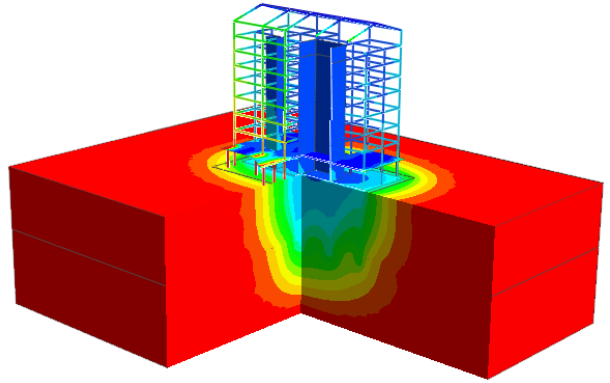
MIDAS *Technical Material*

Tutorial



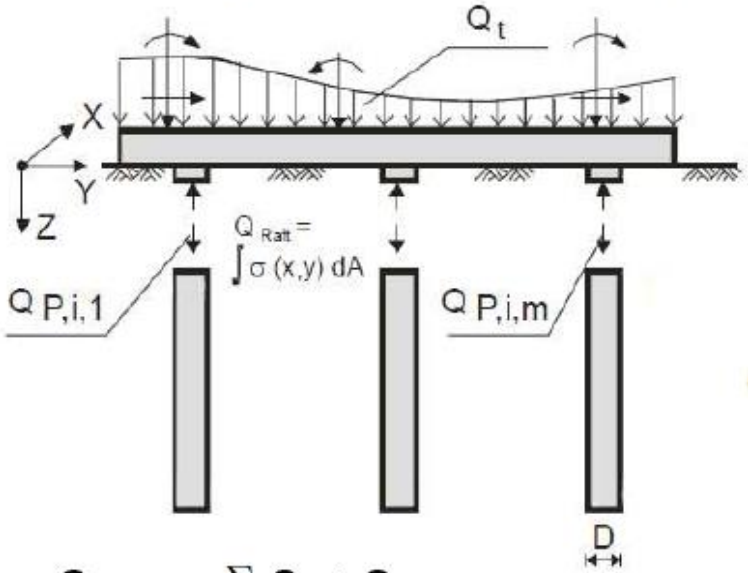
Pile-Raft Foundation Analysis

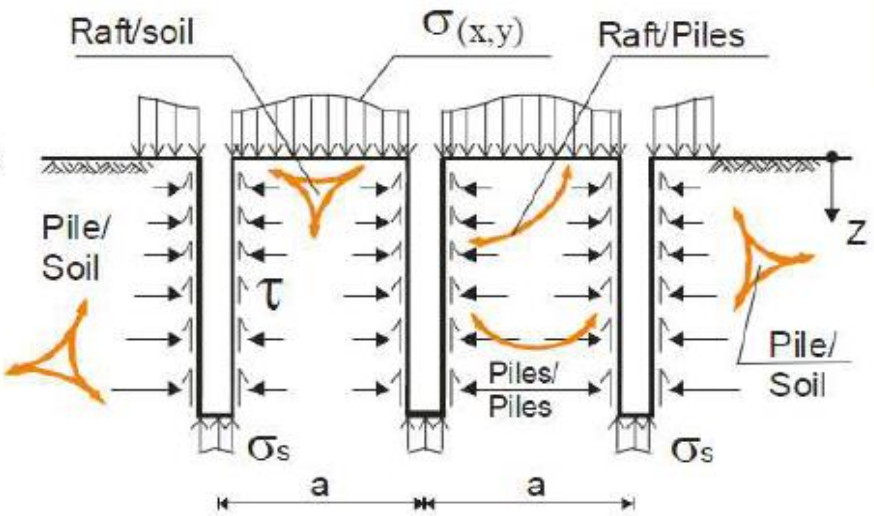
Integrated Solver Optimized for the next generation 64-bit platform

Finite Element Solutions for Geotechnical Engineering

Pile-Raft Foundation

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

Learning Objective

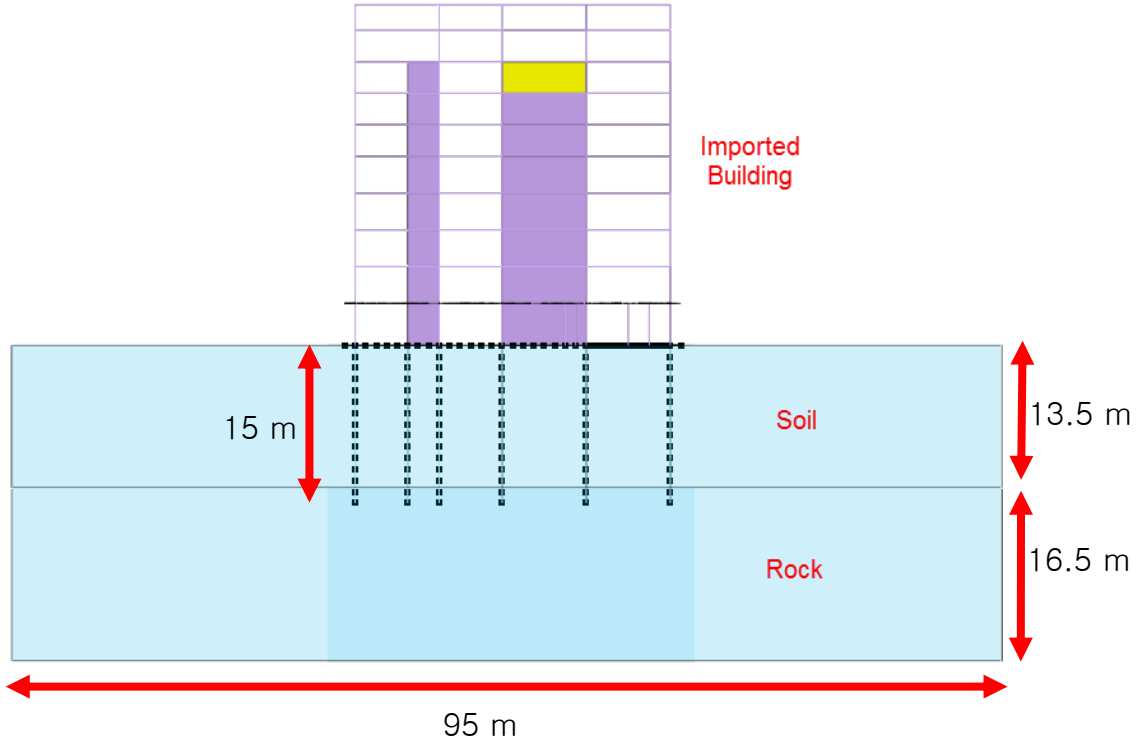
Overview

- This tutorial will explain the steps to be followed to perform Stress Analysis for Pile Raft Foundation.
- The modelling of interaction between the piles and the soil will be dealt.

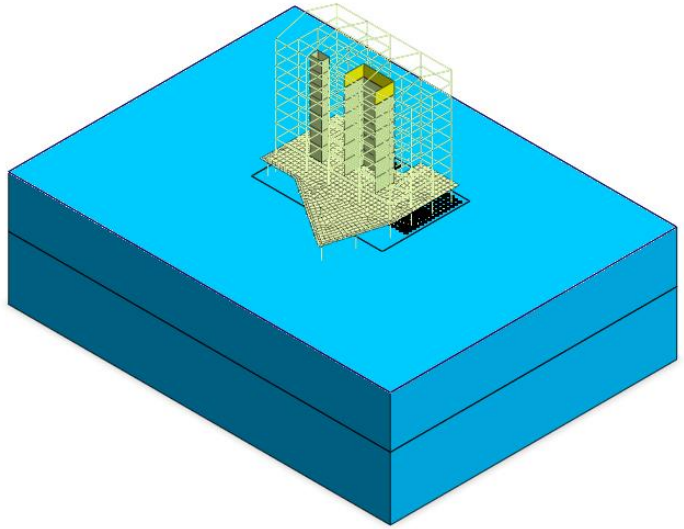
Workflow

STEP 1	STEP 2	STEP 3	STEP 4	STEP 5	STEP 6	STEP 7
- Import the Building and the Load Set - Geometric Modelling of the Soil Layers	Material and Property Definition	Meshing	Load and Boundary Conditions	Construction Stage Definition	Analysis Case Definition	Perform and Analyse Results

Building Model and Subsurface Layers



Imported Building and Soil Profile – 2D section



3D model

Material Properties

Table-1: Material Properties

Material	Model Type	Elastic Modulus (kN/m^2)	Poisson's Ratio	Unit Weight (kN/m^3)	Saturated Unit Weight (kN/m^3)	Initial Void Ratio	Cohesion (kN/m^2)	Friction angle (degrees)
Soil	Isotropic Mohr-Coulomb	50,000	0.3	20	21	0.5	30	36
Siltstone	Isotropic Mohr-Coulomb	210,000	0.3	22	22	0.5	205	27

Table-2: Interface Properties

Material	Model Type	Ultimate Shear Force (kN/m^2)	Shear Stiffness Modulus (kN/m^3)	Normal Stiffness Modulus (kN/m^3)
Pile Interface_Soil	Interface and Pile	320	32120	76923
Pile Interface_Rock	Interface and Pile	2753	275317.7	323076.9

Note: **If the resistance due to skin friction from rock layer is to be exempted, then the values of ultimate shear force and ultimate shear modulus for Pile Interface_Rock may be inputted as 0.**

Step **01**

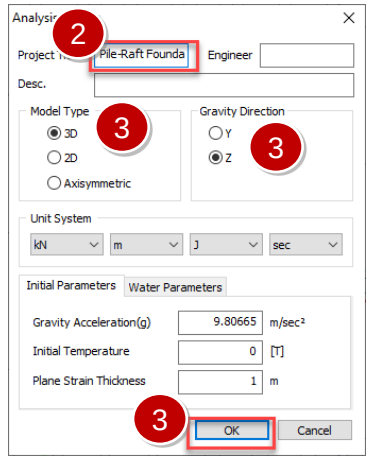
1-1 Import Building from Midas Gen

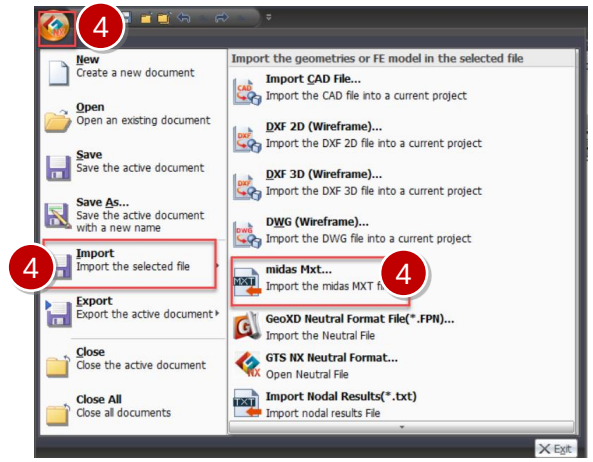
Procedure

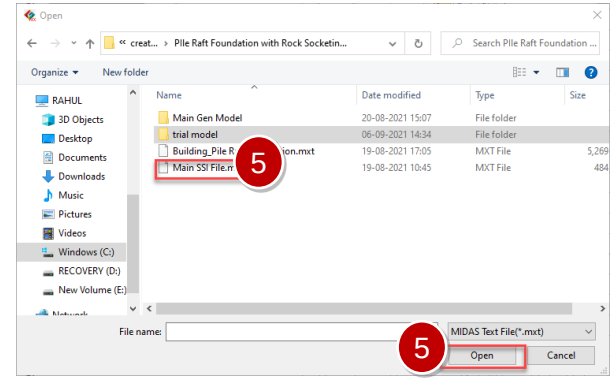
1. Open GTS NX.
2. Project Title: enter “**Pile-Raft Foundation Analysis**” , set units as **kN, m and sec**
3. Select Model type **3D** and Gravity **direction Z** and click **‘OK’**
4. Click on GTS NX icon on the top left corner of Interface and select **Import > midas Mxt.**
5. Select **“Main SSI File.mxt”** and click ‘Open’.
6. In the ‘Model-workstree’, expand **Mesh** and select/click **Point Spring.**
7. Press **Delete** on keyboard and click **‘Ok’.**

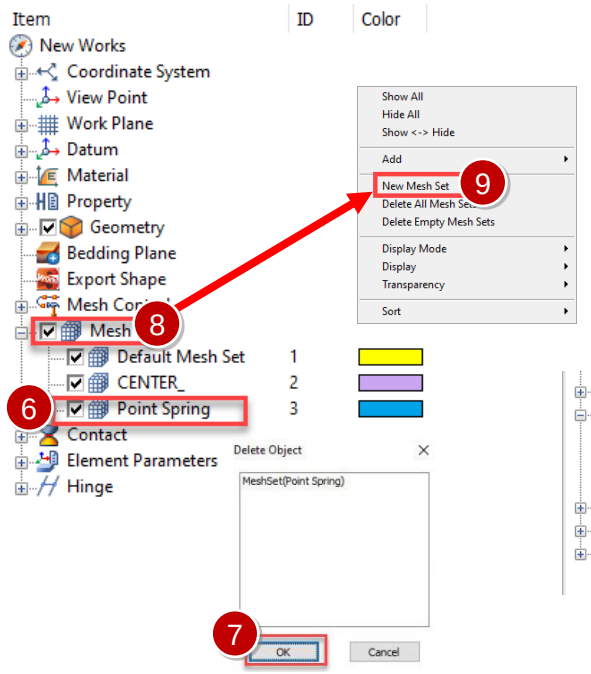
Now we will be separating the Piles and include it in a new Mesh Set.

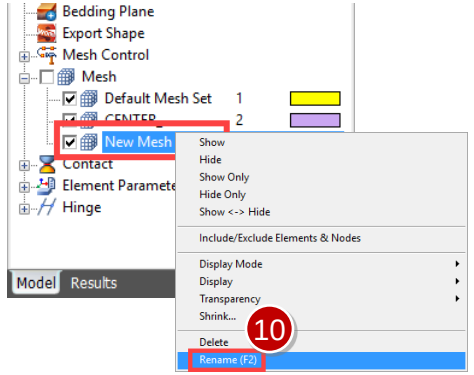
8. Right click on the **Mesh** using the cursor.
9. Click on ‘New Mesh Set’
10. Right click on **‘New Mesh Set’** and click on **‘Rename (F2)’** and change the name to **‘Piles’.**
11. Then right click again on the **‘Piles’** Mesh set and click **on ‘Include/Exclude Elements and Nodes’**
12. Click on **‘Front view’**
13. Click on ‘**Select Objects’**
14. Click and enable **‘Intersect’**
15. Select Piles by using the cursor and make sure that the raft is not being included.
16. Click **‘OK’**
17. Disable **‘Intersect’** by clicking on it.

