

Overview

▪ 3-D Linear Static Analysis

▪ Model

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

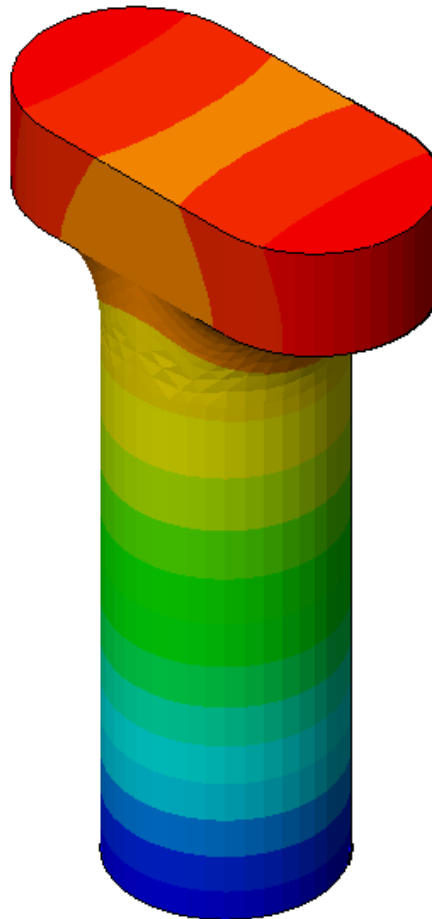
▪ Load & Boundary Condition

- Body Force
- Pressure
- Constraint

▪ Result Evaluation

- Deformation
- 3D Element Principal

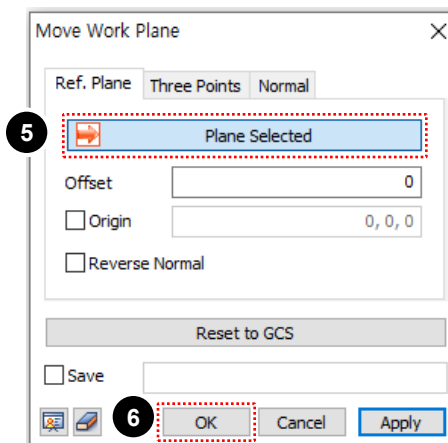
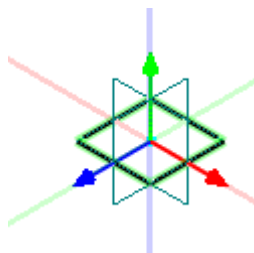
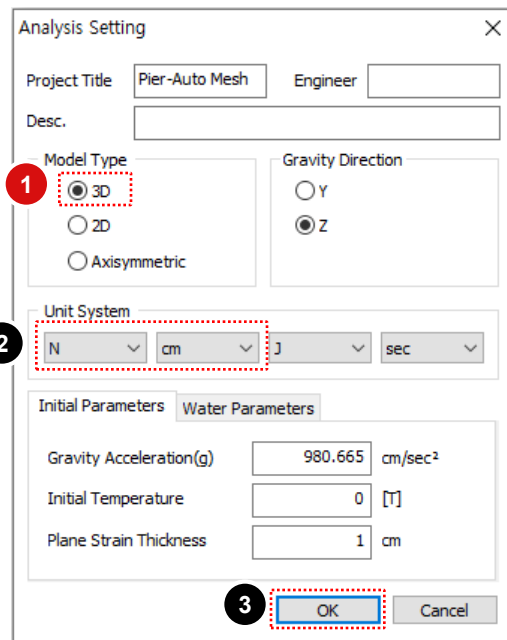
Pier - AutoMesh



Analysis Setting

Procedure

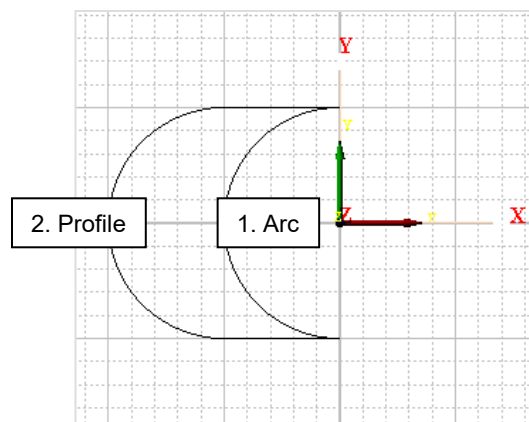
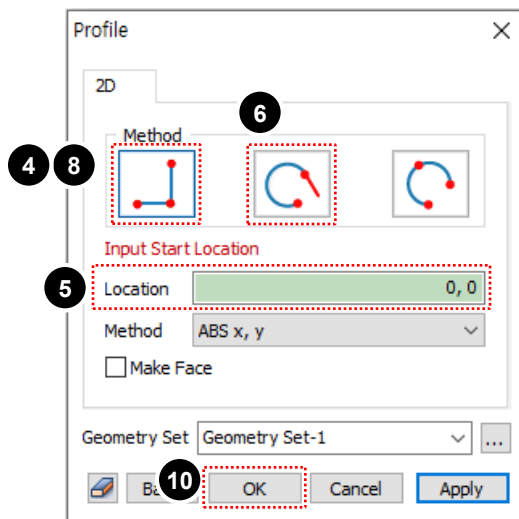
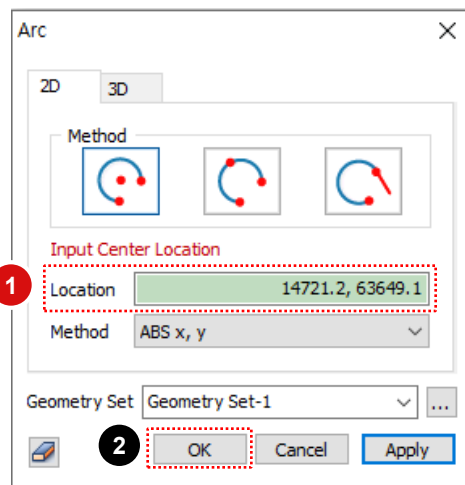
- 1 Model Type : [3D]
- 2 Unit System : [N, cm]
- 3 Click [OK] Button
- 4 Move Work Plane
- 5 Select [XY-Plane]
- 6 Click [OK] Button
- 7 Define Grid
- 8 Width : "20"
- 9 Check on [User Defined] : "30"
- 10 Click [OK] Button



