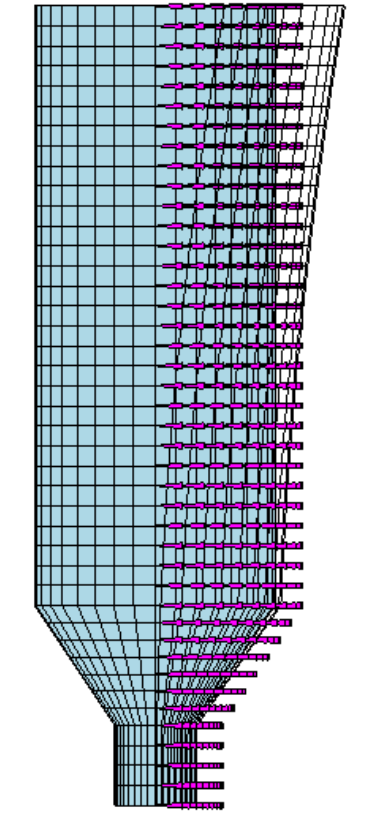
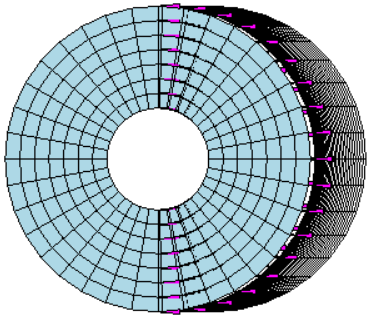
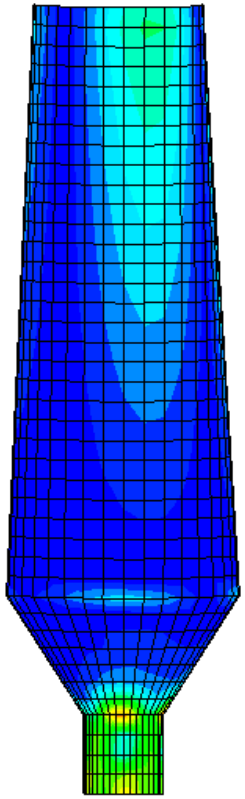
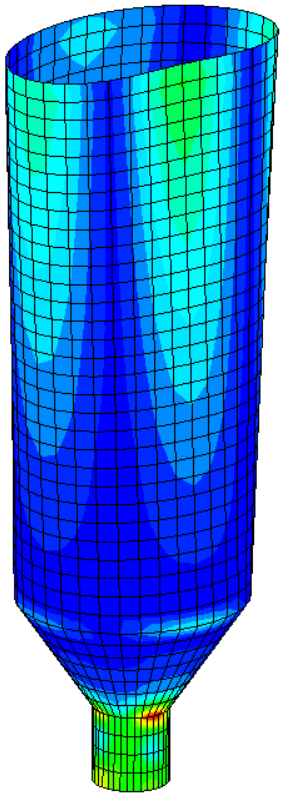
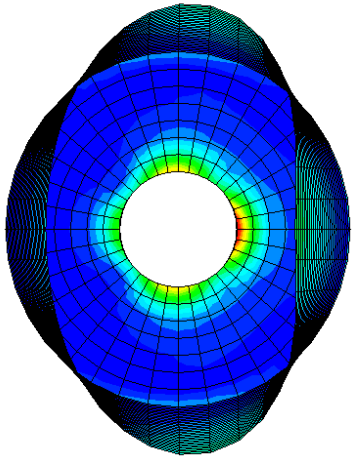
LS-3. Silo







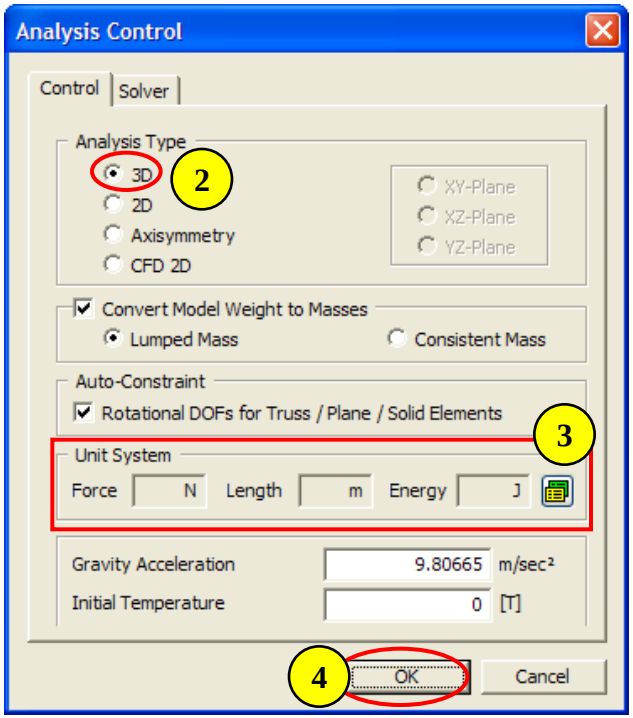


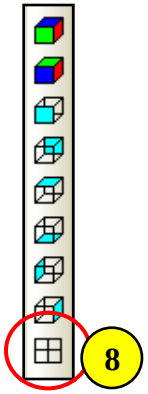


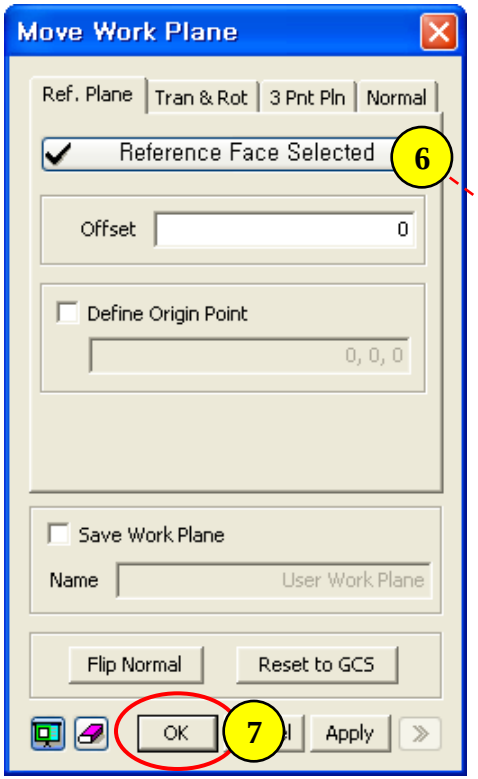
Overview

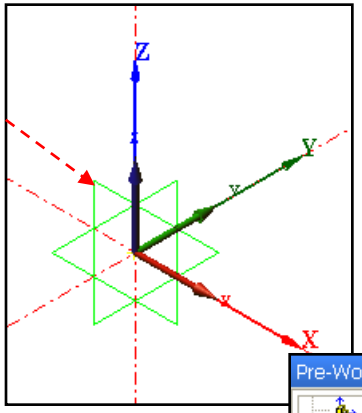
- **3-D Linear Static Analysis**
- **Model**
  - *Unit : N, m*
  - *Isotropic Elastic Material*
  - *Plate Elements*
- **Load & Boundary Condition**
  - *Self Weight*
  - *Face Pressure*
  - *Constraint*
- **Result Evaluation**
  - *Deformation*
  - *Maximum Shear Stress at Element Mid-Plane*

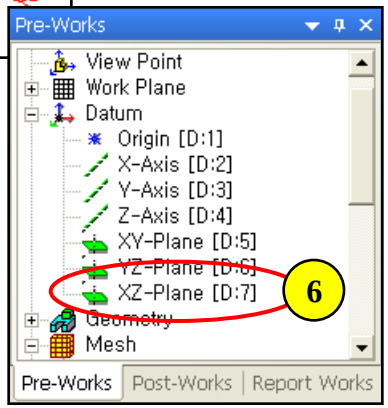
Step 1.







