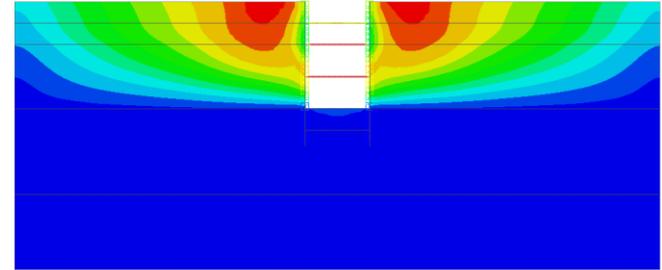


MIDAS *Technical  
Material*

# Tutorial



Deep Excavations Stability Using EUROCODE 7

## Contents

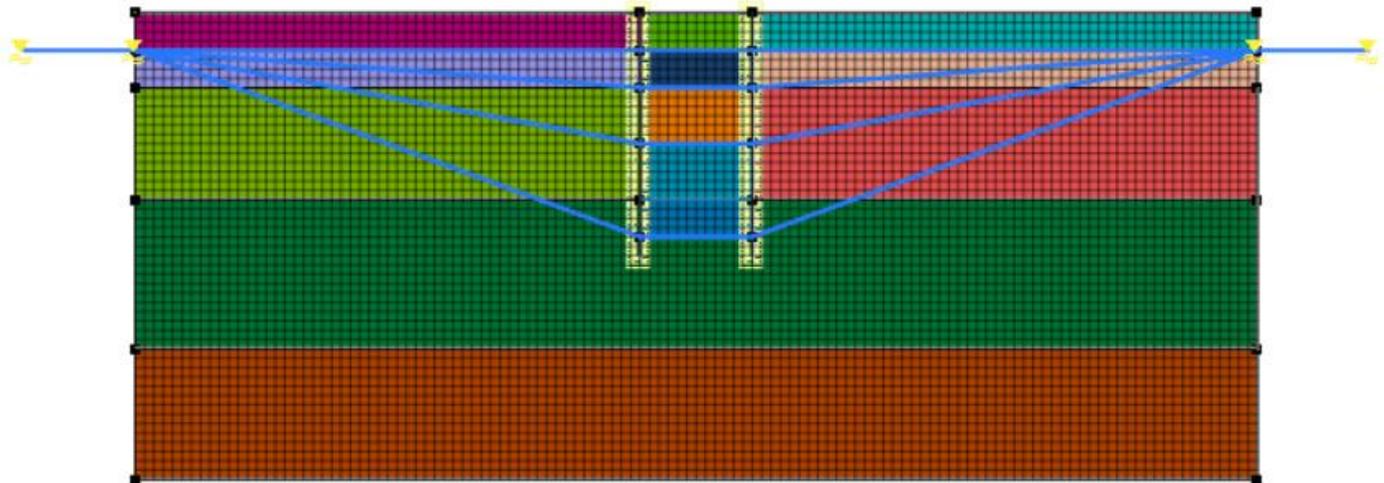
- Step 1: Initial Setting
- Step 2: Defining Soil material
- Step 3: Defining Property
- Step 4: Geometric Modeling
- Step 5: Meshing
- Step 6: Boundary Condition
- Step 7: Loads Definition
- Step 8: Analysis Case
- Step 9: Analysis
- Step 10: Results
- Step 11: Certification Task

# Numerical Model Design

## Overview

This model tutorial is intended to show the user the geogrid design features and its effect on slope reinforcement.

## Modelling



## Contents

- Step 1: Initial Setting
- Step 2: Defining Soil material
- Step 3: Defining Property
- Step 4: Geometric Modeling
- Step 5: Meshing
- Step 6: Boundary Condition
- Step 7: Loads Definition
- Step 8: Analysis Case
- Step 9: Analysis
- Step 10: Results

## 2D Deep Excavation

### Material Properties

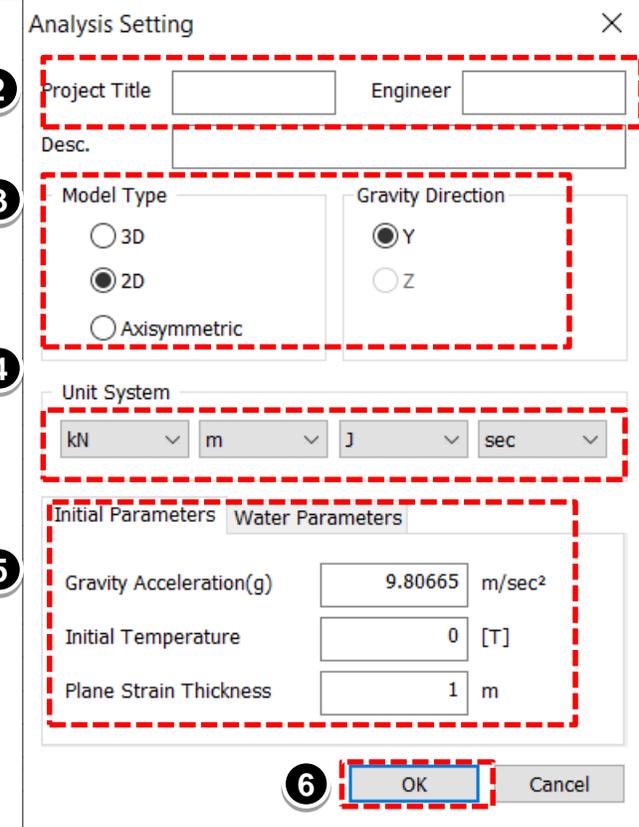
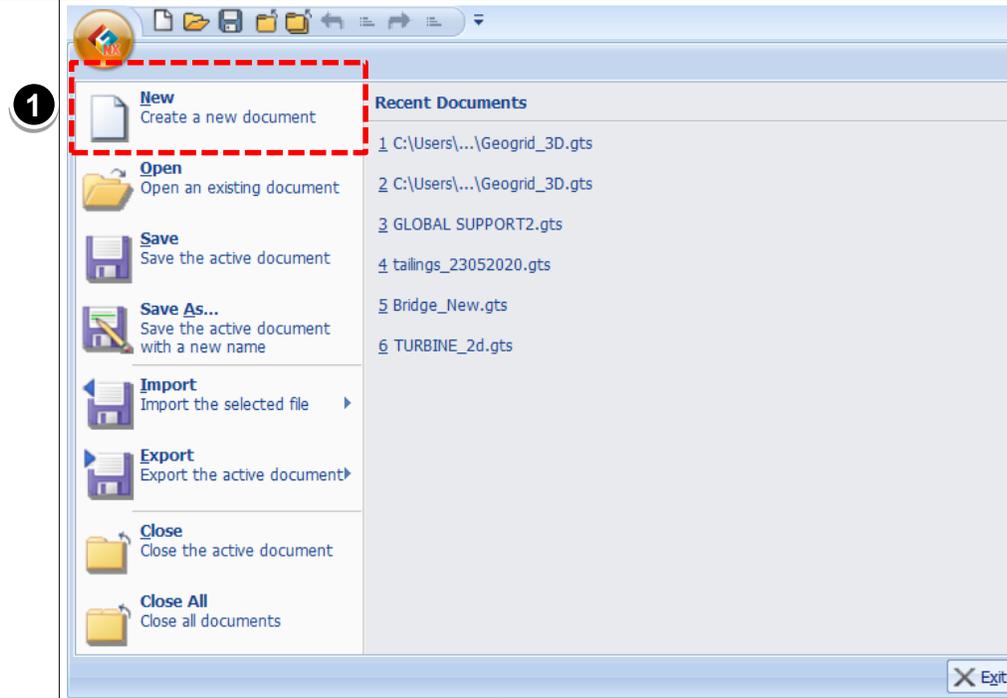
Name	Material Model	Modulus of Elasticity (kN/m <sup>2</sup> )	Poisson's ratio	Unit Weight (kN/m <sup>3</sup> )	Saturated unit Weight (kN/m <sup>3</sup> )	Cohesion (kN/m <sup>2</sup> )	Friction angle
TOP SOIL	Mohr-Coulomb	35000	0.3	18	19	5	32
SAND	Mohr-Coulomb	45000	0.27	18.5	20	3	30
GLACIAL TILL	Mohr-Coulomb	60000	0.32	20	21	7	33
DENSE SAND	Mohr-Coulomb	150000	0.32	21	22.5	25	38
BEDROCK	Mohr-Coulomb	2500000	0.32	26	27	25	39

Name	Material Model	Modulus of Elasticity (kN/m <sup>2</sup> )	Poisson's ratio	Unit Weight (kN/m <sup>3</sup> )
CONCRETE	Elastic	3.50E+08	0.33	24
STEEL	Elastic	2.10E+09	0.33	77

## Procedure

Starting Midas GTS NX

- ① Click on GTS NX icon > **New Project**
- ② Enter the Project name as 3D Geogrid & Engineer XYZ.
- ③ Select Model Type as **2D**.
- ④ Select **kN ,m and sec** in The Unit System.
- ⑤ Use the Default values for Initial parameters.
- ⑥ Click **OK**.



## Procedure

### Defining Soil Materials

- ① Go to Mesh > Material
- ② Click on Create. Select **Isotropic**
- ③ Select Model type as **Mohr-Coulomb**
- ④ In General tab, Enter the value of **Elastic modulus (E) =35000 kN/m<sup>2</sup>**
- ⑤ Enter **Poisson's ratio= 0.3, Unit weight = 18 kN/m<sup>3</sup>**.

The screenshot illustrates the software interface for defining a soil material. The ribbon menu at the top shows the 'Mesh' tab selected, with the 'Material' button highlighted. The 'Add/Modify Material' dialog box is open, showing a 'Create...' dropdown menu with 'Isotropic' selected. The 'Material' dialog box is also open, showing the 'General' tab with the following parameters:

Parameter	Value	Unit
Elastic Modulus(E)	35000	kN/m <sup>2</sup>
Inc. of Elastic Modulus	0	kN/m <sup>3</sup>
Inc. of Elastic Modulus Ref. Height	0	m
Poisson's Ratio(v)	0.3	
Unit Weight( $\gamma$ )	18	kN/m <sup>3</sup>
Ko Determination	0.470080736	
Thermal Coefficient	1e-006	1/[T]
Molecular vapor diffusion coefficient	0	m <sup>2</sup> /sec
Thermal diffusion enhancement	0	
Damping Ratio	0.05	
Cohesion(C)	30	kN/m <sup>2</sup>
Frictional Angle( $\phi$ )	36	[deg]
Tensile Strength	0	kN/m <sup>2</sup>

## 2-2 Defining Soil Material

### Procedure

- ➊ Go to Porous Tab > Enter Unit weight (saturated) =19 kN/m<sup>3</sup>
- ➋ Keep Drainage Parameters as Drained
- ➌ Go to Non-Linear Tab > Enter the value of Cohesion = 5 kN/m<sup>2</sup>
- ➍ Input the value of Frictional angle =32°
- ➎ Enter the name as TOP SOIL.
- ➏ Click OK

Follow the same procedure to define the material properties of the other soil layers.

Material

ID 1 Name TOP SOIL Color  

Model Type Mohr-Coulomb  Structure

➊ **Porous** General Non-Linear Thermal Time Dependent

Unit Weight(Saturated) 19 kN/m<sup>3</sup>

Initial Void Ratio(eo) 0.5

➋  Unsaturated Property

Drainage Parameters

Drained

Undrained Poisson's Ratio 0.495

Skempton's B Coefficient 0.97826087

Seepage & Consolidation Parameters

Permeability Coefficients

kx	ky	kz	
1e-005	1e-005	1e-005	m/sec

Void Ratio Dependency of Permeability(ck) 0.5

Specific Storativity(Ss) 5.23021e-1/m Auto

OK Cancel Apply

Material

ID 1 Name TOP SOIL Color  

Model Type Mohr-Coulomb  Structure

General Porous **Non-Linear** Thermal Time Dependent

➋ Cohesion(C) 5 kN/m<sup>2</sup>

Inc. of Cohesion 0 kN/m<sup>3</sup>

Inc. of Cohesion Ref. Height 0 m

➌ Frictional Angle( $\phi$ ) 32 [deg]

Dilatancy Angle 36 [deg]

Tension Cut-off

Tensile Strength 0 kN/m<sup>2</sup>

Cut-off Yield Surface

Pressure  Rankine

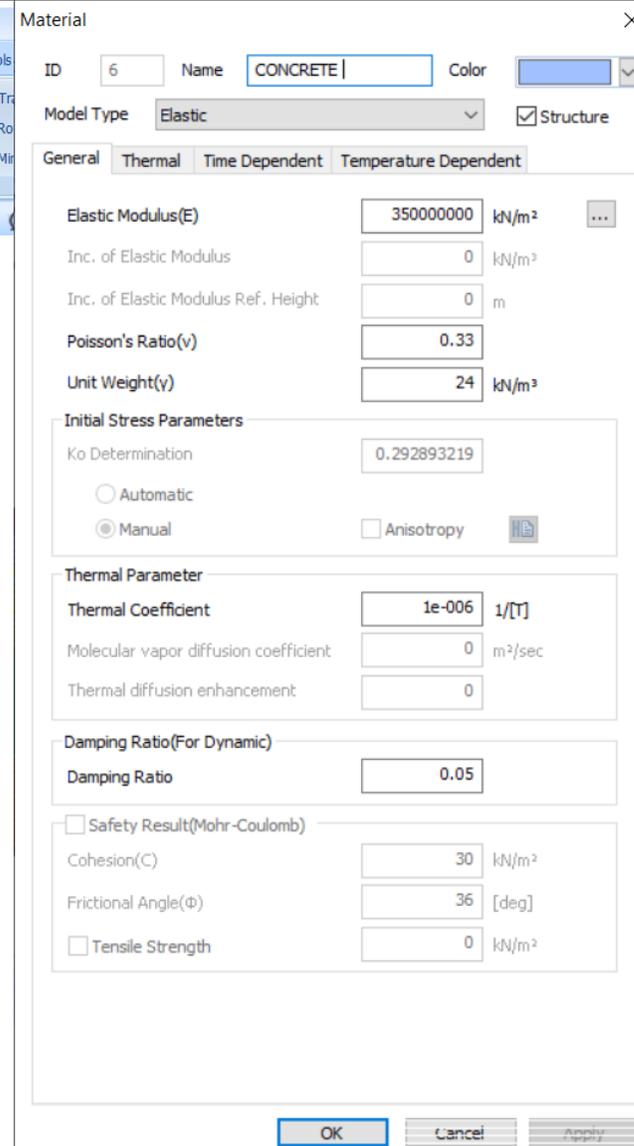
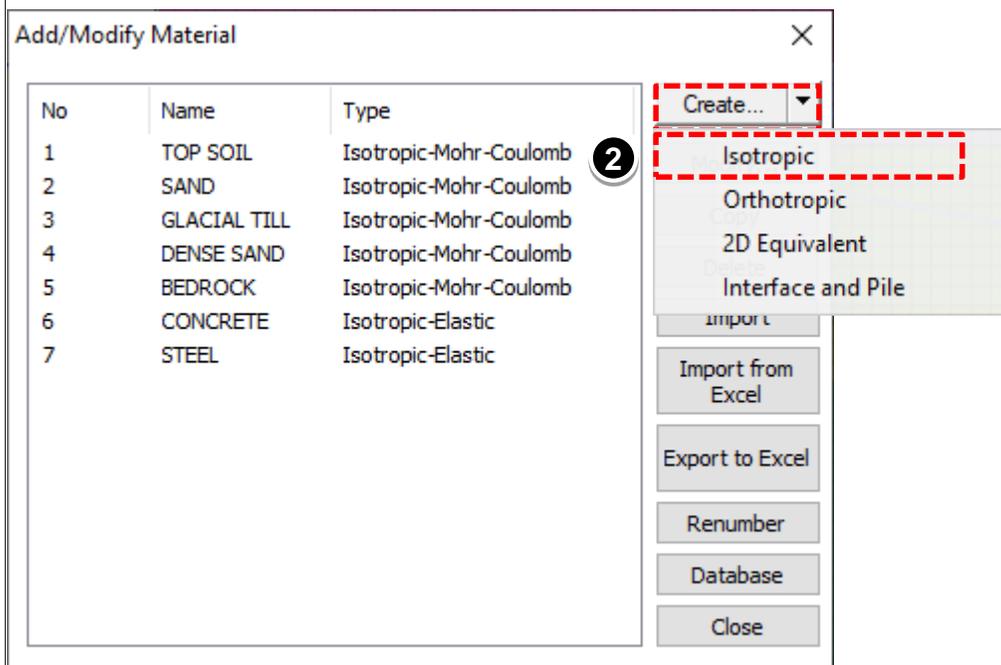
➎ **OK** Cancel Apply

## Procedure

### Defining Soil Materials

- ① Go to Mesh > Material
- ② Click on Create. Select **Isotropic**.
- ③ Select Model type as **CONCRETE**.
- ④ Click OK.

Follow the same procedure to define the material properties of STEEL.



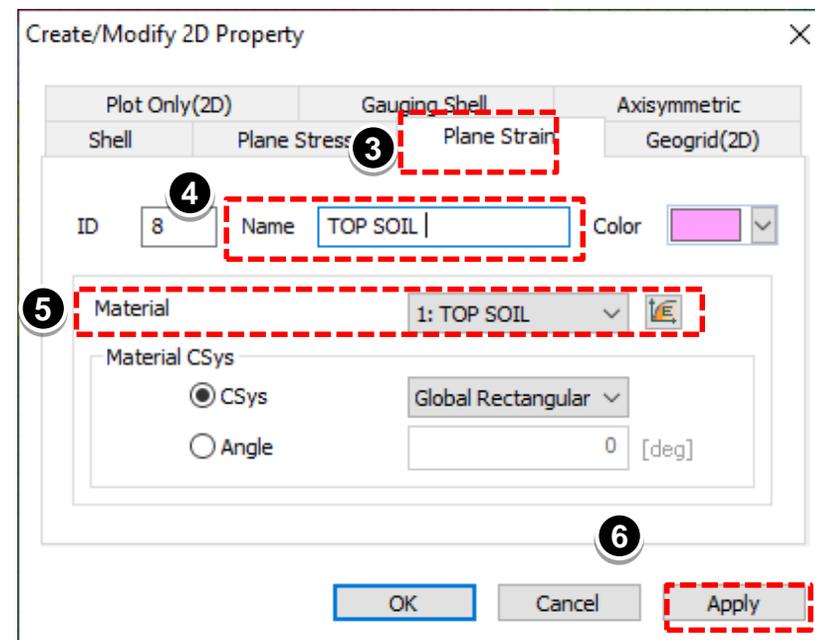
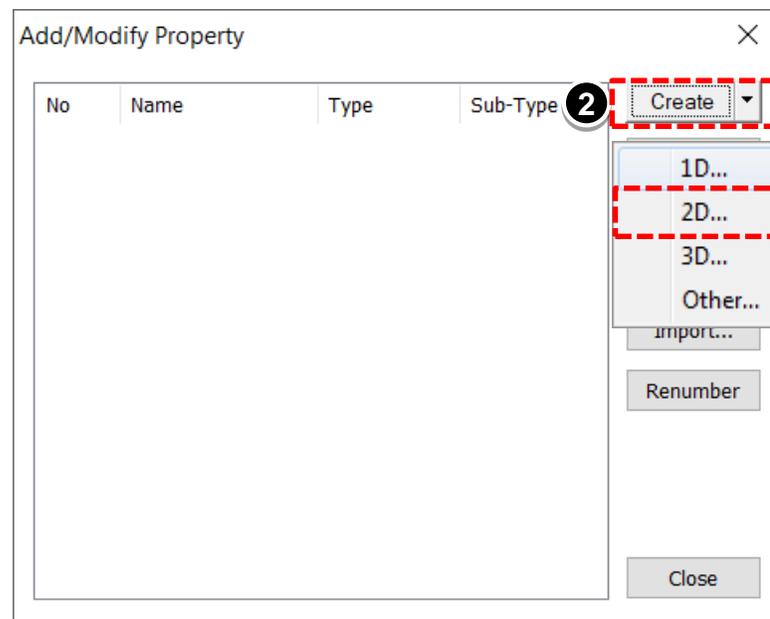
## 3-1 Defining Property (SOIL)

### Procedure

#### Defining Property

- ① Go to Mesh>Click on Property
- ② Click on Create. Select **2D**
- ③ Select **Plane Strain**
- ④ Name it as **TOP SOIL**
- ⑤ Select **TOP SOIL** from Material drop-down menu.
- ⑥ Click **Apply**.

Similarly Define the 2D property for other soil layers.