Procedure

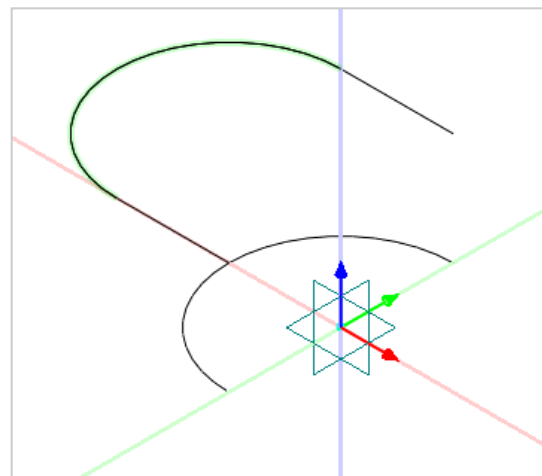
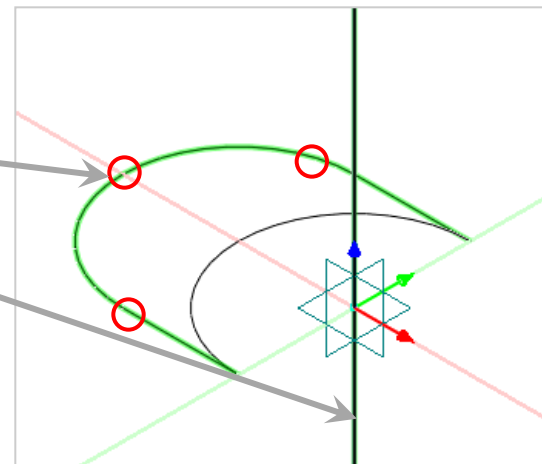
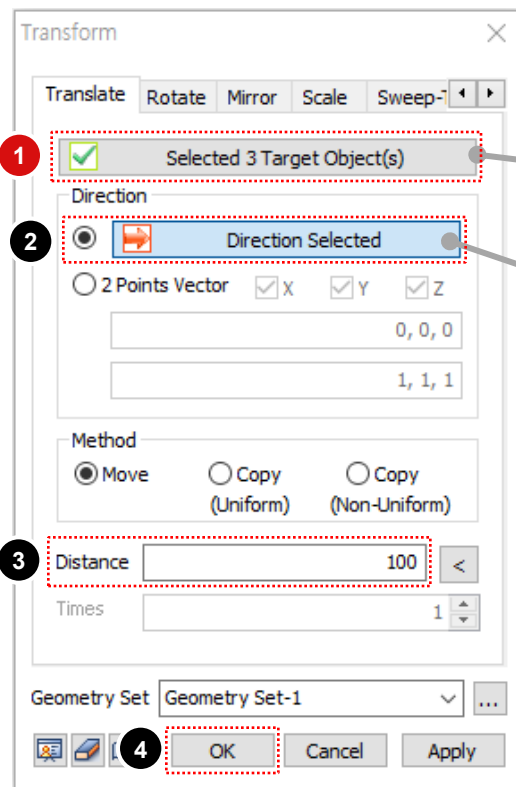
- 1 Location : “(0, (0, 100), (0, -100))”
- 2 Click **[OK]** Button
- 3 Geometry > Point & Curve > **[Profile]**
- 4 Select **[Polyline]**
- 5 Location : “(0, 100) , <-100>”
- 6 Select **[Tangential Arc]**
- 7 Location : “(100, 180)”
- 8 Select **[Polyline]**
- 9 Location : “<-100,0>”
- 10 Click **[OK]** Button

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



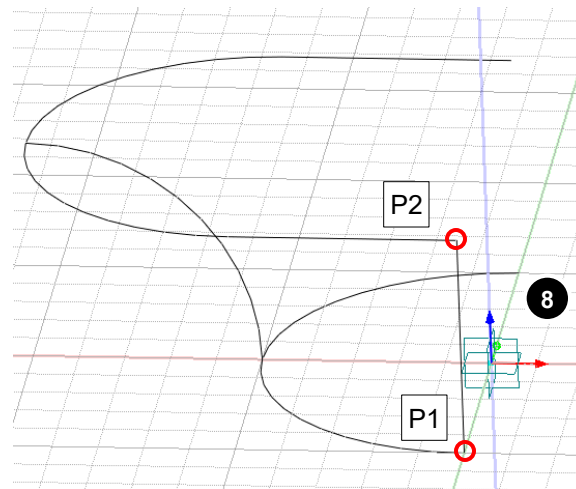
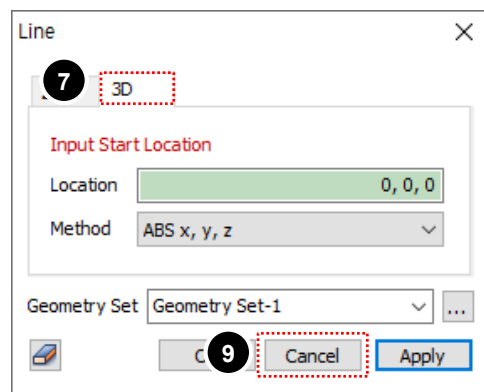
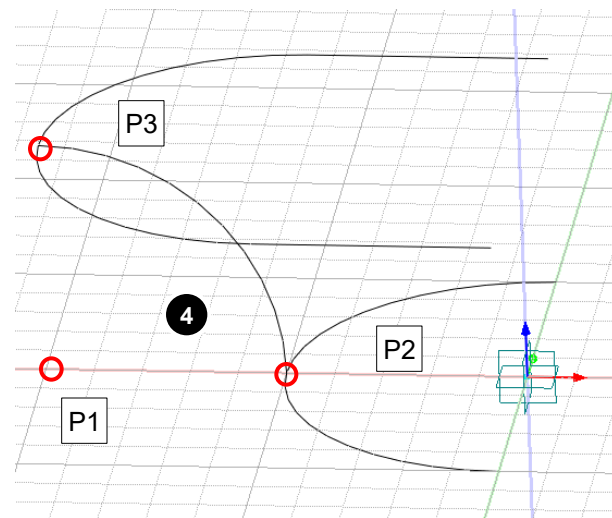
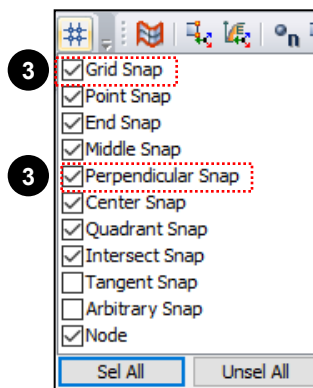
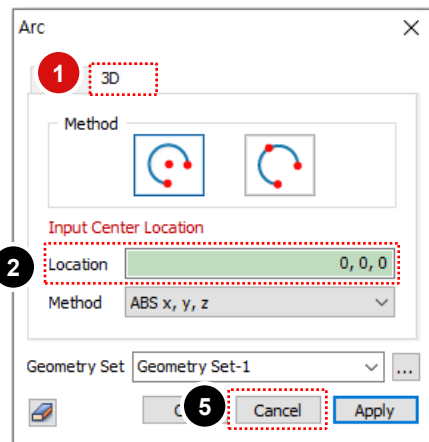
Procedure

- 1 Select Object Shape marked by [] (See Figure)
- 2 Direction : [Z-Axis]
- 3 Distance : "100"
- 4 Click [OK] Button




