

The background features a complex network of interconnected nodes and lines, resembling a mesh or a molecular structure. The nodes are small circles, and the lines are thin, connecting the nodes in a non-uniform pattern. The overall color scheme is a gradient from light green on the left to bright orange on the right.

GTS NX

New eXperience of Geo-Technical analysis System

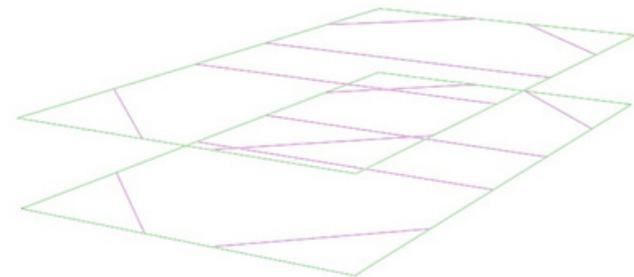
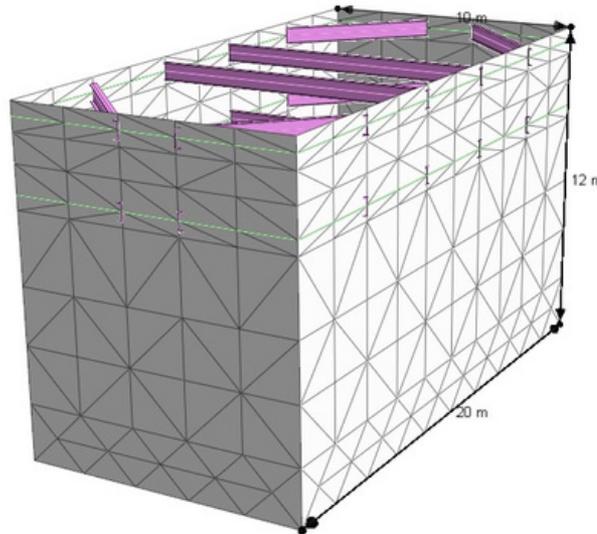
Excavation

Advanced Course [Session 1]

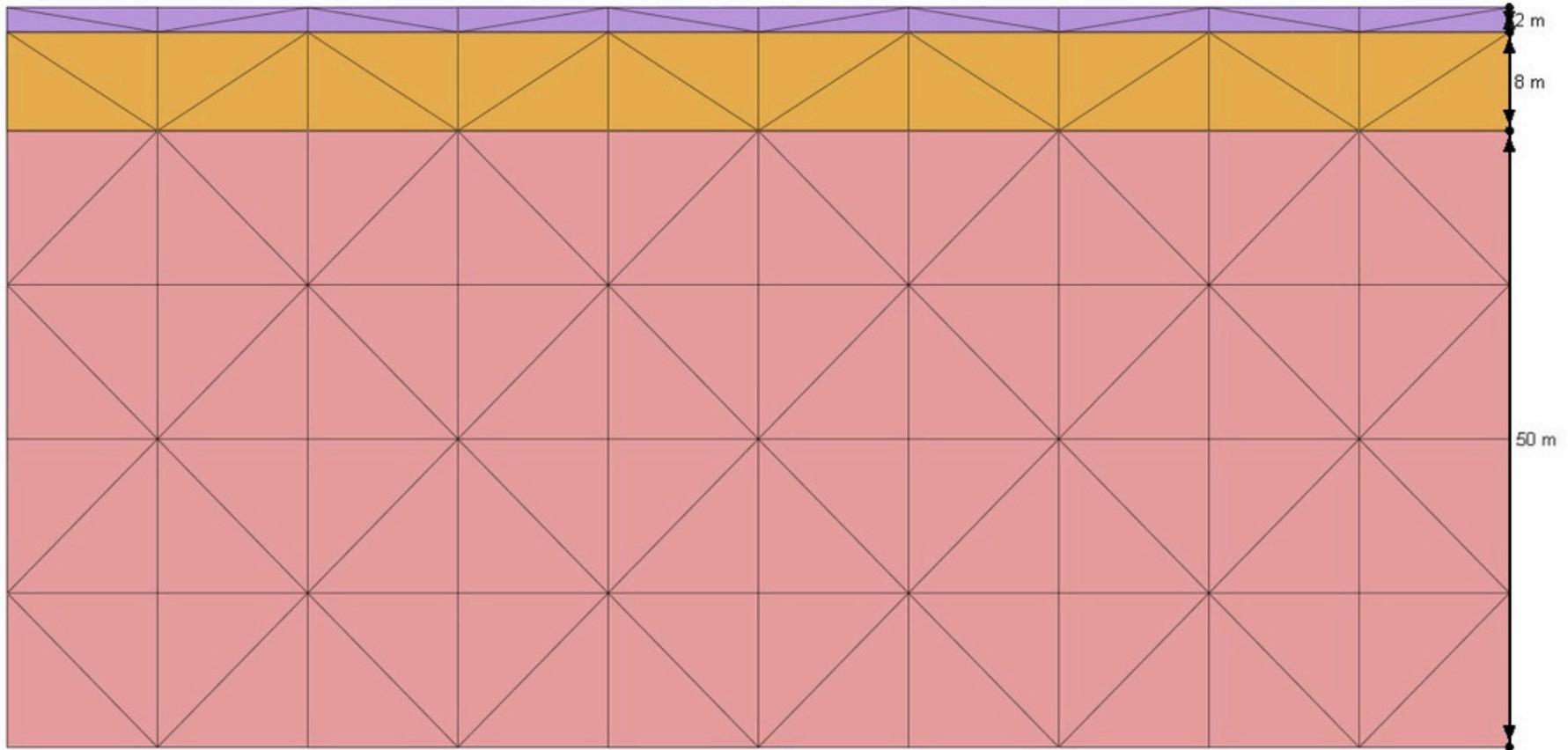
Modeling and Analysis Summary

The model represents 10mX20m excavation area with temporary facilities which are diaphragm wall, waler and strut. The excavation process consists of 3 stages(1.0, 3.5, 6.0 b.g.l.(m)). And, total supports(waler and strut) are installed at 2 levels (0.5, 3.0 b.g.l.(m)) with interval distance of 2.5m. The toe level of retaining wall(D-Wall) is 12 b.g.l.(m).

The ground is consisting of 3 layers and depths are respectively: Buried layer 2.0 m, Colluviums 8.0 m, the rest part is weathered soil. Surcharge and water level are considered as well in this case.



Modeling and Analysis Summary



Ground

[unit: kN, m]

Name	Buried Layer	Colluvium	Weathered Soil
Material	Isotropic	Isotropic	Isotropic
Model Type	Hardening Soil	Hardening Soil	Hardening Soil
General			
Poisson's Ratio (ν)	0.25	0.25	0.25
Unit Weight (γ) [kN/m ³]	16	17	20
Ko	0.5	0.7	0.7
Porous			
DrainageParameters	Drained	Undrained	Drained
Permeability k [m/sec]	1e-007	1e-007	1e-007
Non-Linear			
Secant Stiffness in Tri-axial Test (E50ref) [kN/m ²]	10,000	60,000	100,000
Tangential Stiffness Primary Oedometer Test (Eoedref) [kN/m ²]	10,000	60,000	100,000
Unloading/Reloading Stiffness (Eurref) [kN/m ²]	30,000	180,000	300,000
Cohesion (c) [kN/m ²]	1	1	4
Frictional Angle (Φ) [deg]	30	34	38

Structure

[unit: kN, m]

Name	Conc'	Steel
Material	Isotropic	Isotropic
Model Type	Elastic	Elastic
Elastic Modulus (E) [kN/m ²]	28,000,000	205,000,000
Poisson's Ratio (ν)	0.15	0.15
Unit Weight (γ) [kN/m ³]	24	74

Property (Structure)

[unit: kN, m]

Name	Material	Type
Retaining Wall (1.0m thk.D-Wall)	Conc'	Shell
Waler(UB 610x229x101 kg/m)	Steel	Beam
Strut (UB 610x229x101 kg/m)	Steel	Truss

GTS NX

New eXperience of Geo-Technical analysis System

Geometry Modeling

MIDAS

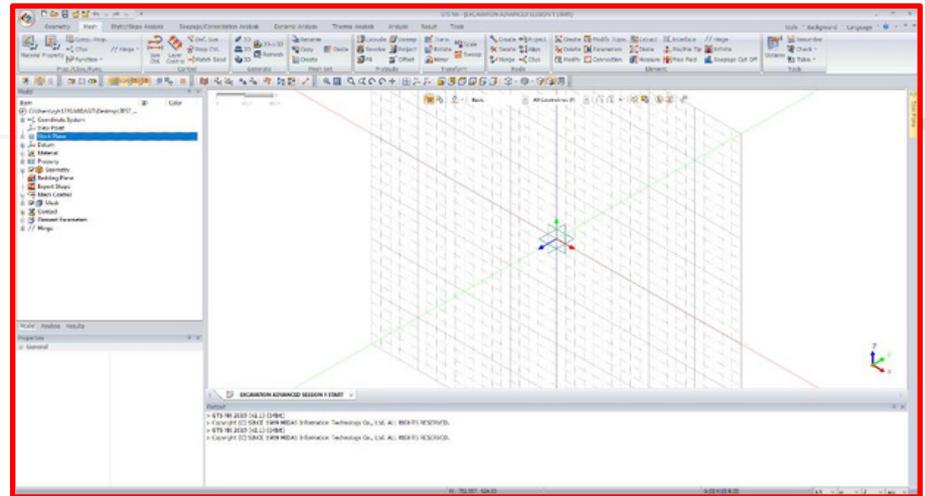
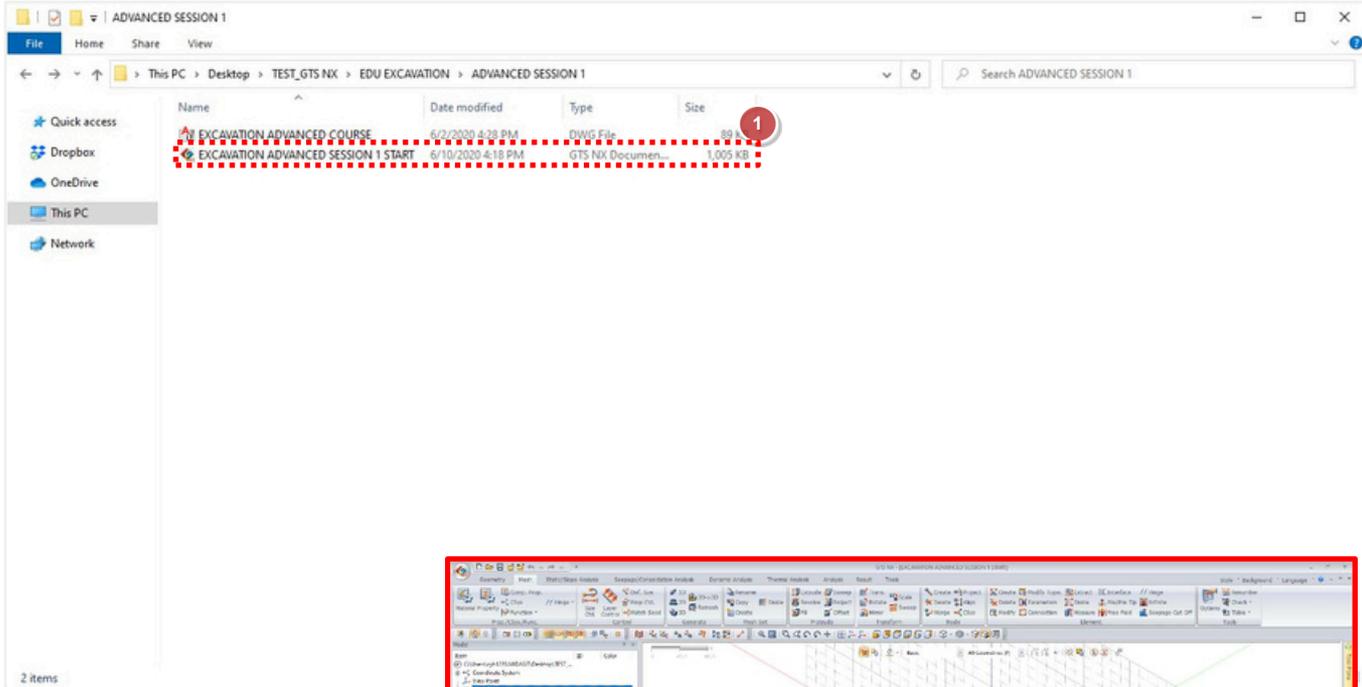
03 Geometry Modeling

Starting

Procedure

1 Open the start file

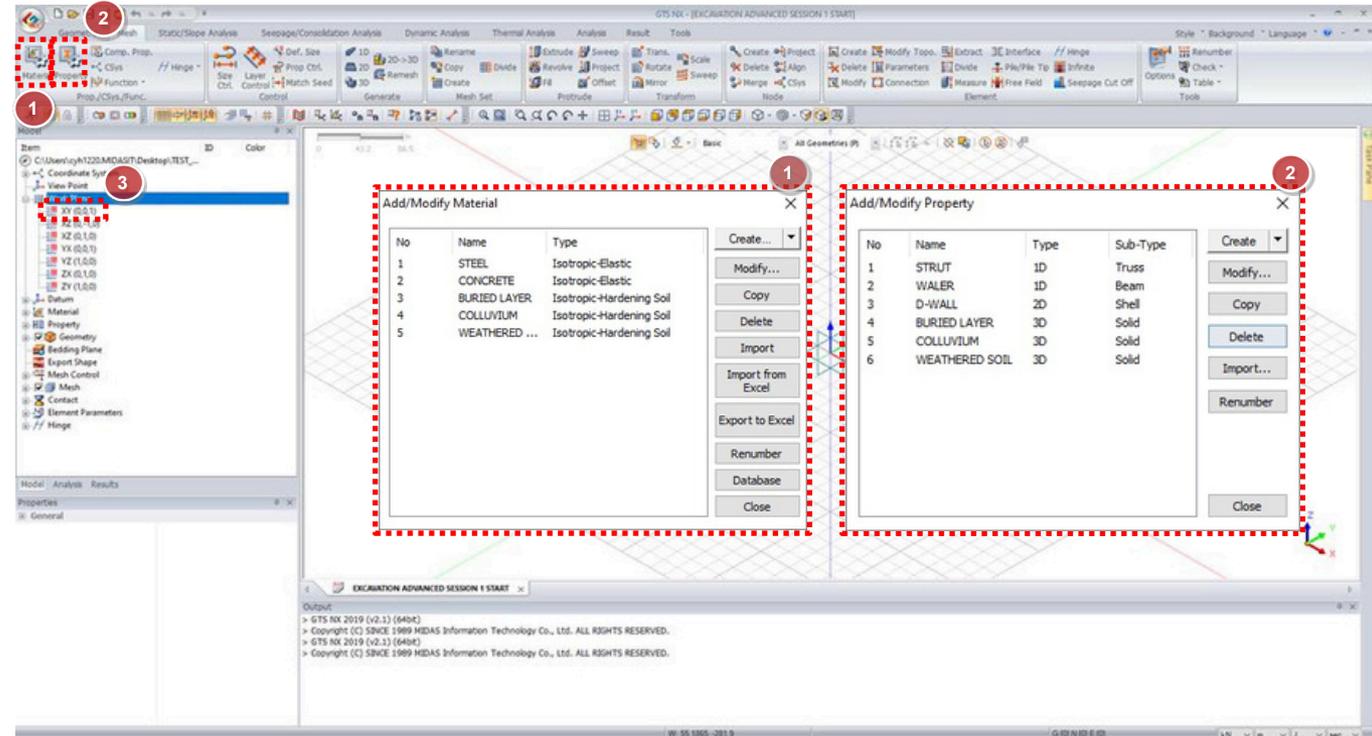
“EXCAVATION ADVANCED SESSION 1 START”



Check the properties

Procedure

- 1 Check existing materials (Mesh > Prop./CSys./Func. > Material)
- 2 Check existing properties (Mesh > Prop./CSys./Func. > Property)
- 3 Open "Work Plane" tree
Double click XY (0,0,1) to change work plane

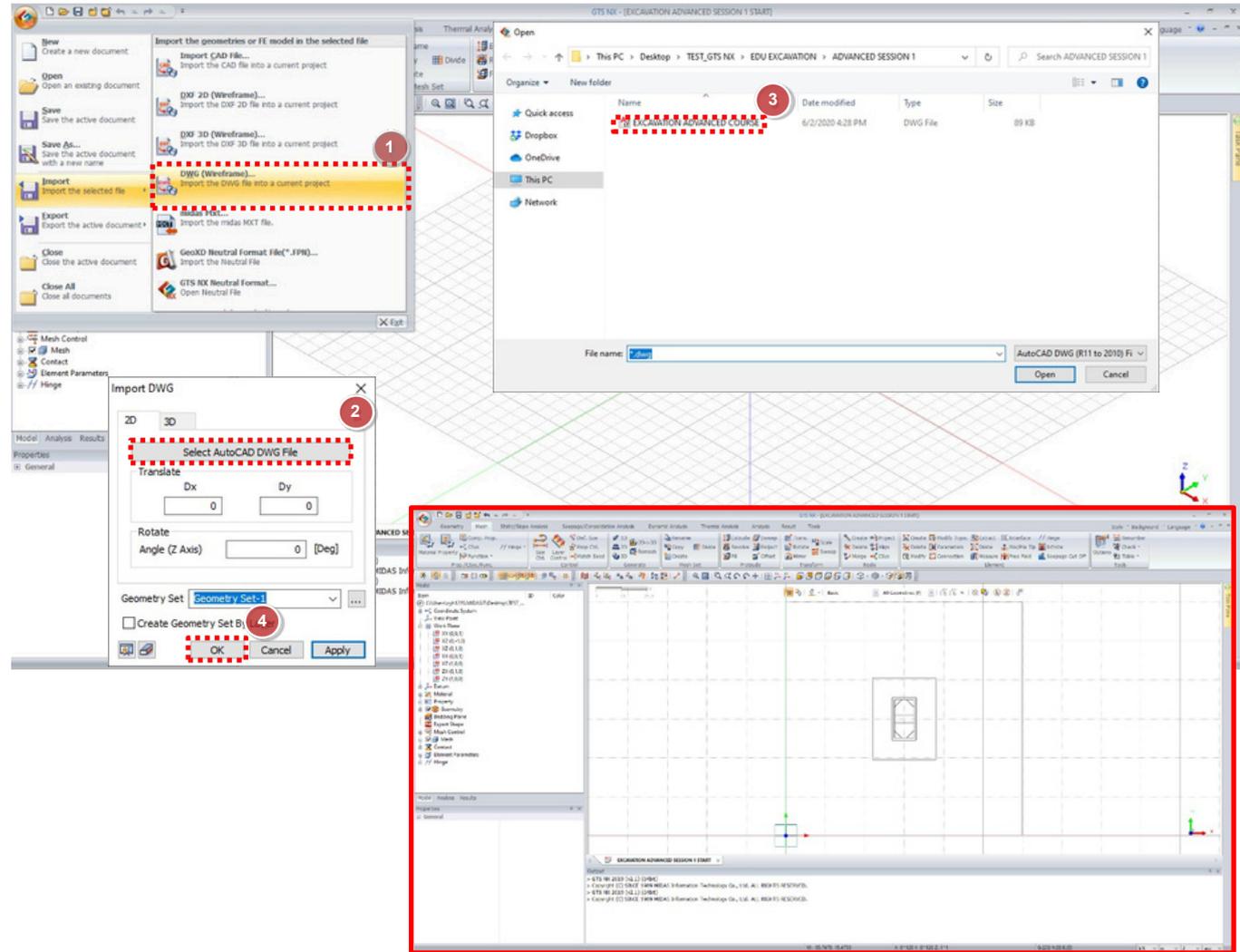


03 Geometry Modeling

ImportCAD file

Procedure

- 1 Click “DWG(Wireframe)...”
- 2 Click
“Select AutoCAD DWG File”
- 3 Double click “EXCAVATION
ADVANCED COURSE” Click
“OK”

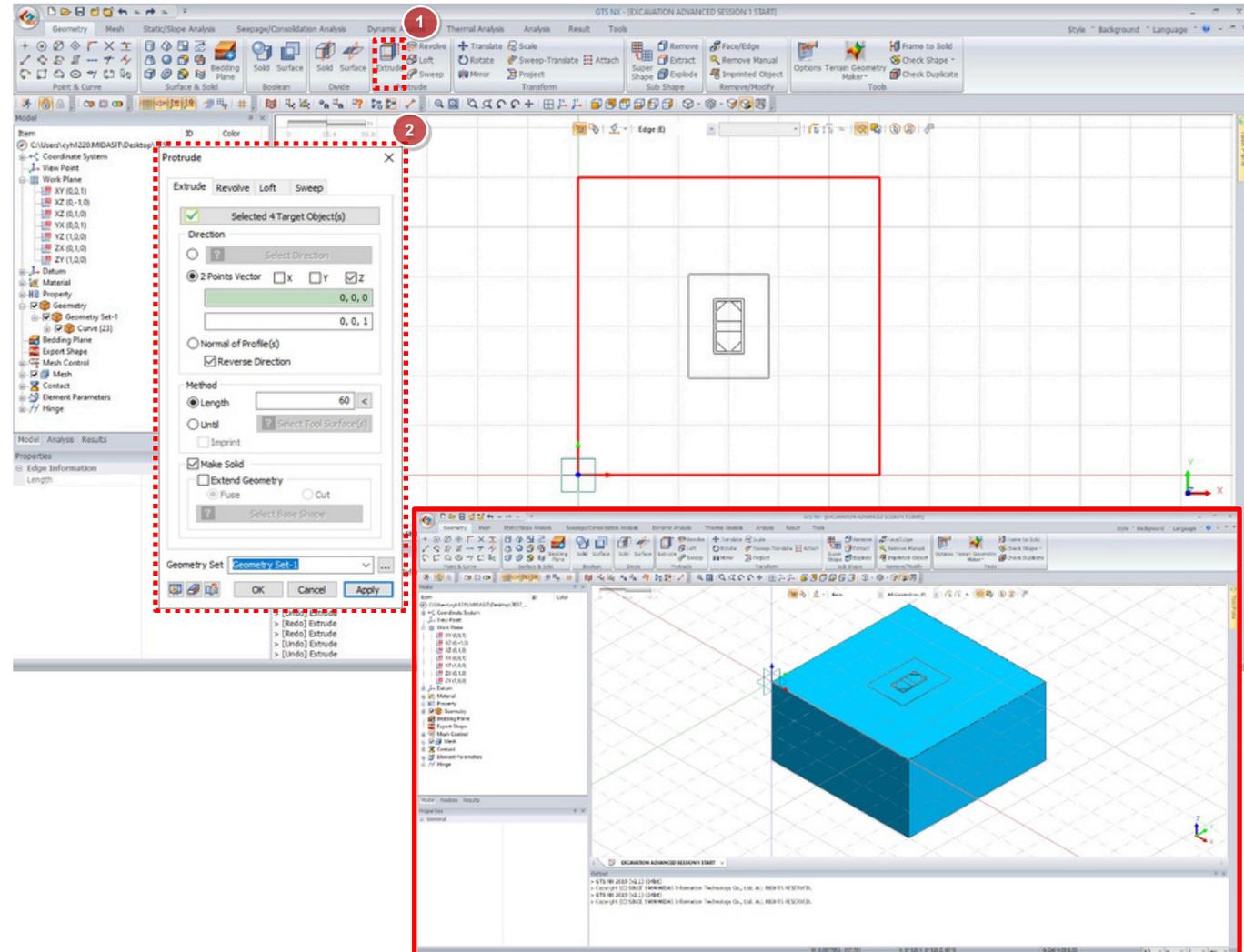


03 Geometry Modeling

Makethesolid for ground part

Procedure

- 1 Click "Extrude"
Select out lines (4 lines)
- 2 (Press 'E' to change selection method to edge from face)
Check on "2 Points Vector"
Check on Z only
Check on "Reverse Direction"
Length "60"
Click "OK"



03 Geometry Modeling

Makethesolid for excavation(retaining wall) part

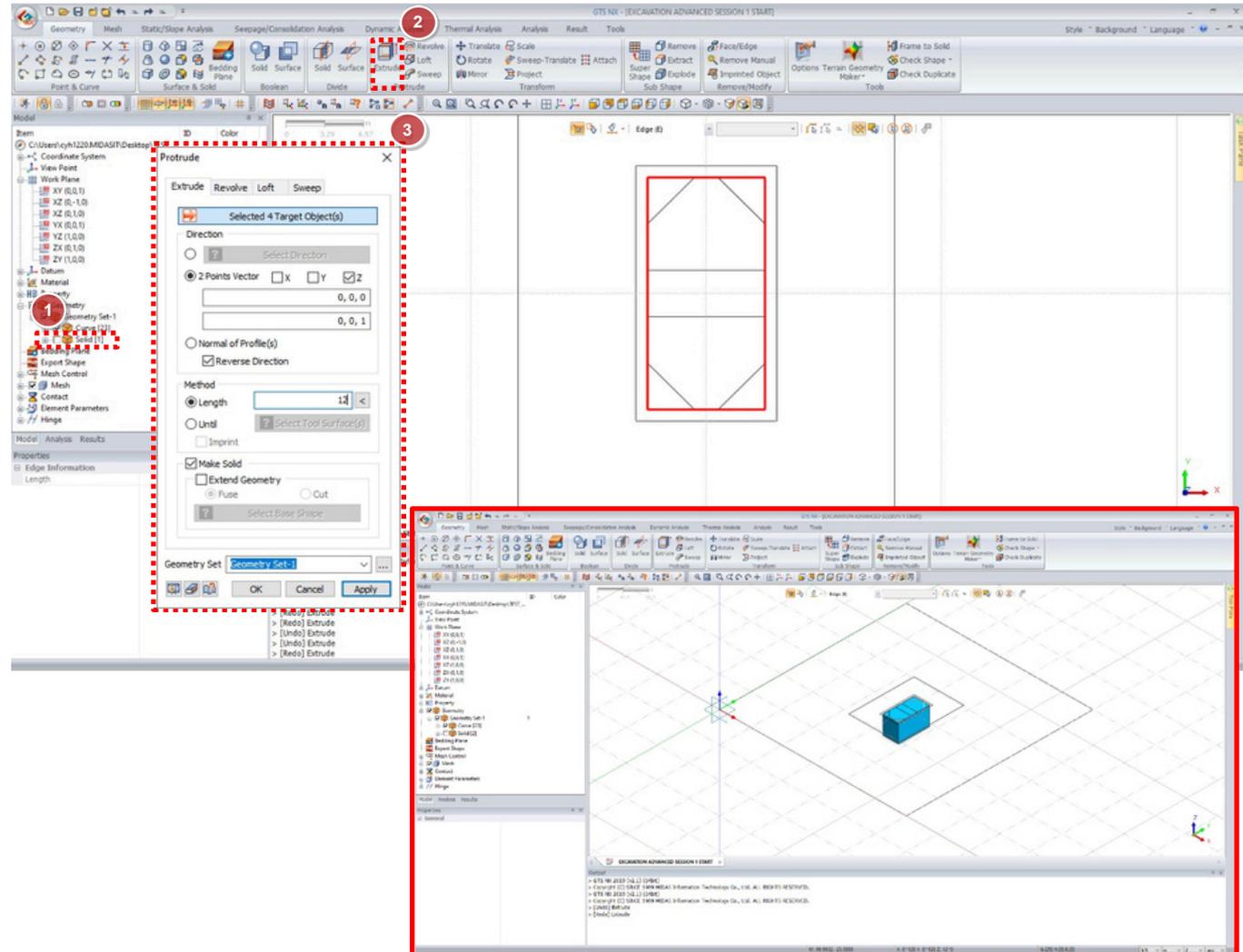
Procedure

- 1 Open Geometry tree
Hide Solid
Click "Extrude"
- 2 Select out lines (4 lines)
(Press 'E' to change selection method to edge from face)
- 3 Check on "2 Points Vector"

Check on Z only

Length "12"

Click "OK"



03 Geometry Modeling

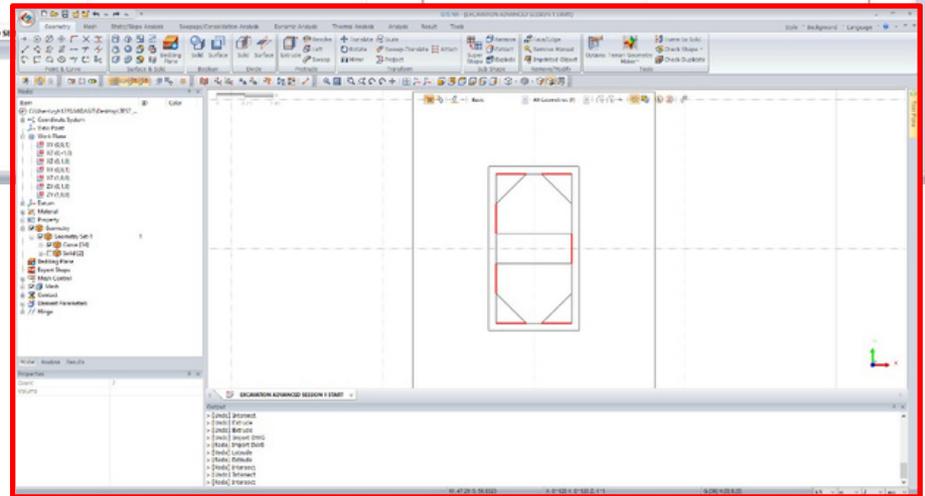
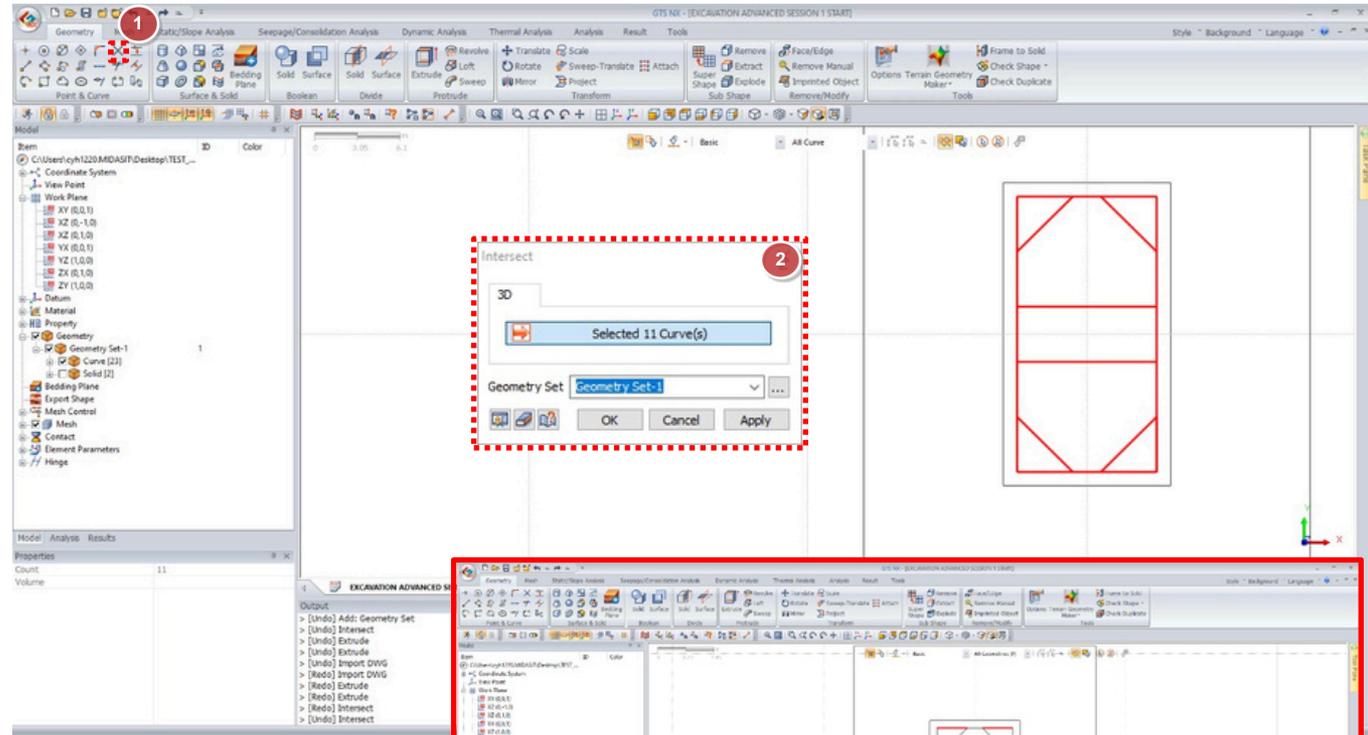
Strut & Waler

