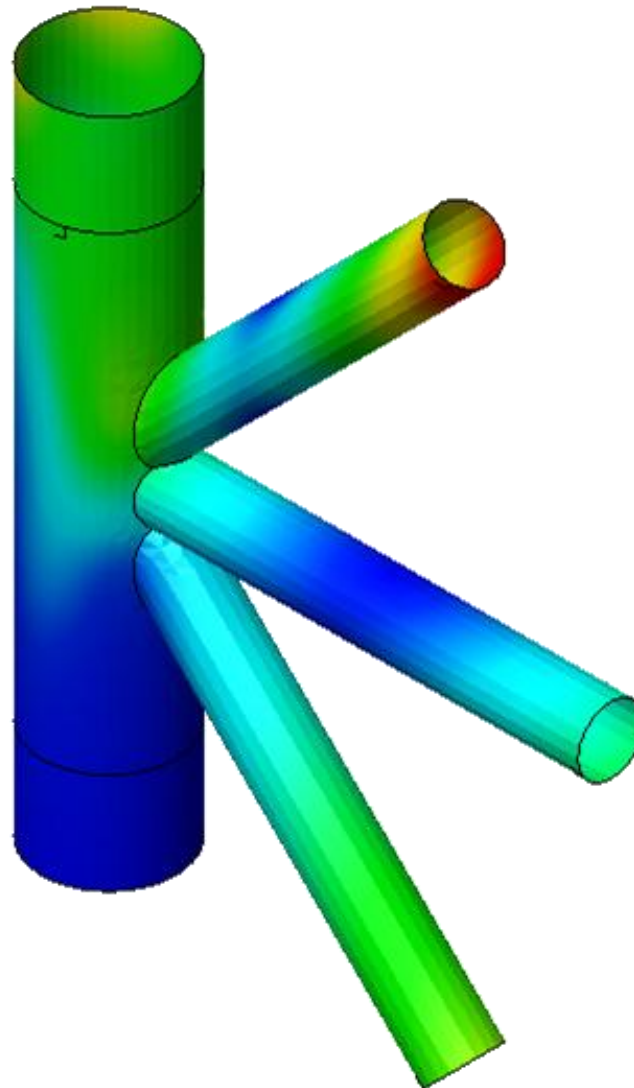


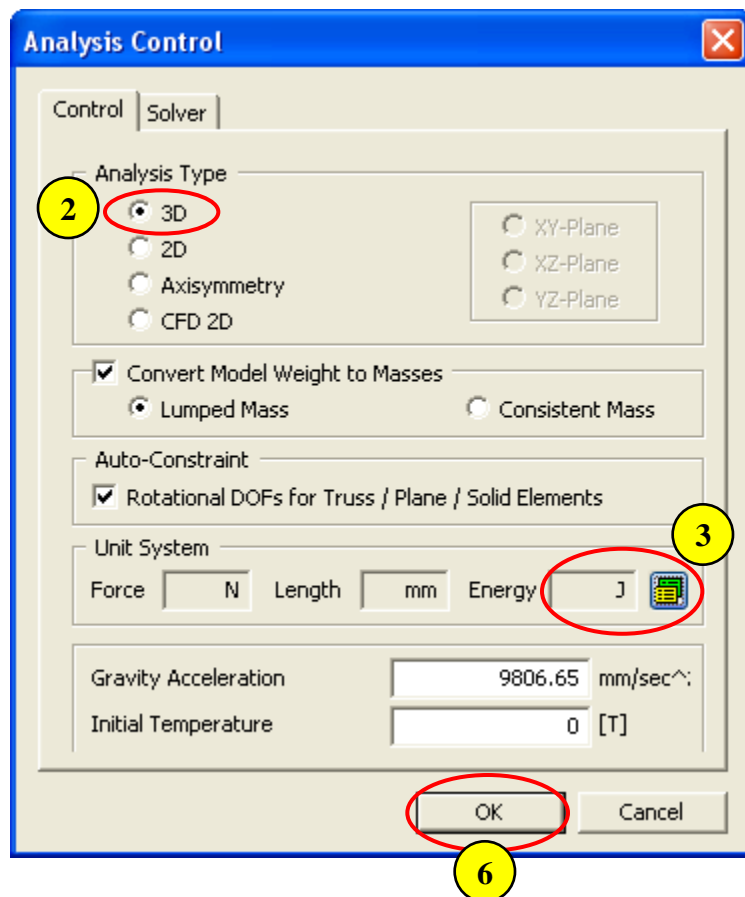
BK-1. Buckling Pipe



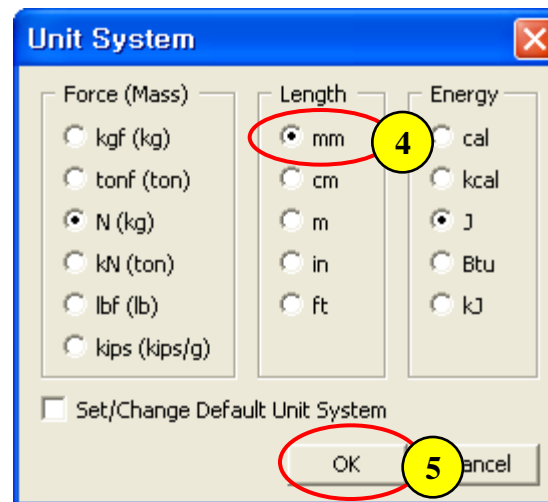
Overview

- 3-D Linear Buckling Analysis
- Model
 - Unit : N, mm
 - Isotropic Elastic Material
 - Plate Element
- Load & Boundary Conditions
 - Pressure
 - Constraint
- Result Evaluation
 - Buckling Mode Shapes

Step 1.

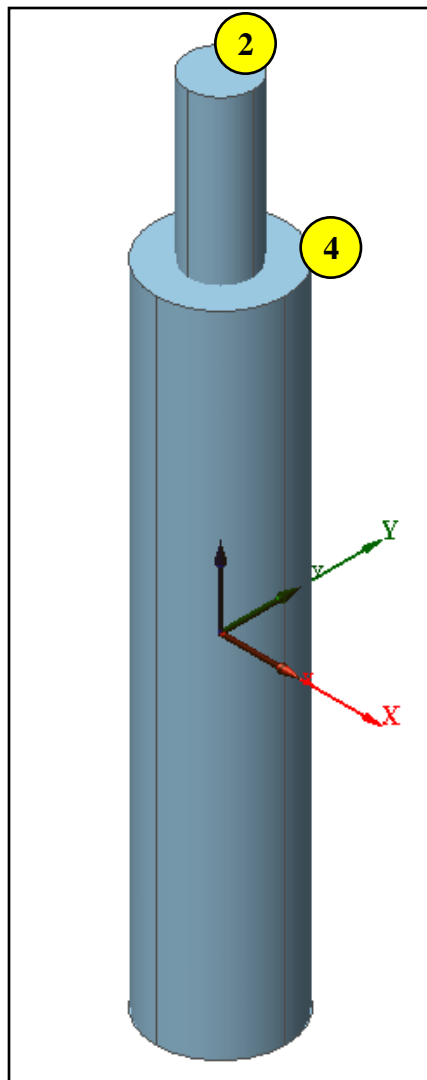
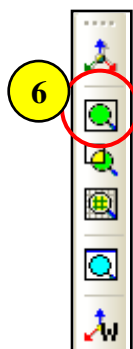
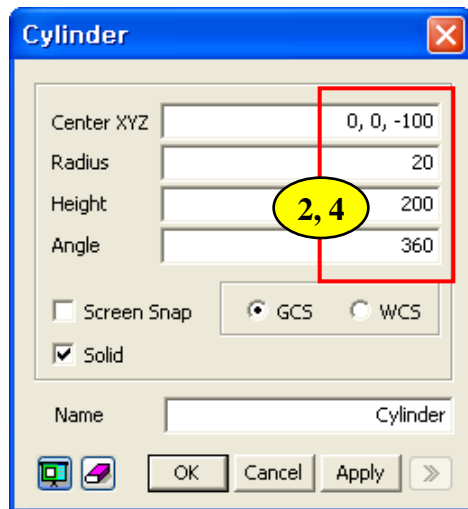


1. Analysis > Analysis Control – “Control” tab
2. Analysis Type : 3D
3. Click Button (Unit System)
4. Length : mm
5. Click [OK] Button
6. Click [OK] Button



Analysis Control Dialog is automatically activated at startup.

Step 2.