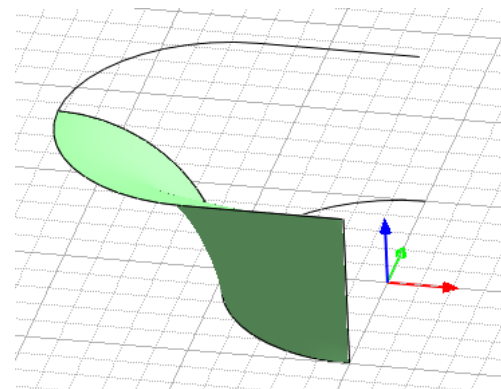
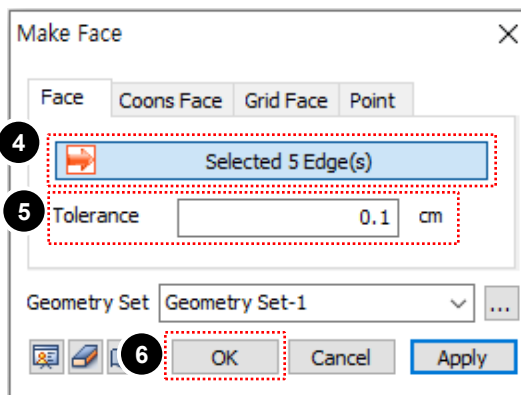
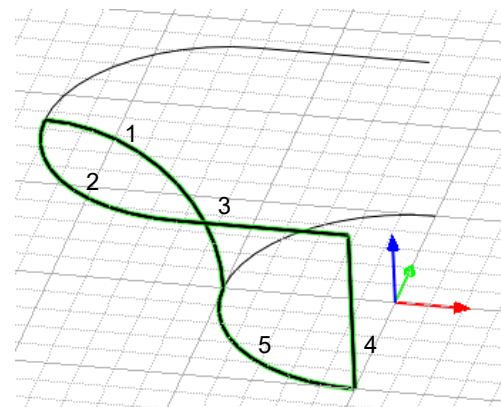
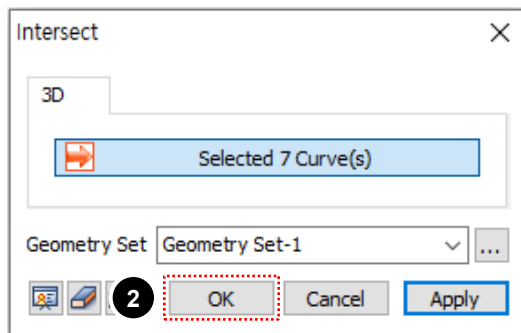
Procedure

- 1 Click **[3D]** tab
- 2 Location : “(-200, 0, 0), (-100, 0, 0), (-200, 0, 100)”
- 3 Toggle on **[Grid Snap]** and **[Quadrant Snap]**
- 4 Select **[P1, P2]** and **[P3]** in sequential order (See Figure)
- 5 Click **[Cancel]** Button
- 6 Geometry > Point & Curve > **[Line]**
- 7 Click **[3D]** tab
- 8 Select **[P1]** and **[P2]**
- 9 Click **[Cancel]** Button



Procedure

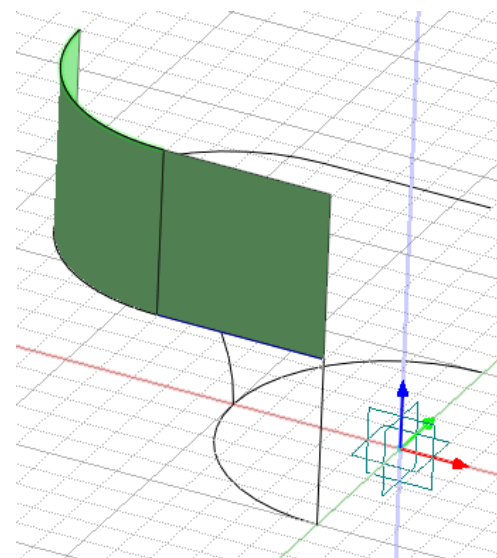
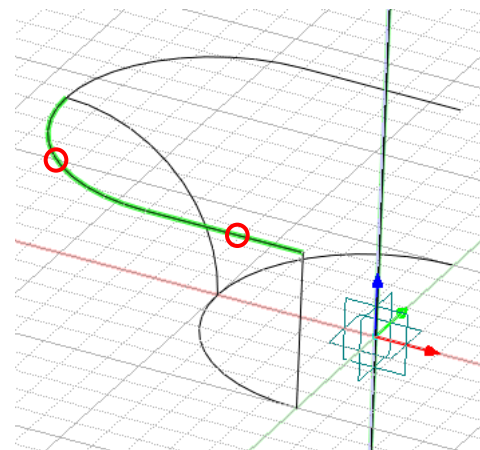
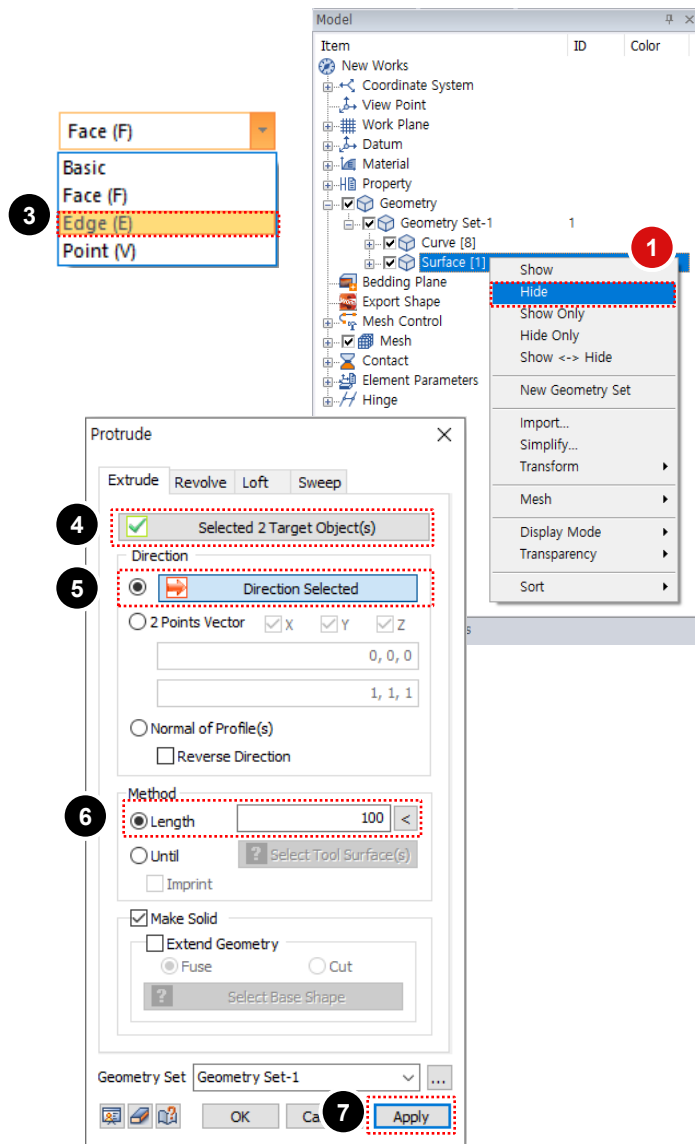
- 1 Select [] Select All
- 2 Click [OK] Button
- 3 Geometry > Surface & Solid >
[Make Face]
- 4 Select [5 Lines] (See Figure)
- 5 Tolerance : "0.1"
- 6 Click [OK] Button



-
- 1 "Enter" as shortcut for "Apply"
 - 2 "Ctrl+A" as shortcut for "Select Displayed".