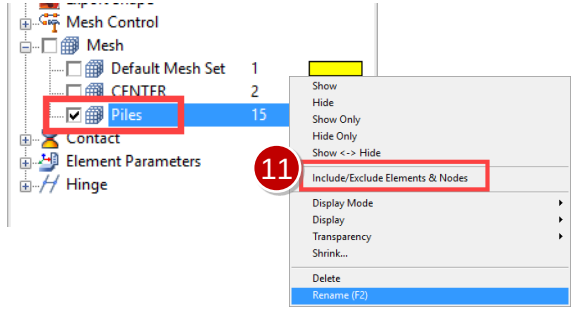


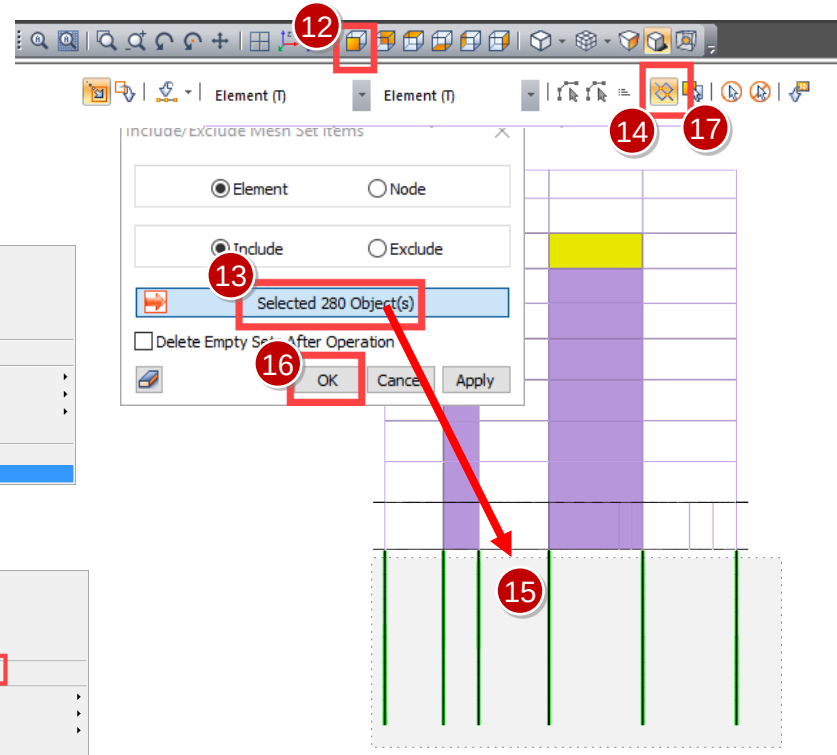










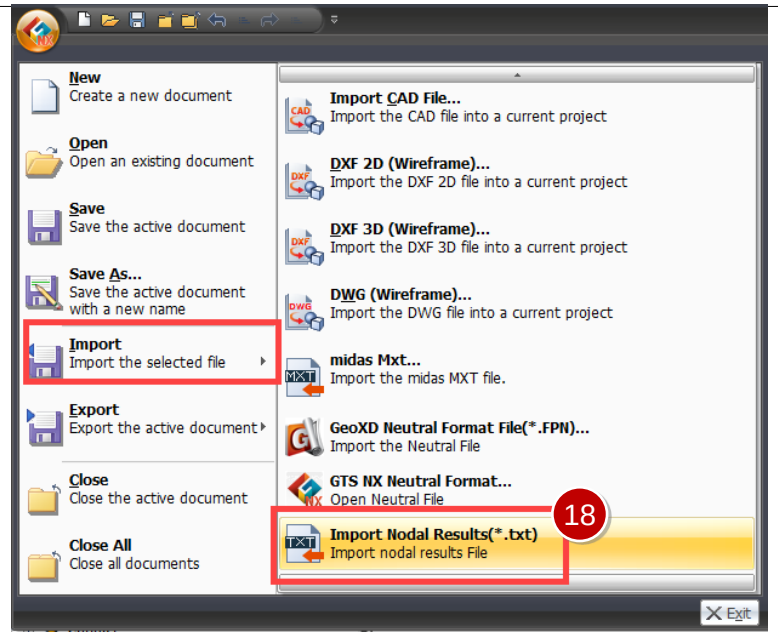


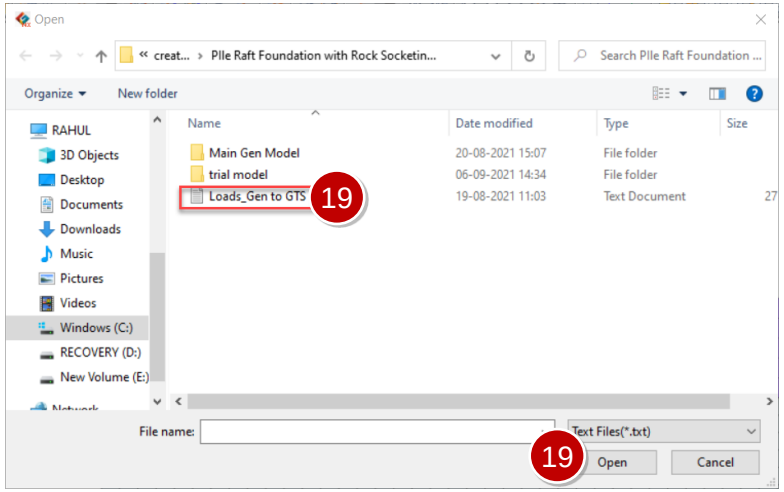
Step 01

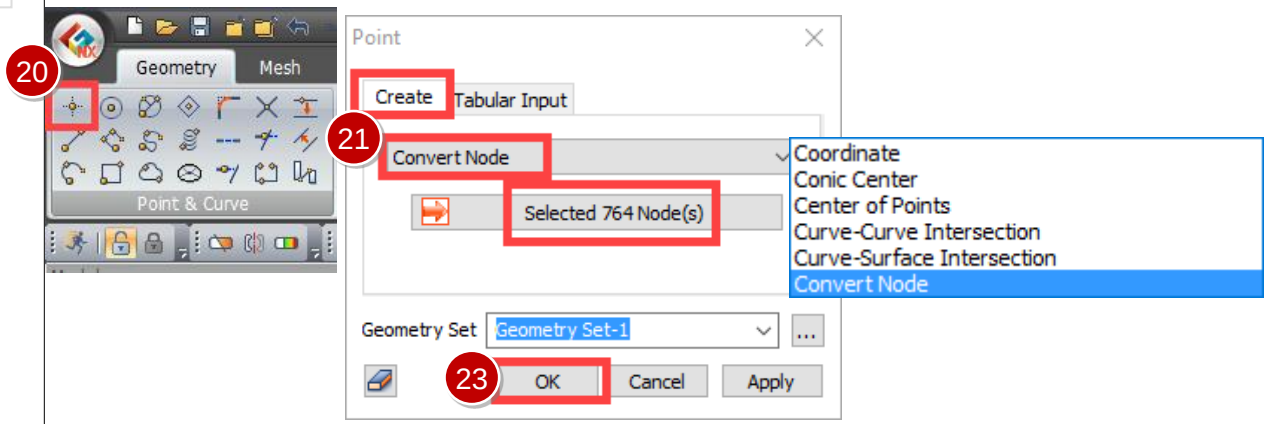
1-2 Import Load from Midas Gen

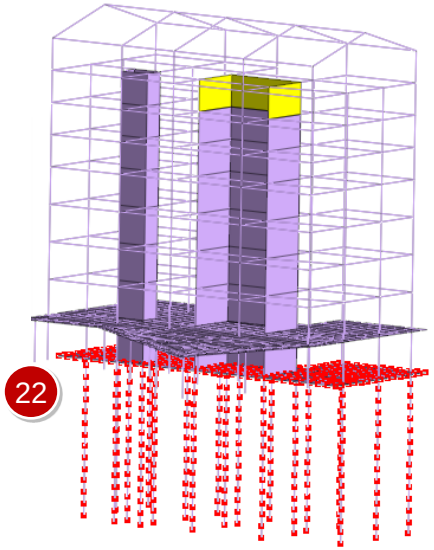
Procedure

18. Click on **GTS NX Icon**>**Import > Import Nodal Results (*.txt)**
19. Select **“Loads_Gen to GTS”** and click **‘Open’**
20. Go to Geometry > Point & Curves > **Point**
21. Click on ‘**Create**’ and select ‘**Convert Node**’ from the dropdown menu.
22. Select all the nodes of Raft and the Pile as shown in the Picture.
23. Click **‘OK’**





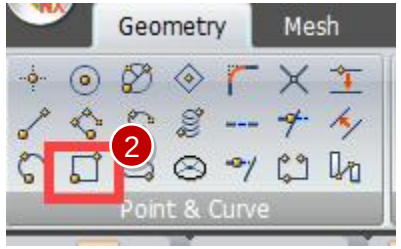


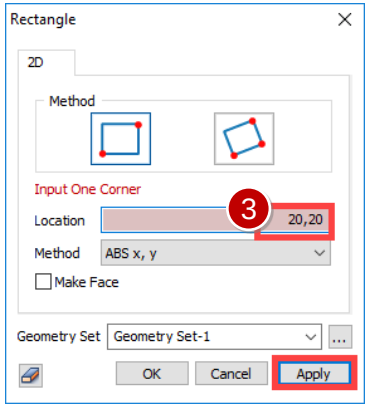


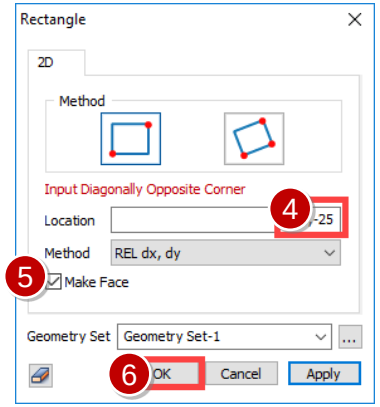
1-3 Soil Strata Modeling

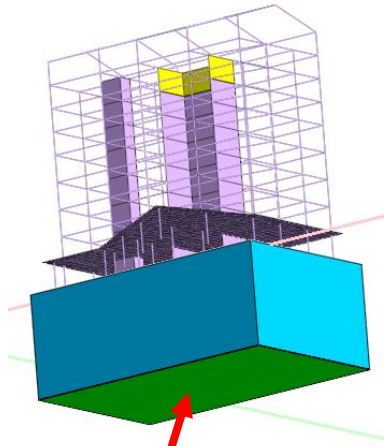
Procedure

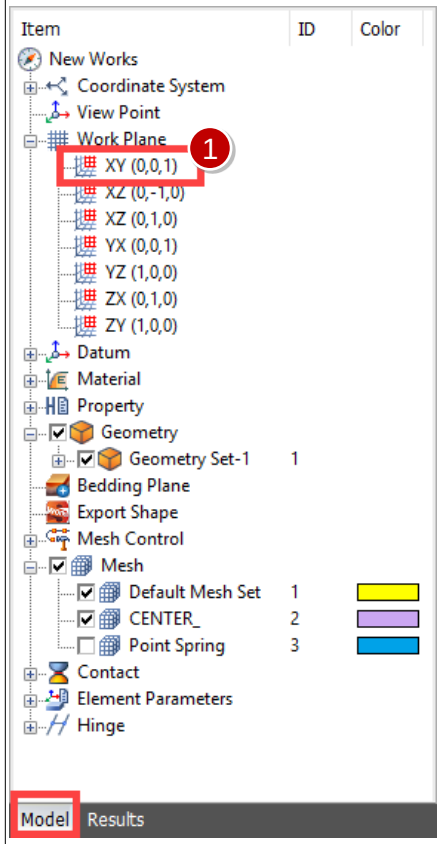
1. Go to Works tree > Model > Work Plane > **Double click on 'XY(0,0,1).**
2. Go to **Geometry> Point and Curve > Rectangle.**
3. Input the first corner of the rectangle as **(20,20) and** click 'Apply'
4. Input the opposite corner of the rectangle as **(40,-25)**
5. Enable **'Make Face'**
6. Click **'OK'**
7. **Go to Geometry > Protrude > Extrude**
8. Select the Surface as shown in the figure.
9. Select the **Z direction** using the Cursor.
10. Input the length as **-13.5m**
11. Click **'Apply'.**
12. Select the bottom surface of the Extruded Solid
13. Select **'Z'** direction
14. Input the length as **-16.5m**
15. Click **'OK'**

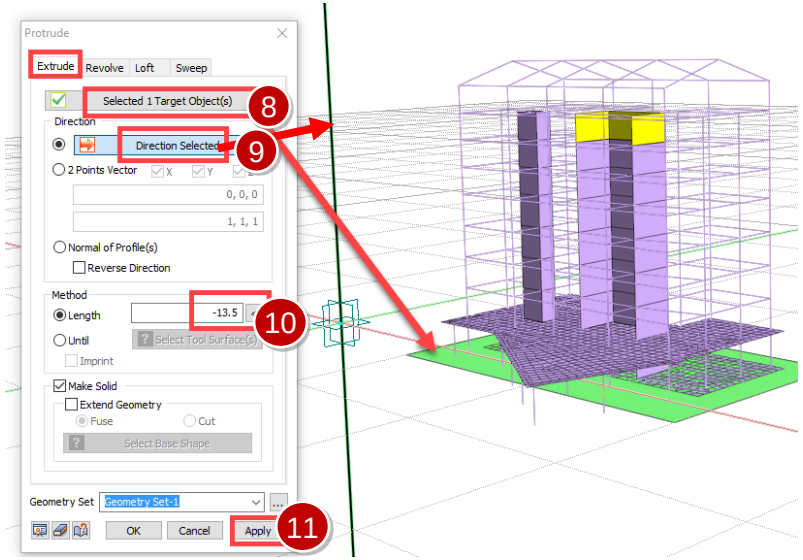


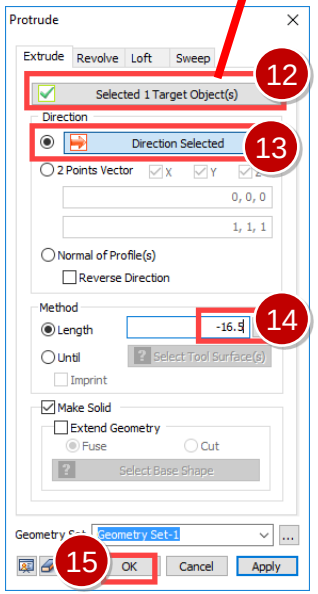








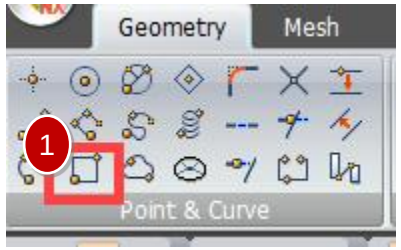


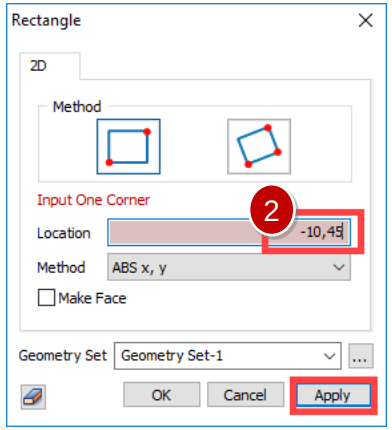


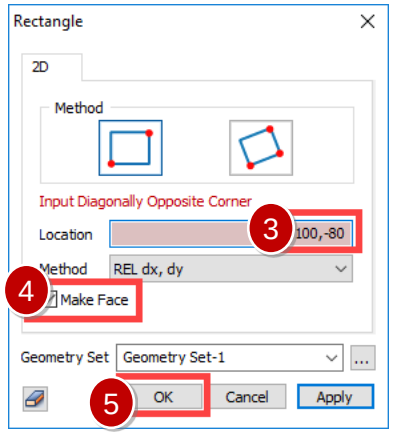
1-4 Soil Strata Modeling

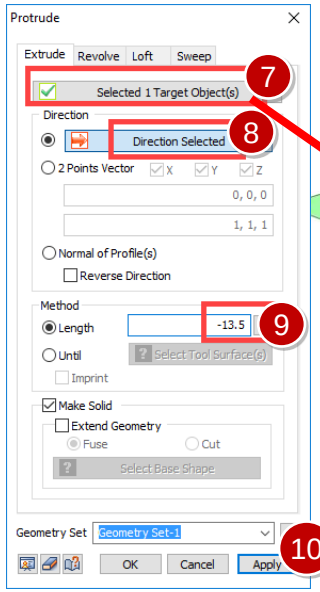
Procedure

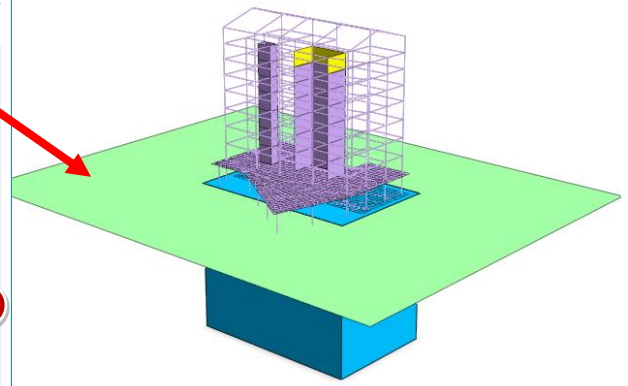
1. Go to Geometry > Point & Curve > **Rectangle**
2. Input the first corner of the rectangle as **(-10,45) and** click 'Apply'
3. Input the opposite corner of the rectangle as **(100,-80)**
4. Enable **'Make Face'**
5. Click **'OK'**
6. Go to Geometry > Protrude **> Extrude**
7. Select the Surface as shown in the figure.
8. Select the **Z direction** using the Cursor.
9. Input the length as **-13.5m**
10. Click **'Apply'.**
11. Select the bottom surface of the Extruded Solid
12. Select **'Z'** direction
13. Input the length as **-16.5m**
14. Click **'OK'**

