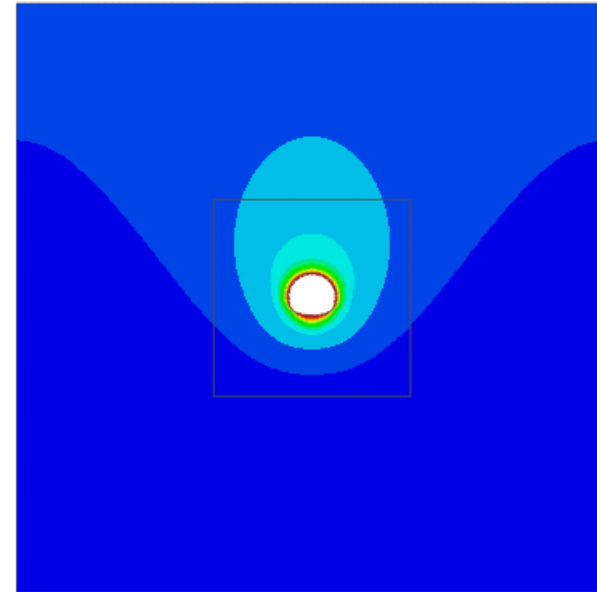


MIDAS *Technical Material* Tutorial

Stress relaxation in NATM Tunnel



MIDAS



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: Construction stage Definition
- Step 9: Analysis Case
- Step 10: Analysis
- Step 11: Results

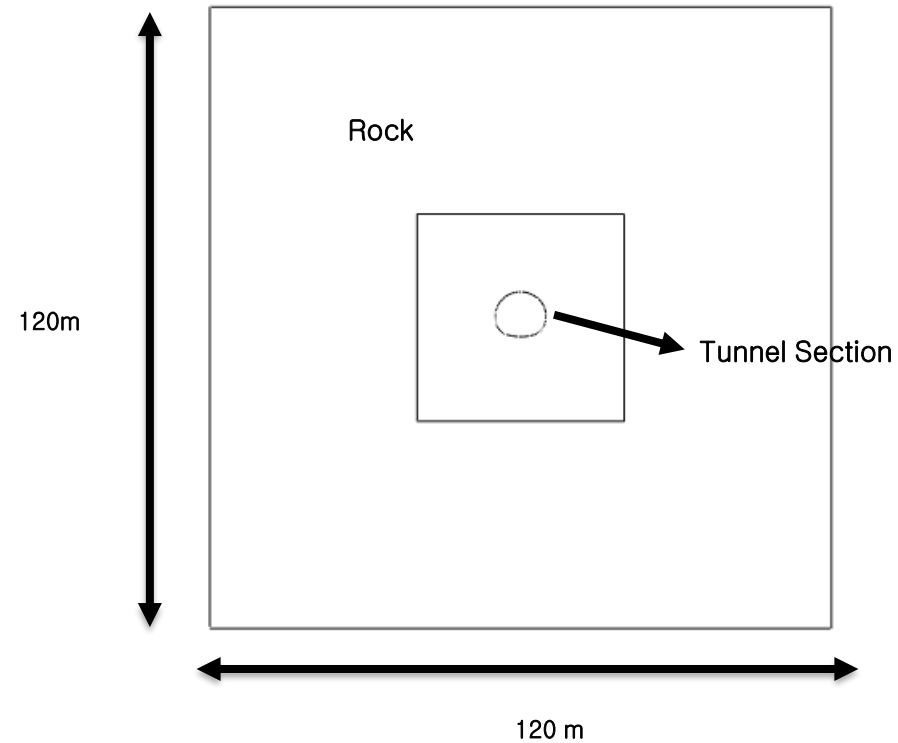
Program Version	GTS NX 2023(v.1.1.)
Revision Date	May 09,2023

NATM Tunnel

Overview

This example problem is meant to demonstrate the general work-flow to perform construction stage analysis and simulate stress relaxation in NATM Tunnel.

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: Construction stage Definition
- Step 9: Analysis Case
- Step 10: Analysis
- Step 11: Results

NATM 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
Rock	Mohr-Coulomb	3e6	0.3	25	25	2200	30
Initial Liner	Elastic	2e7	0.2	25	–	–	–
Final Liner	Elastic	1.3e7	0.23	25	–	–	–

Program Version GTS NX 2023(v.1.1.)

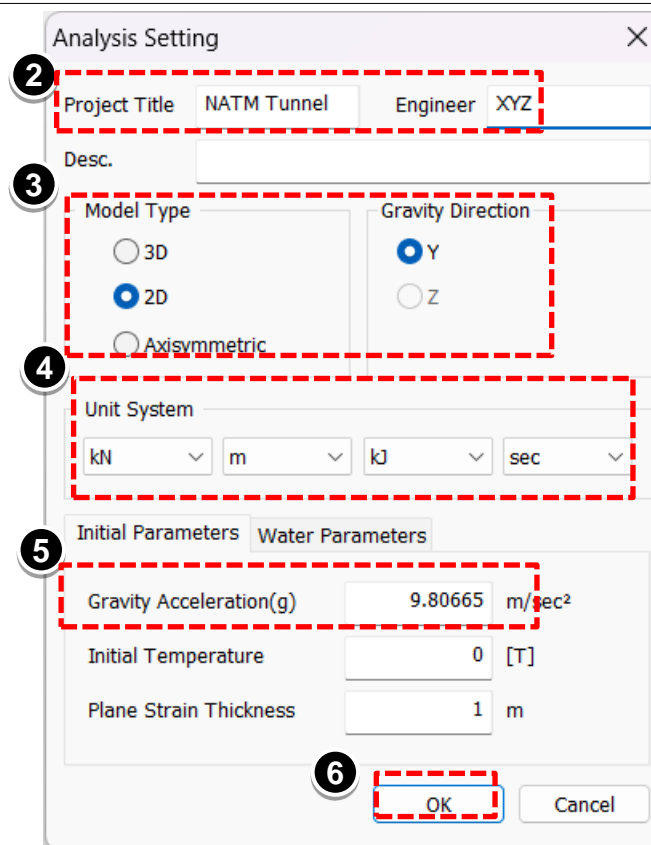
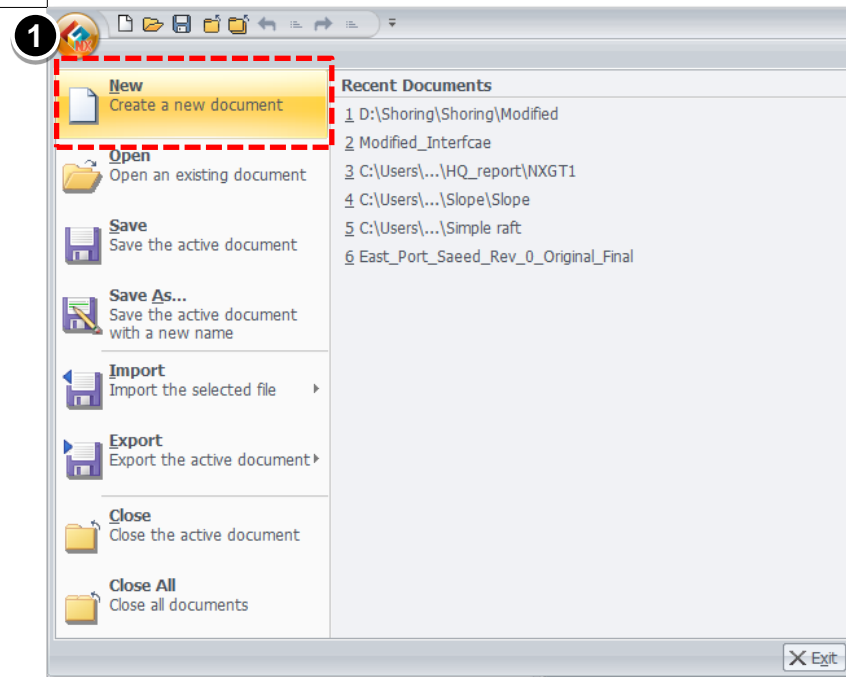
Revision Date May 09,2023

1-1 Initial Setting

Procedure

Starting Midas GTS NX

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



Procedure

Defining Soil Materials

- ① Go to **Mesh > Material**
- ② Click on **Create**. Select **Isotropic**
- ③ Select Model type as **Mohr-Coulomb**
- ④ In General tab, Enter the value of **Elastic modulus (E) = 3000000 kN/m²**
- ⑤ Enter **Poisson's ratio = 0.3**, **Unit weight = 25 kN/m³**.
- ⑥ Select **Manual** in **Ko Determination** and input the value as **1**.

The screenshot illustrates the software interface for defining a soil material, following the steps outlined in the procedure. The interface is divided into several tabs: Geometry, Mesh, Static/Slope Analysis, Seepage/Consolidation Analysis, Dynamic Analysis, and Analysis. The **Mesh** tab is active, and the **Material** option is selected. A red dashed box highlights the **Create...** button in the **Add/Modify Material** dialog box, which is open. The **Create...** dropdown menu shows **Isotropic** as the selected option. The **Material** dialog box is also shown, with the following settings:

- ID:** 1
- Name:** Rockmass
- Color:** Cyan
- Model Type:** Mohr-Coulomb
- General Tab:**
 - Elastic Modulus(E):** 3000000 kN/m²
 - Inc. of Elastic Modulus:** 0 kN/m³
 - Inc. of Elastic Modulus Ref. Height:** 0 m
 - Poisson's Ratio(v):** 0.3
 - Unit Weight(γ):** 25 kN/m³
- Initial Stress Parameters:**
 - Ko Determination:** 1
 - Automatic:** Selected
 - Manual:** Selected
 - Anisotropy:** Unchecked
- 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

The **DB** button is located at the bottom right of the dialog box. The **OK**, **Cancel**, and **Apply** buttons are at the bottom of the dialog box.

2-2 Defining Soil Material

Procedure

- ➊ Go to Porous Tab > Enter Unit weight (saturated) = 25 kN/m^3
- ➋ Keep Drainage Parameters as Drained
- ➌ Go to Non-Linear Tab > Enter the value of Cohesion = 2200 kN/m^2
- ➍ Input the value of Frictional angle = 30°
- ➎ Enter the name as Rockmass
- ➏ Click OK

Material dialog box showing the Porous and Non-Linear tabs. The following parameters are defined:

- ID: 1
- Name: Rockmass
- Model Type: Mohr-Coulomb
- Unit Weight (Saturated): 25 kN/m^3
- Initial Void Ratio (eo): 0.5
- Drainage Parameters: Drained
- Frictional Angle (Φ): 30°
- Unsaturation Property: Unsaturation Property (checked)
- Skempton's B Coefficient: 0.97826087
- Seepage & Consolidation Parameters:
 - Permeability Coefficients: $k_x = 1e-005$, $k_y = 1e-005$, $k_z = 1e-005$ m/sec
 - Void Ratio Dependency of Permeability (ck): 0.5
 - Specific Storativity (Ss): 5.230213 1/m

Material dialog box showing the Non-Linear tab. The following parameters are defined:

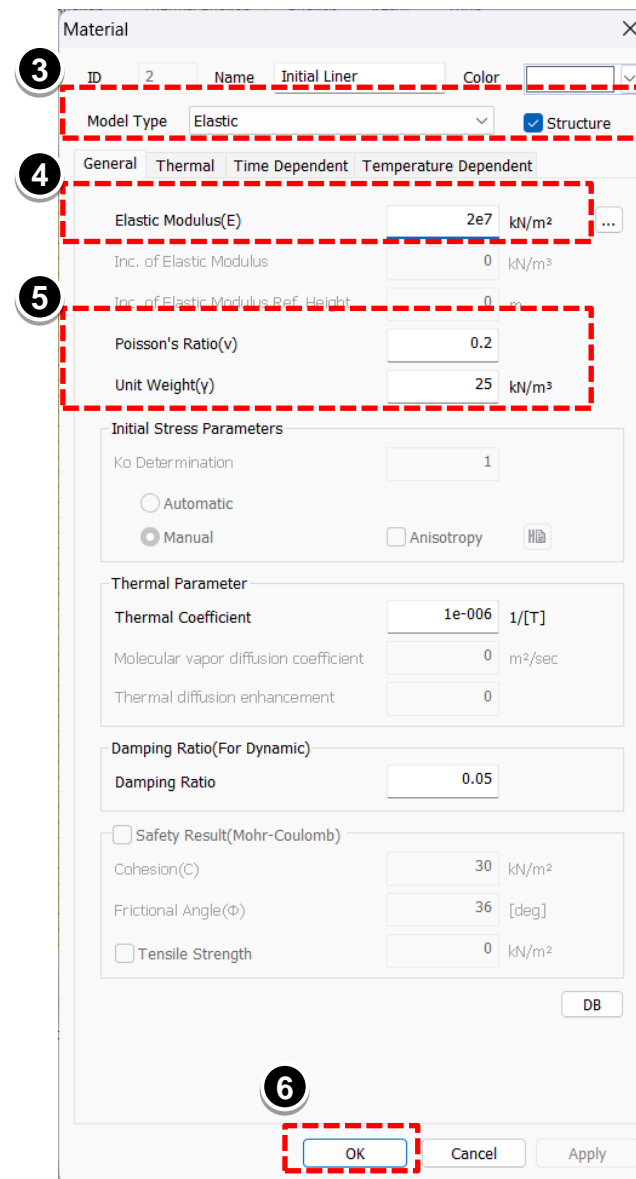
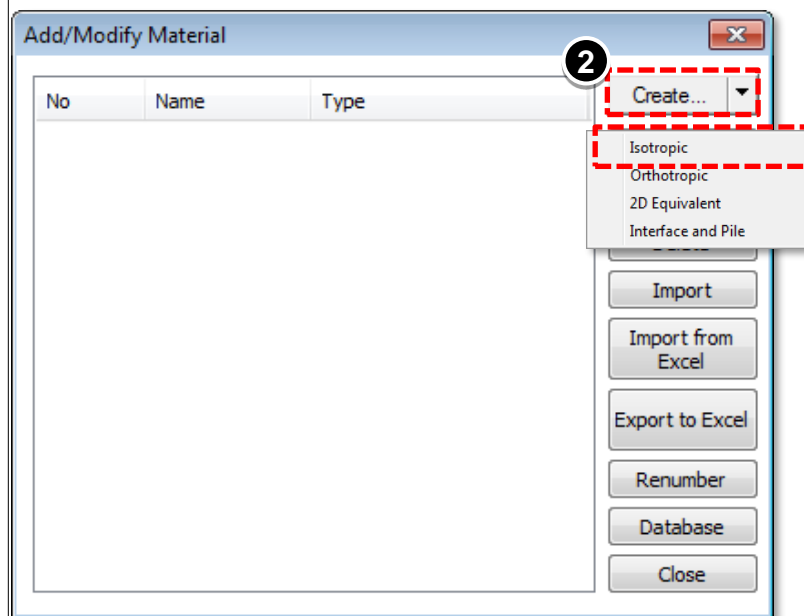
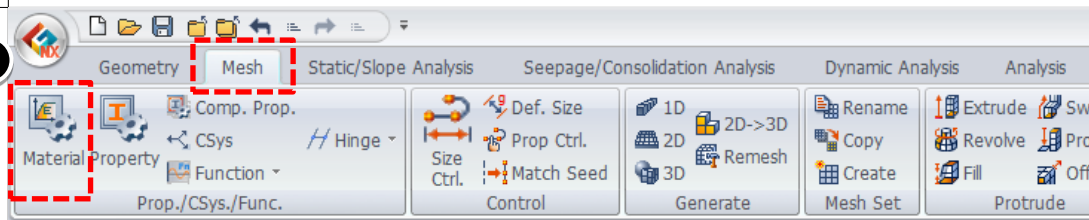
- ID: 1
- Name: Rockmass
- Model Type: Mohr-Coulomb
- Cohesion (C): 2200 kN/m^2
- Inc. of Cohesion: 0 kN/m^3
- Inc. of Cohesion Ref. Height: 0 m
- Frictional Angle (Φ): 30°
- Dilatancy Angle: 36 $^\circ$
- Tension Cut-off: Tension Cut-off (checked)
- Tensile Strength: 0 kN/m^2
- Cut-off Yield Surface: Rankine

2-3 Defining Structural Material

Procedure

Defining Structural Materials

- ① Go to **Mesh > Material**
- ② Click on **Create**. Select **Isotropic**
- ③ Select Model type as **Elastic**. Check on **structure**. Enter name **Initial Liner**
- ④ In General tab, Enter the value of **Elastic modulus (E) = 2e7 kN/m²**
- ⑤ Enter **Poisson's ratio = 0.2**, **Unit weight = 25 kN/m³**.
- ⑥ Click **OK**
Similarly enter the material data for **Final Liner**

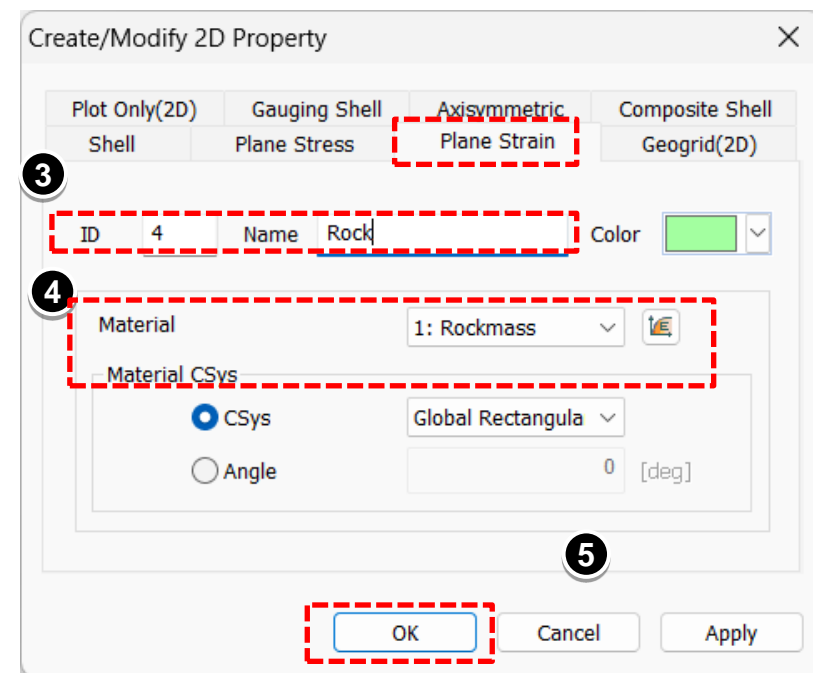
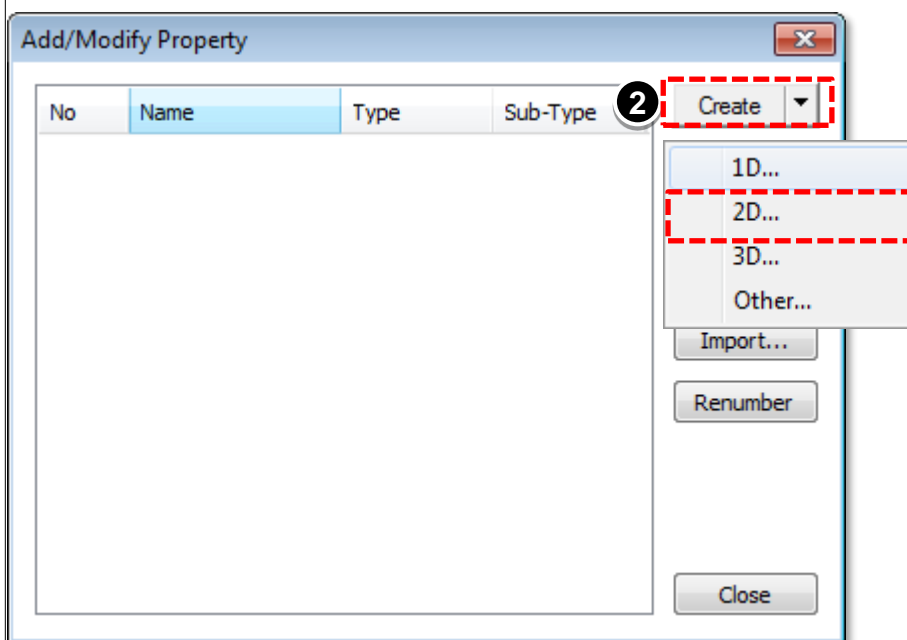
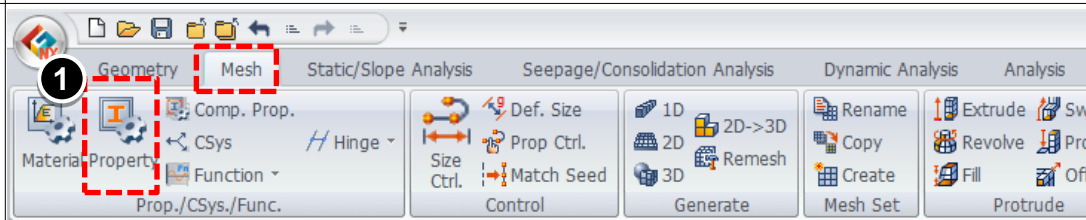


