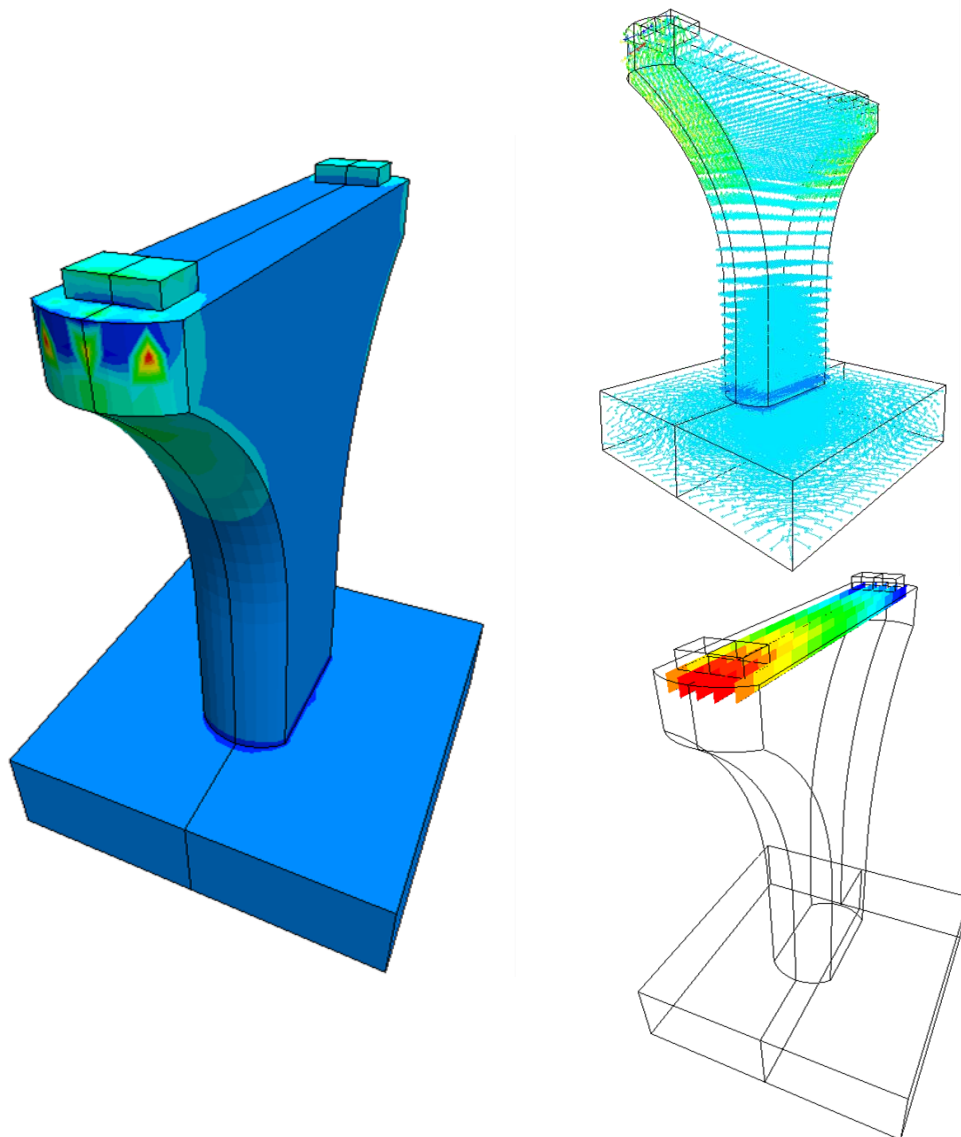


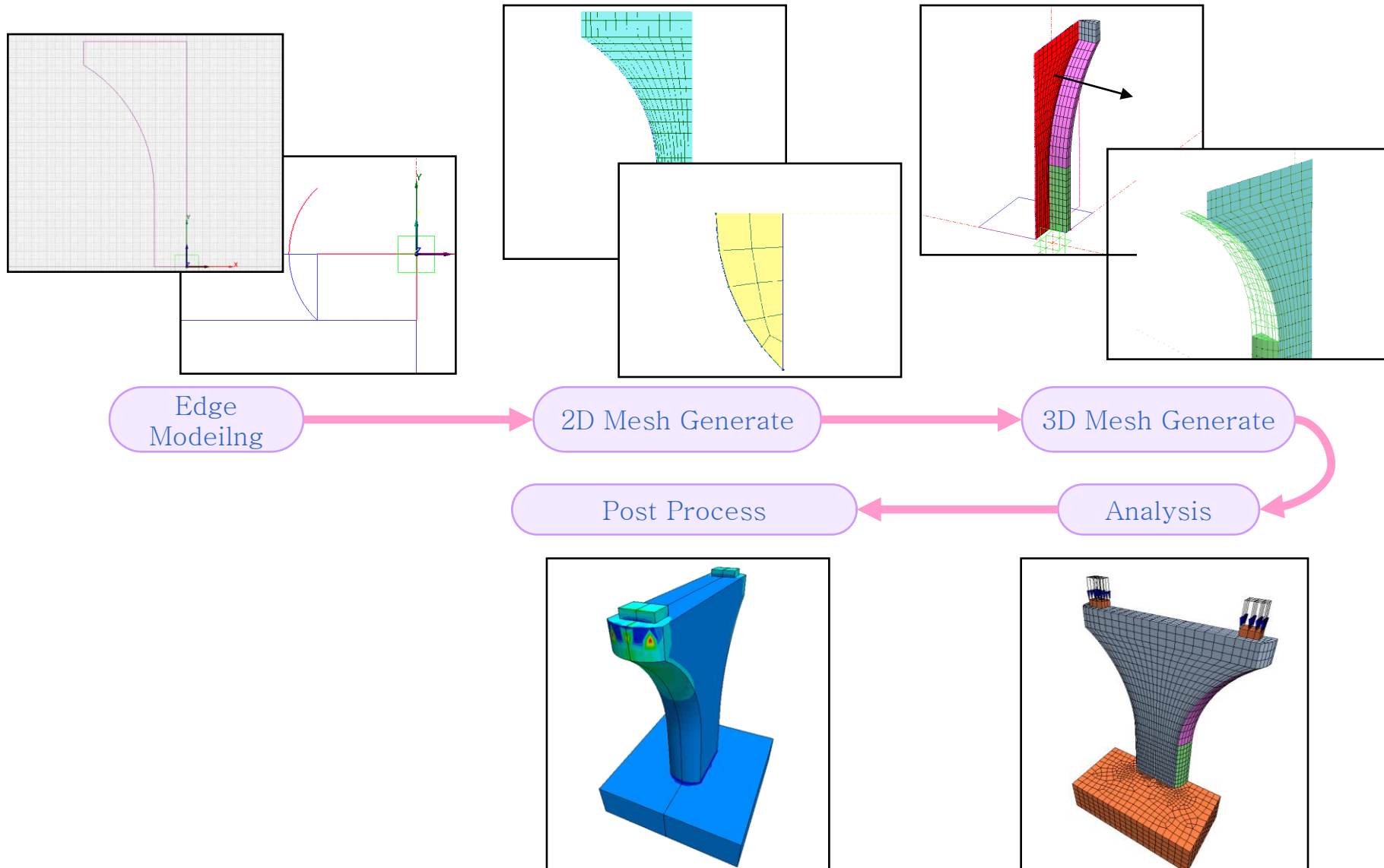
LS-17. Pier with Tendon



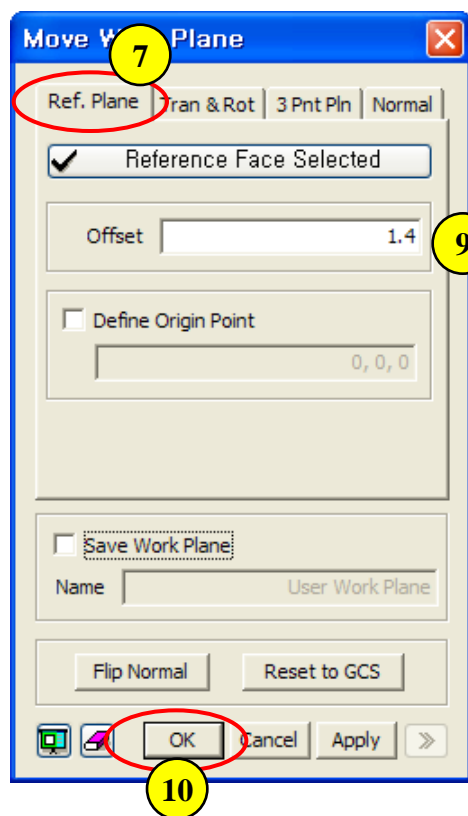
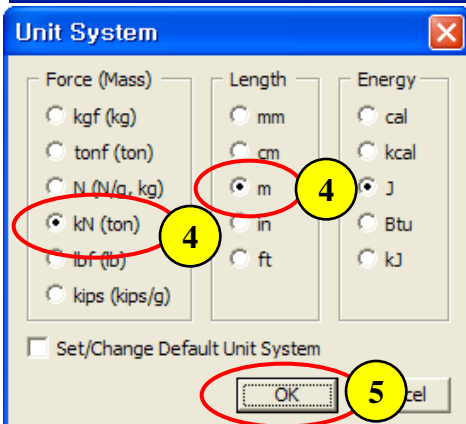
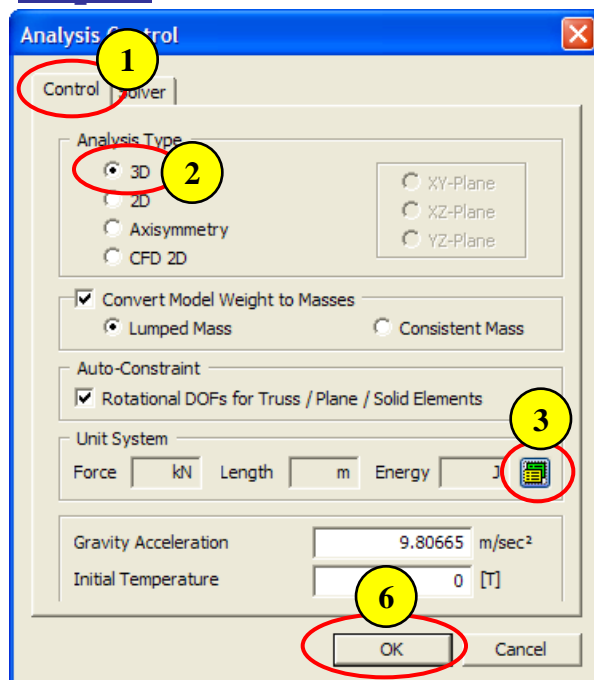
Overview

- 3-D Linear Static Analysis
- Model
 - $\frac{1}{2}$ Symmetric Model
 - Unit : kN, m
 - Isotropic Elastic Material
 - Reinforcement Element
 - Solid Element
- Load & Boundary Condition
 - Body Force
 - Pressure
 - Prestress
 - Symmetric Condition
- Result Evaluation
 - Reinforcement Stress
 - Principal Stress Vector Plot

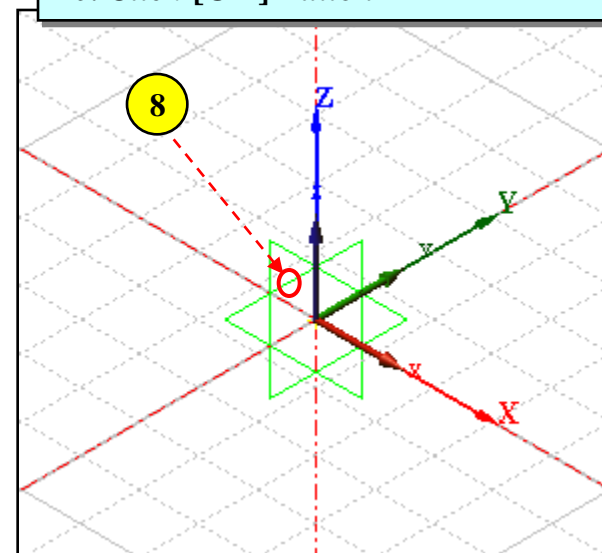
Modeling Flow



Step 1.

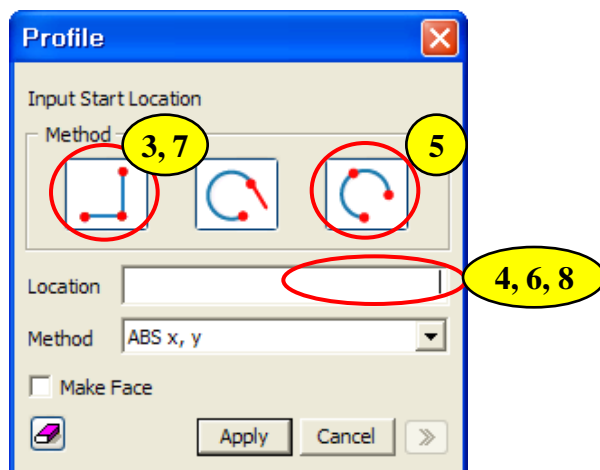
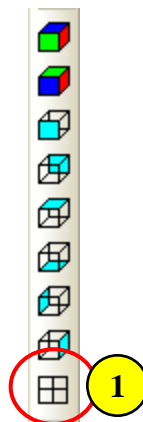


1. Analysis > Analysis Control
– “Control” tab
2. Analysis Type : 3D
3. Click Button
4. Unit : kN , m
5. Click [OK] Button
6. Click [OK] Button
7. Geometry > Work Plane > Move
– “Ref. Plane” tab
8. Select “XZ Plane”
9. Offset: 1.4
10. Click [OK] Button

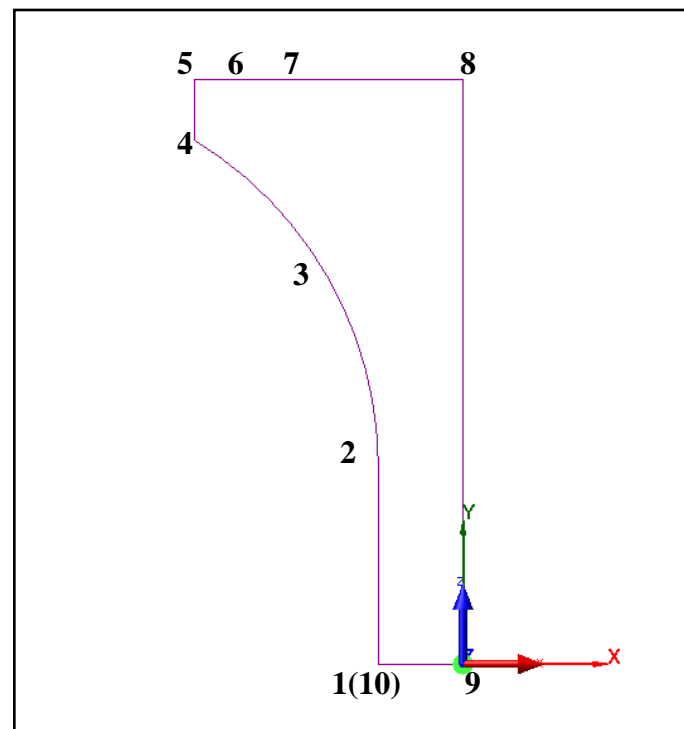


Analysis Control Dialog is automatically activated at startup.

Step 2.

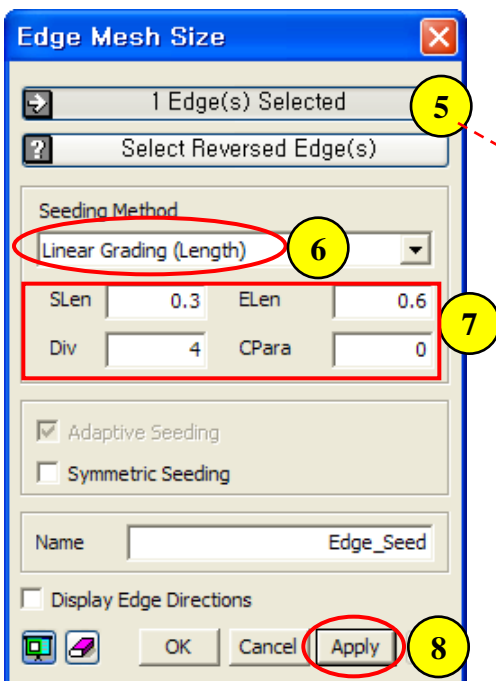
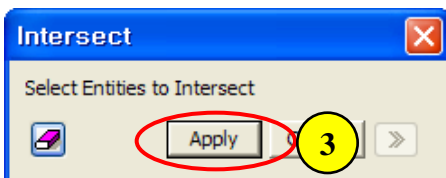


1. Click "Normal View"
2. Geometry > Curve > Create on WP > Profile (Wire)
3. Method : Line
4. Location : $(-2.1, 0), <0, 5.0>$
5. Method : 3 Point Arc
6. Location : $<-1.242, 4.588>, <-3.308, 3.412>$
7. Method : Line
8. Location : $<0, 1.5>, <0.25>, <1.8>, <4.6>, <0, -14.5>, <-2.1>$



$\diamond ()$: "ABS x, y", $<>$: "REL dx, dy"

Step 3.



1. Geometry > Curve > Intersect...

2. Toggle on "Displayed"

3. Click [Apply] Button

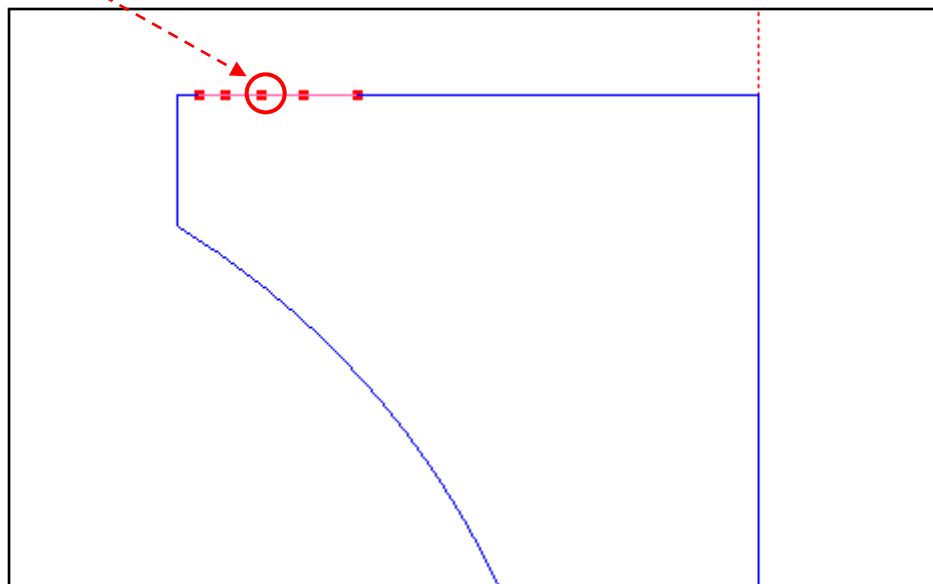
4. Mesh > Size Control > Along Edge...