Procedure

1 Click "Intersect"
Select 11 curves

2 Click "OK"

[Information]
Lines will be divided by point
with intersect function

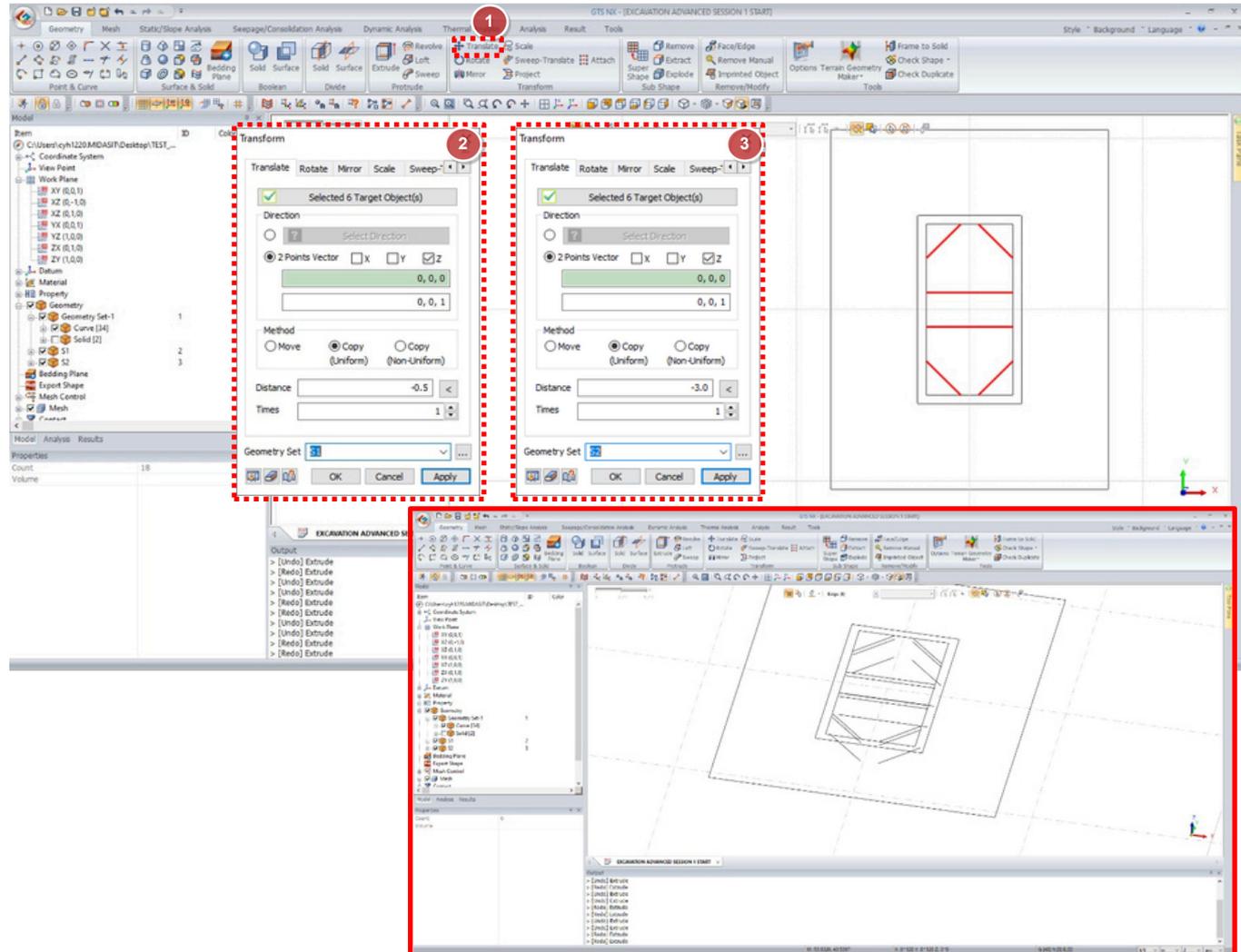


03 Geometry Modeling

Strut & Waler

Procedure

- 1 Click "Translate"
Select 6 curves
- 2 Check on "2 Points Vector"
Select Z only
Select "Copy" from method
Distance "-0.5"
Geometry Set "S1"
Click "Apply"
- 3 Distance "-3.0"
Geometry Set "S2"
Click "OK"

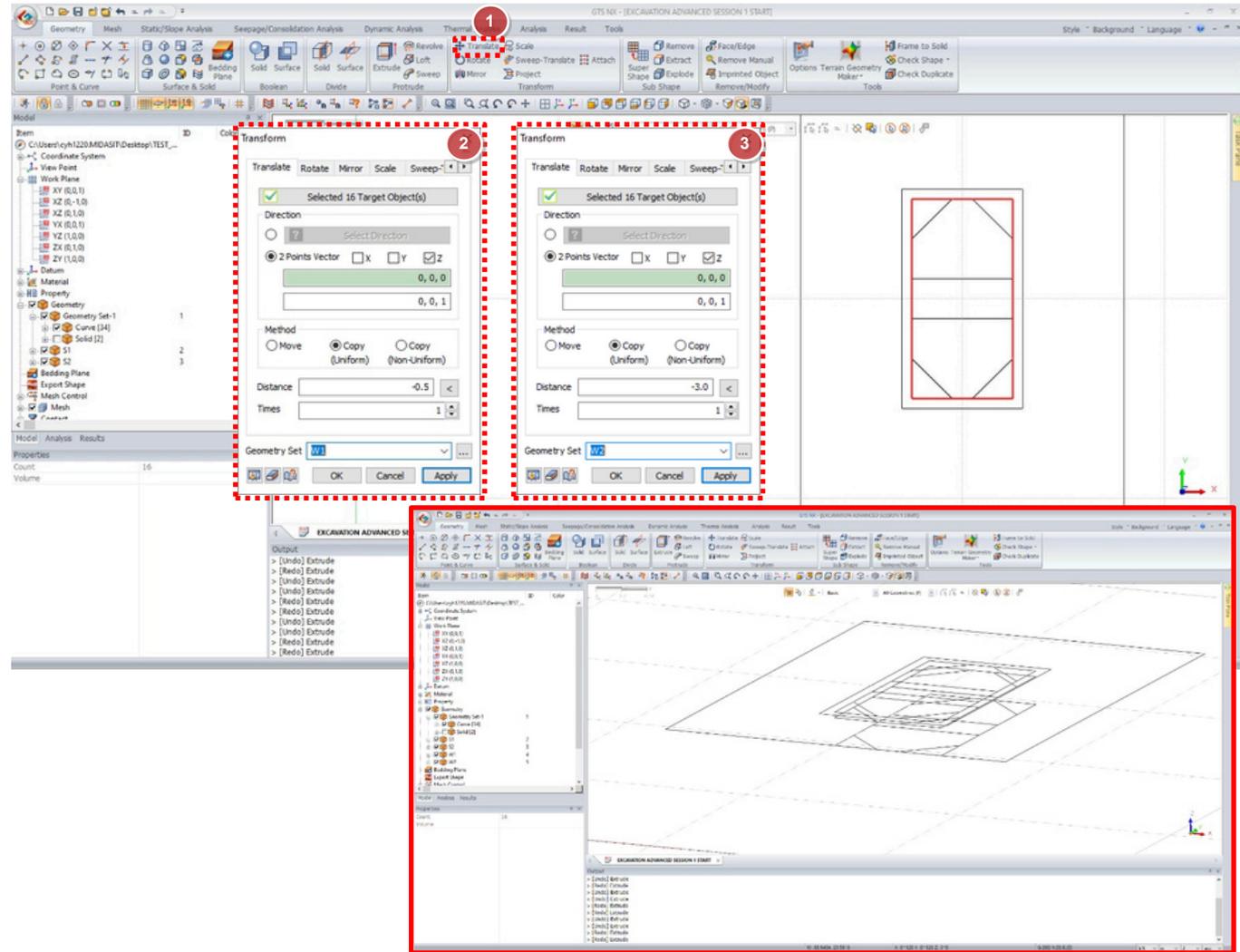


03 Geometry Modeling

Strut & Waler

Procedure

- 1 Click "Translate"
Select 16 curves
- 2 Check on "2 Points Vector"
Select Z only
Select "Copy" from method
Distance "-0.5"
Geometry Set "W1"
Click "Apply"
- 3 Distance "-3.0"
Geometry Set "W2"
Click "OK"

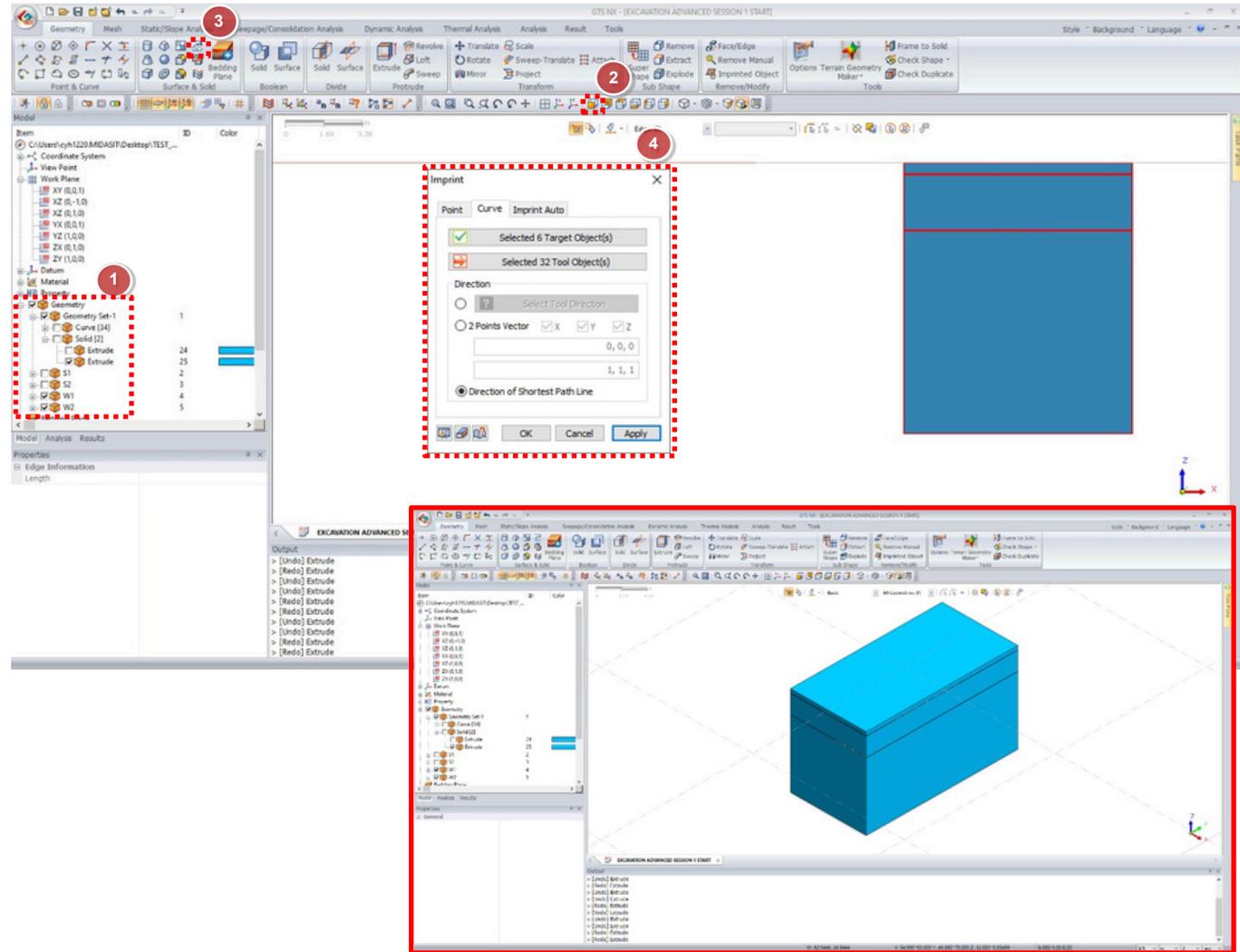


03 Geometry Modeling

Waterimprint to retaining wall part

Procedure

- 1 Hide Curve / S1 / S2 Show W1 / W2 / Extrude(2nd)
Click "front view"
- 2 Click "Imprint" Go to "Curve"
tab Ctrl+A to select all faces
- 3 Select W1 & W2 Curves
Select "Direction of shortest.."
- 4 Click "OK"

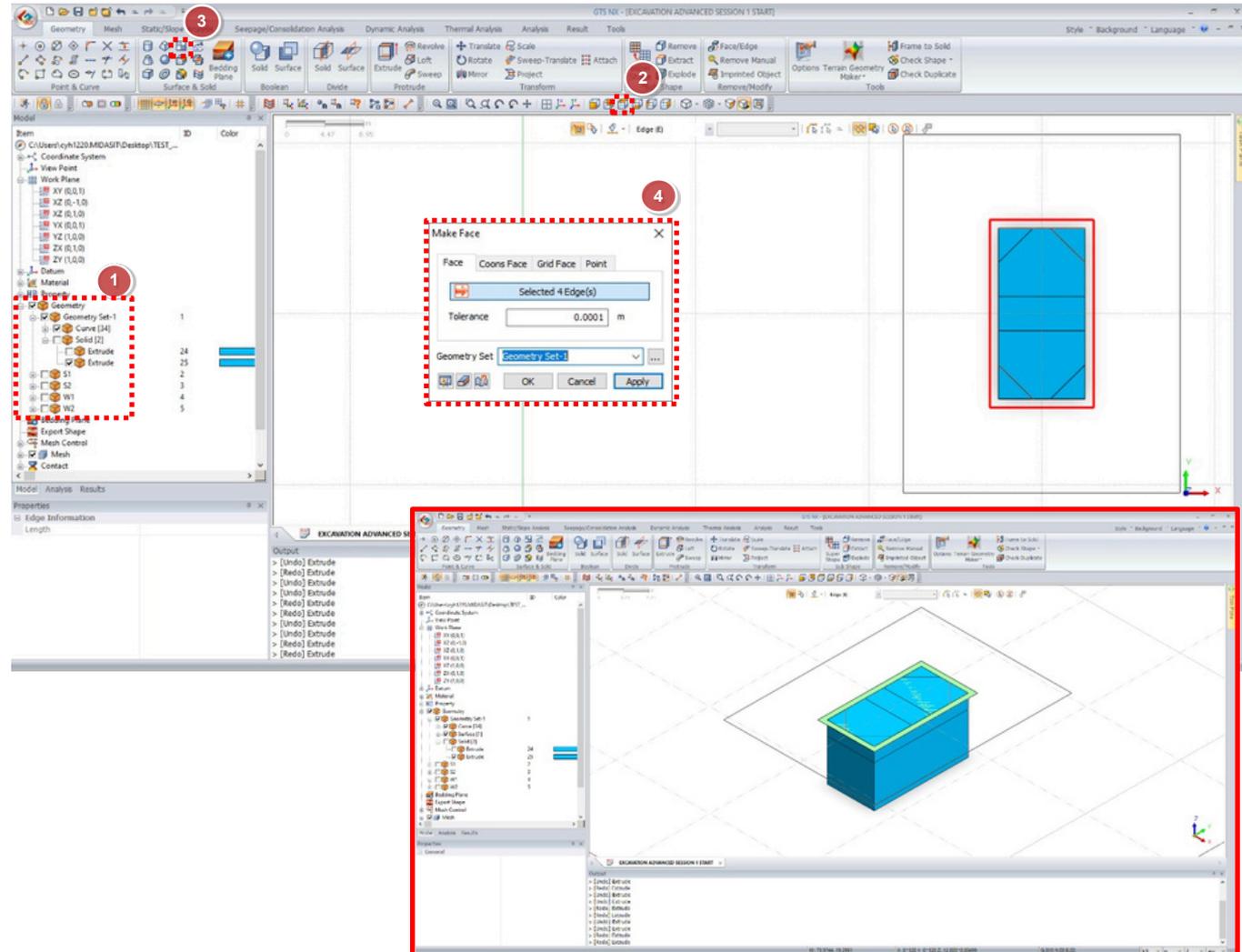


03 Geometry Modeling

Define the excavation depth

Procedure

- 1 Hide W1 / W2
Show Curve
Click "Top View"
- 2 Click "Make Face"
Select 4 edges
- 3 Geometry Set "Geometry Set-1"
Click "OK"
- 4

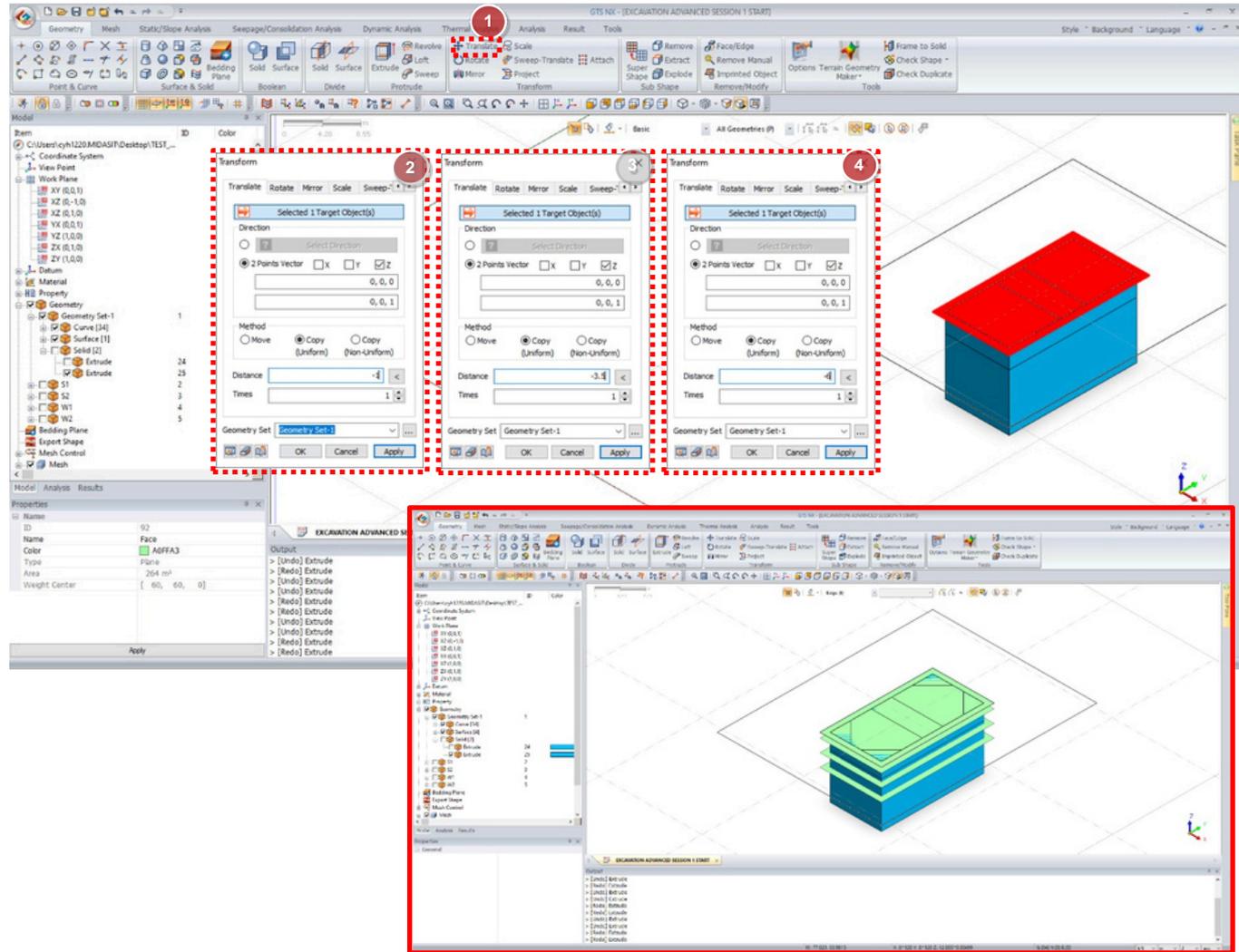


03 Geometry Modeling

Definetheexcavation depth

Procedure

- 1 Click "Translate"
Select face
Select "2 Point Vector"
Distance "-1.0"
Geometry Set "Geometry Set-1"
Click "Apply"
Distance "-3.5"
- 3 Click "Apply"
- 4 Distance "-6.0"
Click "OK"

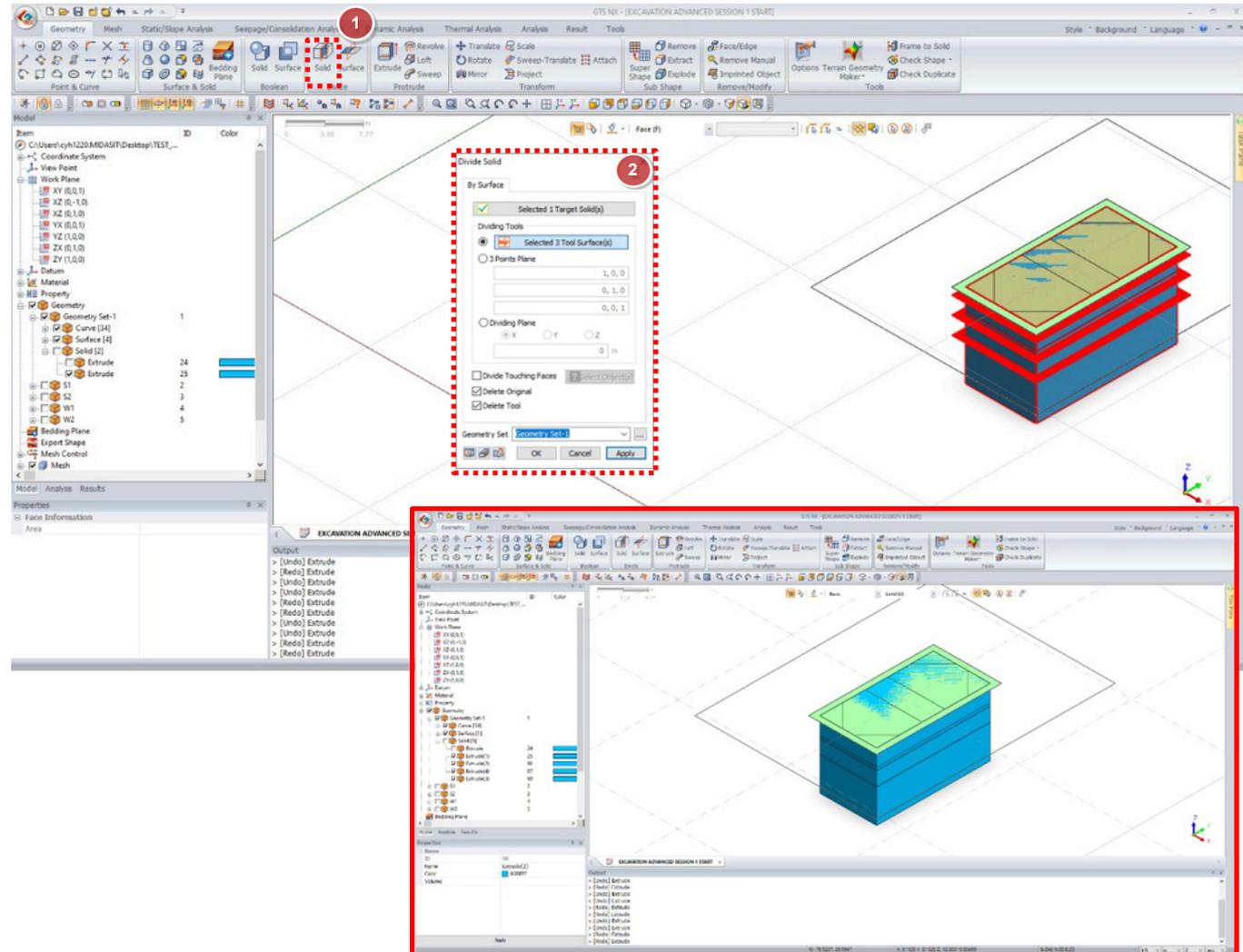


03 Geometry Modeling

Define the excavation depth

Procedure

- 1 Click "Solid" from Divide
Select Solid (1 Solid)
- 2 Select faces (3 faces)
Click "OK"
[Information]
Solid will be divided by selected 3 faces.

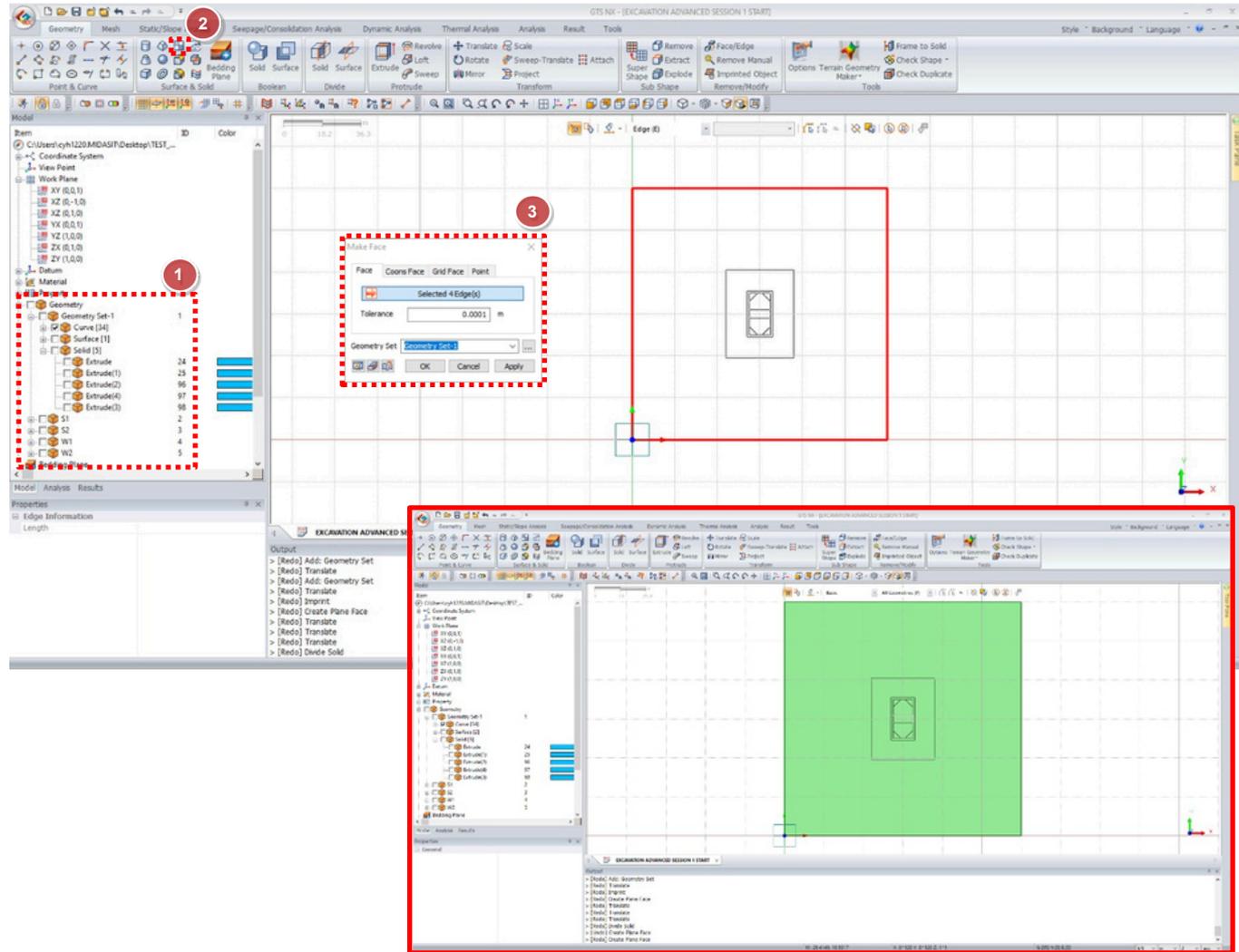


03 Geometry Modeling

Definethesoil layers

Procedure

- 1 Only show Curve
- 2 Click "Make Face"
- 3 Select 4 edges
Click "OK"

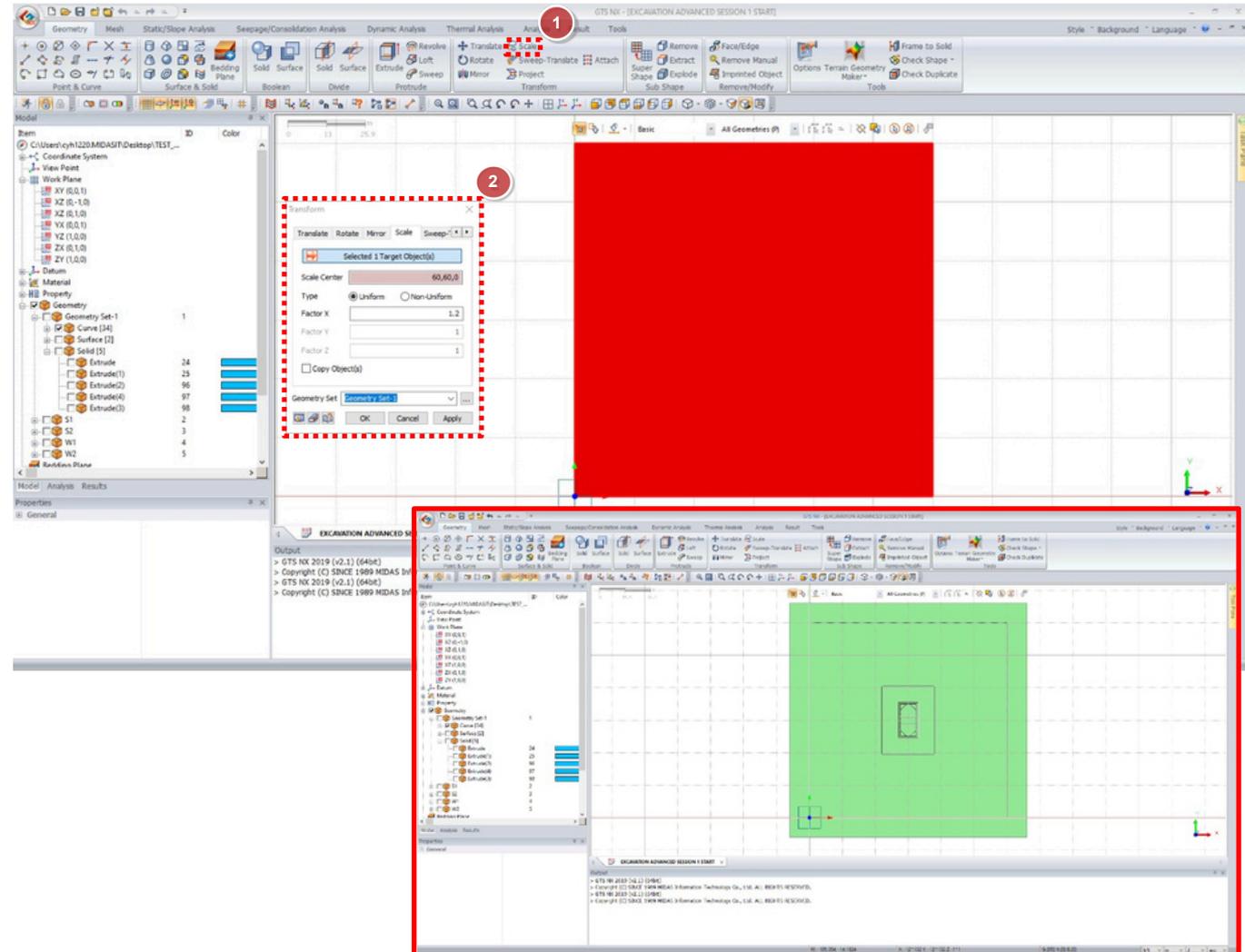


03 Geometry Modeling

Definethesoil layers

Procedure

- 1 Click "Scale"
- 2 Select 1 face
Scale Center "60,60,0"
Factor X "1.2"
Click "OK"

