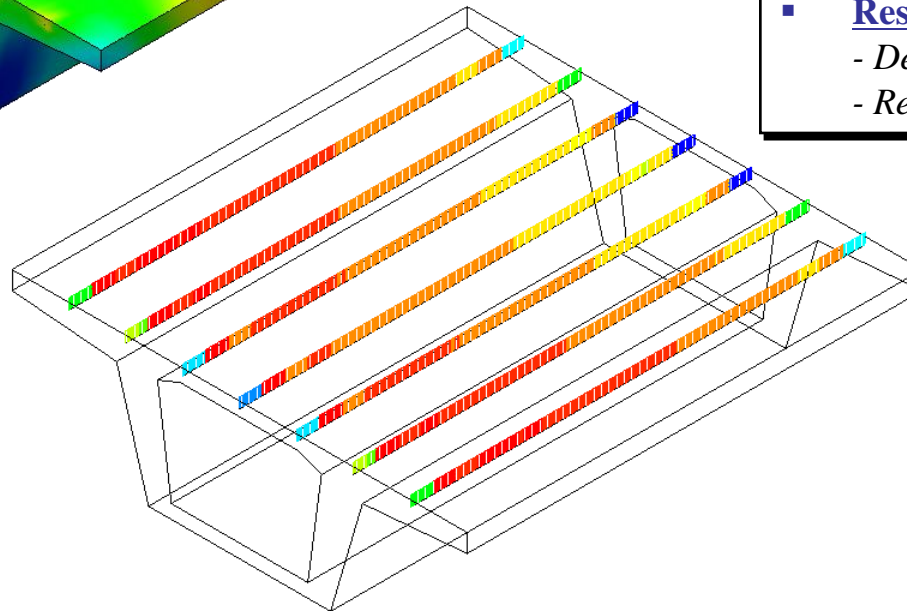
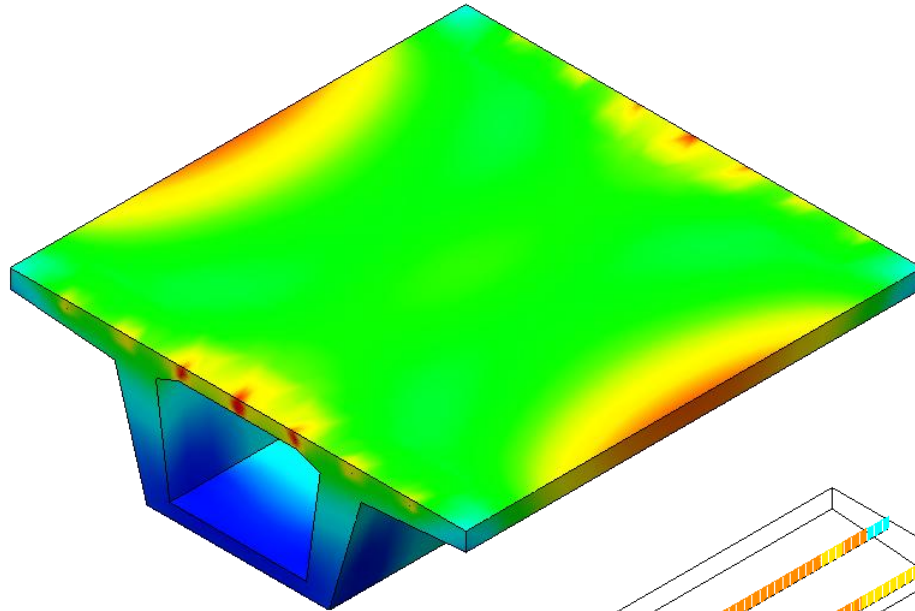


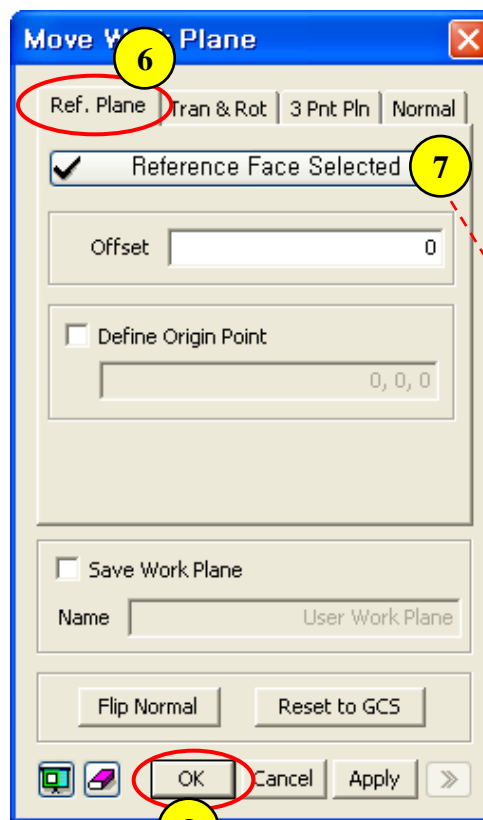
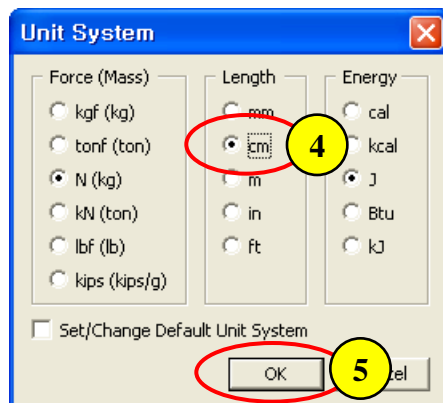
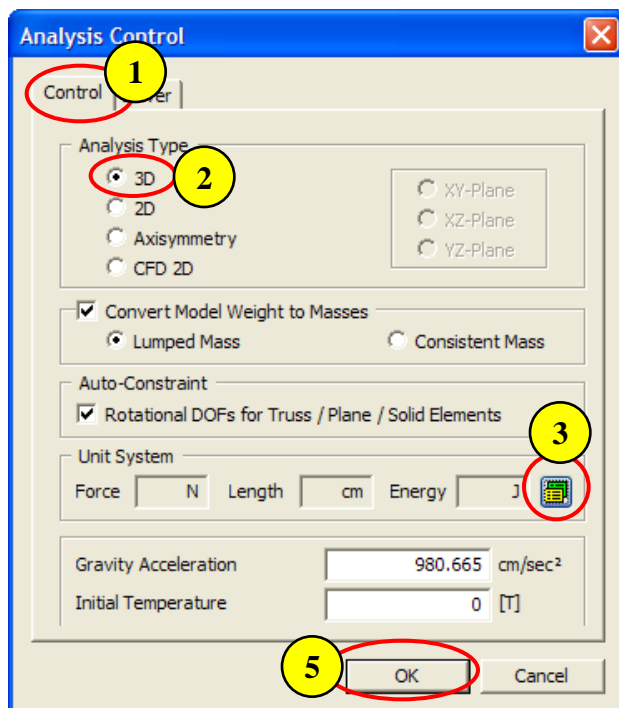
## LS-13. Analysis of a Box Girder



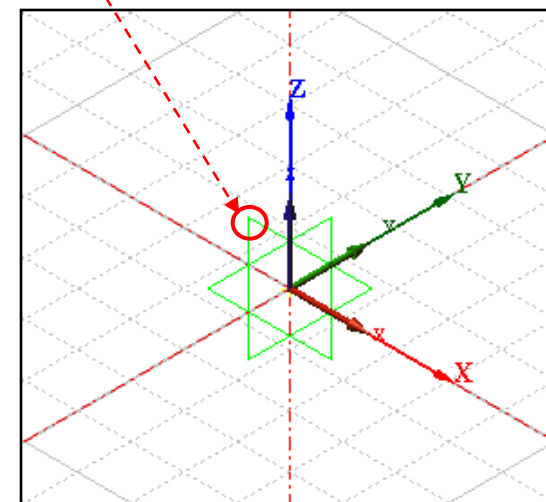
### Overview

- 3-D Linear Static Analysis
- Model
  - Unit : N, cm
  - Isotropic Elastic Material
  - Reinforcement Element
  - Solid Element
- Load & Boundary Condition
  - Body Force
  - Pressure
  - Prestress for Reinforcement
  - Constraint
- Result Evaluation
  - Deformation
  - Reinforcement Stress

## Step 1.

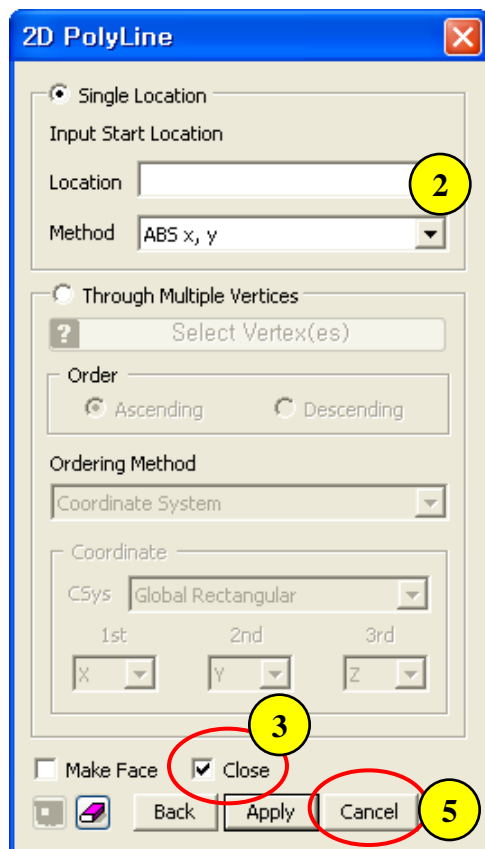


1. Analysis > Analysis Control – “Control” tab
2. Analysis Type : 3D
3. Click Button
4. Length : cm
5. Click [OK] Button
6. Geometry > Work Plane > Move – “Ref. Plane” Tab
7. Select “XZ Plane”
8. Click [OK] Button

