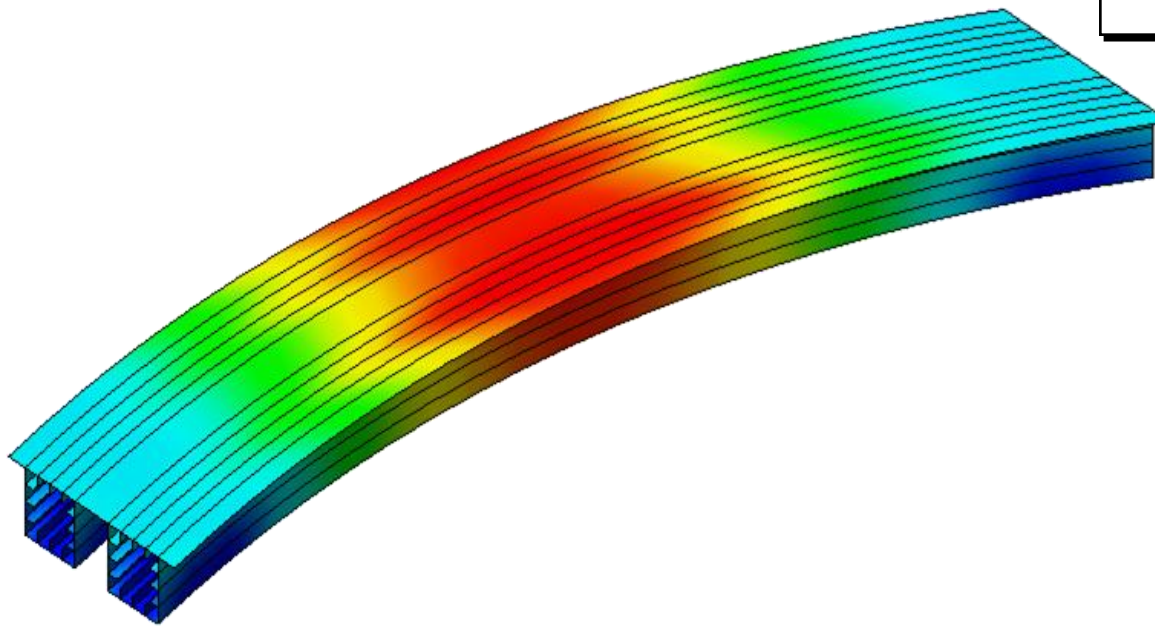


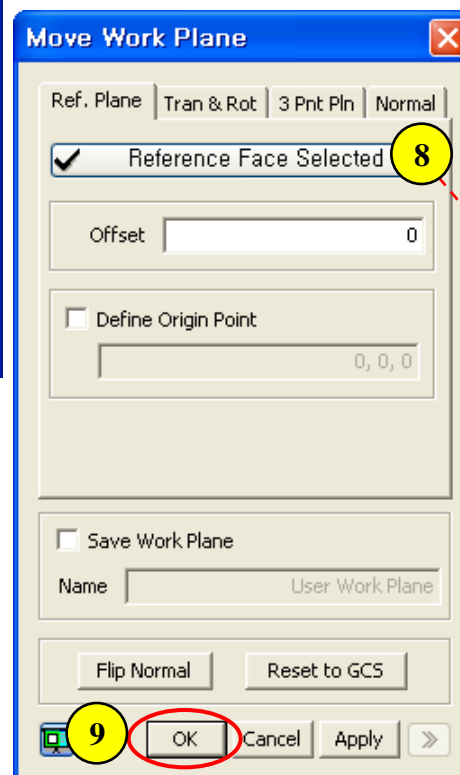
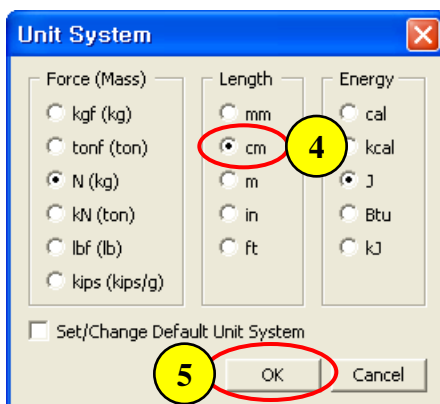
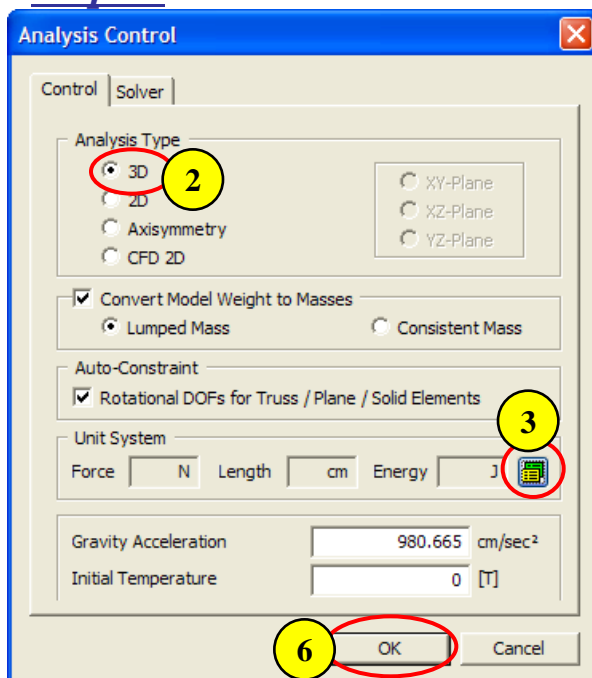
LS-11. Analysis of a Steel Box Bridge


Overview

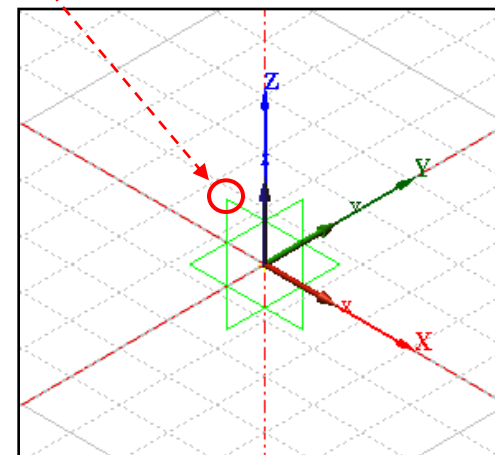
- 3-D Linear Static Analysis
- Model
 - Unit : N, cm
 - Isotropic Elastic Material
 - Plate Element
- Load & Boundary Condition
 - Body Force
 - Pressure
 - Constraint



Step 1.

