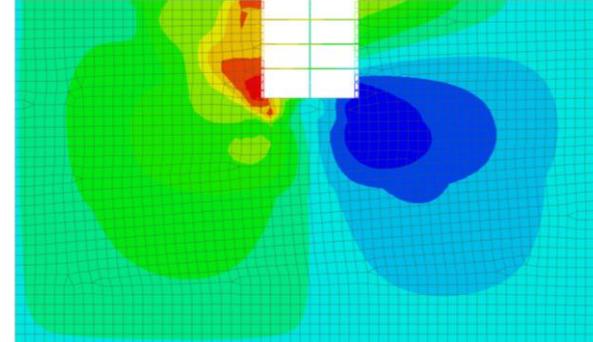


MIDAS *Technical
Material*

Tutorial



Deep Excavation Analysis

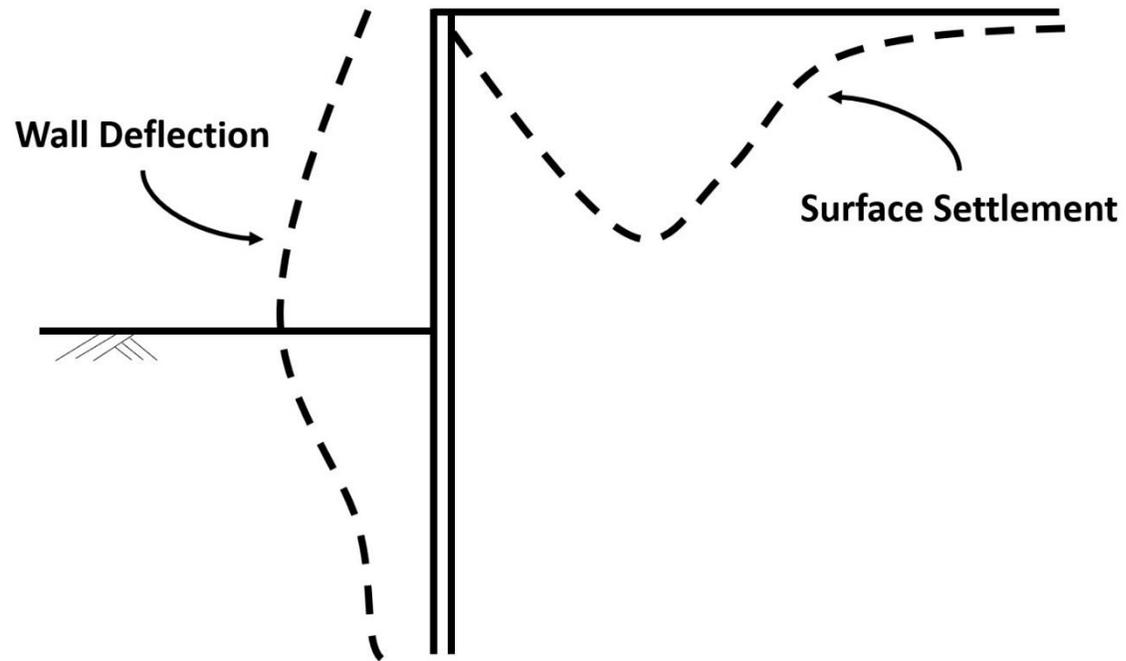


Integrated Solver Optimized for the next generation 64-bit platform
Finite Element Solutions for Geotechnical Engineering



Deep Excavation

Behaviors of Soil When It's under excavation construction



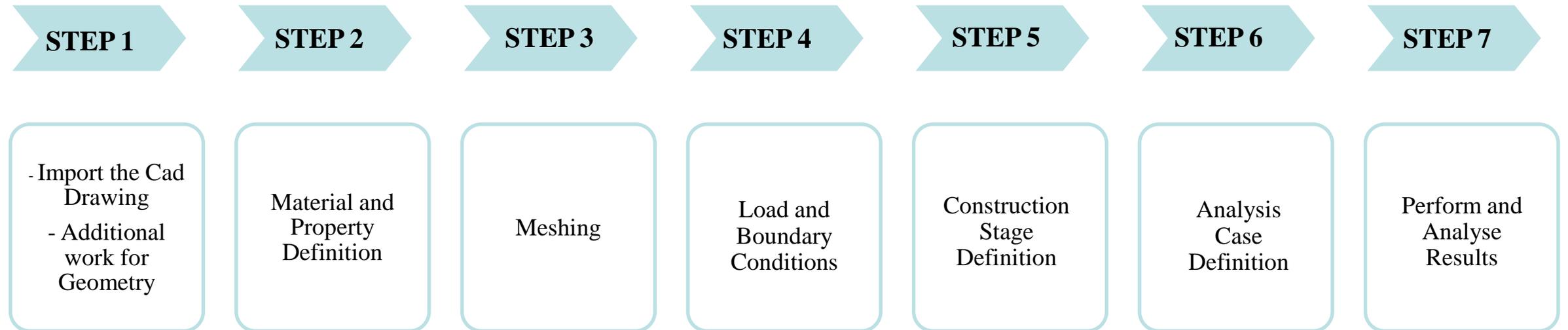
Expected Results

Learning Objective

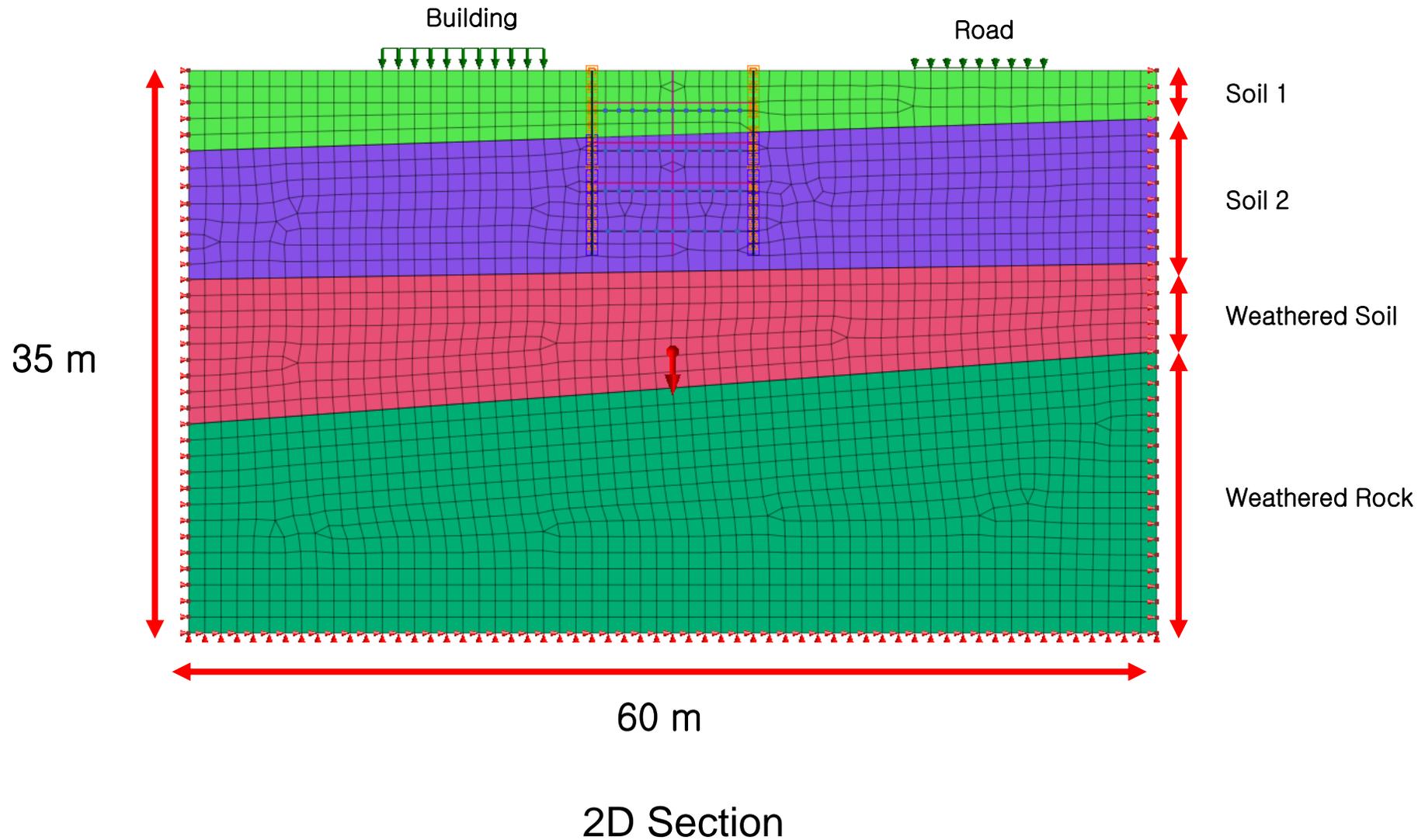
Overview

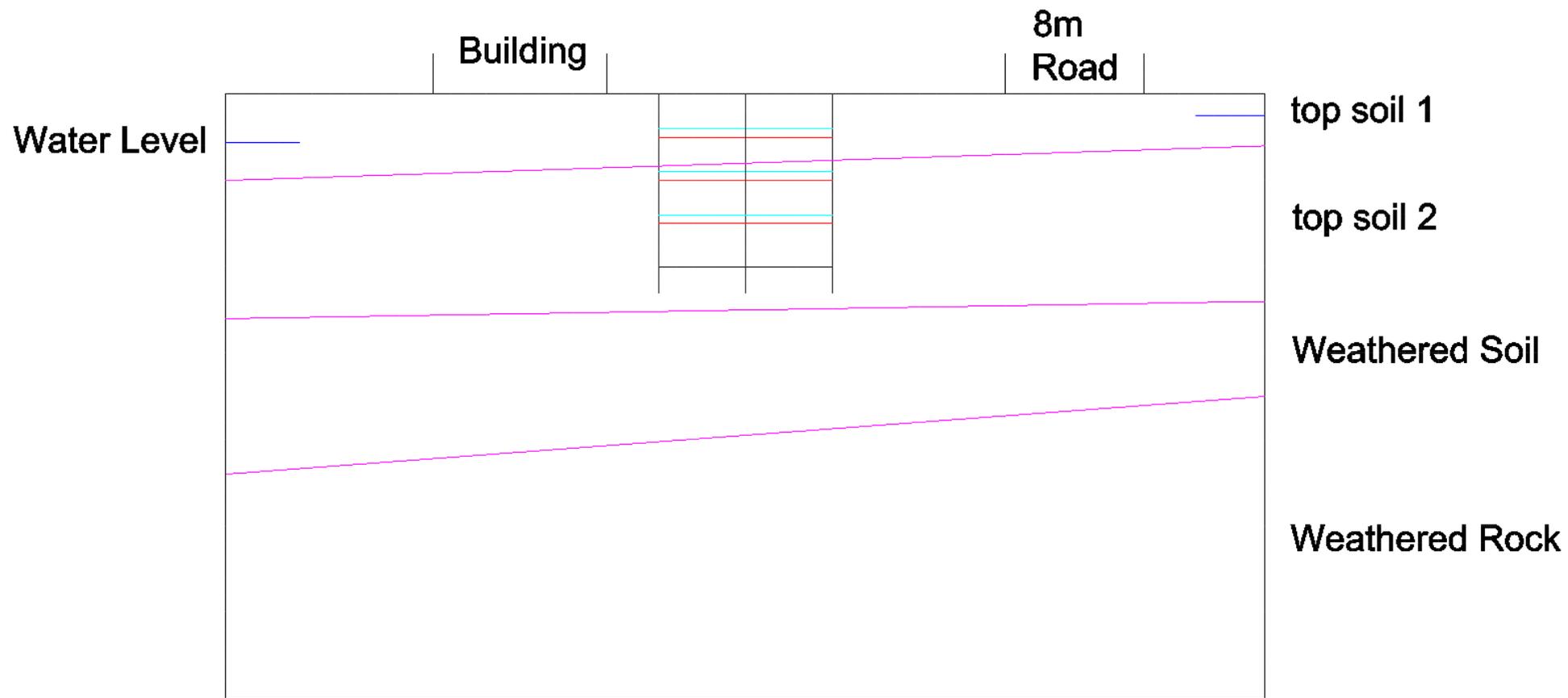
- This tutorial will explain the steps to be followed to perform Seepage-Stress Coupled Analysis for Deep Excavation.
- Partial Factor, Seepage-Stress Coupled analysis and Generate Report functions will be dealt.

Workflow



Building Model and Subsurface Layers





2D Section

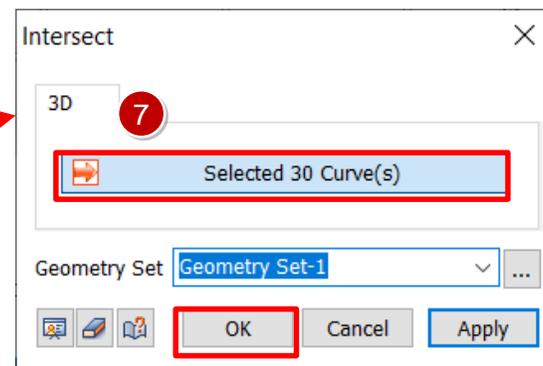
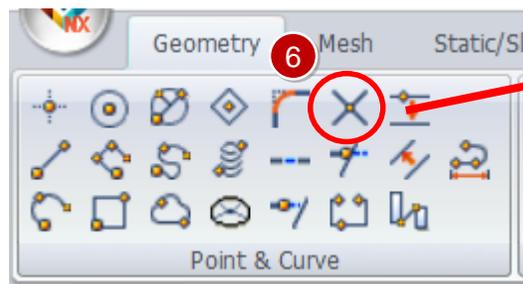
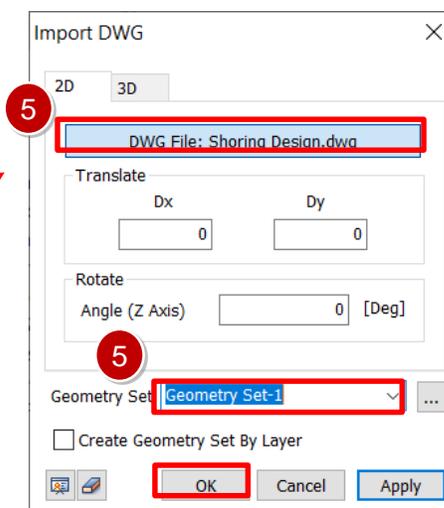
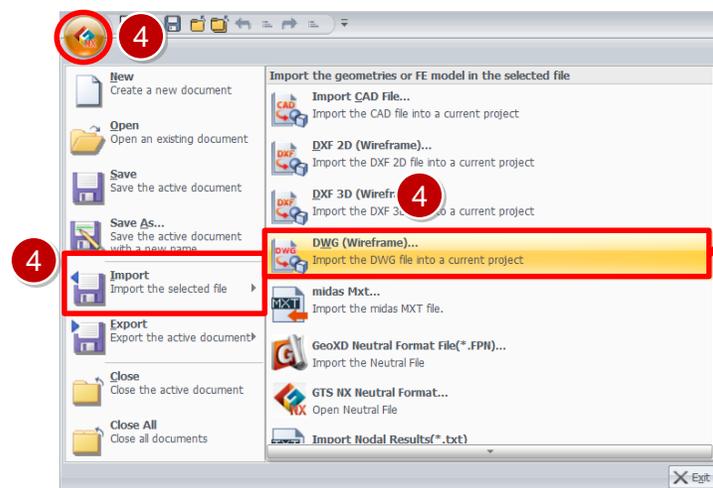
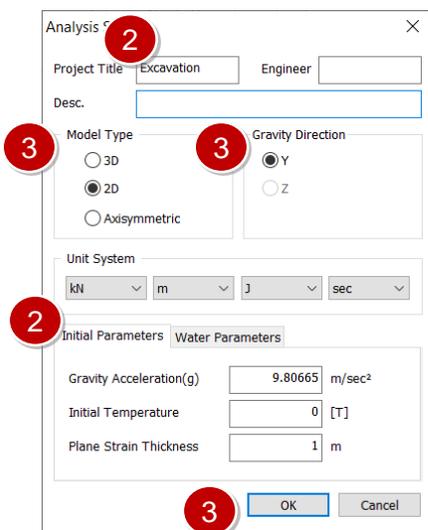
Material Properties