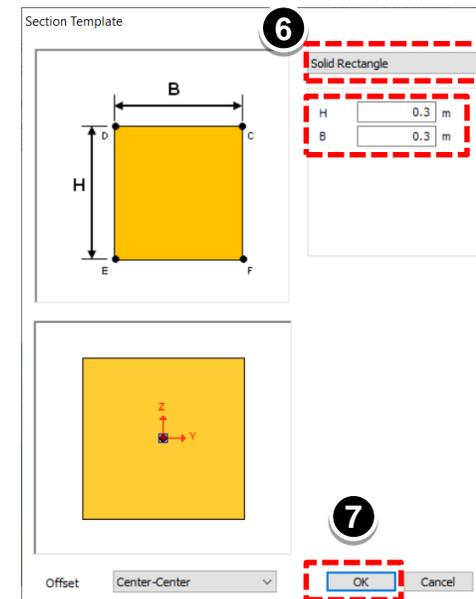
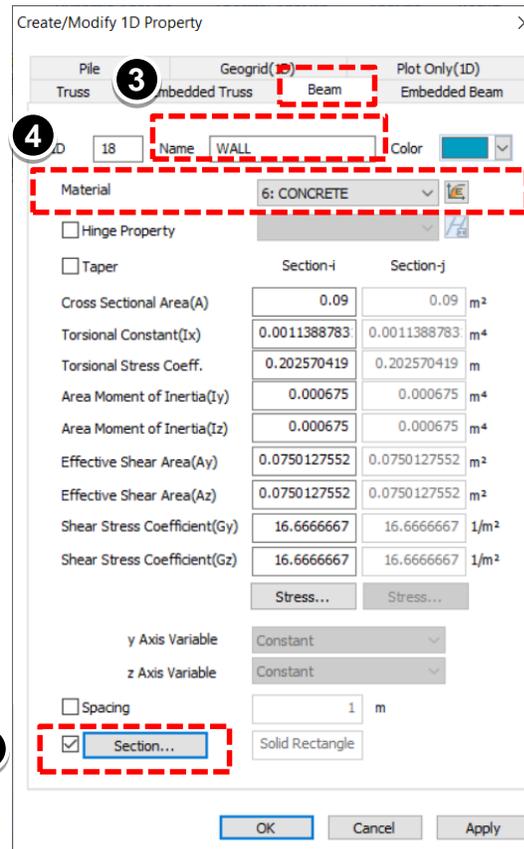
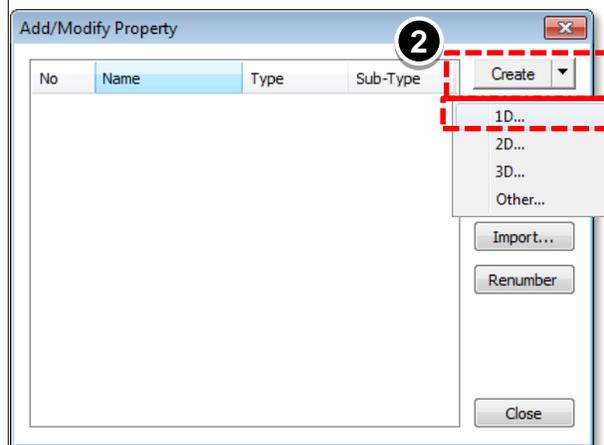


## 3-2 Defining RE WALL Property

## Procedure

## Defining Property

- ① Go to Mesh>Click on Property
- ② Click on Create. Select 1D
- ③ Click on **Beam**
- ④ Name it as **Wall** and select material as **CONCRETE**.
- ⑤ Select **SECTION**.
- ⑥ Select **Solid Rectangle** and enter **H = 0.3m** and **B = 0.3 m**.
- ⑦ Click **OK**.

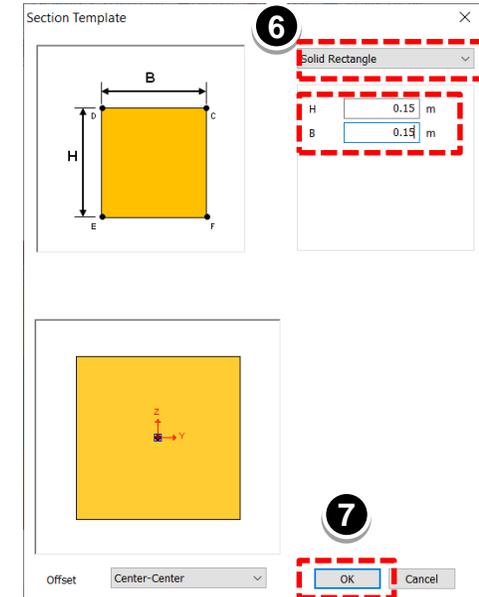
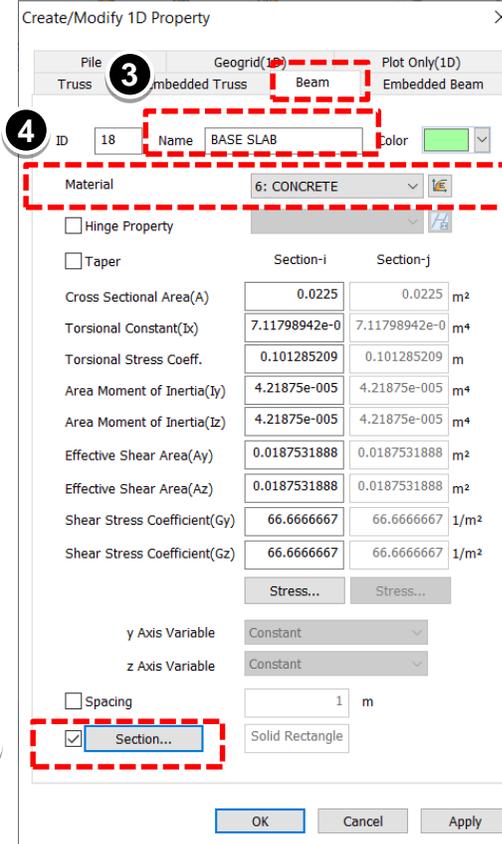
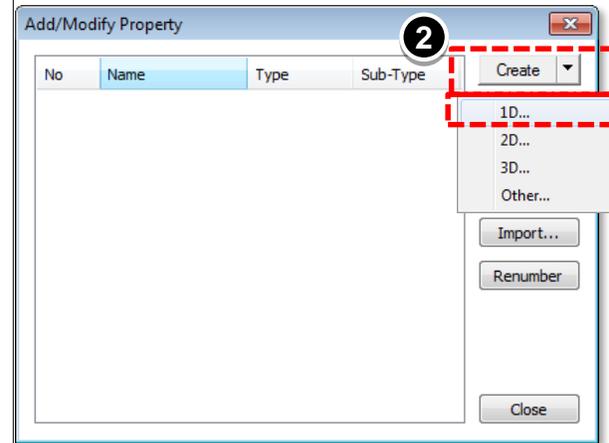


## 3-3 Defining BASE SLAB Property

### Procedure

#### Defining Property

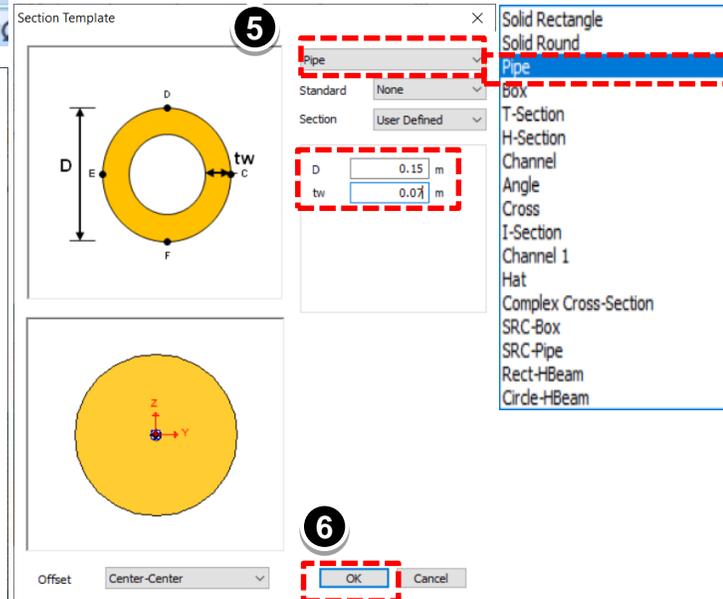
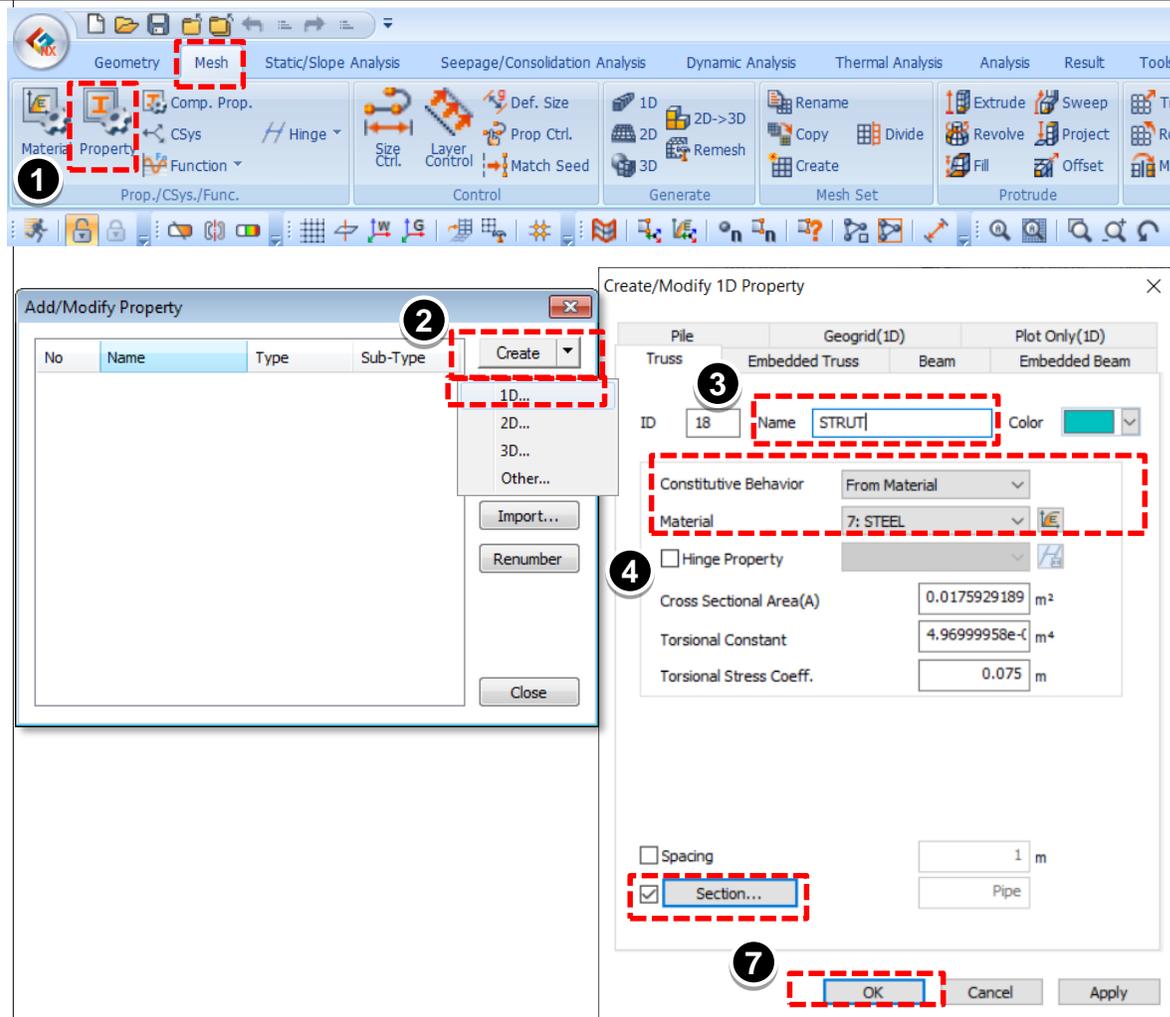
- ① Go to Mesh>Click on Property
- ② Click on Create. Select 1D
- ③ Click on **Beam**
- ④ Name it as **BASE SLAB** and select material as **CONCRETE**.
- ⑤ Select **SECTION**.
- ⑥ Select **Solid Rectangle** and enter **H = 0.15 m** and **B = 0.15 m**.
- ⑦ Click **OK**.



## Procedure

### Defining Property

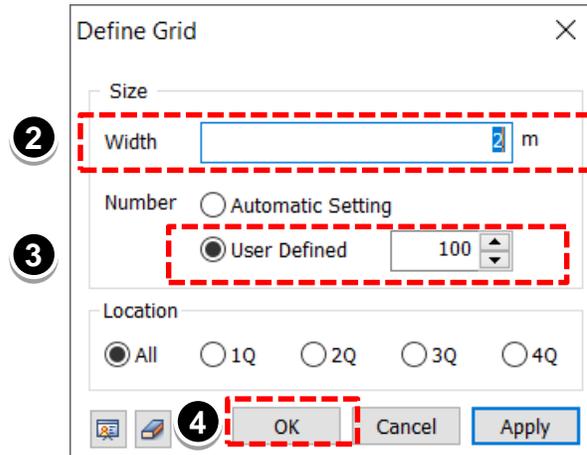
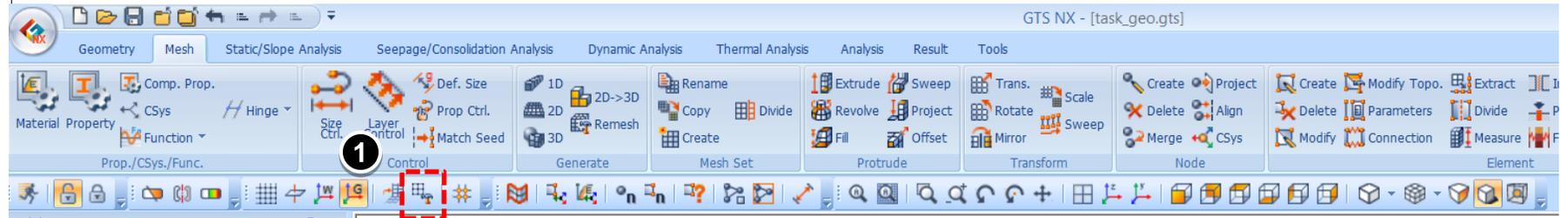
- ① Go to Mesh>Click on Property
- ② Click on Create. Select 1D
- ③ Click on **TRUSS**.
- ④ Name it as **STRUT** and select material as **STEEL**.
- ⑤ Check on **SECTION**.
- ⑥ In the drop down select **PIPE**.  
**D = 0.15m, tw = 0.07m.**
- ⑦ Click **OK**.



## 4-1 Geometric Modelling- Redefining Grid Size

### Procedure

- 1 Go to Geometry > Point
- 2 Enter width as **2m**.
- 3 Click **user defined** and enter **100**.
- 4 Click **OK**.



## 4-2 Geometric modelling-Creation of Points

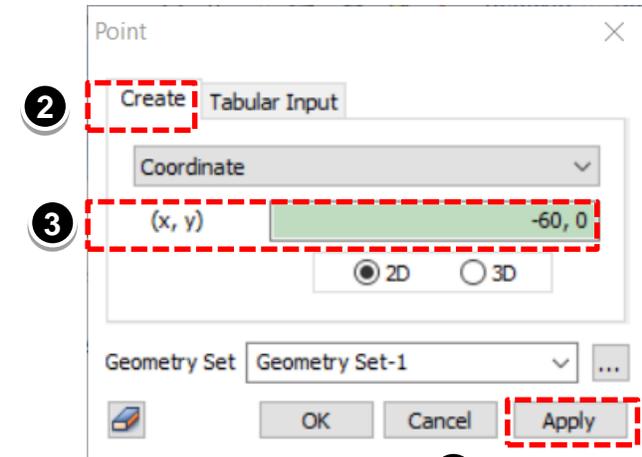
### Procedure

- ❶ Go to Geometry > Point
  - ❷ Click Create.
  - ❸ Enter the points as shown in the table.
  - ❹ Click Apply after entering each point.
- After all the points are created, click OK.



No.	x	y
1	-60	0
2	-6	0
3	6	0
4	60	0
5	-60	-4
6	-6	-4
7	6	-4
8	60	-4
9	-60	-8
10	-6	-8
11	6	-8
12	60	-8
13	-6	-14
14	6	-14
15	-60	-20
16	-6	-20
17	6	-20
18	60	-20
19	-6	-24
20	6	-24
21	-6	-26
22	6	-26

No.	x	y
23	-60	-36
24	60	-36
25	-60	-50
26	60	-50



## 4-3 Geometric modelling-Creation of Lines

### Procedure

- ① Go to Geometry > Point
  - ② Select method as 2D.
  - ③ Input Start corner at Location -60,0
  - ④ Click on method as (Abs X,Y).
  - ⑤ Input End Location -6,0 ( Abs X,Y) and click **Apply**.
  - ⑥ Click **OK**.
- Repeat the same procedure for creating all the lines.

The screenshot displays the software's ribbon interface and two 'Line' dialog boxes. The ribbon includes tabs for Geometry, Mesh, Static/Slope Analysis, Seepage/Consolidation Analysis, Dynamic Analysis, Thermal Analysis, Analysis, Result, and Tools. The 'Geometry' tab is active, showing various tools like Point & Curve, Surface & Solid, Boolean, Divide, Protrude, and Transform.

The first 'Line' dialog box (top) is for creating a line with the following settings:
 

- Method: 2D
- Input Start Location: -60, 0
- Method: ABS x, y
- Geometry Set: Geometry Set-1
- Buttons: OK, Cancel, Apply (highlighted with a red dashed box and circled 5)

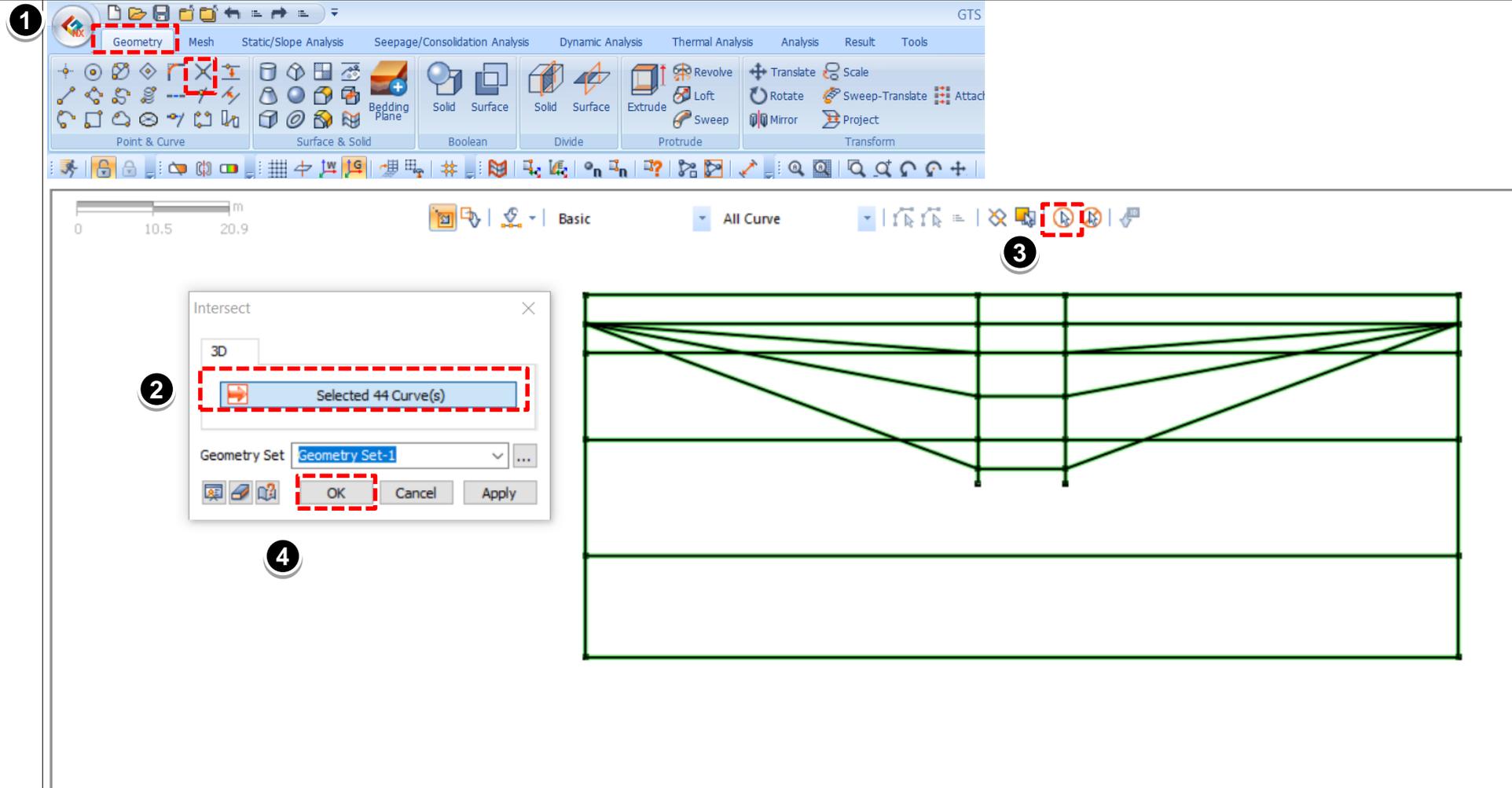
The second 'Line' dialog box (bottom) is for creating a line with the following settings:
 

- Method: 2D
- Input End Location: -6, 0
- Method: ABS x, y
- Geometry Set: Geometry Set-1
- Buttons: OK (highlighted with a red dashed box and circled 6), Cancel, Apply

To the right of the dialog boxes is a grid of 26 numbered points (1-26) arranged in a rectangular pattern. Lines connect the points to form a series of horizontal and vertical lines, with diagonal lines connecting the top-left and bottom-right corners to the top-right and bottom-left corners respectively.

