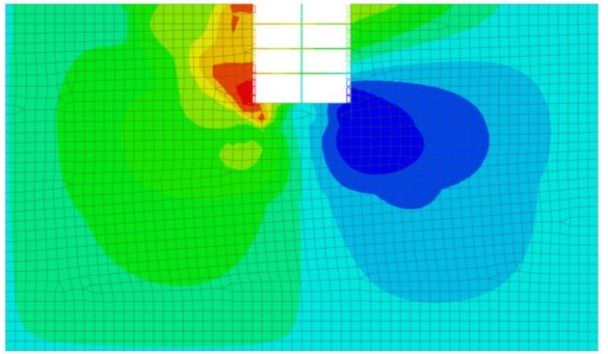
MIDAS *Technical Material*

Tutorial



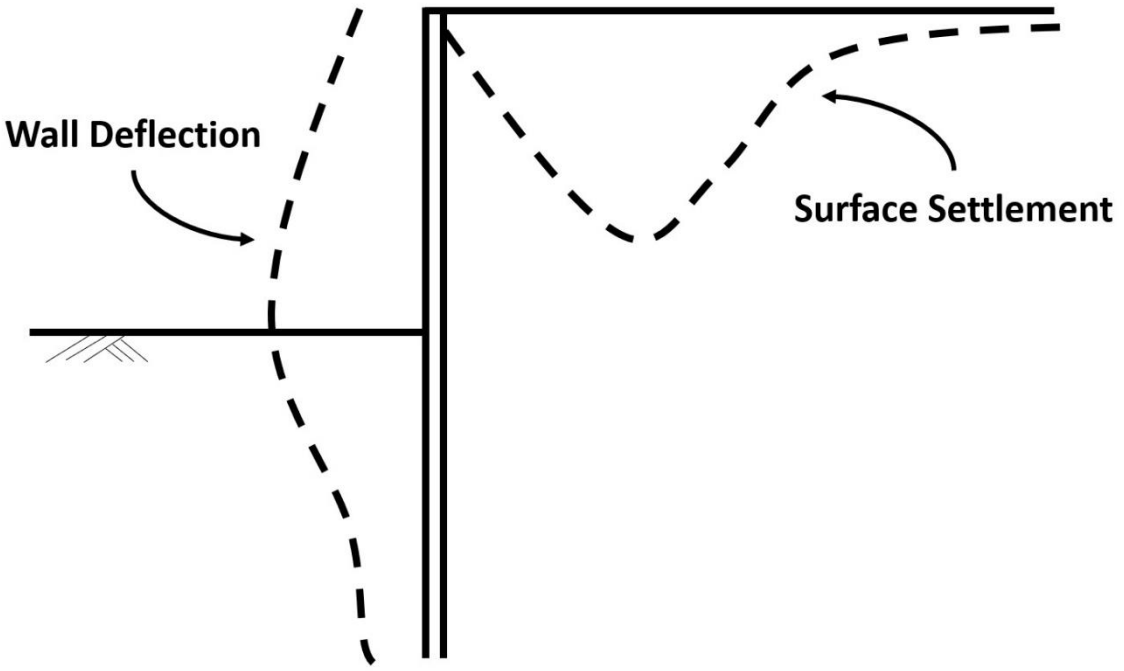
Deep Excavation Analysis

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



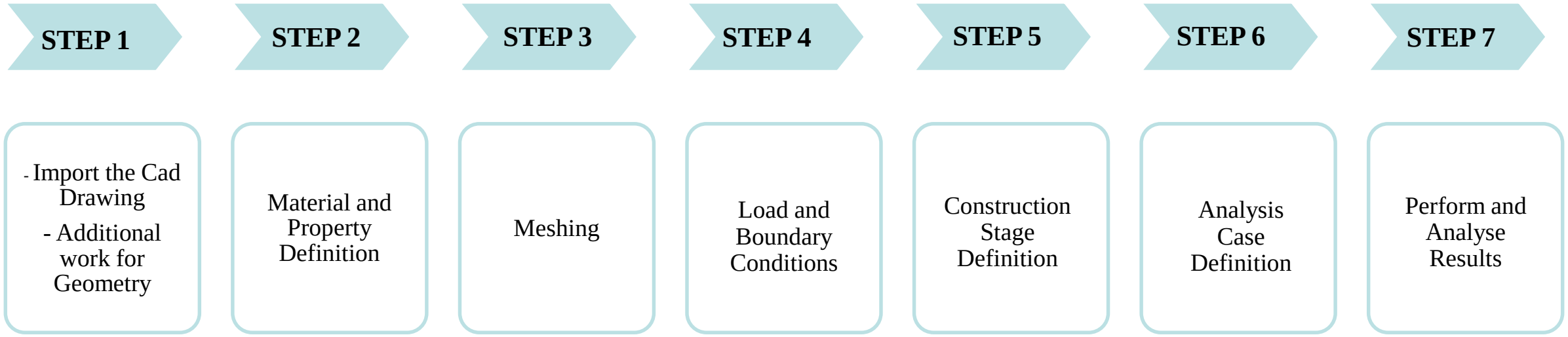
Expected Results

Learning Objective

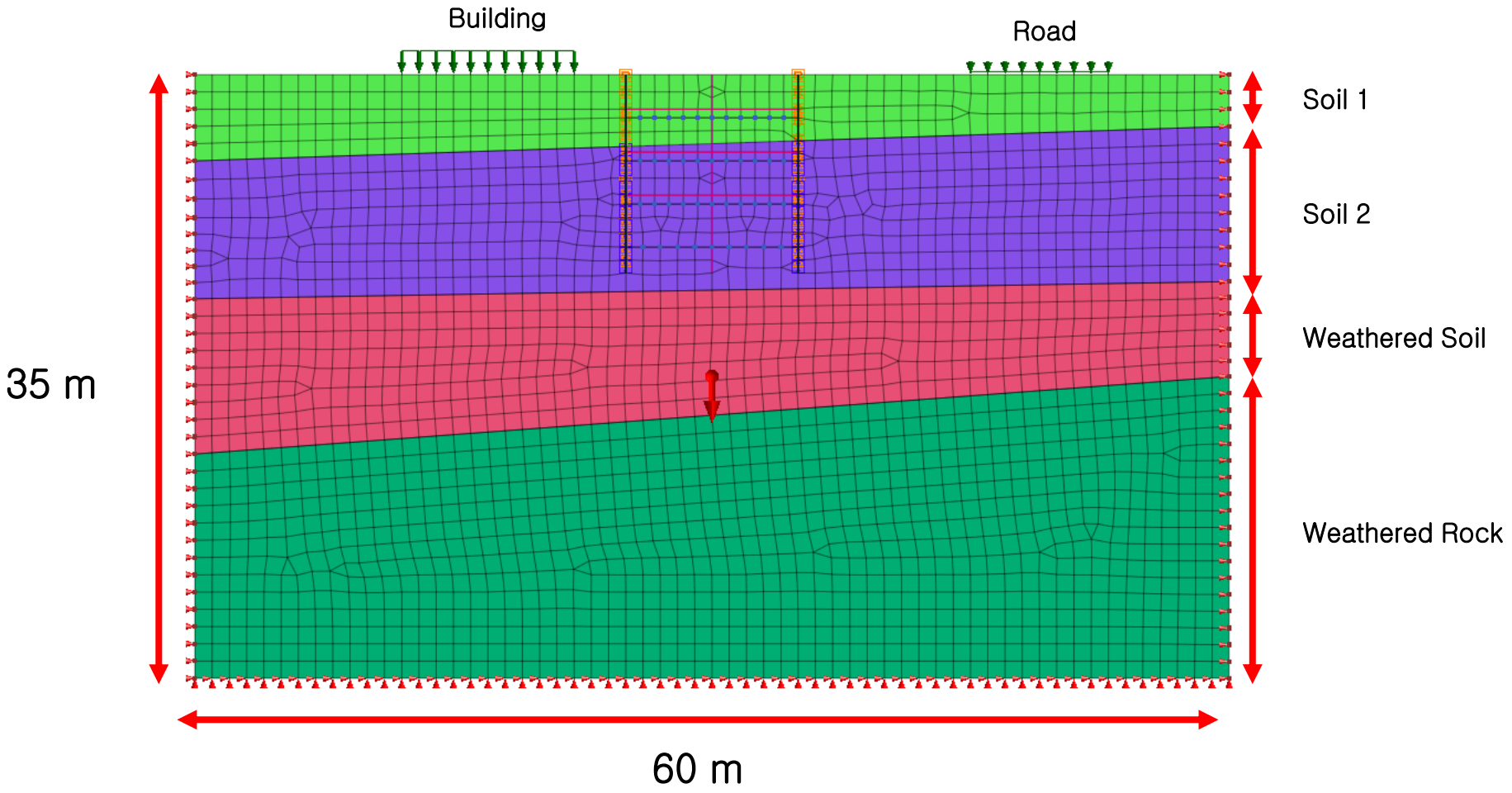
Overview

- 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

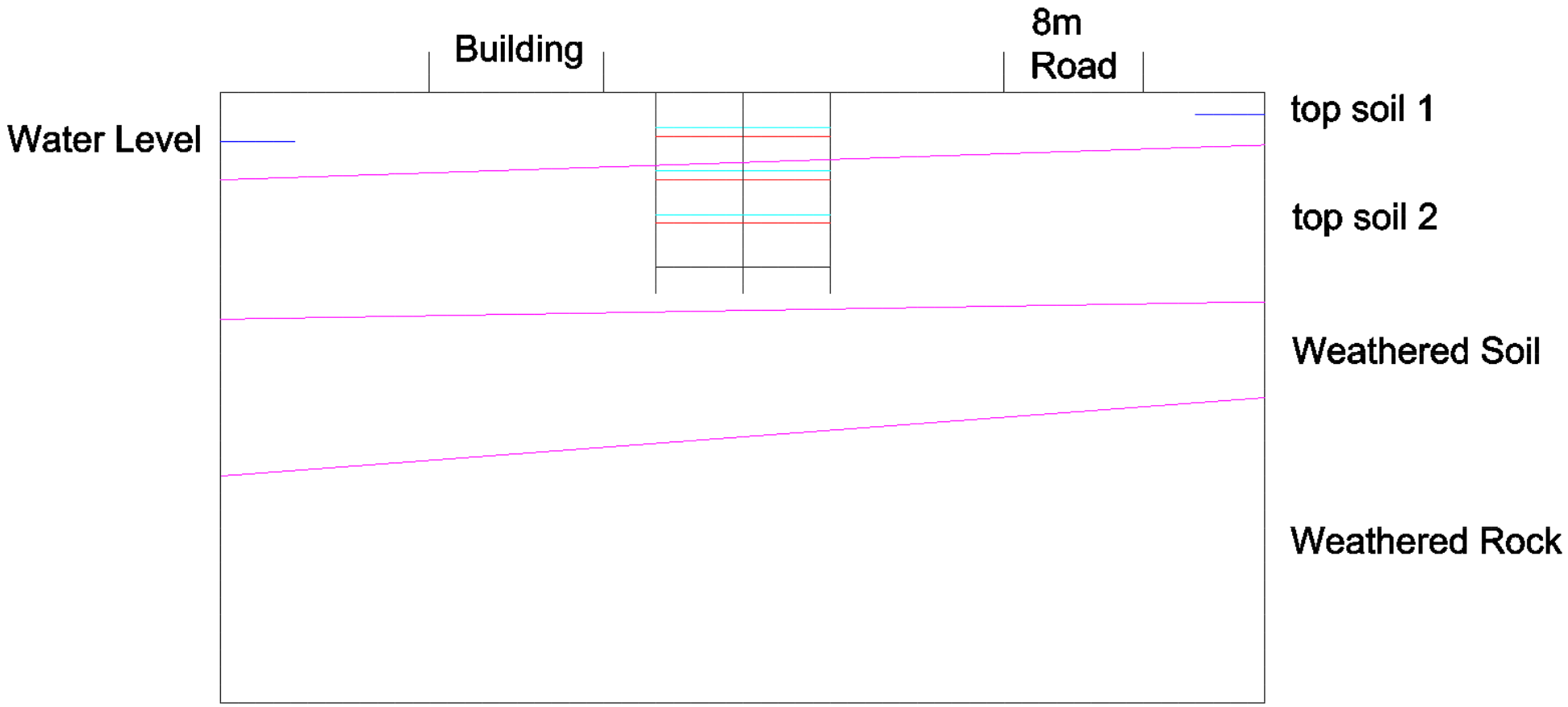


Building Model and Subsurface Layers



2D Section

Shoring Design



2D Section

Material Properties

Material Properties

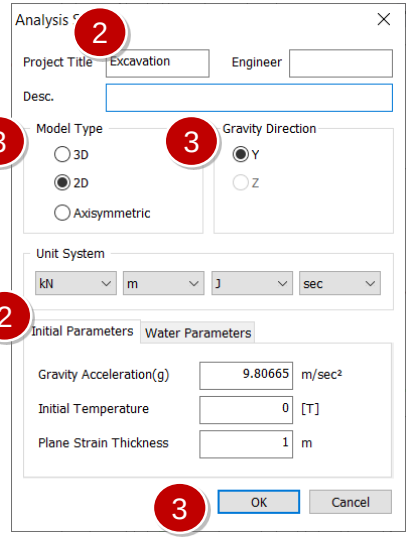
Material	Model Type	Elastic Modulus (kN/m^2)	Poisson's Ratio	Unit Weight (kN/m^3)	Saturated Unit Weight (kN/m^3)	Initial Void Ratio	Cohesion (kN/m^2)	Friction angle (degrees)	Permeability (m/sec)
Soil 1	Isotropic Mohr-Coulomb	17,000	0.35	17	18	0.5	7	25	9.78e-6
Soil 2	Isotropic Mohr-Coulomb	20,000	0.35	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	-	-	-	-	-

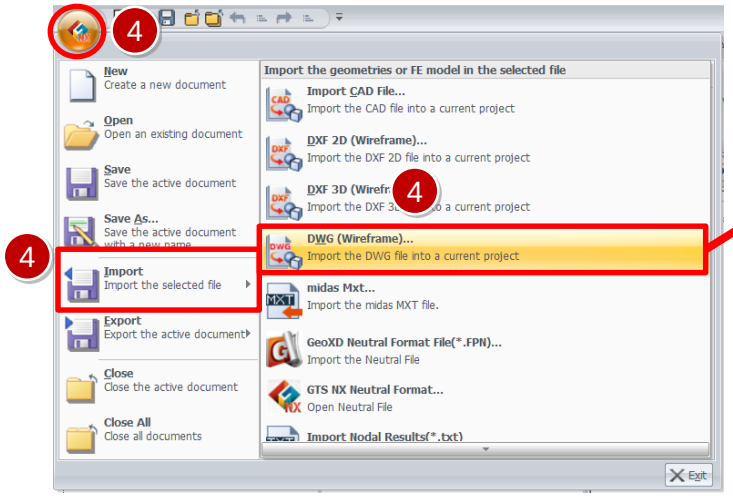
Step **01**

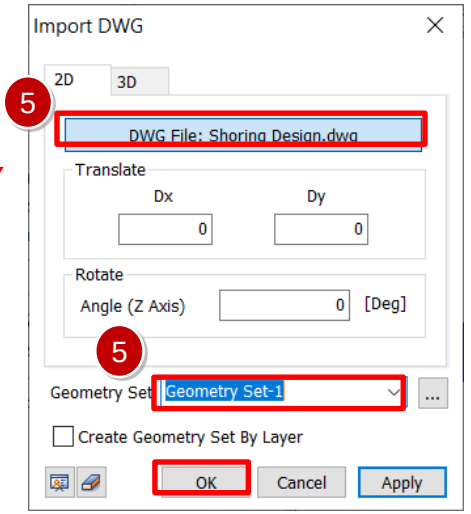
1-1 Import CAD drawing and Divide the Edges

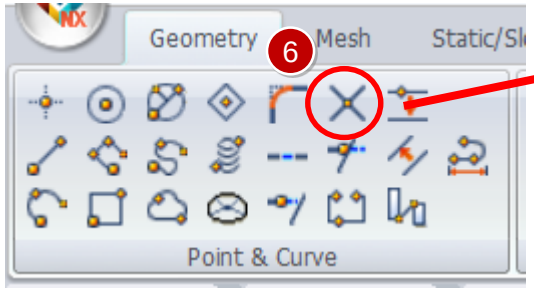
Procedure

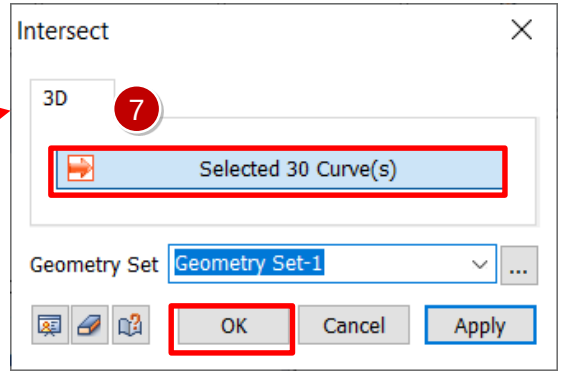
1. Open GTS NX.
2. Project Title: enter “**Excavation**” , set units as **kN, m and sec**
3. Select Model type **2D** and Gravity **dir ection Y** and click **‘OK’**
4. Click on GTS NX icon on the top left corner of Interface and select **Import > DWG.**
5. Select **“Select AutoCAD DWG File ”** and select “**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’.**











