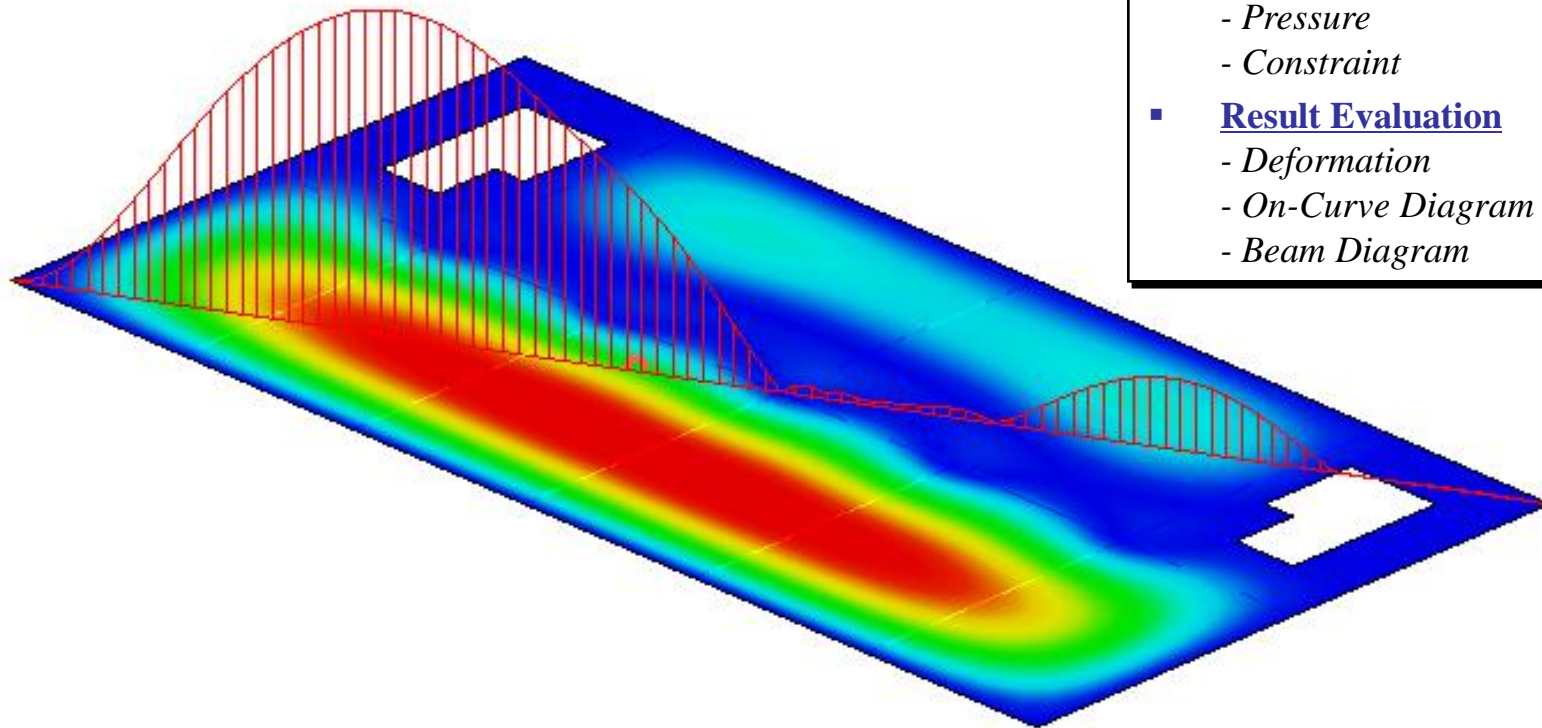


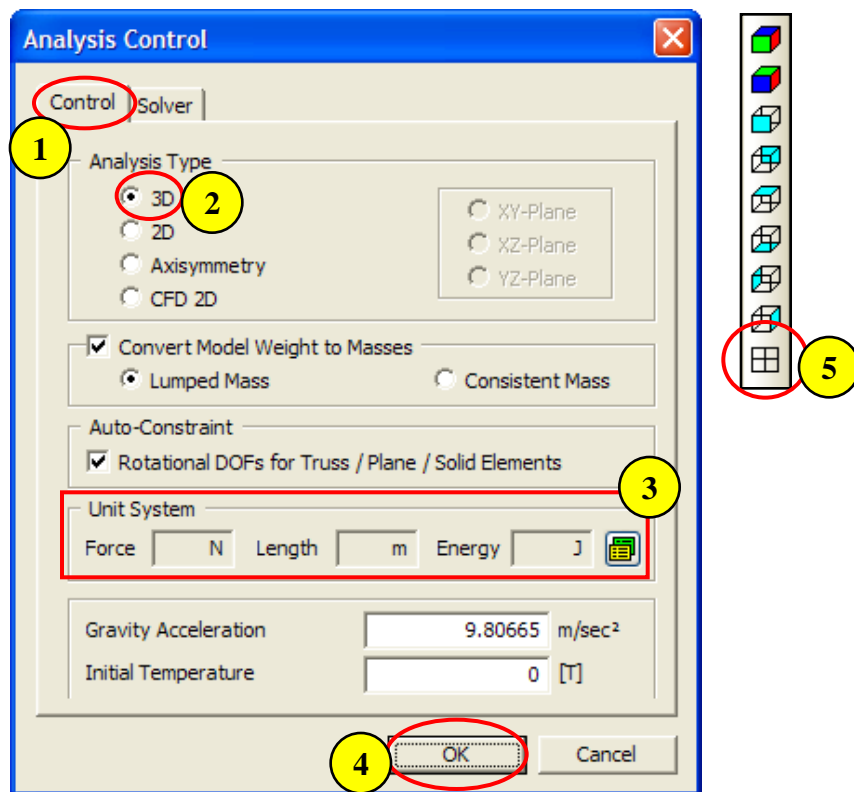
LS-6. Concrete Floor

Overview

- 3-D Linear Static Analysis
- Model
 - Unit : N, m
 - Isotropic Elastic Material
 - Plate and Beam Elements
- Load & Boundary Condition
 - Body Force
 - Pressure
 - Constraint
- Result Evaluation
 - Deformation
 - On-Curve Diagram
 - Beam Diagram



Step 1.



1. Analysis > Analysis Control – Control tab

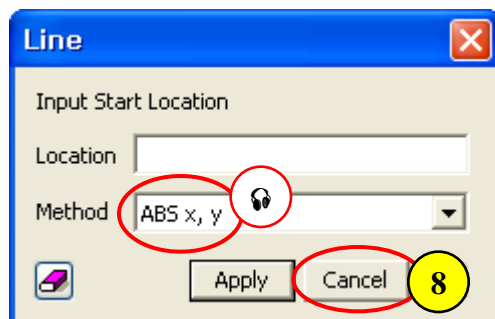
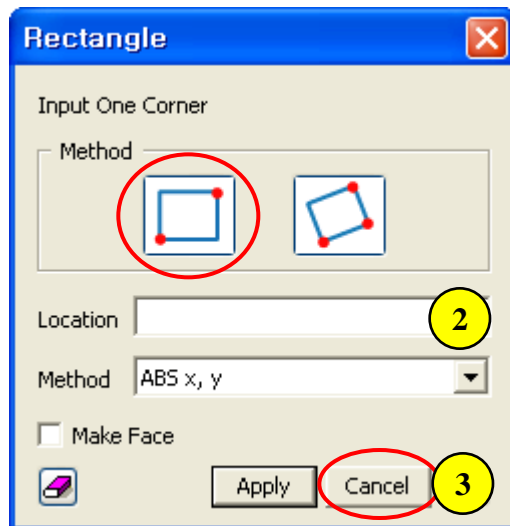
2. Analysis Type : 3D

3. Unit : N, m

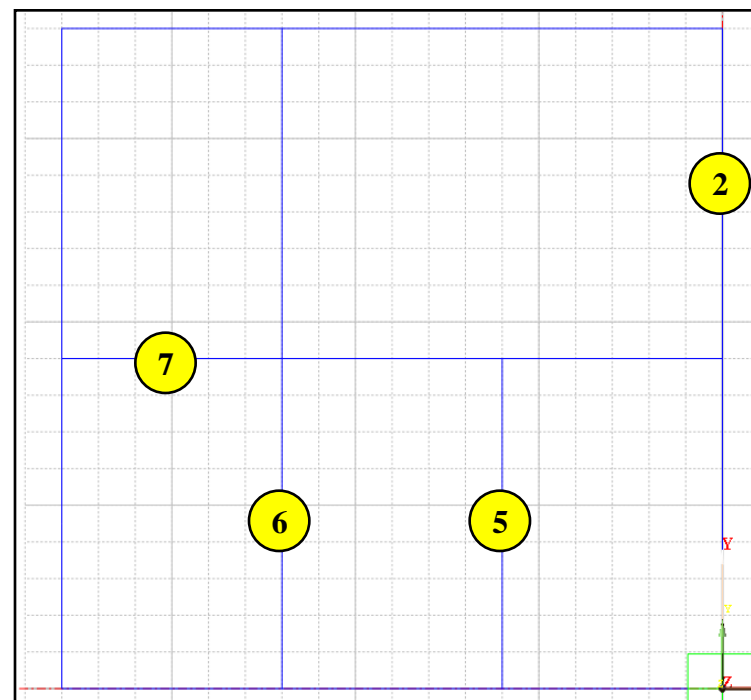
4. Click [OK] Button

5. Click “Normal View”

Step 2.



1. *Geometry > Curve > Create on WP > Rectangle (Wire)...*
2. *Location : (0) , <-18, 18> 🗣*
3. *Click [Cancel] Button 🗣*
4. *Geometry > Curve > Create on WP > Line...*
5. *L1 : SL(-6) , EL<0, 9> 🗣*
6. *L2 : SL(-12) , EL<0, 18> 🗣*
7. *L3 : SL(0, 9) , EL<-18> 🗣*
8. *Click [Cancel] Button*



🗣 [Esc] as shortcut for [Cancel].

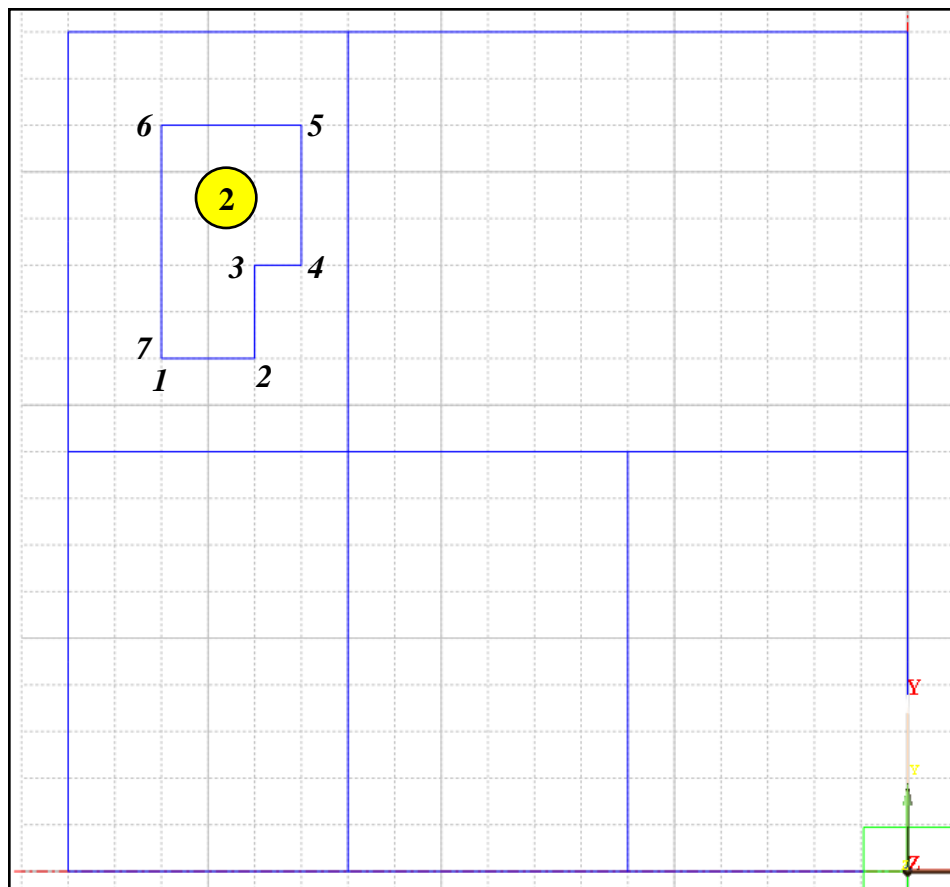
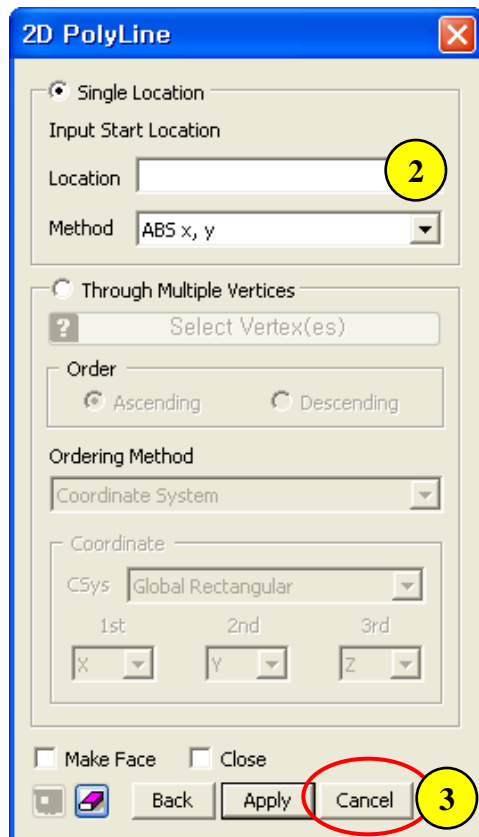
🗣 (): "ABS x, y", <>: "REL dx, dy"
<-18> same as <-18, 0>

Step 3.

