click the **Zoom Box** button In the **Graphics** toolbar and then use the mouse to zoom in on the contact zone where the mesh is the densest.

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** check box.

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 1 (sol1)

1 In the **Study** toolbar, click **Show Default Solver**.

2 In the **Model Builder** window, expand the **Solution 1 (sol1)** node.

3 In the **Model Builder** window, expand the **Study 1>Solver Configurations> Solution 1 (sol1)>Stationary Solver 1** node, then click **Parametric 1**.

4 In the **Settings** window for **Parametric**, click to expand the **Continuation** section.

5 From the **Predictor** list, choose **Constant**.

Restrict the step size to 0.5 to better capture the development of the plastic zone.

6 Select the **Tuning of step size** check box.

7 In the **Initial step size** text field, type `0.5`.

8 In the **Maximum step size** text field, type `0.5`.

9 In the **Study** toolbar, click **Compute**.

RESULTS

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.

1 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 In the **Displacement** toolbar, click **Plot**.

4 Click the **Zoom Extents** button in the **Graphics** toolbar.

Plastic Region

1 In the **Model Builder** window, under **Results** click **Equivalent Plastic Strain (solid)**.

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

Contour 1

1 In the **Model Builder** window, expand the **Plastic Region** node, then click **Contour 1**.

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

3 In the **Expression** text field, type `solid.epe>0`.

4 In the **Plastic Region** toolbar, click **Plot**.

Wall Deflection

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

2 In the **Settings** window for **1D 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. Select the **x-axis label** check box.

6 In the associated text field, type `Horizontal displacement (mm)`.

7 Select the **y-axis label** check box.

8 In the associated text field, type `Depth(m)`.

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

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

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

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

9 Find the **Include** subsection. In the **Prefix** text field, type `Depth=`.

10 In the **Wall Deflection** toolbar, click **Plot**.

Surface Settlement

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

2 In the **Settings** window for **1D 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. Select the **x-axis label** check box.

6 In the associated text field, type `Distance from the diaphragm (m)`.

7 Select the **y-axis label** check box.

8 In the associated text field, type `Vertical displacement (mm)`.

9 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 8 only.

3 In the **Settings** window for **Line Graph**, locate the **y-Axis Data** section.

4 In the **Expression** text field, type `v`.

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

6 Locate the **Legends** section. Select the **Show legends** check box.

7 Find the **Include** subsection. In the **Prefix** text field, type `Depth=`.

8 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 **RainbowLight**.

Deformation 1

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

Displacement 3D

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