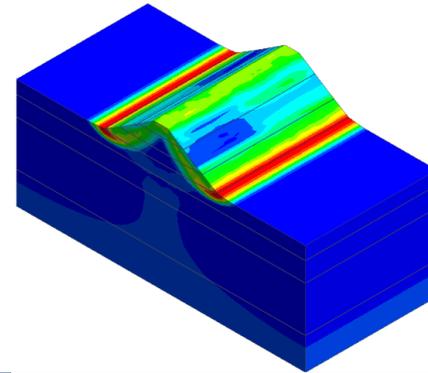


MIDAS *Technical
Material*

Tutorial



2D Geogrid Reinforced Slope Stability Tutorial

Contents

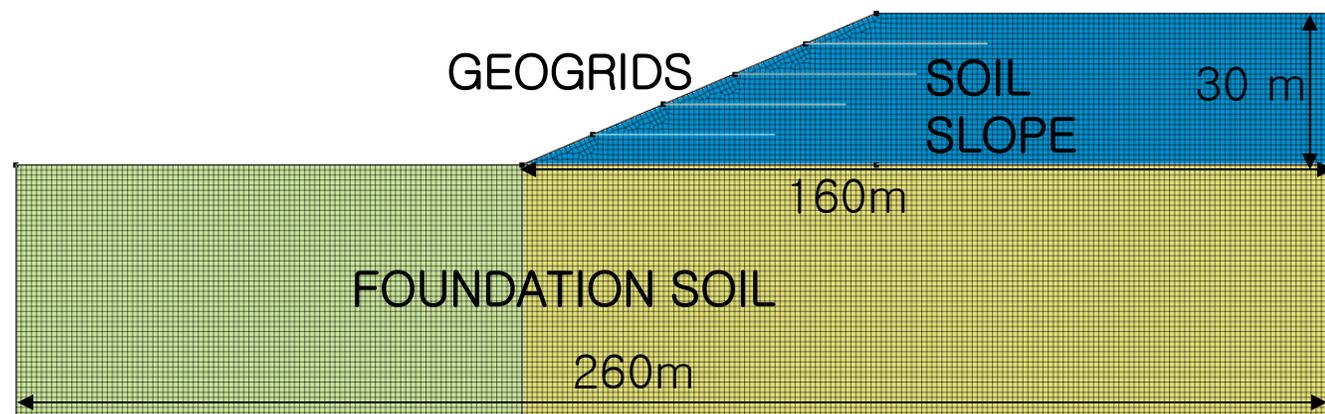
- Step 1: Initial Setting
- Step 2: Defining Soil material
- Step 3: Defining Property
- Step 4: Geometric Modeling
- Step 5: Meshing
- Step 6: Boundary Condition
- Step 7: Loads Definition
- Step 8: Analysis Case
- Step 9: Analysis
- Step 10: Results
- Step 11: Certification Task

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 Geogrid

Material Properties

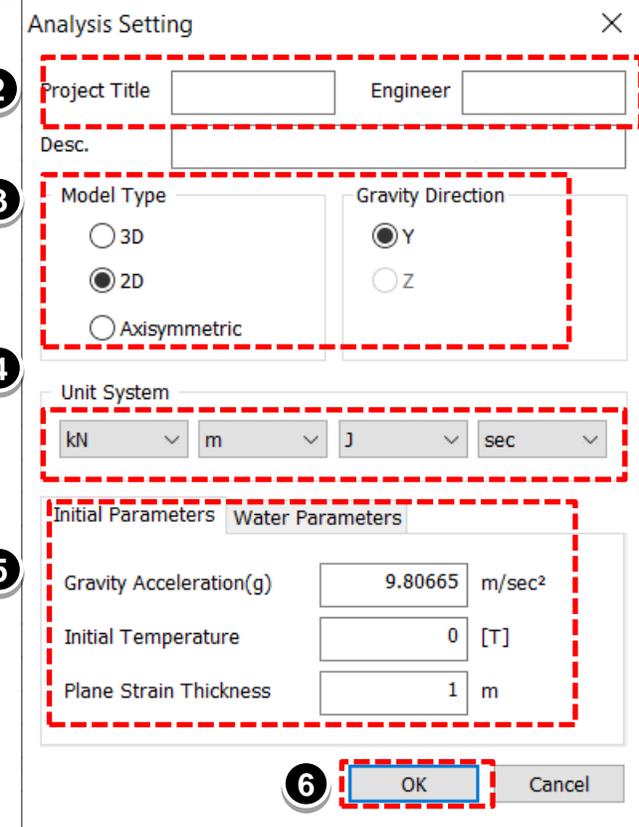
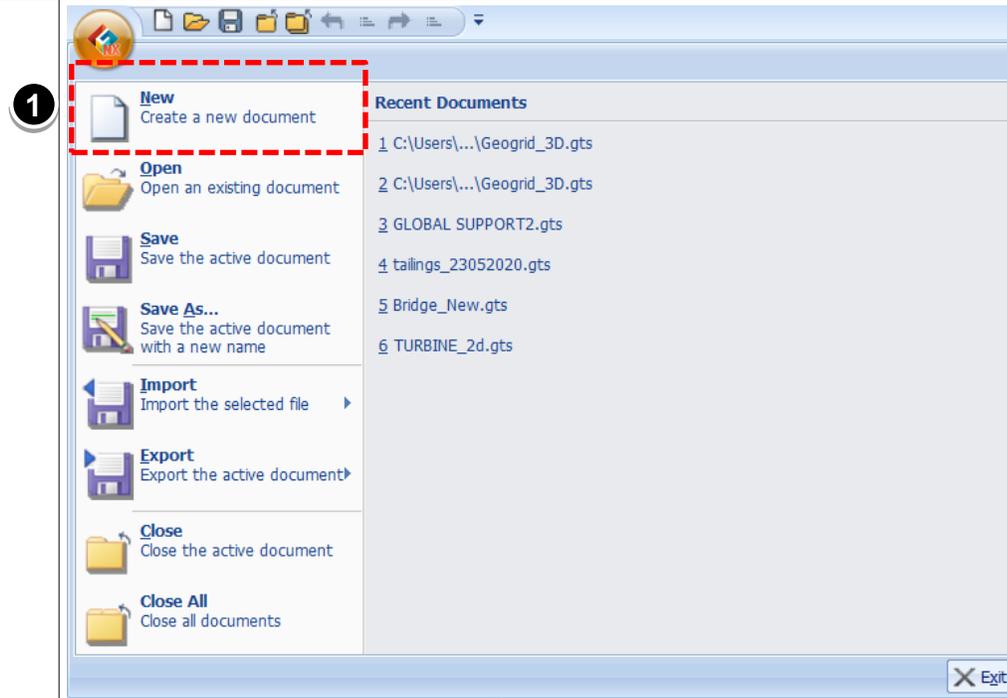
Name	Material Model	Modulus of Elasticity (kN/m ²)	Poisson's ratio	Unit Weight (kN/m ³)	Saturated unit Weight (kN/m ³)	Cohesion (kN/m ²)	Friction angle
Soil	Mohr-Coulomb	35000	0.3	18	19	1	26
Foundation	Mohr-Coulomb	48000	0.15	19	20	5	32

Name	Material Model	Modulus of Elasticity (kN/m ²)	Modulus of Elasticity (kN/m ²)	Shear Modulus (kN/m ²)	Tensile Strength 1 (kN/m ²)	Tensile Strength 2 (kN/m ²)	Unit Weight (kN/m ³)
Geogrid	Orthotropic (Geogrid)	120000	120000	100000	85000	85000	21.5

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

- ➊ Go to Mesh > Material
- ➋ Click on Create. Select **Isotropic**
- ➌ Select Model type as **Mohr-Coulomb**
- ➍ In General tab, Enter the value of **Elastic modulus (E) =35000 kN/m²**
- ➎ Enter **Poisson's ratio= 0.3, Unit weight = 18 kN/m³**.

The screenshot illustrates the software interface for defining a soil material. The process is guided by five numbered steps:

- ➊ **Mesh > Material**: The user navigates to the Material property tool in the Mesh tab of the software's ribbon.
- ➋ **Create... dropdown menu**: The user selects the **Isotropic** material model from the dropdown menu.
- ➌ **Material dialog box**: The user enters the material name as **SOIL** and selects the **Mohr-Coulomb** model type.
- ➍ **General tab**: The user defines the material properties:

Property	Value	Unit
Elastic Modulus(E)	35000	kN/m ²
Inc. of Elastic Modulus	0	kN/m ³
Inc. of Elastic Modulus Ref. Height	0	m
Poisson's Ratio(v)	0.3	
Unit Weight(γ)	18	kN/m ³

Procedure

- ① Go to Porous Tab > Enter Unit weight (saturated) =19 kN/m³
- ② Keep Drainage Parameters as Drained
- ③ Go to Non-Linear Tab > Enter the value of Cohesion = 1 kN/m²
- ④ Input the value of Frictional angle =26°
- ⑤ Enter the name as SOIL.
- ⑥ Click OK

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

Material

ID 1 Name SOIL Color

Model Type Mohr-Coulomb Structure

① General Porous Non-Linear Thermal Time Dependent

Unit Weight(Saturated) kN/m³

Initial Void Ratio(eo)

② Unsaturated Property ...

