



Chapter 2

3D Excavation with Retaining System

Workflow

| | | |
|------------------|---|-----------|
| Section 1 | Overview | 1 |
| 1.1 | Learning Purpose | 1 |
| 1.2 | Modeling and Analysis Summary | 2 |
| Section 2 | Analysis Setting | 3 |
| 2.1 | Analysis Setting | 3 |
| Section 3 | Material and Property | 4 |
| 3.1 | Material Definition for Ground and Structures | 4 |
| 3.2 | Define Properties | 6 |
| Section 4 | Modeling | 7 |
| 4.1 | Geometry Modeling | 7 |
| 4.2 | Generate Mesh | 11 |
| Section 5 | Analysis | 17 |
| 5.1 | Setting Load Condition | 17 |
| 5.2 | Setting Boundary Condition | 18 |
| 5.3 | Define Construction Stage | 19 |
| 5.4 | Setting Analysis Case | 23 |
| Section 6 | Result Verification | 24 |
| 6.1 | Result Verification | 24 |
| 6.2 | Verify Displacement | 24 |
| 6.3 | Verify Stresses | 27 |
| 6.4 | Verify Friction/Relative Displacement of Wall Interface | 29 |

3D Excavation with Retaining System

Section 1

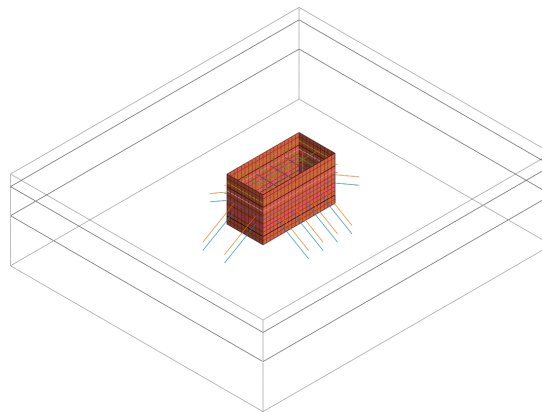
Overview

1.1 Learning Purpose

This tutorial focuses on analyzing the construction stages of 3D retaining wall excavation to identify ground-structure interaction. The objective is to assess retaining wall stability using the finite element method to calculate displacement and stress based on the elastic-plastic characteristics of the ground and the interaction between the retaining wall and structure members. This approach allows verification of stress and displacement not only of the retaining wall but also of its influence on the surrounding ground and adjacent structures simultaneously.

In contrast to 2D analysis, the retaining wall installed in the 3D excavation model is significantly affected by the structure's direction and boundary conditions. Specifically, it enables detailed review of stress distributions on cross-sections, which is not feasible in 2D models. Additionally, an interface is included between the ground and retaining wall to simulate ground-structure interaction more realistically.

► Analysis Model Overview



In this tutorial, we will cover the following main concepts:

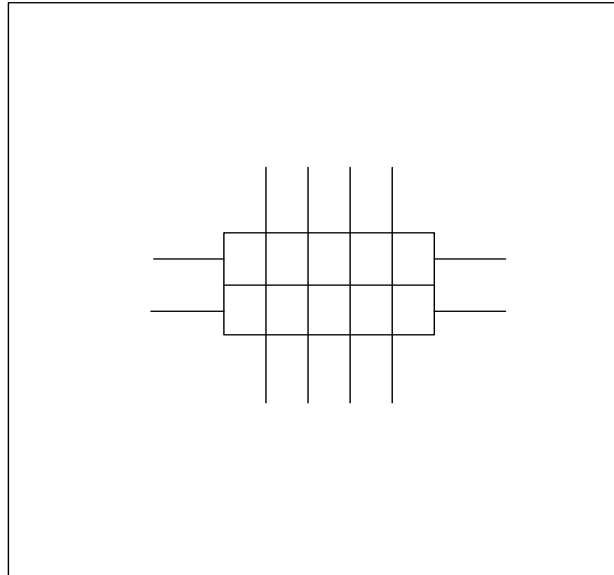
- Applying the Modified Mohr-Coulomb model to simulate ground hardening.
- Modeling sheet piles, beams, trusses, and anchors.
- Implementing interface elements to simulate the separation behavior between the wall and ground.
- Applying anchor pretension.
- Verifying the ground stress, member force, and deformation results.



1.2 Modeling and Analysis Summary

The model represents a 10m x 20m excavation area with temporary facilities including sheet piles, walling, struts, pegs, and anchors. The maximum excavation depth is 10m. The excavation process consists of 5 stages at depths of 3m, 5m, 7m, 9m, and 10m. Total supports (walling and stiffener) are installed at 4 levels with interval distances of 2m (at depths of 2m, 4m, 6m, and 8m). Depending on the ground properties, struts are set at levels 1 and 2, and anchors are set at levels 3 and 4 to reinforce each level. The height of the retaining wall (sheet pile) is 12m, with a distance of 2m from the bottom of the excavation. The ground consists of 3 layers with depths as follows: buried layer (3m), colluviums (10m), and weathering soil (22m) from the surface.

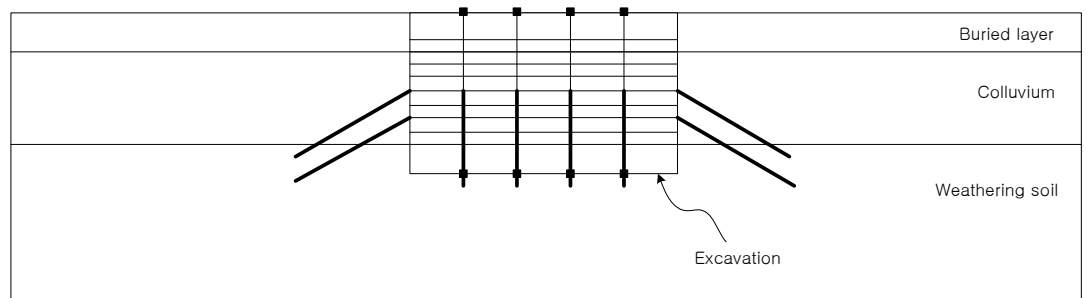
► Plane View



The width of the total ground area should be 1.5 to 3 times wider than the excavation width to minimize the influence of the boundary conditions.

Below is the illustration of the ground composition and excavation stages:

► Cross-Section






Section 2

2.1 Analysis Setting

Analysis Setting

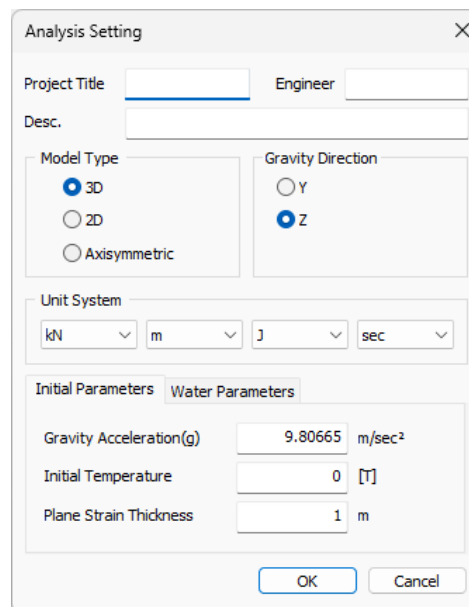
[ : Analysis → Analysis Case → Setting]

[Open the attached start file (02 3D Excavation_Start)]

Set the model type, gravity direction, and initial parameters. Verify the unit system that will be applied to the analysis. The unit system can be adjusted both during the modeling process and after performing the analysis. Input parameters will automatically be converted to the correct unit system.

For this tutorial, a 3D model will be used with the Z gravity direction, and the SI unit system (kN, m) will be applied.

► Analysis Setting Window



The screenshot shows the 'Analysis Setting' dialog box with the following fields and options:

- Project Title**: [Empty text box]
- Engineer**: [Empty text box]
- Desc.**: [Empty text box]
- Model Type**:
 - ☒ 3D
 - ☐ 2D
 - ☐ Axisymmetric
- Gravity Direction**:
 - ☐ Y
 - ☒ Z
- Unit System**:
 - Force: kN (dropdown)
 - Length: m (dropdown)
 - Energy: J (dropdown)
 - Time: sec (dropdown)
- Initial Parameters** (selected tab):
 - Gravity Acceleration(g): 9.80665 m/sec²
 - Initial Temperature: 0 [°C]
 - Plane Strain Thickness: 1 m
- Water Parameters** (unselected tab): [Empty]
- Buttons**: OK, Cancel



Section 3

Material and Property

3.1 Material Definition for Ground and Structures

For the material model type, apply 'Hardening Soil' for the ground and 'Elastic' for the structure. The 'Hardening Soil' model is a material model that follows the 'Power-law'. It can be used to simulate the combined behavior of nonlinear elastic models and elasto-plastic models. Especially by defining elastic modulus during loading and unloading processes, we can minimize the uplift of the excavated surface caused by excavation (unloading process). The 'Elastic' model does not consider material nonlinearity.

It is necessary to consider the nonlinearity of the interface elements which simulate the separation behavior of the ground and retaining wall.

