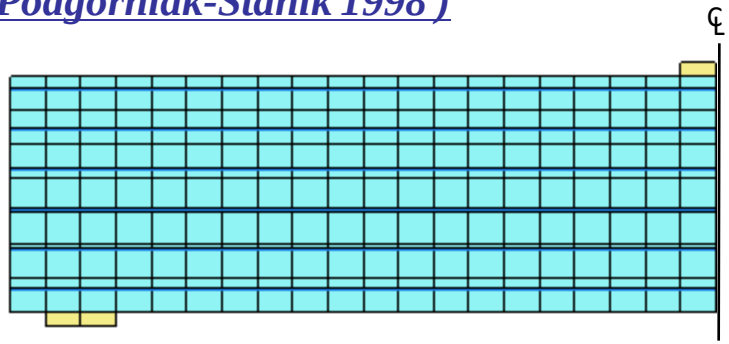
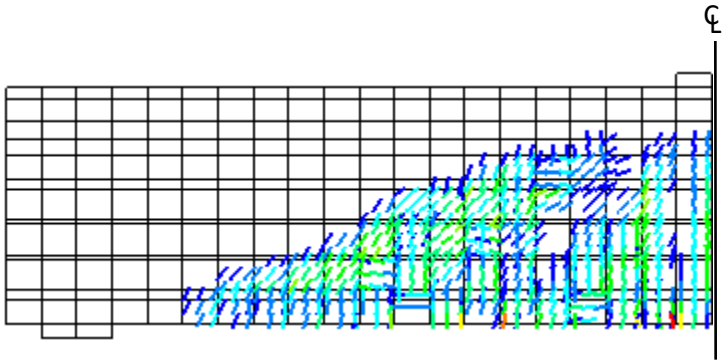
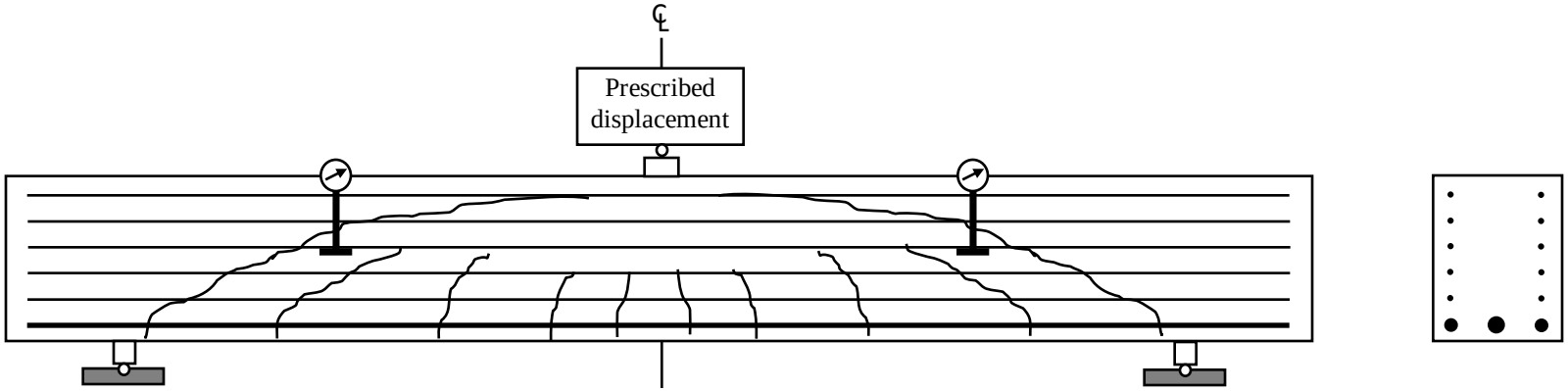
NL-4. Shear Crack Failure in an RC beam

(Podgorniak-Stanik 1998)





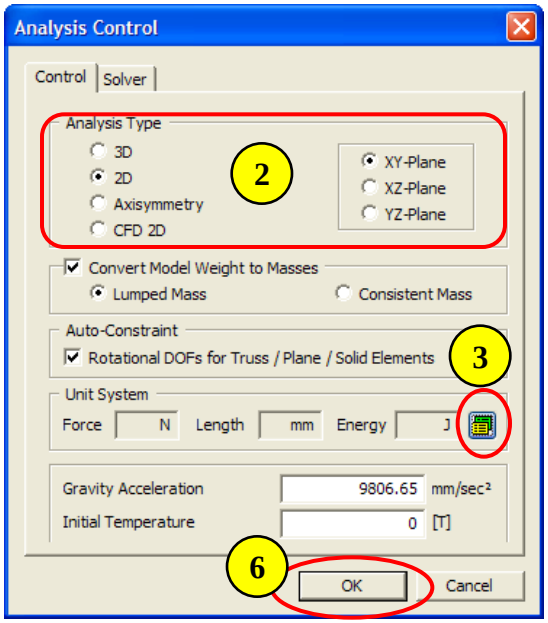


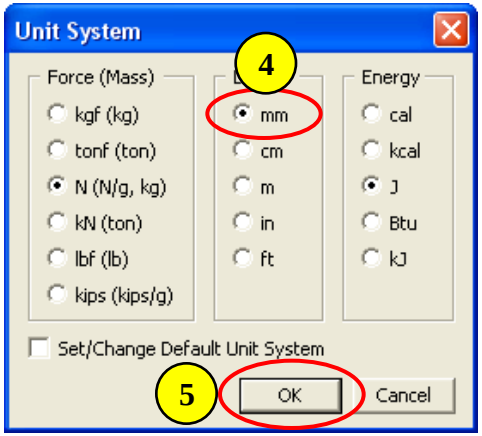
Overview

- **2D Nonlinear Analysis**
- **Model**
  - *A symmetric RC beam with reinforcement element*
  - *Cracking material model*
  - *Unit : N, mm*
- **Load & Boundary Condition**
  - *Prescribed displacement*
  - *Simply constraint*

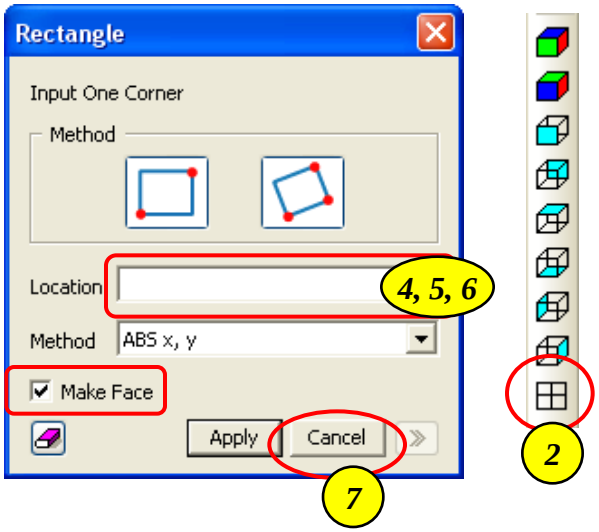
Step 1.

1. *Analysis > Analysis Control – Control tab*
2. *Analysis Type : 2D and XY - Plane*
3. *Click*  *Button (Unit System)*
4. *Length : mm*
5. *Click on* **[OK]** *Button*
6. *Click on* **[OK]** *Button*





Step 2.

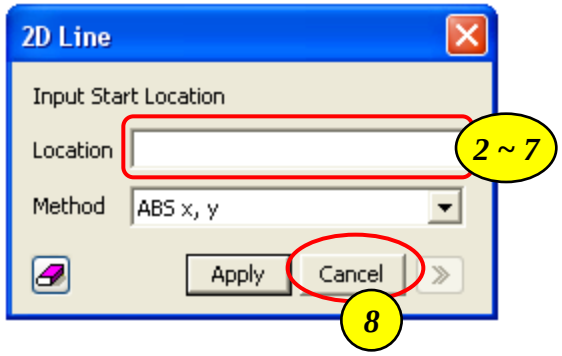


1. *Toggle off "Toggle Grid"*
2. *Click "Normal View"*
3. *Geometry > Curve > Create on WP > Rectangle (Wire)...*
4. *Location : (0), <1500, 500>*
5. *Location : (75,0), <150, -30>*
6. *Location : (1500,500), <-75, 30>*
7. *Click on [Cancel] Button*

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

[Esc] as shortcut for [Cancel]

Step 3.



1. Geometry > Curve > Create on WP >Line ...
2. Location : (0, 50), <1500, 0>
3. Location : (0, 135), <1500, 0>
4. Location : (0, 220), <1500, 0>
5. Location : (0, 305), <1500, 0>
6. Location : (0, 390), <1500, 0>
7. Location : (0, 475), <1500, 0>
8. Click on [Cancel] Button

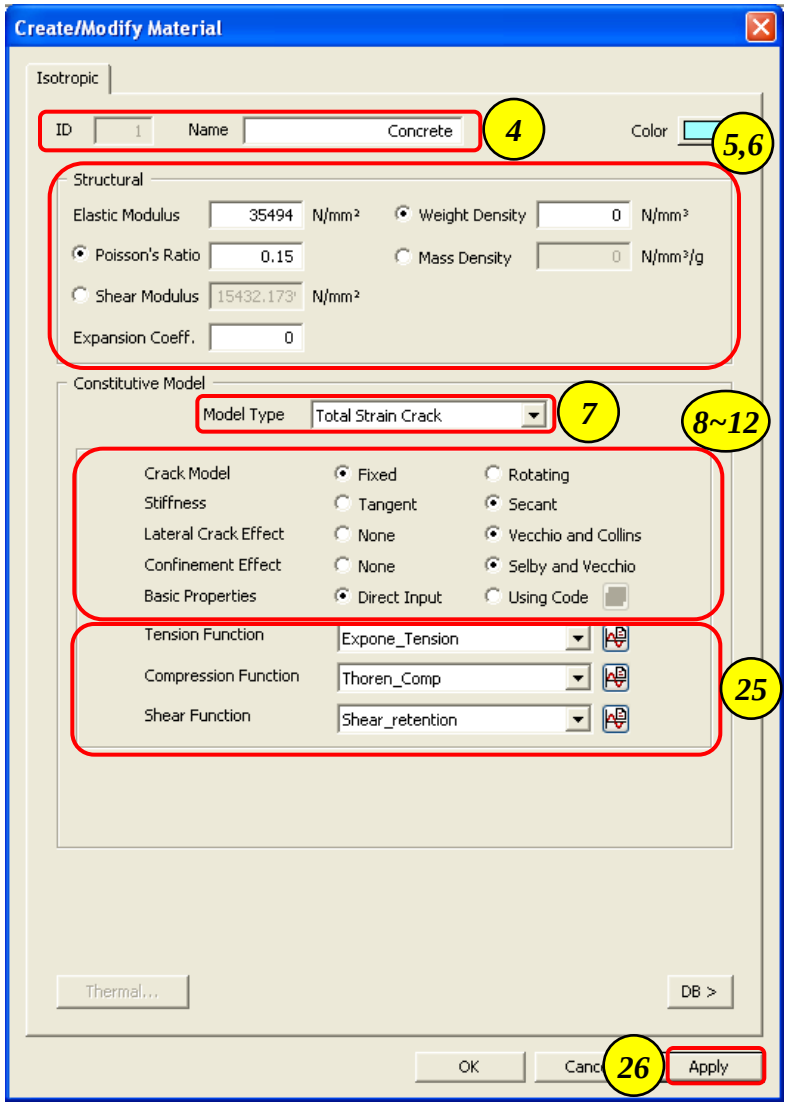
() : "ABS x, y" , <> : "REL dx, dy"
(0) same as (0, 0)

[Esc] as shortcut for [Cancel]

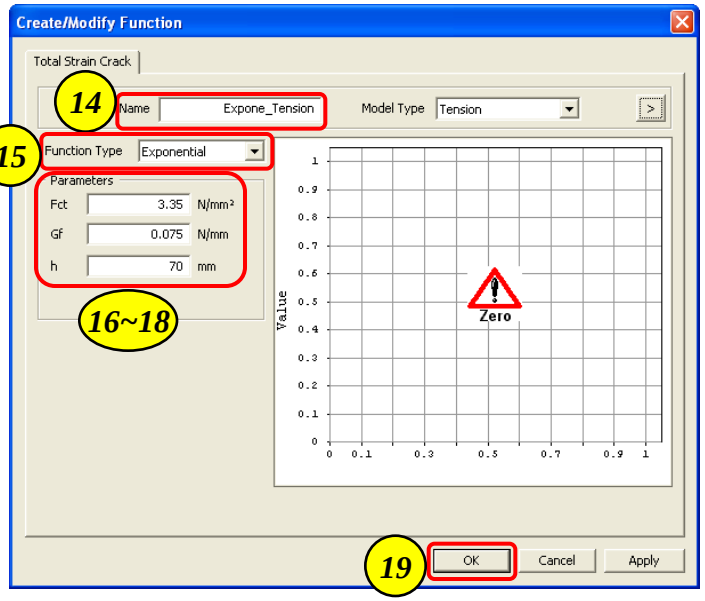
Step 4.

