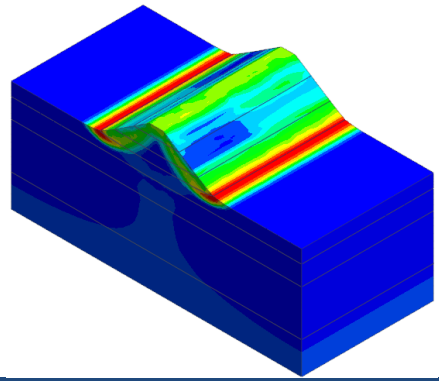
MIDAS *Technical Material*

Tutorial



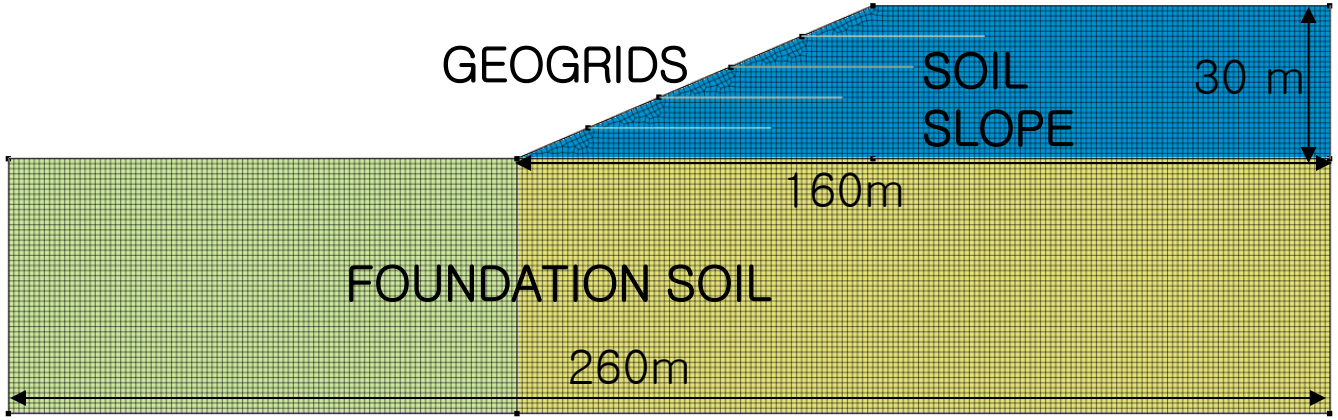
2D Geogrid Reinforced Slope Stability Tutorial

Contents

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

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

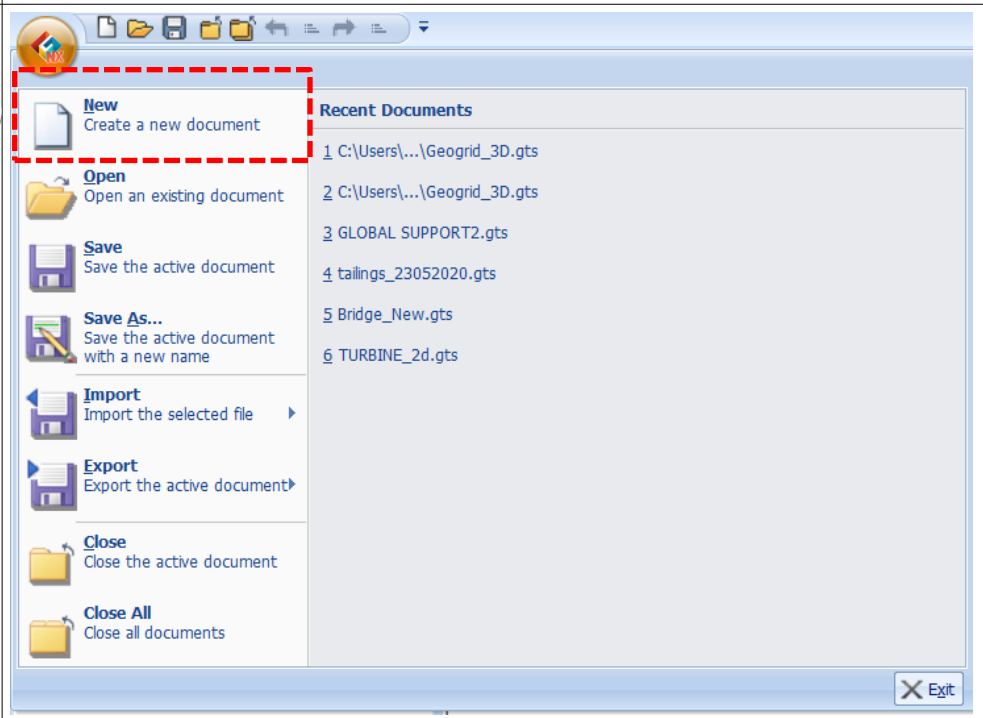
Name	Material Model	Modulus of Elasticity (kN/m^2)	Modulus of Elasticity (kN/m^2)	Shear Modulus (kN/m^2)	Tensile Strength 1 (kN/m^2)	Tensile Strength 2 (kN/m^2)	Unit Weight (kN/m^3)
Geogrid	Orthotropic (Geogrid)	120000	120000	100000	85000	85000	21.5

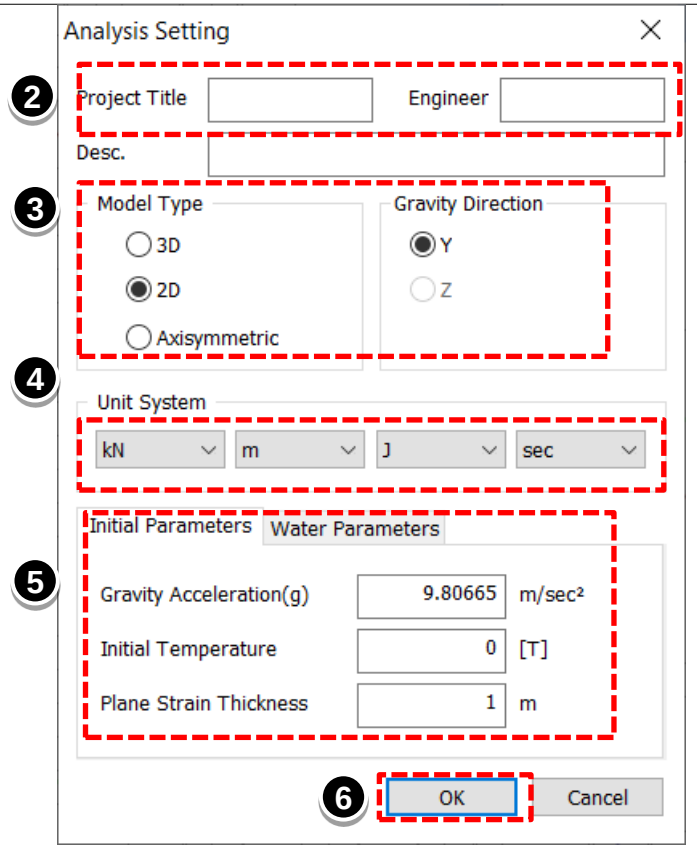
1-1 Initial Setting

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**.





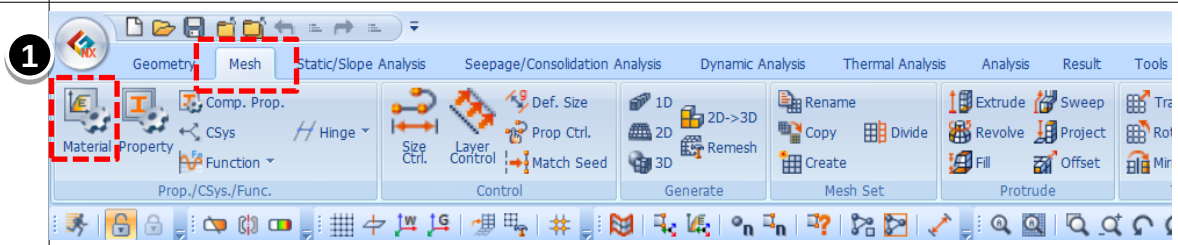
Step 02

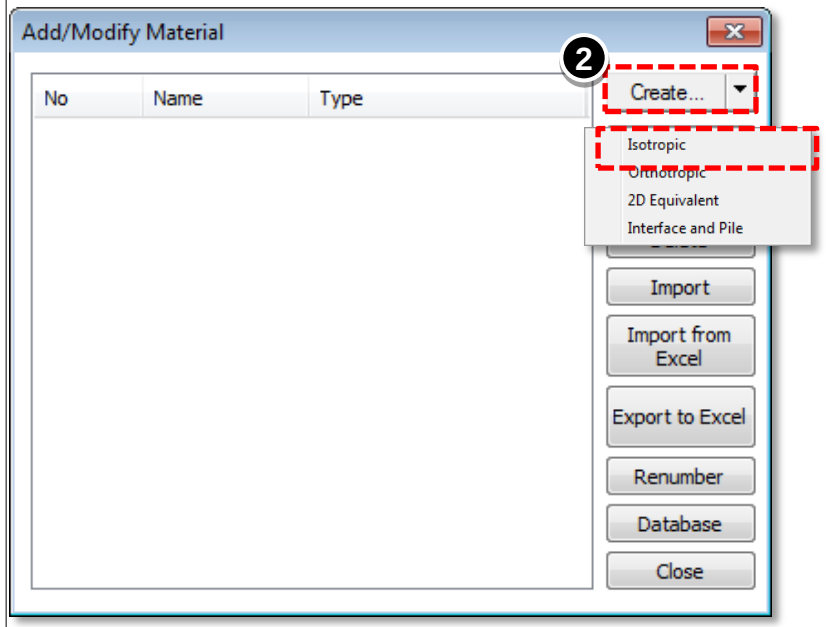
2-1 Defining Soil Material

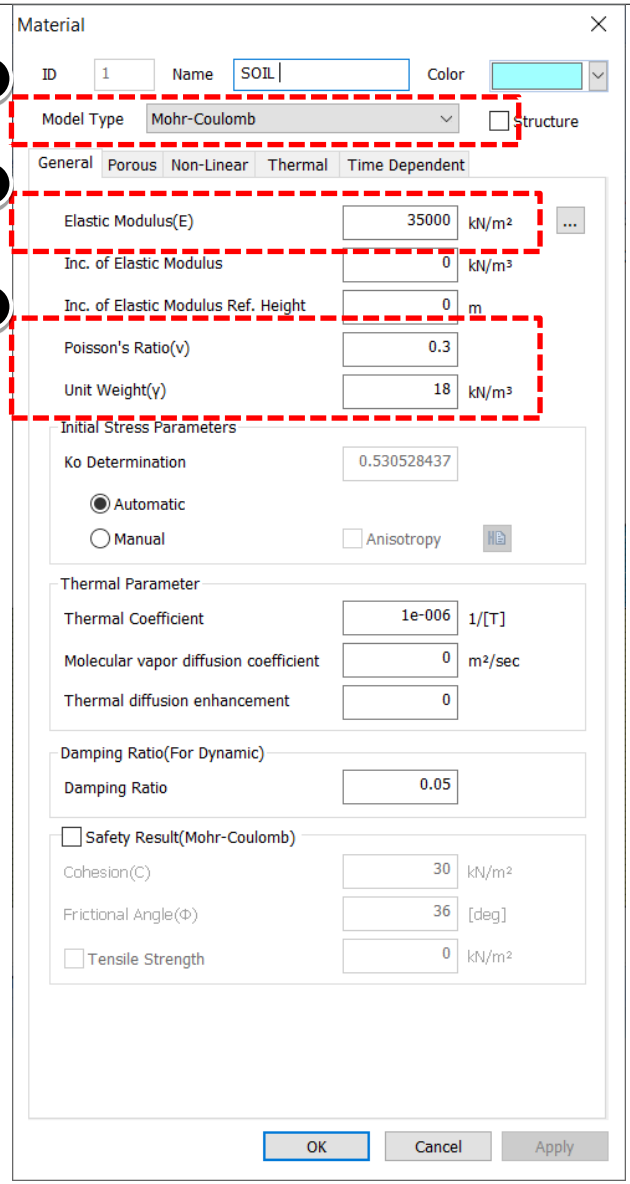
Procedure

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/m²**
5. Enter **Poisson's ratio= 0.3**, **Unit weight = 18 kN/m³**.





