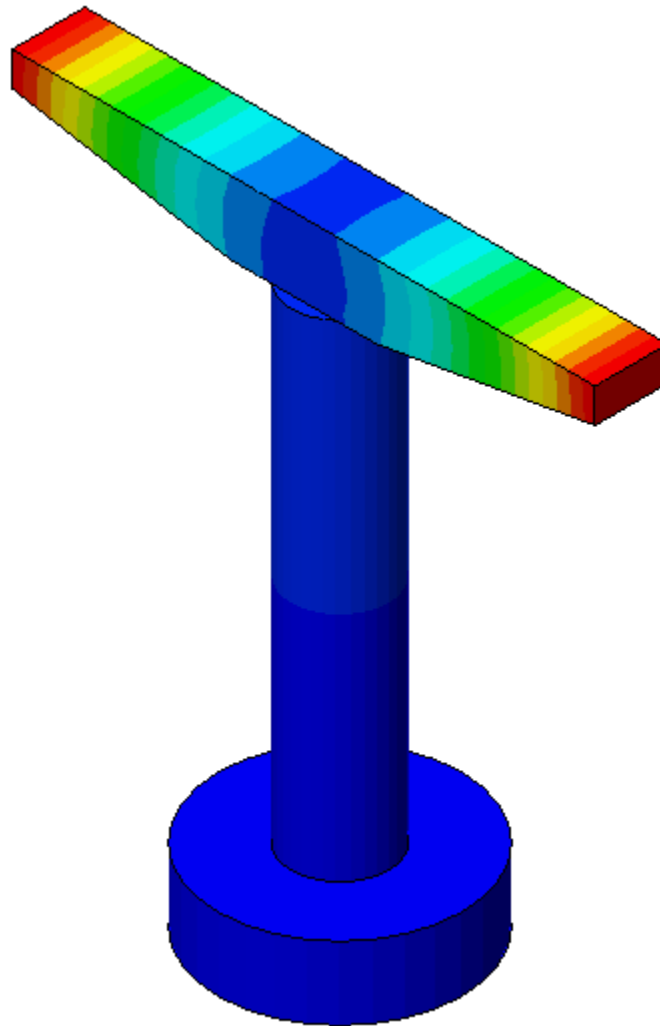


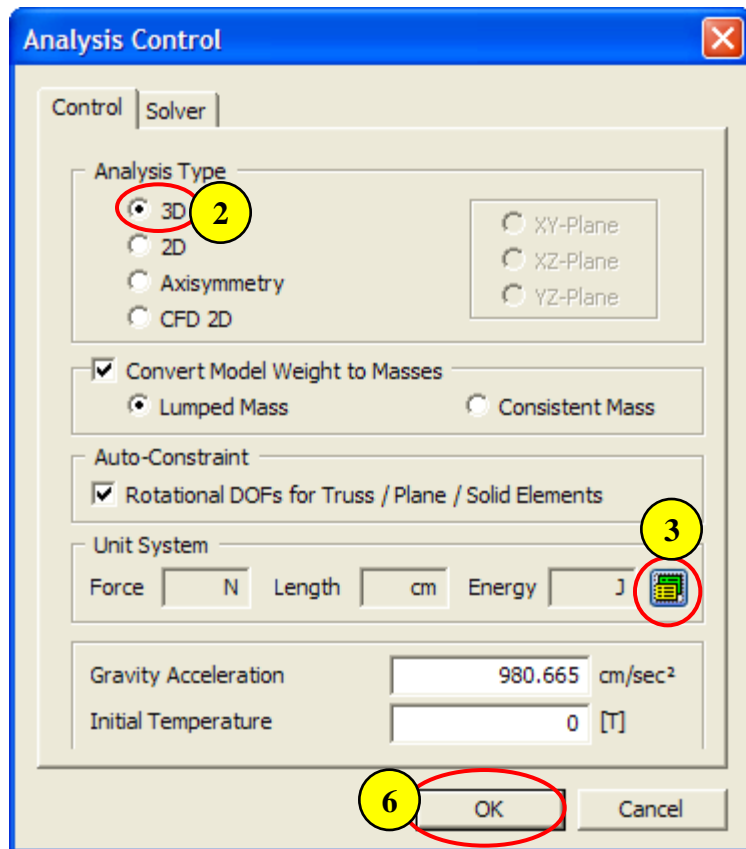
LS-9. Pier – MapMesh



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

Step 1.



1. Analysis > Analysis Control – “Control” tab

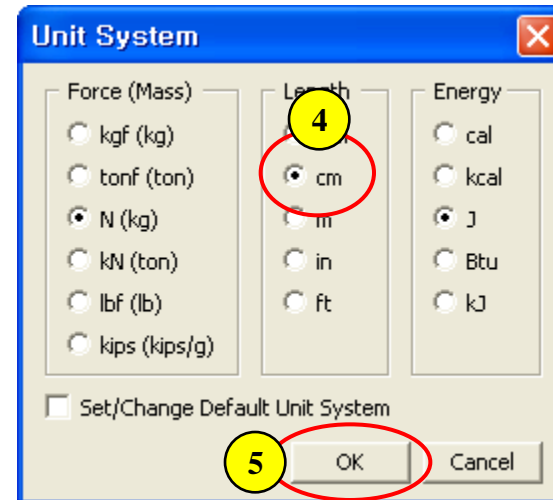
2. Analysis Type : 3D

3. Click  Button (Unit System)

4. Length : cm

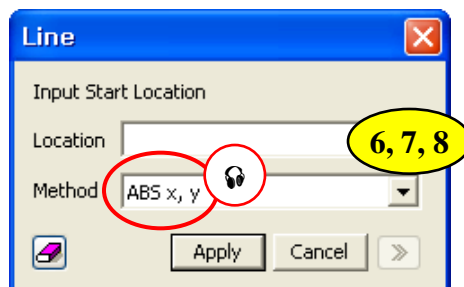
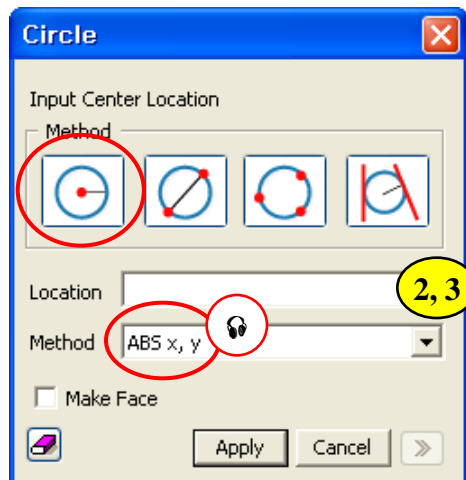
5. Click [OK] Button

6. Click [OK] Button

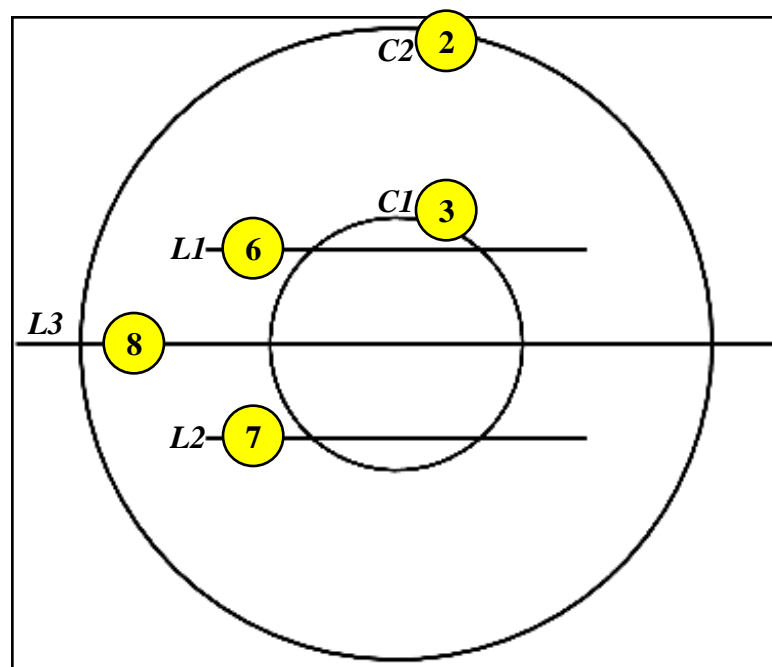


 Analysis Control Dialog is automatically activated at startup.

Step 2.



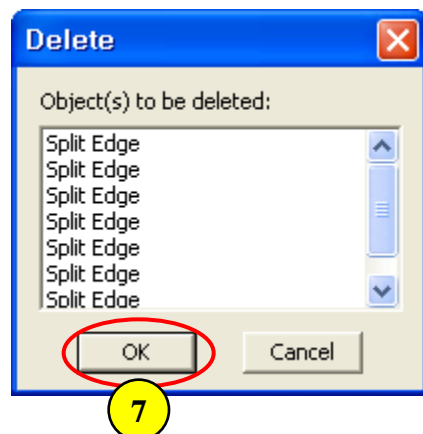
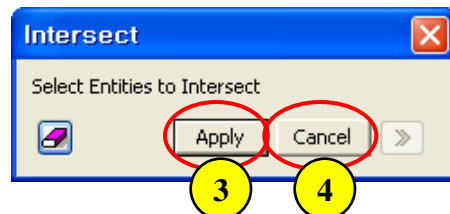
1. *Geometry > Curve > Create on WP > Circle ...*
2. *Location - C1 : Center(0) , R=40*
3. *Location - C2 : Center(0) , R=100*
4. *Click [Cancel] Button*
5. *Geometry > Curve > Create on WP > Line ...*
6. *Location - L1 : (-60, 30), <120>*
7. *Location - L2 : (-60, -30), <120>*
8. *Location - L3 : (-120), <240>*
9. *Click [Cancel] Button*