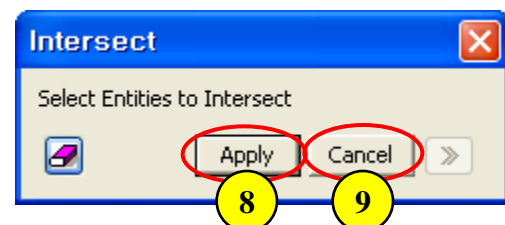
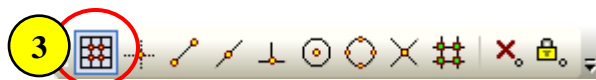
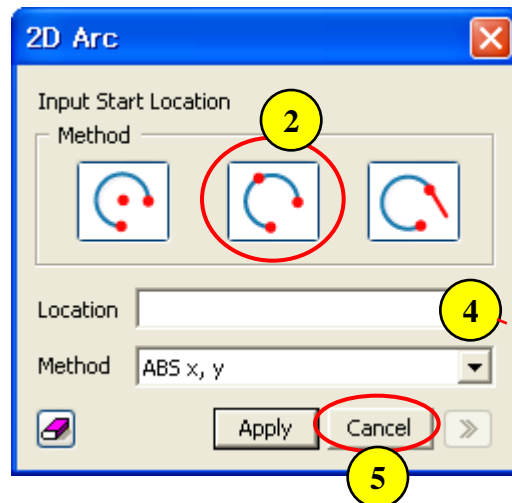
1. *Geometry > Curve > Create on WP > Polyline (Wire)...*


2. *Location : (-16, 11) , <2> , <0, 2> , <1> , <0, 3> , <-3> , <0, -5>*

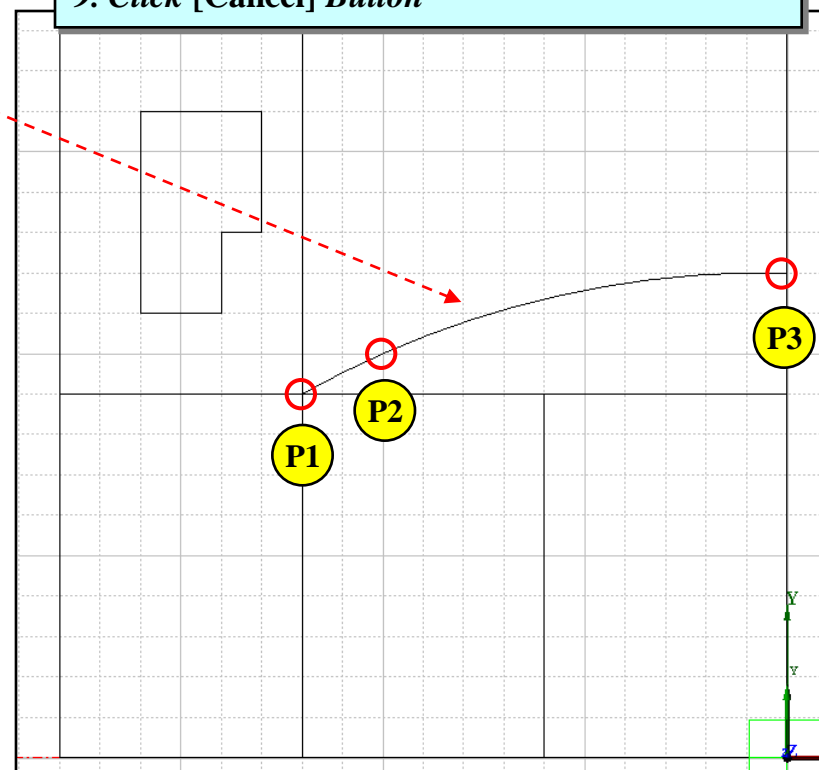
3. *Click [Cancel] Button*



Step 4.



1. Geometry > Curve > Create on WP > Arc...
2. Select "3-Point Arc"
3. Toggle on "Grid Snap"
4. Start (P1) , Second (P2) , End (P3)
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 5.

1. Mesh > Auto Mesh > Planar Area...

2. Select  "Displayed"

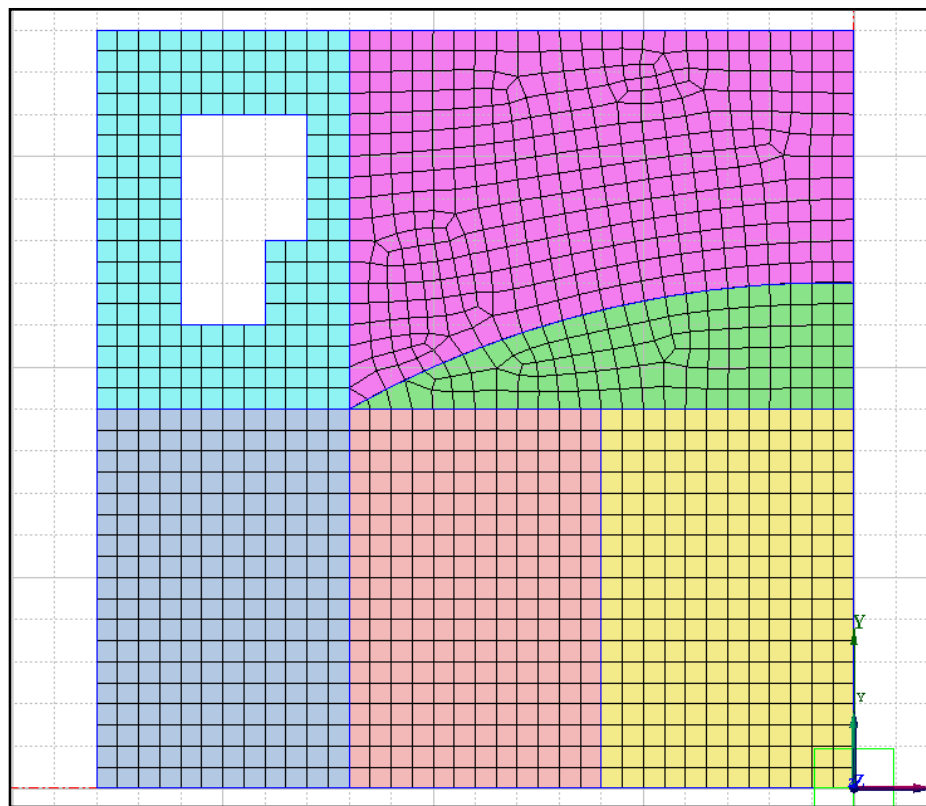
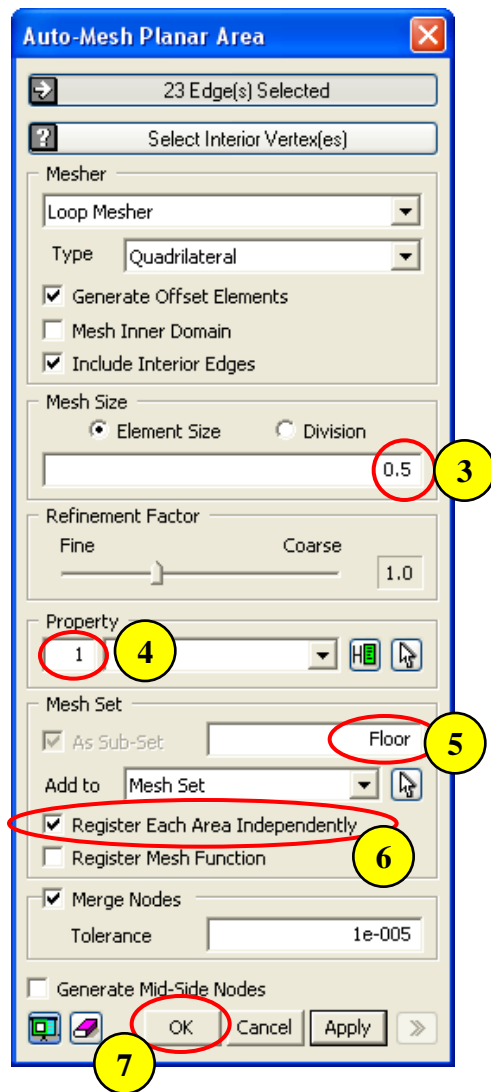
3. Mesh Size - Element Size : 0.5

