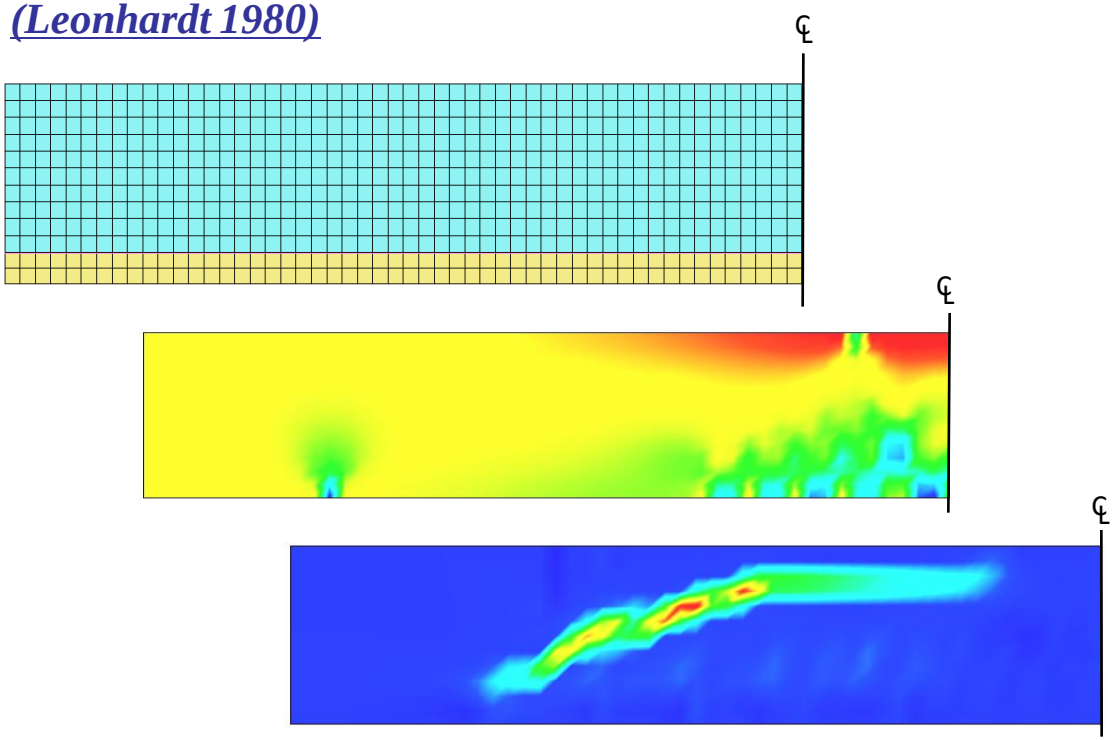
NL-3. Shear Crack Failure in an RC beam

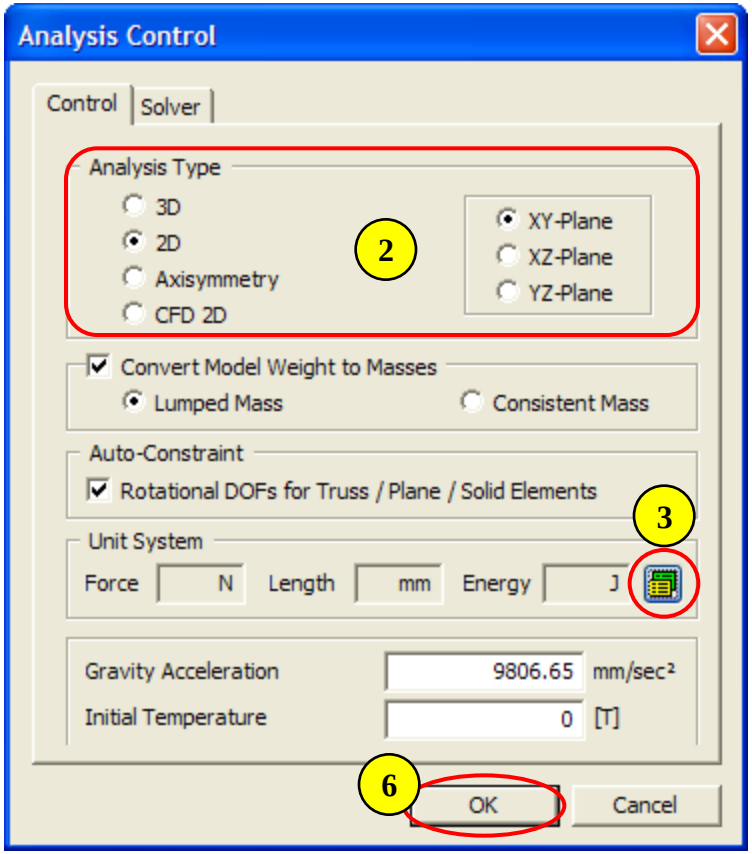
(Leonhardt 1980)



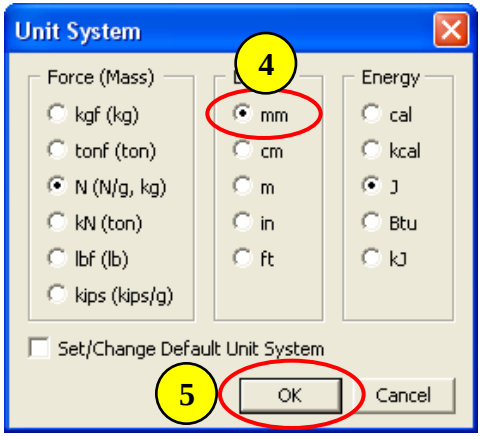
Overview

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

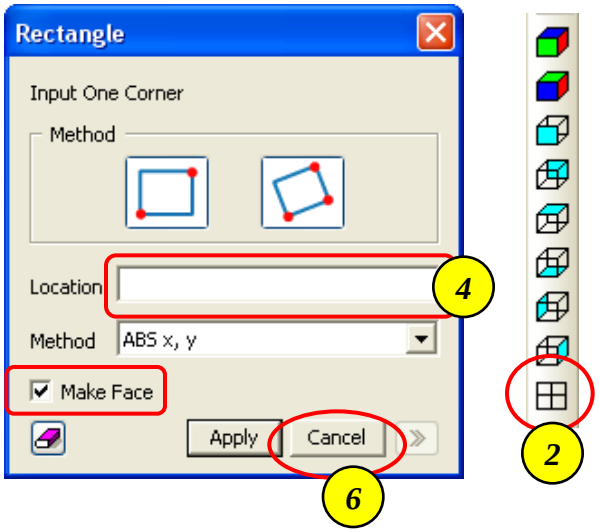
Step 1.