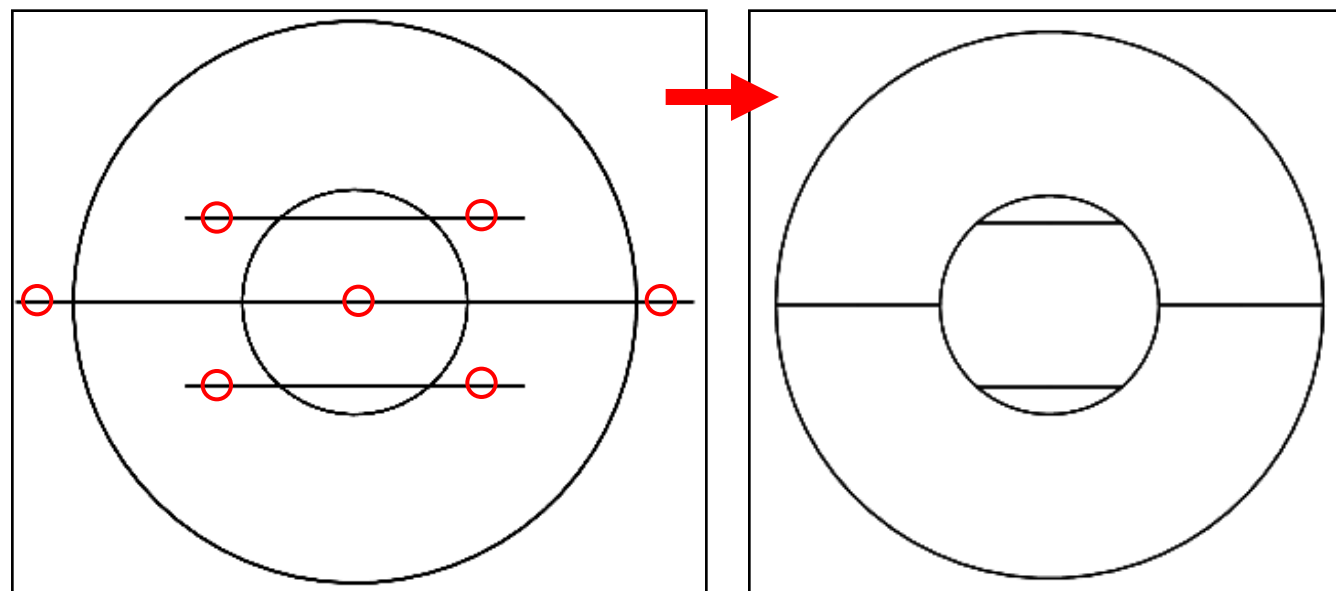
() : “ABS x, y”, <> : “REL dx, dy”
 (0) same as (0, 0), <120> same as <120, 0>

“Esc” as shortcut for “Cancel”.

Step 3.



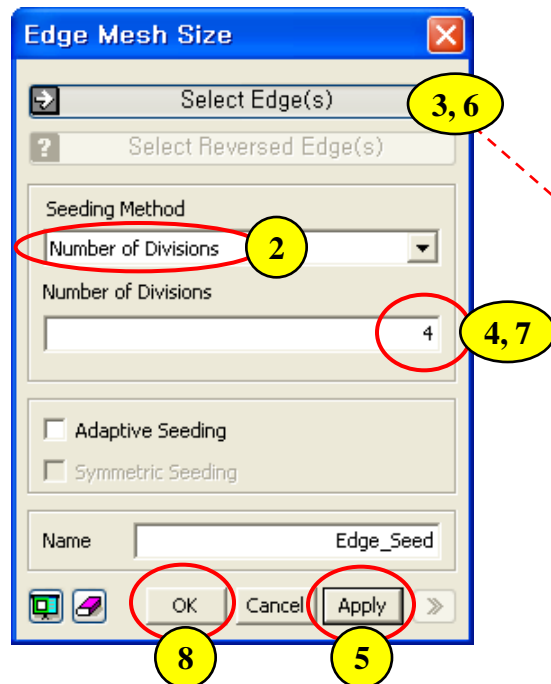
1. *Geometry > Curve > Intersect ...*
2. Select  “Displayed” 
3. Click [Apply] Button 
4. Click [Cancel] Button
5. Select Unnecessary Edges marked by “O” (See Figure)
6. Press “Delete” Key
7. Click [OK] Button



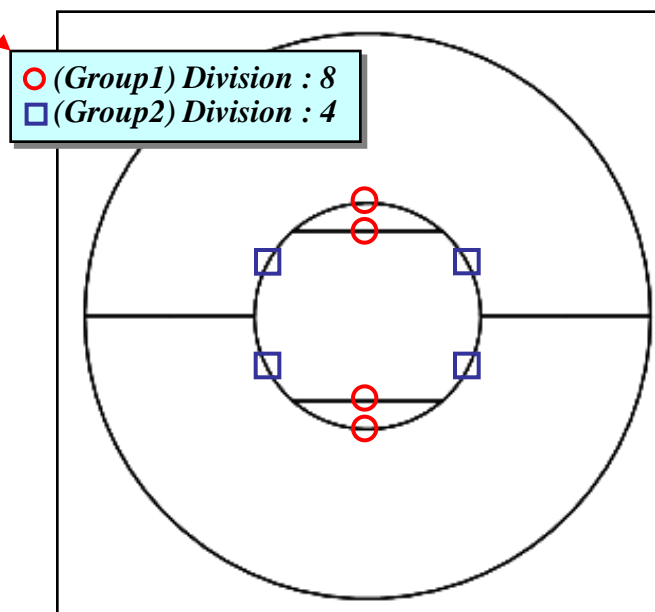
 “Ctrl+A” as shortcut for “Select Displayed”.

 “Enter” as shortcut for “Apply”.

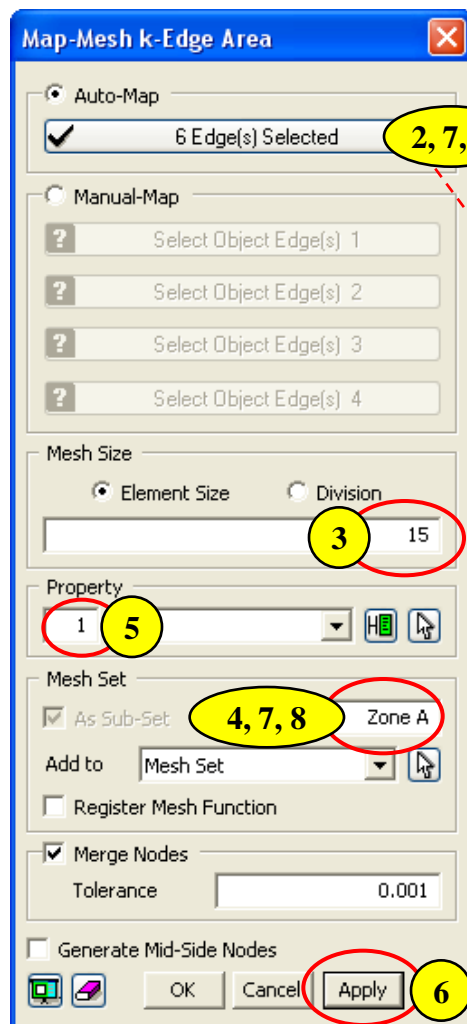
Step 4.



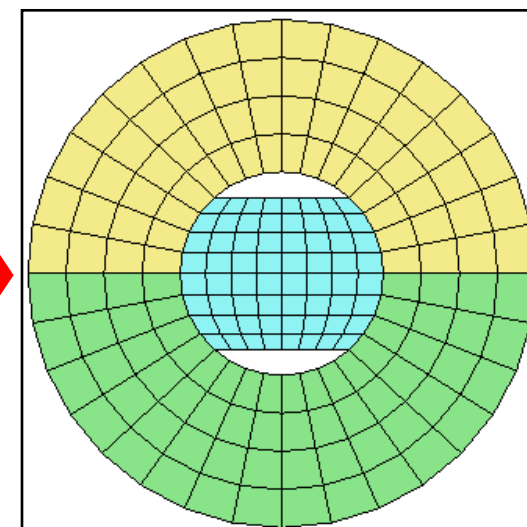
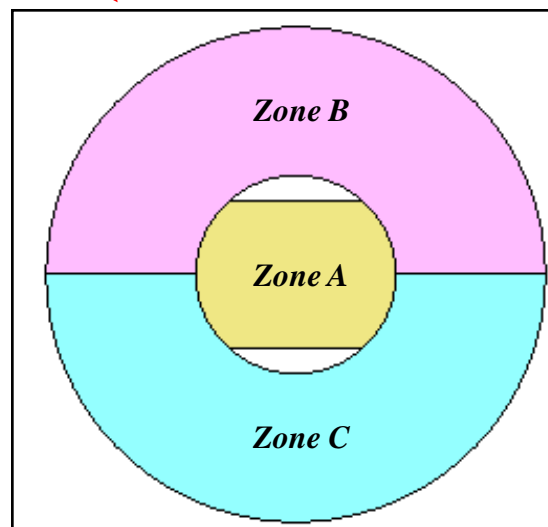
1. Mesh > Size Control > Along Edge ...
2. Seeding Method : "Number of Divisions"
3. Select "Group1" Edges (See Figure)
4. Number of Divisions : 8
5. Click [Apply] Button
6. Select "Group2" Edges (See Figure)
7. Number of Divisions : 4
8. Click [OK] Button



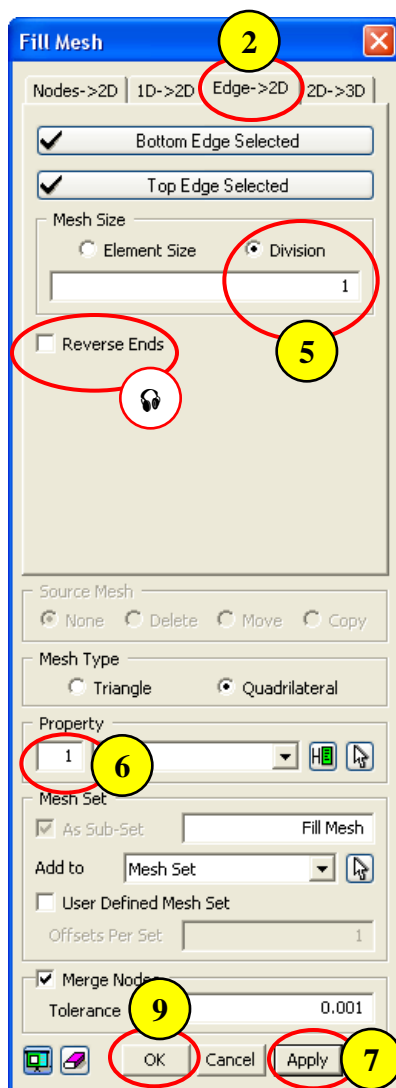
Step 5.