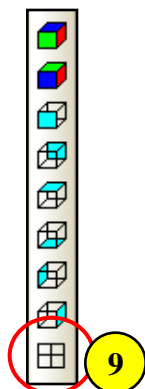
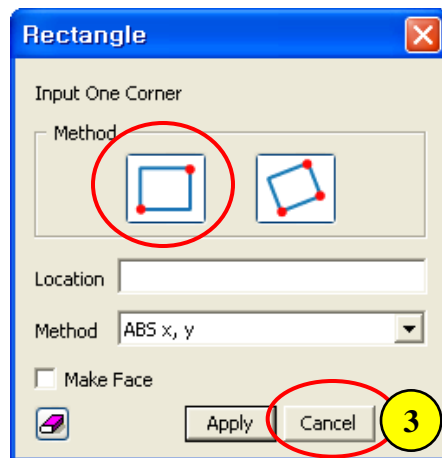


1. Analysis > Analysis Control - "Control" tab
2. Analysis Type : 3D
3. Click  Button (Unit System)
4. Length : cm
5. Click [OK] Button
6. Click [OK] Button
7. Geometry > Work Plane > Move ...
8. Select "XZ Plane"
9. Click [OK] Button

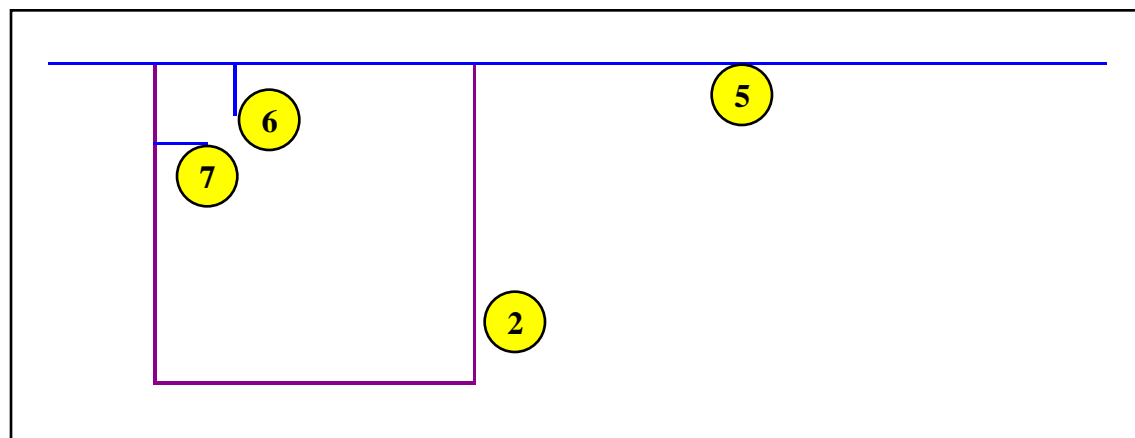
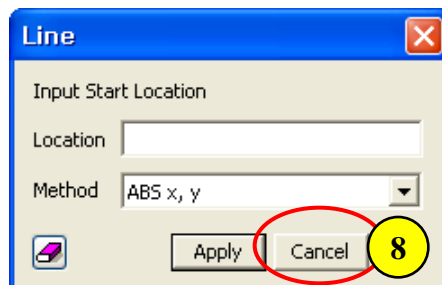


 Analysis Control Dialog is automatically activated at startup.

Step 2.



1. *Geometry > Curve > Create on WP > Rectangle (Wire) ...*
2. *Location : (20), <60, -60> Ⓜ*
3. *Click [Cancel] Button Ⓜ*
4. *Geometry > Curve > Create on WP > Line ...*
5. *Location : (0), <200>*
6. *Location : (35), <0, -10>*
7. *Location : (20, -15), <10>*
8. *Click [Cancel] Button*
9. *Click “Normal View”*

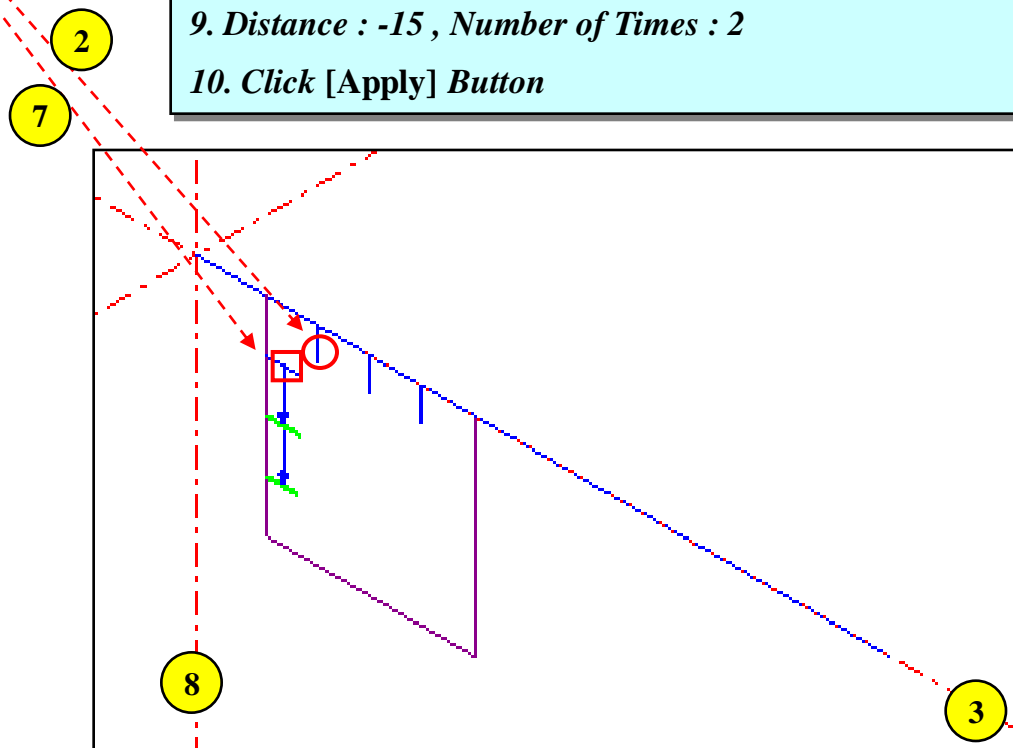
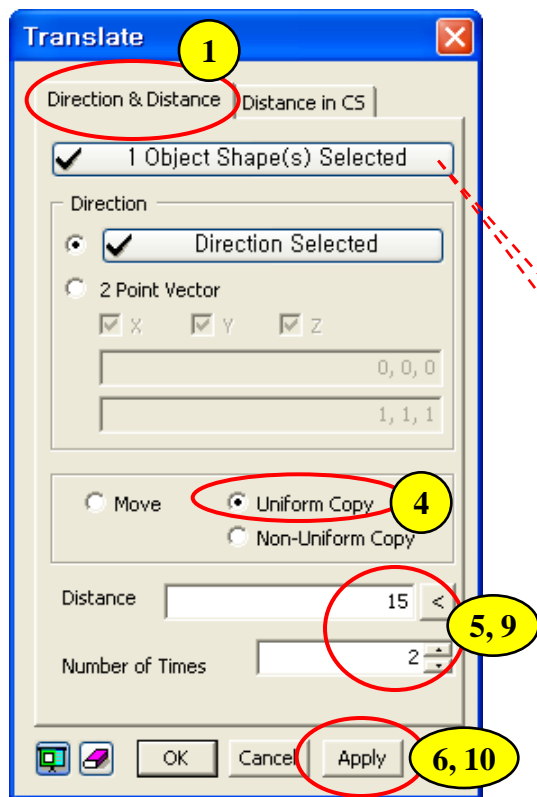


Ⓜ () : “ABS x, y”, <> : “REL dx, dy”
(20) same as (20, 0)

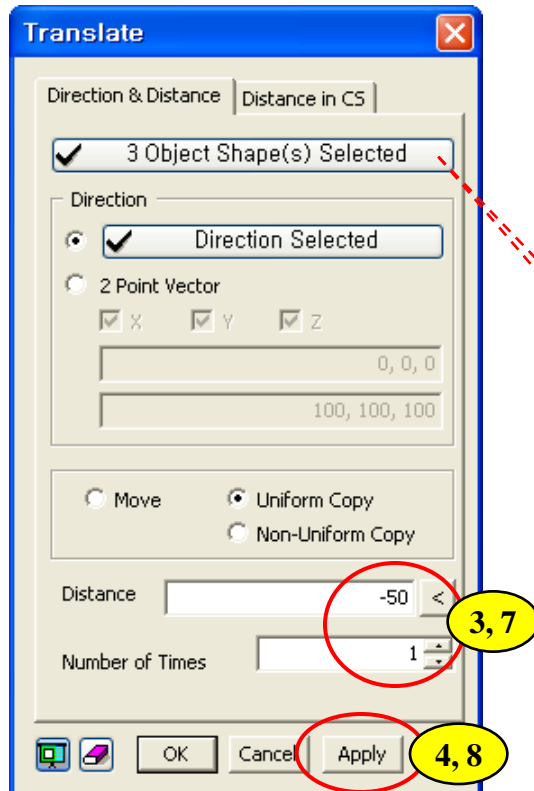
Ⓜ “Esc” as shortcut for “Cancel”.