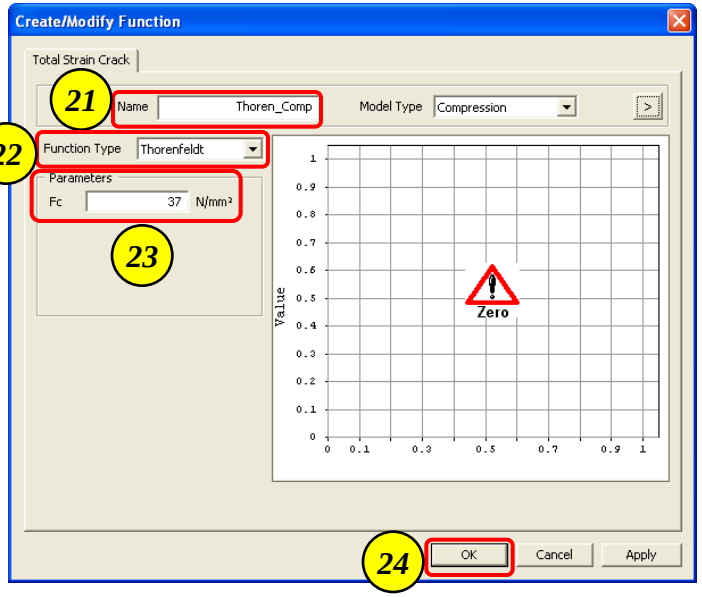
1. Analysis > Material …
2. Click [Create] Button
3. Select “Isotropic” tab
4. ID : 1 , Name : Concrete
5. Elastic Modulus : 35494 N/mm^2
6. Poisson’s Ratio : 0.15
7. Model Type : Total Strain Crack
8. Crack model: Fixed
9. Stiffness: Secant
10. Lateral Crack Effect: Vecchio and Collins
11. Confinement Effect: None
12. Basic Properties: Direct Input
13. Click on  to define Tension Function
14. Name: Expone_tension
15. Function Type: Exponential
16. Fct = 3.35 N/mm^2
17. Gf = 0.075 N/mm
18. h = 70 mm
19. Click on [OK] Button
20. Click on  to define Compression Function
21. Name: Thoren_comp
22. Function Type: thorenfeldt
23. Fc = 37 N/mm^2
24. Click on [OK] Button
25. Click on  to define Shear Function
26. Name: Shear_retention
27. Function Type: Constant
28. Beta = 0.05
29. Click on [OK] Button
30. Select the tension, compression, shear functions from the drop lists
31. Click on [Apply] Button
32. ID : 2 , Name : Steel
33. Elastic Modulus : 200000 N/mm^2
34. Poisson’s Ratio : 0.15
35. Model Type : Von Mises
36. Yield stress : 480 N/mm2
37. Click on [Apply] Button
38. ID : 3 , Name : Wood
39. Elastic Modulus : 2000 N/mm^2
40. Poisson’s Ratio : 0.15
41. Model Type : Elastic
42. Click on [OK] Button
43. Click on [Close] Button

Step 5.

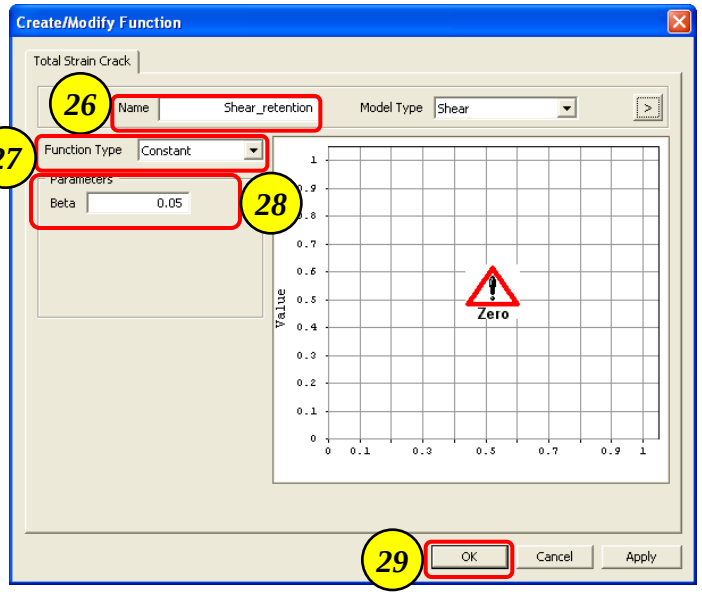


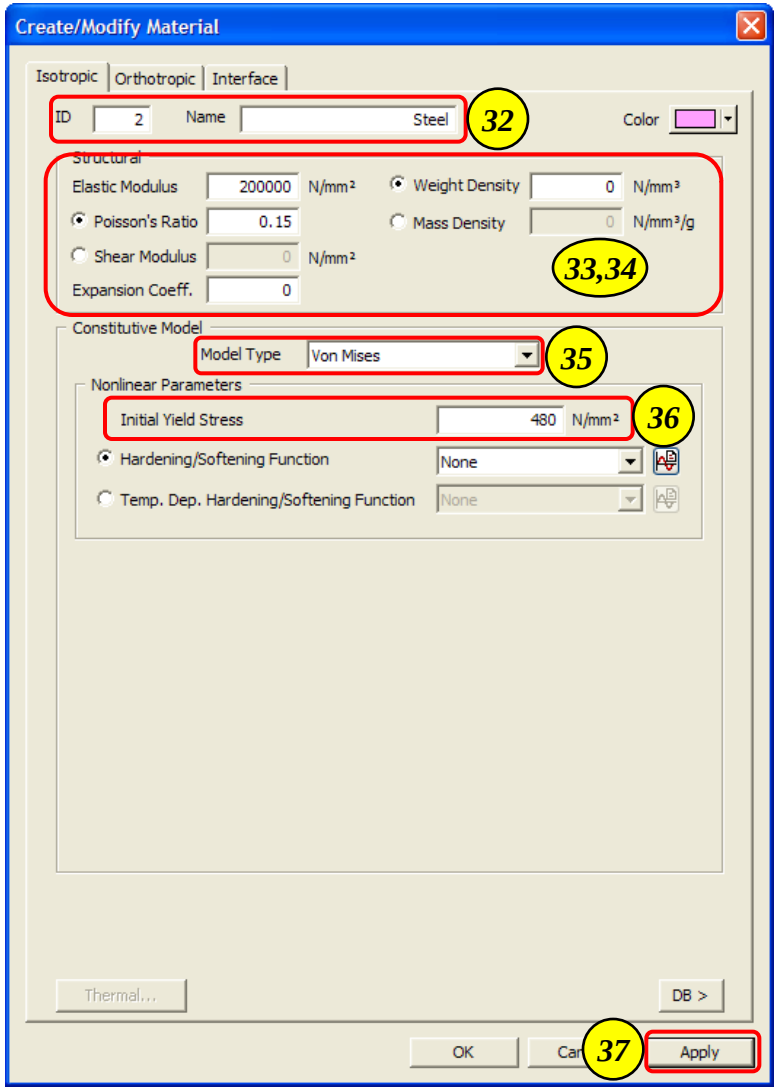
Concrete mechanical properties obtained from CEB-FIP MC 90



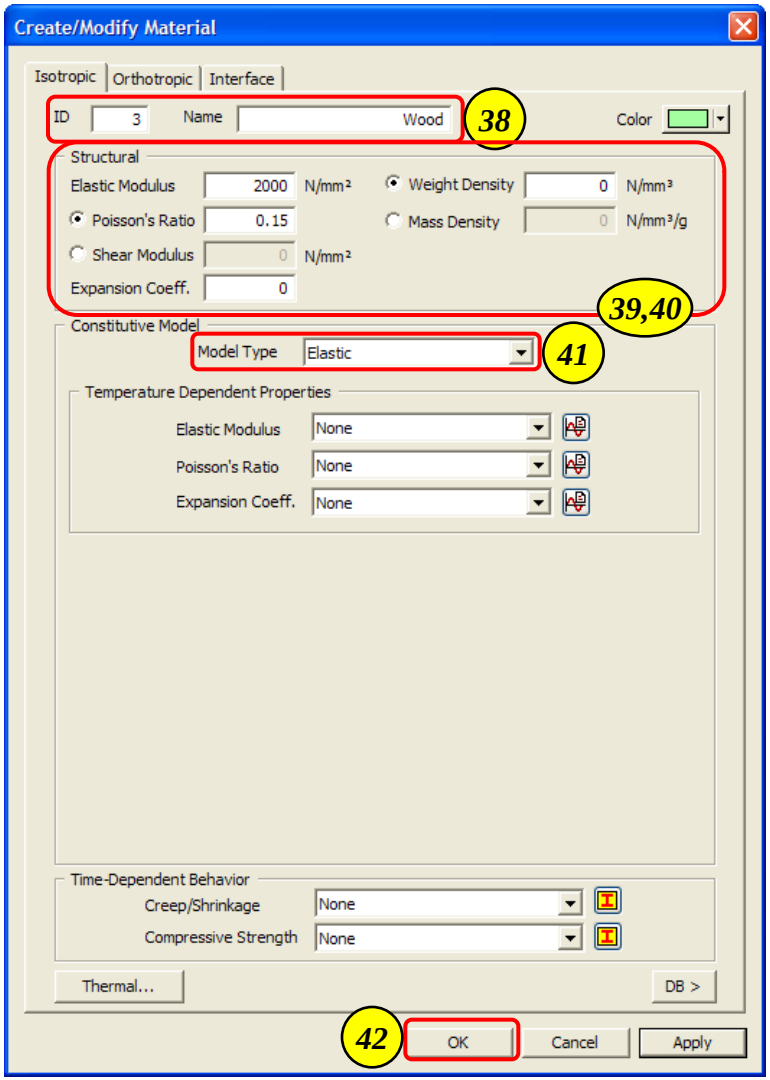


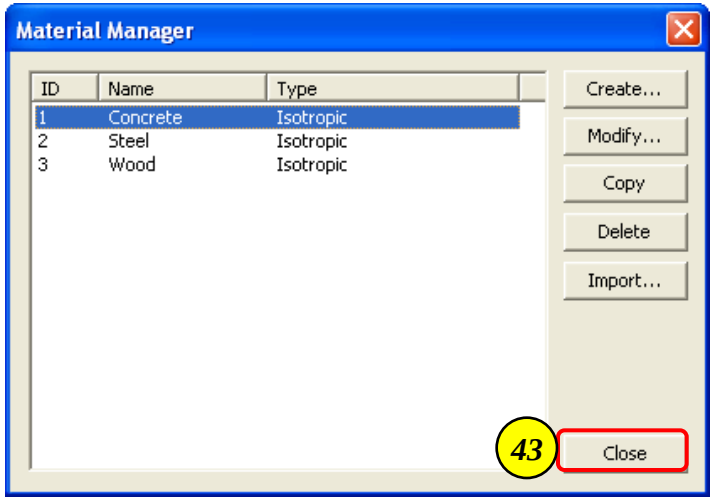
Step 6.


Create/Modify Function
Total Strain Crack
26
Name
Shear_retention
Model Type
Shear
27
Function Type
Constant
Parameters
Beta
0.05
28
Value
Zero
29
OK
Cancel
Apply


Create/Modify Material
Isotropic
Orthotropic
Interface
ID
2
Name
Steel
32
Color
Structural
Elastic Modulus
200000
N/mm²
Weight Density
0
N/mm³
Poisson's Ratio
0.15
Mass Density
N/mm³/g
Shear Modulus
N/mm²
Expansion Coeff.
0
33,34
Constitutive Model
Model Type
Von Mises
35
Nonlinear Parameters
Initial Yield Stress
480
N/mm²
36
Hardening/Softening Function
None
Temp. Dep. Hardening/Softening Function
Thermal...
DB >
OK
37
Apply

Step 7.