Drainage Parameters

Drained

Undrained Poisson's Ratio

Skempton's B Coefficient

Seepage & Consolidation Parameters

Permeability Coefficients

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

Void Ratio Dependency of Permeability(ck)

Specific Storativity(Ss) 1/m Auto

OK Cancel Apply

Material

ID 1 Name SOIL Color

Model Type Mohr-Coulomb Structure

General Porous Non-Linear Thermal Time Dependent

③ Cohesion(C) kN/m²

Inc. of Cohesion kN/m³

Inc. of Cohesion Ref. Height m

④ Frictional Angle(Φ) [deg]

Dilatancy Angle [deg]

Tension Cut-off

Tensile Strength kN/m²

Cut-off Yield Surface

Pressure Rankine

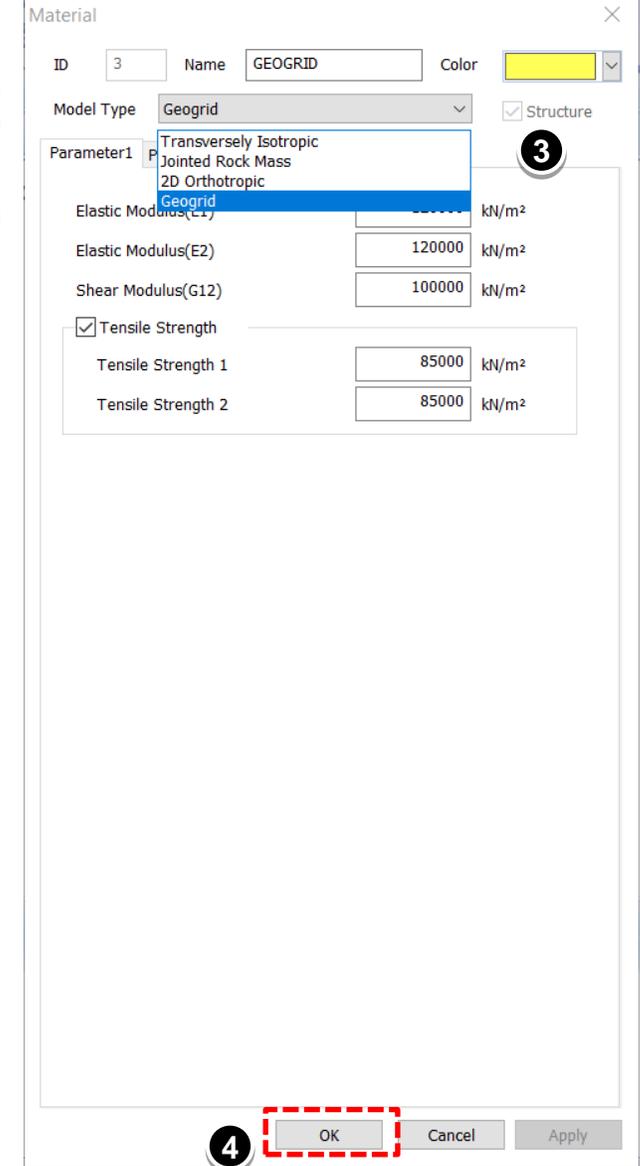
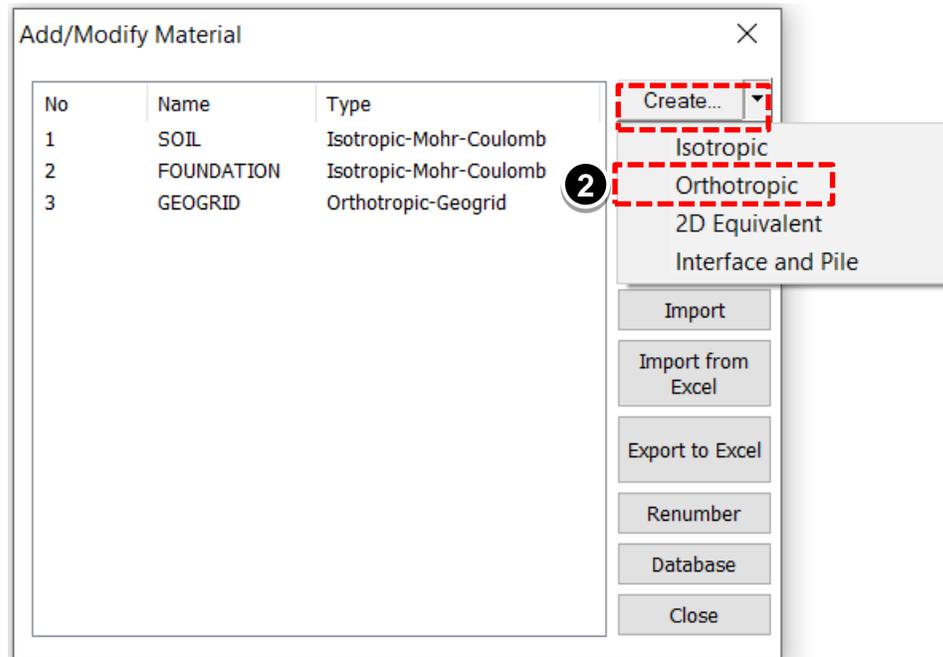
⑥ OK Cancel Apply

2-3 Defining Geogrid Material

Procedure

Defining Soil Materials

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



2-3 Defining Geogrid Material

Procedure

Defining Soil Materials

- 1 Select parameter 1.
- 2 In Parameter 1 tab, Enter the value of **E1 = 120000**, **E2 = 120000**, **G12 = 100000**, **kN/m²**
- 3 Click the Tensile Strength Box, and enter value of **Tensile Strength 1 = 85000**, **Tensile Strength 2 = 85000 kN/m²**
- 4 Enter **Unit weight = 21.5**.
- 5 Keep the Thermal Parameters as **default**.
- 6 Click **OK**

Material

ID 3 Name GEOGRID Color

Model Type Geogrid Structure

Parameter1 Parameter2 Thermal

Elastic Modulus(E1) kN/m²

Elastic Modulus(E2) kN/m²

Shear Modulus(G12) kN/m²

Tensile Strength

Tensile Strength 1 kN/m²

Tensile Strength 2 kN/m²

OK Cancel Apply

Material

ID 3 Name GEOGRID Color

Model Type Geogrid Structure

Parameter1 Parameter2 Thermal

Unit Weight(γ) kN/m³

Initial Stress Parameters

Ko Anisotropy

Thermal Parameter

Thermal Coefficient1 1/[T]

Thermal Coefficient2 1/[T]

Thermal Coefficient3 1/[T]

Molecular vapor diffusion coefficient m²/sec

Thermal diffusion enhancement

Damping Ratio(For Dynamic)

Damping Ratio

OK Cancel Apply

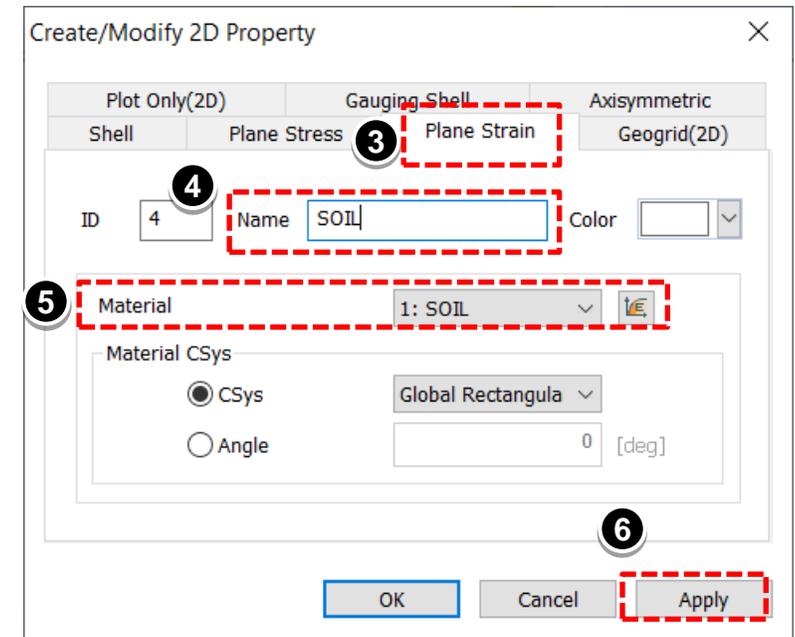
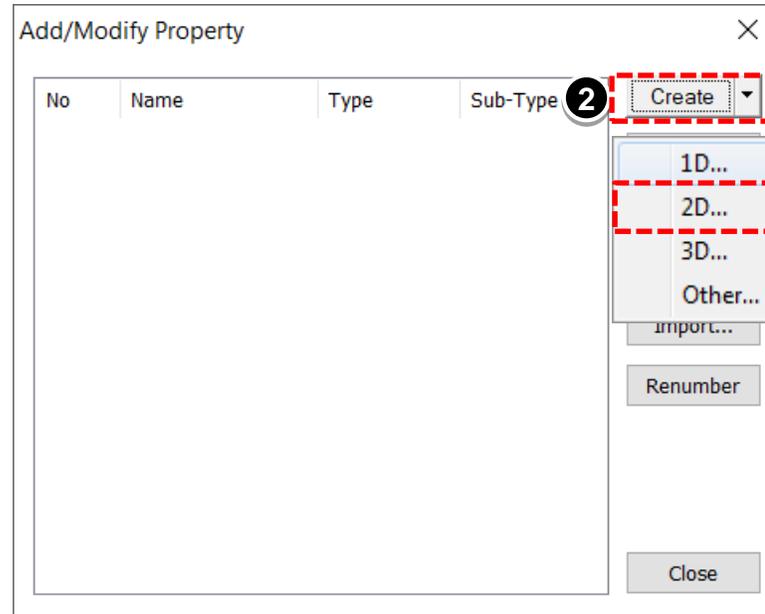
3-1 Defining Property

Procedure

Defining Property

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

Similarly Define the 2D property for Foundation layer.