3-1 Defining Property

Procedure

Defining Property

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

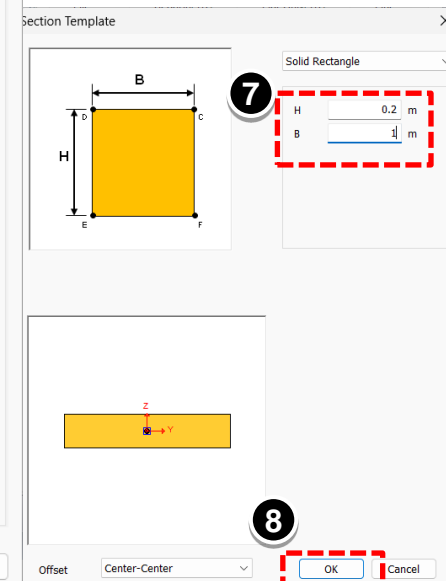
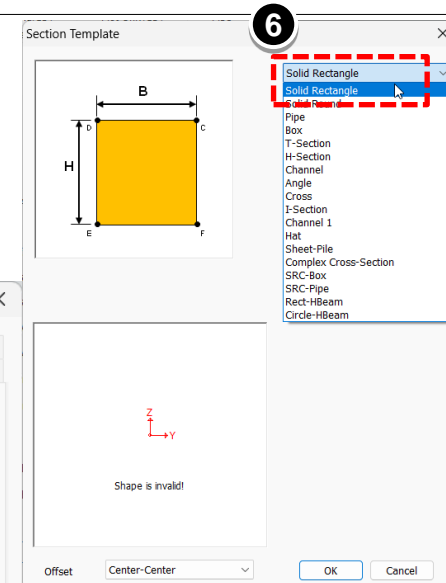
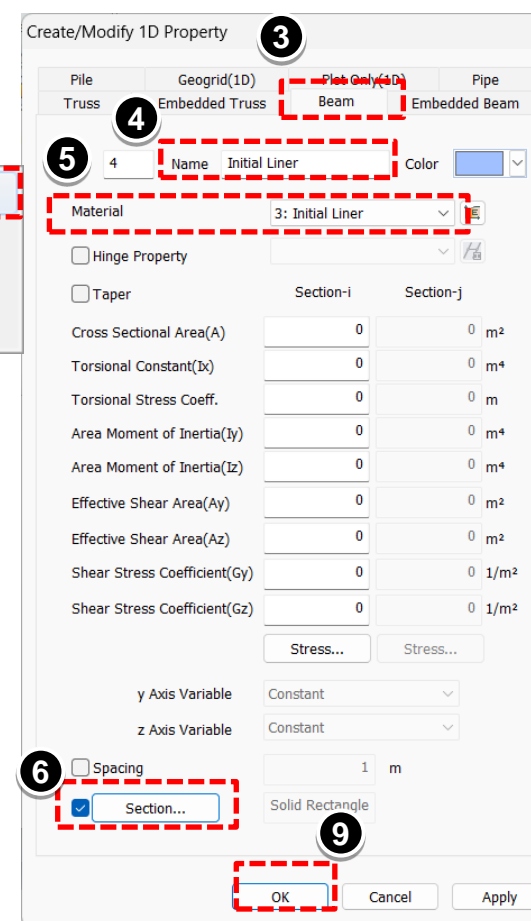
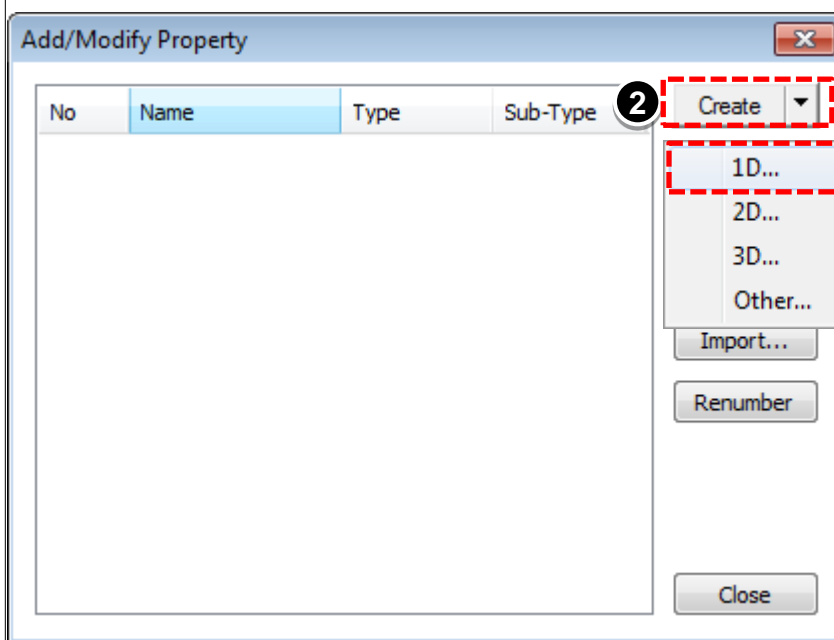
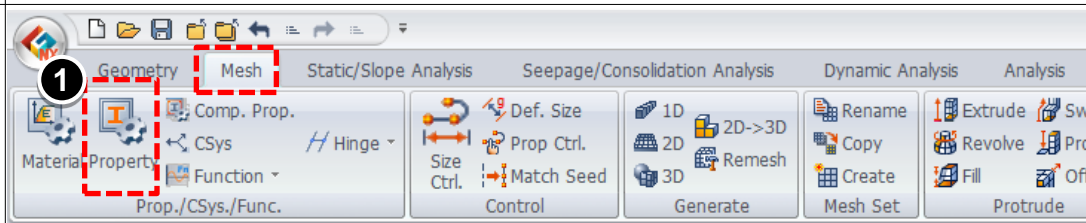


3-2 Defining Liner as Beam Element

Procedure

Defining Liner as Beam element.

- ① Go to **Mesh**>Click on **Property**
- ② Click on **Create**. Select **1D**
- ③ Click on **Beam**
- ④ Name it as **Initial Liner**.
- ⑤ Select material as **Initial Liner**.
- ⑥ Click **Section** and select **Solid Rectangle** from drop-down.
- ⑦ Enter **H=0.2m** and **B=1m**.
- ⑧ Click **OK**.
- ⑨ Click **OK**.

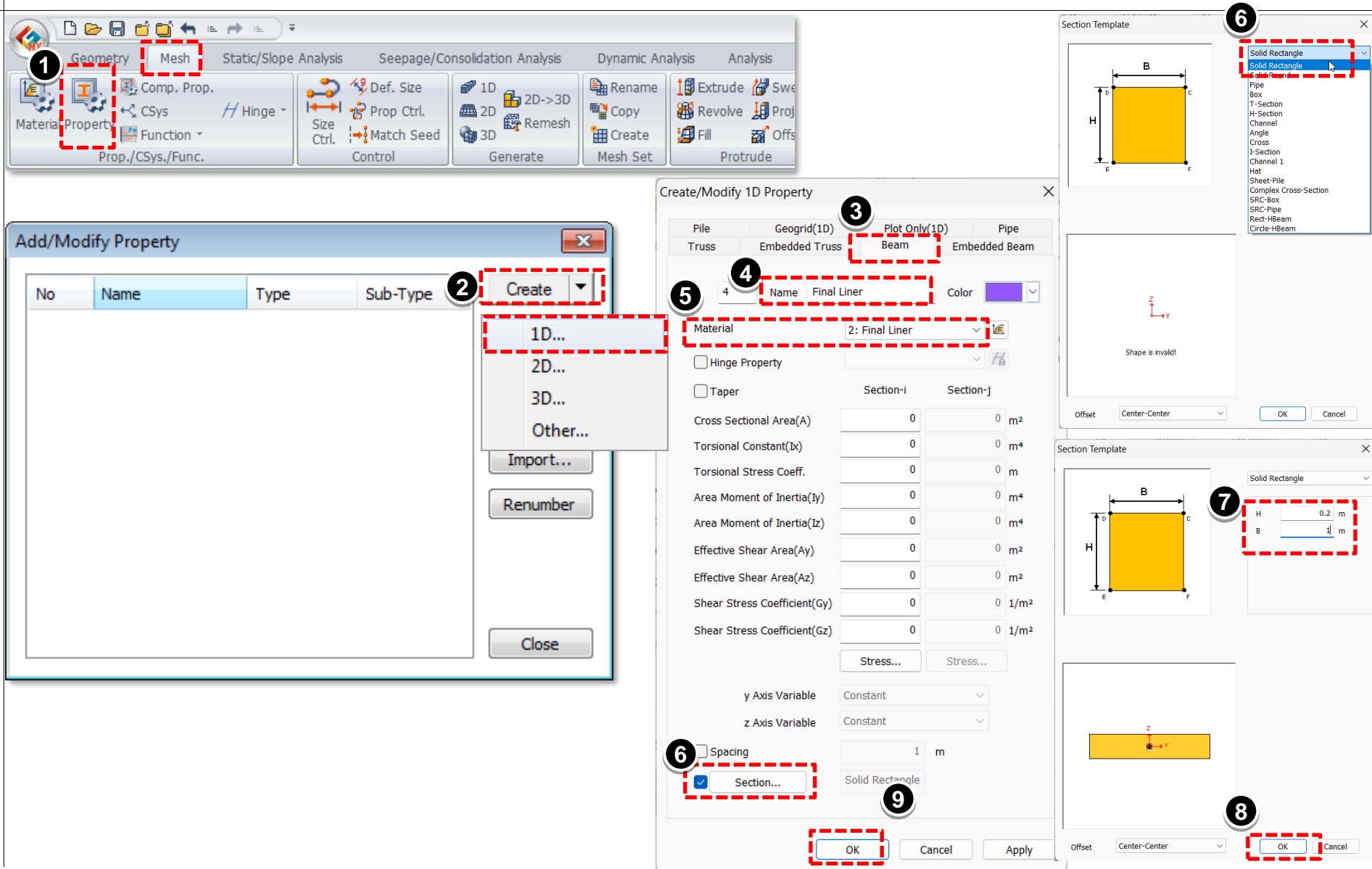


3-3 Defining Liner as Beam Element

Procedure

Defining Liner as Beam element.

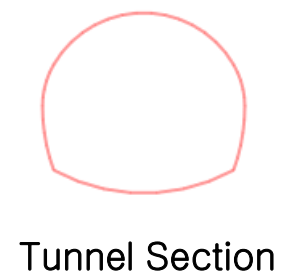
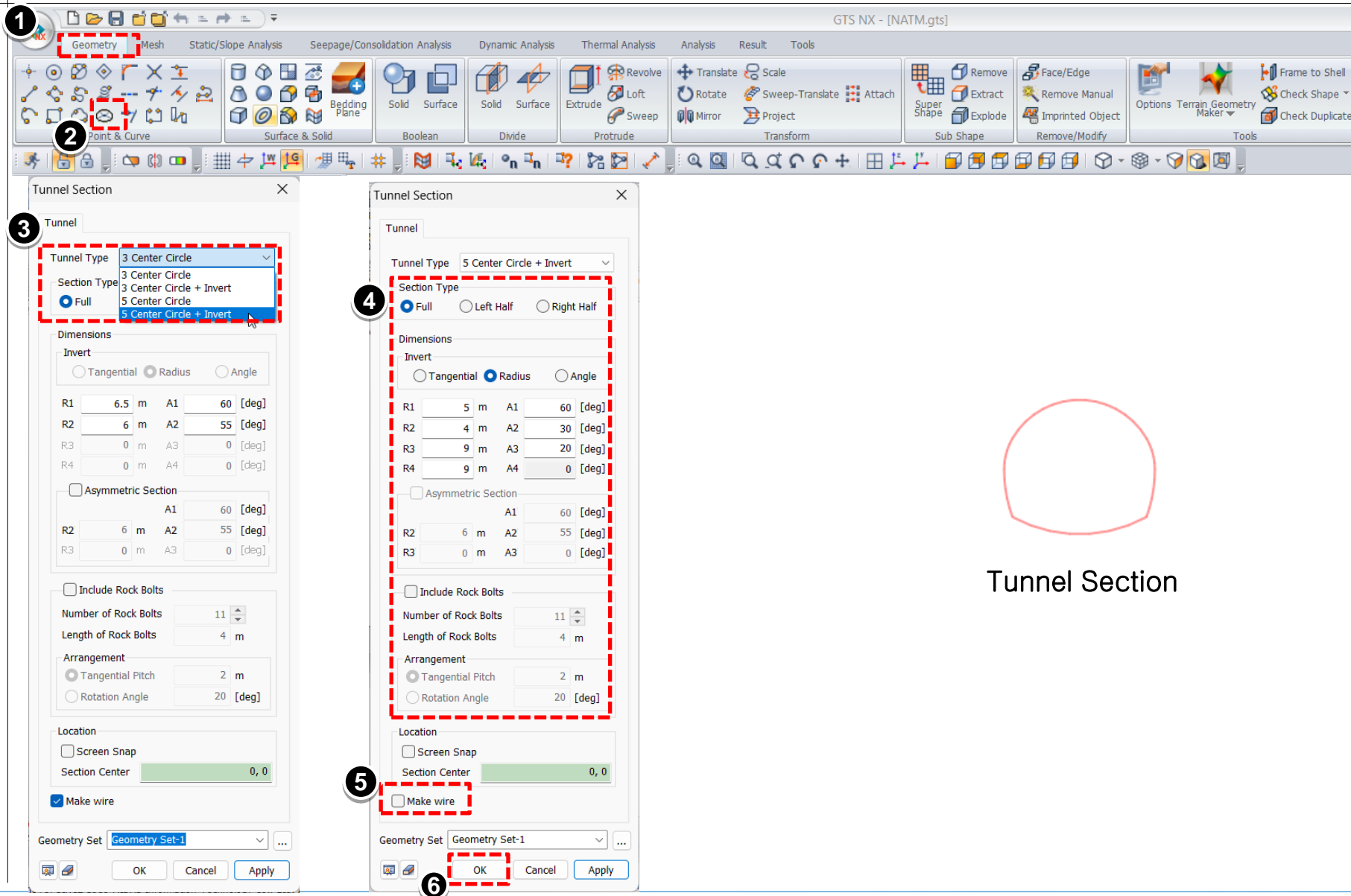
- ① Go to **Mesh**>Click on **Property**
- ② Click on **Create**. Select **1D**
- ③ Click on **Beam**
- ④ Name it as **Final Liner**.
- ⑤ Select material as **Final Liner**.
- ⑥ Click **Section** and select **Solid Rectangle** from drop-down.
- ⑦ Enter **H=0.3m** and **B=1m**.
- ⑧ Click **OK**.
- ⑨ Click **OK**.



4-1 Geometric modelling-Tunnel Section

Procedure


- ① Click on **Geometry**
- ② Click **Tunnel Section**
- ③ Select **5 Center Circle + Invert** as the tunnel Type.
- ④ Enter the dimensions as shown.
- ⑤ Uncheck **Make wire**.
- ⑥ Click **OK**.

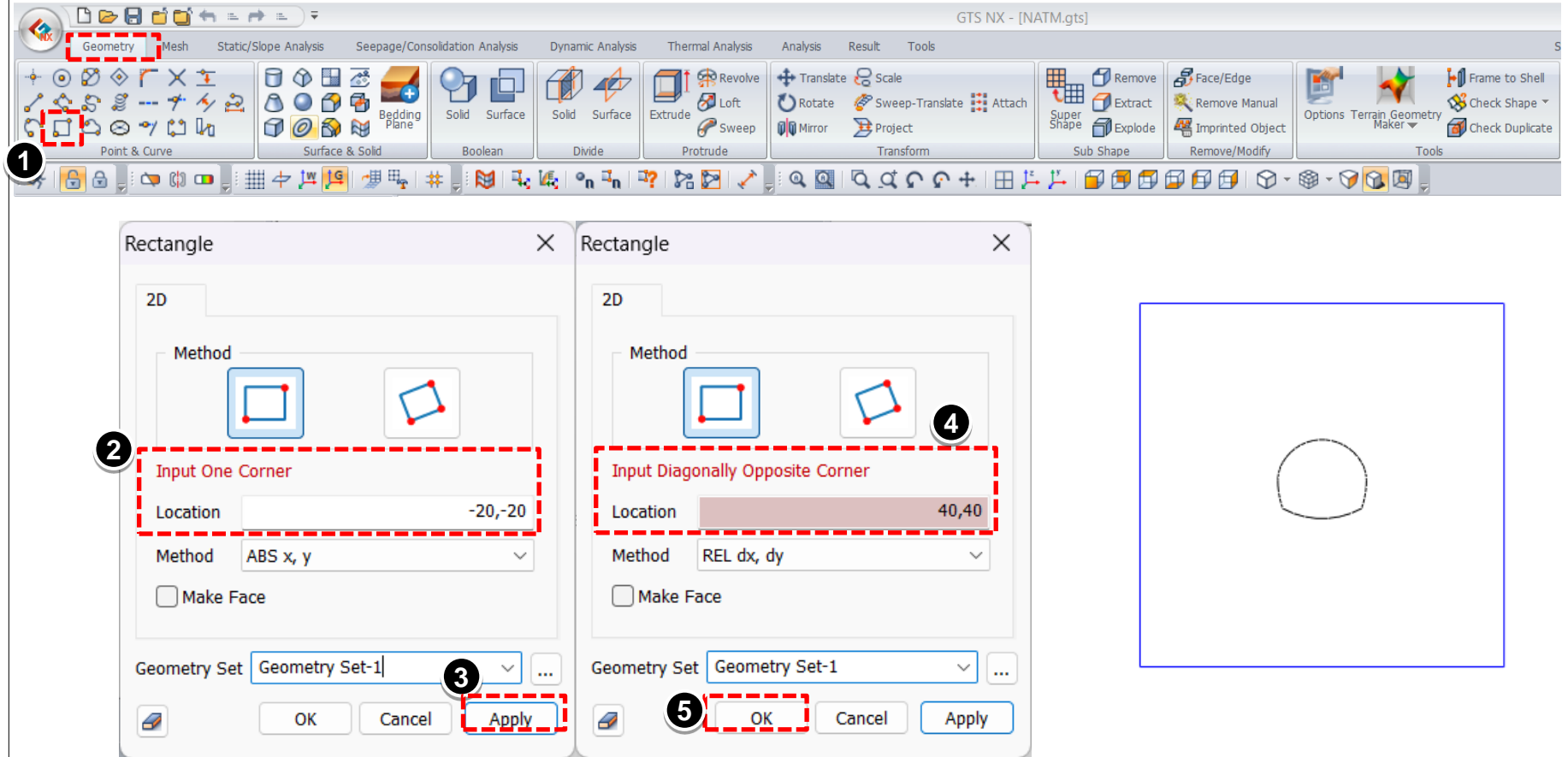


Tunnel Section

4-1 Geometric modelling-Ground Domain


Procedure

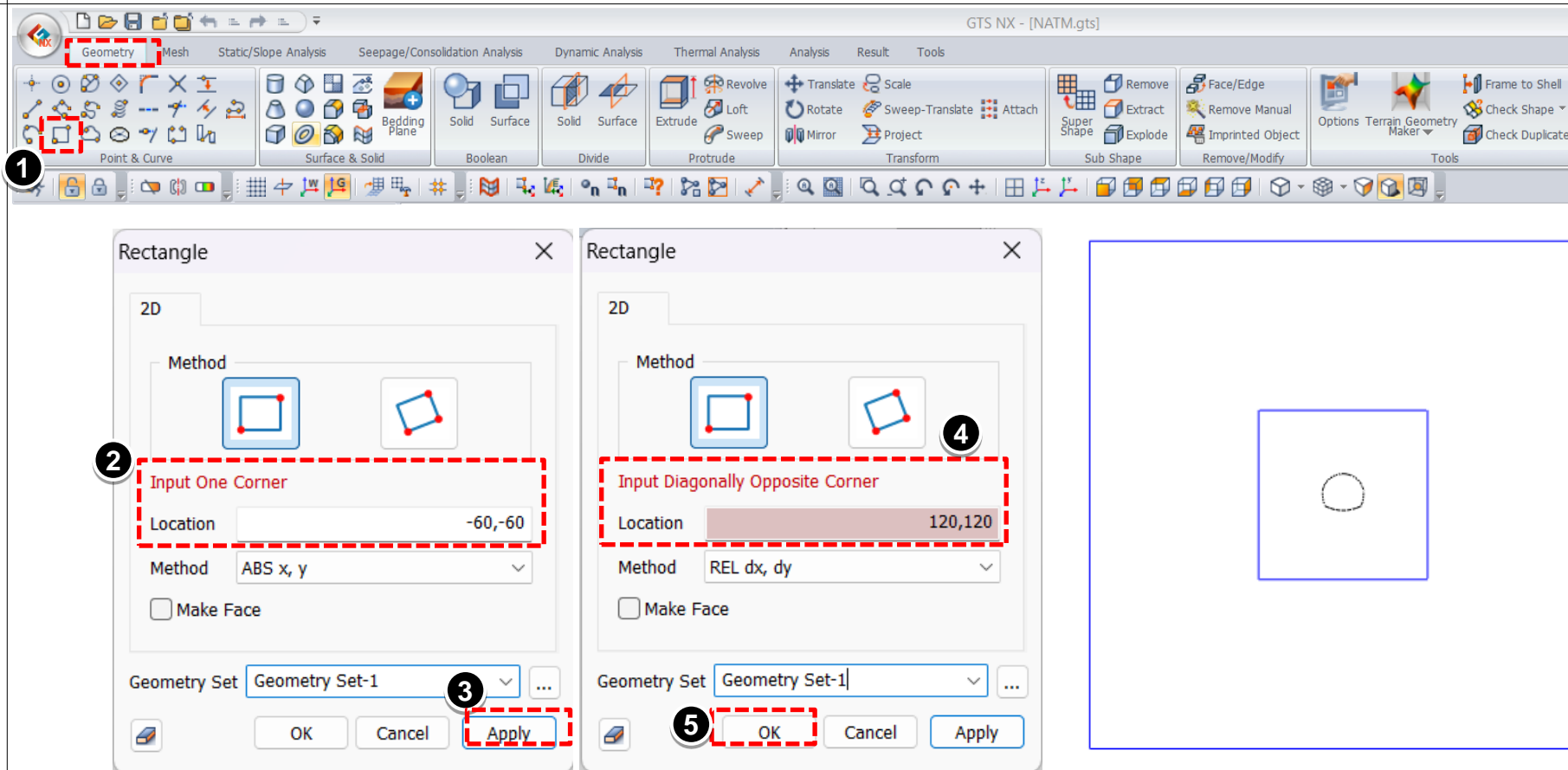
- ❶ Click **Rectangle** 
- ❷ Enter corner Location as **-20,-20**.
- ❸ Click **Apply**.
- ❹ Enter Diagonally Opposite Corner Location as **40,40**.
- ❺ Click **OK**.