Refer to the table below for the material properties of each ground and structural member. For the interface material, use the parameters calculated automatically by the Wizard.

► Ground Material

| Name | Buried layer | Colluvium | Weathering soil |
|----------------------------------|----------------|----------------|-----------------|
| Material | Isotropic | Isotropic | Isotropic |
| Model Type | Hardening Soil | Hardening Soil | Hardening Soil |
| Poisson's Ratio | 0.333 | 0.306 | 0.384 |
| Unit Weight | 16 | 17 | 20 |
| K0 | 0.5 | 0.441 | 0.74 |
| Unit Weight (Saturated) | 20 | 20 | 22 |
| Initial Void Ratio | 0.5 | 0.5 | 0.5 |
| Drainage Parameters | Drained | Drained | Drained |
| Permeability | 1 | 1 | 1 |
| E50ref | 22,000 | 43,000 | 150,000 |
| Eoedref | 22,000 | 43,000 | 150,000 |
| Eurref | 66,000 | 129,000 | 450,000 |
| Failure Ratio | 0.9 | 0.9 | 0.9 |
| Reference Pressure | 12 | 47 | 110 |
| Power of Stress Level Dependency | 0.5 | 0.5 | 0.5 |
| Friction Angle | 30 | 34 | 38 |
| K0nc | 0.5 | 0.441 | 0.384 |
| Dilatancy Angle | 0 | 4 | 5 |
| Cohesion | 5 | 10 | 15 |

► Structure Material

| Name | Structure material 1 | Structure material 2 |
|-----------------|----------------------|----------------------|
| Material | Isotropic | Isotropic |
| Model Type | Elastic | Elastic |
| Elastic Modulus | 210,000,000 | 200,000,000 |
| Poisson's Ratio | 0.3 | 0.3 |
| Unit Weight | 76.98 | 76.98 |

► TIP
Interface Element

When generating interface elements using the interface wizard, input the parameters as below so that the material properties will be automatically calculated according to the properties of the adjacent ground elements.

The wizard will calculate material properties using the following method:

Apply the 'virtual thickness factor (t_v)' and 'Strength Reduction Factor (R)' by using the stiffness and nonlinear parameters of the adjacent elements. Depending on the stiffness of the surrounding or structural members, the parameters and stiffness of the interface material are applied differently.

$$K_n = E_{oed,i} / t_v$$

$$K_t = G_i / t_v$$

$$C_i = R \times C_{soil}$$

Here,

$$E_{oed,i} = 2 \times G_i \times (1 - \nu_i) / (1 - 2 \times \nu_i)$$

ν_i = interface Poisson's ratio = 0.45 (Interface is for simulating incompressibility friction behavior. To prevent numerical errors, use 0.45 to calculate Interface Poisson's ratio.)

t_v = Virtual thickness factor (Generally use a value in the range of 0.01~0.1. If the stiffness is high, use a smaller value.)

$G_i = R \times G_{soil}$ ($G_{soil} = E / (2(1 + \nu_{soil}))$), R = Strength reduction factor

General strength reduction factors according to structural members and adjacent ground properties are listed as below:

Sandy soil/Steel material = R : 0.6~0.7

Clay/Steel material = R : 0.5

Sandy soil/Concrete = R : 1.0~0.8

Clay/Concrete = R : 1.0~0.7



3.2 Define Properties

Properties represent physical attributes of the meshes and will be assigned to mesh sets during mesh generation. While defining ground and structure properties, firstly choose the material to be used. For structure properties, structure types and cross-section shapes (cross-section stiffness) should be further defined.

Use 'beam element' for sheet piles since they are continuous walls with thickness. 'Beam element' is also used for walling, pegs, and struts since they need to resist axial, shearing, and bending forces.

Use 'embedded truss element' for anchors, which only resist axial force. Struts are usually assumed as 'truss elements' which only resist axial force. However, in the case of a model with pegs, it is reasonable to assume that they also resist shearing and bending forces.

'Embedded truss element' is only for buried structural members. It behaves like a 'truss element' but does not need to be connected to adjacent elements with nodes in 3D analysis, making it easy to apply.

Refer to the table below for the ground properties. For interface material properties, use the parameters calculated automatically through the Wizard.

► Ground Property

| Name | Interface (Buried layer) | Interface (Colluvium) | Interface (Weathering Soil) | Buried layer | Colluvium | Weathering Soil |
|-------------------|-----------------------------|--------------------------|-----------------------------------|--------------|-----------|--------------------|
| Type | Other | Other | Other | 3D | 3D | 3D |
| Model Type | Interface | Interface | Interface | - | - | - |
| Interface Type | Face | Face | Face | - | - | - |
| Material | Buried layer | Colluvium | Weathering Soil | Buried layer | Colluvium | Weathering Soil |

The structure properties are detailed in the following table. The rigidity of the cross-section will be automatically calculated once the cross-section shape is defined.

► Structure Property

| Name | Sheet Pile | Walling, Plus peg, Strut | Anchor |
|--------------|-------------------------|--------------------------|---------------------------------------|
| Type | 2D | 1D | 1D |
| Model Type | Shell | Beam | Embedded Truss (linear elasticity) |
| Material | Structure material 1 | Structure material 1 | Structure material 2 |
| Section | - | H-Section | Solid Round |
| Section Size | Uniform Thickness : 0.1 | 300x300x10/15 | 0.025 |




Section 4

Modeling

4.1 Geometry Modeling

Since the purpose of this tutorial is to study 3D geometry, mesh generation, analysis workflow, and results checking, you can begin by opening the start file where basic materials and properties have already been predefined.

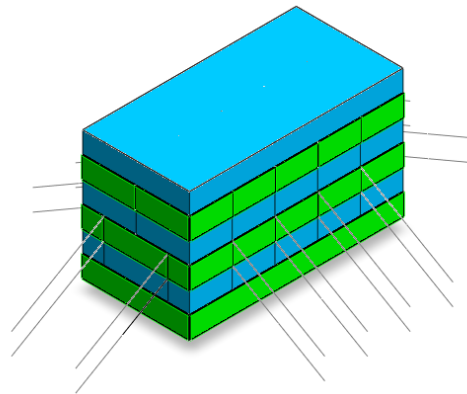
[: Geometry → Protrude → Extrude]

This process involves creating lines, faces, or solids by extruding from geometries of lower dimensions, such as points, edges, or faces. With lines that form a closed domain, it is possible to extrude a solid directly.

To create the 3D ground and excavation area:

- ① Change the selection filter to edge and select the lines (68) of the 'Excavation and wall' as the target object.
- ② Choose the Z axis as the direction and check the [Reverse Direction] option.
- ③ Enter the length as 2 (m), which signifies the height for the struts.
- ④ Click [Apply] and verify the generated solids on the work window.

► Solid for Excavation



► TIP Extrude Function

For beam elements in a 3D model, the ground elements and nodes need to be connected. Beam elements have to be generated using the [Extrude] function after ground meshing. Therefore, spaces are needed in the solid surfaces to extrude beam elements. Additionally, anchor elements must be connected to the nodes of walling/walls. Therefore, the solid surfaces must be divided so that the elements can be generated on these locations. When creating solids by extruding from closed lines, the intersections are locations for the beam elements extrusion later.

► TIP Selection Method

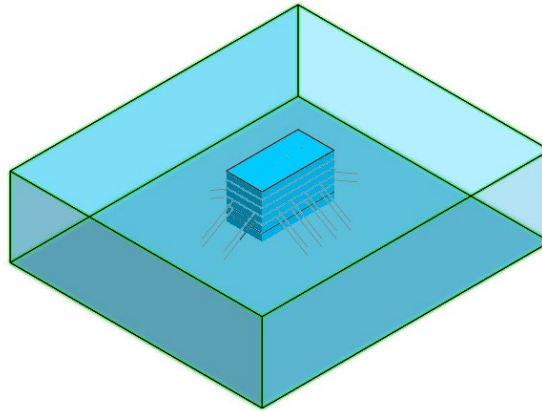
It is possible to select target objects both in the work tree and directly on the screen.




Basic Tutorials

- ① Generate solid of the 'Ground area'.
- ② Select face(1) of the ground area.
- ③ Set the direction to the Y axis. Since the extrude direction is the same as the GCS, uncheck [Reverse Direction].
- ④ Enter 80 (m) for the whole length of the ground area. Click on [OK].

► Overall Solid Geometry

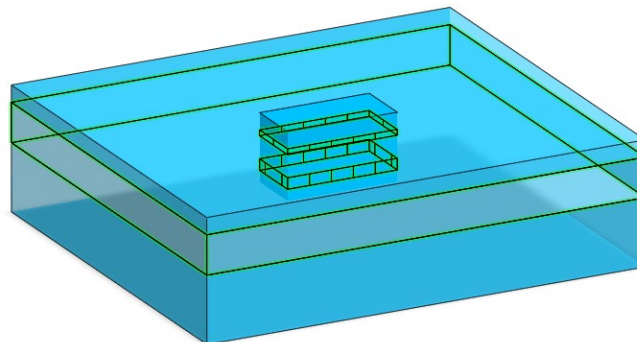


[ : Geometry → Divide → Solid]

This process divides the ground solid and excavation area using bedding planes and cutting surfaces of construction stages.