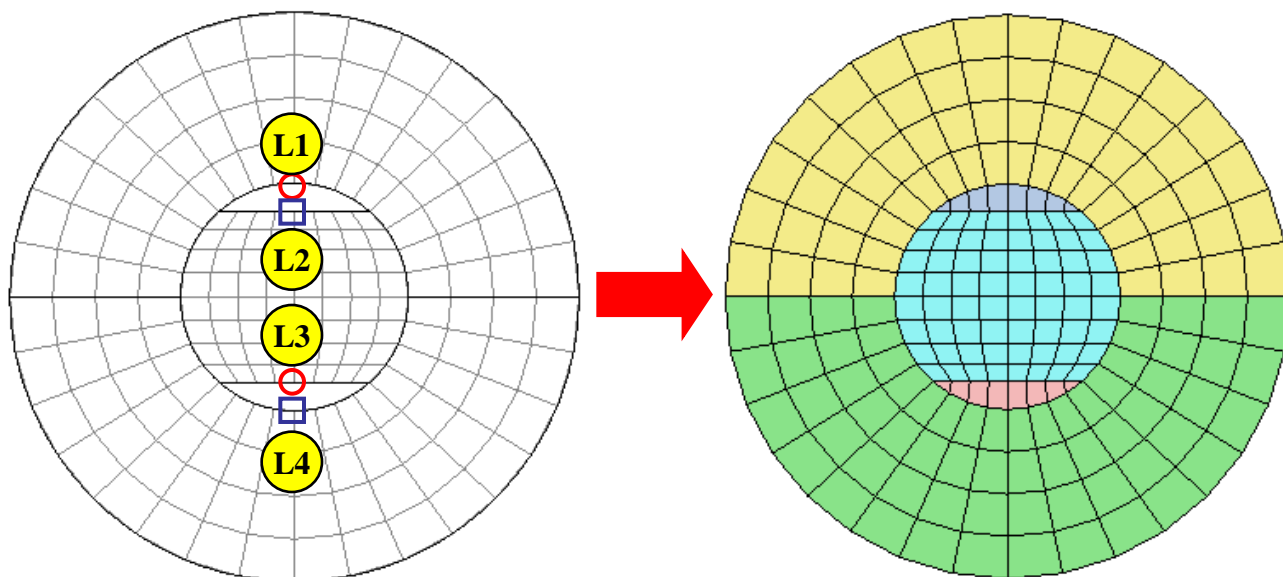
1. Mesh > Map Mesh > k-Edge Area...
2. Select 6 Edges of "Zone A" (See Figure)
3. Element Size : 15
4. Mesh Set : Zone A
5. Property(1)
6. Click [Apply] Button
7. Repeat Step 2~6 for "Zone B"
8. Repeat Step 2~6 for "Zone C"



Step 6.

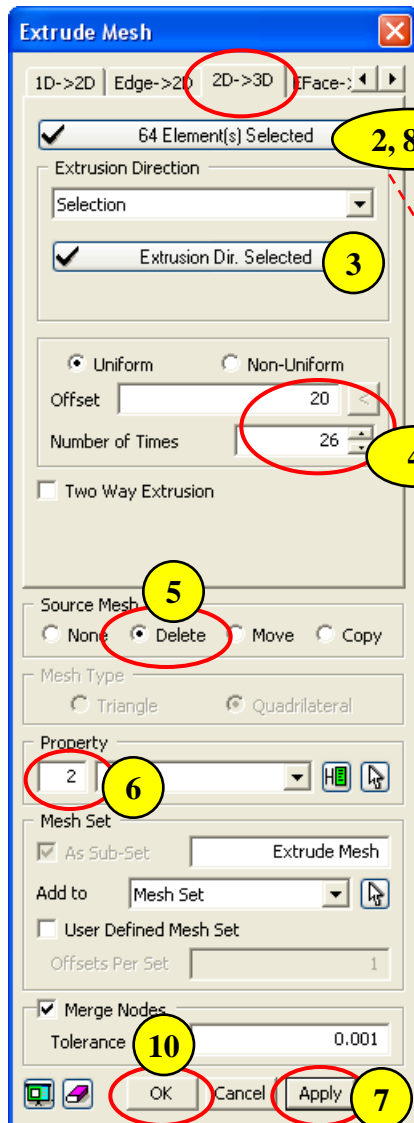


1. Mesh > Protrude Mesh > Fill...
2. Select “Edge->2D” Tab
3. Select “L1” for Top Edge marked by “○” (See Figure)
4. Select “L2” for Bottom Edge marked by “□” (See Figure)
5. Mesh Size : Division (1)
6. Property (1)
7. Click [Apply] Button
8. Repeat Step 3 to 6 for “L3” & “L4”
9. Click “OK” Button

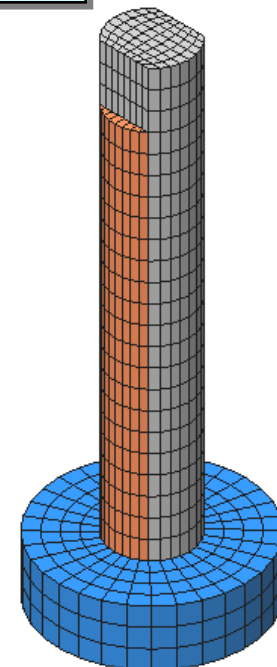
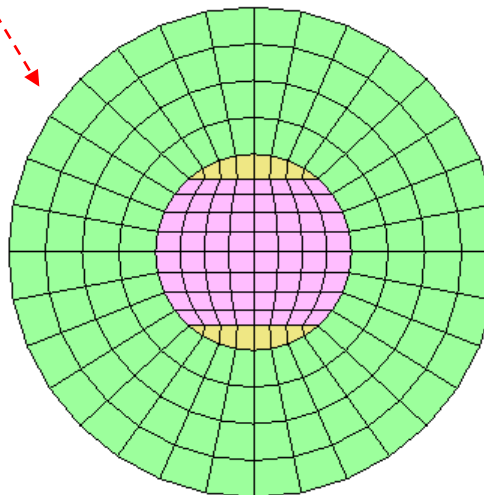


🔊 If result is wrong, check on “Reverse Ends”.

Step 7.

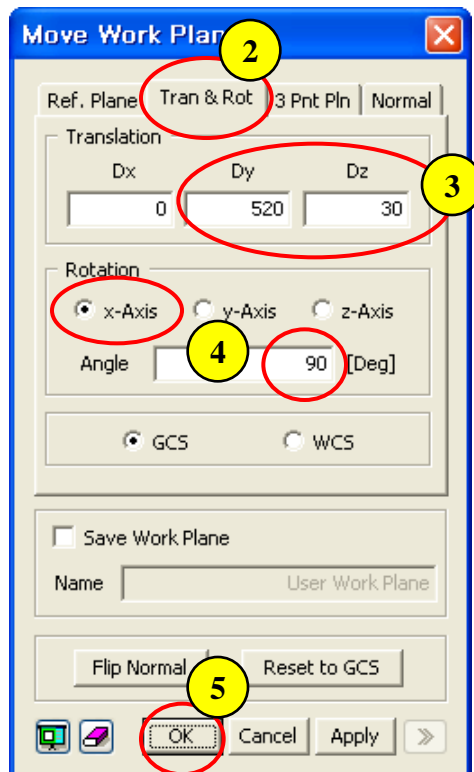


1. Mesh > Protrude Mesh > Extrude : “2D->3D” tab
2. Select Mesh Set (1) (See Figure)
3. Extrusion Direction : Z-axis
4. Offset : 20 , Number of Times : 26
5. Source Mesh : Delete
6. Property (2)
7. Click [Apply] Button
8. Repeat Step 2 to 7 for Mesh Set (2) (See Figure)
9. Repeat Step 2 to 7 for Mesh Set (3) (See Figure)
10. Click “OK” Button

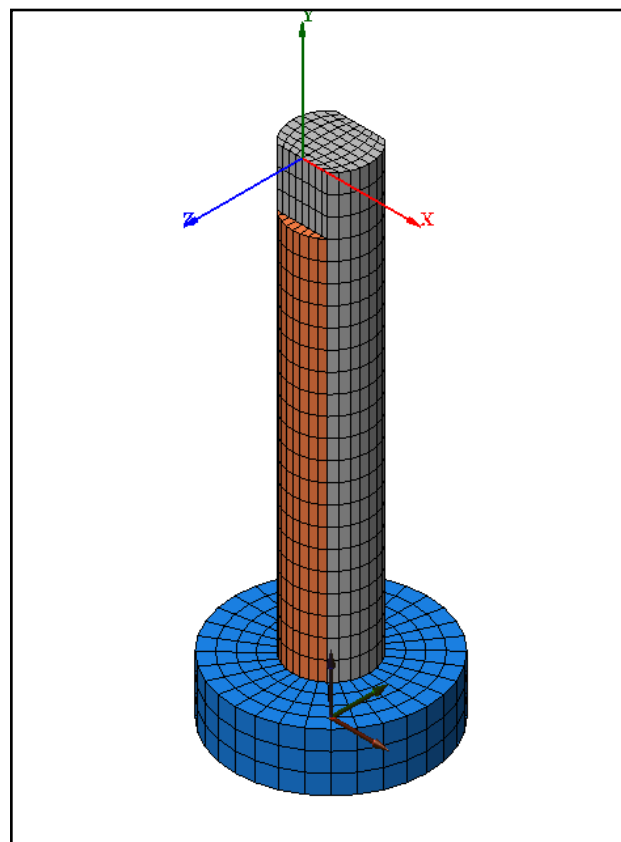


1. Number of Times : 26
2. Number of Times : 23
3. Number of Times : 3

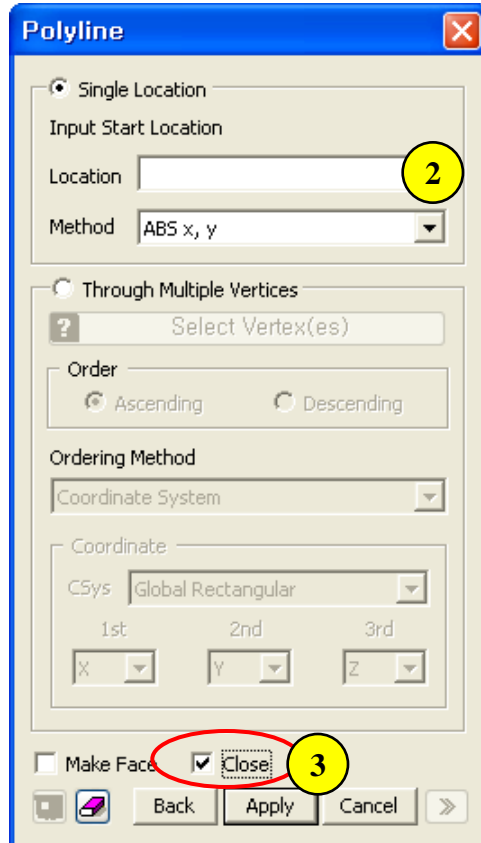
Step 8.