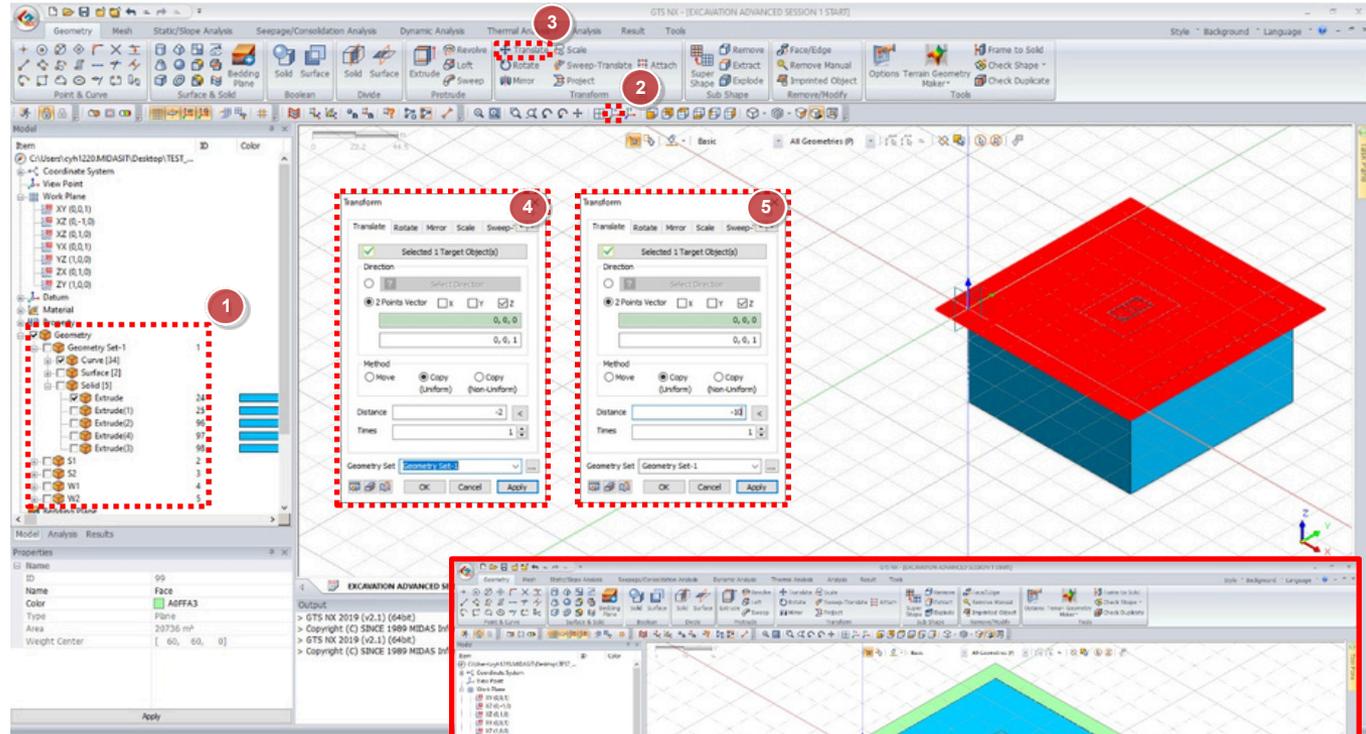


03 Geometry Modeling

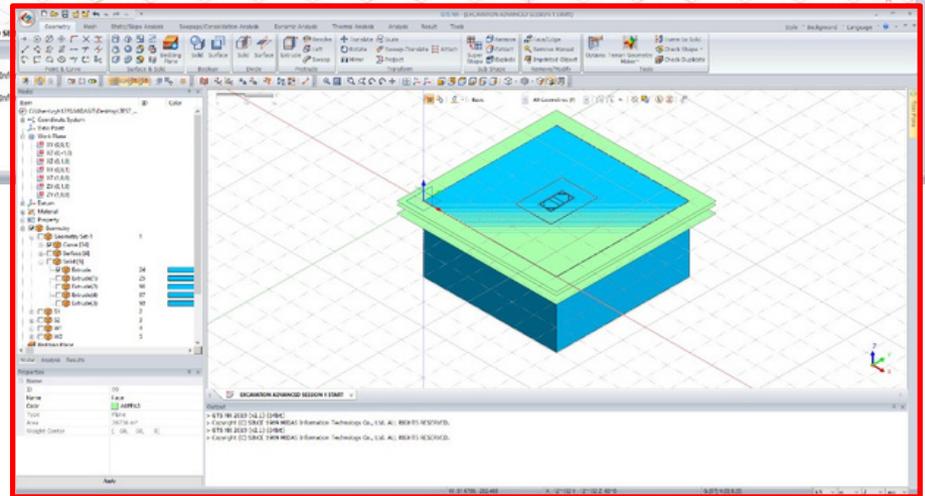
Definethesoil layers

Procedure

- 1 Show extrude solid (1st)
Click "Isometric 1"
- 2 Click "Translate"
Select 1 face
- 3 Select "2 Points Vector"
Select "Z" only
- 4 Select "Copy"
Distance "-2"
Click "Apply"



5

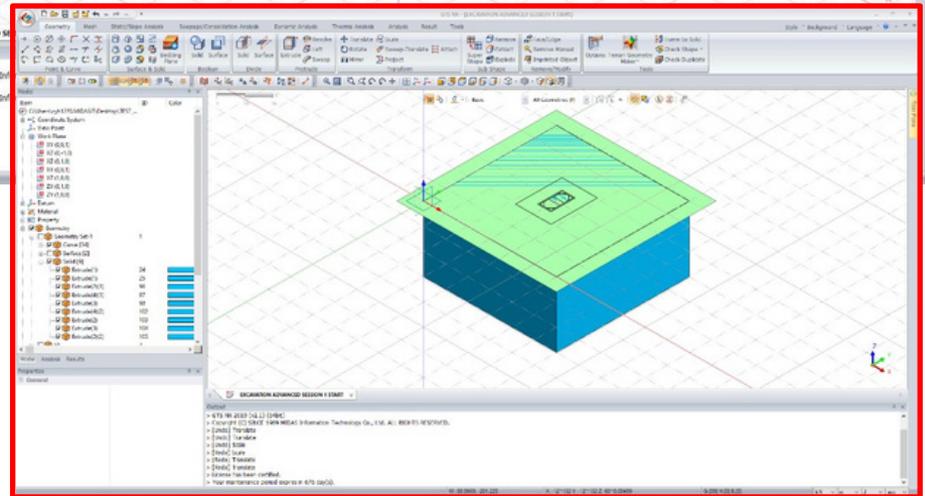
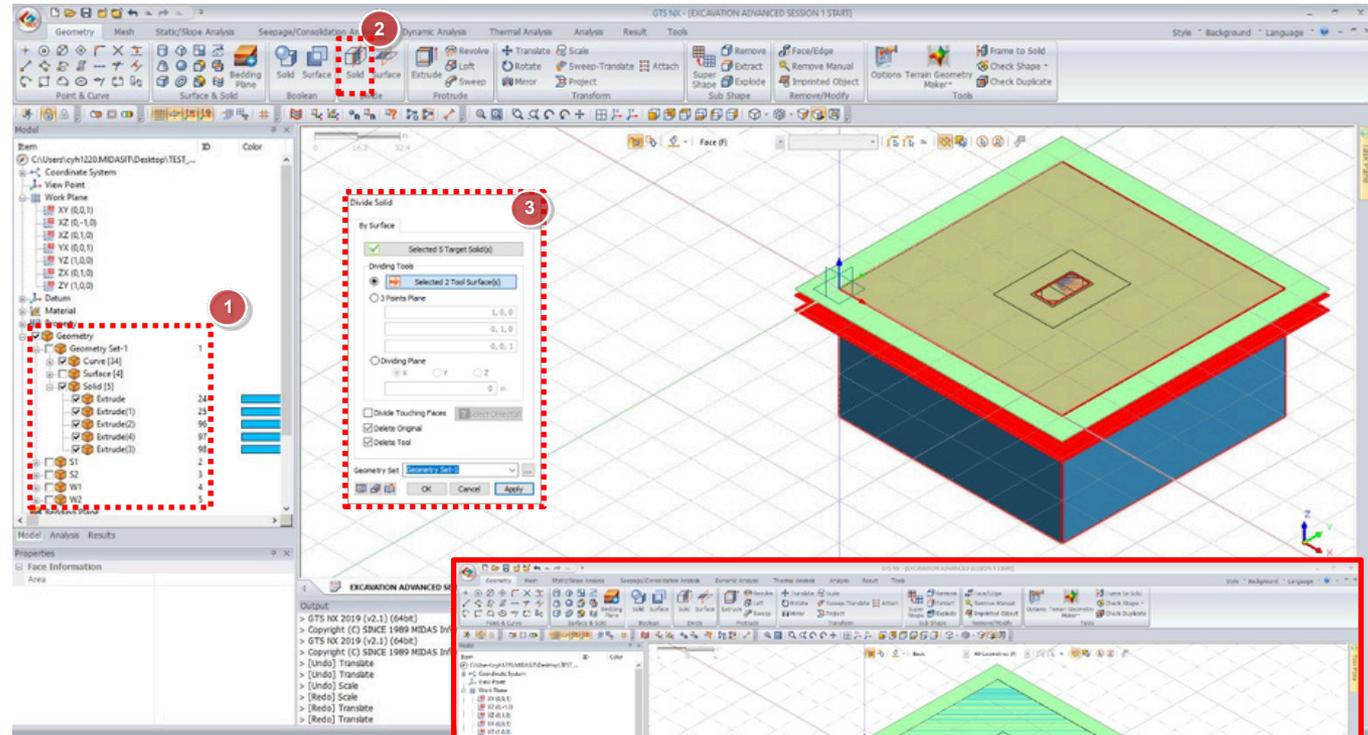


03 Geometry Modeling

Definethesoil layers

Procedure

- 1 Show every solid
- 2 Click "Solid"
- 3 Select 5 Solids
Select 2 Faces
Click "OK"



03 Geometry Modeling

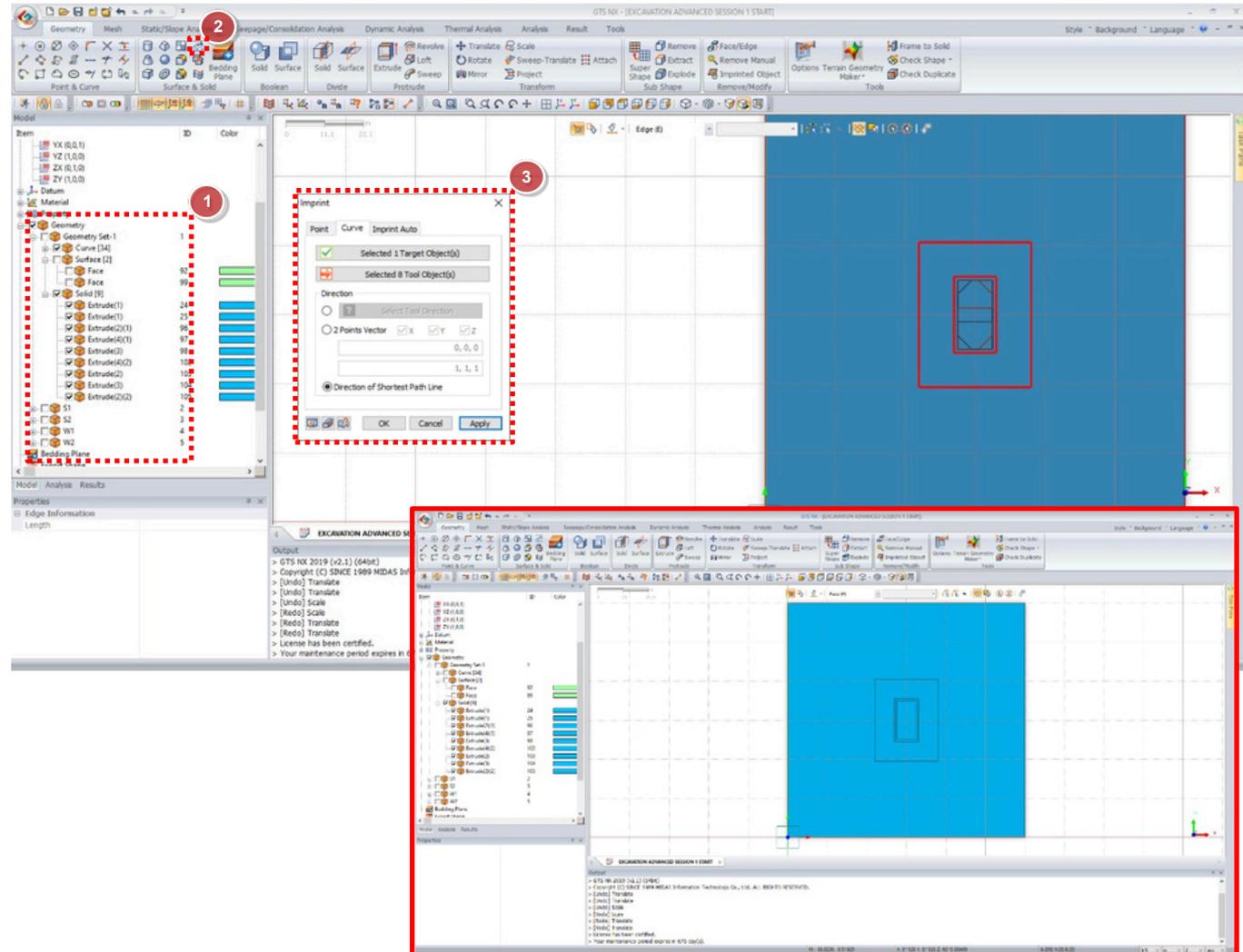
Imprintsurcharge area to ground surface

Procedure

- 1 Show every Solid & Curve
- 2 Click "Imprint"
- 3 Go to "Curve" tab
Select ground surface (1 face)
Select 8 edges
Select "Direction of shortest.."
Click "OK"

[Information]

You can see imprinted lines on the ground surface when you hide every curve which means the ground surface is divided by lines.

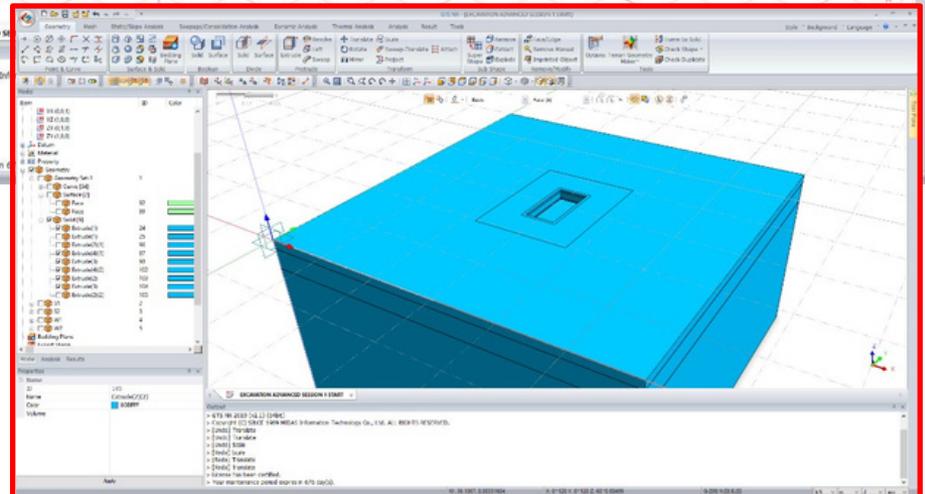
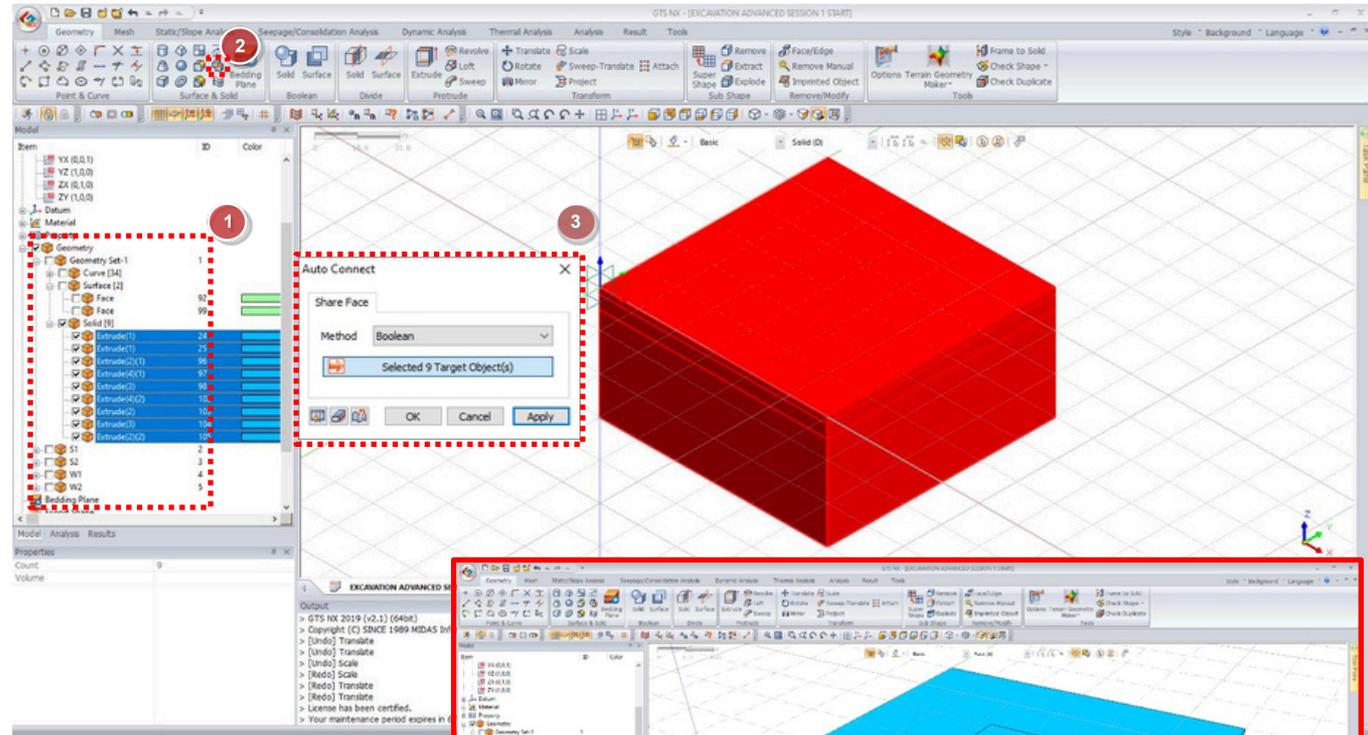


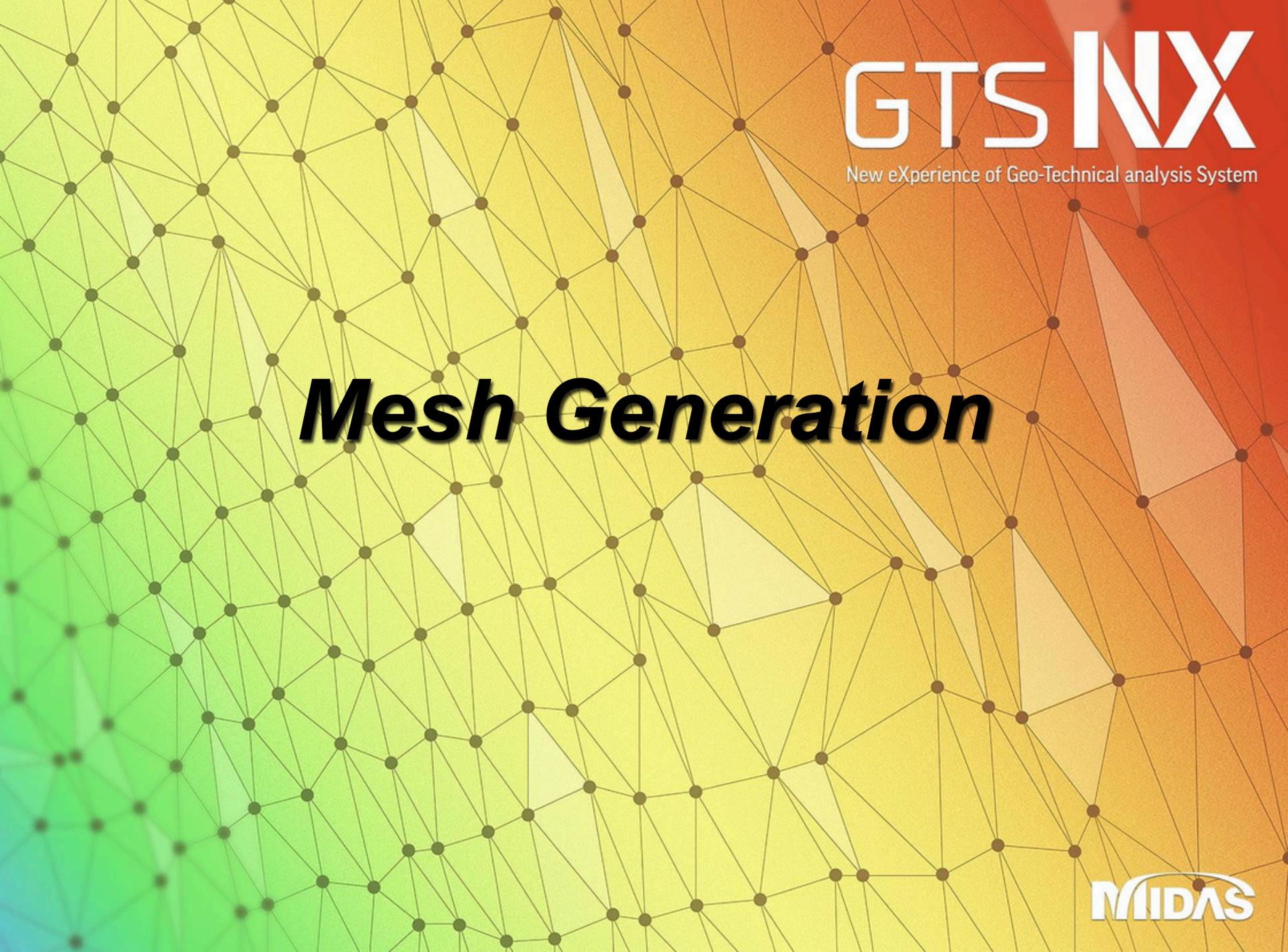
03 Geometry Modeling

Eliminate the overlapping solids

Procedure

- 1 Show every Solid only
- 2 Click "Auto Connect"
- 3 Method "Boolean"
Select 9 solids (ctrl+a, select all)
Click "OK"



The background of the slide is a complex, interconnected mesh of thin grey lines forming a network of irregular polygons. Small dark grey dots are placed at the vertices of this network. The background color transitions from a light green on the left to a bright yellow in the center, and finally to a deep orange-red on the right.

GTS NX

New eXperience of Geo-Technical analysis System

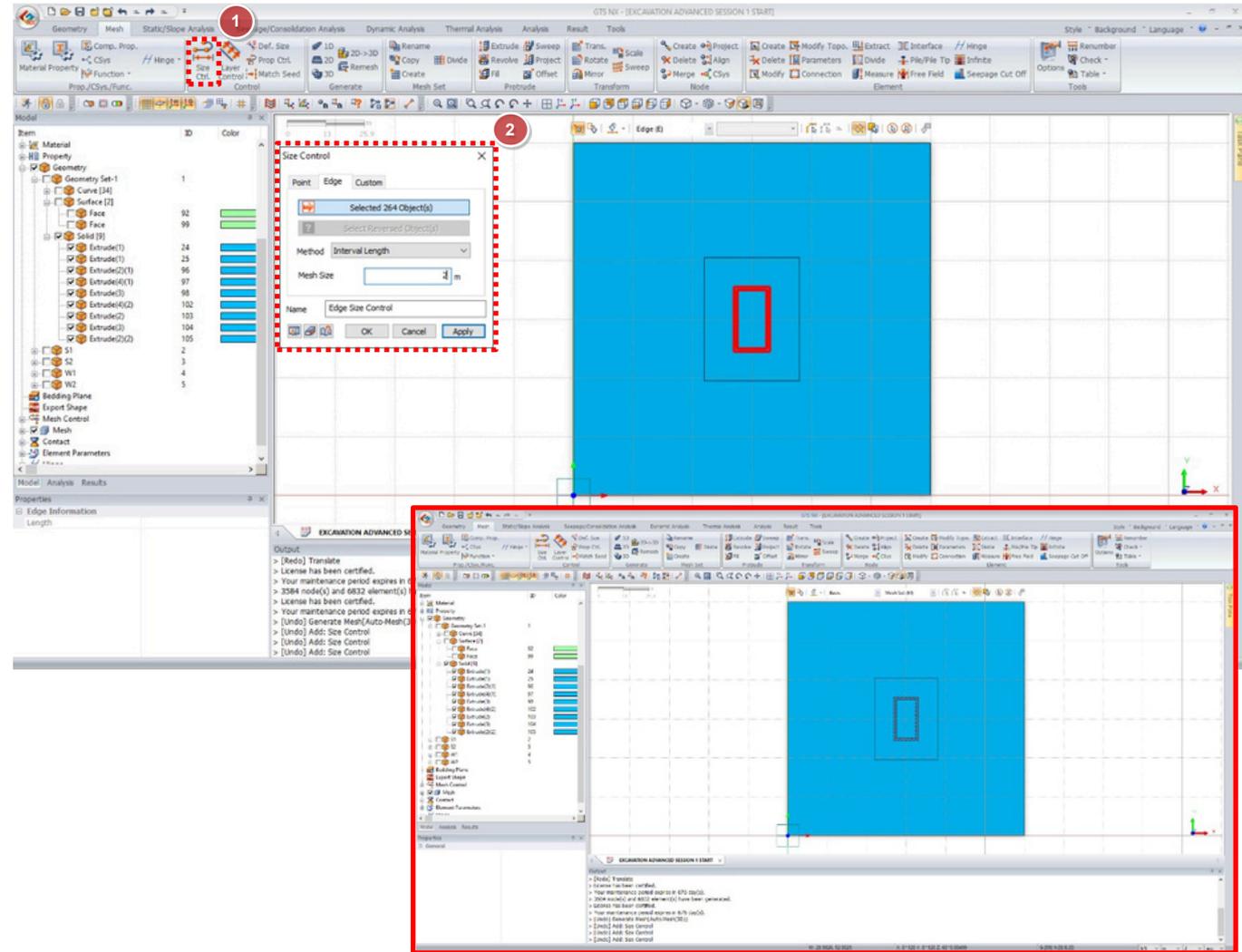
Mesh Generation

04 Mesh Generation

Definemesh size before meshing

Procedure

- 1 Click "Size Ctrl."
Select 264 edges
- 2 Mesh Size "2"
Click "OK"
[Information]
You can define mesh size when you generate the mesh also.

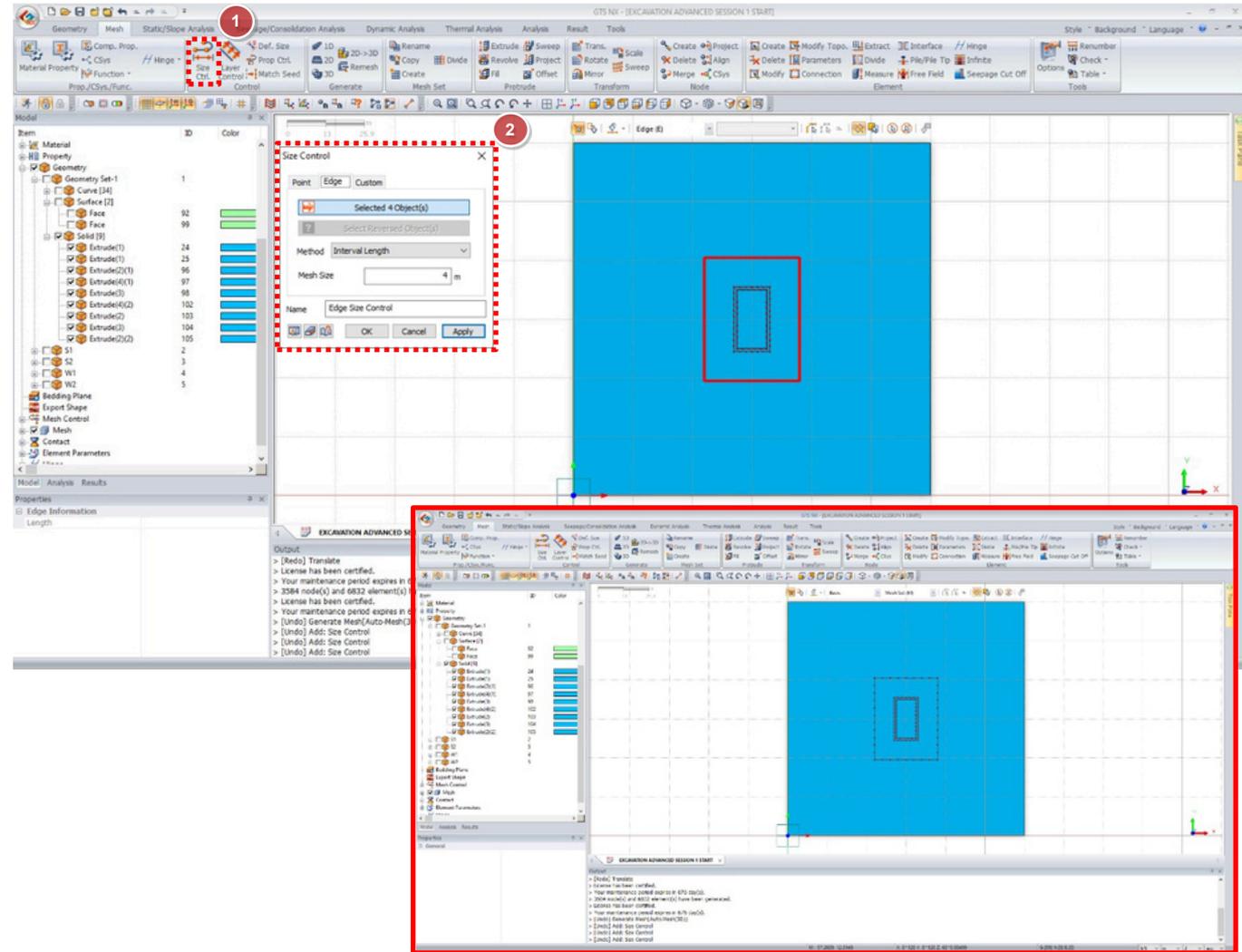


04 Mesh Generation

Definemesh size before meshing

Procedure

- 1 Click "Size Ctrl."
Select 4 edges
- 2 Mesh Size "4"
Click "OK"
[Information]
You can define mesh size when you generate the mesh also.