- ① First, divide the ground and excavation solids by bedding planes.
- ② Select the entire solid (7) as the target object.
- ③ For the tool object, select 'Bedding plane' (2).
- ④ Click on [Apply].

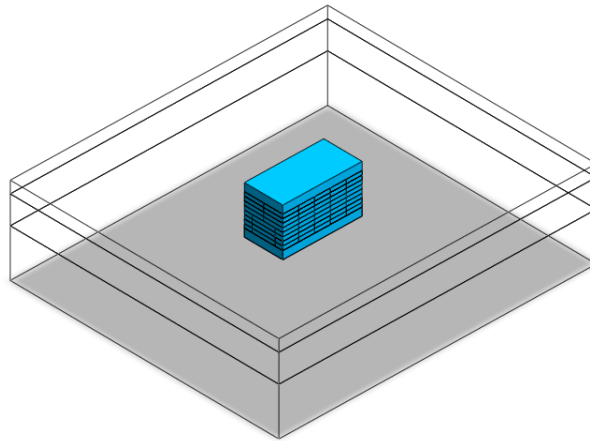
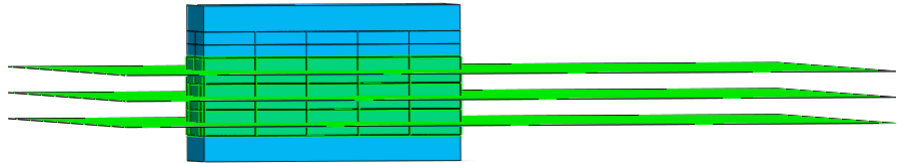
► Divided Bedding Plane





- ① Divide the excavation solid by cutting planes.
- ② Select the solids (3) which will be divided by cutting planes. (Since the other solids cannot be divided even if they are selected in target objects, you can just select all the solids.)
- ③ Select cutting plane (3) as a tool object.
- ④ Click [OK].

► Dividing for
Excavation



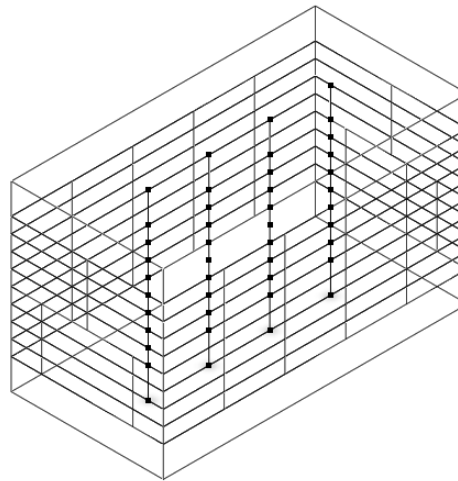



[ : Geometry → Surface & Solid → Imprint]

When you generate beam elements in a 3D model, if the elements are located on the surface of the solid, connecting nodes are automatically generated by the extruding function. However, in the case of plus pegs which pass through the solid, the [Imprint] function needs to be used to recognize the locations of beam elements on the solid surface. Use [Imprint Auto] to generate nodes on all the excavation surfaces through which the plus pegs go.

- ① Select the [Imprint Auto] tab.
- ② Select the entire excavation solids (10) as the target object.
- ③ Select plus pegs (4) as the tool object.
- ④ Click [OK].

► Auto Imprint Function

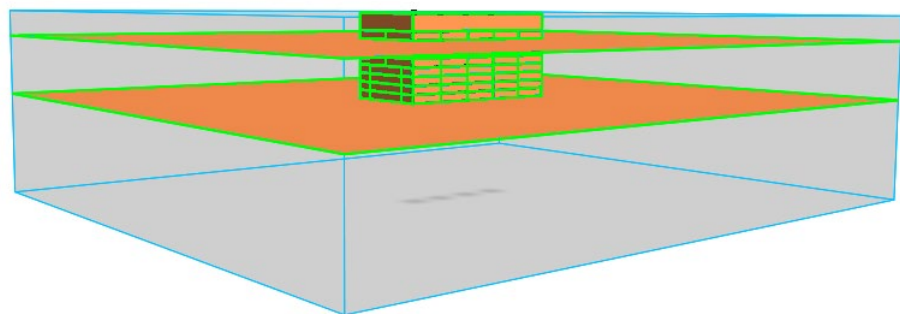


[ : Geometry → Surface & Solid → Auto Connect]

This process automatically generates shared faces after deleting duplicate parts in the entire solid. It is a necessary step before mesh generation, as nodes need to be connected to transfer forces.

- ① Select all the solids (13) and click on [OK].

► Auto Connect
Function






4.2 Generate Mesh

Mesh shape and quality are crucial aspects of finite element analysis. Typically, smaller mesh sizes result in better mesh quality. However, reducing mesh size can also increase analysis time. Therefore, it's advisable to choose a mesh size that balances accuracy and computational efficiency.

When generating the mesh, you have two approaches:

- Assign properties to each solid first and then mesh them individually.
- Mesh the entire model first and then assign properties to each mesh using the [Parameter] function.

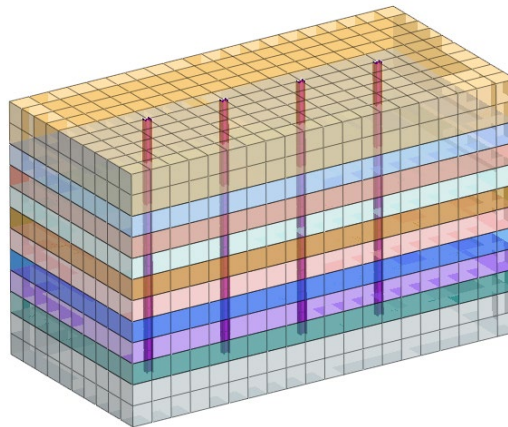
[ : Mesh → Generate → 3D]

This process generates 3D meshes for the ground area. When generating meshes for edges passing through a solid, you can enable the 'Interior Edge' option to generate meshes simultaneously with the solid. This ensures that elements and nodes are connected to the ground.

Here are the steps to generate the excavation solid mesh:

- ① Select the excavation and ground solids.
- ② Go to the 'Auto-Solid' tab and choose the excavation solids.
- ③ Enter '1' for the mesh size and select 'Hybrid Mesher (Hexahedron centered)' from the dropdown menu.
- ④ Set advanced options by clicking the button.
- ⑤ Check the 'Interior Edge/Point' option and select the lines (40) of the imprints as interior edges.
- ⑥ Check the 'Consider Imprinting Shape on Face' option to generate meshes with nodes connected to interior edges.
- ⑦ Select the 'Plus peg' property to assign to the interior edge.
- ⑧ Click [OK] to close the advanced options window and then click [Apply] to generate meshes.

► Generate Mesh for
Excavation and Plus Peg

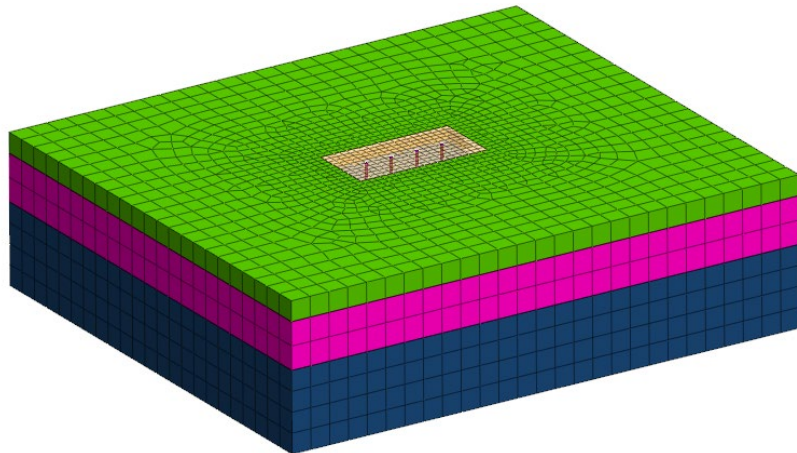





To generate the mesh for the ground area solid, follow these steps:

- ① Select the ground area solid (3) and enter '3' as the mesh size.
- ② Click [OK] to confirm and generate the meshes.

► Generate Mesh for
Ground

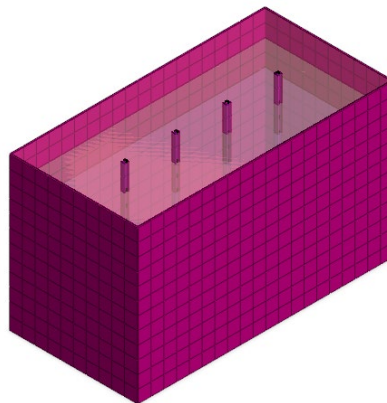


[: Mesh → Element → Extract]

To generate the 'Sheet Pile' element, follow these steps:

- ① Under the Geometry tab, select the 'Face' type.
- ② Select entire faces around the excavation solid in the direction of the X and Y axes.
- ③ Name the mesh set as 'Sheet Pile' and click on [Apply] to confirm.

