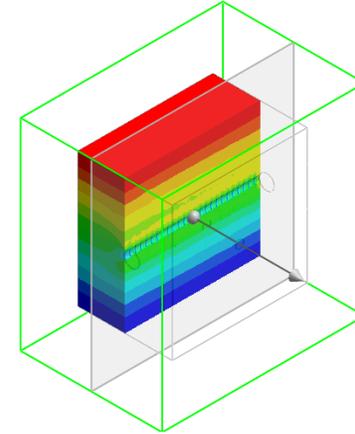


MIDAS *Technical
Material*

Tutorial



3D SHIELD TBM TUNNEL

Contents

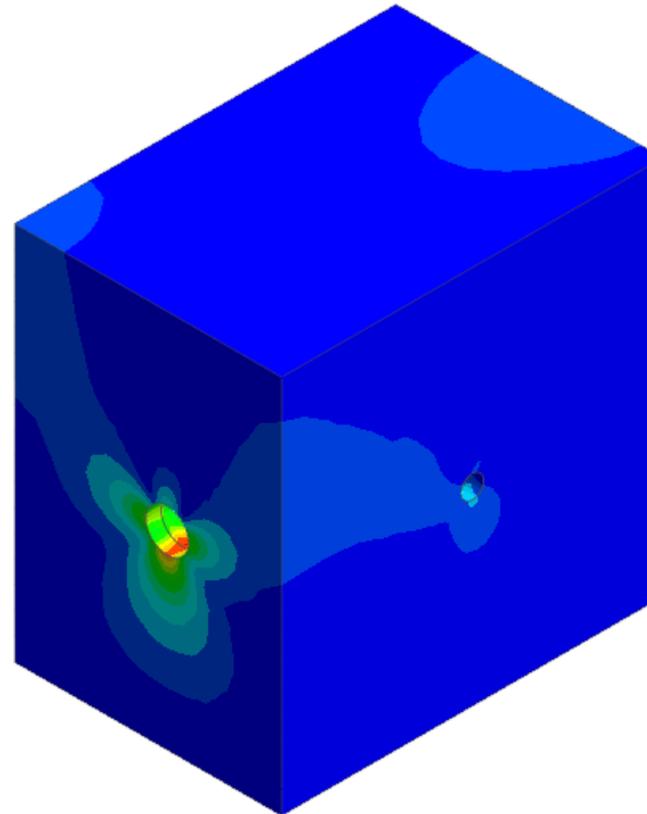
- **Step 1:** Initial Setting
- **Step 2:** Defining Soil material
- **Step 3:** Defining Property
- **Step 4:** Geometric Modeling
- **Step 5:** Meshing
- **Step 6:** Boundary Condition
- **Step 7:** Loads Definition
- **Step 8:** Analysis Case
- **Step 9:** Analysis
- **Step 10:** Results
- **Step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

Numerical Model Design

Overview

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

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

3D Shield Tunnel

Material Properties

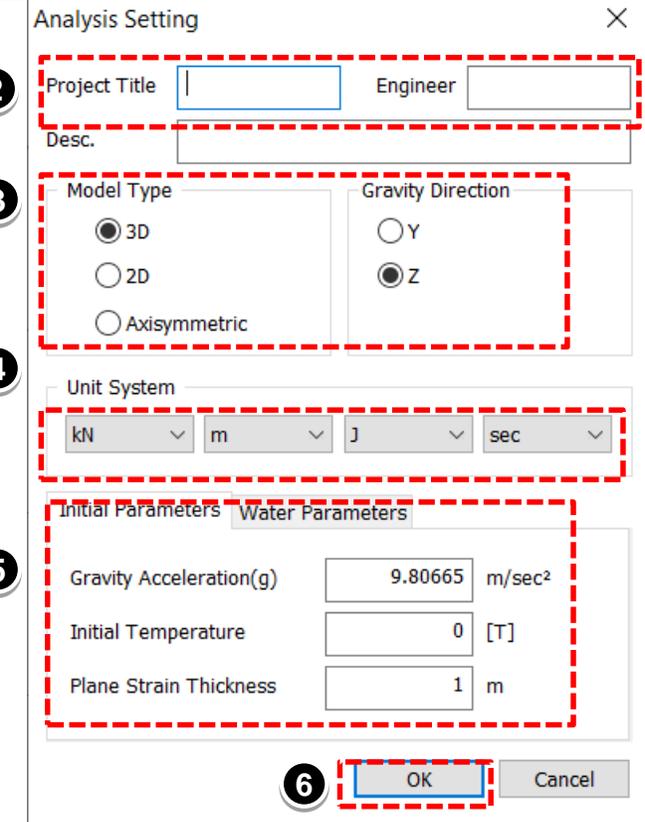
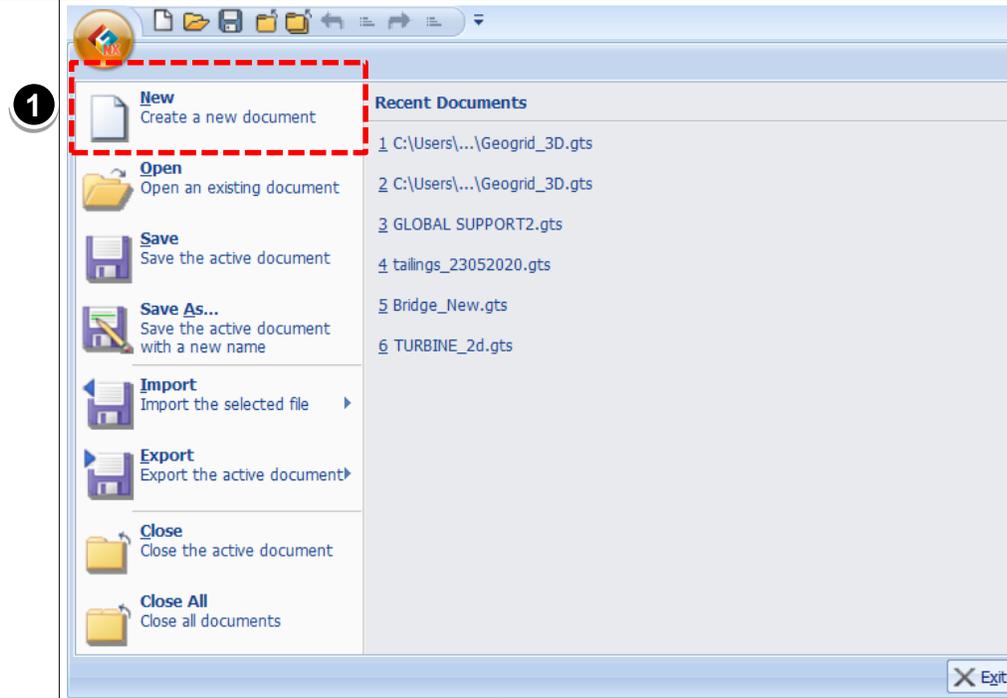
Name	Material Model	Modulus of Elasticity (kN/m ²)	Poisson's ratio	Unit Weight (kN/m ³)	Saturated unit Weight (kN/m ³)	Cohesion (kN/m ²)	Friction angle
SOFT ROCK	Mohr-Coulomb	250000	0.3	22	23	20	38

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

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**.



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**.

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...' button and a dropdown menu with 'Isotropic' selected. The 'Material' properties dialog box is also open, showing the 'General' tab with the following settings:

- ID: 2
- Name: SOFT ROCK
- Color: (pink)
- Model Type: Mohr-Coulomb
- Structure:
- General tab selected
- Elastic Modulus(E): 250000 kN/m²
- Inc. of Elastic Modulus: 0 kN/m³
- Inc. of Elastic Modulus Ref. Height: 0 m
- Poisson's Ratio(ν): 0.3
- Unit Weight(γ): 22 kN/m³
- Initial Stress Parameters:
 - Ko Determination: 0.384338525
 - Automatic:
 - Manual:
 - Anisotropy:
- Thermal Parameter:
 - Thermal Coefficient: 1e-006 1/[T]
 - Molecular vapor diffusion coefficient: 0 m²/sec
 - Thermal diffusion enhancement: 0
- Damping Ratio(For Dynamic):
 - Damping Ratio: 0.05
- Safety Result(Mohr-Coulomb):
 - Cohesion(C): 30 kN/m²
 - Frictional Angle(Φ): 36 [deg]
 - Tensile Strength: 0 kN/m²

2-2 Defining Soil Material

Procedure

- ① Go to Porous Tab > Enter Unit weight (saturated) = 23 kN/m³
- ② Keep Drainage Parameters as Drained
- ③ Go to Non-Linear Tab > Enter the value of Cohesion = 20 kN/m²
- ④ Input the value of Frictional angle = 38°
- ⑤ Enter the name as SOFT ROCK.
- ⑥ Click OK

Material

ID: 2 Name: SOFT ROCK 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

Unified Failure Ratio

Skempton's B Coefficient

Seepage & Consolidation Parameters

Permeability Coefficients

lx	ky	kz	
<input type="text" value="1e-005"/>	<input type="text" value="1e-005"/>	<input type="text" value="1e-005"/>	m/sec

Void Ratio Dependency of Permeability(ck)

Specific Storativity(Ss) 1/m

Material

ID: 2 Name: SOFT ROCK 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

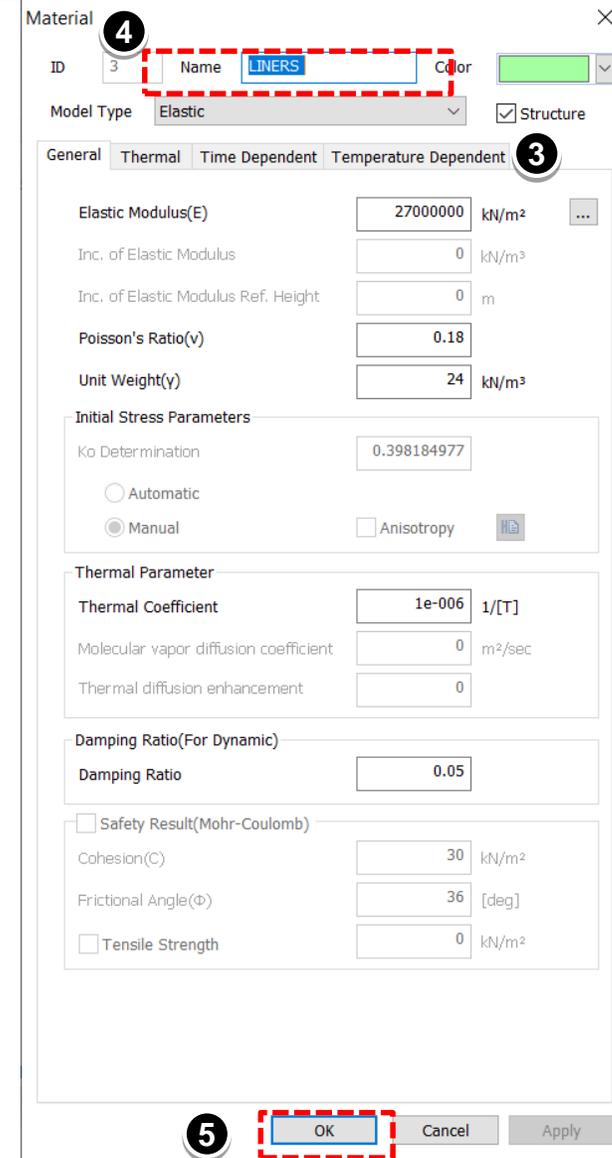
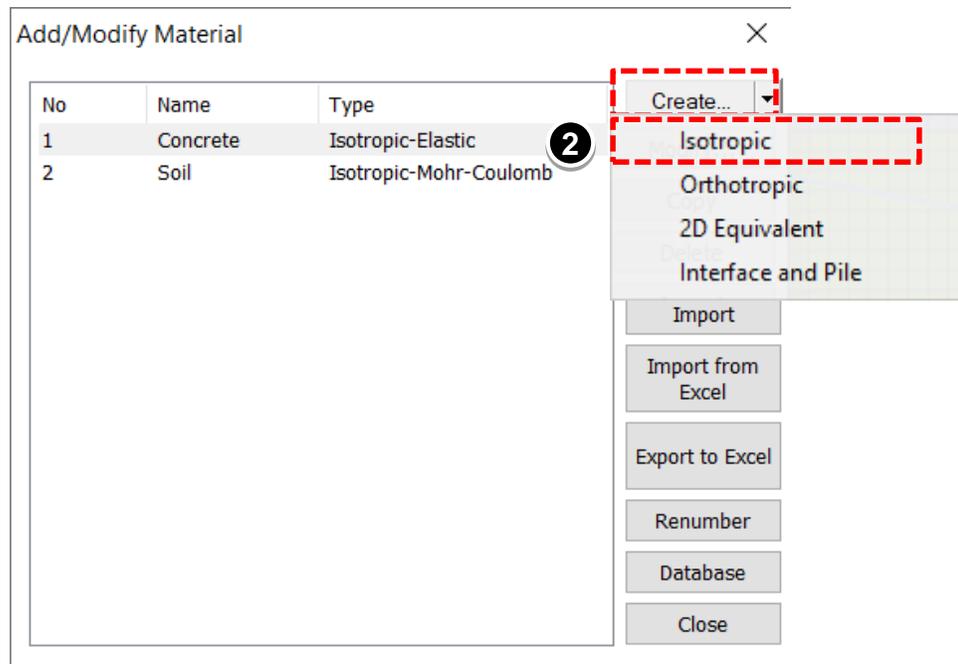
⑥

Procedure

Defining Soil Materials

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

SIMILARLY CREATE MATERIALS FOR STEEL SHIELD & GROUT.



