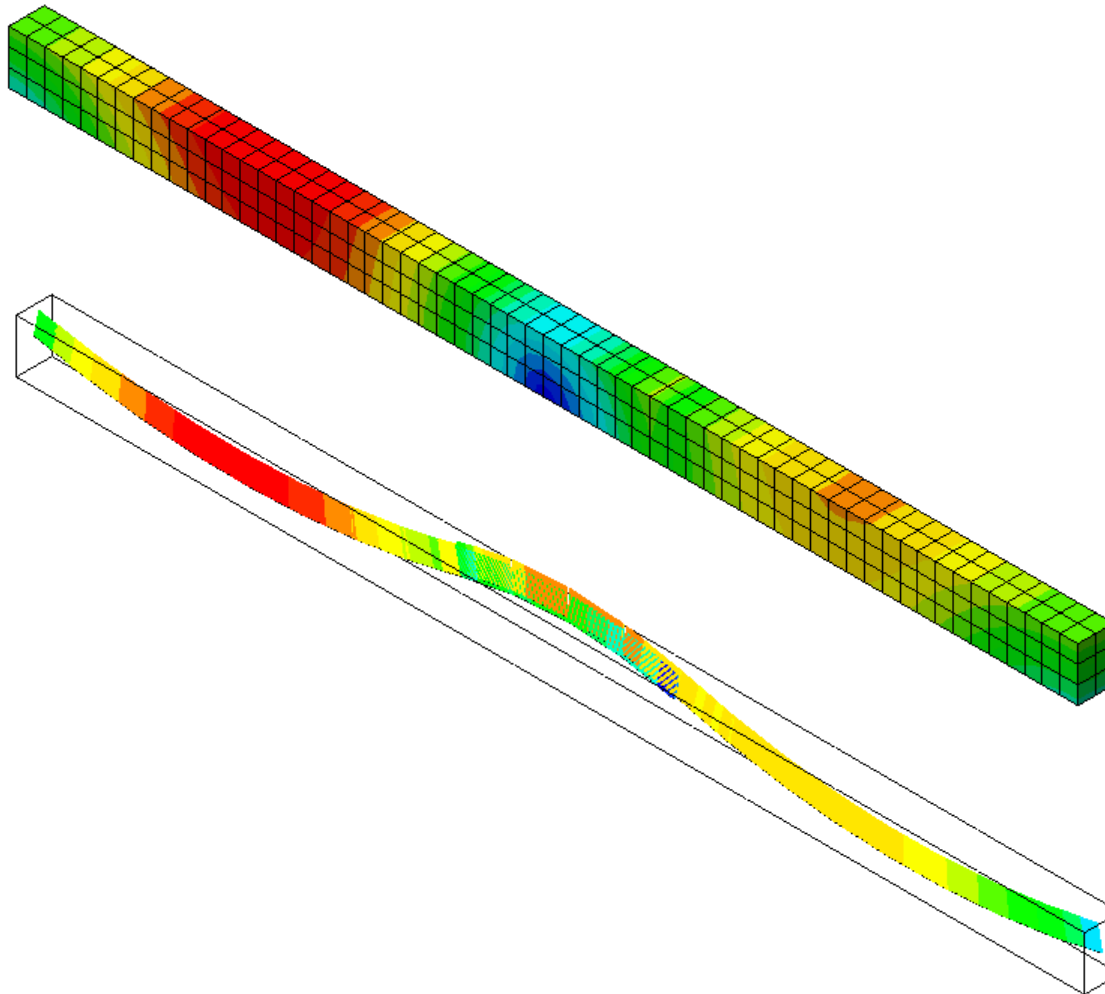


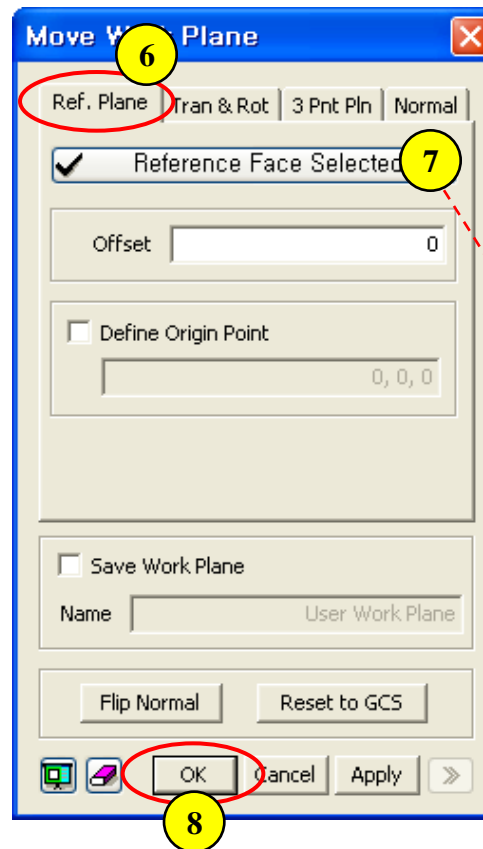
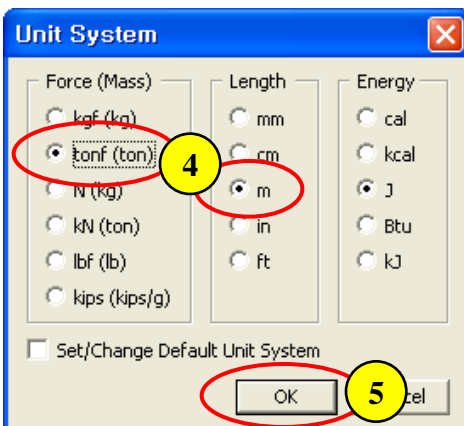
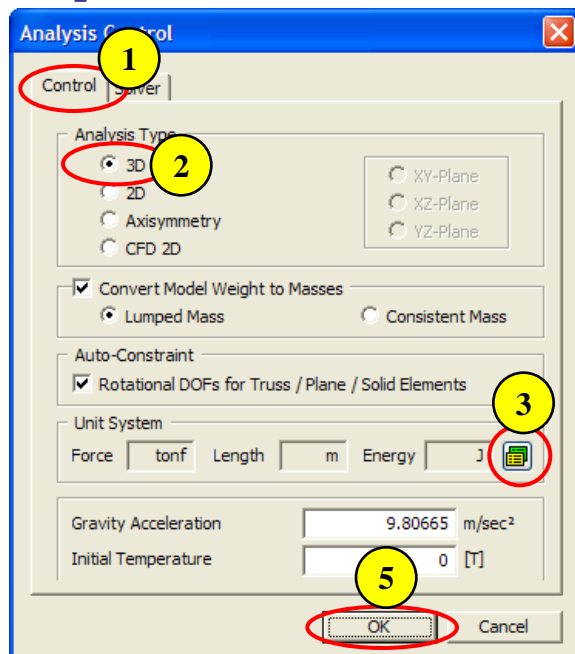
CS-1. PSC Beam



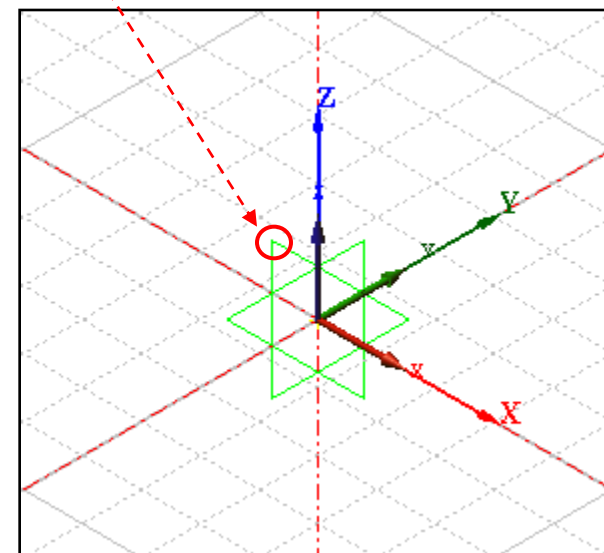
Overview

- 3-D Construction Stage Analysis
- Model
 - Unit : tonf, m
 - Isotropic Elastic Material
 - Time Dependent Material
 - Reinforcement Element
 - Solid Element
- Load & Boundary Conditions
 - Body Force
 - Prestress for Reinforcement
 - Constraint
 - Construction Stage
- Result Evaluation
 - Deformation
 - Reinforcement Stress

Step 1.

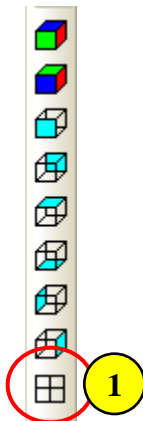
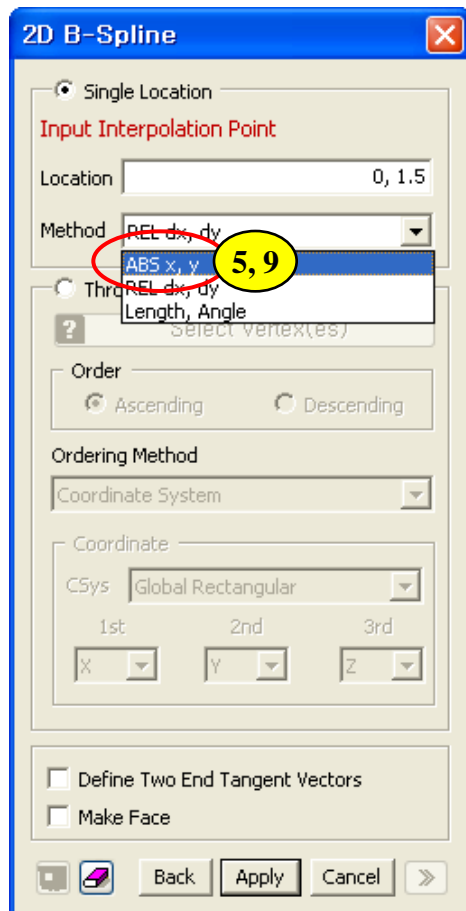


1. Analysis > Analysis Control
– “Control” tab
2. Analysis Type : 3D
3. Click Button
4. Unit : tonf , m
5. Click [OK] Button
6. Geometry > Work Plane > Move
– “Ref. Plane” tab
7. Select “XZ Plane”
8. Click [OK] Button

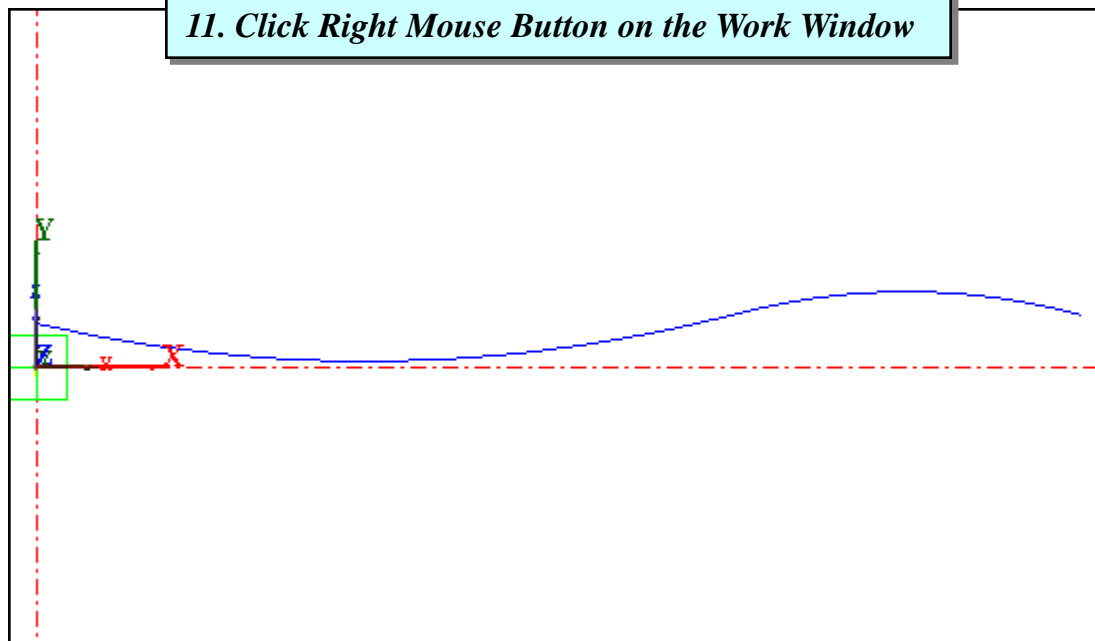


Analysis Control Dialog is automatically activated at startup.

Step 2.

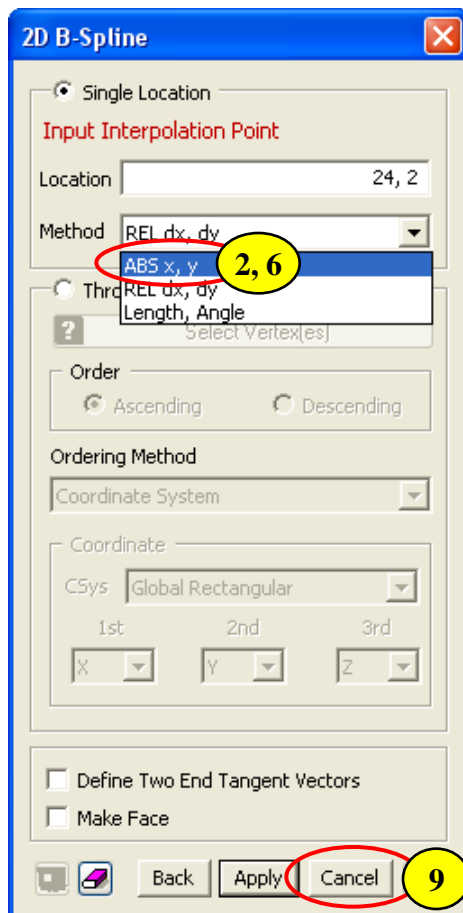


1. Click "Normal View"
2. Toggle off "Toggle Grid"
3. Geometry > Curve > Create on WP > B-Spline...
4. Location : (0, 1.5) ⚙
5. Change Method to "ABS x, y"
6. Location : (12, 0.2), (24, 1.8)
7. Click Right Mouse Button on the Work Window
8. Location : (24, 1.8)
9. Change Method to "ABS x, y"
10. Location : (30, 2.6), (36, 1.8)
11. Click Right Mouse Button on the Work Window

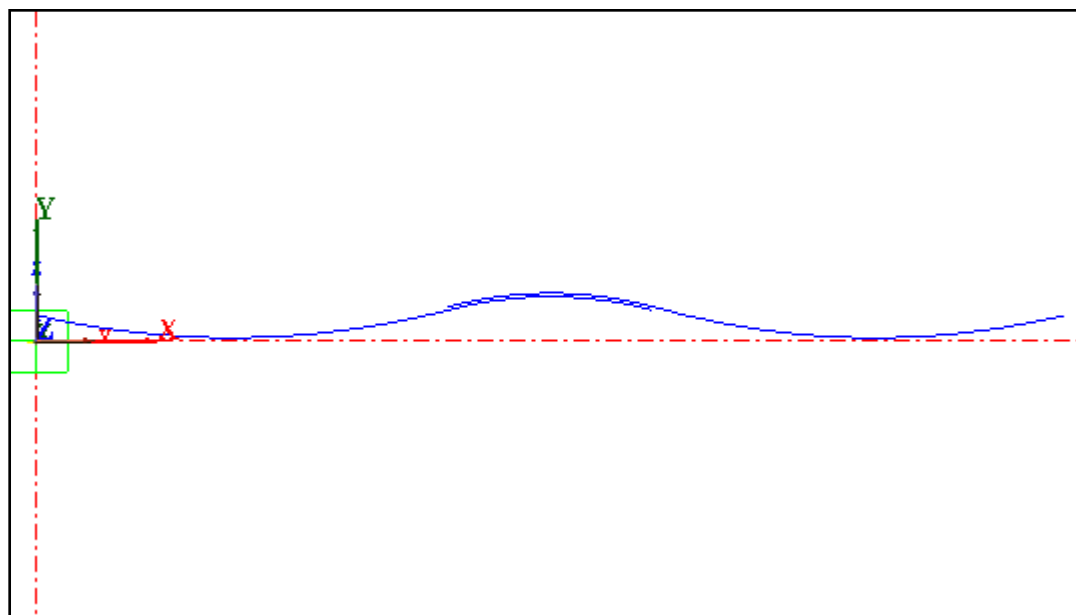


⚙ () : "ABS x, y", ⚙ : "REL dx, dy"

Step 3.