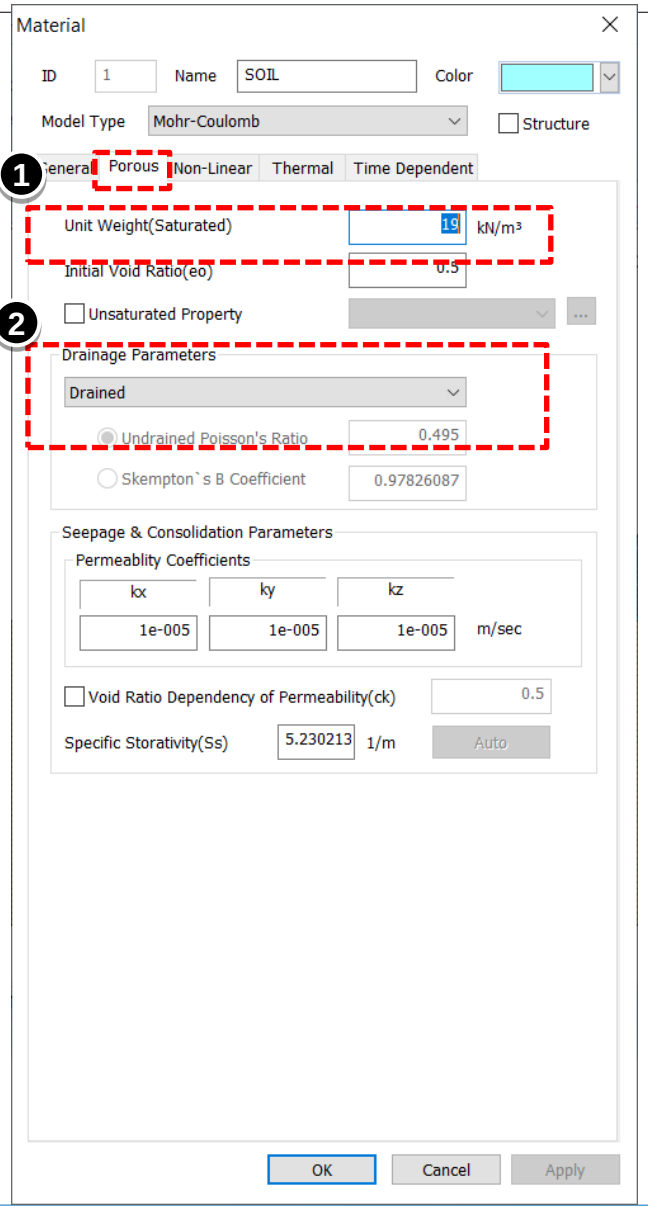


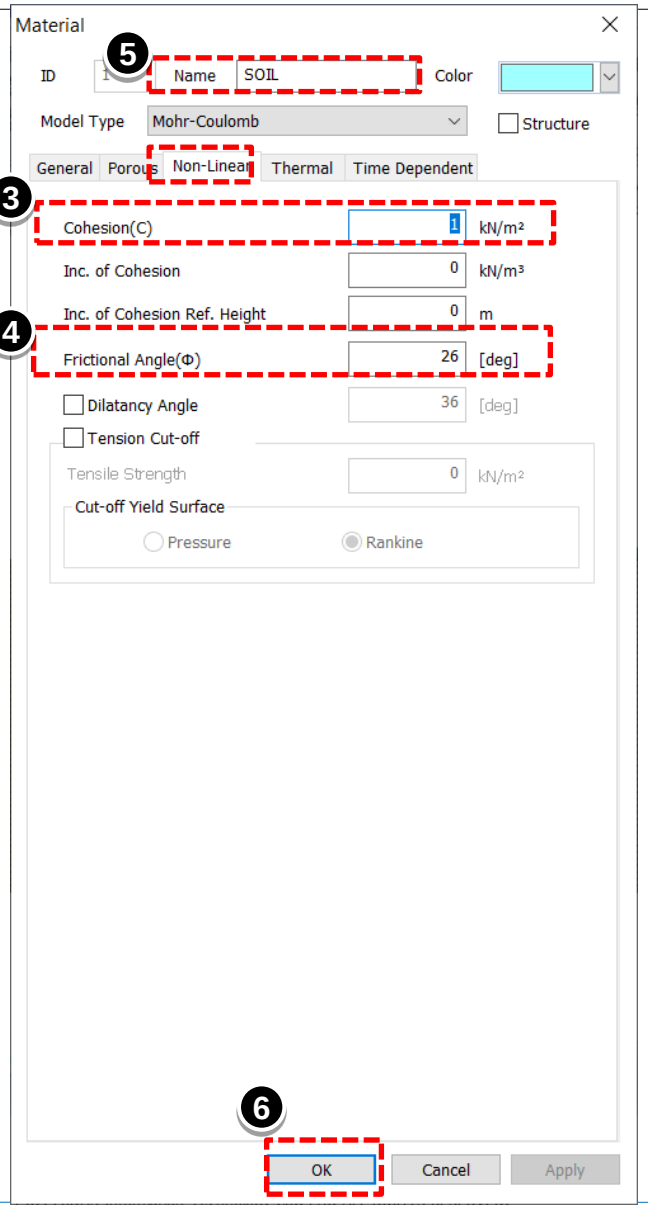
2-2 Defining Soil Material

Procedure

1. Go to Porous Tab > **Enter Unit weight (saturated) =19 kN/m^3**
2. Keep Drainage Parameters as **Drained**
3. Go to Non-Linear Tab > Enter the value of **Cohesion = 1 kN/m^2**
4. Input the value of **Frictional angle =26^0**
5. Enter the name as **SOIL**.
6. Click **OK**

