PLAXIS

3D Analysis of a High-Rise Building Foundation

PLAXIS 3D 2024.3

Last updated: June, 2025

Seequent, The Bentley Subsurface Company

Table of Contents

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 × 61 m² raft supported by 134 barrettes, as illustrated in Figure 1.

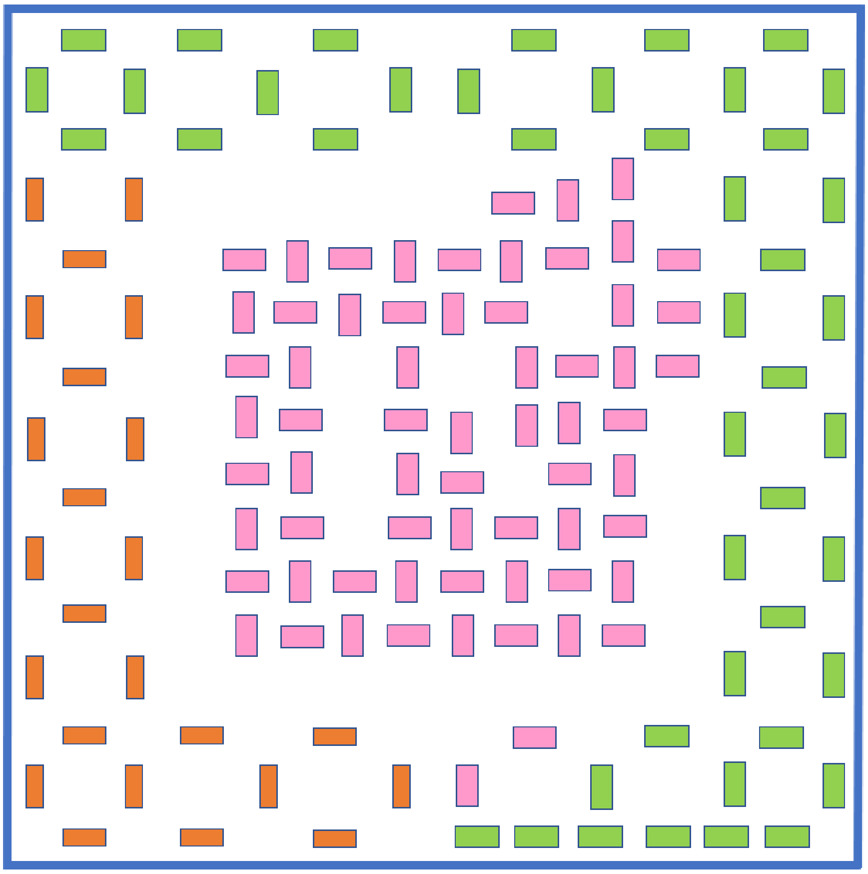


Figure 1: Footprint representation and barrettes layout

The barrettes are distributed across three groups:

- Type 1 located in the core area. Section 2800x1500 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 2800x1500 mm (same as type 1) with a base at the level -66.5 m (59.2 m long).
- Type 3 located in the lower/left half of peripheral ring. Section 2800x1000 mm with a base at the level -58 m (50.7 m long)

An important aspect of this study is the validation of the embedded beam modelling approach. This was addressed in a **companion exercise entitled *"Vertical Behaviour Assessment on an In-Group Barrette Pile"***, in which both volume piles and embedded beam piles were modelled and compared in a representative pile group configuration. The results of that study served to calibrate the embedded beam parameters used in the present model.

The main objective of this study is to build a comprehensive PLAXIS 3D geotechnical model to evaluate whether the foundation system satisfies serviceability limit state (SLS) criteria. Additionally, the model is used to provide meaningful geotechnical input to the structural design team. In particular, we focus on the **derivation of equivalent spring stiffnesses** that characterize the pile group behaviour and can be transferred to the structural model. This process highlights the **iterative nature of geotechnical–structural interaction**, where foundation stiffness from the geotechnical model feeds into the structural analysis, and loads from the structural model are fed back into the geotechnical simulation.

To streamline the model construction, an Excel-based automation process was used. One spreadsheet includes barrette geometries and locations, while another specifies pile head loads based on structural model outputs. These spreadsheets generate PLAXIS 3D command lines, significantly reducing model preparation time and ensuring consistency. This automation strategy will be showcased throughout the study.

Through this analysis, we also aim to demonstrate how a detailed geotechnical simulation, beyond regulatory compliance, can contribute meaningfully to multidisciplinary foundation design workflows.

Keywords: PLAXIS 3D, Barrette piles, Pile group, Soil structure Interaction, Equivalent pile stiffness, Automation

2.

Model Construction

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

The lateral model dimensions (300 m × 300 m) have been chosen large enough compared to the raft dimensions to avoid boundary effects.

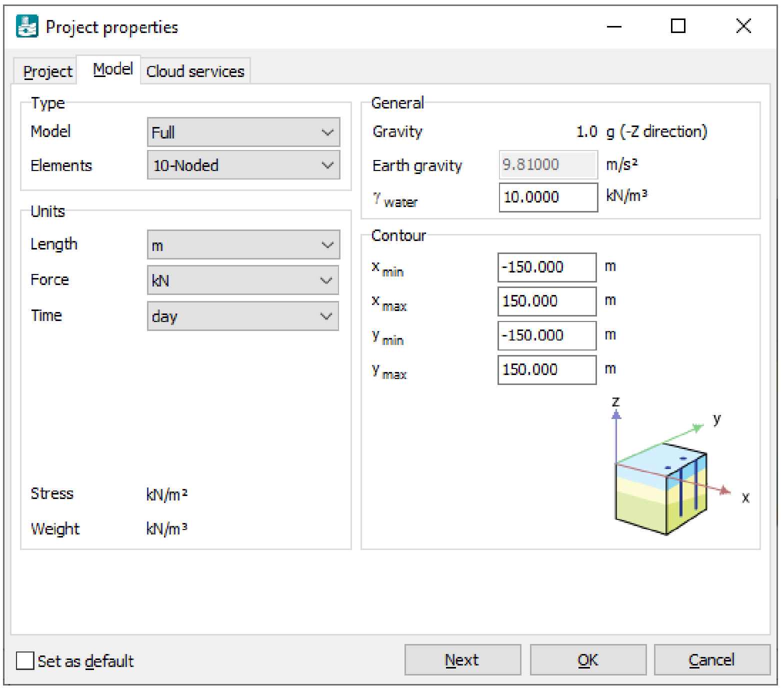


Figure 2: Model dimension definition in Project properties

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 7 soil layers from 3 m to -7.3 m, -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	**Silty and Medium Sand**	**Dense Sand**	**Calcarenite**	**Calcisiltite/ Siltstone**	**Mudstone**
Soil model	Linear Elastic	MC	MC	MC	MC
Drainage type	Drained	Drained	Drained	Drained	Drained
Unsaturated weight γ_{unsat}	18 kN/m^3	18 kN/m^3	19 kN/m^3	19 kN/m^3	19 kN/m^3
Saturated weight γ_{sat}	18 kN/m^3	18 kN/m^3	19 kN/m^3	19 kN/m^3	19 kN/m^3
Mechanical properties					
Young's modulus E_{ref}	30E3 kPa	50E3 kPa	300E3 kPa	1.7E6	1.7E6
Poisson's ratio ν	0.33	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
Initial stress properties					
Initial stress ratio K_0	Auto	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
Mechanical properties		
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
Initial stress properties		
Initial stress ratio K_0	Auto	Auto