Step 3.

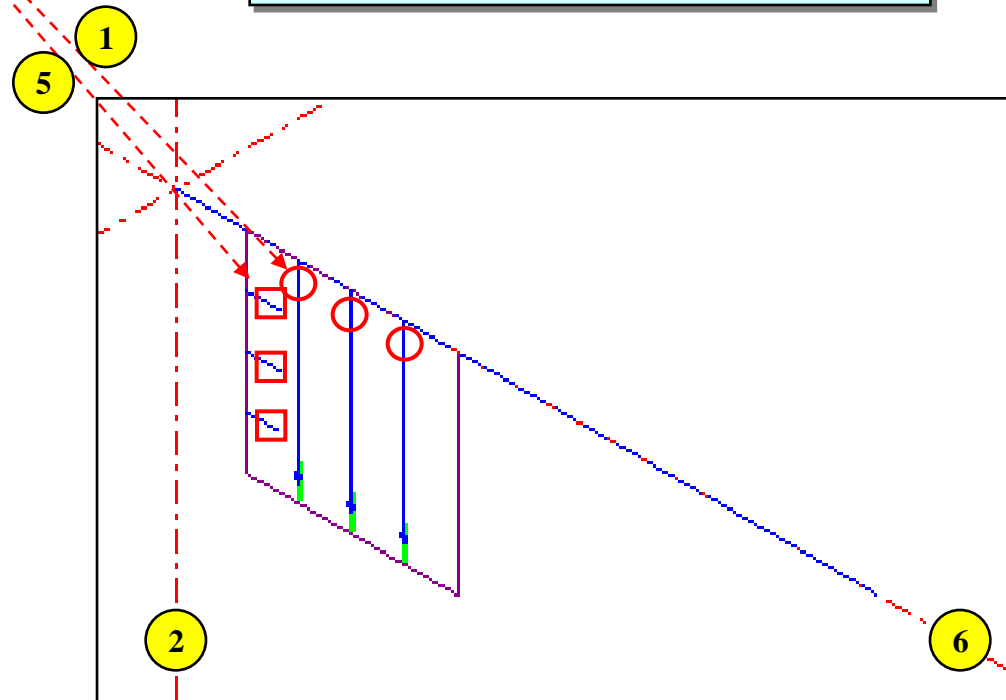
1. Geometry > Transform > Translate - “Direction & Distance” tab
2. Select Edge marked by “O” (See Figure)
3. Direction : X-Axis
4. Check on “Uniform Copy”
5. Distance : 15 , Number of Times : 2
6. Click [Apply] Button
7. Select Edge marked by “□” (See Figure)
8. Direction : Z-Axis
9. Distance : -15 , Number of Times : 2
10. Click [Apply] Button



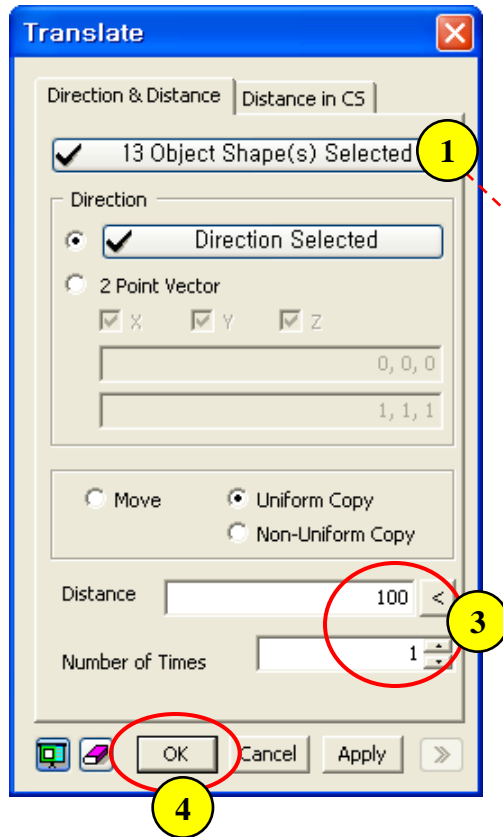
Step 4.



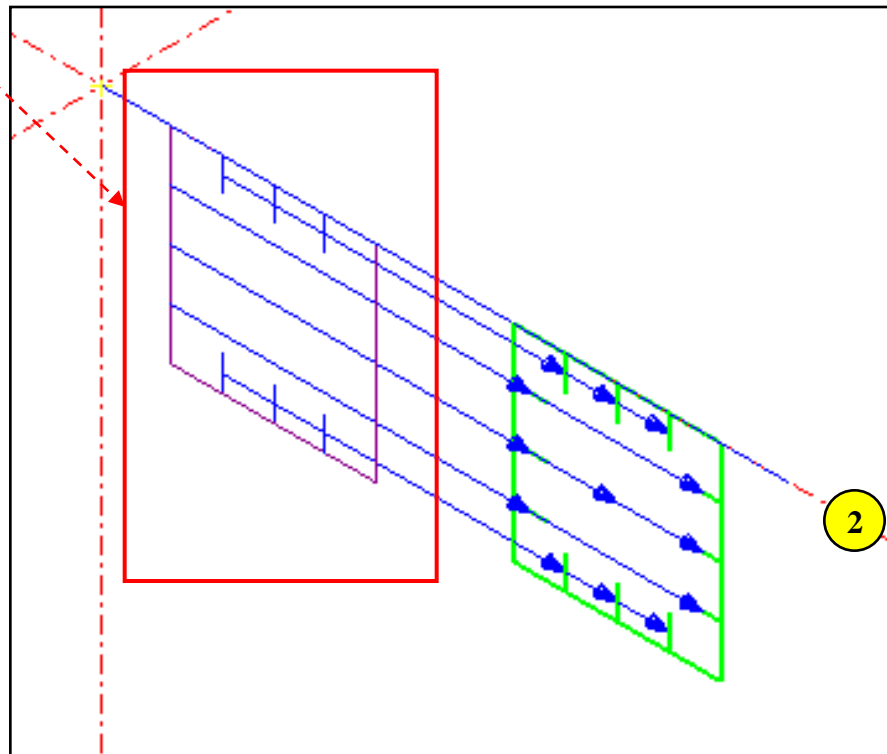
1. Select Edges marked by “○” (See Figure)
2. Direction : Z-Axis
3. Distance : -50 , Number of Times : 1
4. Click [Apply] Button
5. Select Edges marked by “□” (See Figure)
6. Direction : X-Axis
7. Distance : 50 , Number of Times : 1
8. Click [Apply] Button