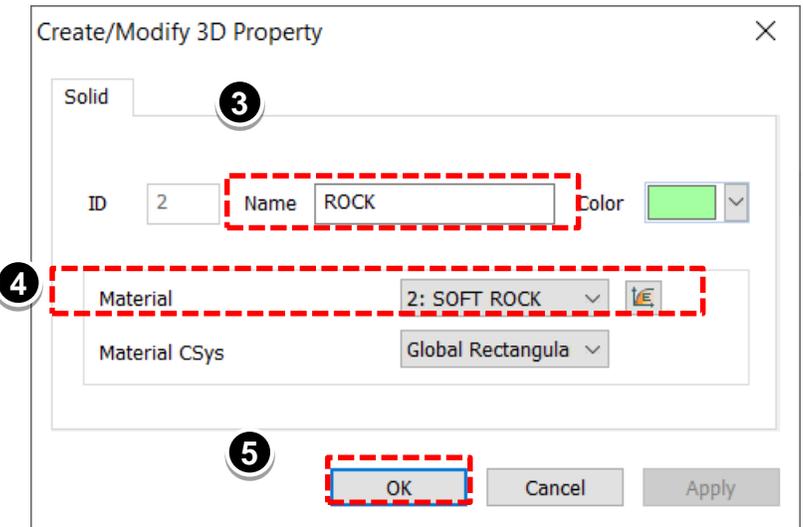
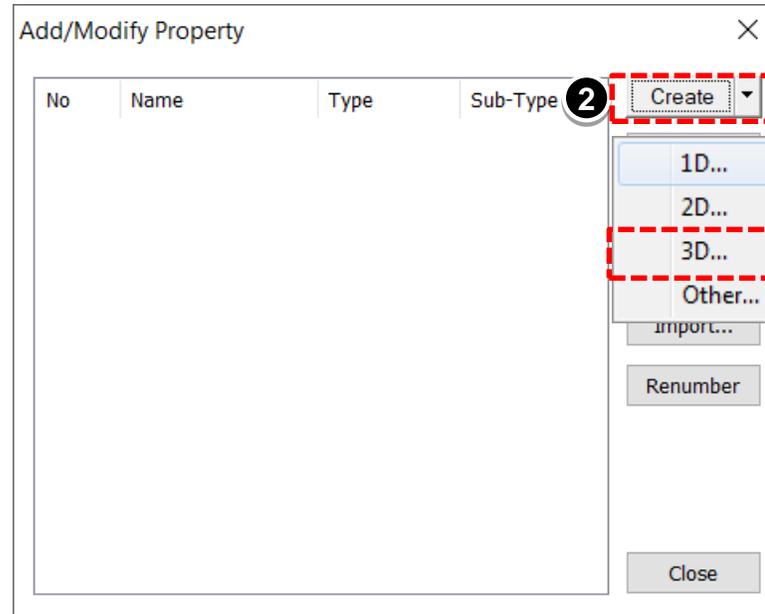
3-1 Defining 3D Property (SOIL)

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.



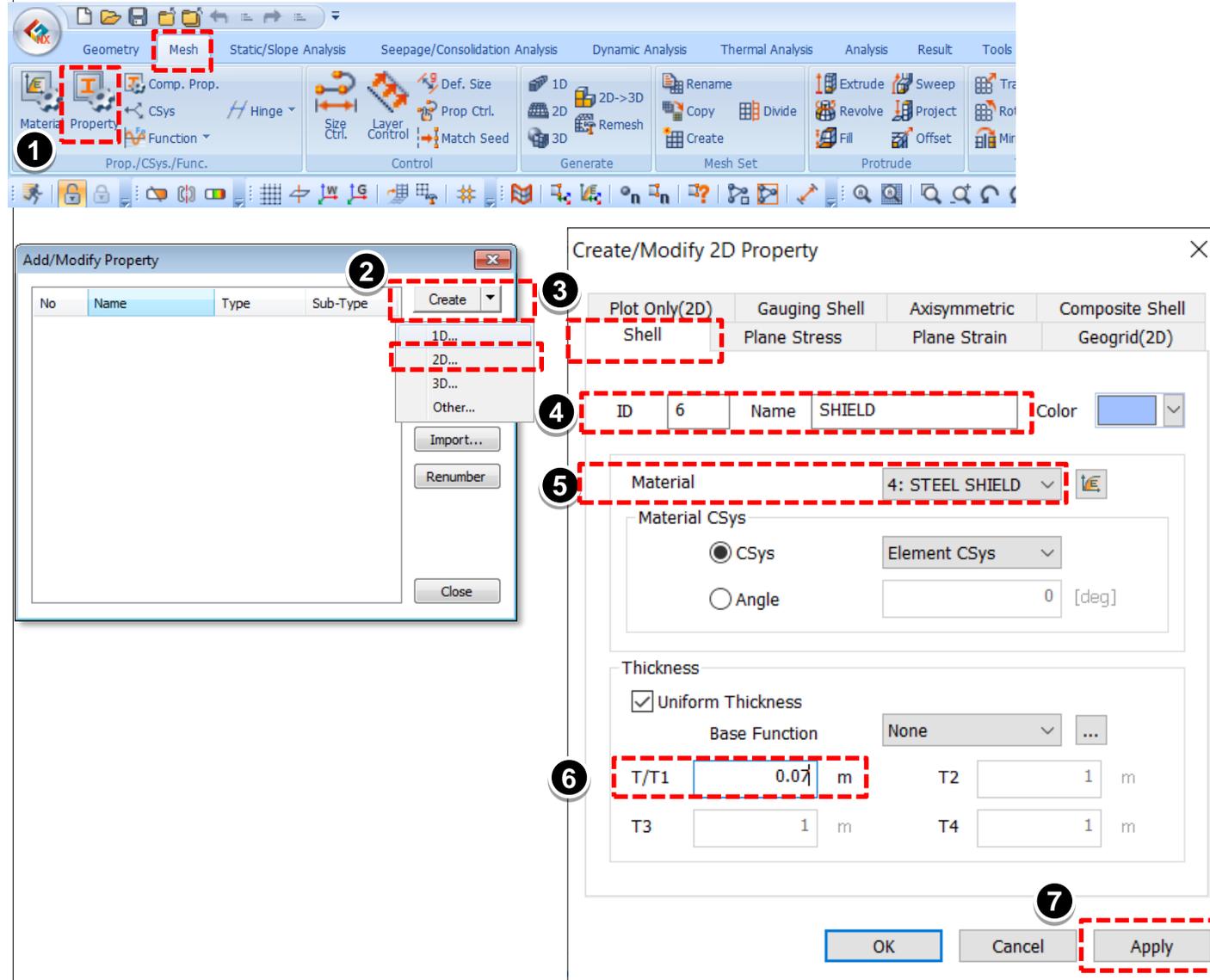
3-2 Defining 2D Shield Property

Procedure

Defining Property

- ➊ Go to Mesh>Click on Property
- ➋ Click on Create. Select 2D
- ➌ Click on **Shell**.
- ➍ Name it as **SHIELD**.
- ➎ Select material as **STEEL SHIELD**.
- ➏ Enter T/T1 as **0.07 m**.
- ➐ 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.

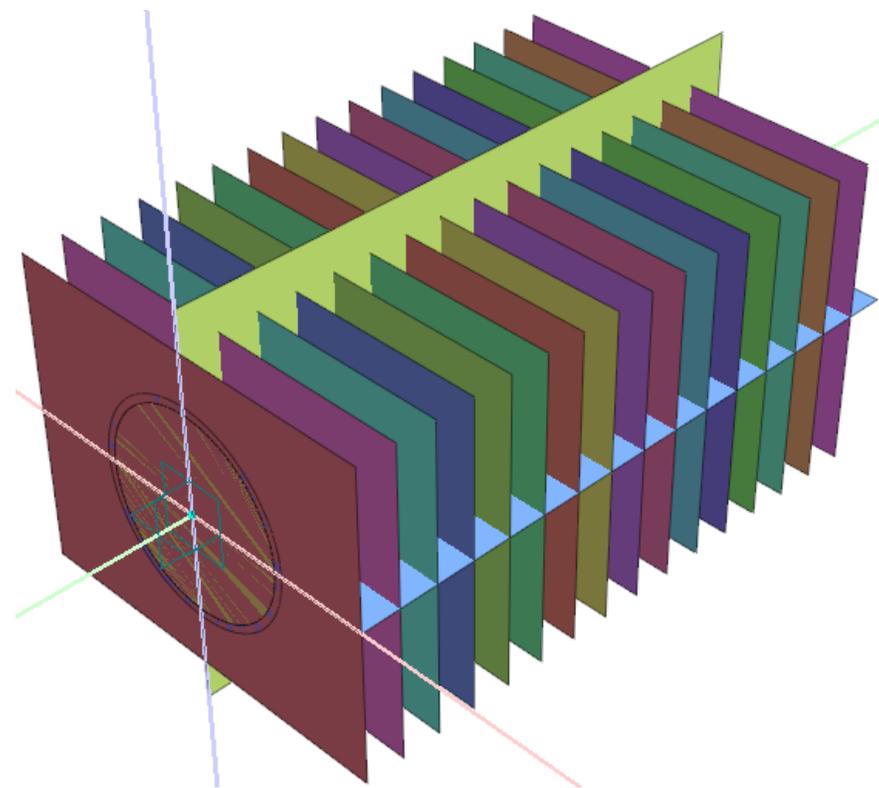
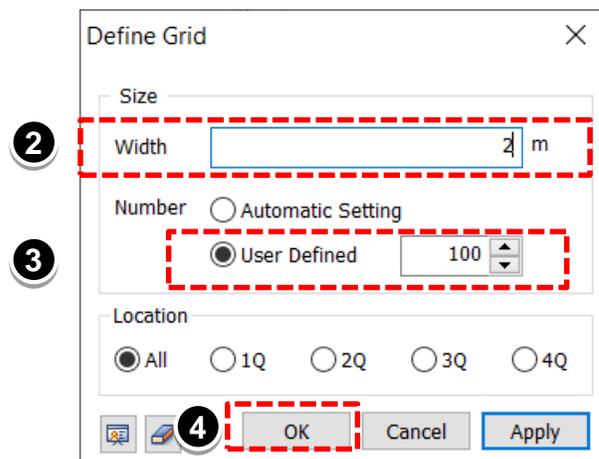
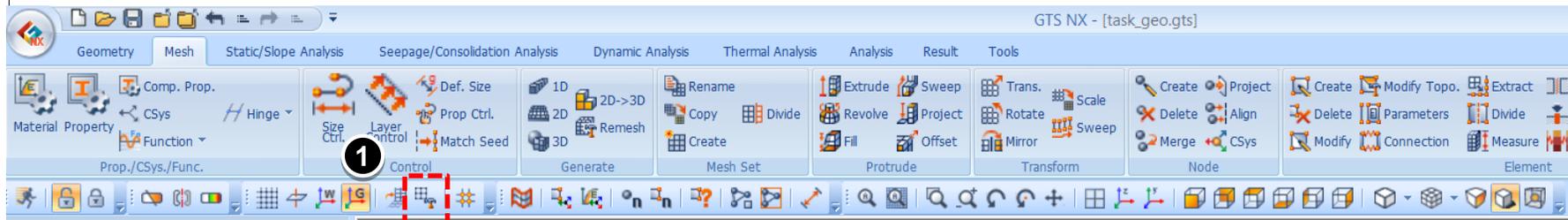


4-1 Geometric Modelling- Start File & Redefining Grid Size

Procedure

Open the Start File “**shield_start**”

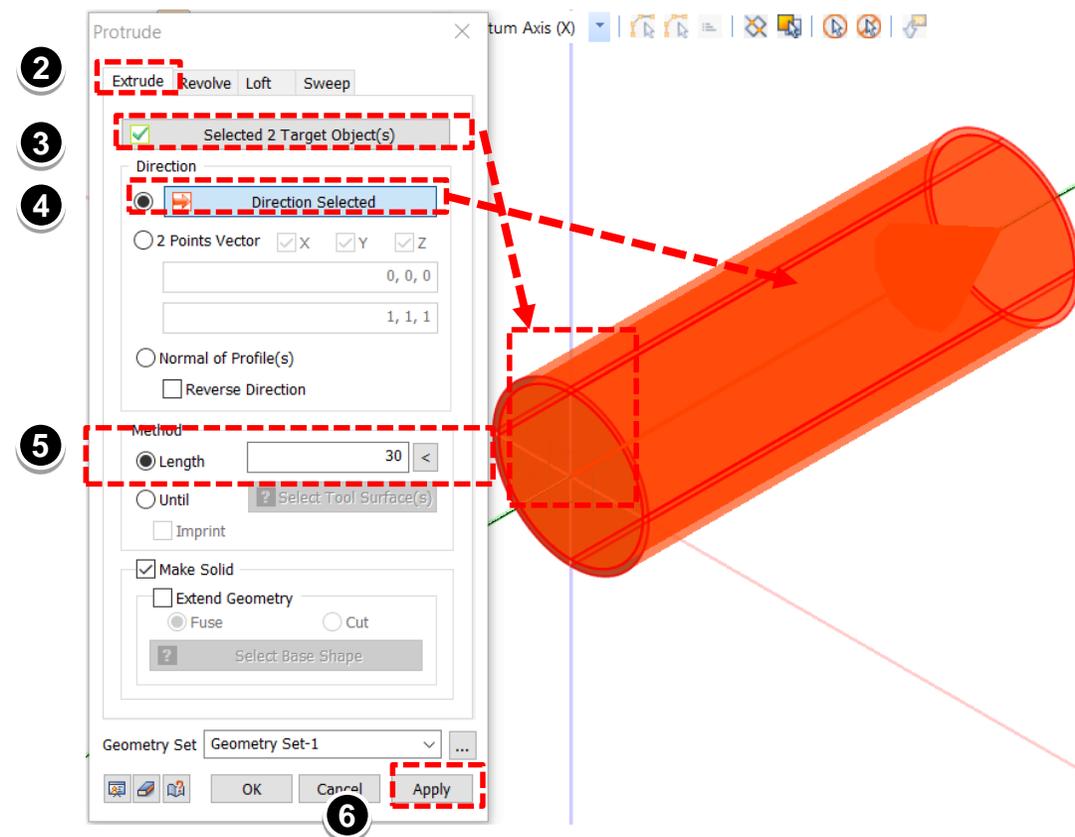
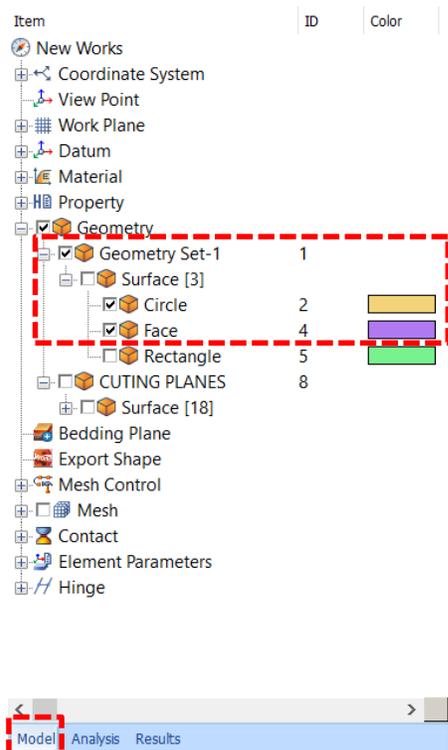
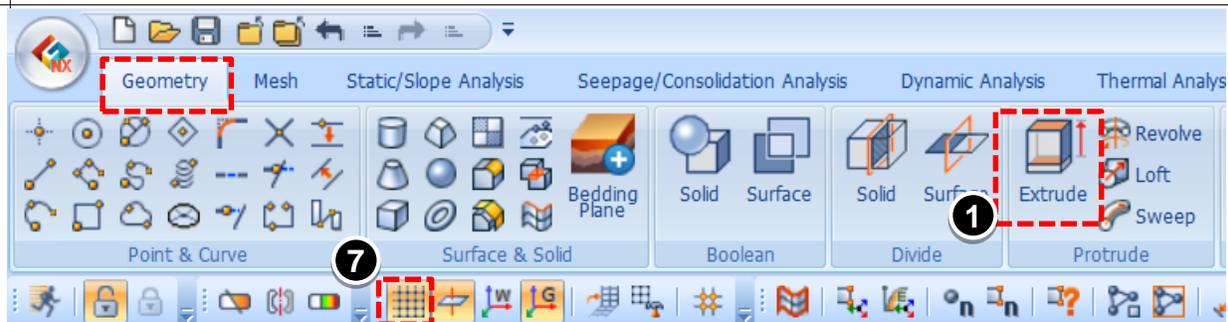
- 1 Go to Grid Size.
- 2 Enter width as **2m**.
- 3 Click **user defined** and enter **100**.
- 4 Click **OK**.



