

PLAXIS

Vertical Behaviour Assessment of an In-Group Barrette Pile Part I: Pile as Volume Elements

PLAXIS 3D 2024.2



Table of Contents

1 Introduction.....	1
2 Model Construction	3
3 Main Calculation Results.....	9
4 Mesh Sensitivity Analysis.....	13
5 Appendix.....	15

1.

Introduction

This case study presents the geotechnical finite element analysis of a pile-raft foundation system supporting a high-rise residential building. The foundation comprises a $61 \times 61 \text{ m}^2$ raft supported by 134 barrettes, as illustrated in Figure 1.

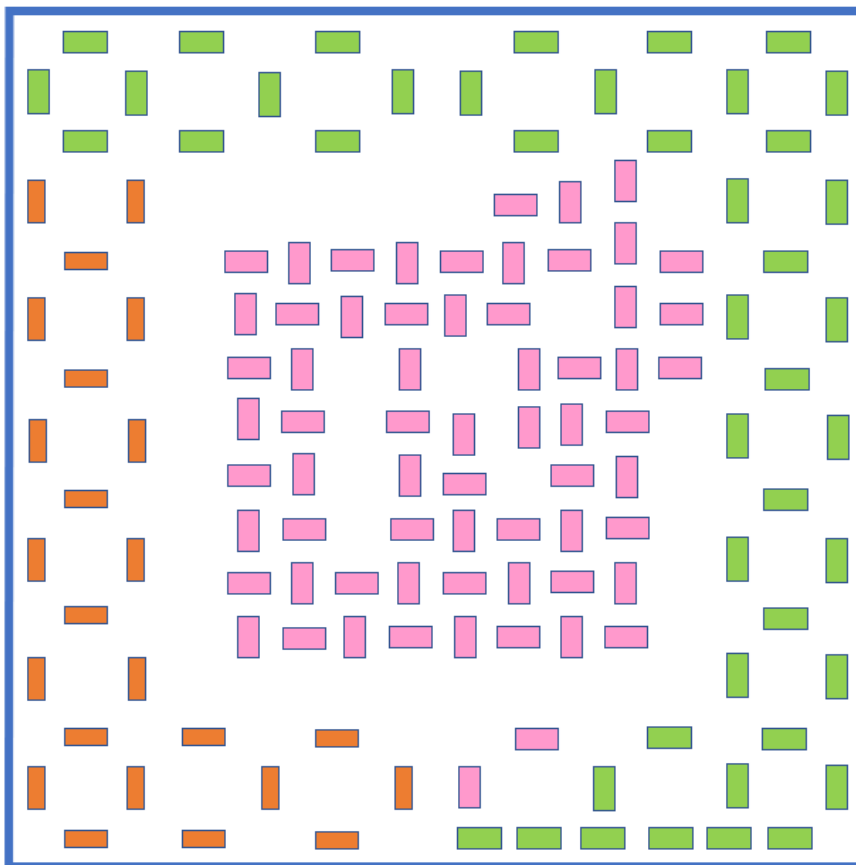





Figure 1: Footprint representation and barrettes layout

Barrettes are divided into 3 types:

-  Type 1 located in the core area. Section $2800 \times 1500 \text{ mm}$ with a base at the level -77 m (69.7 m long).
-  Type 2 positioned in the upper/right half of peripheral ring. Section $2800 \times 1500 \text{ mm}$ (same as type 1) with a base at the level -66.5 m (59.2 m long)
-  Type 3 in the lower/left half of peripheral ring area. Section $2800 \times 1000 \text{ mm}$ with a base at the level -58 m (50.7 m long)

The finite element analysis of the entire piled raft foundation is addressed in a separate study entitled “*3D Analysis of a High-Rise Building Foundation*”.

The objective of the present example is to evaluate the **vertical behaviour of a single Type 3 barrette pile** embedded within a representative group configuration. This is achieved using appropriate cyclic symmetry boundary conditions, along with a mesh sensitivity study aimed at:

- Establishing the **minimum mesh density requirements**, and
- **Optimizing the trade-off between computational cost and accuracy** for the full 3D foundation model.

In this first part, the barrette is modelled using a **volume element approach**, and the soil constitutive parameters are those previously calibrated against experimental results from a pile load test, as described in the article “*3D Back-Analysis of an O-Cell Pile Load Test*.”

A second model, presented in the follow-up document “*Vertical Behaviour Assessment of an In-Group Barrette in PLAXIS 3D – Part II: Pile as Embedded Beam Elements*,” adopts an **embedded beam representation** for the pile. That second part details the modelling strategy and parameter settings specific to the embedded beam formulation, allowing direct comparison between both modelling approaches.

Keywords: PLAXIS 3D, Barrette pile, Volume pile, Pile group, Cyclic symmetry, Mesh sensitivity

2.

Model Construction

Start a new project and define the appropriate Contour dimensions according to the size of the geometry (see Figure 2).

