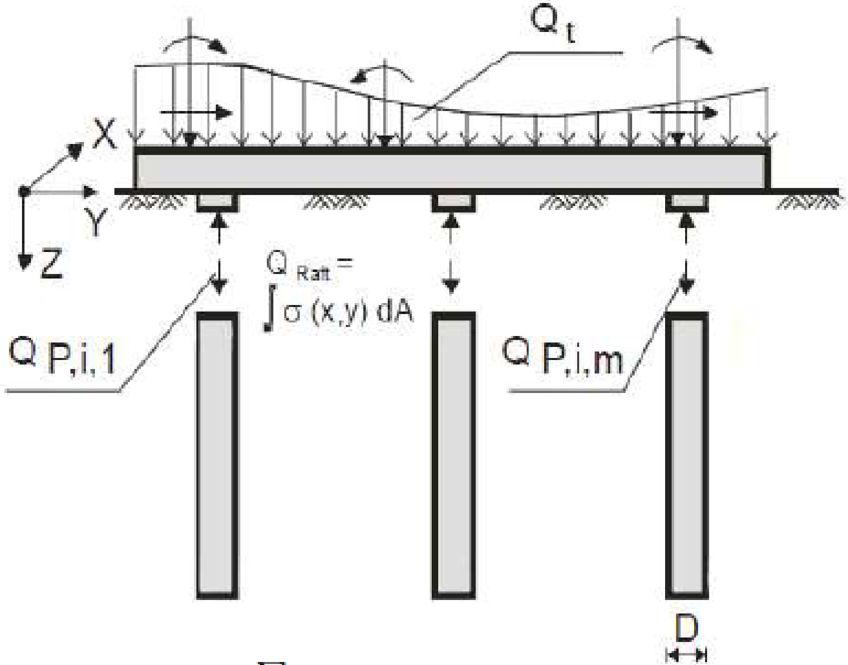
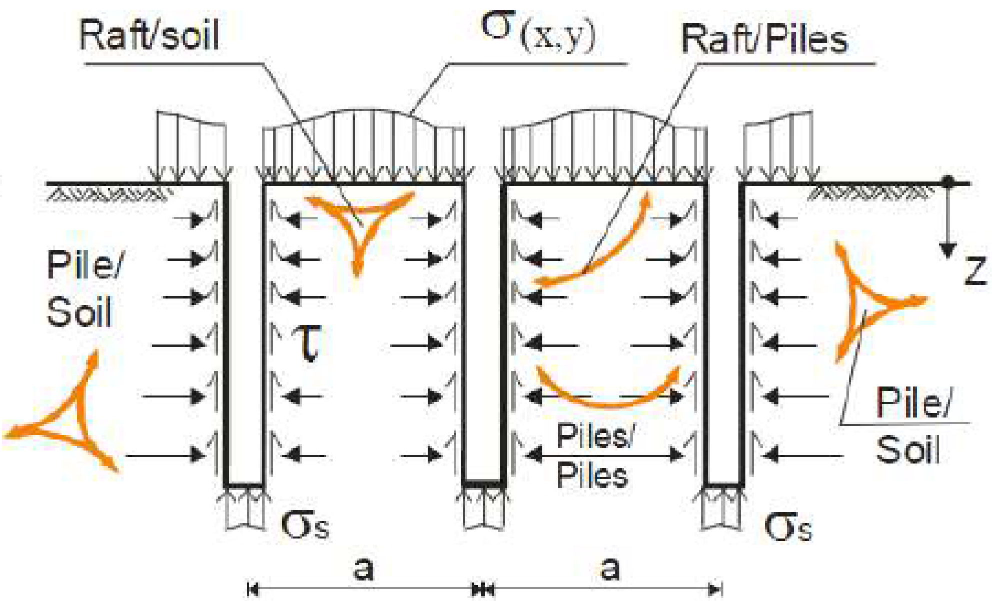
3D Piled Raft Foundation

Integrated Solver Optimized for the next generation 64-bit platform

Finite Element Solutions for Geotechnical Engineering

Bearing Behavior of a Piled Raft





$$Q_{tot} = \Sigma\, Q_P + Q_R$$

$$Q_P = Q_b + Q_s$$

$$Q_R = \int \sigma(x,y)\, dA$$

$$Q_{tot} \geq \eta \cdot \Sigma\, S_{tot}$$

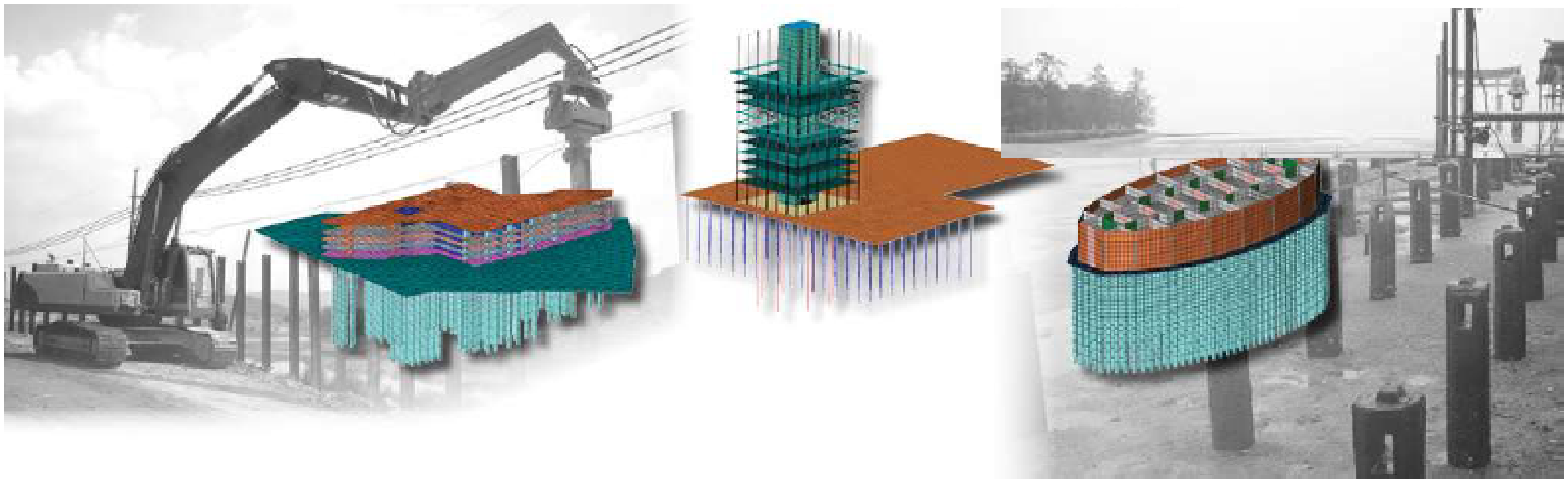
Interaction influences:

- Pile-Soil interaction
- Pile-Pile interaction
- Raft-Soil interaction
- Pile-Raft interaction

Pile Modeling in GTS NX

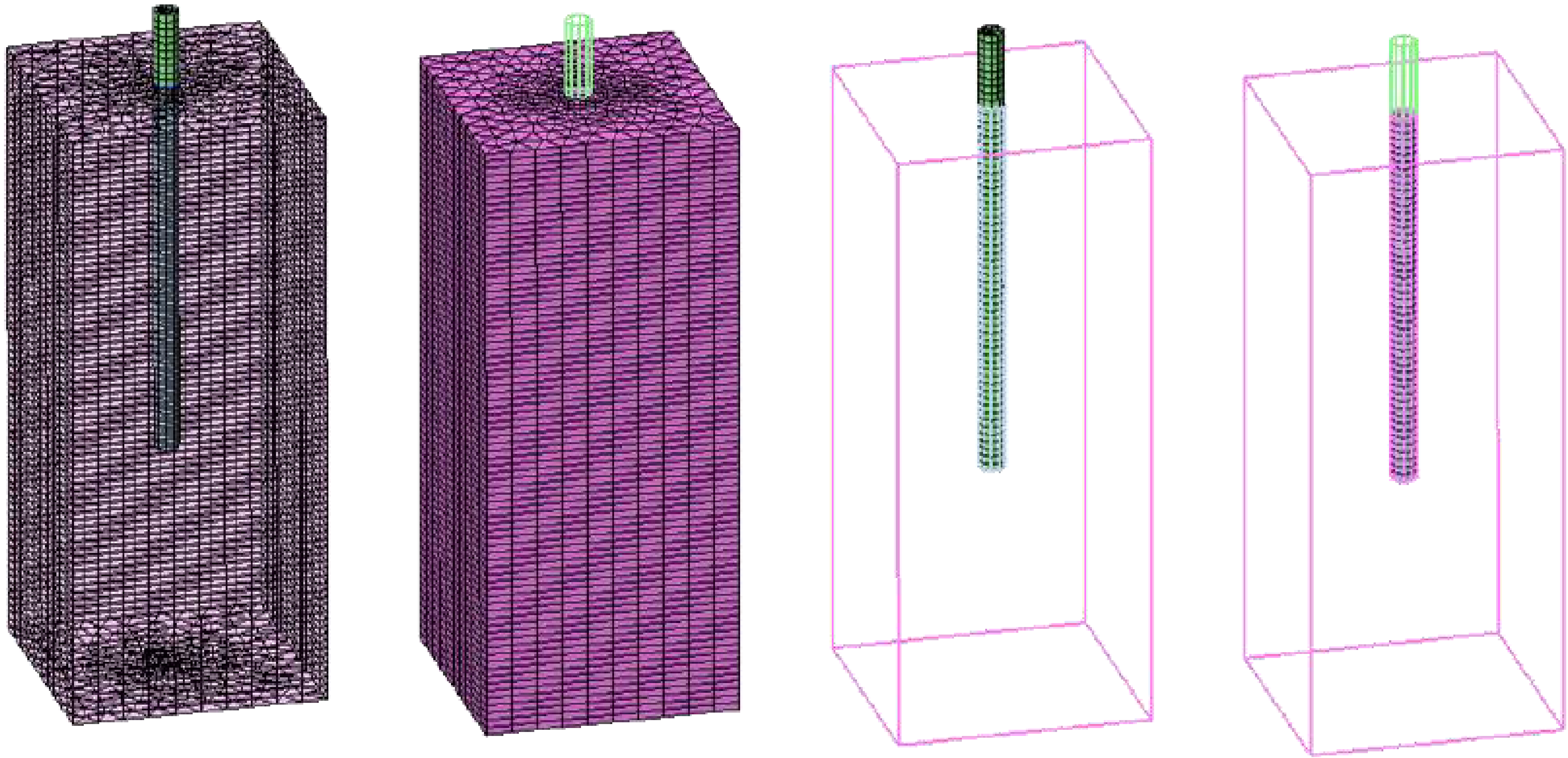
Three pile modeling approaches are available:

- Solid Element Model
- Beam-Solid Connectivity Model
- Line-to-Solid Interface Model (embedded pile)



Solid Element Model

GTS NX

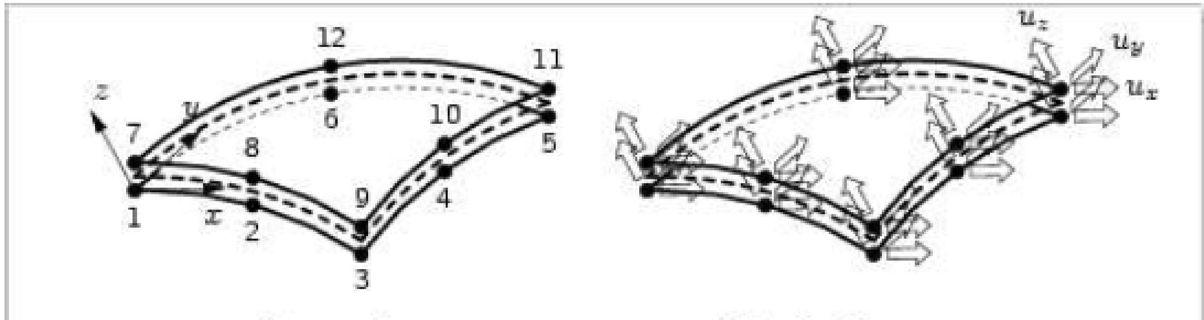


Model = Soil (solid) + Pile (solid) + Interface (surface)

Nodal connectivity is required on pile outer surface

Solid Element Model

Surface interface elements for solid-to-solid connection:



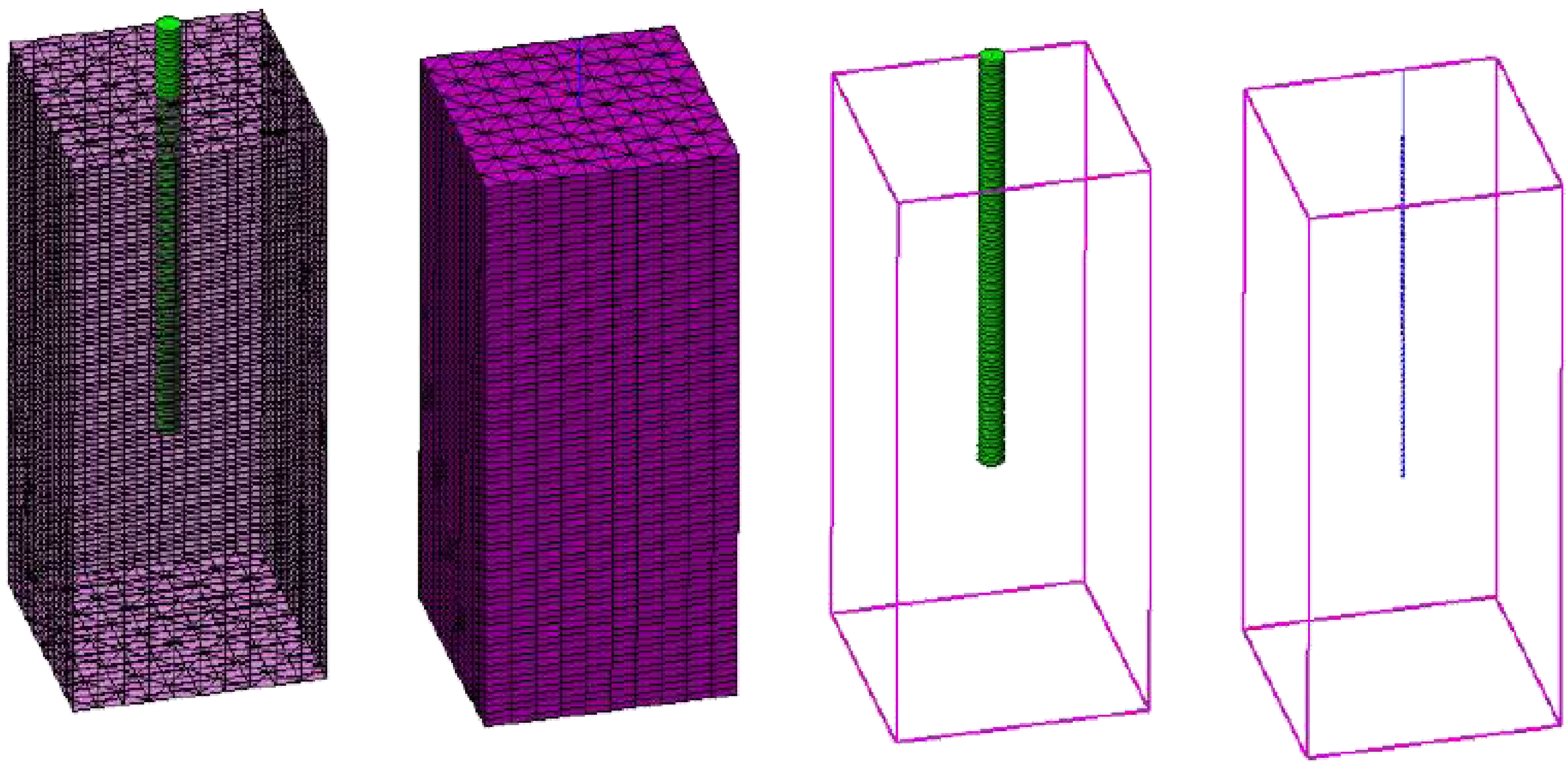
(a) topology (b) displacements

Points of attention for solid element models:

- Model definition and mesh-generation could be elaborative for large number of piles
- Many elements in model → large computation times
- Pile forces and moments are not directly available in post-processing
- Interface behavior: elastic, nonlinear elastic, coulomb friction, and user-supplied material