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.



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.

The screenshot displays the Midas User software interface. The top toolbar shows the 'Geometry' tab selected, with the 'Divide' > 'Solid' tool highlighted. The works tree on the left shows the following structure:

- Item
- ID
- Color
- C:\Users\Chinmoy\Desktop\MID...
- Coordinate System
- View Point
- Work Plane
- Datum
- Material
- Property
- Geometry
 - Geometry Set-1 1
 - Surface [3]
 - Circle 2
 - Face 4
 - Rectangle 5
 - Solid [2]
 - Extrude 602
 - Extrude 603
 - CUTTING PLANES 8
 - Surface [18]
 - Bedding Plane
 - Export Shape
 - Mesh Control
 - Mesh
 - Contact
 - Element Parameters
 - Hinge

The 'Divide Solid' dialog box is open, showing the following settings:

- By Surface
 - Selected 1 Target Solid(s)
- Dividing Tools
 - Selected 18 Tool Surface(s)
 - 3 Points Plane
 - X: 1, 0, 0
 - Y: 0, 1, 0
 - Z: 0, 0, 1
 - Dividing Plane
 - X Y Z
 - Distance: 0 m
- Divide Touching Faces (Select Object(s))
- Delete Original
- Delete Tool
- Geometry Set: segments
- Buttons: OK, Cancel, Apply

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.

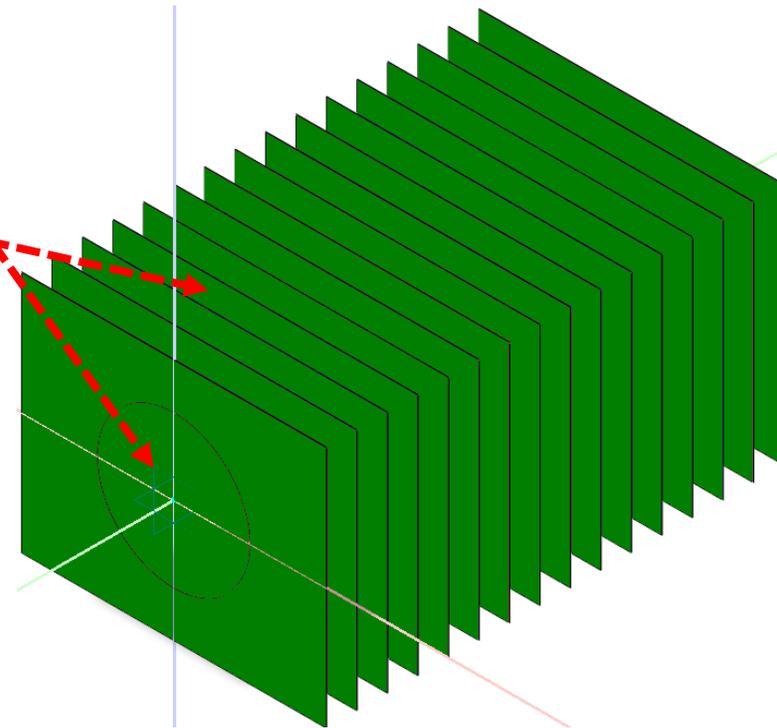
④ Click on **both** options.

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.

The screenshot shows the Midas Gen software interface. The top ribbon has tabs for Geometry, Mesh, Static/Slope Analysis, Seepage/Consolidation Analysis, Dynamic Analysis, and Thermal Analysis. The 'Divide Solid' dialog box is open, showing the 'By Surface' tab. The 'Selected 1 Target Solid(s)' and 'Selected 16 Tool Surface(s)' options are checked. The 'Delete Original' and 'Delete Tool' options are also checked. The 'Geometry Set' is set to 'TUNNEL SECTIONS'. The 'OK' button is highlighted. The works tree shows 'Geometry Set-1' with 'Surface [3]' and 'Solid [1]' selected. The 'Cutting Planes' folder is expanded, showing 16 'Rectangle' surfaces. The 'Divide Solid' dialog box is annotated with numbered red dashed boxes corresponding to the procedure steps.

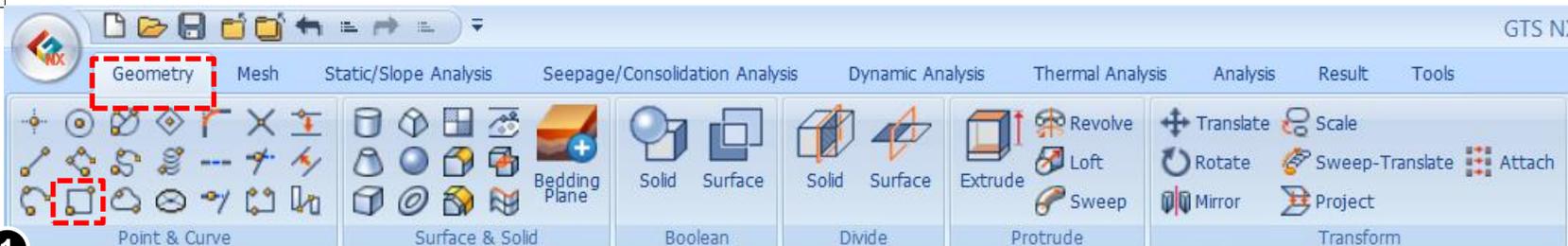


4-5 GROUND FACE & SOLIDS CREATION

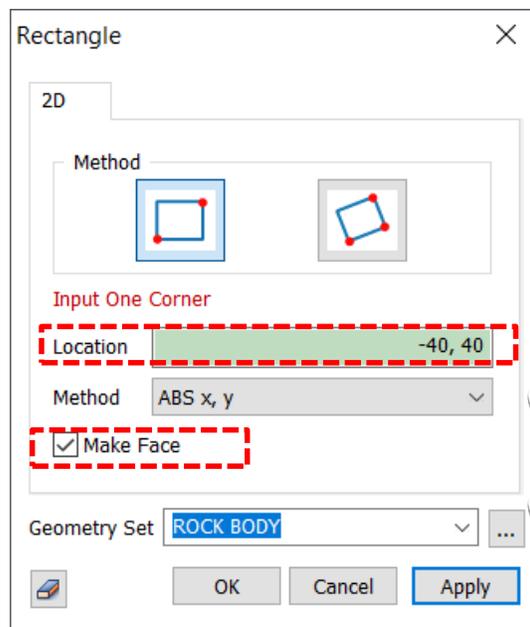
Procedure

- 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.

