3

Loading of a suction pile

3.1 Introduction

In this tutorial a suction pile in an offshore foundation will be considered. A suction pile is a hollow steel pile with a large diameter and a closed top, which is installed in the seabed by pumping water from the inside. The resulting pressure difference between the outside and the inside is the driving force behind this installation.

This exercise will investigate the displacement of the suction pile under working load conditions. Four different angles of the working load will be considered. The installation process itself will not be modelled.

Objectives

- Using the polycurve designer
- Using rigid body objects
- Undrained effective stress analysis with undrained strength parameters
- Undrained shear strength increasing with depth
- Copying material data sets
- Changing settings in Output
- Helper objects for local mesh refinements

3.2 Geometry

In this exercise, the length of the suction pile is 10 m and the diameter is 5.0 m. An anchor chain is attached on the side of the pile, 7 m from the top. The water depth at the considered location is 50 m above the ground. Only one symmetric half will be modelled.

The soil consists of clay but because of the short duration of the load, an undrained stress analysis with undrained strength parameters will be performed. The geometry for the problem is shown below.

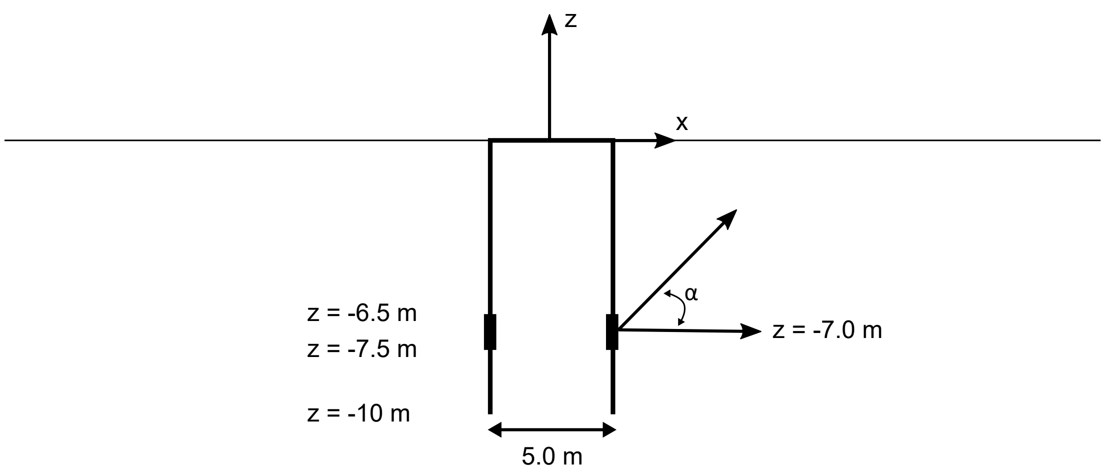

Figure 3–1: Geometry of the suction pile

3.3 Create a new project

An area of 30 m wide and 60 m long with half of the suction pile will be modelled in this example. With these dimensions the model is sufficiently large to avoid any influence from the model boundaries. To define the geometry for this exercise, follow these steps:

1. Start the **Input** program and select **New project** from the **Create/Open project** dialog box.
2. Enter an appropriate title for the project.
3. Keep the standard units and set the model dimensions to:
   - **a.** x_{min} =-30.0 and x_{max} = 30.0,
   - **b.** y_{min} = 0.0 and y_{max} = 30.0.
4. Click **OK**.

3.4 Define the soil stratigraphy

In the current example only one horizontal soil layer is present. A single borehole is sufficient to define it.

1. Click the **Create borehole** button and create a borehole at (0 0 0).

   The **Modify soil layers** window pops up.

2. In the **Modify soil layers** window add a soil layer with top boundary at z = 0 m and bottom boundary at z = -30 m.

3. The water depth at the considered location is 50 m. This would imply that the head is set to 50 m, but the results will be equal as long as the whole geometry is below the water level (the water above the ground is a total load and not an effective load). Hence, a head of 1.0 m is sufficient. Set the **Head** to 1.0 m.