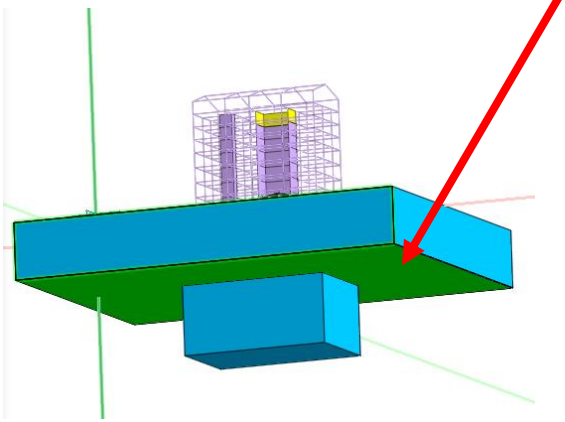


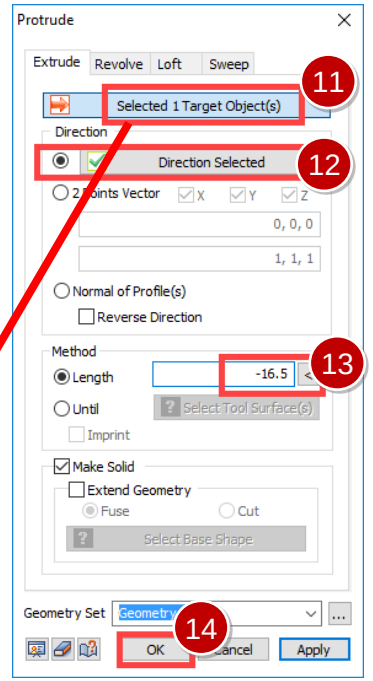










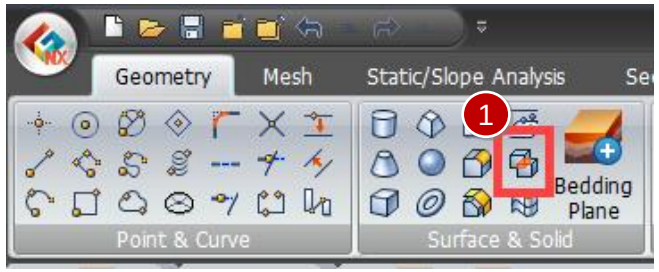


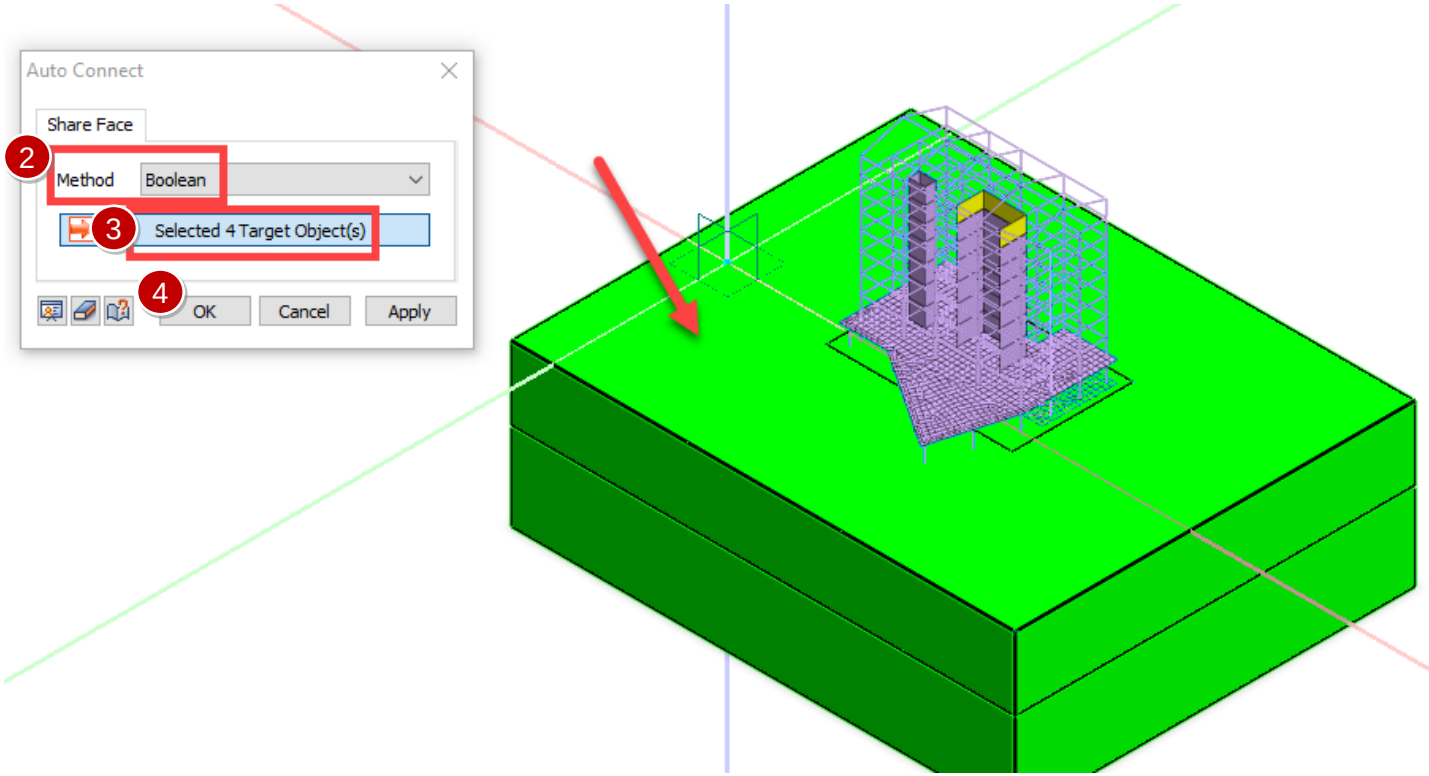
The reason behind the modeling of two separate inner and outer solid domains is to facilitate the finer mesh for the inner solids as it interact with Piles.

1-5 AutoConnect

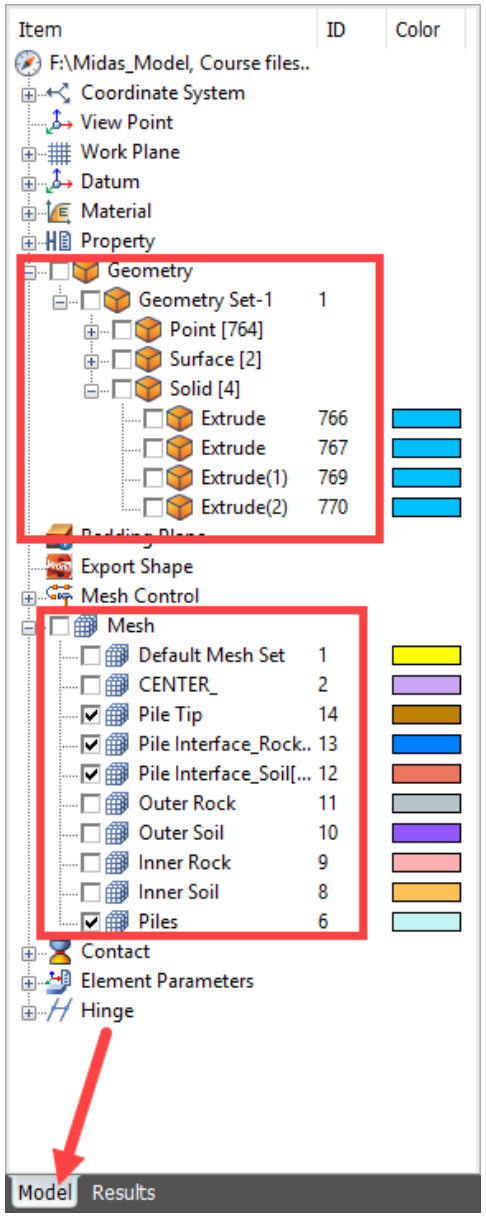
Procedure

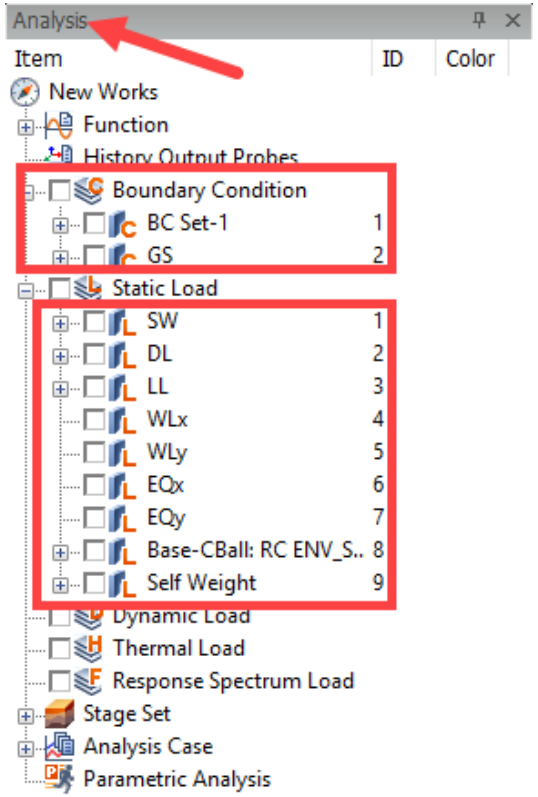
1. Go to Geometry > Surface & Solid> **AutoConnect**
2. Choose the Method as ‘Boolean’
3. Select all the four Solids.
4. Click **‘OK’**





Additional Information





- All the Geometry Objects (Points, Lines, Surfaces, Solids) and Mesh Sets are saved in the **Model** tab of the WorksTree.
- The Boundary Conditions and Load Sets are saved in the **Analysis** Tab of the WorksTree.

Kindly **Hide or Unhide** them as per your convenience while **Modelling** and **Viewing** Results.

- For the current tutorial, we would recommend you hiding the load sets after importing it for better Modeling Experience.

2-1 Material Definition

Procedure

1. Go to **Mesh** > **Prop/Csys/Func >Material**
2. Click on **Create** > **Isotropic**
3. Select Model type > **Mohr-Coulomb**
4. Enter the Name as **'Soil'**
5. In **General tab**

   Elastic Modulus > **50000 kN/m^2**

   Poisson's Ratio > **0.3**

   Unit Weight > **20 kN/m^3**
6. In **Porous tab**

   Saturated unit weight > **21 kN/m^3**

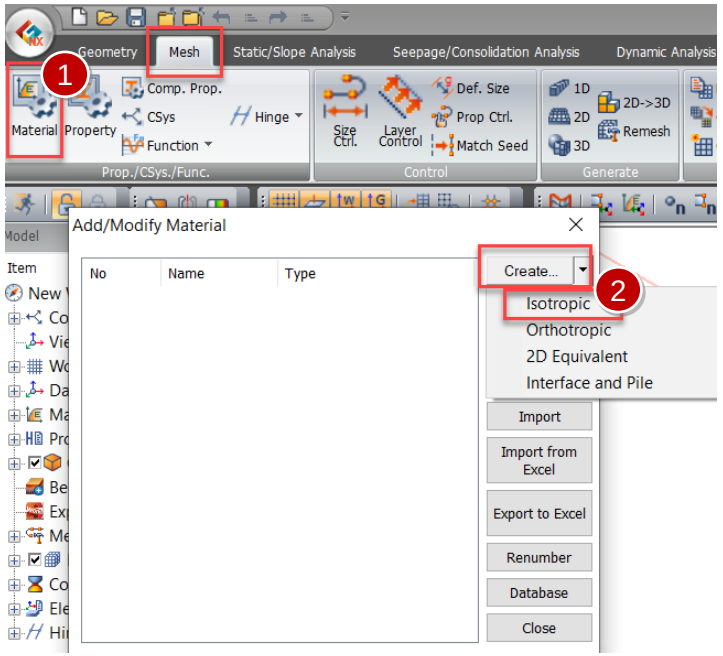
   Initial Void Ratio > **0.5**
7. In **Non-Linear tab**

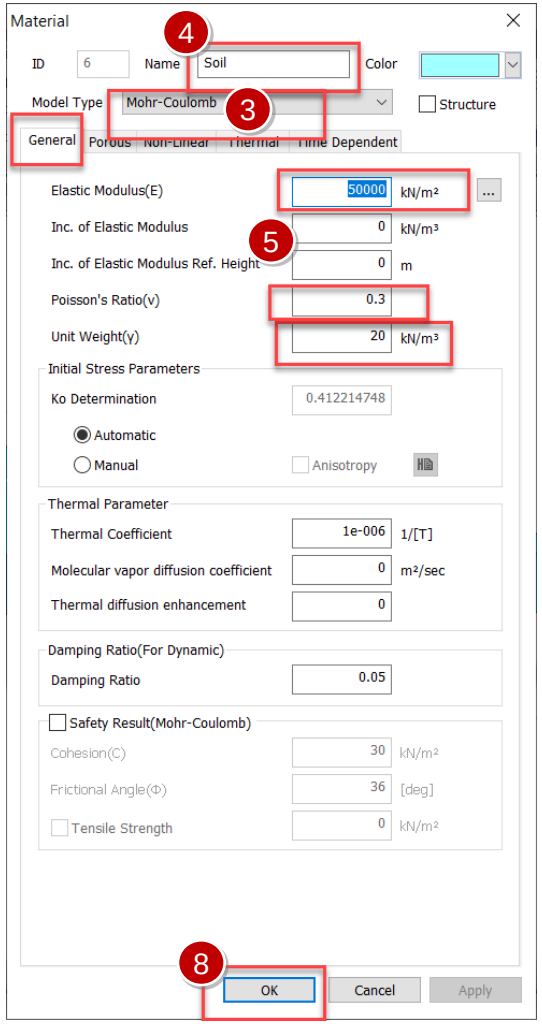
   cohesion > **30 kN/m^2**

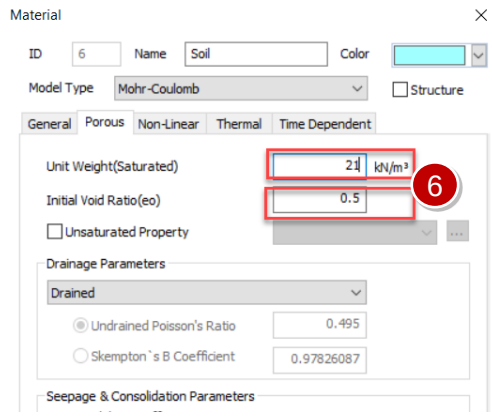
   Friction angle > **36 degrees**
8. Click **'OK'**

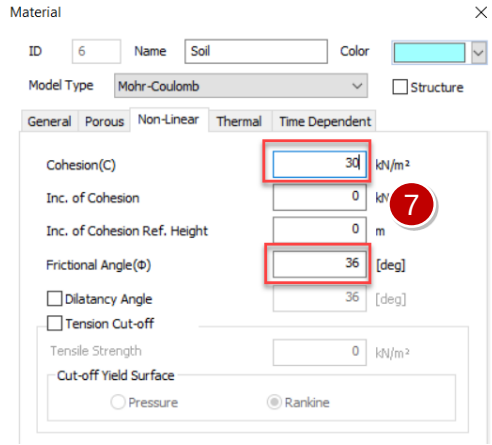
Similarly define the inputs for Siltstone Material as given in Tables 1.

