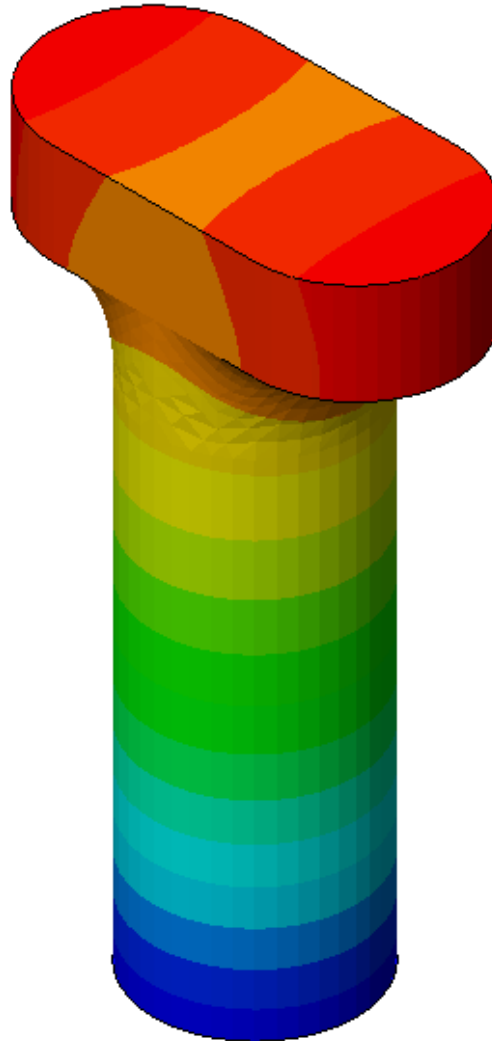


## LS-8. Pier – AutoMesh



### 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 Stress

## Step 1.

1. Analysis > Analysis Control – “Control” tab

2. Analysis Type : 3D

3. Click  Button (Unit System)

4. Length : cm

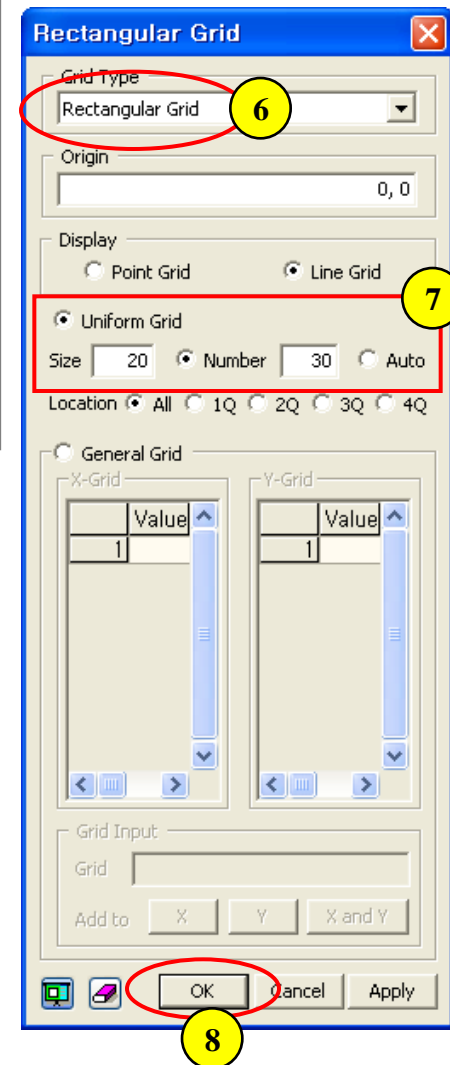
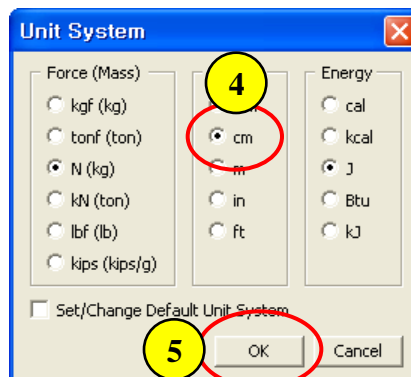
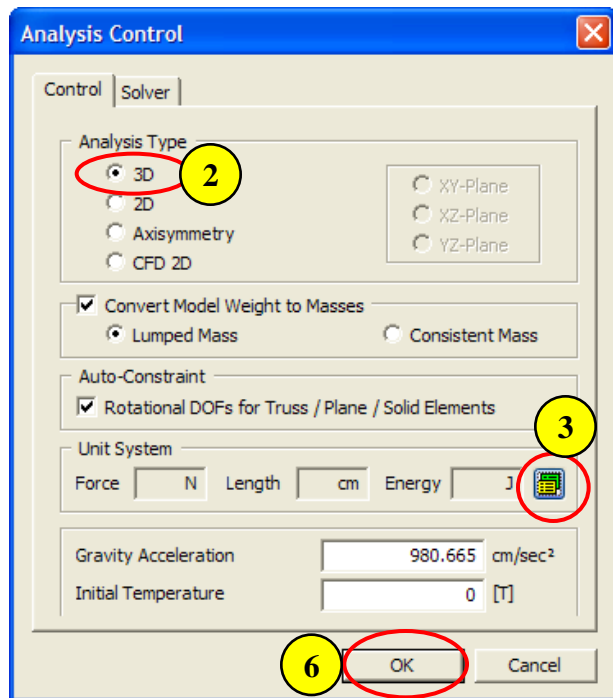
5. Click “OK” Button

6. Geometry > Work Plane > Grid Setting ...

7. Grid Type : Rectangular Grid

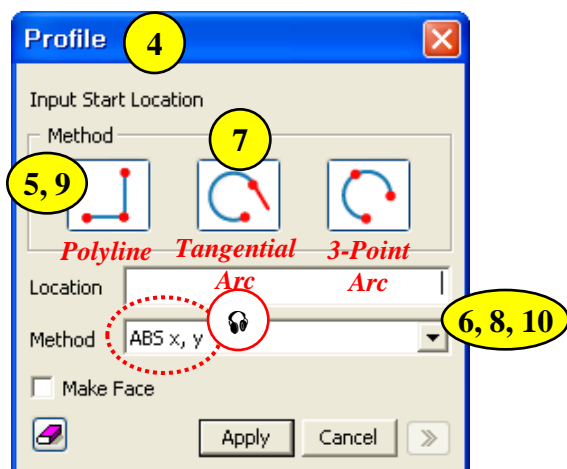
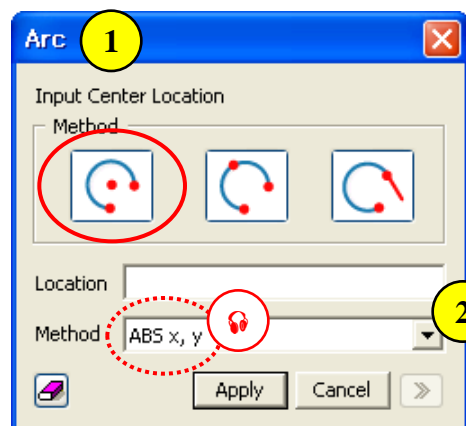
8. Size : 20 , Number : 30

9. Click “OK” Button



 Analysis Control Dialog is automatically activated at startup.

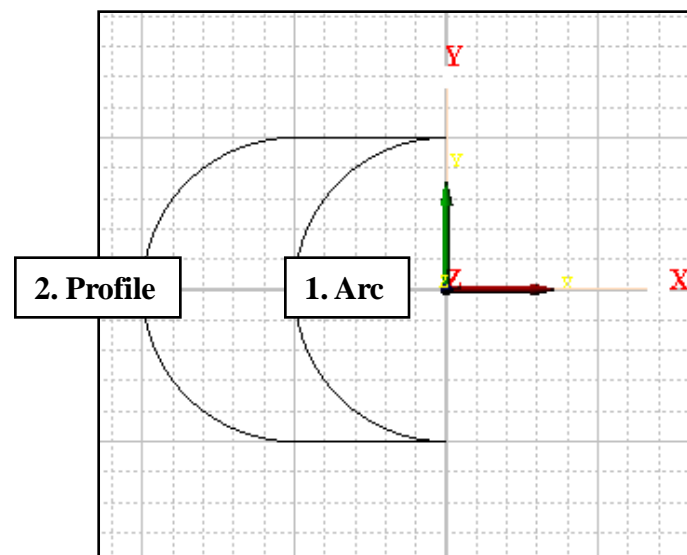
## Step 2.



