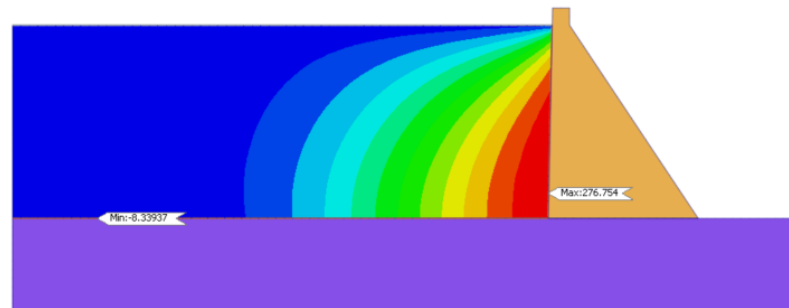


MIDAS *Technical Material*

Tutorial



Sloshing Fluid Element

Contents

- **Step 1:** Initial Setting
- **Step 2:** Geometry
- **Step 3:** Material Definition
- **Step 4:** Property Definition
- **Step 5:** Meshing
- **Step 6:** Boundary Condition
- **Step 7:** Loads Definition
- **Step 8:** Analysis Case
- **Step 9:** Analysis
- **Step 10:** Results

Numerical Model Design**Overview**

The seismic event in the case of the Dam Structure with reservoir can result in a hydrodynamic force acting on the dam (also can be termed as sloshing action of the fluid). This Fluid Structure Interaction can result in frequency dependent hydrodynamic pressures that can be interpreted as an added force, an added mass, and an added damping.

The following methods are widely used to consider the fluid-structure interaction in the case of dams,

1. Westergaard's added mass
2. C.N. Zargar's added mass method:
3. Sloshing Fluid Medium

In this tutorial, we will go through the process of modeling the reservoir using the Sloshing Fluid Medium. In the case of a seismic event, this Sloshing medium results in the application of the hydrodynamic loads on the dam. This Sloshing Fluid Medium is only applicable in Eigenvalue Analysis and Time History Analysis.

Sloshing Fluid

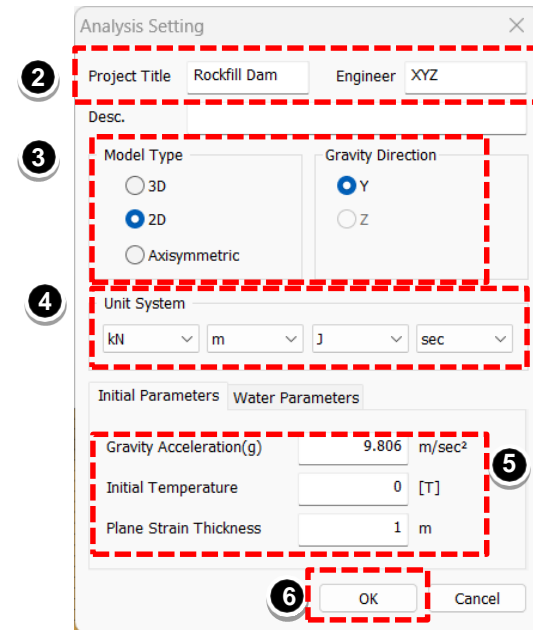
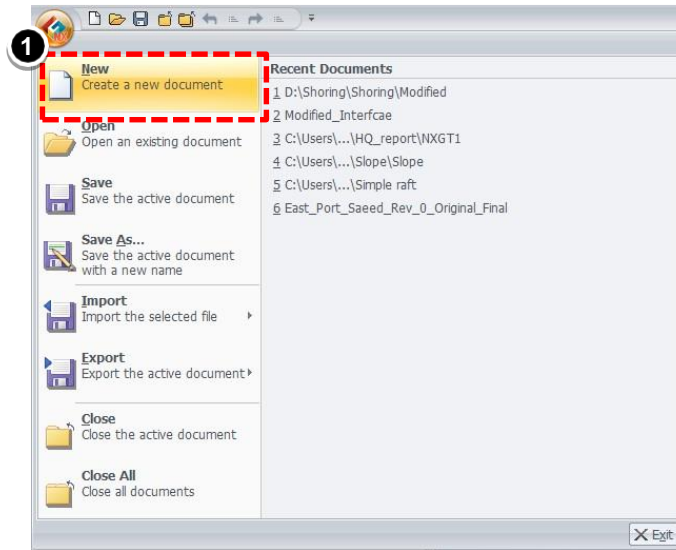
Material Properties

| Name | Material Model | Modulus of Elasticity (kN/m²) | Bulk Modulus (kN/m²) | Poisson's ratio | Unit Weight (kN/m³) |
|-------------------|-----------------|-------------------------------|----------------------|-----------------|----------------------------|
| Concrete (M25) | Elastic | 22410000 | - | 0.2 | 24.35 |
| Foundation (Rock) | Elastic | 22400000 | - | 0.2 | 0 (Massless foundation) |
| Reservoir Water | Sloshing Medium | - | 2200000 | - | 9.80665 |

Procedure

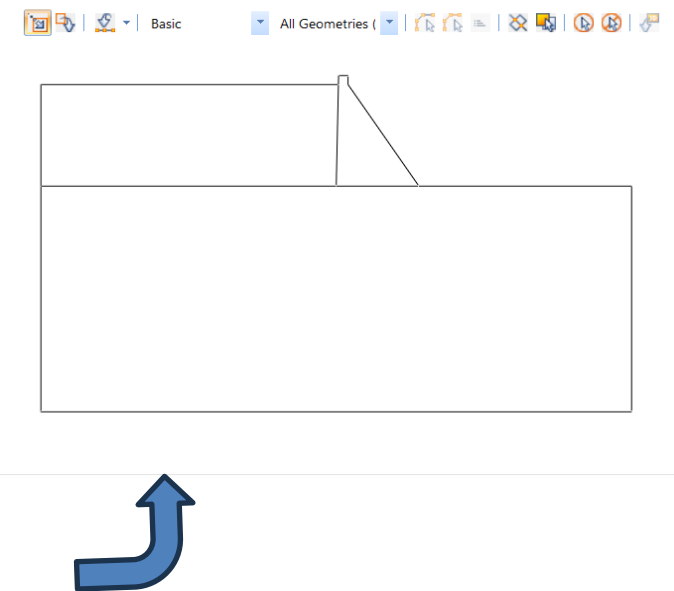
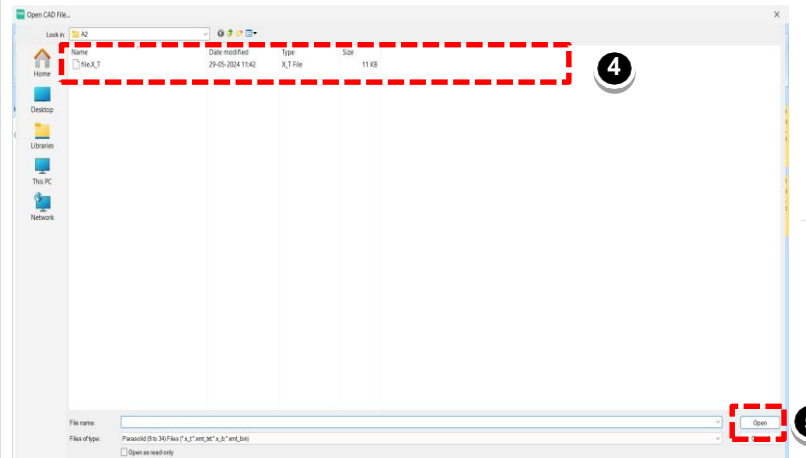
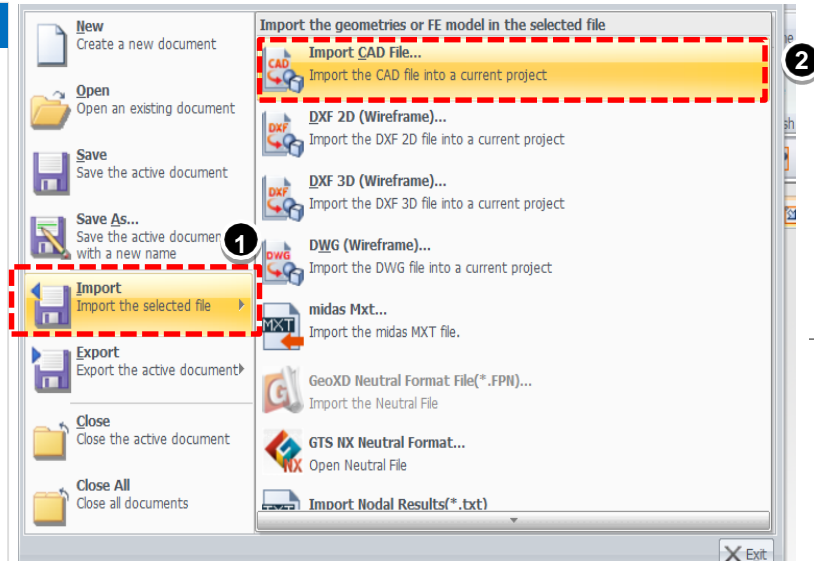
Starting Midas GTS NX