4. Property : 1

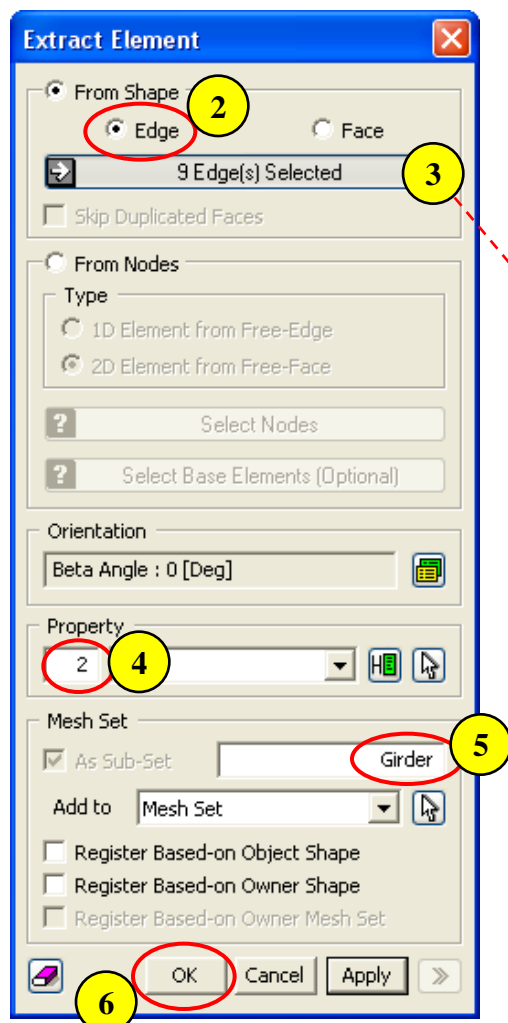
5. Mesh Set : Floor

6. Check on "Register Each Area Independently"

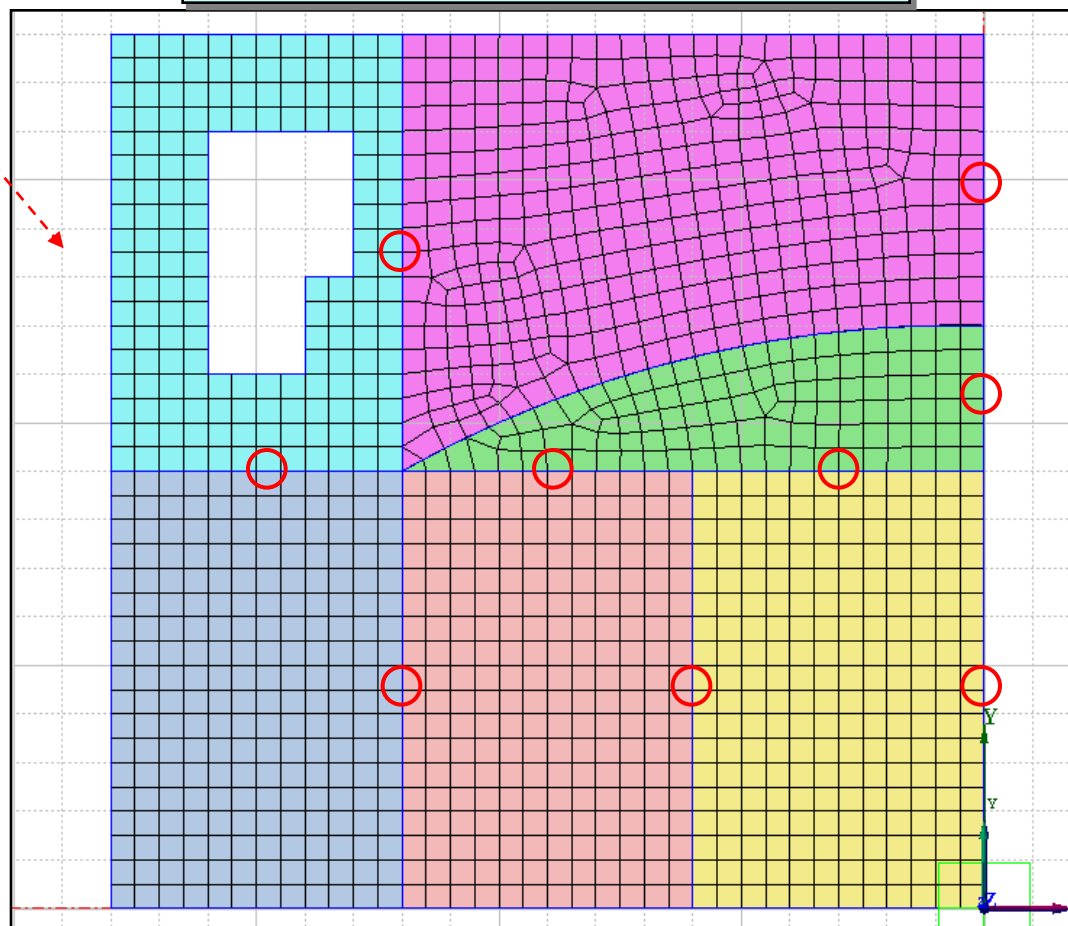
7. Click [OK] Button



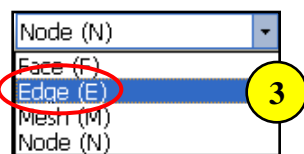
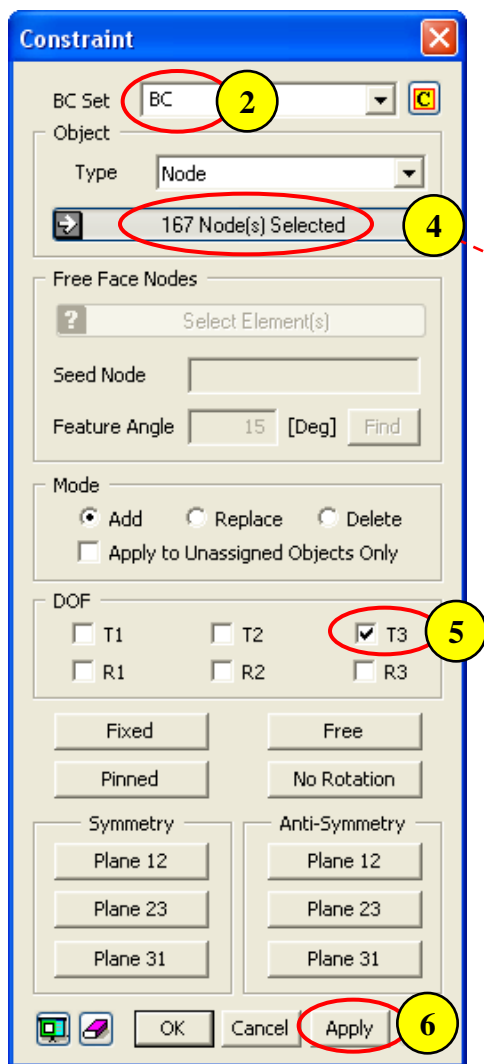
Step 6.



1. Mesh > Element > Extract Element...
2. From Shape - Edge
3. Select 9 Edges Marked by "O" (See Figure)
4. Property : 2
5. Mesh Set : Girder
6. Click [OK] Button



Step 7.



1. Analysis > BC > Constraint ...

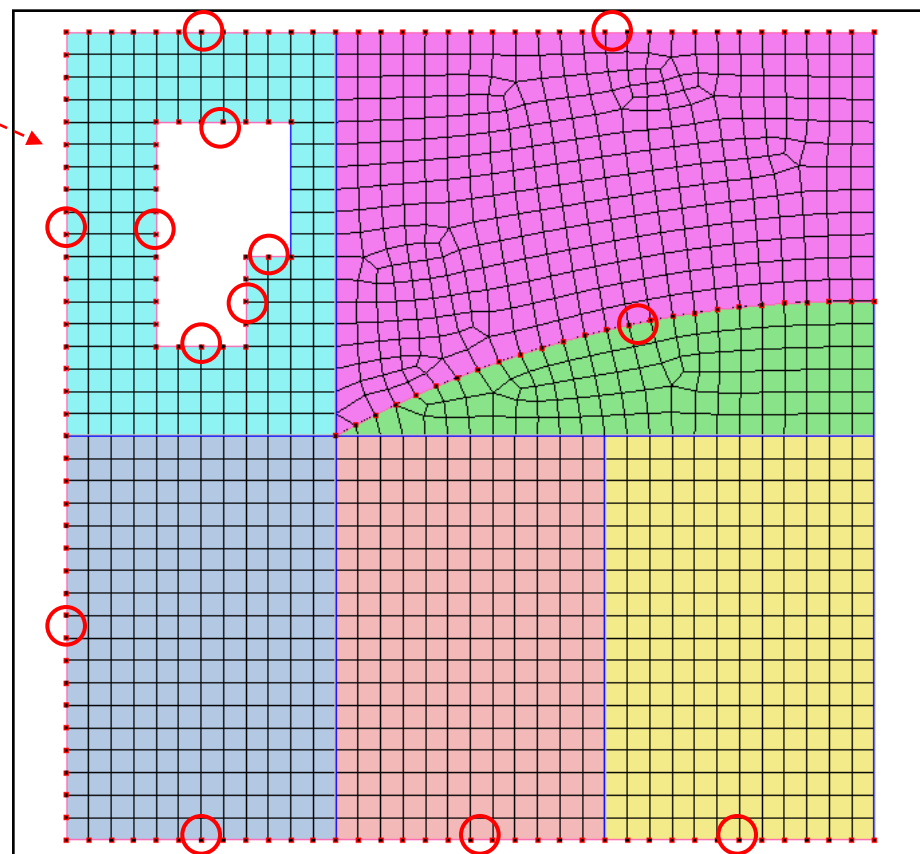
2. BC Set : BC

3. Change Selection Filter to "Edge (E)"

4. Select 13 Edges Marked by "O" (See Figure) *

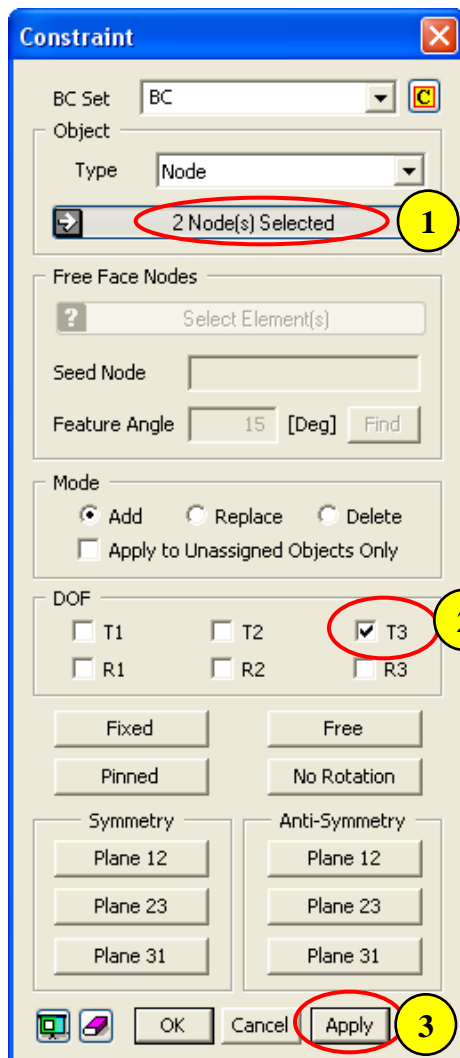
5. Check on "T3"

6. Click [Apply] Button

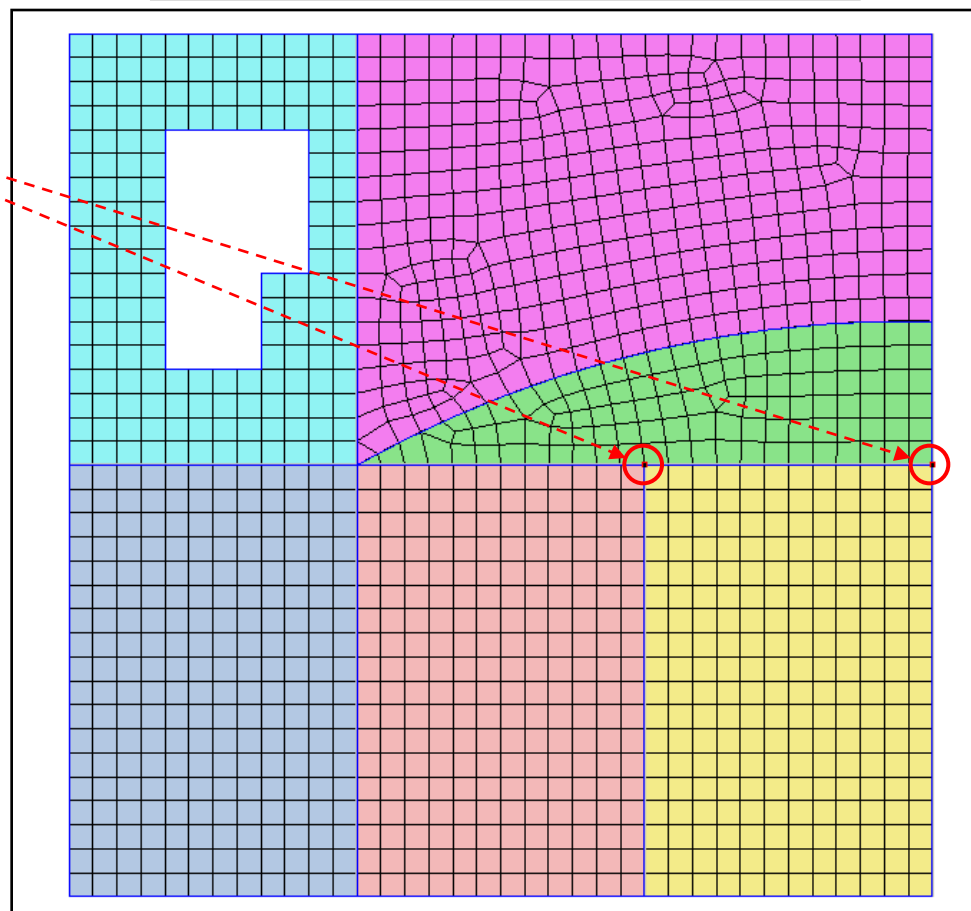


* Nodes generated on the selected edges are selected.

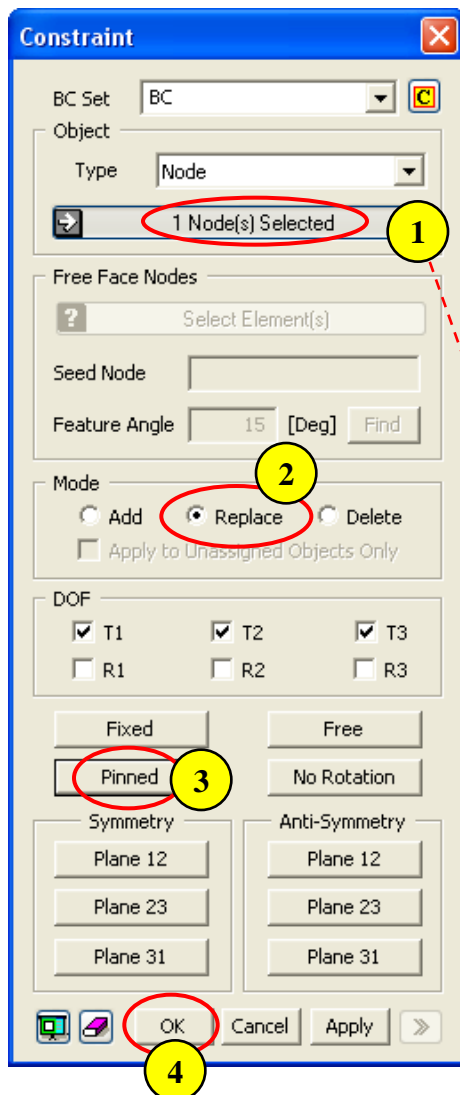
Step 8.