1. *Geometry > Primitive Feature > Cylinder...*

2. *C1 : Center XYZ (0, 0, -100)*

Radius : 20

Height : 200

3. *Click [Apply] Button*

4. *C2 : Center XYZ (0, 0, 0)*

Radius : 10

Height : 150

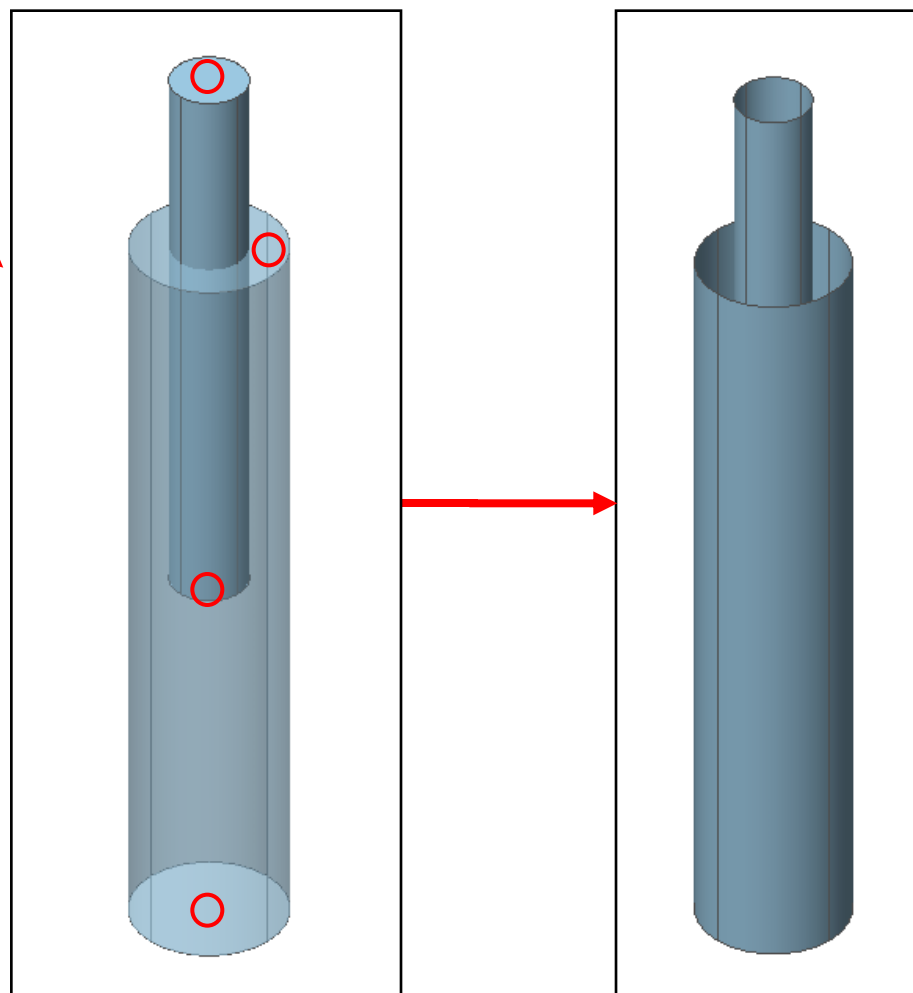
5. *Click [OK] Button*

6. *Click "Zoom All"*


Step 3.

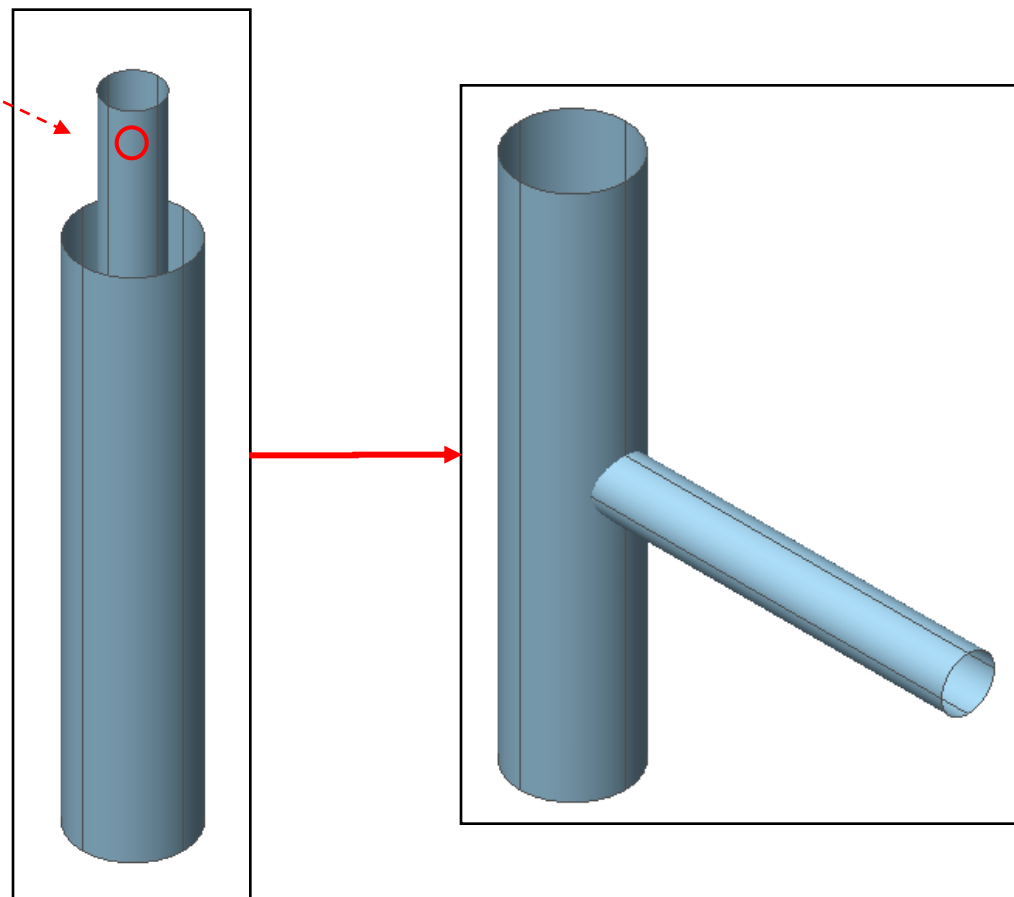
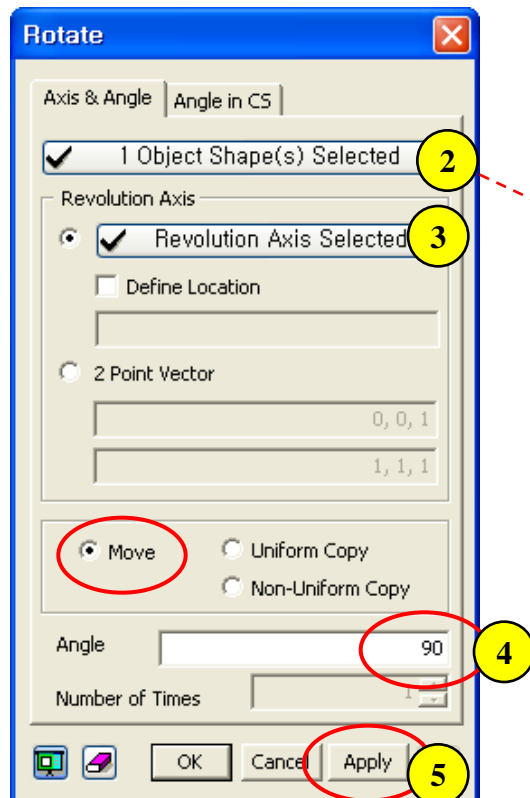


1. *Geometry > Remove > Sub-face ...*
2. *Select Top & Bottom Faces of 2 Solids marked by “O” (See Figure)*
3. *Click [OK] Button*



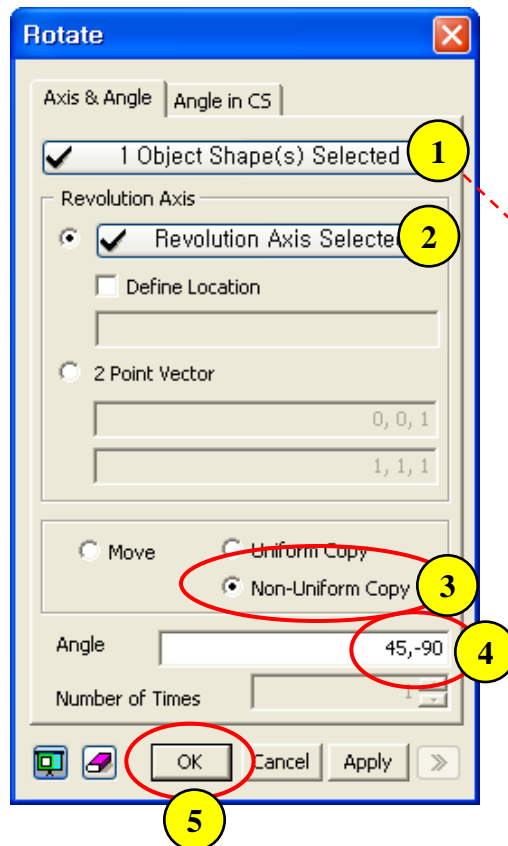
Step 4.