Beam-Solid Connectivity Model

GTS NX



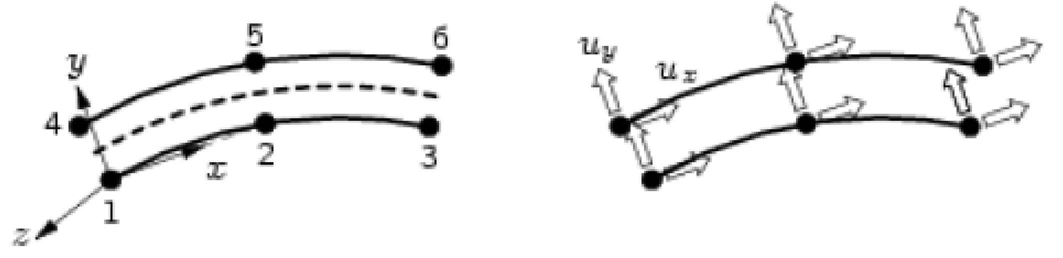
Model = Soil (solid) + Pile (beam) + Interface (line)

Nodal connectivity is required along pile length

Beam-Solid Connectivity Model

GTS NX

Line interface elements for beam-to-solid connection:

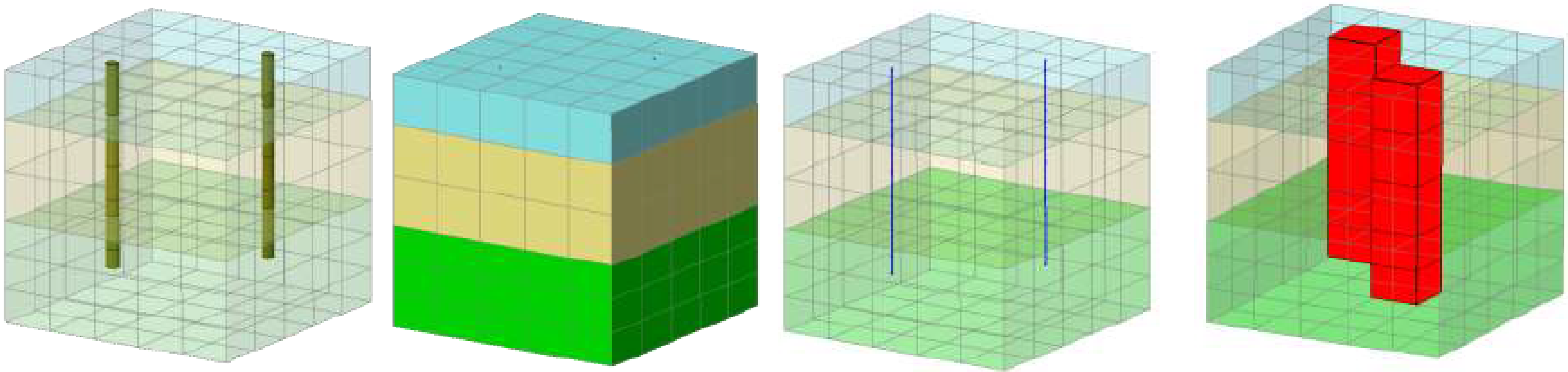


(a) topology (b) displacements

Points of attention for beam-solid element models:

- The nodal compatibility requirement makes geometry modeling and meshing of the soil elaborative.
- For piled rafts with large number of piles, this technique produces large models → large computation times

Line-to-Solid Interface Model



Model = Soil (solid) + Pile (beam) + Interface (line-to-solid)

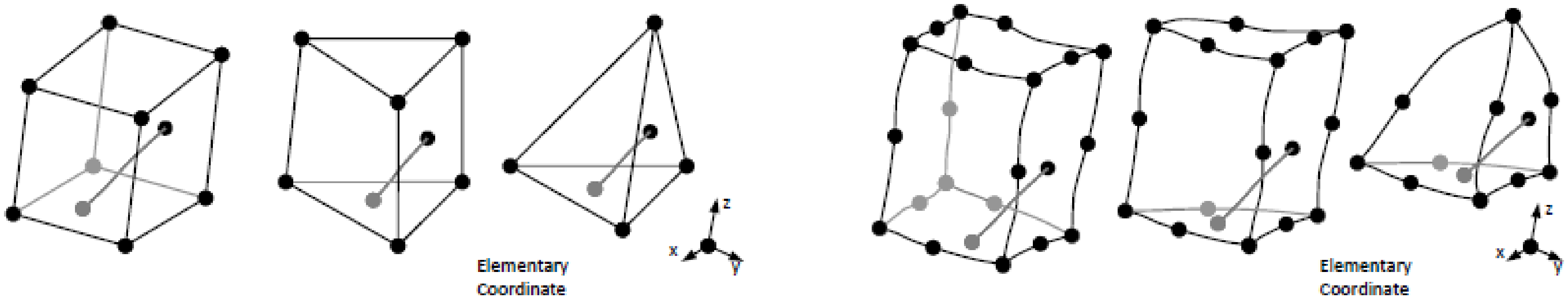
No nodal connectivity required. => well-suited for PR

Sadek & Shahrour (2004):
A three dimensional embedded beam element for reinforced geomaterials
Shear interaction between beam element and surrounding soil.

Line-to-Solid Interface Model

GTS NX

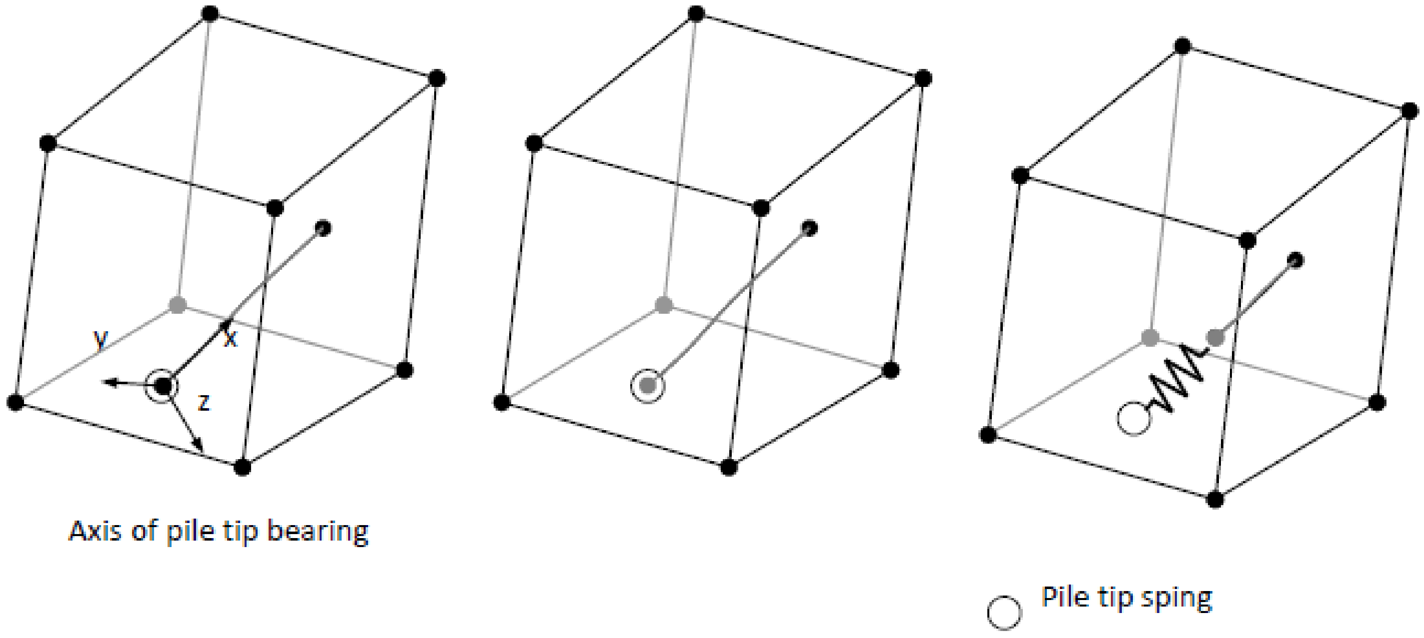
Line-to-solid interface elements for beam-to-solid connection:



Characteristics of line-to-solid interface modeling

- Pile and soil geometries and meshes can be specified independently
- Intersections of line and solid elements are calculated automatically
- Nonlinear friction-slip properties for line-solid interface elements
- Mesh refinement requirements for the soil are minimum
  → reduced computation time

Point-to-Solid Interface Model



Model = Soil (solid) + Pile (beam) + Interface (point-to-solid)

Characteristics of point-to-solid interface modeling

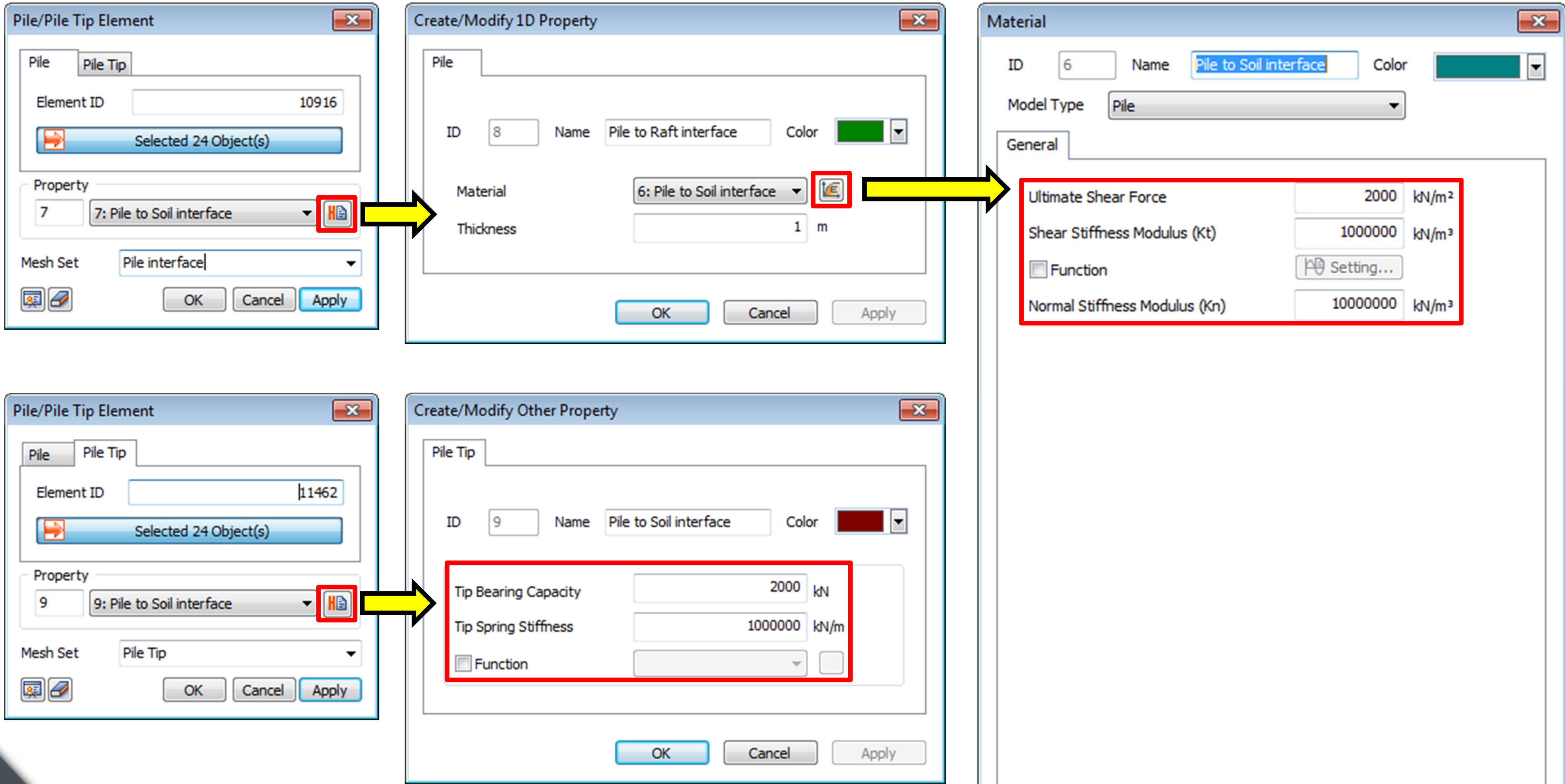
- The pile tip can be arbitrarily placed in the solid element automatically
- Nonlinear properties for point-solid interface elements

Line-to-Solid Interface Elements

GTS NX

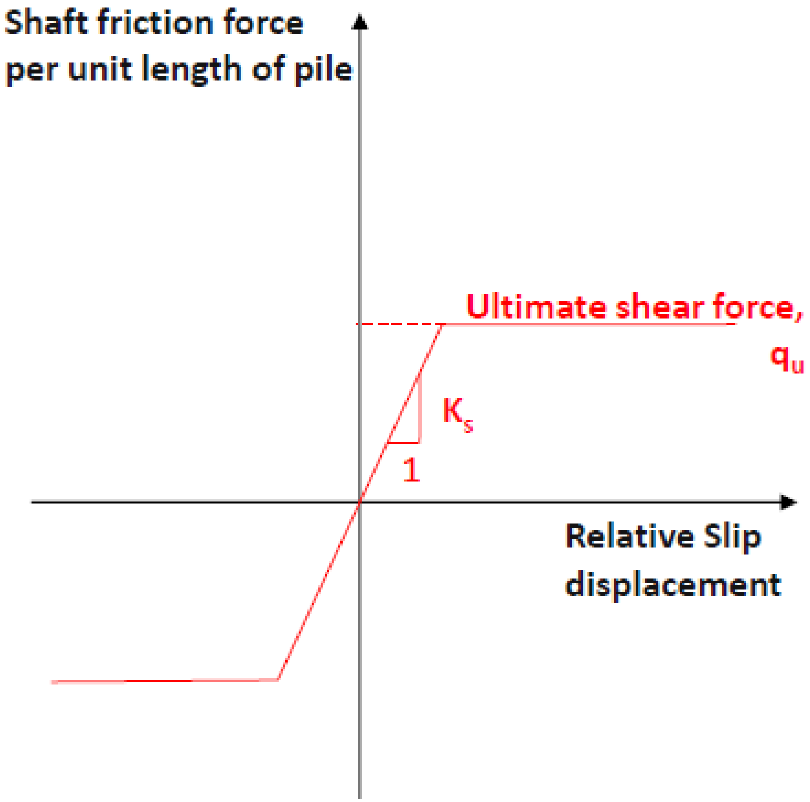
Pile bearing capacity is an input and not a result of FEM calculation.

→ To check global pile response from soil modelling and pile-soil interaction (Displacements, bending moments, axial forces, shaft friction, tip force, etc.)



Pile Element Parameters

GTS NX



Input parameters:

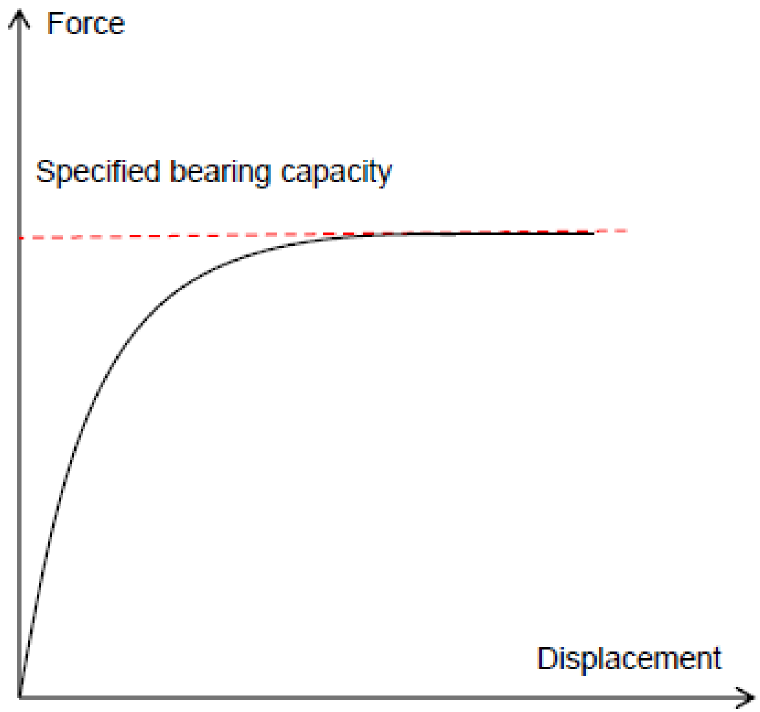
- **Ultimate shear force, q_u** [kN/m]
per unit length of the pile, at reference depth.
- **Shear Stiffness Modulus, K_s** [kPa]
Linear elastic penalty stiffness of the interface in the longitudinal direction of the pile.
- **Normal Stiffness Modulus, K_n,K_t** [kPa]
Linear elastic penalty stiffness of the interface in the transversal direction.