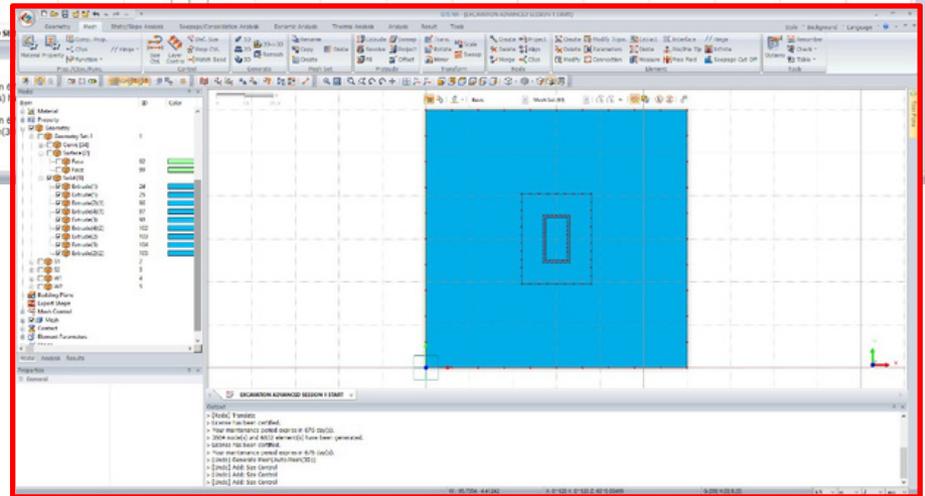
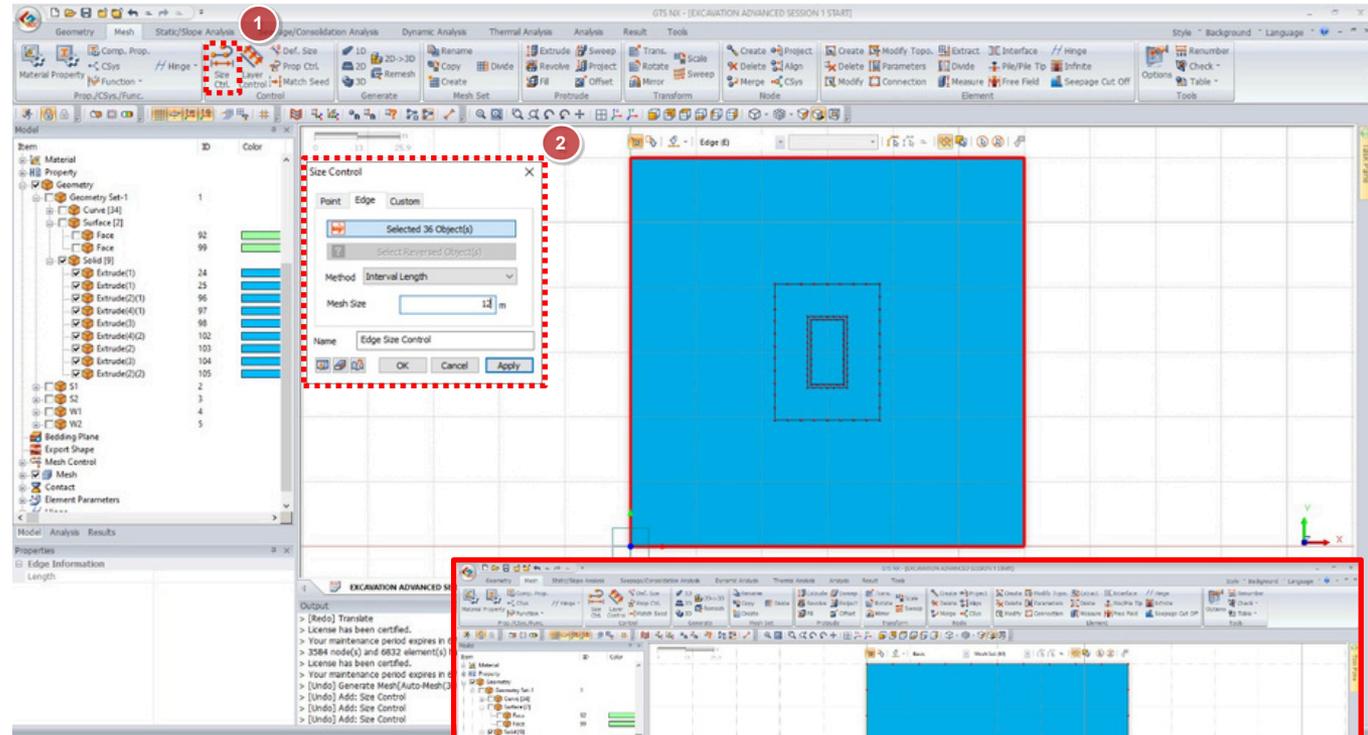


04 Mesh Generation

Definemesh size before meshing

Procedure

- 1 Click "Size Ctrl." Select 36 edges Mesh Size "12" Click "OK"
- 2 [Information] You can define mesh size when you generate the mesh also.

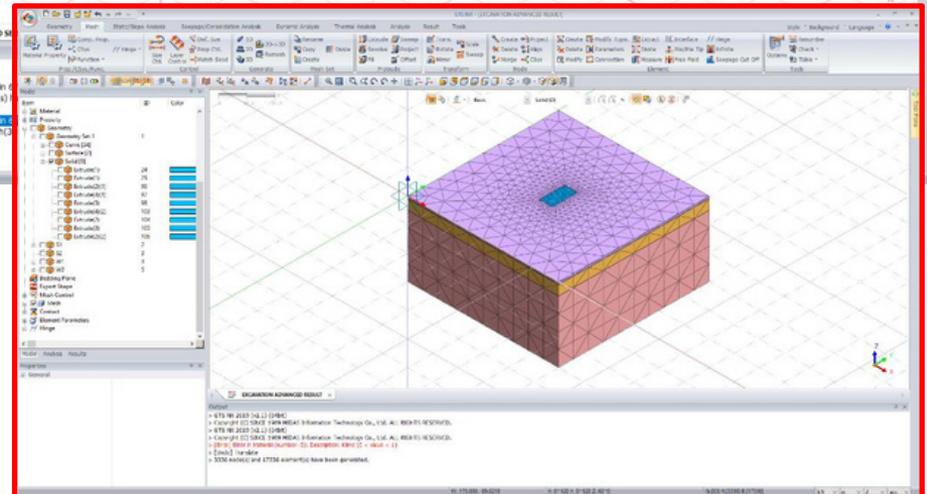
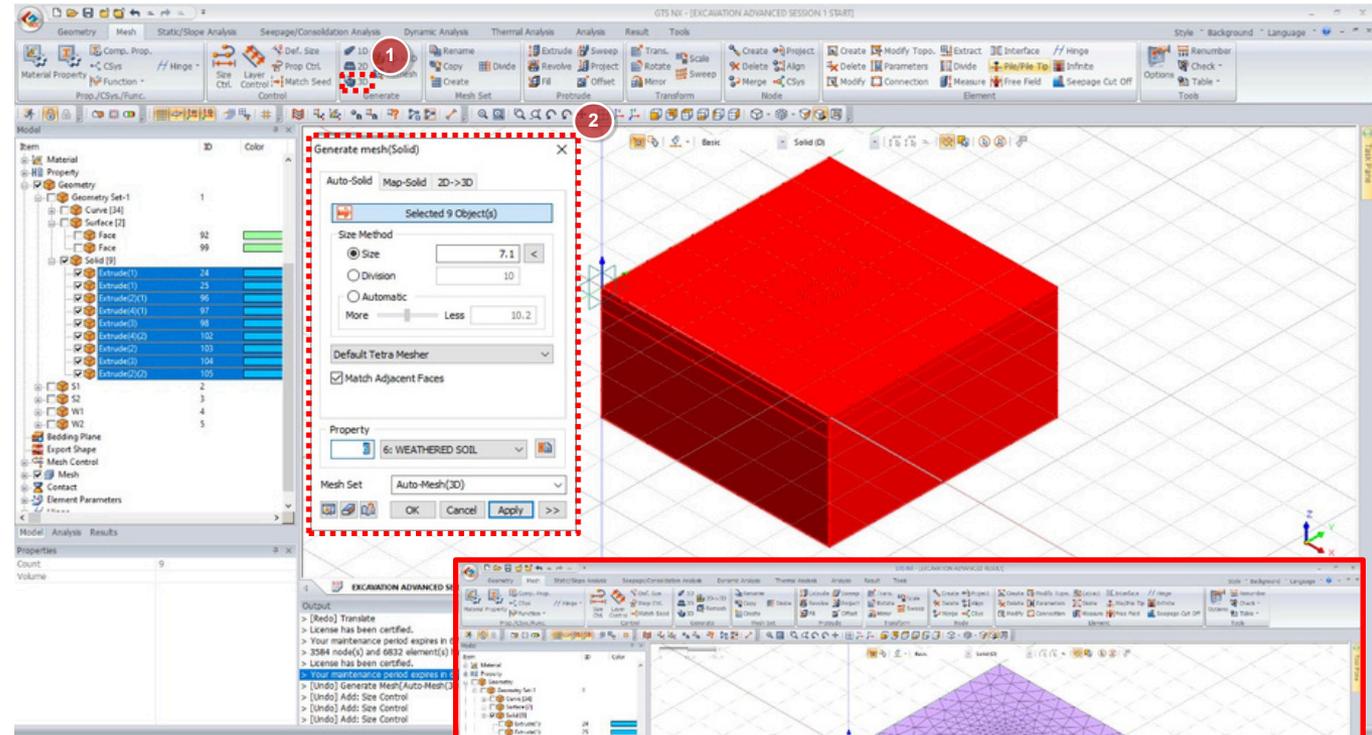


04 Mesh Generation

Generate mesh for ground

Procedure

- 1 Click "3D"
Select 9 solids (ctrl+a, select all)
- 2 Select "Default Tetra Mesher"
Property "WEATHERED SOIL"
Click "OK"
[Information]
Mesh size has defined previous stage so that the mesh will be automatically generated with assigned mesh size.



04 Mesh Generation

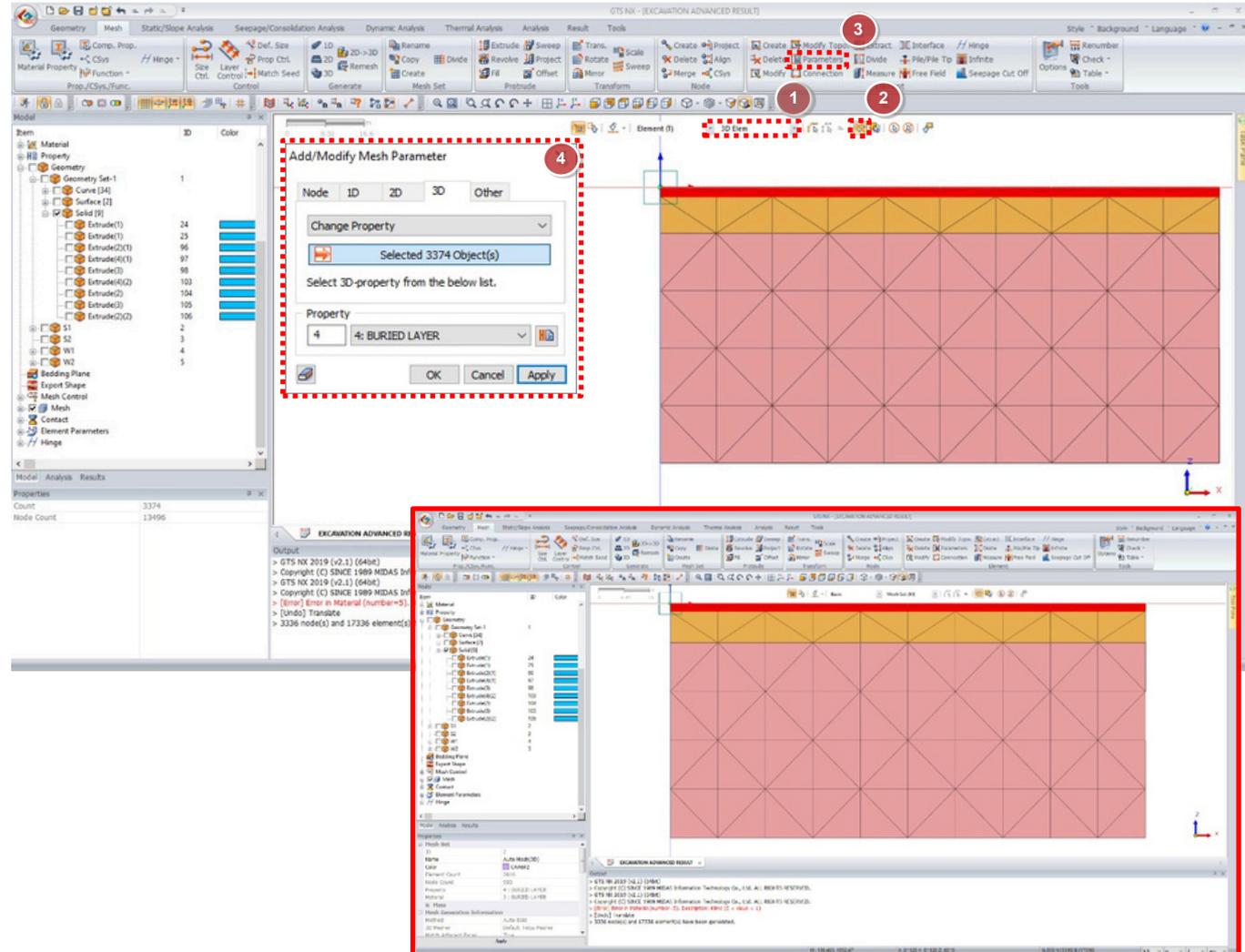
Define properties for soil layer

Procedure

- 1 Change selection method to "Mesh Set"
Turn on "Intersect"
- 2 Click "Parameters" Go to "3D" tab Select first layer
- 3 Property "BURRIED LAYER"

4

Click "Apply"

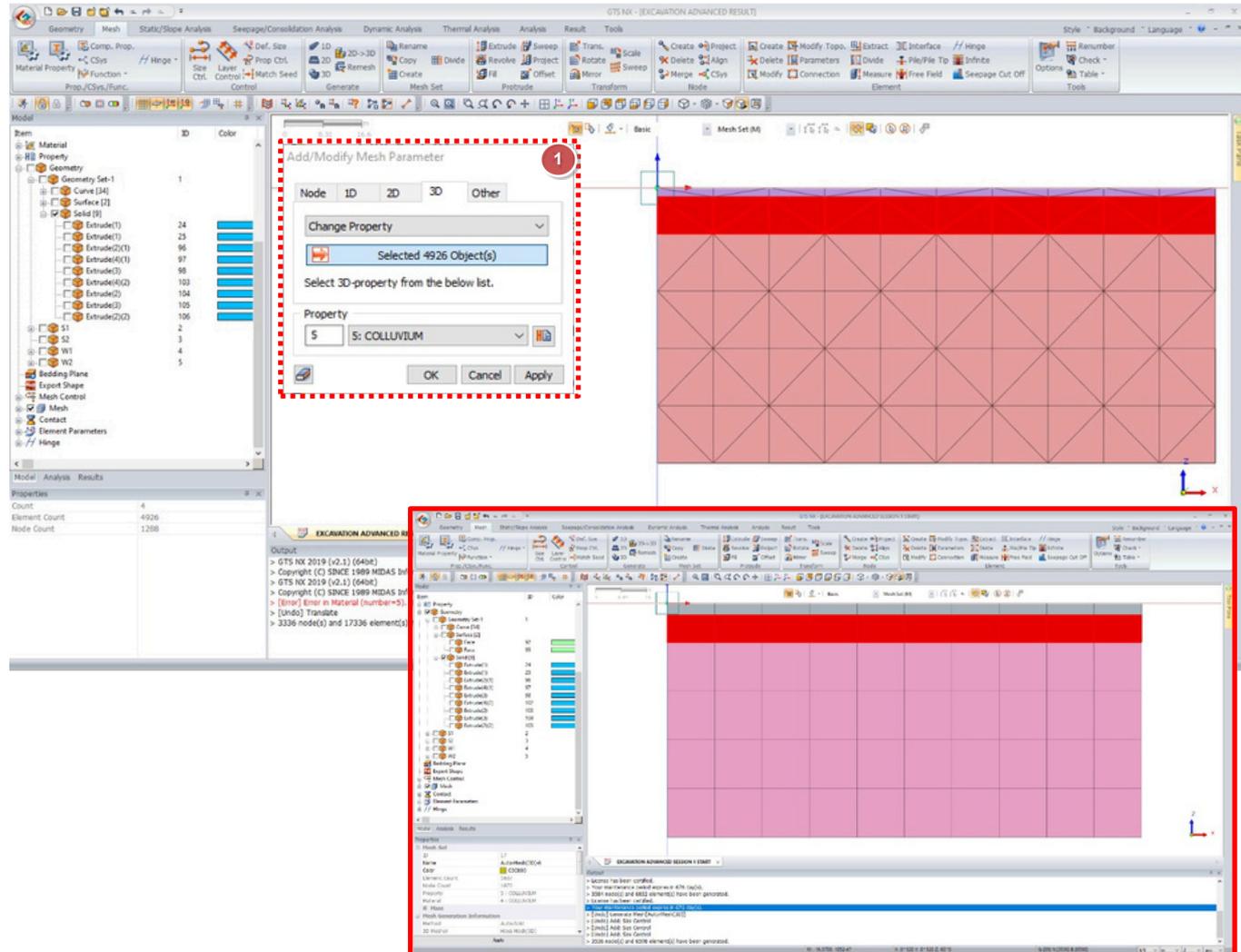


04 Mesh Generation

Define properties for soil layer

Procedure

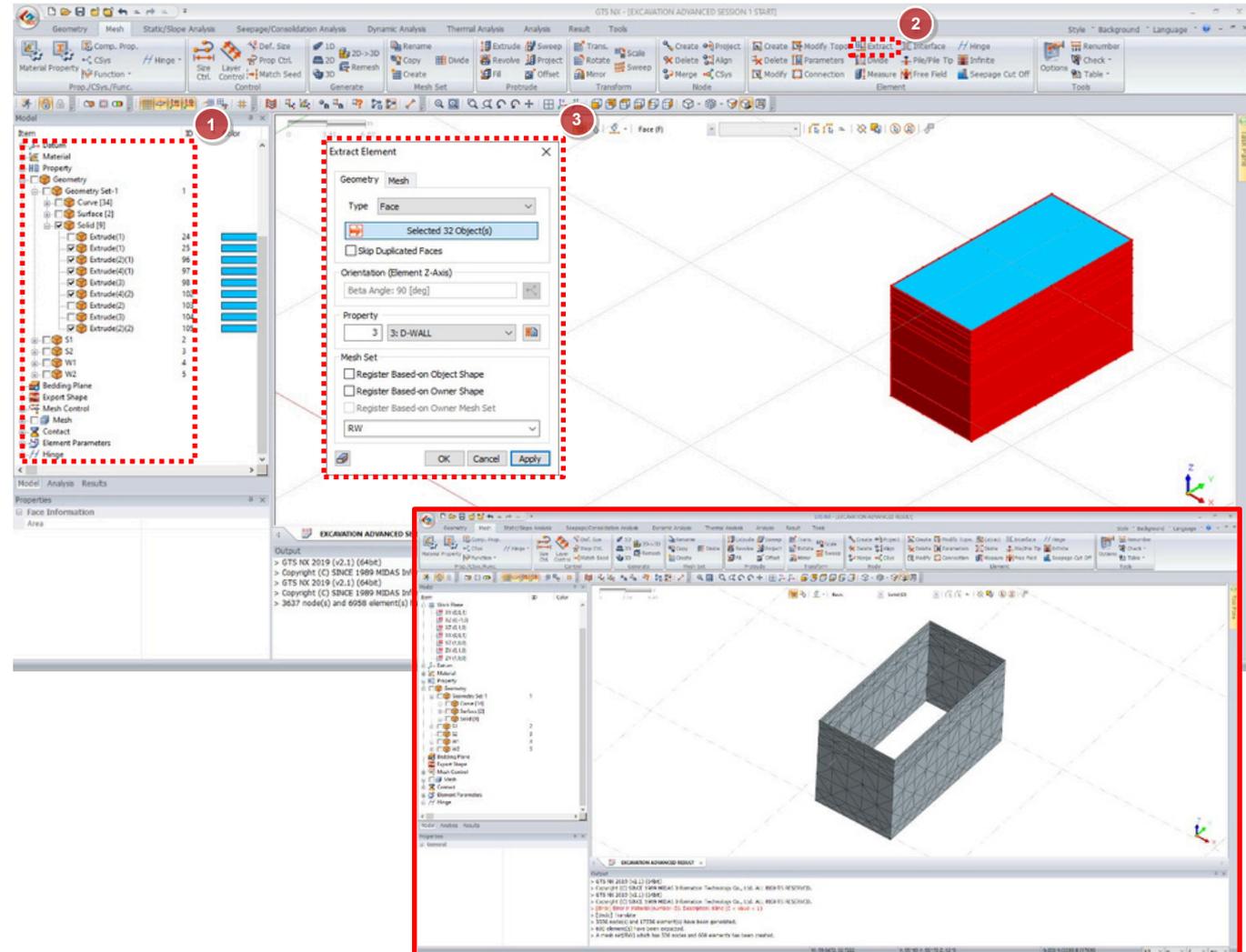
- 1 Select second layer
Property "COLLUVIUM"
Click "OK"
[Information]
The rest part already assigned with "WEATHERED SOIL" when you generate mesh.
You can check assigned property under properties window which is located left below.



Generate mesh for retaining wall

Procedure

- 1 Hide every mesh set
Show only inside solids
Click "Extract"
- 2
- 3 Type "Face"
Select 32 faces
Property "D-WALL"
Mesh set name "RW"
Click "OK"

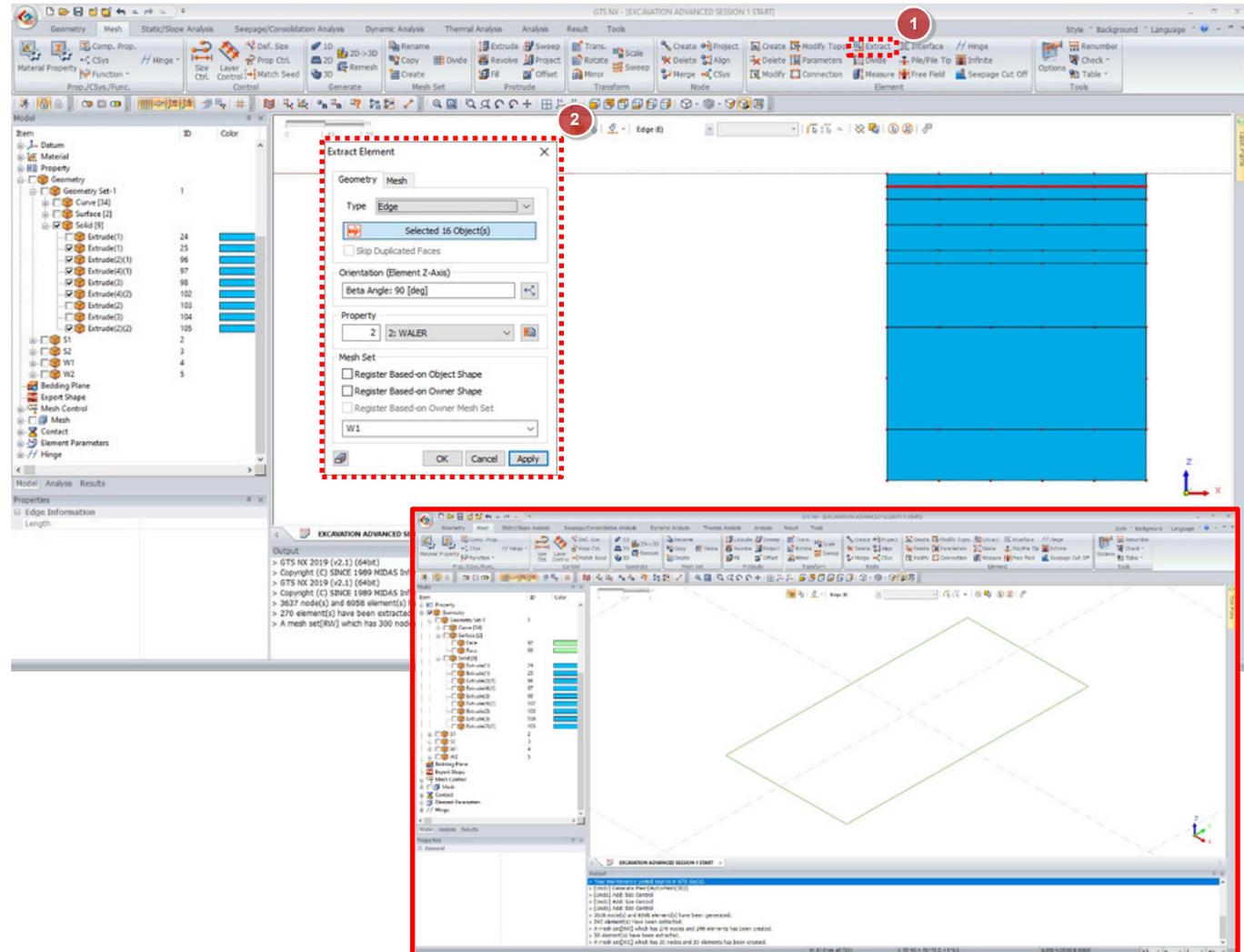


04 Mesh Generation

Generate mesh for water (1st layer)

Procedure

- 1 Click “Extract”
Type “Edge”
- 2 Select 16 lines
Property “WALER”
Mesh set name “W1”
Click “Apply”

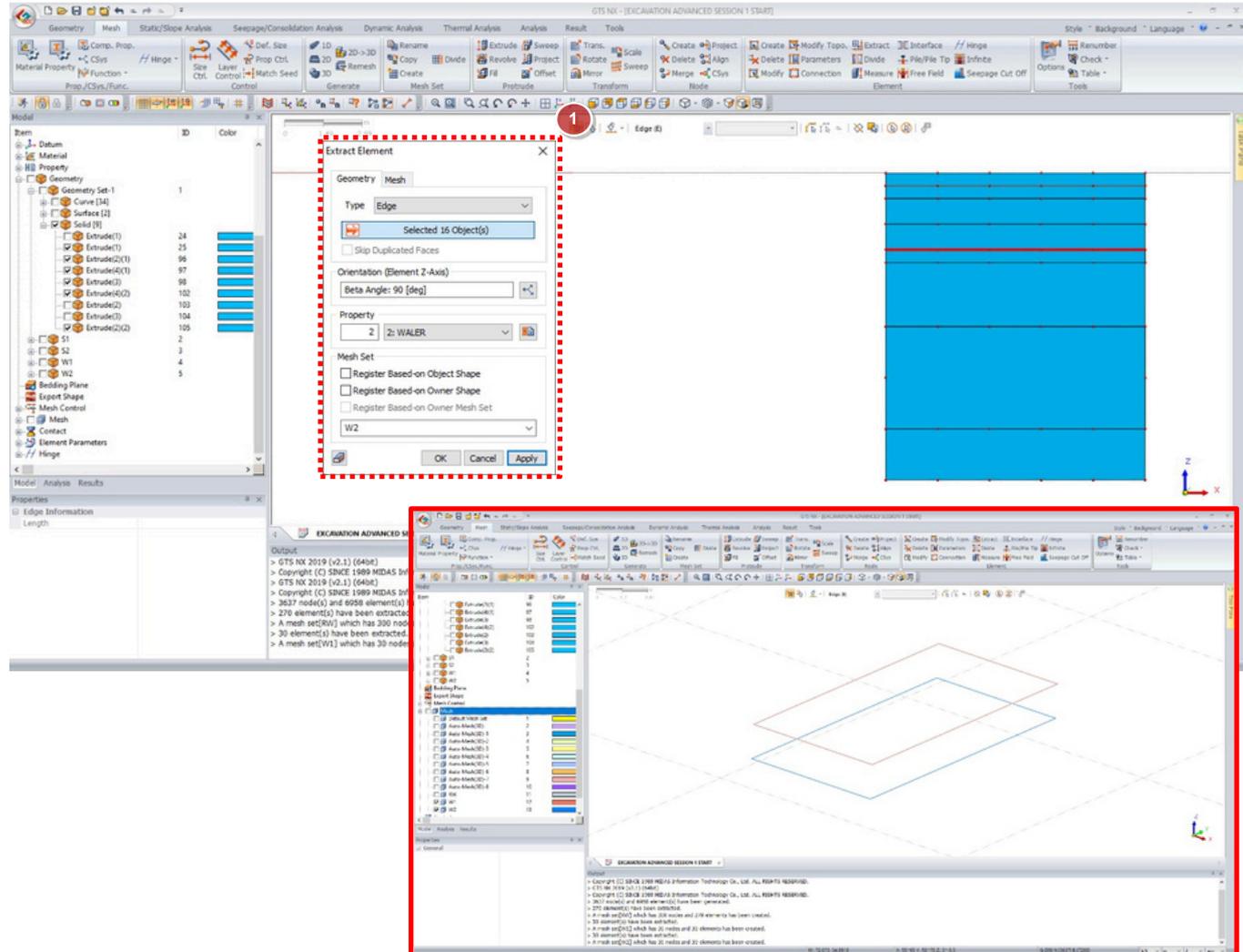


04 Mesh Generation

Generate mesh for water (2nd layer)

Procedure

- 1 Type "Edge"
Select 16 lines
Property "WALER"
Mesh set name "W2"
Click "OK"