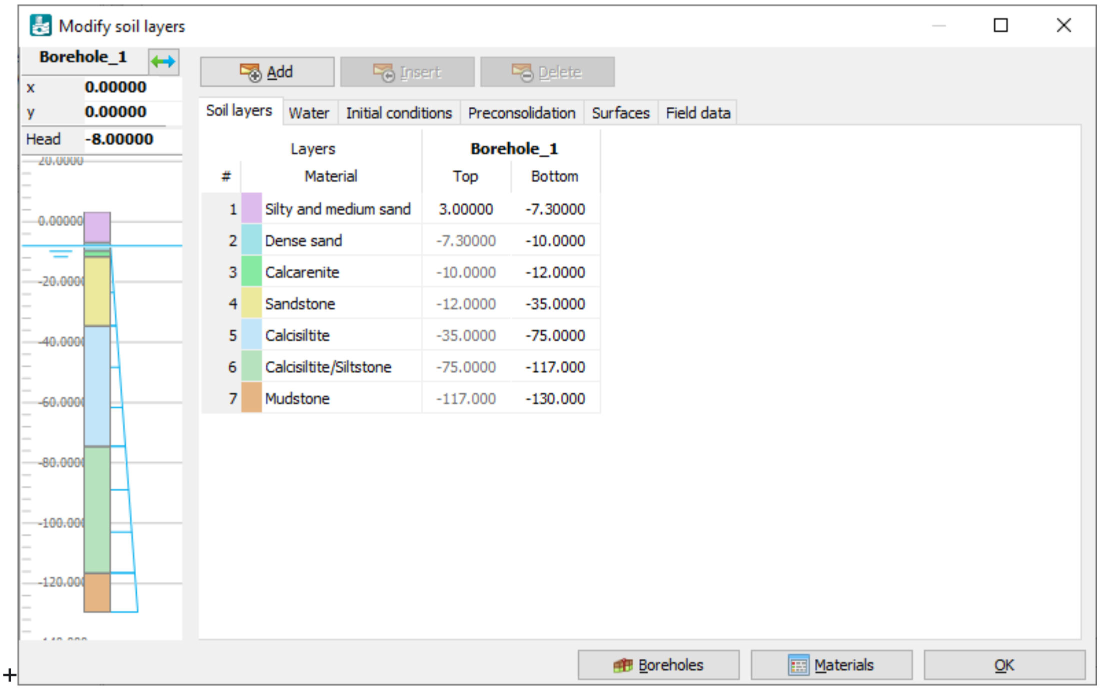


Figure 3: Borehole definition

A linear elastic material model is assigned to the upper silty and medium sand layer to facilitate the initial excavation stage down to –7.3 m. This approach allows the top layer to temporarily self-support its own weight without collapsing. While this behaviour is not realistic, it simplifies the model setup by removing the need to include a retaining wall system. Given that the sole objective of this analysis is to study the pile group foundation behaviour, we consider this assumption acceptable, as the self-weight of the top layer is still correctly accounted for.

Structures mode

Set the properties of the embedded beams that will be used to model the piles using values proposed in Table 2 and Table 3. The geometrical properties correspond to those of the 1.5 m × 2.8 m and 1.0 m × 2.8 m barrette cross-sections.

An excel spreadsheet entitled "*Command lines geometry.xlsx*" is provided. It contains for each barrette the following data:

- absolute top position of the barrette
- relative top position calculated from a user-defined horizontal translation vector (Shift X, Shift Y)
- the pile type (type I, type II or type III)
- orientation along global X or Y axis (0 and 1 respectively).

The corresponding relevant commands to be imported and read in from the *PLAXIS 3D Command Runner* are being highlighted in *Light blue* .(the commands in *Orange* correspond to the ones to be considered for a model creation using volume pile representation instead of embedded beams). Many sets of commands must be prepared to define the embedded beams elements, their orientation (as we are considering rectangular barrettes, the moment of inertia are different around local axis 2 and 3), their property and their applied point load the value of which is defined from the structural model and corresponding to the force in the pile spring using the calculated stiffness from the previous iteration. All commands generated in the Excel Spreadsheet have been copied into the log file *Pile_Group_foundation_EB.log.*

Table 2: Summary of embedded beam properties.

Material Name	Pile – Type I	Pile – Type II	Pile – Type III
Material type	Elastic	Elastic	Elastic
Unit weight	24 kN/m^3	24 kN/m^3	24 kN/m^3
Cross-section type	User-defined	User-defined	User-defined
A	4.2 m^2	4.2 m^2	2.8 m^2
I_2	2.744 m^4	2.744 m^4	1.829 m^4
I_3	0.7875 m^4	0.7875 m^4	0.2333 m^4
Stiffness *E*	20E6 kN/m^2	20E6 kN/m^2	20E6 kN/m^2
Axial skin resistance	Multi-linear	Multi-linear	Multi-linear
Multi-linear axial skin resistance	*8 rows*	*8 rows*	*8 rows*
Base resistance	100E3 kN	100E3 kN	100E3 kN

Table 3: Skin resistance evolution along embedded beams

	Pile Type I		Pile Type II		Pile Type III	
	Distance	Axial skin resistance	Distance	Axial skin resistance	Distance	Axial skin resistance
1	0 m	860 kN/m	0 m	860 kN/m	0 m	760 kN/m
2	2.7 m	860 kN/m	2.7 m	860 kN/m	2.7 m	760 kN/m
3	2.7 m	3440 kN/m	2.7 m	3440 kN/m	2.7 m	3040 kN/m
4	4.7 m	3440 kN/m	4.7 m	3440 kN/m	4.7 m	3040 kN/m
5	4.7 m	1935 kN/m	4.7 m	1935 kN/m	4.7 m	1710 kN/m
6	27.7 m	1935 kN/m	27.7 m	1935 kN/m	27.7 m	1710 kN/m
7	27.7 m	3440 kN/m	27.7 m	3440 kN/m	27.7 m	3040 kN/m
8	69.7 m	3440 kN/m	59.2 m	3440 kN/m	50.7 m	3040 kN/m

First start copying and pasting into the *Command runner* (accessible from *Expert → Run Commands* … menu item), the 134 commands corresponding to the creation of the embedded beam:

Pile_Group_Foundation_EB.log

```
Embeddedbeam    13.38   18.18   -7.300  13.38   18.18   -77
Embeddedbeam    5.66    16.63   -7.300  5.66    16.63   -77
Embeddedbeam    9.58    16.63   -7.300  9.58    16.63   -77
….
128 lines skipped
….
Embeddedbeam    -24.75  -28.15  -7.300  -24.75  -28.15  -60
Embeddedbeam    -16.45  -28.15  -7.300  -16.45  -28.15  -60
Embeddedbeam    -7.05   -28.15  -7.300  -7.05   -28.15  -60
```

Then start assigning *AxisFunction* to *Manual:*

Pile_Group_Foundation_EB.log

---

set line_1.AxisFunction "Manual"

set line_2.AxisFunction "Manual"

set line_3.AxisFunction "Manual"

….
128 lines skipped

….
set line_132.AxisFunction "Manual"

set line_133.AxisFunction "Manual"

set line_134.AxisFunction "Manual"

---

and set their manual axis to the proper direction depending on the orientation of each barrette:

Pile_Group_Foundation_EB.log

---

set line_1.AxisVectorX 1

set line_2.AxisVectorY 1

set line_3.AxisVectorX 1

….
128 lines skipped

….
set line_132.AxisVectorY 1

set line_133.AxisVectorY 1

set line_134.AxisVectorY 1

Attach the appropriate embedded beam property set using the following set of commands:

Pile_Group_Foundation_EB.log

---

set EmbeddedBeam_1.Material PileTypeI

set EmbeddedBeam_2.Material PileTypeI

set EmbeddedBeam_3.Material PileTypeI

….
128 lines skipped

….
set EmbeddedBeam_132.Material PileTypeIII

set EmbeddedBeam_133.Material PileTypeIII

set EmbeddedBeam_134.Material PileTypeIII

Rename all previously 134 created *line_i* by *Pile_i*

Pile_Group_Foundation_EB.log

```
rename line_1 "Pile1"
rename line_2 "Pile2"
rename line_3 "Pile3"
….
128 lines skipped
….
rename line_132 "Pile132"
rename line_133 "Pile133"
rename line_134 "Pile134"
```

Select all created piles and create a group named *All_Piles*. This group will be helpful during results post-processing for computing equivalent pile stiffnesses.