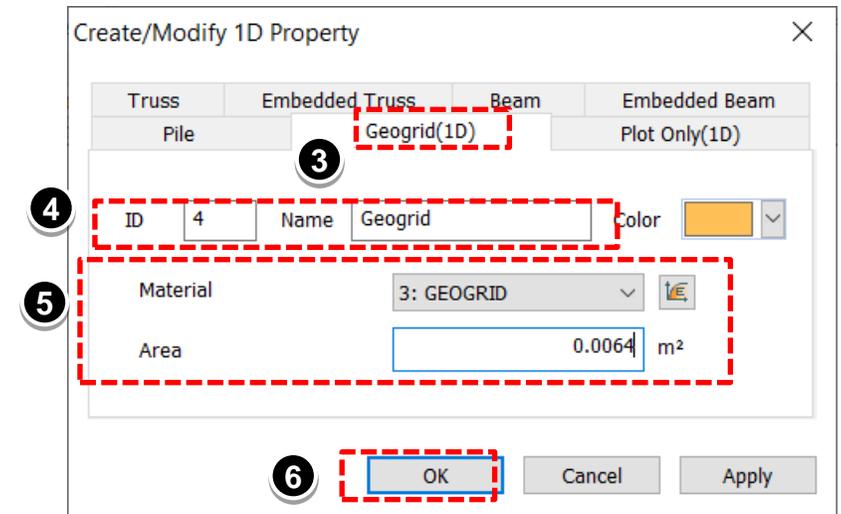
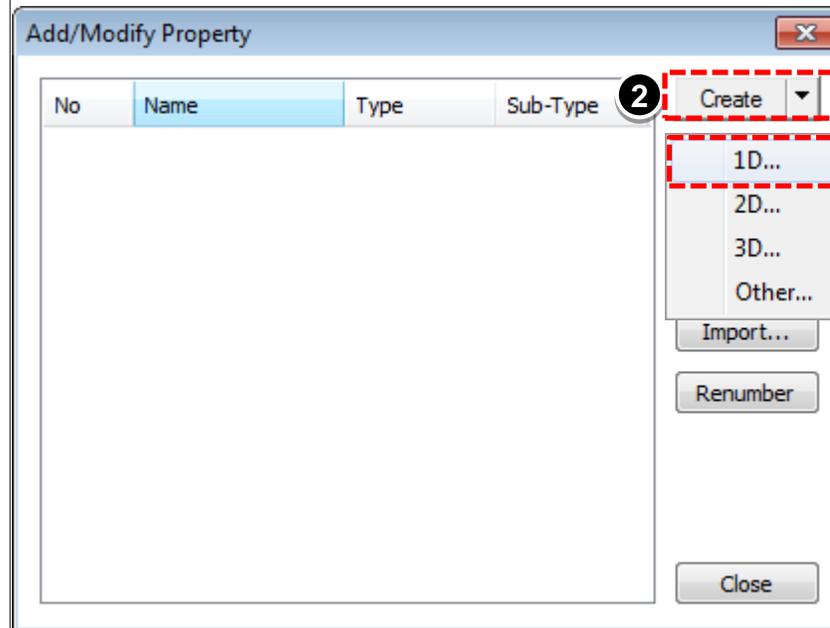


3-2 Defining Geogrid Property

Procedure

Defining Property

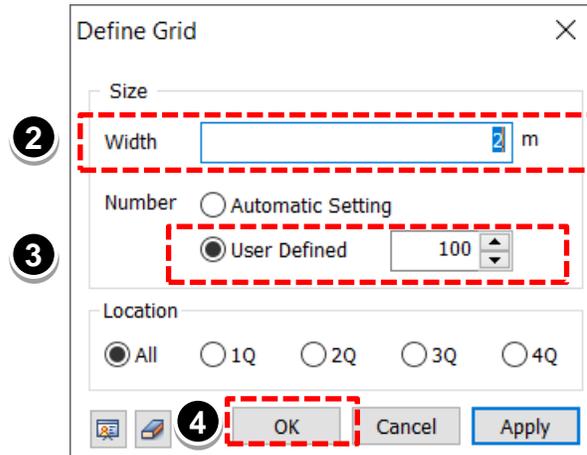
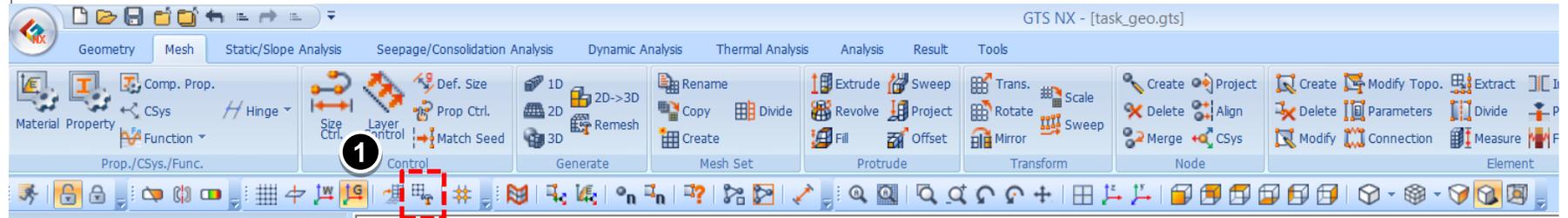
- ① Go to Mesh>Click on Property
- ② Click on Create. Select 1D
- ③ Click on **Geogrid (1D)**
- ④ Name it as **Geogrid**.
- ⑤ Check on Area. Enter the value as **0.0064 m²**
- ⑥ Click **OK**



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**.



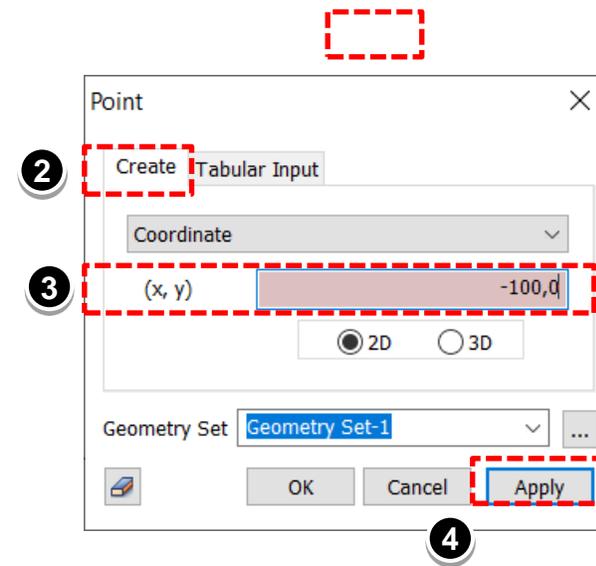
4-2 Geometric modelling-Creation of Points

Procedure

- ① Go to Geometry > Point
 - ② Click Create.
 - ③ Enter the points as shown in the table.
 - ④ Click Apply after entering each point.
- After all the points are created, click OK.



No.	x	y
1	-100	0
2	0	0
3	160	0
4	160	30
5	70	30
6	-100	-50
7	0	-50
8	160	-50
9	14	6
10	50	6
11	28	12
12	60	12
13	42	18
14	78	18
15	56	24
16	92	24



4-3 Geometric modelling-Creation of Lines

Procedure

- ① Go to Geometry > Point
- ② Select method as 2D, Input Start corner at Location **-60,0 (Abs X,Y)**
- ③ Click **Apply**
- ④ Input End Location **160,0.(Abs X,Y)** and click **Apply**
- ⑤ Select Input Start Location **-60,0 (Abs X,Y)**.
- ⑥ Select Input End Location **-60,-85 (Abs X,Y)**.
- ⑦ Click **OK**.

Repeat the same procedure for creating all the lines.

