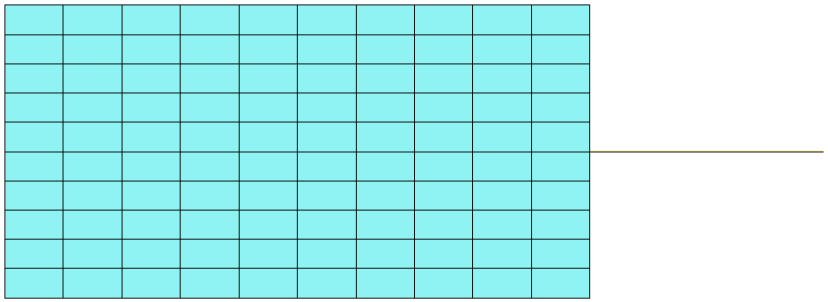
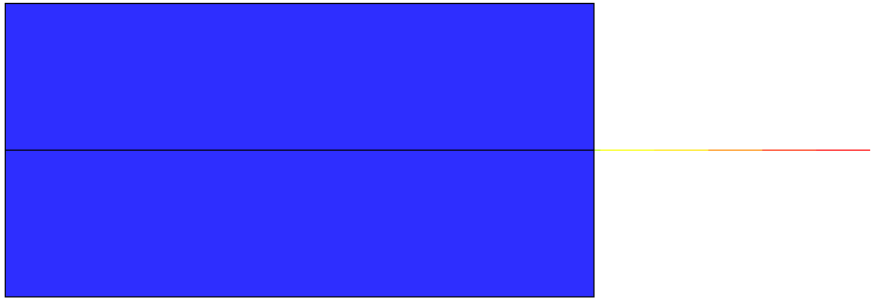
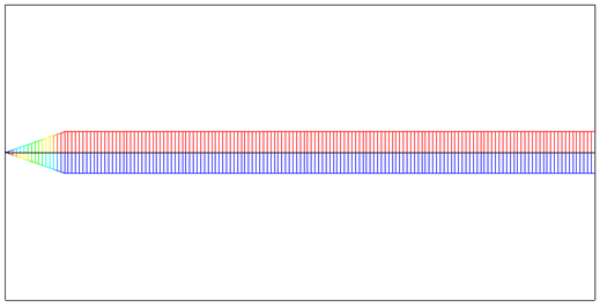
NL-1. An embedded bar in a concrete block (pull out test for bond-slip interface model)





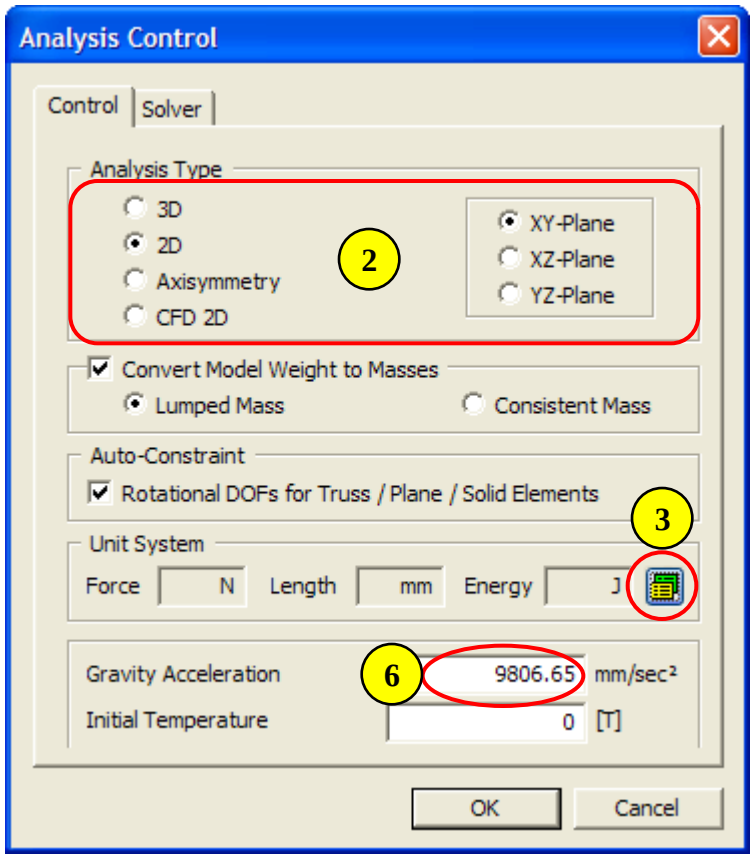


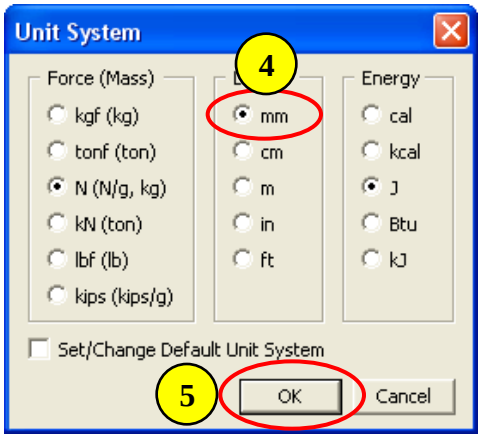
Overview

- **2D Nonlinear Analysis**
- **Model**
  - *An embedded bar in a concrete block*
  - *Bond slip nonlinear interface*
  - *Unit : N, mm*
  - *Isotropic Elastic Material*
- **Load & Boundary Condition**
  - *Prescribed displacement*
  - *Constraint*
- **Result Evaluation**
  - *Displacement*
  - *Interface stress*

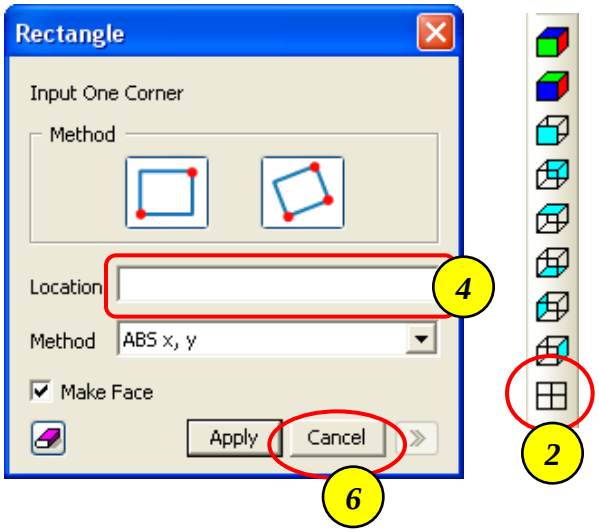
Step 1.

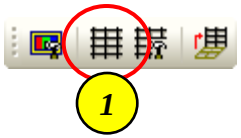
1. *Analysis > Analysis Control – Control tab*
2. *Analysis Type : 2D and XY - Plane*
3. *Click*  *Button (Unit System)*
4. *Length : mm*
5. *Click on* **[OK]** *Button*
6. *Click on* **[OK]** *Button*



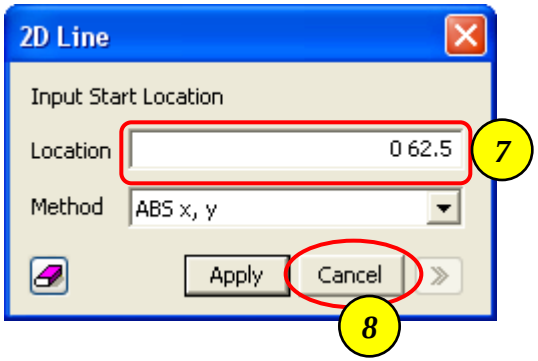


Step 2.