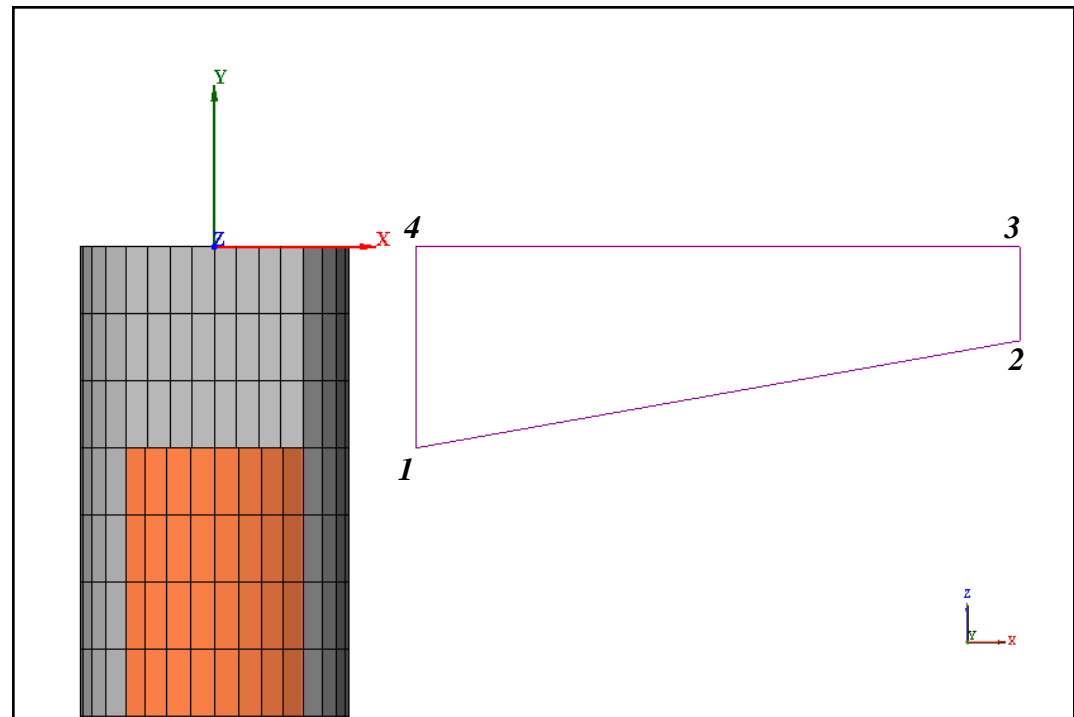
1. Geometry > Work Plane > Move ...
2. Select "Tran & Rot" Tab
3. Translation - Dy : 520 , Dz : 30
4. Rotation - X-axis , Angle : 90
5. Click [OK] Button



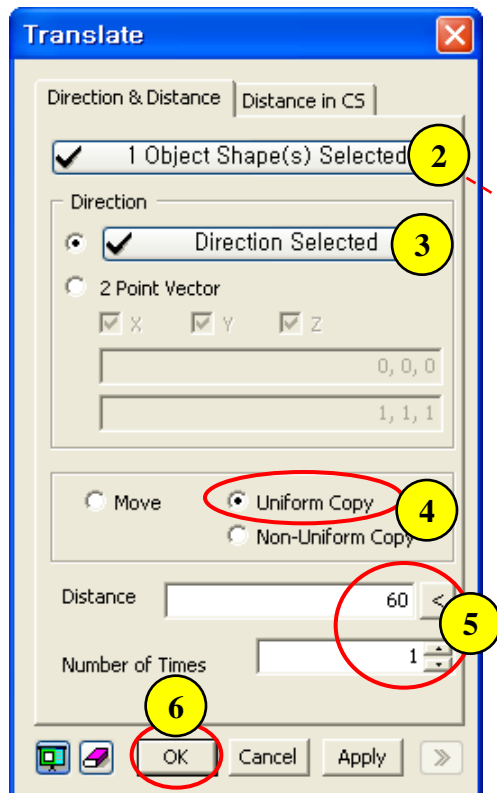
Step 9.



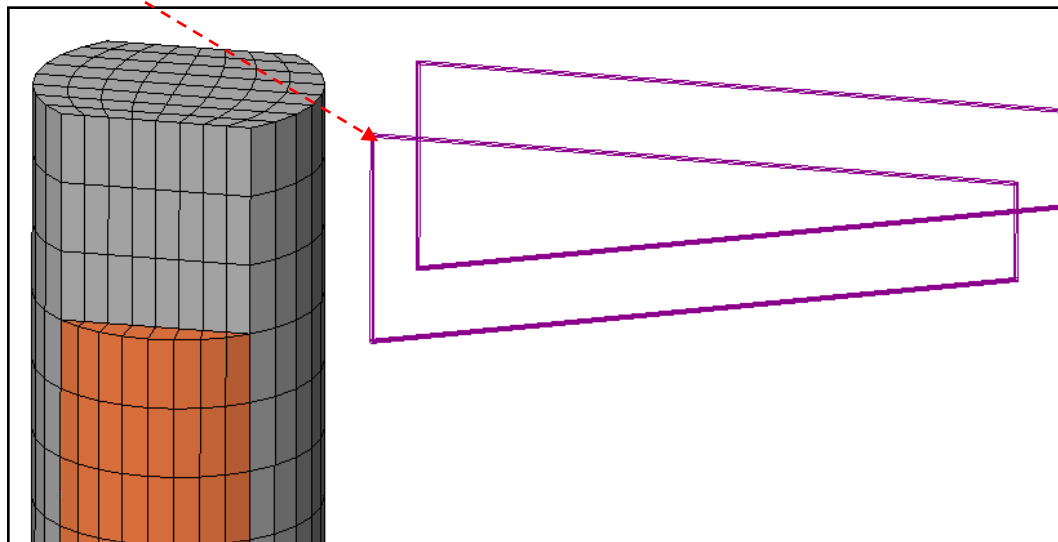
1. *Geometry > Curve > Create on WP > Polyline (Wire) ...*
2. *Location : (60, -60), <180, 32>, <0, 28>, <-180>*
3. *Check on “Close”*
4. *Click Right Mouse Button on Work Window*
5. *Click [Cancel] Button*



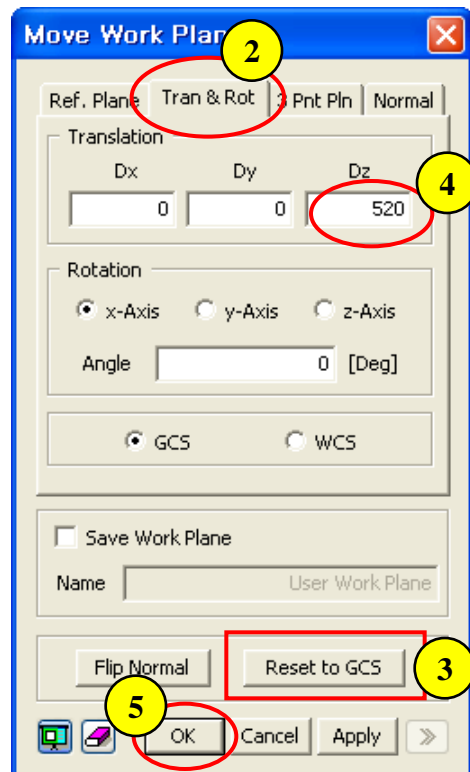
Step 10.



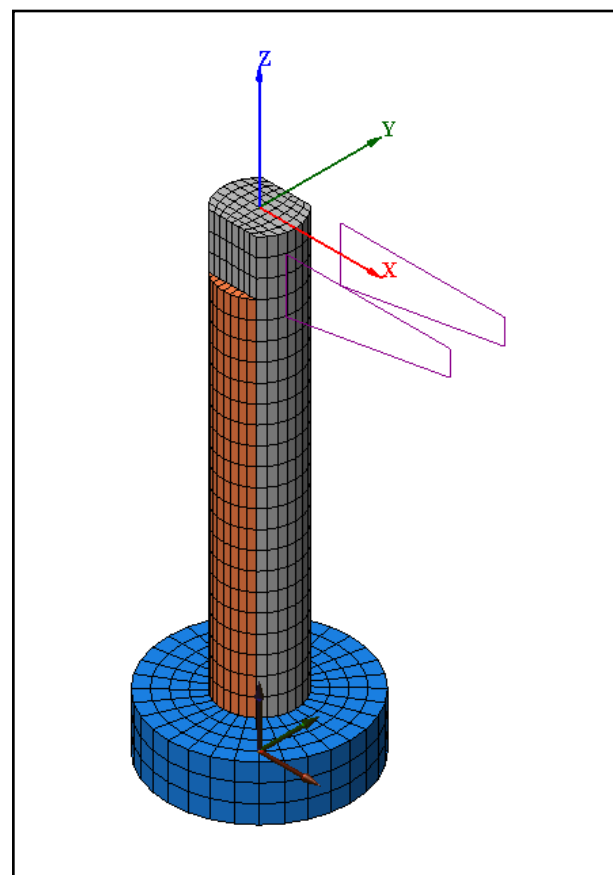
1. Geometry > Transform > Translate...
2. Select Polyline (See Figure)
3. Direction : Y-axis
4. Check on “Uniform Copy”
5. Distance : 60 , Number of Times : 1
6. Click “OK” Button



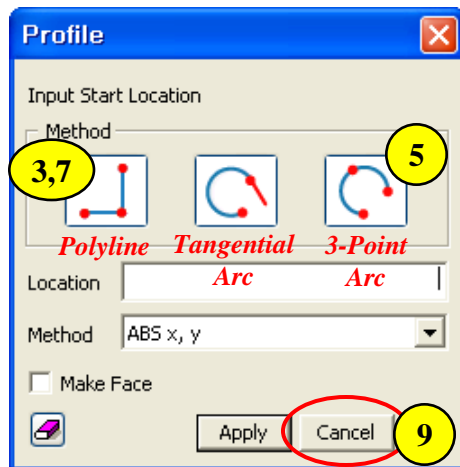
Step 11.