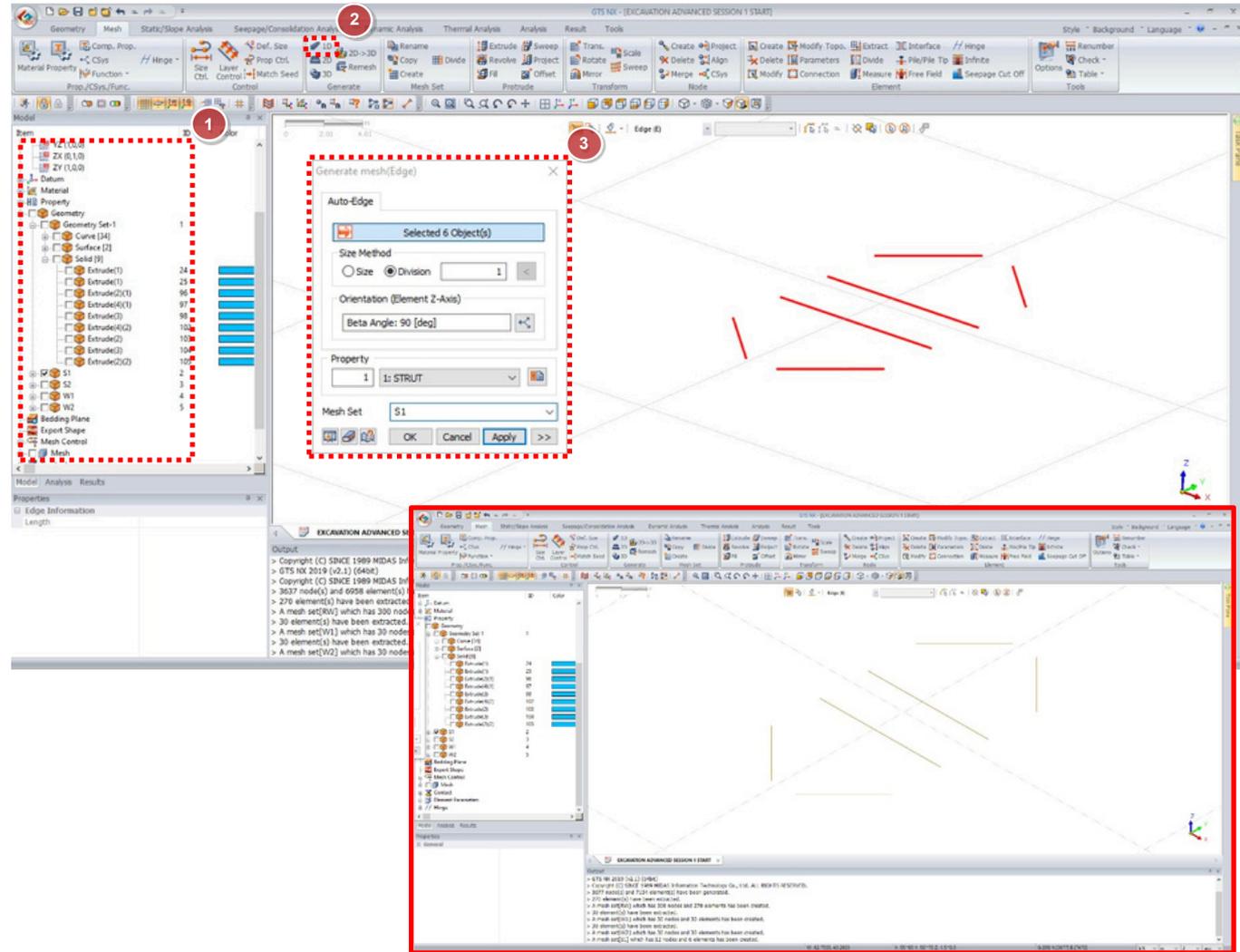


04 Mesh Generation

Generatemesh for Strut (1st layer)

Procedure

- 1 <Mesh tree>
Hide every mesh set
<Geometry tree>
Show S1 only
- 2 Click "1D"
Select 6 lines (ctrl+a, select all)
- 3 Division "1"
Property "Strut"
Mesh Set name "S1"
Click "Apply"



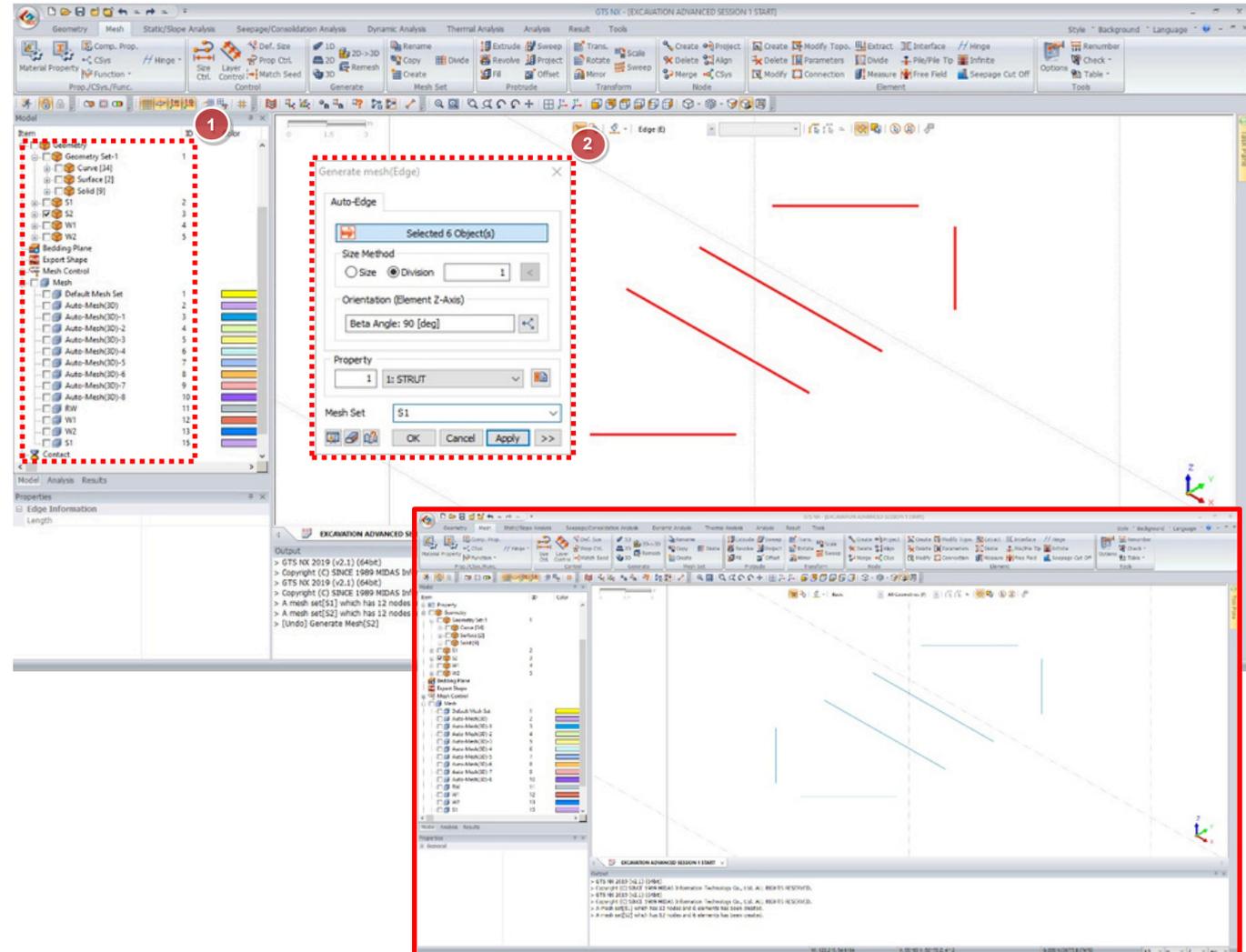
04 Mesh Generation

Generate mesh for Strut (2nd layer)

Procedure

- 1 <Mesh tree>
Hide every mesh set
<Geometry tree>
Show S2 only

Select 6 lines (ctrl+a, select all)
Division "1"
Property "Strut"
Mesh Set name "S2"
Click "OK"



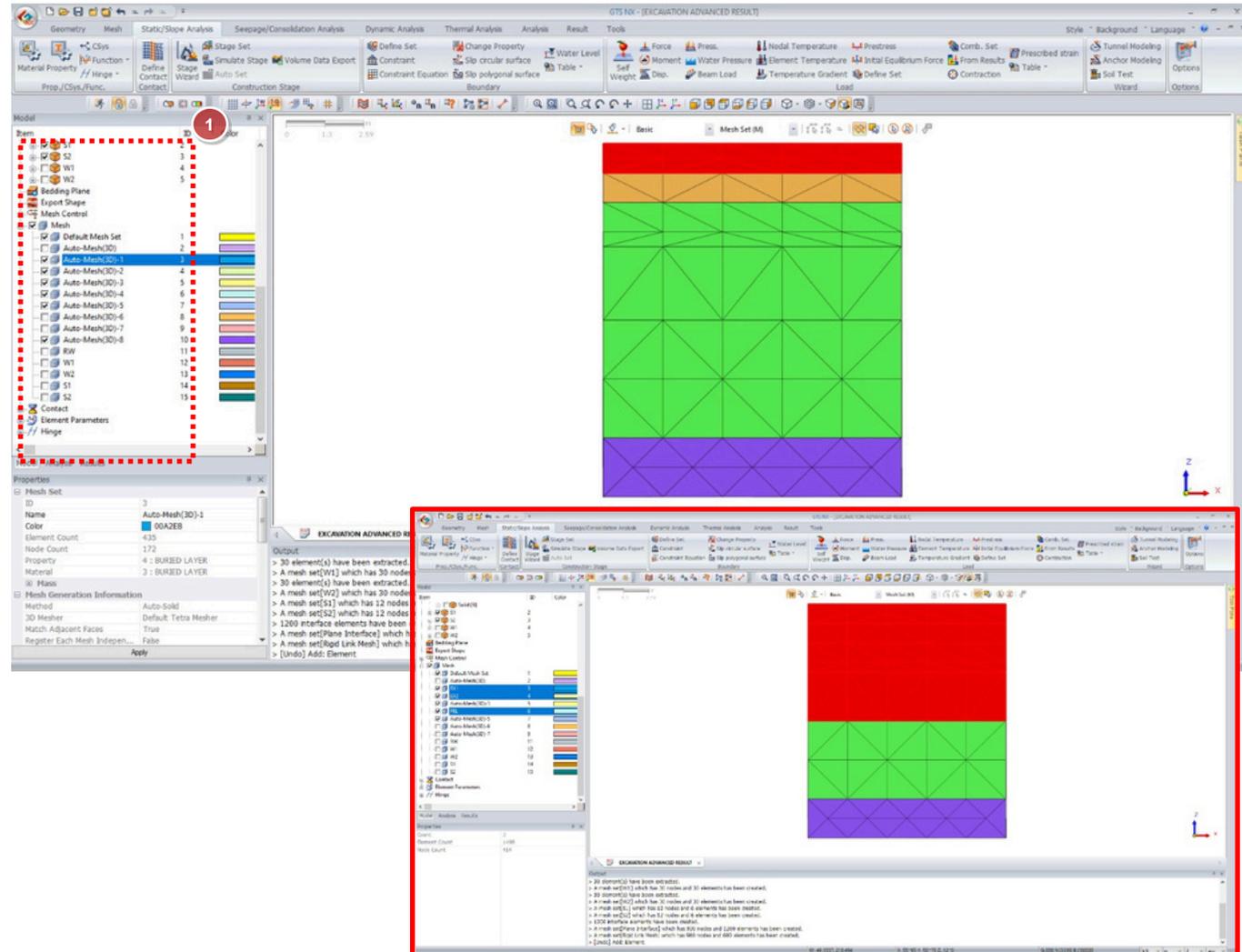
Mergemesh set and change name of mesh set

Procedure

- <Mesh tree>
 Show only excavation part
 <Geometry tree>
 Hide every geometry

Selection method change “Mesh se” press “M”
 Select first layer

Change mesh set name to “EX 1” under mesh tree using “F2” key
 Select 2nd& 3rdlayer
 Right click
 Select “Merge”
 Change mesh set name to “EX2” under mesh tree using “F2” key
 Select 4thlayer
 Change mesh set name to “FEL” under mesh tree using “F2”key



GTS NX

New eXperience of Geo-Technical analysis System

Load & Boundary Condition

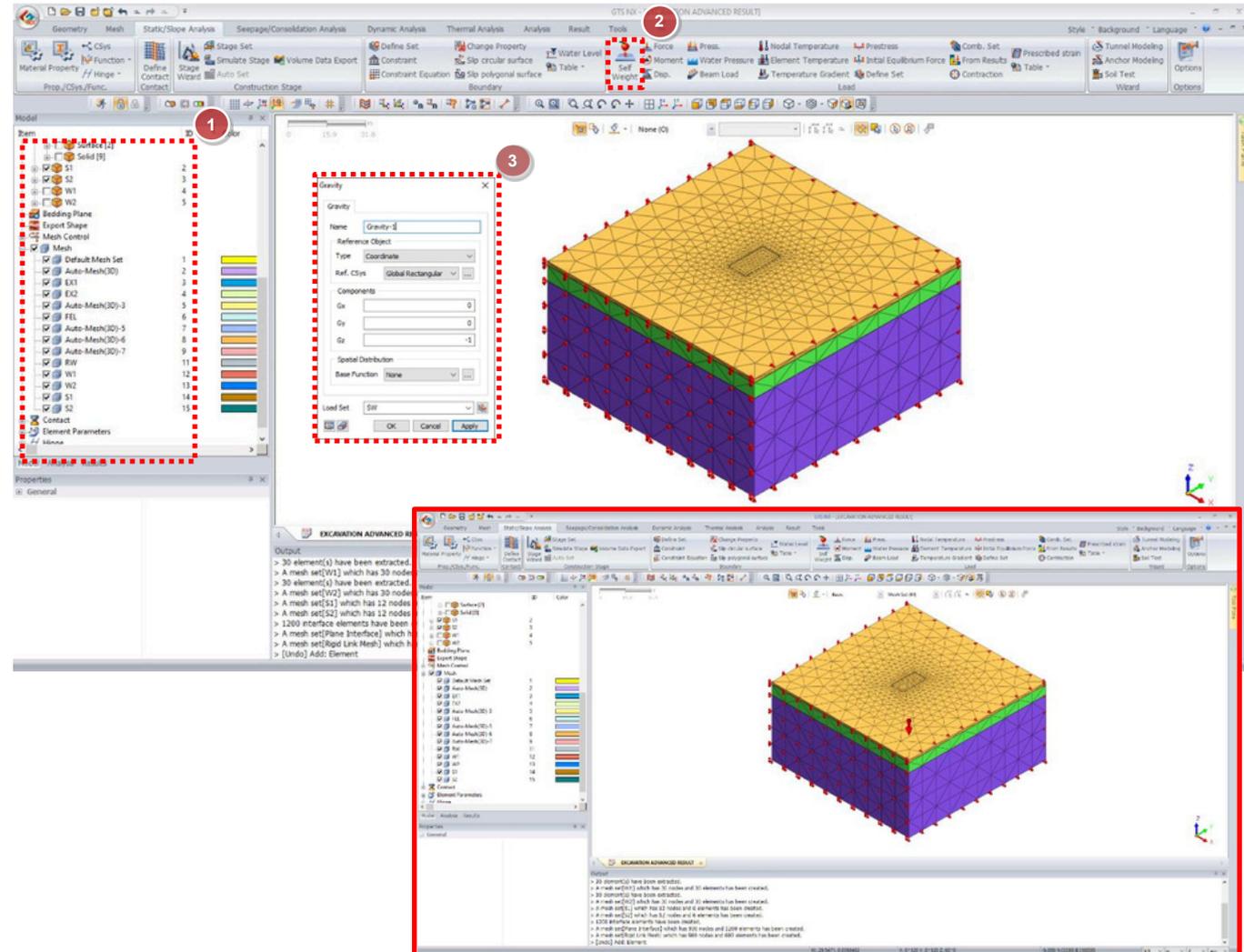
MIDAS

05 Load & Boundary Condition

Make self weight

Procedure

- 1 <Mesh tree>
Show every mesh set
<Geometry tree>
Hide every geometry
- 2 Click "Self Weight"
Load Set name "SW"
- 3 Click "OK"



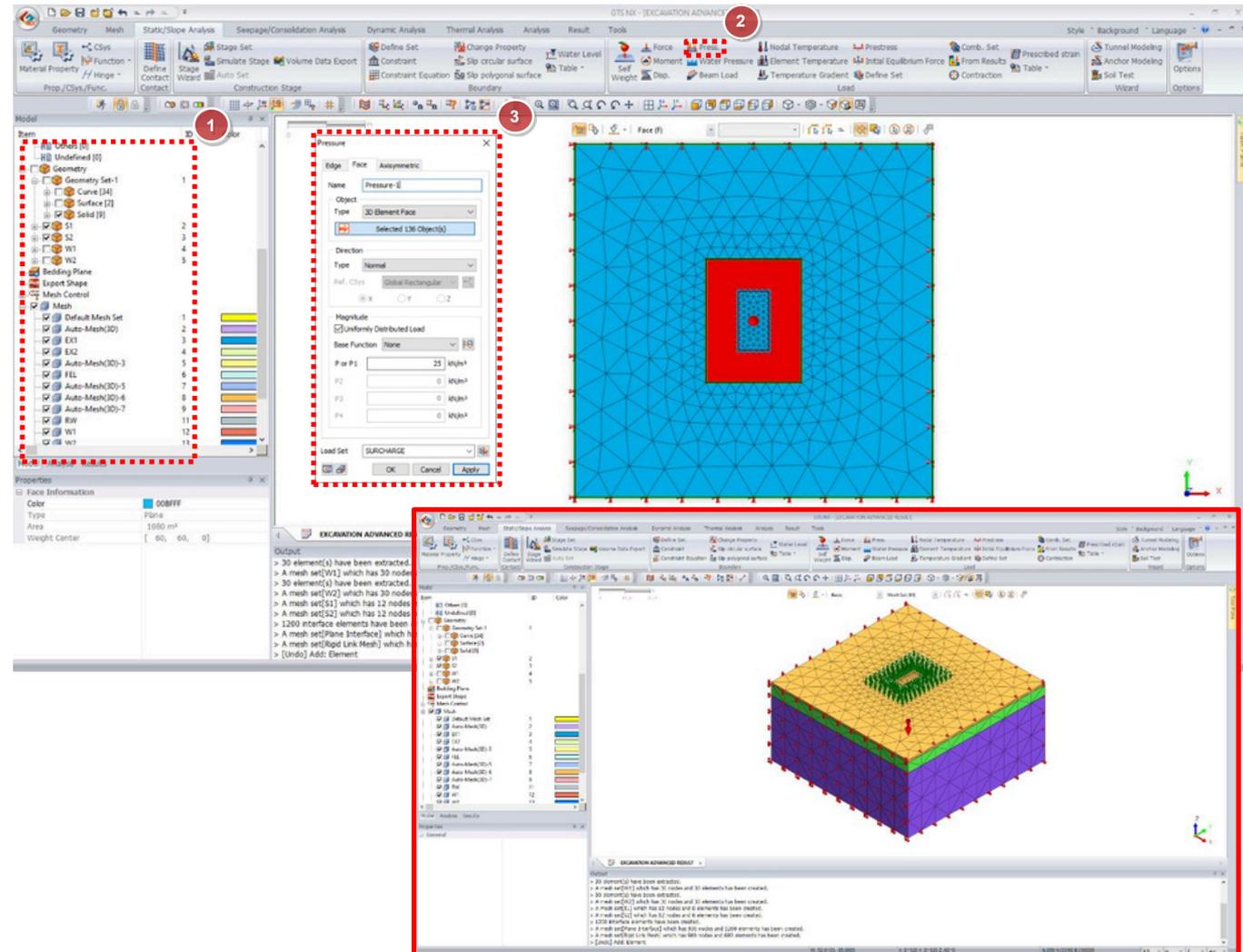
05 Load & Boundary Condition

Make surcharge

Procedure

- 1 <Mesh tree>
Show every mesh set
<Geometry tree>
Show every solid
- 2 Click "Press."
Go to "Face" tab
- 3 Object type "3D Element Face"

Change selection method
change to "Face" Press "F"
P or P1 "25"
Load Set name "SURCHARGE"
Click "OK"



The background features a complex network of interconnected nodes and lines, resembling a mesh or a molecular structure. The nodes are small dark circles, and the lines are thin, light-colored. The overall color scheme transitions from a vibrant green on the left to a warm orange and red on the right.

GTS NX

New eXperience of Geo-Technical analysis System

Water Level

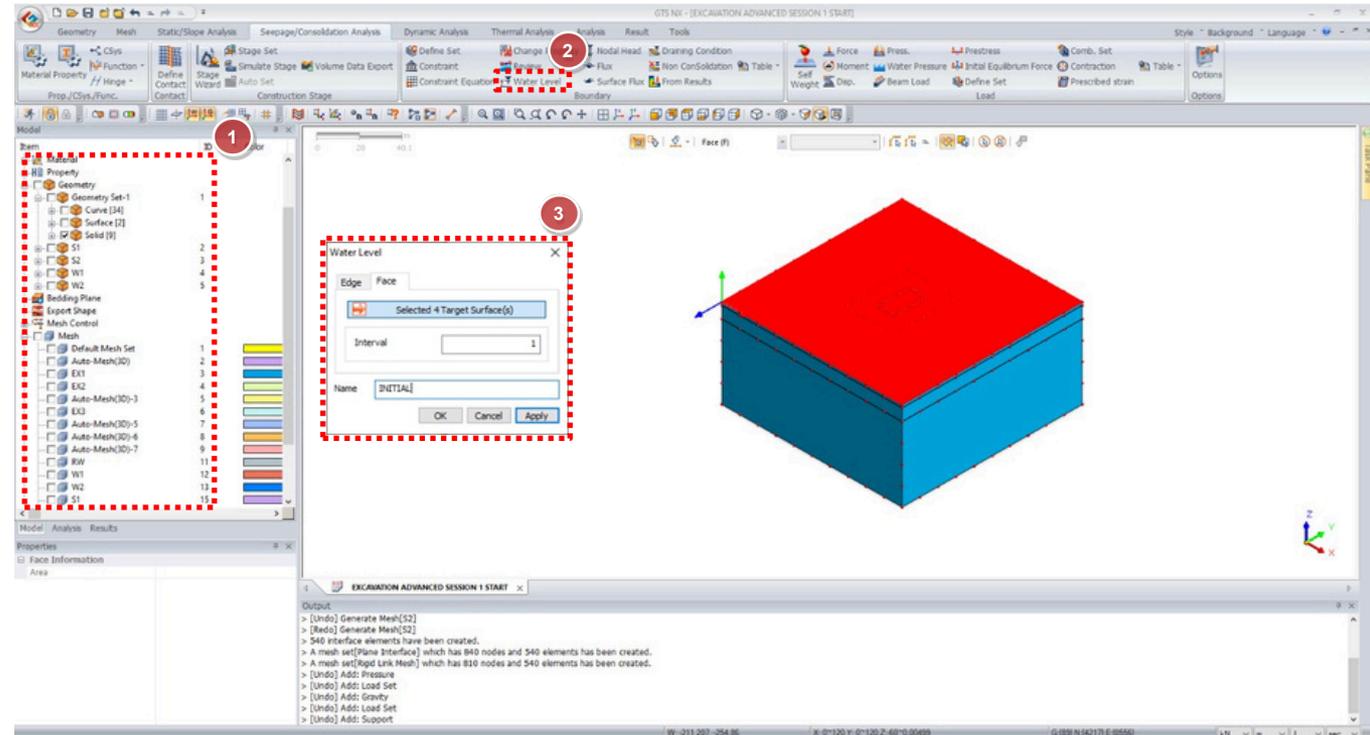
MIDAS

Define water levels for each excavation stage

Procedure

- 1 <Mesh tree>
Hide every mesh set
<Geometry tree>
Show every solid
- 2 Click “Water Level”
Go to “Face” tab
- 3 Select ground surface
Name “Initial”

Click “OK”

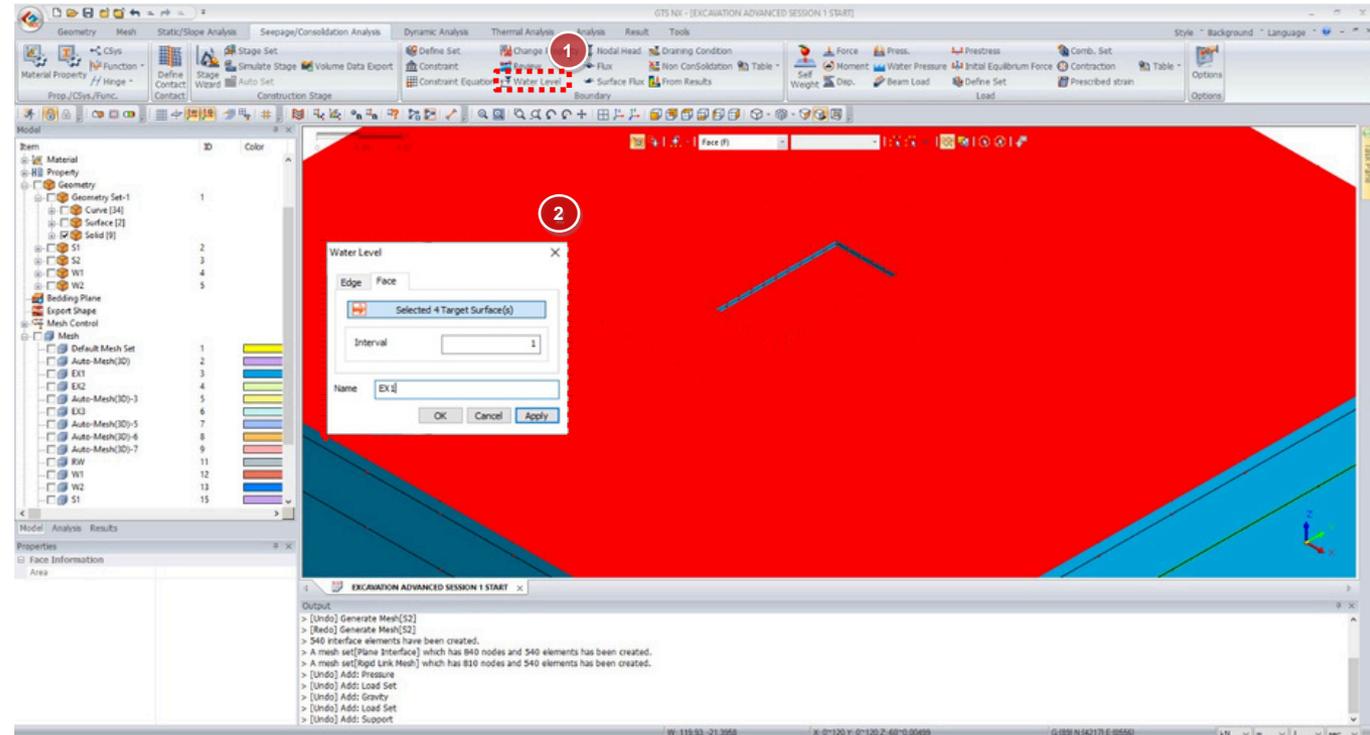


Define water levels for each excavation stage

Procedure

- 1 Click “Water Level”
Go to “Face” tab

Select ground surface and EX1 floor
Name “EX1”
Click “OK”
[Information]
Excavation floor need to be selected at the end.

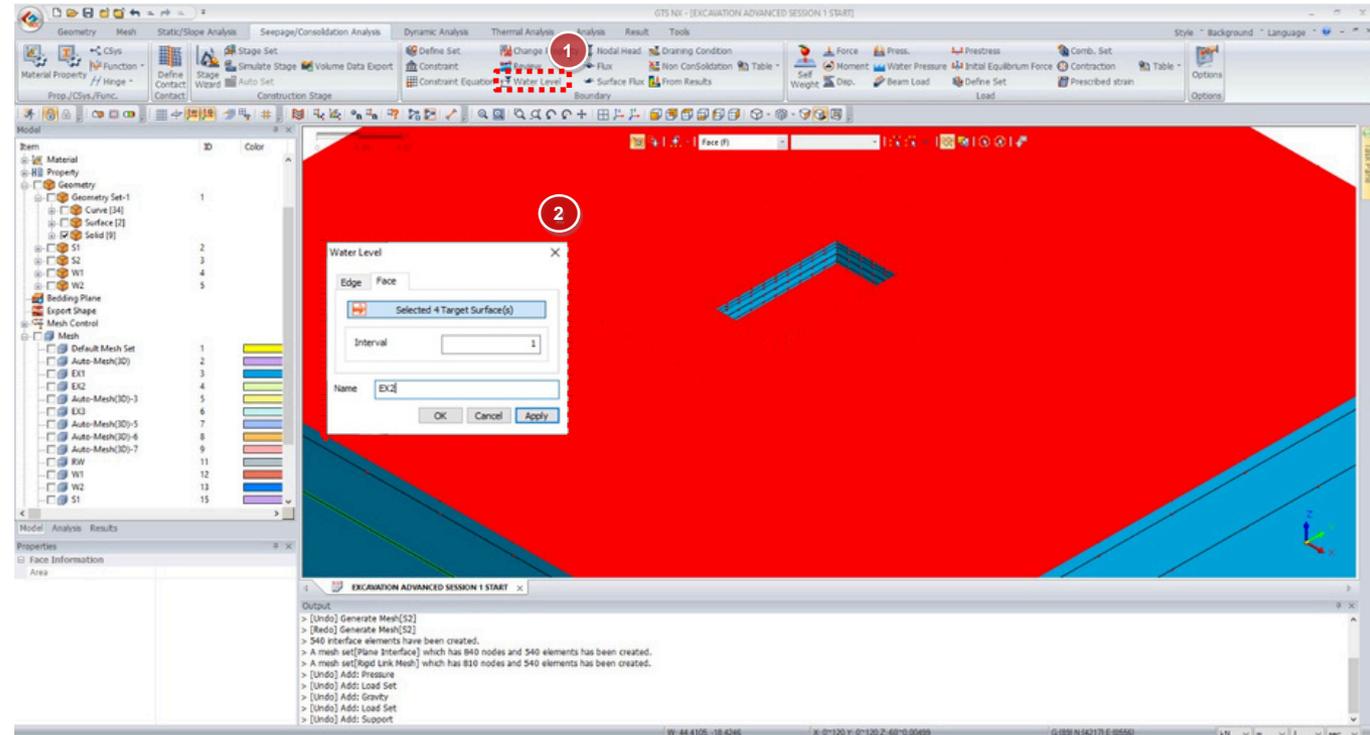


Define water levels for each excavation stage

Procedure

- 1 Click “Water Level”
Go to “Face” tab

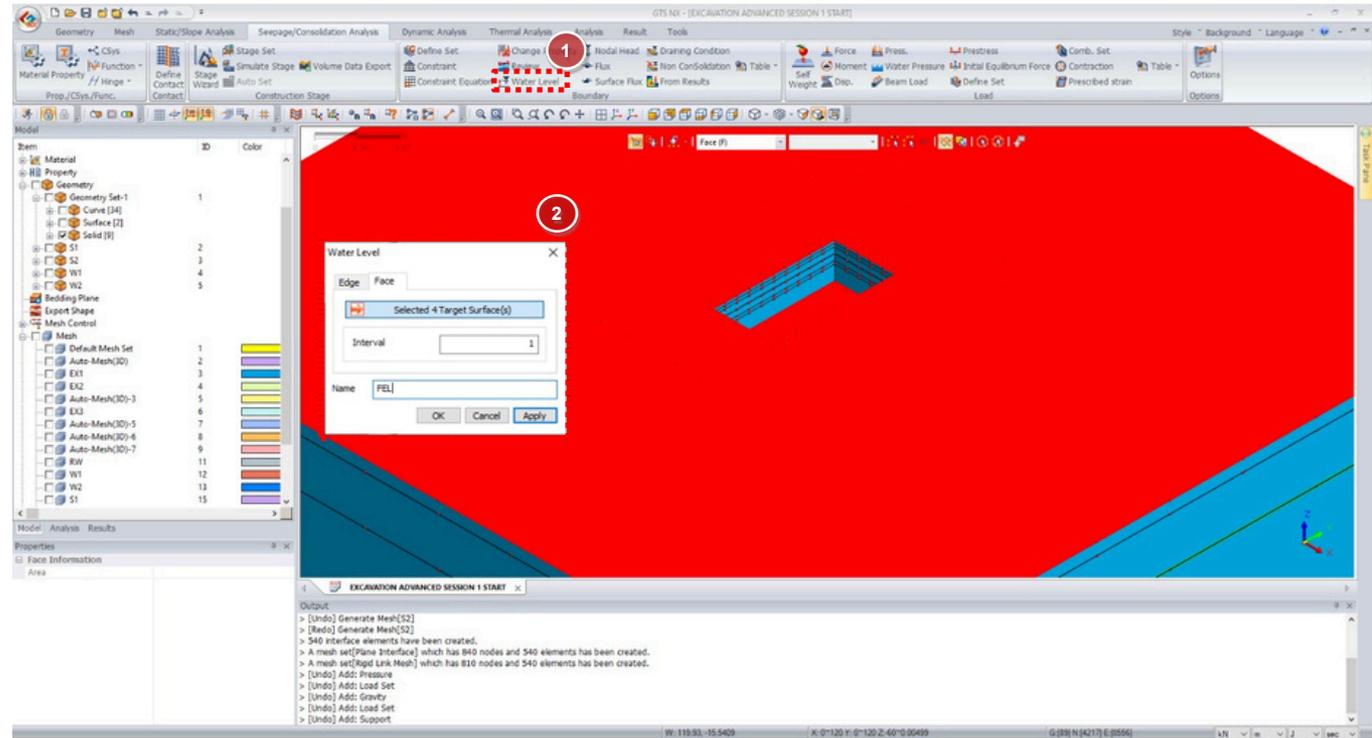
Select ground surface and EX2 floor
Name “EX2”
Click “OK”
[Information]
Excavation floor need to be selected at the end.