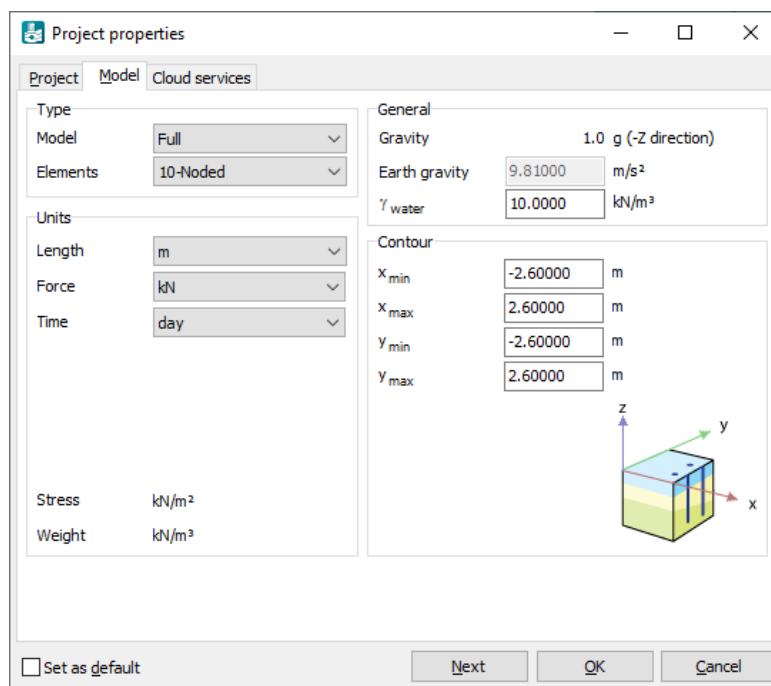


Figure 2: Model dimension definition in Project properties

The lateral model dimensions (5,2 m × 5,2 m) have been chosen from the average horizontal barrette spacing in the denser area of the pile group (taken from the centre part of the piled-raft foundation). This way as it is representative of the behaviour of a barrette in a pile group situation through the consideration of the cyclic symmetry introduced by the boundary conditions.

Soil mode

First create the soil material properties that will be used in this project as summarized in Table 1.

Click the create borehole button in the side (vertical) toolbar to start defining the soil stratigraphy and click at $x = 0$ in the drawing area to locate the borehole. The *Modify soil layers* window will appear as shown in Figure 3:

- Add 6 soil layers from -7.3 m to -10 m, -10 m to -12 m, -12 m to -35 m, -35 m to -75 m, -75 m to -117 m and -117 m to -130 m,
- Assign to each respective layer the soil material set parameters as defined in Table 1,
- Set water level head to -8 m.

Table 1: Soil material set parameters

Parameter	Dense Sand	Calcarenite	Calcisiltite/ Siltstone	Mudstone
Soil model	Mohr-Coulomb	Mohr-Coulomb	Mohr-Coulomb	Mohr-Coulomb
Drainage type	Drained	Drained	Drained	Drained
Unsaturated weight γ_{unsat}	18 kN/m ³	19 kN/m ³	19 kN/m ³	19 kN/m ³
Saturated weight γ_{sat}	18 kN/m ³	19 kN/m ³	19 kN/m ³	19 kN/m ³
<i>Mechanical properties</i>				
Young's modulus E_{ref}	50E3 kPa	300E3 kPa	1.7E6	1.7E6
Poisson's ratio ν	0.35	0.25	0.25	0.25
Cohesion c'_{ref}	1 kPa	35 kPa	38 kPa	38 kPa
Friction angle ϕ'	40°	32°	27°	27°
Dilatancy angle ψ	0°	0°	0°	0°
Tensile strength f_t	0 kPa	0 kPa	0 kPa	0 kPa
<i>Initial stress properties</i>				
Initial stress ratio K_0	Auto	Auto	Auto	Auto

Parameter	Sandstone	Calcisiltite
Soil model	HSSmall	HSSmall
Drainage type	Drained	Drained
Unsaturated weight γ_{unsat}	19 kN/m ³	20 kN/m ³
Saturated weight γ_{sat}	19 kN/m ³	20 kN/m ³
<i>Mechanical properties</i>		
Ref. secant modulus E_{50}^{ref}	200E3 kPa	500E3 kPa
Ref. oedometer modulus E_{oed}^{ref}	200E3 kPa	500E3 kPa
Ref. Young's modulus E_{ur}^{ref}	600E3 kPa	1.5E6 kPa
Poisson's ratio ν_{ur}	0.1	0.1
Power coefficient m	0	0
Reference pressure p_{ref}	100 kPa	100 kPa
Ref. dynamic shear modulus G_0^{ref}	750E3 kPa	1.5E6 kPa
Shear Strain 70%	0.4E-3	0.4E-3
Cohesion c'_{ref}	25 kPa	40 kPa
Friction angle ϕ'	35°	38°
Dilatancy angle ψ	0°	0°
Tensile strength f_t	0 kPa	0 kPa
<i>Initial stress properties</i>		
Initial stress ratio K_0	Auto	Auto