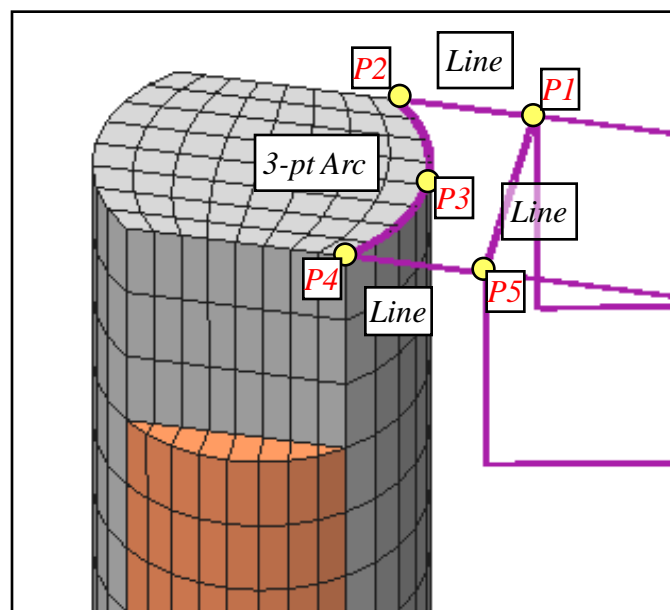
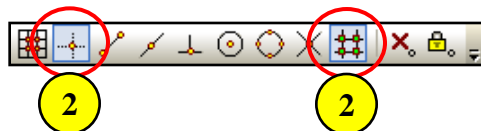
1. Geometry > Work Plane > Move ...
2. Select "Tran & Rot" Tab
3. Click [Reset to GCS] Button
4. Translation - Dz : 520
5. Click [OK] Button



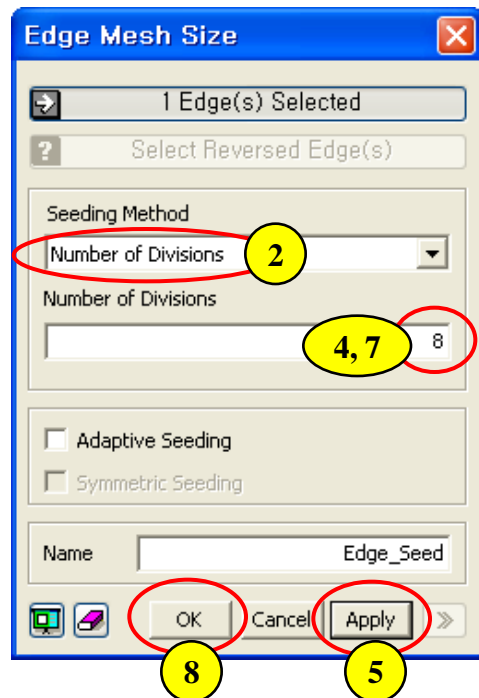
Step 12.



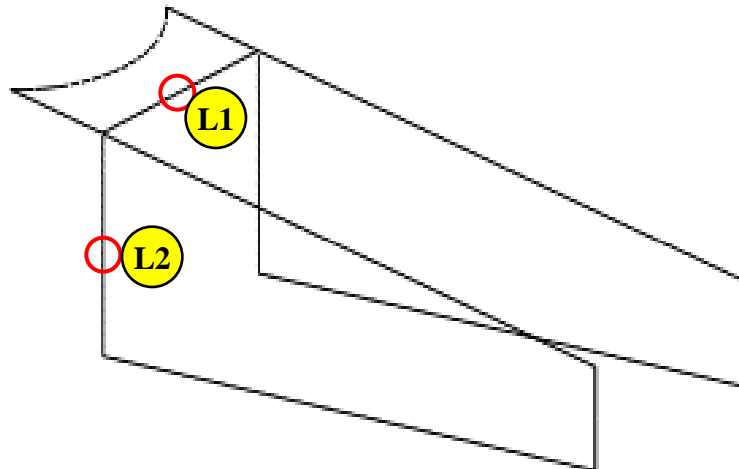
1. Geometry > Curve > Create on WP > Profile ...
2. Toggle on “Vertex Snap” & “Node Snap”
3. Select “Polyline” method
4. Select P1 , P2 in sequential order
5. Select “3-Point Arc” method
6. Select P3 , P4 in sequential order
7. Select “Polyline” method
8. Select P4 , P5 , P1 in sequential order
9. Click [Cancel] Button



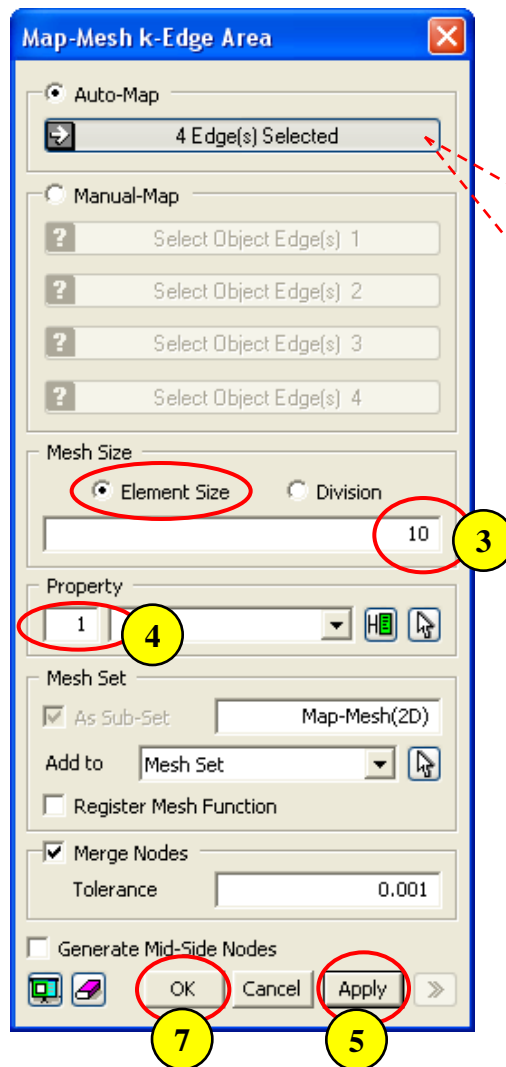
Step 13.



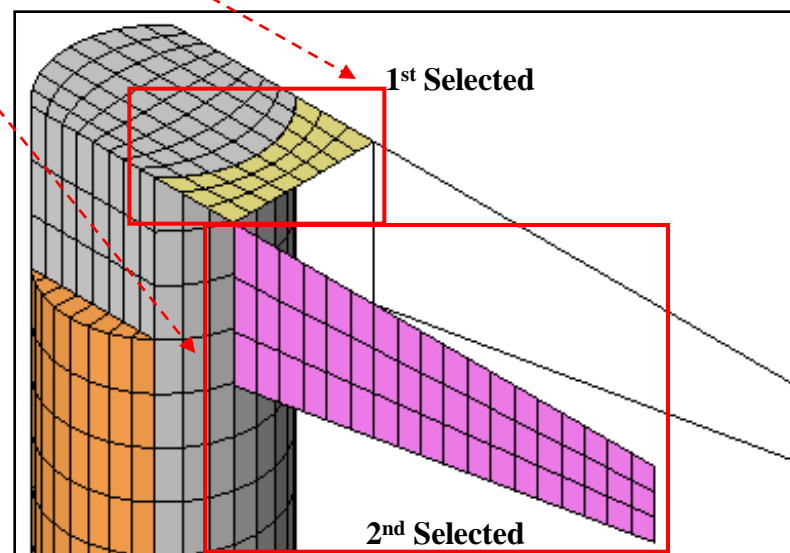
1. Mesh > Size Control > Along Edge ...
2. Seeding Method : “Number of Divisions”
3. Select “L1” Edge (See Figure)
4. Number of Divisions : 8
5. Click “Apply” Button
6. Select “L2” Edge (See Figure)
7. Number of Divisions : 3
8. Click “OK” Button



Step 14.