①

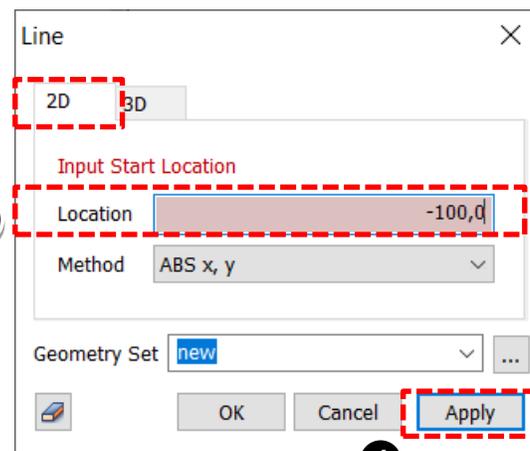


X	Y
14	6
50	6
28	12
60	12
42	18
78	18
56	24
92	24

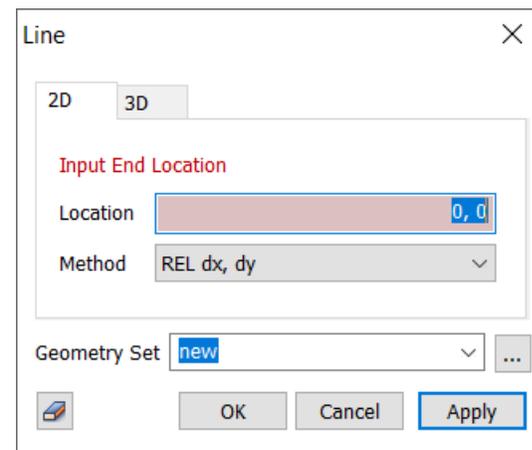
x1	y1	x2	y2
14	6	50	6
28	12	60	12
42	18	78	18
56	24	92	24

②

③

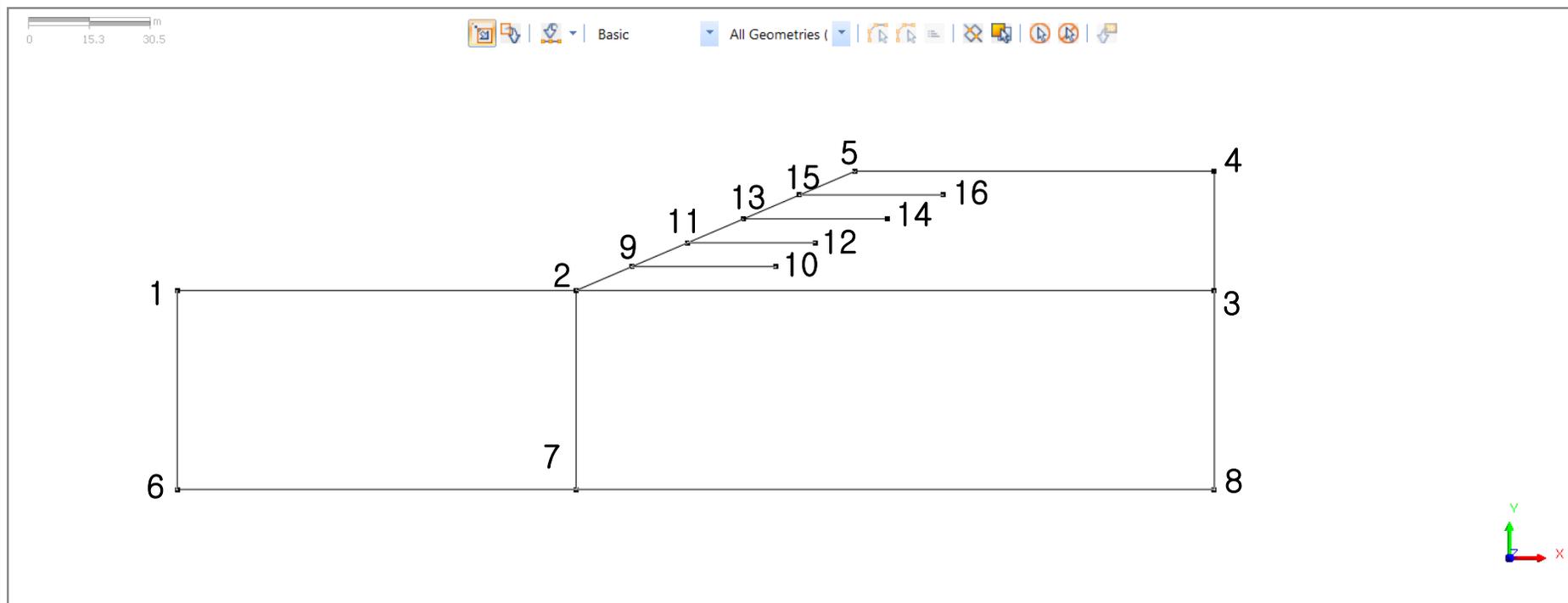


④



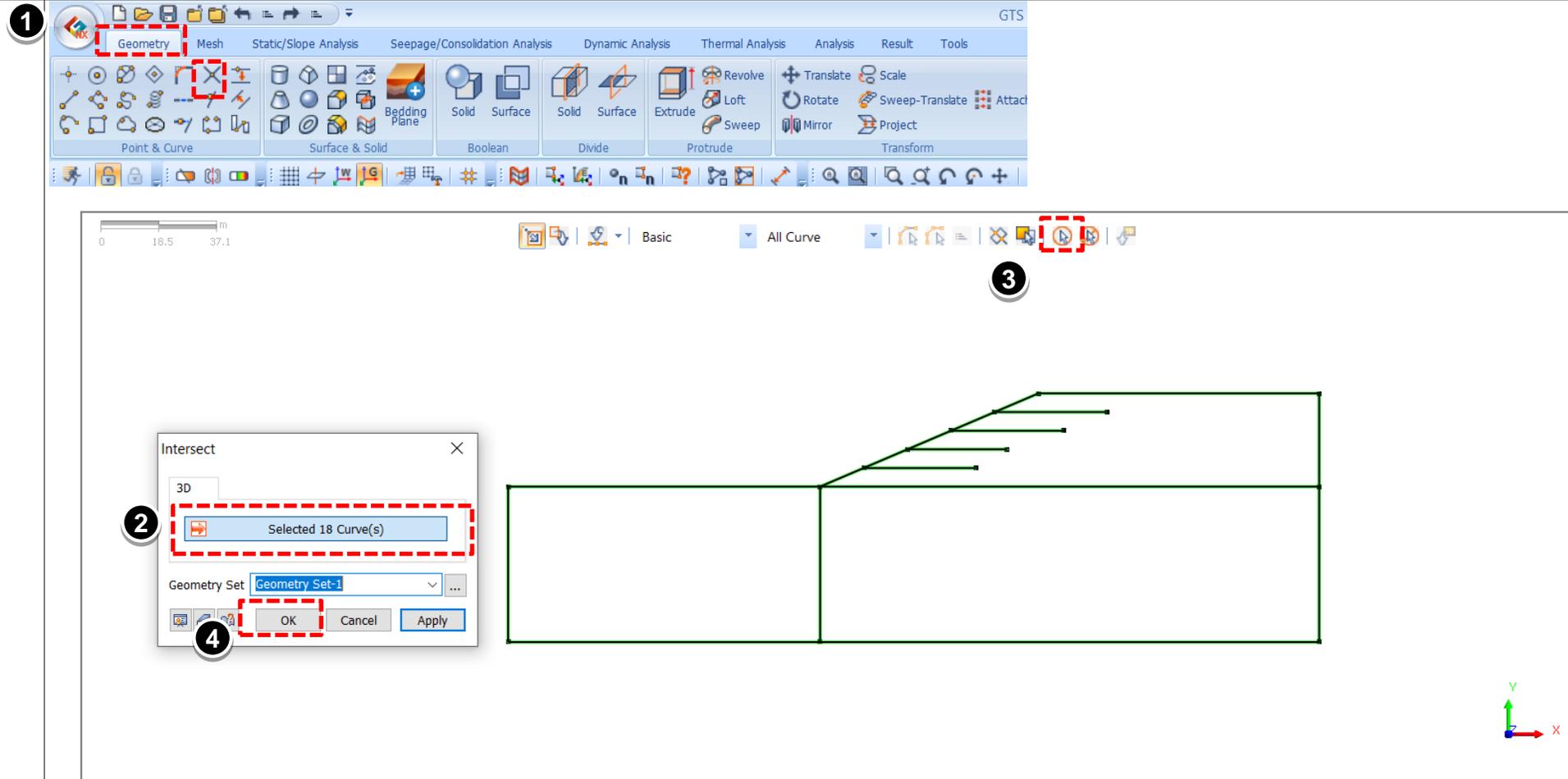
Procedure

- 1 The points presented in Section 4-2 are shown in the picture here.
- 2 The lines connect the points as in the cross section shown here.



