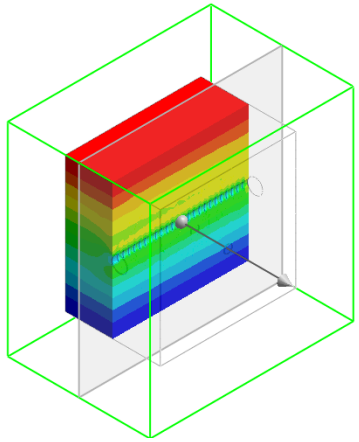
MIDAS *Technical Material*

Tutorial



3D SHIELD TBM TUNNEL

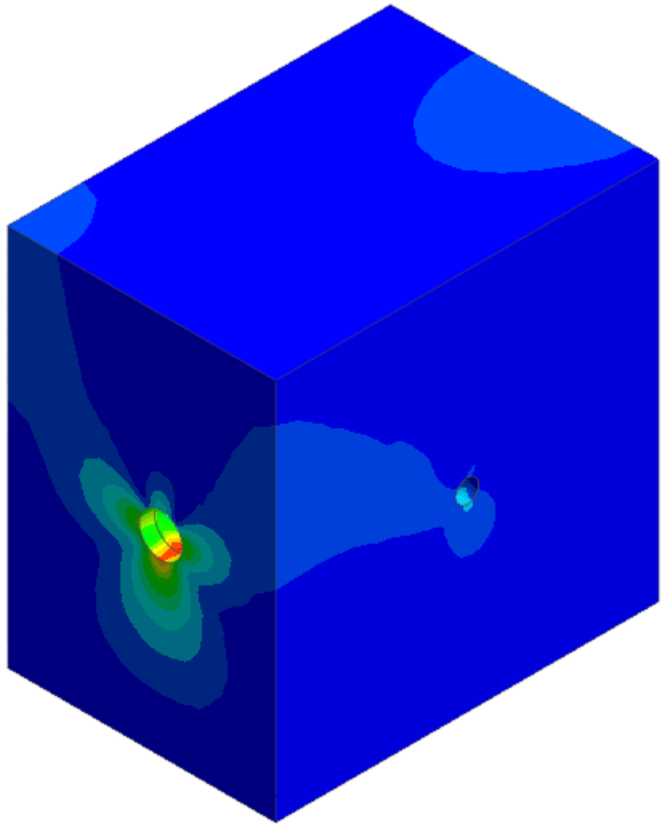
Contents

Numerical Model Design

Overview

This model tutorial is intended to show the user the method of simulating the advancement of a shield TBM and analyze the various forces and deformations associated with the design methodology.

Modelling



Contents

3D Shield Tunnel

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
SOFT ROCK	Mohr-Coulomb	250000	0.3	22	23	20	38

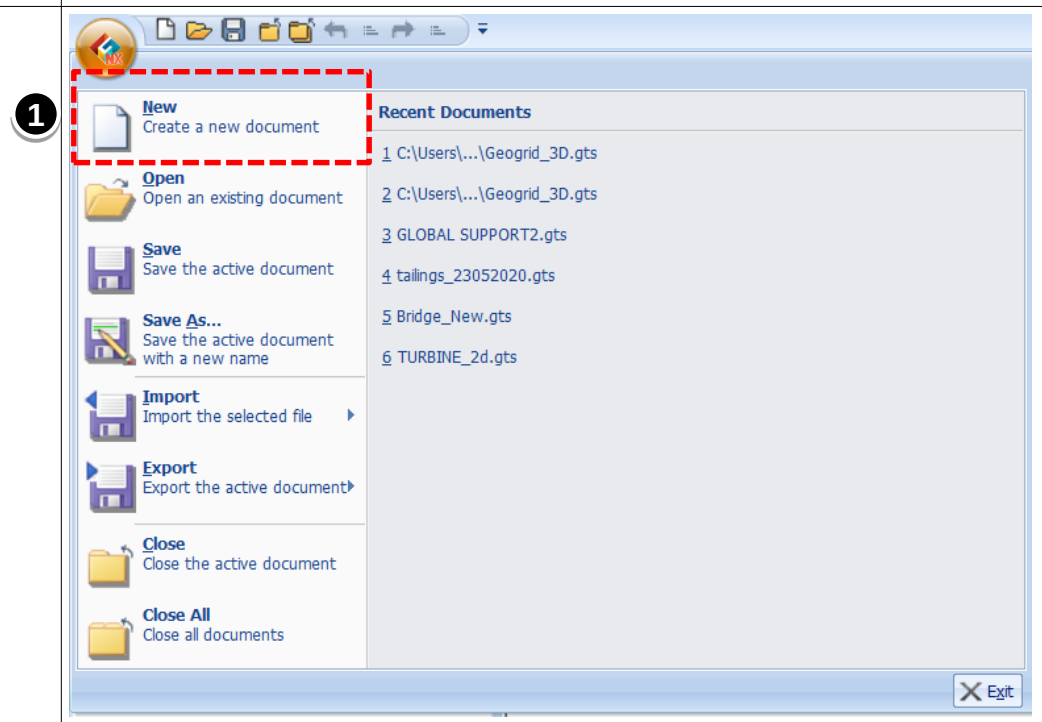
Name	Material Model	Modulus of Elasticity (kN/m^2)	Poisson's ratio	Unit Weight (kN/m^3)
LINERS	Elastic	2.7E+07	0.18	24
STEEL SHIELD	Elastic	2.10E+08	0.24	77
GROUT	Elastic	1.80E+07	0.3	23

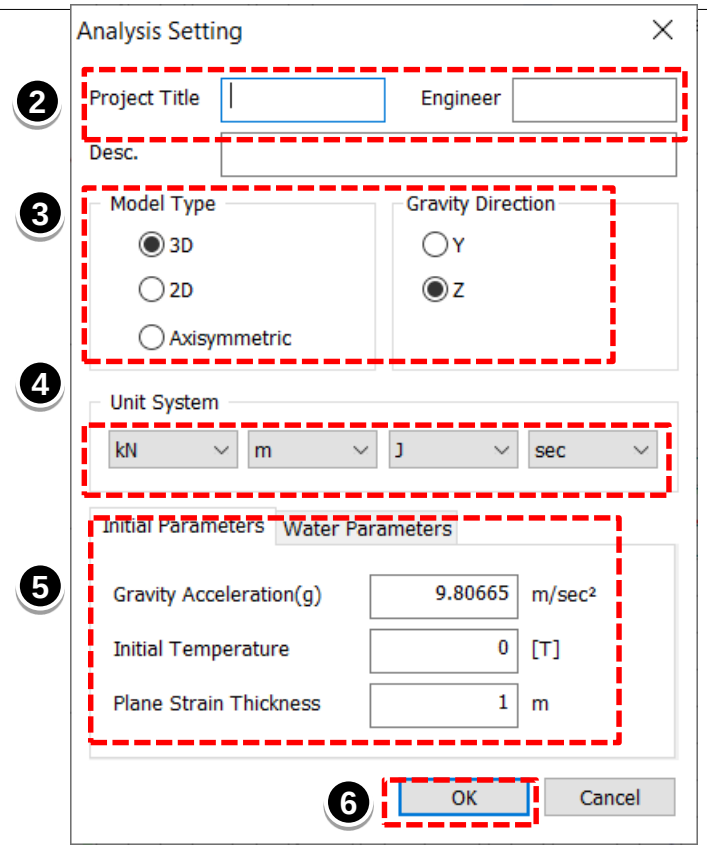
1-1 Initial Setting

Procedure

Starting Midas GTS NX

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



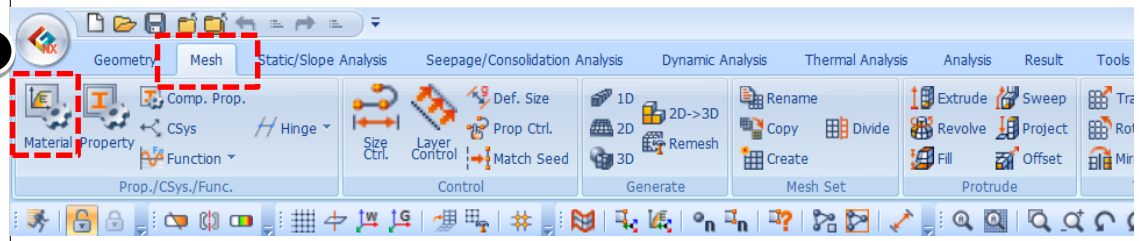


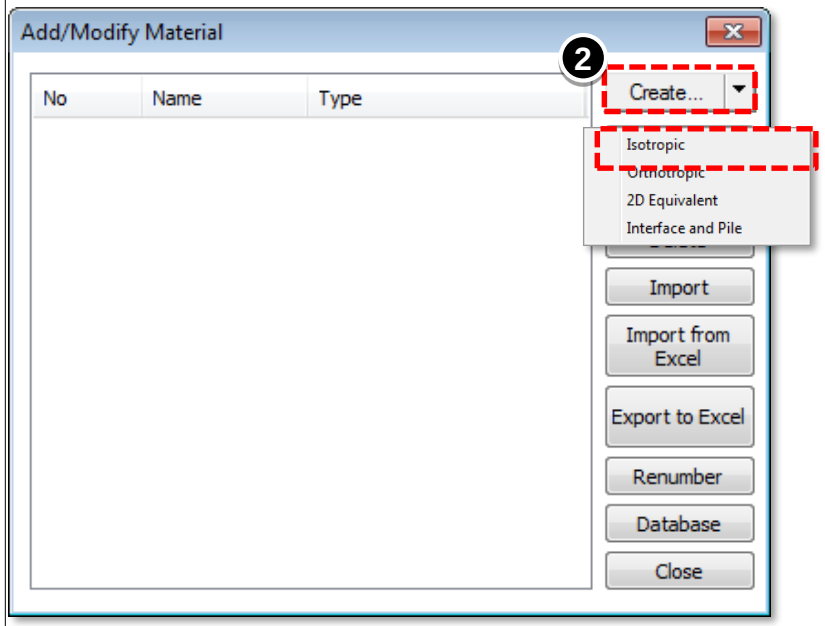
2-1 Defining Soil Material

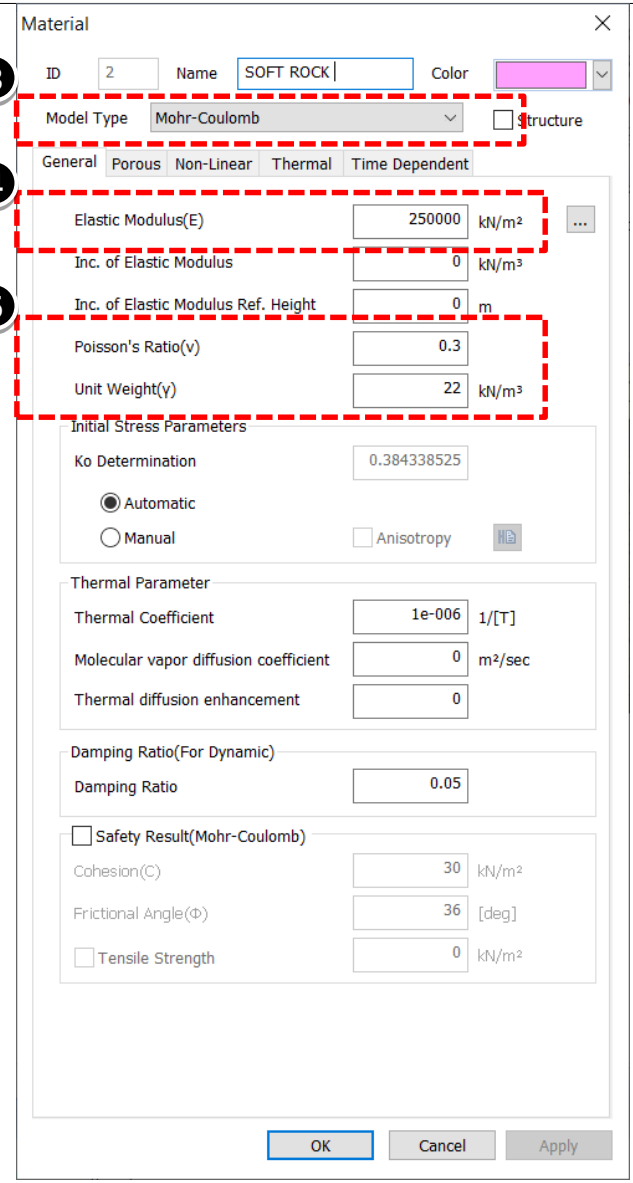
Procedure

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) =250000 kN/m²**
5. Enter **Poisson's ratio= 0.3**, **Unit weight = 22 kN/m³**.





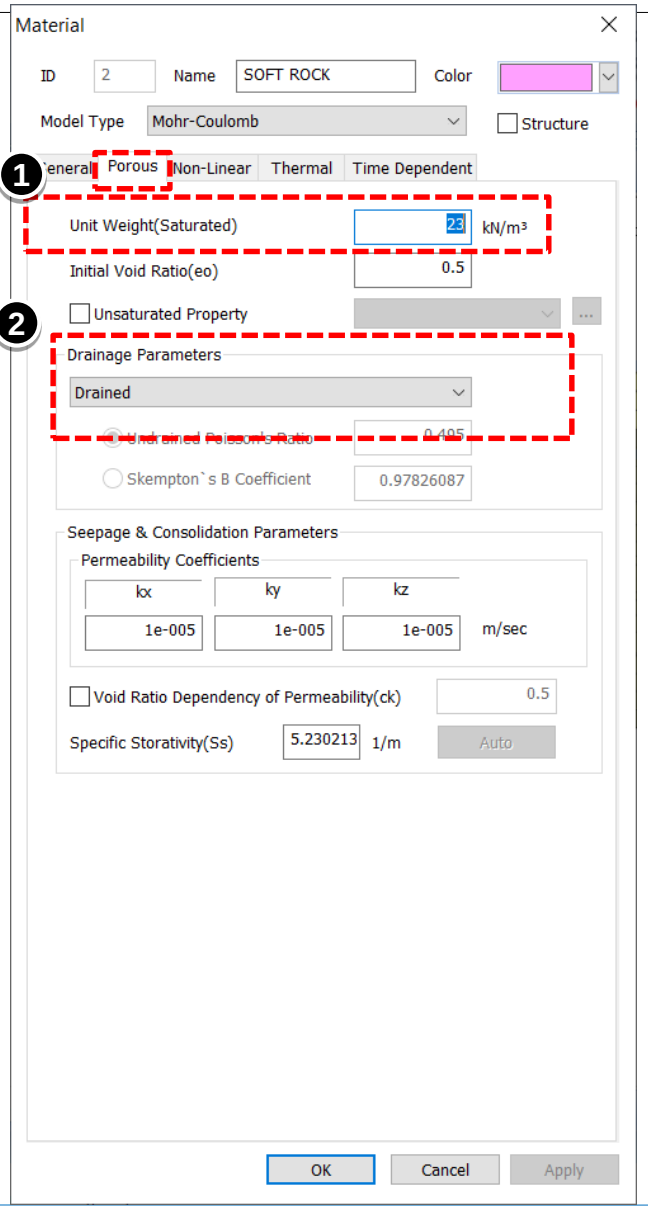


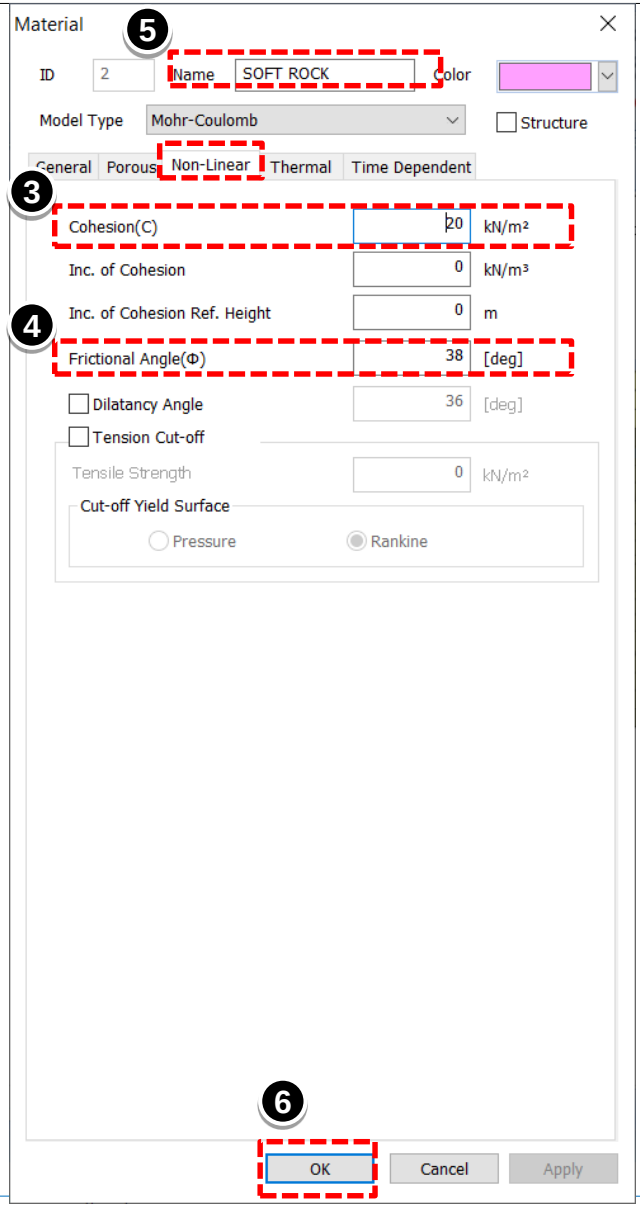
Step 02

2-2 Defining Soil Material

Procedure

1. Go to Porous Tab > **Enter Unit weight (saturated) =23 kN/m^3**
2. Keep Drainage Parameters as **Drained**
3. Go to Non-Linear Tab > Enter the value of **Cohesion = 20 kN/m^2**
4. Input the value of **Frictional angle = 38^0**
5. Enter the name as **SOFT ROCK**.
6. Click **OK**





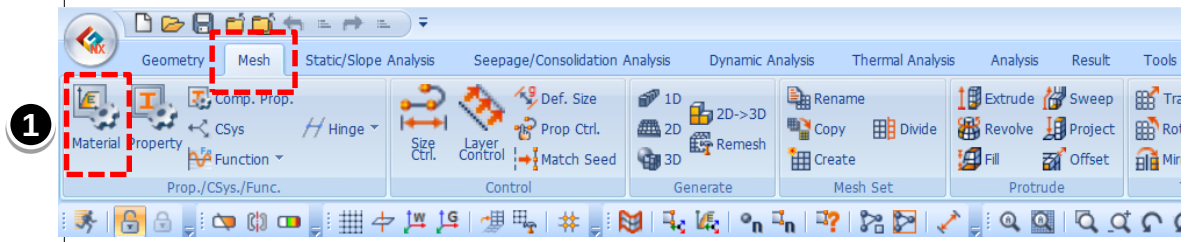
2-3 Defining Structural Materials

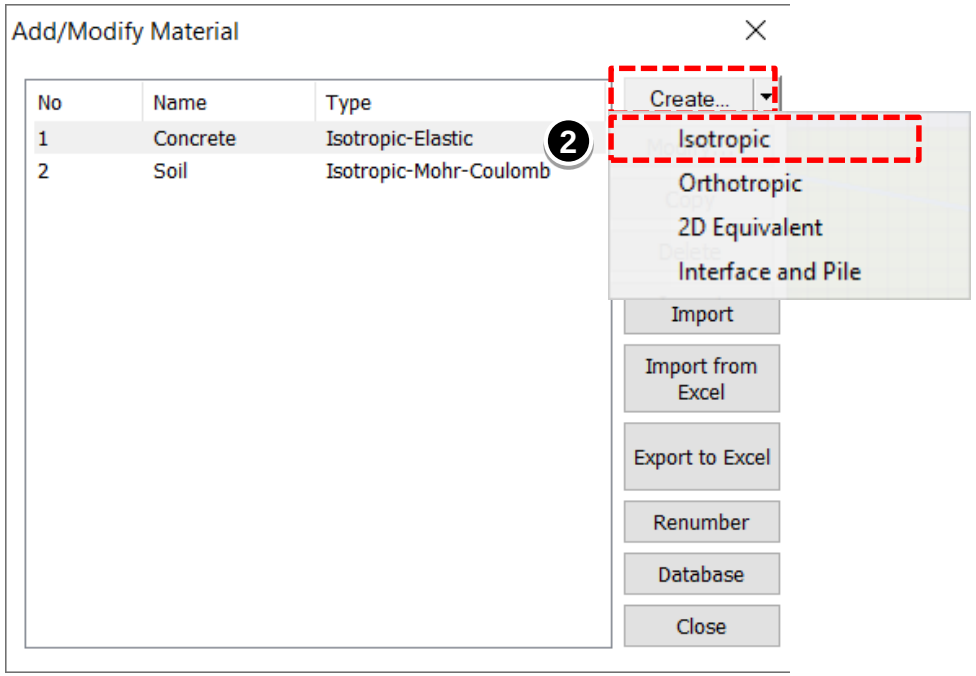
Procedure

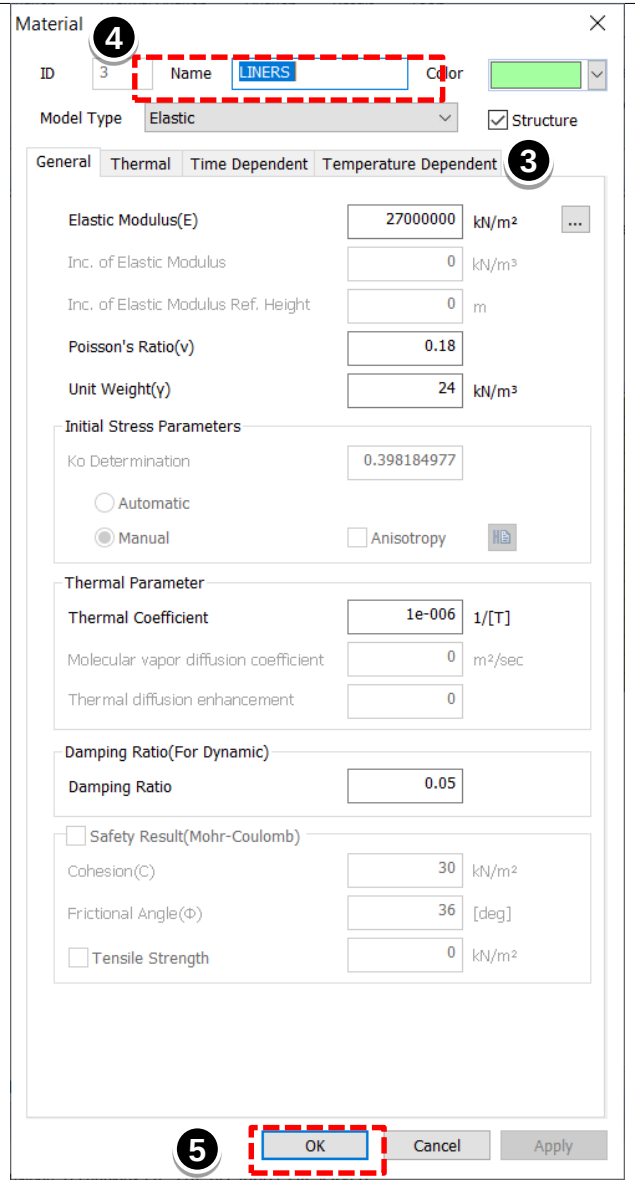
Defining Soil Materials

1. Go to Mesh > Material
2. Click on Create. Select **Isotropic.**
3. Select Model type as **ELASTIC &** Click on **STUCTURE** BOX**.**
4. Name it as **LINERS.**
5. Click **OK.**

SIMILARLY CREATE MATERIALS FOR STEEL SHIELD & GROUT.