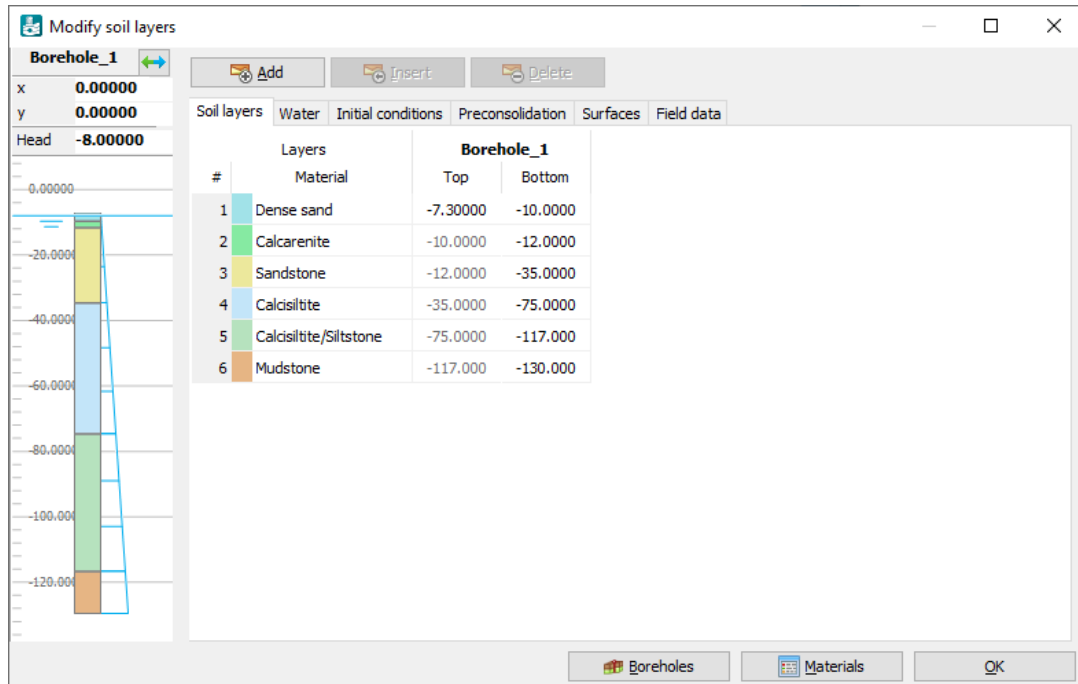


Figure 3: Borehole definition

Structure mode:

Start defining a surface at top model level by creating a polygon using the following surface points:

- (-1.4, -0.5, -7.3),
- (1.4, -0.5, -7.3),
- (1.4, 0.5, -7.3),
- (-1.4, 0.5, -7.3).

Consider the previously created surface *Polygon_1* and from the right-mouse click context menu select *Decompose into outlines*.

Select the previously created contour lines and extrude them downwards by an extrusion vector of (0, 0, -50.7 m).

The initial surface *Polygon_1* must be copied to represent the bottom surface of the barrette. Select *Polygon_1* and create an array with "1D, in z direction" shape with number of columns set to 2 and -50.7 m (barrette bottom position) as a distance between columns.

Select the outmost surfaces in X and Y direction (resulting from the line extrusion process) and select *Create positive interface*. Select the bottom interface and *Create negative interface*.

Finally, the pressure load at the top of the barrette will be created by creating a surface load on *Polygon_1*. Set the pressure to $\sigma_z = -30000$ kPa

Mesh mode

Go to *Mesh mode* and

- Select the four top clusters surrounding the barrette lateral outer contour and set their respective coarseness factor to 0.2 (see Figure 4(a)).
- Select the four inner soil clusters corresponding to the barrette set their respective coarseness factor to 0.1 (see Figure 4(b)).

- Click on *Generate mesh* and set the *Element distribution* to *Medium*. The generated mesh is presented in Figure 4(c).