5. Select 1 Edge marked by "O" (See Figure)

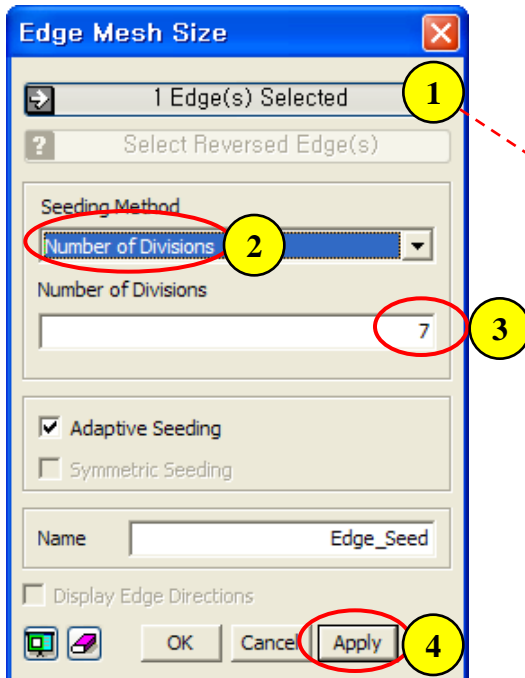
6. Seeding Method : Linear Grading (Length)

7. Slen : 0.3, Elen : 0.6, Div : 4

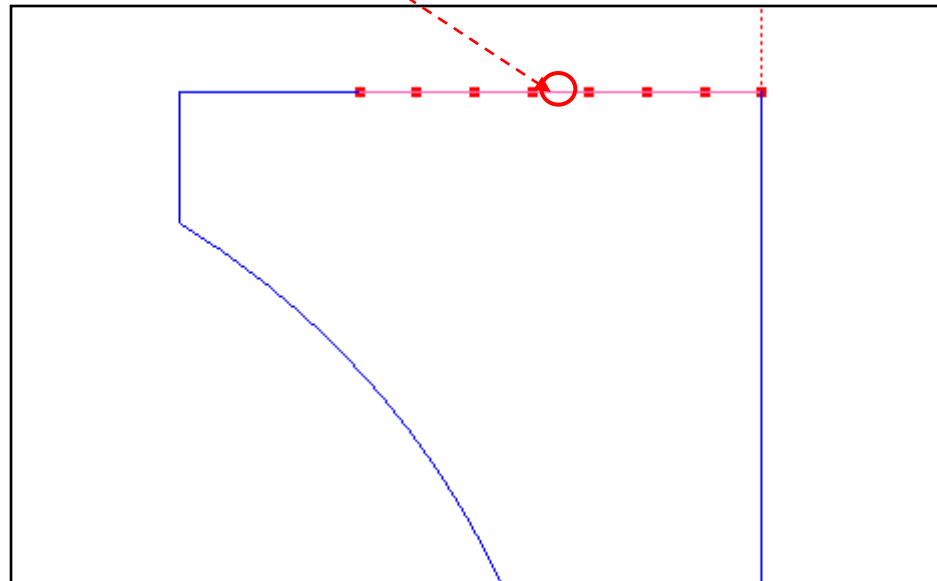
8. Click [Apply] Button



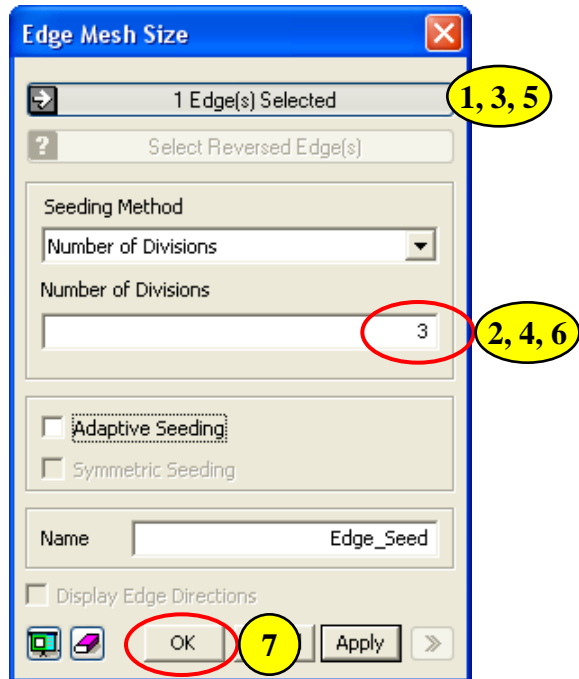
Step 4.



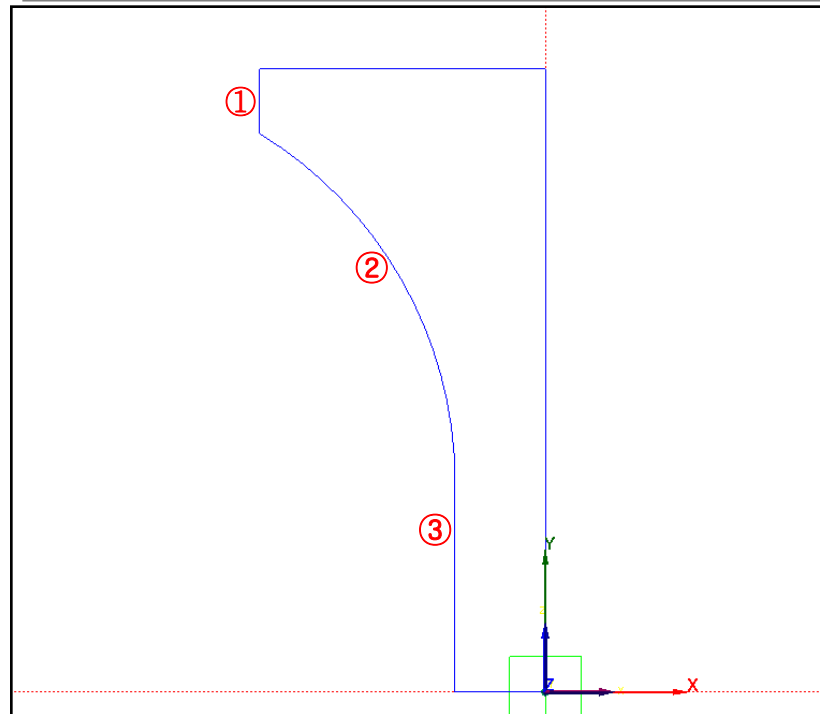
1. Select 1 Edge marked by “○” (See Figure)
2. Seeding Method : Number of Divisions
3. Number of Divisions : 7
4. Click [Apply] Button



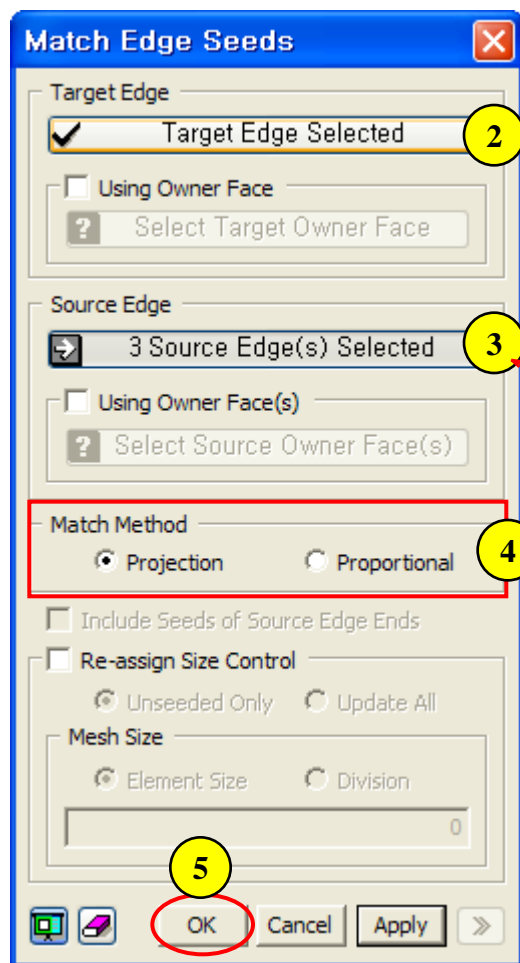
Step 5.



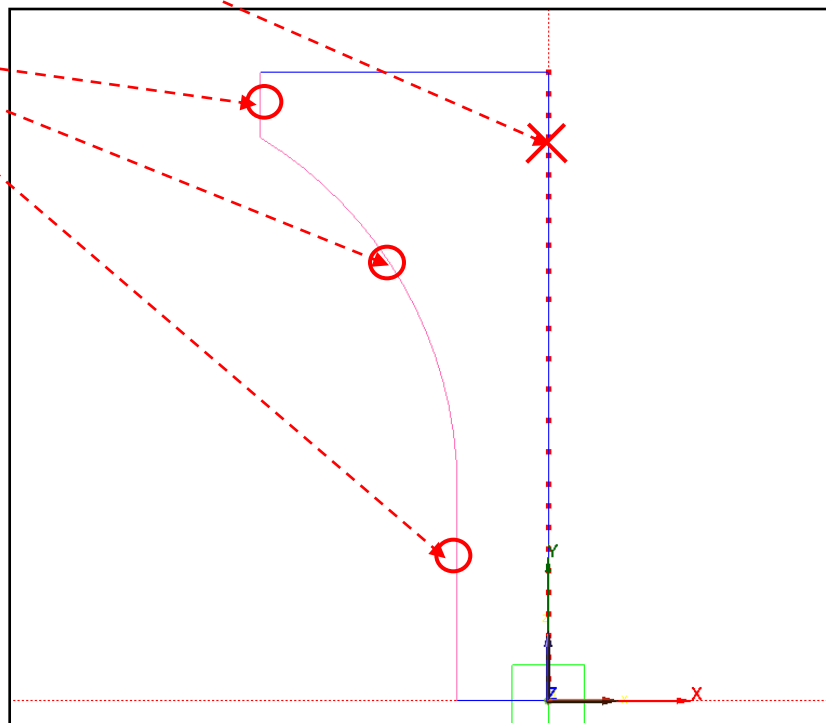
1. Select 1 Edge marked by “①” (See Figure)
2. Number of Divisions : 3
3. Select 1 Edge marked by “②” (See Figure)
4. Number of Divisions : 13
5. Select 1 Edge marked by “③” (See Figure)
6. Number of Divisions : 10
7. Click [OK] Button



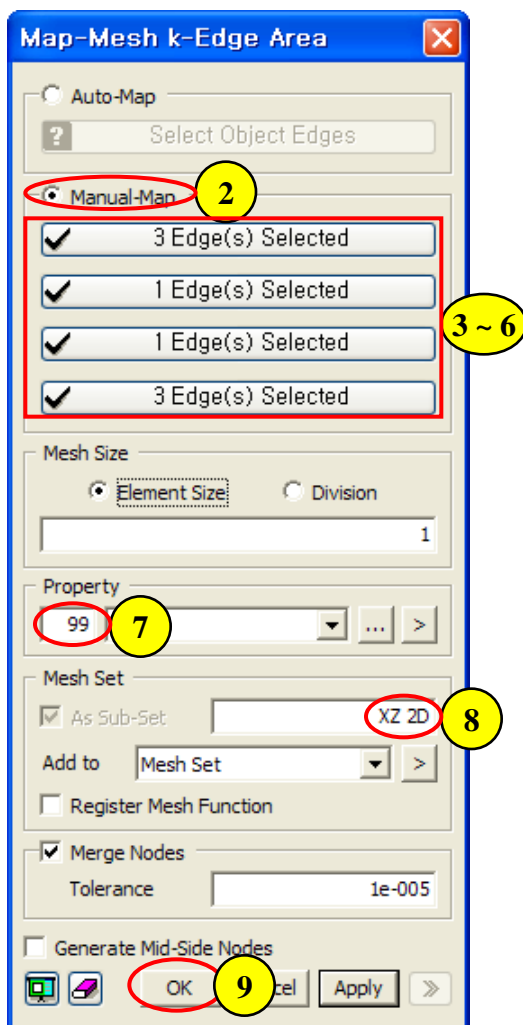
Step 6.



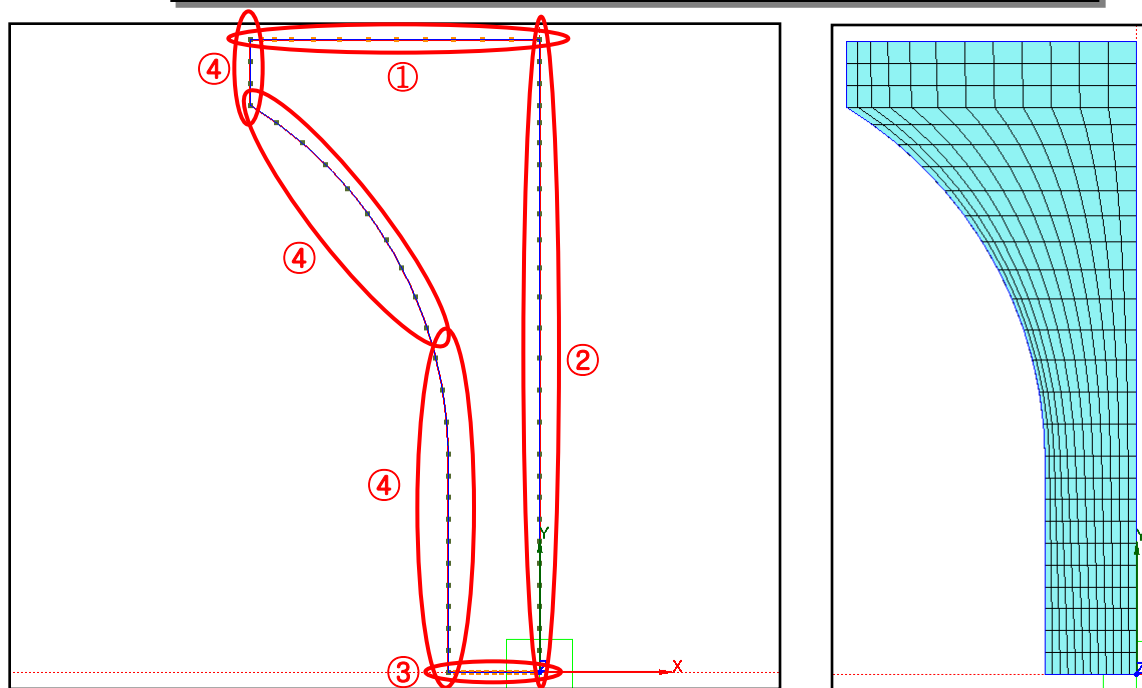
1. Mesh > Size Control > Match Edge Seeds...
2. Select 1 Target Edge marked by "X" (See Figure)
3. Select 3 Source Edge marked by "O" (See Figure)
4. Match Method : Projection
5. Click [OK] Button



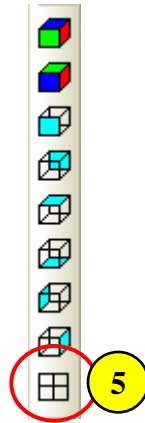
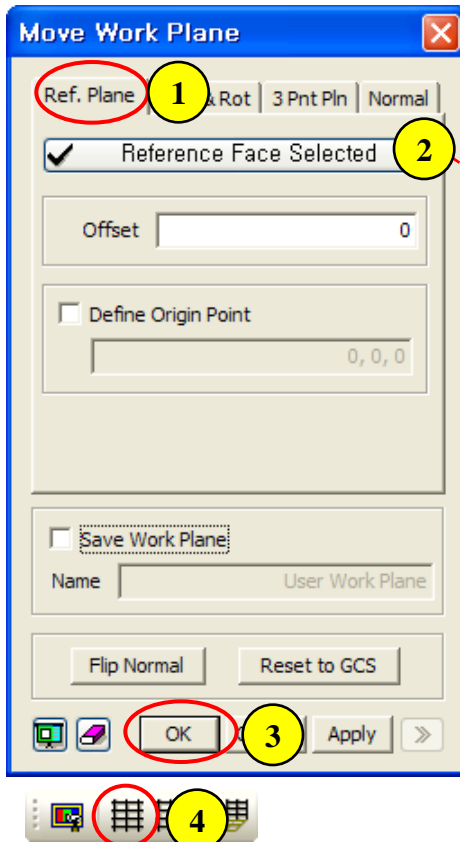
Step 7.



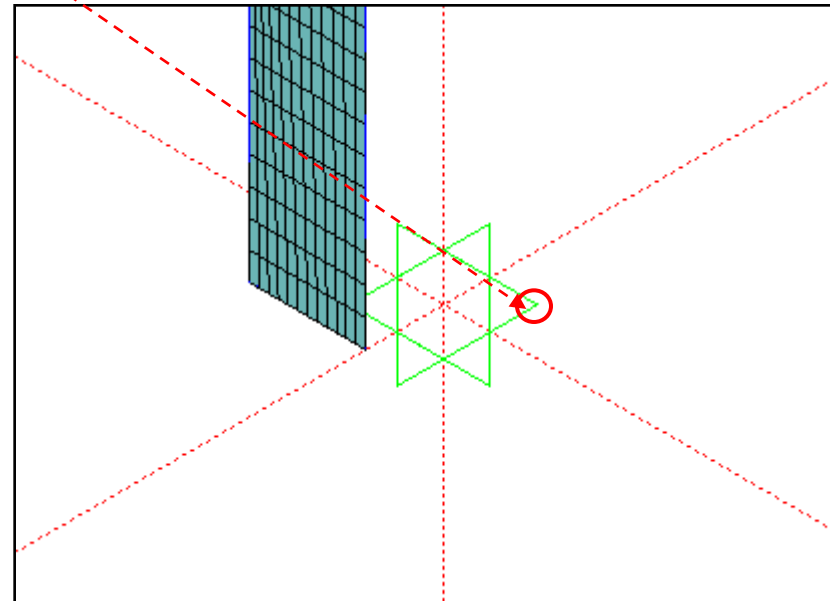
1. Mesh > Map Mesh > K-Edge Area...
2. Check on : Manual-Map
3. Select 3 Edges marked by “①” (See Figure)
4. Select 1 Edge marked by “②” (See Figure)
5. Select 1 Edge marked by “③” (See Figure)
6. Select 3 Edges marked by “④” (See Figure)
7. Property : 99
8. Mesh Set : “XZ 2D”
9. Click [OK] Button



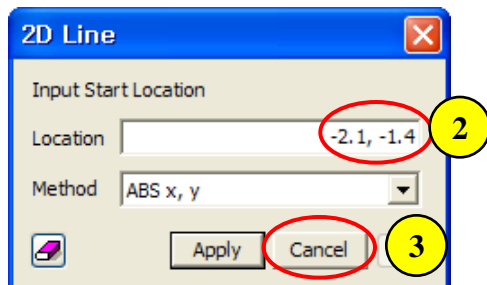
Step 8.



1. Geometry > Work Plane > Move – “Ref. Plane” tab
2. Select “XY Plane”
3. Click [OK] Button
4. Toggle on “Grid”
5. Toggle on “Normal View”



Step 9.