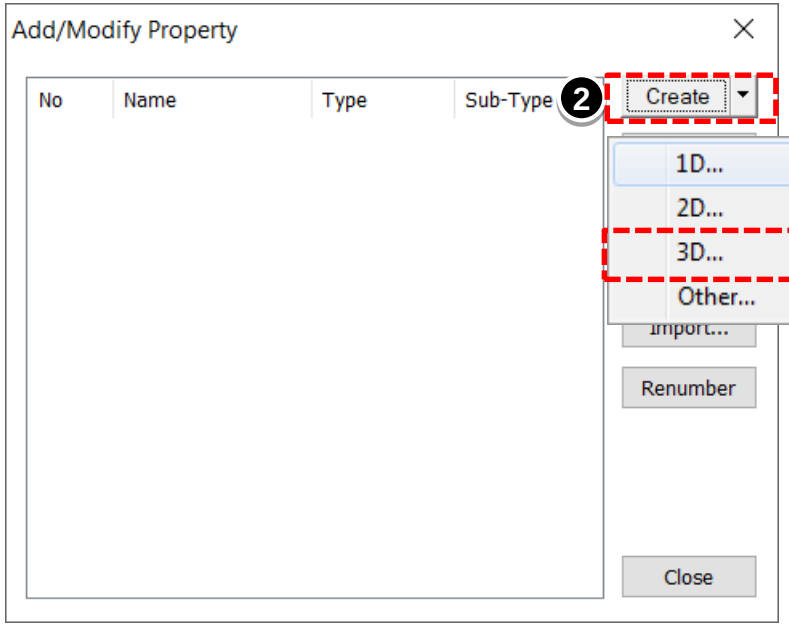
3-1 Defining 3D Property (SOIL)

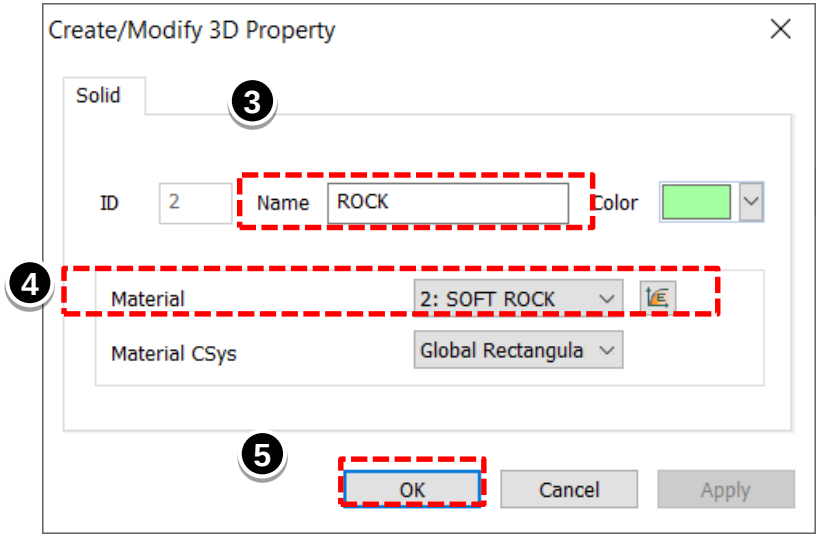
Procedure

Defining Property

1. Go to Mesh>Click on Property
2. Click on Create. Select **3D**
3. Name it as **SOFT ROCK**
4. Select **SOFT ROCK** from Material drop-down menu.
5. Click **OK.**

SIMILARLY CREATE 3D PROPERTY FOR LINERS.





Step 03

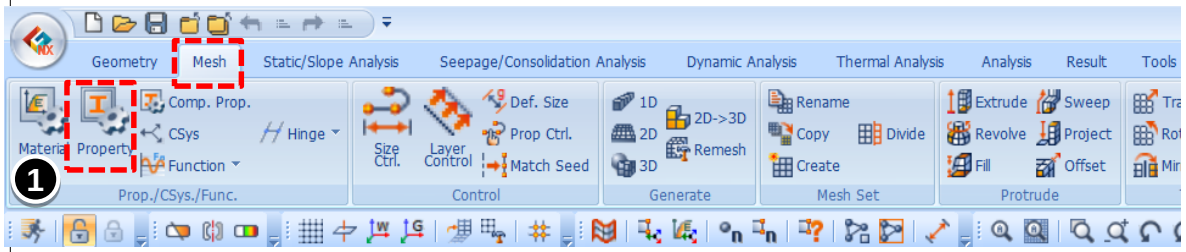
3-2 Defining 2D Shield Property

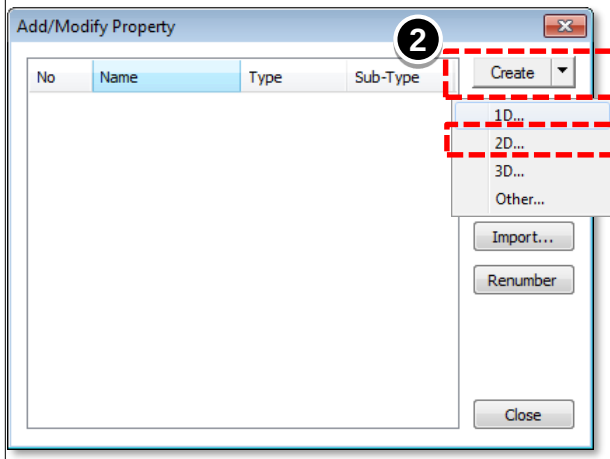
Procedure

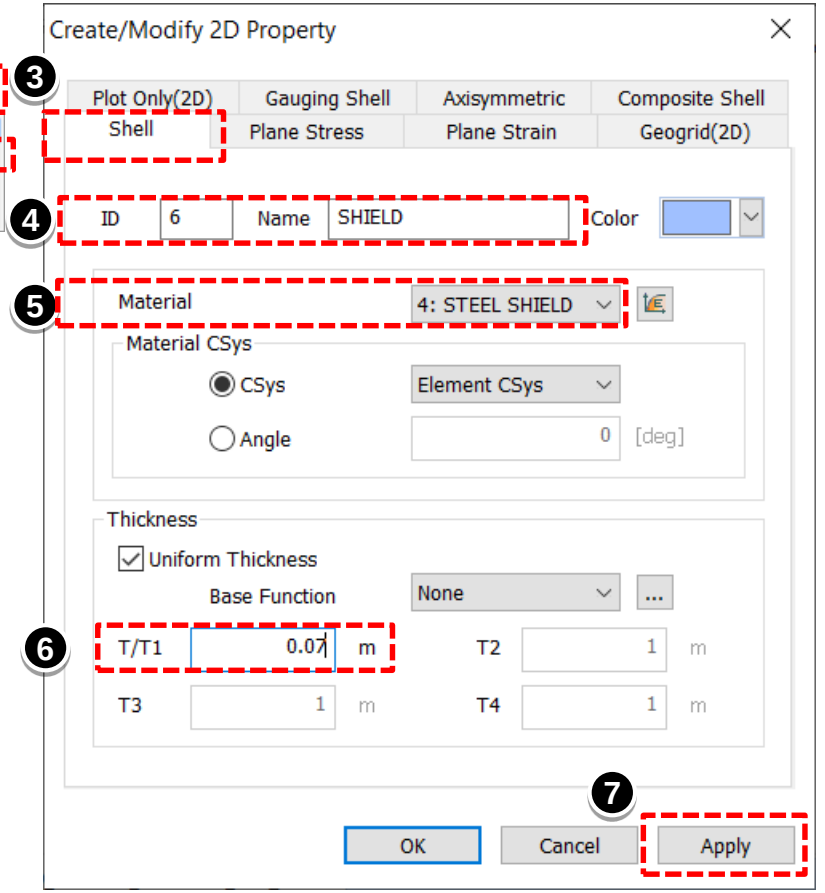
Defining Property

1. Go to Mesh>Click on Property
2. Click on Create. Select **2D**
3. Click on **Shell.**
4. Name it as **SHIELD.**
5. Select material as **STEEL SHIELD.**
6. Enter T/T1 as **0.07 m.**
7. **Click Apply.**

SIMILARLY CREATE SHELL PROPERTY AND NAME IT AS GROUT, SELECT MATERIAL AS GROUT AND ENTER T/T1 AS 0.07 m AND CLICK OK.





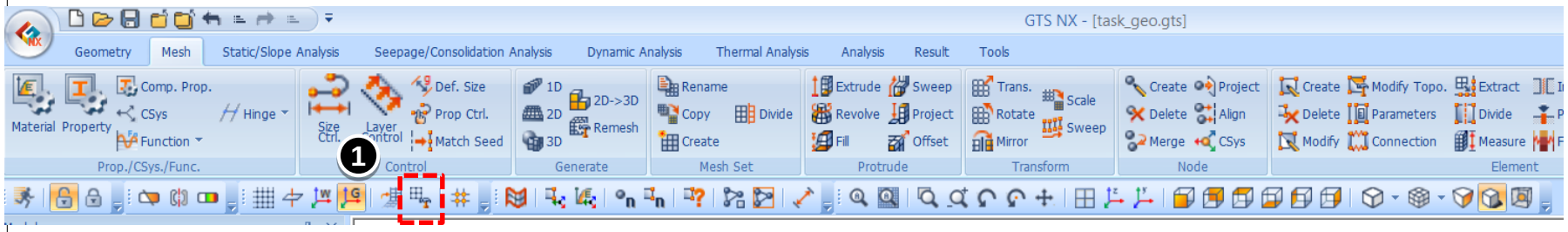


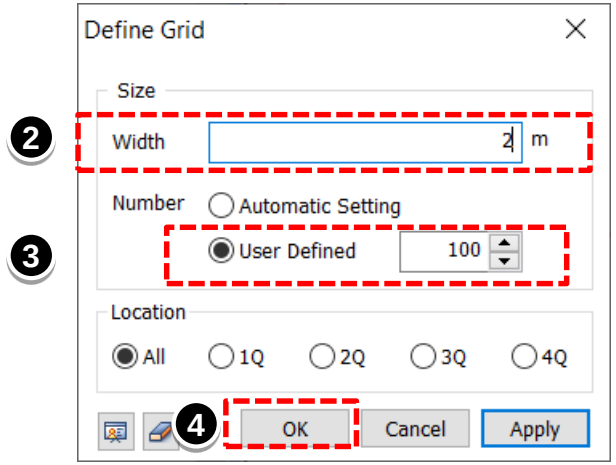
4-1 Geometric Modelling- Start File & Redefining Grid Size

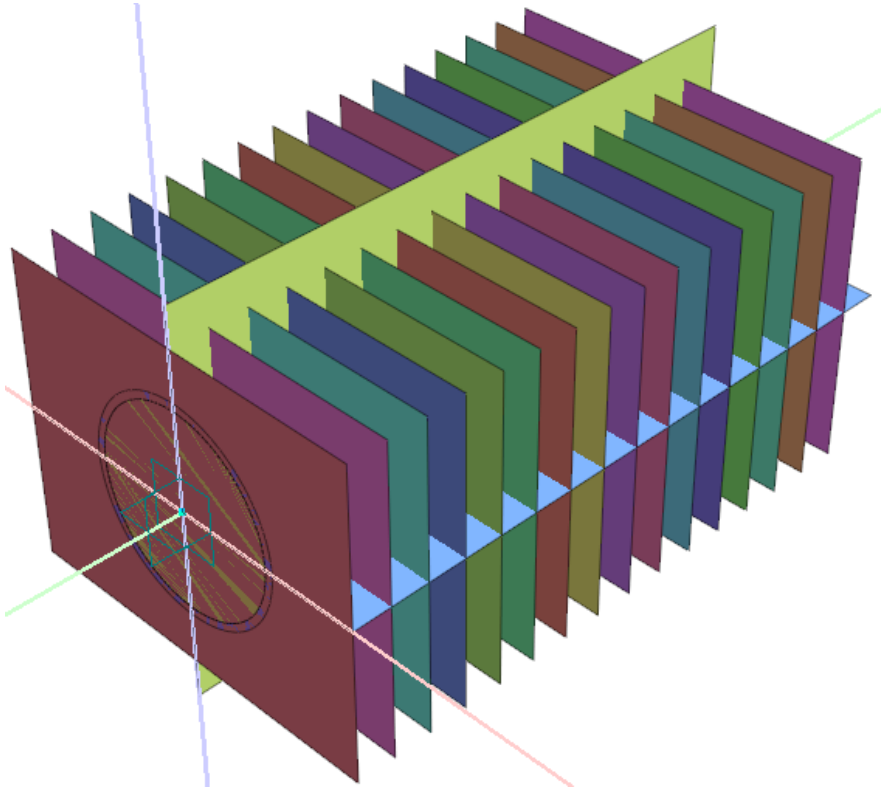
Procedure

Open the Start File **“shield_start”**

1. Go to Grid Size.
2. Enter width as **2m.**
3. Click **user defined** and enter **100**.
4. Click **OK**.





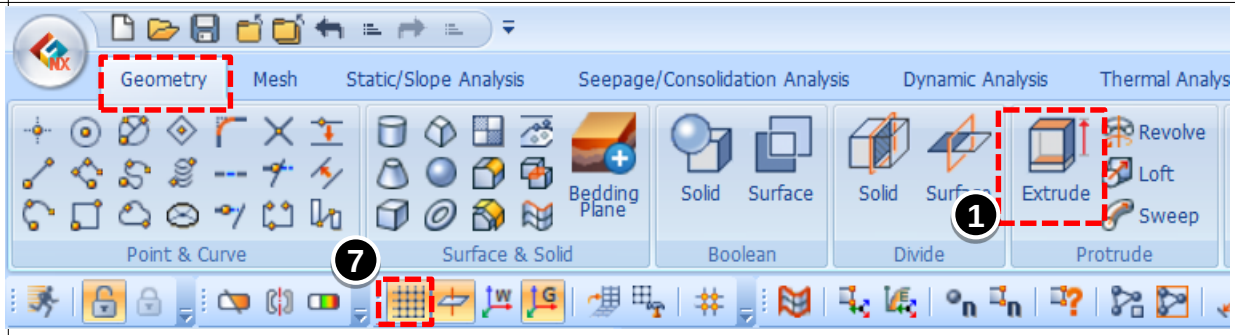


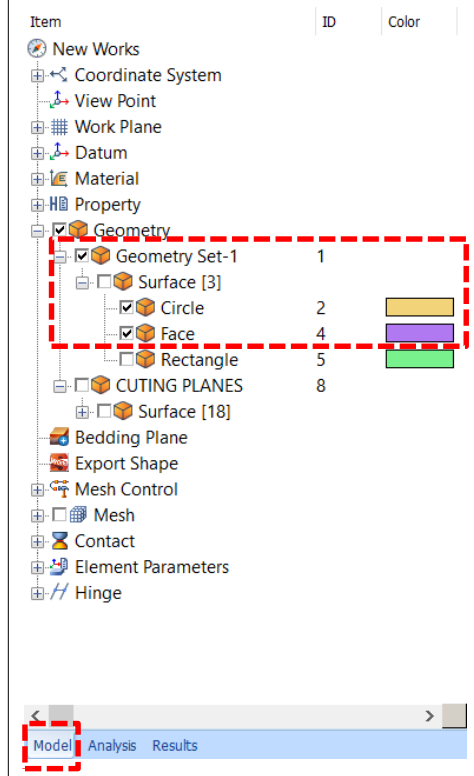
4-2 EXTRUSION – 2D TO 3D SOLIDS CREATION

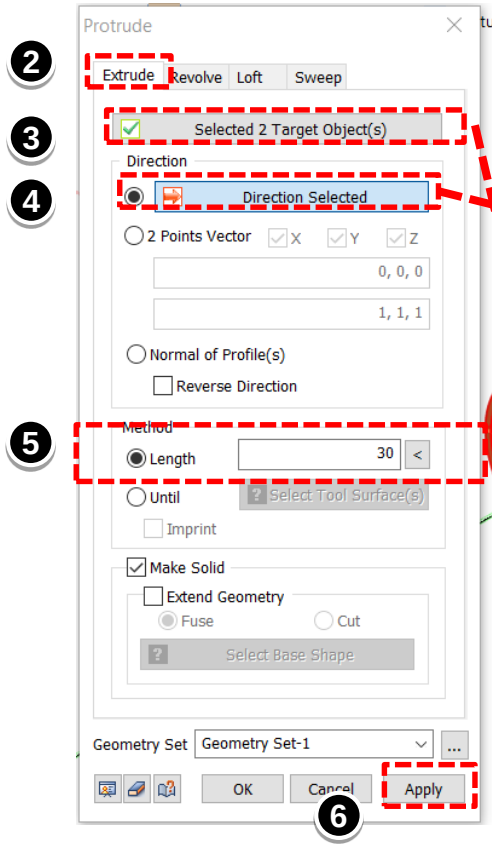
Procedure

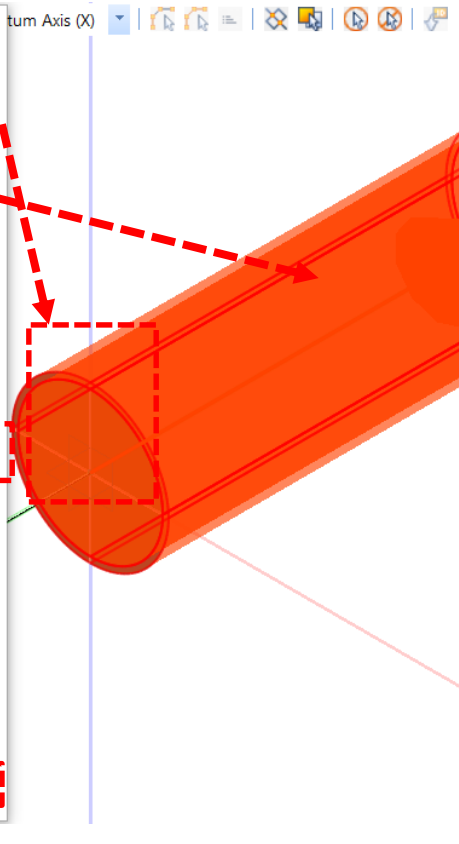
In the works tree select Model>Geometry>Geometry Set 1>Surface & Activate the Circle & Face

1. Go to Geometry>Protrude>Extrude
2. Select Extrude.
3. Select the **circle & face** as target objects.
4. Click on the **Y-axis** as selected direction.
5. Enter length as **30m.**
6. **Click Apply.**
7. Click to activate/deactivate the grid.









