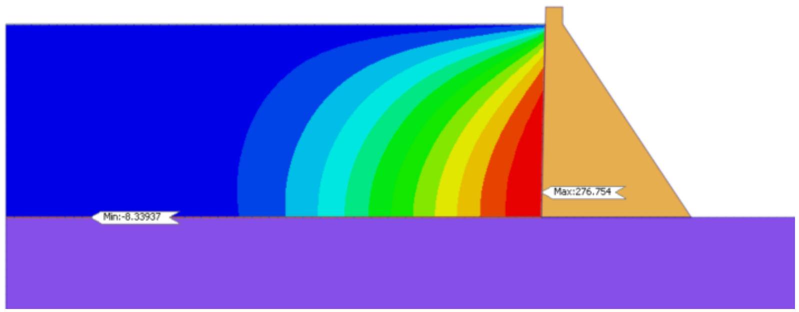
MIDAS *Technical Material*

Tutorial



Sloshing Fluid Element

Contents

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 appliable in Eigenvalue Analysis and Time History Analysis.

Sloshing Fluid

Material Properties

Name	Material Model	Modulus of Elasticity (kN/m^2)	Bulk Modulus (kN/m^2)	Poisson's ratio	Unit Weight (kN/m^3)
Concrete (M25)	**Elastic**	22410000	-	0.2	24.35
Foundation (Rock)	**Elastic**	22400000	-	0.2	0 (Massless foundation)
Reservoir Water	**Sloshing Medium**	-	2200000	-	9.80665

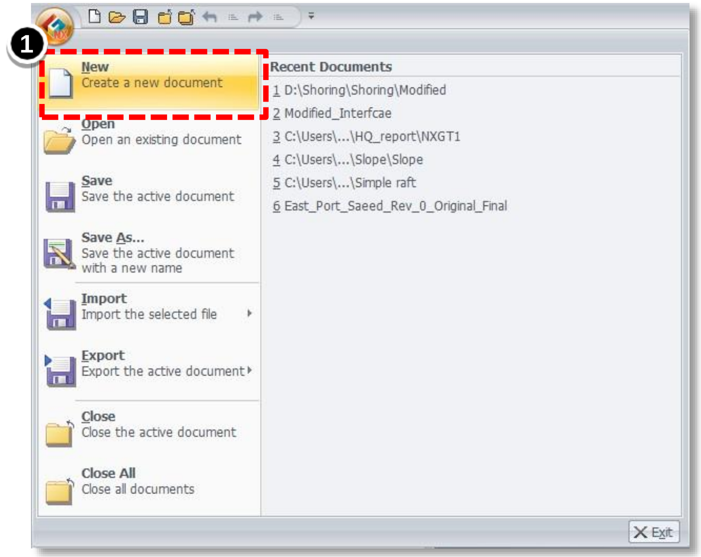
Step 01

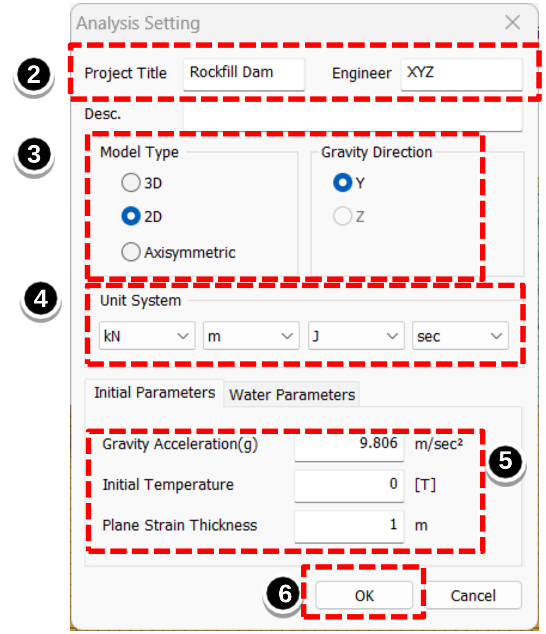
1-1 Initial Setting

Procedure

Starting Midas GTS NX

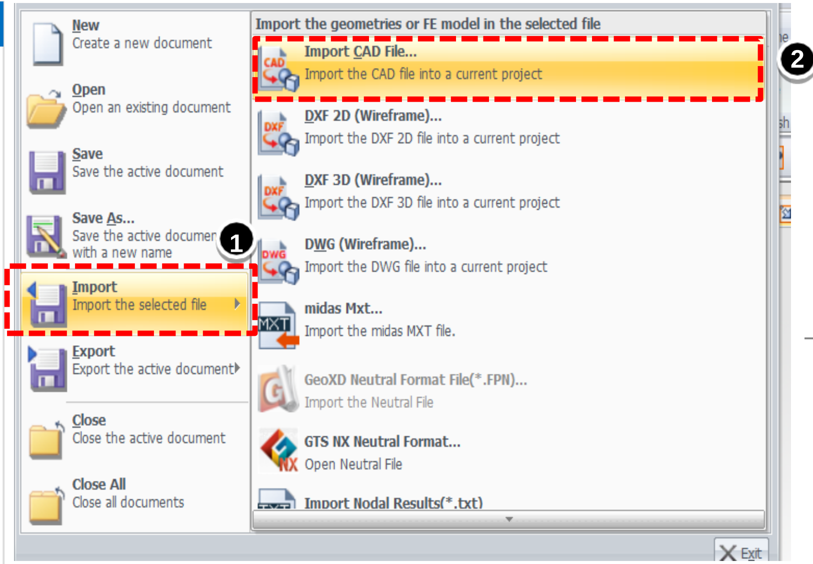
1. Click on GTS NX icon > New Project
2. Enter the Project name as Sloshing Fluid
3. **Select Model Type as 2D and Gravity Direction as Y**
4. **Select kN ,m and sec in The Unit System**
5. **Use the Default values for Initial parameters**
6. Click **OK**

