

**Overview****▪ 2-D Linear Static Analysis****▪ Model**

- $\frac{1}{4}$  Symmetric Model
- Unit : N, mm
- Isotropic Elastic Material
- Plane Stress Elements

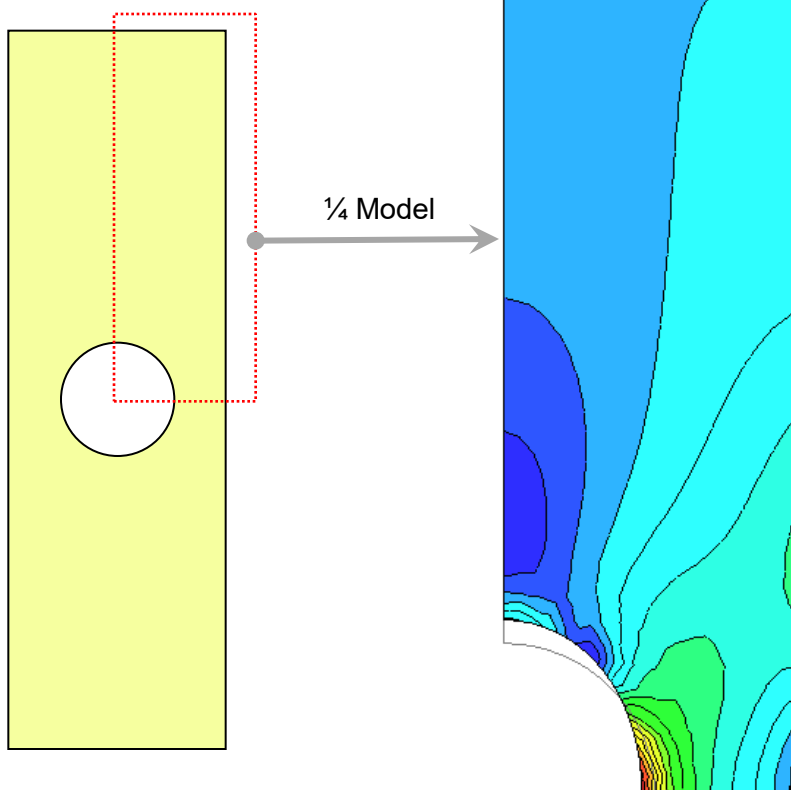
**▪ Load & Boundary Condition**

- Edge Pressure
- Constraint (Symmetry)

**▪ Result Evaluation**

- von Mises Stress
- Principal Stress Vector
- Probe Result

# Plate with a Hole



# Analysis Setting


## Procedure

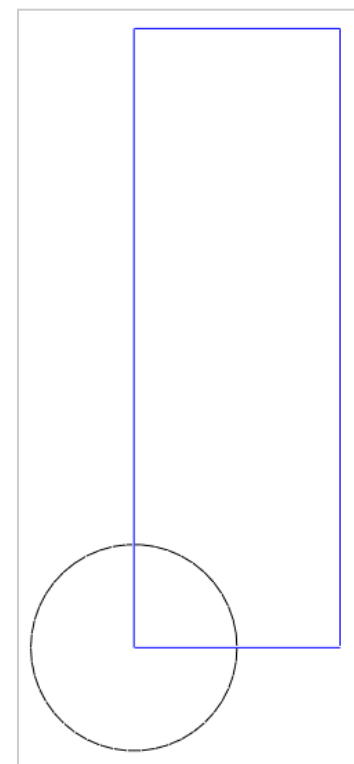
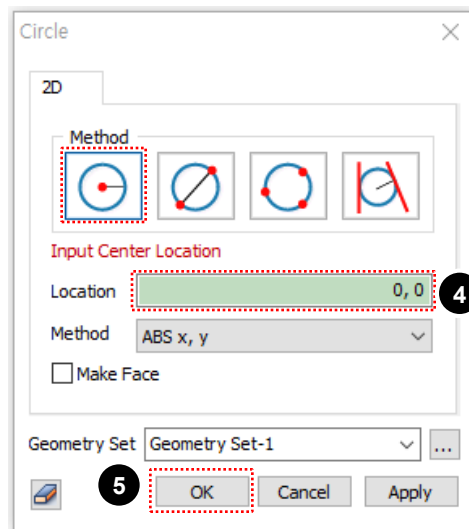
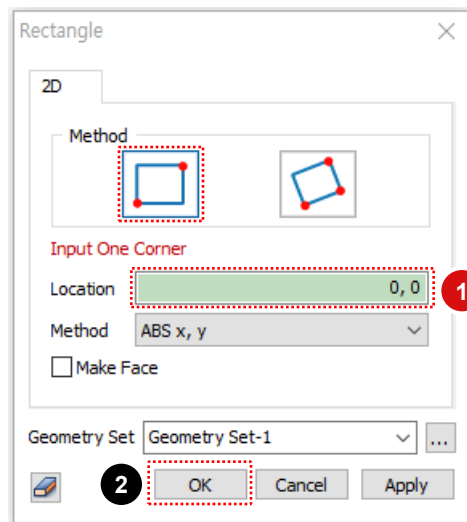
- 1 Model Type : **[2D]**
- 2 Unit System : **[N, cm]**
- 3 Click **[OK]** Button
- 4 Click Right Mouse Button in Work Window and Select **[Hide All Guiders]**


