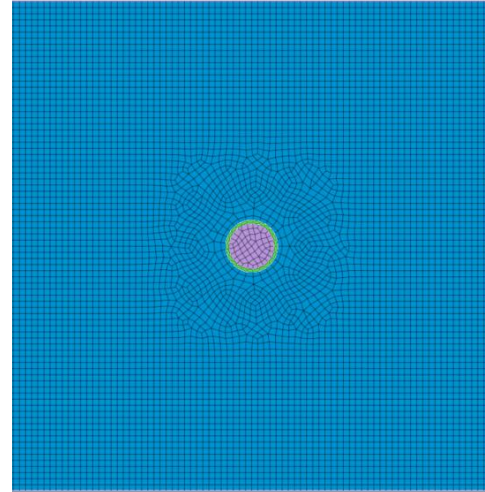


MIDAS *Technical
Material*

Tutorial



Support Capacity Plots

OVERVIEW

This tutorial provides a comprehensive guide for creating Support Capacity Plots for Tunnel Lining in MIDAS GTS-NX. These plots are essential tools in Tunnel Engineering, used to assess the performance and stability of tunnel under various loading conditions.

The tutorial covers the step-by-step procedure for generating support capacity plots, enabling users to effectively analyze and design tunnel linings.

OBJECTIVE

- Analyze the structural forces on each part of the tunnel lining and create support capacity interaction diagrams.
- Equivalent Section approach based on Carranza-Torres & Diederich is used to calculate Support Capacity (i.e. Moment Capacity, Thrust Capacity and Shear Capacity) for Tunnel Lining in this tutorial.
- Provide a detailed, procedural approach for constructing support capacity plots.

Proposed Tunnel Support Section Properties

Steel Properties

	Material Property	Section Property
Modulus of Elasticity (kN/m ²)	2.1e8	-
Compressive Strength (kN/m ²)	5e5	-
Tensile Strength (kN/m ²)	5e5	-
Poisson's Ratio	0.25	-
Steel Set Spacing (m)	-	0.6
Steel Set Height (m)	-	0.162
Area of steel (m ²)	-	0.00475
Moment of Inertia (m ⁴)	-	2.23e-5

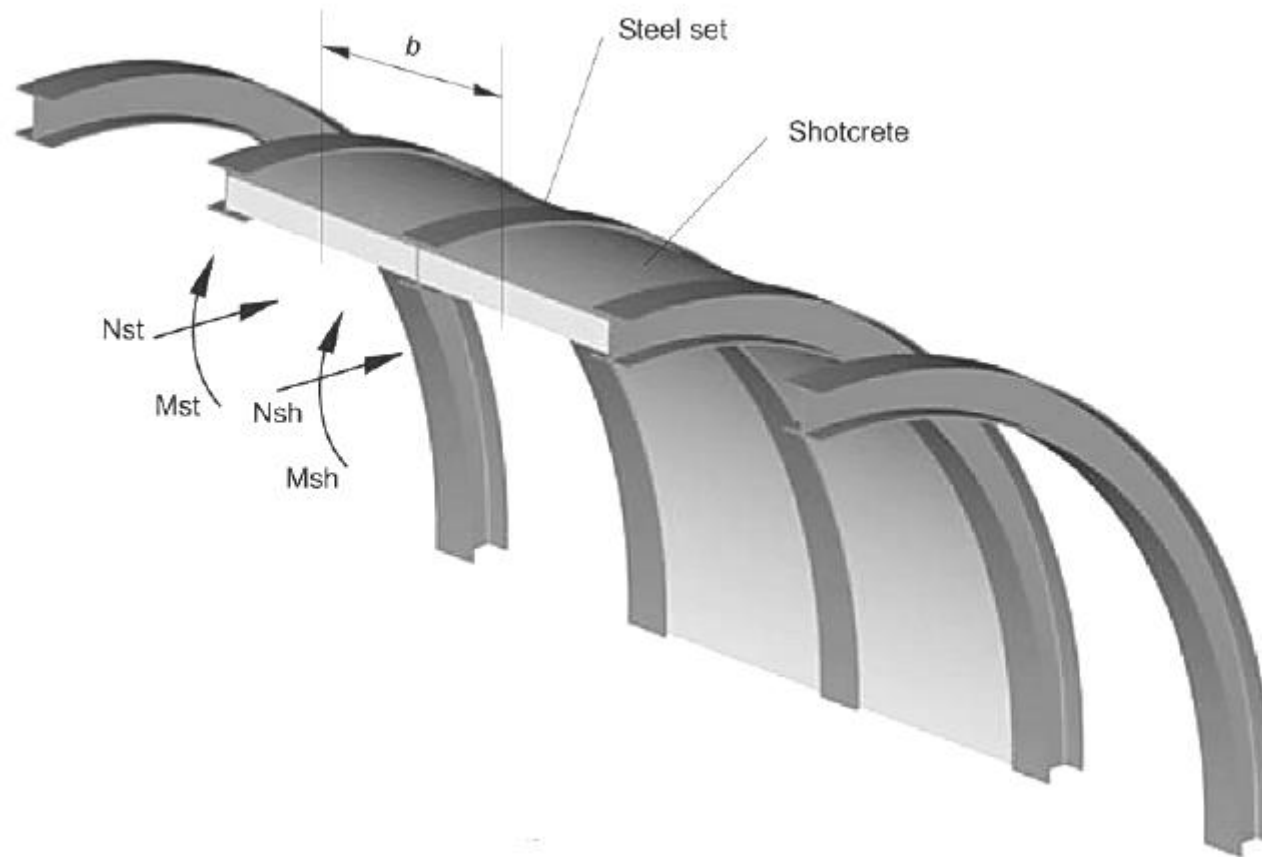
Shotcrete Properties

	Material Property	Section Property
Modulus of Elasticity (kN/m ²)	3e7	-
Compressive Strength (kN/m ²)	4e4	-
Tensile Strength (kN/m ²)	2.5e3	-
Poisson's Ratio	0.15	-
Shotcrete Thickness (m)	-	0.2
Area of Shotcrete (m ²)	-	0.12
Moment of Inertia (m ⁴)	-	0.0004

General Parameter

Radius of Tunnel (m)	2
Total Width of Lining (m)	0.6

Tunnel Support Section



A section of width b in a composite lining consisting of steel sets, spaced at a distance s , embedded in shotcrete. Moments M_{st} and axial thrusts N_{st} are induced in the steel sets and moments M_{sh} and thrusts N_{sh} are induced in the shotcrete shell.

Equivalent Section Properties

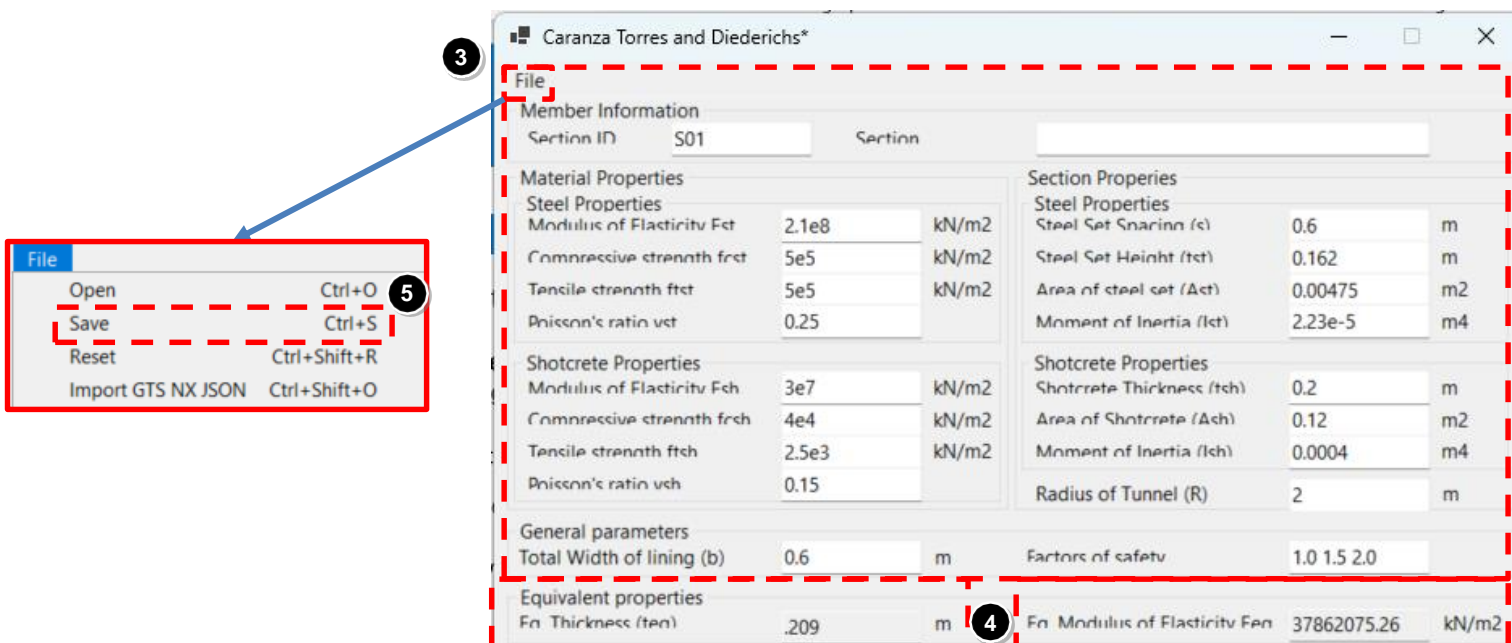
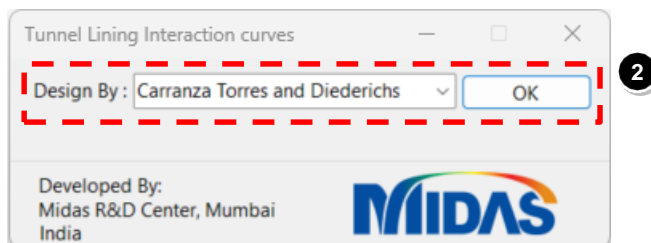
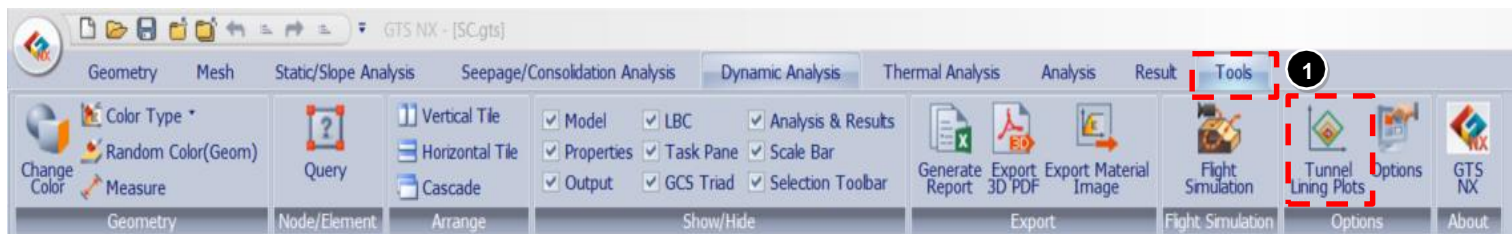
Procedure

Equivalent Section properties are required prior to generating curves.

1. Go to Tools > Tunnel Lining Plots
2. Select Carranza Torres and Diederichs > Click OK
3. Enter the Material and Section Properties of the proposed tunnel support.
4. Note down the Equivalent Section Properties (i.e. Thickness and Modulus of Elasticity)
5. Save the file >Json. We will use this in later part for generating curves.