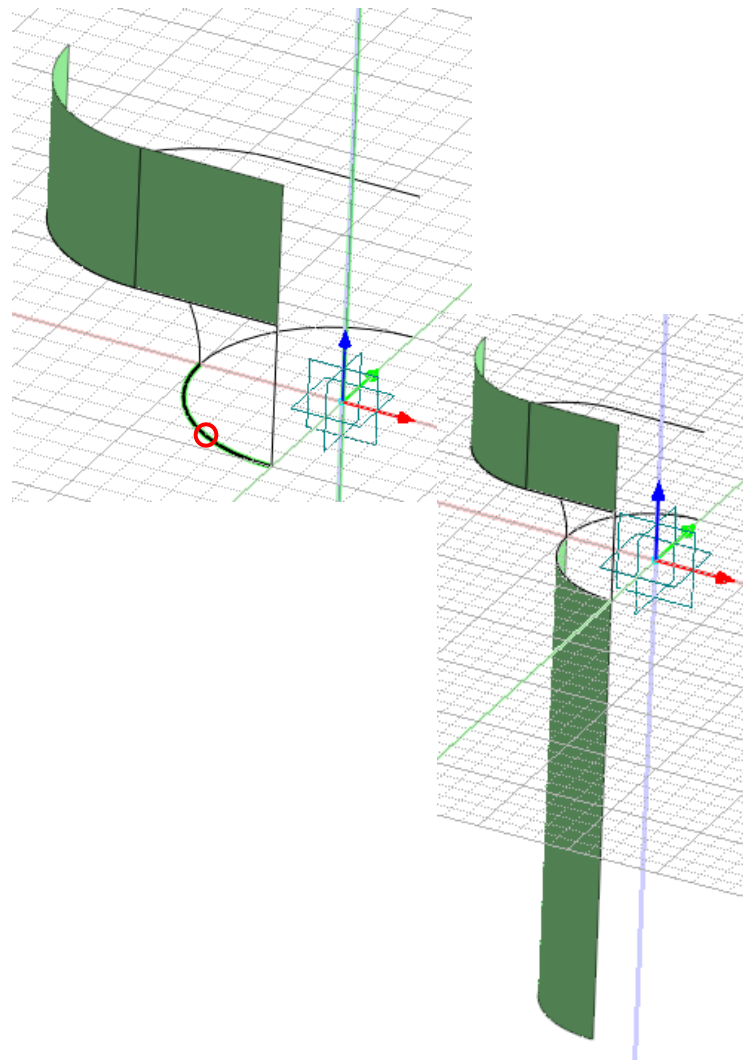
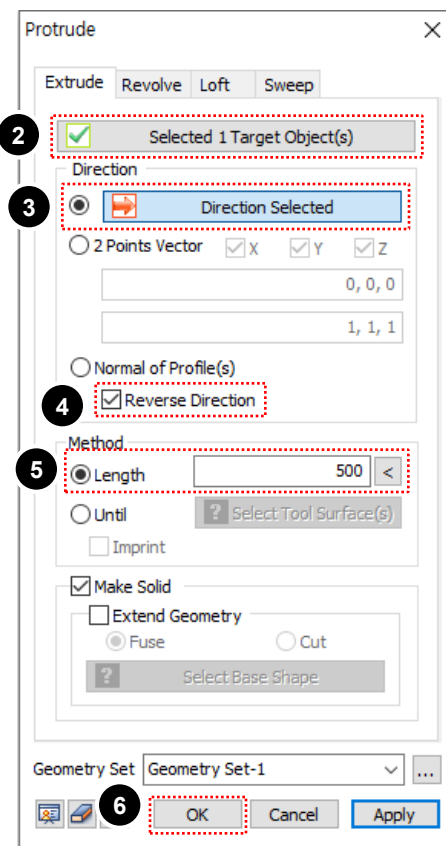
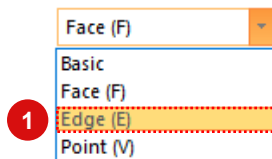
Procedure

- 1 Click Right Mouse Button and Select **[Hide]**
- 2 Geometry > Protrude > **[Extrude]**
- 3 Change Selection Filter to **[Edge (E)]**
- 4 Select **[2 Lines]** (See figure)
- 5 Extrusion Direction : **[Z-axis]**
- 6 Length : **"100"**
- 7 Click **[Apply]** Button





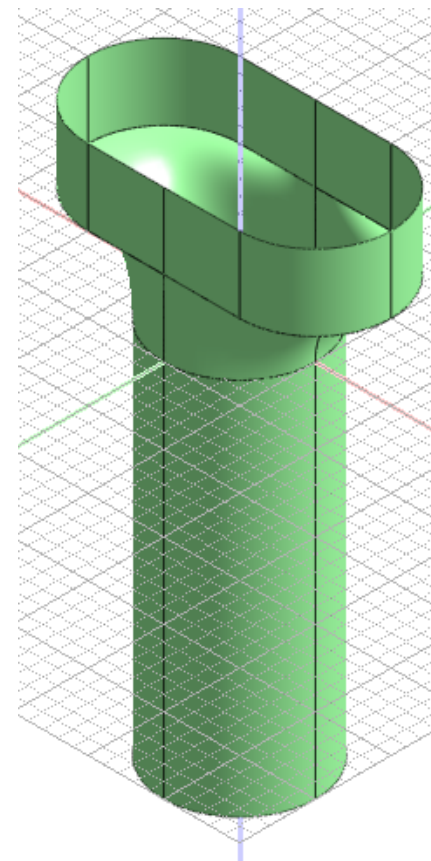
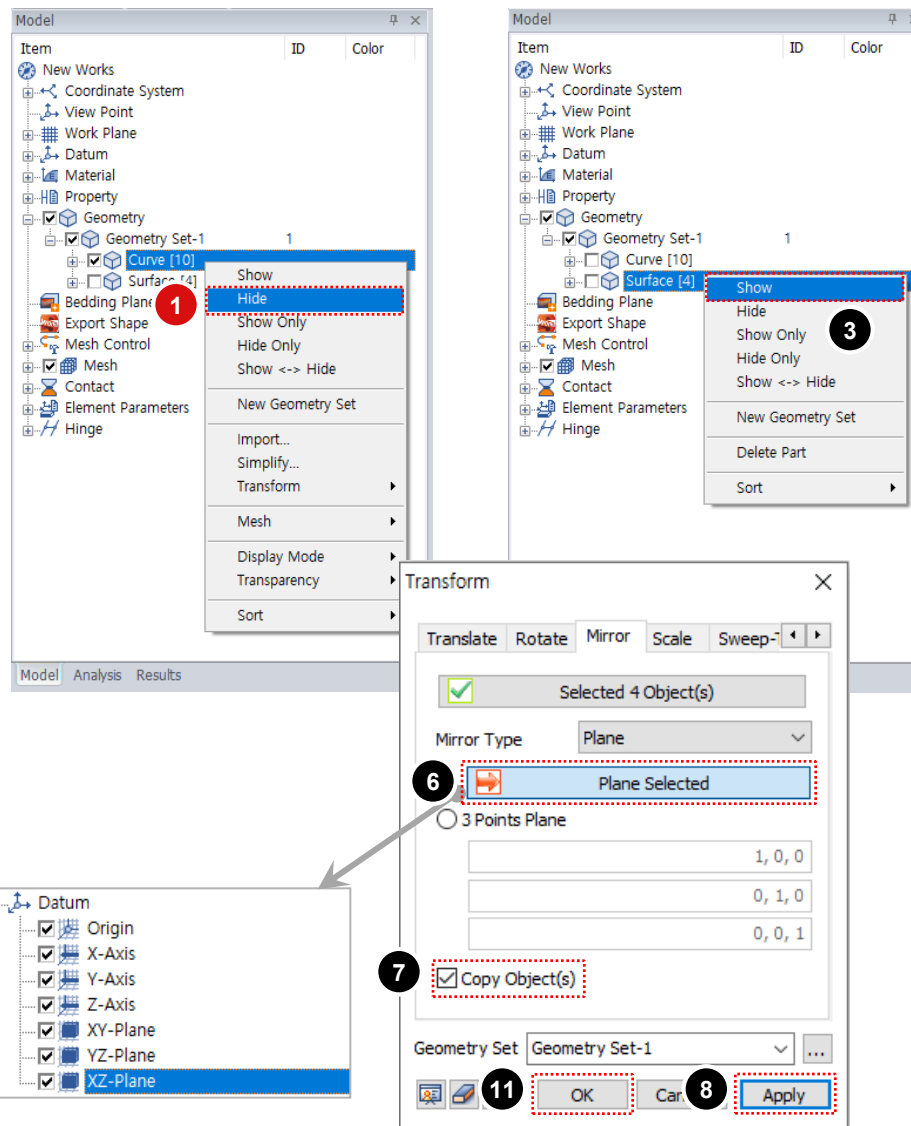
Procedure