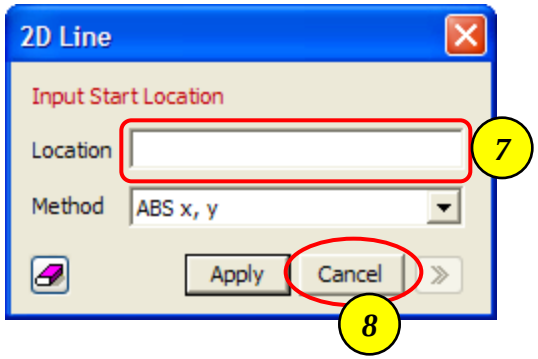
1. *Analysis > Analysis Control – Control tab*
2. *Analysis Type : 2D and XY - Plane*
3. *Click*  *Button (Unit System)*
4. *Length : mm*
5. *Click on* **[OK]** *Button*
6. *Click on* **[OK]** *Button*

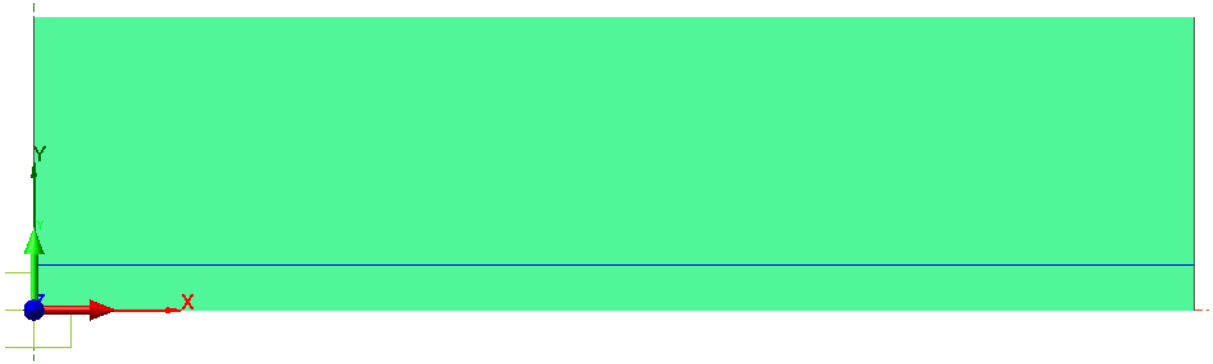


Step 2.



1. *Toggle off "Toggle Grid"*
2. *Click "Normal View"*
3. *Geometry > Curve > Create on WP > Rectangle (Wire)...*
4. *Location : (0), <1275, 320>*
5. *Click on [Cancel] Button*
6. *Geometry > Curve > Create on WP >Line ...*
7. *Location : (0, 50), <1275, 0>*
8. *Click on [Cancel] Button*





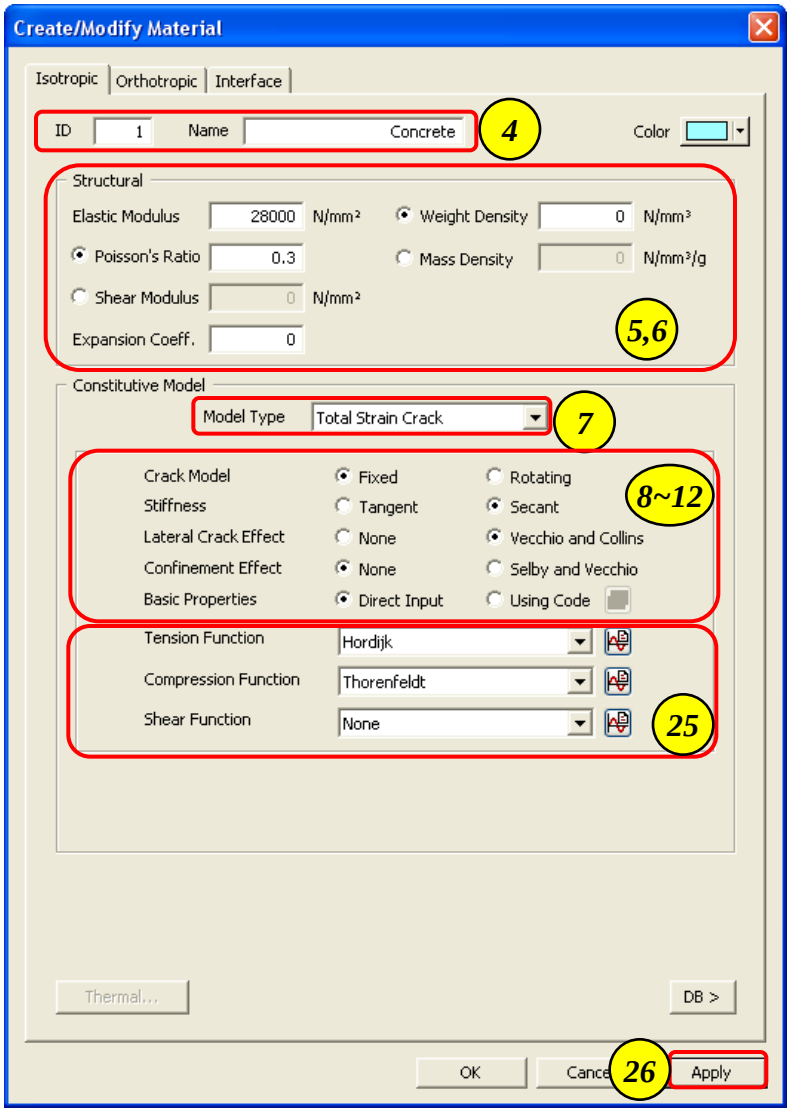
() : "ABS x, y" , <> : "REL dx, dy"
(0) same as (0, 0)

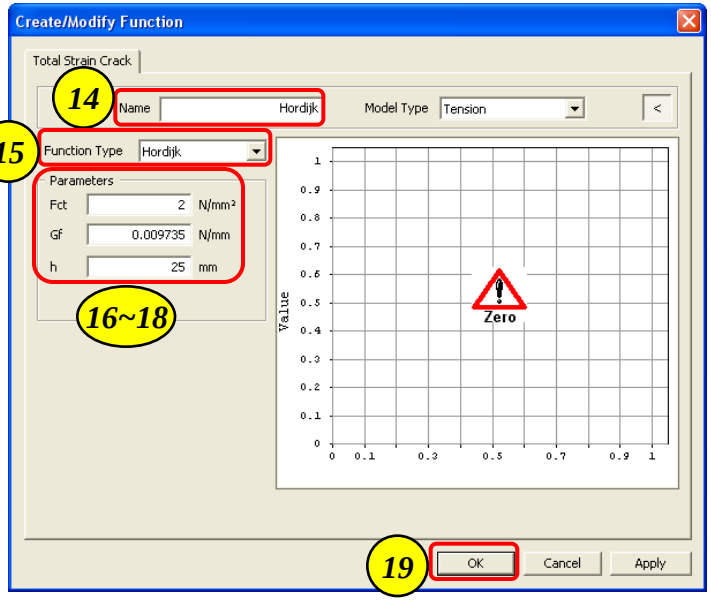
[Esc] as shortcut for [Cancel]

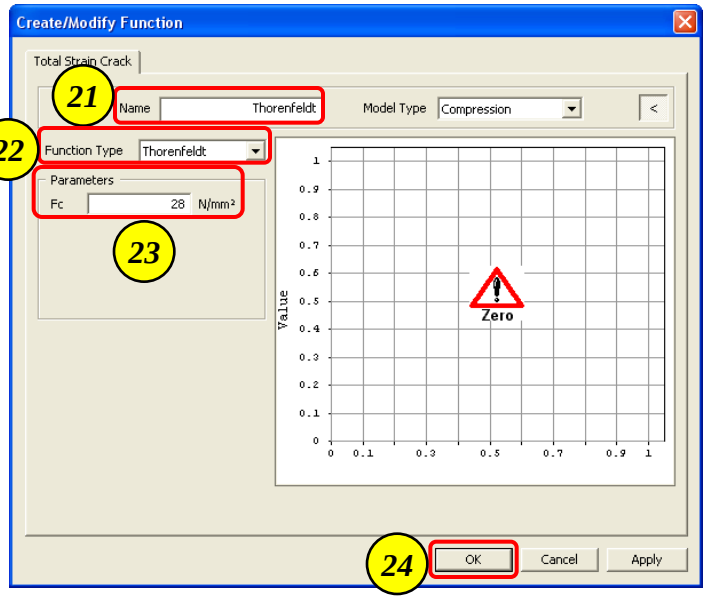
Step 3.