Here,

1. Equivalent thickness = 0.209m
2. Eq. Elastic Modulus (Conc) = 37862075.26 kN/m²



Contents

- **Step 1:** Initial Setting
- **Step 2:** Geometric Modelling
- **Step 3:** Material Definition
- **Step 4:** Property Definition
- **Step 5:** Meshing
- **Step 6:** Boundary Definition
- **Step 7:** Load Definition
- **Step 8:** Construction Stage Definition
- **Step 9:** Analysis Case
- **Step 10:** Analysis
- **Step 11:** Results

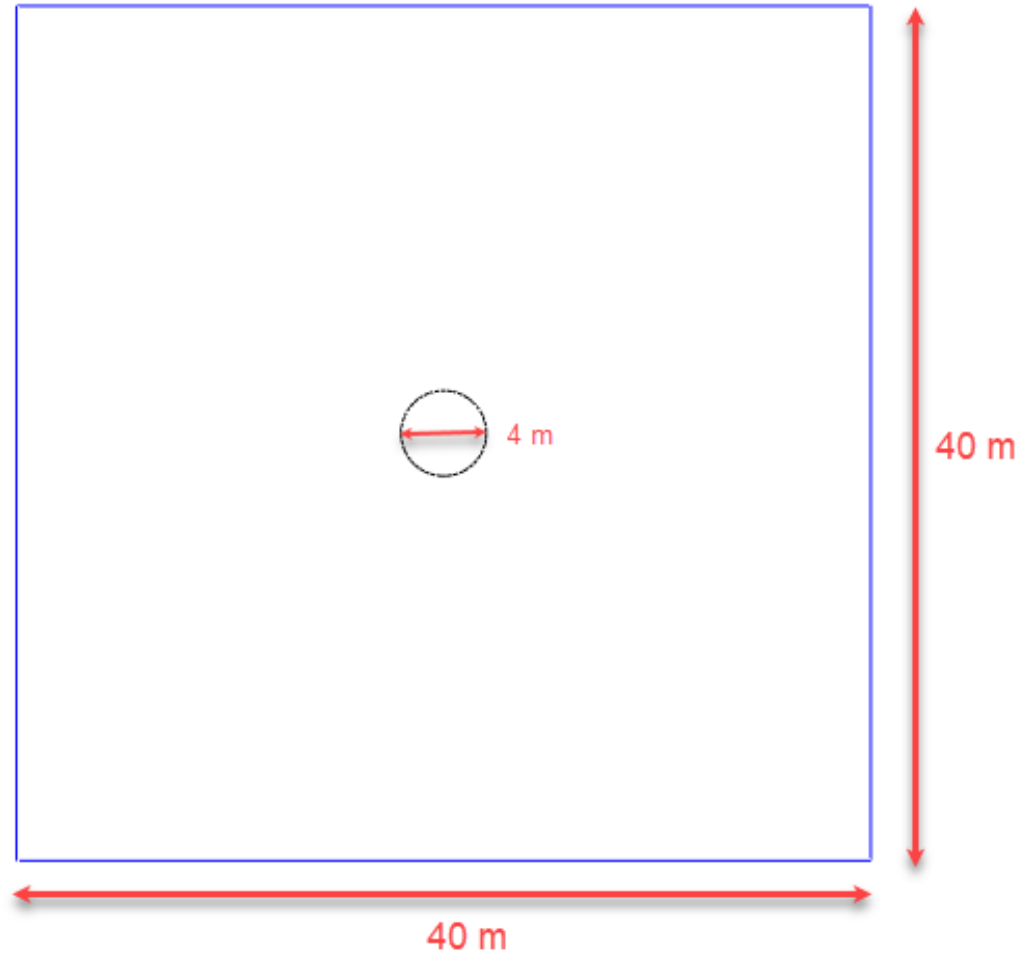
Support Capacity Curves

Material Properties

Name	Rock	Equivalent Property (Tunnel Support)
Material	Isotropic	Isotropic
Model Type	Mohr-Coulomb	Elastic
Elastic Modulus (kN/m ²)	4000000	37862075
Poisson's Ratio (v)	0.3	0.3
Unit Weight (kN/m ³)	27	25
Ko	Auto	-
Saturated Unit weight (kN/m ³)	28	-
Cohesion (kN/m ²)	1000	-
Phi (deg)	35	-

Overview – Front View

A 2m radius tunnel is to be excavated in rock mass. Tunnel support with a shotcrete of 0.2m thick and a lattice girder with 0.6m spacing is proposed.

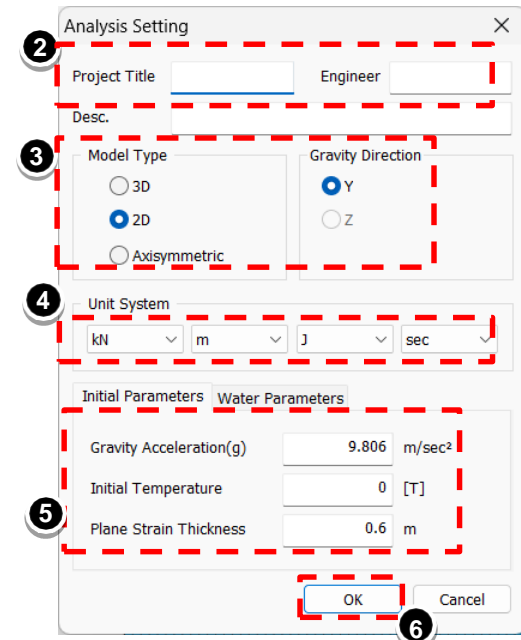
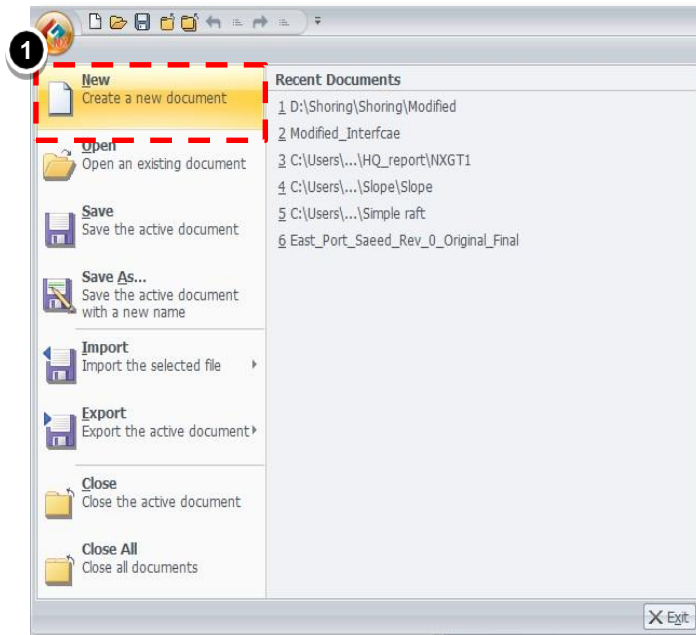


Procedure

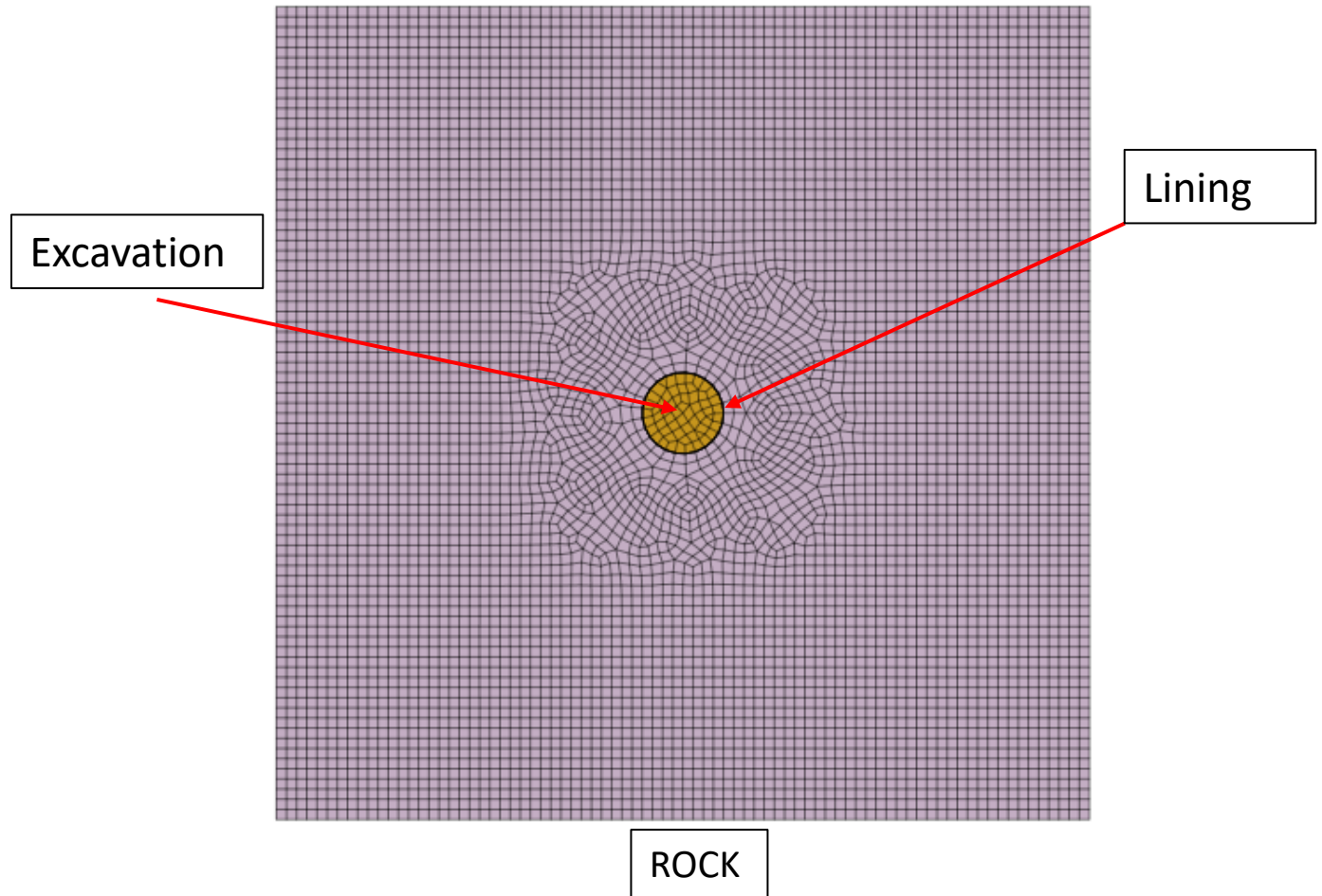
Starting Midas GTS NX

- 1 Click on GTS NX icon > New Project
- 2 Enter the Project Title
- 3 Select Model Type as **2D** and Gravity Direction as **Y**
- 4 Select **KN, m and sec** in The Unit System
- 5 Consider Plane Strain Thickness as **0.6m**
- 6 Click **OK**

Here, Plane Strain Thickness of 0.6m will be the spacing between lattice girder. (Since we have used this in calculating our equivalent section properties)



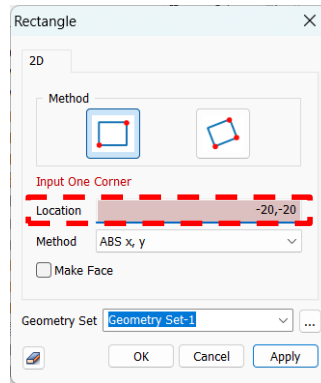
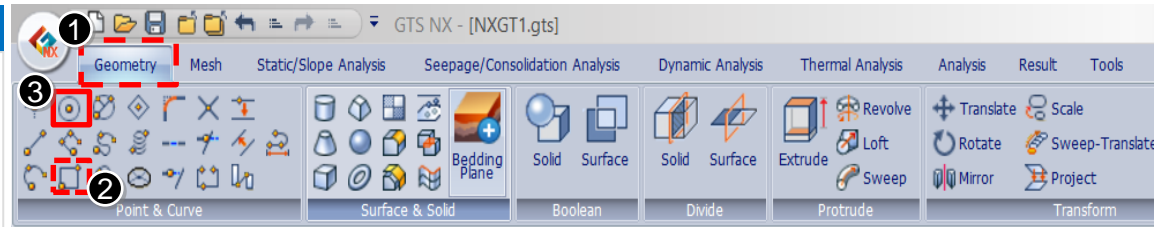
Procedure



