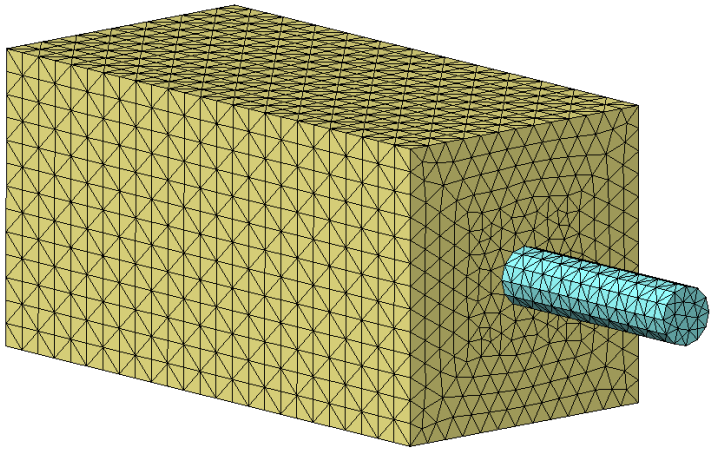
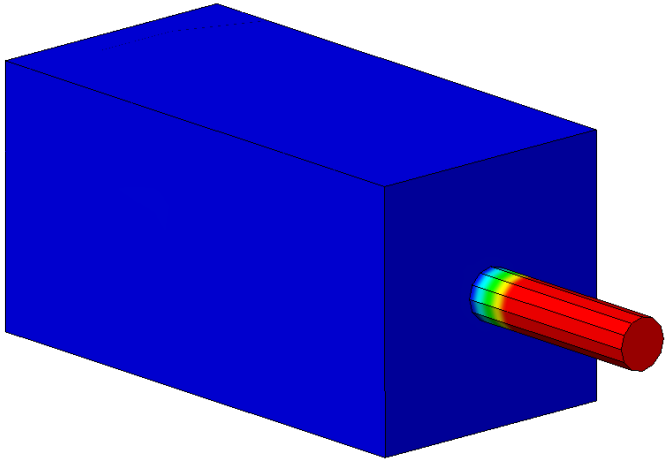
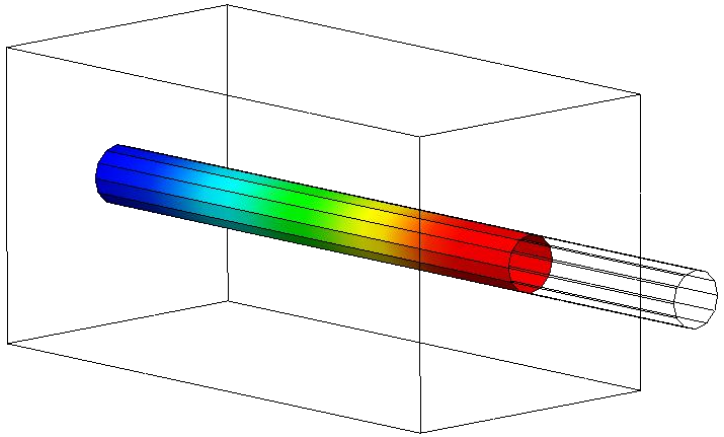
NL-2. An embedded bar in a concrete block (pull out test for bond-slip interface model)