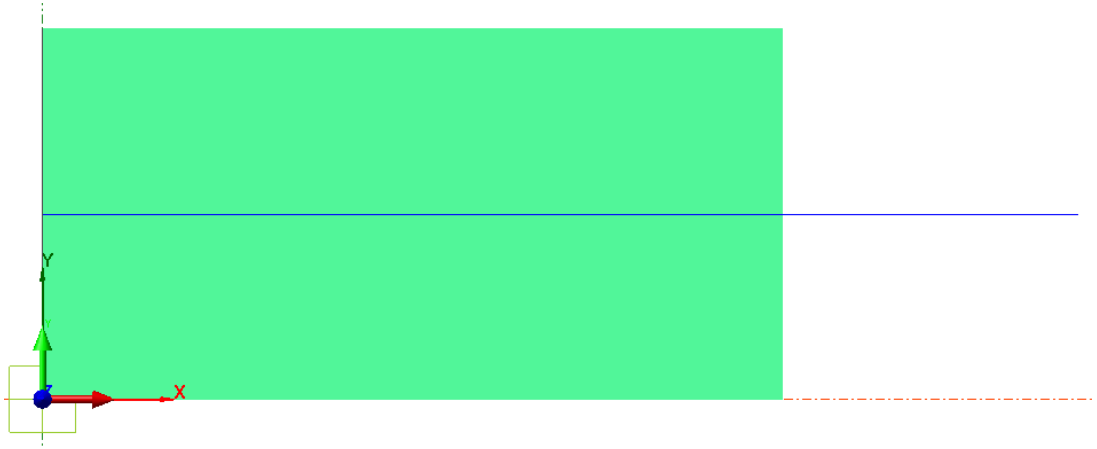




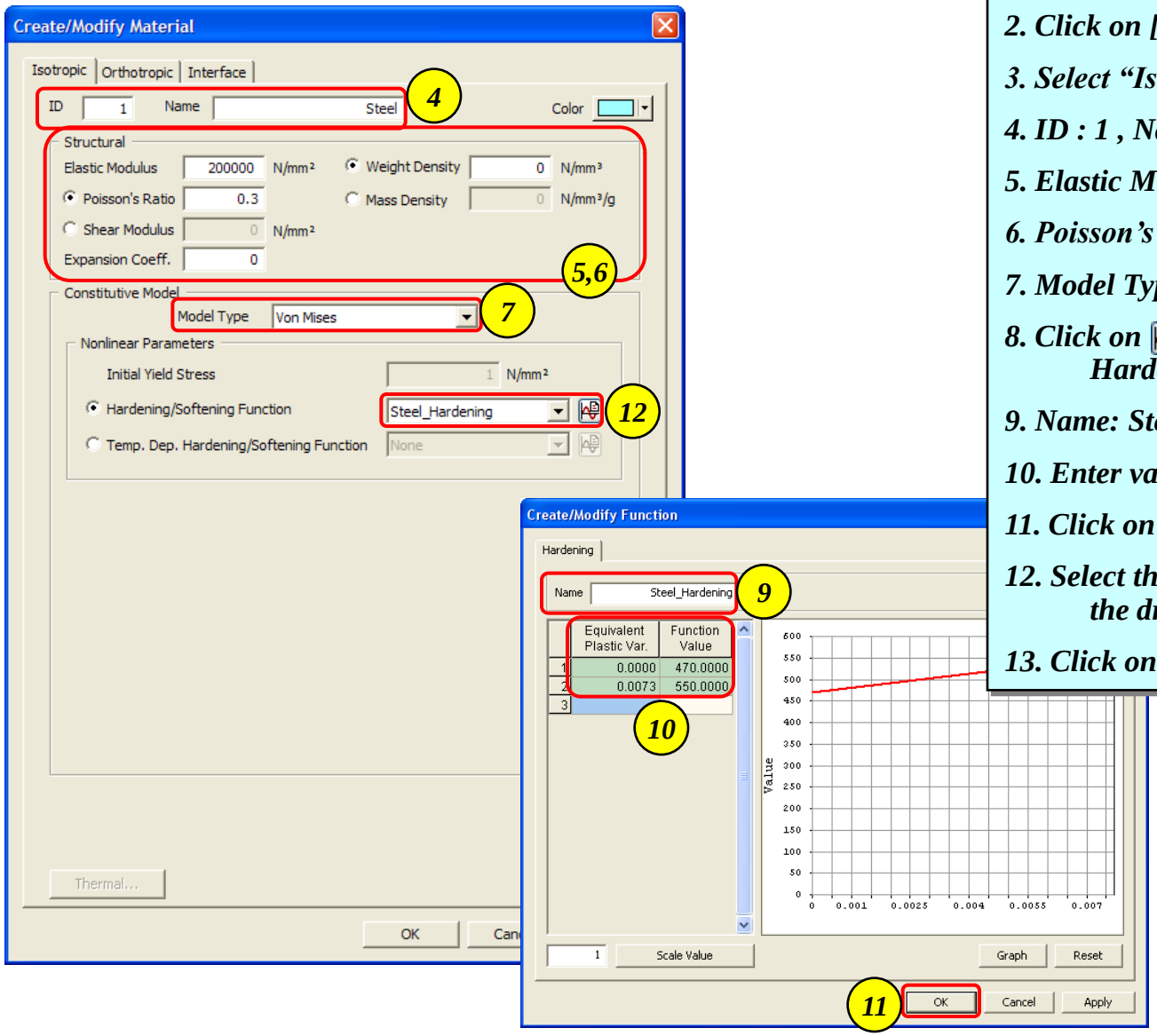
1. *Toggle off "Toggle Grid"*
2. *Click "Normal View"*
3. *Geometry > Curve > Create on WP > Rectangle (Wire)...*
4. *Location : (0), <250, 125>*
5. *Click on [Cancel] Button*
6. *Geometry > Curve > Create on WP >Line ...*
7. *Location : (0, 62.5), <350, 0>*
8. *Click on [Cancel] Button*



- *() : "ABS x, y" , <> : "REL dx, dy"*
  (0) same as (0, 0)
- *[Esc] as shortcut for [Cancel]*



Step 3.

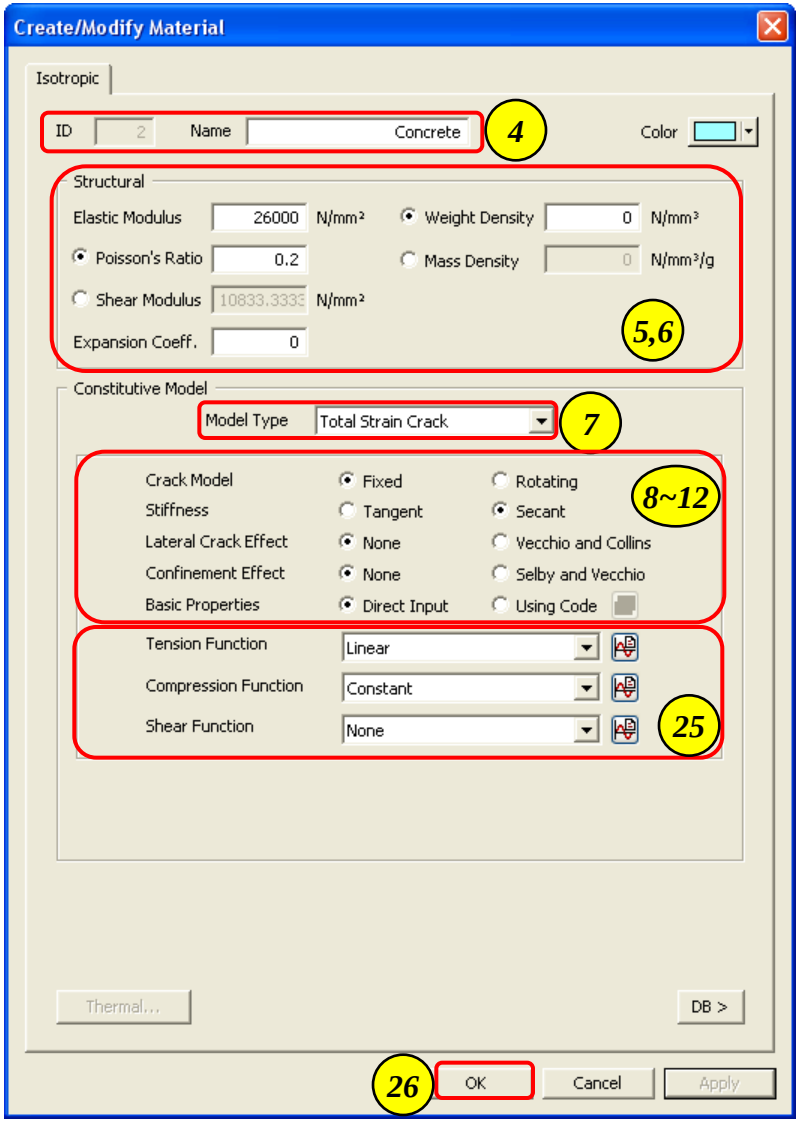


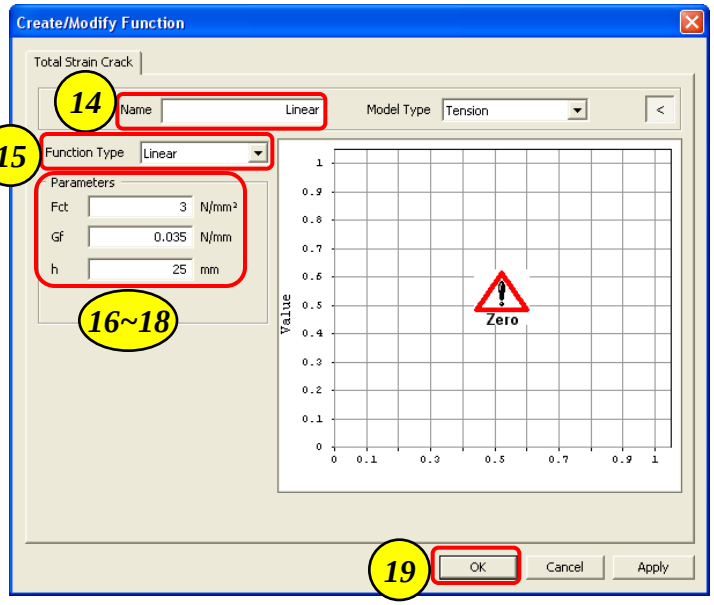
1. *Analysis > Material …*
2. *Click on [Create] Button*
3. *Select "Isotropic" tab*
4. *ID : 1 , Name : Steel*
5. *Elastic Modulus : 200000 N/mm²*
6. *Poisson's Ratio : 0.3*
7. *Model Type : Von Mises*
8. *Click on to define Hardening/Softening Function*
9. *Name: Steel_Hardening*
10. *Enter values as shown in the picture*
11. *Click on [OK] Button*
12. *Select the hardening function from the drop list*
13. *Click on [Apply] Button*

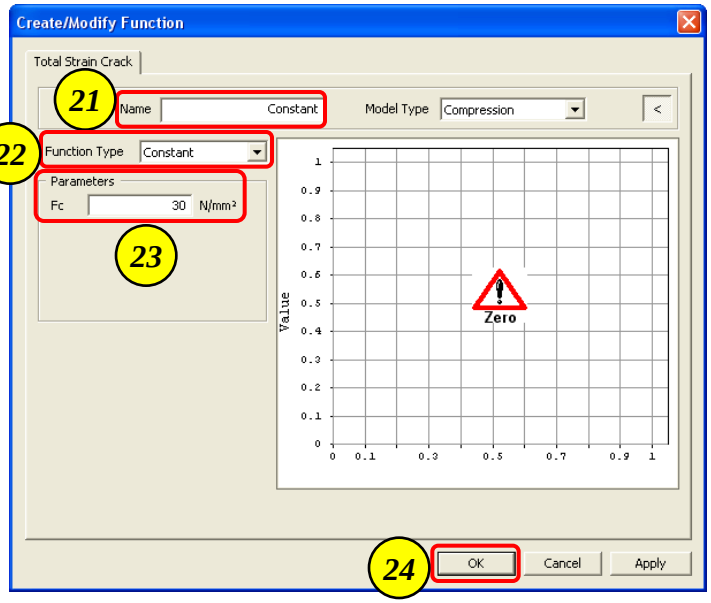
Step 4.

