

## Overview

### ▪ 3-D Linear Static Analysis Model

- Unit : N, cm
- Isotropic Elastic Material
- Solid Element

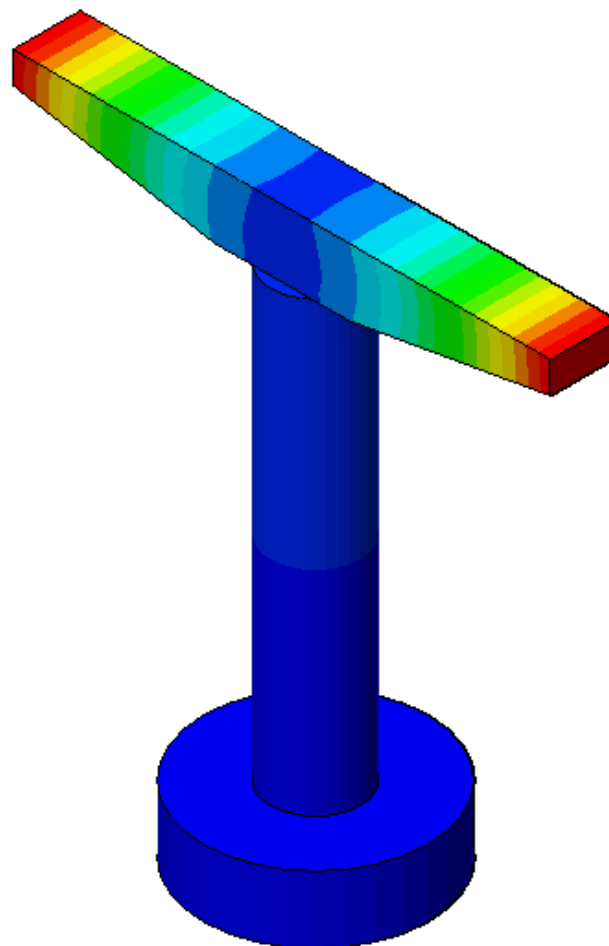
### ▪ Load & Boundary Condition

- Body Force
- Pressure
- Constraint

### ▪ Result Evaluation

- Deformation
- 3D Element Principal Stress

# Pier - MapMesh



**Procedure**

- 1 Analysis Type : **[3D]**
- 2 Gravity Direction : **[Y]**
- 3 Force : **[N]** , Length : **[cm]**
- 4 Click **[OK]** Button

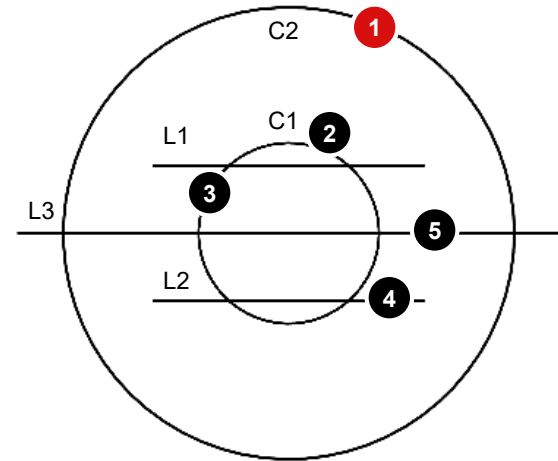
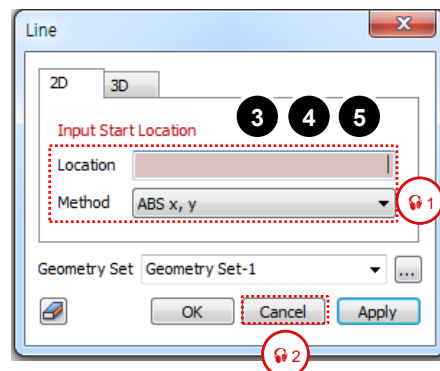
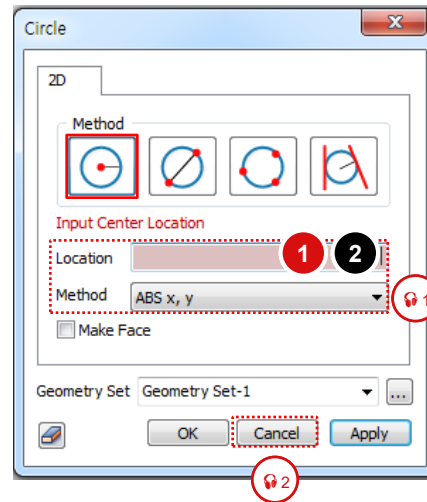
The screenshot shows the 'Analysis Setting' dialog box. It has fields for 'Project Title' and 'Engineer'. Below these is a 'Desc.' field. The 'Model Type' section has three radio buttons: '3D' (selected and circled with a red dashed box and labeled '1'), '2D', and 'Axisymmetric'. The 'Gravity Direction' section has three radio buttons: 'Y' (selected and circled with a red dashed box and labeled '2'), 'Z', and 'X'. The 'Unit System' section has three dropdown menus: 'N' (circled with a red dashed box and labeled '3'), 'cm', and 'J'. The 'Initial Parameters' tab is active, showing 'Gravity Acceleration(g)' as 980.665 cm/sec², 'Initial Temperature' as 0 [T], and 'Plane Strain Thickness' as 1 cm. At the bottom, the 'OK' button is circled with a red dashed box and labeled '4'. A 'Cancel' button is also present.



Analysis Control Dialog is automatically activated at startup.

**Procedure**



- 1 Location - C1 : “**Center(0) , R=40**”
- 2 Location - C2 : “**Center(0) , R=100**”
- 3 Click Cancel Button
- 4 Location - L1 : “**(-60, 30), <120>**”
- 5 Location - L2 : “**(-60, -30), <120>**”
- 6 Location - L3 : “**(-120), <240>**”
- 7 Click Cancel Button

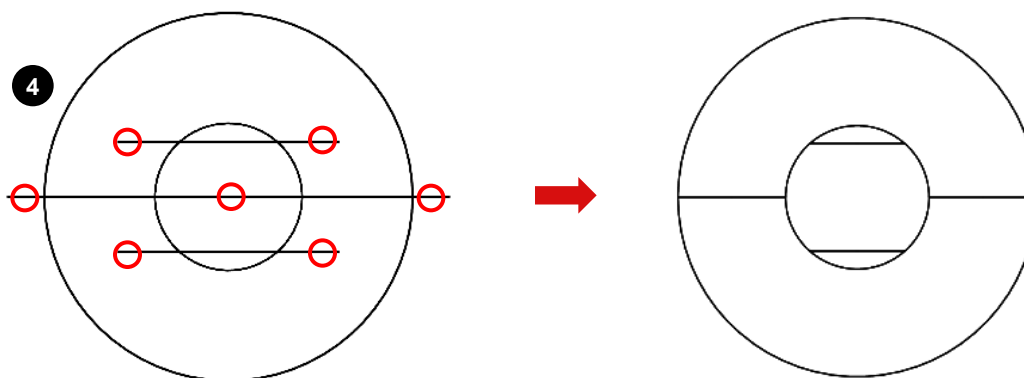
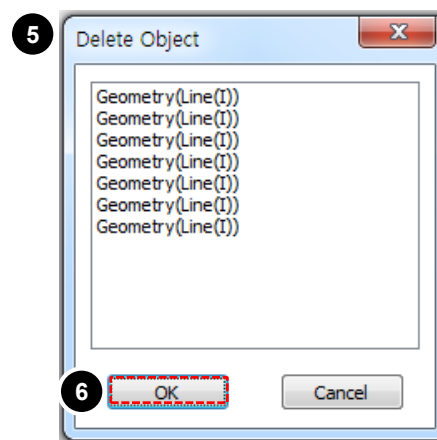
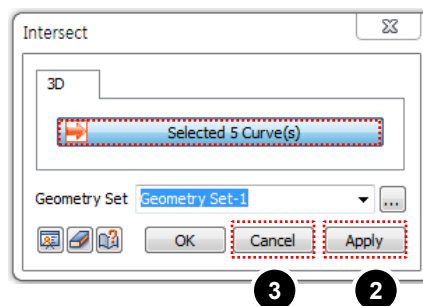


1 ( ): “ABS x, y”,  
 < >: “REL dx, dy”  
 ( 0 ) same as (0, 0),  
 <120> same as <120,0>


2 “Esc” as shortcut for “Cancel”.