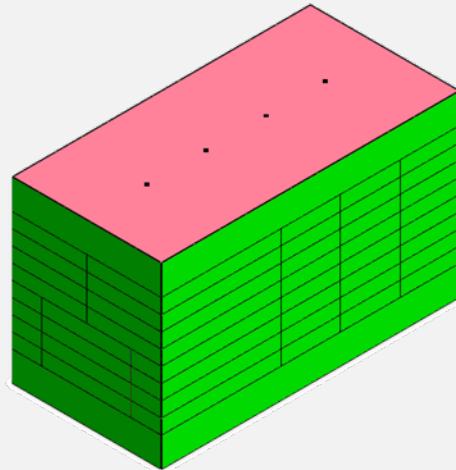
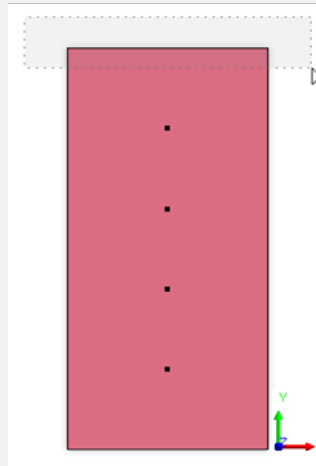
► Extrude Sheet Pile
Mesh





► Tip
Select the Faces

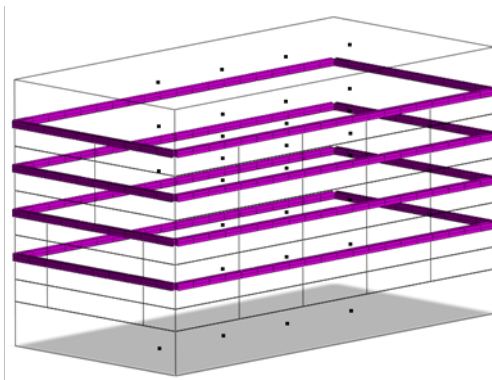
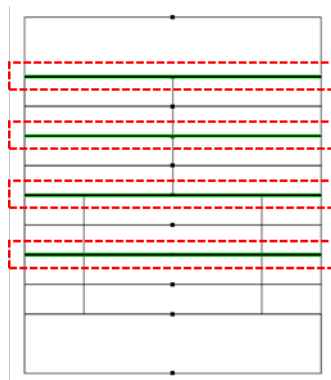
The geometry needs to be displayed on the work window to enable extrusion. You can select a portion of the geometry shape as follows: From the view toolbar, choose the top view. Then, drag and select all the faces around the excavation solid in the X,Y axis direction.




To select the boundary lines of the Walling and extrude elements separately for each boundary line, follow these steps:

- ① Change the selection type to 'Edge'.
- ② Select the boundary lines located at Stage 1 Walling.
- ③ Assign the property to 'Walling' and name it 'Stage 1 Walling'.
- ④ Generate meshes for 'Stage 2~4 Walling' by selecting each walling in order.
- ⑤ Later, assign each walling mesh set to the relevant construction stage referring to the name.

► Extrude Walling Mesh

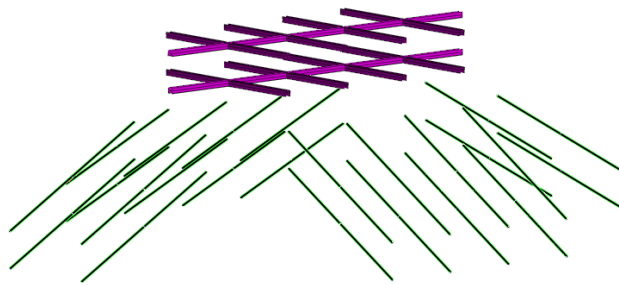




[: Mesh → Generate → 1D]

- ① To generate strut elements and anchors separately for each construction stage, follow these steps:
- ② Select 'Stage 1 strut' (13).
- ③ Enter '1' for division and select the property 'Strut'.
- ④ Name the mesh set as 'Stage 1 strut' and click on [Apply].
- ⑤ Similarly, generate 'Stage 2 strut', 'Stage 3 anchor', and 'Stage 4 anchor' by separating the names and properties accordingly.

► Generate Structure
Mesh

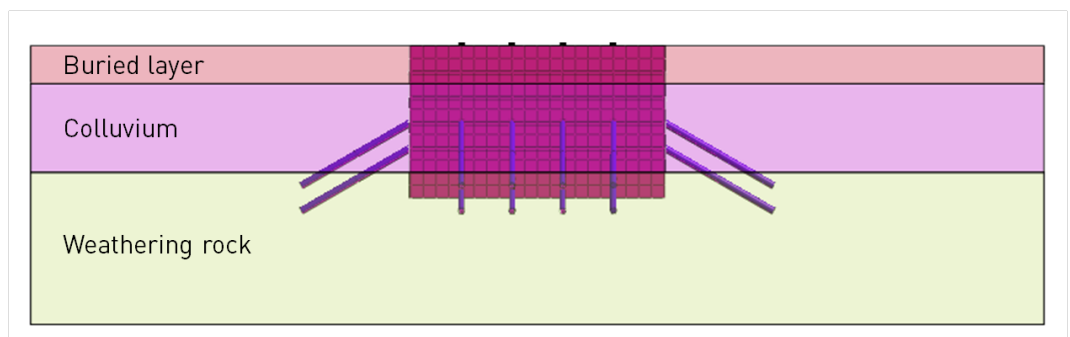


[: Mesh → Element → Parameters]

To check and assign properties to mesh sets, follow these steps:

- ① Select the [3D] tab.
- ② Refer to the image provided, and select each 3D mesh set one by one.
- ③ Assign the appropriate properties to each mesh set.
- ④ Click on the [Apply] button.
- ⑤ Verify the properties of each mesh set by clicking on them in the work tree and checking the property window.

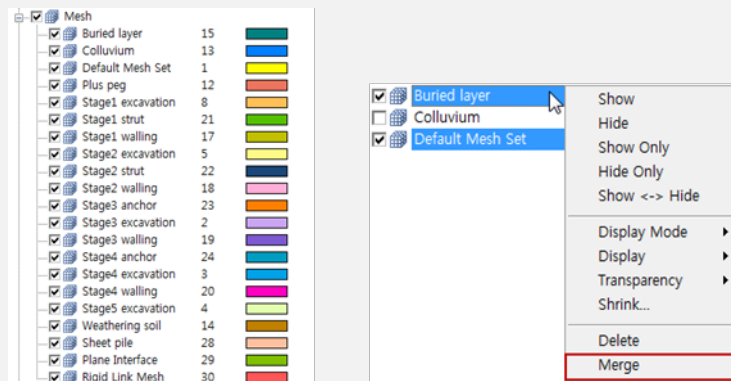
► Stratum Distribution
Outline



► Tip

Change Mesh Set Name
Merge Mesh Set

The mesh sets will be automatically separated by each solid. Select the mesh set in the model tree and adjust the parameters for each mesh set. You can also rename the mesh sets for the construction stages. To change the mesh set name in the work tree, use the [F2] key. If meshes that are activated/deactivated at the same time in a construction stage are divided into several mesh sets, you can merge those mesh sets using the [Merge] function from the context menu by right-clicking the mouse.



[Icon]: Mesh → Element → Interface]

This process generates 3D interface elements to simulate the separation behavior between the ground and the wall. It uses the generated sheet piles (shell elements) to create interface elements at the sides and rear end of the excavation part.

The [Interface] function works as follows: Immediately after the interface elements are generated, connected nodes are automatically detached at the spots of the interface. Then, between the detached nodes, it creates elements with specific rigidity in normal and tangent directions. For stages where the interface elements are not yet activated (e.g., foundation), rigid links must be applied to connect the nodes to prevent errors. However, for stages where the interface elements are activated, rigid links should be excluded. In this tutorial, the material properties of the interface elements are automatically set in the wizard by calculating them from the surrounding material properties.

Here are the steps:

- ① Select the [Plane] tab.
- ② Choose the 'From shell' type.
- ③ Select the 'Sheet Pile' elements (720) and choose 'Both' directions.
- ④ Check the [Merge Nodes] option and select the nodes (60) at the bottom part of the sheet pile as shown in the image.
- ⑤ Choose 'Wizard' and enter parameters (tv:0.1, R:0.65) as shown below.
- ⑥ Check the [Create Rigid Link Element] option.
- ⑦ Press the [Ok] button to generate the interface elements.
- ⑧ The tree of interface material/property for each stratum is generated automatically.