.









Step 02

2-2 Pile Interface - Material Definition

Procedure

Go to **Mesh** > **Prop/Csys/Func >Material**

1. Click on **Create** > **Interface and Pile**
2. Select Model type > **Pile**
3. Enter the Name as **'Pile Interface_Soil'**
4. In **General tab**

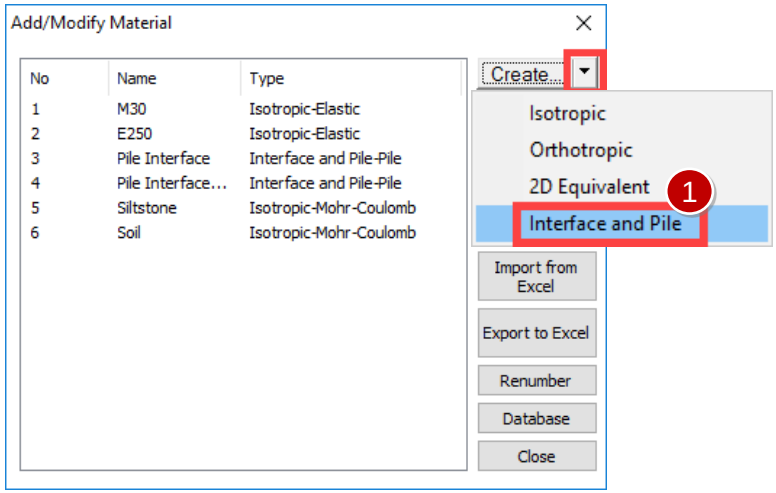
   Ultimate Shear Force > **320 kN/m^2**

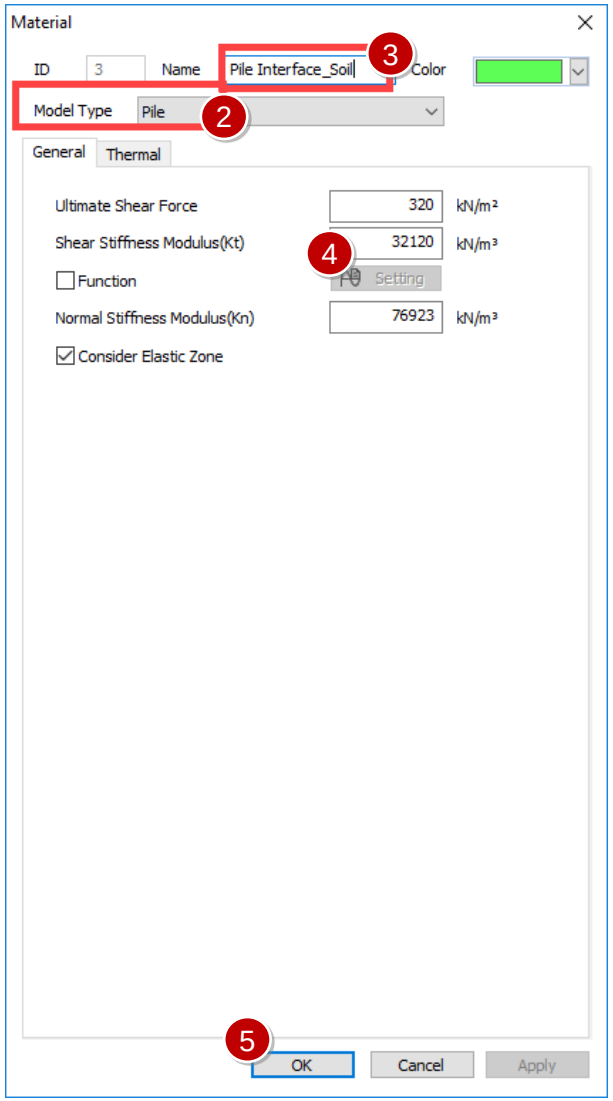
   Shear Stiffness Modulus **> 32120 kN/m3**

   Normal Stiffness Modulus **> 76923 kN/m3**
5. Click **'OK'**

Similarly define the inputs for 'Pile Interface_Rock' Material as given in Table 2.

.





Step 02

2-3 Defining Property

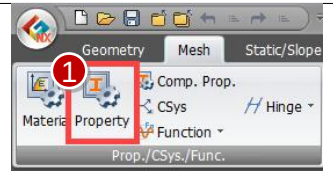
Procedure

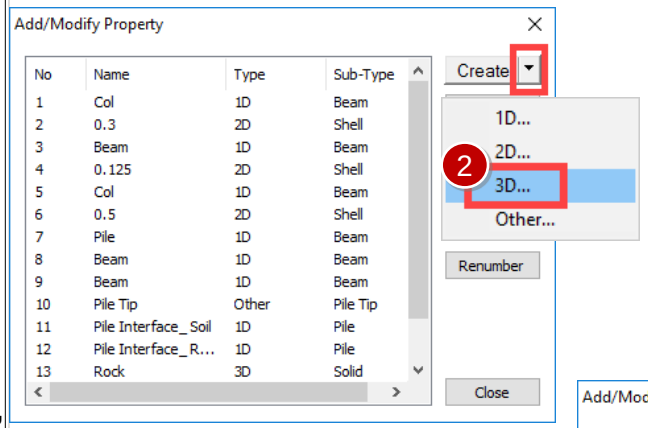
1. Go to **Mesh > Prop/Csys/Func > Property**
2. Select **Create > 3D**
3. Select **Material > Soil**
4. Enter property name **'Soil'**
5. Click **'Apply'**
6. Select **Material > Siltstone**
7. Enter property name **'Rock'**
8. Click **'OK'**
9. Select **Create > 1D**
10. Select **'Pile'**
11. Select the Material **'Pile Interface_Soil'**
12. Enter the property name **'Pile Interface_Soil'.**
13. Click **'Apply'**

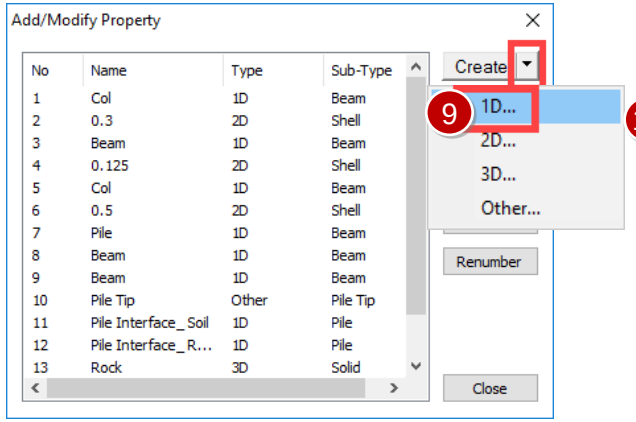
Similarly define the inputs for 'Pile Interface_Rock' Material as given in Table-2.

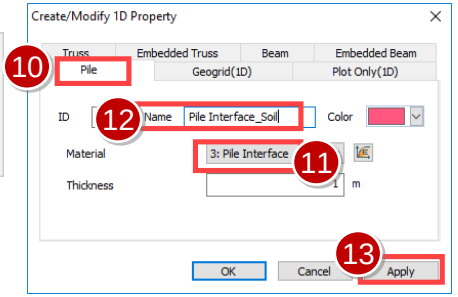
14. Select **Create > Other**
15. Select **'Pile Tip'**
16. Enter name **'Pile Tip'**
17. Input Tip Bearing Capacity**> 8138 kN**

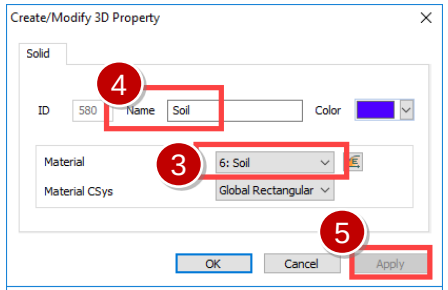
    Tip Spring Stiffness**> 521285 kN/m**
18. Click **'OK'**
19. Click **'OK'**

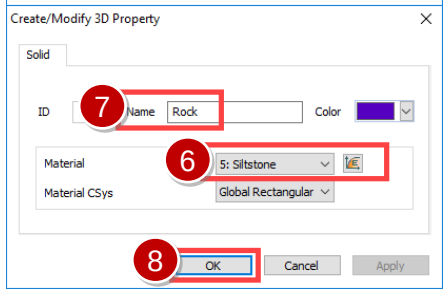


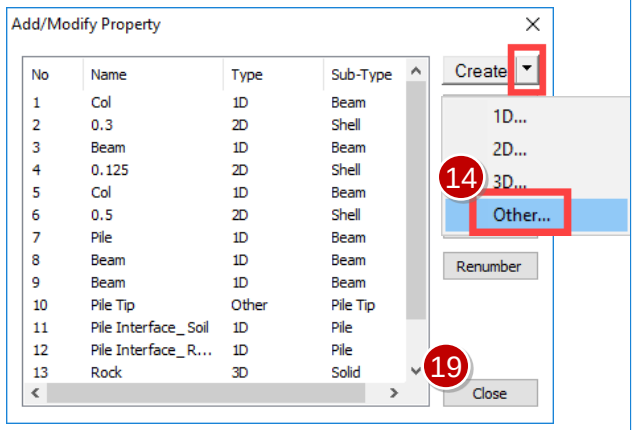


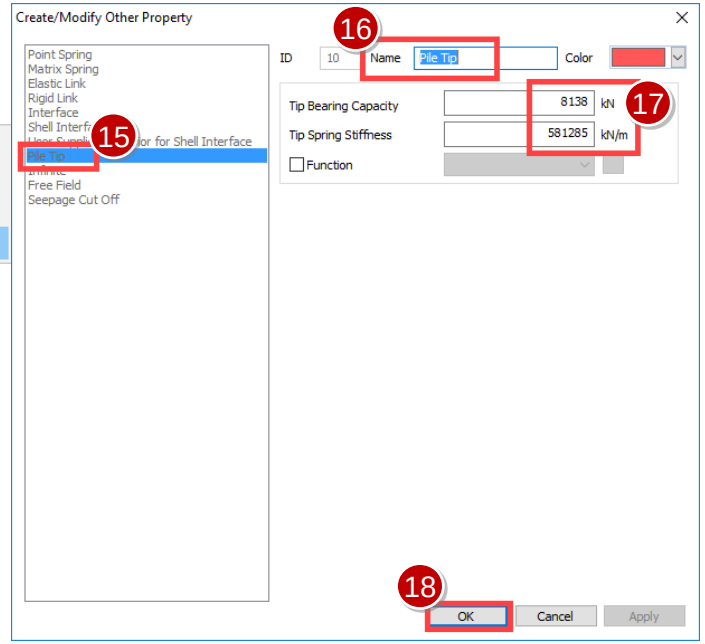










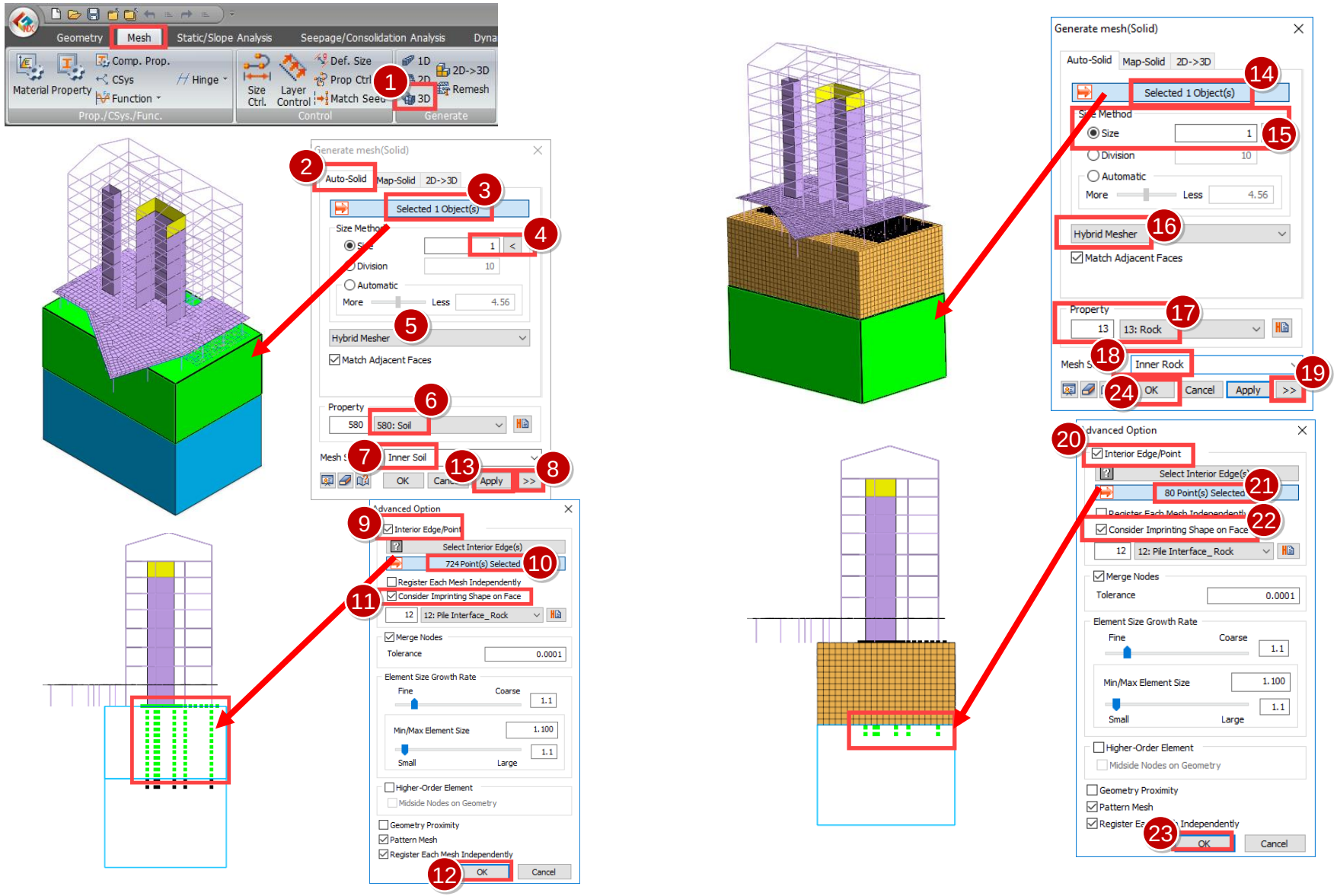


Step 03

3-1 Meshing Soil and Rock Layers

Procedure

1. Go to **Mesh** > **Generate > 3D**
2. Go to '**AutoSolid**' Tab
3. Select the **Inner Soil - Solid**
4. Input the Mesh Size **1m.**
5. Select '**Hybrid Mesher**'
6. Select the '**Soil**' **property.**
7. Input name '**Inner Soil**'
8. Go to '**Advance Option**'
9. Enable '**Inner Edge/Points**'
10. Select the **Points** inside the Soil Layer
11. Enable '**Consider Imprinting Shape on Face**'
12. Click '**OK**'
13. Click '**Apply**'
14. Select the **Inner Rock - Solid**
15. Input the Mesh Size **1m.**
16. Select '**Hybrid Mesher**'
17. Select the '**Rock**' **property.**
18. Input name '**Inner Rock**'
19. Go to '**Advance Option**'
20. Enable '**Inner Edge/Points**'
21. Select the **Points** inside the Soil Layer
22. Enable '**Consider Imprinting Shape on Face**'
23. Click '**OK**'
24. Click '**OK**'