- ① Click on GTS NX icon > New Project
- ② Enter the Project name as Sloshing Fluid
- ③ Select Model Type as 2D and Gravity Direction as Y
- ④ Select kN ,m and sec in The Unit System
- ⑤ Use the Default values for Initial parameters
- ⑥ Click OK



Procedure

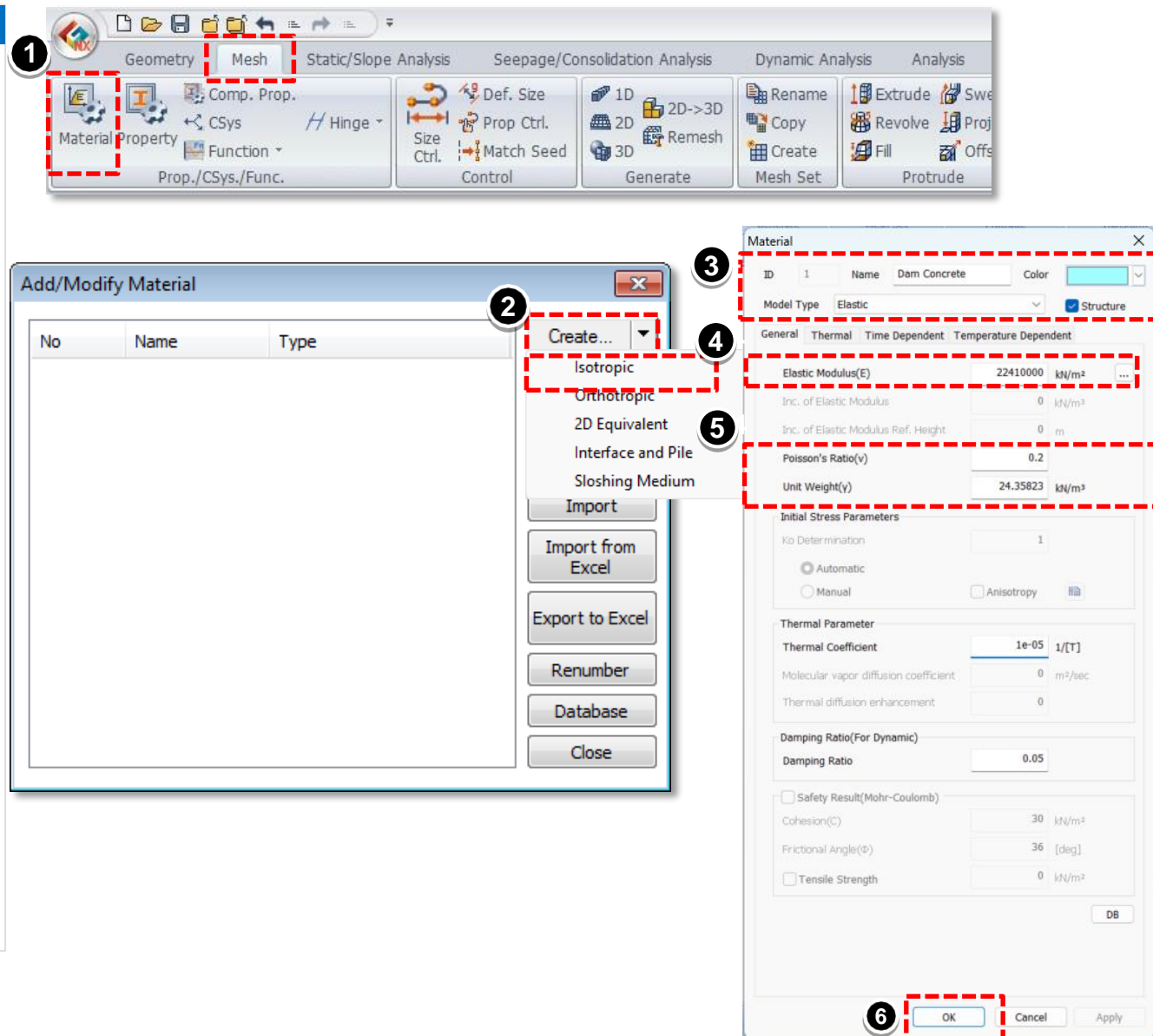
- 1 Go to GTS ICON > Import Projects
- 2 Select Import CAD File
- 3 Select .X_T file from desired location
- 4 Click Open



Procedure

- 1 Go to Mesh > Material
- 2 Click on Create dropdown. Select **Isotropic**
- 3 Name it as **Dam Concrete** and Select Model type as **Elastic**
- 4 In General tab, Enter the value of **Elastic modulus (E) =22410000 kN/m²**
- 5 Enter **Poisson's ratio= 0.2**,
Unit weight = 24.3528 kN/m³.
- 6 Click on 'OK'

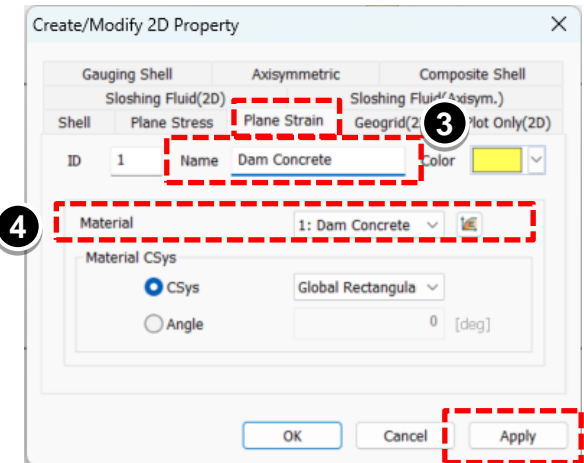
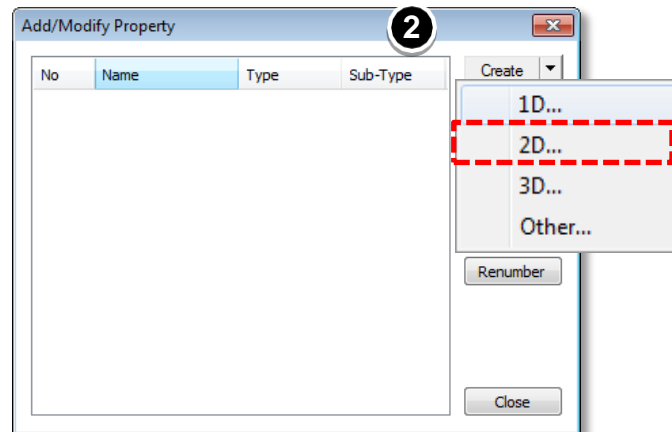
Create the same for Foundation (Rock) & Fluid (Sloshing Medium).
For the definition of the Fluid Material, select 'Sloshing Medium' and not 'Isotropic'.



Procedure

- ❶ Go to Mesh>Click on Property
- ❷ Click on Create. Select **2D**
- ❸ Select Plane Strain Element and name it as **Dam Concrete**
- ❹ Select **Dam Concrete** from Material drop down
- ❺ Click **Apply**
- ❻ Similarly Define the property for **Rock and Sloshing Fluid**