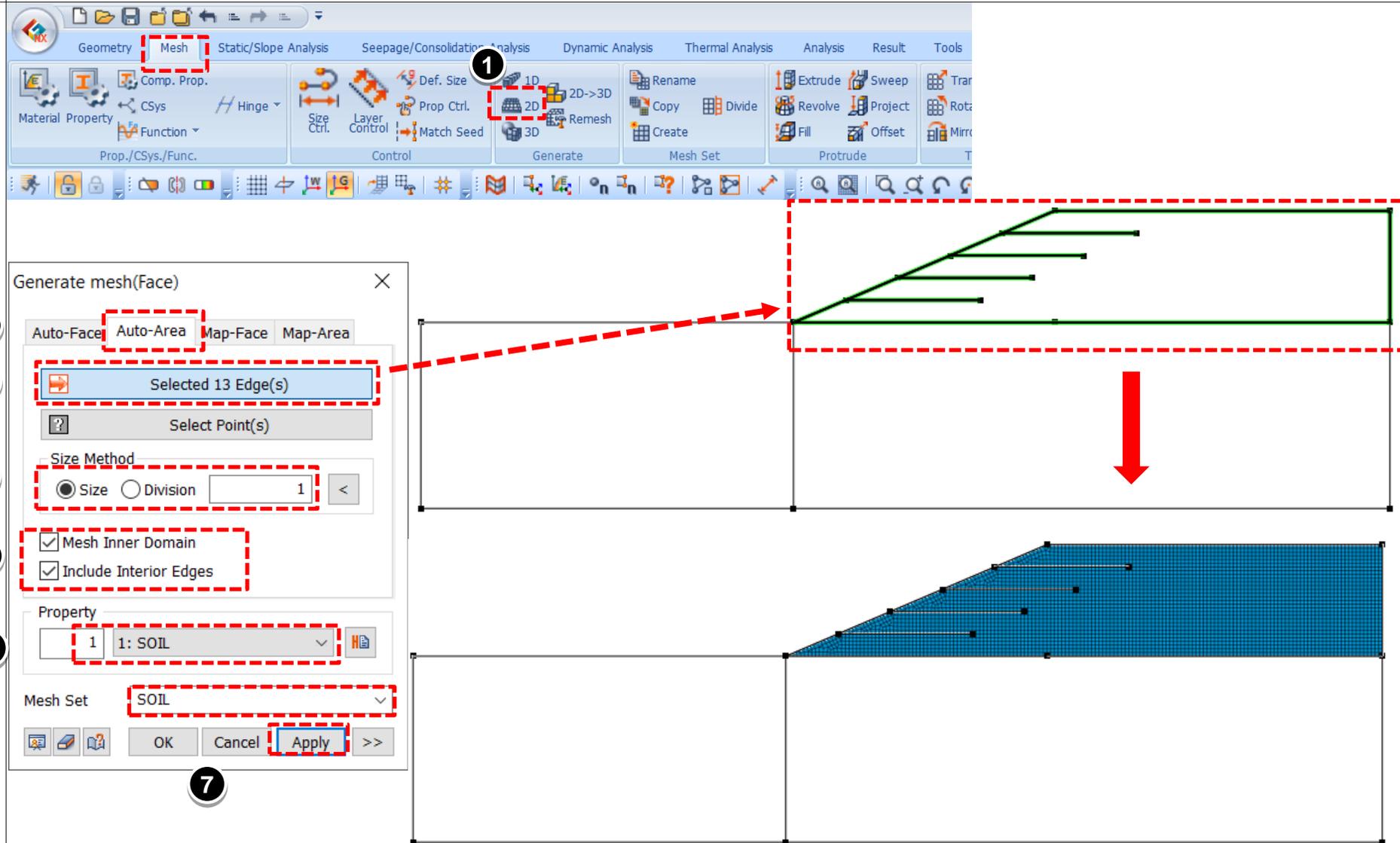
Procedure

- 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.



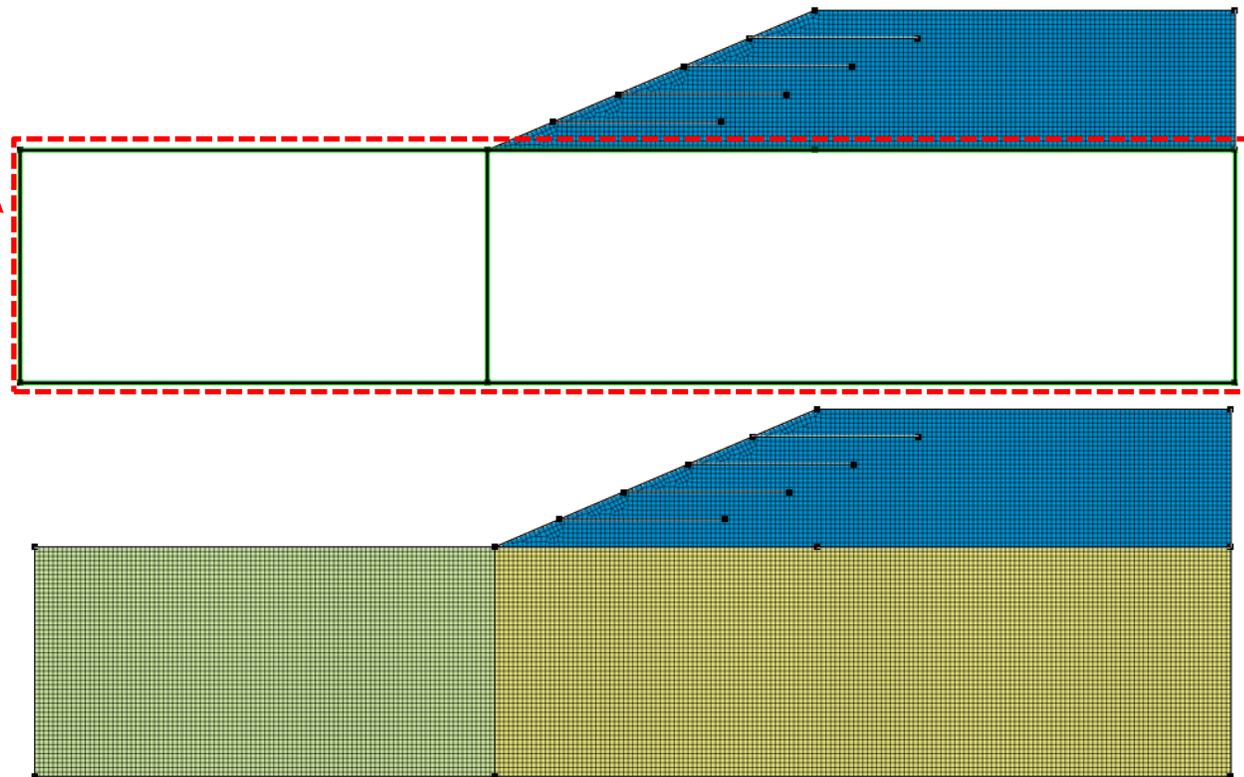
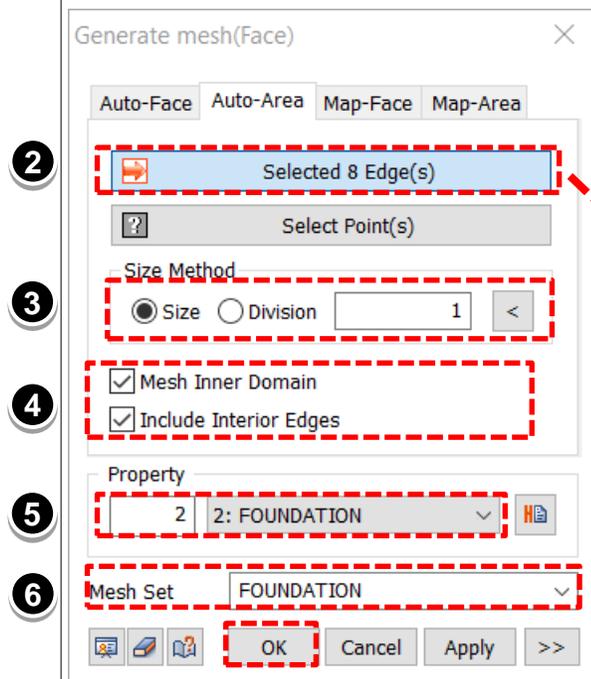
Procedure

- ❶ Go to Mesh > Generate > 2D
- ❷ Select the lines as shown in the box that constitute parts of the embankment.
- ❸ Enter the size as 1 m.
- ❹ Tick on the “Mesh Inner Domain”.
- ❺ The property is SOIL.
- ❻ Give the Mesh set name “SOIL.”
- ❼ Click **Apply**.



Procedure

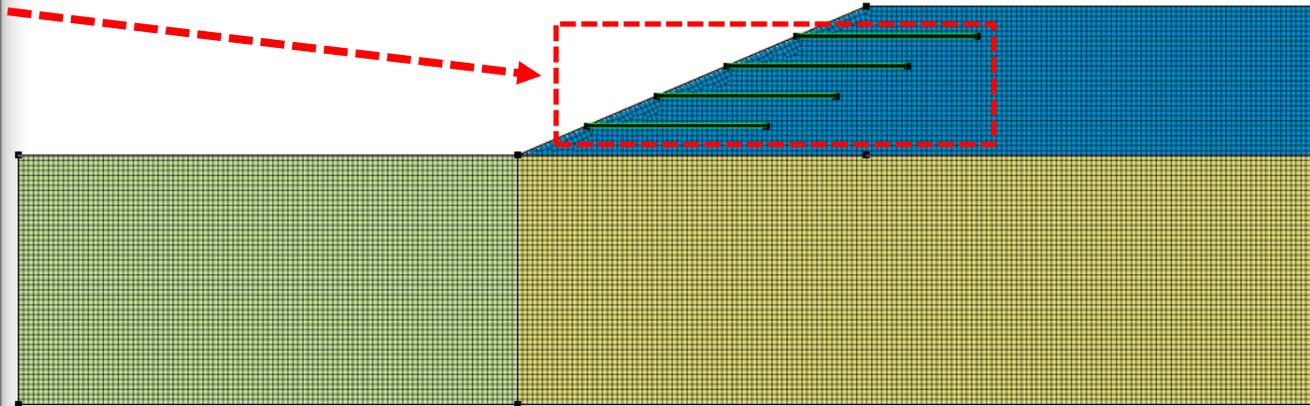
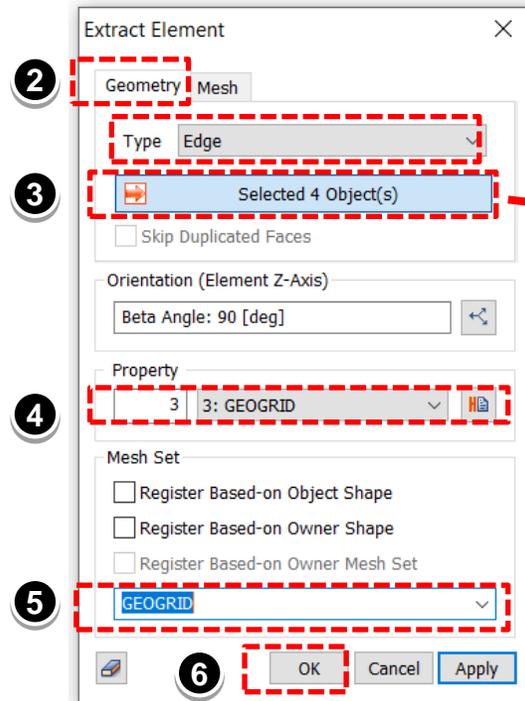
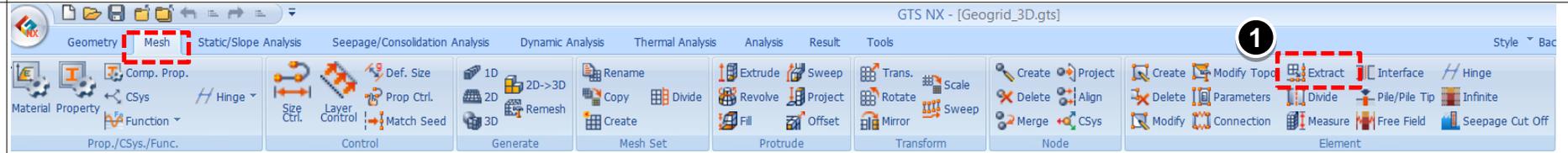
- ❶ Go to Mesh > Generate > 2D
- ❷ Select the lines as shown.
- ❸ In the Size method give size 1 m.
- ❹ Tick on the “Mesh Inner Domain”.
- ❺ In the property drop-down menu, select **Foundation**.
- ❻ Give the Mesh set name “**FOUNDATION.**”
- ❼ Click **OK**.