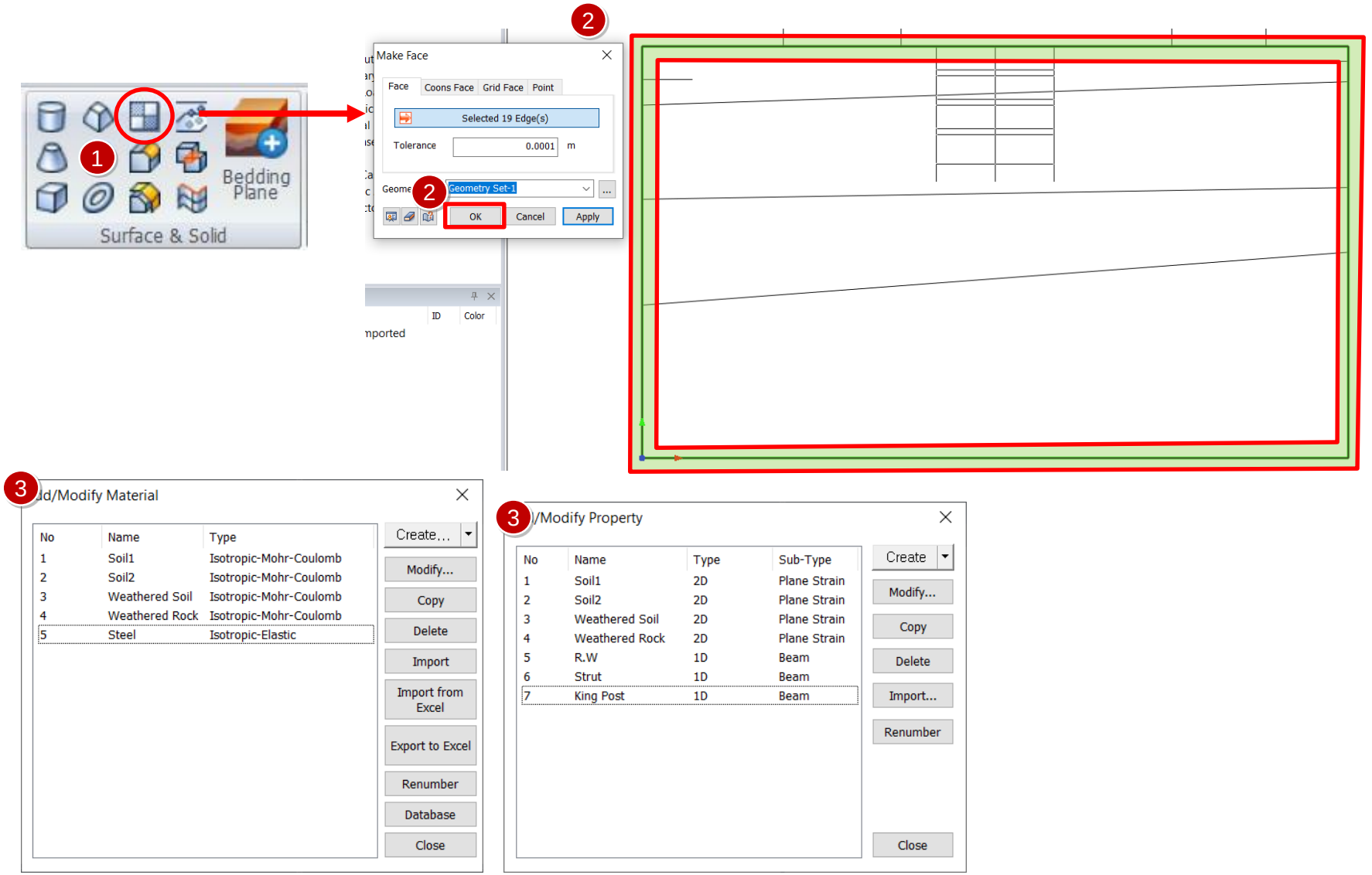
Step
01

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.*



Step 02

2-0 Material Definition

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**

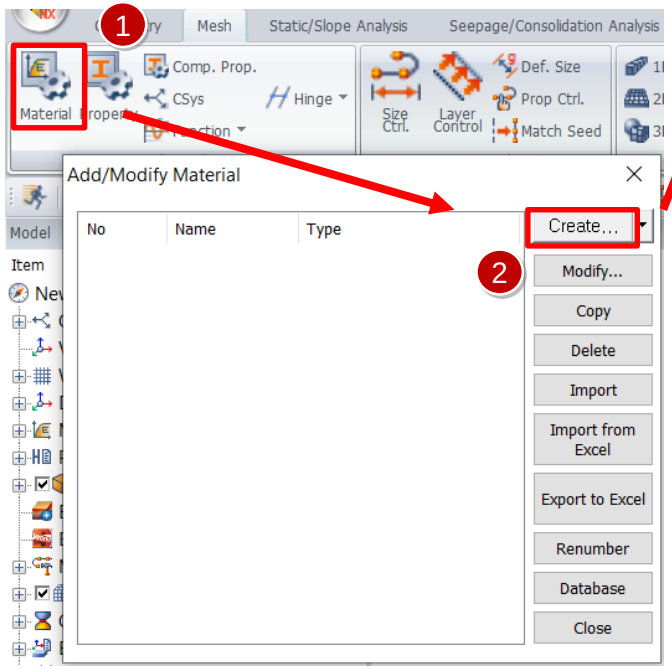
   Initial Void Ratio > **0.5**

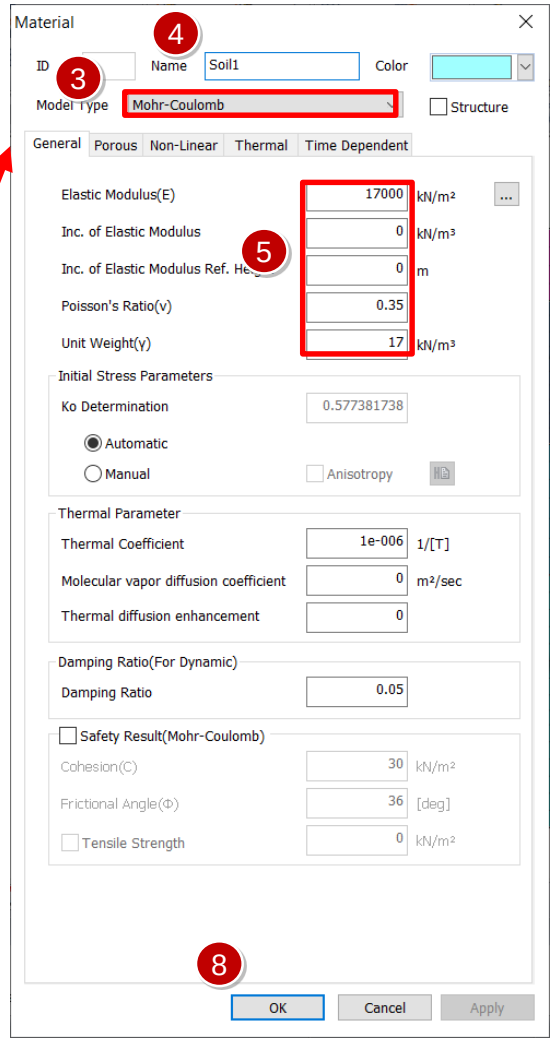
   Permeability > **9.78e-6 m/sec**
7. In **Non-Linear tab**

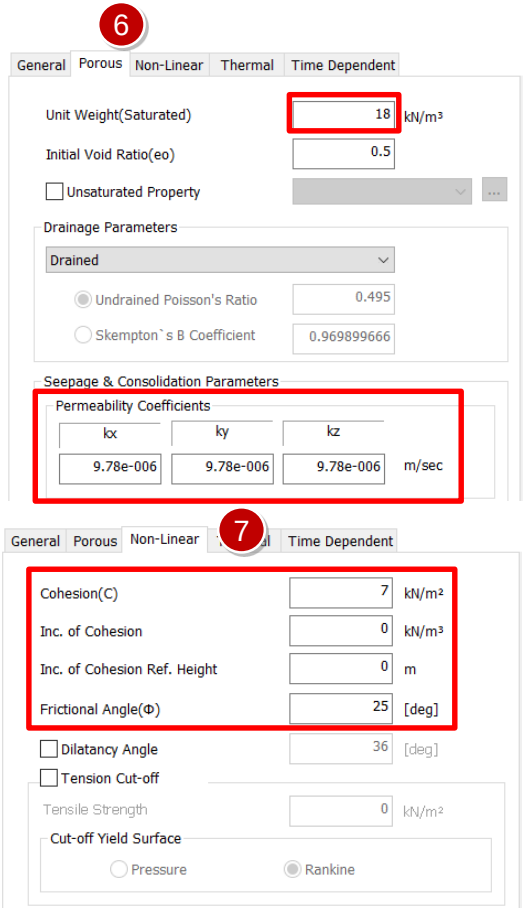
   cohesion > **7 kN/ m³**

   Friction angle > **25 degrees**
8. Click **'OK'**

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







2-0 Property Definition

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'**

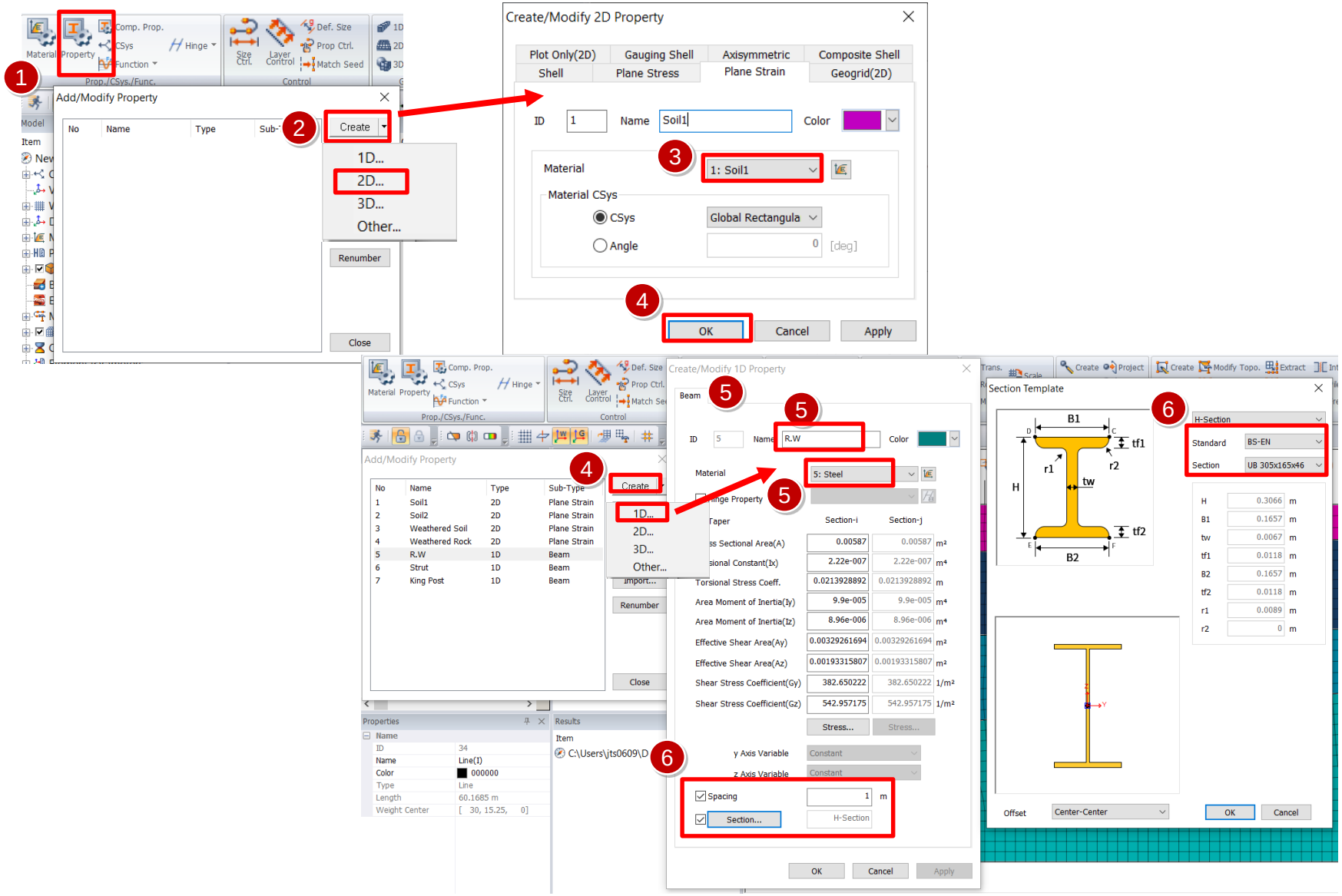
Repeatedly, Define all the Soil Properties following the 5th page.

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



Step
01

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

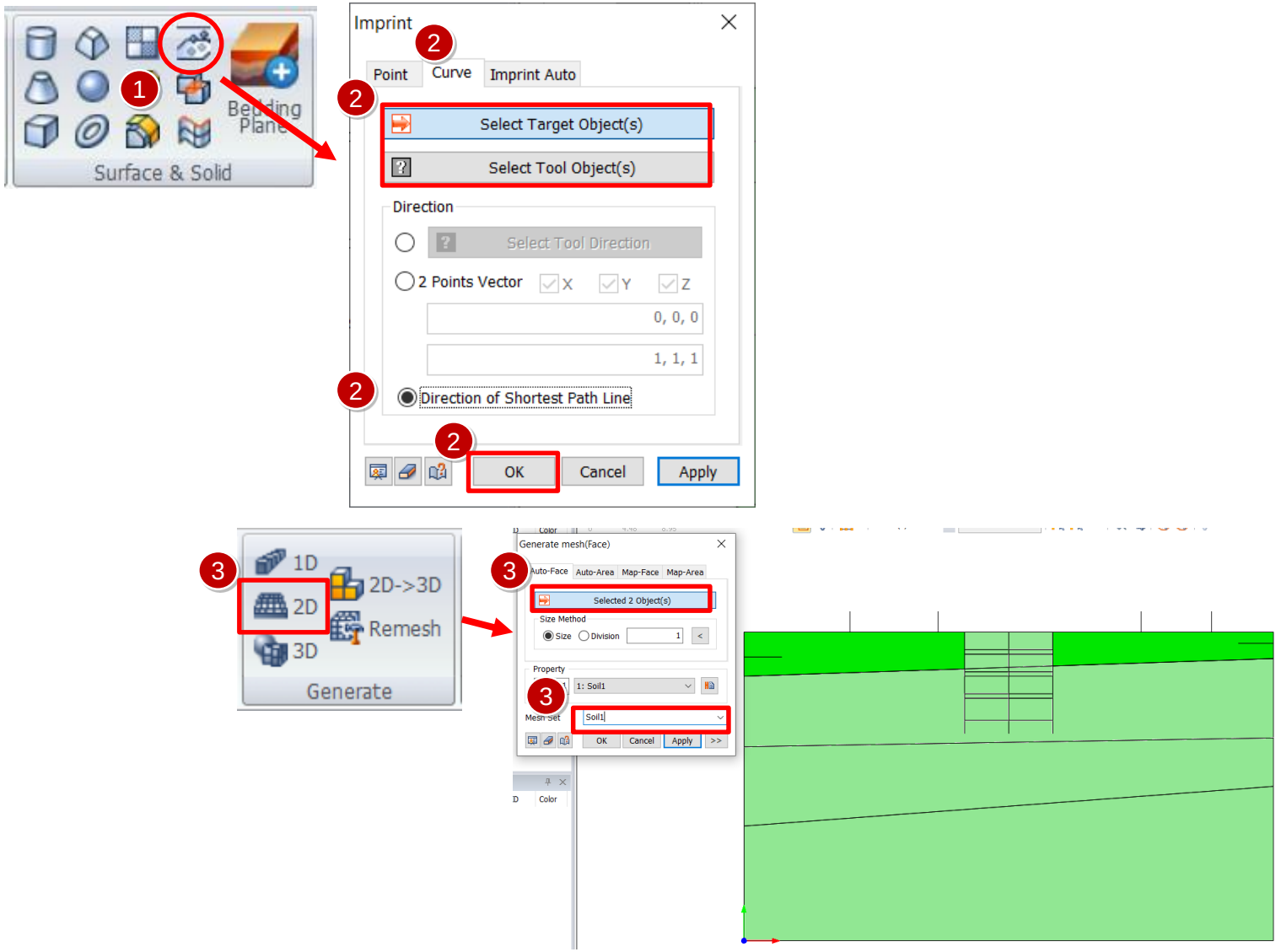
Procedure

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.



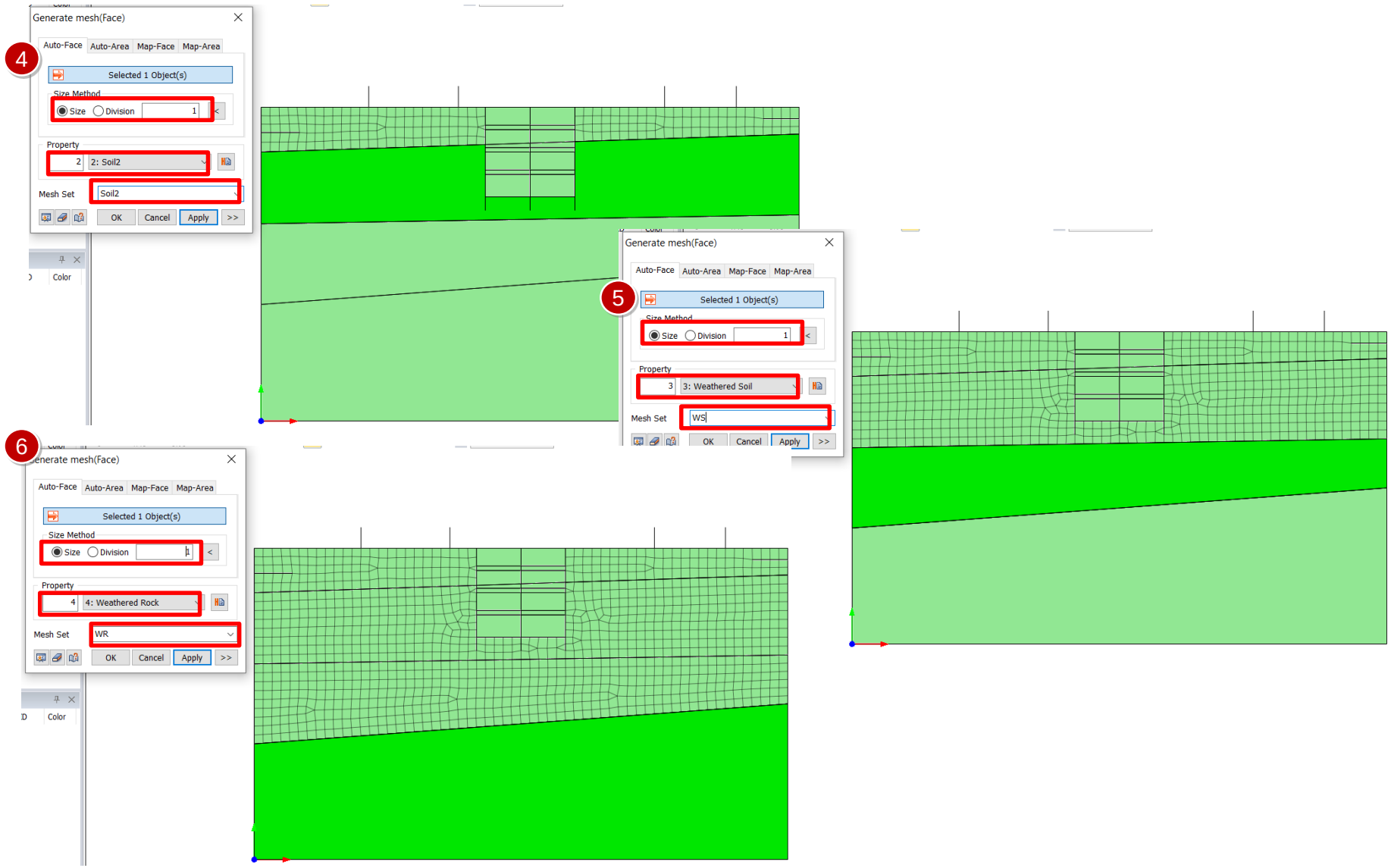
Step
02

2-1 Generate Mesh Set – Soil Elements

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.



Step
02

2-1 Generate Mesh Set – Soil Elements

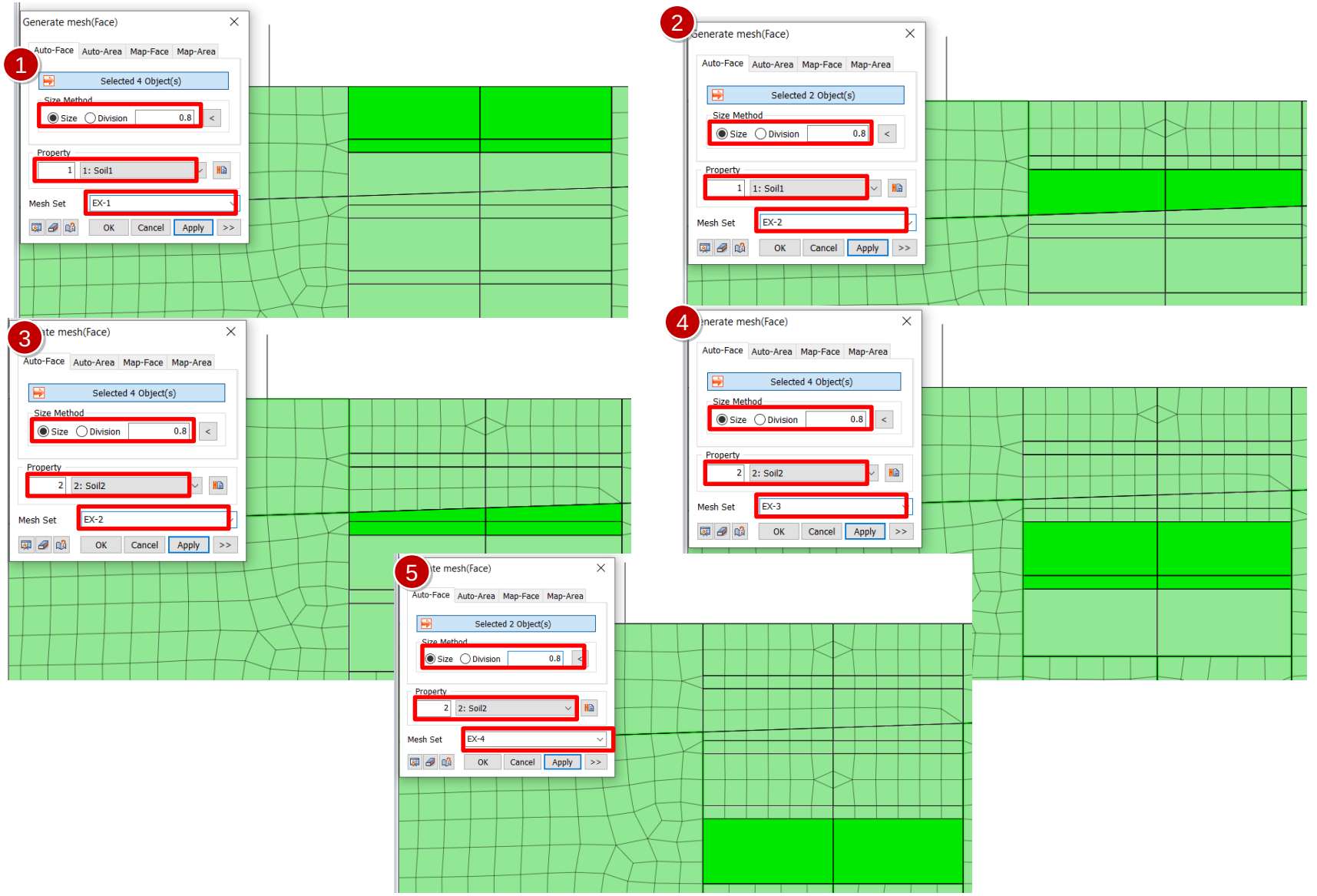
Procedure

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.