🔊  $()$  : “ABS x, y”,  $\langle \rangle$  : “REL dx, dy”  
 $(0)$  same as  $(0, 0)$

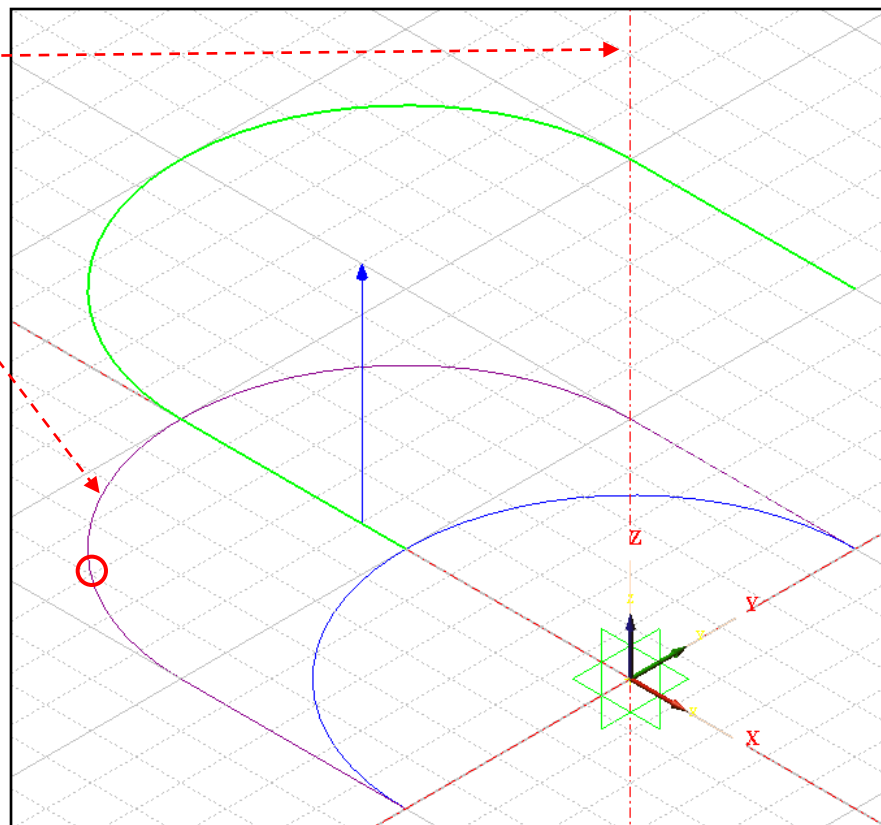
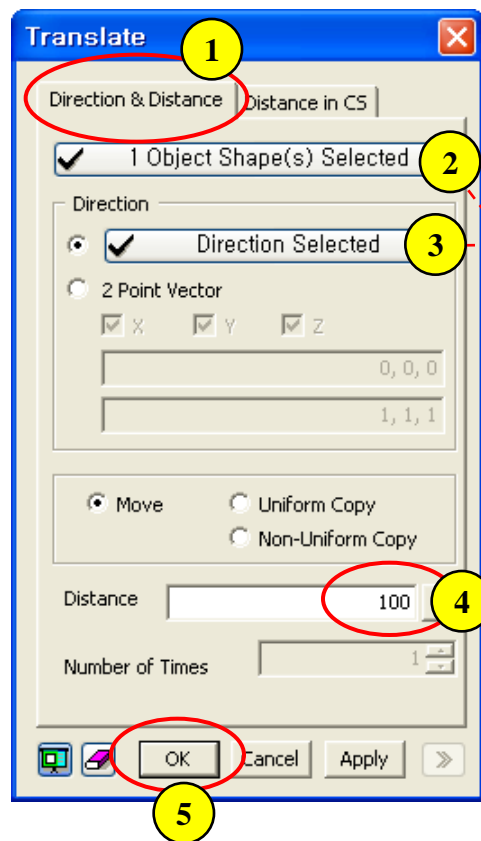
🔊 “Esc” as shortcut for “Cancel”.

1. Geometry > Curve > Create on WP > Arc...
2. Location :  $(0), (0, 100), (0, -100)$  🗣
3. Click [Cancel] Button 🗣
4. Geometry > Curve > Create on WP > Profile (Wire)...
5. Select “Polyline”
6. Location :  $(0, 100), \langle -100 \rangle$
7. Select “Tangential Arc”
8. Location :  $(100, 180)$
9. Select “Polyline”
10. Location :  $\langle 100, 0 \rangle$
11. Click [Cancel] Button

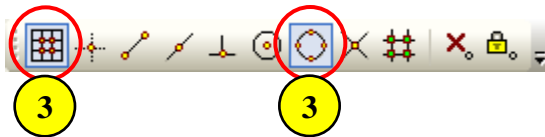
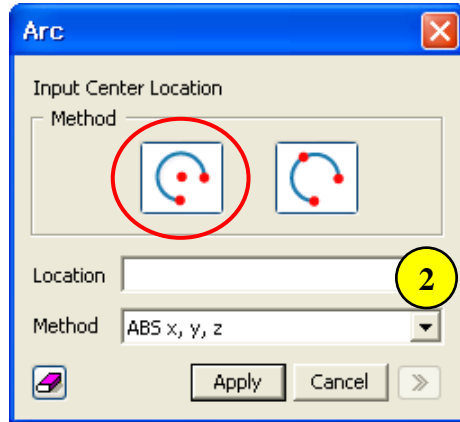


### Step 3.

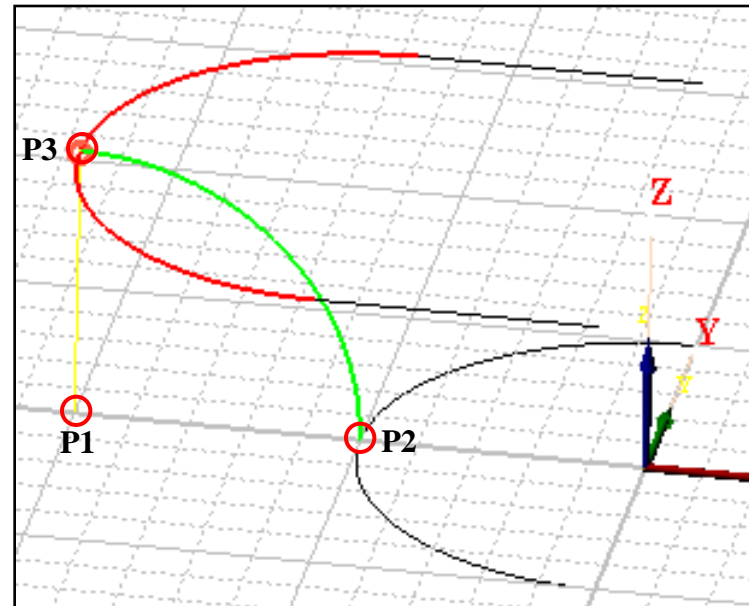
1. Geometry > Transform > Translate : “Direction & Distance” tab
2. Select Object Shape marked by “O” (See Figure)
3. Direction : Z-Axis
4. Distance : 100
5. Click [OK] Button



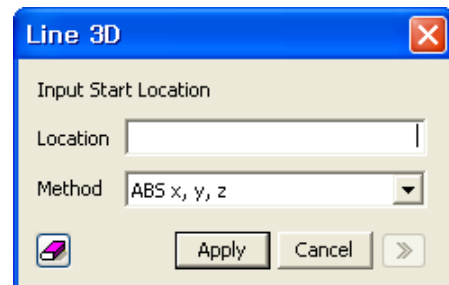
## Step 4.