3.5 Create and assign the material data sets

The material properties for the data sets are shown in Table 3–1 (p. 61).

Table 3–1: Material properties for the soil and interface

Property	Name	Clay	Interface	Unit
General				
Soil model	Model	Mohr-Coulomb	Mohr-Coulomb	-
Drainage type	Type	Undrained B	Undrained B	-
Unsaturated unit weight	γ_{unsat}	20	20	kN/m^3
Saturated unit weight	γ_{sat}	20	20	kN/m^3
Mechanical				
Young's modulus	E'_{ref}	1000	1000	kN/m^2
Poisson's ratio	ν (nu)	0.35	0.35	-
Increase in stiffness	E'_{inc}	1000	1000	$kN/m^2/m$
Reference level	z_{ref}	0.0	0.0	m
Undrained shear strength	$s_{u,ref}$	1.0	1.0	kN/m^2
Increase in undrained shear strength	$s_{u,inc}$	4.0	4.0	$kN/m^2/m$
Tension cut-off	-	Inactive	Inactive	-

Interfaces				
Strength determination	-	Rigid	Rigid	-
Interface reduction factor	R_{inter}	1.0	1.0	-
Initial				
K_0 determination	-	Manual	Manual	-
Initial lateral earth pressure coefficient	$K_{0,x}$, $K_{0,y}$	0.5	0.5	-

1. Click the **Materials** button .
2. Create the data sets given in Table 3–1 (p. 61). In the **Mechanical** tabsheet in the advanced parameters for strength deselect the **Tension cut-off** option. In this exercise, the permeability of the soil will not influence the results. Instead of using effective strength properties, the cohesion parameter will be used in this example to model undrained shear strength.

> **Note:**
>
> The **Interface** data set can be quickly created by copying the 'Clay' data set.

3. Assign the 'Clay' material data set to the soil layer and close the **Material sets** window.

3.6 Define the structural elements

The suction pile is modelled in the **Structures mode** as half a cylindrical surface and this is then defined as a rigid body. Also, a helper object is set for local mesh refinements.

3.6.1 Create a suction pile

In the **Structures mode** the suction pile as a rigid body will be defined. This is done by creating a polycurve at the soil surface and extruding it downward.

1. Click the **Structures** tab to proceed to **Structures mode** .
2. Click the **Start designer** button and then select the **Create polycurve** in the side toolbar.
3. Click at (2.5 0 0) on the drawing area to define the insertion point.

> **Note:** Select the menu **Options** > **Visualization settings** and set the **Intervals** to 2, while leaving the **Spacing** to 1 m. This allows to move the mouse with 0.5 m interval.

The **Polycurve designer** window pops up.

4 The polycurve is drawn in the xy-plane (see the Figure 3–2 (p. 63)). Hence the default orientation axes are valid for this example.

Note: For more information refer to "*Create polycurve*" in the Reference Manual.