1. *Geometry > Transform > Rotate ...*
2. *Select Pipe marked by “O” (See Figure)*
3. *Revolution Axis : Y-axis*
4. *Angle (90)*
5. *Click [Apply] Button* 

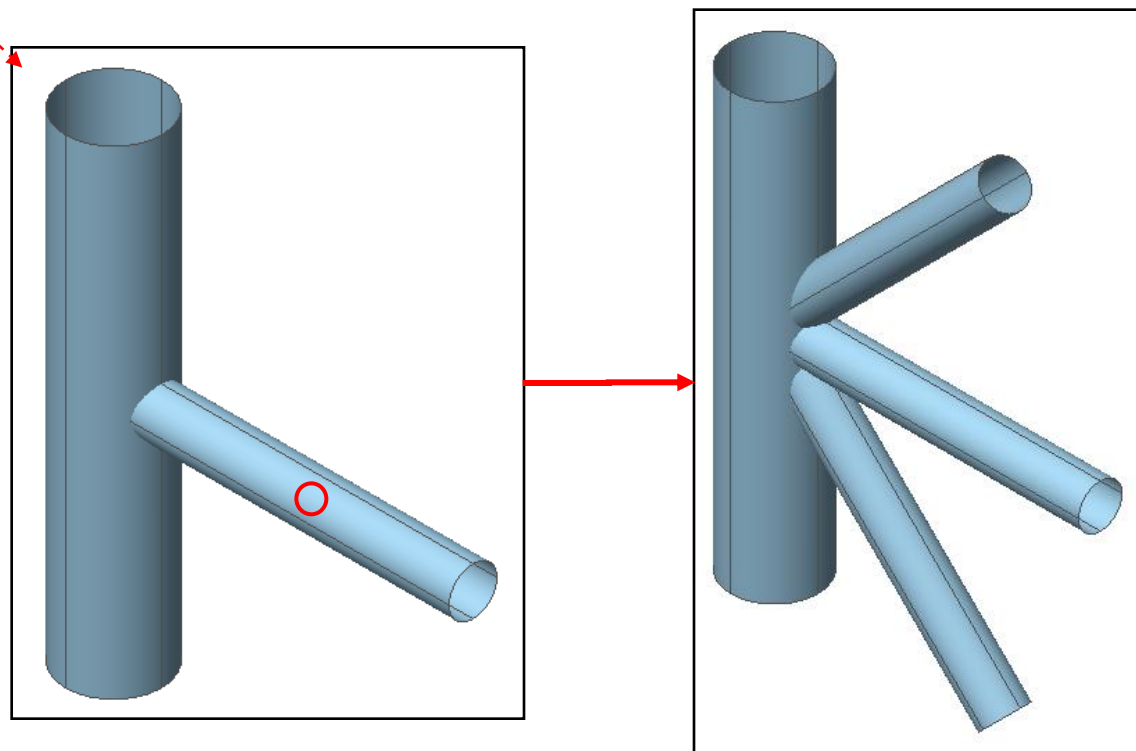


 “Enter” as shortcut for “Apply”.

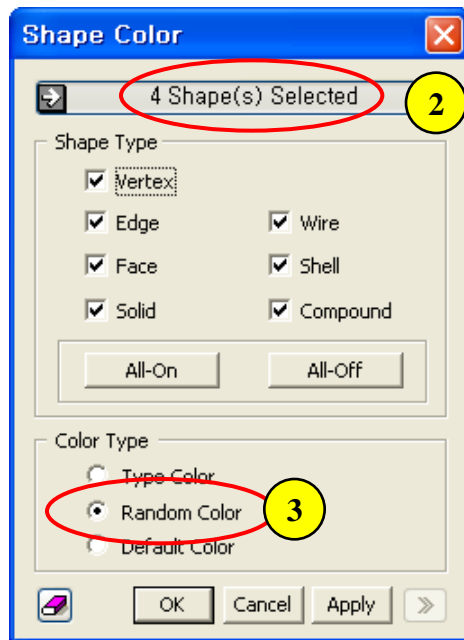
Step 5.




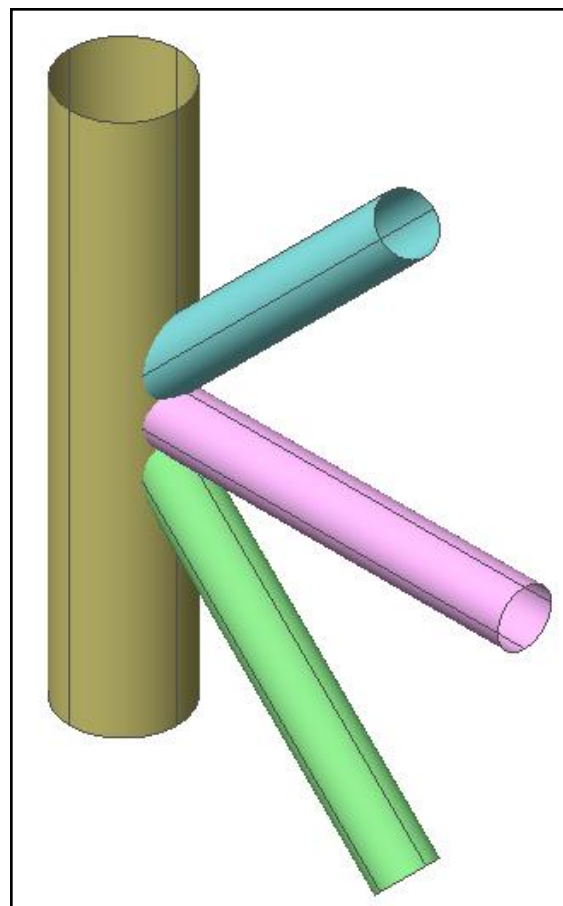
1. Select Pipe marked by "O" (See Figure)
2. Revolution Axis : Y-axis
3. Option : "Non-Uniform Copy"
4. Angle (45, -90)
5. Click [OK] Button




Step 6.

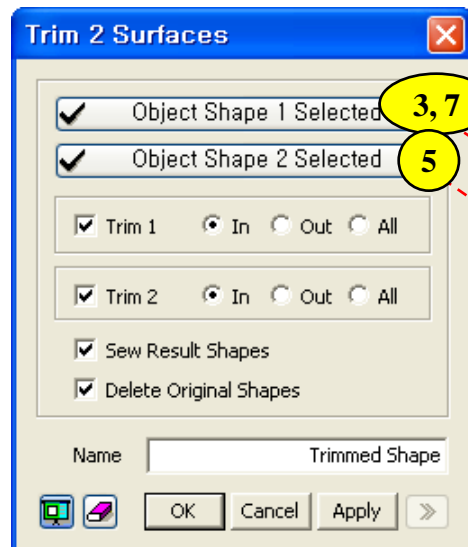


1. Geometry > Shape Color ...
2. Select  "Displayed" ⁹
3. Color Type : Random Color
4. Click [OK] Button

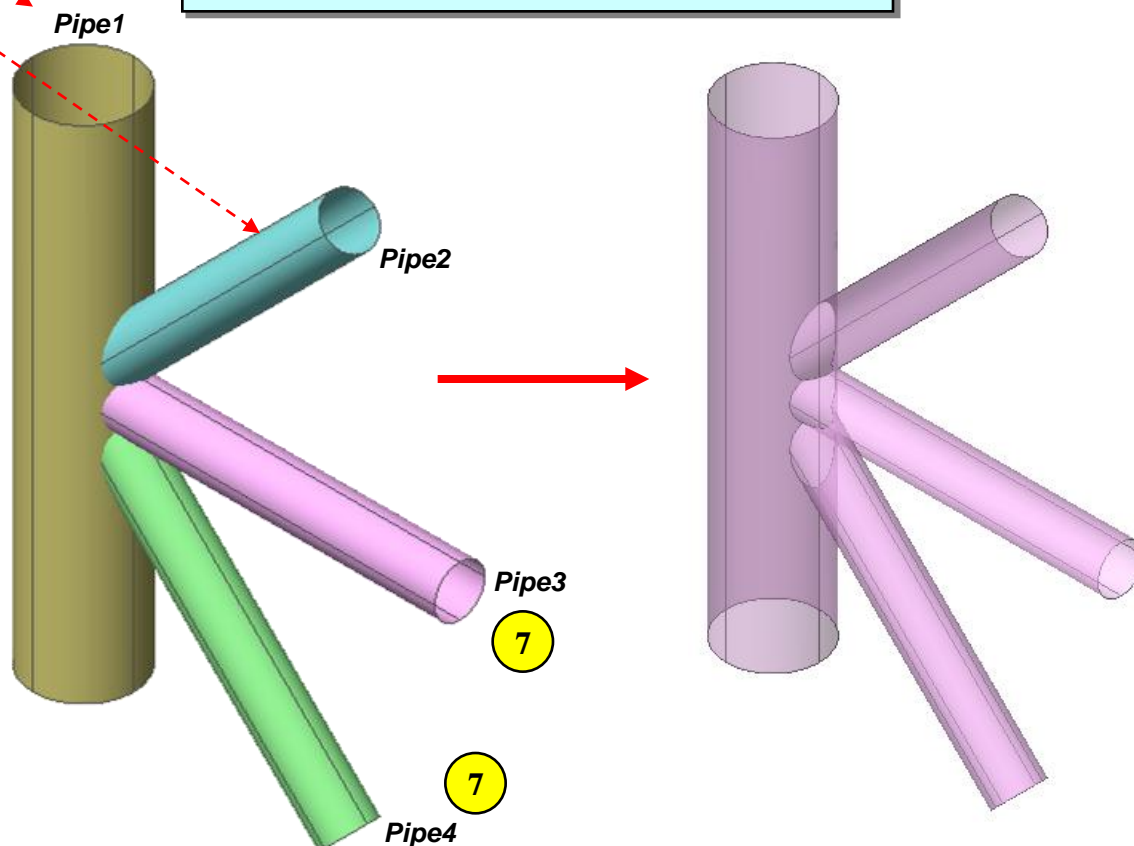
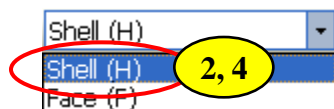


 "Ctrl+A" as shortcut for "Select Displayed".