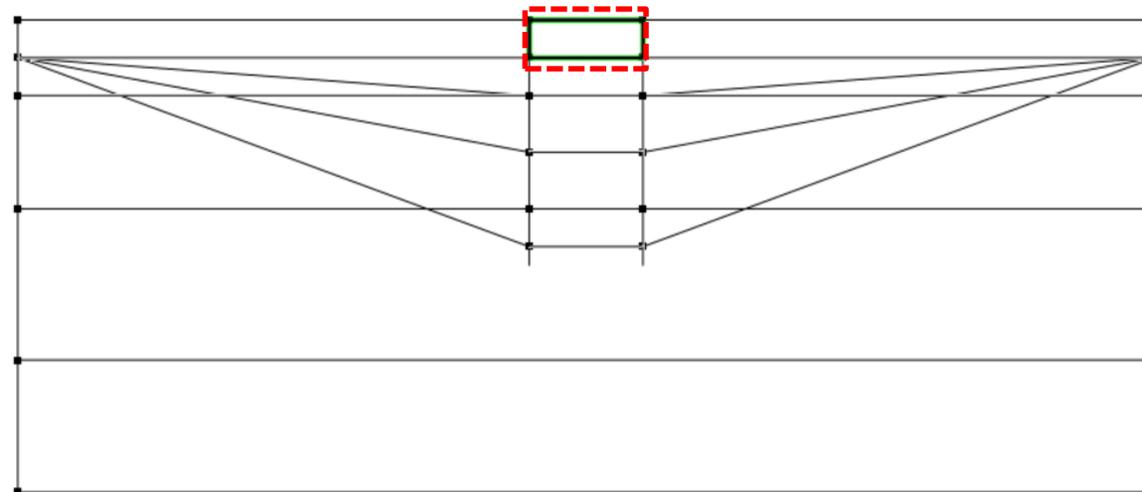
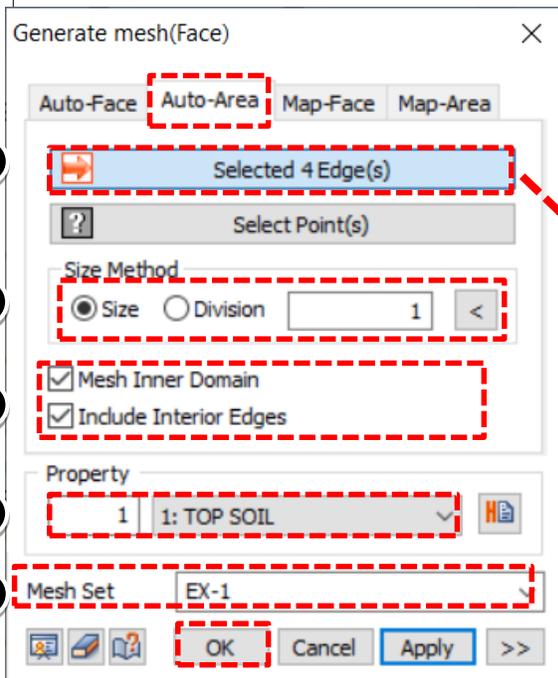
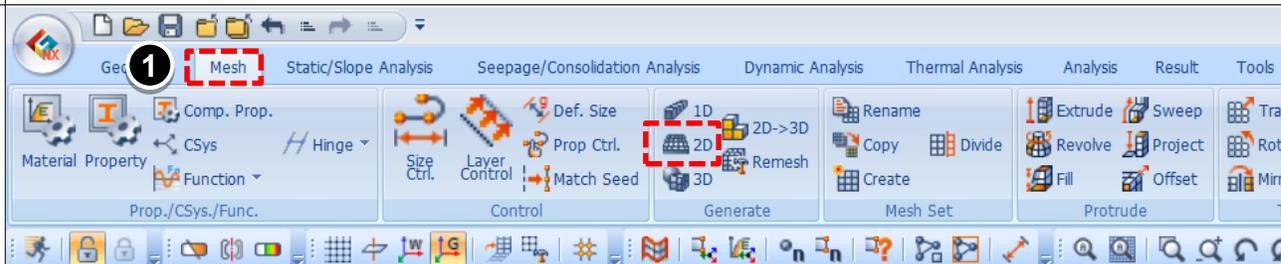
**Procedure**

- 1 Go to Geometry > Intersect
- 2 Select the curves drawn in the previous steps.
- 3 The **Select All** option can also be used to select all the curves..
- 4 Click OK.



### Procedure

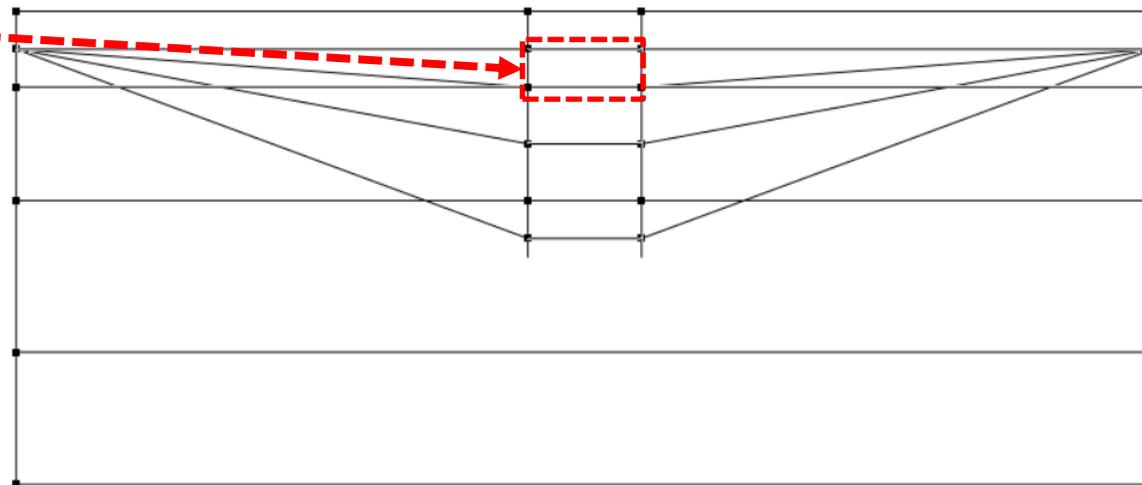
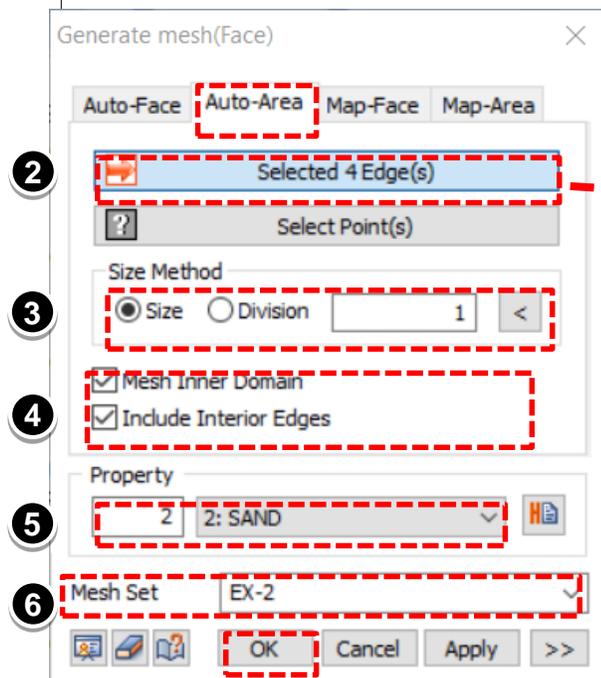
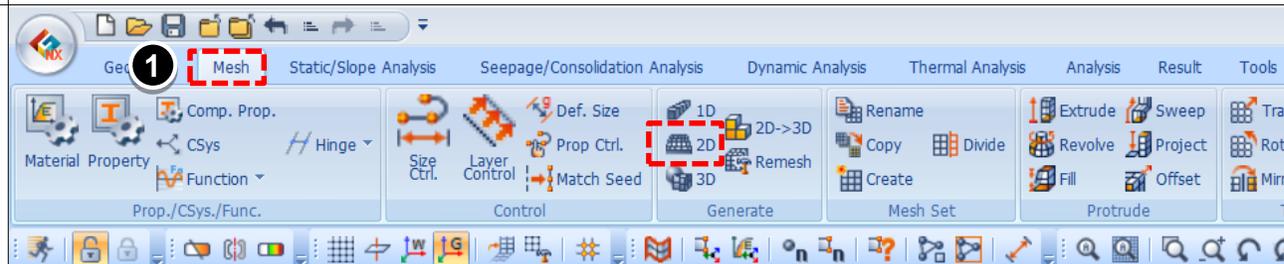
- ① Go to Mesh > Generate > 2D
- ② Select the lines as shown.
- ③ In the Size method give size 1 m.
- ④ Tick on the “Mesh Inner Domain”.
- ⑤ In the property drop-down menu, select TOP SOIL.
- ⑥ Give the Mesh set name “EX-1.”
- ⑦ Click OK.



⑦

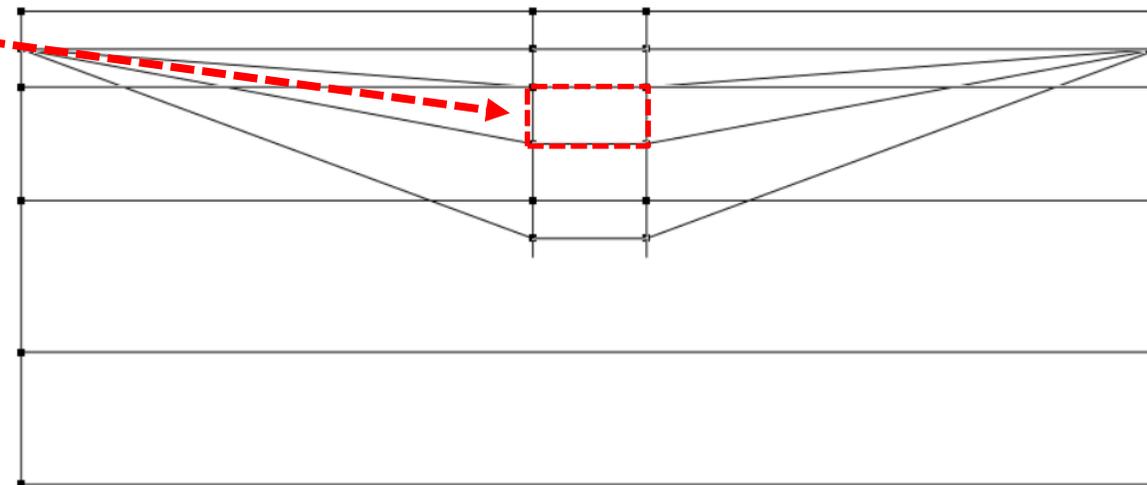
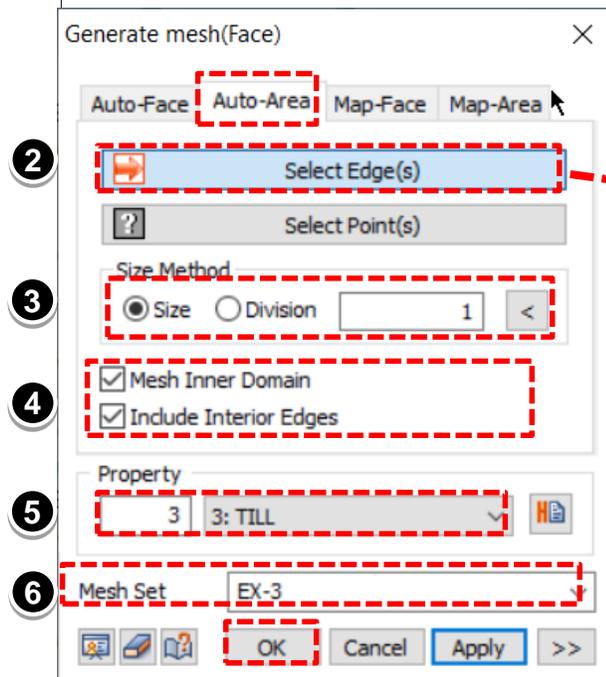
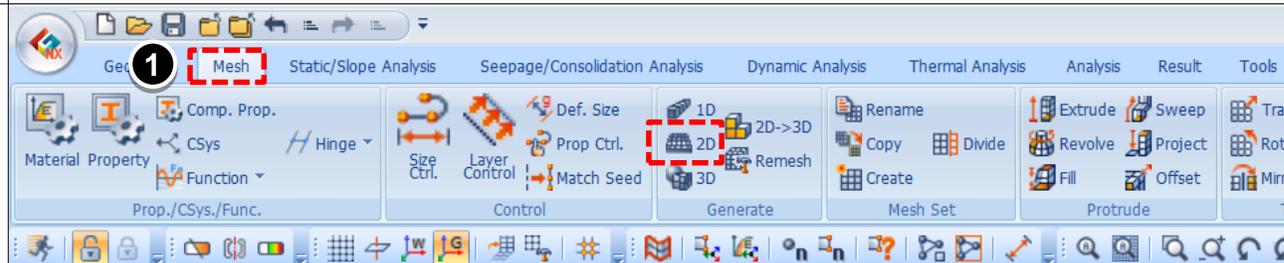
### Procedure

- 1 Go to Mesh > Generate > 2D
- 2 Select the lines as shown.
- 3 In the Size method give size 1 m.
- 4 Tick on the “Mesh Inner Domain”.
- 5 In the property drop-down menu, select **SAND**.
- 6 Give the Mesh set name “EX-2.”
- 7 Click OK.



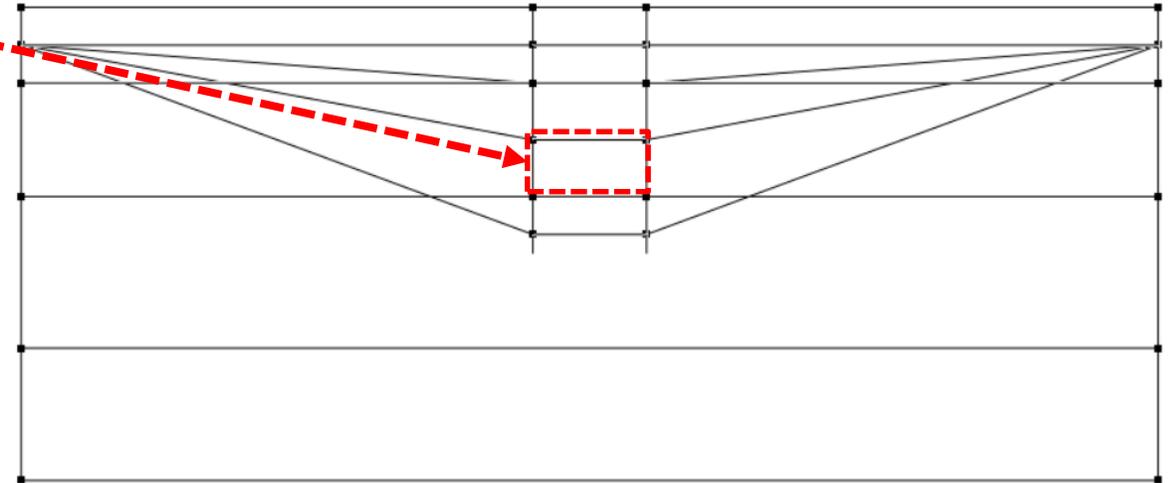
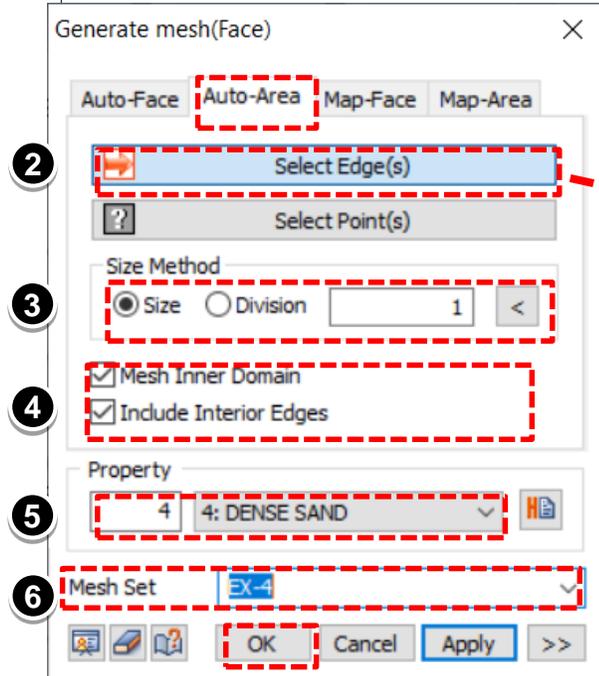
### Procedure

- 1 Go to Mesh > Generate > 2D
- 2 Select the lines as shown.
- 3 In the Size method give size 1 m.
- 4 Tick on the “Mesh Inner Domain”.
- 5 In the property drop-down menu, select TILL.
- 6 Give the Mesh set name “EX-3.”
- 7 Click OK.



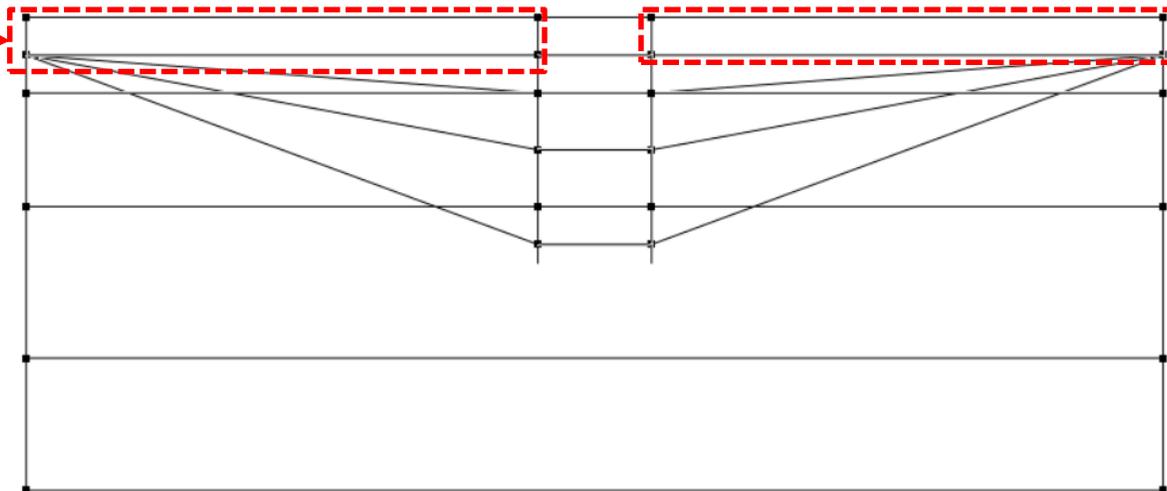
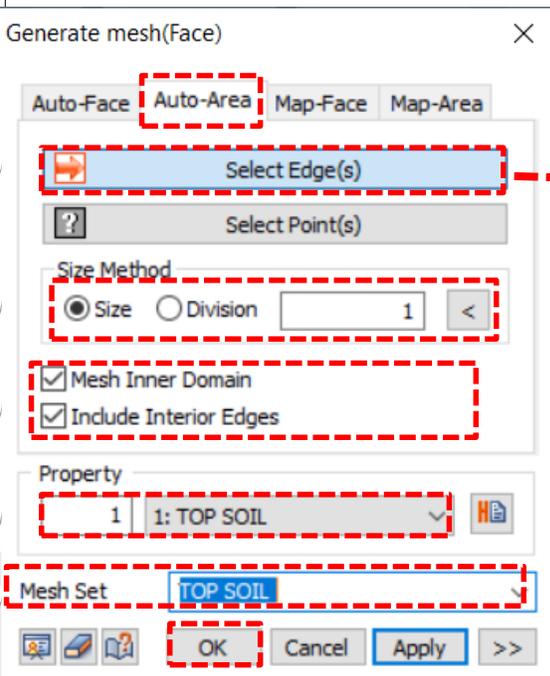
### Procedure

- 1 Go to Mesh > Generate > 2D
- 2 Select the lines as shown.
- 3 In the Size method give size 1 m.
- 4 Tick on the “Mesh Inner Domain”.
- 5 In the property drop-down menu, select **DENSE SAND**.
- 6 Give the Mesh set name “EX-4.”
- 7 Click OK.