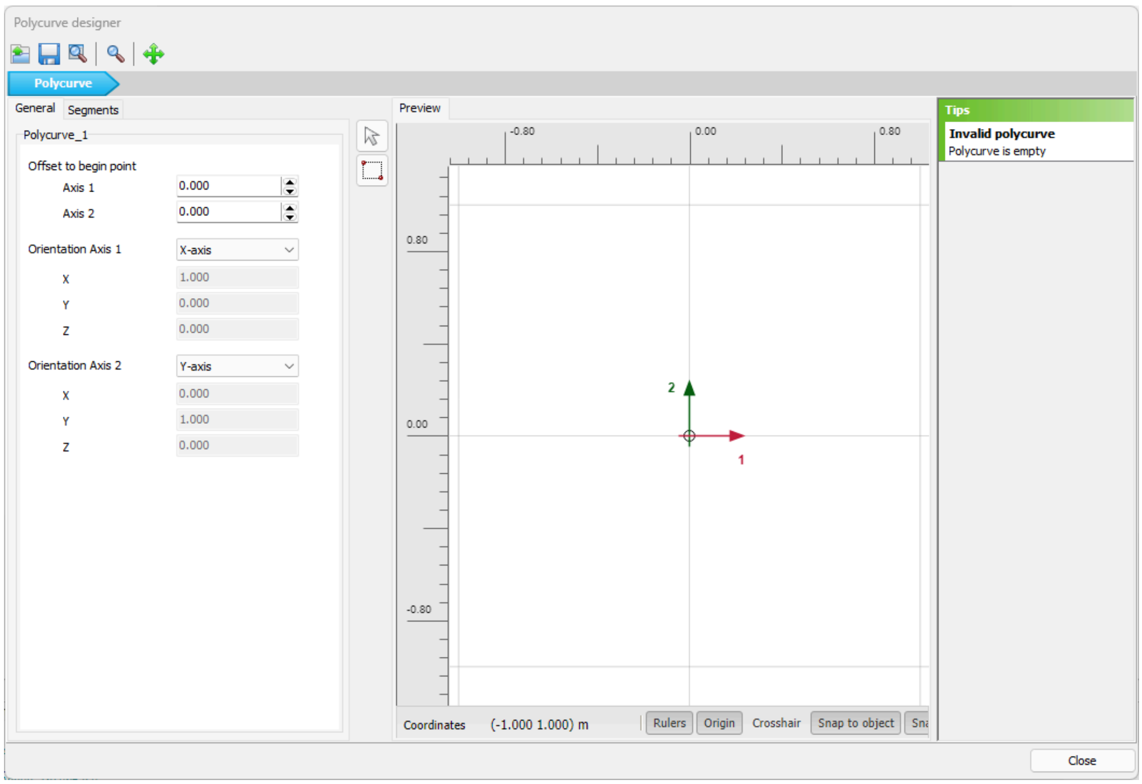

Figure 3–2: General tabsheet of the Polycurve designer

5 In the **Segments** tabsheet, click on the **Add section** button in the side toolbar.

6 In the Segment explorer - for the newly created element - set the **Segment type** to **Arc**, the **Relative start angle** to 90°, the **Radius** to 2.5 m and the **Segment angle** to 180°. The geometry will appear automatically in the **Preview area**.