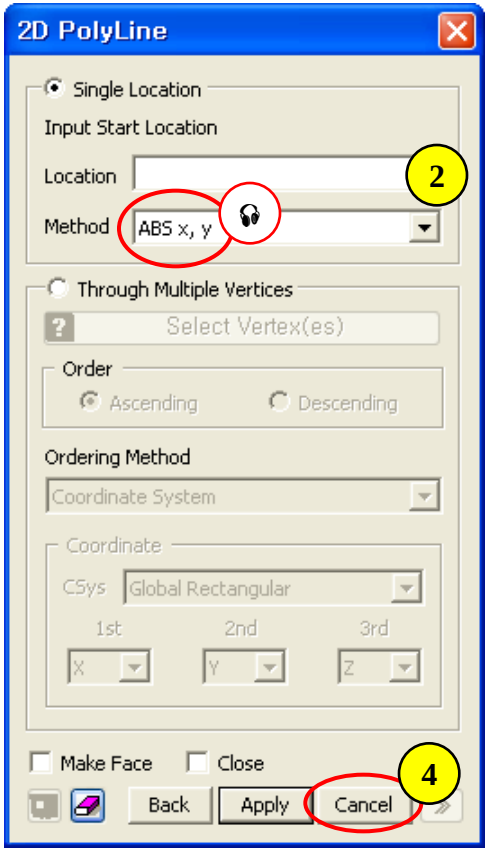


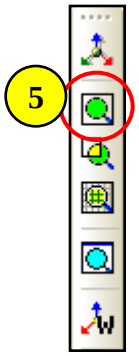


1. *Analysis > Analysis Control – Control tab*
2. *Analysis Type : 3D*
3. *Unit System : N , m*
4. *Click* **[OK]** *Button*
5. *Geometry > Work Plane > Move…*
6. *Select "XZ-Plane"*
7. *Click* **[OK]** *Button*
8. *Click "Normal View"*

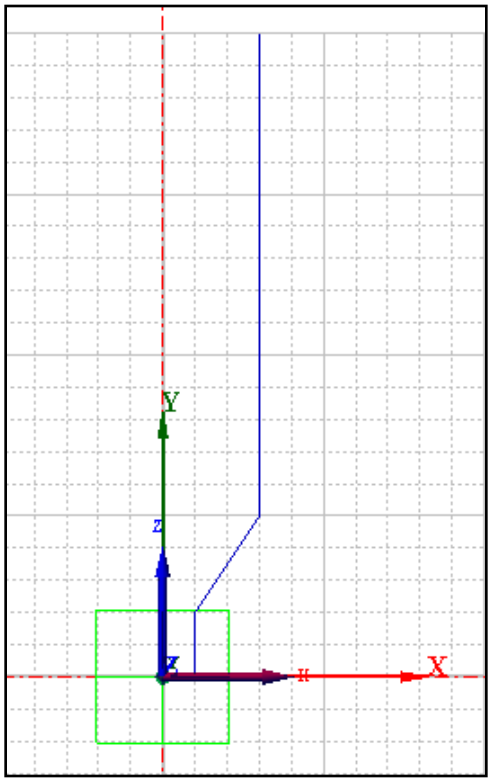
Select "XZ-Plane" in Work Window or Pre-Works Tree

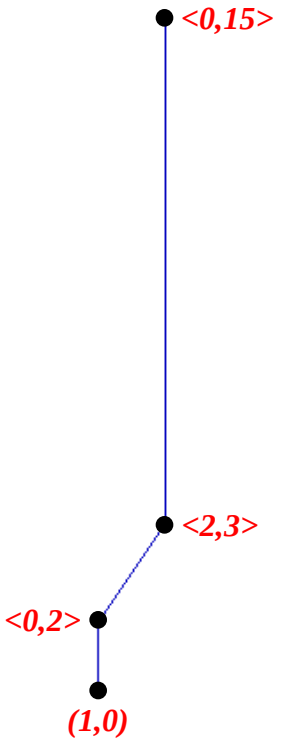
Step 2.





1. *Geometry > Curve > Create on WP > Polyline (Wire)…*
2. *Location : (1) , <0, 2> , <2, 3> , <0, 15>*
3. *Click Right Mouse Button in Work Window (to Stop Polyline Drawing)*
4. *Click* **[Cancel]** *Button*
5. *Click "Zoom All"*

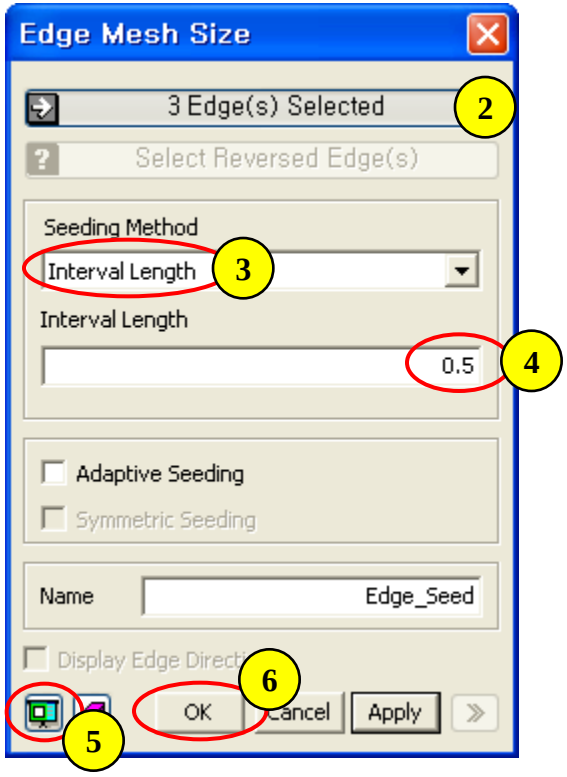


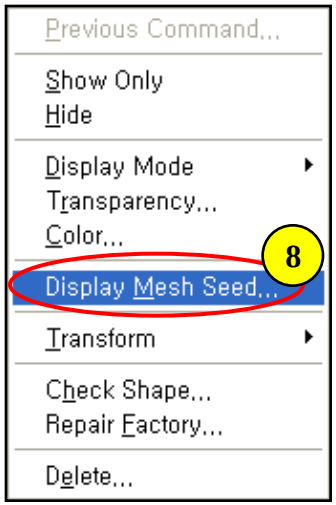


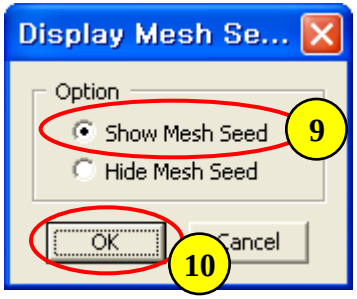
- *(): "ABS x, y", <>: "REL dx, dy"*
  (1) same as (1, 0)
- **[Esc]** *as shortcut for* **[Cancel].**

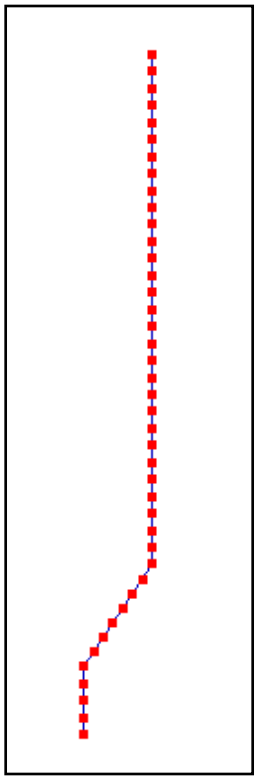
Step 3.