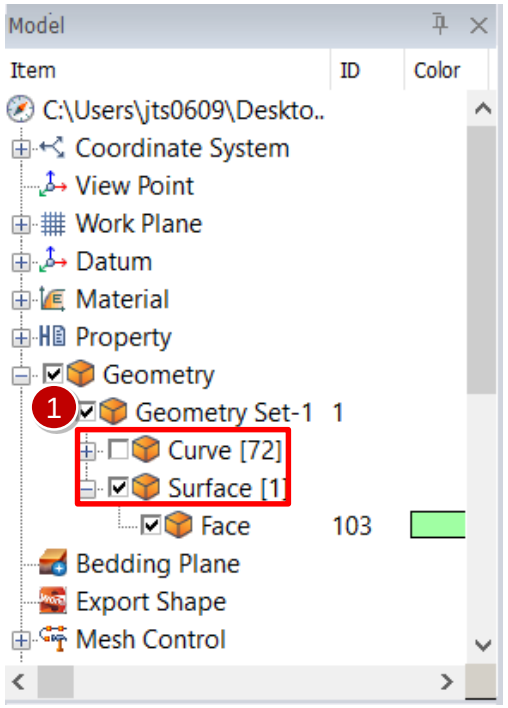


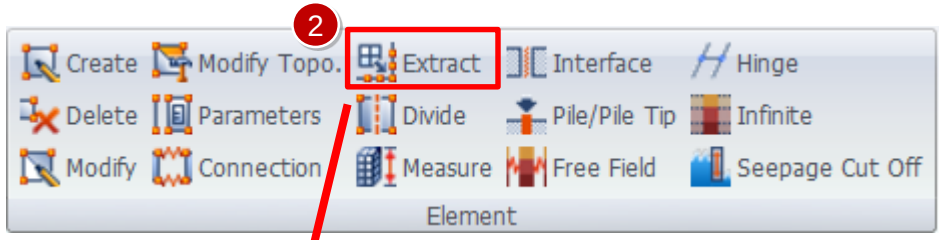
2-2 Generate Mesh Set – Structural Elements

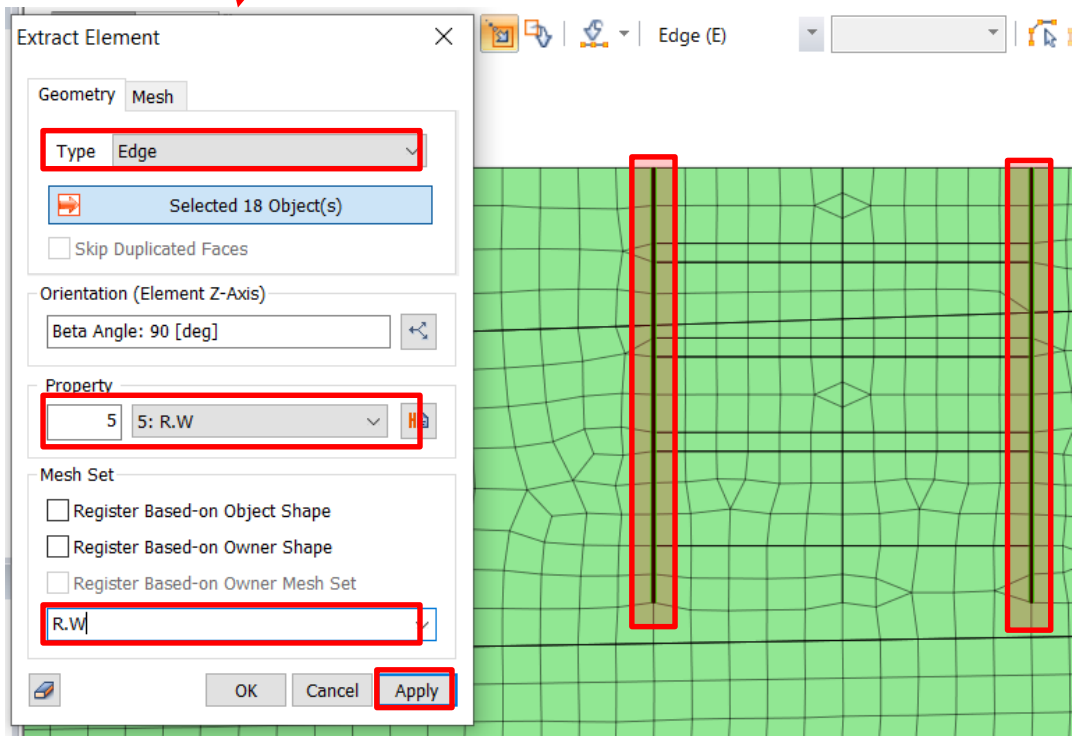
Procedure

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**'.







Step 02

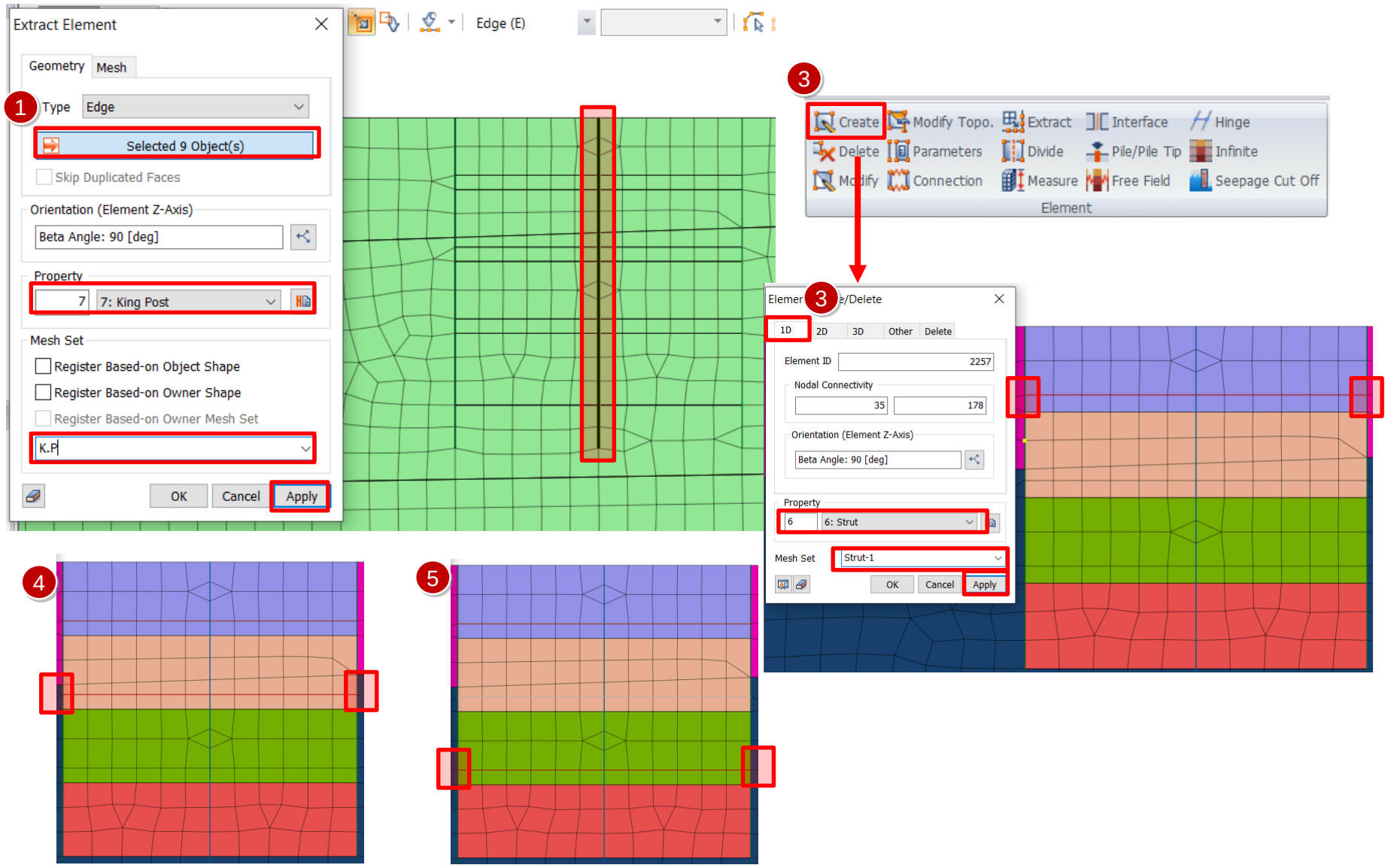
2-2 Generate Mesh Set – Structural Elements

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.,

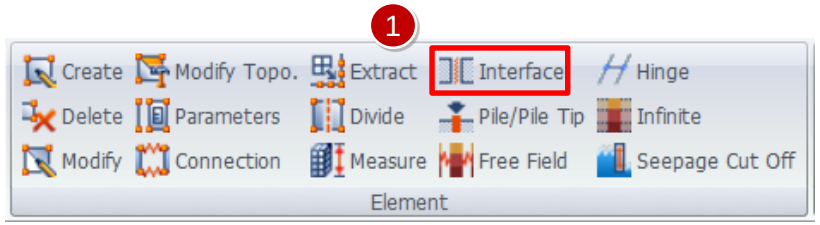
Make **4. Strut-2 and 5. Strut-3**

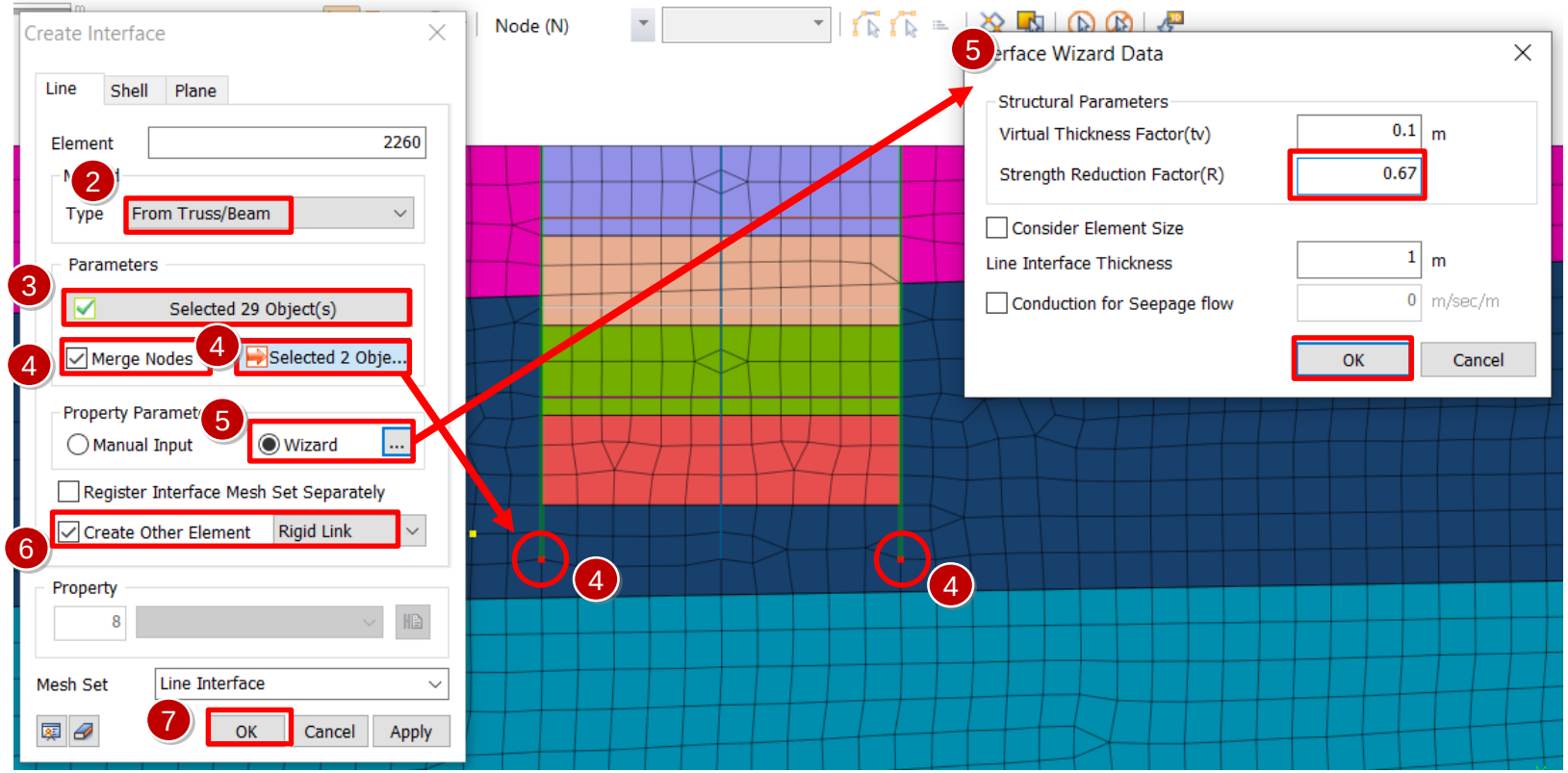


2-2 Generate Mesh Set – Interface Element

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**.



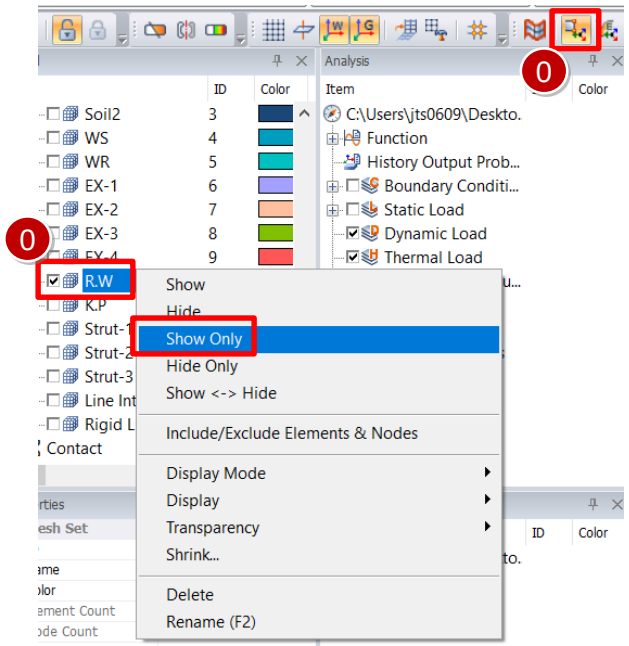


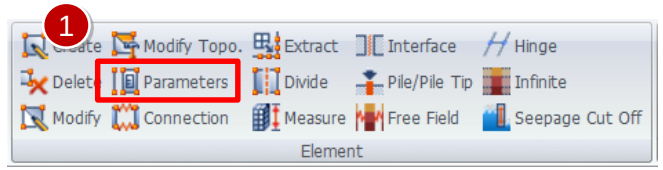
Step 02

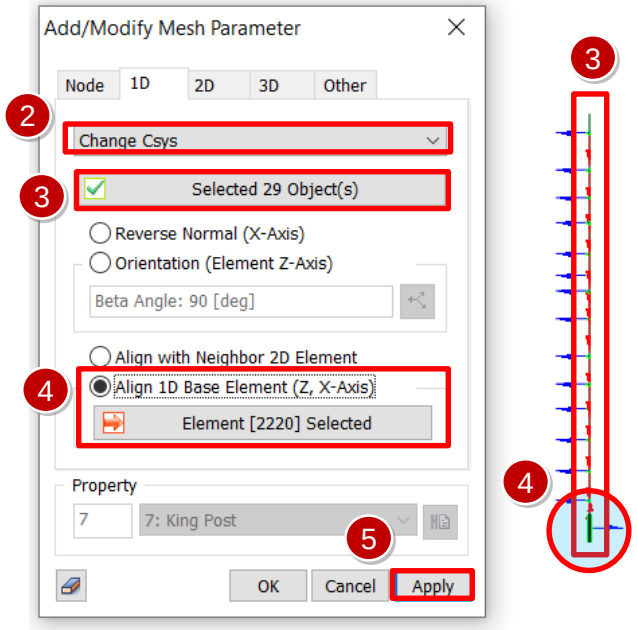
2-3 Generate Mesh Set – Align Element Coordinate System(Retaining Wall)

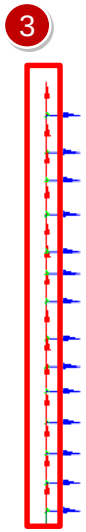
Procedure

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**.







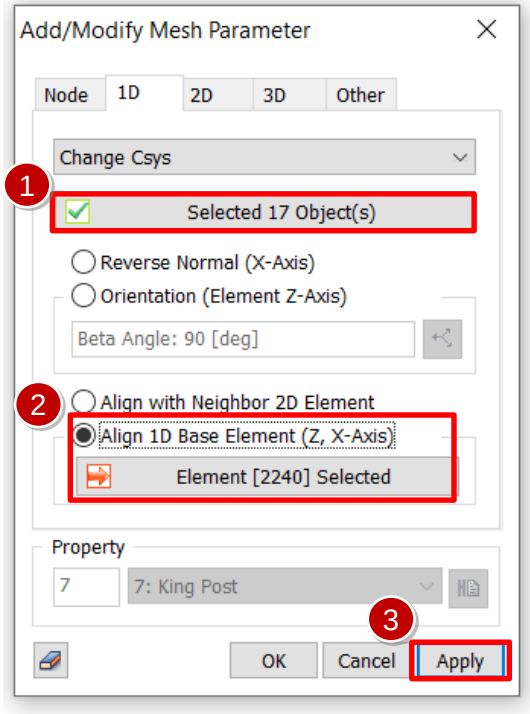


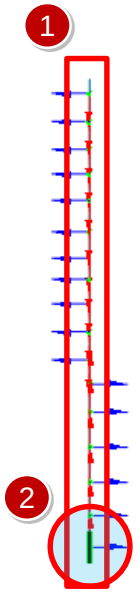
Step 02

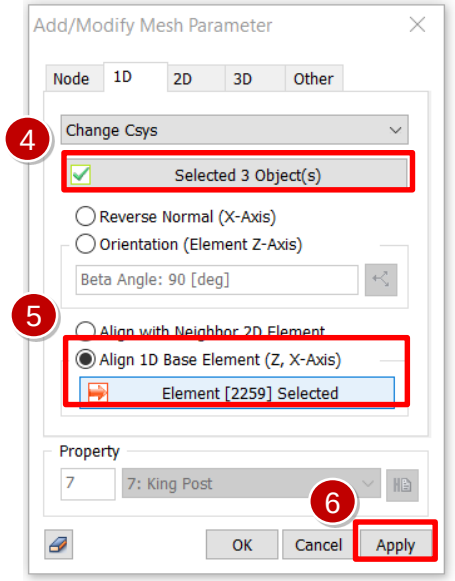
2-4 Generate Mesh Set – Align Element Coordinate System(King Post&Struts)

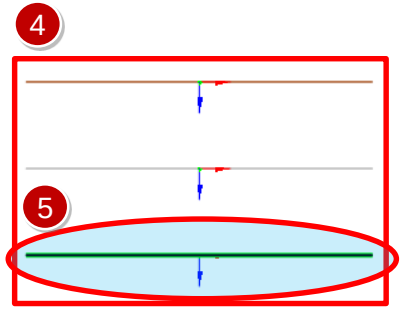
Procedure

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**.