1. Analysis > Material …
2. Click [Create] Button
3. Select “Isotropic” tab
4. ID : 2 , Name : Concrete
5. Elastic Modulus : 26000 N/mm^2
6. Poisson’s Ratio : 0.2
7. Model Type : Total Strain Crack
8. Crack model: Fixed
9. Stiffness: Secant
10. Lateral Crack Effect: None
11. Confinement Effect: None
12. Basic Properties: Direct Input
13. Click on  to define Tension Function
14. Name: Linear
15. Function Type: Linear
16. Fct = 3 N/mm^2
17. Gf = 0.035 N/mm
18. h = 25 mm
19. Click on [OK] Button
20. Click on  to define Compression Function
21. Name: Constant
22. Function Type: Constant
23. Fc = 30 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 : 3 , Name : Interface
28. Interface Nonlinearities: Bond Slip
29. Normal Stiffness Modulus (Kn): 260000 N/mm^3
30. Shear Stiffness Modulus (Kt): 26000 N/mm^3
31. Select Cubic Function
32. Constant (c): 15
33. Shear Slip: 0.1 mm
34. Click on [OK] Button
35. Click on [Close] Button

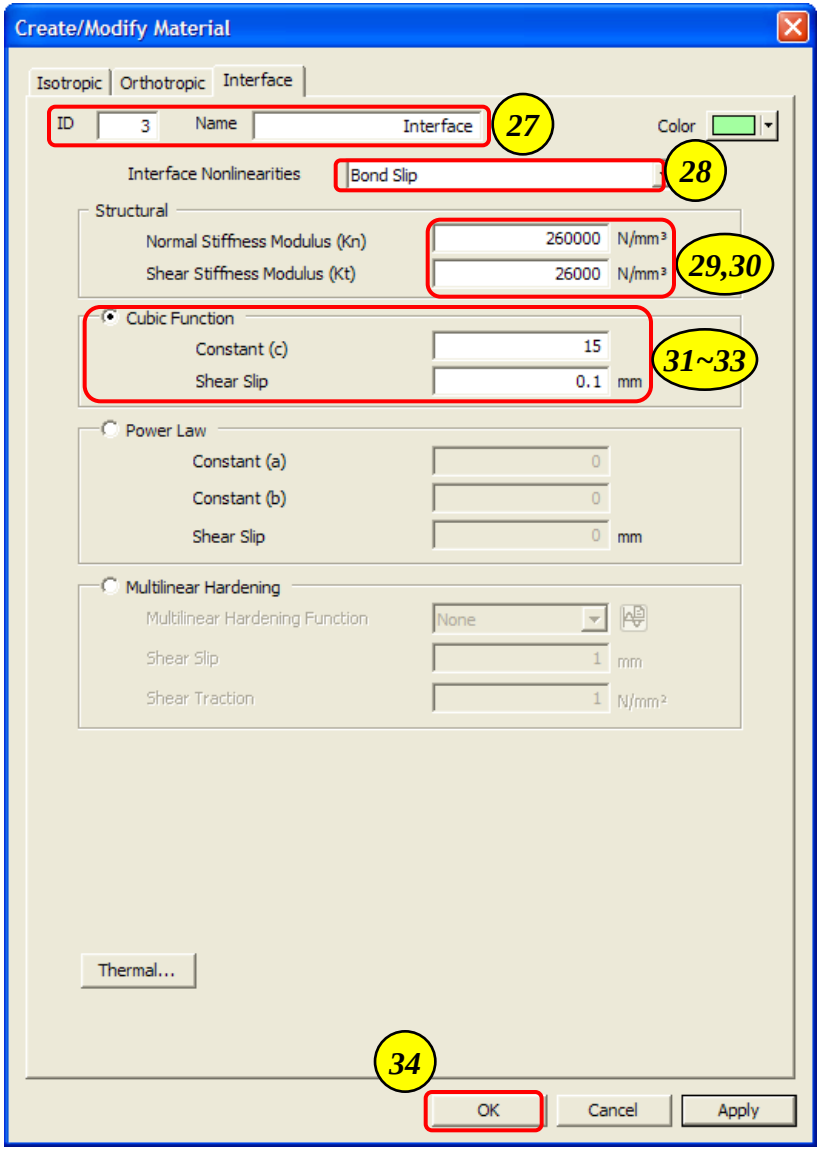
Step 5.

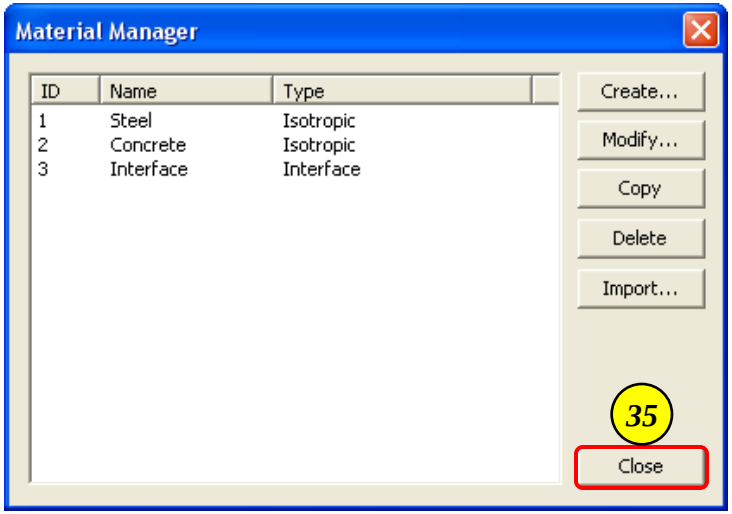

Create/Modify Material
Isotropic
ID 2
Name Concrete
4
Color
Structural
Elastic Modulus 26000 N/mm²
Weight Density 0 N/mm³
Poisson's Ratio 0.2
Mass Density 0 N/mm³/g
Shear Modulus 10833.3333 N/mm²
Expansion Coeff. 0
5,6
Constitutive Model
Model Type Total Strain Crack
7
Crack Model Fixed Rotating
Stiffness 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 Linear
Compression Function Constant
Shear Function None
25
Thermal...
DB >
26
OK
Cancel
Apply


Create/Modify Function
Total Strain Crack
14
Name Linear
Model Type Tension
15
Function Type Linear
Parameters
Fct 3 N/mm²
Gf 0.035 N/mm
h 25 mm
16~18
Value
Zero
19
OK
Cancel
Apply

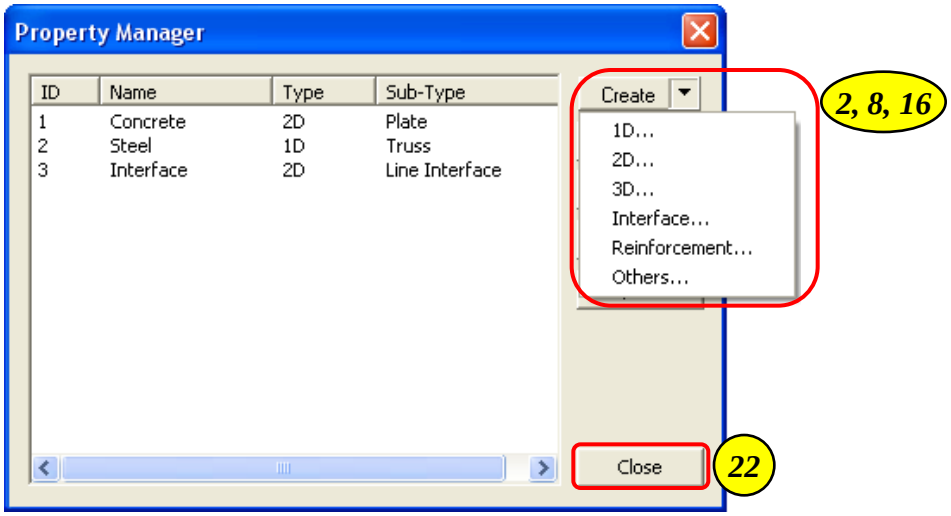

Create/Modify Function
Total Strain Crack
21
Name Constant
Model Type Compression
22
Function Type Constant
Parameters
Fc 30 N/mm²
23
Value
Zero
24
OK
Cancel
Apply

Step 6.