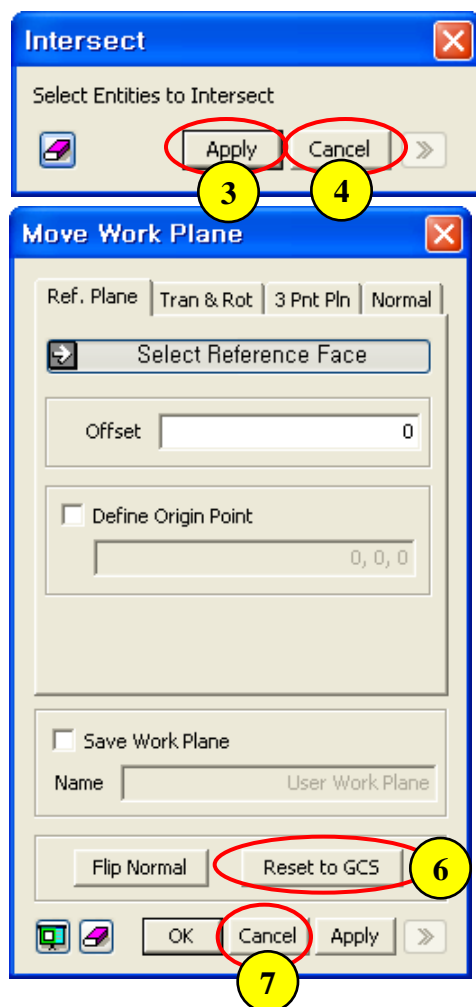
Step 5.






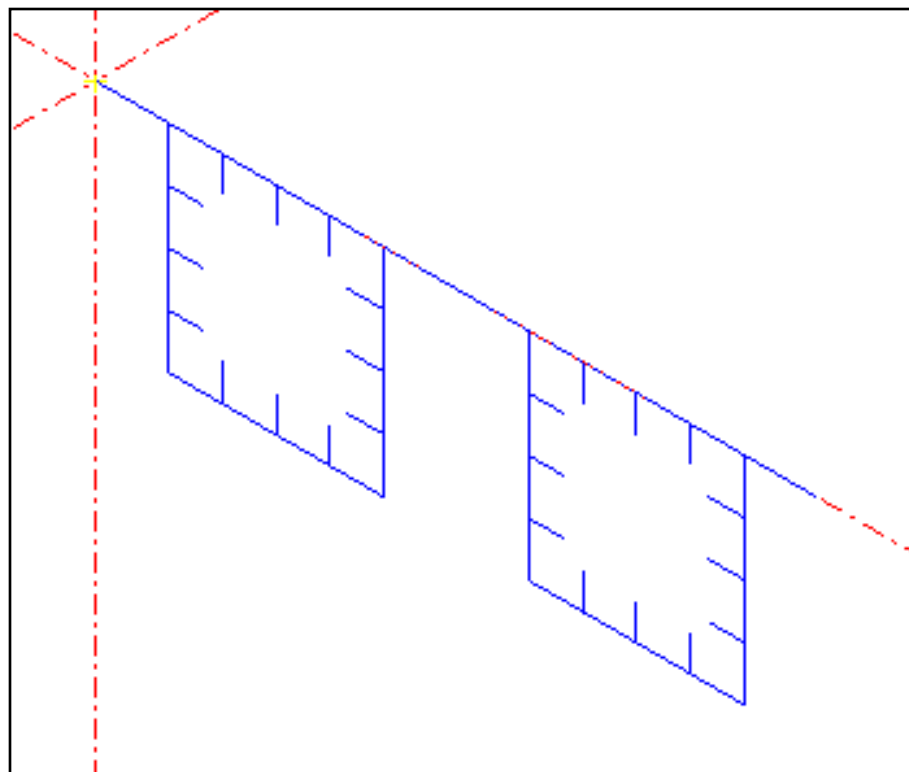
1. Select 13 Edges (See Figure)
2. Direction : X-Axis
3. Distance : 100 , Number of Times : 1
4. Click [OK] Button





Step 6.

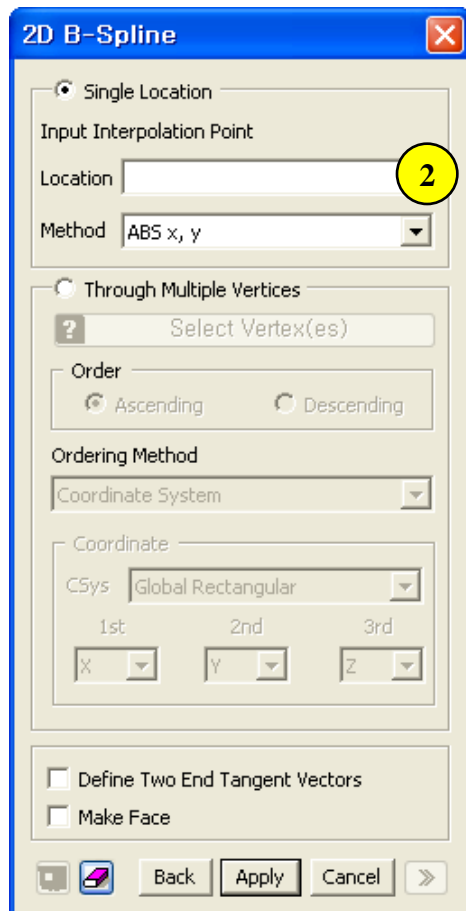


1. *Geometry > Curve > Intersect ...*
2. Select  "Displayed" 
3. Click [Apply] Button 
4. Click [Cancel] Button
5. *Geometry > Work Plane > Move ...*
6. Click [Reset to GCS] Button
7. Click [Cancel] Button