4-1 Geometric modelling-Ground Domain

Procedure


- ① Click **Rectangle** 
- ② Enter corner Location as **-60,-60**.
- ③ Click **Apply**.
- ④ Enter Diagonally Opposite Corner Location as **120,120**.
- ⑤ Click **OK**.

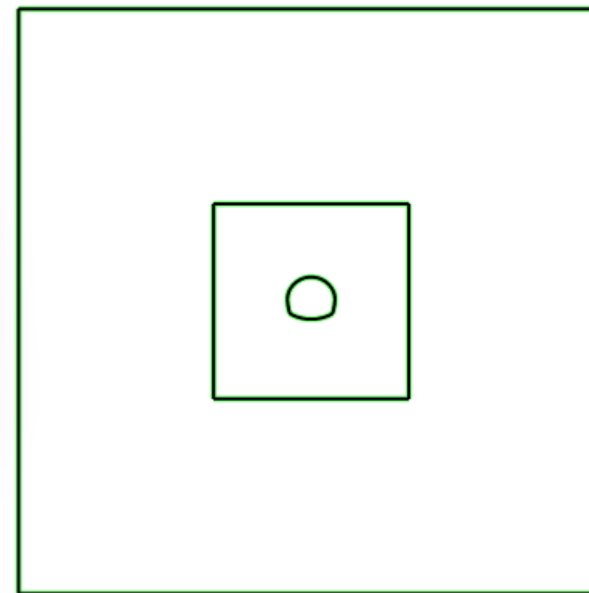
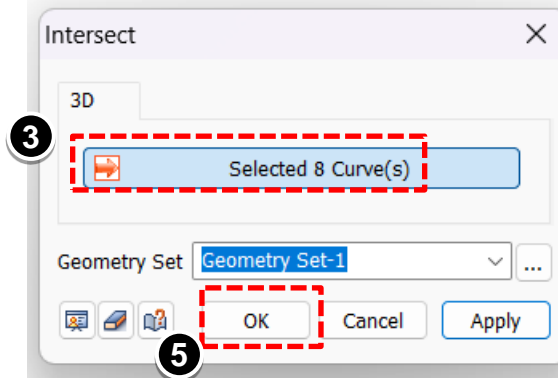
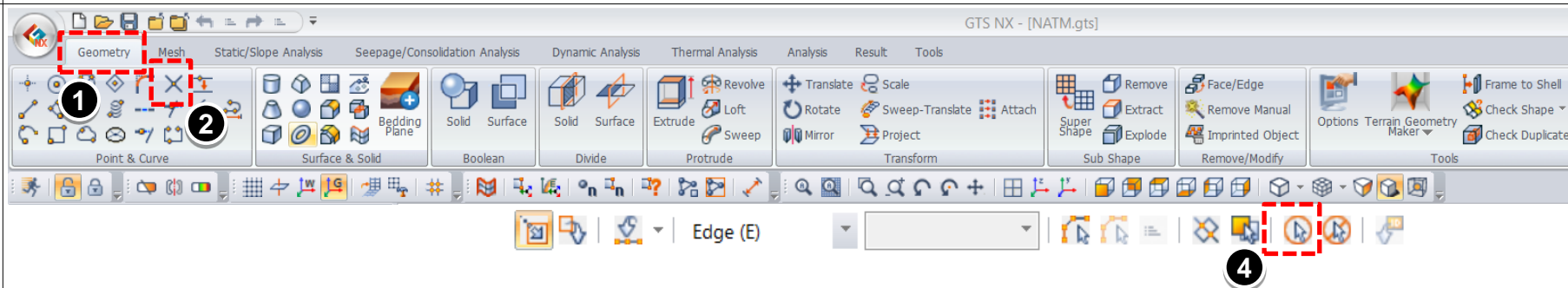


Model Geometry

4-2 Geometric modelling-Intersect

Procedure

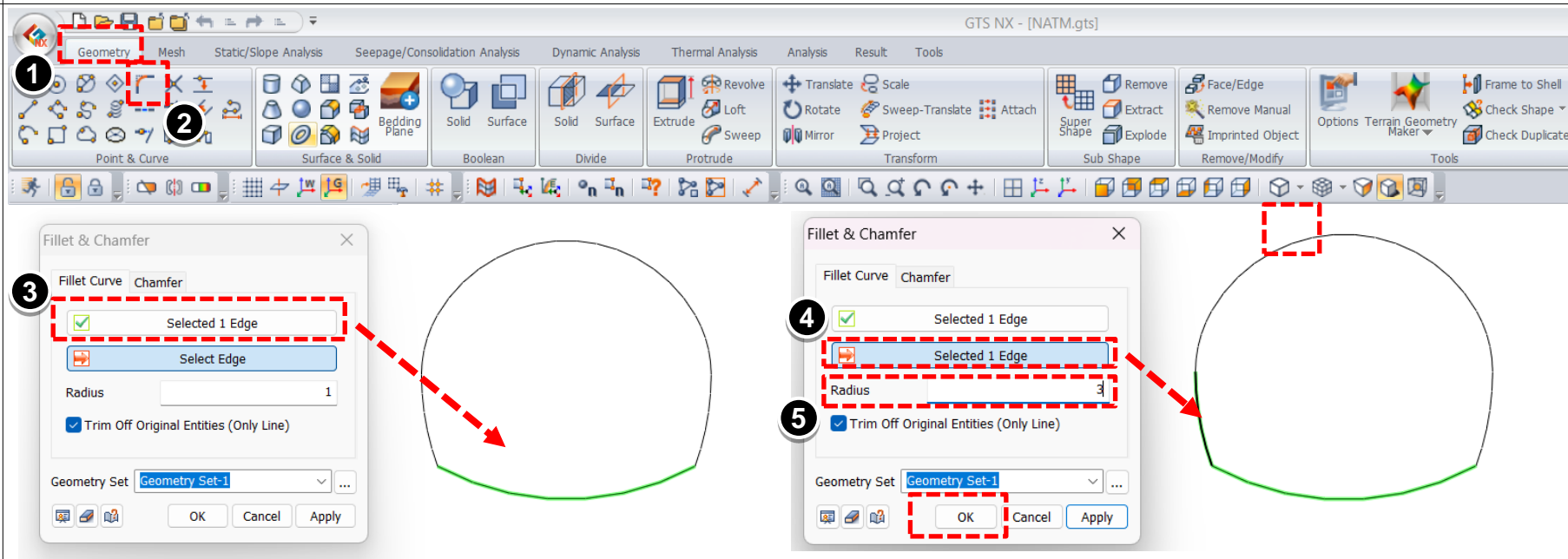
- 1 Go to **Geometry**
- 2 Click **Intersect**
- 3 Select all the curves.
- 4 Click  to select all the curves.
- 5 Click **OK**.



4-3 Geometric modelling-Fillet Curve

Procedure

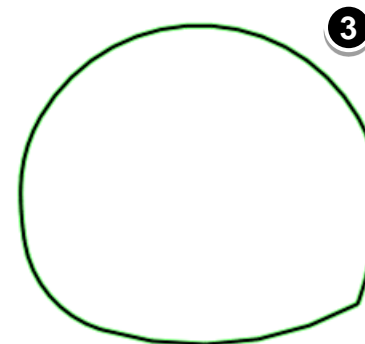
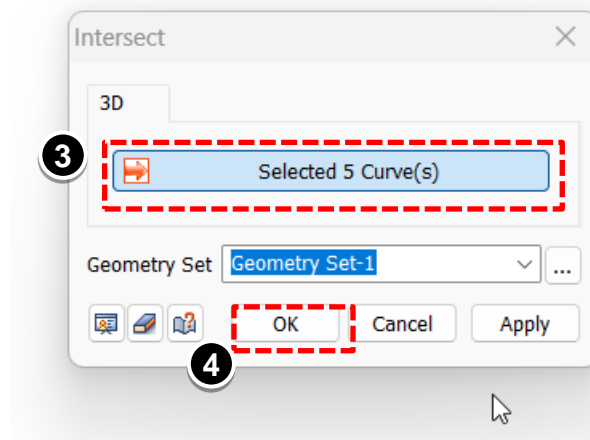
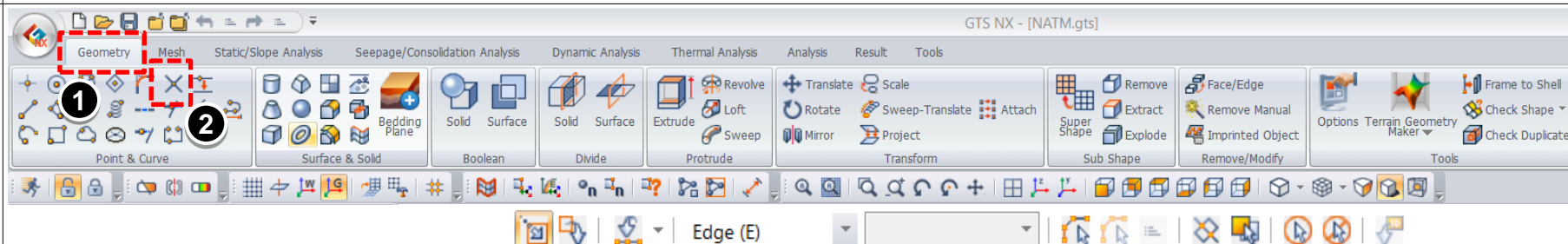
- ❶ Go to **Geometry**
- ❷ Click **Fillet Curve**
- ❸ Select the invert edge as shown.
- ❹ Select the tunnel edge as shown.
- ❺ Enter Radius as 3.
- ❻ Click **OK**.



4-4 Geometric modelling-Intersect

Procedure

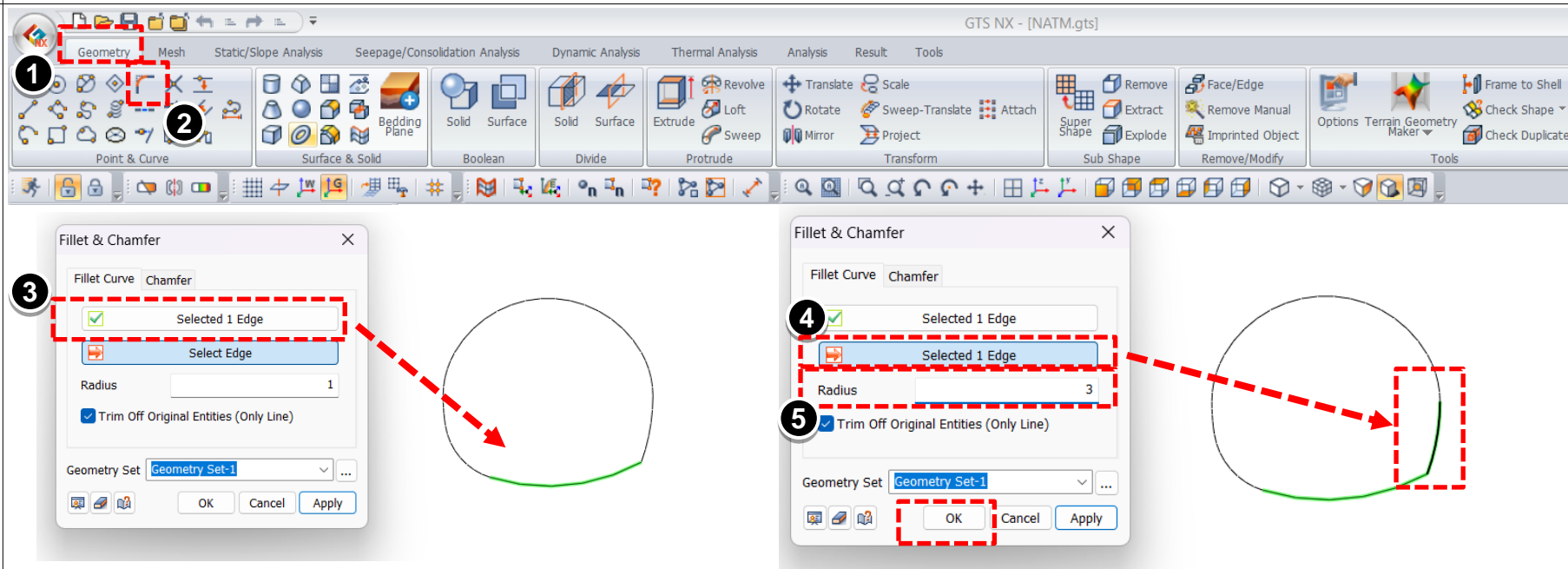
- 1 Go to **Geometry**
- 2 Click **Intersect**
- 3 Select the tunnel section curves.
- 4 Click **OK**.



4-5 Geometric modelling-Fillet Curve

Procedure

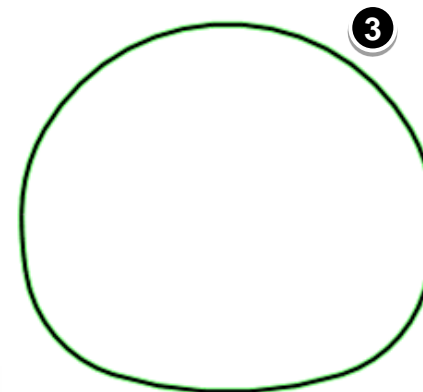
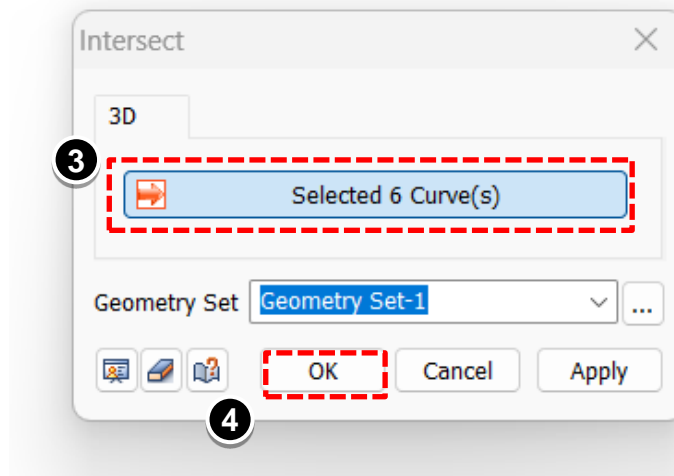
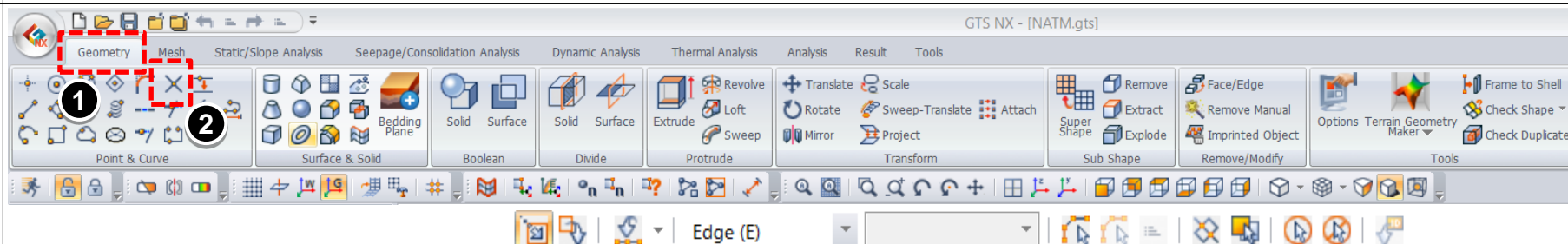
- 1 Go to **Geometry**
- 2 Click **Fillet Curve**
- 3 Select the invert edge as shown.
- 4 Select the tunnel edge as shown.
- 5 Enter Radius as 3.
- 6 Click **OK**.



4-6 Geometric modelling-Intersect

Procedure

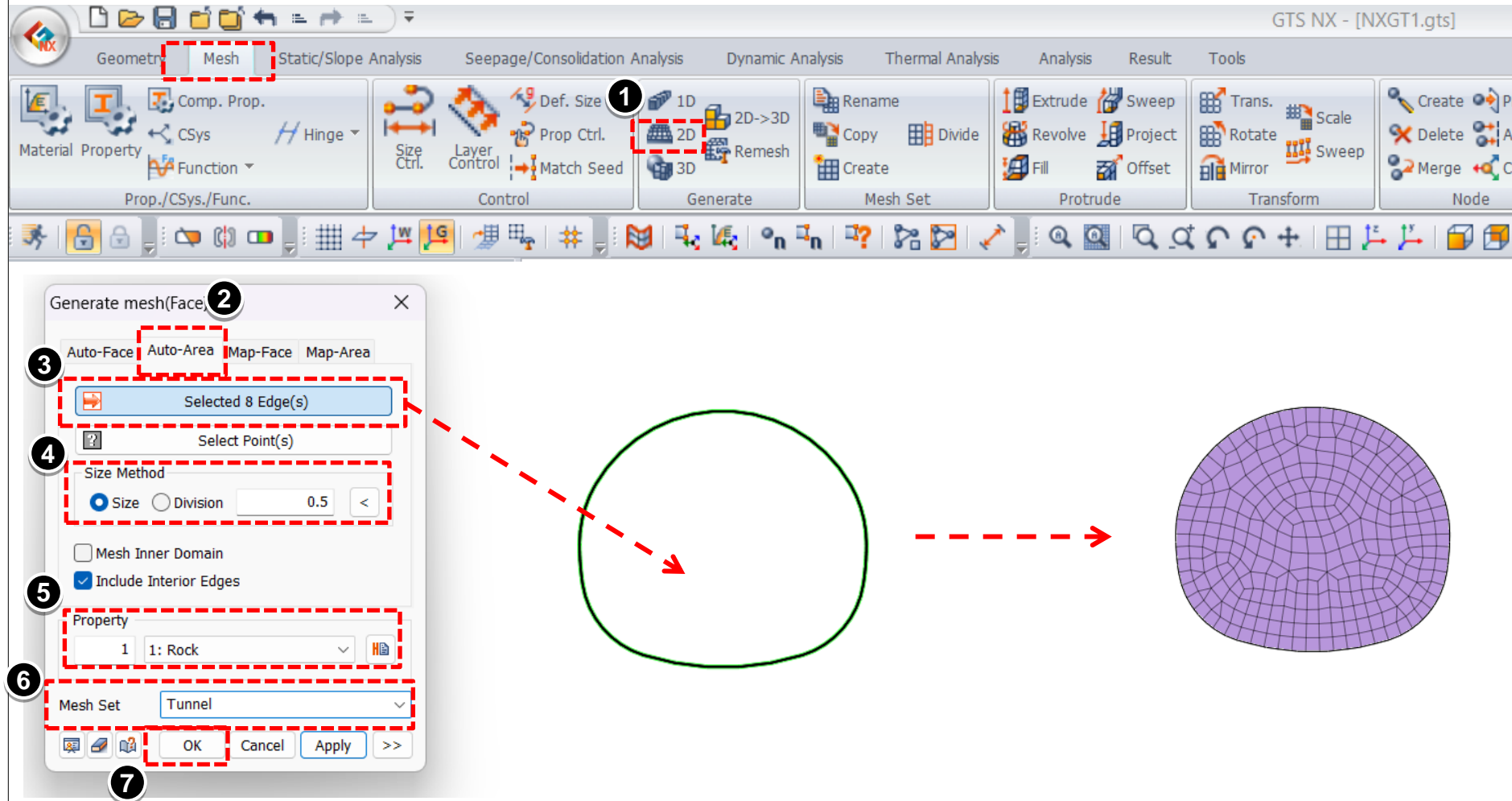
- 1 Go to **Geometry**
- 2 Click **Intersect**
- 3 Select the tunnel section curves.
- 4 Click **OK**.



