

PLAXIS

Vertical Behaviour Assessment of an In-Group Barrette Pile

Part II: Pile as Embedded Beam Elements

PLAXIS 3D 2024.2



Table of Contents

1 Introduction.....	1
2 Model Construction	3
3 Main Calculation Results.....	8
4 Mesh Sensitivity Analysis.....	11
5 Appendix.....	13

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.

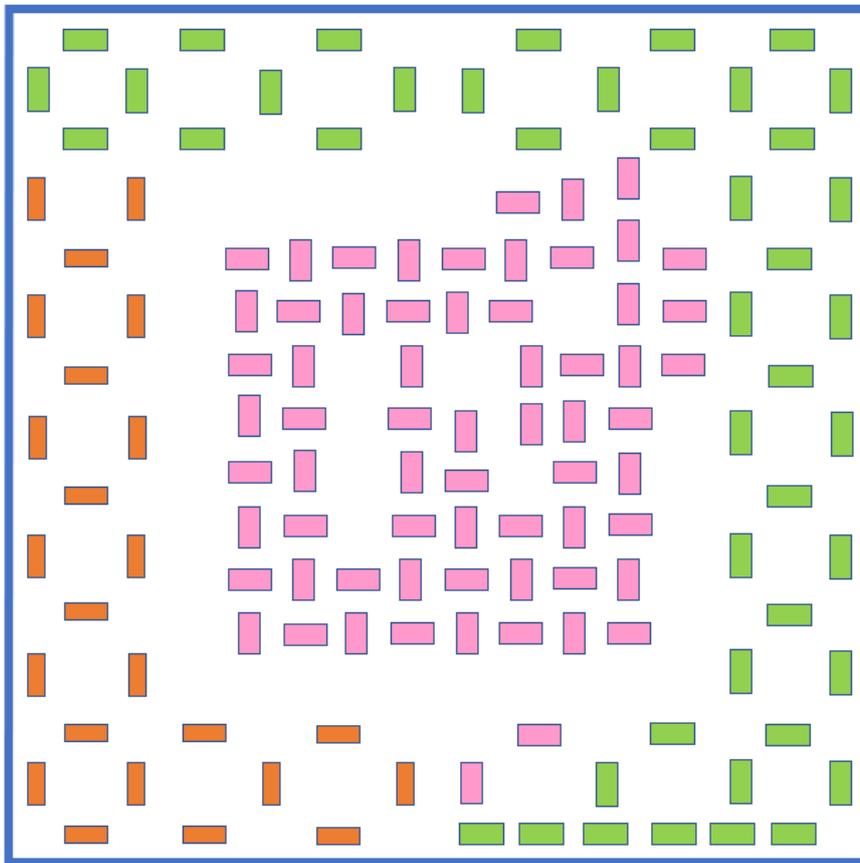


Figure 1: Footprint representation and barrettes layout

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 study is to evaluate the **vertical response of a single Type 3 barrette pile** within a group configuration, using appropriate cyclic symmetry boundary conditions. A **mesh sensitivity analysis** is also performed to:

- Determine the **minimum mesh density requirements**, and
- **Optimize the balance between computational cost and accuracy** for the full 3D foundation model.

In this analysis, the barrette is represented using an **embedded beam element** approach.

This work serves as a continuation of the previous study entitled “*Vertical Behaviour Assessment of an In-Group Barrette in PLAXIS 3D – Part I: Pile as Volume Elements*.” The same configuration and loading conditions are used here, but the pile is modelled with embedded beam elements instead of volume elements, allowing for a direct comparison between both modelling strategies.

Keywords: PLAXIS 3D, Barrette piles, Embedded beam, 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).