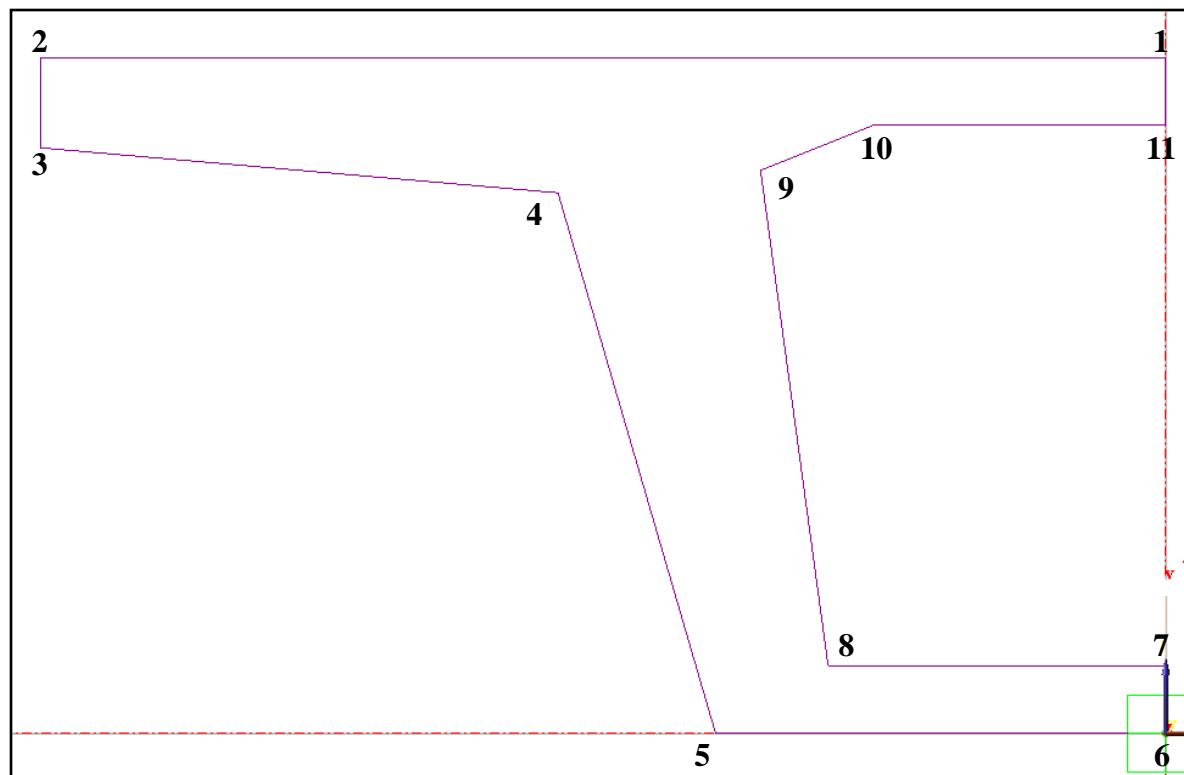


Analysis Control Dialog is automatically activated at startup.

## Step 2.



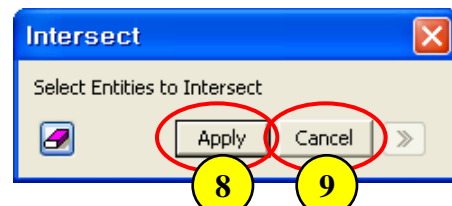
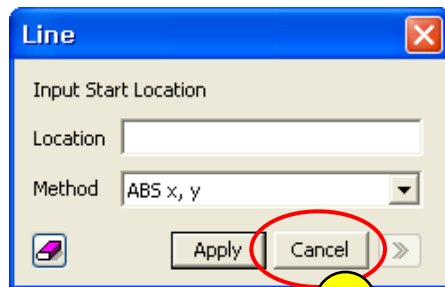
1. **Geometry > Curve > Create on WP > Polyline (Wire)...**
2. **Location :** (0, 300), <-500>, <0, -40>, <230, -20>, <70, -240>, <200>, <0, 30>, <-150>, <-30, 220>, <50, 20>, <130> Ⓜ
3. **Check on “Close”**
4. **Click Right Mouse Button on the Work Window**
5. **Click [Cancel] Button** Ⓜ
6. **Click “Normal View”**




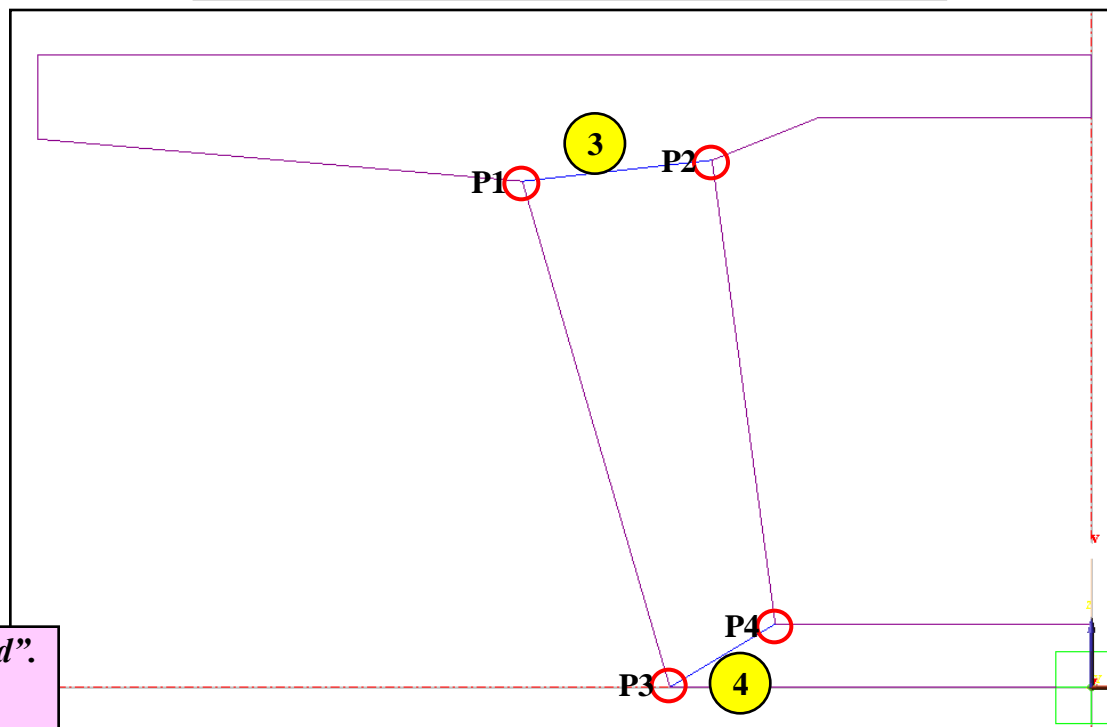
Ⓜ () : “ABS x, y”, <> : “REL dx, dy”  
(-500) same as (-500, 0)

Ⓜ [Esc] as shortcut for [Cancel].

### Step 3.

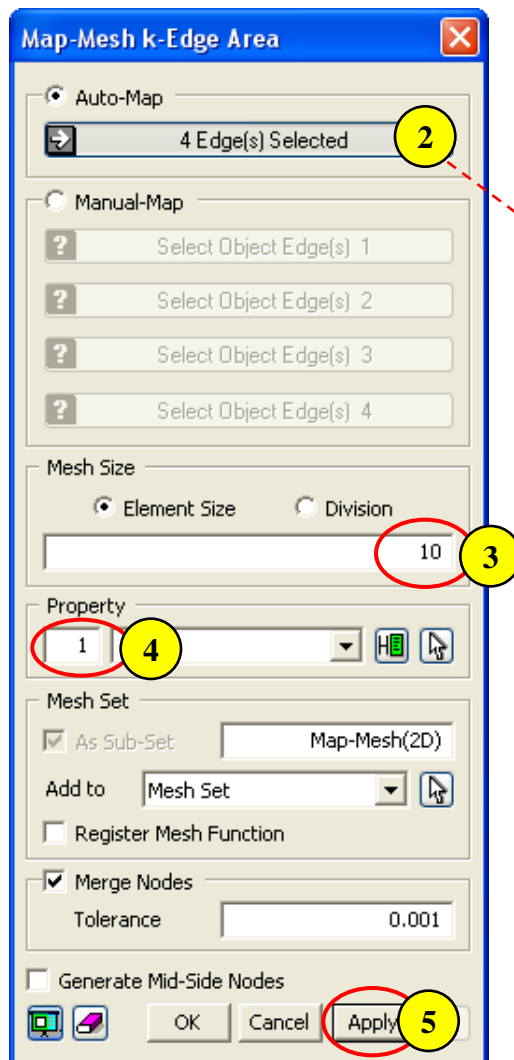


1. Geometry > Curve > Create on WP > Line...
2. Toggle on "Vertex Snap"
3. Select P1 & P2 (See Figure)
4. Select P3 & P4 (See Figure)
5. Click [Cancel] Button
6. Geometry > Curve > Intersect ...
7. Select  "Displayed"
8. Click [Apply] Button
9. Click [Cancel] Button