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)	Permeability (m/sec)
Soil 1	Isotropic Mohr-Coulomb	17,000	0.35	17	18	0.5	7	25	9.78e-6
Soil 2	Isotropic Mohr-Coulomb	20,000	0.35	19	20	0.5	15	30	9.78e-6
Weathered Soil	Isotropic Mohr-Coulomb	30,000	0.35	17	18	0.5	46.7	25	9.58e-7
Weathered Rock	Isotropic Mohr-Coulomb	65,000	0.3	20	21	0.5	100	30	4.3e-7
Steel	Isotropic Mohr-Coulomb	205,939,600	0.16	78.5	-	-	-	-	-
Con'c	Isotropic Mohr-Coulomb	24,645,000	0.18	23.53965	-	-	-	-	-

1-1 Import CAD drawing and Divide the Edges

Procedure

1. Open GTS NX.
2. Project Title: enter **"Excavation"**, set units as **kN, m and sec**
3. Select Model type **2D** and Gravity **direction Y** and click **'OK'**
4. Click on GTS NX icon on the top left corner of Interface and select **Import > DWG**.
5. Select **"Select AutoCAD DWG File"** and select **"Shoring Design.dwg"**. And click **'OK'**.
6. Go to **Geometry > Point & Curve > Intersect**.
7. **'Ctrl+A'** on the keyboard to select all the edges on screen and click **'OK'**.



1-2 Make Face & Check Materials and Properties

Procedure

1. Go to **Geometry > Surface & Solid > Make Face**
2. Select outlines of the model and Click 'OK'
3. Go to **Mesh > Prop./CSys./Func. > Materials / Properties**. Every materials and properties has been input already. Just check the parameters inside page number 5.

***If you want to know defining the material and property, Please refer to the 8th and 9th page.*

1 Surface & Solid

2 Make Face

Face Coons Face Grid Face Point

Selected 19 Edge(s)

Tolerance 0.0001 m

Geometry Set-1

OK Cancel Apply

3 Material/Modify Material

No	Name	Type
1	Soil1	Isotropic-Mohr-Coulomb
2	Soil2	Isotropic-Mohr-Coulomb
3	Weathered Soil	Isotropic-Mohr-Coulomb
4	Weathered Rock	Isotropic-Mohr-Coulomb
5	Steel	Isotropic-Elastic

3 Property/Modify Property

No	Name	Type	Sub-Type
1	Soil1	2D	Plane Strain
2	Soil2	2D	Plane Strain
3	Weathered Soil	2D	Plane Strain
4	Weathered Rock	2D	Plane Strain
5	R.W	1D	Beam
6	Strut	1D	Beam
7	King Post	1D	Beam

Procedure

1. Go to **Mesh > Prop/Csys/Func > Material**
2. Click on **Create > Isotropic**
3. Select Model type > **Mohr-Coulomb**
4. Name as **'Soil1'**
5. Under the **General tab**
Elastic Modulus > **17000 kN/m²**
Poisson's Ratio > **0.35**
Unit Weight > **17 kN/ m³**
6. In **Porous tab**
Saturated unit weight > **18 kN/ m³**
Initial Void Ratio > **0.5**
Permeability > **9.78e-6 m/sec**
7. In **Non-Linear tab**
cohesion > **7 kN/ m³**
Friction angle > **25 degrees**
8. Click **'OK'**

Repeatedly, Define all the Materials following the 5th page.

The screenshot illustrates the material definition process in a software application. It is divided into several panels and tabs, with red boxes and numbers highlighting key steps:

- 1:** The **Material** icon in the software toolbar is highlighted.
- 2:** The **Create...** button in the **Add/Modify Material** dialog is highlighted.
- 3:** The **ID** field in the **Material** dialog is highlighted with the value **Soil1**.
- 4:** The **Model Type** dropdown menu is highlighted, showing **Mohr-Coulomb** selected.
- 5:** The **General** tab in the **Material** dialog is highlighted, showing the following values:
 - Elastic Modulus(E): 17000 kN/m²
 - Inc. of Elastic Modulus: 0 kN/m³
 - Inc. of Elastic Modulus Ref. Height: 0 m
 - Poisson's Ratio(ν): 0.35
 - Unit Weight(γ): 17 kN/m³
- 6:** The **Porous** tab in the **Material** dialog is highlighted, showing the following values:
 - Unit Weight(Saturated): 18 kN/m³
 - Initial Void Ratio(e_0): 0.5
 - Permeability Coefficients (kx, ky, kz): 9.78e-006 m/sec
- 7:** The **Non-Linear** tab in the **Material** dialog is highlighted, showing the following values:
 - Cohesion(C): 7 kN/m²
 - Inc. of Cohesion: 0 kN/m³
 - Inc. of Cohesion Ref. Height: 0 m
 - Frictional Angle(Φ): 25 [deg]
- 8:** The **OK** button at the bottom of the **Material** dialog is highlighted.

Procedure

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

1. Click on **Create > 2D**
2. Select Model type > **Plane Strain**
3. Name as **'Soil1'**
4. Click **'OK'**

Repeatedly, Define all the Soil Properties following the 5th page.

For the structural element property,

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

5. Click on **Create > 1D**, Select **Type > Beam**, Name as **R.W** and Assign **'Steel'** Property.
6. Tick **'Spacing'**, Tick **'Section'** and select the section **'UB 305x165x46'** for Retaining Wall.

Strut(Truss) : UB 305x165x46
King Post(Beam) : J 254x203x82

The screenshot illustrates the software interface for defining properties. It shows three overlapping dialog boxes:

- 1. Add/Modify Property:** A table with columns 'No', 'Name', 'Type', and 'Sub-Type'. It lists existing properties like 'Soil1', 'Soil2', 'Weathered Soil', etc.
- 2. Create/Modify 2D Property:** A dialog for defining a 2D property. It has tabs for 'Shell', 'Plane Stress', 'Plane Strain', and 'Geogrid(2D)'. The 'Plane Strain' tab is active. Fields include ID (1), Name (Soil1), Color (purple), and Material (1: Soil1). The 'OK' button is highlighted.
- 3. Create/Modify 1D Property:** A dialog for defining a 1D property. It has tabs for 'Beam', 'Taper', and 'Hinge Property'. The 'Beam' tab is active. Fields include ID (5), Name (R.W), Color (green), and Material (5: Steel). The 'Spacing' and 'Section' checkboxes are checked. The 'Section' dropdown is set to 'H-Section'. The 'OK' button is highlighted.
- 4. Section Template:** A dialog for selecting a section template. It shows a diagram of an I-beam with dimensions B1, B2, H, r1, r2, tf1, tf2, and tw. The 'Standard' is set to 'BS-EN' and the 'Section' is 'UB 305x165x46'. The 'OK' button is highlighted.

1-3 Imprint edges into the face & 2-1 Generate Mesh Set – Soil Elements

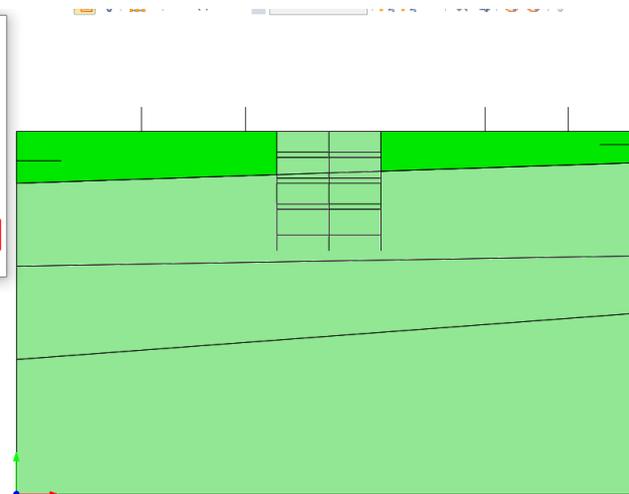
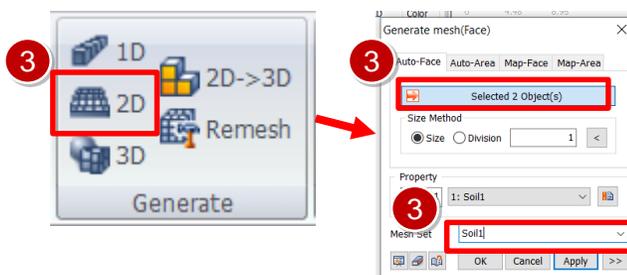
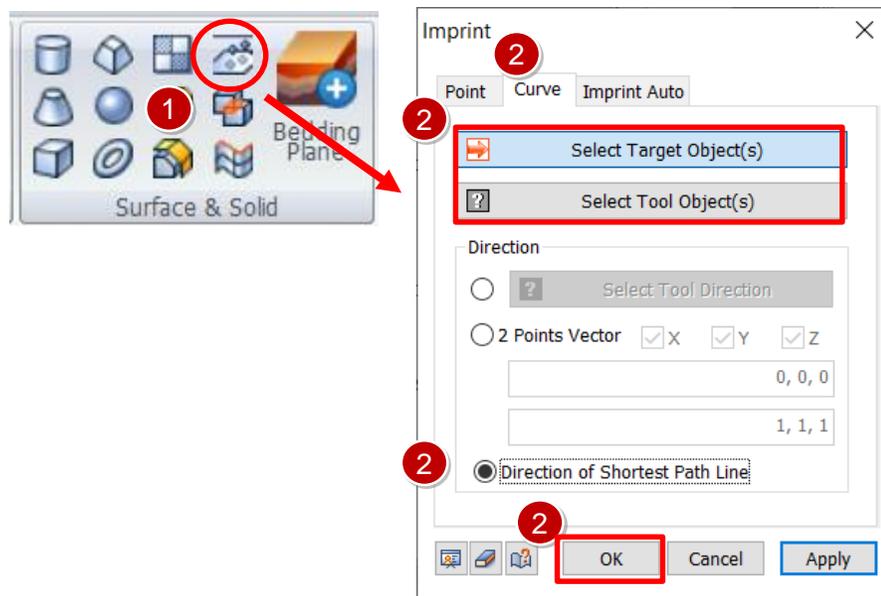
Procedure

1. Go to **Geometry > Surface & Solid > Imprint**
2. Go to **Curve** type, Select the face as a Target object. And Select all edges on the screen as tool object. Then, Choose '**Direction of Shortest Path Line**'. Click '**OK**'

All setting for the mesh is done.

Now We're making meshes.

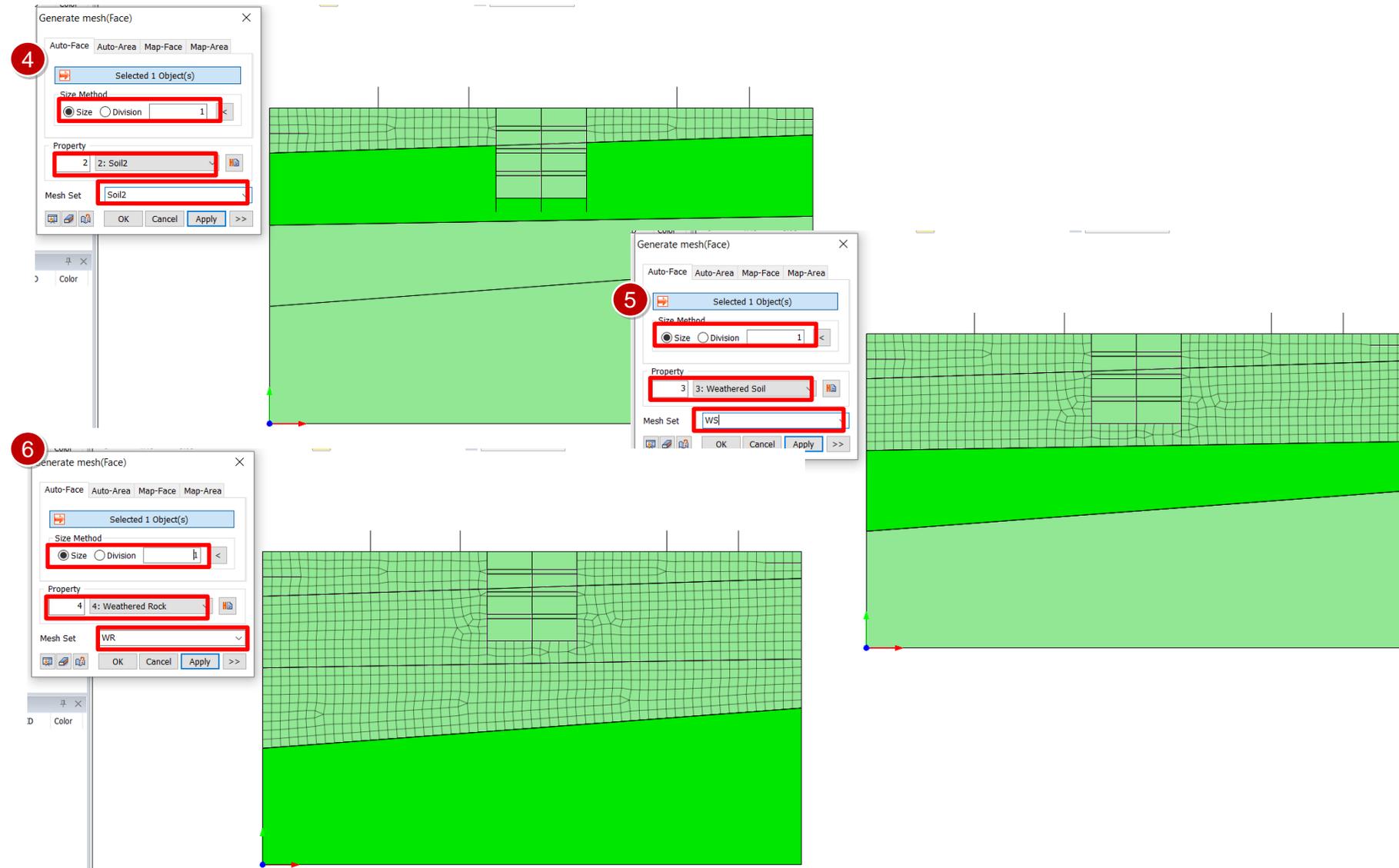
3. Go to **Mesh > Generate > 2D**. Go to 'Auto-Face', Click the faces for the Soil1 layer. Give 1 meter size and assign Soil1 property. Name Soil1 and Click OK.