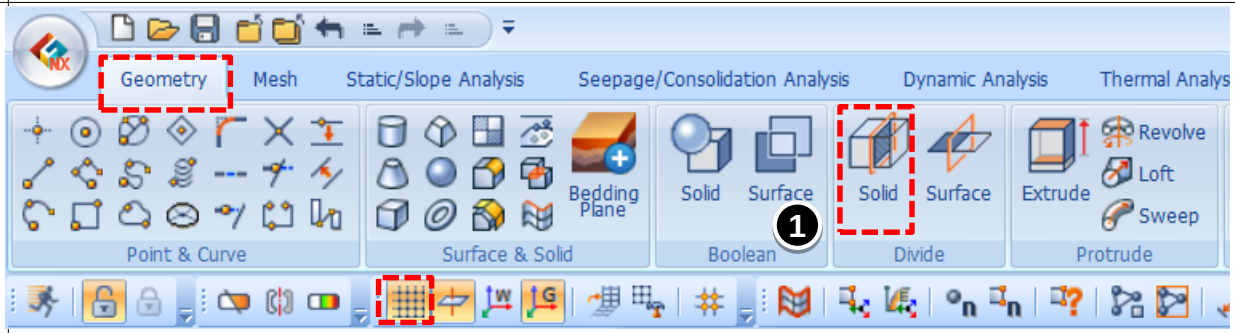
4-3 CONNECTING FINAL SOLID SECTIONS

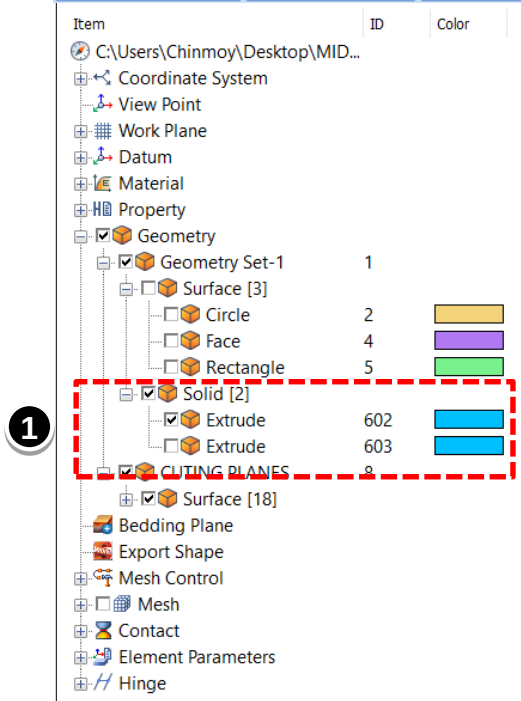
Procedure

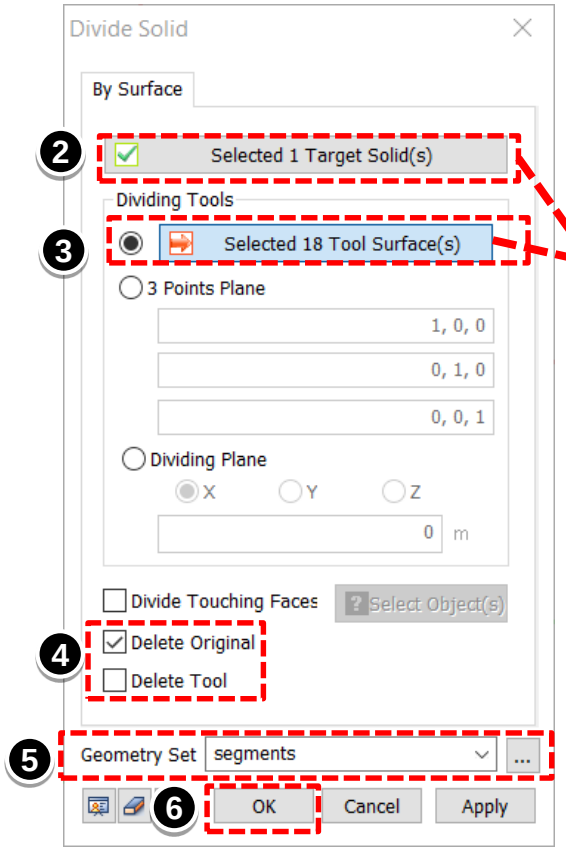
In the works tree select Model>Geometry>Geometry Set 1>Surface & Activate outer solid and the Cutting Planes Geometry Set & Deactivate all the other Surfaces in Geometry Set -1.

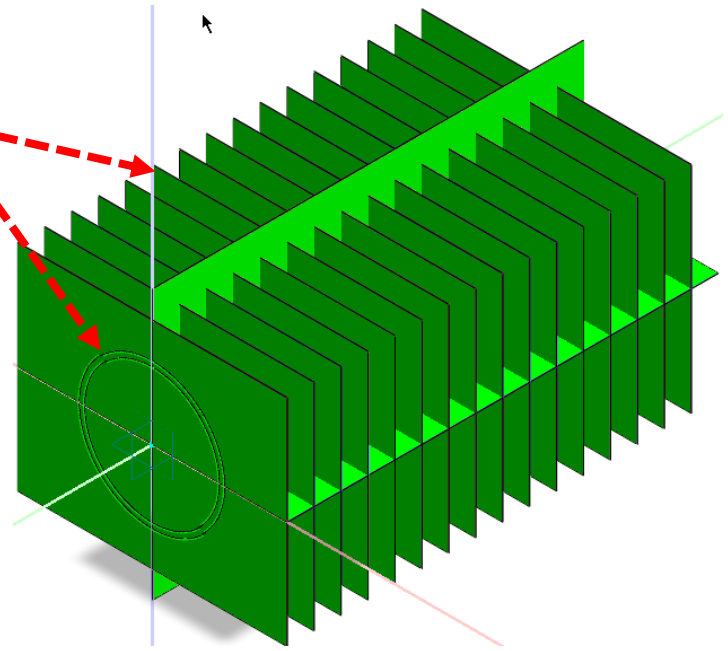
1. Go to Geometry>Divide>Solid..
2. Select the outer activated solid.
3. Select all the 18 tools in the **Cutting Planes** Geometry Set.
4. Click on **Delete Original** and untick the **Delete Tool** option.
5. Enter the name of geometry set as **“segments”.**
6. Click **OK.**

Drag and drop any remaining part of the segments in Geometry Set -1 to segments folder.









Step
04

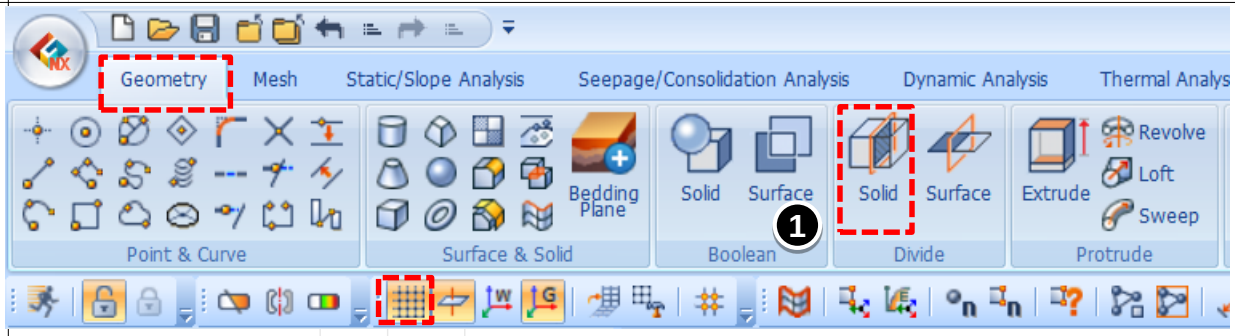
4-4 CONNECTING FINAL SOLID SECTIONS

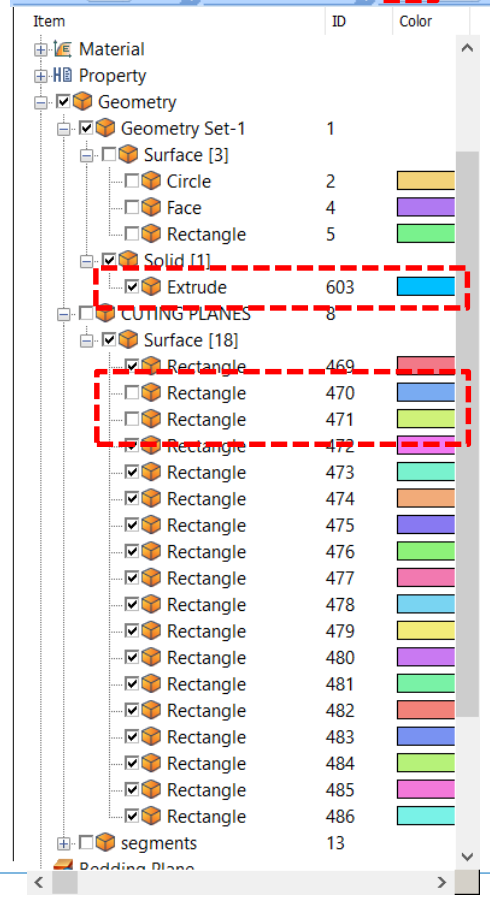
Procedure

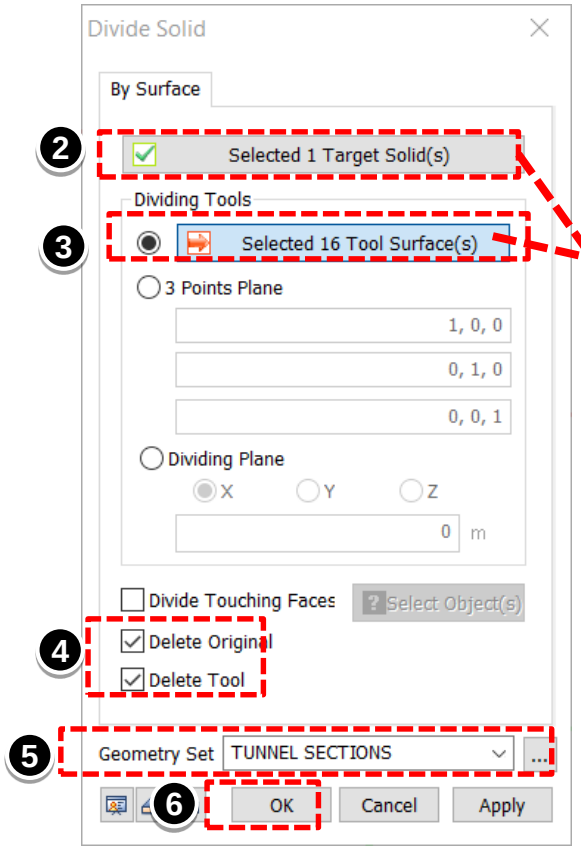
In the works tree select Model>Geometry>Geometry Set 1>Surface & Activate outer solid and the Cutting Planes Geometry Set except the two faces shown in the box & Deactivate all the other Surfaces in Geometry Set -1.

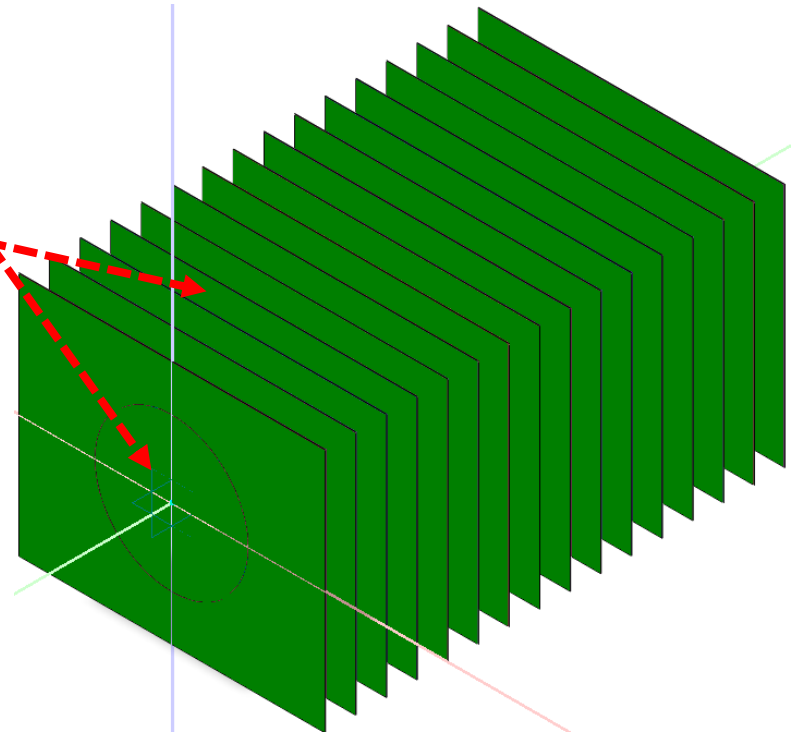
1. Go to Geometry>Divide>Solid..
2. Select the inner activated solid.
3. Select all the 16 tools in the **Cutting Planes** Geometry Set.
4. Click on **both** options.
5. Enter the name of geometry set as **“Tunnel Sections”.**
6. Click **OK.**

Drag and drop any remaining part of the tunnel sections in Geometry Set -1 to the Tunnel Sections Folder.









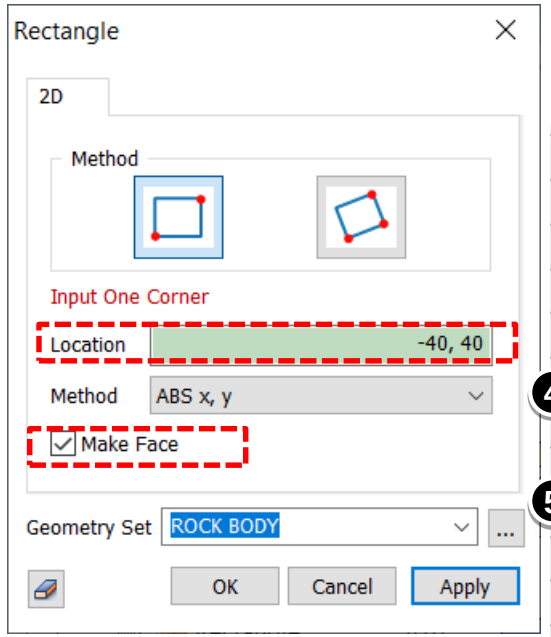
Step
04

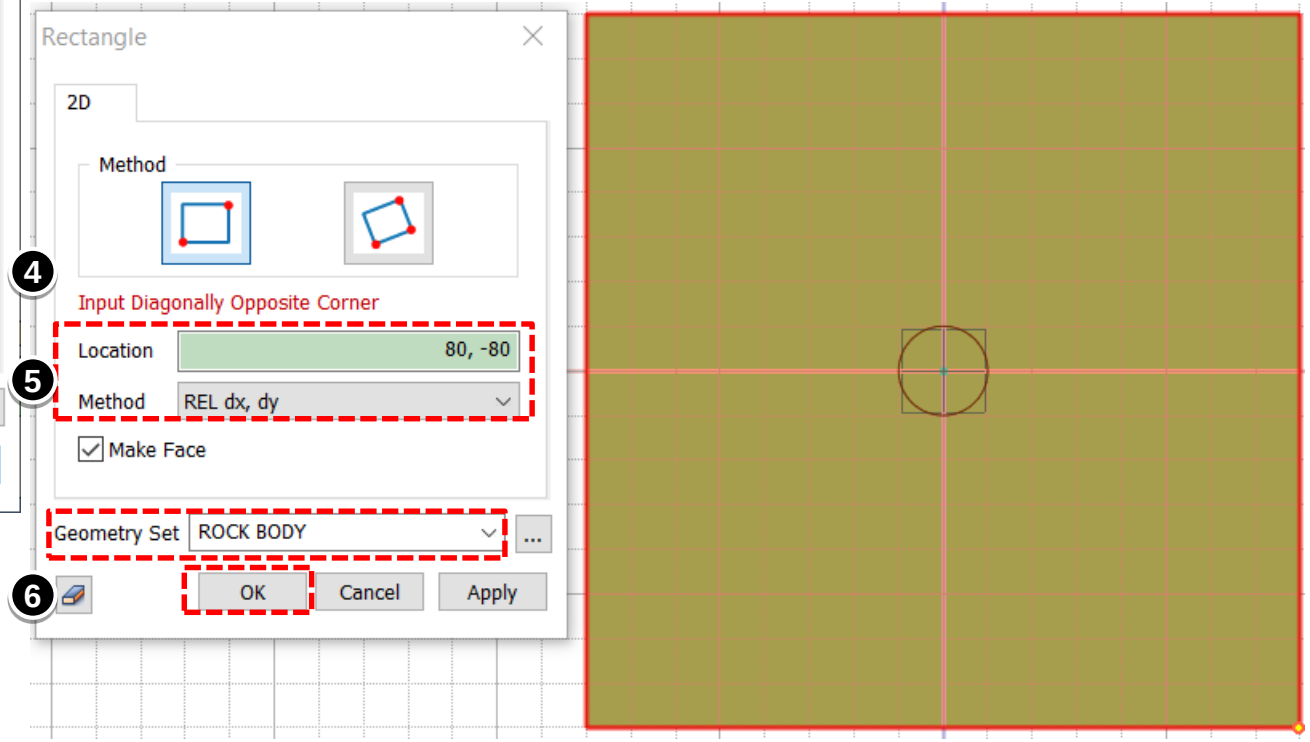
4-5 GROUND FACE & SOLIDS CREATION

Procedure

1. Go to Geometry>Rectangle.
2. Enter the input location as (-40,40).
3. Click on Make Face.
4. Input the corners as (**80,-80**) and select the method as **REL dx,dy.**
5. Enter the geometry set name as **ROCK BODY.**
6. Click **OK.**

FOLLOW THE STEPS IN SECTION/SLIDE 4-2 AND CREATE THE GROUND SOLID WITH THE LENTH IN "Y" DIRECTION AS 30m.

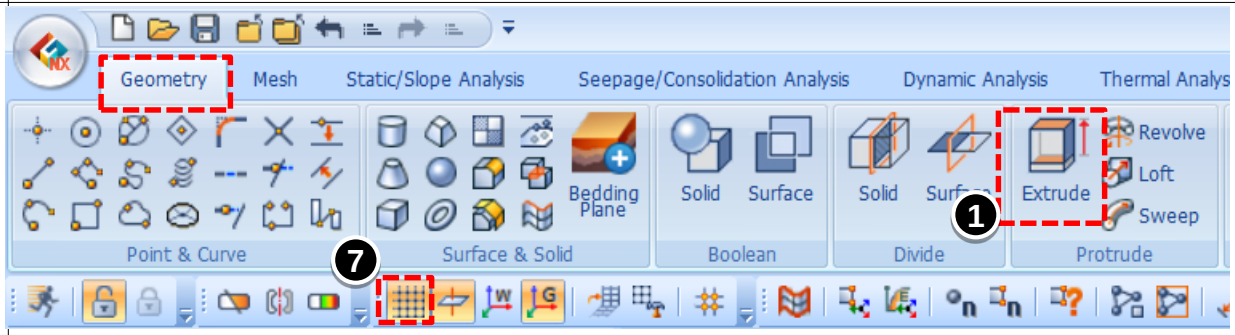


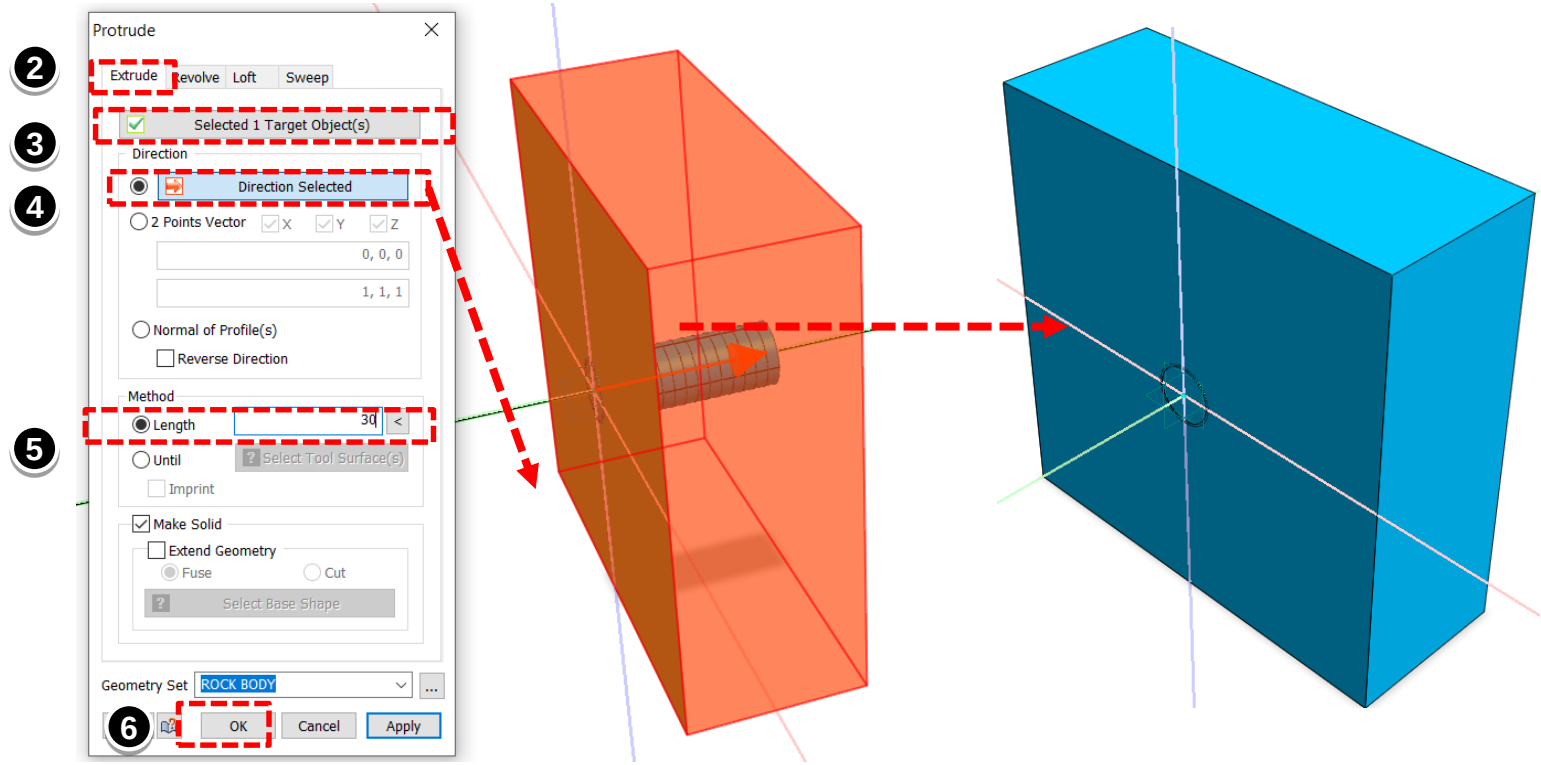


4-6 EXTRUSION – 2D TO 3D SOLIDS CREATION

Procedure

1. Go to Geometry>Protrude>Extrude
2. Select Extrude.
3. Select the **ground face** as target object.
4. Click on the **Y-axis** as selected direction.
5. Enter length as **30m.**
6. **Click OK.**
7. Click to activate/deactivate the grid.