1. *Mesh > Size Control > Along Edge…*
2. *Select “Displayed”*
3. *Seeding Method : Interval Length*
4. *Interval Length : 0.5*
5. *Click “Preview” Button*
6. *Click* [OK] *Button*
7. *Select “Displayed”*
8. *Click Right Mouse Button and Select “Display Mesh Seed…”*
9. *Select “Show Mesh Seed”*
10. *Click* [OK] *Button*



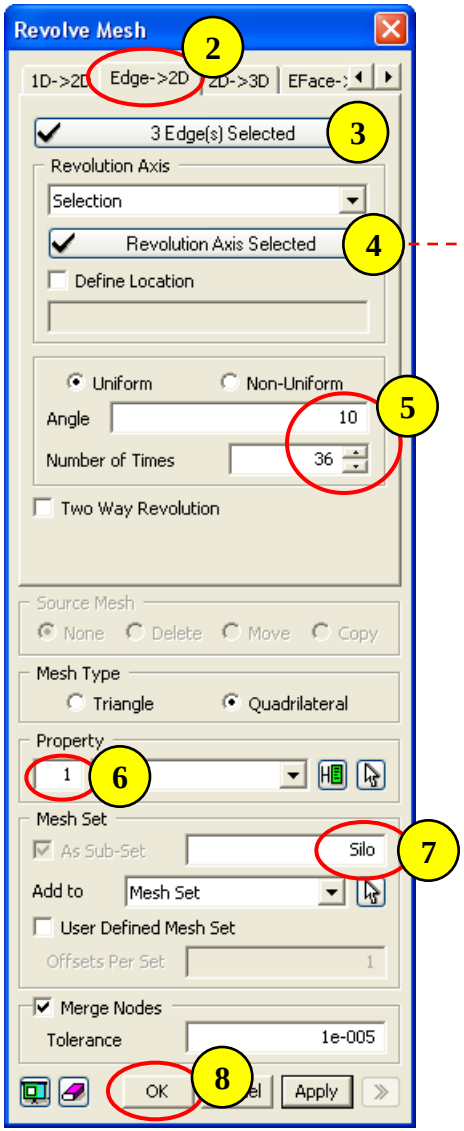


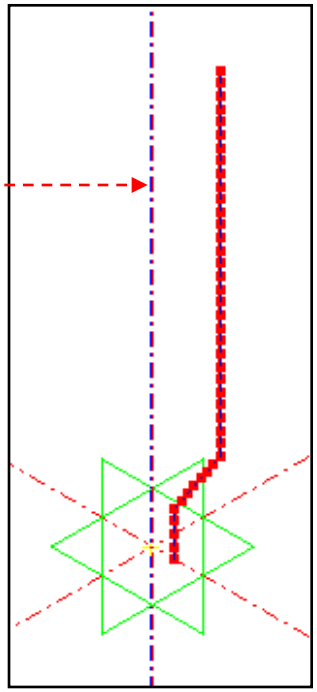




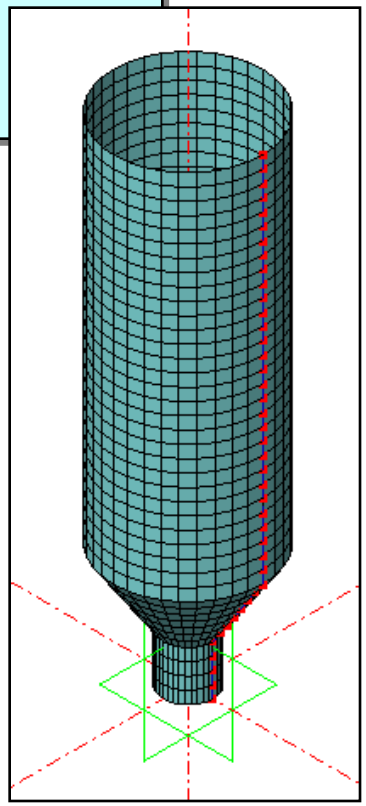
“Ctrl+A” as shortcut for “Select Displayed”.

Step 4.



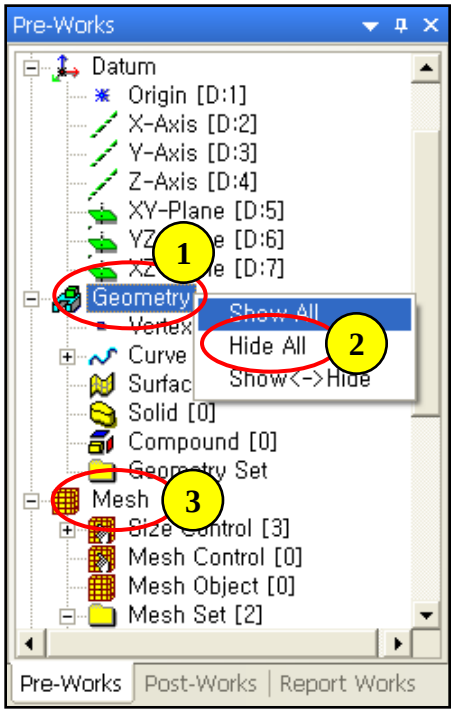


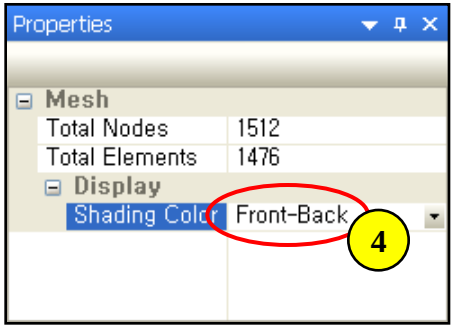
1. *Mesh > Protrude Mesh > Revolve…*
2. *Select "Edge->2D" tab*
3. *Select "Displayed"*
4. *Revolution Axis : Z Axis*
5. *Angle : 10 , Number of Times : 36*
6. *Property : 1*
7. *Mesh Set : Silo*
8. *Click* **[OK]** *Button*



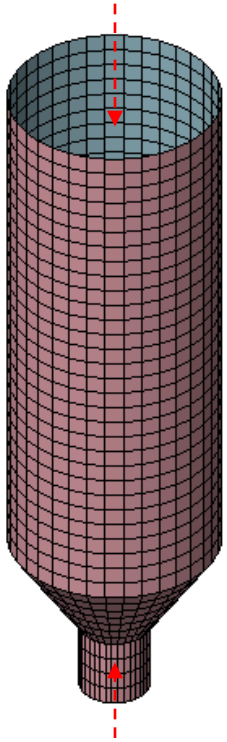
Step 5.