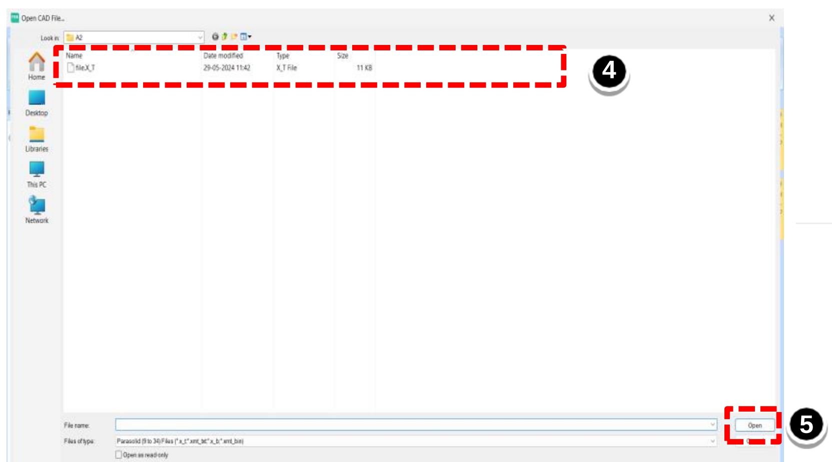


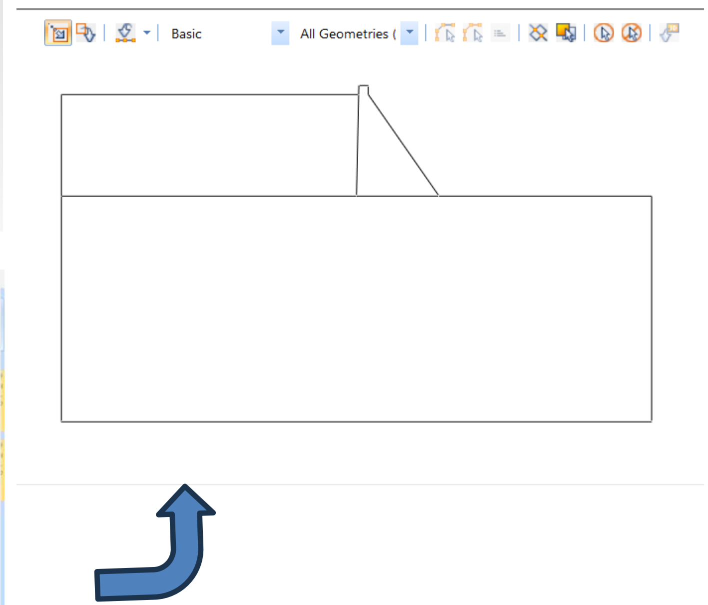


Procedure

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







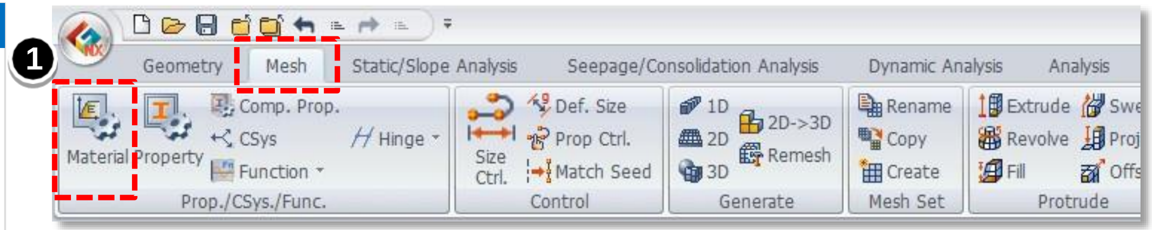
Step
03

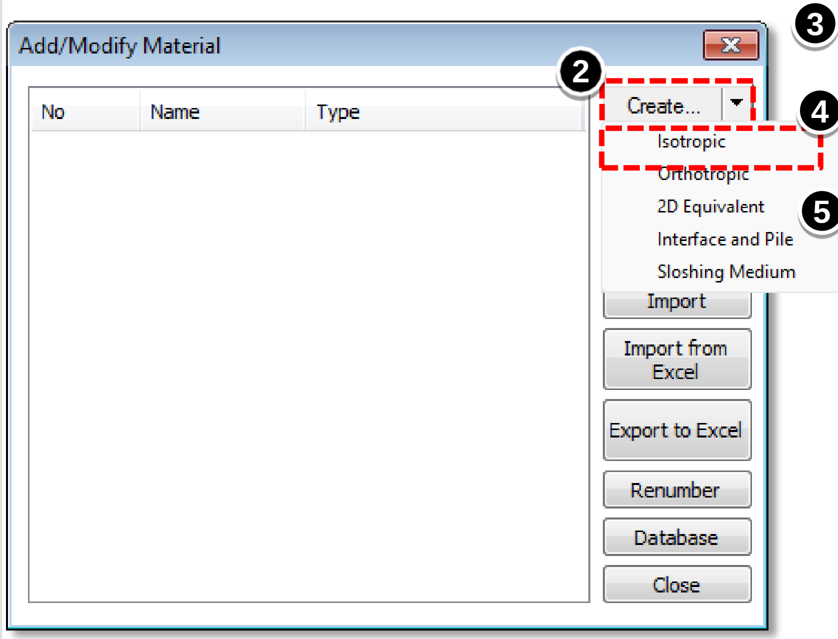
3-1 Material Definition

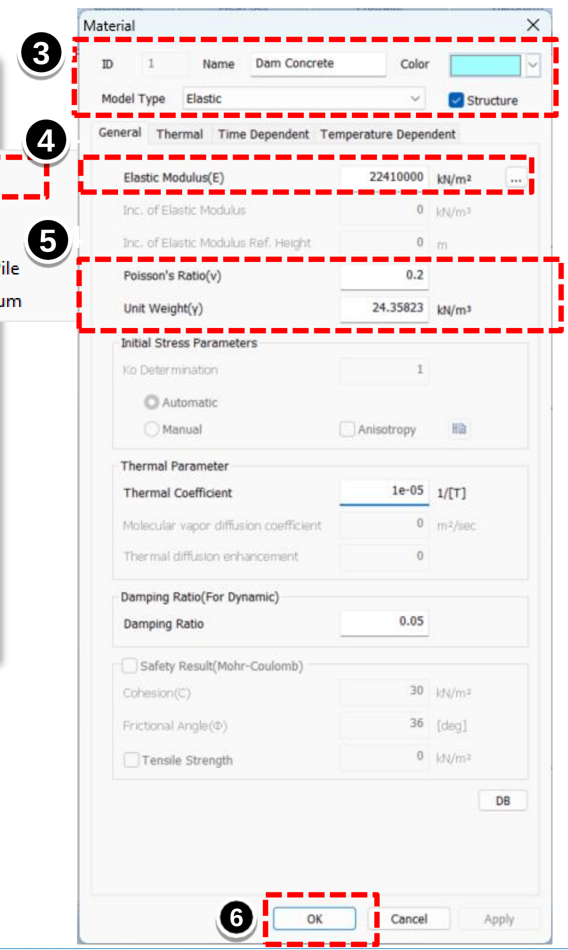
Procedure

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'.





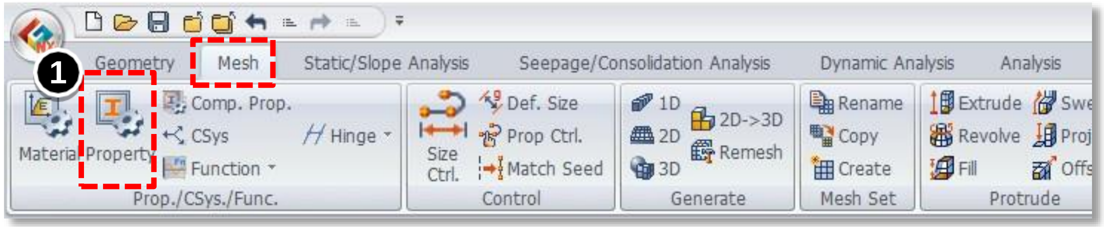


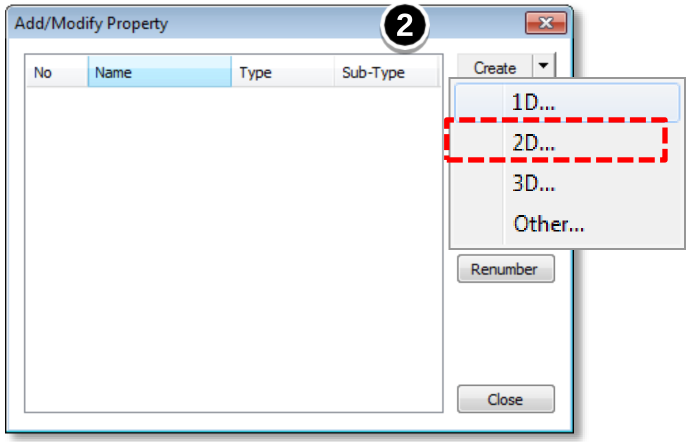
4-1 Property Definition

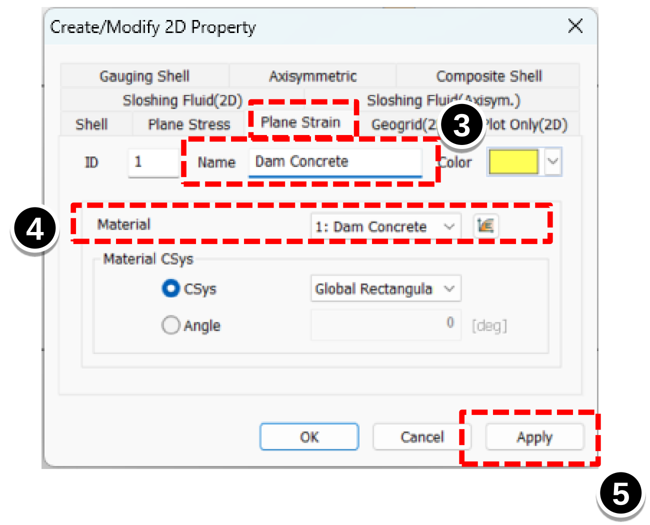
Procedure

1. Go to Mesh>Click on Property
2. Click on Create. Select **2D**
3. Select Plane Strain Element and name it as **Dam Concrete**
4. Select **Dam Concrete** from Material drop down
5. Click **Apply**
6. Similarly Define the property for **Rock and Sloshing Fluid**