**Procedure**

- 1 Select **[Displayed]** 
- 2 Click **[Apply]** Button 
- 3 Click **[Cancel]** Button
- 4 Select Unnecessary Edges marked by **[○]** (See Figure) Select
- 5 Press **[Delete]** Key
- 6 Click **[OK]** Button

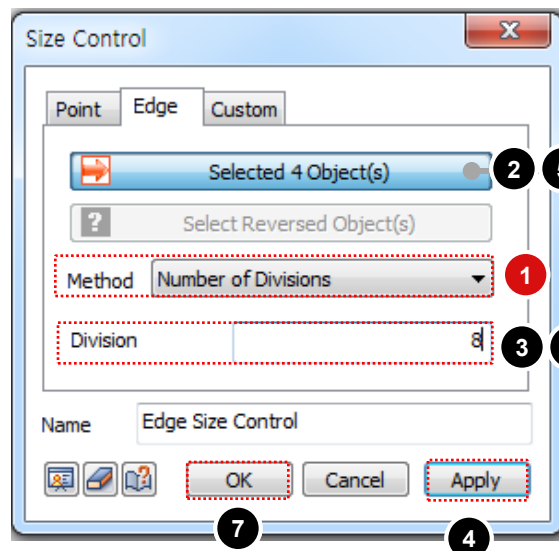


 "Ctrl+A"  
"as shortcut for "Select Displayed".

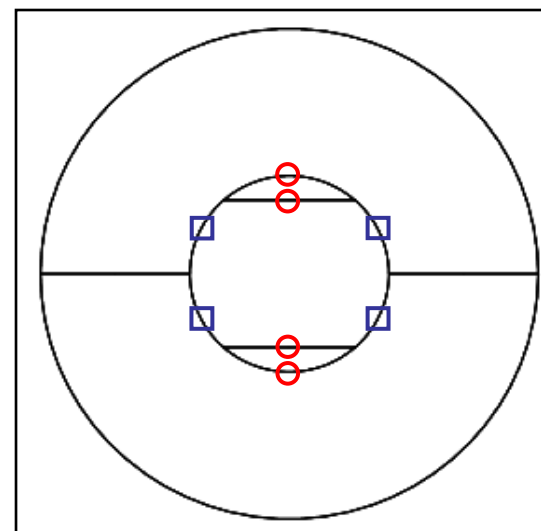
 "Enter" as shortcut for "Apply".

**Procedure**

- 1 Seeding Method :  
**[Number of Divisions]**
- 2 Select **[Group1]** Edges (See Figure)
- 3 Number of Divisions : "8"
- 4 Click **[Apply]** Button
- 5 Select **[Group2]** Edges (See Figure)
- 6 Number of Divisions : "4"
- 7 Click **[OK]** Button

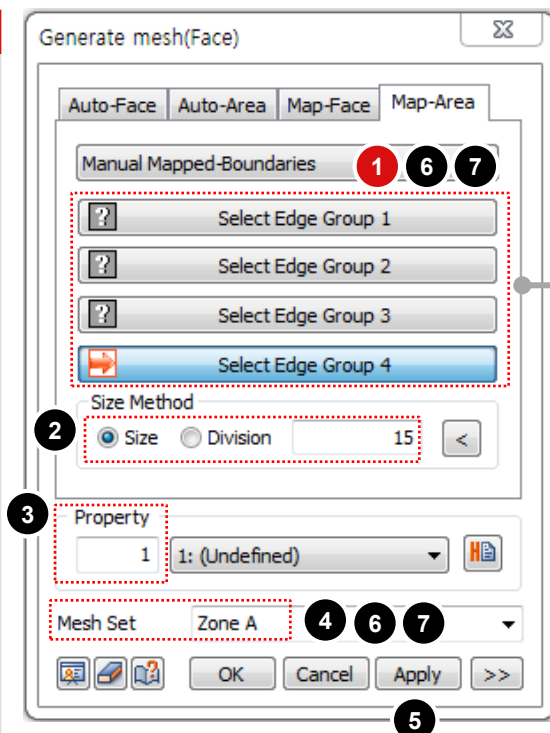


- (Group1) Division : 8  
□ (Group2) Division : 4



**Procedure**

- 1 Select 6 Edges of **[Zone A]**  
(See Figure)
- 2 Element Size : "15"
- 3 Property : "1"
- 4 Mesh Set : **[Zone A]**
- 5 Click **[Apply]** Button
- 6 Repeat Step 2~6 for **[ Zone B ]**
- 7 Repeat Step 2~6 for **[ Zone C ]**

**Zone A**

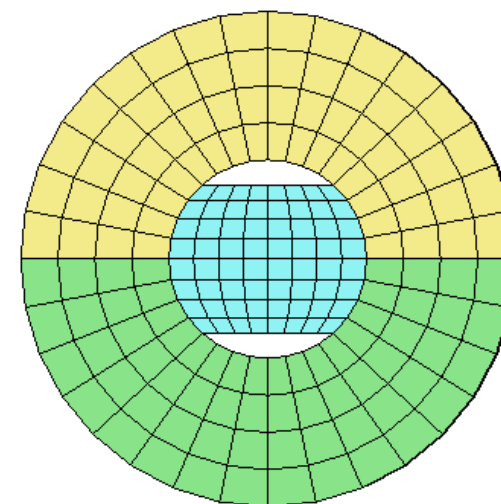
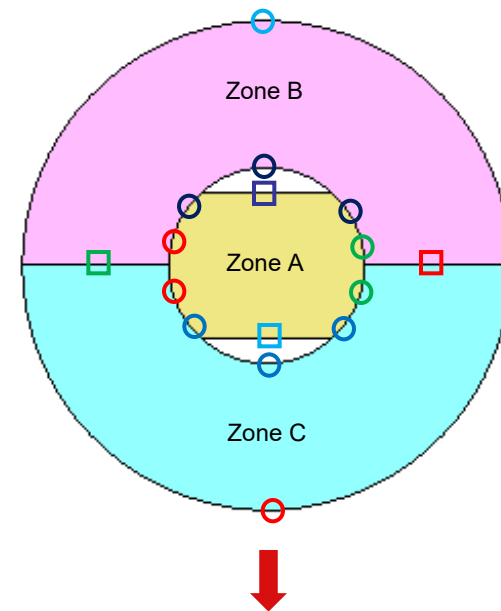
- (Group1) : ○  
 (Group2) : □  
 (Group3) : ●  
 (Group4) : □

**Zone B**



- (Group1) : ○  
 (Group2) : □  
 (Group3) : ●  
 (Group4) : □

**Zone B**

- (Group1) : ○  
 (Group2) : □  
 (Group3) : ●  
 (Group4) : □

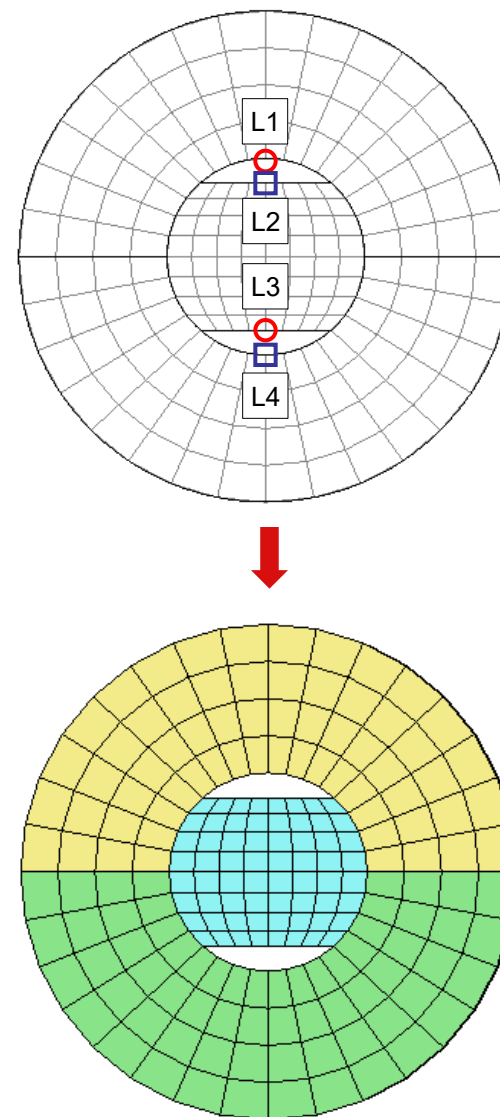
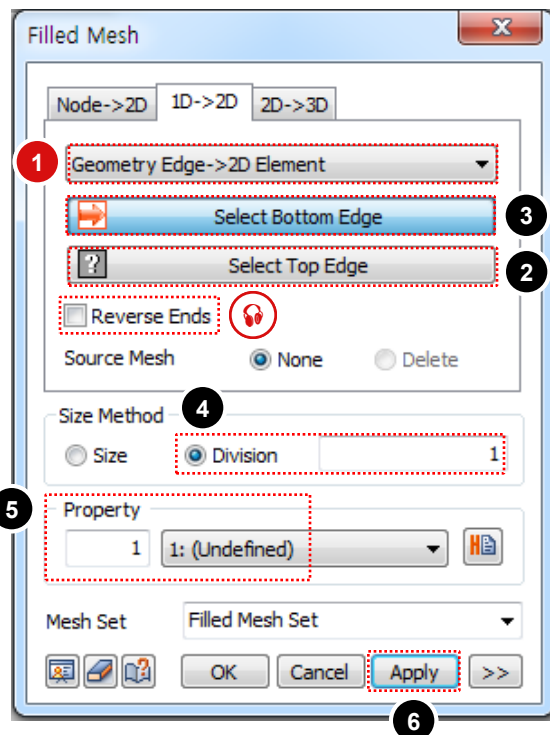