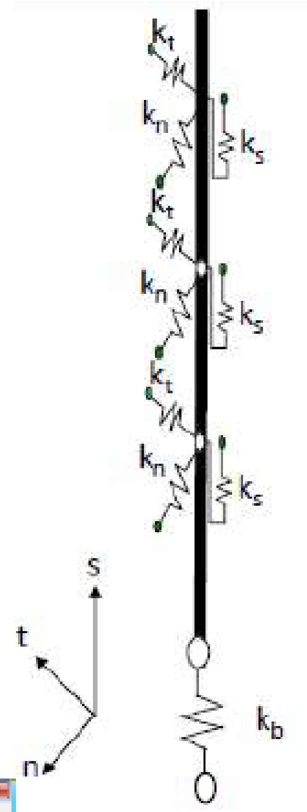
These input parameters are best extracted from SLT results after separating shaft friction and base bearing behavior from the total response

Pile bearing capacity is input in pile elements and not a result of the calculation!

=> Deformation behavior

Pile Element Parameters



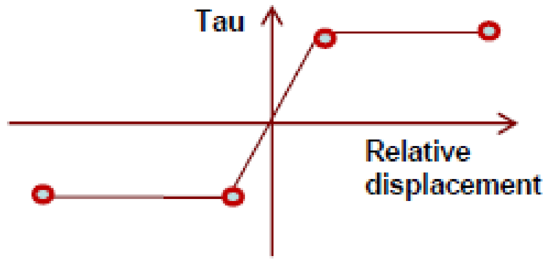


Skin tractions

t_s = q_s/length = k_s * (Δu) $\leq q_{ult}$

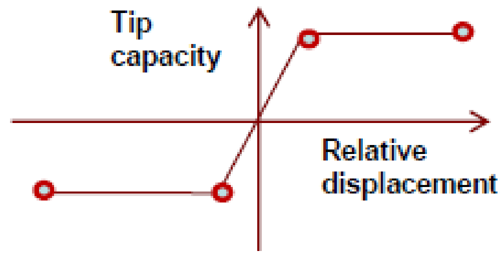
t_n = q_n/length = k_n * (Δu)

t_t = q_t/length = k_t * (Δu)



Base (tip) bearing capacity

q_b = k_b * (Δu) $\leq q_{bult}$

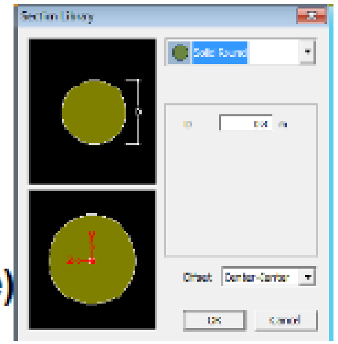


Pile beam properties

Cross section properties

Length

Material properties (e.g. Concrete)



Pile Element Parameters

GTS NX

Because the pile size, length, neighboring ground material properties all affect the pile element parameters, it is ideal to use the results of a loading test.

However, if there are no test results, the Ultimate Shaft Resistance, Lateral Subgrade Reaction Modulus and End Bearing Capacity of a pile can be calculated using the formulas proposed by the design code and the neighboring ground parameters (unit weight, cohesion, friction angle etc.).

1. Using loading test results

For example, if a load of 1000kN was found before failure from the pile loading test and the pile length was 10m, the Ultimate shear force is [1000kN/10m/1m] = 100kN/m^2. Here, the 1m is the unit length of the input pile element thickness. The Shear stiffness modulus is the slope of the linear section on the relationship graph with relative displacement until the 1000kN load is applied. If we assume that the relative displacement at 1000kN is 0.01m, the Shear stiffness modulus becomes [100kN/m^2 / 0.01m] = 10000kN/m^3.

2. Using the Ultimate Shaft Resistance results

For each design code, various formulas are suggested to predict the Ultimate bearing capacity of a pile according to ground and pile section properties. For example, if the calculated Ultimate shaft resistance is 50kN/m^2 and the Shaft surface area of the pile(equivalent circumference) is 3m, input 50kN/m^2 for the Ultimate shear force and 3m for the Pile element thickness respectively, or input 150kN/m^2 for the Ultimate shear force and the unit length of the Pile element thickness 1m. The Pile element length is automatically taken into account.

Because the load-relative displacement has no relationship, if the allowable settlement is assumed to occur at the Ultimate bearing capacity, the Shear stiffness modulus applied to the analysis can be inferred with reference to the allowable settlement. If the allowable settlement is 0.025m for the example above, the Shear stiffness modulus is [150kN/m^2 / 0.025m] = 6000kN/m^3. If the Pile element thickness is 3m, entering [50kN/m^2 / 0.025m] = 2000kN/m^3 still gives the same results. However, if a different numerical value from the unit length is input for the Pile element thickness, be aware that the same Normal stiffness modulus is equally applied.

Pile Modelling in GTS NX

GTS NX

Type	Axial behaviour		Lateral behaviour (Pile head loading)		Lateral behaviour (Horizontal soil displacement)	
Modelling approach	**Pile group**	**Single pile**	**Pile group**	**Single pile**	**Pile group**	**Single pile**
Solid elements	**A**	**A**	**A**	**A**	**A**	**A**
Pile elements	**A**	**A**	**C**	**C**	**B**	**B**

A : The best modelling choice for the correspondent application

B : Reasonable modelling

C : First order approximation

Limitations of pile elements

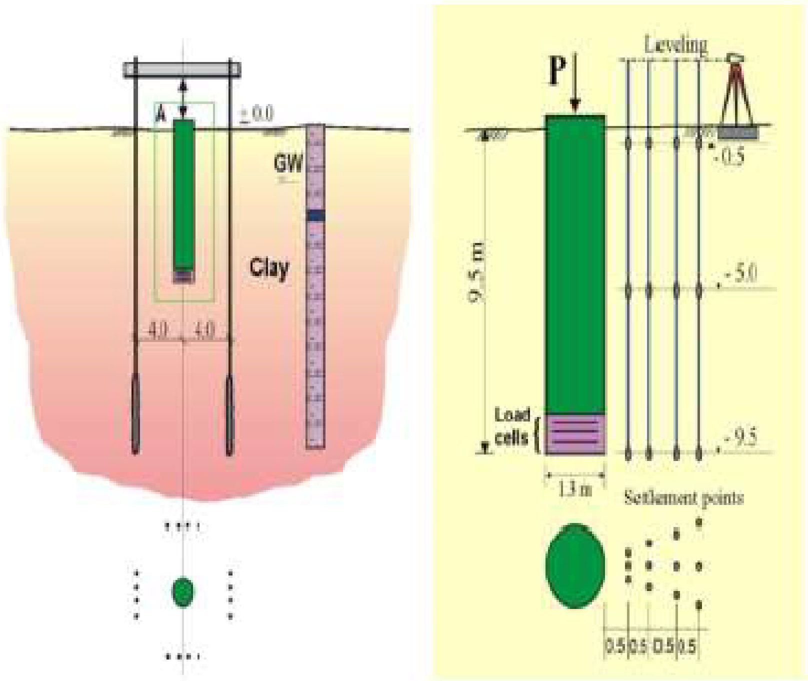
1. Pile elements have capabilities for lateral loading behaviour in case of rough pile-soil contact (full bonding) and small soil displacements.
2. When using 'standard' mesh around pile elements (no local refinement), stiffness and lateral capacity are over-estimated.
3. The pile diameter determines the size of the elastic zone in the soil around the pile in which plastic soil behaviour is excluded. By considering this in the modelling, it makes the pile elements behave almost like a solid elements.

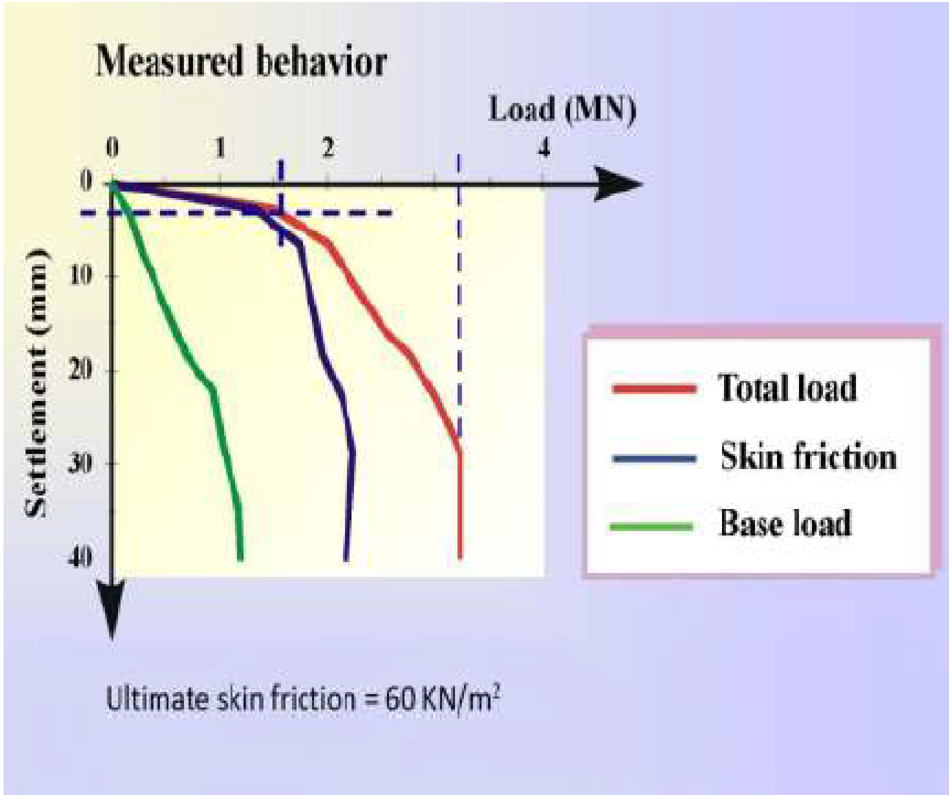
Verification

Single pile analysis of the Alzey Bridge pile loading test

The pile load test was conducted by Sommer & Hammabach in 1974 to optimize the foundation design of Alzey Highway Bridge in Germany (El-Mossallamy 1999)

Alzey bridge Pile-Load test after EL-Mossallamy (1999)

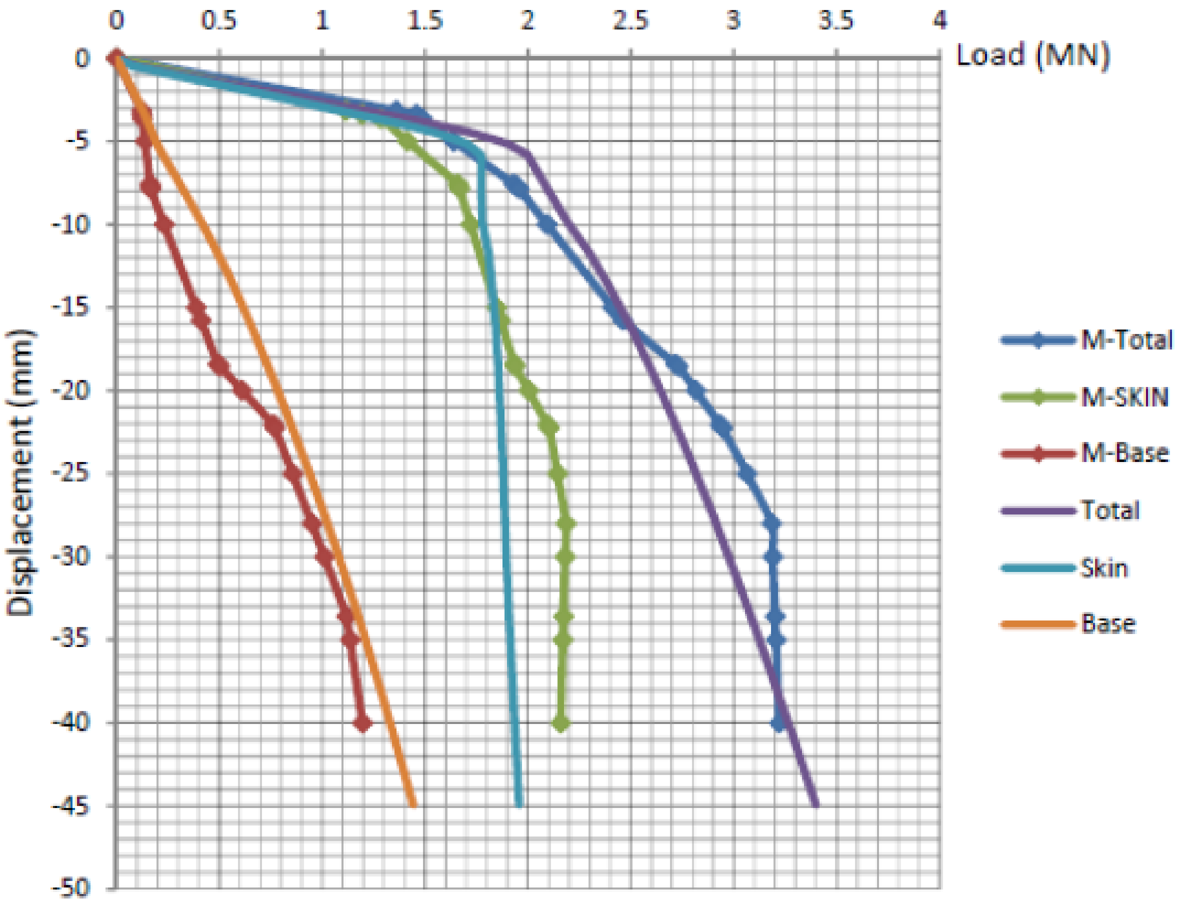




Verification

GTS NX

Single pile analysis of the Alzey Bridge pile loading test



Elkadi, O. (2010)

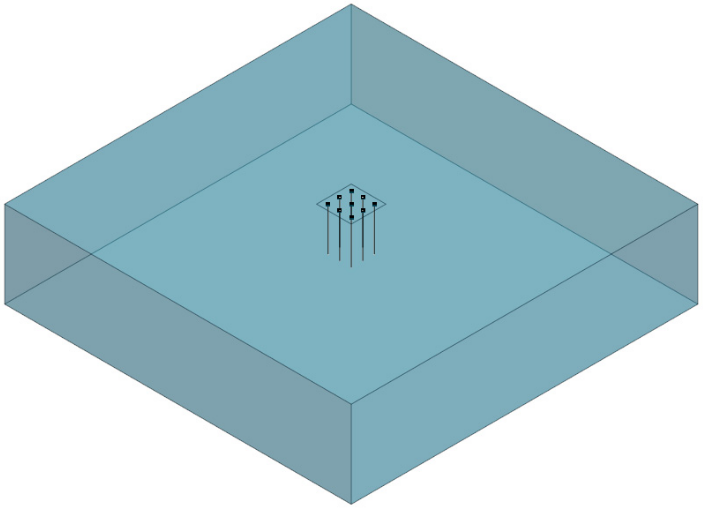
Procedure

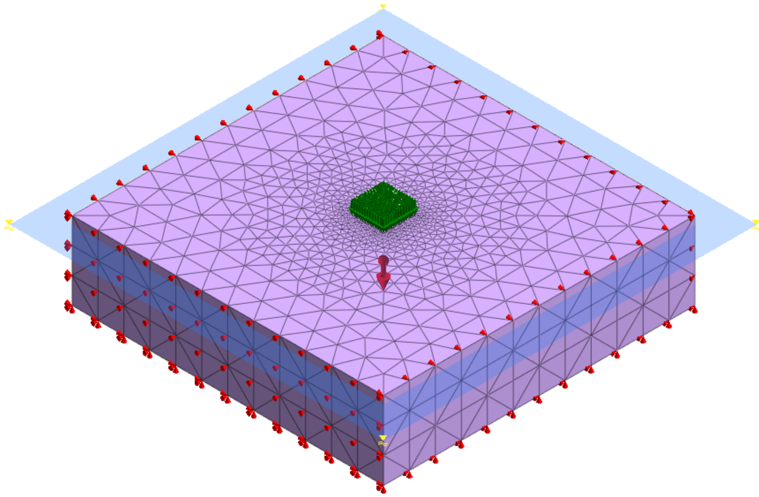
• This tutorial demonstrates some of the support systems by modelling a piled raft foundation.

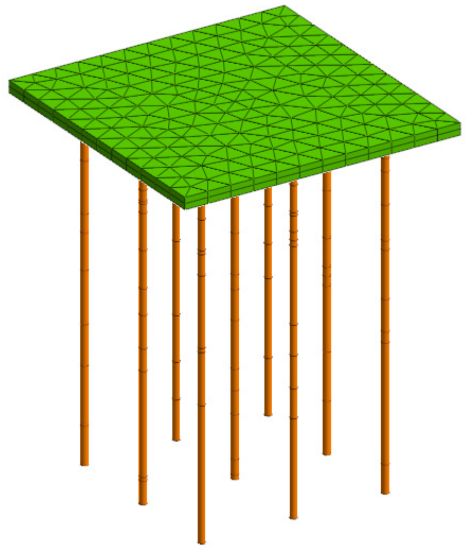
• It is possible to review in detail the stress distributions on cross-sections, which is not possible in 2D models.

• The evaluation of the soil-structure behavior is done by using pile elements, not by simple beam elements.