► Interface Wizard

Interface Wizard Data

Structural Parameters

Strength Reduction Factor(R) 0.67

☐ Virtual Thickness Factor(tv) 0.1 m

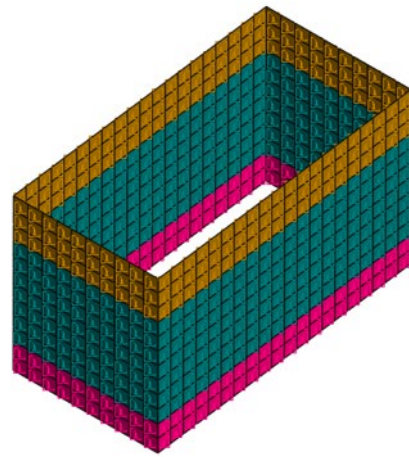
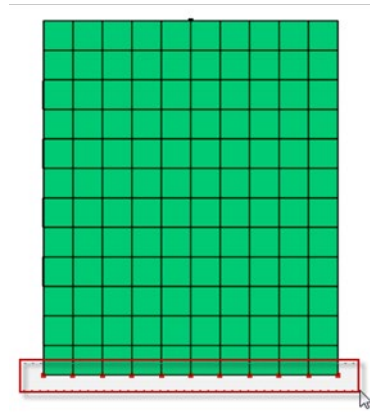
☐ Consider Element Size

Line Interface Thickness 1 m

☐ Conduction for Seepage flow 0 m/sec/m

OK Cancel

► Generate Interface Mesh



► Tip
Interface

If you check the [Consider Element Size] option, the wizard will calculate the interface material properties by considering the average length (for lines) and average area (for faces) of the adjacent ground elements. In other words, it calculates the interface rigidity in the normal and tangent directions by multiplying the average length (l) and average area (A) to the virtual thickness using the following relation:

$$K_n = E_{\text{oad},i} / (l \text{ or } \sqrt{A} \times t_v)$$

$$K_t = G_i / (l \text{ or } \sqrt{A} \times t_v)$$

If you uncheck the option, it applies a unit length (area).

For a line interface, you can define the thickness separately. The thickness is an important factor when using the interface for ground material that exhibits hardening (Modified Mohr-Coulomb). Generally, it is determined by considering the diameter of the adjacent ground. However, if there is no exact value available, you can use the default value set in the program. You don't need to manually input the thickness for the plane interface in a 3D model like in this tutorial.


When defining the seepage rigidity for the interface elements, 'Seepage Flow' can be defined similarly to the ground permeability coefficient. However, if you uncheck [Conduction for Seepage flow], it is assumed to be an impermeable layer.

Section 5

5.1 Setting Load Condition

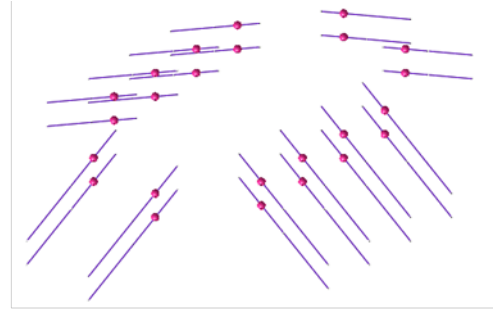
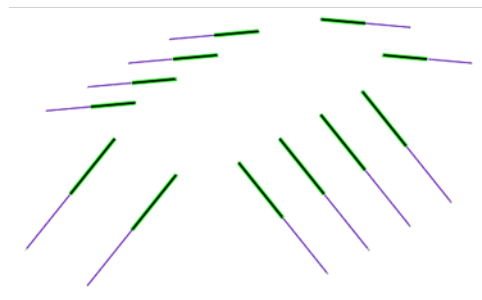
Analysis

This process sets pretension for the anchor-free face. It allows controlling ground displacement by applying pretension (prestress) to Truss/Embedded Truss elements.

[: Static/Slope Analysis → Load → Prestress]

- ① Select the element type 'Truss/Embedded Truss'.
- ② Choose the free face (12) of 'Stage3 anchor' as shown in the image below.
- ③ Enter 200 kN in the load components.
- ④ Change the load set name to 'Stage3 anchor tension'.
- ⑤ Similarly, set 200 kN of prestress for the free face element of 'Stage4 anchor'.

► Setting Prestress for Anchor






5.2 Setting Boundary Condition

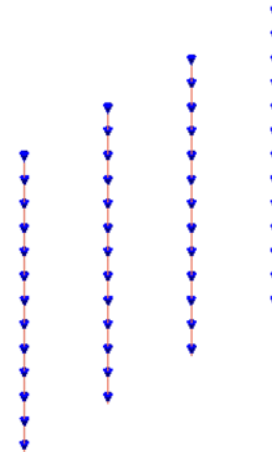
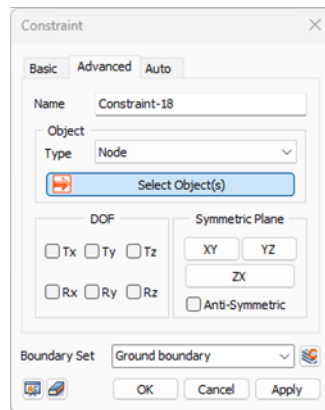
This process sets boundary conditions to prevent internal deformation or rotation based on the Global Coordinate System (GCS).

For the boundaries of the entire model, constraints for left/right/bottom displacements according to GCS are automatically set. Rotation in the Rz direction of plus pegs is constrained to prevent degree of freedom errors.

[: Static/Slope Analysis → Boundary → Constraint]

- ① Select the [Auto] tab.
- ② Check the [Consider All Mesh Sets] option and enter the boundary set name as 'Ground boundary'.
- ③ Click on [Apply].
- ④ Ensure all meshes are shown on the work window, and verify the generated boundaries on the screen.
- ⑤ Select the [Advanced] tab.
- ⑥ Choose the 'Node' type. Select all nodes of the generated plus peg elements and check 'Rz'.
- ⑦ Name the boundary set as 'Constraint rotation' and click on [OK].

► Setting Constraint Pile Rotation



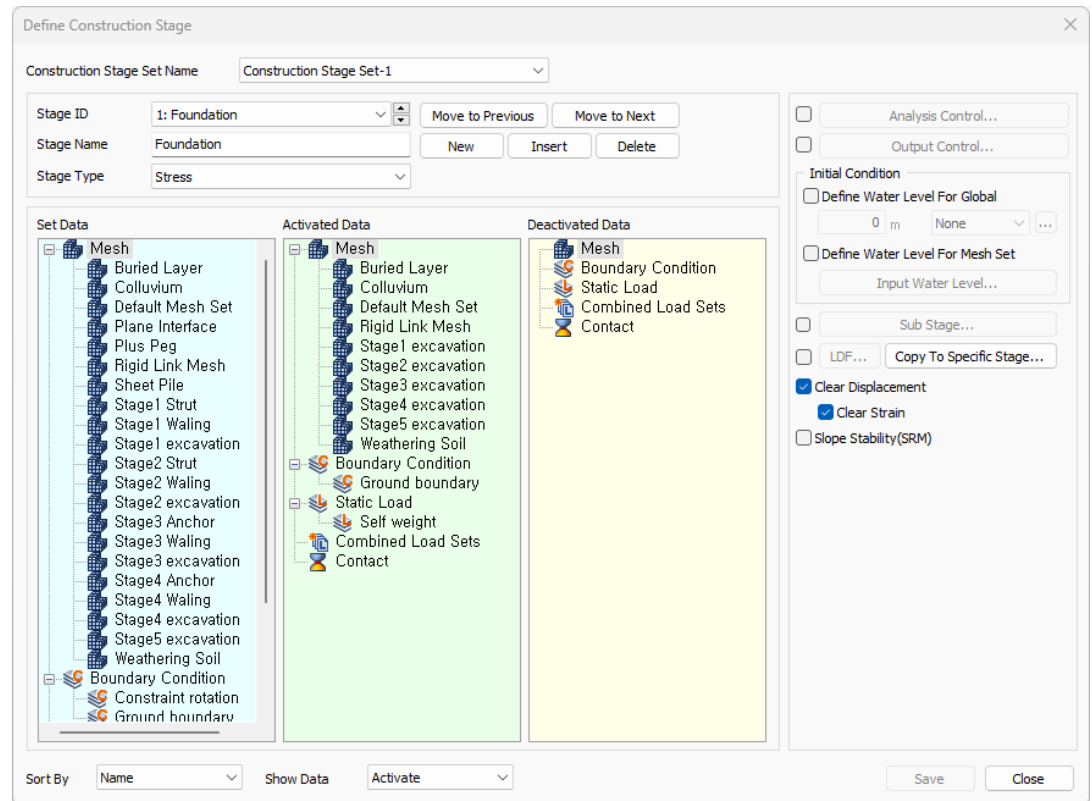
5.3 Define Construction Stage

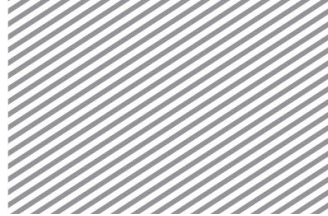
This process defines construction stages to analyze the results at different stages of construction: foundation under initial stress conditions, excavation (or banking), support structure installations, loading, etc. Mesh sets should be separated beforehand according to each construction stage.