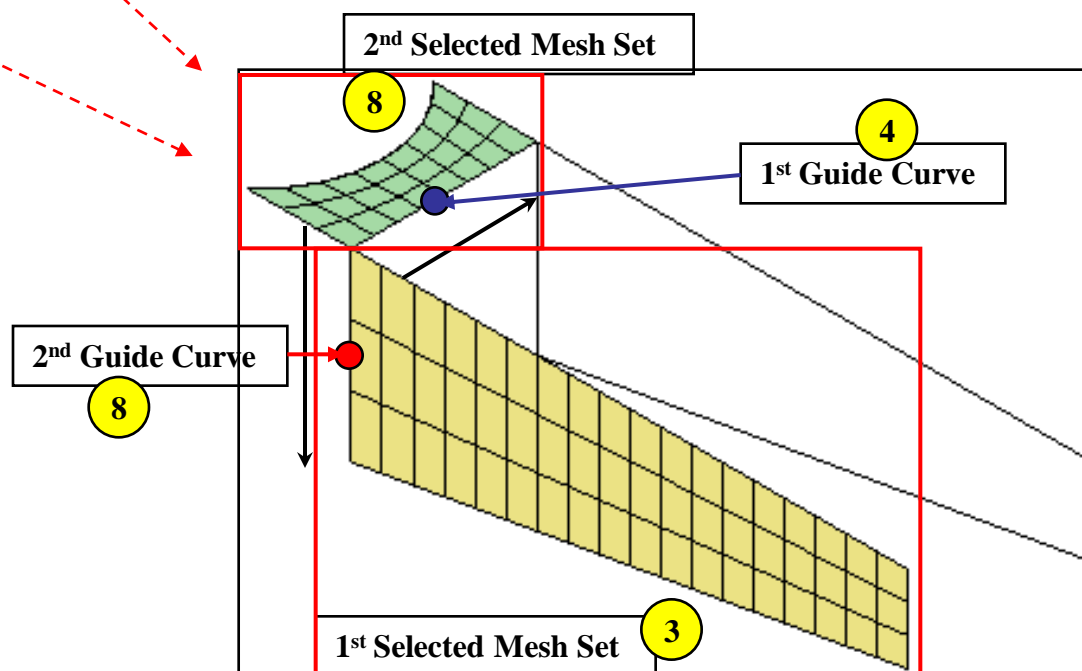
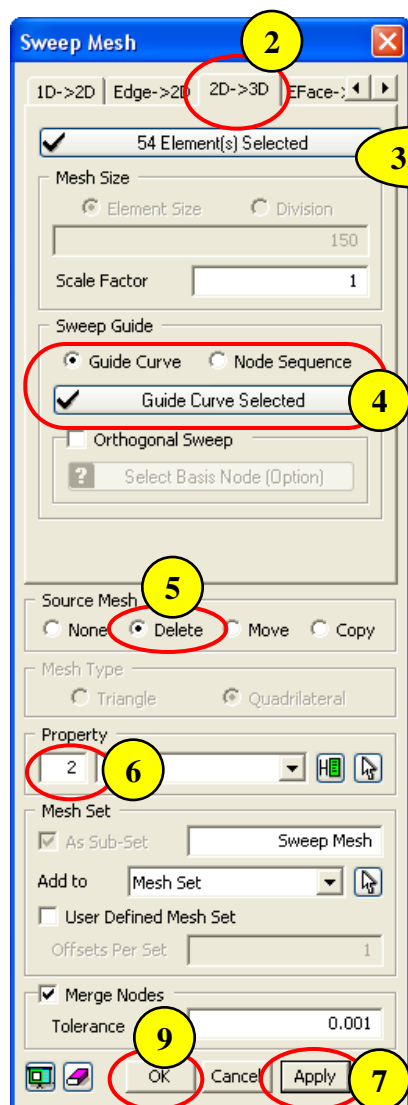


1. Mesh > Map Mesh > k-Edge Area ...
2. Select Edges (See Figure)
3. Mesh Size - Element Size : 10
4. Property (1)
5. Click [Apply] Button
6. Repeat Step 2~4 for 2nd Edges (See Figure)
7. Click [OK] Button

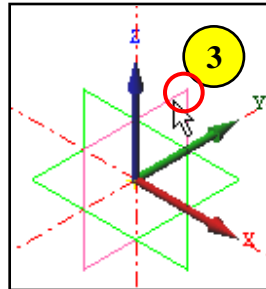
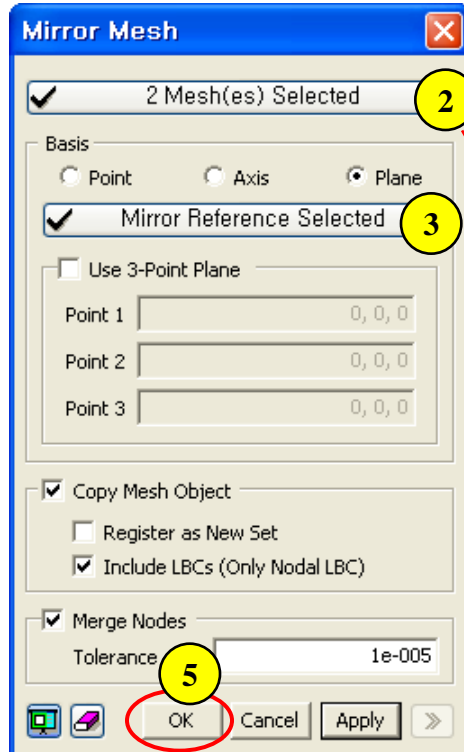


Step 15.

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

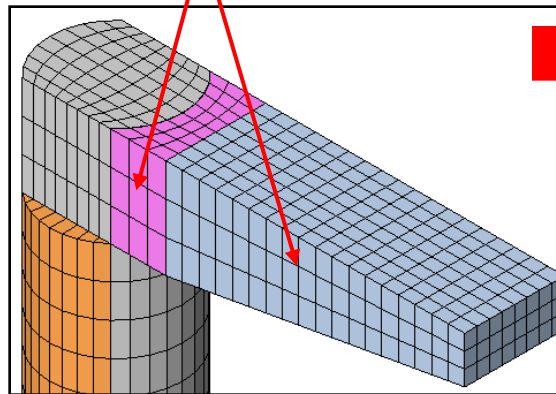


Step 16.

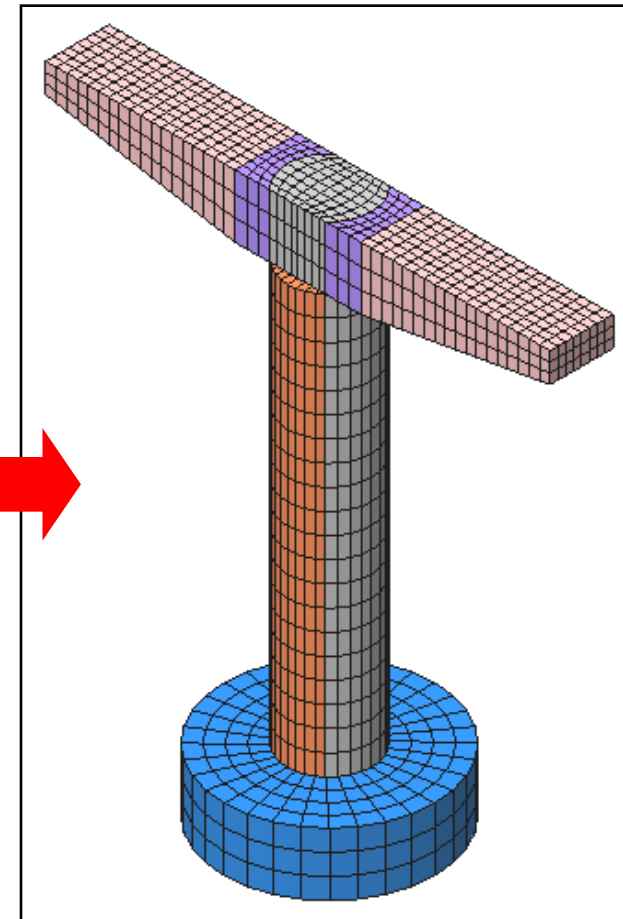


Select "YZ-Plane"
in Work Window
or Pre-Works Tree

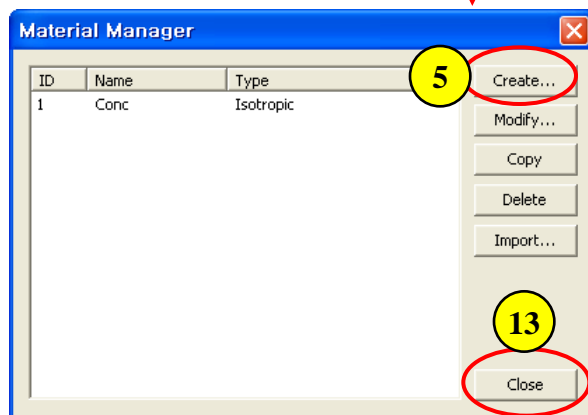
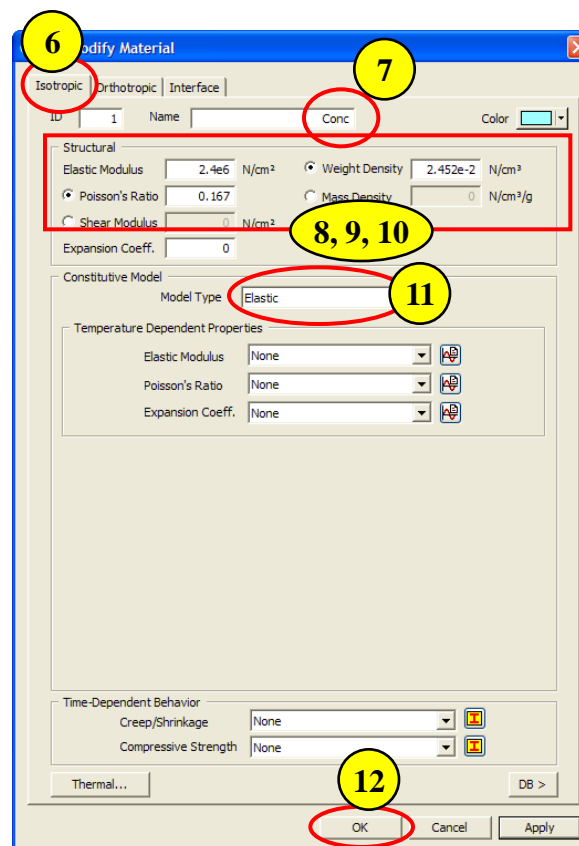
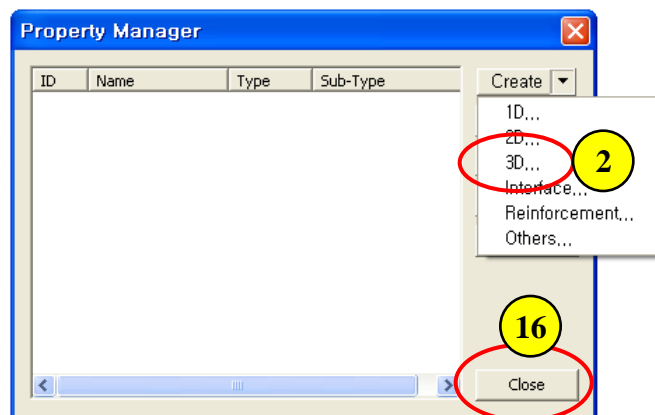
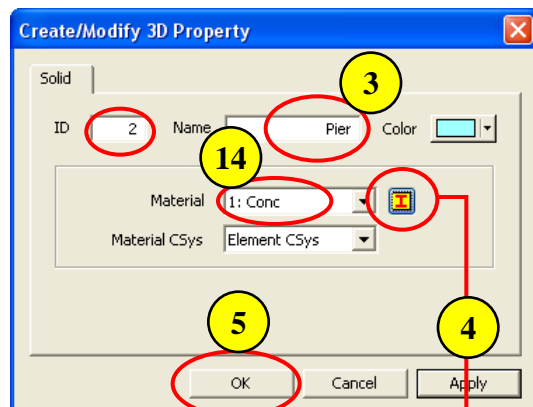
Selected Mesh Set