5-1 Meshing-Tunnel

Procedure

- ❶ Go to **Mesh > Generate > 2D**
- ❷ Click **Auto-Area**.
- ❸ Select the **Tunnel** curves.
- ❹ Give size as **0.5**
- ❺ Select property as **Rock**.
- ❻ Give the Mesh set name **"Tunnel"**
- ❼ Click **OK**.

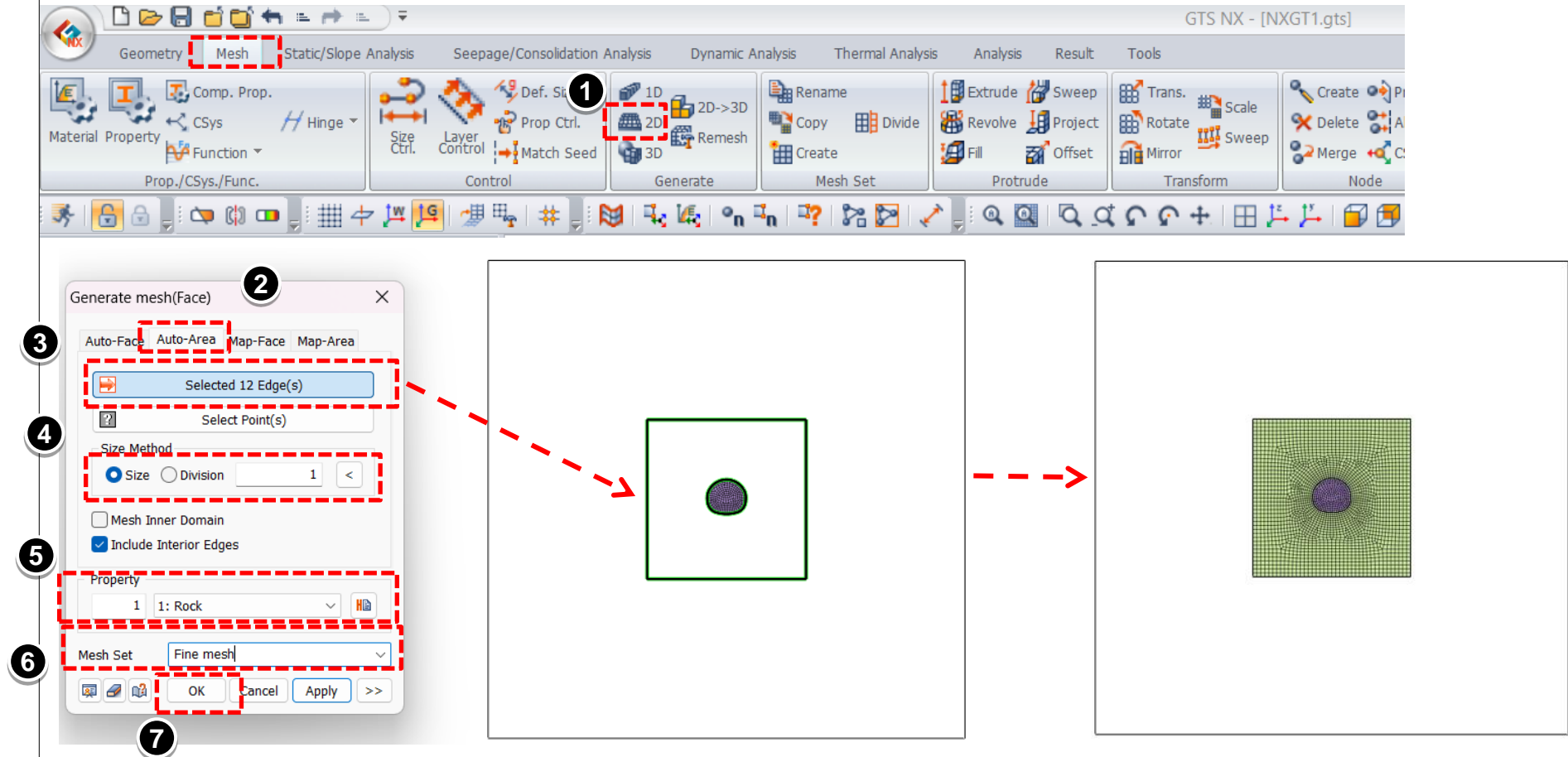


5-2 Meshing-Fine mesh area

Procedure

- ❶ Go to **Mesh > Generate > 2D**
- ❷ Click **Auto-Area**.
- ❸ Select the curves as shown.
- ❹ Give size as 1
- ❺ Select property as **Rock**.
- ❻ Give the Mesh set name "**Fine mesh**"
- ❼ Click **OK**.

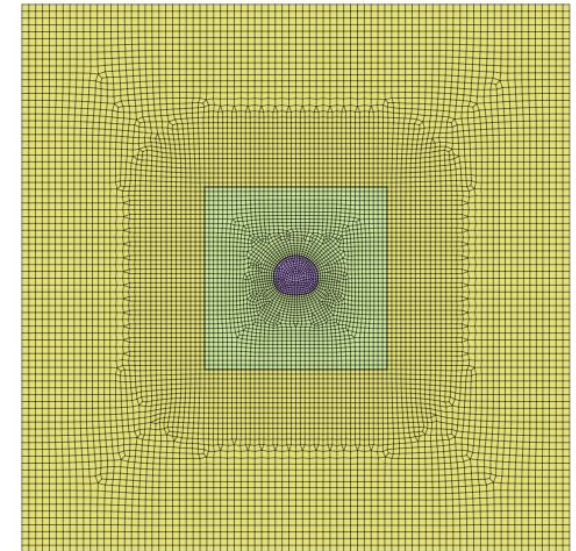
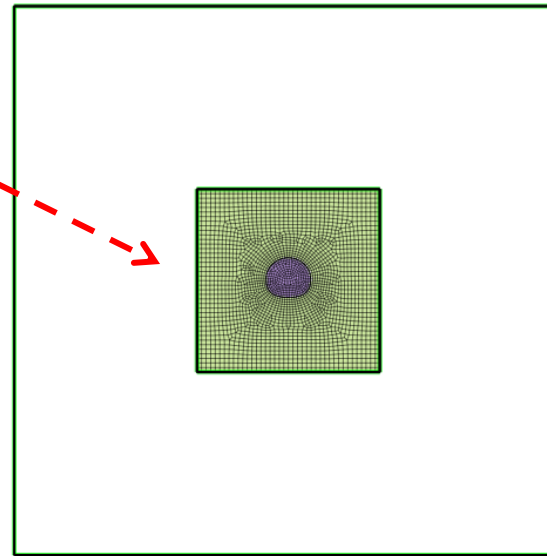
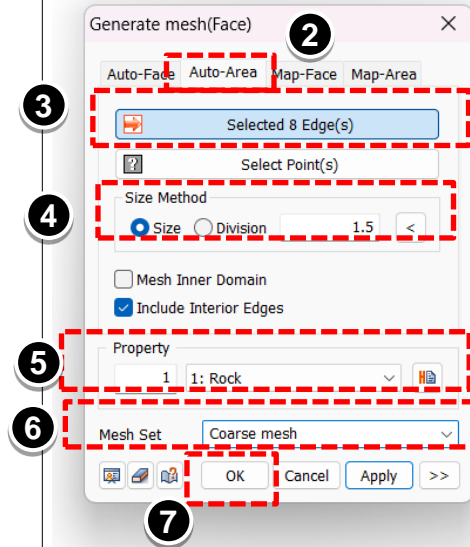
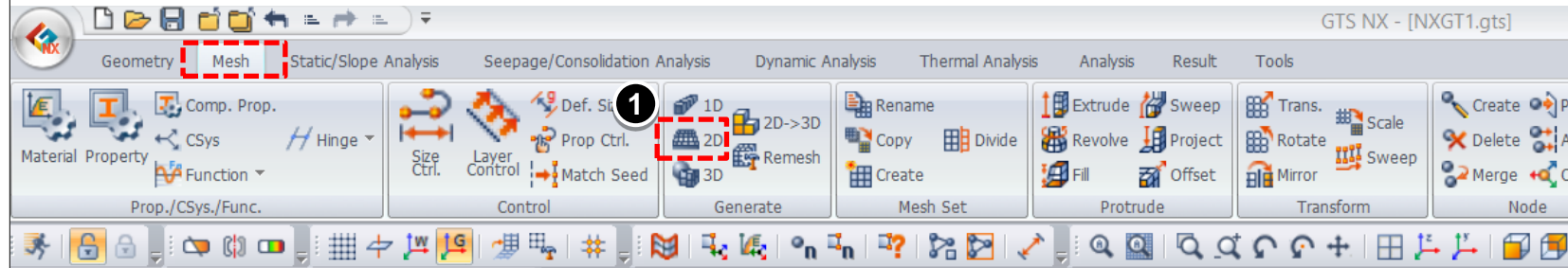
Fine mesh is assigned to some area near the tunnel section where stress concentration is expected. For the remaining area, relatively coarse mesh could be used.



5-3 Meshing-Coarse mesh area

Procedure

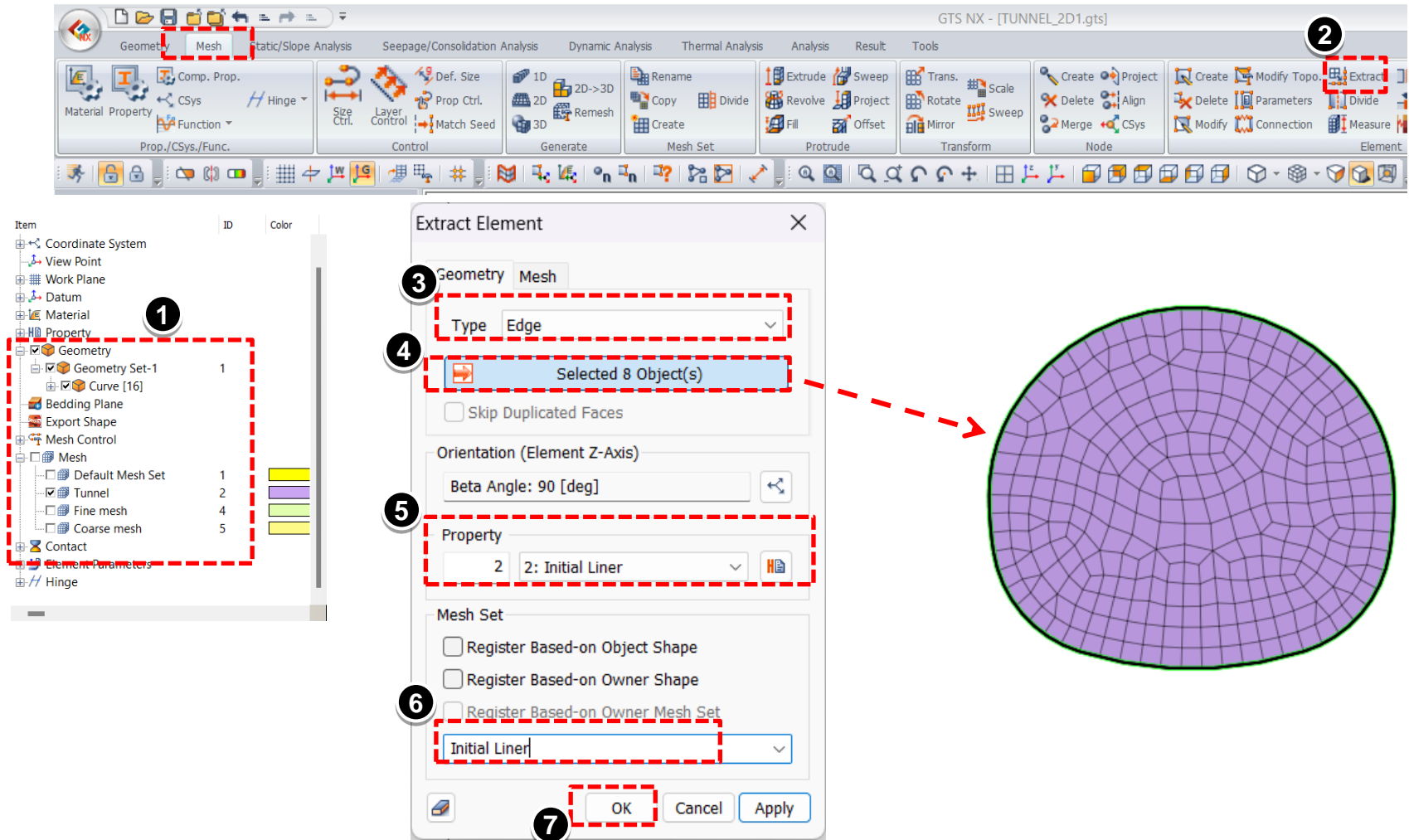
- ❶ Go to **Mesh > Generate > 2D**
- ❷ Click **Auto-Area**.
- ❸ Select the boundary curves as shown.
- ❹ Give size as **1.5**
- ❺ Select property as **Rock**.
- ❻ Give the Mesh set name **"Coarse mesh"**
- ❼ Click **OK**.



5-4 Meshing-Extract Liner

Procedure

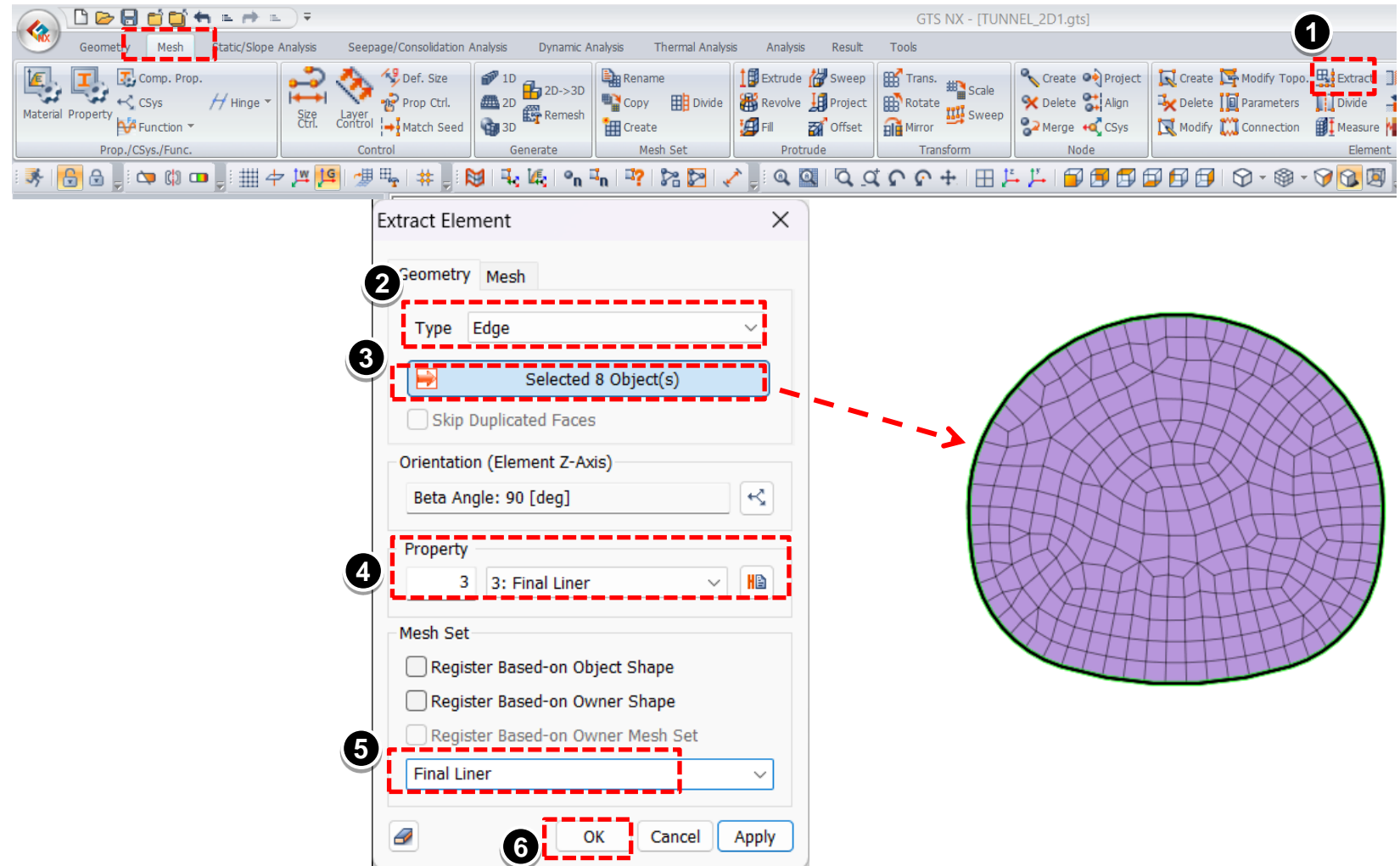
- ① Enable all the geometry curves and only the Tunnel mesh.
- ② Go to **Mesh > Element > Extract**
- ③ Select the Type as **Edge**.
- ④ Select the **tunnel section edges**.
- ⑤ Select the property as **Initial Liner**.
- ⑥ Give the Mesh set name "**Initial Liner**"
- ⑦ Click **OK**.



5-5 Meshing-Extract Liner

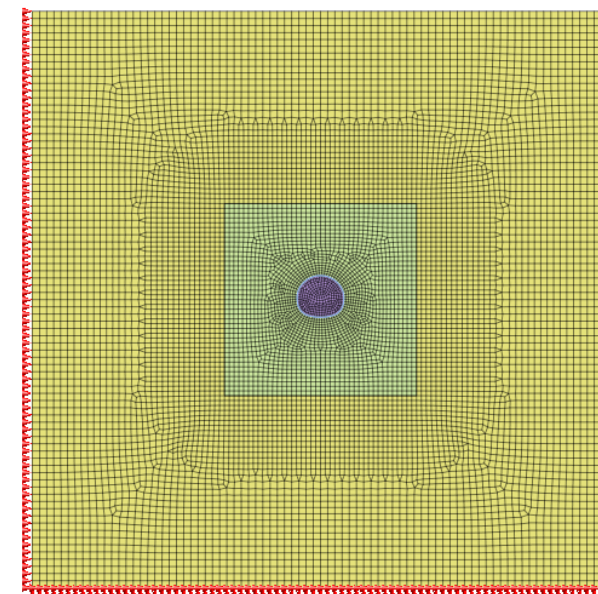
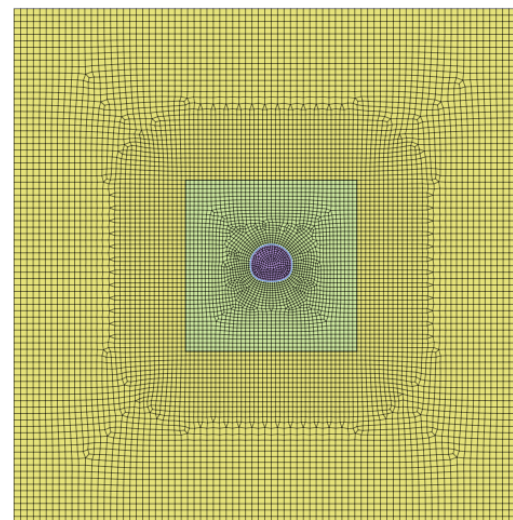
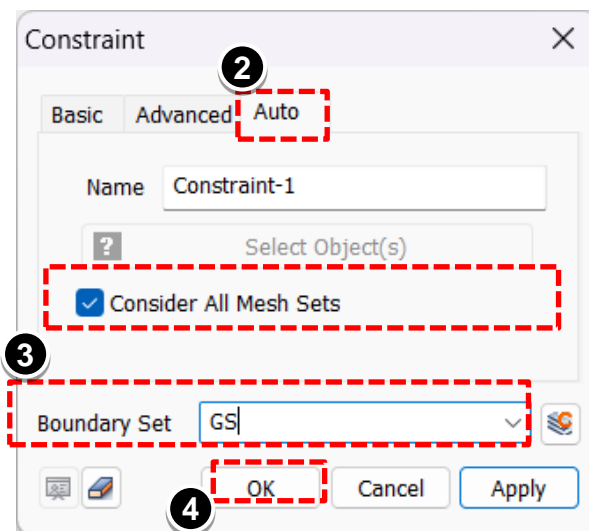
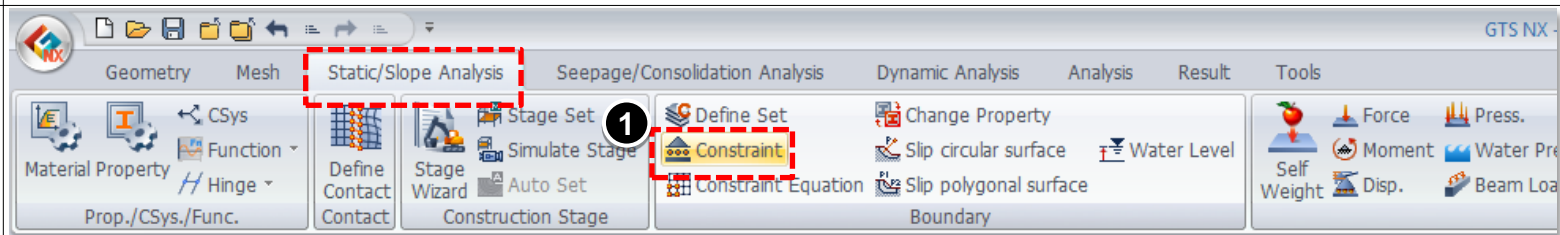
Procedure

- 1 Go to **Mesh > Element > Extract**
Select the Type as **Edge**.
- 2 Select the **tunnel section edges**.
- 3 Select the property as **Final Liner**.
- 4 **Liner**.
- 5 Give the Mesh set name "**Final Liner**".
- 6 Click **OK**.