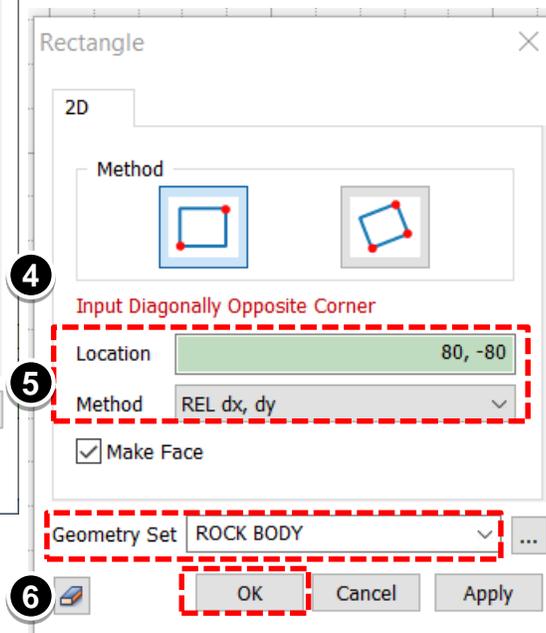


1



2

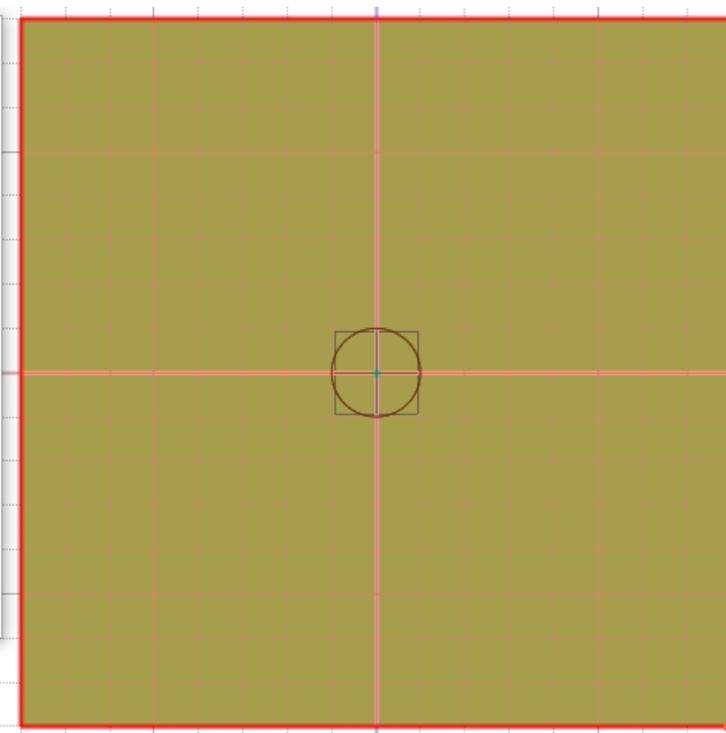
3



4

5

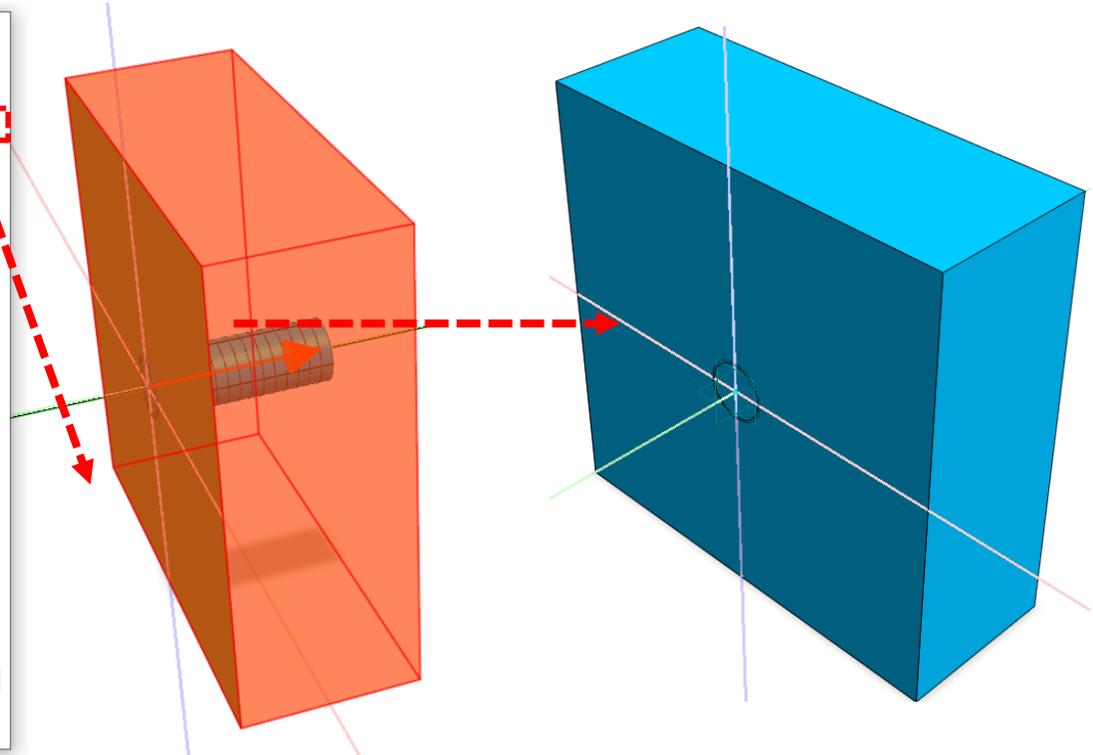
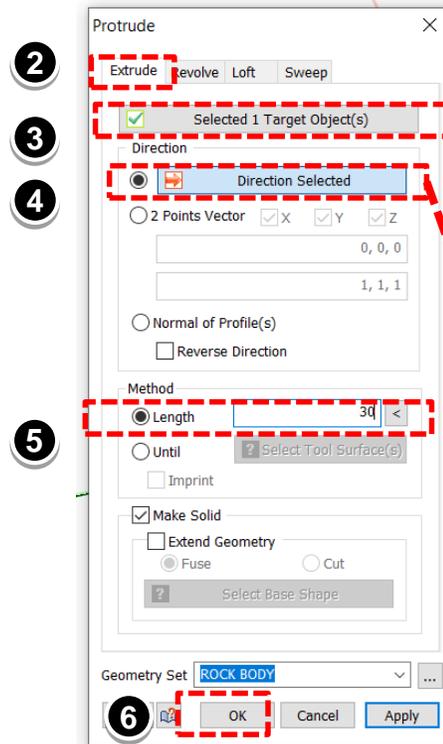
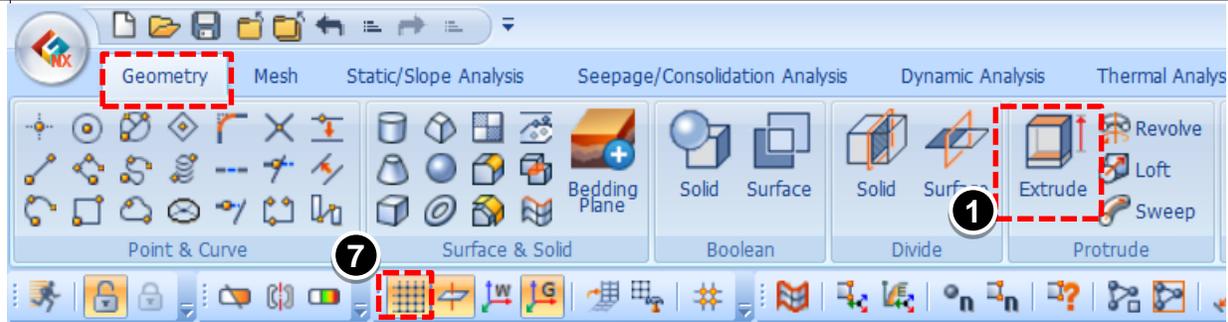
6



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

Procedure

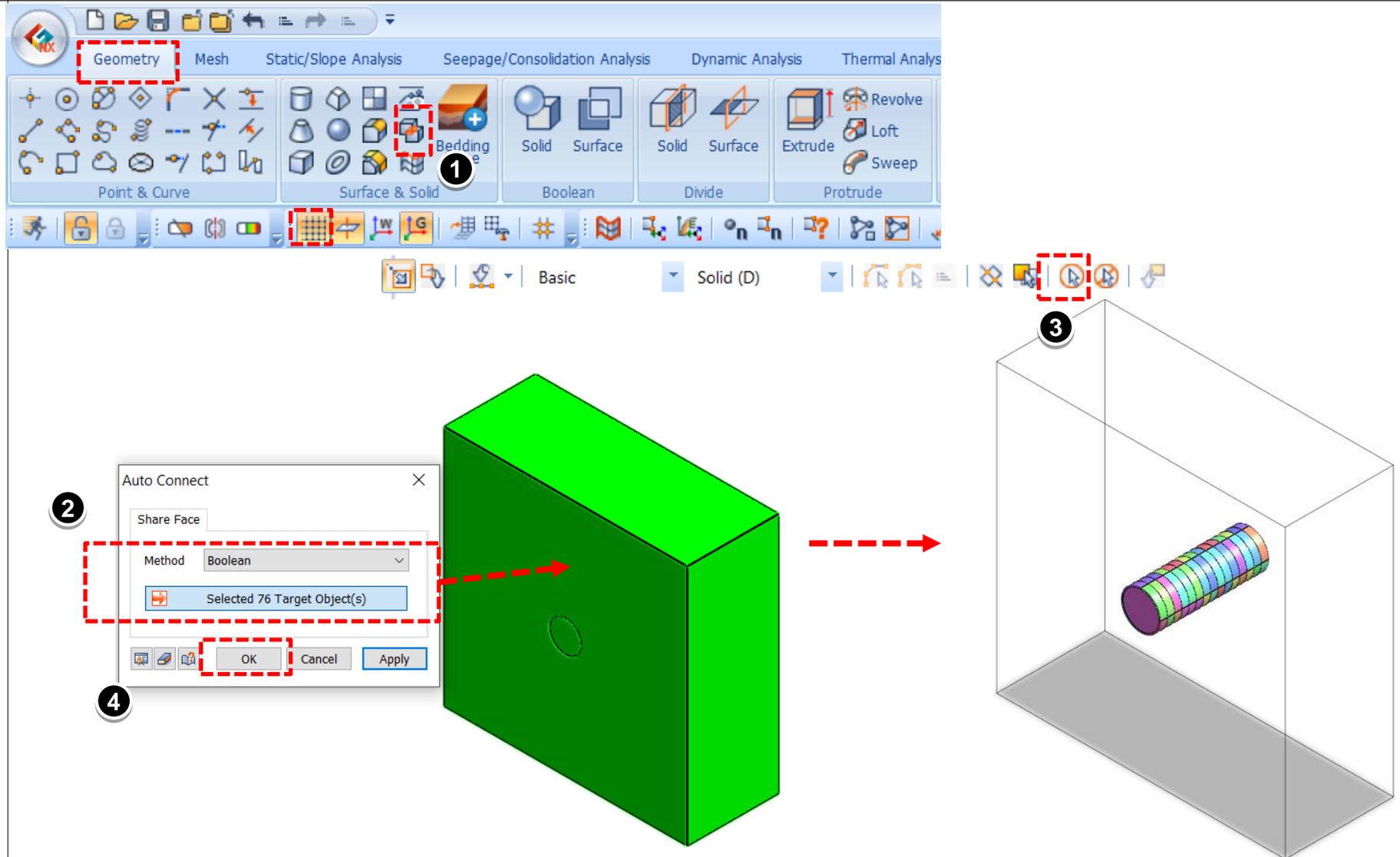
- ① Go to Geometry>Protrude>Extrude
- ② Select Extrude.
- ③ Select the **ground face** as target object.
- ④ Click on the **Y-axis** as selected direction.
- ⑤ Enter length as **30m**.
- ⑥ Click **OK**.
- ⑦ Click to activate/deactivate the grid.



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.

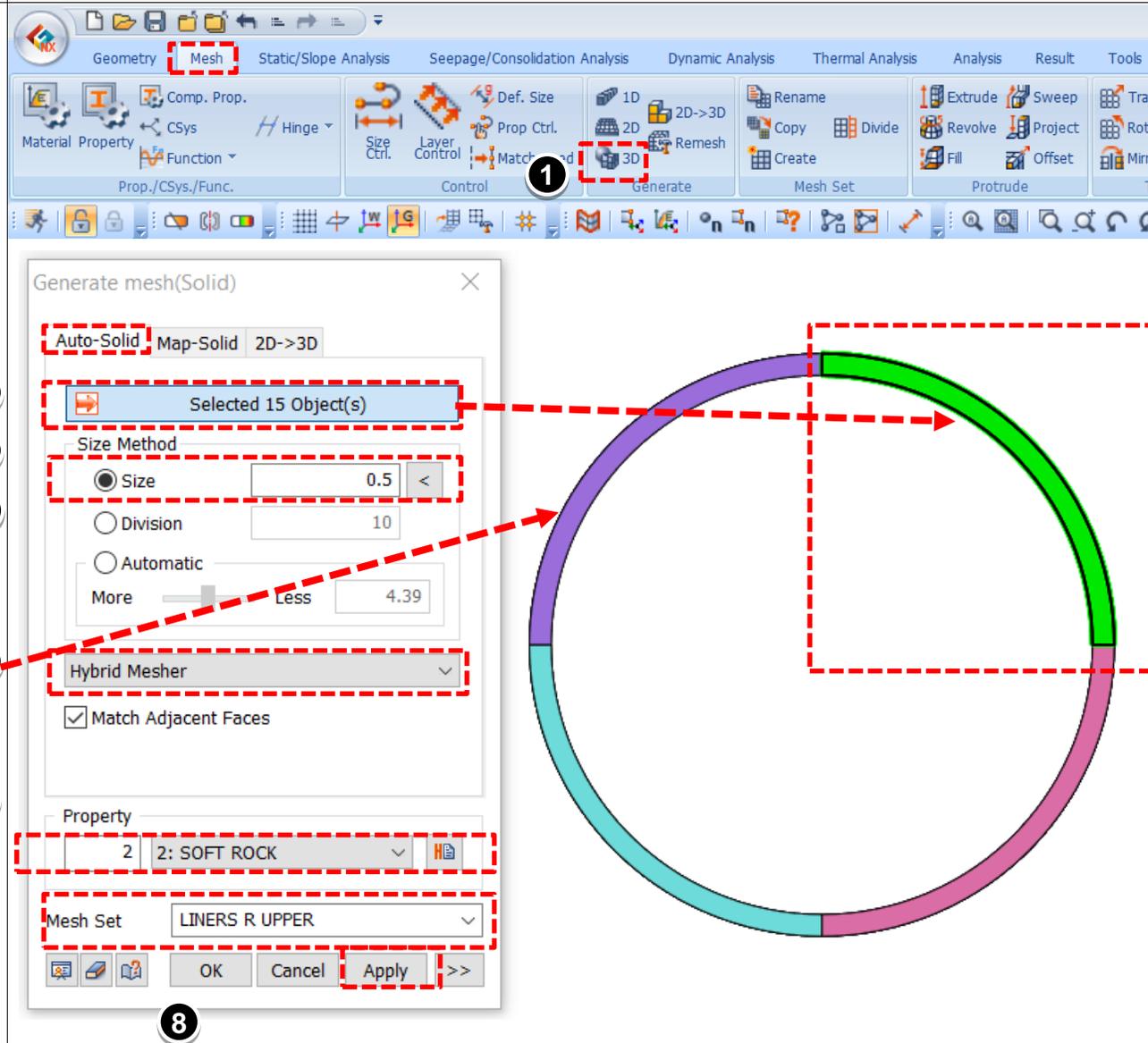


5-1 Meshing Liner Segments

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.

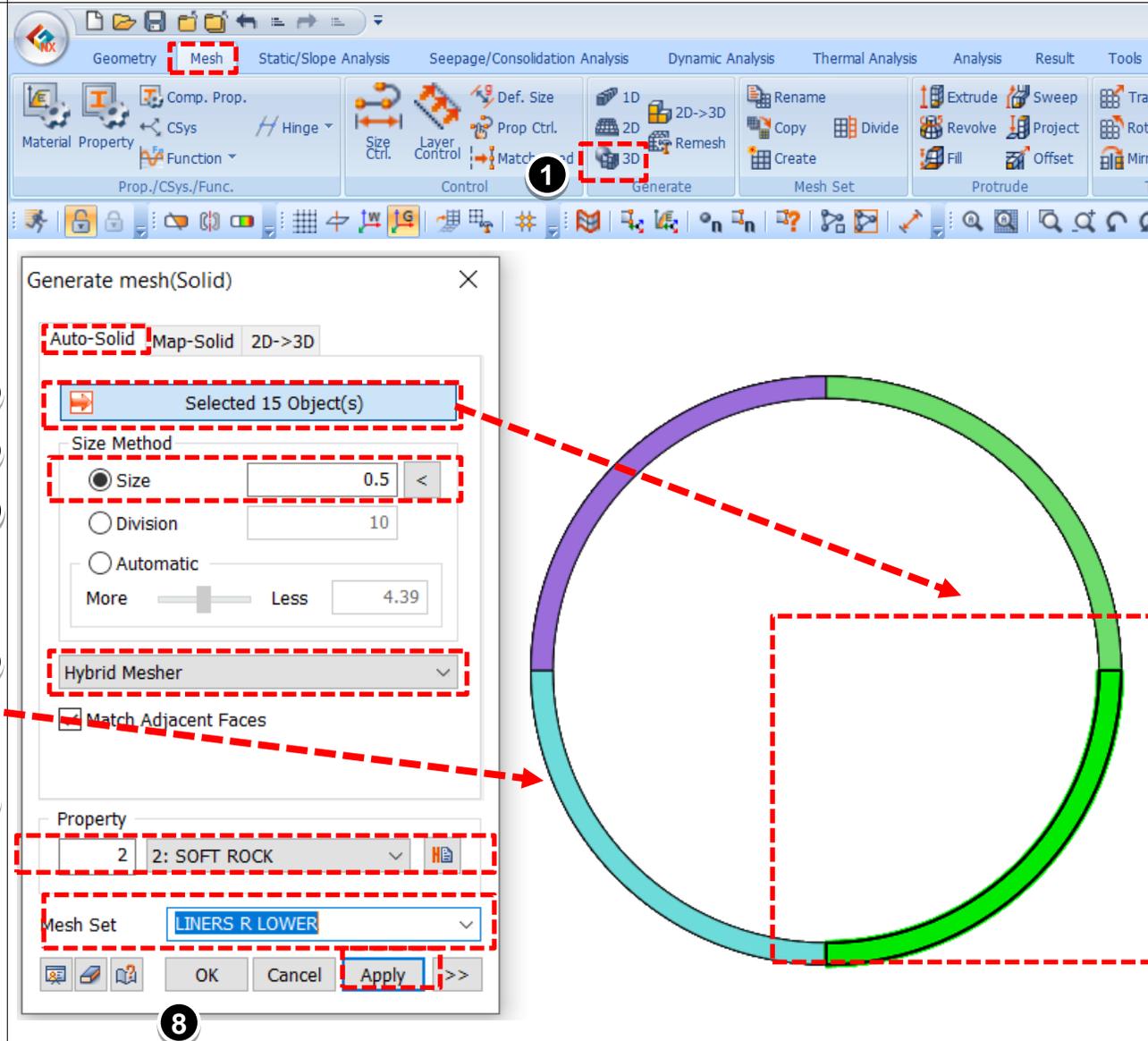


5-2 Meshing Liner Segments

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**.