### Procedure

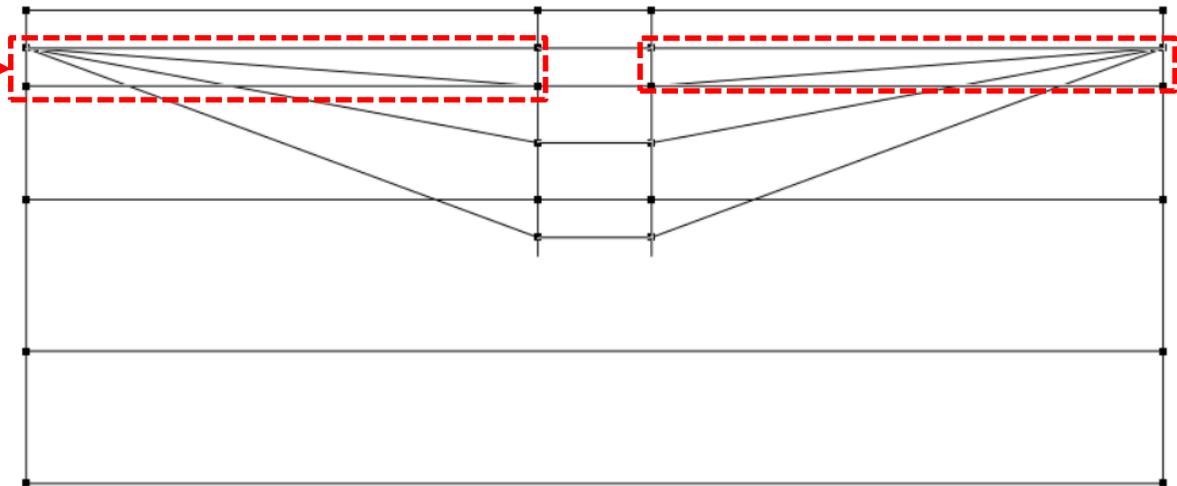
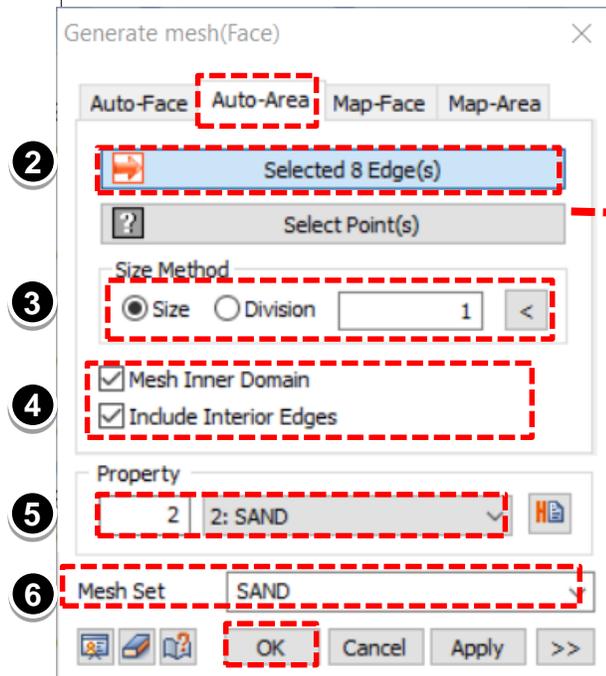
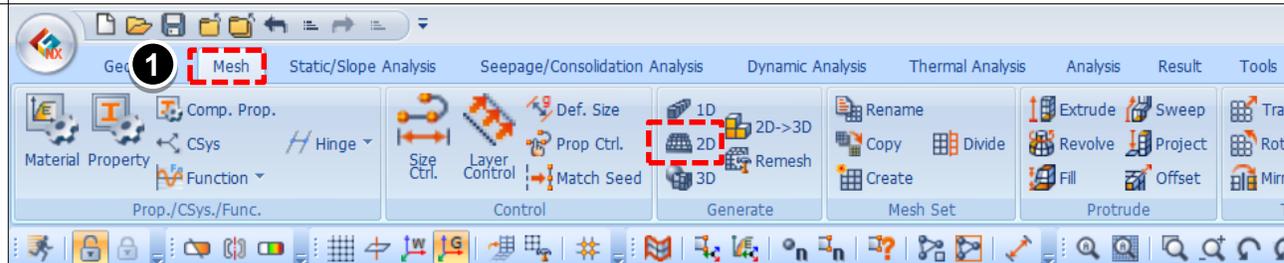
- 1 Go to Mesh > Generate > 2D
- 2 Select the lines as shown.
- 3 In the Size method give size 1 m.
- 4 Tick on the "Mesh Inner Domain".
- 5 In the property drop-down menu, select TOP SOIL.
- 6 Give the Mesh set name "TOP SOIL."
- 7 Click OK.



7

### Procedure

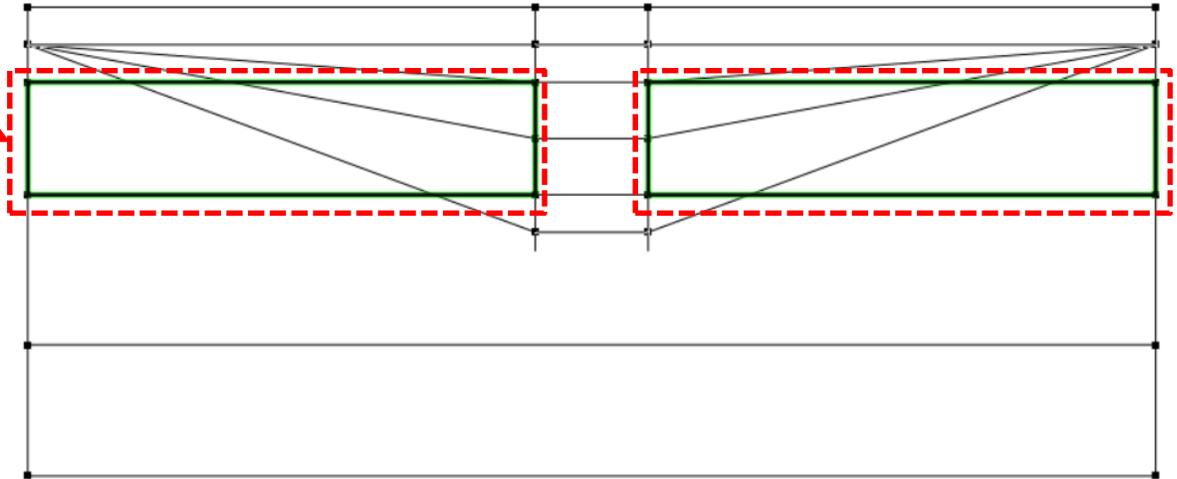
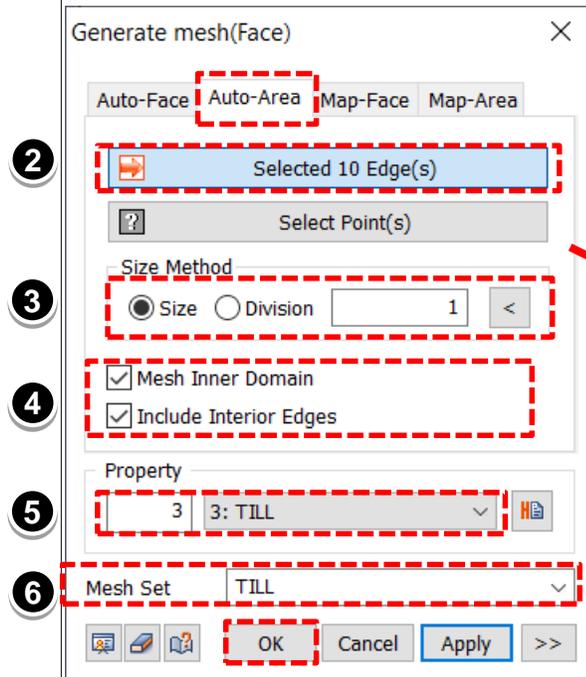
- 1 Go to Mesh > Generate > 2D
- 2 Select the lines as shown.
- 3 In the Size method give size 1 m.
- 4 Tick on the "Mesh Inner Domain".
- 5 In the property drop-down menu, select **SAND**.
- 6 Give the Mesh set name "SAND."
- 7 Click OK.



7

### Procedure

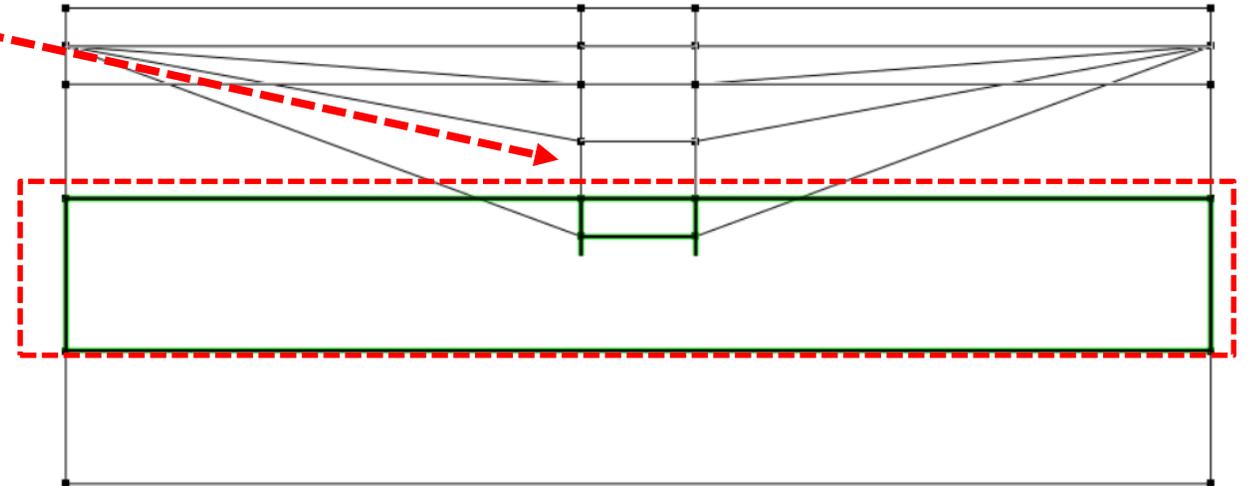
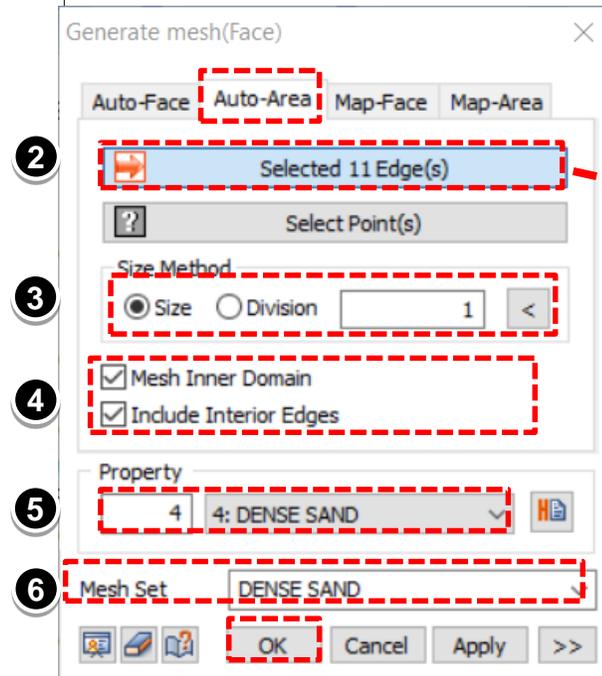
- 1 Go to Mesh > Generate > 2D
- 2 Select the lines as shown.
- 3 In the Size method give size 1 m.
- 4 Tick on the "Mesh Inner Domain".
- 5 In the property drop-down menu, select TILL.
- 6 Give the Mesh set name "TILL."
- 7 Click OK.



7

### Procedure

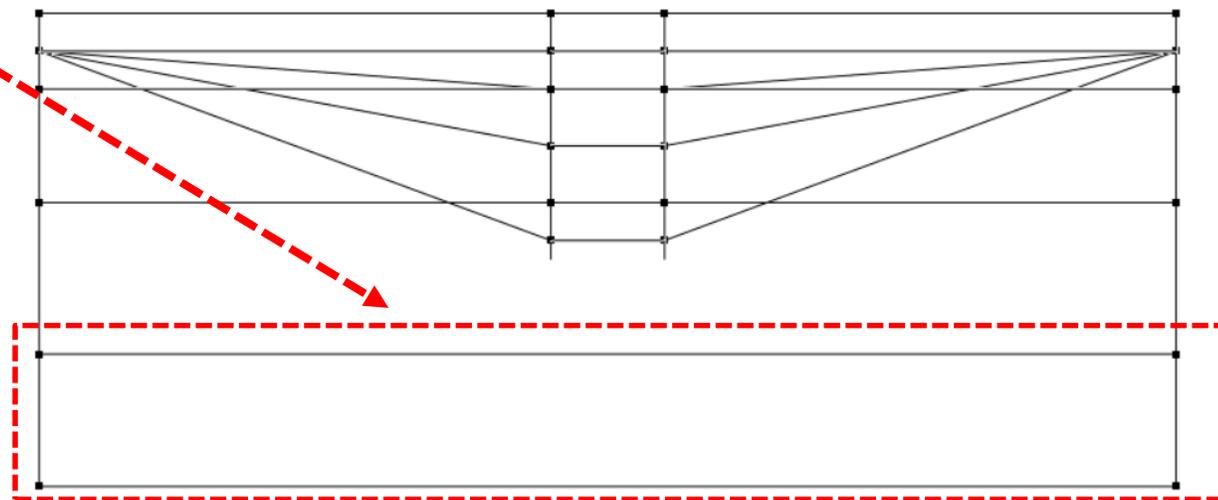
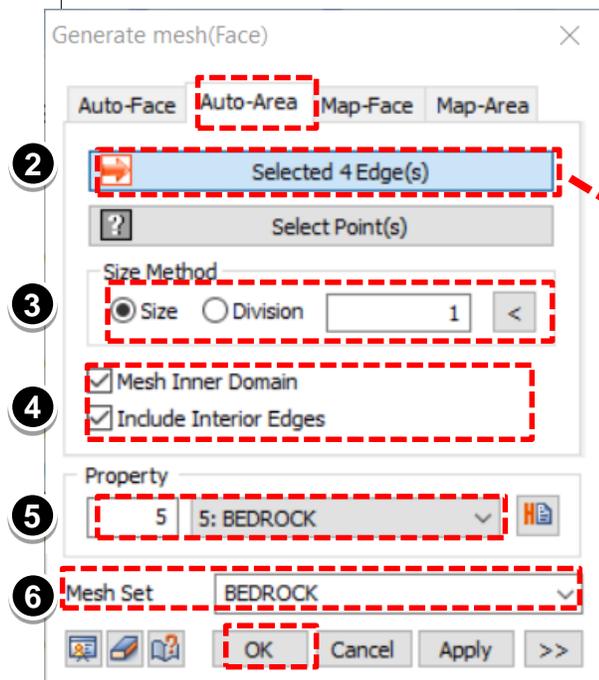
- ① Go to Mesh > Generate > 2D
- ② Select the lines as shown.
- ③ In the Size method give size 1 m.
- ④ Tick on the “Mesh Inner Domain”.
- ⑤ In the property drop-down menu, select **DENSE SAND**.
- ⑥ Give the Mesh set name “DENSE SAND.”
- ⑦ Click OK.



⑦

### Procedure

- 1 Go to Mesh > Generate > 2D
- 2 Select the lines as shown.
- 3 In the Size method give size 1 m.
- 4 Tick on the “Mesh Inner Domain”.
- 5 In the property drop-down menu, select **BEDROCK**.
- 6 Give the Mesh set name “BEDROCK.”
- 7 Click OK.

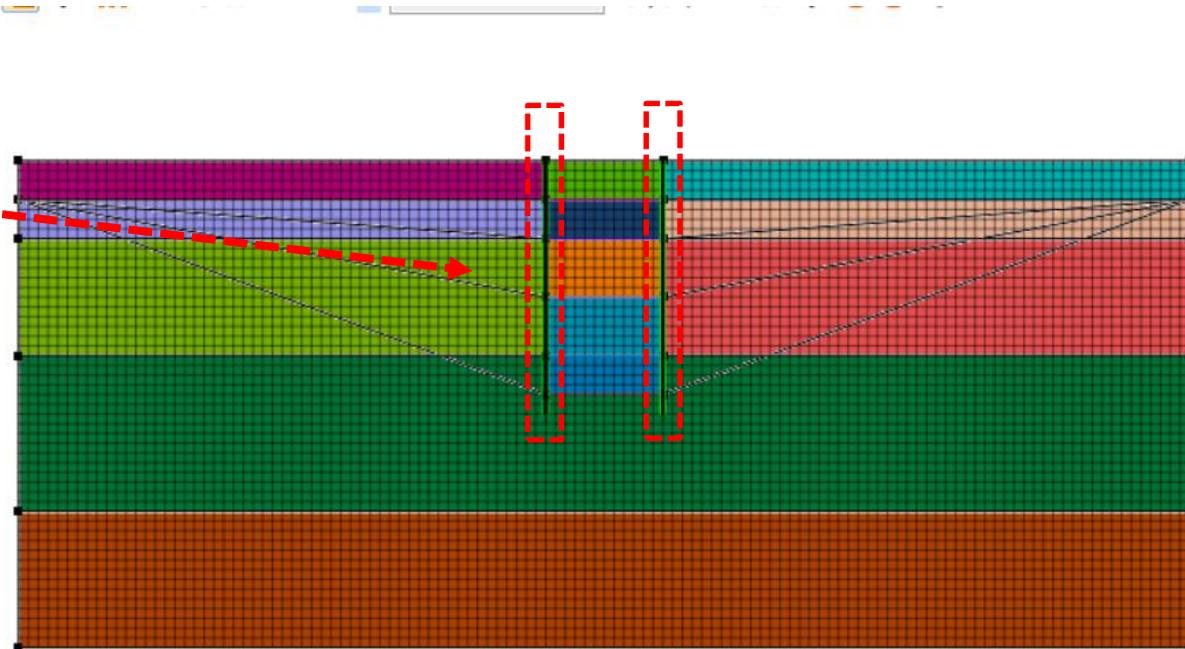
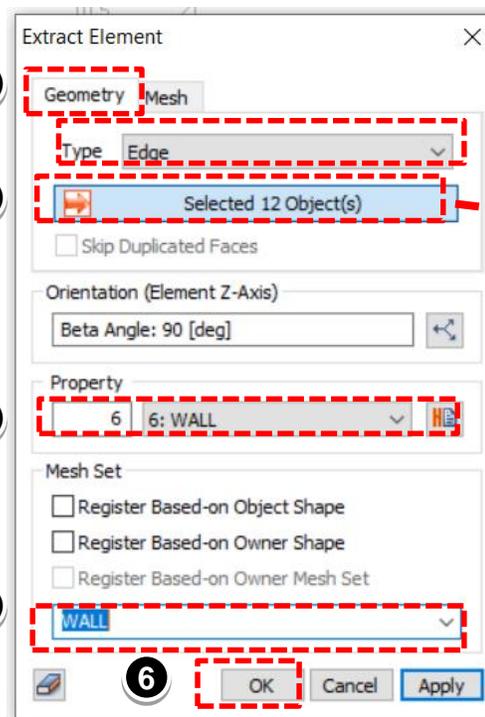
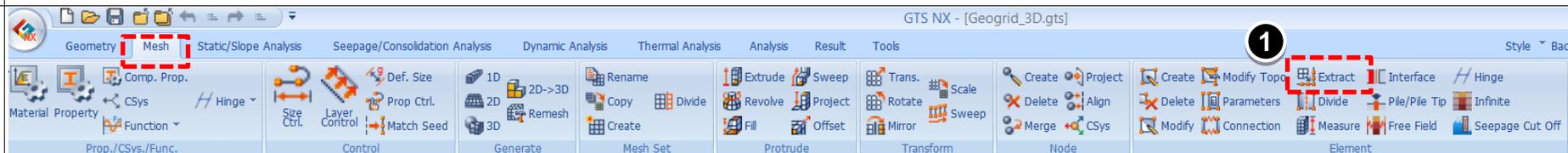


7

# 5-10 Meshing-Extraction of RE WALL Elements

## Procedure

- ① Go to Mesh>Element>Extract
- ② Change the type to **Edge**.
- ③ Select the four edges as shown in the enclosed box.
- ④ The property is **WALL**.
- ⑤ Name retaining wall mesh set as **WALL**.
- ⑥ Click **OK**.

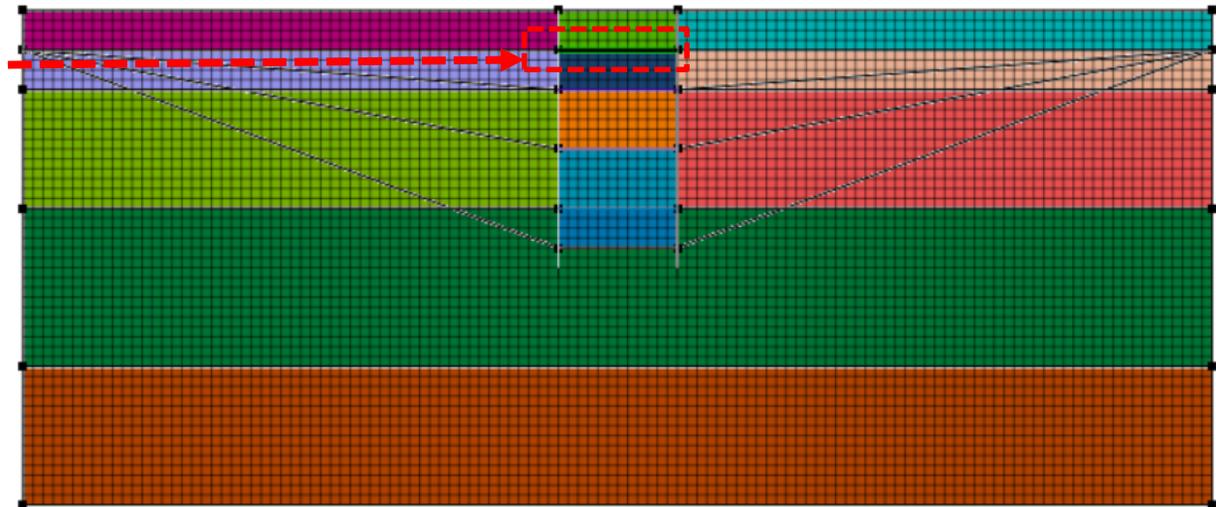
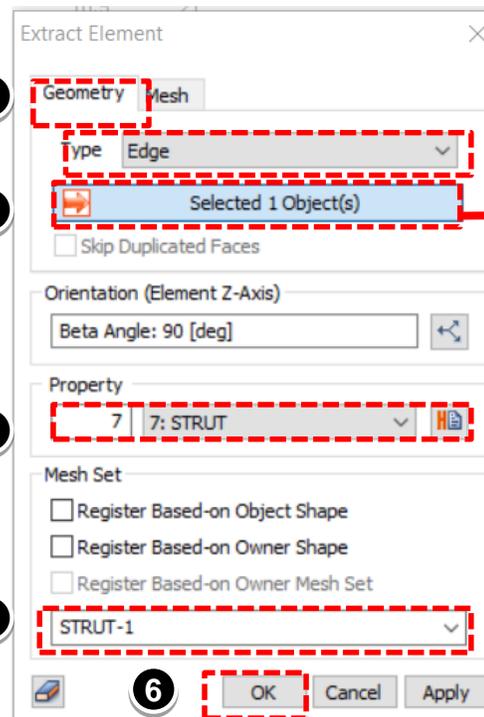
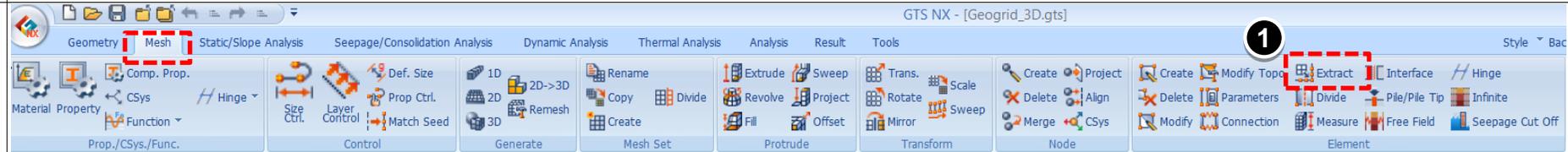


7

# 5-11 Meshing-Extraction of STRUT Elements

## Procedure

- ① Go to Mesh>Element>Extract
- ② Change the type to **Edge**.
- ③ Select the four edges as shown in the enclosed box.
- ④ The property is **STRUT**.
- ⑤ Name STRUT mesh set as **STRUT-1**.
- ⑥ Click **OK**.