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



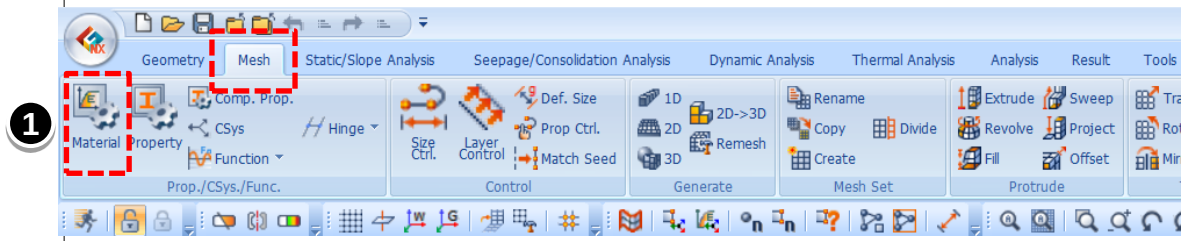


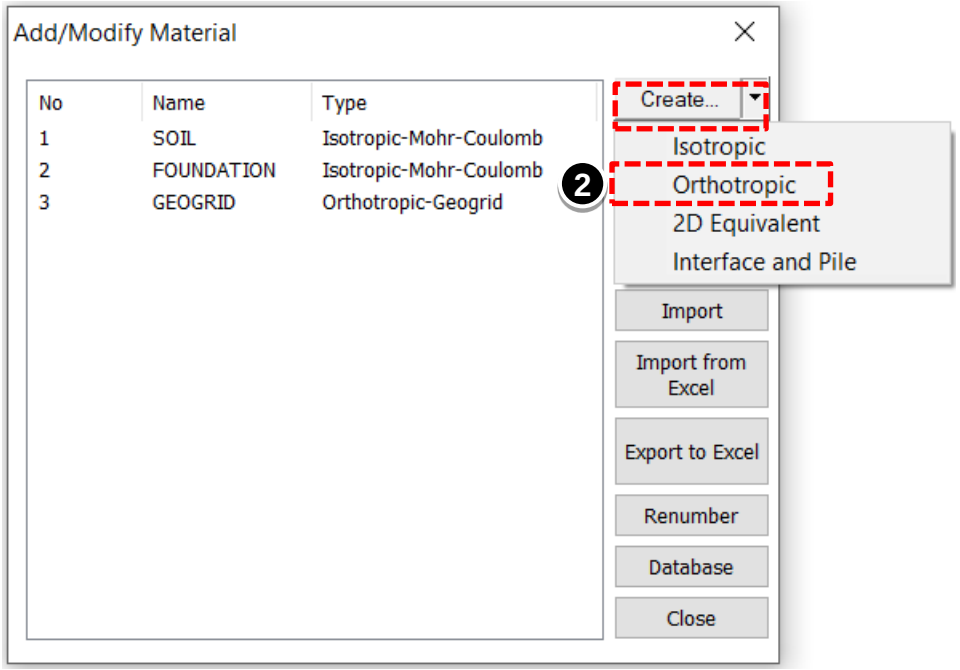
2-3 Defining Geogrid Material

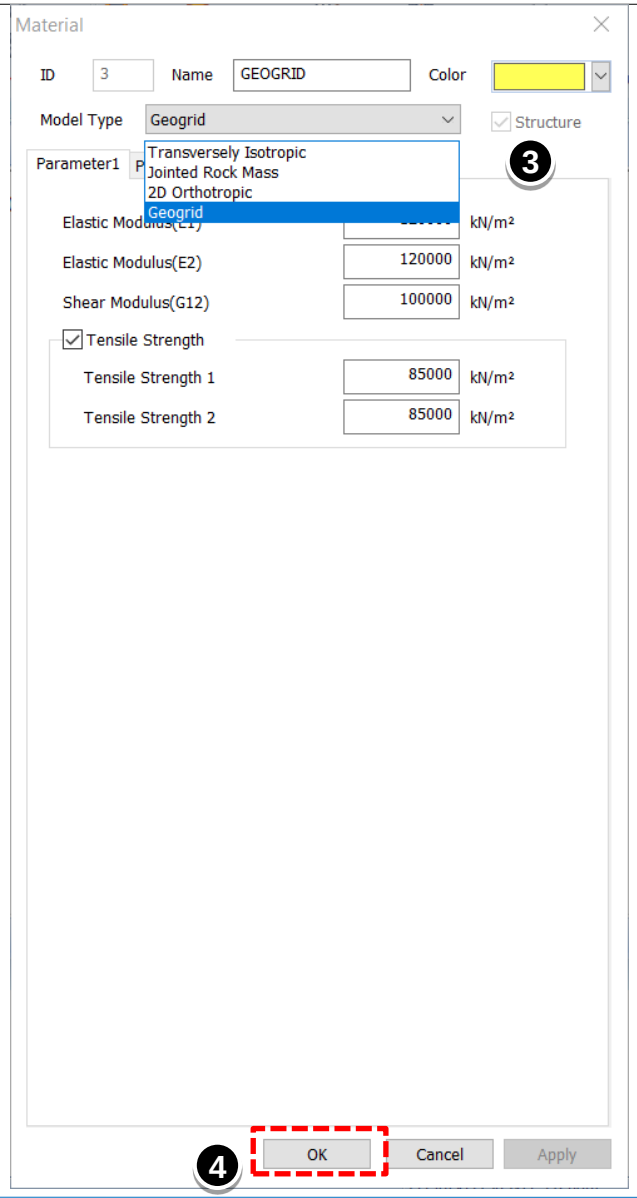
Procedure

Defining Soil Materials

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







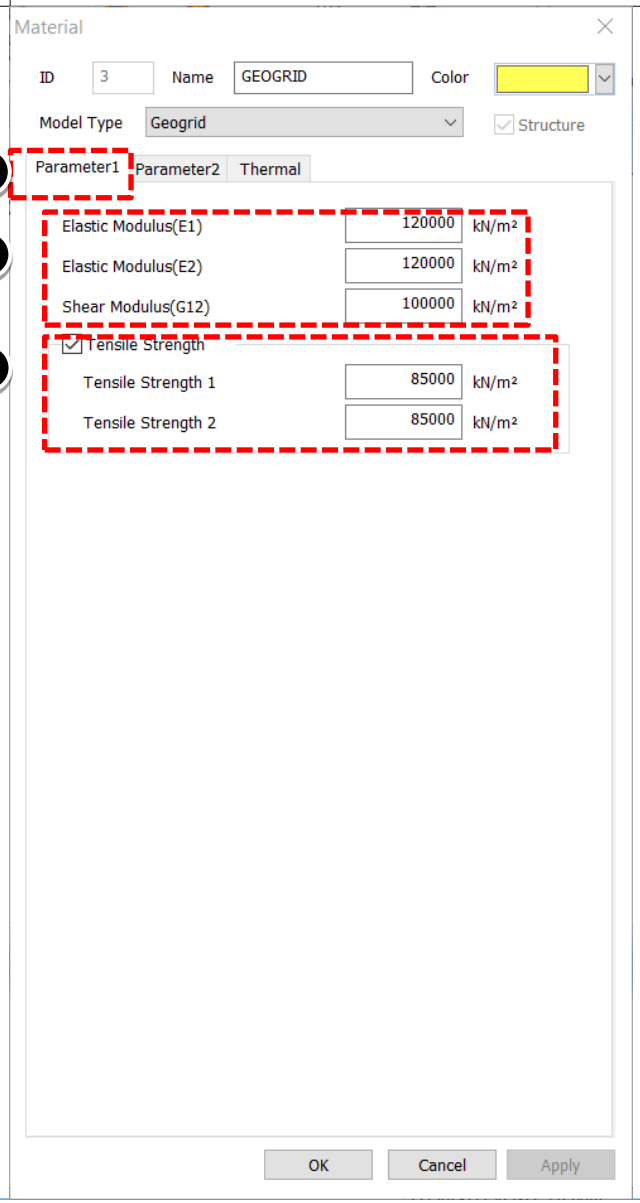
Step 02

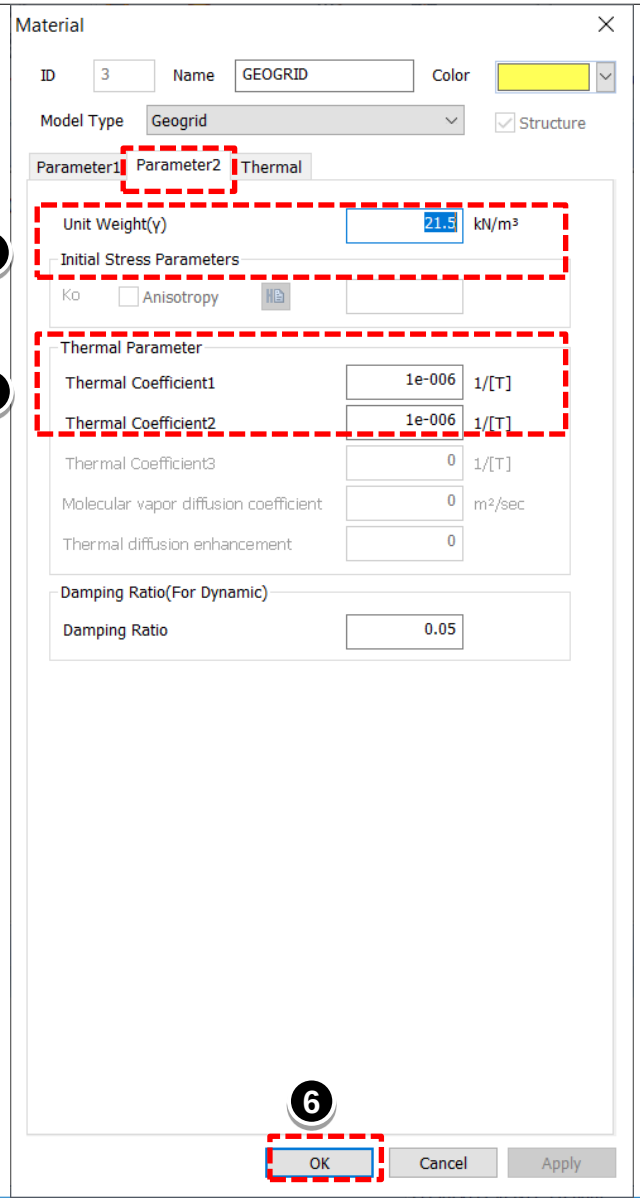
2-3 Defining Geogrid Material

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 kN/m³**.
5. Keep the Thermal Parameters as **default**.
6. Click **OK**





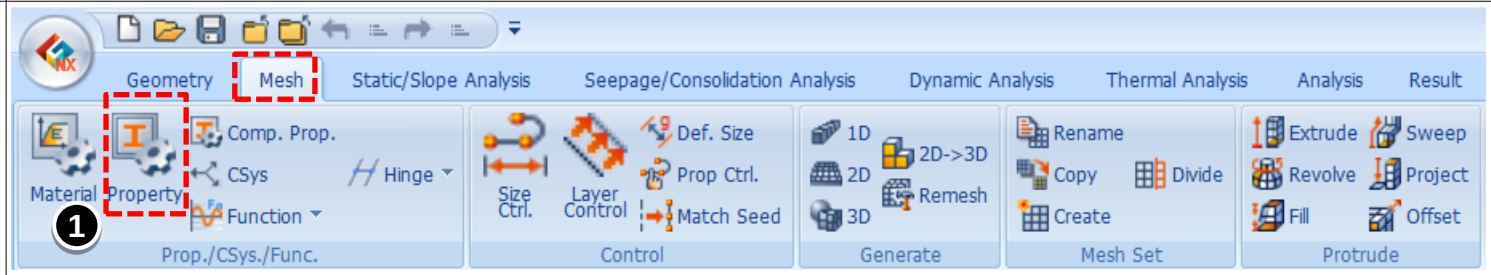
3-1 Defining Property

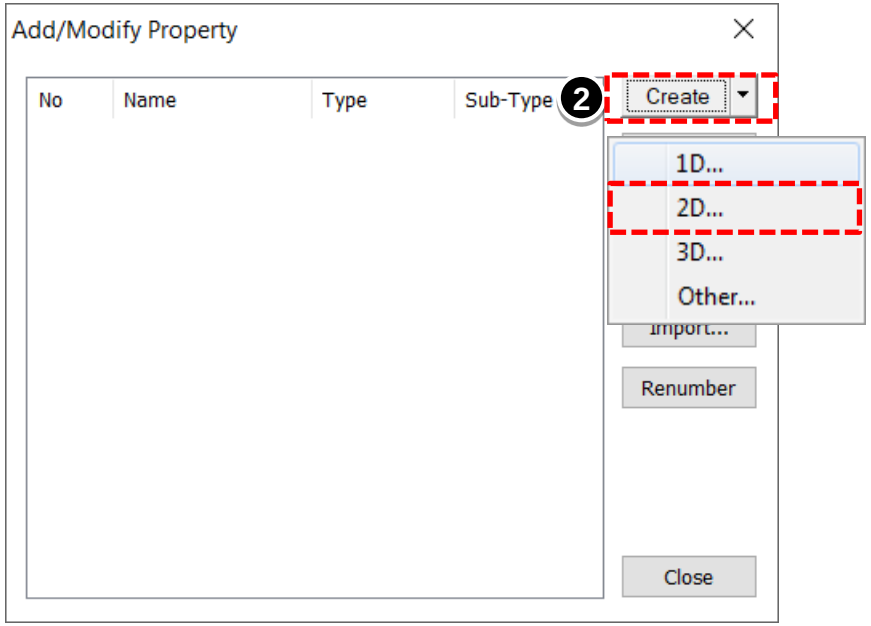
Procedure

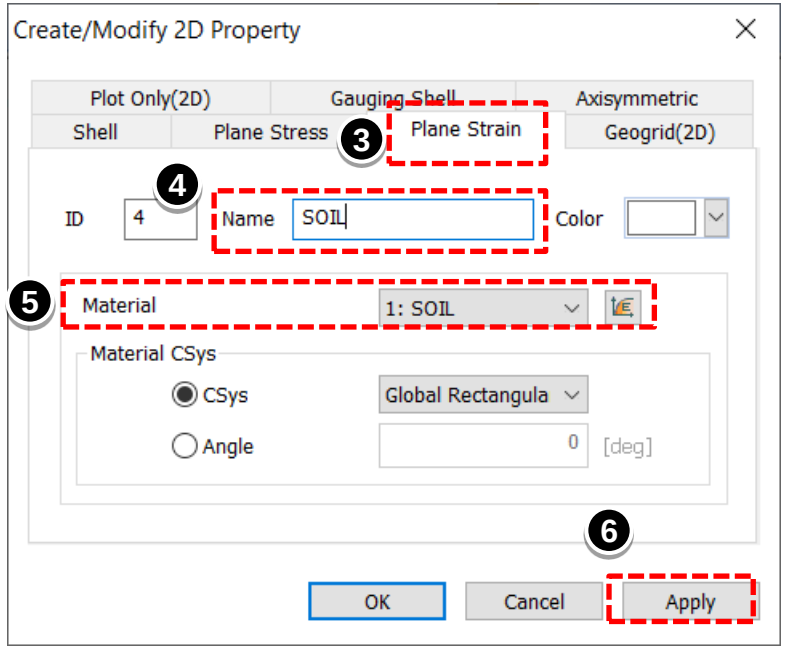
Defining Property

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.







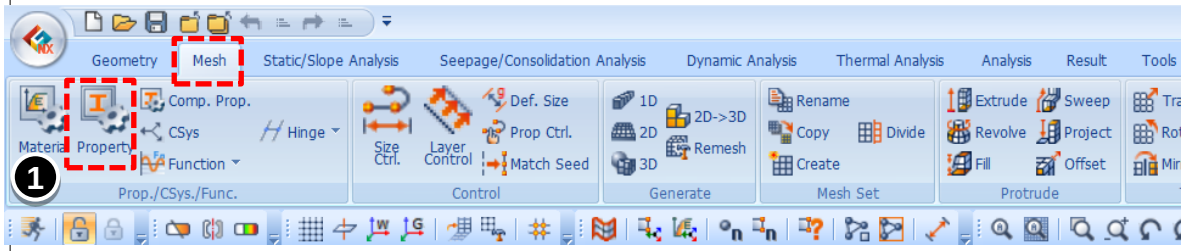
Step 03

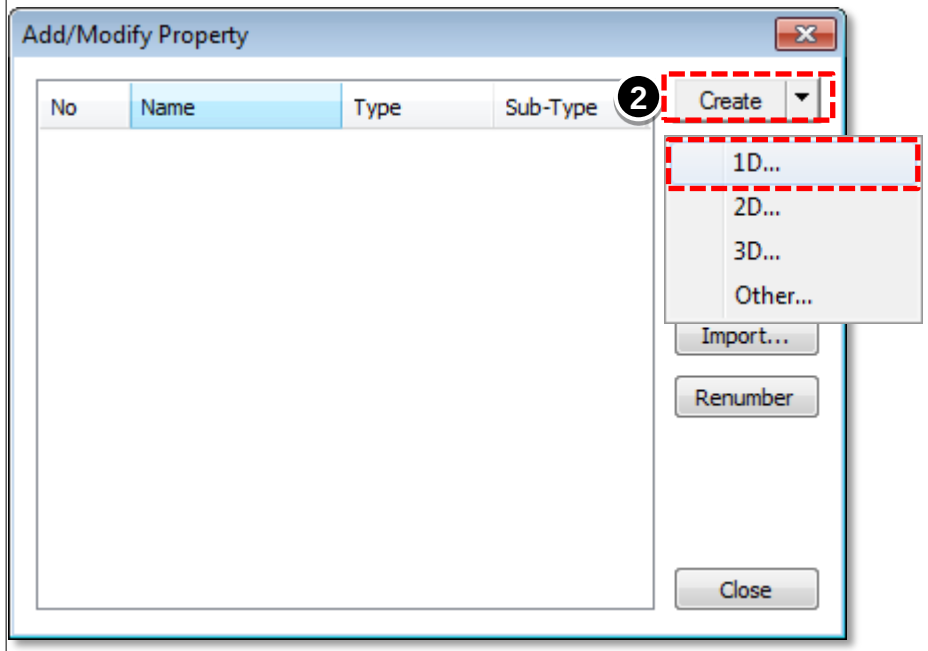
3-2 Defining Geogrid Property

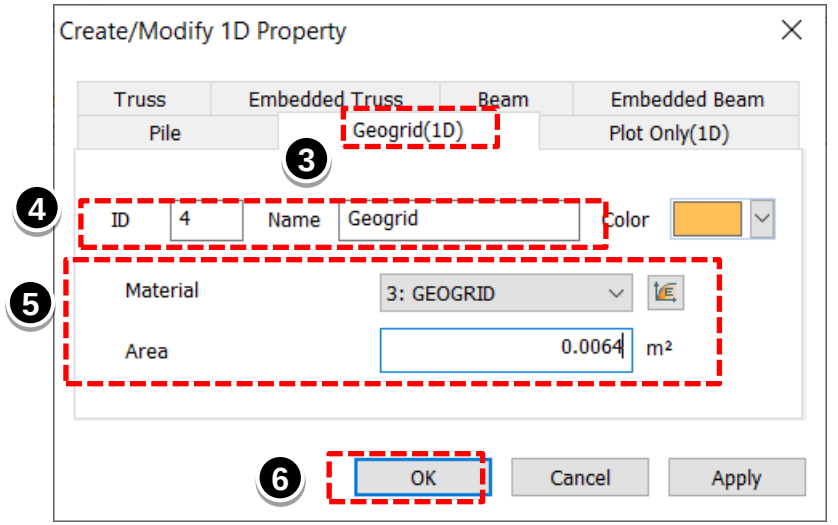
Procedure

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**





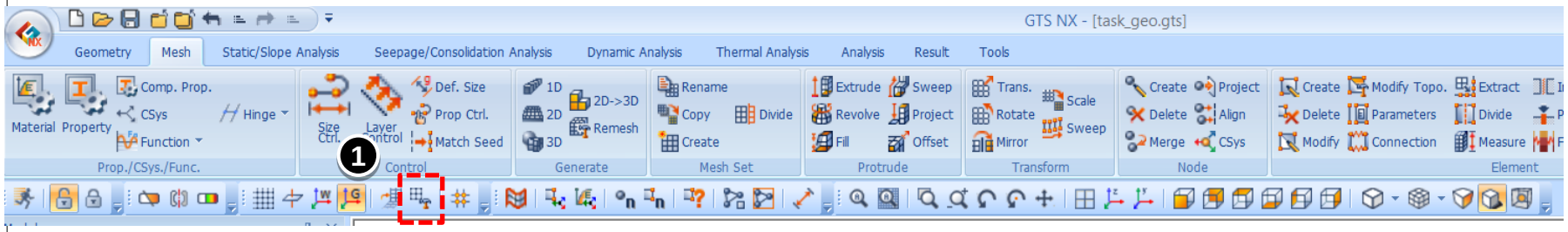


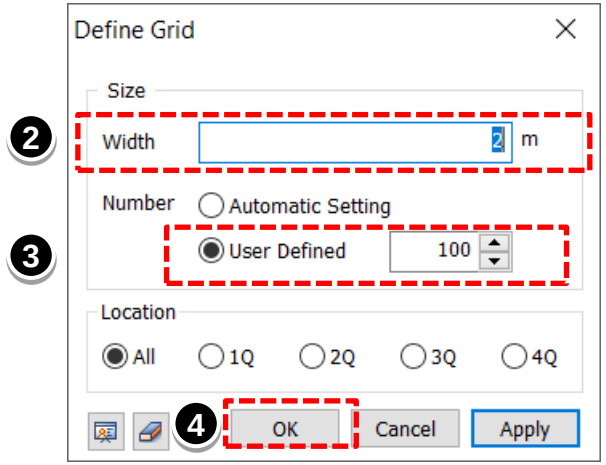
Step 04

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**.