1. Mesh > Transform > Mirror ...
2. Select 2 Mesh Sets (See Figure)
3. Mirror Plane : YZ-Plane
4. Click [OK] Button

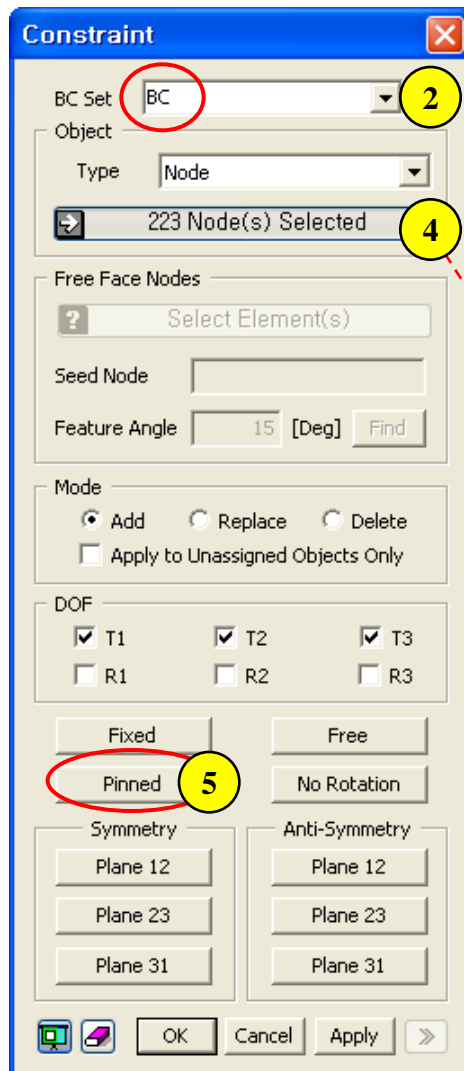


Step 17.

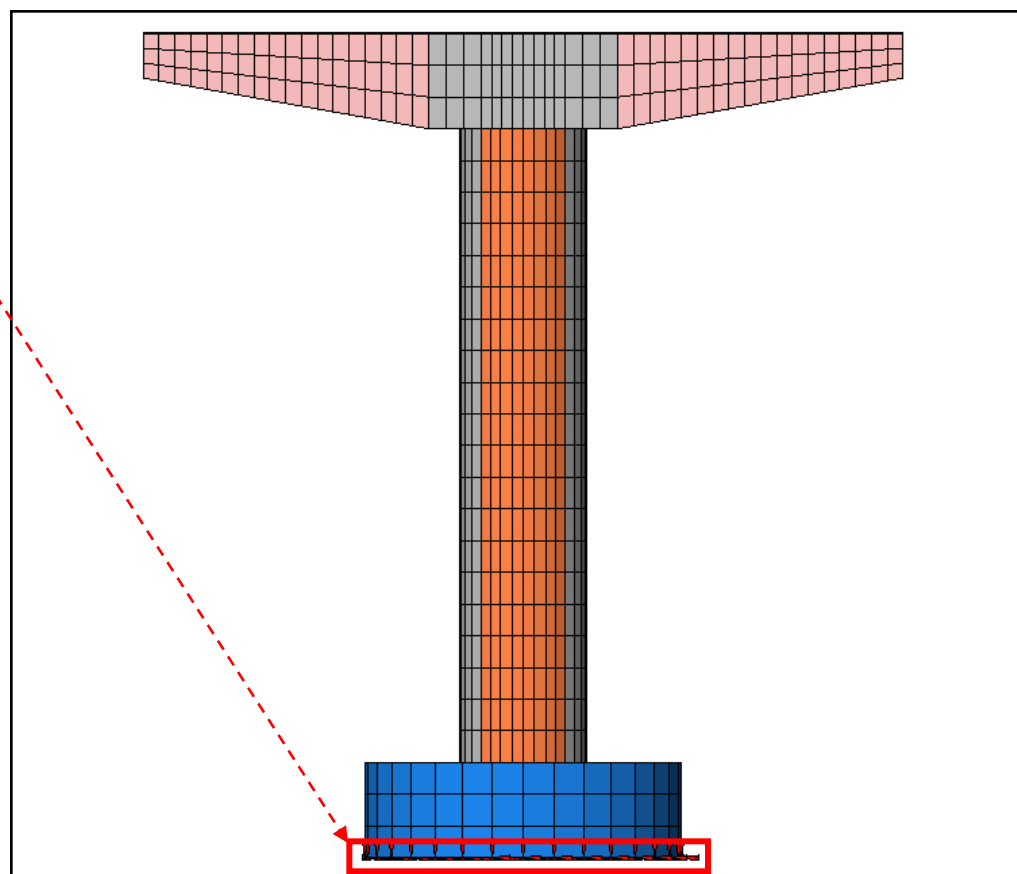


1. Analysis > Property ...
2. Create 3D ...
3. ID : 1 , Name : Pier
4. Click  Button
5. Click [Create] Button
6. Select "Isotropic" Tab
7. ID : 1 , Name : Conc
8. Elastic Modulus : $2.4e6 \text{ N/cm}^2$
9. Poisson's Ratio : 0.167
10. Weight Density : $2.452e-2 \text{ N/cm}^3$
11. Model Type : Elastic
12. Click [OK] Button
13. Click [Close] Button
14. Select "1: Conc" for Material
15. Click [OK] Button
16. Click [Close] Button

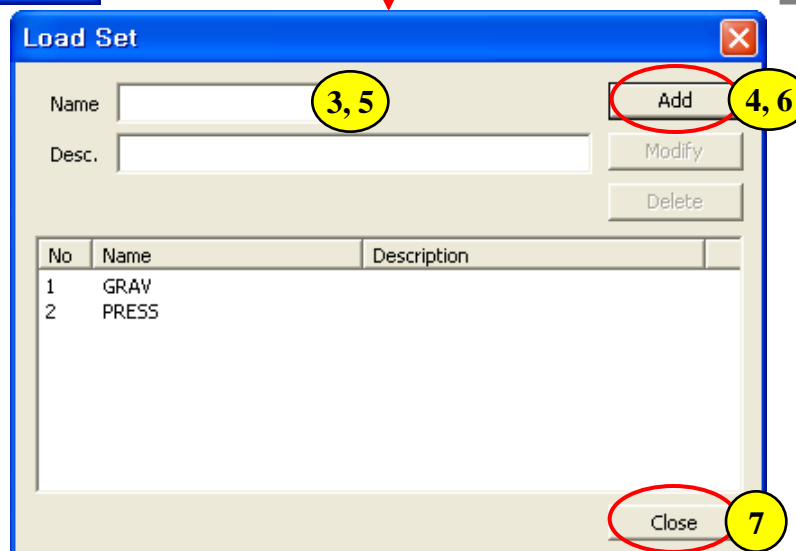
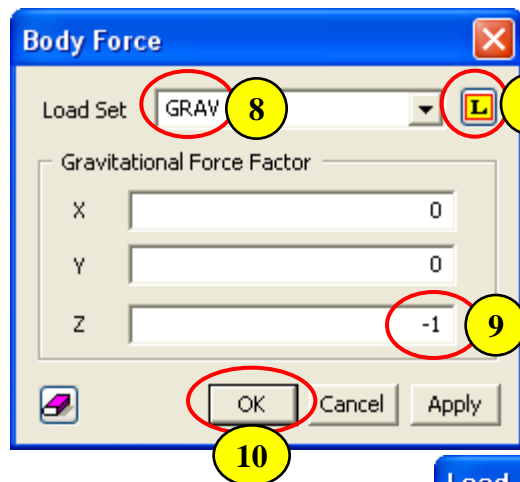
Step 18.



1. Analysis > BC > Constraint ...
2. BC Set : BC
3. Click “Front View”
4. Select 223 Nodes (See Figure)
5. Click [Pinned] Button
6. Click [OK] Button

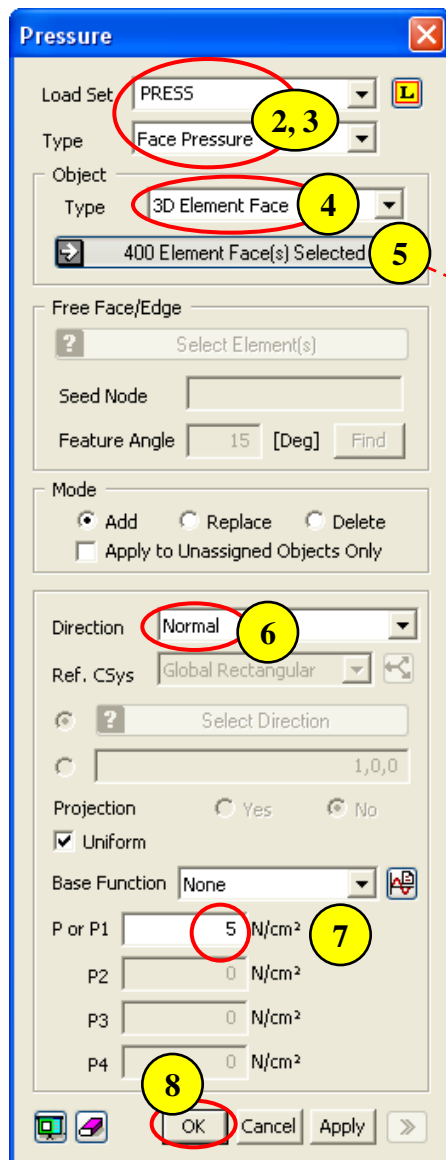


Step 19.