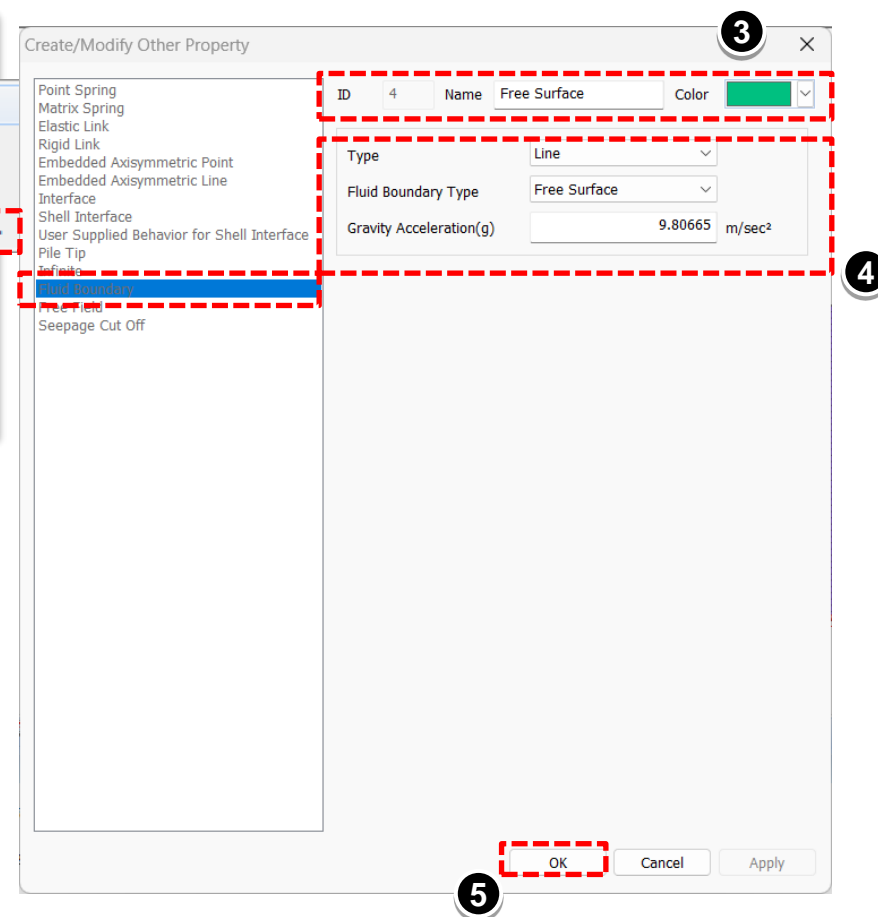
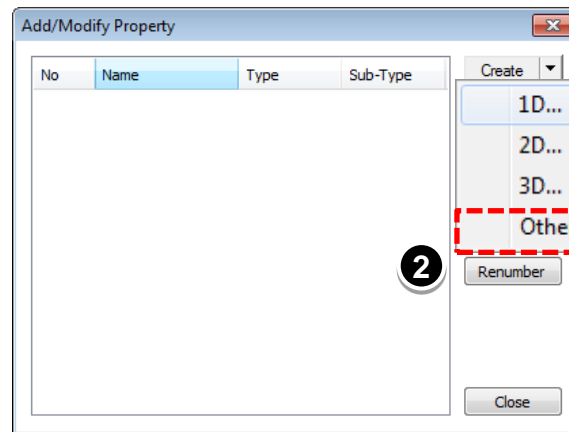
For the definition of the 'Sloshing Fluid' element, select the 'Sloshing Fluid (2D)'



4-2 Fluid Boundary Property

Procedure

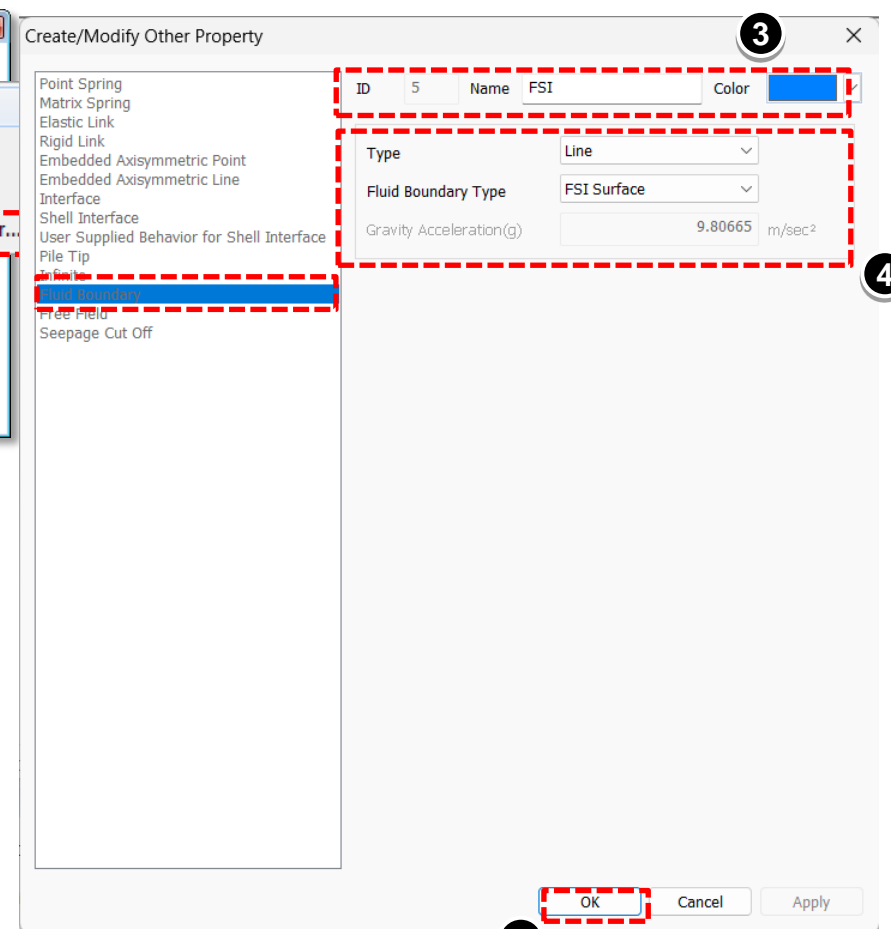
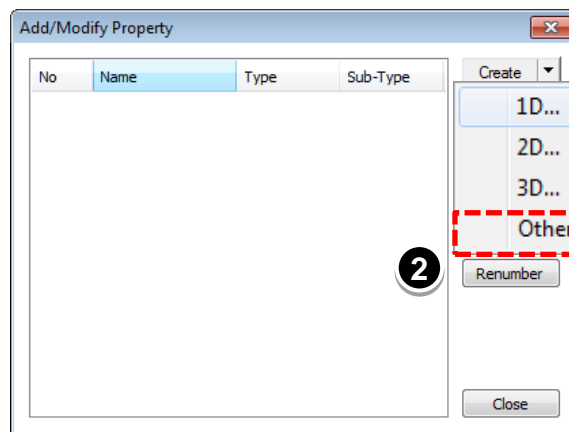
- 1 Go to Mesh>Click on Property
- 2 Click on Create. Select **Others**
- 3 Name it as **Free Surface**
- 4 Select **Fluid Boundary** type as **Free Surface** from Material drop down
- 5 Click **Apply**



4-3 Fluid Boundary Property

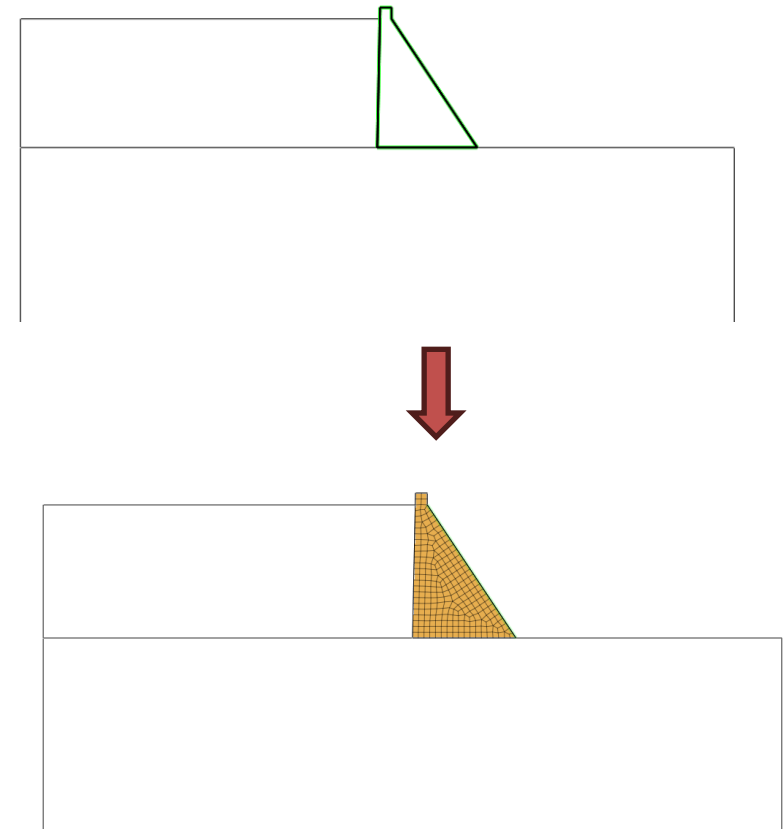
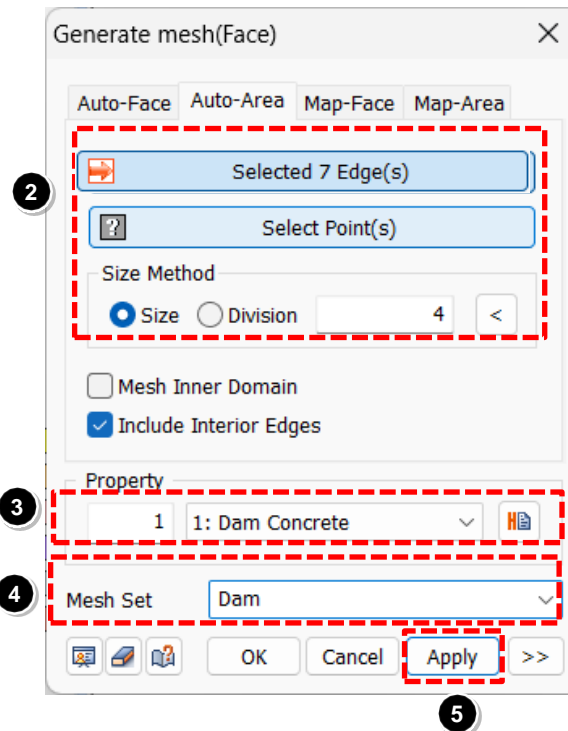
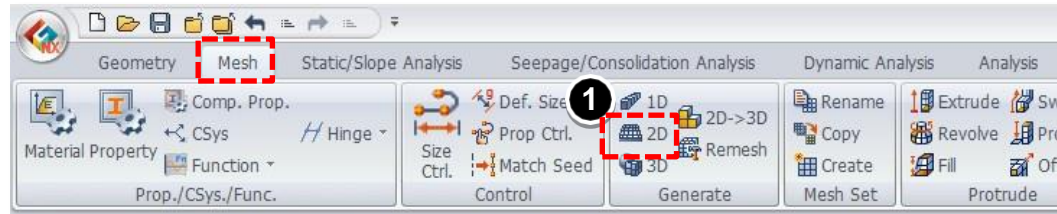
Procedure