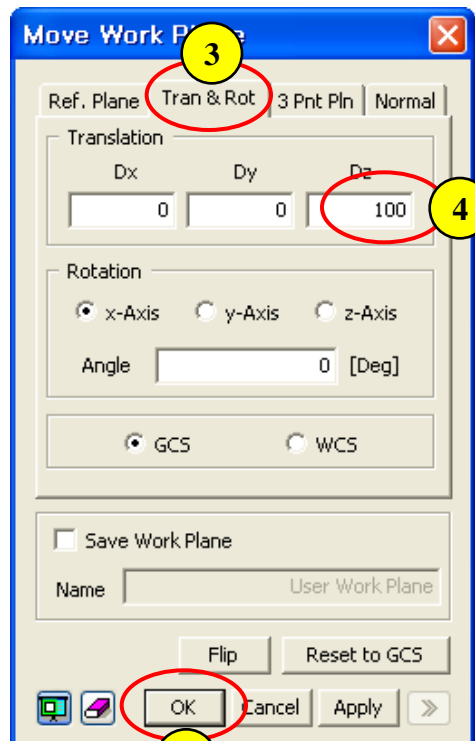
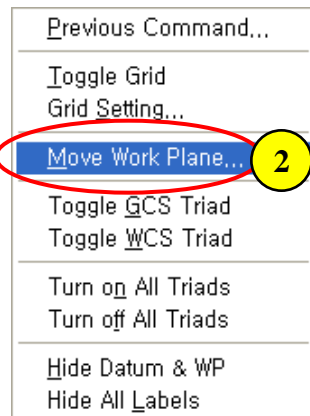
Step 7.



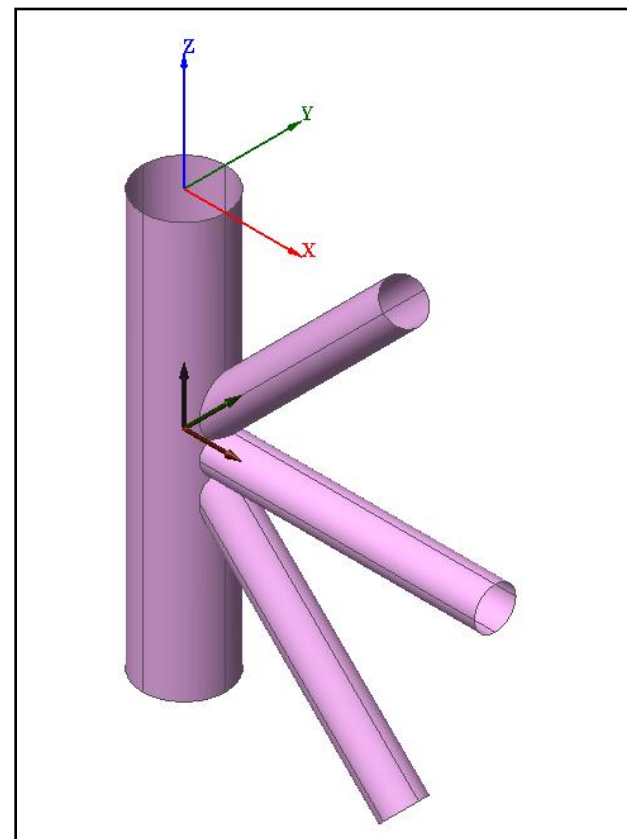
1. Geometry > Surface > Trim 2 Surfaces ...
2. Change Selection Filter to "Shell (H)"
3. Select Pipe1
4. Change Selection Filter to "Shell (H)"
5. Select Pipe2
6. Click [Apply] Button
7. Repeat steps 2-6 for Pipe3 & 4



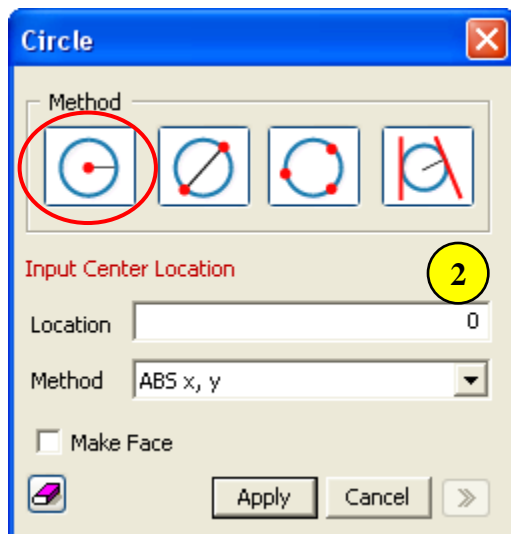
Step 8.



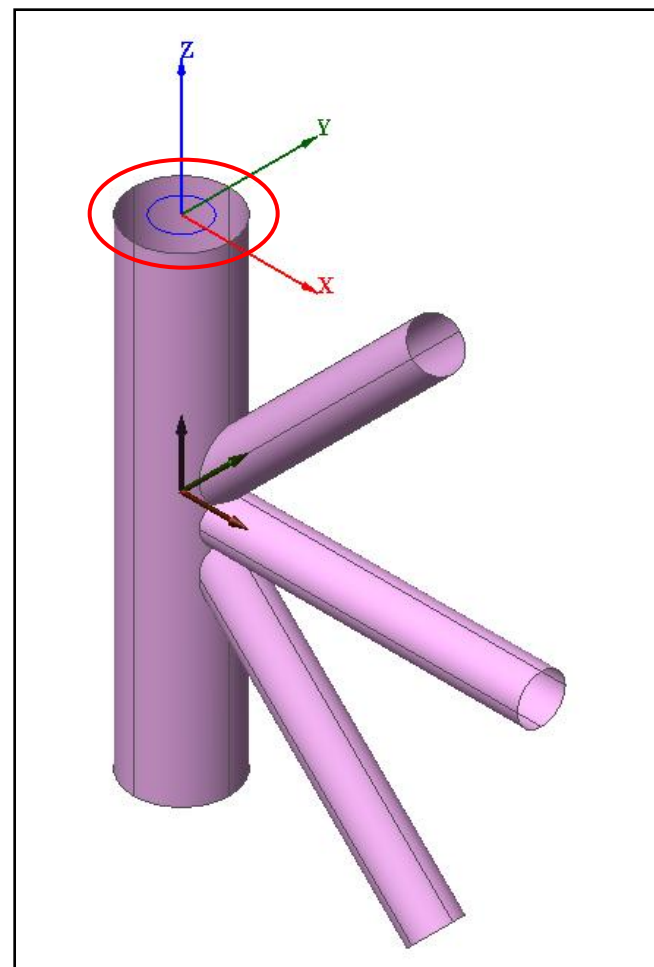
1. Click Right Mouse Button on Work Window
2. Select "Move Work Plane ..."
3. Select "Tran & Rot" Tab
4. Translation : Dz (100)
5. Click [OK] Button



Step 9.