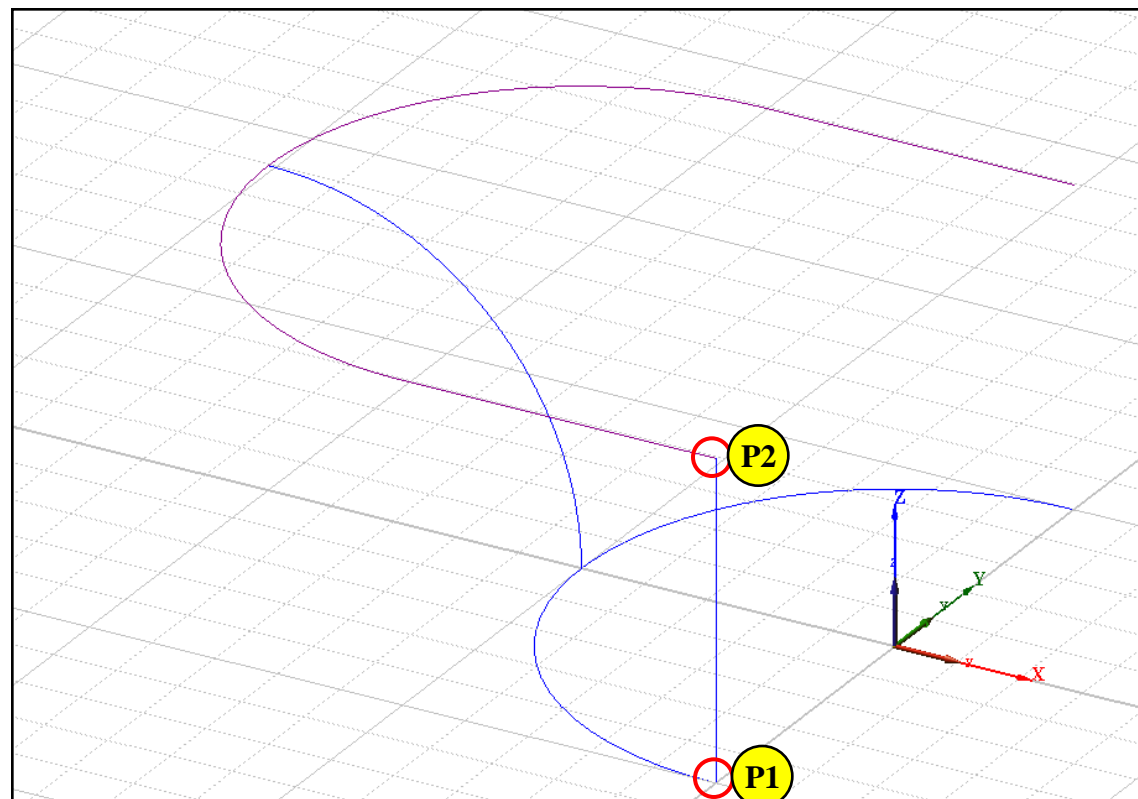
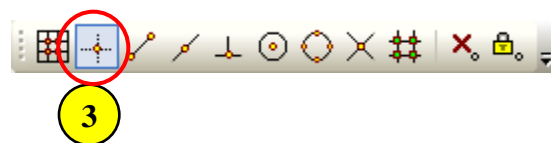
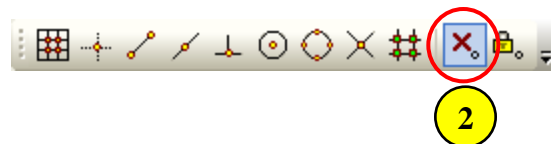
1. *Geometry > Curve > Create 3D > Arc...*
2. *Location : (-200, 0, 0), (-100, 0, 0), (-200, 0, 100)*
- Or
3. *Toggle on “Grid Snap” and “Quadrant Snap”*
4. *Select P1, P2 and P3 in sequential order (See Figure)*
5. *Click “Cancel” Button*



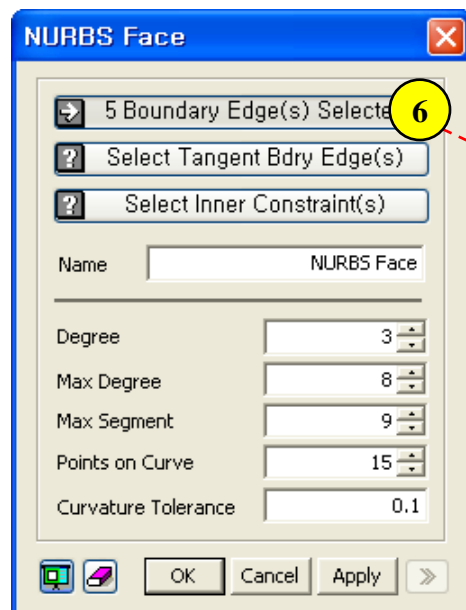
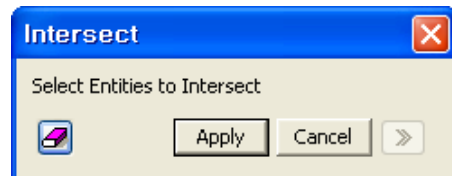
## Step 5.




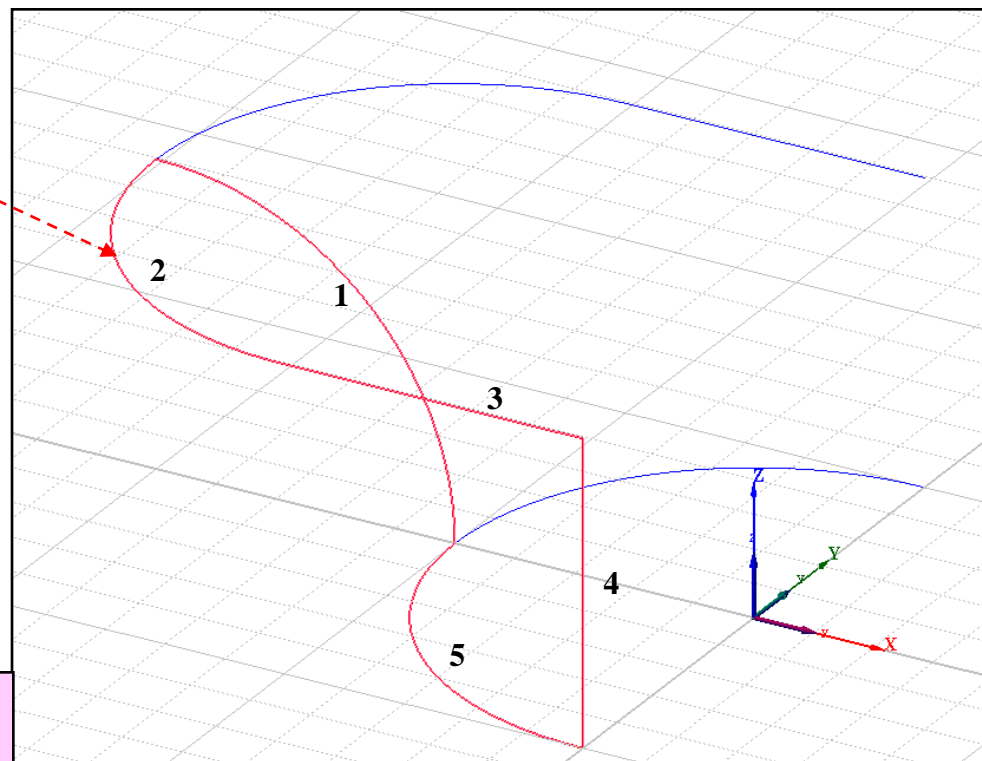
1. Geometry > Curve > Create 3D > Line ...
2. Click “Off All Snaps” Button
3. Toggle on “Vertex Snap”
4. Select “P1” and “P2”
5. Click [Cancel] Button



## Step 6.

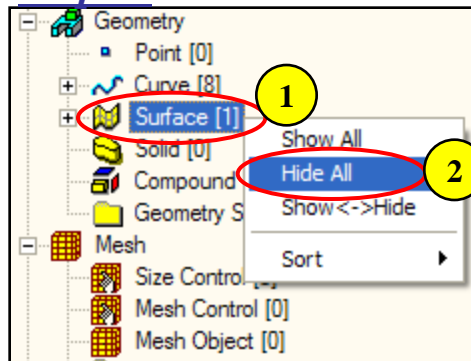


1. **Geometry > Curve > Intersect ...**
2. Select  **“Displayed”**
3. Click [Apply] Button
4. Click [Cancel] Button
5. **Geometry > Surface > Create > NURBS Face...**
6. **Select 5 Lines (See Figure)**
7. Click [OK] Button

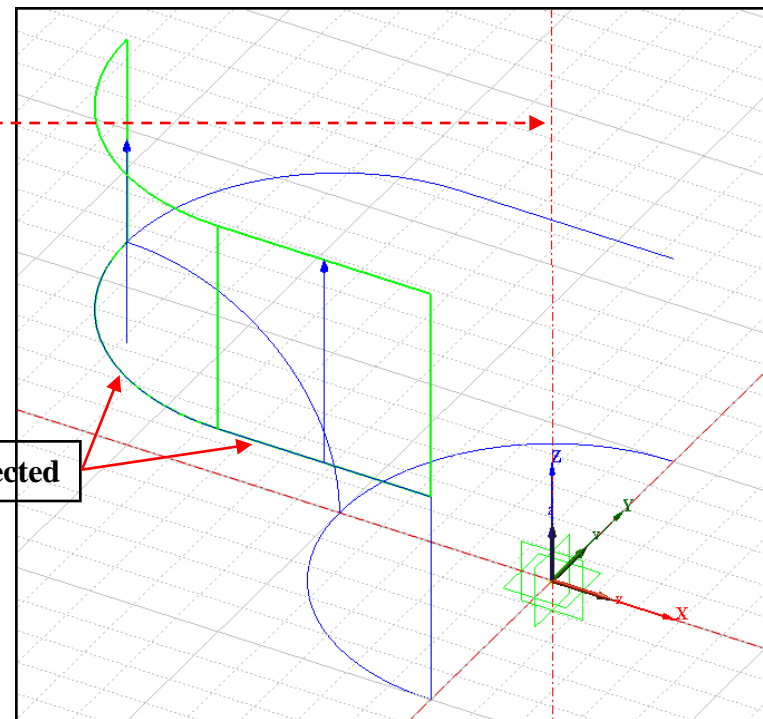
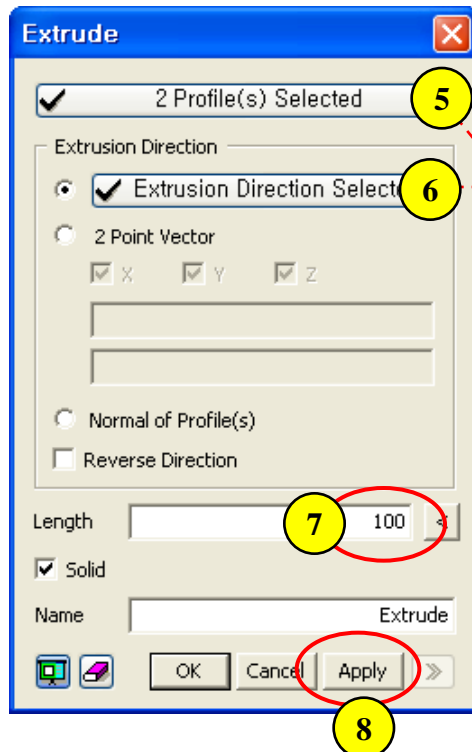