Overview

- 3D 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*

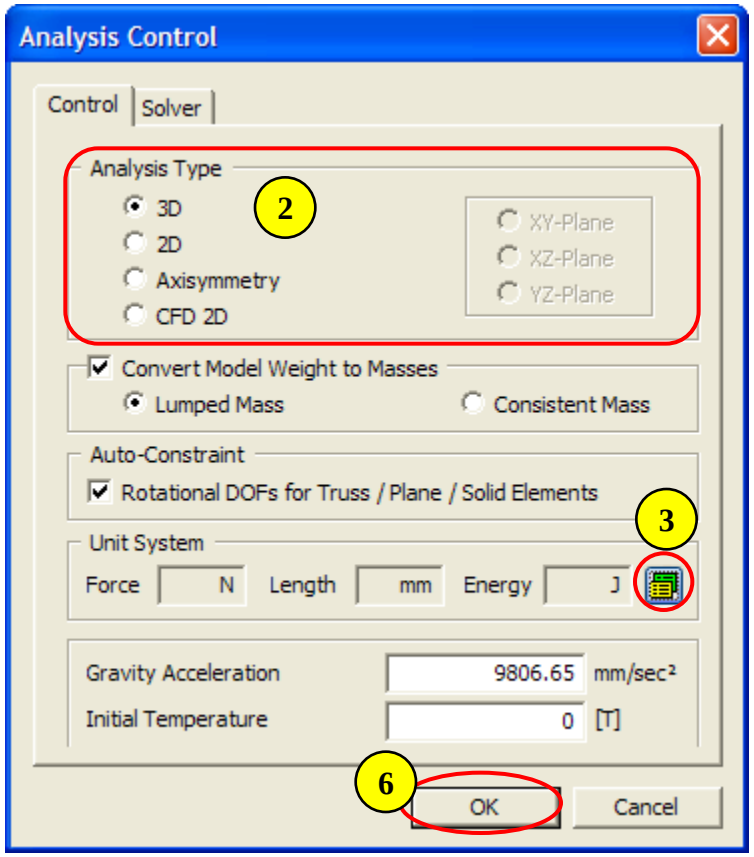


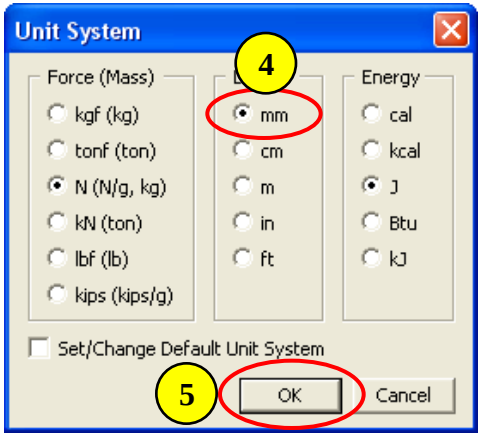




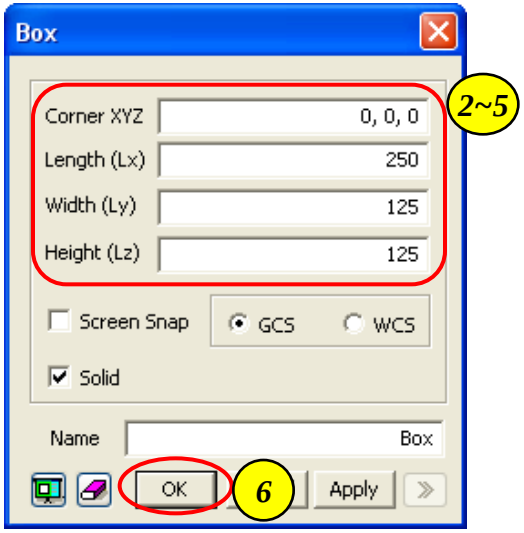
Step 1.