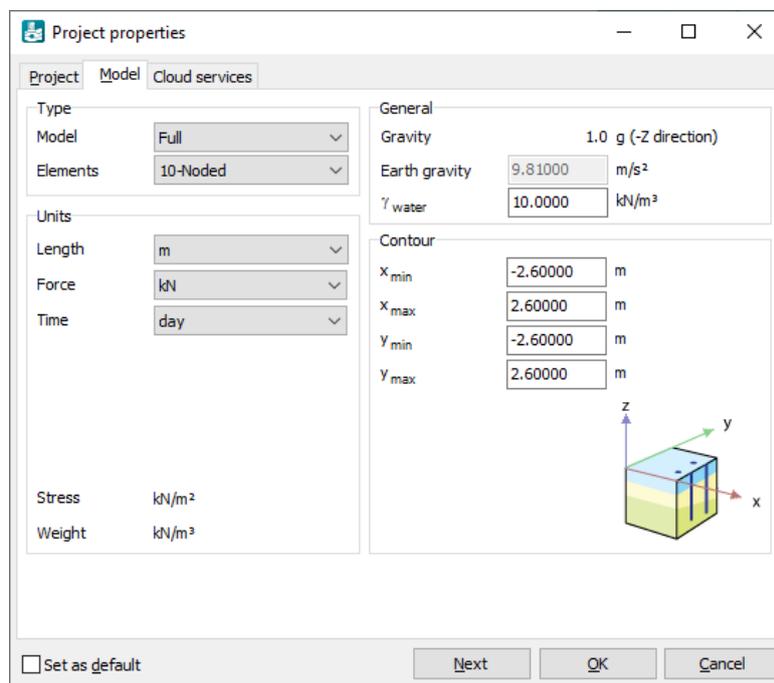


Figure 2: Model dimension definition in Project properties

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

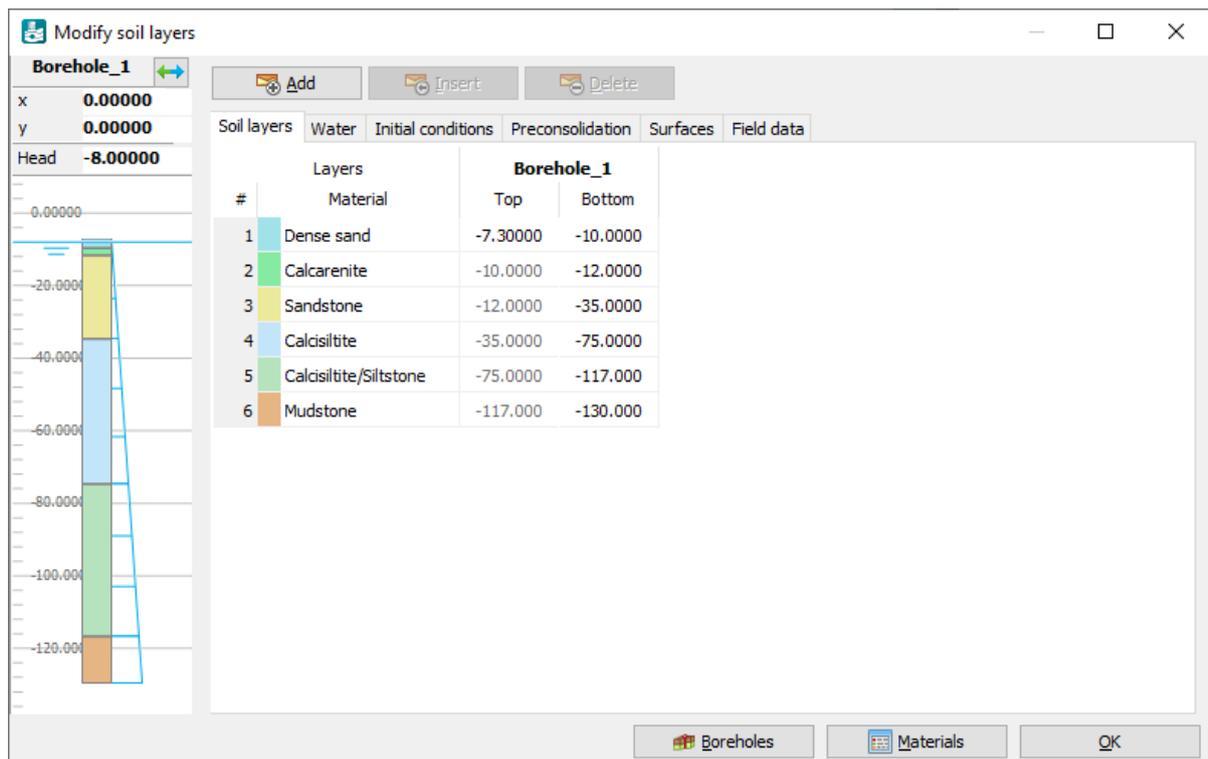


Figure 3: Borehole definition

Table 2: Summary of embedded beam properties for the embedded beam

Material Name		Pile – Type III	
Material type	Elastic		
Unit weight	24 kN/m ³		
Cross-section type	User-defined		
A	2.8 m ²		
I ₂	1.829 m ⁴		
I ₃	0.2333 m ⁴		
Stiffness <i>E</i>	20E6 kN/m ²		
Axial skin resistance	Multi-linear		
		Distance [m]	Axial skin resistance (kN/m)
		0.00	760
		2.70	760
		2.71	3040
		4.70	3040
		4.71	1710
		27.70	1710
		27.71	3040
		50.70	3040
Base resistance	100.E3 kN		

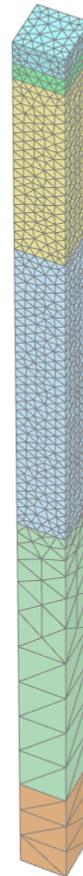
Structures mode:

Go to Structures mode and:

- Define an embedded beam going from (0, 0, -7.3) down to (0, 0, -58) and set the connection type to *Free*
- Set the property of the embedded beam using values proposed in Table 2. The geometrical properties correspond to those of the 1.0 m × 2.8 m barrette cross-section. The skin resistance values have been computed by integration of the respective interface resistance q_s within each soil layer over the barrette perimeter. Finally, F_{\max} has been taken arbitrarily large such it is never reached under considered load value.
- Assign the previously defined embedded beam property set to the embedded beam elements.
- Create a point load at (0, 0, -7.3) and set the vertical force value to $F_z = -84000$ kN (corresponding to a load pressure $\sigma_z = 30000$ kN over the pile top area $A = 2.8$ m²).



(a) Mesh refinement



(b) Generated mesh

Figure 4: Mesh definition

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

Staged construction mode

Go to the *Staged construction mode*

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* and activate all embedded beam elements.

Phase 2: Loading

Create a second phase entitled *Loading*:

- Start with activating *Reset displacements to zero* from the *Phase settings* dialog box
- Activate the point load

Before starting the calculation process, two nodes belonging to respectively the upper and lower part 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 singleGroupEBPileMediumMesh.p3d and then open the output program by viewing the results for the Loading phase [Phase_2].

Vertical displacement field

After filtering out soil elements from lowest soil layers (mudstone and calcisiltite/siltstone) and hiding elements with $Y < 0$ (using the *Filter* command from the *Geometry* menu), one can evaluate the vertical displacement results as shown in Figure 5(a) showing good agreement with the volume pile approach (see Figure 5(b)).