[]: Static/Slope Analysis → Construction Stage → Stage Set]

- ① Set the [Stage type] to 'Stress'.
- ② Click on [Add] to create a construction stage set.
- ③ Click on [Define Construction stage] to specify the construction stages.
- ④ The construction stages should be defined as follows:

► Stage 1.
Foundation





Basic Tutorials

► Stage 2. Install Walling and Plus Peg

Define Construction Stage

Construction Stage Set Name: Construction Stage Set-1

Stage ID: 2: Install Walling and Plus peg | Move to Previous | Move to Next

Stage Name: Install Walling and Plus peg | New | Insert | Delete

Stage Type: Stress

| Set Data | Activated Data | Deactivated Data |
|--|---|---|
| <ul style="list-style-type: none">Mesh<ul style="list-style-type: none">Buried LayerColluviumDefault Mesh SetPlane InterfacePlus PegRigid Link MeshSheet PileStage1 StrutStage1 WallingStage1 excavationStage2 StrutStage2 WallingStage2 excavationStage3 AnchorStage3 WallingStage3 excavationStage4 AnchorStage4 WallingStage4 excavationStage5 excavationWeathering SoilBoundary ConditionConstraint rotationGround boundary | <ul style="list-style-type: none">Mesh<ul style="list-style-type: none">Plane InterfacePlus PegSheet PileBoundary ConditionConstraint rotationStatic LoadCombined Load SetsContact | <ul style="list-style-type: none">Mesh<ul style="list-style-type: none">Rigid Link MeshBoundary ConditionStatic LoadCombined Load SetsContact |

Sort By: Name | Show Data | Activate

Save | Close

Analysis Control... | Output Control...

Initial Condition

☐ Define Water Level For Global | 0 m | None | ...

☐ Define Water Level For Mesh Set | Input Water Level...

☐ Sub Stage...

☐ LDF... | Copy To Specific Stage...

☐ Clear Displacement

☒ Clear Strain

☐ Slope Stability(SRM)

► Stage 3. Excavation and install stage1 strut

Define Construction Stage

Construction Stage Set Name: Construction Stage Set-1

Stage ID: 3: Stage1 excavation and install stage1 strut | Move to Previous | Move to Next

Stage Name: Stage1 excavation and install stage1 strut | New | Insert | Delete

Stage Type: Stress

| Set Data | Activated Data | Deactivated Data |
|--|---|---|
| <ul style="list-style-type: none">Mesh<ul style="list-style-type: none">Buried LayerColluviumDefault Mesh SetPlane InterfacePlus PegRigid Link MeshSheet PileStage1 StrutStage1 WallingStage1 excavationStage2 StrutStage2 WallingStage2 excavationStage3 AnchorStage3 WallingStage3 excavationStage4 AnchorStage4 WallingStage4 excavationStage5 excavationWeathering SoilBoundary ConditionConstraint rotationGround boundary | <ul style="list-style-type: none">Mesh<ul style="list-style-type: none">Stage1 StrutStage1 WallingBoundary ConditionStatic LoadCombined Load SetsContact | <ul style="list-style-type: none">Mesh<ul style="list-style-type: none">Stage1 excavationBoundary ConditionStatic LoadCombined Load SetsContact |

Sort By: Name | Show Data | Activate

Save | Close

Analysis Control... | Output Control...

Initial Condition

☐ Define Water Level For Global | 0 m | None | ...

☐ Define Water Level For Mesh Set | Input Water Level...

☐ Sub Stage...

☐ LDF... | Copy To Specific Stage...

☐ Clear Displacement

☒ Clear Strain

☐ Slope Stability(SRM)

► Stage 4.
Excavation and install
stage2 strut

Define Construction Stage

Construction Stage Set Name: Construction Stage Set-1

Stage ID: 4: Stage2 excavation and install sta... Move to Previous Move to Next

Stage Name: Stage2 excavation and install stag2 strut New Insert Delete

Stage Type: Stress

| Set Data | Activated Data | Deactivated Data |
|---|---|---|
| <ul style="list-style-type: none"> Mesh Buried Layer Colluvium Default Mesh Set Plane Interface Plus Peg Rigid Link Mesh Sheet Pile Stage1 Strut Stage1 Waling Stage1 excavation Stage2 Strut Stage2 Waling Stage2 excavation Stage3 Anchor Stage3 Waling Stage3 excavation Stage4 Anchor Stage4 Waling Stage4 excavation Stage5 excavation Weathering Soil Boundary Condition Constraint rotation Ground boundar... | <ul style="list-style-type: none"> Mesh Stage2 Strut Stage2 Waling Boundary Condition Static Load Combined Load Sets Contact | <ul style="list-style-type: none"> Mesh Stage2 excavation Boundary Condition Static Load Combined Load Sets Contact |

Sort By: Name Show Data: Activate

Save Close

Analysis Control... Output Control...

Initial Condition

Define Water Level For Global: 0 m None

Define Water Level For Mesh Set: Input Water Level...

Sub Stage... LDF... Copy To Specific Stage...

Clear Displacement Clear Strain Slope Stability(SRM)

► Stage 5.
Excavation and install
stage3 anchor

Define Construction Stage

Construction Stage Set Name: Construction Stage Set-1

Stage ID: 5: Stage3 excavation and install sta... Move to Previous Move to Next

Stage Name: Stage3 excavation and install stag3 anchor New Insert Delete

Stage Type: Stress

| Set Data | Activated Data | Deactivated Data |
|---|---|---|
| <ul style="list-style-type: none"> Mesh Buried Layer Colluvium Default Mesh Set Plane Interface Plus Peg Rigid Link Mesh Sheet Pile Stage1 Strut Stage1 Waling Stage1 excavation Stage2 Strut Stage2 Waling Stage2 excavation Stage3 Anchor Stage3 Waling Stage3 excavation Stage4 Anchor Stage4 Waling Stage4 excavation Stage5 excavation Weathering Soil Boundary Condition Constraint rotation Ground boundar... | <ul style="list-style-type: none"> Mesh Stage3 Anchor Stage3 Waling Boundary Condition Static Load Stage3 anchor tension Combined Load Sets Contact | <ul style="list-style-type: none"> Mesh Stage3 excavation Boundary Condition Static Load Combined Load Sets Contact |

Sort By: Name Show Data: Activate

Save Close

Analysis Control... Output Control...

Initial Condition

Define Water Level For Global: 0 m None

Define Water Level For Mesh Set: Input Water Level...

Sub Stage... LDF... Copy To Specific Stage...

Clear Displacement Clear Strain Slope Stability(SRM)



Basic Tutorials

- Stage 6.
Excavation and install
stage4 anchor

Define Construction Stage

Construction Stage Set Name: Construction Stage Set-1

Stage ID: 6: Stage4 excavation and install sta... Move to Previous Move to Next

Stage Name: Stage4 excavation and install stag4 anchor New Insert Delete

Stage Type: Stress

| Set Data | Activated Data | Deactivated Data |
|--|---|---|
| <ul style="list-style-type: none">Mesh<ul style="list-style-type: none">Buried LayerColluviumDefault Mesh SetPlane InterfacePlus PegRigid Link MeshSheet PileStage1 StrutStage1 WalingStage1 excavationStage2 StrutStage2 WalingStage2 excavationStage3 AnchorStage3 WalingStage3 excavationStage4 AnchorStage4 WalingStage4 excavationStage5 excavationWeathering SoilBoundary ConditionConstraint rotationGround boundar... | <ul style="list-style-type: none">Mesh<ul style="list-style-type: none">Stage4 AnchorStage4 WalingBoundary ConditionStatic LoadStage4 anchor tensionCombined Load SetsContact | <ul style="list-style-type: none">Mesh<ul style="list-style-type: none">Stage4 excavationBoundary ConditionStatic LoadCombined Load SetsContact |

Sort By: Name Show Data: Activate

Save Close

Analysis Control... Output Control...

Initial Condition

☐ Define Water Level For Global 0 m None ...

☐ Define Water Level For Mesh Set Input Water Level...

☐ Sub Stage...

☐ LDF... Copy To Specific Stage...

☐ Clear Displacement ☒ Clear Strain

☐ Slope Stability(SRM)

- Stage 7.
Final excavation

Define Construction Stage

Construction Stage Set Name: Construction Stage Set-1

Stage ID: 7: Final excavation Move to Previous Move to Next

Stage Name: Final excavation New Insert Delete

Stage Type: Stress

| Set Data | Activated Data | Deactivated Data |
|--|---|---|
| <ul style="list-style-type: none">Mesh<ul style="list-style-type: none">Buried LayerColluviumDefault Mesh SetPlane InterfacePlus PegRigid Link MeshSheet PileStage1 StrutStage1 WalingStage1 excavationStage2 StrutStage2 WalingStage2 excavationStage3 AnchorStage3 WalingStage3 excavationStage4 AnchorStage4 WalingStage4 excavationStage5 excavationWeathering SoilBoundary ConditionConstraint rotationGround boundar... | <ul style="list-style-type: none">Mesh<ul style="list-style-type: none">Boundary ConditionStatic LoadCombined Load SetsContact | <ul style="list-style-type: none">Mesh<ul style="list-style-type: none">Stage5 excavationBoundary ConditionStatic LoadCombined Load SetsContact |

Sort By: Name Show Data: Activate

Save Close

Analysis Control... Output Control...

Initial Condition

☐ Define Water Level For Global 0 m None ...

☐ Define Water Level For Mesh Set Input Water Level...

☐ Sub Stage...