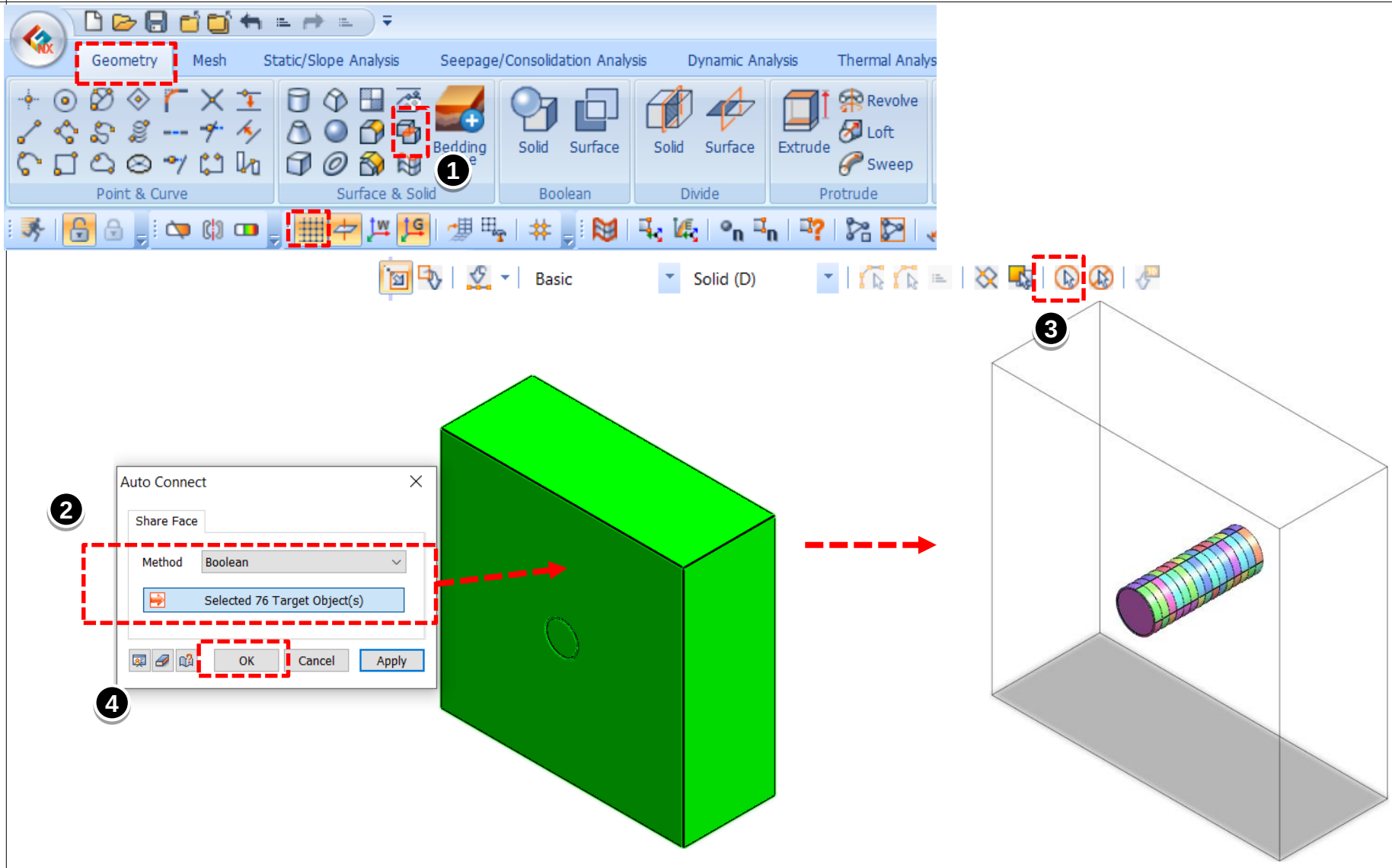




4-7 CONNECTING ALL SOLIDS SEGMENTS

Procedure

1. Go to Geometry>Autoconnect.
2. Select the solids in the model.
3. Click Select All.
4. Click **OK.**

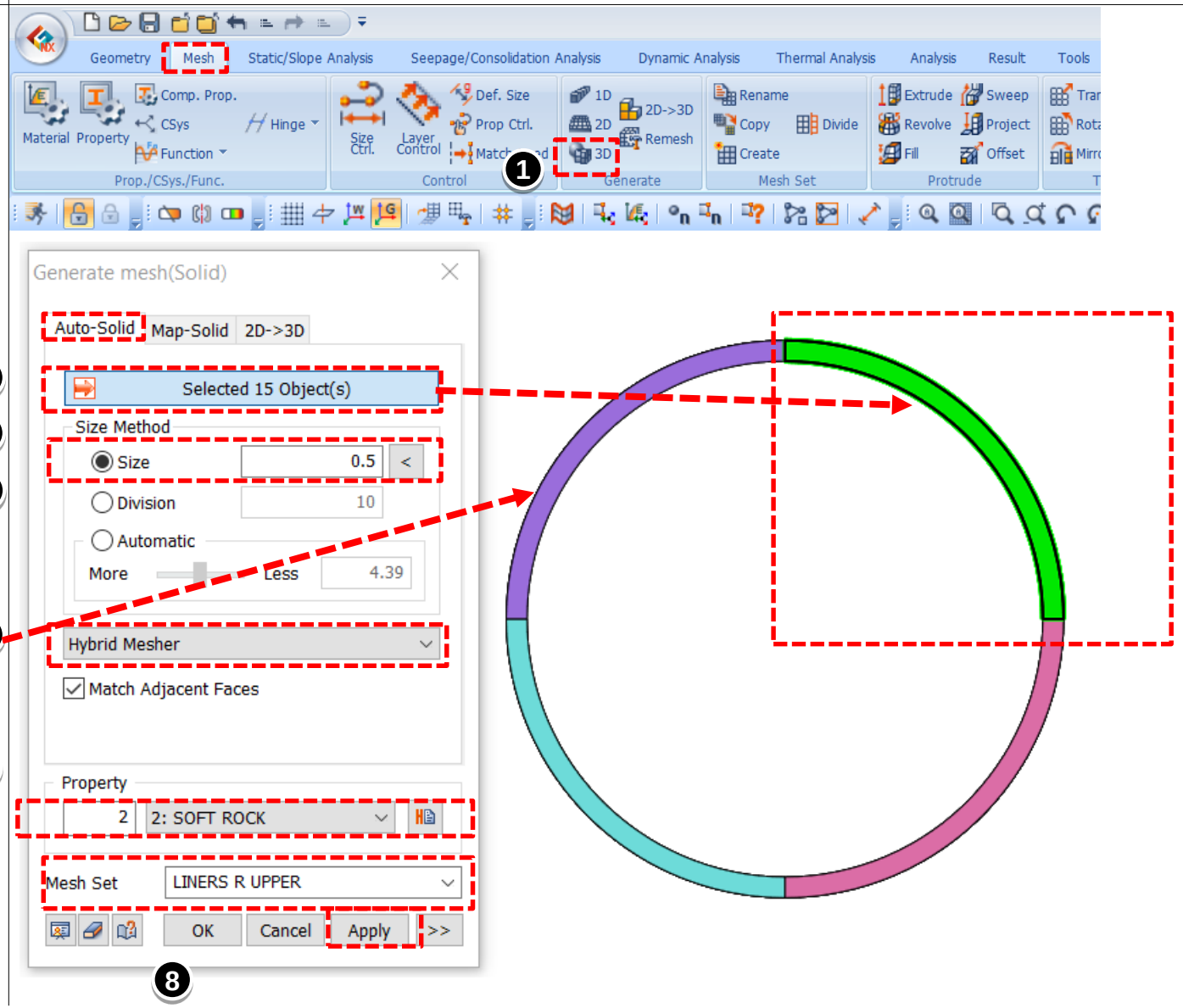


5-1 Meshing Liner Segments

Procedure

1. Go to Mesh > Generate > 3D
2. Select Auto-Solid.
3. Select the 15 segments as shown in the picture.
4. In the Size method give size **0.5 m.**
5. **Select Hybrid Mesher as the mesher type.**
6. In the property drop-down menu, select **Soft Rock**.
7. Give the Mesh set name "**Liners R Upper**." GTS NX will automatically assign the names to the solid sections selected.
8. Click **Apply**.

SIMILARLY CREATE THE MESH SETS FOR LINER IN LOWER UPPER SECTION & NAME THE MESH SET AS LINERS L UPPER.

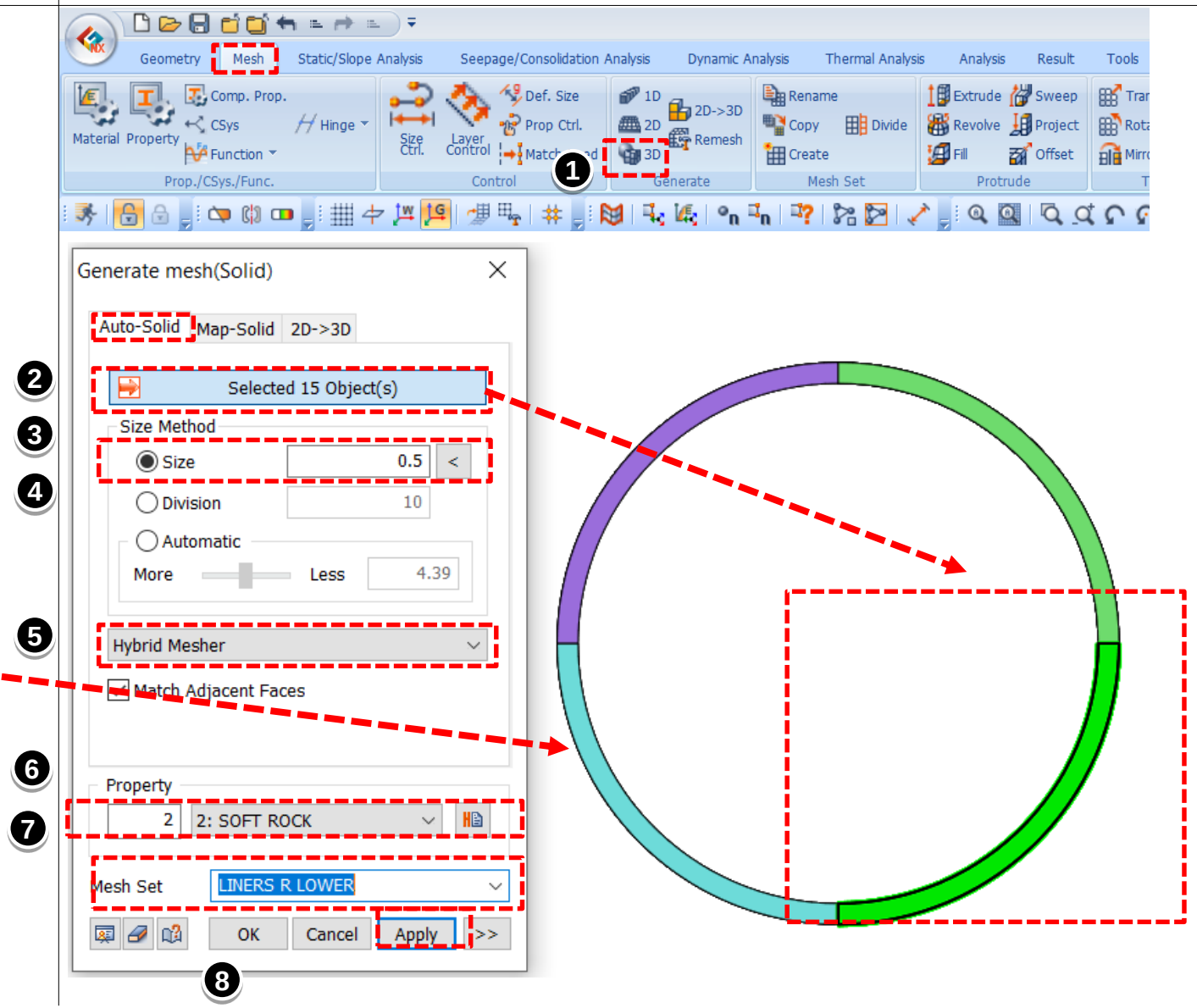


5-2 Meshing Liner Segments

Procedure

1. Go to Mesh > Generate > 3D
2. Select Auto-Solid.
3. Select the 15 segments as shown in the picture.
4. In the Size method give size **0.5 m.**
5. **Select Hybrid Mesher as the mesher type.**
6. In the property drop-down menu, select **Soft Rock**.
7. Give the Mesh set name "**LINERS R LOWER**." GTS NX will automatically assign the names to the solid sections selected.
8. Click **Apply**.

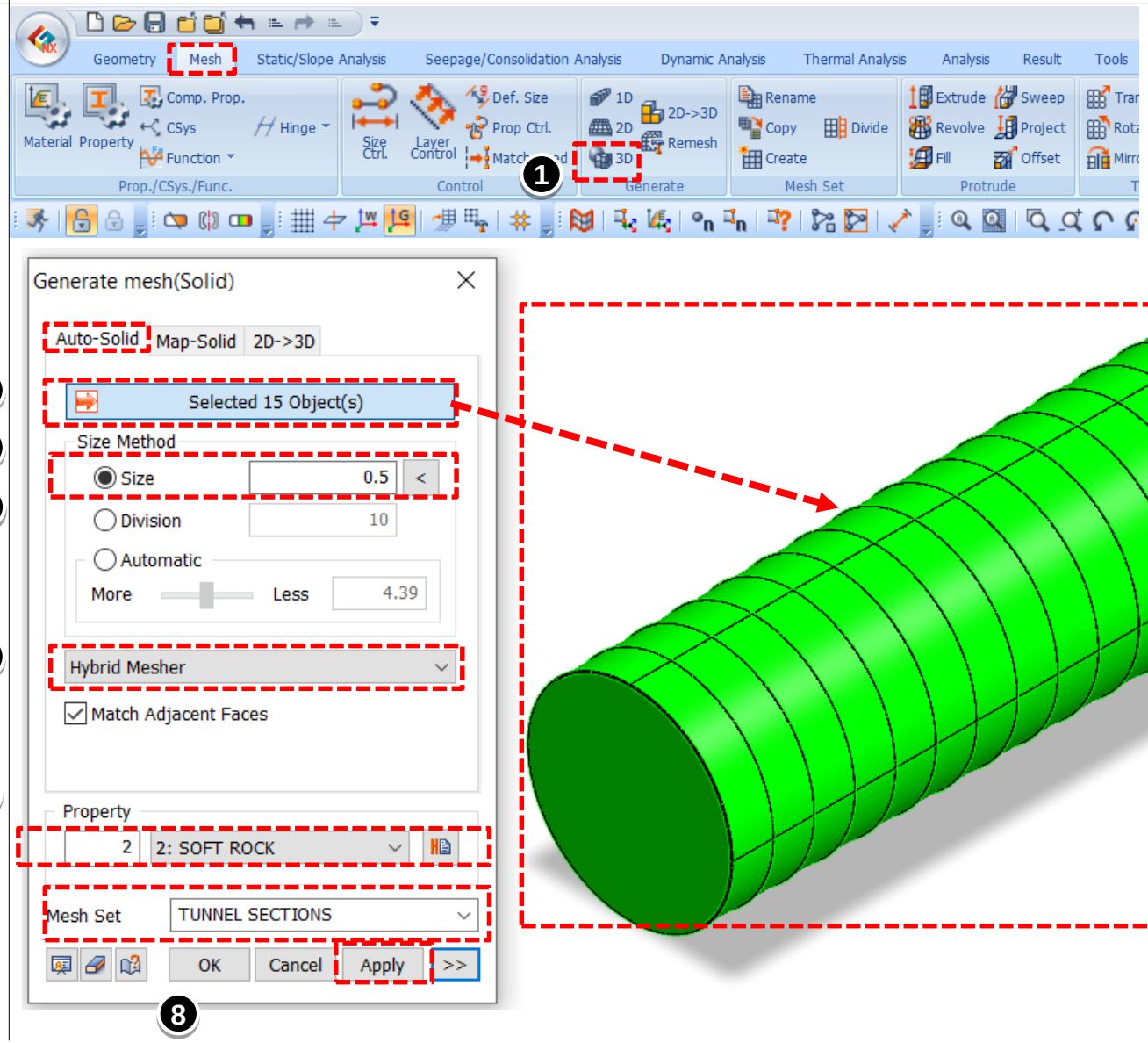
SIMILARLY CREATE THE MESH SETS FOR LINER IN LOWER UPPER SECTION & NAME THE MESH SET AS LINERS L LOWER.



5-3 Meshing Tunnel Sections

Procedure

1. Go to Mesh > Generate > 3D
2. Select Auto-Solid.
3. Select the 15 segments as shown in the picture.
4. In the Size method give size **0.5 m.**
5. **Select Hybrid Mesher as the mesher type.**
6. In the property drop-down menu, select **Soft Rock**.
7. Give the Mesh set name "**TUNNEL SECTIONS**." GTS NX will automatically assign the names to the solid sections selected.
8. Click **Apply**.