- 🔊 **“Enter” as shortcut for “Apply”**
- 🔊 **“Ctrl+A” as shortcut for “Select Displayed”.**

## Step 7.

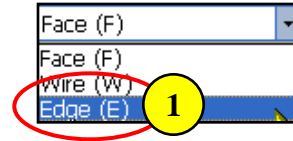
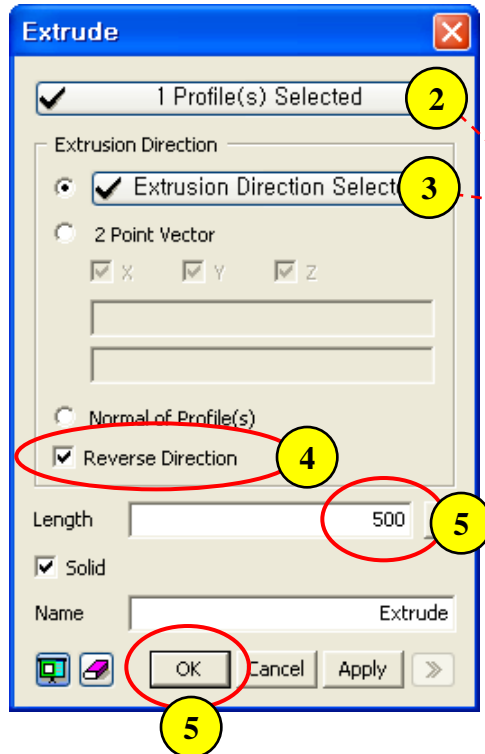


1. Pre-Works Tree : Geometry > Surface
2. Click Right Mouse Button and Select “Hide All”
3. Geometry > Generator Feature > Extrude ...
4. Change Selection Filter to “Edge (E)”
5. Select 2 Lines (See figure)
6. Extrusion Direction : Z-axis
7. Length : 100
8. Click [Apply] Button

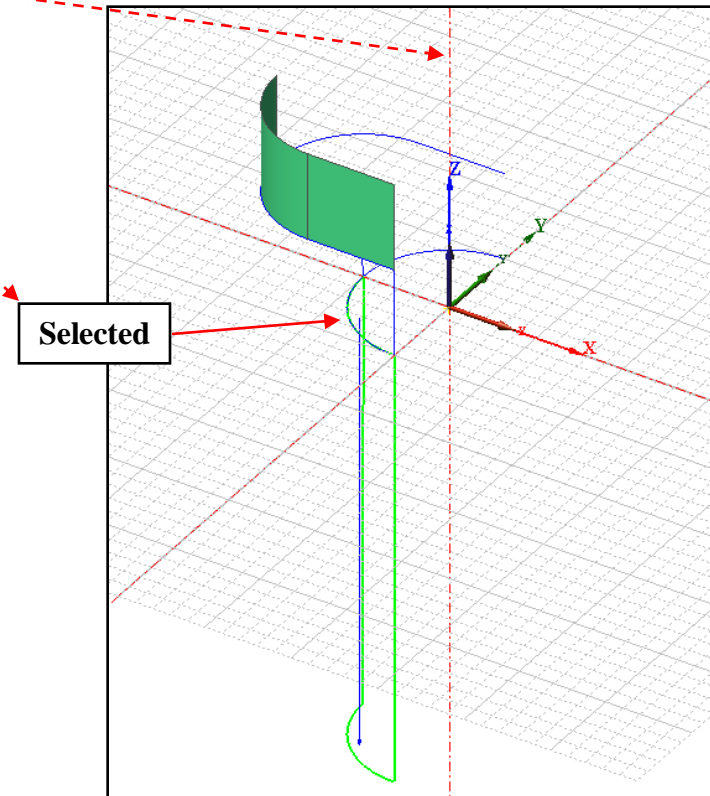




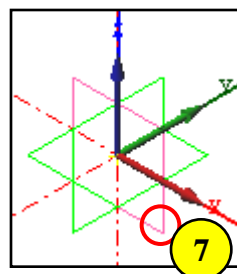
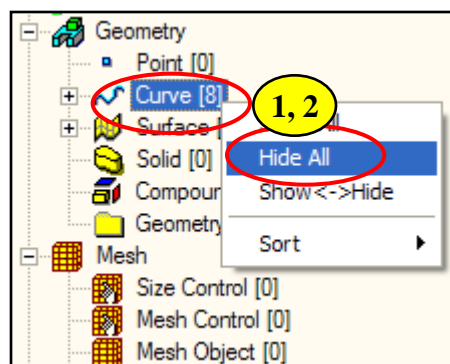
## Step 8.




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

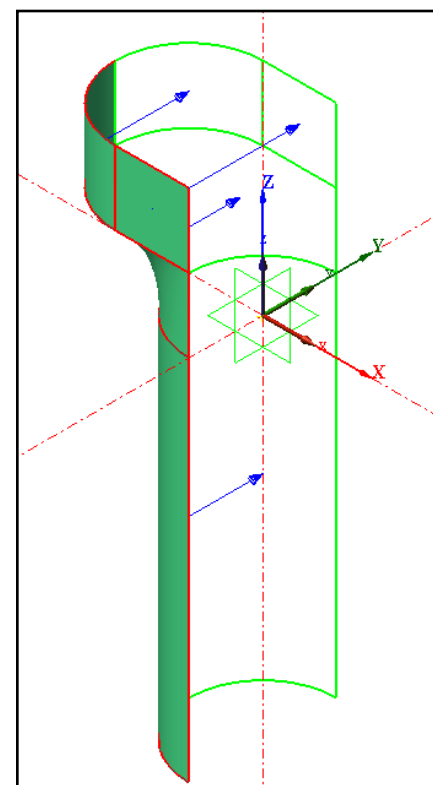
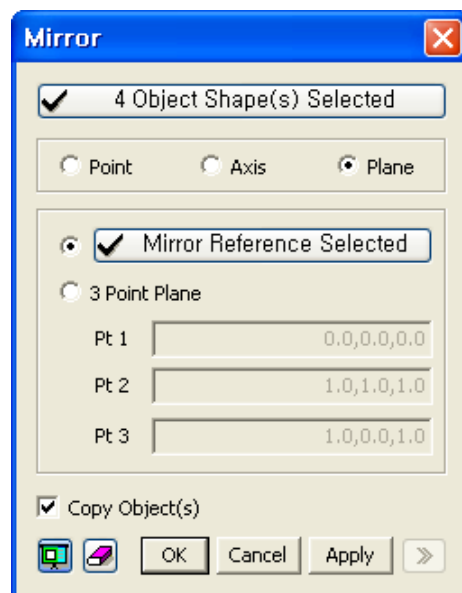


## Step 9.

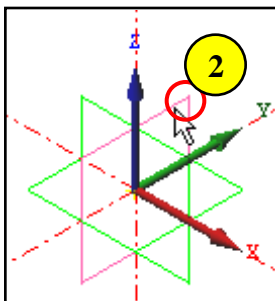
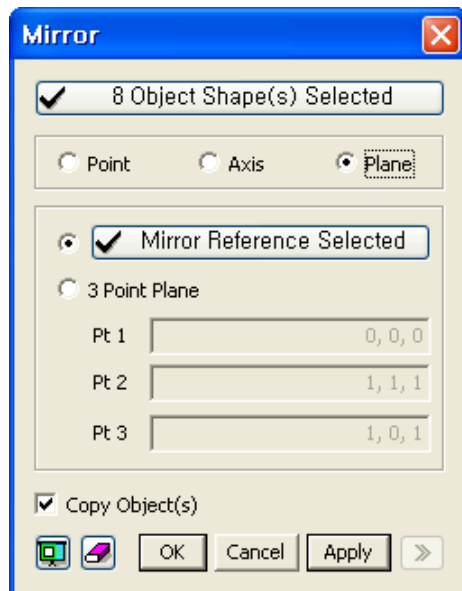