The creation of the point load system will be eased by renaming top embedded point consecutively by:

Pile_Group_Foundation_EB.log

```
rename point_1 "Loading_Point_1"
rename point_3 "Loading_Point_2"
rename point_5 "Loading_Point_3"
….
128 lines skipped
….
rename point_263 "Loading_Point_132"
rename point_265 "Loading_Point_133"
rename point_267 "Loading_Point_134"
```

The loads will be applied directly on the pile head as no raft will be modelled in the PLAXIS 3D geotechnical model (the raft will be part of the structural model). Therefore, all embedded beam top point connection type should be set to *Free.*

Pile_Group_Foundation_EB.log

```
set Pile1.EmbeddedBeam.Connection “Free”
set Pile2.EmbeddedBeam.Connection “Free”
set Pile3.EmbeddedBeam.Connection “Free”
….
128 lines skipped
….
set Pile132.EmbeddedBeam.Connection “Free”
set Pile133.EmbeddedBeam.Connection “Free”
set Pile134.EmbeddedBeam.Connection “Free”
```

Finally top point load can be created:

Pile_Group_Foundation_EB.log

```
pointload Loading_Point_1
pointload Loading_Point_2
pointload Loading_Point_3
….
```

128 lines skipped

```
….
pointload Loading_Point_132
pointload Loading_Point_133
pointload Loading_Point_134
```

The exact point load values will be set during phase settings definition as explained earlier.

Some additional surfaces must be created for the sake of identifying the excavated soil mass as well as providing some partitioning means to the geometry:

- Start defining a top surface at top model level by creating a polygon using the following surface points:
  - (-30.5, 30.5, 3),
  - (30.5, 30.5, 3),
  - (30.5, -30.5, 3),
  - (-30.5, -30.5, 3).
- The initial polygon *Polygon_1* should be copied multiple times in the downwards direction to represent the different characteristic levels that will be introduced (excavation, mesh refining zones). Select *Polygon_1* and create arrays of "*1D, in z direction*" shape with number of columns set to 2 each time with the following distance between columns:
  - -10.3 m (excavation level)
  - -63 m (bottom of pile type III level)
  - -69.5 m (bottom of pile type II level)
  - -80 m (bottom of pile type I level)
- *Decompose* the latest polygon (*Polygon_5*) *into outlines*
- Extrude the four previously created lines upwards using a (0, 0, 80) extrusion vector.
- Select the latest created polygon Polygon_5 and reset its corner point coordinates to:
  - (-40, 40, -77),
  - (40, 40, -77),
  - (40, -40, -77),
  - (-40, -40, -77).
- *Decompose* again the updated polygon (*Polygon_5*) *into outlines*
- Extrude the four previously created lines upwards using a (0, 0, 80) extrusion vector.

Final model geometry is presented in Figure 4.

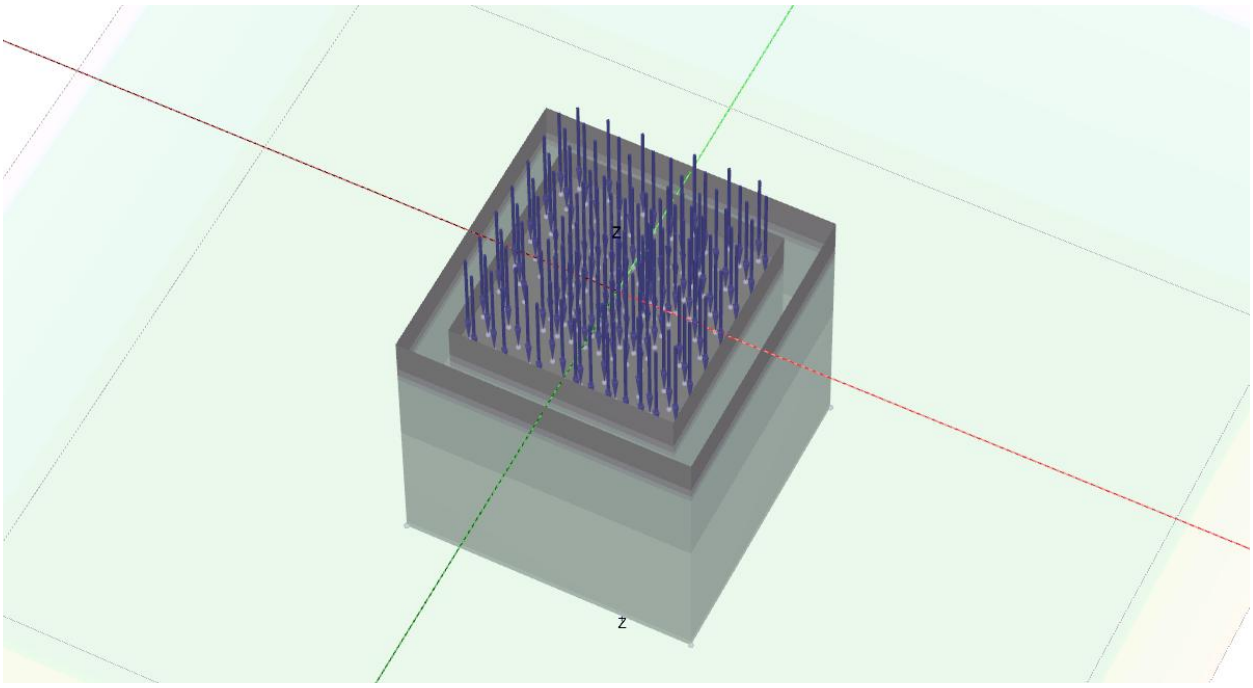


Figure 4: Final geometry presentation

Mesh mode

Go to the *Mesh* mode and select the 8 top inner soil clusters through which the pile group has been defined and set their respective coarseness factor to 0.15 (see Figure 5(a)).

Click on *Generate* mesh and set the *Element distribution* to *Medium*. The generated mesh is presented in Figure 5(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.

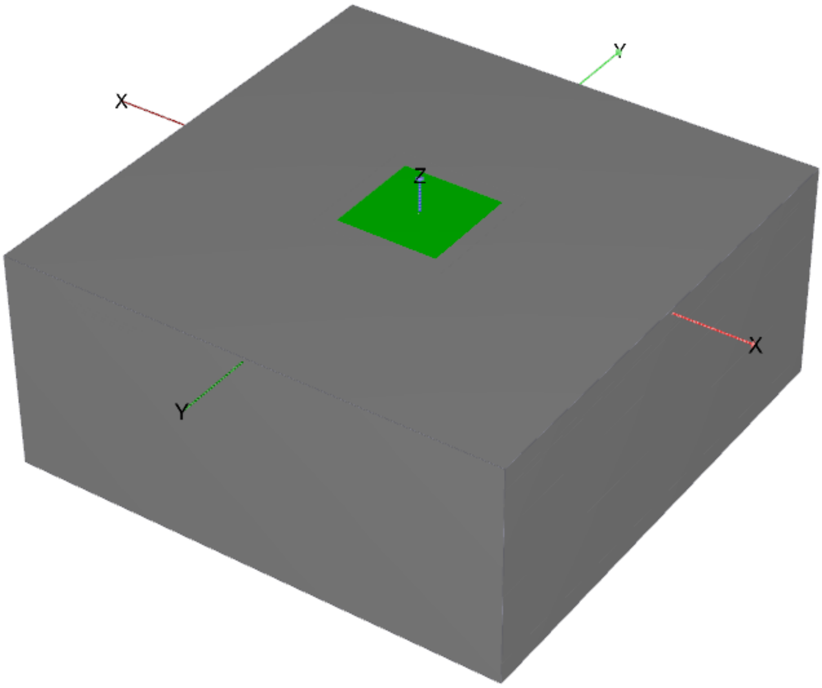
Phase 1: Excavation

Create a first phase and name it *Excavation*.