- 1 Change Selection Filter to **[Edge(E)]**
- 2 Select **[1 Line]** (See figure)
- 3 Extrusion Direction : **[Z-axis]**
- 4 Check on **[Reverse Direction]**
- 5 Length : **"500"**
- 6 Click **[OK]** Button



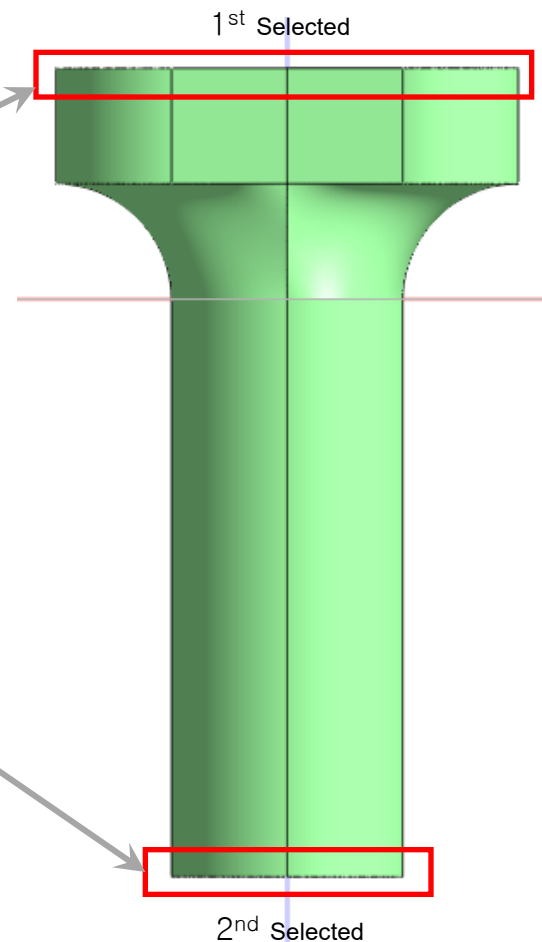
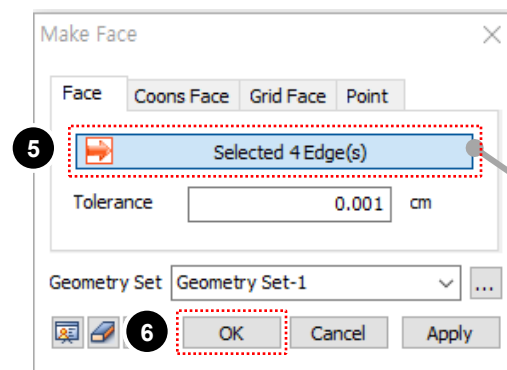
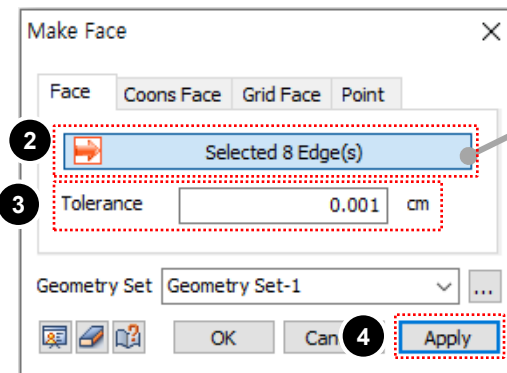
Procedure

- 1 Click Right Mouse Button and Select **[Hide]**
- 2 Model : Geometry > **[Surface]**
- 3 Click Right Mouse Button and Select **[Show]**
- 4 Geometry > Transform > **[Mirror]**
- 5 Select [] Displayed
- 6 Mirror Plane : **[XZ-Plane]**
- 7 Check on **[Copy Object (s)]**
- 8 Click **[Apply]** Button
- 9 Select [] Displayed
- 10 Mirror Plane : **[YZ-Plane]**
- 11 Click **[OK]** Button





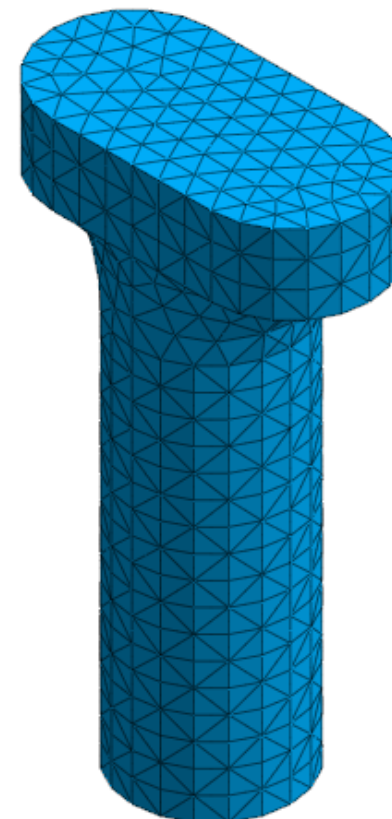
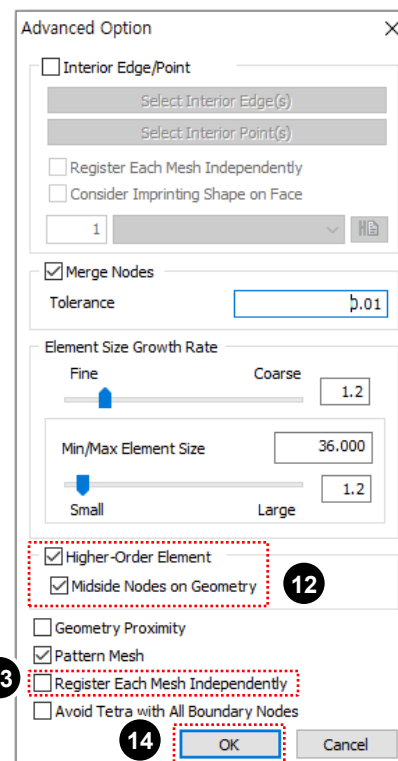
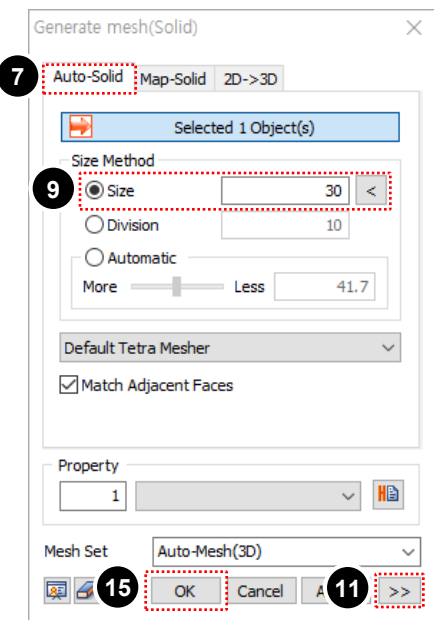
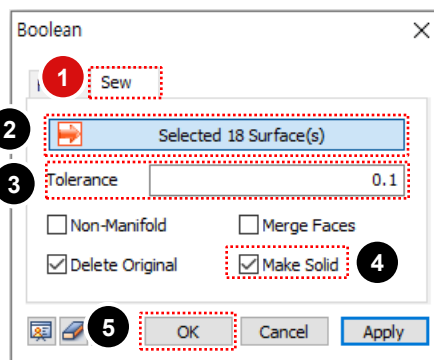
Procedure