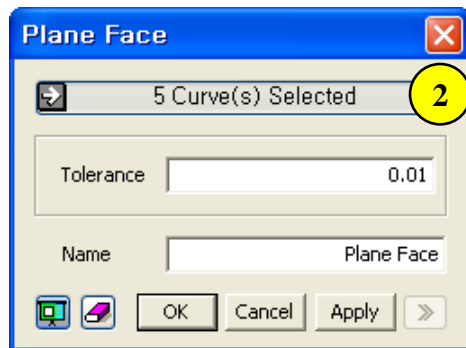


1. **Geometry > Curve > Create on WP > Circle ...**
2. **Location : (0), (10)**
3. **Click [Cancel] Button** ⚙

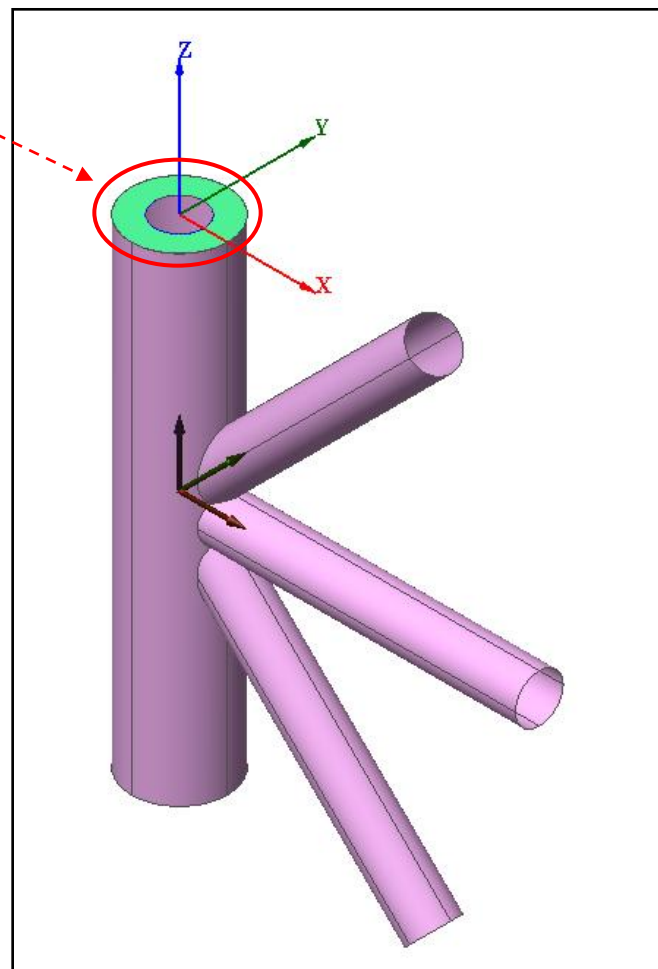


⚙ **"Esc" as shortcut for "Cancel".**

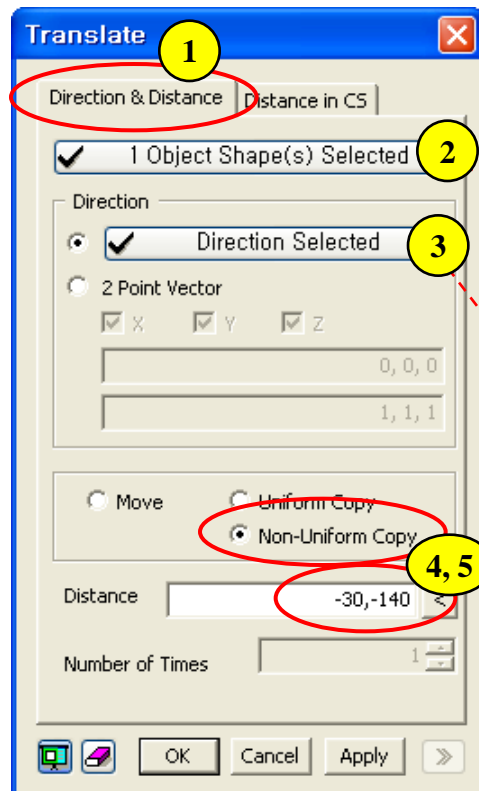
Step 10.



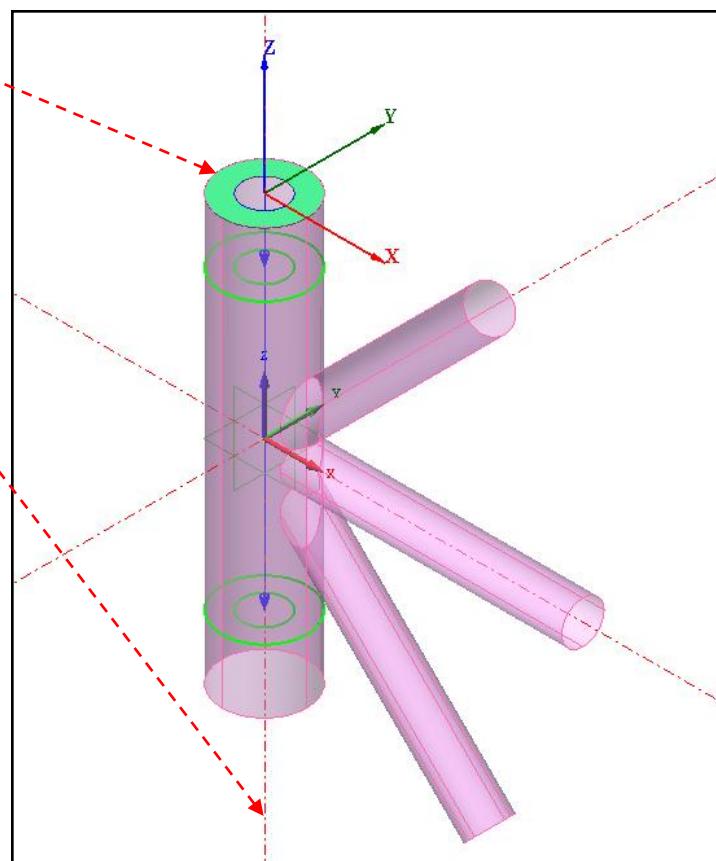
1. *Geometry > Surface > Create > Plane Face ...*
2. *Select 5 Curves (See Figure)*
3. *Click [OK] Button*




Step 11.

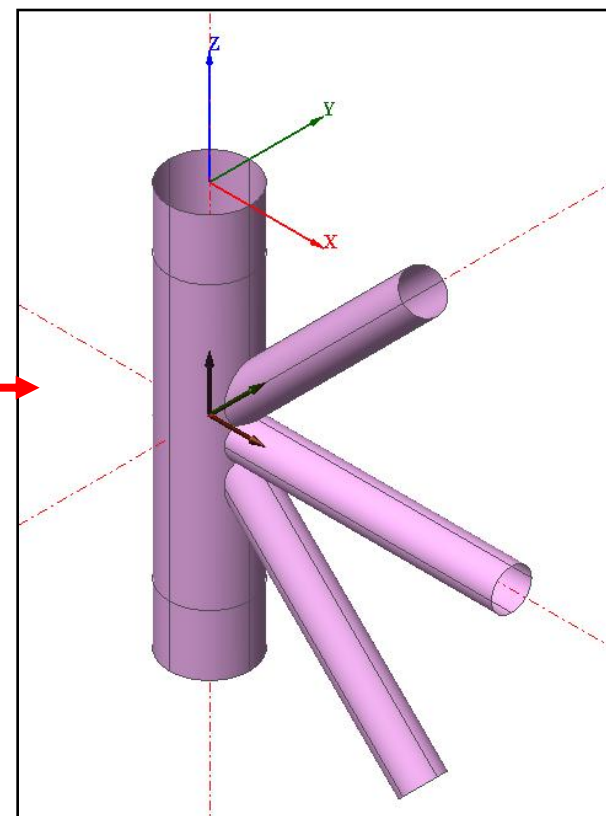
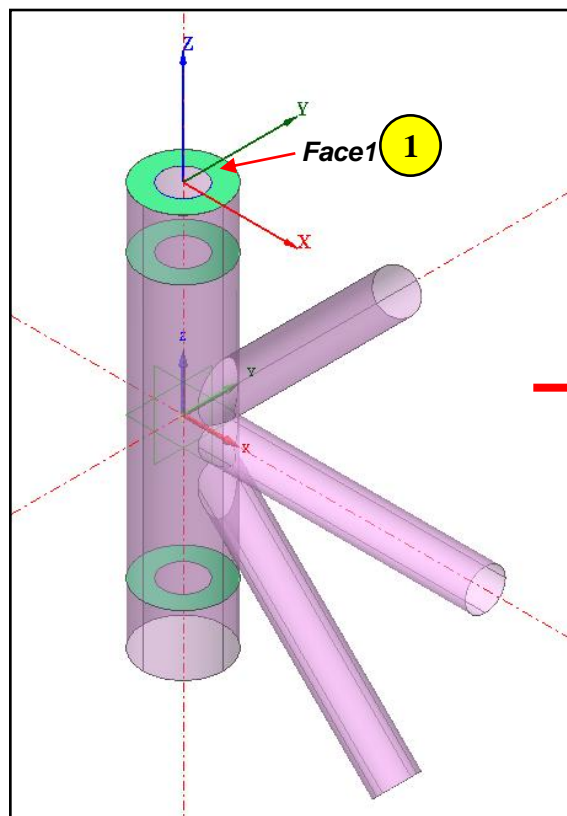
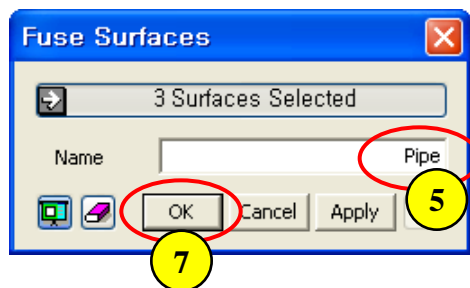
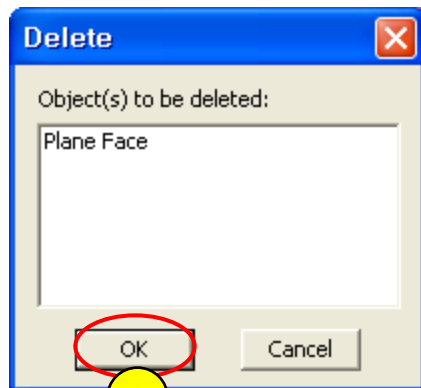