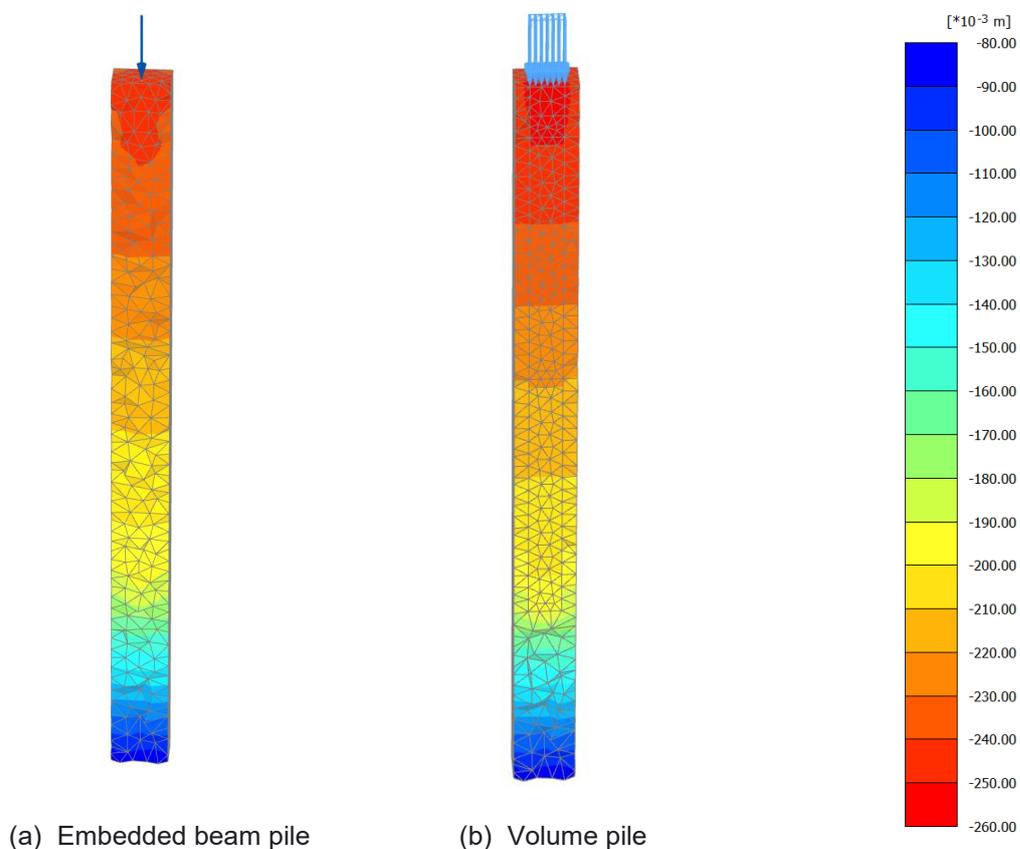


Figure 5: Vertical displacement contour plots for vertically loaded pile

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*.

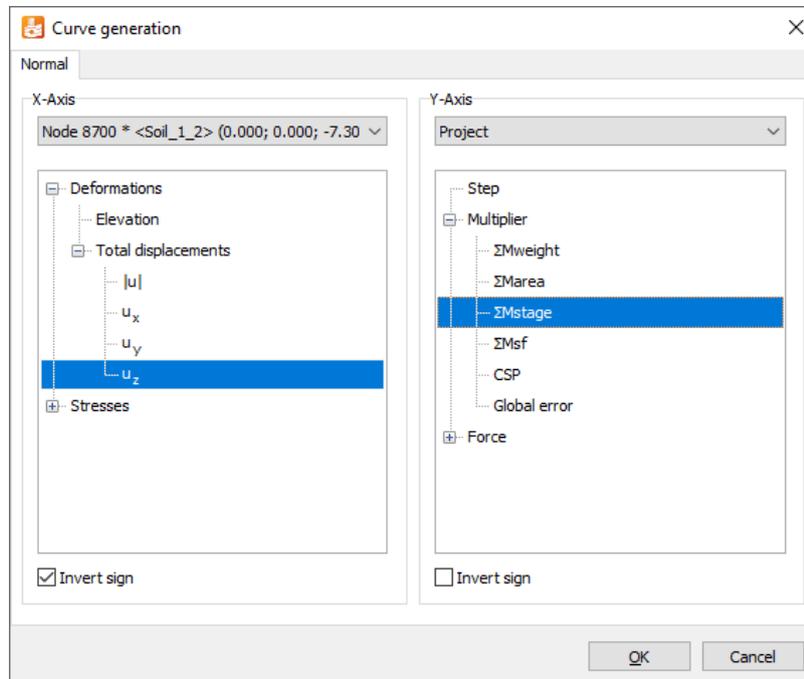


Figure 6: Curve generation settings for load displacement curve

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 6).