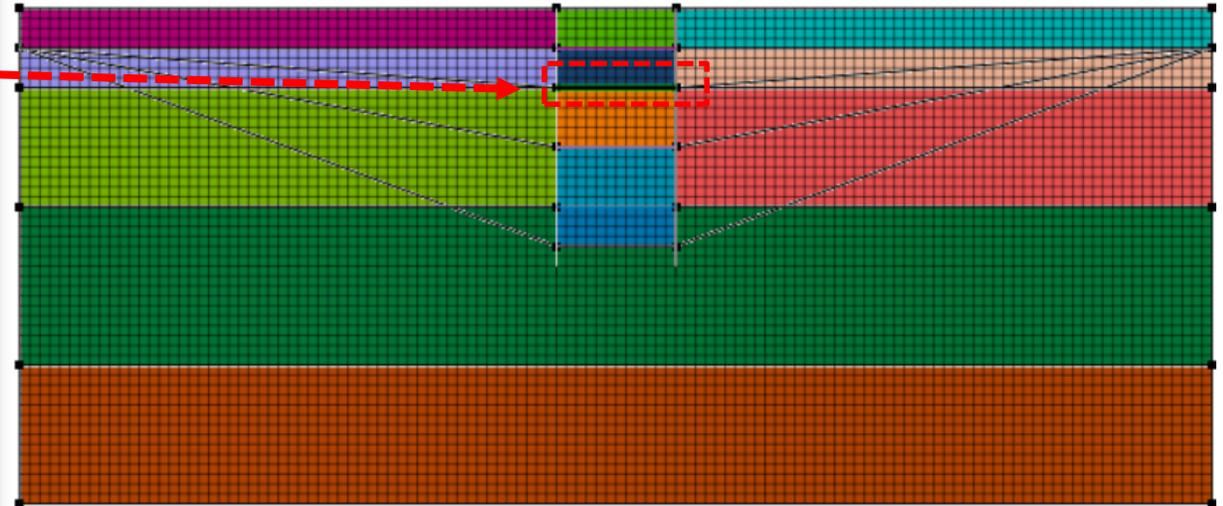
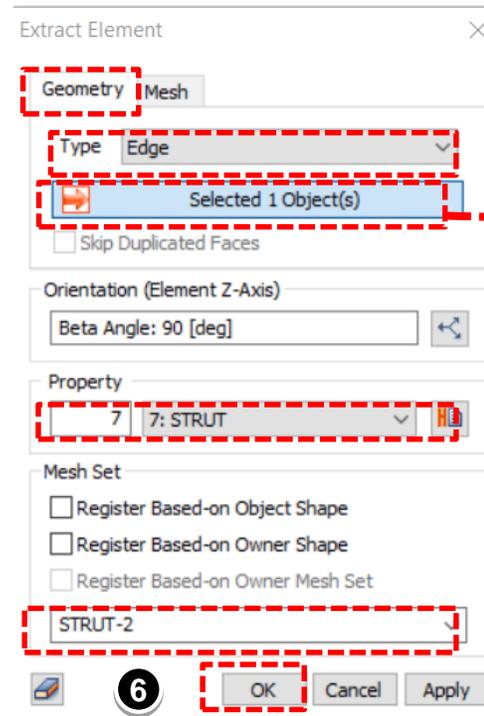
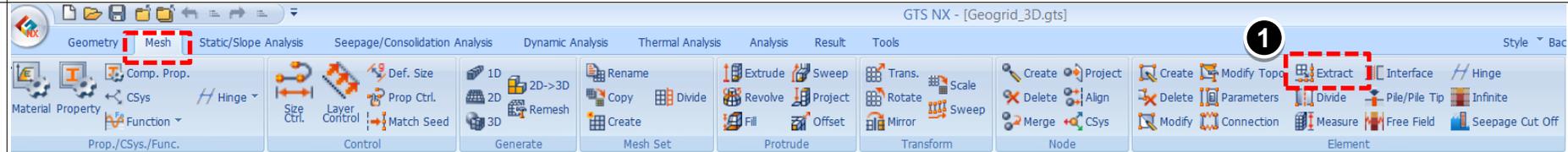


7

# 5-12 Meshing-Extraction of STRUT Elements

## Procedure

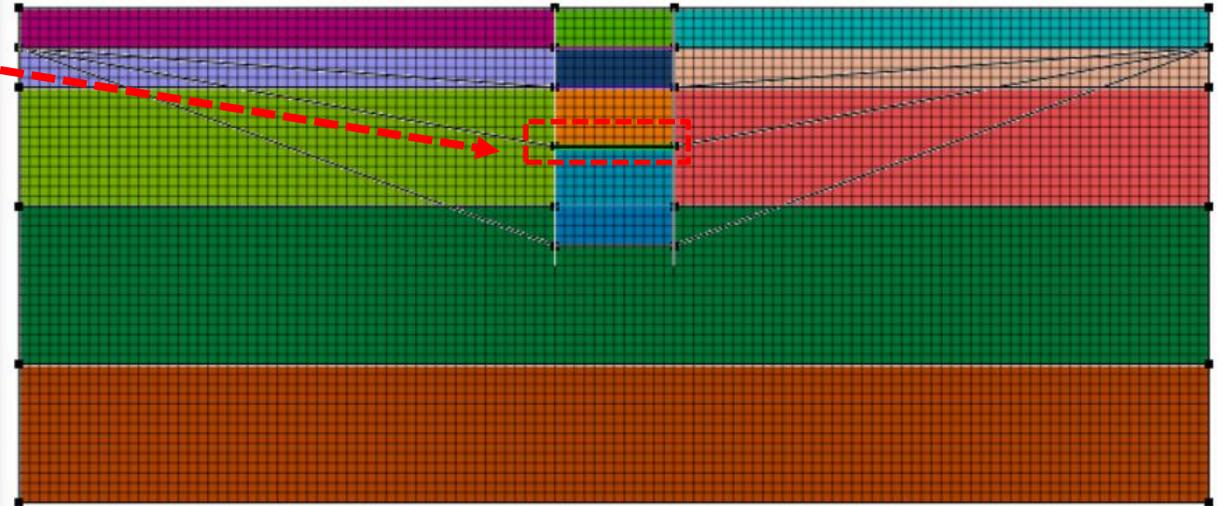
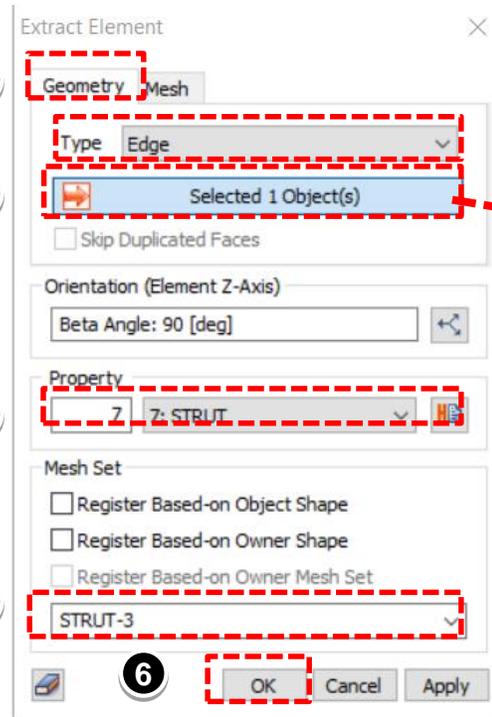
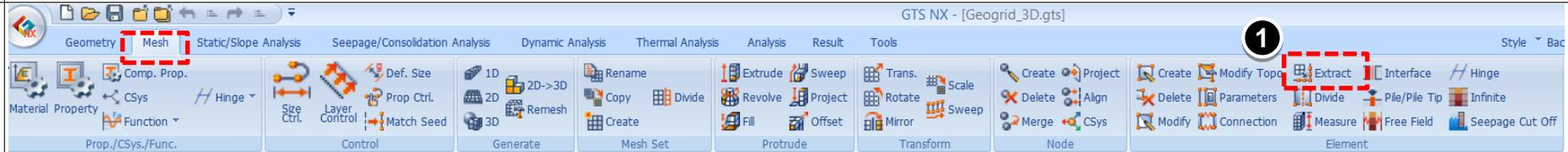
- 1 Go to Mesh>Element>Extract
- 2 Change the type to **Edge**.
- 3 Select the four edges as shown in the enclosed box.
- 4 The property is **STRUT**.
- 5 Name STRUT mesh set as **STRUT-2**.
- 6 Click **OK**.



# 5-13 Meshing-Extraction of STRUT Elements

## Procedure

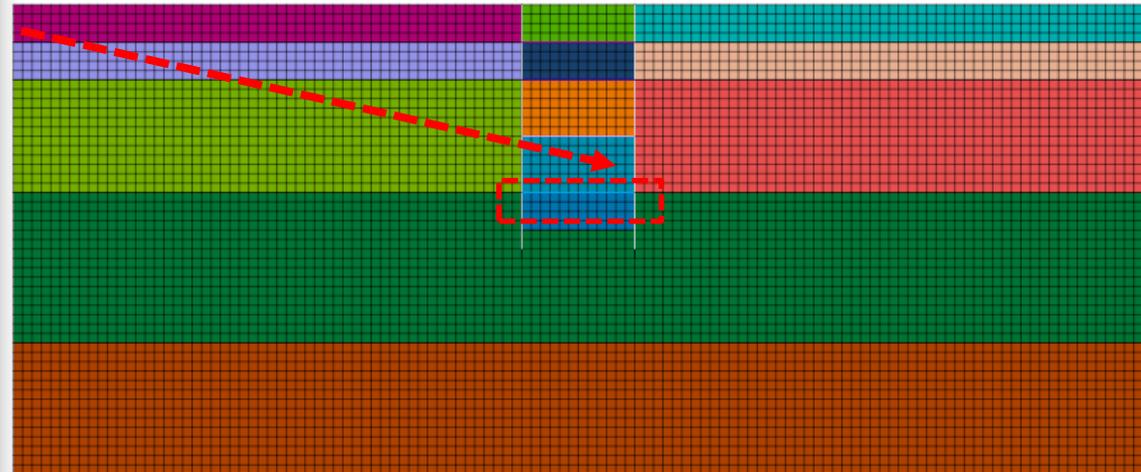
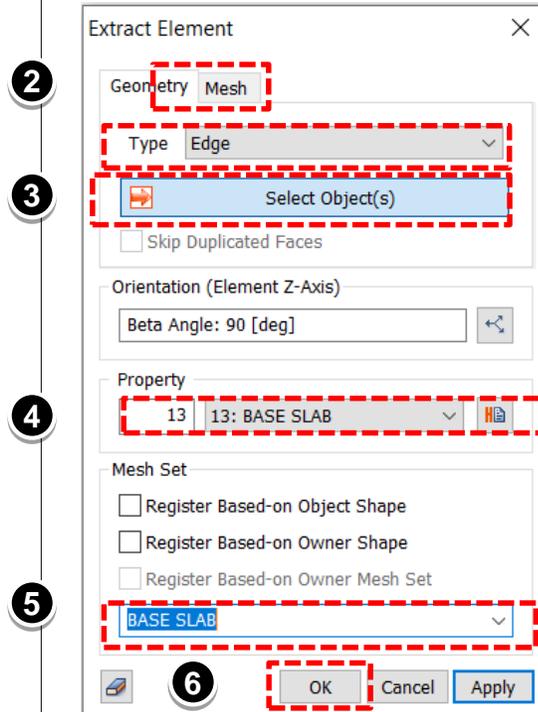
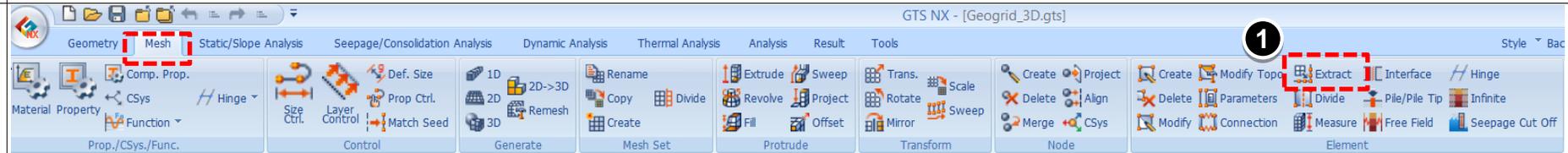
- ① Go to Mesh>Element>Extract
- ② Change the type to **Edge**.
- ③ Select the four edges as shown in the enclosed box.
- ④ The property is **STRUT**.
- ⑤ Name STRUT mesh set as **STRUT-3**.
- ⑥ Click **OK**.



## 5-13 Meshing-Extraction of BASE SLAB.

### Procedure

- ① Go to Mesh>Element>Extract
- ② Change the type to **Edge**.
- ③ Select the four edges as shown in the enclosed box.
- ④ The property is **BASE SLAB**.
- ⑤ Name mesh set as **BASE SLAB**.
- ⑥ Click **OK**.



# 5-14 Meshing- Creation of Interface Elements

## Procedure

- 1 Go to Mesh>Element>Interface.
- 2 Change the type to **LINE**.
- 3 In the drop down menu select "**From Truss/Beam**".
- 4 Select the **Wall** Mesh elements as shown in the picture.
- 5 In property parameters, SELECT **Wizard**.
- 6 Enter the value of Strength Reduction Factor as **0.65**.
- 7 Click **OK**.
- 8 Click on "**Create Other Element**" to create **Rigid Link**.
- 9 Name mesh set as **Line Interface**.
- 10 Click **OK**.

The screenshot displays the GTS NX software interface for creating an interface element. The main window shows the 'Interface' dialog box with the following settings:

- Type:** Line
- Method:** From Truss/Beam
- Parameters:** Selected 52 Object(s)
- Property Parameters:** Wizard (selected)
- Create Other Element:** Rigid Link
- Mesh Set:** Line Interface

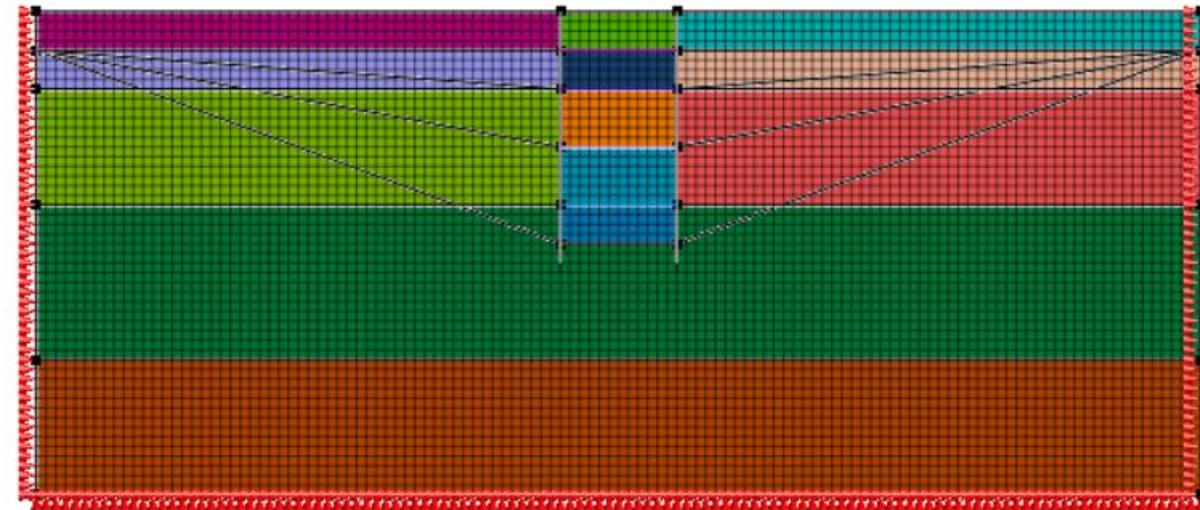
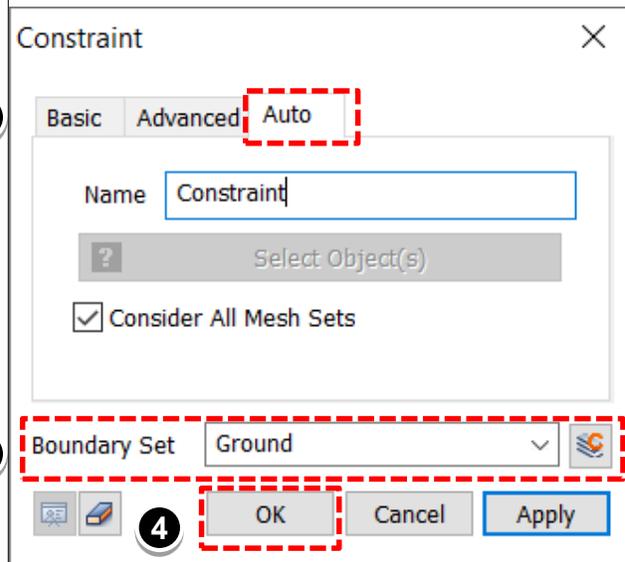
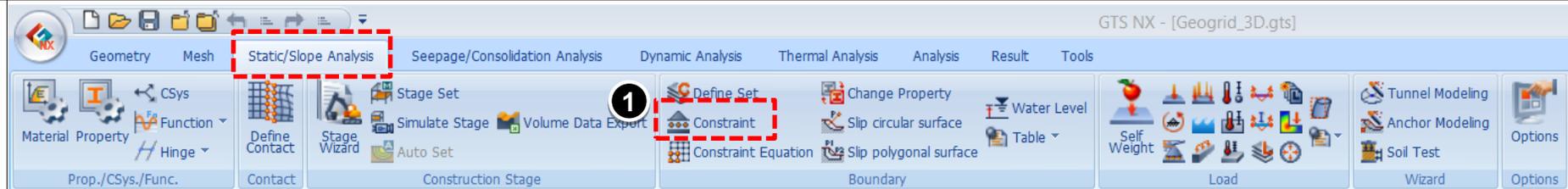
The 'Interface Wizard Data' dialog box is also visible, showing the following settings:

- Structural Parameters:**
  - Virtual Thickness Factor (tv): 0.1 m
  - Strength Reduction Factor (R): 0.65
- Line Interface Thickness:** 1 m
- Conduction for Seepage flow:** 0 m/sec/m

The 3D model on the right shows a meshed structure with a red dashed box highlighting the selected wall mesh elements and the interface location.