Select "XZ-Plane" in Work Window or Pre-Works Tree

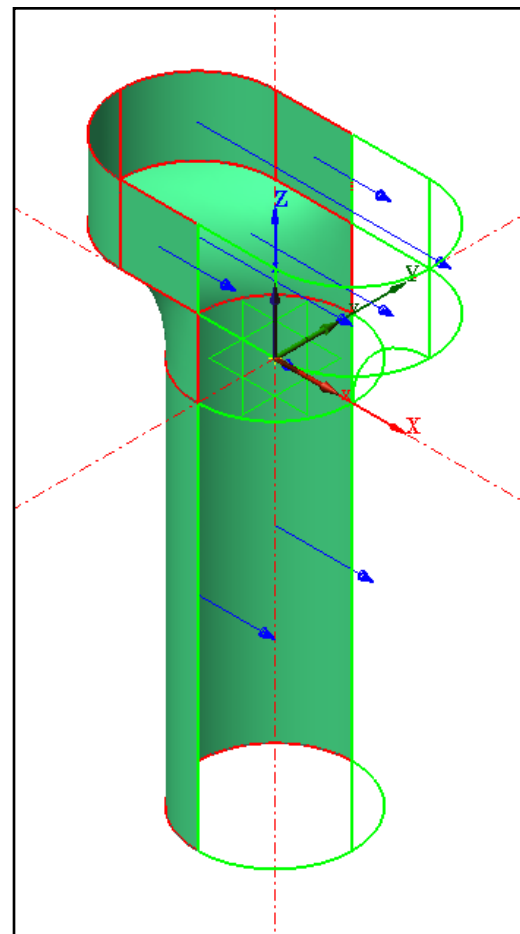
1. Pre-Works Tree : Geometry > Curve
2. Click Right Mouse Button and Select "Hide All"
3. Pre-Works Tree : Geometry > Surface
4. Click Right Mouse Button and Select "Show All"
5. Geometry > Transform > Mirror ...
6. Select  "Displayed"
7. Mirror Plane : XZ-Plane
8. Click [Apply] Button




## Step 10.

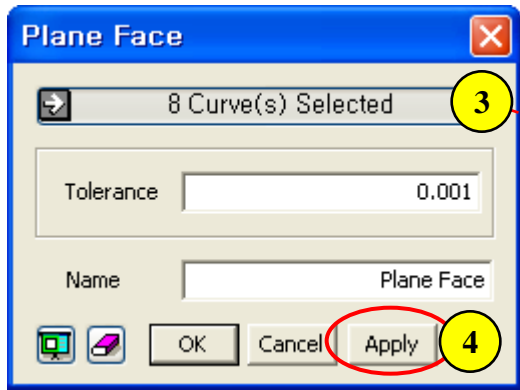


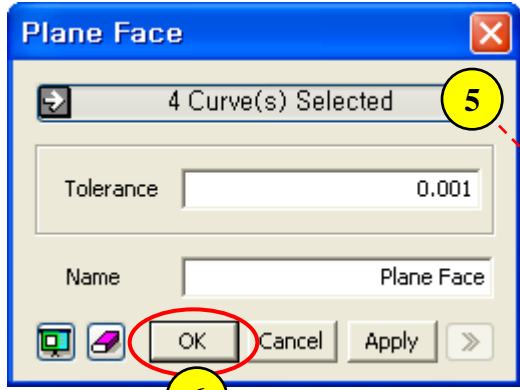
1. Select  "Displayed"
2. Mirror Plane : YZ-Plane
3. Click "OK" Button



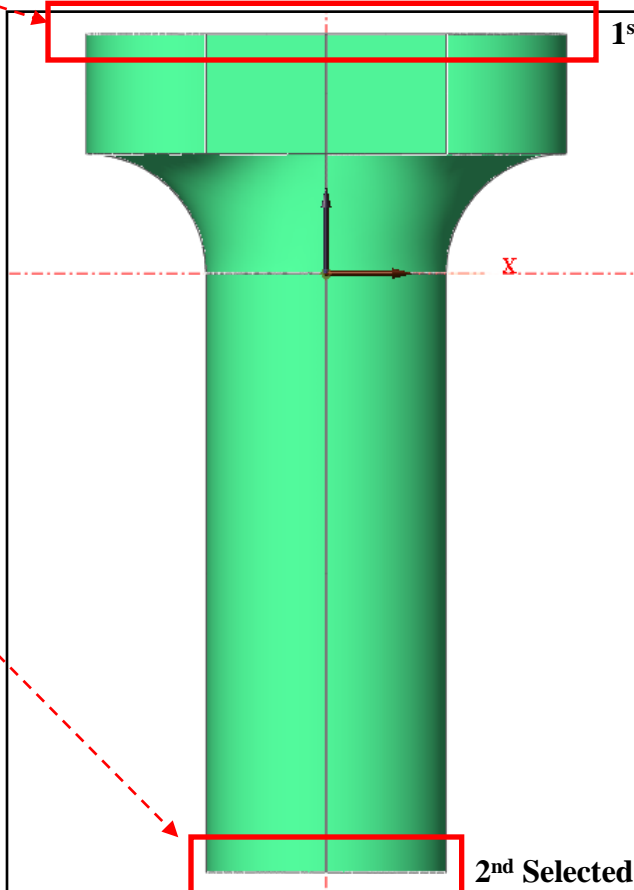
## Step 11.



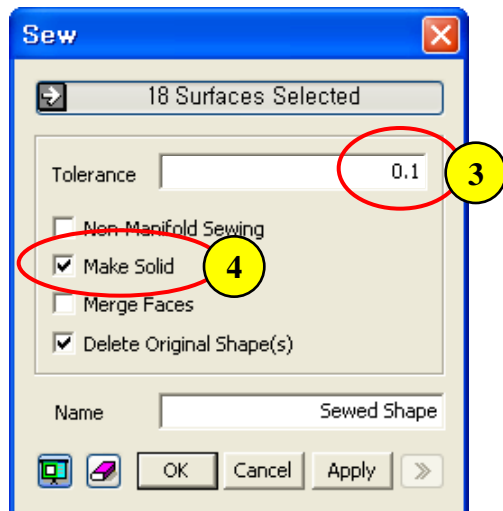





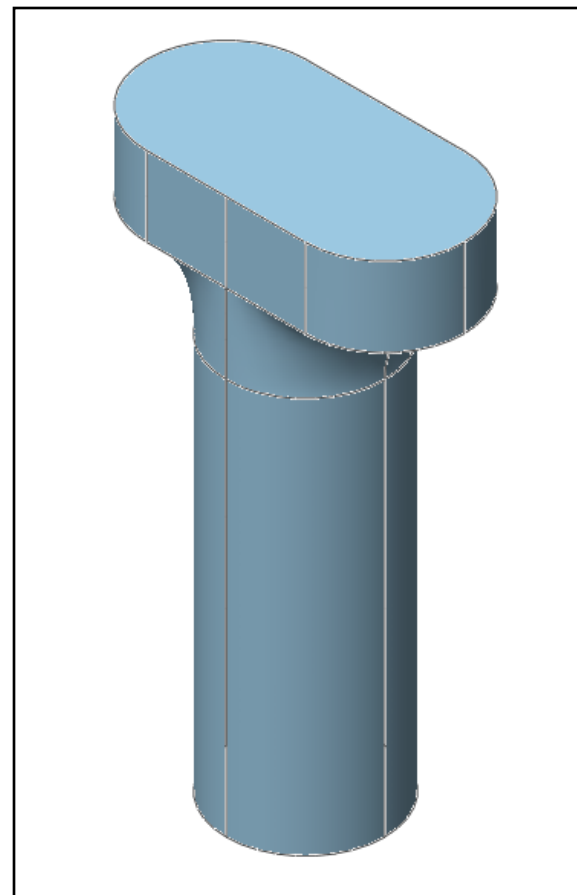
1. Geometry > Surface > Create > Plane Face ...
2. Click "Front View"
3. Select 8 Curves (See Figure)
4. Click [Apply] Button
5. Select 4 Curves (See Figure)
6. Click [OK] Button



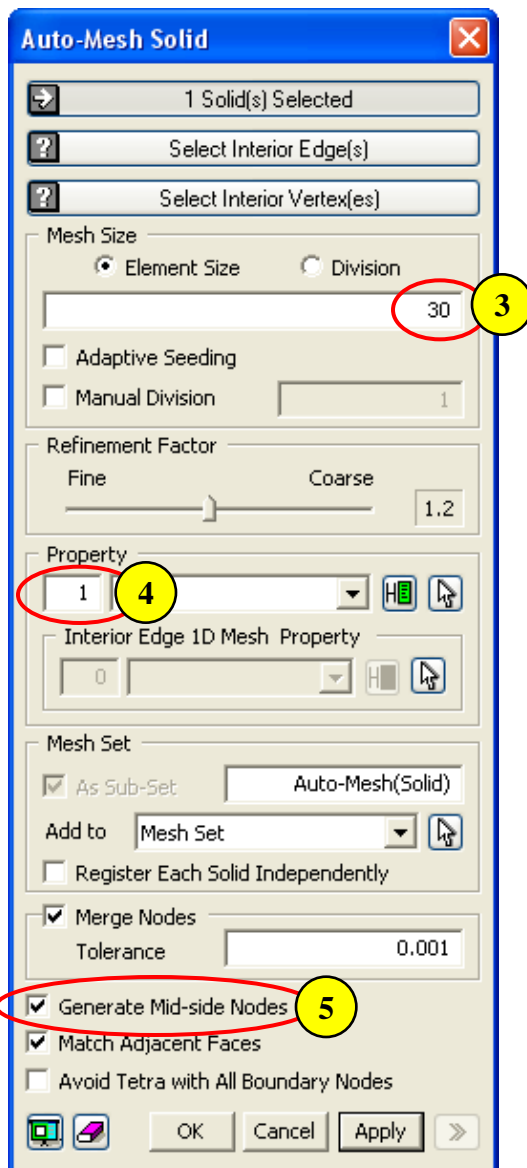
## Step 12.