1. *Pre-Works Tree : Geometry ...*
2. *Click Right Mouse Button and Select "Hide All"*
3. *Pre-Works Tree : Mesh…*
4. *Property Window : Display - Shading Color (Front-Back)*
5. *Mesh > Element > Change Parameter…*
6. *Select "Reverse Normal" Option*
7. *Select "Displayed"*
8. *Click* **[OK]** *Button*

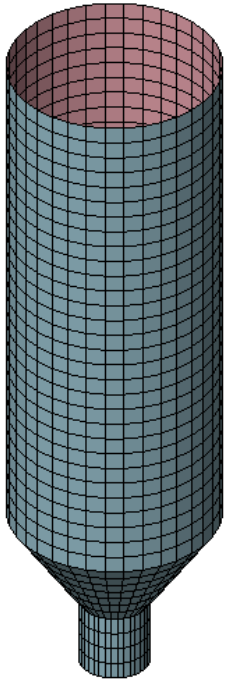


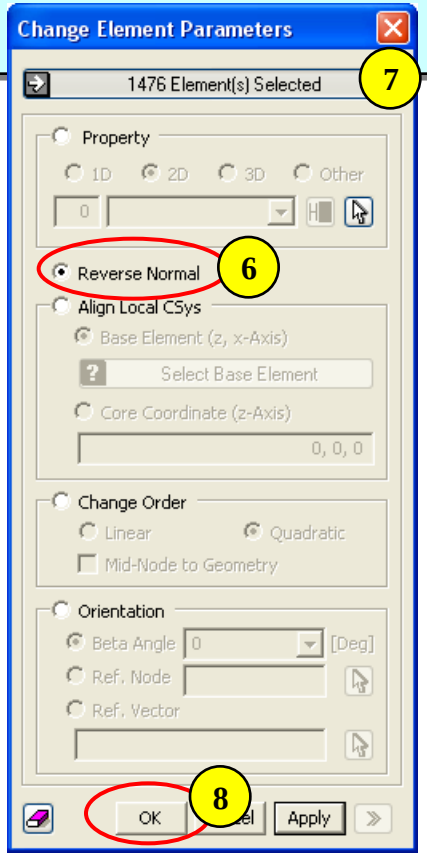


Blue Color : Front
(Positive Normal Side)

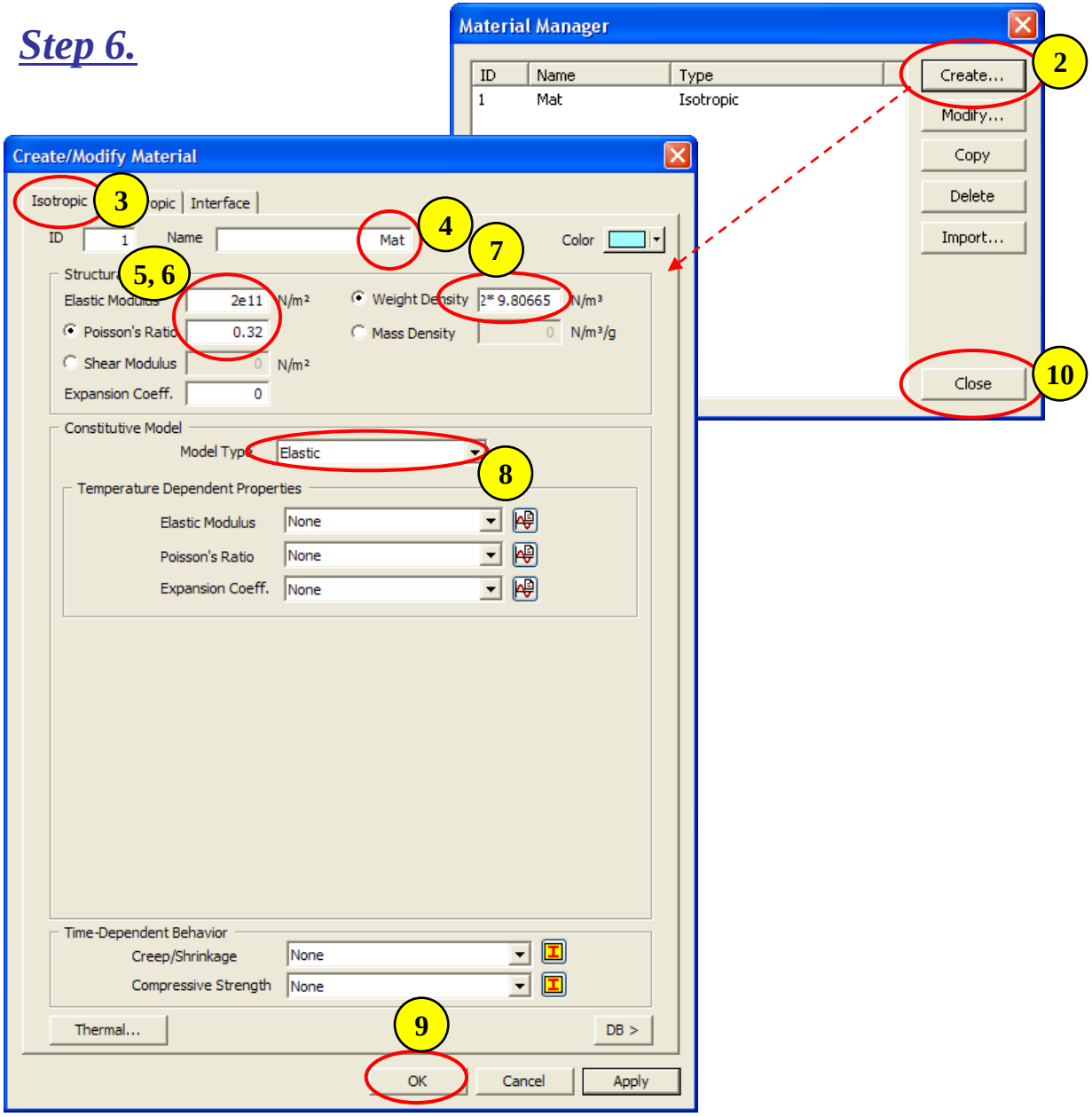


Red Color : Back
(Negative Normal Side)