1. Analysis > Material …
2. Click [Create] Button
3. Select “Isotropic” tab
4. ID : 1 , Name : Concrete
5. Elastic Modulus : 28000 N/mm^2
6. Poisson’s Ratio : 0.3
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: Hordijk
15. Function Type: Hordijk
16. Fct = 2 N/mm^2
17. Gf = 0.009735 N/mm
18. h = 25 mm
19. Click on [OK] Button
20. Click on  to define Compression Function
21. Name: Thorenfeldt
22. Function Type: Thorenfeldt
23. Fc = 28 N/mm^2
24. Click on [OK] Button
25. Select the tension and compression functions from the drop lists
26. Click on [Apply] Button
27. ID : 2 , Name : Steel
28. Elastic Modulus : 200000 N/mm^2
29. Poisson’s Ratio : 0.3
30. Model Type : Von Mises
31. Click on  to define Hardening/Softening Function
32. Name: Steel_Hardening
33. Enter values as shown in the picture
34. Click on [OK] Button
35. Select the hardening function from the drop list
36. Click on [OK] Button
37. Click on [Close] Button

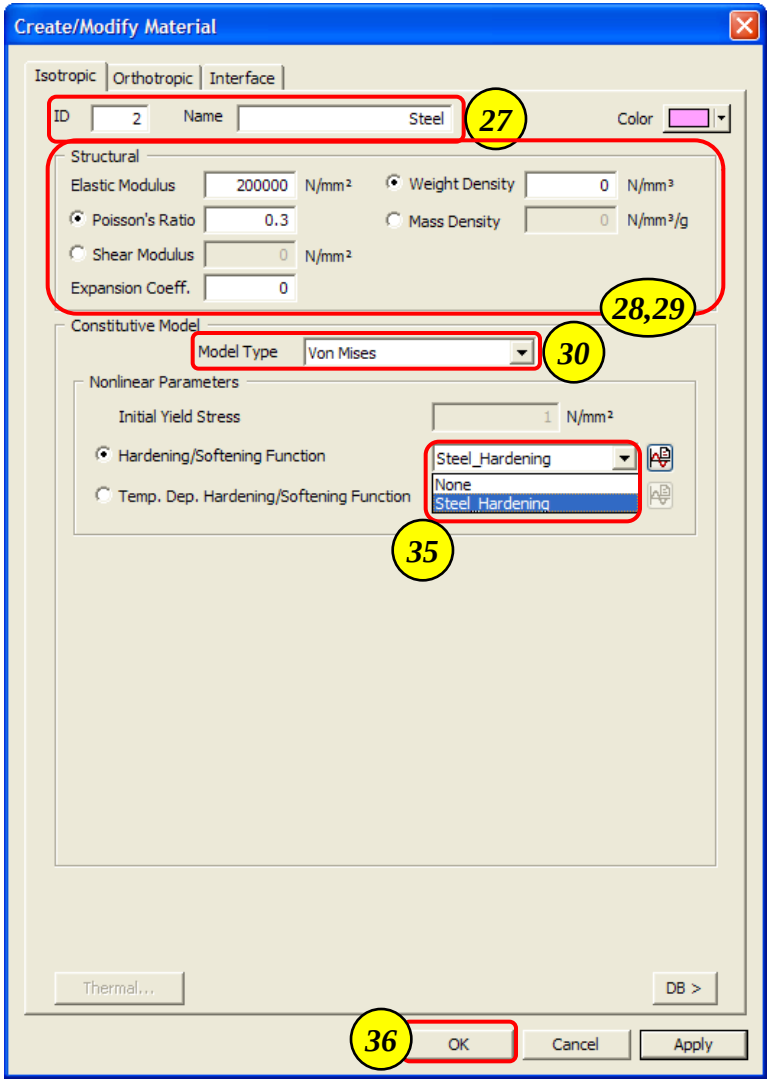
Step 4.

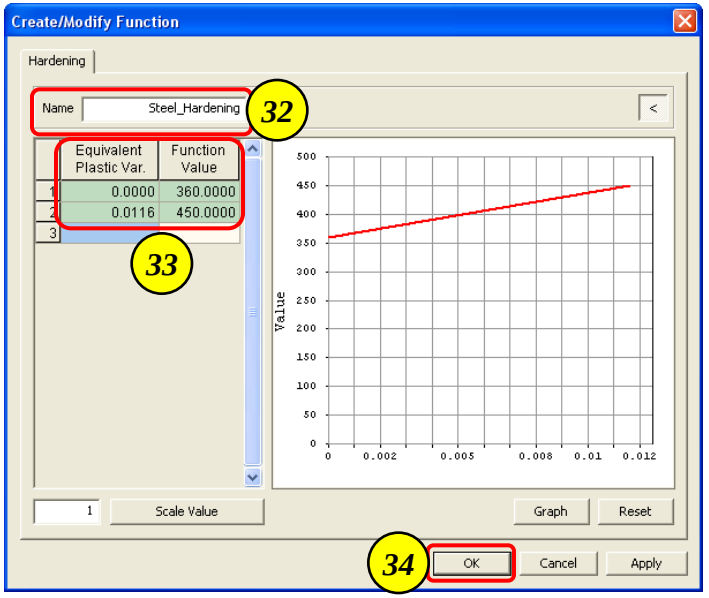

Create/Modify Material
Isotropic
Orthotropic
Interface
ID
1
Name
Concrete
4
Color
Structural
Elastic Modulus
28000
N/mm²
Weight Density
0
N/mm³
Poisson's Ratio
0.3
Mass Density
0
N/mm³/g
Shear Modulus
0
N/mm²
Expansion Coeff.
0
5,6
Constitutive Model
Model Type
Total Strain Crack
7
Crack Model
Fixed
Rotating
Stiffness
Tangent
Secant
Lateral Crack Effect
None
Vecchio and Collins
Confinement Effect
None
Selby and Vecchio
Basic Properties
Direct Input
Using Code
8~12
Tension Function
Hordijk
Compression Function
Thorenfeldt
Shear Function
None
25
Thermal...
DB >
OK
Cancel
26
Apply

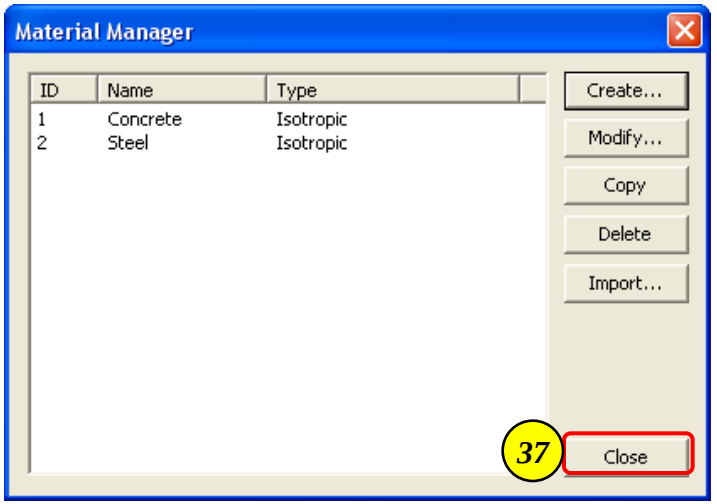

Create/Modify Function
Total Strain Crack
14
Name
Hordijk
Model Type
Tension
15
Function Type
Hordijk
Parameters
Fct
2
N/mm²
Gf
0.009735
N/mm
h
25
mm
16~18
Value
Zero
19
OK
Cancel
Apply


Create/Modify Function
Total Strain Crack
21
Name
Thorenfeldt
Model Type
Compression
22
Function Type
Thorenfeldt
Parameters
Fc
28
N/mm²
23
Value
Zero
24
OK
Cancel
Apply