1. *Geometry > Curve > Create on WP > Line...*

2. *Location : (-2.1, -1.4), <0, 1.4>*

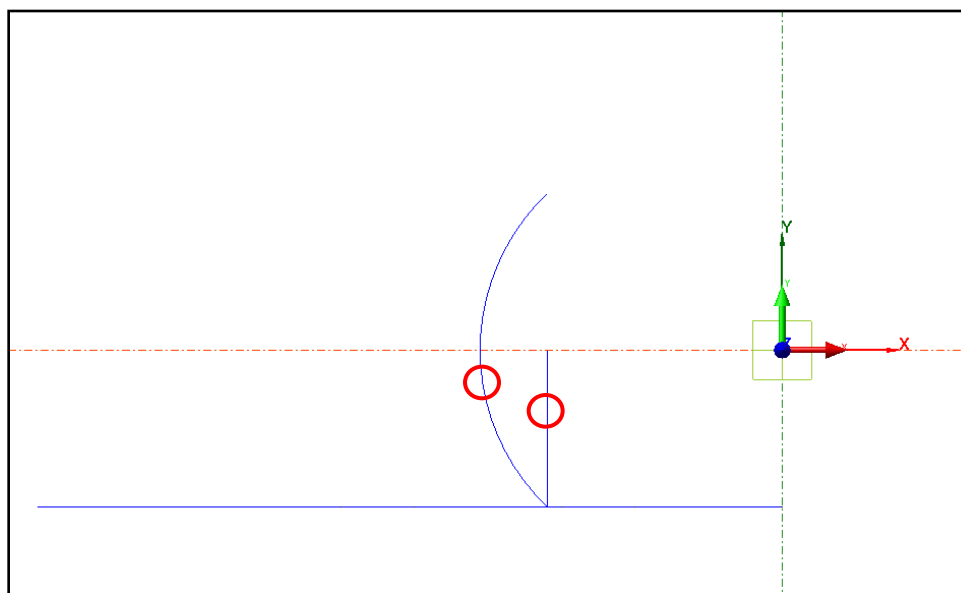
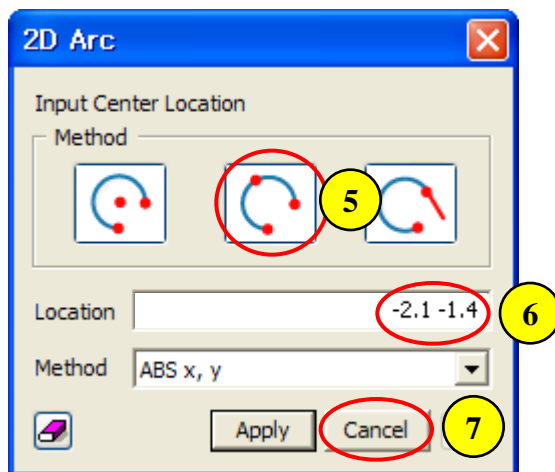
3. *Click [Cancel] Button*

4. *Geometry > Curve > Create on WP > Arc...*

5. *Select 3 Point Arc*

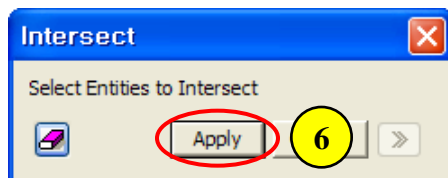
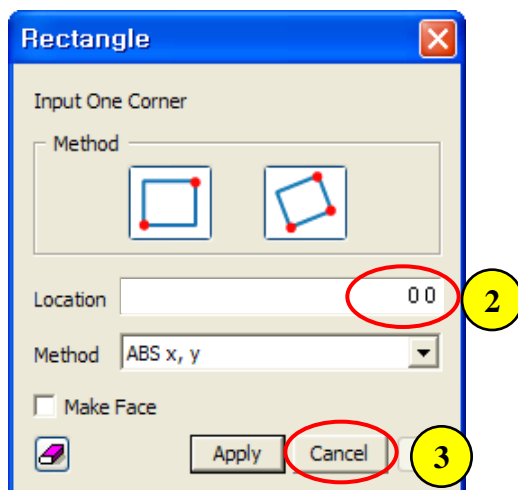
6. *Location : (-2.1, -1.4), (-2.7, 0), (-2.1, 1.4)*

7. *Click [Cancel] Button*

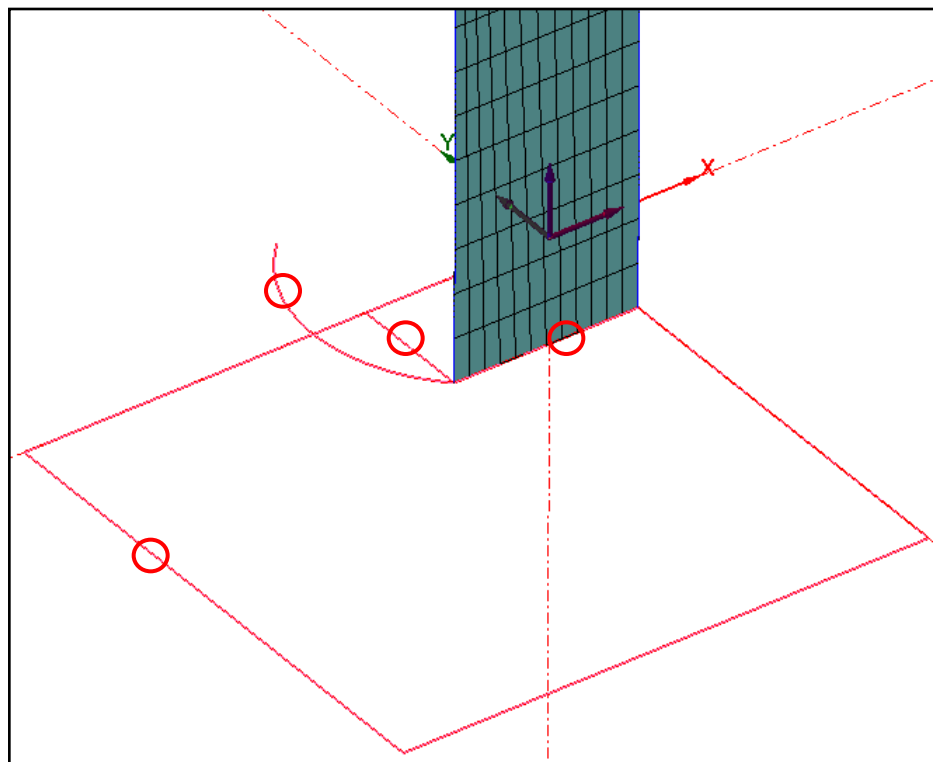


⓪ : "ABS x, y", <> : "REL dx, dy"

Step 10.

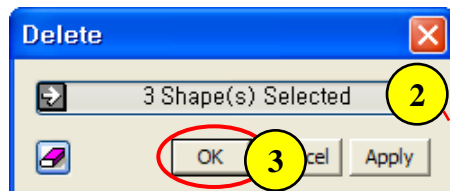


1. *Geometry > Curve > Create on WP > Rectangle (Wire)*
2. *Location : (0, 0), <-6.0, -6.0>*
3. *Click [Cancel] Button*
4. *Geometry > Curve > Intersect...*
5. *Select 3 Edges and 1 Wire marked by “○” (See Figure)*
6. *Click [Apply] Button*

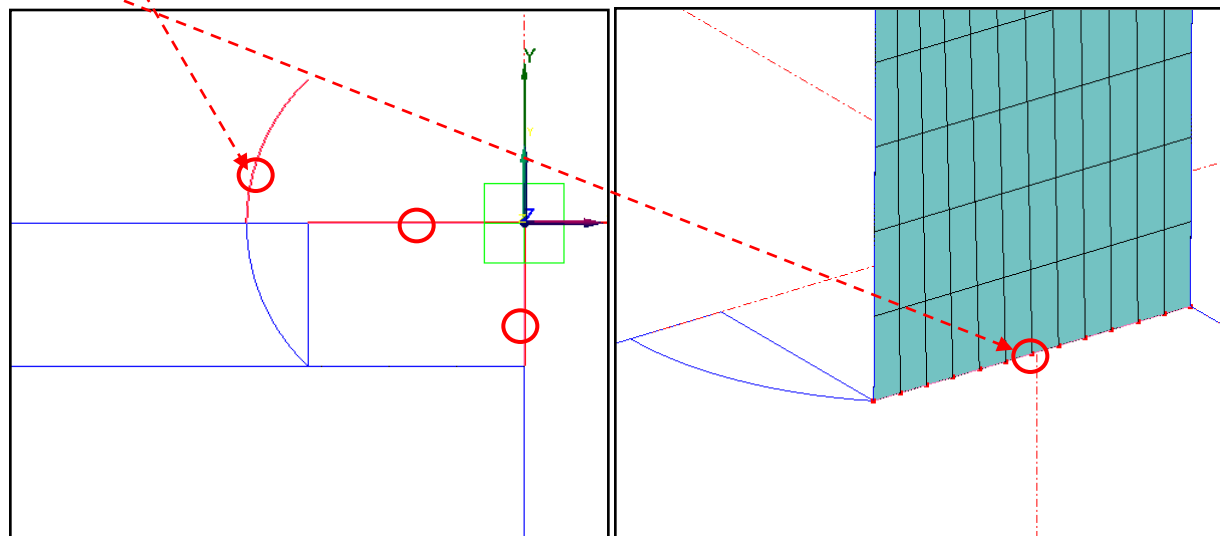
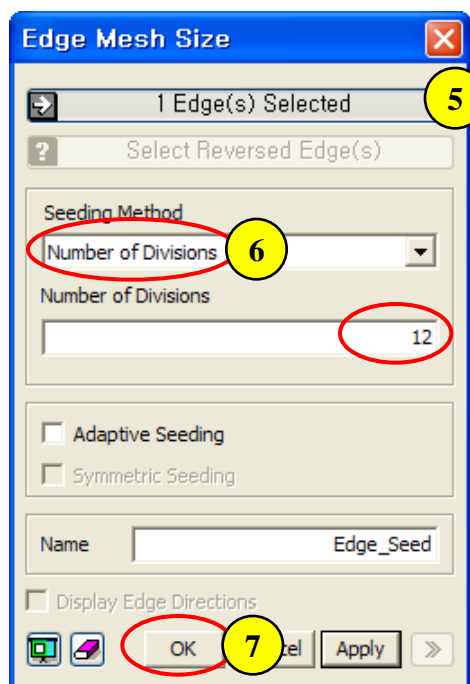


○ : “ABS x, y”, ◇ : “REL dx, dy”

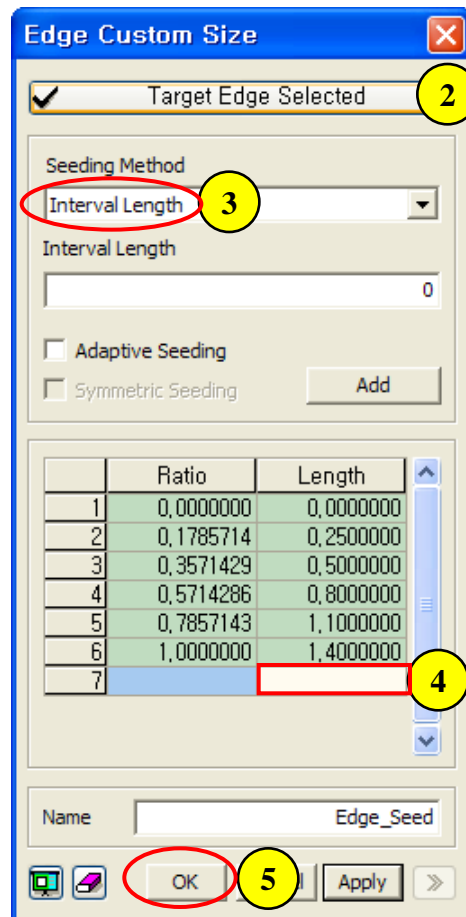
Step 11.



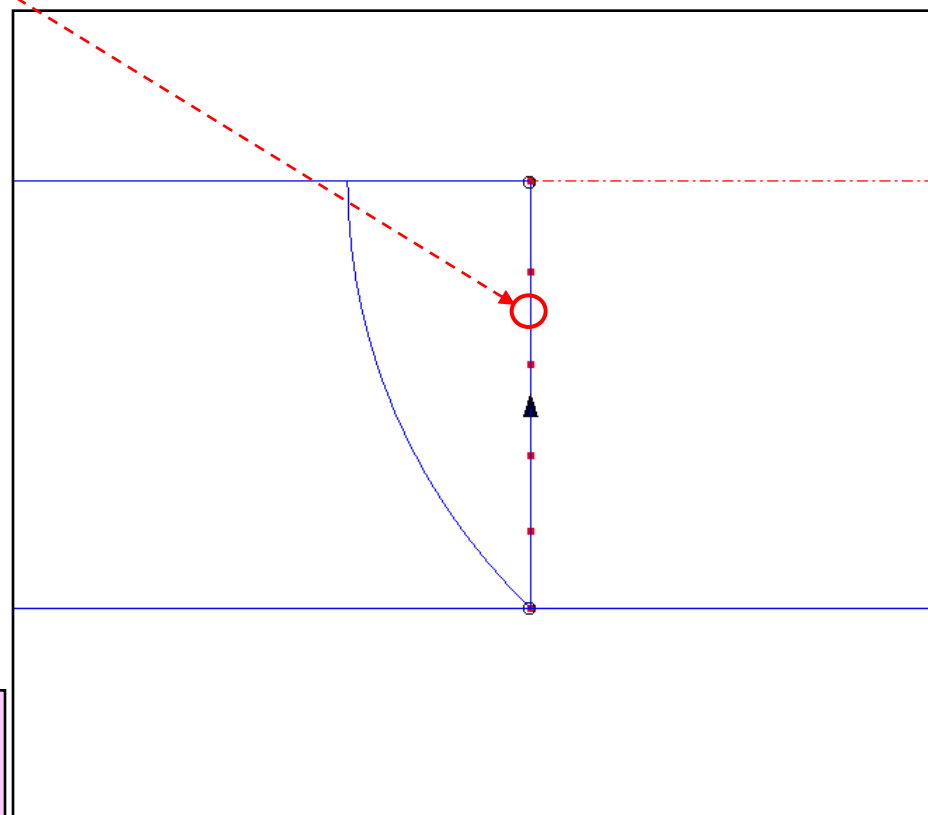
1. Geometry > Delete...
2. Select 3Edges marked by "O" (See Figure)
3. Click [OK] Button
4. Mesh > Size Control > Along Edge...
5. Select 1Edge marked by "O" (See Figure)
6. Number of Divisions : 12
7. Click [OK] Button



Step 12.

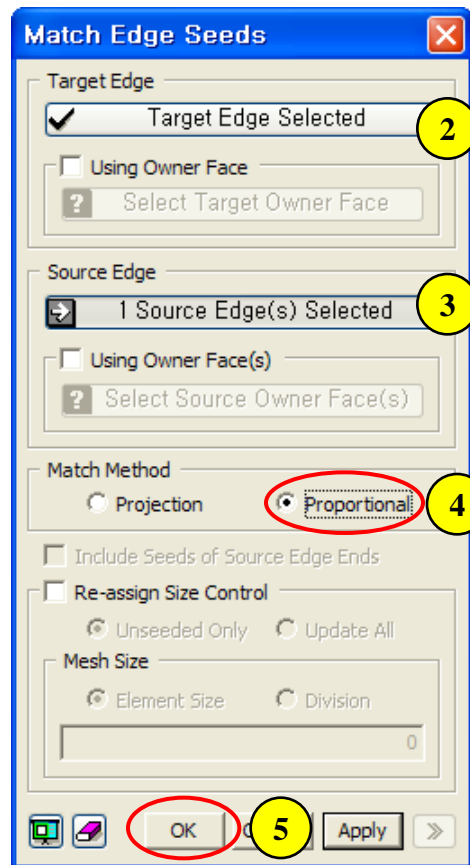


1. Mesh > Size Control > Custom Size Along Edge...
2. Select 1 Target Edge marked by "O" (See Figure)
3. Seeding Method : Interval Length
4. Length : 0.25, 0.5, 0.8, 1.1[Ⓢ]
5. Click [OK] Button

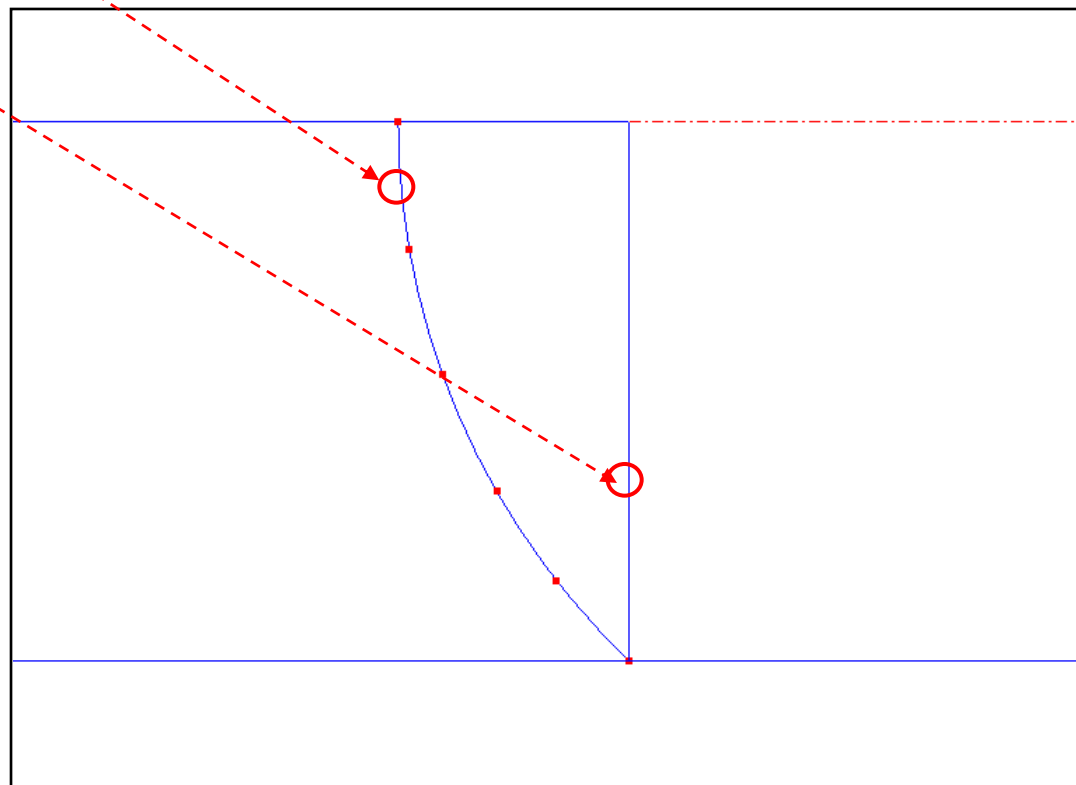


[Ⓢ] The inserted values for length will be automatically sorted in the Length column and the corresponding Ratio will be calculated.

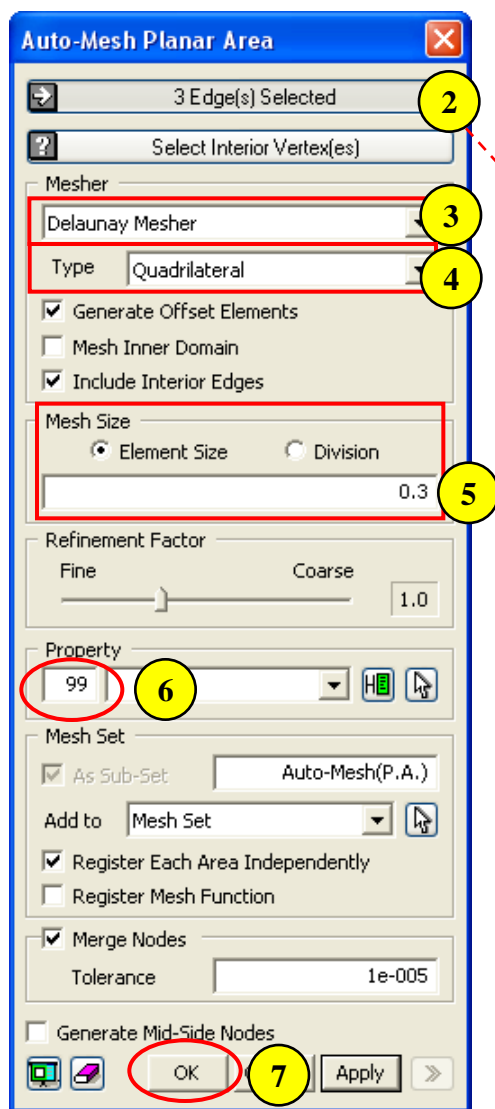
Step 13.