Procedure

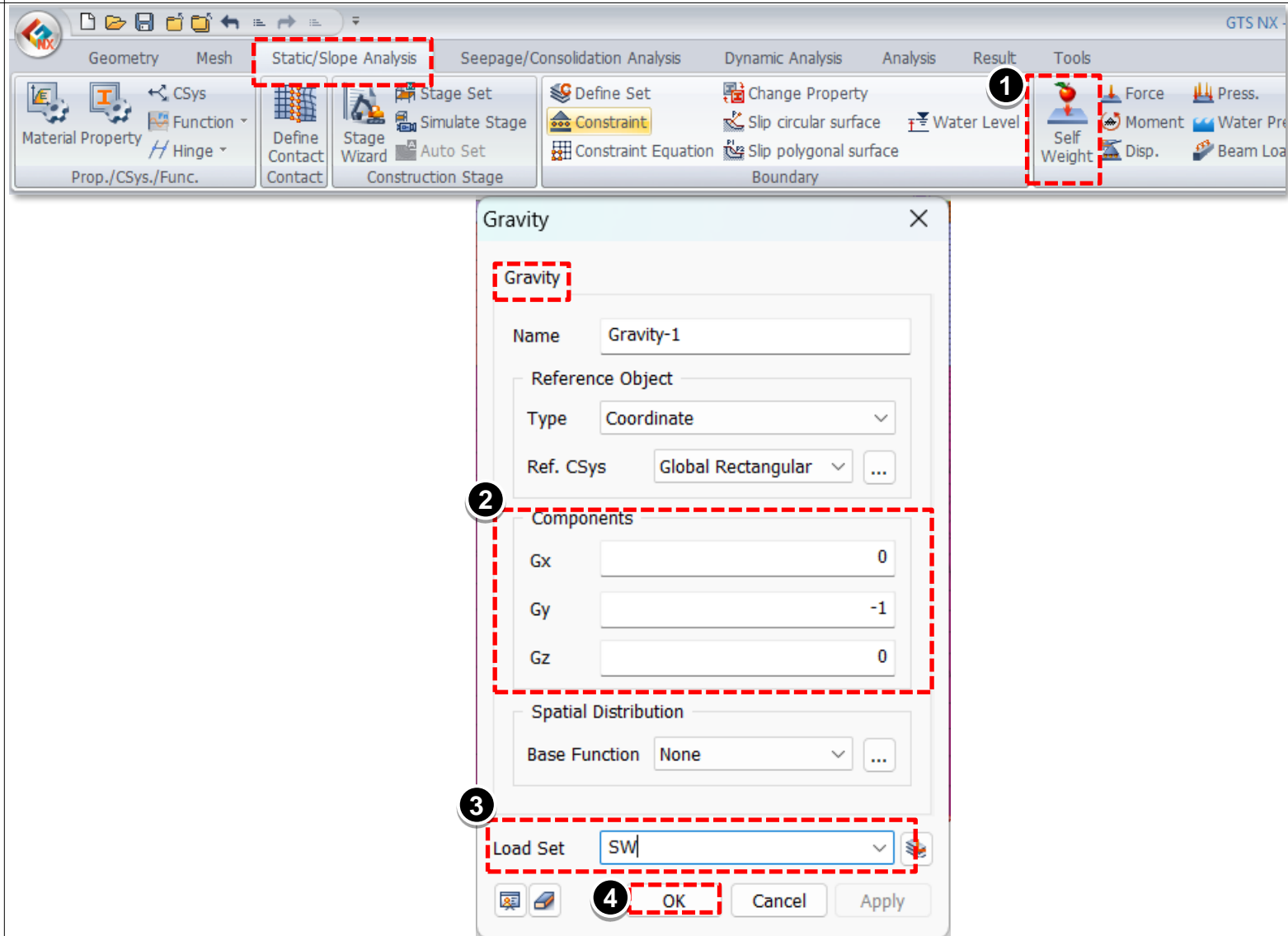
- ❶ Go to **Static/Slope Analysis > Constraint**
- ❷ Select **Auto**, “**Consider All mesh Sets**” is already checked on.
- ❸ Name the boundary Set as “**GS**”
- ❹ Click **OK**



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**

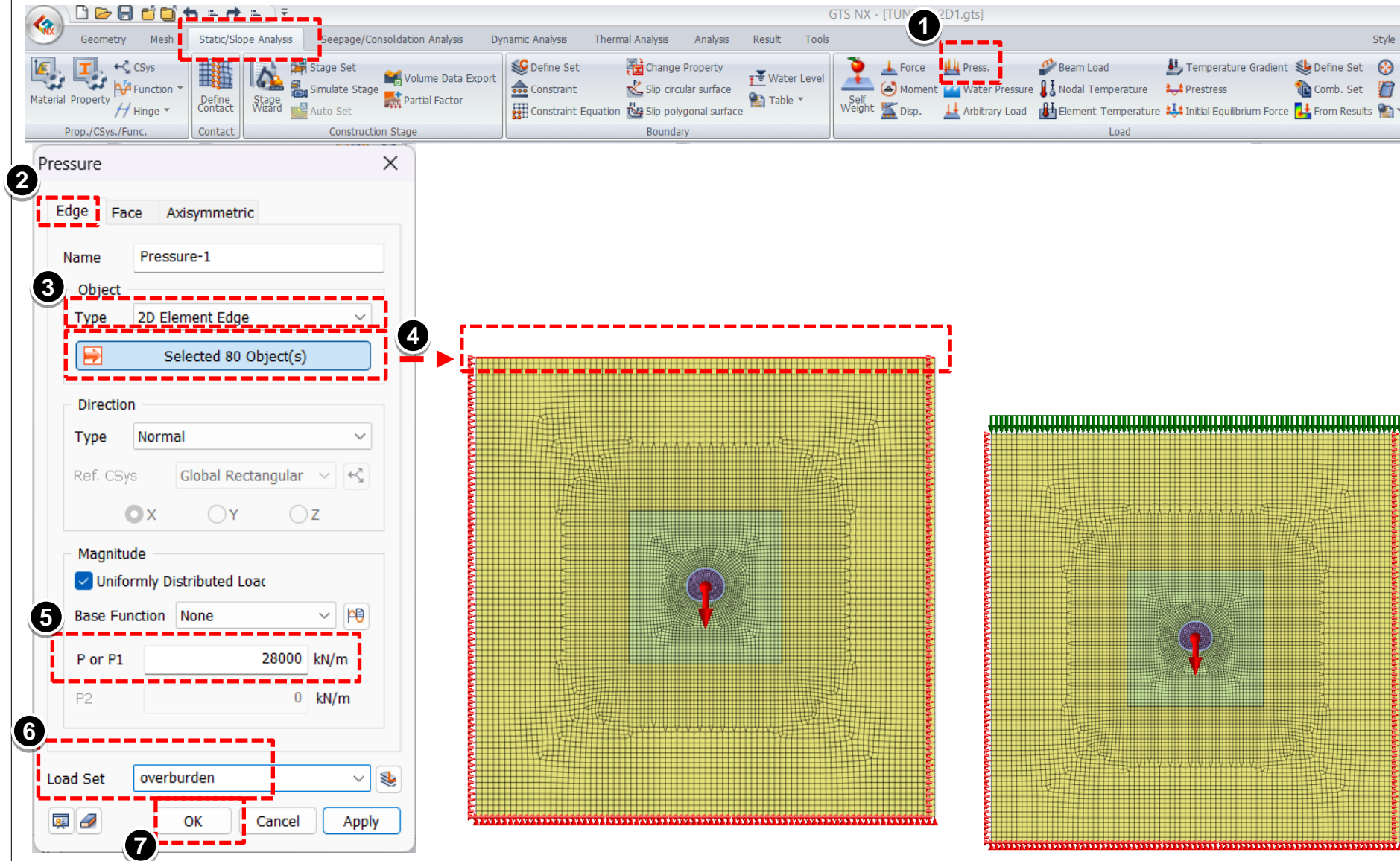


7-2 Load Definition-Overburden Pressure

Procedure

- ➊ Go to **Static/Slope Analysis > Load > Pressure**
- ➋ Go to **Edge**
- ➌ Select Type as **2D Element Edge**
- ➍ Select the element edges as shown.
- ➎ Enter value as **28000**.
- ➏ Name the Load Set as **"overburden"**
- ➐ Click **OK**

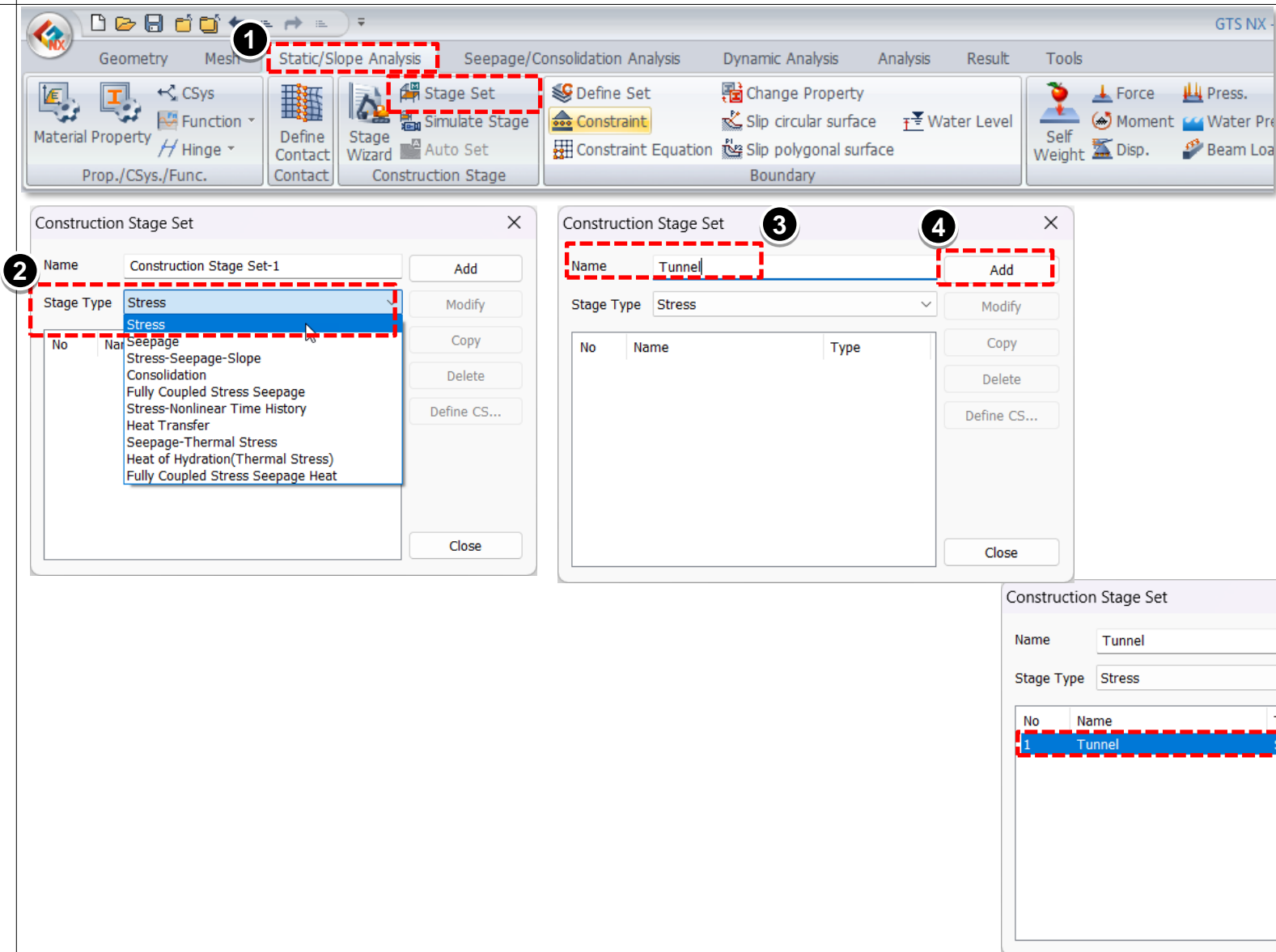
Since we are not modelling the entire overburden in the model, we are applying it as an equivalent pressure load on the top.



8-0 Construction stage Definition

Procedure

- ❶ Go to **Static/Slope Analysis > Construction Stage-> Stage Set**
- ❷ Select the Stage Type as **Stress**
- ❸ Enter the name as **Tunnel**
- ❹ Click **Add**
- ❺ Select the Stage Definition created and click on **Define CS.**



8-1 Construction stage Definition

Procedure

- ❶ Put Stage name as **In situ**
- ❷ Drag and drop **Coarse mesh, Fine mesh and Tunnel** Mesh sets into Activated data column.
- ❸ Similarly activate the **GS** Boundary condition and loads **SW** and **overburden**.
- ❹ Click on **Save**
- ❺ Click **New**.

Define Construction Stage

Construction Stage Set Name: Tunnel

Stage ID: 1: Construction Stage-1

Stage Name: In situ

Stage Type: Stress

Buttons: Move to Previous, Move to Next, New, Insert, Delete

Initial Condition:

- ☐ Define Water Level For Global: 0 m, None
- ☐ Define Water Level For Mesh Set: Input Water Level...
- ☐ Sub Stage...
- ☐ LDF... Copy To Specific Stage...
- ☐ Clear Displacement
- ☐ Slope Stability(SRM)
- ☐ Slope Stability(SAM)

Set Data:

- Mesh
 - Coarse mesh
 - Default Mesh Set
 - Final Liner
 - Fine mesh
 - Initial Liner
 - Tunnel
- Boundary Condition
 - GS
- Static Load
 - SW
 - overburden
- Contact

Activated Data:

- Mesh
 - Coarse mesh
 - Fine mesh
 - Tunnel
- Boundary Condition
 - GS
- Static Load
 - SW
 - overburden
- Contact

Deactivated Data:

- Mesh
 - Boundary Condition
 - Static Load
 - Contact

Sort By: Name

Show Data: All

Buttons: Save, Close

8-2 Construction stage Definition

Procedure

- 1 Put Stage name as **Clear Displacement**
- 2 Check on **Clear Displacement**.
- 3 Click on **Save**
- 4 Click **New**.

Define Construction Stage

Construction Stage Set Name: Tunnel

Stage ID: 2: Construction Stage-2

Stage Name: Clear Displacement

Stage Type: Stress

Buttons: Move to Previous, Move to Next, New, Insert, Delete

Set Data:

- Mesh
 - Coarse mesh
 - Default Mesh Set
 - Final Liner
 - Fine mesh
 - Initial Liner
 - Tunnel
- Boundary Condition
- GS
- Static Load
- SW
- overburden
- Contact

Activated Data:

- Mesh
- Boundary Condition
- Static Load
- Contact

Deactivated Data:

- Mesh
- Boundary Condition
- Static Load
- Contact

Initial Condition:

- ☐ Define Water Level For Global
 - 0 m
 - None
 - ...
- ☐ Define Water Level For Mesh Set
 - Input Water Level...
- ☐ Sub Stage...
- ☐ LDF...
 - Copy To Specific Stage...
- ☒ Clear Displacement
- ☐ Slope Stability(SRM)
- ☐ Slope Stability(SAM)

Sort By: Name

Show Data: All

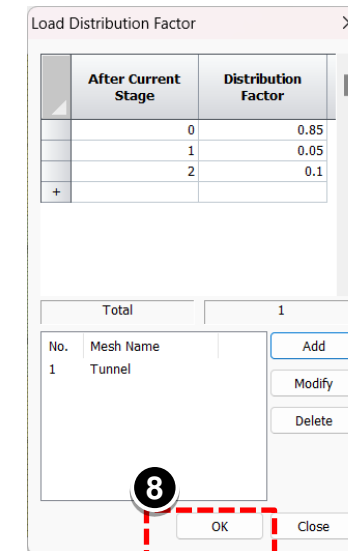
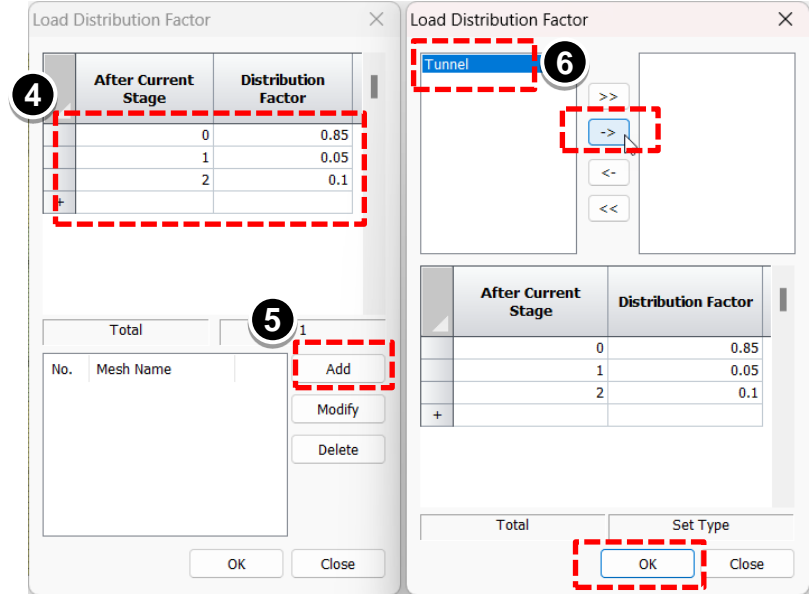
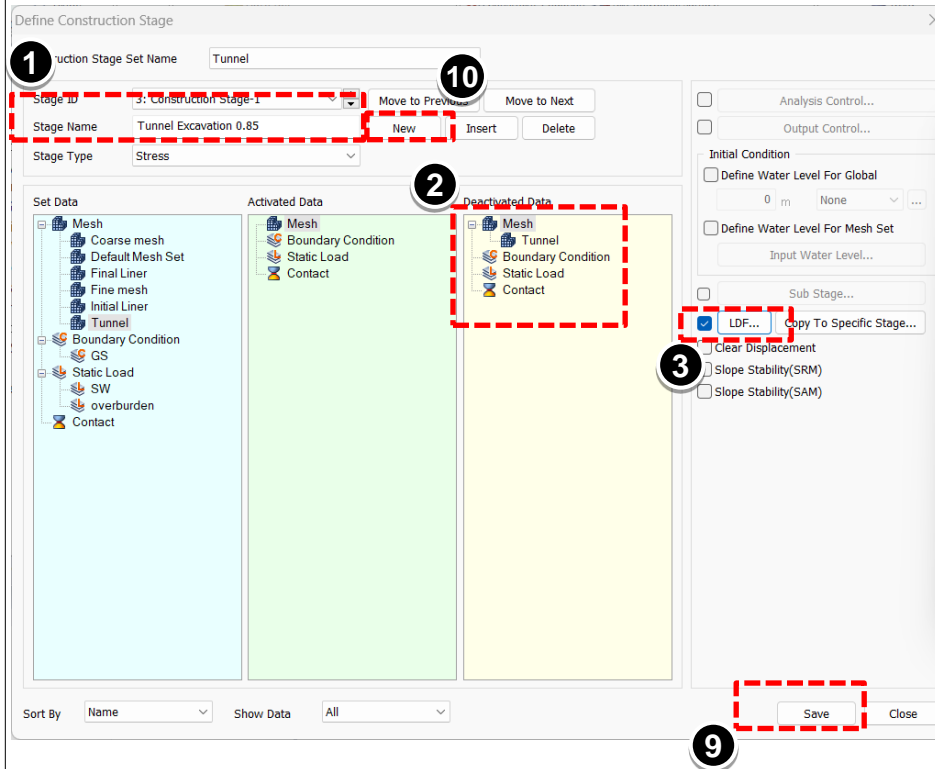
Buttons: Save, Close

8-3 Construction stage Definition

Procedure

- ❶ Put Stage name as **Tunnel Excavation 0.85**
- ❷ Drag **Tunnel** mesh into Deactivated Data.
- ❸ Check on and click 'LDF'.
- ❹ Enter the distribution factors as shown.
- ❺ Click **Add**.
- ❻ Click on **Tunnel** mesh and to activate it.
- ❼ Click **OK**.
- ❽ Click **OK**.
- ❾ Click **Save**.
- ❿ Click **New**.