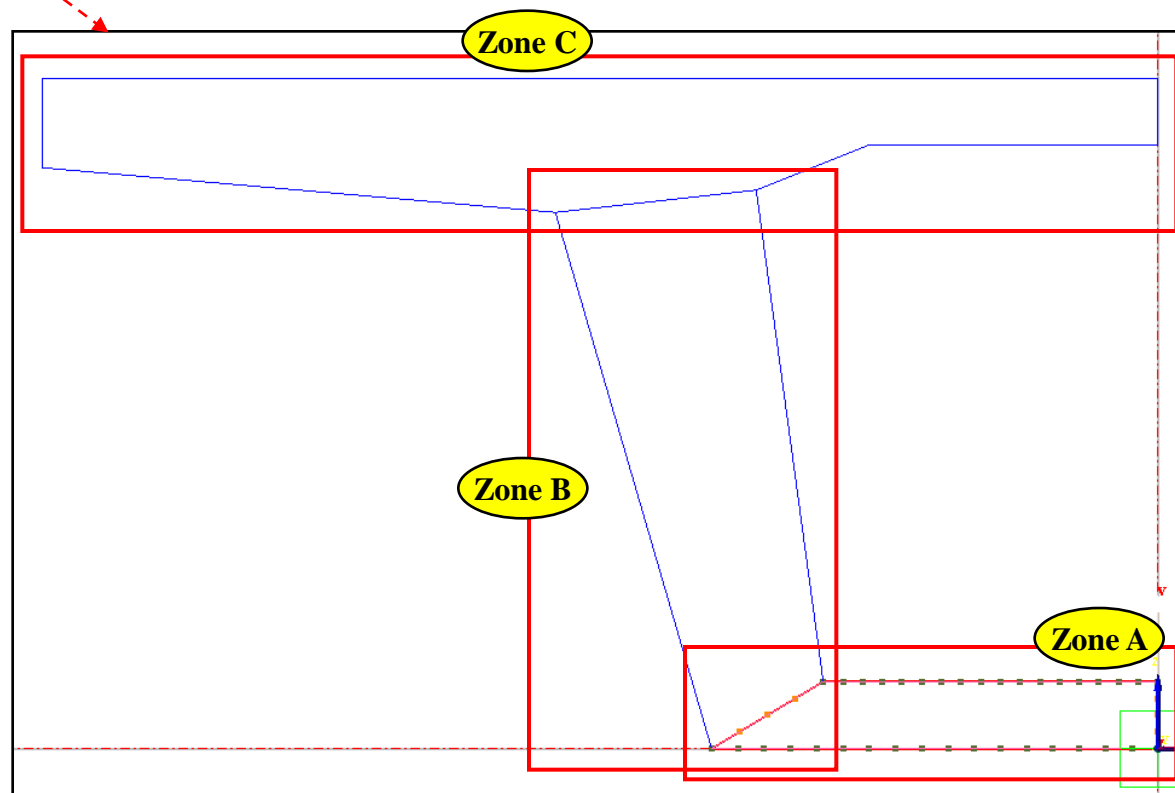


- 🔊 "Ctrl+A" as shortcut for "Select Displayed".
- 🔊 [Enter] as shortcut for [Apply].

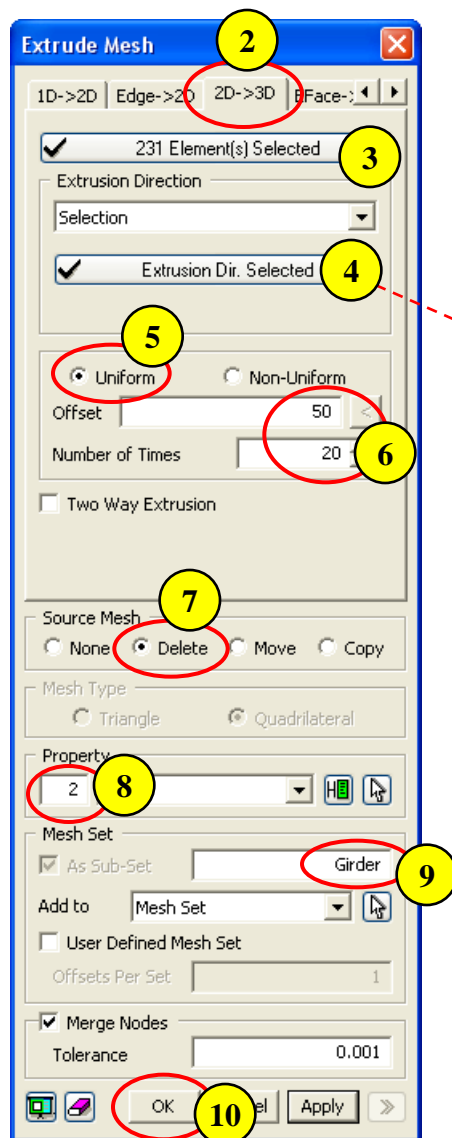
## Step 4.




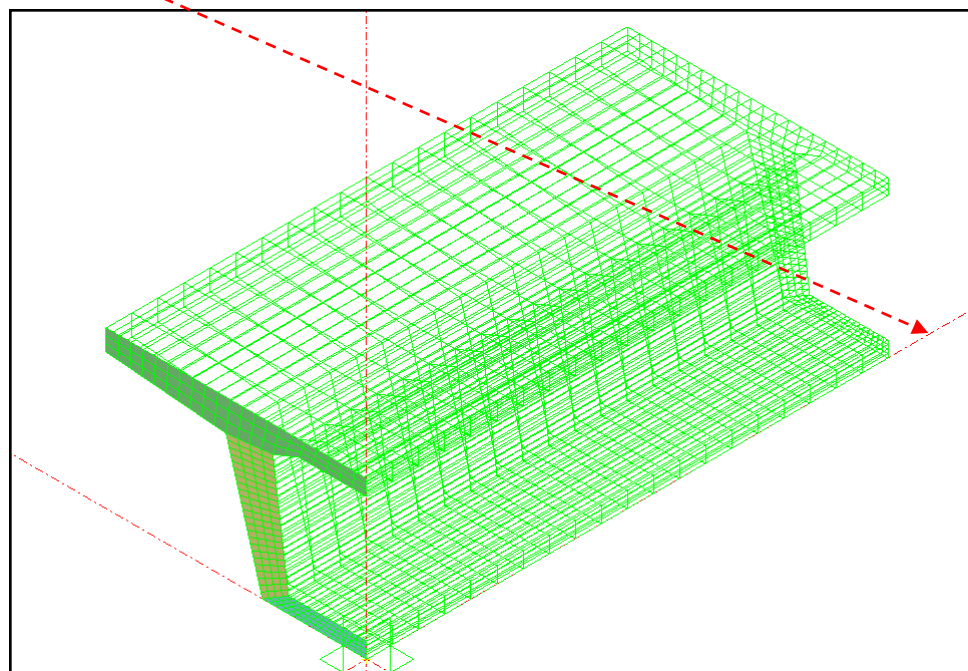
1. Mesh > Map Mesh > k-Edge Area ...
2. Select "Zone A"
3. Mesh Size - Element Size : 10
4. Property (1)
5. Click [Apply] Button
6. Repeat step 2~5 for "Zone B" & "Zone C"



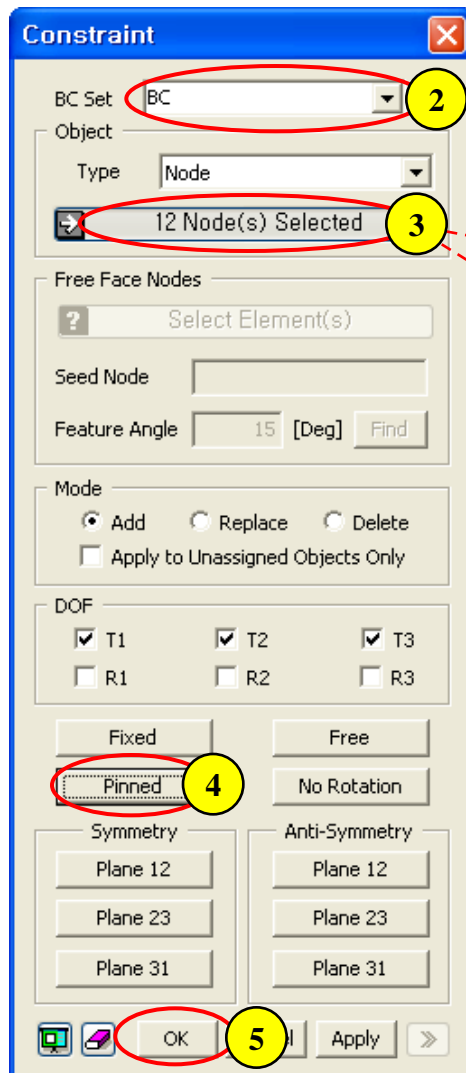
## Step 5.



1. Click "Isometric 1 View"
2. Mesh > Protrude Mesh > Extrude - "2D->3D" tab
3. Select  "Displayed All"
4. Extrusion Direction : Y-Axis
5. Check on "Uniform"
6. Offset : 50 , Number of Times : 20
7. Source Mesh : Delete
8. Property (2)
9. Mesh Set : Girder
10. Click [OK] Button



## Step 6.



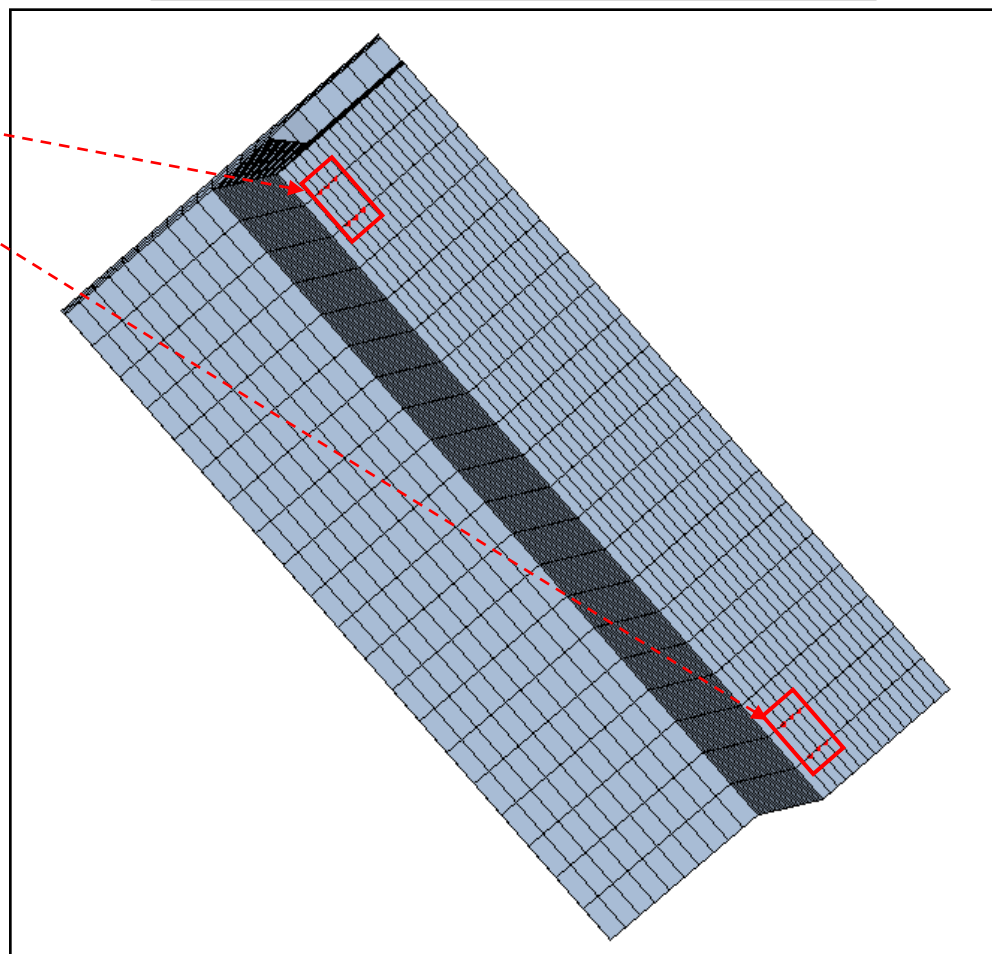
1. Analysis > BC > Constraint ...