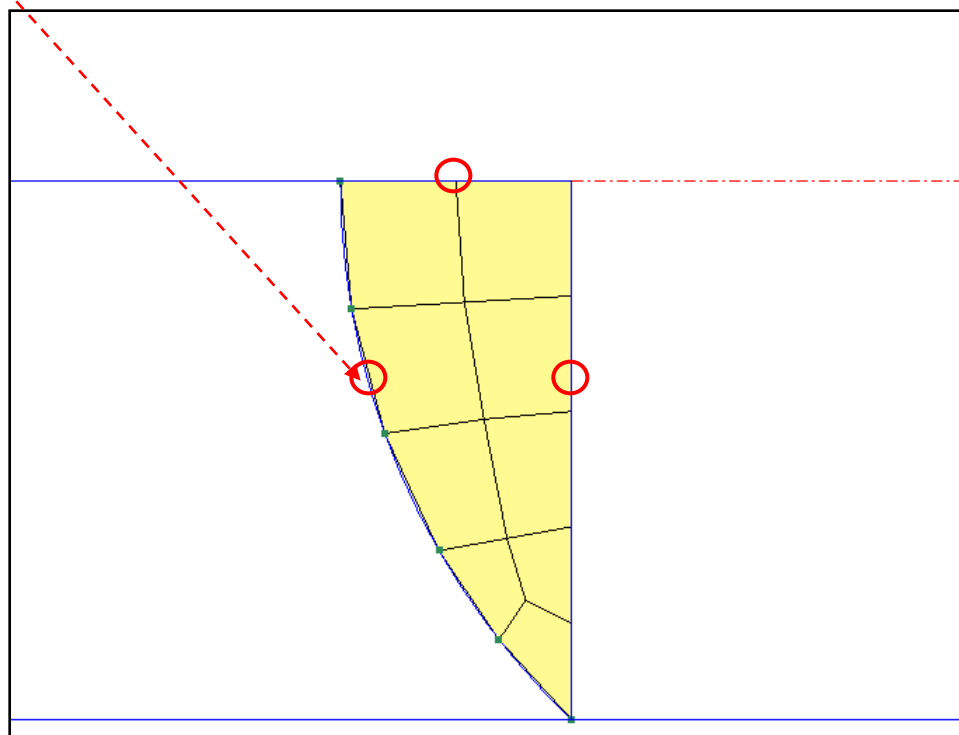
1. Mesh > Size Control > Match Edge Seeds...
2. Select 1 Target Edge marked by "O" (See Figure)
3. Select 1 Source Edge marked by "O" (See Figure)
4. Match Method : Proportional
5. Click [OK] Button



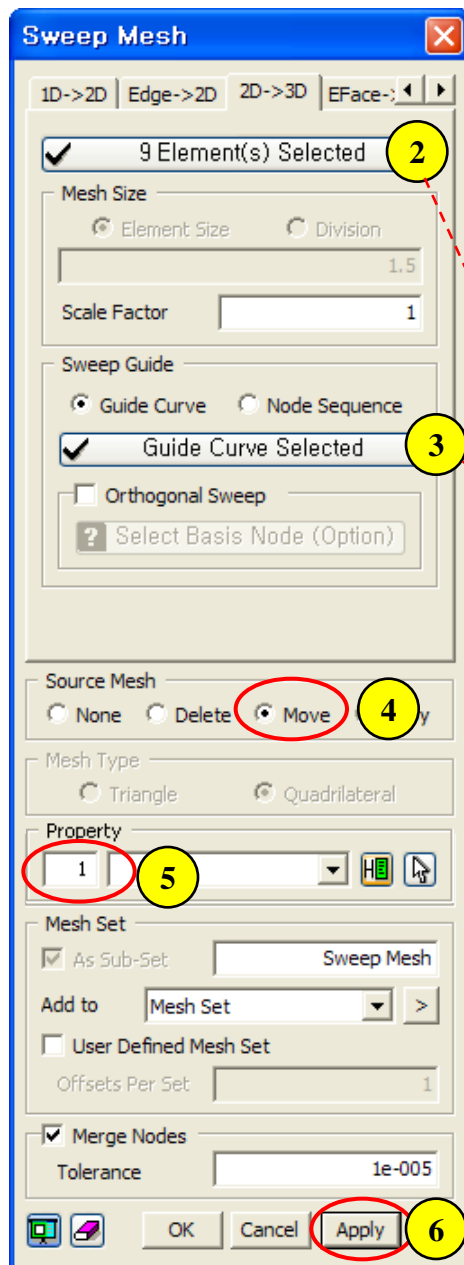
Step 14.



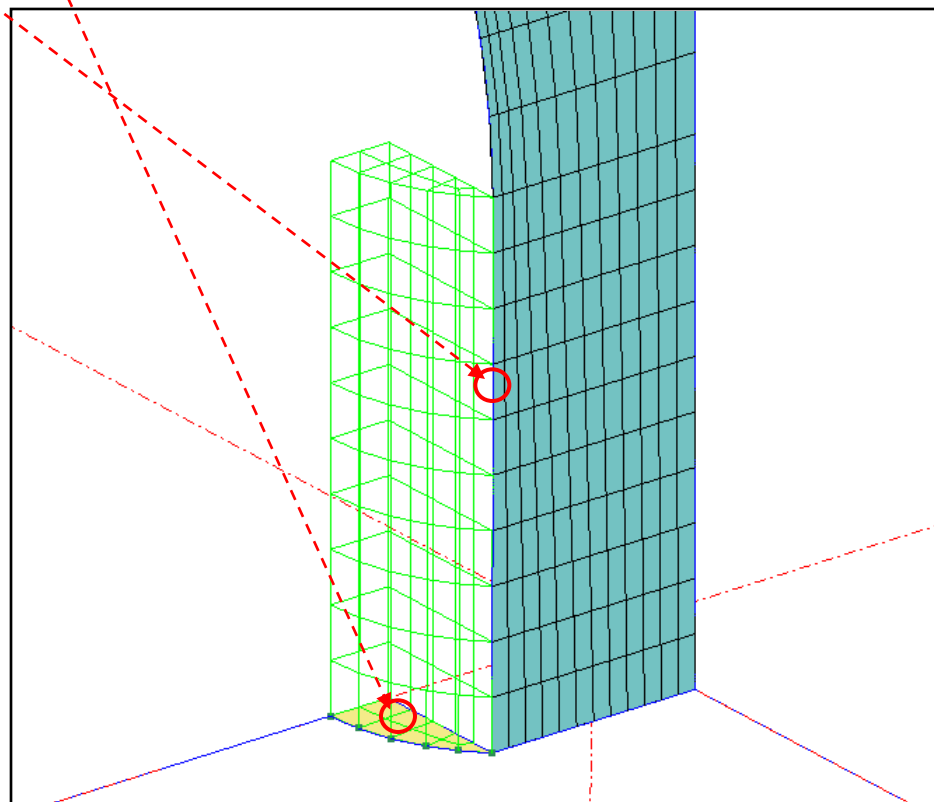
1. Mesh > Auto Mesh > Planar Area...
2. Select 3 edges marked by "O" (See Figure)
3. Mesher : Delaunay Mesher
4. Type : Quadrilateral
5. Mesh Size : Element Size (0.3)
6. Property : 99
7. Click [OK] Button



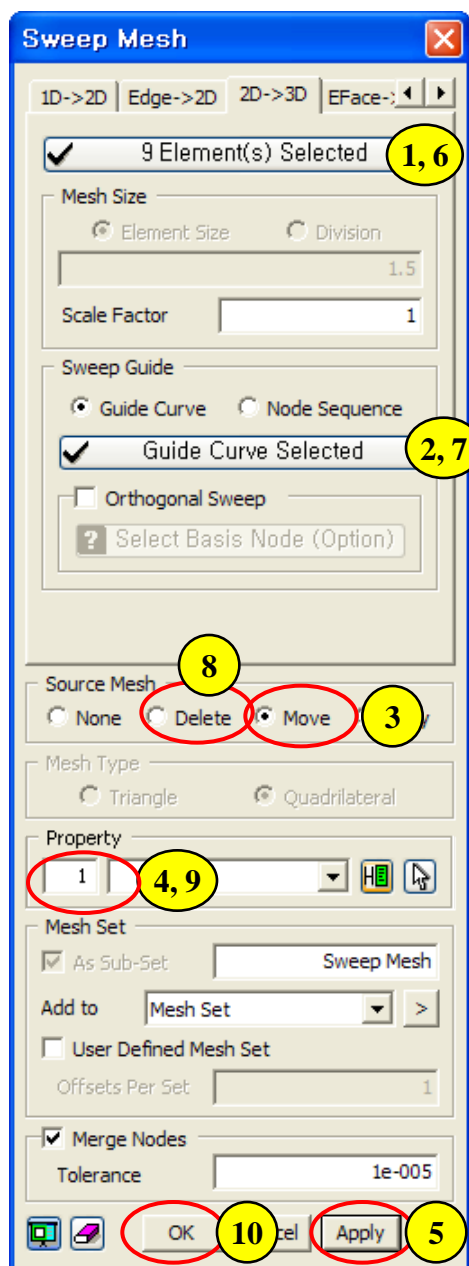
Step 15.



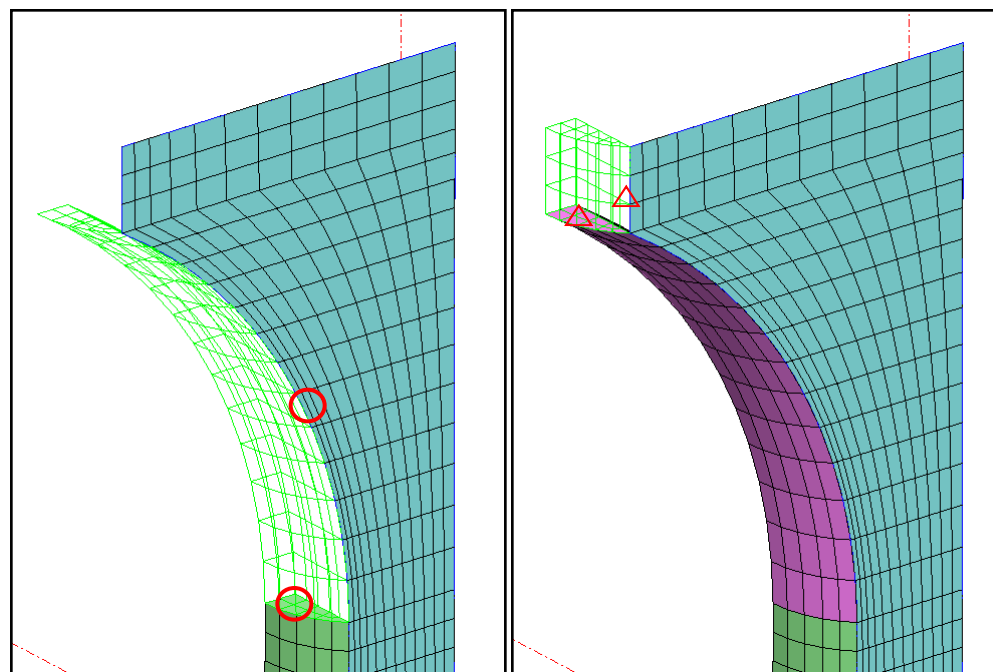
1. Mesh > Protrude Mesh > Sweep...
2. Select 1 Mesh set marked by "O" (See Figure)
3. Select 1 Edge marked by "O" (See Figure)
4. Source Mesh : Move
5. Property : 1
6. Click [Apply] Button



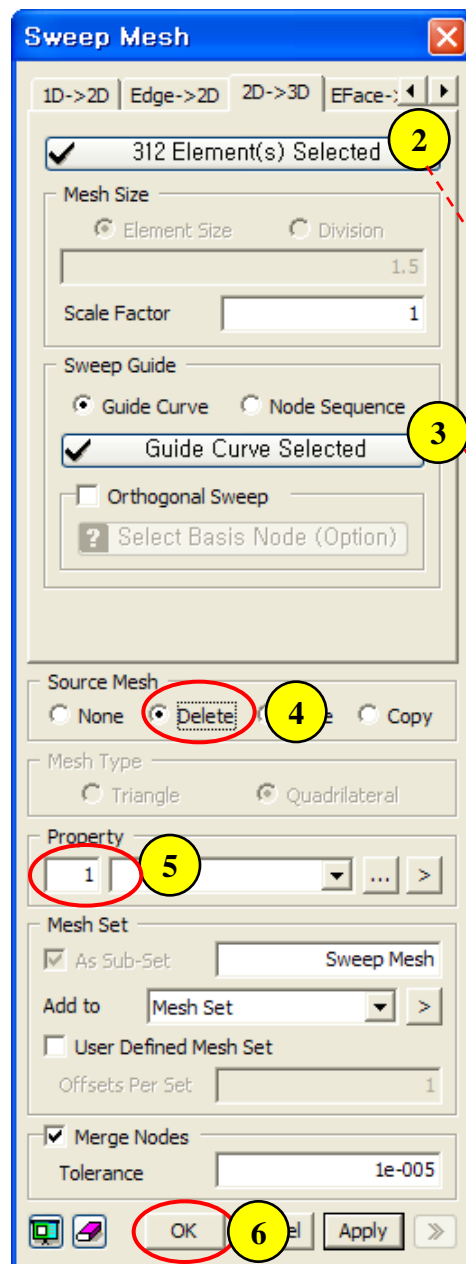
Step 16.



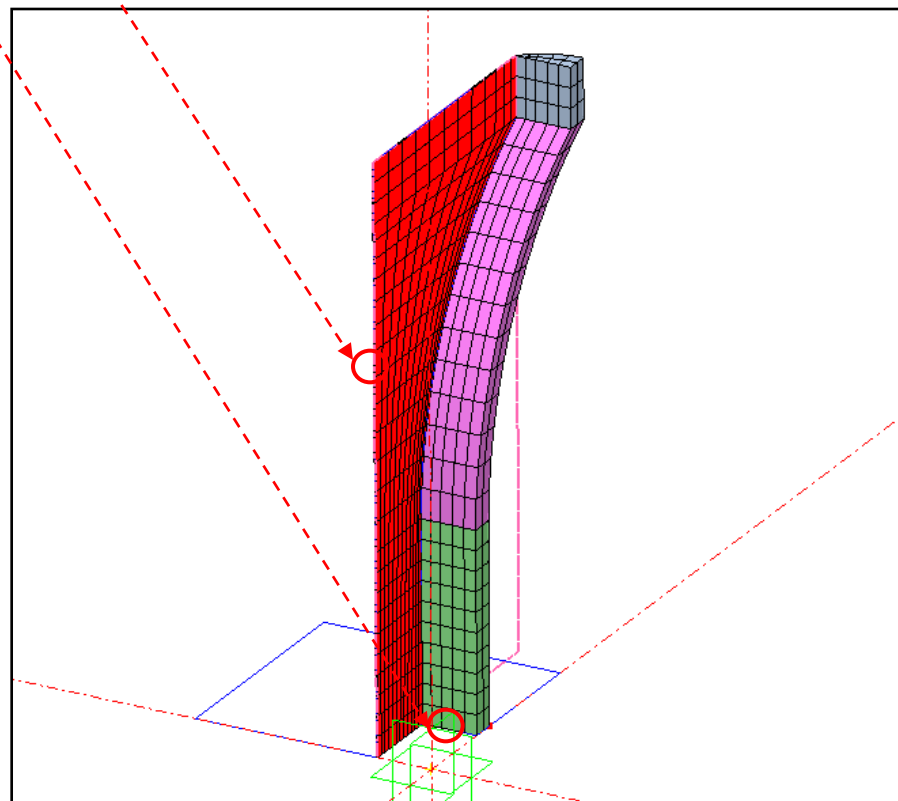
1. Select 1 Mesh set marked by “O” (See Figure)
2. Select 1 Edge marked by “O” (See Figure)
3. Source Mesh : Move
4. Property : 1
5. Click [Apply] Button
6. Select 1 Mesh set marked by “△” (See Figure)
7. Select 1 Edge marked by “△” (See Figure)
8. Source Mesh : Delete
9. Property : 1
10. Click [OK] Button



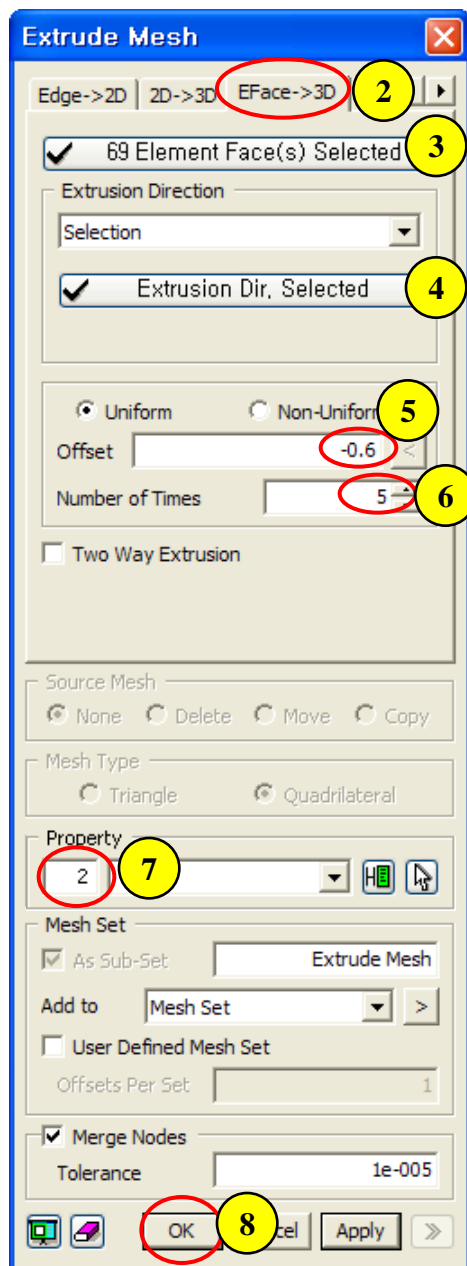
Step 17.