Procedure

Repeatedly, Make Soil layers.

4. Select the face for the Soil2 layer. Give 1 meter size, assign Soil2 property and name Soil2. And, Apply.
5. Select the face for the Weathered Soil layer. Give 1 meter size, assign Weathered Soil property and name W.S. And, Apply.
6. Select the face for the Weathered Rock layer. Give 1 meter size, assign Weathered Rock property and name W.R. And, Apply.



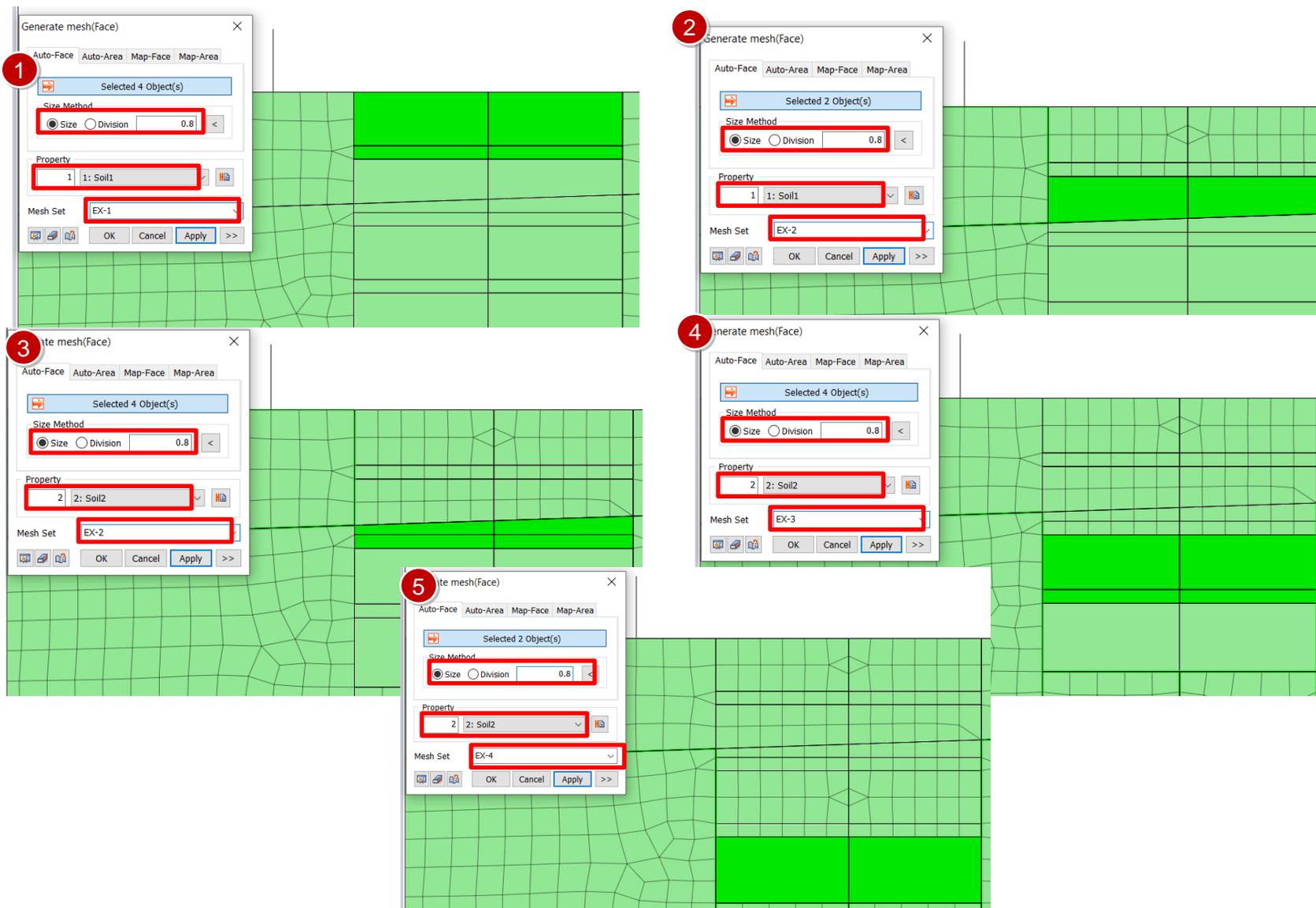
Procedure

Make Excavation Part.

1. Select the face for the EX-1. Give 0.8 meter size, assign Soil1 property and name EX-1. And, Apply.
2. Select the face for the EX-2. Give 0.8 meter size, assign Soil1 property and name EX-2. And, Apply.

***Be careful : Second Excavation part is divided with the different soil layers. So, the user has to assign different soil properties.*

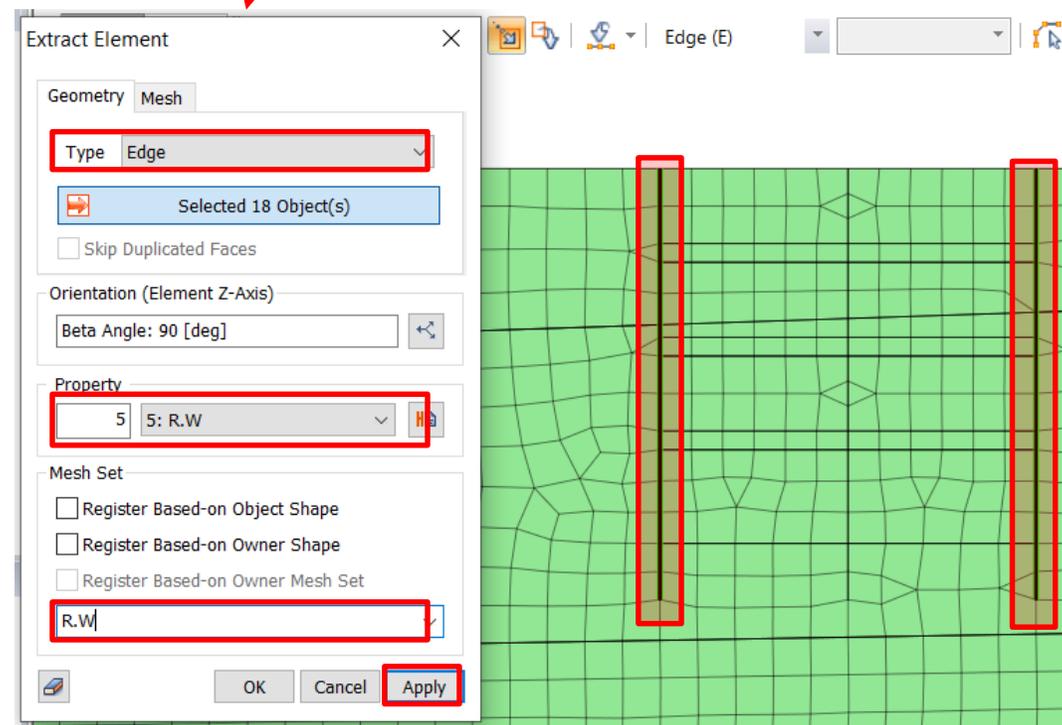
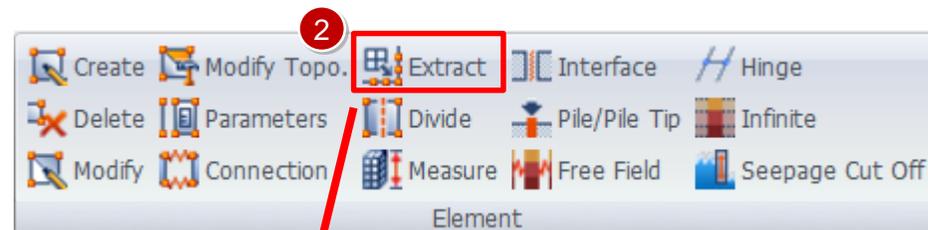
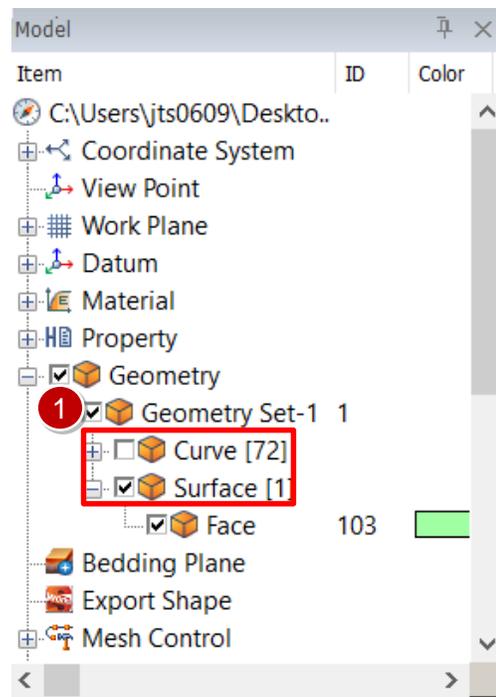
3. Select the face for the EX-2. Give 0.8 meter size, assign Soil2 property and name EX-2. And, Apply.
4. Select the face for the EX-3. Give 0.8 meter size, assign Soil2 property and name EX-3. And, Apply.
5. Select the face for the EX-4. Give 0.8 meter size, assign Soil2 property and name EX-4. And, Apply.



Procedure

Make Structural Elements.

1. Model > Geometry, **Hide** all the **Curves** and **Show only Face**.
2. Go to **Mesh > Element > Extract**, Select Type : **Edge**, Select **Retaining Wall lines**. Assign **R.W Property**, give name as **R.W** and click '**Apply**'.



2-2 Generate Mesh Set – Structural Elements

Procedure

1. Select **King Post** lines. Assign **King Post Property**, give name as **K.P** and click 'Apply'.
2. **Hide All Geometry set.**
3. Go to **Mesh > Element > Create**. Select 1D type, Grab the **Starting Point & End Point** of the First Strut, Give **Strut Property** and Name it as 'Strut-1'. Apply.

With the same process from 3.,
Make **4. Strut-2** and **5. Strut-3**

The image illustrates the software workflow for creating structural elements. It shows the 'Extract Element' dialog box with the following settings:

- Type: Edge
- Selected 9 Object(s)
- Orientation (Element Z-Axis): Beta Angle: 90 [deg]
- Property: 7: King Post
- Mesh Set: Register Based-on Object Shape, Register Based-on Owner Shape, Register Based-on Owner Mesh Set
- Property Name: K.P

The 'Element Create/Delete' dialog box shows the following settings:

- Element ID: 2257
- Nodal Connectivity: 35, 178
- Orientation (Element Z-Axis): Beta Angle: 90 [deg]
- Property: 6: Strut
- Mesh Set: Strut-1

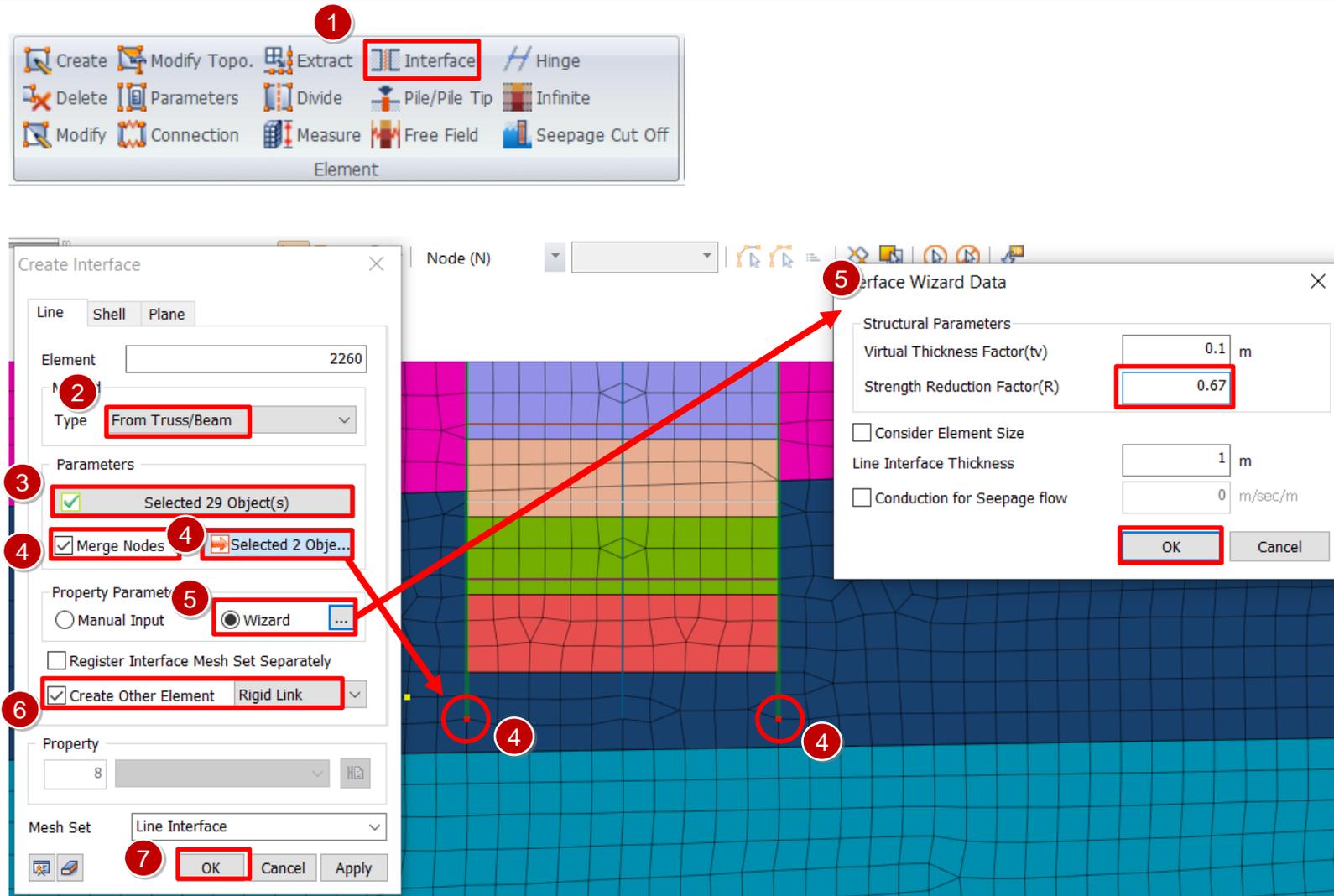
The meshed structure is shown in three stages:

1. The initial mesh with the King Post highlighted in red.
2. The King Post highlighted in red, with the mesh set hidden.
3. The King Post highlighted in red, with the mesh set hidden, and the first strut highlighted in red.