**Procedure**

- 1 Select **[Geometry Edge→2D Element]** Tab
  - 2 Select [ L1 ] for **Top Edge** marked by [  ] (See Figure)
  - 3 Select [ L2 ] for **Bottom Edge** marked by [  ] (See Figure)
  - 4 Mesh Size : **[Division]** "1"
  - 5 Property : "1"
  - 6 Click **[Apply]** Button
  - 7 Repeat Step 3 to 6 for [ L3 ] & [ L4 ]
- Click **[OK]** Button



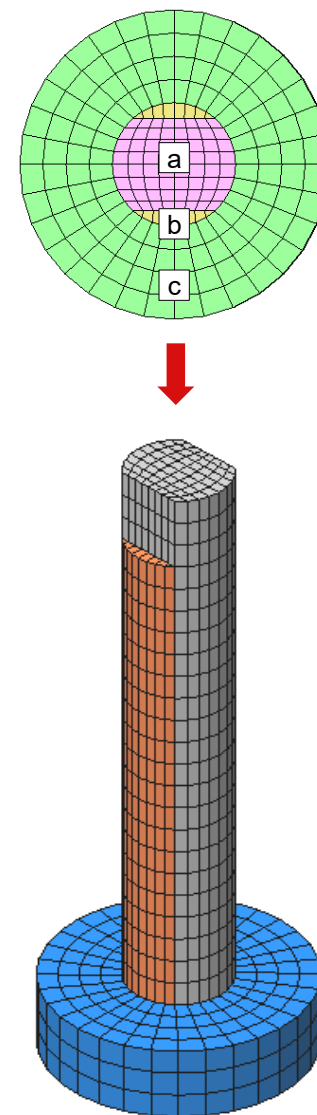
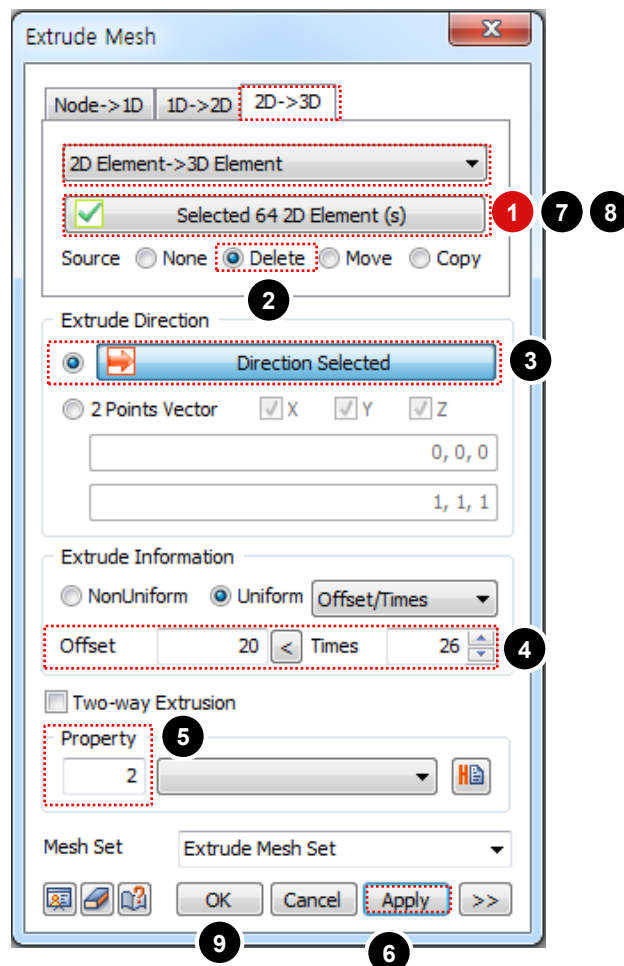
If result is wrong, check on "Reverse Ends".



**Procedure**

- 1 Select **[Mesh Set]** (a) (See Figure)
- 2 Source Mesh : **[Delete]**
- 3 Extrusion Direction : **[Z-axis Click]**
- 4 Offset : **"20"**, Number of Times : **"26"**
- 5 Property : **"2"**
- 6 Click **[Apply]** Button
- 7 Repeat Step **[1 to 6]** for Mesh Set (b)  
(See Figure)
- 8 Repeat Step **[1 to 6]** for Mesh Set (c)  
(See Figure)
- 9 Click **[OK]** Button

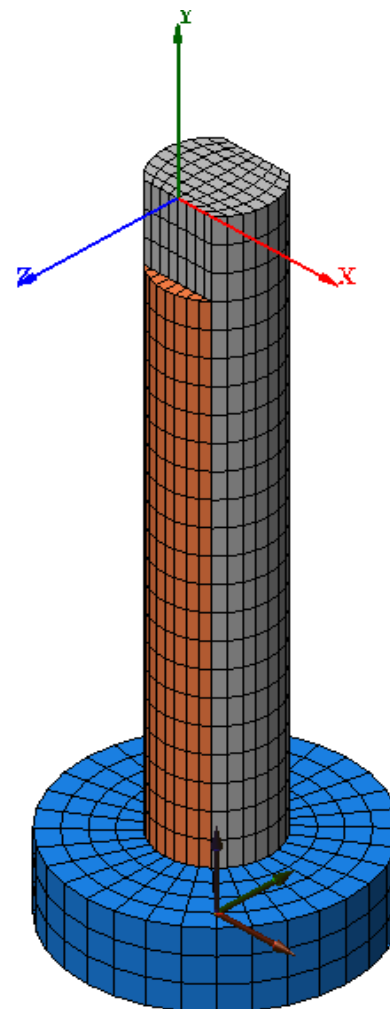
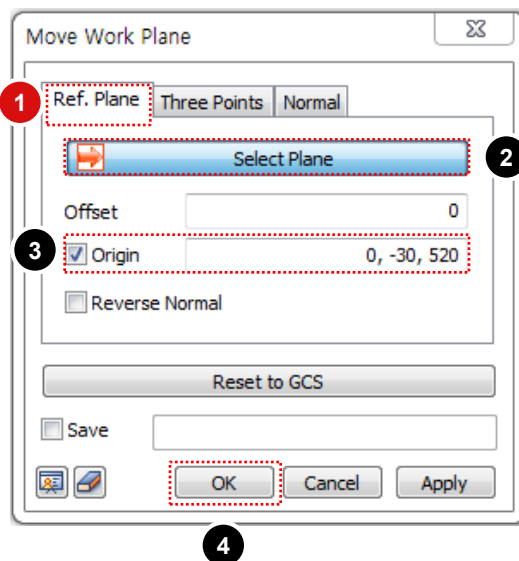
- a Number of Times : 26  
b Number of Times : 23  
c Number of Times : 3