Create/Modify Material
Isotropic
Orthotropic
Interface
ID
3
Name
Interface
27
Color
Interface Nonlinearities
Bond Slip
28
Structural
Normal Stiffness Modulus (Kn)
260000
N/mm³
Shear Stiffness Modulus (Kt)
26000
N/mm³
29,30
Cubic Function
Constant (c)
15
Shear Slip
0.1
mm
31~33
Power Law
Constant (a)
Constant (b)
Shear Slip
mm
Multilinear Hardening
Multilinear Hardening Function
None
Shear Slip
1
mm
Shear Traction
1
N/mm²
Thermal...
34
OK
Cancel
Apply


Material Manager
ID
Name
Type
1
Steel
Isotropic
2
Concrete
Isotropic
3
Interface
Interface
Create...
Modify...
Copy
Delete
Import...
35
Close

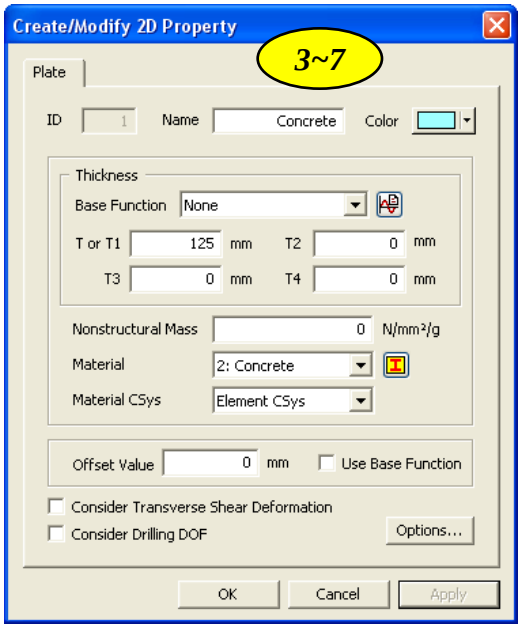
Step 7.

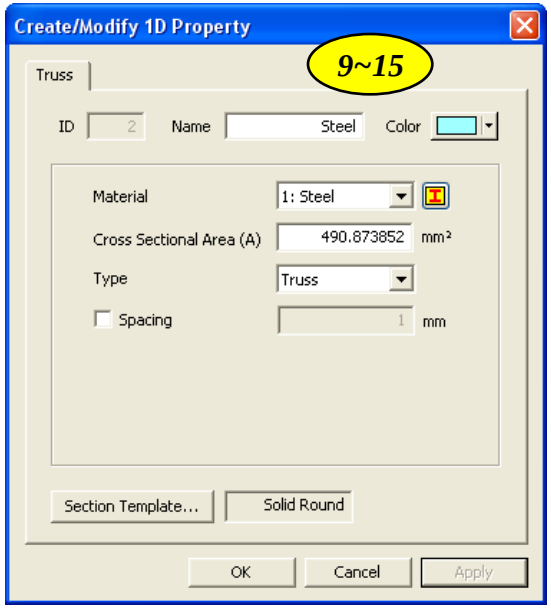


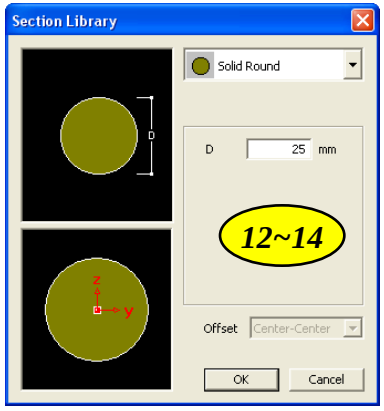
1. Analysis > Property...
2. Create 2D…
3. ID : 1 , Name : Concrete
4. Thickness: 125 mm
5. Material : (2: Concrete)
6. Do not Consider Transverse Shear Deformation
7. Click on [OK] Button
8. Create 1D…
9. ID : 2 , Name : Steel
10. Material : (1: Steel)
11. Click on Section Template …
12. Select Solid Round
13. D = 25 mm
14. Click on [OK] Button
15. Click on [OK] Button

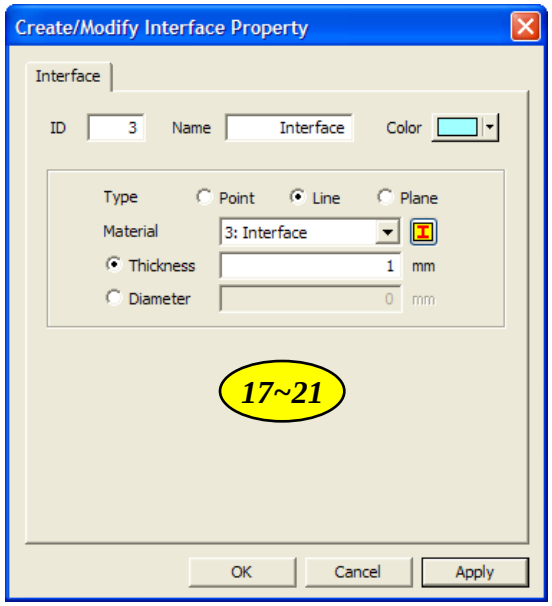
16. Create Interface…
17. ID : 3 , Name : Interface
18. Type: Line
19. Material : (3: Interface)
20. Thickness or Perimeter: 1mm
21. Click on [OK] Button
22. Click on [Close] Button

Step 8.

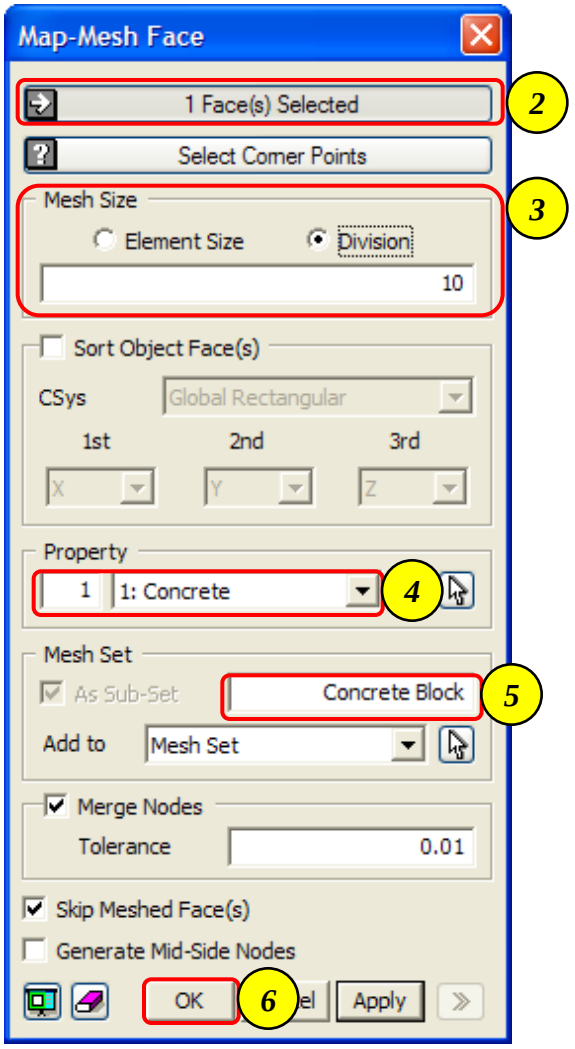

Create/Modify 2D Property
3~7
Plate
ID 1
Name Concrete
Color
Thickness
Base Function None
T or T1 125 mm
T2 0 mm
T3 0 mm
T4 0 mm
Nonstructural Mass 0 N/mm²/g
Material 2: Concrete
Material CSys Element CSys
Offset Value 0 mm
Use Base Function
Consider Transverse Shear Deformation
Consider Drilling DOF
Options...
OK
Cancel
Apply


Create/Modify 1D Property
9~15
Truss
ID 2
Name Steel
Color
Material 1: Steel
Cross Sectional Area (A) 490.873852 mm²
Type Truss
Spacing 1 mm
Section Template...
Solid Round
OK
Cancel
Apply

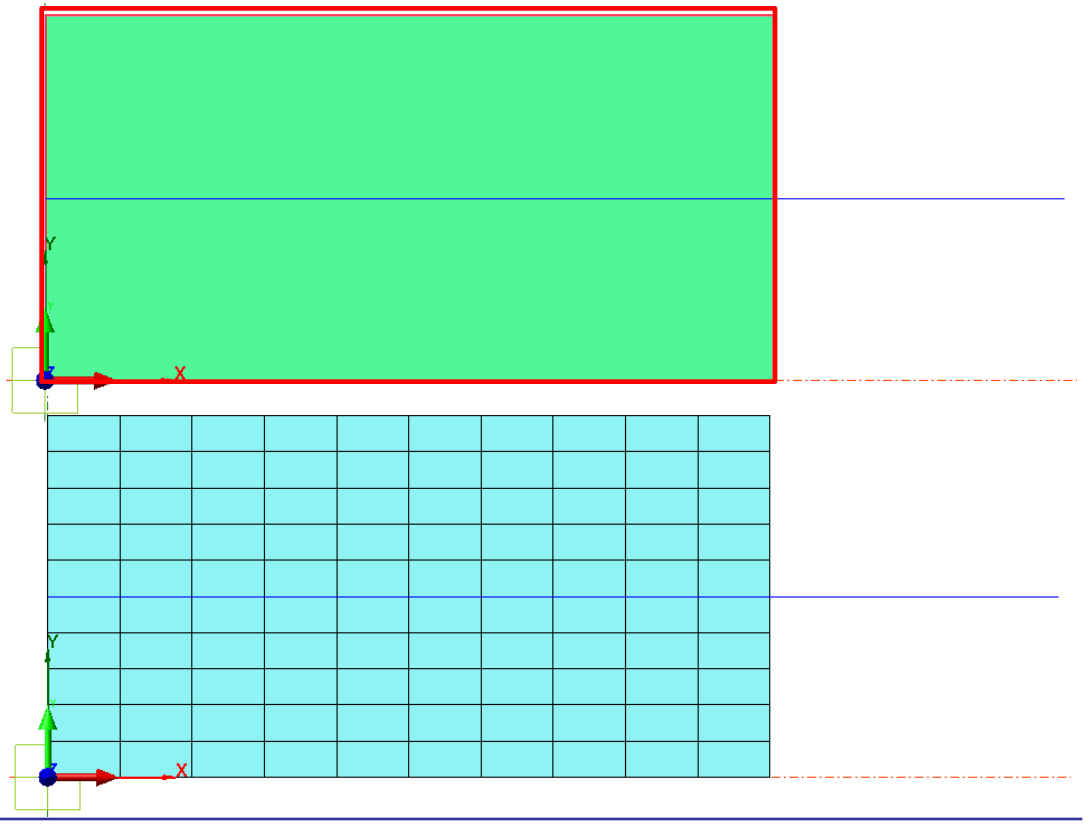

Section Library
Solid Round
D 25 mm
12~14
Offset Center-Center
OK
Cancel


Create/Modify Interface Property
Interface
ID 3
Name Interface
Color
Type Point Line Plane
Material 3: Interface
Thickness 1 mm
Diameter 0 mm
17~21
OK
Cancel
Apply

