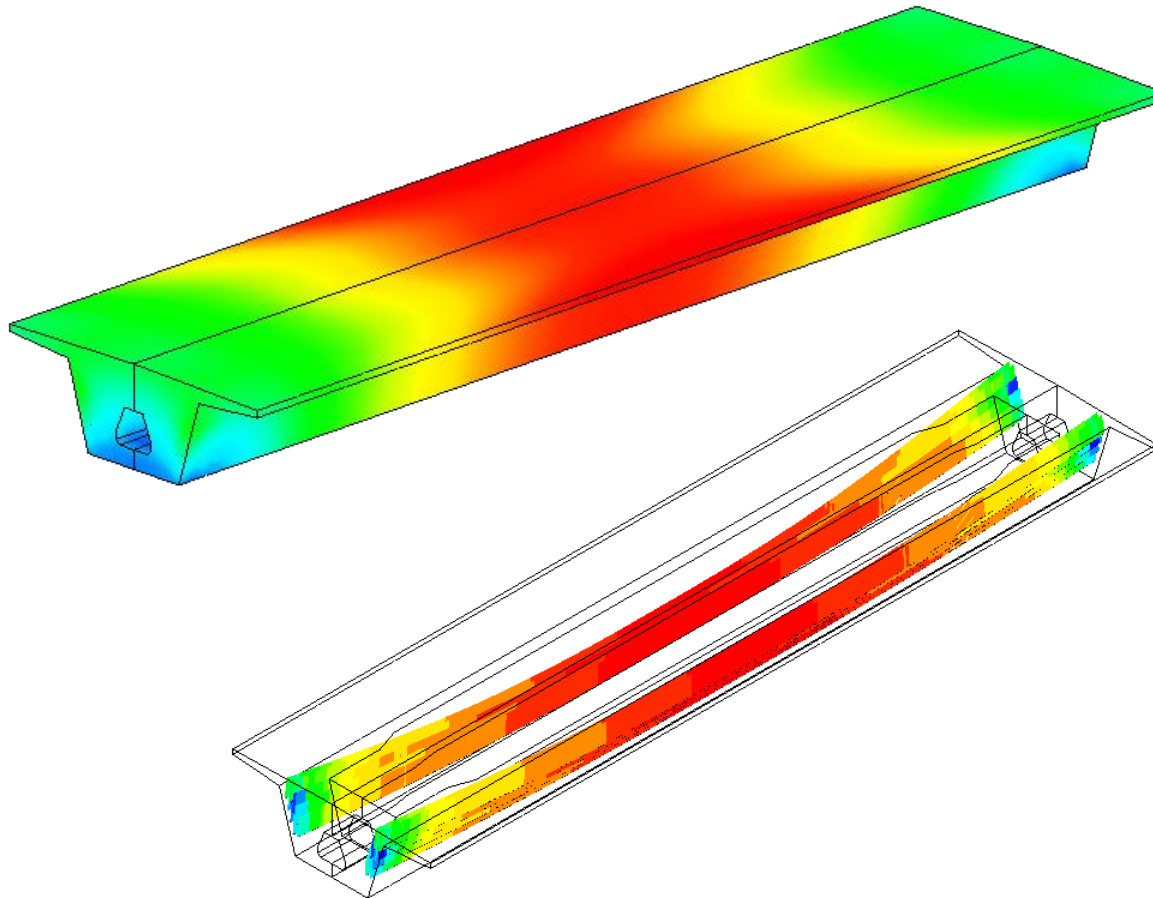


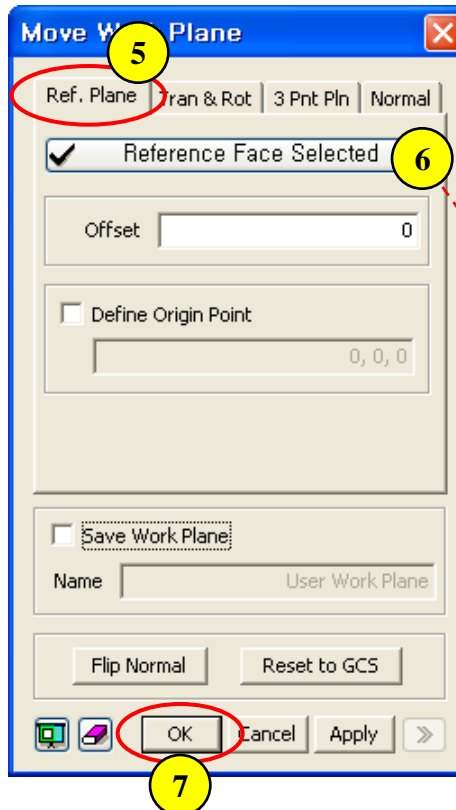
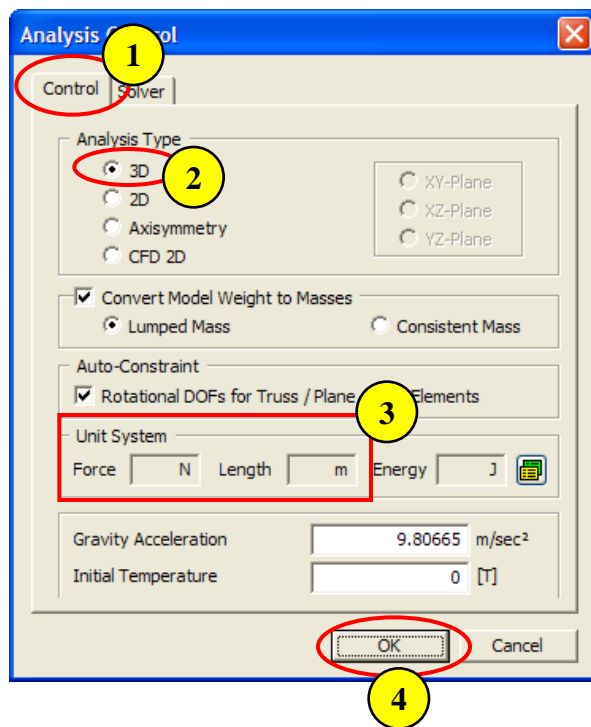
LS-16. FSM Bridge



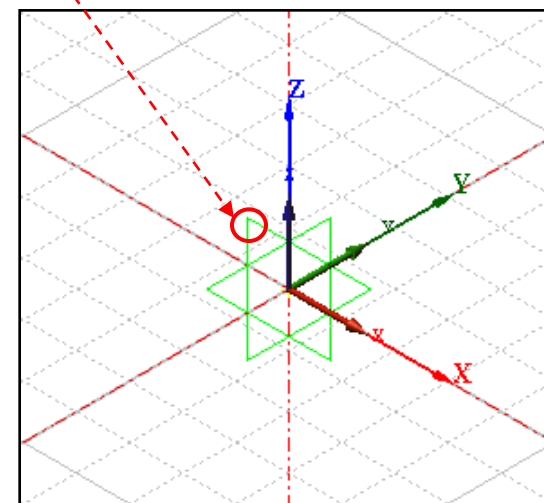
Overview

- 3-D Linear Static Analysis
- Model
 - $\frac{1}{2}$ Symmetric Model
 - Unit : N, m
 - Isotropic Elastic Material
 - Reinforcement Element
 - Solid Element
- Load & Boundary Condition
 - Body Force
 - Pressure
 - Arbitrary Load
 - Prestress for Reinforcement
 - Constraint
 - Symmetric Condition
- Result Evaluation
 - Deformation
 - Mirror Mode
 - Reinforcement Stresses

Step 1.

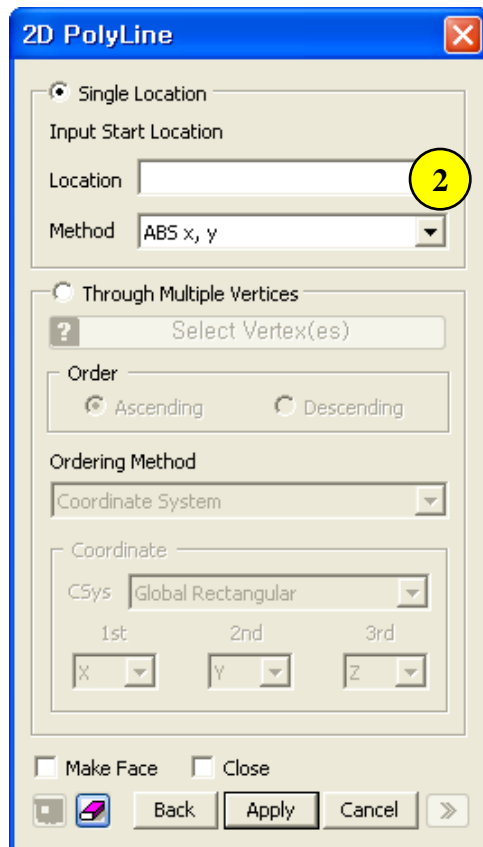


1. Analysis > Analysis Control – “Control” tab
2. Analysis Type : 3D
3. Unit : N , m
4. Click [OK] Button
5. Geometry > Work Plane > Move – “Ref. Plane” tab
6. Select “XZ Plane”
7. Click [OK] Button



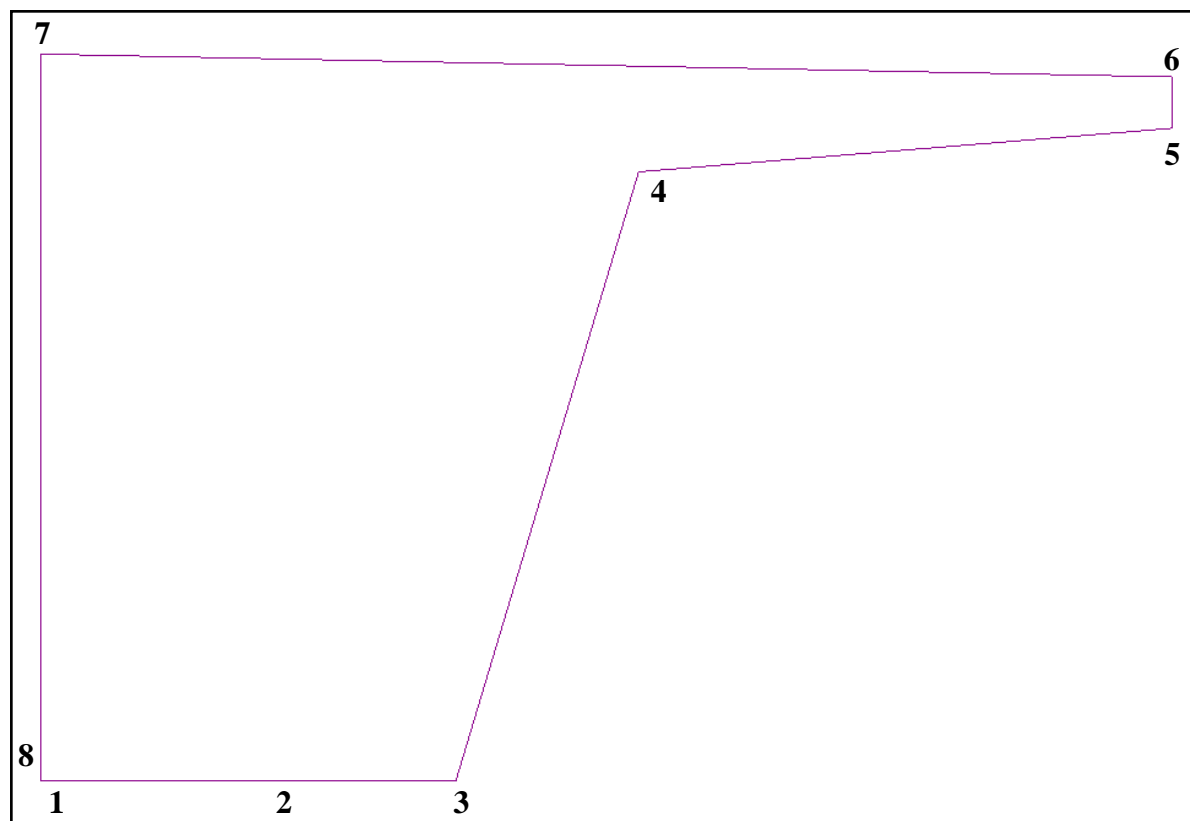
🔊 Analysis Control Dialog is automatically activated at startup.

Step 2.



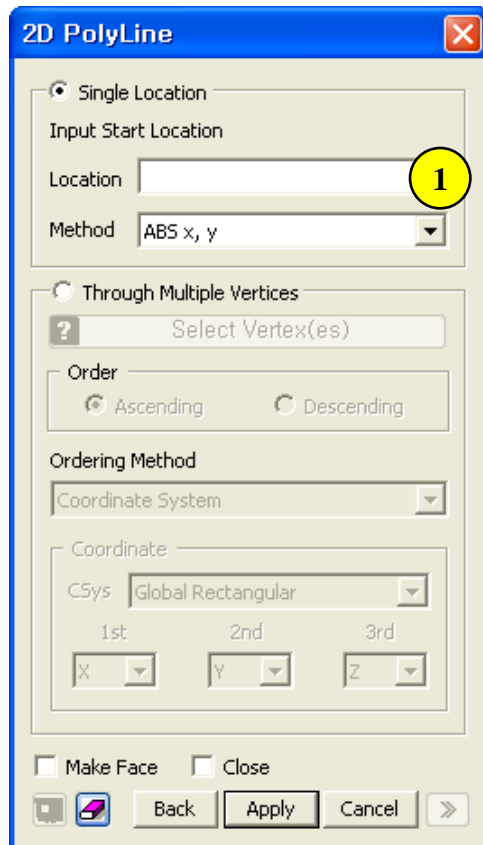
1. Geometry > Curve > Create on WP > Polyline (Wire)...

2. Location : (0) , <1.25> , <0.75> , <0.88, 2.933> , <2.57, 0.21> , <0, 0.247> , <-5.45, 0.11> , <0, -3.5> Ⓜ



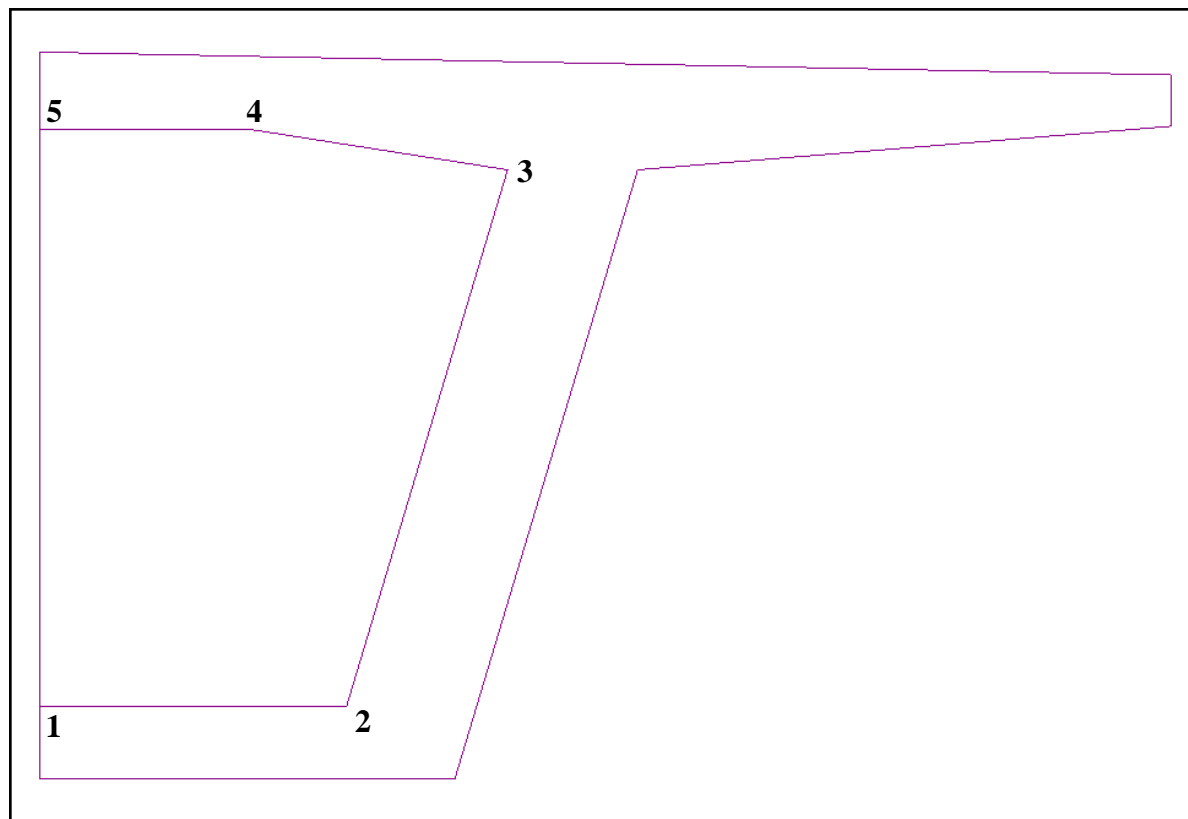
Ⓜ (): "ABS x, y", <>: "REL dx, dy"
(0) same as (0, 0)

Step 3.



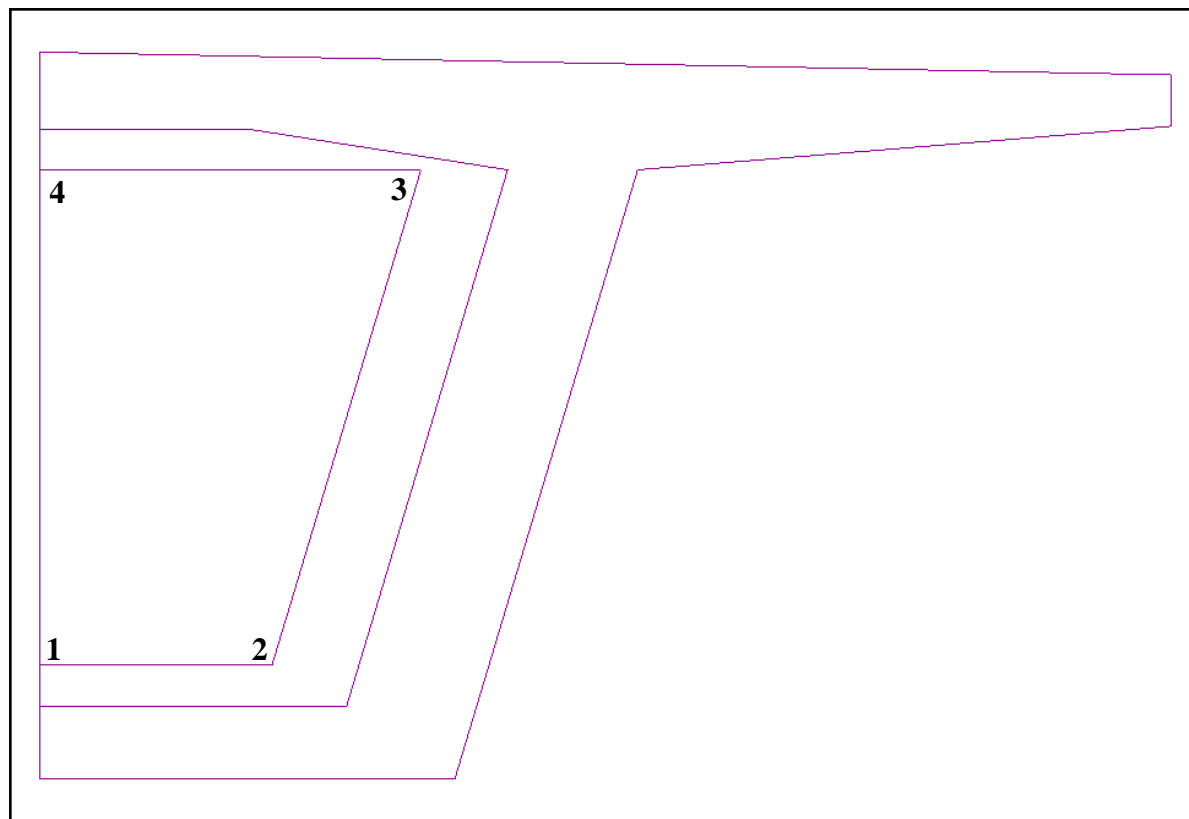
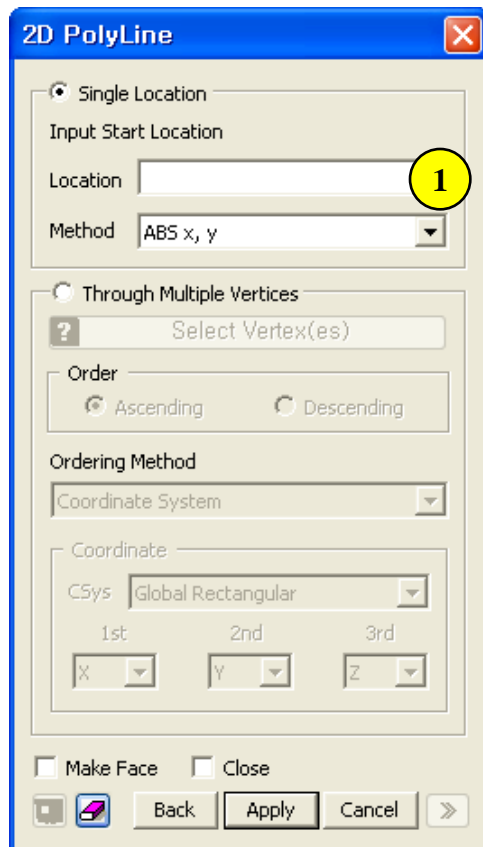
1. Location : (0, 0.35) , <1.479> , <0.775, 2.583> , <-1.254, 0.197> , <-1>

2. Click Right Mouse Button on the Work Window

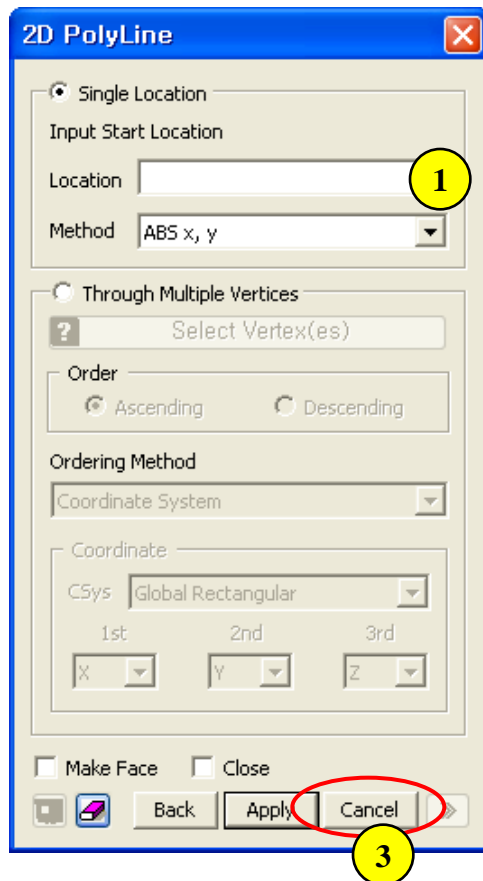



Step 4.

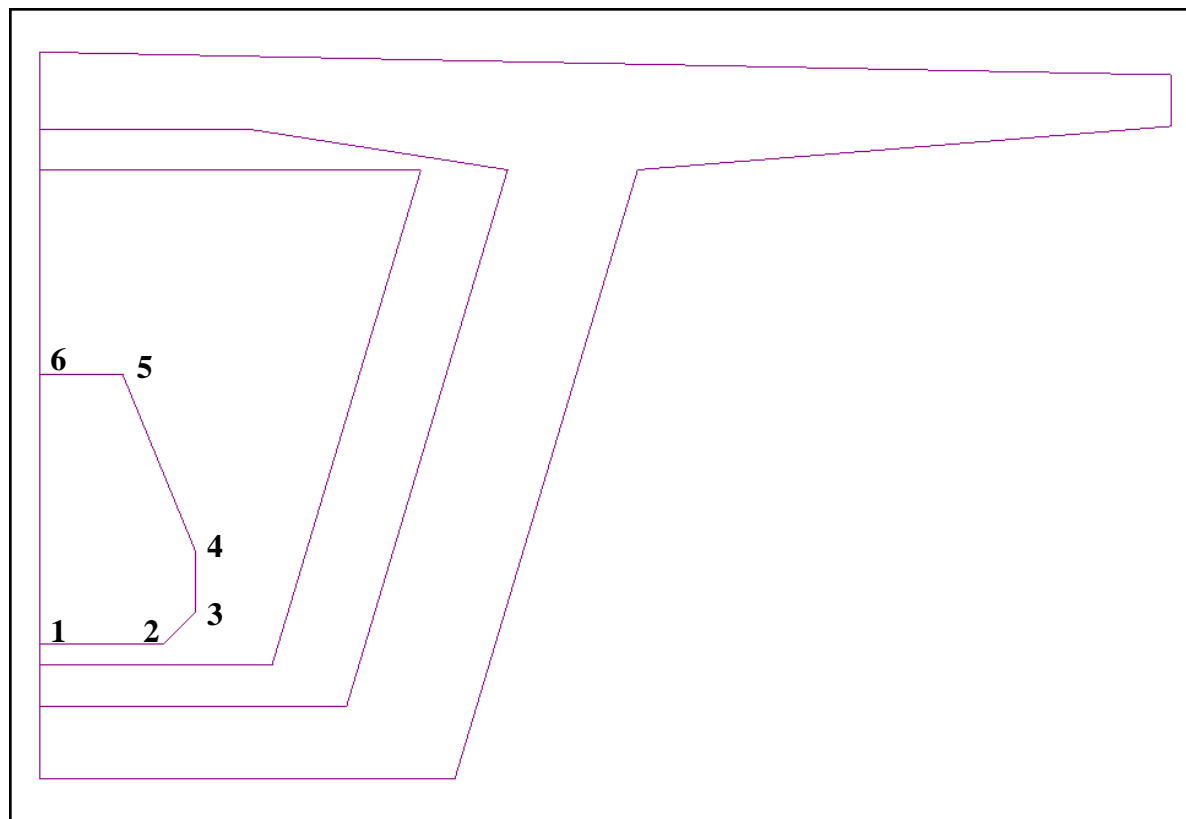
1. Location : (0, 0.55) , <1.121> , <0.715, 2.38> , <-1.836>
2. Click Right Mouse Button on the Work Window



Step 5.