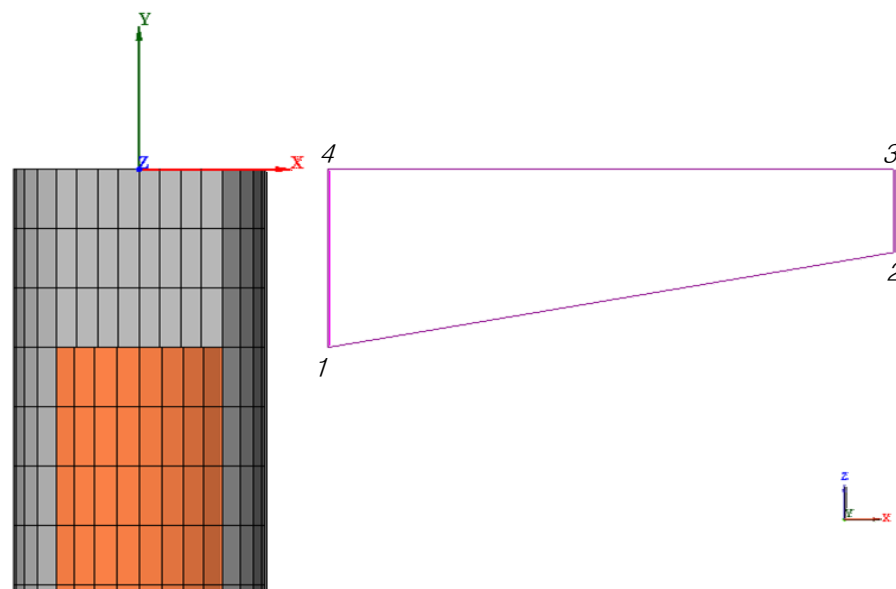
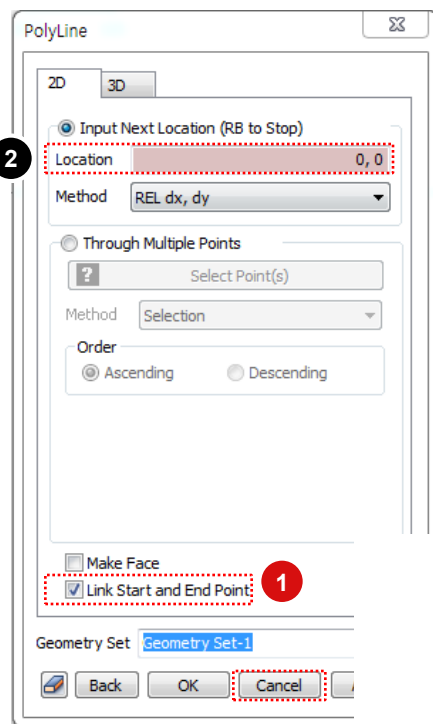
**Procedure**

- 1 Select **[Ref.Plane]** Tab
- 2 Select Plane : **"XZ-Plane"**
- 3 Origin : **[0,-30,520]**
- 4 Click **[OK]** Button



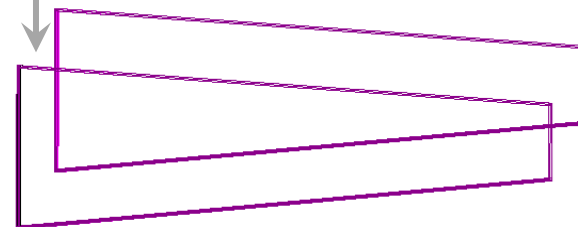
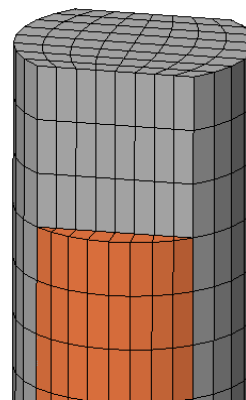
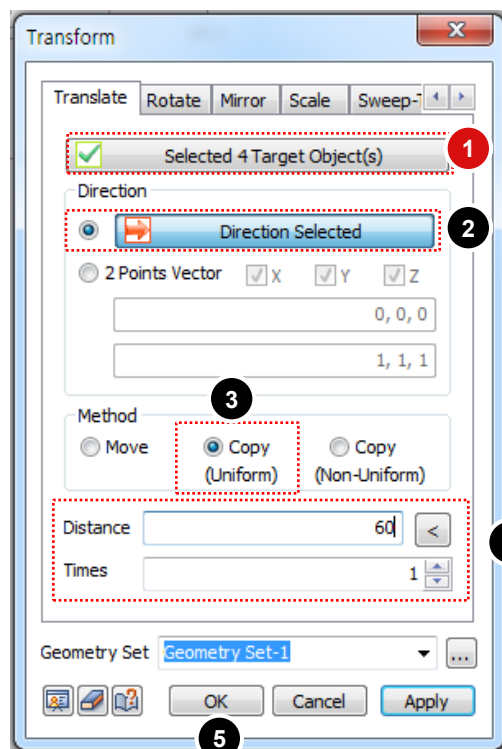
**Procedure**

- 1 Check on  
**[Link Start And End Point]**
- 2 Location : “(60, -60), <180, 32>,  
<0, 28>, <-180>”
- 3 Click **[Cancel]** Button



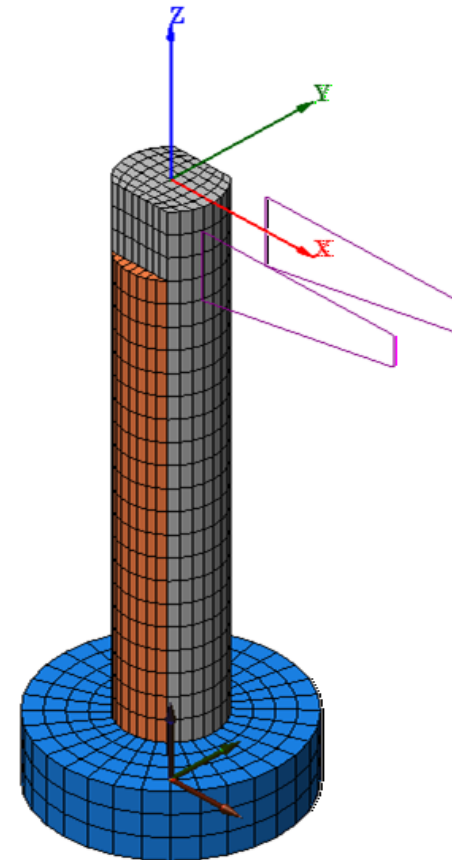
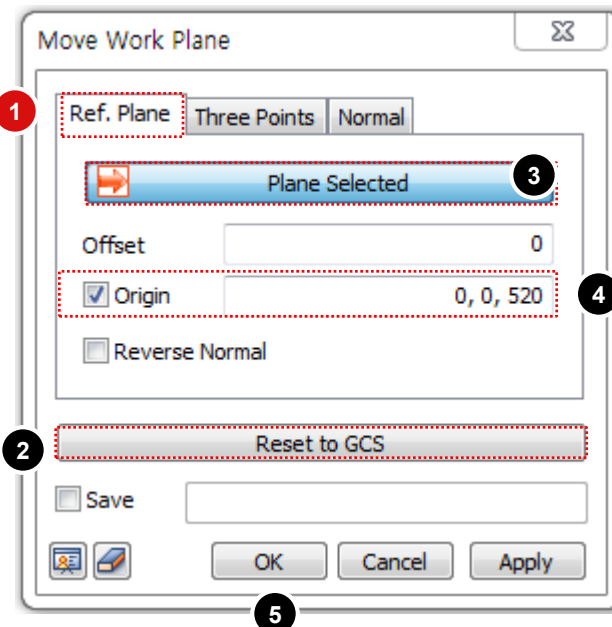
**Procedure**

- 1 Select **[Polyline]** (See Figure)
- 2 Direction : **[Y-axis]**
- 3 Check on **[Uniform Copy]**
- 4 Distance : "**60**", Number of Times : "**1**"
- 5 Click **[OK]** Button



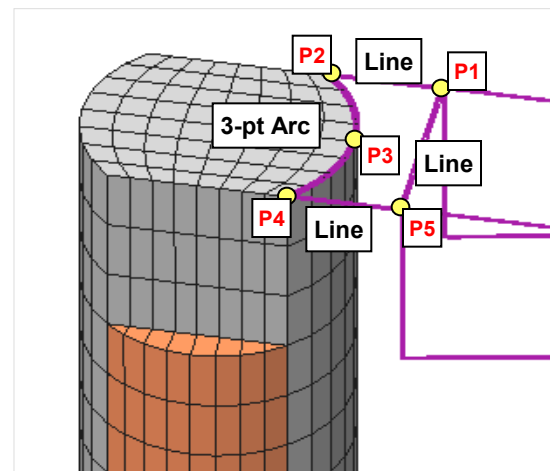
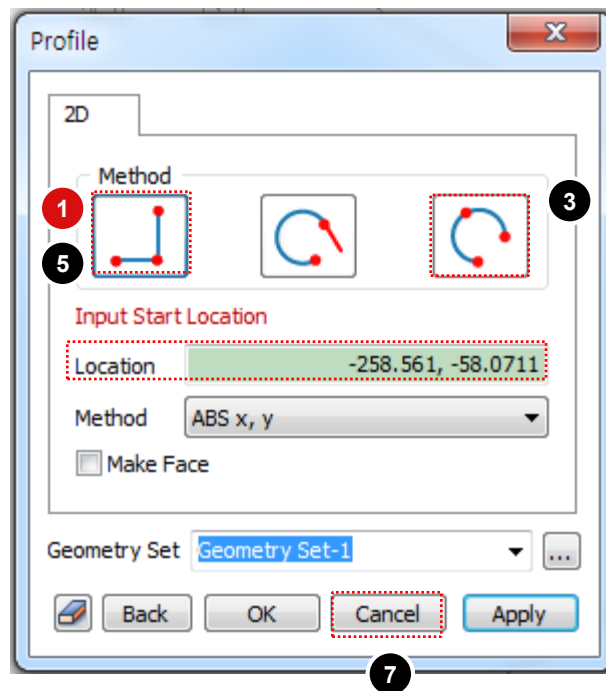
**Procedure**