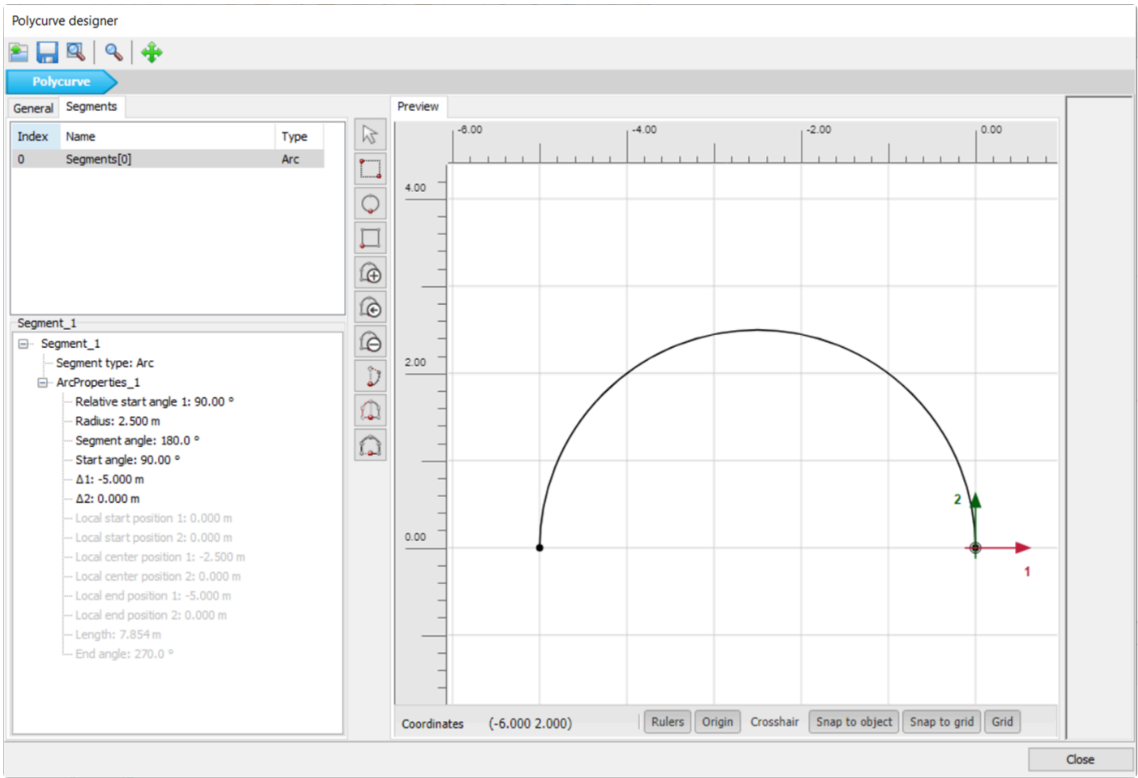

Figure 3–3: Segment tabsheet of the Polycurve designer

7 Click on **Close**, the polycurve will be automatically inserted into the model.

8 Click on the created polycurve and select the **Extrude object** and set the z value to -10 m.

9 Right-click the created surface, and select create positive interface to create an interface on the inside of the the suction pile. Similarly create a negative interface for the outside of the suction pile.

10 Right-click the polycurve and select **Close** from the appearing menu. Further, right click the closed polycurve and select **Create** > **Create surface**.

This creates the top surface of the suction pile.

11 Right-click the top surface and create a negative interface.

12 In the **Model explorer** select each **Interface**. Once the interfaces are selected, in the **Selection Explorer** inside the **Material mode** select **Custom** from in the displayed dropdown menu (see Figure 3–4 (p. 65)).

13 For the **Material** option from the dropdown menu select **Interface**.

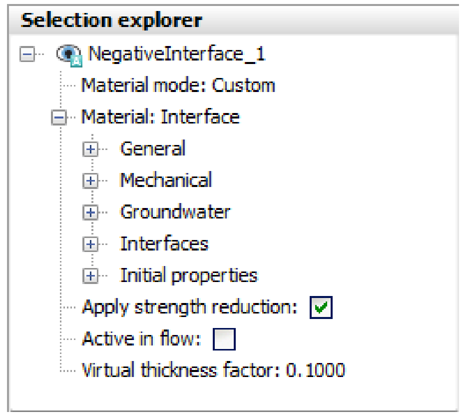

Figure 3–4: *Interface* material assignment in *Selection explorer*

14 Multi-select the top and the curved surface. Right-click on the selected surfaces and select the option **Create** > **Create rigid body** from the appearing menu (see Figure 3–5 (p. 65)).

> **Note:**
>
> To create the suction pile, the Rigid body functionality is used. For more information on Rigid bodies, refer to the corresponding section in the Reference Manual.