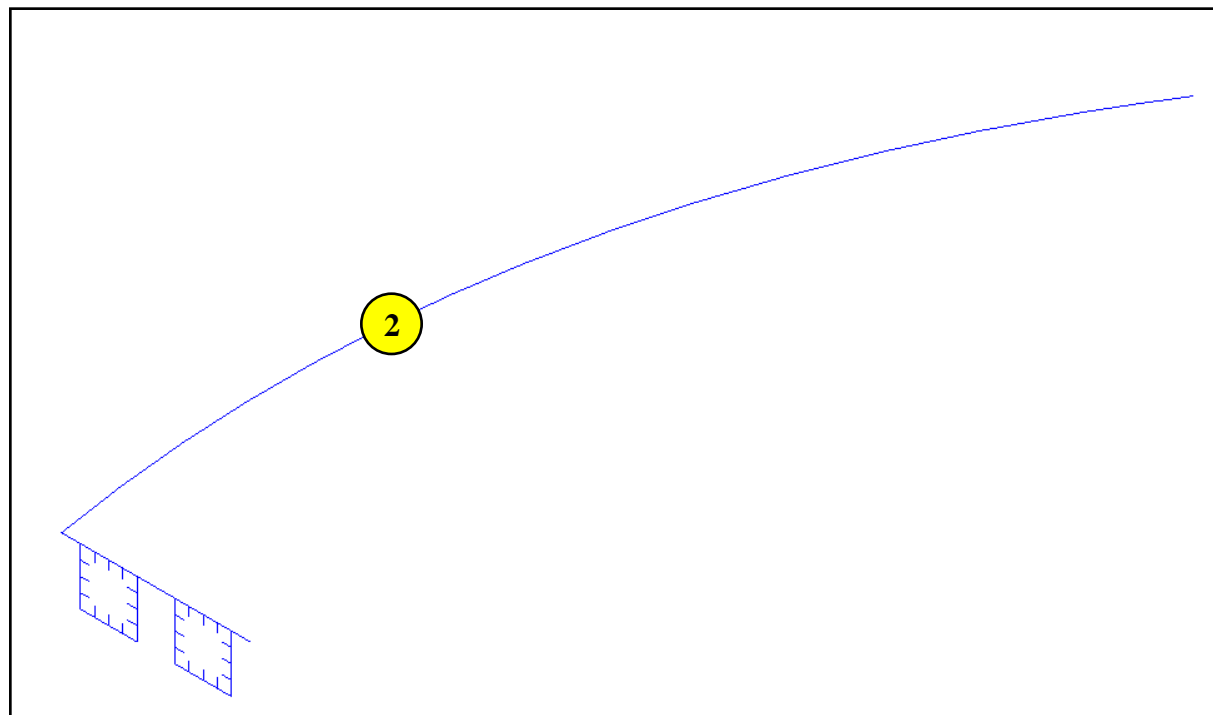


-  "Ctrl+A" as shortcut for "Select Displayed All".
-  "Enter" as shortcut for "Apply"

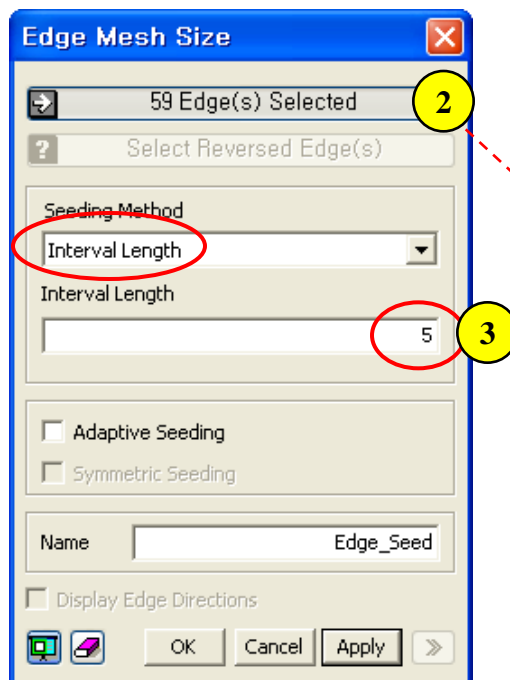
Step 7.



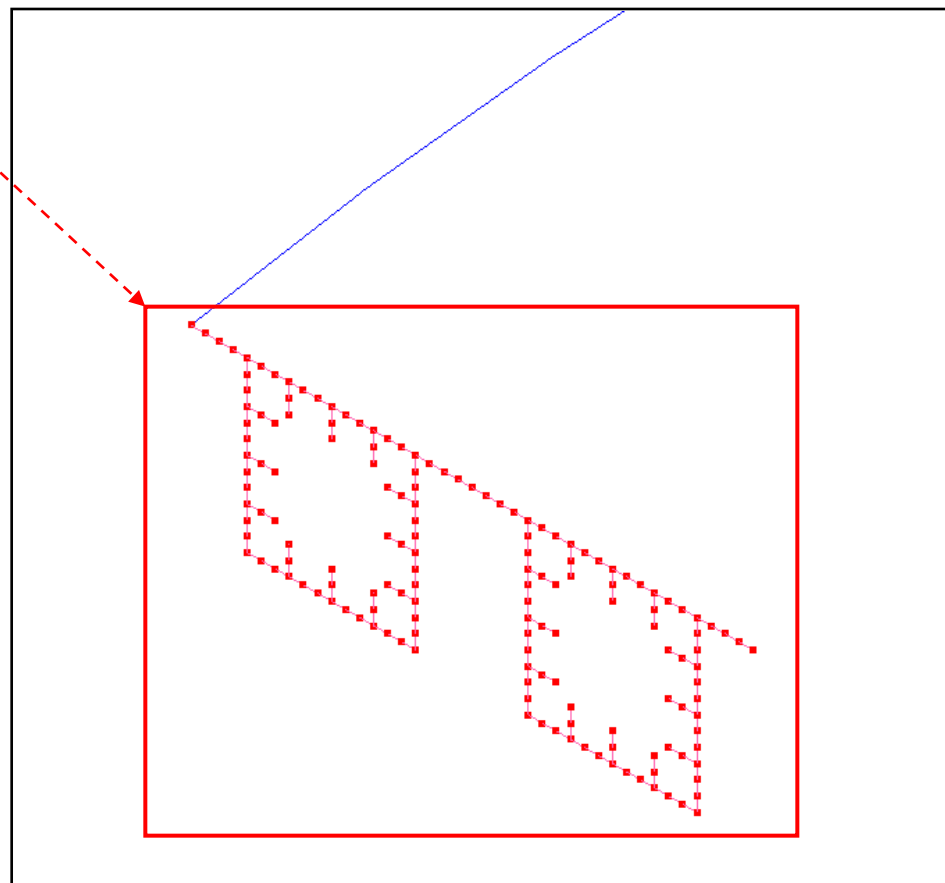
1. **Geometry > Curve > Create WP > B-Spline ...**
2. **Location : (0), <0, 500>, <200, 500>**
3. **Click Right Mouse Button on the Work Window**
4. **Click [Cancel] Button**



Step 8.