The image shows the 'Analysis Setting' dialog box. It has a title bar with a close button. The 'Project Title' is 'Plate with a Hole' and 'Engineer' is empty. The 'Desc.' field is empty. Under 'Model Type', there are three radio buttons: '3D', '2D' (selected and circled with a red dashed line and labeled '1'), and 'Axisymmetric'. Under 'Gravity Direction', there are two radio buttons: 'Y' (selected) and 'Z'. Under 'Unit System', there are two dropdown menus: the first is set to 'N' (circled with a red dashed line and labeled '2') and the second is set to 'cm'. At the bottom, there are two tabs: 'Initial Parameters' and 'Water Parameters'. The 'Initial Parameters' tab is active, showing 'Gravity Acceleration(g)' as 980.665 cm/sec<sup>2</sup>, 'Initial Temperature' as 0 [T], and 'Plane Strain Thickness' as 0.1 cm. At the bottom right, there are 'OK' and 'Cancel' buttons. The 'OK' button is circled with a red dashed line and labeled '3'.

The image shows a right-click context menu. It has a title bar and a close button. The menu items are: 'Show All', 'Hide All', 'Show All Geometries', 'Hide All Geometries', 'Show All Meshes', 'Hide All Meshes', 'Move Work Plane', 'Show/Hide Guiders' (with a right arrow), 'Show All Guiders', 'Hide All Guiders' (highlighted with a blue background and circled with a red dashed line and labeled '4'), 'Show All Labels', and 'Hide All Labels'.




**Procedure**


- 1 Location : “(0) , <10, 30>” 
- 2 Click [OK] Button
- 3 Geometry > Point & Curve > **Circle**
- 4 Center : “(0)”, Radius : “5”
- 5 Click [OK] Button
- 6 Click [Zoom All]

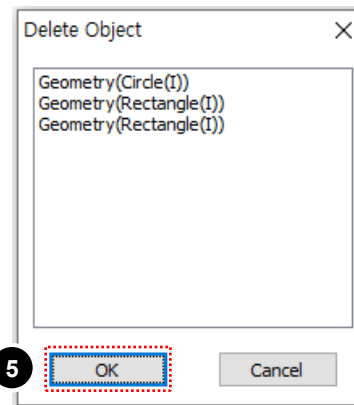
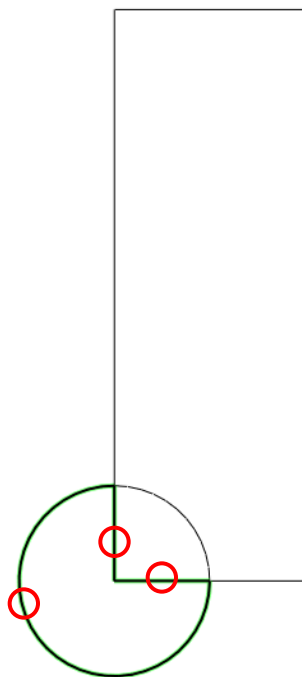
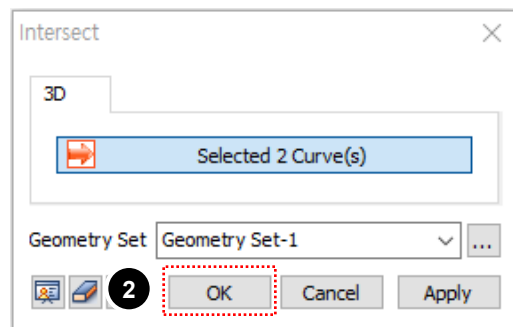