2. BC Set : BC

3. Select 12 Nodes of Bottom Face (See Figure)

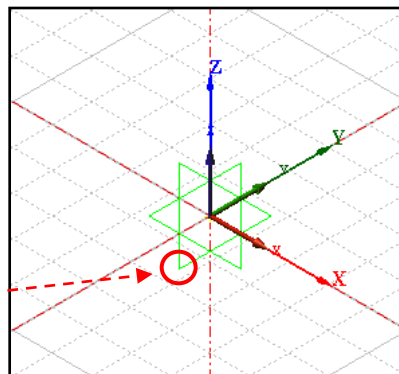
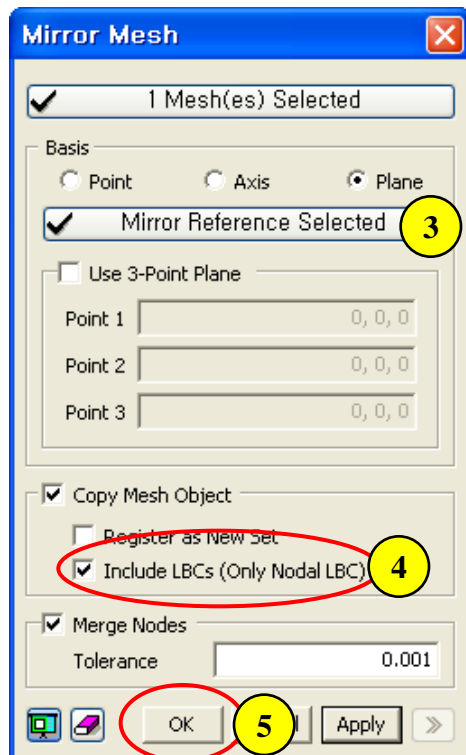
4. Click [Pinned] Button


5. Click [OK] Button

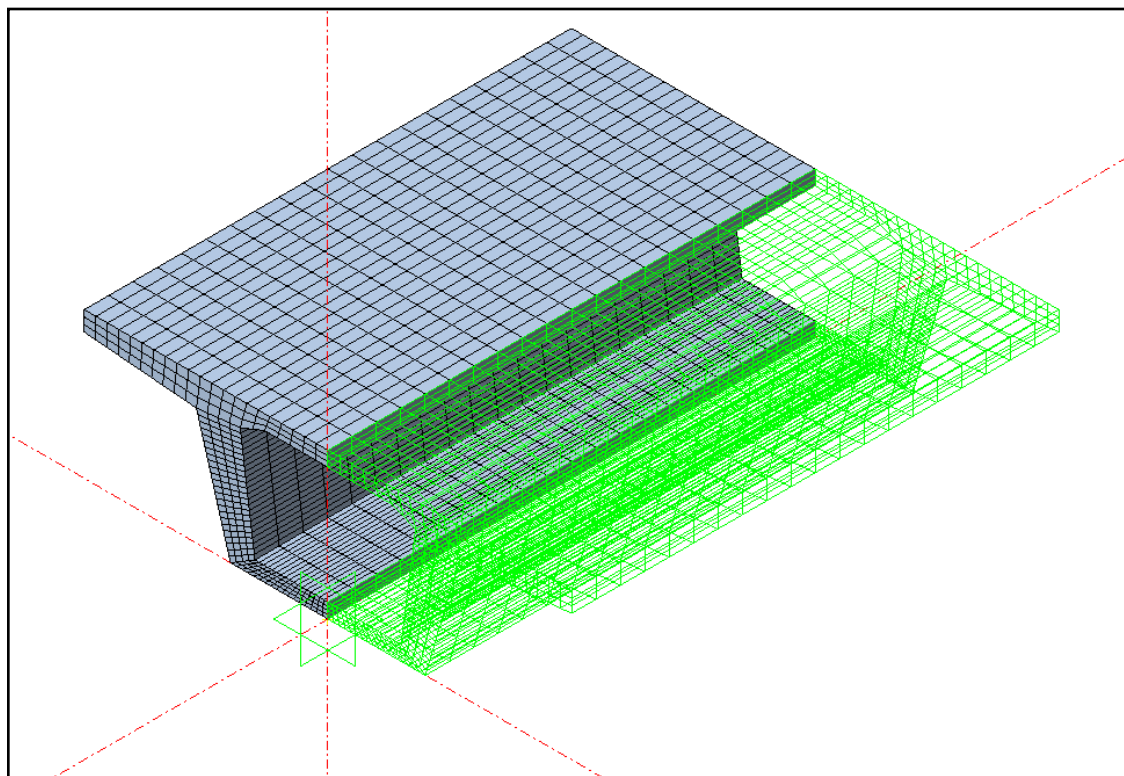




## Step 7.

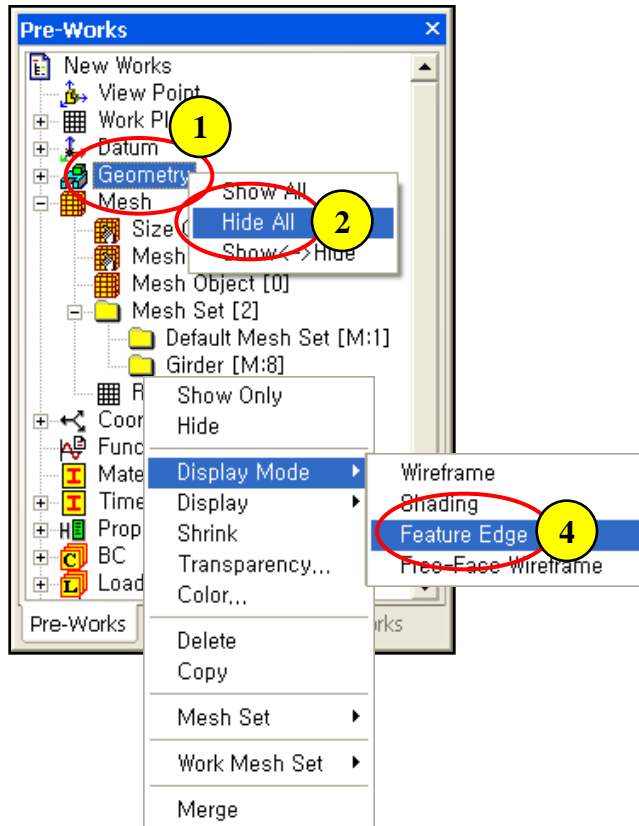


1. Mesh > Transform > Mirror ...
2. Select  "Displayed"
3. Mirror Plane : YZ Plane
4. Check on "Include LBCs"
5. Click [OK] Button





## Step 8.



1. Pre-Works Tree : Geometry

2. Click Right Mouse Button and Select “Hide All”

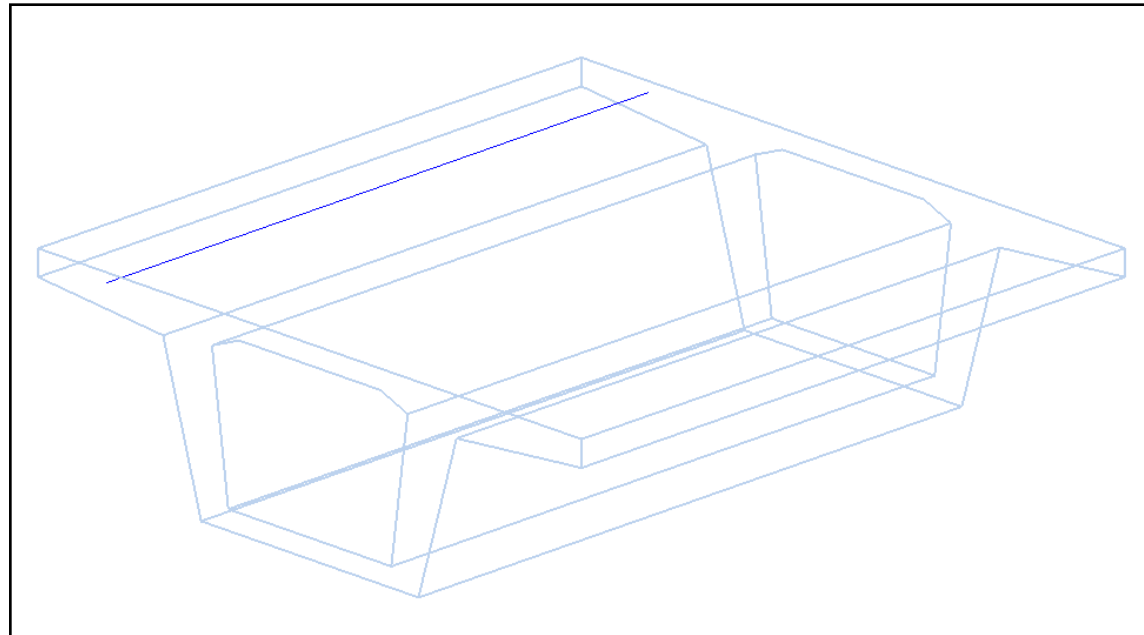
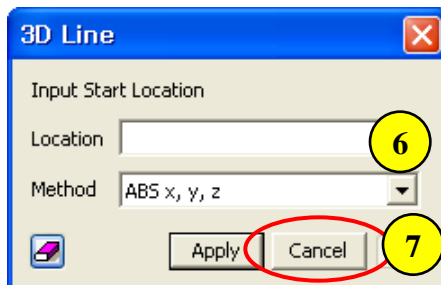
3. Pre-Works Tree : Mesh > Mesh Set > Girder