1. Geometry > Transform > Translate - “Direction & Distance” tab
2. Select Face (See Figure)
3. Direction : Z-axis
4. Check on “Non-Uniform Copy”
5. Distance : (-30, -140)
6. Click [OK] Button

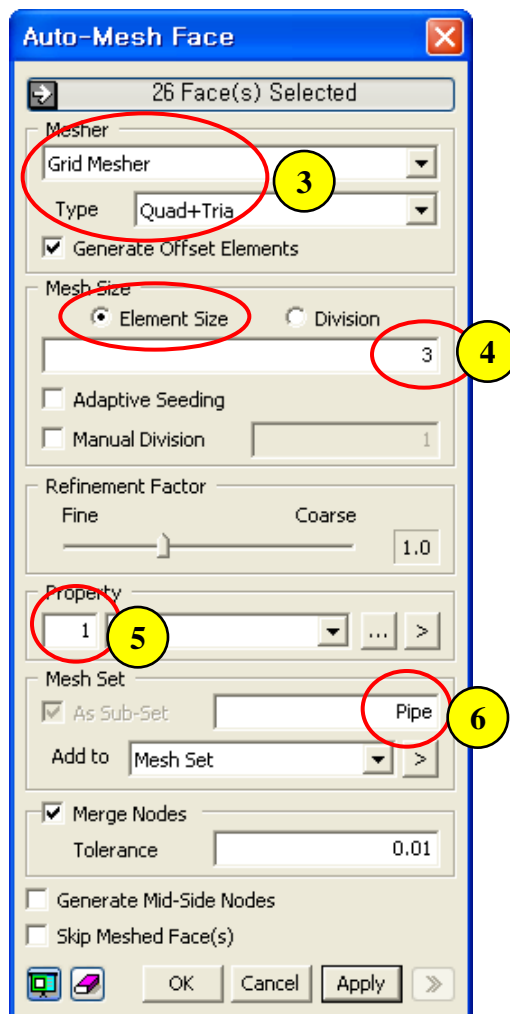



Step 12.

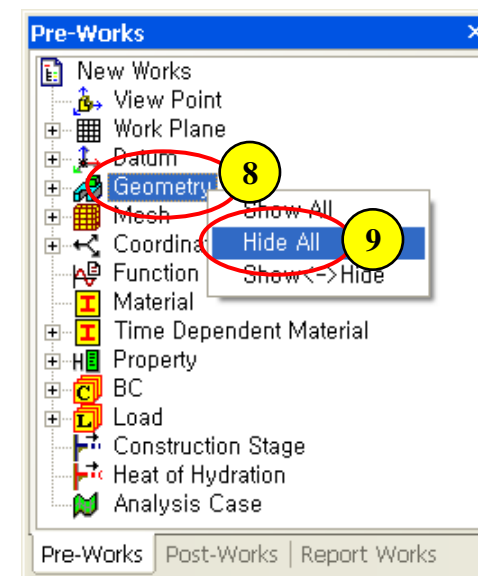
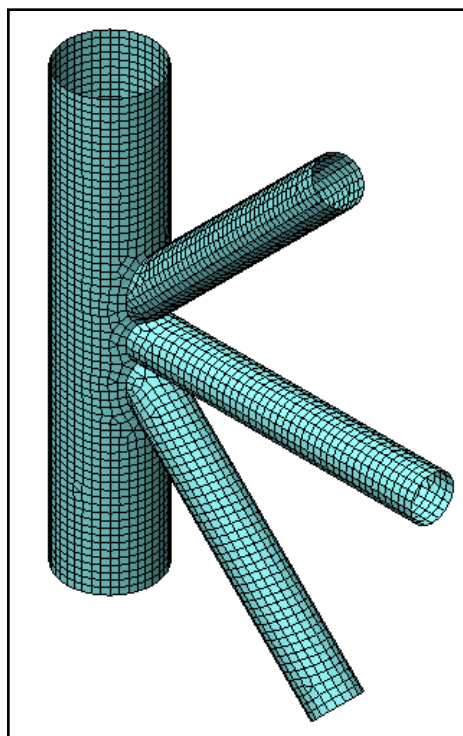
1. Select Face1 (See Figure)
2. Press "Delete" key
3. Click [OK] Button
4. Geometry > Surface > Fuse ...
5. Select  "Displayed"
6. Name : Pipe
7. Click [OK] Button



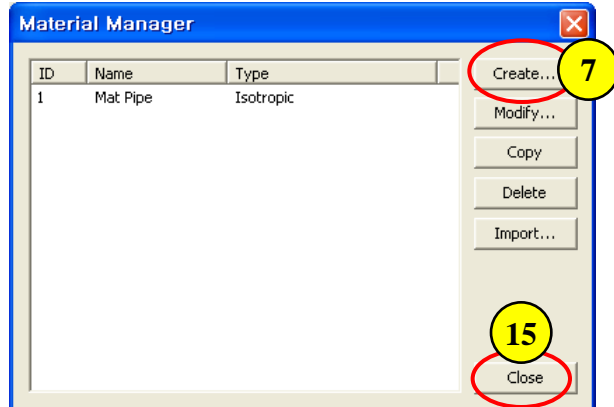
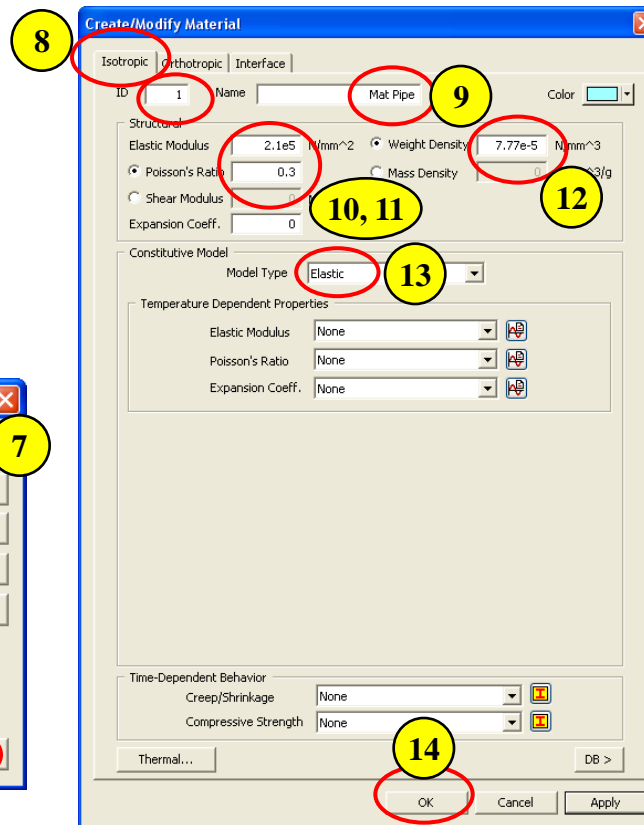
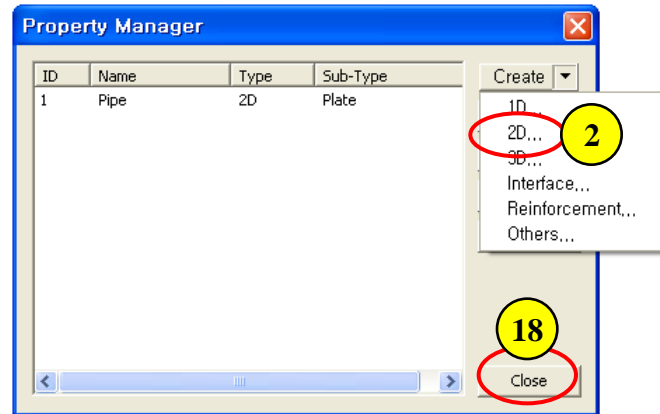
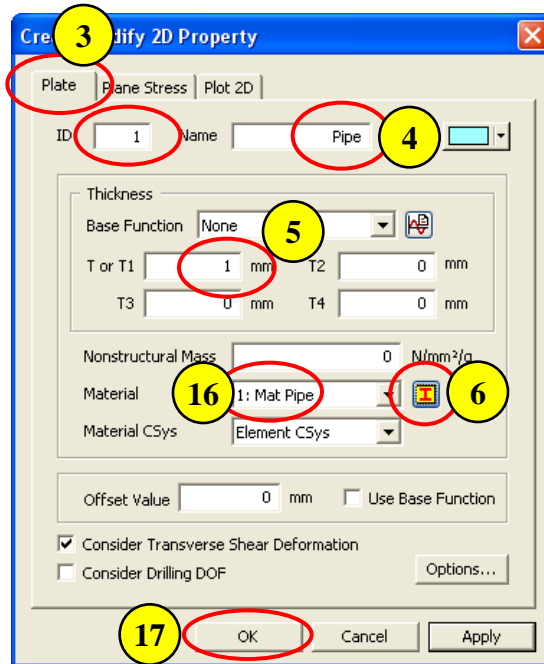
Step 13.