LDF is to account for stress relaxation in the tunnel section. We are allowing 85% relaxation in this stage, and next 5% and 10% in the subsequent stages.



8-4 Construction stage Definition

Procedure

- 1 Put Stage name as **Tunnel Excavation 0.9**
- 2 Click on **Save**
- 3 Click **New**.

Define Construction Stage

Construction Stage Set Name: Tunnel

1 Stage ID: 4: Construction Stage-1

Stage Name: Tunnel Excavation 0.9

Stage Type: Stress

3 New

Move to Previous Move to Next

Insert Delete

Set Data

- Mesh
 - Coarse mesh
 - Default Mesh Set
 - Final Liner
 - Fine mesh
 - Initial Liner
 - Tunnel
- Boundary Condition
- GS
- Static Load
- SW
- overburden
- Contact

Activated Data

- Mesh
 - Boundary Condition
 - Static Load
 - Contact

Deactivated Data

- Mesh
 - Boundary Condition
 - Static Load
 - Contact

Initial Condition

☐ Define Water Level For Global

0 m None ...

☐ Define Water Level For Mesh Set

Input Water Level...

☐ Sub Stage...

☐ LDF... Copy To Specific Stage...

☐ Clear Displacement

☐ Slope Stability(SRM)

☐ Slope Stability(SAM)

2 Save

Close

Sort By: Name

Show Data: All

8-5 Construction stage Definition

Procedure

- 1 Put Stage name as **Initial Liner**
- 2 Activate **Initial Liner mesh**.
- 3 Click on **Save**
- 4 Click **New**.

Define Construction Stage

Construction Stage Set Name: Tunnel

1 Stage ID: 5: Construction Stage-1 4 Move to Previous Move to Next

Stage Name: Initial Liner New Insert Delete

Stage Type: Stress

2

Set Data

- Mesh
 - Coarse mesh
 - Default Mesh Set
 - Final Liner
 - Fine mesh
 - Initial Liner
 - Tunnel
- Boundary Condition
 - GS
- Static Load
 - SW
 - overburden
- Contact

Activated Data

- Mesh
 - Initial Liner
 - Boundary Condition
 - Static Load
 - Contact

Deactivated Data

- Mesh
 - Boundary Condition
 - Static Load
 - Contact

Initial Condition

☐ Define Water Level For Global

0 m None ...

☐ Define Water Level For Mesh Set

Input Water Level...

☐ Sub Stage...

☐ LDF... Copy To Specific Stage...

☐ Clear Displacement

☐ Slope Stability(SRM)

☐ Slope Stability(SAM)

3 Save Close

Sort By: Name Show Data: All

8-6 Construction stage Definition

Procedure

- ❶ Put Stage name as **Final Liner**
- ❷ Activate **Final Liner** mesh and deactivate **Initial Liner** mesh.
- ❸ Click on **Save** and then **Close**.

The initial liner, in many cases, is shotcrete which could be eroded over time, so we are not going to consider any resistance due to the initial liner, we will deactivate it such that entire load is applied on the final liner.

Define Construction Stage

Construction Stage Set Name: Tunnel

❶ Stage ID: 6: Construction Stage-1

Stage Name: Final Liner

Stage Type: Stress

Move to Previous Move to Next

New Insert Delete

Set Data

❷ Activated Data

Deactivated Data

Save Close

Sort By: Name Show Data: All

Analysis Control... Output Control...

Initial Condition

Define Water Level For Global: 0 m None

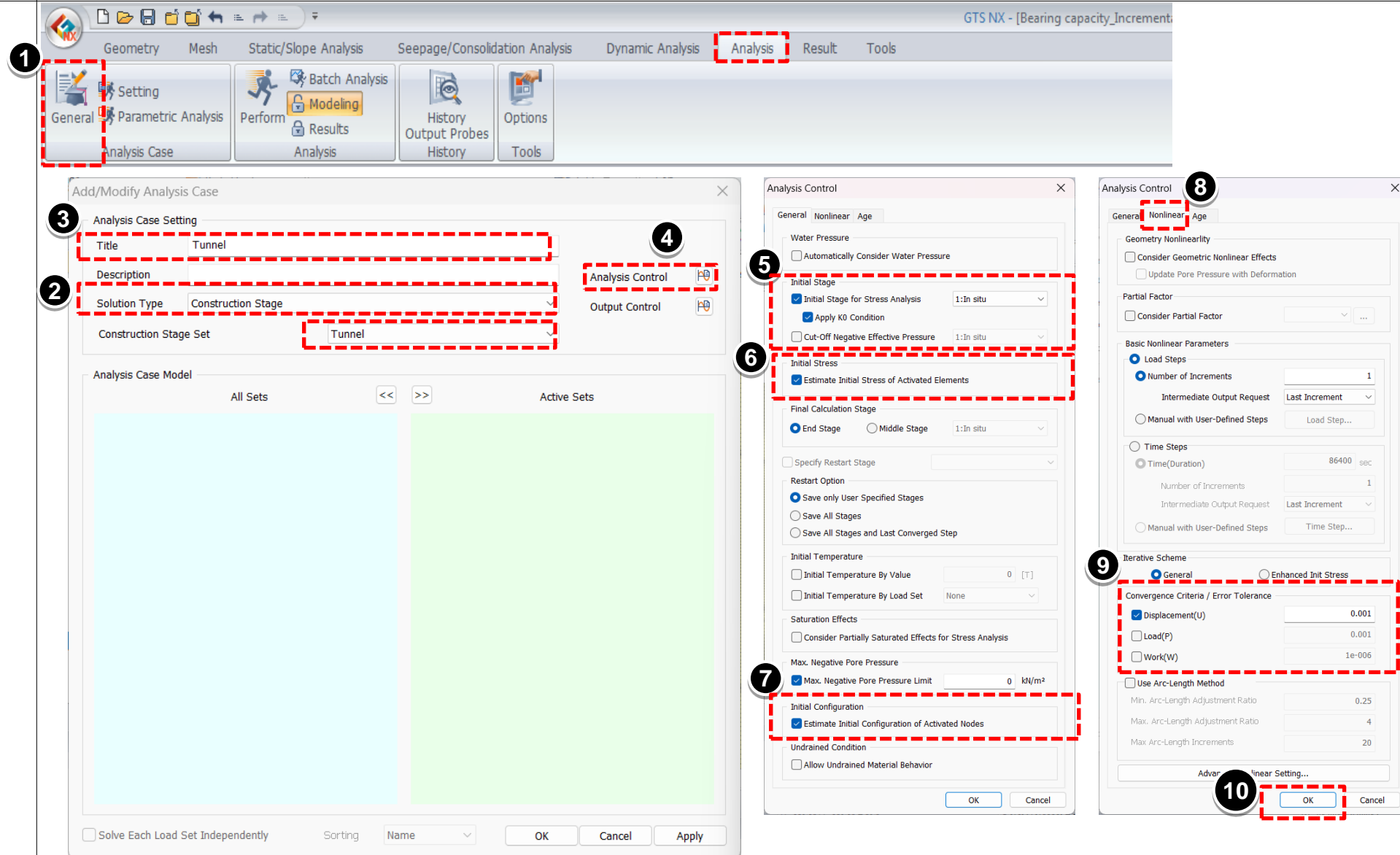
Define Water Level For Mesh Set: Input Water Level...

Sub Stage... LDF... Copy To Specific Stage...

Clear Displacement Slope Stability(SRM) Slope Stability(SAM)

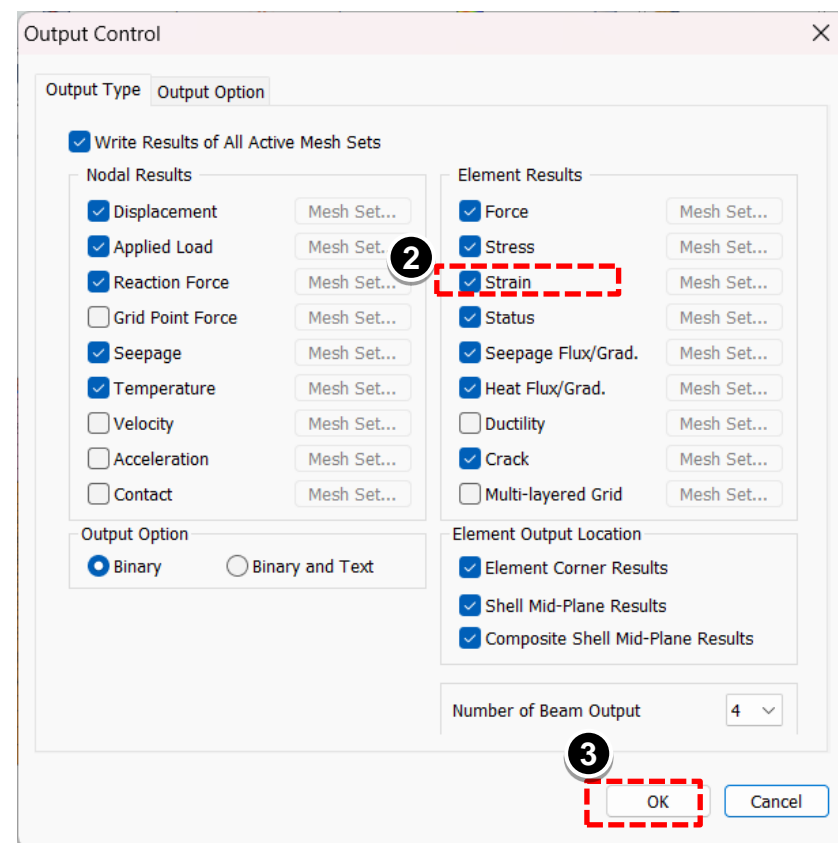
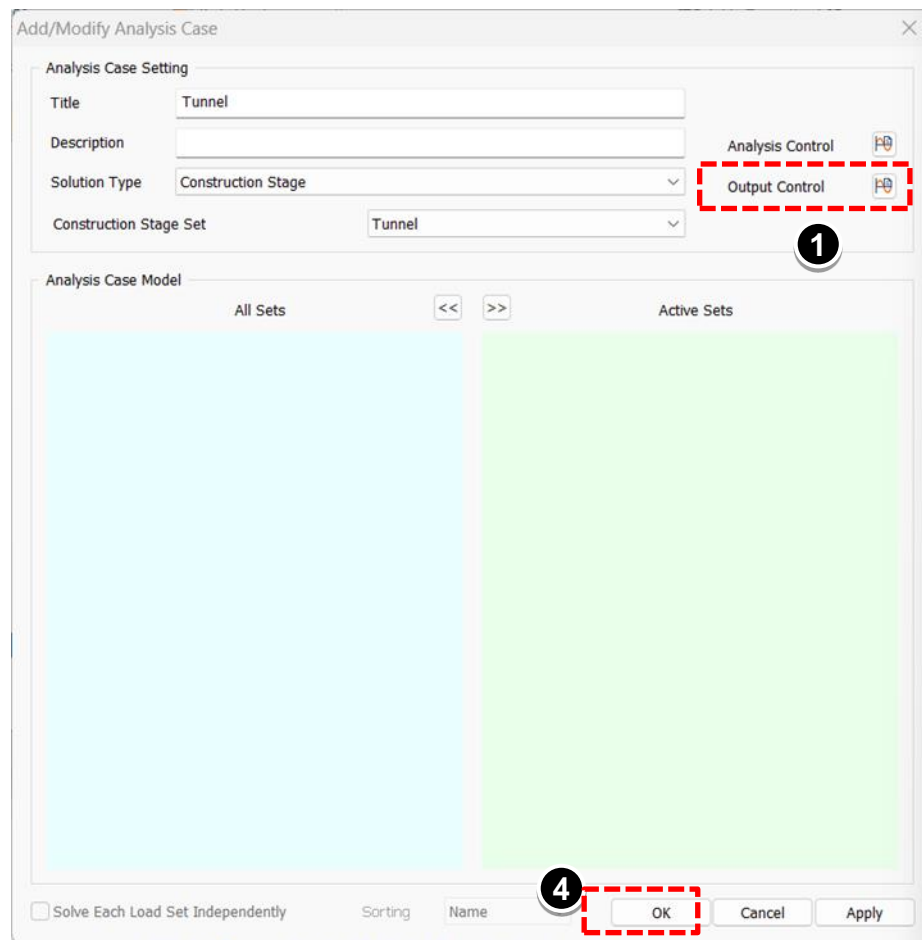
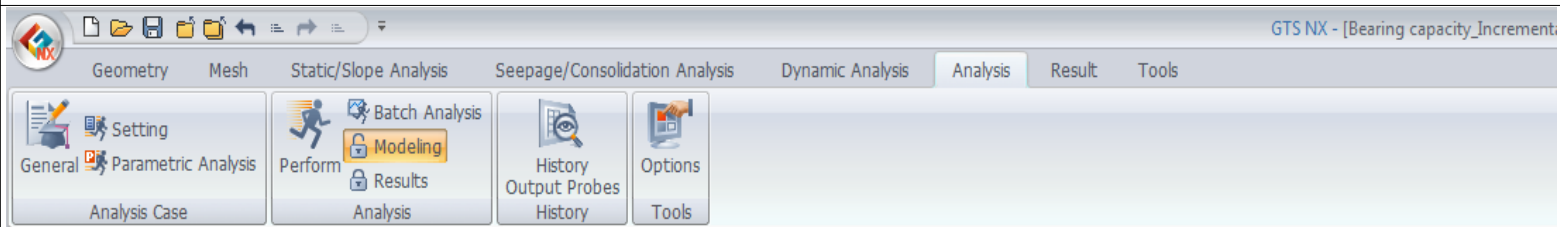
Procedure

- ❶ Go to **Analysis > General**
- ❷ Select Solution type as **Construction Stage**.
- ❸ Enter Title '**Tunnel**'
- ❹ Click on **Analysis Control**
- ❺ Check on **Initial Stage for stress analysis** and Select **In situ**. Check on **Apply K0 condition**.
- ❻ Check on **Estimate Initial Stress of Activated Elements**.
- ❼ Check on **Estimate Initial Configuration of Activated Nodes**.
- ❽ Click on **Nonlinear**.
- ❾ Change the Convergence Criteria to **Displacement**.
- ❿ Click **OK**.



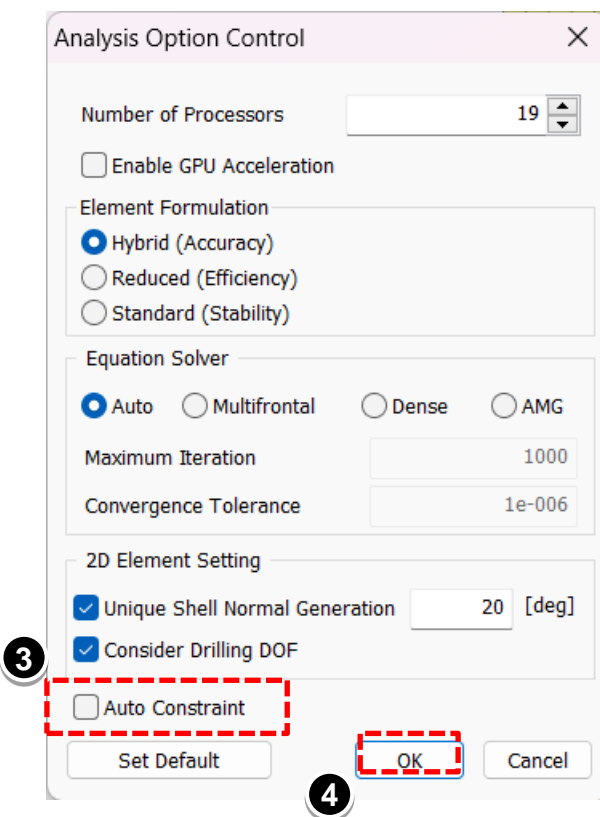
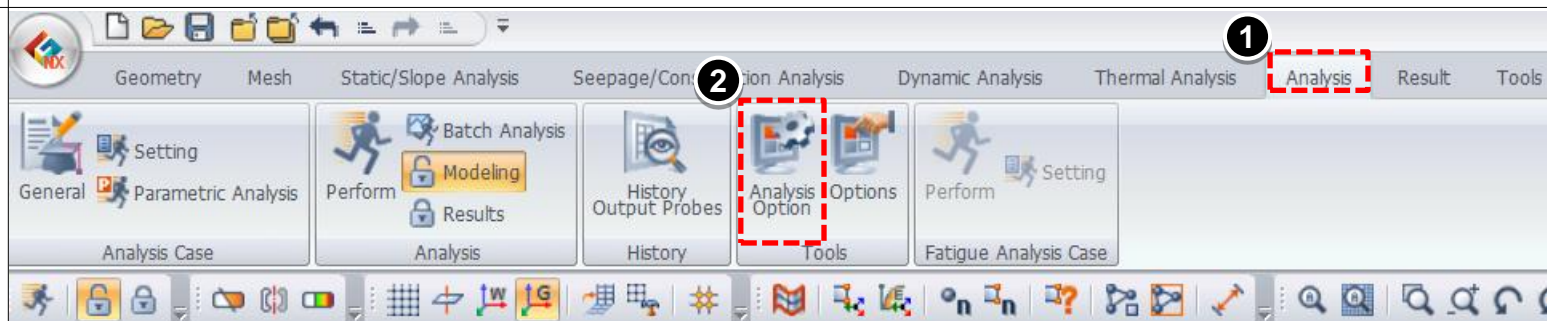
Procedure

- ① Click on **Output Control**
- ② Check on **Strain**.
- ③ Click **OK**
- ④ Click **OK**.



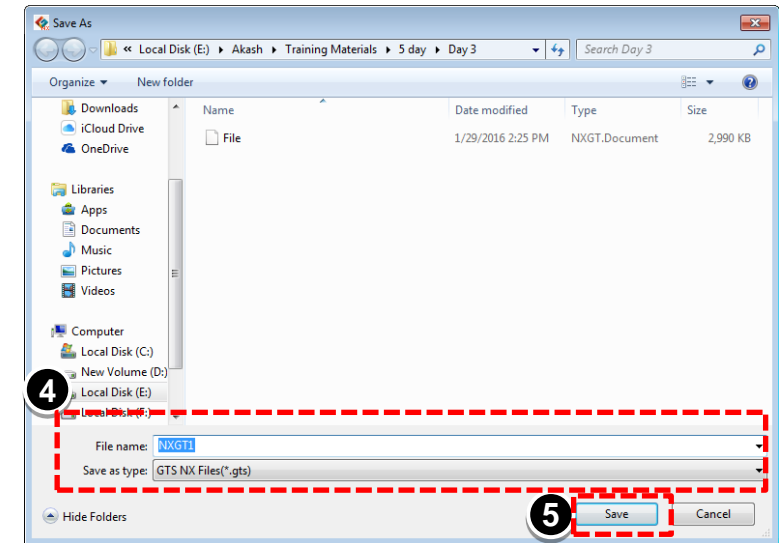
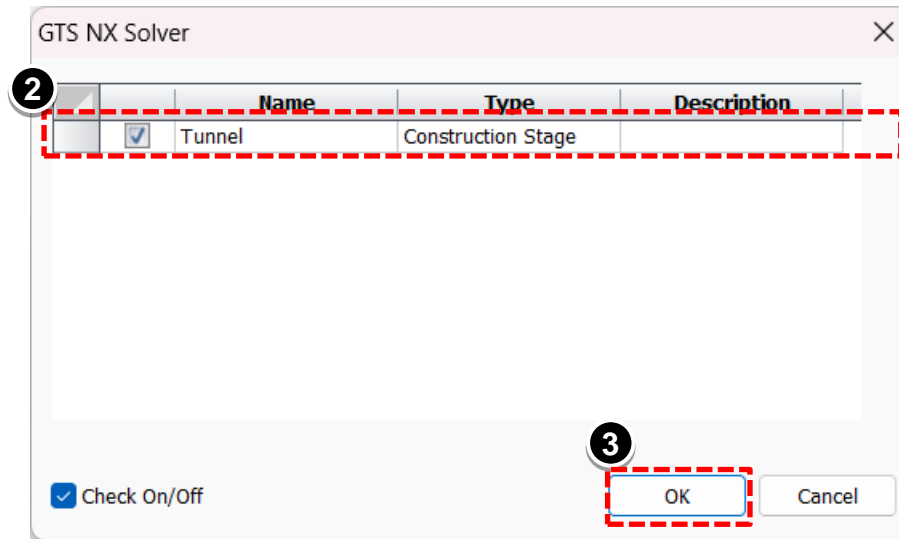
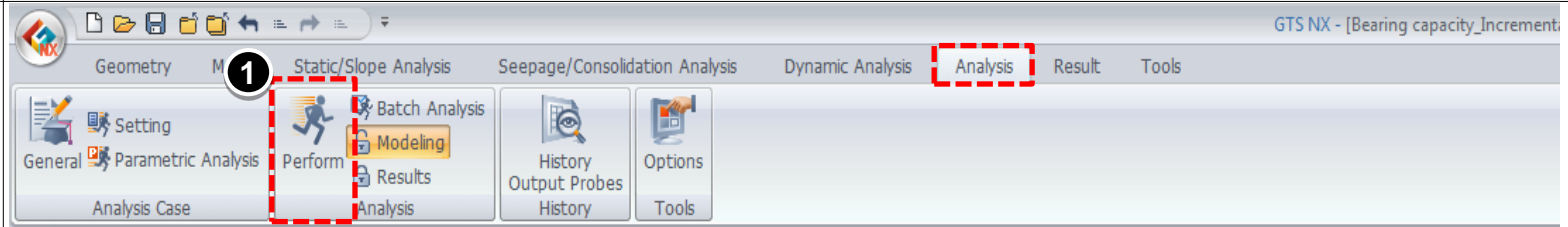
Procedure

- ❶ Go to **Analysis**
- ❷ Go to **Analysis Option**
- ❸ Uncheck **Auto Constraint**
- ❹ Click **OK**.