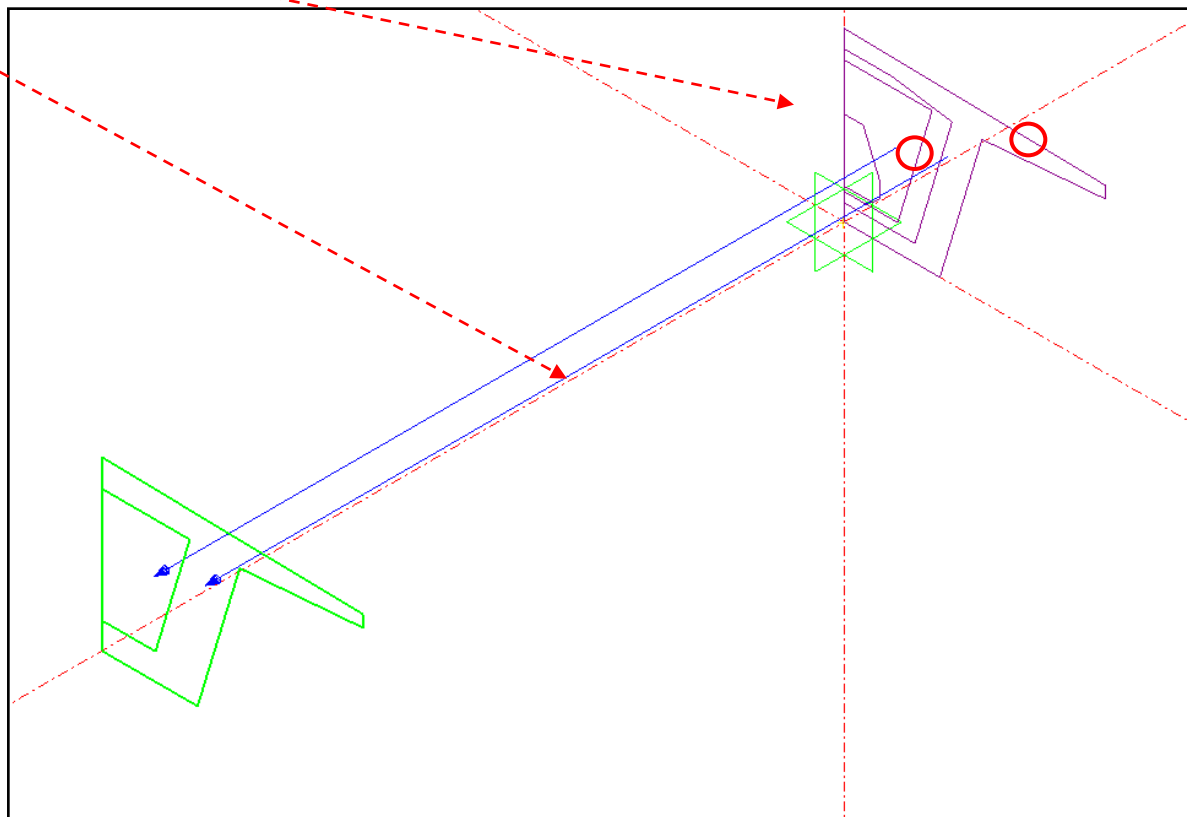
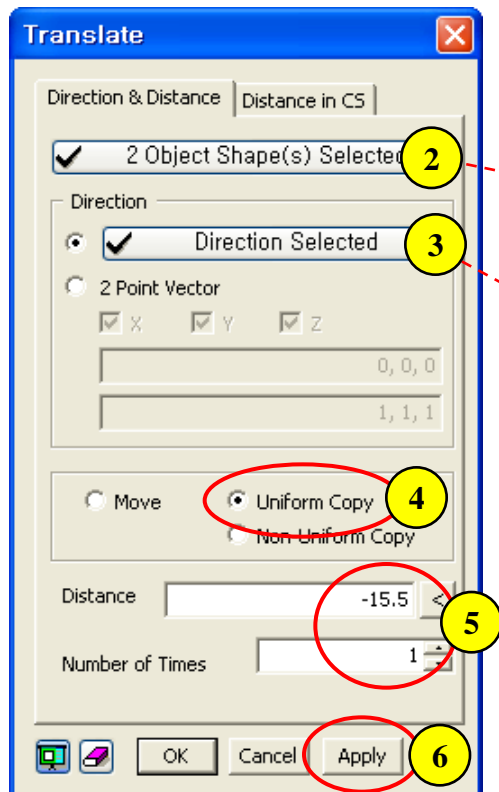
1. Location : (0, 0.65), <0.6>, <0.15, 0.15>, <0, 0.3>, <-0.35, 0.85>, <-0.4>
2. Click Right Mouse Button on the Work Window
3. Click [Cancel] Button 



 [Esc] as shortcut for [Cancel].

Step 6.

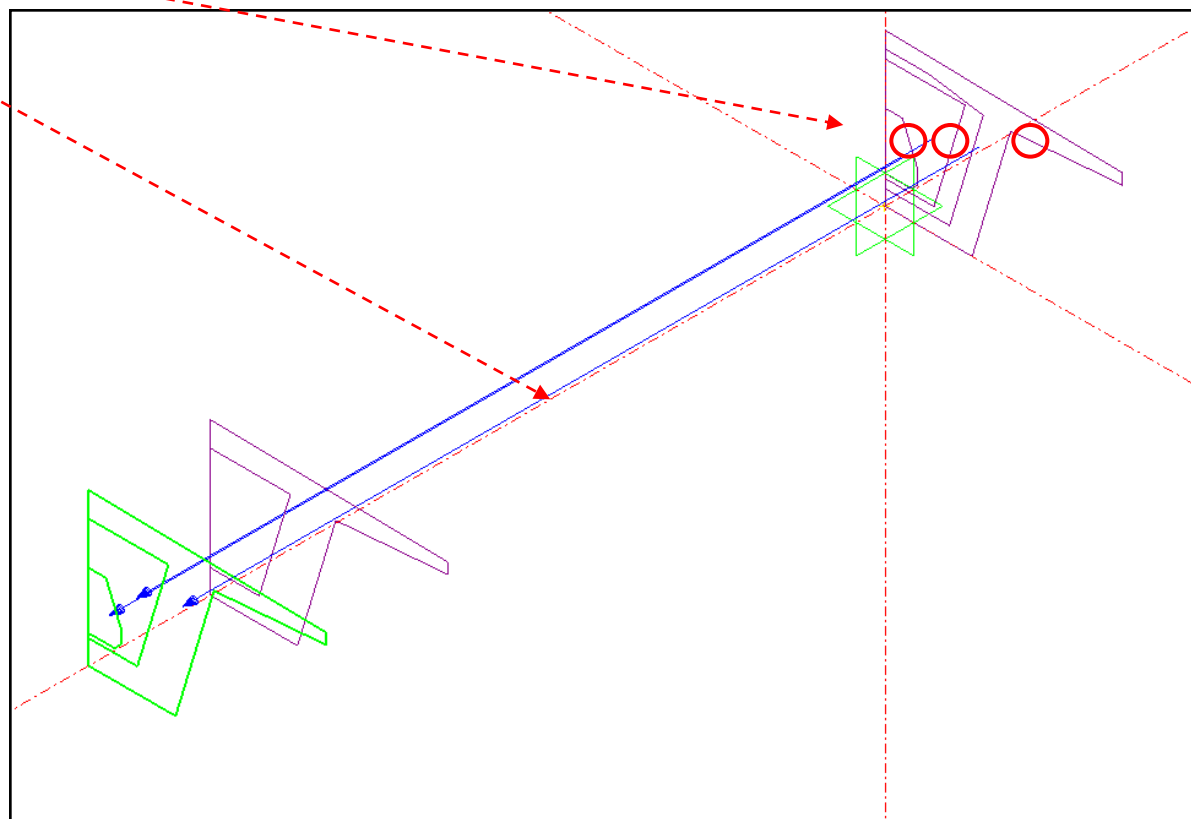
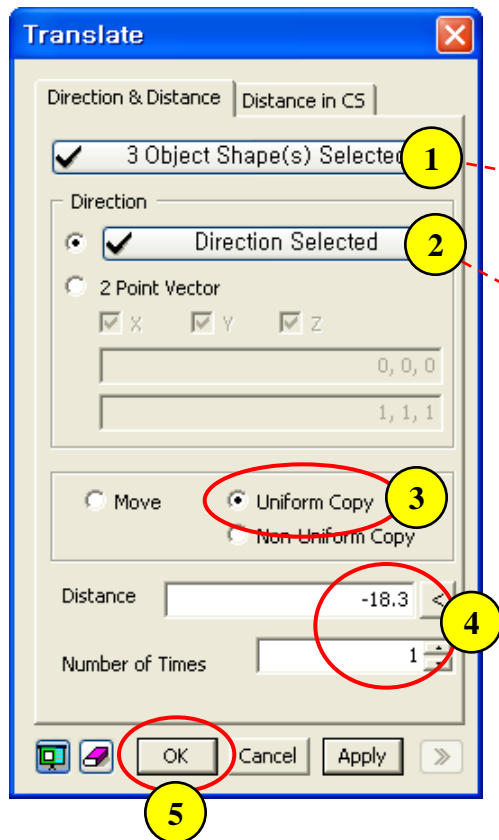
1. Geometry > Transform > Translate ...
2. Select 2 Shapes(Wires) marked by "O" (See Figure)
3. Click "Select Direction" Button and Select "Y-Axis"
4. Check on "Uniform Copy"
5. Distance (-15.5) , Number of Times (1)
6. Click [Apply] Button 



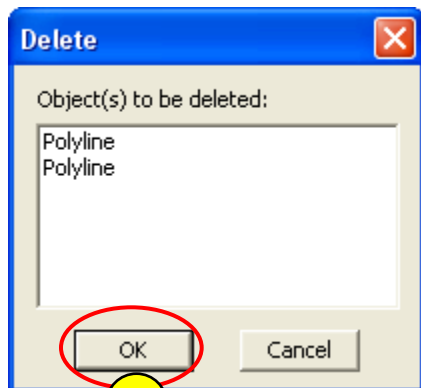
 [Enter] as shortcut for [Apply].


Step 7.

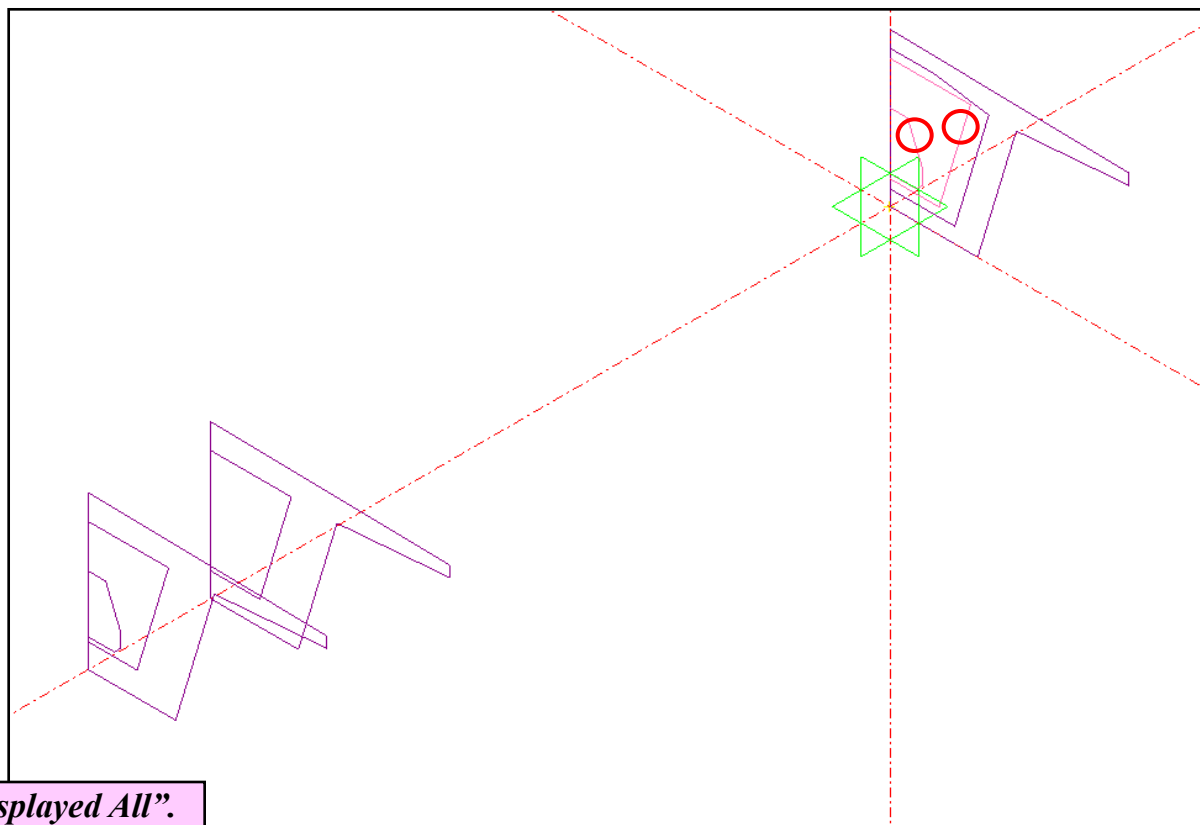
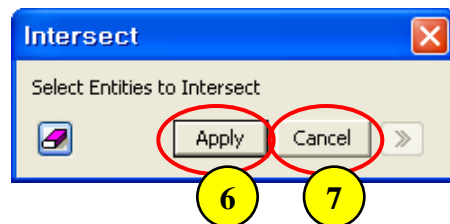
1. Select 3 Shapes(Wires) marked by “O” (See Figure)
2. Click “Select Direction” Button and Select “Y-Axis”
3. Check on “Uniform Copy”
4. Distance (-18.3) , Number of Times (1)
5. Click [OK] Button




Step 8.

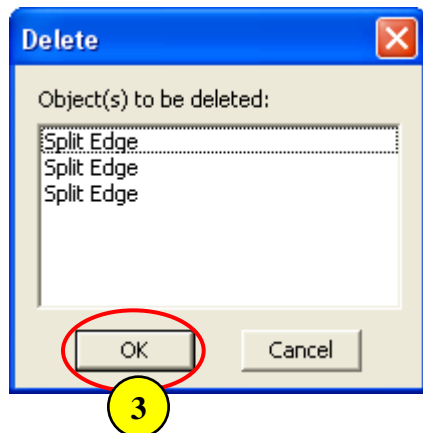


1. Select 2 Shapes(Wires) marked by “O” (See Figure)
2. Press [Delete] Key
3. Click [OK] Button
4. Geometry > Curve > Intersect ...
5. Select  “Displayed All”
6. Click [Apply] Button
7. Click [Cancel] Button

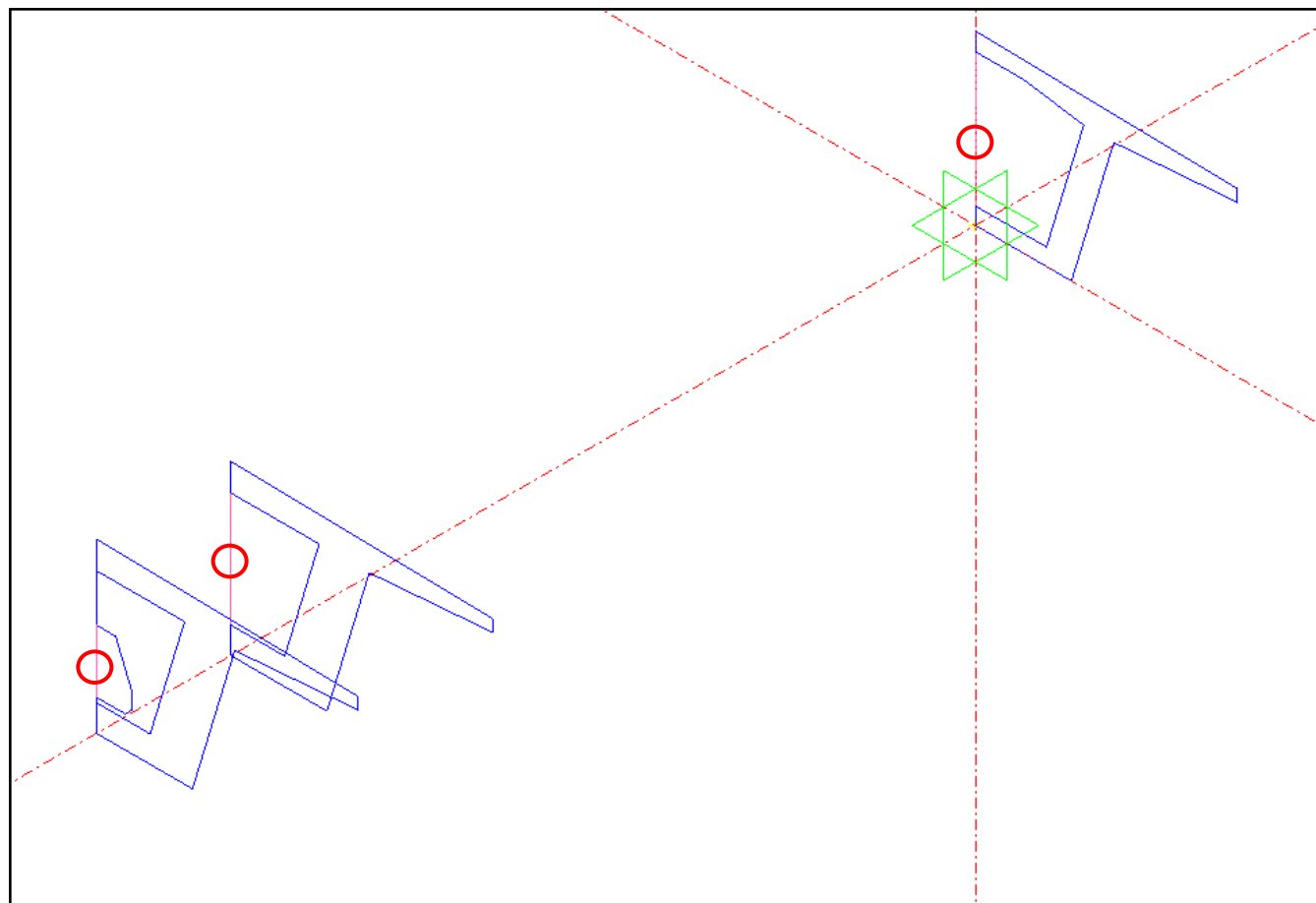


 “Ctrl+A” as shortcut for “Select Displayed All”.

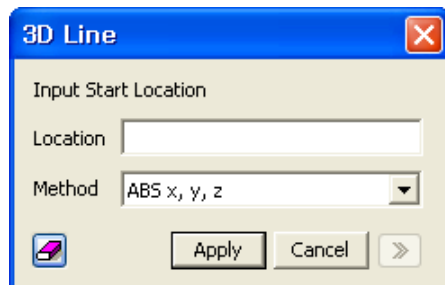
Step 9.



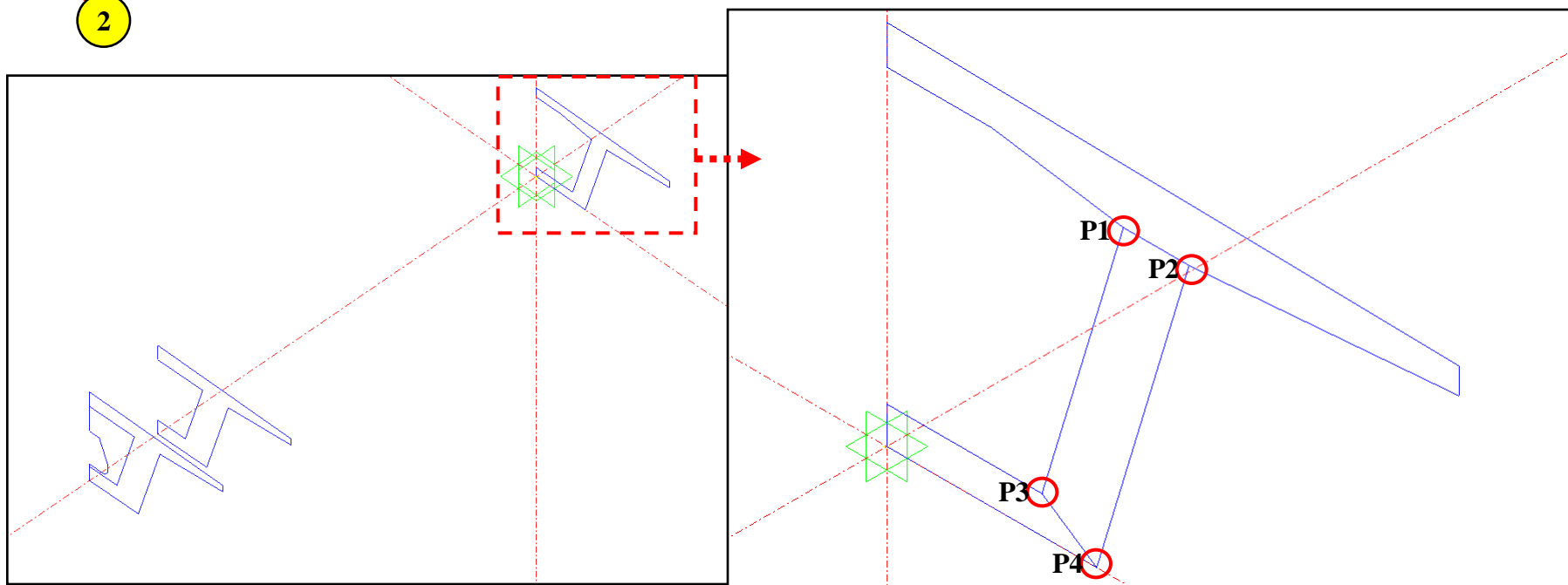
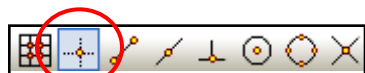
1. Select 3 Shapes(Edges) marked by “○” (See Figure)
2. Press [Delete] Key
3. Click [OK] Button



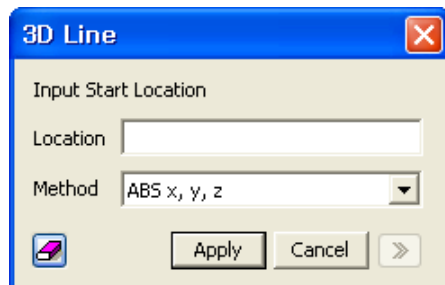
Step 10.



1. *Geometry > Curve > Create 3D > Line...*
2. *Toggle on “Vertex Snap”*
3. *Select P1 & P2 in sequential order (See Figure)*
4. *Select P3 & P4 in sequential order (See Figure)*

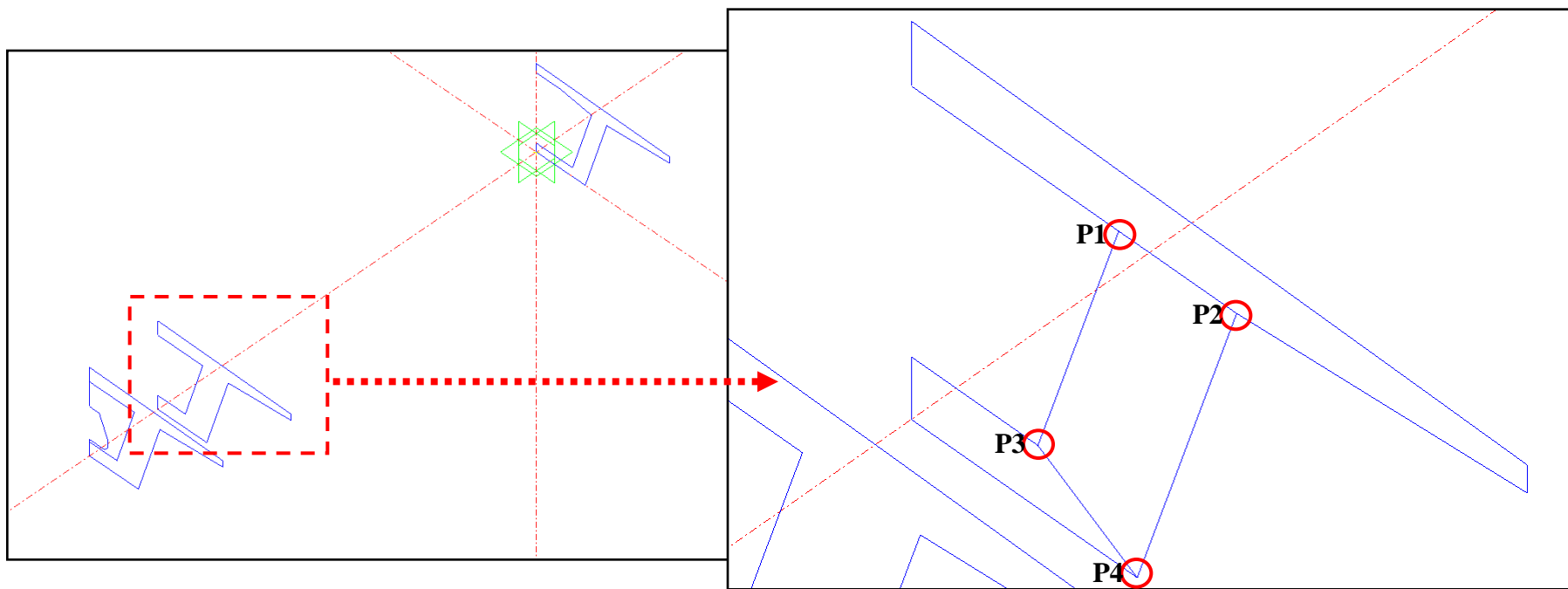


Step 11.

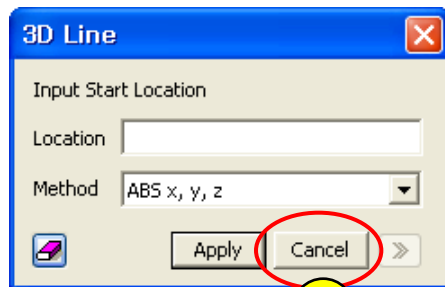


1. Select P1 & P2 in sequential order (See Figure)

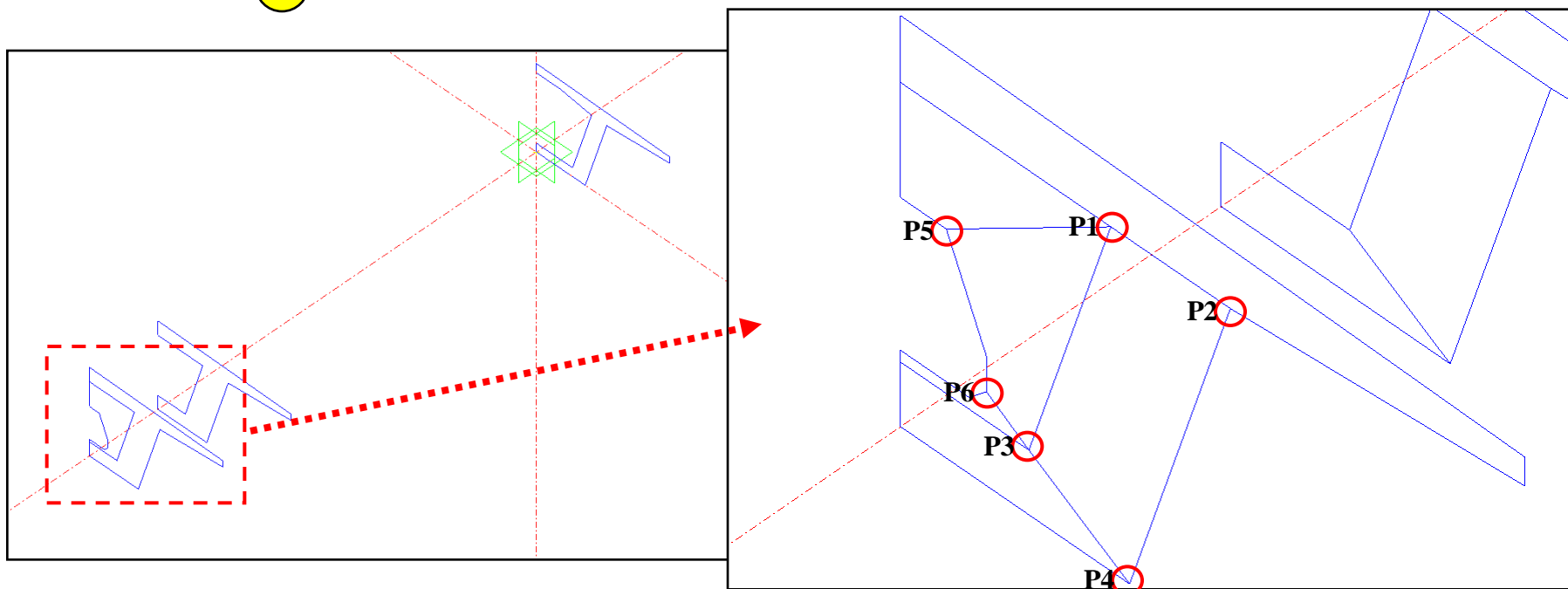
2. Select P3 & P4 in sequential order (See Figure)



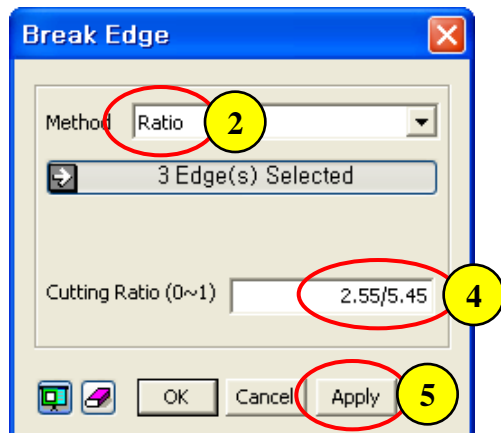
Step 12.