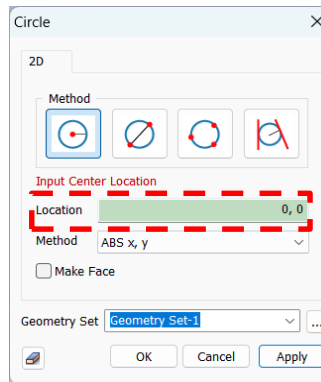
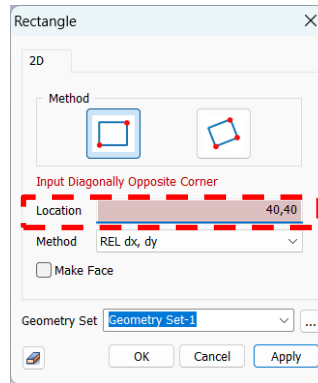
2-1 Geometric Modelling

Procedure

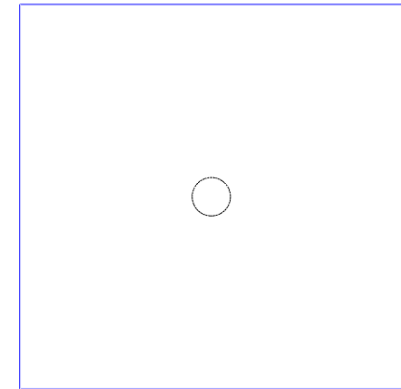
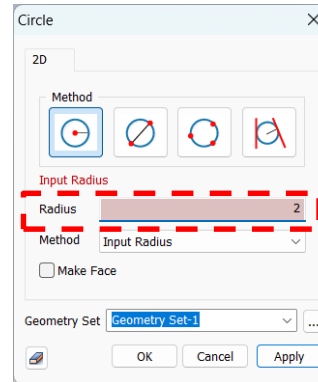
- ① Go to Geometry > Rectangle
- ② Enter Location **-20,-20** and enter Diagonally Opposite Corner location **40,40**. Click on **Apply**
- ③ Select **Circle**. Enter first location as **0,0** and then enter radius as **2m**. Click on **Apply**.



②

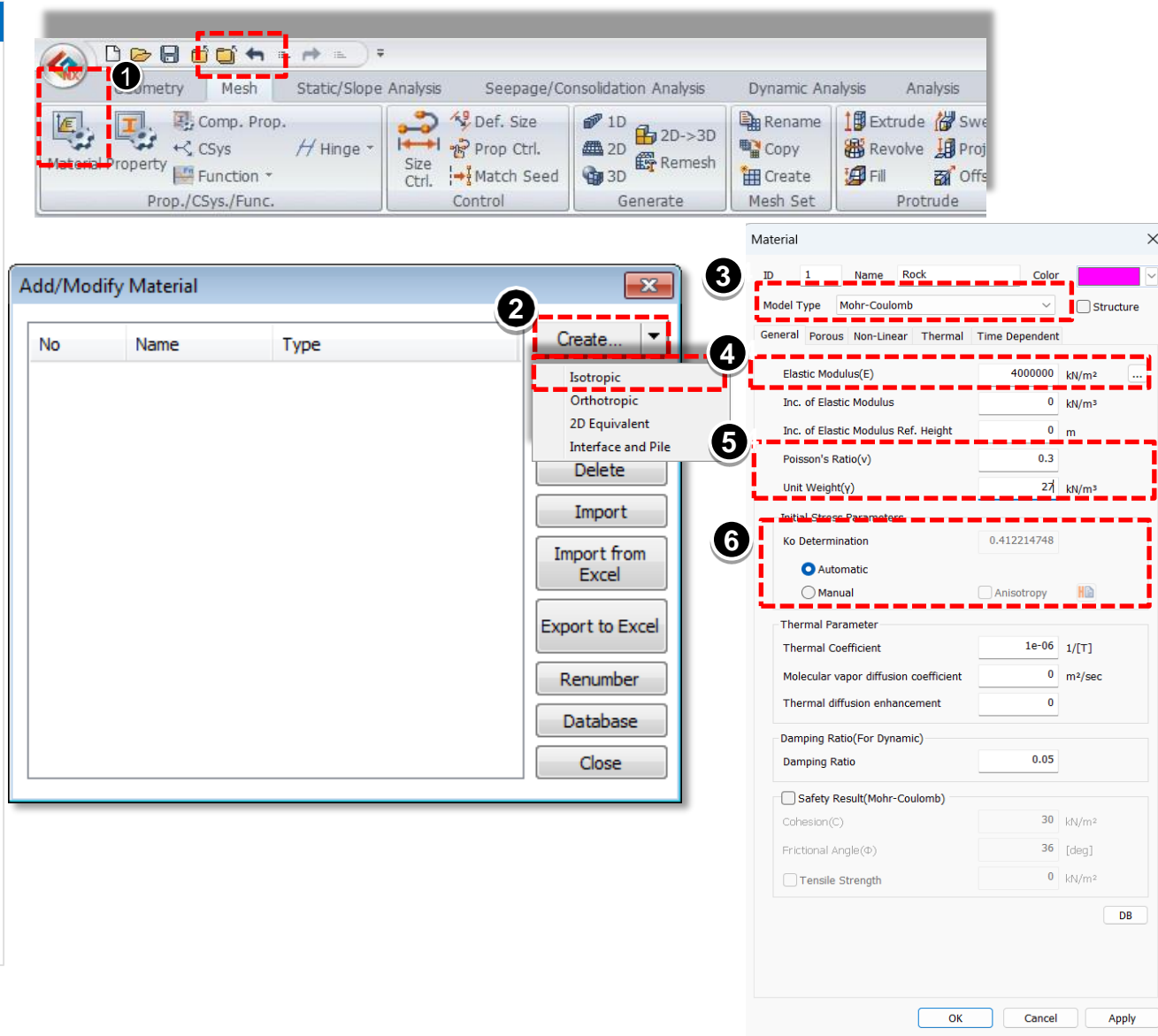


③



Procedure

- ❶ Go to Mesh > Material
- ❷ Click on Create dropdown. Select **Isotropic**
- ❸ Name it as **Rock** and Select Model type as **Mohr- Coulomb**
- ❹ In General tab, Enter the value of **Elastic Modulus (E) = 4000000 kN/m²**
- ❺ Enter **Poisson's ratio= 0.3**,
Unit weight = 27 kN/m³.
- ❻ Set **K0** to Auto



3-2 Material Definition

Procedure

- 1 Go to Porous Tab > Enter **Unit Weight (saturated)=21 kN/m³**
- 2 Keep Drainage Parameters as **Drained**.
- 3 Enter **kx= ky= kz= 1e-05 m/sec**
- 4 Go to Non Linear Tab > Enter the value of **Cohesion = 1000 kN/m²**
- 5 Input the value of **Frictional angle =35°**
- 6 Enter the name as **Rock**
- 7 Click **OK**

Material dialog box, Porous tab. The dialog shows ID 1, Name Rock, and Model Type Mohr-Coulomb. The Porous tab is active. Red dashed boxes and numbers 1-3 highlight the input fields:

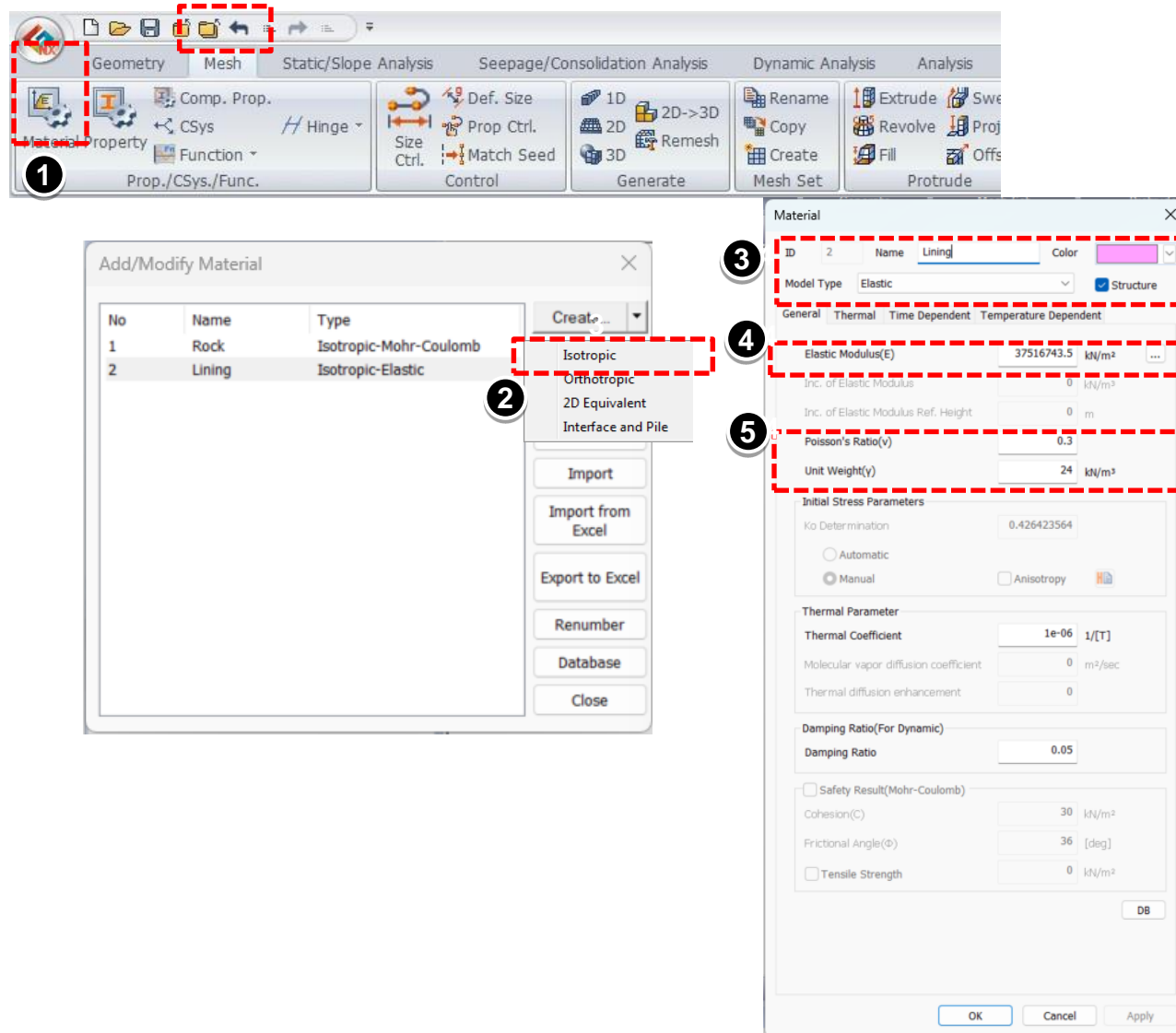
- 1. Unit Weight (Saturated) = 21 kN/m³, Initial Void Ratio (eo) = 0.5.
- 2. Drainage Parameters: Drained.
- 3. Seepage & Consolidation Parameters: Permeability Coefficients (kx, ky, kz) all set to 1e-005 m/sec.

Material dialog box, Non-Linear tab. The dialog shows ID 1, Name Rock, and Model Type Mohr-Coulomb. The Non-Linear tab is active. Red dashed boxes and numbers 4-7 highlight the input fields:

- 4. Cohesion (C) = 1000 kN/m².
- 5. Frictional Angle (Φ) = 35 [deg].
- 6. Name field (Rock).
- 7. OK button.