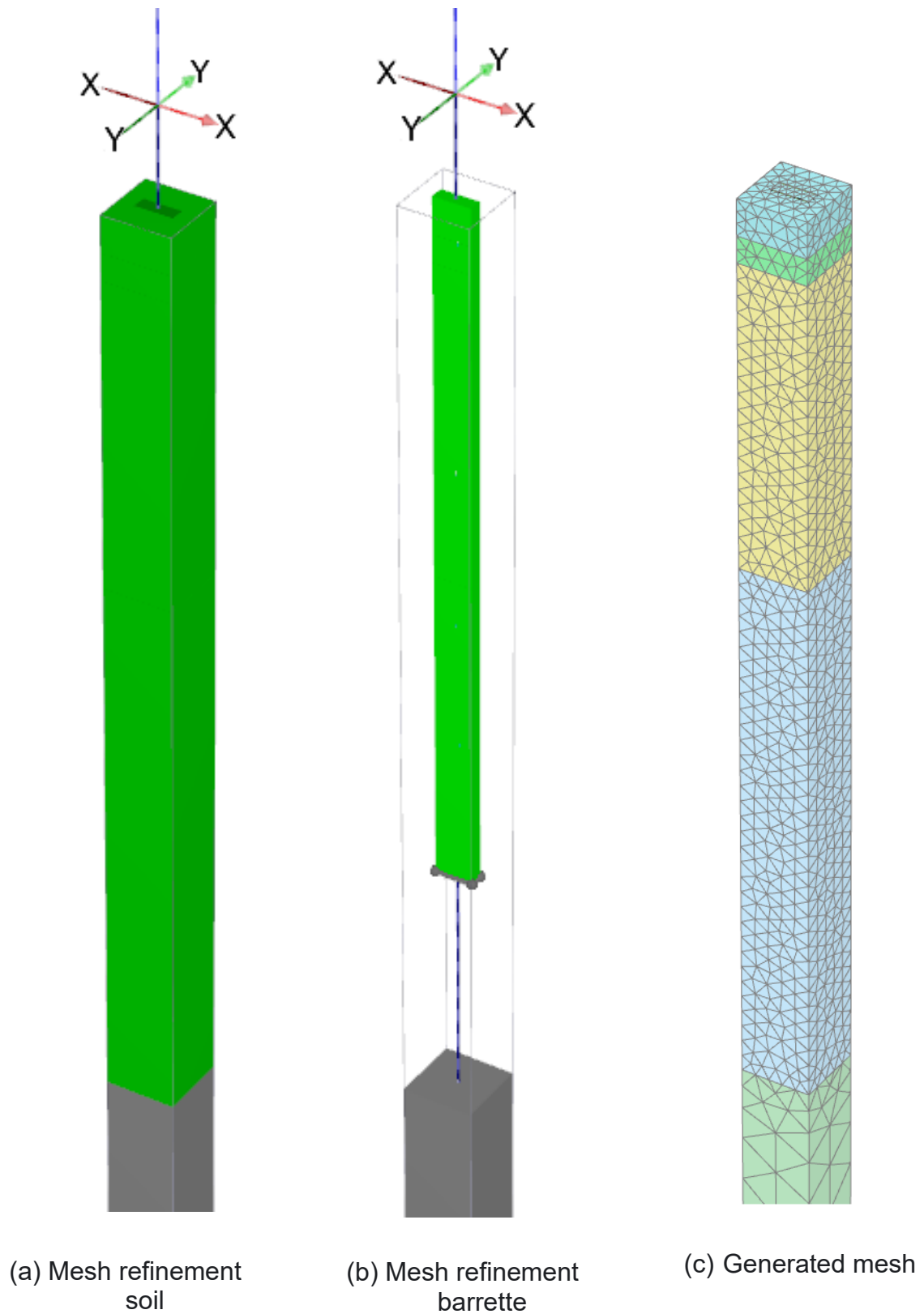


Figure 4: Mesh definition

Staged construction mode

Go to the *Staged construction* mode. Start creating the material property sets for the concrete materials as well as the barrette/soil interface properties about to be activated as given in Table 2 and Table 3 respectively.

Table 2: Concrete material properties

Material Name	Concrete
Soil model	Linear Elastic
Unit weight unsaturated γ_{unsat}	24 kN.m ⁻³
Drainage type	Non-porous
<i>Mechanical properties</i>	
Young modulus E_{ref}	20E6 kPa
Poisson ratio ν	0.2

Table 3: Interface material set parameters

Parameter	Interface Dense Sand	Interface Calcarenite	Interface Sandstone	Interface Calcisiltite
Soil model	Mohr-Coulomb	Mohr-Coulomb	Mohr-Coulomb	Mohr-Coulomb
Drainage type	Drained	Drained	Drained	Drained
Unsaturated weight γ_{unsat}	18 kN/m ³	19 kN/m ³	19 kN/m ³	20 kN/m ³
Saturated weight γ_{sat}	18 kN/m ³	19 kN/m ³	19 kN/m ³	20 kN/m ³
<i>Mechanical properties</i>				
Young's modulus E_{ref}	50E3 kPa	300E3 kPa	600E3	1.5E6
Poisson's ratio ν	0.35	0.25	0.25	0.25
Cohesion c'_{ref}	100 kPa	400 kPa	225 kPa	400 kPa
Friction angle ϕ'	0°	0°	0°	0°
Dilatancy angle ψ	0°	0°	0°	0°
Tensile strength f_t	0 kPa	0 kPa	0 kPa	0 kPa

Initial conditions

For the initial phase, K0-procedure will be used which is the default phase type in PLAXIS so no specific action is required at that stage.

Phase 1: Excavation

Create a first phase and name it *Pile Installation*:

- Select the 4 soil clusters corresponding to the barrette (from $z = -7.5$ m down to $z = -58$ m) and set their material to *Concrete* as defined in Table 2. Set their *WaterConditions* to *Dry* as well.
- Activate all interface elements around the barrette volume.

- Assign proper interface properties depending on whether the barrette is contacting the dense sand, the calcarenite, the calcisiltite or the sandstone as shown in Figure 5.



Figure 5: Activation and property assignment of interface elements

Phase 2: Loading

Create a second phase entitled *Loading*:

- Start with activating *Reset displacements to zero* from the *Phase settings* dialog box
- Activate surface loads.

Before starting the calculation process, two nodes belonging to respectively the upper and lower parts of the pile must be selected for being able to display the characteristic load-displacement curves:

- Click the *Select points for curves* button. The model and *Select points* window will be displayed in the *Output* program.
- Define (0, 0, -7.3) and (0, 0, -58) as *Point-of-interest coordinates*

The calculation can be launched by clicking on *Calculate*.

3.

Main Calculation Results

Once the calculation has completed, save the PLAXIS 3D model for instance as singleGroupPileMediumMesh.p3d and then open the output program by viewing the results for the Loading phase [Phase_2].

Deformed mesh and vertical displacement field

After filtering out soil elements from lowest soil layers (mudstone and calcisiltite/siltstone) along with interface elements and hiding elements with $Y < 0$ (using the *Filter* command from the *Geometry* menu), one can evaluate deformed mesh and vertical displacement results as shown in Figure 6 and Figure 7.



Deformed mesh |u| (scaled up 100 times)

Maximum value = 0.2579 m (at Node 60)

Figure 6: Deformed mesh for vertically loaded pile (scale factor = 100)