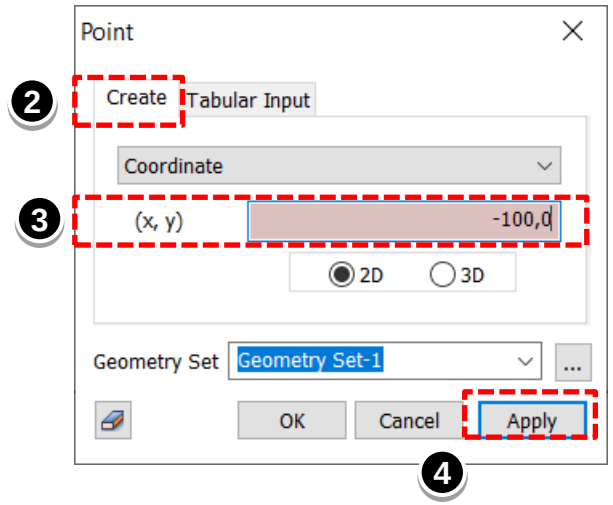
4-2 Geometric modelling-Creation of Points

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

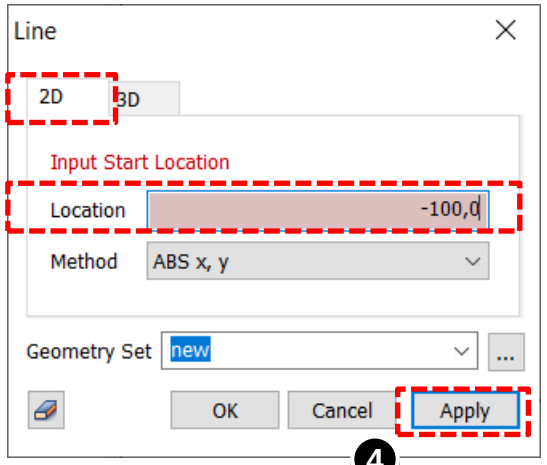
Procedure

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**.

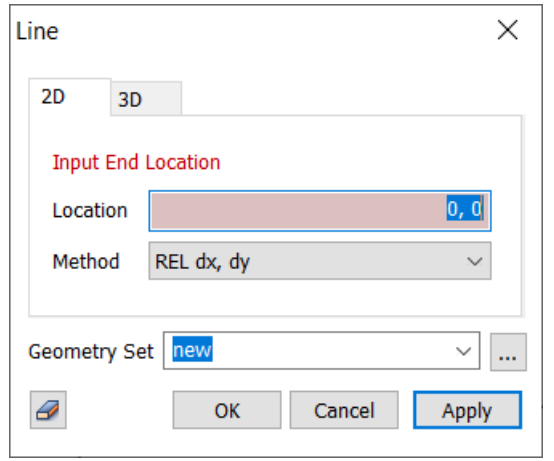
Repeat the same procedure for creating all the lines.

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

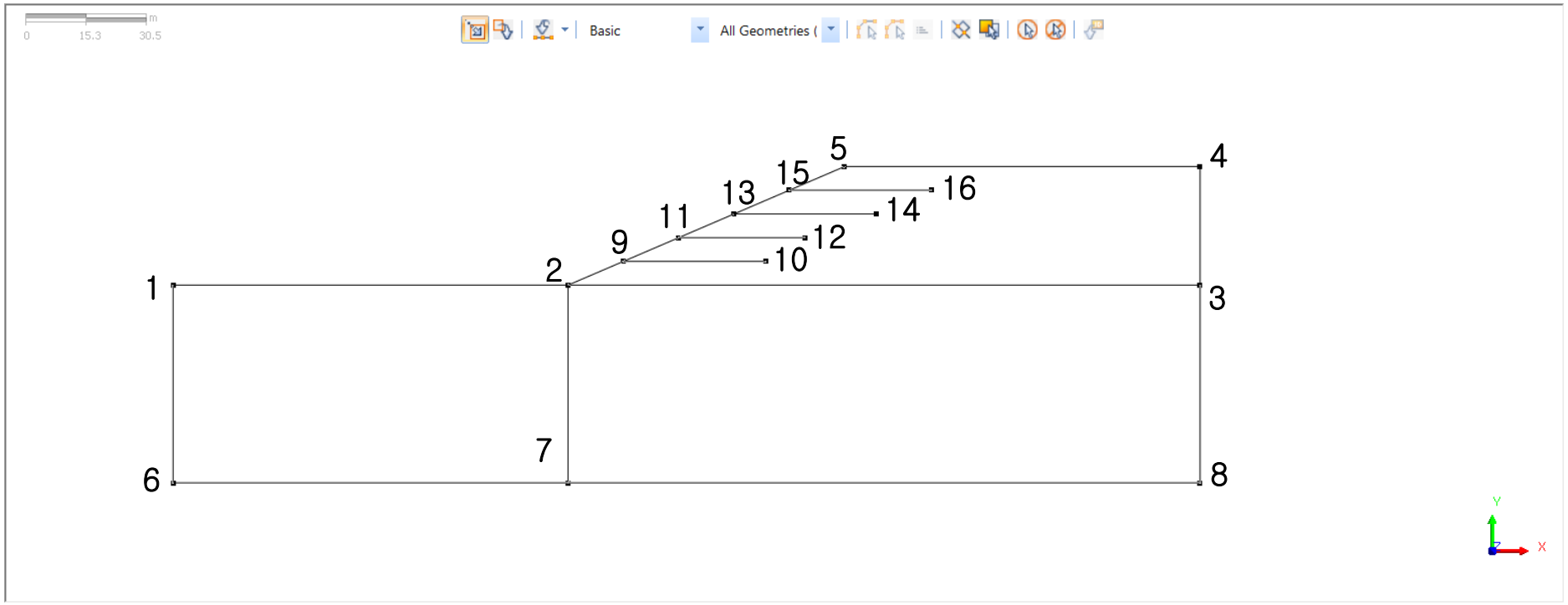


Step
04

4-4 Geometric modelling- Finished Geometry

Procedure

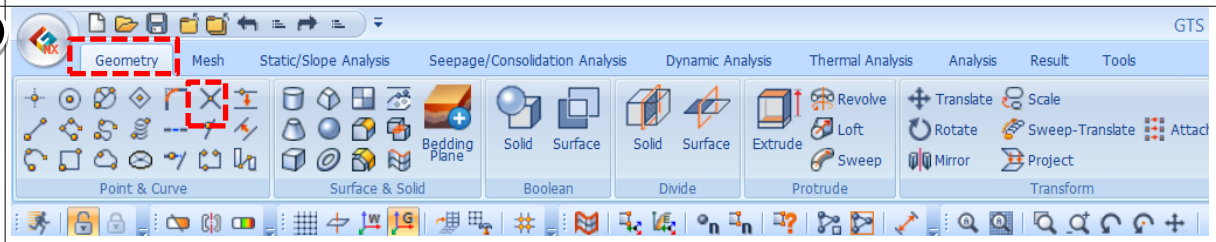
1. The points presented in Section 4-2 are shown in the picture here.
2. The lines connect the points as in the cross section shown here.

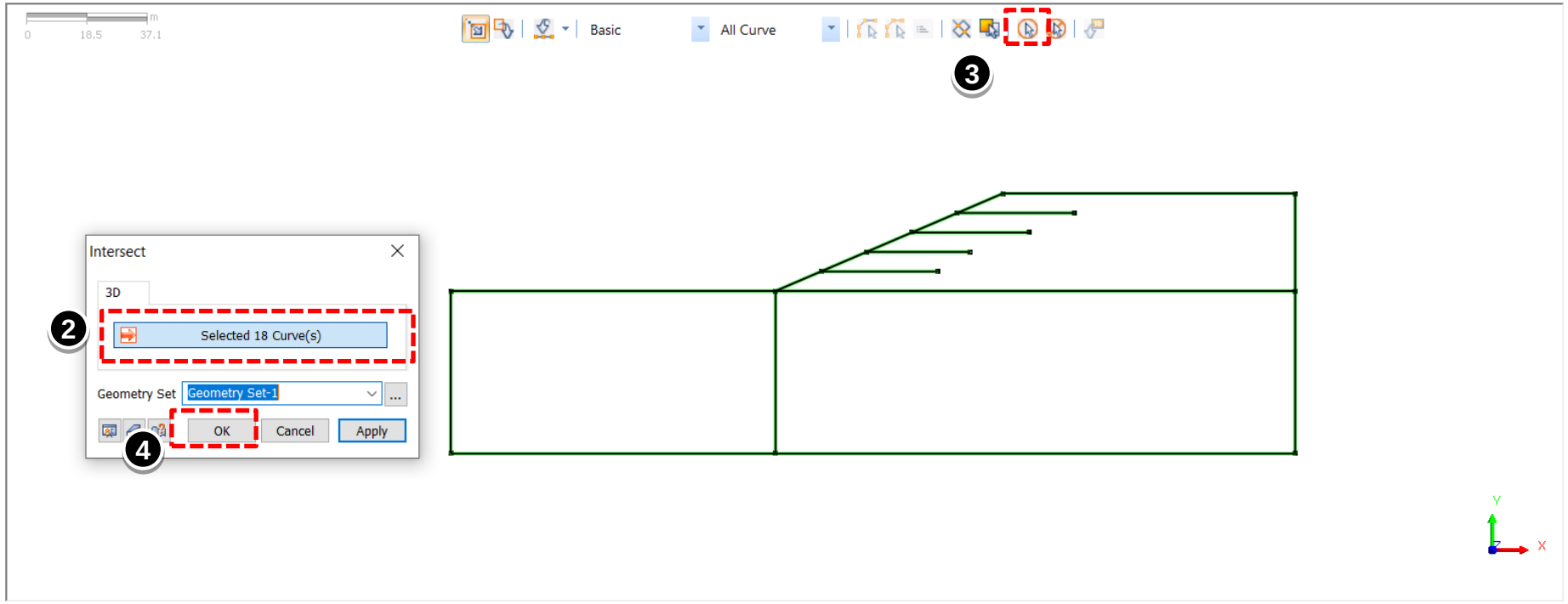


4-7 Geometric modelling- Connection & Intersection

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**.