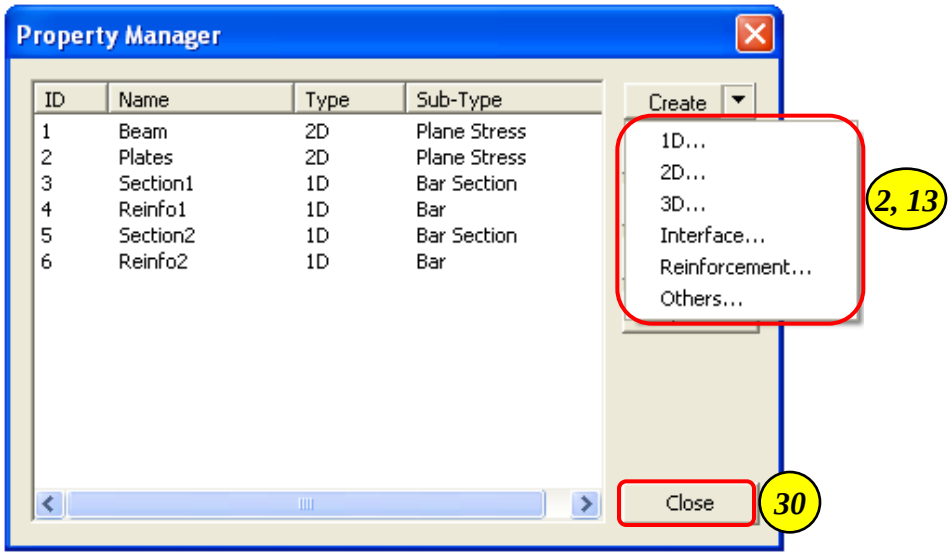

Create/Modify Material
Isotropic
Orthotropic
Interface
ID
3
Name
Wood
38
Color
Structural
Elastic Modulus
2000
N/mm²
Weight Density
0
N/mm³
Poisson's Ratio
0.15
Mass Density
0
N/mm³/g
Shear Modulus
0
N/mm²
Expansion Coeff.
0
39,40
Constitutive Model
Model Type
Elastic
41
Temperature Dependent Properties
Elastic Modulus
None
Poisson's Ratio
None
Expansion Coeff.
None
Time-Dependent Behavior
Creep/Shrinkage
None
Compressive Strength
None
Thermal...
DB >
42
OK
Cancel
Apply

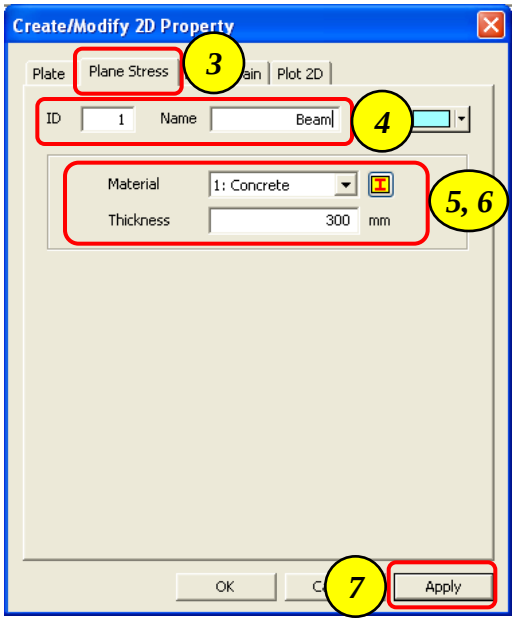

Material Manager
ID
Name
Type
1
Concrete
Isotropic
2
Steel
Isotropic
3
Wood
Isotropic
Create...
Modify...
Copy
Delete
Import...
43
Close

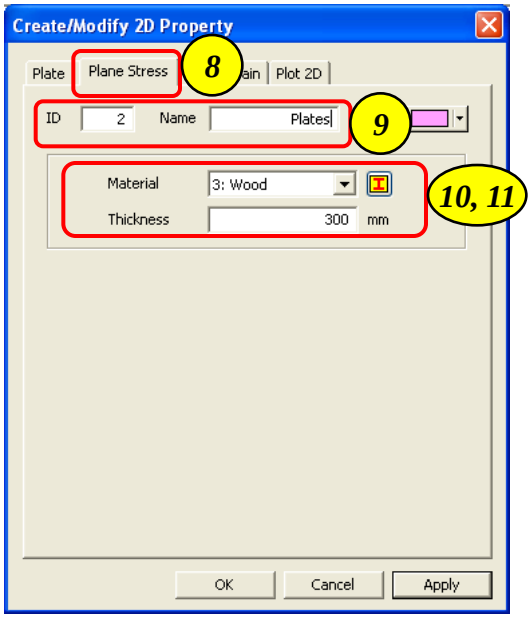
Step 8.

1. Analysis > Property...
2. Create 2D…
3. Select “Plane Stress” tab
4. ID : 1 , Name : Beam
5. Thickness: 300 mm
6. Material : (1: Concrete)
7. Click on [Apply] Button
8. Select “Plane Stress” tab
9. ID : 2 , Name : Plates
10. Thickness: 300 mm
11. Material : (3: Wood)
12. Click on [OK] Button
13. Create Reinforcement…
14. Select “Bar Section” tab
15. ID : 3 , Name : Section1
16. Click on [Apply] Button
17. Select “Bar” tab
18. ID : 4 , Name : Reinfo1
19. Material : (2: Steel)
20. Cross Sectional Area: 1100 mm^2
21. Select “reinforcement”
22. Click on [Apply] Button
23. Select “Bar Section” tab
24. ID : 5 , Name : Section2
25. Click on [Apply] Button
26. Select “Bar” tab
27. ID : 6 , Name : Reinfo2
28. Material : (2: Steel)
29. Cross Sectional Area: 100 mm^2
30. Select “reinforcement”
31. Click on [OK] Button
32. Click on [Close] Button

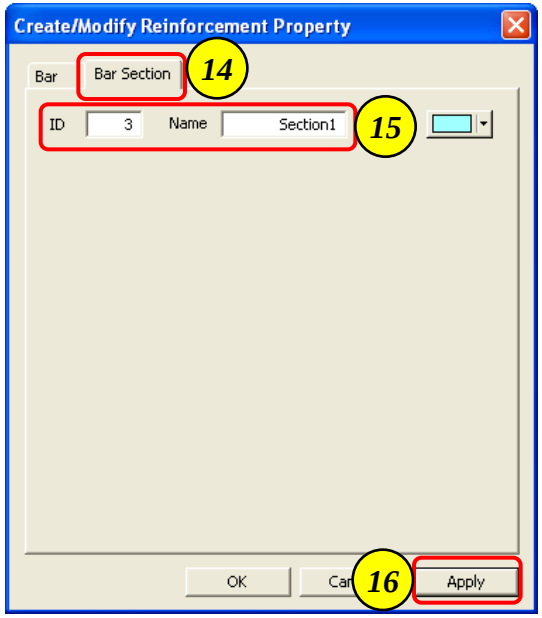
Step 9.

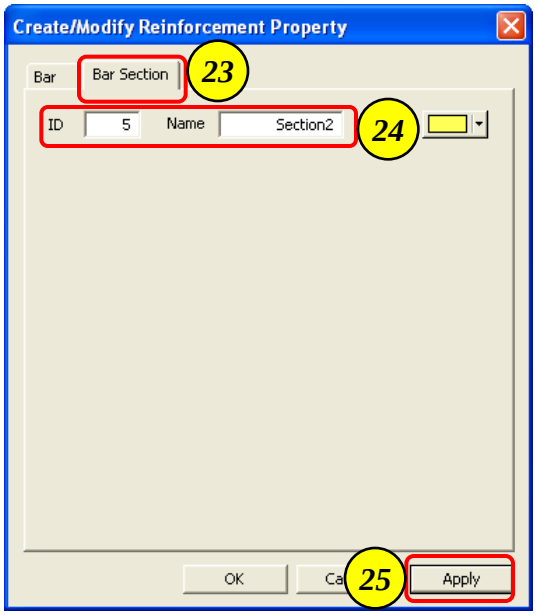

Property Manager
ID
Name
Type
Sub-Type
1 Beam 2D Plane Stress
2 Plates 2D Plane Stress
3 Section1 1D Bar Section
4 Reinfo1 1D Bar
5 Section2 1D Bar Section
6 Reinfo2 1D Bar
Create
1D...
2D...
3D...
Interface...
Reinforcement...
Others...
2, 13
Close
30

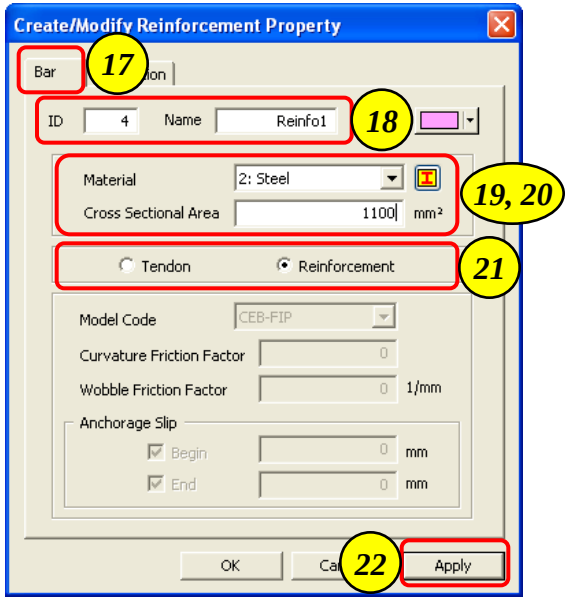

Create/Modify 2D Property
Plate
Plane Stress
3
Plot 2D
ID 1
Name Beam
4
Material 1: Concrete
Thickness 300 mm
5, 6
OK
7
Apply

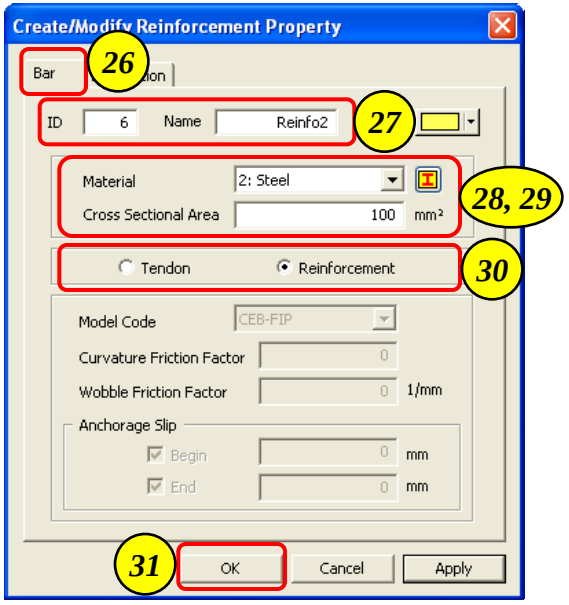

Create/Modify 2D Property
Plate
Plane Stress
8
Plot 2D
ID 2
Name Plates
9
Material 3: Wood
Thickness 300 mm
10, 11
OK
Cancel
Apply

Step 10.

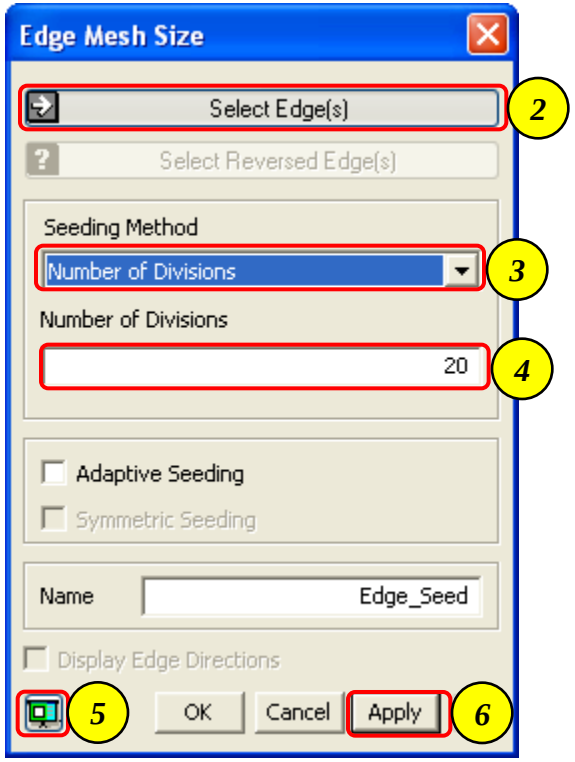

Create/Modify Reinforcement Property
Bar
Bar Section
14
ID
3
Name
Section1
15
OK
Cancel
16
Apply


Create/Modify Reinforcement Property
Bar
Bar Section
23
ID
5
Name
Section2
24
OK
Cancel
25
Apply

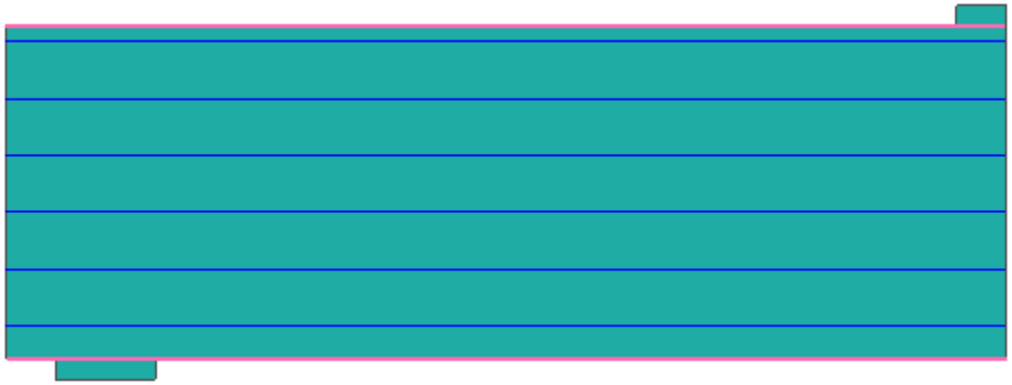

Create/Modify Reinforcement Property
17
Bar
ID
4
Name
Reinfo1
18
Material
2: Steel
Cross Sectional Area
1100
mm²
19, 20
Tendon
Reinforcement
21
Model Code
CEB-FIP
Curvature Friction Factor
0
Wobble Friction Factor
0
1/mm
Anchorage Slip
Begin
End
mm
OK
Cancel
22
Apply


Create/Modify Reinforcement Property
26
Bar
ID
6
Name
Reinfo2
27
Material
2: Steel
Cross Sectional Area
100
mm²
28, 29
Tendon
Reinforcement
30
Model Code
CEB-FIP
Curvature Friction Factor
0
Wobble Friction Factor
0
1/mm
Anchorage Slip
Begin
End
mm
31
OK
Cancel
Apply

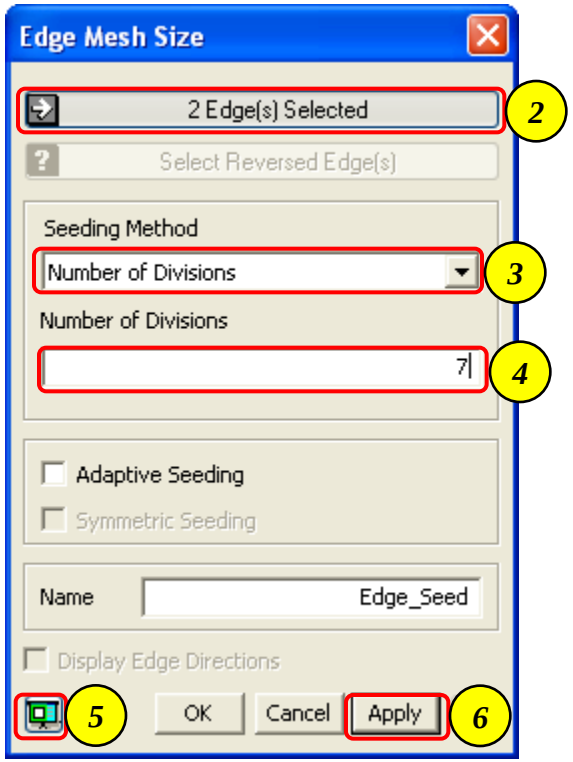
Step 11.