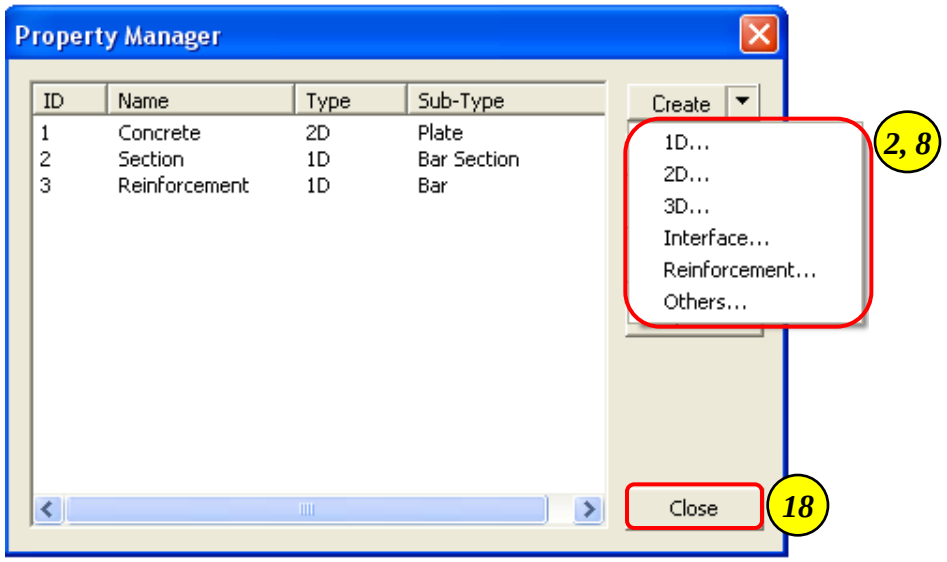
Step 5.


Create/Modify Material
Isotropic
Orthotropic
Interface
ID
2
Name
Steel
27
Color
Structural
Elastic Modulus
200000
N/mm²
Weight Density
0
N/mm³
Poisson's Ratio
0.3
Mass Density
0
N/mm³/g
Shear Modulus
0
N/mm²
Expansion Coeff.
0
28,29
Constitutive Model
Model Type
Von Mises
30
Nonlinear Parameters
Initial Yield Stress
1
N/mm²
Hardening/Softening Function
Steel_Hardening
None
Steel_Hardening
Temp. Dep. Hardening/Softening Function
35
Thermal...
DB >
36
OK
Cancel
Apply


Create/Modify Function
Hardening
Name
Steel_Hardening
32
Equivalent Plastic Var.
Function Value
0.0000
360.0000
0.0116
450.0000
33
Value
Scale Value
Graph
Reset
34
OK
Cancel
Apply

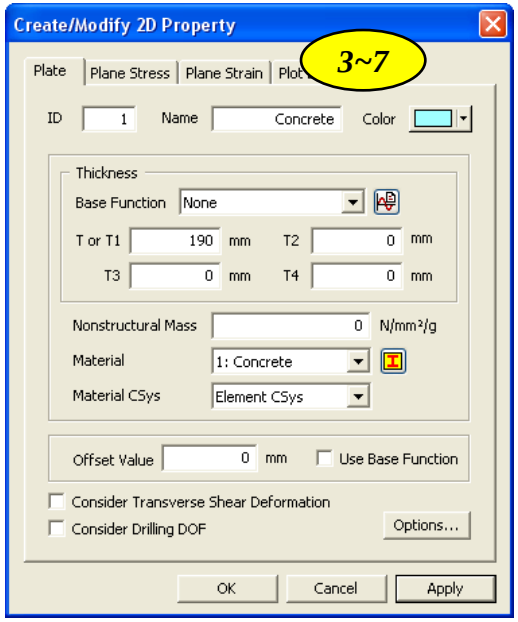

Material Manager
ID
Name
Type
1
Concrete
Isotropic
2
Steel
Isotropic
Create...
Modify...
Copy
Delete
Import...
37
Close

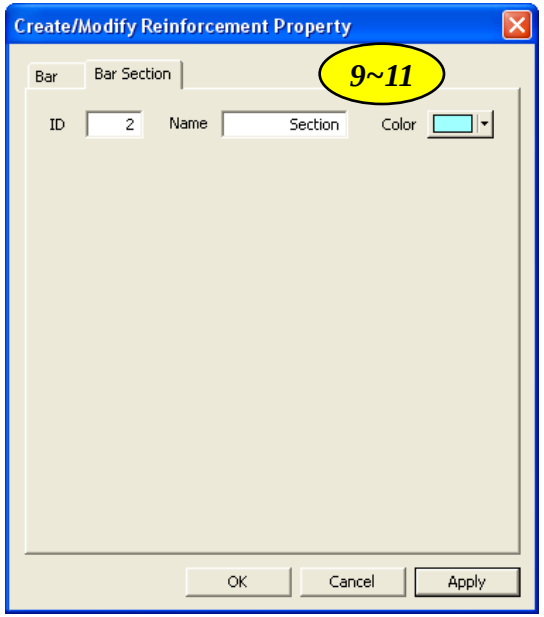
Step 6.

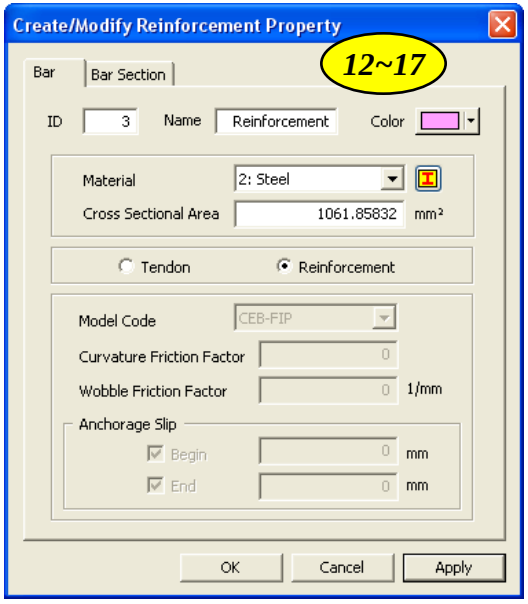


1. Analysis > Property...
2. Create 2D…
3. ID : 1 , Name : Concrete
4. Thickness: 190 mm
5. Material : (1: Concrete)
6. Do not Consider Transverse Shear Deformation
7. Click on [OK] Button
8. Create Reinforcement…
9. Click on Bar Section tab
10. ID : 2 , Name : Section
11. Click on [Apply] Button
12. Click on Bar tab
13. ID : 3 , Name : Reinforcement
14. Material : (2: Steel)
15. Cross Sectional Area: 1061.85832 mm^2
16. Select Reinforcement
17. Click on [OK] Button
18. Click on [Close] Button

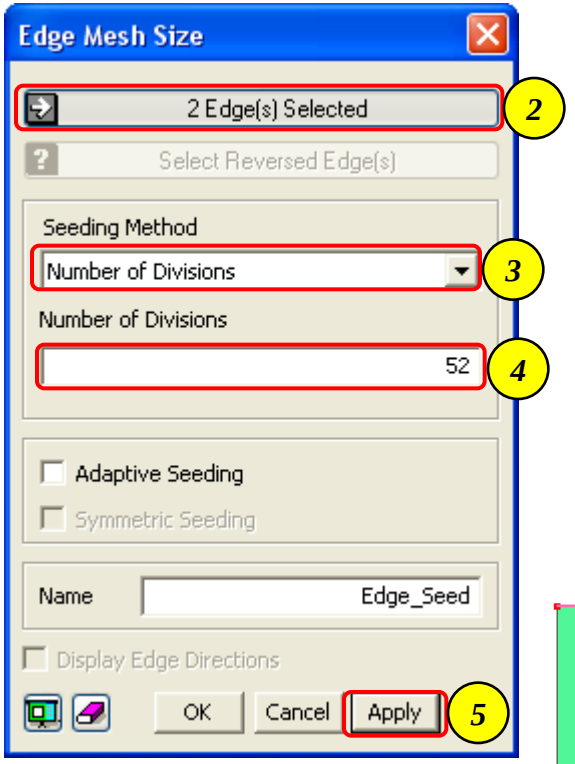
Step 7.