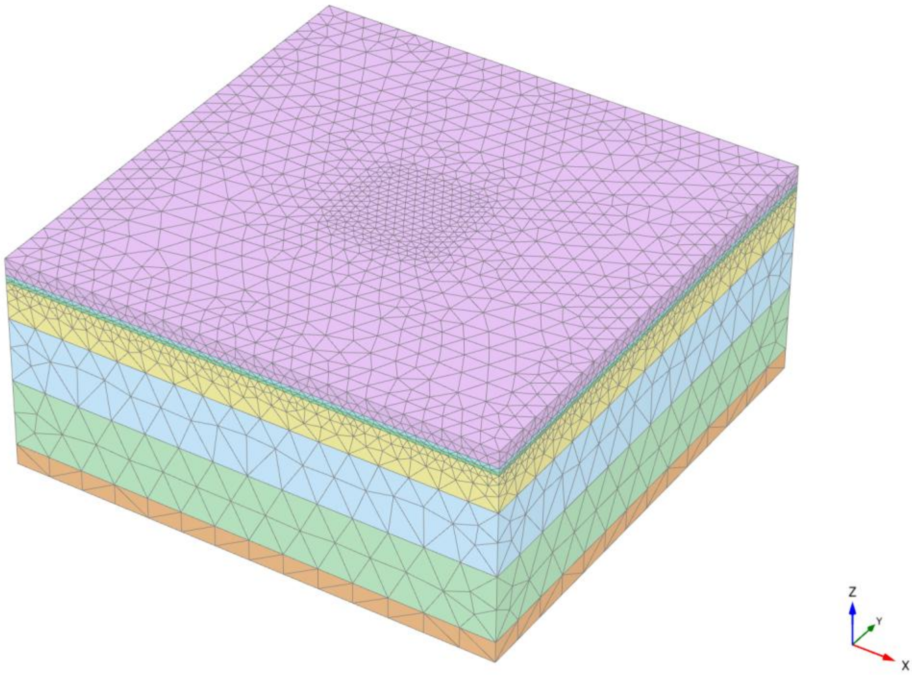
Select the top center soil cluster (from z = 3 m down to z = -7.3 m and deactivate it. Also set its *WaterConditions* to *Dry* as well.

Phase 2: Pile installation

Create a second phase entitled *Pile installation* in which one simply needs to activate all embedded beams.



(a) Mesh refinement



(b) generated mesh

Figure 5: Mesh definition

Phase 3: Dead Load - SLS

In the excel spreadsheet *Loadcases.xlsx*, the load values have been defined for each pile based on the result of the structural model from the previous iteration. Both Volume piles and Embedded beams configurations have been considered but only the embedded beam approach will be retained here. We have also generated a set of commands for the ULS configuration by multiplying all load values by 1.35 (but it wont be used in this exercise). All commands generated in the Excel Spreadsheet have also been copied into the log file *Pile_Group_foundation_EB.log*

Create a third phase entitled *Dead Load - SLS*:

- Start with activating *Reset displacements to zero*,
- Activate all point loads,
- Set their respective value by reading in the relevant commands from the *Loadcases.xlsx* spreadsheet.

Pile_Group_Foundation_EB.log

```
set PointLoad_5_1.Fz phase_3 -58412.585
set PointLoad_6_1.Fz phase_3 -64309.0385
set PointLoad_7_1.Fz phase_3 -62515.3037
….
```

128 lines skipped

```
….
set PointLoad_2_1.Fz phase_3 -52694.8674
set PointLoad_3_1.Fz phase_3 -53238.7397
set PointLoad_107_1.Fz phase_3 -47872.4236
```

Start the calculation by clicking on *Calculate*. Once the model is complete, save the project as *3D Foundation Analysis.p3d*

3.

Main Calculation Results

Once the calculation has completed, save the PLAXIS 3D model for instance as 3D Foundation Analysis.p3d and then open the *Output* program by viewing the results for the phase *Dead Load* - *SLS* [Phase_3].

Deformed mesh and vertical displacement field – SLS Design check

After filtering out soil elements for which coordinates Y <0 (using the *Filter* command from the *Geometry* menu), one can evaluate deformed mesh as shown in Figure 6.

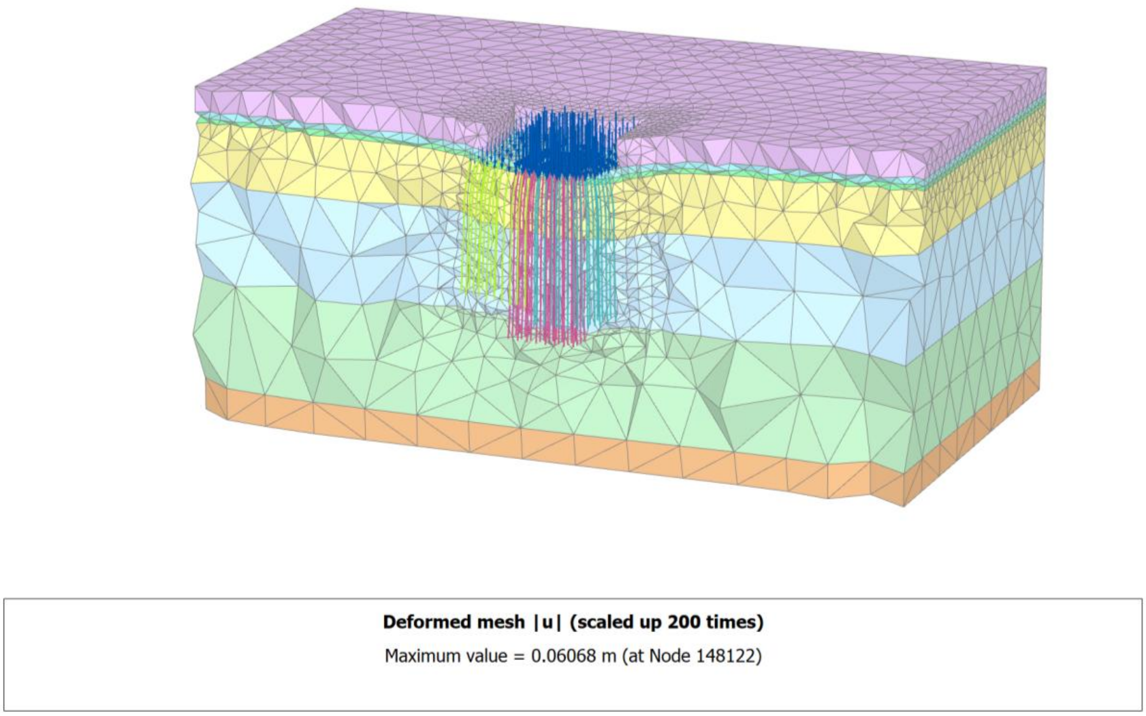


Figure 6: Deformed mesh for vertically loaded pile under dead weight

Vertical displacement field

In the same way, one can also evaluate the vertical displacements as shown in Figure 7.