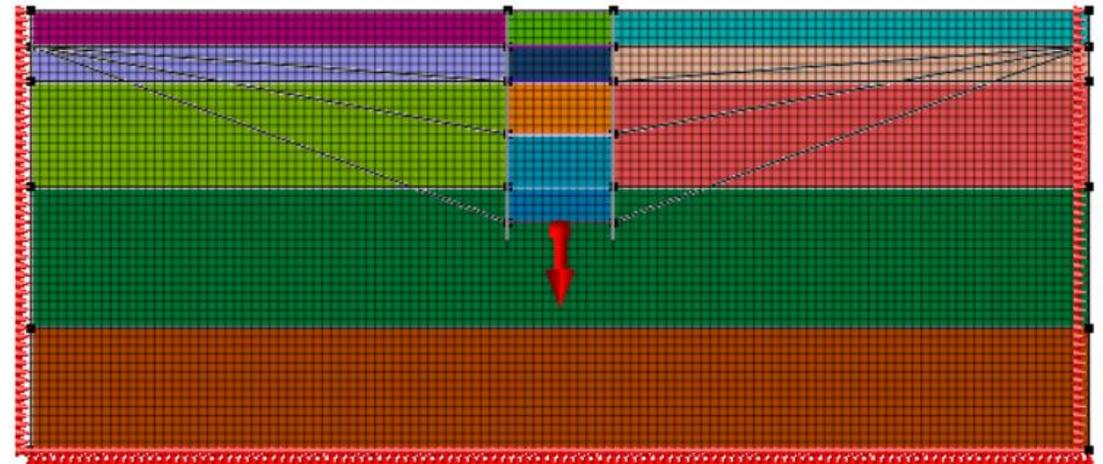
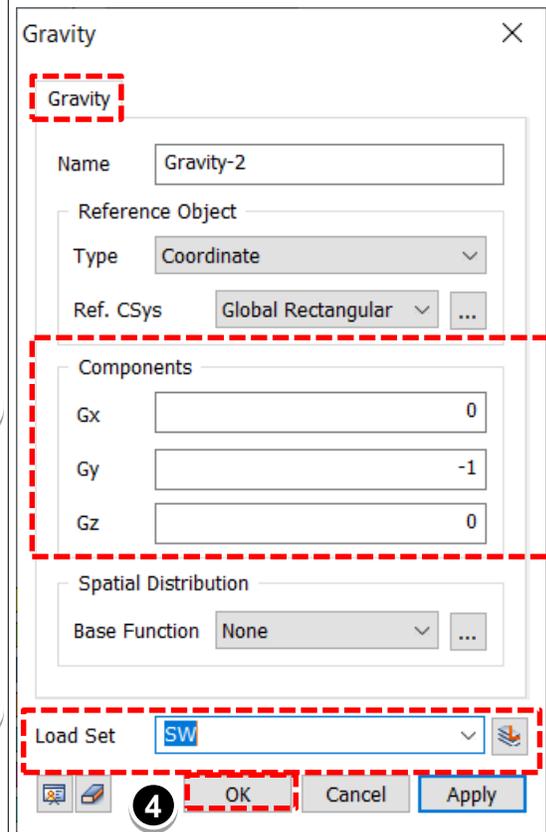
## Procedure

- ① Go to Static/Slope Analysis > Constraint
- ② Select Auto, check on “**Consider All mesh Sets**”
- ③ Name the boundary Set as “**Ground**”
- ④ Click **OK**



### Procedure

- ❶ Go to Static/Slope Analysis > Load > Self Weight
- ❷ Check for gravity Direction,  $G_y$  should be -1
- ❸ Name the Load Set as "SW"
- ❹ Click OK



## 7-2 Load Definition- Surcharge/ Pressure Load

### Procedure

- ➊ Go to Static/Slope Analysis > Load > Pressure Load
- ➋ Select **EDGE**.
- ➌ Select object type as **2D Element Edge**.
- ➍ Select the 2D elements as shown in the picture.
- ➎ Enter the value **50kN/m** in **P** or **P1**.
- ➏ Name the load set as **LOAD**.
- ➐ Click **OK**

GTS NX - [Midas model.gts]

Geometry Mesh **Static/Slope Analysis** Seepage/Consolidation Analysis Dynamic Analysis Thermal Analysis Analysis Result Tools

Material Property Function CSys Hinge Define Contact Stage Wizard Stage Set Simulate Stage Volume Data Export Auto Set Define Set Change Property Water Level Self Weight Tunnel Modeling Anchor Modeling Soil Test Wizard Options

Prop./CSys./Func. Contact Construction Stage Boundary Load Wizard Options

**1**

Pressure

**2** Edge Face Axisymmetric

Name Pressure-1

Object

**3** Type 2D Element Edge

**4** Selected 18 Object(s)

Direction

Type Normal

Ref. CSys Global Rectangular

X  Y  Z

Magnitude

Uniformly Distributed Load

Base Function None

**5** P or P1 50 kN/m

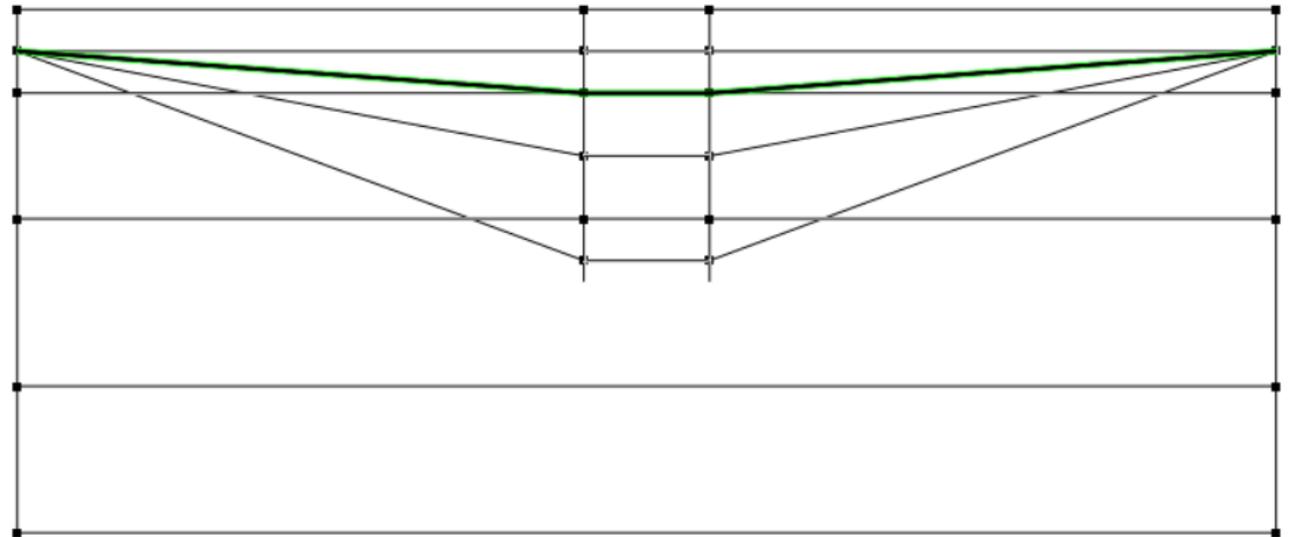
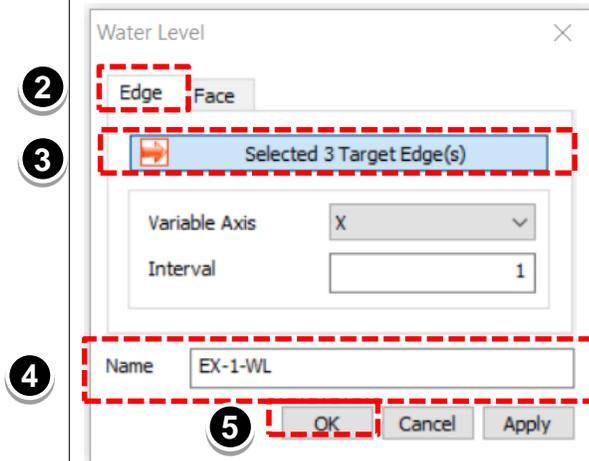
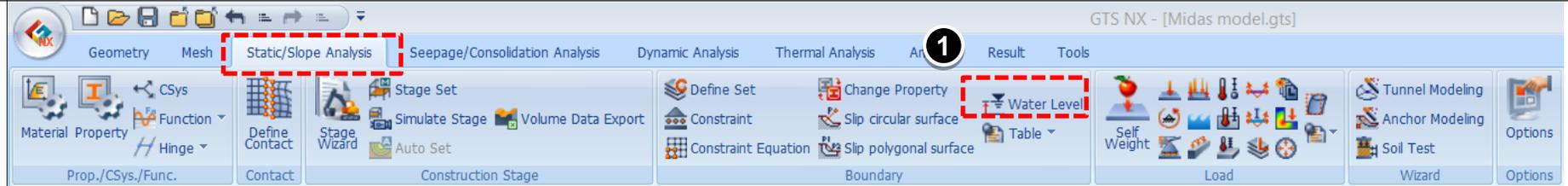
P2 0 kN/m

**6** Load Set LOAD

**7** OK Cancel Apply

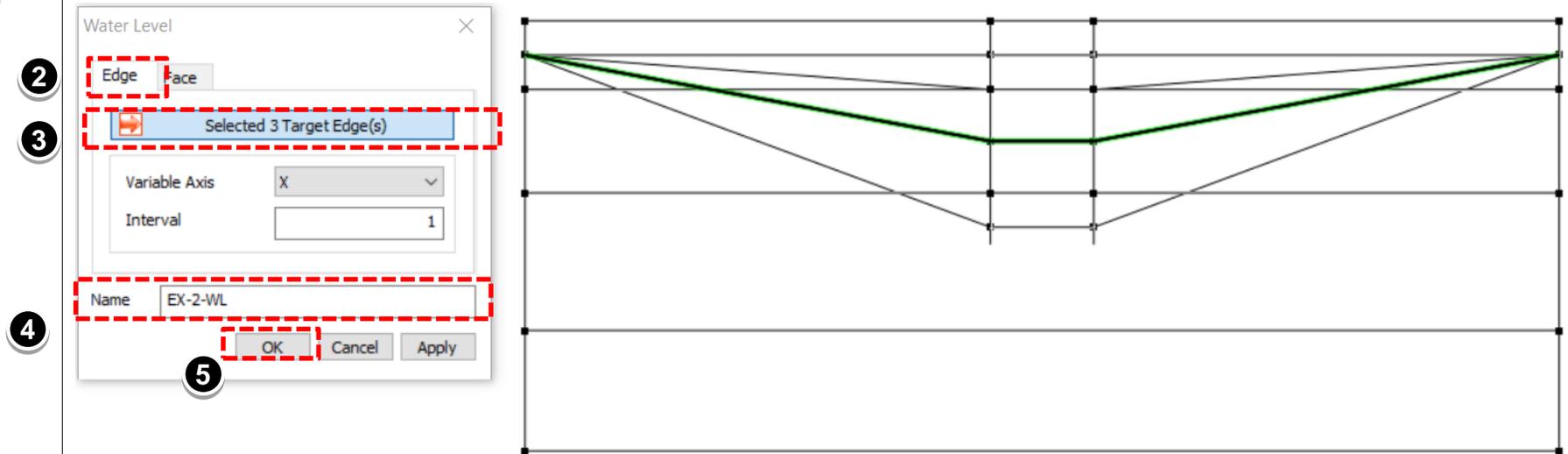
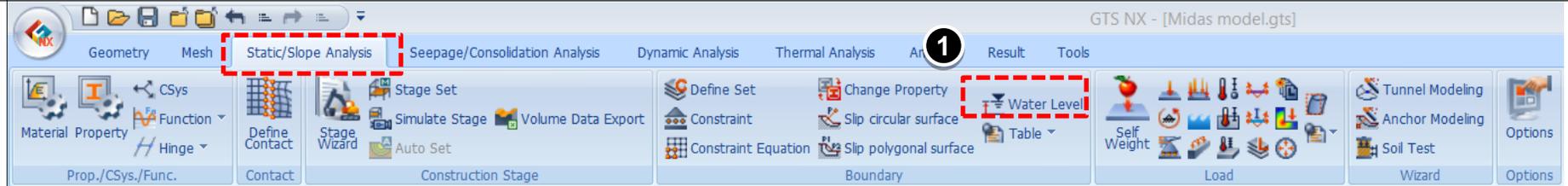
### Procedure

- ❶ Go to Static/Slope Analysis > Boundary > Water level
- ❷ Select **Edge**.
- ❸ Select the edges as shown in the picture.
- ❹ Name it as **EX-1-WL**.
- ❺ Click **OK**



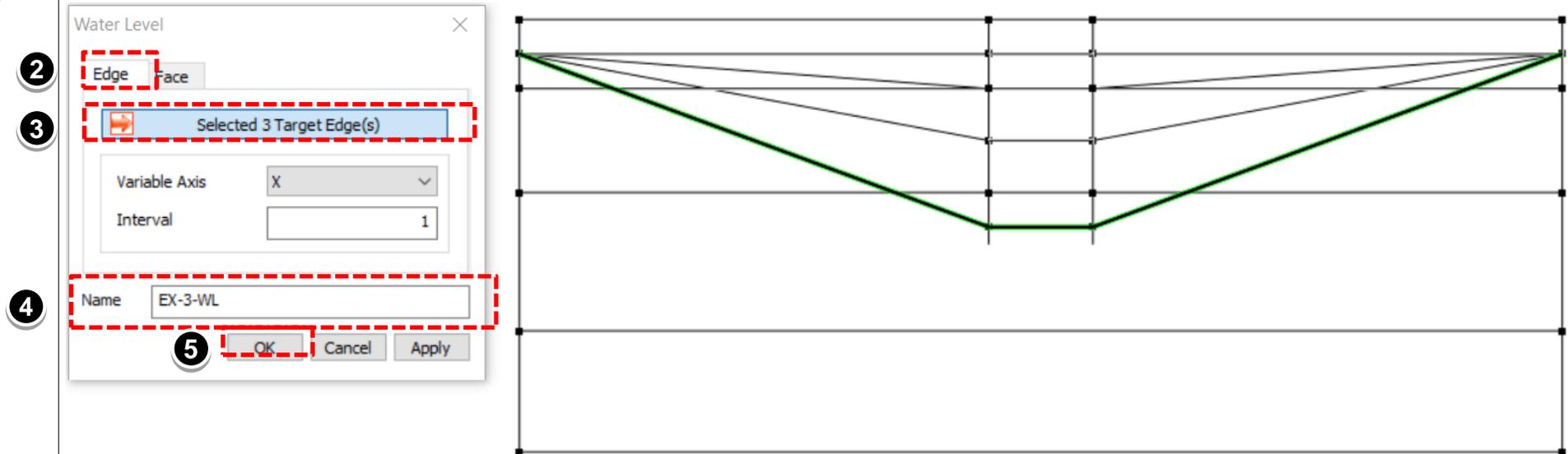
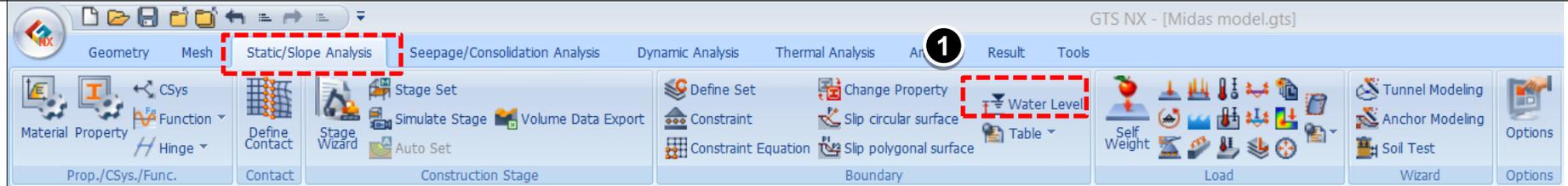
### Procedure

- ① Go to Static/Slope Analysis > Boundary > Water level
- ② Select **Edge**.
- ③ Select the edges as shown in the picture.
- ④ Name it as **EX-2-WL**
- ⑤ Click **OK**



### Procedure

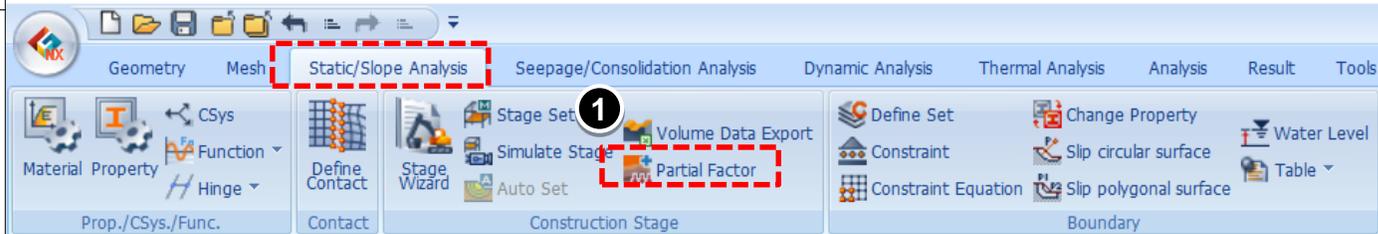
- ❶ Go to Static/Slope Analysis > Boundary > Water level
- ❷ Select **Edge**.
- ❸ Select the edges as shown in the picture.
- ❹ Name it as **EX-3-WL**
- ❺ Click **OK**



## 7-6 Partial Factors Definition

## Procedure

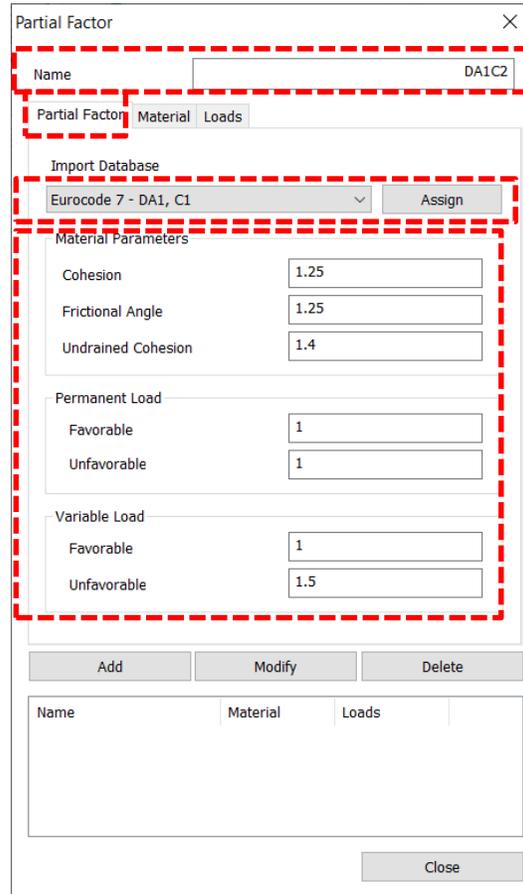
- 1 In the Static/Slope Analysis Tab, Click on **Partial Factor**.
  - 2 Name the case as **DA1C2**.
  - 3 Click on **Partial factor** tab.
  - 4 In the Solution Type Drop-Down Menu, **DA1C2 and press Assign**. The parameters will be automatically assigned.
  - 5 Click on **Materials** Tab.
  - 6 In the materials tab, select all the soil materials and the interface materials.
  - 7 Click on **Load** Tab.
  - 8 Select the **LOAD** and in the Factor section, select **Permanent Unfavorable**.
  - 9 Click **Add**.
  - 10 Click **Close**.
- Follow the same procedure for creating one more partial factor case for DA1C1.