1. Mesh > Auto Mesh > Face ...
2. Select  "Displayed"
3. Mesher : Grid Mesher , Type : Quad+Tria
4. Mesh Size : Element Size (3)
5. Property (1)
6. Mesh Set : Pipe
7. Click [OK] Button
8. Pre-Works Tree : Geometry
9. Click Right Mouse Button & Select "Hide All"

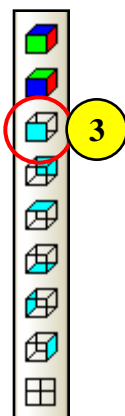
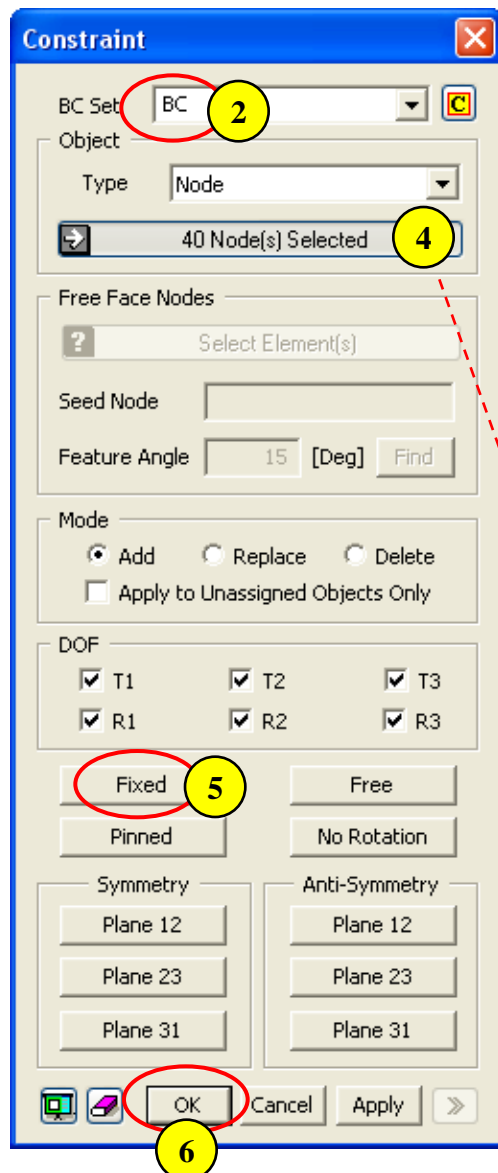


Step 14.

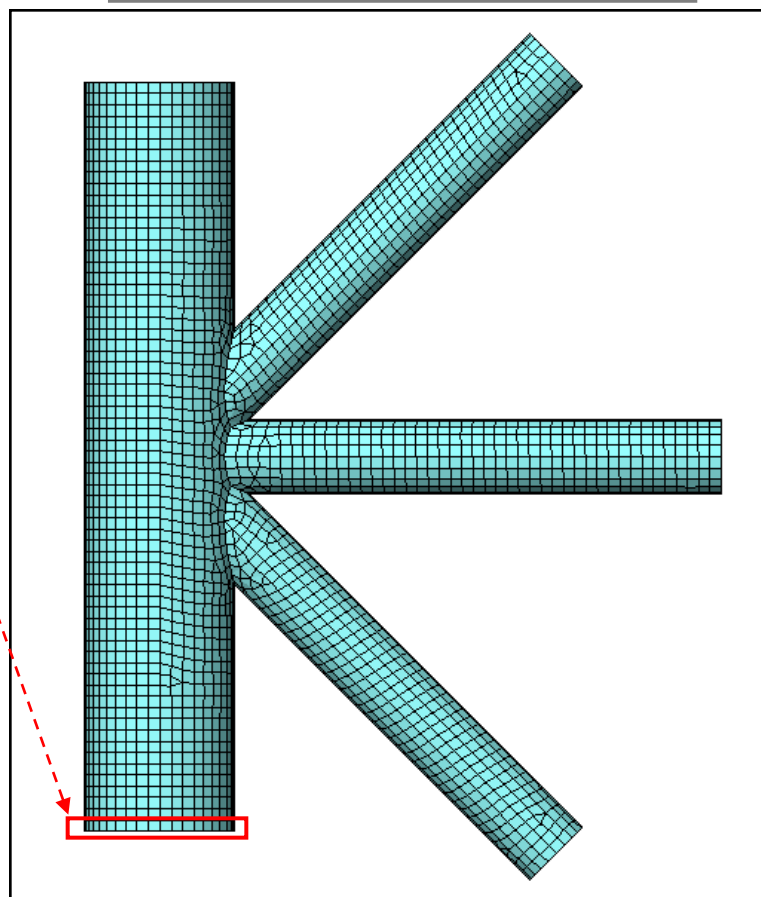


1. Analysis > Property ...
2. Create 2D ...
3. Select "Plate" tab
4. ID : 1 , Name : Pipe
5. T or T1 : 1 mm
6. Click  Button (Material)
7. Click [Create] Button
8. Select "Isotropic" tab
9. ID : 1 , Name : Mat Pipe
10. Elastic Modulus : 2.1e5 N/mm²
11. Poisson's Ratio : 0.3
12. Weight Density : 7.77e-5 N/mm³
13. Model Type : Elastic
14. Click [OK] Button
15. Click [Close] Button
16. Select "1: Mat Pipe" for Material
17. Click [OK] Button
18. Click [Close] Button

Step 15.

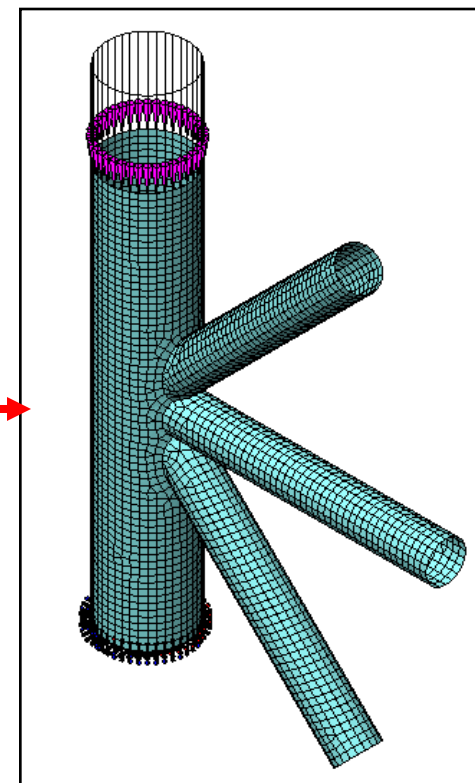
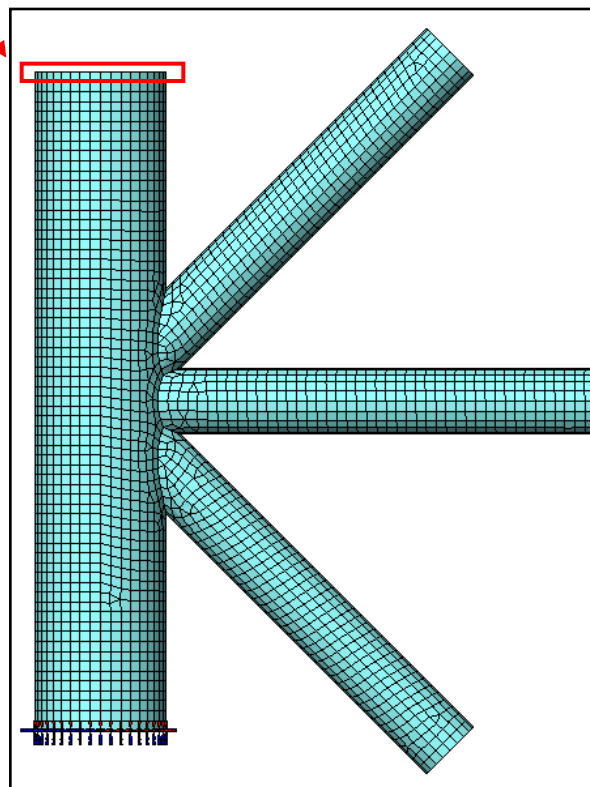
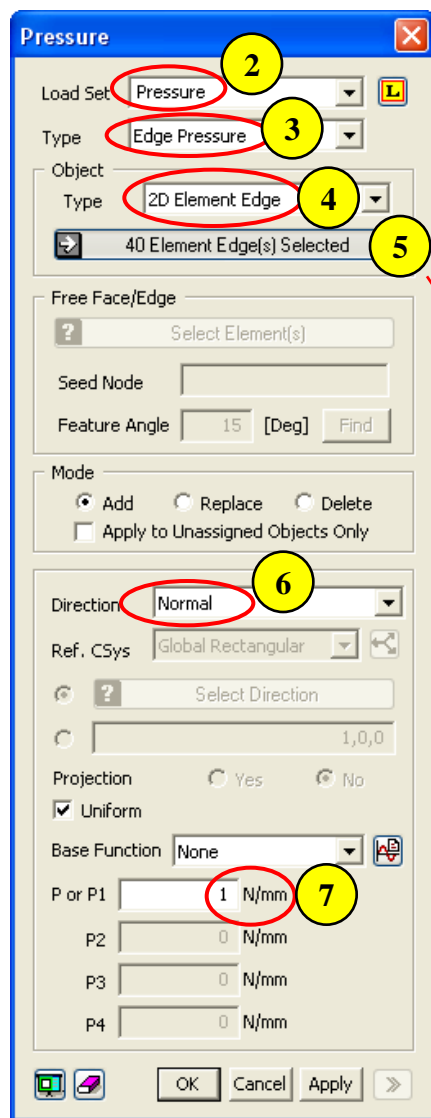