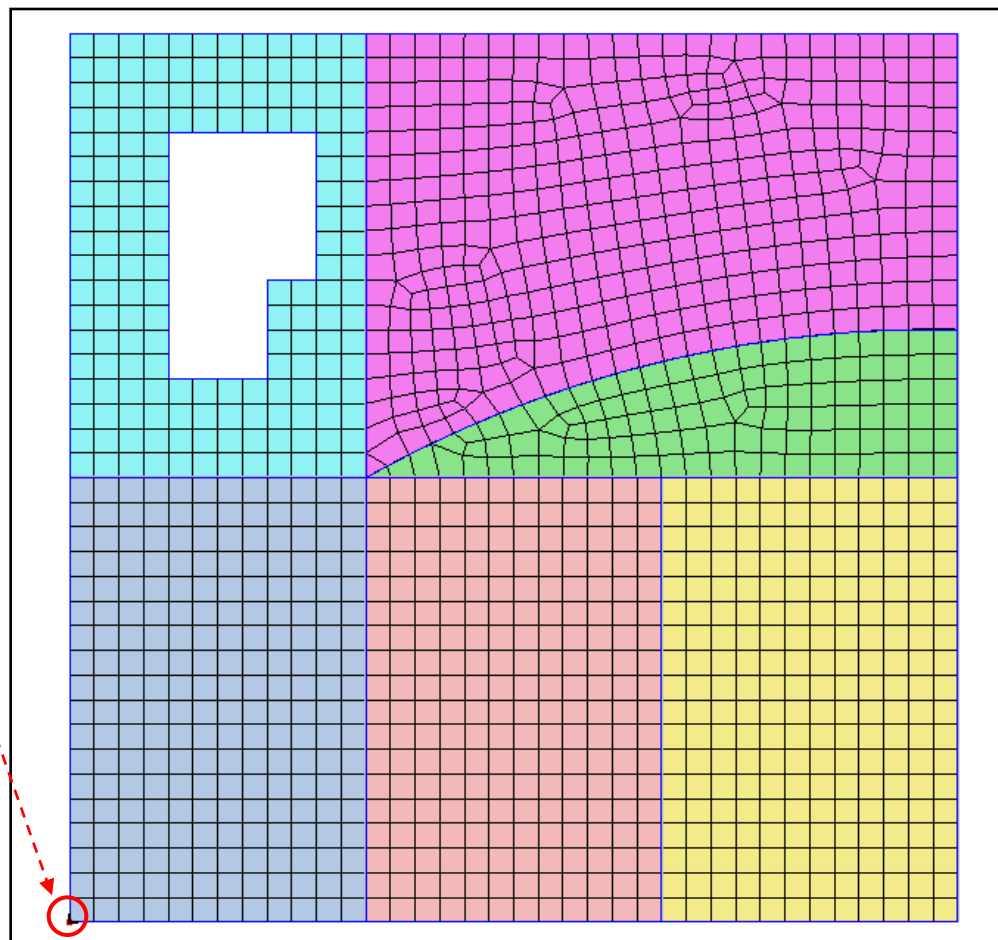
1. Select 2 Nodes Marked by "O" (See Figure)
2. Check on "T3"
3. Click [Apply] Button



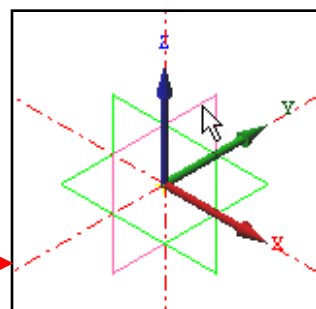
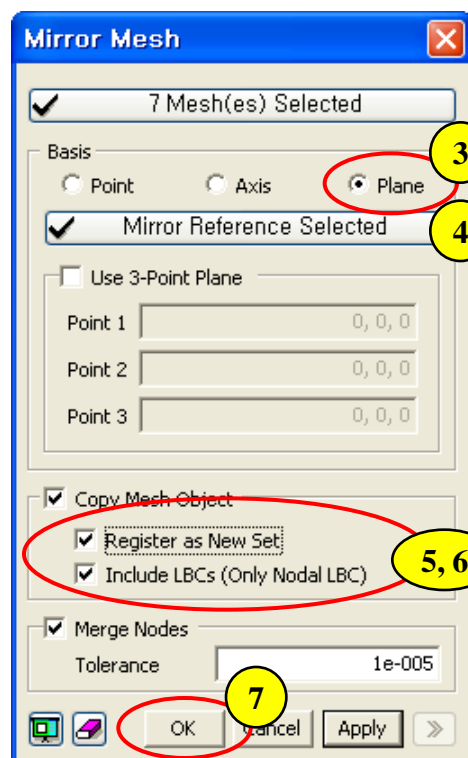
Step 9.



1. Select Node Marked by “O” (See Figure)
2. Mode : Replace
3. Click [Pinned] Button
4. Click [OK] Button

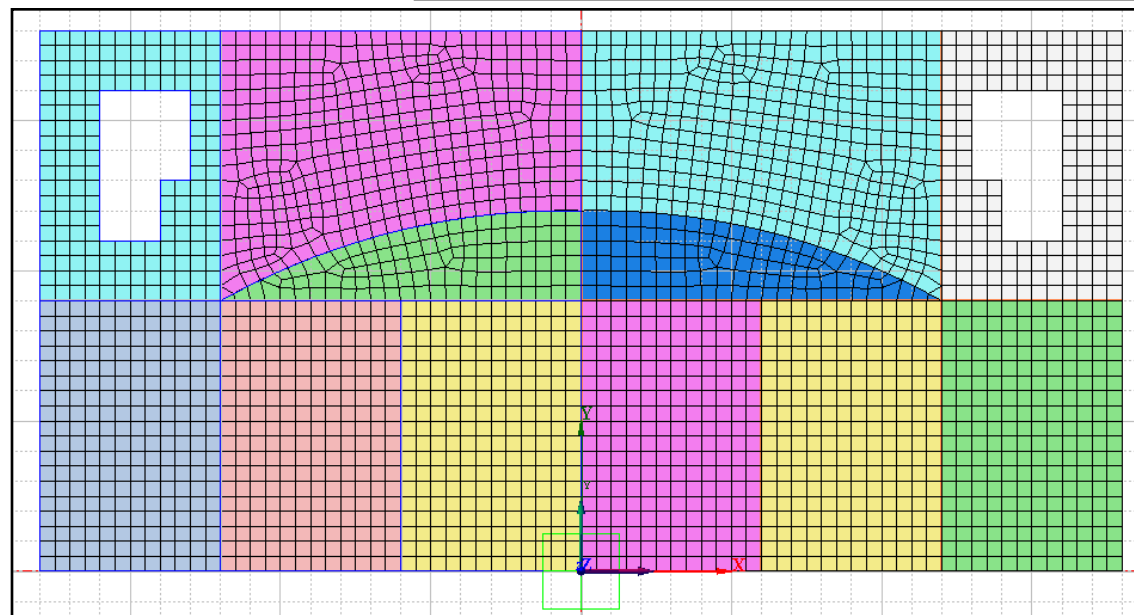
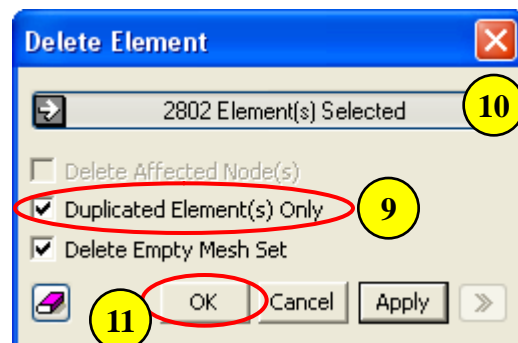


Step 10.




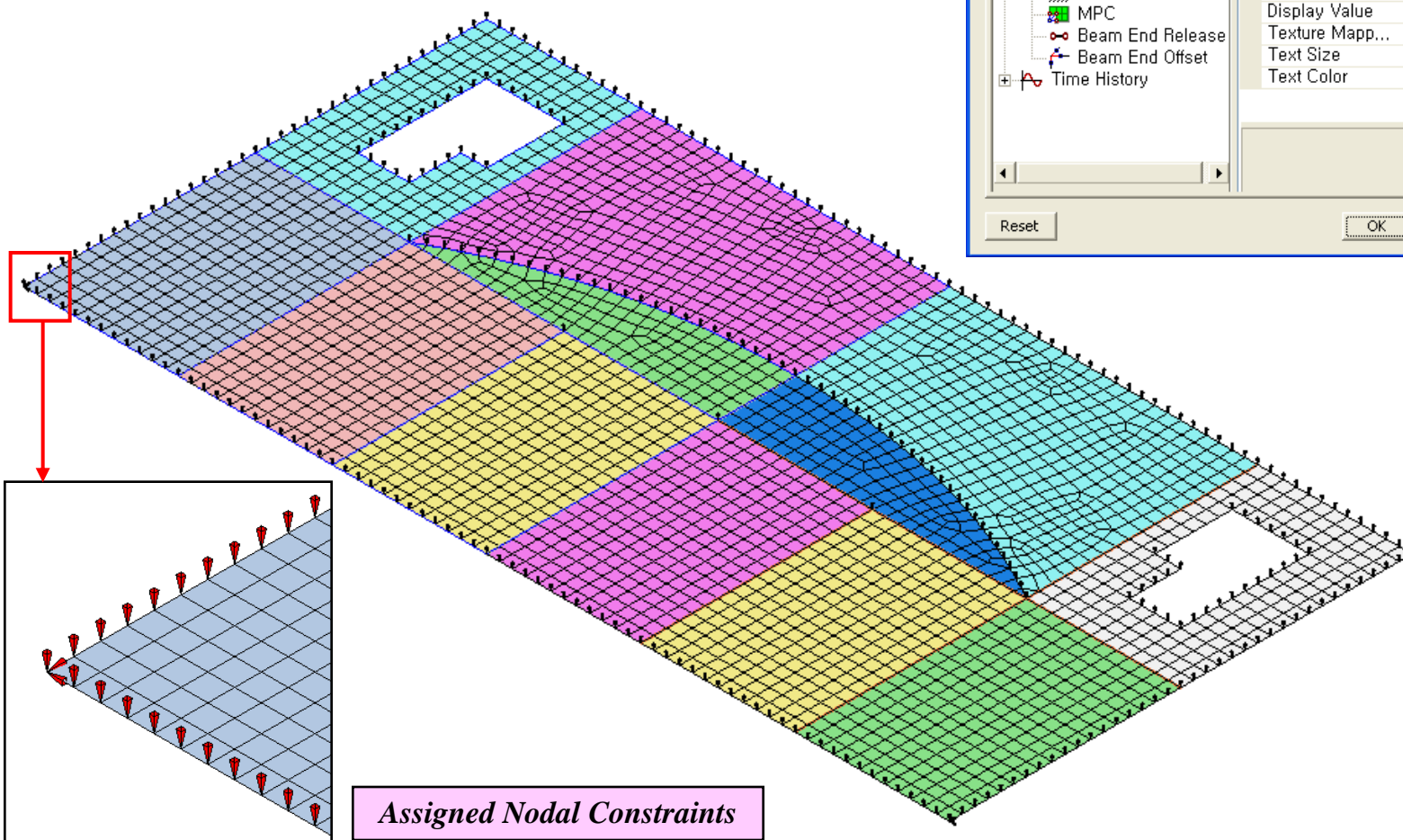
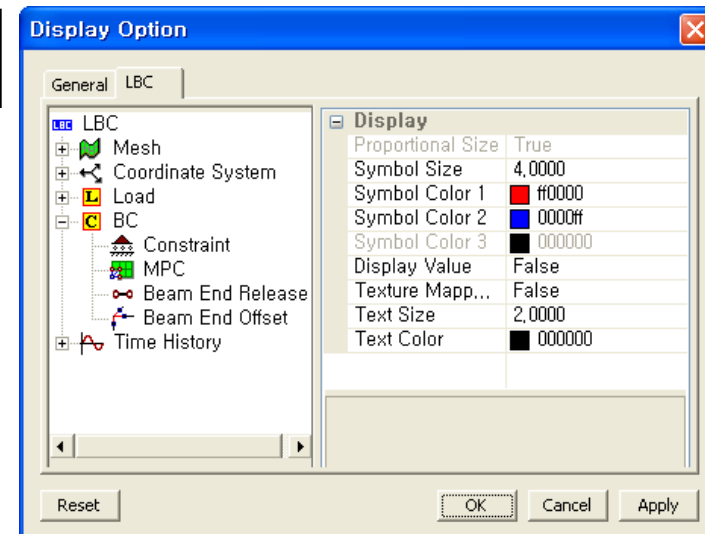
Select "YZ-Plane"
in Work Window
or Pre-Works Tree

1. Mesh > Transform > Mirror...
2. Select "Displayed"
3. Basis : Plane
4. Select "YZ-Plane"
5. Check on "Register as New Set"
6. Check on "Include LBCs (Only Nodal LBC)"
7. Click [OK] Button
8. Mesh > Element > Delete ...
9. Check on "Duplicated Element(s) Only"
10. Select "Displayed"
11. Click [OK] Button