- 1 Go to Mesh > Click on Property
- 2 Click on Create. Select **Others**
- 3 Name it as **FSI**
- 4 Select **Fluid Boundary type as FSI Surface** from Material drop down
- 5 Click **Apply**



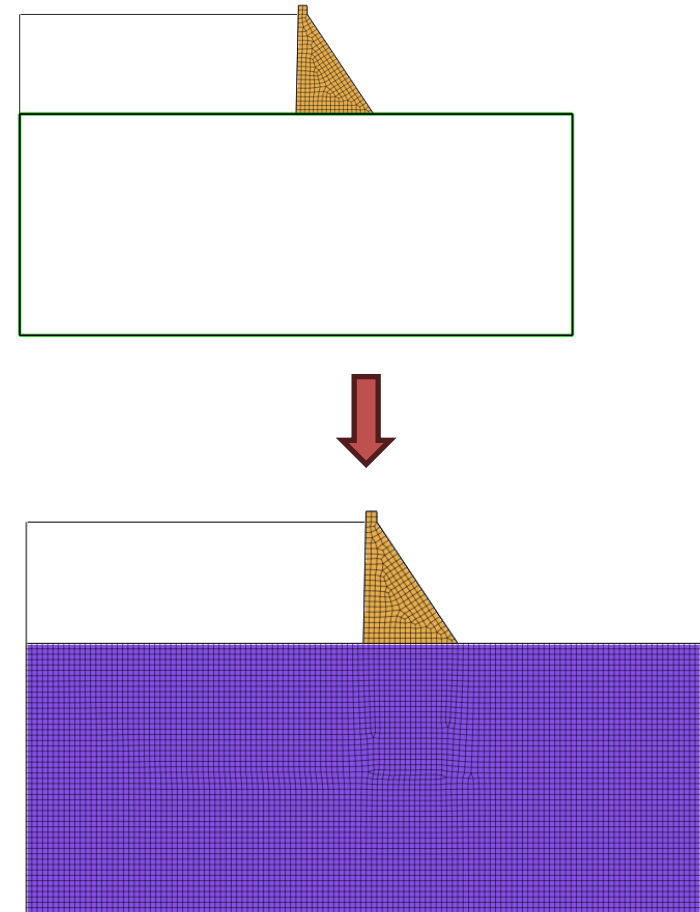
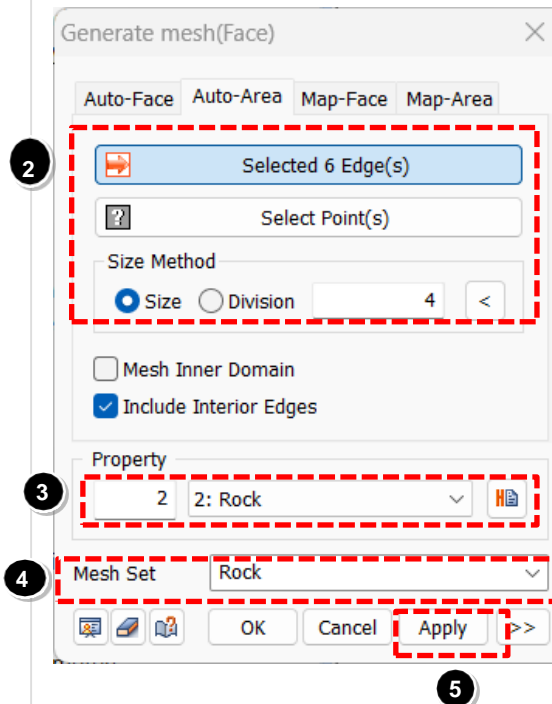
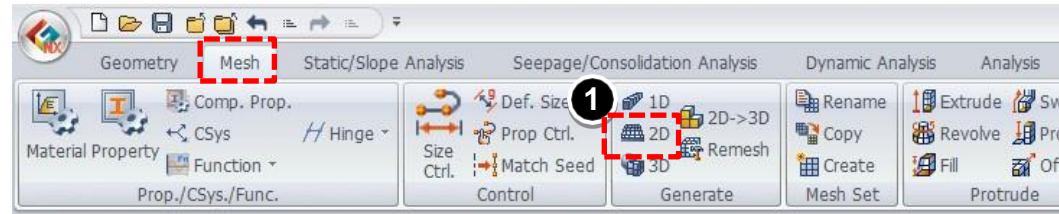
Procedure

- 1 Go to Mesh > Generate > 2D
- 2 Select the edges as shown.
Give mesh size 4 m
- 3 Select property as **Dam Concrete**
- 4 Name the Mesh set **Dam**
- 5 Click **Apply**



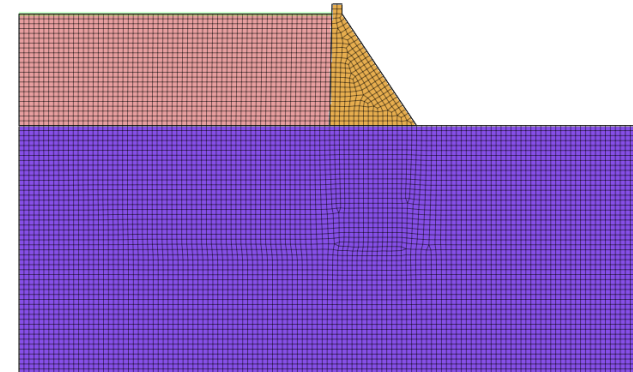
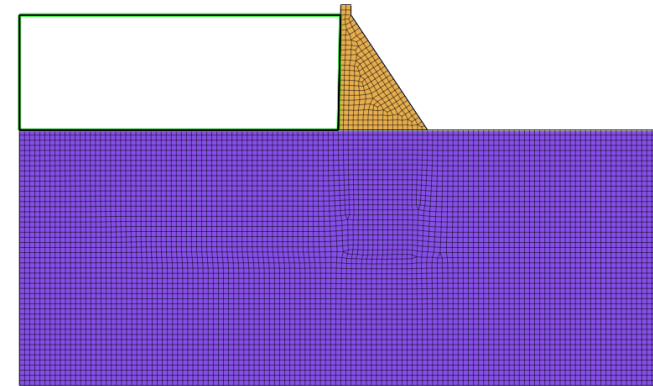
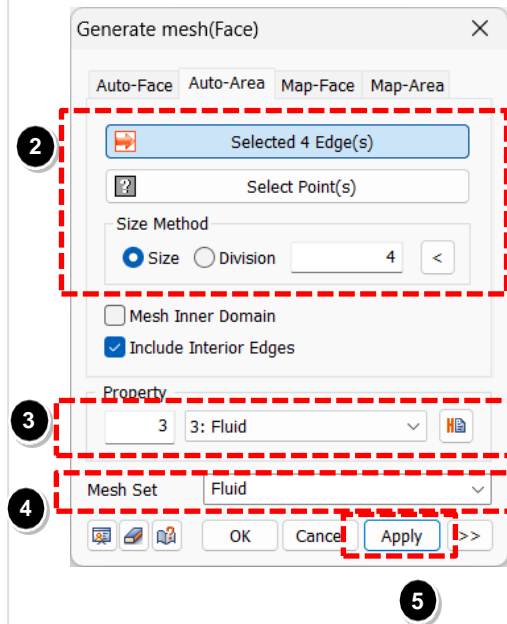
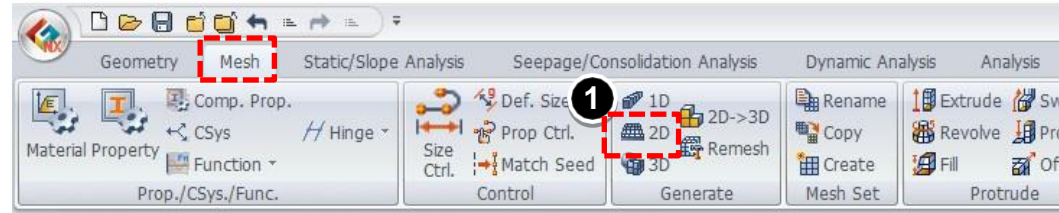
Procedure