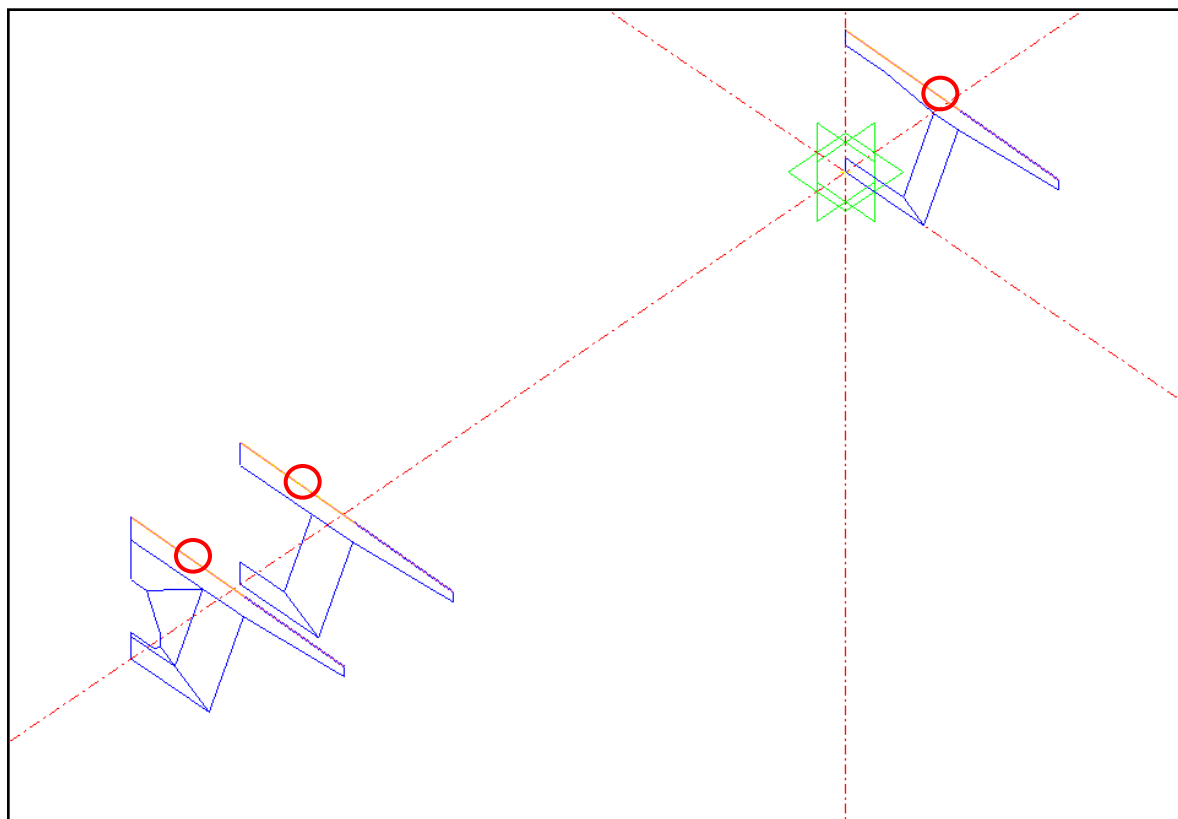
1. Select P1 & P2 in sequential order (See Figure)
2. Select P3 & P4 in sequential order (See Figure)
3. Select P5 & P1 in sequential order (See Figure)
4. Select P6 & P3 in sequential order (See Figure)
5. Click [Cancel] Button



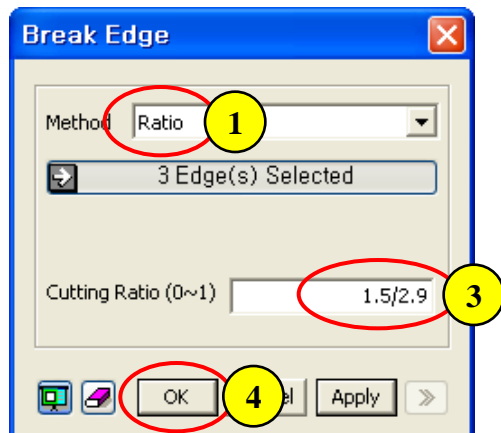
Step 13.



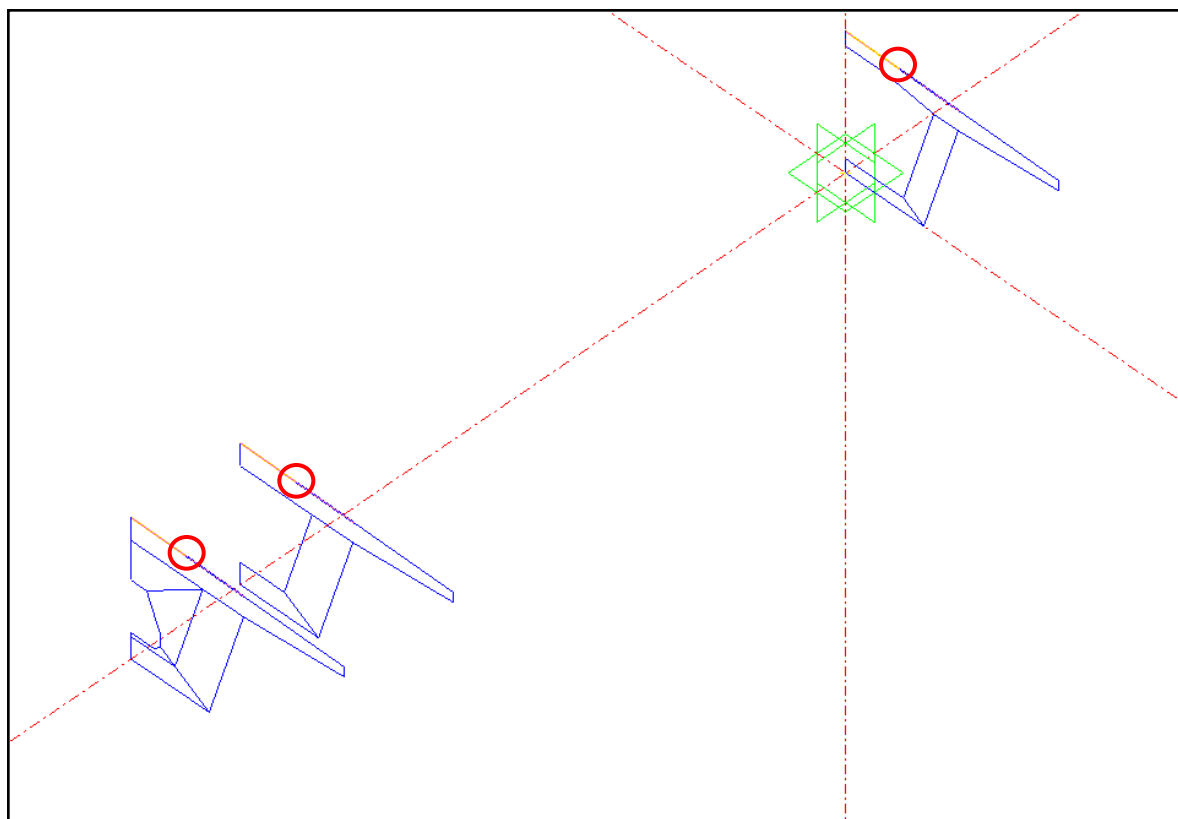
1. *Geometry > Curve > Break ...*
2. *Method : Ratio*
3. *Select 3 Edges marked by “○” (See Figure)*
4. *Cutting Ratio (0~1) : 2.55/5.45*
5. *Click [Apply] Button*



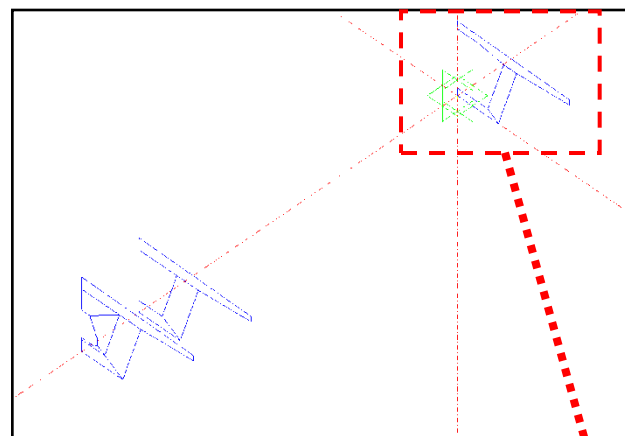
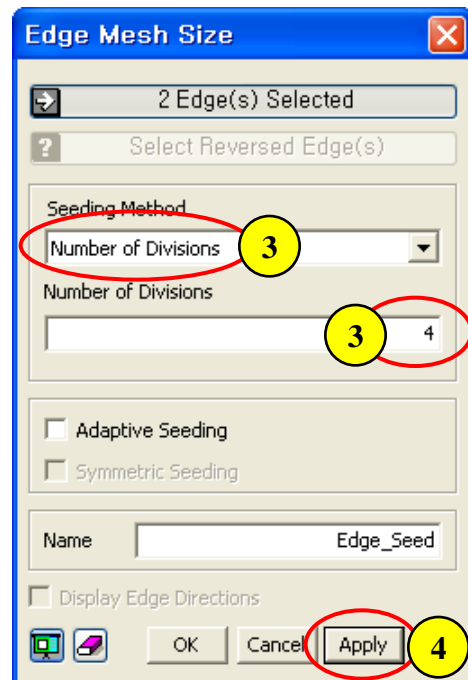
Step 14.



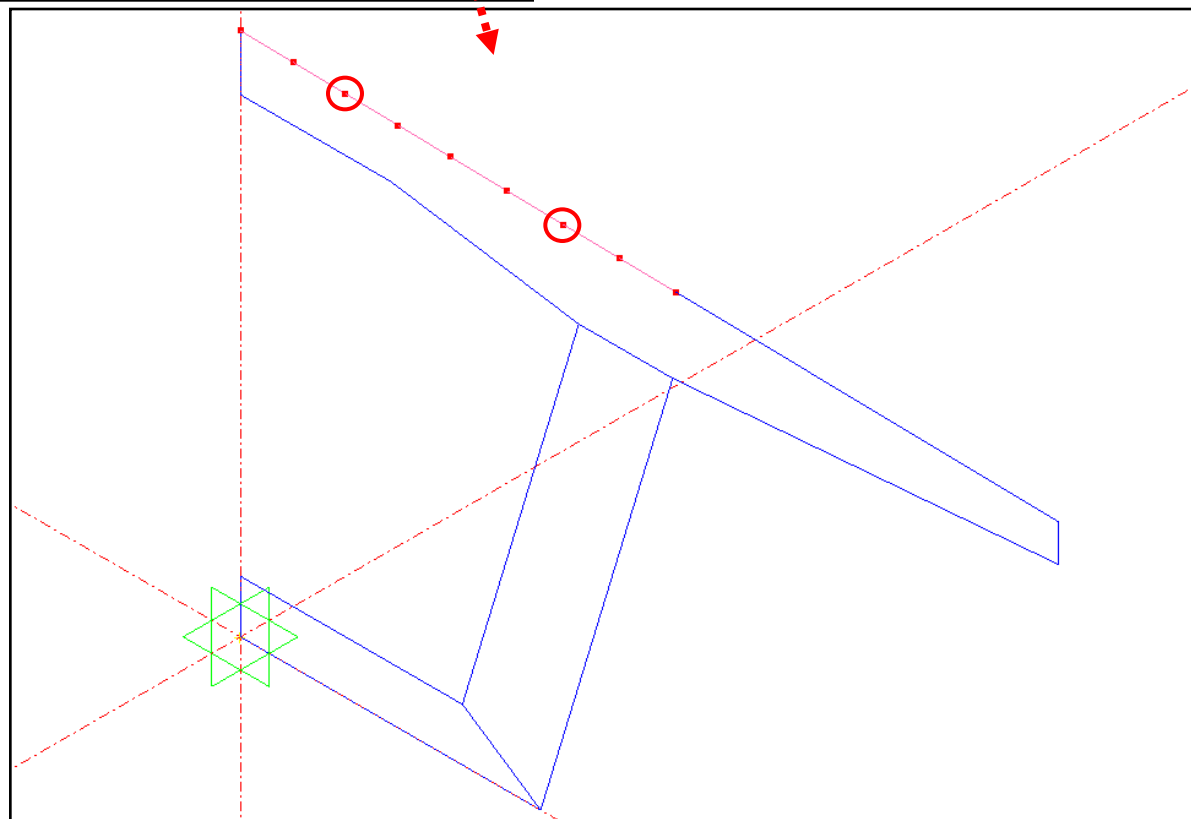
1. Method : Ratio
2. Select 3 Edges marked by “O” (See Figure)
3. Cutting Ratio (0~1) : 1.5/2.9
4. Click [OK] Button



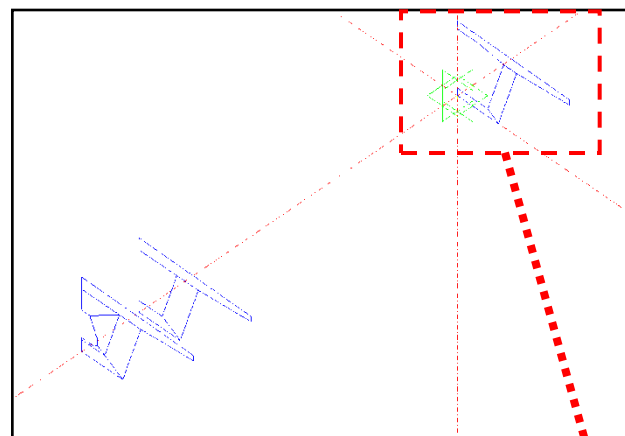
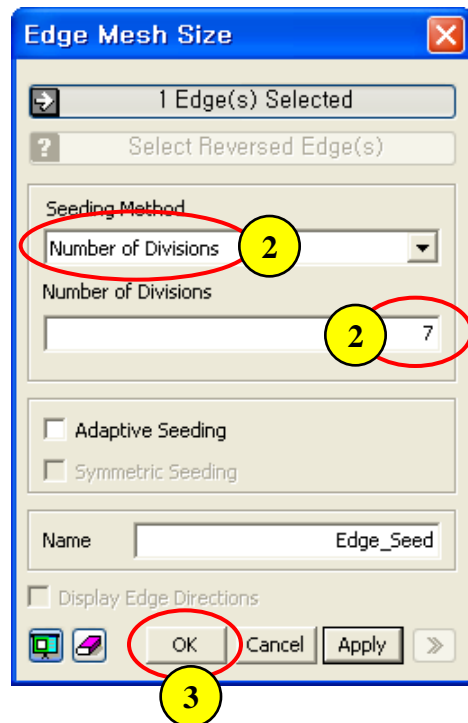
Step 15.



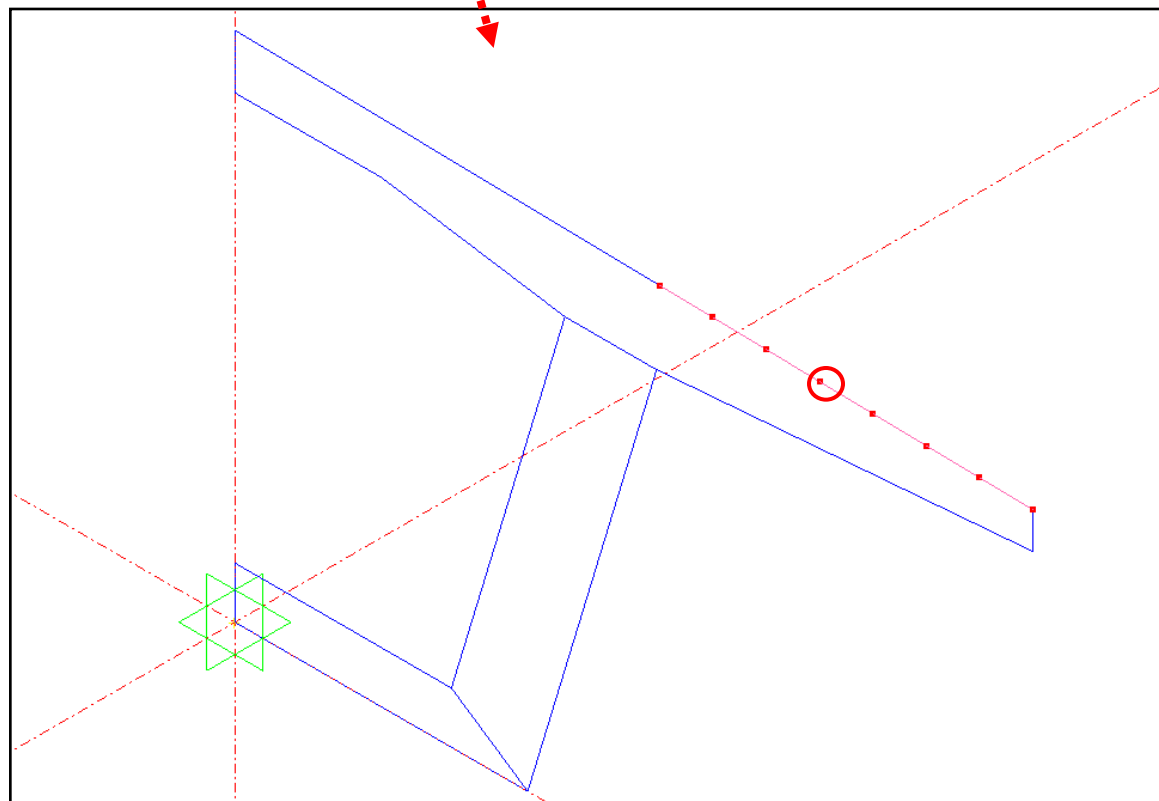
1. Mesh > Size Control > Along Edge ...
2. Select 2 Edges marked by "O" (See Figure)
3. Number of Divisions (4)
4. Click [Apply] Button



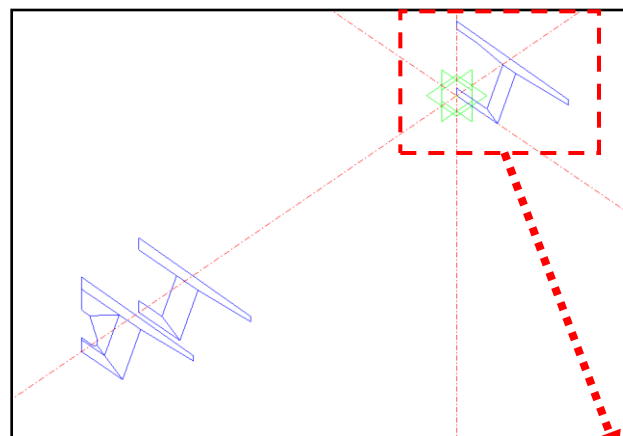
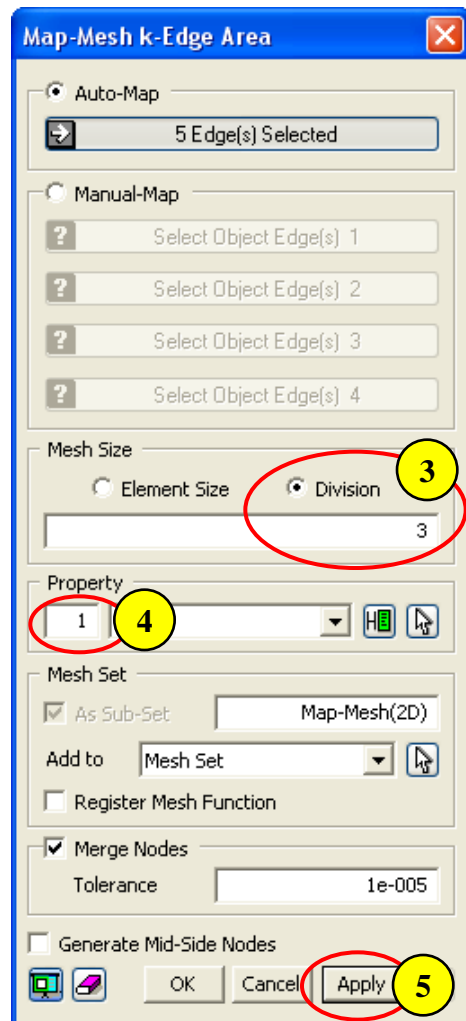
Step 16.



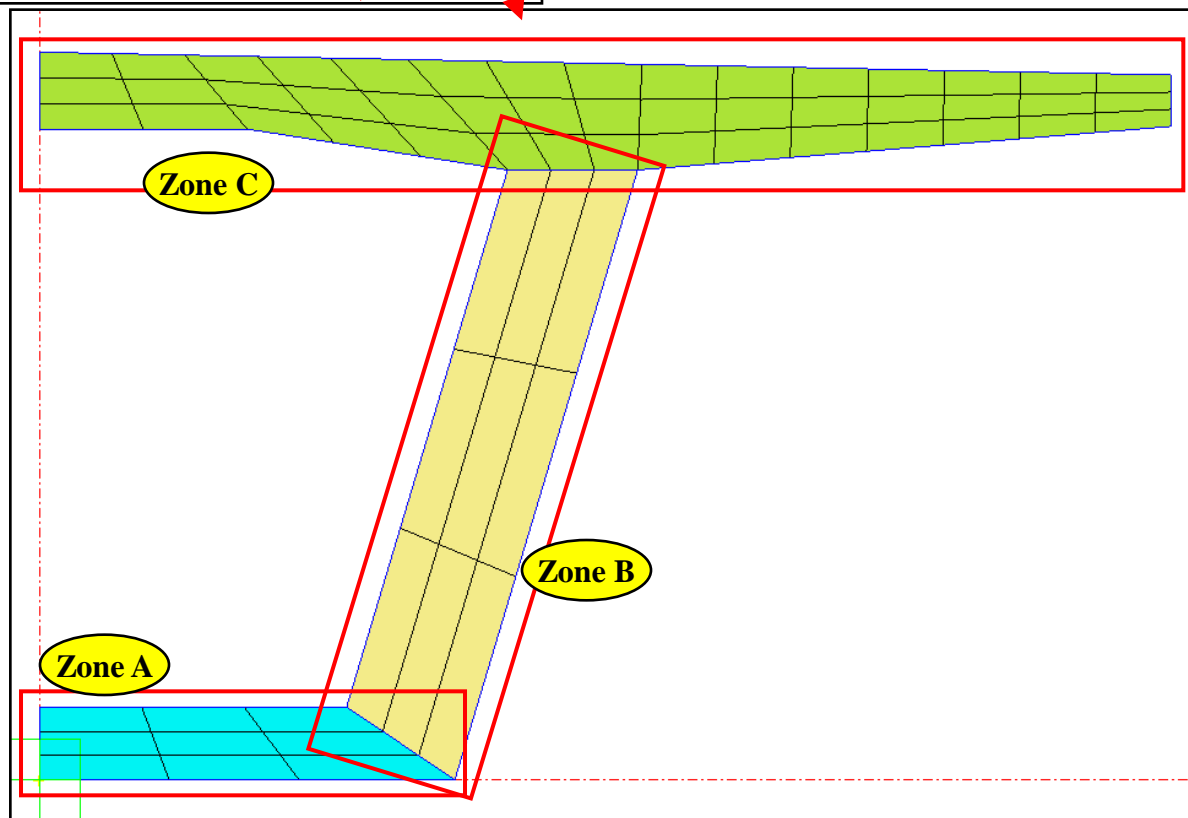
1. Select 1 Edge marked by "O" (See Figure)
2. Number of Divisions (7)
3. Click [OK] Button



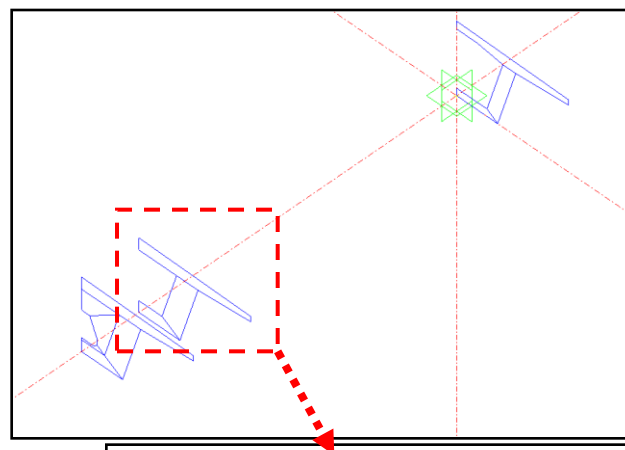
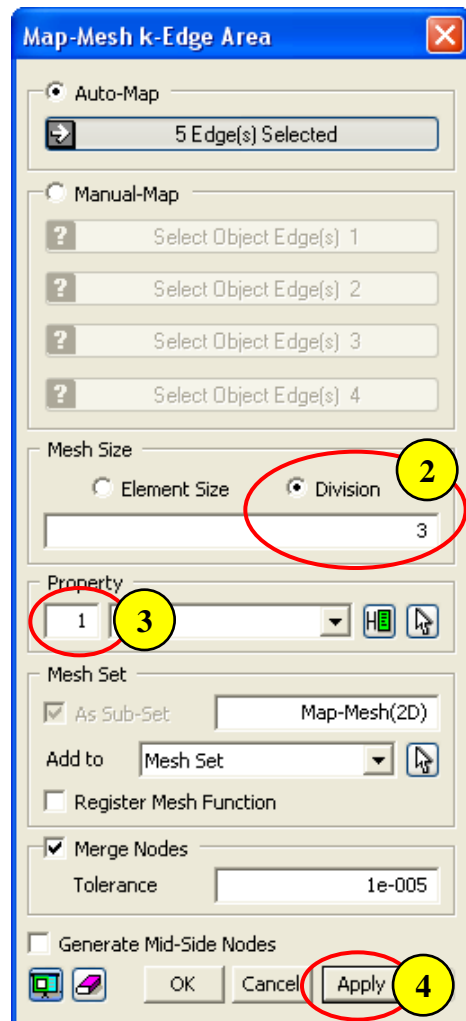
Step 17.