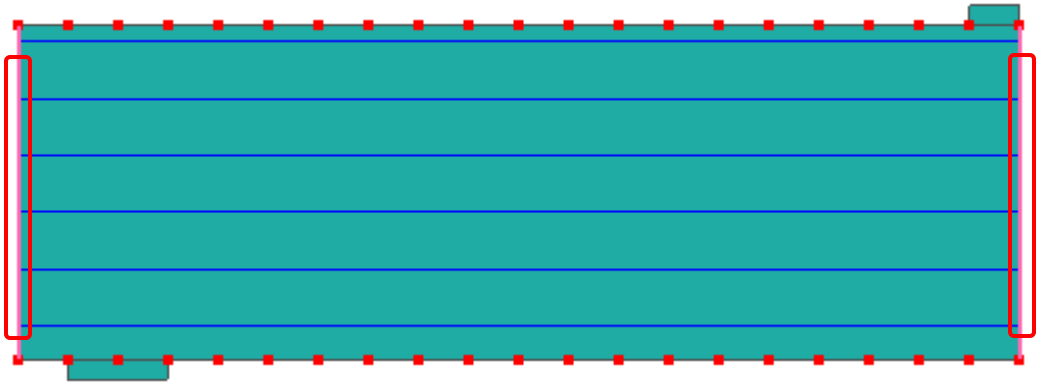
1. *Mesh > Size Control > Along Edge ...*
2. *Select two edges as shown in figure*
3. *Seeding Method: Number of Divisions*
4. *Number of Divisions: 20*
5. *Click on*  *to preview seeding*
6. *Click on [Apply] Button*



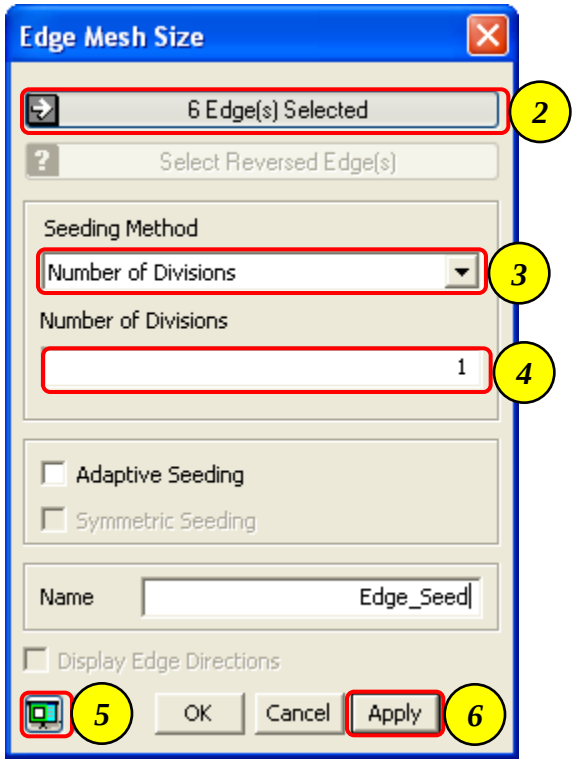
Step 12.



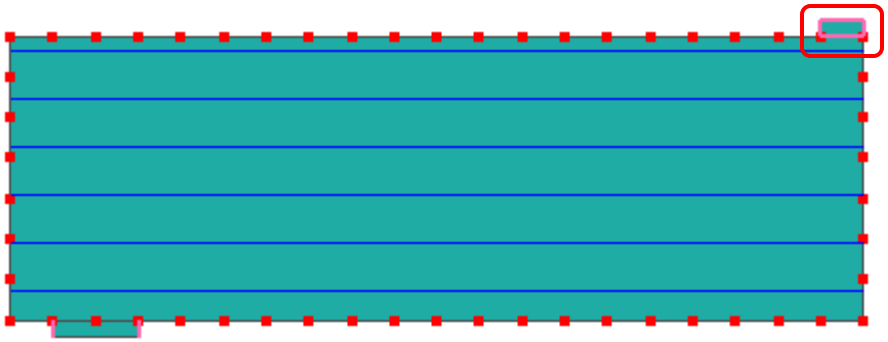
1. *Mesh > Size Control > Along Edge ...*
2. *Select two edges as shown in figure*
3. *Seeding Method: Number of Divisions*
4. *Number of Divisions: 7*
5. *Click on*  *to preview seeding*
6. *Click on [Apply] Button*



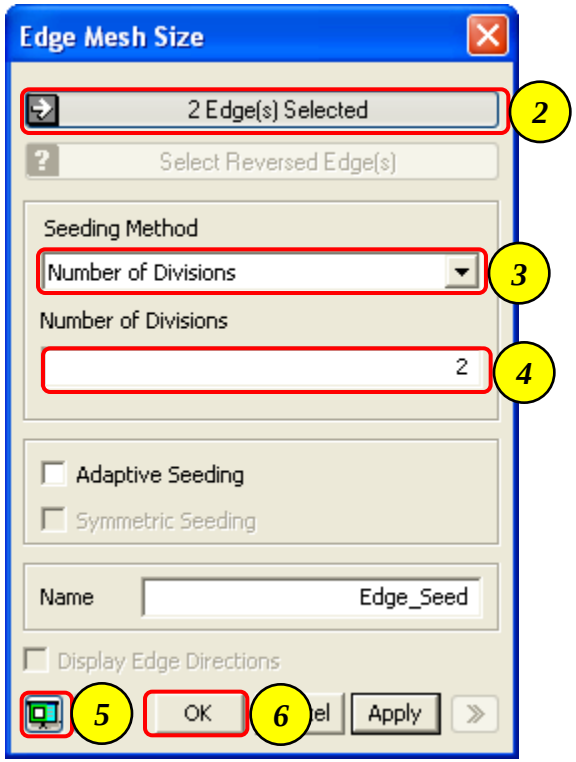
Step 13.



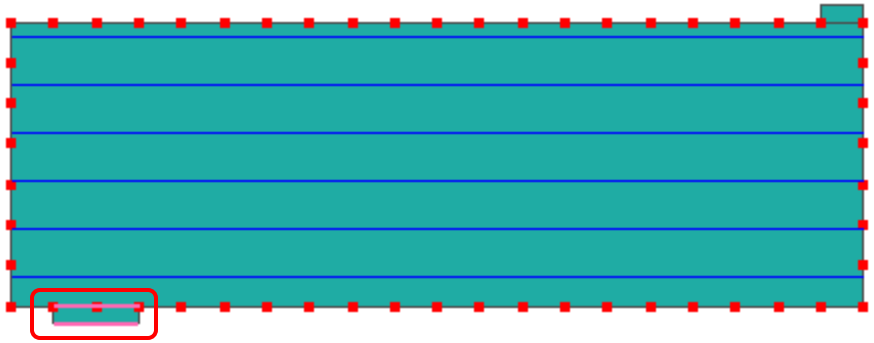
1. Mesh > Size Control > Along Edge ...
2. Select two edges as shown in figure
3. Seeding Method: Number of Divisions
4. Number of Divisions: 1
5. Click on  to preview seeding
6. Click on [Apply] Button



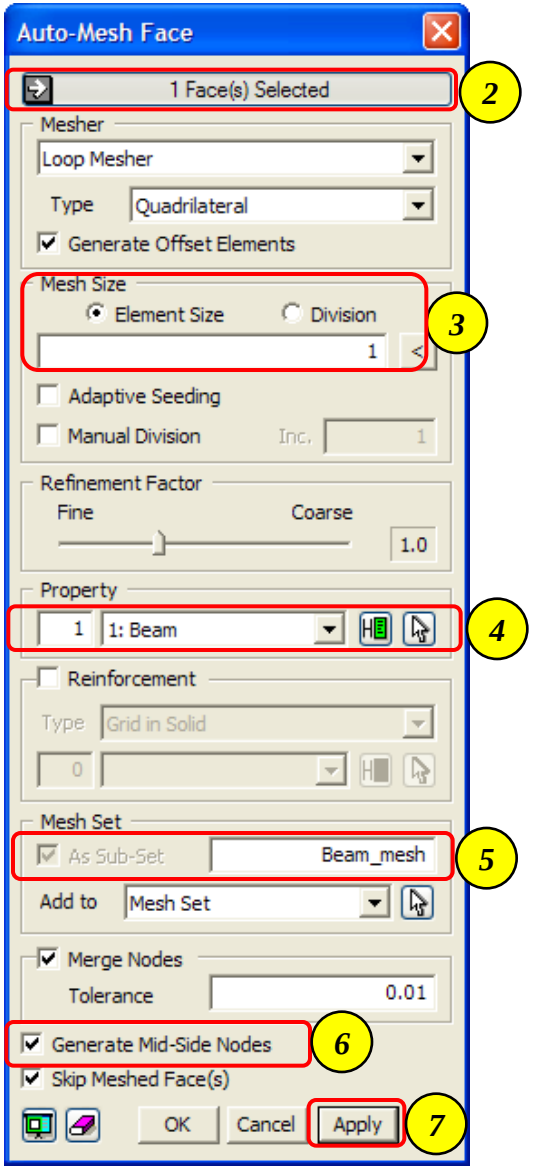
Step 14.



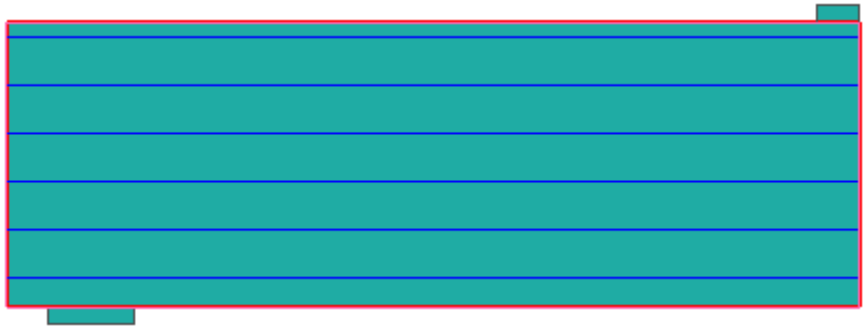
1. Mesh > Size Control > Along Edge ...
2. Select two edges as shown in figure
3. Seeding Method: Number of Divisions
4. Number of Divisions: 2
5. Click on  to preview seeding
6. Click on [OK] Button

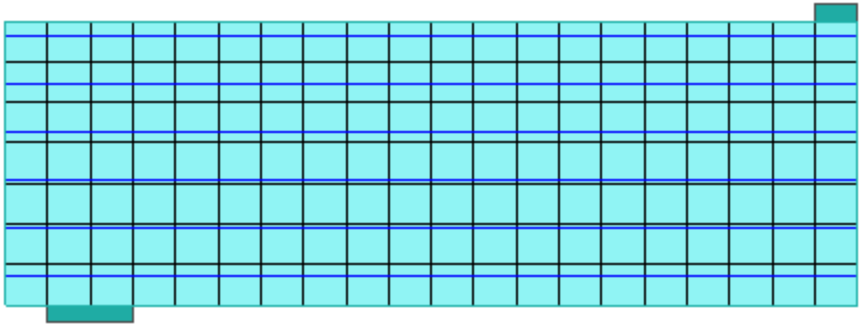


Step 15.