- 1 Go to Mesh > Generate > 2D
- 2 Select the edges and enter size as **4m**
- 3 Select property as **Rock**
- 4 Name the Mesh set **Rock**
- 5 Click **Apply**



Procedure

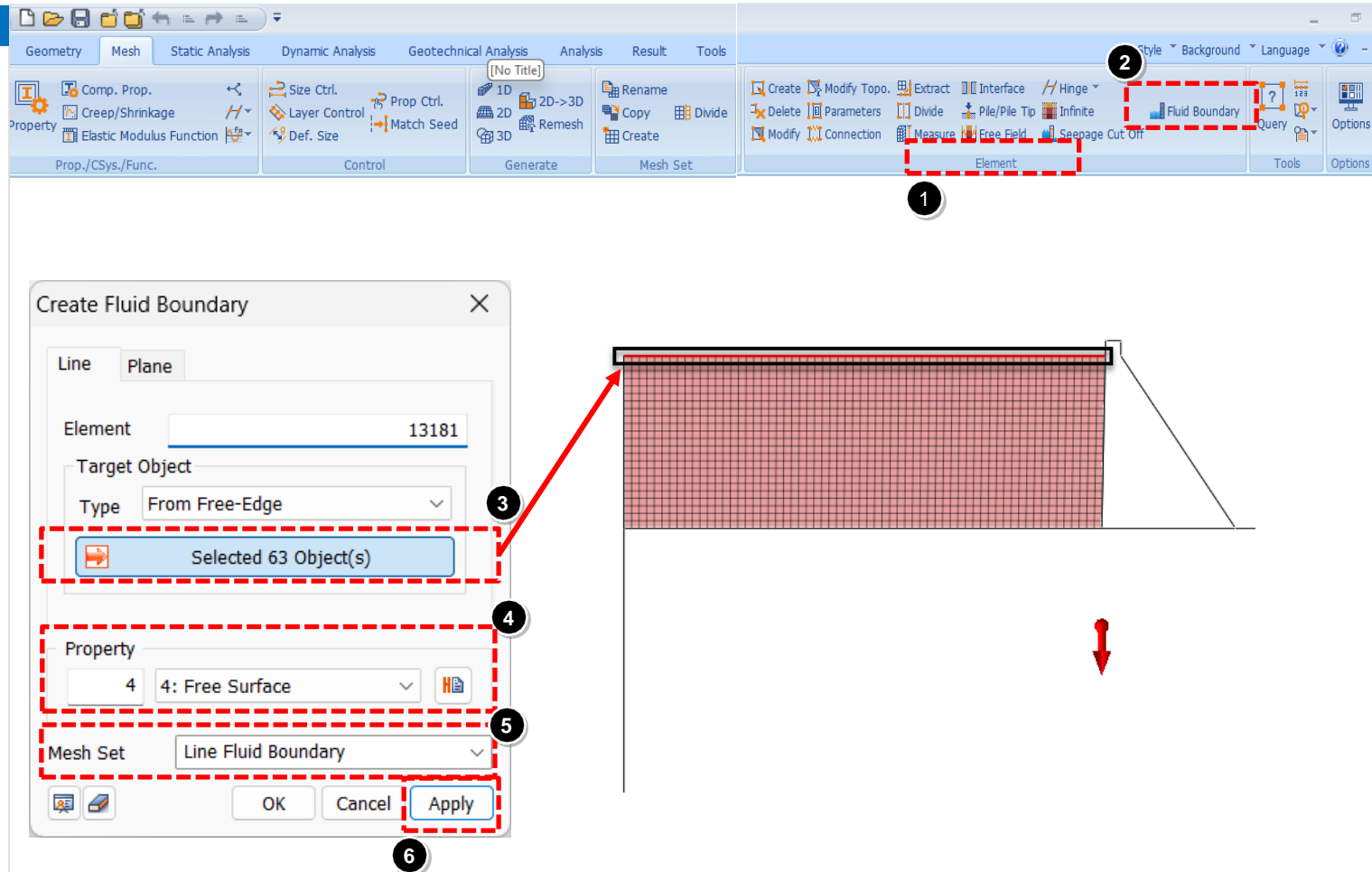
- 1 Go to Mesh > Generate > 2D
- 2 Select the edges as shown.
Give mesh size 4 m
- 3 Select property as **Fluid**
- 4 Name the Mesh set **Fluid**
- 5 Click **Apply**



6-1 Fluid Boundary Condition

Procedure

- 1 Go to Mesh > Elements
- 2 Select Fluid Boundary
- 3 Select the top surface of sloshing fluid
- 4 Assign **free surface** property
- 5 Enter Mesh set name
- 6 Click Apply

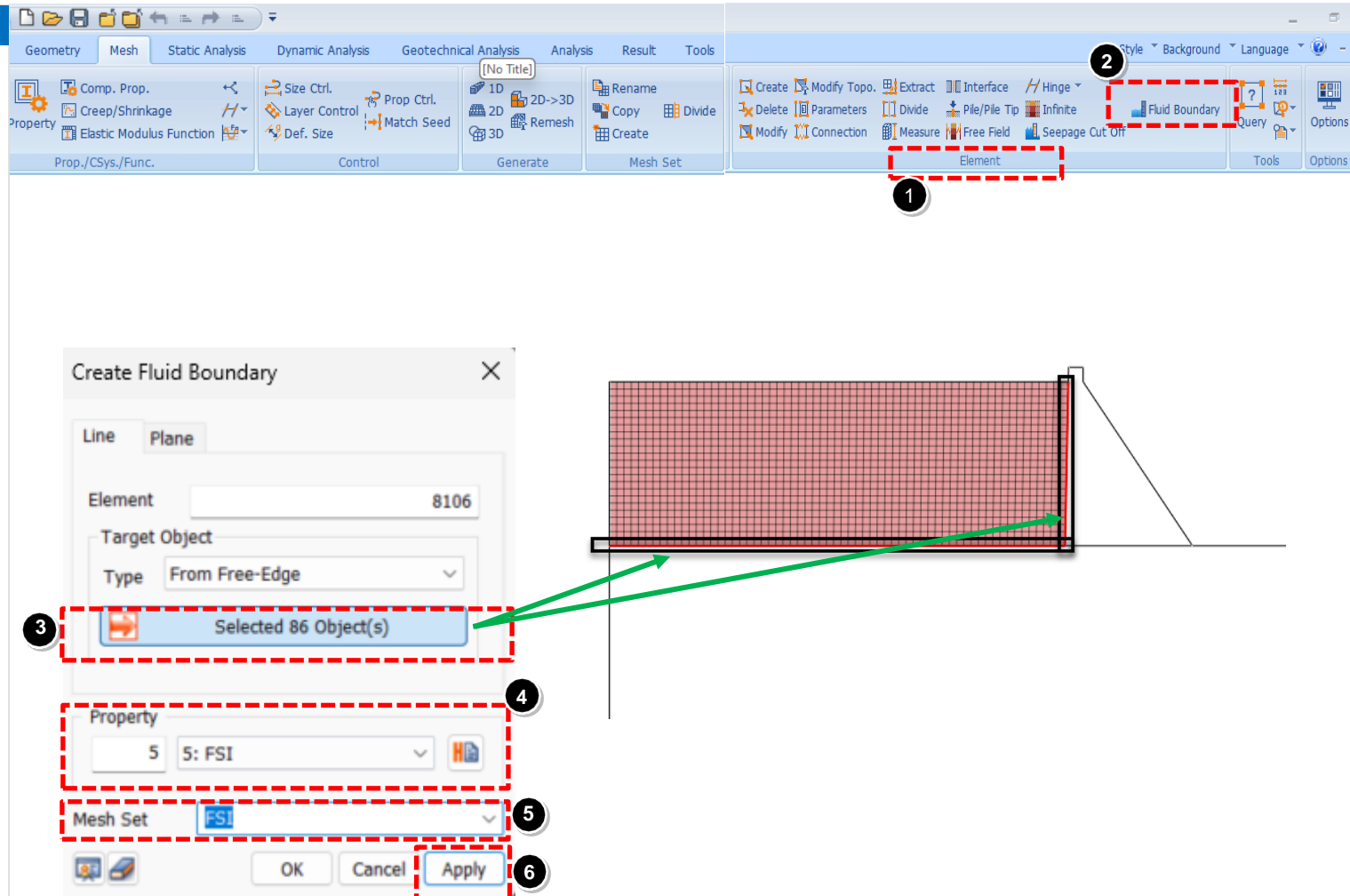


6-2 Fluid Boundary Condition

Procedure

Before moving further, we will hide rock mesh set from work-tree.