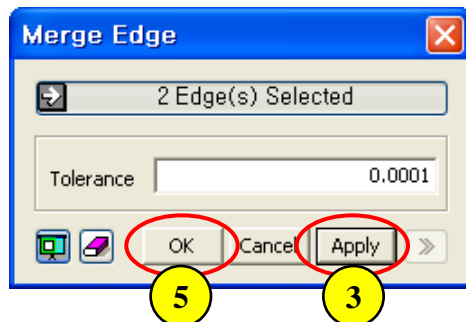


1. Location : (24, 2)
2. Change Method to “ABS x, y”
3. Location : (30, 2.8), (36, 2)
4. Click Right Mouse Button on the Work Window
5. Location : (36, 2)
6. Change Method to “ABS x, y”
7. Location : (48, 0.2), (60, 1.5)
8. Click Right Mouse Button on the Work Window
9. Click [Cancel] Button

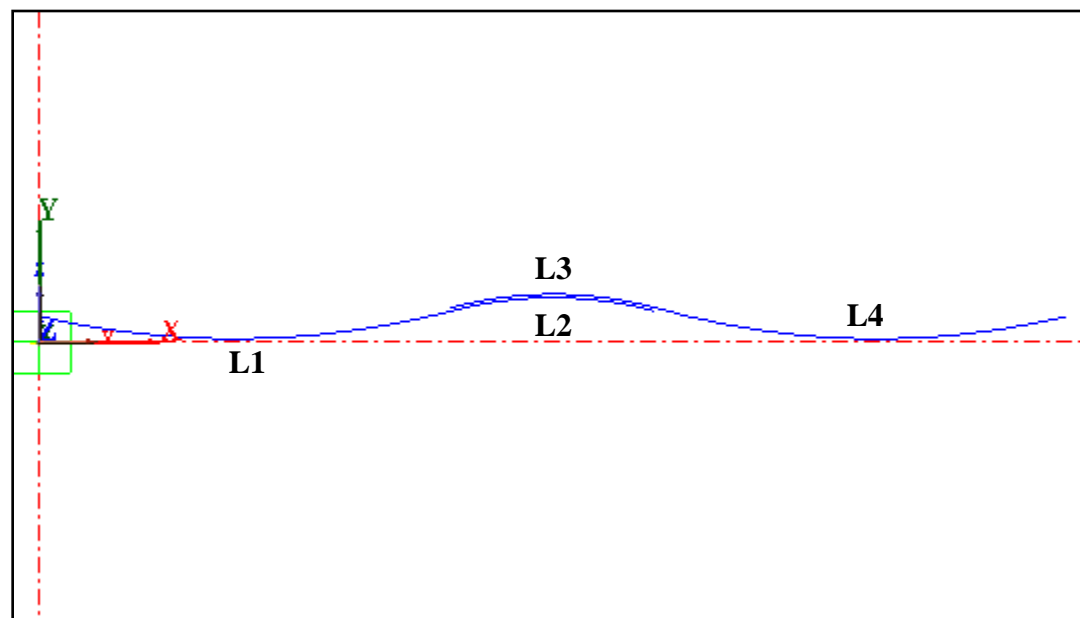


[Esc] as shortcut for [Cancel].

Step 4.

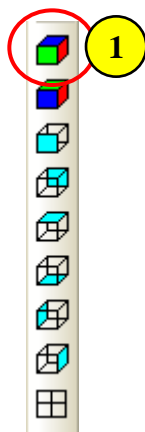
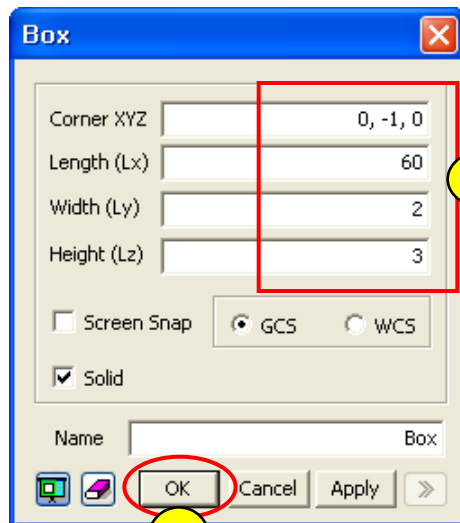


1. Geometry > Curve > Merge...
2. Select L1 & L2 (See Figure)
3. Click [Apply] Button
4. Select L3 & L4 (See Figure)
5. Click [OK] Button

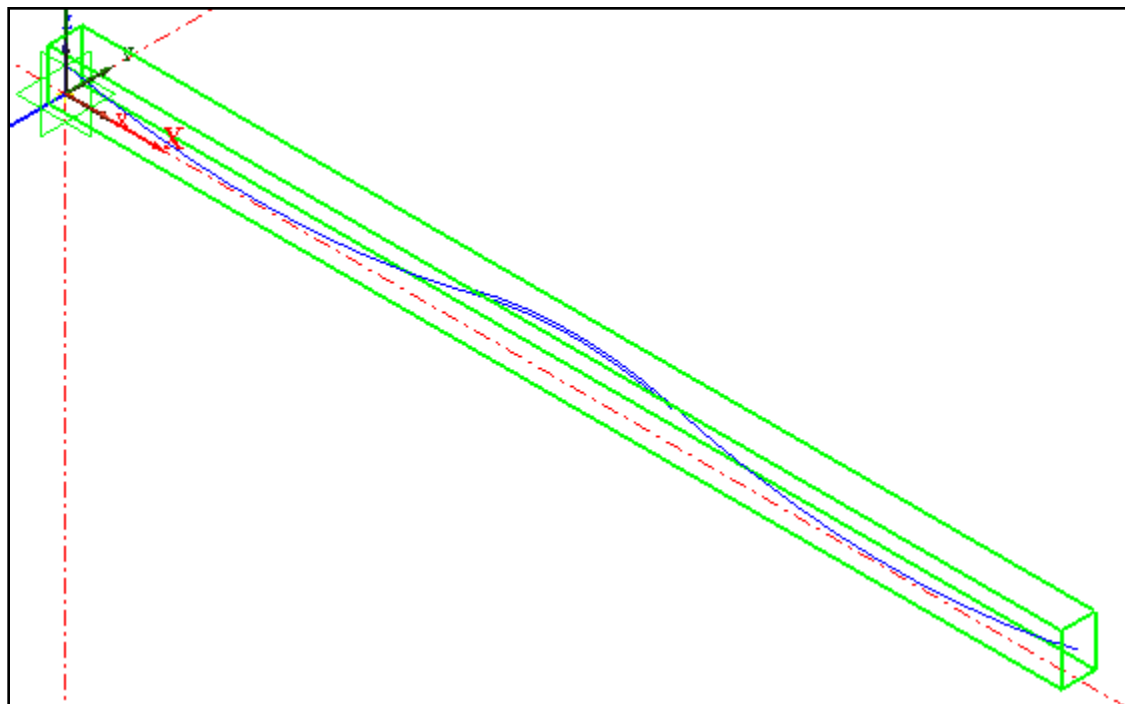


[Enter] as shortcut for [Apply].

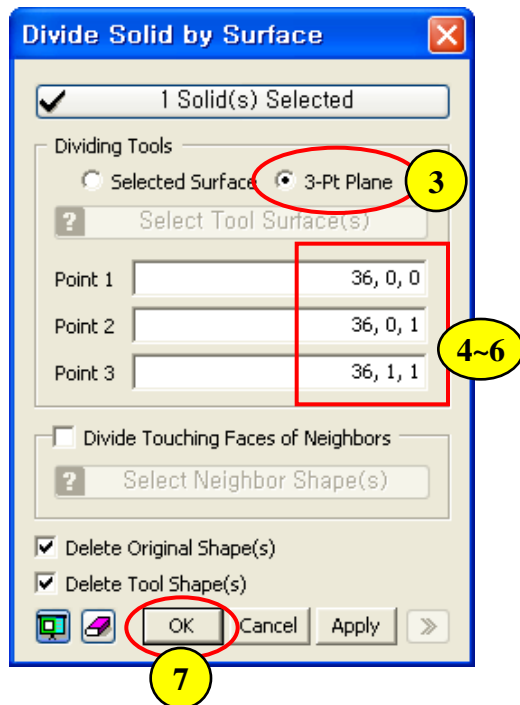
Step 5.



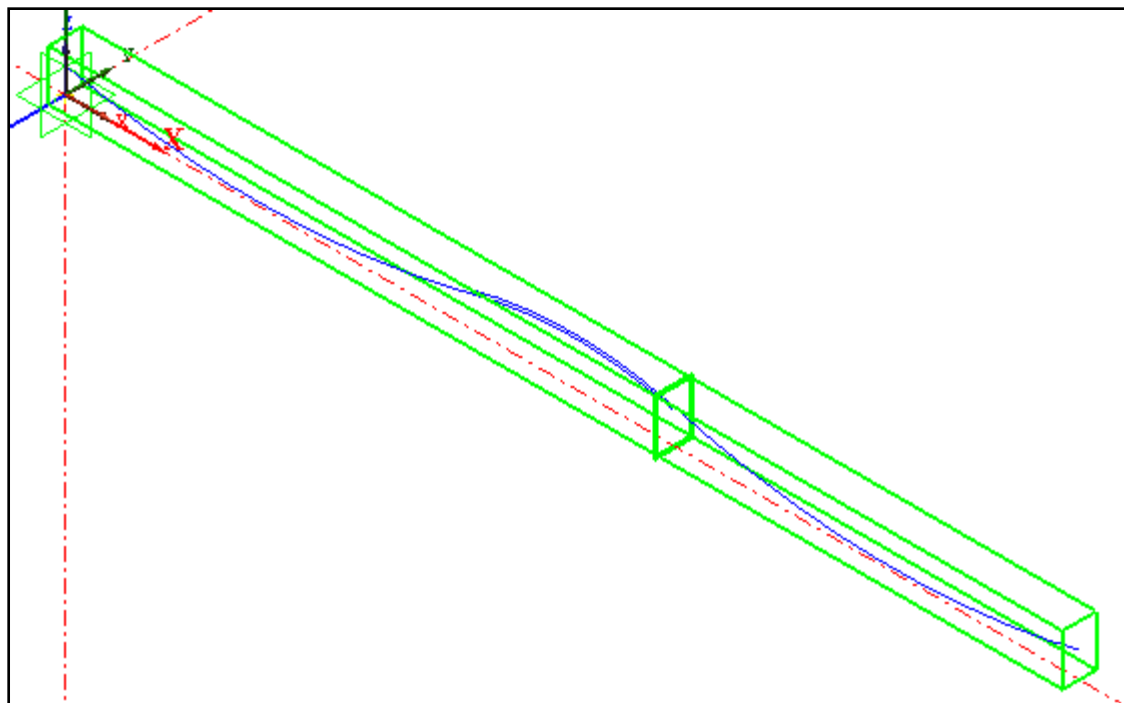
1. Click "Isometric 1 View"
2. Geometry > Primitive Feature > Box...
3. Corner XYZ : (0, -1, 0)
4. Length (Lx) : 60
5. Width (Ly) : 2
6. Height (Lz) : 3
7. Click [OK] Button



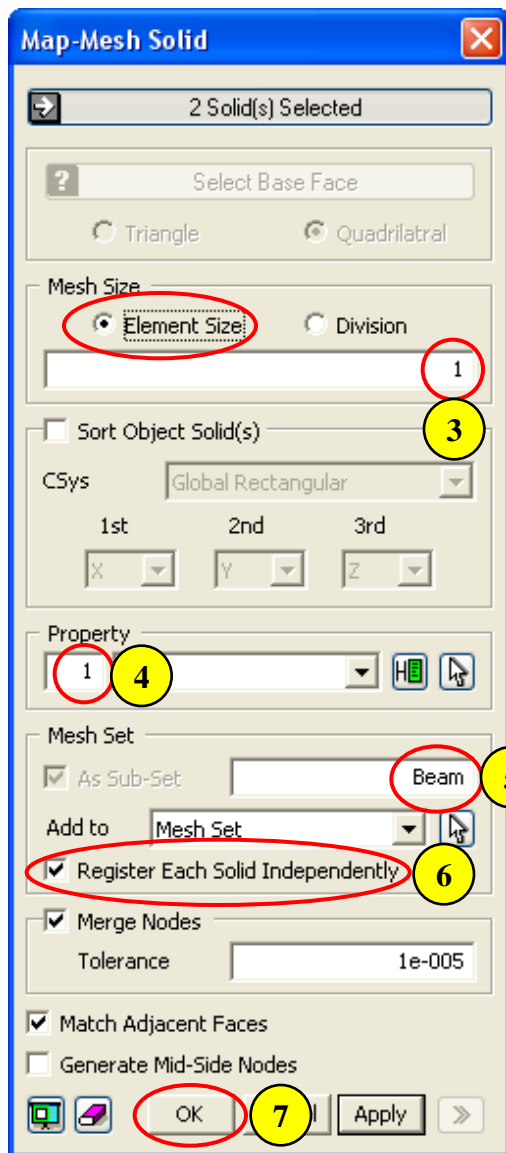
Step 6.



