Cantilever Retaining Wall
Outline

1 Description 3
   1.1 Properties 4
   1.2 Modeling Approach 5

2 Finite Element Model 6
   2.1 Geometry 8
   2.2 Properties 10
      2.2.1 Soil 10
      2.2.2 Retaining Wall 11
   2.3 Boundary Conditions 13
   2.4 Loads 15
   2.5 Mesh 16

3 Staged Construction Analysis 21
   3.1 Commands 21
   3.2 Results 28
      3.2.1 Vertical Displacements 28
      3.2.2 Distributed Moment 29

Appendix A Convergence Tolerance 30
Appendix B Additional Information 32
1 Description

Cantilevered retaining walls are typically used in cut excavations [Fig. 1] [Fig. 2]. Nevertheless, they can be used also for fill applications. Due to their narrow base width, cantilevered retaining walls are suitable for those situations with tight space constraints or right-of-way constraints.
1.1 Properties

The geometry of the model problem is represented in Figure 3. The retaining wall, made of concrete, is assumed isotropic and linear elastic. The mechanical response of the soil is modeled according to the Mohr-Coulomb model. The material properties for the concrete and the soil are summarized in Table 1.

![Figure 3: Schematic representation of the problem](https://dianafea.com)

<table>
<thead>
<tr>
<th>Parameters</th>
<th>Concrete</th>
<th>Soil</th>
</tr>
</thead>
<tbody>
<tr>
<td>Density $\rho$</td>
<td>2.4 T/m$^3$</td>
<td>1.9 T/m$^3$</td>
</tr>
<tr>
<td>Young’s modulus $E$</td>
<td>30000 N/mm$^2$</td>
<td>55 N/mm$^2$</td>
</tr>
<tr>
<td>Poisson’s ratio $\nu$</td>
<td>0.2</td>
<td>0.3</td>
</tr>
<tr>
<td>Friction angle $\phi$</td>
<td>-</td>
<td>35°</td>
</tr>
<tr>
<td>Dilatancy angle $\psi$</td>
<td>-</td>
<td>0.0°</td>
</tr>
<tr>
<td>Cohesion $c$</td>
<td>-</td>
<td>15 kN/m$^2$</td>
</tr>
<tr>
<td>Lateral pressure ratio $K_0$</td>
<td>-</td>
<td>0.65</td>
</tr>
</tbody>
</table>
1.2 Modeling Approach

The following aspects are considered:

- 2D plane strain is used for the modeling of this problem
- the lateral boundaries of the model (denoted by gray dashed lines in Figure 4) are constrained for normal displacements
- the retaining wall is modeled using sheet pile wall elements;
- the wall-soil interactions are modeled via automatic interface elements
- a staged construction analysis is performed for this geomechanical example, involving three stages: i) stress initialization in the soil and application of the surface load, ii) installation of the retaining wall and iii) analysis of the stresses in the soil after excavation
- the soil is divided into three domains: i) excavated soil, ii) passive soil region and iii) the remaining part of the soil [Fig. 4]
- the goal of the simulation is to investigate the displacement of the soil after excavation and calculate the bending moments along the axis of the retaining wall

![Diagram of retaining wall modeled in DIANAIE](https://dianafea.com)
2 Finite Element Model

For the modeling session we start a new project for structural analysis and plane strain conditions [Fig. 5]. We use quadratic finite elements. We choose quadrilateral elements to be dominantly used in the mesh.

Main menu ➔ File ➔ New

Figure 5: New project dialog
We choose meter for the length unit, kilonewton for force unit and degree for the angle.

Geometry browser ➔ Reference system ➔ Units  [Fig. 6]
Property Panel  [Fig. 7]
2.1 Geometry

To model the geometry of the soil, we create a first polygon sheet representing the entire soil model and two smaller polygon sheets for the excavated and passive portions [Fig. 4].

Main menu → Geometry → Create → Add polygon sheet [Fig. 8] – [Fig. 10]

Main menu → Viewer → Viewpoints → Top view

Main menu → Viewer → Fit all

Figure 8: Add polygon sheet soil
Figure 9: Add polygon sheet excavated
Figure 10: Add polygon sheet passive
Figure 11: Geometry top view
Then, we subtract the *excavated* and *passive* shapes from the *soil* one. Here, the *excavated* and *passive* shapes are not be removed at the end of the operation [Fig. 12].

Figure 12: Subtract shapes

Figure 13: Geometry top view
2.2 Properties

2.2.1 Soil

We assign the material properties to the soil. As mentioned in Section 1.1, the soil is modeled using the Mohr-Coulomb model (material properties are listed in Table 1) including a tension cut-off criterion to prevent the soil to resist to tensile stresses. Moreover, we select the initial stress option to create an initial state of stress in the soil based on K0 [Fig. 15] [Fig. 16].
2.2.2 Retaining Wall

We assign the properties to the retaining wall. We do this by first creating a sheet pile wall set, which we call Wall, and defining properties for it, and then we create the geometry of the retaining wall. In this way, the retaining wall geometry is automatically placed in the sheet pile wall set and the corresponding properties are automatically assigned to it.