2

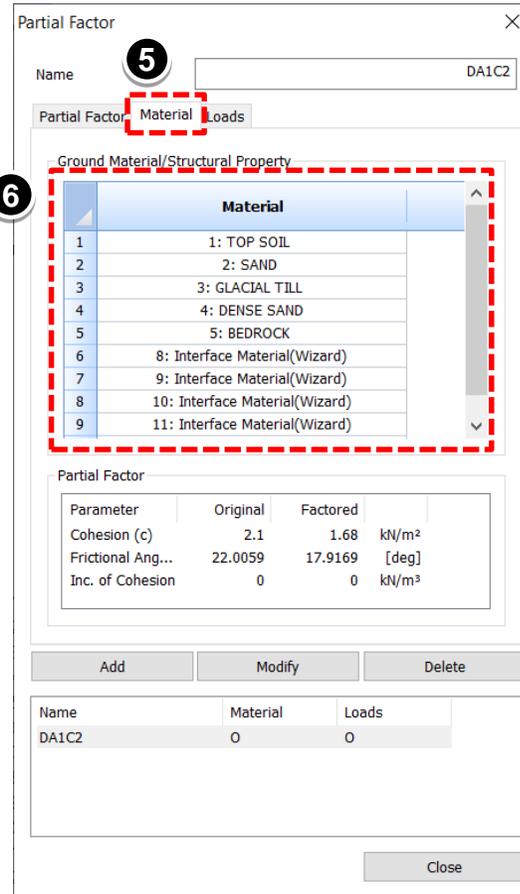
3

4



5

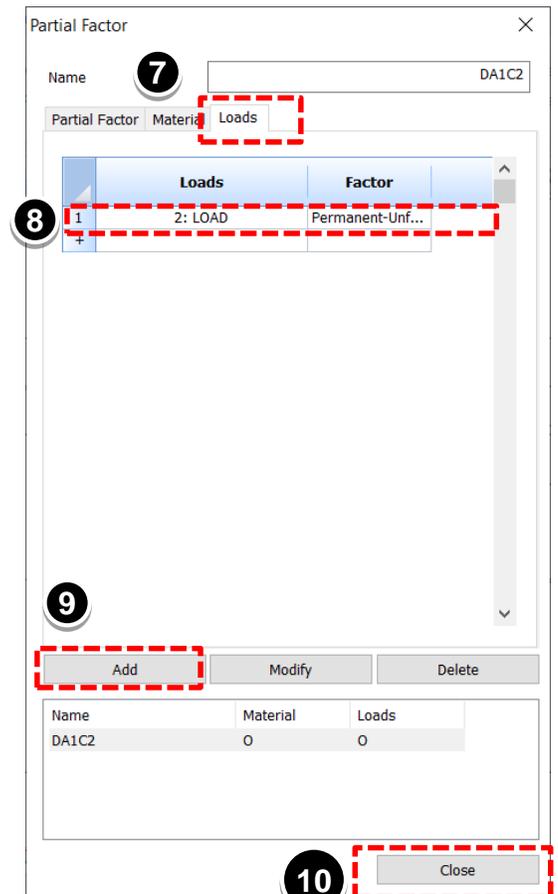
6



7

8

9



10

Partial Factor ×

Name

Partial Factor **Material** Loads

Import Database  
Eurocode 7 - DA1, C1

Material Parameters

Cohesion

Frictional Angle

Undrained Cohesion

Permanent Load

Favorable

Unfavorable

Variable Load

Favorable

Unfavorable

Name	Material	Loads
DA1C1	0	0
DA1C2	0	0

Eurocode 7 - DA1, C1

Eurocode 7 - DA1, C2

Eurocode 7 - DA2

Eurocode 7 - DA3

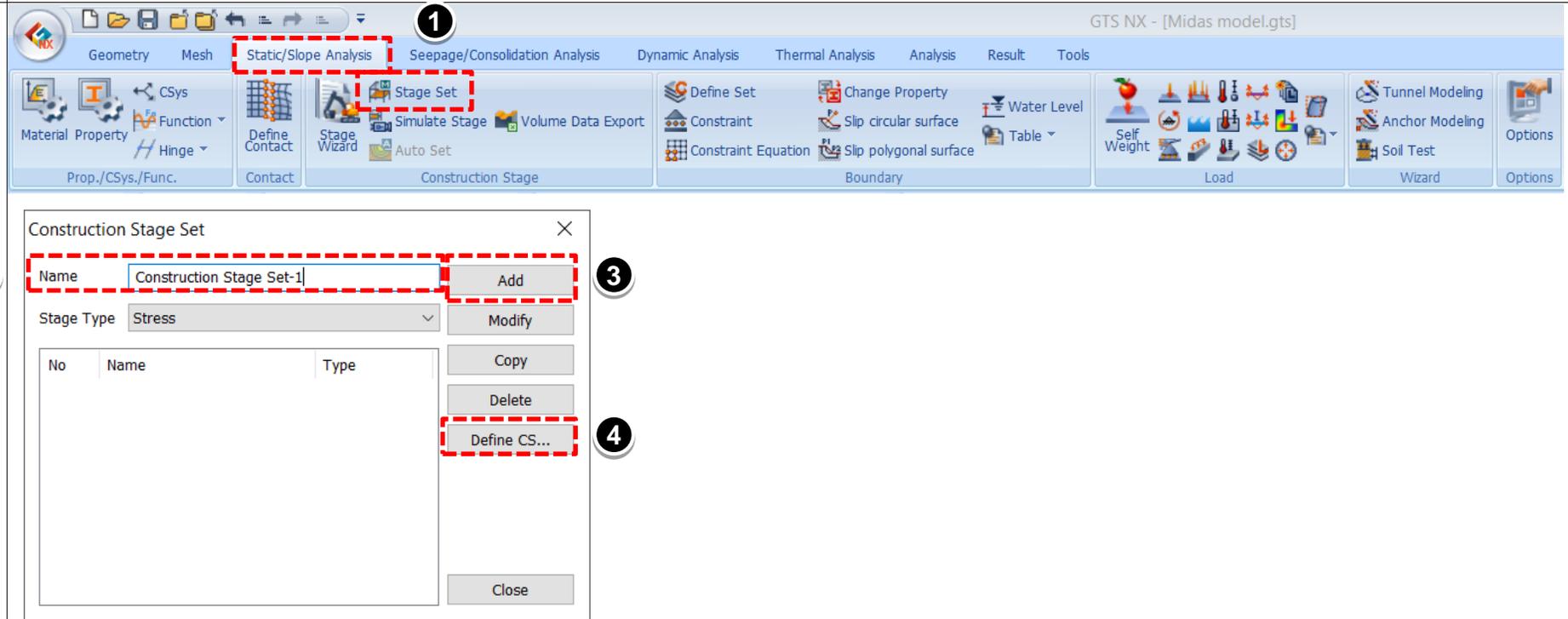
- **DA1, C1:** Partial factor will apply to load only.
- **DA1, C2:** Partial factor will apply to load and soil material.
- **DA2:** DA2 is similar with DA1, C1. But, the factors for pile and footing are different.
- **DA3:** DA3 is similar with DA1, C2. But, the factor for load (Unfavorable under Variable) is different

Values of Partial Factor	Permanent		Variable		Soil		
	Fav.	Unfav.	Fav.	Unfav.	Effective Cohesion (c)	tan $\Phi'$	Undrained Strength (su)
Eurocode 7 - DA1, C1	1.000	1.350	1.000	1.500	1.000	1.000	1.000
Eurocode 7 - DA1, C2	1.000	1.000	1.000	1.300	1.250	1.250	1.400
Eurocode 7 - DA2	1.000	1.350	1.000	1.500	1.000	1.000	1.000
Eurocode 7 - DA3	1.000	1.350	1.000	1.500	1.250	1.250	1.400

# 8-1 Construction Stage Analysis

## Procedure

- 1 Go to Static/Slope Analysis/Stage Set.
- 2 Enter the desired name for the stage set.
- 3 Click **Add**.
- 4 Click on **Define CS Stage Set**.  
The next steps are defined in the subsequent steps.



## 8-2 Construction Stage Analysis

### Procedure

- ① Name the stage as Initial.
- ② Activate the mesh sets, boundary condition and loading condition as shown in the picture.
- ③ Click in **Define Water level of Global** and enter **-4m**.
- ④ Click on **Clear Displacement**.
- ⑤ Click **Save**.
- ⑥ Click **New**.

The screenshot displays the 'Define Construction Stage' dialog box in GTS NX. The 'Static/Slope Analysis' tab is active in the top toolbar. The dialog box contains the following elements:

- Construction Stage Set Name:** Construction Stage Set-1
- Stage ID:** 1: Initial (Step 1)
- Stage Name:** Initial (Step 2)
- Stage Type:** Stress (Step 6)
- Set Data:** A tree view showing various mesh and boundary condition sets.
- Activated Data:** A list of sets including BEDROCK, DENSE SAND, DENSE SAND-1, EX-1 to EX-4, Rigid Link Mesh, SAND, SAND-1, TILL, TILL-1, TOP SOIL, TOP SOIL -1, Boundary Condition, BC, Static Load, and SW (Step 2).
- Deactivated Data:** A list of sets including Mesh, Boundary Condition, Static Load, and Contact.
- Initial Condition:**
  - Define Water Level For Global (Step 3): -4 m
  - Define Water Level For Mesh Set
- Other Options:**
  - Analysis Control...
  - Output Control...
  - Sub Stage...
  - LDF...
  - Clear Displacement (Step 4)
  - Slope Stability(SRM)
  - Slope Stability(SAM)
- Buttons:** Move to Previous, Move to Next, New, Insert, Delete, Save (Step 5), and Close.

# 8-3 Construction Stage Analysis

## Procedure

- ① Name the stage as WALL.
- ② Activate and deactivate the mesh sets, as shown in the picture.
- ③ Click **Save**.
- ④ Click **New**.

The screenshot displays the 'Define Construction Stage' dialog box in the GTS NX software. The dialog is titled 'Define Construction Stage' and has a close button (X) in the top right corner. It contains several sections:

- Construction Stage Set Name:** Construction Stage Set-1
- Stage ID:** 2: WALL
- Stage Name:** WALL (highlighted with a red dashed box and a circled '1')
- Stage Type:** Stress
- Buttons:** Move to Previous, Move to Next, New (highlighted with a red dashed box and a circled '4'), Insert, Delete
- Set Data:** A tree view of model elements, including Mesh (BASE SLAB, BEDROCK, DENSE SAND, DENSE SAND-1, Default Mesh Set, EX-1, EX-2, EX-3, EX-4, Line Interface, Rigid Link Mesh, SAND, SAND-1, STRUT-1, STRUT-2, STRUT-3, TILL, TILL-1, TOP SOIL, TOP SOIL -1, WALL), Boundary Condition (BC), Static Load (LOAD, SW), and Contact.
- Activated Data:** A list of elements to be active in this stage, including Mesh (Line Interface, WALL), Boundary Condition (Boundary Condition), Static Load (Static Load), and Contact (LOAD, Contact). This list is highlighted with a red dashed box and a circled '2'.
- Deactivated Data:** A list of elements to be inactive in this stage, including Mesh (Rigid Link Mesh), Boundary Condition (Boundary Condition), Static Load (Static Load), and Contact (Contact).
- Initial Condition:** Define Water Level For Global (0 m, None), Define Water Level For Mesh Set (Input Water Level...), Sub Stage..., LDF..., Clear Displacement, Slope Stability(SRM), Slope Stability(SAM).
- Buttons:** Save (highlighted with a red dashed box and a circled '3'), Close

The top toolbar shows the 'Static/Slope Analysis' tab selected, with 'Stage Set' and 'Simulate Stage' buttons highlighted. The bottom right corner has 'Save' and 'Close' buttons.

# 8-4 Construction Stage Analysis

## Procedure

- ① Name the stage as **EX-1**.
- ② Deactivate the mesh sets, as shown in the picture.
- ③ Click in **Define Water level of Global** and enter **1m** and select **Ex-1-WL**.
- ④ Click on **Sub Stage**.
- ⑤ Type **11** and select **DA1C2**.
- ⑥ Click **OK**.
- ⑦ Click **SAVE**.

GTS NX - [Midas model.gts]

Geometry Mesh **Static/Slope Analysis** Seepage/Consolidation Analysis Dynamic Analysis Thermal Analysis Analysis Result Tools

Material Property CSys Function Hinge Define Contact Stage Wizard Simulate Stage Volume Data Export Prop./CSys./Func. Contact Construction Stage Define Set Change Property Water Level Self Weight Tunnel Modeling Anchor Modeling Soil Test Options

Constraint Constraint Equation Slip circular surface Slip polygonal surface Table

Define Construction Stage

Construction Stage Set Name Construction Stage Set-1

Stage 1: 3: EX-1 Move to Previous Move to Next

Stage Name EX-1 New Insert Delete

Stage Type Stress

Set Data Activated Data Deactivated Data

Mesh BASE SLAB BEDROCK DENSE SAND DENSE SAND-1 Default Mesh Set EX-1 EX-2 EX-3 EX-4 Line Interface Rigid Link Mesh SAND SAND-1 STRUT-1 STRUT-2 STRUT-3 TILL TILL-1 TOP SOIL TOP SOIL-1 WALL Boundary Condition BC Static Load LOAD SW Contact

Mesh Boundary Condition Static Load Contact

Mesh EX-1 Boundary Condition Static Load Contact

Initial Condition

Define Water Level For Global

1 m EX-1-WL

Define Water Level For Mesh Set

Input Water Level...

Sub Stage...

LDF...

Clear Displacement

Slope Stability(SRM)

Slope Stability(SAM)

Sub Stage

Name	Partial Factor
1	11 DA1C2

OK Cancel

Sort By Name Show Data Activate

Save Close

# 8-5 Construction Stage Analysis

## Procedure

- ① Name the stage as **EX-2**.
- ② Deactivate the mesh sets, as shown in the picture.
- ③ Click on **Sub Stage**.
- ④ Type **12** and select **DA1C2**.
- ⑤ Click **OK**.
- ⑥ Click **SAVE**.

The screenshot shows the 'Define Construction Stage' dialog box in GTS NX. The 'Construction Stage Set Name' is 'Construction Stage Set-1'. The 'Stage ID' is 'EX-2' and the 'Stage Name' is 'EX-2'. The 'Stage Type' is 'Stress'. The 'Set Data' column lists various mesh and boundary condition sets. The 'Activated Data' column shows 'STRUT-1' and 'Boundary Condition' sets. The 'Deactivated Data' column shows 'EX-2' and 'Boundary Condition' sets. The 'Sub Stage...' button is checked. The 'Save' button is highlighted.

The 'Sub Stage' dialog box is also shown, with a table containing the following data:

Name	Partial Factor
12	DA1C2

# 8-6 Construction Stage Analysis

## Procedure

- ① Name the stage as **EX-3**.
- ② Deactivate the mesh sets, as shown in the picture.
- ③ Click in **Define Water level of Global** and enter **1m** and select **Ex-2-WL**.
- ④ Click on **Sub Stage**.
- ⑤ Type **13** and select **DA1C2**.
- ⑥ Click **OK**.
- ⑦ Click **SAVE**.

Define Construction Stage

Construction Stage Set Name: Construction Stage Set-1

Stage ID: ① S: EX-3

Stage Name: ② EX-3

Stage Type: Stress

Set Data

Set Data	Activated Data	Deactivated Data
Mesh	Mesh	Mesh
BASE SLAB	STRUT-2	EX-3
BEDROCK	Boundary Condition	Boundary Condition
DENSE SAND	Static Load	Static Load
DENSE SAND-1	Contact	Contact
Default Mesh Set		
EX-1		
EX-2		
EX-3		
EX-4		
Line Interface		
Rigid Link Mesh		
SAND		
SAND-1		
STRUT-1		
STRUT-2		
STRUT-3		
TILL		
TILL-1		
TOP SOIL		
TOP SOIL -1		
WALL		
Boundary Condition		
BC		
Static Load		
LOAD		
SW		
Contact		

Initial Condition

Define Water Level For Global

1 m EX-2-WL

Define Water Level For Mesh Set

Input Water Level...

Sub Stage...

LDF...

Clear Displacement

Slope Stability(SRM)

Slope Stability(SAM)

Sub Stage

Name	Partial Factor
1	13 DA1C2
+	

OK Cancel

Sort By: Name Show Data: Activate

Save Close

# 8-7 Construction Stage Analysis

## Procedure

- ① Name the stage as **EX-4**.
- ② Deactivate and activate the mesh sets, as shown in the picture.
- ③ Click in **Define Water level of Global** and enter **1m** and select **Ex-3-WL**.
- ④ Click on **Sub Stage**.
- ⑤ Type **14** and select **DA1C2**.
- ⑥ Click **OK**.
- ⑦ Click **SAVE**.

**REPEAT THE SAME STEPS FROM 8-1 EXCEPT ACTIVATION OF ANY SUBSTAGE AND CREATE ONE MORE CONSTRUCTION STAGE SETS. THESE STAGES WILL BE USED FOR COMPARISON STUDY.**

**OR THE STEPS DESCRIBED IN SLIDE 8-8 CAN ALSO BE FOLLOWED.**

The screenshot displays the GTS NX software interface for defining a construction stage. The main window shows the 'Define Construction Stage' dialog box with the following details:

- Construction Stage Set Name:** Construction Stage Set-1
- Stage ID:** 6: EX-4
- Stage Name:** EX-4
- Stage Type:** Stress
- Set Data:**
  - Mesh:** BASE SLAB, BEDROCK, DENSE SAND, DENSE SAND-1, Default Mesh Set, EX-1, EX-2, EX-3, EX-4, Line Interface, Rigid Link Mesh, SAND, SAND-1, STRUT-1, STRUT-2, STRUT-3, TILL, TILL-1, TOP SOIL, TOP SOIL -1, WALL
  - Boundary Condition:** BC
  - Static Load:** LOAD, SW
  - Contact:** Contact
- Analysis Control:**  Analysis Control...
- Output Control:**  Output Control...
- Water Level:**
  - Define Water Level For Global (1 m, EX-4)
  - Define Water Level For Mesh Set
- Sub Stage:**  Sub Stage...
- Other Options:**  LDF...,  Clear Displacement,  Slope Stability(SRM),  Slope Stability(SAM)

The 'Sub Stage' dialog box shows a table with the following data:

Name	Partial Factor
14	DA1C2

At the bottom of the 'Define Construction Stage' dialog, the 'Save' button is highlighted, and the 'Sub Stage' dialog has an 'OK' button highlighted.

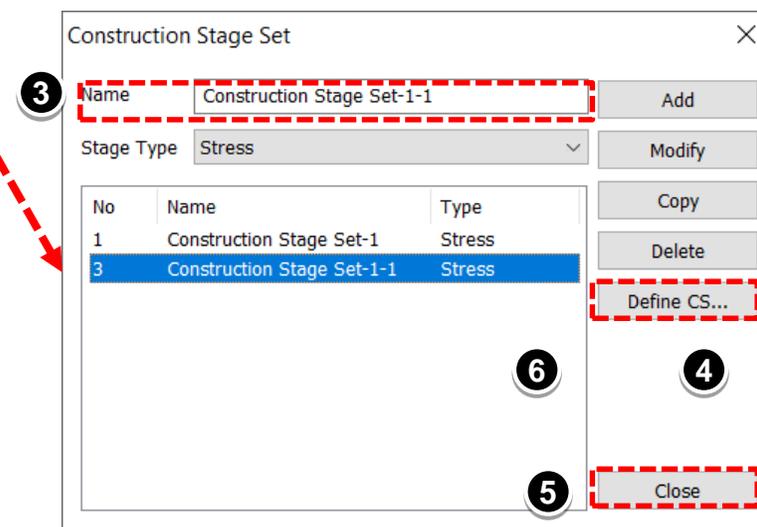
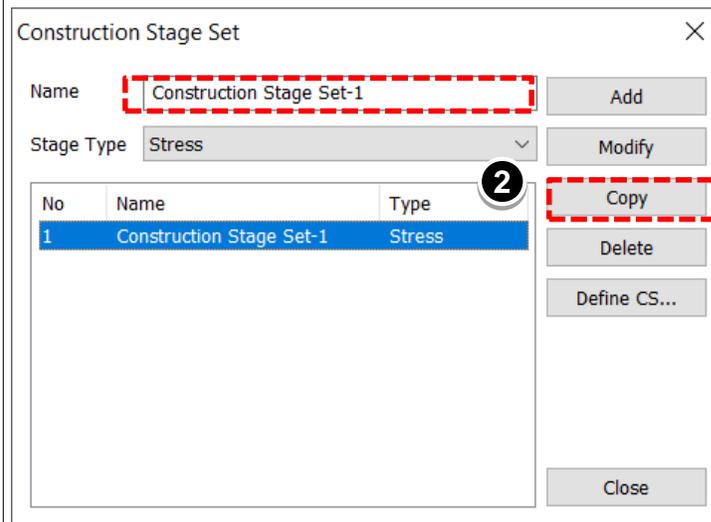
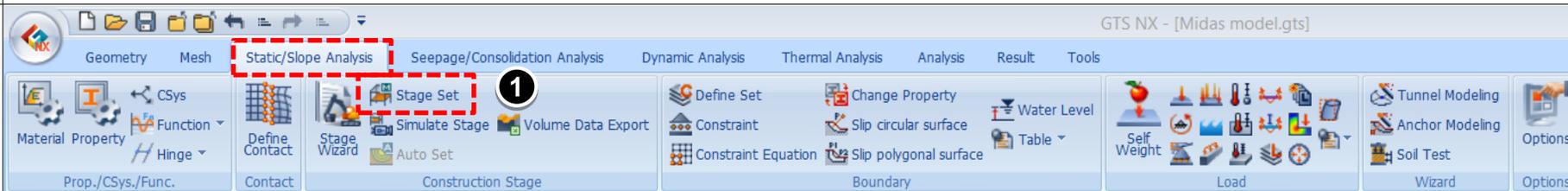
# 8-8 Construction Stage Analysis HINTS

## Procedure

- ❶ Go to Static/Slope Analysis > Stage Set.
- ❷ Click in the previously created construction stage set.
- ❸ Click **Copy**.
- ❹ The new stage will appear like as shown in the image.
- ❺ Click **Define CS**.

**THIS NEW CONSTRUCTION STAGE SET WILL HAVE THE SAME STAGES AS THE BUILT BEFORE, SO THE SUBSTAGES WILL HAVE TO BE MANUALLY DEACTIVATED FOR THIS STAGE SET.**

**MANUALLY DEACTIVATE THE SUBSTAGES IN STAGES EX—1-1, EX-2-1, EX-3-1, EX-4-1.**



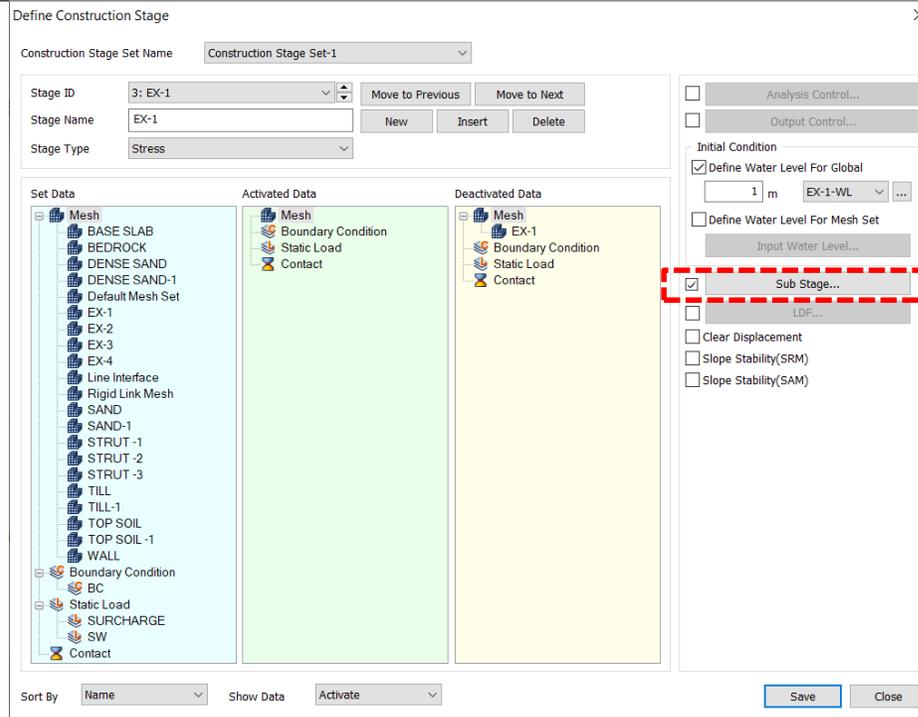
❷

# 8-9 Construction Stage Analysis HINTS

## Procedure

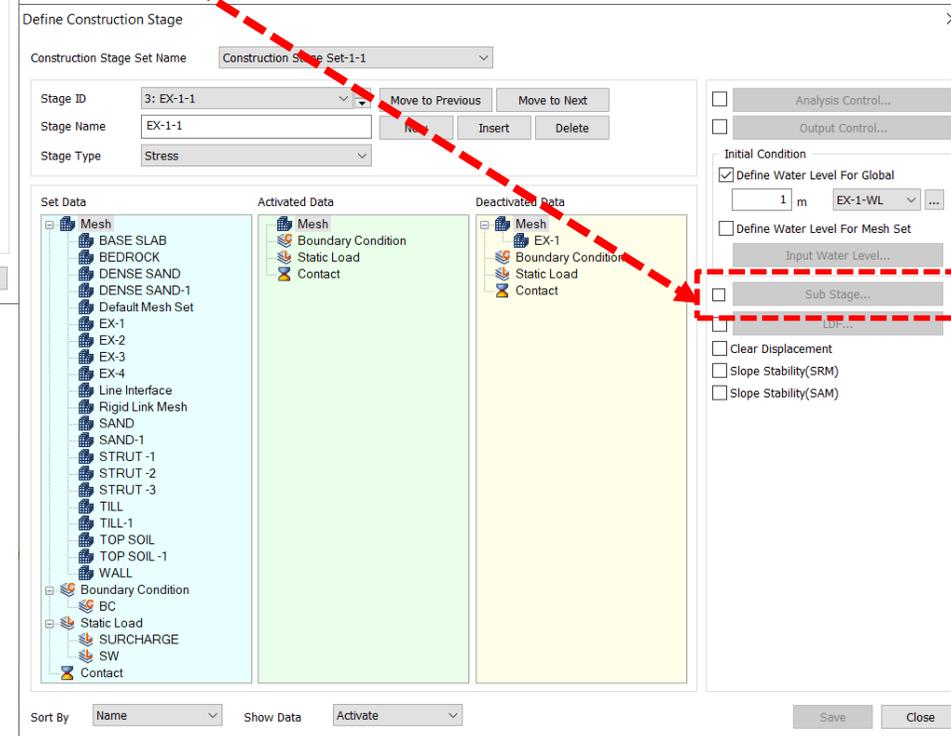
THE TWO PICTURES IN THIS SLIDE ILLUSTRATE THE DIFFERENCE BETWEEN THE CONSTRUCTION STAGES OF THE TWO SETS.