Define water levels for each excavation stage

Procedure

- 1 Click “Water Level”
Go to “Face” tab
- Select ground surface and FEL floor
- Name “FEL”
- Click “OK”
- [Information] Excavation floor need to be selected at the end.



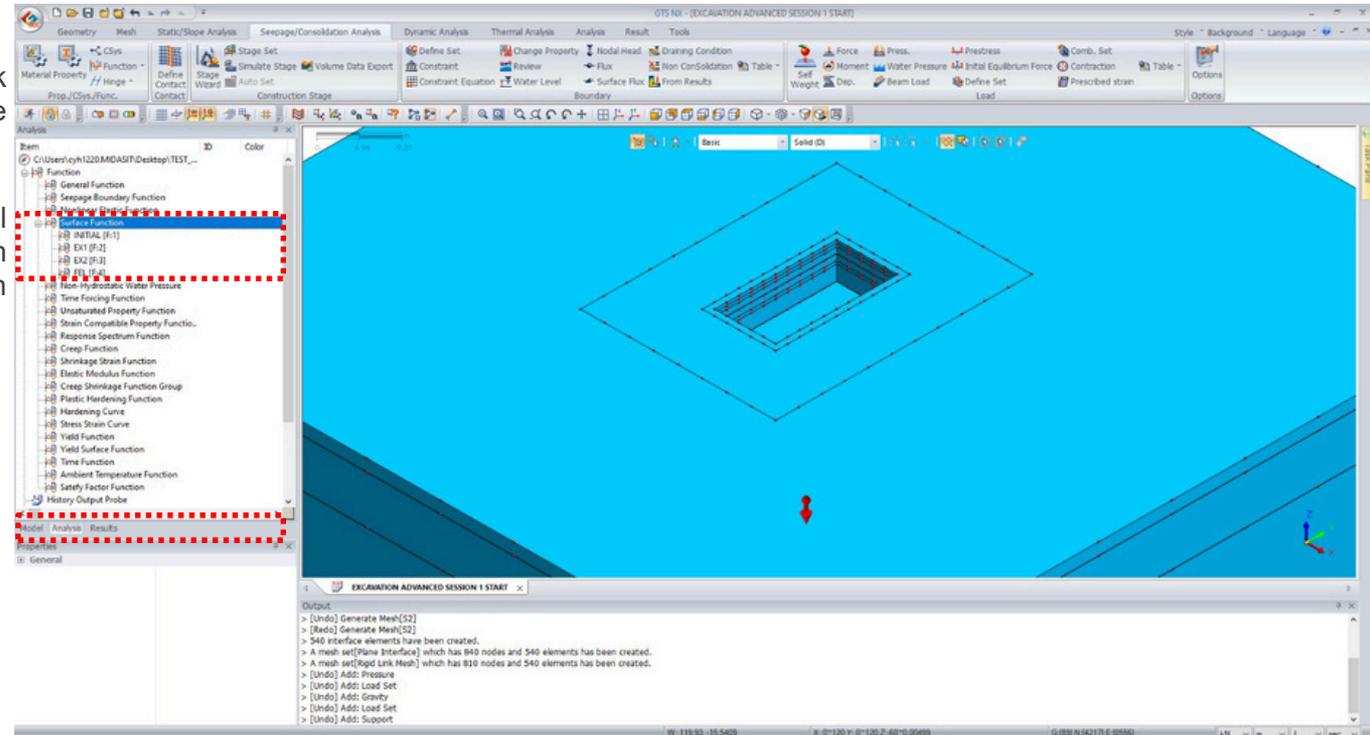
Define water levels for each excavation stage

Procedure

[Information] You can check created water level from “Surface Function”

under “Analysis” tab

And, you can control water level manually with coordinate system with table using right click on created water level



GTS NX

New eXperience of Geo-Technical analysis System

Thank you!

MIDAS

The background features a complex network of interconnected nodes and lines, resembling a mesh or a molecular structure. The nodes are small circles, and the lines are thin, connecting the nodes in a non-uniform pattern. The overall color scheme is a gradient from green on the left to orange and red on the right.

GTS NX

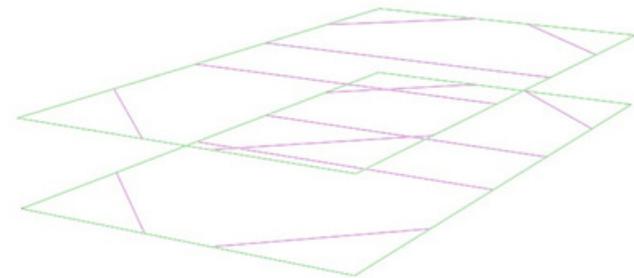
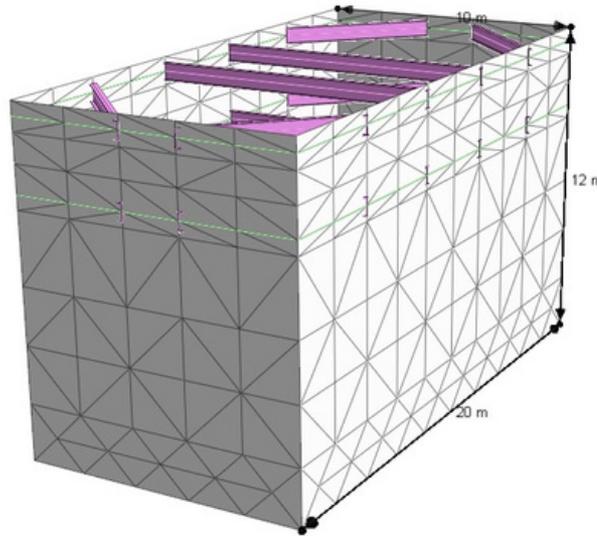
New eXperience of Geo-Technical analysis System

Excavation

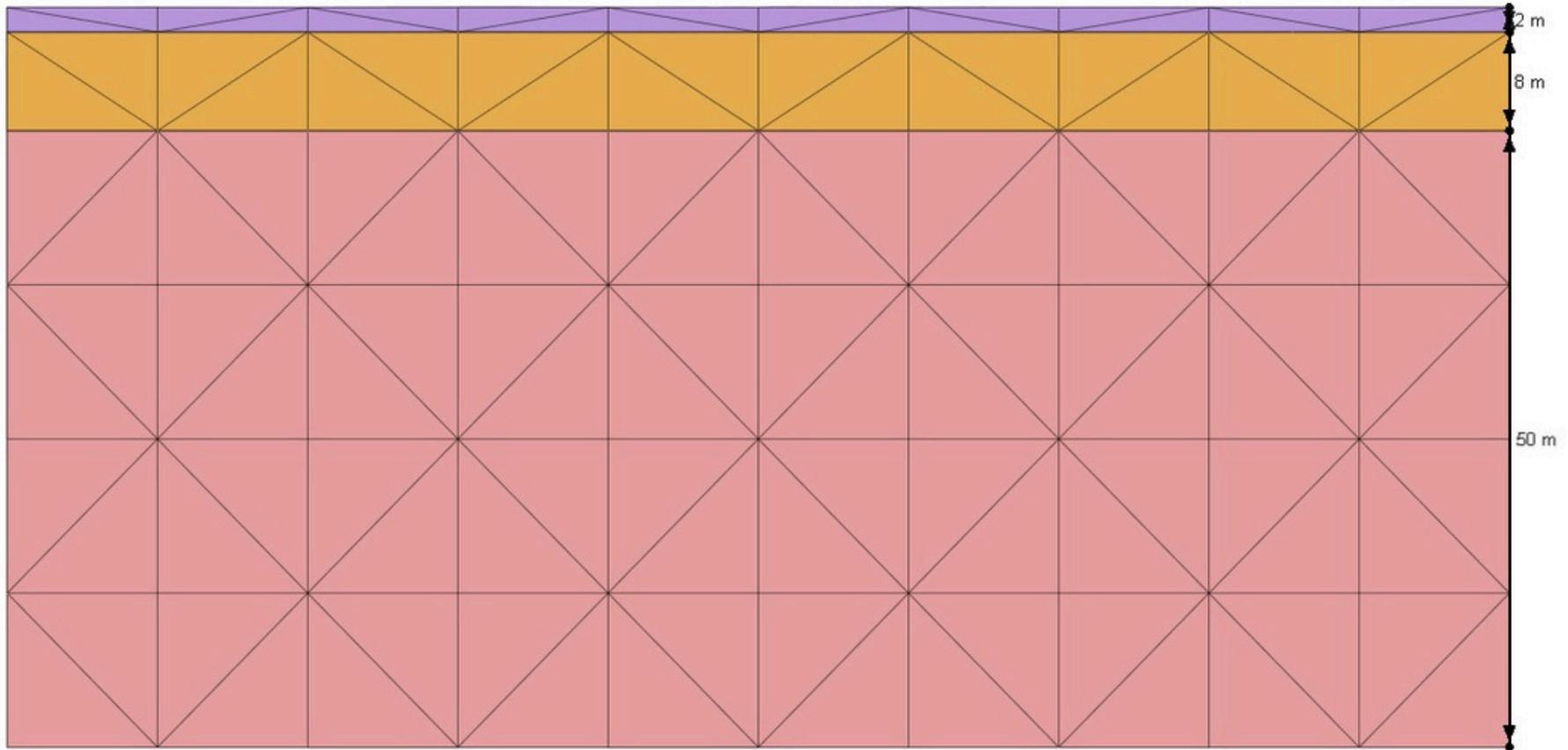
Advanced Course [Session 2]

Modeling and Analysis Summary

The model represents 10mX20m excavation area with temporary facilities which are diaphragm wall, water and strut. The excavation process consists of 3 stages(1.0, 3.5, 6.0 b.g.l.(m)). And, total supports(waler and strut) are installed at 2 levels (0.5, 3.0 b.g.l.(m)) with interval distance of 2.5m. The toe level of retaining wall(D-Wall) is 12 b.g.l.(m). The ground is consisting of 3 layers and depths are respectively: Buried layer 3.0 b.g.l.(m), Colluviums 7.0 b.g.l.(m), the rest part is weathered soil. Surcharge and water level are considered as well in this case.



Modeling and Analysis Summary



Ground

[unit: kN, m]

Name	Buried Layer	Colluvium	Weathered Soil
Material	Isotropic	Isotropic	Isotropic
Model Type	Hardening Soil	Hardening Soil	Hardening Soil
General			
Poisson's Ratio (ν)	0.25	0.25	0.25
Unit Weight (γ) [kN/m ³]	16	17	20
Ko	0.5	0.7	0.7
Porous			
DrainageParameters	Drained	Undrained	Drained
Permeability k [m/sec]	1e-007	1e-007	1e-007
Non-Linear			
Secant Stiffness in Tri-axial Test (E50ref) [kN/m ²]	10,000	60,000	100,000
Tangential Stiffness Primary Oedometer Test (Eoedref) [kN/m ²]	10,000	60,000	100,000
Unloading/Reloading Stiffness (Eurref) [kN/m ²]	30,000	180,000	300,000
Cohesion (c) [kN/m ²]	1	1	4
Frictional Angle (Φ) [deg]	30	34	38

Structure

[unit: kN, m]

Name	Conc'	Steel
Material	Isotropic	Isotropic
Model Type	Elastic	Elastic
Elastic Modulus (E) [kN/m ²]	28,000,000	205,000,000
Poisson's Ratio (ν)	0.15	0.15
Unit Weight (γ) [kN/m ³]	24	74

Property (Structure)

[unit: kN, m]

Name	Material	Type
Retaining Wall (1.0m thk.D-Wall)	Conc'	Shell
Waler(UB 610x229x101 kg/m)	Steel	Beam
Strut (UB 610x229x101 kg/m)	Steel	Truss

GTS NX

New eXperience of Geo-Technical analysis System

Define Analysis Stage

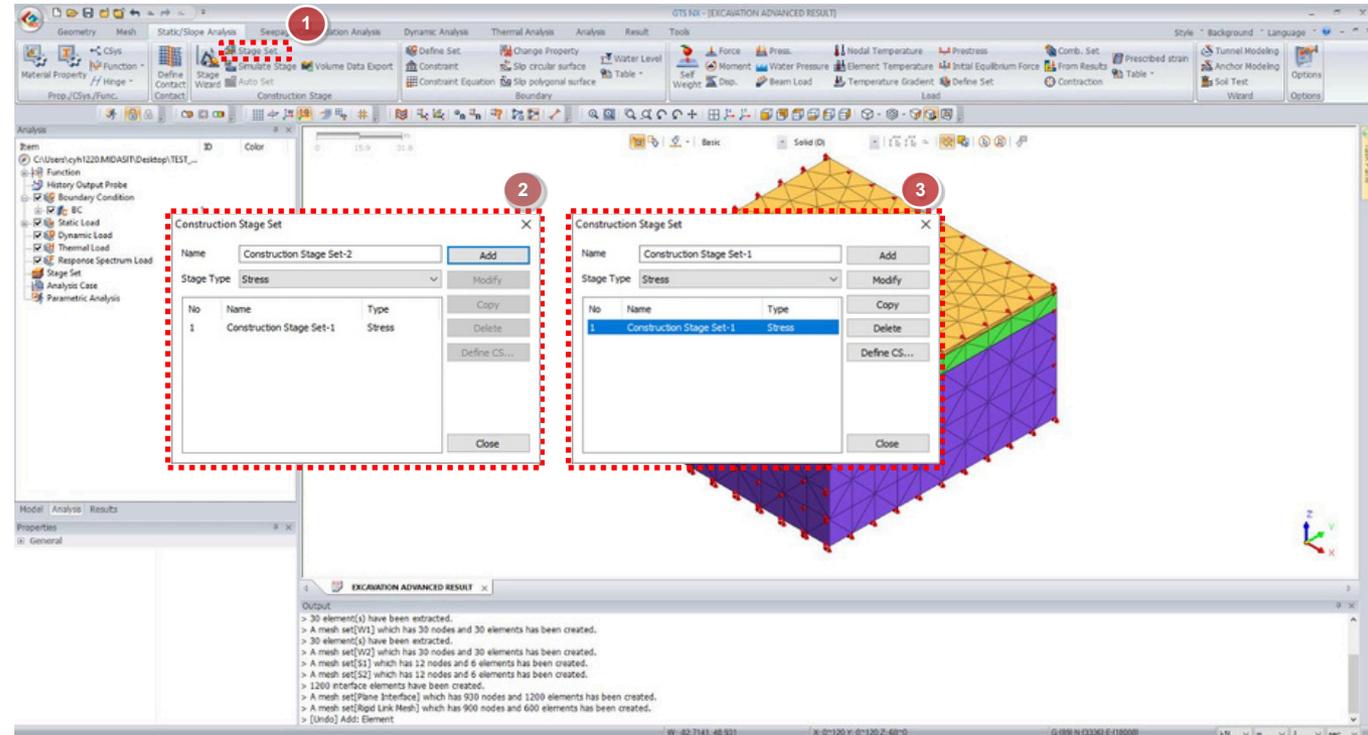
MIDAS

07 Define Analysis Stage

Definestage type

Procedure

- 1 Click "Stage Set" Stage Type "Stress"
- 2 Click "Add"
- 3 Select created stage set Click "Define CS..."

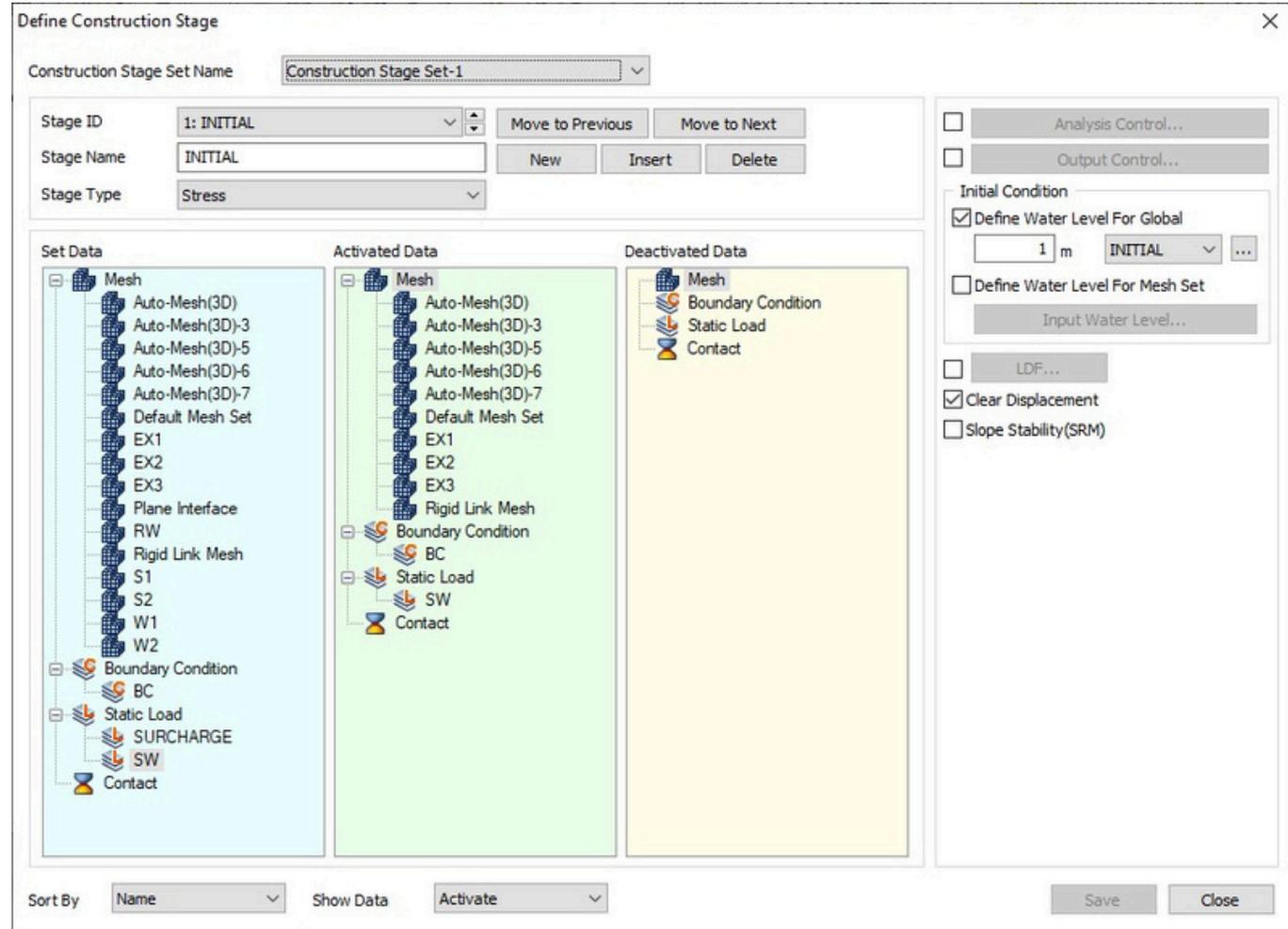


07 Define Analysis Stage

InitialStage

Procedure

- Stage Name "INITIAL"
- <Activate>
- Every mesh set of ground
- Rigid Link Mesh
- BC
- SW
- <Deactivate>
-
- <Water level>
- Check on "Define Water Level"
- "1" m & Select "Initial"
- Check on "Clear Displacement"
- Click "Save"
- Click "New"



Define Construction Stage

Construction Stage Set Name: Construction Stage Set-1

Stage ID: 1: INITIAL

Stage Name: INITIAL

Stage Type: Stress

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

Set Data (Deactivated): Mesh, Auto-Mesh(3D), Auto-Mesh(3D)-3, Auto-Mesh(3D)-5, Auto-Mesh(3D)-6, Auto-Mesh(3D)-7, Default Mesh Set, EX1, EX2, EX3, Plane Interface, RW, Rigid Link Mesh, S1, S2, W1, W2, Boundary Condition, BC, Static Load, SURCHARGE, SW, Contact

Activated Data (Activated): Mesh, Auto-Mesh(3D), Auto-Mesh(3D)-3, Auto-Mesh(3D)-5, Auto-Mesh(3D)-6, Auto-Mesh(3D)-7, Default Mesh Set, EX1, EX2, EX3, Rigid Link Mesh, Boundary Condition, BC, Static Load, SW, Contact

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

Initial Condition:

- Define Water Level For Global: 1 m, INITIAL
- Define Water Level For Mesh Set
- LDF...
- Clear Displacement
- Slope Stability(SRM)

Buttons: Save, Close

07 Define Analysis Stage

Surcharge

Procedure

Stage Name "SURCHARGE"

<Activate>

SURCHARGE

<Deactivate>

-

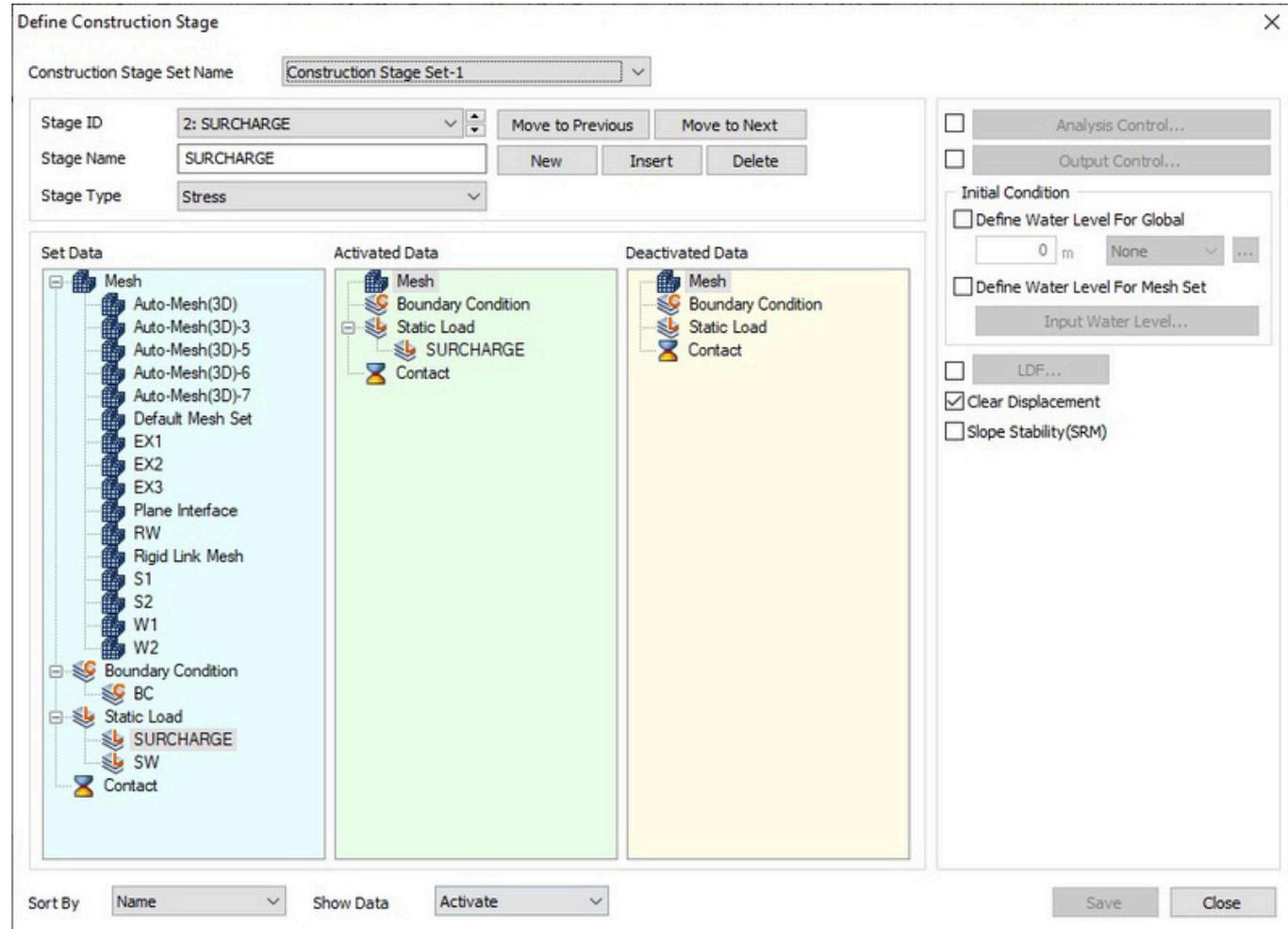
<Water level>

-

Check on "Clear Displacement"

Click "Save"

Click "New"



Define Construction Stage

Construction Stage Set Name: Construction Stage Set-1

Stage ID: 2: SURCHARGE

Stage Name: SURCHARGE

Stage Type: Stress

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

Set Data	Activated Data	Deactivated Data
Mesh	Mesh	Mesh
Auto-Mesh(3D)	Boundary Condition	Boundary Condition
Auto-Mesh(3D)-3	Static Load	Static Load
Auto-Mesh(3D)-5	SURCHARGE	Static Load
Auto-Mesh(3D)-6	Contact	Contact
Auto-Mesh(3D)-7		
Default Mesh Set		
EX1		
EX2		
EX3		
Plane Interface		
RW		
Rigid Link Mesh		
S1		
S2		
W1		
W2		
Boundary Condition		
BC		
Static Load		
SURCHARGE		
SW		
Contact		

Initial Condition

Define Water Level For Global

0 m None ...

Define Water Level For Mesh Set

Input Water Level...

LDF...

Clear Displacement

Slope Stability(SRM)

Sort By: Name Show Data: Activate

Buttons: Save, Close

07 Define Analysis Stage

Retainingwall

Procedure

Stage Name "RW"

<Activate>

RW

Plane Interface

<Deactivate>

Rigid Link Mesh

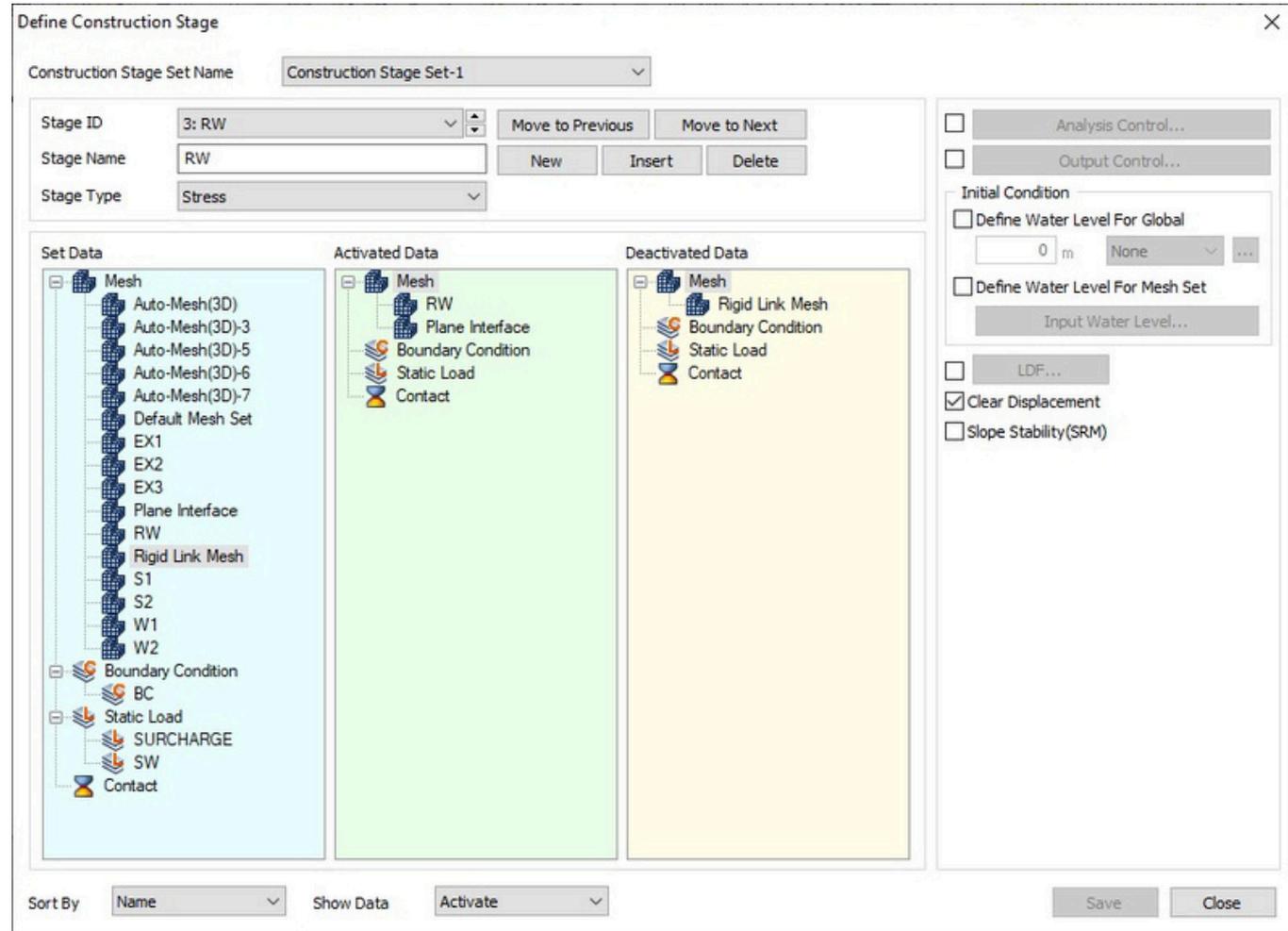
<Water level>

-

Check on "Clear Displacement"

Click "Save"

Click "New"



Define Construction Stage

Construction Stage Set Name: Construction Stage Set-1

Stage ID: 3: RW (Move to Previous, Move to Next)

Stage Name: RW (New, Insert, Delete)

Stage Type: Stress

Set Data	Activated Data	Deactivated Data
Mesh	Mesh	Mesh
Auto-Mesh(3D)	RW	Rigid Link Mesh
Auto-Mesh(3D)-3	Plane Interface	Boundary Condition
Auto-Mesh(3D)-5	Boundary Condition	Static Load
Auto-Mesh(3D)-6	Static Load	Contact
Auto-Mesh(3D)-7	Contact	
Default Mesh Set		
EX1		
EX2		
EX3		
Plane Interface		
RW		
Rigid Link Mesh		
S1		
S2		
W1		
W2		
Boundary Condition		
BC		
Static Load		
SURCHARGE		
SW		
Contact		

Initial Condition

Define Water Level For Global (0 m, None, ...)