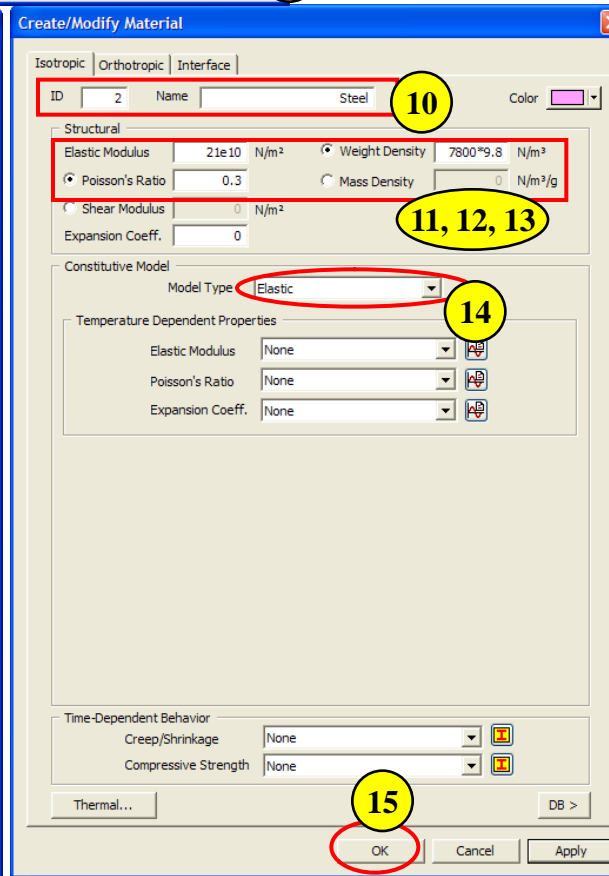
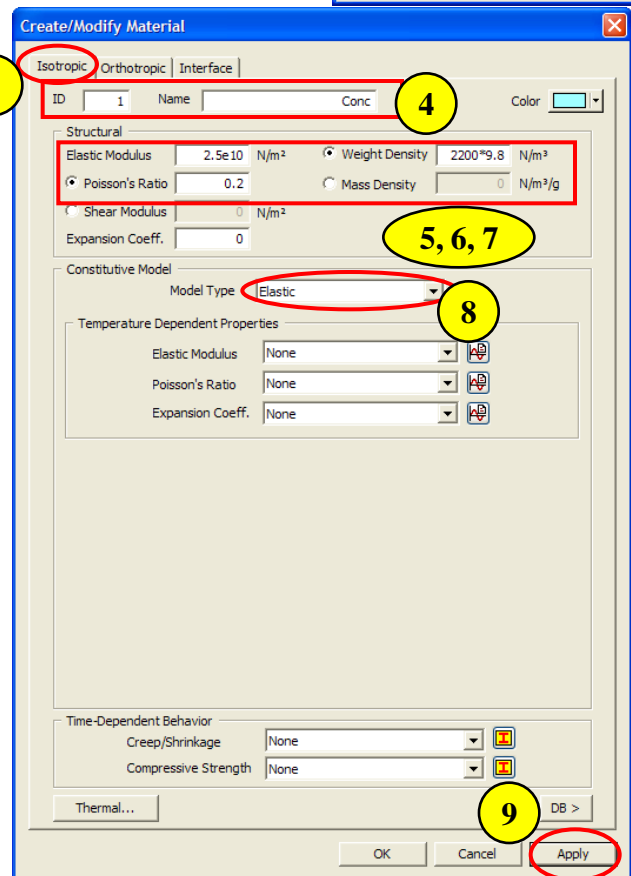
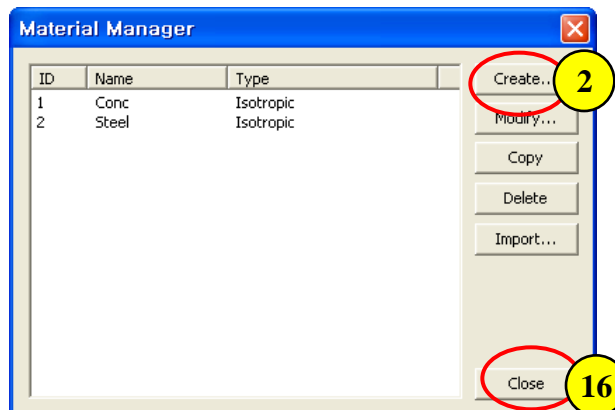
Step 11.

Symbol size can be changed at  “Display Option” – “LBC” tab.



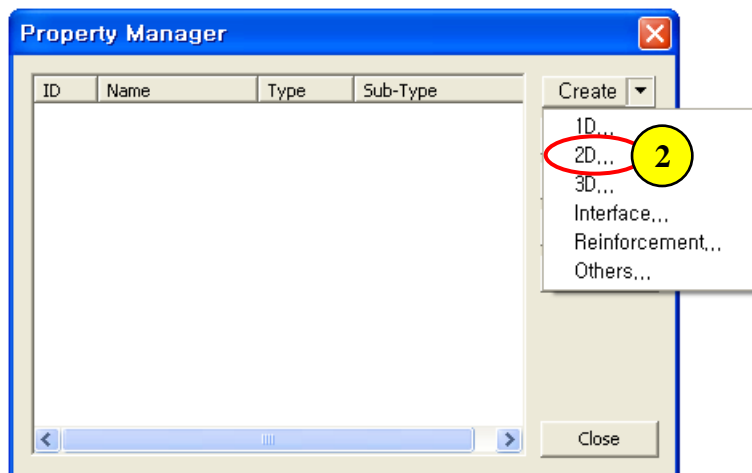
Assigned Nodal Constraints

Step 12.

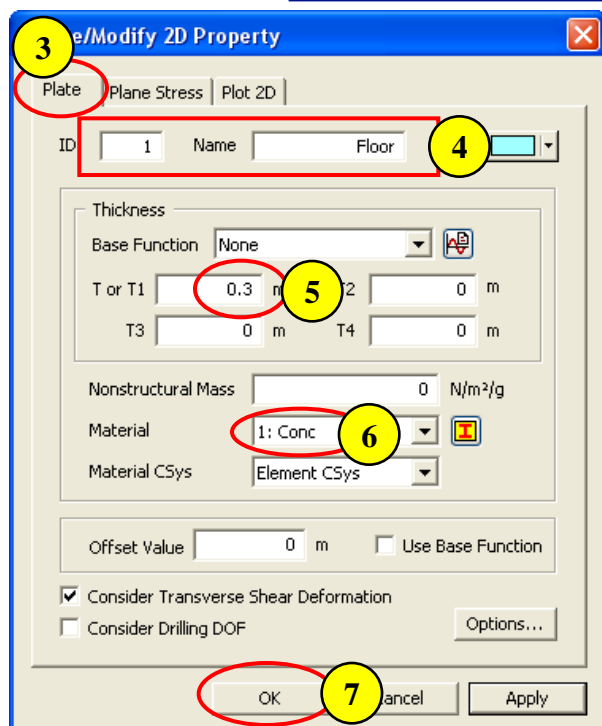


1. Analysis > Material ...
2. Click [Create...] Button
3. Select "Isotropic" tab
4. ID : 1 , Name : Conc
5. Elastic Modulus : $2.5e10 \text{ N/m}^2$
6. Poisson's Ratio : 0.2
7. Weight Density : $2200*9.8 \text{ N/m}^3$
8. Model Type : Elastic
9. Click [Apply] Button
10. ID : 2 , Name : Steel
11. Elastic Modulus : $21e10 \text{ N/m}^2$
12. Poisson's Ratio : 0.3
13. Weight Density : $7800*9.8 \text{ N/m}^3$
14. Model Type : Elastic
15. Click [OK] Button
16. Click [Close] Button

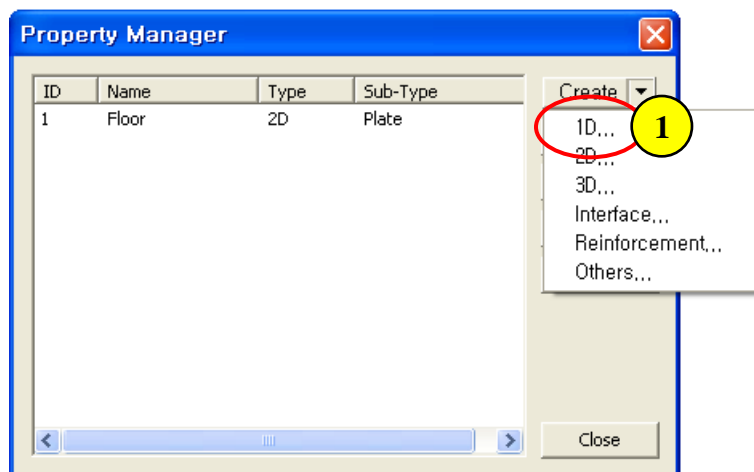
Step 13.



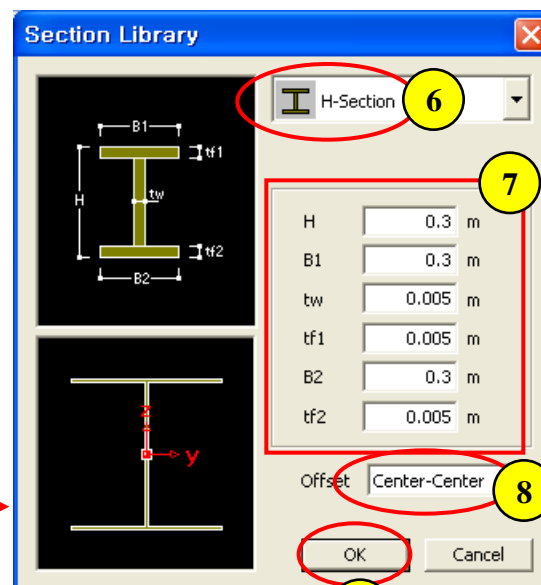
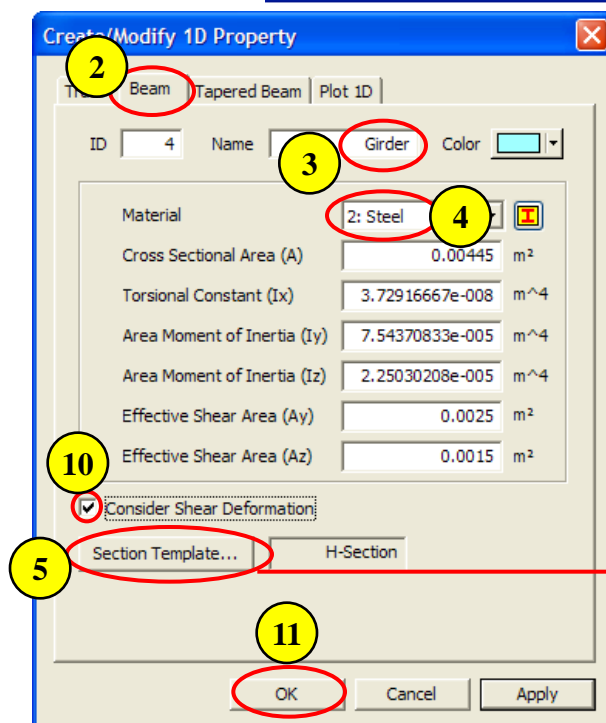
1. Analysis > Property ...
2. Create 2D ...
3. Select "Plate" tab
4. ID : 1 , Name : Floor
5. T or T1 : 0.3 m
6. Select "1: Conc" for Material
7. Click [OK] Button



Step 14.