ENSURE THAT ALL THE SUBSTAGES ARE DEACTIVATED FOR THE NEW STAGE SET.



CONSTRUCTION STAGE FOR NORMAL + DA1C2 ANALYSIS CASE

CONSTRUCTION STAGE FOR DA1C1 ANALYSIS CASE



## Procedure

- 1 In the Analysis Tab, Click on General
- 2 Name the case as **NORMAL+DA1C2**.
- 3 In the Solution Type Drop-Down Menu, **Construction Stage**.
- 4 Click on **Analysis Control**.
- 5 Click in **Initial Stage**, and select **both** the options.
- 6 Click **OK**.
- 7 Click **OK**.

The screenshot displays the software's analysis case definition workflow. The top ribbon shows the 'Analysis' tab selected, with the 'General' sub-tab active. The 'Add/Modify Analysis Case' dialog box is open, showing the following settings:

- Title:** NORMAL + DA1C2
- Description:** (empty)
- Solution Type:** Construction Stage
- Construction Stage Set:** Construction Stage Set-1
- Analysis Case Model:** Two panels, 'All Sets' (light blue) and 'Active Sets' (light green), are visible.
- Buttons:** 'OK', 'Cancel', and 'Apply' are at the bottom.

The 'Analysis Control' dialog box is also open, with the 'General' tab selected. The following options are checked:

- Water Pressure:**  Automatically Consider Water Pressure
- Initial Stage:**
  - Initial Stage for Stress Analysis
  - Apply K0 Condition
  - Cut-Off Negative Effective Pressure
- Initial Stress:**  Estimate Initial Stress of Activated Elements
- Final Calculation Stage:**  End Stage,  Middle Stage
- Restart Option:**  Save only User Specified Stages,  Save All Stages
- Max. Negative Pore Pressure:**  Max. Negative Pore Pressure Limit

The 'OK' button in the 'Analysis Control' dialog is highlighted.

## Procedure

- 1 In the Analysis Tab, Click on General
- 2 Name the case as DA1C1.
- 3 In the Solution Type Drop-Down Menu, **Construction Stage**.
- 4 Click on **Analysis Control**.
- 5 Click in **Initial Stage**, and select both the options.
- 6 In the Nonlinear tab, click on **Consider Partial Factor** and select DA1C1.
- 7 Click **OK**.
- 8 Click **OK**.
- 9 Click **OK**.

The screenshot illustrates the software interface for defining an analysis case. The top menu bar shows the 'Analysis' tab selected. The 'Add/Modify Analysis Case' dialog box is open, showing the 'Analysis Case Setting' section with the following fields:

- Title: DA1C1
- Description: (empty)
- Solution Type: Construction Stage
- Construction Stage Set: Construction Stage Set-2

The 'Analysis Control' dialog box is also open, showing the 'General' tab with the following settings:

- Water Pressure:  Automatically Consider Water Pressure
- Initial Stage:  Initial Stage for Stress Analysis (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 (1:Initial)
- Restart Option:  Save only User Specified Stages,  Save All Stages
- Initial Temperature:  Initial Temperature By Value (0 [T]),  Initial Temperature By Load Set (None)
- Saturation Effects:  Consider Partially Saturated Effects for Stress Analysis
- Max. Negative Pore Pressure:  Max. Negative Pore Pressure Limit (0 kN/mm<sup>2</sup>)
- Initial Configuration:  Estimate Initial Configuration of Activated Nodes

The 'Analysis Control (Nonlinear)' dialog box is open, showing the 'Nonlinear' tab with the following settings:

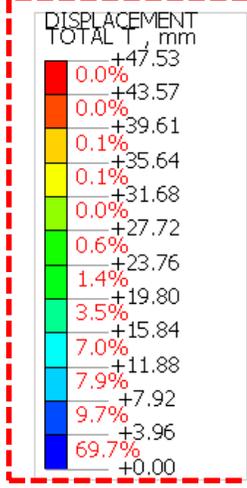
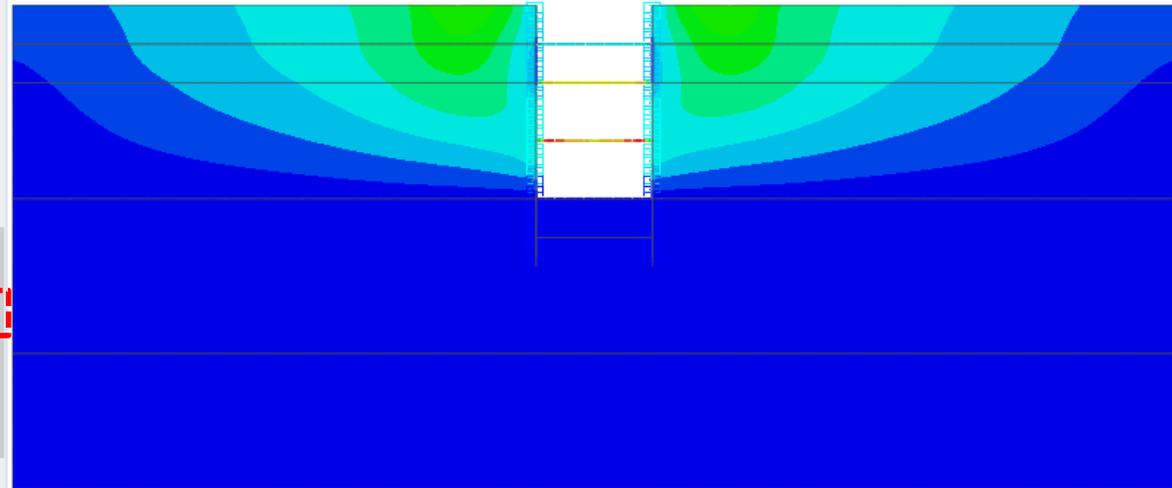
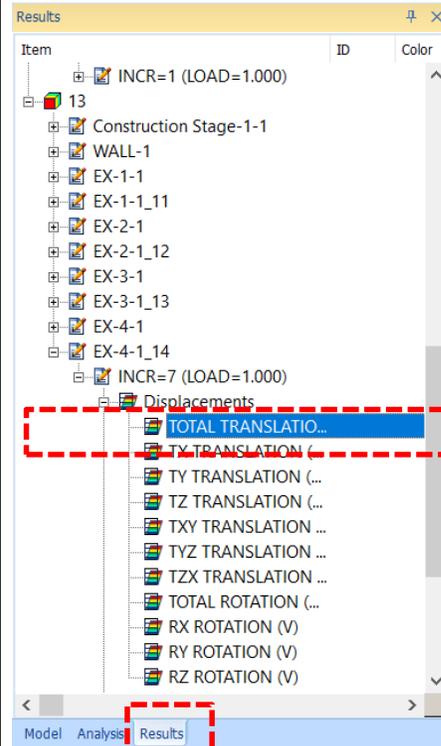
- Geometry Nonlinearity:  Consider Geometric Nonlinear Effects,  Update Pore Pressure with Deformation
- Partial Factor:  Consider Partial Factor (DA1C1)
- Basic Nonlinear Parameters:  Load Steps,  Time Steps
- Iterative Scheme:  General,  Enhanced Init Stress
- Convergence Criteria / Error Tolerance:  Displacement(U) (0.001),  Load(P) (0.001),  Work(W) (1e-006)

The 'Add/Modify Analysis Case' dialog box has an 'OK' button highlighted with a red dashed box and a numbered circle 9. The 'Analysis Control' dialog box has an 'OK' button highlighted with a red dashed box and a numbered circle 8. The 'Analysis Control (Nonlinear)' dialog box has an 'OK' button highlighted with a red dashed box and a numbered circle 7.

# 10-1 Result (DISPLACEMENTS)

## Procedure

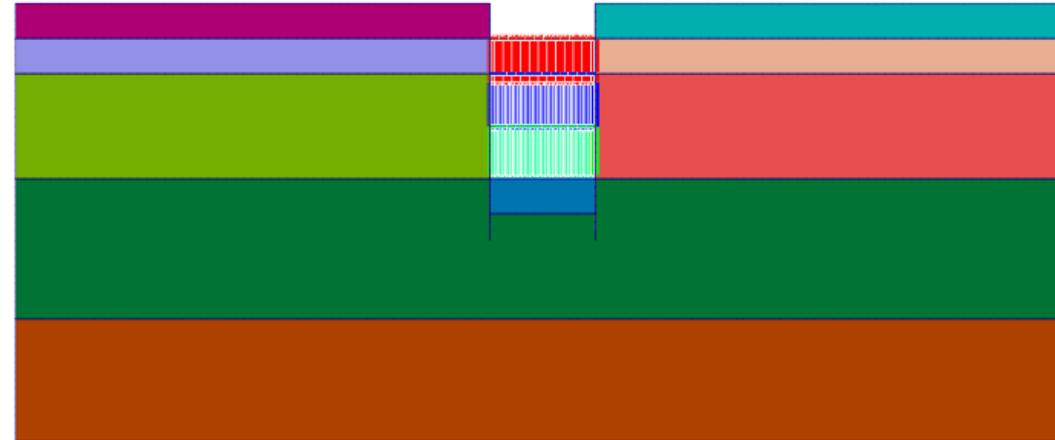
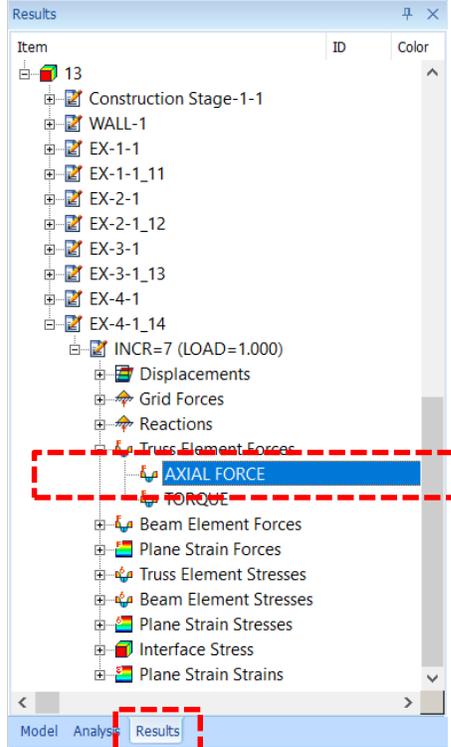
- 1 Go to Results>
- 2 Click on Total Displacements.
- 3 Legend shows the displacement around the model.



# 10-2 Result (AXIAL FORCES ON STRUTS)

## Procedure

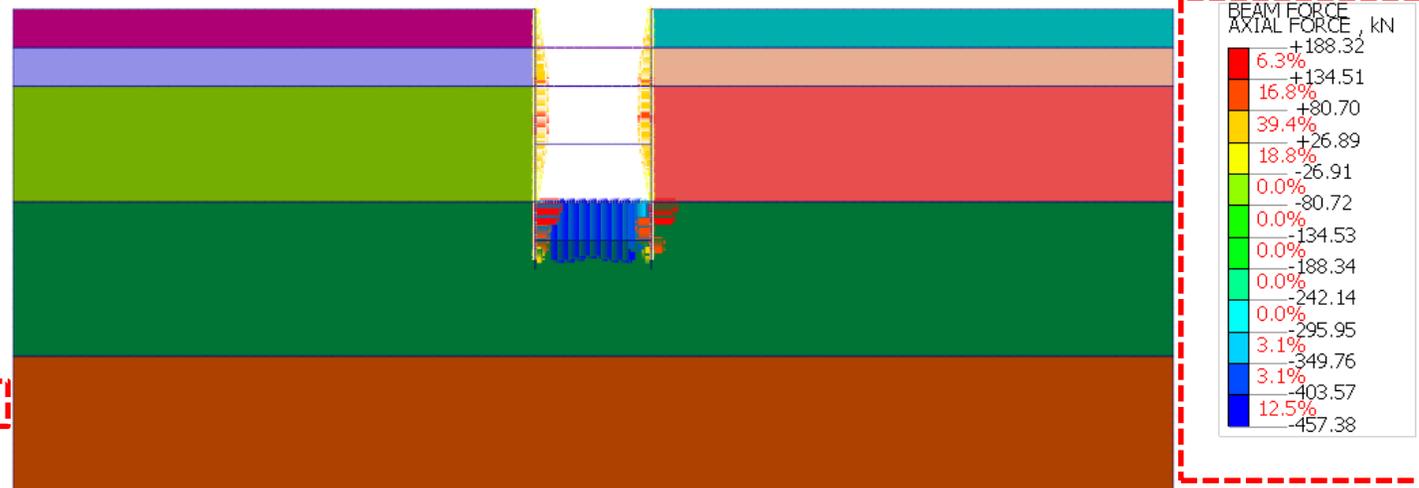
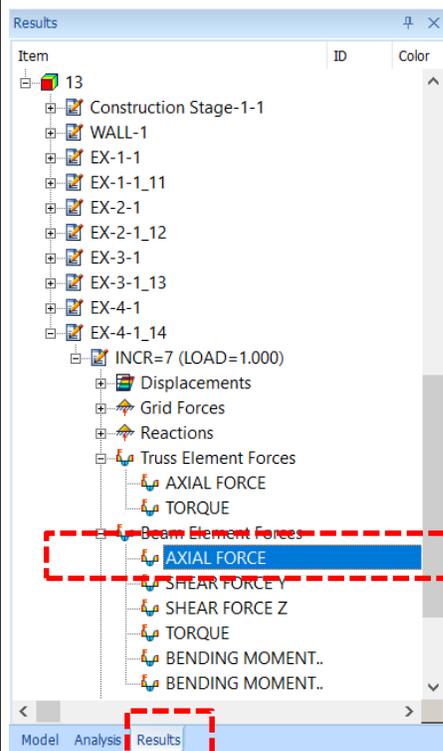
- 1 Go to Results> EX-4-1
- 2 Click on TRUSS ELEMENT FORCES.
- 3 Legend shows the forces acting on the struts.



# 10-3 Result (AXIAL FORCES ON WALL & BASE SLAB)

## Procedure

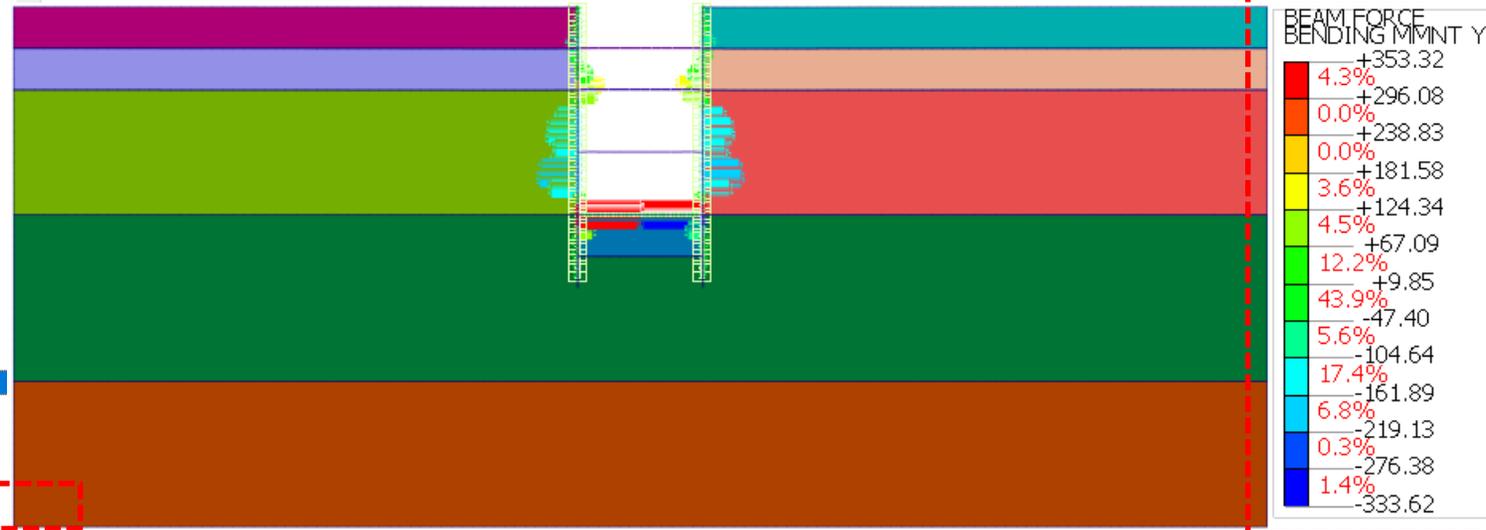
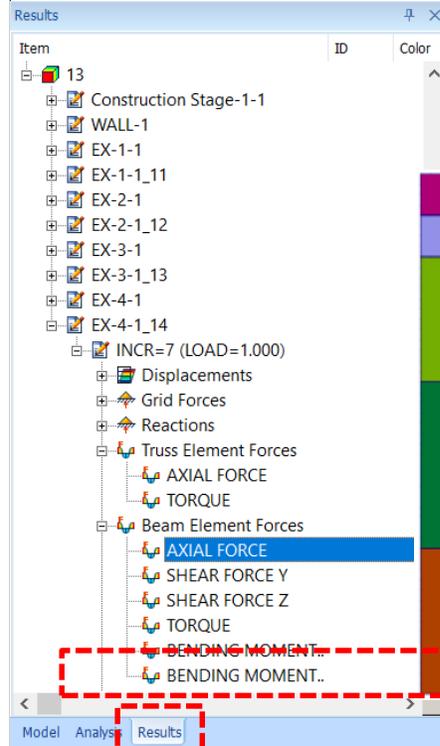
- 1 Go to Results> EX-4-1
- 2 Click on BEAM ELEMENT FORCES.
- 3 Legend shows the forces acting on the wall & base slab.



# 10-3 Result (BENDING MOMENTS ON WALL)

## Procedure

- 1 Go to Results> EX-4-1
- 2 Click on BEAM ELEMENT FORCES & select **Bending Moment Y**.
- 3 Legend shows the bending moments acting on the wall & base slab.



1

2

3

- GTS NX Certification task for participants involves submission of file (.docx or .pdf) with :
  - Generating the model as shown in tutorial.
    - Images/3D PDF report in the output.
  - Short summary of model creation, and results.

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

# CERTIFICATION TASK GUIDELINES

## Online Course Assignment

You can download the model files and a report template for your assignment mission. Please click the download button below.

In addition, please be noted that the assignment submission policy has been changed. Click the 'Go To Submit' button to hand in your assignment. Submission through email is not accepted.

Upon completion of your submission, you will receive a confirmation of completion and the uploaded assignment will be automatically counted.

- **Deadline**
- **Submission:** Click **[Go To Submit]**
- **Format:** Only the provided report in docx or pdf file format.

Download Assignment

Go to Submit

MIDAS GEOTECH | ONLINE COURSE 2021

### ONLINE COURSE ASSIGNMENT

First name\*  
Hyebin

Last name\*  
Cha

Organization\*  
TEST

Country\*  
Albania

Email\*  
onb0218@midasit.com

Select the title of the online course\*  
[Session 3] Deep Excavation Analysis according to Eurocode 7

Files\*  
Please attach your assignment report here. pdf or docx format only. (see details below)  
[파일선택] 선택된 파일 없음

I agree to submit the file and personal information to MIDAS ENG SOLUTION.\*

By clicking submit below, you consent to allow MIDAS ENG SOLUTION to store and process the personal information submitted above to provide you the content requested.

**MODEL DATA & TASK FILES INSTALLATION DATE – 14<sup>TH</sup> APRIL**

**SUBMISSION ONLY VIA LINK, MAIL SUBMISSIONS NOT ACCEPTED**

**SUBMISSION DATE : 23<sup>RD</sup> APRIL**