 ( ): “ABS x, y”, < >: “REL dx, dy”  
(0) same as (0, 0)


**Procedure**

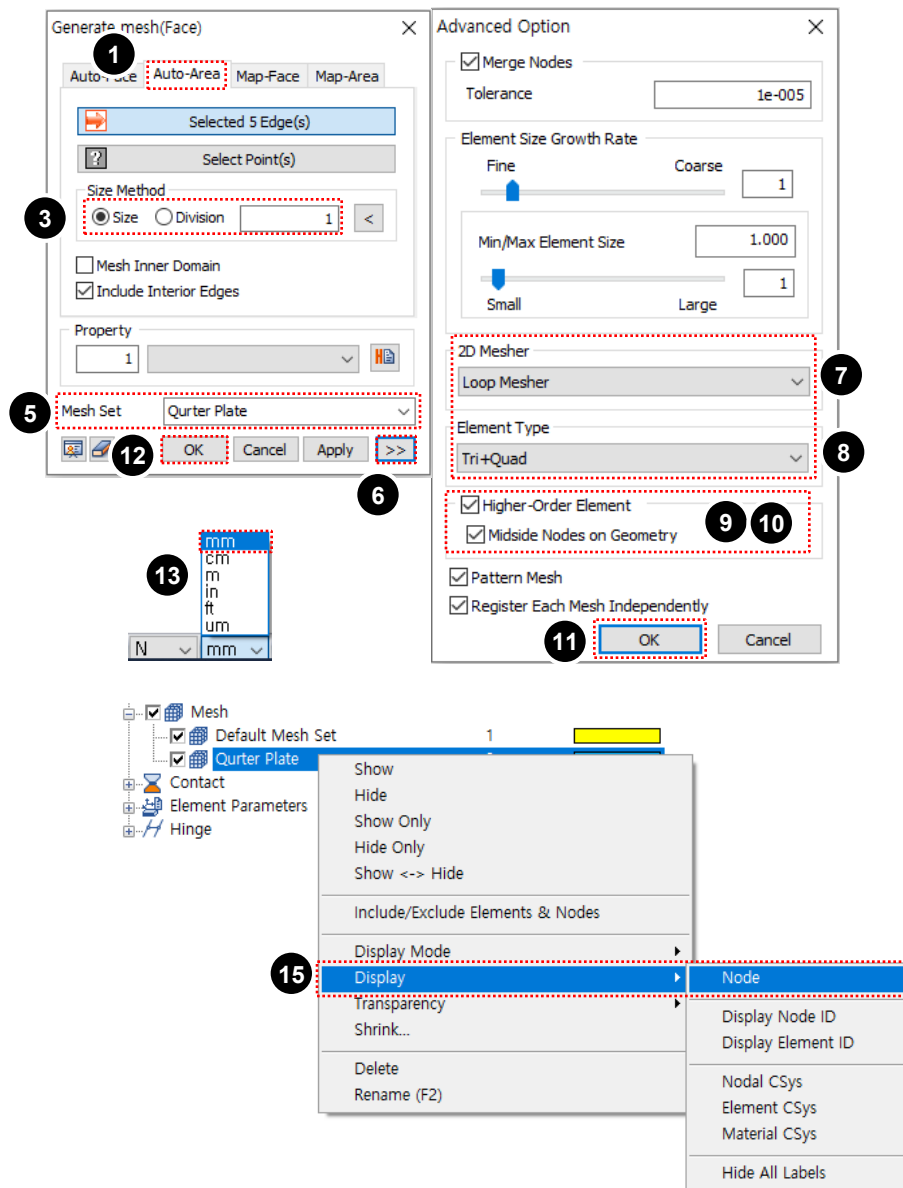
- 1 Select [  ] Select All 
- 2 Click [OK] Button
- 3 Select **3 Edges** Marked by [  ]  
(See Figure)
- 4 Press [Delete] Key
- 5 Click [OK] Button

 "Ctrl+A" as shortcut for "Select Displayed".