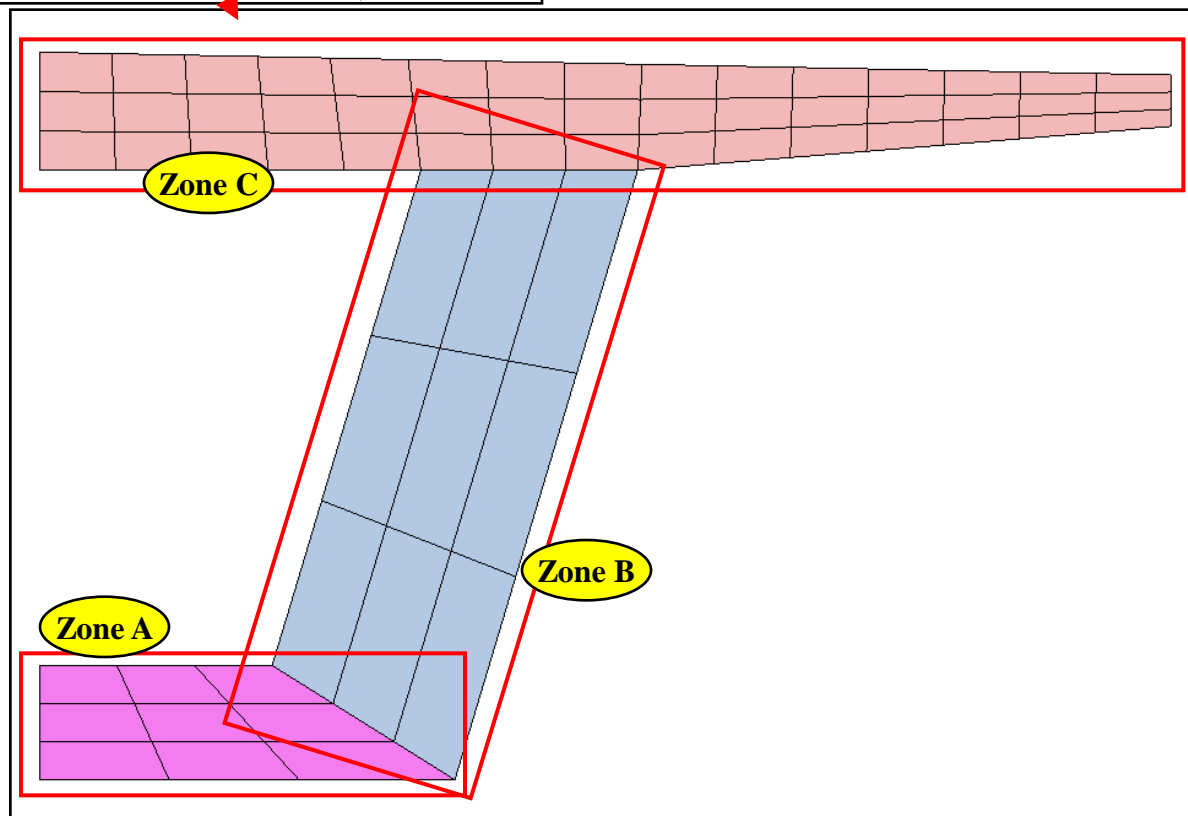
1. Mesh > Map Mesh > k-Edge Area ...
2. Select 5 Edges of "Zone A" (See Figure)
3. Mesh Size : Division (3)
4. Property (1)
5. Click [Apply] Button
6. Repeat step 2~5 for "Zone B" & "Zone C"



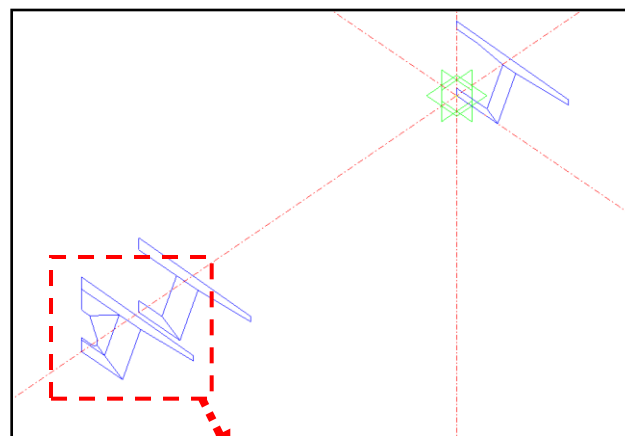
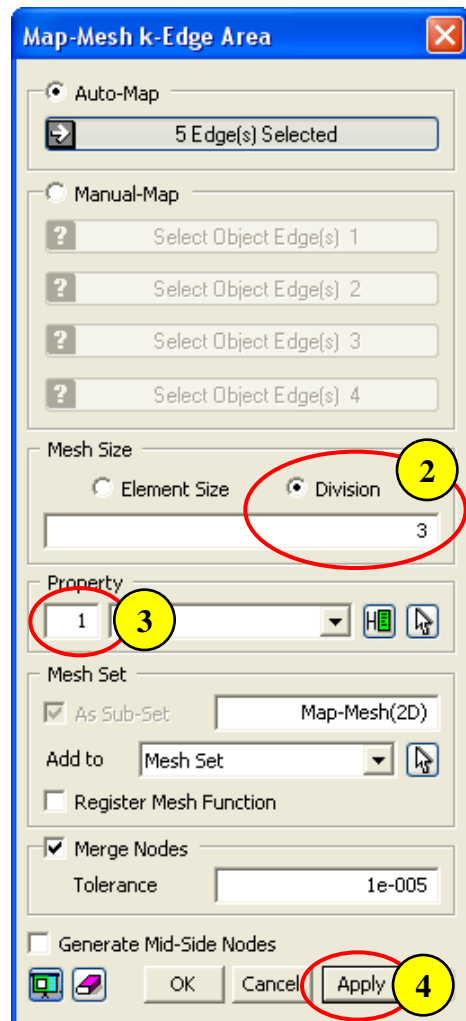
Step 18.



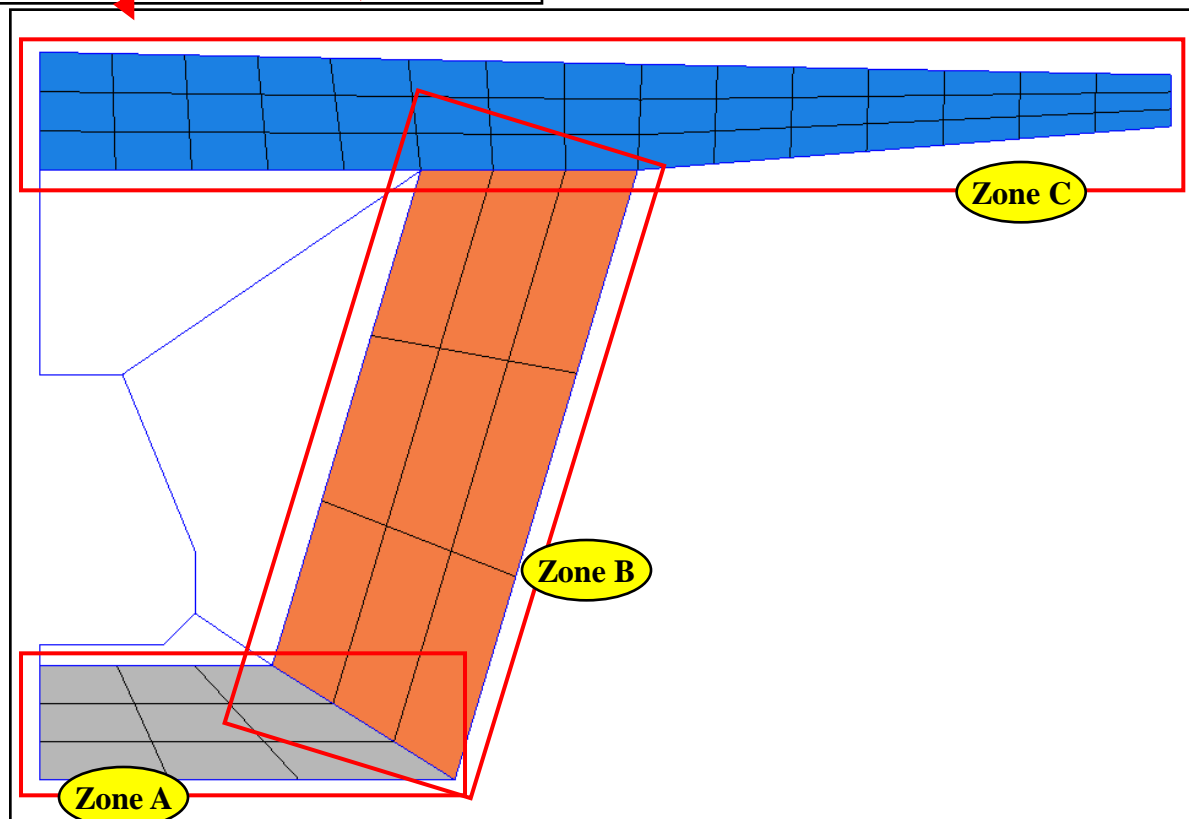
1. Select 5 Edges of "Zone A" (See Figure)
2. Mesh Size : Division (3)
3. Property (1)
4. Click [Apply] Button
5. Repeat step 1~4 for "Zone B" & "Zone C"



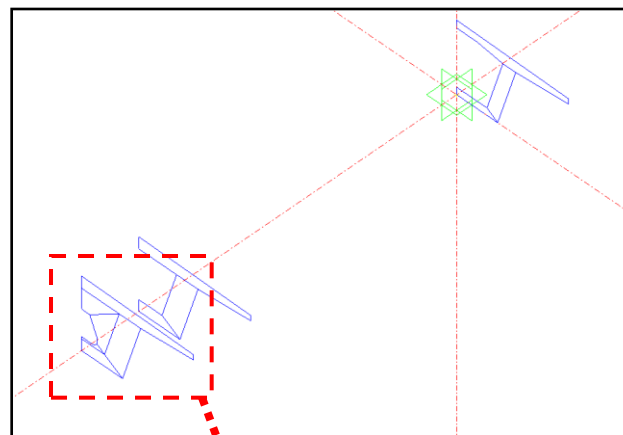
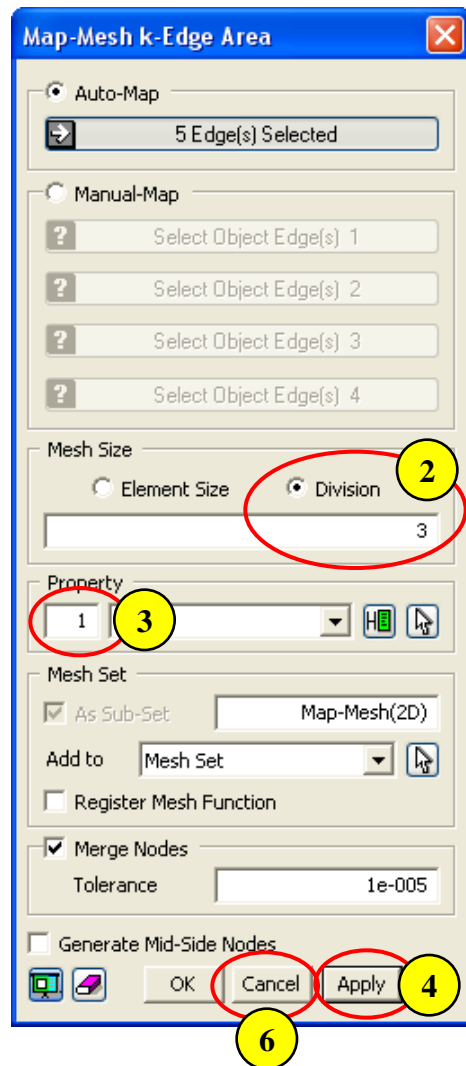
Step 19.



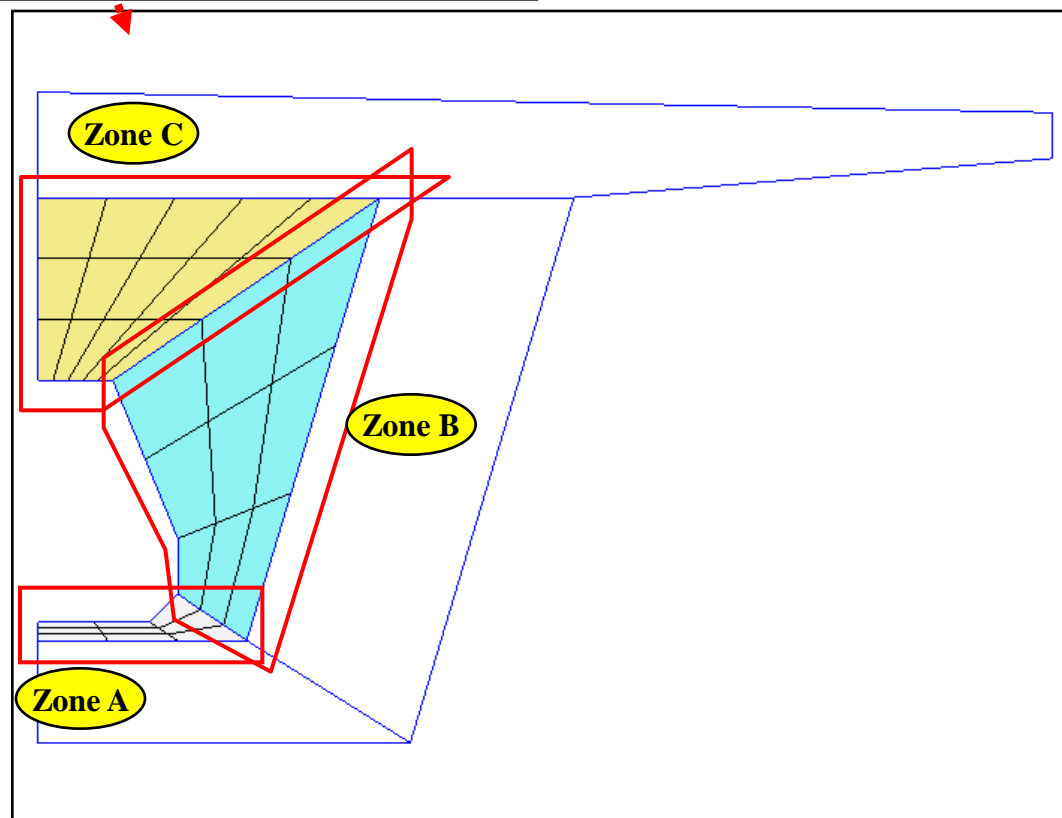
1. Select 5 Edges of "Zone A" (See Figure)
2. Mesh Size : Division (3)
3. Property (1)
4. Click [Apply] Button
5. Repeat step 1~4 for "Zone B" & "Zone C"



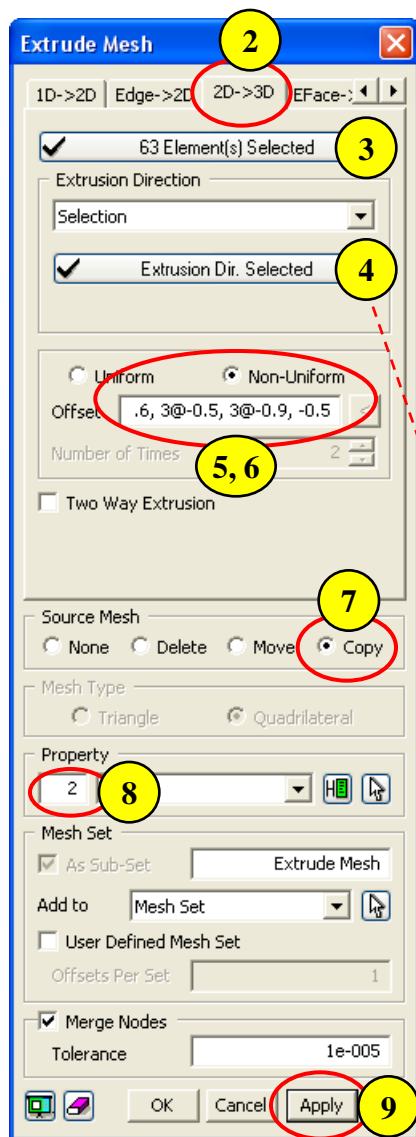
Step 20.



1. Select 5 Edges of "Zone A" (See Figure)
2. Mesh Size : Division (3)
3. Property (1)
4. Click [Apply] Button
5. Repeat step 1~4 for "Zone B" & "Zone C"
6. Click [Cancel] Button



Step 21.



1. Mesh > Protrude Mesh > Extrude ...

2. Select "2D->3D" tab

3. Select 63 Elements (See Figure)

4. Extrusion Direction : Y-Axis

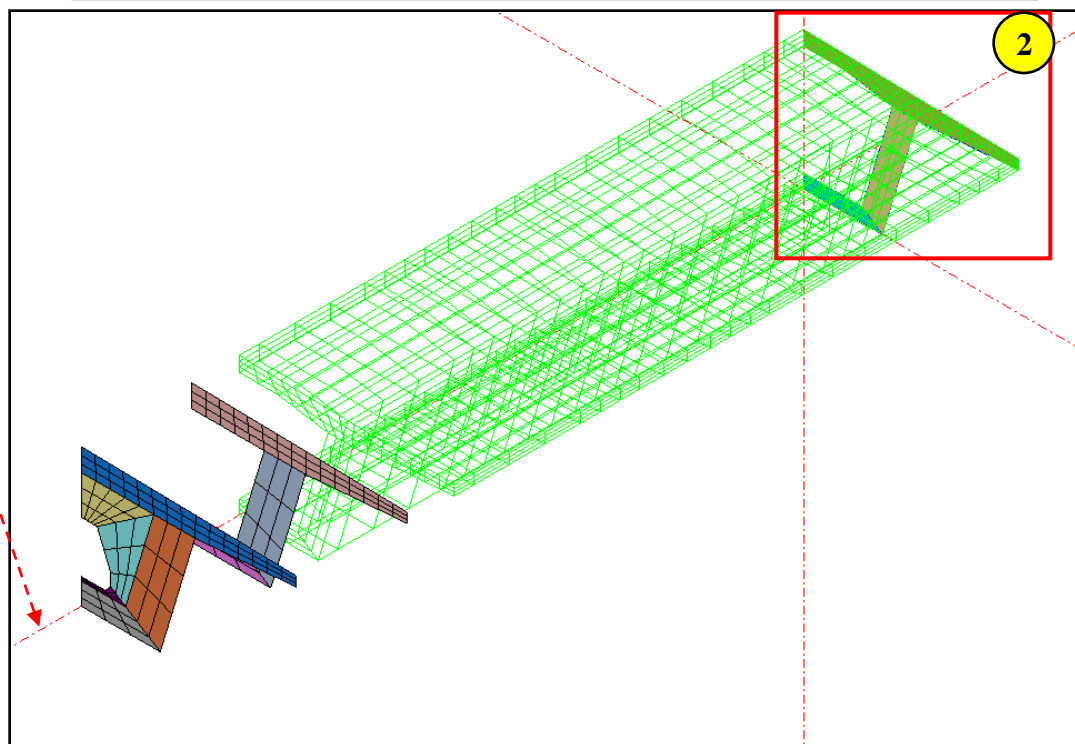
5. Check on "Non-Uniform"

6. Offset : 3@-0.5, 6@-0.8, 3@-0.5, 3@-0.6, 3@-0.5, 3@-0.9, -0.5

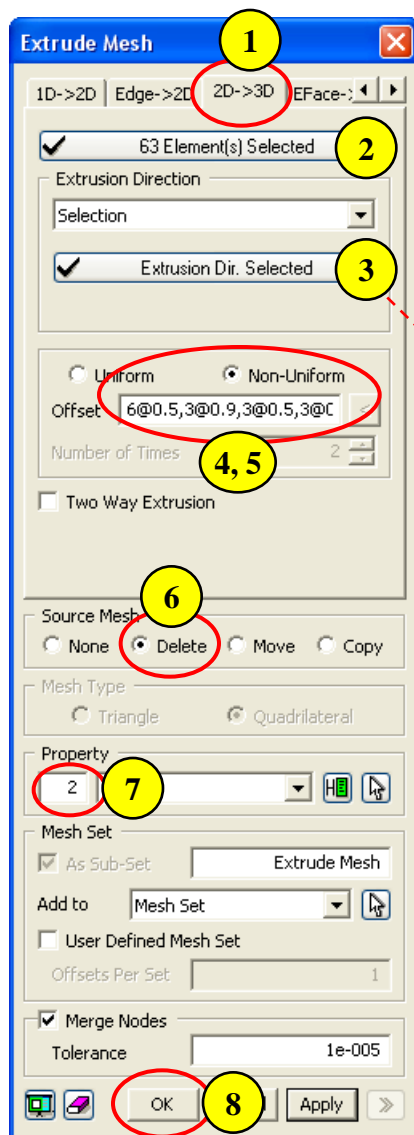
7. Source Mesh : Copy

8. Property (2)

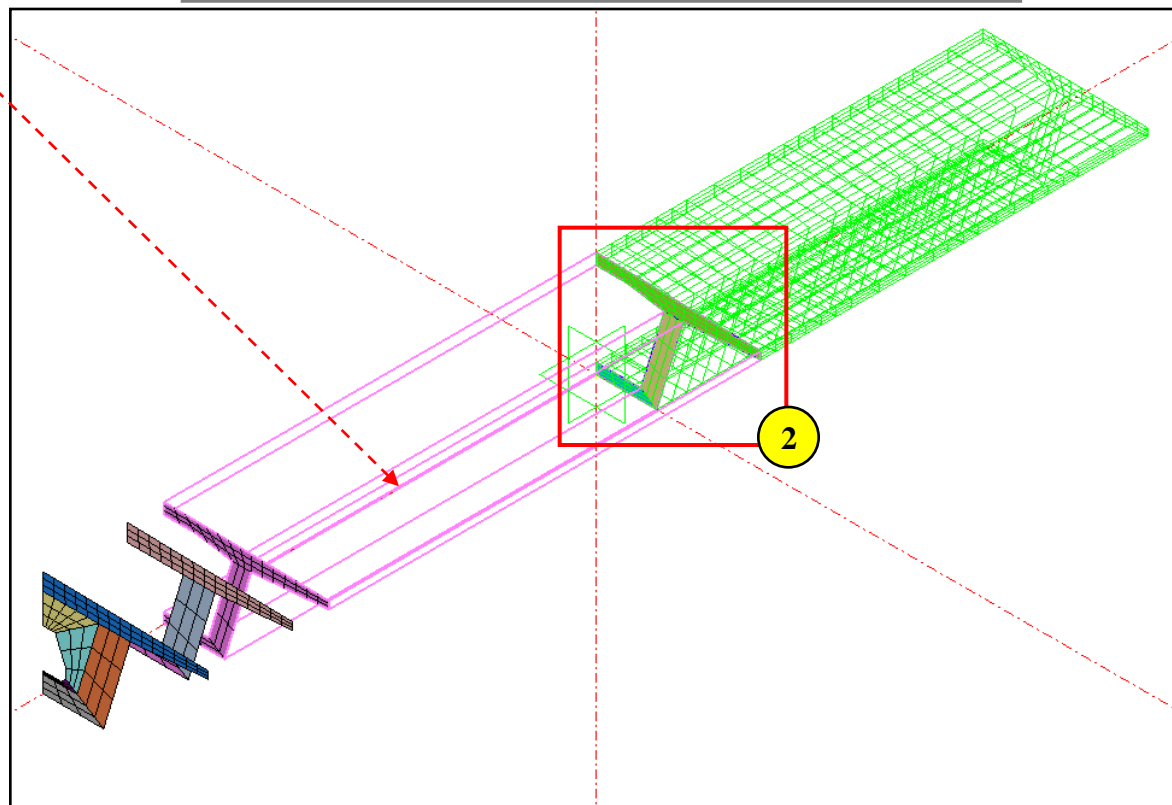
9. Click [Apply] Button



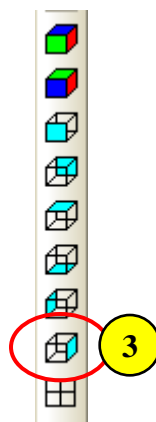
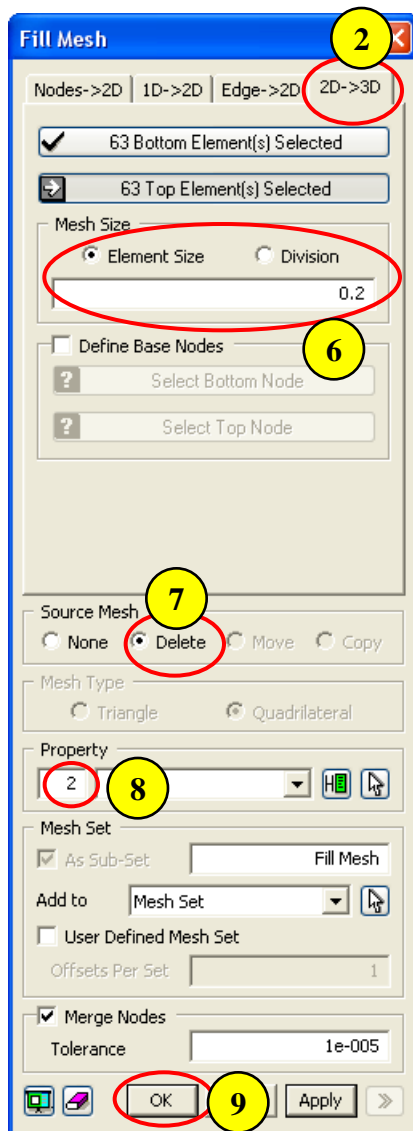
Step 22.



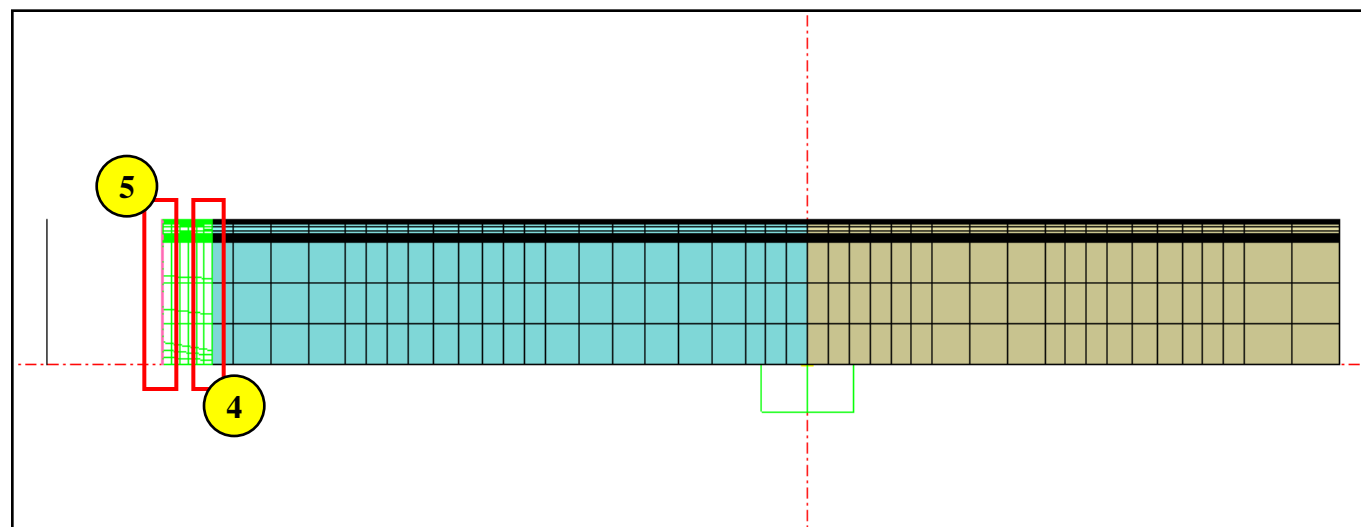
1. Select "2D->3D" tab
2. Select 63 Elements (See Figure)
3. Extrusion Direction : Y-Axis
4. Check on "Non-Uniform"
5. Offset : 6@0.5, 3@0.9, 3@0.5, 3@0.6, 6@0.5, 2@1.15
6. Source Mesh : Delete
7. Property (2)
8. Click [OK] Button



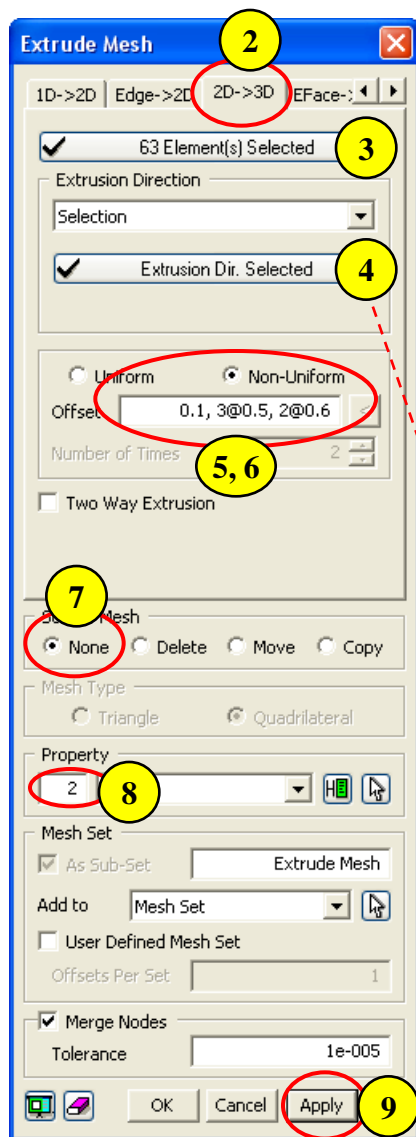
Step 23.