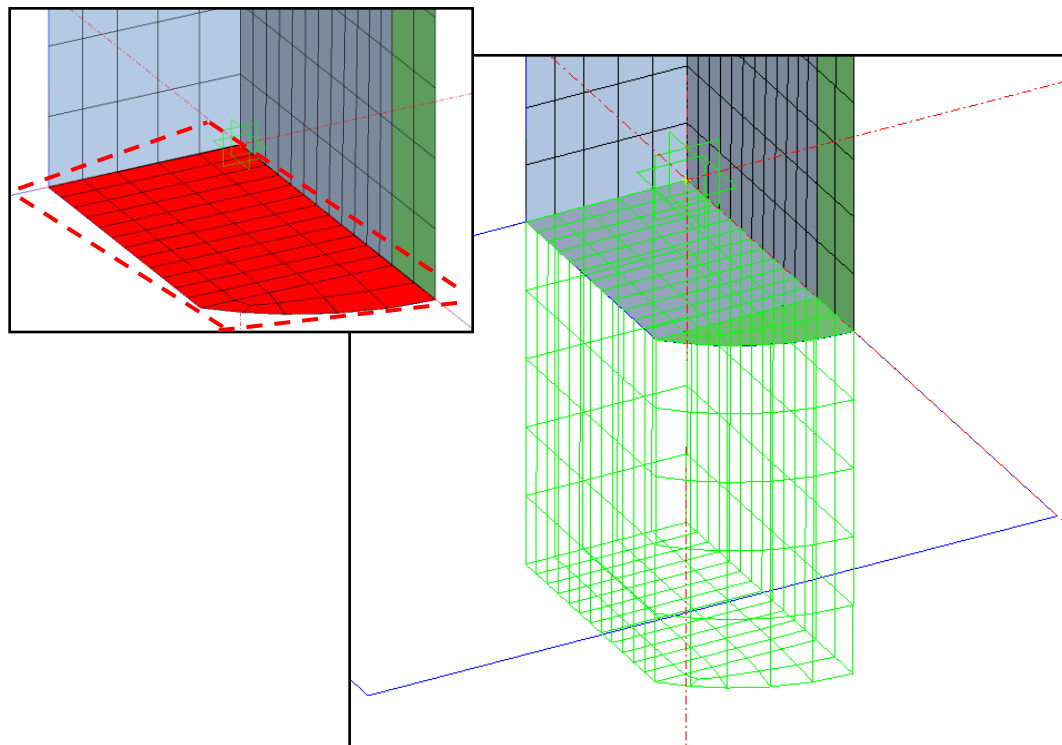
1. Mesh > Protrude Mesh > Sweep...
2. Select 1 Mesh set marked by "O" (See Figure)
3. Select 1 Edge marked by "O" (See Figure)
4. Source Mesh : Delete
5. Property : 1
6. Click [OK] Button



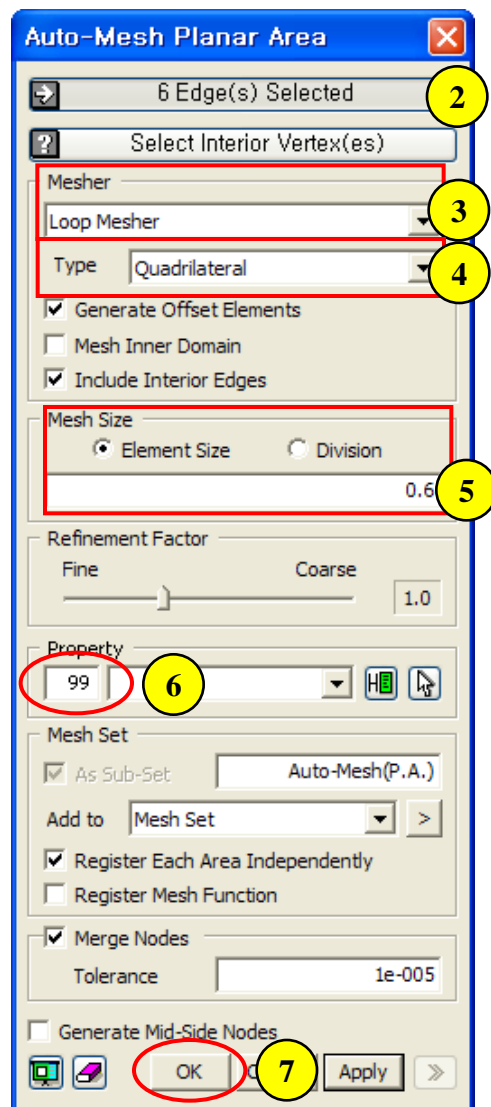
Step 18.



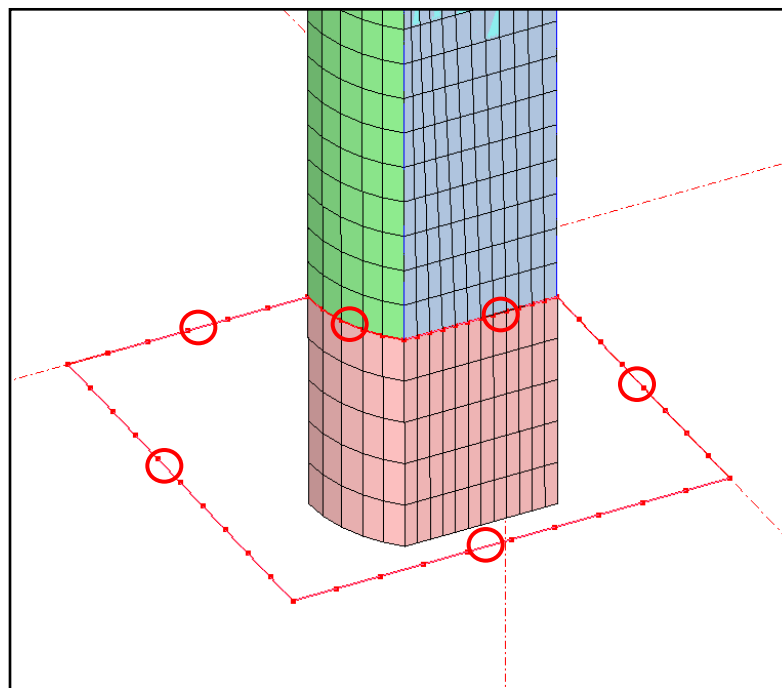
1. Mesh > Protrude Mesh > Extrude...
2. Select "Eface → 3D" Tab
3. Select 69 Element-Faces marked by "□" (See Figure)
4. Select Extrusion Direction : Z Axis
5. Offset : -0.6
6. Number of Times : 5
7. Property : 2
8. Click [OK] Button



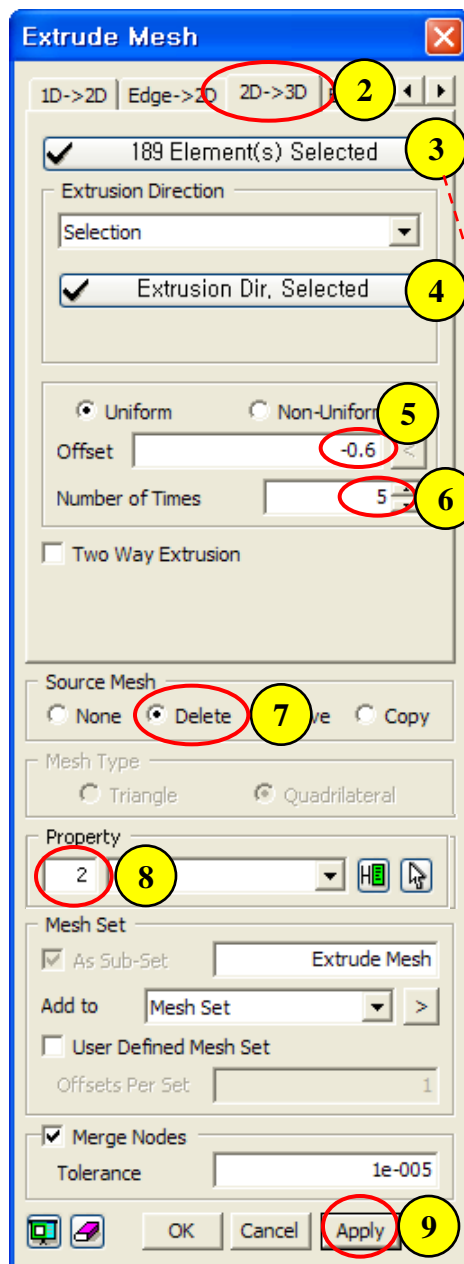
Step 19.



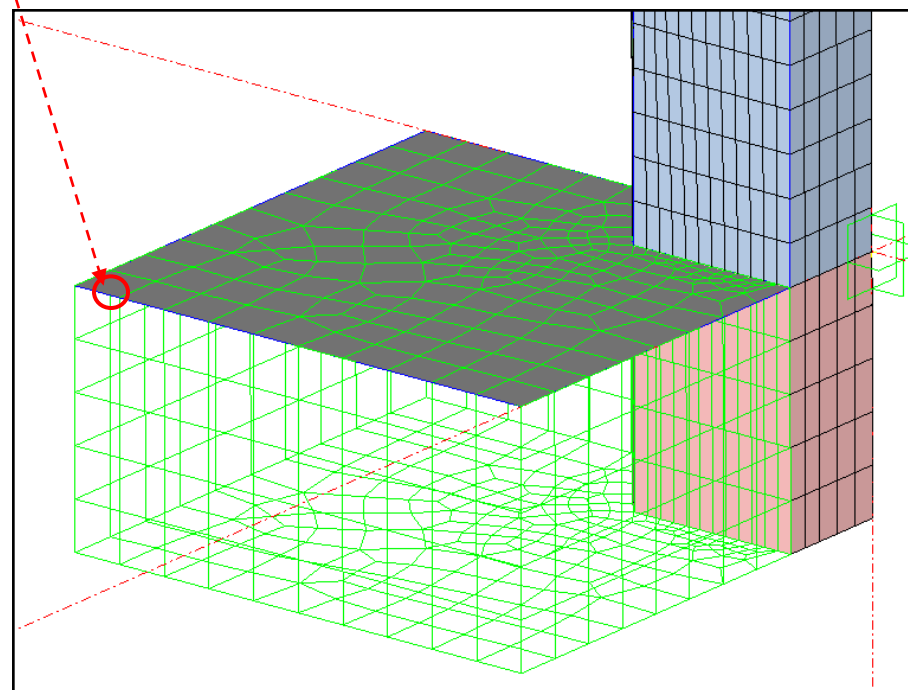
1. Mesh > Auto Mesh > Planar Area...
2. Select 6 Edges marked by "O" (See Figure)
3. Mesher : Loop Mesher
4. Type : Quadrilateral
5. Mesh Size : Element Size (0.6)
6. Property : 99
7. Click [OK] Button



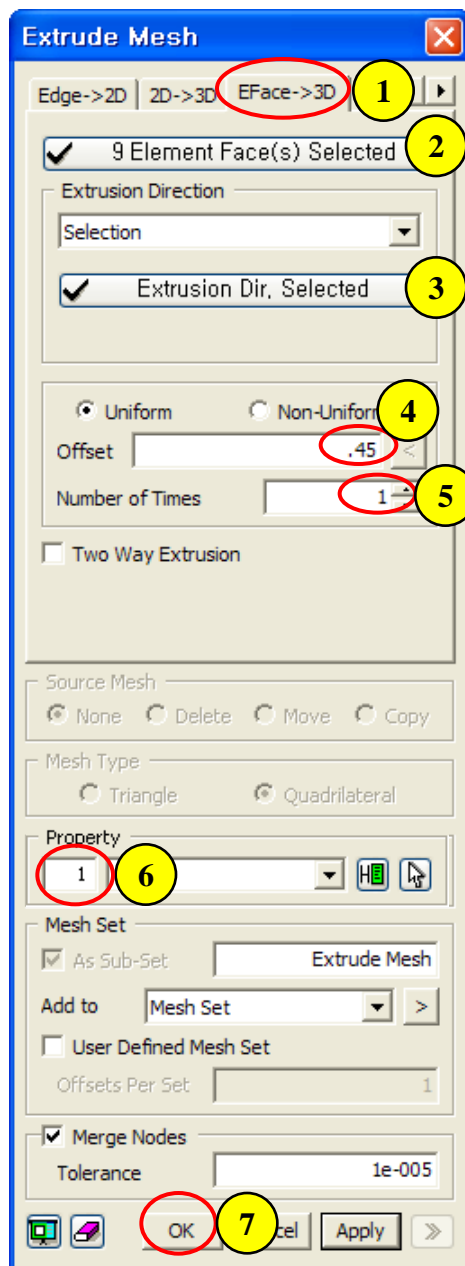
Step 20.



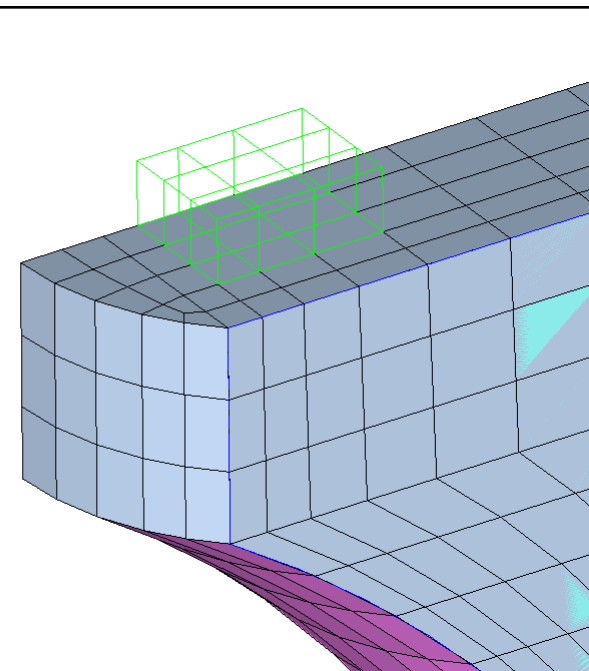
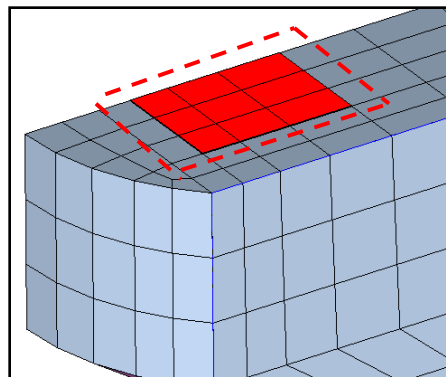
1. Mesh > Protrude Mesh > Extrude...
2. Select "2D→3D" Tab
3. Select 1 Mesh Set marked by "○" (See Figure)
4. Select Extrusion Direction : Z Axis
5. Offset : -0.6
6. Number of Times : 5
7. Source Mesh : Delete
8. Property : 2
9. Click [Apply] Button



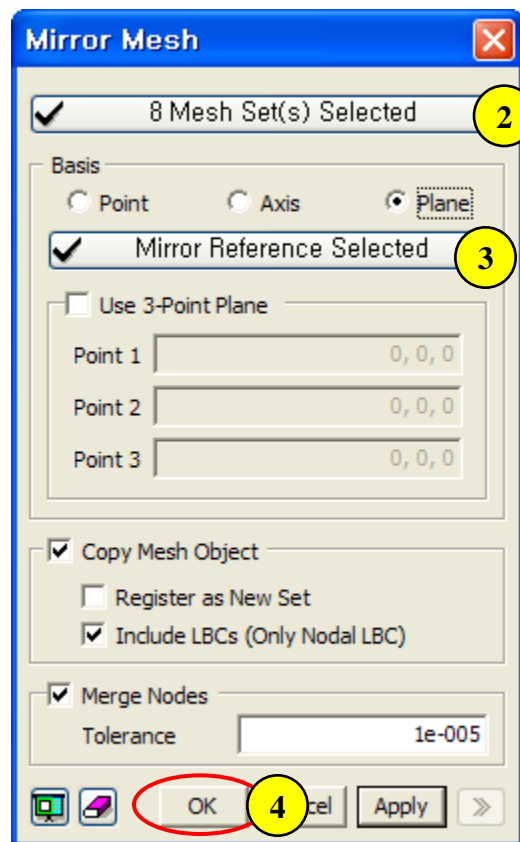
Step 21.



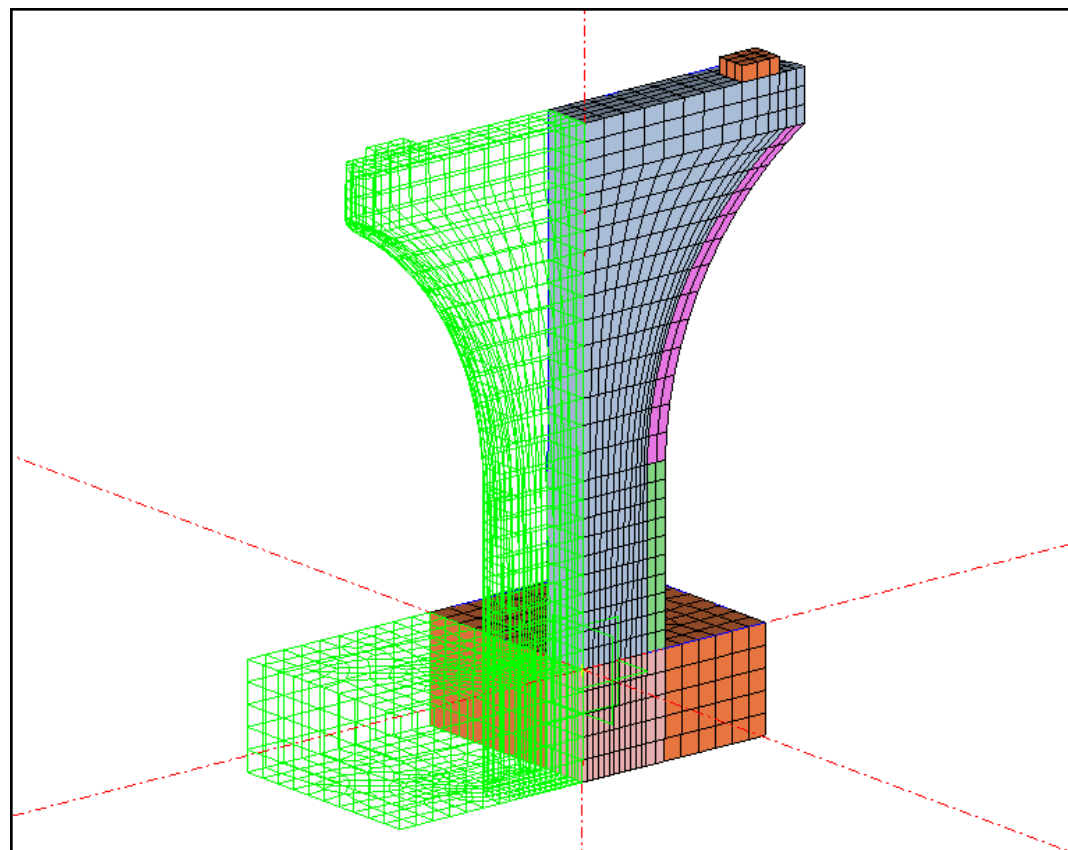
1. Select "EFace → 3D" Tab
2. Select 9 Element-Faces marked by "□" (See Figure)
3. Select Extrusion Direction : Z Axis
4. Offset : 0.45
5. Number of Times : 1
6. Property : 1
7. Click [OK] Button



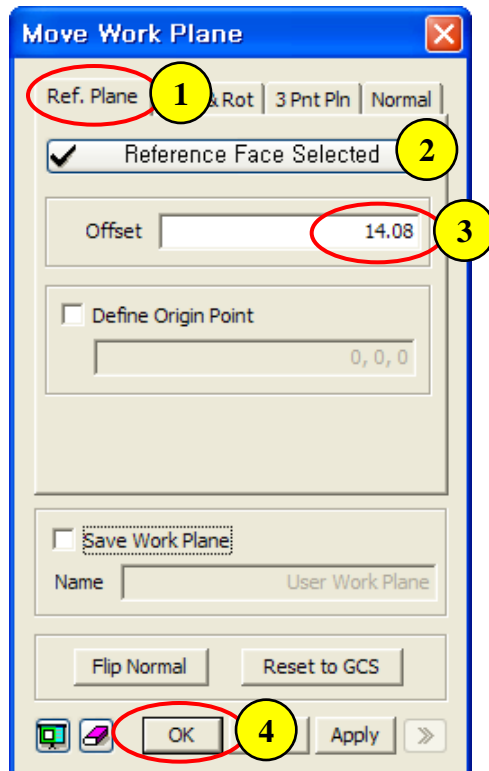
Step 22.