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





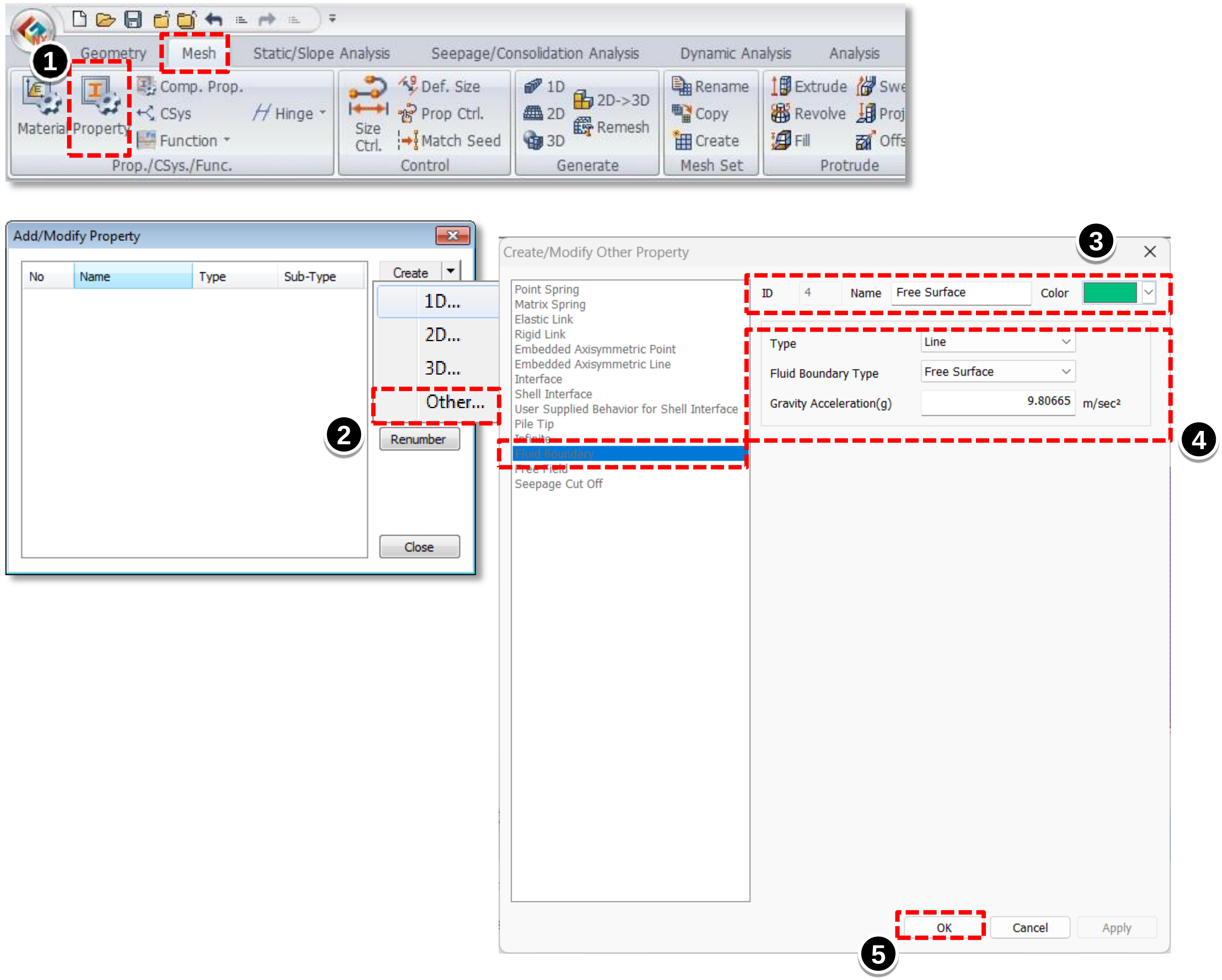


Step 04

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**

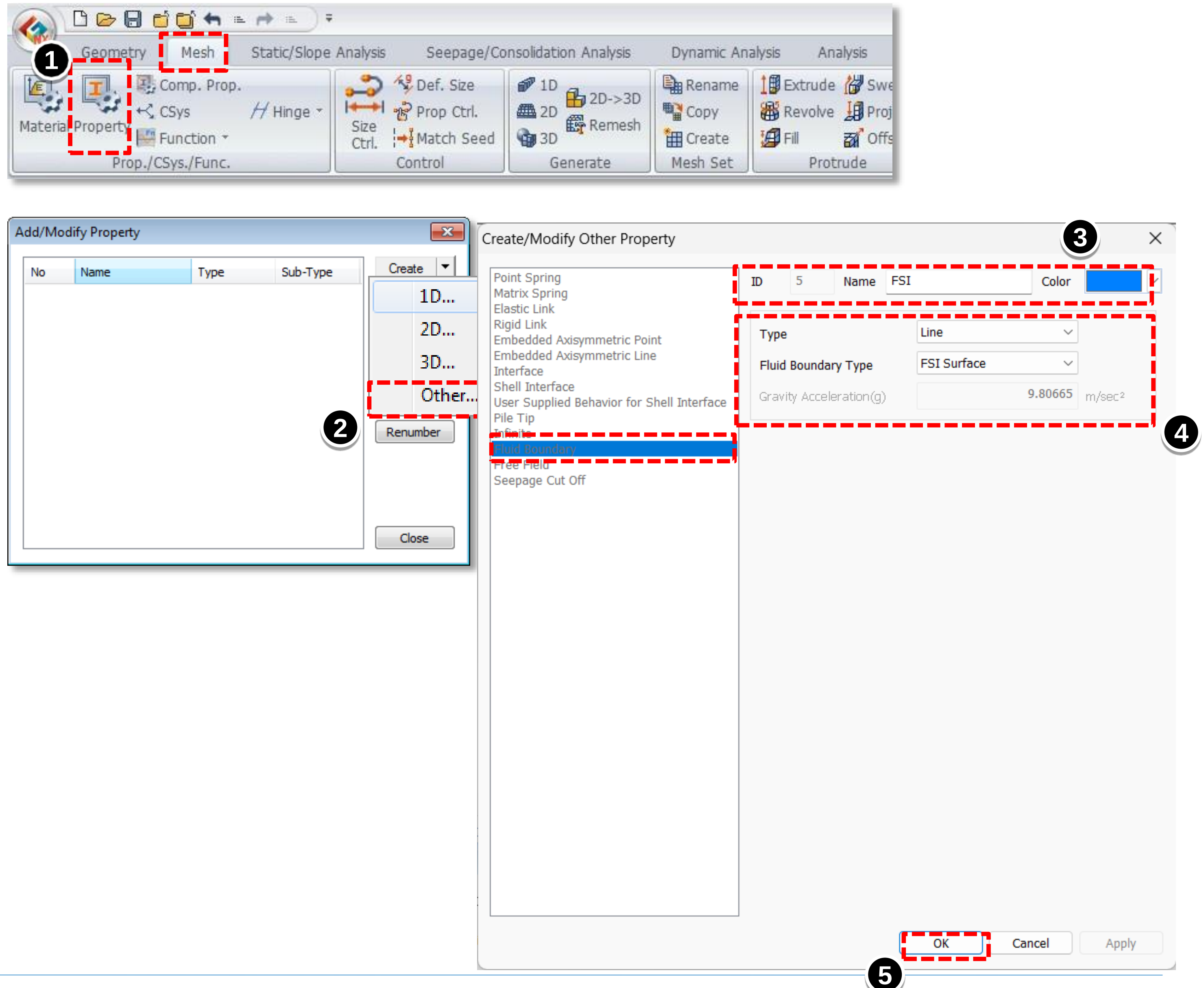


Step
04

4-3 Fluid Boundary Property

Procedure

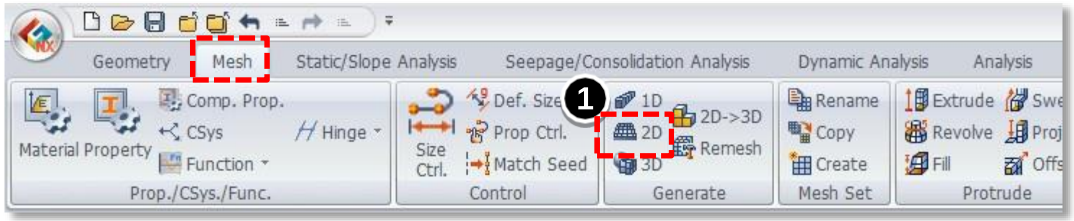
1. Go to Mesh > Click on Property
2. Click on Create. Select **Others**
3. Name it as **FSI**
4. Select **Fluid Boundary type as FSI Surface** from Material drop down
5. Click **Apply**