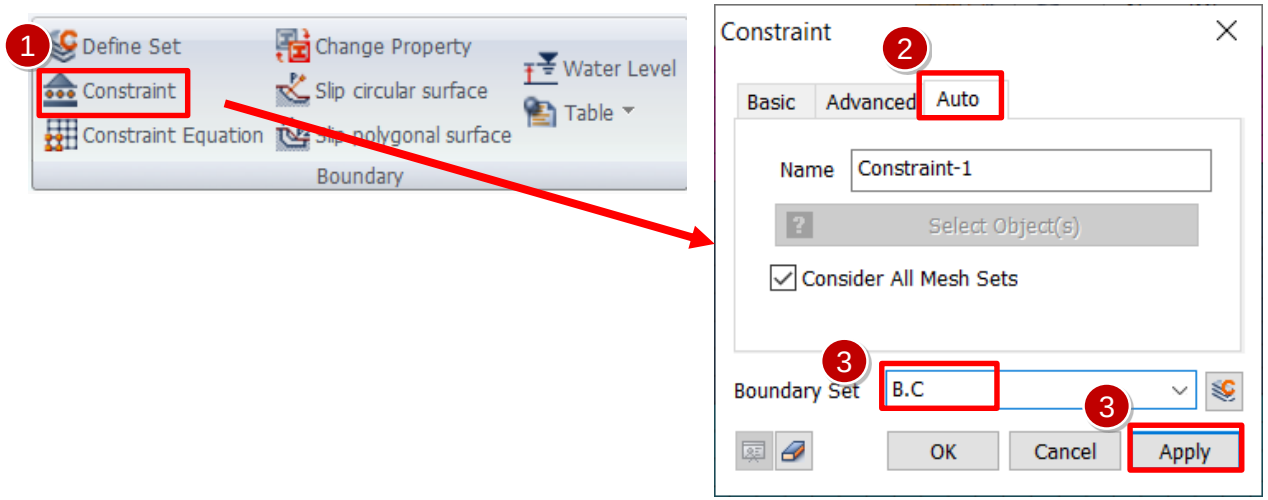
Step 03

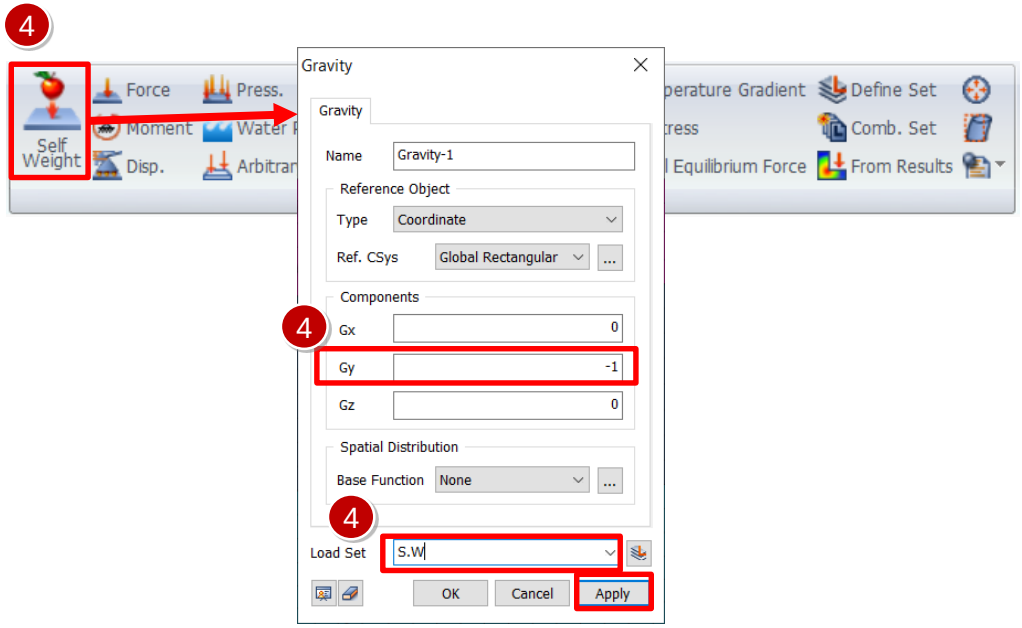
3-1 Assign Boundary/Load Condition(Constraint, Self Weight)

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**.



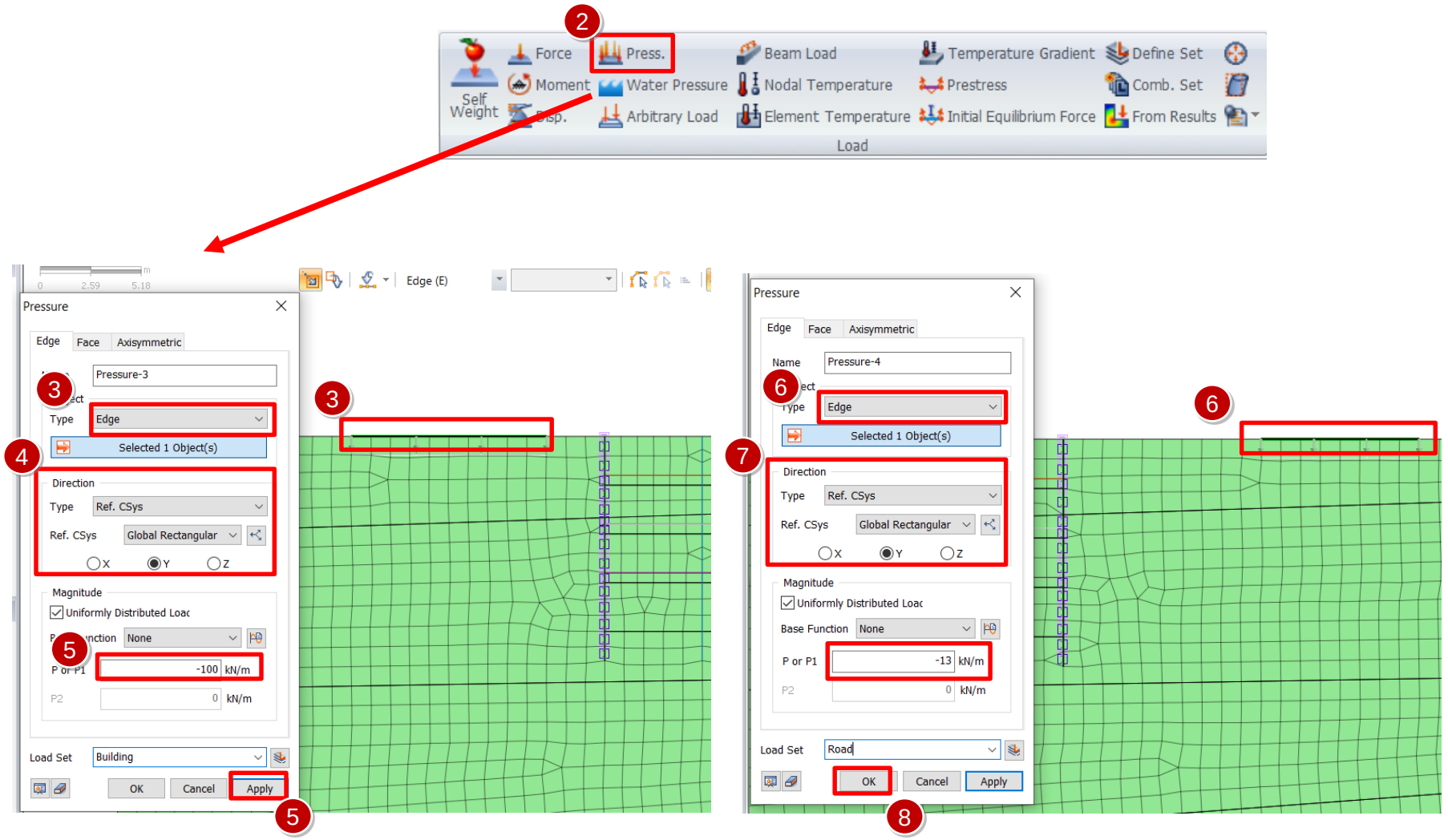


Step 03

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 **'-100**kN/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 **'-13**kN/m' and **Name as a Road** and **Apply.**

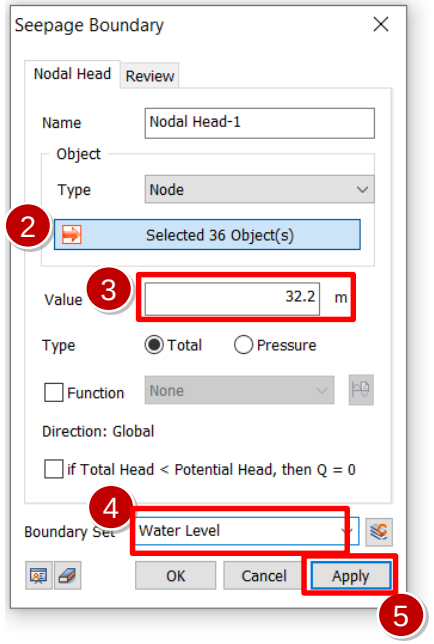


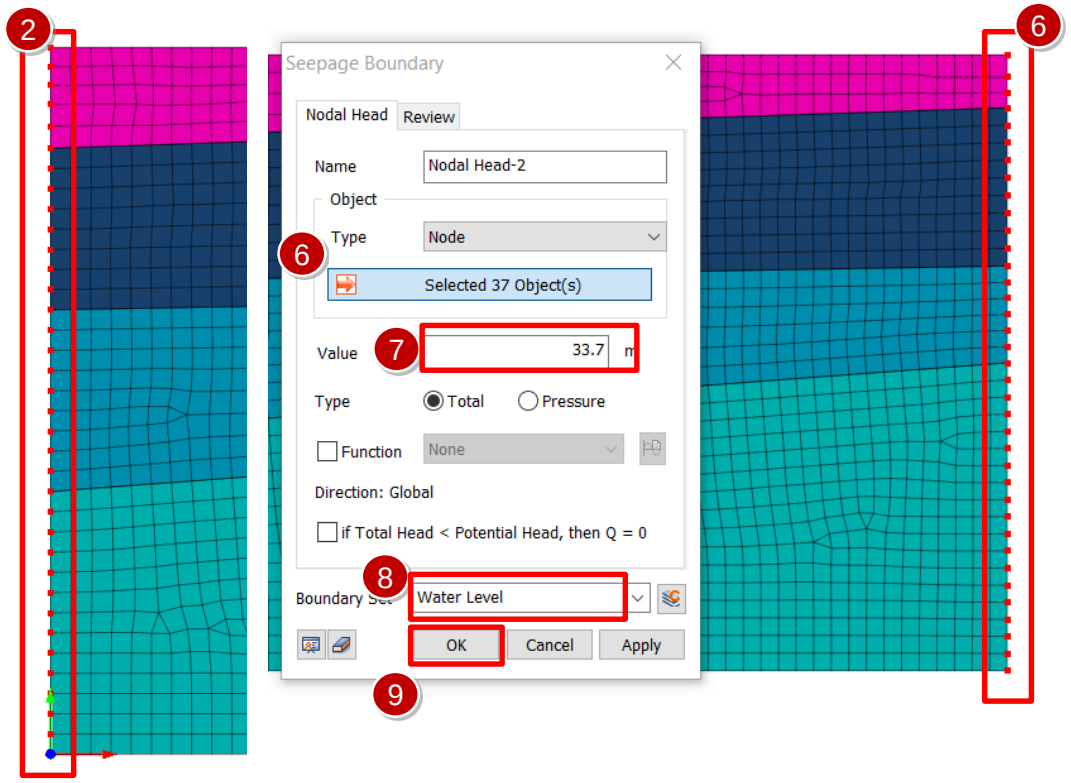
Step
03

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.



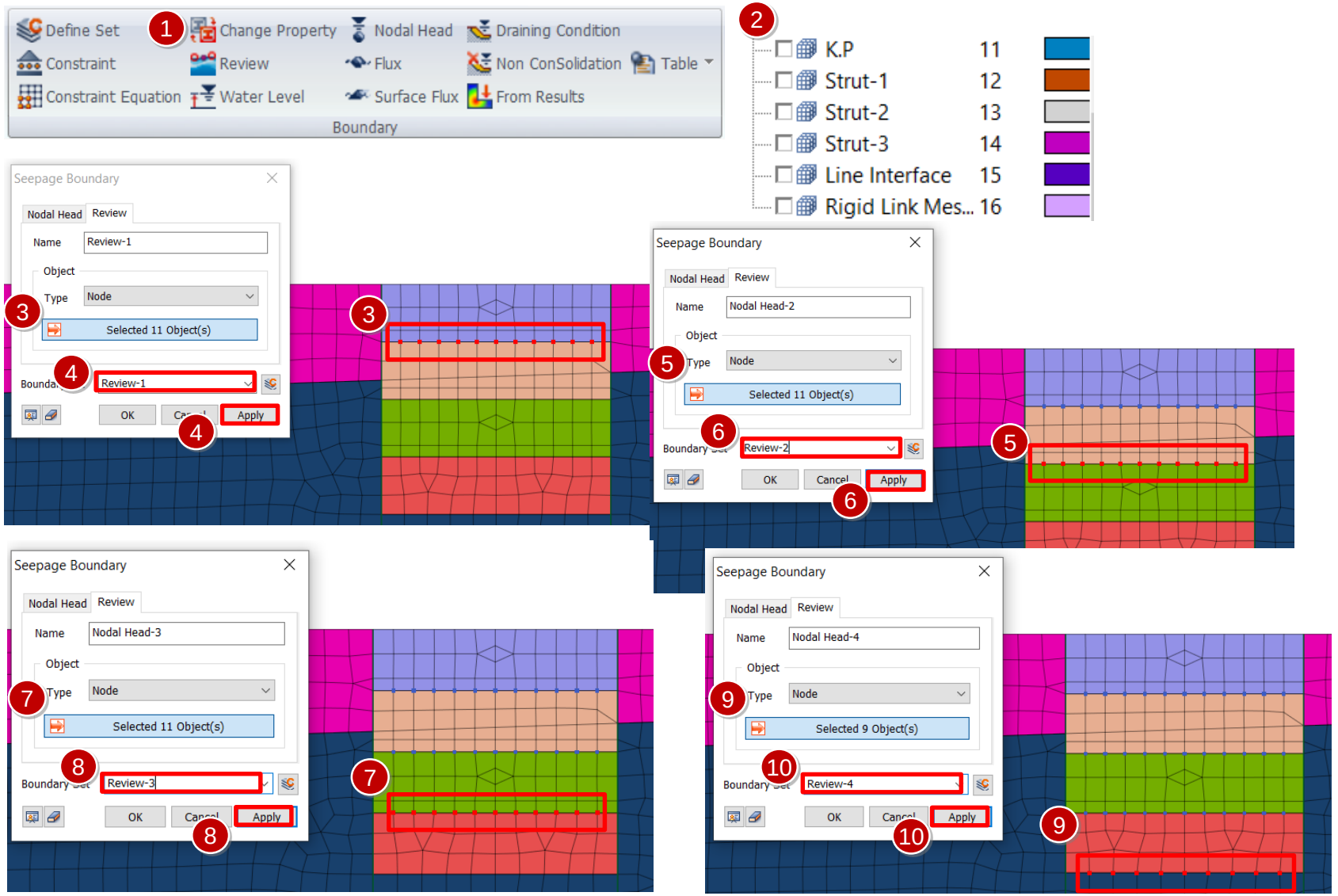


Step 03

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.

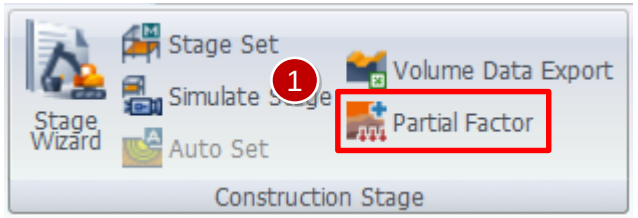


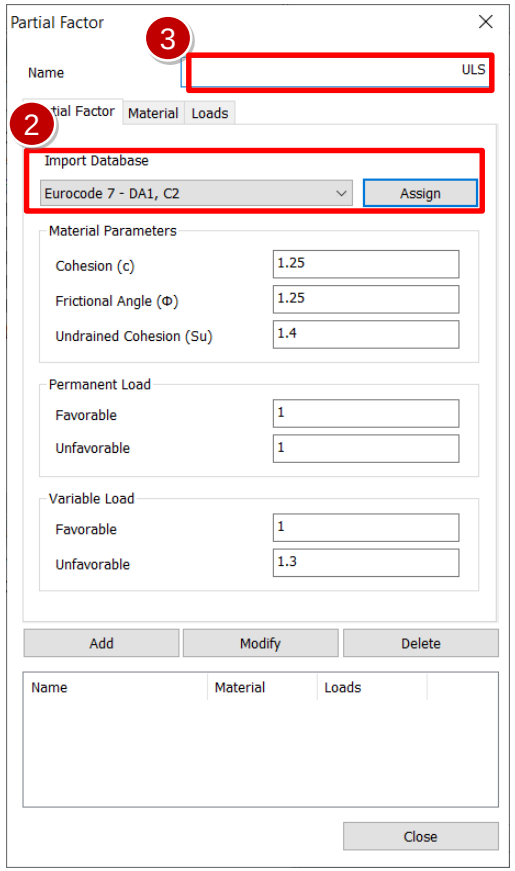
Step 03

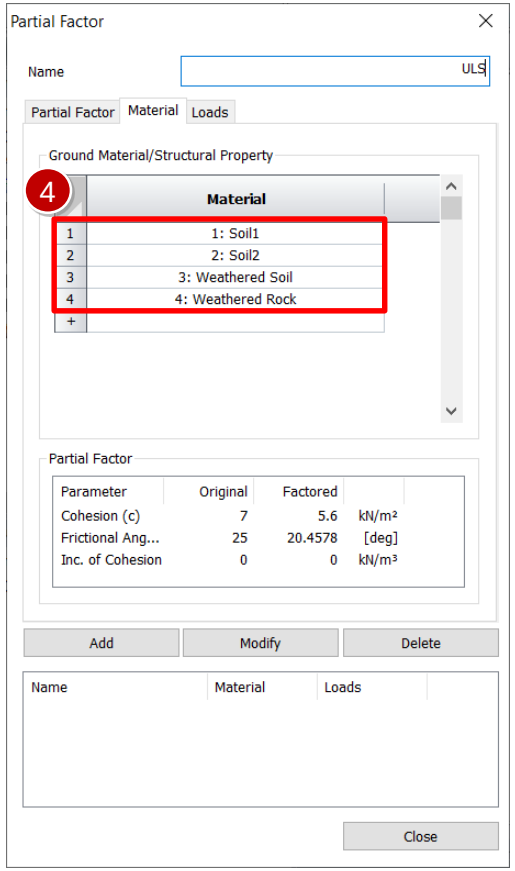
3-5 Assign Partial Factor

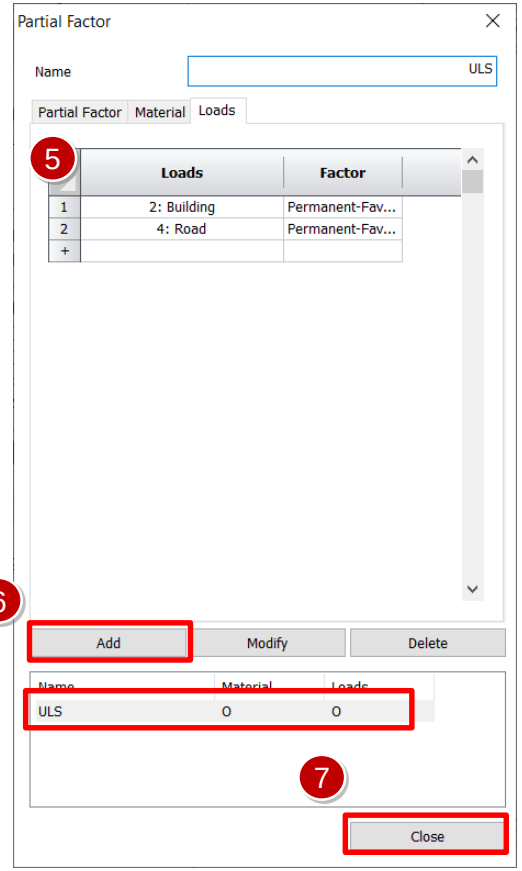
Procedure

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.