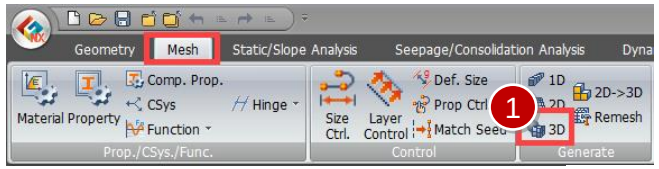


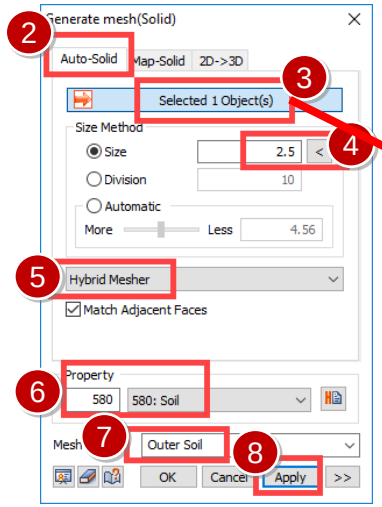
Step 03

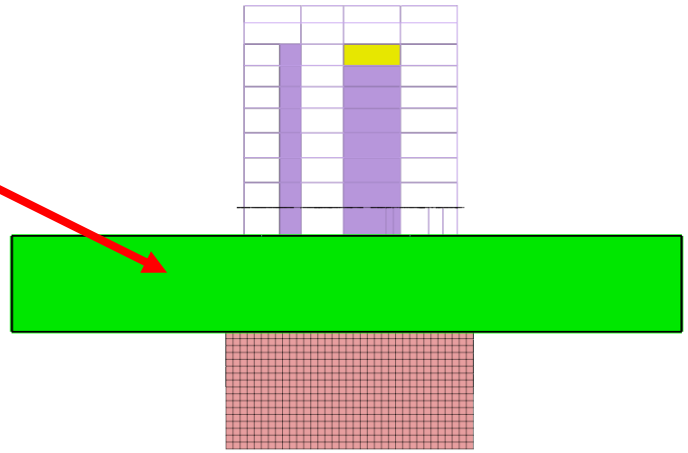
3-2 Meshing Soil and Rock Layers

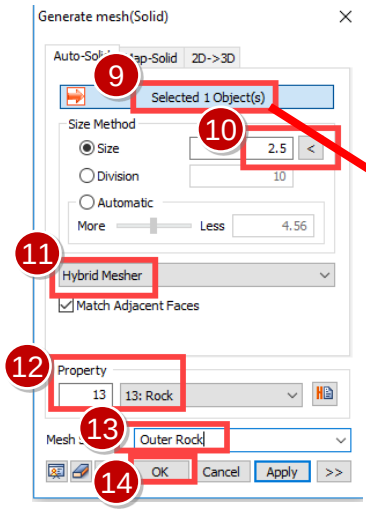
Procedure

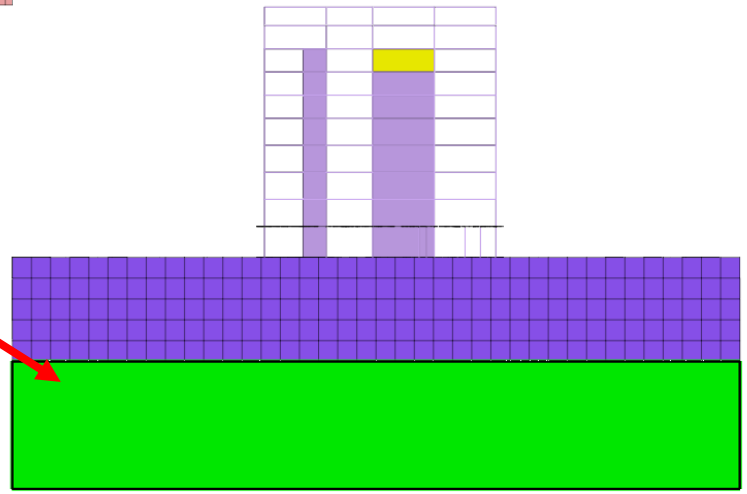
1. Go to **Mesh** > **Generate > 3D**
2. Go to '**AutoSolid**' Tab
3. Select the **Outer Soil - Solid**
4. Input the Mesh Size **2.5m.**
5. Select **'Hybrid Mesher'**
6. Select the **'Soil' property.**
7. Input name **'Outer Soil'**
8. Click **'Apply'**
9. Select the **Outer Rock - Solid**
10. Input the Mesh Size **2.5m.**
11. Select **'Hybrid Mesher'**
12. Select the **'Rock' property.**
13. Input name **'Outer Rock'**
14. Click **'OK'**









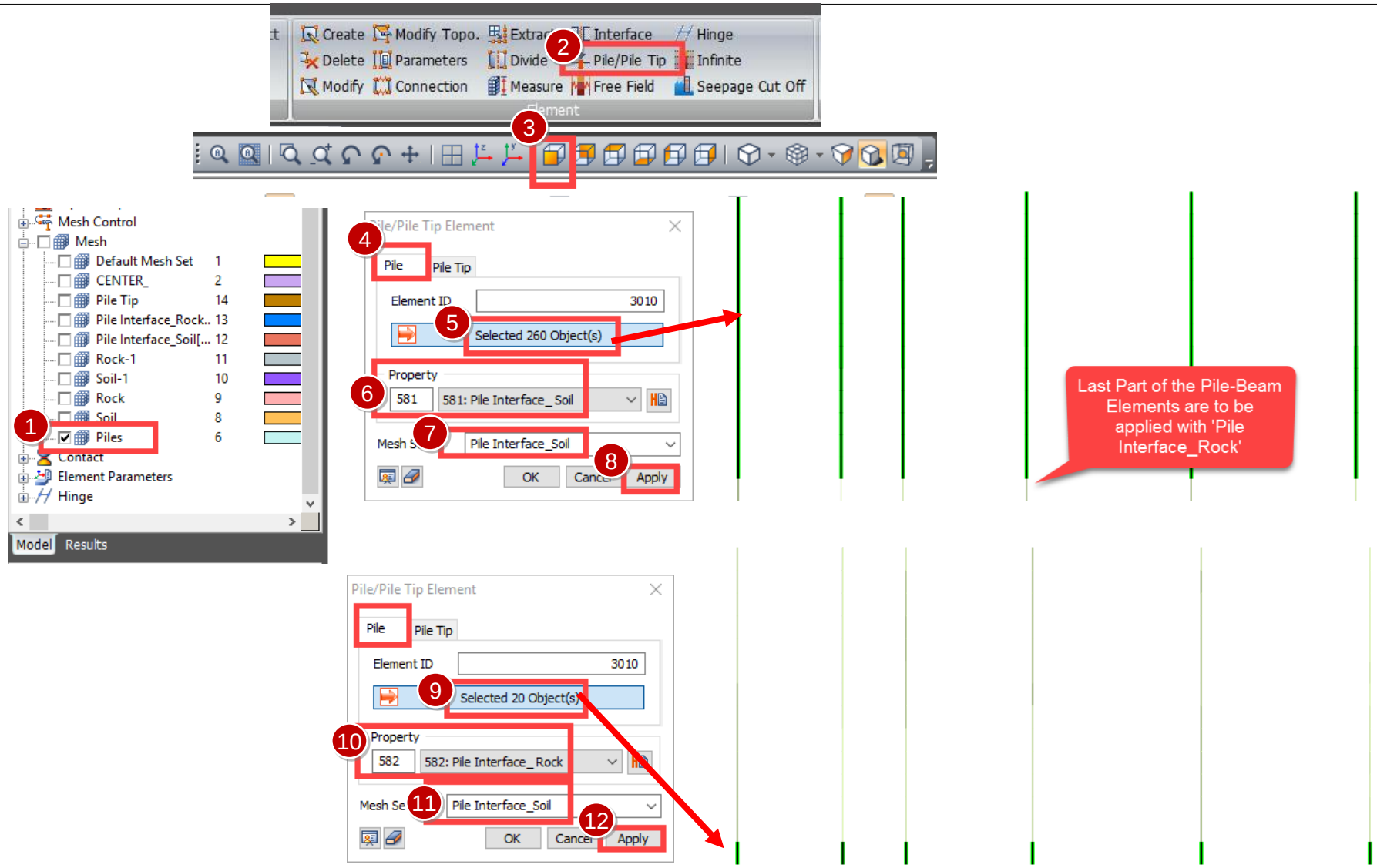


Step
03

3-2 Assigning Pile Interface

Procedure

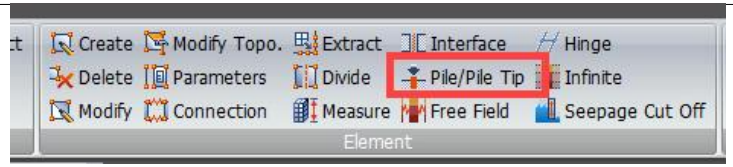
1. Hide all the mesh sets except **'Piles'**
2. Go to **Mesh** > **Element > Pile/Pile Tip**
3. Click **'Front View'**
4. Go to **Pile Tab**.
5. Select all pile elements present in the soil layer.
6. Select Property **Pile Interface_Soil**
7. Name Mesh Set **Pile Interface_Soil**
8. Click **'Apply'**
9. Select all pile elements present in the Rock layer.
10. Select Property **Pile Interface_Rock**
11. Name Mesh Set **Pile Interface_Rock**
12. Click **'Apply'**

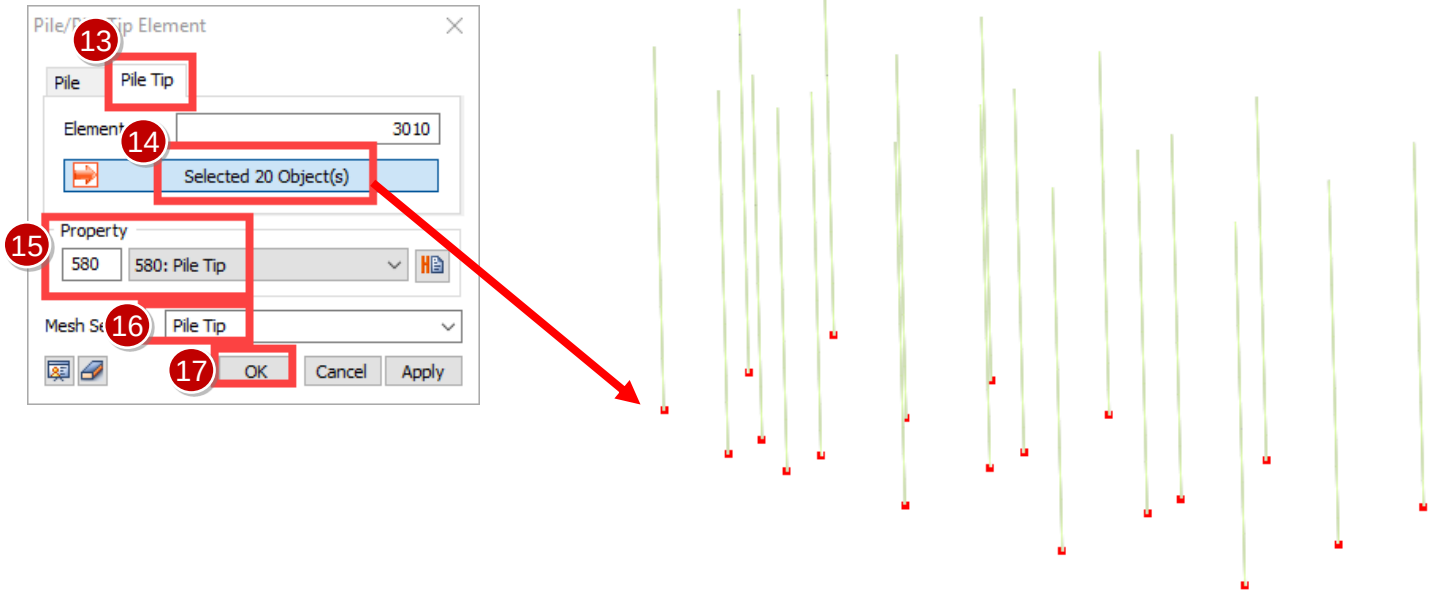


3-2 Assigning Pile Tip

Procedure

13. Go to **Pile Tip** tab.
14. Select the bottom nodes of all the piles as shown in picture.
15. Select the property **'Pile Tip'.**
16. Enter the name as **Pile Tip**
17. Click **'OK'**



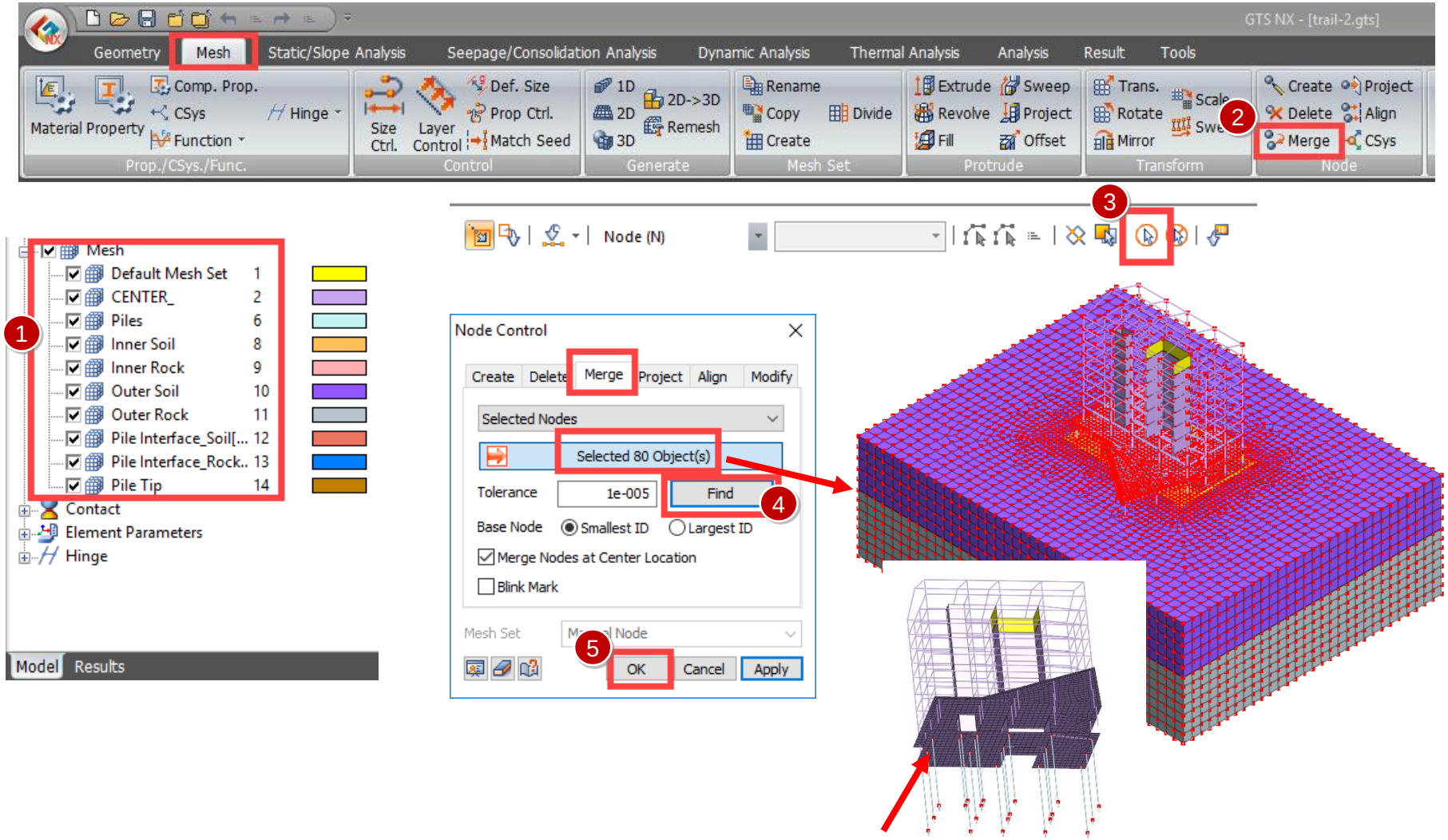


3-3 Modeling Rigid Connection b/w Piles and Raft

Procedure

After creation of the Pile Interface, the common node between the Raft and the Pile is be divided into two and are connected by the interface element. This results in non-rigid behaviour between Pile and the Raft. Hence, we need to merge those nodes to maintain rigid connection.