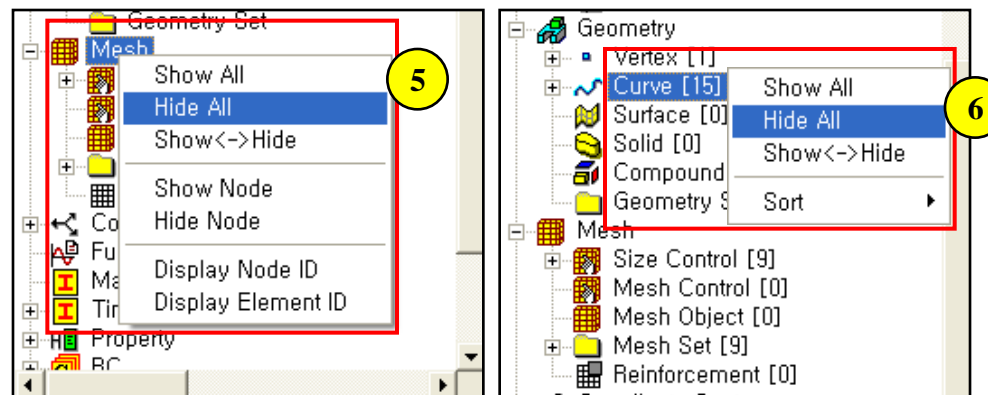
1. *Mesh > Transform > Mirror...*
2. *Select All Mesh Set (See Figure)*
3. *Select Mirror Plane : “YZ Plane”*
4. *Click [OK] Button*



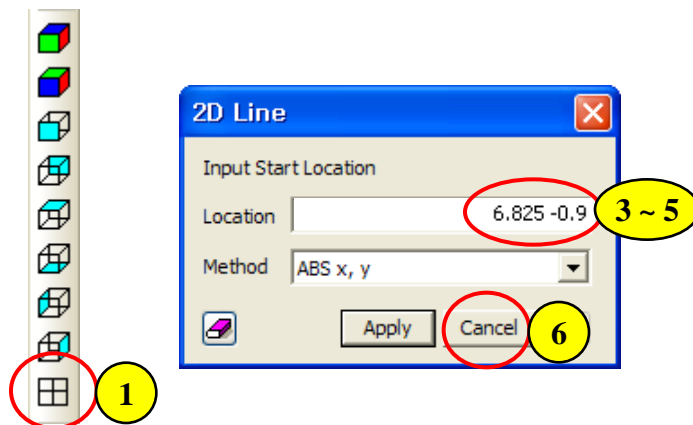
Step 23.



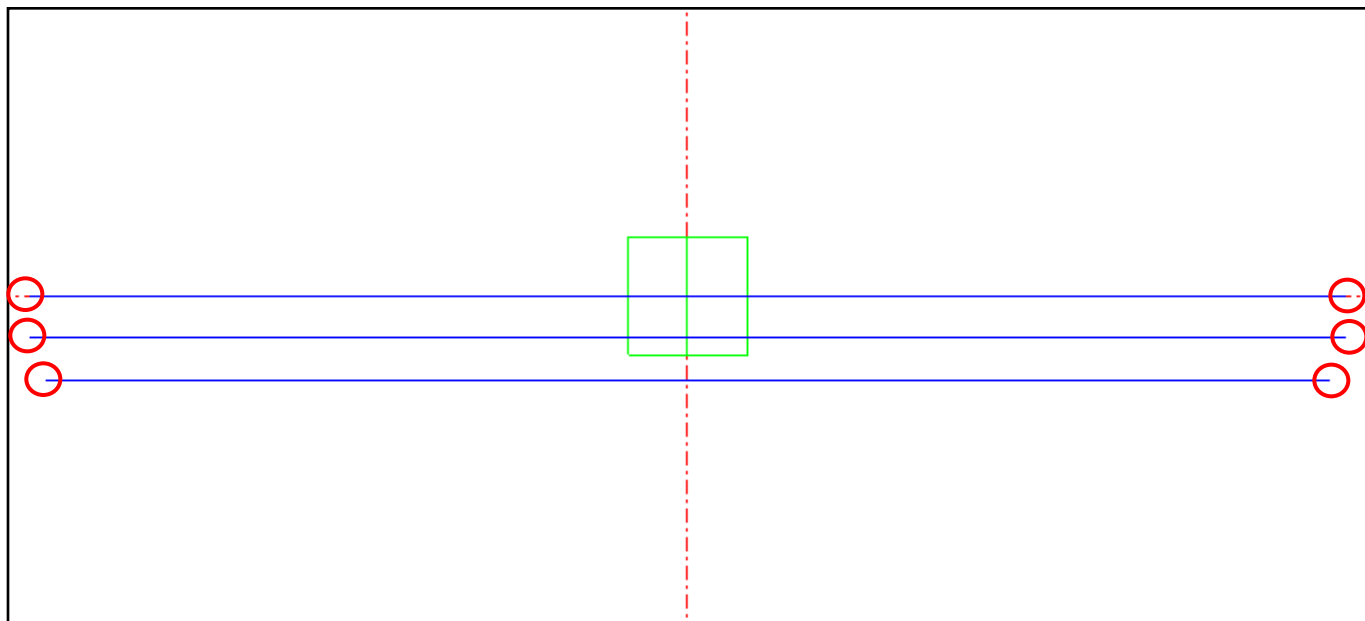
1. Geometry > Work Plane > Move – “Ref. Plane” tab
2. Select “XY Plane”
3. Offset : 14.08
4. Click [OK] Button
5. Select “Mesh” in Tree Menu and Hide All (Click Right Mouse Button)
6. Select “Curve” in Tree Menu and Hide All (Click Right Mouse Button)



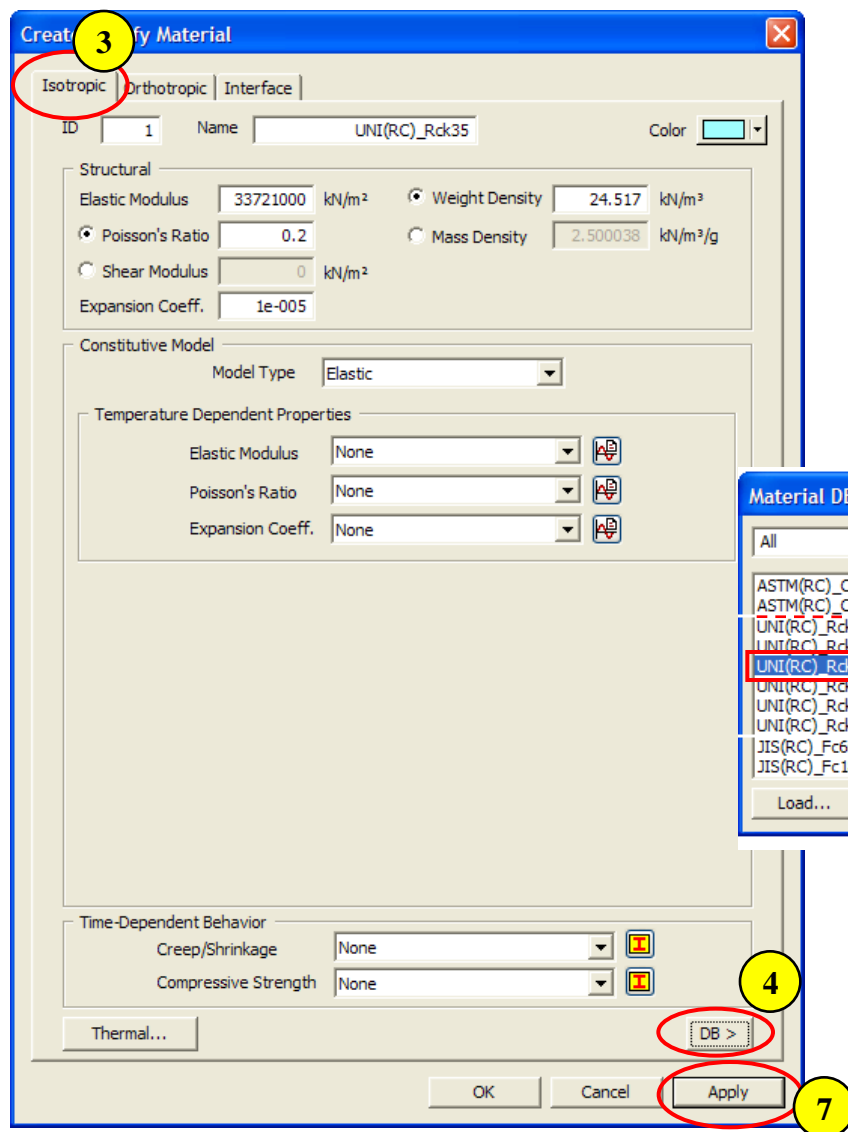
Step 24.



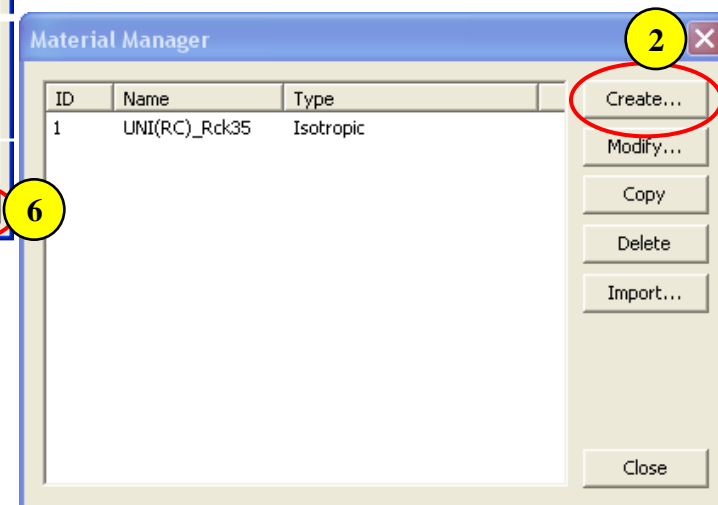
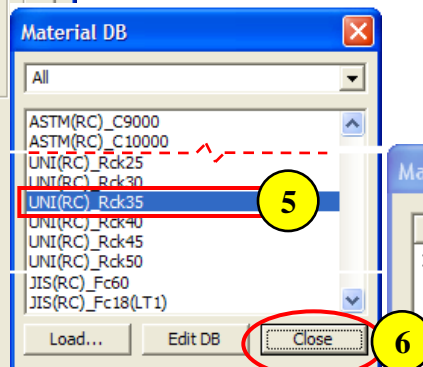
1. Click "Normal View"
2. Geometry > Curve > Create on WP > Line...
3. Location : (6.825, -0.9), <-13.65>
4. Location : (7, -0.45), <-14>
5. Location : (7, 0), <-14>
6. Click [Cancel] Button



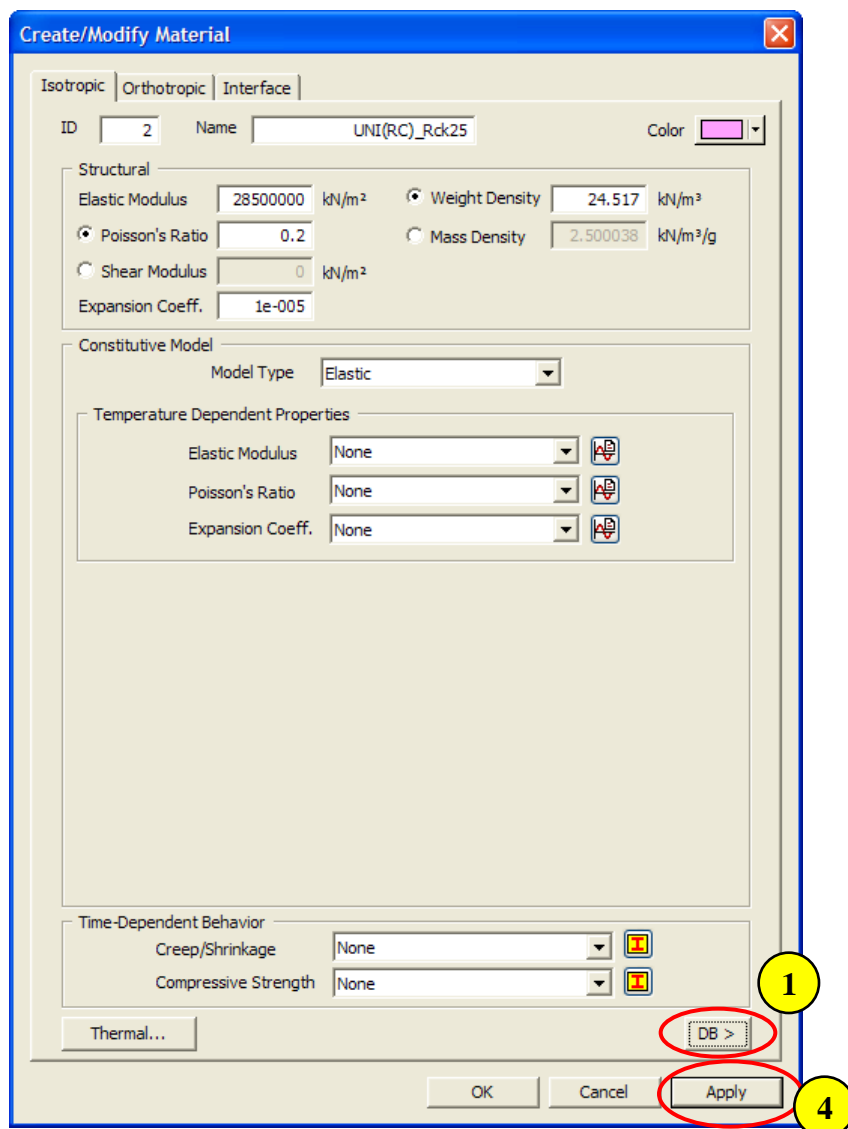
Step 25.



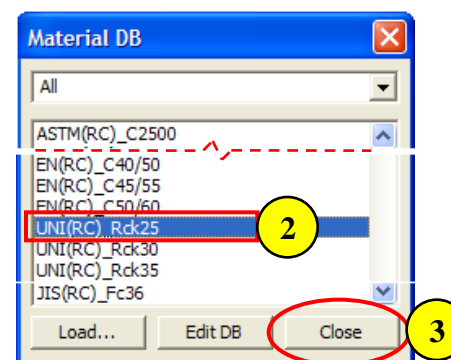
1. Analysis > Material ...
2. Click [Create...] Button
3. Select "Isotropic" tab
4. Click [DB] Button
5. Select "UNI(RC)_Rck35"
6. Click [Close] Button
7. Click [Apply] Button



Step 26.



1. Click [DB] Button
2. Select "UNI(RC)_Rck25"
3. Click [Close] Button
4. Click [Apply] Button



Step 27.

Create/Modify Material

Isotropic | Orthotropic | Interface

ID: 3 Name: Tendon Color: [Green]

Structural

Elastic Modulus: 200000000 kN/m² Weight Density: 78.5 kN/m³

Poisson's Ratio: 0.3 Mass Density: 2.500038 kN/m³/g

Shear Modulus: 0 kN/m²

Expansion Coeff.: 1e-005

Constitutive Model

Model Type: Elastic

Temperature Dependent Properties

Elastic Modulus: None

Poisson's Ratio: None

Expansion Coeff.: None

Time-Dependent Behavior

Creep/Shrinkage: None

Compressive Strength: None

Thermal... DB >

OK Cancel Apply

1. Name : Tendon
2. Elastic Modulus : 200,000,000 kN/m²
3. Poisson Ratio : 0.3
4. Weight Density : 78.5kN/m³
5. Click [OK] Button
6. Click [Close] Button

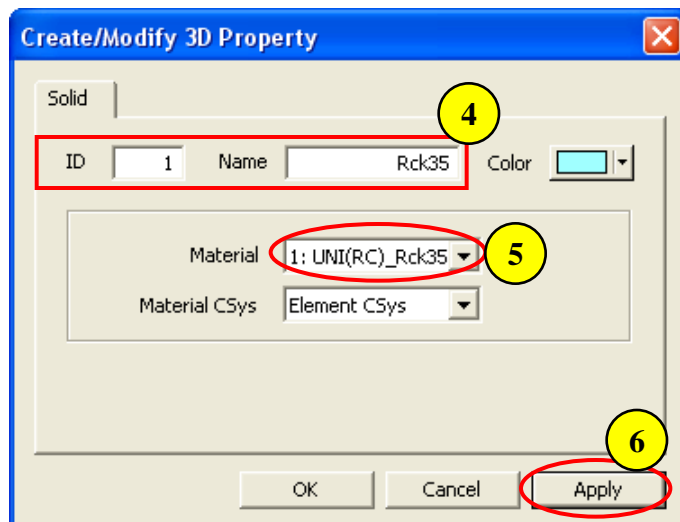
Material Manager

ID	Name	Type
1	UNI(RC)_Rck35	Isotropic
2	UNI(RC)_Rck25	Isotropic
3	Tendon	Isotropic

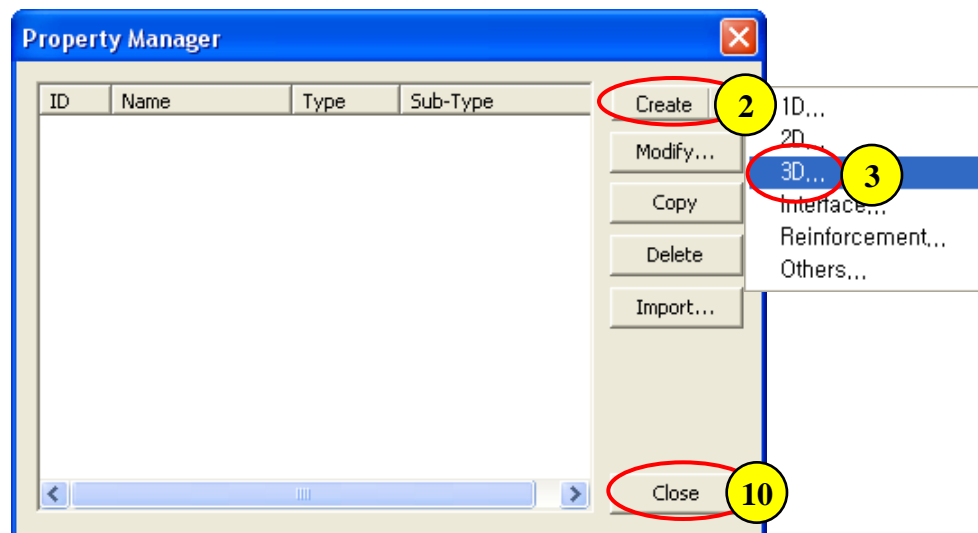
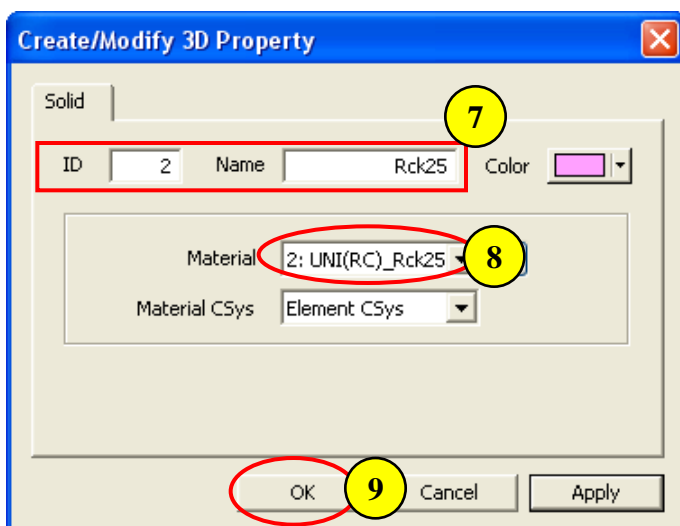
Create... Modify... Copy Delete Import...