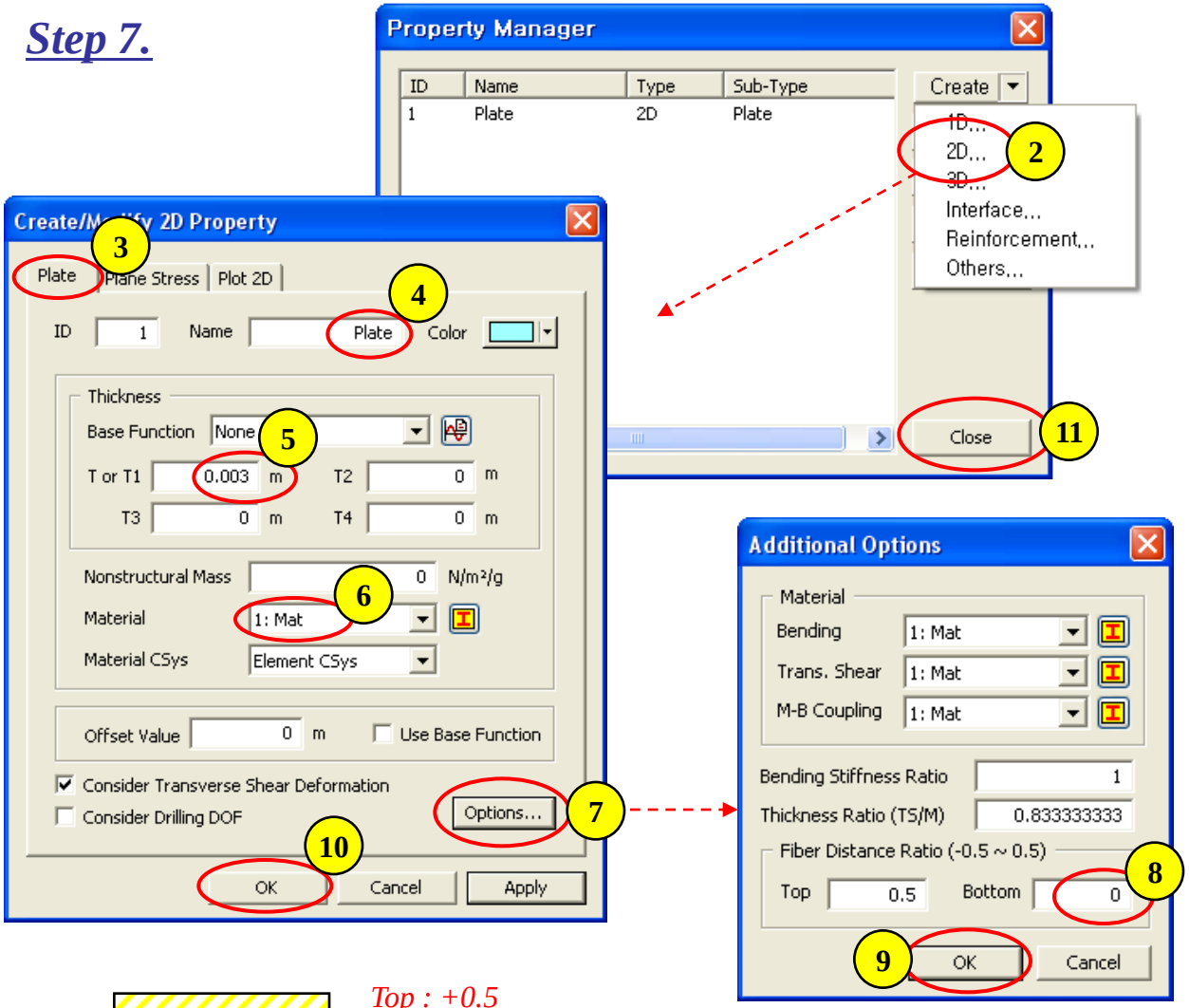


Step 6.

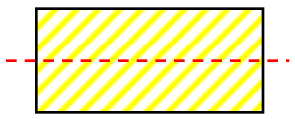


1. *Analysis > Material …*
2. *Click* **[Create...]** *Button*
3. *Select "Isotropic" tab*
4. *ID : 1 , Name : Mat*
5. *Elastic Modulus : 2e11 N/m²*
6. *Poisson's Ratio : 0.32*
7. *Weight Density :*
   802 9.80665 N/m³*
8. *Model Type : Elastic*
9. *Click* **[OK]** *Button*
10. *Click* **[Close]** *Button*

Step 7.



1. *Analysis > Property …*
2. *Create 2D…*
3. *Select "Plate" tab*
4. *ID : 1 , Name : Plate*
5. *T or T1 : 0.003*
6. *Material : 1: Mat*
7. *Click* **[Option…]** *Button*
8. *Fiber Distance*
   – Bottom : 0
9. *Click* **[OK]** *Button*
10. *Click* **[OK]** *Button*
11. *Click* **[Close]** *Button*

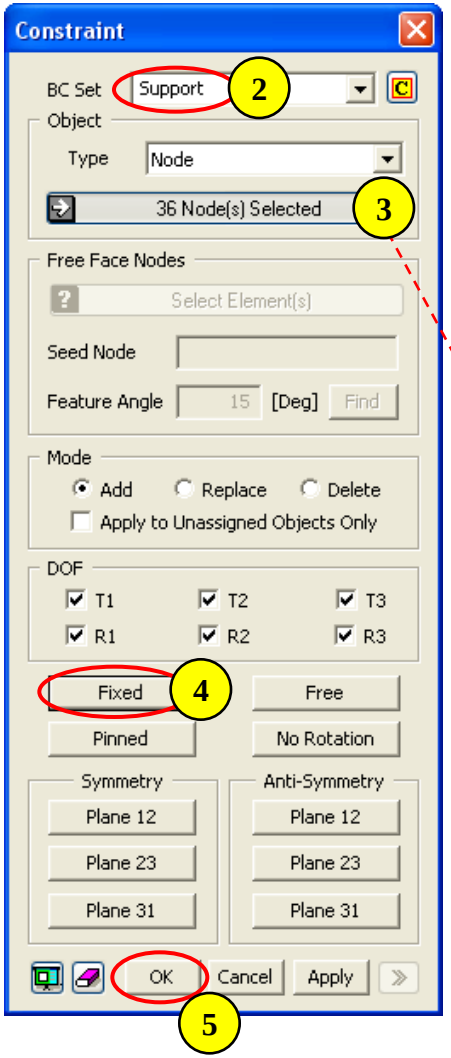


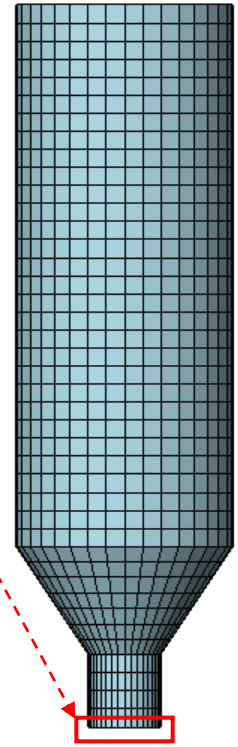
Top : +0.5
Middle : 0.0
Bottom : -0.5

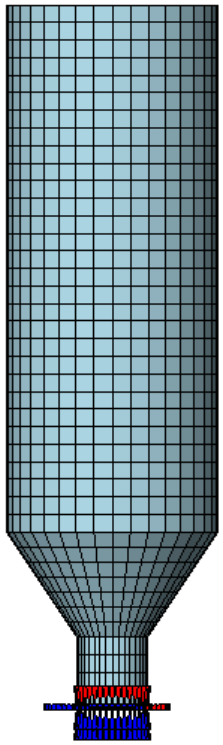
<Normalized Distance in Plate Section>

Fiber Distance : Stress Calculation Location (Normalized Distance)
Setting Bottom as '0' makes Bottom Stress to be Mid-Plane Stress.

Step 8.