5-3 Meshing-Extraction of Geogrid 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 **Geogrid**.
- ⑤ Name geogrid mesh set as **Geogrid**.
- ⑥ Click **OK**.



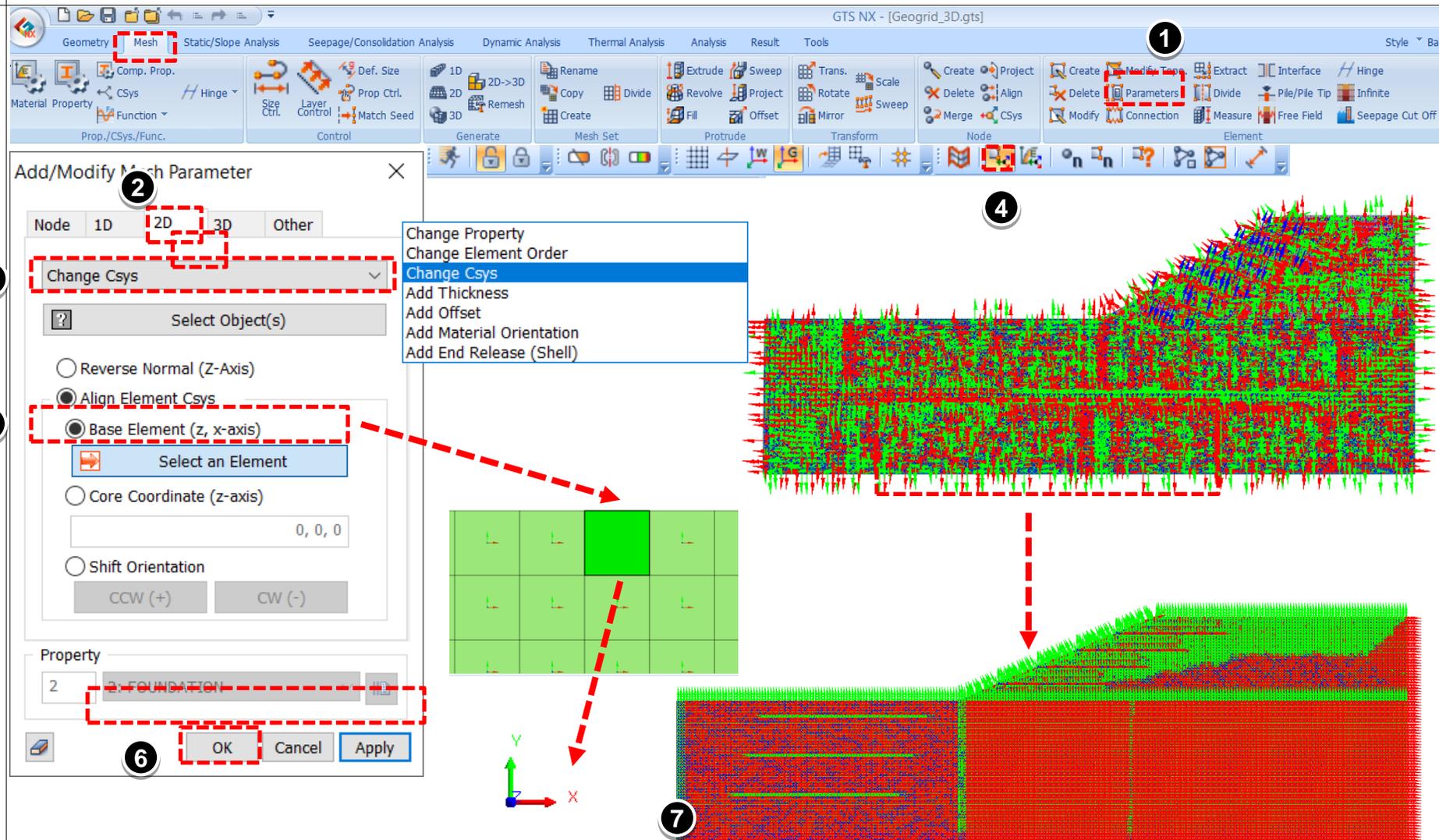
7

5-3 Meshing- Aligning the csys of Elements

Procedure

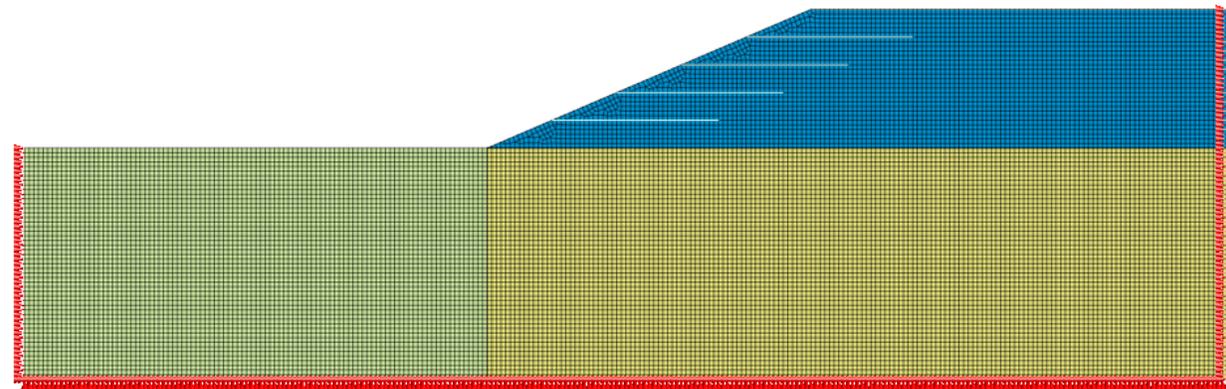
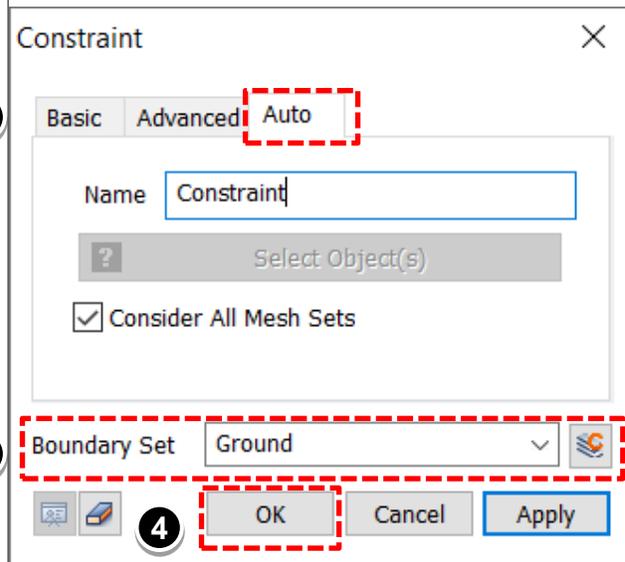
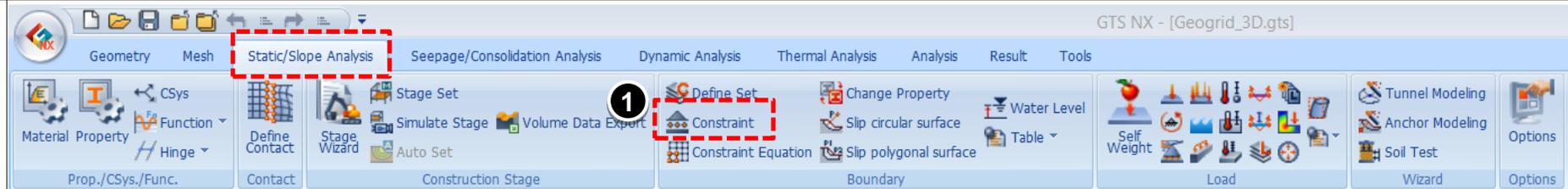
- ① Go to Mesh>Element>parameters.
- ② Go to 2D.
- ③ Change the type to **Change Csyz**.
- ④ Click on **Show Element Csyz**.
- ⑤ Select the Base Element which Csyz resembles the global csyz as shown below.
- ⑥ Click **OK**.