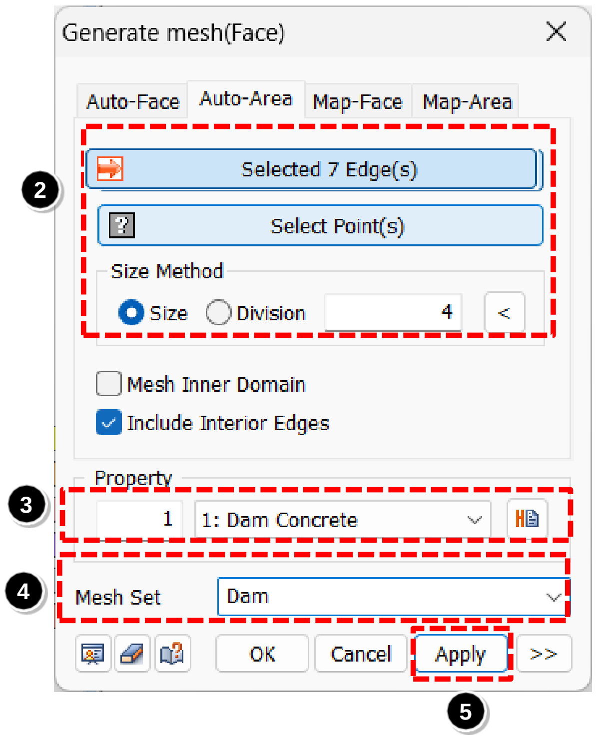


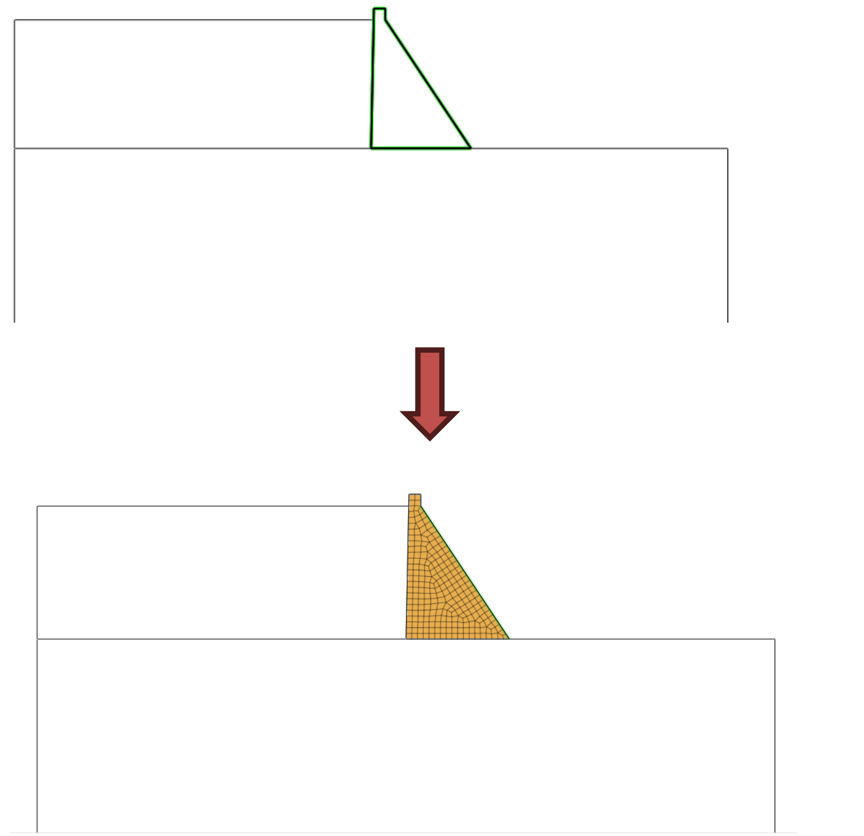
5-1 Meshing

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**



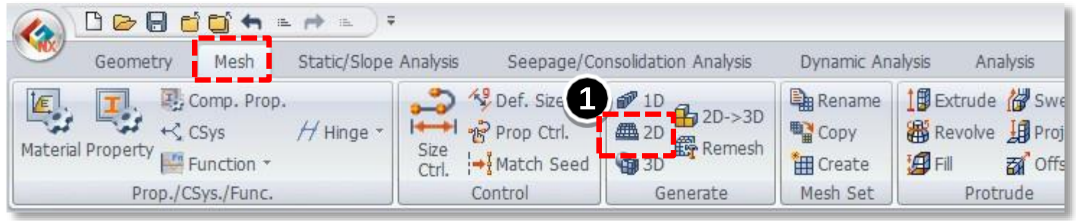


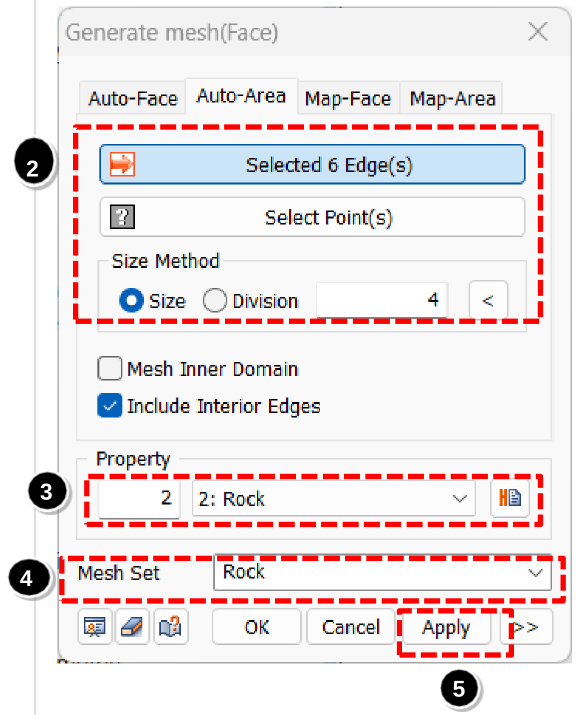


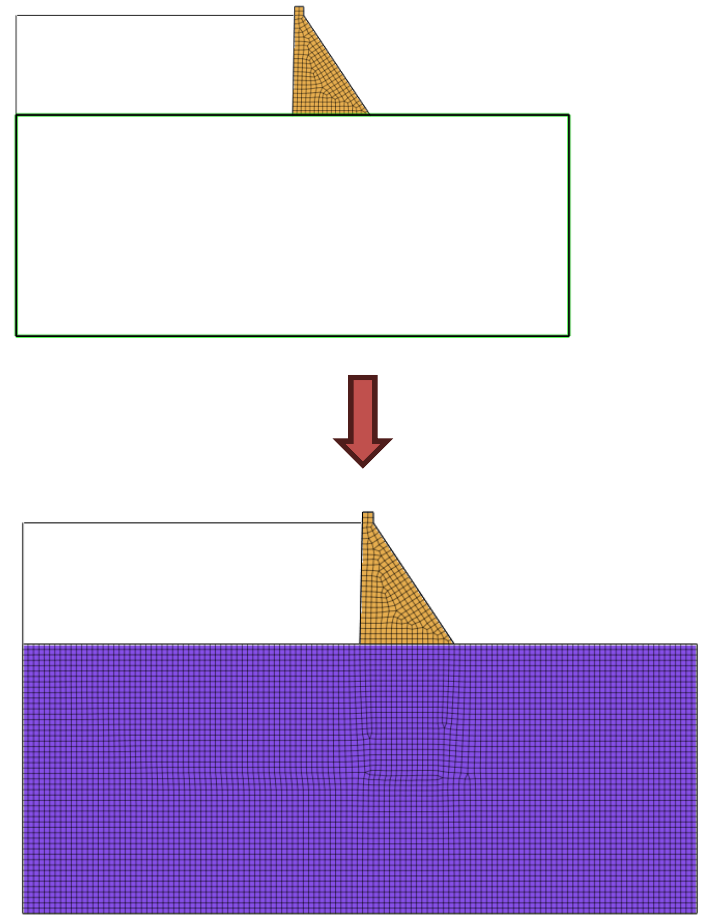
5-2 Meshing

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**



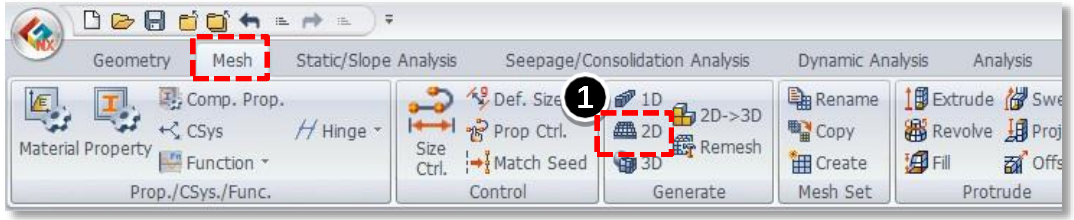


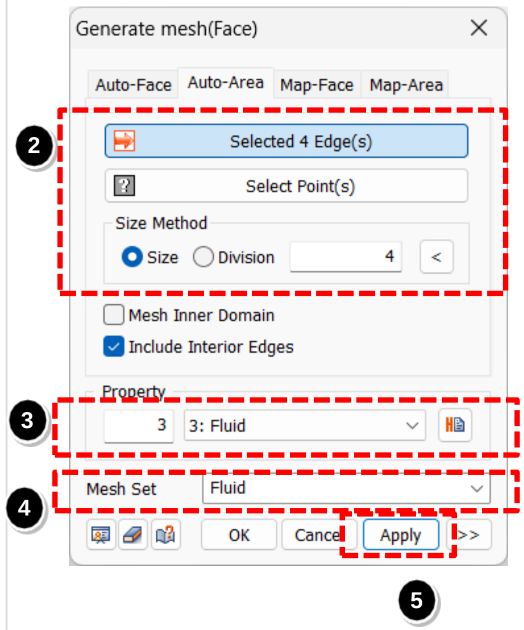


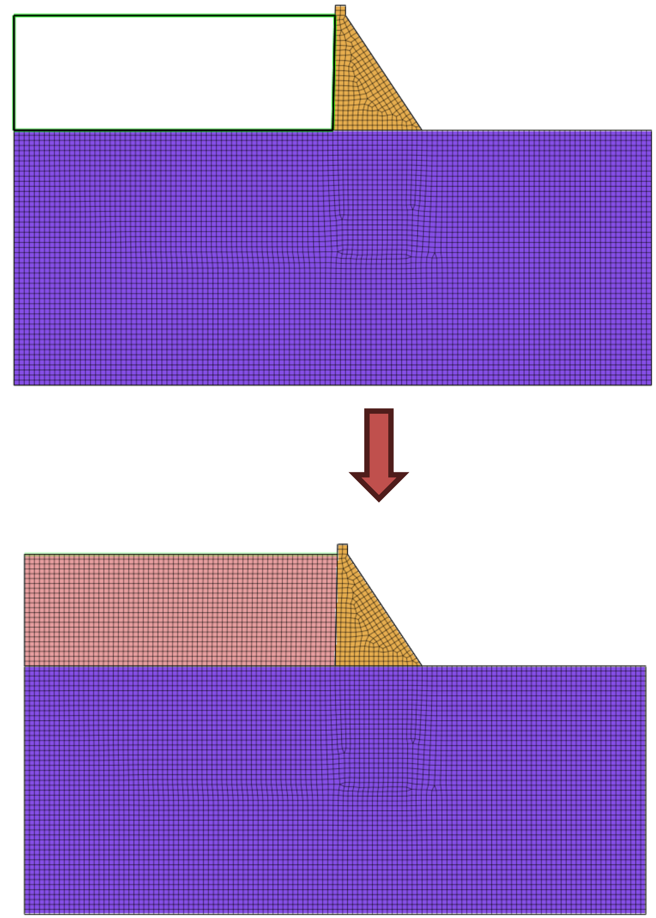
5-3 Meshing