Define Water Level For Mesh Set (Input Water Level...)

LDF...

Clear Displacement

Slope Stability(SRM)

Sort By: Name (Show Data, Activate)

Save Close

07 Define Analysis Stage

1stExcavation

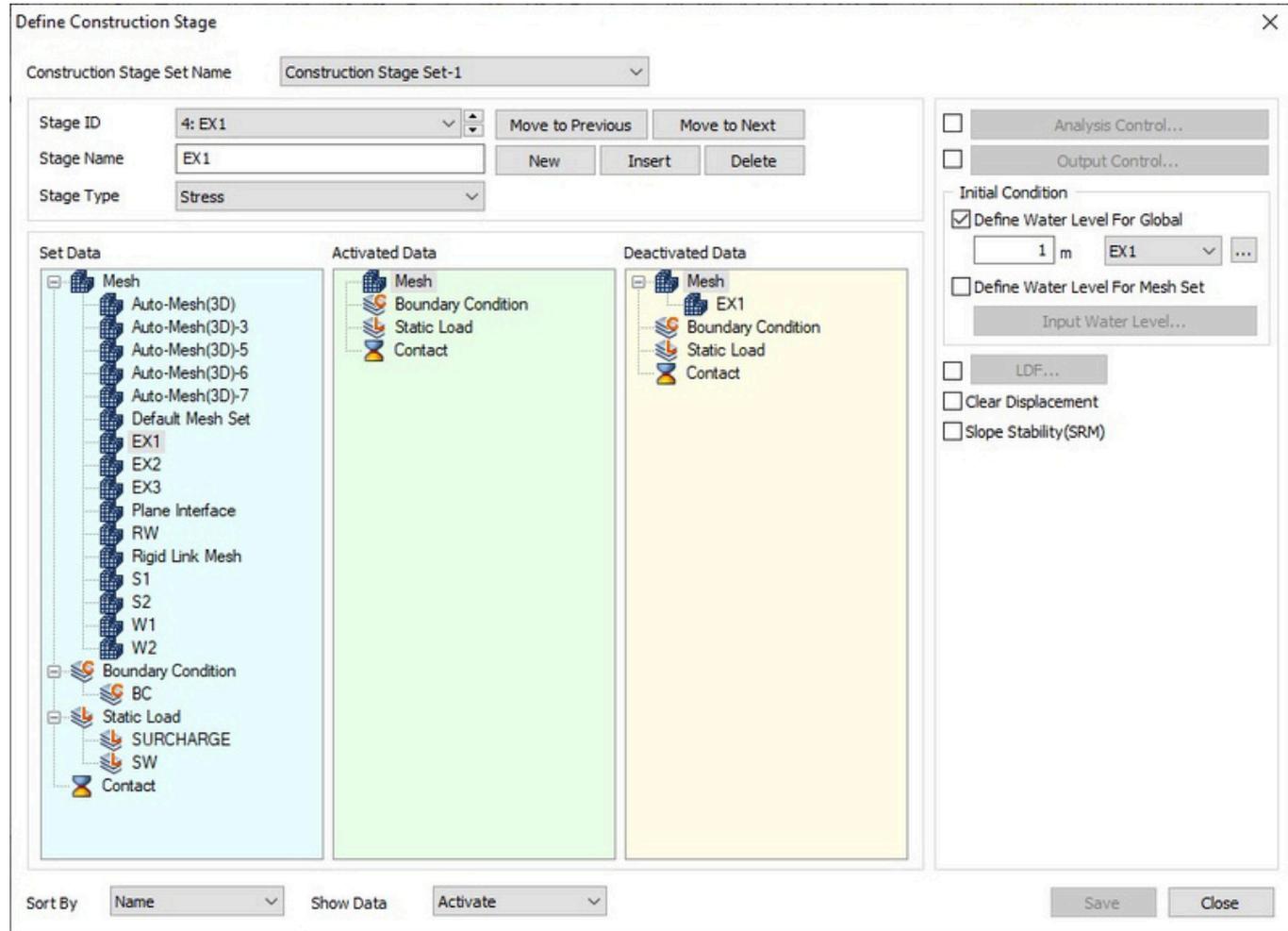
Procedure

Stage Name "EX1"
<Activate>
-
<Deactivate>
EX1
<Water level>
Check on "Define Water Level"

"1" m & Select "EX1"

Click "Save"

Click "New"



Define Construction Stage

Construction Stage Set Name: Construction Stage Set-1

Stage ID: 4: EX1 (Move to Previous, Move to Next)

Stage Name: EX1 (New, Insert, Delete)

Stage Type: Stress

Set Data:

- Mesh
 - Auto-Mesh(3D)
 - Auto-Mesh(3D)-3
 - Auto-Mesh(3D)-5
 - Auto-Mesh(3D)-6
 - Auto-Mesh(3D)-7
 - Default Mesh Set
 - EX1
 - EX2
 - EX3
 - Plane Interface
 - RW
 - Rigid Link Mesh
 - S1
 - S2
 - W1
 - W2
- Boundary Condition
 - BC
- Static Load
 - SURCHARGE
 - SW
- Contact

Activated Data:

- Mesh
- Boundary Condition
- Static Load
- Contact

Deactivated Data:

- Mesh
 - EX1
- Boundary Condition
- Static Load
- Contact

Initial Condition:

- Define Water Level For Global (1 m, EX1)
- Define Water Level For Mesh Set (Input Water Level...)

LDF...

Clear Displacement

Slope Stability(SRM)

Sort By: Name (Show Data, Activate)

Save Close

07 Define Analysis Stage

1st Reinforcement

Procedure

Stage Name "S1"

<Activate>

S1

W1

<Deactivate>

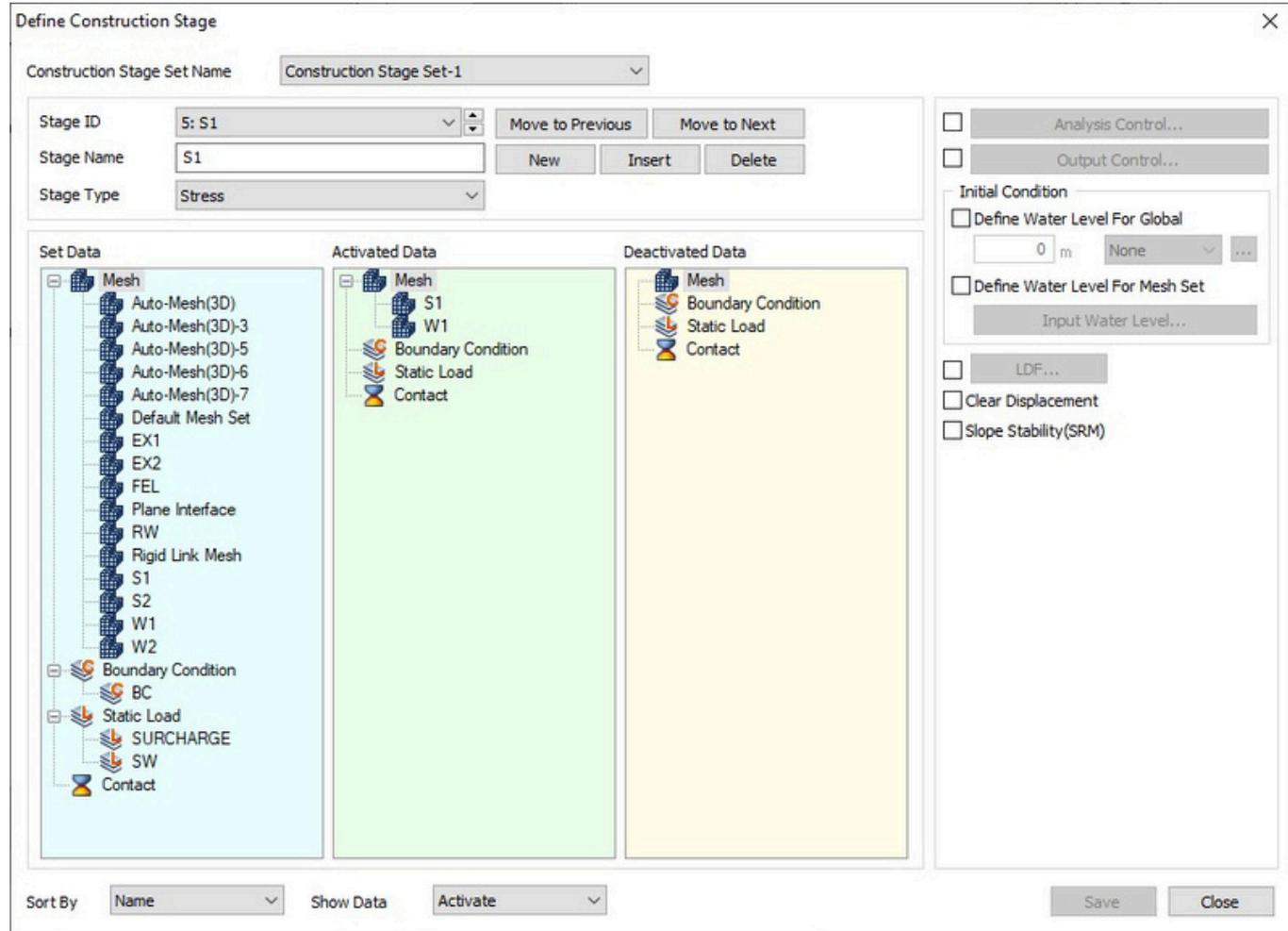
-

<Water level>

-

Click "Save"

Click "New"



Define Construction Stage

Construction Stage Set Name: Construction Stage Set-1

Stage ID: 5: S1 | Move to Previous | Move to Next

Stage Name: S1 | New | Insert | Delete

Stage Type: Stress

Set Data

- Mesh
 - Auto-Mesh(3D)
 - Auto-Mesh(3D)-3
 - Auto-Mesh(3D)-5
 - Auto-Mesh(3D)-6
 - Auto-Mesh(3D)-7
 - Default Mesh Set
 - EX1
 - EX2
 - FEL
 - Plane Interface
 - RW
 - Rigid Link Mesh
 - S1
 - S2
 - W1
 - W2
- Boundary Condition
 - BC
- Static Load
 - SURCHARGE
 - SW
- Contact

Activated Data

- Mesh
 - S1
 - W1
- Boundary Condition
- Static Load
- Contact

Deactivated Data

- Mesh
- Boundary Condition
- Static Load
- Contact

Initial Condition

- Define Water Level For Global
0 m | None | ...
- Define Water Level For Mesh Set
Input Water Level...

LDF... | Clear Displacement | Slope Stability(SRM)

Sort By: Name | Show Data | Activate

Save | Close

07 Define Analysis Stage

2ndExcavation

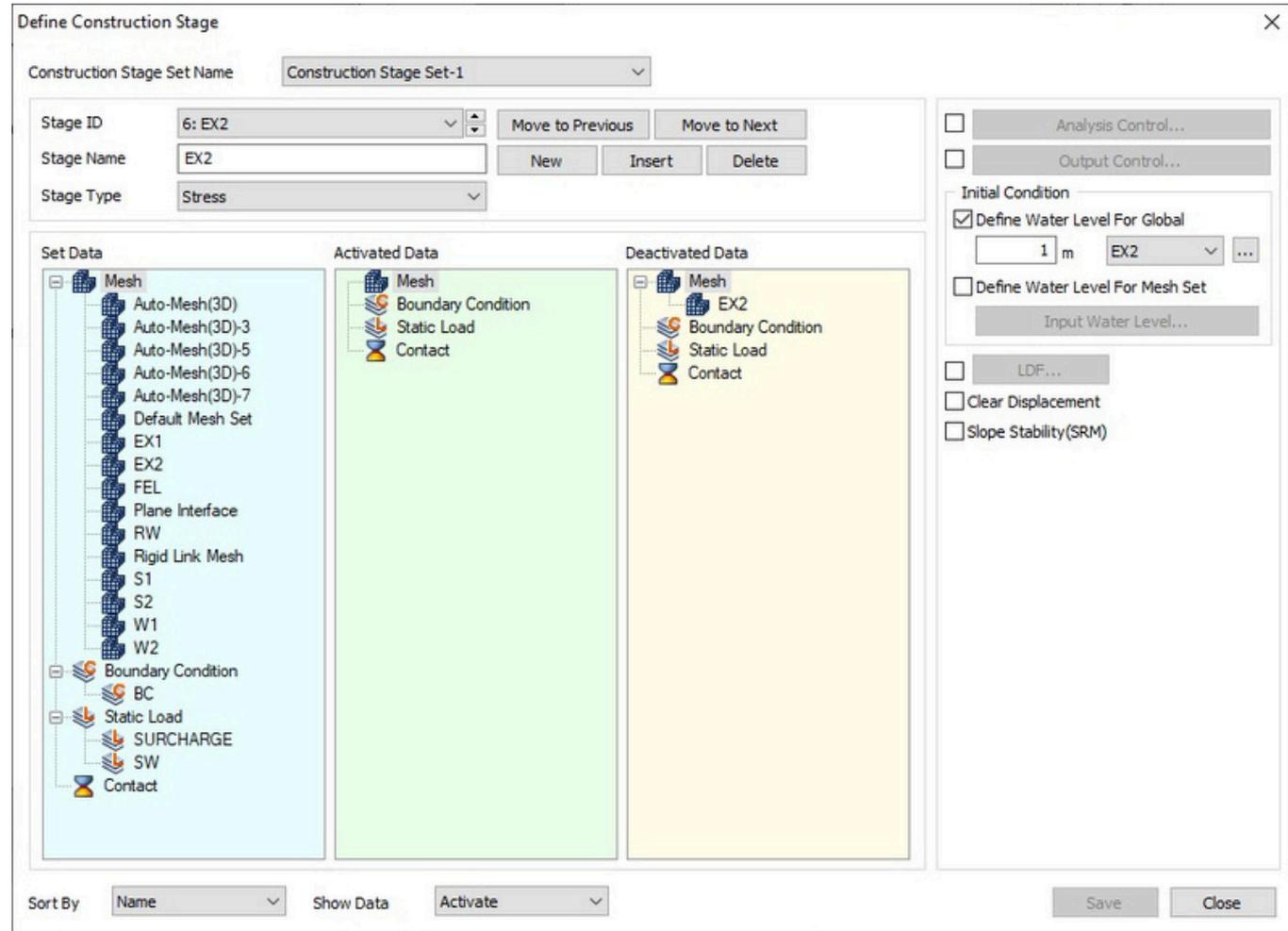
Procedure

Stage Name "EX2"
<Activate>
-
<Deactivate>
EX2
<Water level>
Check on "Define Water Level"

"1" m & Select "EX2"

Click "Save"

Click "New"



Define Construction Stage

Construction Stage Set Name: Construction Stage Set-1

Stage ID: 6: EX2 Move to Previous Move to Next

Stage Name: EX2 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">Auto-Mesh(3D)Auto-Mesh(3D)-3Auto-Mesh(3D)-5Auto-Mesh(3D)-6Auto-Mesh(3D)-7Default Mesh SetEX1EX2FELPlane InterfaceRWRigid Link MeshS1S2W1W2Boundary Condition<ul style="list-style-type: none">BCStatic Load<ul style="list-style-type: none">SURCHARGESWContact	<ul style="list-style-type: none">MeshBoundary ConditionStatic LoadContact	<ul style="list-style-type: none">Mesh<ul style="list-style-type: none">EX2Boundary ConditionStatic LoadContact

Initial Condition

Define Water Level For Global
1 m EX2 ...

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

LDF...

Clear Displacement

Slope Stability(SRM)

Sort By: Name Show Data Activate By: Save Close

07 Define Analysis Stage

2nd Reinforcement

Procedure

Stage Name "S2"

<Activate>

S2

W2

<Deactivate>

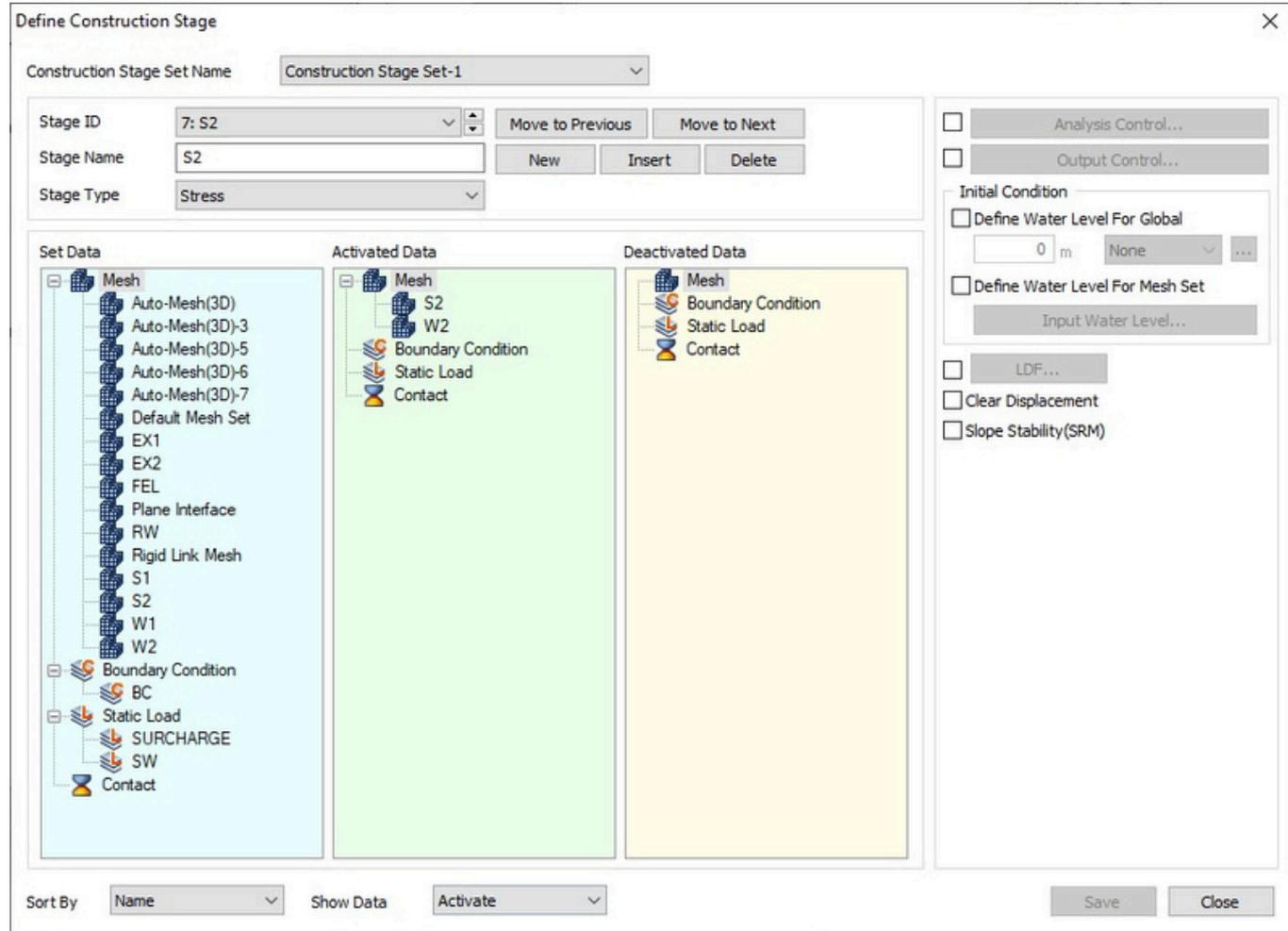
-

<Water level>

-

Click "Save"

Click "New"



Define Construction Stage

Construction Stage Set Name: Construction Stage Set-1

Stage ID: 7: S2

Stage Name: S2

Stage Type: Stress

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

Set Data:

- Mesh
 - Auto-Mesh(3D)
 - Auto-Mesh(3D)-3
 - Auto-Mesh(3D)-5
 - Auto-Mesh(3D)-6
 - Auto-Mesh(3D)-7
 - Default Mesh Set
 - EX1
 - EX2
 - FEL
 - Plane Interface
 - RW
 - Rigid Link Mesh
 - S1
 - S2
 - W1
 - W2
- Boundary Condition
 - BC
- Static Load
 - SURCHARGE
 - SW
- Contact

Activated Data:

- Mesh
 - S2
 - W2
- Boundary Condition
- Static Load
- Contact

Deactivated Data:

- Mesh
- Boundary Condition
- Static Load
- Contact

Initial Condition:

- Define Water Level For Global (0 m)
- Define Water Level For Mesh Set (Input Water Level...)

Other options:

- LDF...
- Clear Displacement
- Slope Stability(SRM)

Sort By: Name | Show Data: Activate

Buttons: Save, Close

07 Define Analysis Stage

Final Excavation

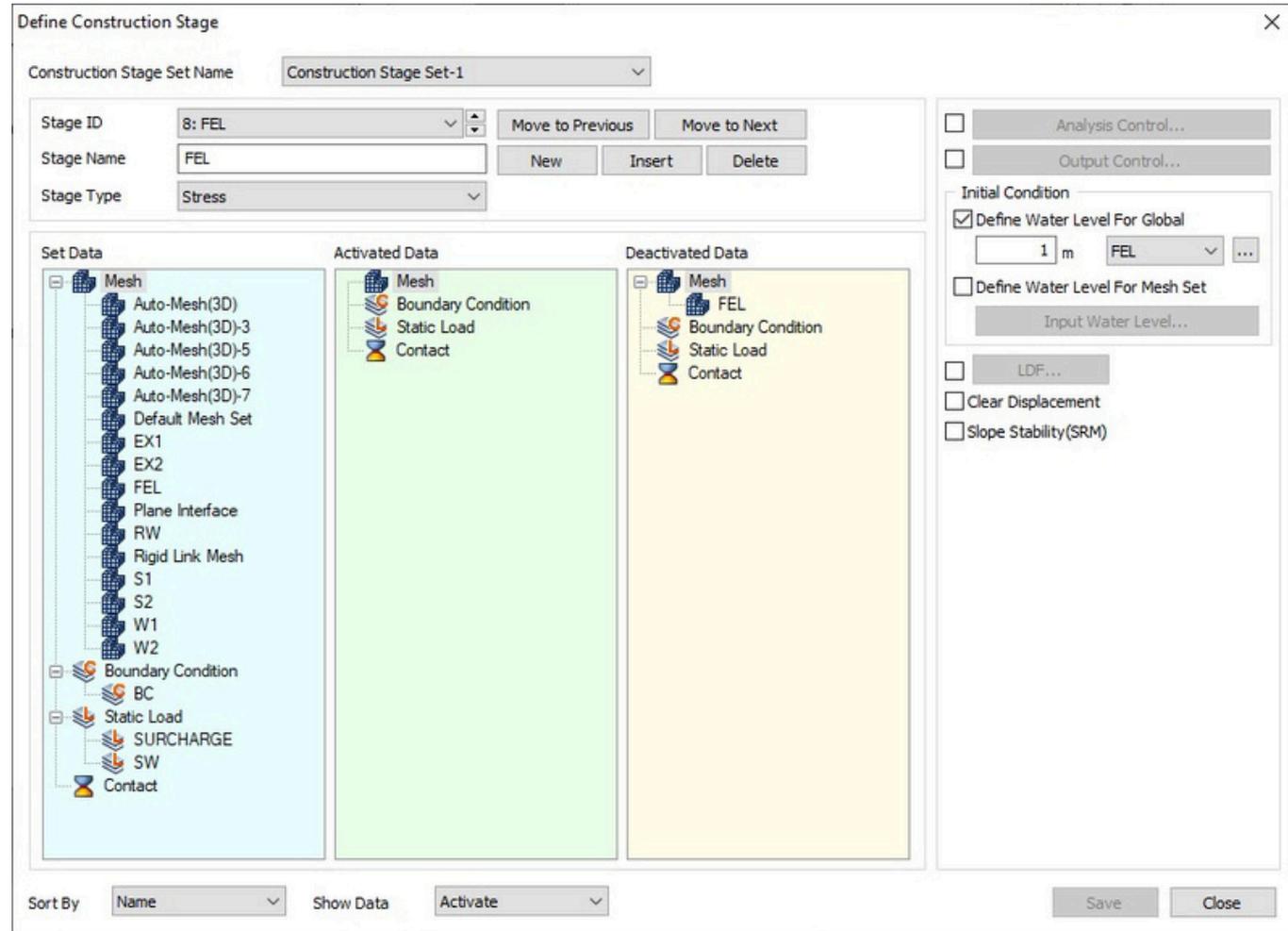
Procedure

Stage Name "FEL"
<Activate>
-
<Deactivate>
FEL
<Water level>
Check on "Define Water Level"

"1" m & Select "FEL"

Click "Save"

Click "New"



Define Construction Stage

Construction Stage Set Name: Construction Stage Set-1

Stage ID: 8: FEL (Move to Previous, Move to Next)

Stage Name: FEL (New, Insert, Delete)

Stage Type: Stress

Set Data	Activated Data	Deactivated Data
Mesh <ul style="list-style-type: none">Auto-Mesh(3D)Auto-Mesh(3D)-3Auto-Mesh(3D)-5Auto-Mesh(3D)-6Auto-Mesh(3D)-7Default Mesh SetEX1EX2FELPlane InterfaceRWRigid Link MeshS1S2W1W2	Mesh <ul style="list-style-type: none">Boundary ConditionStatic LoadContact	Mesh <ul style="list-style-type: none">FELBoundary ConditionStatic LoadContact

Initial Condition

Define Water Level For Global (1 m, FEL)

Define Water Level For Mesh Set (Input Water Level...)

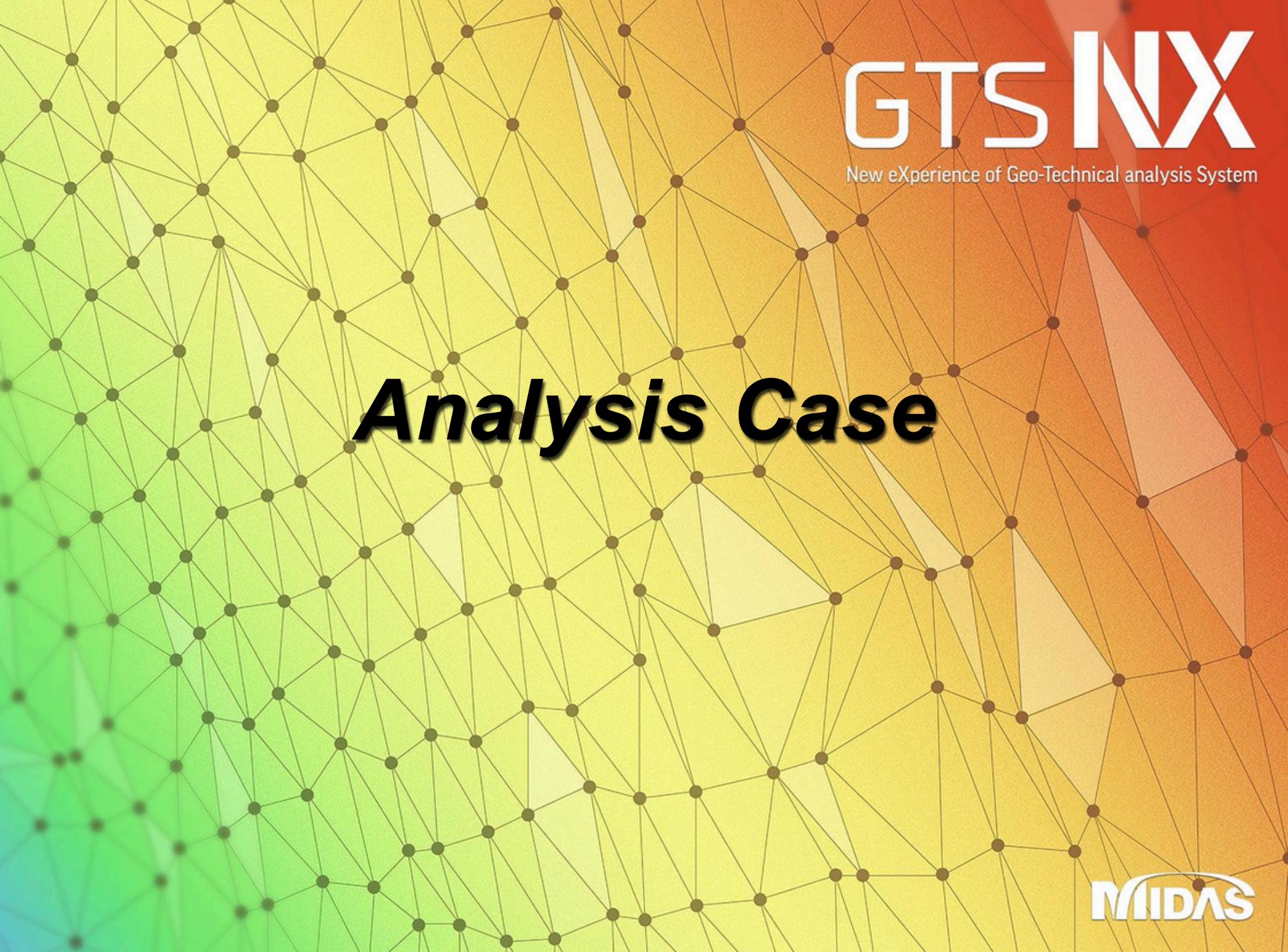
LDF...

Clear Displacement

Slope Stability(SRM)

Sort By: Name (Show Data, Activate)

Save Close

The background features a complex network of interconnected nodes and lines, resembling a mesh or a molecular structure. The nodes are small dark circles, and the lines are thin, light-colored. The overall color scheme transitions from a light green on the left to a bright orange on the right.