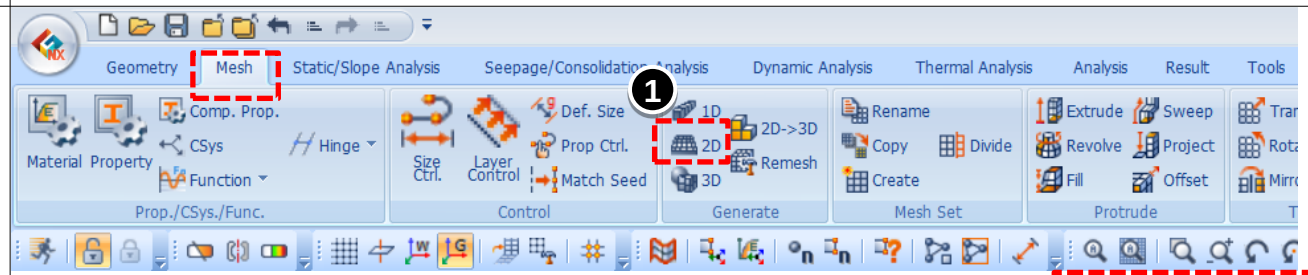


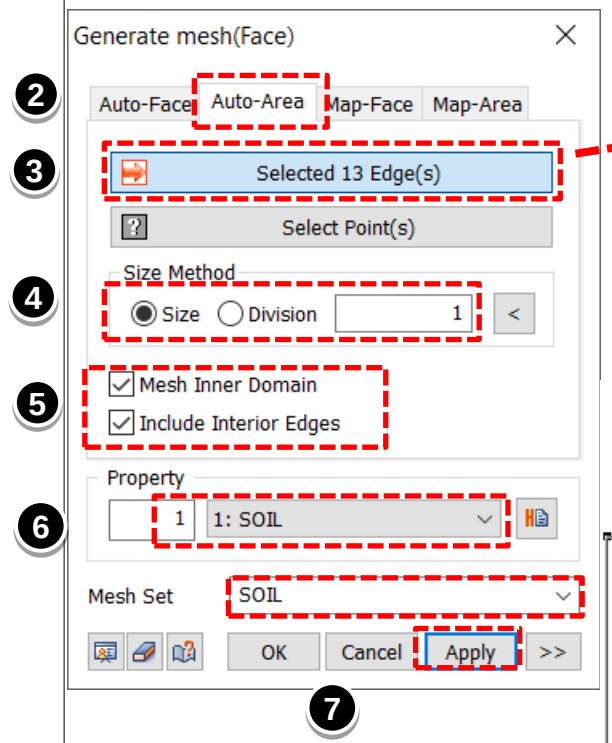


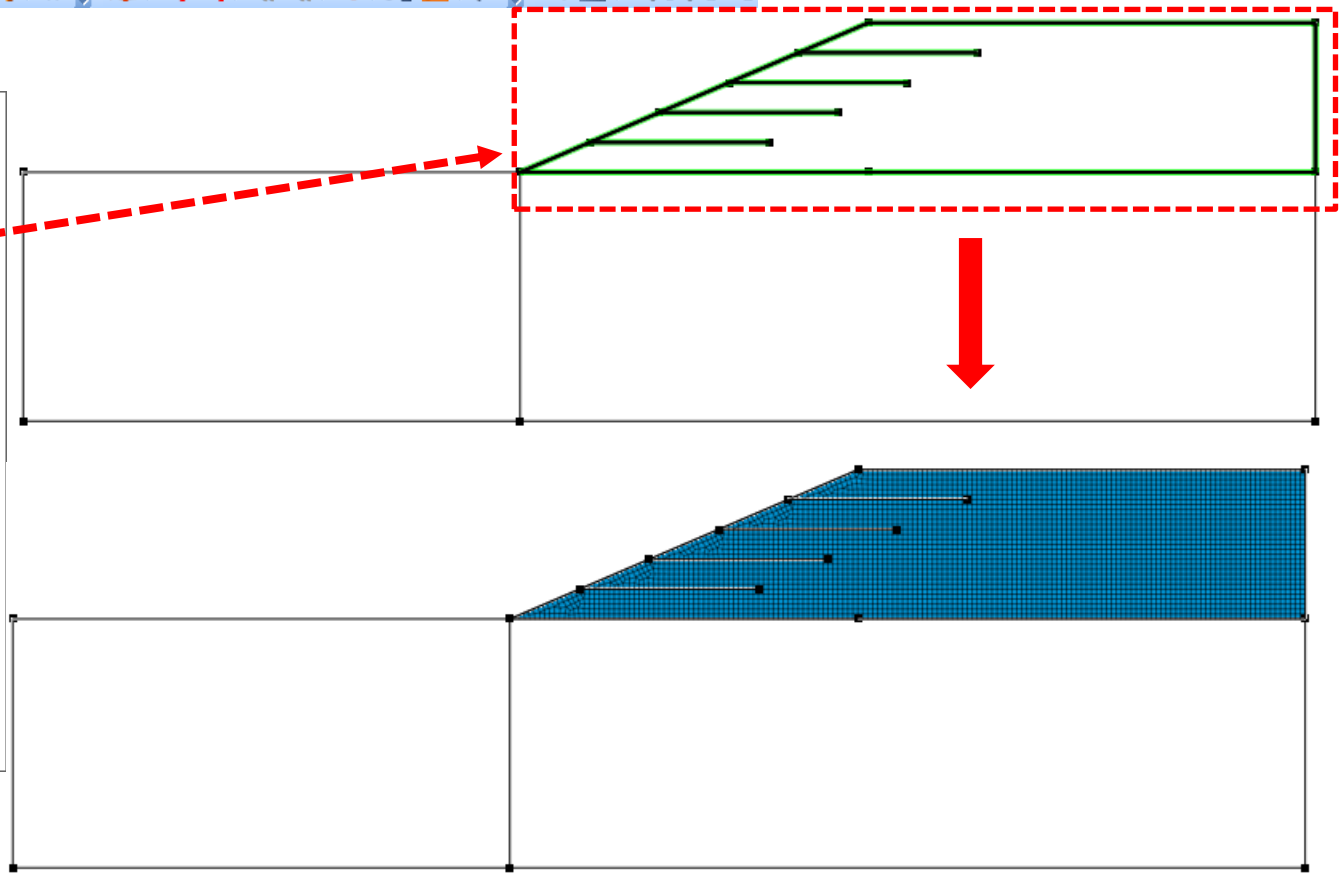
5-1 Meshing

Procedure

1. Go to Mesh > Generate > 2D
2. Select the lines as shown in the box that constitute parts of the embankment.
3. Enter the size as **1 m**.
4. Tick on the "**Mesh Inner Domain**".
5. The property is **SOIL.**
6. Give the Mesh set name "**SOIL**."
7. Click **Apply**.



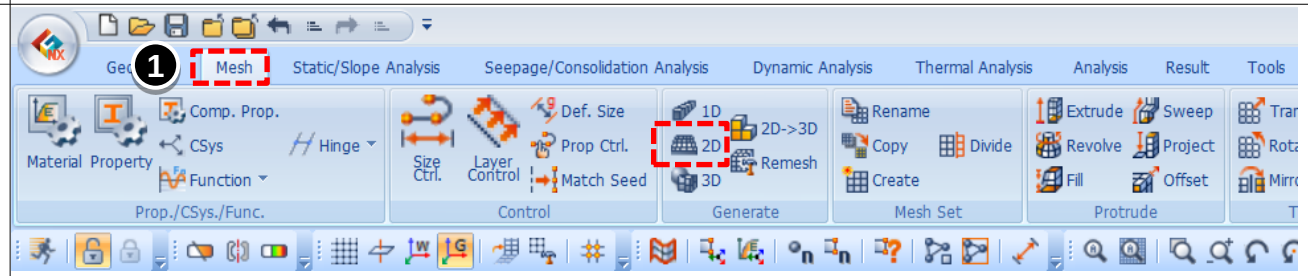


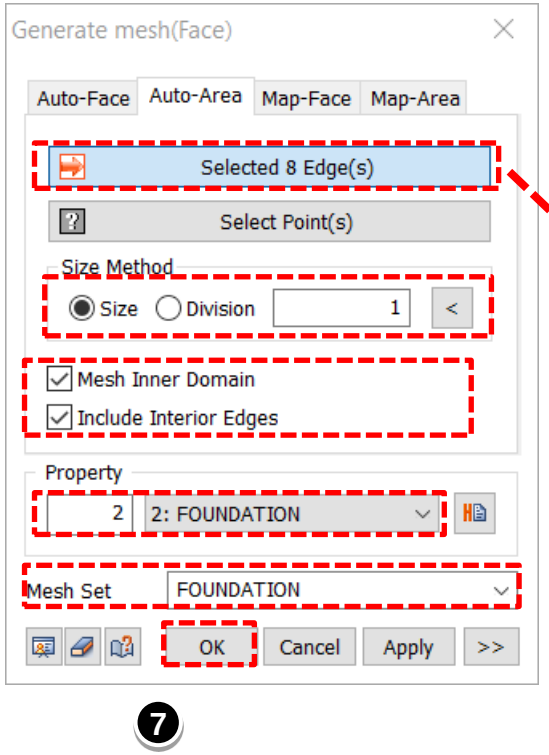


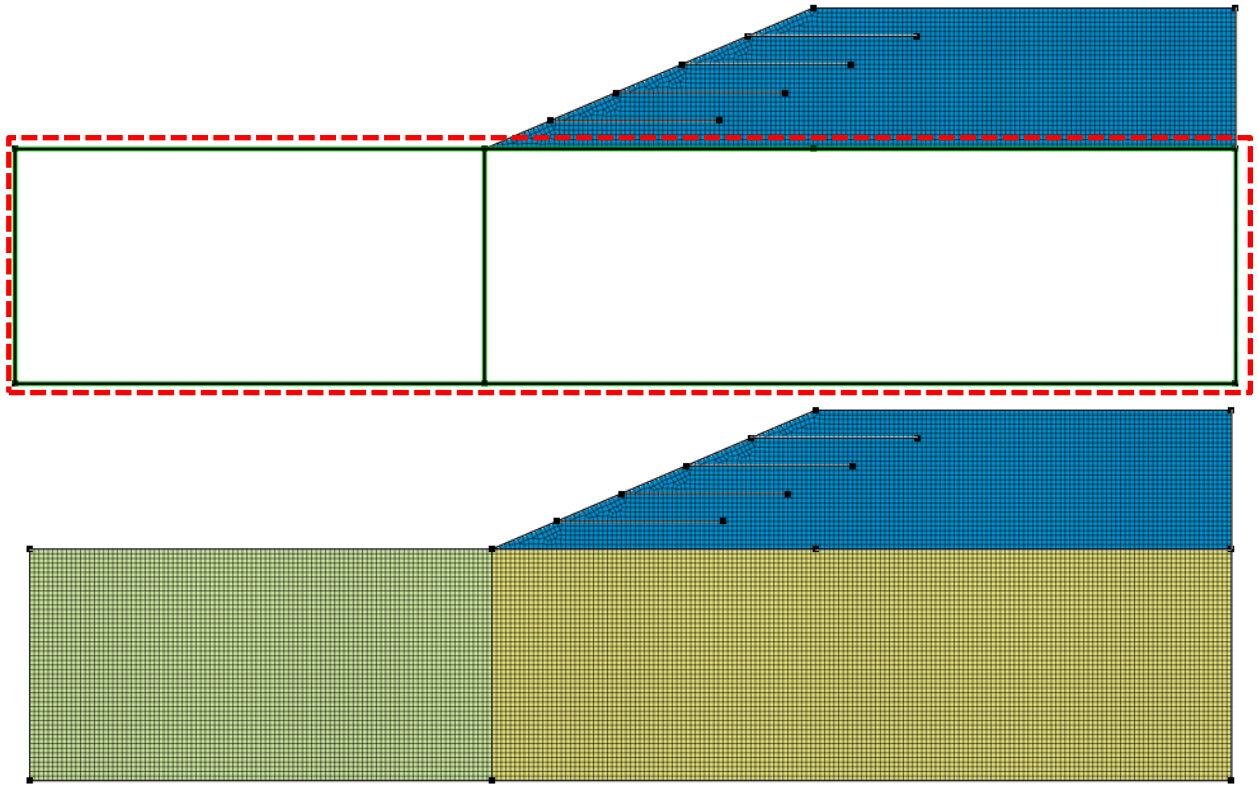
5-2 Meshing

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 **Foundation**.
6. Give the Mesh set name "**FOUNDATION**."
7. Click **OK**.