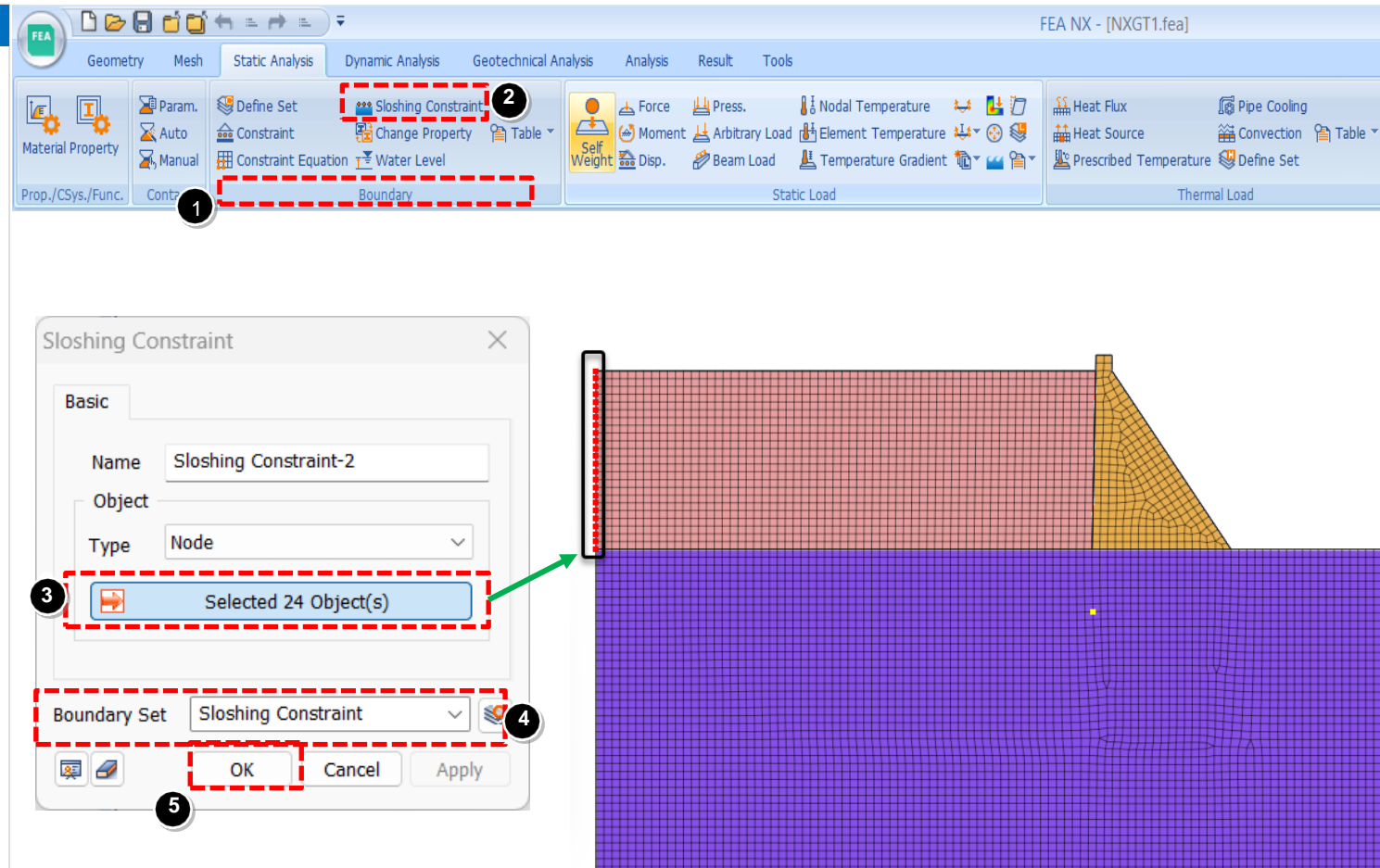
- ① Go to Mesh > Elements
- ② Select Fluid Boundary
- ③ Select the surface of reservoir (sloshing fluid)
- ④ Assign 'FSI' property
- ⑤ Enter Mesh set name as **FSI**
- ⑥ Click Apply



6-3 Fluid Boundary Condition

Procedure

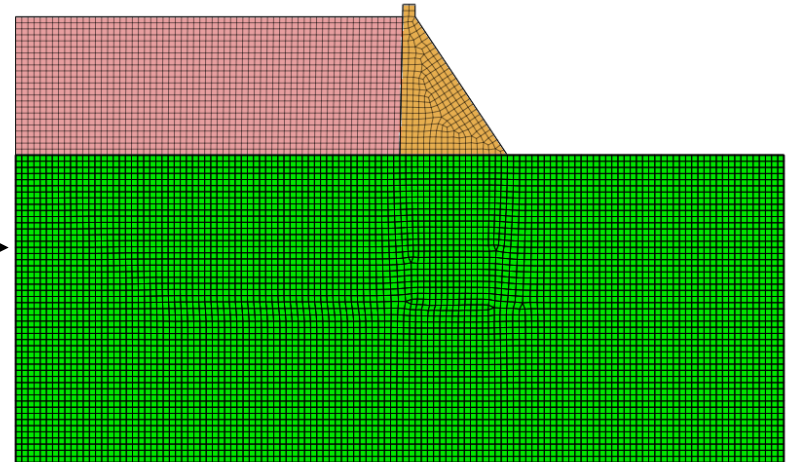
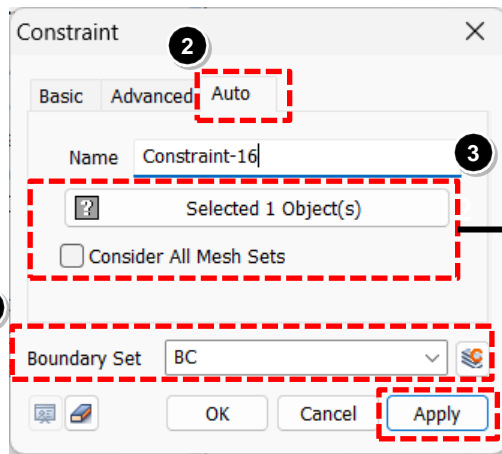
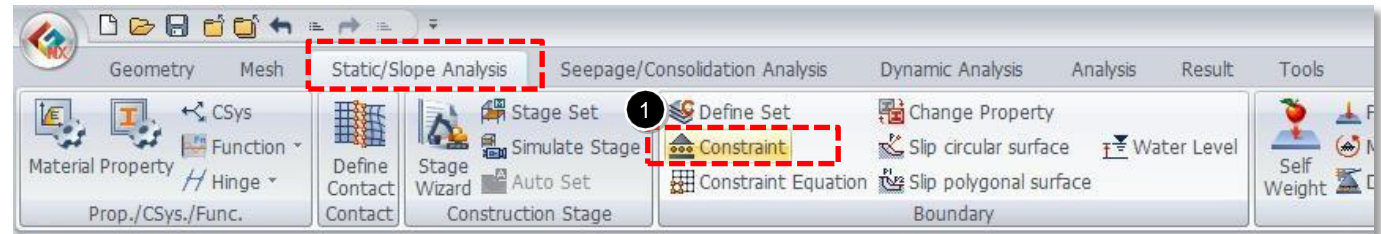
- 1 Go to Static/Slope Analysis > Boundary
- 2 Select Sloshing Constraint
- 3 Select the nodes as shown
- 4 Name the Boundary Set as **Sloshing Constraints**
- 5 Click **OK**



6-4 Boundary Condition

Procedure

- ❶ Go to Mesh > Constraint
- ❷ Select Auto
- ❸ Uncheck **Consider All Mesh Set** and **Select the mesh set of rock**
- ❹ Name the Boundary Set as **BC**
- ❺ Click **Apply**.