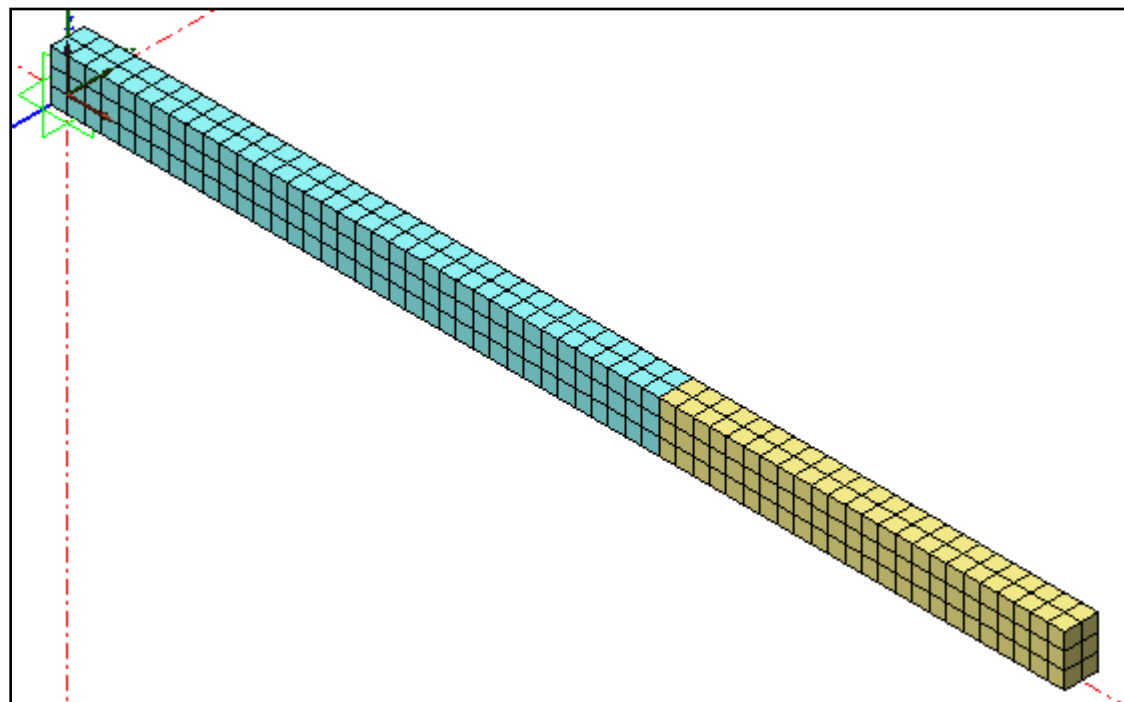
1. Geometry > Solid > Divide...
2. Select Solid Box
3. Check on "3-Pt Plane"
4. Point 1 : (36, 0, 0)
5. Point 2 : (36, 0, 1)
6. Point 3 : (36, 1, 1)
7. Click [OK] Button



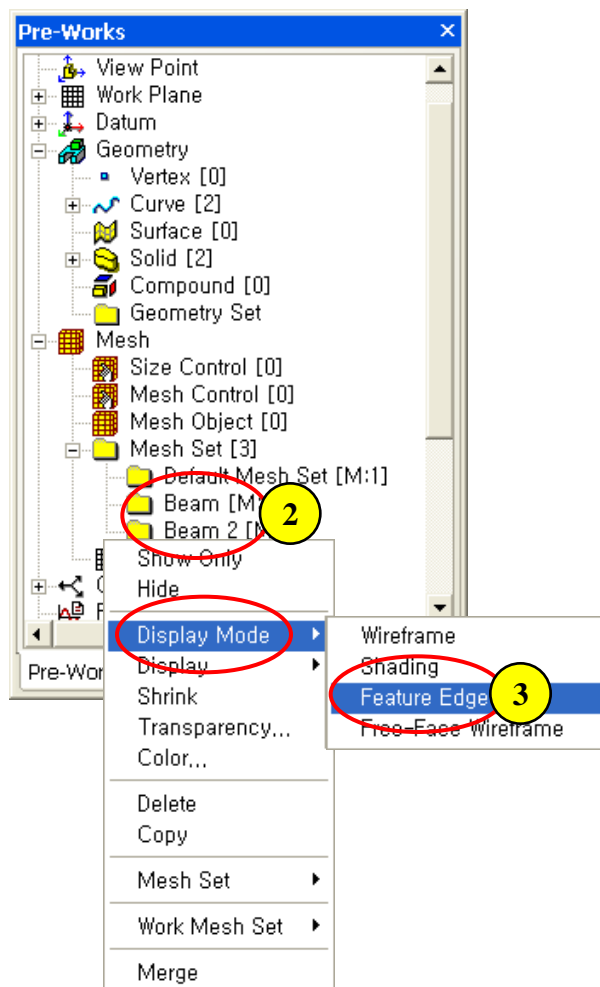
Step 7.



1. *Mesh > Map Mesh > Solid...*
2. *Select 2 Solids*
3. *Mesh Size : Element Size (1)*
4. *Property : 1*
5. *Mesh Set : Beam*
6. *Check on "Register Each Solid Independently"*
7. *Click [OK] Button*



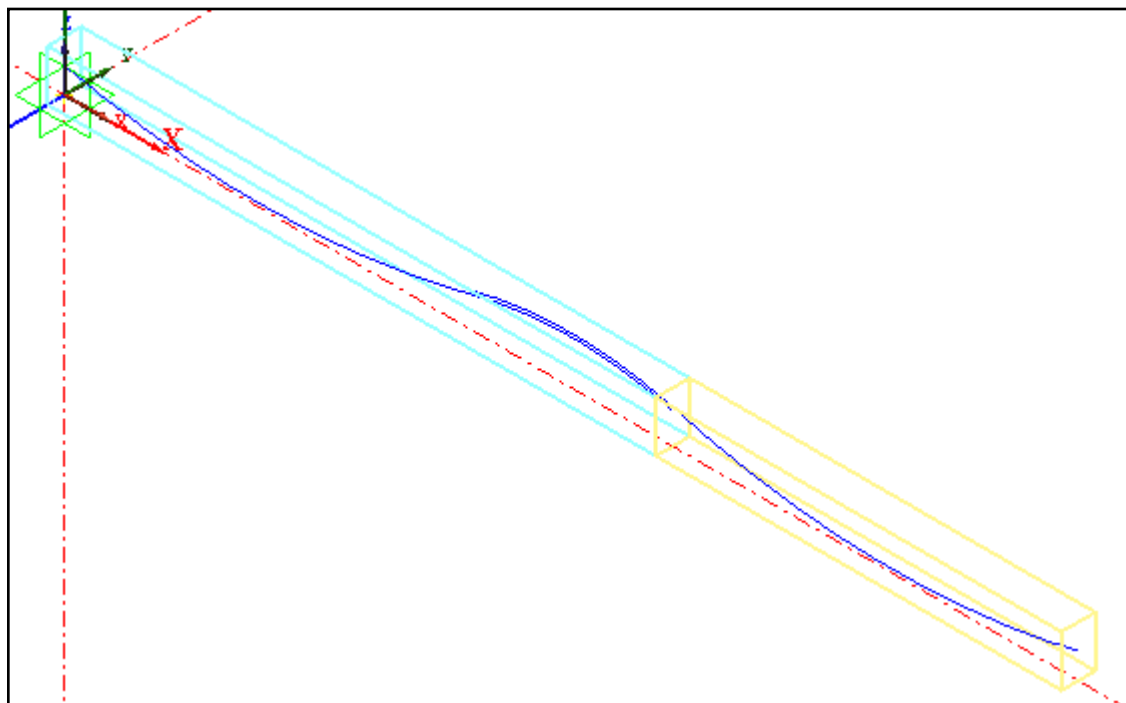
Step 8.



1. Pre-Works Tree : Mesh > Mesh Set...

2. Select "Beam" & "Beam 2"

3. Click Right Mouse Button and Select "Display Mode > Feature Edge"



Step 9.

Time-Dependent Material (Creep/Shrinkage)

Name: Creep/Shrinkage Code: CEB-FIP

CEB

Compressive Strength of Concrete at Age of 28 Days: 4000 tonf/m²

Relative Humidity of Ambient Environment (40 ~ 99): 70 %

Notational Size of Member: 1.2 m

$h = 2 * A_c / u$ (A_c : Section Area, u : Perimeter in Contact with Atmosphere)

Type of Cement

☐ Rapid Hardening High Strength Cement (RS)

☒ Normal or Rapid Hardening Cement (N, R)

☐ Slowly Hardening Cement (SL)

Age of Concrete at Beginning of Shrinkage: 3 day

Show Result... **OK** Cancel Apply

1. Analysis > Time-Dependent Material > Creep/Shrinkage...

2. Name : Creep/Shrinkage

3. Code : CEB-FIP

4. Compressive Strength of Concrete at Age of 28 Days : 4000 tonf/m²

5. Relative Humidity of Ambient Environment (40~99) : 70 %

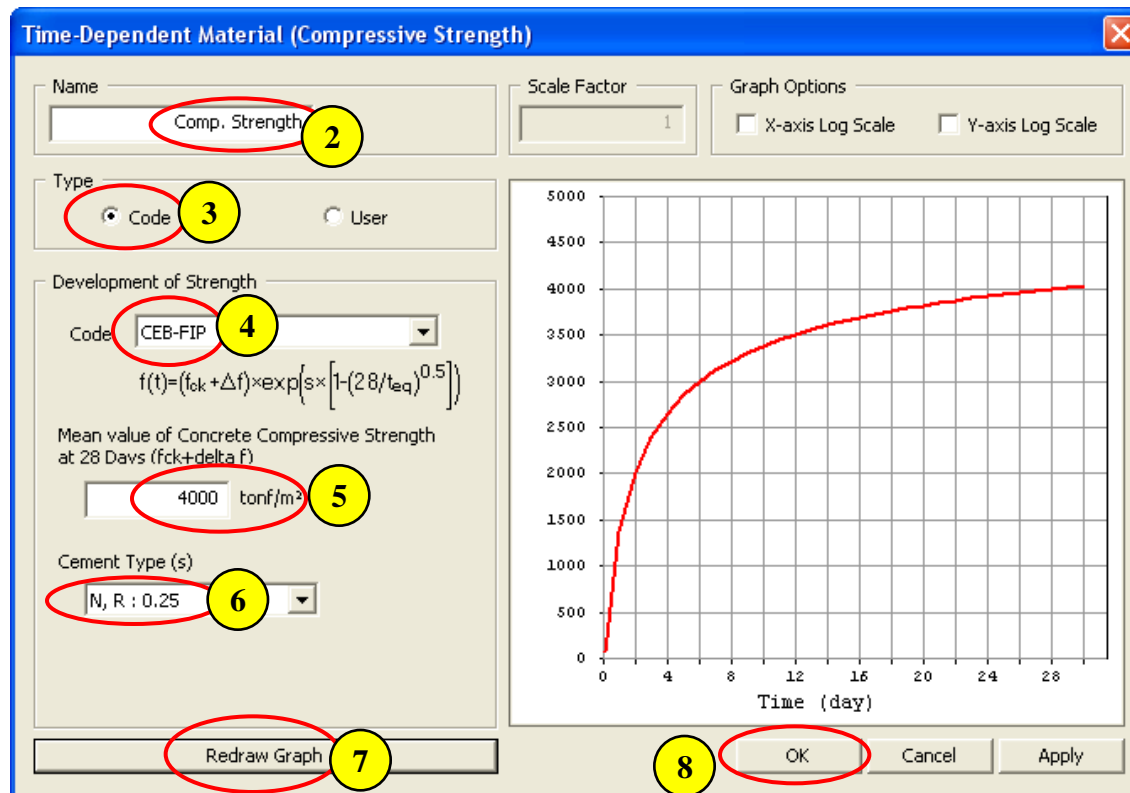
6. Notational Size of Member : 1.2 m

7. Type of Cement : Normal or Rapid Hardening Cement (N, R)

8. Age of Concrete at Beginning of Shrinkage : 3 Days

9. Click [OK] Button

Step 10.



1. Analysis > Time-Dependent Material
> Compressive Strength...

2. Name : Comp. Strength

3. Type : Code

4. Code : CEB-FIP

5. Compressive Strength of Concrete
at Age of 28 Days : 4000 tonf/m²

6. Cement Type (s)- N, R : 0.25

7. Click [Redraw Graph] Button

8. Click [OK] Button

Step 11.

3 Isotropic

ID 1 Name Conc_C400 Color

4, 13

4, 13

6~9 15~18

5, 14

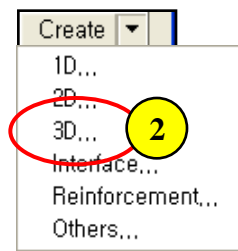
9, 10, 19

20

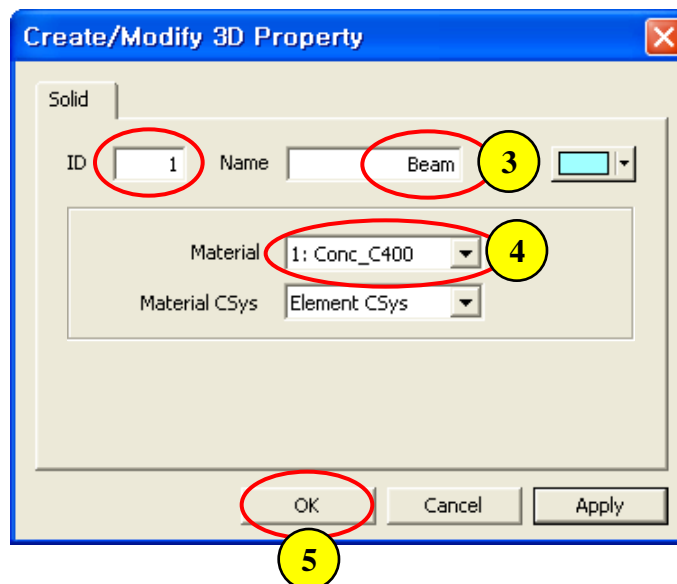
12

1. Analysis > Material ...
2. Click [Create] Button
3. Select "Isotropic" tab
4. ID : 1 , Name : Conc_C400
5. Model Type : Elastic
6. Elastic Modulus : 2.793e6 tonf/m²
7. Poisson's Ratio : 0.167
8. Expansion Coeff. : 1e-5
9. Weight Density : 2.5 tonf/m³
10. Creep/Shrinkage : Creep/Shrinkage
11. Compressive Strength : Comp. Strength
12. Click [Apply] Button
13. ID : 2 , Name : Tendon
14. Model Type : Elastic
15. Elastic Modulus : 2.0e7 tonf/m²
16. Poisson's Ratio : 0
17. Expansion Coeff. : 0
18. Weight Density : 7.85 tonf/m³
19. Creep/Shrinkage & Compressive Strength : None
20. Click [OK] Button
21. Click [Close] Button

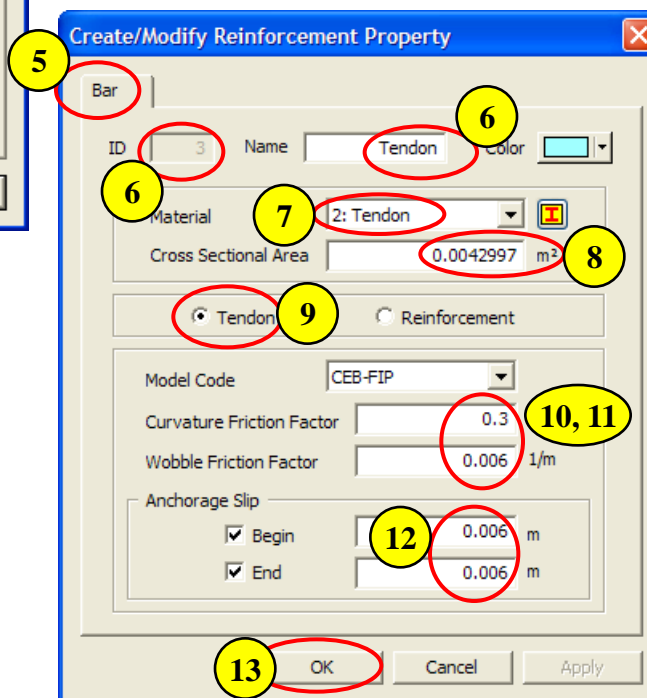
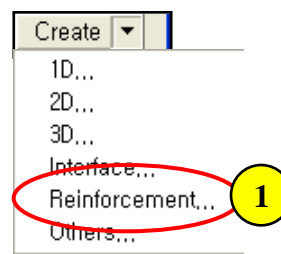
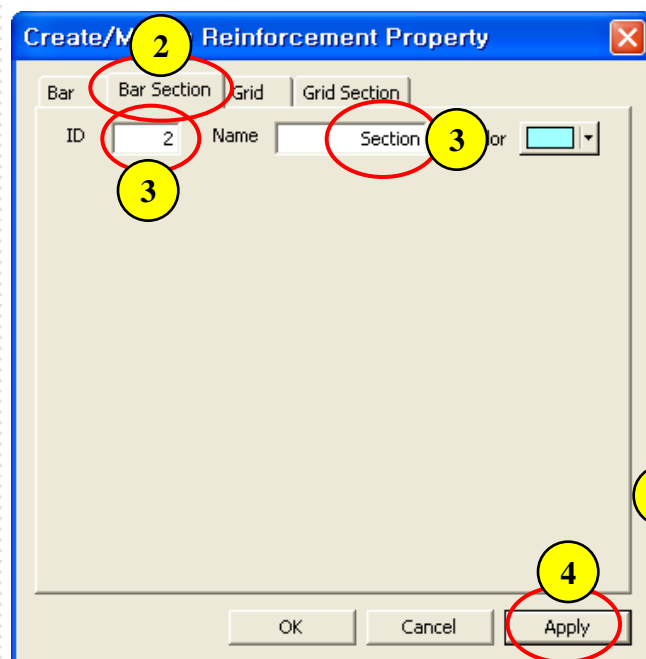
Step 12.