• Also, pile elements are perfect for evaluating the soil-structure behavior because it can consider the influence of interface between pile and adjacent ground.







GTS NX

Soil

[unit : kN, m]

Name	Clay
Material	Isotropic
Model Type	Mohr-Coulomb
General	
Elastic Modulus(E) [kN/m^2]	5,000
Poisson's Ratio(v)	0.35
Unit Weight(γ) [kN/m^3]	18
Unit Weight(γ_{sat}) [kN/m^3]	21
Ko	Auto
Non-Linear	
Cohesion(c) [kN/m^2]	4
Frictional Angle(Φ) [deg]	30
Dilatancy Angle(Φ) [deg]	0
Tensile Strength [kN/m^2]	0

GTS NX

Structure

[unit : kN, m]

Name	Raft	Pile
Material	Isotropic	Isotropic
Model Type	Elastic	Elastic
Elastic Modulus (E) [kN/m^2]	3.5e+07	3.5e+07
Poisson's Ratio (v)	0.2	0.2
Unit Weight (γ) [kN/m^3]	25	25
Element Type	Shell	Beam
Thickness [m]	0.5	-
Diameter [m]	-	0.3

Pile

[unit : kN, m]

Name	Pile Interface
Model Type	Pile
Ultimate Shear Force [kN/m^2]	100
Shear Stiffness Modulus (Kt) [kN/m^3]	5e+03
Normal Stiffness Modulus (Kn) [kN/m^3]	5e+04
Tip Bearing Capacity [kN]	100
Tip Spring Stiffness [kN/m]	5e+04

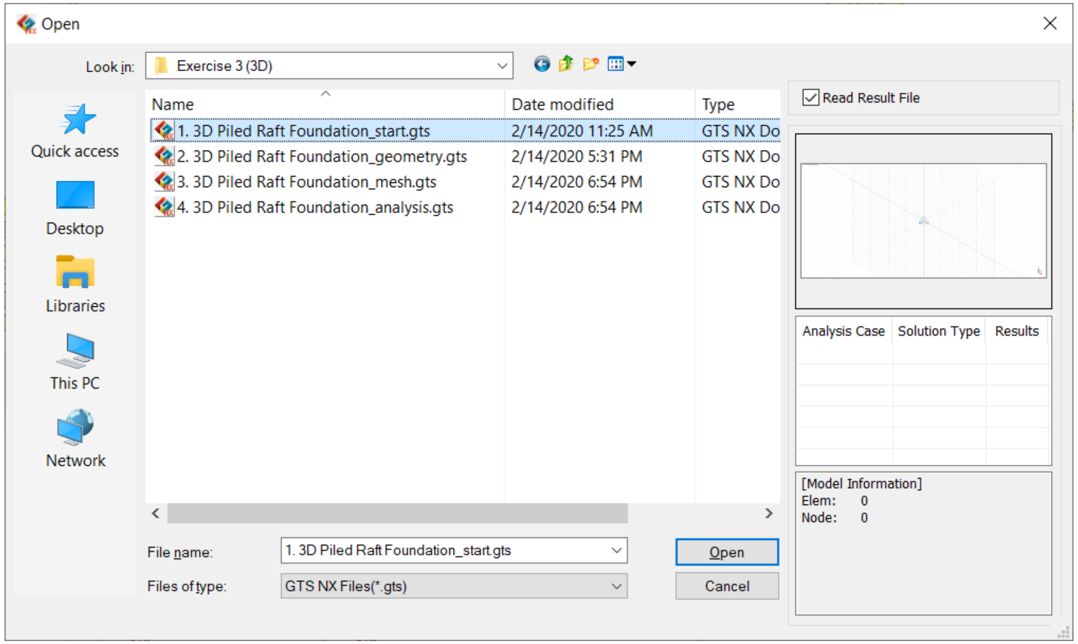
Procedure

Start the tutorial by opening the start file in which basic materials and properties have already been predefined.

1 **File > Open**

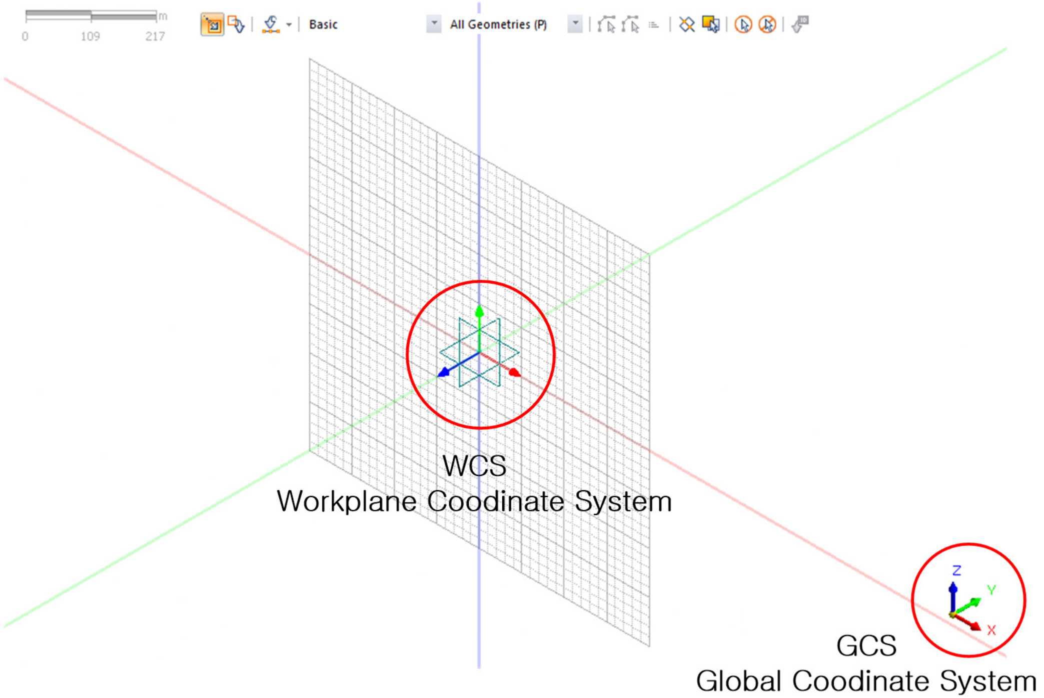
- Select **'1. 3D Piled Raft Foundation_start.gts'**
- Open

1



In GTS NX, 2 types of coordinate system are used: **Global coordinate system (GCS)** and **Work plane coordinate system (WCS)**. GCS is a fixed coordinate system indicated in **red(X-axis)**, **green(Y-axis)**, and **blue (Z-axis)** colors at the right bottom corner on the screen. WCS is a coordinate system which locates at center of the working window and moves with the work plane. So if the work plane changes, WCS will also change. 3D absolute coordinates are necessary to locate geometry in the space, but practically relative coordinates are commonly used, for example to indicate length. You can process modeling by inputting 2D coordinates (XY plane in WCS) after moving the work plane to proper location.

Keep in mind that in case of inputting direction for extruding geometry or defining load/boundary conditions, it always follows Global coordinate system.

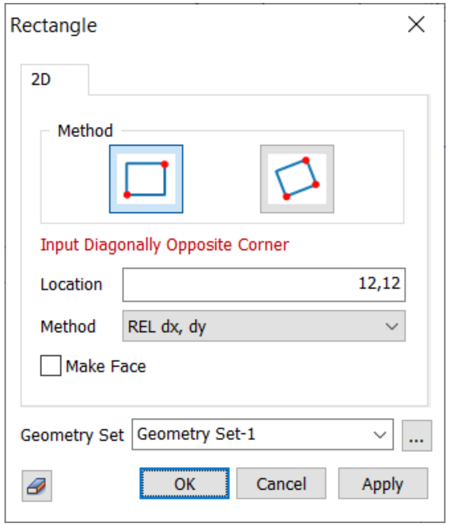


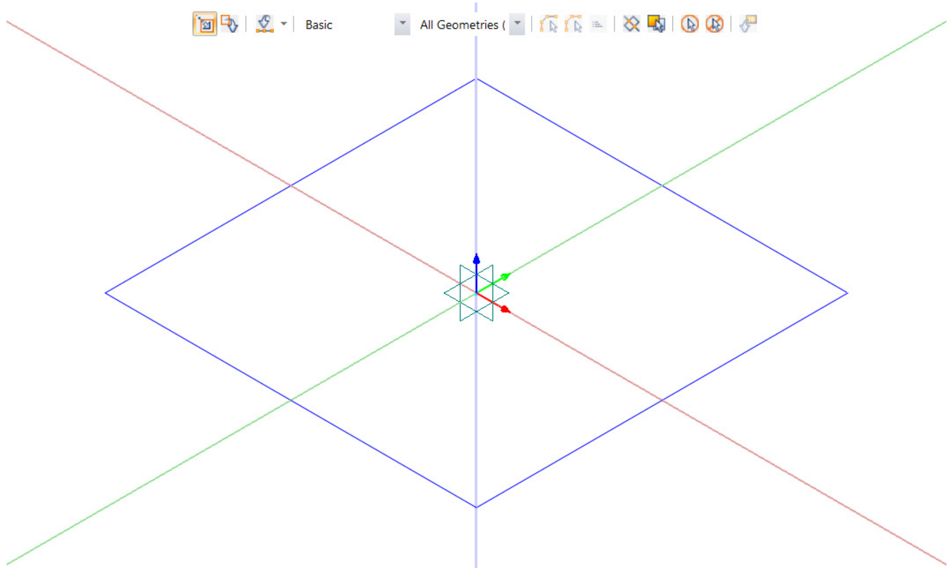
Procedure

Create a rectangle for raft and points for pile head.

1 **Geometry > Point & Curve > Rectangle**

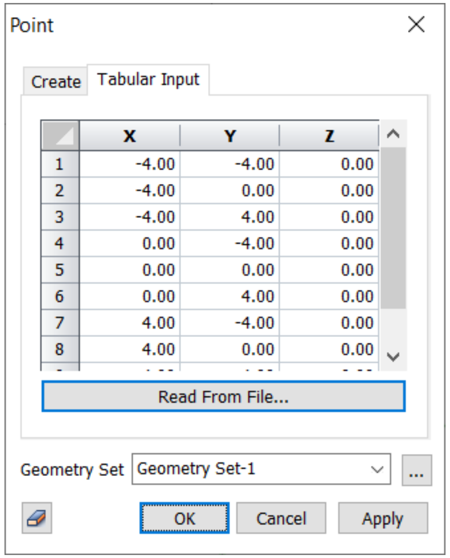
- Double click **'Work-Tree > Work Plane > XY(0,0,1)'**
- Location: **(-6,-6) <12,12>**
- OK

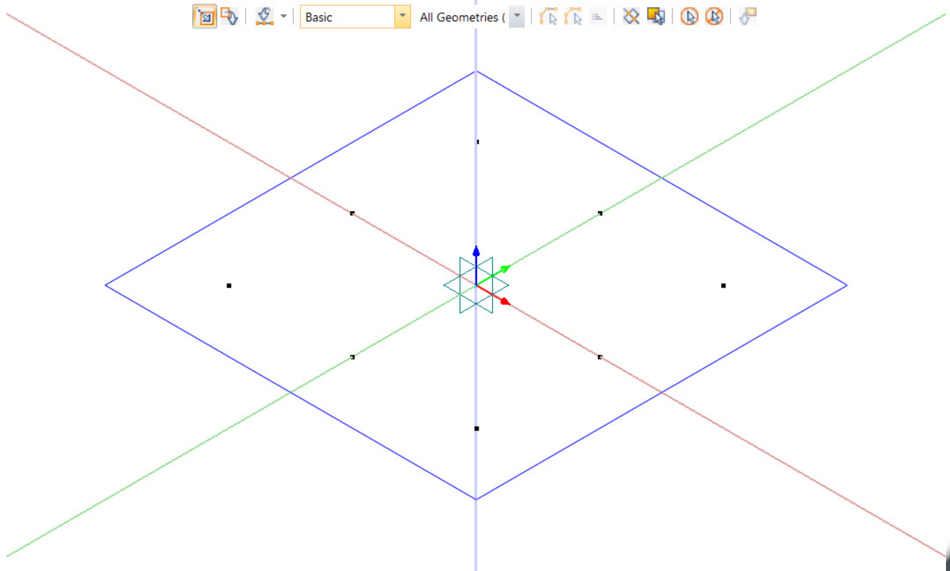




2 **Geometry > Point & Curve > Point**

- Tabular Input tab
- Read From File…
- **'Location of Piles.txt'**
- OK





( ): 'ABS x, y'
< >: 'REL dx, dy'

Procedure

Create lines for pile and ground solid.

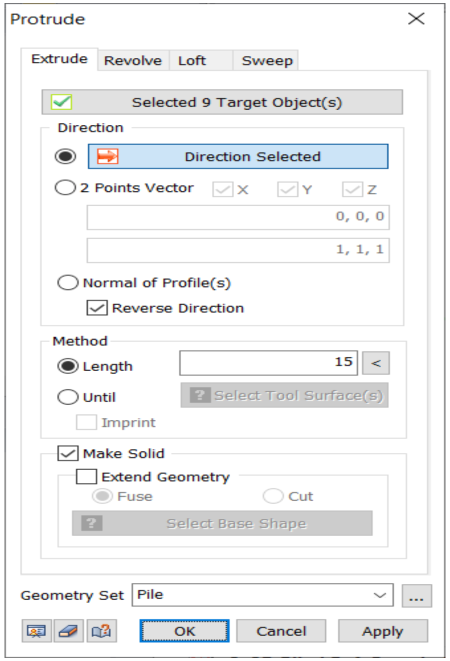
1 **Geometry > Protrude > Extrude**

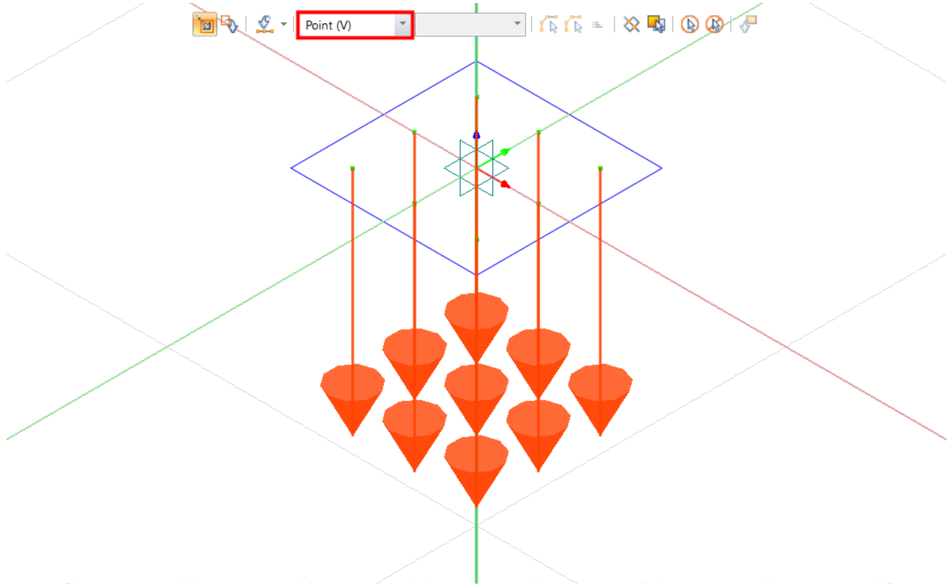
- Extrude tab
- Selection filter: **Point (V)**
- Select: **9 points (pile head)**
- Direction: **Z-axis**
- Reverse Direction: **Check on**
- Length: **15**
- Geometry Set: **Pile**
- OK

2 **Geometry > Surface & Solid > Box**

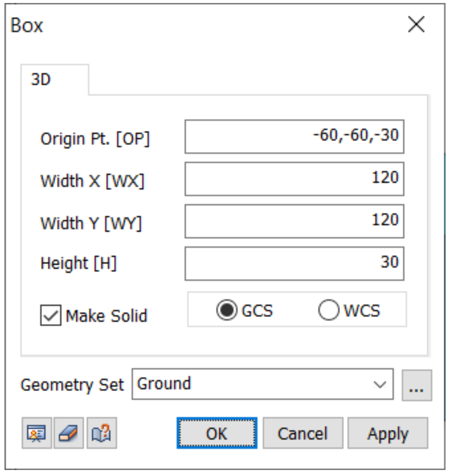
- Origin Point: **-60,-60,-30**
- Width X: **120**
- Width Y: **120**
- Height: **30**
- Geometry Set: **Ground**
- OK