☐ LDF... Copy To Specific Stage...

☐ Clear Displacement ☒ Clear Strain

☐ Slope Stability(SRM)

5.4 Setting Analysis Case

This process configures the analysis method and model data for the analysis. The analysis and output types can be controlled using the advanced options. For construction stage analysis, since the analysis data has already been set up, the [Analysis Case Model] is deactivated.

[: Analysis → Analysis Case → General]

- ① Enter the name of the analysis case and select 'Construction Stage' as the solution type.
- ② Navigate to Analysis > General > Initial Stage, and set Initial Stage for Stress Analysis to '1:Foundation'. Ensure that [Apply K0 Condition] is checked.
- ③ Click on [OK].

5.5 Perform Analysis

Perform the analysis and output the results. Once the analysis is completed, the software will automatically switch to [Post-Mode] for checking the results. You can switch back to [Pre-Mode] for further modifications to the model and options.

[: Analysis → Analysis → Perform]

- ① Click on 'Perform' function
- ② Click on [OK]

► Tip
Real-Time Check

During the analysis, you can monitor the calculation process in real-time. Messages indicating convergence status, warnings, and errors can be viewed through the [Output Window].



Section 6

Results

6.1 Result Verification

After the analysis, you can review the results such as displacements, stresses, and member forces for each construction stage in the Result Tree. Results can be visualized in contour plots, tables, and graphs. The key result items to check in this tutorial are listed below:

- Sheet Pile – Horizontal displacement, bending stress, shear stress
- Vertical displacement at the bottom part of excavation and ground solids
- Walling and Strut – Bending stress, shear stress
- Anchor – Maximum axial force
- Interface – Relative displacement and friction between the wall and ground

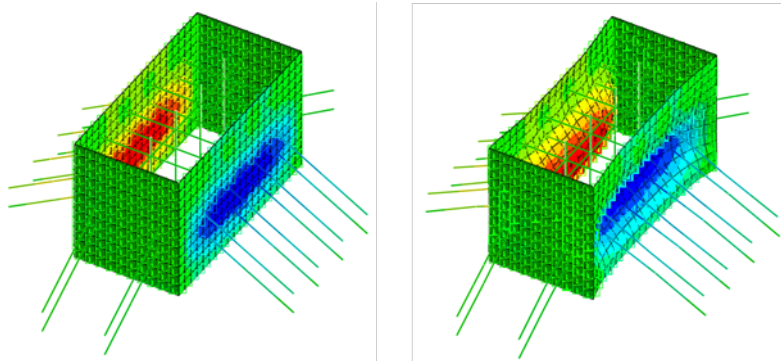
6.2 Verify Displacement

TX, TY, TZ represent displacements in the X, Y, Z directions respectively. Horizontal displacement and settlement tendency according to banking and surface loading can be verified using TX and TZ. '(V)' refers to the result items which can be represented by both contour and vector simultaneously. In GTS NX, it is possible to display contour and vector simultaneously for displacements and principal stresses.

To check the results at the last stage of 'Final excavation':

- Select the last stage in the Result Tree.
- Navigate to Displacement > TX TRANSLATION (V).
- From Result > General > Deform, you can directly visualize the deformation in the X direction. (The scale of the deformation shape can be adjusted in the property window. You can see the difference by checking [Actual Deformation] in the Result > Show/Hide.)

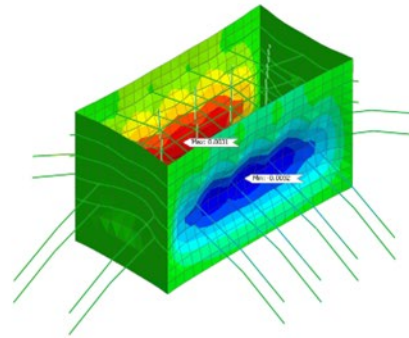
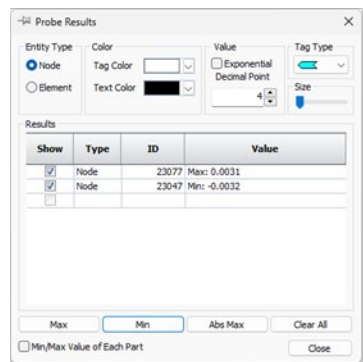
► Horizontal
Displacement
(Undeformed &
Deformed)





It is possible to view the values of specific elements or nodes using Result > Advanced > Probe. Additionally, you can locate the maximum/minimum/absolute maximum values on the model.

► Probe



By moving the simulation bar at the bottom of the work window, you can simulate the changes in results throughout the entire process of construction stages.

► Stage Bar

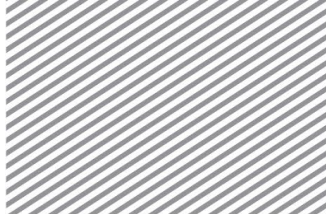


Add/Create the result items that you want to see.

- To create custom result items, navigate to Result > Result > Calculation.
- Choose XY to view deformations in two directions and combine the results to create a new result item.
- Select [Step: final excavation], [Result type: Displacement]. Add data of TX and TY to the list.
- Enter the following formula to create the TXY displacement result item:

$$\text{SQRT}([A][A]+[B][B])$$

- Click on [OK] to add the new result item according to the formula. You can view the results in the form of contour, graph, table, etc.



Basic Tutorials

► Generate XY Direction Displacement

Result Calculation

Target Result Set (File)
☒ Create ☐ Add

Source Result
Analysis Set: 1
Step: Foundation:INCR=1 (LOAD=1.000)
Result: Displacements
Data: TY TRANSLATION (V)

Calculation Information

| Data | Symbol |
|--------------------|--------|
| TX TRANSLATION (V) | [A] |
| TY TRANSLATION (V) | [B] |

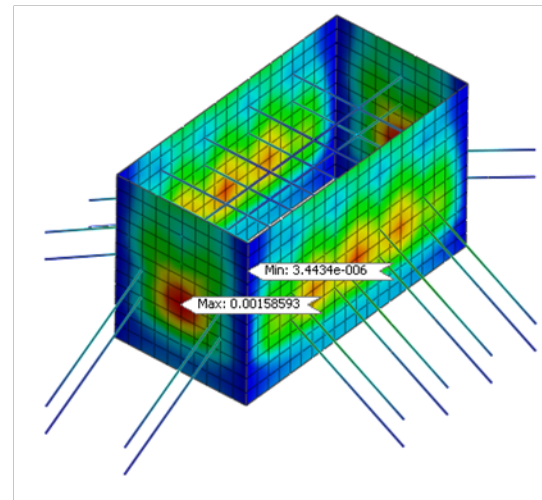
Exp.

(ex) $\text{SQRT}([A]^2 + [B]^2) / [C]$

* Usable operation : +, -, *, /
* Usable function : SQRT, SIN, COS, TAN, ASIN, ACOS, ATAN, SINH, COSH, COTAN, LN, LOG, ABS

Description

☐ Generate All Steps



Utilize the [Extract] function to export maximum displacement results in each construction stage. Right-click the mouse to plot a graph based on the selected data in the table.

► Extract Result, Table and Graph

Extract Results

Output Data
Analysis Set: 1
Result Type: Displacements
Results: TX TRANSLATION (V)

Steps: Results
☒ Foundation:INCR=1 (LOAD=1.000):TX TRANSLATION (V)
☒ Install Walling and Plus peg:INCR=1 (LOAD=1.000):TX TRANSLATION (V)
☒ Stage1 excavation and install stag1 strut:INCR=1 (LOAD=1.000):TX TRANSLATION (V)
☒ Stage2 excavation and install stag2 strut:INCR=1 (LOAD=1.000):TX TRANSLATION (V)
☒ Stage3 excavation and install stag3 anchor:INCR=1 (LOAD=1.000):TX TRANSLATION (V)
☒ Stage4 excavation and install stag4 anchor:INCR=1 (LOAD=1.000):TX TRANSLATION (V)

Order
☒ Step ☐ Node/Element

Object
☒ Node ☐ Element

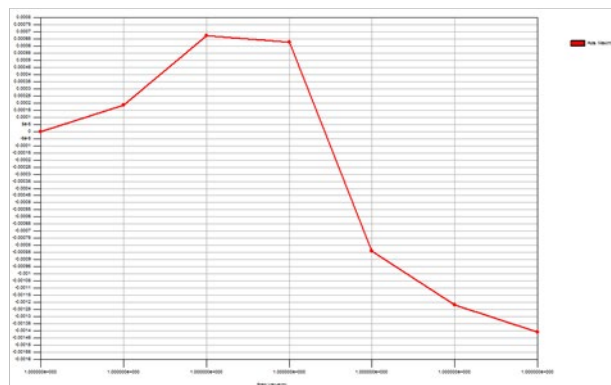
Nodal Results Extraction
☐ User Defined
Select Object
Sort: X Y Z ☐ Ascending