1. Analysis > Property...
2. Create 3D...
3. ID : 1 , Name : Beam
4. Select "1: Conc_C400" for Material
5. Click [OK] Button

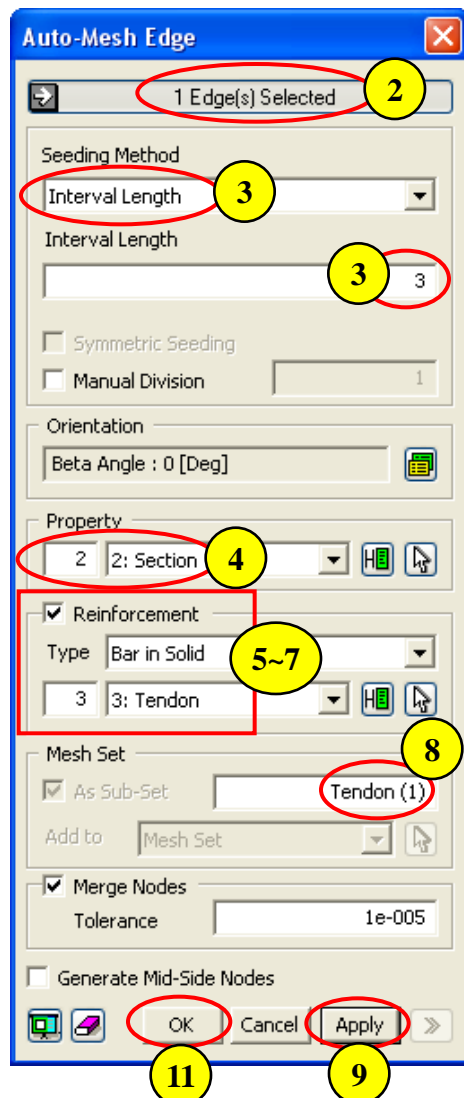


Step 13.

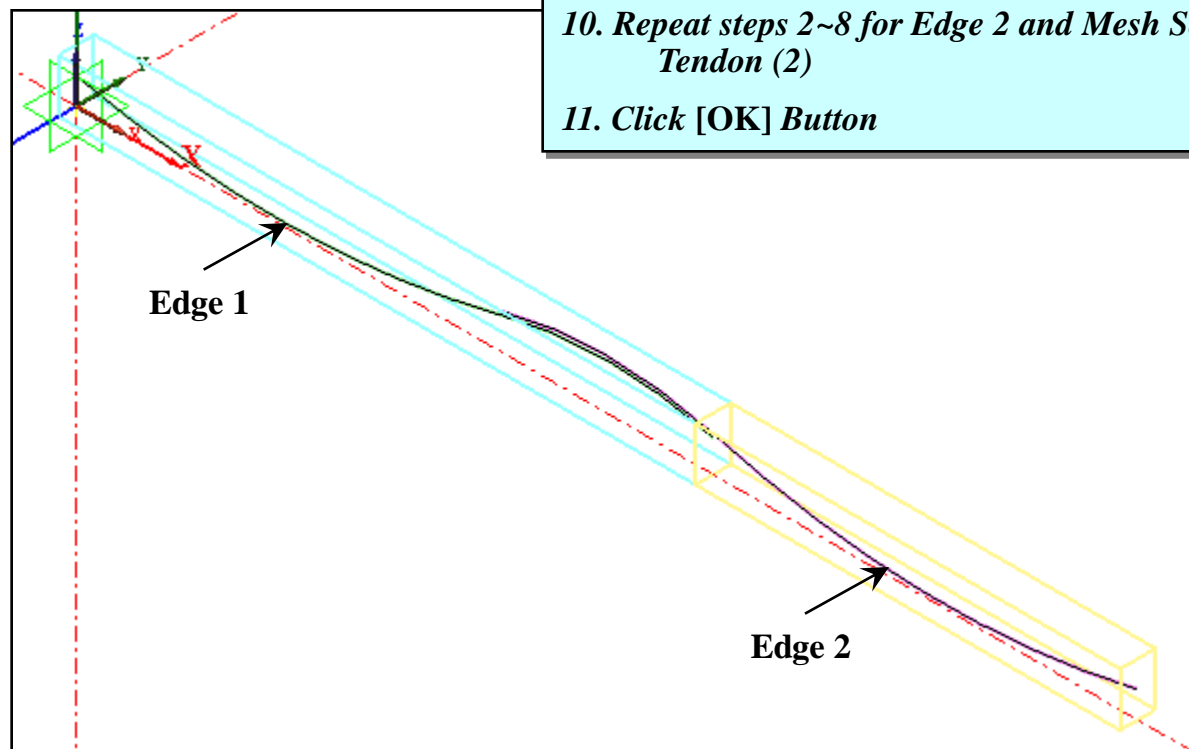


1. Create Reinforcement ...
2. Select "Bar Section" tab
3. ID : 2 , Name : Section
4. Click [Apply] Button
5. Select "Bar" tab
6. ID : 3 , Name : Tendon
7. Select "2: Tendon" for Material
8. Cross Section Area : 4.2997e-3 m²
9. Check on "Tendon"
10. Curvature Friction Factor : 0.3
11. Wobble Friction Factor : 0.006
12. Anchorage Slip : 0.006 m
13. Click [OK] Button
14. Click [Close] Button

Step 14.



1. Mesh > Auto Mesh > Edge ...
2. Select Edge 1
3. Seeding Method : Interval Length (3)
4. Property : (2: Section)
5. Check on "Reinforcement"
6. Type : Bar in Solid
7. Property : (3: Tendon)
8. Mesh Set : Tendon (1)
9. Click [Apply] Button
10. Repeat steps 2~8 for Edge 2 and Mesh Set: Tendon (2)
11. Click [OK] Button

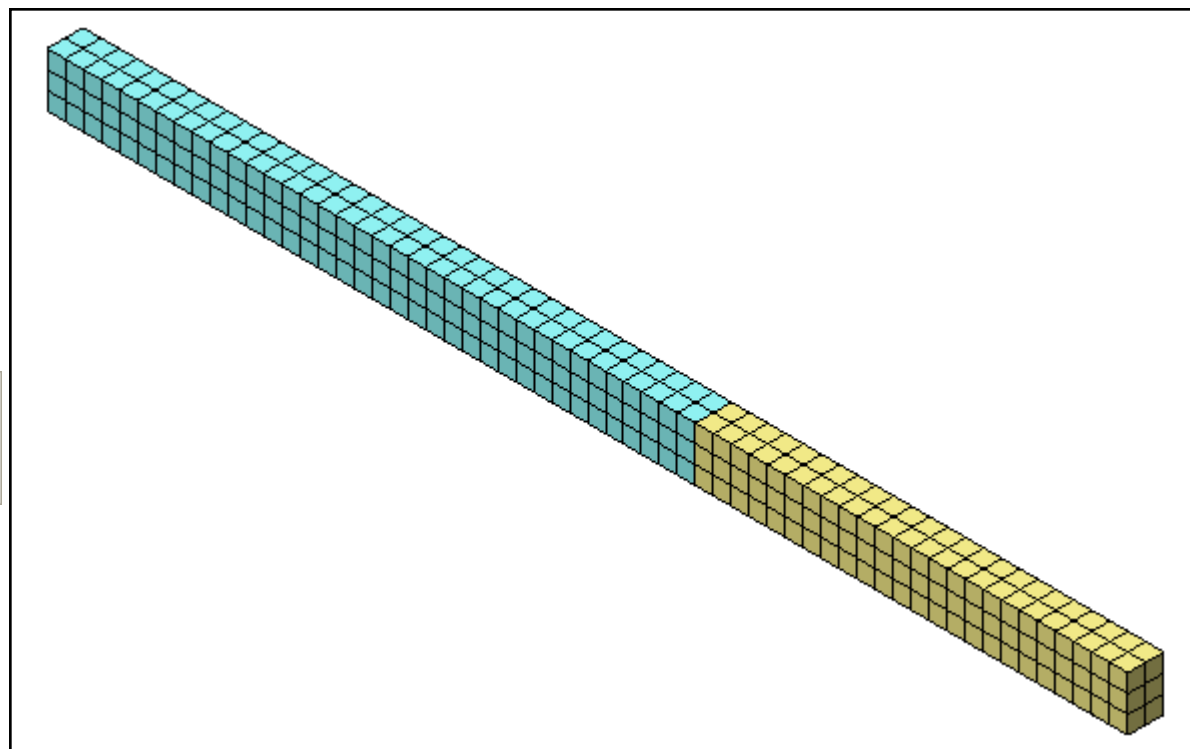
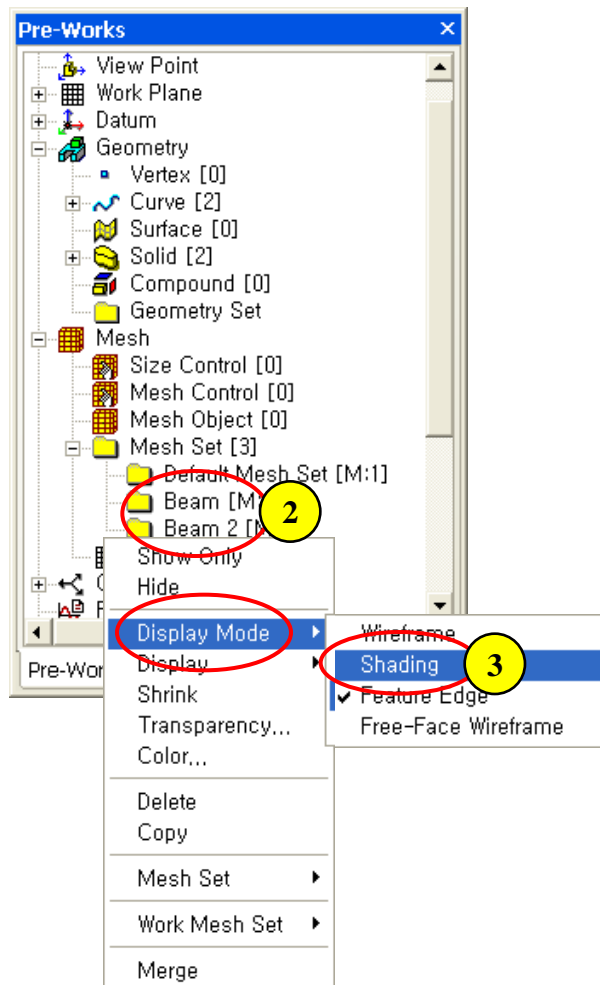


Step 15.

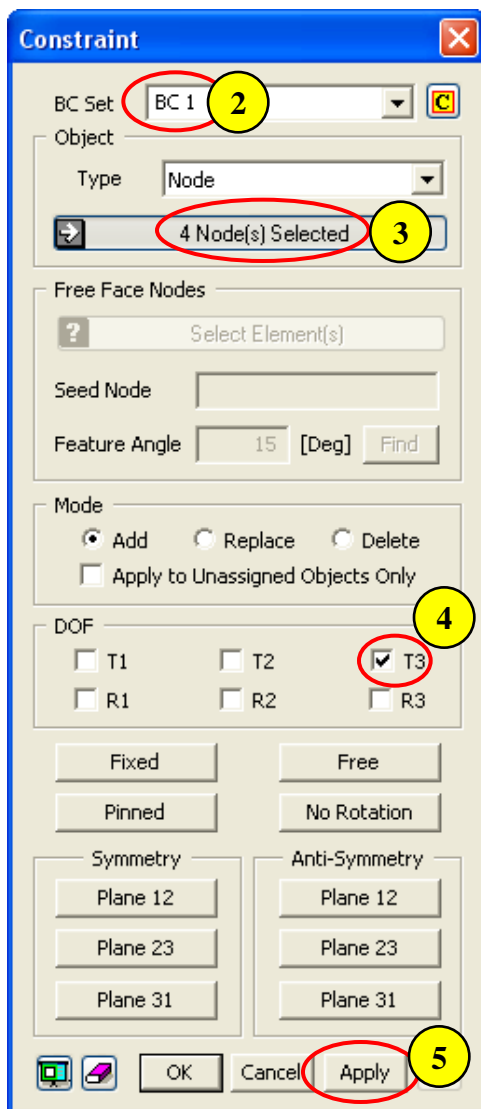
1. Pre-Works Tree : Mesh > Mesh Set...

2. Select "Beam" & "Beam 2"

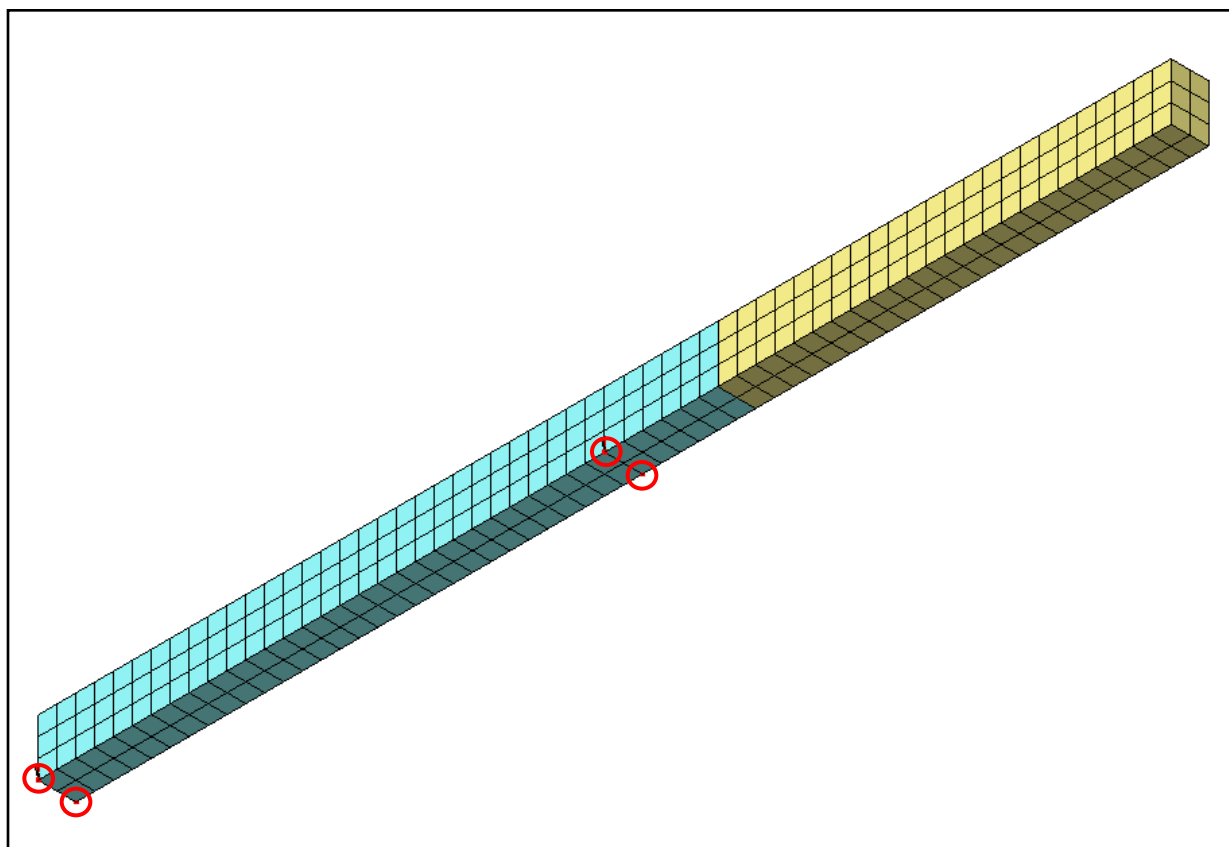
3. Click Right Mouse Button and Select "Display Mode > Shading"



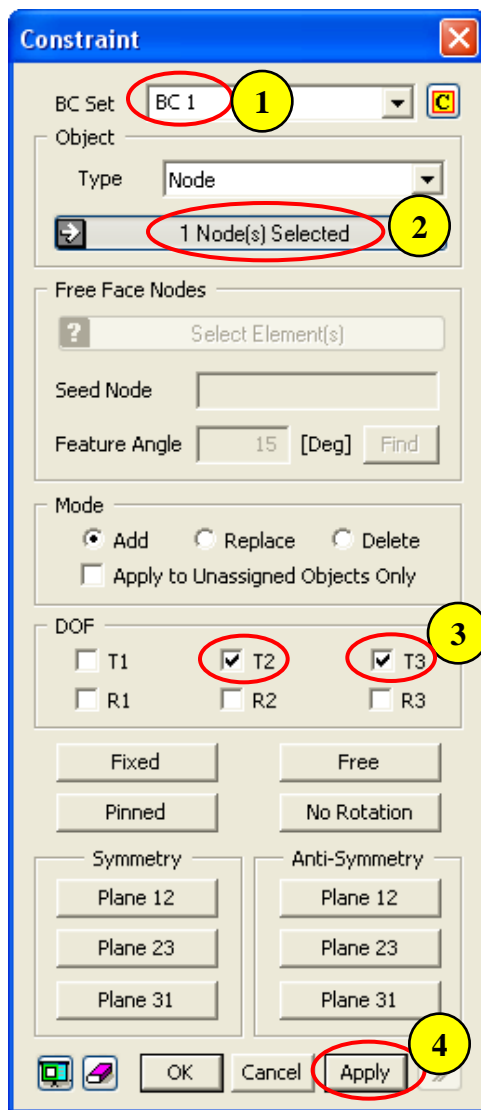
Step 16.