1. Analysis > BC > Constraint ...
2. BC Set : BC
3. Click "Front View"
4. Select 40 Nodes (See Figure)
5. Click [Fixed] Button
6. Click [OK] Button

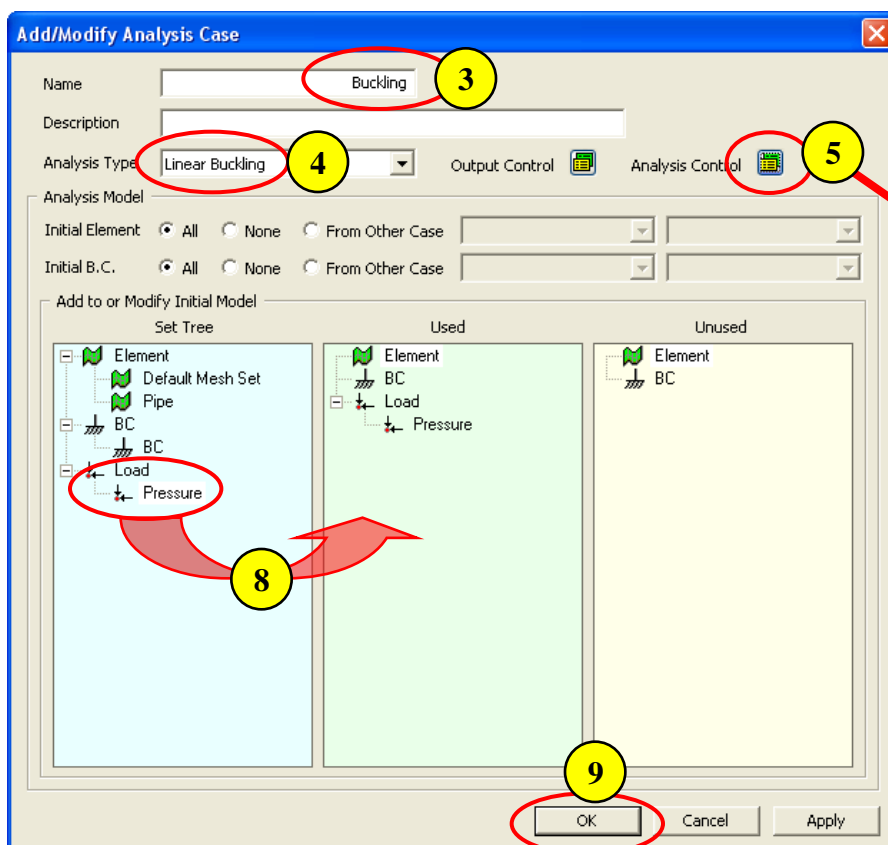
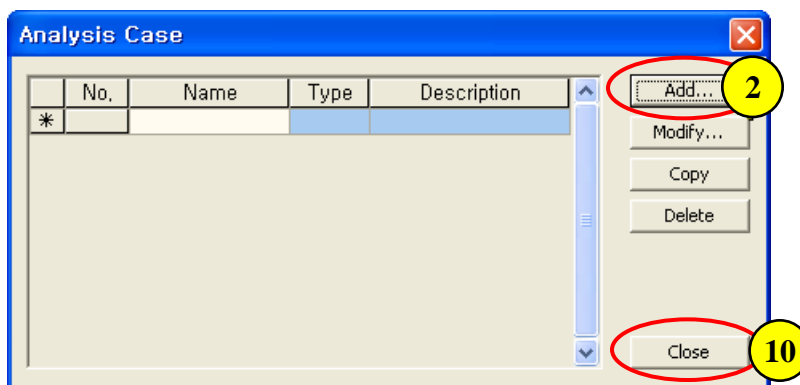



Step 16.

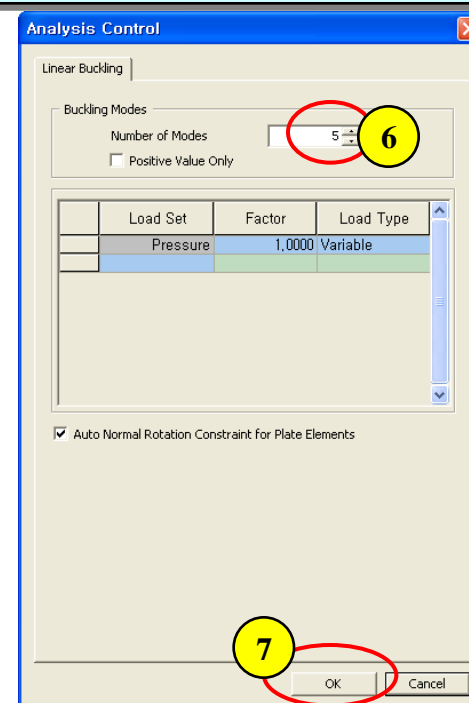
1. Analysis > Load > Pressure ...
2. Load Set : Pressure
3. Type : Edge Pressure
4. Object Type : 2D Element Edge
5. Select 40 Element Edges (See Figure)
6. Direction : Normal
7. P or P1: 1 N/mm
8. Click [OK] Button



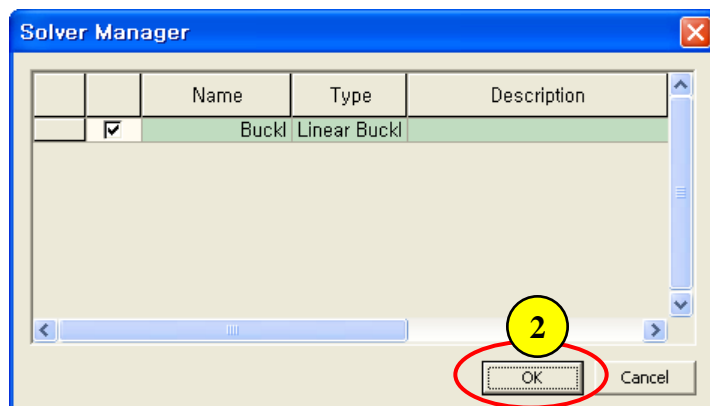
Step 17.



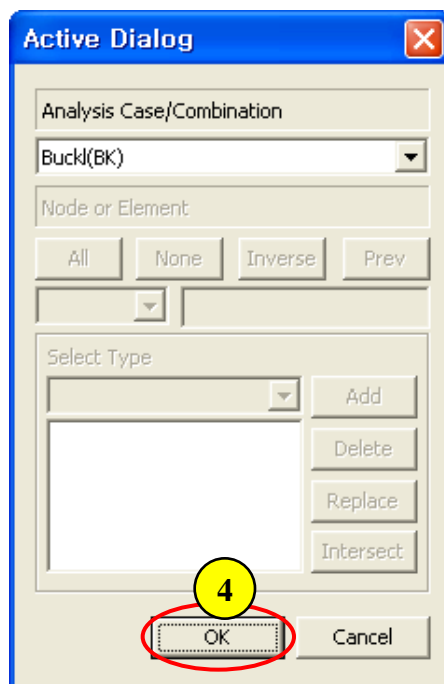
1. Analysis > Analysis Case ...
2. Click [Add] Button
3. Name : Buckling
4. Analysis Type : Linear Buckling
5. Click  Button (Analysis Control)
6. Number of Modes : 5
7. Click [OK] Button
8. Drag & Drop "Load" to "Used" Window
9. Click [OK] Button
10. Click [Close] Button



Step 18.



1. File > Save... (Pipe.feb)
2. Analysis > Solve ...
3. Click [OK] Button
4. Post > Buckling Load Factor ...
5. Click [OK] Button



BUCKLING ANALYSIS					
Mode	Load Factor	Tolerance			
1	1750.363037	0.0000e+000			
2	2004.106445	0.0000e+000			
3	2185.906006	0.0000e+000			
4	2203.235107	0.0000e+000			
5	2292.452637	0.0000e+000			