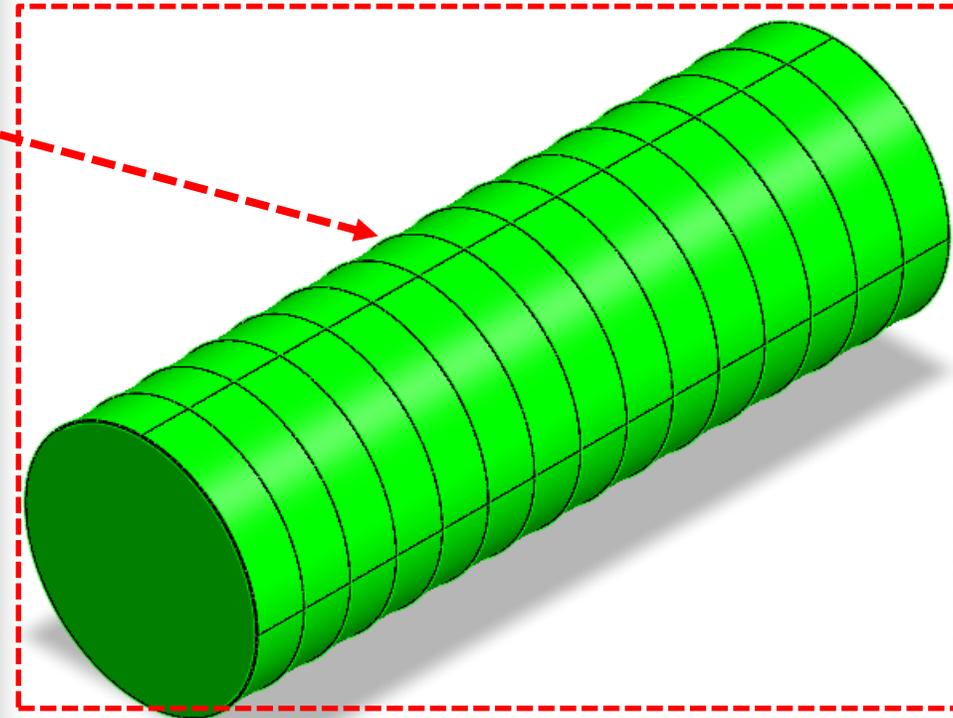
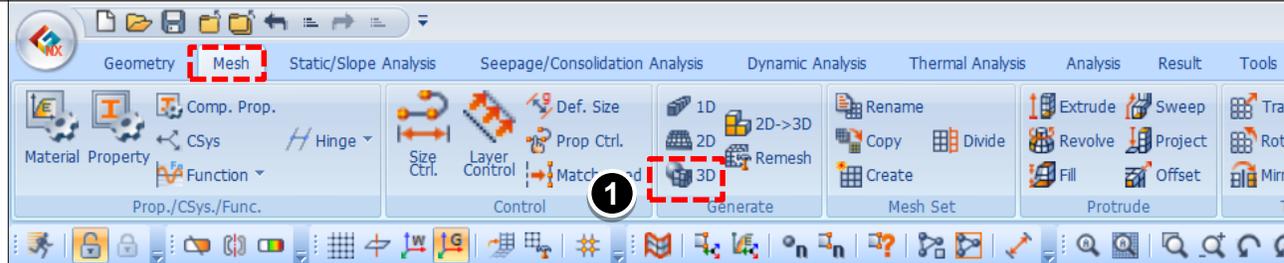
SIMILARLY CREATE THE MESH SETS FOR LINER IN LOWER UPPER SECTION & NAME THE MESH SET AS LINERS L LOWER.



5-3 Meshing Tunnel Sections

Procedure

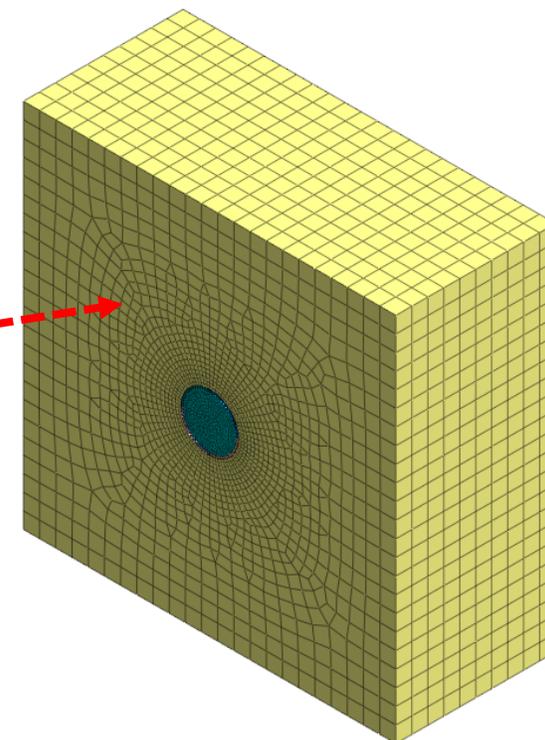
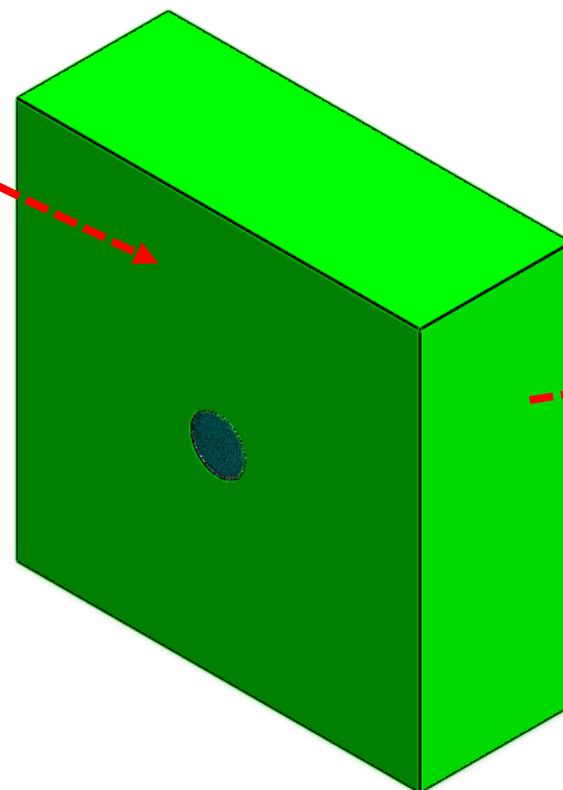
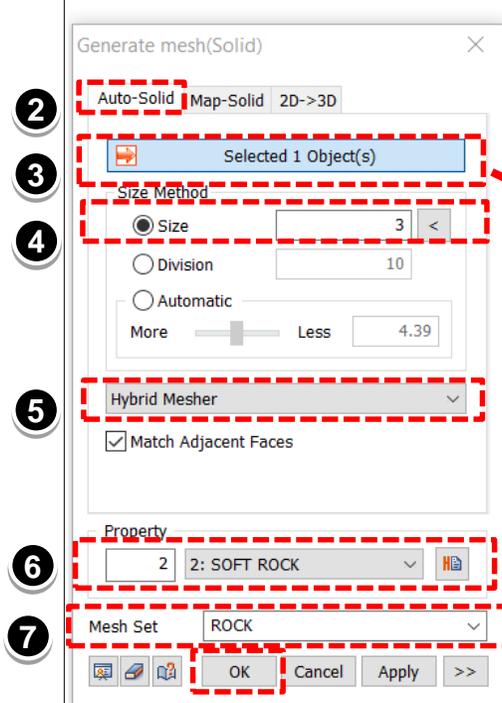
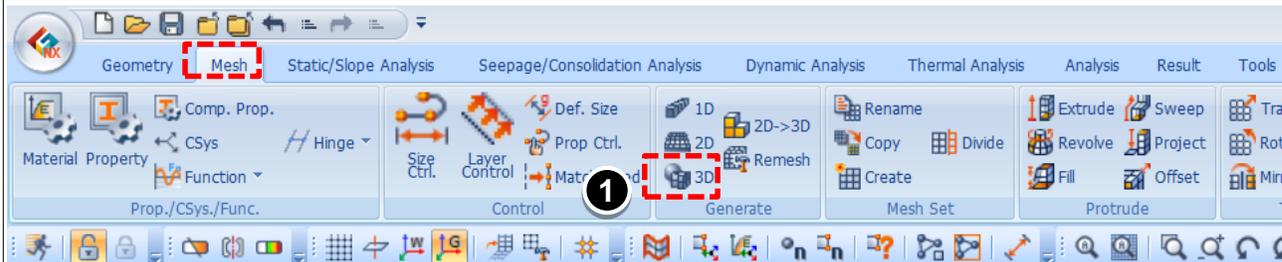
- ❶ Go to Mesh > Generate > 3D
- ❷ Select Auto-Solid.
- ❸ Select the 15 segments as shown in the picture.
- ❹ In the Size method give size 0.5 m.
- ❺ Select Hybrid Mesher as the mesher type.
- ❻ In the property drop-down menu, select **Soft Rock**.
- ❼ Give the Mesh set name "TUNNEL SECTIONS." GTS NX will automatically assign the names to the solid sections selected.
- ❽ Click **Apply**.



5-4 Meshing Ground Body

Procedure

- ❶ Go to Mesh > Generate > 3D
- ❷ Select Auto-Solid.
- ❸ Select the 9 segments of cross passage.
- ❹ In the Size method give size 3 m.
- ❺ Select Hybrid Mesher as the mesher type.
- ❻ In the property drop-down menu, select **SOFT ROCK**.
- ❼ Give the Mesh set name **ROCK**.
- ❽ Click **OK**.



8

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.

The screenshot displays the GTS NX software interface for a file named 'Geogrid_3D.gts'. The 'Mesh' tab is active in the top ribbon. A red dashed box highlights the 'Extract' button in the 'Element' group. Below the ribbon, the 'Extract Element' dialog box is open, with several fields highlighted by red dashed boxes and numbered steps:

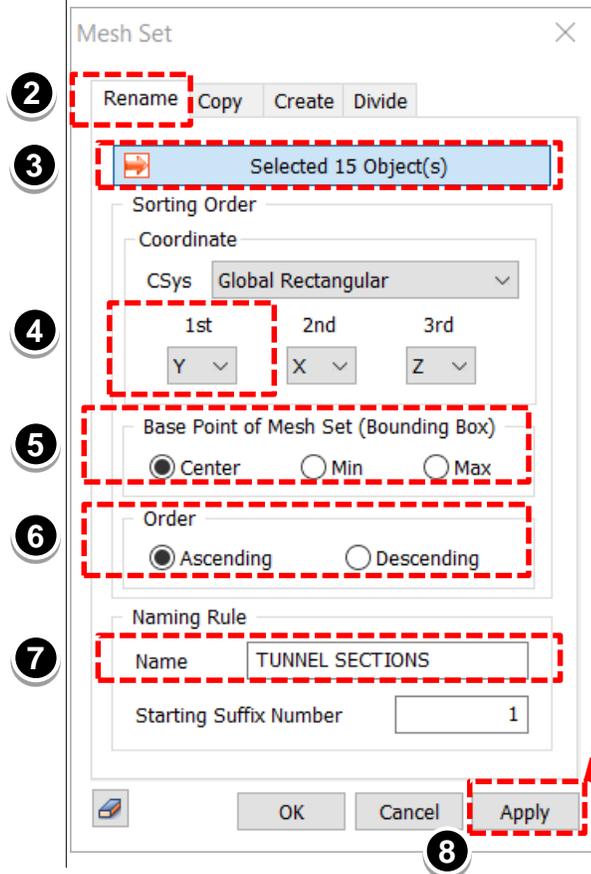
- Step 1:** The 'Mesh' tab in the ribbon is highlighted.
- Step 2:** The 'Geometry' tab in the dialog is highlighted.
- Step 3:** The 'Type' dropdown menu is set to 'Face'.
- Step 4:** The 'Selected 4 Object(s)' field is highlighted, with red dashed arrows pointing to the selected faces on the 3D model.
- Step 5:** The 'Property' dropdown menu is set to '1: SHIELD'.
- Step 6:** The 'Mesh Set' dropdown menu is set to 'SHIELD 1'.
- Step 7:** The 'OK' button is highlighted.

The 3D model shows a cylindrical tunnel shield with a meshed section on its left end. The mesh is composed of green and purple elements. The shield is divided into colored segments (green, purple, blue, yellow, red, orange, brown) along its length.

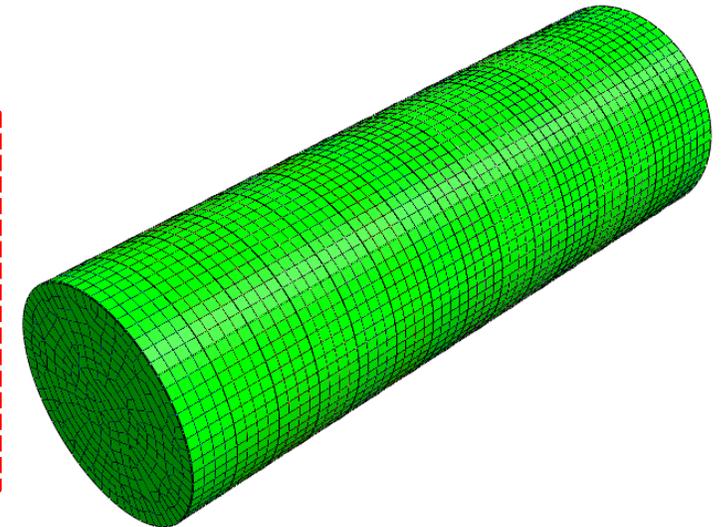
5-6 Renaming Main Tunnel Segments

Procedure

- ❶ Go to Mesh > Rename
- ❷ Click on Rename.
- ❸ Select the 25 mesh sets for the **main tunnel segments**.
- ❹ In sorting order, start with **Y axis** as the 1st priority.
- ❺ Leave the base point as Center.
- ❻ Set the order as **Ascending**.
- ❼ Enter the name as **TUNNEL SECTIONS**.
- ❽ Click **Apply**.