Procedure

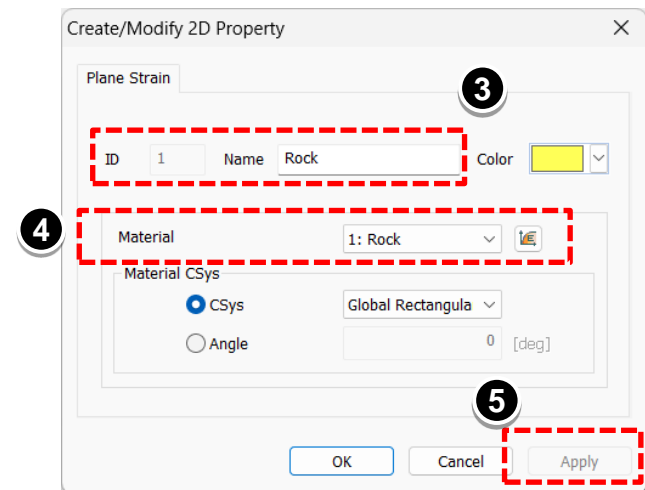
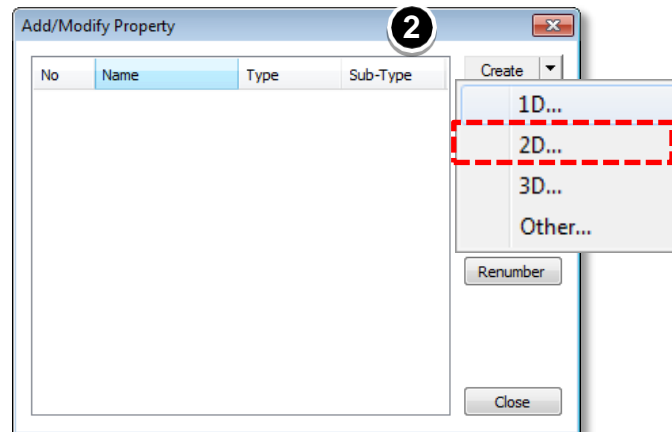
- 1 Go to Mesh > Material
- 2 Click on Create dropdown. Select **Isotropic**
- 3 Name it as **Lining** and Select Model type as **Elastic**
- 4 In General tab, Enter the value of **Elastic Modulus (E) = 37862075.26 kN/m²**
(Equivalent properties derived at starting of tutorial)
- 5 Enter **Poisson's ratio= 0.3**,
Unit weight = 24 kN/m³.



4-1 Property Definition

Procedure

- ❶ Go to Mesh > Click on Property
- ❷ Click on Create. Select 2D
- ❸ Name it as **Rock**
- ❹ Select **Rock** from Material drop down
- ❺ Click **Apply**



4-2 Property Definition

Procedure

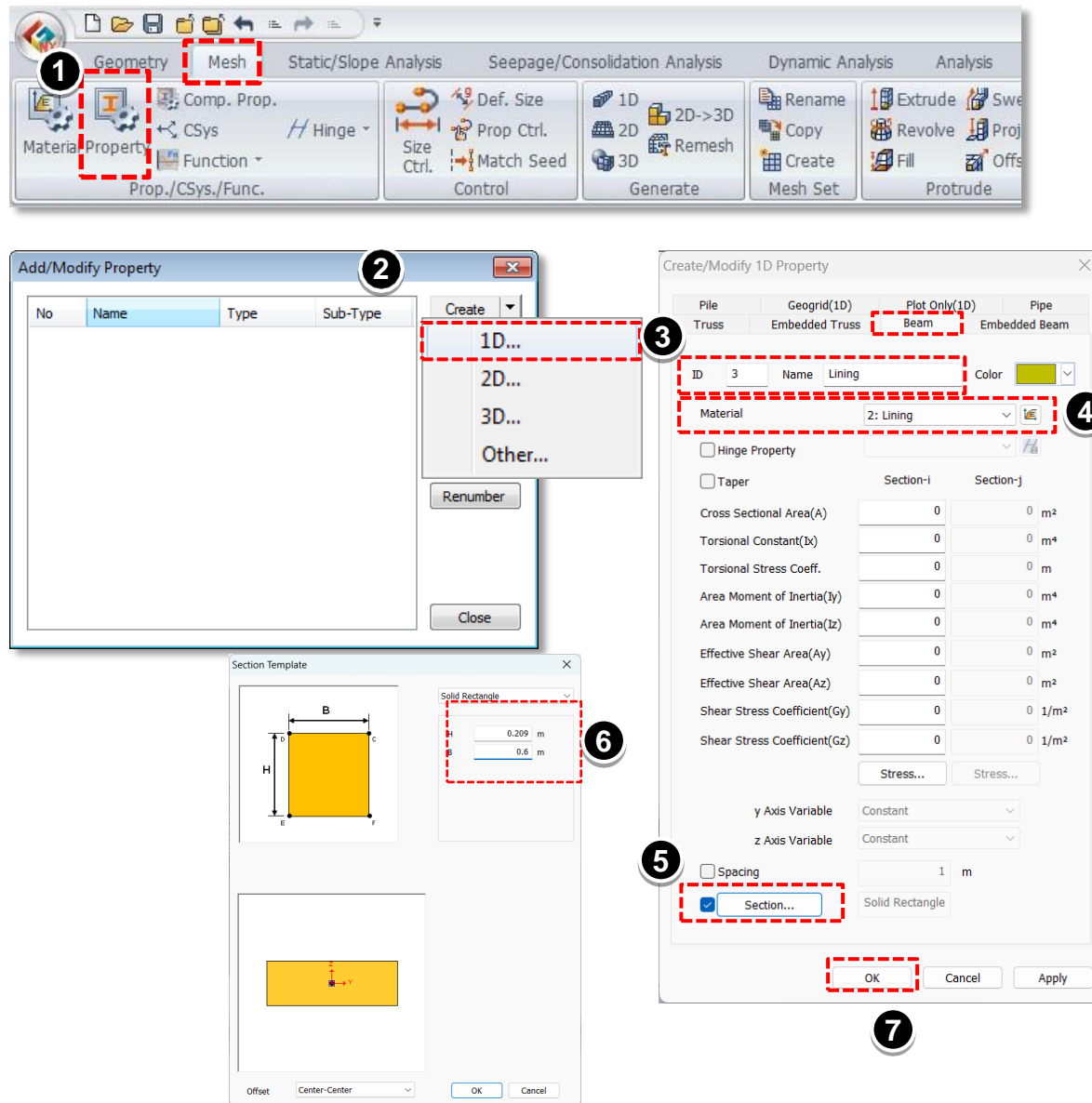
- ❶ Go to Mesh > Property
- ❷ Click on Create > 1D > Beam
- ❸ Name it as **Lining**
- ❹ Select **Lining** from Material drop down.
- ❺ Click on Section at the bottom
- ❻ Select Solid Rectangle > **H = 0.209 m** and **B = 0.6 m**.
- ❼ Click OK.

Here,

H represents the thickness of an equivalent section

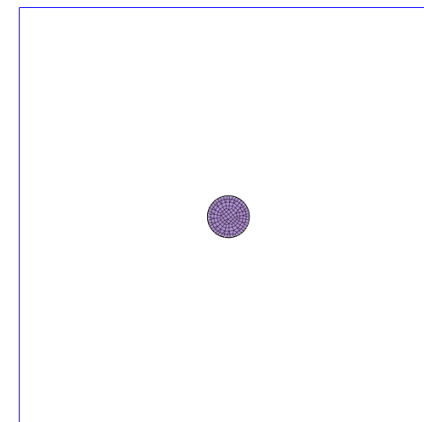
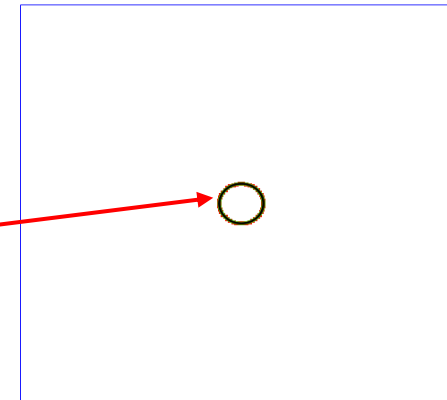
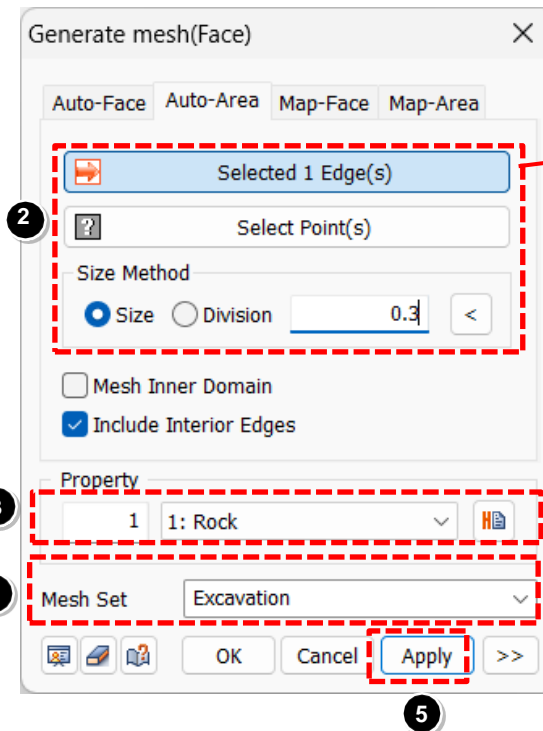
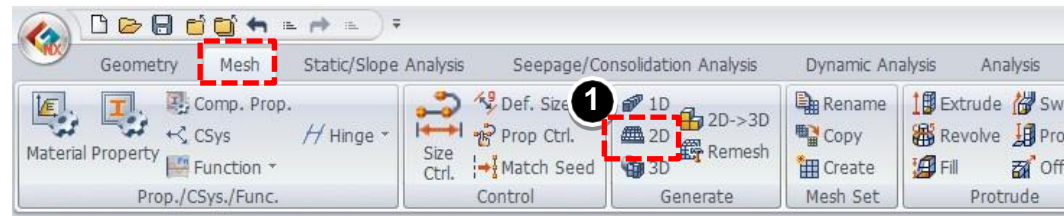
&

B represents Width of lining equal to the plain strain element width.