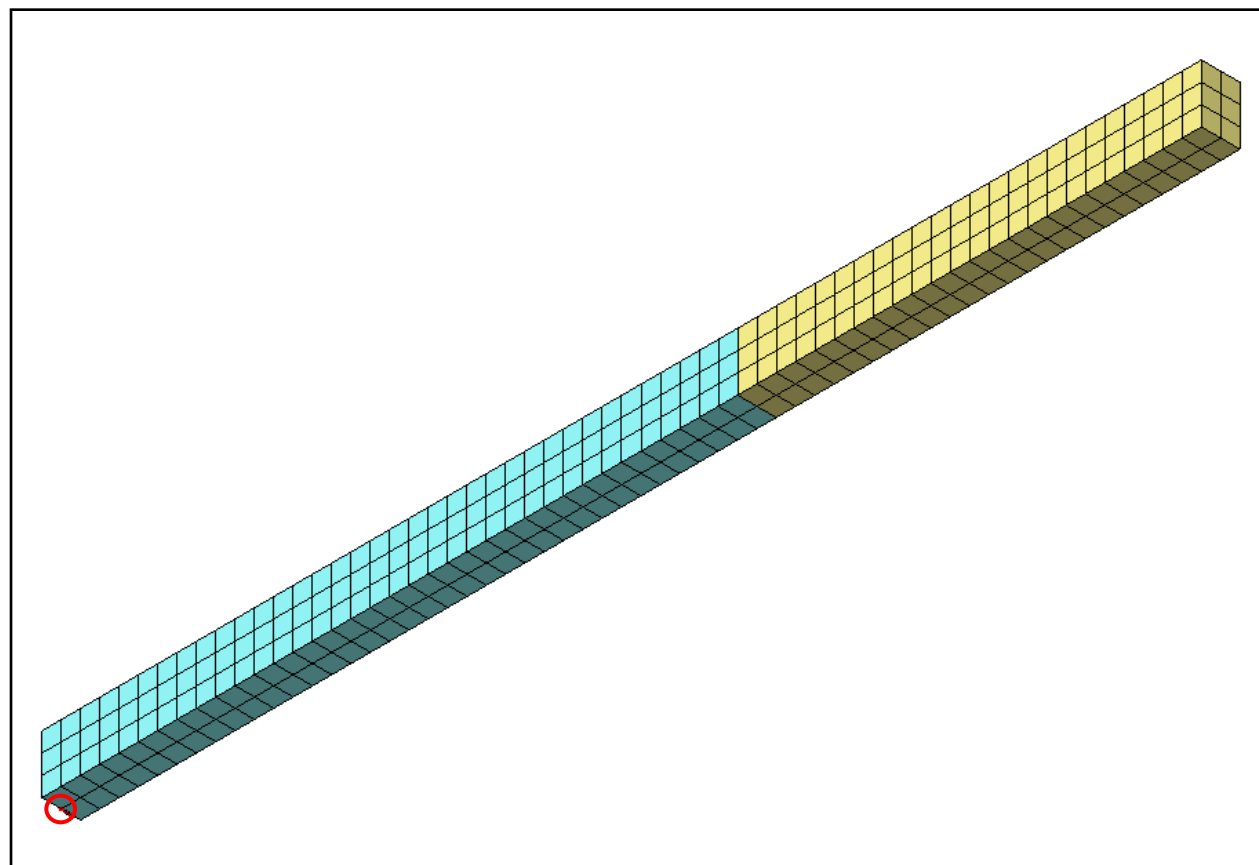
1. Analysis > BC > Constraint ...
2. BC Set : BC 1
3. Select 4 Nodes Marked by "O" (See Figure)
4. Check on "T3"
5. Click [Apply] Button



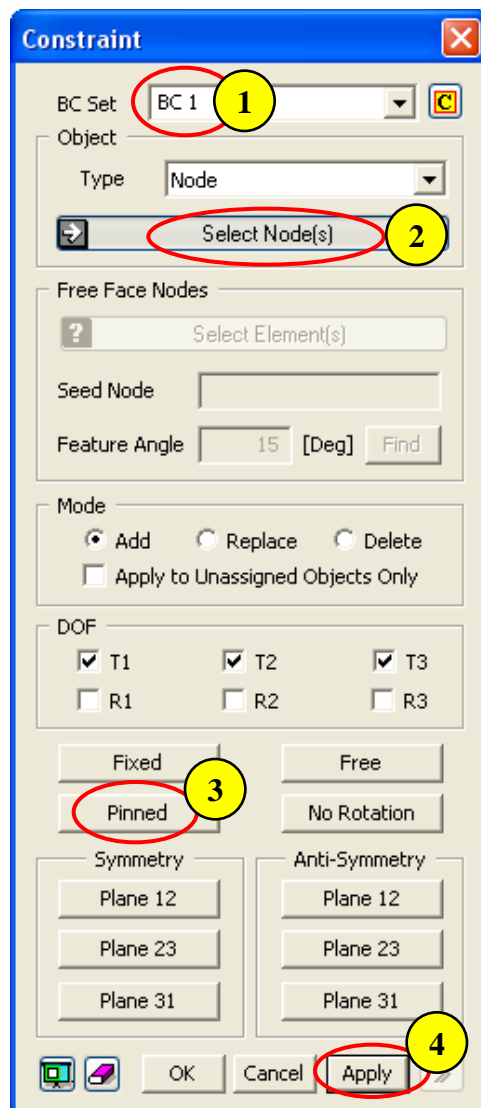
Step 17.



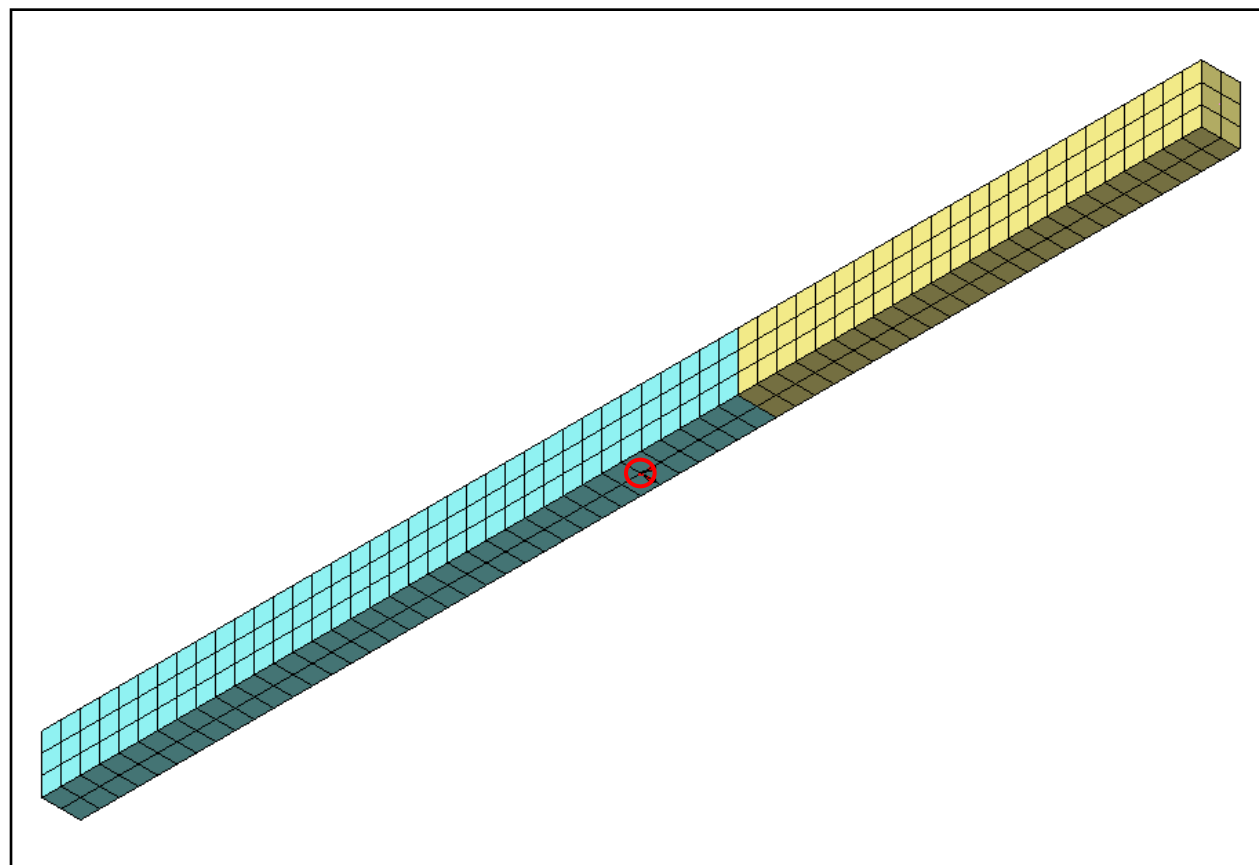
1. BC Set : BC 1
2. Select 1 Node Marked by “O” (See Figure)
3. Check on “T2” & “T3”
4. Click [Apply] Button



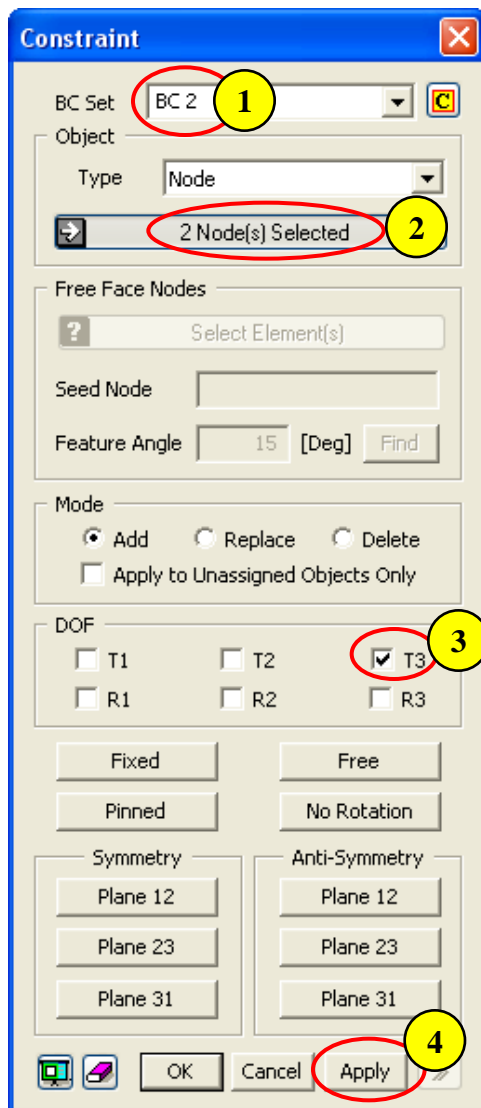
Step 18.



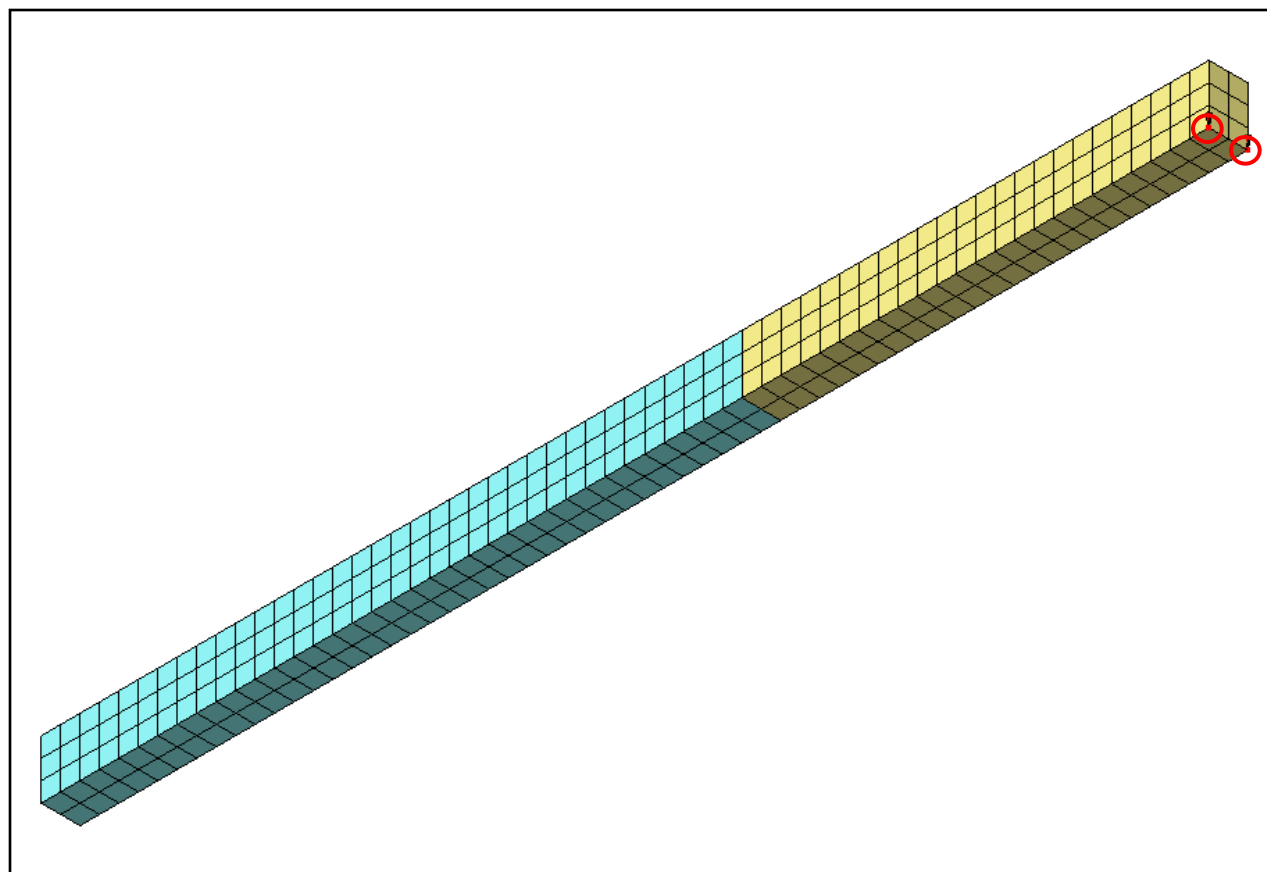
1. BC Set : BC 1
2. Select 1 Node Marked by "O" (See Figure)
3. Click [Pinned] Button
4. Click [Apply] Button