From the *Curves Settings* only select last phase (*Vertical Loading [Phase 2]*) calculation results to be displayed as shown in Figure 7.

Repeat these operations for bottom node at (0, 0, -58) as well as for top and bottom nodes for the reference model using volume elements to represent the pile. Generated load-displacement curves at both pile top and bottom locations are finally shown in Figure 8.

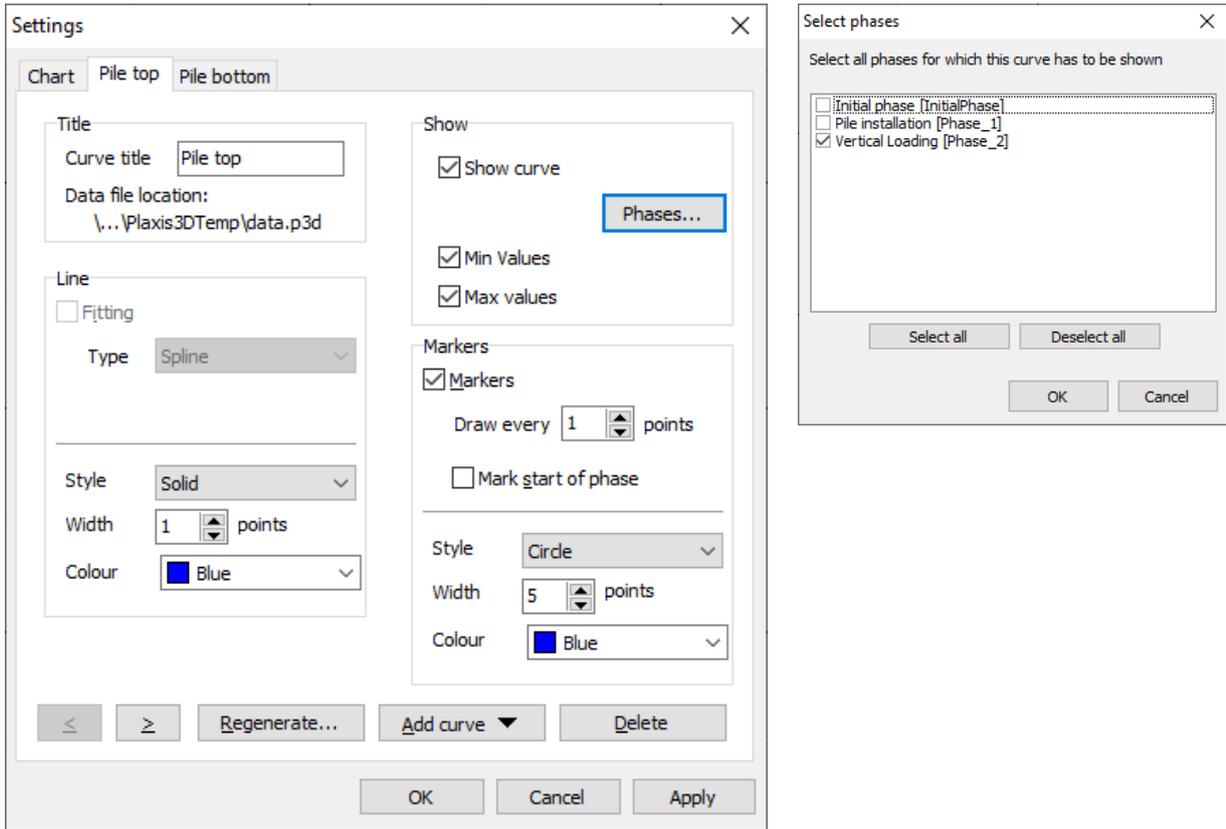


Figure 7: Curve settings for phase selection

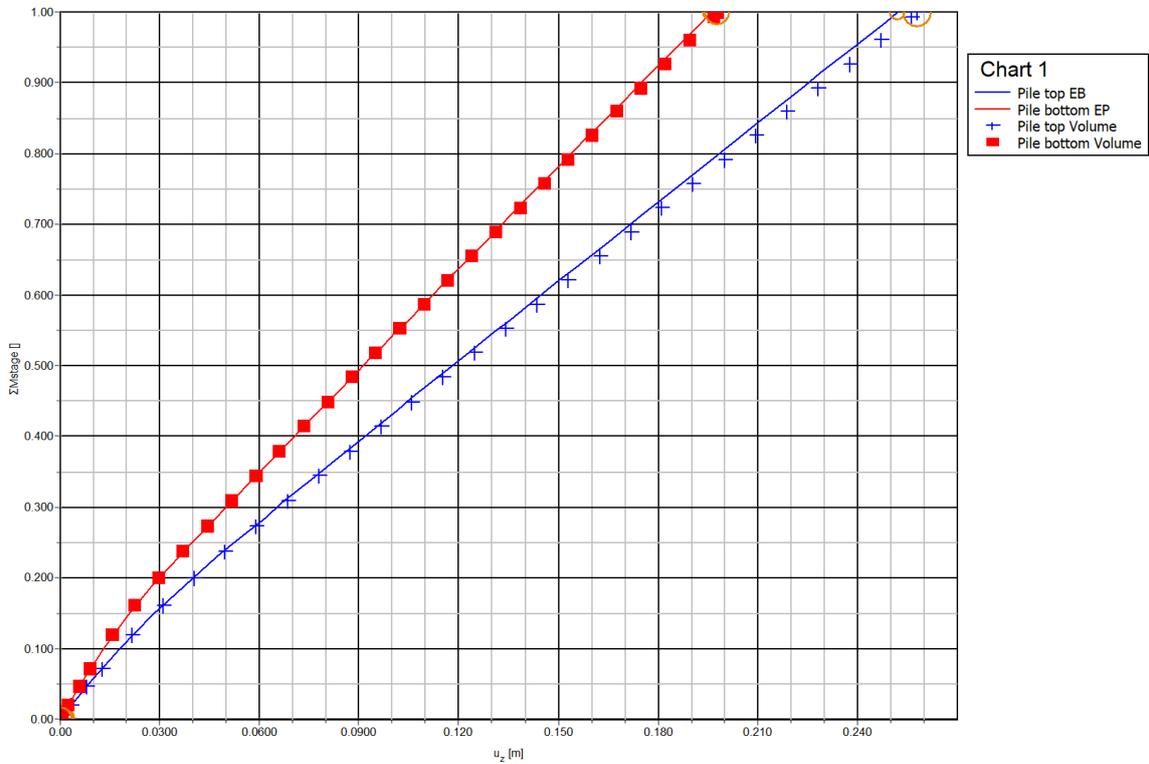


Figure 8: Calculated load-displacement curves at both pile top and bottom locations

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 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 9.