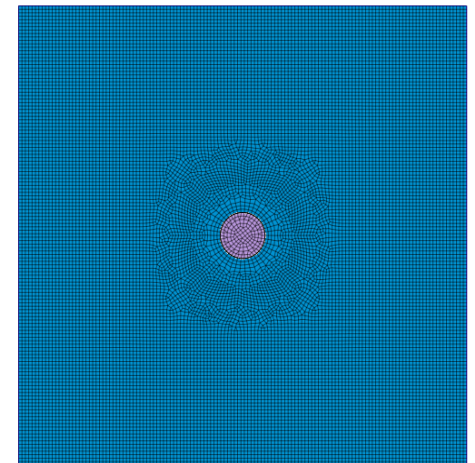
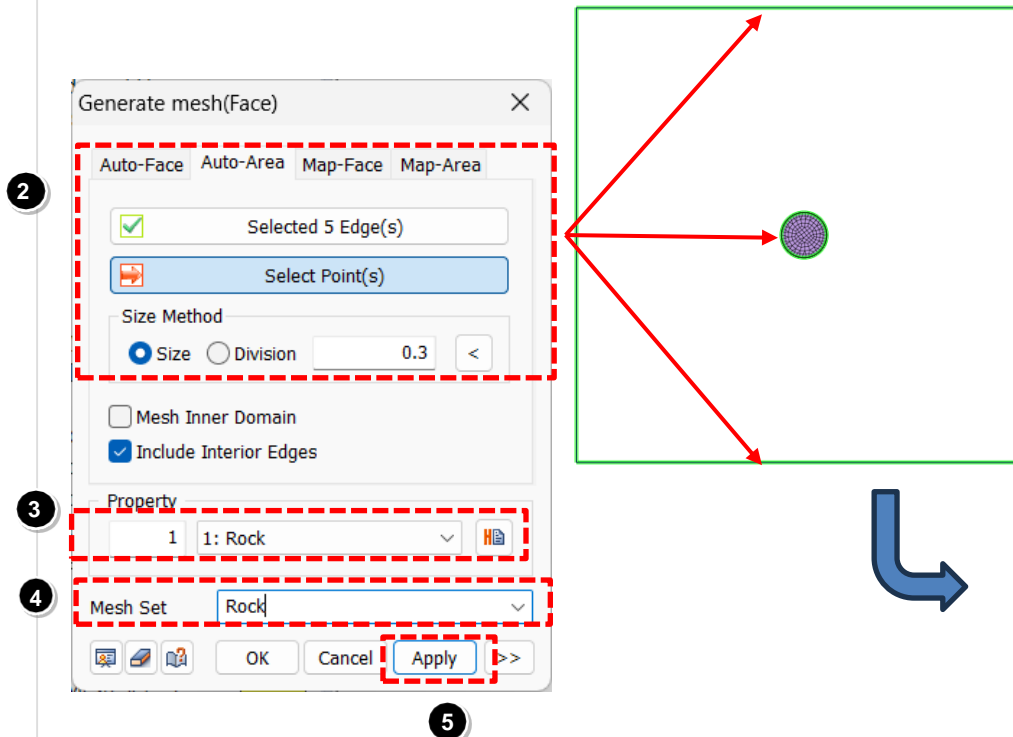
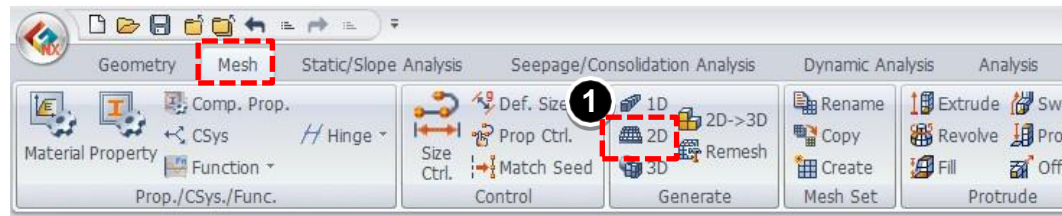
Procedure

- ❶ Go to Mesh > Generate > 2D
- ❷ Select the edges as shown.
Give mesh size **0.3 m**
- ❸ Select property as **Rock**
- ❹ Name the Mesh set **Excavation**
- ❺ Click **Apply**



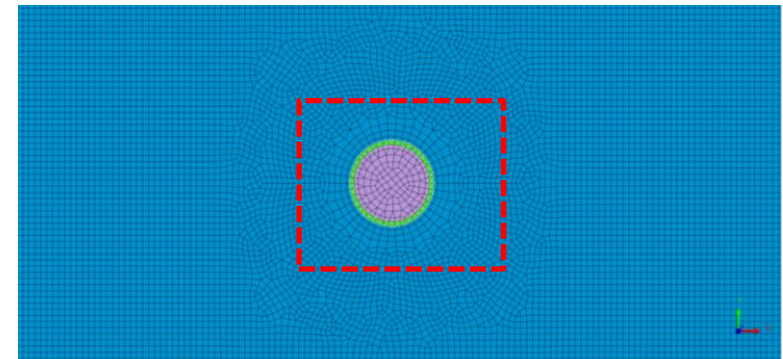
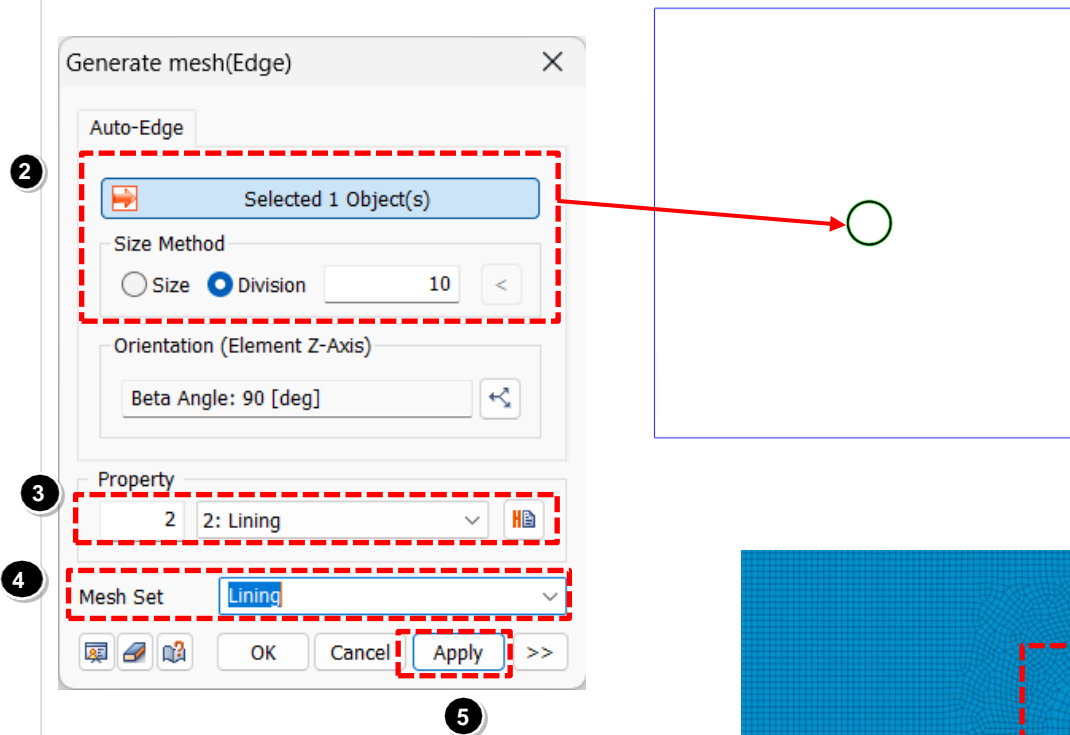
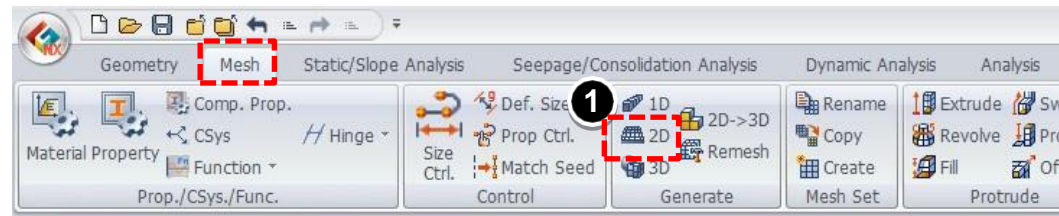
Procedure

- ❶ Go to Mesh > Generate > 2D
- ❷ Select the edges and enter size as **0.3m**
- ❸ Select property as **Rock**
- ❹ Name the Mesh set **Rock**
- ❺ Click **Apply**



Procedure

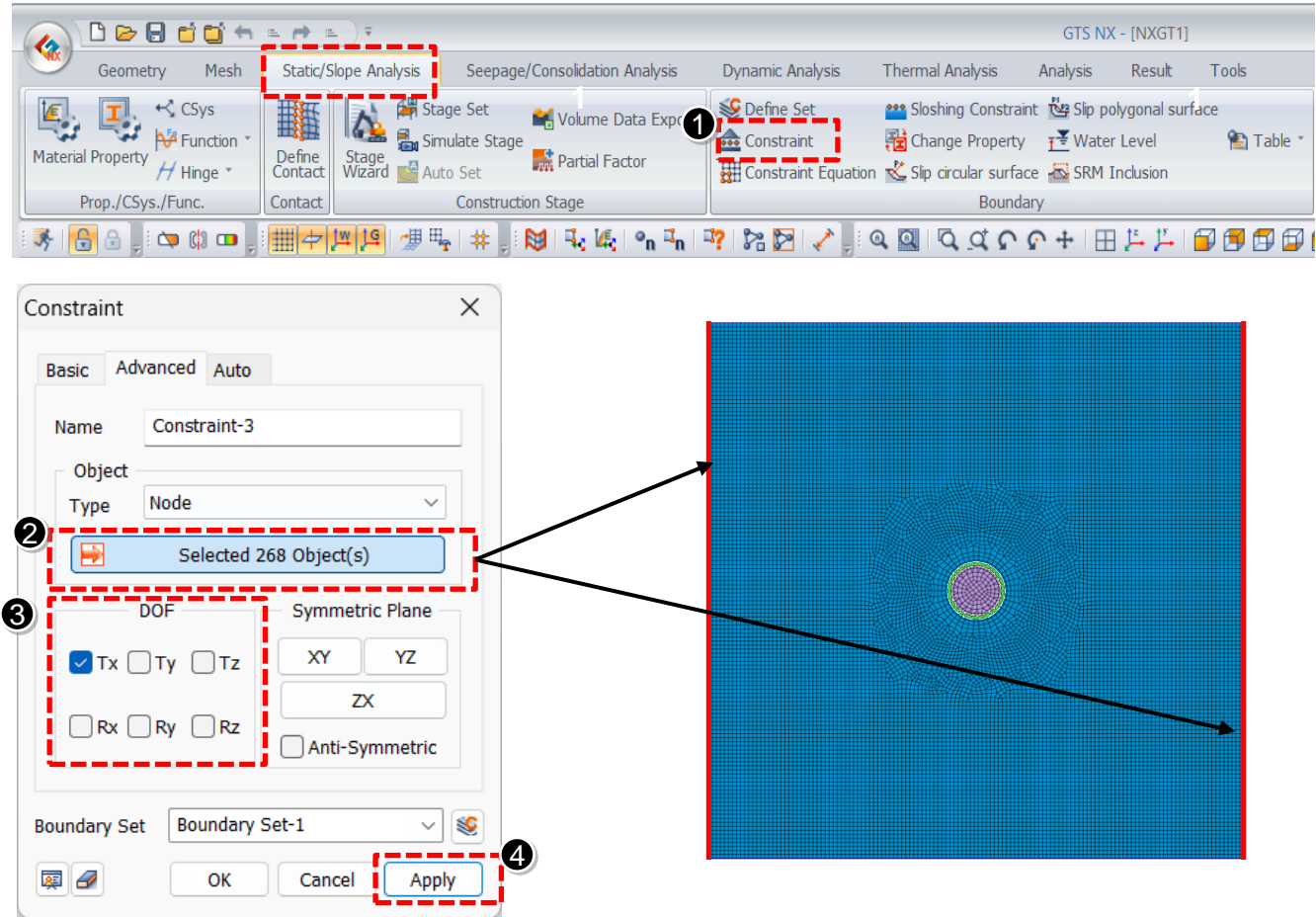
- ❶ Go to Mesh > Generate > 1D
- ❷ Select the edges and enter division as 10
- ❸ Select property as **Lining**
- ❹ Name the Mesh set **Lining**
- ❺ Click **Apply**