4. Click Right Mouse Button and Select “Display Mode > Feature Edge”

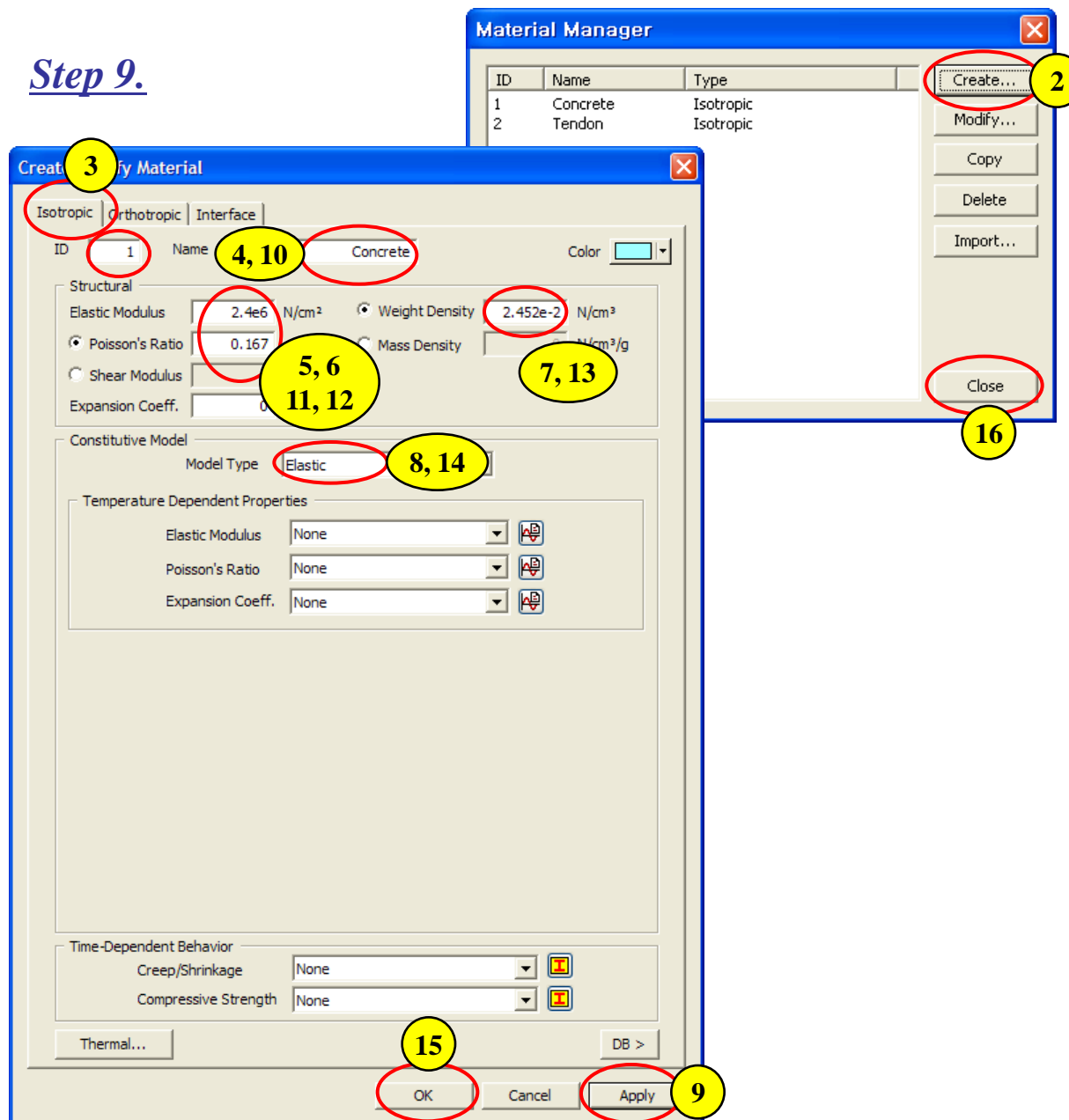
5. Geometry > Curve > Create 3D > Line...

6. Location : (-375, 0, 285), <0, 1000, 0>

7. Click [Cancel] Button

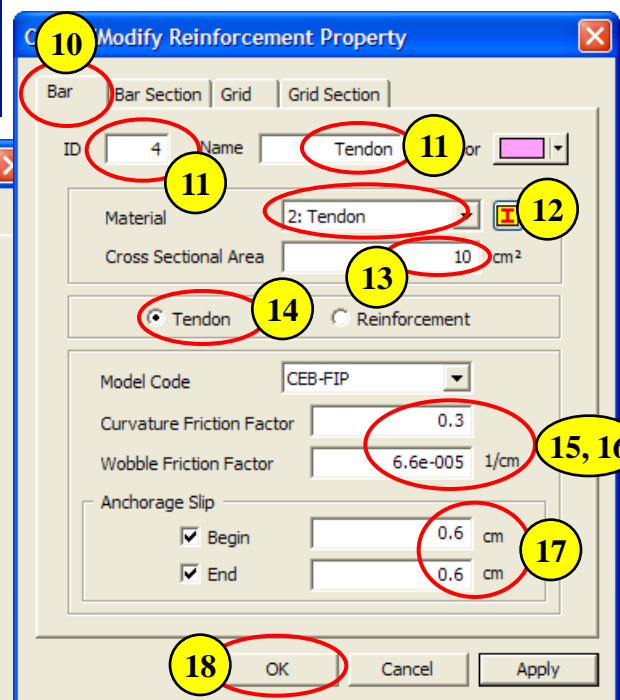
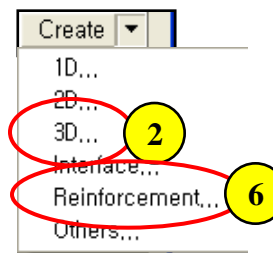
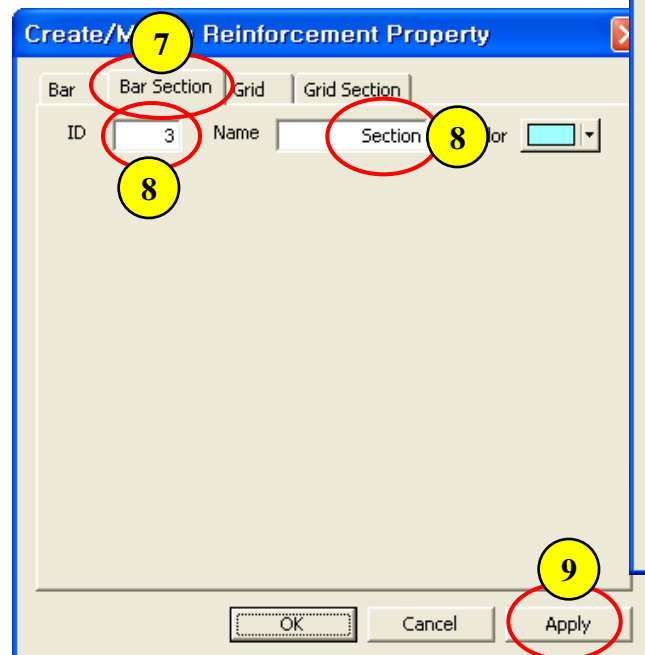
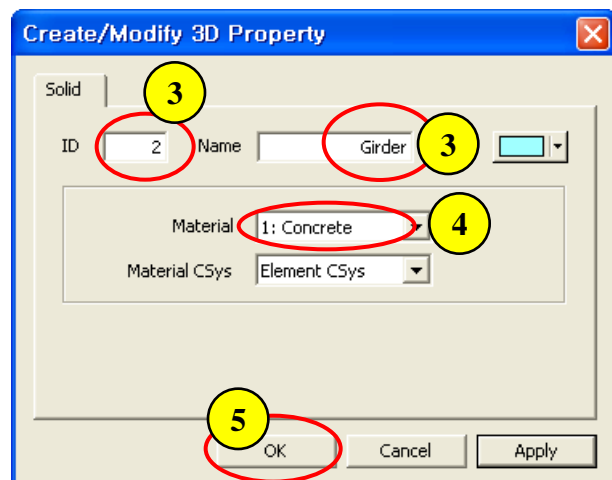


## Step 9.



1. Analysis > Material...
2. Click [Create...] Button
3. Select "Isotropic" tab
4. ID : 1 , Name : Concrete
5. Elastic Modulus :  $2.4e6 \text{ N/cm}^2$
6. Poisson's Ratio : 0.167
7. Weight Density :  $2.452e-2 \text{ N/cm}^3$
8. Model Type : Elastic
9. Click [Apply] Button
10. ID : 2 , Name : Tendon
11. Elastic Modulus :  $1.96e7 \text{ N/cm}^2$
12. Poisson's Ratio : 0.3
13. Weight Density :  $7.698e-2 \text{ N/cm}^3$
14. Model Type : Elastic
15. Click [OK] Button
16. Click [Close] Button

## Step 10.



1. Analysis > Property ...
2. Create 3D ...
3. ID : 2 , Name : Girder
4. Select "1:Concrete" for Material
5. Click [OK] Button
6. Create Reinforcement ...
7. Select "Bar Section" tab
8. ID : 3 , Name : Section
9. Click [Apply] Button
10. Select "Bar" tab
11. ID : 4 , Name : Tendon
12. Select "2: Tendon" for Material
13. Cross Section Area : 10 cm<sup>2</sup>
14. Select "Tendon"
15. Curvature Friction Factor : 0.3
16. Wobble Friction Factor : 6.6e-5
17. Anchorage Slip : 0.6 cm
18. Click [OK] Button
19. Click [Close] Button