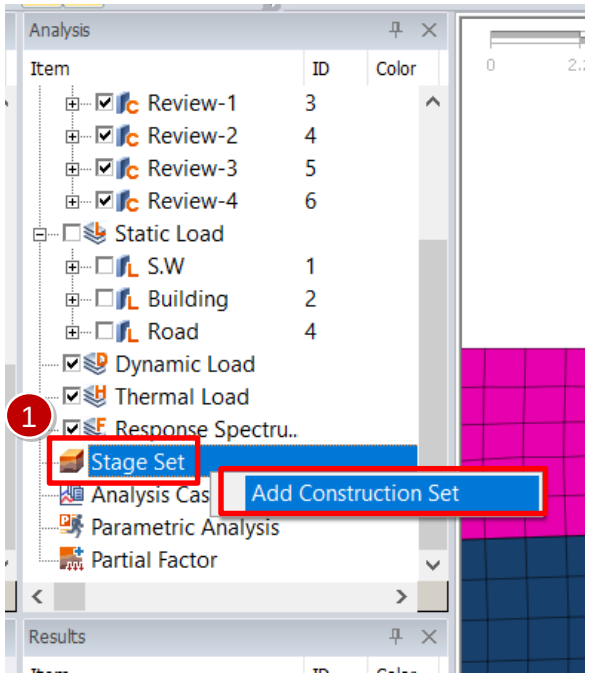


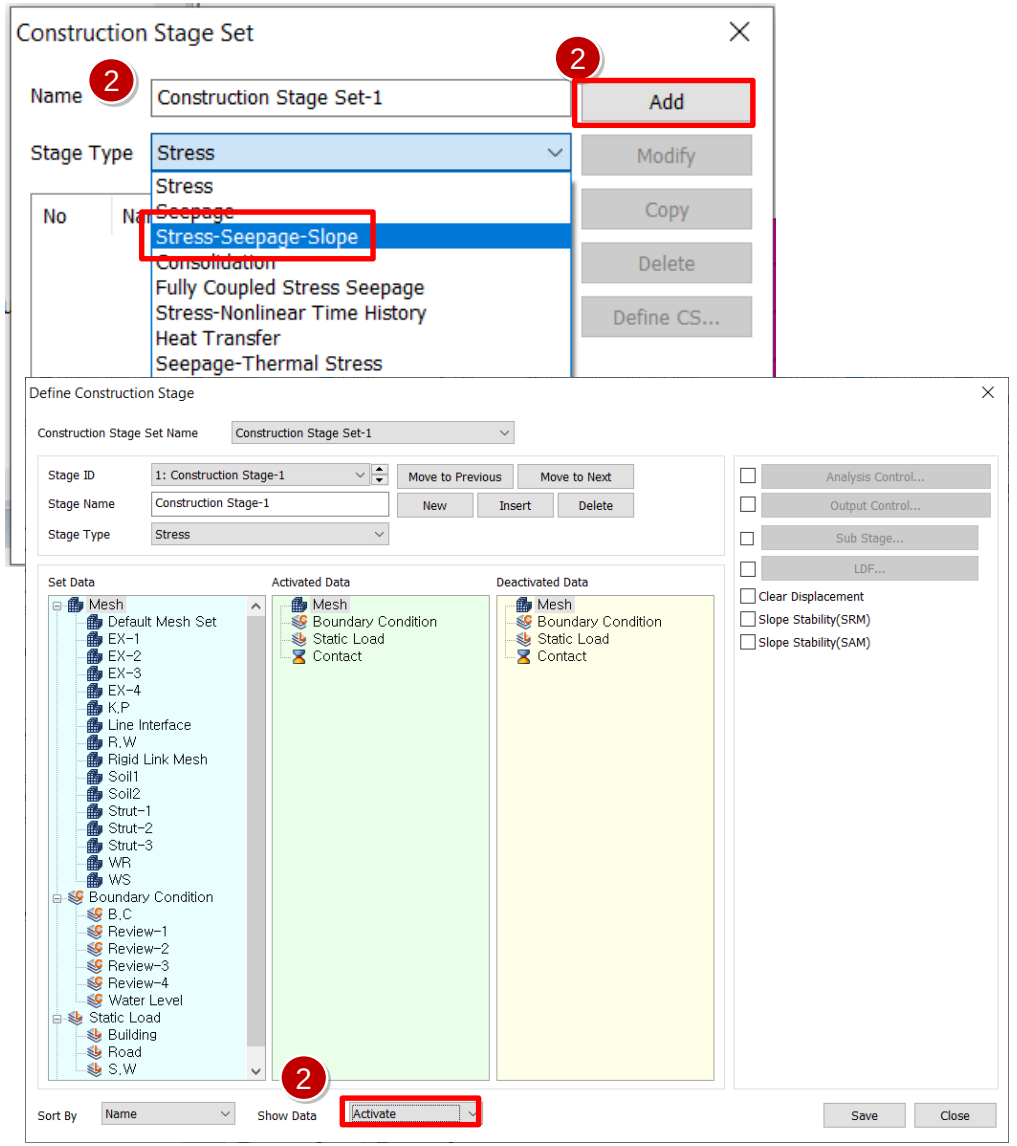
Step 04

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**'.



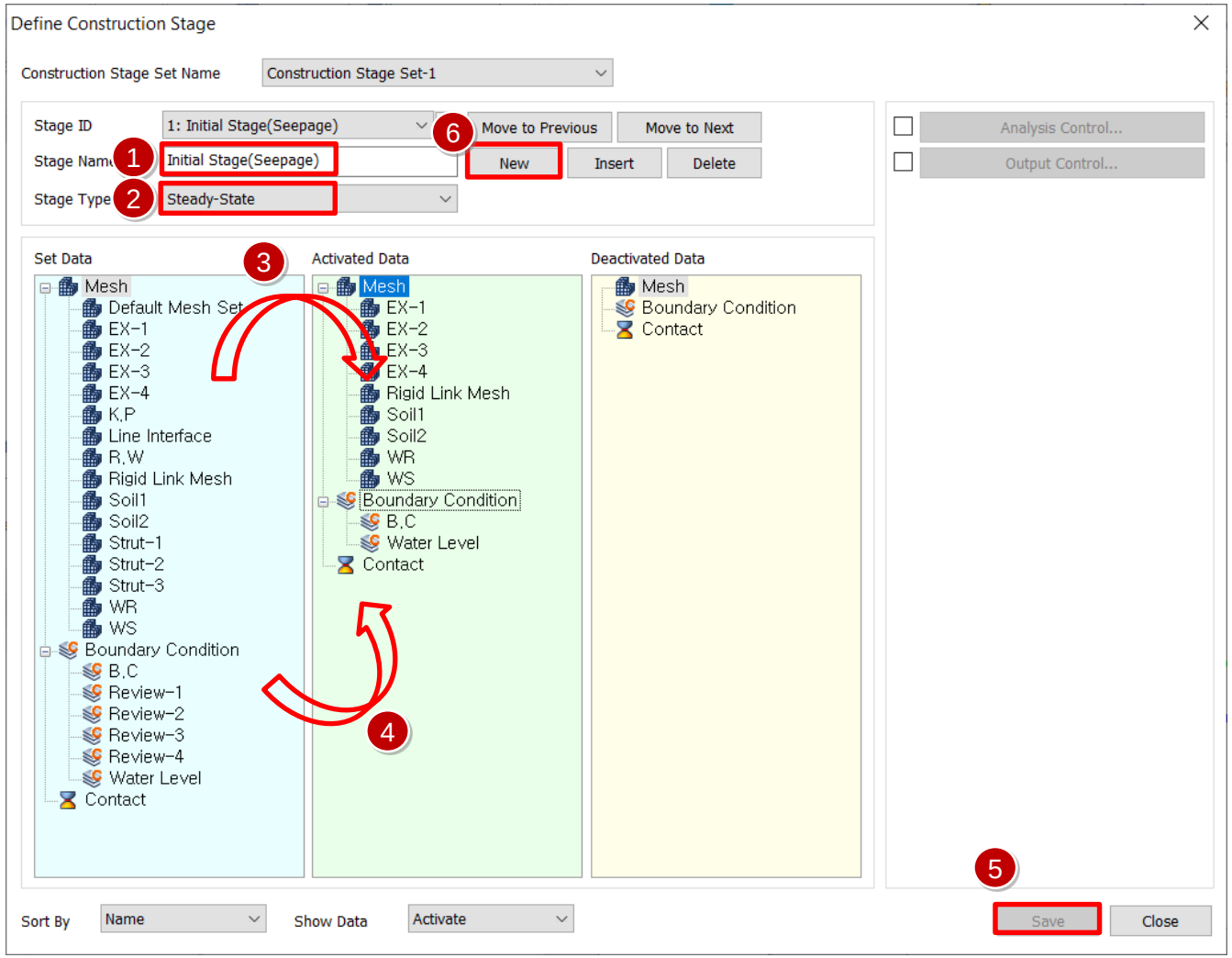


Step 04

4-1 Define Construction Stage

Procedure

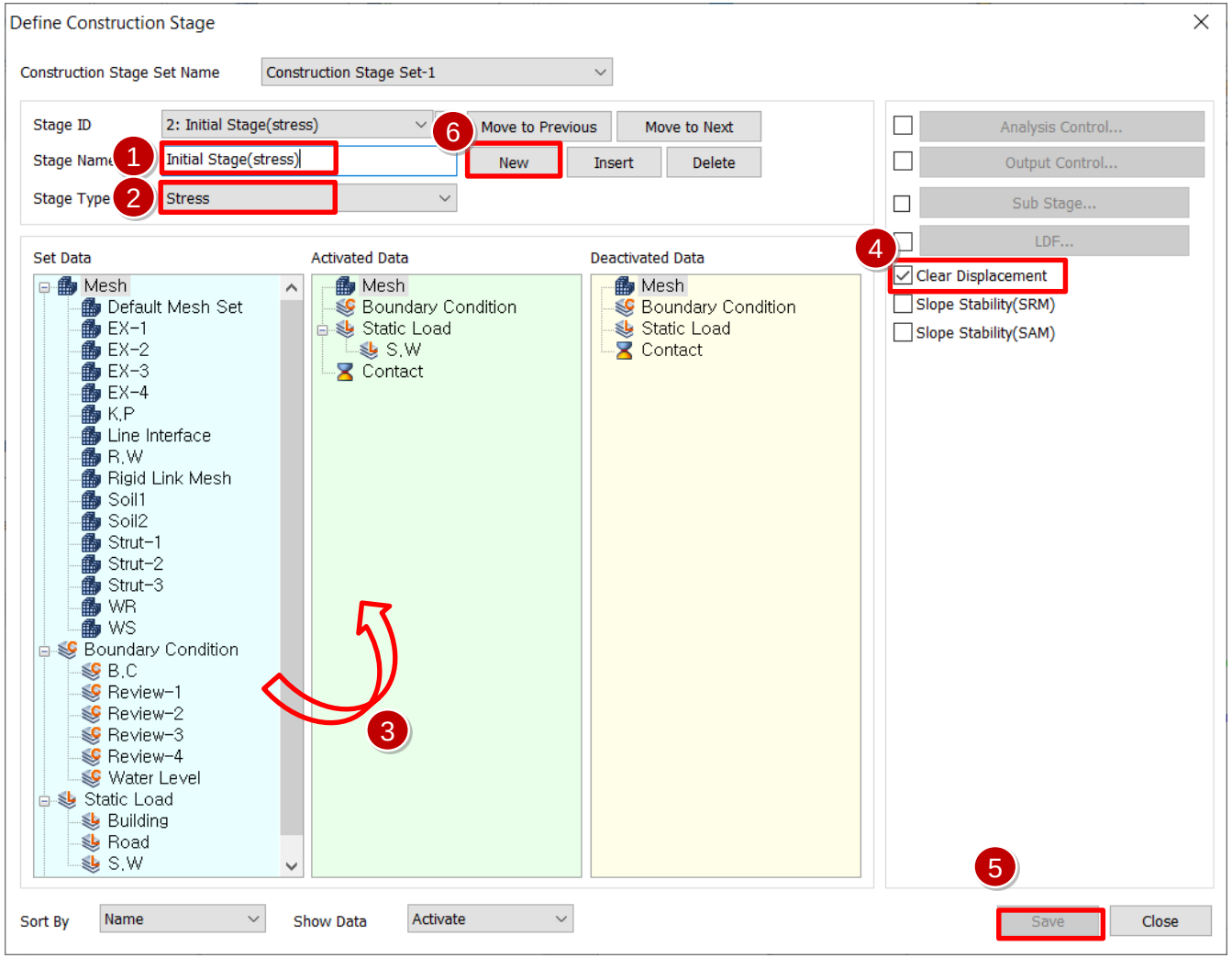
1. Give Name as a '**Initial stage(seepage)**'.
2. Change to '**Steady-State**'.
3. Activate the elements sets(**EX-1, EX-2, EX-3, EX-4, Rigid Link, Soil1, Soil2, WS, WR**) by using Drag&Drop from Set Data to Activated Data.
4. Activate the boundary conditions(**Boundary Condition, Water Level**) by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



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**'.

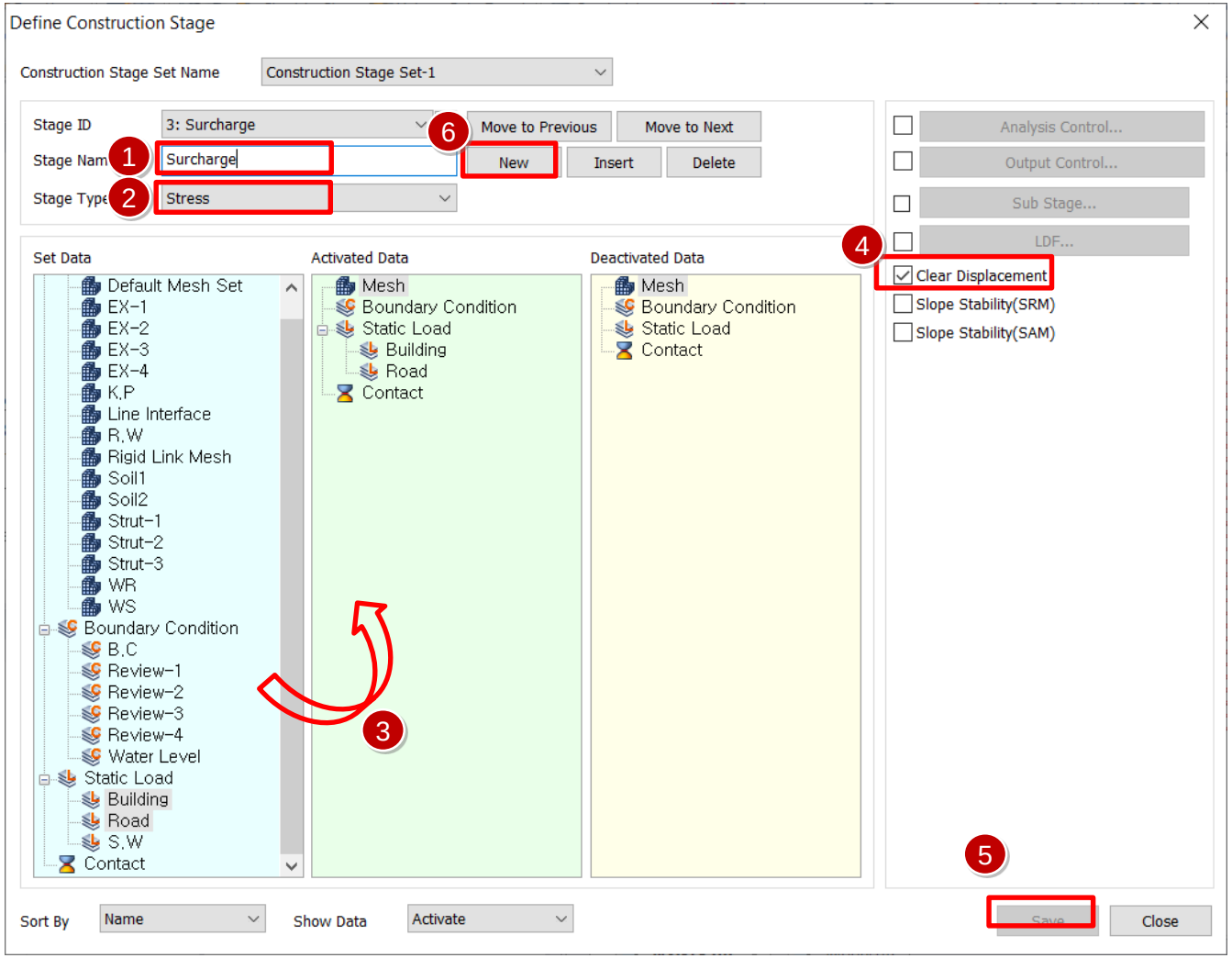


Step 04

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**'.

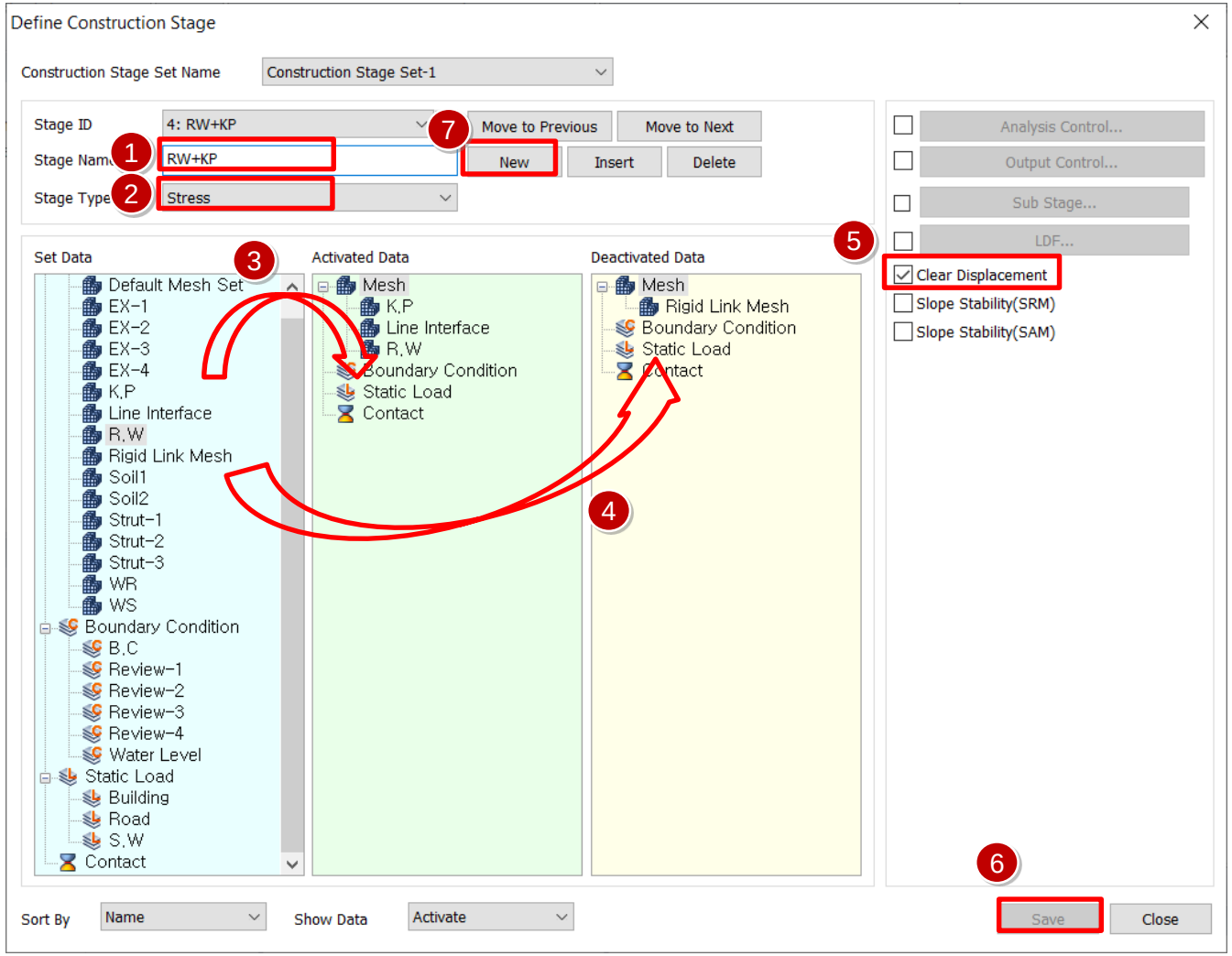


Step 04

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**'.