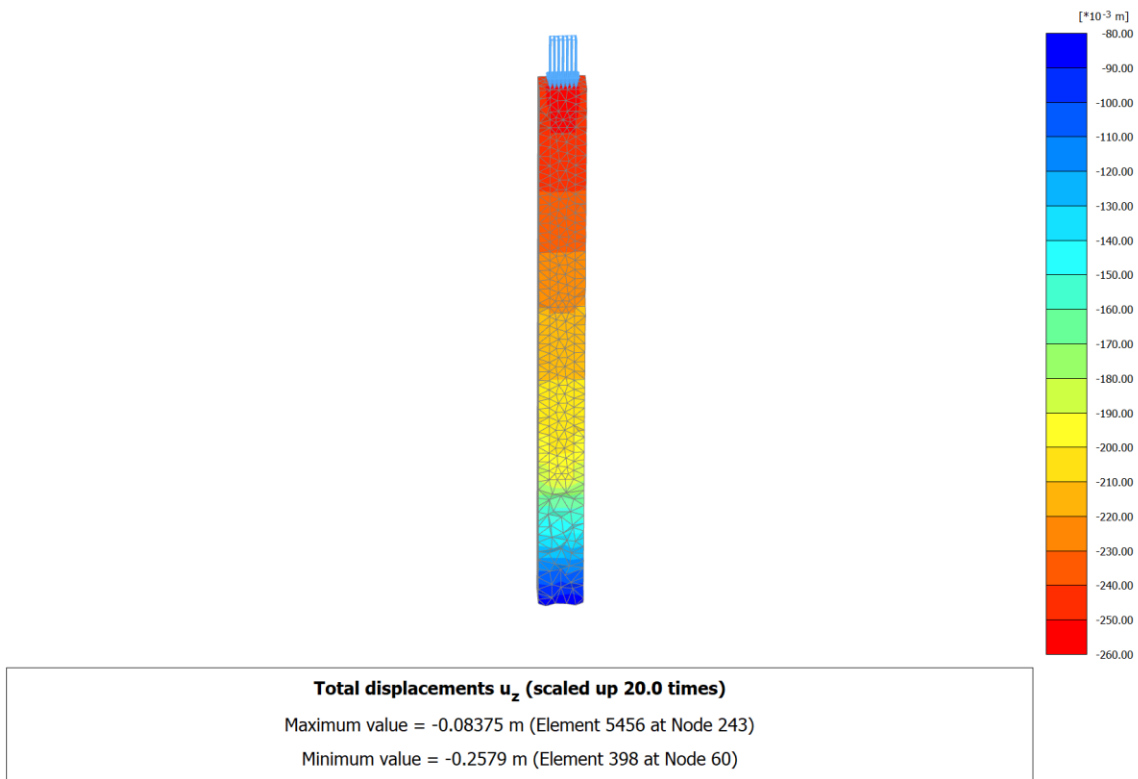


Figure 7: Vertical displacement contour plots for vertically loaded pile

Load-displacement curves

The load-displacement curves can be generated for the two selected curves points (at top and bottom of the barrette) by using the *Curve manager*.

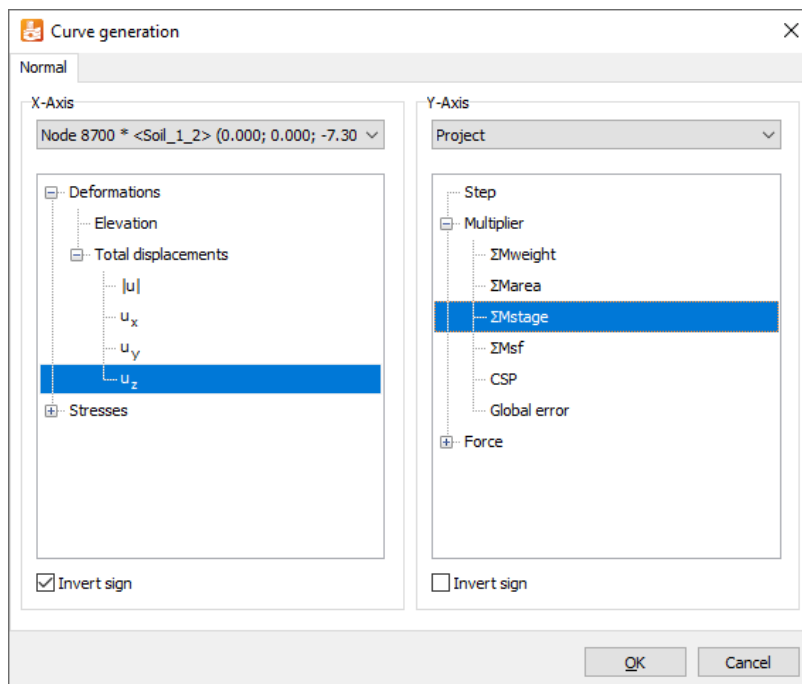


Figure 8: Curve generation settings for load displacement curve

Select the *Curves manager* tool and click on *New* in the *Charts* tabsheet to create a new curve. In the *Curve generation* dialog box, select the vertical displacement u_z for the bottom node at (0, 0, -7.3) as the X-Axis result (also activating the *Invert sign* option) and the ΣM_{stage} Project result as a Y-Axis result (see Figure 8).

From the *Curves Settings* only select last phase (*Vertical Loading [Phase 2]*) calculation results to be displayed as shown in Figure 9.