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

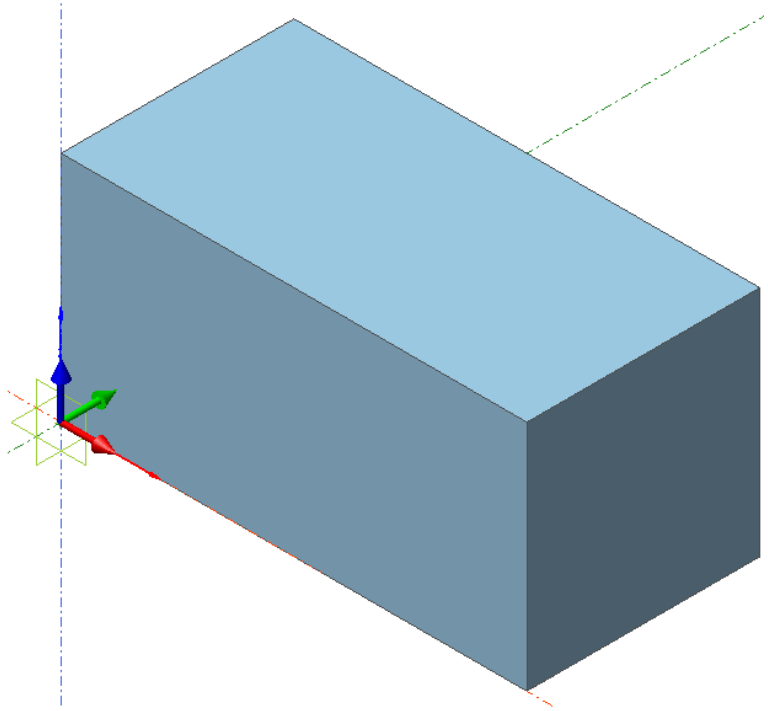




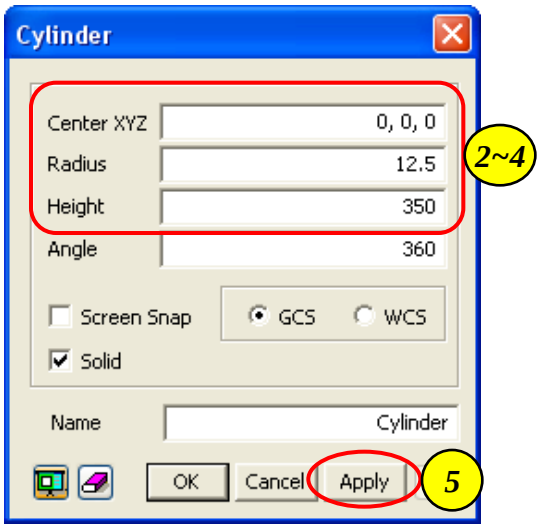
Step 2.