Procedure

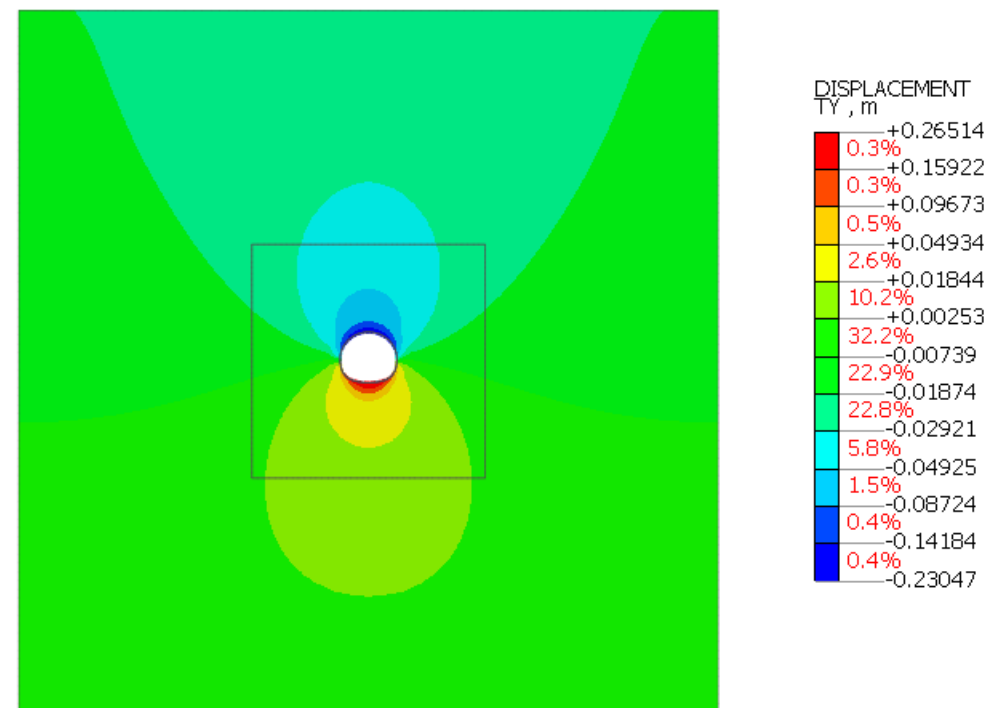
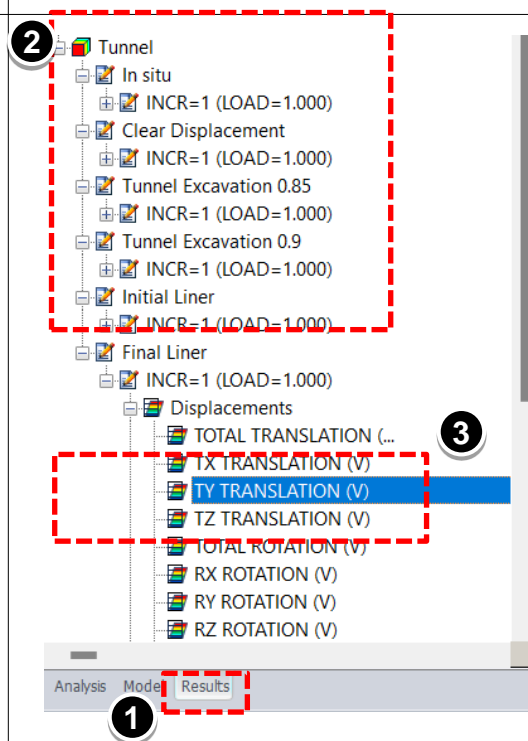
- ❶ Go to **Analysis > Perform**
- ❷ Select **Tunnel**
- ❸ Click **OK**
- ❹ Give the File name
- ❺ Save it at desired location



Procedure

- ① Go to Works Tree Window > **results**
- ② Open **Tunnel** results
- ③ Click on  icon to open up different contours. Double click on **TY Translation** to see the contour plot

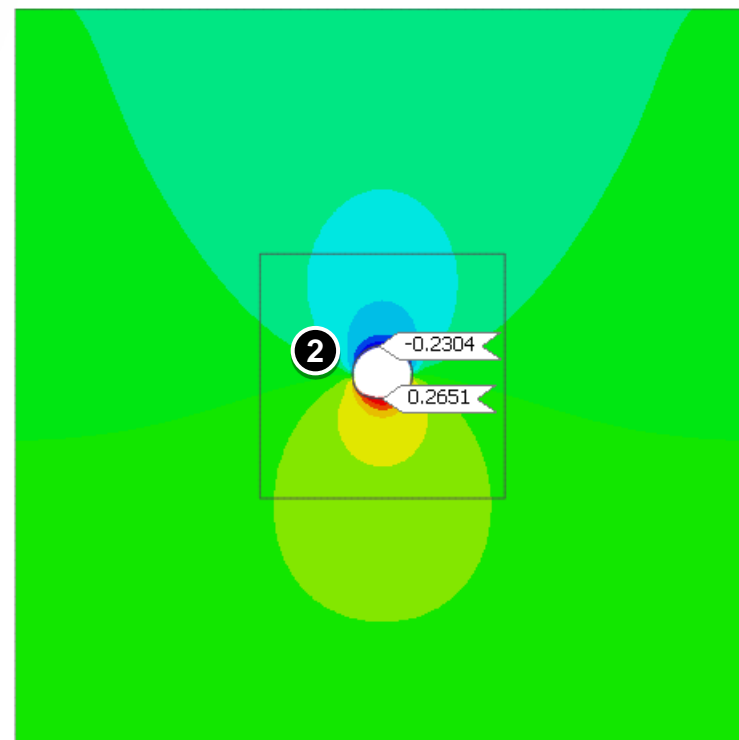
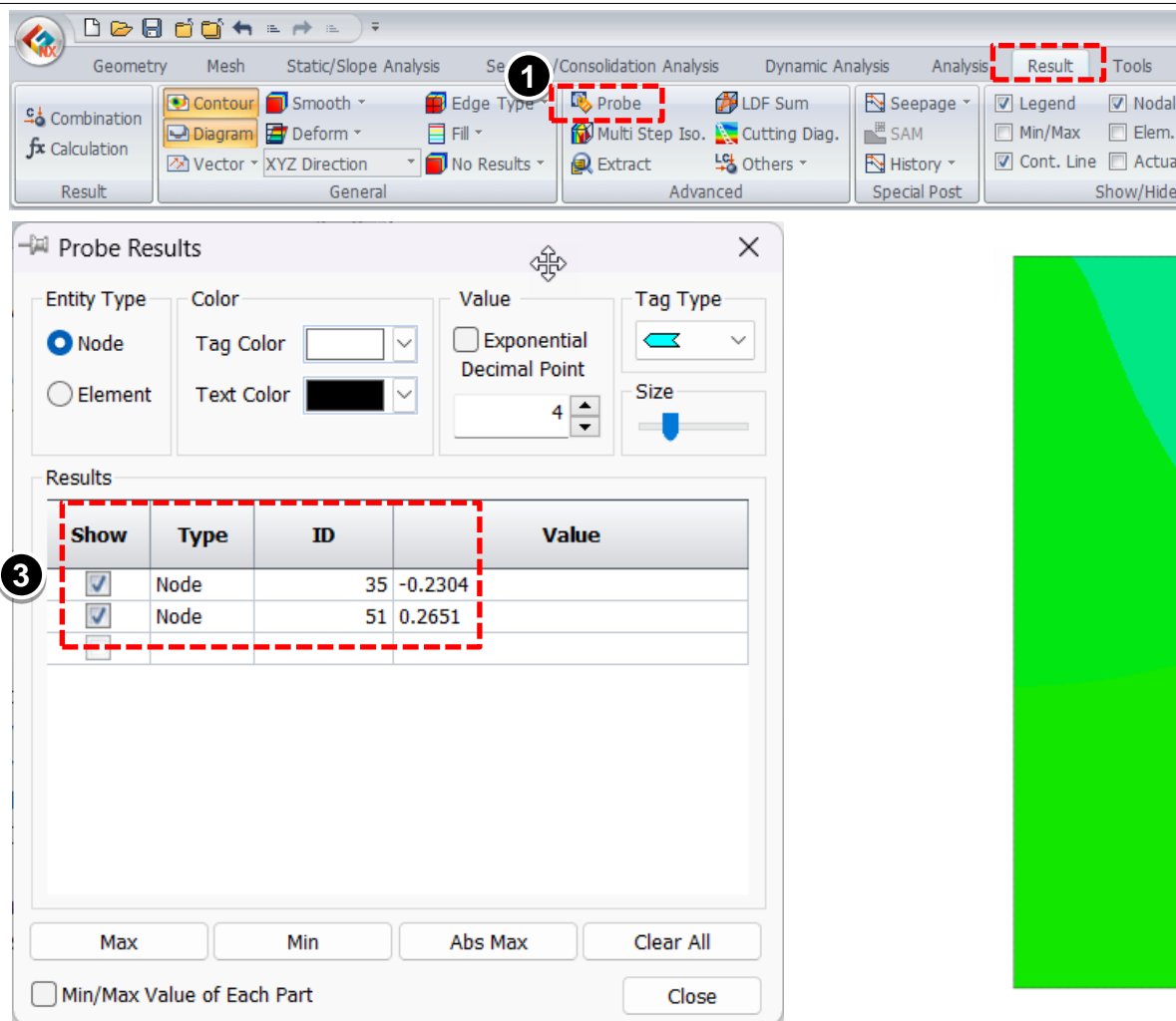
Final settlements can be checked using **TY Translation** of 'Final Liner' stage.



Settlement in the last stage

Procedure

- ❶ Go to **Results> probe**
- ❷ Click on the model on Node of interest
- ❸ One can see the probe value in the model itself or in Probe results tab.
- ❹ Close it

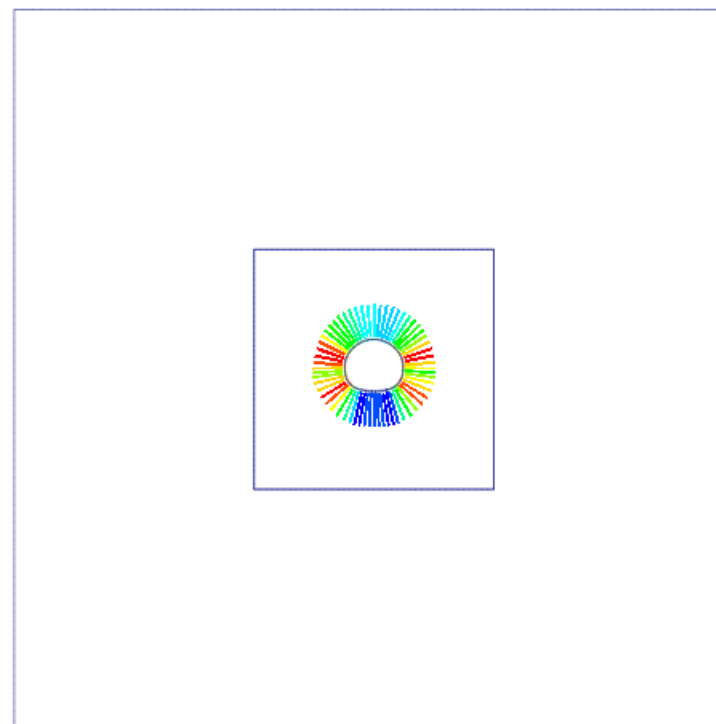
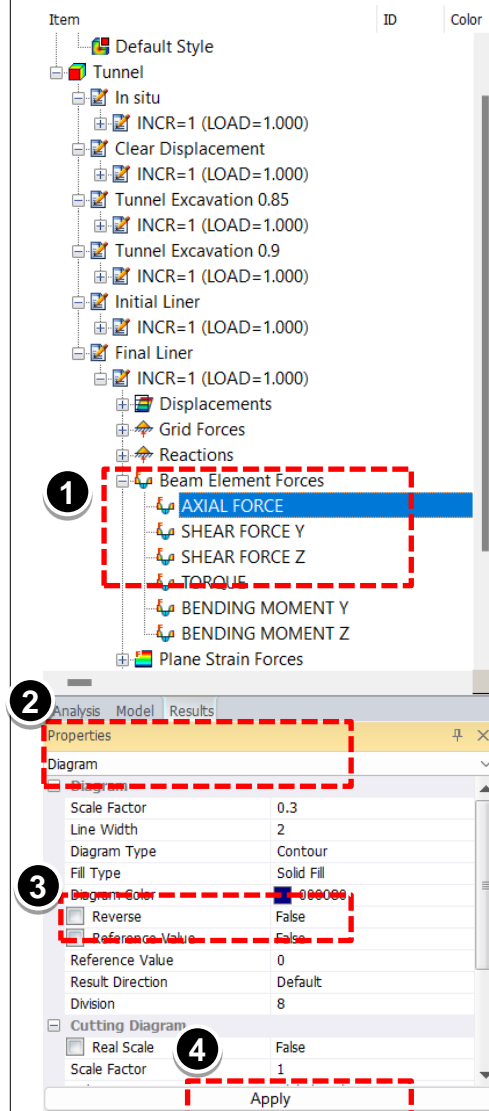


Probe Result

Procedure

- ❶ Double click **Axial Force** under 'Beam Element Forces' to check for axial force variation in Liner.
- ❷ Go to **Properties**. Select **Diagram** from the drop down.
- ❸ Uncheck **Reverse**.
- ❹ Click **Apply**.

This is used to change the view of the result output.

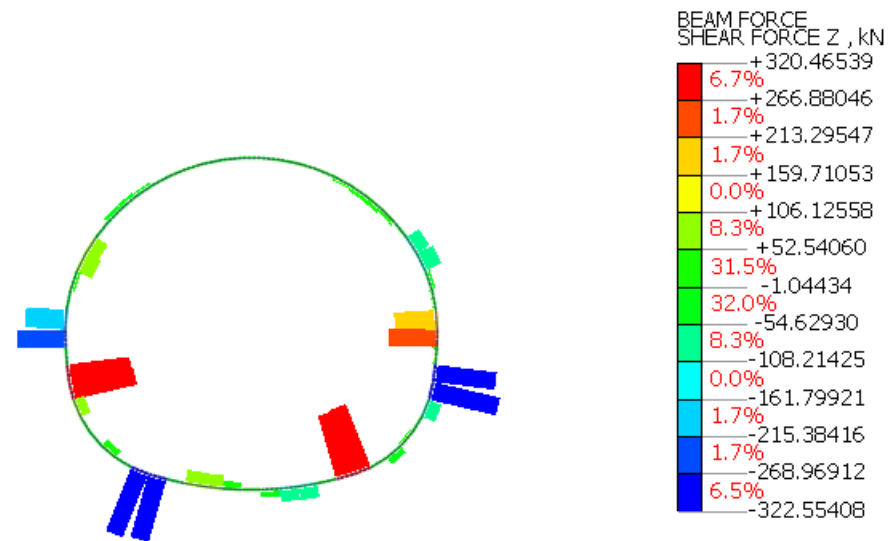
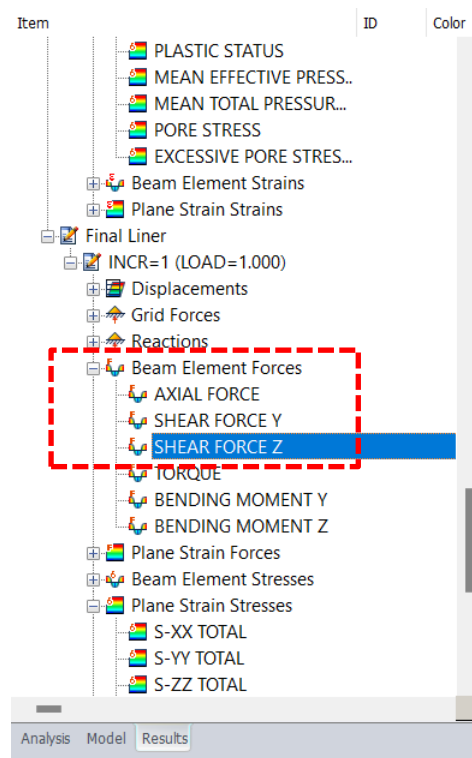


BEAM FORCE	
AXIAL FORCE , kN	
8.7%	-10991.30371
8.1%	-11155.46973
9.8%	-11229.35254
7.0%	-11464.13477
8.0%	-11541.80664
8.5%	-11675.65527
8.3%	-11987.76660
8.5%	-12243.88281
8.0%	-12368.43652
8.3%	-12415.49707
8.3%	-12458.11523
8.3%	-12544.03711
8.3%	-12624.78027

Axial Force variation in Liner

Procedure

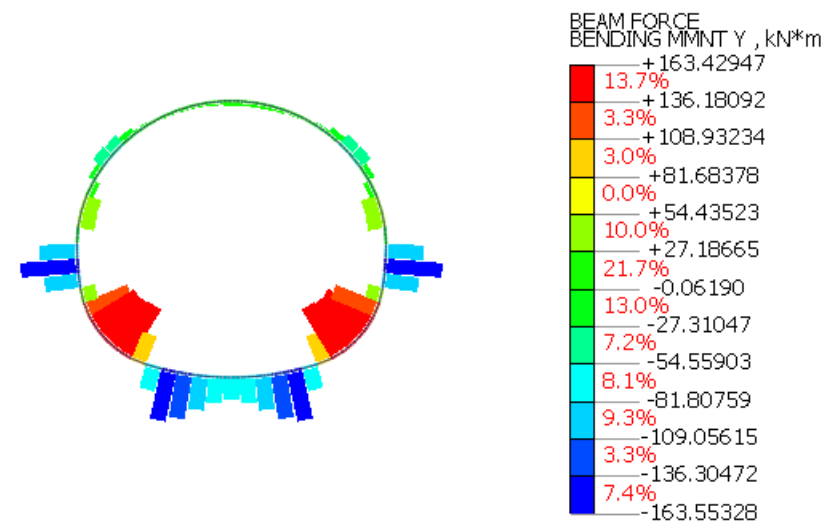
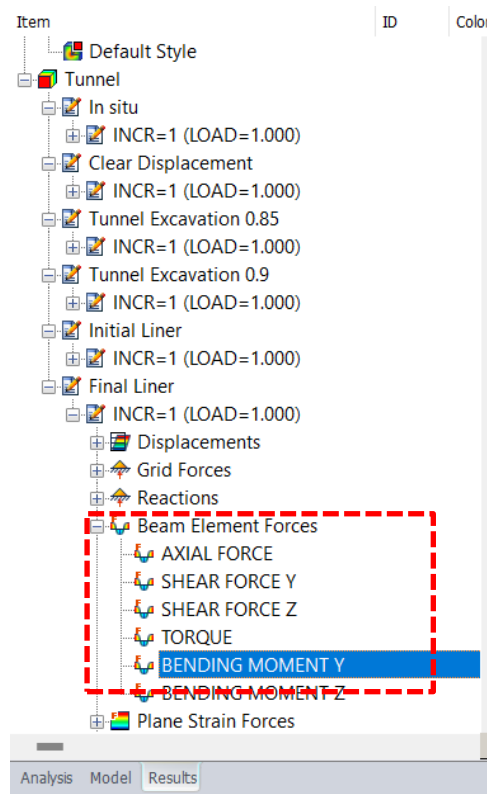
- 1 Double click **Shear Force** under '**Beam Element Forces**' to check for axial force variation in Liner.