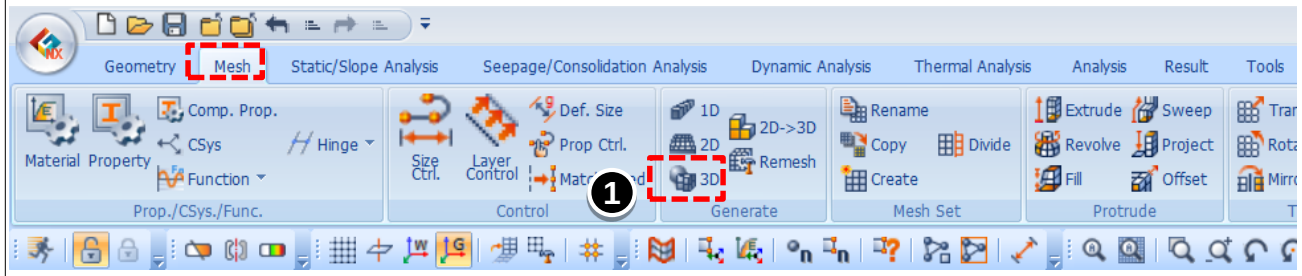


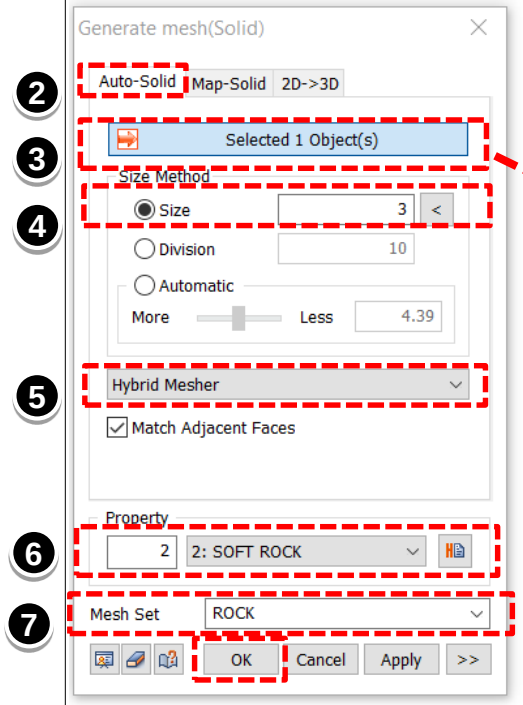
Step
05

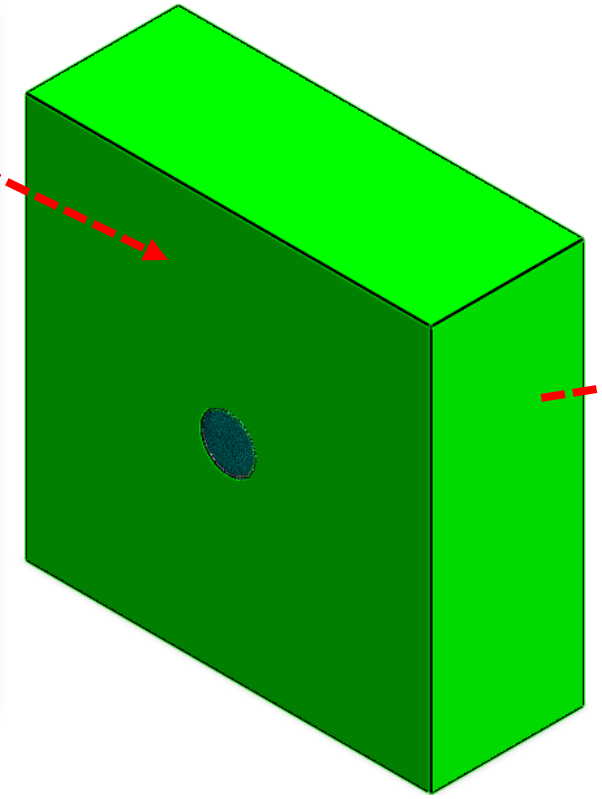
5-4 Meshing Ground Body

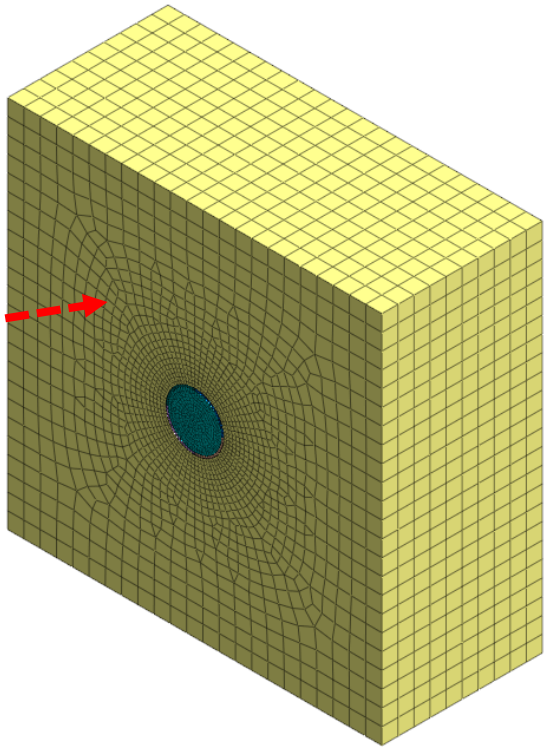
Procedure

1. Go to Mesh > Generate > 3D
2. Select Auto-Solid.
3. Select the 9 segments of cross passage.
4. In the Size method give size **3 m.**
5. **Select Hybrid Mesher as the mesher type.**
6. In the property drop-down menu, select **SOFT ROCK**.
7. Give the Mesh set name **ROCK.**
8. Click **OK**.









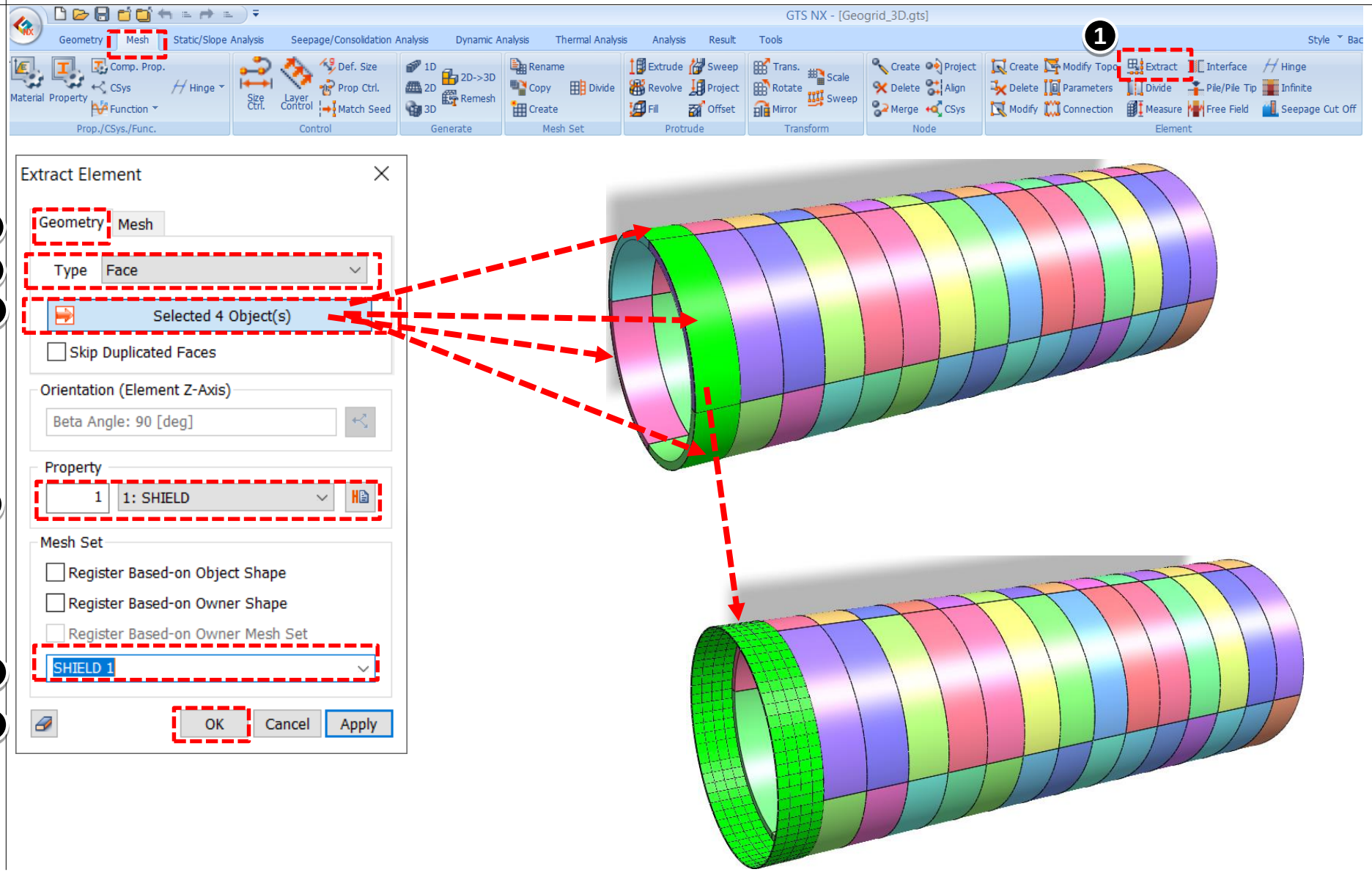
Step **05**

5-5 Meshing - Extraction of Shell Elements (TUNNEL SHIELD)

Procedure

1. Go to Mesh>Element>Extract.
2. Select Geometry.
3. Change the type to **FACE**.
4. Select the two faces as shown in the enclosed box.
5. The property is **SHIELD**.
6. Name retaining wall mesh set as **SHIELD 1**.
7. Click **OK**.

NOTE: ENSURE THAT YOU RENAME THE EXTRACTED MESH SETS AFTER THE FACE HAVE BEEN SELECTED, FAILURE TO DO SO WILL CREATE A COMBINED MESH SET.

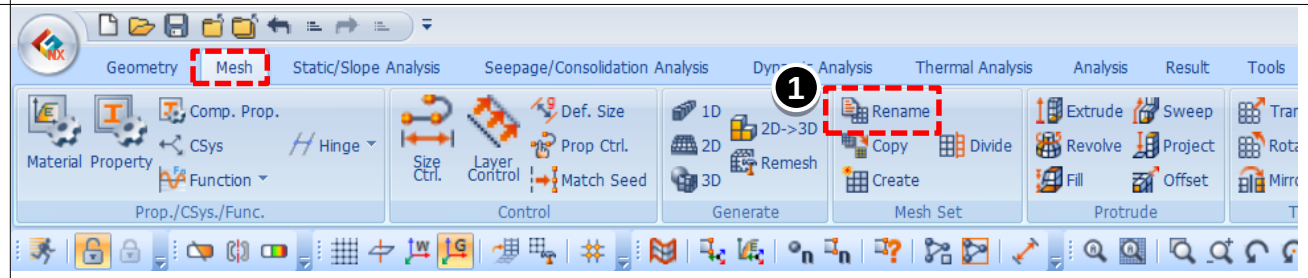


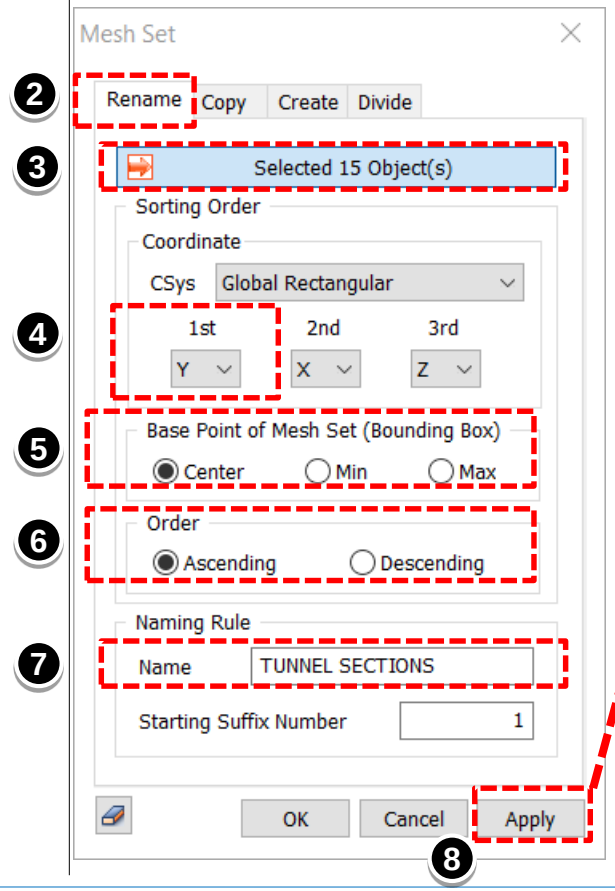
Step 05

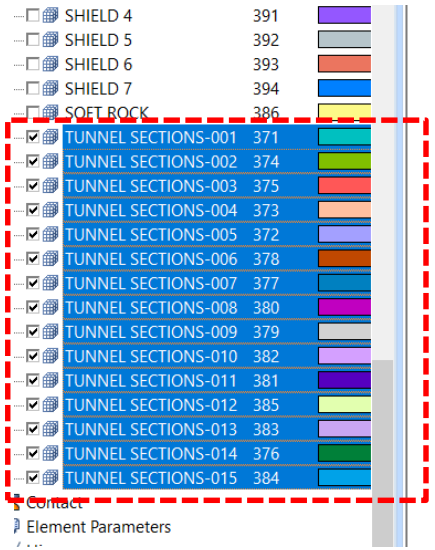
5-6 Renaming Main Tunnel Segments

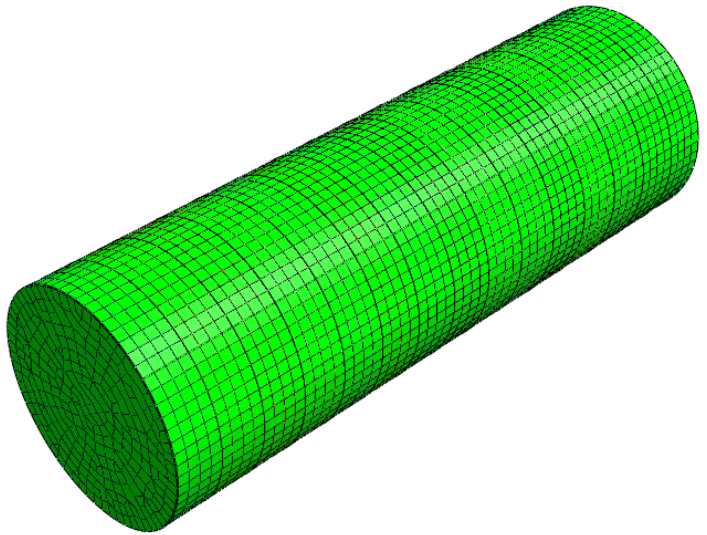
Procedure

1. Go to Mesh > Rename
2. Click on Rename.
3. Select the 25 mesh sets for the **main tunnel segments**.
4. In sorting order, start with **Y axis** as the 1st priority.
5. Leave the base point as Center.
6. Set the order as **Ascending**.
7. Enter the name as **TUNNEL SECTIONS.**
8. Click **Apply**.









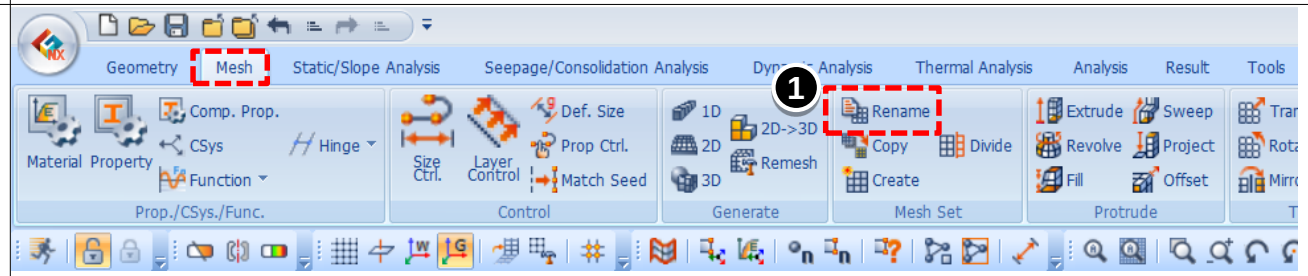
Step 05

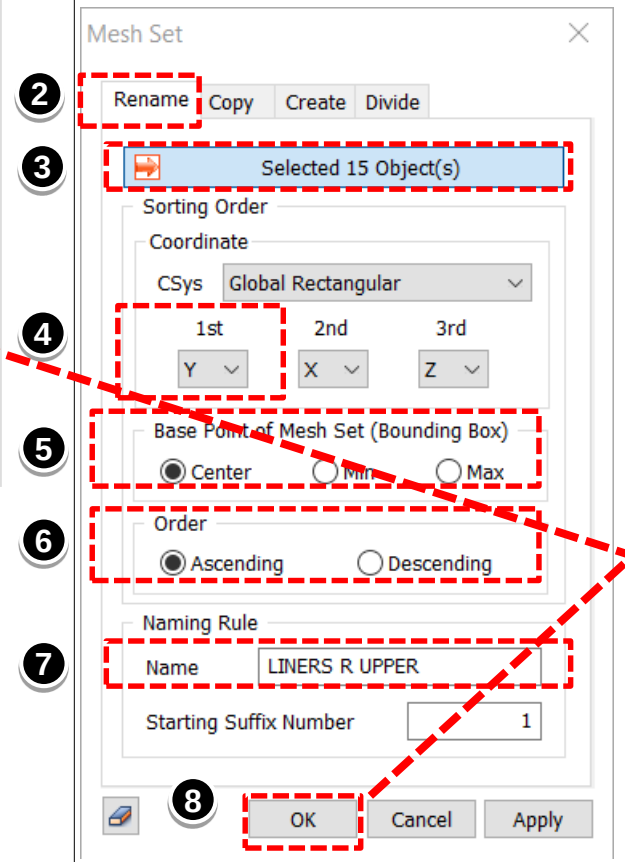
5-7 Renaming Liner Segments

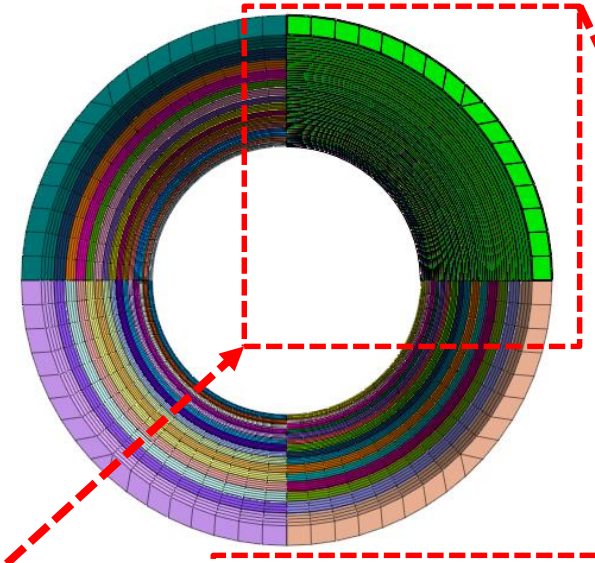
Procedure

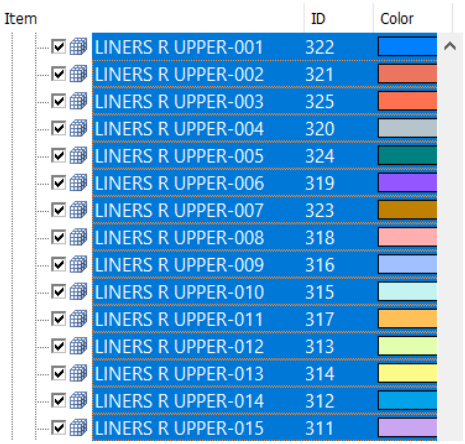
1. Go to Mesh > Rename
2. Click on Rename.
3. Select the 15 mesh sets for the **Liners R Upper Mesh Sets.**
4. In sorting order, start with **Y axis** as the 1st priority.
5. Leave the base point as Center.
6. Set the order as **Ascending**.
7. Enter the name as “**LINERS R UPPER”.**
8. Click **OK**.

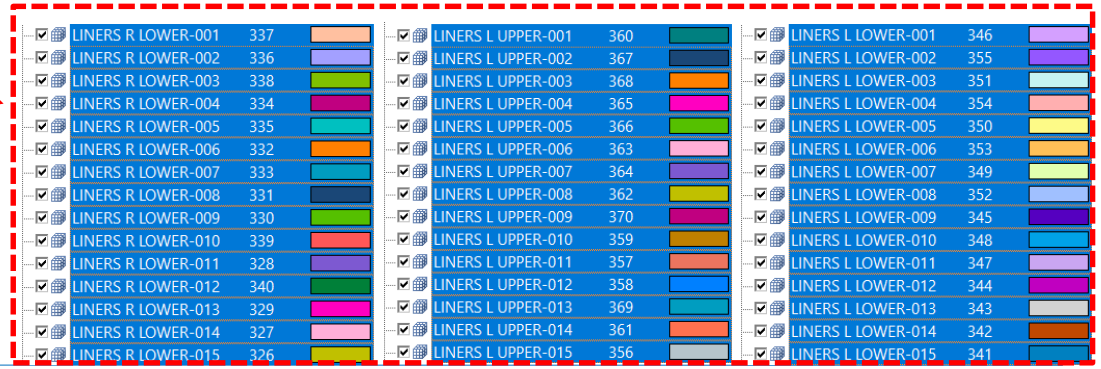
SIMILARLY RENAME THE MESH SETS “LINERS L LOWER”, “LINERS L UPPER” “LINERS R LOWER”.