Create/Modify 2D Property
3~7
Plate
Plane Stress
Plane Strain
ID 1
Name Concrete
Color
Thickness
Base Function None
T or T1 190 mm
T2 0 mm
T3 0 mm
T4 0 mm
Nonstructural Mass 0 N/mm²/g
Material 1: Concrete
Material CSys Element CSys
Offset Value 0 mm
Use Base Function
Consider Transverse Shear Deformation
Consider Drilling DOF
Options...
OK
Cancel
Apply

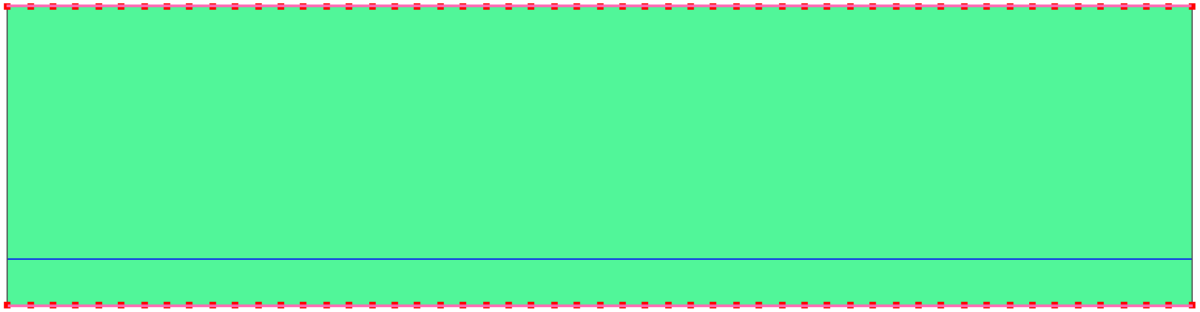

Create/Modify Reinforcement Property
9~11
Bar
Bar Section
ID 2
Name Section
Color
OK
Cancel
Apply


Create/Modify Reinforcement Property
12~17
Bar
Bar Section
ID 3
Name Reinforcement
Color
Material 2: Steel
Cross Sectional Area 1061.85832 mm²
Tendon
Reinforcement
Model Code CEB-FIP
Curvature Friction Factor 0
Wobble Friction Factor 0 1/mm
Anchorage Slip
Begin 0 mm
End 0 mm
OK
Cancel
Apply

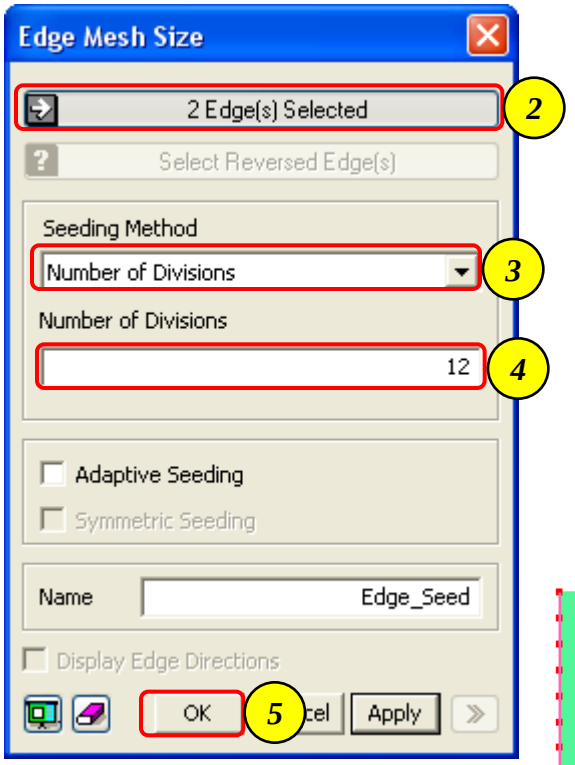
Step 8.



1. *Mesh > Size Control > Along Edge ...*
2. *Select two edges as shown in figure*
3. *Seeding Method: Number of Divisions*
4. *Number of Divisions: 52*
5. *Click on [Apply] Button*

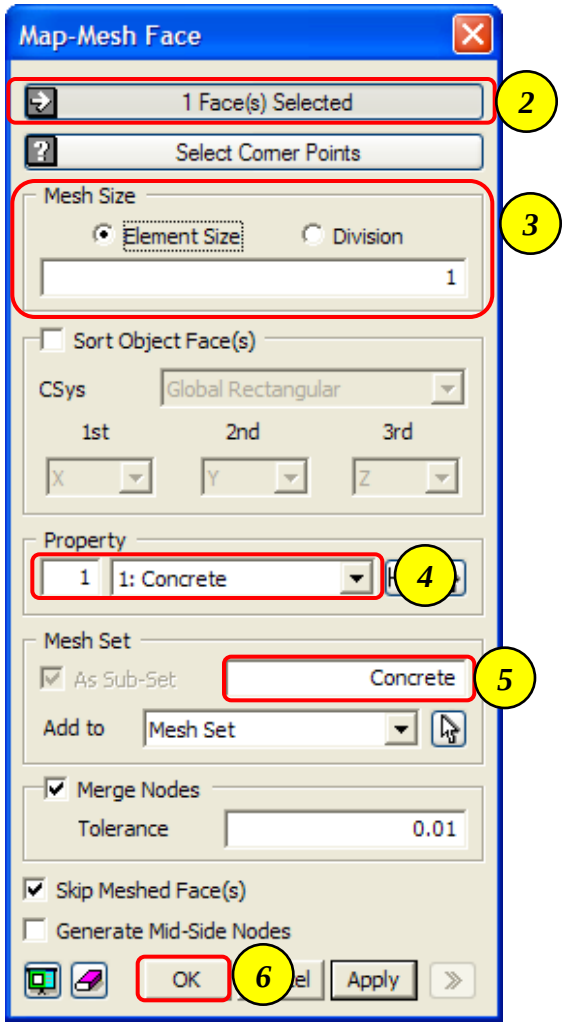


Step 9.

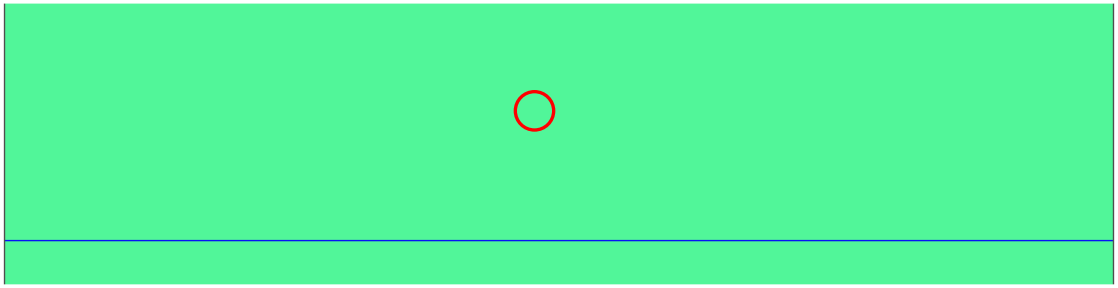


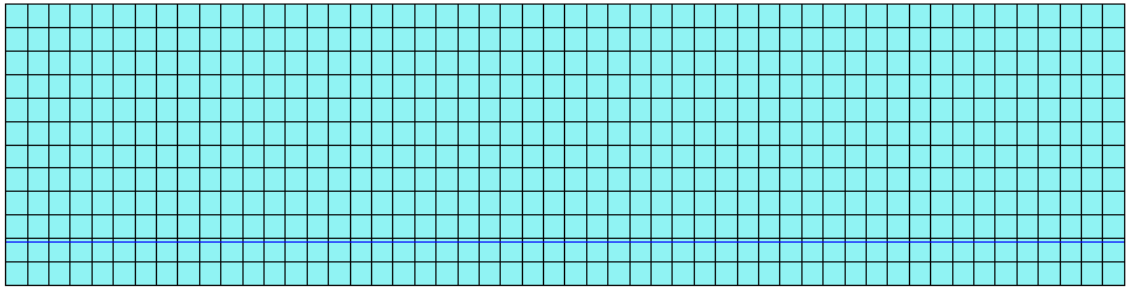
1. Mesh > Size Control > Along Edge ...
2. Select two edges as shown in figure
3. Seeding Method: Number of Divisions
4. Number of Divisions: 12
5. Click on [OK] Button

Step 10.