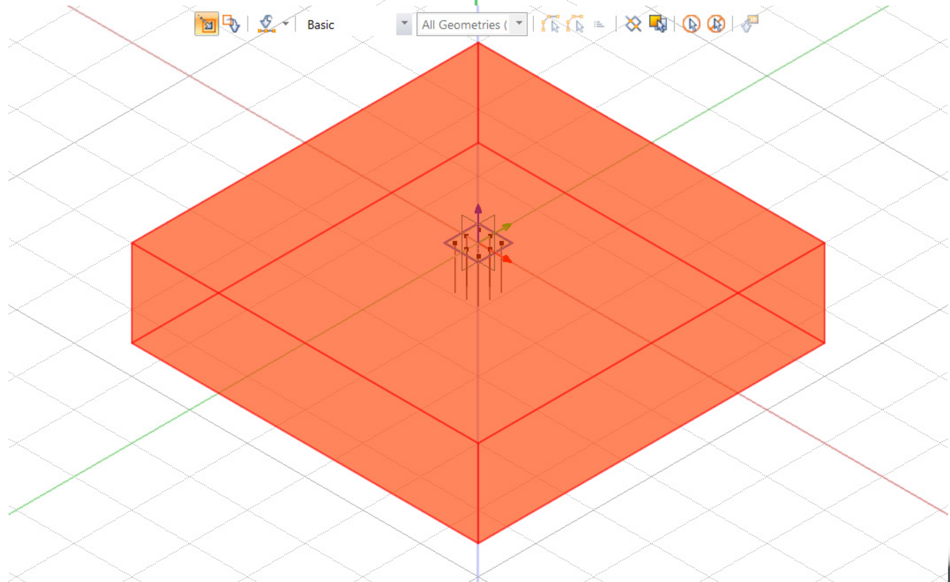
1





2





Procedure

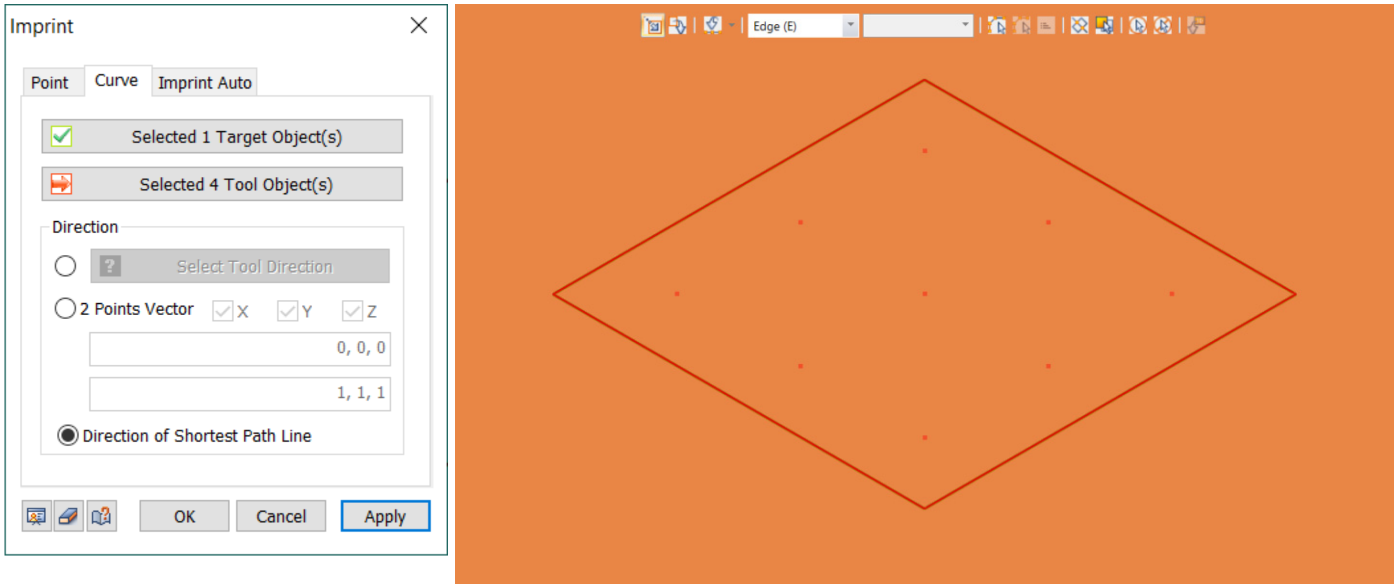
Project pile location and the raft area on the top surface of solid.

1. **Geometry > Surface & Solid > Imprint**
   - Point tab
   - Select Target: **Top Surface**
   - Select Tool: **9 points for piles**
   - Select: **Direction of Shortest Path Line**
   - Apply

2. - Curve tab
   - Select Target: **Top Surface**
   - Select Tool: **4 edges for raft**
   - Select: **Direction of Shortest Path Line**
   - OK

1

2



Procedure

Specify size of elements for the raft foundation and generate ground mesh.

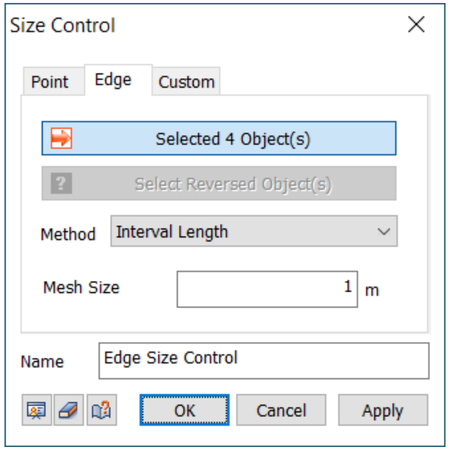
1 Mesh > Control > Size Ctrl.

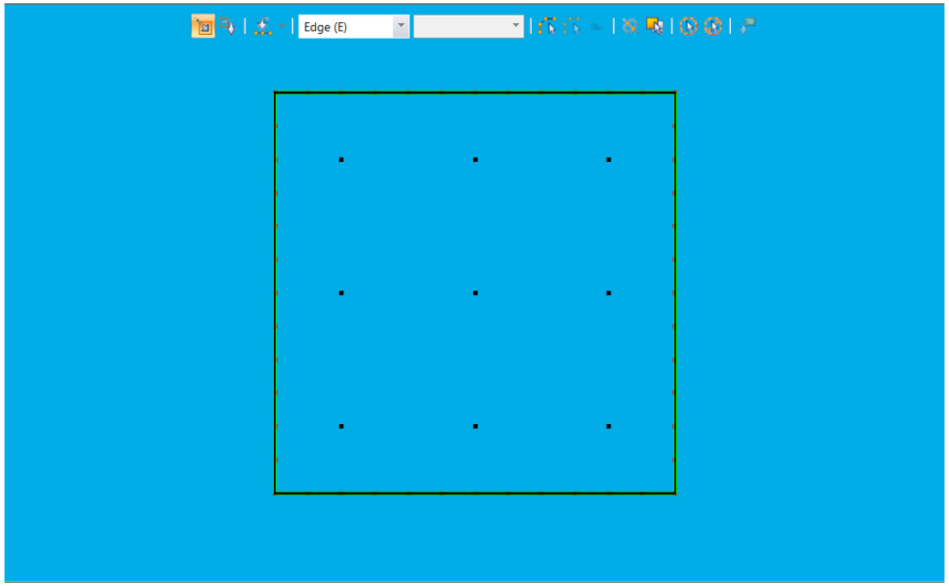
- Edge tab
- View Toolbar: **Top**
- Select: **4 edges for the raft foundation**
- Method: **Interval Length**
- Mesh Size: **1**
- OK

2 Mesh > Generate > 3D

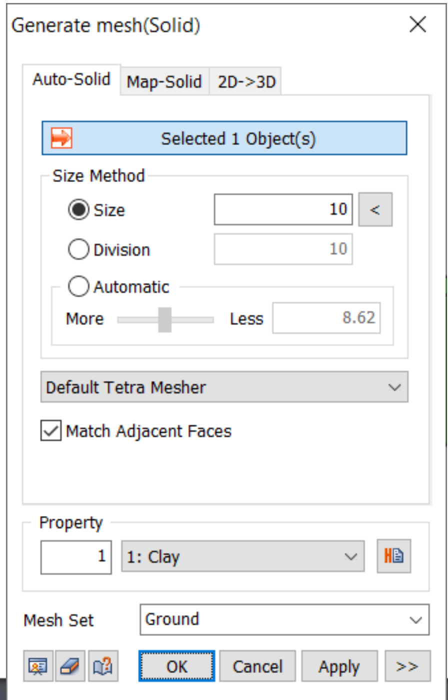
- Auto-Solid tab
- Select: **Ground (Solid)**
- Size: **10**
- Property: **1: Clay**
- Mesh Set: **Ground**
- OK

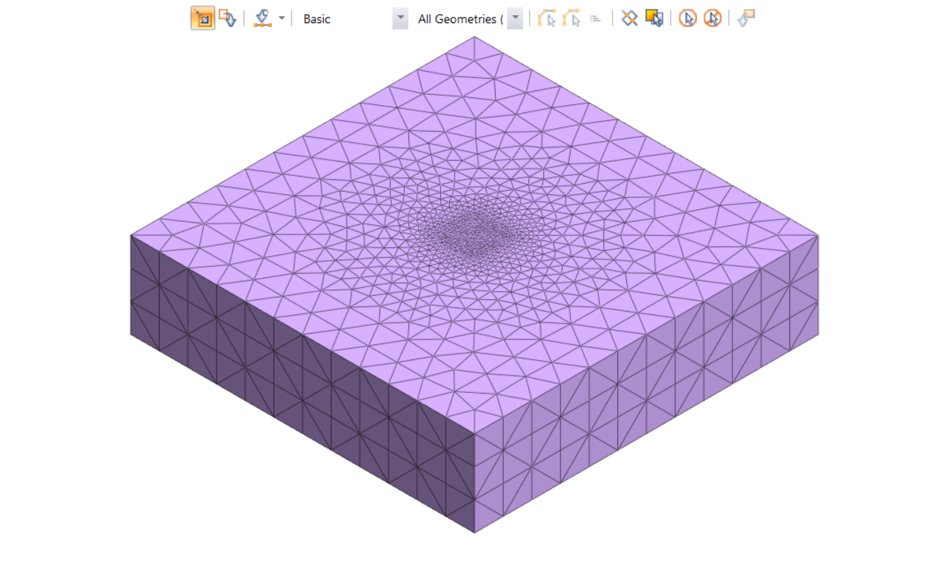
1





2





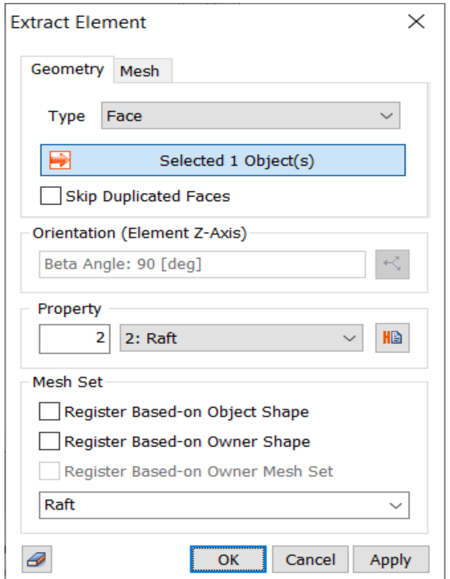
GTS NX

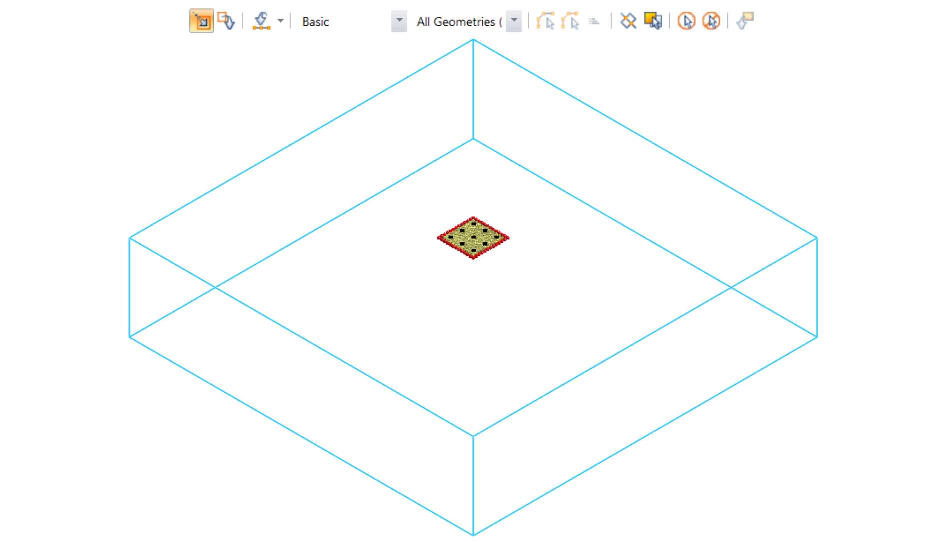
Procedure

Generate mesh for the raft foundation and piles.

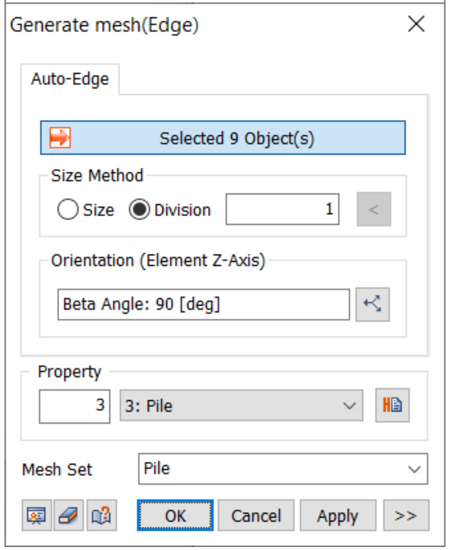
1 - Show only the geometry solid for Ground
- Select it and right click to select '**Display Mode > Line Only**'

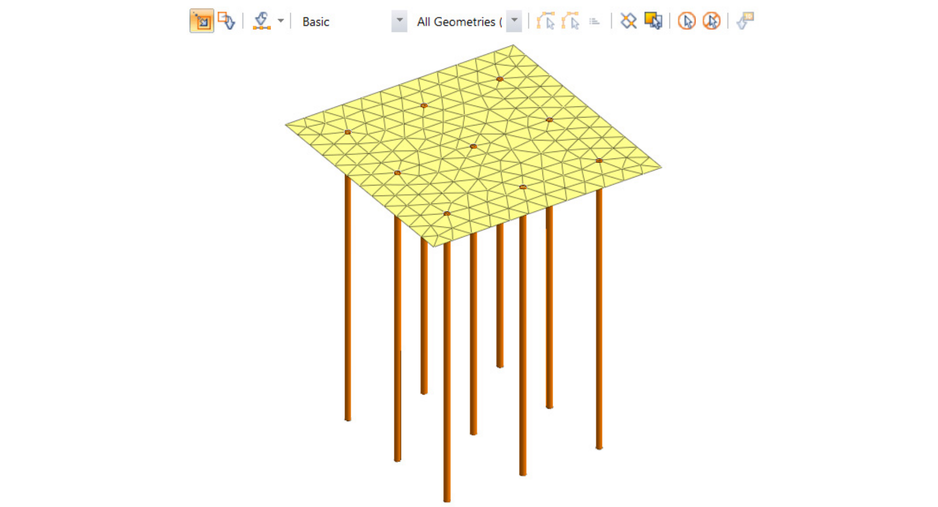
Mesh > Element > Extract
- Geometry tab
- Type: **Face**
- Select: **1 surface for the raft foundation**
- Property: **2: Raft**
- Mesh Set: **Raft**
- OK





2 **Mesh > Generate > 1D**
- Show **geometry lines for pile**
- Select: **all lines**
- Division: **1**
- Property: **3: Pile**
- Mesh Set: **Pile**
- OK





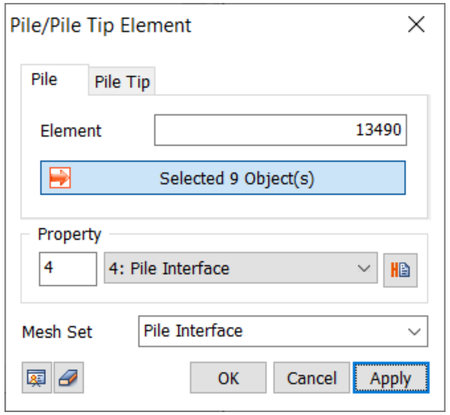
Procedure

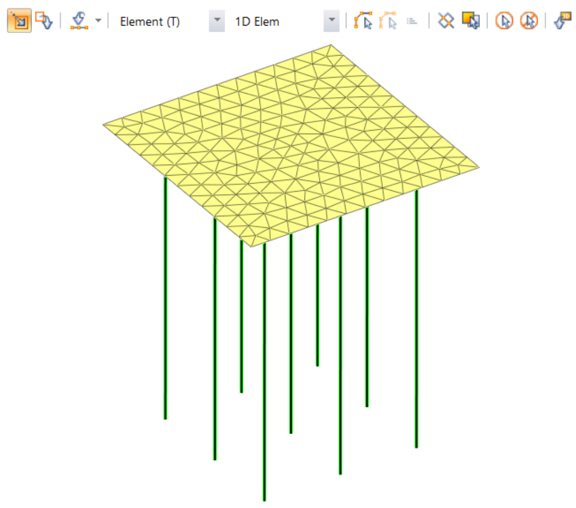
Generate mesh for the pile interface and pile tip.