2-2 Generate Mesh Set – Interface Element

Procedure

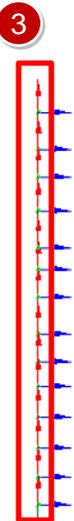
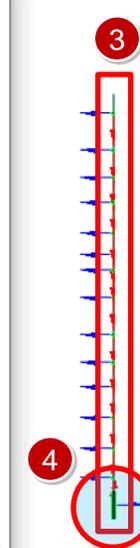
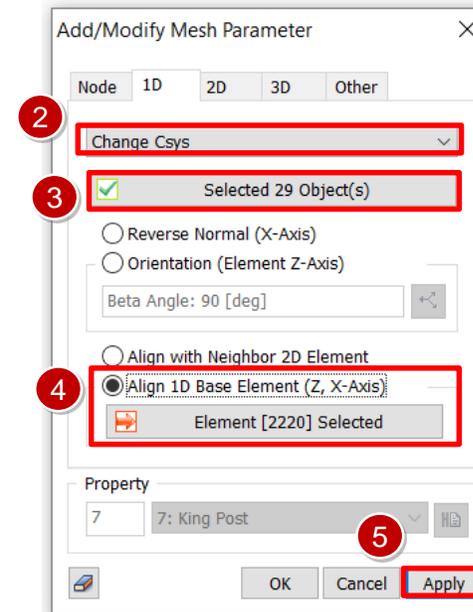
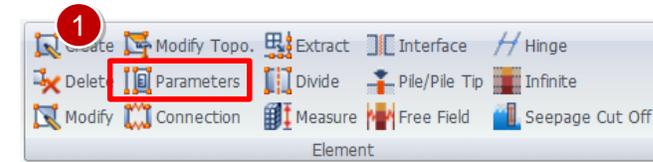
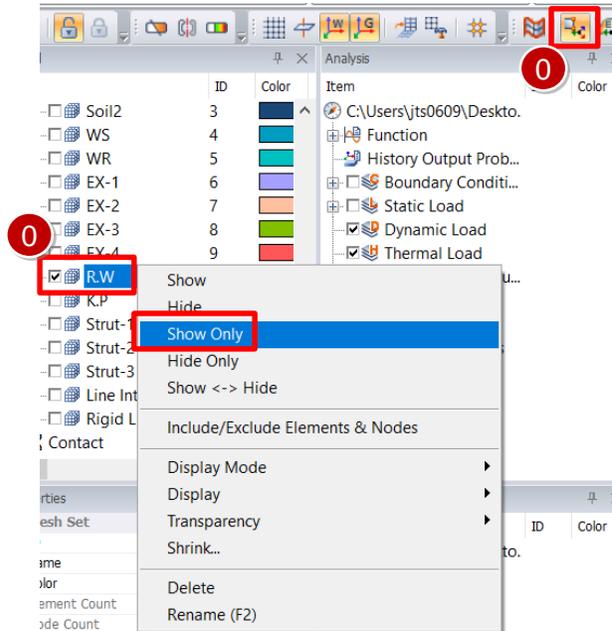
1. Go to **Mesh > Element > Interface**.
2. Select **'From Truss/Beam'** Type.
3. Select the **'R.W'** Mesh Set.
4. Tick **'Merge Nodes'** and Select the End Points of the Retaining Wall.
5. Click **'Wizard'** and Assign Strength Reduction Factor(R) as **'0.67'**.
6. Tick **'Create Other Element'** and select **'Rigid Link'**.
7. Click **OK**.



2-3 Generate Mesh Set – Align Element Coordinate System(Retaining Wall)

Procedure

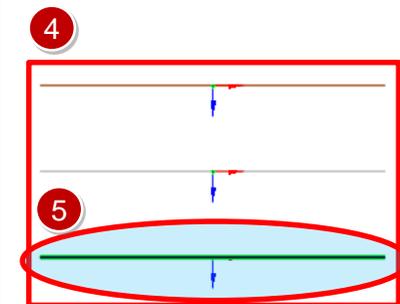
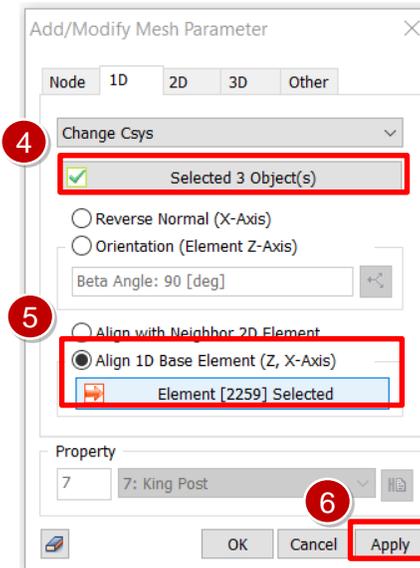
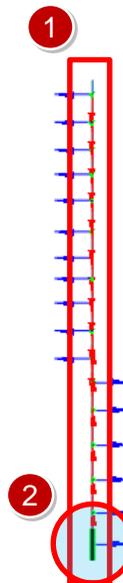
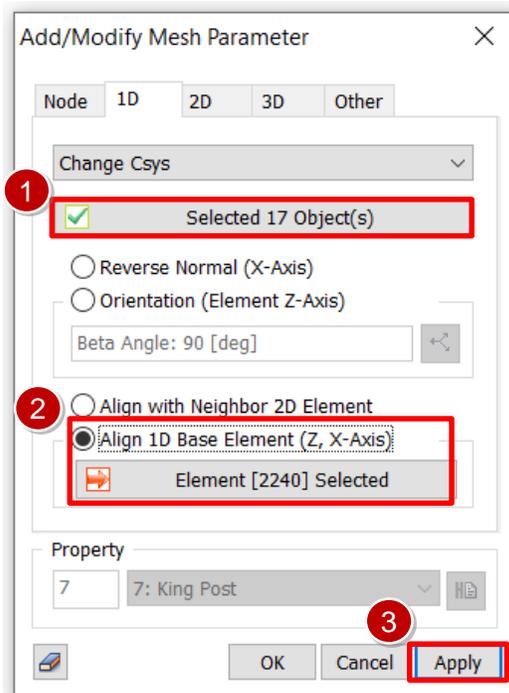
0. Click the right button on the **R.W** mesh set, Click **Show Only** and **Show/Hide Element CSys**
1. Go to **Mesh > Element > Parameters**.
2. Select **1D > Change Csys**.
3. Select the '**R.W**' Mesh Set.
4. Tick '**Align 1D Base Element(Z, X-Axis)**'. And, **Select the bottom one** from the left Retaining Wall.
5. Click **Apply**.



2-4 Generate Mesh Set – Align Element Coordinate System(King Post&Struts)

Procedure

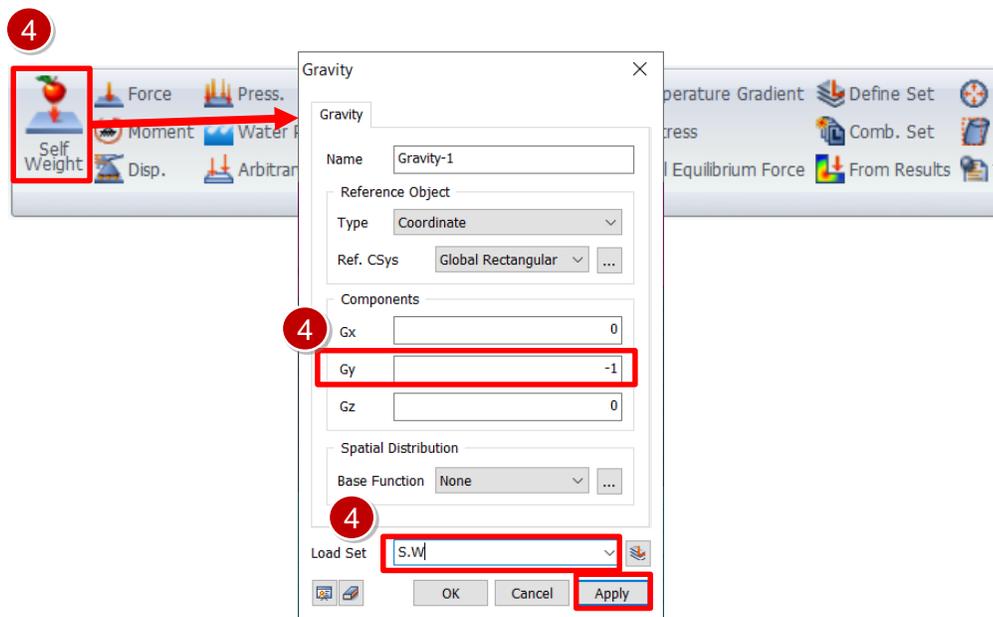
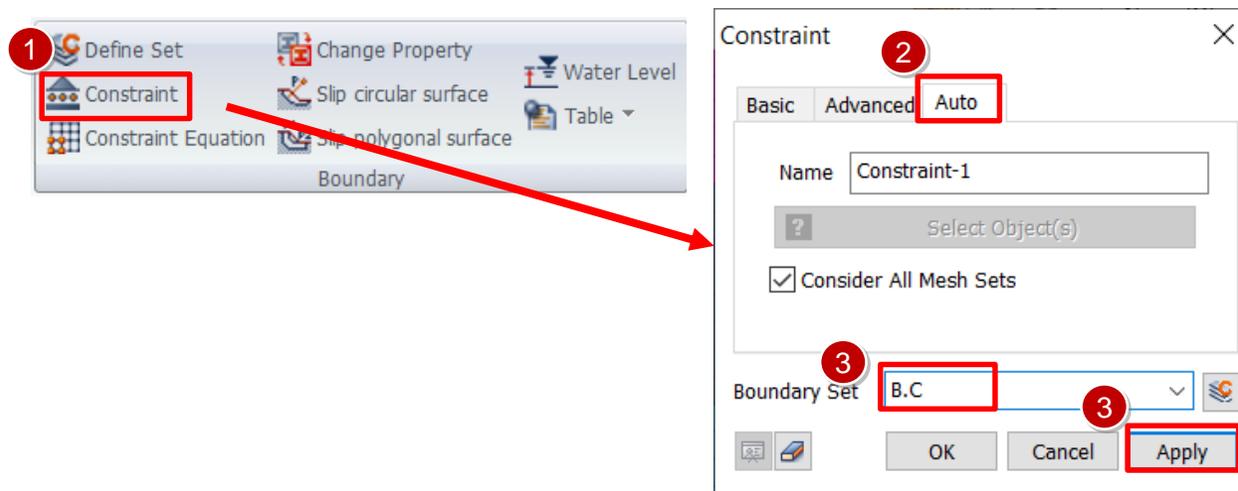
0. Click the right button on the **K.P** mesh set, Click **Show Only** and **Show/Hide Element CSys**
1. Select the 'K.P' Mesh Set.
2. Tick 'Align 1D Base Element(Z, X-Axis)'. And, **Select the bottom one.**
3. Click **Apply**.
4. Select the 'Strut-1, 2, 3' Mesh Set.
5. Tick 'Align 1D Base Element(Z, X-Axis)'. And, Select the 'Strut-3'.
6. Click **Apply**.



3-1 Assign Boundary/Load Condition(Constraint, Self Weight)

Procedure

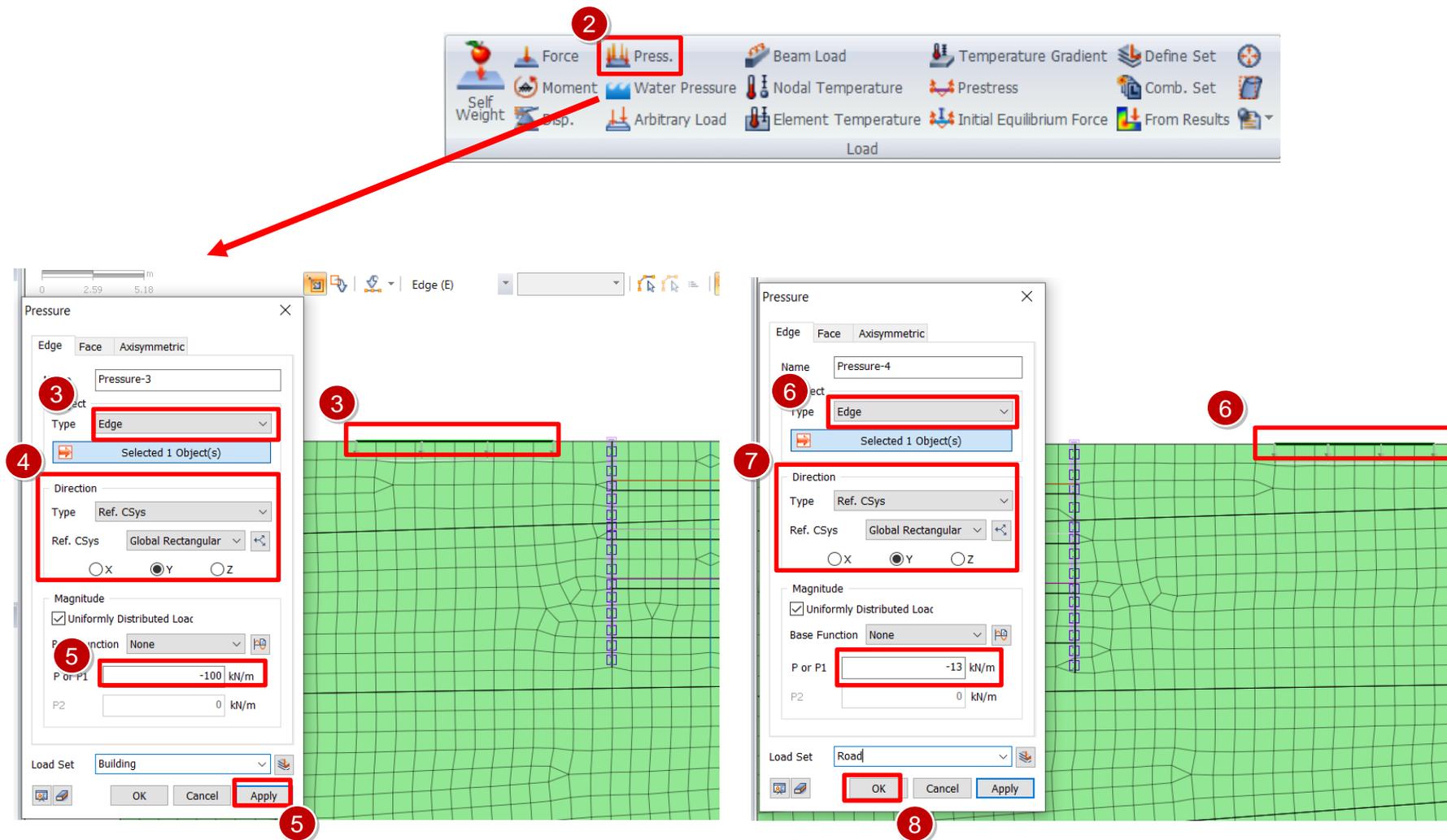
1. Go to **Static/Slope Analysis > Boundary > Constraint**.
2. Select Type : **'Auto'**.
3. Give **Name as a B.C.** and **Apply**.
4. Click 'Self Weight' under the **Static/Slope Analysis > Boundary > Load**. And, Check the Components whether it goes with Gy. Give name as **S.W** and **Apply**.