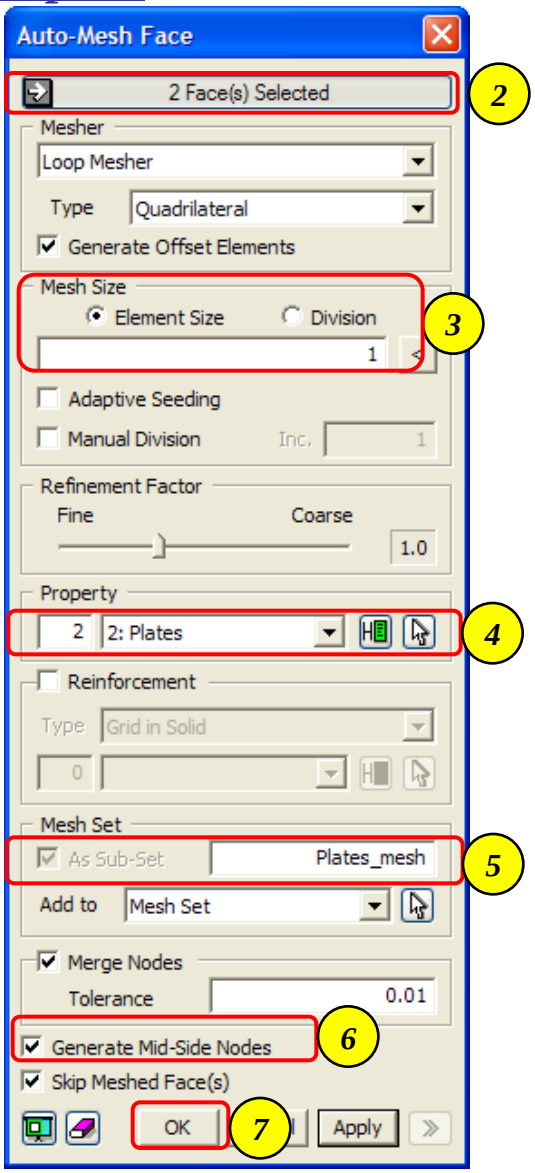


1. *Mesh > Auto Mesh > Face ...*
2. *Select the highlighted face*
3. *Element Size: 1*
4. *Property: 1: Beam*
5. *Mesh Set: Beam_mesh*
6. *Check on "Generate Mid-Side Nodes" box*
7. *Click on [Apply] Button*

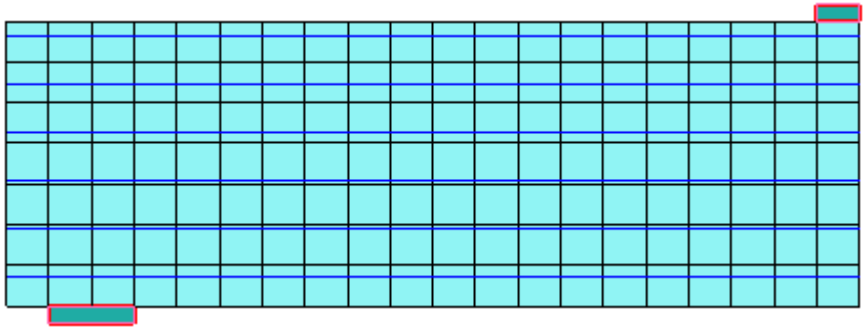


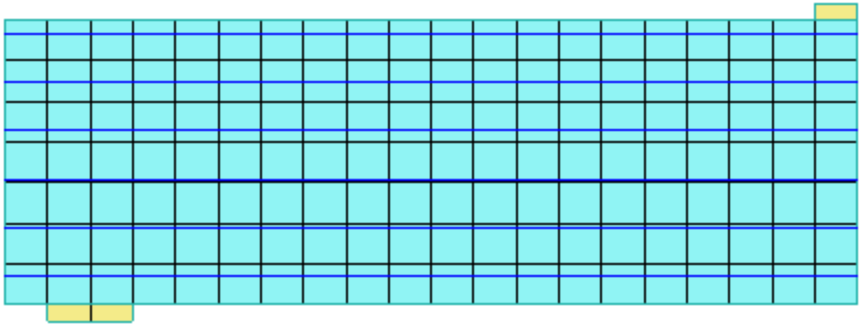


Step 16.

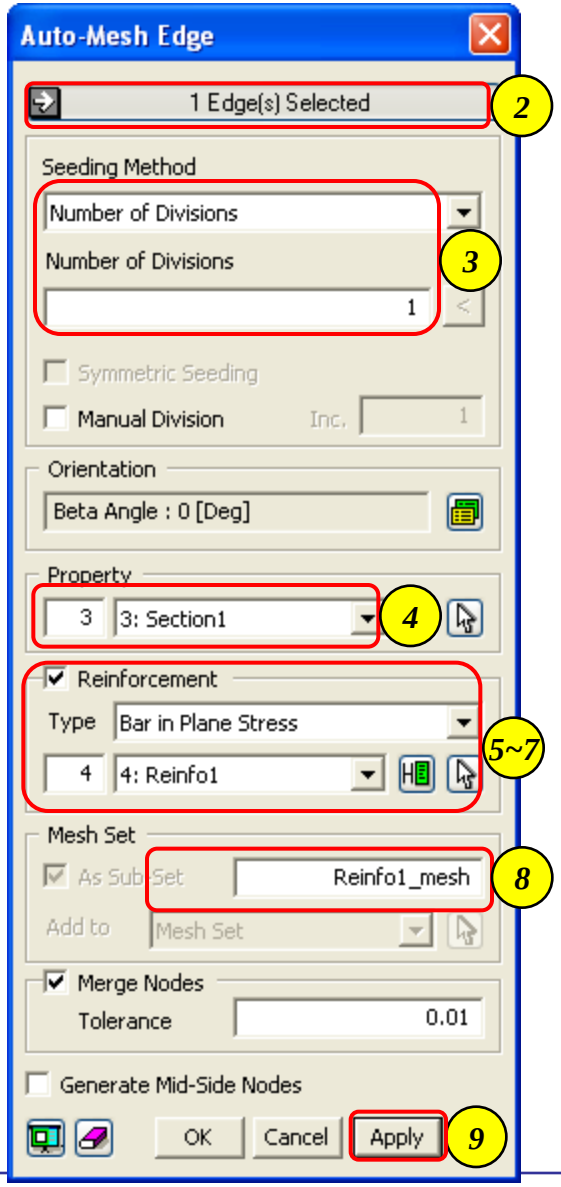


1. *Mesh > Auto Mesh > Face ...*
2. *Select the highlighted faces*
3. *Element Size: 1*
4. *Property: 2: Plates*
5. *Mesh Set: Plates_mesh*
6. *Check on "Generate Mid-Side Nodes" box*
7. *Click on [OK] Button*



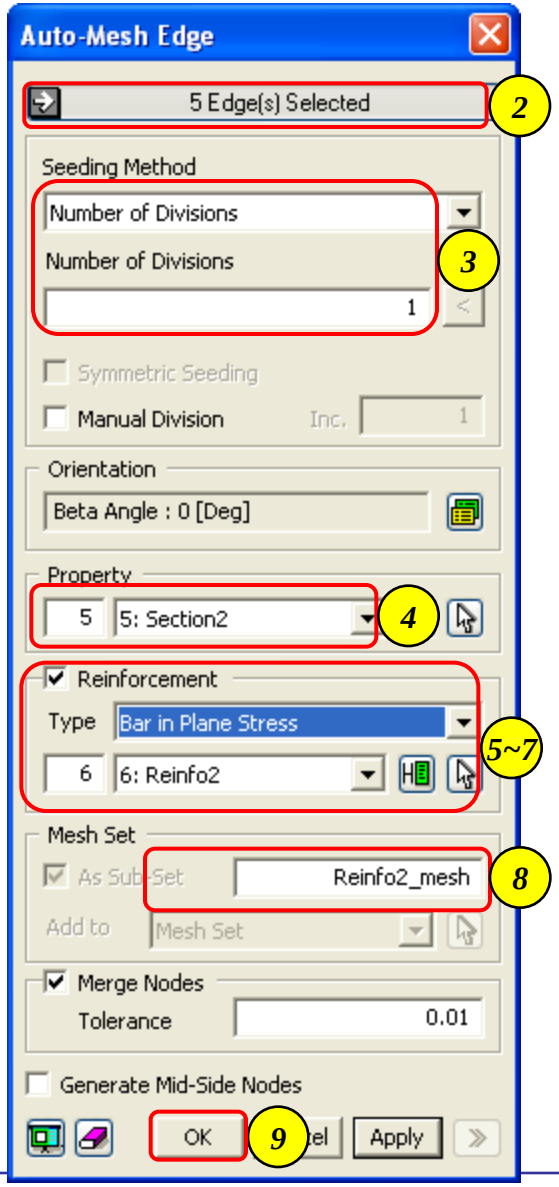


Step 17.

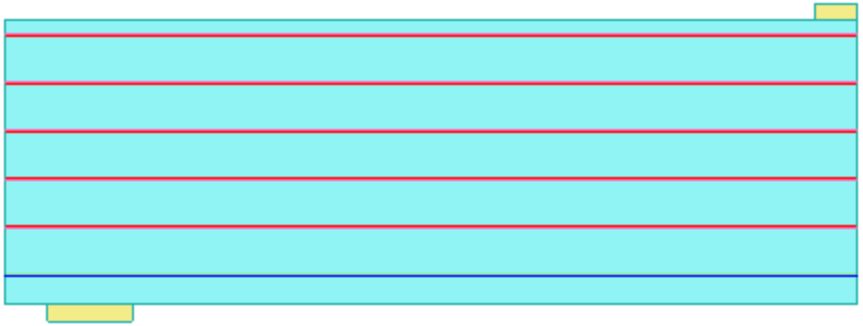


1. *Mesh > Auto Mesh > Edge ...*
2. *Select the highlighted edge in the Figure*
3. *Seeding method – Number of Divisions : 1*
4. *Property: 3: Section1*
5. *Tick on Reinforcement*
6. *Type: Bare in "Plane Stress"*
7. *Property: 4: Reinfo1*
8. *Mesh Set: Reinfo1_mesh*
9. *Click on [Apply] Button*

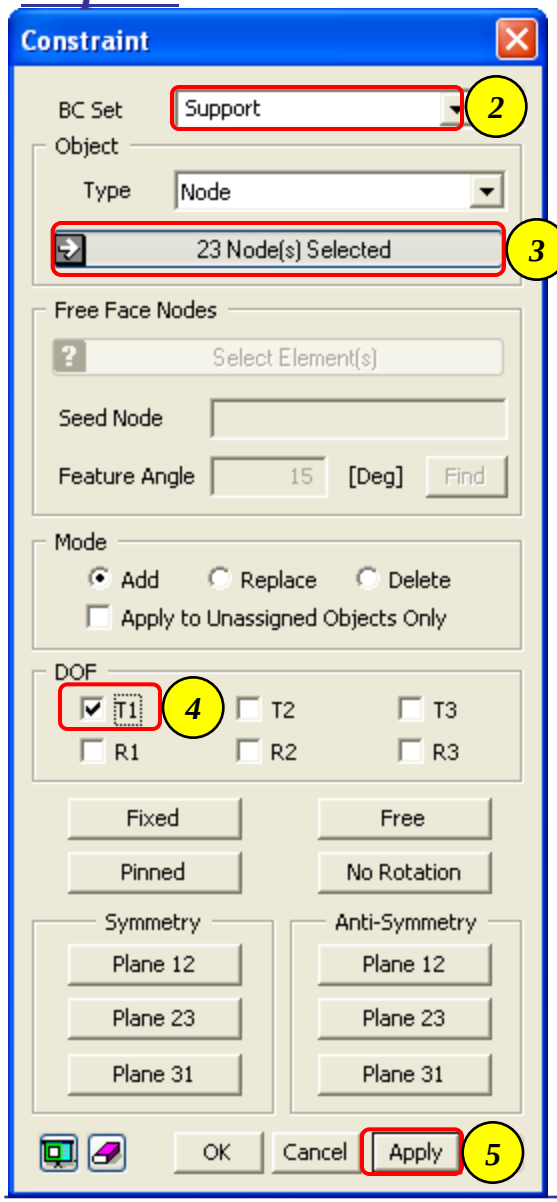
Step 18.