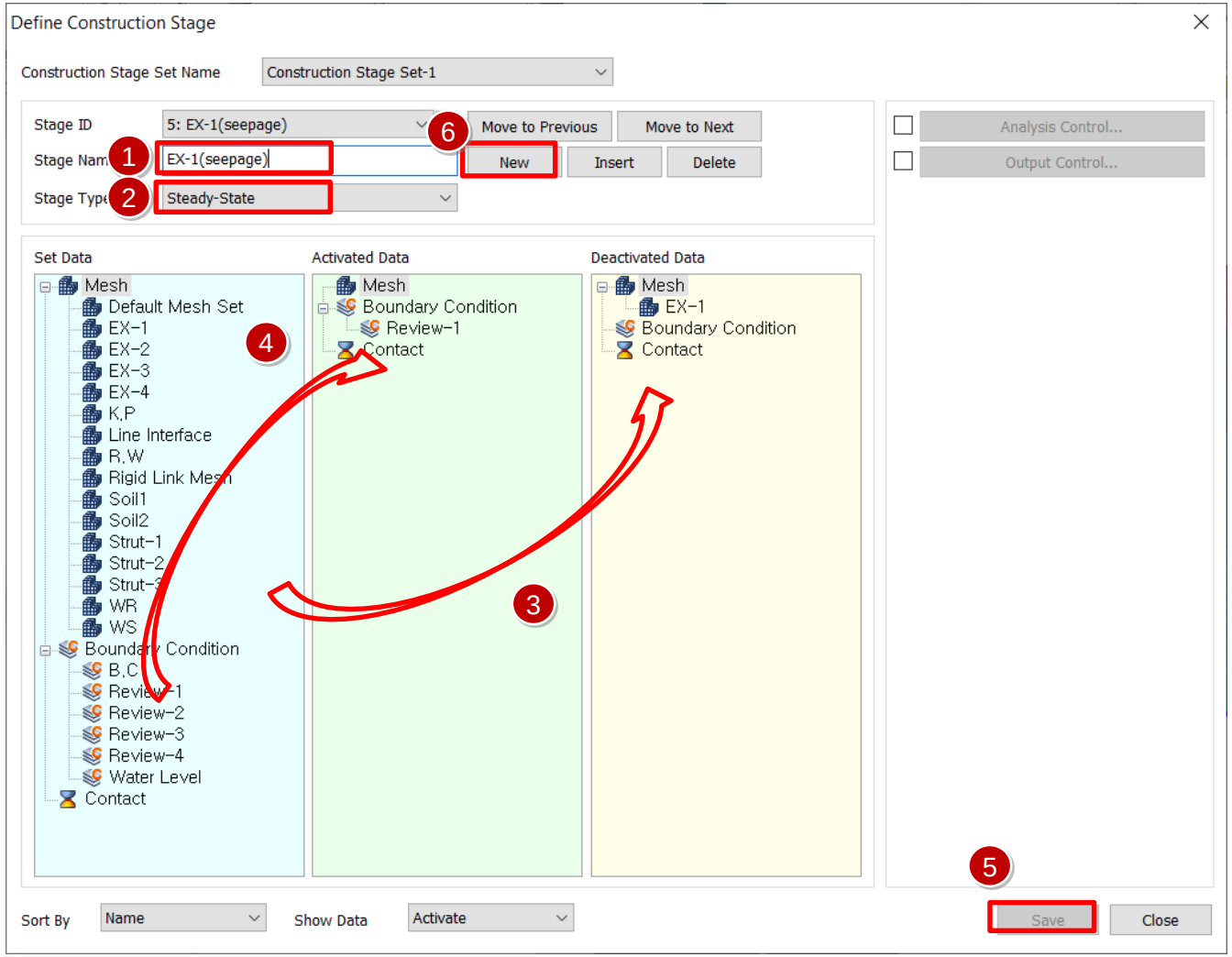


Step 04

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**'.

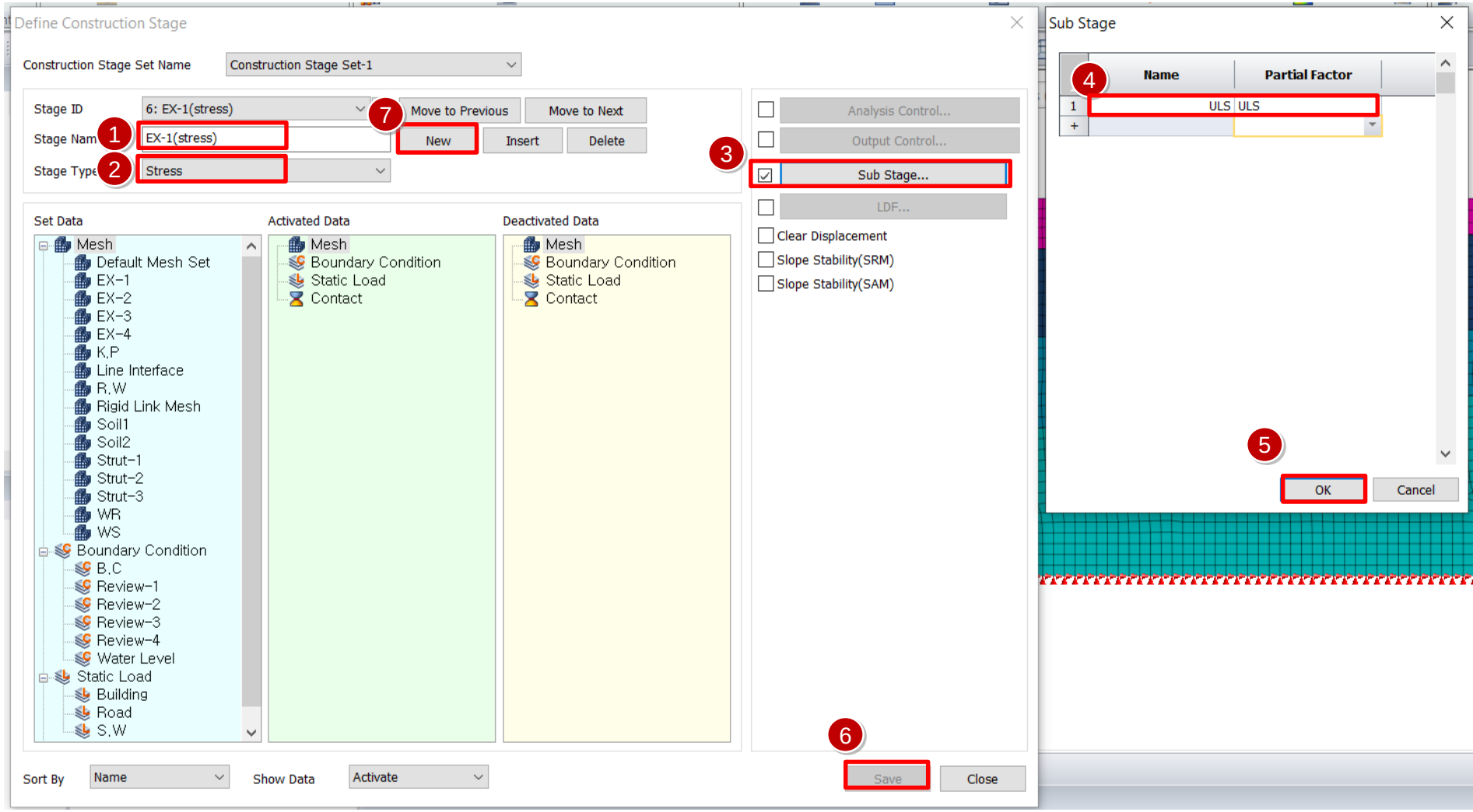


Step 04

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**'.

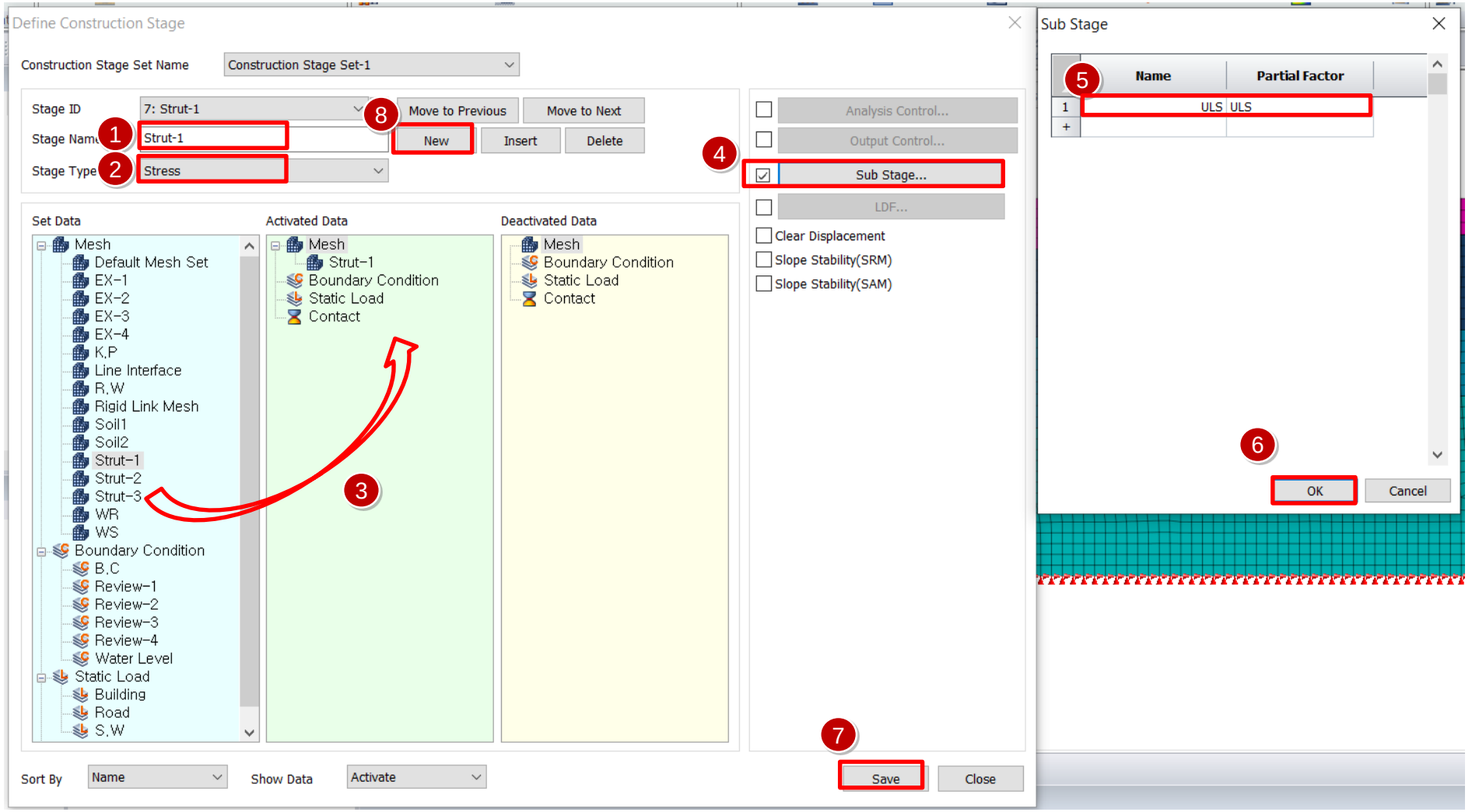


Step 04

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**'.

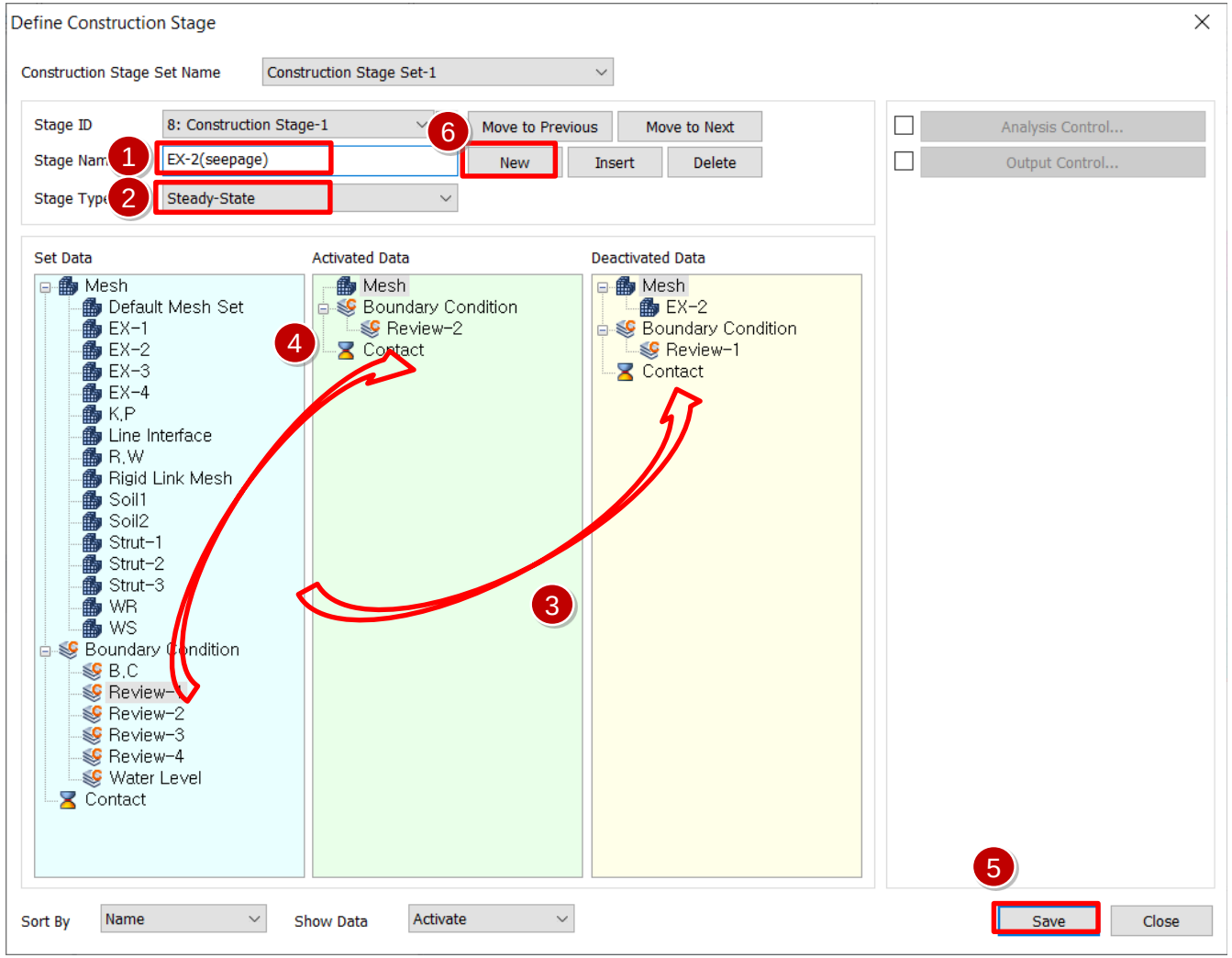


Step 04

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**'.

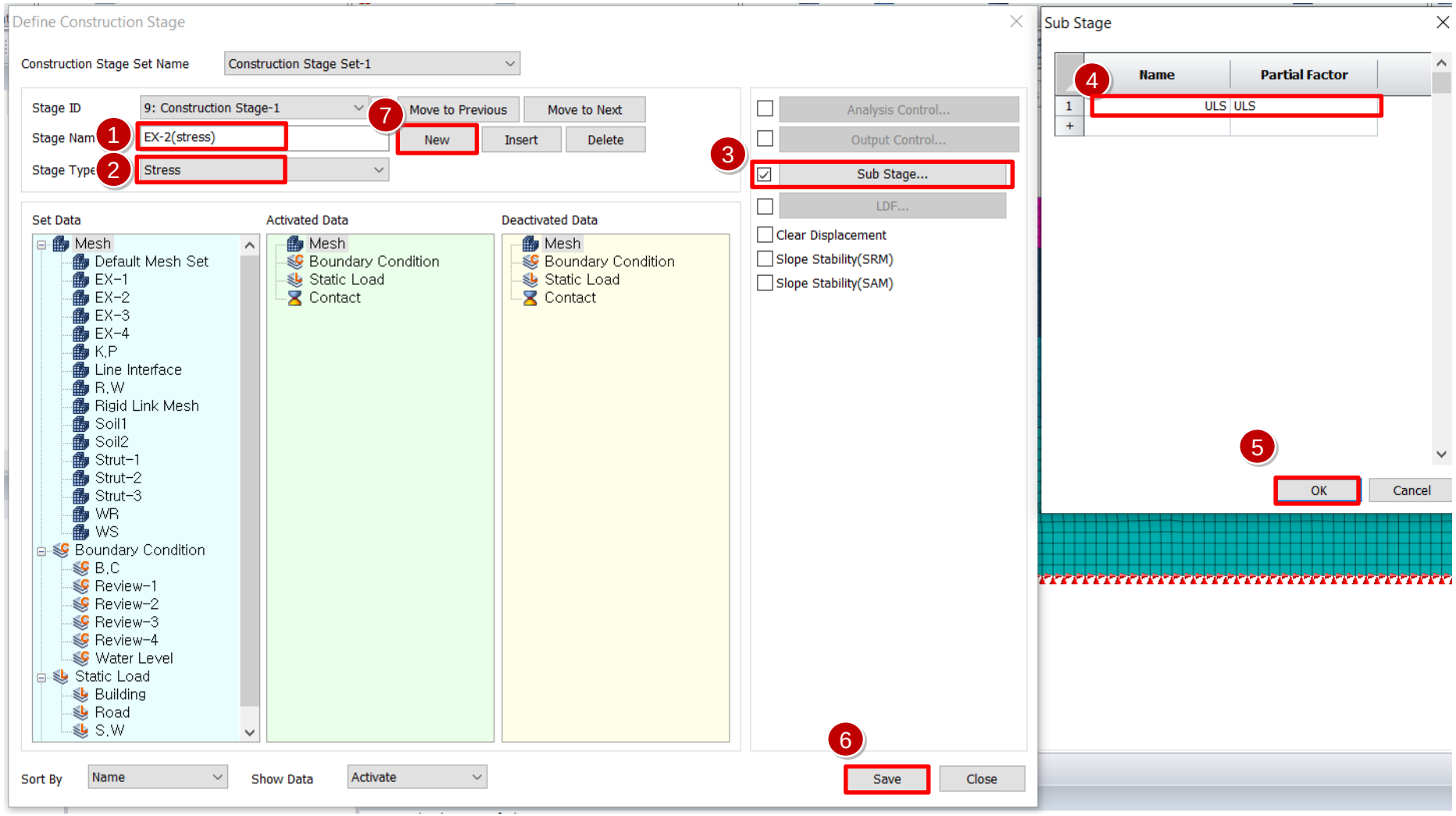


Step 04

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**'.