Close

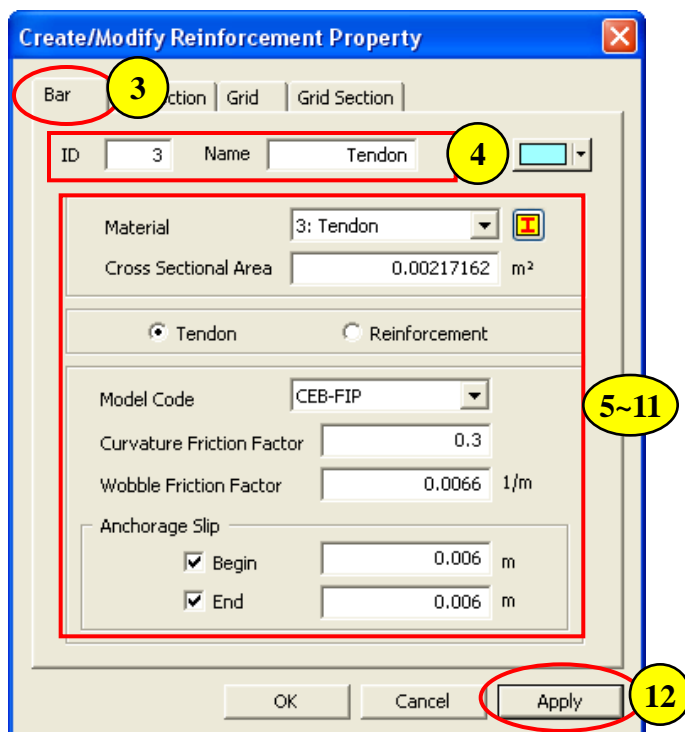
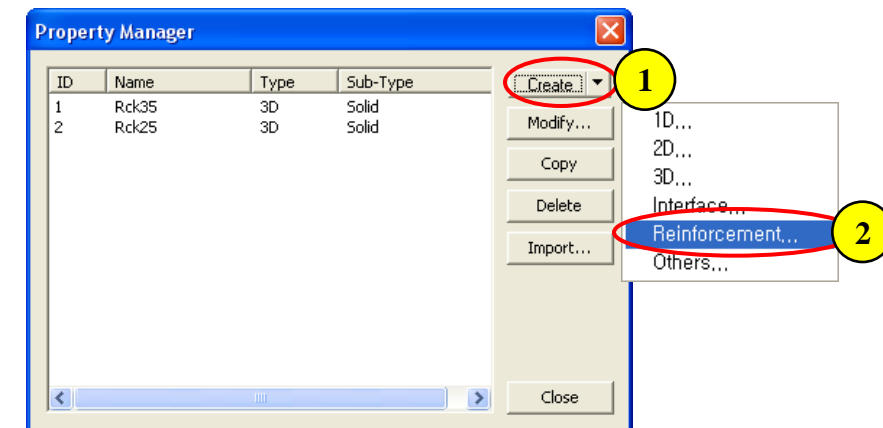
Step 28.



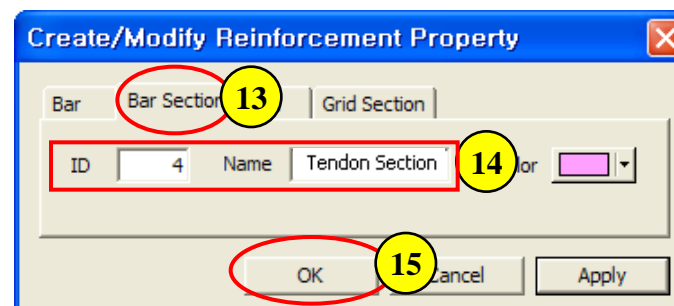
1. Analysis > Property...
2. Click "Create" Button
3. Select "3D"
4. ID : 1, Name : Rck35
5. Material : "1: UNI(RC)_Rck35"
6. Click [Apply] Button
7. ID : 2, Name : Rck25
8. Material : "2: UNI(RC)_Rck25"
9. Click [OK] Button
10. Click [Close] Button



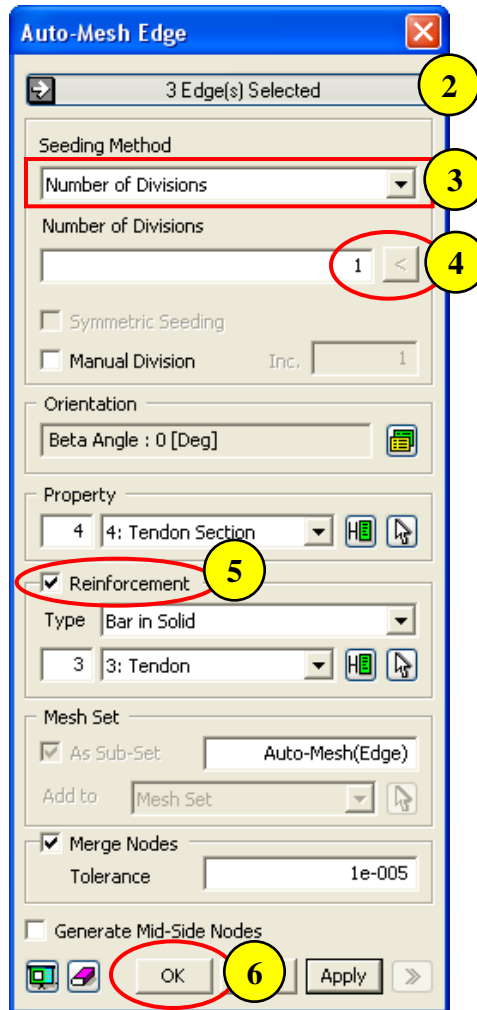
Step 29.



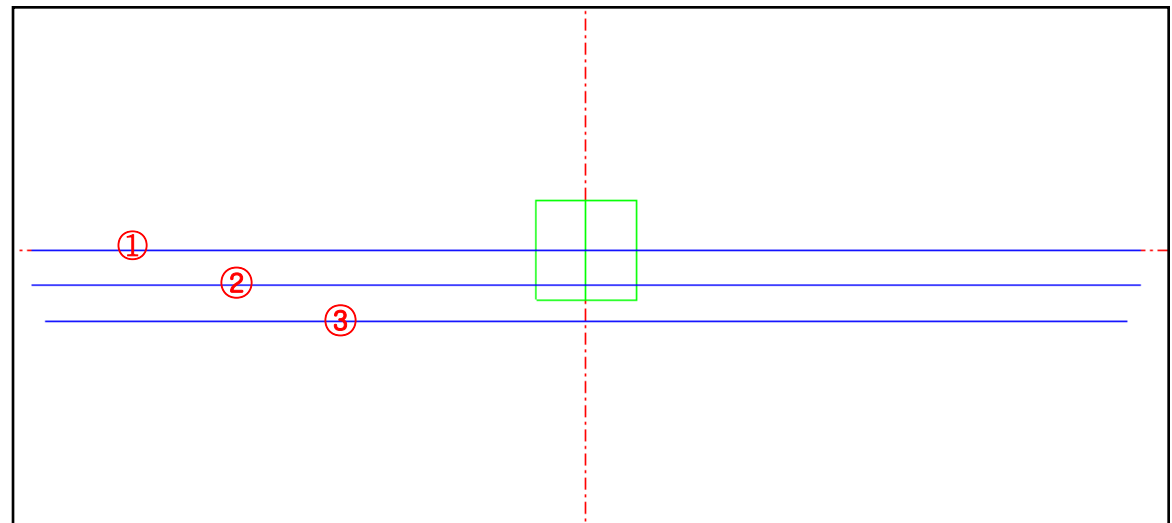
1. Click "Create" Button
2. Select "Reinforcement"
3. Select "Bar" Tab
4. ID : 3, Name : Tendon
5. Material : "3: Tendon"
6. Cross Sectional Area : 0.00217162m^2
7. Check on "Tendon"
8. Model Code: "CEB-FIP"
9. Curvature Friction Factor : 0.3
10. Wobble Friction Factor : 0.0066
11. Anchorage Slip : Begin : 0.006
End : 0.006
12. Click [Apply] Button
13. Select "Bar Section" Tab
14. ID : 4, Name : Tendon Section
15. Click [OK] Button



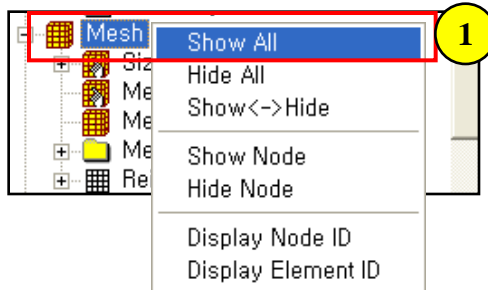
Step 30.



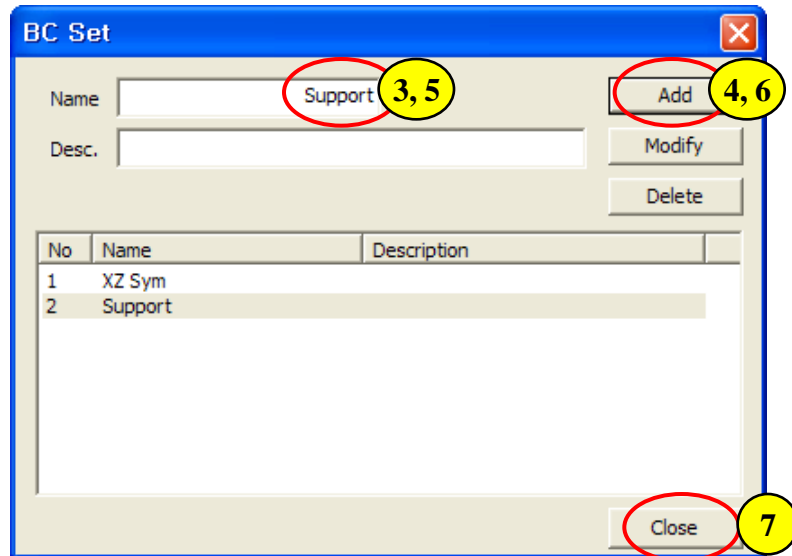
1. Mesh > Auto Mesh > Edge...
2. Select 3Edges shown on the figure
3. Seeding Method : Number of Divisions
4. Number of Divisions : 1
5. Check on "Reinforcement"
6. Click [OK] Button



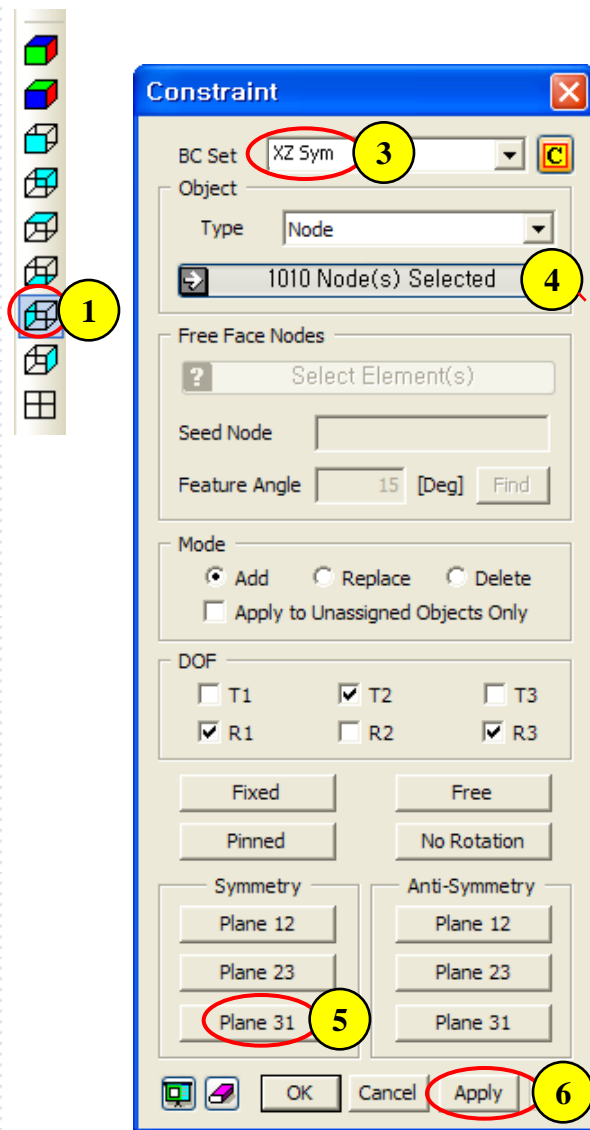
Step 31.



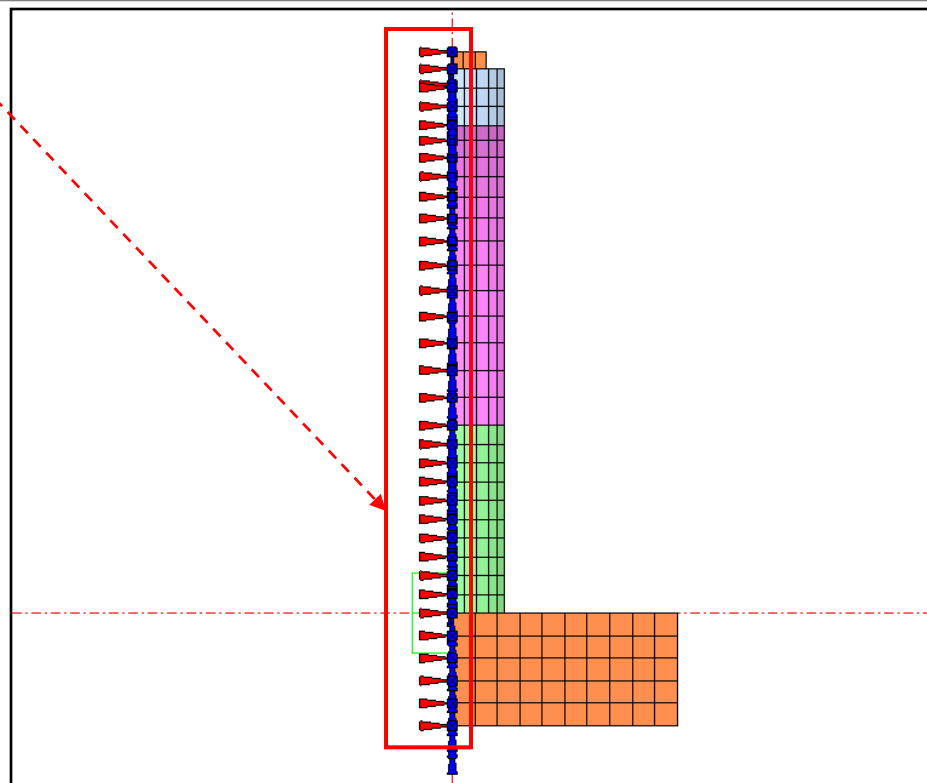
1. Select "Mesh" in Tree Menu and Show All (Click Right Mouse Button)
2. Analysis > BC > Set...
3. Name : XZ Sym
4. Click [Add] Button
5. Name : Support
6. Click [Add] Button
7. Click [Close] Button



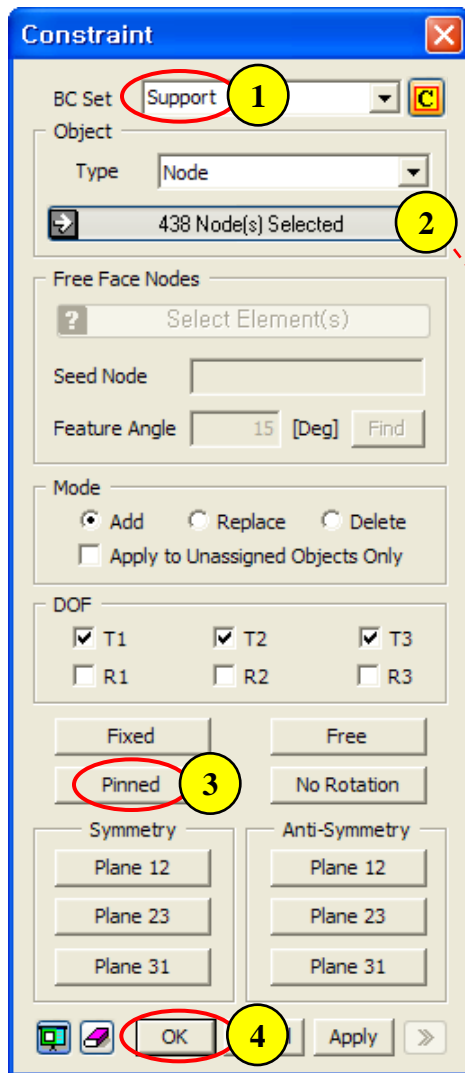
Step 32.



1. Toggle on "Left View"
2. Analysis > BC > Constraint...
3. BC Set : "XZ Sym"
4. Select Nodes marked by "□" (See Figure)
5. Click [Plane31] Button in Symmetry
6. Click [Apply] Button



Step 33.

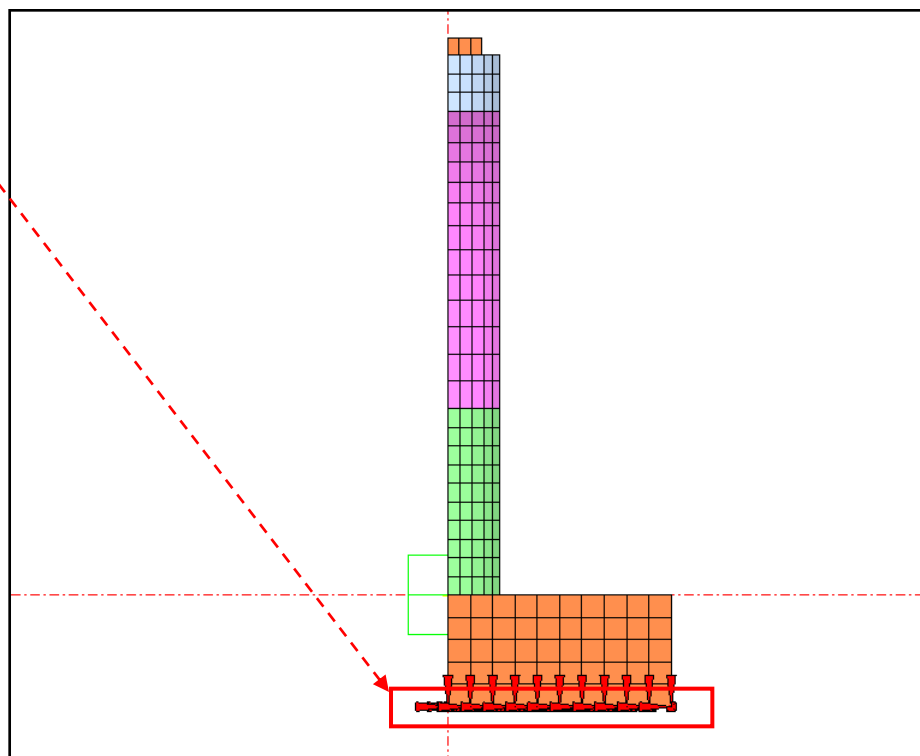


1. BC Set : “Support”

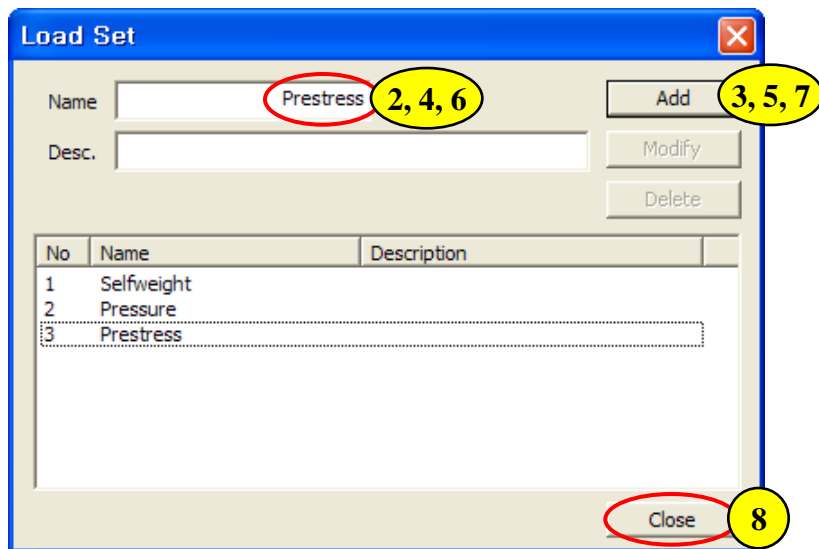
2. Select Nodes marked by “□” (See Figure)

3. Click [Pinned] Button in Symmetry

4. Click [OK] Button



Step 34.