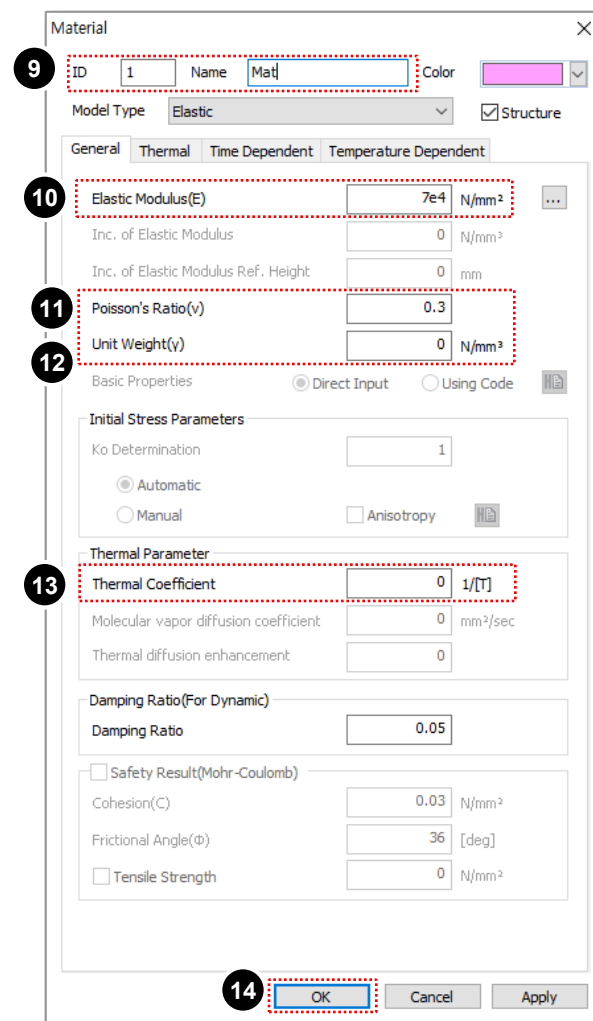
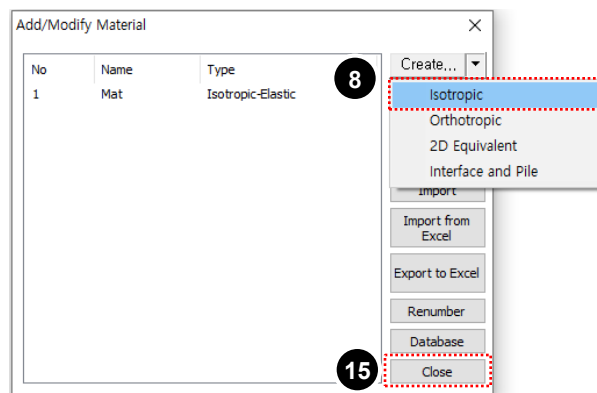
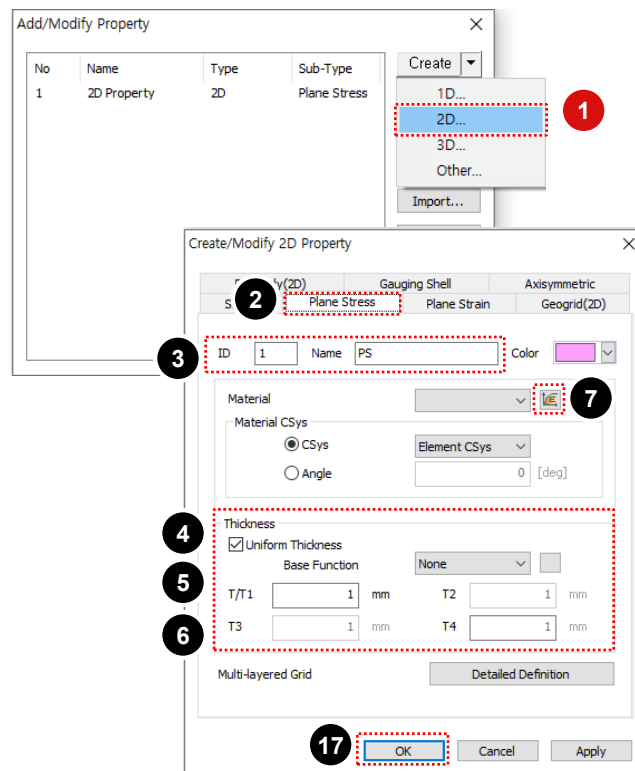
## Procedure