For the geogrid elements, select 1D and repeat the same procedure and select the base element whose Csyz is aligned with the global Csyz as shown below.



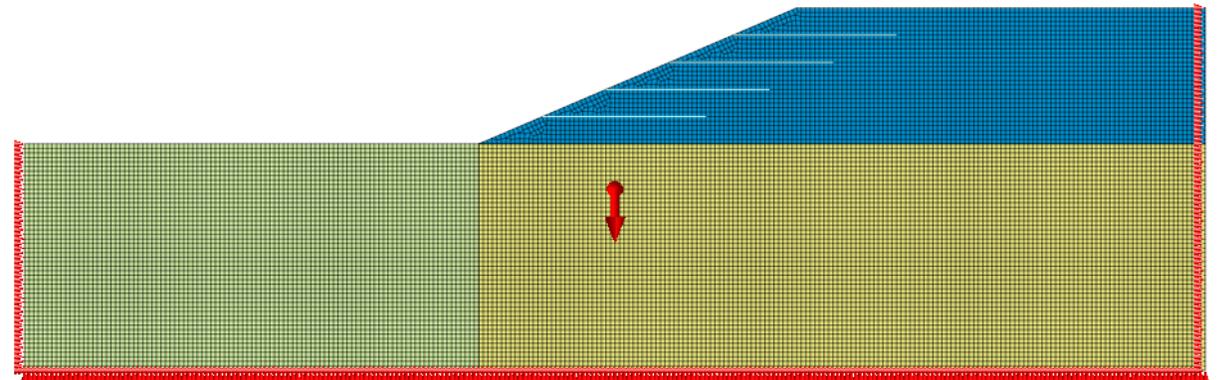
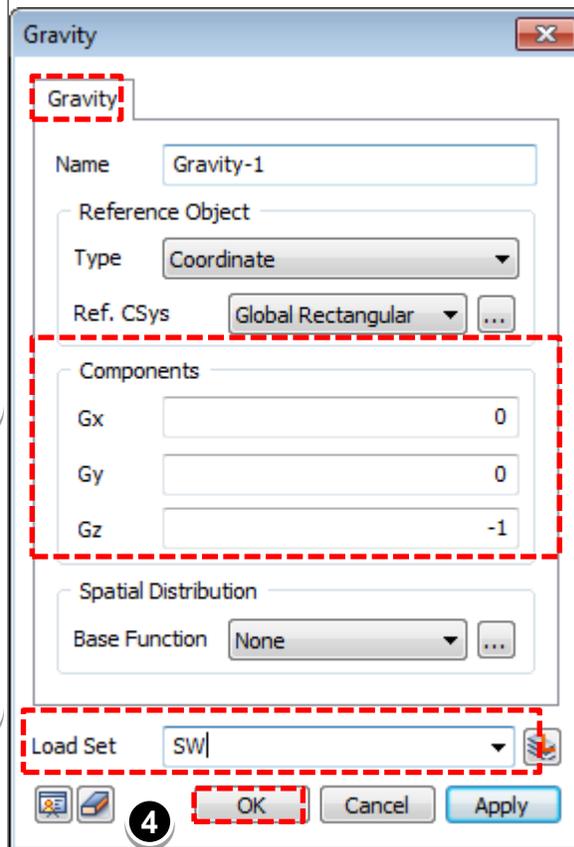
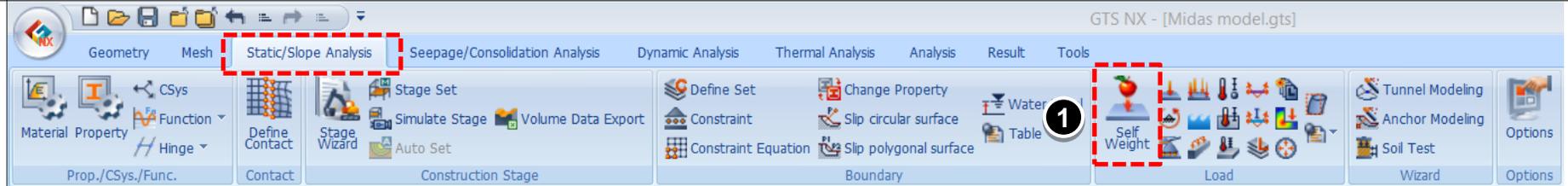
Procedure

- 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

- ① Go to Static/Slope Analysis > Load > Self Weight
- ② Check for gravity Direction, G_y should be -1
- ③ Name the Load Set as "SW"
- ④ Click OK



8-1 Analysis Case Definition

Procedure

- 1 In the Analysis Tab, Click on General
- 2 Name the case as **Slope_No_Geogrid**.
- 3 In the Solution Type Drop-Down Menu, **Select Slope Stability (SRM)**.
- 4 Drag and drop the all the **soil** and **embankment** mesh sets along with the **ground boundary** condition and the **self weight**.
- 5 Click **OK**.

The screenshot illustrates the 'Add/Modify Analysis Case' dialog box in a software application. The 'Analysis' tab is active in the top menu bar. The dialog box is titled 'Add/Modify Analysis Case' and contains the following elements:

- Analysis Case Setting:**
 - Title:** SLOPE_no_geogrid
 - Description:** (empty)
 - Solution Type:** Slope Stability(SRM)
 - Construction Stage Set:** (empty)
 - Analysis Control:** (icon)
 - Output Control:** (icon)
- Analysis Case Model:**
 - All Sets:** Mesh (Default Mesh Set, FOUNDATION, GEOGRID, SOIL), Boundary Condition (GROUND), Static Load, SW, Contact Pair.
 - Active Sets:** Mesh (Default Mesh Set, FOUNDATION, SOIL), Boundary Condition (GROUND), Static Load, SW, Contact Pair.
- Buttons:** Solve Each Load Set Independently (checkbox), Sorting (Name), OK, Cancel, Apply.

To the right of the dialog box, a dropdown menu for 'Slope Stability(SRM)' is open, displaying a list of analysis options. The 'Slope Stability(SRM)' option is highlighted in blue.

8-2 Analysis Case Definition

Procedure

- 1 In the Analysis Tab, Click on General
- 2 Name the case as **Slope_Geogrid**.
- 3 In the Solution Type Drop-Down Menu, **Select Slope Stability (SRM)**.
- 4 Drag and drop the all the **soil, geogrid and foundation** mesh sets along with the **ground** boundary condition and the **self weight**.
- 5 Click **OK**.

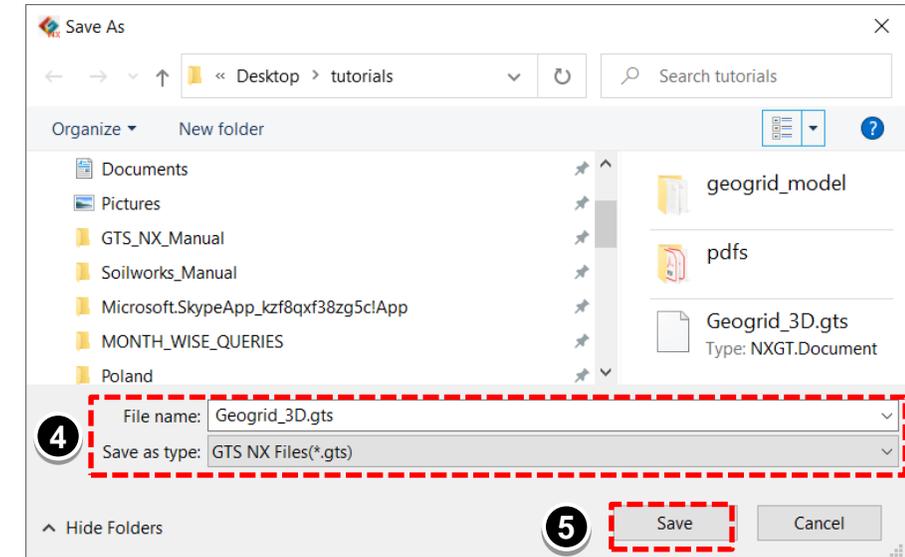
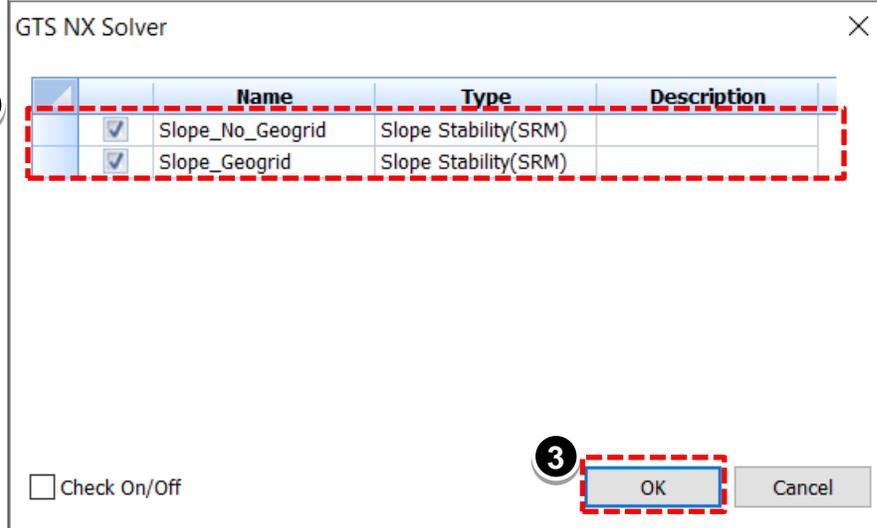
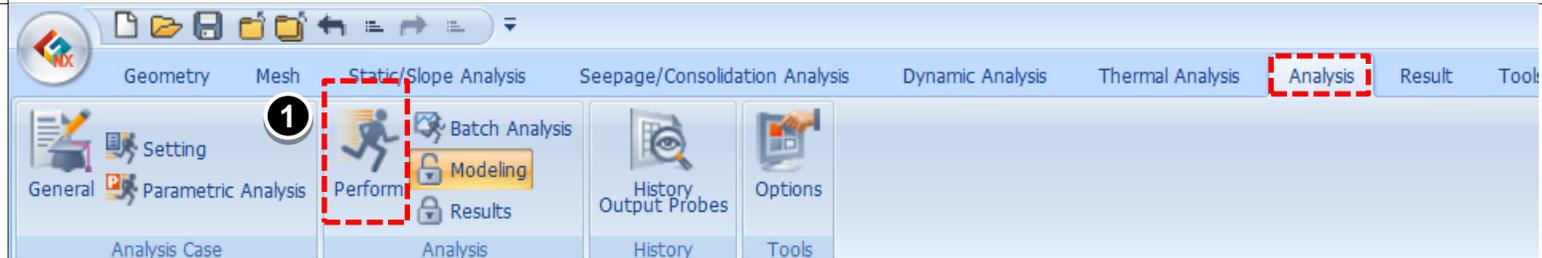
The screenshot illustrates the 'Add/Modify Analysis Case' dialog box in a software application. The 'Analysis' tab is selected in the top menu. The dialog box is titled 'Add/Modify Analysis Case' and contains the following elements:

- Analysis Case Setting:**
 - Title:** SLOPE_geogrid
 - Description:** (empty)
 - Solution Type:** Slope Stability(SRM)
 - Construction Stage Set:** (empty)
 - Analysis Control:** (icon)
 - Output Control:** (icon)
- Analysis Case Model:**
 - All Sets:** Mesh (Default Mesh Set, FOUNDATION, GEOGRID, SOIL), Boundary Condition (GROUND), Static Load (SW), Contact Pair.
 - Active Sets:** Mesh (Default Mesh Set, FOUNDATION, GEOGRID, SOIL), Boundary Condition (GROUND), Static Load (SW), Contact Pair.

A red dashed box highlights the 'Active Sets' panel. A red dashed arrow points from the 'Slope Stability(SRM)' option in the Solution Type dropdown menu to the 'Active Sets' panel. The 'OK' button is highlighted with a red dashed box.

Procedure

- ❶ Go to Analysis > Perform
- ❷ Select both the analysis cases.
- ❸ Click **OK**
- ❹ Give the File name
- ❺ Save it at desired location



10-1 Slope Factor Of Safety Result (Without Geogrid)

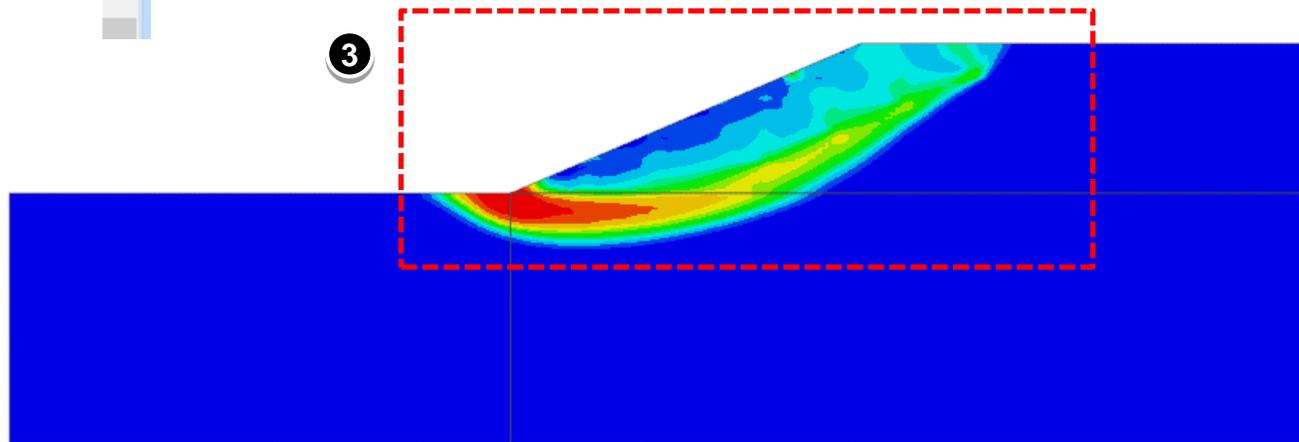
Procedure

- ① Go to Results>
- ② Click on **Solid Strains**
- ③ One can see the localization of strains by clicking on the **E Equivalent** tab.
- ④ The **Safety Factor** tab shows the calculated critical factor of safety of slope .

NOTE: As Slope FoS is less than 1.5, hence it requires reinforcement .

Item	ID	Color
Slope Stability(SRM)		
INCR=5 (FOS=1.3004)		
Displacements		
Grid Forces		
Plane Strain Forces		
Plane Strain Stresses		
Plane Strain Strains		
E-XX		
E-YY		
E-XY		
E-MAJOR PRINCI...		
E-MINOR PRINCI...		
E-VON MISES		
E-MAX SHEAR		
E-VOLUMETRIC		
E-DEVIATORIC		
E-EFFECTIVE PLA		
E-EQUIVALENT		
Safety Factor		
1.30039 [Slope Stability.		
SLOPE geogrid		

③



②

④

①

10-2 Slope Factor Of Safety Result (With Geogrid)

Procedure

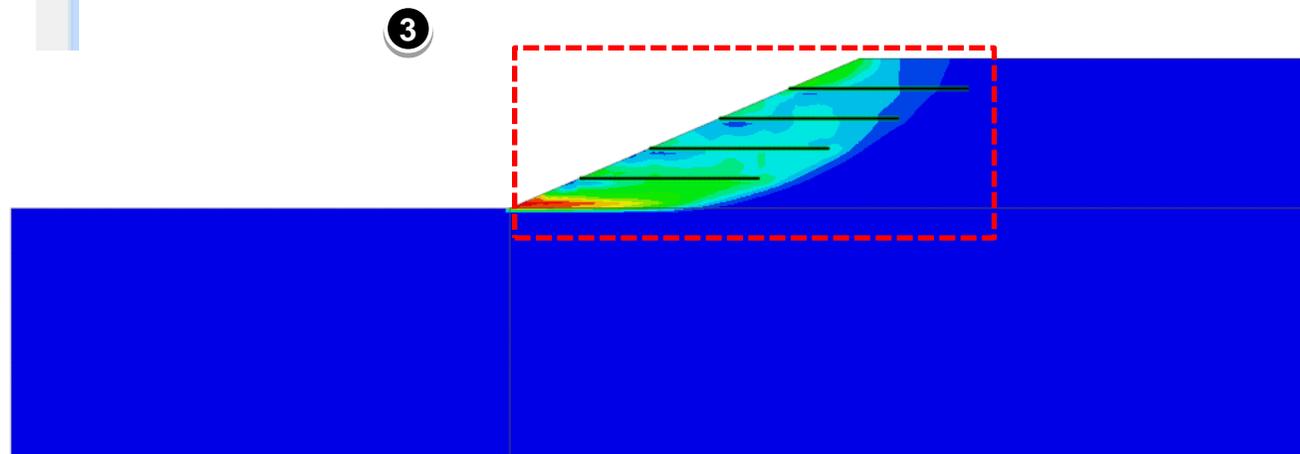
- ① Go to Results>
- ② Click on **Plain Strain Strains**
- ③ One can see the localization of strains by clicking on the **E Equivalent** tab.
- ④ The **Safety Factor** tab shows the calculated critical factor of safety of slope.

NOTE: The critical slope factor of safety increases by 0.37 with the addition of geogrid.

Results

Item	ID	Color
1.30039 [Slope Stability.]		
SLOPE_geogrid		
Slope Stability(SRM)		
INCR=14 (FOS=1.6750)		
Displacements		
Grid Forces		
Truss Element Forces		
Plane Strain Forces		
Truss Element Stress...		
Plane Strain Stresses		
Truss Element Strain...		
Plane Strain Strains		
E-XX		
E-YY		
E-XY		
E-MAJOR PRINCI...		
E-MINOR PRINCI...		
E-VON MISES		
E-MAX SHEAR		
E-VOLUMETRIC		
E-DEVIATORIC		
E-EFFECTIVE PLA...		
E-EQUIVALENT		
Safety Factor		
1.675 [Slope Stability(S..		

Model Analysis Results



②

④

①

- 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.