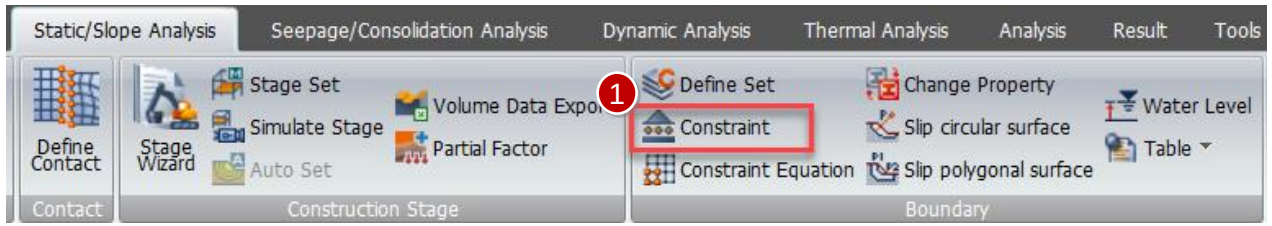
1. **Unhide** all the mesh sets
2. Go to Mesh > Node > **Merge**
3. Select **All** the nodes**.**
4. Click **Find**
5. Click **'OK'**

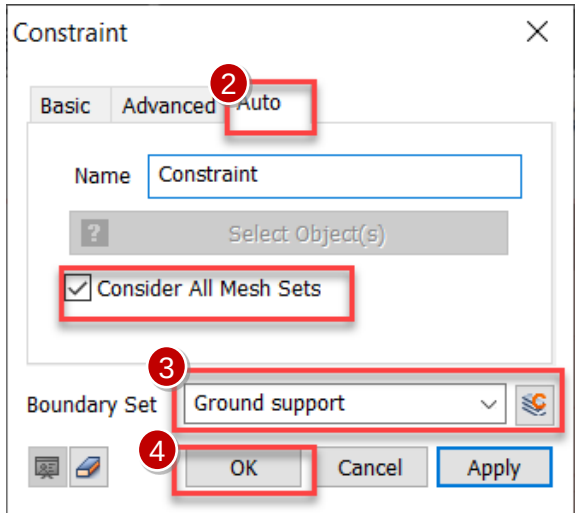


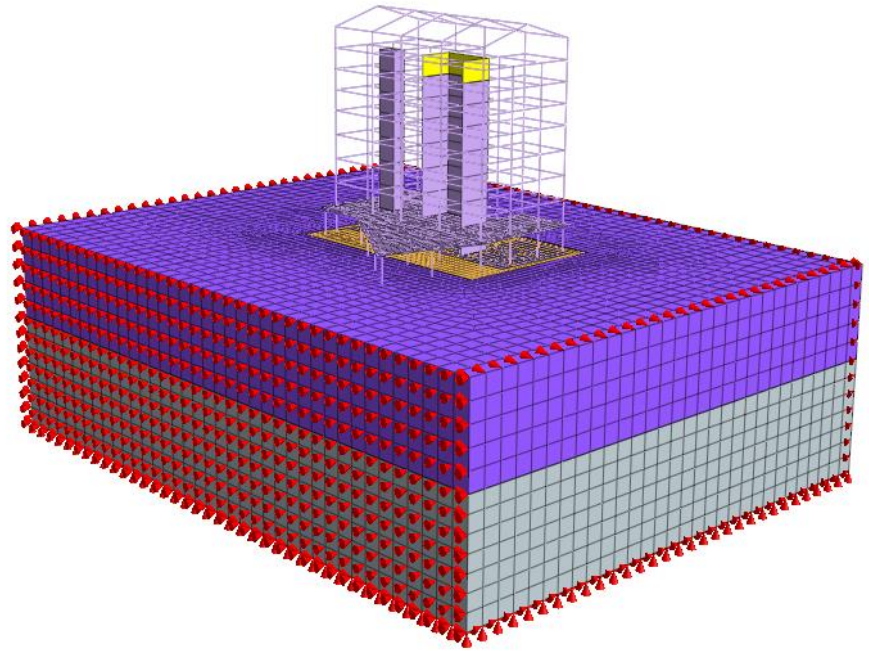
4-1 Assigning Boundary Conditions

Procedure

1. Select **Static/Slope Analysis > Boundary > Constraint**
2. Go to **Auto** tab Check on **Consider All Mesh Sets**
3. Name Boundary Set > **GS**
4. Click **OK**



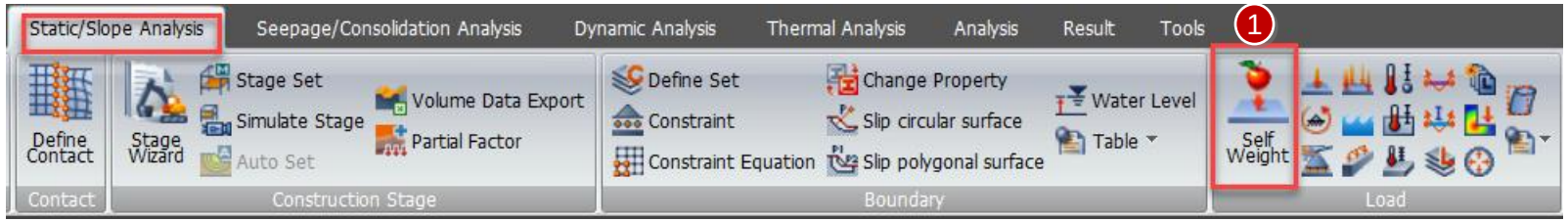


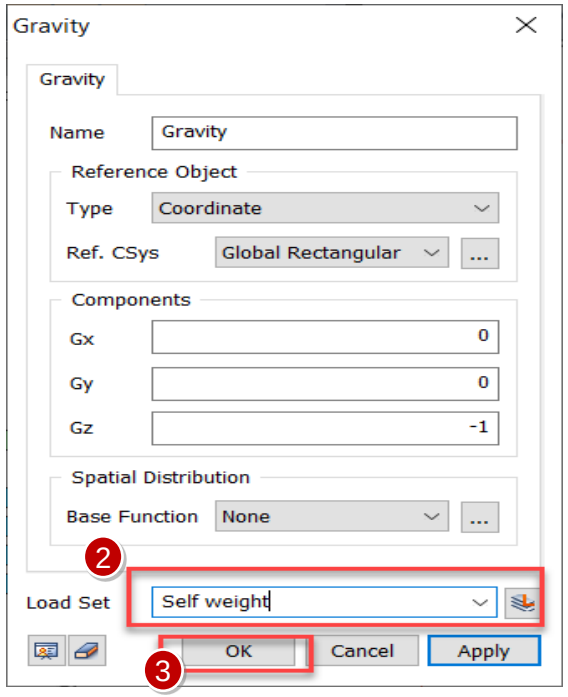


4-2 Assigning Self Weight

Procedure

1. Go to **Static/Slope Analysis > Load > Self Weight**
2. Enter **Load Set** > **Self weight**
3. Click **'OK'**



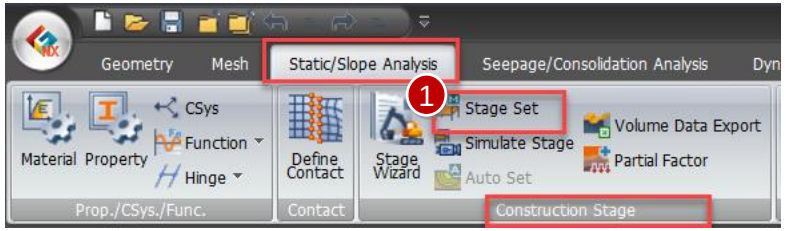


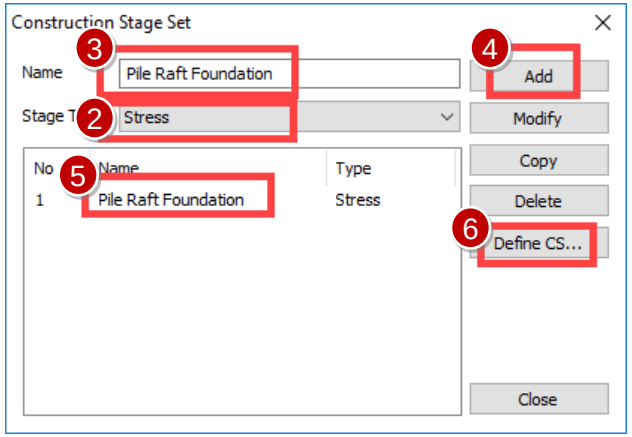
Step 05

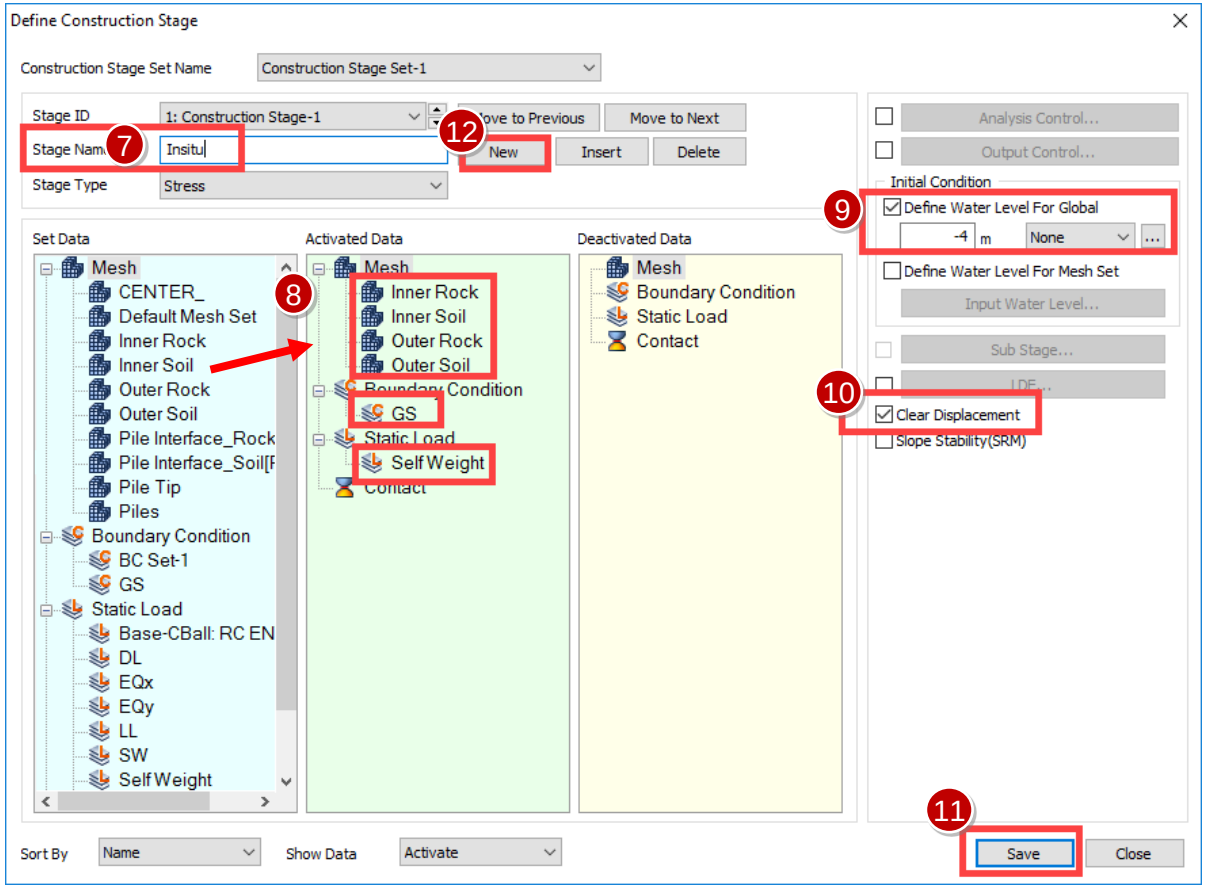
5-1 Construction Stage Definition

Procedure

1. Go to **Static/Slope Analysis > Construction Stage > Stage Set**
2. Set the stage type as **stress.**
3. Enter the name as **Pile Raft Foundation.**
4. Click **'ADD'**
5. Select the generated Construction Stage Set.
6. Click on **'Define CS'**
7. Enter Stage Name > **In-Situ**
8. For the In-situ stage activate the **Inner Soil, Outer Soil, Outer Soil and Outer Rock mesh sets.** Also Activate **Ground Support** and **Self Weight.**
9. Enable Define Water for Global. Enter **'-4' m** in water level column.
10. Enable **'Clear Displacement'**
11. Click **'Save'**
12. Click **'New'**