3-2 Assign Boundary/Load Condition(Pressure)

Procedure

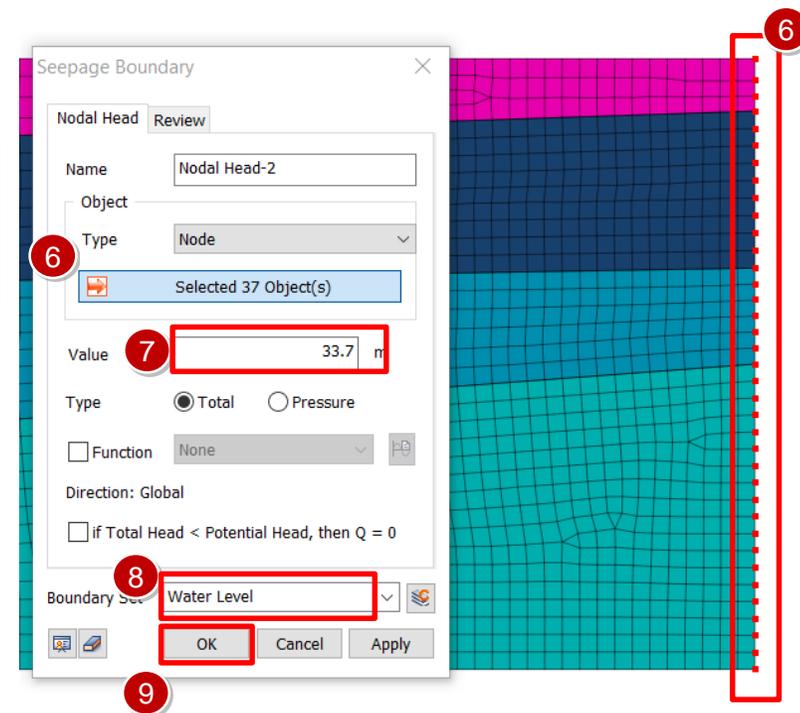
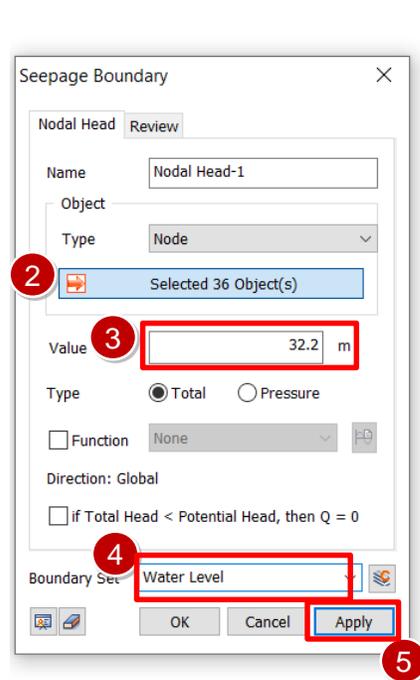
1. Show Surface
2. Go to **Static/Slope Analysis > Load > Pressure**.
3. Select Type : **'Edge'**. And Select the Building Edge.
4. Change the type to Ref. CSys and choose Y direction.
5. Give **'-100kN/m'** and **Name as a Building** and **Apply**.
6. Select Type : **'Edge'**. And Select the Road Edge.
7. Change the type to Ref. CSys and choose Y direction.
8. Give **'-13kN/m'** and **Name as a Road** and **Apply**.



3-3 Assign Boundary/Load Condition(Nodal Head)

Procedure

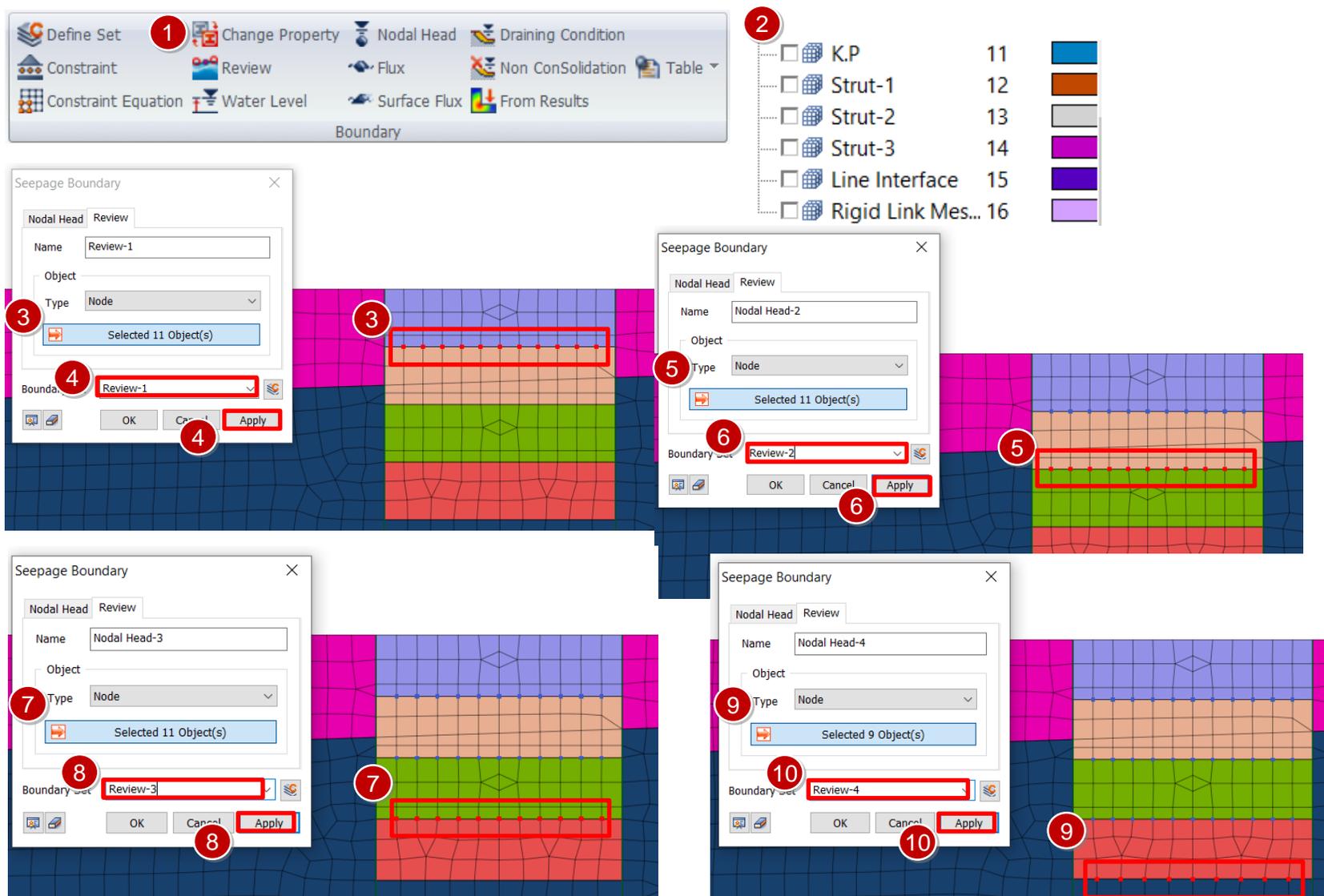
1. Go to **Seepage/Consolidation Analysis > Boundary > Nodal Head**.
2. Select the nodes of the **left outline** of the whole model.
3. Give value '**32.2**'.
4. Name as a '**Water Level**'.
5. Apply.
6. Select the nodes of the **right outline** of the whole model.
7. Give value '**33.7**'.
8. Name as a '**Water Level**'.
9. OK.



3-4 Assign Boundary/Load Condition(Review Boundary)

Procedure

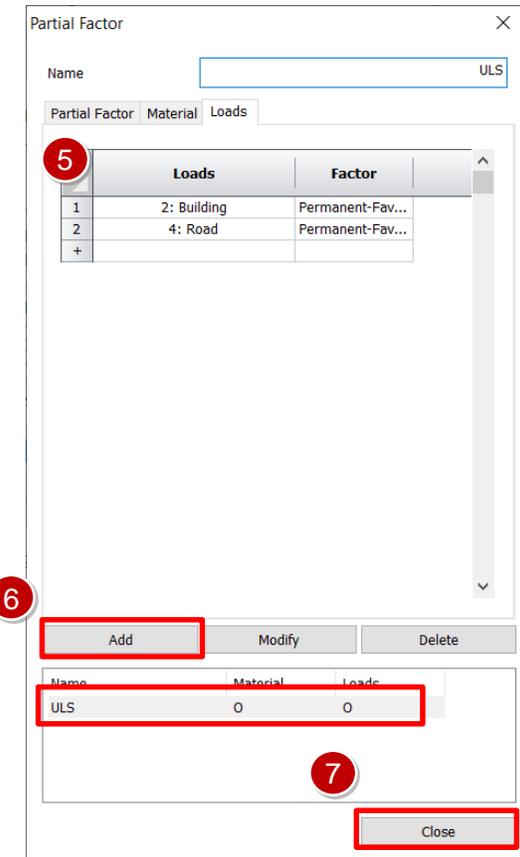
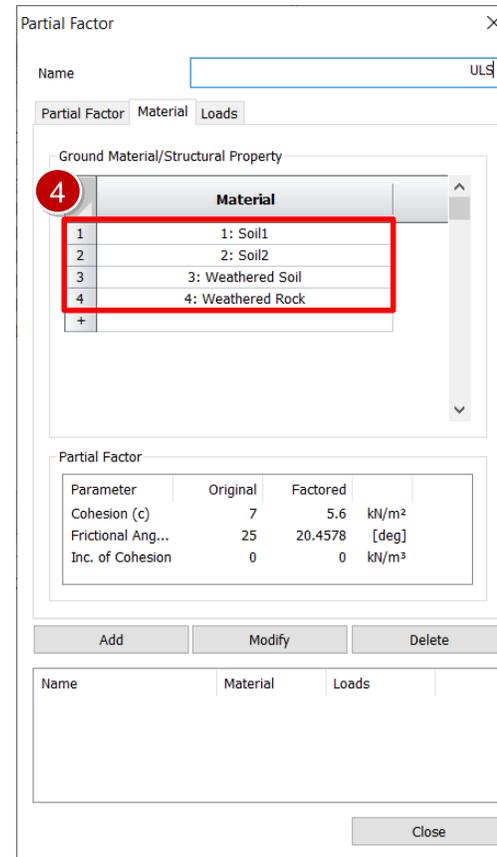
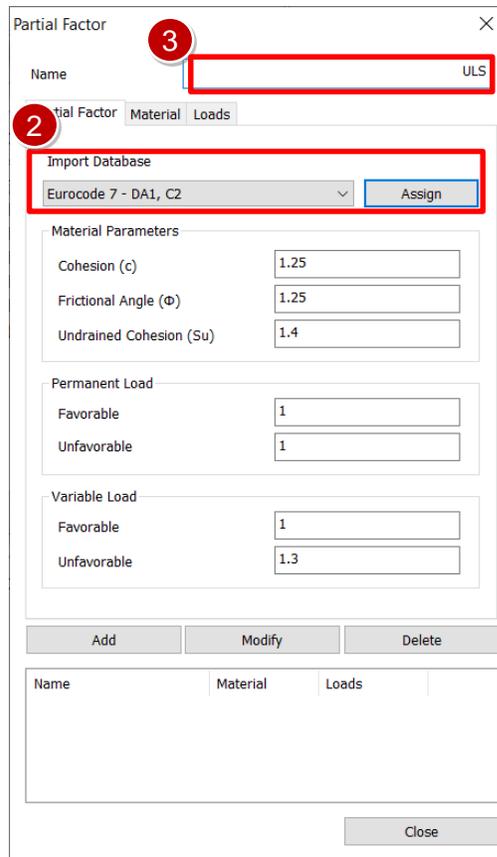
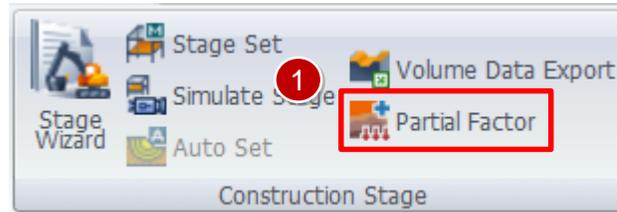
1. Go to **Seepage/Consolidation Analysis > Boundary > Review**
2. Hide the structural elements sets such as King Post, Struts, Line Interface, Rigid Link.
3. Select the nodes of the first excavation part's bottom.(without the nodes of the retaining wall locations.)
4. Give name as a **'Review-1'**. And Apply.
5. Select the nodes of the second excavation part's bottom.(without the nodes of the retaining wall locations.)
6. Give name as a **'Review-2'**. And Apply.
7. Select the nodes of the third excavation part's bottom.(without the nodes of the retaining wall locations.)
8. Give name as a **'Review-3'**. And Apply.
9. Select the nodes of the fourth excavation part's bottom.(without the nodes of the retaining wall locations.)
10. Give name as a **'Review-4'**. And Apply.