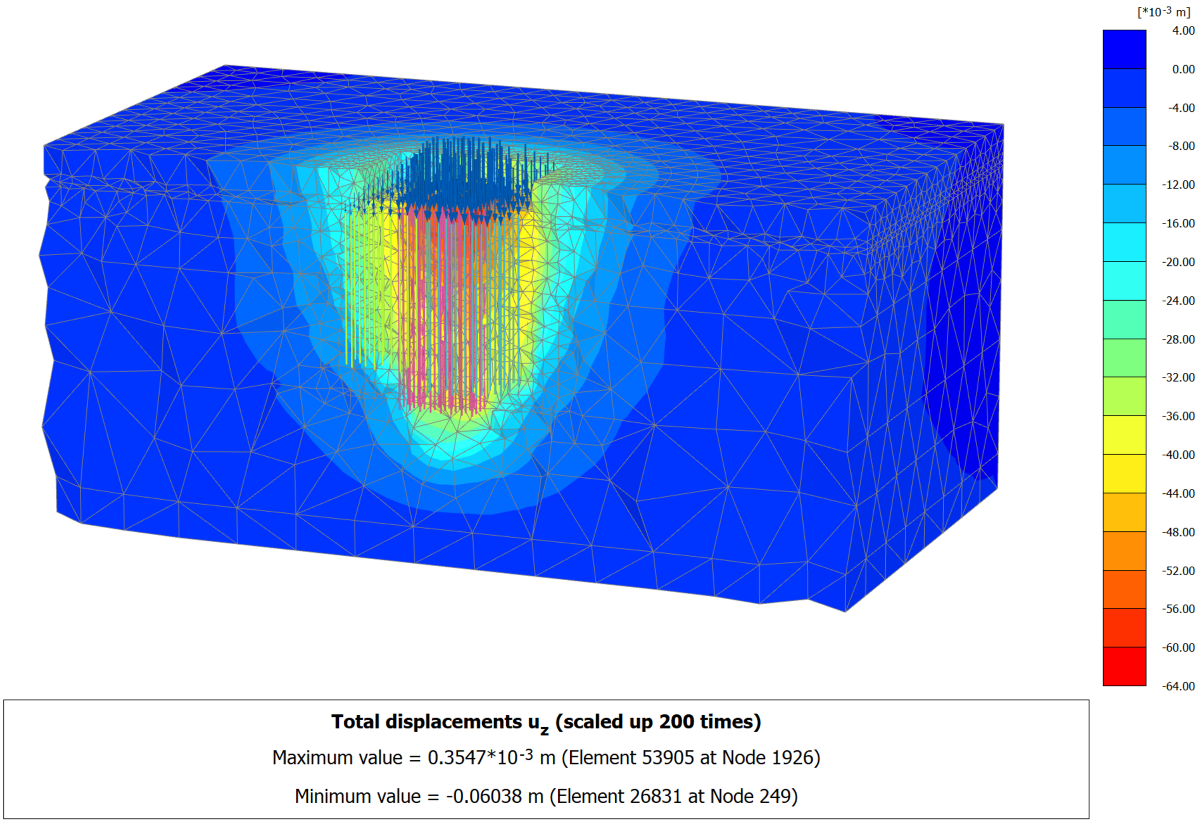


Figure 7: Vertical displacement contour plots for vertically loaded pile under dead weight

The maximum deformation under the raft will approximately be 6 cm which is in the range of what is being accepted for such type of structures

One can also check the differential settlement of the raft. For that:

- First make visible all soil elements (clicking on the eye symbol from the *Geometry* item of the model tree)
- By using the *Toggle visibility* tool, hide all soil clusters except the one under the raft. This will reproduce the view shown in Figure 8.

The differential settlement can be roughly evaluated by considering the difference between the maximum and minimum vertical displacement over half the raft diagonal length which would provide

$$A = (0.06038 - 0.01815)/\left(\sqrt{30.5^2 + 30.5^2}/2\right) \approx 1.96\,‰$$

which also falls right below usual threshold values for such type of structures.

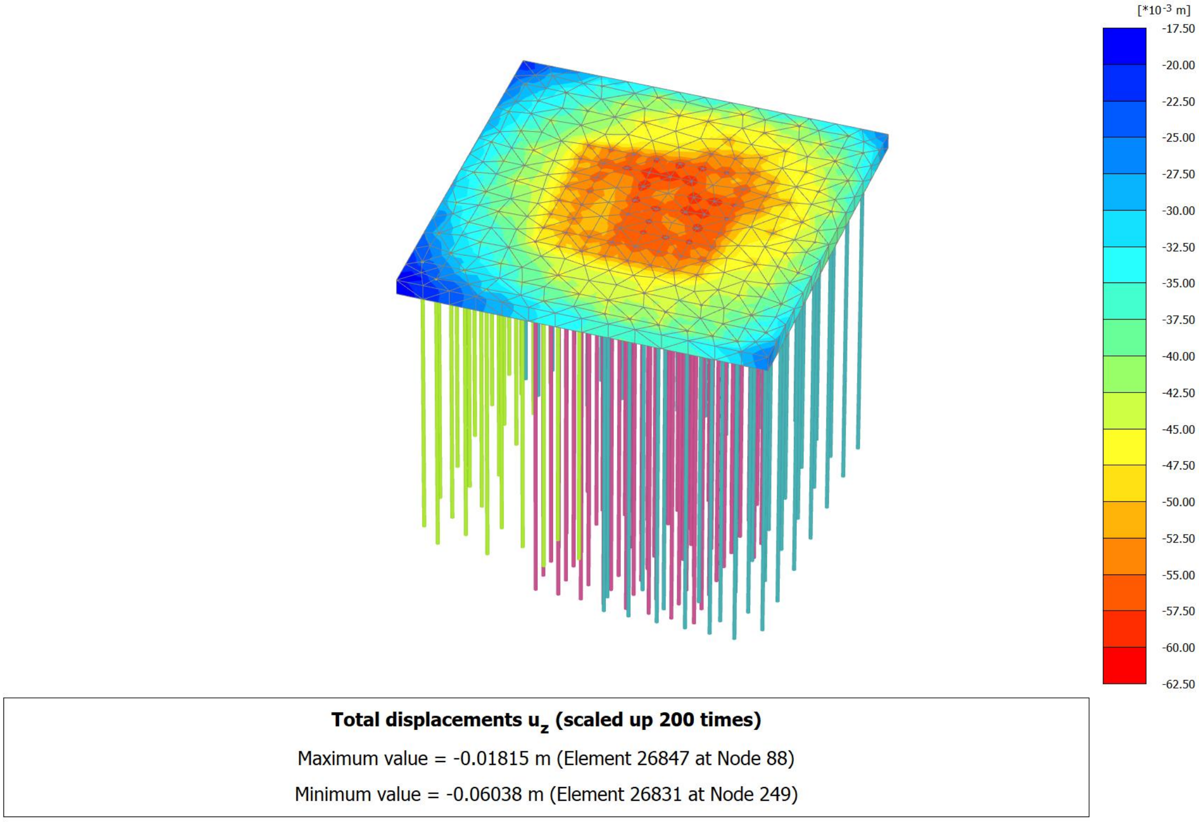


Figure 8: Soil vertical displacement just under the raft

Equivalent pile stiffness in ParaView

The structural model, which includes both the building and the raft, must account for the contribution of each individual barrette. This is typically done by representing each barrette as an equivalent unidirectional spring. The stiffness of these springs is usually determined through an iterative process between the structural model and the geotechnical model we have just discussed. In this context, it is essential for geotechnical engineers to provide the corresponding equivalent pile stiffness values—this aspect will be addressed in this chapter.

PLAXIS supports export of results in **VTK** format (Visualization Toolkit), which is compatible with ParaView. This allows users to perform high-quality 3D post-processing, including advanced rendering, custom animations, slicing, volume plots, and more in addition to its high-quality visualization capabilities.

ParaView is also a powerful tool for generating user-defined results based on primary outputs. This makes it particularly useful for advanced post-processing tasks that go beyond the standard functionalities of PLAXIS Output.

More specifically, when evaluating **equivalent pile stiffness**, which is defined as the ratio of the **axial force in the pile** to the **displacement at the pile head**, ParaView allows users to compute this directly by combining the exported primary results.

In PLAXIS 3D Output, select *Export to ParaView* (*Tech. Preview*) from the *File* menu. *The Export to ParaView* dialog box pops up as shown in Figure 9. Note that the creation of the temporary database can take up to a couple of minutes to prepare the data in the proper format.

From the *Nodal results* category, select both *Displacement* and *Normal force 1* and click on *Export.* Select a folder where the ParaView files should be created.