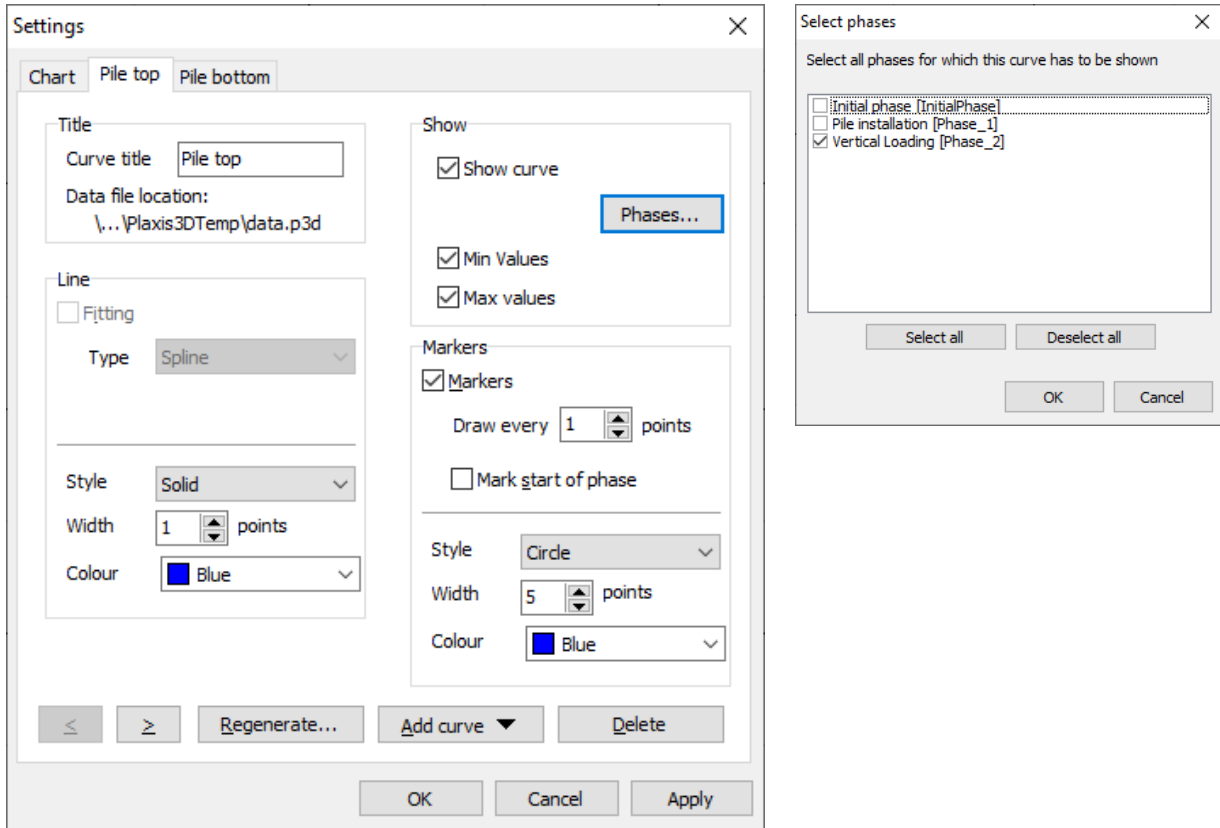
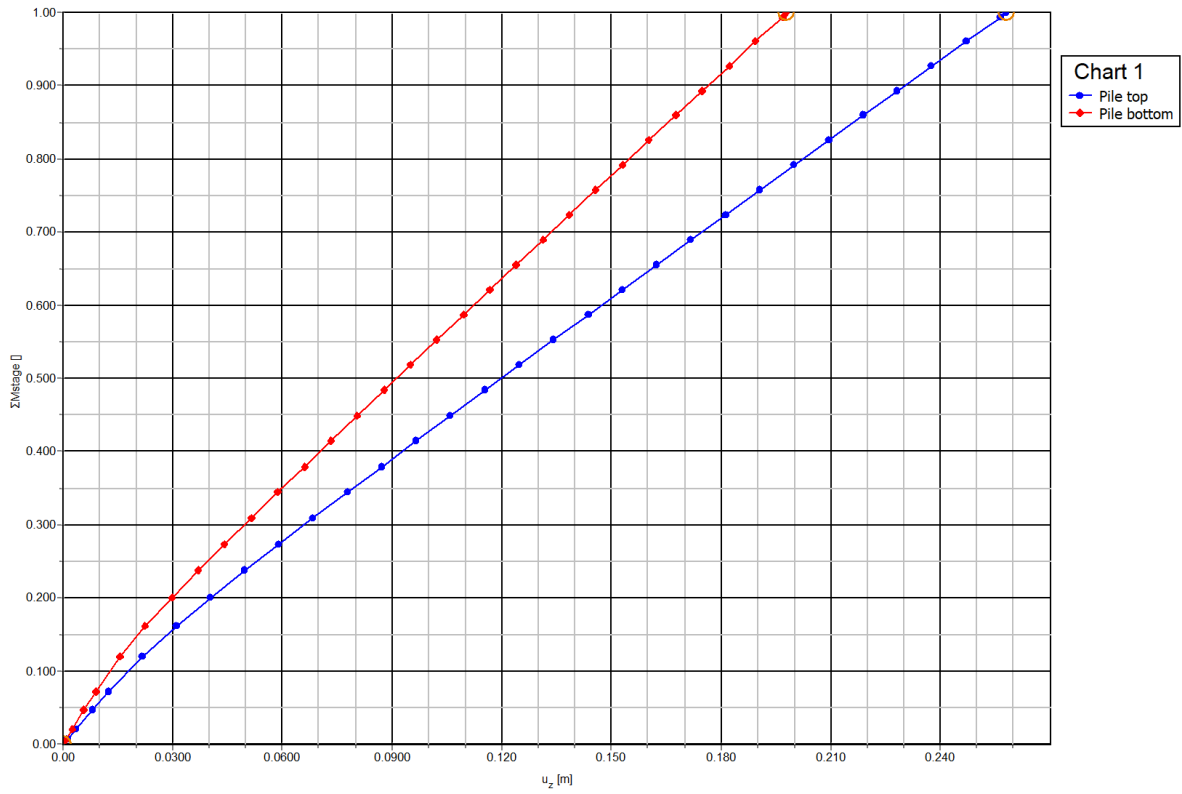


Figure 9: Curve settings for phase selection

Repeat these operations for bottom node at (0, 0, -58). Generated load-displacement curves at both pile top and bottom locations are finally shown in Figure 10.



Calculated load-displacement curves at both pile top and bottom locations

4.