## Step 11.

1. Mesh > Auto Mesh > Edge ...

2. Select Edge marked by "O" (See Figure)

3. Seeding Method - Number of Divisions : 1

4. Property : (3: Section)

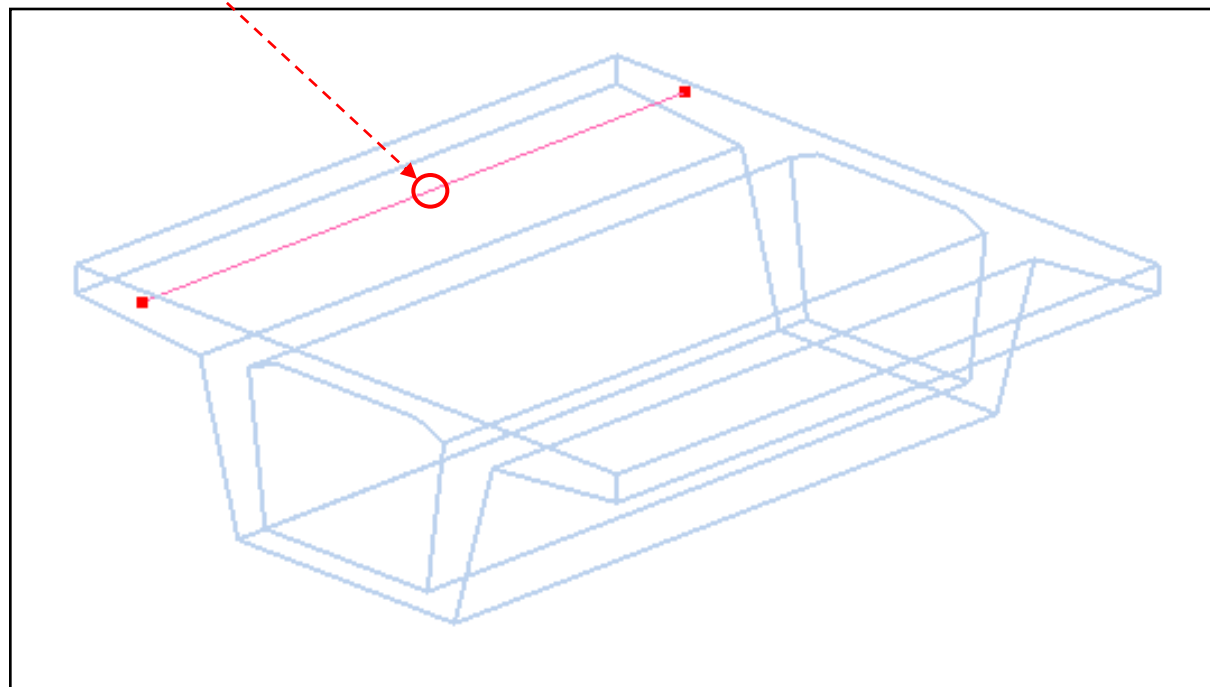
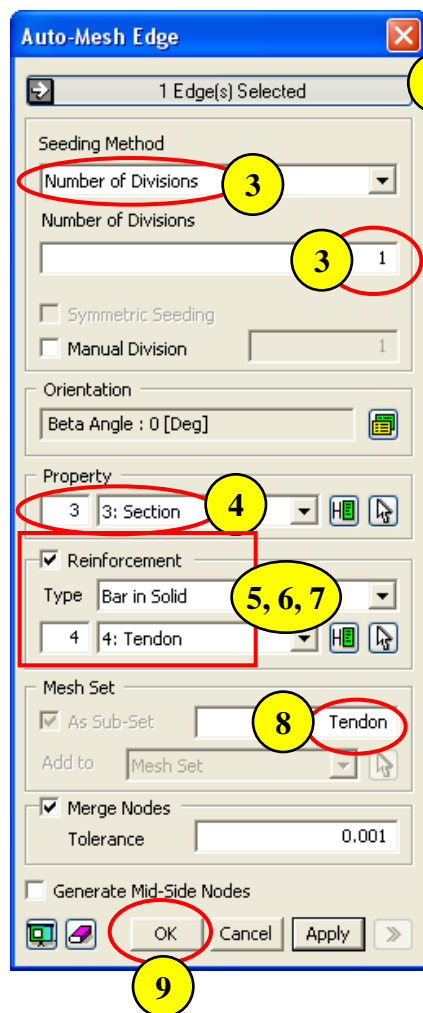
5. Check on "Reinforcement"

6. Type : Bar in Solid

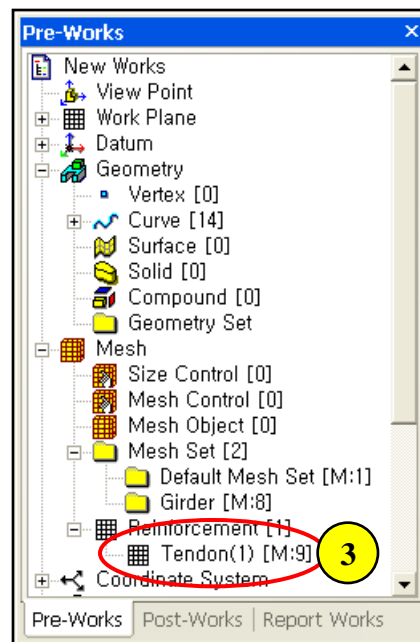
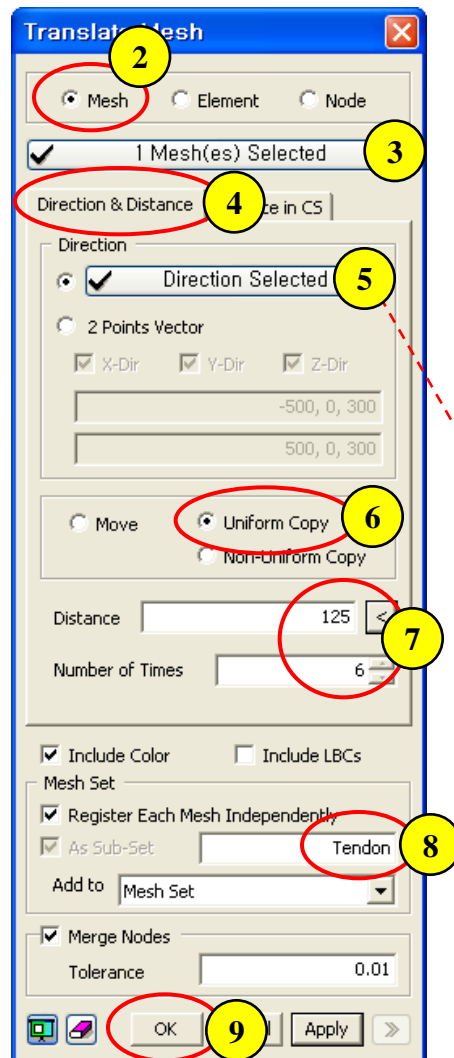
7. Property : (4: Tendon)

8. Mesh Set : Tendon

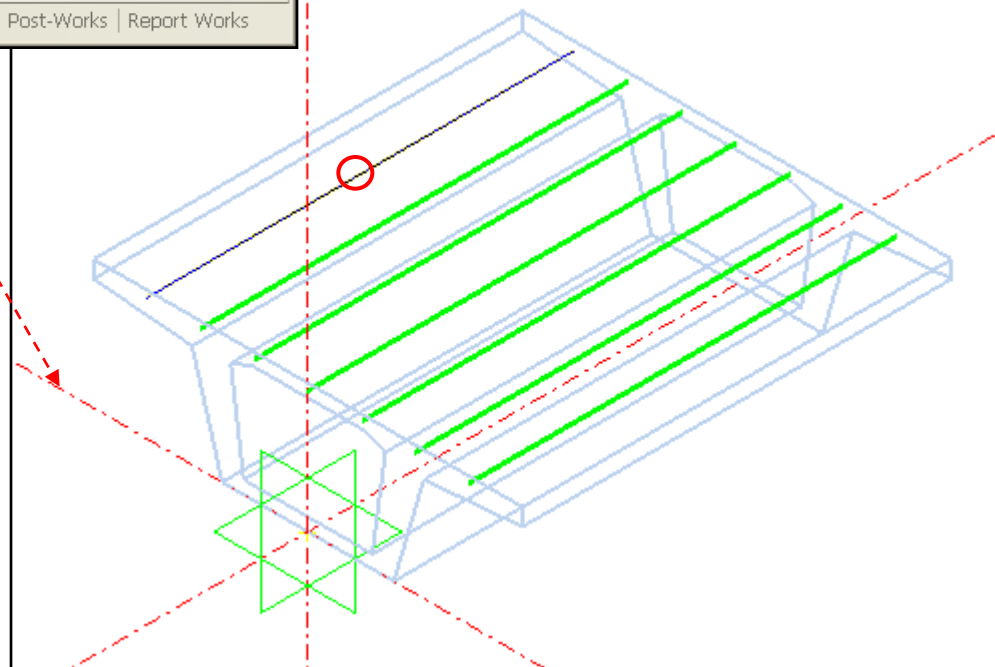
9. Click [OK] Button



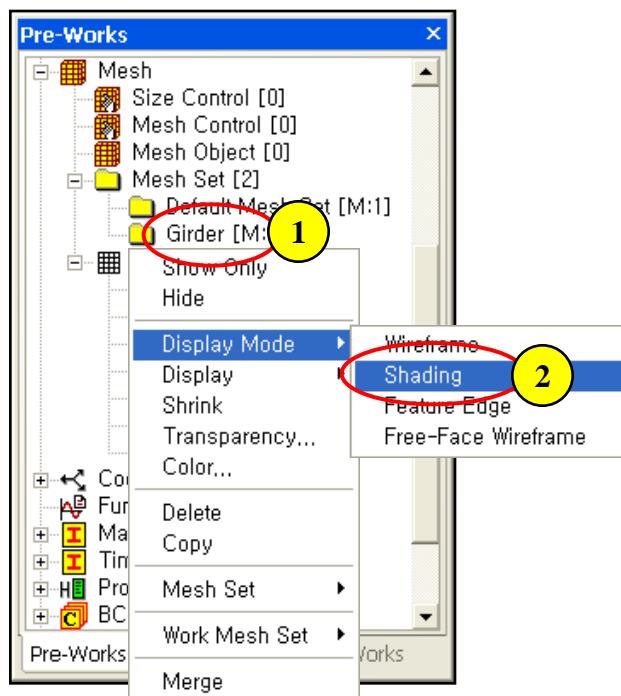
## Step 12.