3-5 Assign Partial Factor

Procedure

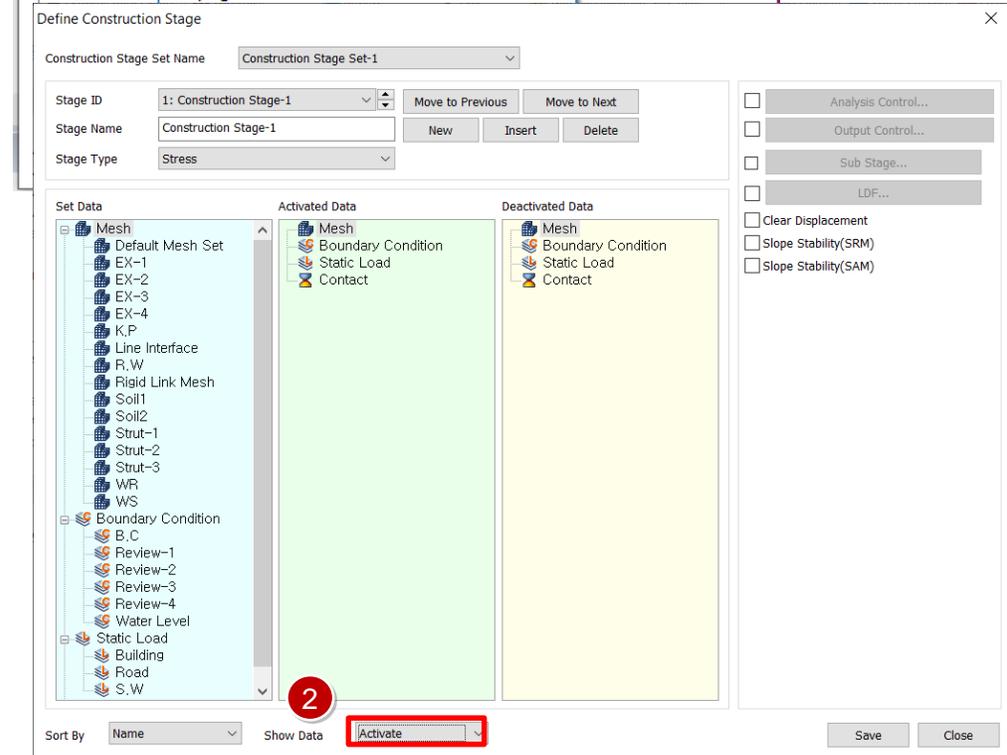
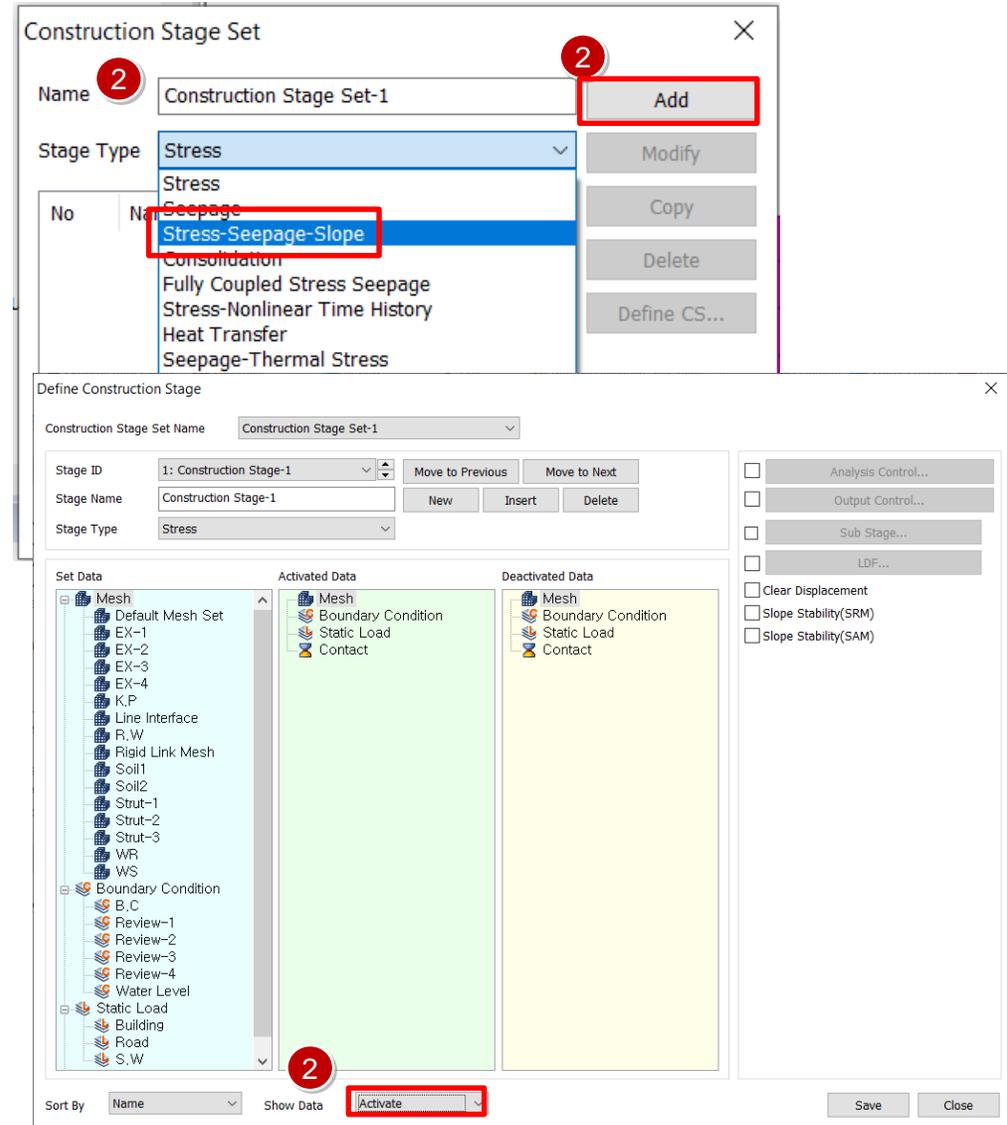
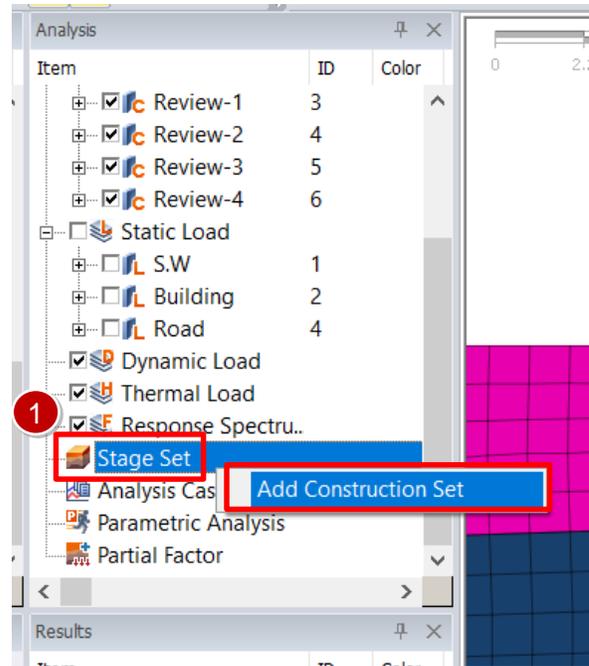
1. Go to **Static/Slope Analysis > Construction Stage > Partial Factor**.
2. Import Database '**Eurocode 7 – DA1, C2**'. And Click '**Assign**'.
3. Give name as a '**ULS**'
4. Assign the factor to the materials.
5. Assign the factor to the loads.
6. Click **Add**.
7. Close.



4-1 Define Construction Stage

Procedure

1. Click the right button on the Stage Set from Analysis work-tree. 'Add Construction Set'.
2. Select the 'Stress-Seepage-Slope' Stage type. Add it.
3. Change the Show Data to 'Active'.



4-1 Define Construction Stage

Procedure

1. Give Name as a 'Initial stage(seepage)'.
Initial stage(seepage)'.
2. Change to 'Steady-State'.
3. Activate the elements sets(**EX-1, EX-2, EX-3, EX-4, Rigid Link, Soil1, Soil2, WS, WR**) by using Drag&Drop from Set Data to Activated Data.
4. Activate the boundary conditions(**Boundary Condition, Water Level**) by using Drag&Drop from Set Data to Activated Data.
5. Save it.
6. Click 'New'.

Define Construction Stage

Construction Stage Set Name: Construction Stage Set-1

Stage ID: 1: Initial Stage(Seepage) 6 Move to Previous Move to Next

Stage Name: 1 Initial Stage(Seepage) 6 New Insert Delete

Stage Type: 2 Steady-State

Set Data 3 Activated Data Deactivated Data

Set Data: Mesh, Default Mesh Set, EX-1, EX-2, EX-3, EX-4, K,P, Line Interface, R.W, Rigid Link Mesh, Soil1, Soil2, Strut-1, Strut-2, Strut-3, WR, WS, Boundary Condition, B.C, Review-1, Review-2, Review-3, Review-4, Water Level, Contact

Activated Data: Mesh, EX-1, EX-2, EX-3, EX-4, Rigid Link Mesh, Soil1, Soil2, WR, WS, Boundary Condition, B.C, Water Level, Contact

Deactivated Data: Mesh, Boundary Condition, Contact

Sort By: Name Show Data: Activate

5 Save Close

4-1 Define Construction Stage

Procedure

1. Give Name as a 'Initial stage(stress)'.
2. Change to 'Stress'.
3. Activate the boundary conditions(**Self Weight**) by using Drag&Drop from Set Data to Activated Data.
4. Tick the 'Clear Displacement'.
5. Save it.
6. Click 'New'.

Define Construction Stage

Construction Stage Set Name: Construction Stage Set-1

Stage ID: 2: Initial Stage(stress) 6 Move to Previous Move to Next

Stage Name: 1 Initial Stage(stress) 6 New Insert Delete

Stage Type: 2 Stress

Set Data: Mesh, Default Mesh Set, EX-1, EX-2, EX-3, EX-4, K,P, Line Interface, R,W, Rigid Link Mesh, Soil1, Soil2, Strut-1, Strut-2, Strut-3, WR, WS, Boundary Condition, B,C, Review-1, Review-2, Review-3, Review-4, Water Level, Static Load, Building, Road, S,W

Activated Data: Mesh, Boundary Condition, Static Load, S,W, Contact

Deactivated Data: Mesh, Boundary Condition, Static Load, Contact

4 Clear Displacement
 Slope Stability(SRM)
 Slope Stability(SAM)

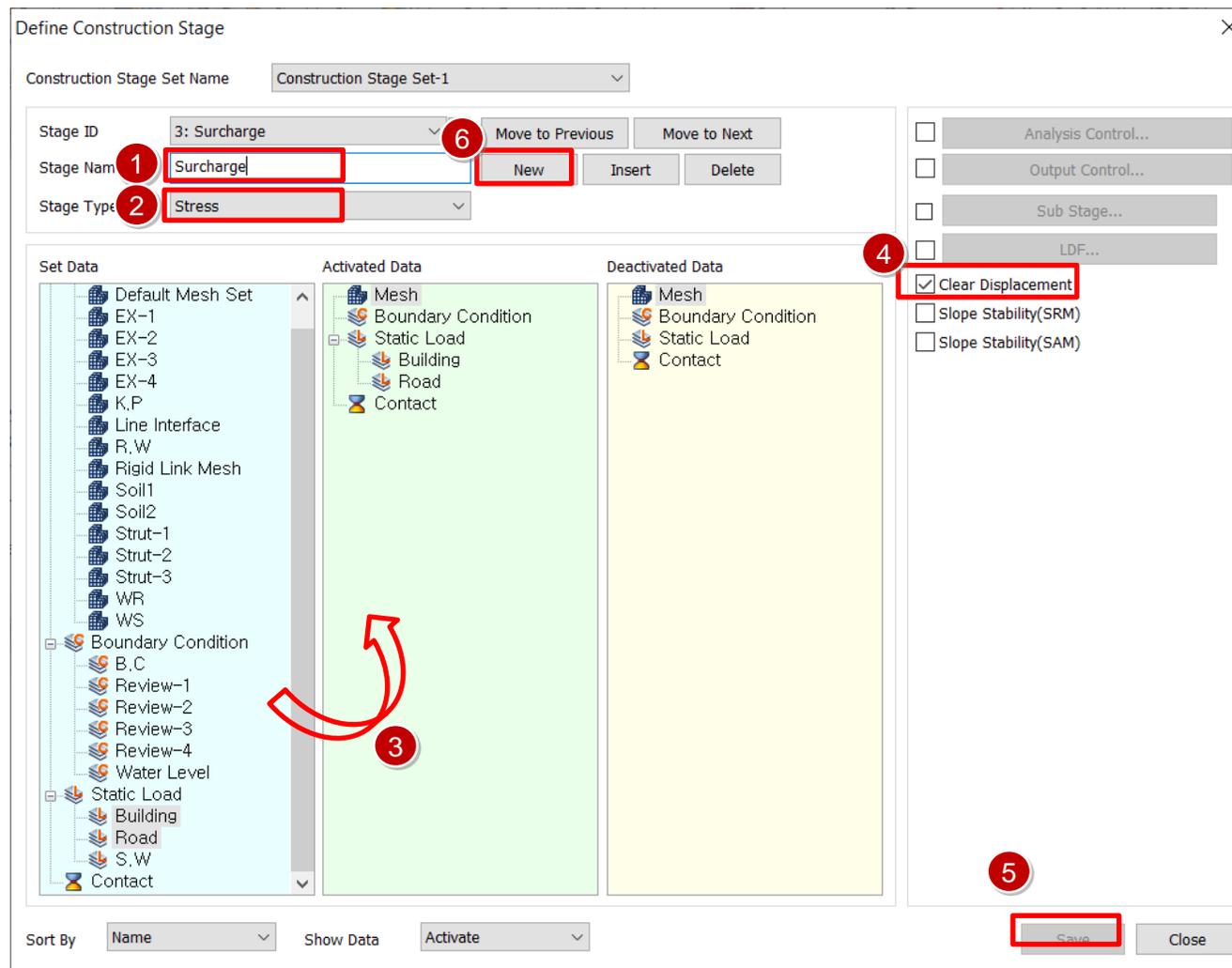
Sort By: Name Show Data: Activate

5 Save Close

4-1 Define Construction Stage

Procedure

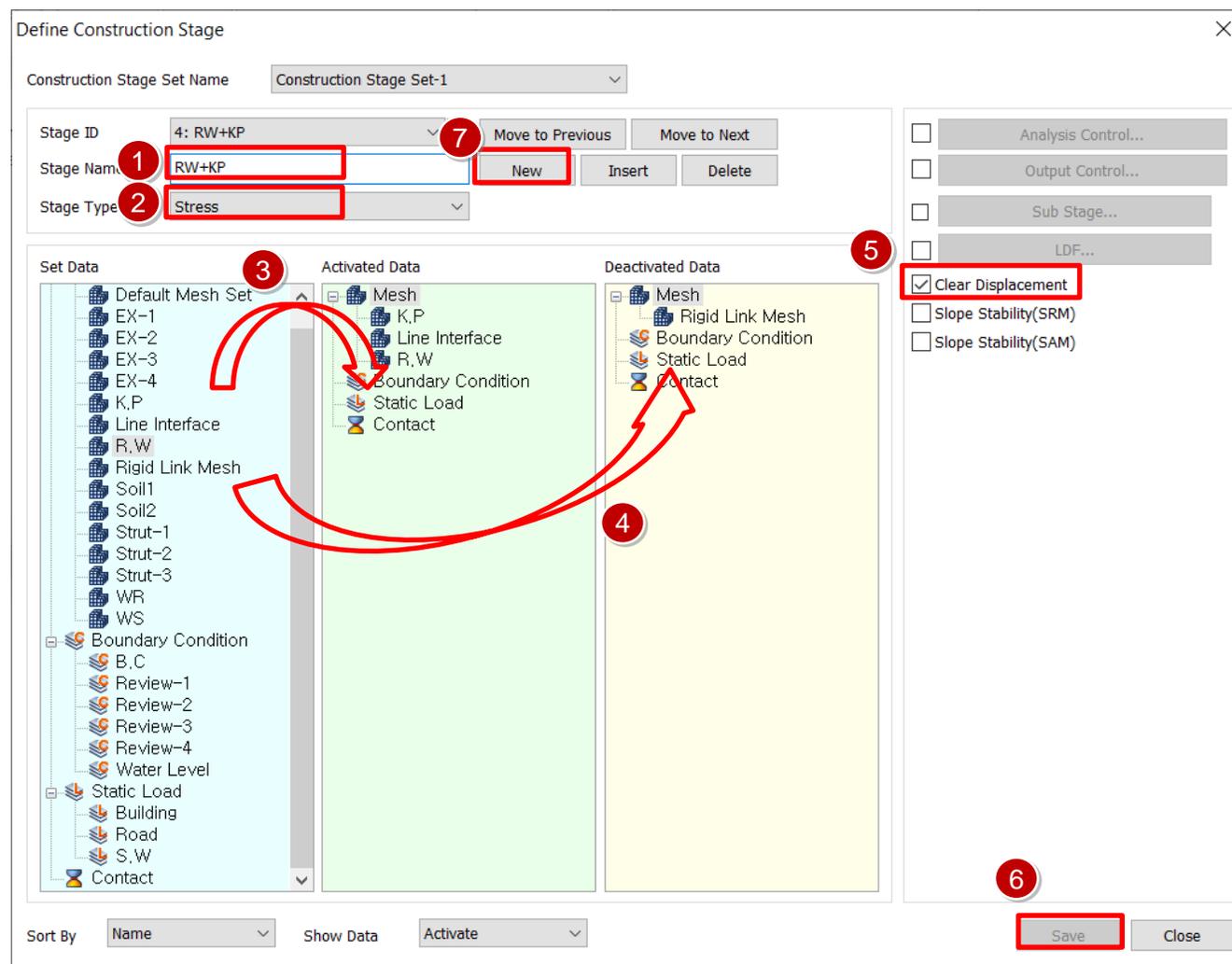
1. Give Name as a 'Surcharge'.
2. Change to 'Stress'.
3. Activate the Static loads(**Building, Road**) by using Drag&Drop from Set Data to Activated Data.
4. Tick the 'Clear Displacement'.
5. Save it.
6. Click 'New'.