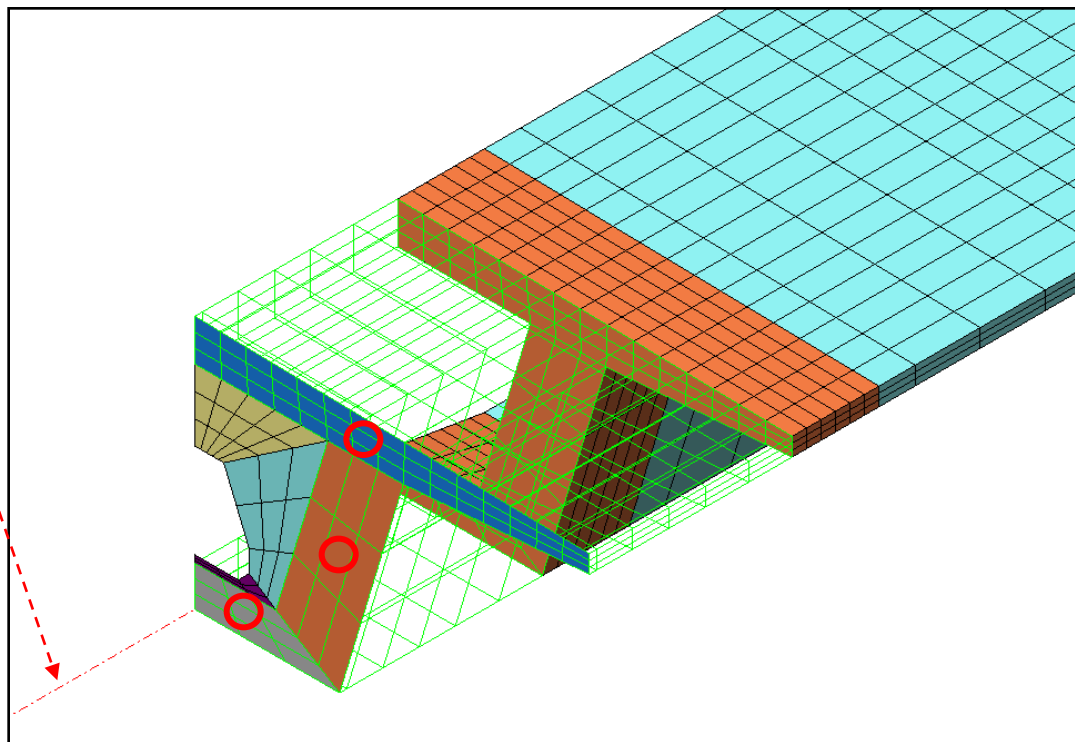
1. Mesh > Protrude Mesh > Fill ...
2. Select "2D->3D" tab
3. Click "Right View" Button
4. Select 63 Elements for Bottom (See Figure)
5. Select 63 Elements for Top (See Figure)
6. Mesh Size : Element Size (0.2)
7. Source Mesh : Delete
8. Property (2)
9. Click [OK] Button



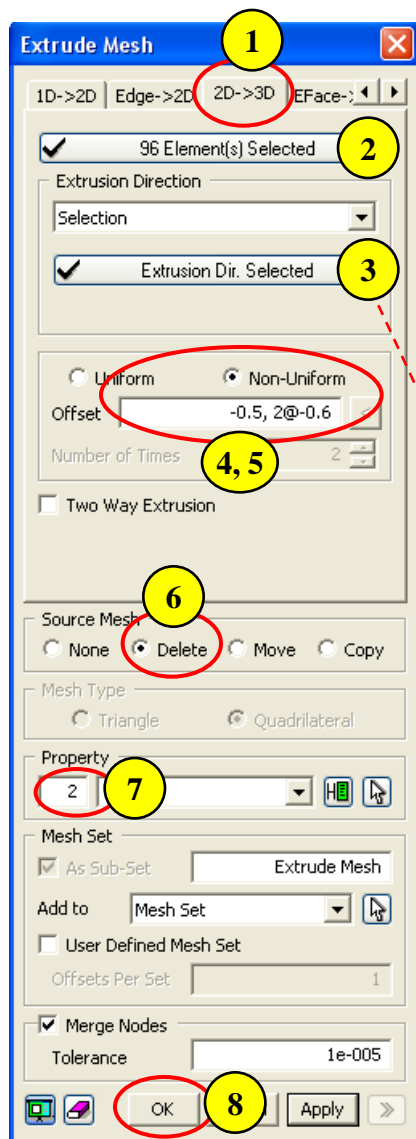
Step 24.



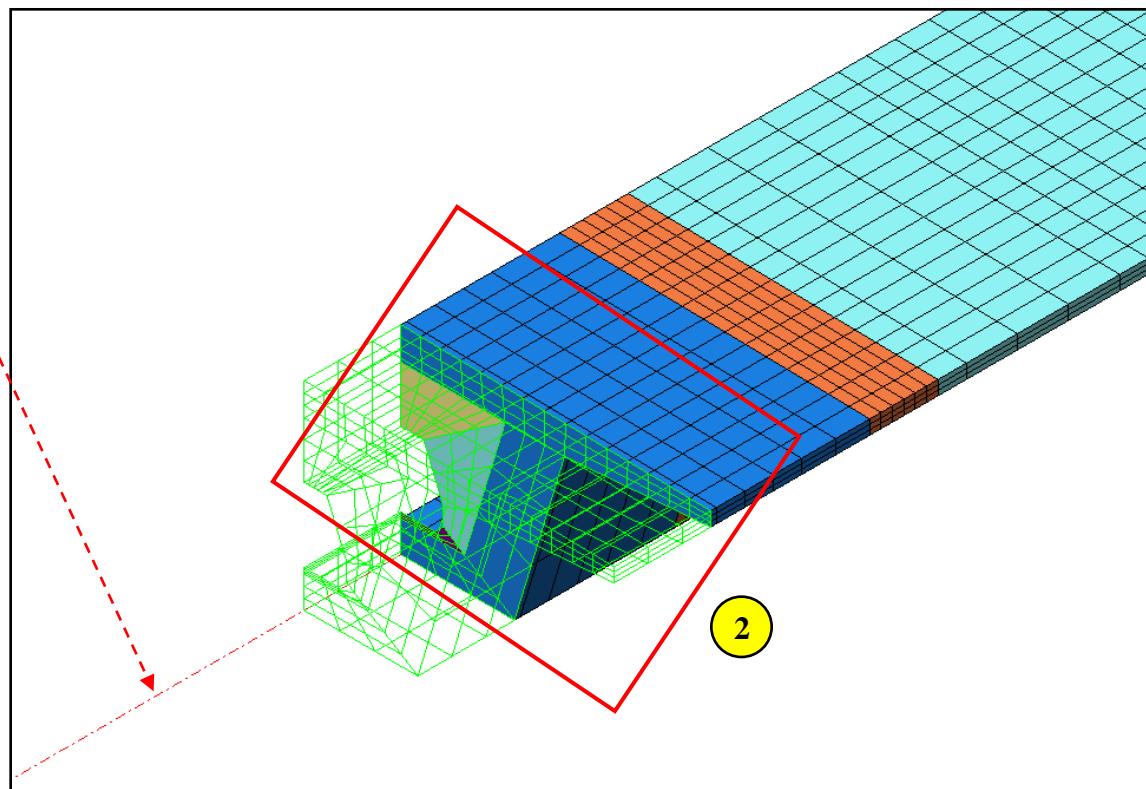
1. Mesh > Protrude Mesh > Extrude ...
2. Select "2D->3D" tab
3. Select 3 Mesh Set (63 Elements) marked by "O" (See Figure)
4. Extrusion Direction : Y-Axis
5. Check on "Non-Uniform"
6. Offset : 0.1, 3@0.5, 2@0.6
7. Source Mesh : None
8. Property (2)
9. Click [Apply] Button



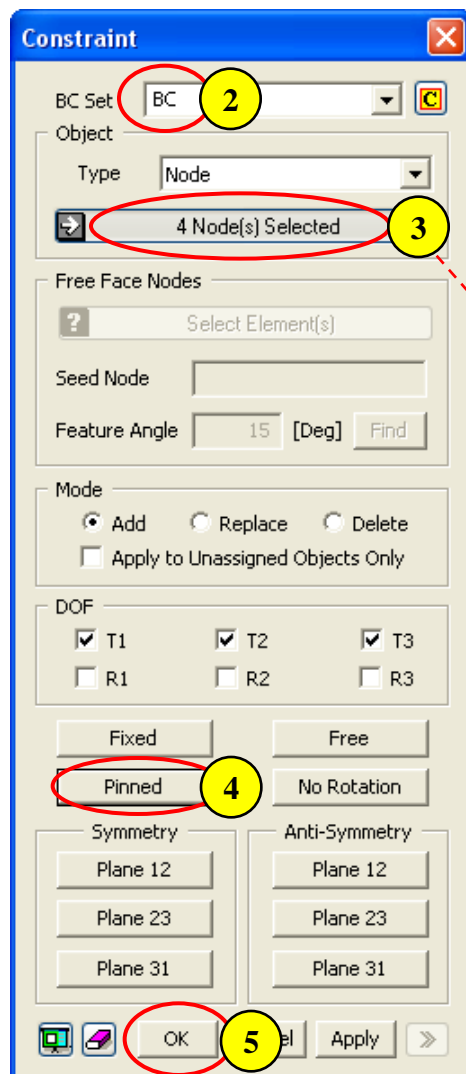
Step 25.



1. Select "2D->3D" tab
2. Select 96 Elements (See Figure)
3. Extrusion Direction : Y-Axis
4. Check on "Non-Uniform"
5. Offset : -0.5, 2@-0.6
6. Source Mesh : Delete
7. Property (2)
8. Click [OK] Button



Step 26.



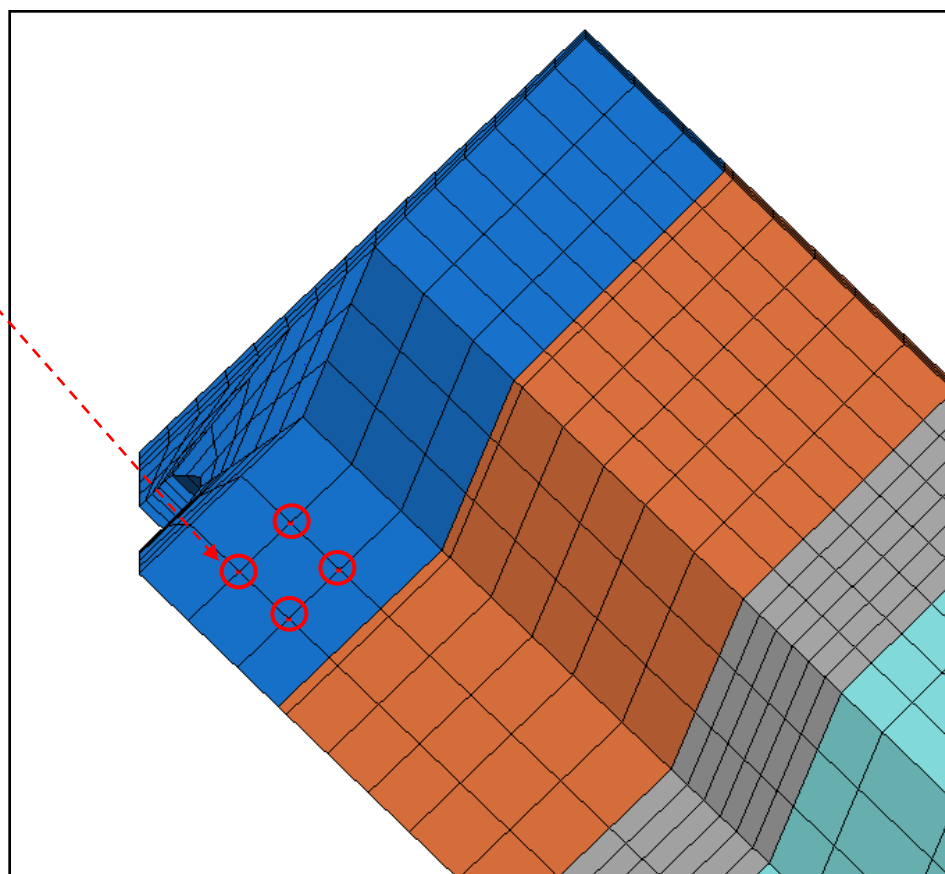
1. Analysis > BC > Constraint ...

2. BC Set : BC

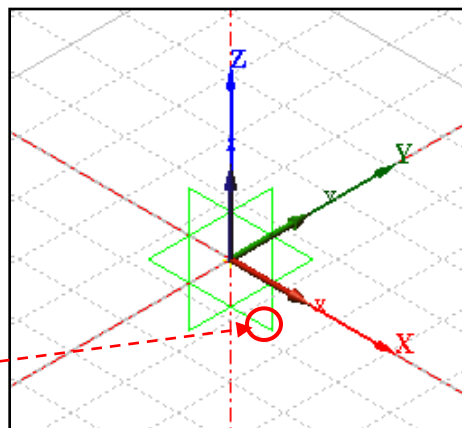
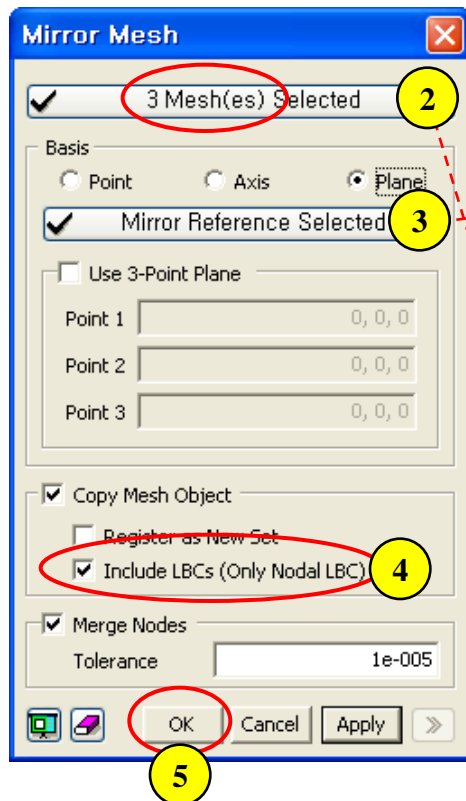
3. Select 4 Nodes of Bottom marked by "O" (See Figure)

4. Click [Pinned] Button

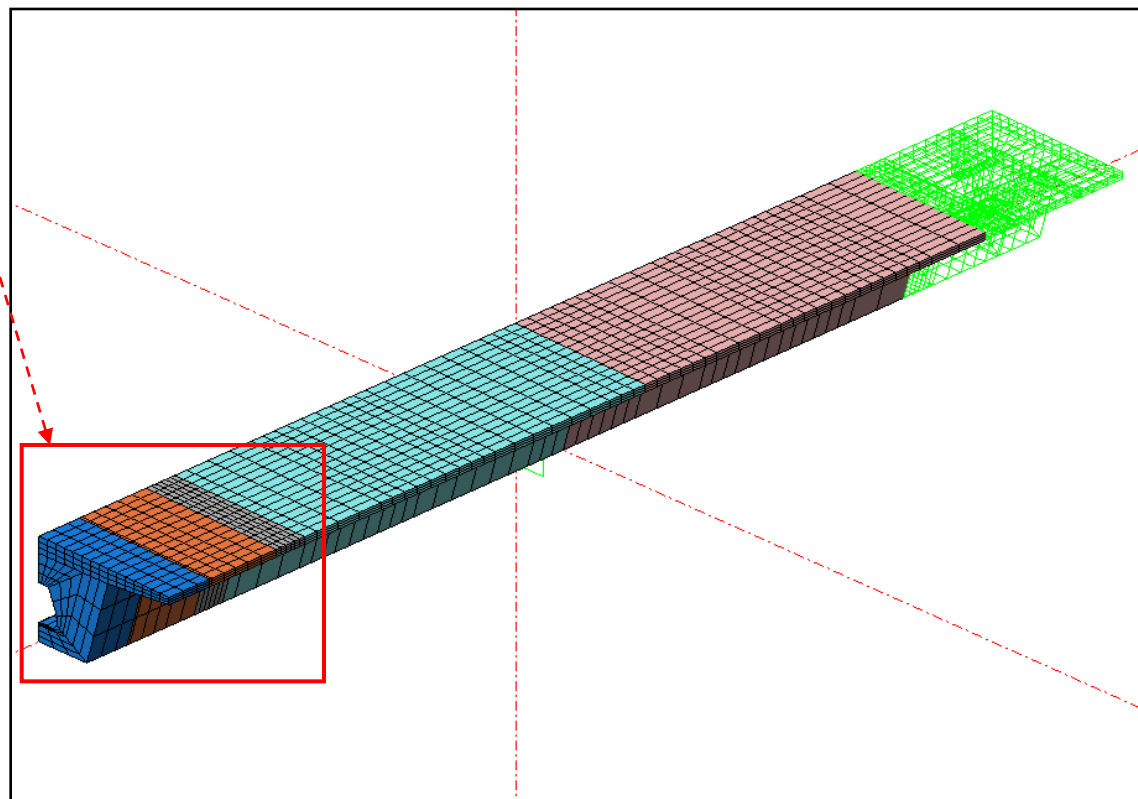
5. Click [OK] Button



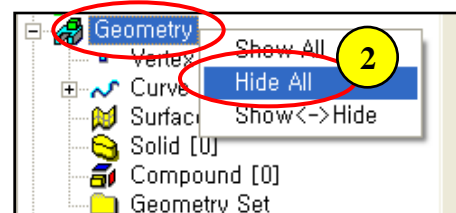
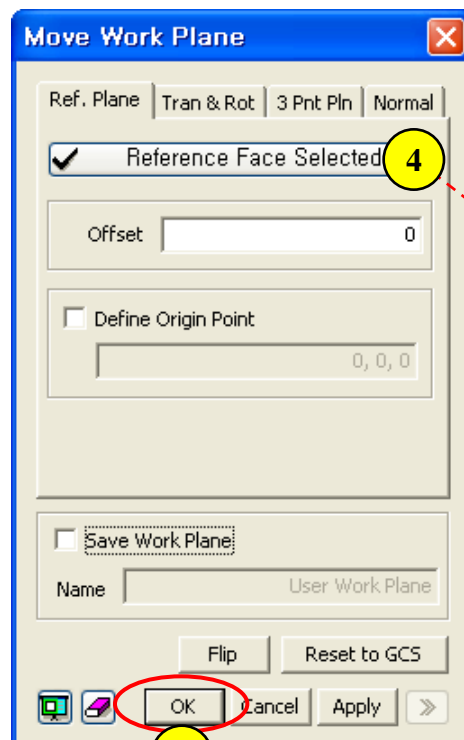
Step 27.



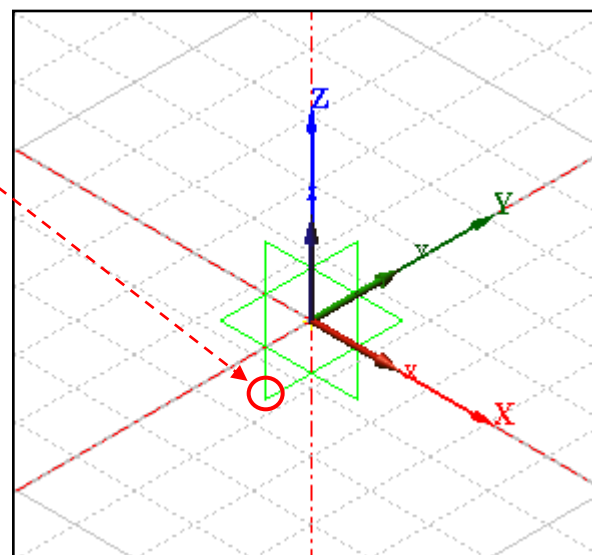
1. Mesh > Transform > Mirror ...
2. Select 3 Meshes (See Figure)
3. Mirror Plane : XZ-Plane
4. Check on "Include LBCs ..."
5. Click [OK] Button



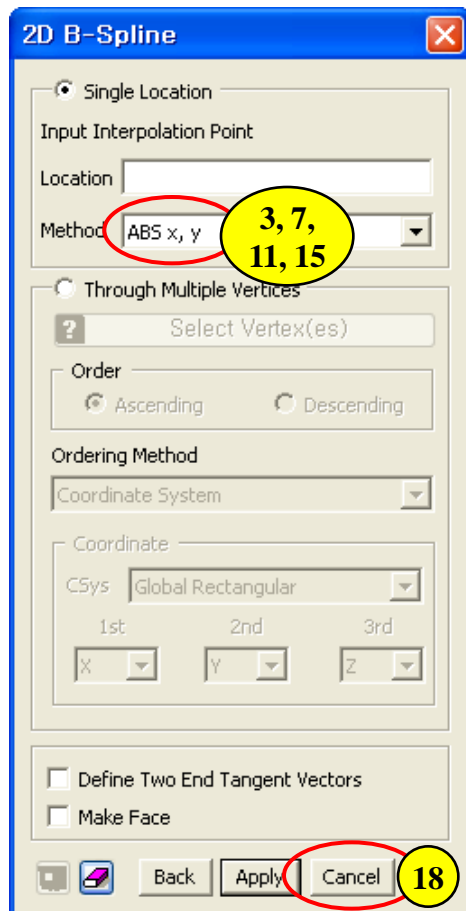
Step 28.



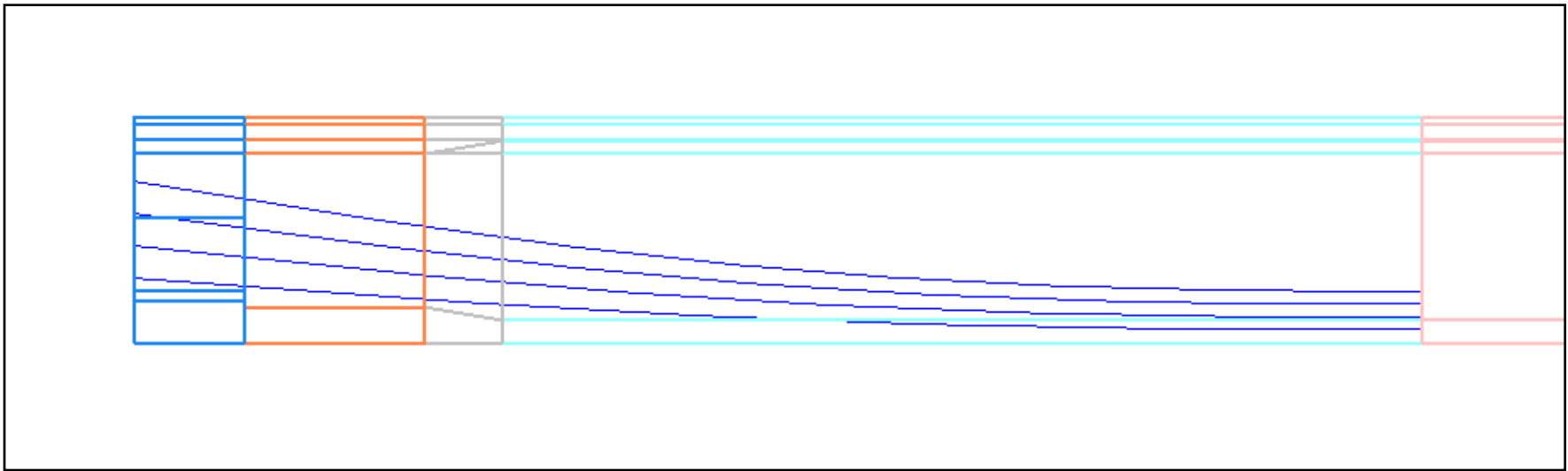
1. Pre-Works Tree : Geometry ...
2. Click Right Mouse Button and Select "Hide All"
3. Geometry > Work Plane > Move ...
4. Select "YZ-Plane"
5. Click [OK] Button



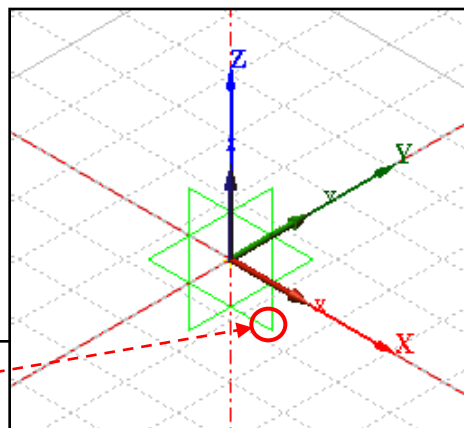
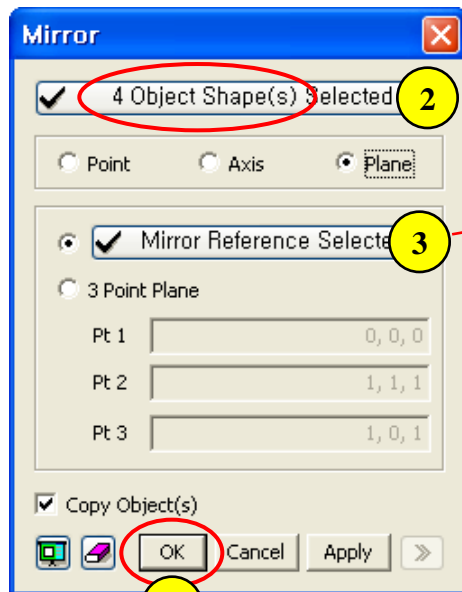
Step 29-1.