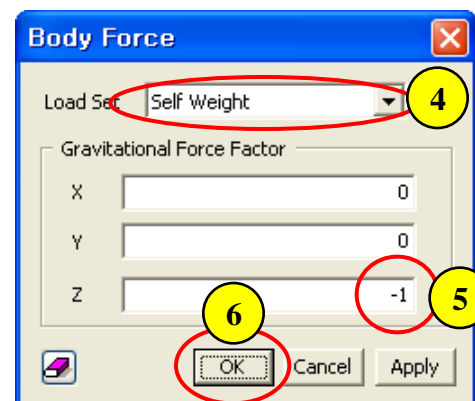
1. Mesh > Transform > Translate ...
2. Check on "Mesh"
3. Select "Reinforcement > Tendon(1)" Mesh Set in Pre-Works Tree (See Figure)
4. Select "Direction & Distance" tab
5. Direction "X-Axis"
6. Select "Uniform Copy"
7. Distance : 125 , Number of Times : 6
8. Mesh Set : Tendon
9. Click [OK] Button



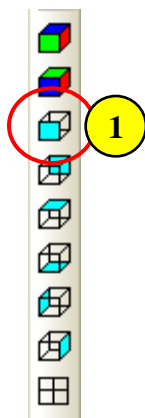
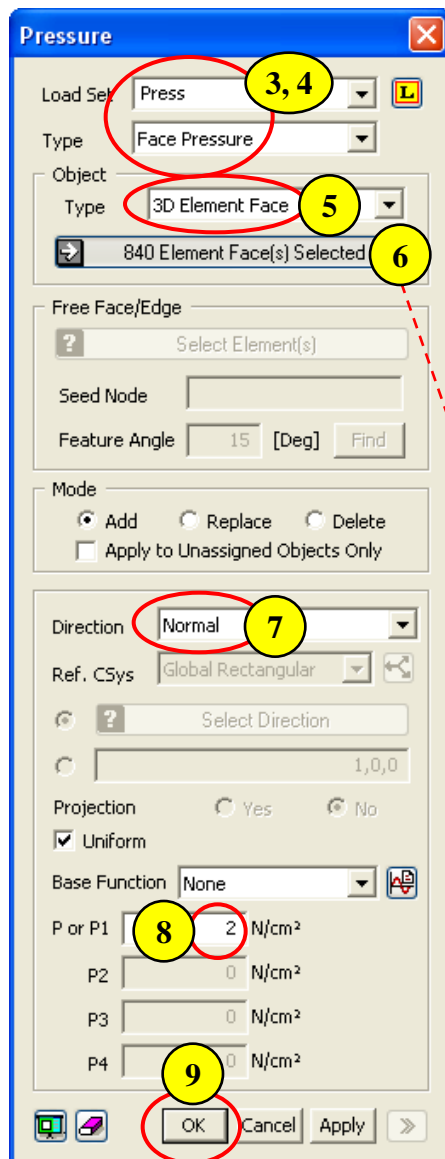
### Step 13.



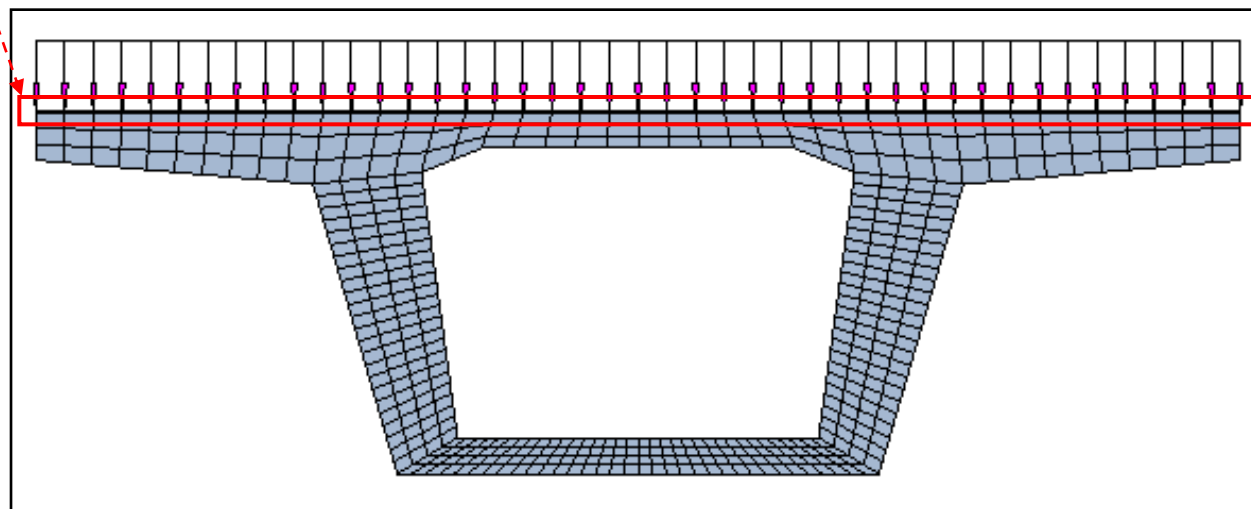
1. Pre-Works Tree : Mesh > Mesh Set > Girder
2. Click Right Mouse Button and Select "Display Mode > Shading"
3. Analysis > Load > Body Force ...
4. Load Set : Self Weight
5. Gravitational Force Factor : Z (-1)
6. Click [OK] Button



## Step 14.

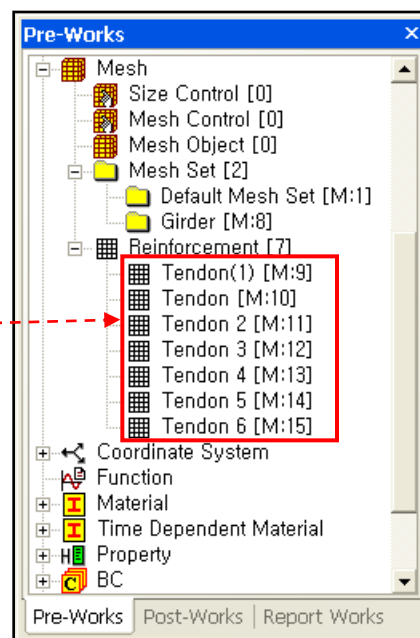
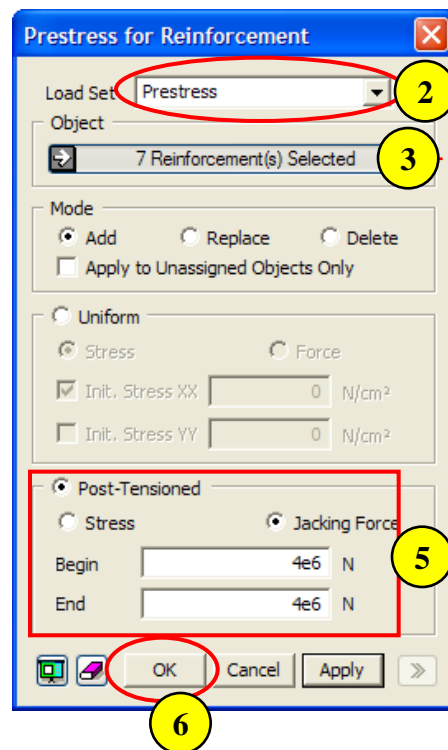


1. Click "Front View"
2. Analysis > Load > Pressure ...
3. Load Set : Press
4. Type : Face Pressure
5. Object Type : 3D Element Face
6. Select Top Element Faces of Girder (See Figure)
7. Direction : Normal
8. P or P1 : 2 N/cm<sup>2</sup>
9. Click [OK] Button