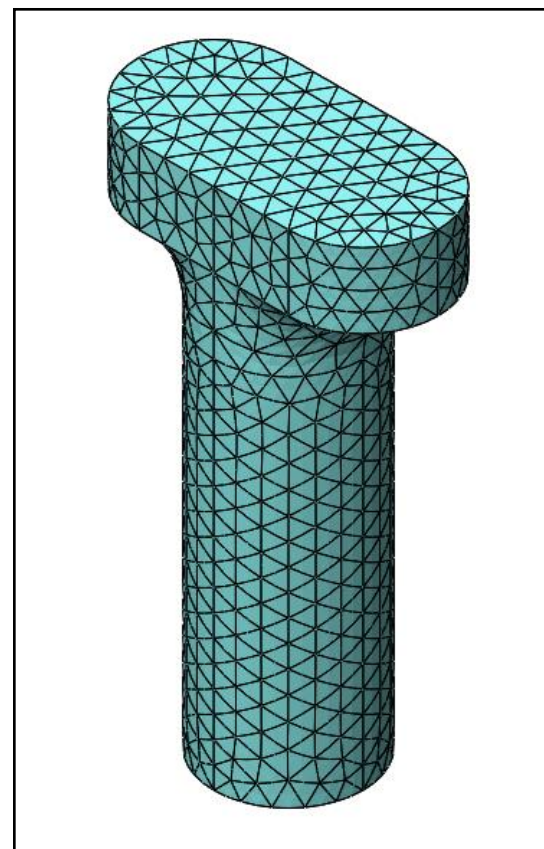
1. *Geometry > Surface > Sew ...*
2. Select  “Displayed”
3. *Tolerance : 0.1*
4. *Check on “Make Solid”*
5. *Click “OK” Button*



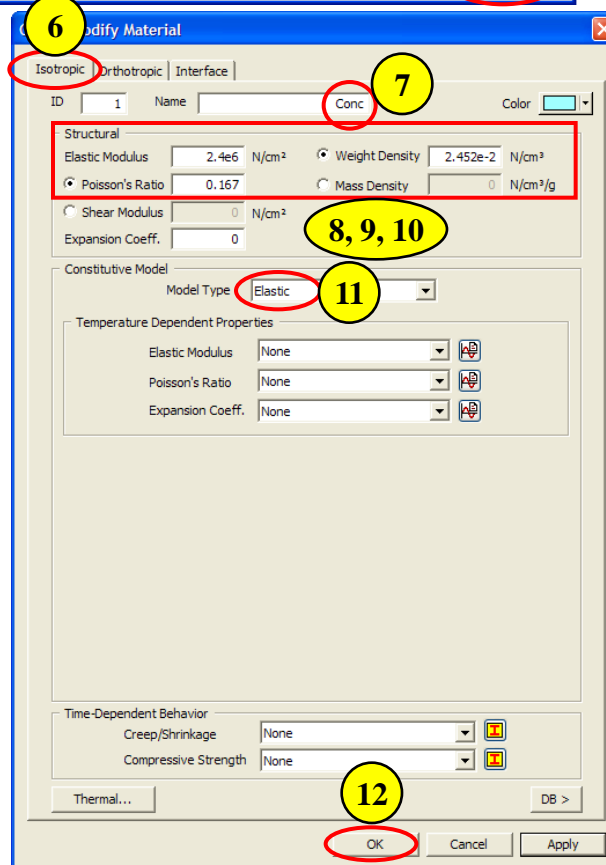
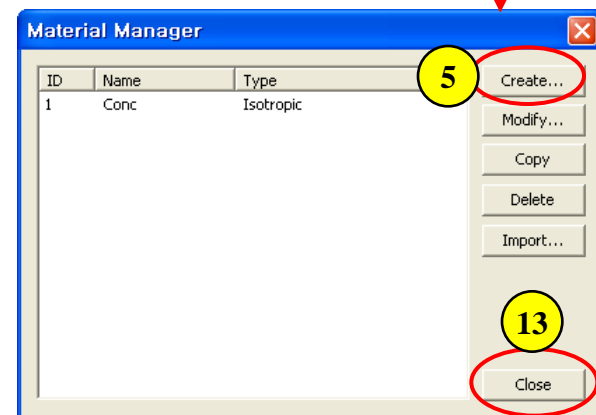
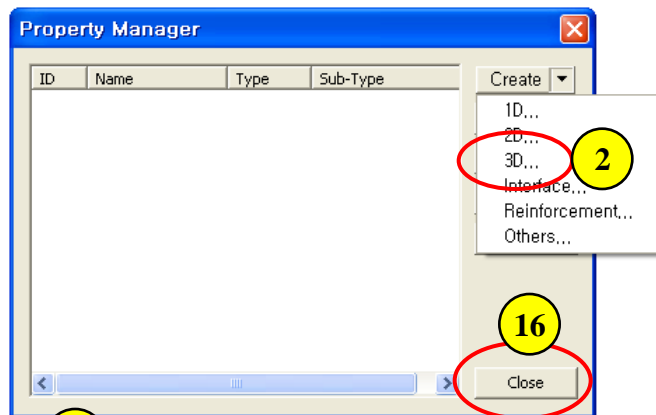
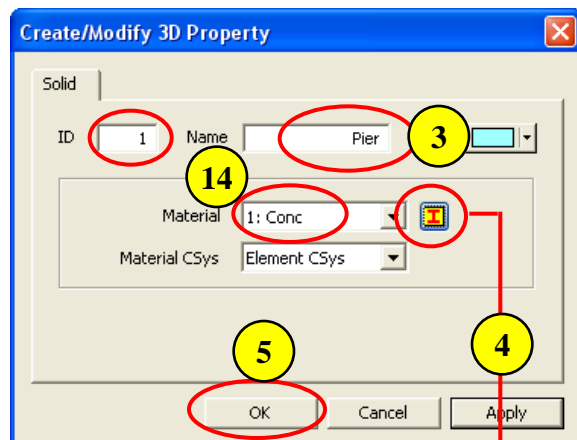
## Step 13.



1. *Mesh > Auto Mesh > Solid ...*
2. *Select Solid*
3. *Mesh Size : Element Size(30)*
4. *Property (1)*
5. *Check on “Generate Mid-Side Nodes”*
6. *Click [OK] Button*