- 1 Click **[Front]**
- 2 Select **[8 Curves]** (See Figure)
- 3 Tolerance : "**0.001**"
- 4 Click **[Apply]** Button
- 5 Select **[4 Curves]** (See Figure)
- 6 Click **[OK]** Button




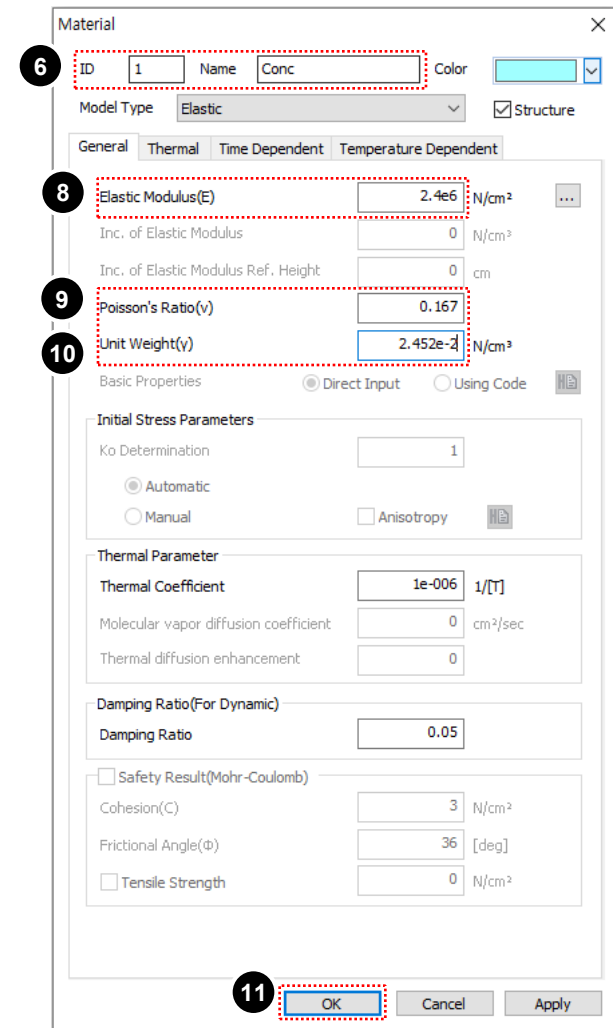
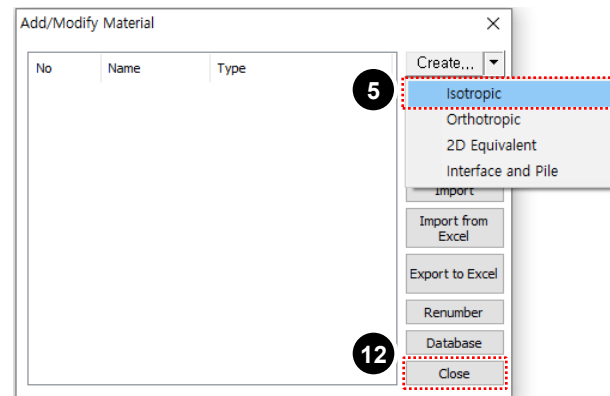
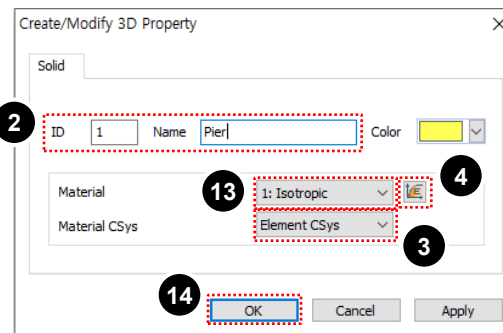
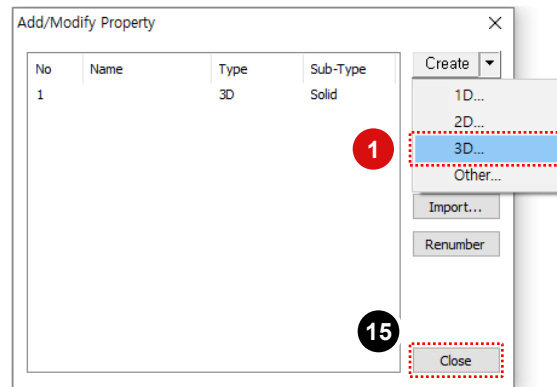
Procedure

- 1 Select **[Sew]** Tab
- 2 Select [] Select All
- 3 Tolerance : “0.1”
- 4 Check on **[Make Solid]**
- 5 Click **[OK]** Button
- 6 Mesh > Generate > **[2D→3D]**
- 7 Select **[Auto-Solid]** Tab
- 8 Select [] Select All
- 9 Size Method : **[Size] “30”**
- 10 Property : “1”
- 11 Click [**>>**] Button
- 12 Check on **[Higher-Order Element]** and **[Midside Nodes on Geometry]**
- 13 Check off **[Register Each Mesh Independently]**
- 14 Click **[OK]** Button
- 15 Click **[OK]** Button



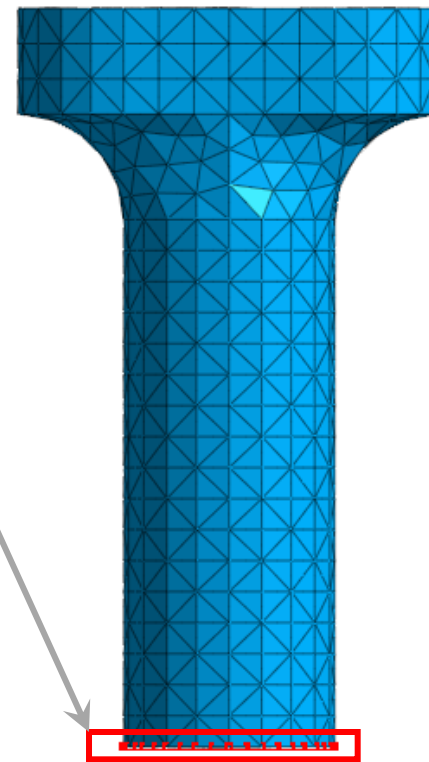
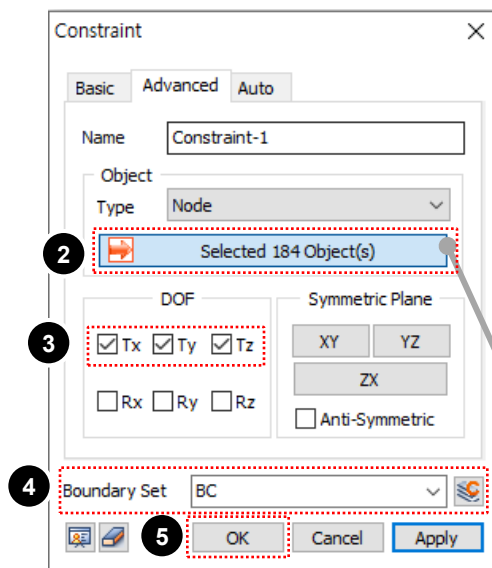
Procedure