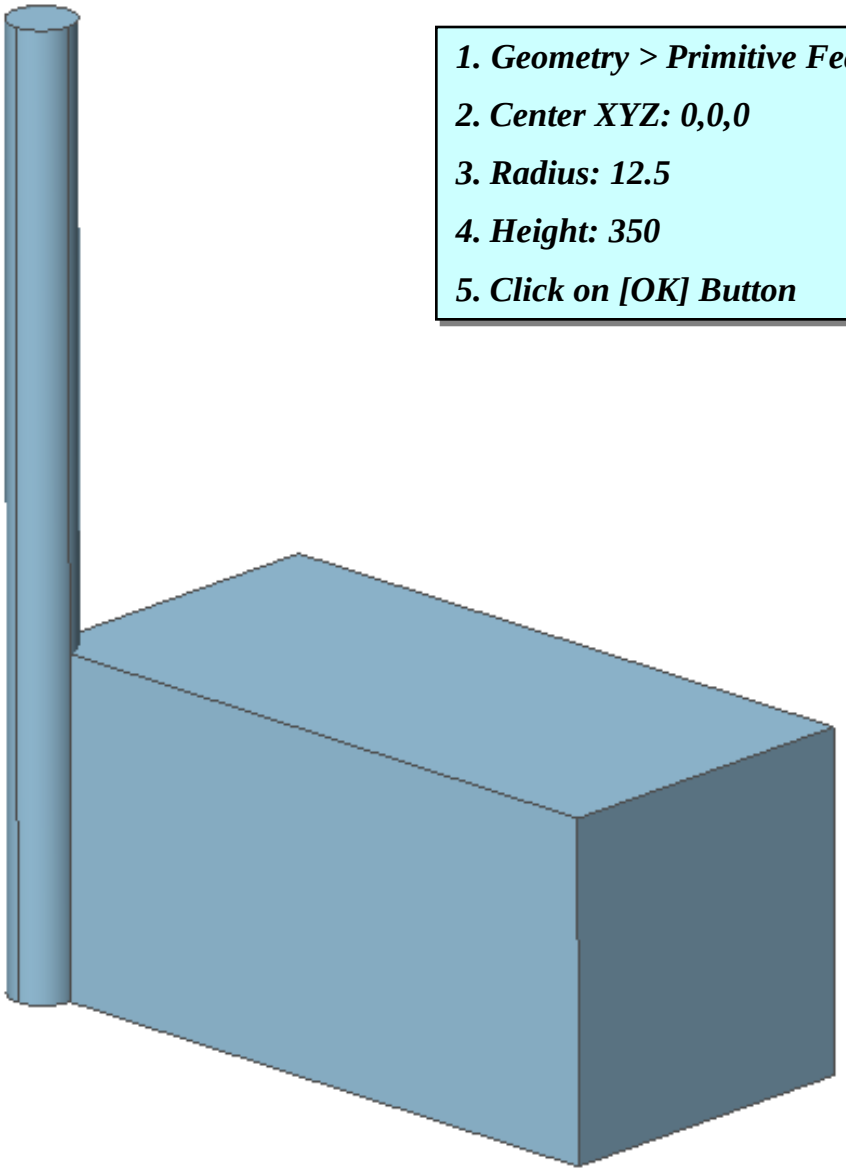


1. *Geometry > Primitive Feature > Box ...*
2. *Corner XYZ: 0,0,0*
3. *Length (Lx): 250*
4. *Length (Ly): 125*
5. *Length (Lz): 125*
6. *Click on [OK] Button*



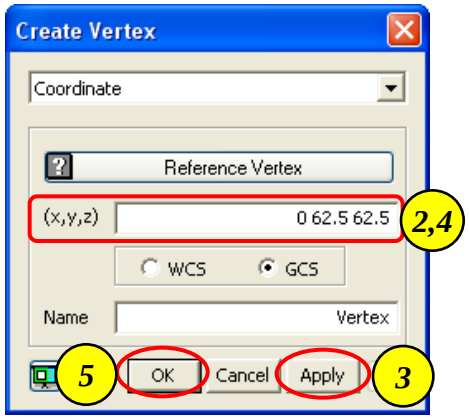
Step 3.