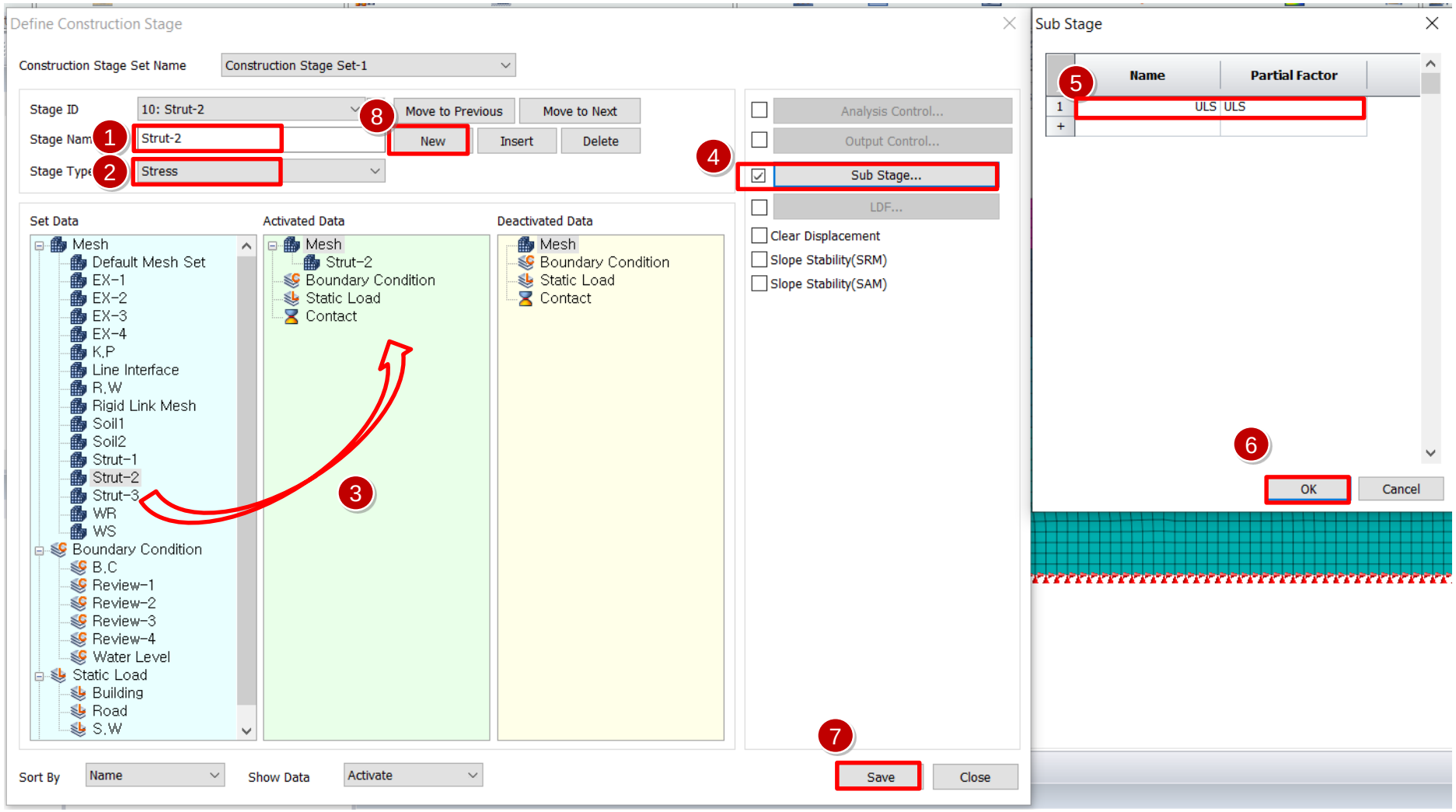


Step 04

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**'.

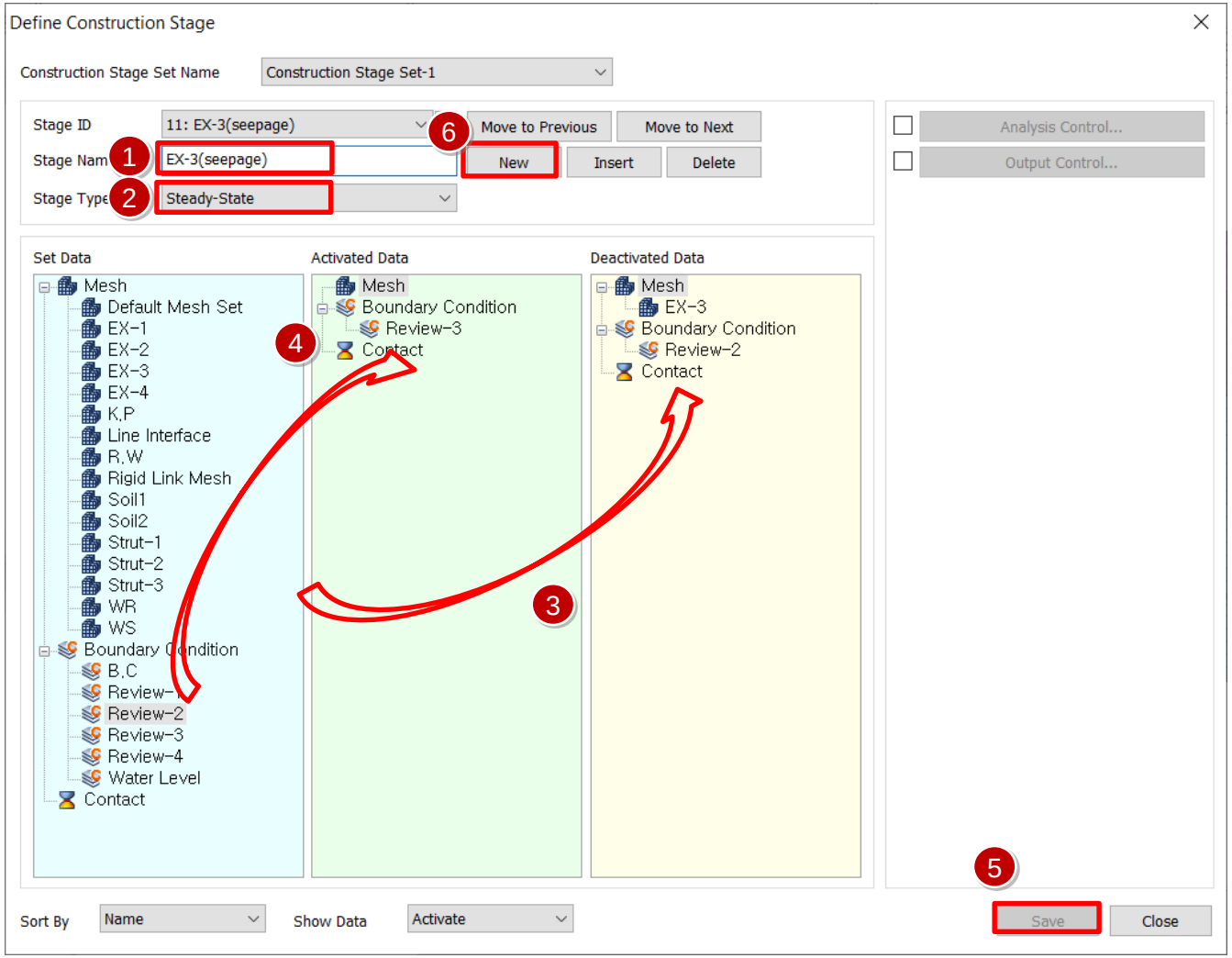


Step 04

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**'.

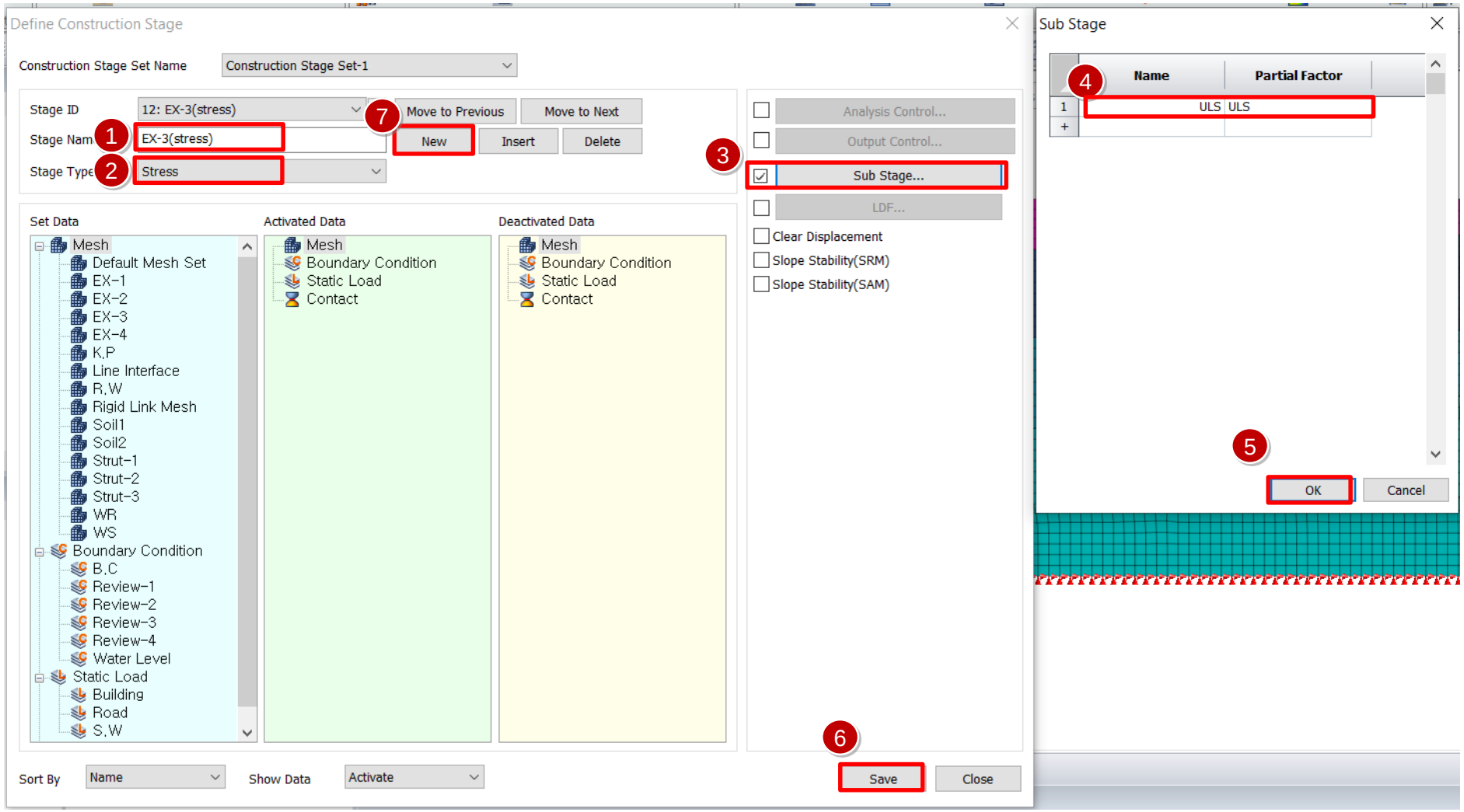


Step 04

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**'.

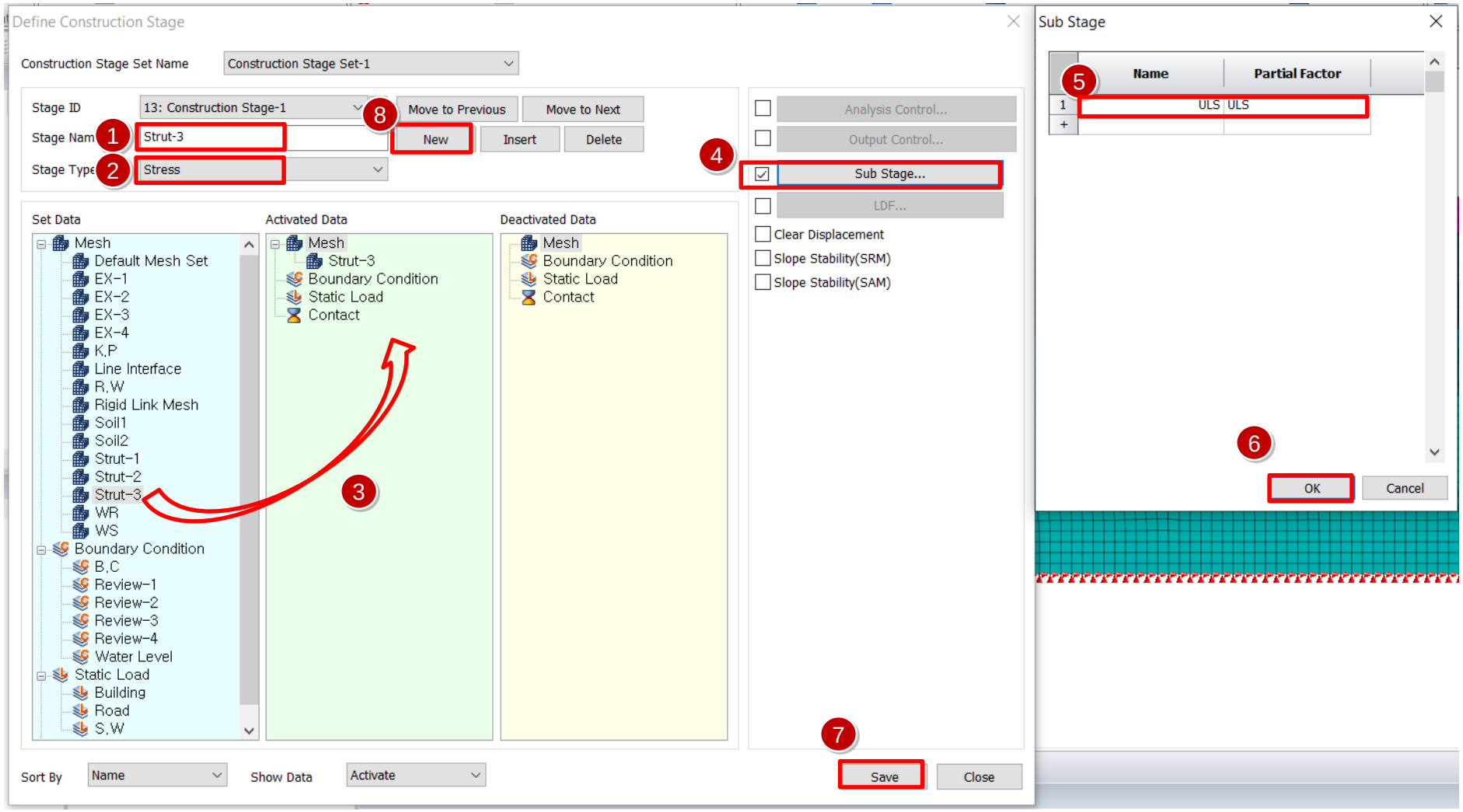


Step 04

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**'.

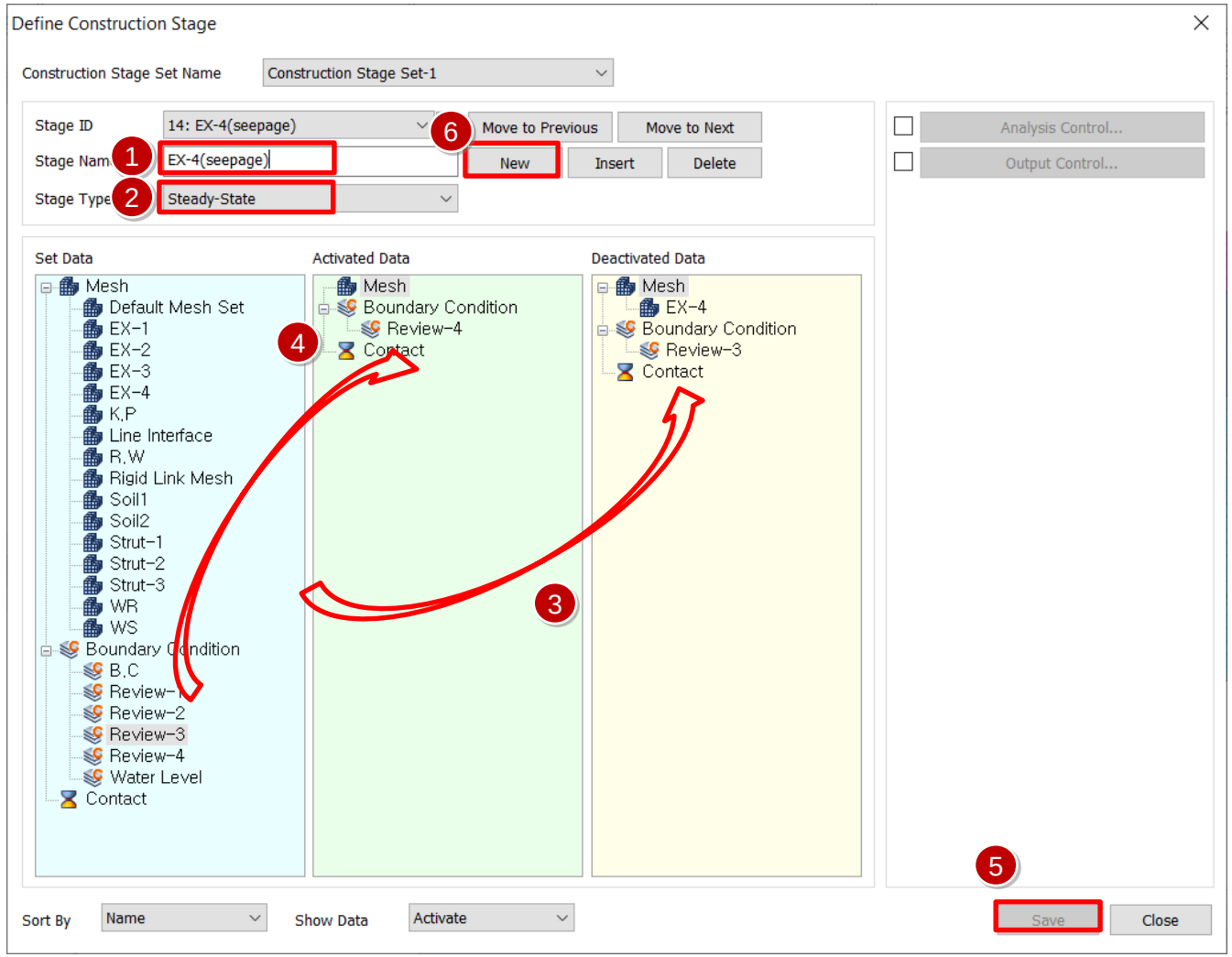


Step 04

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**'.