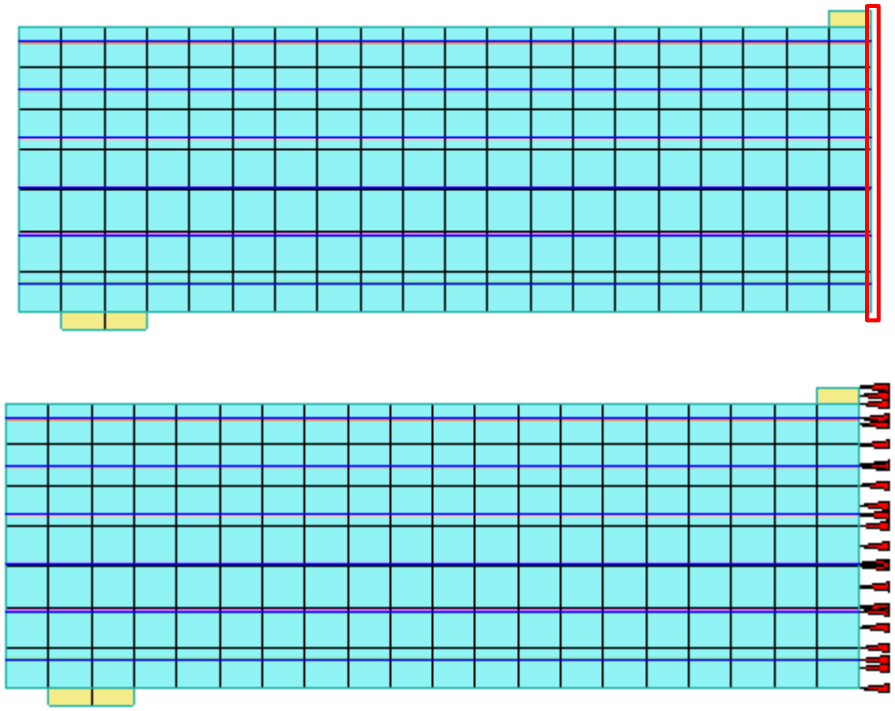
1. *Mesh > Auto Mesh > Edge ...*
2. *Select the highlighted edges in the Figure*
3. *Seeding method – Number of Divisions : 1*
4. *Property: 5: Section2*
5. *Tick on Reinforcement*
6. *Type: Bare in "Plane Stress"*
7. *Property: 6: Reinfo2*
8. *Mesh Set: Reinfo2_mesh*
9. *Click on [OK] Button*



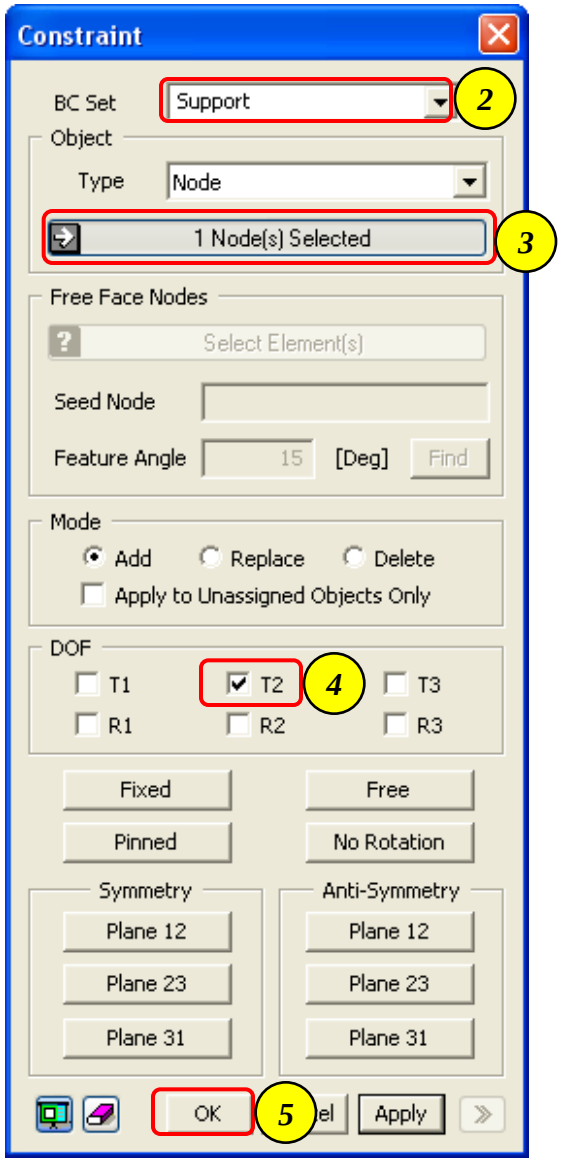
Step 19.



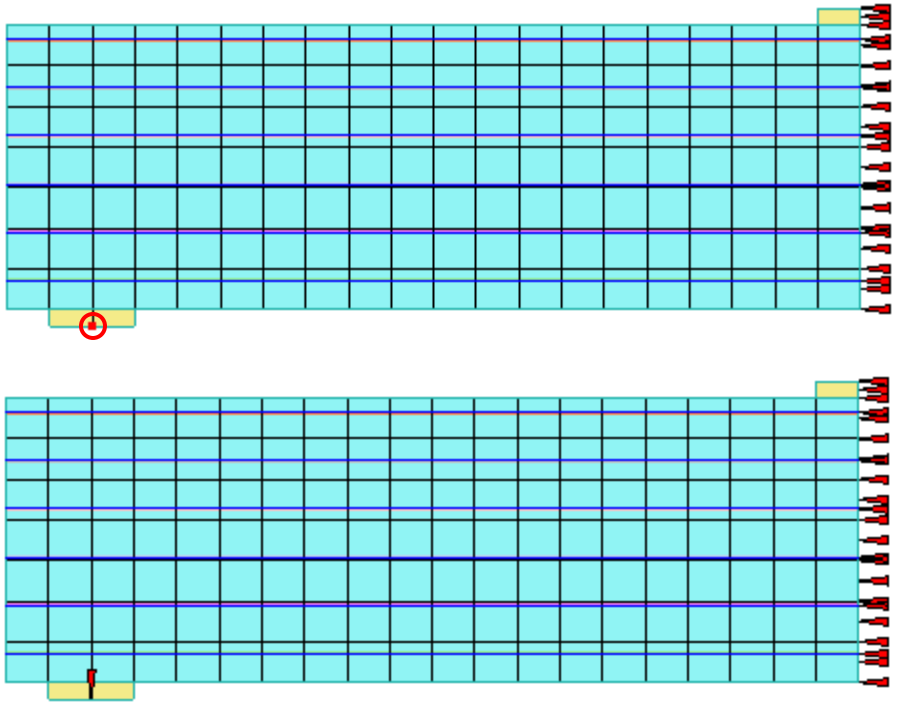
1. *Analysis > BC > Constraint …*
2. *BC Set : Support*
3. *Select 23 Nodes (See Figure)*
4. *Click on "T1"*
5. *Click [Apply] Button*



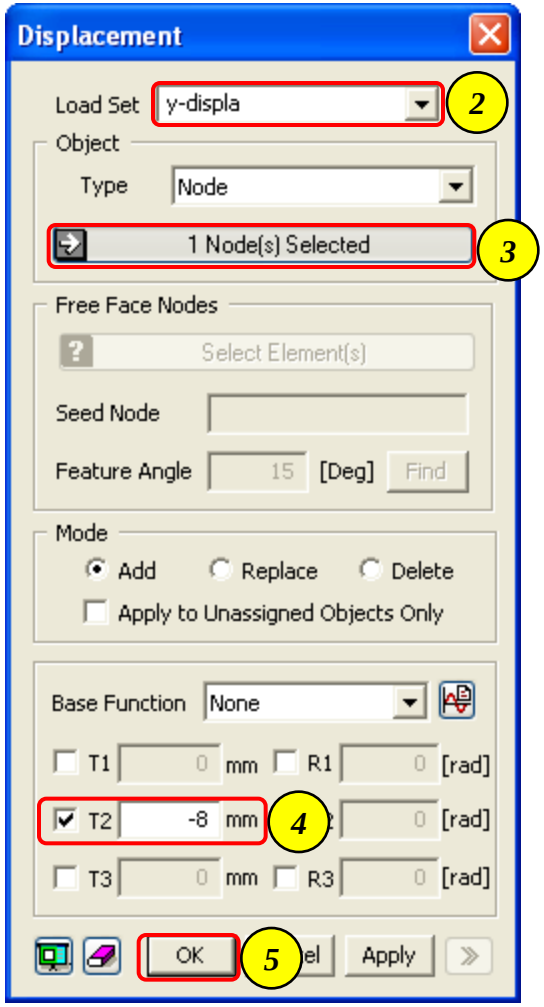
Step 20.



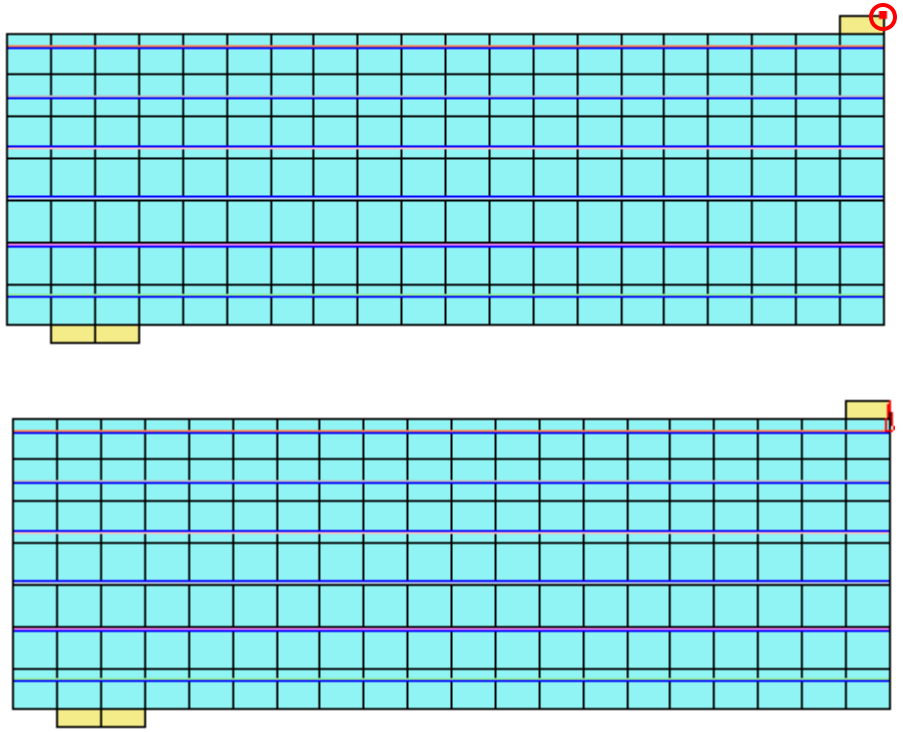
1. *Analysis > BC > Constraint …*
2. *BC Set : Support*
3. *Select the node (See Figure)*
4. *Click "T2"*
5. *Click [OK] Button*



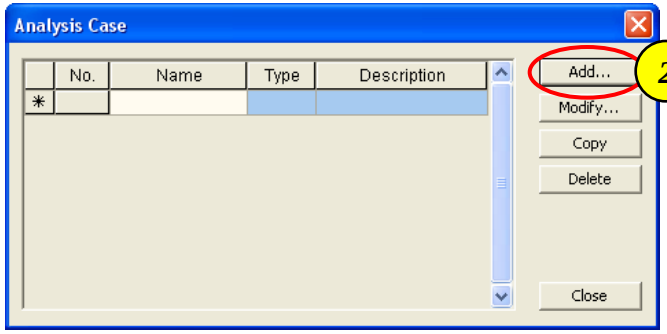
Step 21.



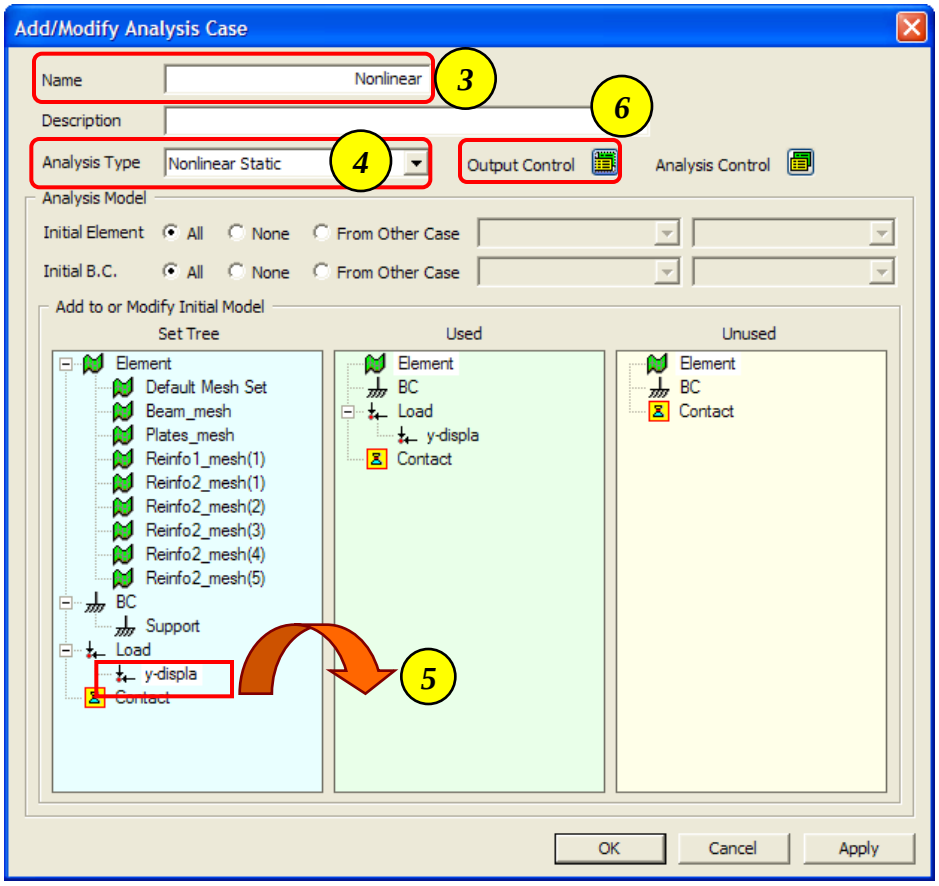
1. *Analysis > Load > Displacement …*
2. *Load Set : y-displa*
3. *Select the Node (See Figure)*
4. *Click on "T2" : -8 mm*
5. *Click [OK] Button*

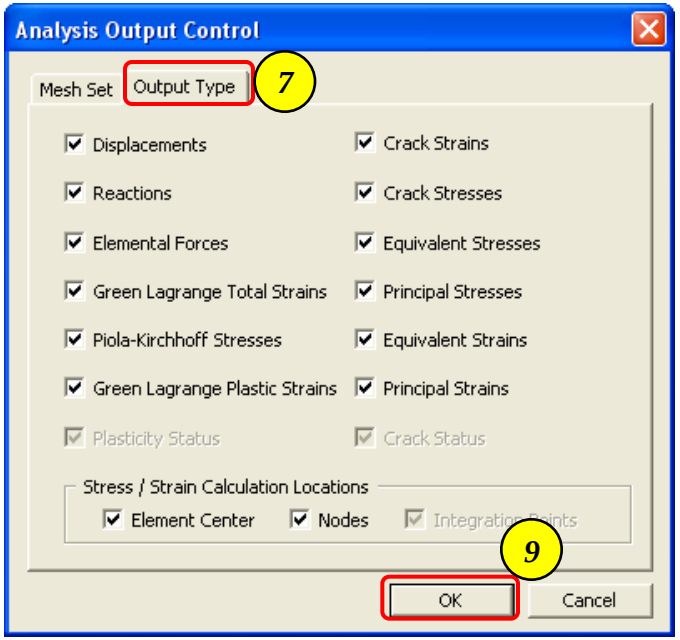


Step 22.