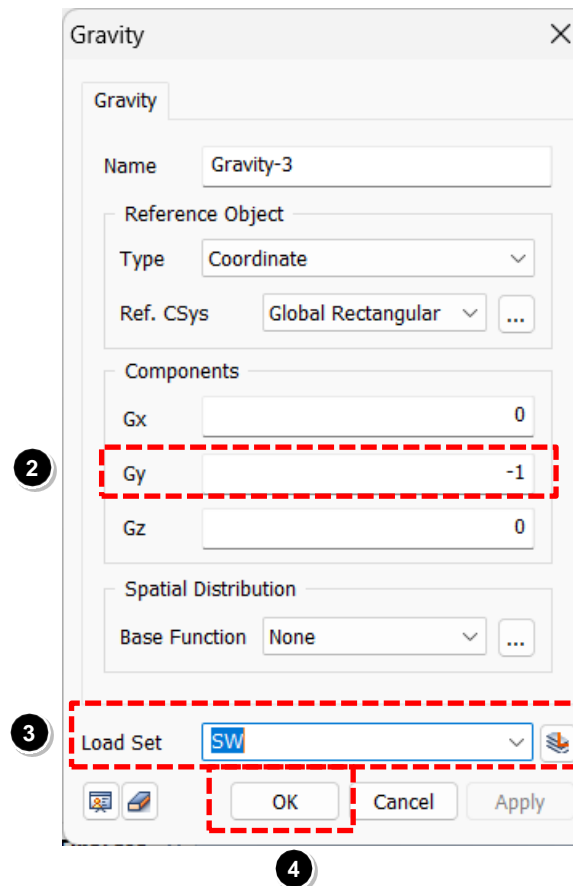
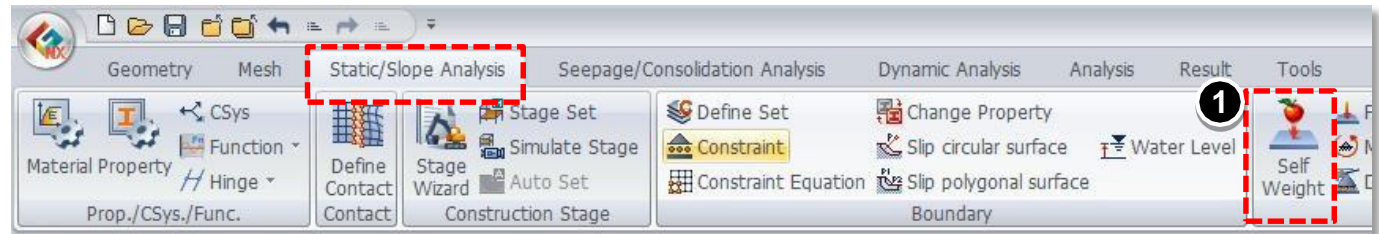


7-1 Load Definition – Self Weight

Procedure

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

From GTS NX 2024 version onwards, the Self-Weight Function is defined and present by default. So, if the 'Default Self-Weight' is present in the model, there is no need to perform this step.



7-2 Load Definition – Ground Acceleration

Procedure

- 1 Go to Dynamic Analysis > Load > Ground Acceleration
- 2 Check X Direction > Click on function symbol
- 3 Click on **Add Time Function**
- 4 Click on Earthquake
- 5 Select El Centro Earthquake
- 6 Click OK

The screenshot shows the FEA NX software interface with the following components:

- Menu Bar:** FEA, Geometry, Mesh, Static Analysis, **Dynamic Analysis** (highlighted with a red dashed box and circle 1), Geotechnical Analysis, Analysis, Result, Tools.
- Ribbon:**
 - Dynamic Analysis:** Material Property, Param. Auto, Manual, Define Set, Constraint, Constraint Equation, Sloshing Constraint, Transmitting, Table, **Ground Acceleration** (highlighted with a red dashed box and circle 1), Response Spectrum, Dynamic Nodal, Dynamic Surface, Load to Mass, Train Dynamic Load Table, Table.
 - Load:** (highlighted with a red dashed box and circle 1)
 - Tools:** Seismic Data Generator, Dynamic Load Data Generator, Free Field Analysis, Artificial Earthquake, Material Evaluator, Options.
- Ground Acceleration Dialog Box:**
 - Name: Ground Acceleration-1
 - ☒ **X Direction** (highlighted with a red dashed box and circle 2)
 - Function: None (Constant) (highlighted with a red dashed box and circle 2)
 - Scale Factor: 1
 - Arrival Time: 0 sec
 - ☐ Y Direction
 - ☐ Z Direction
 - Dynamic Load Set: Dynamic Load Set-1
 - Buttons: OK, Cancel, Apply
- Time Forcing Function Dialog Box:**
 - Buttons: **Add Time Function** (highlighted with a red dashed box and circle 3), Add Time Sinusoidal, Modify/Show, Delete.
- Generate Earthquake Acceleration Record Dialog Box:**
 - Earthquake: **1940, El Centro Site, 270 Deg** (highlighted with a red dashed box and circle 5)
 - Amplitude Scale: 1
 - Time Scale: 1
 - Text: 1940, El Centro Site, 270 Deg, Peak = 0.3569 g, Duration = 53.72 sec
 - Buttons: **OK** (highlighted with a red dashed box and circle 6), Cancel
- Time History Load Function Dialog Box:**
 - Name: (highlighted with a red dashed box and circle 4)
 - Time Function Data Type: Normalized Acceleration
 - Scaling: ☒ Scale Factor 1
 - Self Weight: 9.80665 m/sec²
 - Graph Option: ☐ X-axis Log Scale, ☐ Y-axis Log Scale
 - Baseline Correction: ☒ Original, ☐ Consider
 - Buttons: OK, Cancel, Apply

7-3 Load Definition – Ground Acceleration

Procedure

On Selecting OK

- 1 Time History Load function is generated. Rename it as shown. Select OK
- 2 We will see Time Forcing Function named “Elcent_h” added. Close the tab.
- 3 In Ground acceleration tab, Select function as ‘Elcent_h’
- 4 Click on OK

The screenshot displays the NX FEA software interface for defining ground acceleration. The top menu bar includes options like Geometry, Mesh, Static Analysis, Dynamic Analysis, Geotechnical Analysis, Analysis, Result, and Tools. Below the menu is a toolbar with icons for Material Property, Param. Auto, Manual, Define Set, Constraint, Constraint Equation, Sloshing Constraint, Transmitting, Table, Define Set, Response Spectrum, Ground Acceleration, Time Varying Static, Dynamic Nodal, Load to Mass, Train Dynamic Load Table, Table, Seismic Data Generator, Dynamic Load Data Generator, Free Field Analysis, Artificial Earthquake, Material Evaluator, and Options.

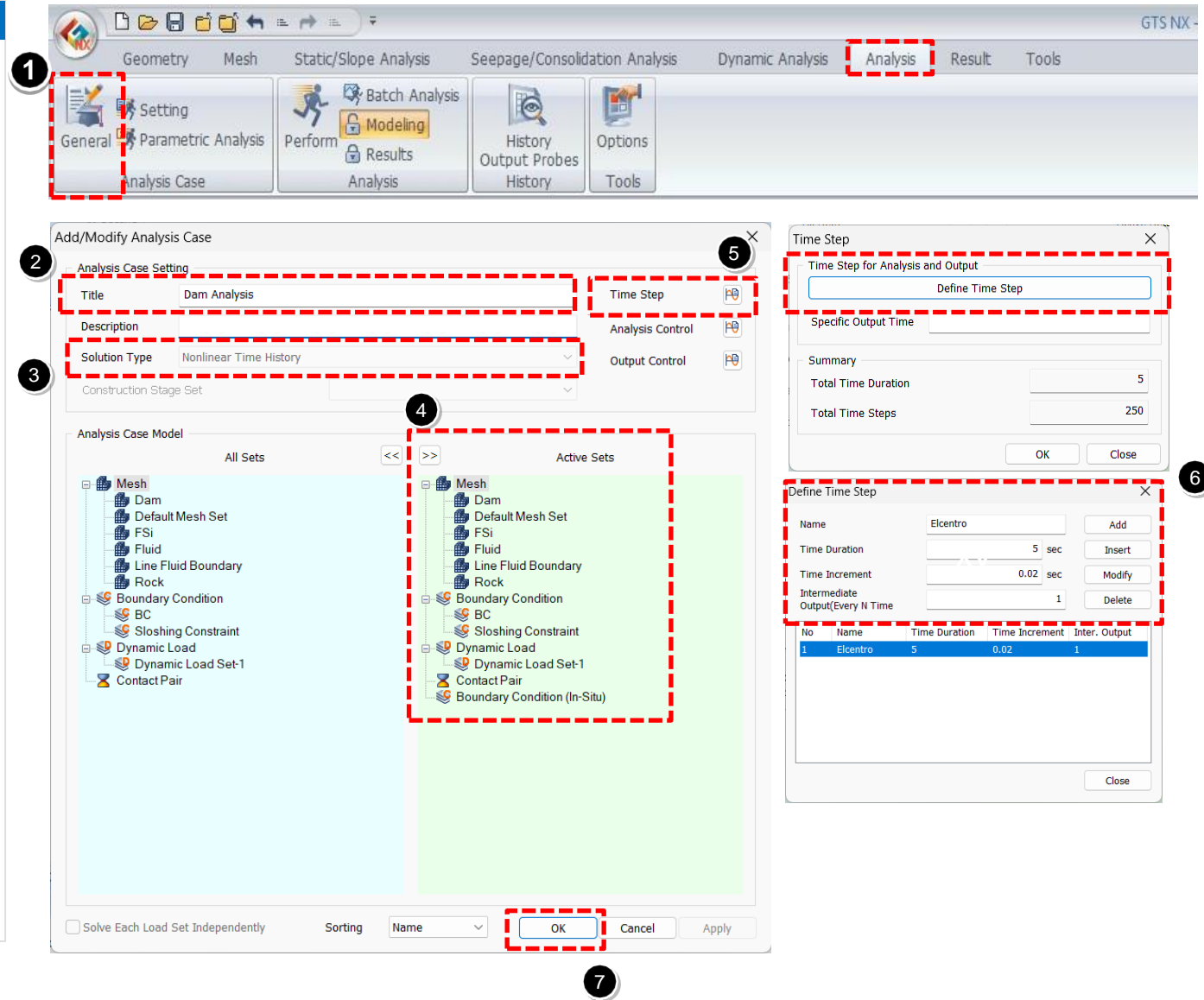
The main workspace shows the 'Time History Load Function' dialog box. It has a 'Name' field set to 'Elcent_h' and a 'Time Function Data Type' set to 'Normalized Acceleration'. The 'Scaling' section has 'Scale Factor' set to 1 and 'Self Weight' set to 9.80665 m/sec². The 'Graph Option' section has 'X-axis Log Scale' and 'Y-axis Log Scale' both unchecked. A table of 'Time (sec)' and 'Value (g)' data is shown, with a graph of the time history. The 'Baseline Correction' section has 'Original' selected. The 'Description' field is set to '1940, El Centro Site, 270 Deg'. The 'OK', 'Cancel', and 'Apply' buttons are at the bottom.