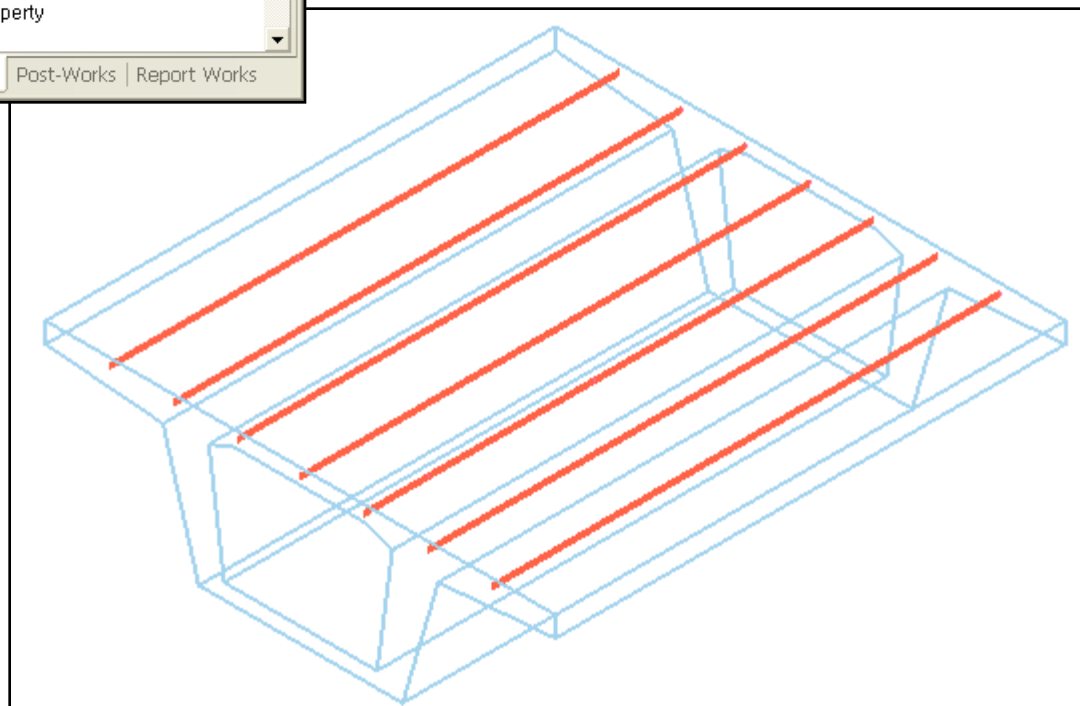




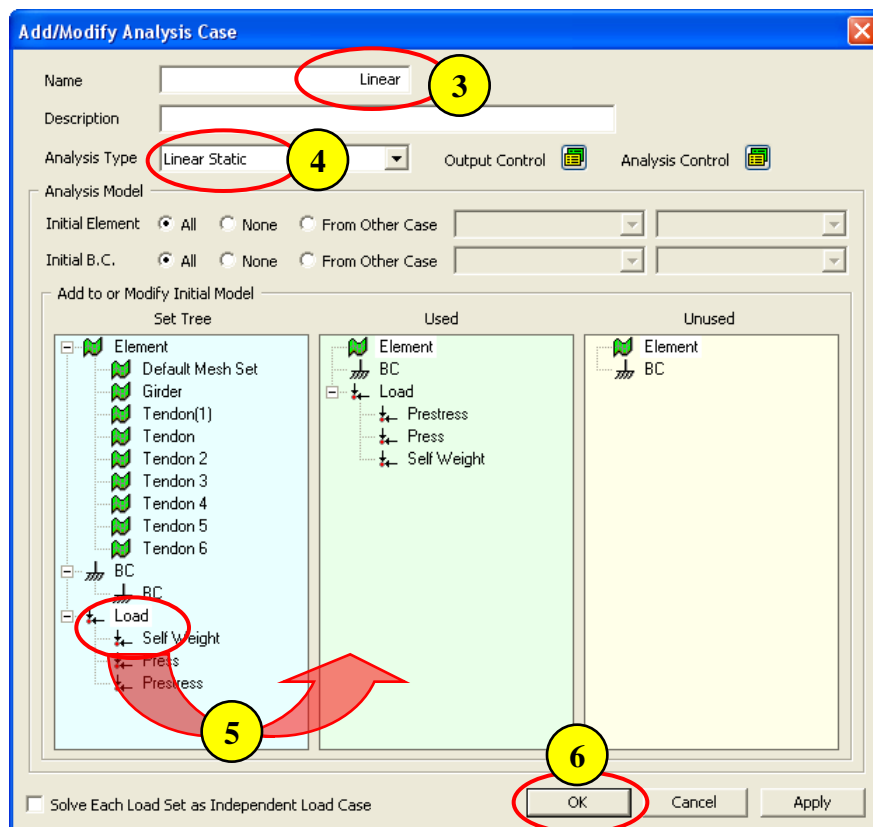
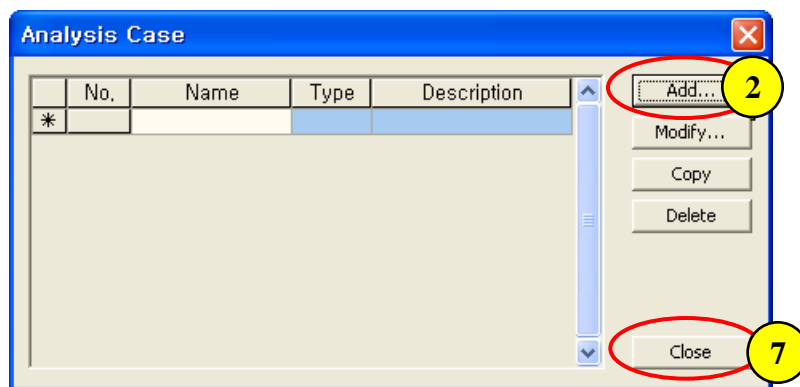
## Step 15.



1. Analysis > Load > Prestress for Reinforcement ...
2. Load Set : Prestress
3. Select 7 Reinforcements in Pre-Works Tree (See Figure)
4. Select "Post Tensioned (Jacking Force)"
5. Begin & End : 4e6 N
6. Click [OK] Button



## Step 16.



1. Analysis > Analysis Case ...

2. Click [Add] Button

3. Name : Linear

4. Analysis Type : Linear Static

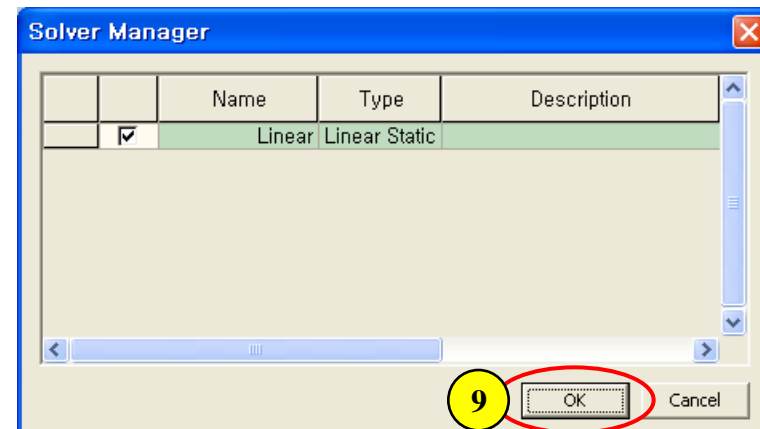
5. Drag & Drop "Load" to "Used" Window

6. Click [OK] Button

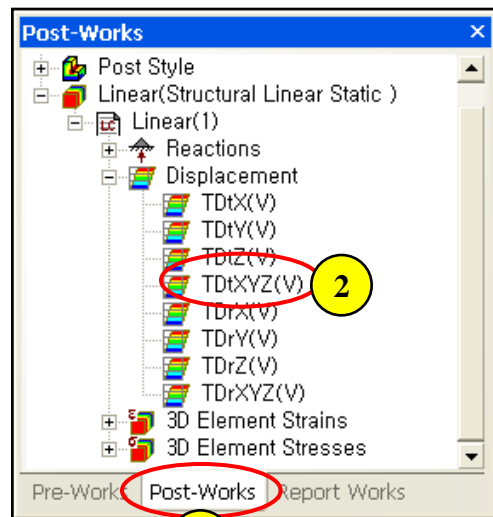
7. Click [Close] Button

8. Analysis > Solve ...

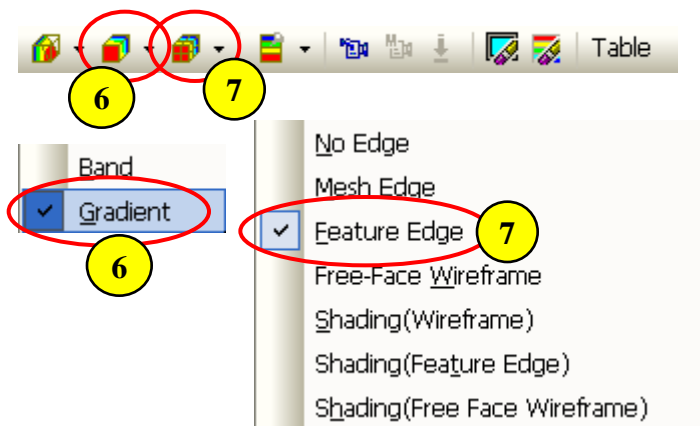
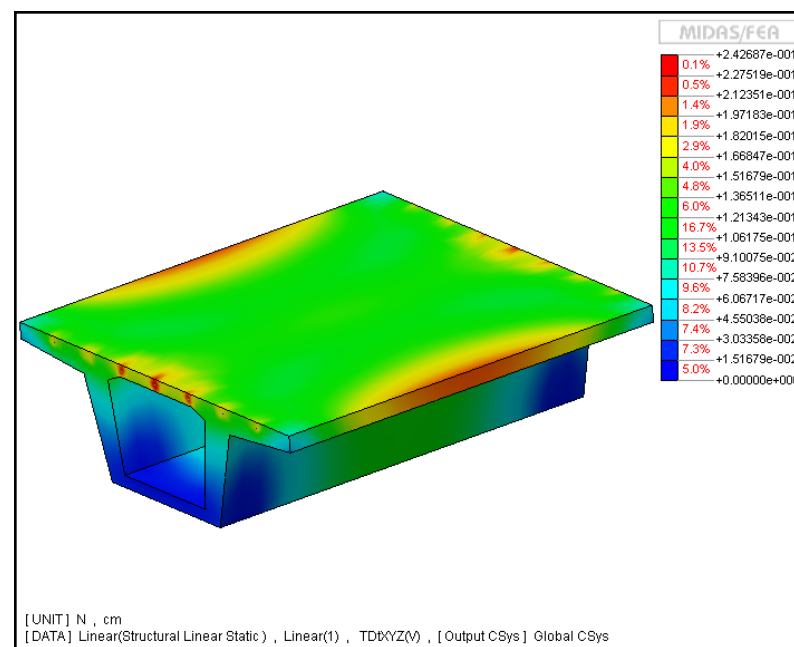
9. Click [OK] Button



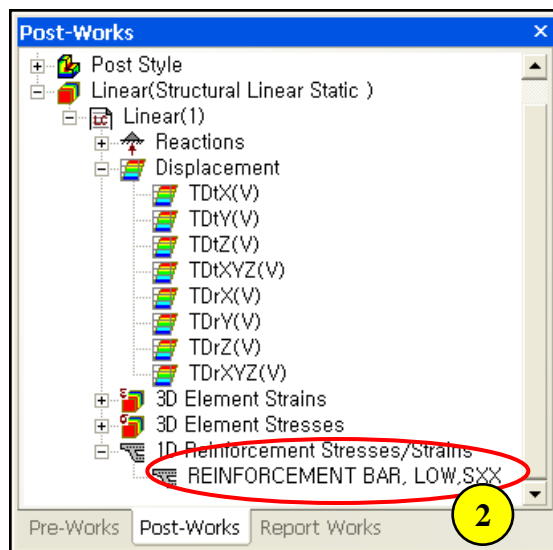
# Step 17.



1. Post-Works Tree : Linear (Structural Linear Static) > Linear(1) > Displacement
2. Double Click “TDrXYZ(V)”
3. Click “Sens.” Button
4. Select “Undeformed” for Mesh Shape (See Figure)
5. Click “Post Style” Toolbar
6. Select “Gradient” for Contour Type
7. Select “Feature Edge” for Edge Type



## Step 18.



1. Post-Works Tree : Linear (Structural Linear Static) > Linear(1)  
> 1D Reinforcement Stresses/Strains

2. Double Click “REINFORCEMENT BAR, LOW,SXX”

3. Property Window : Contour

4. No Result Entity : Feature Edge

5. Click [Apply] Button