Mesh Sensitivity Analysis

Mesh sensitivity can be evaluated by re-running both a coarser and a finer mesh version of the previously created FE model.

Rerun the analysis using respectively a coarseness factor of 1 and 0.1 for the four top clusters surrounding the barrette lateral outer contour initially set to a coarseness factor of 0.2 (operation to be carried out from the mesh mode followed by a mesh generation). Corresponding mesh configurations are summarized in Figure 10.

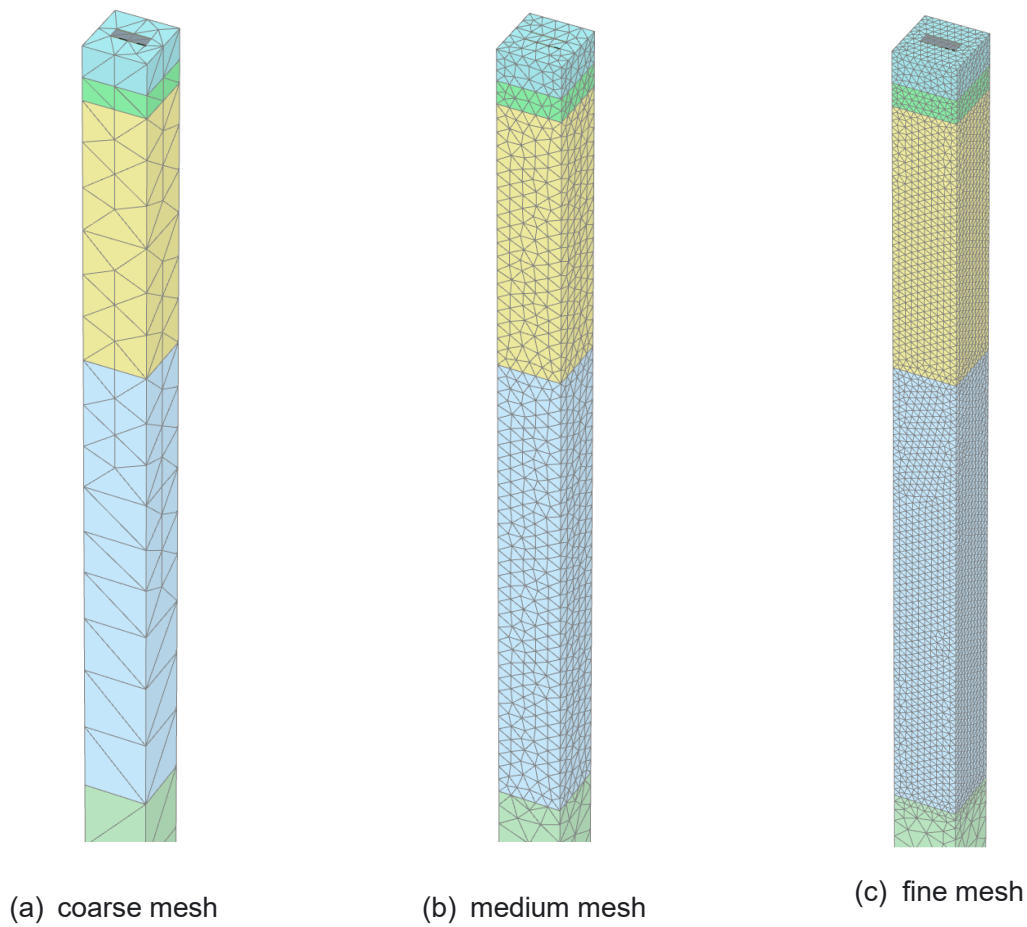
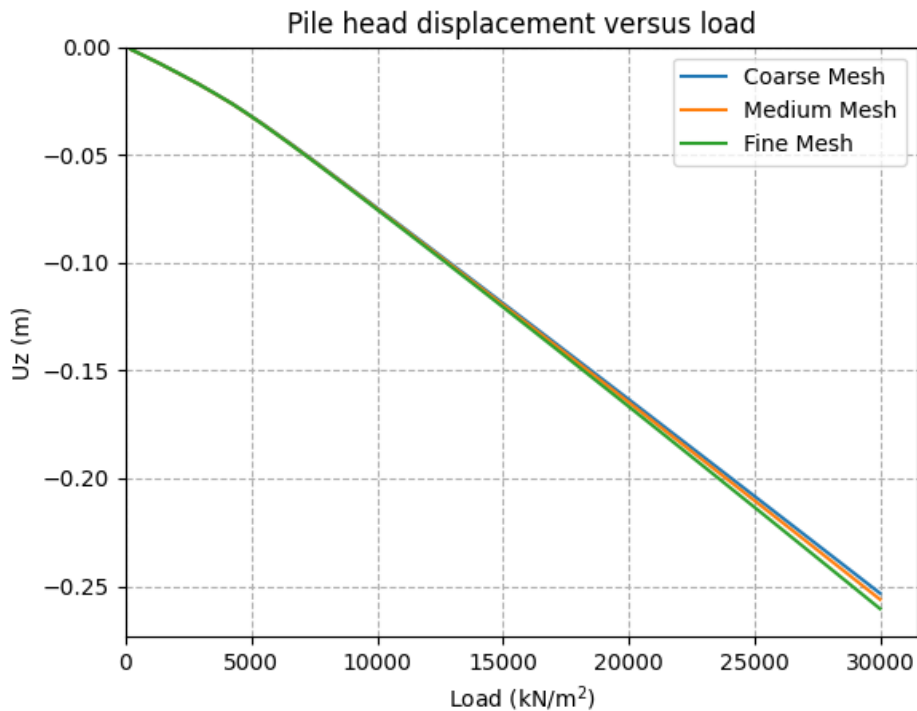