6-1 Boundary Condition

Procedure

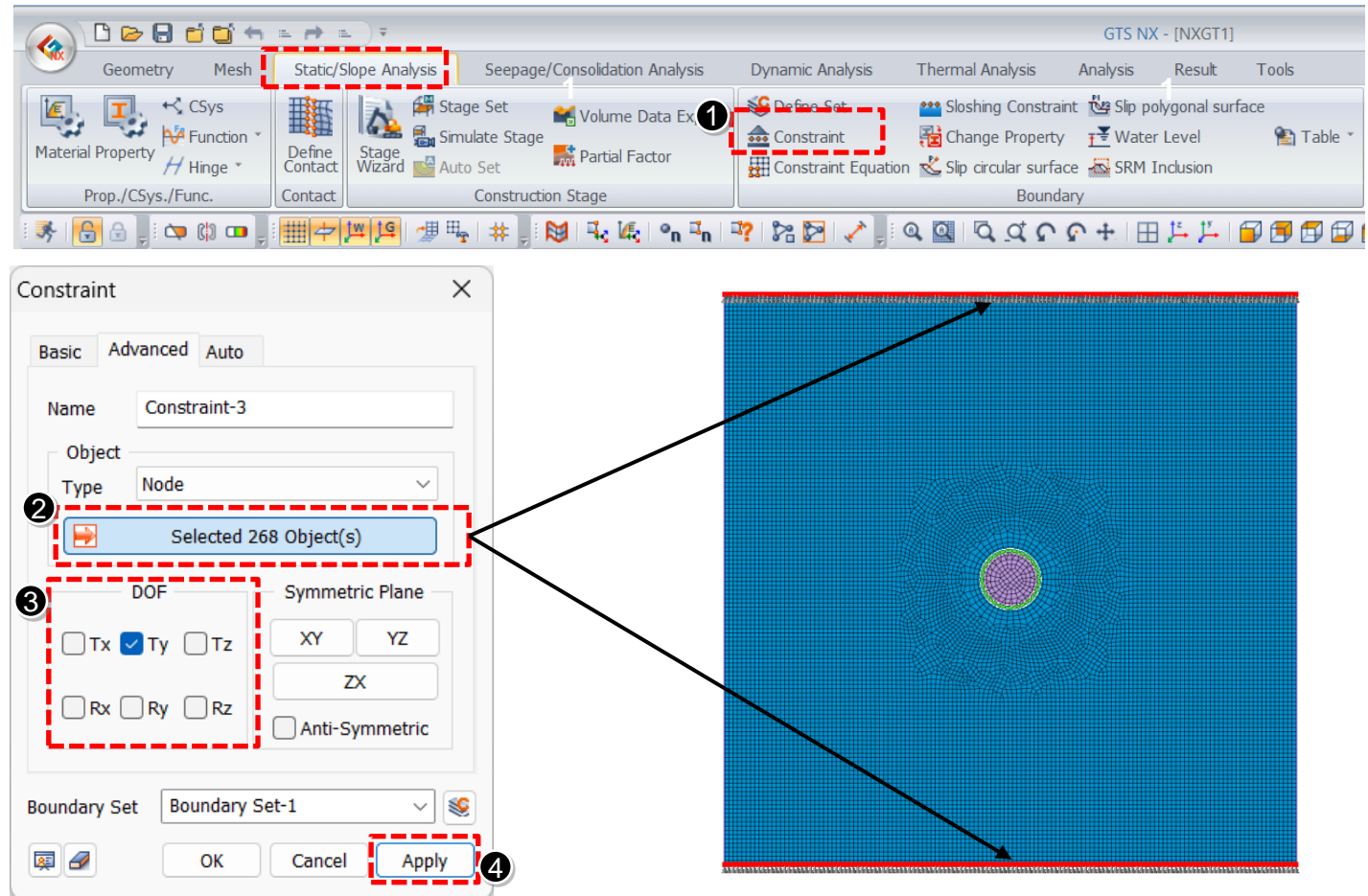
- ❶ Go to Static/Slope > **Constraints**
- ❷ Select the nodes as shown in the image
- ❸ Select **Tx** as **DOF**
- ❹ Click Apply



6-2 Boundary Condition

Procedure

- ① Go to Static/Slope > Constraints
- ② Select the nodes as shown in the image
- ③ Select Ty as DOF
- ④ Click Apply



7 Load Definition : In-situ Stress

Procedure

- 1 Go to Static/Slope Analysis > Load > **Initial Equilibrium Force**
- 2 Select Element type as **Plane Strain/Stress**
- 3 Select all the elements as shown
- 4 Enter the **Sxx = -4000 kN/m²** and **Syy = -2000 kN/m²** as shown
- 5 Enter Load Set Name as **Insitu Stress**
- 6 Click **OK**

The screenshot displays the GTS NX - [NXGT1] software interface. The 'Static/Slope Analysis' tab is selected in the top menu. The 'Load' button in the 'Load' group is highlighted with a red dashed box and a red arrow. The 'Initial Equilibrium Force' dialog box is open, showing the following settings:

- Name:** Initial Equilibrium Force-1
- Element Type:** Plane Strain/Plane Stress
- Object:** Selected 6428 Object(s)
- Ref. CSys:** Global Rectangular
- Components:**

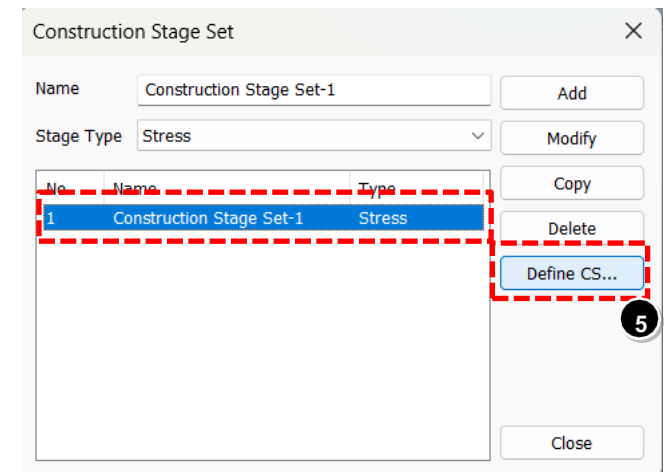
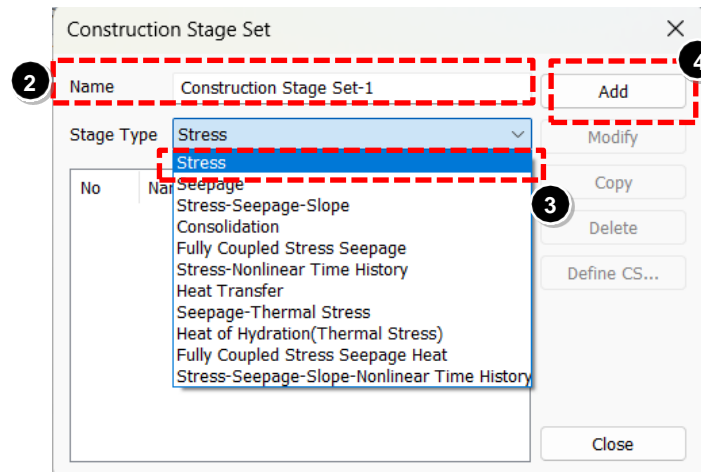
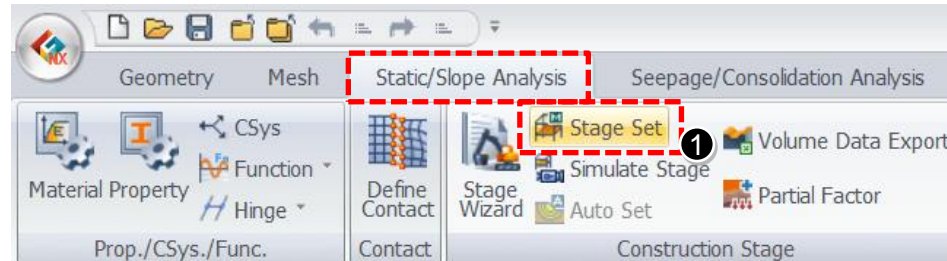
Component	Value	Unit	Base Function
Sxx	-4000	kN/m ²	None
Syy	-2000	kN/m ²	None
Szz	0	kN/m ²	None
Sxy	0	kN/m ²	None
- Self-Weight Consideration:** ☐
- Load Set:** Insitu Stress
- Buttons:** OK, Cancel, Apply

Below the dialog box, two mesh visualizations are shown. The left mesh is green, representing the initial state. The right mesh is red, representing the state after applying the initial equilibrium force. A blue arrow points from the green mesh to the red mesh.

8-1 Construction Stage Definition

Procedure

- ❶ Go to Static/Slope Analysis > Construction Stage > Stage Set
- ❷ Give the Name as **"Construction Stage 1"**
- ❸ Select the stage type as **Stress**
- ❹ Click **Add**
- ❺ Select the defined stage and click **Define CS**



8-2 Construction Stage Set

Procedure

- 1 Give stage name as 'S1' for 'Construction Stage 1'
- 2 Select the Stage type as 'Stress State'
- 3 Activate mesh set , boundary condition and static load as shown
- 4 Click Save

Define Construction Stage

Construction Stage Set Name: Construction Stage Set-1

Stage ID: 1: S1

Stage Name: S1

Stage Type: Stress

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

Initial Condition:

- ☐ Define Water Level For Global
- ☐ Define Water Level For Mesh Set
- ☐ Sub Stage...
- ☐ LDF...
- ☐ Clear Displacement
- ☐ Slope Stability(SRM)
- ☐ Slope Stability(SAM)

Set Data:

- Mesh
- Default Mesh Set
- Excavation
- Excavation-1
- Lining
- Rock
- Rock-1
- Boundary Condition
- Boundary Set-1
- Static Load
- Insitu Stress
- Contact

Activated Data:

- Mesh
- Excavation
- Rock
- Boundary Condition
- Boundary Set-1
- Static Load
- Insitu Stress
- Contact

Deactivated Data:

- Mesh
- Boundary Condition
- Static Load
- Contact

Sort By: Name

Show Data: Activate

Buttons: Save, Close

8-3 Construction Stage Set

Procedure

- 1 Give stage name as '**S2**' for '**Construction Stage 1**'
- 2 Select the Stage type as '**Stress State**'
- 3 Activate **lining mesh set**
- 4 Deactivate **Excavation**
- 5 Click **Save**

Define Construction Stage

Construction Stage Set Name: Construction Stage Set-1

1 Stage ID: 2: S2 (Red dashed box)

Stage Name: S2 (Red dashed box)

Stage Type: Stress (Red dashed box)

2 Move to Previous Move to Next New Insert Delete

Set Data Activated Data Deactivated Data

3 4

Sort By: Name Show Data: Activate

5 Save Close

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...

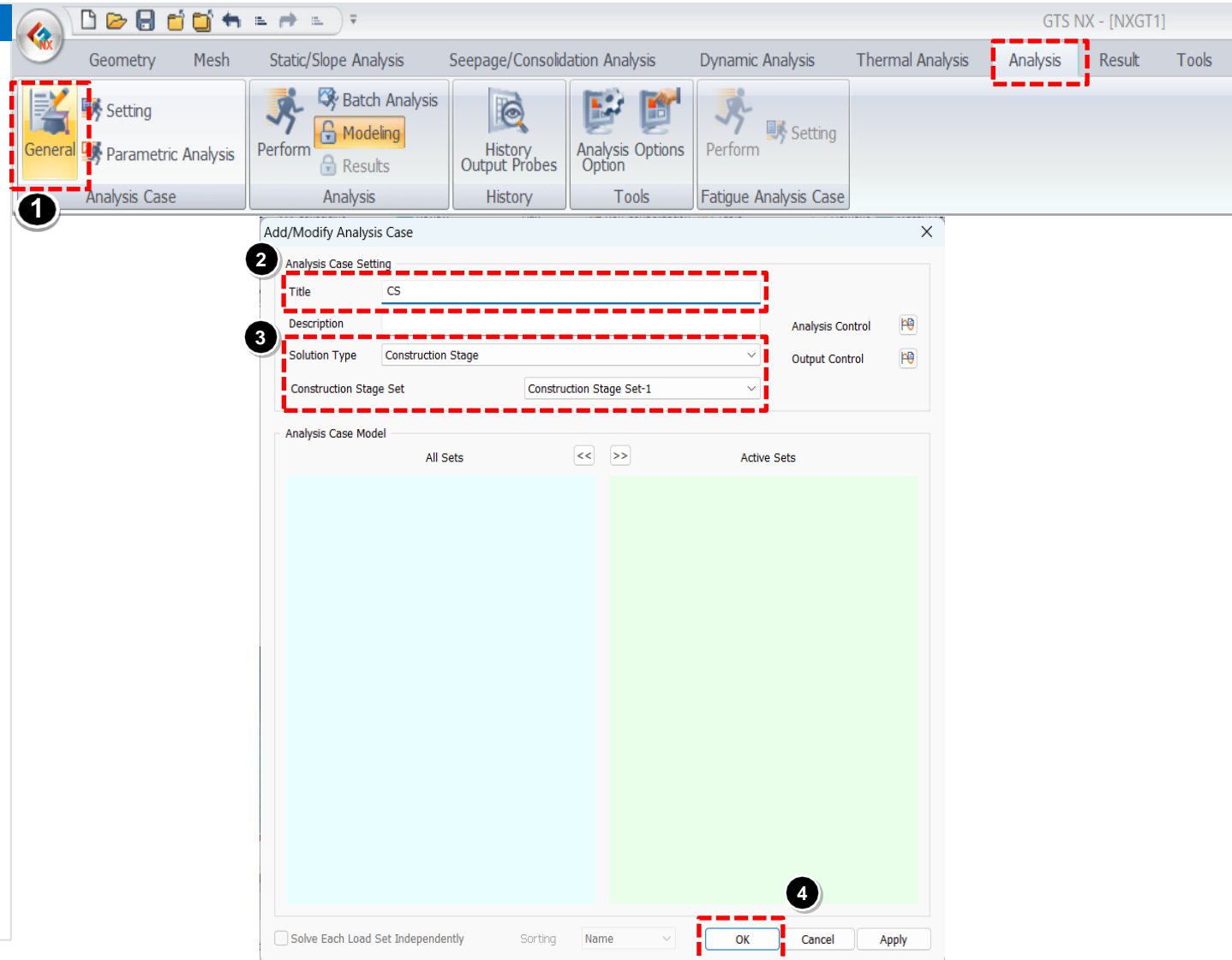
☐ Clear Displacement

☐ Slope Stability(SRM)