1. Analysis > Load > Set...

2. Name : "Self weight"

3. Click [Add] Button

4. Name : "Pressure"

5. Click [Add] Button

6. Name : "Prestress"

7. Click [Add] Button

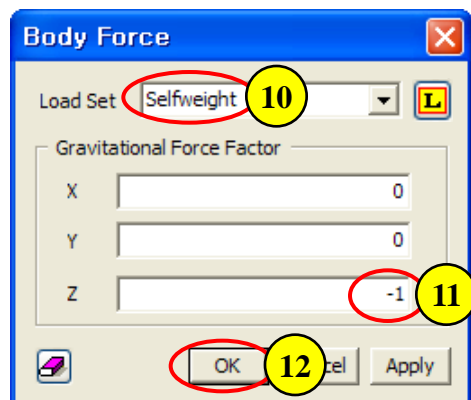
8. Click [Close] Button

9. Analysis > Load > Body Force...

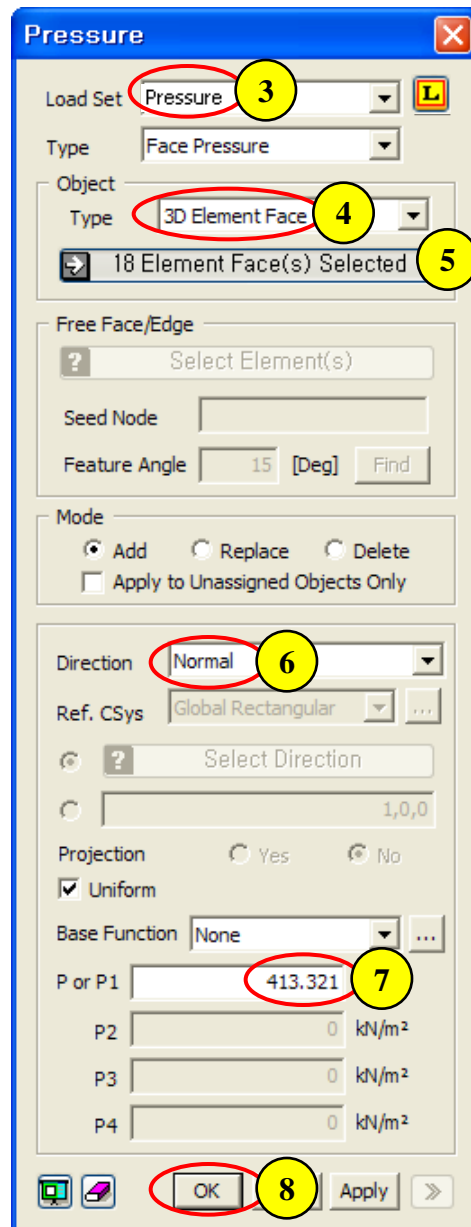
10. Load Set : Self weight

11. Gravitational Force Factor : Z : -1

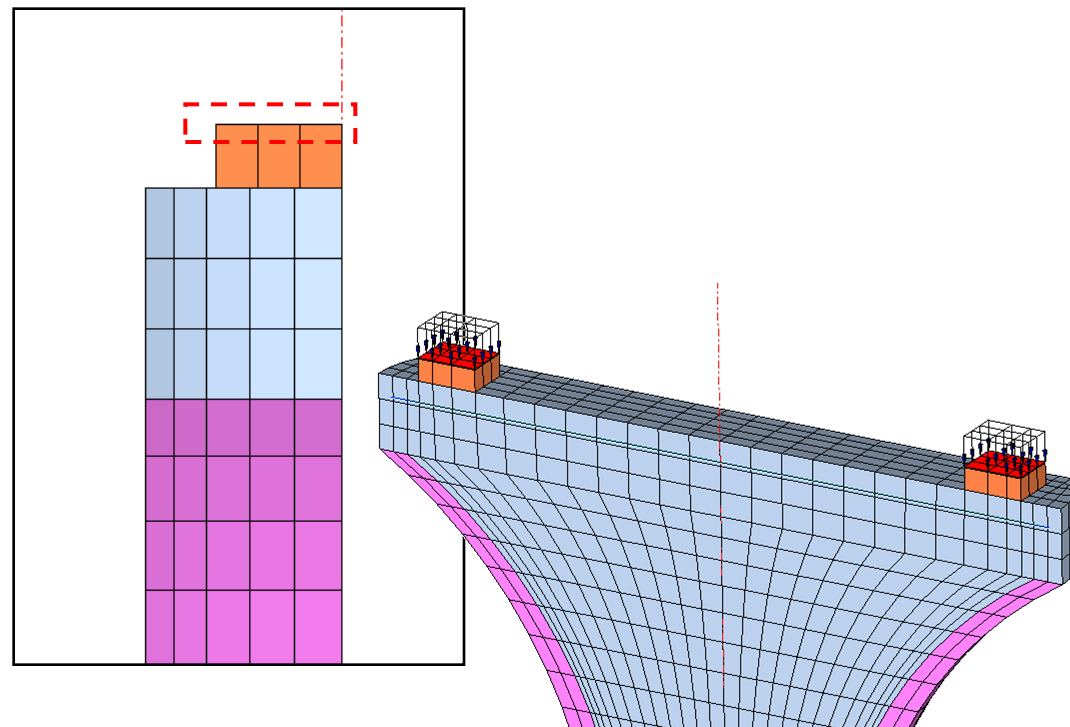
12. Click [OK] Button



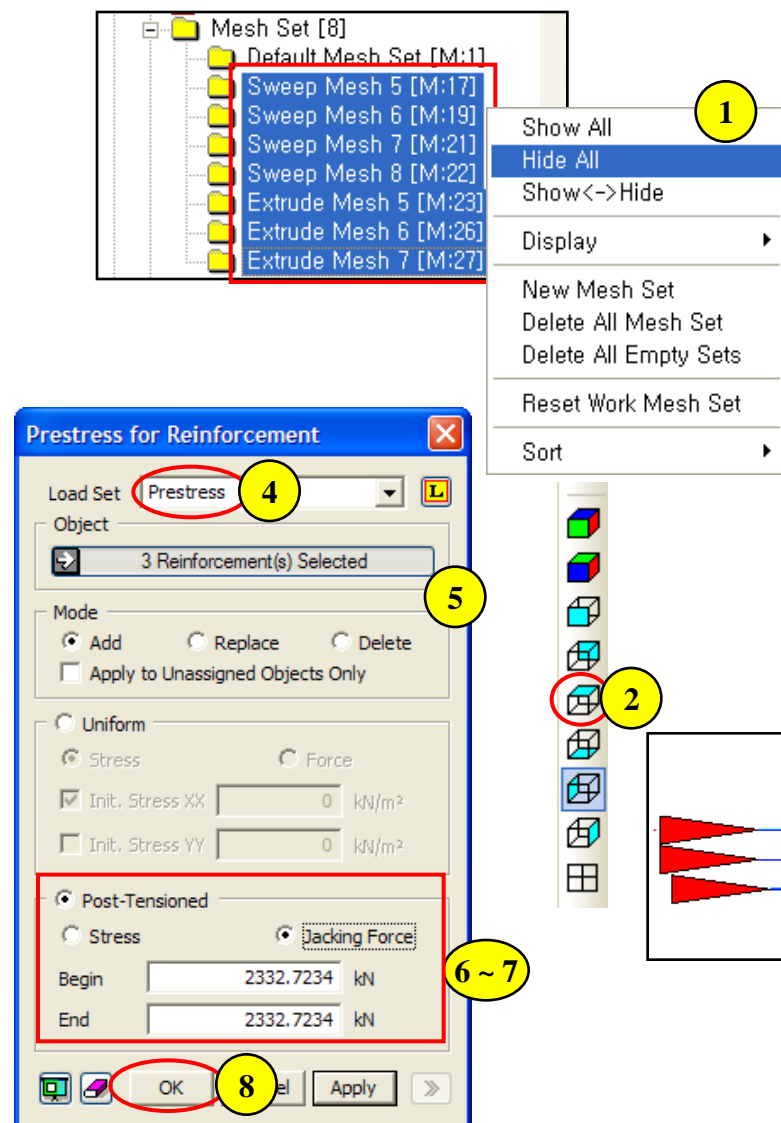
Step 35.



1. Toggle on "Right View"
2. Analysis > Load > Pressure...
3. Load Set : "Pressure"
4. Object Type : "3D Element Face"
5. Select 3D Element Faces marked " " (See Figure)
6. Direction : Normal
7. P or P1 : 413.321 kN/m²
8. Click [OK] Button

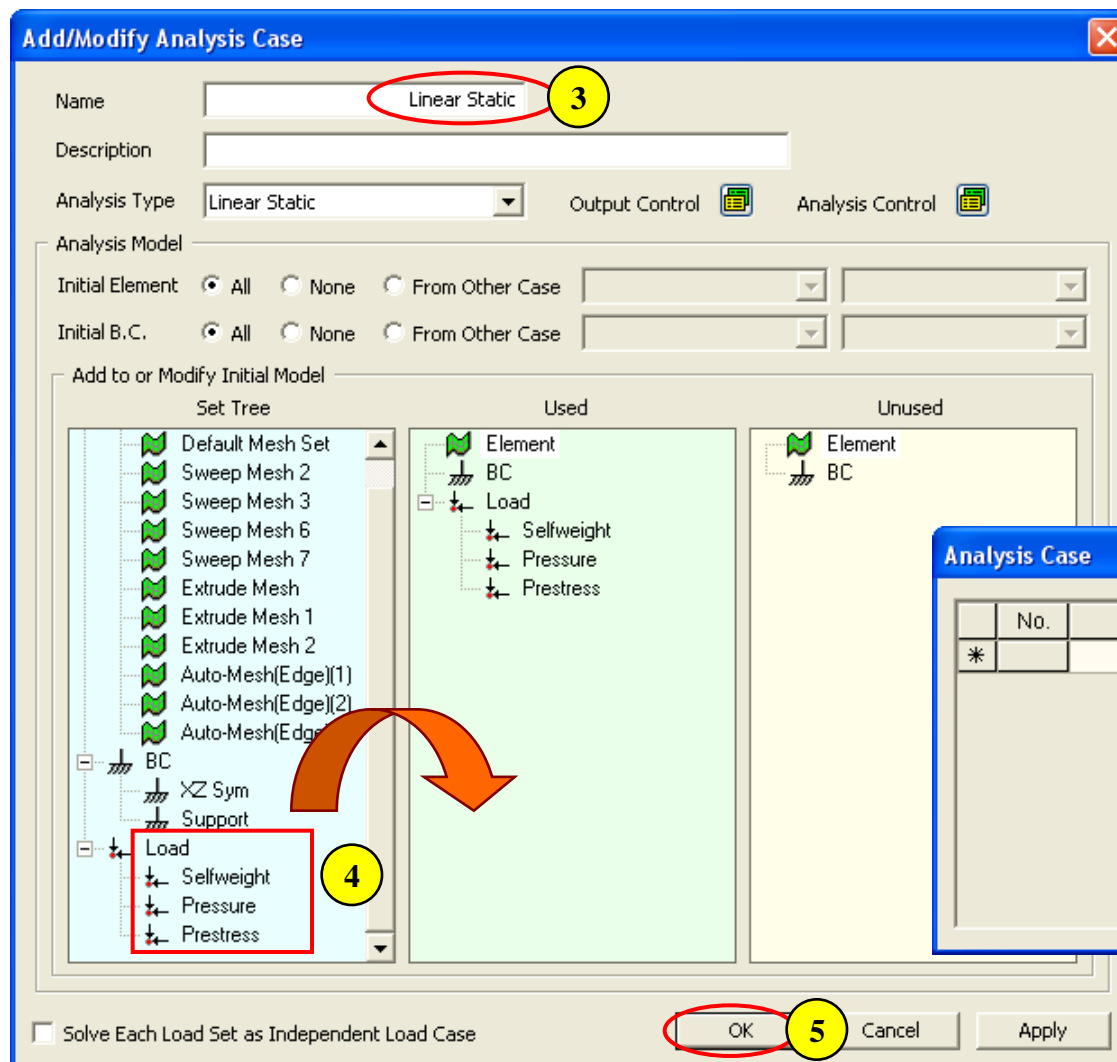


Step 36.



1. Select 7 Mesh Sets in Tree Menu and Hide All (Click Right Mouse Button)
2. Toggle on “Top View”
3. Analysis > Load > Prestress for Reinforcement...
4. Load Set : Prestress
5. Select Mesh Sets as shown in figure
6. Check on “Post-Tensioned (Jacking Force)”
7. Begin : 2332.7234, End : 2332.7234
8. Click [OK] Button

Step 37.



1. Analysis > Analysis Case...

2. Click [Add] Button

3. Name : "Linear Static"

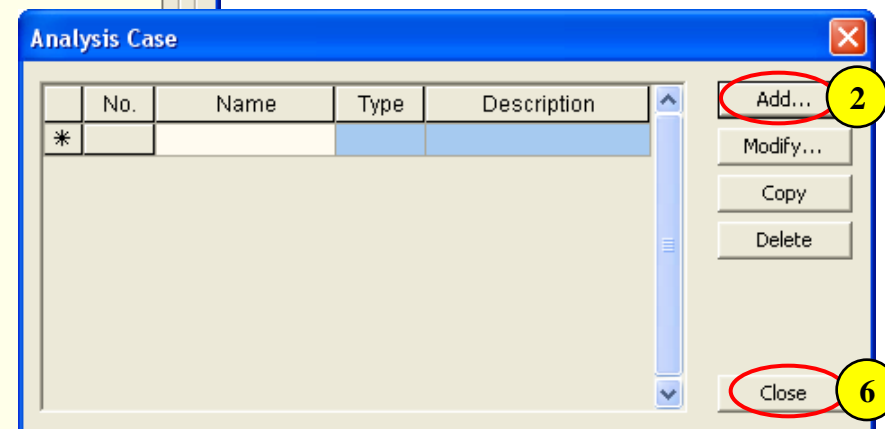
4. Drag & Drop Load Sets marked by "□"
(See Figure)

5. Click [OK] Button

6. Click [Close] Button

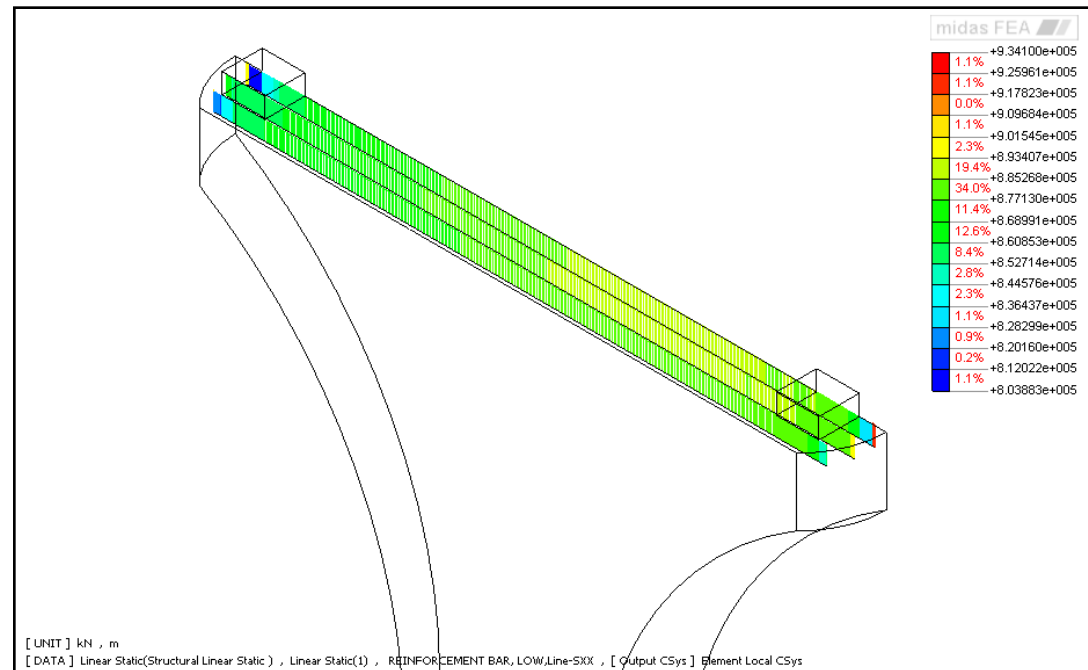
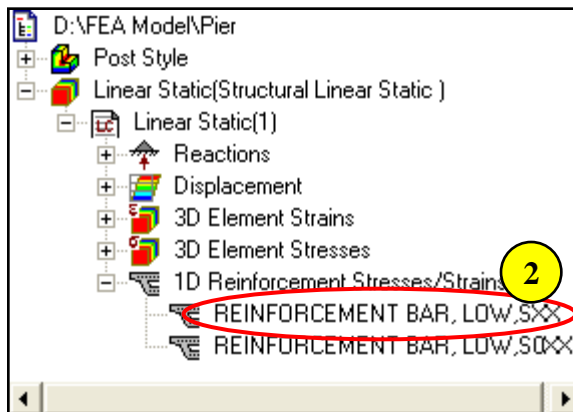
7. Analysis > Solve ...

8. Click [OK] Button

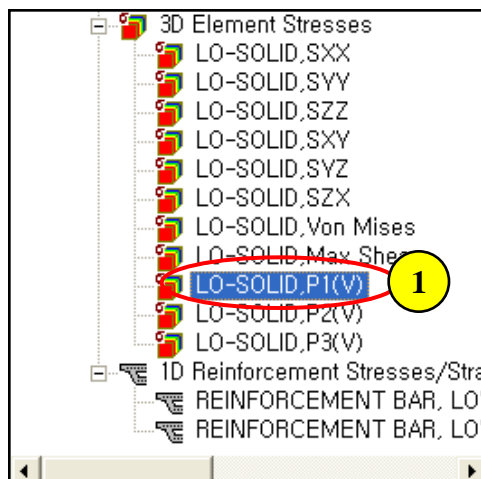


Step 38.

1. Post-Works Tree : Linear Static (Structural Linear Static) > Linear Static(1)
> 1D Reinforcement Stresses/Strains
2. Double Click on “REINFORCEMENT BAR, LOW,SXX”



Step 39.



1. Double Click on “LO-SOLID, P1(V)”

2. Property Window > Vector > Vector Type : Contour

3. Click [Apply] Button

4. Post Data Tab > Plot Type > Vector Plot

5. Post Style Tab > Edge Type > Feature Edge