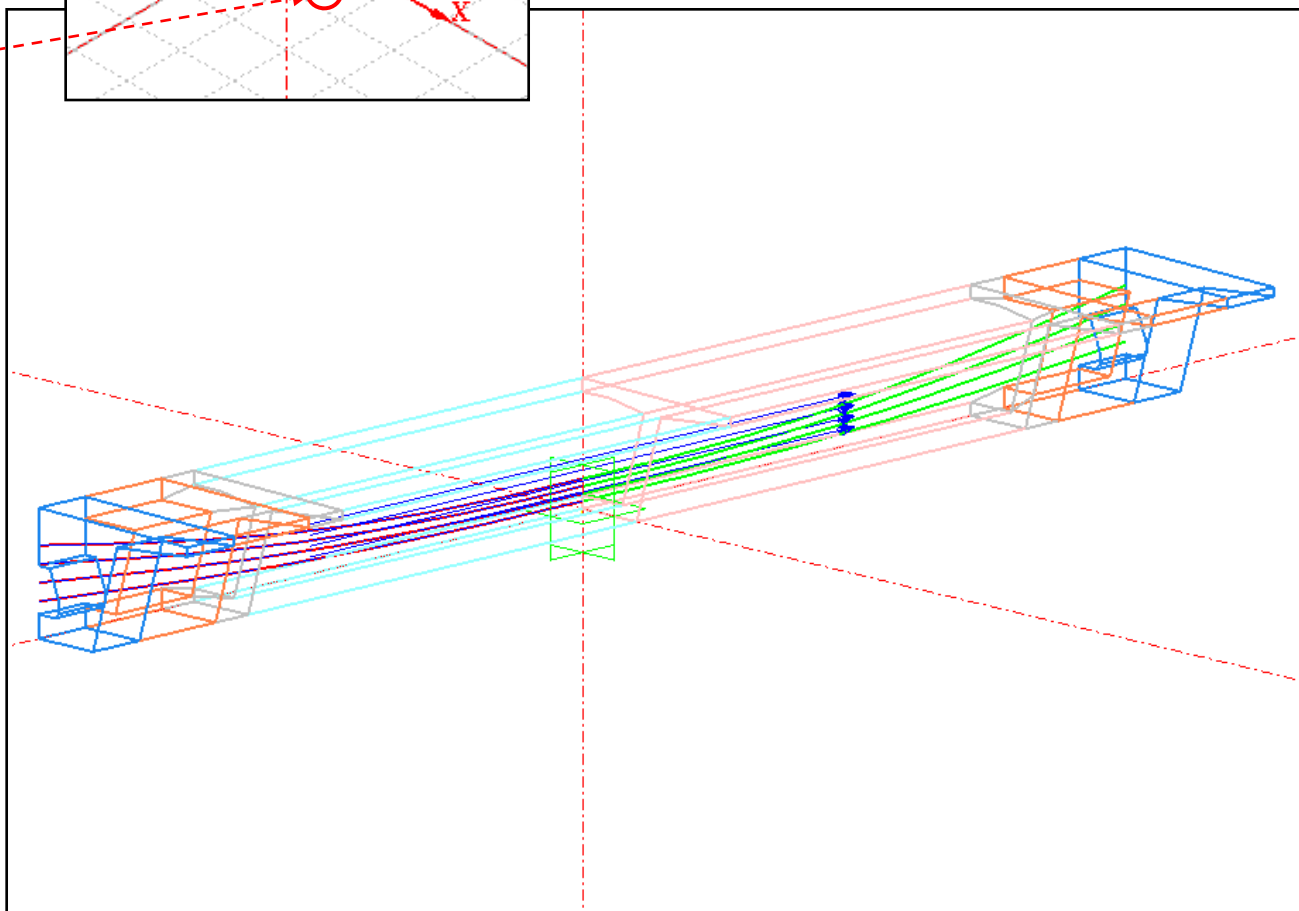
1. Geometry > Curve > Create on WP > B-Spline ...
2. Location : (-20, 2.5)
3. Change Method to "ABS x, y"
4. Location : (-18, 2.175), (-10, 1.147), (-2, 0.8), (0, 0.8)
5. Click Right Mouse Button on the Work Window
6. Location : (-20, 2)
7. Change Method to "ABS x, y"
8. Location : (-18, 1.732), (-10, 0.886), (-2, 0.6), (0, 0.6)
9. Click Right Mouse Button on the Work Window
10. Location : (-20, 1.5)
11. Change Method to "ABS x, y"
12. Location : (-18, 1.290), (-10, 0.625), (-2, 0.4), (0, 0.4)
13. Click Right Mouse Button on the Work Window
14. Location : (-20, 1)
15. Change Method to "ABS x, y"
16. Location : (-18, 0.847), (-10, 0.364), (-2, 0.2), (0, 0.2)
17. Click Right Mouse Button on the Work Window
18. Click [Cancel] Button

Step 29-2.

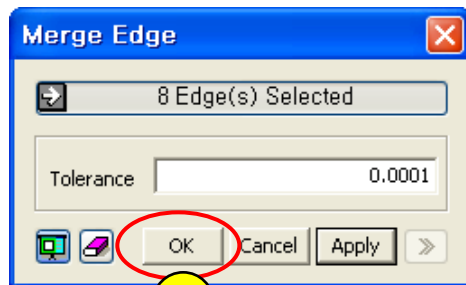
Step 30.




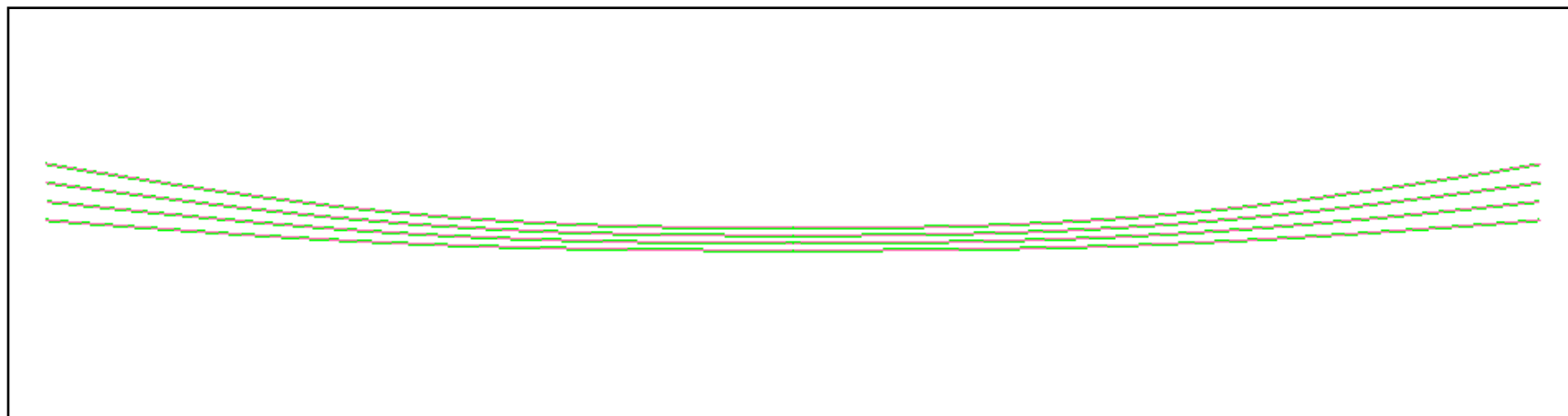
1. Geometry > Transform > Mirror ...
2. Select 4 B-Spline Curves
3. Mirror Plane : XZ-Plane
4. Click [OK] Button



Step 31.

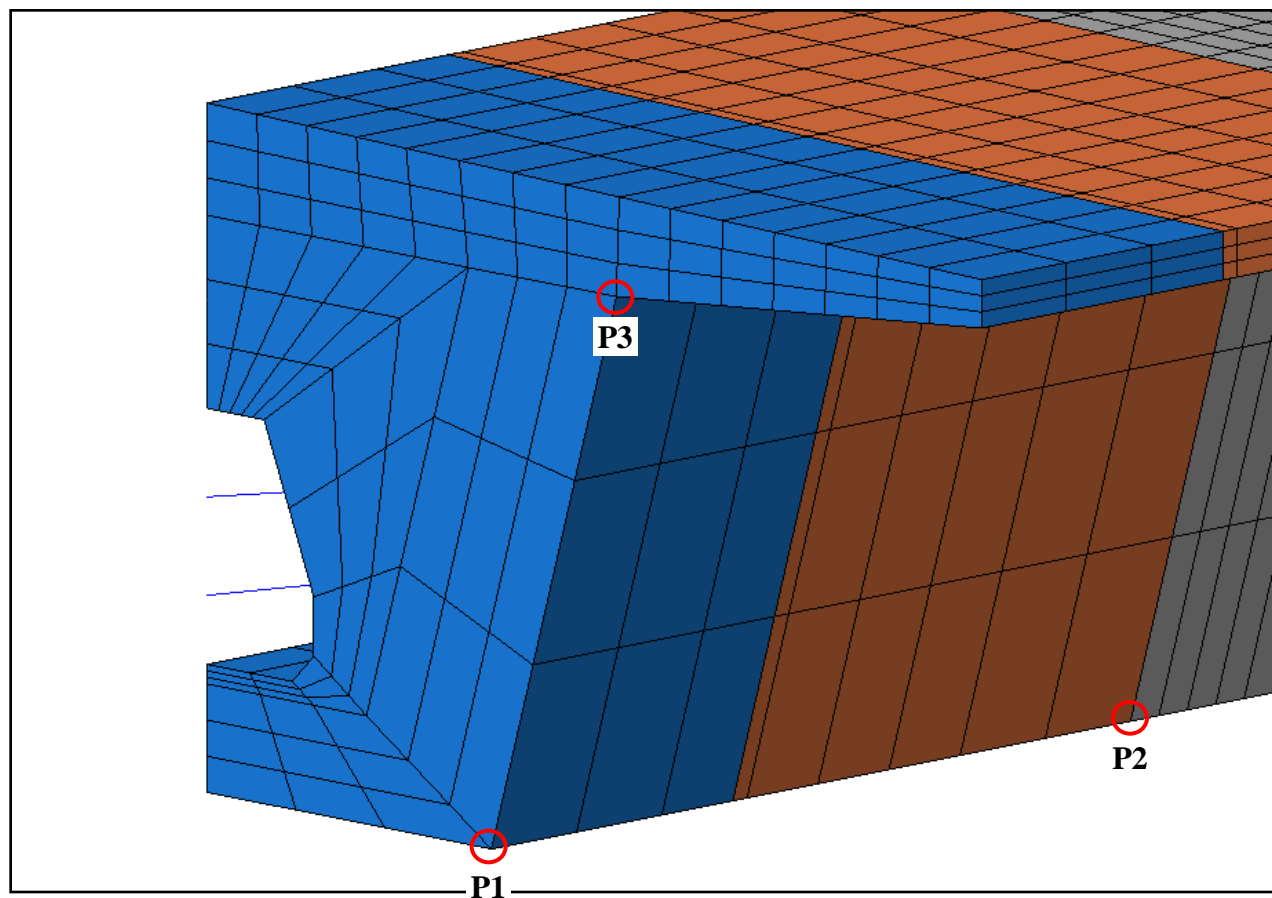
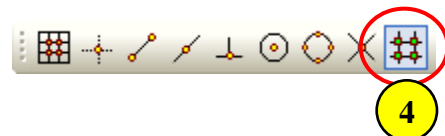
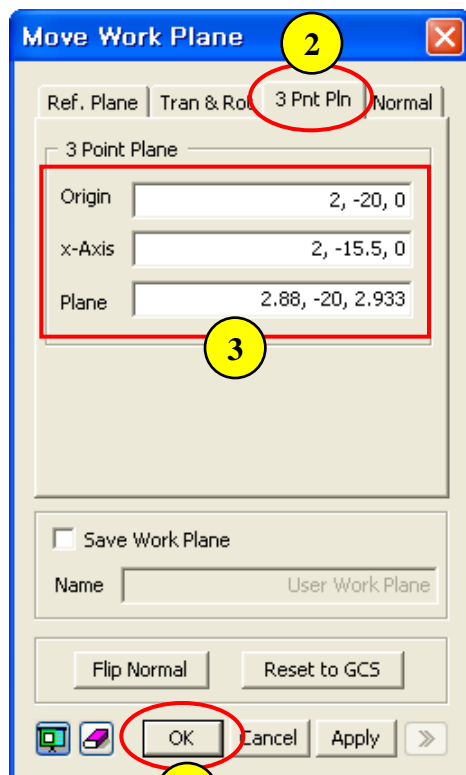


1. *Geometry > Curve > Merge ...*
2. Click  "Displayed All"
3. Click [OK] Button

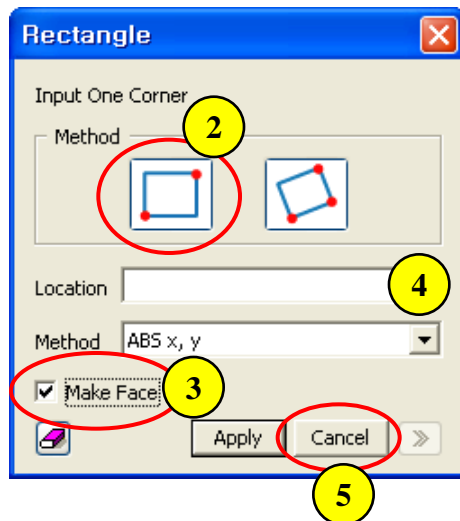


Step 32.

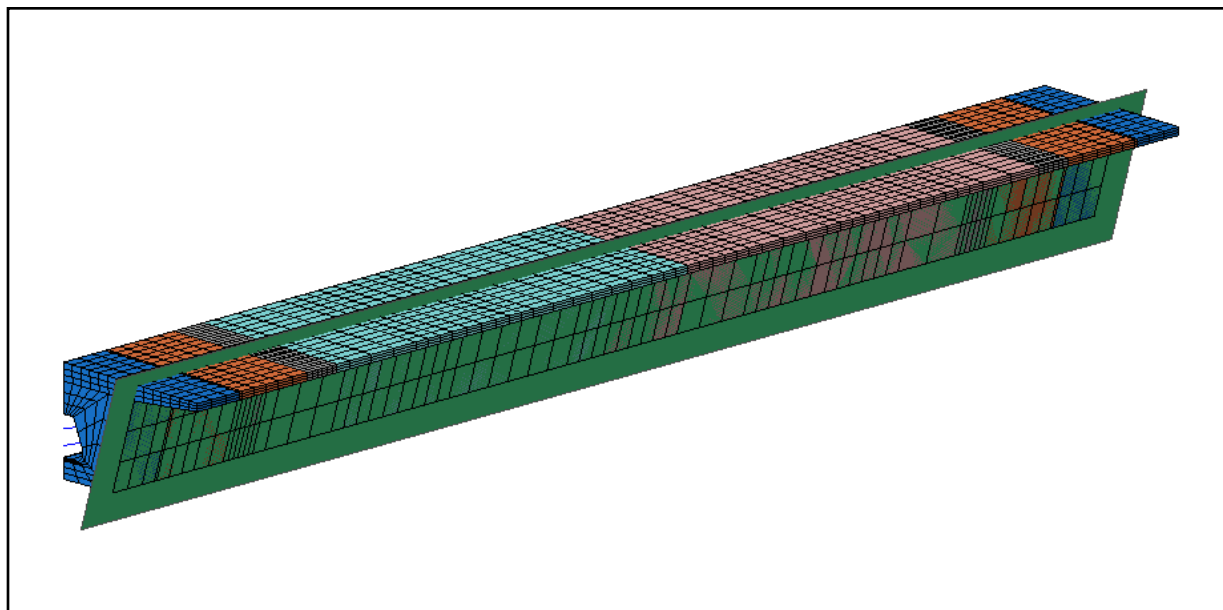
1. Geometry > Work Plane > Move ...
2. Select "3 Pnt Pln" tab
3. Toggle on only "Node Snap"
4. Select P1~P3 in sequential order (See Figure)
5. Click [OK] Button



Step 33.

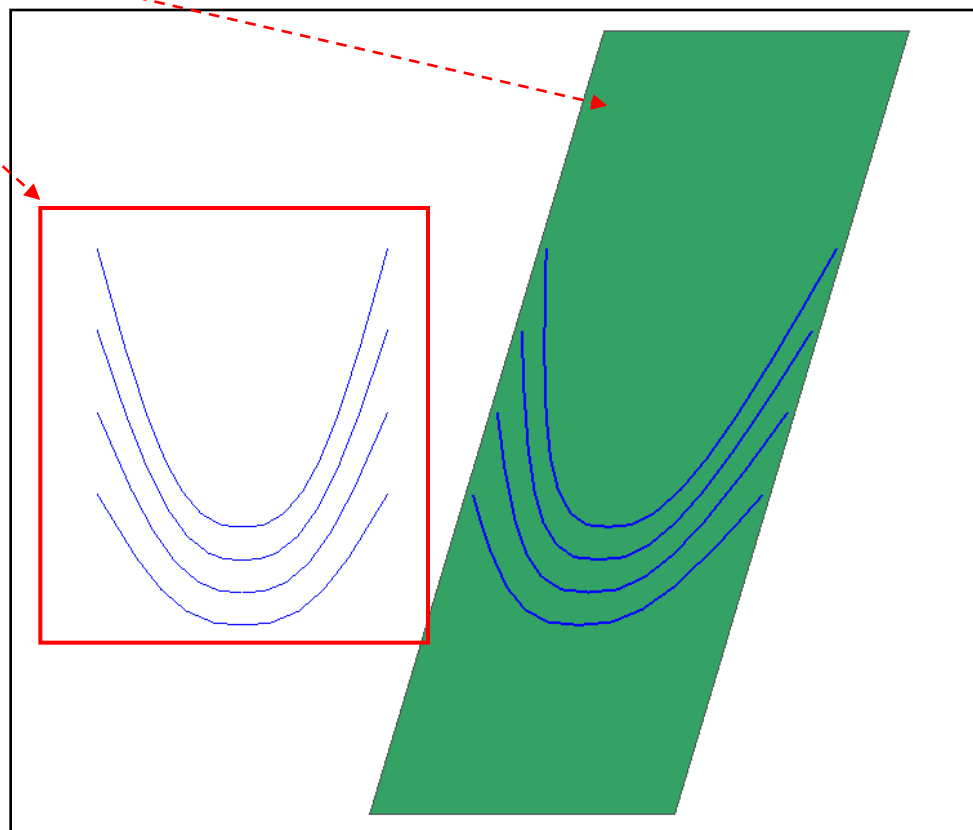
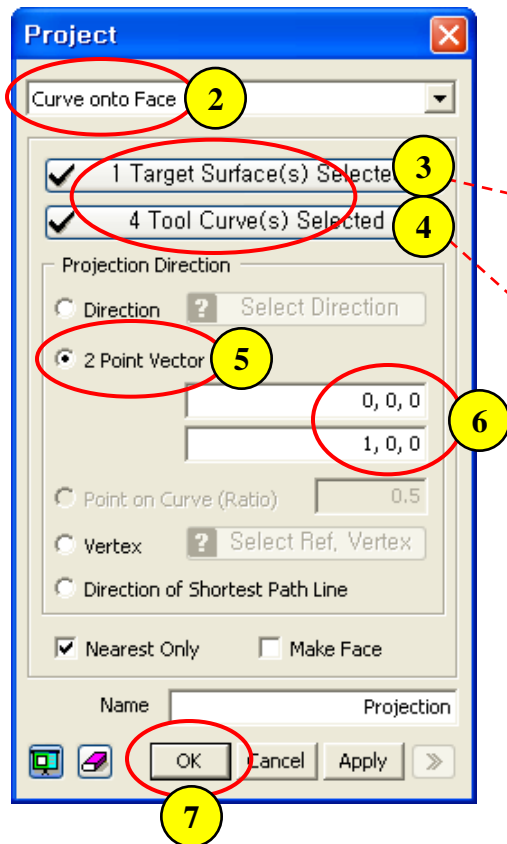


1. *Geometry > Curve > Create on WP > Rectangle ...*
2. *Select 1st Method*
3. *Check on “Make Face”*
4. *Location : (-1, -1), <42, 5>*
5. *Click [Cancel] Button*

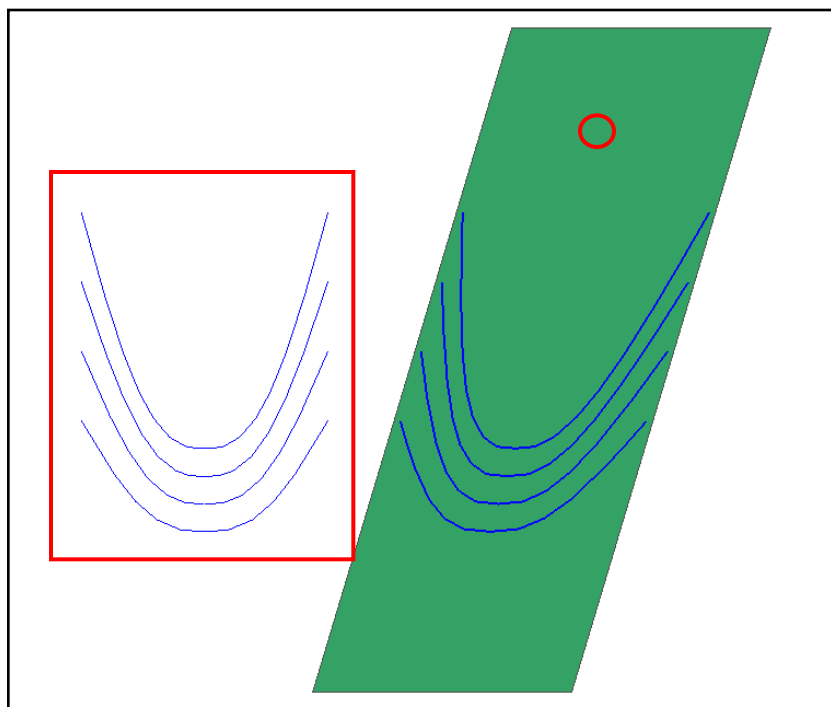
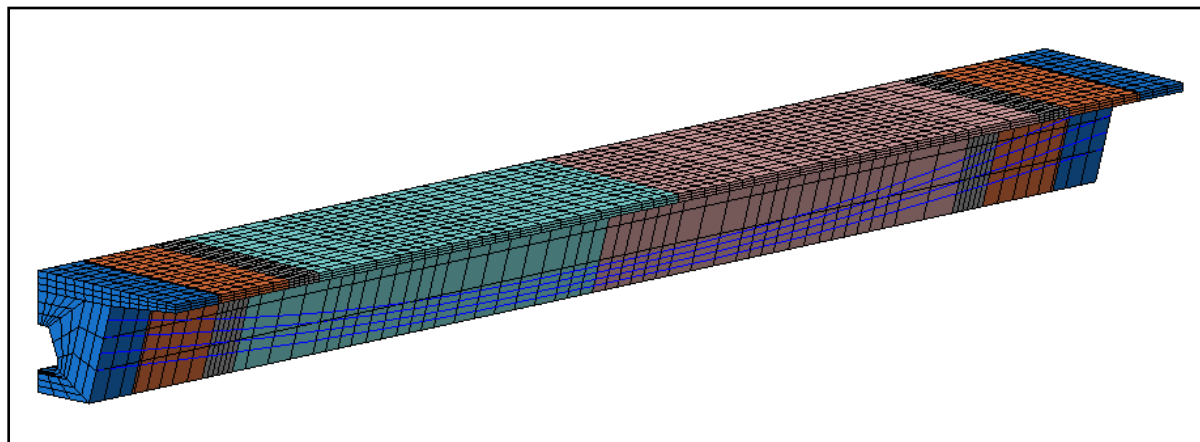
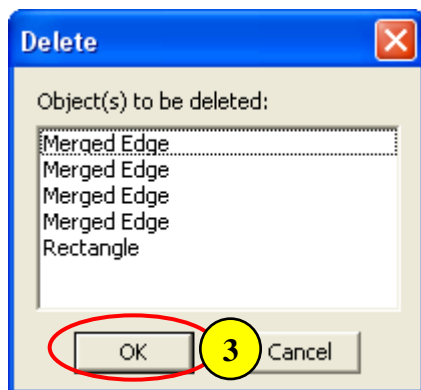


Step 34.

1. Geometry > Transform > Project ...
2. Select "Curve onto Face"
3. Select Target Surface
4. Select 4 Merged B-Spline Curves for Tool Curves
5. Check on "2 Point Vector"
6. Enter : (0, 0, 0), (1, 0, 0)
7. Click [OK] Button

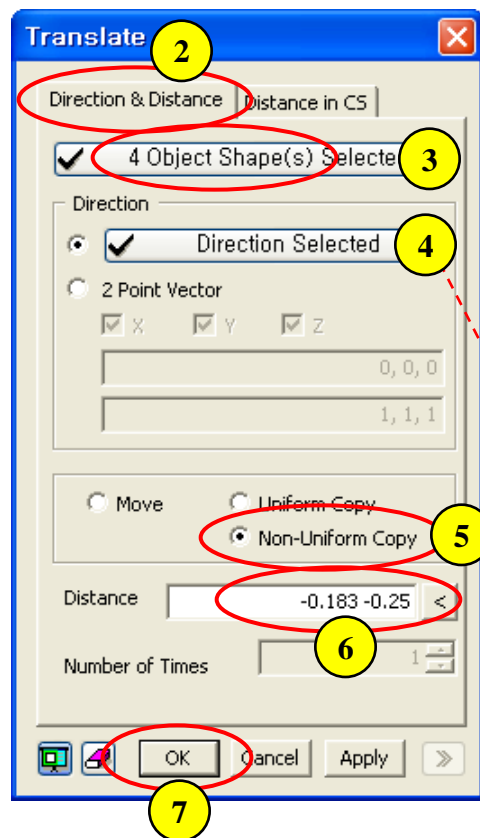



Step 35.

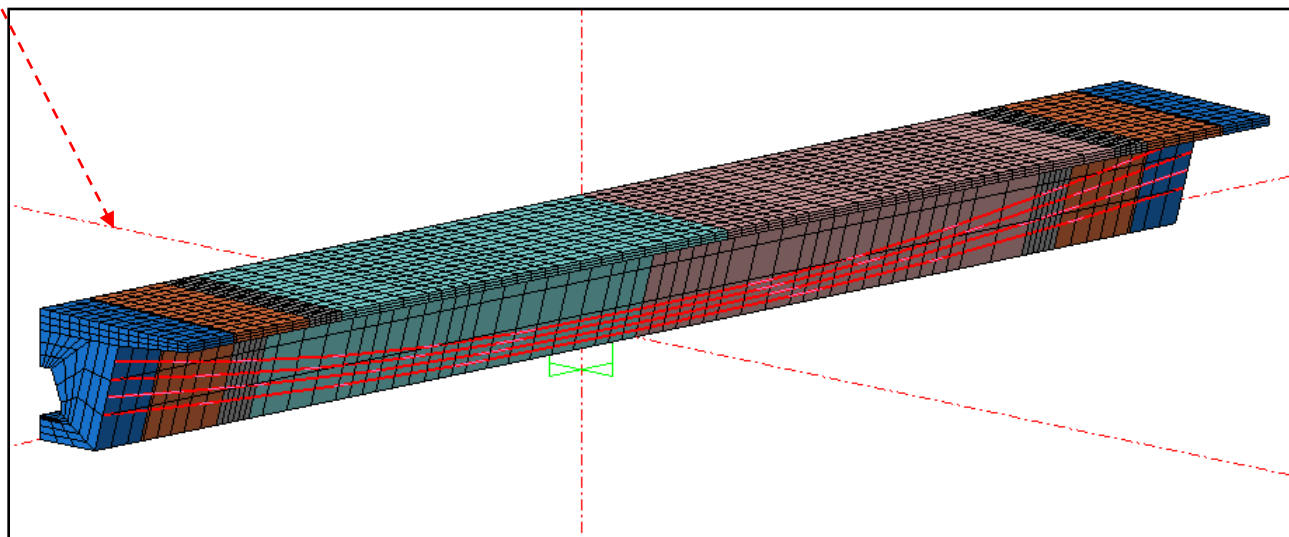


1. *Select 4 Merged B-Spline Curves and Surface*
2. *Press [Delete] Key*
3. *Click [OK] Button*

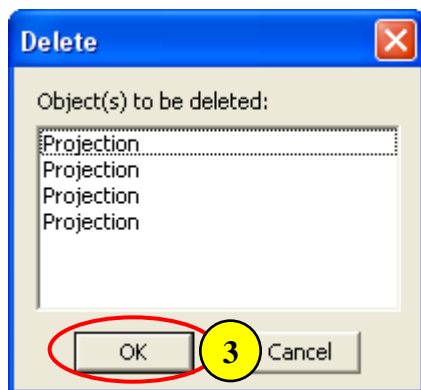
Step 36.