GTS NX

New eXperience of Geo-Technical analysis System

Analysis Case

MIDAS

Definestage type

Procedure

- 1 Click "General"
Title "3D EXCAVATION"
- 2 Click "Analysis Control"
- 3 General tab
Check on "Initial Stage for..."
Check on "Apply K0 Condition"
Go to "Nonlinear" tab
Select "Enhanced Init Stress"
- 4 Click "OK"

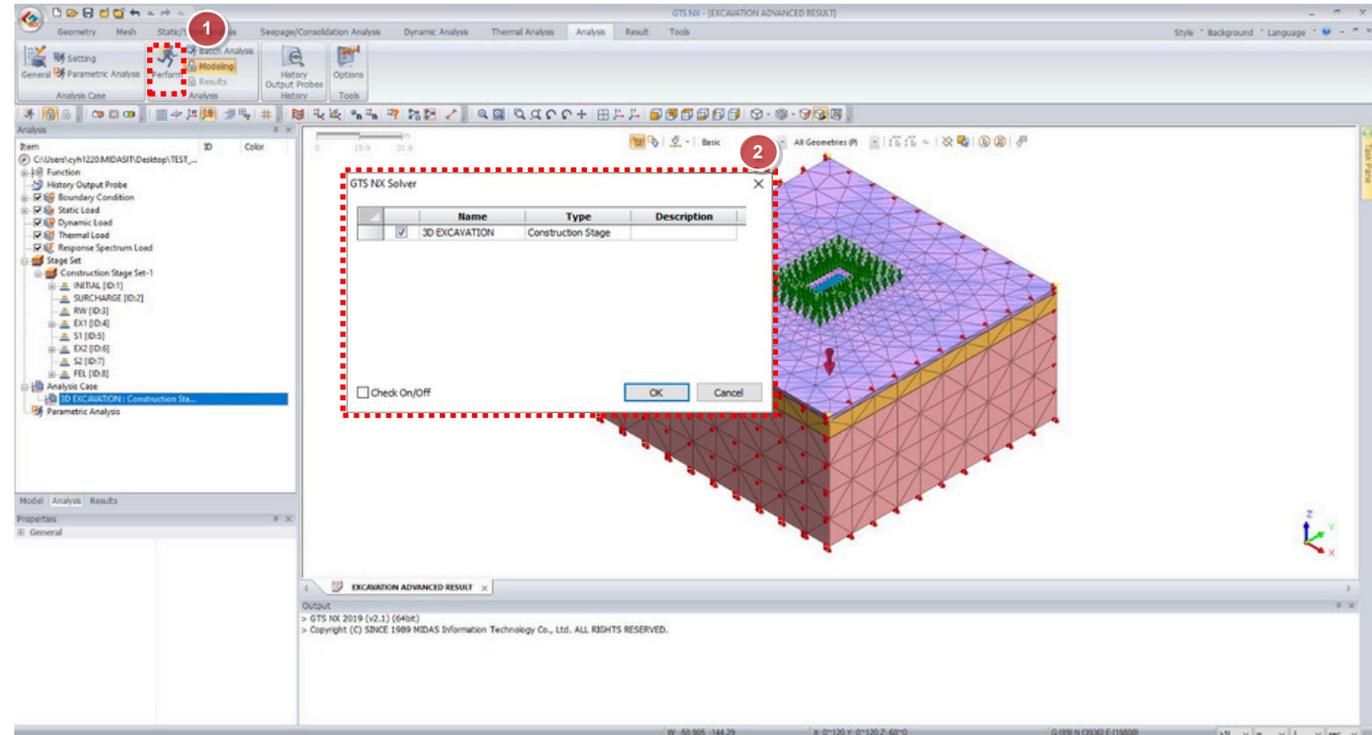
The screenshot displays the GTS NX software interface for defining an analysis case. The 'Analysis Case' dialog box is open, showing the following settings:

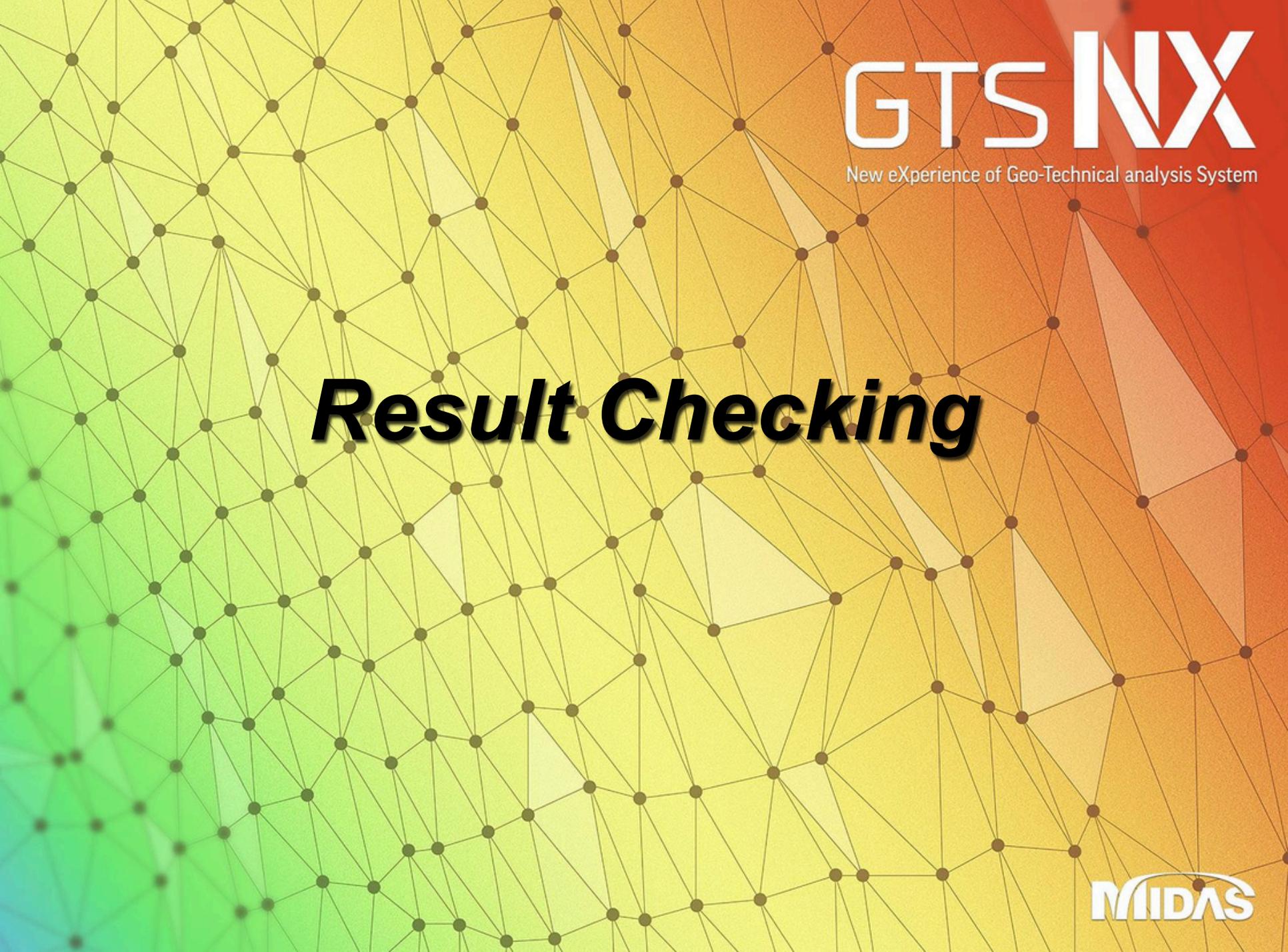
- General Tab (Step 1):** Title is set to "3D EXCAVATION".
- Analysis Control Tab (Step 2):**
 - General:**
 - Water Pressure: Automatically Consider Water Pressure
 - Initial Stage: Initial Stage for Stress Analysis (set to 1:INITIAL)
 - Apply K0 Condition
 - Cut-Off Negative Effective Pressure
 - Initial Stress: Estimate Initial Stress of Activated Elements
 - Final Calculation Stage:** End Stage, Middle Stage (set to 1:INITIAL)
 - Specify Restart Stage
 - Restart Option:** Save only User Specified Stages, Save All Stages
 - Initial Temperature:**
 - Initial Temperature By Value (set to 0 [T])
 - Initial Temperature By Load Set (set to None)
 - Saturation Effects:** Consider Partially Saturated Effects for Stress Analysis
 - Max. Negative Pore Pressure:** Max. Negative Pore Pressure Limit (set to 0 kN/m²)
 - Initial Configuration:** Estimate Initial Configuration of Activated Nodes
- Nonlinear Tab (Step 3):**
 - Geometry Nonlinearity:** Consider Geometric Nonlinear Effects
 - Update Pore Pressure with Deformation
 - Basic Nonlinear Parameters:**
 - Load Steps:
 - Number of Increments: 1
 - Intermediate Output Request: Last Increment
 - Manual with User-Defined Steps
 - Time Steps:
 - Time (Duration): 86400 sec
 - Number of Increments: 1
 - Intermediate Output Request: Last Increment
 - Manual with User-Defined Steps
 - Iterative Scheme:** General, Enhanced Init Stress
 - Convergence Criteria / Error Tolerance:**
 - Displacement(U): 0.03
 - Load(P): 0.03
 - Work(W): 1e-006
- Step 4:** The 'OK' button is highlighted.

Perform the analysis

Procedure

- 1 Click "Perform"
Check on 3D EXCAVATION
- 2 Click "OK"



The background features a complex network of interconnected nodes and lines, resembling a mesh or a molecular structure. The nodes are small dark circles, and the lines are thin, light-colored. The overall color scheme transitions from a light green on the left to a bright orange on the right.

GTS INX

New eXperience of Geo-Technical analysis System

Result Checking

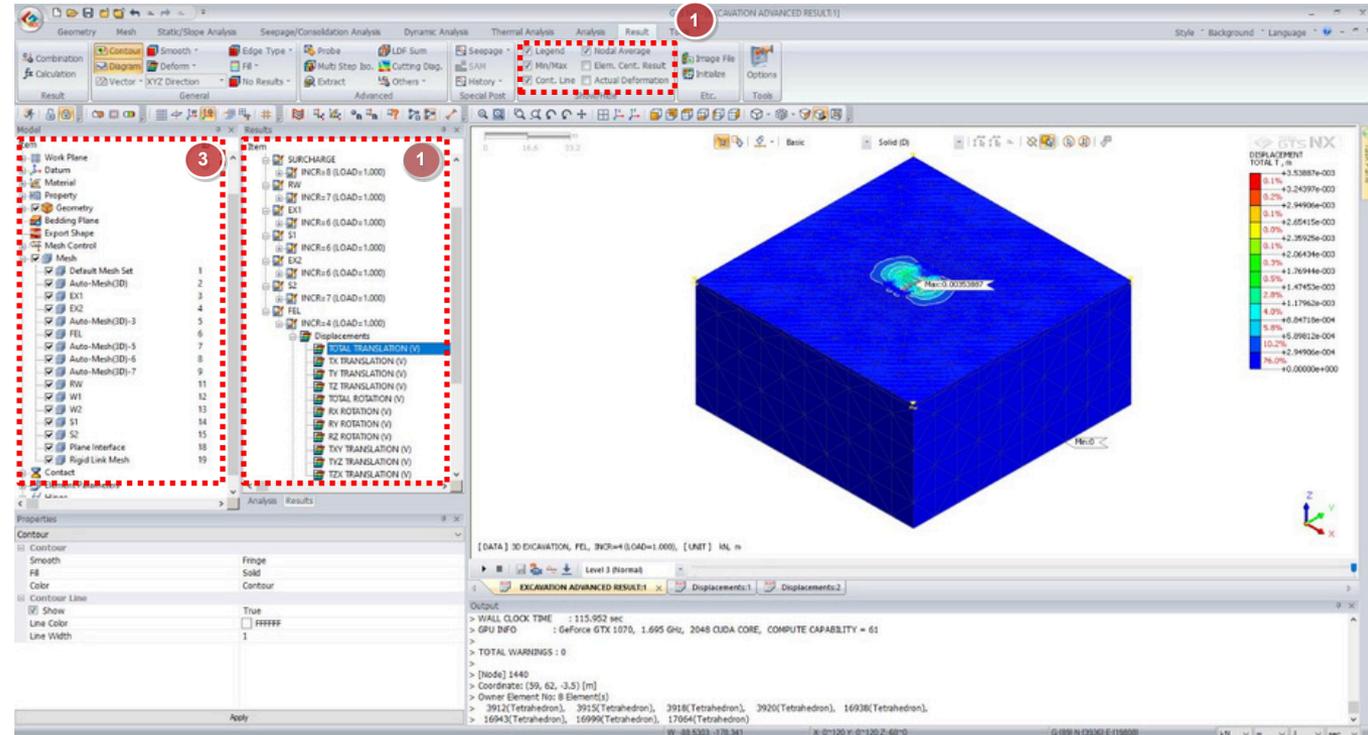
MIDAS

09 Result Checking

Total Displacement

Procedure

- 1 Click "Displacements" > "Total Translation(v)"
Check on Min/Max
- 2
- 3 <Mesh Tree>
Show every mesh set



GroundSettlement

Procedure

- 1 Click "Extract"
- 2 Results "TZ Translation (V)"
- 3 Turn on "Front Selection" to select view side only
Click Table

The screenshot displays the GTS NX software interface for result checking. The 'Extract Results' dialog box is open, showing the following settings:

- Output Data:** 3D EXCAVATION
- Analysis Set:** 3D EXCAVATION
- Result Type:** Displacements
- Results:** TZ TRANSLATION (V)
- Step: Results:** A list of results is shown, with 'FEL:INCR=4 (LOAD=1.000):TZ TRANSLATI' selected.
- Order:** Node/Element
- Object:** Node
- Nodal Results Extraction:** User Defined
- Select Object:** 1 3 6 7 9 12to14 16 17 19 20
- Sort:** X, Y, Z, Ascending
- Extraction Position in Element:** Table

The 3D visualization shows a blue mesh representing the ground settlement results. A red circle highlights the 'Extract Results' dialog box, and a red square highlights the 'Table' button. A red circle with the number '3' is also visible in the top right corner of the interface.

09 Result Checking

GroundSettlement

Procedure

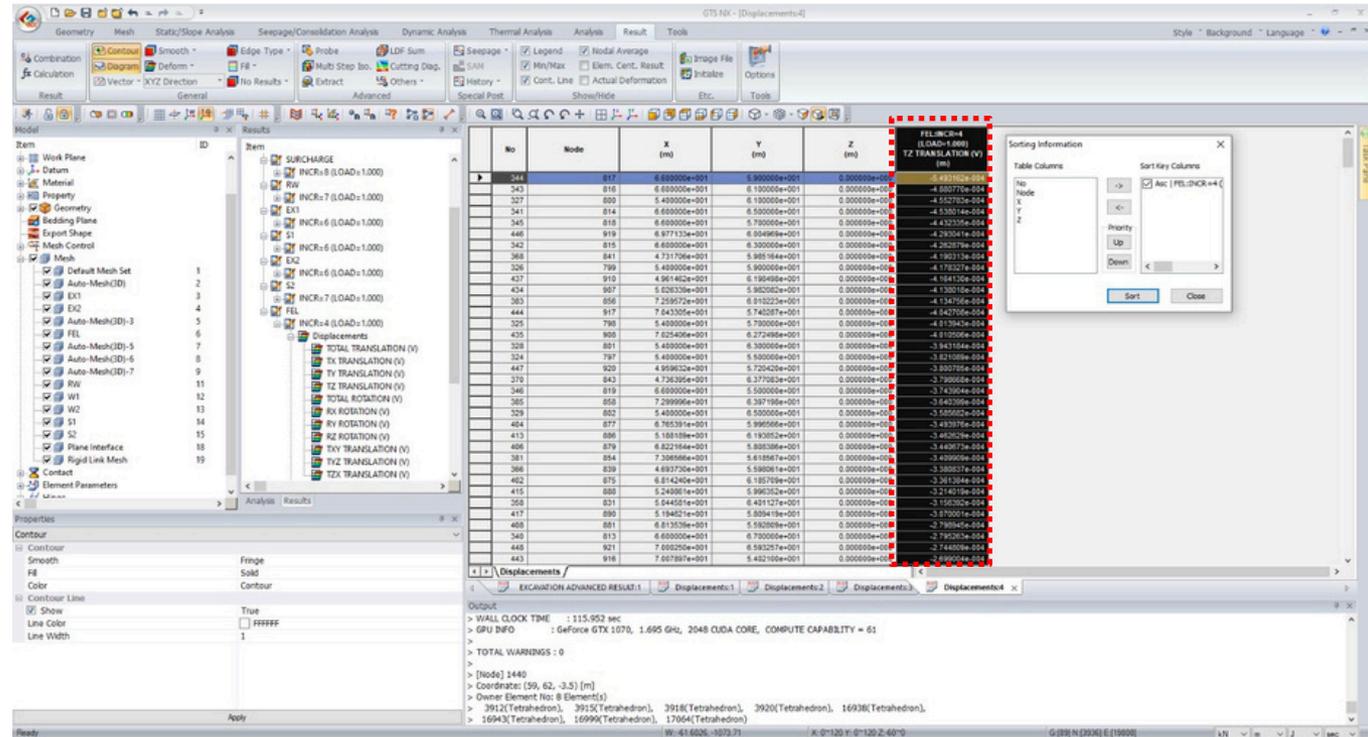
Right Click on the values

Click "Sorting Dialog"

Move "FEL:INCR=4..." to right side

Click "Sort"

Find maximum value

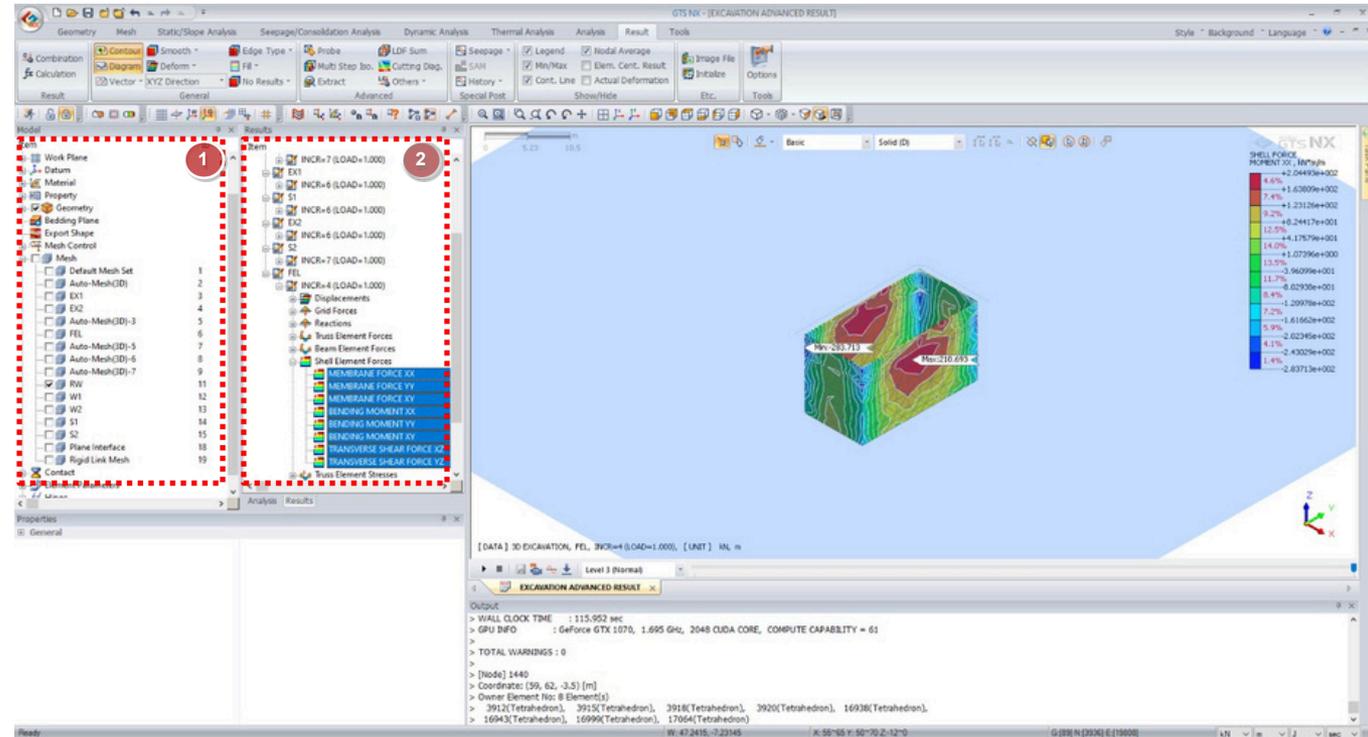


No	Node	X (m)	Y (m)	Z (m)	FEL:INCR=4 (LOAD=1.000) TZ TRANSLATION (V) (m)
344	817	8.690000e-001	8.990000e-001	0.000000e+000	-5.843100e-004
343	816	6.600000e-001	6.100000e-001	0.000000e+000	-4.800770e-004
327	800	5.400000e-001	6.100000e-001	0.000000e+000	-4.027100e-004
341	814	8.690000e-001	6.500000e-001	0.000000e+000	-4.133050e-004
345	818	6.600000e-001	5.700000e-001	0.000000e+000	-4.432050e-004
448	919	8.971330e-001	6.804890e-001	0.000000e+000	-4.293041e-004
342	815	6.600000e-001	6.300000e-001	0.000000e+000	-4.020370e-004
368	841	4.731700e-001	5.988164e-001	0.000000e+000	-4.190313e-004
326	799	5.400000e-001	5.900000e-001	0.000000e+000	-4.178327e-004
437	910	4.811463e-001	6.188490e-001	0.000000e+000	-4.084100e-004
424	907	5.820330e-001	5.982032e-001	0.000000e+000	-4.133050e-004
383	858	7.209702e-001	6.810233e-001	0.000000e+000	-4.134760e-004
444	917	7.643300e-001	6.740287e-001	0.000000e+000	-4.042100e-004
325	798	5.400000e-001	5.700000e-001	0.000000e+000	-4.115030e-004
435	900	7.625400e-001	6.272490e-001	0.000000e+000	-4.010500e-004
328	801	5.400000e-001	6.300000e-001	0.000000e+000	-3.941164e-004
324	797	5.400000e-001	5.600000e-001	0.000000e+000	-3.821000e-004
447	920	4.999630e-001	5.720420e-001	0.000000e+000	-3.800700e-004
370	843	4.736300e-001	6.377024e-001	0.000000e+000	-3.790600e-004
340	819	8.690000e-001	5.500000e-001	0.000000e+000	-3.743000e-004
385	859	7.209990e-001	6.397160e-001	0.000000e+000	-3.840300e-004
329	802	5.400000e-001	6.500000e-001	0.000000e+000	-3.550020e-004
484	877	6.765391e-001	5.999560e-001	0.000000e+000	-3.493970e-004
413	880	5.188180e-001	4.183051e-001	0.000000e+000	-2.400200e-004
406	879	6.822164e-001	5.808380e-001	0.000000e+000	-3.446070e-004
381	854	7.308960e-001	6.818867e-001	0.000000e+000	-3.409000e-004
388	839	4.803700e-001	5.580610e-001	0.000000e+000	-3.300010e-004
402	875	6.814240e-001	6.185700e-001	0.000000e+000	-3.361300e-004
415	888	5.240810e-001	6.996362e-001	0.000000e+000	-3.214010e-004
358	831	5.844000e-001	6.401127e-001	0.000000e+000	-3.100200e-004
417	890	5.194021e-001	5.908190e-001	0.000000e+000	-3.070010e-004
408	881	6.813530e-001	5.920000e-001	0.000000e+000	-2.780040e-004
340	813	6.600000e-001	6.700000e-001	0.000000e+000	-2.795200e-004
448	921	7.602250e-001	6.582251e-001	0.000000e+000	-2.740000e-004
443	918	7.607890e-001	5.402100e-001	0.000000e+000	-2.696000e-004

RetainingWall Checking

Procedure

- 1 <Mesh Tree>
Show only RW mesh set
- 2 Click “Shell Element Forces” to check forces of shell (RW)
[Information]
RW has assumed by Shell element so that RW can be verified from shell element forces
You can check deflection of retaining wall from “Displacement”



The screenshot displays the GTS NX software interface. The top menu bar includes options like Geometry, Mesh, Static/Slope Analysis, Seepage/Consolidation Analysis, Dynamic Analysis, Thermal Analysis, Analysis, Result, and Tools. The left sidebar shows the Mesh Tree with a red dashed box highlighting the 'RW' (Retaining Wall) mesh set. The main window shows a 3D model of a retaining wall with a color-coded stress distribution. The bottom status bar indicates the current analysis results, including the element type (FEL) and the load (LOAD=1.000).

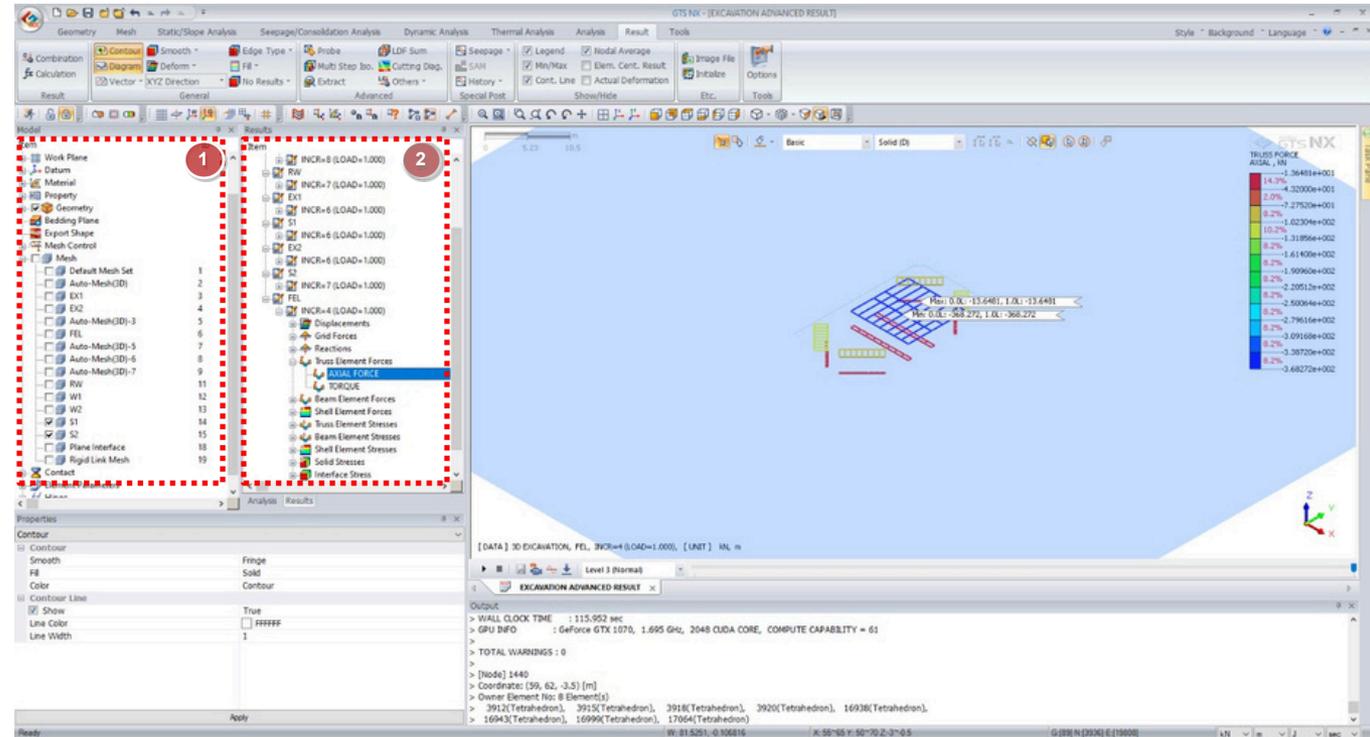
Output:

```
> WALL CLOCK TIME : 115.952 sec
> GPU INFO : GeForce GTX 1070, 1.695 GHz, 2048 CUDA CORE, COMPUTE CAPABILITY = 61
> TOTAL WARNINGS : 0
> [Node] 1440
> Coordinate: (59, 62, -3.5) [m]
> Owner Element No: 8 Element(s)
> 3912(Tetrahedron), 3915(Tetrahedron), 3918(Tetrahedron), 3920(Tetrahedron), 16938(Tetrahedron),
> 16943(Tetrahedron), 16999(Tetrahedron), 17004(Tetrahedron)
```

Axial force of strut

Procedure

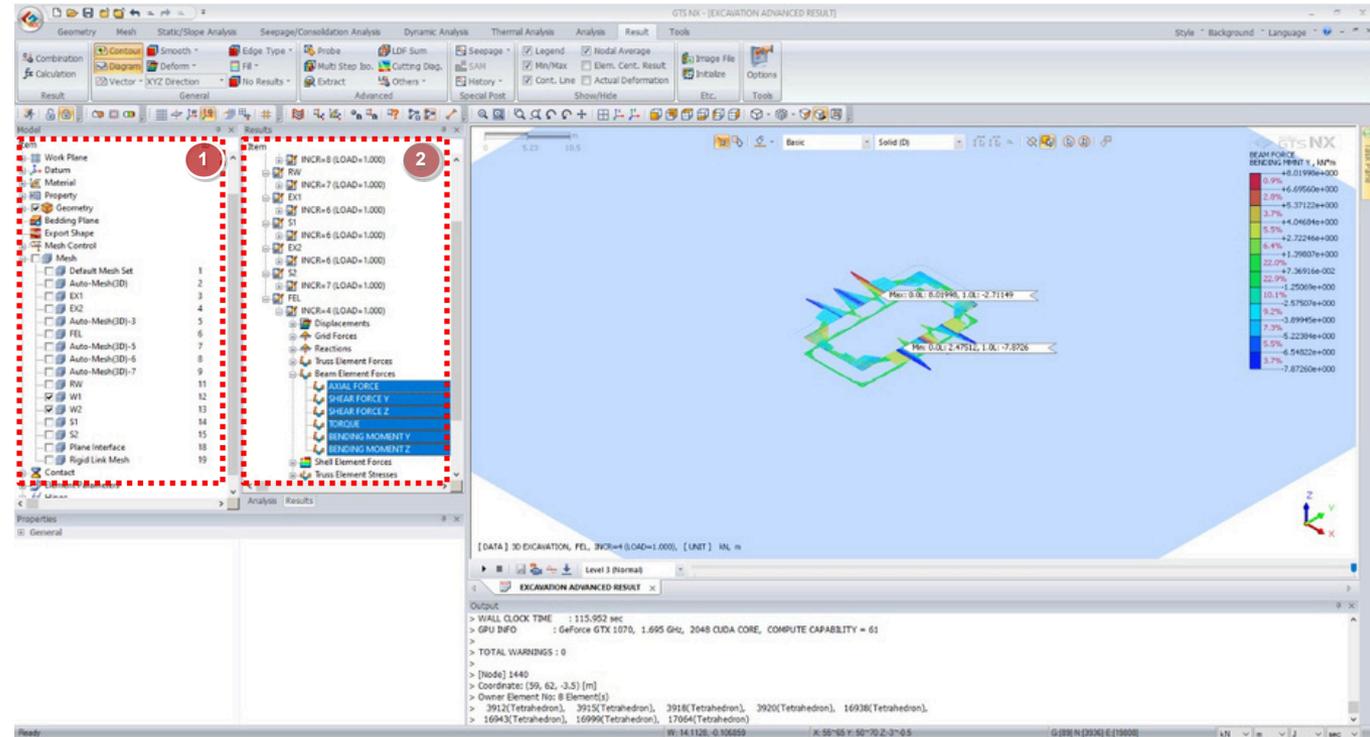
- 1 <Mesh Tree>
Show S1 & S2 mesh set
- 2 Click "TrussElementForces"
to check axial force of struts
[Information]
Strut has assumed by truss
element



GroundSettlement

Procedure

- 1 <Mesh Tree>
Show W1 & W2 mesh set
- 2 Click "BeamElementForces" to check forces of Waler [Information]
Waler has assumed by beam element



GTS NX

New eXperience of Geo-Technical analysis System

Thank you!

MIDAS