4-1 Define Construction Stage

Procedure

1. Give Name as a 'R.W+K.P'.
2. Change to 'Stress'.
3. Activate the elements sets(**King Post, Line Interface, Retaining Wall**) by using Drag&Drop from Set Data to Activated Data.
4. Deactivate the elements sets(**Rigid Link**) by using Drag&Drop from Set Data to Deactivated Data.
5. Click 'Clear Displacement'.
6. Save it.
7. Click 'New'.



4-1 Define Construction Stage

Procedure

1. Give Name as a 'EX-1(seepage)'.
2. Change to 'Steady-State'.
3. Deactivate the elements sets(EX-1) by using Drag&Drop from Set Data to Deactivated Data.
4. Activate the boundary conditions(Review-1) by using Drag&Drop from Set Data to Activated Data.
5. Save it.
6. Click 'New'.

Define Construction Stage

Construction Stage Set Name: Construction Stage Set-1

Stage ID: 5: EX-1(seepage) 6 Move to Previous Move to Next

Stage Name: 1 EX-1(seepage) 6 New Insert Delete

Stage Type: 2 Steady-State

Set Data: Mesh, Default Mesh Set, EX-1, EX-2, EX-3, EX-4, K,P, Line Interface, R,W, Rigid Link Mesh, Soil1, Soil2, Strut-1, Strut-2, Strut-3, WR, WS, Boundary Condition, B.C, Review-1, Review-2, Review-3, Review-4, Water Level, Contact

Activated Data: Mesh, Boundary Condition, Review-1, Contact

Deactivated Data: Mesh, EX-1, Boundary Condition, Contact

Sort By: Name Show Data: Activate

5 Save Close

4-1 Define Construction Stage

Procedure

1. Give Name as a 'EX-1(stress)'.
2. Change to 'Stress'.
3. Tick the 'Sub Stage...'
4. Name as a 'ULS' using ULS partial factor.
5. Click OK.
6. Save it.
7. Click 'New'.

Define Construction Stage

Construction Stage Set Name: Construction Stage Set-1

Stage ID: 6: EX-1(stress)

Stage Name: EX-1(stress)

Stage Type: Stress

Buttons: Move to Previous, Move to Next, New, Insert, Delete

Sub Stage...

Set Data:

- Mesh
 - Default Mesh Set
 - EX-1
 - EX-2
 - EX-3
 - EX-4
 - K,P
 - Line Interface
 - R,W
 - Rigid Link Mesh
 - Soil1
 - Soil2
 - Strut-1
 - Strut-2
 - Strut-3
 - WR
 - WS
- Boundary Condition
 - B,C
 - Review-1
 - Review-2
 - Review-3
 - Review-4
 - Water Level
- Static Load
 - Building
 - Road
 - S,W

Activated Data:

- Mesh
- Boundary Condition
- Static Load
- Contact

Deactivated Data:

- Mesh
- Boundary Condition
- Static Load
- Contact

Buttons: Analysis Control..., Output Control..., Sub Stage..., LDF..., Clear Displacement, Slope Stability(SRM), Slope Stability(SAM)

Buttons: Save, Close

Sub Stage

Name	Partial Factor
ULS	ULS

Buttons: OK, Cancel

4-1 Define Construction Stage

Procedure

1. Give Name as a 'Strut-1'.
2. Change to 'Stress'.
3. Activate the elements sets(**Strut-1**) by using Drag&Drop from Set Data to Activated Data.
4. Tick the 'Sub Stage...'.
5. Name as a 'ULS' using ULS partial factor.
6. Click OK.
7. Save it.
8. Click 'New'.

Define Construction Stage

Construction Stage Set Name: Construction Stage Set-1

Stage ID: 7: Strut-1

Stage Name: **Strut-1**

Stage Type: **Stress**

Set Data

- Mesh
 - Default Mesh Set
 - EX-1
 - EX-2
 - EX-3
 - EX-4
 - K,P
 - Line Interface
 - R,W
 - Rigid Link Mesh
 - Soil1
 - Soil2
 - Strut-1
 - Strut-2
 - Strut-3
 - WR
 - WS
- Boundary Condition
 - B,C
 - Review-1
 - Review-2
 - Review-3
 - Review-4
 - Water Level
- Static Load
 - Building
 - Road
 - S,W

Activated Data

- Mesh
 - Strut-1
- Boundary Condition
 - Static Load
- Contact
 - Contact

Deactivated Data

- Mesh
 - Boundary Condition
 - Static Load
 - Contact

Sub Stage

Name	Partial Factor
ULS	ULS

OK

Save

4-1 Define Construction Stage

Procedure

1. Give Name as a 'EX-2(seepage)'.
2. Change to 'Steady-State'.
3. Deactivate the elements sets(EX-2) and boundary condition(Review-1) by using Drag&Drop from Set Data to Deactivated Data.
4. Activate the boundary conditions(Review-2) by using Drag&Drop from Set Data to Activated Data.
5. Save it.
6. Click 'New'.

Define Construction Stage

Construction Stage Set Name: Construction Stage Set-1

Stage ID: 8: Construction Stage-1

Stage Name: EX-2(seepage)

Stage Type: Steady-State

Buttons: Move to Previous, Move to Next, New, Insert, Delete

Analysis Control...
Output Control...

Set Data: Mesh, Default Mesh Set, EX-1, EX-2, EX-3, EX-4, K,P, Line Interface, R,W, Rigid Link Mesh, Soil1, Soil2, Strut-1, Strut-2, Strut-3, WR, WS, Boundary Condition, B.C, Review-1, Review-2, Review-3, Review-4, Water Level, Contact

Activated Data: Mesh, Boundary Condition, Review-2, Contact

Deactivated Data: Mesh, EX-2, Boundary Condition, Review-1, Contact

Sort By: Name

Show Data: Activate

Buttons: Save, Close

4-1 Define Construction Stage

Procedure

1. Give Name as a 'EX-2(stress)'.
2. Change to 'Stress'.
3. Tick the 'Sub Stage...'.
4. Name as a 'ULS' using ULS partial factor.
5. Click OK.
6. Save it.
7. Click 'New'.

Define Construction Stage

Construction Stage Set Name: Construction Stage Set-1

Stage ID: 9: Construction Stage-1

Stage Name: EX-2(stress)

Stage Type: Stress

Set Data:

- Mesh
 - Default Mesh Set
 - EX-1
 - EX-2
 - EX-3
 - EX-4
 - K,P
 - Line Interface
 - R,W
 - Rigid Link Mesh
 - Soil1
 - Soil2
 - Strut-1
 - Strut-2
 - Strut-3
 - WR
 - WS
- Boundary Condition
 - B,C
 - Review-1
 - Review-2
 - Review-3
 - Review-4
 - Water Level
- Static Load
 - Building
 - Road
 - S,W

Activated Data:

- Mesh
- Boundary Condition
- Static Load
- Contact

Deactivated Data:

- Mesh
- Boundary Condition
- Static Load
- Contact

Options:

- Analysis Control...
- Output Control...
- Sub Stage...
- LDF...
- Clear Displacement
- Slope Stability(SRM)
- Slope Stability(SAM)

Sub Stage

	Name	Partial Factor
1	ULS	ULS
+		

Buttons: New, Insert, Delete, Move to Previous, Move to Next, OK, Cancel, Save, Close.

4-1 Define Construction Stage

Procedure

1. Give Name as a 'Strut-2'.
2. Change to 'Stress'.
3. Activate the elements sets(**Strut-2**) by using Drag&Drop from Set Data to Activated Data.
4. Tick the 'Sub Stage...'.
5. Name as a 'ULS' using ULS partial factor.
6. Click OK.
7. Save it.
8. Click 'New'.

Define Construction Stage

Construction Stage Set Name: Construction Stage Set-1

Stage ID: 10: Strut-2

Stage Name: **1** Strut-2 **8** New

Stage Type: **2** Stress **4** Sub Stage...

Set Data: Mesh, Default Mesh Set, EX-1, EX-2, EX-3, EX-4, K,P, Line Interface, R,W, Rigid Link Mesh, Soil1, Soil2, Strut-1, **3** Strut-2, Strut-3, WR, WS, Boundary Condition, B,C, Review-1, Review-2, Review-3, Review-4, Water Level, Static Load, Building, Road, S,W

Activated Data: Mesh, Strut-2, Boundary Condition, Static Load, Contact

Deactivated Data: Mesh, Boundary Condition, Static Load, Contact

Sub Stage

Name	Partial Factor
5 1 ULS	ULS
+	

6 OK Cancel

7 Save Close

4-1 Define Construction Stage

Procedure

1. Give Name as a 'EX-3(seepage)'.
2. Change to 'Steady-State'.
3. Deactivate the elements sets(EX-3) and boundary condition(Review-2) by using Drag&Drop from Set Data to Deactivated Data.
4. Activate the boundary conditions(Review-3) by using Drag&Drop from Set Data to Activated Data.
5. Save it.
6. Click 'New'.

Define Construction Stage

Construction Stage Set Name: Construction Stage Set-1

Stage ID: 11: EX-3(seepage) 6 Move to Previous Move to Next

Stage Name: 1 EX-3(seepage) 6 New Insert Delete

Stage Type: 2 Steady-State

Set Data: Mesh, Default Mesh Set, EX-1, EX-2, EX-3, EX-4, K,P, Line Interface, R,W, Rigid Link Mesh, Soil1, Soil2, Strut-1, Strut-2, Strut-3, WR, WS, Boundary Condition, B.C, Review-1, Review-2, Review-3, Review-4, Water Level, Contact

Activated Data: Mesh, Boundary Condition, Review-3, Contact 4

Deactivated Data: Mesh, EX-3, Boundary Condition, Review-2, Contact 3

Sort By: Name Show Data: Activate

5 Save Close

Analysis Control...
Output Control...

4-1 Define Construction Stage

Procedure

1. Give Name as a 'EX-3(stress)'.
2. Change to 'Stress'.
3. Tick the 'Sub Stage...'.
4. Name as a 'ULS' using ULS partial factor.
5. Click OK.
6. Save it.
7. Click 'New'.

The screenshot shows the 'Define Construction Stage' dialog box with the following settings:

- Construction Stage Set Name: Construction Stage Set-1
- Stage ID: 12: EX-3(stress)
- Stage Name: EX-3(stress)
- Stage Type: Stress
- Buttons: Move to Previous, Move to Next, New, Insert, Delete
- Options: Analysis Control..., Output Control..., Sub Stage... (checked), LDF..., Clear Displacement, Slope Stability(SRM), Slope Stability(SAM)
- Set Data: Mesh, Boundary Condition, Static Load, Contact
- Activated Data: Mesh, Boundary Condition, Static Load, Contact
- Deactivated Data: Mesh, Boundary Condition, Static Load, Contact
- Buttons: Save, Close

The 'Sub Stage' dialog box shows the following table:

	Name	Partial Factor
1	ULS	ULS
+		

Buttons: OK, Cancel

4-1 Define Construction Stage

Procedure

1. Give Name as a 'Strut-3'.
2. Change to 'Stress'.
3. Activate the elements sets(**Strut-3**) by using Drag&Drop from Set Data to Activated Data.
4. Tick the 'Sub Stage...'.
5. Name as a 'ULS' using ULS partial factor.
6. Click OK.
7. Save it.
8. Click 'New'.

Define Construction Stage

Construction Stage Set Name: Construction Stage Set-1

Stage ID: 13: Construction Stage-1

Stage Name: Strut-3

Stage Type: Stress

Set Data:

- Mesh
 - Default Mesh Set
 - EX-1
 - EX-2
 - EX-3
 - EX-4
 - K,P
 - Line Interface
 - R,W
 - Rigid Link Mesh
 - Soil1
 - Soil2
 - Strut-1
 - Strut-2
 - Strut-3
 - WR
 - WS
- Boundary Condition
 - B,C
 - Review-1
 - Review-2
 - Review-3
 - Review-4
 - Water Level
- Static Load
 - Building
 - Road
 - S,W

Activated Data:

- Mesh
 - Strut-3
- Boundary Condition
- Static Load
- Contact

Deactivated Data:

- Mesh
 - Boundary Condition
 - Static Load
 - Contact

Sub Stage

	Name	Partial Factor
1	ULS	ULS
+		

OK

Save

4-1 Define Construction Stage

Procedure

1. Give Name as a 'EX-3(seepage)'.
2. Change to 'Steady-State'.
3. Deactivate the elements sets(EX-4) and boundary condition(Review-3) by using Drag&Drop from Set Data to Deactivated Data.
4. Activate the boundary conditions(Review-4) by using Drag&Drop from Set Data to Activated Data.
5. Save it.
6. Click 'New'.

Define Construction Stage

Construction Stage Set Name: Construction Stage Set-1

Stage ID: 14: EX-4(seepage) 6 Move to Previous Move to Next

Stage Name: 1 EX-4(seepage) 6 New Insert Delete

Stage Type: 2 Steady-State

Analysis Control...
Output Control...

Set Data	Activated Data	Deactivated Data
<ul style="list-style-type: none"> Mesh <ul style="list-style-type: none"> Default Mesh Set EX-1 EX-2 EX-3 EX-4 K,P Line Interface R,W Rigid Link Mesh Soil1 Soil2 Strut-1 Strut-2 Strut-3 WR WS Boundary Condition <ul style="list-style-type: none"> B,C Review-1 Review-2 Review-3 Review-4 Water Level Contact 	<ul style="list-style-type: none"> Mesh <ul style="list-style-type: none"> Boundary Condition Review-4 Contact 	<ul style="list-style-type: none"> Mesh <ul style="list-style-type: none"> EX-4 Boundary Condition Review-3 Contact

Sort By: Name Show Data: Activate

5 Save Close

4-1 Define Construction Stage

Procedure

1. Give Name as a 'EX-4(stress)'.
2. Change to 'Stress'.
3. Tick the 'Sub Stage...'.
4. Name as a 'ULS' using ULS partial factor.
5. Click OK.
6. Save it.
7. Click 'New'.