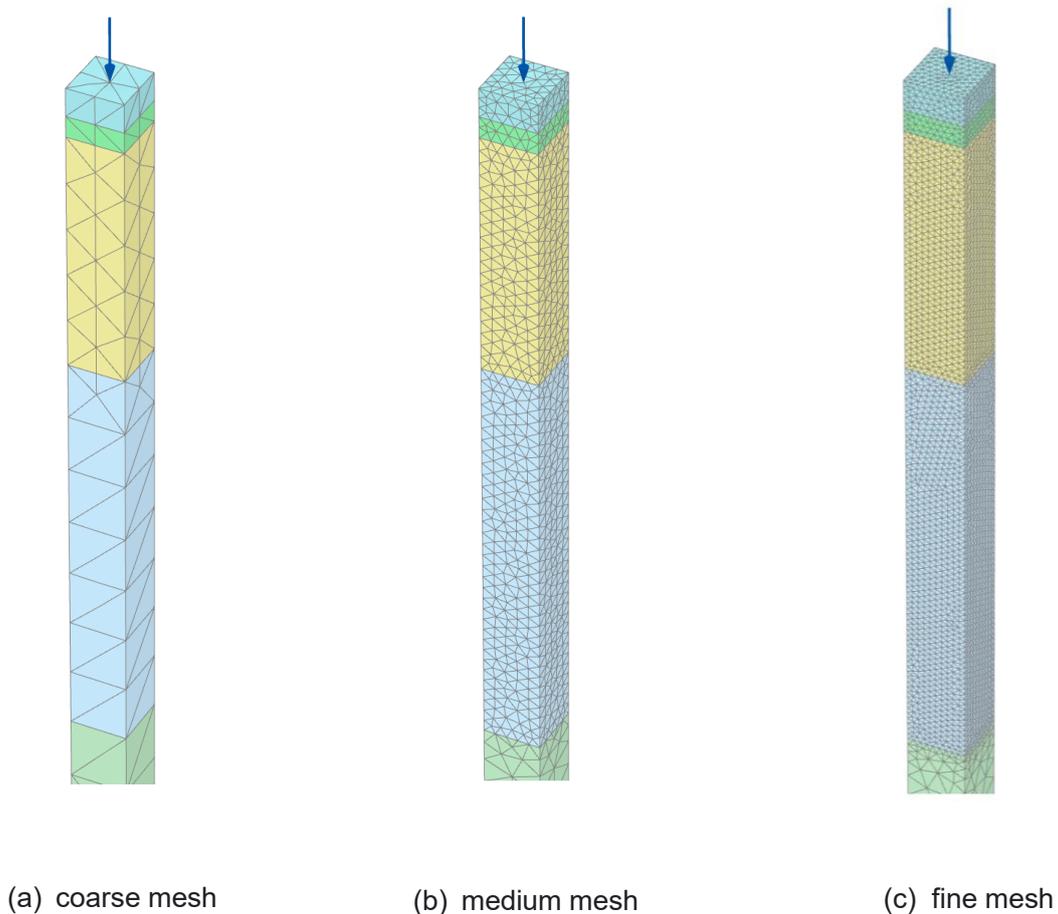
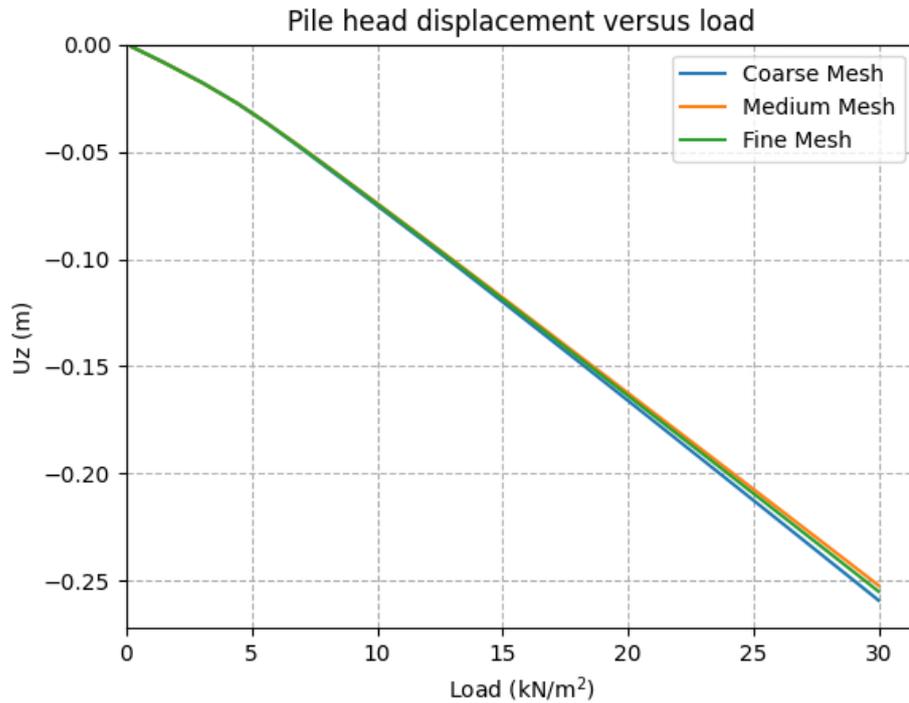