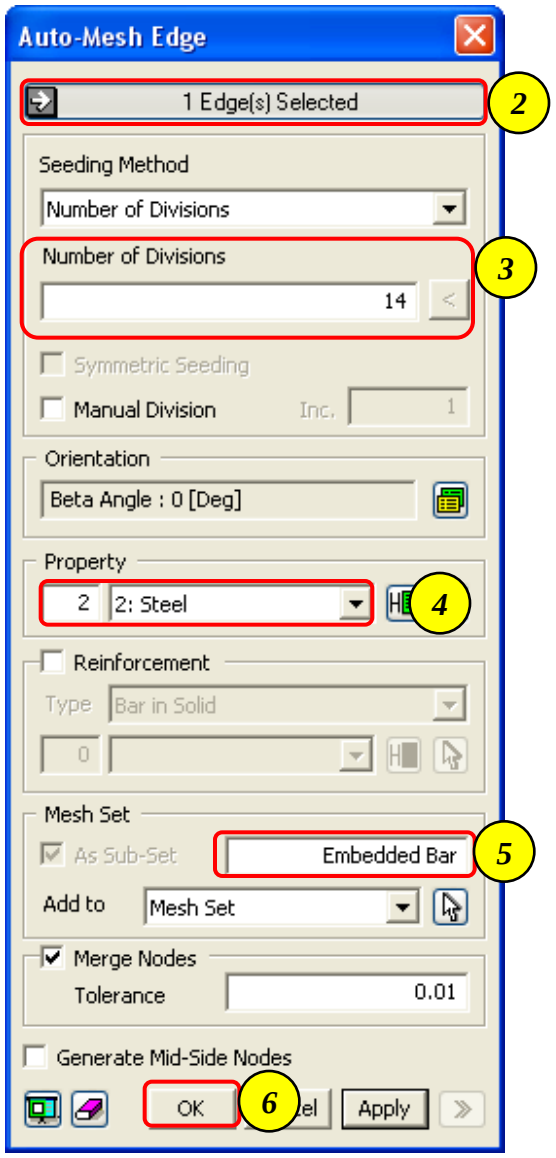
Step 9.



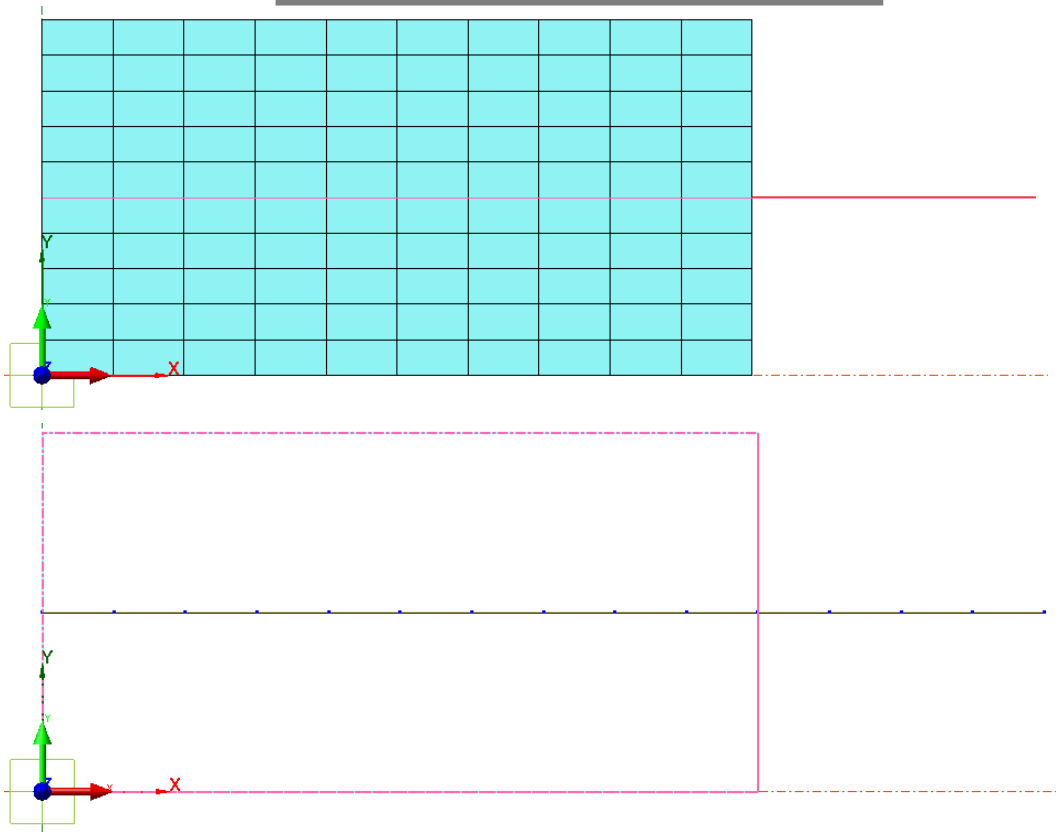
1. *Mesh > Map Mesh > Face ...*
2. *Select the highlighted face (see Figure)*
3. *Mesh Size: Division , 10*
4. *Property: 1 : Concrete*
5. *Mesh Set: Concrete Block*
6. *Click on [OK] Button*



Step 10.

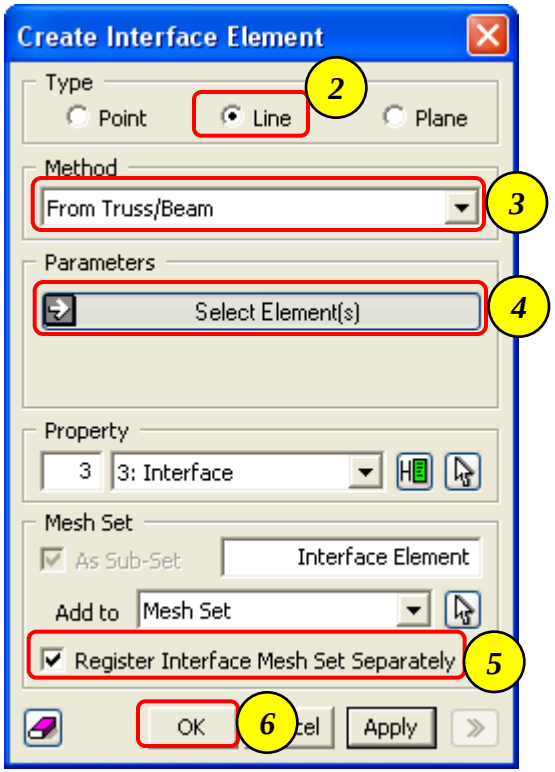


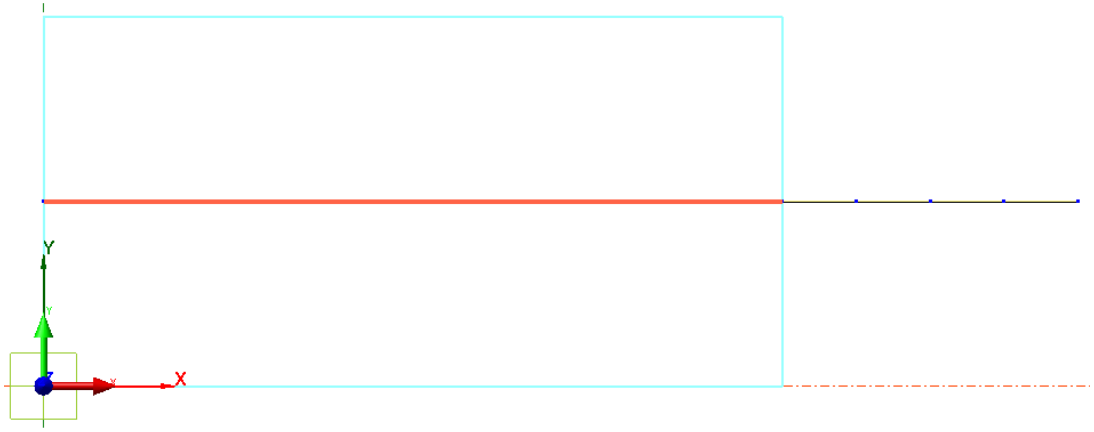
1. *Mesh > Auto Mesh > Edge ...*
2. *Select the highlighted edge (see Figure)*
3. *Number of Divisions: 14*
4. *Property: 2 : Steel*
5. *Mesh Set: Embedded Bar*
6. *Click on [OK] Button*

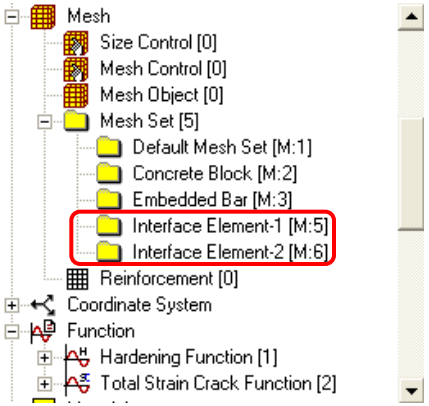


Step 11.