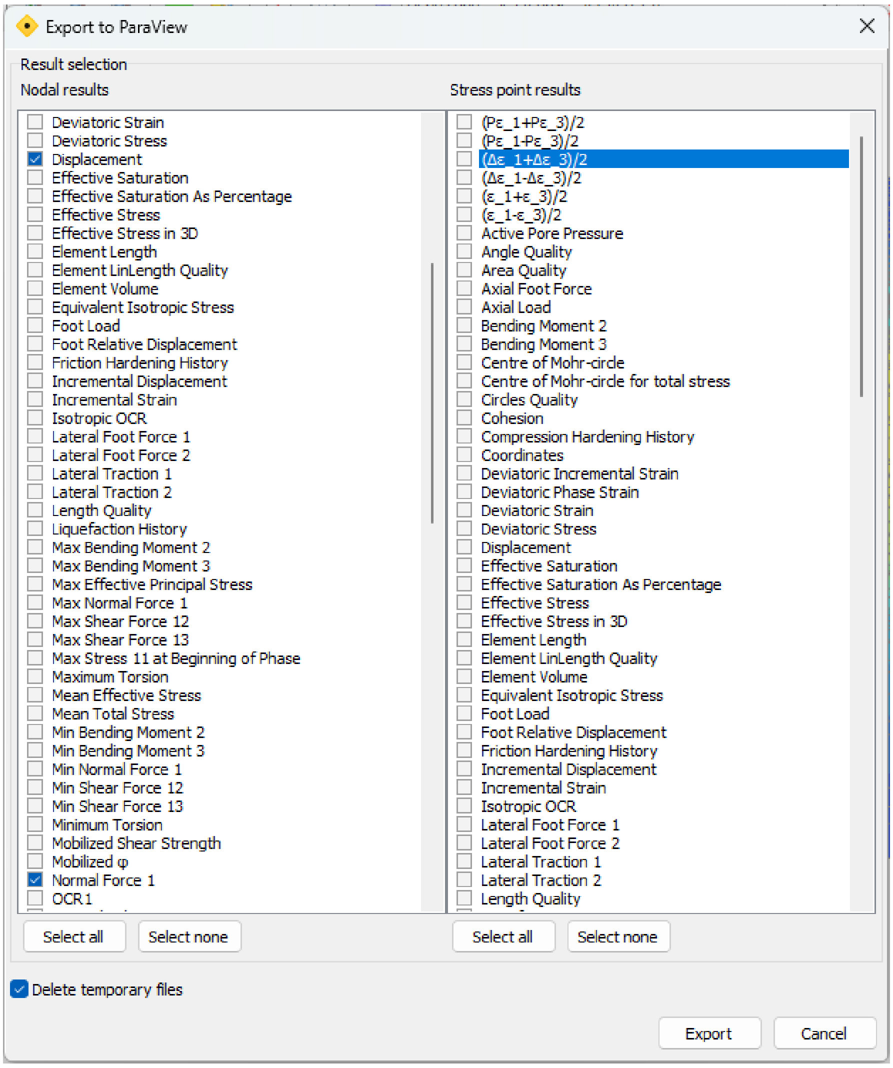


Figure 9: Export to ParaView dialog box

The creation of the ParaView files may require some time (which is increasing with the number of results selected to be exported and the number of phases the PLAXIS model has calculated)

Once the export is complete, open ParaView (available as a free download from the official website: https://www.paraview.org/download/). Then, load the .vtu file generated for the structural elements.

This file is located in the directory:

`3D Foundation Analysis - ParaViewExport/1_4_0_Dead Load - SLS [Phase_3]_data_nodal`

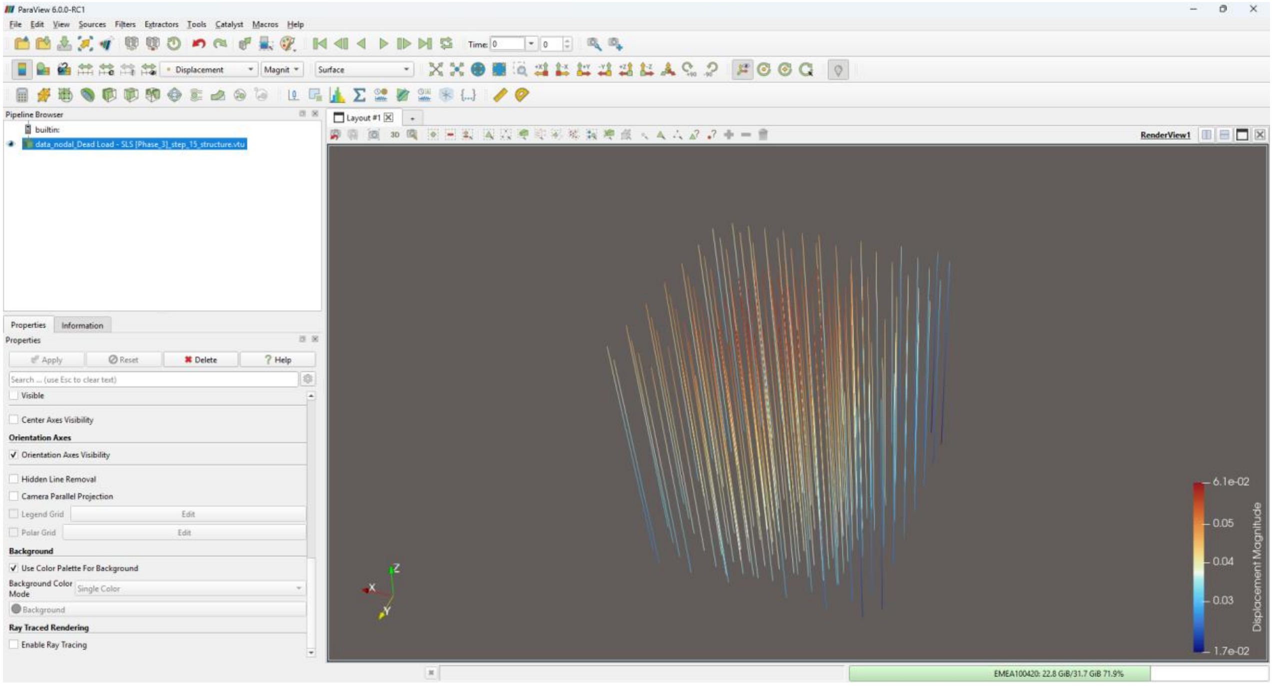
Look for a file named:

`data_nodal_Dead Load SLS [Phase_3]_step_xxx_structure.vtu`

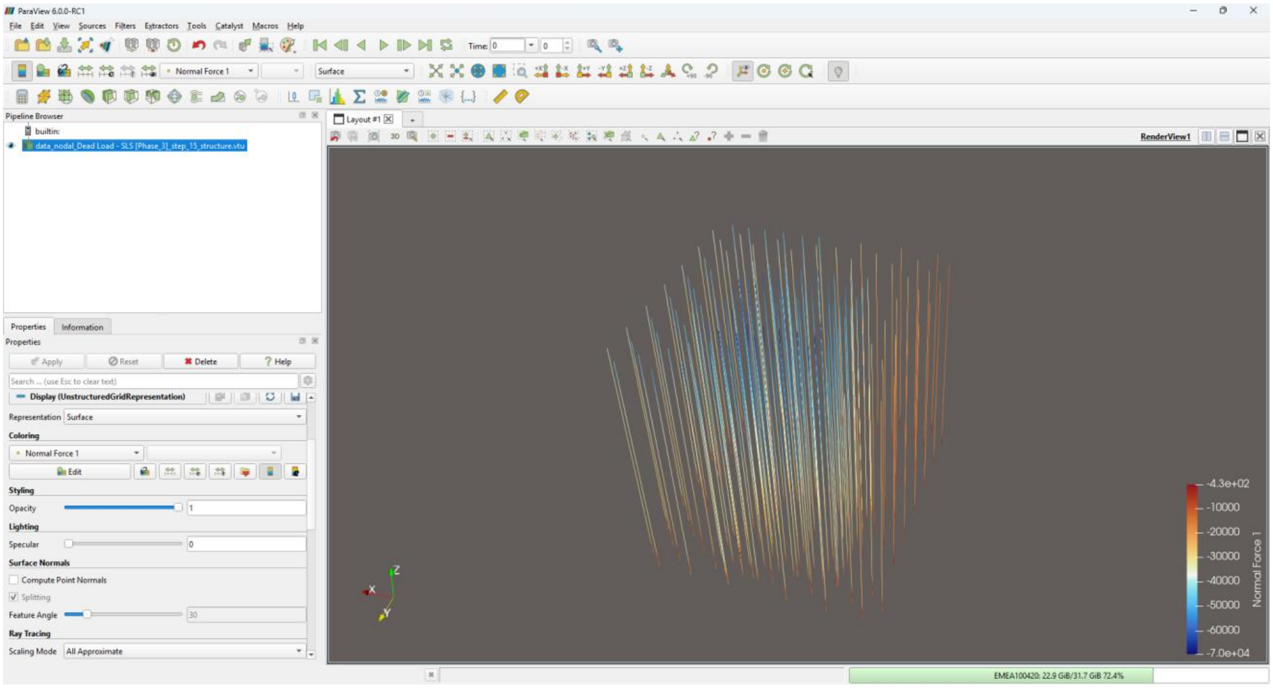
where `xxx` corresponds to the final step number of *Phase_3*.