- 1 Select **[Auto-Area]**
- 2 Select [  ] Select all
- 3 Size Method - Mesh Size : “1”
- 4 Property : “1”
- 5 Mesh Set : **[Quarter Plate]**
- 6 Click [ >> ] Button
- 7 2D Mesher : **[Loop Mesher]**
- 8 Element Type : **[Tri+Quad]**
- 9 Check on **[Higher-Order Element]**
- 10 and **[Midside Nodes on Geometry]**
- 11 Click **[OK]** Button
- 12 Click **[OK]** Button
- 13 Change Unit System : **[N, mm]**
- 14 Model: **[Mesh - Quarter Plate]**
- 15 Click Right Mouse Button and Select **[Display > Node]**



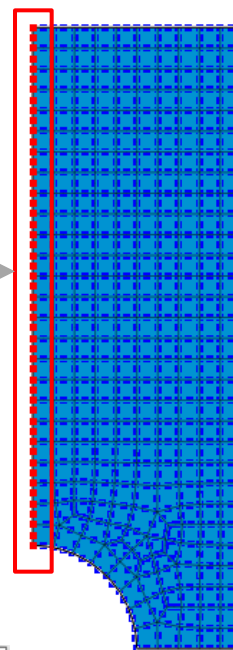
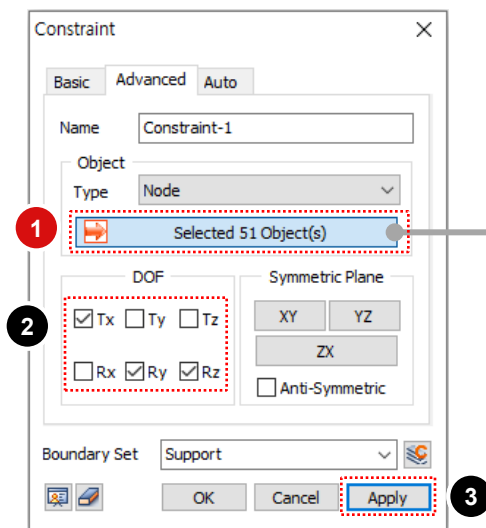
## Procedure


- 1 Create [2D]
- 2 Select [Plane Stress] tab
- 3 ID : "1", Name : "PS"
- 4 Check off [Uniform Thickness]
- 5 T/T1, T2, T3, T4 : "1"
- 6 Check on [Uniform Thickness]
- 7 Click [Material] Button
- 8 Click [Create Isotropic]
- 9 ID : "1", Name : "Mat"
- 10 Elastic Modulus : "7e4" N/mm<sup>2</sup>
- 11 Poisson's Ratio : "0.3"
- 12 Unit Weight : "0"
- 13 Thermal Coefficient : "0"
- 14 Click [OK] Button
- 15 Click [Close] Button
- 16 Select "1: Mat" for Material
- 17 Click [OK] Button
- 18 Click [Close] Button