- 1 Create [3D]
- 2 ID : "1", Name : "Pier"
- 3 Select : "Element Csys"
- 4 Click  Button (Material)
- 5 Click [Create Isotropic] Button
- 6 ID : "1", Name : "Conc"
- 7 Model Type : "Elastic"
- 8 Elastic Modulus : "2.4e6 N/cm²"
- 9 Poisson's Ratio : "0.167"
- 10 Unit Weight : "2.452e-2 N/cm²"
- 11 Click [OK] Button
- 12 Click [Close] Button
- 13 Select : "1: Isotropic" for Material
- 14 Click [OK] Button
- 15 Click [Close] Button



Procedure

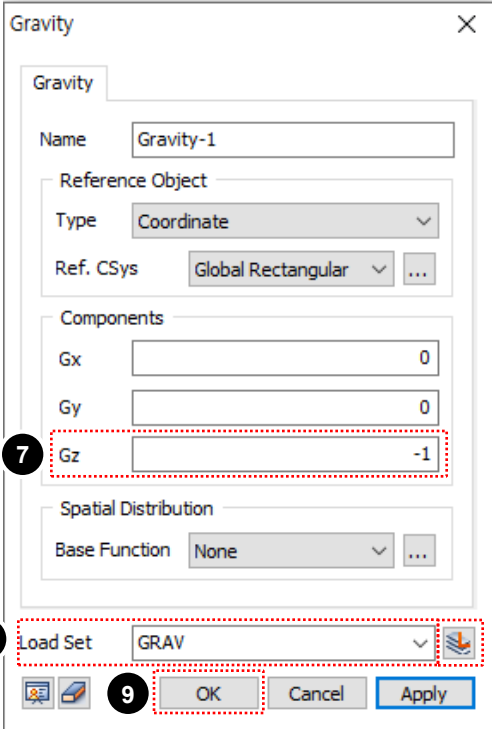
- 1 Click **[Front View]**
- 2 Select **[184 Nodes]** (See Figure)
- 3 Click **[Tx, Ty, Tz]** Button
- 4 BC Set : **[BC]**
- 5 Click **[OK]** Button



Selected

Procedure

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



Gravity dialog box showing settings for Gravity-1. The Gz component is set to -1. The Load Set is set to GRAV. The OK button is highlighted.

Gravity

Name: Gravity-1

Reference Object

Type: Coordinate

Ref. CSys: Global Rectangular

Components

Gx: 0

Gy: 0

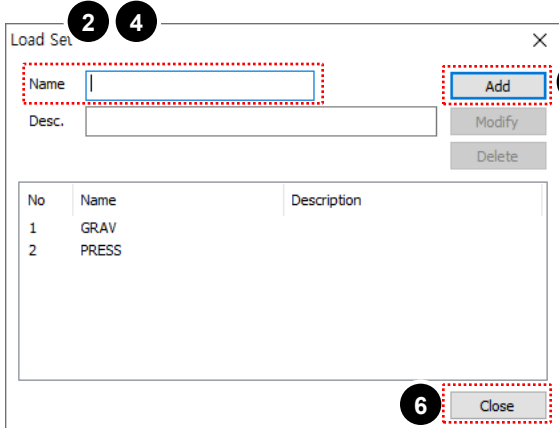
Gz: -1

Spatial Distribution

Base Function: None

Load Set: GRAV

OK Cancel Apply



Load Set dialog box showing the list of load sets. The Name field is empty. The Add button is highlighted. The Close button is highlighted.

Load Set

Name:

Desc:

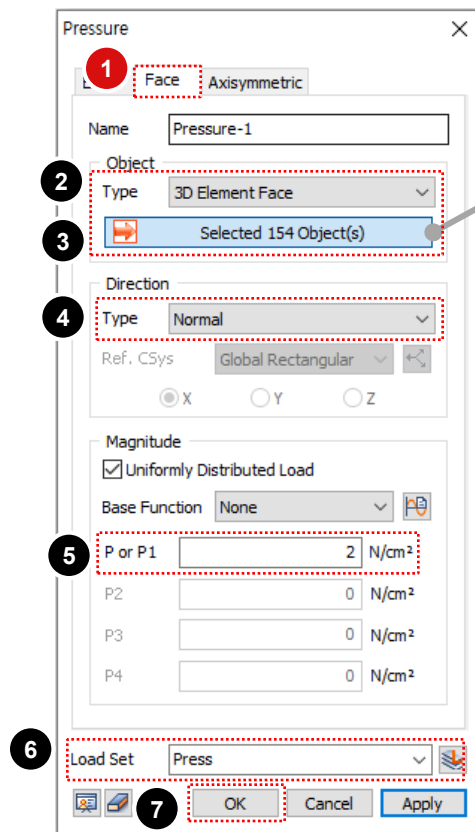
Add Modify Delete

No	Name	Description
1	GRAV	
2	PRESS	

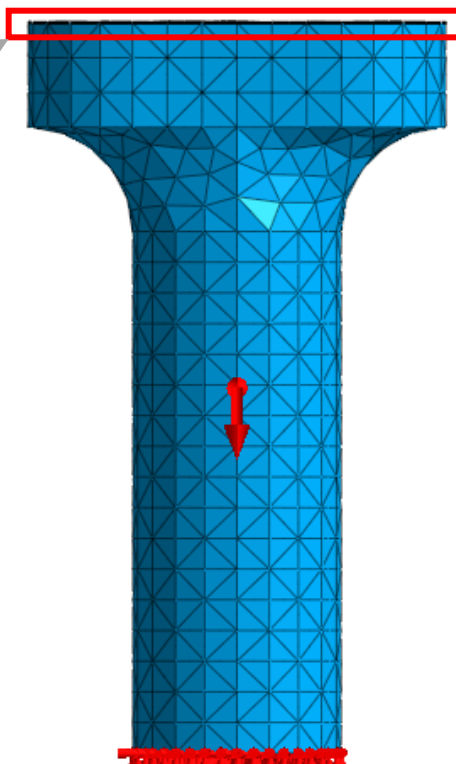
Close

Procedure

- 1 Select **"Face"**
- 2 Object Type : **"3D Element Face"**
- 3 Select 154 Elements (See Figure)
- 4 Direction : **"Normal"**
- 5 P or P1 : **"2"**
- 6 Load Set : **"Press"**
- 7 Click **[OK]** Button

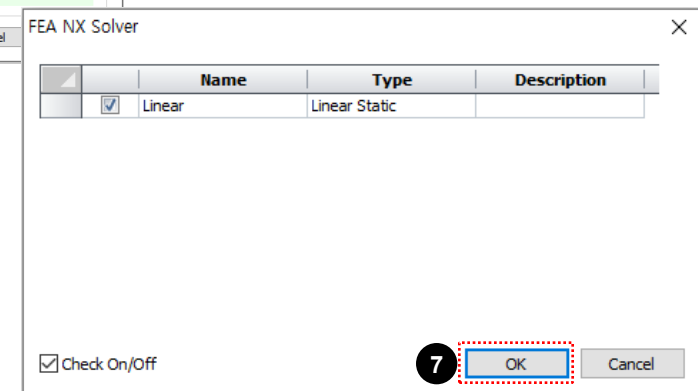
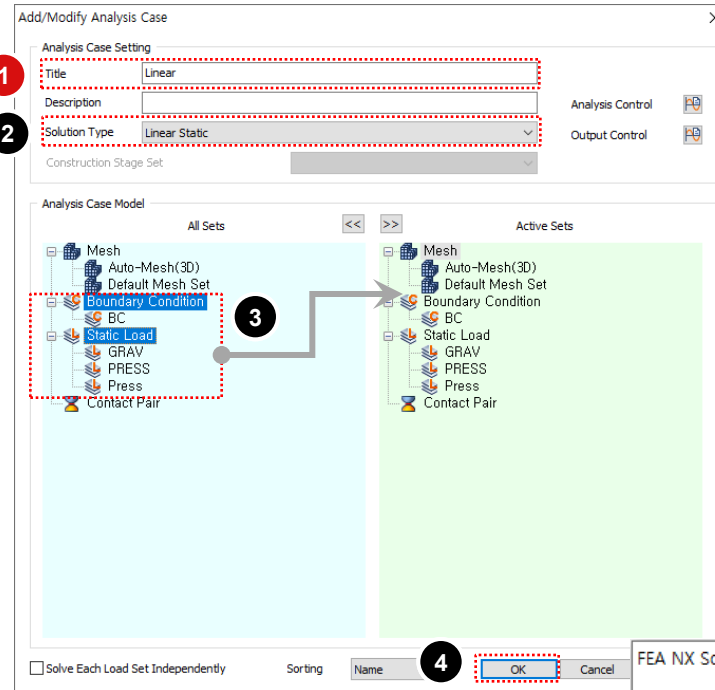