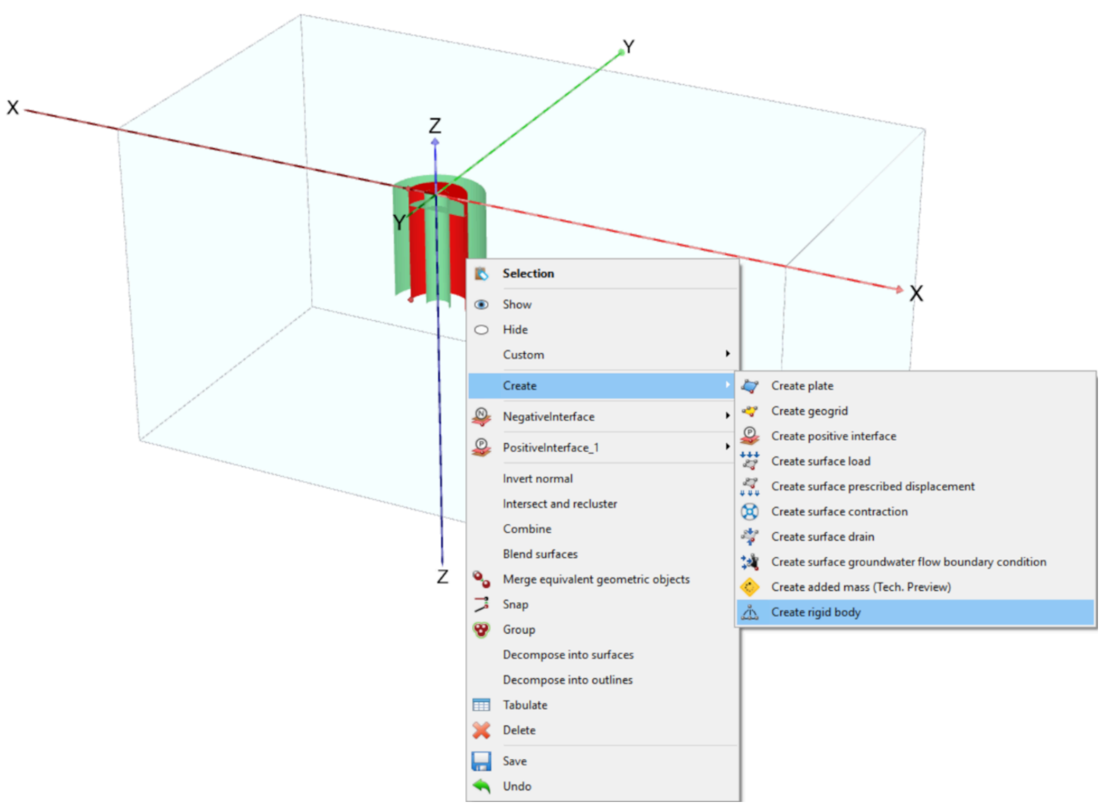

Figure 3–5: *Rigid body* creation

15 Select the created rigid body and in the **Selection explorer** set the reference point as (2.5 0 -7) assigning the values to x_{ref}, y_{ref} and z_{ref}.

16 As displayed in Figure 3–6 (p. 66) set the Translation condition$_y$ to **Displacement**, the Rotation condition$_x$ and Rotation condition$_z$ to **Rotation**. Their corresponding values are $u_y = \phi_x = \phi_z = 0$.

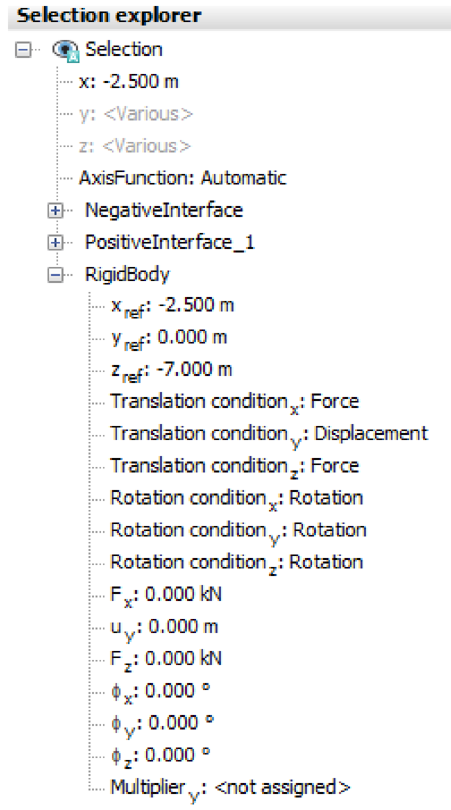

Figure 3–6: *Rigid body* in the *Selection explorer*

3.6.2 Create helper objects for local mesh refinements

A surface is created around the suction pile to achieve better mesh refinements. This is done by creating a circular surface around the suction pile using the **Polycurve designer**.

1 Click the Start designer button select the **Create polycurve** button in the side toolbar and click on (7.5 0 0) in the drawing area.

2 In the **General** tabsheet the default orientation axes (x-axis, y-axis) are valid for this polycurve.

3 In the **Segments** tabsheet, click on the **Add segment** in the top toolbar. Set the **Segment type** to **Arc**, **Relative start angle** to 90°, **Radius** to 7.5 m and **Segment angle** to 180°.