# Step 14.



1. Analysis > Property ...

2. Create 3D ...

3. ID : 1 , Name : Pier

4. Click  Button

5. Click [Create] Button

6. Select "Isotropic" Tab

7. ID : 1 , Name : Conc

8. Elastic Modulus :  $2.4e6 \text{ N/cm}^2$

9. Poisson's Ratio : 0.167

10. Weight Density :  $2.452e-2 \text{ N/cm}^3$

11. Model Type : Elastic

12. Click [OK] Button

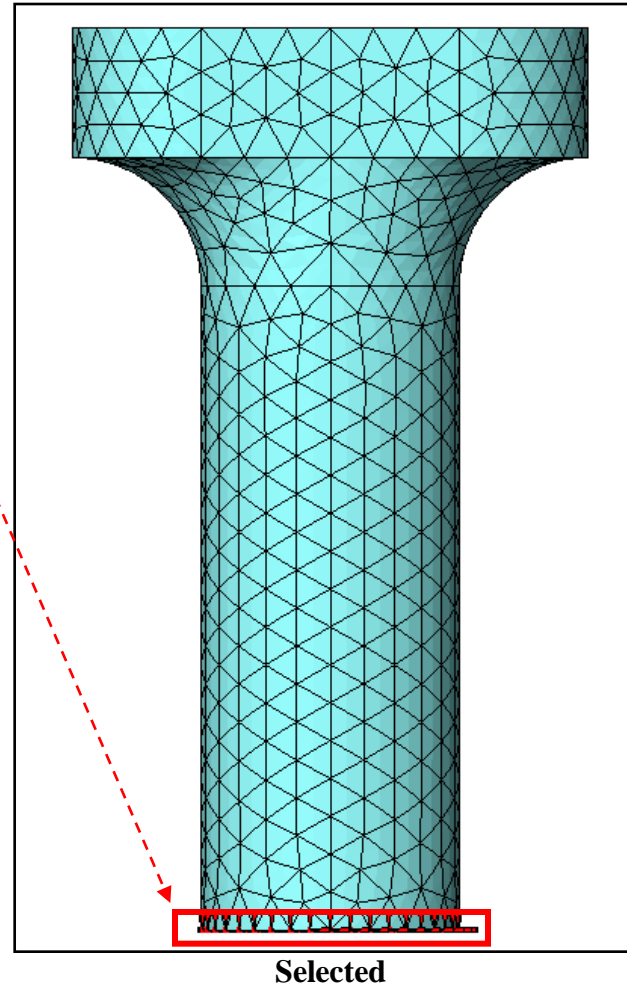
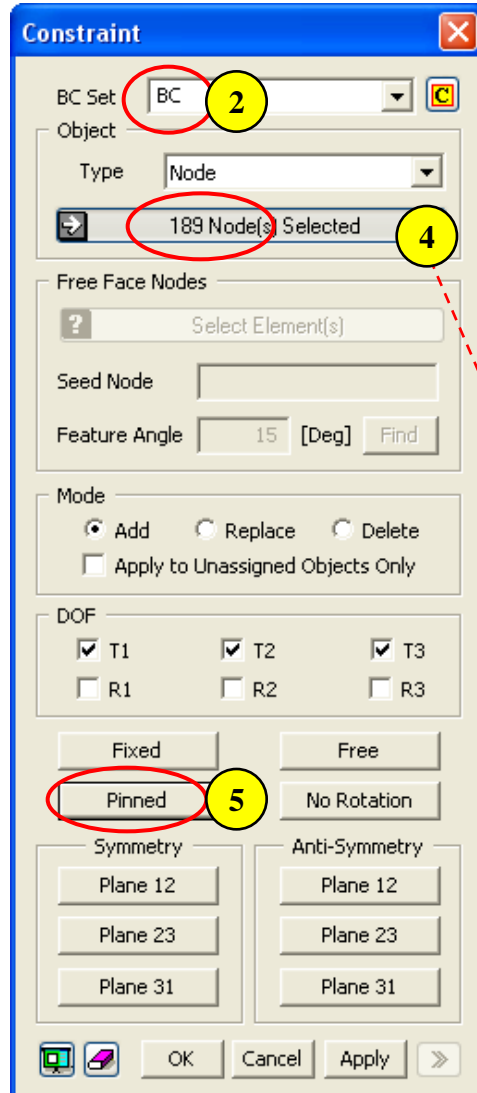
13. Click [Close] Button

14. Select "1: Conc" for Material

15. Click [OK] Button

16. Click [Close] Button

## Step 15.

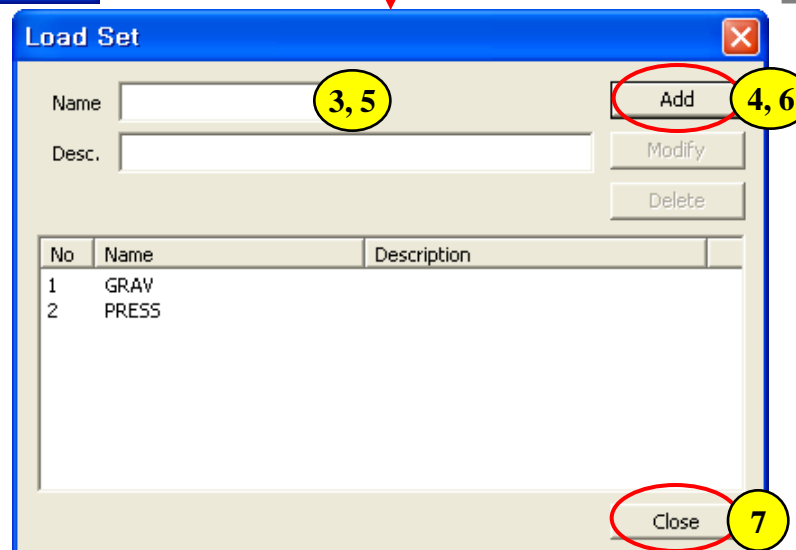
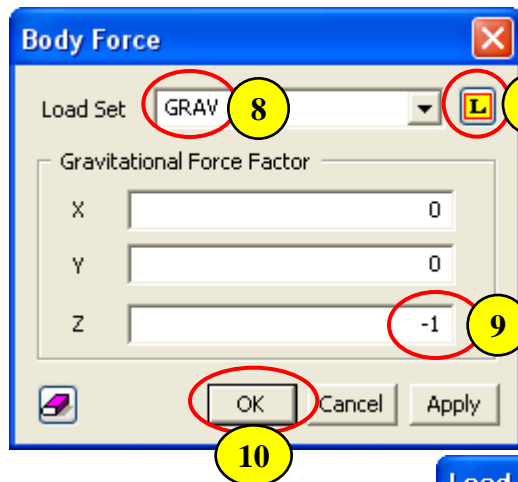


1. Analysis > BC > Constraint ...
2. BC Set : BC
3. Click "Front View"
4. Select 189 Nodes (See Figure)
5. Click [Pinned] Button
6. Click [OK] Button



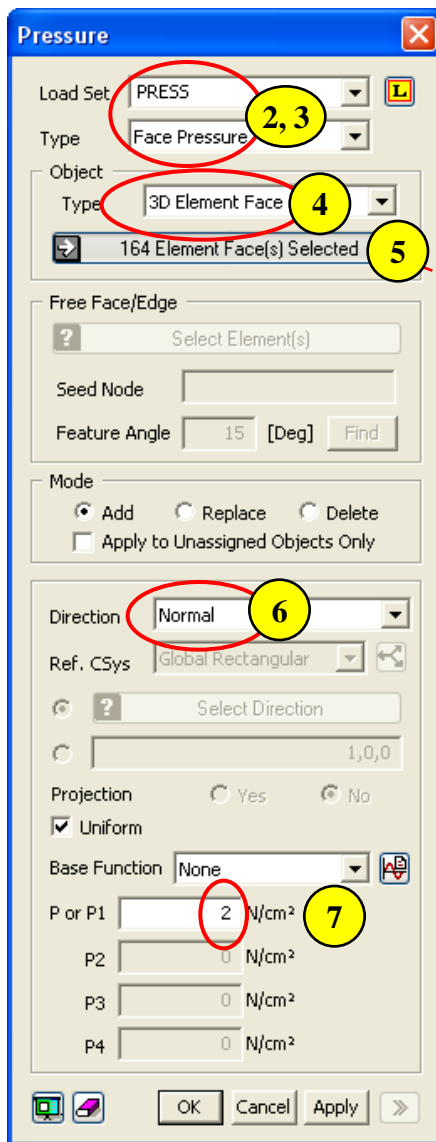


## Step 16.



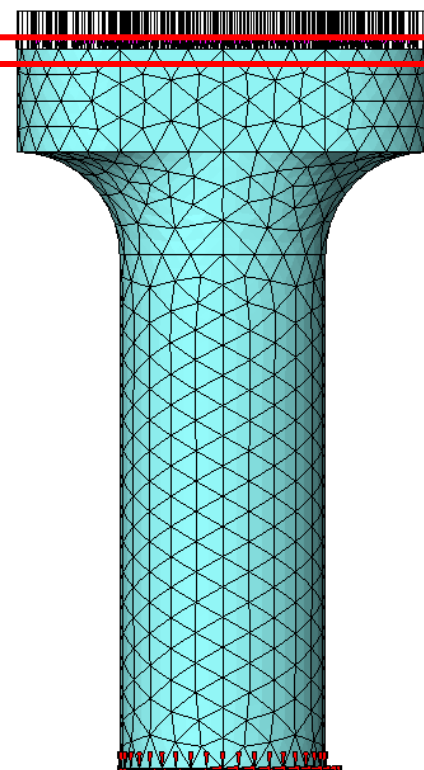
1. Analysis > Load > Body Force ...
2. Click Button
3. Name : GRAV
4. Click [Add] Button
5. Name : PRESS
6. Click [Add] Button
7. Click [Close] Button
8. Load Set : GRAV
9. Gravitational Force Factor : Z (-1)
10. Click [OK] Button