In ParaView make sure the data are visible, by clicking on the visibility symbol  associated with the previously open data file in the *Pipeline Browser*

In the *Properties* tabsheet, change the data to be displayed from *Solid Color* to *Displacement/Z* and *Normal Force 1* from the *Display (UnstructuredGridRepresentation)/Coloring* section as shown in Figure 10(a) and Figure 10 (b) respectively.



(a) Displacement Z



(b) Normal Force 1

Figure 10: Embedded beam result presentation in ParaView as Surface representation

The strategy to obtain the pile equivalent pile stiffness is to first select the top nodes of the pile and

First starts changing the presentation of the *Display* from *Surface* to *Points* and set the camera view such as to obtain a lateral view of the pile group as shown in Figure 11 .

Then *Select Points Through(g)*  from the active layout toolbox and draw a selection polygon just around all top nodes of the piles.

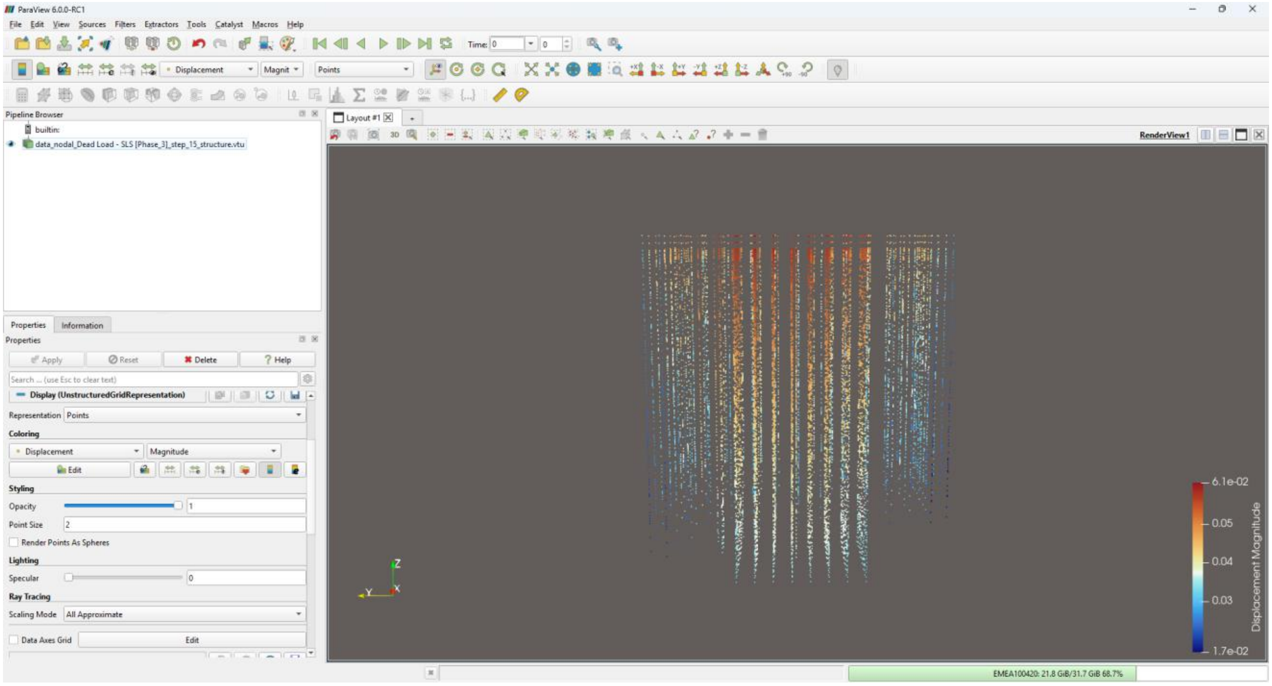


Figure 11: Displacement Z embedded beam result presentation in ParaView as Points representation

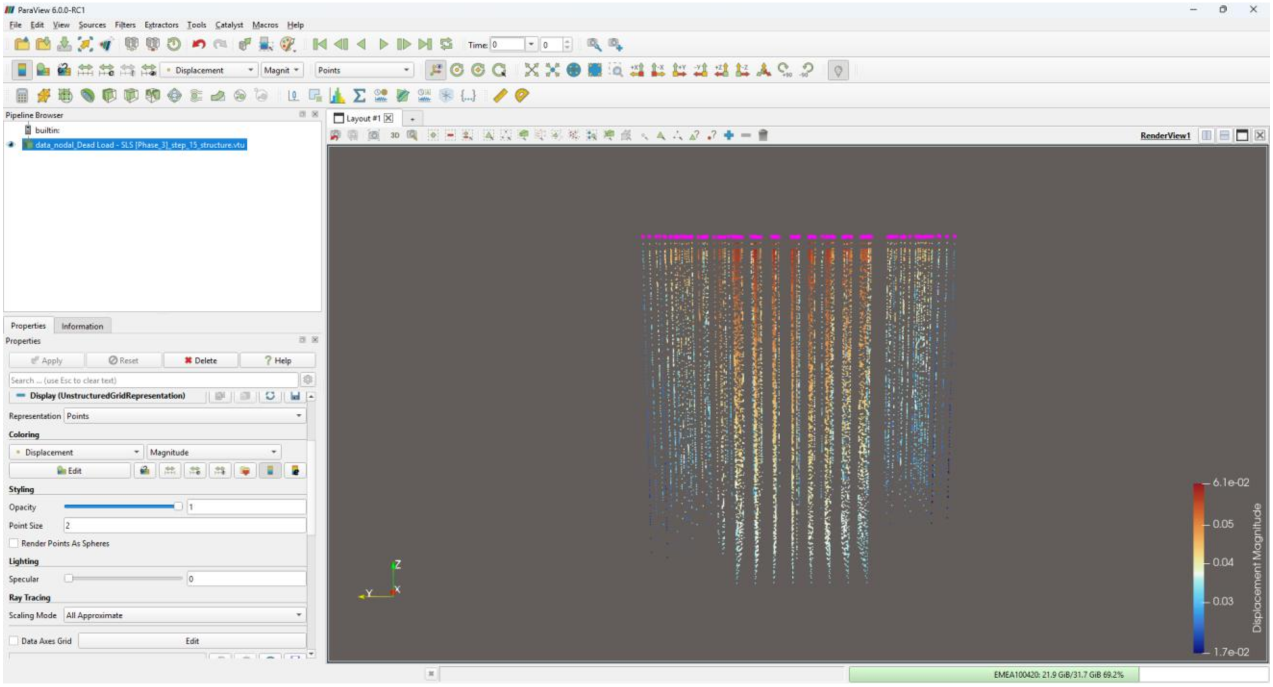


Figure 12: Selected pile top nodes in ParaView

Click on *Extract Selection* to create a new data set. Click on *Apply* in the *Properties* tabsheet to refresh the visualization. Make sure to turn off visibility of the original data set and turn it on for the new one.