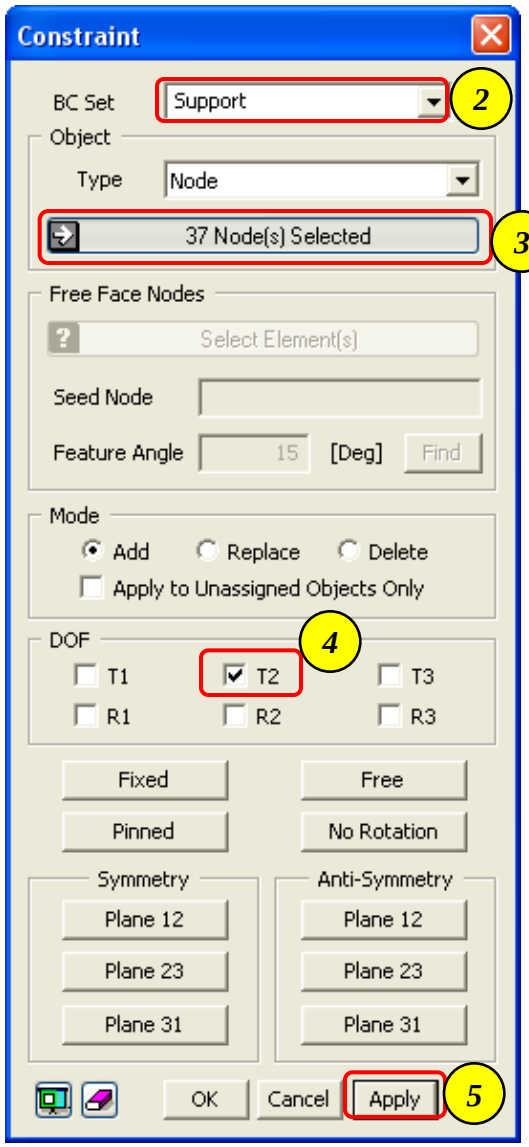
1. *Mesh > Element > Create Interface Element ...*
2. *Type: Line*
3. *Method: From Truss/Beam*
4. *Select the highlighted elements (see Figure)*
5. *Check on Register Interface Mesh Set Separately*
6. *Click on [OK] Button*



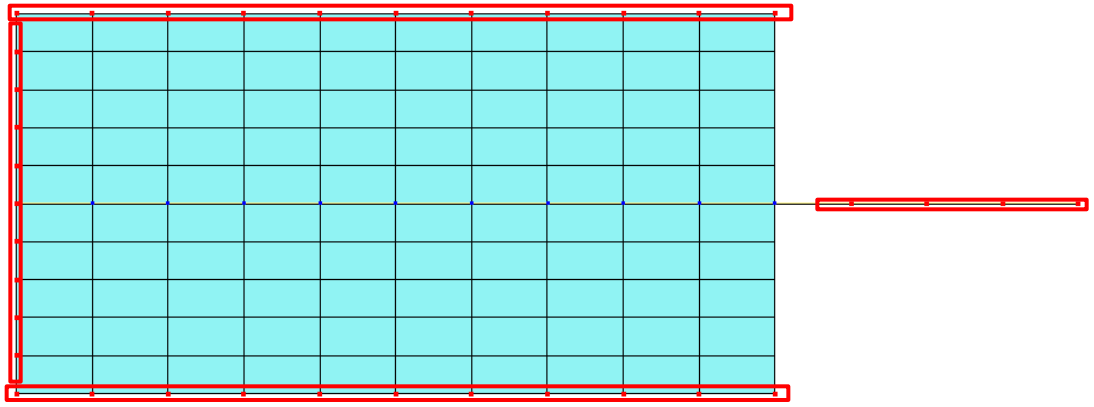




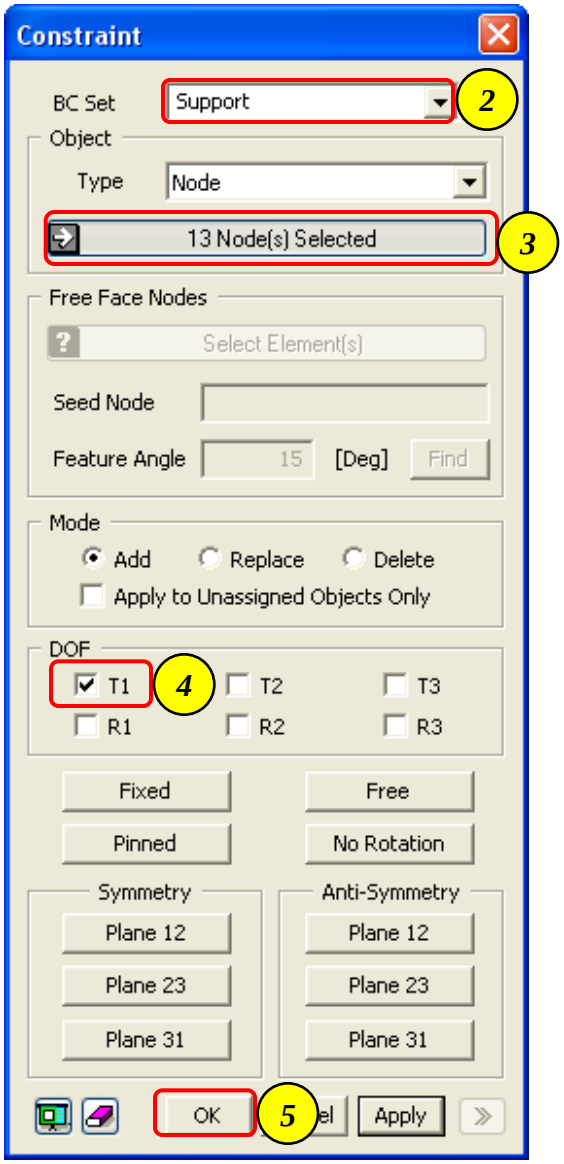
Step 12.



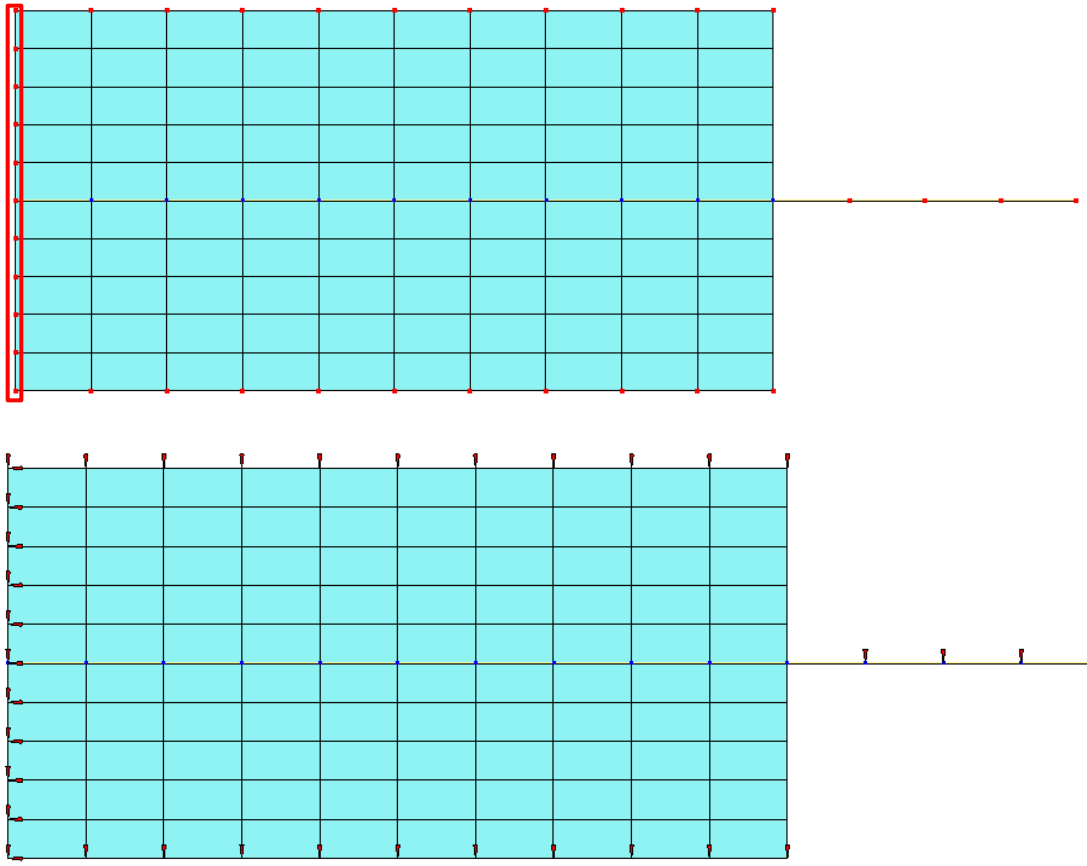
1. *Analysis > BC > Constraint …*
2. *BC Set : Support*
3. *Select 37 Nodes (See Figure)*
4. *Click "T2"*
5. *Click [Apply] Button*