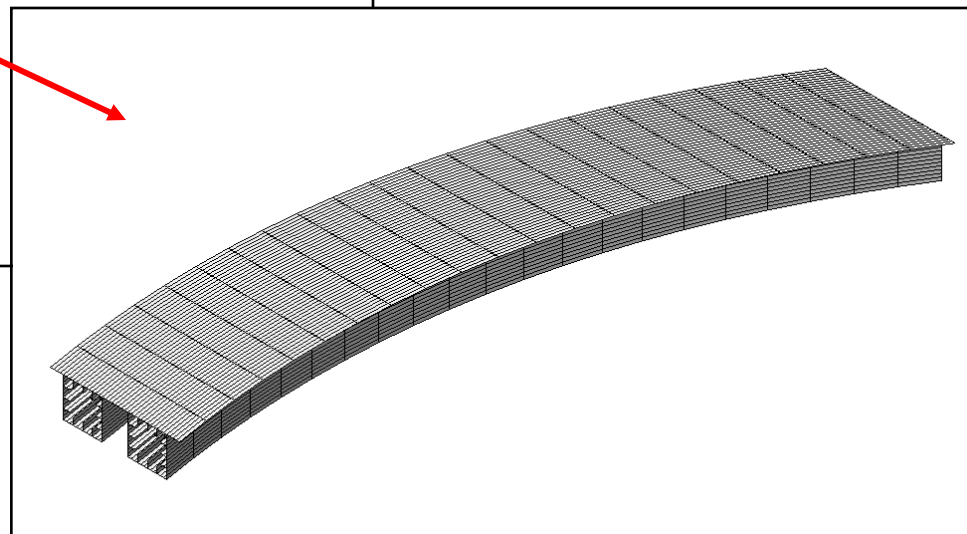
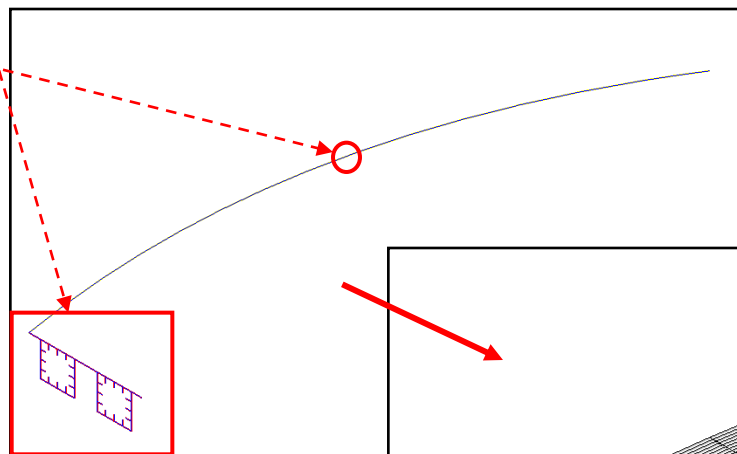
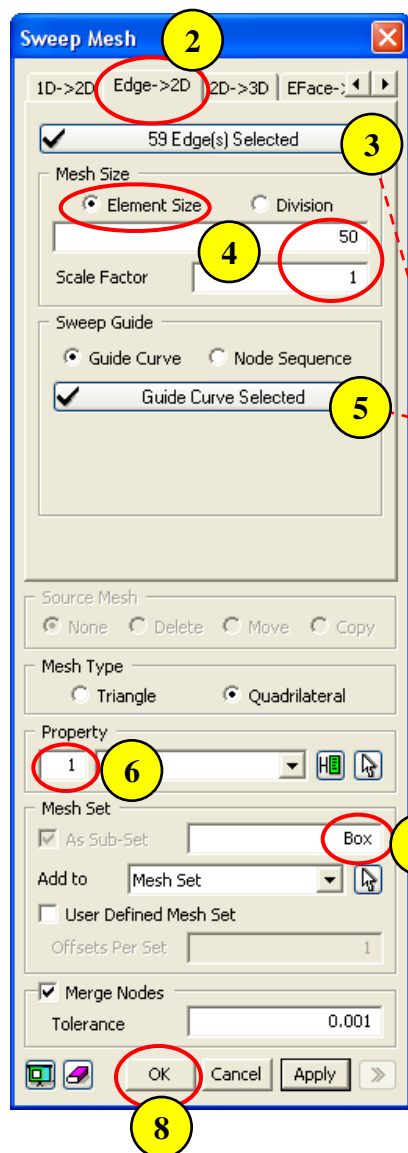


1. Mesh > Size Control > Along Edge ...
2. Select 59 Edges (See Figure)
3. Seeding Method - Interval Length : 5
4. Click [OK] Button

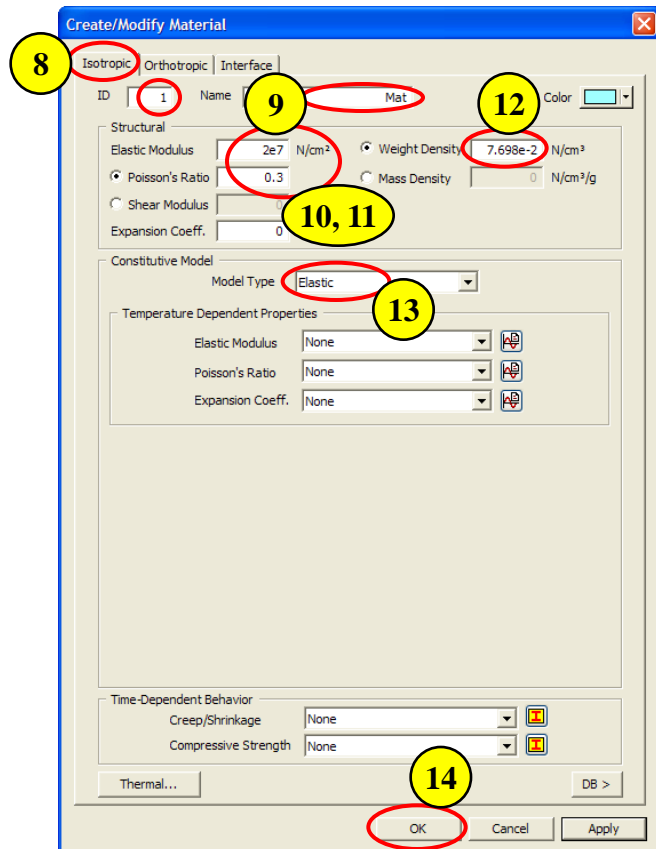
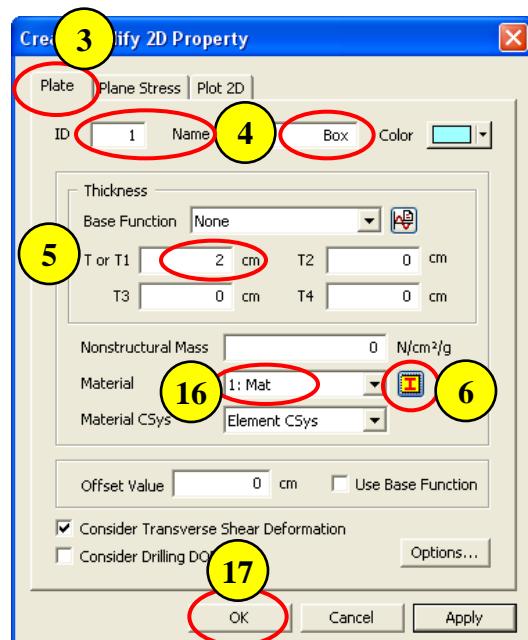
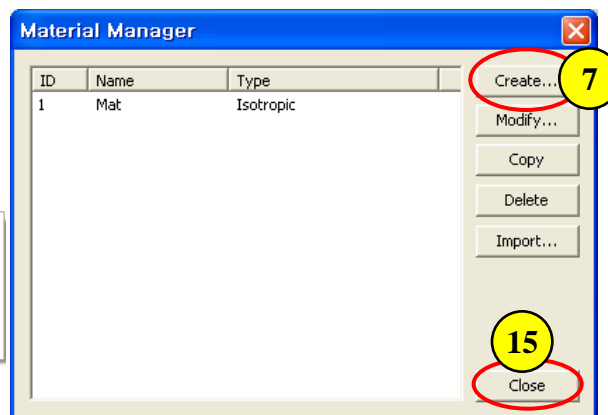
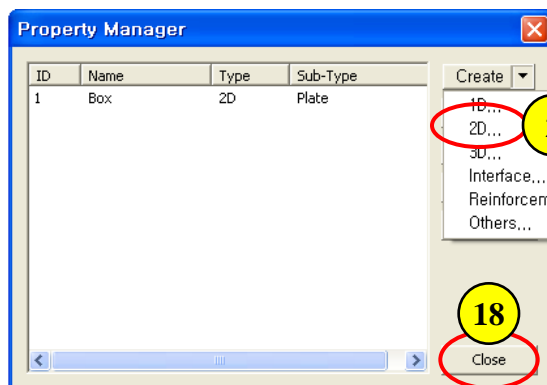



Step 9.