One may also consider changing the presentation of the data on the new data set to *Points* and increasing the *Point Size* to 5 as shown in Figure 13

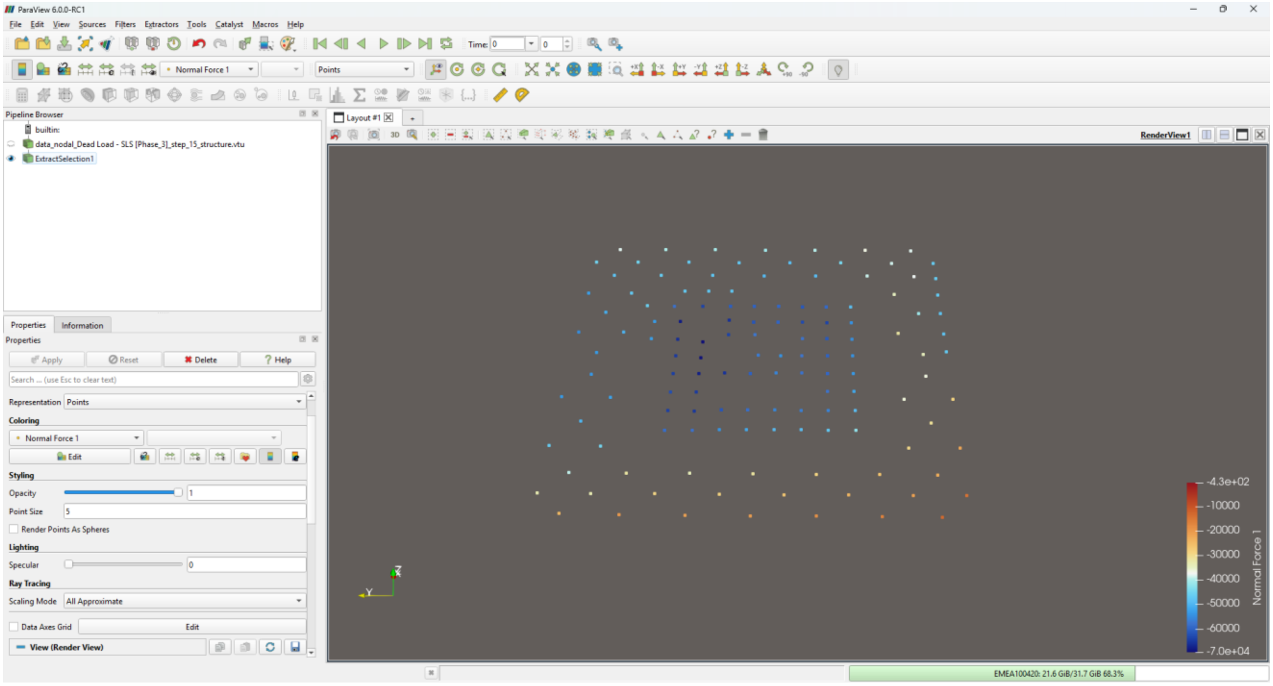


Figure 13: *Normal Force 1* Points representation on *Extractselection* dataset

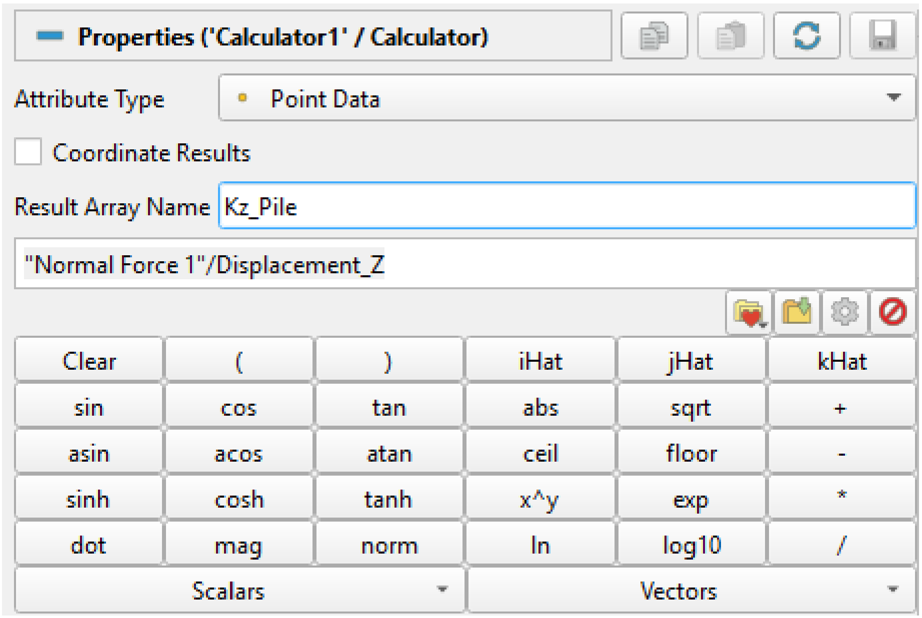


Figure 14: Creation of a new result array *Kz_Pile*

Make sure the *ExtractSelection* dataset is selected in the *Pipeline browser* and click on the *Compute new attributes array* button  followed by *Apply* in the *Properties* tabsheet.

In the *Properties* section of the new created *Calculator* dataset enter the following expression: "Normal Force 1"/Displacement_Z both item available from the *Scalars* dropdown menu and name the new Result Array *Kz_Pile* (see Figure 14). Click *Apply*

One can now visualize the calculated the new result (as *Point Gaussian* representation for instance) and query any value by using the *Hover point on* tool as shown in Figure 15.

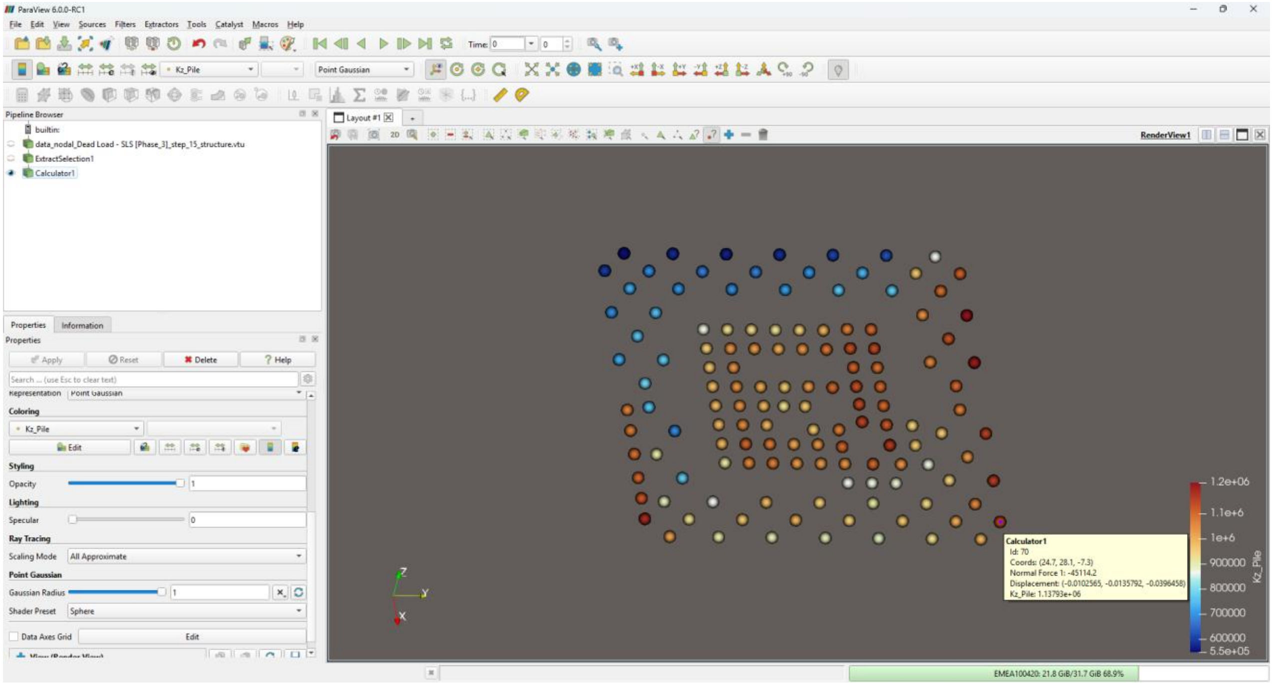


Figure 15: *Kz_Pile* Point Gaussian representation on *Calculator* dataset