<input type="checkbox"/>	SHIELD 4	391	
<input type="checkbox"/>	SHIELD 5	392	
<input type="checkbox"/>	SHIELD 6	393	
<input type="checkbox"/>	SHIELD 7	394	
<input type="checkbox"/>	SOFT ROCK	386	
<input checked="" type="checkbox"/>	TUNNEL SECTIONS-001	371	
<input checked="" type="checkbox"/>	TUNNEL SECTIONS-002	374	
<input checked="" type="checkbox"/>	TUNNEL SECTIONS-003	375	
<input checked="" type="checkbox"/>	TUNNEL SECTIONS-004	373	
<input checked="" type="checkbox"/>	TUNNEL SECTIONS-005	372	
<input checked="" type="checkbox"/>	TUNNEL SECTIONS-006	378	
<input checked="" type="checkbox"/>	TUNNEL SECTIONS-007	377	
<input checked="" type="checkbox"/>	TUNNEL SECTIONS-008	380	
<input checked="" type="checkbox"/>	TUNNEL SECTIONS-009	379	
<input checked="" type="checkbox"/>	TUNNEL SECTIONS-010	382	
<input checked="" type="checkbox"/>	TUNNEL SECTIONS-011	381	
<input checked="" type="checkbox"/>	TUNNEL SECTIONS-012	385	
<input checked="" type="checkbox"/>	TUNNEL SECTIONS-013	383	
<input checked="" type="checkbox"/>	TUNNEL SECTIONS-014	376	
<input checked="" type="checkbox"/>	TUNNEL SECTIONS-015	384	

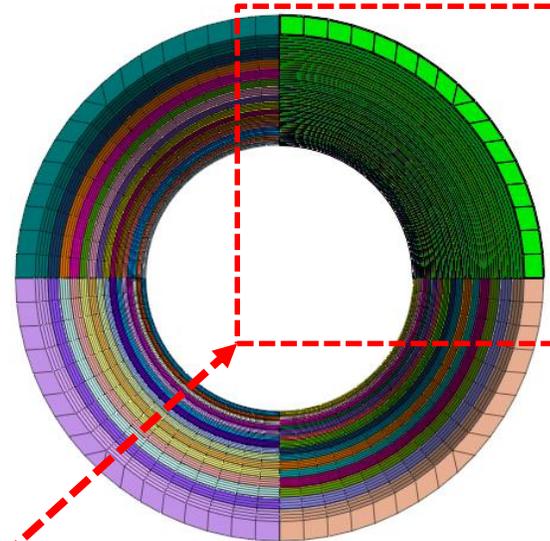
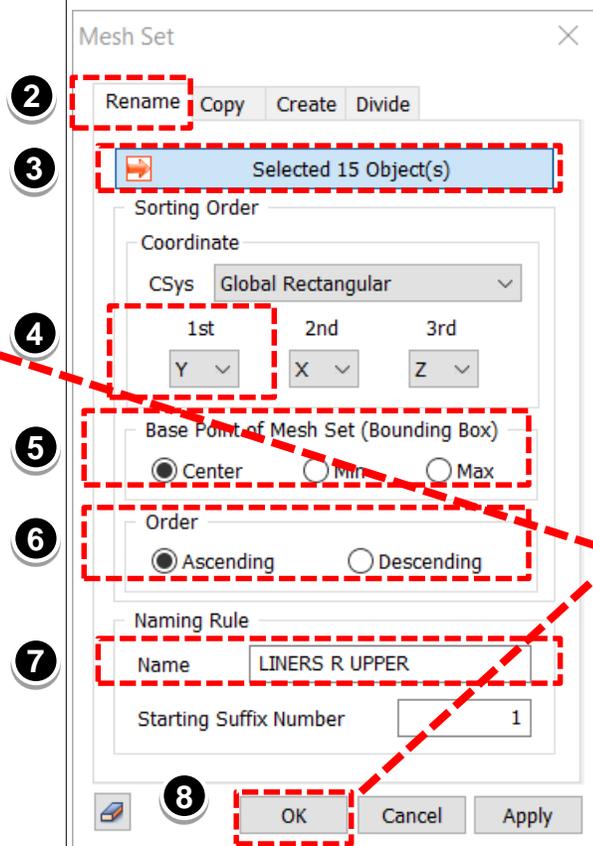


5-7 Renaming Liner Segments

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".



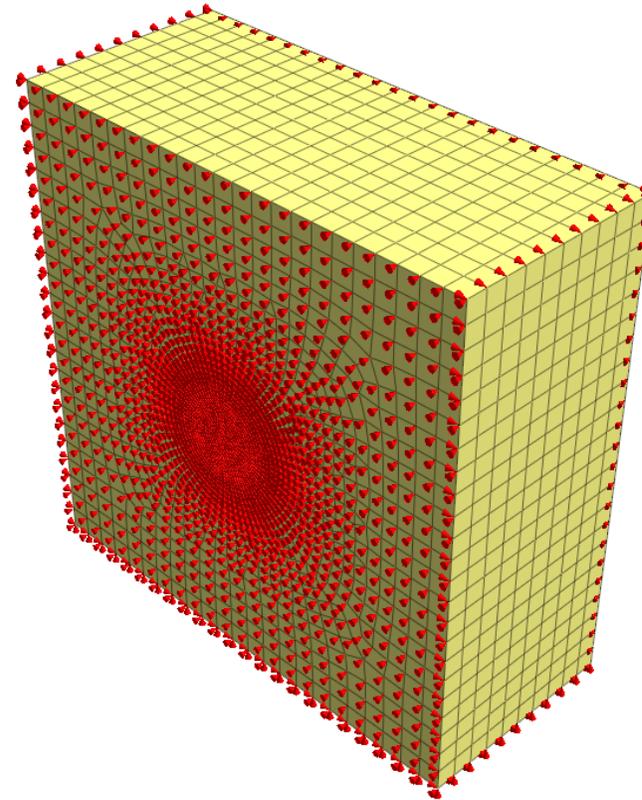
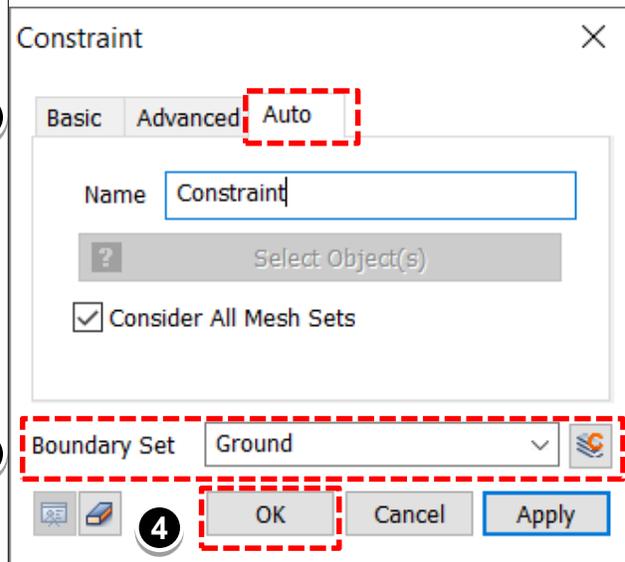
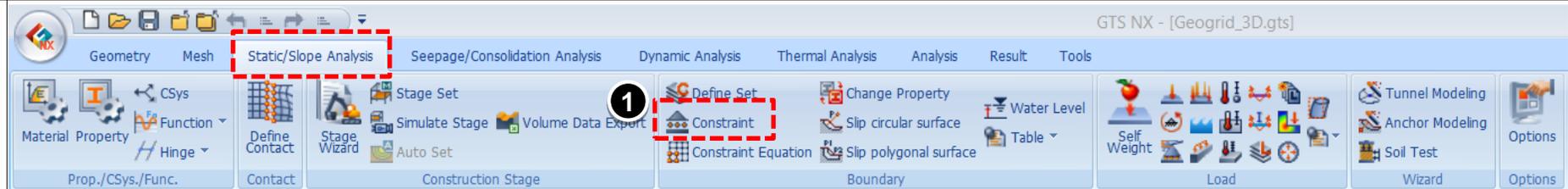
Item	ID	Color
<input checked="" type="checkbox"/> LINERS R UPPER-001	322	Blue
<input checked="" type="checkbox"/> LINERS R UPPER-002	321	Red
<input checked="" type="checkbox"/> LINERS R UPPER-003	325	Orange
<input checked="" type="checkbox"/> LINERS R UPPER-004	320	Grey
<input checked="" type="checkbox"/> LINERS R UPPER-005	324	Green
<input checked="" type="checkbox"/> LINERS R UPPER-006	319	Purple
<input checked="" type="checkbox"/> LINERS R UPPER-007	323	Brown
<input checked="" type="checkbox"/> LINERS R UPPER-008	318	Pink
<input checked="" type="checkbox"/> LINERS R UPPER-009	316	Light Blue
<input checked="" type="checkbox"/> LINERS R UPPER-010	315	Light Green
<input checked="" type="checkbox"/> LINERS R UPPER-011	317	Yellow
<input checked="" type="checkbox"/> LINERS R UPPER-012	313	Light Yellow
<input checked="" type="checkbox"/> LINERS R UPPER-013	314	Yellow
<input checked="" type="checkbox"/> LINERS R UPPER-014	312	Blue
<input checked="" type="checkbox"/> LINERS R UPPER-015	311	Purple

<input checked="" type="checkbox"/> LINERS R LOWER-001	337	Orange
<input checked="" type="checkbox"/> LINERS R LOWER-002	336	Purple
<input checked="" type="checkbox"/> LINERS R LOWER-003	338	Green
<input checked="" type="checkbox"/> LINERS R LOWER-004	334	Pink
<input checked="" type="checkbox"/> LINERS R LOWER-005	335	Cyan
<input checked="" type="checkbox"/> LINERS R LOWER-006	332	Orange
<input checked="" type="checkbox"/> LINERS R LOWER-007	333	Light Blue
<input checked="" type="checkbox"/> LINERS R LOWER-008	331	Dark Blue
<input checked="" type="checkbox"/> LINERS R LOWER-009	330	Light Green
<input checked="" type="checkbox"/> LINERS R LOWER-010	339	Red
<input checked="" type="checkbox"/> LINERS R LOWER-011	328	Purple
<input checked="" type="checkbox"/> LINERS R LOWER-012	340	Green
<input checked="" type="checkbox"/> LINERS R LOWER-013	329	Pink
<input checked="" type="checkbox"/> LINERS R LOWER-014	327	Light Green
<input checked="" type="checkbox"/> LINERS R LOWER-015	326	Yellow
<input checked="" type="checkbox"/> LINERS L UPPER-001	360	Green
<input checked="" type="checkbox"/> LINERS L UPPER-002	367	Dark Blue
<input checked="" type="checkbox"/> LINERS L UPPER-003	368	Orange
<input checked="" type="checkbox"/> LINERS L UPPER-004	365	Pink
<input checked="" type="checkbox"/> LINERS L UPPER-005	366	Light Green
<input checked="" type="checkbox"/> LINERS L UPPER-006	363	Light Blue
<input checked="" type="checkbox"/> LINERS L UPPER-007	364	Purple
<input checked="" type="checkbox"/> LINERS L UPPER-008	362	Yellow
<input checked="" type="checkbox"/> LINERS L UPPER-009	370	Pink
<input checked="" type="checkbox"/> LINERS L UPPER-010	359	Brown
<input checked="" type="checkbox"/> LINERS L UPPER-011	357	Orange
<input checked="" type="checkbox"/> LINERS L UPPER-012	358	Blue
<input checked="" type="checkbox"/> LINERS L UPPER-013	369	Light Blue
<input checked="" type="checkbox"/> LINERS L UPPER-014	361	Orange
<input checked="" type="checkbox"/> LINERS L UPPER-015	356	Light Green
<input checked="" type="checkbox"/> LINERS L LOWER-001	346	Purple
<input checked="" type="checkbox"/> LINERS L LOWER-002	355	Purple
<input checked="" type="checkbox"/> LINERS L LOWER-003	351	Light Blue
<input checked="" type="checkbox"/> LINERS L LOWER-004	354	Pink
<input checked="" type="checkbox"/> LINERS L LOWER-005	350	Yellow
<input checked="" type="checkbox"/> LINERS L LOWER-006	353	Orange
<input checked="" type="checkbox"/> LINERS L LOWER-007	349	Light Green
<input checked="" type="checkbox"/> LINERS L LOWER-008	352	Light Blue
<input checked="" type="checkbox"/> LINERS L LOWER-009	345	Purple
<input checked="" type="checkbox"/> LINERS L LOWER-010	348	Light Blue
<input checked="" type="checkbox"/> LINERS L LOWER-011	347	Purple
<input checked="" type="checkbox"/> LINERS L LOWER-012	344	Purple
<input checked="" type="checkbox"/> LINERS L LOWER-013	343	Light Blue
<input checked="" type="checkbox"/> LINERS L LOWER-014	342	Brown
<input checked="" type="checkbox"/> LINERS L LOWER-015	341	Light Blue

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**



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 BOUNDARY CONDITIONS SHOULD START FROM GROUT - 2 AND END AT GROUT - 15.

The screenshot displays the Midas NX software interface for a 3D model of a shield tunnel. The following steps are illustrated:

- ① The **Static/Slope Analysis** menu is selected, followed by **Boundary** and **Change Property**.
- ② The context is changed to **Basic** and **Mesh Set (M)**.
- ③ The **SHIELD-001** mesh set is selected in the **Model** tree.
- ④ The **Change Property** dialog box is shown with the following settings:
 - Name:** Change Property-82
 - Object:** Selected 272 Object(s)
 - Property:** 5: GROUT
 - Boundary Set:** GROUT -1
- ⑤ The **Apply** button is clicked.