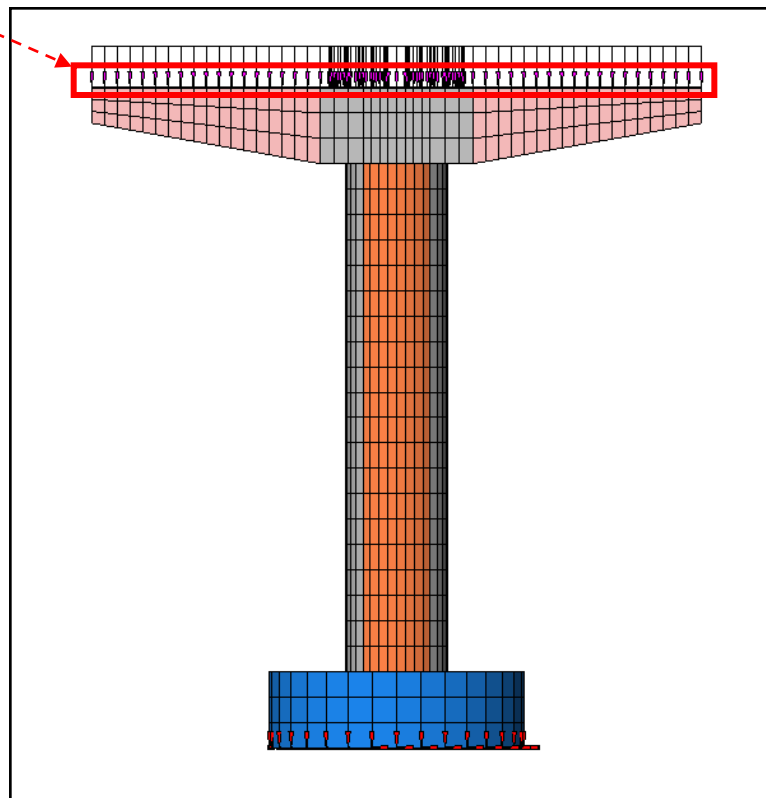


1. Analysis > Load > Body Force ...
2. Click Button
3. Name : GRAV
4. Click [Add] Button
5. Name : PRESS
6. Click [Add] Button
7. Click [Close] Button
8. Load Set : GRAV
9. Gravitational Force Factor : Z (-1)
10. Click [OK] Button

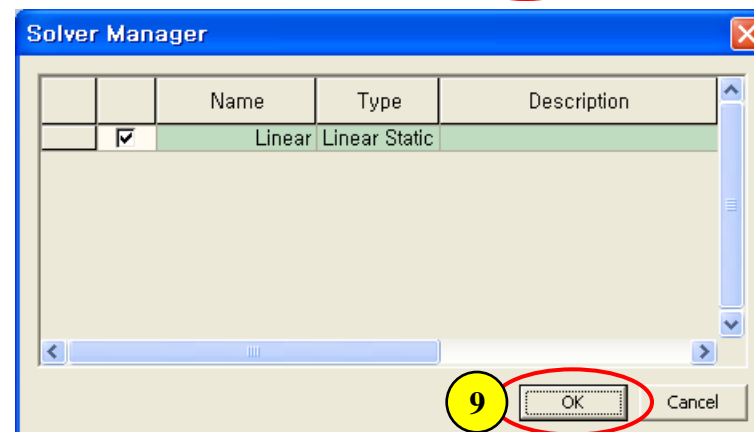
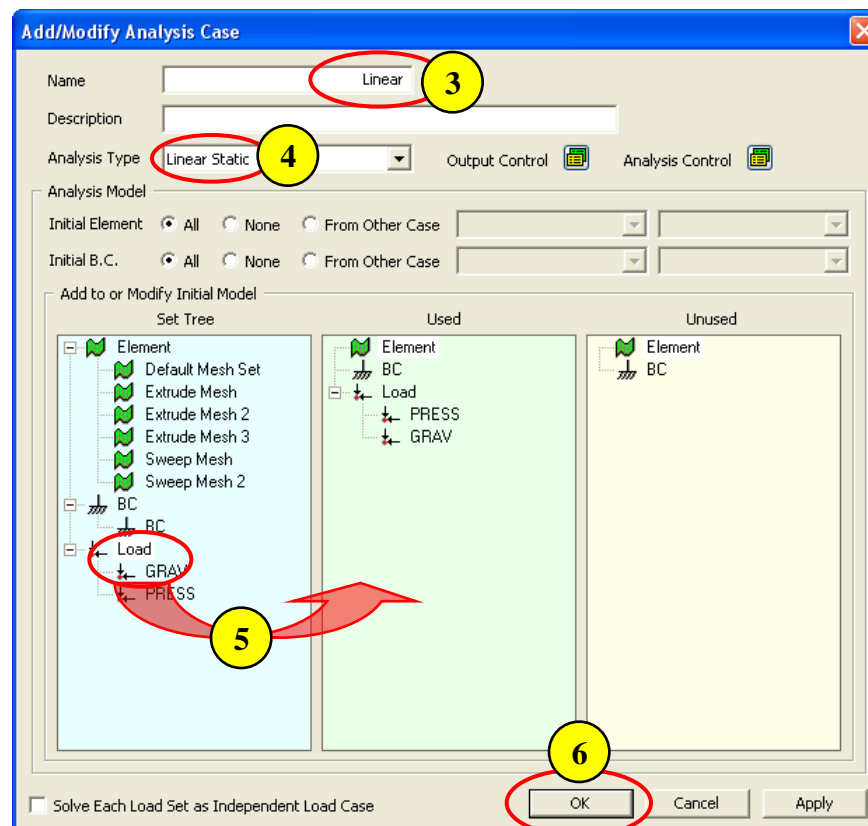
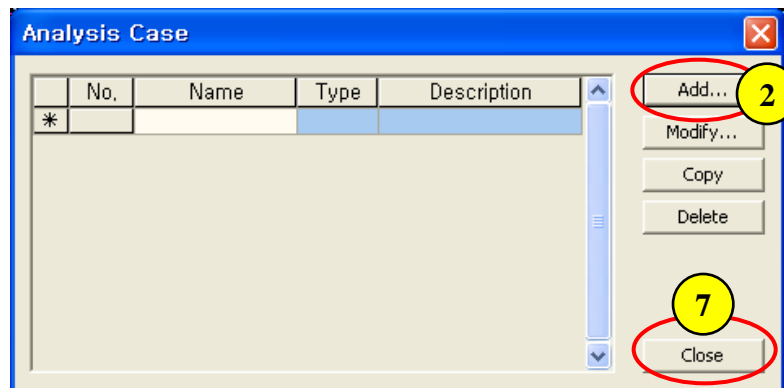
Step 20.



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: 5 N/cm²
8. Click [OK] Button

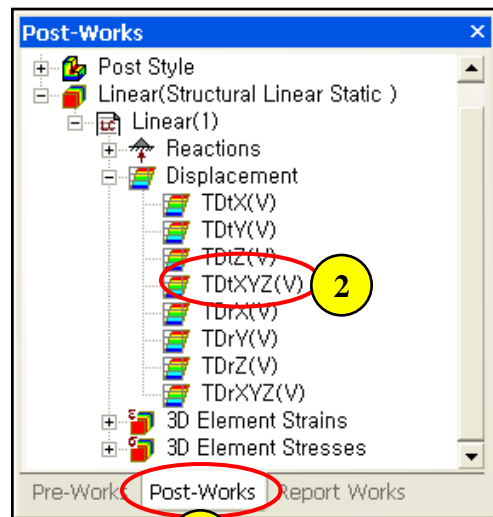


Step 21.

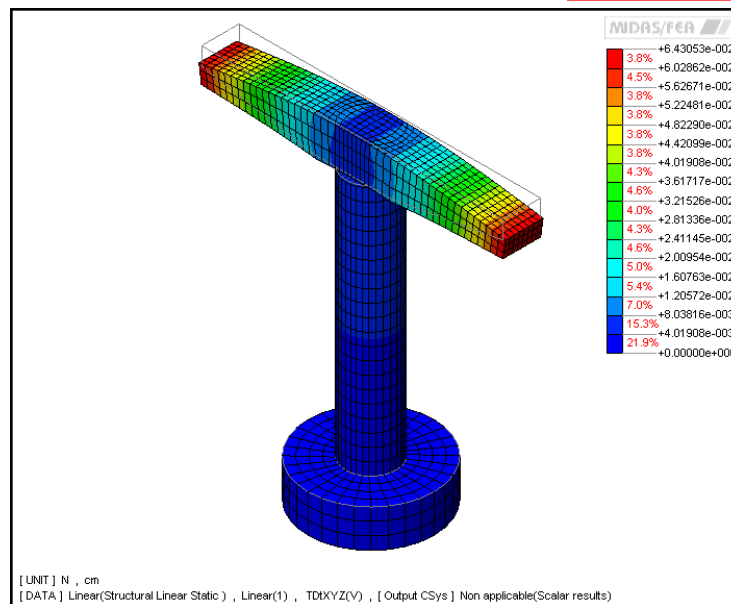
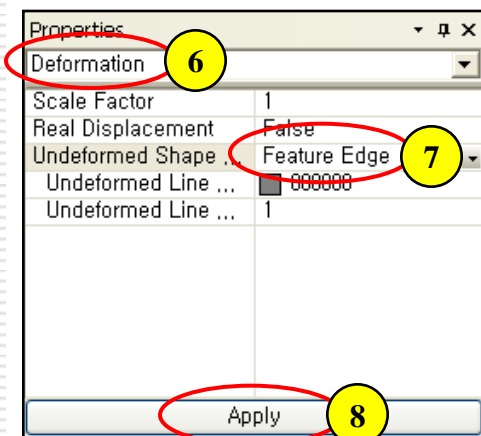
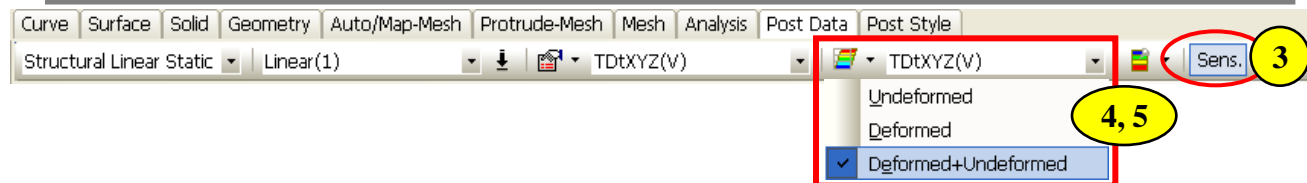


1. Analysis > Analysis Case ...
2. Click [Add] Button
3. Name : Linear
4. Analysis Type : Linear Static
5. Drag & Drop "Load" to "Used" Window
6. Click [OK] Button
7. Click [Close] Button
8. File > Save ... (Pier Map.feb)
9. Analysis > Solve ...
10. Click [OK] Button

Step 22.



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



Step 23.

1. Select “Undeformed” for Mesh Shape

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

3. Double Click “LO-SOLID, P1(V)”