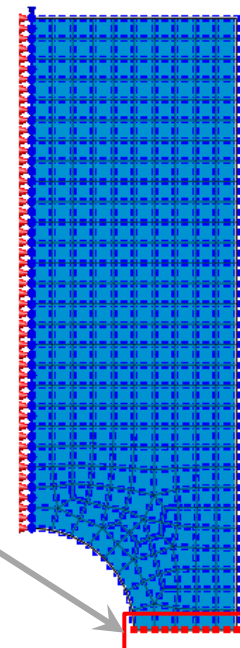
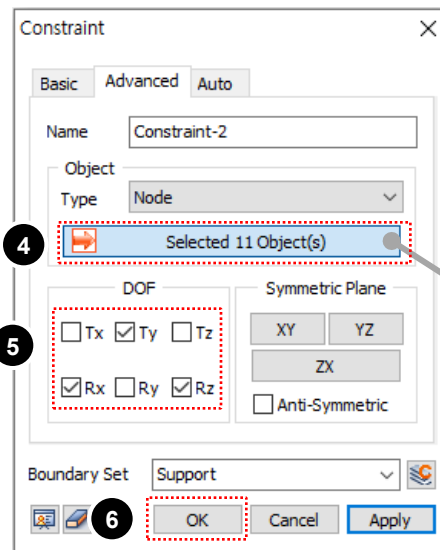


**Procedure**

- 1 Select Left Nodes (51 Nodes)
- 2 Click **[Tx, Ry, Rz]**
- 3 Boundary Set : **[Support]**
- 4 Click **[Apply]** Button
- 5 Select Bottom Nodes (11 Nodes)
- 6 Click **[Ty, Rx, Rz]**
- 7 Click **[OK]** Button

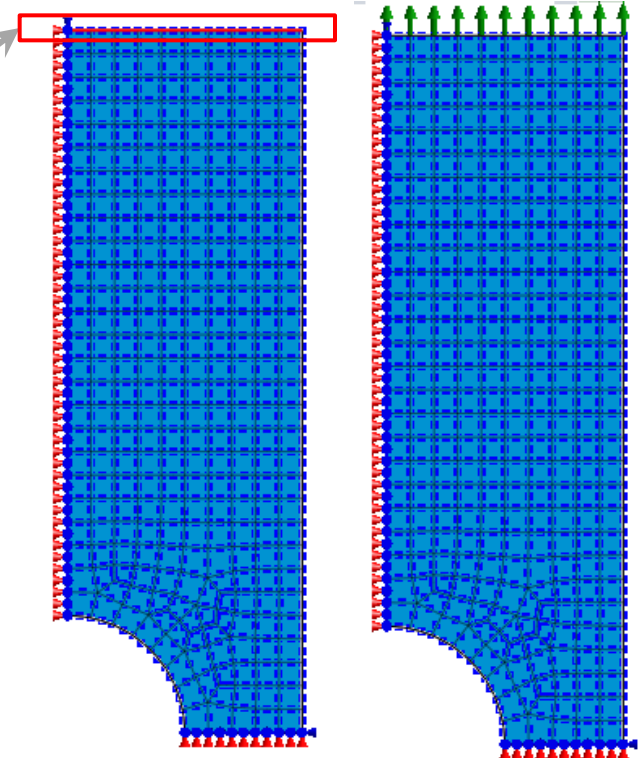
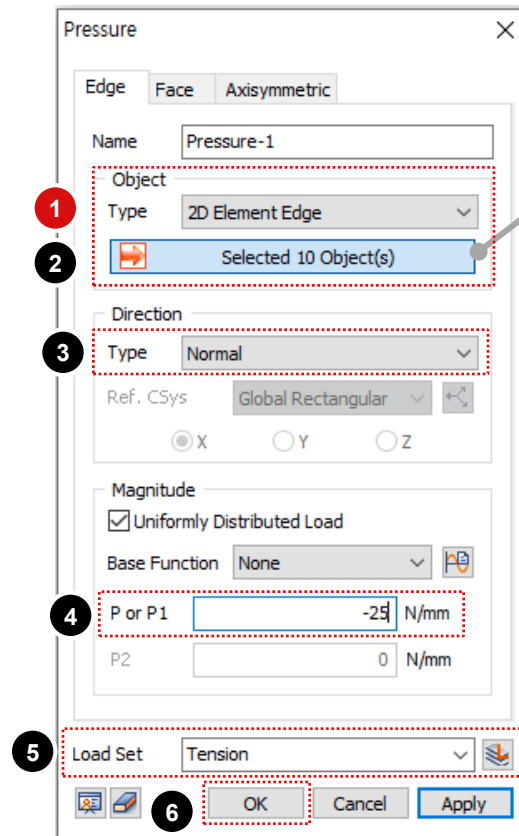