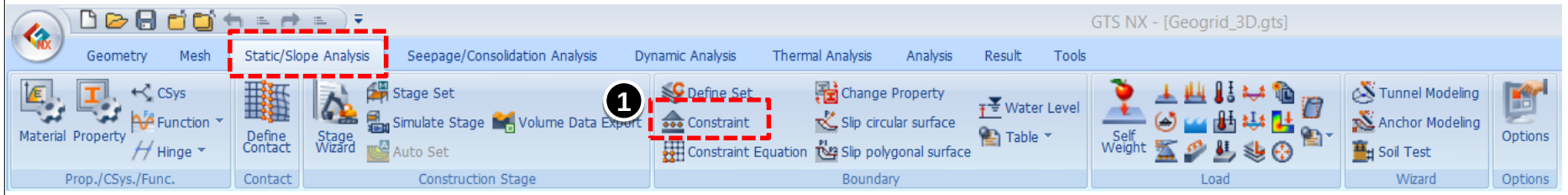
Step
06

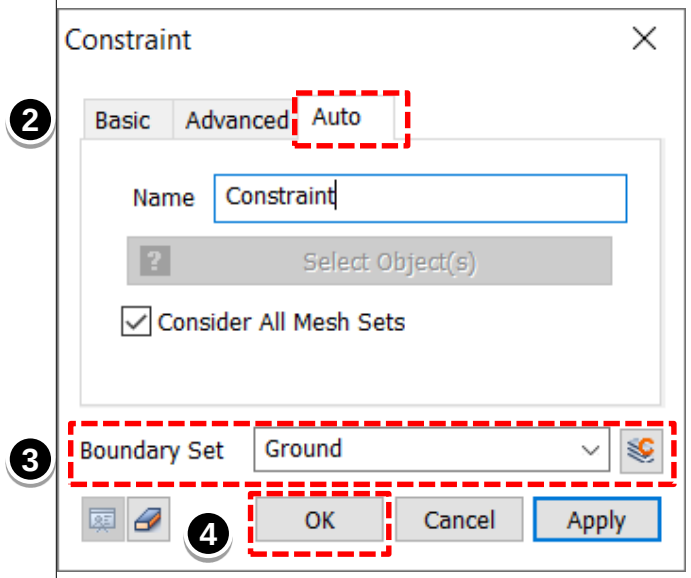
6-1 Boundary Condition

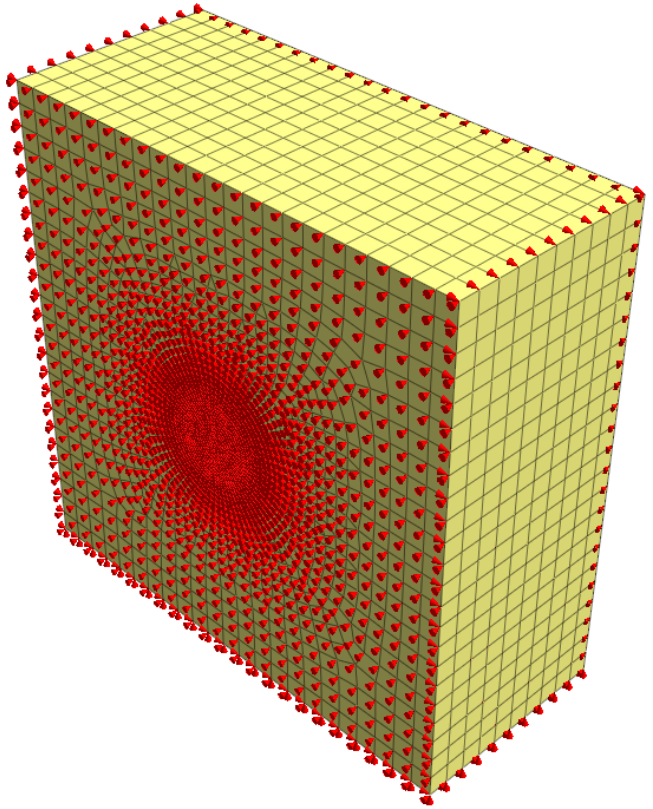
Procedure

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 06

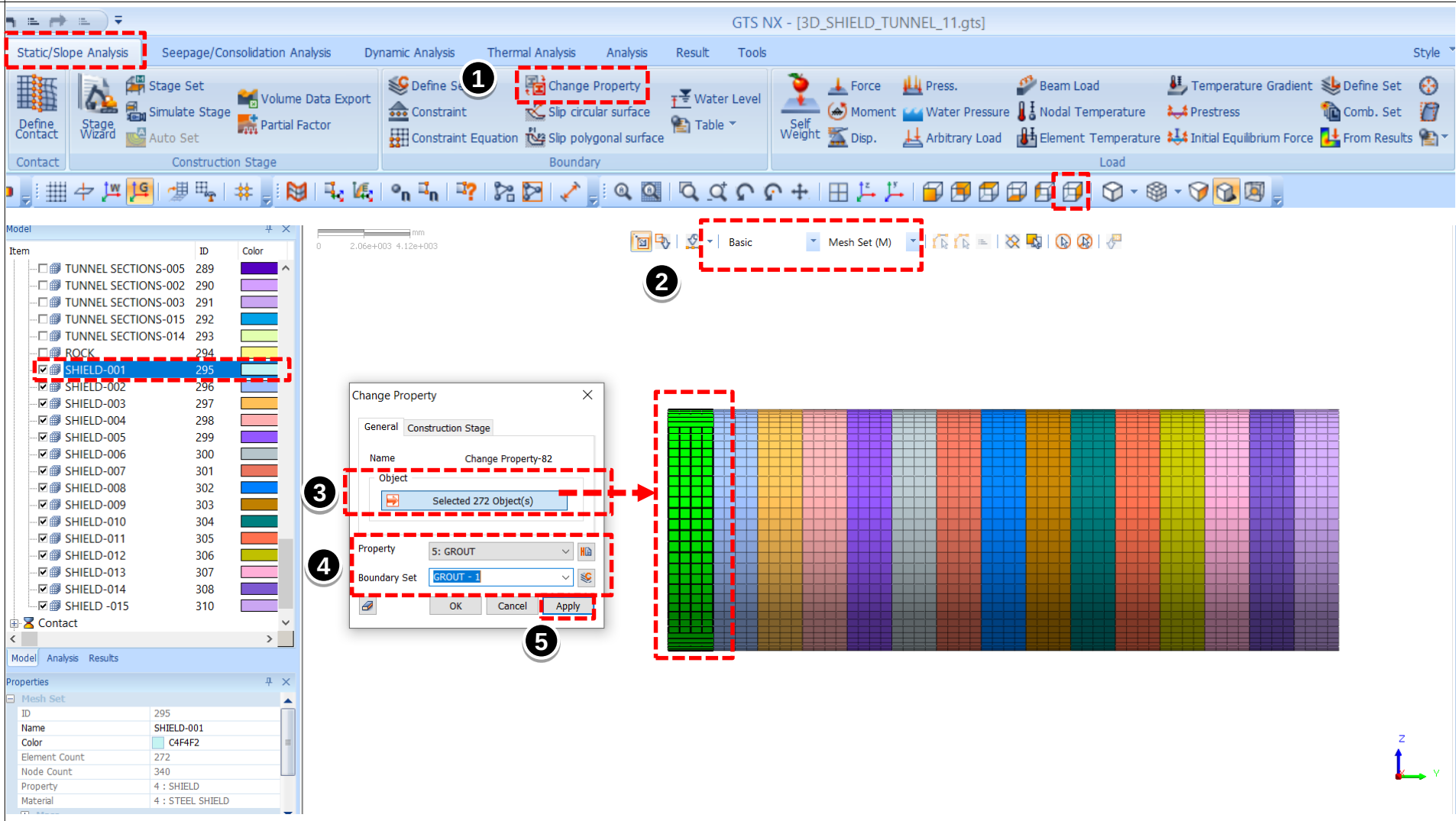
6-2 Boundary Condition (CHANGE ELEMENT ATTRIBUTES FOR SHIELD)

Procedure

1. Go to Static/Slope Analysis > Boundary>Change Property.
2. Change to **“Basic” & “Mesh Set”.**
3. Select “**SHIELD 1**” Mesh Set.

   The property is “**Grout**“.
4. Name the boundary set as **“GROUT -1”.**
5. Click **Apply**.

SIMILARLY CREATE GROUT PROPERTY CHANGE BOUNDARY CONDITION FOR ALL THE SHIELD MESH SETS. THE NAMES OF BOUDNARY CONDITIONS SHOULD START FROM GROUT – 2 AND END AT GROUT – 15.



Step 06

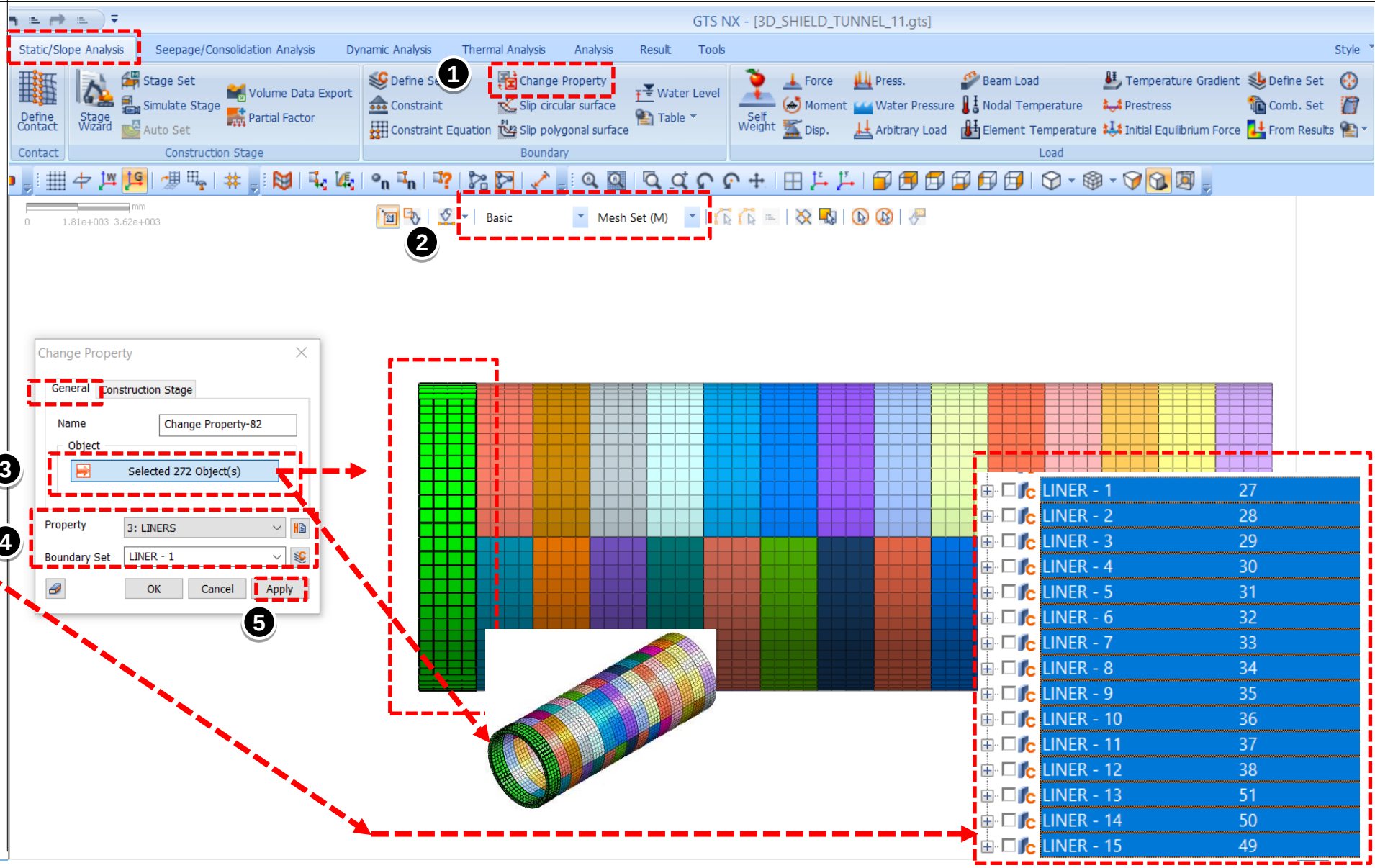
6-3 Boundary Condition (CHANGE ELEMENT ATTRIBUTES FOR LINERS)

Procedure

1. Go to Static/Slope Analysis > Boundary>Change Property.
2. Change to **“Basic” & “Mesh Set”.**
3. Select “**LINER R UPPER- 1, LINERS R LOWER-1, LINER L LOWER-1, LINERS L UPPER-1**” Mesh Sets.
4. The property is “**LINERS**“.

   Name the boundary set as **“LINER -1”.**
5. Click **Apply**.

SIMILARLY CREATE GROUT PROPERTY CHANGE BOUNDARY CONDITION FOR ALL THE SHIELD MESH SETS. THE NAMES OF BOUDNARY CONDITIONS SHOULD START FROM LINER – 2 AND END AT LINER – 15.



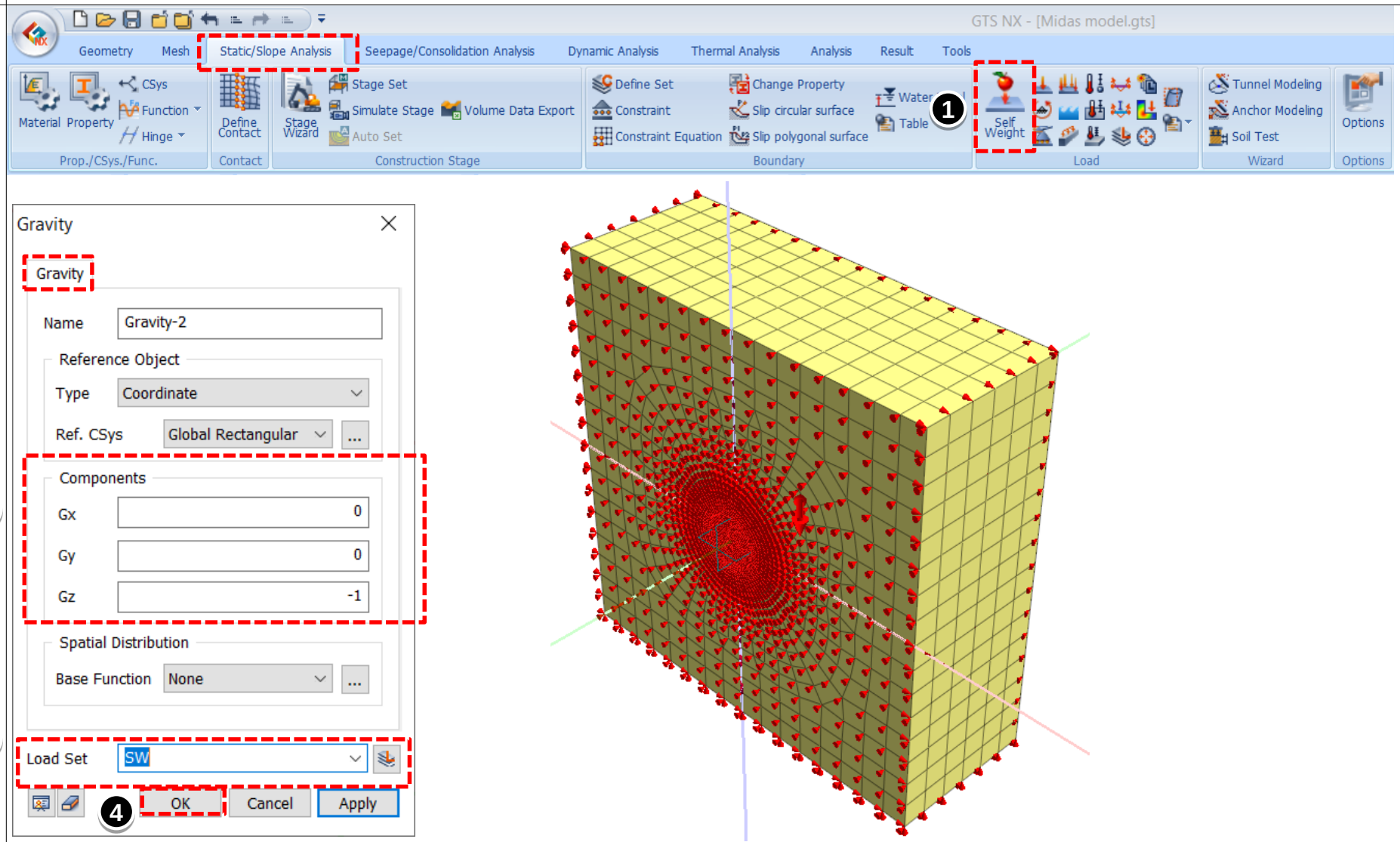
Step
07

7-1 Load Definition - SELF WEIGHT

Procedure

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

.



Step 07

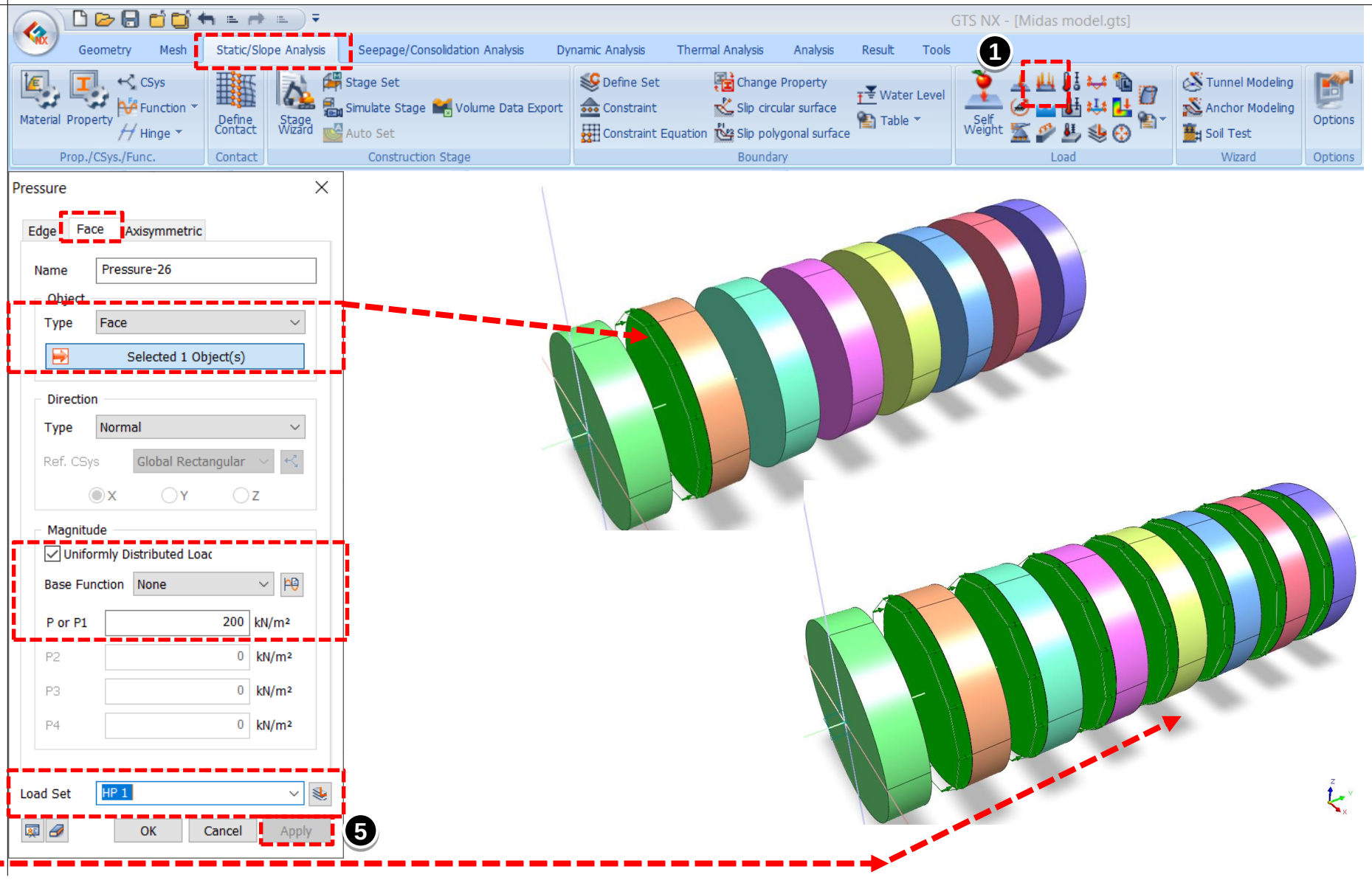
7-2 Load Definition – FACE PRESSURE

Procedure

1. Go to Static/Slope Analysis > Load >Pressure Load.
2. Select “Face” & Select the face of the **3rd Tunnel Section Solid** as shown in the figure.
3. Click on Uniformly Distributed Load.
4. Enter P or P1 value as **“200 kPa”**.
5. Enter the name of load set as “**HP - 1**”.
6. Click **APPLY.**

SIMILARLY SELECT THE FACES OF 5TH, 7TH, 9TH, 11TH, 13TH, 15TH TUNNEL SECTION SOLIDS AS SHOWN IN THE FIGURE AND THE RENAME THE LOAD SETS SARTING FROM HP- 2 TO HP - 7.

.