## Step 17.

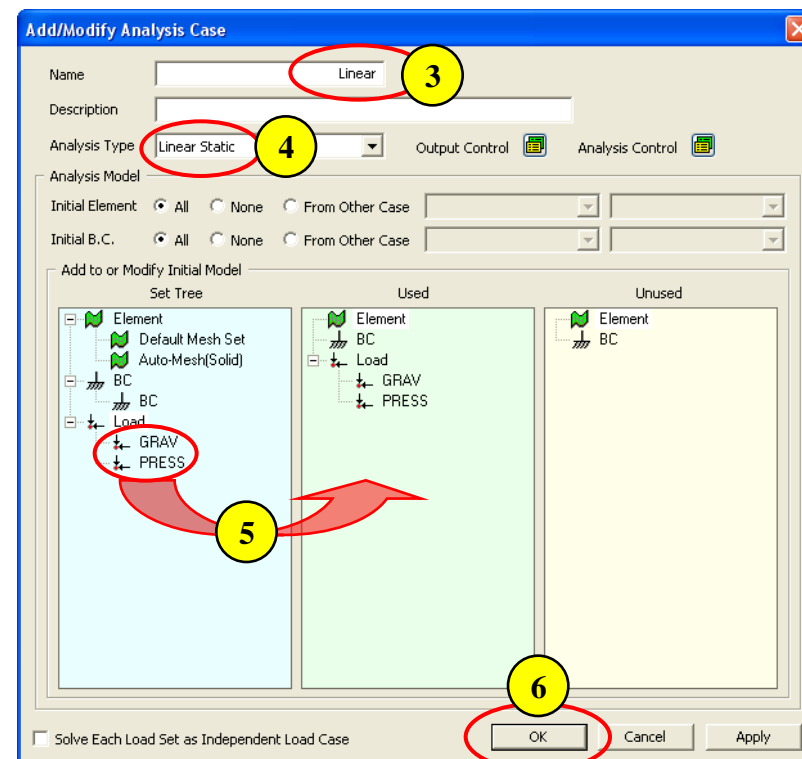
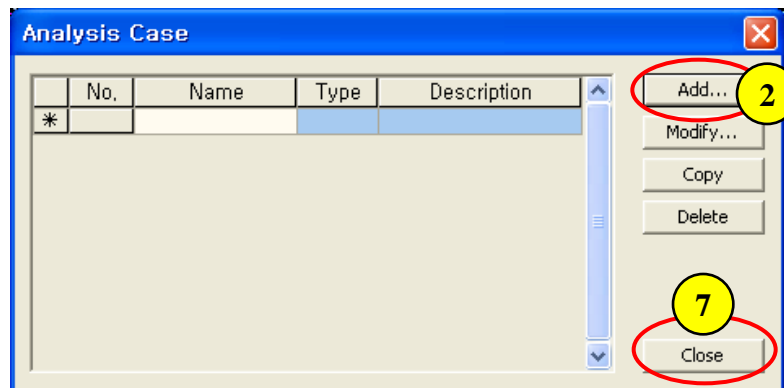


1. Analysis > Load > Pressure ...
2. Load Set : PRESS
3. Type : Face Pressure
4. Object Type : 3D Element Face
5. Select Elements (See Figure)
6. Direction : Normal
7. P or P1: 2 N/cm<sup>2</sup>
8. Click [OK] Button

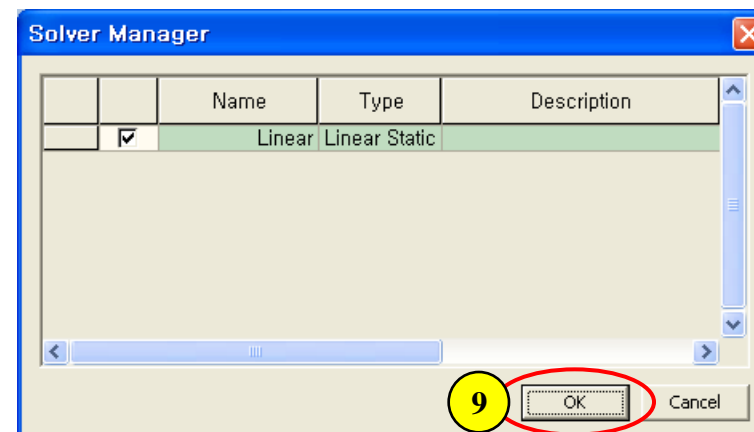
Selected



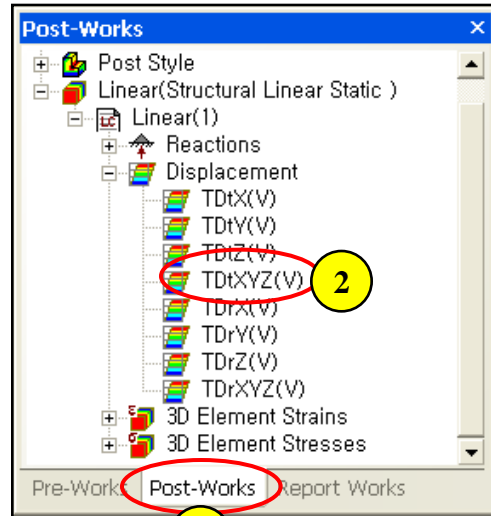
## Step 18.



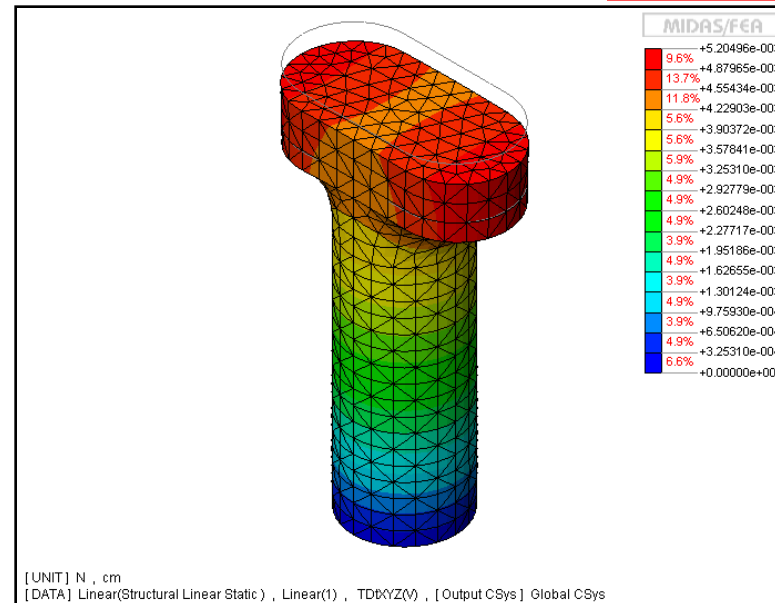
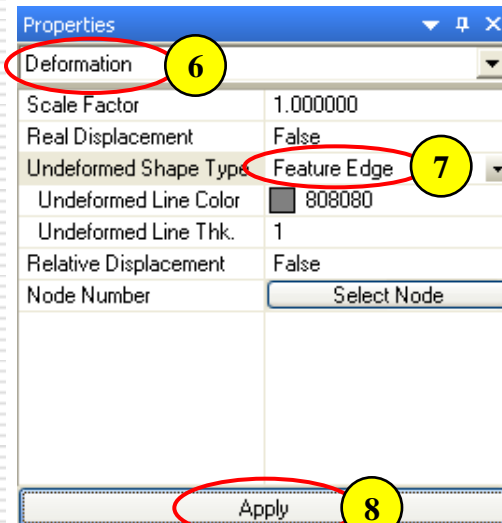
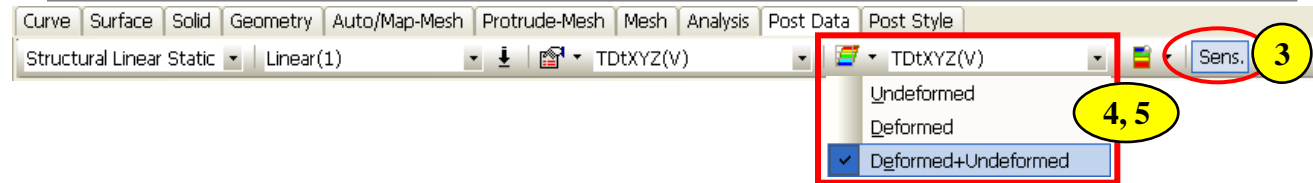
1. Analysis > Analysis Case ...
2. Click [Add] Button
3. Name : Linear
4. Analysis Type : Linear Static
5. Drag & Drop "Load" to "Used" Window
6. Click [OK] Button
7. Click [Close] Button
8. File > Save ... (Pier.feb)
9. Analysis > Solve ...
10. Click [OK] Button



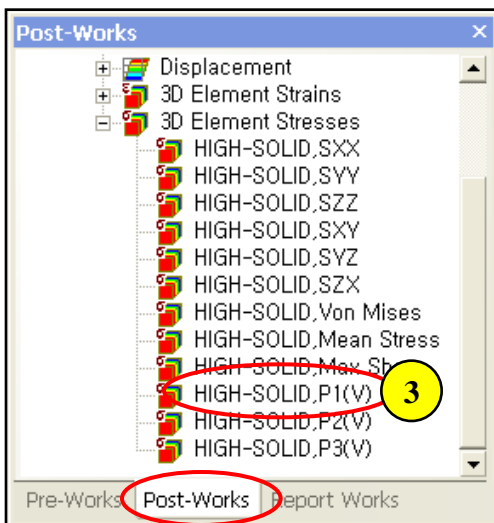
## Step 19.



1. Post-Works Tree : Linear (Structural Linear Static) > Linear(1) > Displacement
2. Double Click “TDtXYZ(V)”
3. Click “Sens.” Button
4. Select “Deformed+Undeformed” for Mesh Shape (See Figure)
5. Select “TDtXYZ(V)” for Deformation Data
6. Property Window : Deformation
7. Undeformed Shape Type : Feature Edge
8. Click [Apply] Button



## Step 20.



1. Select “Undeformed” for Mesh Shape

2. Post-Works Tree : Linear (Structural Linear Static) > Linear(1) > 3D Element Stresses

3. Double Click “HIGH-SOLID, P1(V)”