1 **Mesh > Element > Pile/Pile Tip**
- Pile tab
- Select: **all Pile elements**
- Property: **4: Pile Interface**
- Mesh Set: **Pile Interface**
- Apply

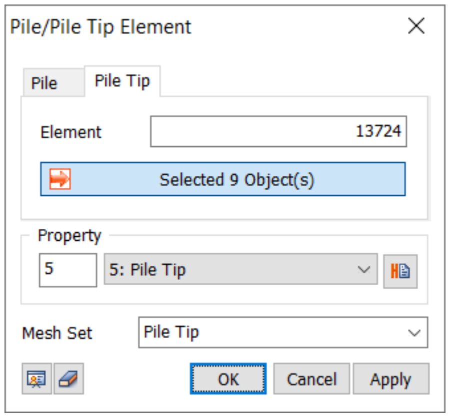
2
- Pile Tip tab
- View Toolbar: **Front**
- Select: **all bottom nodes of Pile elements (as shown in the figure)**
- Property: **5: Pile Tip**
- Mesh Set: **Pile Tip**
- OK

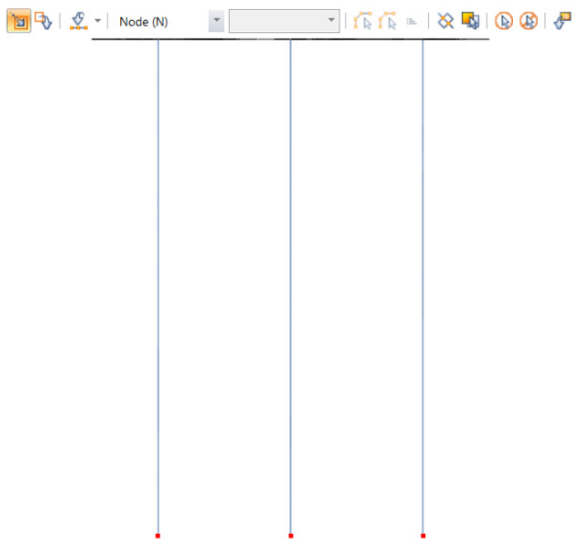
1





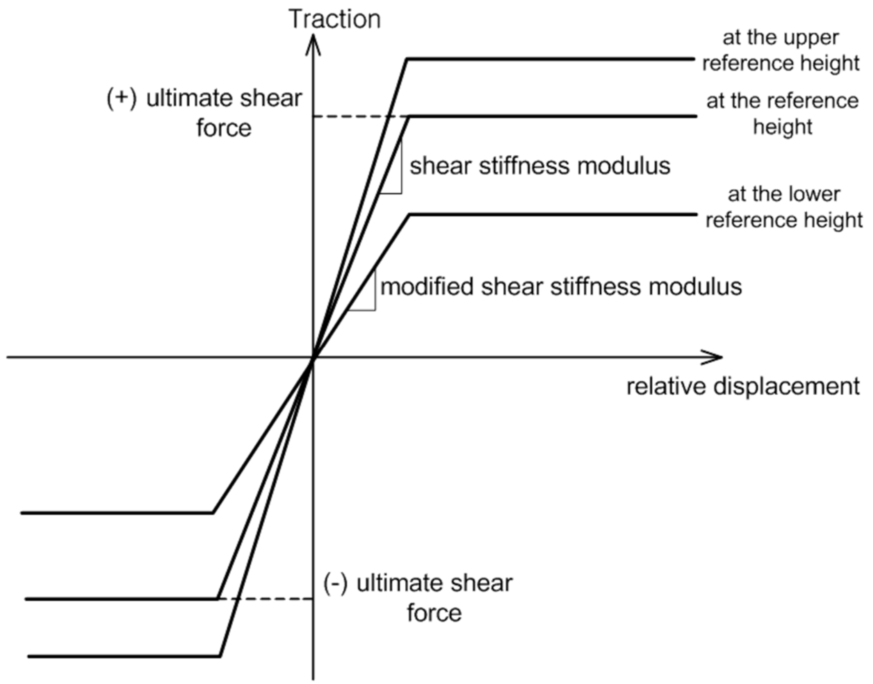
2





The behavior of pile elements can be divided into a normal behavior and a tangential behavior. First, the normal behavior between the pile and the surrounding ground is considered as fixed and rigid, whereas the tangential behavior is a nonlinear elastic behavior. The nonlinear elastic behavior is divided into the yield force and the yield function assigned.

The graph bellow represents the relative displacement between the 2 bodies and the friction when yield force is defined. If the relation is defined by a function, the relation between relative displacement and friction can be defined more precisely.



The Pile tip element works as solid-point interface that presents the relative behavior between the ground elements and pile node. In the element coordinate system of the pile tip element regard the normal direction behavior toward the element as rigid behavior just like a pile behavior. And, regard the tangent direction behavior as nonlinear elastic behavior.

To define the behavior, the material and property of a pile element can be entered based on test data, such as Load Test.

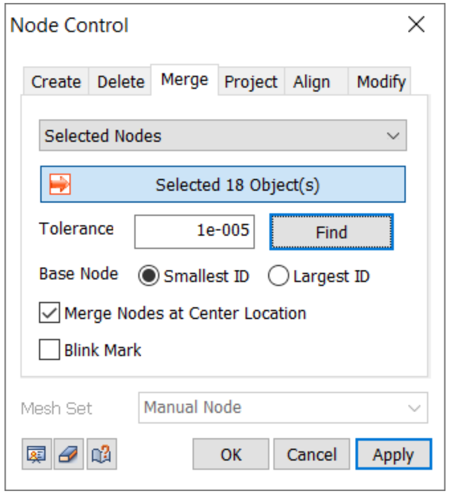
For more information about entering parameters of pile element, look up [User Manual] Ch4 (General Material) or Select F1 for [Online Manual].

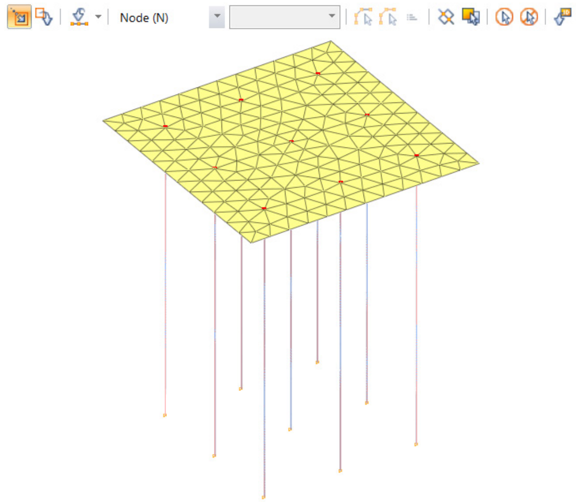
Procedure

Merge nodes between raft and pile head.

1. **Mesh > Node > Merge**
   - Merge tab
   - Select: **all nodes for raft and piles**
   - Click: **Find**
   - OK

1





Pile connection

In the case of the pile connection point coinciding with pile cap elements, the connection behaviour is as follows:

1. Free (Mesh > Element > Pile)

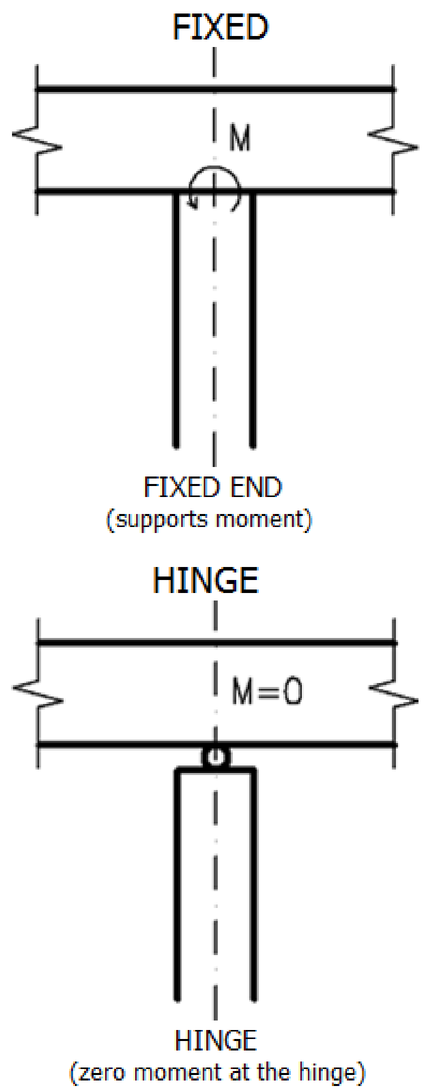
The connection point is free to move and rotate relative to the surrounding soil and the pile cap element. The pile interacts with the surrounding soil and the pile cap elements only through the pile interface elements. Note that the pile cap elements are not free to move and rotate relative to the surrounding soil.

2. Rigid (Mesh > Node > Merge)

The displacement and rotation at the pile connection point are both coupled with the displacement and rotation of the pile cap and/or soil element in which the pile top is located.

3. Hinged (Mesh > Element > Parameters > Beam End Release)

The displacement at the connection point of the pile is directly coupled with the displacement of the pile cap element and/or the surrounding soil in which the pile connection point is located, which means that they undergo exactly the same displacement.



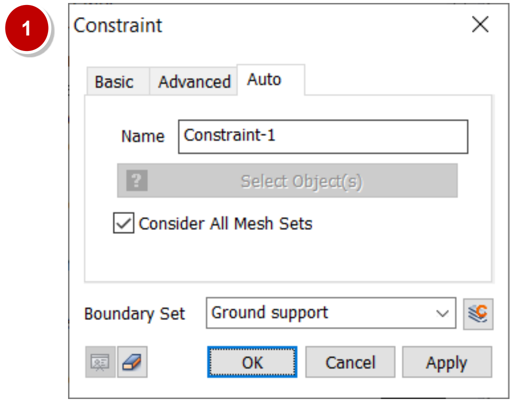
Procedure

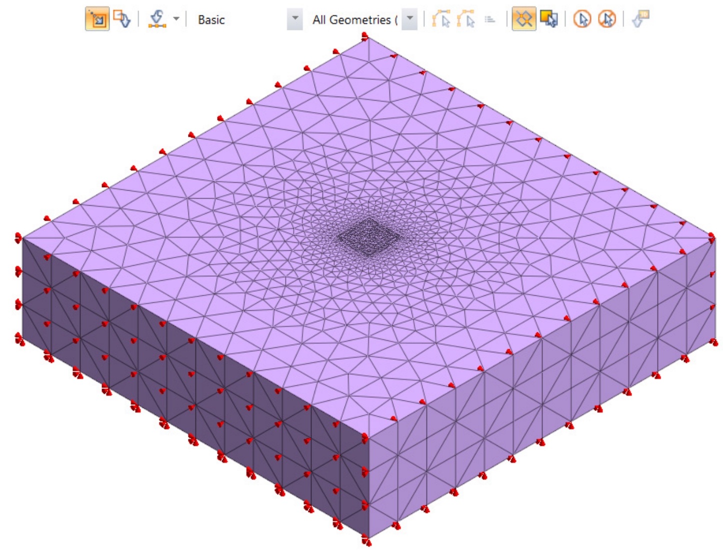
Assign boundary conditions to the external boundary of the model and piles.

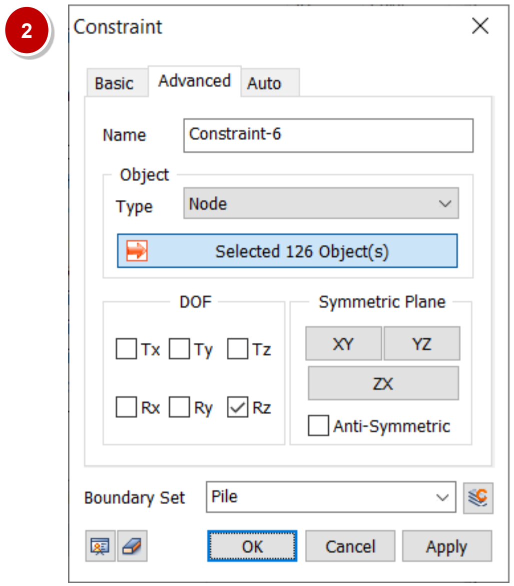
1. **Static/Slope Analysis > Boundary > Constraint**
   - Show **all mesh sets**
   - Auto tab
   - Boundary Set: **Ground support**
   - Apply

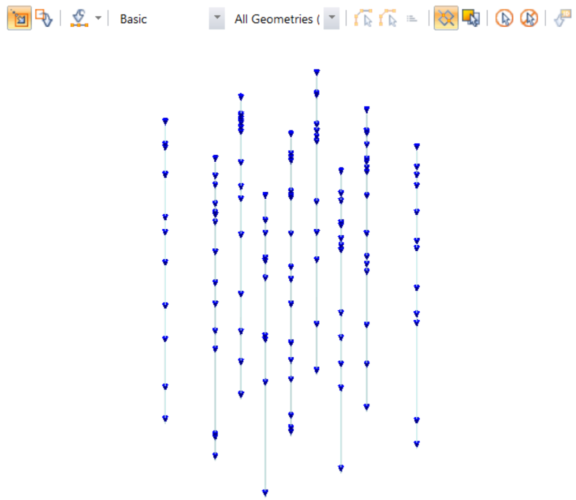
2. - Advanced tab
   - Show only **'Pile' mesh set**
   - Object Type: **Node**
   - Select: **all nodes of Pile elements**
   - DOF: **Rz**
   - Boundary Set: **Pile**
   - OK

Axial rotation constraints to prevent the degree of freedom errors for torsion of beam elements









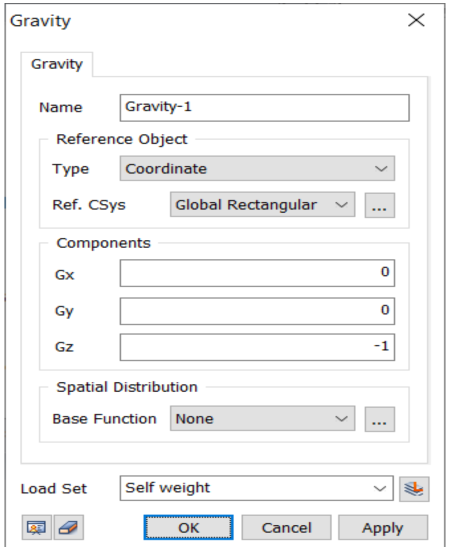
Procedure

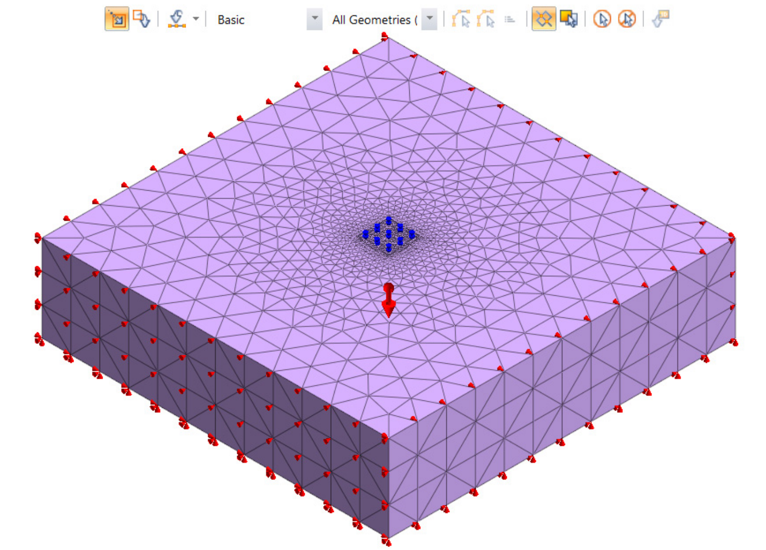
Create self weight and a surface load of the raft foundation.