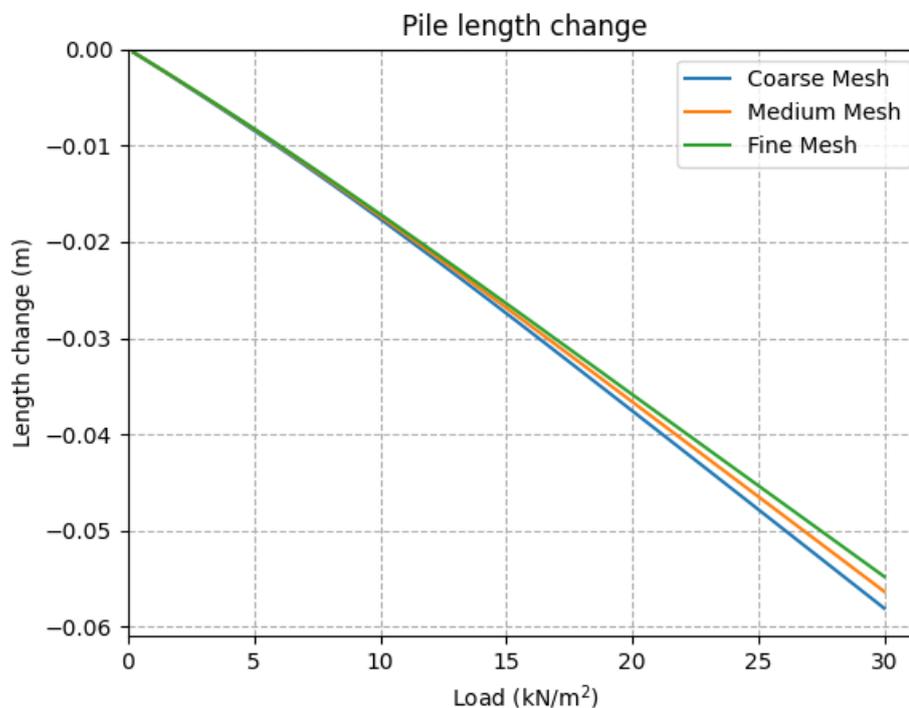
Figure 9: Different mesh configurations for the mesh sensitivity analysis

Run both additional analysis and save the additionally created PLAXIS models as singleGroupEBPileCoarseMesh.p3d and singleGroupEBPileFineMesh.p3d.

Obtained results can be post-processed using the `postProcessSinglePileEB.py` script (see Appendix) such that mesh sensitivity effects on both pile head displacement and pile length change can be evaluated as summarized in Figure 10.



(a) pile head displacement versus load



(b) pile length change versus load

Figure 10: Calculated load-displacement curves for three mesh configurations

5.

Appendix

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

postProcessSinglePileEB.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 = [
    "singleGroupEBPileCoarseMesh.p3d",
    "singleGroupEBPileMediumMesh.p3d",
    "singleGroupEBPileFineMesh.p3d"
]
fileLabels = [
    "Coarse Mesh",
    "Medium Mesh",
    "Fine Mesh"
]

Load = []
topDisplacement = []
UDiff = []
maxLoad = 30

# Fetch results
for filename in filenames:
    print(filename)
    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')
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()
```