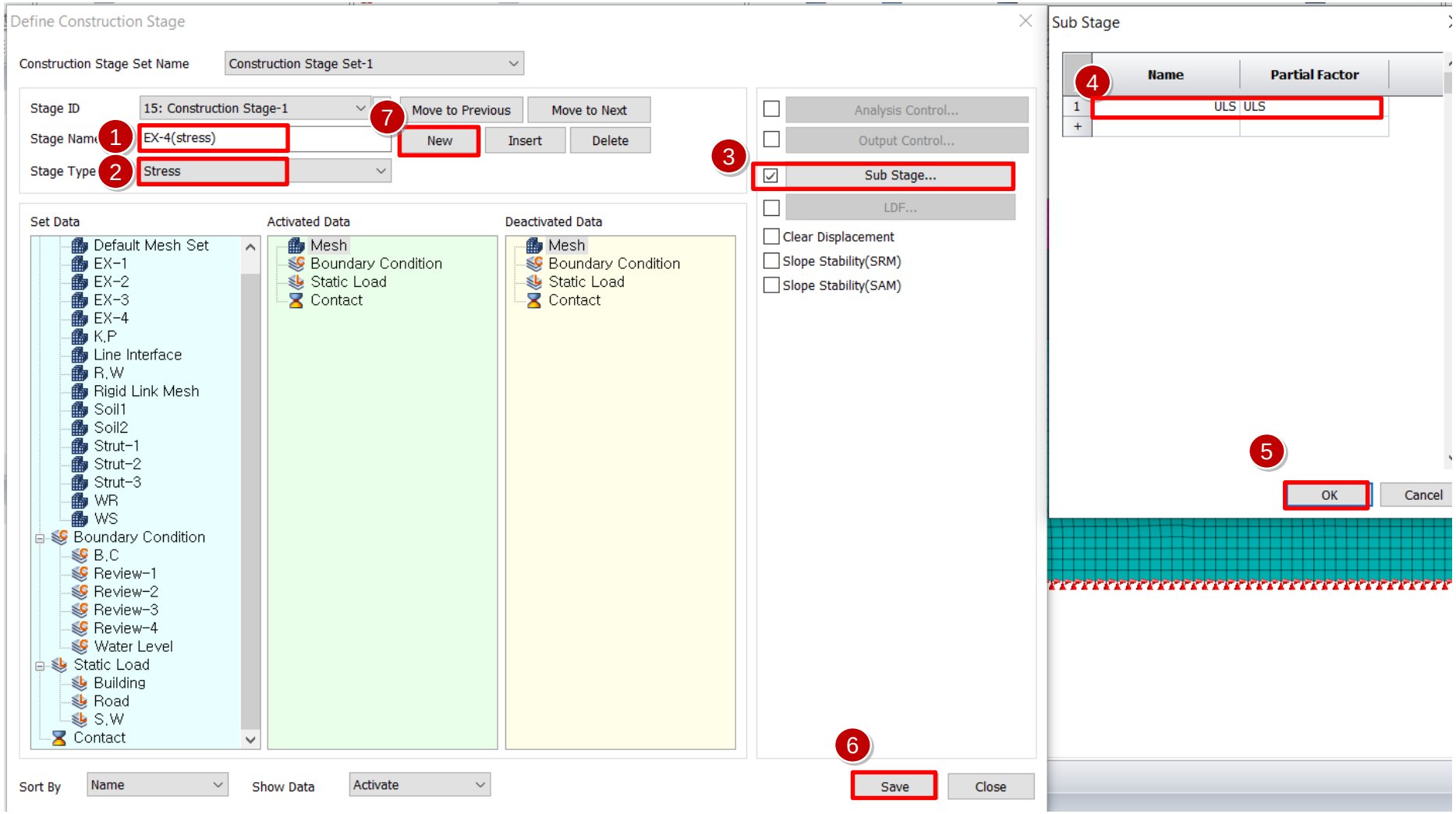


Step 04

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**'.

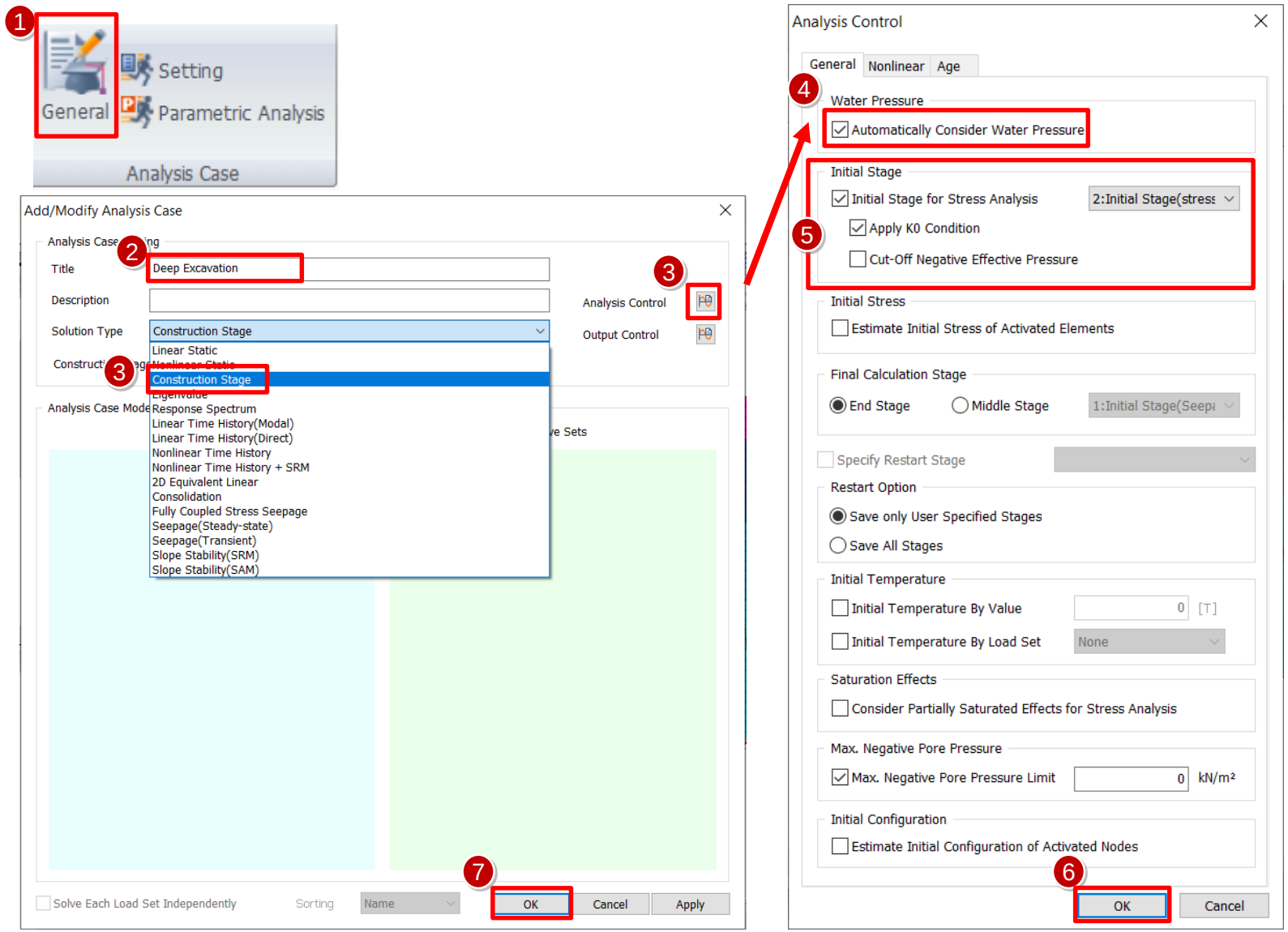


Step 04

4-2 Define Analysis Case

Procedure

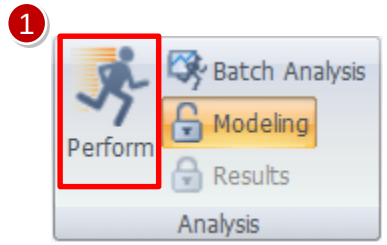
1. Select **Analysis > *Analysis Case*** > **General.**
2. Name as a '**Deep Excavation**'.
3. Change the Solution Type to '**Construction Stage**'.
4. Tick the '**Automatically Consider Water Pressure**'.
5. Tick '**Initial Stage for Stress Analysis**', '**Apply K0 Condition**' and select the '**2:Initial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

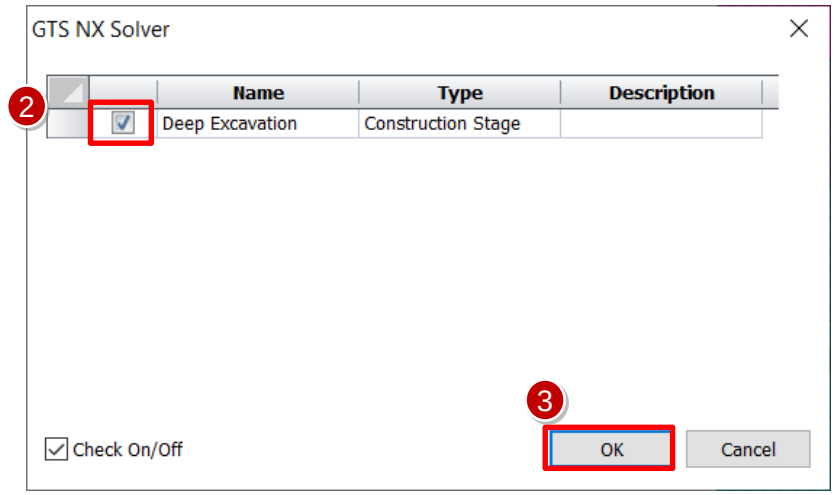


4-3 Analysis

Procedure

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



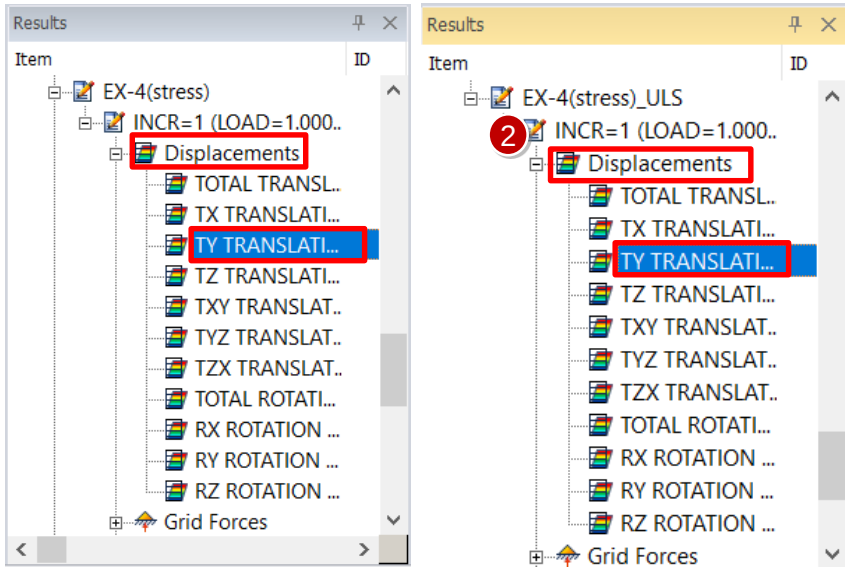


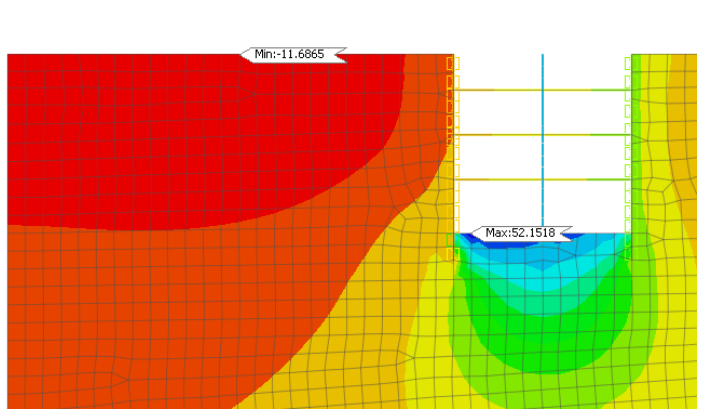
Step
05

Results – TY Translation (mm)

Procedure

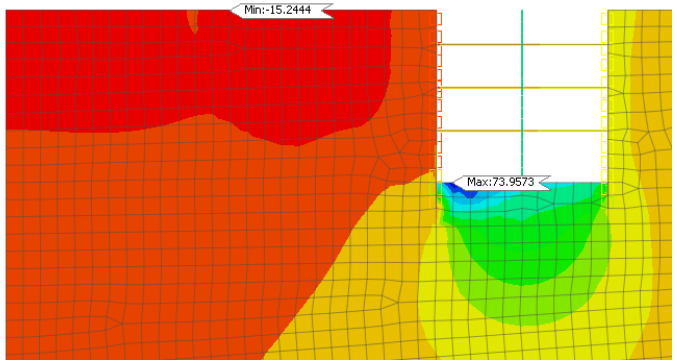
1. Go to **Results Tab,** expand EX-4(stress) construction stage. And Under Displacements, Click on TY Translation (V).
2. Expand EX-4(stress)_ULS construction stage. And Under Displacements, Click on TY Translation (V).
3. Users can **check the results comparing to the normal version and ULS version(partial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





Normal 4th Excavation Stage

3



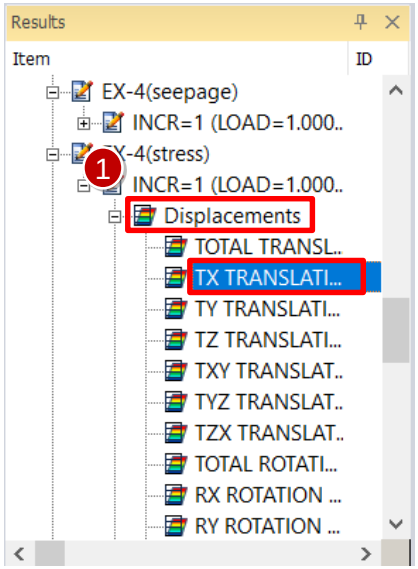
ULS 4th Excavation Stage

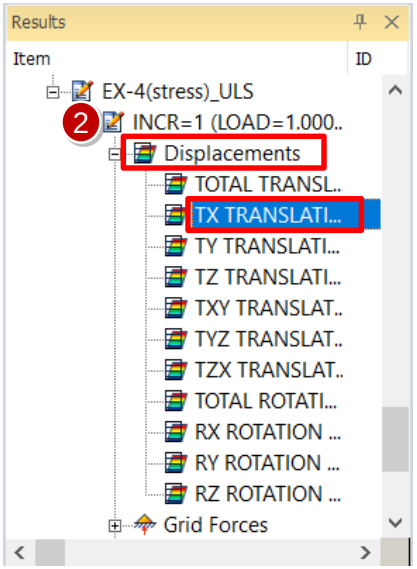
Step
05

Results – TX Translation (mm)

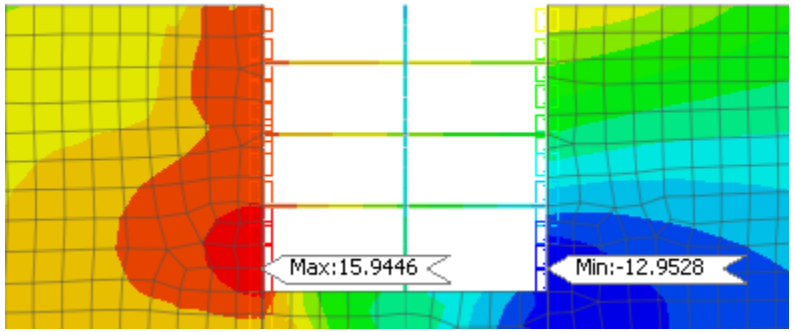
Procedure

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)**

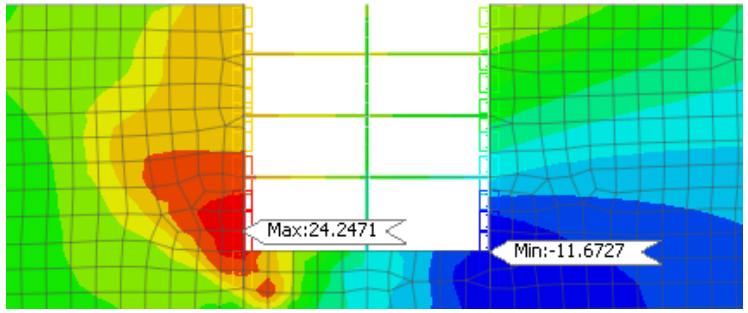




3



Normal 4th Excavation Stage



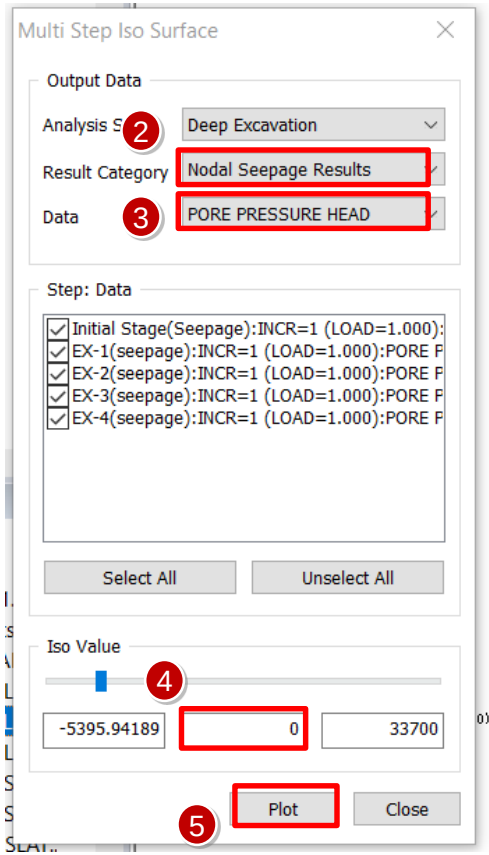
ULS 4th Excavation Stage

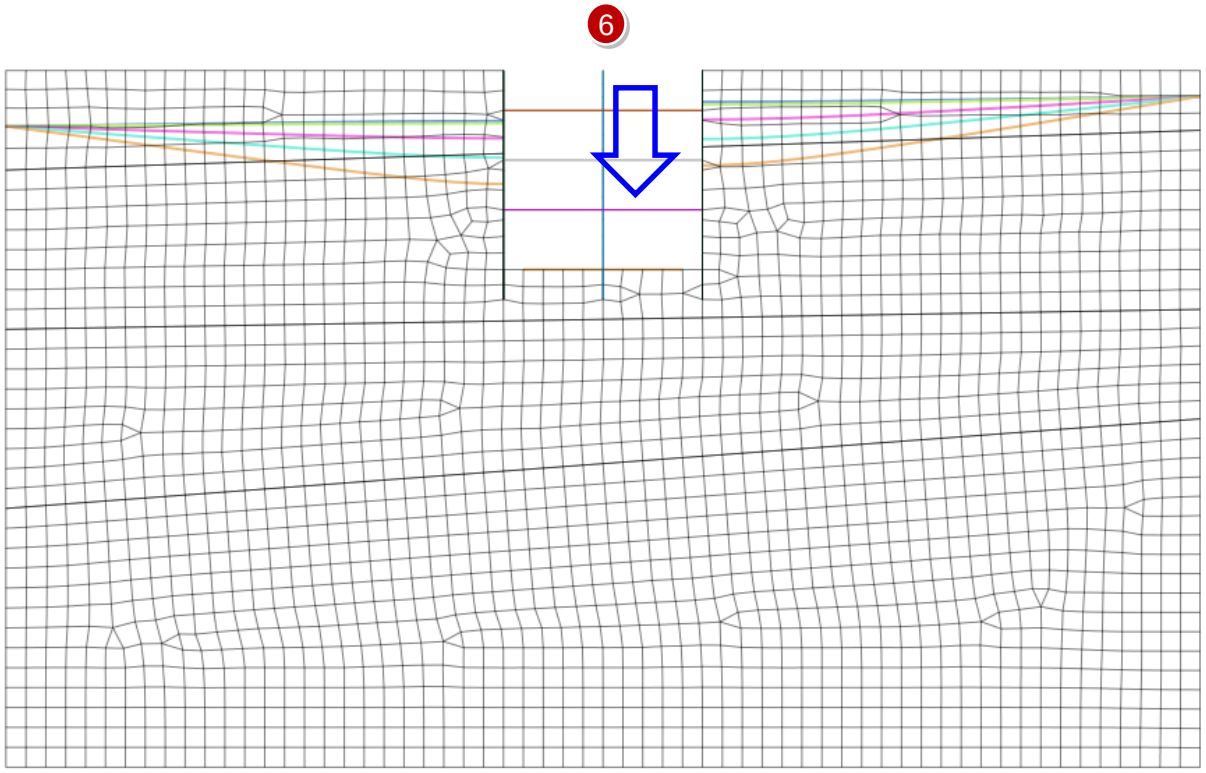
Step
05

Results – Total Translation (m)

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