1. **Static/Slope Analysis > Load > Self Weight**
   - Show **all mesh sets**
   - Gz: **-1**
   - Load Set: **Self weight**
   - OK

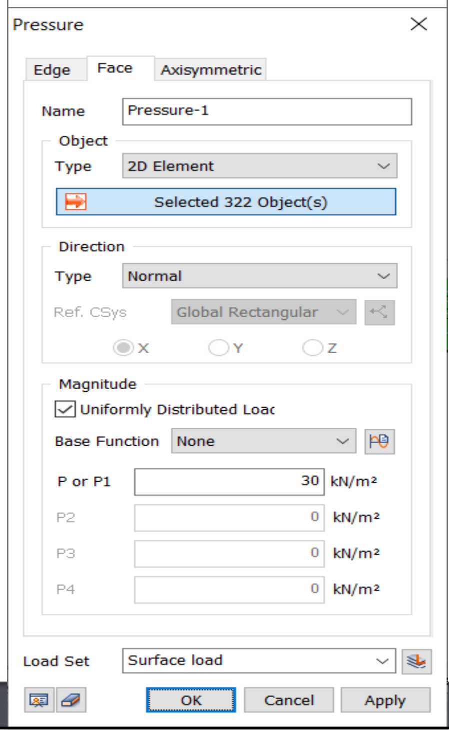
2. **Static/Slope Analysis > Load > Pressure**
   - Show only **'Raft' mesh set**
   - Face tab
   - Object Type: **2D Element**
   - Select: **all elements of Raft**
   - Direction Type: **Normal**
   - P or P1: **30 kN/m²**
   - Load Set: **Surface load**
   - OK

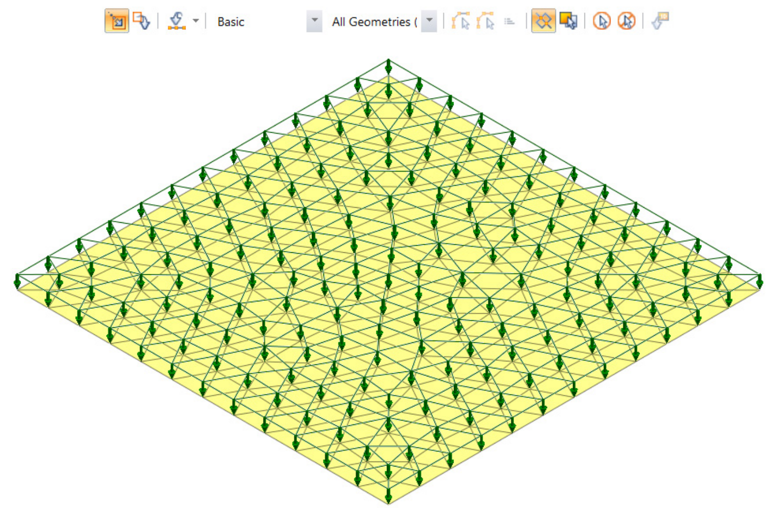
1





2





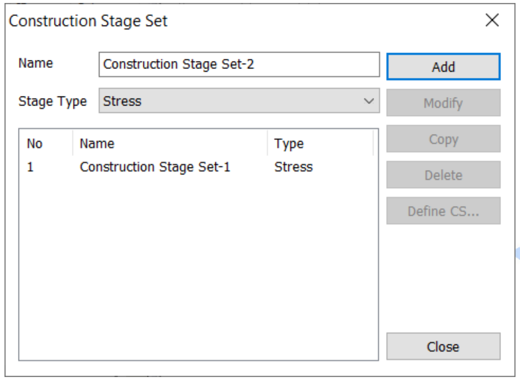
05 Define Construction Stages

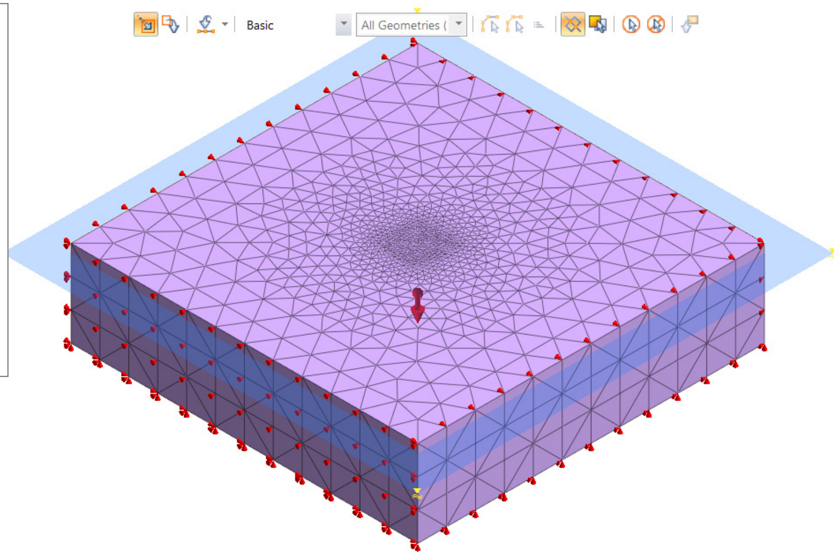
GTS NX

Procedure

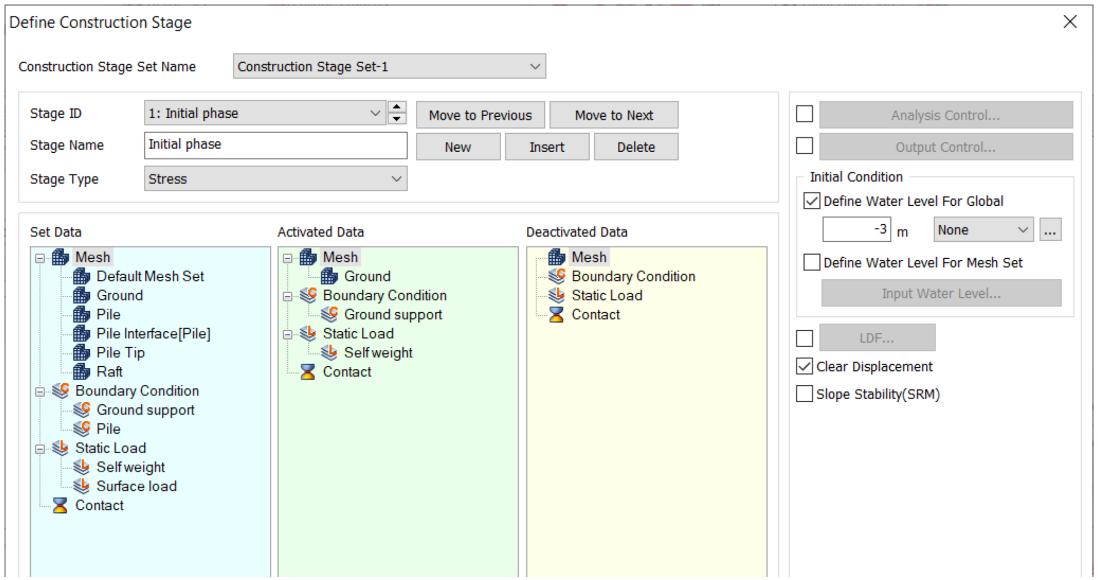
Define a construction stage of the piled raft foundation.

1. **Static/Slope Analysis > Construction Stage > Stage Set**
   - Stage Type: **Stress**
   - Add
   - Double click the **'Construction Stage Set-1'**





2. - Stage Name: **Initial phase**
   - Select mesh, boundary and load sets, and drag & drop them into Activated Data from Set Data. (as shown in the figure)
   - Show Data: **Activate**
   - Define Water Level for Global: **Check on '-3m'**
   - Clear Displacement: **Check on**
   - Save



05 Define Construction Stages

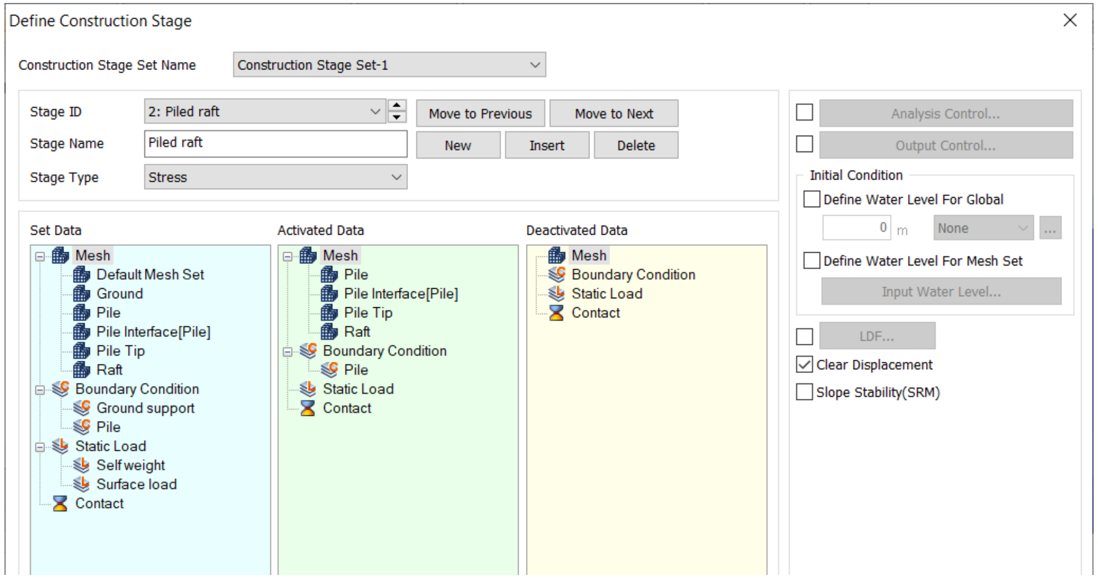
GTS NX

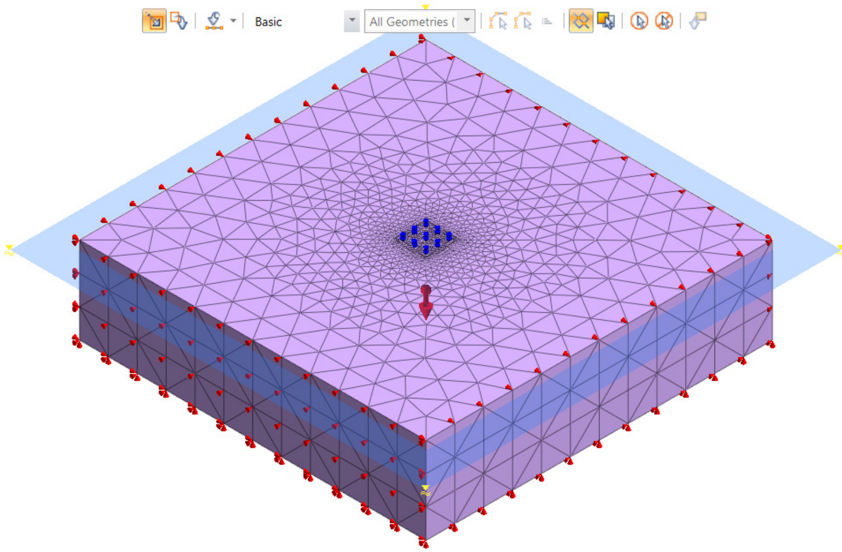
Procedure

1 - New

- Stage Name: **Piled raft**
- Select mesh and boundary sets, and drag & drop them into Activated Data from Set Data. (as shown in the figure)
- Clear Displacement: **Check on**
- Save

1





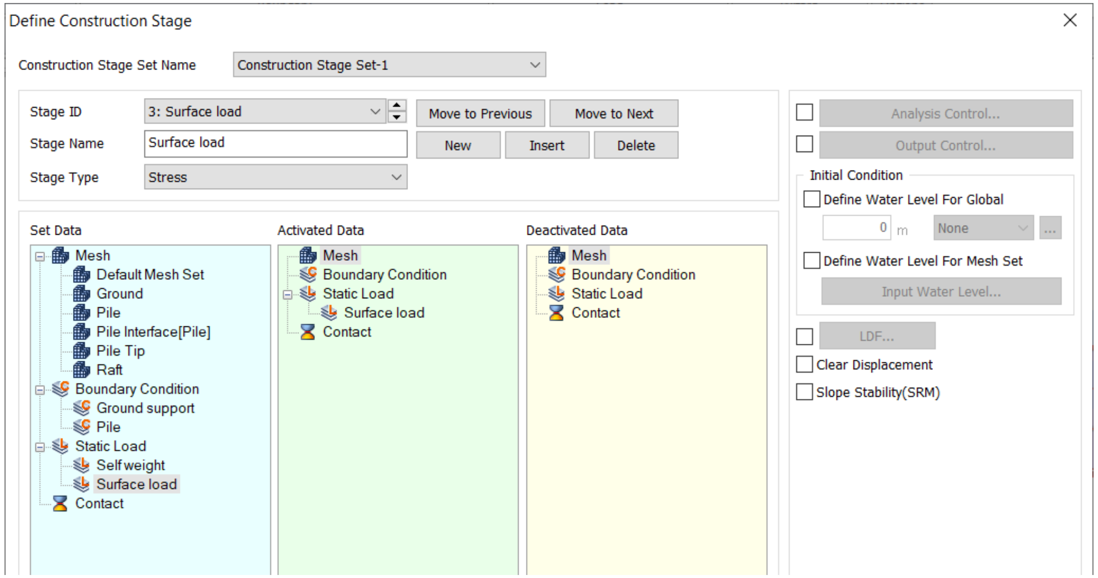
GTS NX

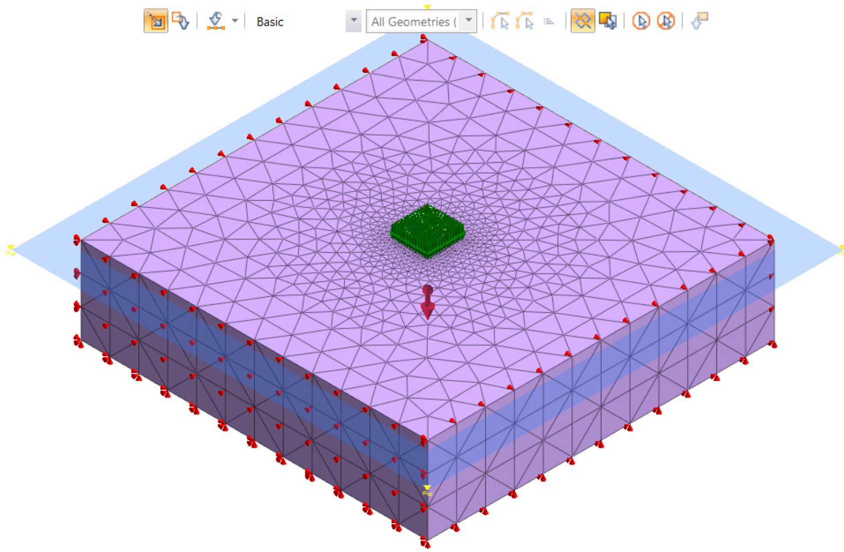
Procedure

1 - New

- Stage Name: **Surface load**
- Select load set, and drag & drop them into Activated Data from Set Data. (as shown in the figure)
- Save

1





06 Analysis Setting

GTS NX

Procedure

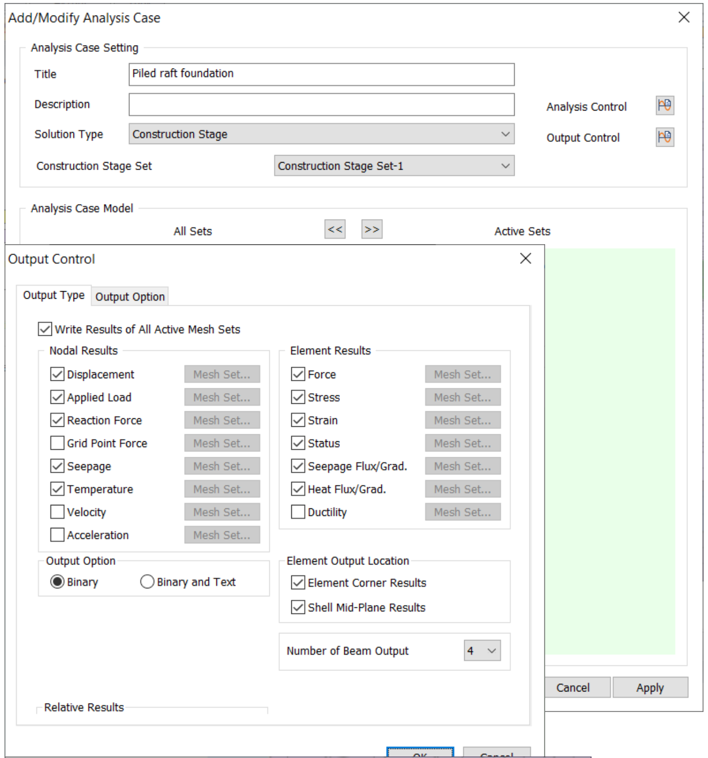
1 **Analysis > Analysis Case > General**

- Title: **Piled raft foundation**
- Solution Type: **Construction Stage**
- Analysis Control
- Initial Stage for Stress Analysis: **Check on**
- Initial Stage: **1:Initial phase**
- Apply K0 Condition: **Check on**
- OK
- Output Control
- Strain: **Check on**
- OK
- OK