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**





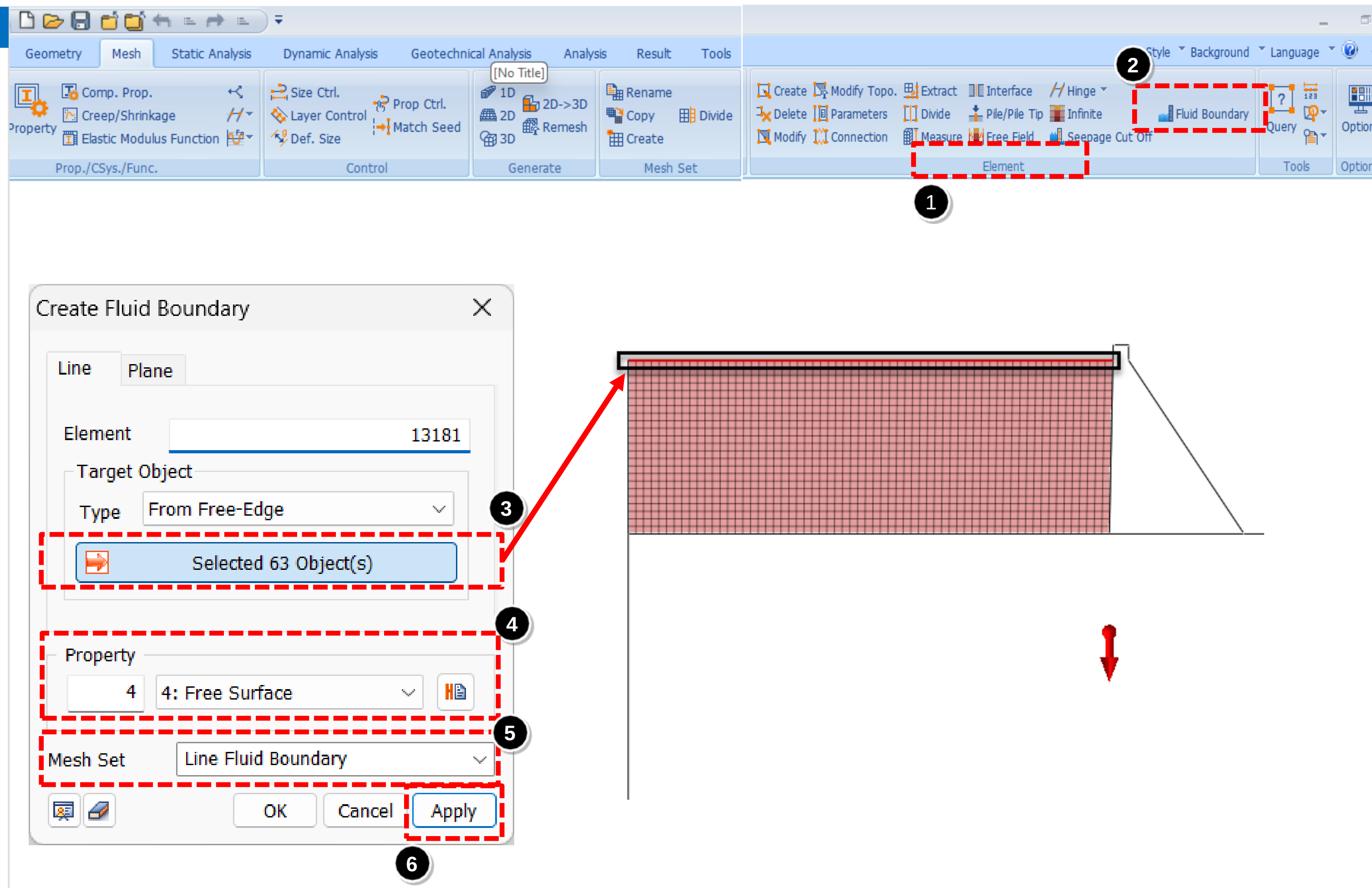


Step
06

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



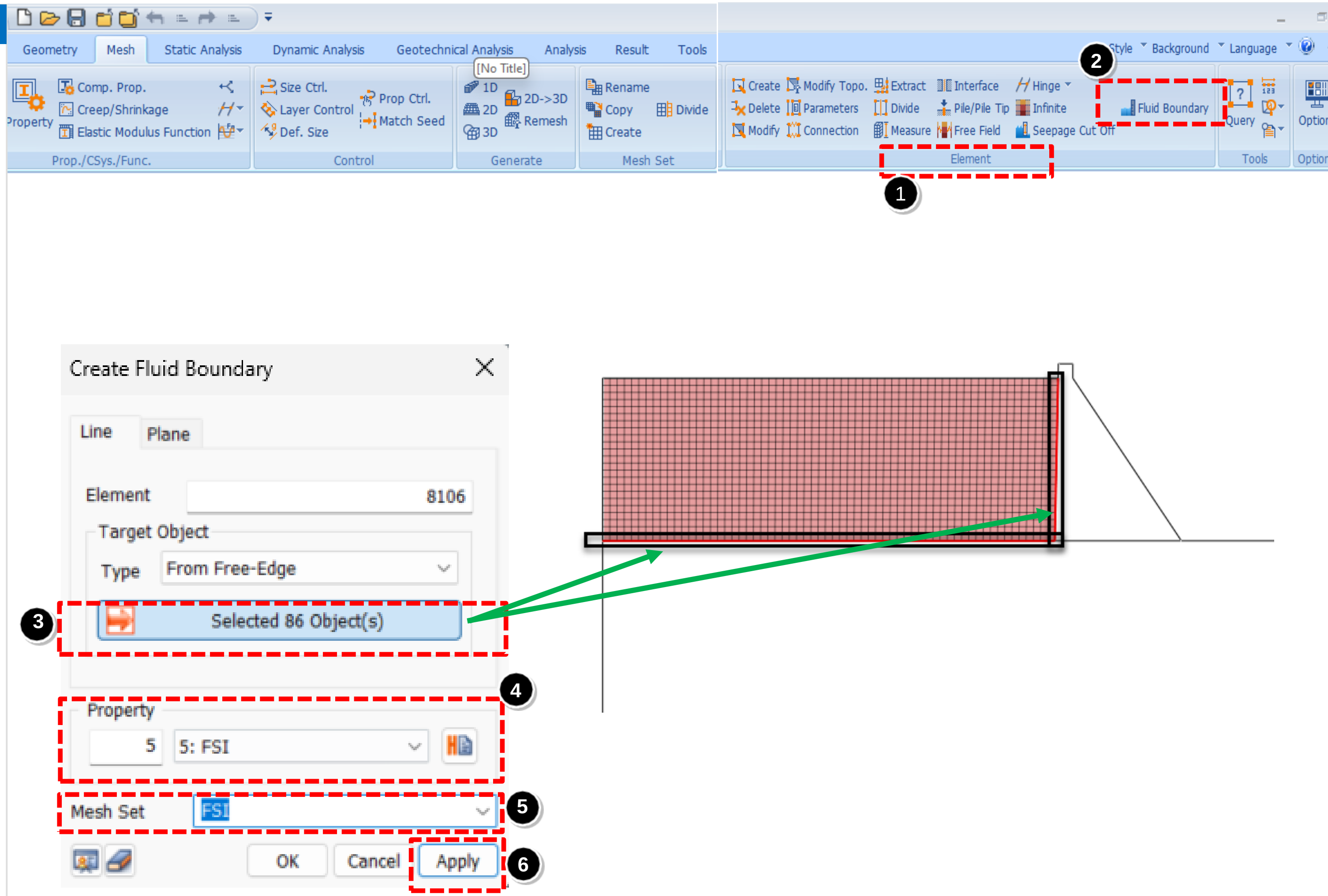
Step **06**

6-2 Fluid Boundary Condition

Procedure

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

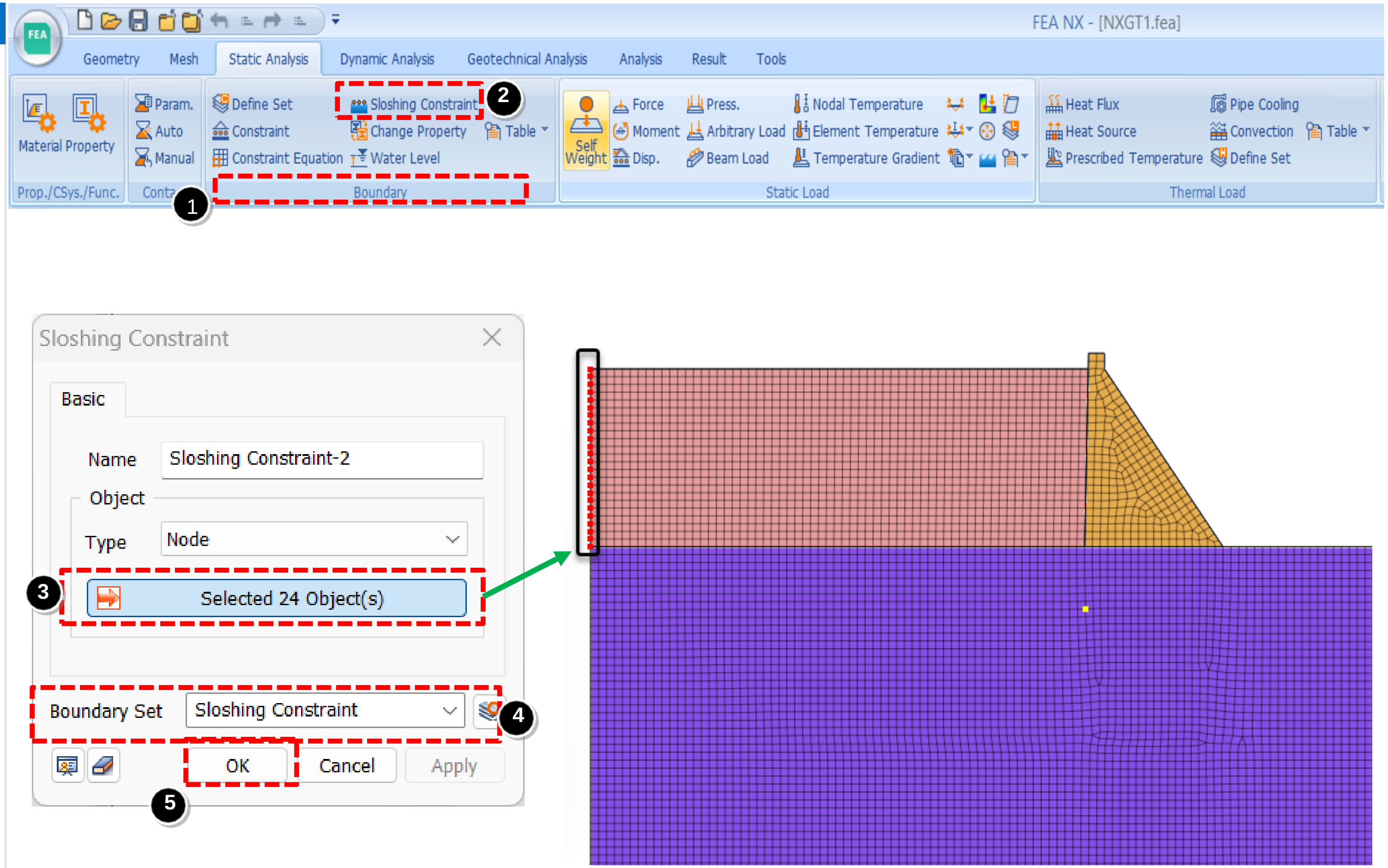
1. Go to Mesh > Elements
2. Select Fluid Boundary
3. Select the surface of reservoir (sloshing fluid)
4. Assign '**FSI**' property
5. Enter Mesh set name as **FSI**
6. Click Apply