The 3D model shows the tunnel shield with the selected mesh set highlighted in green.

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.

The screenshot shows the GTS NX software interface for a 3D shield tunnel model. The 'Change Property' dialog box is open, and the 'General' tab is selected. The dialog box contains the following information:

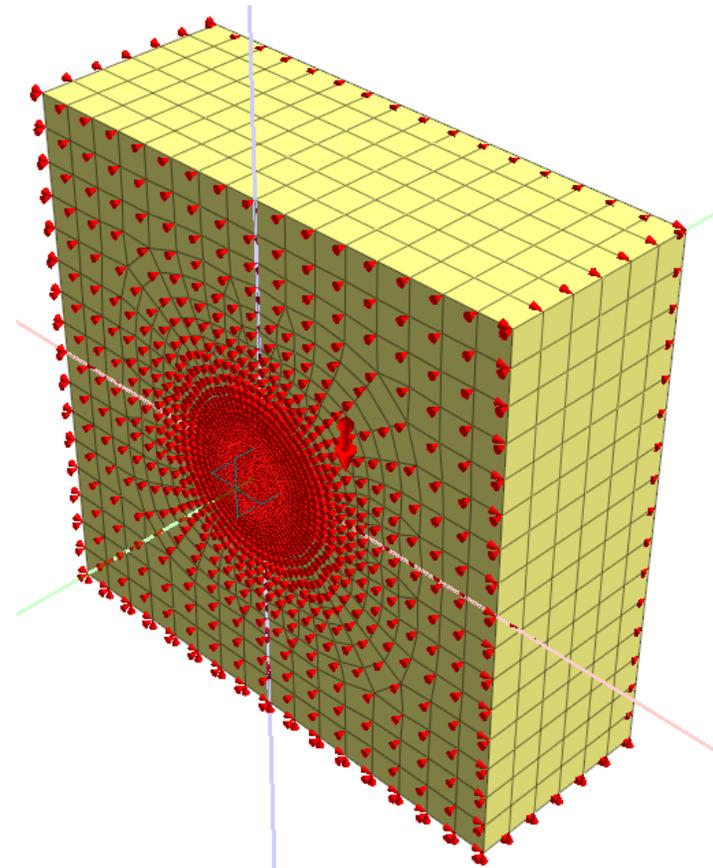
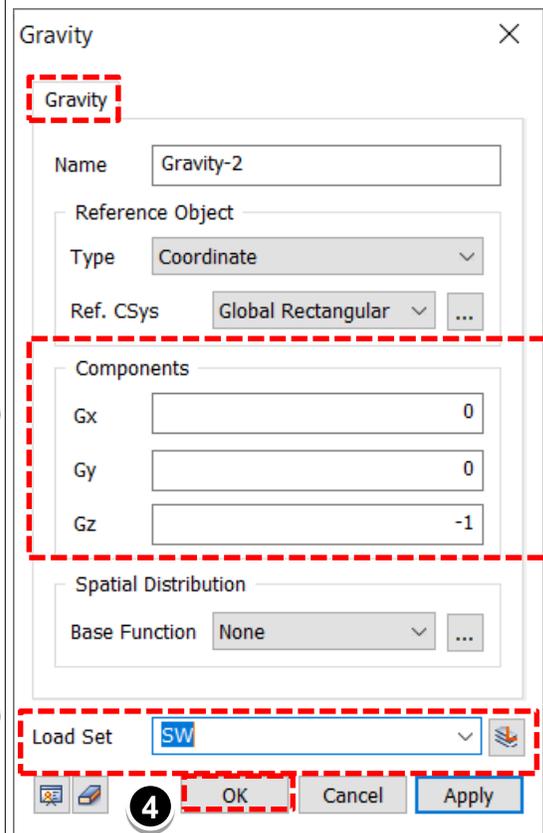
- Name:** Change Property-82
- Object:** Selected 272 Object(s)
- Property:** 3: LINERS
- Boundary Set:** LINER - 1

The 3D model shows a cross-section of the tunnel with various mesh sets highlighted in different colors. A table on the right lists the mesh sets from LINER - 1 to LINER - 15 with their respective object counts.

<input type="checkbox"/>	LINER - 1	27
<input type="checkbox"/>	LINER - 2	28
<input type="checkbox"/>	LINER - 3	29
<input type="checkbox"/>	LINER - 4	30
<input type="checkbox"/>	LINER - 5	31
<input type="checkbox"/>	LINER - 6	32
<input type="checkbox"/>	LINER - 7	33
<input type="checkbox"/>	LINER - 8	34
<input type="checkbox"/>	LINER - 9	35
<input type="checkbox"/>	LINER - 10	36
<input type="checkbox"/>	LINER - 11	37
<input type="checkbox"/>	LINER - 12	38
<input type="checkbox"/>	LINER - 13	51
<input type="checkbox"/>	LINER - 14	50
<input type="checkbox"/>	LINER - 15	49

Procedure

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



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 STARTING FROM HP- 2 TO HP - 7.

The screenshot displays the GTS NX software interface for defining a pressure load. The 'Pressure' dialog box is open, showing the following settings:

- Edge:** Face (selected)
- Name:** Pressure-26
- Object:** Face (selected)
- Type:** Face (selected)
- Direction:** Normal (selected)
- Ref. CSys:** Global Rectangular
- Magnitude:**
 - Uniformly Distributed Load
 - Base Function:** None
 - P or P1:** 200 kN/m²
 - P2:** 0 kN/m²
 - P3:** 0 kN/m²
 - P4:** 0 kN/m²
- Load Set:** HP 1

The 'Apply' button is highlighted, indicating the final step in the procedure. Two 3D models of a tunnel section are shown, illustrating the selection of the 3rd tunnel section solid and its faces.

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 STARTING FROM JT - 2 TO JT - 7.

The screenshot displays the GTS NX software interface with the 'Pressure' dialog box open. The dialog box is divided into several sections, with red dashed boxes and arrows indicating the steps:

- Step 1:** The 'Static/Slope Analysis' tab is selected in the top menu bar.
- Step 2:** The 'Face' option is selected under the 'Edge' dropdown.
- Step 3:** The 'Direction' section is set to 'Ref. Csys' with the 'Y' axis selected.
- Step 4:** The 'Magnitude' section has 'Uniformly Distributed Load' checked.
- Step 5:** The 'Load Set' dropdown is set to 'JT 1'.
- Step 6:** The 'Apply' button is highlighted.

The 3D models show the application of the pressure load to the outer faces of the liner segment rings. The left model shows a single ring with a green face highlighted, and the right model shows a full cylinder with green arrows pointing outwards from the outer surface of each ring.

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.

The screenshot displays the GTS NX software interface for defining a pressure load. The 'Pressure' dialog box is open, showing the following settings:

- Edge:** Face (selected)
- Name:** Pressure-41
- Object:** Type: Face; Selected 8 Object(s)
- Direction:** Type: Normal; Ref. CSys: Global Rectangular; X, Y, Z radio buttons are present.
- Magnitude:** Uniformly Distributed Load; Base Function: None
- P or P1:** 50 kN/m²
- P2:** 0 kN/m²
- P3:** 0 kN/m²
- P4:** 0 kN/m²
- Load Set:** SHIELD PRESSURE 1

Two 3D models of a tunnel liner segment are shown. The left model highlights the selected outer faces in green, with red arrows pointing to the 'Face' tab and the 'Selected 8 Object(s)' field in the dialog. The right model shows the final result with green arrows representing the applied pressure load on the selected faces. Red dashed boxes and arrows indicate the flow of the procedure from the software interface to the 3D models.

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 SEGMENT PRESSURE LOAD SETS WILL END AT SEGMENT PRESSURE - 7.

GTS NX - [Midas model.gts]

Static/Slope Analysis

Pressure

Edge Face Axisymmetric

Name Pressure-55

Object

Type Face

Selected 12 Object(s)

Direction

Type Normal

Ref. CSys Global Rectangular

X Y Z

Magnitude

Uniformly Distributed Load

Base Function None

P or P1 1000 kN/m²

P2 0 kN/m²

P3 0 kN/m²

P4 0 kN/m²

Load Set SEGMENT PRESSURE - 1

OK Cancel Apply

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**.

Element, Boundary, Load Activation Status

Set Type	Set Name Prefix	I.S.	S1	S2	S3	S4	S5	S6	S7	S8	S9	S10	S11	S12	S13	S14	S15	S16	S17	S18	S19
Load Set	SHIELD PRESSURE -		A: 2	A: 3	A: 4	A: 5	A: 6	A: 7	A: 8												
Load Set	SHIELD PRESSURE - 1		A: -																		
Load Set	CONTRACTION																				
Mesh set	Default Mesh Set																				
Boundary Set	GBC	A: -																			
Boundary Set	GROUT -			A: 1,2	A: 3,4	A: 5,6	A: 7,8	A: 9,10	A: 11,12	A: 13,14											
Load Set	HP -		A: 1	A: 2	A: 3	A: 4	A: 5	A: 6	A: 7												
Load Set	JT -				A: 2	A: 3	A: 4	A: 5	A: 6	A: 7	A: 8	A: 9	A: 10	A: 11	A: 12	A: 13	A: 14	A: 15	A: 16	A: 17	A: 18
Load Set	JT - 1				A: -	R: 2	R: 3	R: 4	R: 5	R: 6	R: 7	R: 8	R: 9	R: 10	R: 11	R: 12	R: 13	R: 14	R: 15	R: 16	R: 17
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	A: 3,4	A: 5,6	A: 7,8	A: 9,10	A: 11,12	A: 13to15										
Mesh set	LINERS L UPPER-	A: 1to15	R: 1,2	R: 3,4	A: 1,2	A: 3,4	A: 5,6	A: 7,8	A: 9,10	A: 11,12	A: 13to15										
Mesh set	LINERS R LOWER-	A: 1to15	R: 1,2	R: 3,4	A: 1,2	A: 3,4	A: 5,6	A: 7,8	A: 9,10	A: 11,12	A: 13to15										
Mesh set	LINERS R UPPER-	A: 1to15	R: 1,2	R: 3,4	A: 1,2	A: 3,4	A: 5,6	A: 7,8	A: 9,10	A: 11,12	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: 15												

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.

1

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	<input type="checkbox"/>		1	1	1
Boundary Set	GROUT -	A	1	<input type="checkbox"/>			2	3
Boundary Set	GROUT -	A	2	<input type="checkbox"/>			2	4
Load Set	HP -	A	1	<input type="checkbox"/>			1	1
Load Set	JT -	A	1	<input type="checkbox"/>			1	3
Load Set	JT -	R	1	<input type="checkbox"/>			1	4
Boundary Set	LINER -	A	1	<input checked="" type="checkbox"/>	13	2	3	1
Boundary Set	LINER -	A	2	<input checked="" type="checkbox"/>	1	2	3	1
Boundary Set	LINER -	A	15	<input type="checkbox"/>		1	9	1
Mesh set	LINERS L LOWER-	R	1	<input type="checkbox"/>		2	1	1
Mesh set	LINERS L LOWER-	R	2	<input type="checkbox"/>		2	1	1
Mesh set	LINERS L LOWER-	A	1	<input checked="" type="checkbox"/>	13	2	3	1
Mesh set	LINERS L LOWER-	A	2	<input type="checkbox"/>		2	3	1
Mesh set	LINERS L LOWER-	A	15	<input type="checkbox"/>		1	9	1
Mesh set	LINERS L UPPER-	R	1	<input type="checkbox"/>		2	1	1
Mesh set	LINERS L UPPER-	R	2	<input type="checkbox"/>		2	1	1
Mesh set	LINERS L UPPER-	A	1	<input checked="" type="checkbox"/>	13	2	3	1
Mesh set	LINERS L UPPER-	A	2	<input type="checkbox"/>		2	3	1
Mesh set	LINERS L UPPER-	A	15	<input type="checkbox"/>		1	9	1
Mesh set	LINERS R LOWER-	R	1	<input type="checkbox"/>		2	1	1
Mesh set	LINERS R LOWER-	R	2	<input type="checkbox"/>		2	1	1
Mesh set	LINERS R LOWER-	A	1	<input checked="" type="checkbox"/>	13	2	3	1
Mesh set	LINERS R LOWER-	A	2	<input type="checkbox"/>		2	3	1
Mesh set	LINERS R LOWER-	A	15	<input type="checkbox"/>		1	9	1
Mesh set	LINERS R UPPER-	R	1	<input type="checkbox"/>		2	1	1
Mesh set	LINERS R UPPER-	R	2	<input type="checkbox"/>		2	1	1
Mesh set	LINERS R UPPER-	A	1	<input checked="" type="checkbox"/>	13	2	3	1
Mesh set	LINERS R UPPER-	A	2	<input type="checkbox"/>		2	3	1
Mesh set	LINERS R UPPER-	A	15	<input type="checkbox"/>		1	9	1
Load Set	SEGMENT PRESSURE -	A	1	<input type="checkbox"/>		1	3	1
Mesh set	SHIELD-	A	1	<input type="checkbox"/>		2	1	1
Mesh set	SHIELD-	A	2	<input type="checkbox"/>		2	1	1
Mesh set	TUNNEL SECTIONS-	R	1	<input type="checkbox"/>		2	1	1
Mesh set	TUNNEL SECTIONS-	R	2	<input type="checkbox"/>		2	1	1

2

Apply Assignment Rules

3

OK

Cancel

Element, Boundary, Load Activation Status

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 -											
Load Set	HP -		A: 1	A: 2	A: 3	A: 4	A: 5	A: 6	A: 7			
Load Set	JT -				A: 1	A: 2	A: 3	A: 4	A: 5	A: 6	A: 7	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	R: 5.6	R: 7.8	R: 9.10	R: 11.12	R: 13.14	R: 15	A: 13to15	
Mesh set	LINERS L UPPER-	A: 1to15	R: 1.2	R: 3.4	A: 1.2	A: 3.4	A: 5.6	A: 7.8	A: 9.10	A: 11.12	R: 15	A: 13to15
Mesh set	LINERS R LOWER-	A: 1to15	R: 1.2	R: 3.4	A: 1.2	A: 3.4	A: 5.6	A: 7.8	A: 9.10	A: 11.12	R: 15	A: 13to15
Mesh set	LINERS R UPPER-	A: 1to15	R: 1.2	R: 3.4	A: 1.2	A: 3.4	A: 5.6	A: 7.8	A: 9.10	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		

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**.