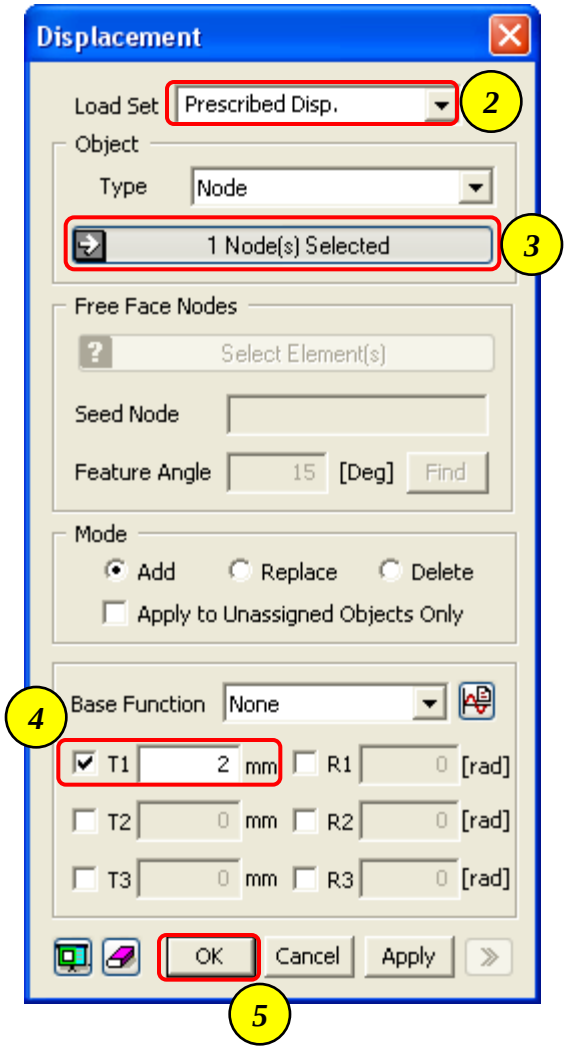
Step 13.



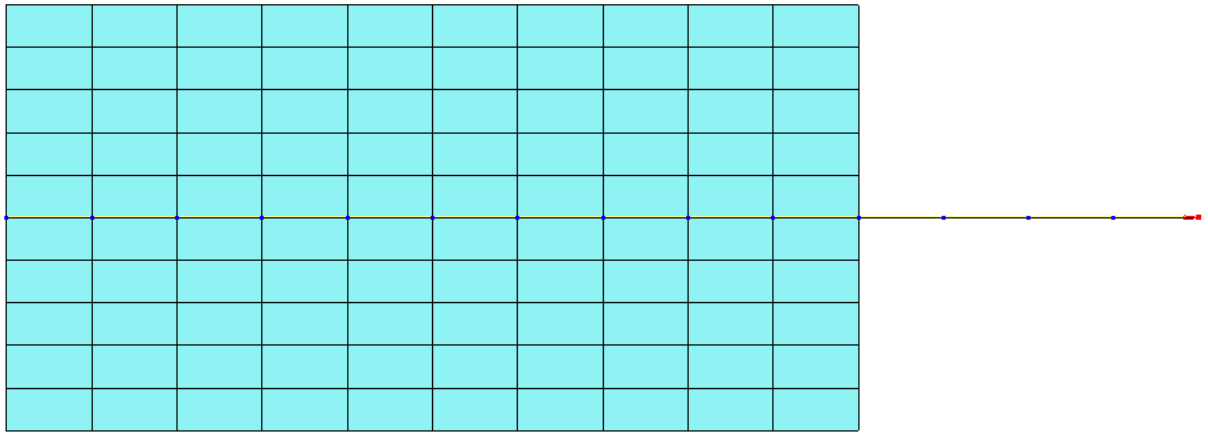
1. *Analysis > BC > Constraint …*
2. *BC Set : Support*
3. *Select 13 Nodes (See Figure)*
4. *Click "T1"*
5. *Click [OK] Button*



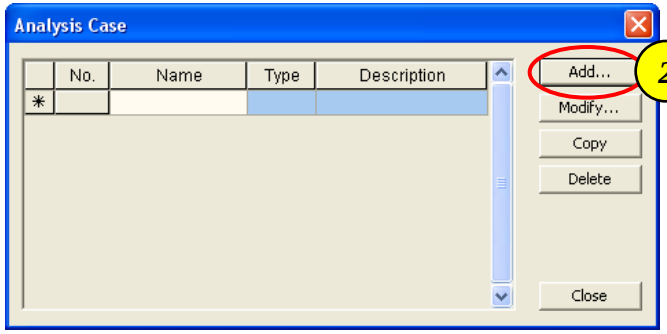
Step 14.



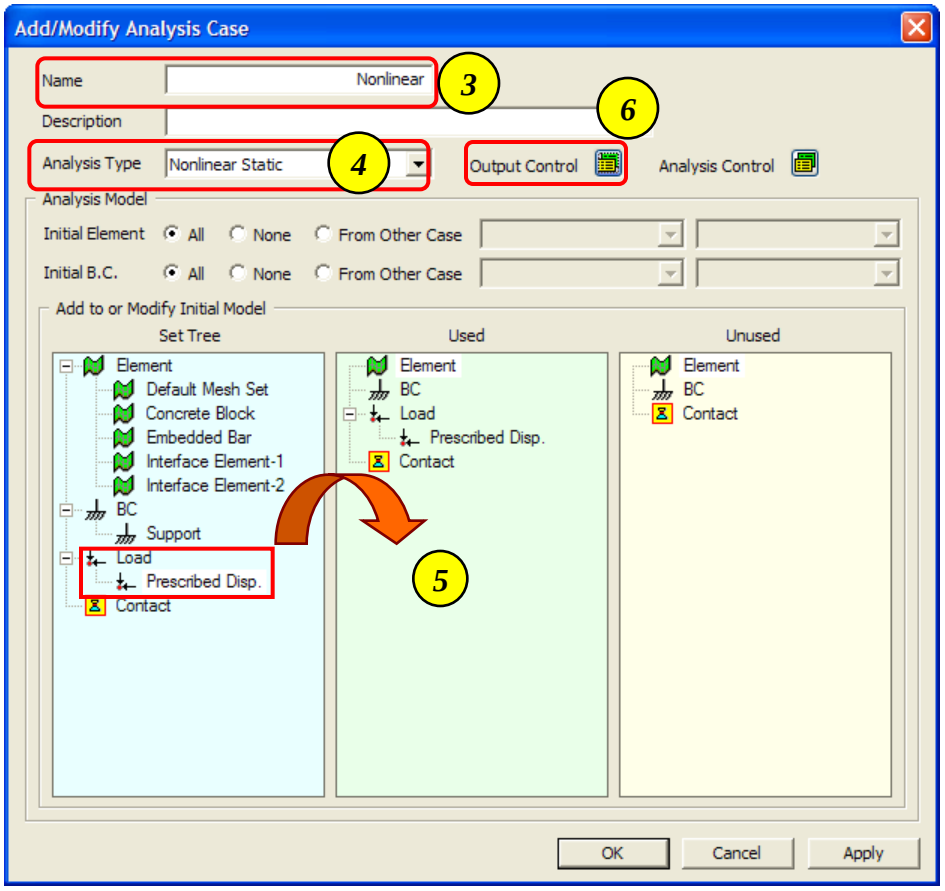
1. *Analysis > Load > Displacement…*
2. *Load Set : Prescribed Disp.*
3. *Select the node as shown in the figure*
4. *T1 = 2 mm*
5. *Click [OK] Button*

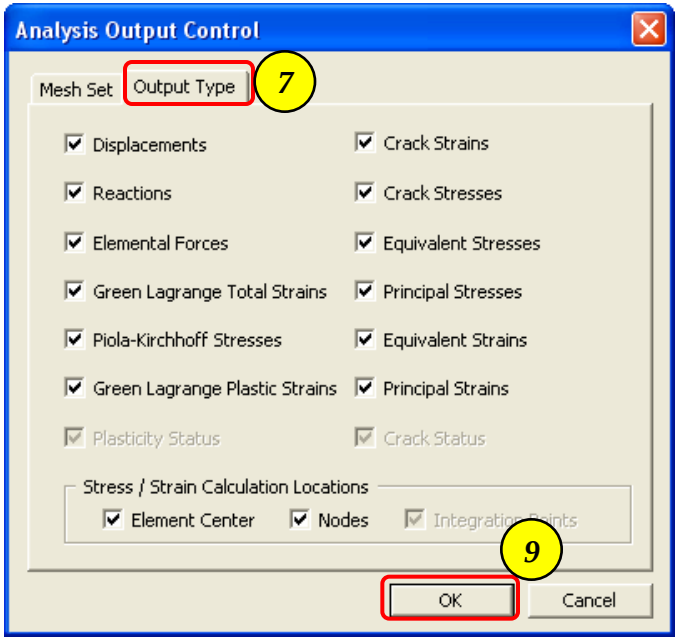


Step 15.