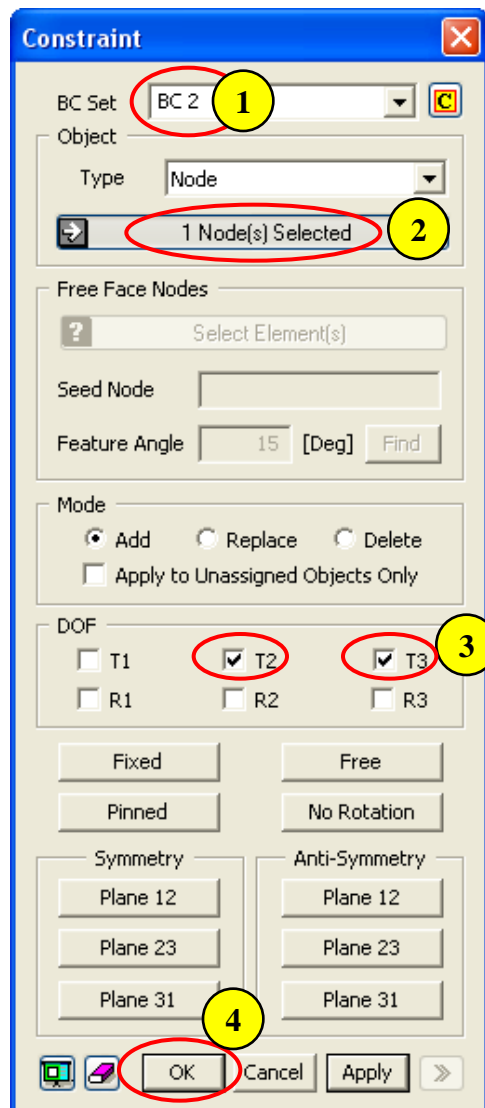
Step 19.



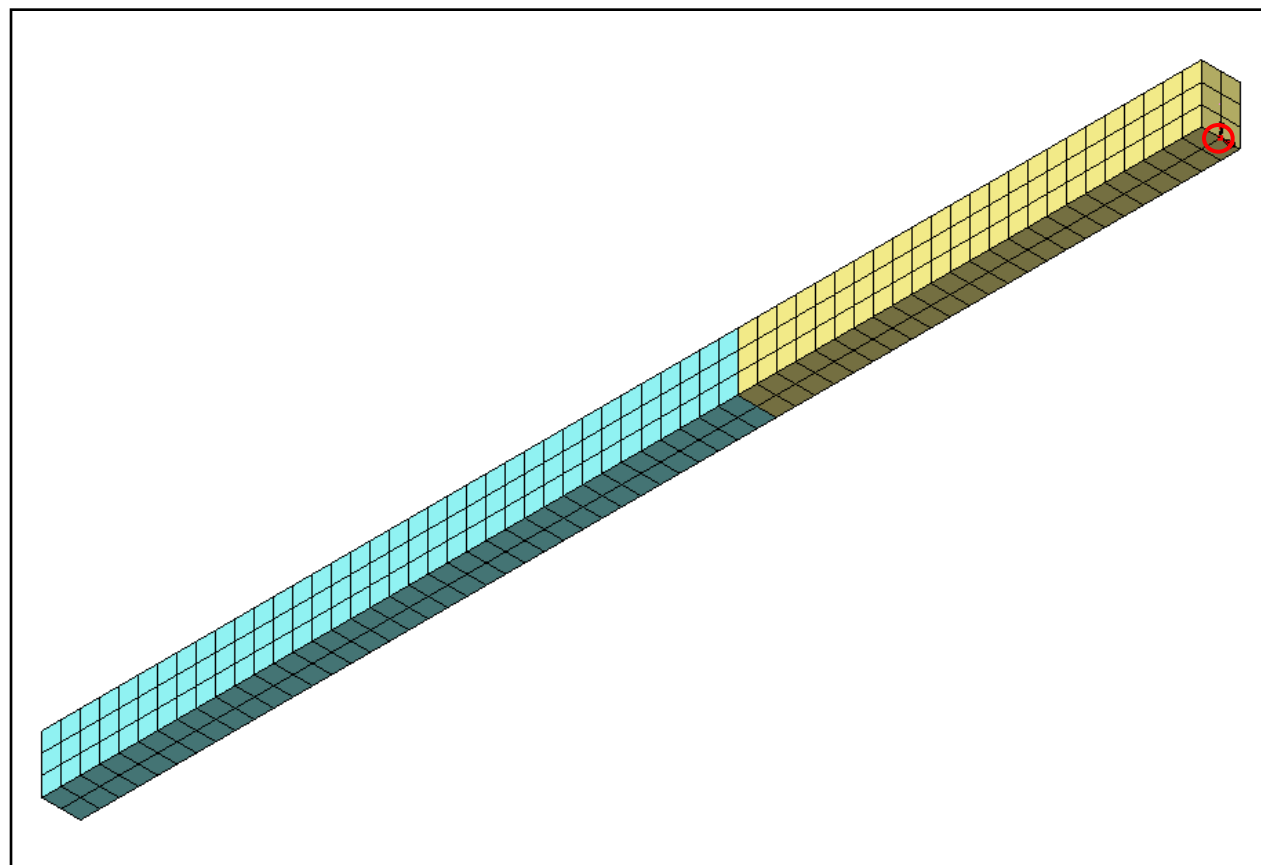
1. BC Set : BC 2
2. Select 2 Nodes Marked by “O” (See Figure)
3. Check on “T3”
4. Click [Apply] Button



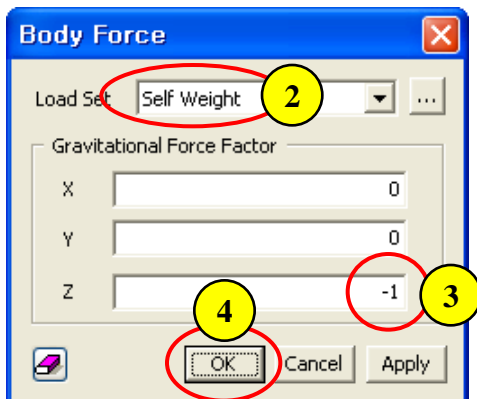
Step 20.



1. BC Set : BC 2
2. Select 1 Node Marked by "O" (See Figure)
3. Check on "T2" & "T3"
4. Click [OK] Button

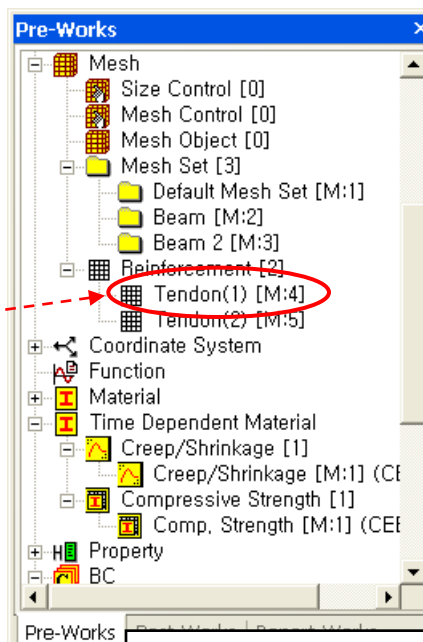
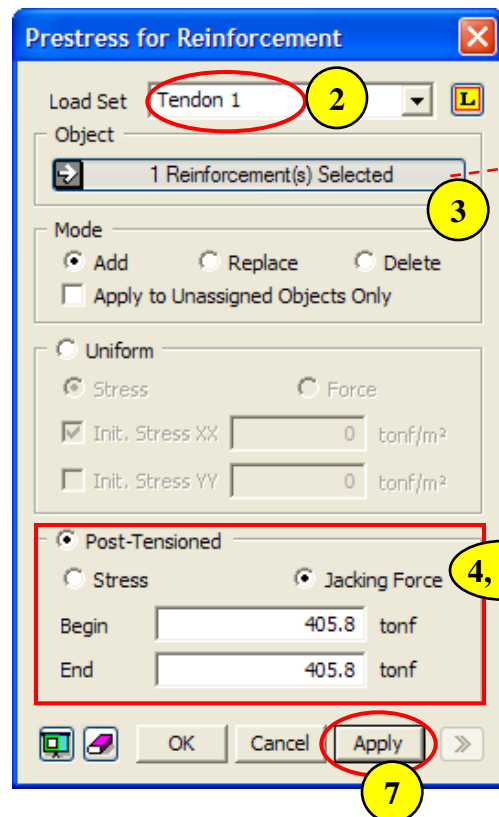


Step 21.

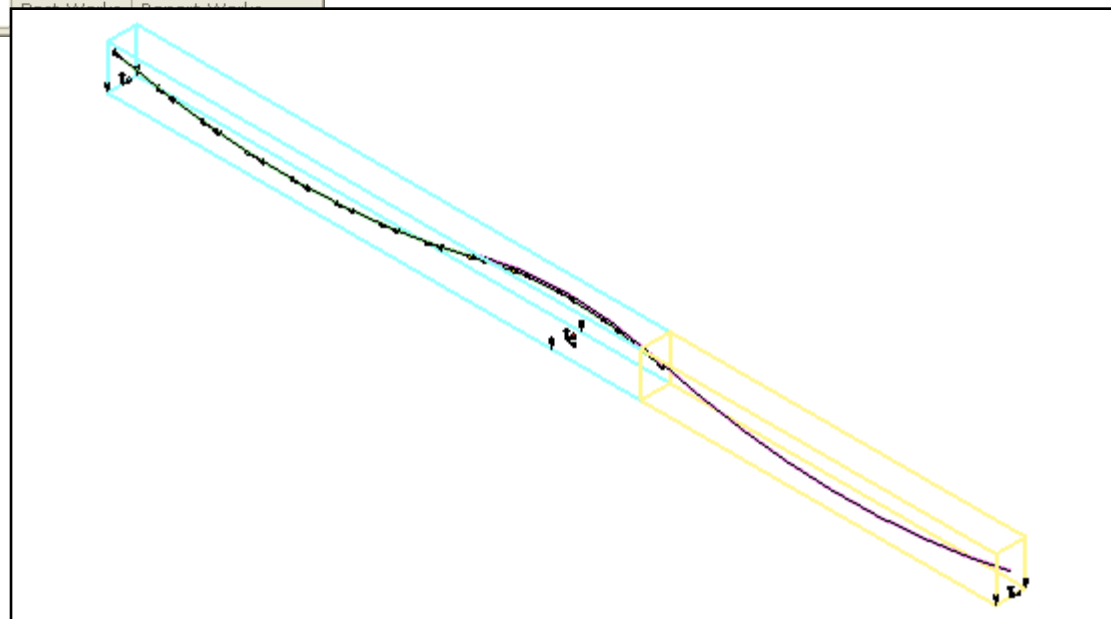


1. Analysis > Load > Body Force ...
2. Load Set : Self Weight
3. Gravitational Force Factor : Z (-1)
4. Click [OK] Button

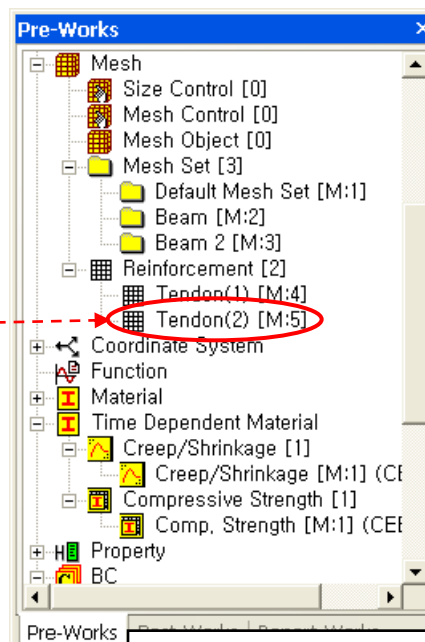
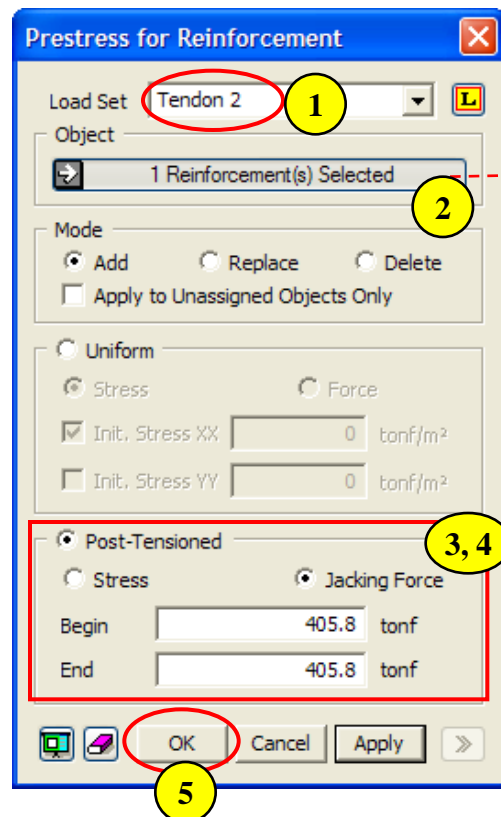
Step 22.