The 'Time Forcing Function' dialog box is also shown. It has a table with columns 'Name', 'Type', and 'Data Type'. The first row is 'Elcent_h', 'Time Function', and 'Norm.Acc.'. The 'Add Time Function', 'Add Time Sinusoidal', 'Modify/Show', 'Delete', and 'Close' buttons are on the right.

The 'Ground Acceleration' dialog box is shown with 'Name' set to 'Ground Acceleration-1'. The 'X Direction' is selected, and the 'Function' is set to 'Elcent_h'. The 'Scale Factor' is 1 and 'Arrival Time' is 0 sec. The 'Y Direction' and 'Z Direction' are unchecked, with 'Function' set to 'None (Constant)', 'Scale Factor' set to 1, and 'Arrival Time' set to 0 sec. The 'Dynamic Load Set' is set to 'Dynamic Load Set-1'. The 'OK', 'Cancel', and 'Apply' buttons are at the bottom.

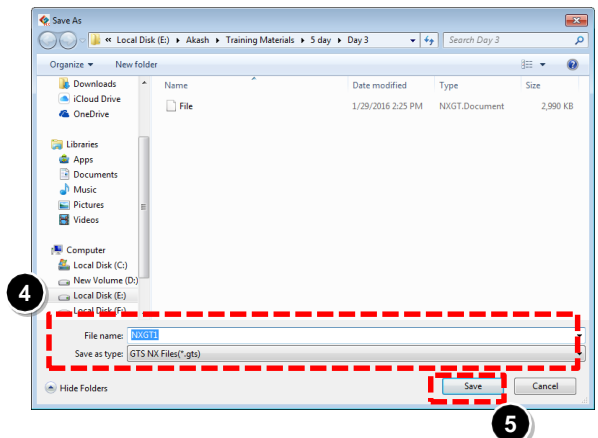
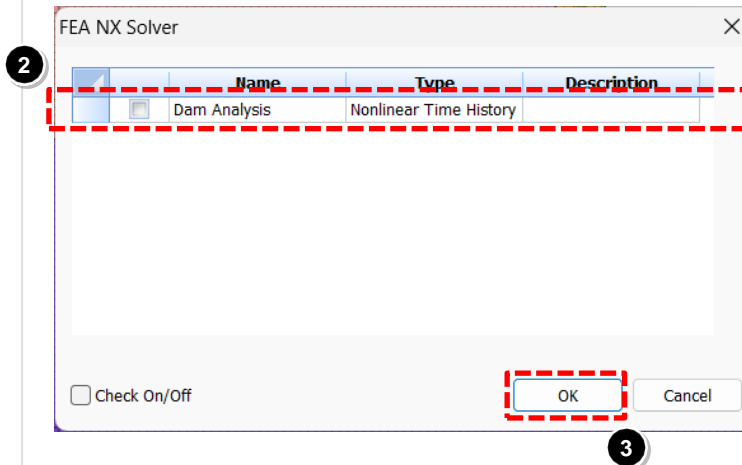
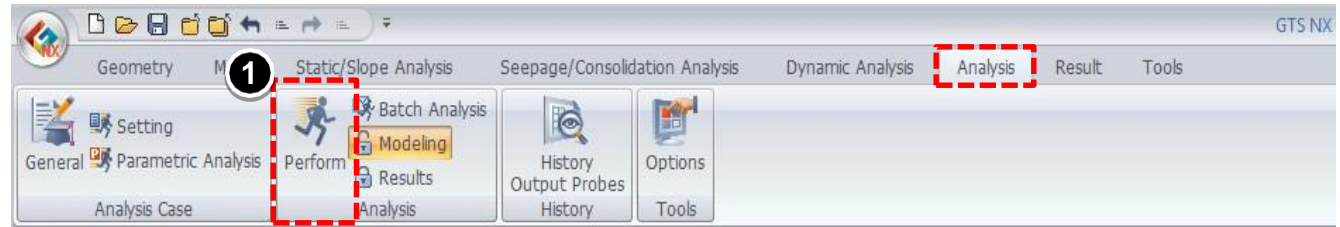
Procedure

- 1 Go to Analysis > General
- 2 Give the name as Dam Analysis
- 3 Select Solution Type as **Nonlinear Time History**
- 4 Activate the mesh set as shown.
- 5 Click on **Time Step**. Define Time Step.
- 6 Enter the details as shown and add the time step.
- 7 Click **Ok**



Procedure

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



10-1 Results > Sloshing Fluid Pressure

Procedure

- 1 Go to works tree and enable mesh to check the results.
- 2 Go to Works Results > Incr. 105

We can find that the fluid pressure is following the parabolic path as proposed by westergaard in Increment 105 etc.,

The screenshot displays the software's Results tree on the left and a 2D pressure plot on the right.

Results Tree:

- Item
- ID
- Color
- INCR=93 (TIME...)
- INCR=94 (TIME...)
- INCR=95 (TIME...)
- INCR=96 (TIME...)
- INCR=97 (TIME...)
- INCR=98 (TIME...)
- INCR=99 (TIME...)
- INCR=100 (TIME...)
- INCR=101 (TIME...)
- INCR=102 (TIME...)
- INCR=103 (TIME...)
- INCR=104 (TIME...)
- INCR=105 (TIME...)
- Displacement.
- Relative Displ.
- Grid Forces
- Reactions
- Velocities
- Relative Velo...
- Accelerations
- Relative Acce..
- Plane Strain F..
- Fluid Bounda...
- Plane Strain S...
- Sloshing Flui...
- 2D PRESS...
- 2D COMP...
- Plane Strain S..
- INCR=106 (TIME...)
- INCR=107 (TIME...)
- INCR=108 (TIME...)
- INCR=109 (TIME...)
- INCR=110 (TIME...)

The item **INCR=105 (TIME...)** is highlighted with a red dashed box and a circled '2'.

2D Pressure Plot:

The plot shows a cross-section of a structure with a sloshing fluid. The fluid pressure is represented by a color gradient from blue (low pressure) to red (high pressure). The pressure distribution is parabolic, with the highest pressure (red) at the top of the fluid and decreasing towards the bottom. The plot is titled "SLOSHING FLUID 2D PRESSURE, kN/m²".

Color Scale:

| Pressure (kN/m ²) |
|-------------------------------|
| 3.1% |
| 2.9% |
| 2.8% |
| 2.9% |
| 3.2% |
| 3.4% |
| 3.5% |
| 4.3% |
| 5.2% |
| 6.2% |
| 7.0% |
| 10.7% |
| 50.6% |
| -0.33937 |

Max: 276.754

Min: -0.33937

[UNIT] kN, m, [Output Csys] Default

Happy Modelling