- 1 Select **[Ref. Plane]** Tab
- 2 Click **[Reset to GCS]** Button
- 3 Plane Selected : **[XY]**
- 4 Origin : **"0,0,520"**
- 5 Click **[OK]** Button



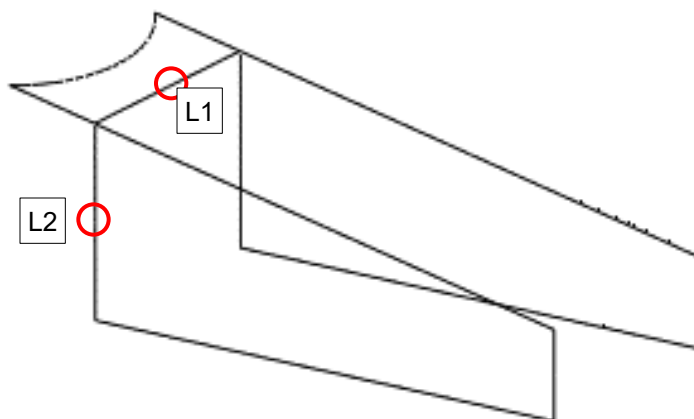
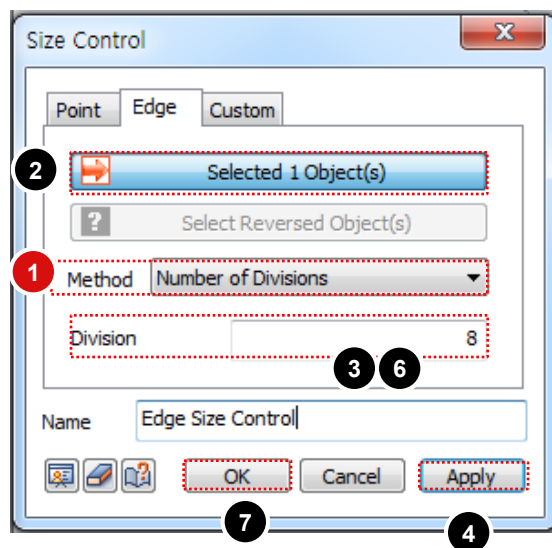
**Procedure**

- 1 Select **[Polyline]** method
- 2 Select **[P1]** , **[P2]** in sequential order
- 3 Select **[3-Point Arc]** method
- 4 Select **[P2],[P3],[P4]**in sequential order
- 5 Select **[Polyline]** method
- 6 Select **[P4],[P5],[P1]** in sequential order
- 7 Click **[Cancel]** Button



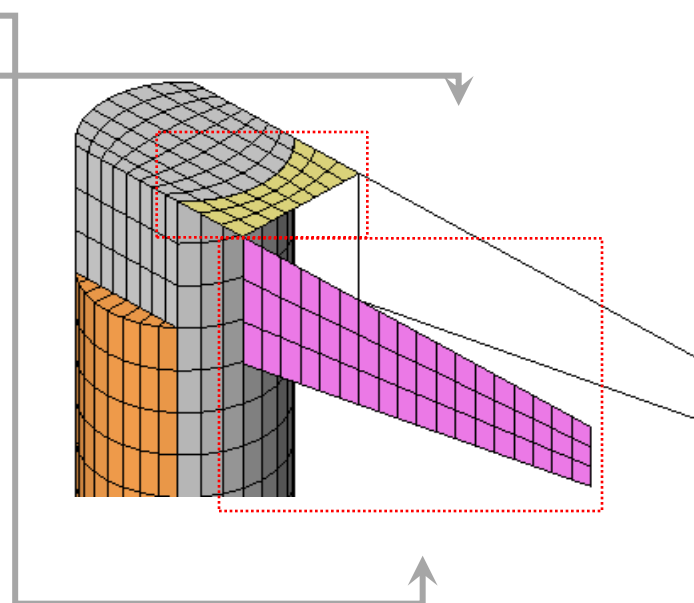
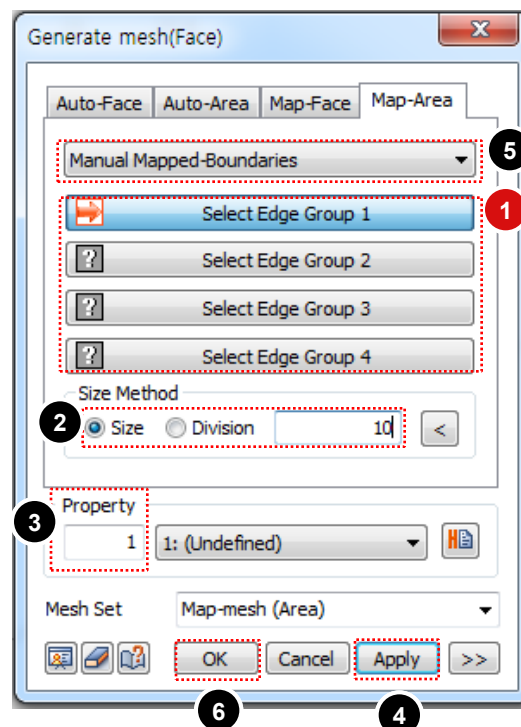
**Procedure**

- 1 Seeding Method :  
**[Number of Divisions]**
- 2 Select **[L1]** Edge (See Figure)
- 3 Number of Divisions : "8"
- 4 Click **[Apply]** Button
- 5 Select **[L2]** Edge (See Figure)
- 6 Number of Divisions : "3"
- 7 Click **[OK]** Button



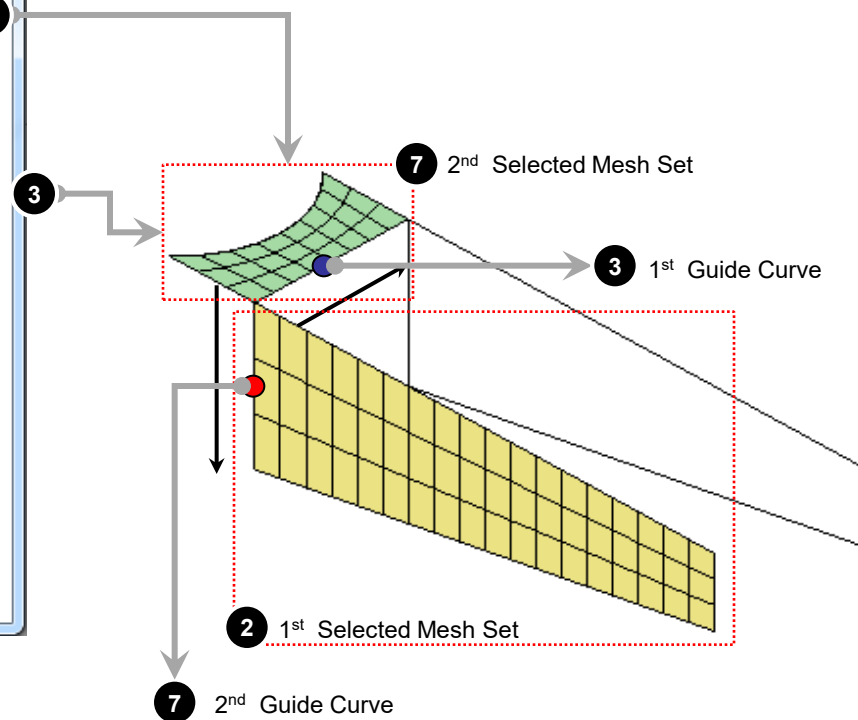
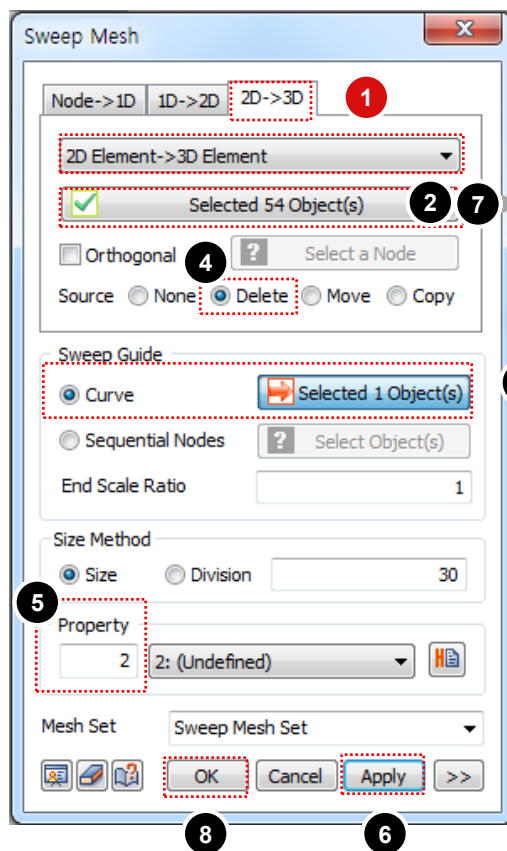
**Procedure**