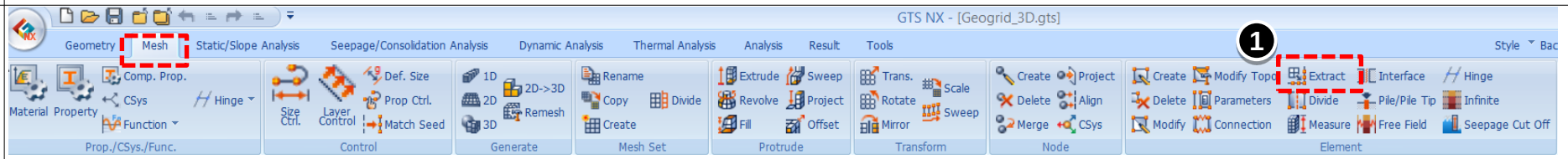


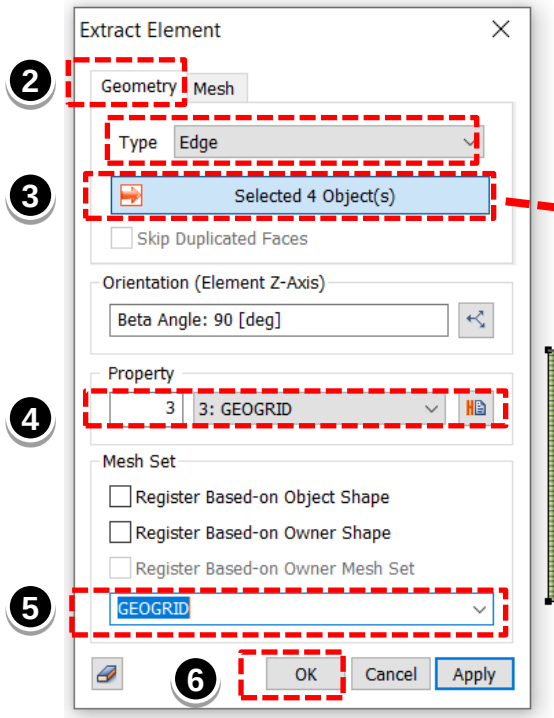


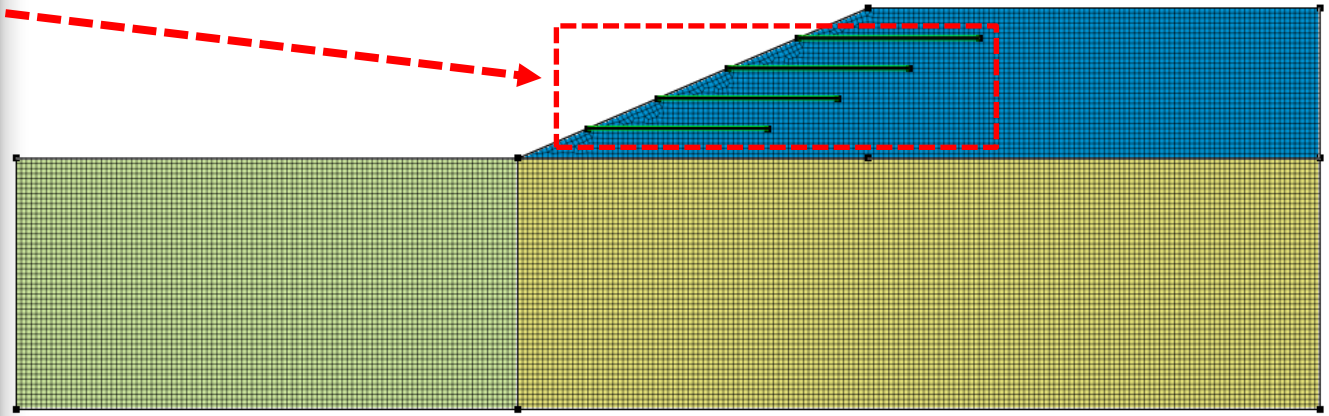
5-3 Meshing-Extraction of Geogrid Elements

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

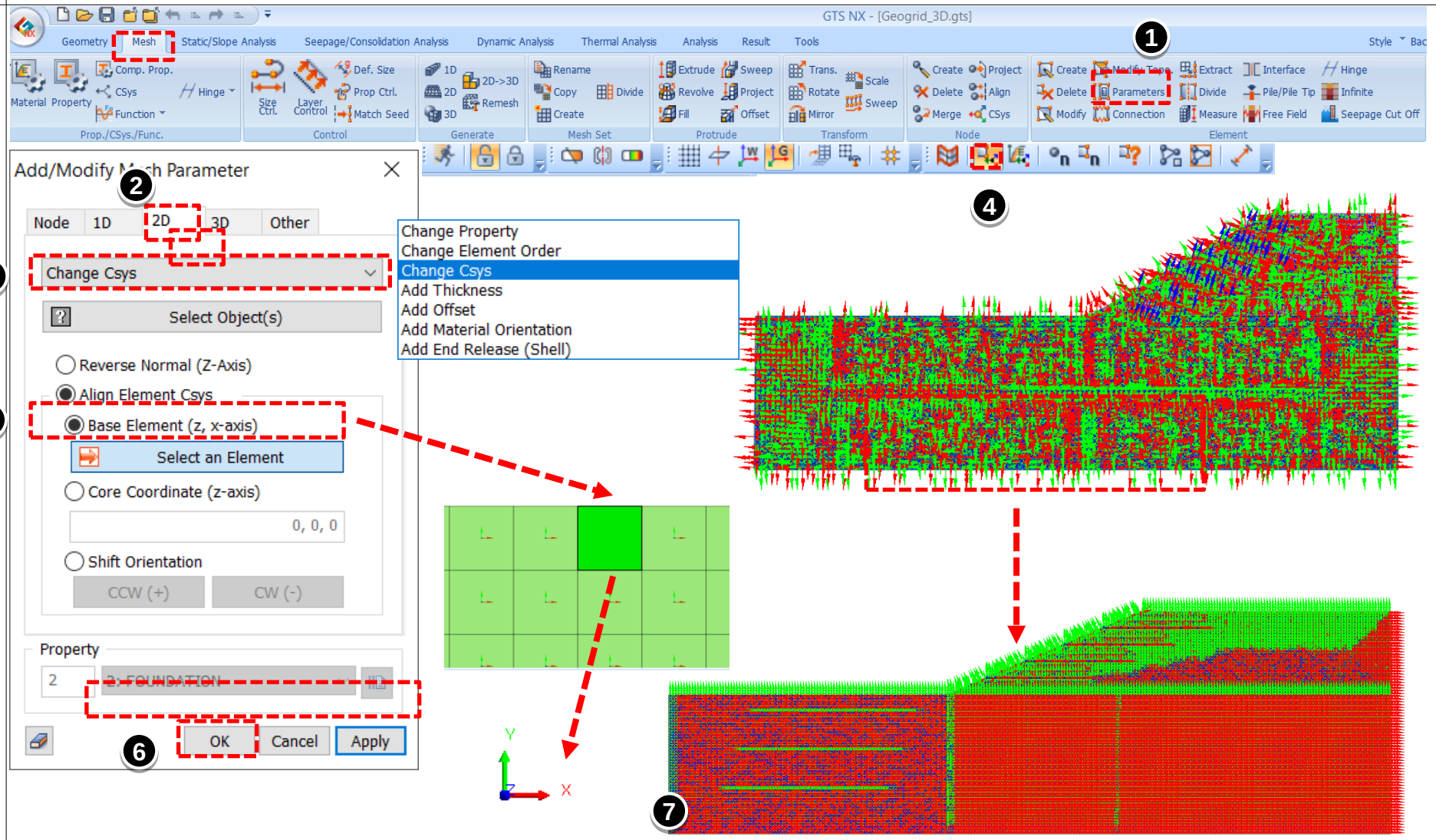
Step
05

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 Csys**.
4. Click on **Show Element Csys**.
5. Select the Base Element which Csys resembles the global csys as shown below.
6. Click **OK**.

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

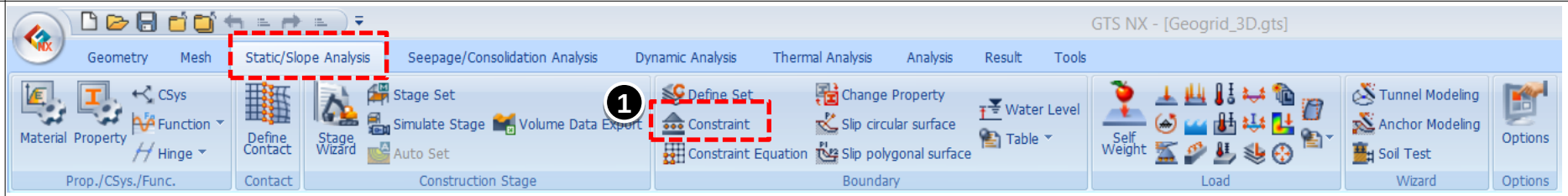


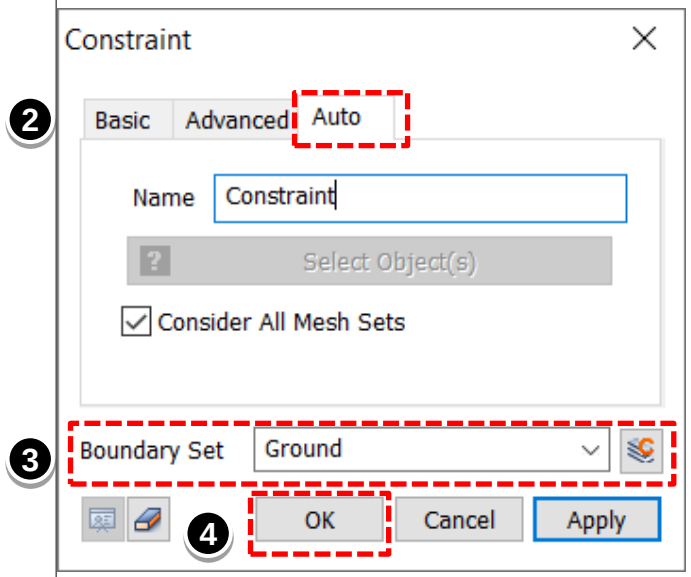
6-0 Boundary Condition

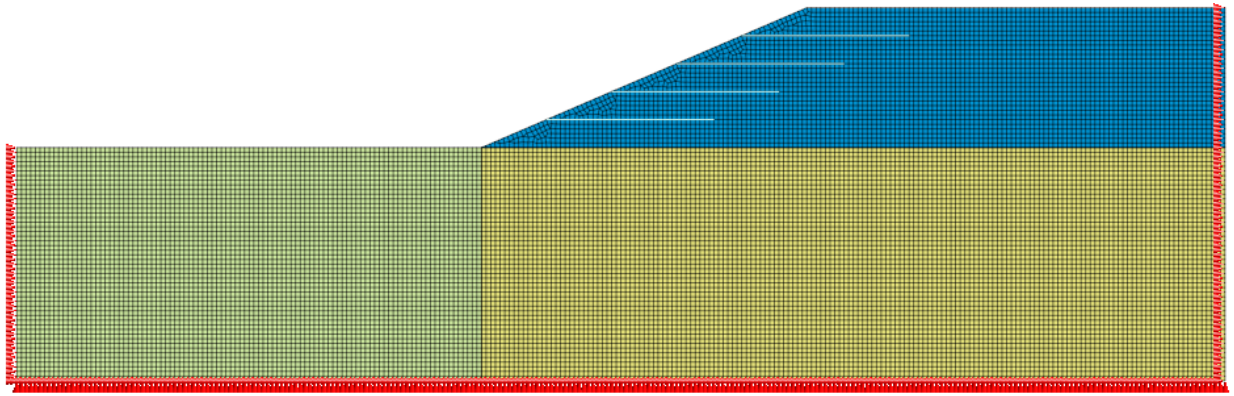
Procedure

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**

.