4 From the toolbar click **Close polycurve** to close the polycurve.

5 Close the **Polycurve designer**.

6 Click on the created polycurve and select the **Extrude object** and set the z value to -15 m.

7 Select the just created polycurve and the polycurve of the suction pile created earlier, right-click and select **Delete** from the appearing menu as shown in Figure 3–7 (p. 67).

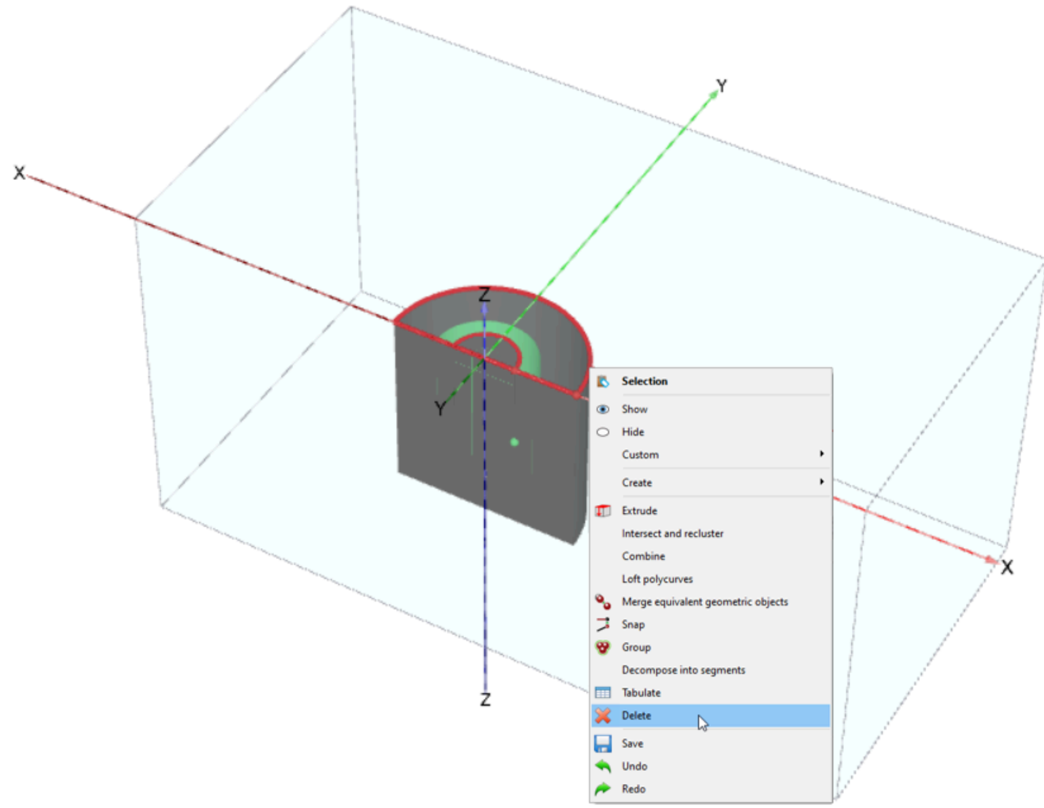

Figure 3–7: Deleting the two created polycurves

The geometry of the project is defined as shown in Figure 3–8 (p. 67).