☐ Maximum ☐ Minimum ☒ Abs. Max

☐ Only Show Node/Element

Extraction Position in Element

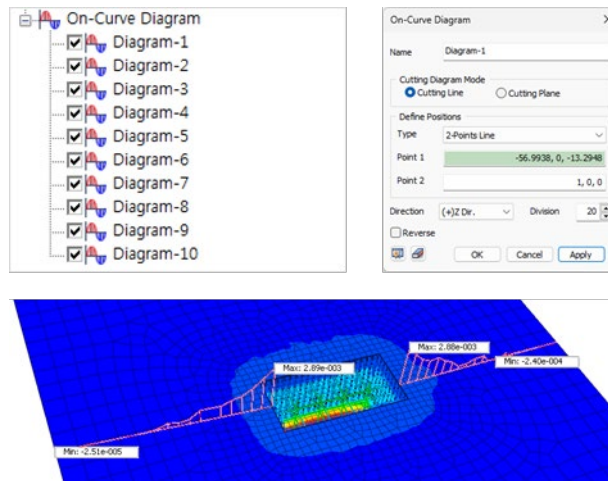
| No | Step | Step Value | Node | Abs. Max (m) |
|----|-------------------------------------|---------------|-------|----------------|
| 1 | Foundation:INCR=1 (LOAD=1.000) | 1.000000e+000 | 22172 | 0.000000e+000 |
| 2 | Install Walling and Plus peg:INCR=1 | 1.000000e+000 | 40245 | 1.837948e-004 |
| 3 | Stage1 excavation and install stag1 | 1.000000e+000 | 46058 | 6.709299e-004 |
| 4 | Stage2 excavation and install stag2 | 1.000000e+000 | 46058 | 6.248550e-004 |
| 5 | Stage3 excavation and install stag3 | 1.000000e+000 | 45229 | -8.391067e-004 |
| 6 | Stage4 excavation and install stag4 | 1.000000e+000 | 45309 | -1.216456e-003 |
| 7 | Final excavation:INCR=4 (LOAD=1) | 1.000000e+000 | 45309 | -1.406522e-003 |





Similarly, verify the settlement of the excavation part by selecting Displacement > TZ TRANSLATION(V). To draw the diagram directly on the model, use Result > Advance > Cutting Diagram. Set the line/point/face to plot the diagram as shown in the image below. Choose the result item, and the result diagram will automatically update. Added diagrams will be registered in the Result Tree. Use the checkbox to Show/Hide each diagram.

► Generate On-Curve Diagram

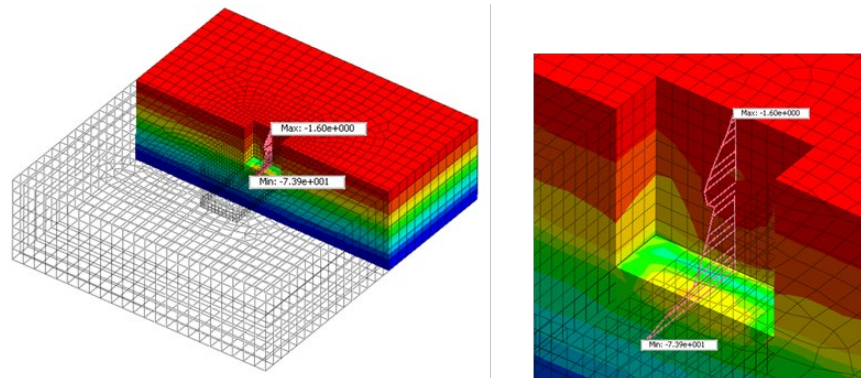


6.3 Verify Stresses

You can view the ground stresses in the 'Solid Stresses' section of the Result Tree. S-XX, S-YY, S-ZZ represent stresses in the X, Y, and Z directions, respectively. Using [On-Curve Diagram], you can plot stress distribution on the cutting line.

- Select Solid Stresses > S-YY of the final excavation stage from the Result Tree.
- You can observe the interior distribution of the ground stresses by using the Advanced View Control Toolbar > Clipping Plane.

► Ground Horizontal Stresses, SYY





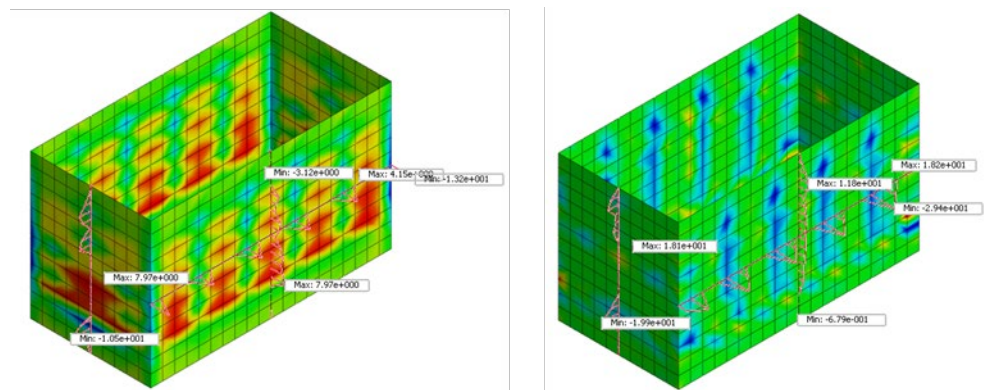
Check member forces/stresses on each structure and sheet pile. Sheet pile can be verified in 'Forces/Stresses' and 'Beam, Truss Element Forces/Stresses' for 1D member. Each result for structure member is plotted based on element coordinate system as by default. If you need to change it, change the coordinate system when you define material/property or in the [Output Control] when you create the analysis case.

Verify the results on the sheet pile of the final excavation stage.

- Verify the moment of the sheet pile by selecting Shell Element Forces > BENDING MOMENT YY in the final excavation stage.
- After that, verify the maximum shear force in TRANSVERSE SHEAR FORCE YZ.
- If you select Result > General > No Results > Exclude, you can hide all the other structures and display only the structure member which you are checking (sheet pile) in the work window.

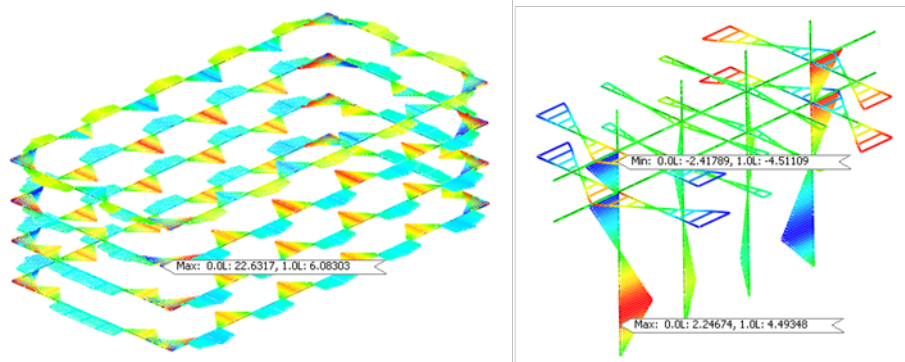
You can see that most maximum member forces concentrate around connections with other structures such as walling, struts, anchors etc.

► Sheet Pile Shear Force and Bending Moment Diagram



Check the maximum moment of the walling, plus pegs, and struts by selecting Beam Element Forces > BENDING MOMENT Y of the final excavation stage in the Result Tree. It is possible to plot the result of each member that you want to see by using the Show/Hide checkbox in the Model Tree.

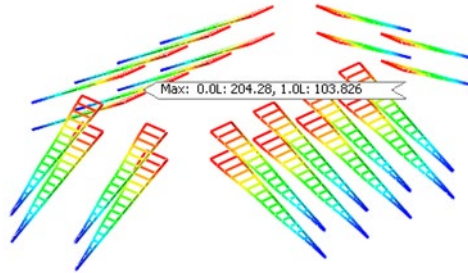
► Walling, Plus Peg and Strut Forces





Verify the axial force on anchors by checking Truss Element Forces > Axial Force of the final excavation stage in the Result Tree.

► Anchor Axial Force

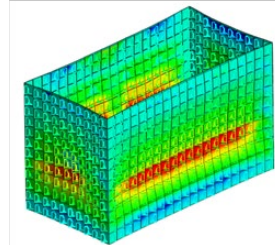
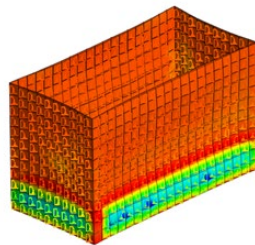


6.4 Verify Friction/Relative Displacement of Wall Interface

Sheet pile and ground have a relatively large difference in rigidity. To simulate the separation behavior of these two, interface elements were applied. You can verify stress and relative displacements in the normal direction and two tangential directions on the wall interface.

- Verify the friction between the wall and ground by selecting Interface Stresses > TANGENTIAL Y of the final excavation stage in the Result Tree. As the excavation progresses, you can observe the generation of significant friction at the bottom of the wall.
- Click on Interface Relative Displacement > PLASTIC TANGENTIAL Y to verify the relative displacement between the ground and wall.
- Compare the displacement of the sheet pile wall with the total displacement at the interface by selecting Displacements > TOTAL TRANSLATION (V) of the final excavation stage.

► Interface Friction and Relative Displacement



► Sheet Pile and Interface Total Displacement