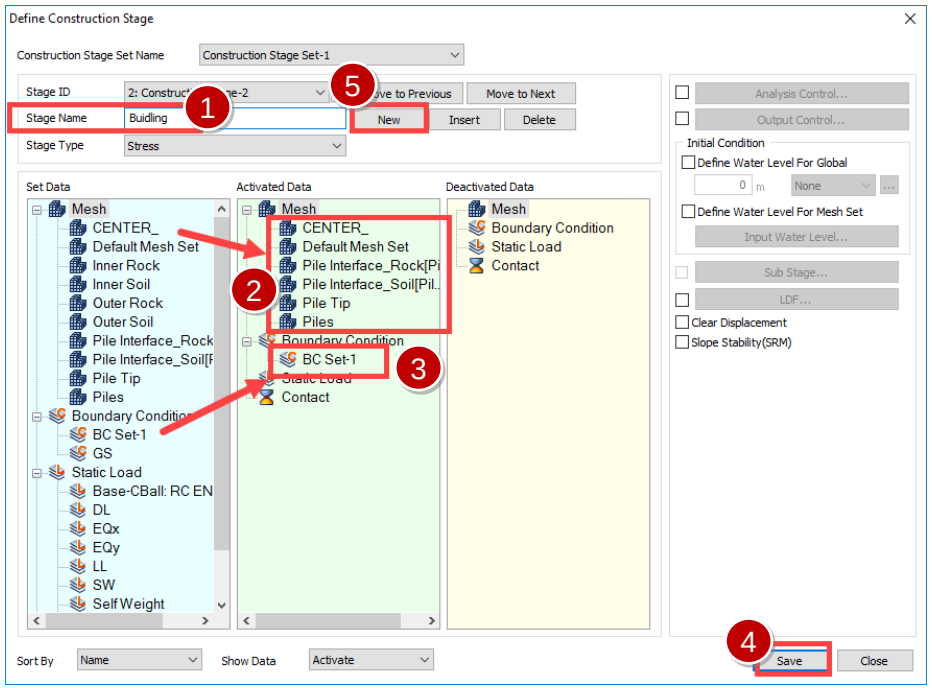


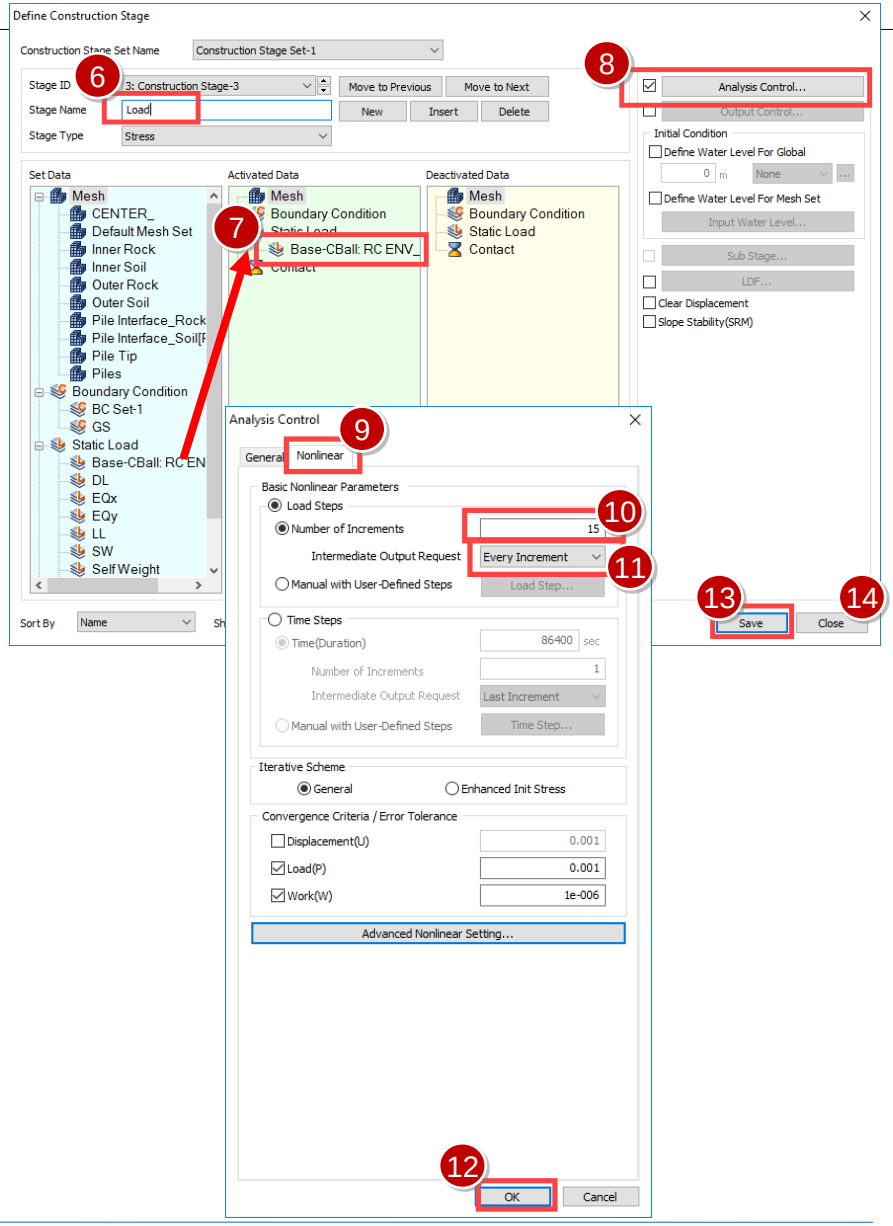
Step 05

5-2 Construction Stage Definition

Procedure

1. Enter Stage Name **Building.**
2. Activate **Center_, Default Mesh set, Piles, Pile Interface_Soil, Pile Interface_Rock** and **Pile Tip** mesh sets.
3. Activate the boundary condition **BC Set-1.**
4. Click **‘Save’**
5. Click **‘New’**
6. Enter Stage Name **Load.**
7. Activate load set **‘Base-CBall: RC ENV_STR’**
8. Enable ‘**Analysis Control**’ and **Open** it.
9. Go to **‘Nonlinear’.**
10. Enter the number of increments **as 15.**
11. Select ‘**Every Increment**’ in the dropdown menu.
12. Click **‘OK’**
13. Click **‘Save’**
14. Click **‘Close’**



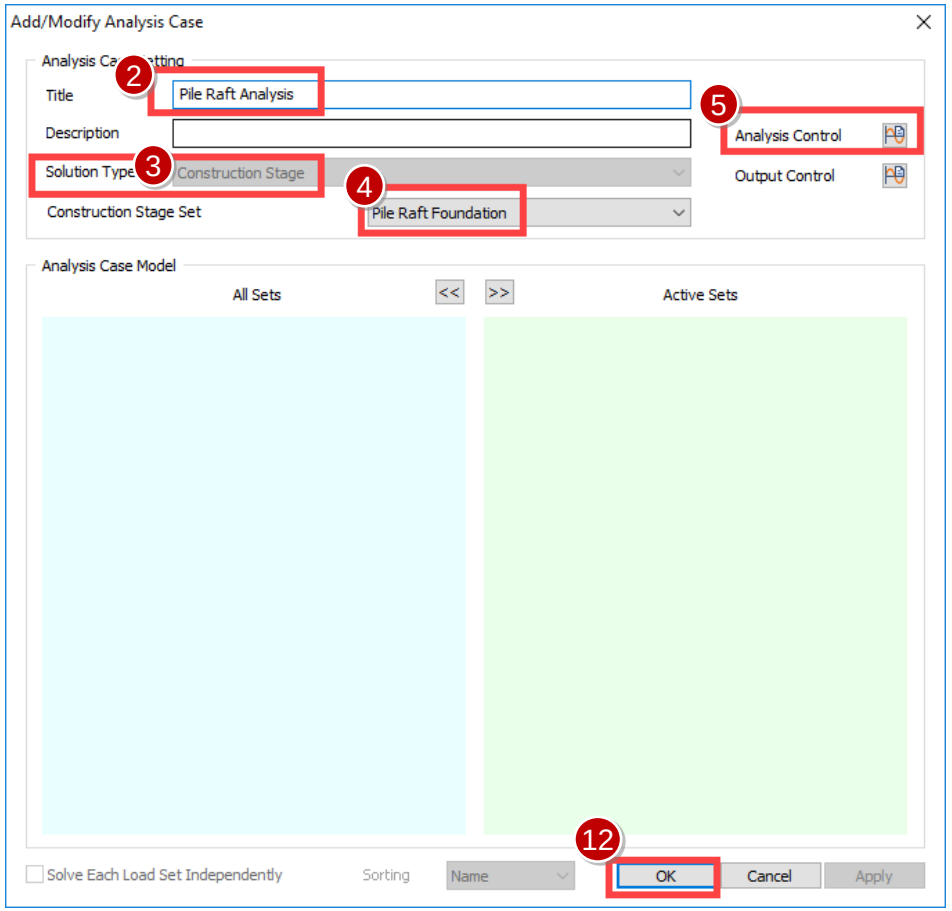


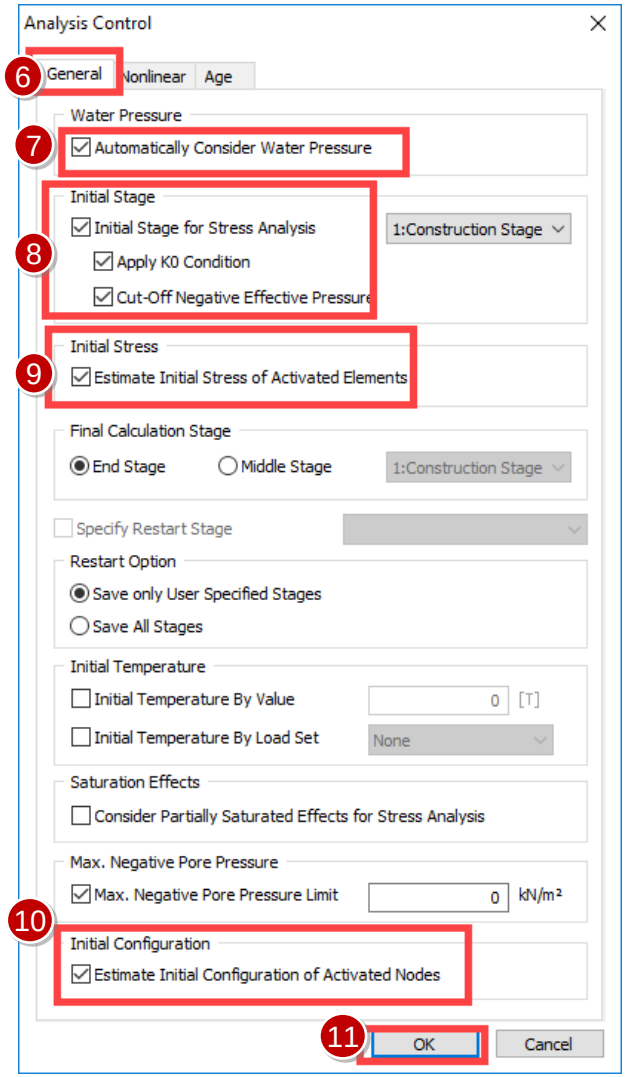
Step 06

6-1 Define Analysis Cases

Procedure

1. Select **Analysis > *Analysis Case* > General**
2. Enter the name as '**Pile Raft Analysis**'
3. Select the solution type as **Construction Stage.**
4. Select '**Pile Raft Foundation**' Construction Stage Set in the drop down menu.
5. Click on **Analysis Control.**
6. Go to **General tab.**
7. Check **Automatically Consider Water Pressure.**
8. Check **Initial Stage for Stress Analysis.** Enable **'Apply K0 Condition'** and **'Cut-off Negative Effective Pressure'.**
9. Enable '**Estimate Initial Stress of Activated Elements**'
10. Enable **Estimate Initial Configuration of Activated Elements.**
11. Click '**OK**'
12. Click '**OK**'

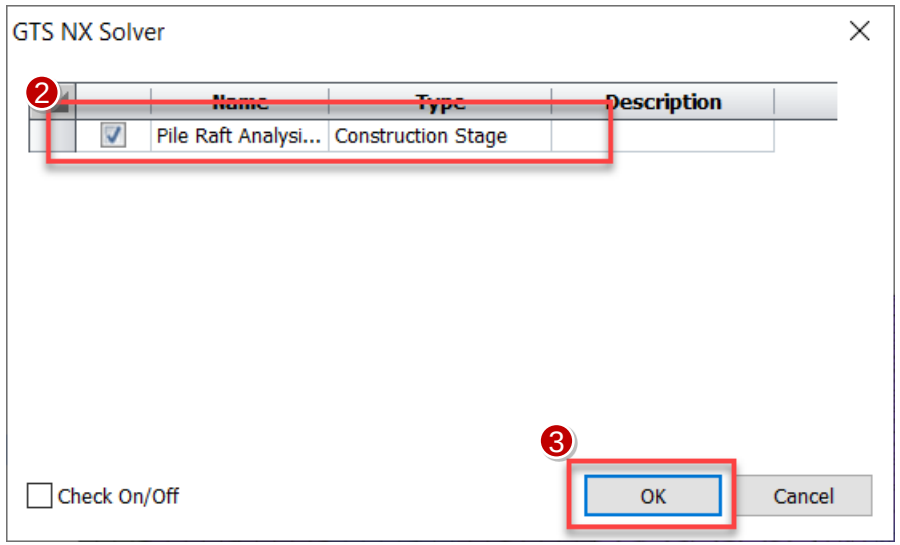




6-2 Analysis

Procedure

1. Select ***Analysis > Perform***
2. Check on **'Pile Raft Analysis'**
3. Click **'OK'**

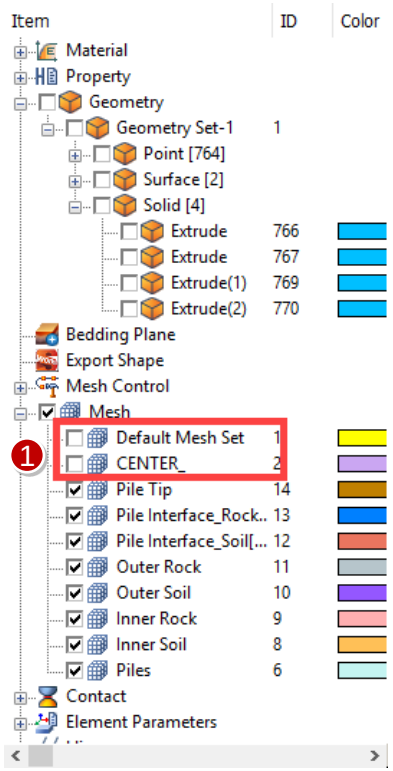


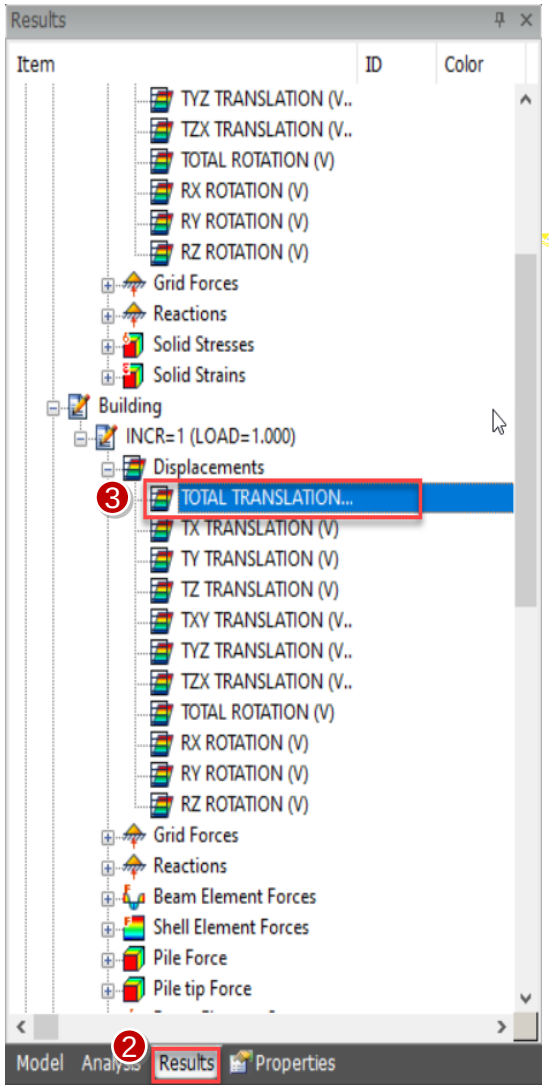
Step
07

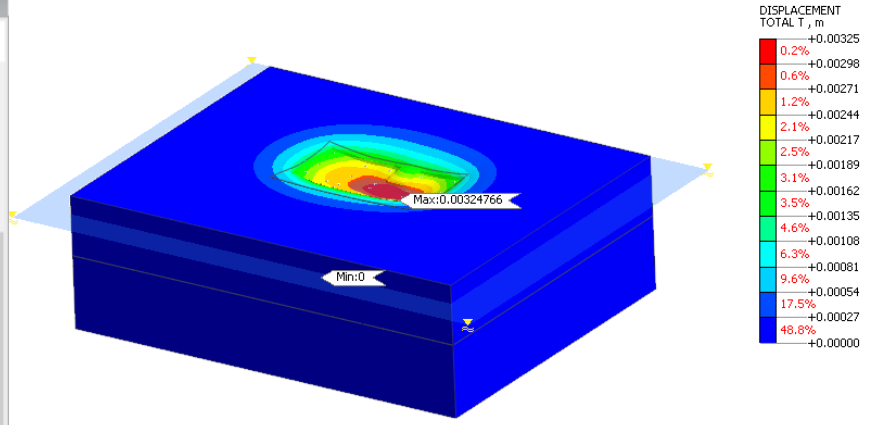
Results – Total Translation (m)

Procedure

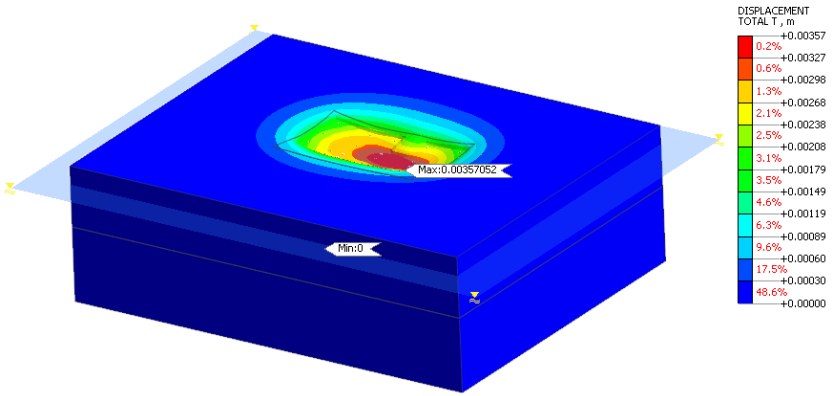
1. Hide the Building Mesh Set in the Model Works tree.
2. Go to **Results Tab,** *expand Building and Load construction stage.*
3. *Under* ***Displacements*** *click on* ***Total Translation (V).***