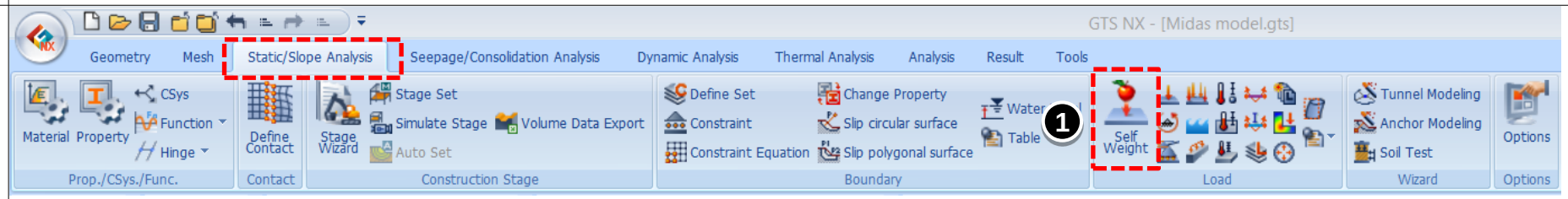
Step
07

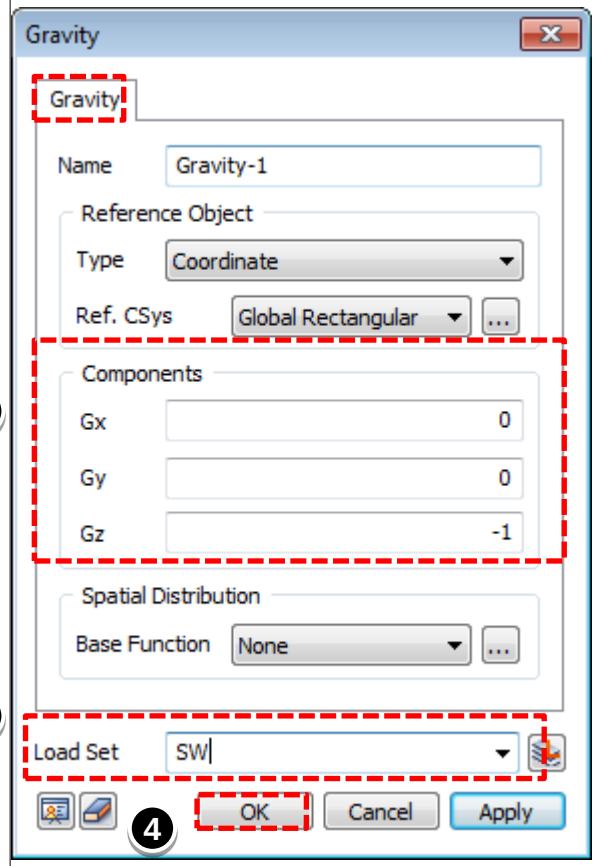
7-1 Load Definition-Self Weight

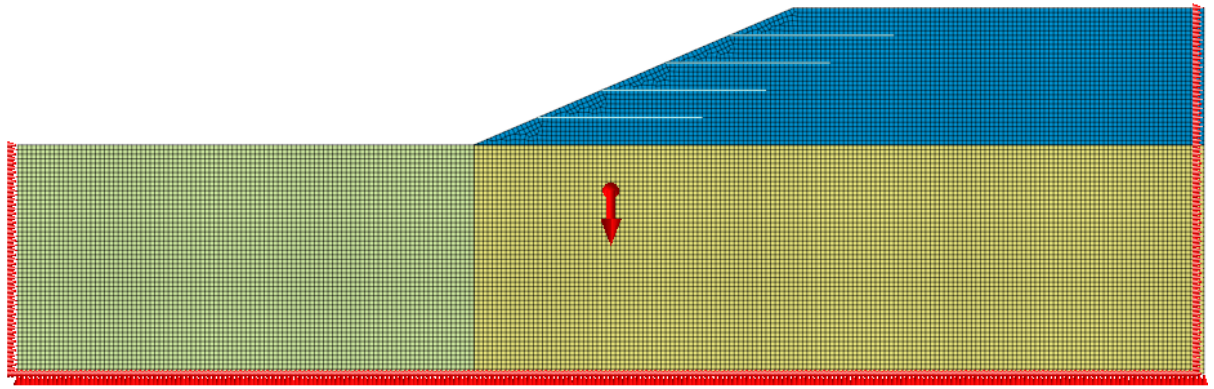
Procedure

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**

.



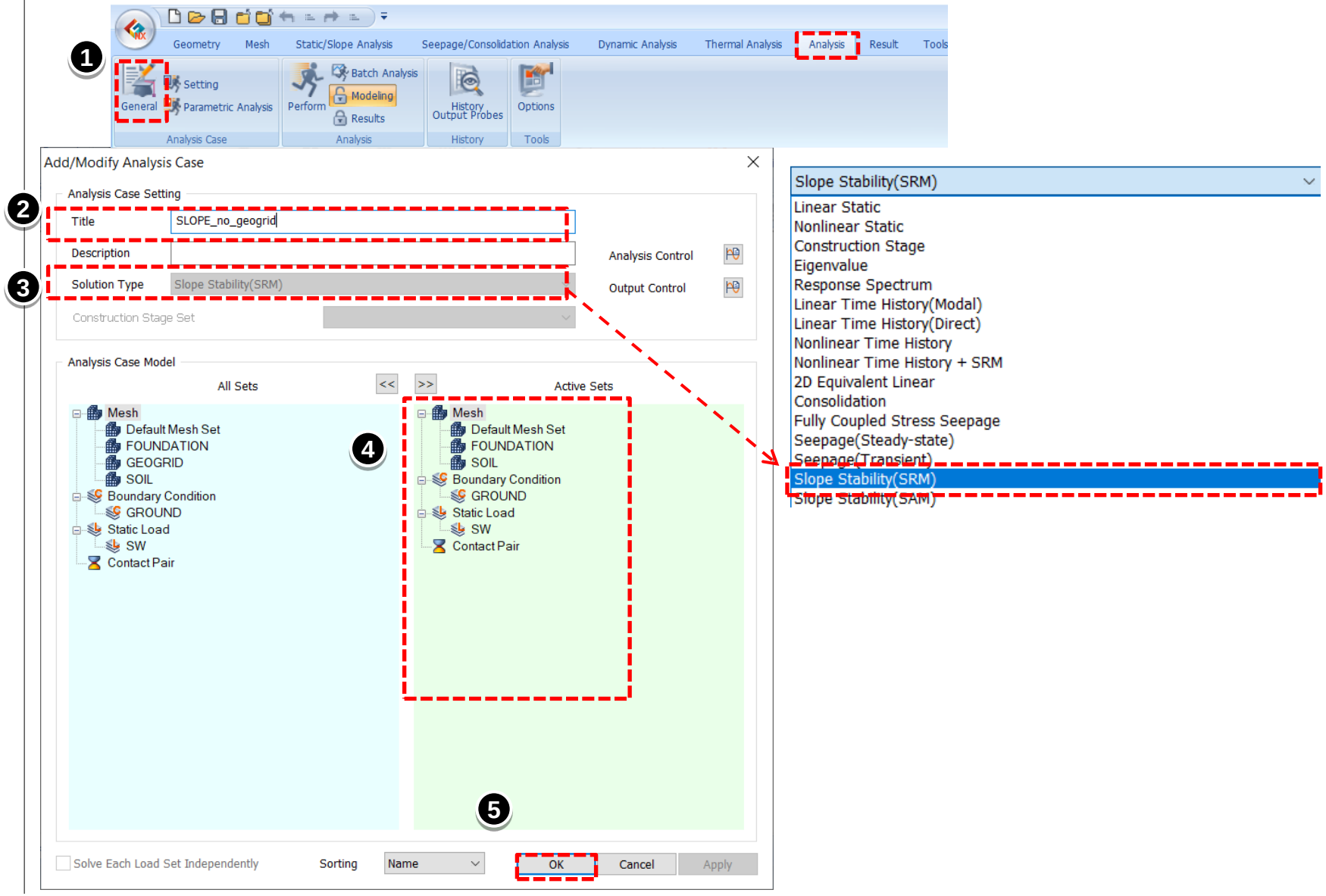




8-1 Analysis Case Definition

Procedure

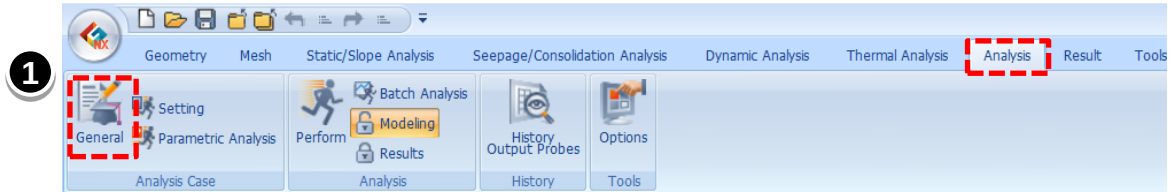
1. In the Analysis Tab, Click on General
2. Name the case as **Slope_No_Geogrid**.
3. In the Solution Type Drop-Down Menu, **Select Slope Stability (SRM).**
4. Drag and drop the all the **soil** and **embankment** mesh sets along with the **ground boundary** condition and the **self weight**.
5. Click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

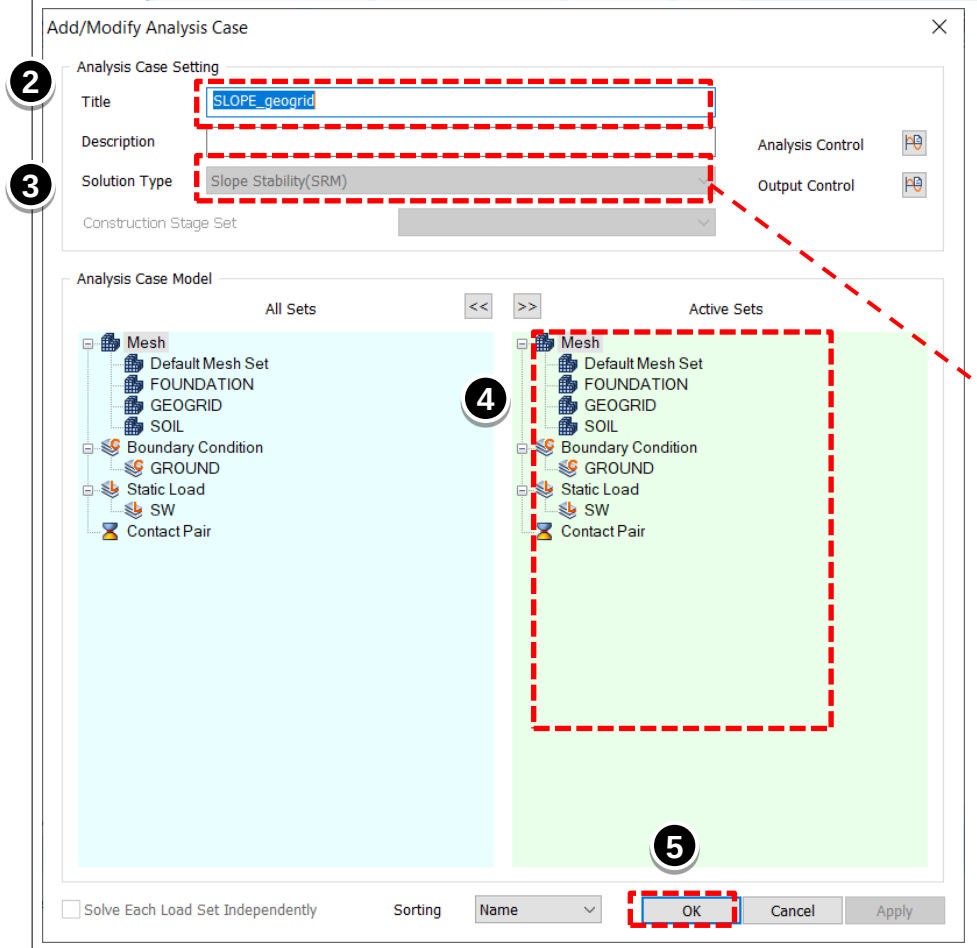


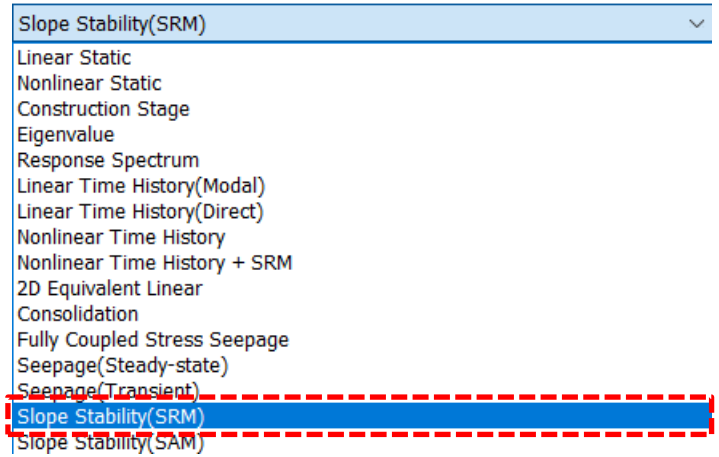
8-2 Analysis Case Definition

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**.