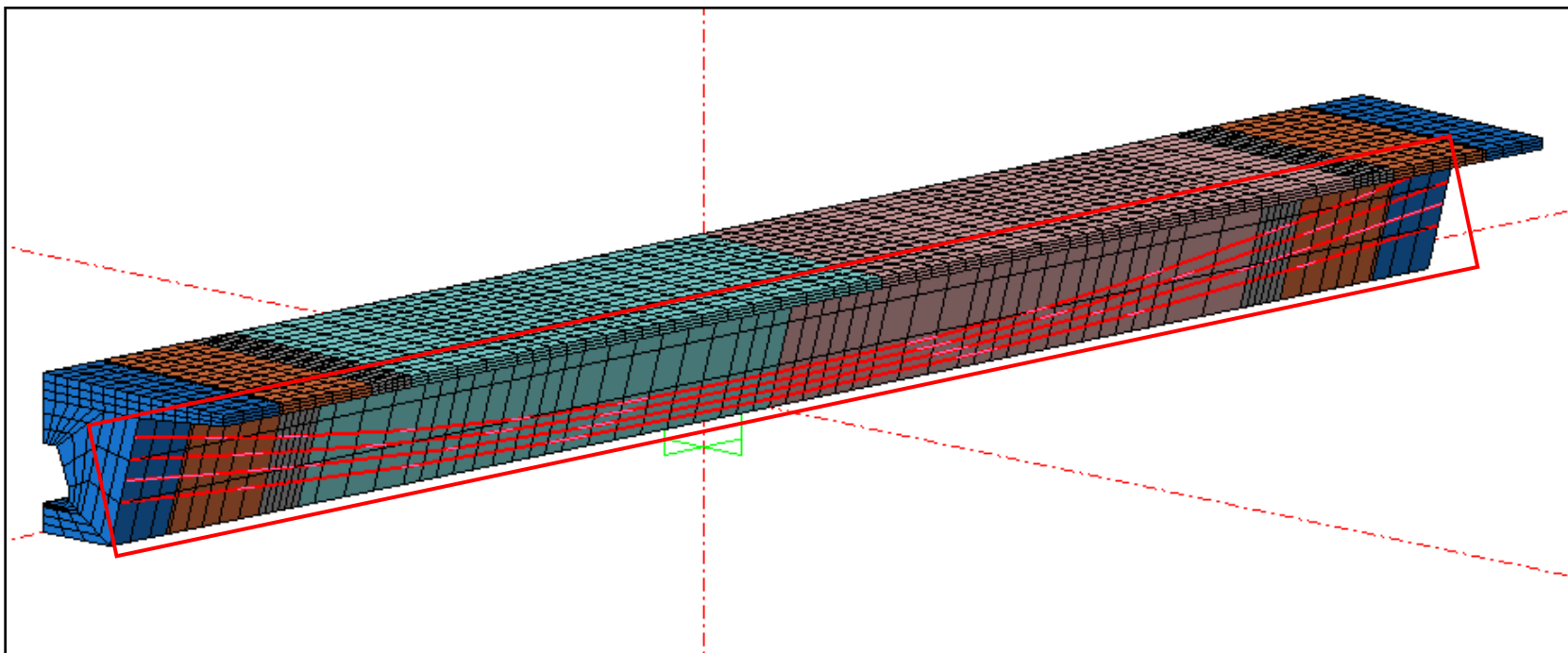
1. Geometry > Transform > Translate ...
2. Select "Direction & Distance" tab
3. Select  "Displayed All" (4 Edges projected)
4. Direction : X-Axis
5. Check on "Non-Uniform Copy"
6. Distance : -0.183, -0.25
7. Click [OK] Button



Step 37.



1. *Select 4 Curves (See Figure)*
2. *Press [Delete] Key*
3. *Click [OK] Button*



Step 38.

Create New Material

Isotropic | Orthotropic | Interface

ID: 1 Name: Concrete Color: [Blue]

Structural

Elastic Modulus: 2.857×10^{10} N/m² Weight Density: 2.452×10^4 N/m³

Poisson's Ratio: 0.167

Expansion Coeff.: [Empty]

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

Material Manager

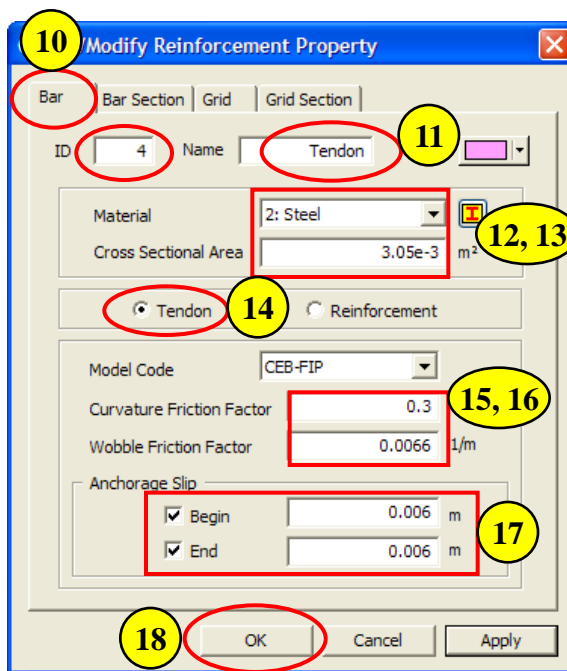
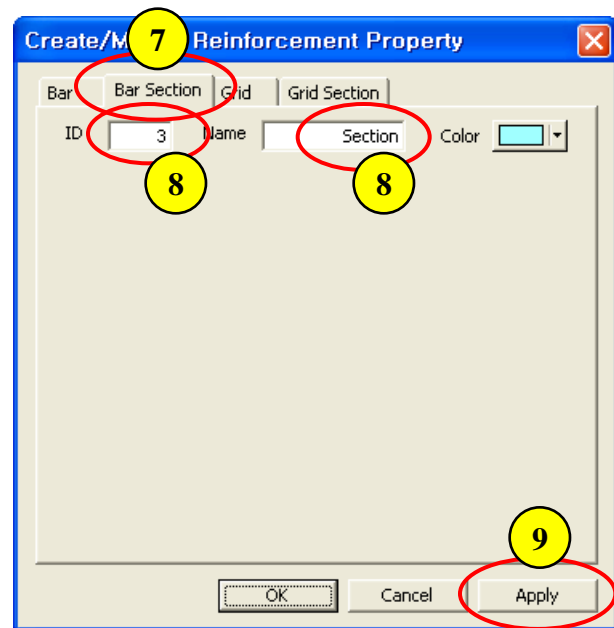
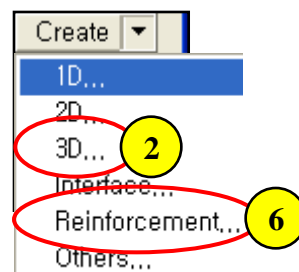
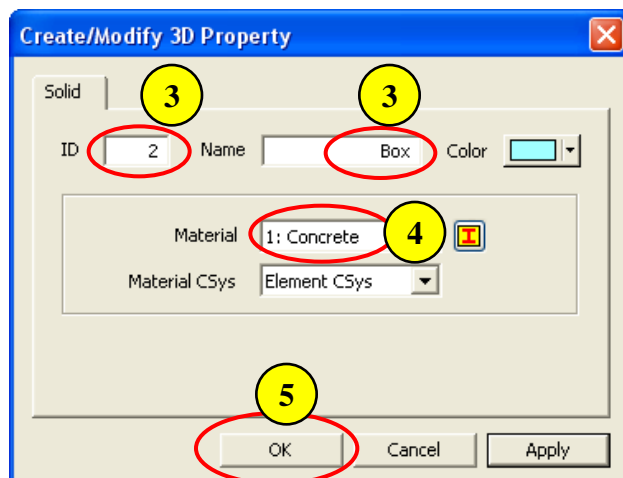
ID	Name	Type
1	Concrete	Isotropic
2	Steel	Isotropic

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

Close

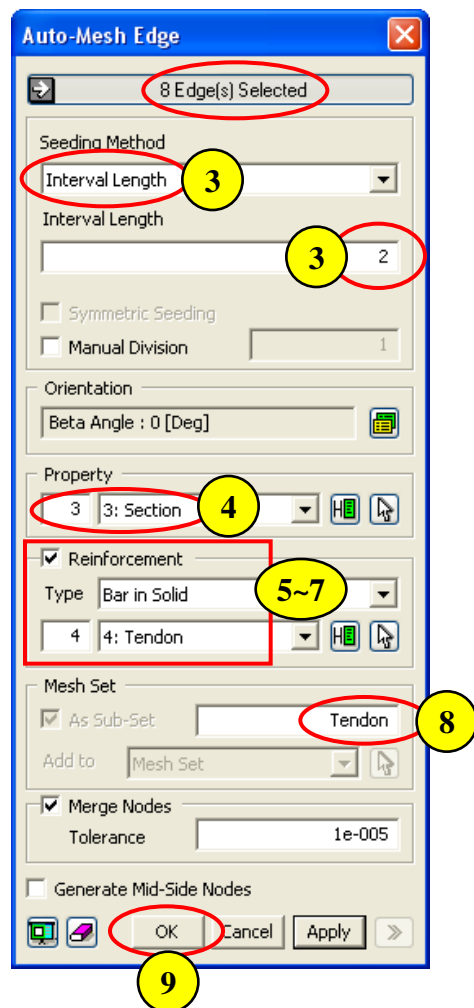
1. Analysis > Material ...
2. Click [Create...] Button
3. Select "Isotropic" tab
4. ID : 1 , Name : Concrete
5. Elastic Modulus : 2.857×10^{10} N/m²
6. Poisson's Ratio : 0.167
7. Weight Density : 2.452×10^4 N/m³
8. Model Type : Elastic
9. Click [Apply] Button
10. ID : 2 , Name : Steel
11. Elastic Modulus : 2×10^{11} N/m²
12. Poisson's Ratio : 0.3
13. Weight Density : 7.698×10^4 N/m³
14. Model Type : Elastic
15. Click [OK] Button
16. Click [Close] Button


Step 39.

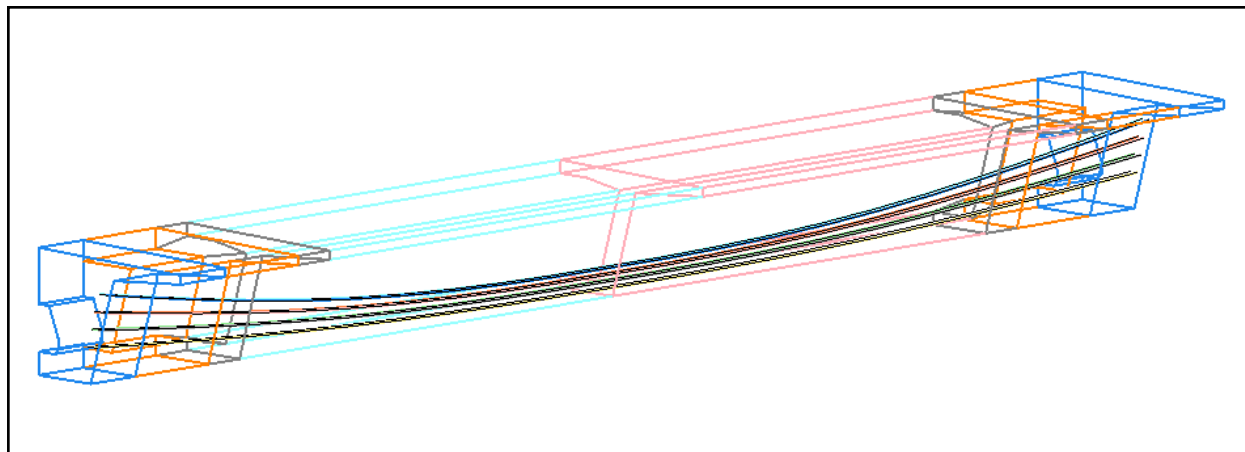


1. Analysis > Property ...
2. Create 3D ...
3. ID : 2 , Name : Box
4. Select "1:Concrete" for Material
5. Click [OK] Button
6. Create Reinforcement ...
7. Select "Bar Section" tab
8. ID : 3 , Name : Section
9. Click [Apply] Button
10. Select "Bar" tab
11. ID : 4 , Name : Tendon
12. Select "2: Steel" for Material
13. Cross Sectional Area : $3.05e-3 \text{ m}^2$
14. Check on "Tendon"
15. Curvature Friction Factor : 0.3
16. Wobble Friction Factor : 0.0066
17. Anchorage Slip
 - Begin : 0.006 m
 - End : 0.006 m
18. Click [OK] Button
19. Click [Close] Button

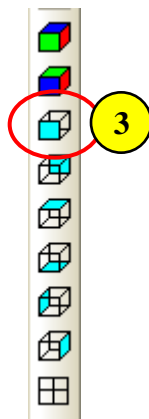
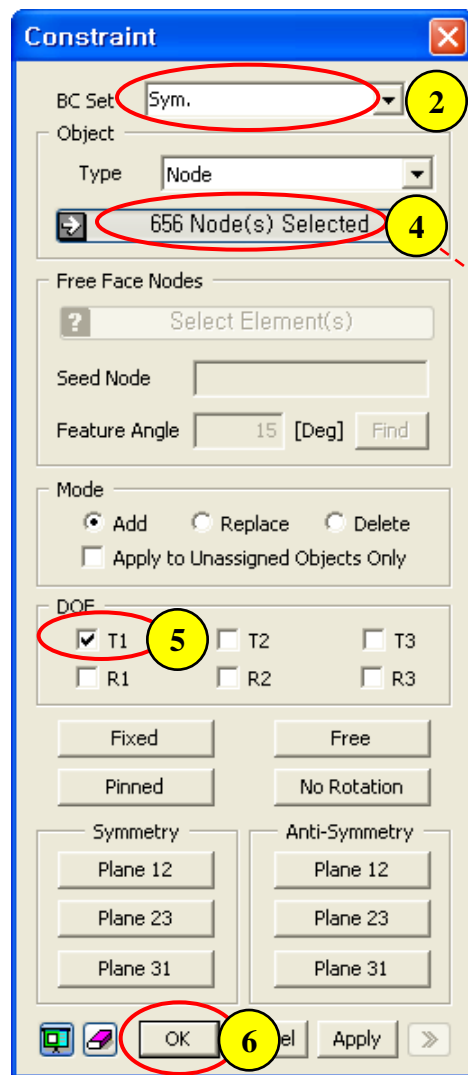
Step 40.



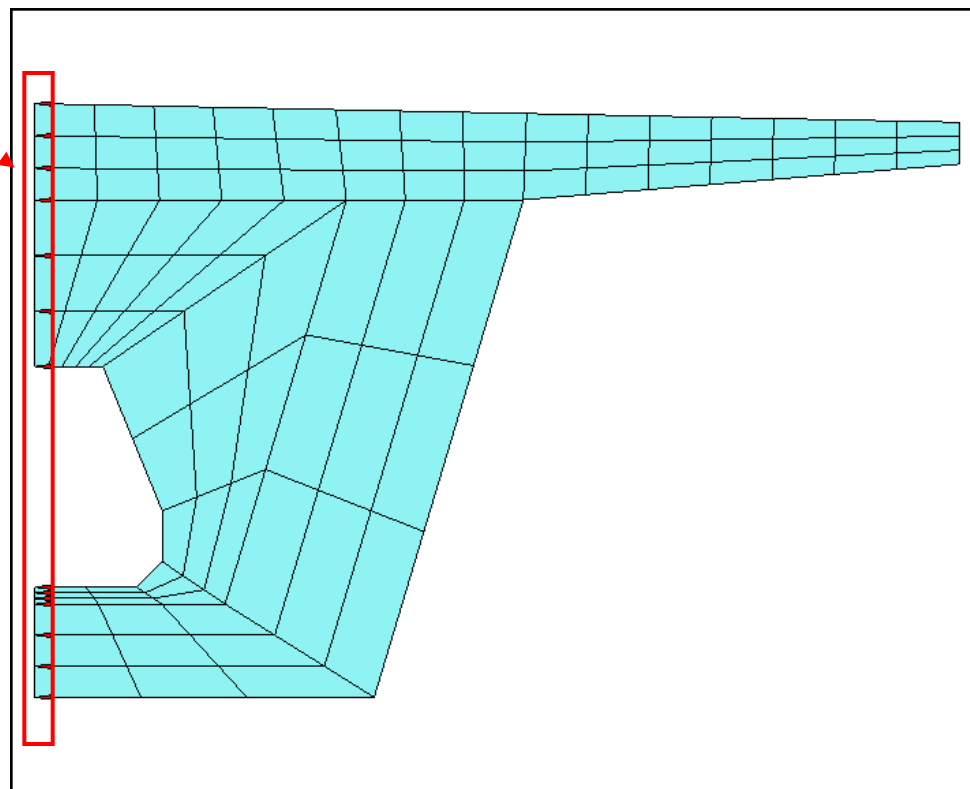
1. Mesh > Auto Mesh > Edge ...
2. Select  "Displayed All"
3. Seeding Method : Interval Length (2)
4. Property : "3: Section"
5. Check on "Reinforcement"
6. Type : Bar in Solid
7. Property : "4: Tendon"
8. Mesh Set : Tendon
9. Click [OK] Button



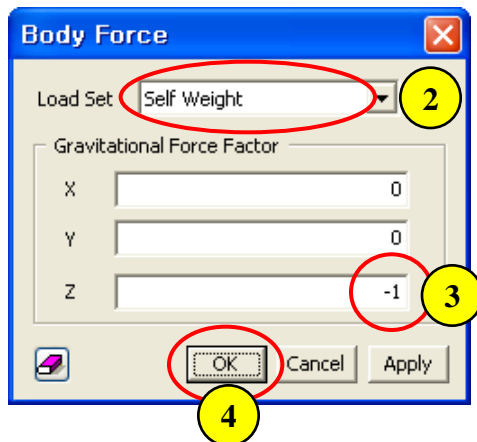
Step 41.



1. Analysis > BC > Constraint ...
2. BC Set : Sym .
3. Click "Front View"
4. Select 656 Nodes (See Figure)
5. Check on "T1"
6. Click [OK] Button



Step 42.



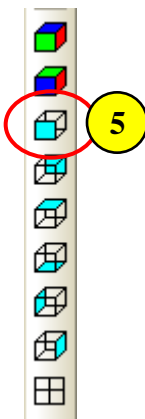
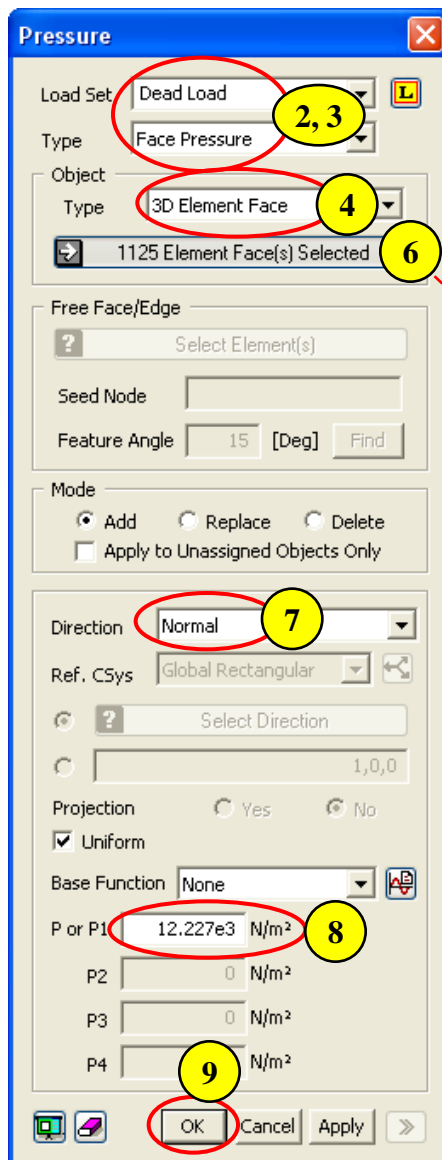
1. Analysis > Load > Body Force ...

2. Load Set : Self Weight

3. Z: -1

4. Click [OK] Button

Step 43.



1. Analysis > Load > Pressure ...

2. Load Set : Dead Load

3. Type : Face Pressure

4. Object Type : 3D Element Face

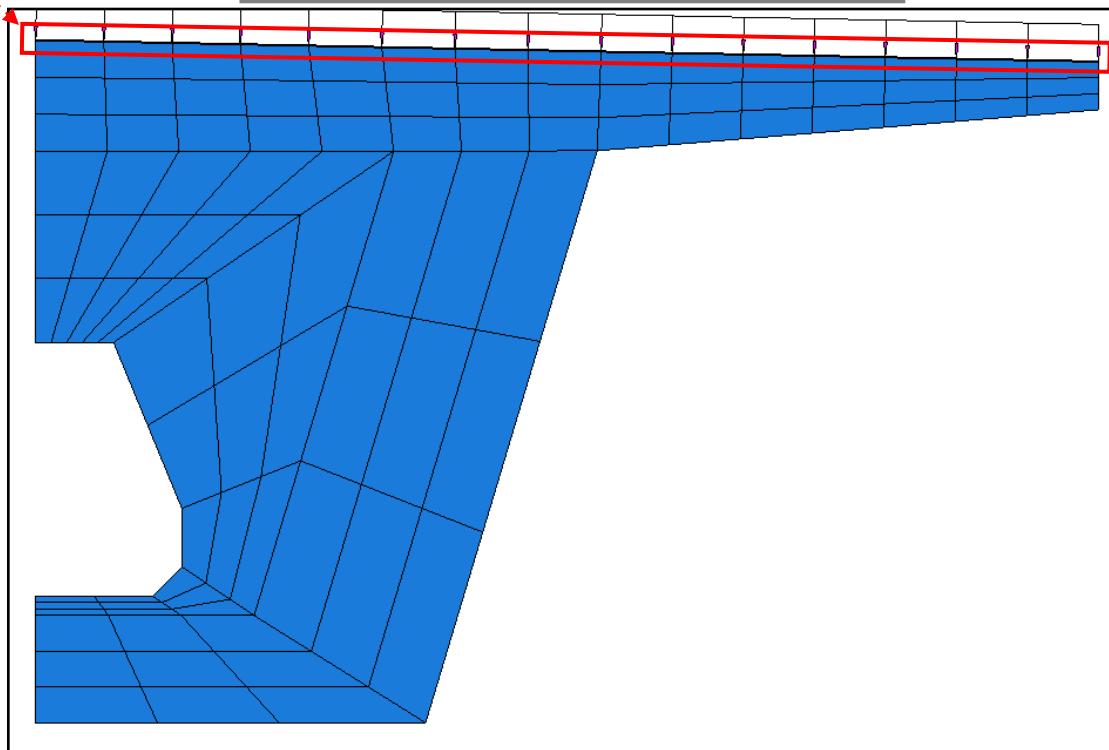
5. Click "Front View"

6. Select 1125 Element Faces (See Figure)

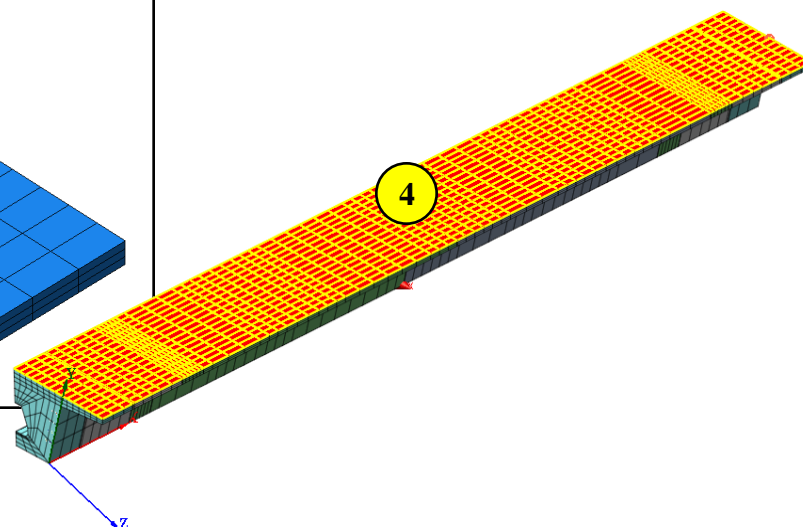
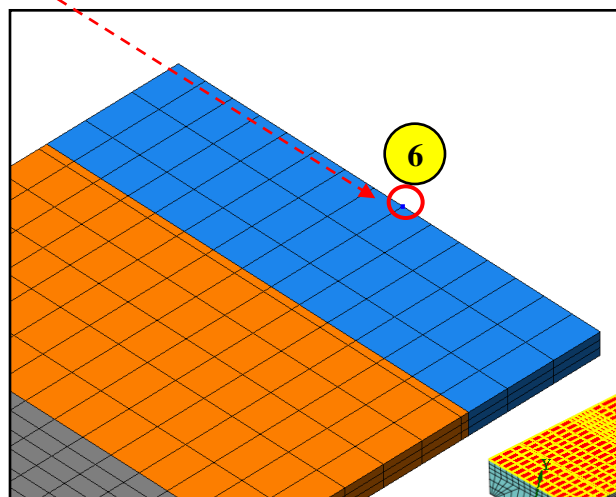
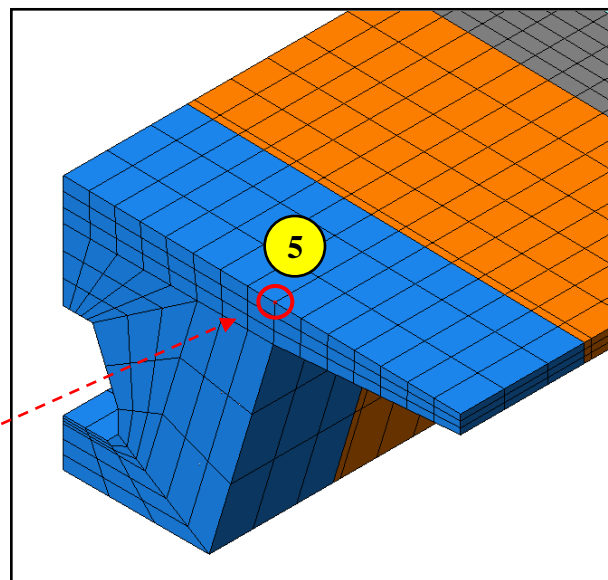
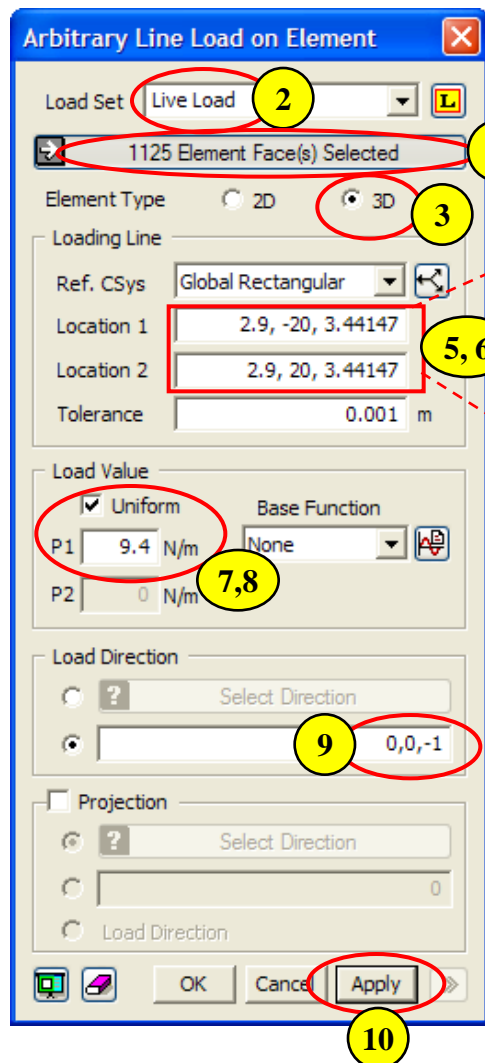
7. Direction : Normal

8. P or P1 : 12.227e3 N/m²

9. Click [OK] Button

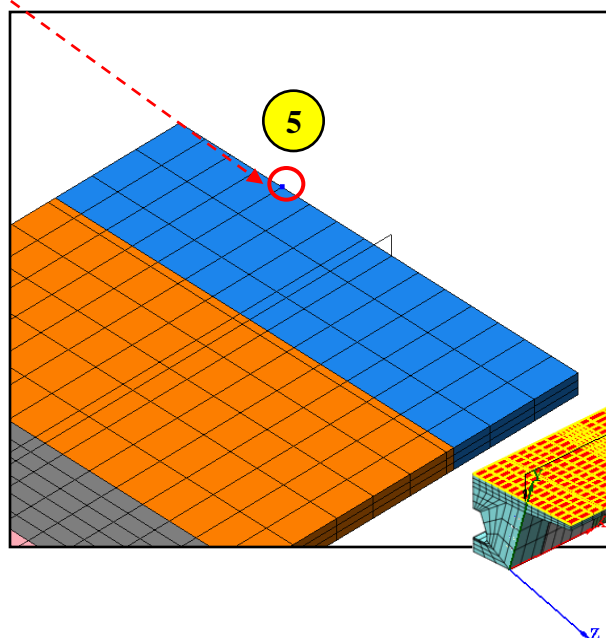
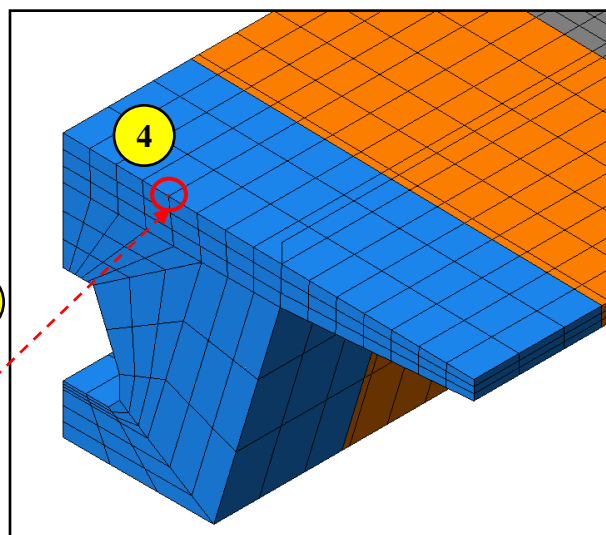
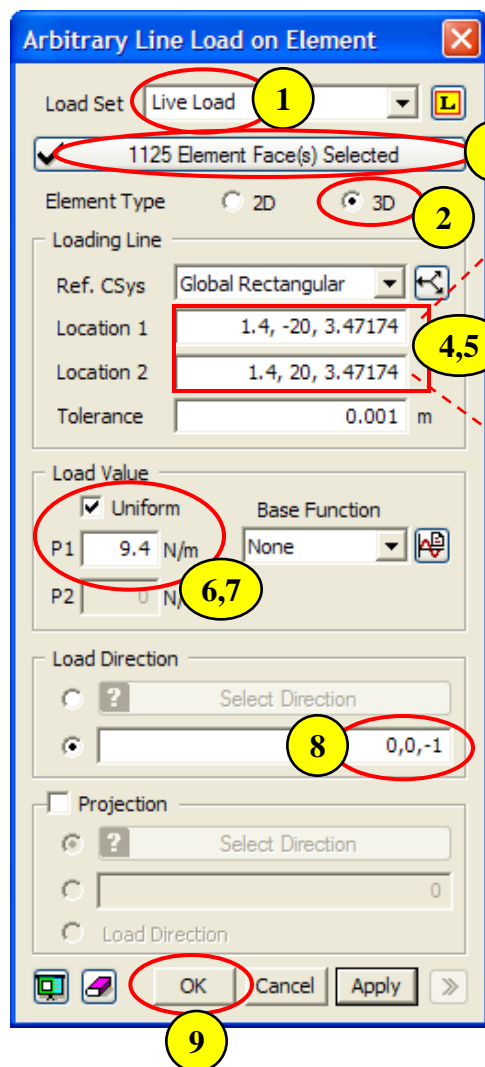


Step 44.