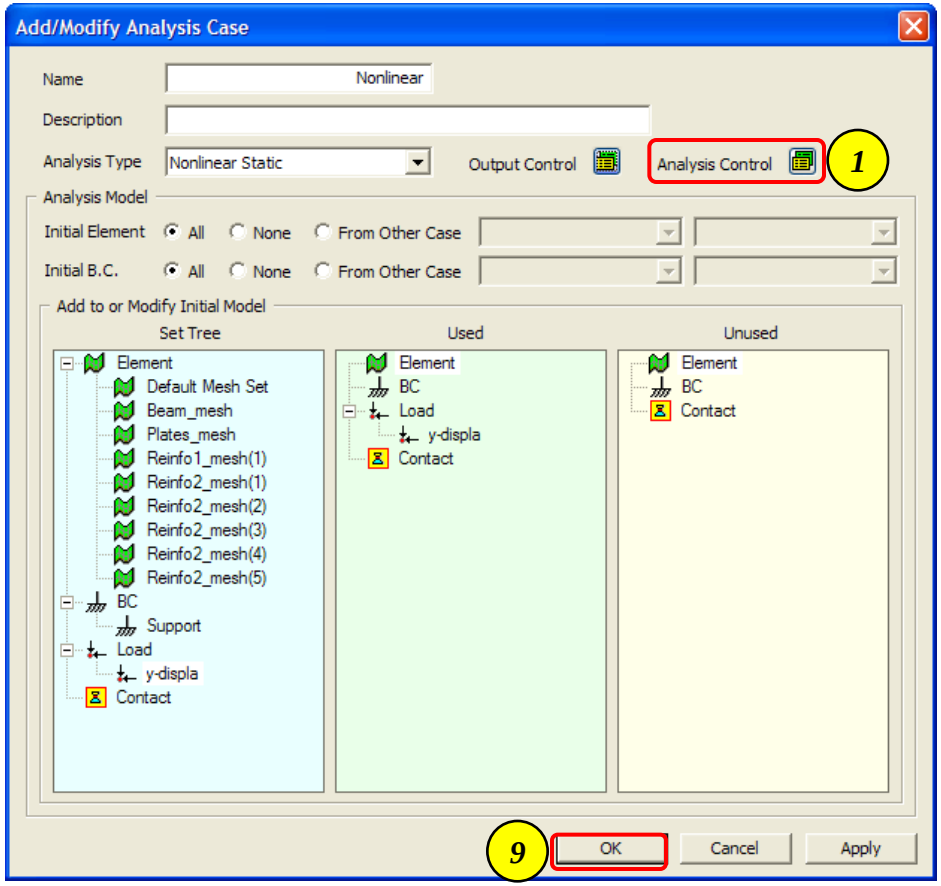


1. *Analysis > Analysis Case…*
2. *Click [Add] Button*
3. *Name : "Nonlinear"*
4. *Analysis Type: Nonlinear Static*
5. *Drag & Drop Load Set marked by "□" (See Figure)*
6. *Click on Output Control*
7. *Select Output Type tab*
8. *Select all the variables*
9. *Click [OK] Button*

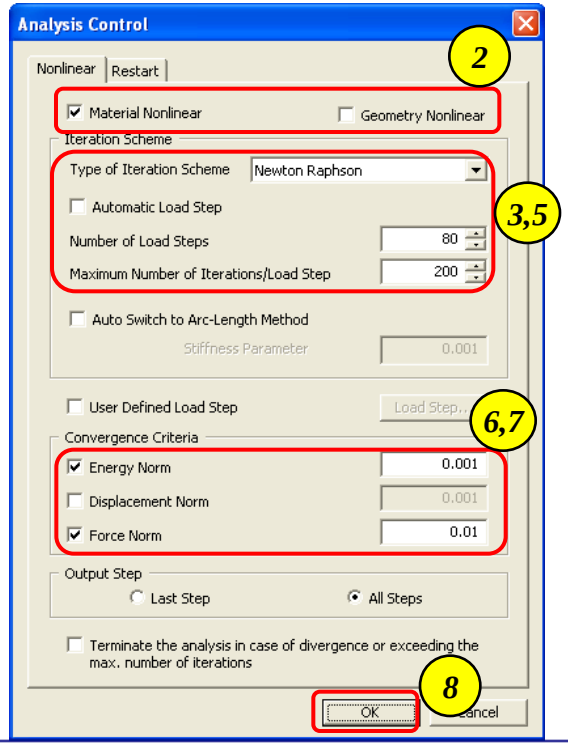




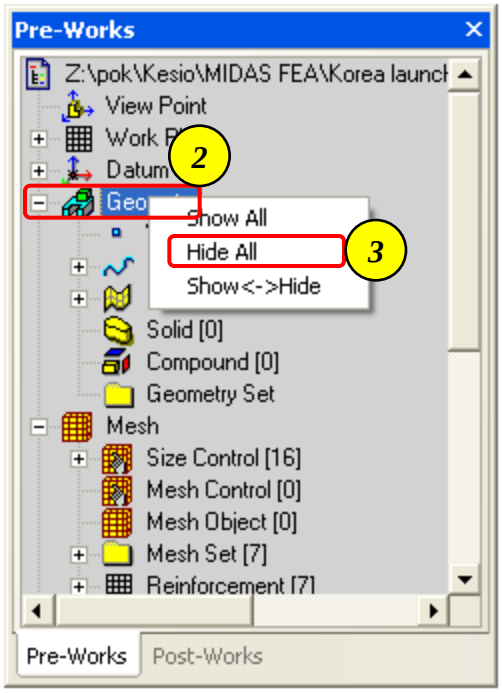
Step 23.



1. Click on Analysis Control button
2. Select Material Nonlinear
3. Type of Iteration Scheme: Newton Raphson
4. Number of Load Steps: 80
5. Maximum Number of Iterations/Load Step: 200
6. Energy Norm: 0.001
7. Force Norm: 0.01
8. Click [OK] Button
9. Click [OK] Button



Step 24.



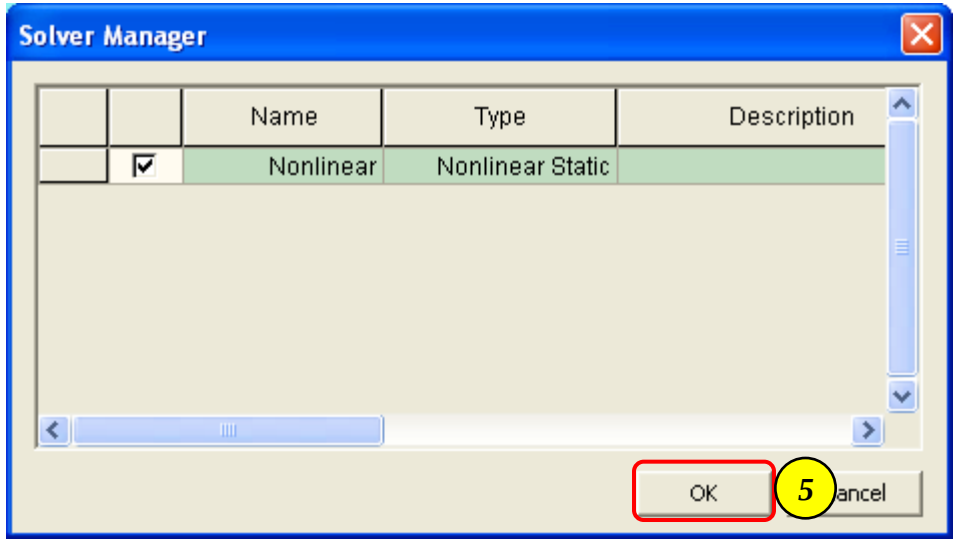
1. File > Save… (NL4 rcbeam.feb)

2. Pre-Works Tree : Geometry…

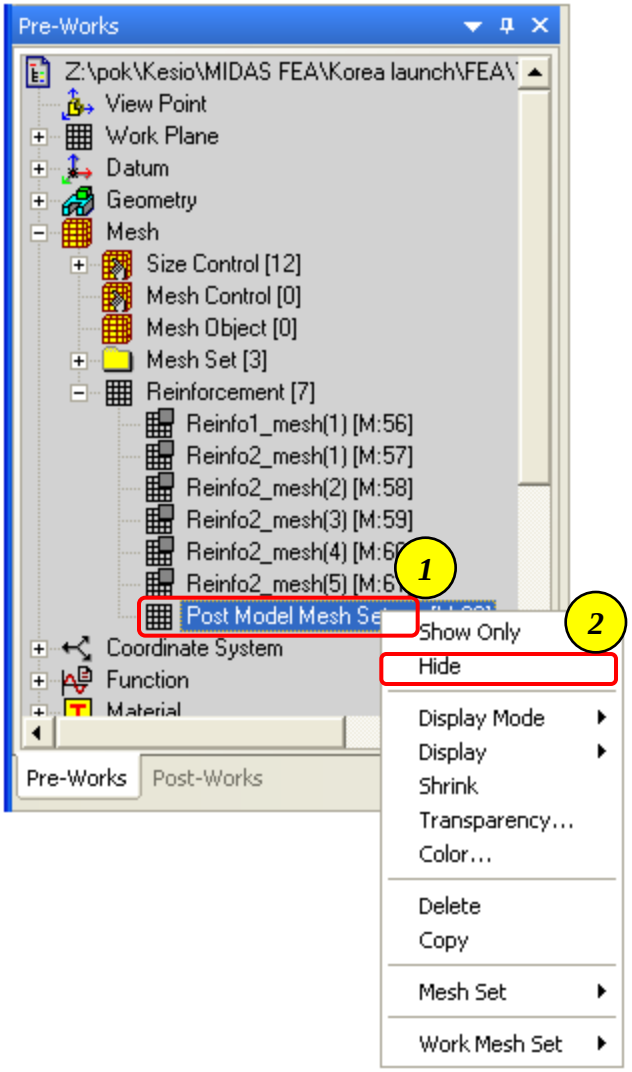
3. Click Right Mouse Button and Select "Hide All"

4. Analysis > Solve…

5. Click [OK] Button

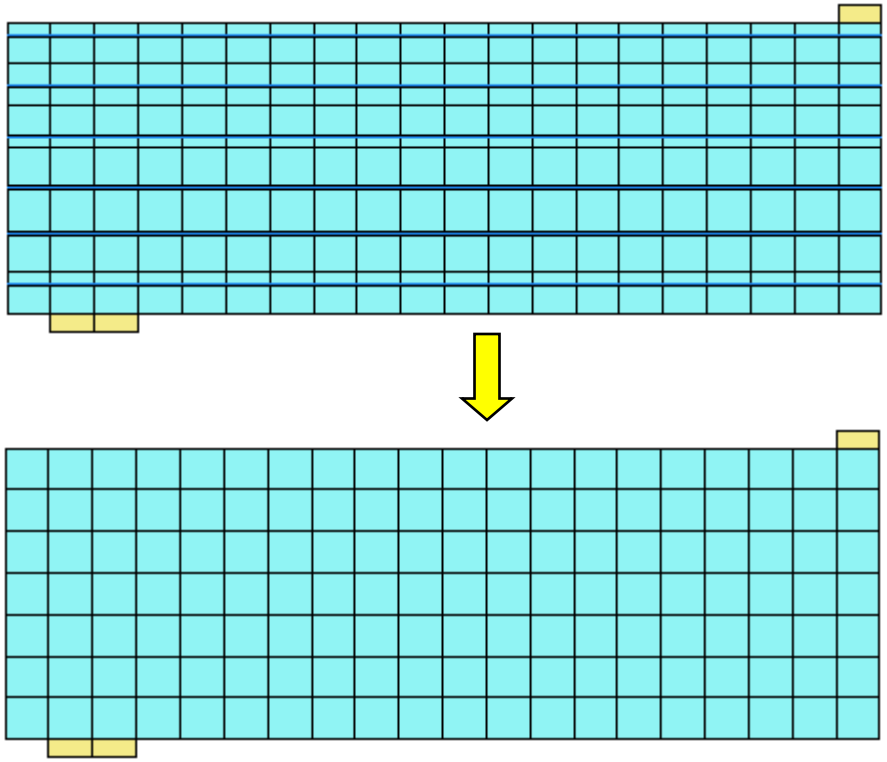


Step 25.