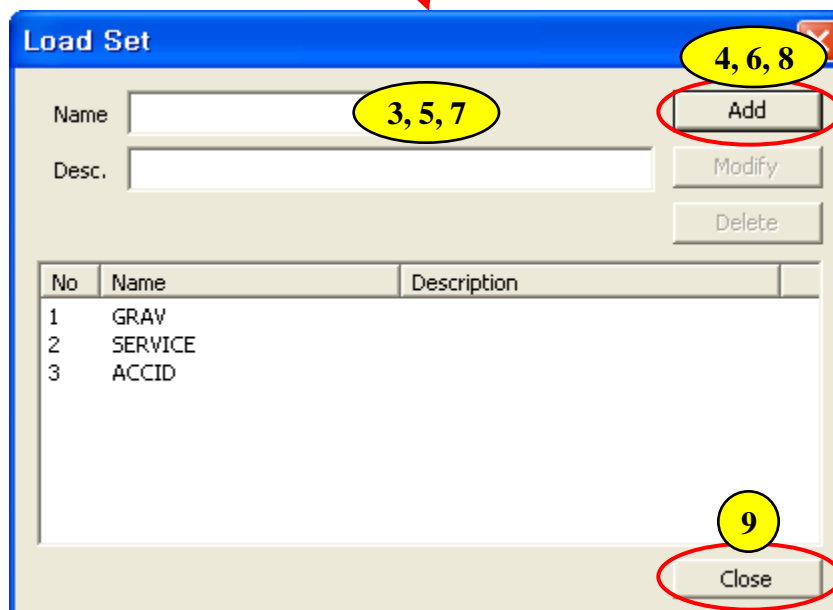
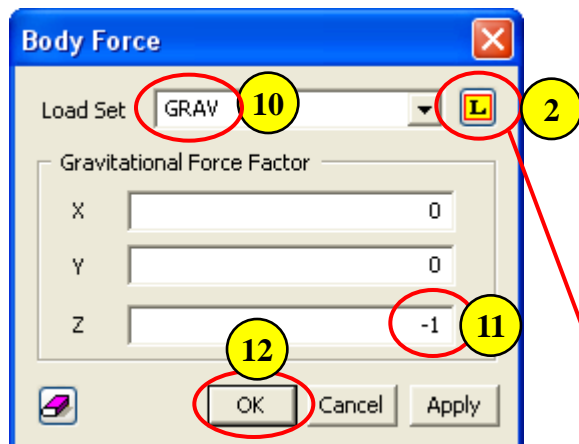
1. Create 1D ...
2. Select "Beam" tab
3. Name : Girder
4. Select "Steel" for Material
5. Click "Section Template" Button
6. Select "H-Section"
7. H (0.3) , B1 (0.3) , tw (0.005) ,
tf1 (0.005) , B2 (0.3) , tf2 (0.005)
8. Offset : Center-Center
9. Click [OK] Button
10. Select "Consider Shear Deformation"
11. Click [OK] Button



Material	2: Steel	...
Cross Sectional Area (A)	0.00445	m ²
Torsional Constant (Ix)	3.72916667e-008	m ⁴
Area Moment of Inertia (Iy)	7.54370833e-005	m ⁴
Area Moment of Inertia (Iz)	2.25030208e-005	m ⁴
Effective Shear Area (Ay)	0	m ²
Effective Shear Area (Az)	0	m ²

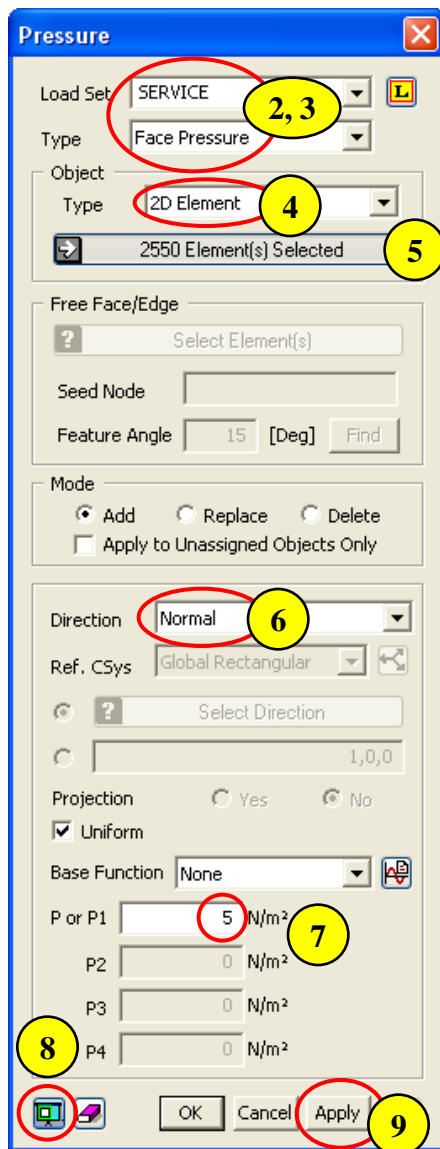
All sectional properties are automatically calculated.

Step 15.

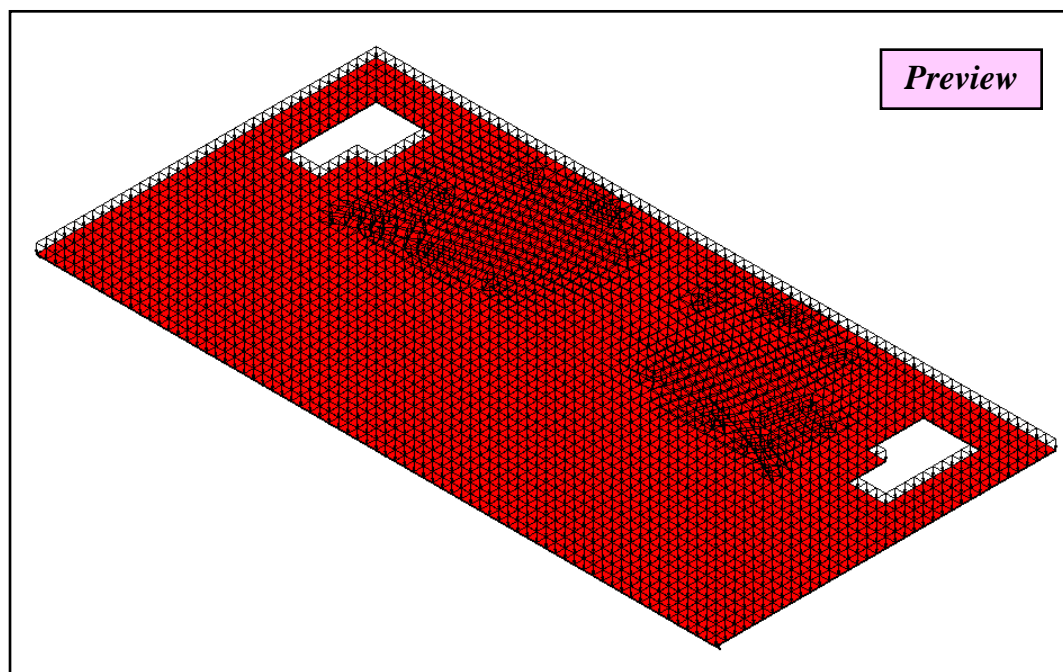


1. Analysis > Load > Body Force...
2. Click Button
3. Name : GRAV
4. Click [Add] Button
5. Name : SERVICE
6. Click [Add] Button
7. Name : ACCID
8. Click [Add] Button
9. Click [Close] Button
10. Load Set : GRAV
11. Gravitational Force Factor : Z (-1)
12. Click [OK] Button

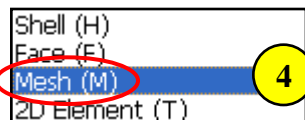
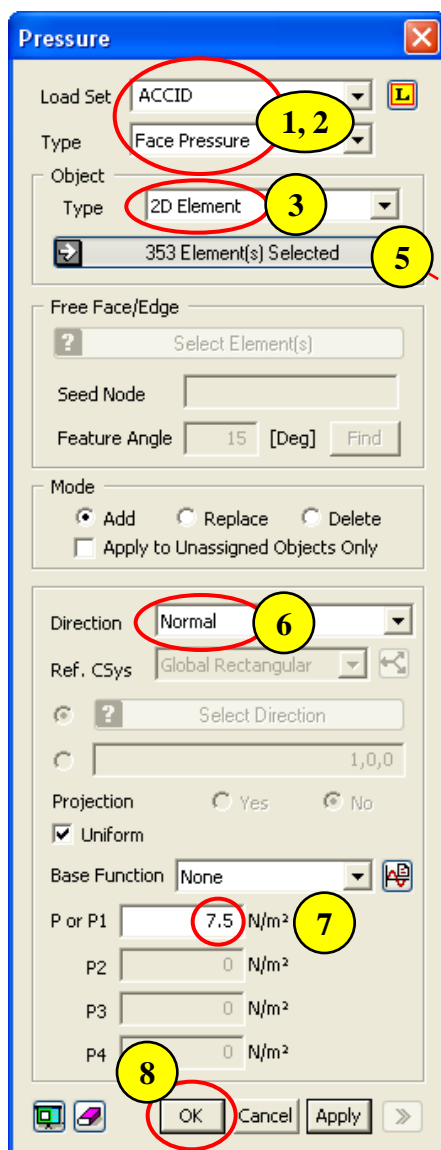
Step 16.



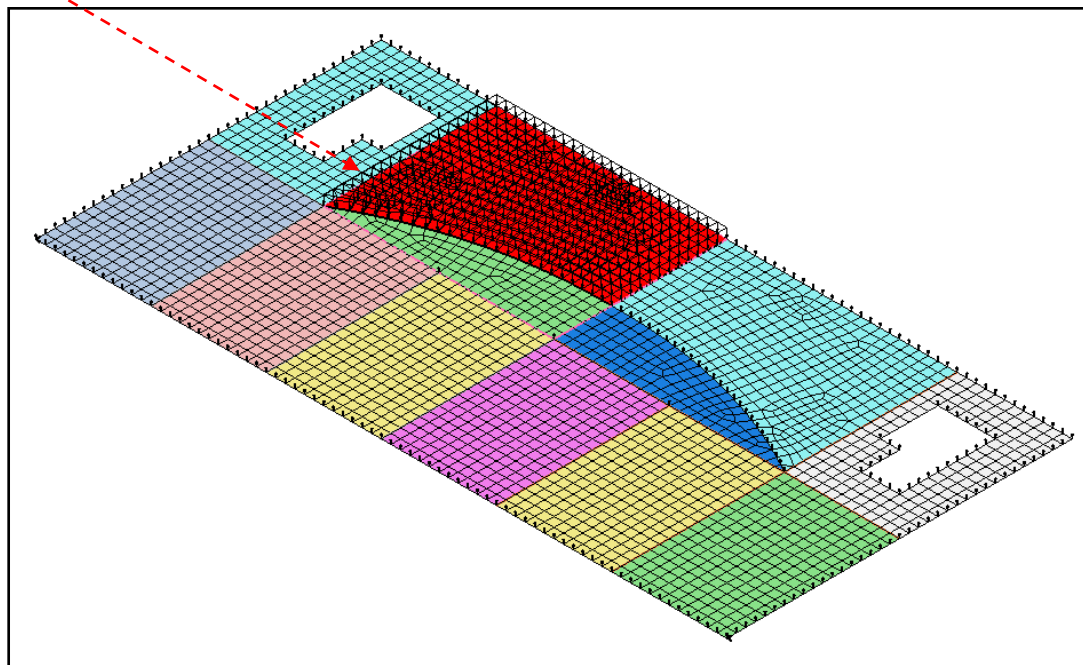
1. Analysis > Load > Pressure ...
2. Load Set : SERVICE
3. Type : Face Pressure
4. Object Type : 2D Element
5. Select "Displayed"
6. Direction : Normal
7. P or P1: 5 N/m²
8. Click "Preview" Button
9. Click [Apply] Button



Step 17.

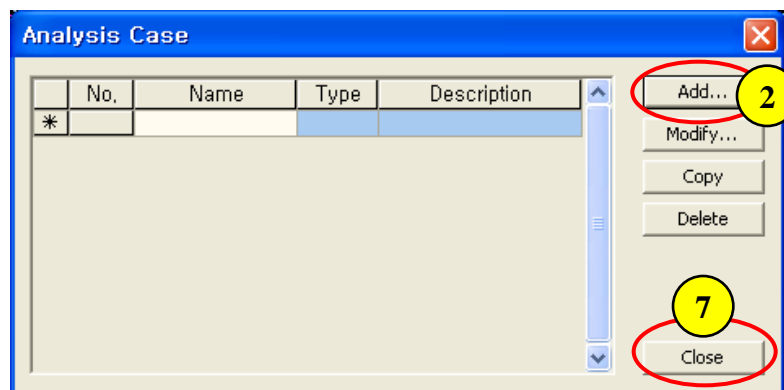


1. Load Set : ACCID
2. Type : Face Pressure
3. Object Type : 2D Element
4. Change Selection Filter to "Mesh (M)"
5. Select Mesh Set (See Figure)
6. Direction : Normal
7. P or P1: 7.5 N/m²
8. Click [OK] Button



Click empty space in Work Window and press shortcut 'M'.

Step 18.