6-3 Fluid Boundary Condition

Procedure

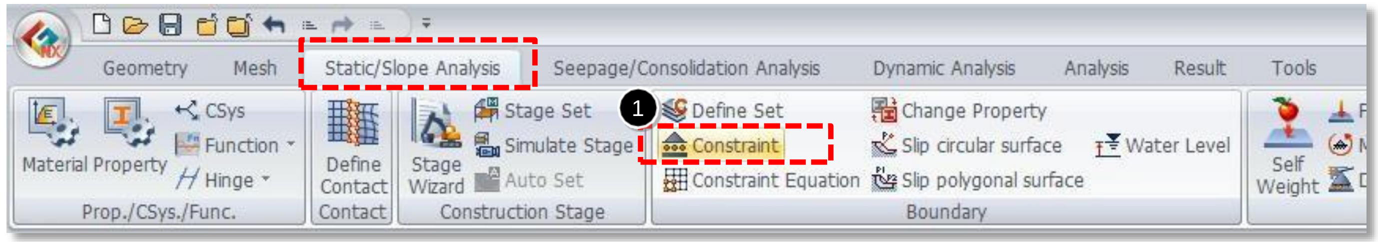
1. Go to Static/Slope Analysis > Boundary
2. Select Sloshing Constraint
3. Select the nodes as shown
4. Name the Boundary Set as **Sloshing Constraints**
5. Click **OK**
.

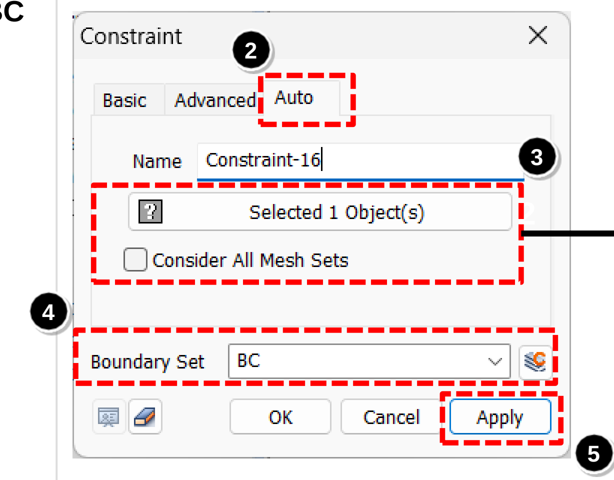


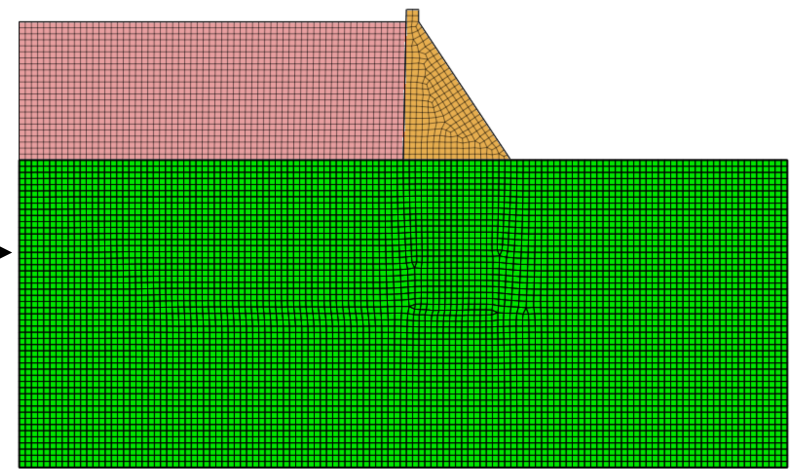
6-4 Boundary Condition

Procedure

1. Go to Mesh > Constraint
2. Select Auto
3. Uncheck **Consider All Mesh Set** and **Select the mesh set of rock**
4. Name the Boundary Set as **BC**
5. **Click Apply**.







Step **07**

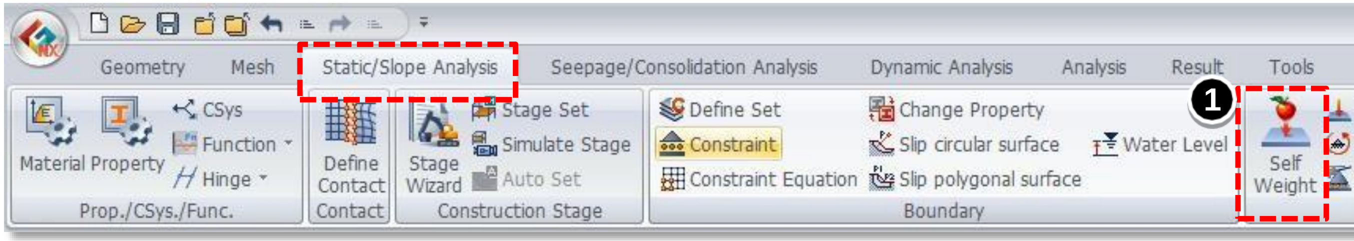
7-1 Load Definition – Self Weight

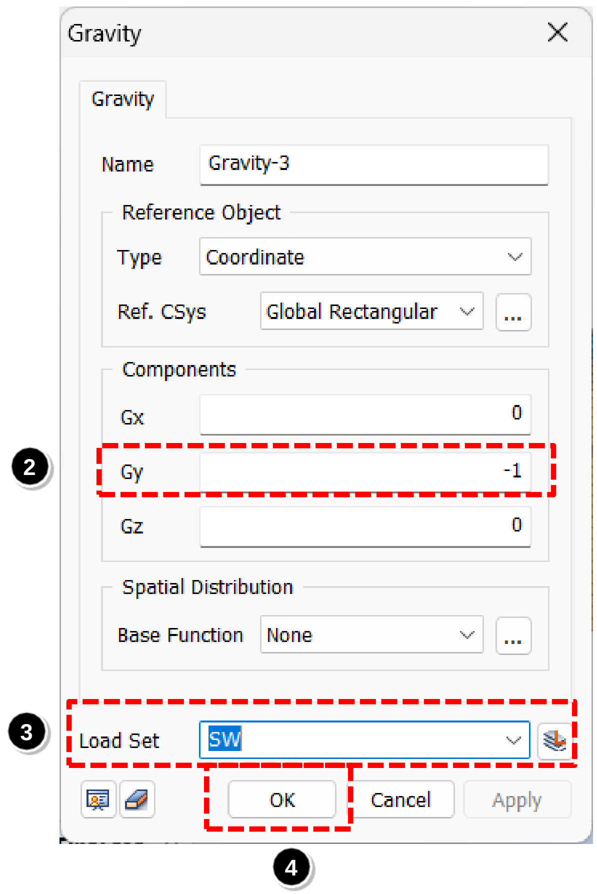
Procedure

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.