Building Stage



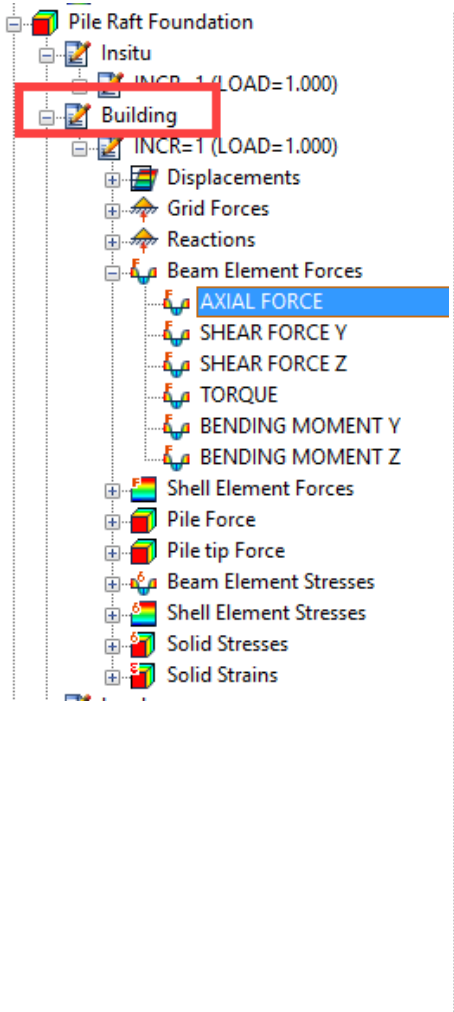
Load Stage

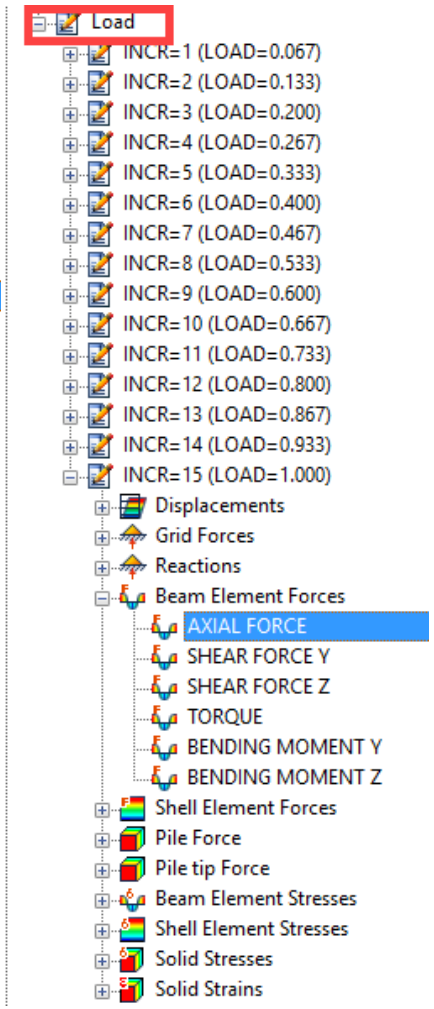
Step
07

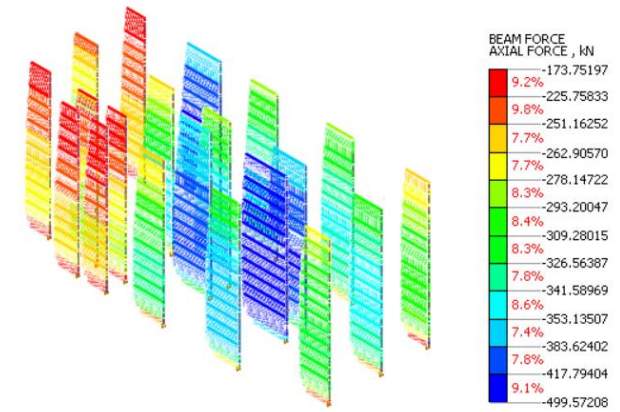
Results – Axial Force in Piles

Procedure

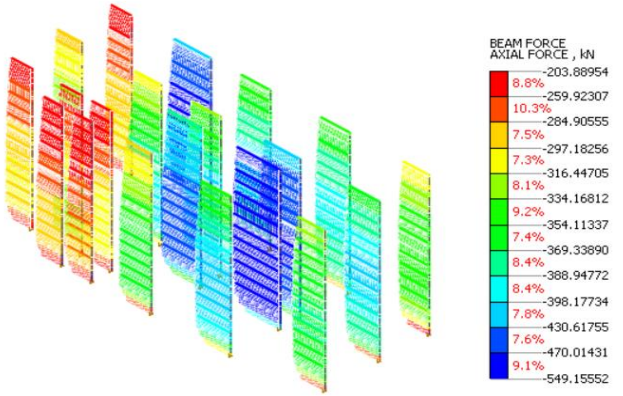
1. Hide Soil and Rock Layers.
2. Go to **Results Tab,** *expand Building and Load construction stage.*
3. *Under* ***Beam Element Forces*** *select* ***Axial Force.***







Building Stage



Load Stage

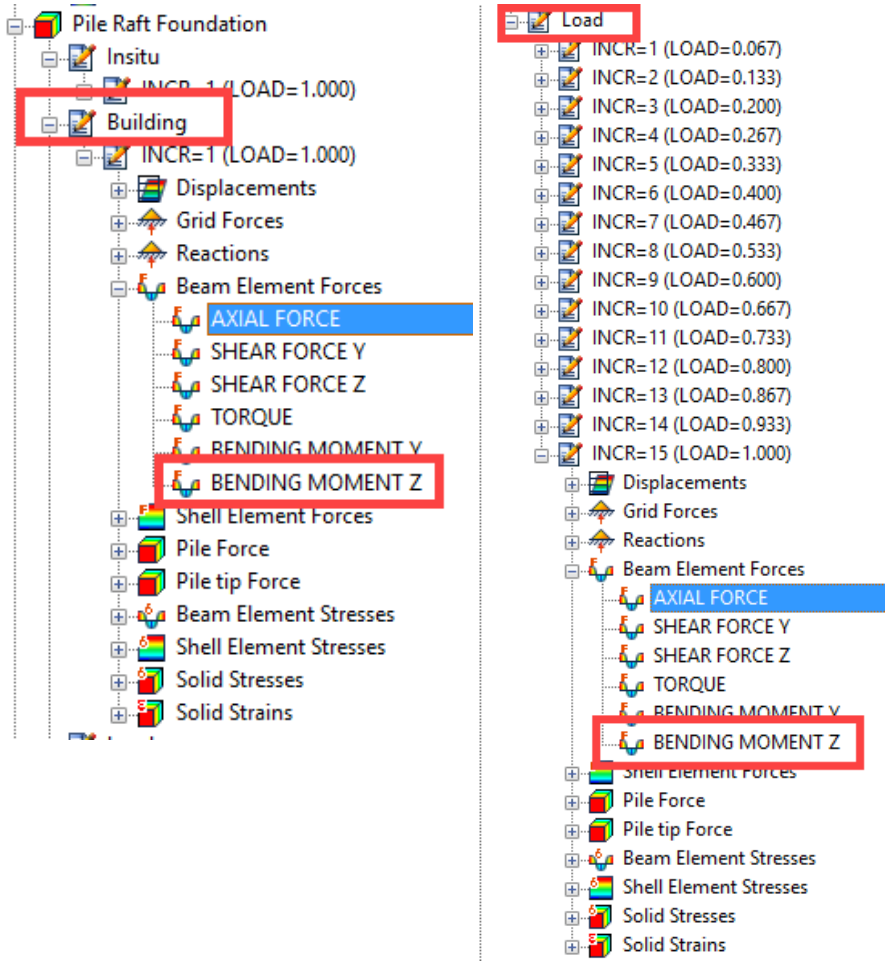
Step
07

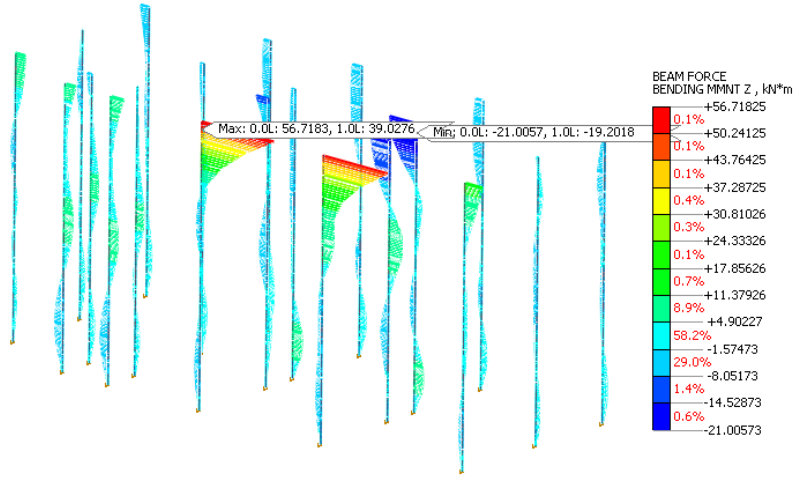
Results – Bending Moment in Piles

Procedure

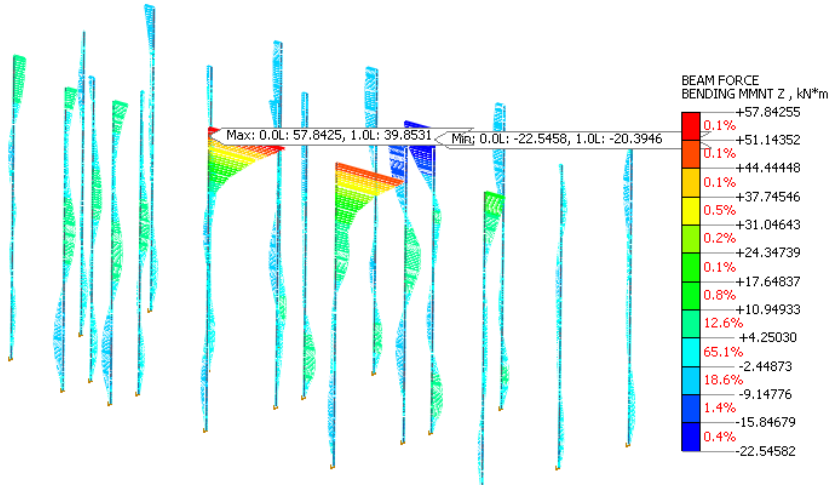
1. Go to **Results Tab,** *expand Building and Load construction stage.*

2. *Under* ***Beam Element Forces*** *select* ***Bending Moment Z.***





Building Stage



Load Stage

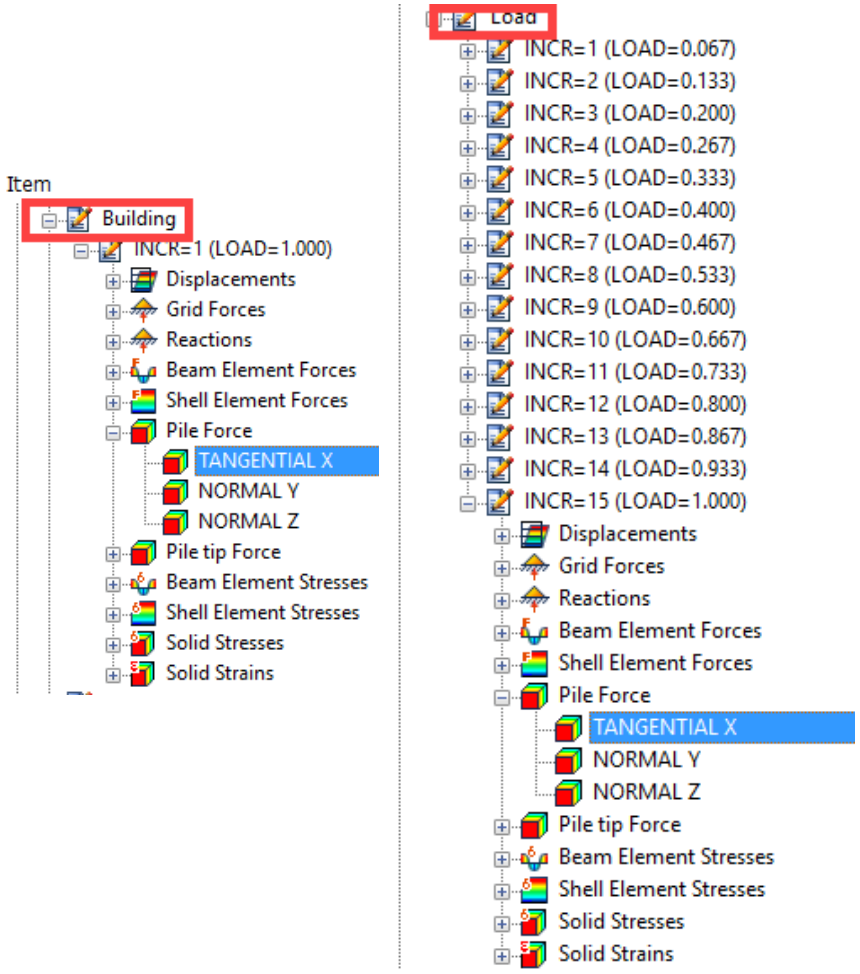
Step
07

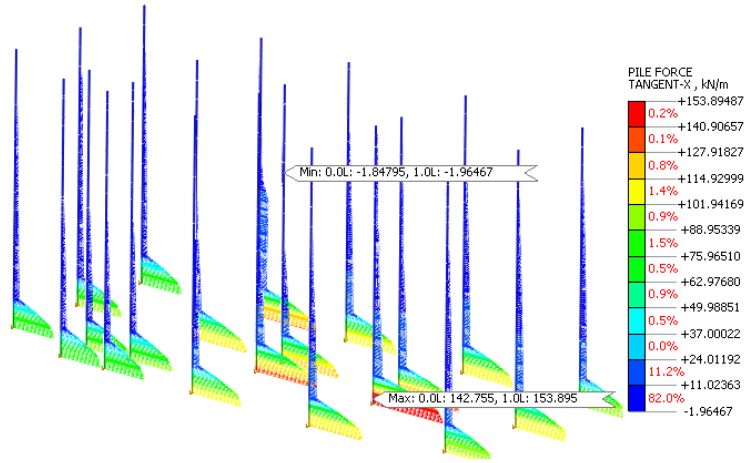
Results – Bending Moment in Piles

Procedure

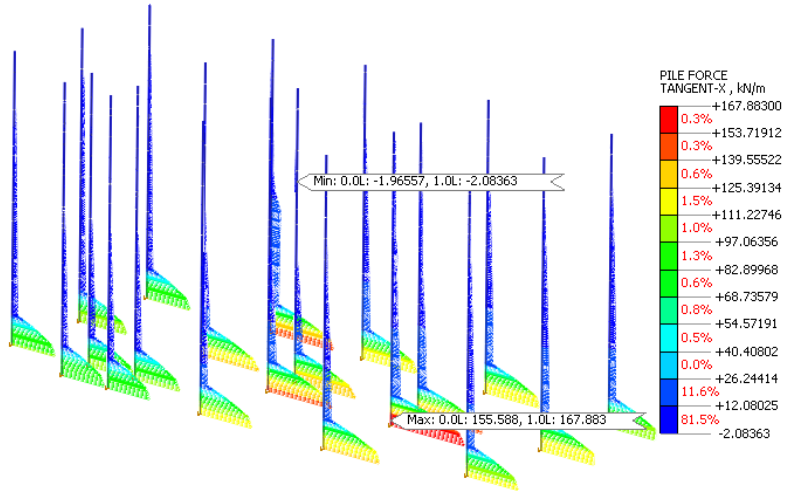
1. Go to **Results Tab,** *expand Building and Load construction stage.*

2. *Under* ***Pile force*** *select* ***Tangential X (Skin Friction)***





Building Stage



Load Stage

Happy Modeling

GTS NX Certification task for participants involves submission of file (.docx or .pdf) with :

- Generating the model as shown in tutorial.
- Settlement vs Time – Graph Picture
- Excess Pore Pressure vs Time – Graph Picture
- Short summary of model creation, and results.

Note: Please enter the name and the country same as you entered in the previous submissions. In case of any name discrepancies among the submissions, the participants will be awarded Zero.

KINDLY SUBMIT YOUR FINAL RESULTS IN THE PROVIDED WORD FILE FORMAT.

The name of the word file should follow "YOUR NAME_COUNTRY.docx/.pdf" format.