-  Drag mouse to select nodes  
(Window Selection)



**Procedure**

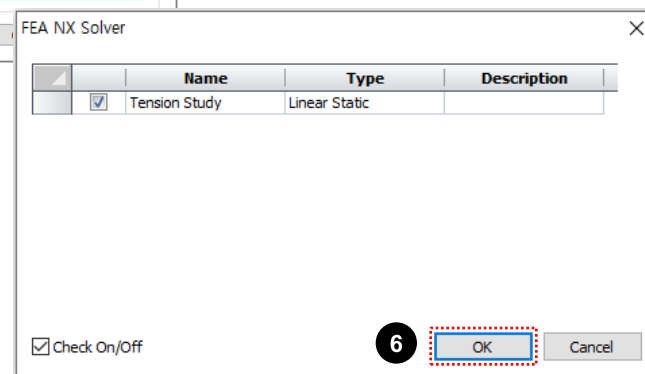
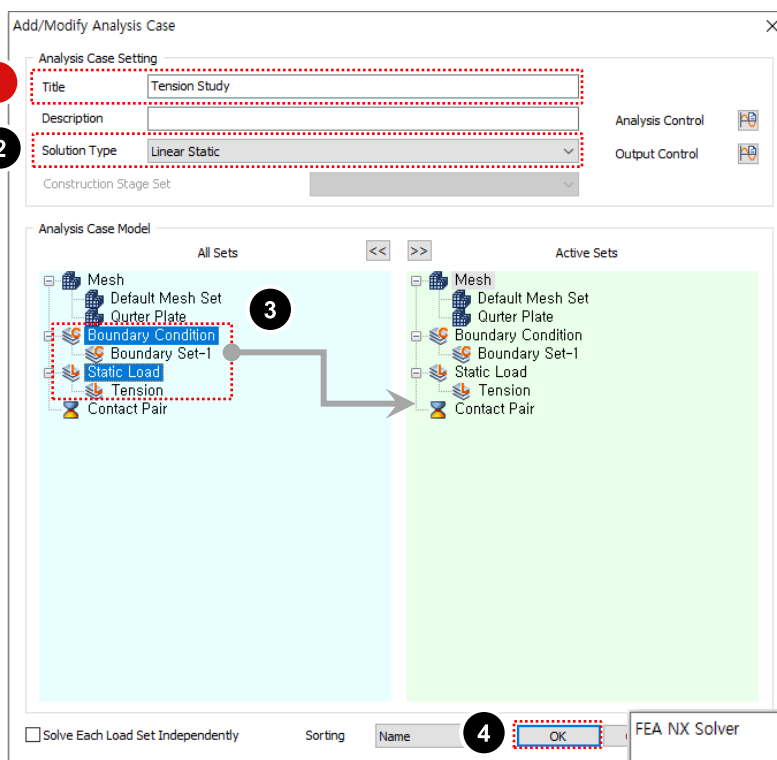
- 1 Object Type: **[2D Element Edge]**
- 2 Select **[10 Element Edges]**
- 3 Direction Type : **[Normal]**
- 4 P or P1 : **"-25" N/mm**
- 5 Load Set : **[Tension]**
- 6 Click **[OK]** Button





**Procedure**

- 1 Title : **"Tension Study"**
- 2 Analysis Type : **[Linear Static]**
- 3 Drag & Drop **[Static Load]** and **[Boundary Condition]** to **[Active Sets]**
- 4 Click **[OK]** Button
- 5 Analysis > **[Perform]**
- 6 Click **[OK]** Button



# 09 Displacements

## Procedure

- 1 Results : Tension Study  
Linear Static > Displacements
- 2 Double Click **[TOTAL  
TRANSLATION (V)]**