1. Analysis > Analysis Case ...

2. Click [Add] Button

3. Name : Linear

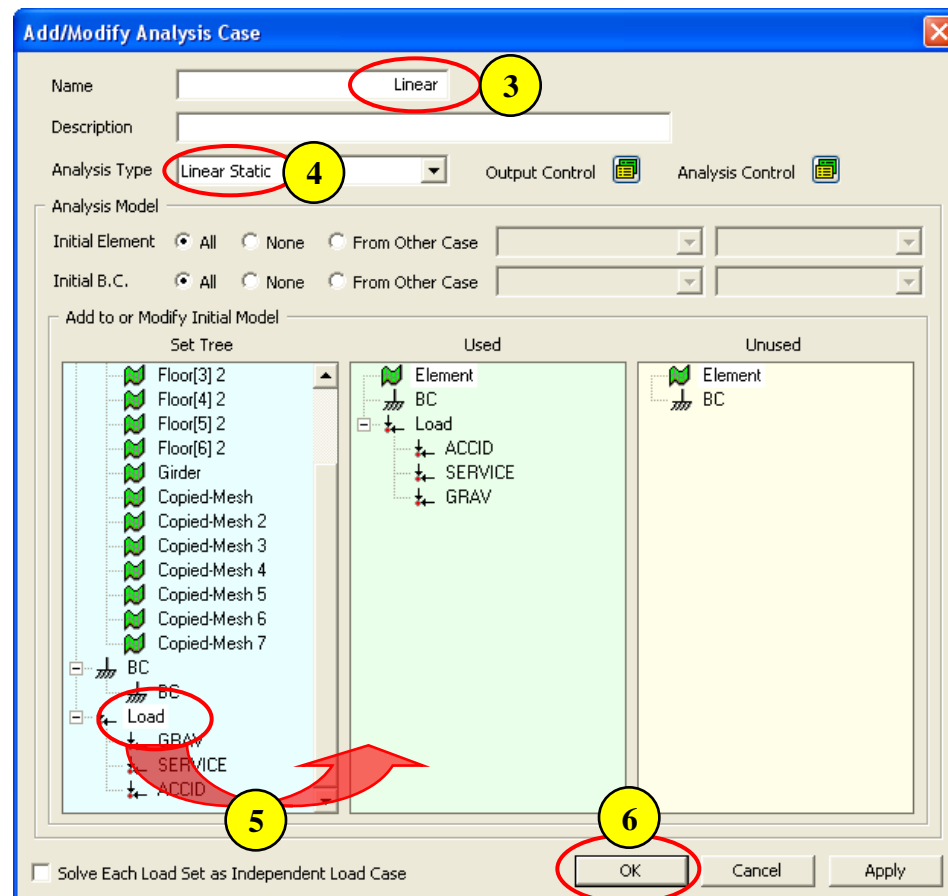
4. Analysis Type : Linear Static

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

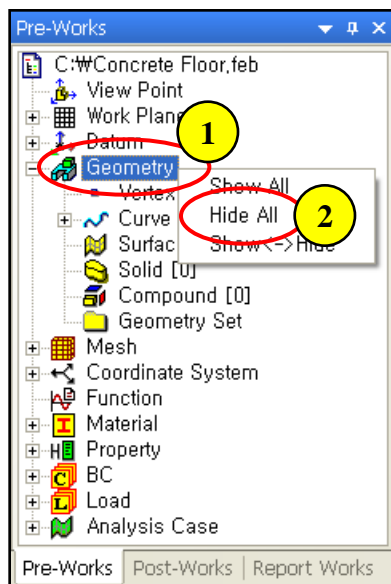
6. Click [OK] Button

7. Click [Close] Button

8. File > Save ... (Floor.feb)



Step 19.

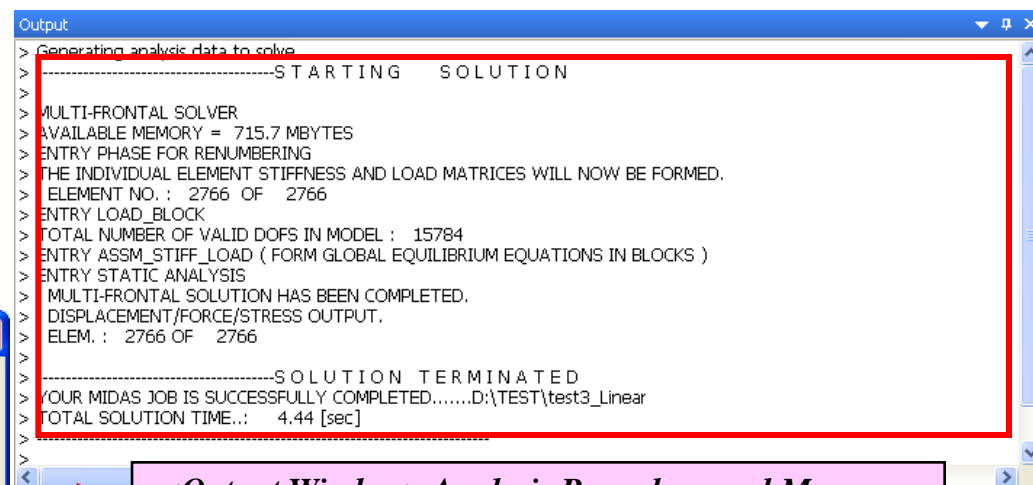


1. Pre-Works Tree : Geometry ...

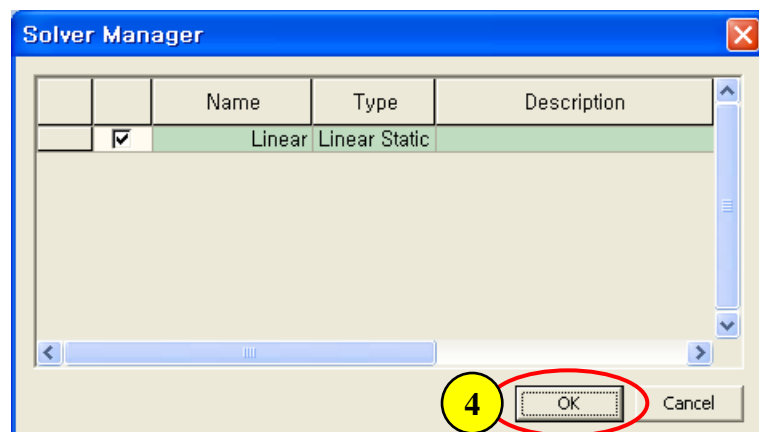
2. Click Right Mouse Button and Select "Hide All"

3. Analysis > Solve ...

4. Click [OK] Button

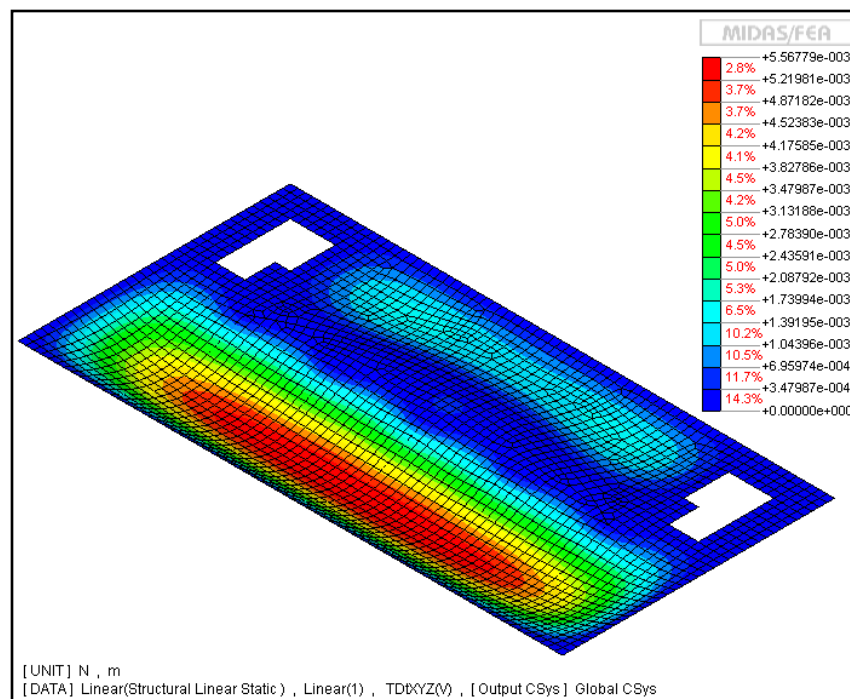
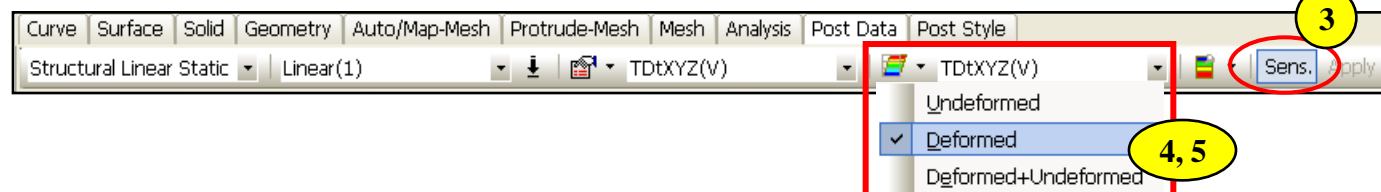
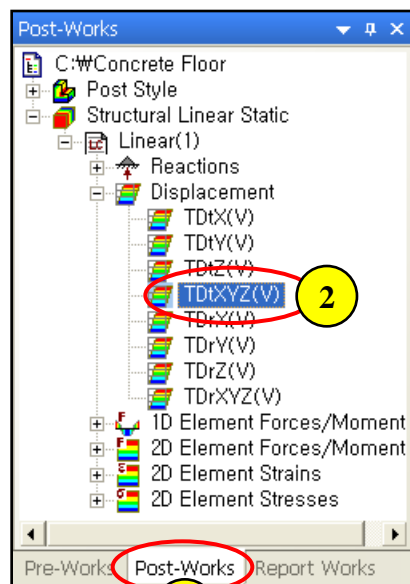


<Output Window> Analysis Procedure and Messages



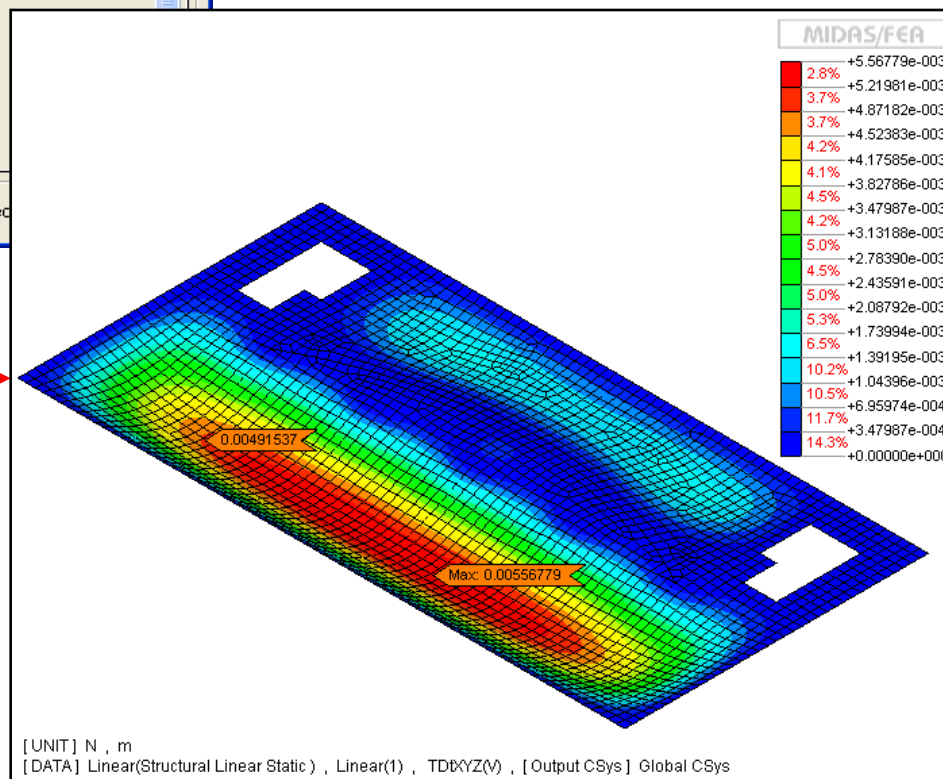
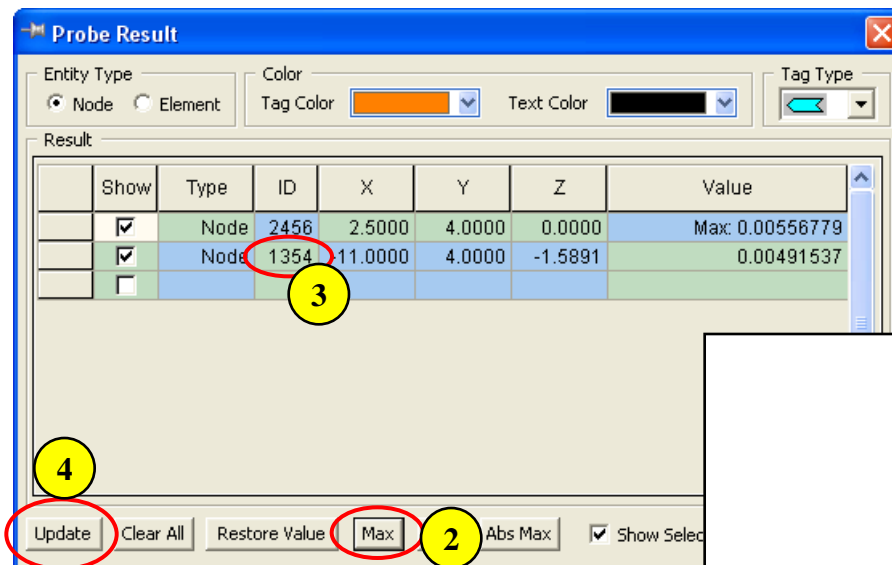
Step 20.