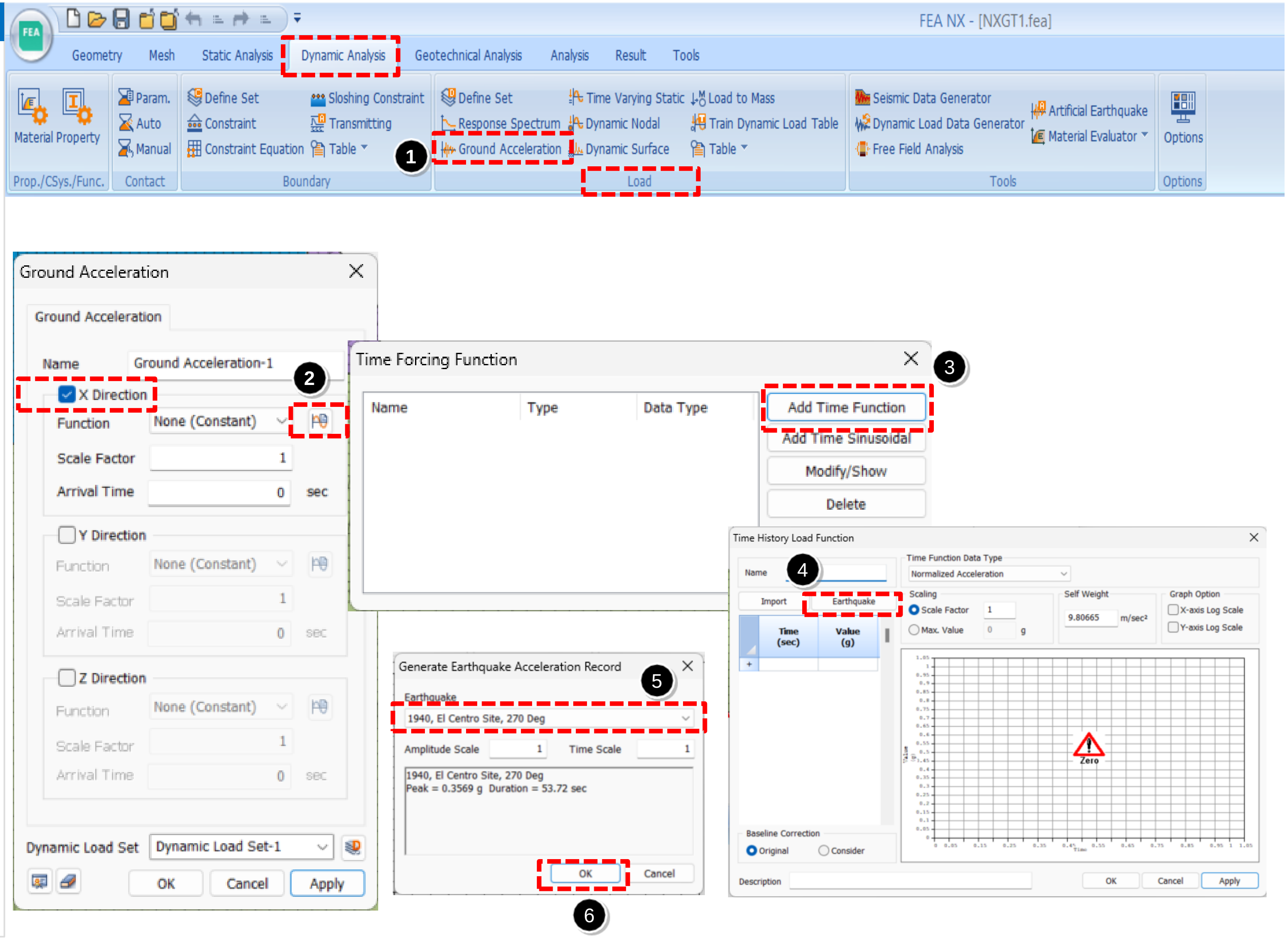


Step 07

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

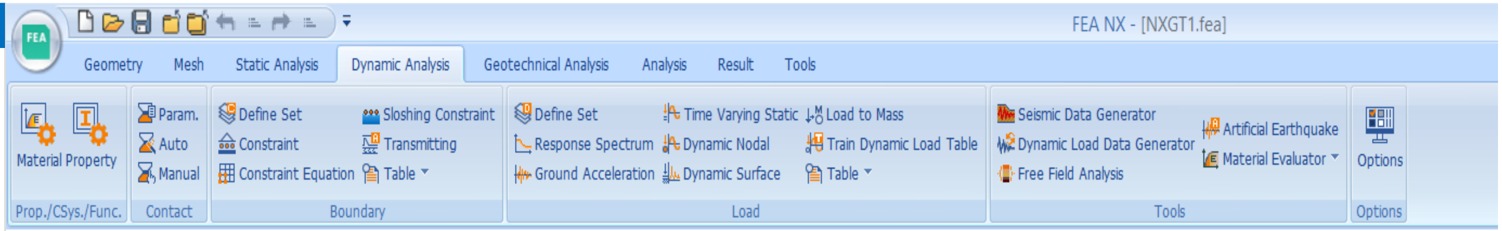


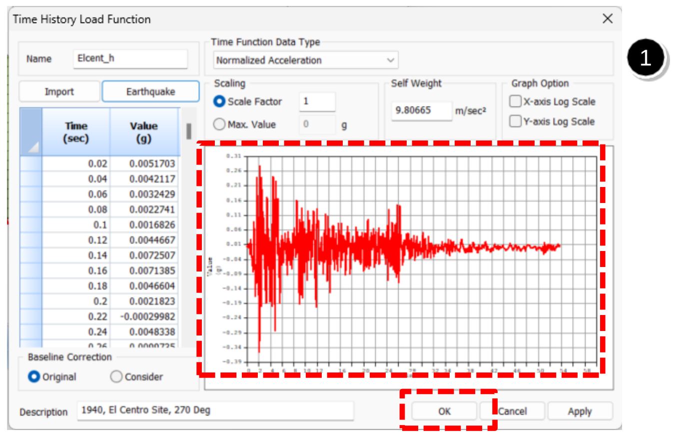
7-3 Load Definition – Ground Acceleration