The screenshot shows the 'Define Construction Stage' dialog box with the following settings:

- Construction Stage Set Name: Construction Stage Set-1
- Stage ID: 15: Construction Stage-1
- Stage Name: EX-4(stress)
- Stage Type: Stress
- Buttons: Move to Previous, Move to Next, New, Insert, Delete
- Sub Stage... checkbox: checked
- Buttons: Analysis Control..., Output Control..., LDF..., Clear Displacement, Slope Stability(SRM), Slope Stability(SAM)
- Buttons: Save, Close

The 'Sub Stage' dialog box shows the following table:

Name	Partial Factor
1 ULS ULS	
+	

Buttons: OK, Cancel

Buttons: Save, Close

4-2 Define Analysis Case

Procedure

1. Select **Analysis > Analysis Case > General**.
2. Name as a **'Deep Excavation'**.
3. Change the Solution Type to **'Construction Stage'**.
4. Tick the **'Automatically Consider Water Pressure'**.
5. Tick **'Initial Stage for Stress Analysis'**, **'Apply K0 Condition'** and select the **'2:Initial Stage(stress)'**.
6. OK.
7. OK.

The image shows two dialog boxes from a software application. The first dialog, 'Add/Modify Analysis Case', has the following elements highlighted with red boxes and numbers:

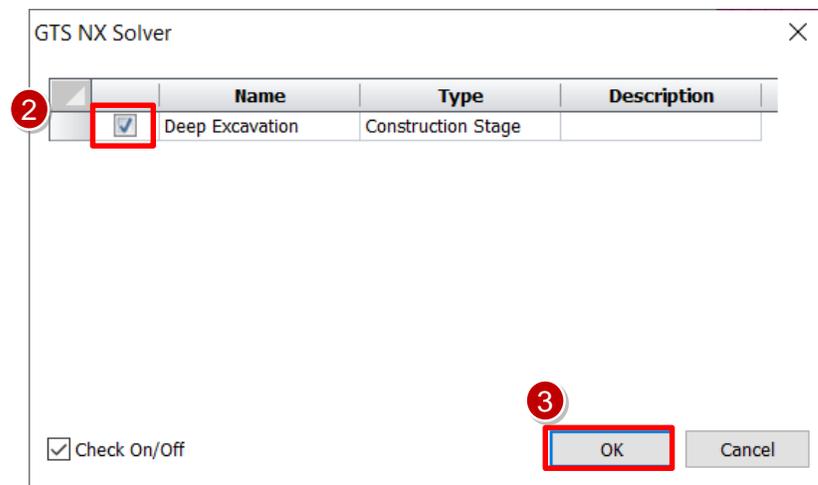
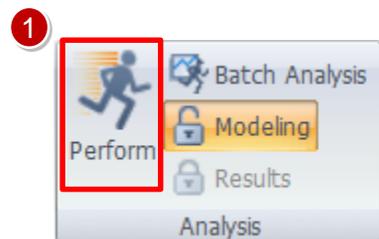
- 1:** The 'General' tab in the top-left corner.
- 2:** The 'Title' field containing the text 'Deep Excavation'.
- 3:** The 'Solution Type' dropdown menu, with 'Construction Stage' selected.
- 7:** The 'OK' button at the bottom.

The second dialog, 'Analysis Control', has the following elements highlighted with red boxes and numbers:

- 4:** The 'Water Pressure' section, with the checkbox 'Automatically Consider Water Pressure' checked.
- 5:** The 'Initial Stage' section, with the checkbox 'Initial Stage for Stress Analysis' checked, the radio button 'Apply K0 Condition' selected, and the dropdown menu set to '2:Initial Stage(stress)'. The checkbox 'Cut-Off Negative Effective Pressure' is unchecked.
- 6:** The 'OK' button at the bottom.

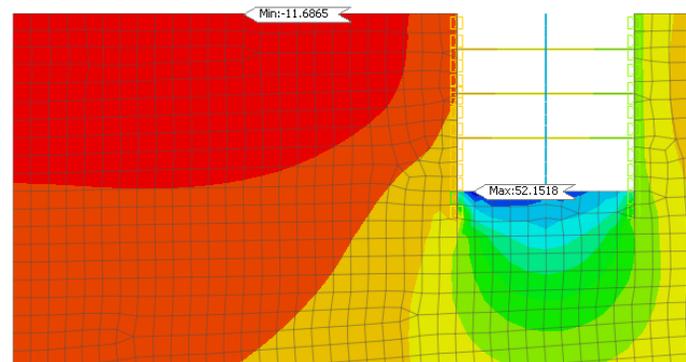
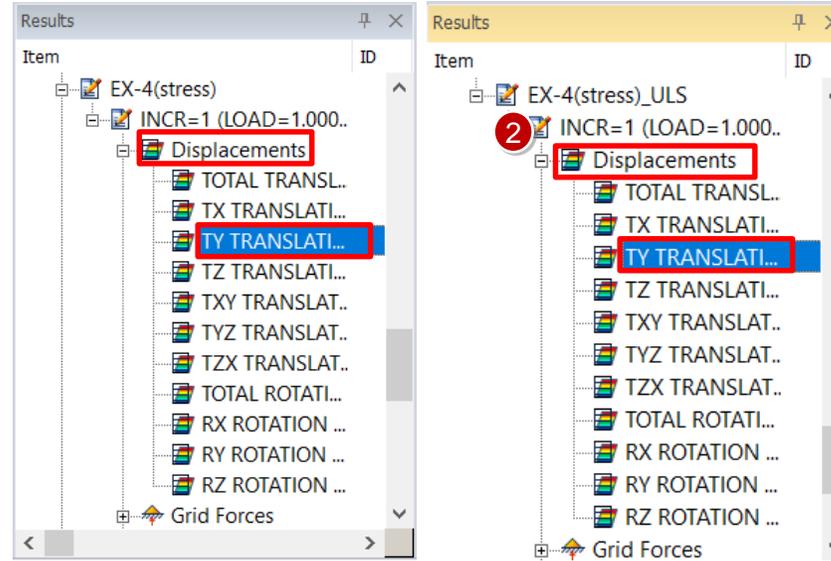
Procedure

1. Select **Analysis > Perform**
2. Check on **'Deep Excavation'**
3. Click **'OK'**



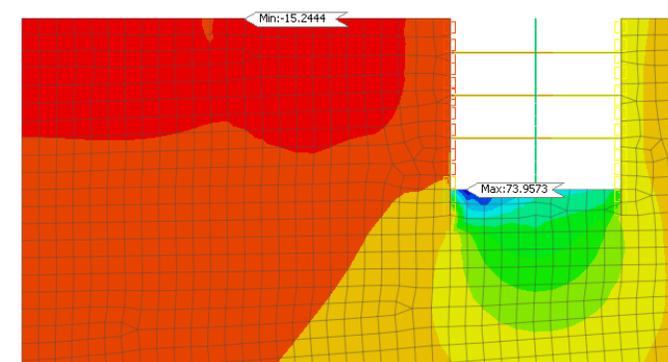
Procedure

1. Go to **Results Tab**, expand EX-4(stress) construction stage. And Under Displacements, Click on TY Translation (V).
2. Expand EX-4(stress)_ULS construction stage. And Under Displacements, Click on TY Translation (V).
3. Users can **check the results comparing to the normal version and ULS version(partial factor)**



Normal 4th Excavation Stage

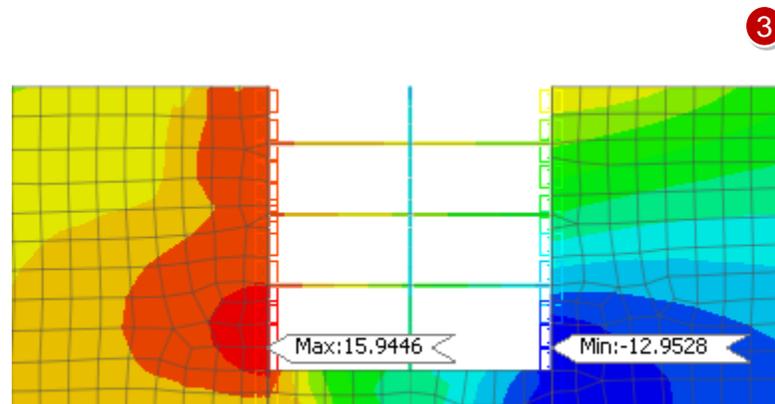
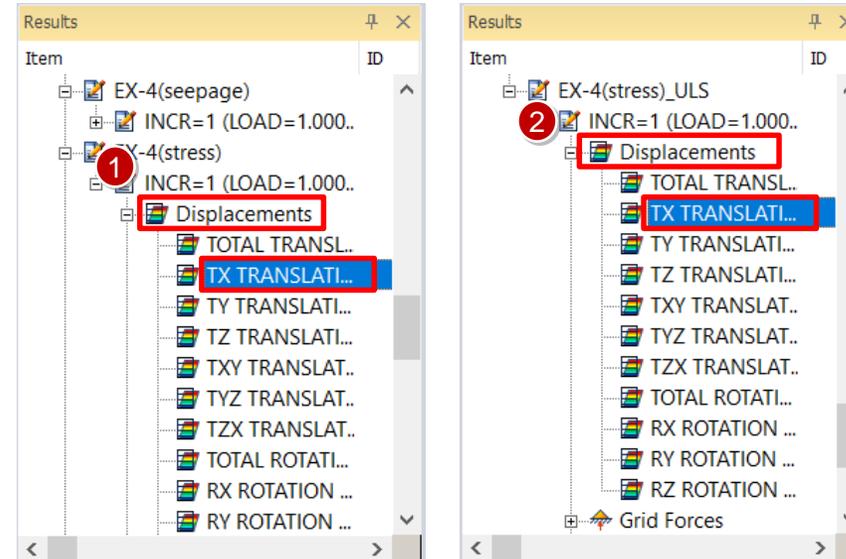
3



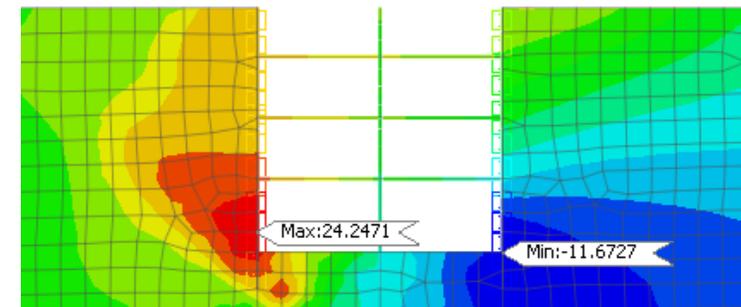
ULS 4th Excavation Stage

Procedure

1. Go to **Results Tab**, expand EX-4(stress) construction stage. And Under Displacements, Click on TX Translation (V).
2. Expand EX-4(stress)_ULS construction stage. And Under Displacements, Click on TX Translation (V).
3. Users can **check the results comparing to the normal version and ULS version(partial factor)**



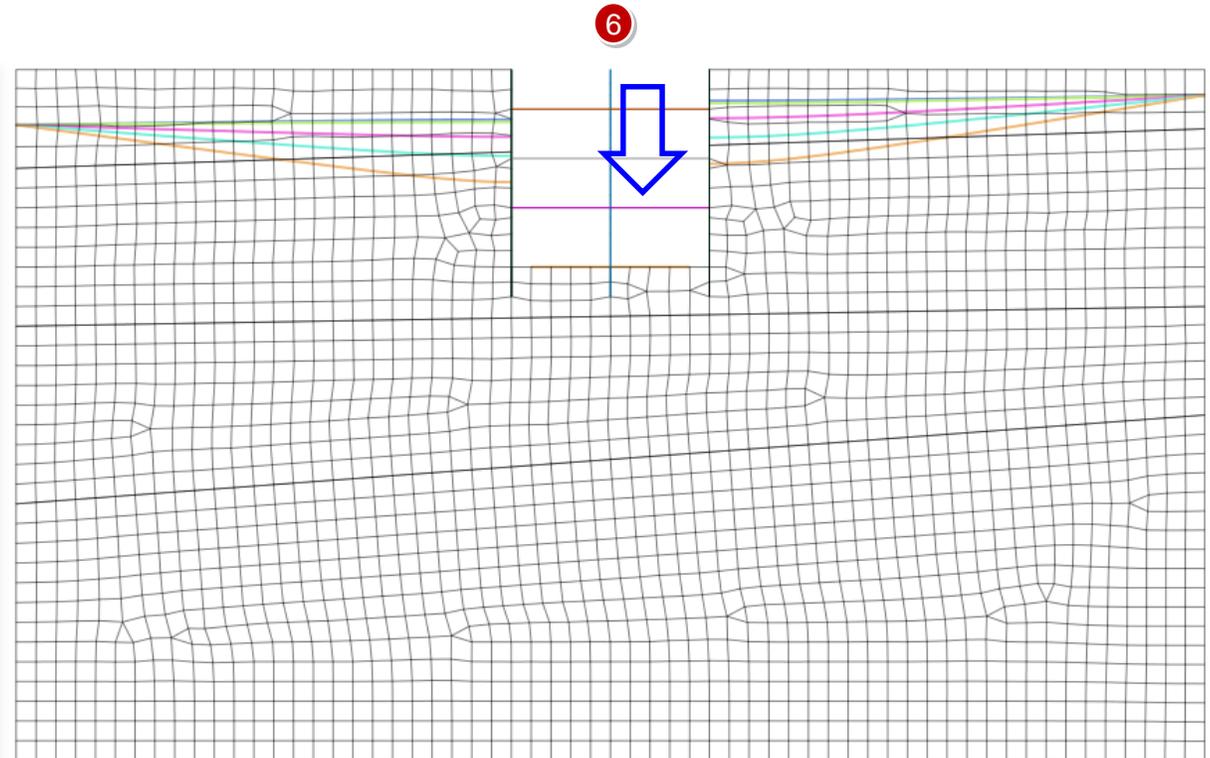
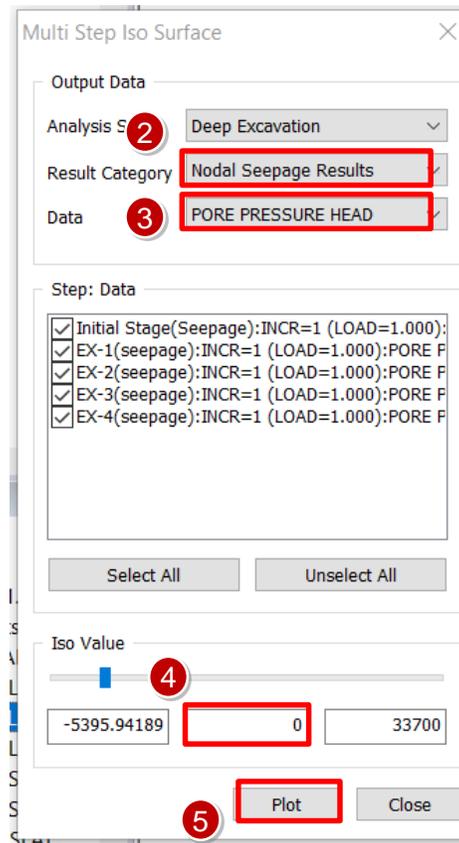
Normal 4th Excavation Stage



ULS 4th Excavation Stage

Procedure

1. Select **Result > Advanced > Multi Step Iso.**
2. Select the **'Nodal Seepage Results'** from Result Category.
3. Select **'PORE PRESSURE HEAD'**.
4. And, give **'0'** value
5. Plot.
6. Look at the water drawdown stage by stage.



Happy Modeling