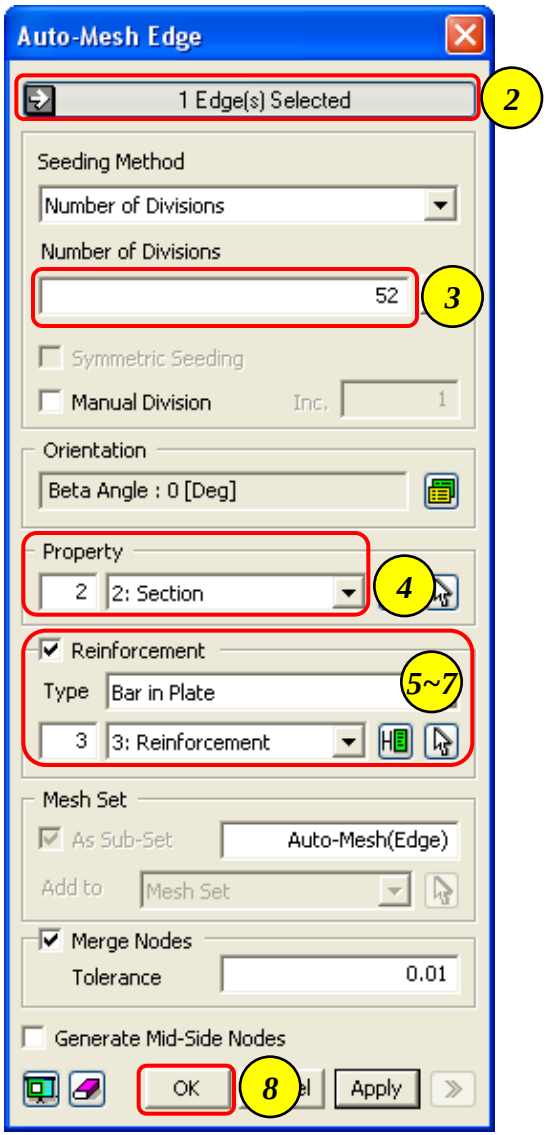


1. *Mesh > Map Mesh > Face ...*
2. *Select the highlighted face*
3. *Element Size: 1*
4. *Property: 1: Concrete*
5. *Mesh Set: Concrete*
6. *Click on [OK] Button*

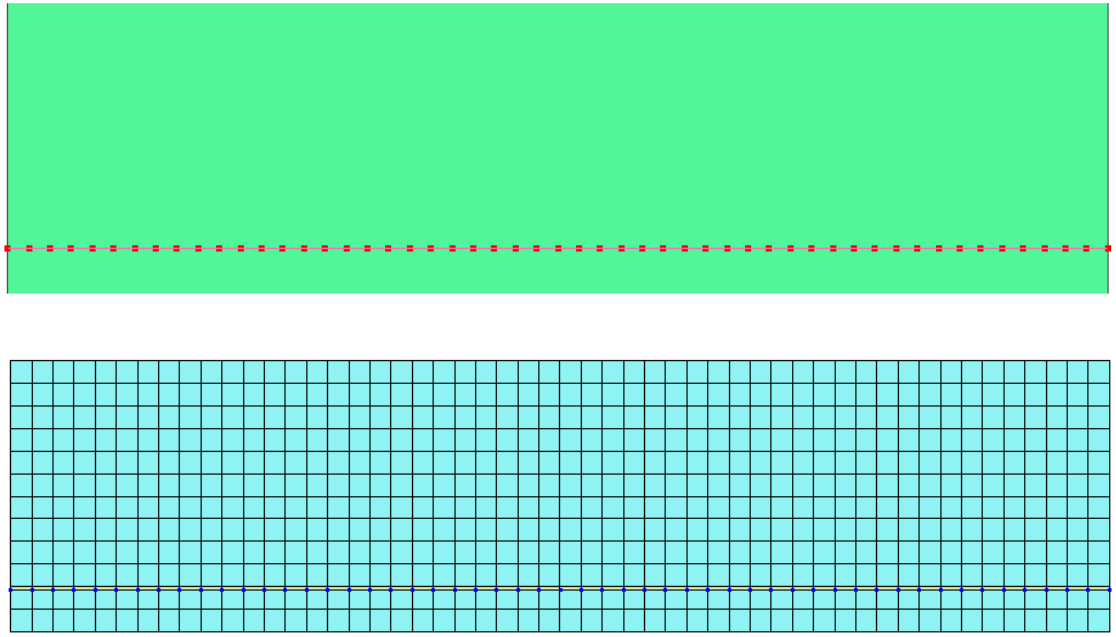




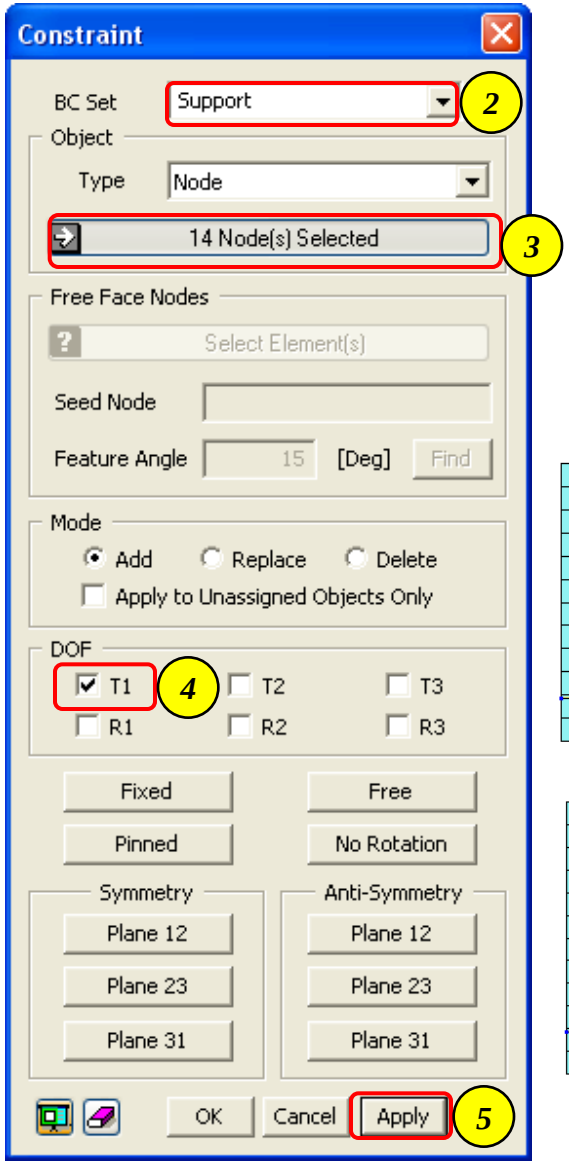
Step 11.