1. Mesh > Protrude Mesh > Sweep ...
2. Select "Edge->2D" Tab
3. Select 59 Edges (See Figure)
4. Mesh Size - Element Size : 50 , Scale Factor : 1
5. Select Guide Curve marked by "O" (See Figure)
6. Property (1)
7. Mesh Set : Box
8. Click [OK] Button

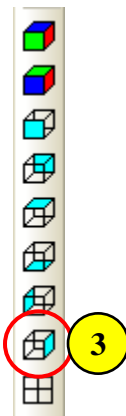
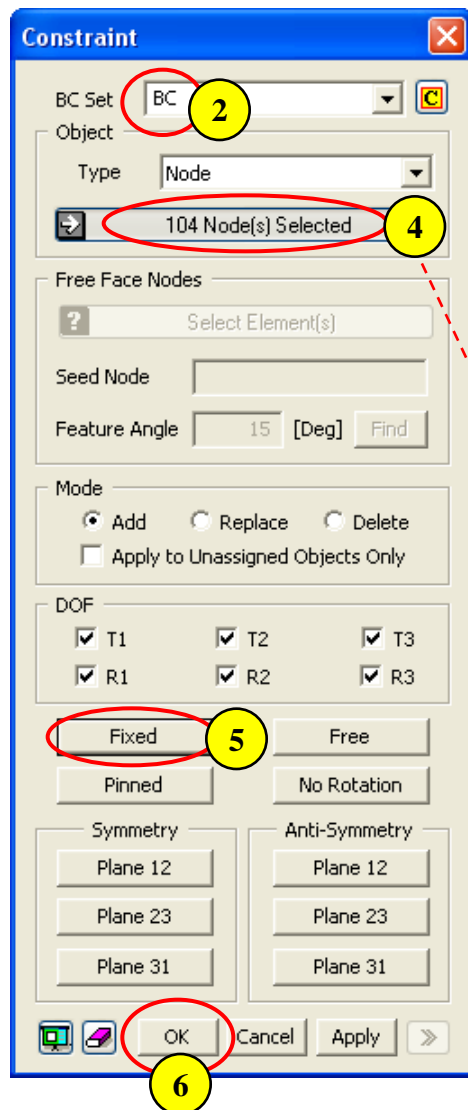


Step 10.

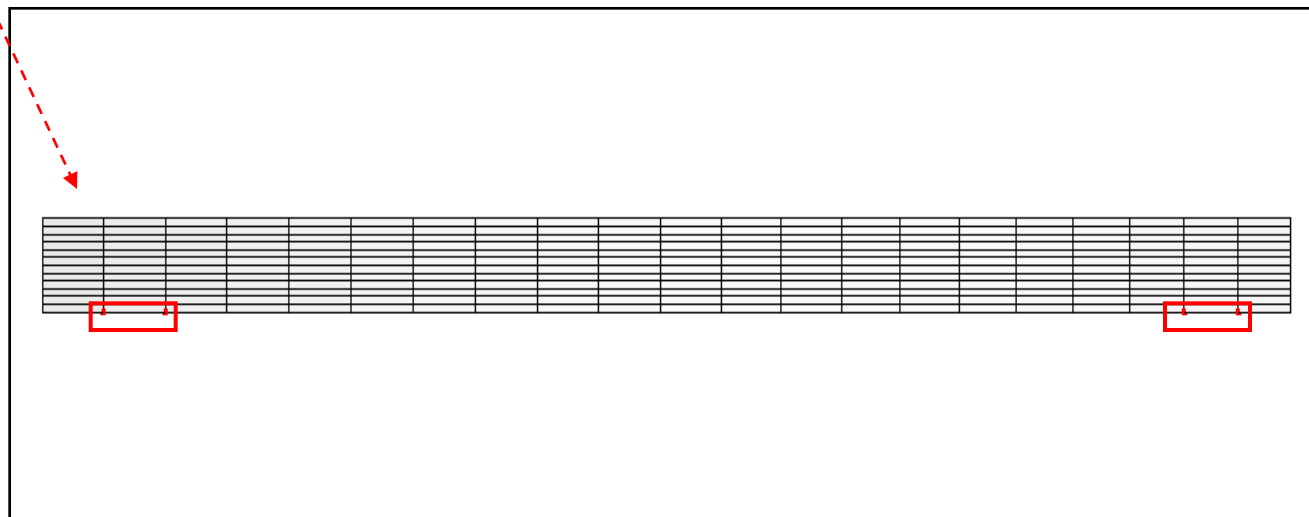


1. Analysis > Property ...
2. Create 2D ...
3. Select "Plate" tab
4. ID : 1 , Name : Box
5. T or T1 : 2 cm
6. Click  Button (Material)
7. Click [Create] Button
8. Select "Isotropic" tab
9. ID : 1 , Name : Mat
10. Elastic Modulus : $2e7 \text{ N/cm}^2$
11. Poisson's Ratio : 0.3
12. Weight Density : $7.698e-2 \text{ N/cm}^3$
13. Model Type : Elastic
14. Click [OK] Button
15. Click [Close] Button
16. Select "1: Mat" for Material
17. Click [OK] Button
18. Click [Close] Button

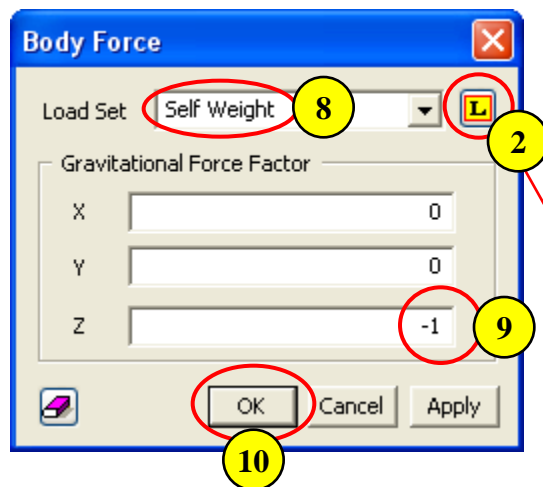
Step 11.




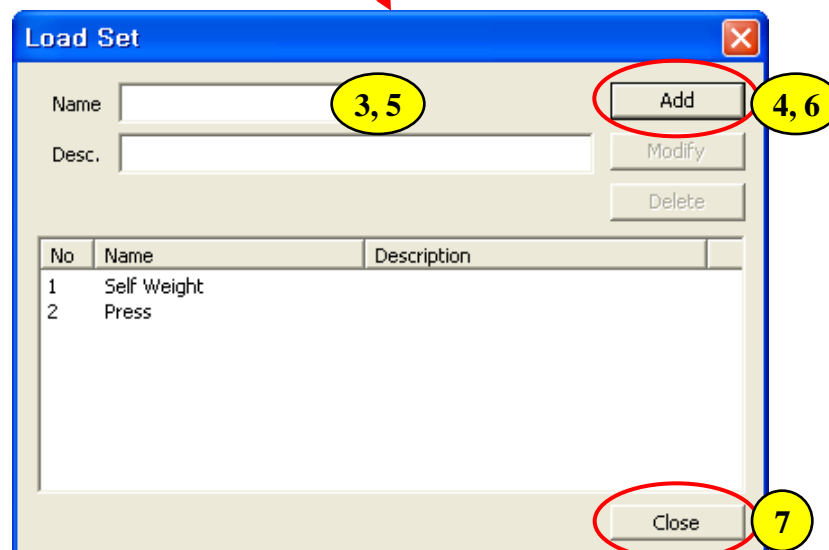
1. Analysis > BC > Constraint ...
2. BC Set : BC
3. Click "Right View"
4. Select 104 Nodes (See Figure)
5. Click [Fixed] Button
6. Click [OK] Button



Step 12.



1. Analysis > Load > Body Force ...
2. Click  Button
3. Name : Self Weight
4. Click [Add] Button
5. Name :Press
6. Click [Add] Button
7. Click [Close] Button
8. Select "Self Weight" for Load Set
9. Z: -1
10. Click [OK] Button



Step 13.