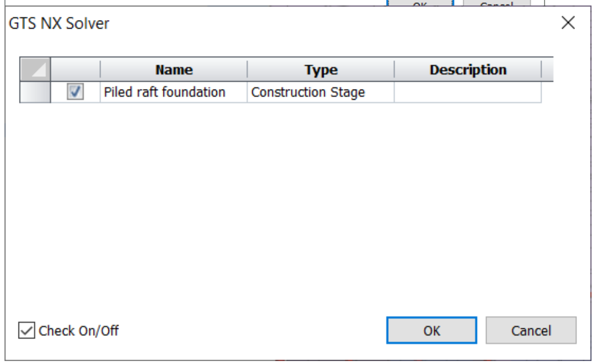
2 **Analysis > Analysis > Perform**

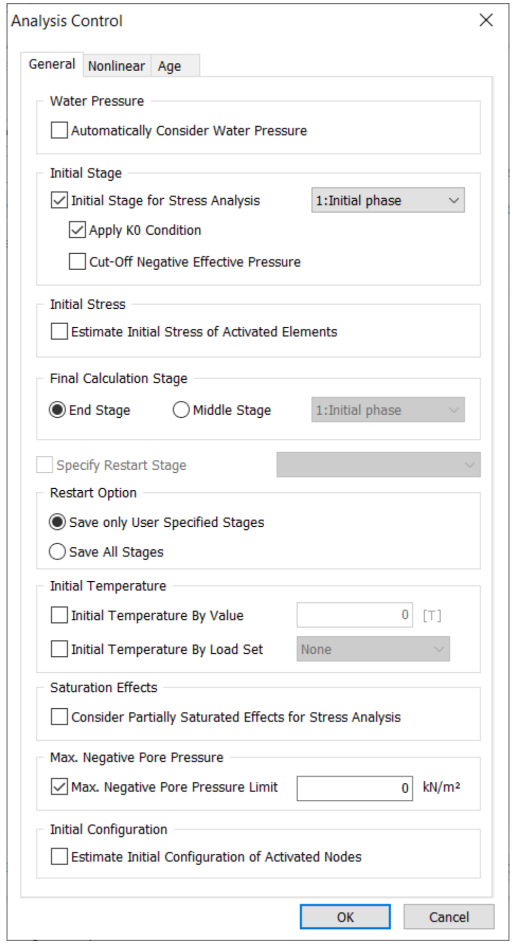
- Analysis Case: **Check on**
- OK

1



2





Procedure

1. Surface load > Displacement > TOTAL TRANSLATION (V)

   Result > General > Smooth: Fringe

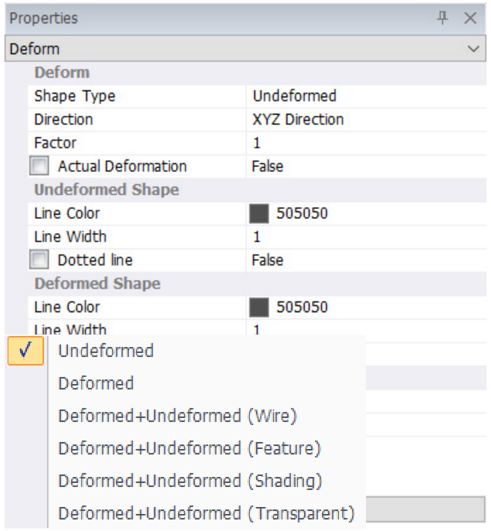
   Result > General > Deform: Undeformed

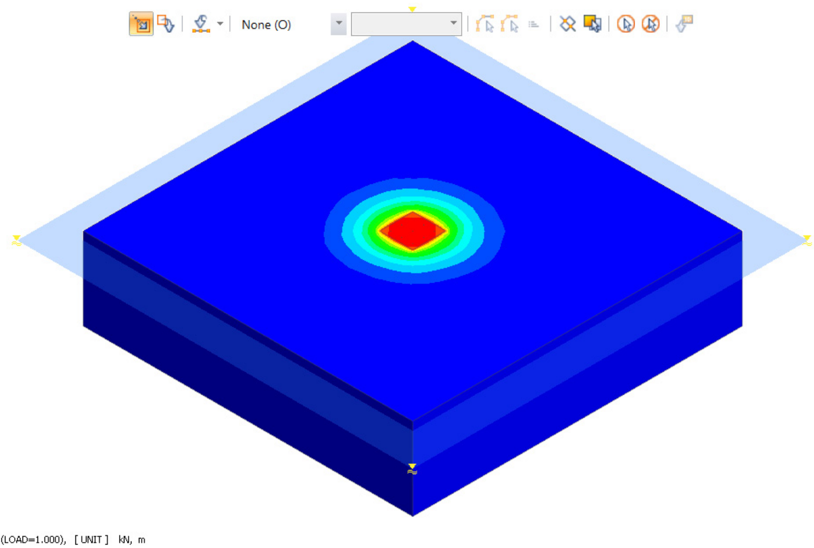
   - Properties: **Deform**

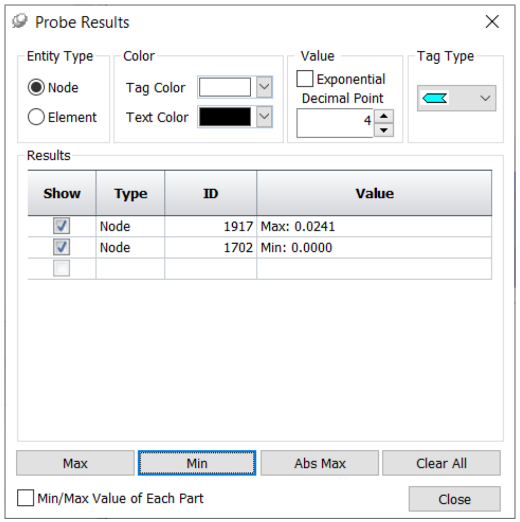
   Result > Show/Hide > Actual Deformation

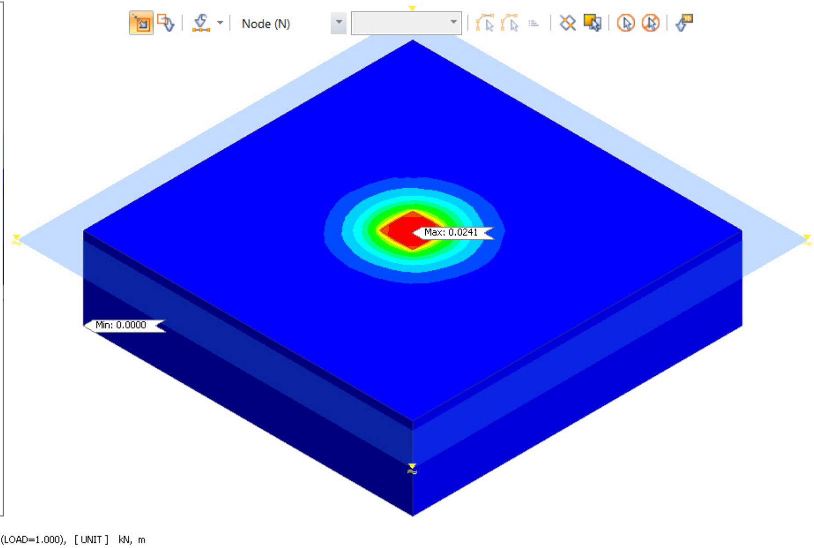
2. **Result > Advanced > Probe**

   - Max
   - Min









GTS NX

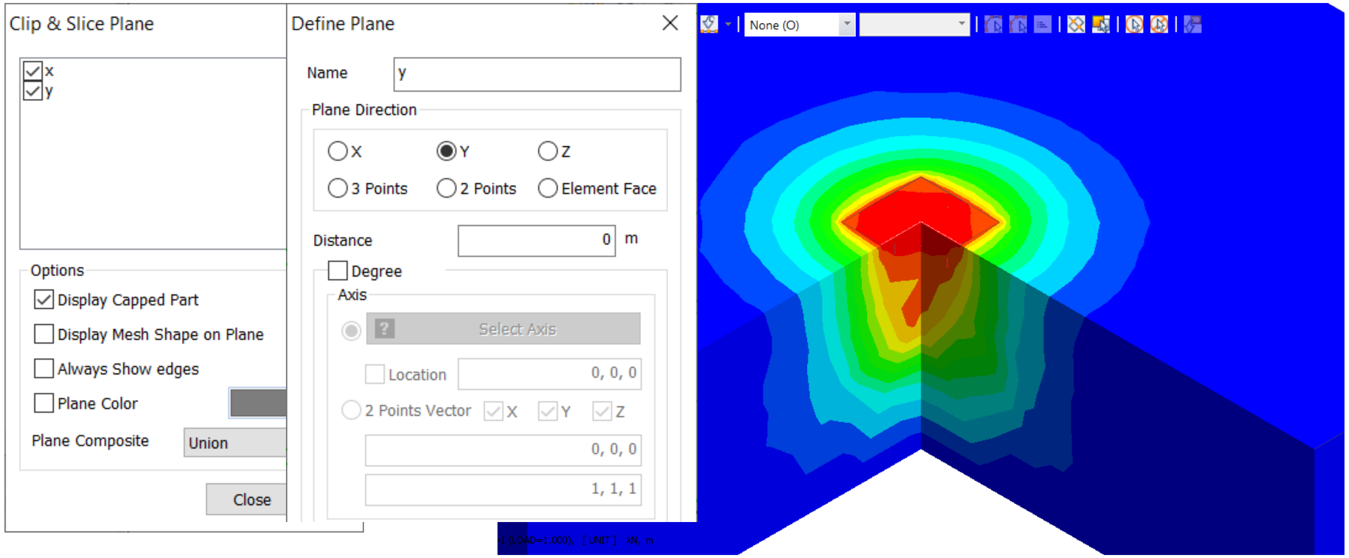
Procedure

1. Advanced View Control: **Clipping Plane**
   - Define Plane
   - X: **0 m**
   - Add
   - Y: **0 m (Reverse)**
   - Add
   - Close
   - Always Show edges: **Check off**
   - Plane Composite: **Union**
   - Close

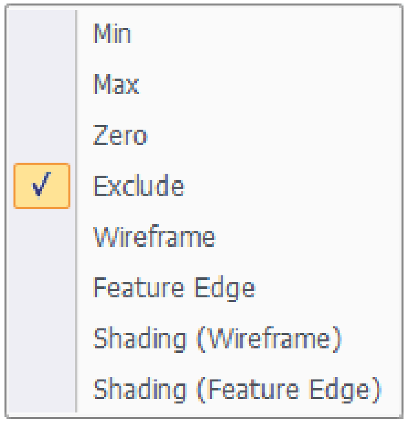
2. Surface load > Beam Element Forces > AXIAL FORCE (The axial force, as expected, decreases with depth into the pile.)

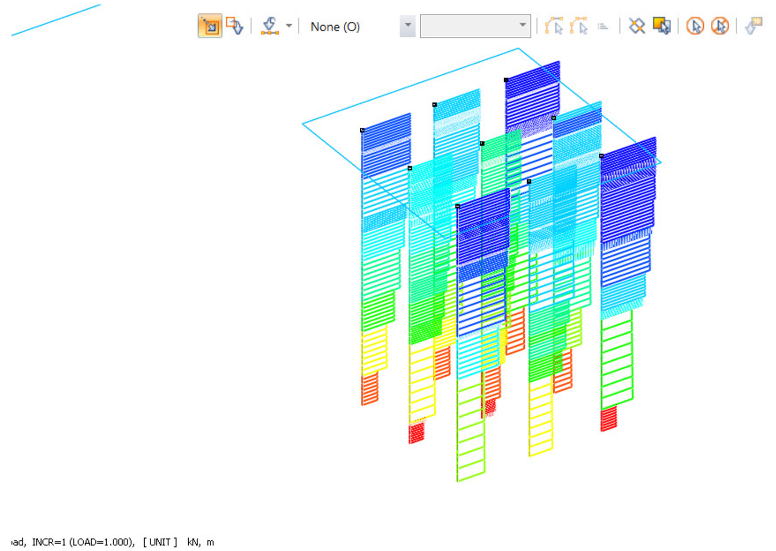
   Result > General > No Results: Exclude

1



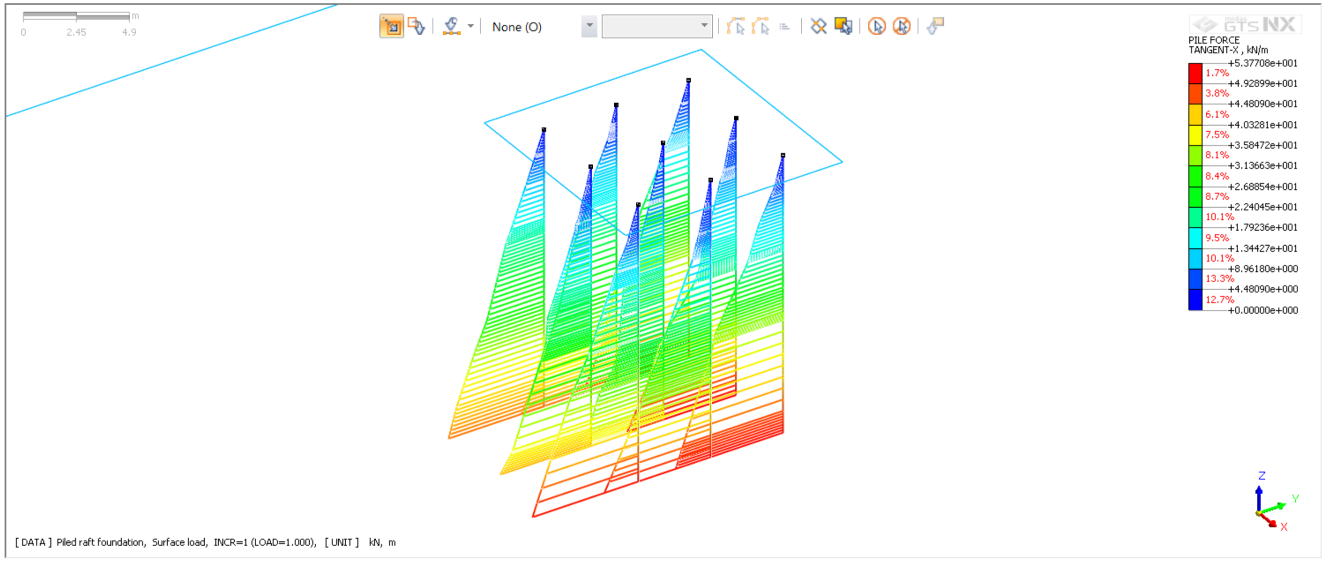
2



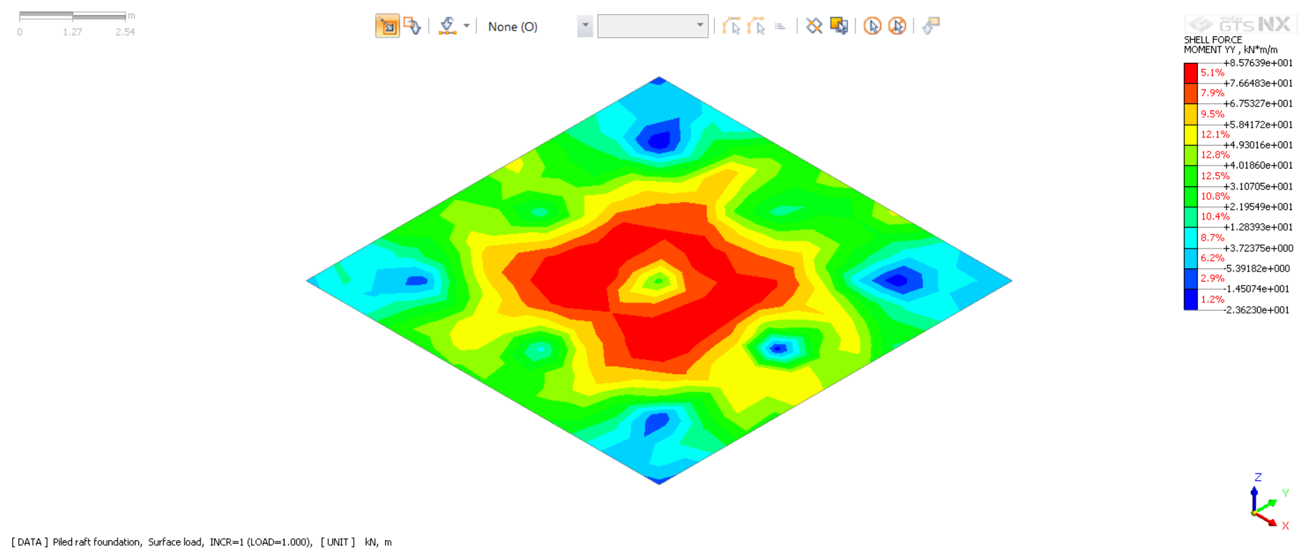


Procedure

1. Surface load > Pile Force > TANGENTIAL X 🎧



2. Surface load > Shell Element Forces > BENDING MOMENT YY



🎧 Shaft friction force between pile and ground