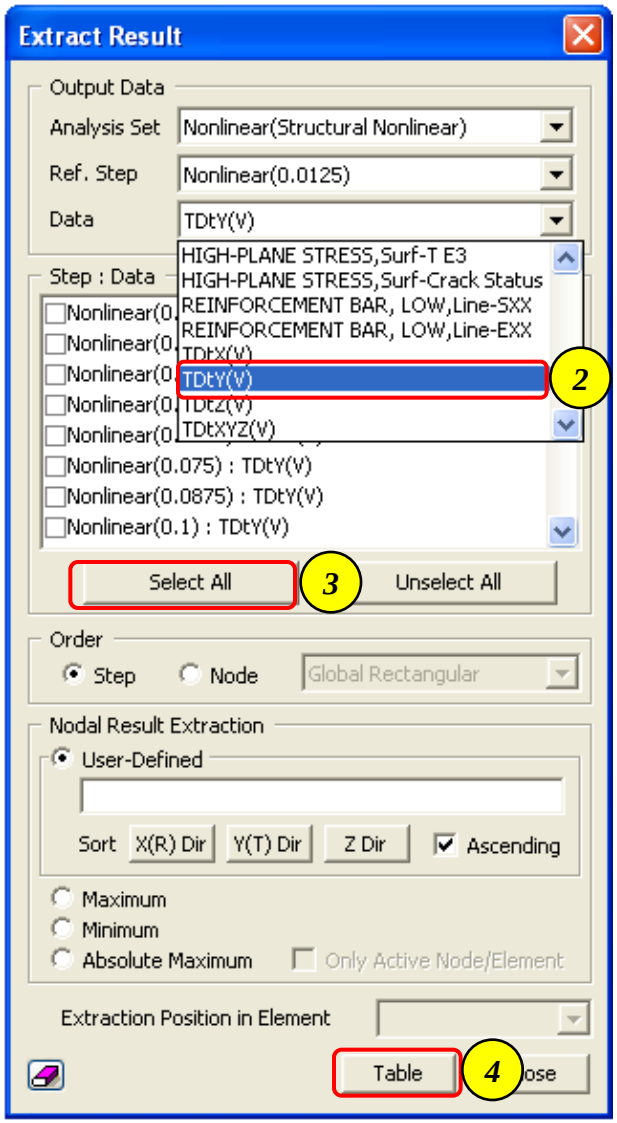
1. Pre-Works Tree : Mesh > Reinforcement > Post Model Mesh Set

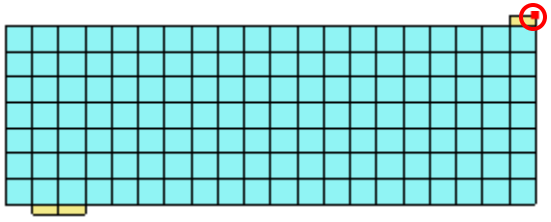
2. Click Right Mouse Button and Select "Hide"

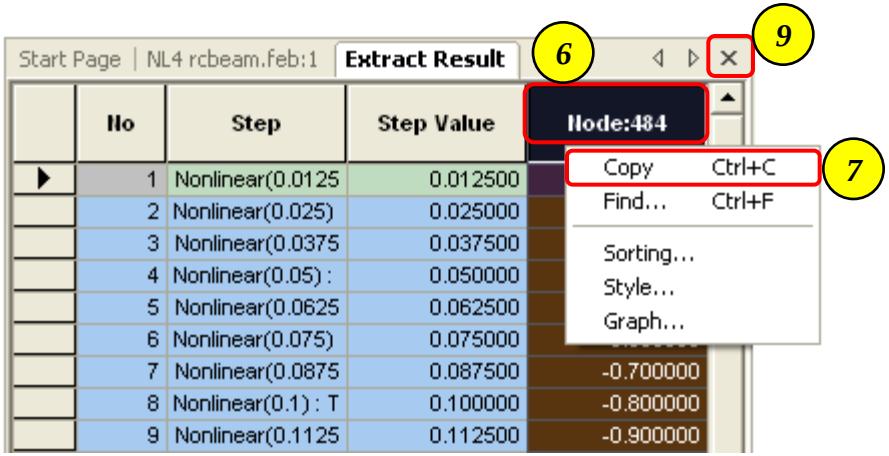


Step 26.

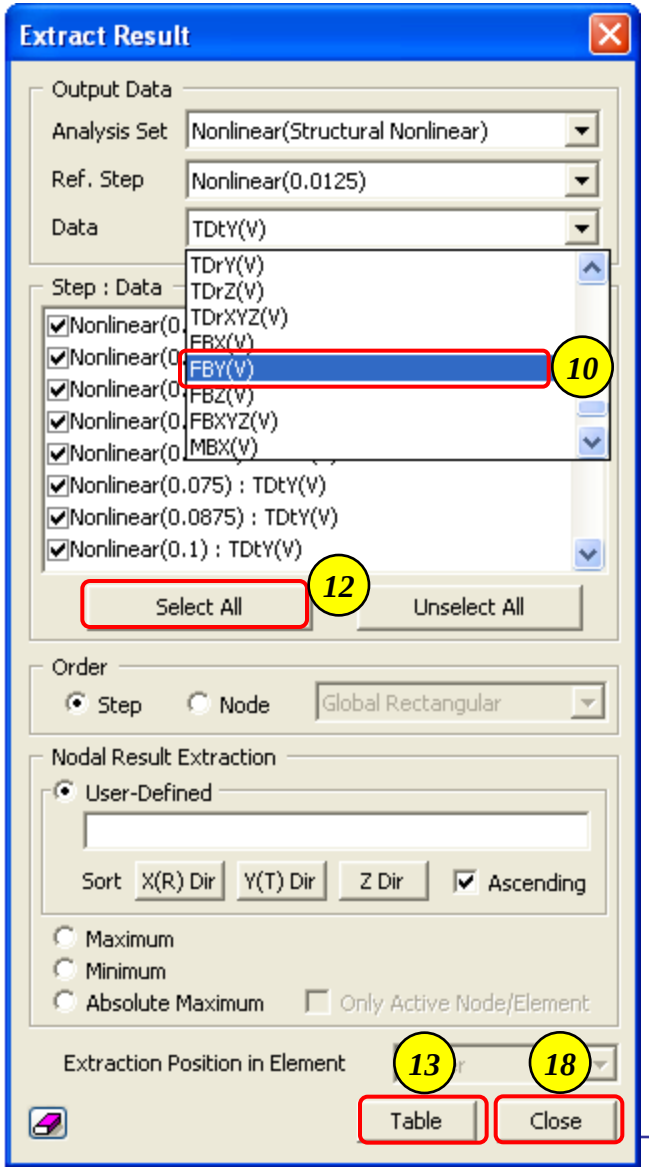
1. *Post > Extract Result…*
2. *Data: Select "TDty" Result*
3. *Select the Node as Shown in the Figure*
4. *Click on [Select All] Button*
5. *Click [Table] Button*
6. *Click on "Node:484" Column*
7. *Click Right Mouse Button and Select "Copy"*
8. *Paste the Results into an Excel Sheet*
9. *Close*







Step 27.



10. Data: Select "FBY(V)" Result

11. Select the Node as Shown in the Figure

12. Click on [Select All] Button

13. Click [Table] Button

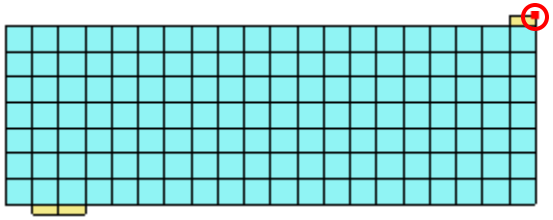
14. Click on "Node:484" Column

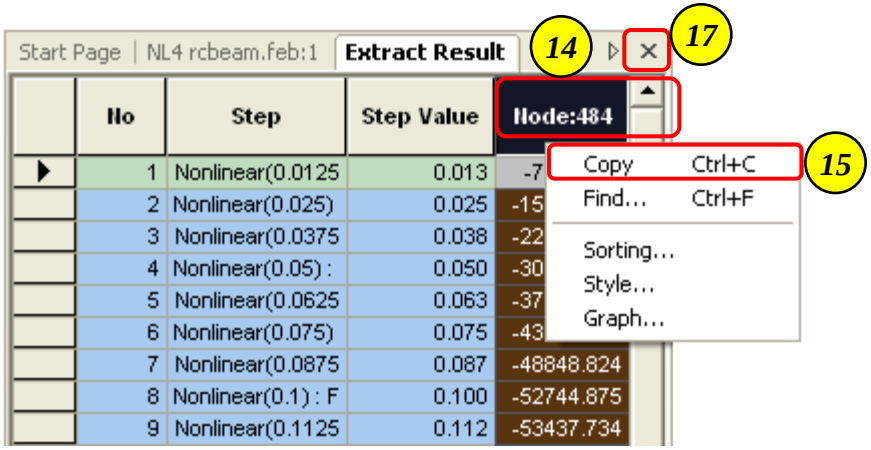
15. Click Right Mouse Button and Select "Copy"

16. Paste the Results into an Excel Sheet

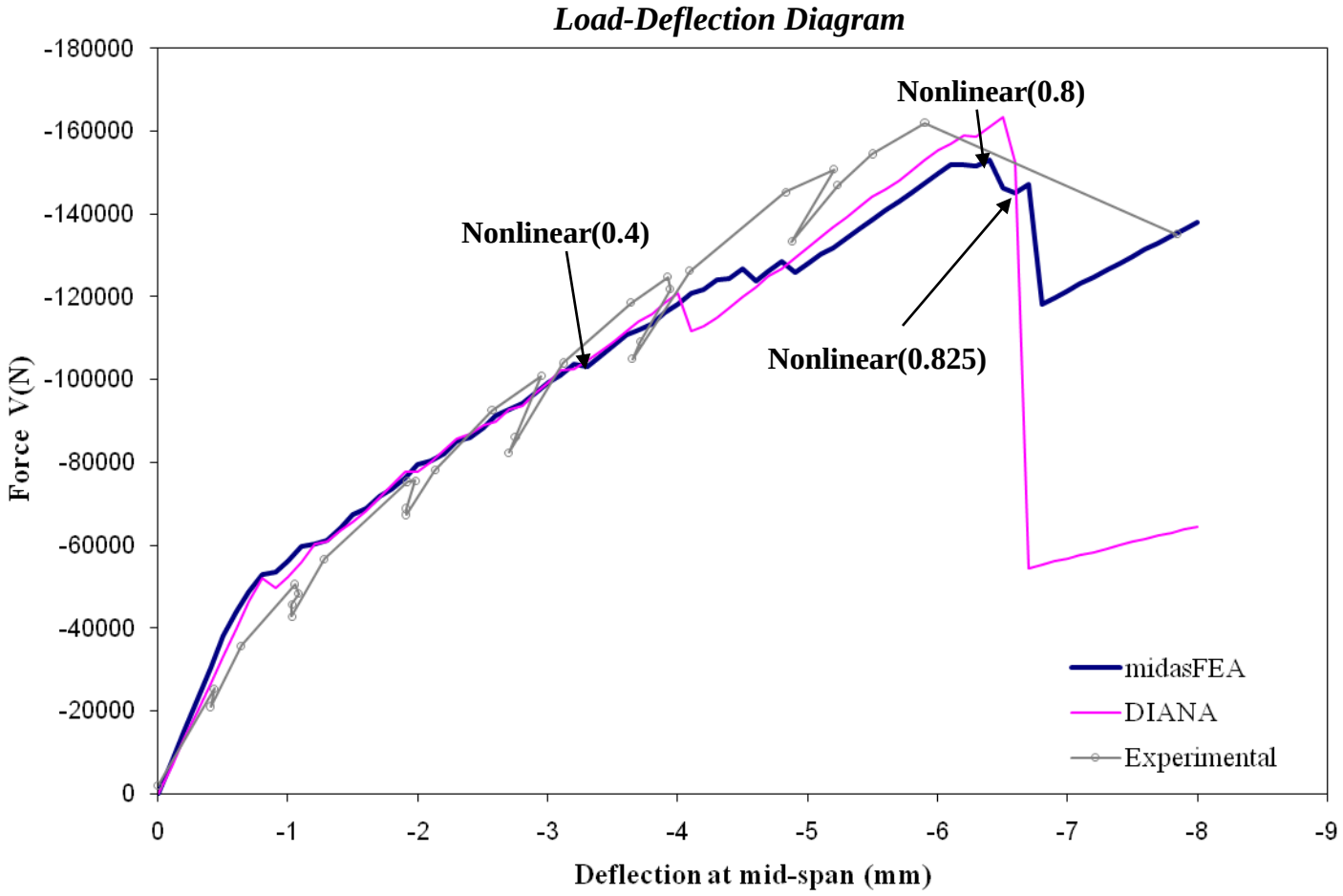
17. Close

18. Click on [Close] Button





Step 28.


Load-Deflection Diagram
Force V(N)
-180000
-160000
-140000
-120000
-100000
-80000
-60000
-40000
-20000
0
0
-1
-2
-3
-4
-5
-6
-7
-8
-9
Deflection at mid-span (mm)
Nonlinear(0.8)
Nonlinear(0.4)
Nonlinear(0.825)
midasFEA
DIANA
Experimental