- 1 Select **Each Edge** (See Figure)
- 2 Mesh Size – [Element Size] : “10”
- 3 Property : “1”
- 4 Click [Apply] Button
- 5 Repeat Step [1~3] for 2<sup>nd</sup> Edges (See Figure)
- 6 Click [OK] Button



**Procedure**

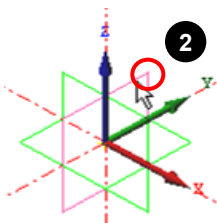
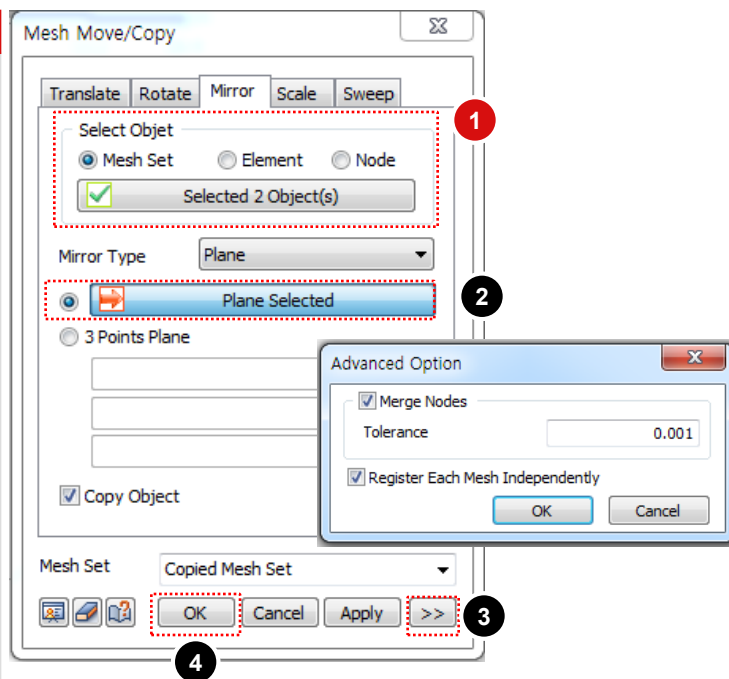
- 1 Select **[2D→3D]** Tab
- 2 Select **[1<sup>st</sup> Mesh Set]** (See Figure)
- 3 Select **[1<sup>st</sup> Guide Curve]**
- 4 Source Mesh : **[Delete]**
- 5 Property : **"2"**
- 6 Click **[Apply]** Button
- 7 Repeat Step **[1~5]** for 2<sup>nd</sup> Mesh Set & Guide Curve (See Figure)
- 8 Click **[OK]** Button



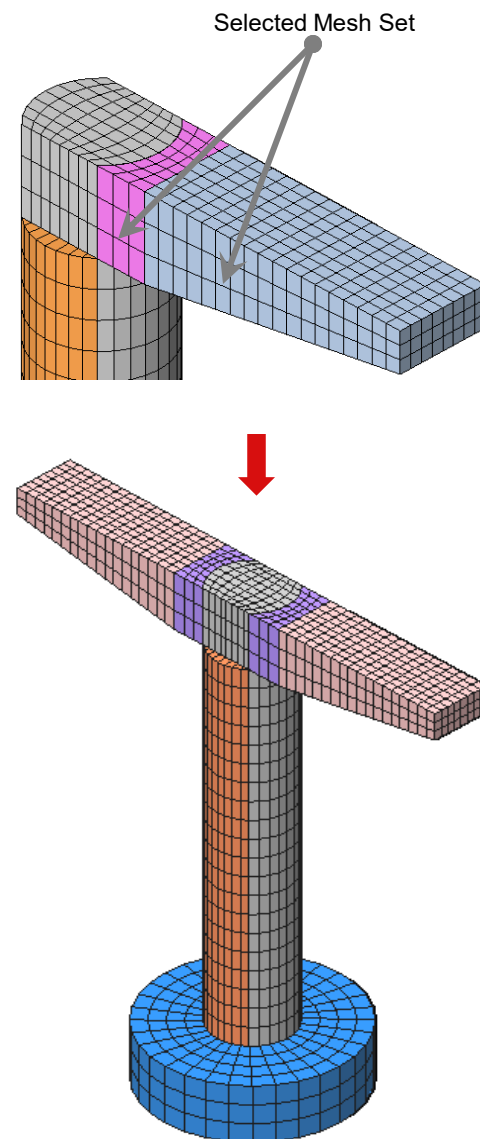


**Procedure**


- 1 Select **[2 Mesh Sets]** (See Figure)
- 2 Mirror Plane : **[YZ-Plane]**
- 3 Advanced Option > **Merge Nodes** On
- 4 Click **[OK]** Button

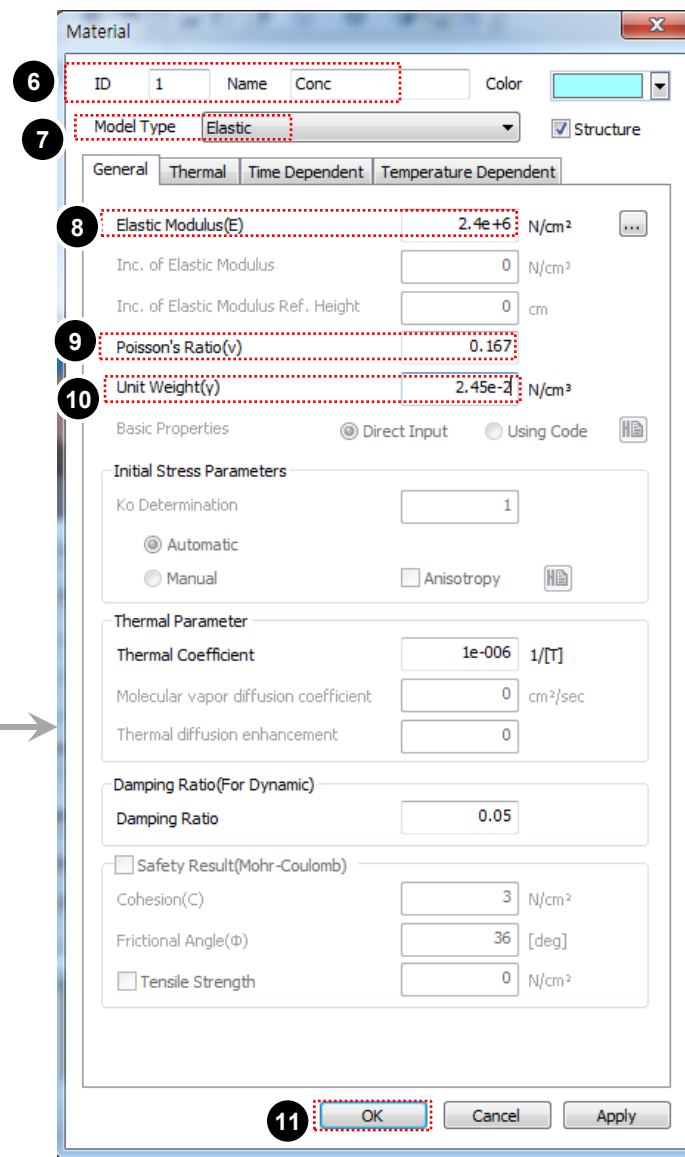
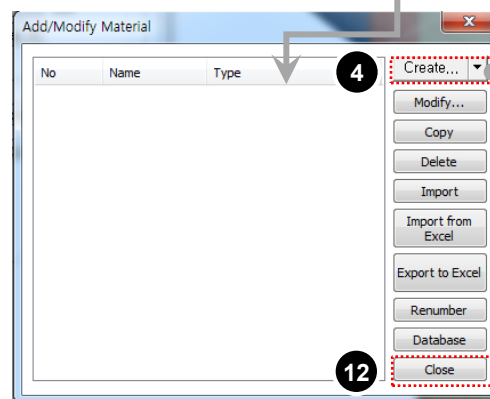
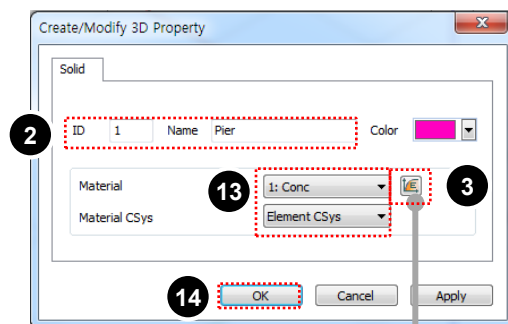
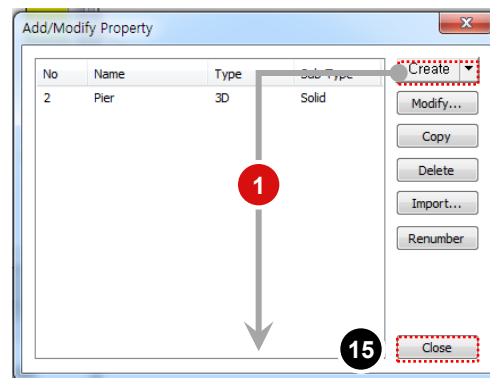