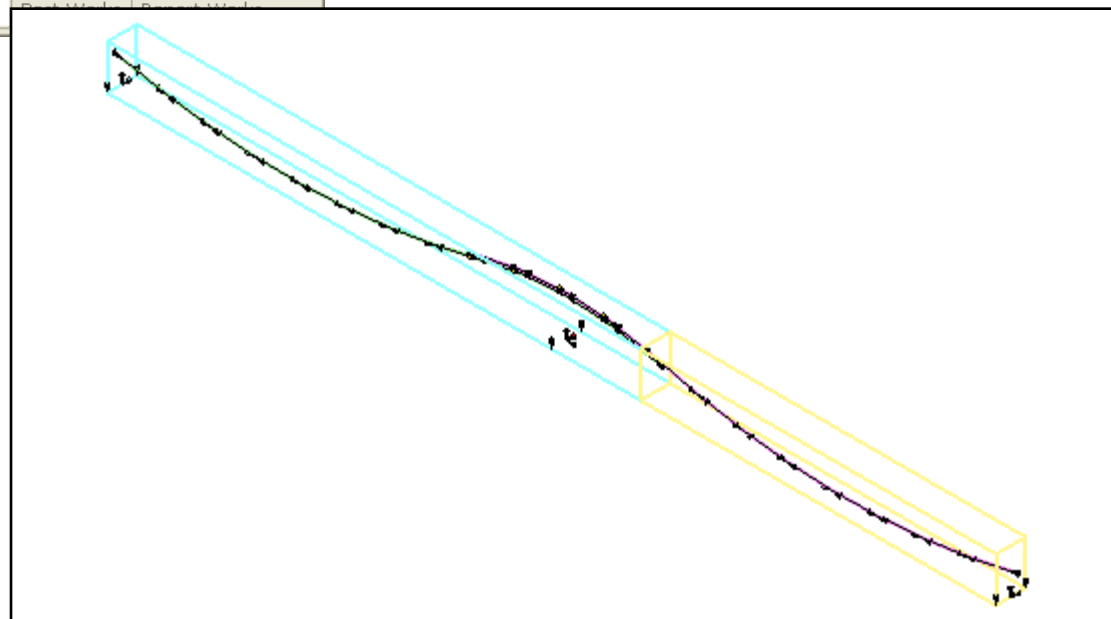
1. Analysis > Load > Prestress for Reinforcement ...
2. Load Set : Tendon 1
3. Select "Reinforcement > Tendon(1)" Mesh Set in Pre-Works Tree (See Figure)
4. Check on "Post Tensioned"
5. Select "Jacking Force"
6. Begin & End : 405.8 tonf
7. Click [Apply] Button



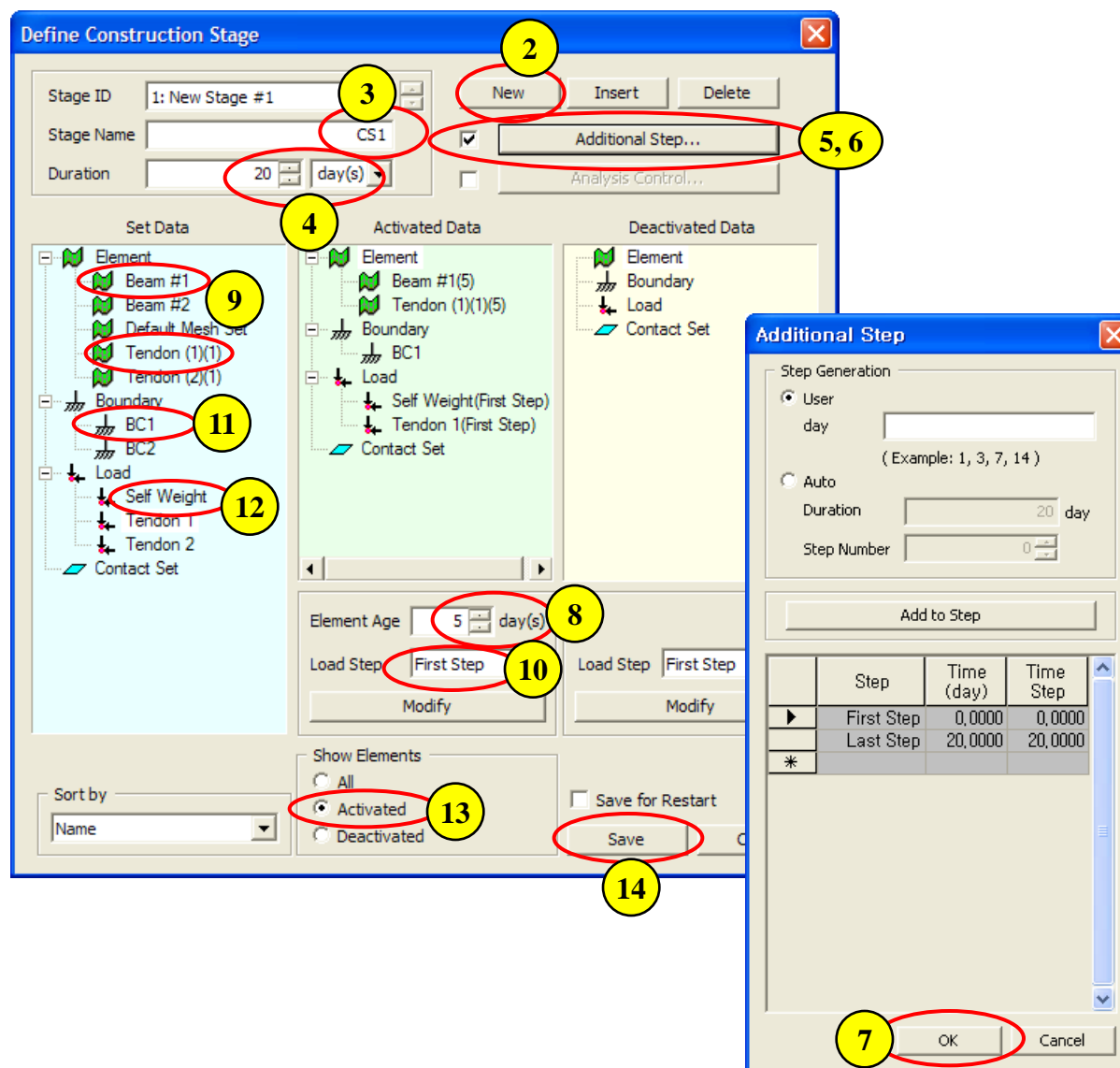
Step 23.



1. Load Set : Tendon 2
2. Select "Reinforcement > Tendon(2)" Mesh Set in Pre-Works Tree (See Figure)
3. Check on "Post Tensioned"
4. Select "Stress"
5. Begin & End : 405.8 tonf
6. Click [OK] Button

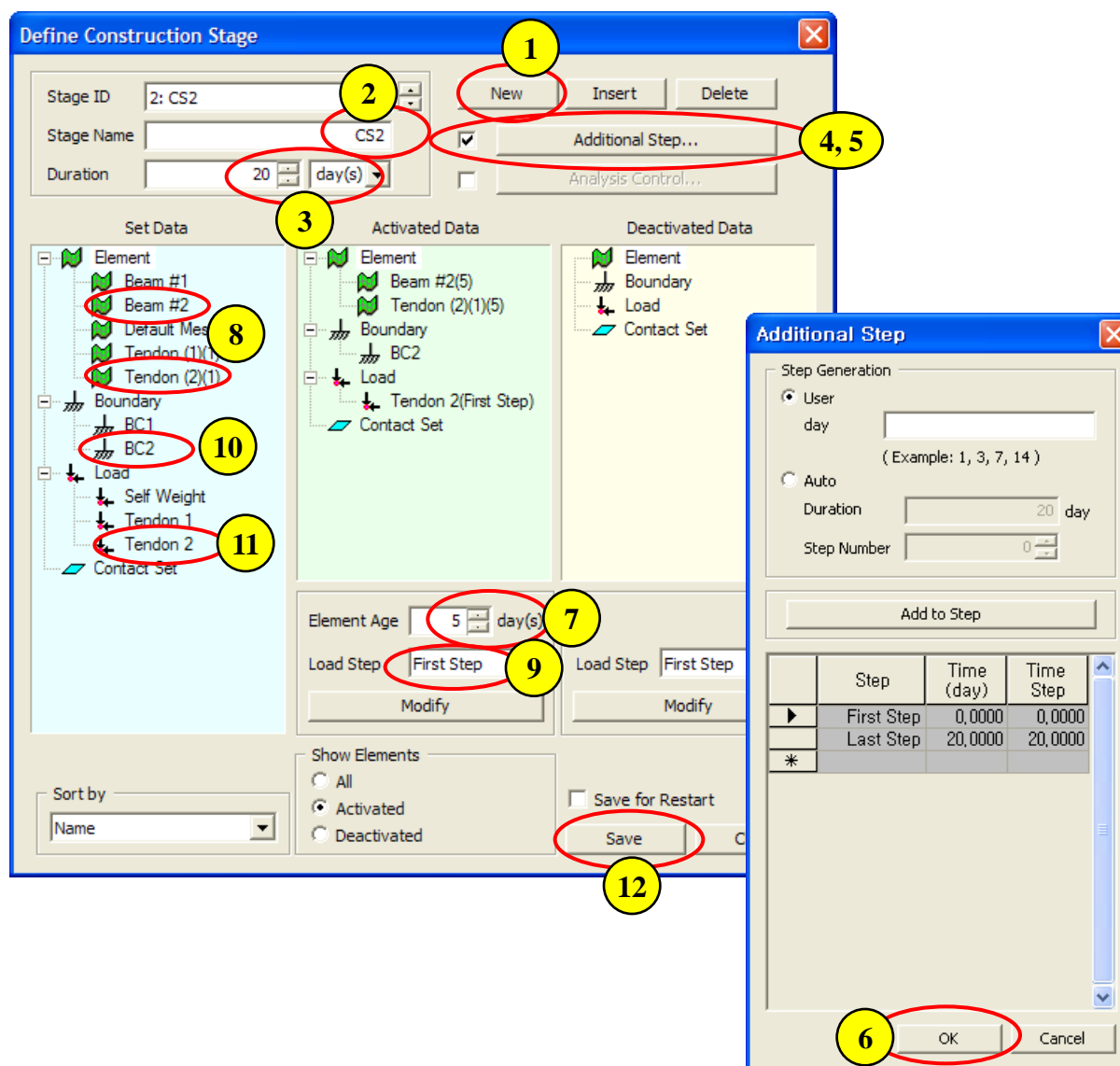


Step 24.



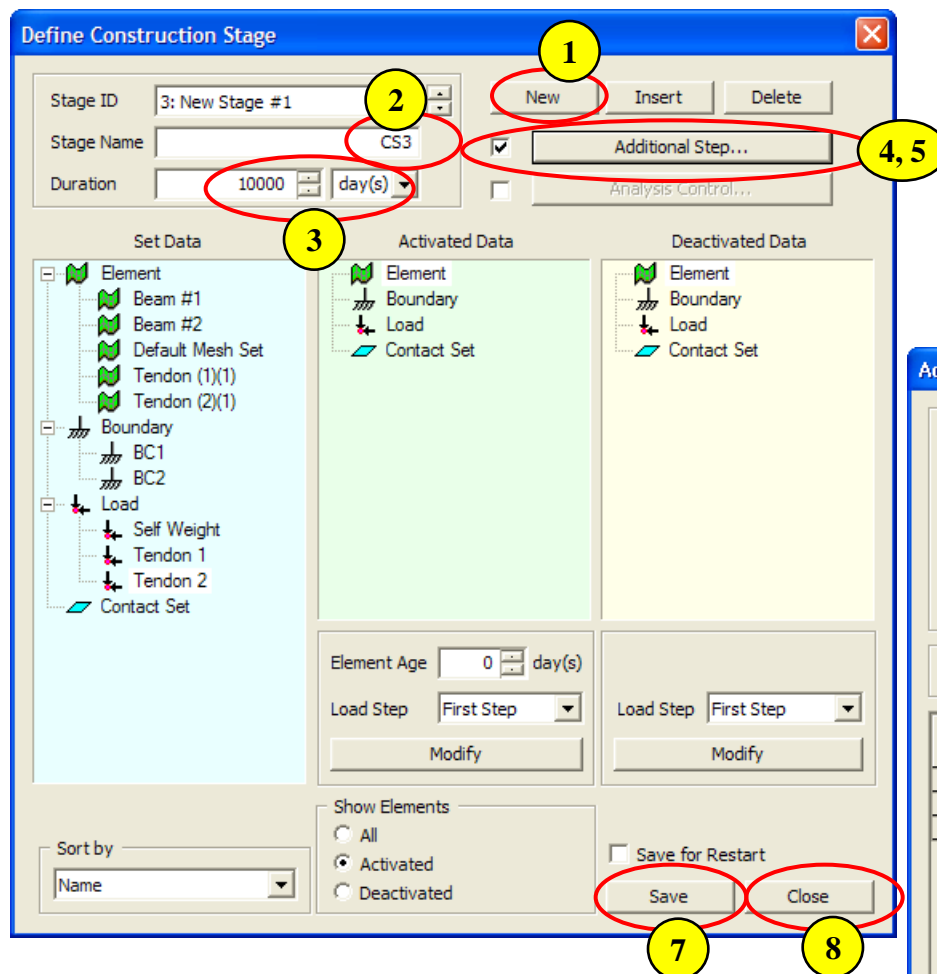
1. Analysis > Construction Stage > Define Construction Stage...
2. Click on [New] Button
3. Stage Name : CS 1
4. Duration : 20 days
5. Check on "Additional Step"
6. Click on [Additional Step] Button
7. Click on [OK] Button
8. Element Age : 5 days
9. Drag & Drop "Beam#1" & "Tendon(1)" to "Activated Data" Window
10. Load Step : First Step
11. Drag & Drop "BC 1" to "Activated Data" Window
12. Drag & Drop "Self Weight" & "Tendon 1" to "Activated Data" Window
13. Check on "Activated"
14. Click on [Save] Button

Step 25.

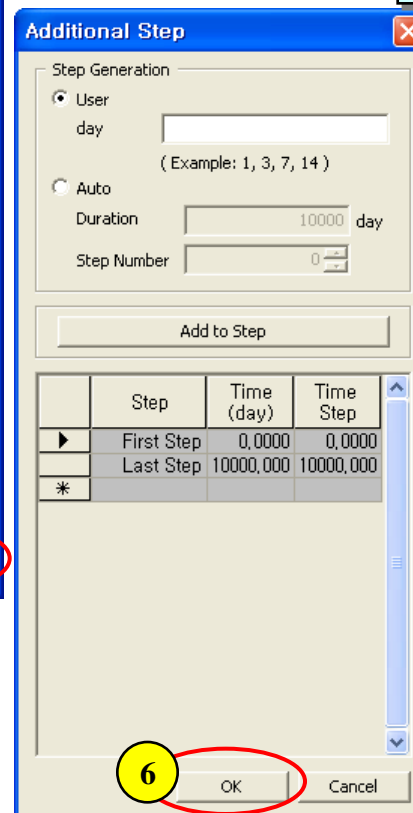


1. Click [New] Button
2. Stage Name : CS 2
3. Duration : 20 days
4. Check on "Additional Step"
5. Click [Additional Step...] Button
6. Click [OK] Button
7. Element Age : 5 days
8. Drag & Drop "Beam# 2" & "Tendon(2)" to "Activated Data" Window
9. Load Step : First Step
10. Drag & Drop "BC 2" to "Activated Data" Window
11. Drag & Drop "Tendon 2" to "Activated Data" Window
12. Click [Save] Button

Step 26.



1. Click [New] Button
2. Stage Name : CS 3
3. Duration : 10000 days
4. Check on "Additional Step"
5. Click [Additional Step...] Button
6. Click [OK] Button
7. Click [Save] Button
8. Click [Close] Button



Step 27.

Add/Modify Analysis Case

Name: (3)

Description:

Analysis Type: (4)

Output Control Analysis Control

Analysis Model

Analysis Control

Analysis: Construction Stage

Final Calculation Stage

☒ End Stage ☐ Middle Stage

☐ Specify Restart Stage

☐ Apply Heat Transfer Result as Thermal Load

Restart Option

☒ Save only user specified stages

☐ If not converged, save its previous stage

☐ Save all stages

Time-Dependent Behavior

Type

☒ Creep ☒ Shrinkage

☒ Variation of Compressive Strength (6, 7)

Load Group

☒ Consider Creep/Shrinkage as Load Group

ID	Name	Descrip...