1. Post-Works Tree : Linear (Structural Linear Static) > Linear(1) > Displacement
2. Double Click "TDtXYZ(V)"
3. Click "Sens." Button
4. Select "Deformed" for Mesh Shape at "Post Data" Toolbar (See Figure)
5. Select "TDtXYZ(V)" for Deformation Data



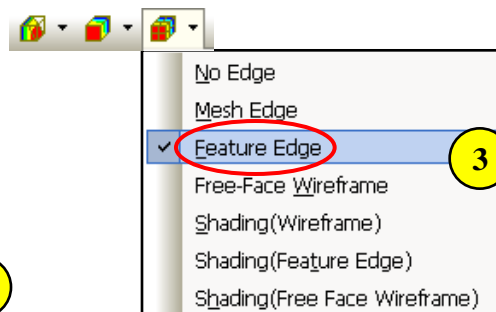
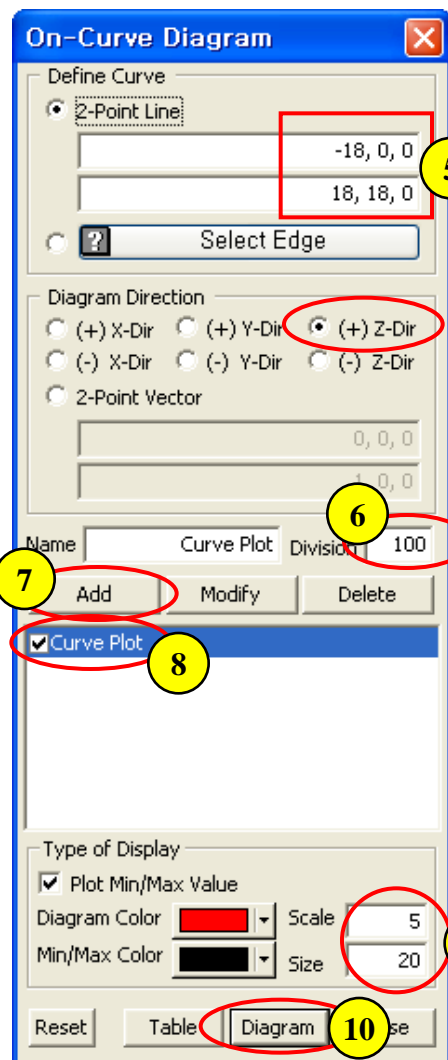
Step 21.

1. Post > Probe Result ...
2. Click [Max] Button
3. Enter Node ID : 1354
4. Click [Update] Button

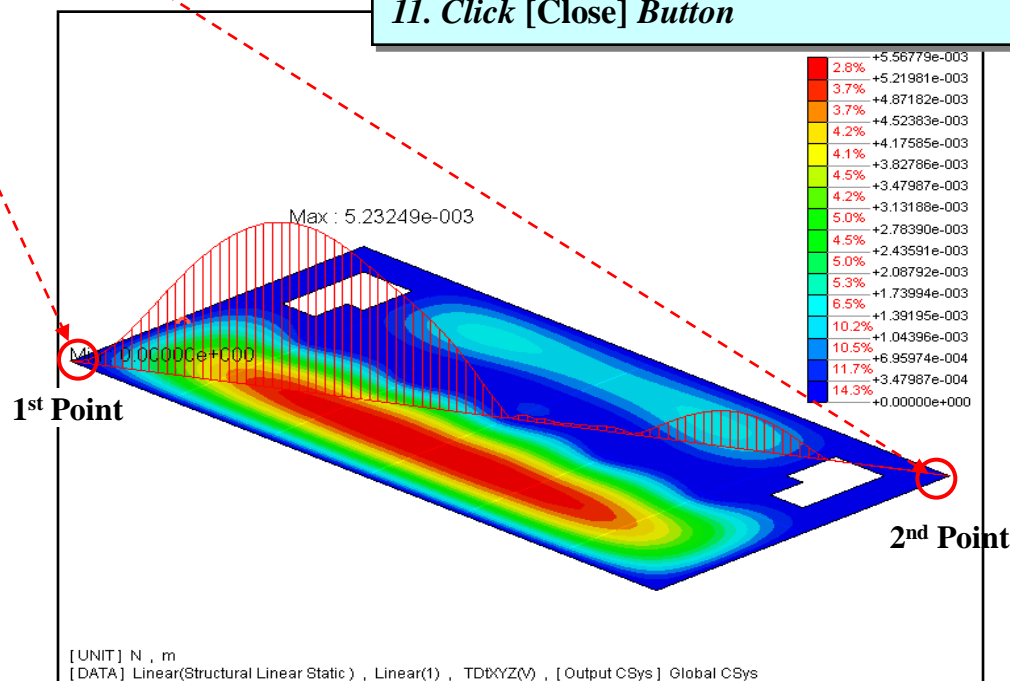


Same as select node in Work Window.

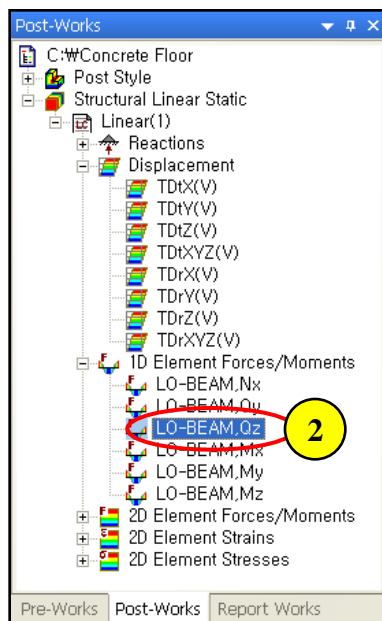
Step 22.



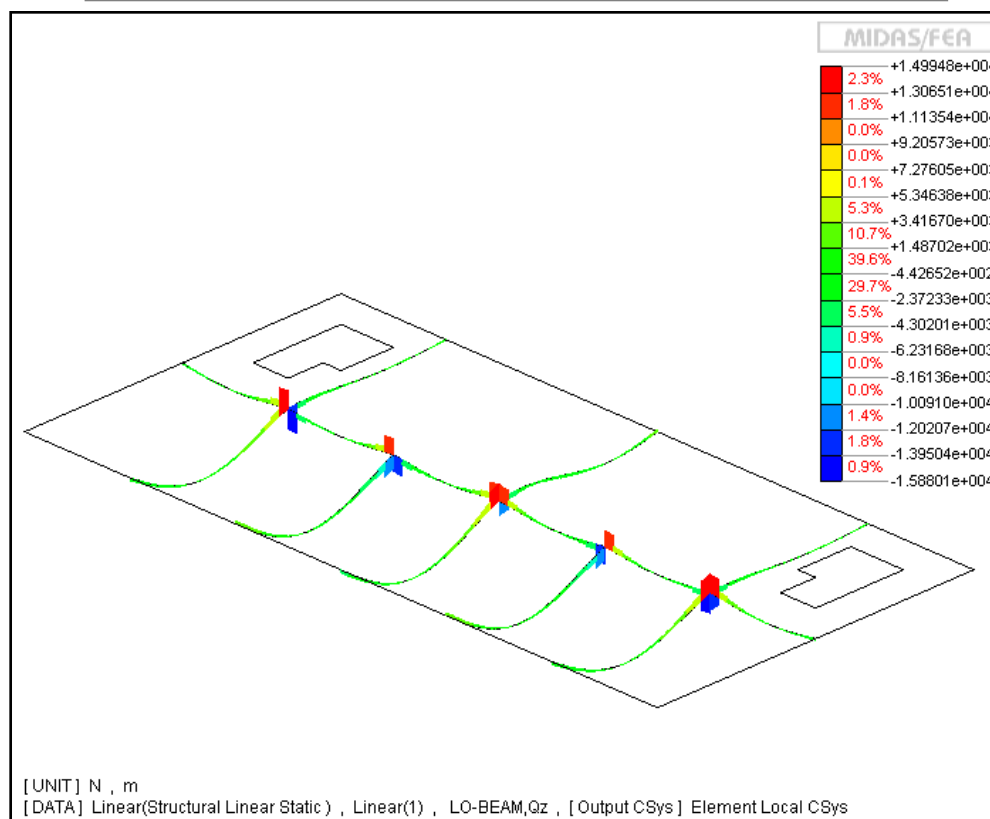
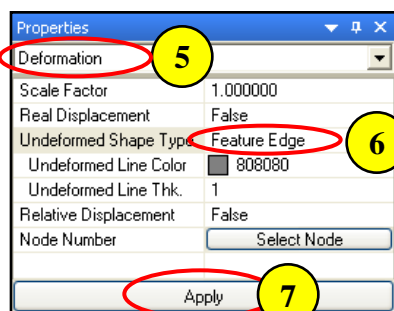
1. Selecte "Undeformed" for Mesh Shape
2. Select "Post Style" Toolbar
3. Select "Feature Edge" For Edge Type
4. Post > On-Curve Diagram ...
5. Select 2 Points (See Figure)
6. Division : 100
7. Click [Add] Button
8. Check on "Curve Plot"
9. Scale : 5 , Size : 20
10. Click [Diagram] Button
11. Click [Close] Button



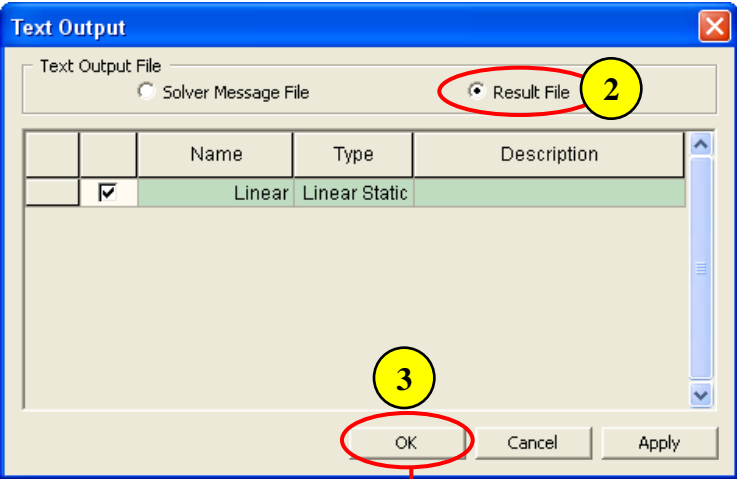
Step 23.



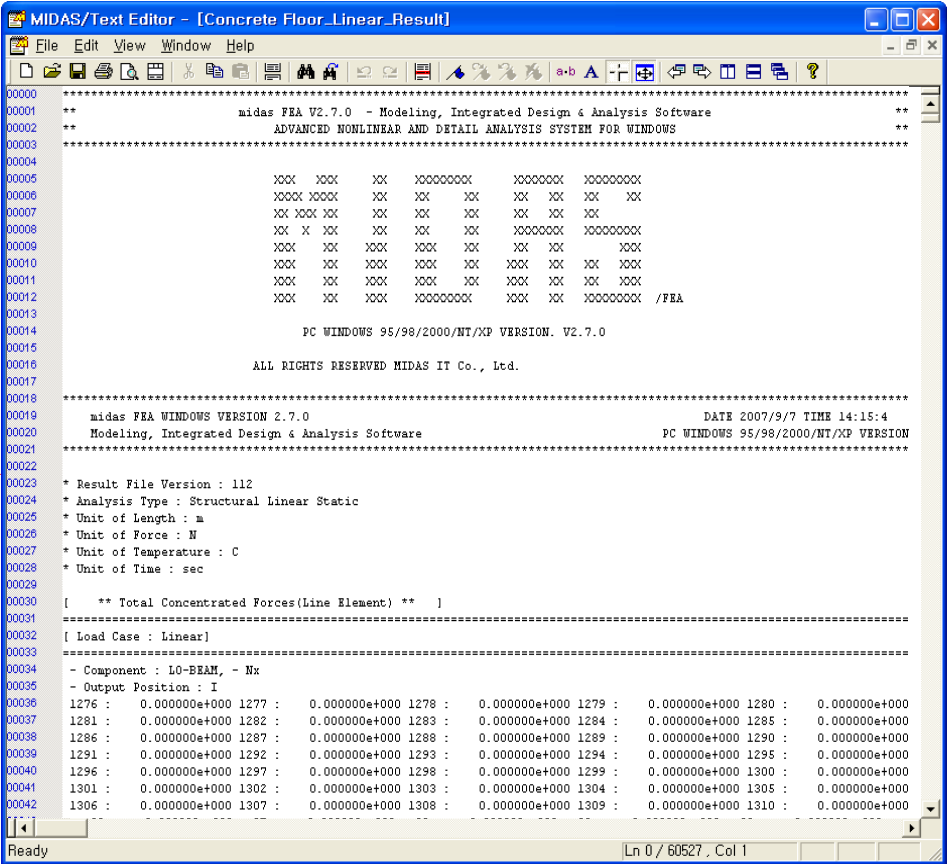
1. Post-Works Tree : Linear (Structural Linear Static) > Linear(1) > 1D Element Forces/Moments ...
2. Double Click “LO-BEAM, Qz”
3. Select “Post Data” Toolbar
4. Select “Deformed+Undeformed” for Mesh Shape
5. Property Window : Deformation
6. Select “Feature Edge” for Undeformed Shape Type
7. Click [Apply] Button



Step 24.



- 1. Post > Text Output ...
- 2. Check on "Result File"
- 3. Click [OK] Button



Text Output of Analysis Results