Step 07

7-3 Load Definition – JACK THRUST

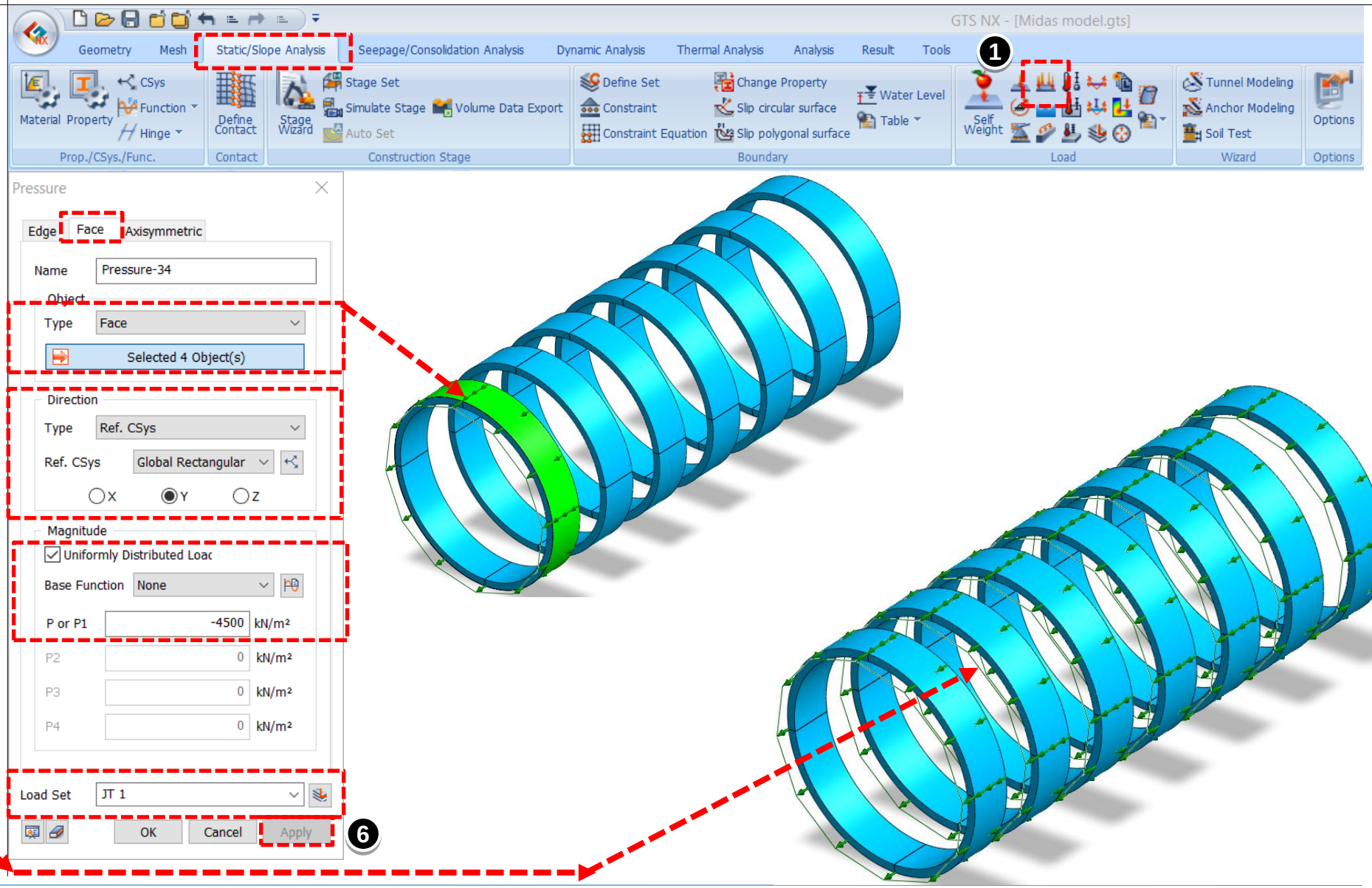
Procedure

1. Go to Static/Slope Analysis > Load >Pressure Load.
2. Select “Face” & Select the Outer faces of the **2nd Liner Segment Solid** as shown in the figure.
3. In the Direction Drop Down Menu, select **“Ref. Csys”** and select **“Y”** Axis.
4. Click on Uniformly Distributed Load.
5. Enter P or P1 value as **“-4500 kPa”**.
6. Enter the name of load set as “**JT - 1**”.

   Click **APPLY.**

   SIMILARLY SELECT THE FACES OF 4th,6th, 8th, 10th, 12th and 14th LINER SEGMENT SOLIDS AS SHOWN IN THE FIGURE AND THE RENAME THE LOAD SETS SARTING FROM JT - 2 TO JT - 7.

   .



Step 07

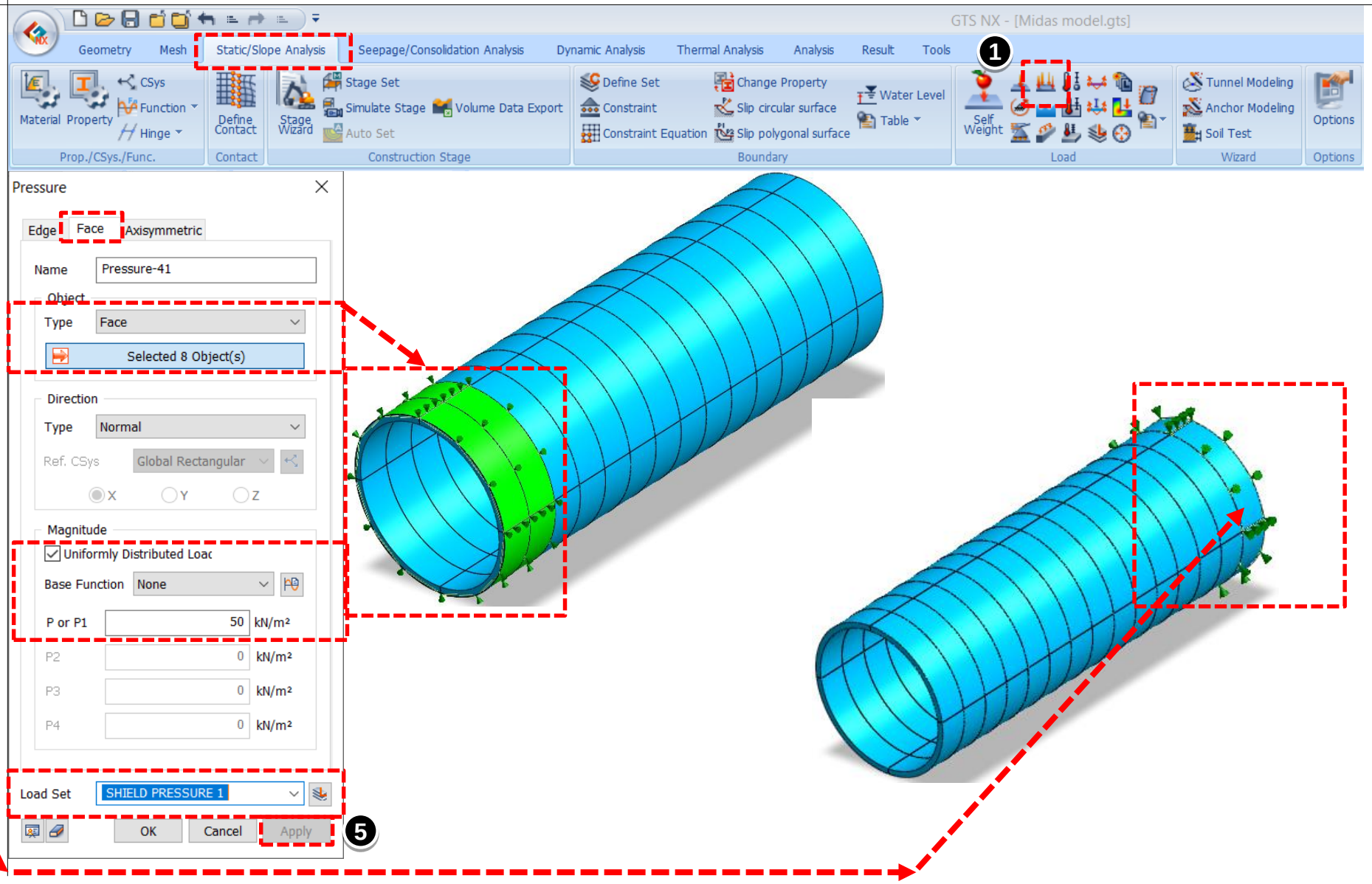
7-4 Load Definition – SHIELD PRESSURE

Procedure

1. Go to Static/Slope Analysis > Load >Pressure Load.
2. Select “Face” & Select the outer faces of the **1st & 2nd Liner Segment Solids** as shown in the figure.
3. Click on Uniformly Distributed Load.
4. Enter P or P1 value as **“50 kPa”**.
5. Enter the name of load set as “**SHIELD PRESSURE - 1**”.
6. Click **APPLY.**

SIMILARLY CREATE OTHER SHIELD PRESSURE STATIC LOADS TILL SHIELD PRESSURE 7, AND FOR THE LAST LINER SEGMENTS, CREATE SHIELD PRESSURE - 8.

.



Step 07

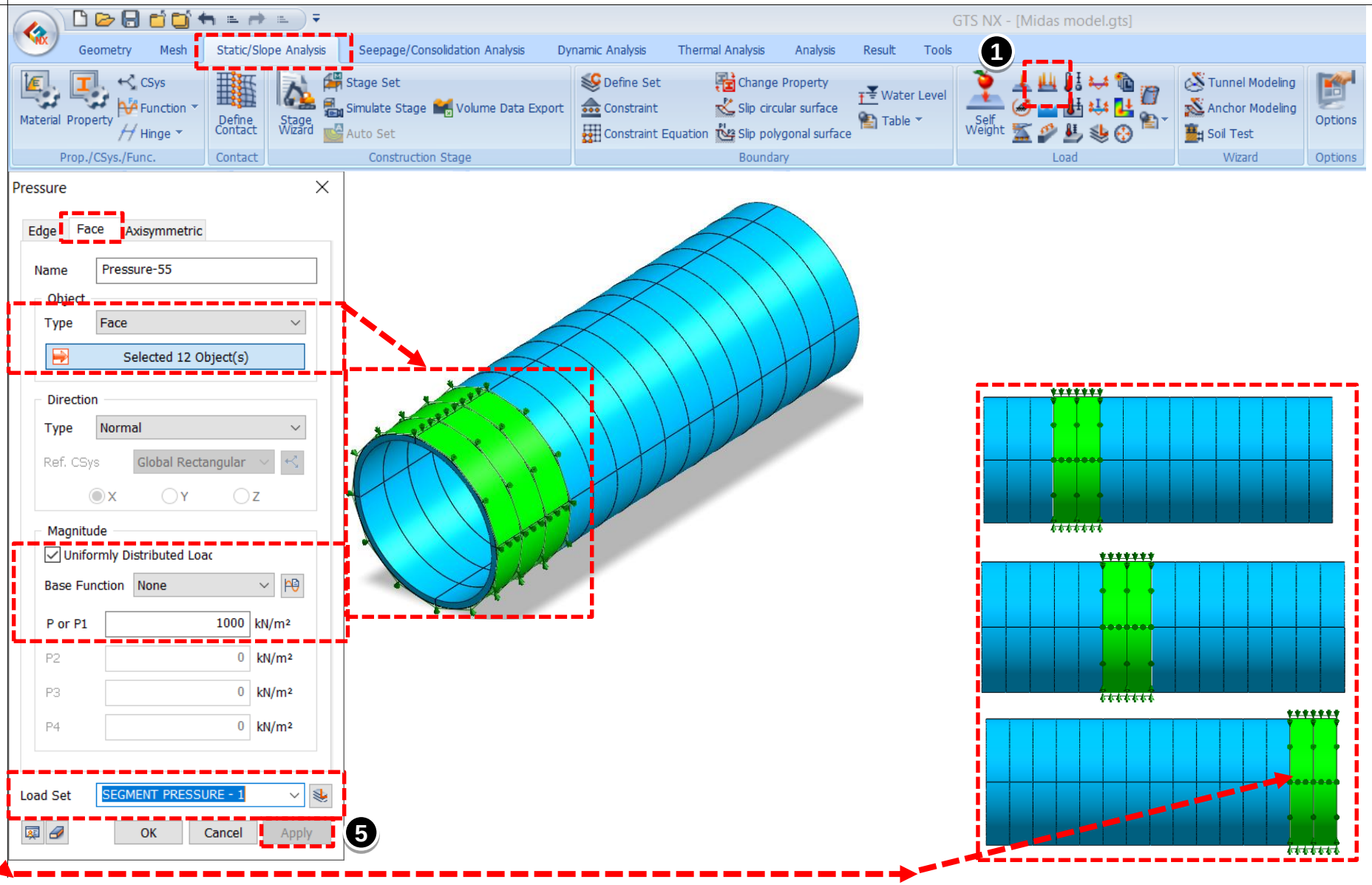
7-5 Load Definition – SEGMENT PRESSURE

Procedure

1. Go to Static/Slope Analysis > Load >Pressure Load.
2. Select “Face” & Select the outer faces of the **1st ,2nd & 3rd Liner Segment Solids** as shown in the figure.
3. Click on Uniformly Distributed Load.
4. Enter P or P1 value as **“1000 kPa”**.
5. Enter the name of load set as “**SEGMENT PRESSURE - 1**”.
6. Click **APPLY.**

SIMILARLY CREATE OTHER SEGMENT PRESSURE STATIC LOADS TILL SHIELD PRESSURE - 7, BY SELECTING TWO ADJACENT OUTER SEGMENTAL SOLID FACES AS SHOWN IN THE FIGURES. THE SEGMETN PRESSURE LOAD SETS WILL END AT SEGMENT PRESSURE - 7.

.



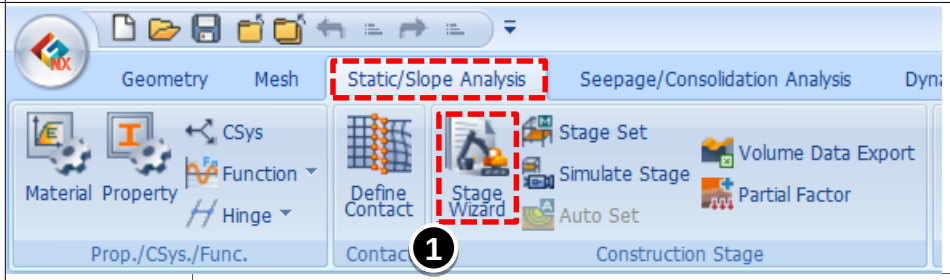
8-1 Construction Stage Wizard

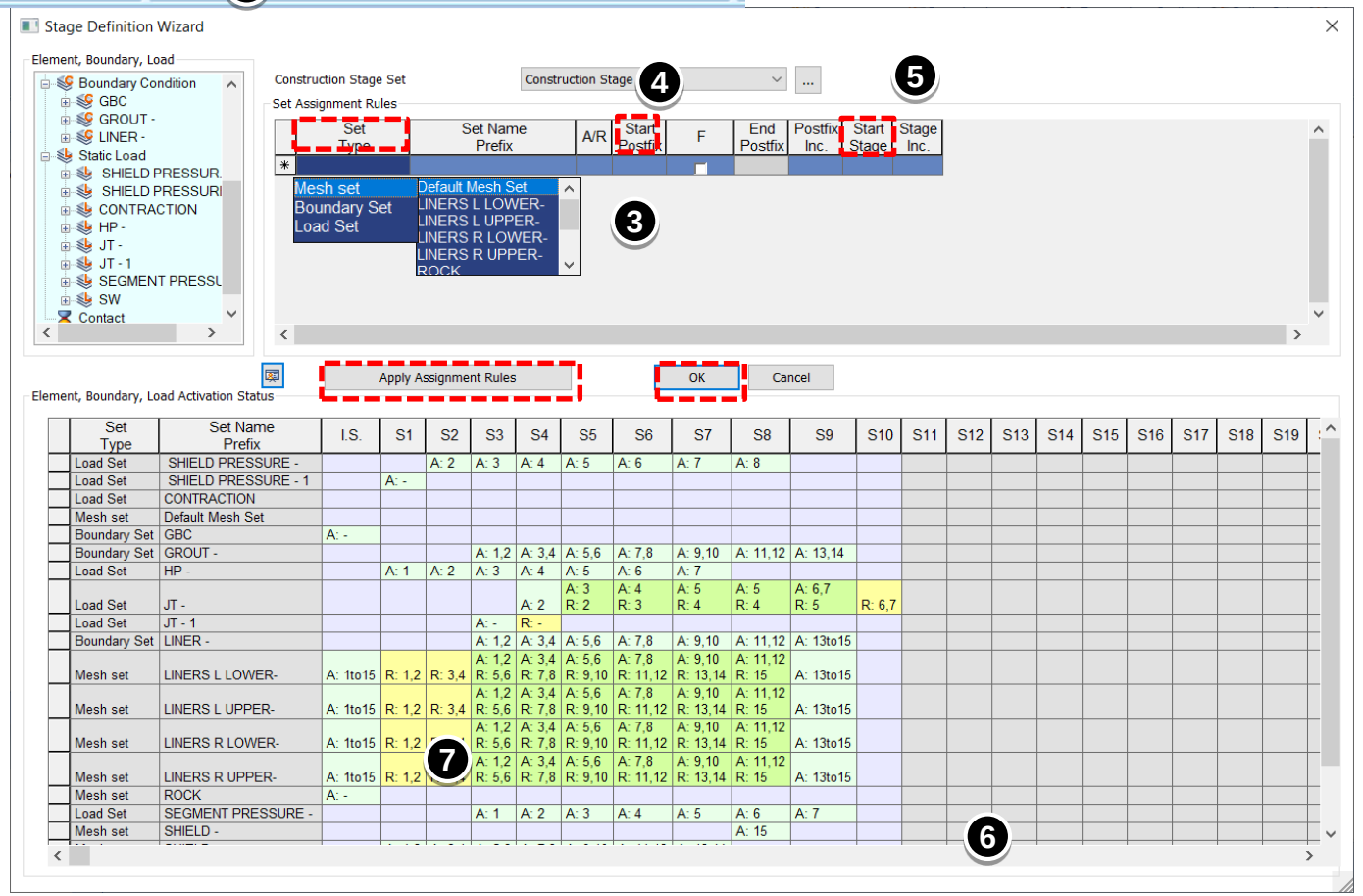
Procedure

1. Go to Static/Slope Analysis> Stage Wizard.
2. In the Set Type Drop Down Menu, **select Mesh Set.**
3. In the Set name Prefix select **The Mesh Sets.**
4. **A/R** stands for **ADD/REMOVE.** Addition will be displayed as **green** in color and Removal will be shown in **Yellow.**
5. **Start Stage** stands for stage in which the particular mesh will be added to the construction stage wizard, if left at 1, then the mesh set will be activated from the initial stage itself.

   Repeat the same process for all mesh, loads and boundaries sets. The sequence is described in the next slide.

   After all the mesh sets are activated and the necessary commands entered, activate the Boundary Condition & Load Set.





8-2 Construction Stage Wizard

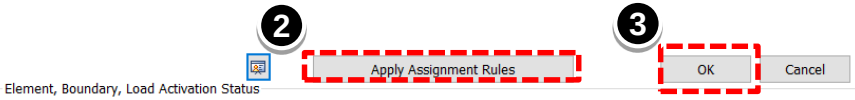
Procedure

1. Enter the data for each mesh, load and boundary set as shown.
2. Click Apply Assignment Rules.
3. Click OK.

   The data should match correctly with the table in the right as shown.

Set Assignment Rules

Set Type	Set Name Prefix	A/R	Start Postfix	F	End Postfix	Postfix Inc.	Start Stage	Stage Inc.
Load Set	SHIELD PRESSURE	A	1	☐		1	1	1
Boundary Set	GROUT -	A	1	☐		2	3	1
Boundary Set	GROUT -	A	2	☐		2	4	1
Load Set	HP -	A	1	☐		1	1	1
Load Set	JT -	A	1	☐		1	3	1
Load Set	JT -	R	1	☐		1	4	1
Boundary Set	LINER -	A	1	☑	13	2	3	1
Boundary Set	LINER -	A	2	☑	1	2	3	1
Boundary Set	LINER -	A	15	☐		1	9	1
Mesh set	LINERS L LOWER-	R	1	☐		2	1	1
Mesh set	LINERS L LOWER-	R	2	☐		2	1	1
Mesh set	LINERS L LOWER-	A	1	☑	13	2	3	1
Mesh set	LINERS L LOWER-	A	2	☐		2	3	1
Mesh set	LINERS L LOWER-	A	15	☐		1	9	1
Mesh set	LINERS L UPPER-	R	1	☐		2	1	1
Mesh set	LINERS L UPPER-	R	2	☐		2	1	1
Mesh set	LINERS L UPPER-	A	1	☑	13	2	3	1
Mesh set	LINERS L UPPER-	A	2	☐		2	3	1
Mesh set	LINERS L UPPER-	A	15	☐		1	9	1
Mesh set	LINERS R LOWER-	R	1	☐		2	1	1
Mesh set	LINERS R LOWER-	R	2	☐		2	1	1
Mesh set	LINERS R LOWER-	A	1	☑	13	2	3	1
Mesh set	LINERS R LOWER-	A	2	☐		2	3	1
Mesh set	LINERS R LOWER-	A	15	☐		1	9	1
Mesh set	LINERS R UPPER-	R	1	☐		2	1	1
Mesh set	LINERS R UPPER-	R	2	☐		2	1	1
Mesh set	LINERS R UPPER-	A	1	☑	13	2	3	1
Mesh set	LINERS R UPPER-	A	2	☐		2	3	1
Mesh set	LINERS R UPPER-	A	15	☐		1	9	1
Load Set	SEGMENT PRESSURE -	A	1	☐		1	3	1
Mesh set	SHIELD-	A	1	☐		2	1	1
Mesh set	SHIELD-	A	2	☐		2	1	1
Mesh set	TUNNEL SECTIONS-	R	1	☐		2	1	1
Mesh set	TUNNEL SECTIONS-	R	2	☐		2	1	1