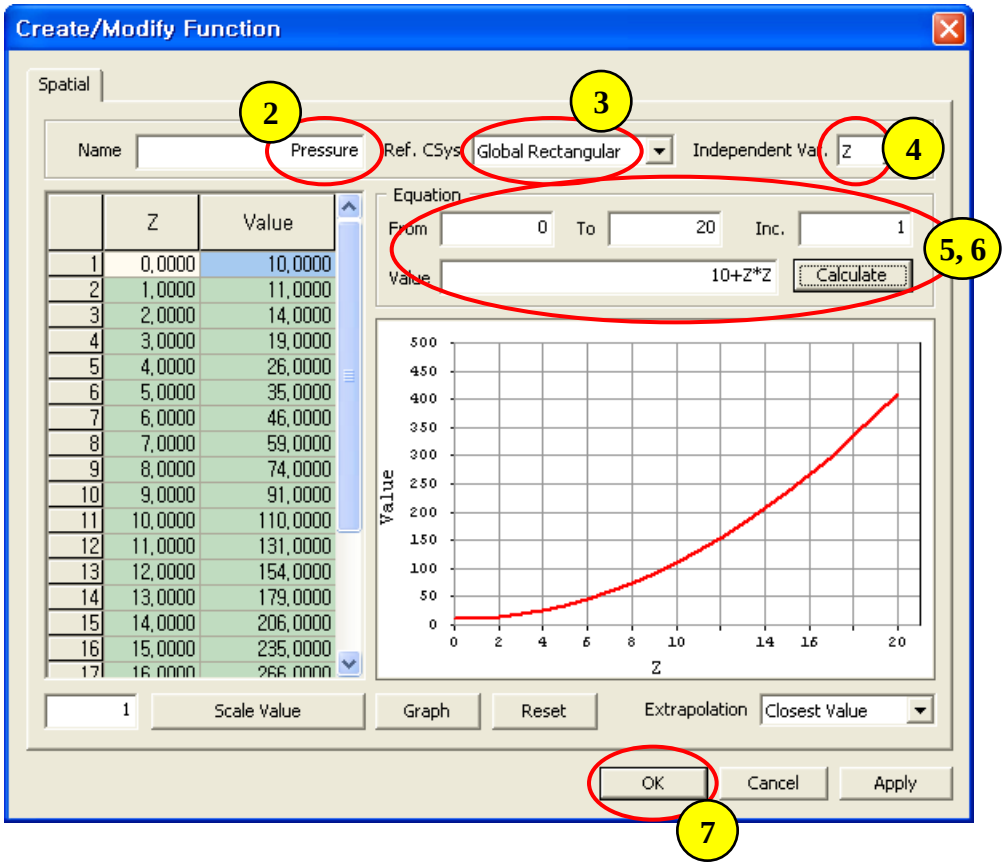




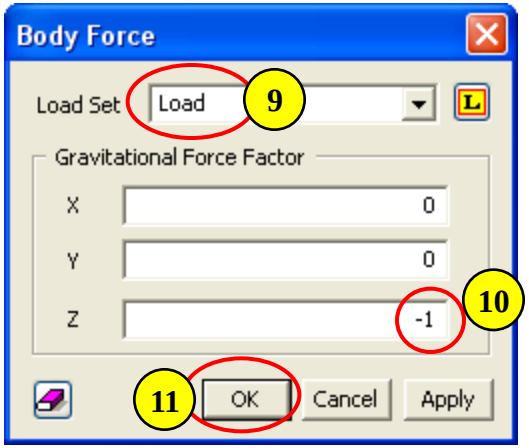


1. *Analysis > BC > Constraint…*
2. *BC Set : Support*
3. *Select 36 Bottom Nodes (See Figure)*
4. *Click* **[Fixed]** *Button*
5. *Click* **[OK]** *Button*

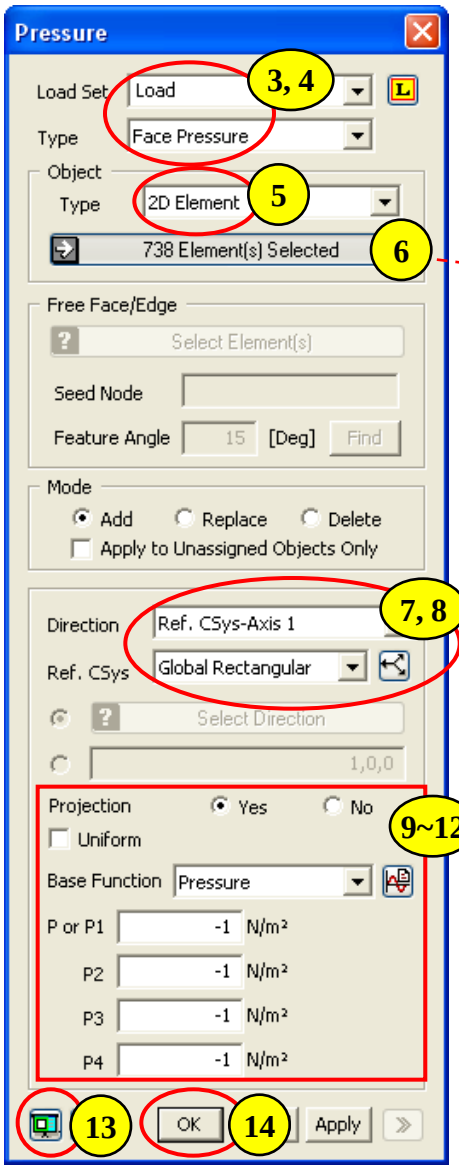
Step 9.

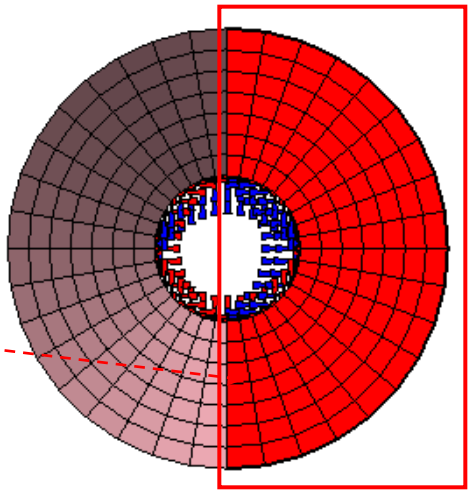


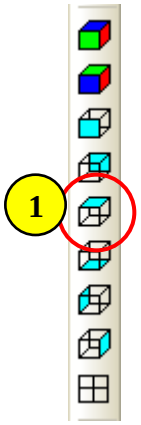
1. *Analysis > General Function…*
2. *Name : Pressure*
3. *Ref. CSys : Global Rectangular*
4. *Independent Var. : Z*
5. *From : 0 , To : 20 , Inc. : 1*
   *Value : 10+Z*Z*
6. *Click* **[Calculate]** *Button*
7. *Click* **[OK]** *Button*
8. *Analysis > Load > Body Force…*
9. *Load Set : Load*
10. *Gravitational Force Factor : -1*
11. *Click* **[OK]** *Button*

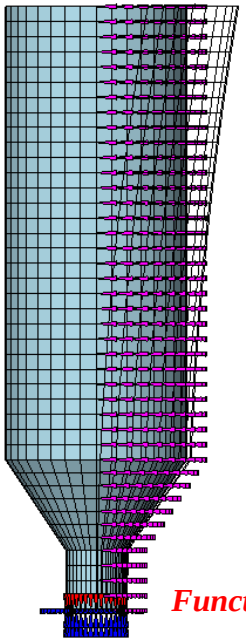


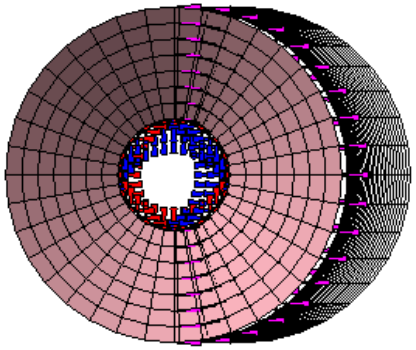
Step 10.