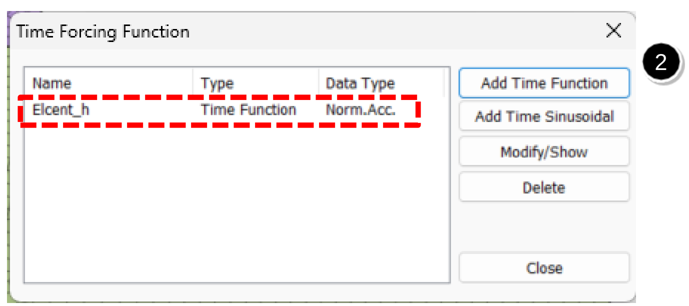
Procedure

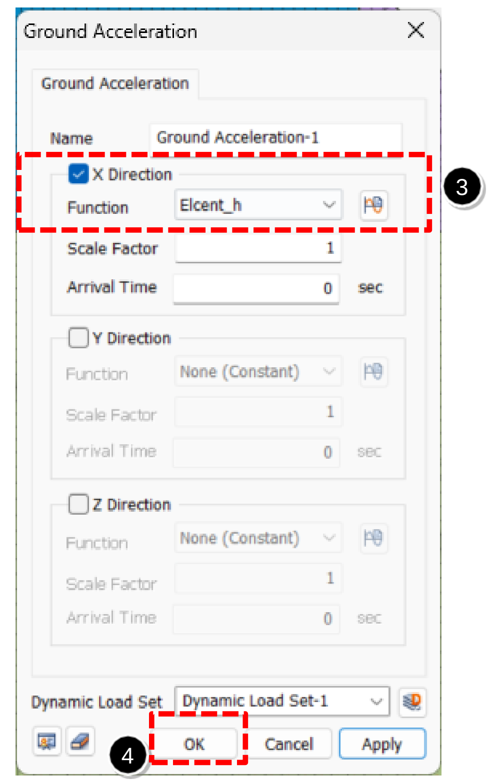
On Selecting OK

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







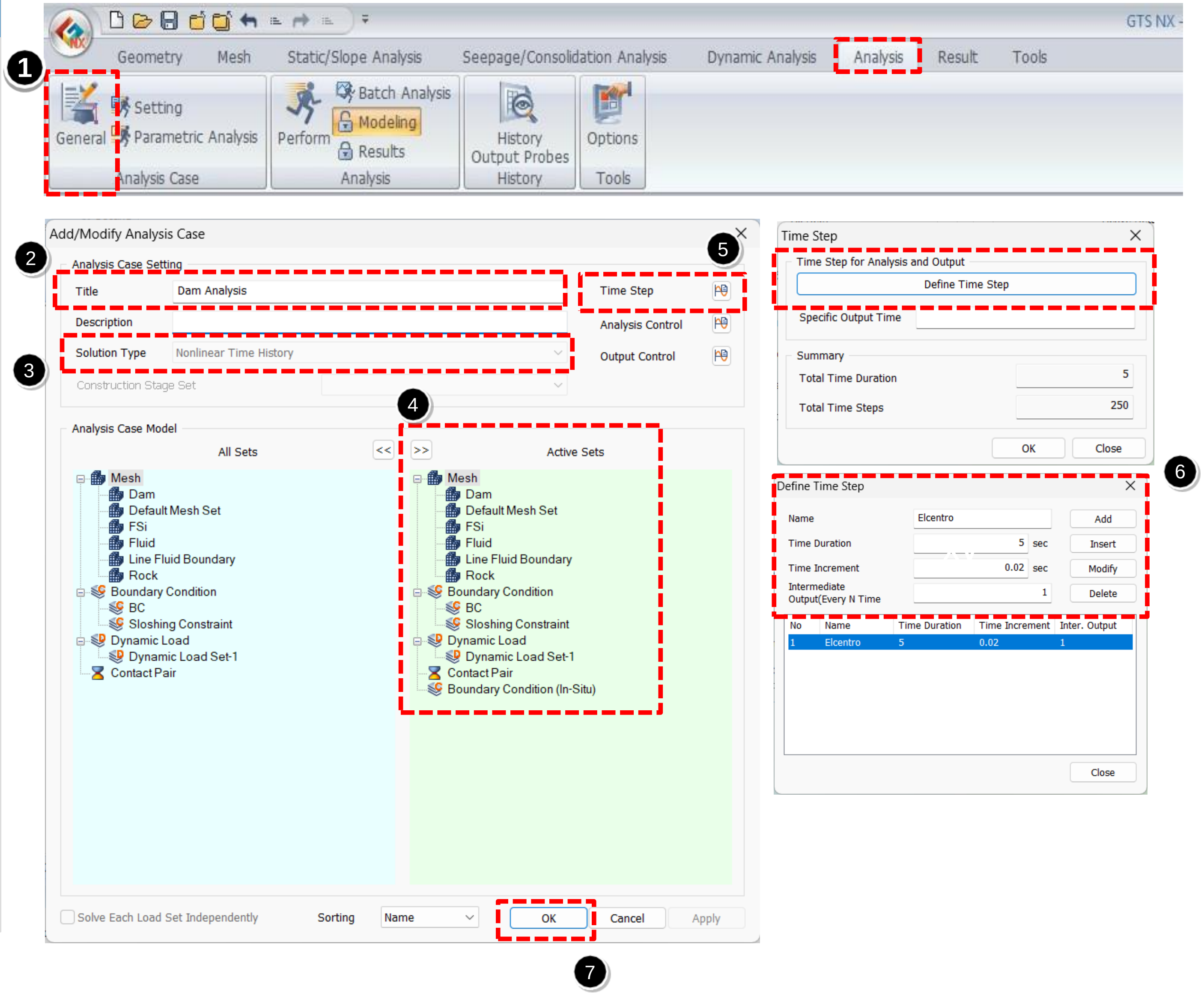


Step 08

8-1 Analysis case

Procedure

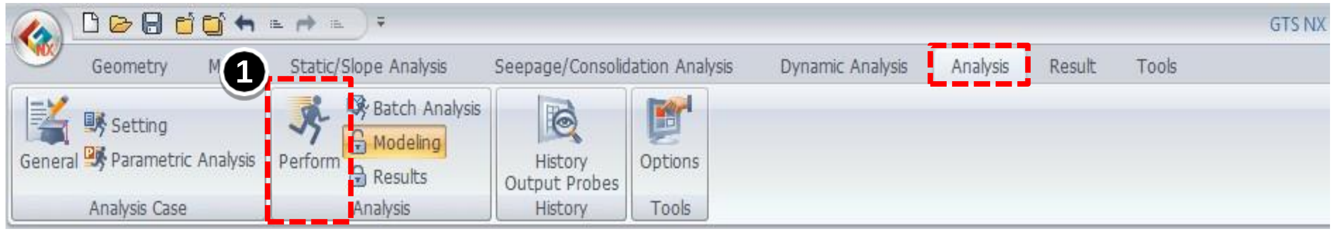
1. Go to Analysis > General
2. Give the name as Dam Analysis
3. Select Solution Type as **Nonlinear Time History**
4. Activate the mesh set as shown.
5. Click on **Time Step.** Define Time Step**.**
6. **Enter the details as shown and add the time step.**
7. Click **Ok**

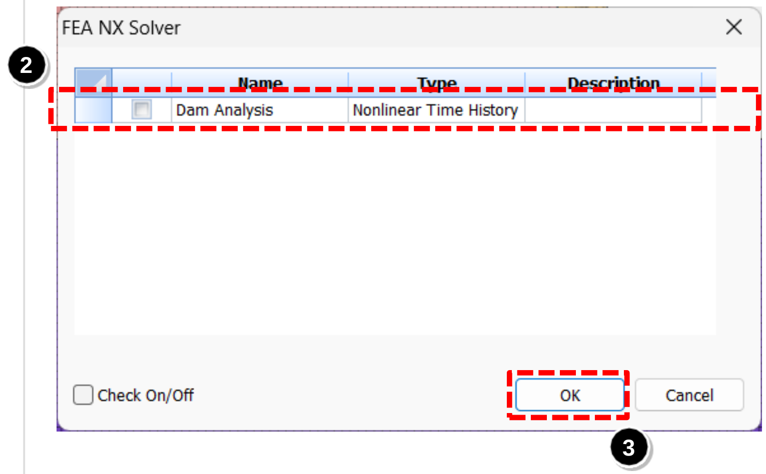


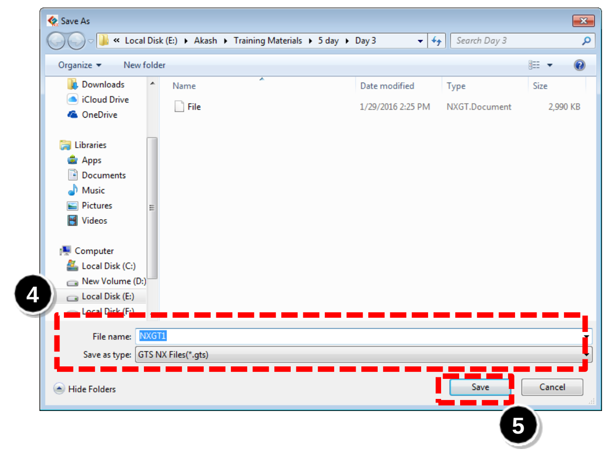
9-1 Analysis

Procedure

1. Go to Analysis > Perform
2. **Select Dam Analysis**
3. Click **OK**
4. **Give the File name as Dam Analysis.**
5. Save it at desired location







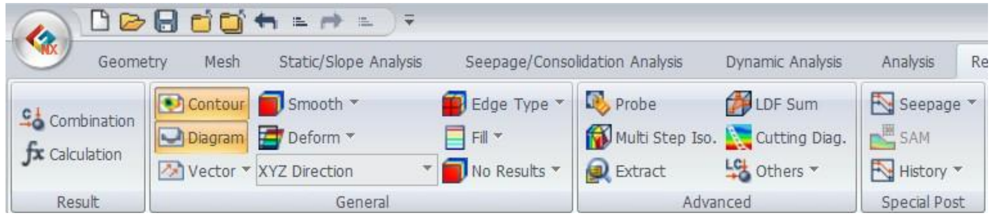
10-1 Results > Sloshing Fluid Pressure

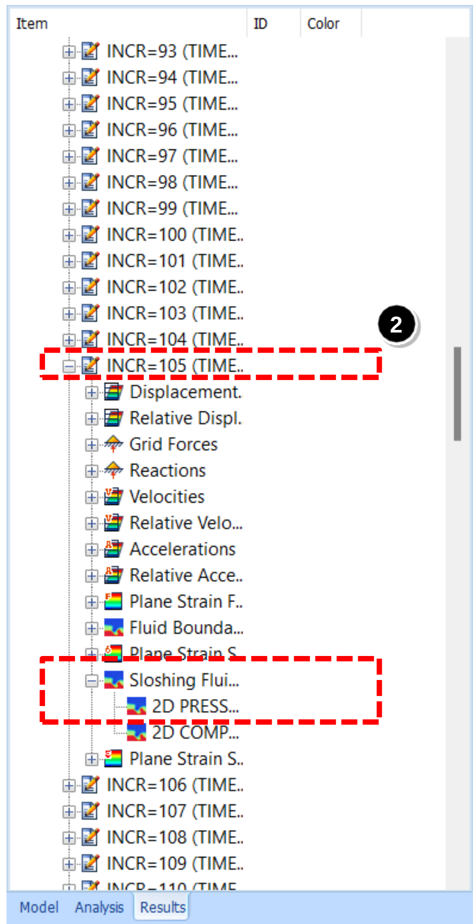
Procedure

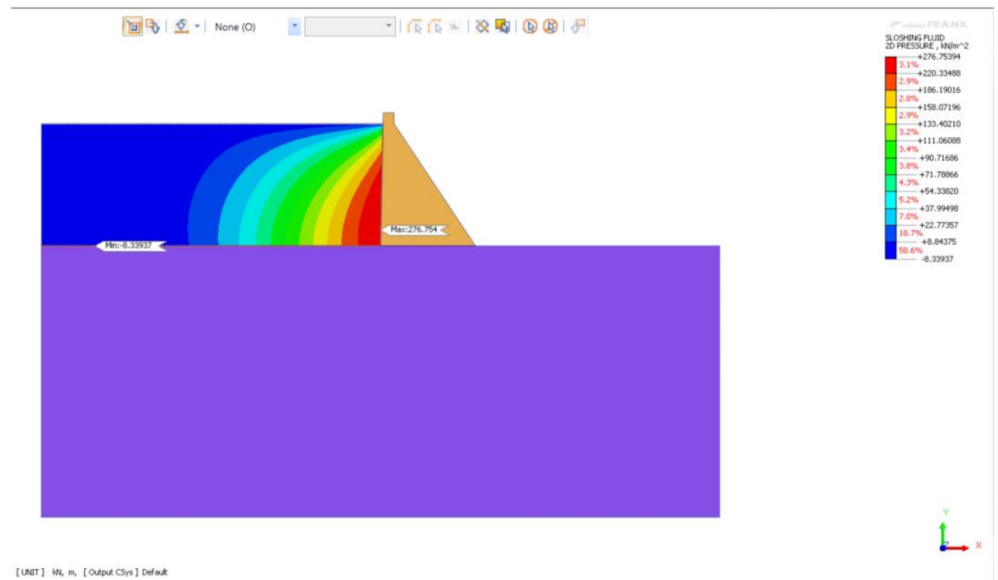
1. Go to works tree and enable mesh to check the results.

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.,







Happy Modelling