The screenshot illustrates the software interface for defining an analysis case. The main window shows the 'Analysis' tab selected, with the 'General' sub-tab active. The 'Add/Modify Analysis Case' dialog is open, showing the following settings:

- Title:** Tunnel Analysis
- Description:** (empty)
- Solution Type:** Construction Stage
- Construction Stage Set:** Cross Tunnel

The 'Analysis Control' dialog is also open, showing the following settings:

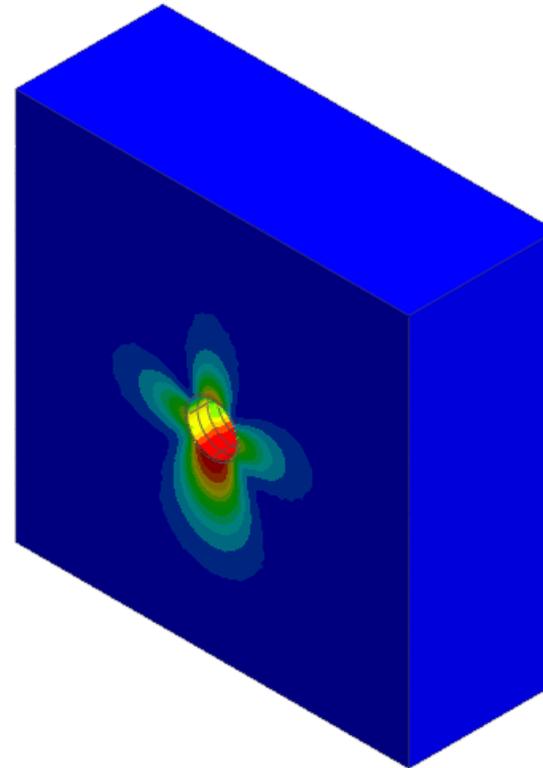
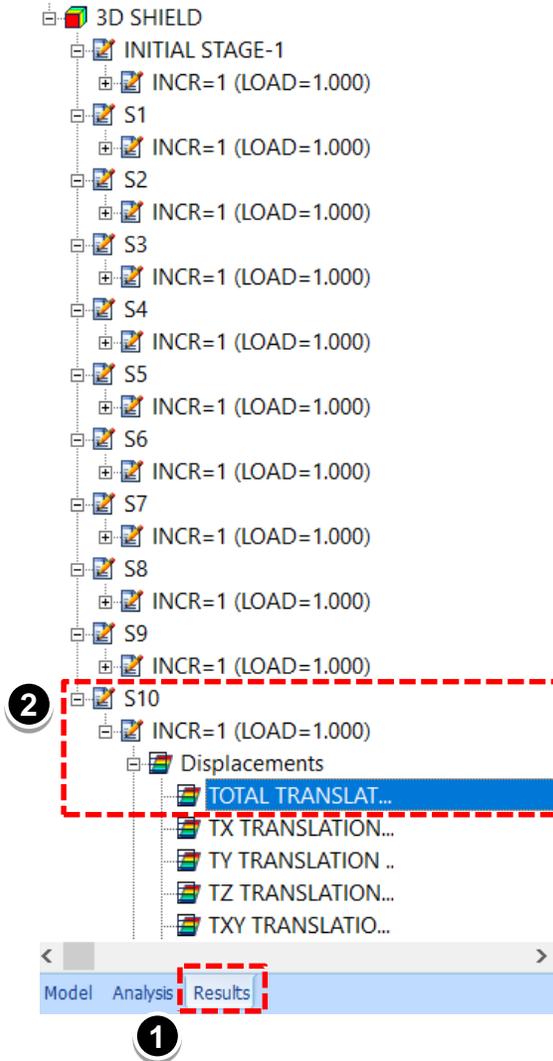
- General Tab:**
 - Water Pressure:** Automatically Consider Water Pressure
 - Initial Stage:**
 - Initial Stage for Stress Analysis (1:I.S.)
 - Apply K0 Condition
 - Cut-Off Negative Effective Pressure
 - Initial Stress:**
 - Estimate Initial Stress of Activated Elements
 - Final Calculation Stage:**
 - End Stage (1:I.S.)
 - Middle Stage (1:I.S.)
 - Specify Restart Stage
 - 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/m²)
 - Initial Configuration:**
 - Estimate Initial Configuration of Activated Nodes

The 'OK' button in the 'Add/Modify Analysis Case' dialog is highlighted with a red dashed box, indicating the final step in the procedure.

10-1 Result (DISPLACEMENTS)

Procedure

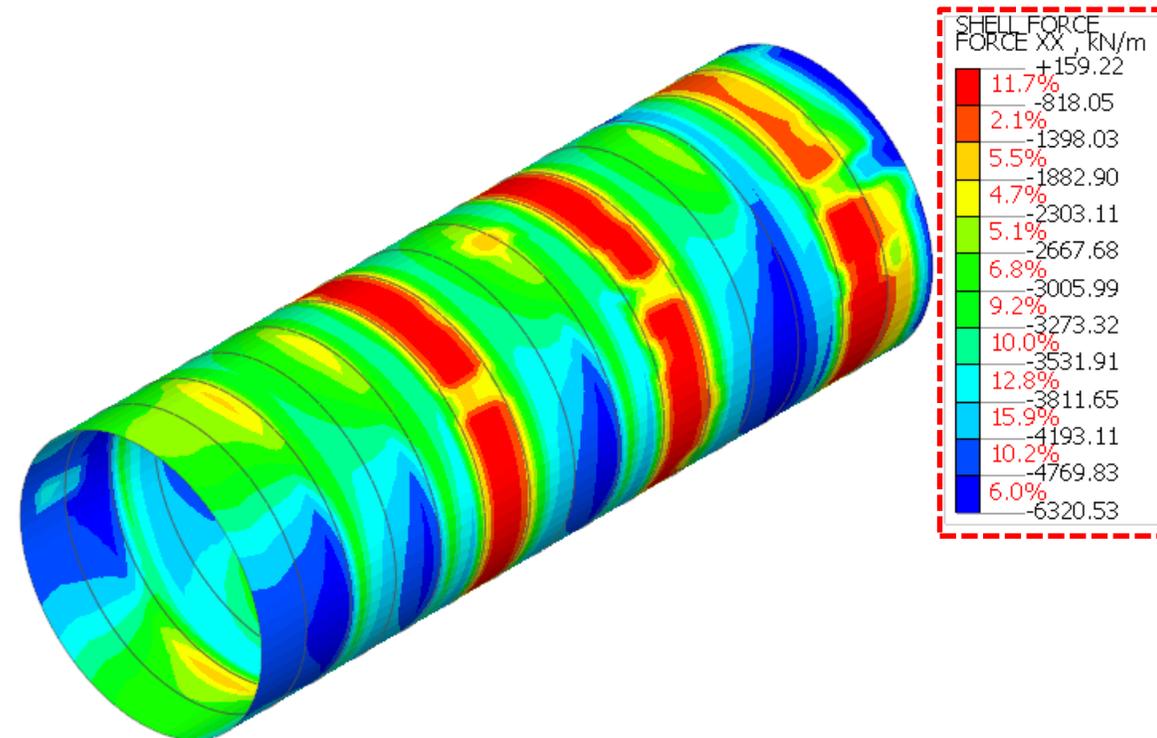
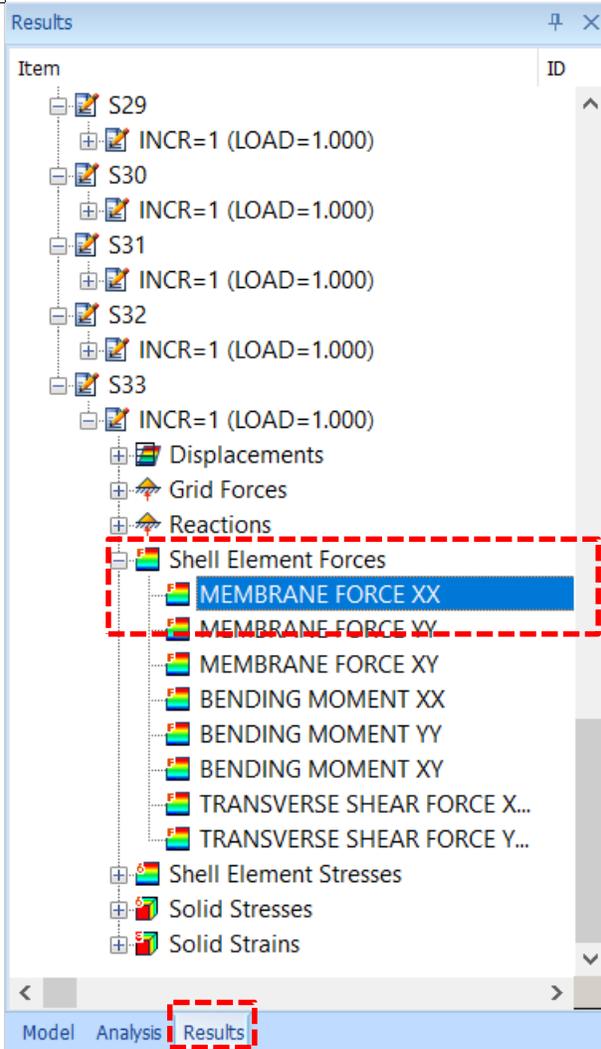
- 1 Go to Results > 3D SHIELD > S10.
- 2 Click on Total Displacements.



10-3 Result (SHELL ELEMENT FORCES)

Procedure

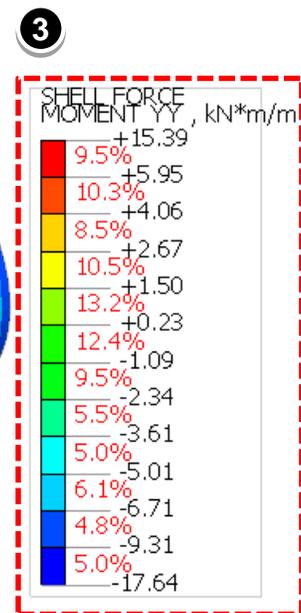
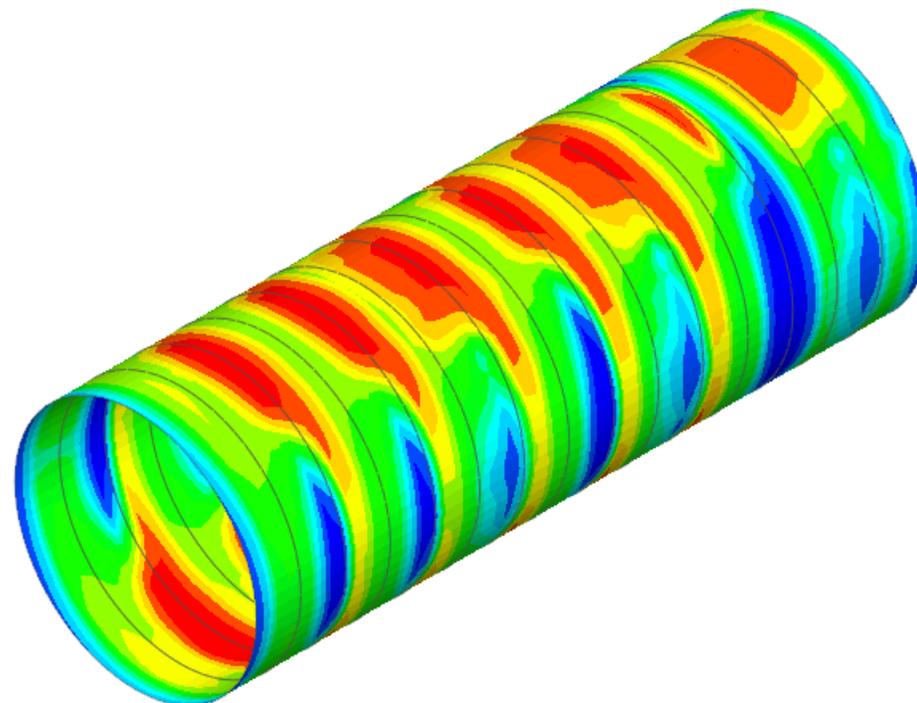
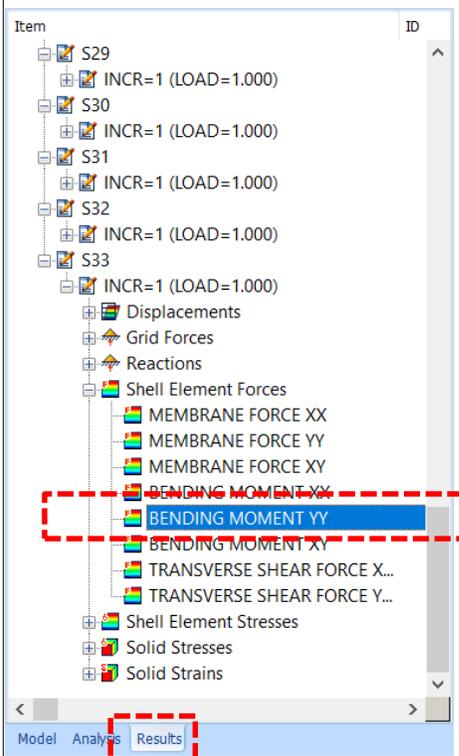
- ① Go to Results > 3D SHIELD > S10.
- ② Click on SHELL ELEMENT FORCES.
- ③ Legend shows the forces acting on the Shell/Shield..



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.



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.

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



MIDAS GEOTECH | ONLINE COURSE 2021

ONLINE COURSE ASSIGNMENT

First name* Last name*

Organization* Country*

Email*

Select the title of the online course*

Files*

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 – 11th NOVEMBER

SUBMISSION ONLY VIA LINK, MAIL SUBMISSIONS NOT ACCEPTED

SUBMISSION DATE : till 26TH NOVEMBER