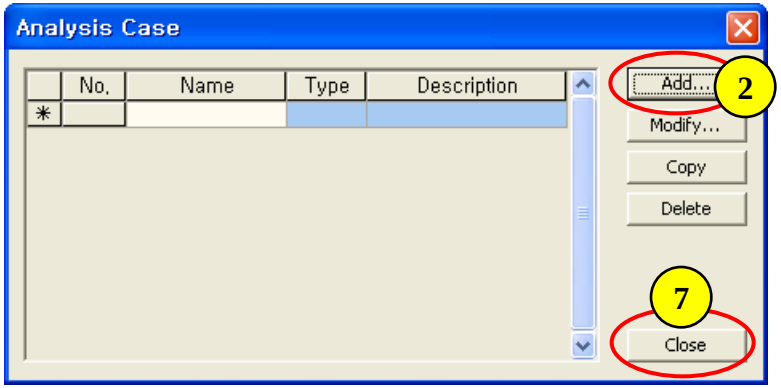
Projected Pressure

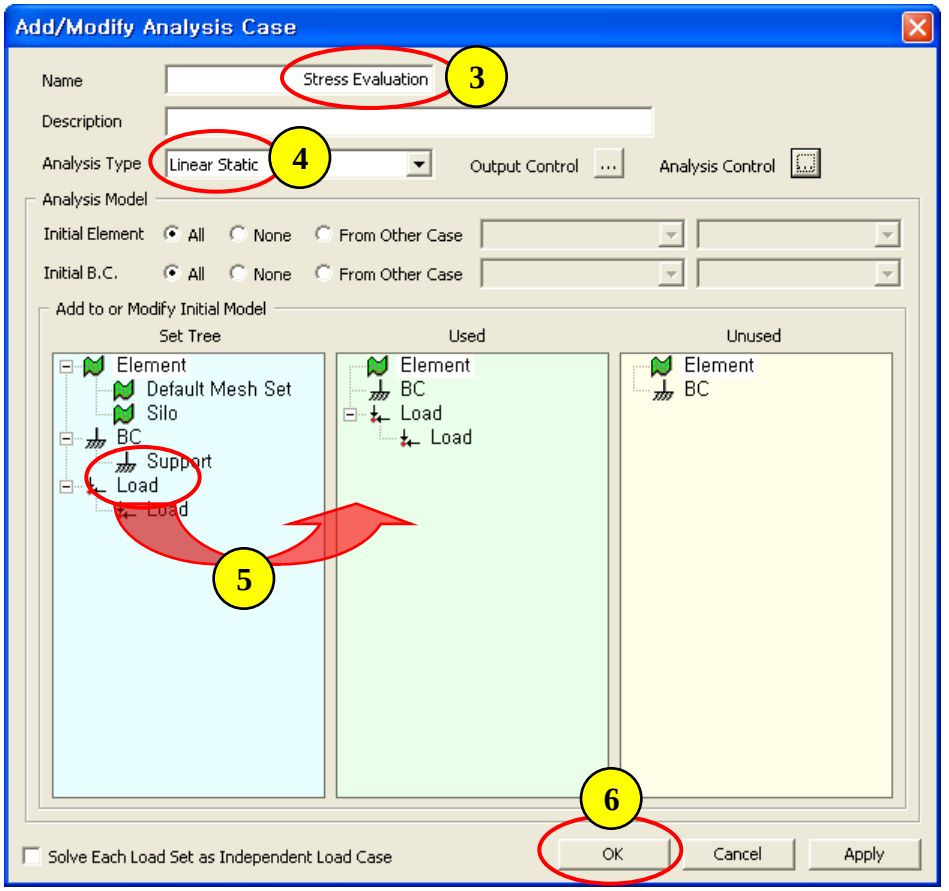
Function-Dependent Pressure

1. *Click "Top View"*
2. *Analysis > Load > Pressure…*
3. *Load Set : Load*
4. *Type : Face Pressure*
5. *Object Type : 2D Element*
6. *Select 738 Elements on the right side of the plan*
7. *Direction : Ref. CSys-Axis 1*
8. *Ref. CSys : Global Rectangular*
9. *Projection : Yes*
10. *Check off "Uniform"*
11. *Base Function : Pressure*
12. *P1~P4 : -1*
13. *Click "Preview" Button*
14. *Click* **[OK]** *Button*

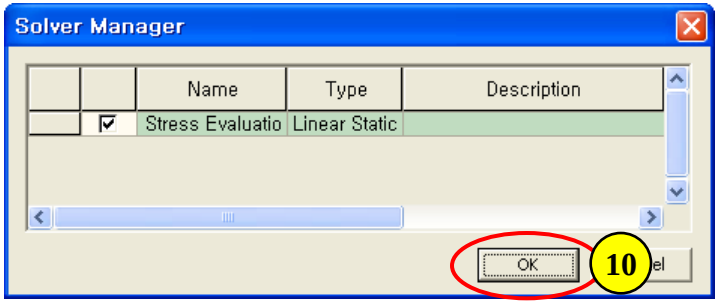
Ref. CSys : Global Rectangular
→ Ref. CSys Axis 1 : Global X Axis

Step 11.



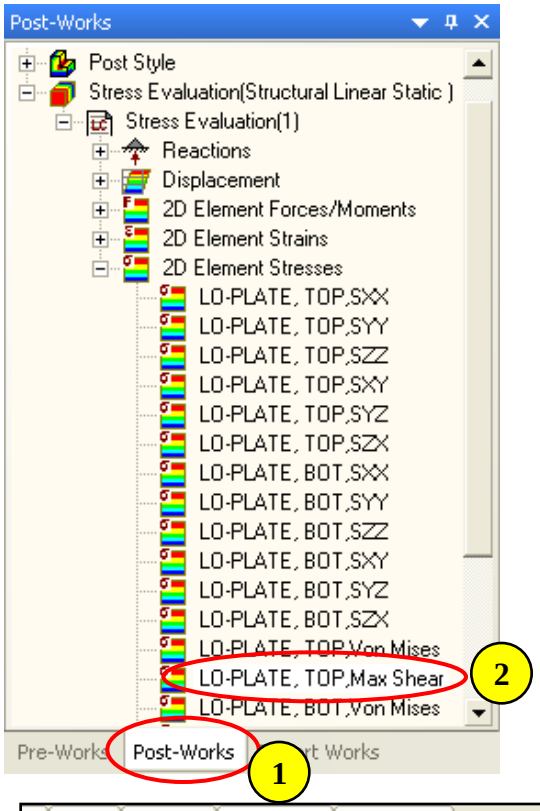


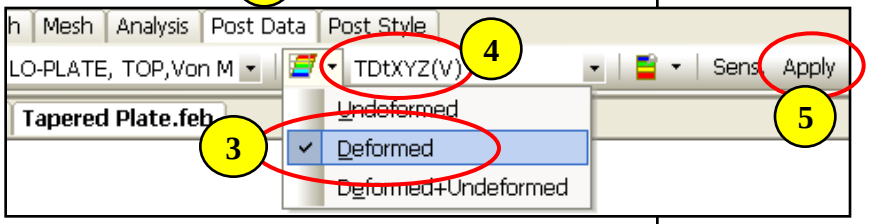
1. *Analysis > Analysis Case …*
2. *Click* [Add] *Button*
3. *Name : Stress Evaluation*
4. *Analysis Type : Linear Static*
5. *Drag & Drop "Load" to "Used" Window*
6. *Click* [OK] *Button*
7. *Click* [Close] *Button*
8. *File > Save … (Silo.feb)*
9. *Analysis > Solve …*
10. *Click* [OK] *Button*