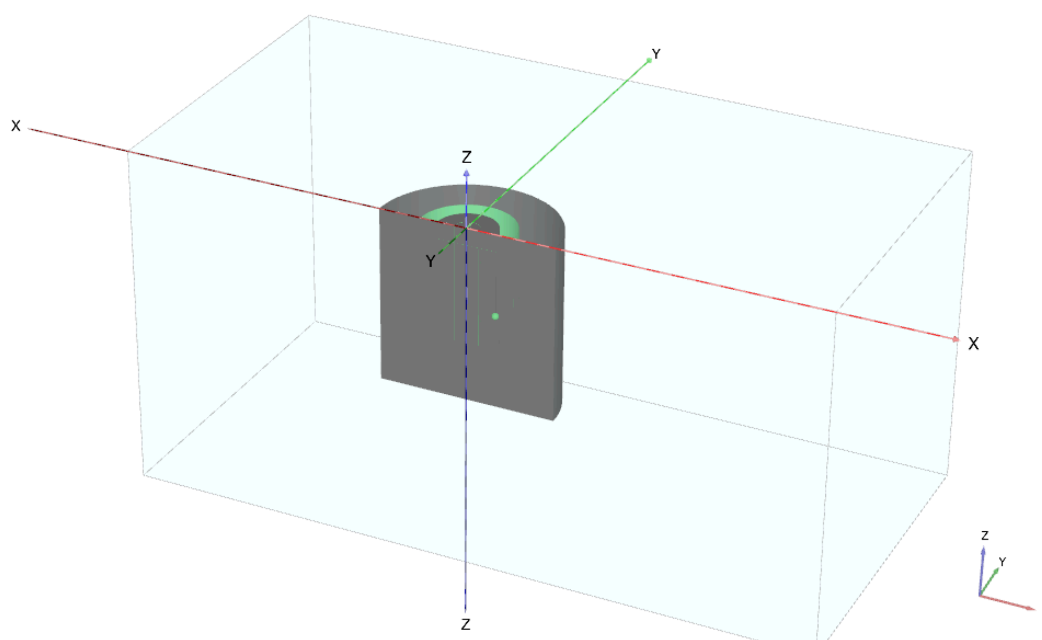

Figure 3–8: Geometry of the suction pile

3.7 Generate the mesh

In order to generate the mesh:

1. Click on the **Mesh** tab to proceed to the **Mesh mode**.
2. Hide the outer surface of the suction pile and the surface that limit the mesh refinement area. Multi-select the suction pile, the surface around the suction pile and the top surface of the suction pile.
3. In the **Selection explorer** set the value of **Coarseness factor** to 0.25.
4. Click the **Generate mesh** button to generate the mesh. The element distribution is **Medium**.
5. Proceed to the **Staged construction mode** .

3.8 Define the calculation

The calculation for this exercise consists of 6 phases. These are the determination of initial conditions, the installation of the suction pile and four different load conditions. The effect of the change of the load direction while keeping the magnitude unchanged will be analysed.

3.8.1 Initial phase: Initial conditions

Click on the **Staged construction** tab to proceed with the definition of the calculation phases.

1. Keep the calculation type of the Initial phase to **K0 procedure**. Ensure that all the structures and interfaces are switched off.

3.8.2 Phase 1: Installation of suction pile

1. Add a new calculation phase and rename it as `Install pile`.
2. For this phase, we use the option of **Ignore undrained behaviour**.
3. Activate all the rigid bodies and interfaces in the project.

3.8.3 Phase 2: Load Pile 30 degrees

1. Add a new phase and rename it as `Load pile 30 degrees`.
2. In the **Phases** window, go to the **Deformation control parameters** subtree and check the **Reset displacements to zero** checkbox.
3. In the **Numerical control parameters** subtree set:
   a. The **Solver type** to **Pardiso (multicore direct)** to enable a faster calculation for this particular project.
   b. Uncheck the **Use default iter parameters** checkbox, which allows you to change advanced settings.
   c. Set the **Max load fraction per step** to 0.1.
4. In the **Model explorer** click on the **Rigid bodies**.
5. In the **Selection explorer** tree, set F_x = 1949 kN and F_z = 1125 kN for the selected rigid bodies.

3.8.4 Phase 3,4,5,6: Load Pile with different direction angles

Define the remaining phases according to the information in Table 3–2 (p. 69). For each phase select the **Reset displacements to zero** option and set **Solver type** to **Pardiso (multicore direct)** and **Max load fraction per step** to 0.1.

Table 3–2: Load information at the chain attachment point

Phase	Start from phase	F_x	F_z
Load pile 30 degrees [Phase_2]	Phase_1	1949 kN	1125 kN
Load pile 40 degrees [Phase_3]	Phase_1	1724 kN	1447 kN
Load pile 50 degrees [Phase_4]	Phase_1	1447 kN	1724 kN
Load pile 60 degrees [Phase_5]	Phase_1	1125 kN	1949 kN

The order of the phases is indicated in the **Phases explorer** (see Figure 3–9 (p. 69)). Calculation of Phase_1 starts after the calculation of Initial phase is completed. The calculation of the remaining phases starts after the calculation of the pile installation phase is completed.