We define the concrete material properties [Table 1] and choose Coulomb friction model for the interfaces with a friction angle $\phi$ of about two thirds of the soil friction angle.

**Main menu** → **Geometry** → **Create** [Fig. 17]

**Geometry browser** → **Sheet pile walls** [Fig. 17] → $\Rightarrow$ → **Sheet pile wall properties** [Fig. 18]

**Sheet pile wall properties** $\Rightarrow$ **Add material** [Fig. 19]

---

**Figure 17:** Geometry browser - sheet pile wall set

**Figure 18:** Sheet pile wall properties

**Figure 19:** Add new material
The retaining wall extends orthogonally to the $XY$ plane, therefore its geometry is defined through a line in this two-dimensional model. We create the line for the retaining wall [Fig. 20].

Main menu → Geometry → Create → Add line [Fig. 20]
2.3 Boundary Conditions

We constrain the displacement along $Y$ at the bottom edge of the model [Fig. 24].

Figure 23: Edit support bottom

Figure 24: Supports at bottom edge
Similarly, we constrain the displacement along $X$ at the left and right edges of the model [Fig. 26].

![Figure 26: Supports at bottom and side edges](https://dianafea.com)
2.4 Loads

We apply a distributed load on the top left edge of the soil model [Fig. 28].

---

**Figure 27:** Edit load *surface load*

**Figure 28:** Distributed load
2.5 Mesh

The mesh is defined by setting mesh properties along the edges. In particular, we create a fine mesh close to the retaining wall that becomes gradually coarser with the increment of distance from it. The characteristic size of the elements along the edges of the generated shapes is presented in Figure 29.

Figure 29: Characteristic size of the elements along the shape edges (arrows denote the gradation from fine to coarse mesh and blue symbols the mesh seeding along the shape edges)
We start with the definition of the mesh size of the retaining wall. According to Figure 29 the size of the retaining wall elements is equal to 0.5 m.

Figure 30: Mesh properties for the wall

Figure 31: Selection of the wall
We set the mesh size along the lateral and bottom edges of the model. The characteristic element size is 5 m [Fig. 29].

**Figure 32:** Mesh properties of the lateral and bottom edges

**Figure 33:** Selection of the lateral and bottom edges
We define a gradation mesh seeding along the horizontal edges of the soil shapes in order to get larger element size moving away from the retaining wall [Fig. 34]. We first select the bottom edges of the *excavated* and *passive* shapes and along the top left edge of the *soil* shape [Fig. 35]. Then we select the top edge of the *excavated* shape. We make use of the preview button in order to check the correct grading of the mesh seeding at the beginning and end of the edges [Fig. 36].
Now, we can generate the finite element mesh.

Figure 37: Finite element mesh
3 Staged Construction Analysis

3.1 Commands

The nonlinear staged construction analysis allows to take into account stress history in the soil and the construction sequence.

For the present case, three different stages are defined [Table 2]: i) stress initializations in the soil based on the K0 coefficient, and application of the surface load, ii) emplacement of the retaining wall in the soil and iii) removal of the excavated portion of soil.

Table 2: Staged analysis

<table>
<thead>
<tr>
<th>Stage number</th>
<th>Stage name</th>
<th>Description</th>
<th>Analysis type</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>K0 and surface load</td>
<td>stress initialization in soil &amp; surface load (wall is not active)</td>
<td>nonlinear analysis</td>
</tr>
<tr>
<td>2</td>
<td>Wall installation</td>
<td>activation of wall</td>
<td>nonlinear analysis</td>
</tr>
<tr>
<td>3</td>
<td>Excavation</td>
<td>deactivation of excavation soil (wall is active)</td>
<td>nonlinear analysis</td>
</tr>
</tbody>
</table>

To set up a staged construction analysis, in general, we follow these steps:

1. add a new Stage (rename accordingly)

2. via the Stage information action panel:
   - select the Shape sets to be active during the stage
   - select the shape material properties (if required)
   - select the Support sets and Tying sets to be active during the phase (if required)
   - set up Static steps (if a load is needed in the stage)

3. define the required analysis commands:
   - initialize stresses if new soil elements are included in the model
   - set up the details for the calculations (e.g., solver, convergence criteria, superposition, ...)

Cantilever Retaining Wall | https://diamafea.com
We set up a staged construction analysis considering the different phases of the excavation.
We start by defining the first construction stage $K_0$ and surface load to initialize the stresses in the soil and apply the surface load (the retaining wall is not considered in this stage).