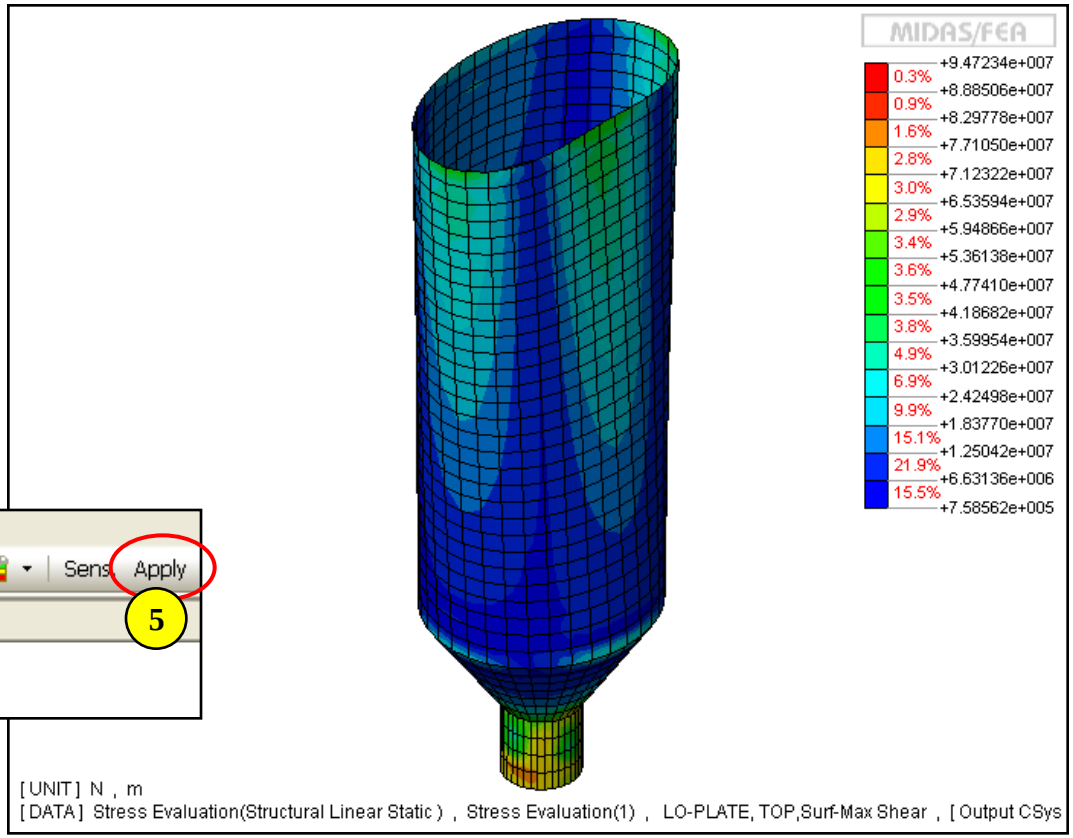


Step 12.

1. *Post-Works Tree : Stress Evaluation(Structural Linear Static) > Stress Evaluation (1) > 2D Element Stresses*
2. *Double Click "LO-PLATE, TOP, Max Shear"*
3. *Select "Deformed" for Mesh Shape at "Post Data" Toolbar*
4. *Select "TDtXYZ(V)" for Deformation Data*
5. *Click "Apply" Button*

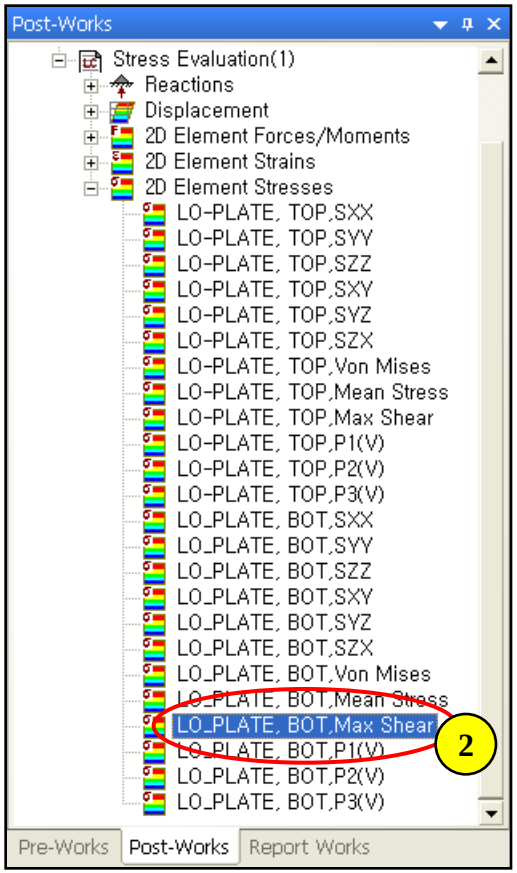


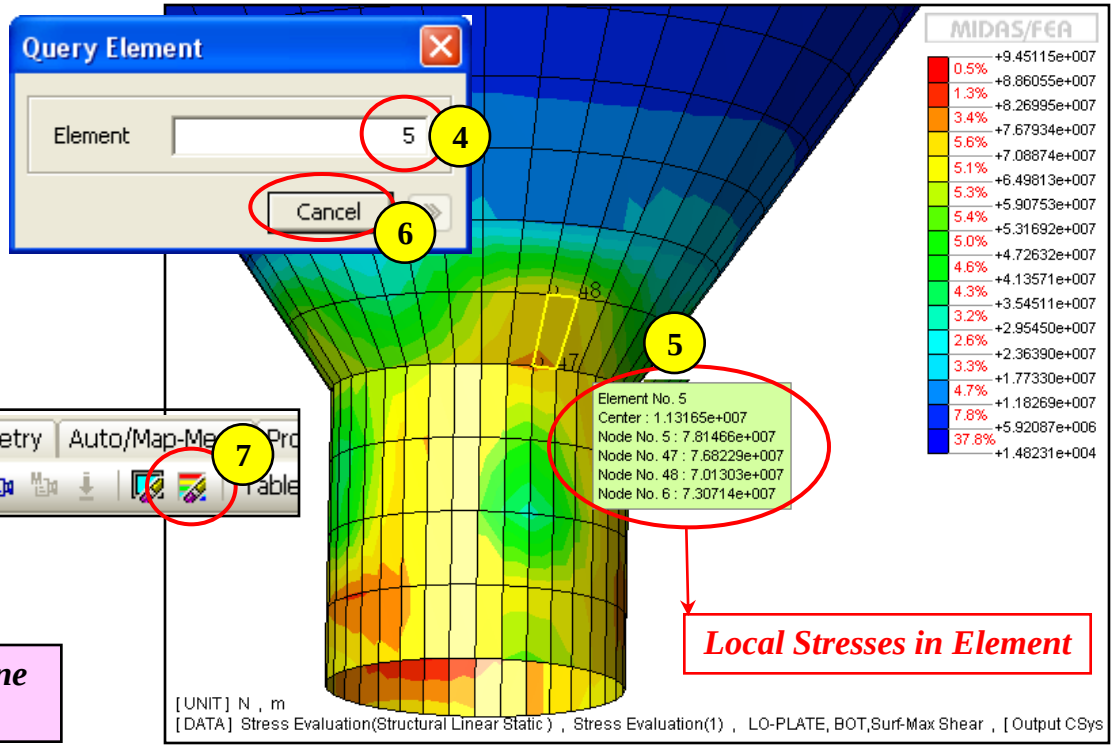




Step 13.

1. *Post-Works Tree : Stress Evaluation(Structural Linear Static) > Stress Evaluation (1) > 2D Element Stresses*
2. *Double Click "LO-PLATE, BOT, Max Shear"*
3. *Mesh > Element > Query…*
4. *Enter "5" and Press* [Enter] *Key*
5. *Move Mouse Cursor on Element 5*
6. *Click* [Cancel] *Button*
7. *Click "Initial Post Style" at "Post Style" Toolbar*





Max. Shear Stress at Element Mid-Plane (Fiber Distance : 0)