Add...
Modify...
Delete

☐ Select Load Set Including Initial Force

☐ Add Initial Force to Element Force

OK (8) Cancel Apply

Analysis Case

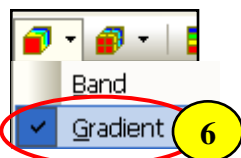
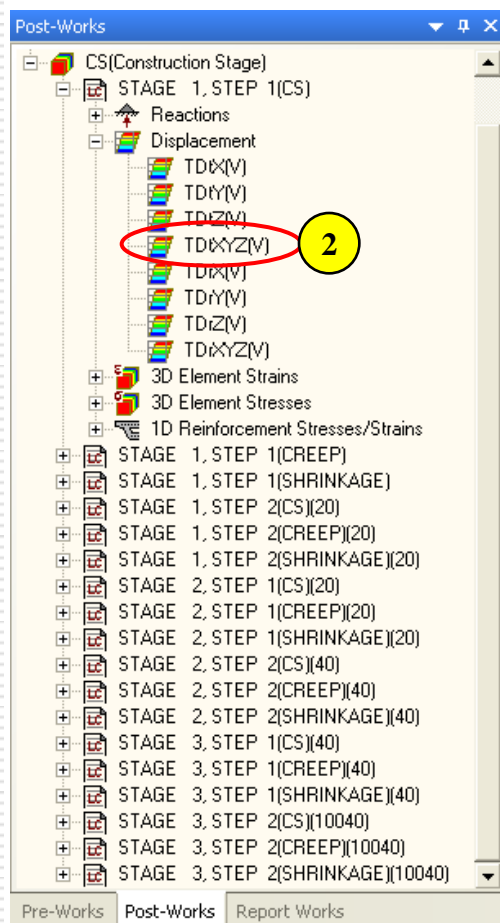
No.	Name	Type	Description
*			

Add... (2)
Modify...
Copy
Delete

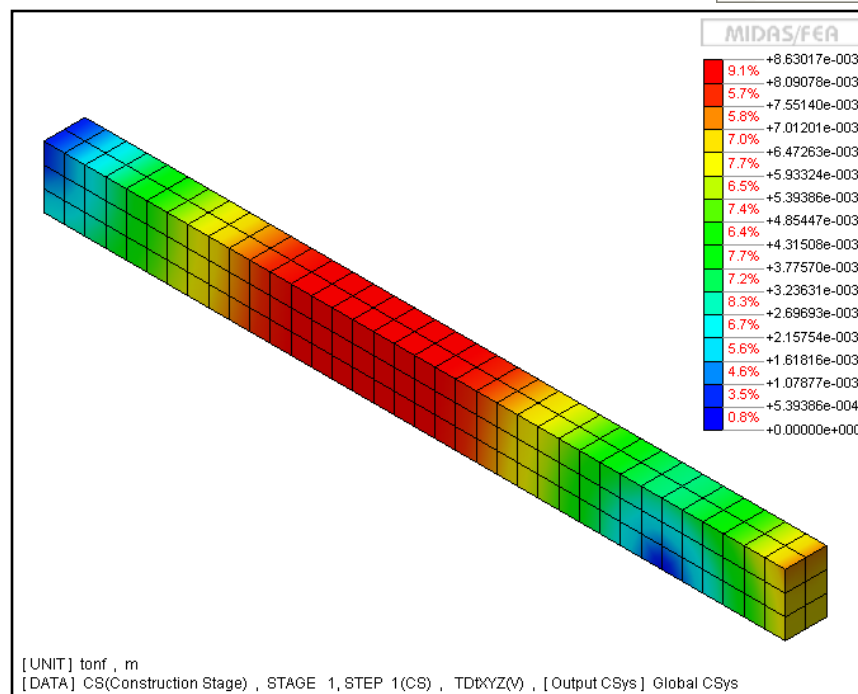
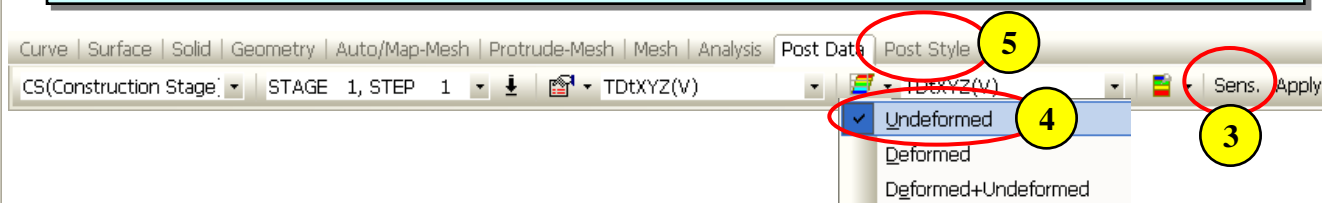
Close (9)

1. Analysis > Analysis Case ...
2. Click [Add] Button
3. Name : CS
4. Analysis Type : Construction Stage
5. Click on Button of Analysis Control
6. Check on "Creep" & "Shrinkage"
7. Check on "Variation of Compressive Strength"
8. Click [OK] Button
9. Click [Close] Button
10. Analysis > Solve...
11. Click [OK] Button

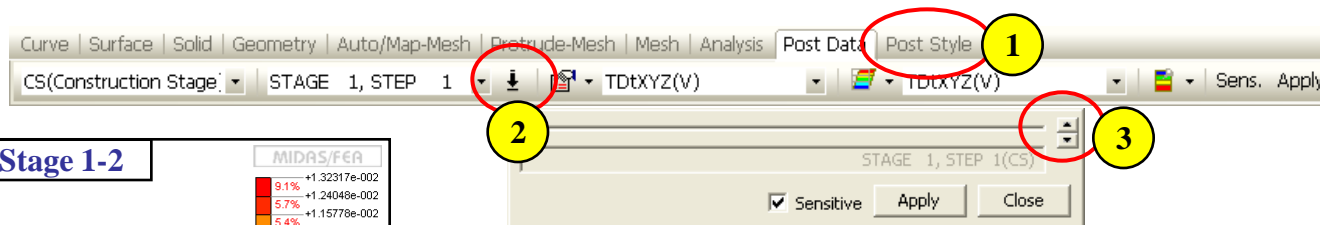
Step 28.



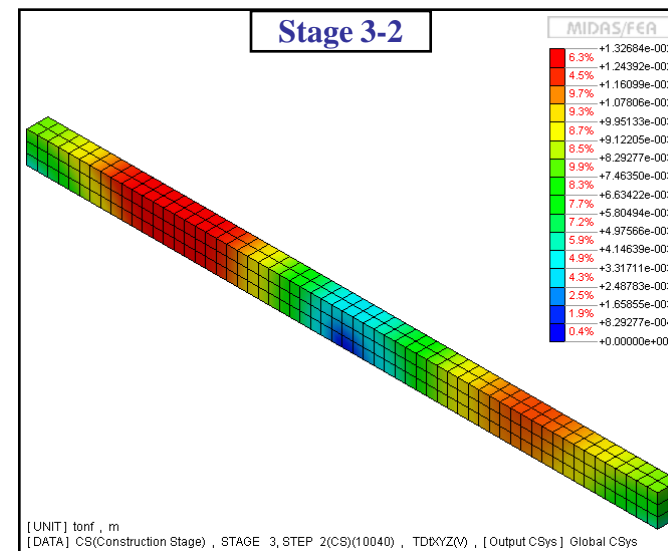
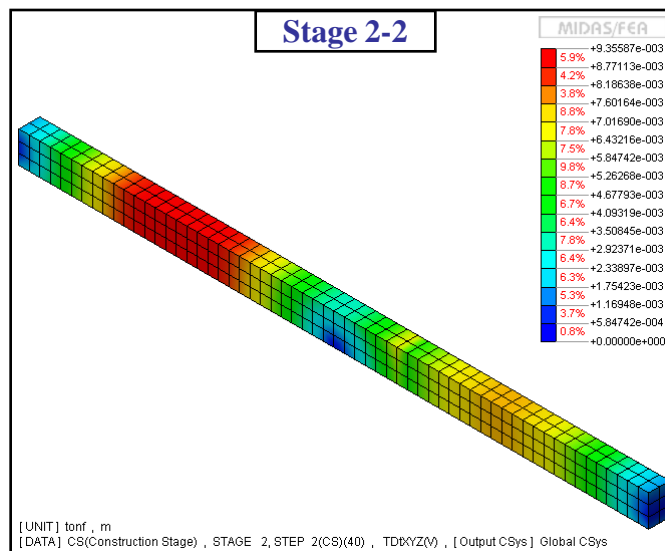
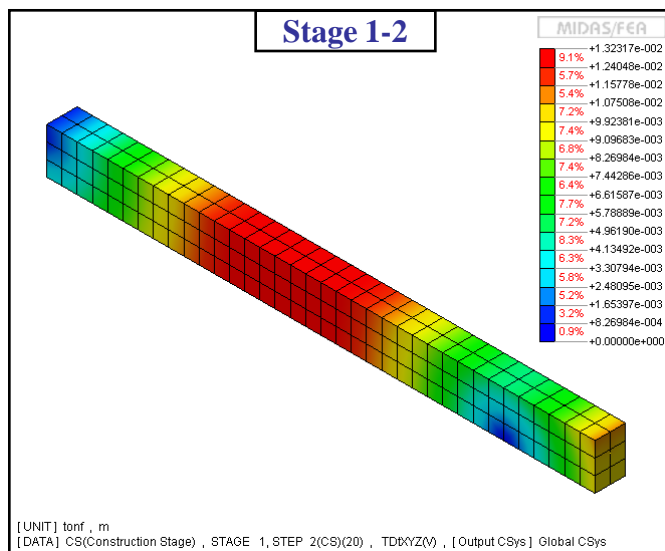
1. Post-Works Tree : CS (Structural Nonlinear) > Stage 1, STEP 1 > Displacement...
2. Double Click "TDtXYZ(V)"
3. Click [Sens.] Button
4. Select "Undeformed" for Mesh Shape (See Figure)
5. Click "Post Style" Toolbar
6. Select "Gradient" for Contour Type



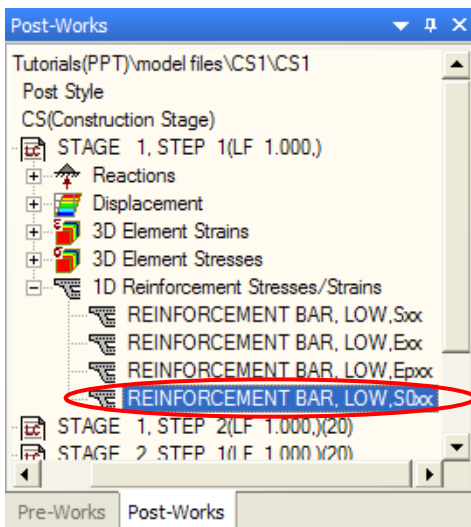
Step 29.



1. Click "Post Data" Toolbar
2. Click "Output Set Slider" Button
3. Click [▲] or [▼] Button to Change Stage



Step 30.



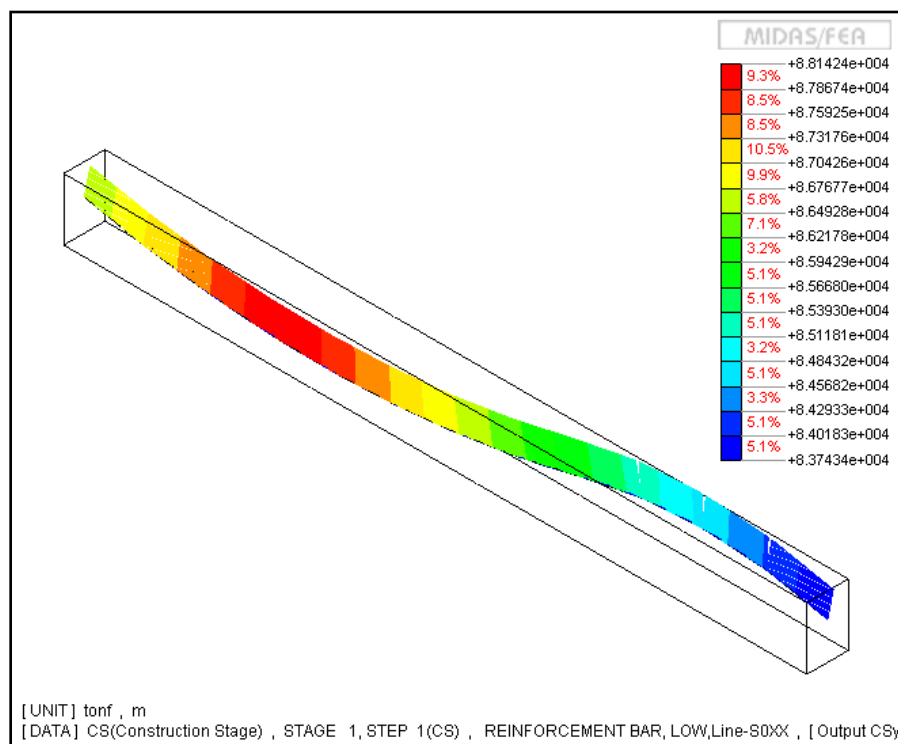
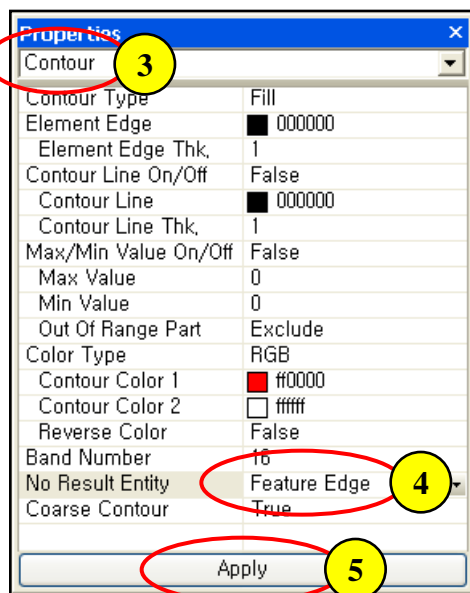
1. Post-Works Tree : CS (Structural Nonlinear) > Stage 1, STEP 1 > 1D Reinforcement Stresses/Strains...

2. Double Click "REINFORCEMENT BAR, LOW, S0XX"

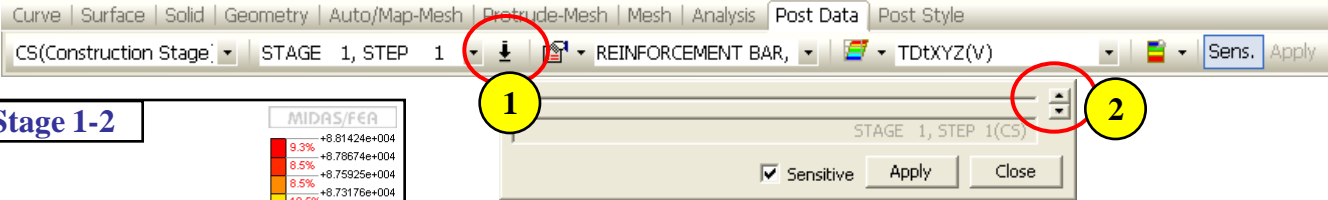
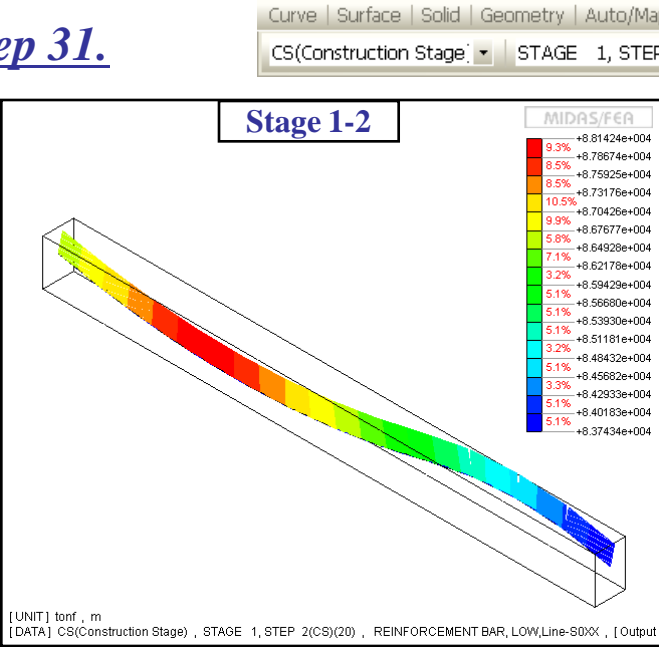
3. Property Window : Contour

4. No Result Entity : Feature Edge

5. Click [Apply] Button



Step 31.



1. Click “Output Set Slider” Button
2. Click [▲] or [▼] Button to Change Stage