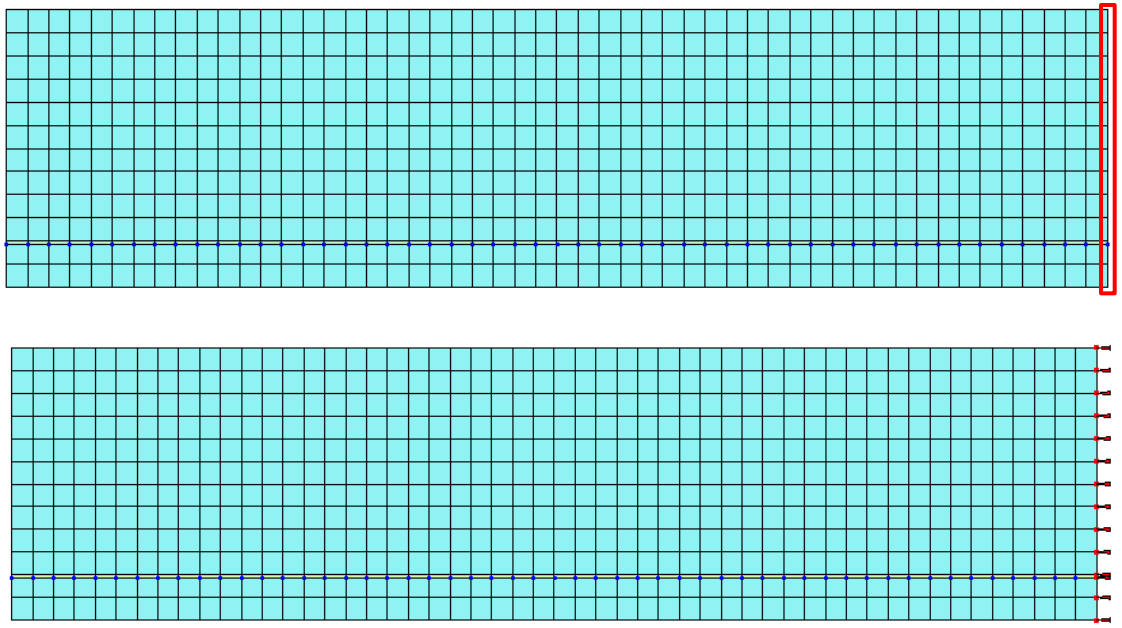
1. *Mesh > Auto Mesh > Edge ...*
2. *Select the highlighted edge*
3. *Number of Divisions: 52*
4. *Property: 2: Section*
5. *Tick on Reinforcement*
6. *Type: Bar in Plate*
7. *Property: 3: Reinforcement*
8. *Click on [OK] Button*



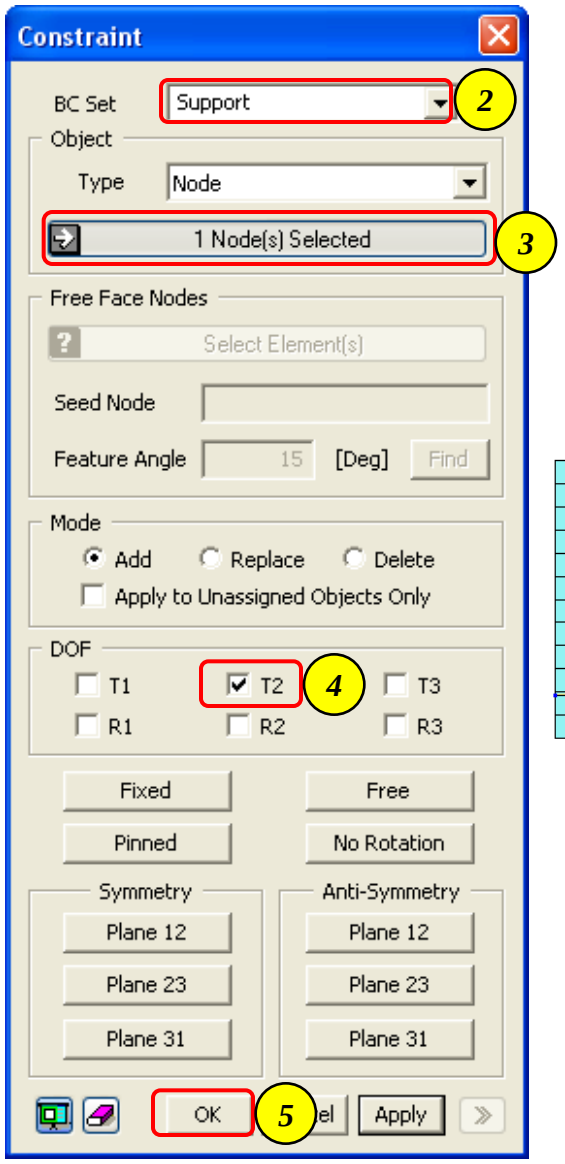
Step 12.



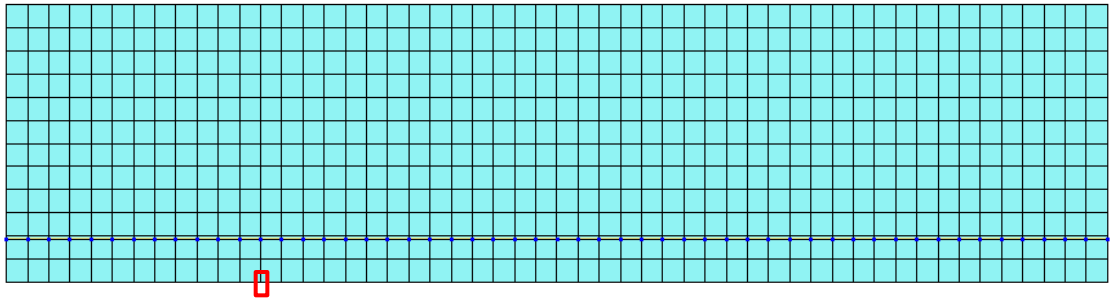
1. *Analysis > BC > Constraint …*
2. *BC Set : Support*
3. *Select 14 Nodes (See Figure)*
4. *Click "T1"*
5. *Click [Apply] Button*

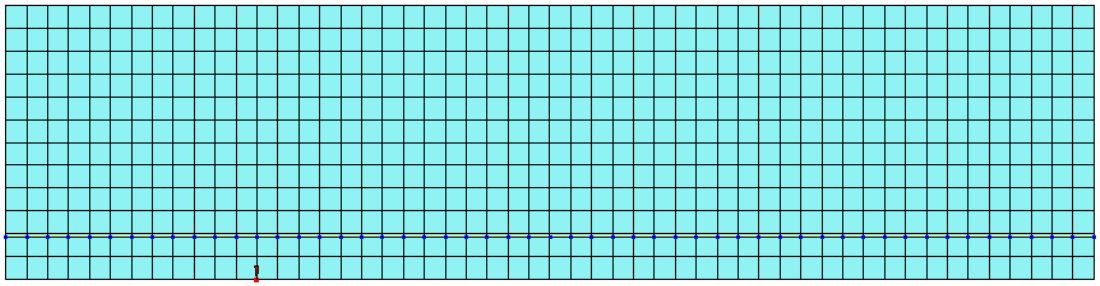


Step 13.