![Figure 41: Analysis browser - add stage](https://dianafea.com)

![Figure 42: Active parts - stage $K_0$ and surface load](https://dianafea.com)

![Figure 43: Loads - stage $K_0$ and surface load](https://dianafea.com)
We create the second construction stage *Wall installation* in which the retaining wall is activated.

**Figure 44:** Analysis browser - add stage

**Figure 45:** Active model parts - stage *Wall installation*
We create the third construction stage *Excavation*. Here the excavated soil is deactivated.

---

**Analysis browser** → **StagedAnalysis** → **Geomechanical staged construction** → "Add stage" → **Excavation** [Fig. 44]

**Analysis browser** → **StagedAnalysis** → **Geomechanical staged construction** → **Excavation** → **Active model parts** [Fig. 47]
We present an overview of the stage settings [Fig. 48] and the active sets in each stage [Fig. 49].

**Analysis browser** ➔ StagedAnalysis ➔ Geomechanical staged construction ➔ Staged construction overview ➔ Stages [Fig. 48]

**Main menu** ➔ Analysis ➔ Edit phases [Fig. 49]

![Figure 48: Staged construction overview](https://dianafea.com)

![Figure 49: Active parts](https://dianafea.com)
We define the convergence criteria. We use the Secant (Quasi-Newton) method and set the convergence tolerance for the displacement equal to 0.001. Such a convergence tolerance is required in order to have realistic results taking into account the adopted mesh, as shown in the comparisons reported in Appendix A.

Finally, we can run the analysis.

**Fig. 50** Analysis browser

**Fig. 51** Iteration commands

**Fig. 52** Convergence settings
3.2 Results

3.2.1 Vertical Displacements

We present a contour plot of the total vertical displacements $TD_{tY}$ after the excavation. As expected, it is observed that, due to the excavation, unloading the surface of the passive part of the soil model (right) has a positive vertical displacement. In the present case, the maximum vertical displacement is around 3.6 cm.

![Figure 53: Result browser](https://dianafea.com)

![Figure 54: Vertical displacement $TD_{tY}$](https://dianafea.com)
3.2.2 Distributed Moment

We display the line diagram of the bending moment $M_{zz}$ along the vertical axis of the retaining wall. The diagram of the bending moment along the upper part of the retaining wall recalls that of a cantilever beam. This is because, due to the excavation, the right side of the exposed side of the retaining wall is not supported. The maximum values of the bending moment are observed slightly below the excavation level.

Figure 55: Result browser

Figure 56: Distributed moment $M_{zz}$
Appendix A  Convergence Tolerance

In order to evaluate the effect of the excavation on the retaining wall we set the convergence tolerance equal to 0.001 (see Figure 52) instead of using the default value 0.01. This stricter tolerance on the acceptance of the results calculated during the analysis improves the solution, specially in the coarser mesh parts found away from the retaining wall. The results obtained using a convergence tolerance equal to 0.01 and to 0.001 are compared in Figure 57 and Figure 58.

As shown in [Fig. 57], the profile of Mzz along the upper part of the retaining wall does not resemble that of a cantilevered beam. Therefore the results obtained with a convergence tolerance equal to 0.01 are not correct. On the contrary, by setting the convergence tolerance to 0.001 we have a correct diagram of Mzz as observed in Figure 58.

![Figure 57: Distributed moment Mzz (convergence tolerance equal to 0.01)](https://dianafea.com)

![Figure 58: Distributed moment Mzz (convergence tolerance equal to 0.001)](https://dianafea.com)
The effect of having a finer mesh and using the default convergence tolerance value of 0.01 is shown in Figure 59, which provides now correct results. These results are in line with what is expected over the retaining wall and match the results provided by the calculation with convergence tolerance equal to 0.001 (see Figure 60). The latter also allows to reduce the calculation time due to the significantly less number of elements.

Figure 59: Distributed moment Mzz (convergence tolerance equal to 0.01 and refined mesh)

Figure 60: Distributed moment Mzz (convergence tolerance equal to 0.001)
Appendix B  Additional Information

Folder: Tutorials/RetainingWall

Number of elements ≈ 500

Keywords:
  ANALYS: nonlin phase physic stagco.
  CONSTR: suppor.
  ELEMEN: c1l2i c19pe cq16e interf pstrai shell spwall struct.
  LOAD: edge elemen force weight.
  MATERI: consta crack cutoff elasti fricti full harden isotro multil nonlin plasti retent smear soften soil spwama strain.
  OPTION: bfgs direct nonsym secant units.
  POST: binary ndiana.
  PRE: dianai.
  RESULT: cauchy displa effect extern force green head moment pressu reacti strain stress total tracti.
Disclaimer: The aim of this technical tutorial is to illustrate various tools, modelling techniques and analysis workflows in DIANA. DIANA FEA BV does not accept any responsibility regarding the presented cases, used parameters, and presented results.