Pressure

Load Set: **Press** (2, 3)

Type: **Face Pressure**

Object Type: **2D Element** (4)

Object: **840 Element(s) Selected** (5)

Free Face/Edge: **Select Element(s)**

Seed Node:

Feature Angle: **15 [Deg]** **Find**

Mode: **Add** **Replace** **Delete**
☐ Apply to Unassigned Objects Only

Direction: **Ref. Csys-Axis 3** (6)

Ref. Csys: **Global Rectangular**

Select Direction

1,0,0

Projection: **Yes** **No**

☒ **Uniform**

Base Function: **None**

P or P1: **-5 N/cm²** (7)

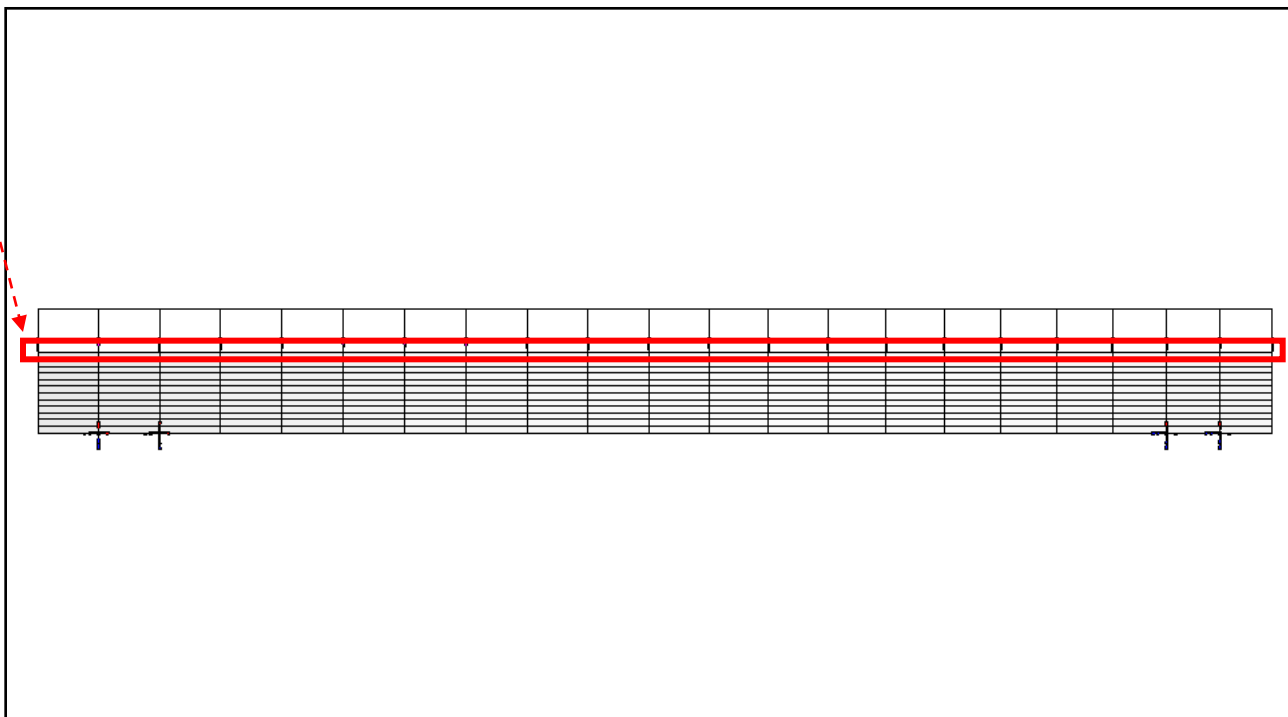
P2: **0 N/cm²**

P3: **0 N/cm²**

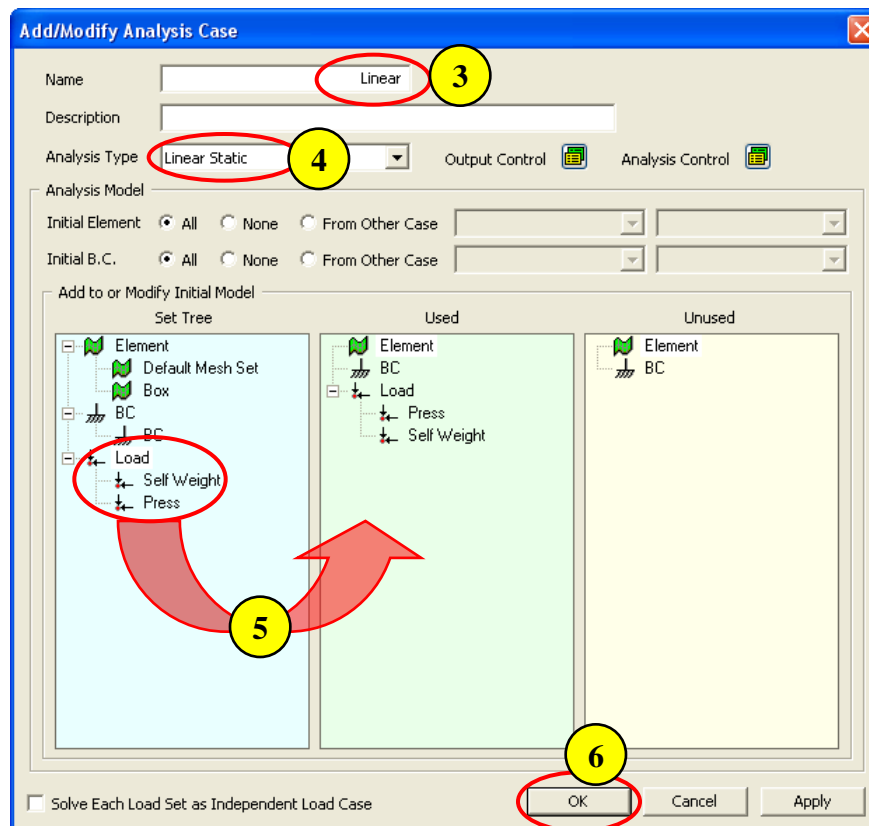
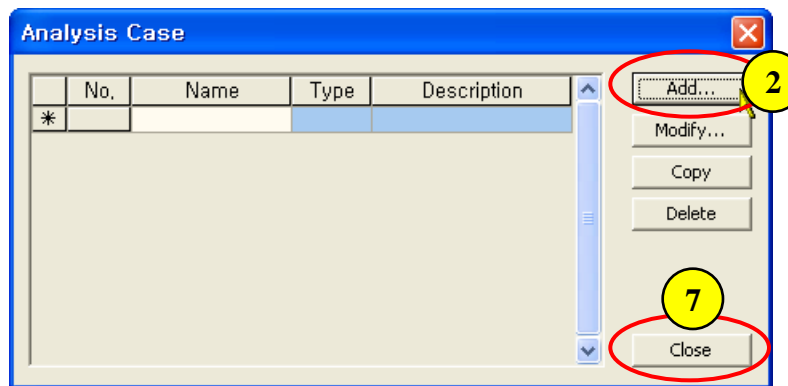
P4: **0 N/cm²**

OK (8) **Apply** **>>**

1. Analysis > Load > Pressure ...
2. Select "Press" for Load Set
3. Type : Face Pressure
4. Object Type : 2D Element
5. Select 840 Elements (See Figure)
6. Direction : Ref. Csys-Axis 3
7. P or P1 : -5 N/cm²
8. Click [OK] Button



Step 14.



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... (Box Girder.feb)

9. Analysis > Solve ...

10. Click [OK] Button