1. Analysis > Load > Arbitrary Load > Line Load ...
2. Load Set : Live Load
3. Element Type : 3D
4. Select the top deck surface elements
5. Select First Node (See Figure)
6. Select Second Node (See Figure)
7. Check on "Uniform"
8. P1 : 9.4 N/m
9. Load Direction : (0, 0, -1)
10. Click [Apply] Button

Step 45.

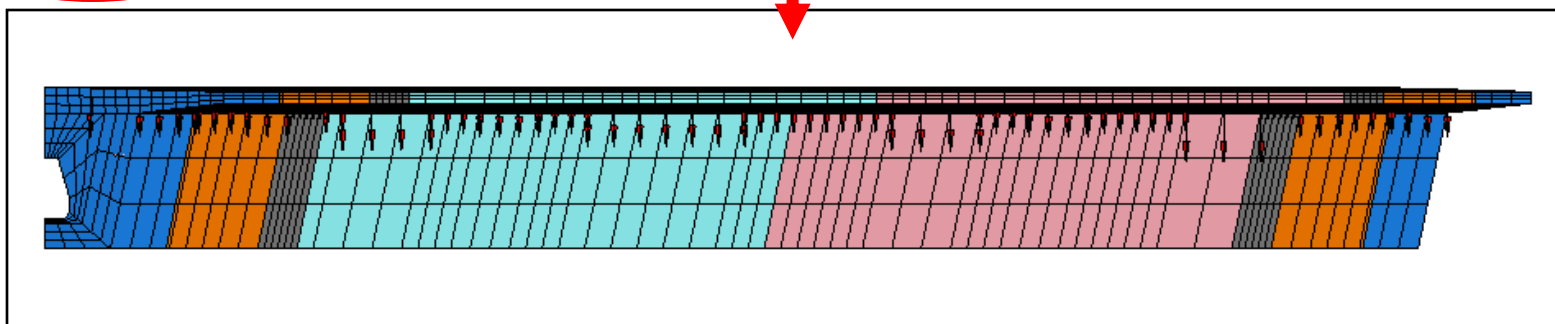
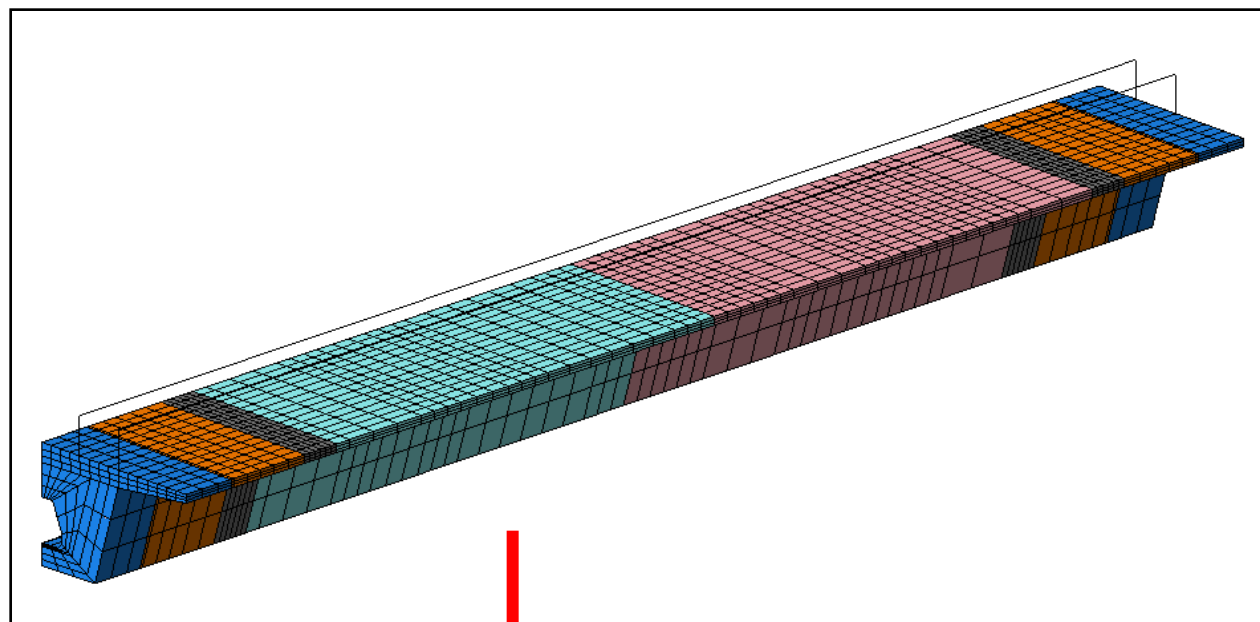
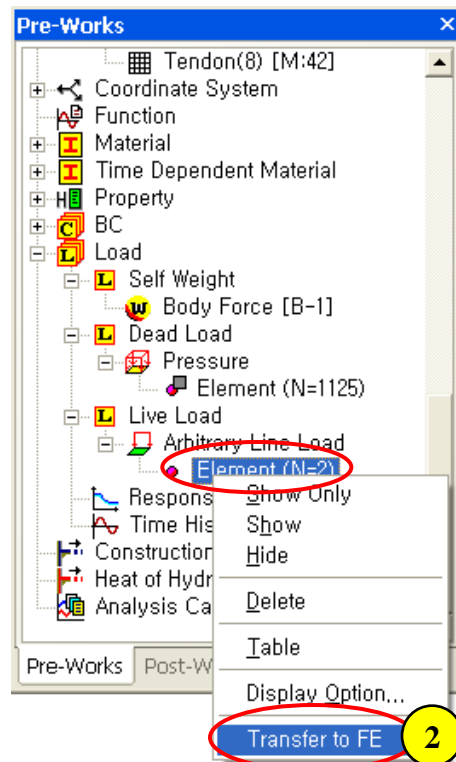


1. Load Set : Live Load
2. Element Type : 3D
3. Select the top deck surface elements
4. Select First Node (See Figure)
5. Select Second Node (See Figure)
6. Check on "Uniform"
7. P1 : 9.4 N/m
8. Load Direction : (0, 0, -1)
9. Click [OK] Button

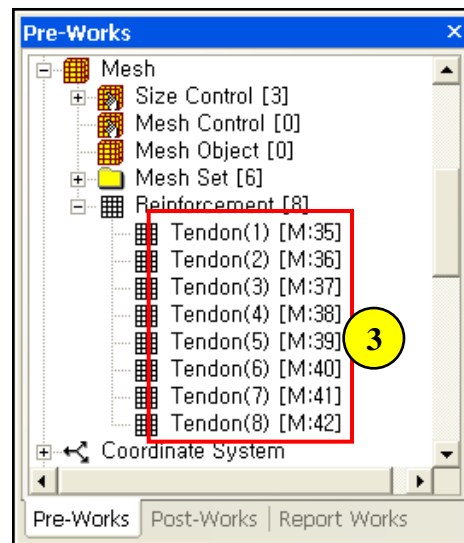
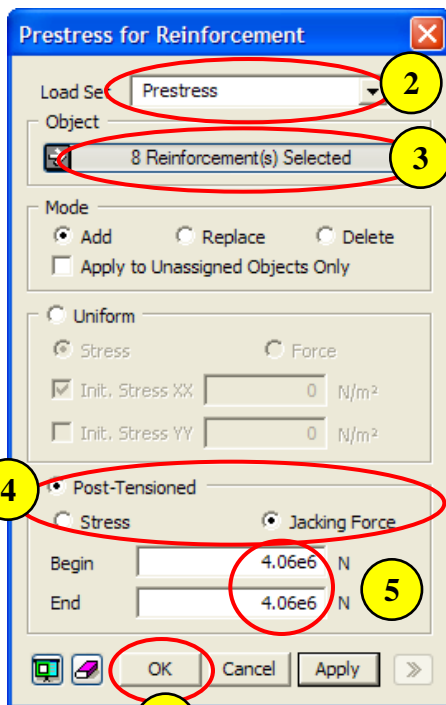
Step 46.

1. Pre-Works Tree : Load > Live Load > Arbitrary Line Load > Element (N=2) ...

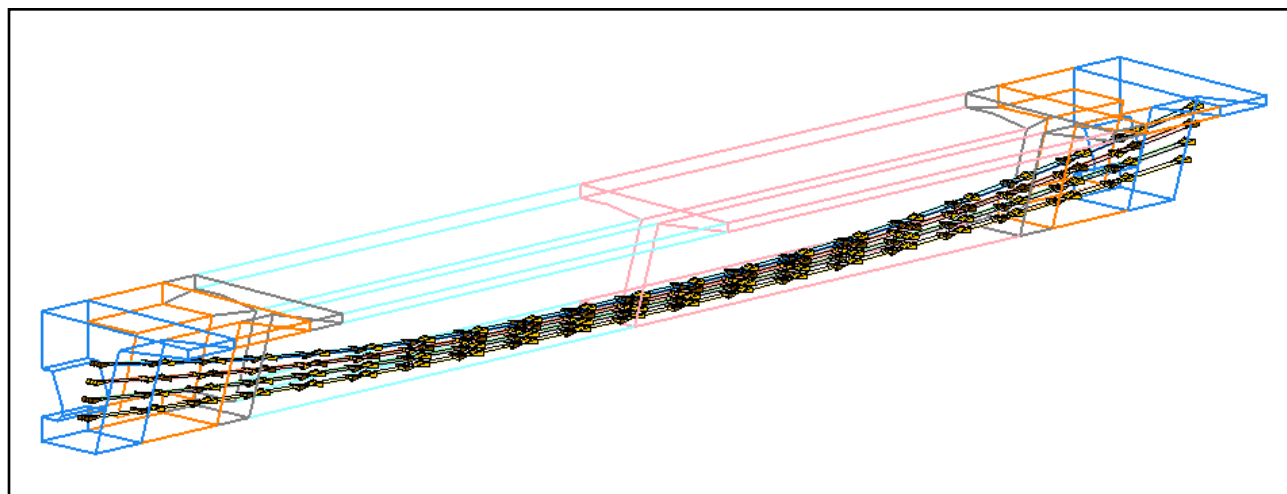
2. Click Right Mouse Button and Select "Transfer to FE"



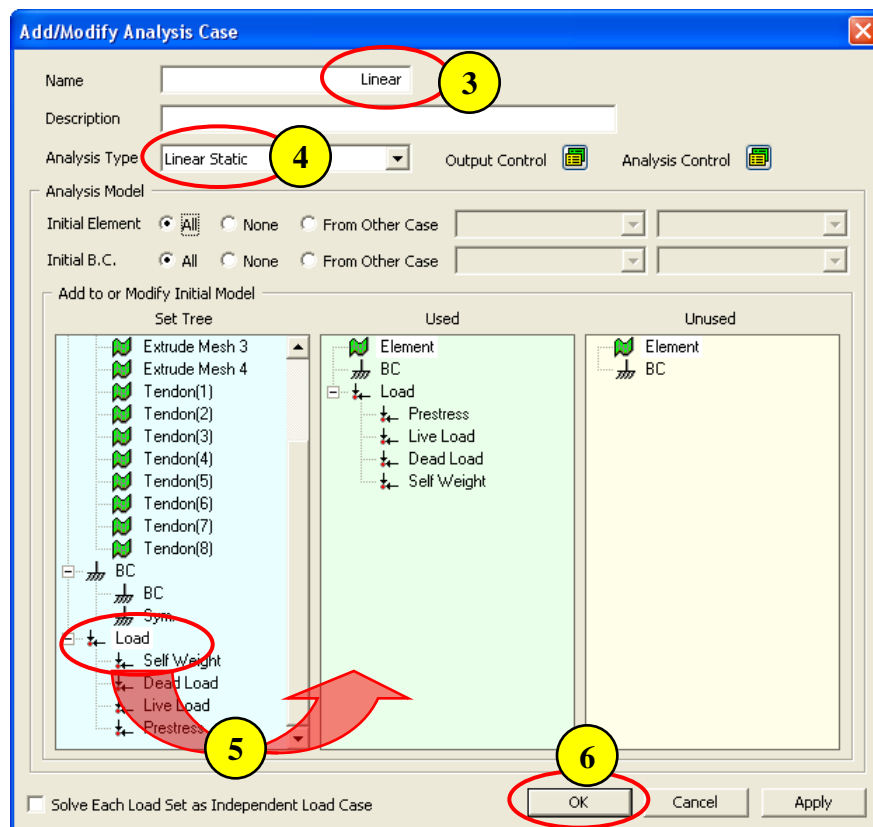
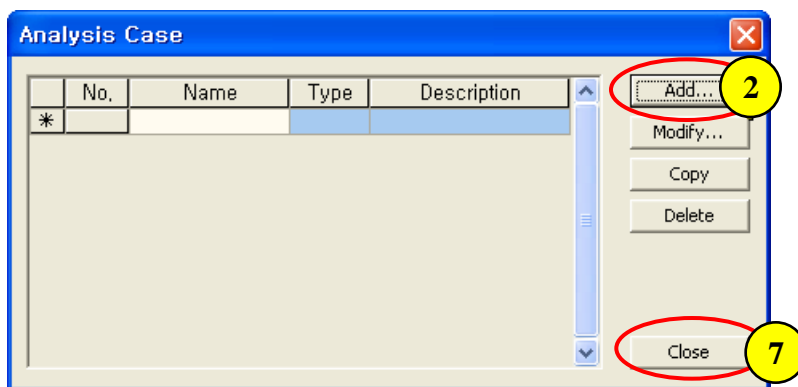
Step 47.



1. Analysis > Load > Prestress for Reinforcement ...
2. Load Set : Prestress
3. Select All Reinforcement in Pre-Works Tree
4. Check on "Post tensioned (Jacking Force)"
5. Begin & End Force : 4.06e6 N
6. Click [OK] Button



Step 48.



1. Analysis > Analysis Case ...

2. Click [Add] Button

3. Name : Linear

4. Analysis Type : Linear Static

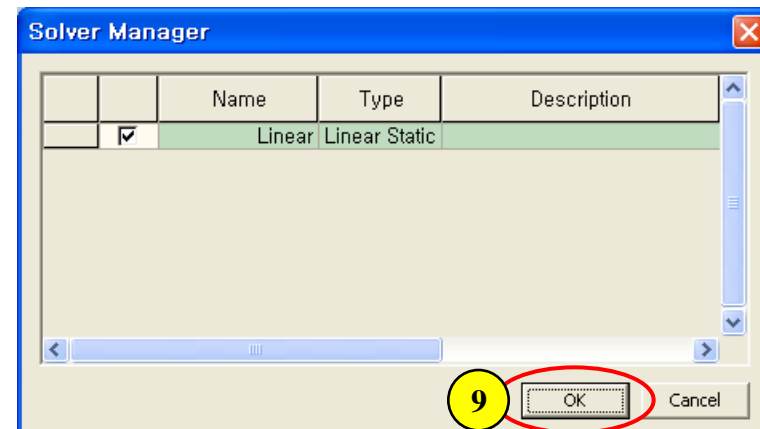
5. Drag & Drop "Load" to "Used" Window

6. Click [OK] Button

7. Click [Close] Button

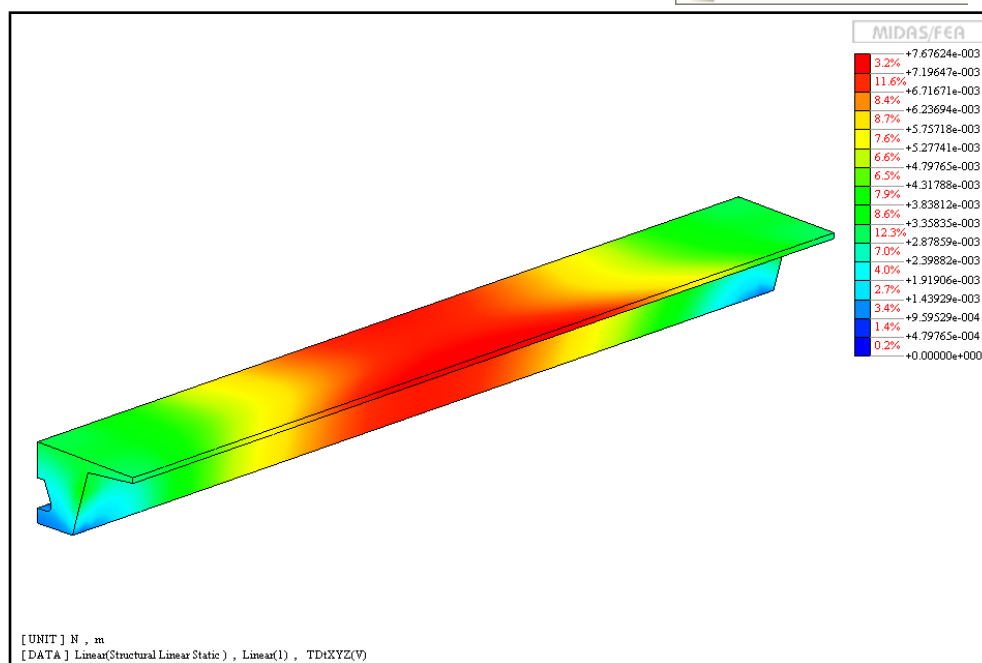
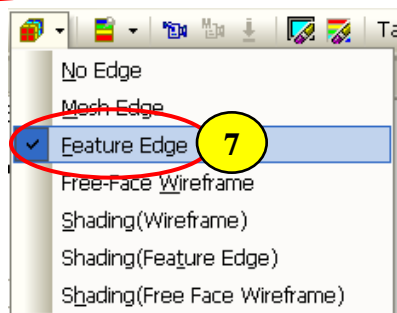
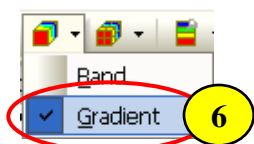
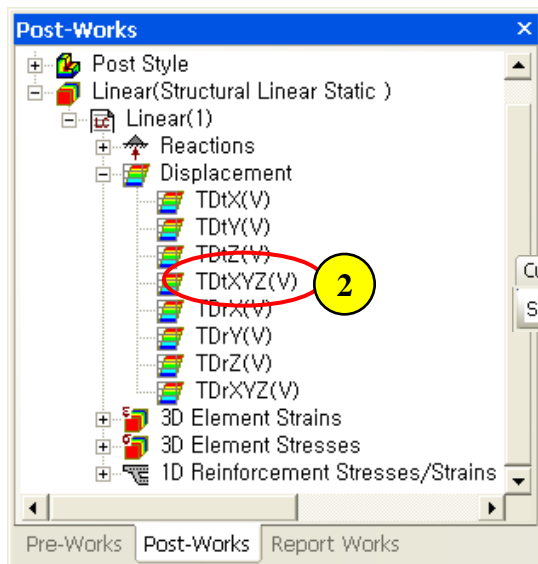
8. Analysis > Solve ...

9. Click [OK] Button

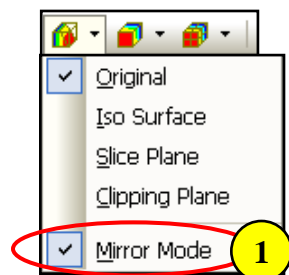


Step 49.

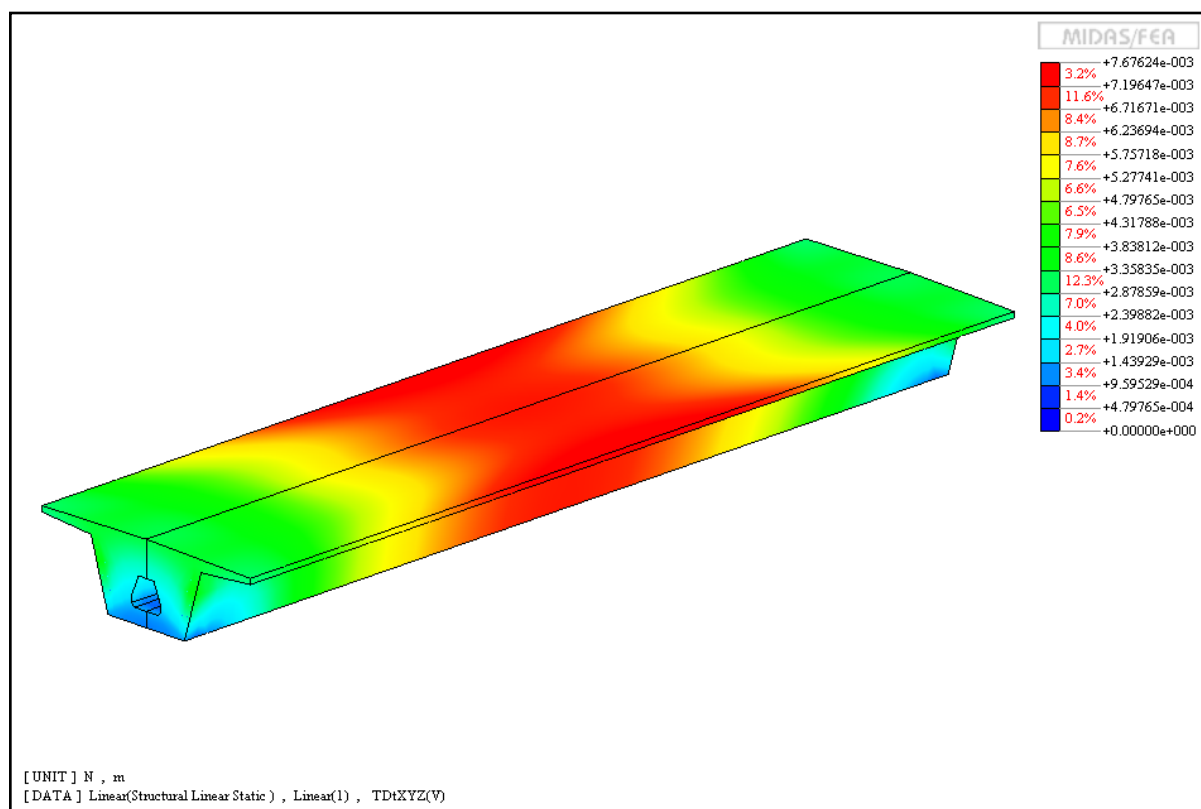
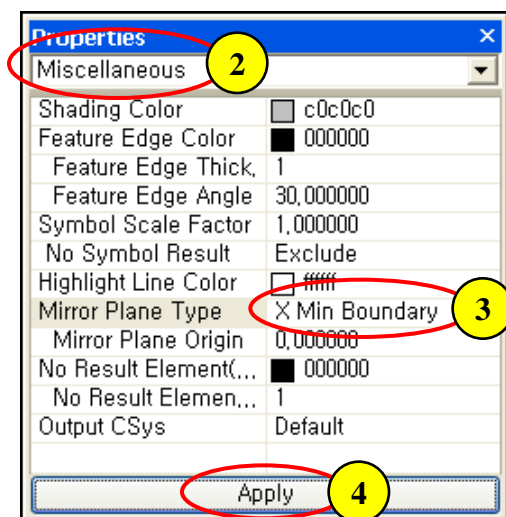
1. Post-Works Tree : Linear (Structural Linear Static) > Linear(1) > Displacement
2. Double Click "TDtXYZ(V)"
3. Click "Sens." Button
4. Select "Undeformed" for Mesh Shape (See Figure)
5. Click "Post Style" Toolbar
6. Select "Gradient" for Contour Type
7. Select "Feature Edge" for Edge Type



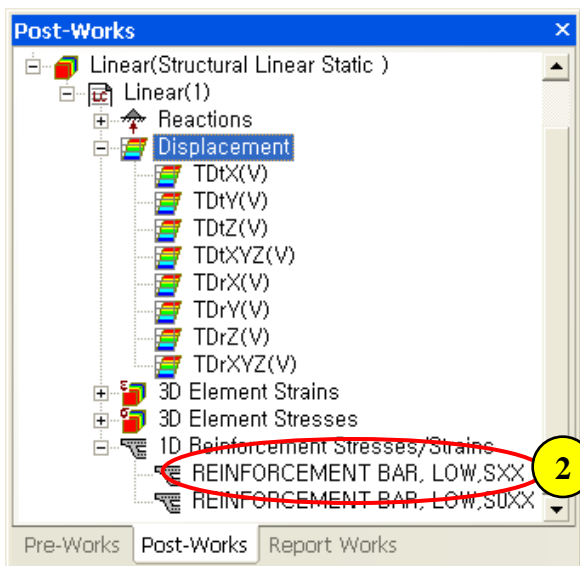
Step 50.



1. Select “Mirror Mode” for Visualization
2. Property Window : Miscellaneous
3. Select “X Min Boundary” for Mirror Plane Type
4. Click [Apply] Button



Step 51.



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