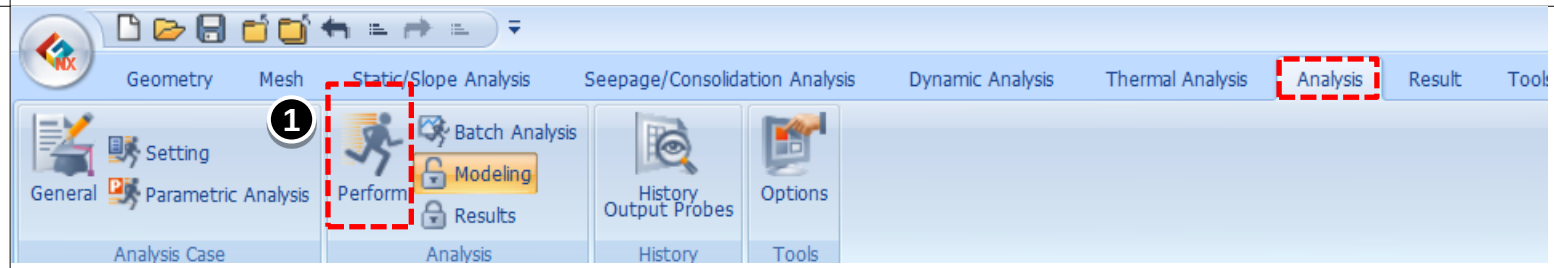


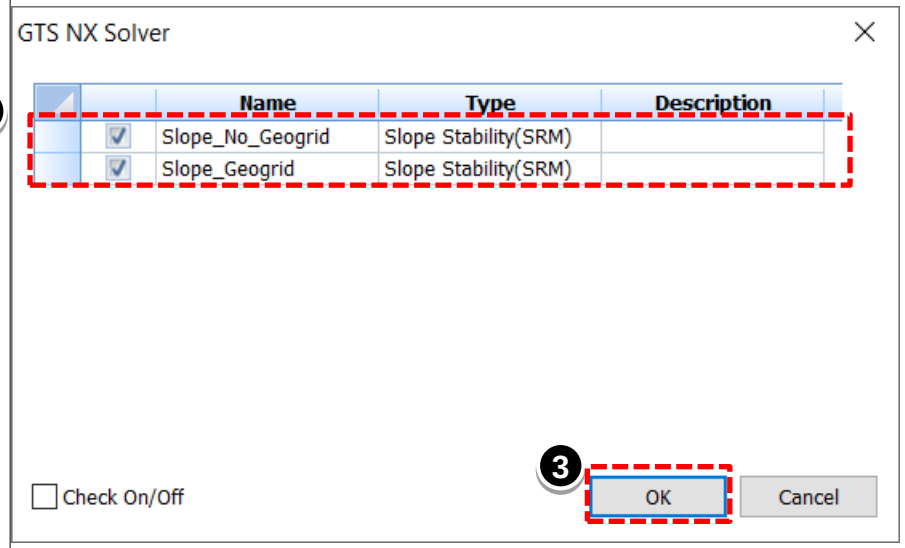


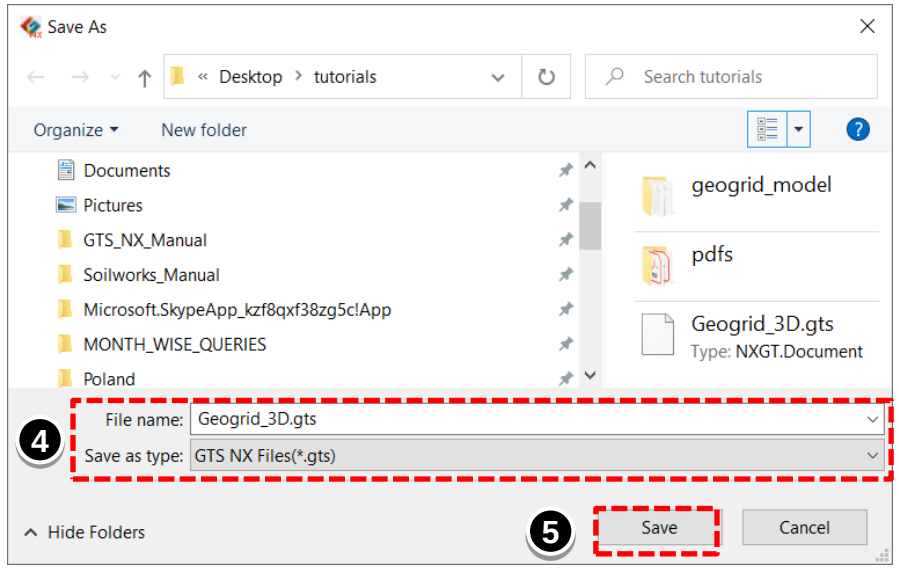
9-1 Analysis

Procedure

1. Go to Analysis > Perform
2. Select both the analysis cases.
3. Click **OK**
4. Give the File name
5. Save it at desired location







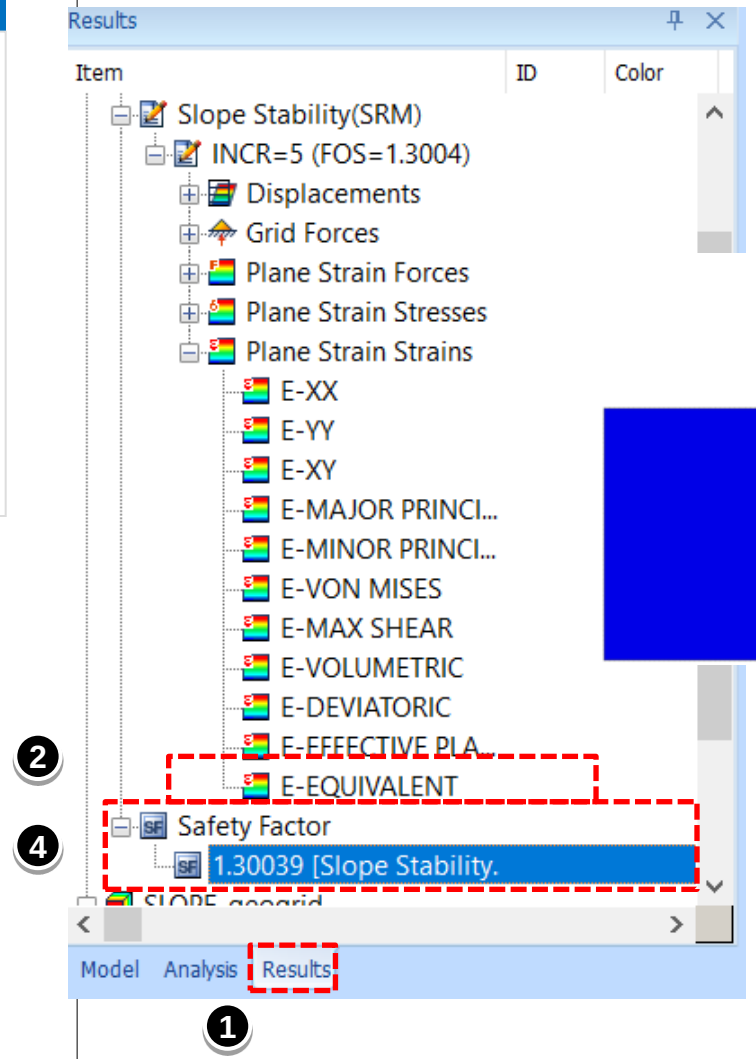
Step
10

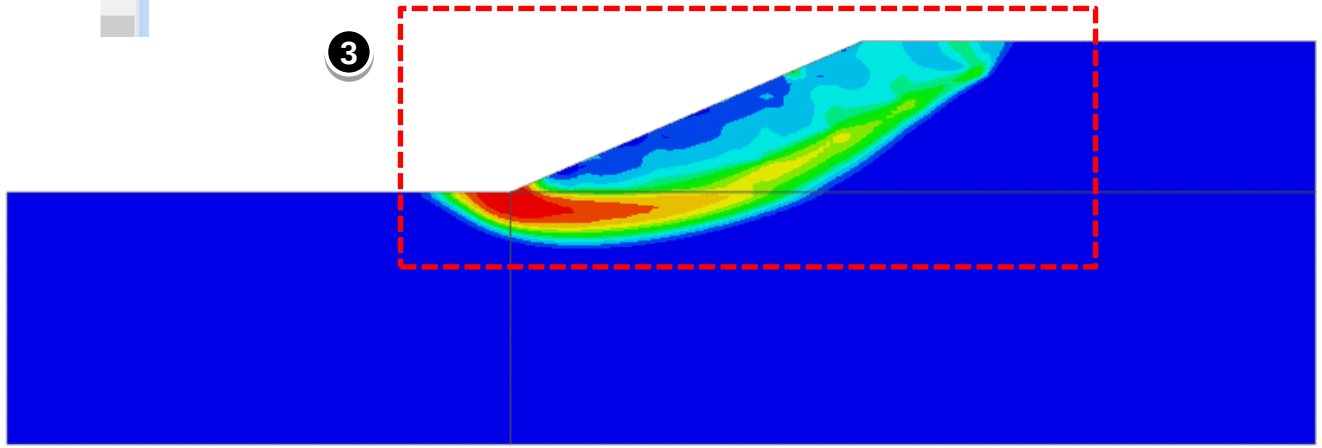
10-1 Slope Factor Of Safety Result (Without Geogrid)

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.





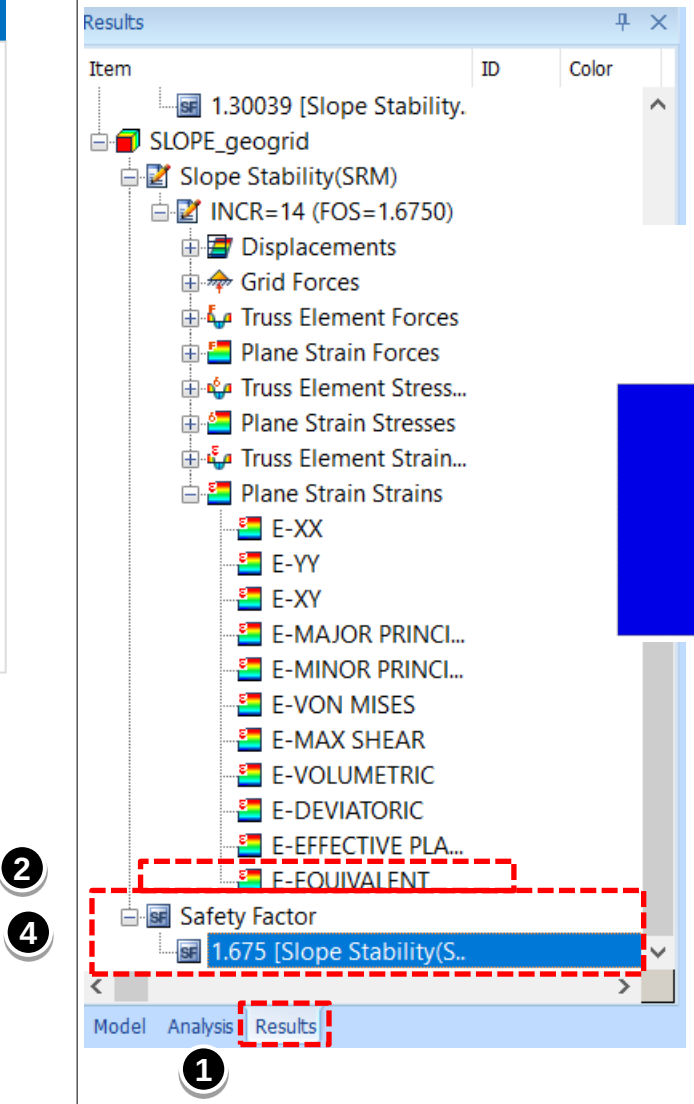
Step 10

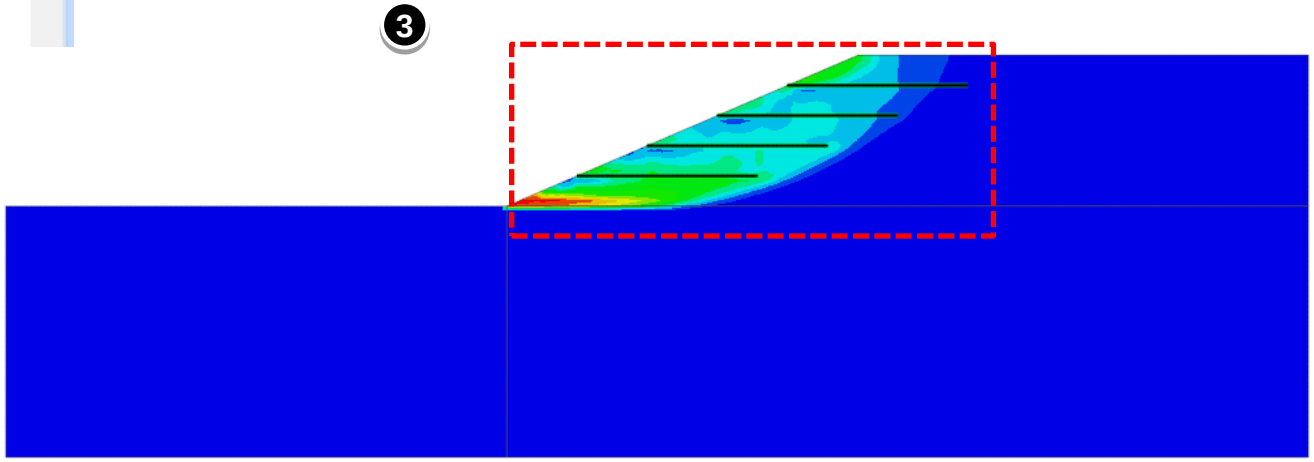
10-2 Slope Factor Of Safety Result (With Geogrid)

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.





CERTIFICATION TASK

- 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.