Selected



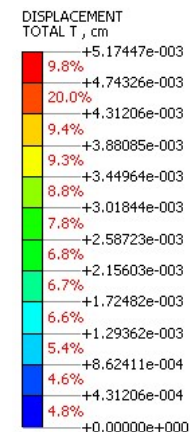
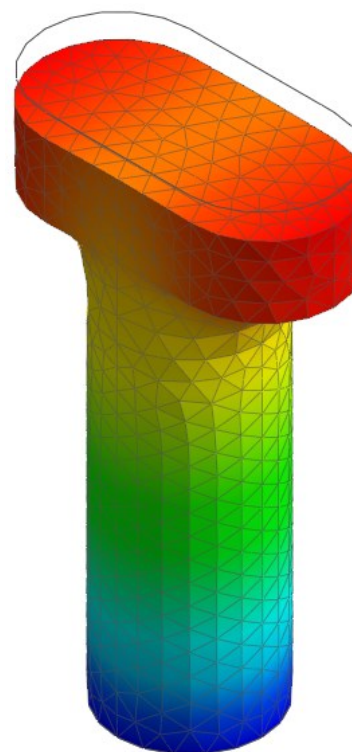
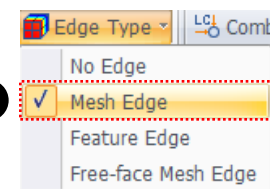
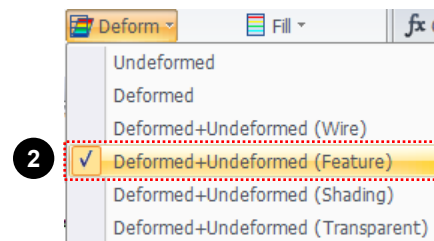
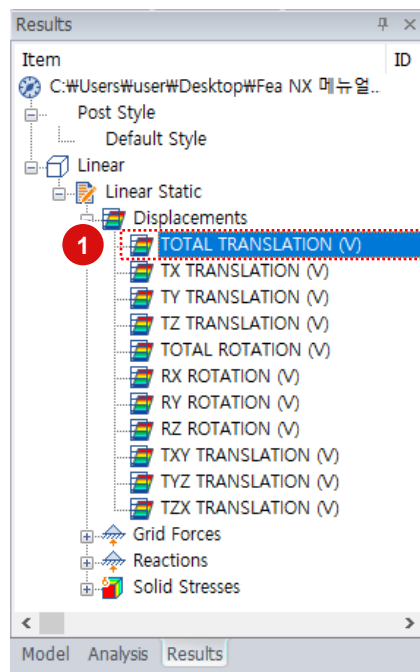
Procedure

- 1 Title : **"Linear"**
- 2 Analysis Type : **[Linear Static]**
- 3 Drag & Drop **[Static Load]** and **[Boundary Condition]** to **[Active Sets]** Window
- 4 Click **[OK]** Buttons
- 5 File > **Save : "Pier-Auto Mesh"**
- 6 Analysis > **Perform**
- 7 Click **[OK]** Button



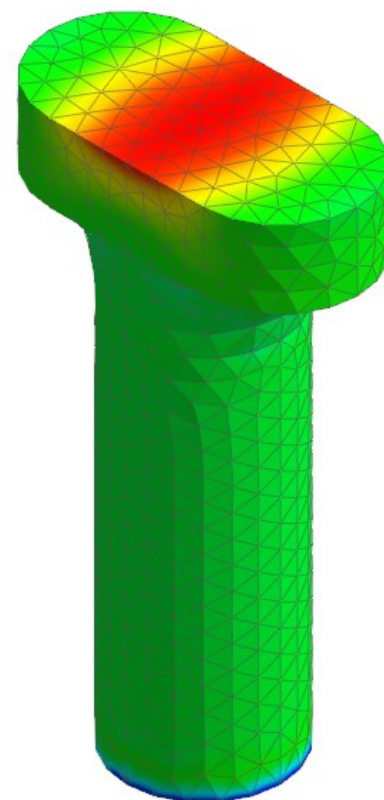
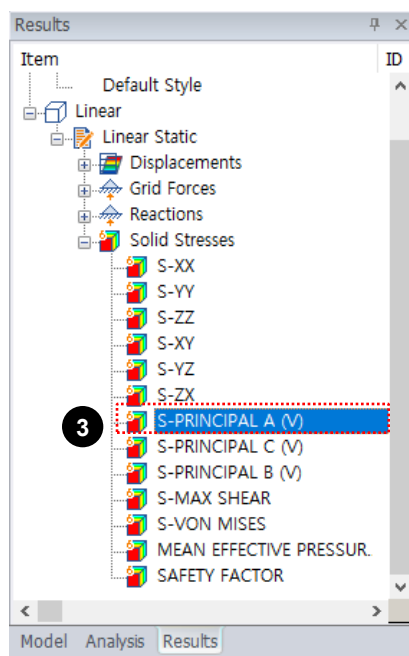
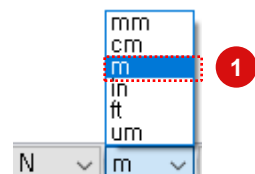
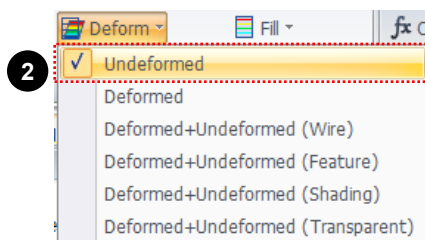
Procedure

- 1 Click **[TOTAL TRANSLATION (V)]**
- 2 Select **[Deformed+Undeformed (Feature)]** for Mesh Shape
- 3 Select **[Feature Edge]**



Procedure

- 1 Unit System : **[N, m]**
- 2 Select **[Undeformed]** for Mesh Shape
- 3 Double Click **[S-PRINCIPAL A (V)]**



SOLID STRESS
S-PRINCIPAL A , N/m²

1.2%	+7.04110e+004
2.1%	+6.01187e+004
3.5%	+4.98263e+004
5.4%	+3.95339e+004
8.6%	+2.92415e+004
15.1%	+1.89492e+004
41.0%	+8.65678e+003
13.7%	-1.63559e+003
4.0%	-1.19280e+004
2.5%	-2.22203e+004
1.8%	-3.25127e+004
1.1%	-4.28051e+004
	-5.30975e+004