☐ Slope Stability(SAM)

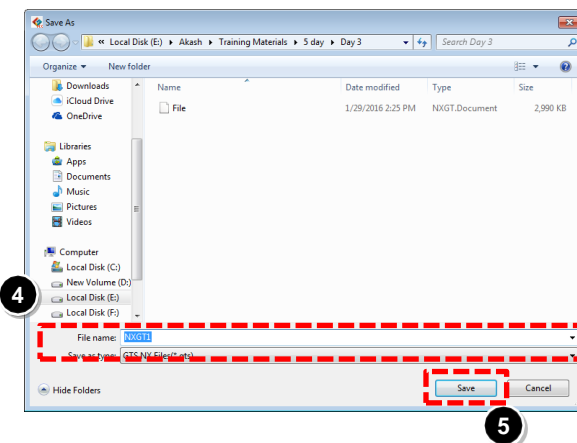
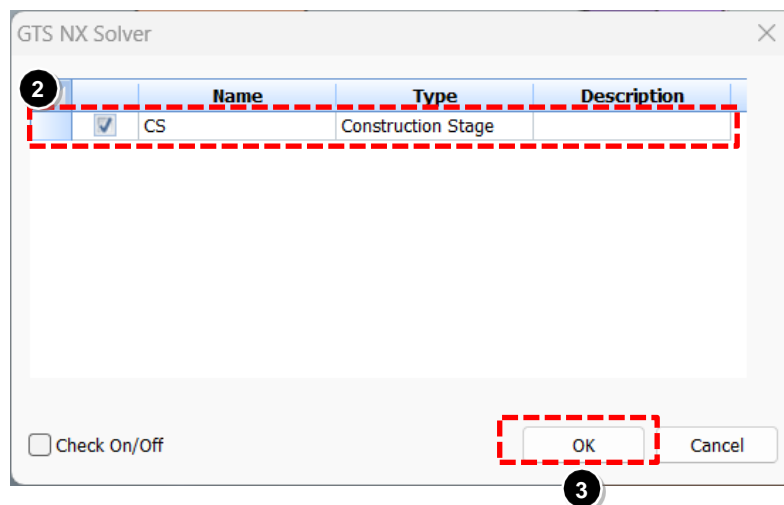
Procedure

- ❶ Go to Analysis > General
- ❷ Give the name as **CS**
- ❸ Select Solution Type as '**Construction Stage**' and Construction Stage Set as '**Construction Stage Set 1**'
- ❹ Click **OK**



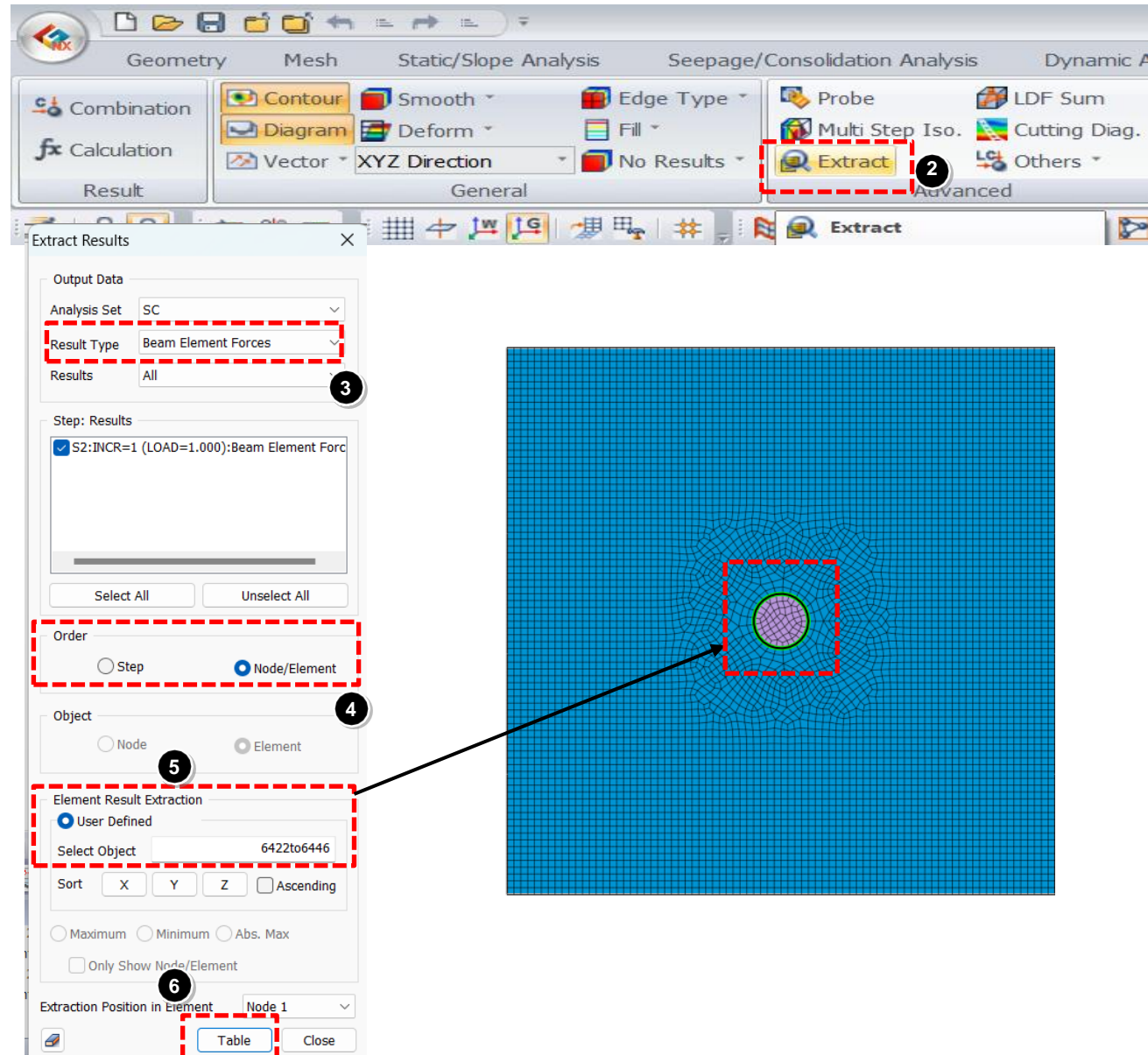
Procedure

- ❶ Go to Analysis > Perform
- ❷ Select **CS (Construction Stage Analysis)**
- ❸ Click **OK**
- ❹ Give the File name as **"MN Curve Analysis"**
- ❺ Save it at desired location



Procedure

- ❶ Go to works tree and Enable mesh sets to check the results.
- ❷ Go to Works Results> Advanced> Extract
- ❸ Select Output Data > Result Type > **Beam Element Forces**
- ❹ Select **Node/Element**
- ❺ Select **all the Elements** in Lining
- ❻ Click **Table**



Procedure

- On clicking table, following table will open
- Select the **Axial Force, Bending Moment and Shear Force** column and right click > Export to Json
- Save the Json file. We will import it for generating curves.

No	Element	X (m)	Y (m)	Z (m)	S2:INCR=1 (LOAD=1.000) AXIAL FORCE X Node 1	S2:INCR=1 (LOAD=1.000) SHEAR FORCE Y Node 1	S2:INCR=1 (LOAD=1.000) SHEAR FORCE Z Node 1	S2:INCR=1 (LOAD=1.000) TORQUE Node 1	S2:INCR=1 (LOAD=1.000) BENDING MOMENT Y	S2:INCR=1 (LOAD=1.000) BENDING MOMENT Z
1	6422	3.718075e-001	1.949083e+000	0.000000e+000	-2.409015e+003	0.000000e+000	7.473831e-001	0.000000e+000	-1.071154e+001	0.000000e+000
2	6423	-1.921891e+000	4.934578e-001	0.000000e+000	-1.311607e+003	0.000000e+000	-4.726542e+000	0.000000e+000	9.426989e+000	0.000000e+000
3	6424	-1.063203e+000	-1.675340e+000	0.000000e+000	-2.095766e+003	0.000000e+000	9.163808e+000	0.000000e+000	-2.896089e+000	0.000000e+000
4	6425	-1.921891e+000	-4.934578e-001	0.000000e+000	-1.315329e+003	0.000000e+000	6.545080e+000	0.000000e+000	1.287190e+001	0.000000e+000
5	6426	-1.738793e+000	-9.559098e-001	0.000000e+000	-1.514996e+003	0.000000e+000	1.262232e+001	0.000000e+000	9.590640e+000	0.000000e+000
6	6427	-6.131606e-001	-1.887114e+000	0.000000e+000	-2.324236e+003	0.000000e+000	9.974755e+000	0.000000e+000	-7.490208e+000	0.000000e+000
7	6428	1.844890e+000	-7.304436e-001	0.000000e+000	-1.378874e+003	0.000000e+000	-4.211339e+000	0.000000e+000	8.005755e+000	0.000000e+000
8	6429	1.264795e+000	1.528875e+000	0.000000e+000	-1.946372e+003	0.000000e+000	1.445835e+001	0.000000e+000	1.042802e+000	0.000000e+000
9	6430	8.448438e-001	-1.795384e+000	0.000000e+000	-2.219098e+003	0.000000e+000	-8.766004e+000	0.000000e+000	-1.064792e+001	0.000000e+000
10	6431	-1.245908e-001	1.980314e+000	0.000000e+000	-2.442578e+003	0.000000e+000	2.475741e+000	0.000000e+000	-1.108622e+001	0.000000e+000
11	6432	1.605275e+000	-1.166301e+000	0.000000e+000	-1.643266e+003	0.000000e+000	-1.362601e+001	0.000000e+000	1.174589e+000	0.000000e+000
12	6433	1.968583e+000	-2.486899e-001	0.000000e+000	-1.237385e+003	0.000000e+000	-4.258276e+000	0.000000e+000	1.011704e+001	0.000000e+000
13	6434	-6.131606e-001	1.887114e+000	0.000000e+000	-2.330659e+003	0.000000e+000	-9.572617e+000	0.000000e+000	-1.232739e+001	0.000000e+000
14	6435	1.605275e+000	1.166301e+000	0.000000e+000	-1.643753e+003	0.000000e+000	1.422004e+001	0.000000e+000	8.171778e+000	0.000000e+000
15	6436	-1.245908e-001	-1.980314e+000	0.000000e+000	-2.436536e+003	0.000000e+000	-3.021370e+000	0.000000e+000	-1.249088e+001	0.000000e+000
16	6437	1.264795e+000	-1.528875e+000	0.000000e+000	-1.946127e+003	0.000000e+000	-1.481614e+001	0.000000e+000	-6.253231e+000	0.000000e+000
17	6438	-1.063203e+000	1.675340e+000	0.000000e+000	-2.100577e+003	0.000000e+000	-9.116863e+000	0.000000e+000	-7.528327e+000	0.000000e+000
18	6439	-1.446441e+000	-1.358298e+000	0.000000e+000	-1.795140e+003	0.000000e+000	1.228474e+001	0.000000e+000	3.262656e+000	0.000000e+000
19	6440	-1.738793e+000	9.559098e-001	0.000000e+000	-1.518279e+003	0.000000e+000	-1.119271e+001	0.000000e+000	3.815714e+000	0.000000e+000
20	6441	-1.446441e+000	1.358298e+000	0.000000e+000	-1.802788e+003	0.000000e+000	-1.351089e+001	0.000000e+000	-2.957743e+000	0.000000e+000
21	6442	1.968583e+000	-2.486899e-001	0.000000e+000	-1.236653e+003	0.000000e+000	4.405777e+000	0.000000e+000	1.225185e+001	0.000000e+000
22	6443	8.448438e-001	-1.795384e+000	0.000000e+000	-2.220838e+003	0.000000e+000	8.987823e+000	0.000000e+000	-6.205645e+000	0.000000e+000
23	6444	-1.98	-1.98	0.000000e+000	-1.240093e+003	0.000000e+000	-2.144971e+000	0.000000e+000	1.179656e+001	0.000000e+000
24	6445	1.84	1.84	0.000000e+000	-1.378871e+003	0.000000e+000	3.732674e+000	0.000000e+000	1.004309e+001	0.000000e+000
25	6446	3.7	3.7	0.000000e+000	-2.405820e+003	0.000000e+000	-6.547558e-001	0.000000e+000	-1.097617e+001	0.000000e+000