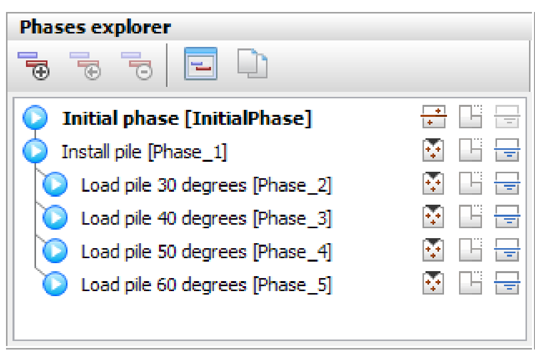

Figure 3–9: *Phases* explorer

3.8.5 Execute the calculation

1. Start the calculation process.
2. Save the project when the calculation is finished.

3.9 | Results

To view the results:

1. View the results of the last calculation phase. The deformed mesh of the whole geometry will be shown. In particular, the displacements of the suction pile itself are of interest.
2. Select the shadings representation and rotate the model such that the Y-axis is perpendicular to the screen.
3. If the axes are not visible, select this option from the **View** menu. It is quite clear that the point force acting on the pile does not disturb the displacement field locally indicating that the pile is sufficiently thick here.
4. In the same manner, the total displacements of the suction pile under a different direction of the load can be inspected by selecting the appropriate phase from the drop-down menu. In particular, Phase_2 is of interest, as in this phase the horizontal part of the load will have the largest value.

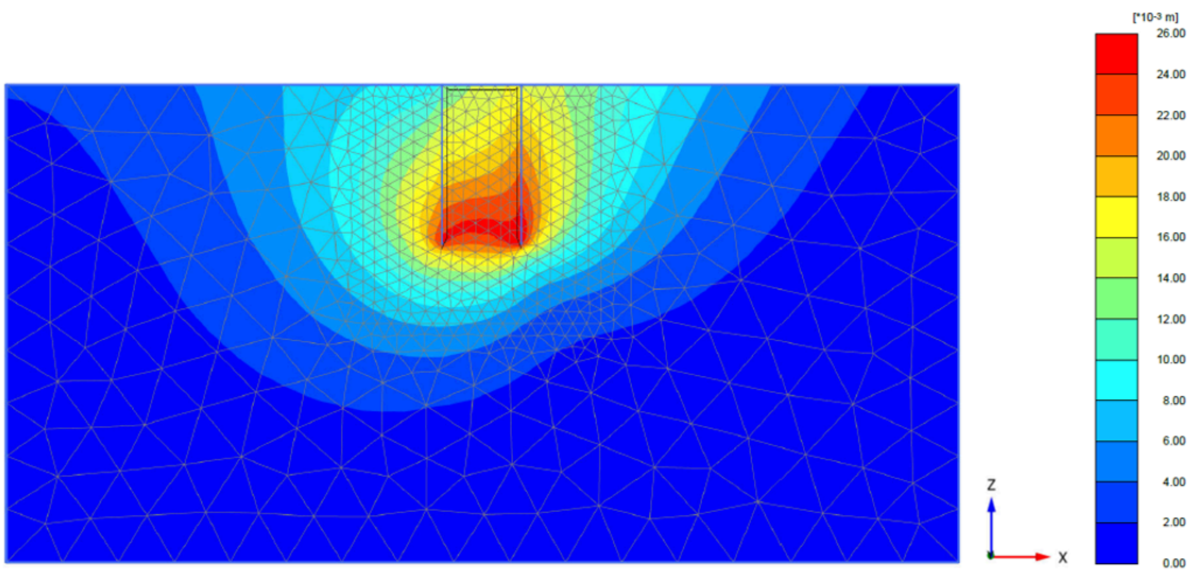

Figure 3–10: Total displacement of the suction pile at the end of *Phase_2*