Shear Force variation in Liner

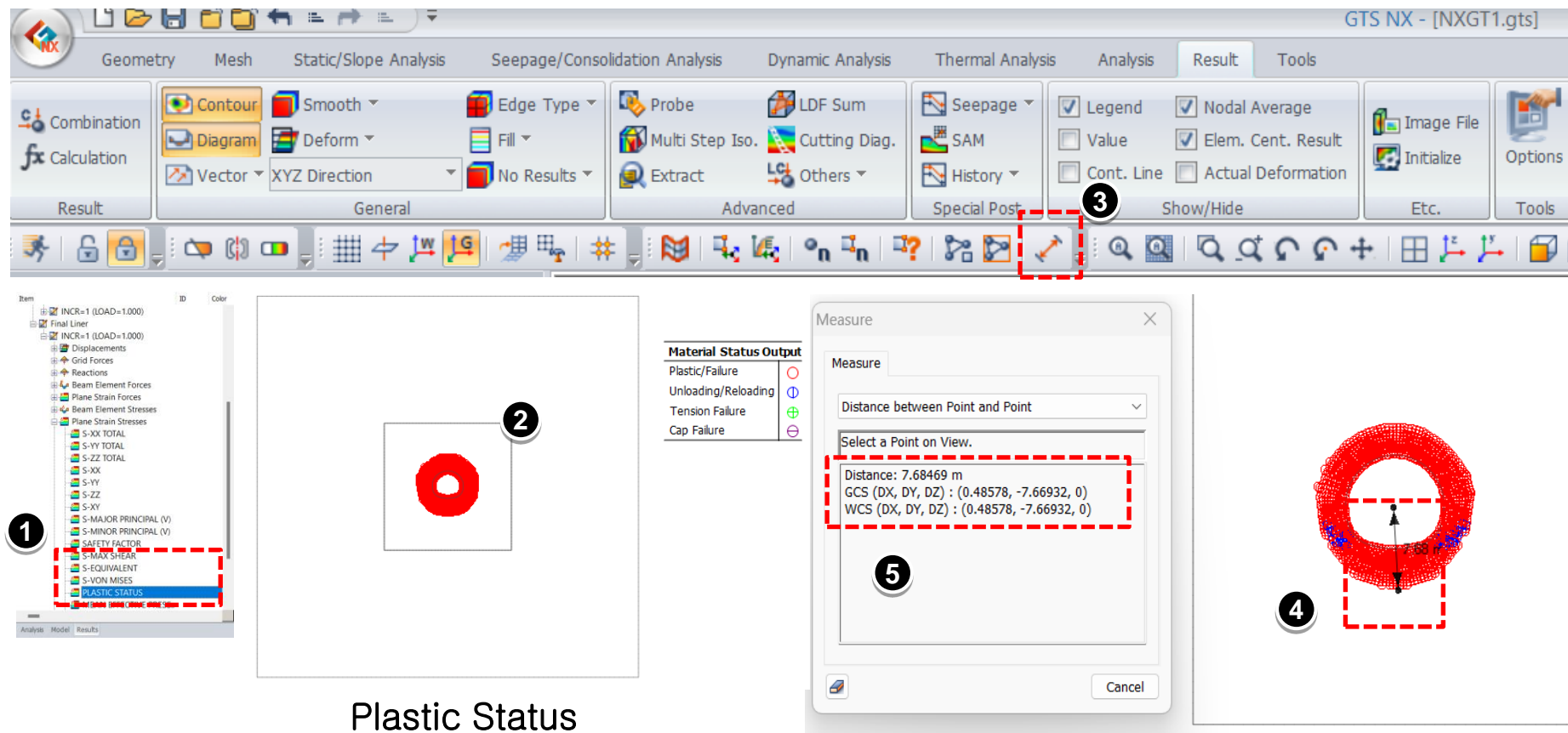
Procedure

- ① You can also check for **bending moment** in tunnel lining.



Bending Moment result for the Liner

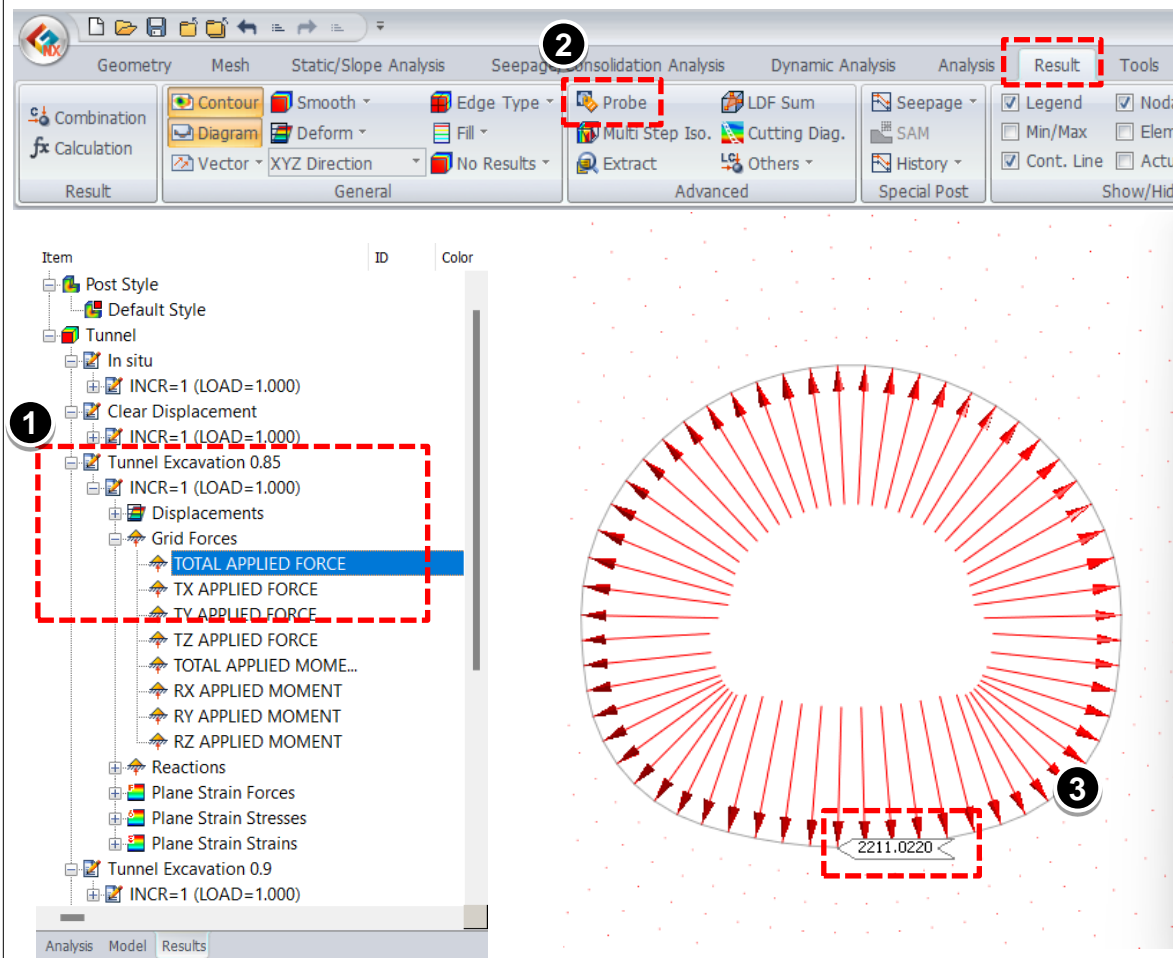
- 1 Go to **Final Liner - Plane Strain Stresses-> Plastic Status** to check for plastic region in the model.
- 2 We can observe plasticity around the tunnel section.
- 3 To check for the radius of plastic region, **Measure** can be used.
- 4 Go to **Measure** and select two points as shown.
- 5 The distance between them will be displayed.



11-1 Result Extraction

Procedure

- ① Click on **Total Applied Force** under **Grid Forces** for the stage **Tunnel Excavation 0.85**.
- ② Go to **Probe** and select any node.
- ③ We can observe that forces have been applied at the tunnel section corresponding to 15% of the stresses such that the stress relaxation is up to 85% based on the LDF input for this stage.



Probe Results

Entity Type: ☒ Node ☐ Element

Color: Tag Color Text Color

Value: Exponential ☐ Decimal Point

Tag Type: Size:

Row	Type	ID	Value
3	Node	50	2211.0220

Max Min Abs Max Clear All

☐ Min/Max Value of Each Part Close

Procedure

- 1 Click on **Total Applied Force** under **Grid Forces** for the stage **Tunnel Excavation 0.9**.

For this stage, grid forces have been applied at the tunnel section corresponding to 10% of the stresses such that allowed stress relaxation is 90% according to LDF input for this stage.

Probe can be used to check the result at the same node.

The screenshot shows the Midas Gen software interface with the following components:

- Menu Bar:** Geometry, Mesh, Static/Slope Analysis, Seepage/Consolidation Analysis, Dynamic Analysis, Analysis, Result, Tools.
- Toolbar:** Includes buttons for Contour, Smooth, Edge Type, Probe (highlighted with a red dashed box and circle 2), LDF Sum, Seepage, Legend, Nodal, etc.
- Left Panel (Tree View):** Lists various analysis items. Under 'Tunnel Excavation 0.9', 'Grid Forces' is expanded, and 'TOTAL APPLIED FORCE' is selected (highlighted with a red dashed box and circle 1).
- Central Model View:** Displays a circular tunnel cross-section with red arrows representing forces. A specific node is highlighted with a red dashed box and circle 3, showing a value of 1475.0715.
- Right Panel (Probe Results Dialog):** A table showing the results for the selected node. The table has columns: Show, Type, ID, and Value. The selected row is highlighted with a red dashed box and circle 3.

Show	Type	ID	Value
<input checked="" type="checkbox"/>	Node	50	1475.0715

[DATA] Tunnel, Tunnel Excavation 0.9, INCR=1 (LOAD=1.000), [UNIT] kN, m

Procedure

- 1 Go to **Result-> Extract**.
- 2 Select **Beam Element Forces** in **Result Type**.
- 3 Select **Axial Force** in **Results**.
- 4 Select the desired steps.
- 5 Keep the **Order** as **Node/Element**.
- 6 Select all the liner elements.
- 7 Click **Table**.
- 8 A Table with the result value, corresponding steps and coordinates of the elements is shown.
- 9 Right click and select **Export to Excel** to export the data to Excel sheet.
- 10 Save the excel file at some location.

You can also extract **Maximum** value of the required result using **Maximum**.

1

2

3

4

5

6

7

8

9

10

No	Element	X (m)	Y (m)	Z (m)	Initial Liner: INCR=1 (LOAD=1.000) AXIAL FORCE Node 1	Final Liner: INCR=1 (LOAD=1.000) AXIAL FORCE Node 1
1	12489	4.790938e+000			-1.161269e+004	-1.161269e+004
2	12490	4.837121e+000			-1.167567e+004	-1.167567e+004
3	12491	4.860241e+000			-1.151153e+004	-1.151153e+004
4	12492	4.860241e+000			-1.145876e+004	-1.145876e+004
5	12493	4.837121e+000			-1.162969e+004	-1.162969e+004
6	12494	4.790938e+000			-1.154181e+004	-1.154181e+004
7	12495	4.345936e+000			-1.121214e+004	-1.121214e+004
8	12496	3.341833e+000			-1.203020e+004	-1.203020e+004
9	12497	4.561590e+000			-1.146414e+004	-1.146414e+004
10	12498	4.707299e+000			-1.152467e+004	-1.152467e+004
11	12499	3.728244e+000			-1.156969e+004	-1.156969e+004
12	12500	4.065791e+000			-1.119815e+004	-1.119815e+004
13	12501	2.916335e+000			-1.227936e+004	-1.227936e+004
14	12502	2.916335e+000			-1.224390e+004	-1.224390e+004
15	12503	4.065791e+000			-1.115547e+004	-1.115547e+004
16	12504	-3.728244e+000			-1.152264e+004	-1.152264e+004
17	12505	-4.561590e+000			-1.135686e+004	-1.135686e+004
18	12506	-3.341833e+000			-1.198778e+004	-1.198778e+004
19	12507	-4.345936e+000			-1.14267e+004	-1.14267e+004
20	12508	-4.707299e+000			-1.143049e+004	-1.143049e+004
21	12509	4.445835e+000			-1.129335e+004	-1.129335e+004
22	12510	4.645636e+000			-1.109848e+004	-1.109848e+004
23	12511	4.780767e+000			-1.107108e+004	-1.107108e+004
24	12512	4.848915e+000			-1.122936e+004	-1.122936e+004
25	12513	-3.209943e+000			-1.209742e+004	-1.209742e+004
26	12514	2.813099e+000			-1.228593e+004	-1.228593e+004
27	12515	-9.894992e-001			-1.236845e+004	-1.236845e+004
28	12516	2.388297e+000			-1.238489e+004	-1.238489e+004

Procedure

- 1 Go to **Result-> Extract**.
- 2 Select **Beam Element Forces** in **Result Type**.
- 3 Select **Shear Force Z**.
- 4 Select the desired steps.
- 5 Keep the **Order** as **Node/Element**.
- 6 Select all the liner elements.
- 7 Click **Table**.
- 8 A Table with the result value, corresponding steps and coordinates of the elements is shown.
- 9 Right click and select **Export to Excel** to export the data to Excel sheet.
- 10 Save the excel file at some location.

You can also extract **Maximum** value of the required result using **Maximum**.

The screenshot shows the GTS NX software interface with the 'Result' tab selected. The 'Extract' button is highlighted in the 'Advanced' section. The 'Extract Results' dialog box is open, showing 'Beam Element Forces' as the Result Type and 'Shear Force Z' as the Results. The 'Order' is set to 'Node/Element'. The 'Table' button is highlighted. A circular mesh of elements is shown, and a table of results is displayed on the right. The table has columns for No, Element, X (m), Y (m), Z (m), and Shear Force Z. The 'Export to Excel' option is highlighted in the right-click context menu.

No	Element	X (m)	Y (m)	Z (m)	Initial Liner: INCR=1 (LOAD=1.000) SHEAR FORCE Z Node 1	Final Liner: INCR=1 (LOAD=1.000) SHEAR FORCE Z Node 1
1	12489	4.790938e+000	-6.374879e-001	0.000000e+000	-2.909216e+002	-1.359589e+001
2	12490	4.837121e+000			2.286765e+002	2.286765e+002
3	12491	4.860241e+000			-2.259358e+002	-2.259358e+002
4	12492	-4.860241e+000			-2.275040e+000	-2.275040e+000
5	12493	-4.837121e+000			2.952693e+002	2.952693e+002
6	12494	-4.790938e+000			-8.982144e+000	-8.982144e+000
7	12495	4.345936e+000			7.773268e+000	7.773268e+000
8	12496	3.341833e+000			-6.484481e+001	-6.484481e+001
9	12497	4.561590e+000			-3.225541e+002	-3.225541e+002
10	12498	4.707299e+000			-3.971454e+001	-3.971454e+001
11	12499	3.728244e+000	-2.692269e+000	0.000000e+000	1.585882e+001	1.585882e+001
12	12500	4.065791e+000	-2.357150e+000	0.000000e+000	-3.204654e+002	-3.204654e+002
13	12501	2.916335e+000	-3.182219e+000	0.000000e+000	-5.759955e+000	-5.759955e+000
14	12502	-2.916335e+000	-3.182219e+000	0.000000e+000	3.913860e+001	3.913860e+001
15	12503	-4.065791e+000	-2.357150e+000	0.000000e+000	5.272605e+001	5.272605e+001
16	12504	-3.728244e+000	-2.692269e+000	0.000000e+000	-5.780103e+000	-5.780103e+000
17	12505	-4.561590e+000	-1.548799e+000	0.000000e+000	4.592041e+000	4.592041e+000
18	12506	-3.341833e+000	-2.969630e+000	0.000000e+000	-3.070437e+002	-3.070437e+002
19	12507	-4.345936e+000	-1.972752e+000	0.000000e+000	-8.462311e+001	-8.462311e+001
20	12508	-4.707299e+000	-1.096017e+000	0.000000e+000	-1.924623e+001	-1.924623e+001
21	12509	4.445835e+000	2.265367e+000	0.000000e+000	1.259565e+001	1.259565e+001
22	12510	4.645636e+000	1.783005e+000	0.000000e+000	1.972141e+002	1.972141e+002
23	12511	4.780767e+000	1.278690e+000	0.000000e+000	-2.571806e+001	-2.571806e+001
24	12512	4.848915e+000	7.610524e-001	0.000000e+000	2.480014e+001	2.480014e+001
25	12513	-3.209943e+000	3.825461e+000	0.000000e+000	-2.300465e+000	-2.300465e+000
26	12514	2.813099e+000	4.126058e+000	0.000000e+000	1.772871e+001	1.772871e+001
27	12515	-9.894992e-001	4.894770e+000	0.000000e+000		
28	12516	2.388297e+000	4.385649e+000	0.000000e+000		

Procedure

- 1 Go to **Result-> Extract**.
- 2 Select **Beam Element Forces** in **Result Type**.
- 3 Select **Shear Force Z**.
- 4 Select the desired steps.
- 5 Keep the **Order** as **Node/Element**.
- 6 Select all the liner elements.
- 7 Click **Table**.
- 8 A Table with the result value, corresponding steps and coordinates of the elements is shown.
- 9 Right click and select **Export to Excel** to export the data to Excel sheet.
- 10 Save the excel file at some location.

You can also extract **Maximum** value of the required result using **Maximum**.

The screenshot illustrates the GTS NX software interface during the result extraction process. The 'Result' menu is open, and the 'Extract' option is highlighted. The 'Extract Results' dialog box is displayed, showing the 'Analysis Set' as 'Tunnel', 'Result Type' as 'Beam Element Forces', and 'Results' as 'BENDING MOMENT Y'. The 'Step: Results' section shows 'Initial Liner: INCR=1 (LOAD=1.000): BENDING M' and 'Final Liner: INCR=1 (LOAD=1.000): BENDING M'. The 'Order' is set to 'Node/Element'. The 'Object' is set to 'Element'. The 'Element Result Extraction' section shows 'User Defined' selected, 'Select Object' as '12489to12548', and 'Sort' as 'X', 'Y', 'Z', 'Ascending'. The 'Extraction Position in Element' is set to 'Node 1'. The 'Table' button is highlighted. A circular mesh of elements is shown with a color gradient. A table of results is displayed on the right, showing columns for No, Element, X, Y, Z, and BENDING MOMENT Y. A context menu is open over the table with 'Export to Excel' selected. A file explorer window is open at the bottom, showing the file name and save location.

No	Element	X (m)	Y (m)	Z (m)	Initial Liner: INCR=1 (LOAD=1.000) BENDING MOMENT Y	Final Liner: INCR=1 (LOAD=1.000) BENDING MOMENT Y
1	12489	4.790938e+000	-6.374879e-001	0.000000e+000	-1.011189e+002	-1.011189e+002
2	12490	4.837121e+000	-	-	-1.643927e+002	-1.643927e+002
3	12491	4.860241e+000	-	-	-1.090709e+002	-1.090709e+002
4	12492	-4.860241e+000	-	-	-1.079568e+002	-1.079568e+002
5	12493	-4.837121e+000	-	-	-1.600247e+002	-1.600247e+002
6	12494	-4.790938e+000	-	-	-9.317599e+001	-9.317599e+001
7	12495	4.345936e+000	-	-	-1.522515e+002	-1.522515e+002
8	12496	3.341833e+000	-	-	-1.639230e+002	-1.639230e+002
9	12497	4.561590e+000	-	-	-1.346379e+002	-1.346379e+002
10	12498	4.707299e+000	-	-	-4.221203e+001	-4.221203e+001
11	12499	3.728244e+000	-2.692269e+000	0.000000e+000	-1.563024e+002	-1.563024e+002
12	12500	4.065791e+000	-2.357150e+000	0.000000e+000	-1.506109e+002	-1.506109e+002
13	12501	2.916335e+000	-3.182219e+000	0.000000e+000	-8.561162e+001	-8.561162e+001
14	12502	-2.916335e+000	-3.182219e+000	0.000000e+000	-8.477039e+001	-8.477039e+001
15	12503	-4.065791e+000	-2.357150e+000	0.000000e+000	-1.466768e+002	-1.466768e+002
16	12504	-3.728244e+000	-2.692269e+000	0.000000e+000	-1.546403e+002	-1.546403e+002
17	12505	-4.561590e+000	-1.548799e+000	0.000000e+000	-1.332805e+002	-1.332805e+002
18	12506	-3.341833e+000	-2.969630e+000	0.000000e+000	-1.625990e+002	-1.625990e+002
19	12507	-4.345936e+000	-1.972752e+000	0.000000e+000	-1.469555e+002	-1.469555e+002
20	12508	-4.707299e+000	-1.096017e+000	0.000000e+000	-4.744640e+001	-4.744640e+001
21	12509	4.445835e+000	2.265367e+000	0.000000e+000	-2.071798e+001	-2.071798e+001
22	12510	4.645636e+000	1.783005e+000	0.000000e+000	-4.789150e+001	-4.789150e+001
23	12511	4.780767e+000	1.278690e+000	0.000000e+000	-4.963137e+001	-4.963137e+001
24	12512	4.848915e+000	7.610524e-001	0.000000e+000	-5.257473e+001	-5.257473e+001
25	12513	-3.209943e+000	3.825461e+000	0.000000e+000	-1.517255e+001	-1.517255e+001
26	12514	2.813099e+000	4.126050e+000	0.000000e+000	-9.548176e+000	-9.548176e+000
27	12515	-9.894952e-001	4.894770e+000	0.000000e+000	-5.465389e+000	-5.465389e+000
28	12516	2.388297e+000	4.385649e+000	0.000000e+000	-1.051267e+000	-1.051267e+000