Select "YZ-Plane" in Work Window or  
Pre-Works Tree



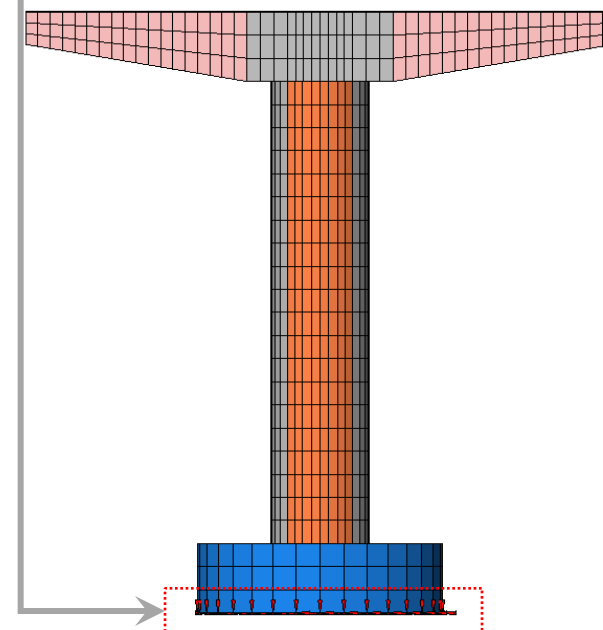
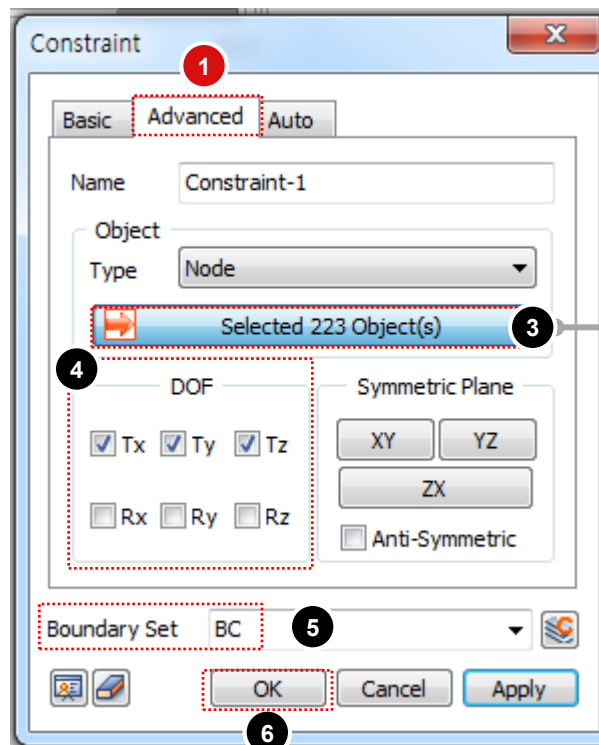
**Procedure**

- 1 Select **[Create 3D ...]**
- 2 ID: "1", Name: "Pier"
- 3 Click  Button
- 4 Click **[Create]** Button
- 5 Select **[Isotropic]** Tab
- 6 ID: "1", Name: "Conc"
- 7 Model Type: **[Elastic]**
- 8 Elastic Modulus: "**2.4e6**" N/cm<sup>2</sup>
- 9 Poisson's Ratio: "**0.167**"
- 10 Unit Weight : "**2.452e-2**" N/cm<sup>3</sup>
- 11 Click **[OK]** Button
- 12 Click **[Close]** Button
- 13 Select **[1: Conc]** for Material
- 14 Click **[OK]** Button
- 15 Click **[Close]** Button



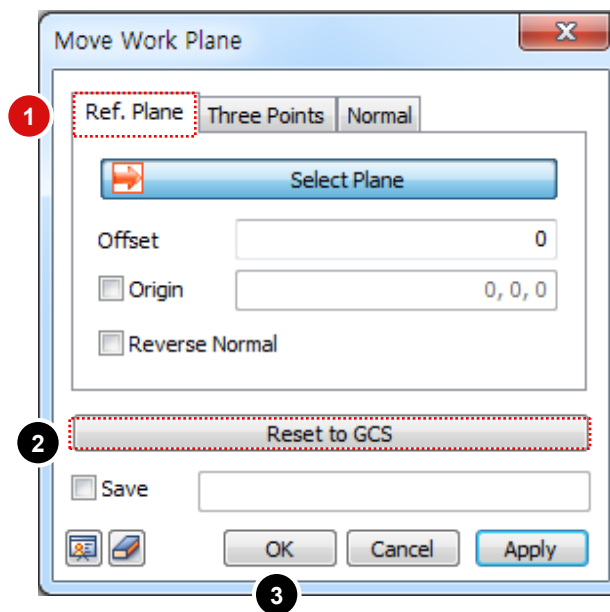
**Procedure**

- 1 [Advanced] tab
- 2 Click [Front View]
- 3 Select [223 Nodes] (See Figure)
- 4 DOF : Tx, Ty, Tz
- 5 BC Set : [BC]
- 6 Click [OK] Button




**Procedure**

- 1 Select **[Ref. Plane]** Tab
- 2 Click **[Reset to GCS]** Button
- 3 Click **[OK]** Button



**Procedure**

- 1 Click  Button
- 2 Name : **[GRAV]**
- 3 Click **[Add]** Button
- 4 Name : **"PRESS"**
- 5 Click **[Add]** Button
- 6 Click **[Close]** Button
- 7 Load Set : **[GRAV]**
- 8 Gravitational Force Factor : **[GZ] "-1"**
- 9 Click **[OK]** Button

**Gravity**

Name: Gravity-1

Reference Object

Type: Coordinate

Ref. CSys: Global Rectangular

Components

Gx: 0

Gy: 0

Gz: -1

Spatial Distribution

Base Function: None

Load Set: GRAV

OK Cancel Apply

**Load Set**

Name: GRAV

Desc.:

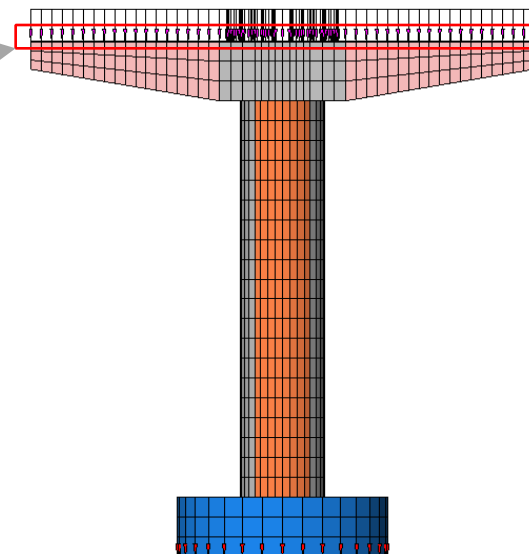
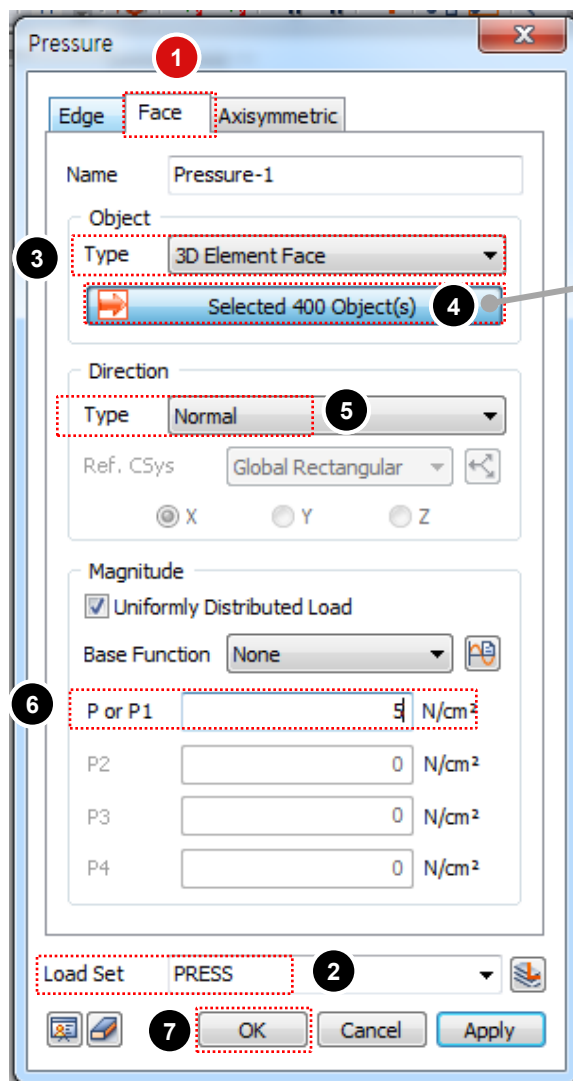
Add Modify Delete

No	Name	Description
----	------	-------------

Close

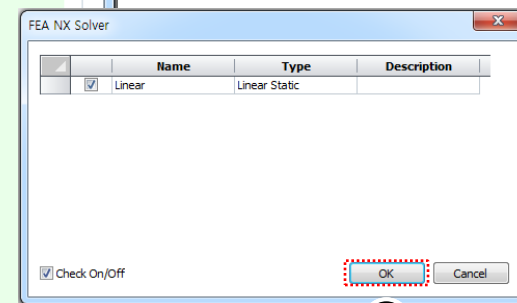
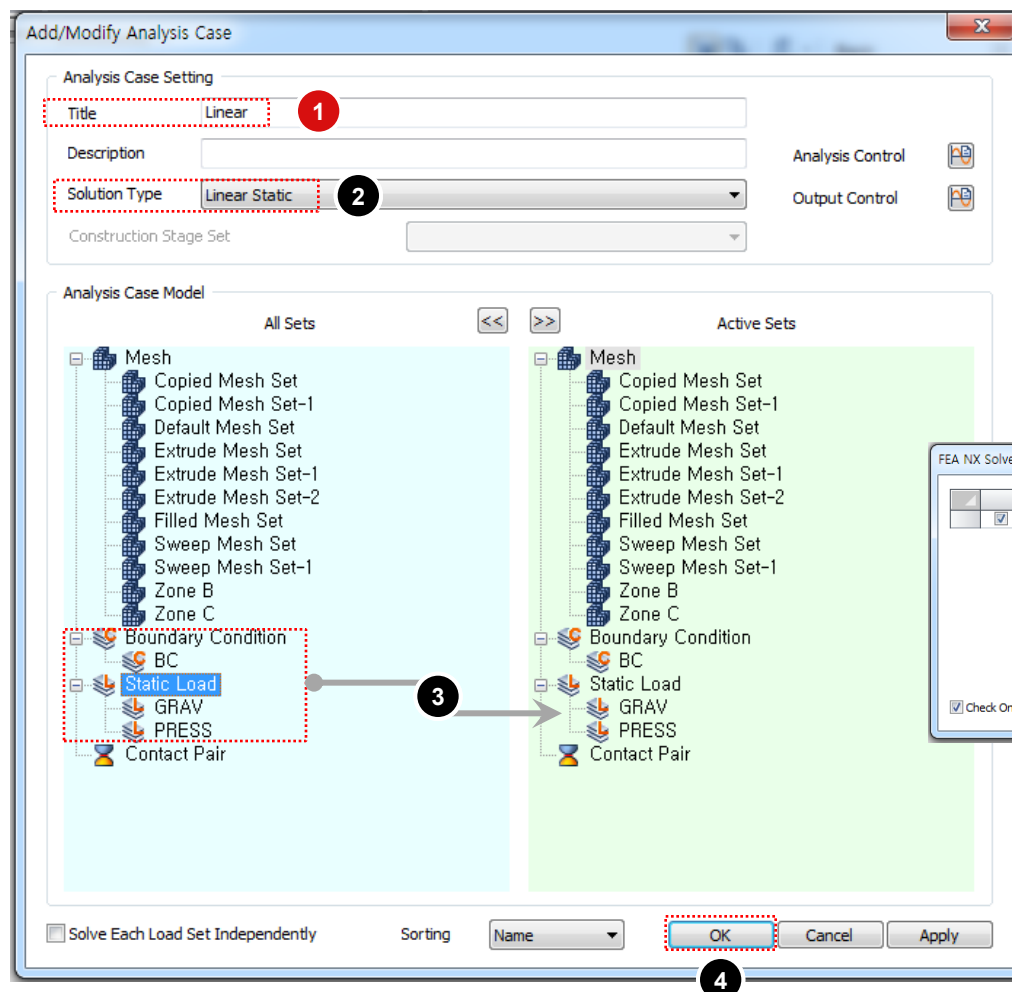
**Procedure**

- 1 [Face] tab
- 2 Load Set : [PRESS]
- 3 Object Type : [3D Element Face]
- 4 Select [Elements] (See Figure)
- 5 Direction : [Normal]
- 6 P or P1: "5" N/cm<sup>2</sup>
- 7 Click [OK] Button



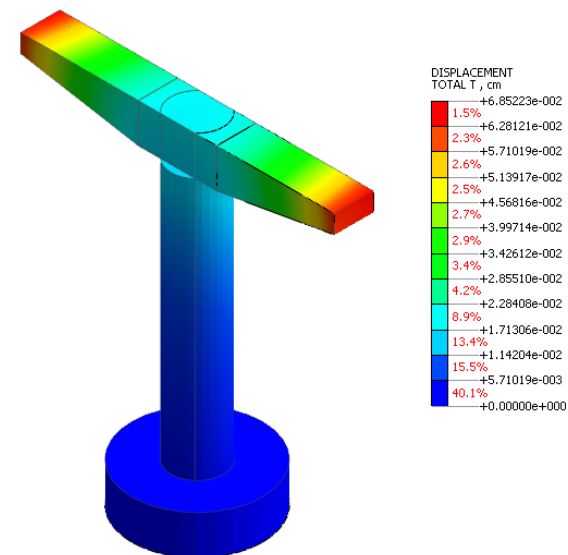
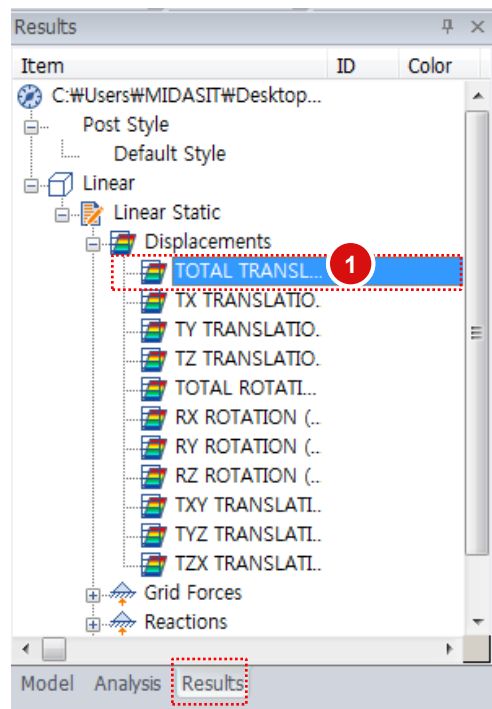
**Procedure**

- 1 Title : **"Linear"**
- 2 Solution Type : **[Linear Static]**
- 3 Drag & Drop **[Boundary Condition]**  
**& [Static Load]**  
to **[Active Sets]** Window
- 4 Click **[OK]** Button
- 5 File > **[Save ... ]**(Pier Map.feb)
- 6 Analysis > **[Perform ...]**
- 7 Click **[OK]** Button



**Procedure**

- 1 Double Click **[TOTAL TRANSL...]**





**Procedure****1 [S-PRINCIPAL A.]**