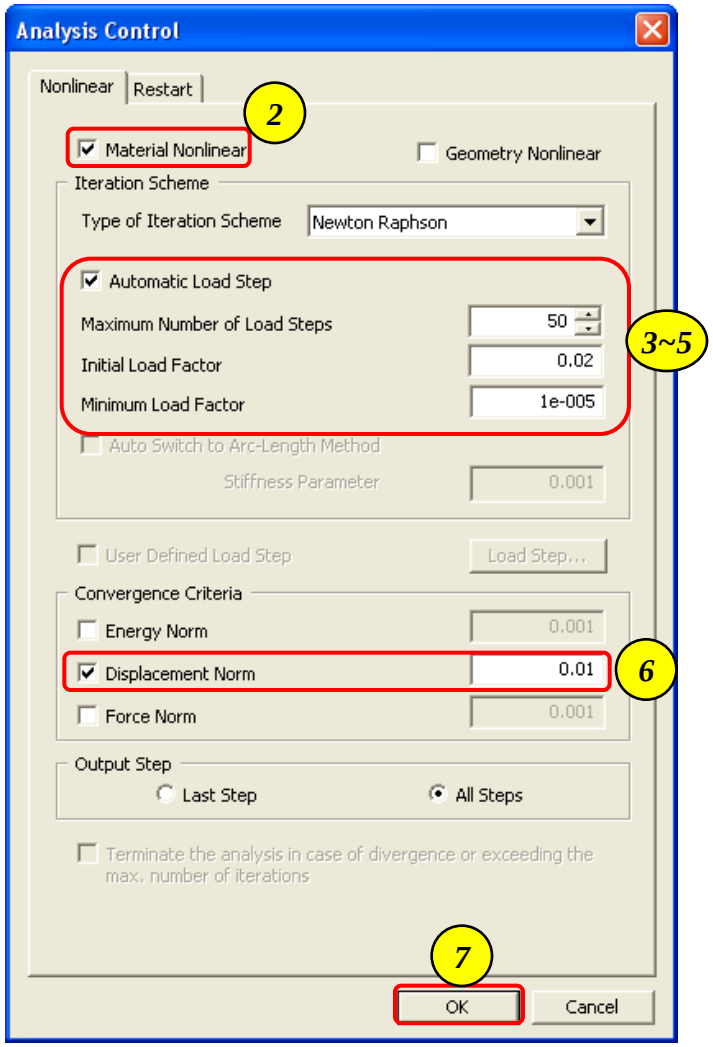
1. *Analysis > Analysis Case…*
2. *Click [Add] Button*
3. *Name : "Nonlinear"*
4. *Analysis Type: Nonlinear Static*
5. *Drag & Drop Load Set marked by "□" (See Figure)*
6. *Click on Output Control*
7. *Select Output Type tab*
8. *Select all the variables*
9. *Click [OK] Button*

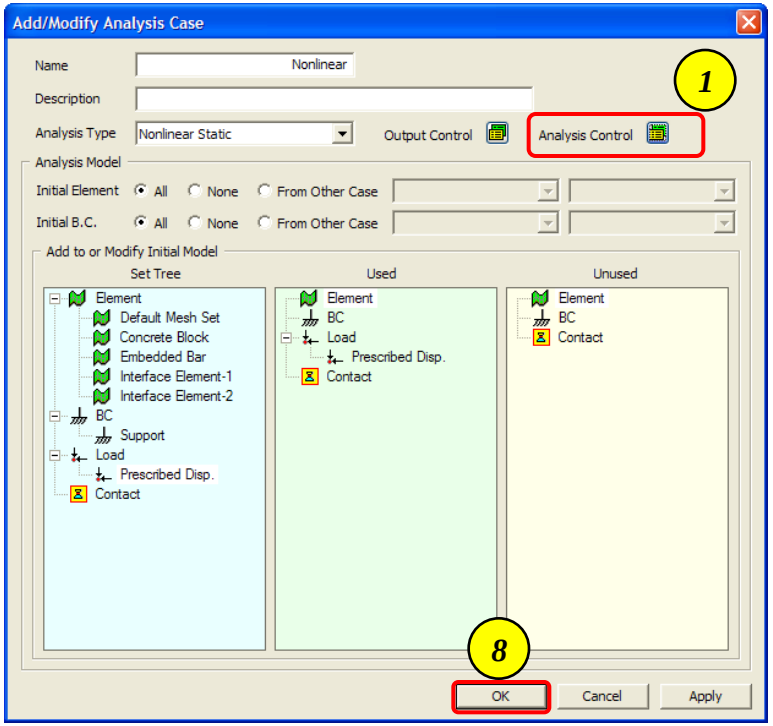




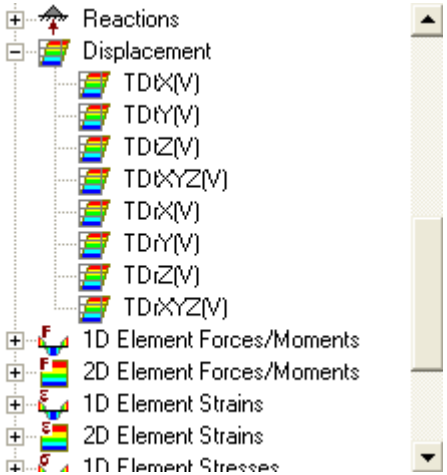
Step 16.

1. Click on Analysis Control
2. Select Material Nonlinear
3. Click on Automatic Load Step
4. Maximum Number of Load Steps: 50
5. Initial Load Factor: 0.02
6. Displacement Norm: 0.01
7. Click [OK] Button
8. Click [OK] Button

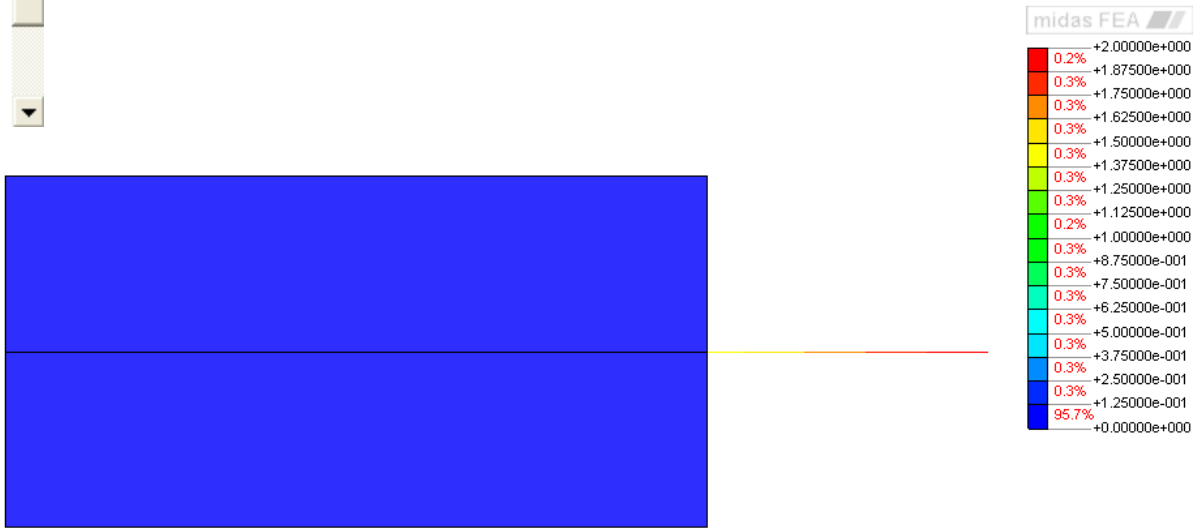




Step 17.

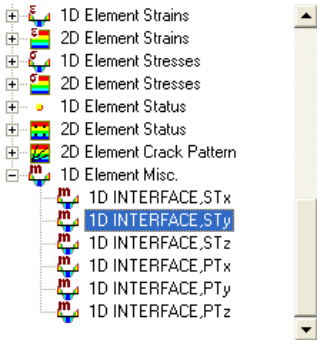


1. *Post-Works Tree : Nonlinear (Structural Nonlinear) > Nonlinear (1) > Displacement*
2. *Double Click on "TDtXYZ(v)"*



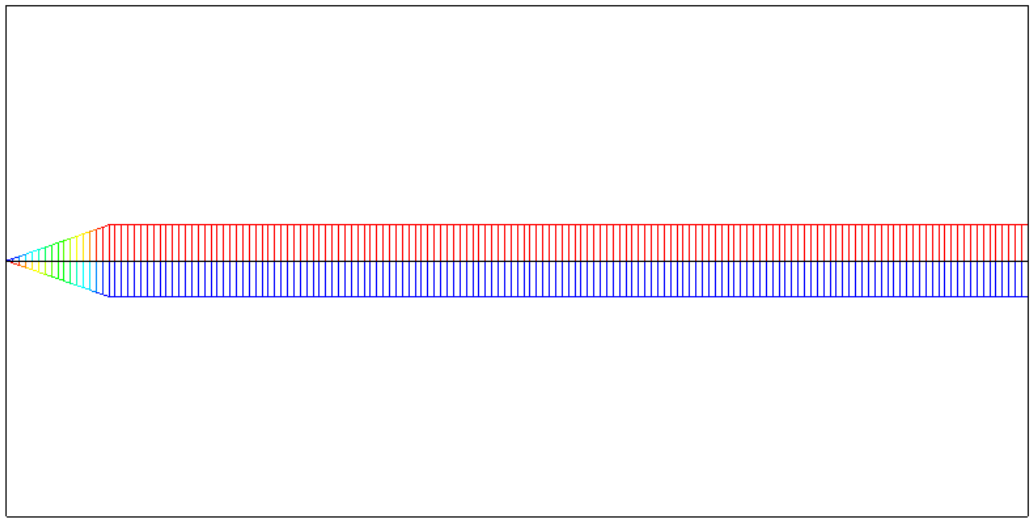
[UNIT] N , mm
[DATA] Nonlinear(Structural Nonlinear) , Nonlinear(1) , TDtX(V) , [Output CSys] Global CSys

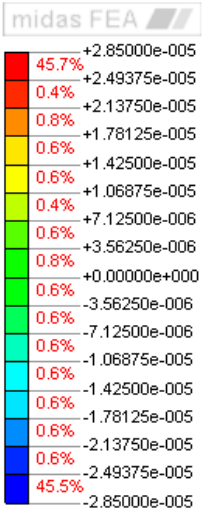
Step 18.



1. Post-Works Tree : Nonlinear (Structural Nonlinear) > Nonlinear (1) > 1D Element Misc.

2. Double Click on "1D INTERFACE, STy"





[UNIT] N , mm
[DATA] Nonlinear(Structural Nonlinear) , Nonlinear(1) , 1D INTERFACE,Line-STy , [Output CSys] Node/Elem Output CSys

The internal variables for interface elements should be presented separately.

The illustrated diagram is an assembled figure.