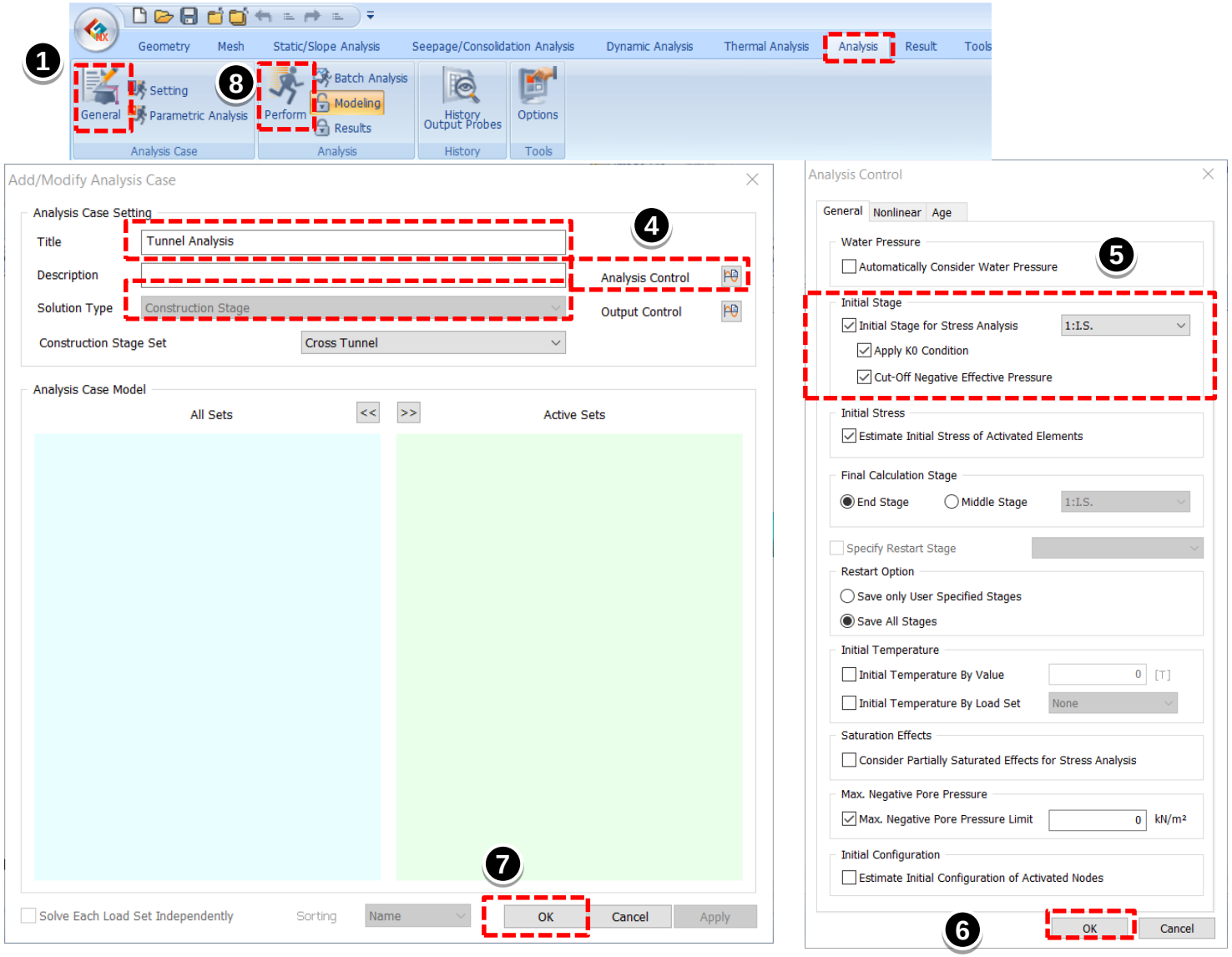
Set Type	Set Name Prefix	I.S.	S1	S2	S3	S4	S5	S6	S7	S8	S9	S10
Load Set	SHIELD PRESSURE -		A: 1	A: 2	A: 3	A: 4	A: 5	A: 6	A: 7	A: 8		
Load Set	CONTRACTION											
Mesh set	Default Mesh Set											
Boundary Set	GBC	A: -										
Boundary Set	GROUT -				A: 1	A: 2,3	A: 4,5	A: 6,7	A: 8,9	A: 10,11	A: 12,13	A: 14,15
Load Set	HP -		A: 1	A: 2	A: 3	A: 4	A: 5	A: 6	A: 7			
Load Set	JT -				A: 1	A: 2 R: 1	A: 3 R: 2	A: 4 R: 3	A: 5 R: 4	A: 6 R: 5	A: 7 R: 6	R: 7
Boundary Set	LINER -				A: 1,2	A: 3,4	A: 5,6	A: 7,8	A: 9,10	A: 11,12	A: 13to15	
Mesh set	LINERS L LOWER-	A: 1to15	R: 1,2	R: 3,4	A: 1,2 R: 5,6	A: 3,4 R: 7,8	A: 5,6 R: 9,10	A: 7,8 R: 11,12	A: 9,10 R: 13,14	A: 11,12 R: 15	A: 13to15	
Mesh set	LINERS L UPPER-	A: 1to15	R: 1,2	R: 3,4	A: 1,2 R: 5,6	A: 3,4 R: 7,8	A: 5,6 R: 9,10	A: 7,8 R: 11,12	A: 9,10 R: 13,14	A: 11,12 R: 15	A: 13to15	
Mesh set	LINERS R LOWER-	A: 1to15	R: 1,2	R: 3,4	A: 1,2 R: 5,6	A: 3,4 R: 7,8	A: 5,6 R: 9,10	A: 7,8 R: 11,12	A: 9,10 R: 13,14	A: 11,12 R: 15	A: 13to15	
Mesh set	LINERS R UPPER-	A: 1to15	R: 1,2	R: 3,4	A: 1,2 R: 5,6	A: 3,4 R: 7,8	A: 5,6 R: 9,10	A: 7,8 R: 11,12	A: 9,10 R: 13,14	A: 11,12 R: 15	A: 13to15	
Mesh set	ROCK	A: -										
Load Set	SEGMENT PRESSURE -				A: 1	A: 2	A: 3	A: 4	A: 5	A: 6	A: 7	
Mesh set	SHIELD-		A: 1,2	A: 3,4	A: 5,6	A: 7,8	A: 9,10	A: 11,12	A: 13,14	A: 15		
Load Set	SW	A: -										
Mesh set	TUNNEL SECTIONS-	A: 1to15	R: 1,2	R: 3,4	R: 5,6	R: 7,8	R: 9,10	R: 11,12	R: 13,14	R: 15		

Step 09

9-1 Analysis Case Definition

Procedure

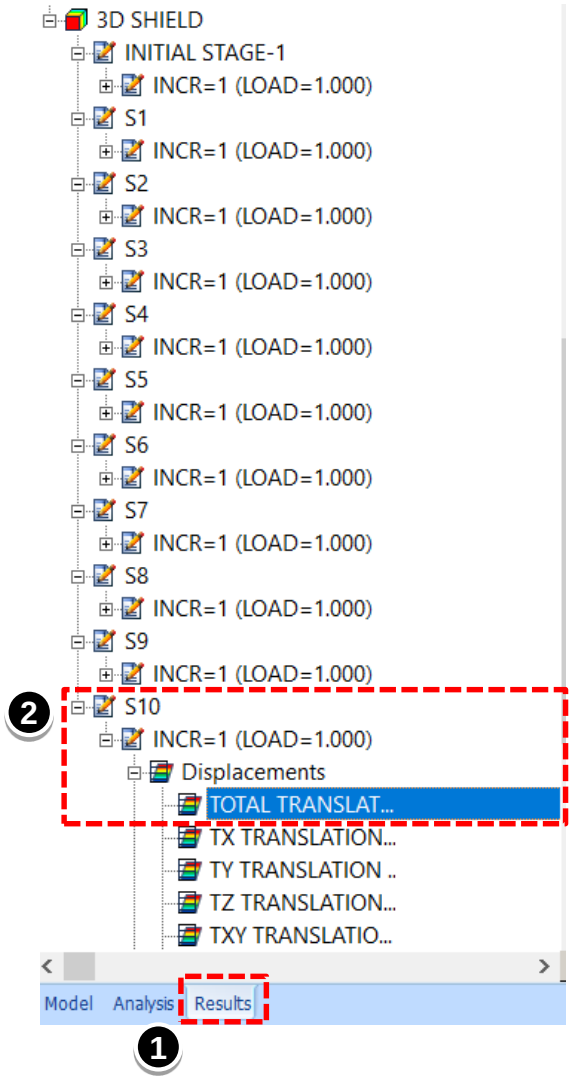
1. In the Analysis Tab, Click on General
2. Name the case as **Tunnel Analysis.**
3. In the Solution Type Drop-Down Menu, **Construction Stage.**
4. Click on **Analysis Control.**
5. Click in **Initial Stage**, and select **both** the options.
6. Click **OK**.
7. Click **OK.**
8. Click **Perform.**

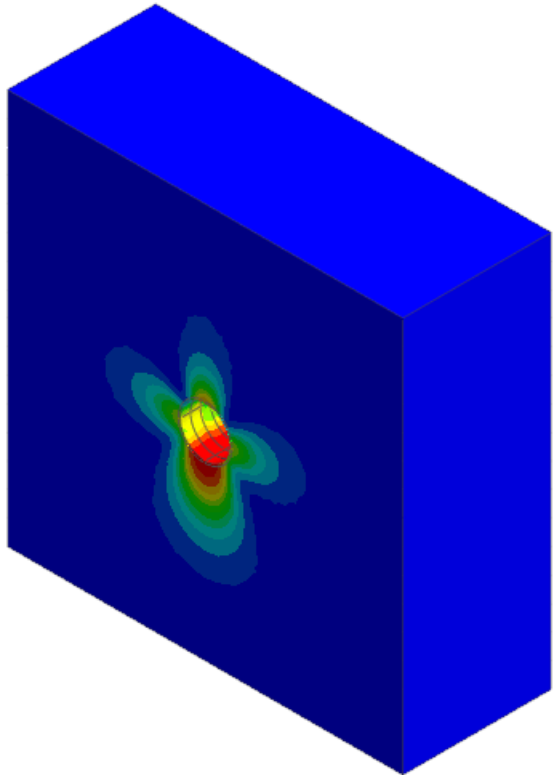


10-1 Result (DISPLACEMENTS)

Procedure

1. Go to Results> 3D SHIELD > S10.
2. Click on Total Displacements.

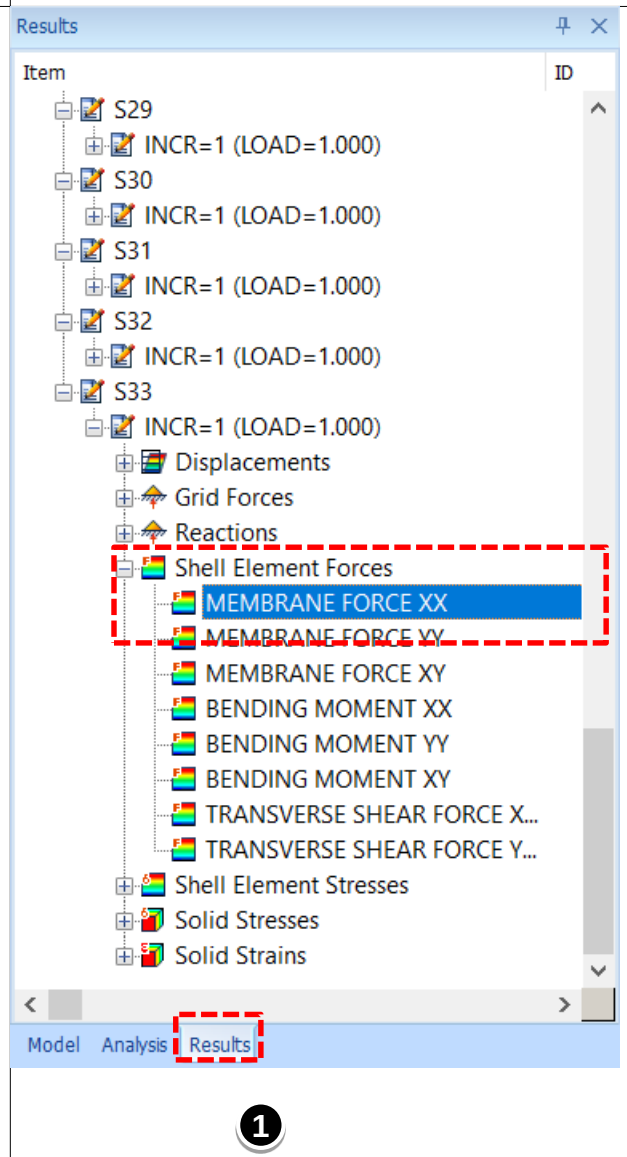


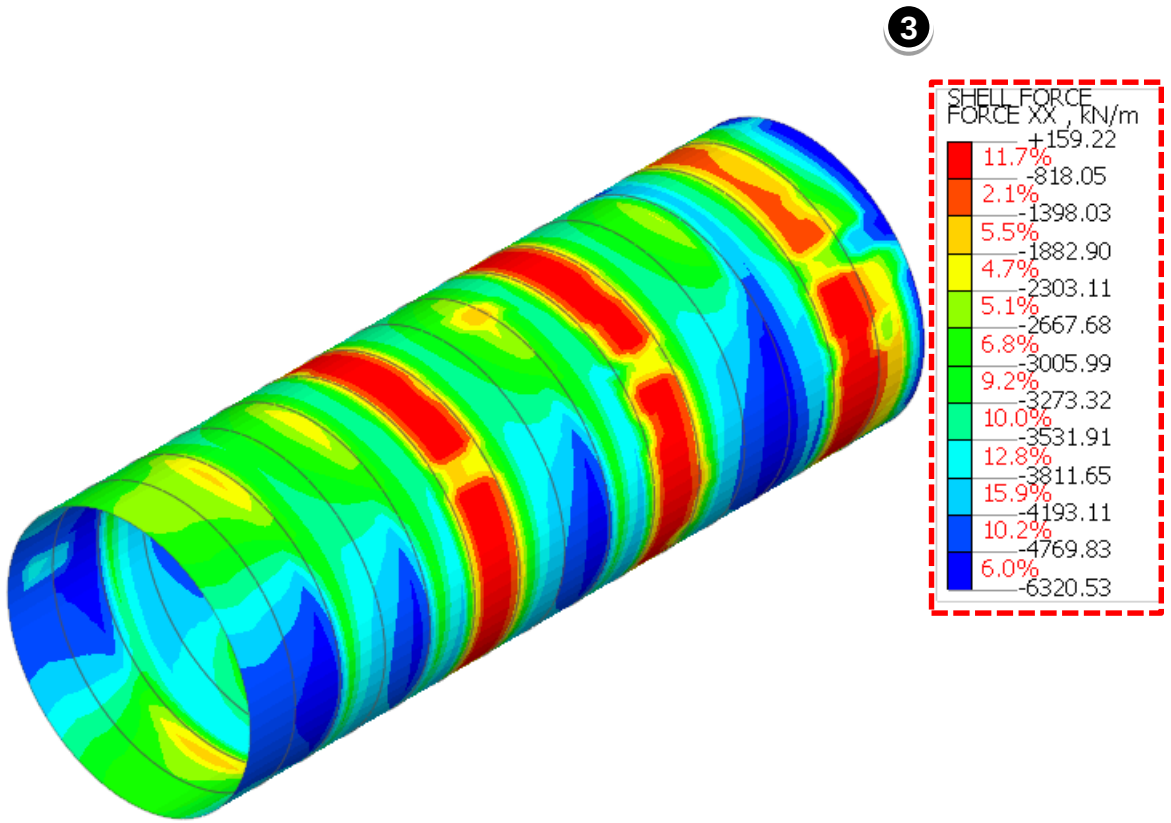


10-3 Result (SHELL ELEMENT FORCES)

Procedure

1. Go to Results> 3D SHIELD > S10.
2. Click on SHELL ELEMENT FORCES.
3. Legend shows the forces acting on the Shell/Shield..

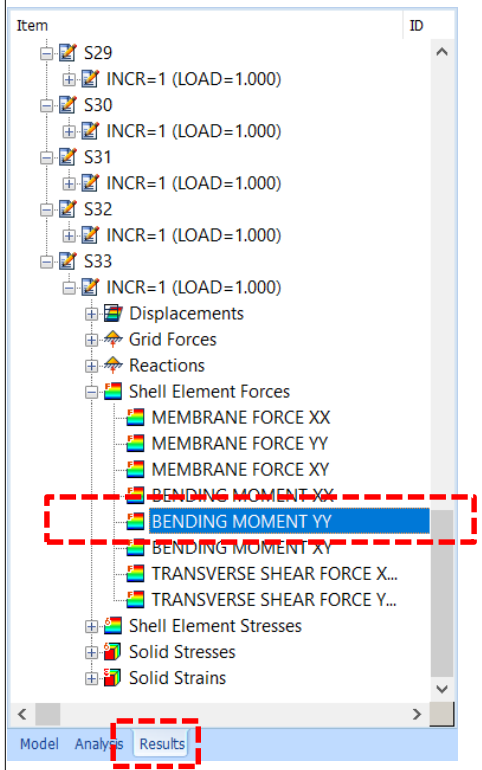


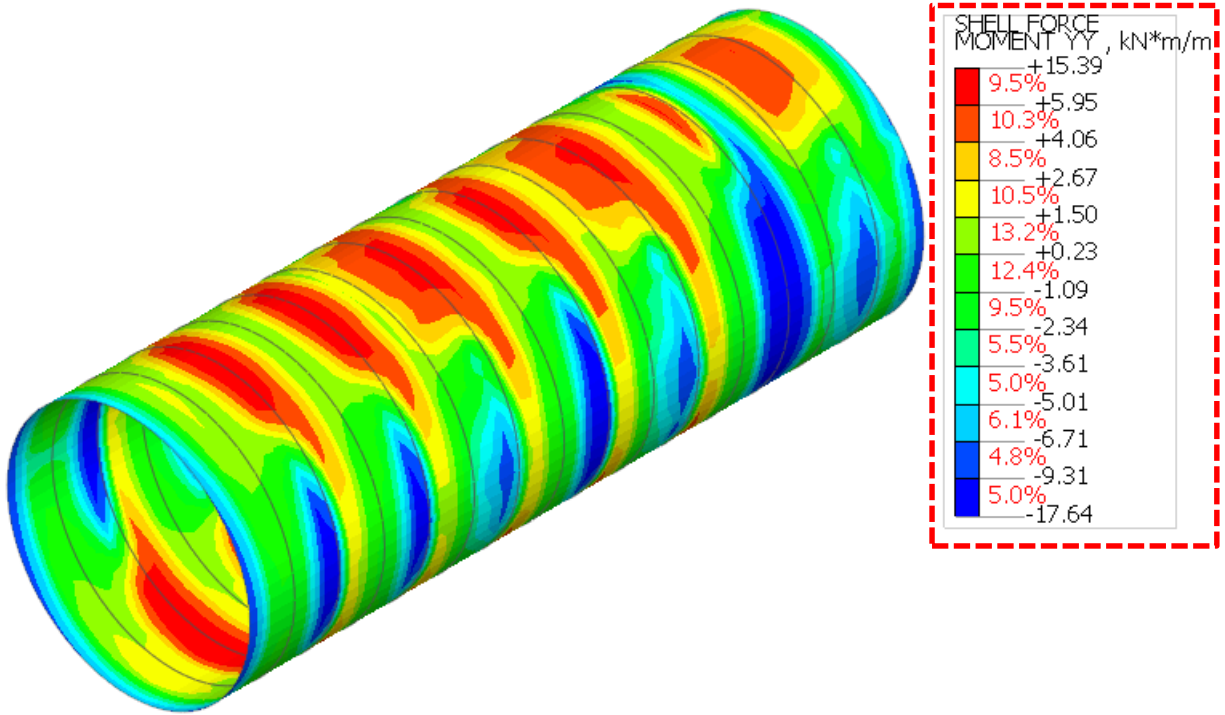


10-3 Result (BENDING MOMENTS ON SHELL)

Procedure

1. Go to Results > 3D SHIELD > S10.
2. Click on BEAM ELEMENT FORCES & select **Bending Moment YY.**
3. Legend shows the bending moments acting on the Shield.



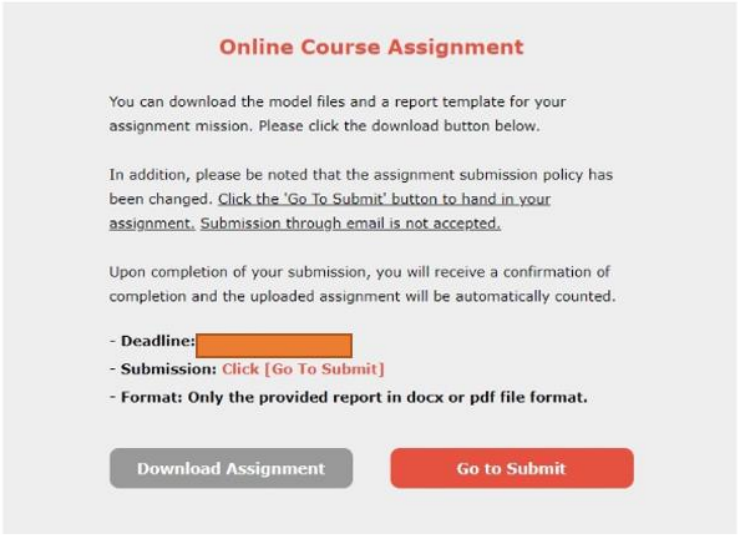


CERTIFICATION TASK

- 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.

CERTIFICATION TASK GUIDELINES



MODEL DATA & TASK FILES & INSTALLATION DATE – 11th NOVEMBER

SUBMISSION ONLY VIA LINK, MAIL SUBMISSIONS NOT ACCEPTED

SUBMISSION DATE : till 26TH NOVEMBER