Figure 10: Different mesh configurations for the mesh sensitivity analysis

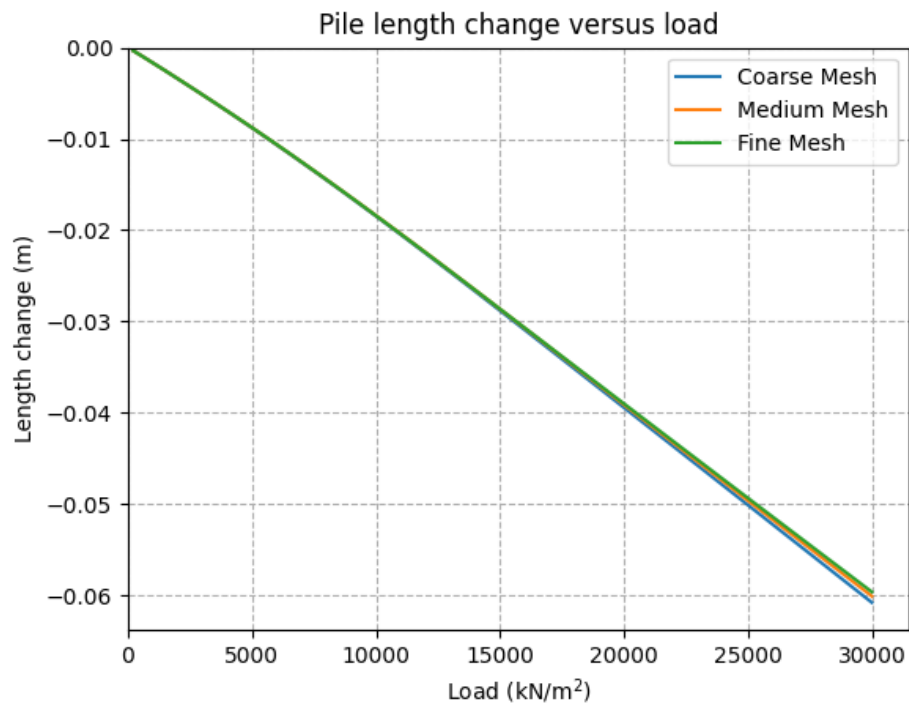
Run both additional analysis and save the additionally created PLAXIS models as singleGroupPileCoarseMesh.p3d and singleGroupPileFineMesh.p3d.

Obtained results can be post-processed using the **postProcessSinglePile.py** script (see Appendix).

Mesh sensitivity effects on both pile head displacement and pile length change are summarized in Figure 11.



(a) pile head displacement versus load



(b) pile length change versus load

Figure 11: Calculated load-displacement curves for three mesh configurations

5.

Appendix

The python code provided in **postProcessSinglePile.py** can be used to generate the characteristic load-displacement curves obtained numerically and how they would compare against the experimental values:

postProcessSinglePile.py

```
from plxscripting.easy import *
import matplotlib.pyplot as plt

# Initialize input scripting server
s_o, g_o = new_server()
#
pathname = 'C:\\Users\\Richard.Witasse\\OneDrive - Bentley Systems, Inc\\Desktop\\'
filenames = [
    "singleGroupPileCoarseMesh.p3d",
    "singleGroupPileMediumMesh.p3d",
    "singleGroupPileFineMesh.p3d"
]
fileLabels = [
    "Coarse Mesh",
    "Medium Mesh",
    "Fine Mesh"
]

Load = []
topDisplacement = []
UDiff = []
maxLoad = 30000

# Fetch results
for filename in filenames:
    filepath = pathname + filename
    s_o.open(filepath)

    unitLength = g_o.GeneralInfo.UnitLength.value
    unitForce = g_o.GeneralInfo.UnitForce.value
    phase = g_o.Phases[-1]
    Load.append([step.Reached.SumMstage.value*maxLoad for step in phase.Steps.value])
    nodeTop = g_o.Nodes[0]
    nodeBottom = g_o.Nodes[1]
    result = g_o.ResultTypes.Soil.Uz
    UTop = [g_o.getcurveresults(nodeTop, step, result) for step in phase.Steps.value]
    UBottom = [g_o.getcurveresults(nodeBottom, step, result) for step in phase.Steps.value]
    topDisplacement.append(UTop)
    UDiff.append([UTop[i] - UBottom[i] for i in range(len(UTop))])
    s_o.close()

# Plot results
fig, ax = plt.subplots()
ax.set_title('Pile head displacement versus load')
ax.set_xlabel("Load (" + unitForce + "/" + unitLength + "$^2$)")
ax.set_ylabel("Uz (" + unitLength + ")")
for i in range(len(topDisplacement)):
    ax.plot(Load[i], topDisplacement[i], label=fileLabels[i])
ax.legend()
plt.xlim(left=0)
plt.ylim(top=0)
plt.grid(True, ls='--')
plt.show()

fig, ax = plt.subplots()
ax.set_title('Pile length change versus load')
ax.set_xlabel("Load (" + unitForce + "/" + unitLength + "$^2$)")
ax.set_ylabel("Length change (" + unitLength + ")")
a = len(UDiff)
for i in range(len(UDiff)):
    ax.plot(Load[i], UDiff[i], label=fileLabels[i])
ax.legend()
plt.xlim(left = 0)
plt.ylim(top = 0)
plt.grid(True, ls='--')
plt.show()
```