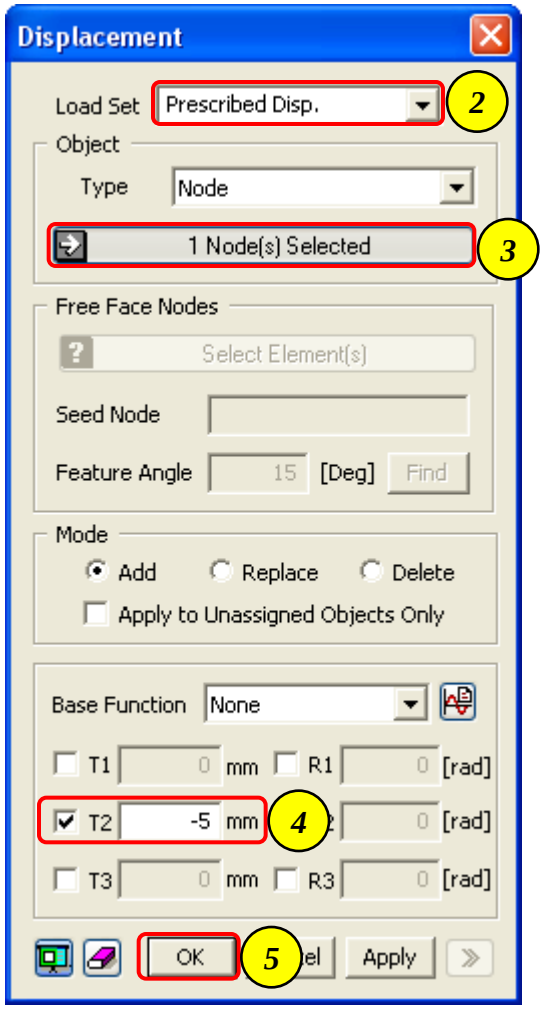


1. *Analysis > BC > Constraint …*
2. *BC Set : Support*
3. *Select 1 Nodes (See Figure)*
4. *Click "T2"*
5. *Click [OK] Button*

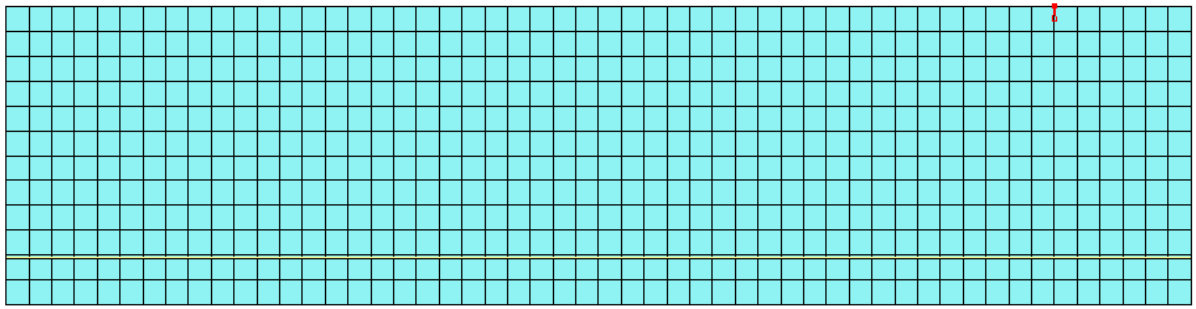




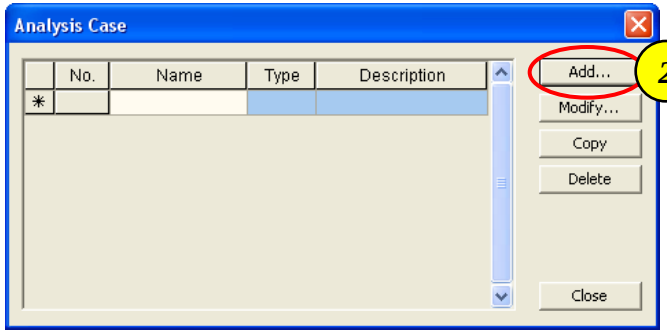
Step 14.



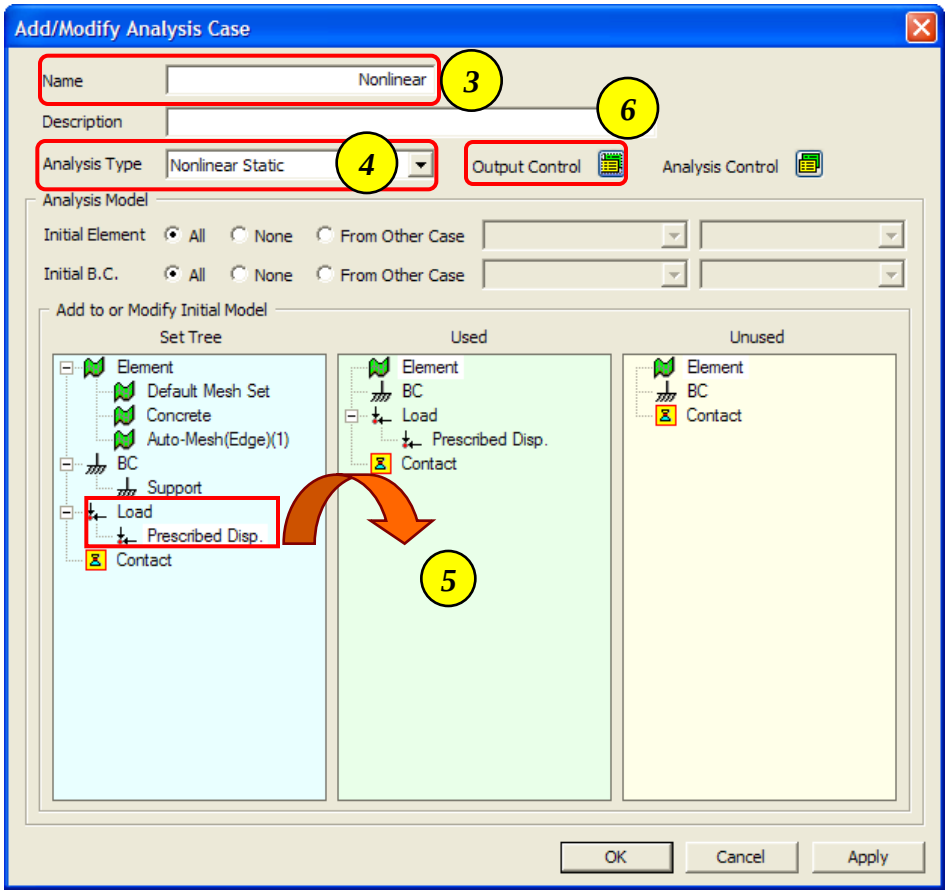
1. *Analysis > Load > Displacement …*
2. *Load Set : Prescribed Disp.*
3. *Select 1 Node (See Figure)*
4. *Click on "T2" : -5 m*
5. *Click [OK] Button*

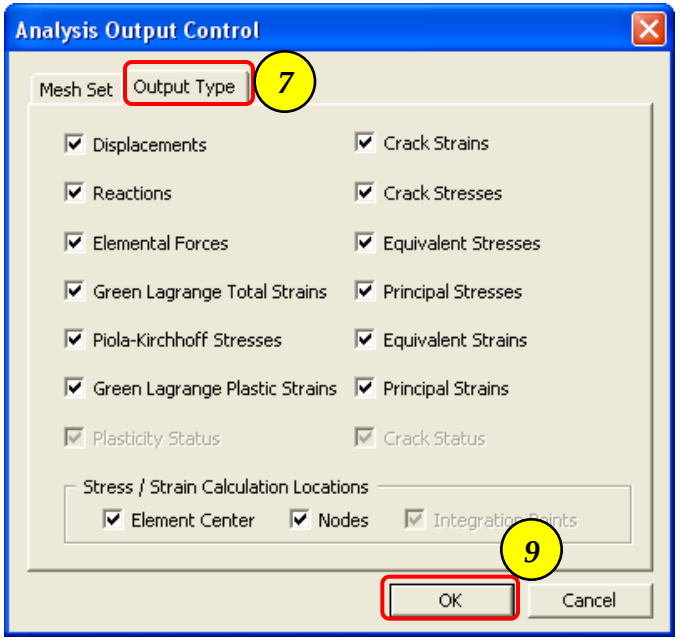


Step 15.



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 16.

1. Click on Analysis Control
2. Select Material Nonlinear
3. Type of Iteration Scheme: Newton Raphson
4. Number of Load Steps: 100
5. Maximum Number of Iterations/Load Step: 200
6. Displacement Norm: 0.01
7. Tick on "Terminate analysis …"
8. Click [OK] Button
9. Click [OK] Button