Sorting Dialog...
Style Dialog...
Show Graph...
Export to Excel
Export to Json

Beam Element Forces /

Axial Force(N)

Shear Force(Q)

Bending Moment(M)

11-3 Results > M-N Curve

- ① Go to Tools tab > Options > **Tunnel Lining Plots**
- ② Click **Tunnel Lining Interaction Curves** > Select **Carranza Torres** and **OK**
- ③ Go to File > Click on Open > **Open Json File** (as saved earlier for equivalent properties)
- ④ Click on File > **Import GTS NX Json file** (for importing input forces)
- ⑤ Click on **Get Redistributed Curves**

1 Go to Tools tab > Options > **Tunnel Lining Plots**

2 Click **Tunnel Lining Interaction Curves** > Select **Carranza Torres** and **OK**

3 Go to File > Click on Open > **Open Json File** (as saved earlier for equivalent properties)

4 Click on File > **Import GTS NX Json file** (for importing input forces)

5 Click on **Get Redistributed Curves**

Tunnel Lining Interaction curves

Design By : Carranza Torres and Diederichs OK

Developed By:
Midas R&D Center, Mumbai
India

Midas

Caranza Torres and Diederichs*

File

Member Information

Section ID: S01 Section:

Material Properties

Steel Properties

Modulus of Elasticity E _{st}	2.1e8	kN/m ²
Compressive strength f _{st}	5e5	kN/m ²
Tensile strength f _{st}	5e5	kN/m ²
Poisson's ratio ν _{st}	0.25	

Shotcrete Properties

Modulus of Elasticity E _{sch}	3e7	kN/m ²
Compressive strength f _{sch}	4e4	kN/m ²
Tensile strength f _{sch}	2.5e3	kN/m ²
Poisson's ratio ν _{sch}	0.15	

Section Properties

Steel Properties

Steel Set Spacing (s)	0.6	m
Steel Set Height (t _{st})	0.162	m
Area of steel set (A _{st})	0.00475	m ²
Moment of Inertia (I _{st})	2.23e-5	m ⁴

Shotcrete Properties

Shotcrete Thickness (t _{sch})	0.2	m
Area of Shotcrete (A _{sch})	0.12	m ²
Moment of Inertia (I _{sch})	0.0004	m ⁴
Radius of Tunnel (R)	2	m

General parameters

Total Width of lining (b) 0.6 m Factors of safety 1.0 1.5 2.0

Equivalent properties

E_n Thickness (t_{en}) .209 m E_n Modulus of Elasticity E_{en} 37862075.26 kN/m²

Input forces from numerical analysis :

M (kNm)	N (kN)	Q (kN)
-1.071154...	-2.409015...	7.47383
9.426989...	-1.311607...	-4.7265
-2.896089...	-2.095766...	9.16380
1.287190...	-1.315329...	6.54508
9.590640...	-1.514996...	1.26223
-7.490208...	-2.324236...	9.97475
8.005755...	-1.378874...	-4.2113
1.042802...	-1.946372...	1.44583
-1.064792...	-2.219098...	-8.7660

Redistributed for :

Mst (kNm)	Nst (kN)	Qst (kN)	Msh	Nsh (kN)

Get Redistributed Forces View Curves Generate Report

11-4 Results > M-N Curve

Procedure

- 1 After Clicking Get Redistributed Force, the following forces will be shown.
- 2 Click on **View Curves** to generate **Support Capacity plots**.
- 3 We can also generate report by clicking on **Generate Report** option
- 4 User can always change the **FOS** to obtain desired plots

Caranza Torres and Diederichs*

File

Member Information
Section ID: S01 Section:

Material Properties

Steel Properties		
Modulus of Elasticity E _{st}	2.1e8	kN/m ²
Compressive strength f _{ct}	5e5	kN/m ²
Tensile strength f _{ct}	5e5	kN/m ²
Poisson's ratio ν _{st}	0.25	

Section Properties

Steel Properties		
Steel Set Spacing (s)	0.6	m
Steel Set Height (t _{st})	0.162	m
Area of steel set (A _{st})	0.00475	m ²
Moment of Inertia (I _{st})	2.23e-5	m ⁴

Shotcrete Properties

Shotcrete Properties		
Modulus of Elasticity E _{ch}	3e7	kN/m ²
Compressive strength f _{ch}	4e4	kN/m ²
Tensile strength f _{ch}	2.5e3	kN/m ²
Poisson's ratio ν _{ch}	0.15	

General parameters

Total Width of lining (b)	0.6	m
Factors of safety	1.0 1.5 2.0	

Equivalent properties

E _n Thickness (t _{en})	.209	m
E _n Modulus of Elasticity E _{en}	37862075.26	kN/m ²

Input forces from numerical analysis :

M (kNm)	N (kN)	Q (kN)
-1.071154...	-2.409015...	7.47383
9.426989...	-1.311607...	-4.7265
-2.896089...	-2.095766...	9.16380
1.287190...	-1.315329...	6.54508
9.590640...	-1.514996...	1.26223
-7.490208...	-2.324236...	9.97475
8.005755...	-1.378874...	-4.2113
1.042802...	-1.946372...	1.44583
-1.064792...	-2.219098...	-8.7660

Redistributed for 1.0 :

M _{st} (kNm)	N _{st} (kN)	Q _{st} (kN)	M _{sh}	N _{sh} (kN)
-3.097967...	540.3243...	0.216156...	-7.613572...	1868.690...
2.726452...	293.6873...	-1.366999...	6.700536...	1017.919...
-0.837600...	469.8559...	2.650336...	-2.058488...	1625.910...
3.722782...	294.4095...	1.892953...	9.149117...	1020.919...
2.773783...	339.2712...	3.650599...	6.816856...	1175.724...
-2.166301...	521.2164...	2.884876...	-5.323906...	1803.019...
2.315406...	308.8113...	-1.217994...	5.690348...	1070.062...
0.301596...	436.2413...	4.181611...	0.741205...	1510.130...
-3.079567...	497.7527...	-2.535284...	-7.568352...	1721.345...

1 Get Redistributed Forces 2 View Curves 3 Generate Report

11-5 Results > M-N Curve

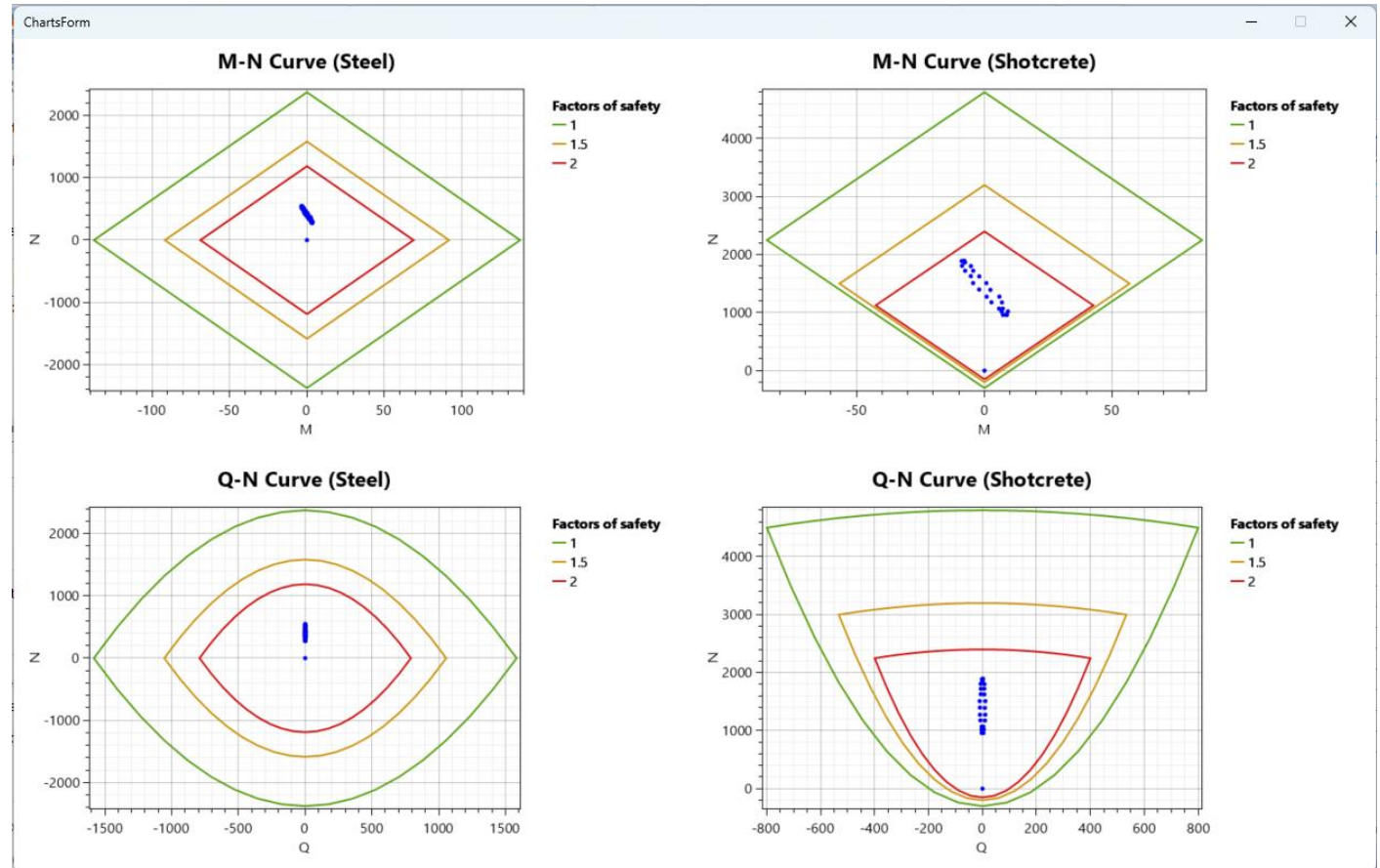
Procedure

View curves will generate following curves.

Blue points represent the **demand**

Green , Orange , Red Curve represent the Capacity for the **FOS 1, 1.5, 2**

As we can see that demand is inside the capacity plots, indicating that the tunnel lining design is safe.



Happy Modelling