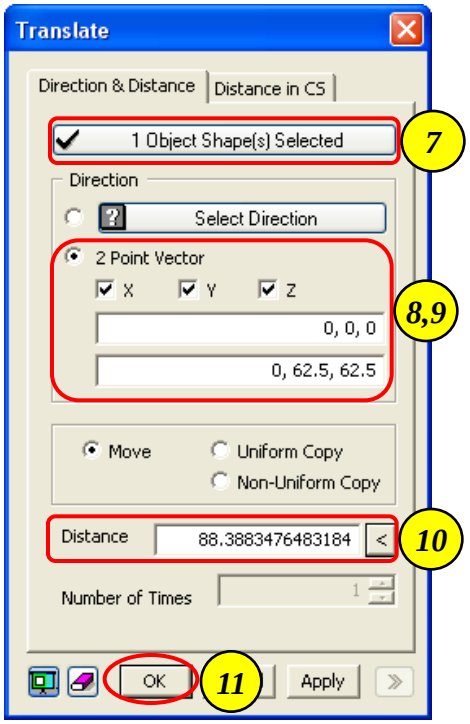


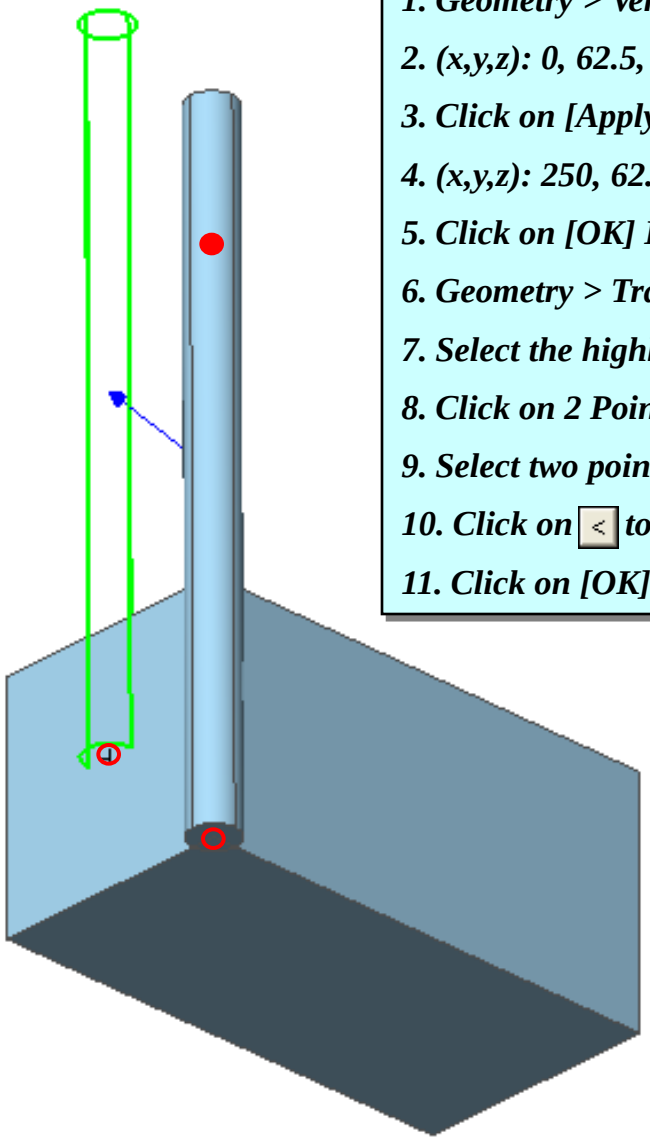


1. *Geometry > Primitive Feature > Cylinder ...*
2. *Center XYZ: 0,0,0*
3. *Radius: 12.5*
4. *Height: 350*
5. *Click on [OK] Button*

Step 4.

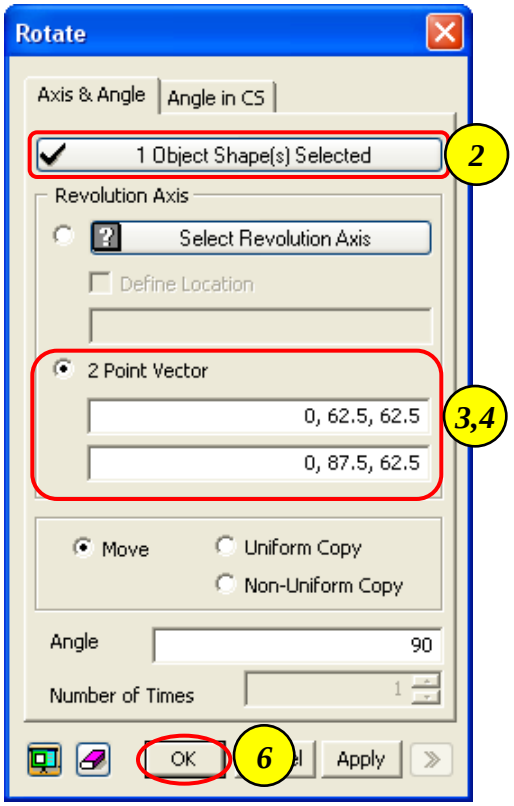


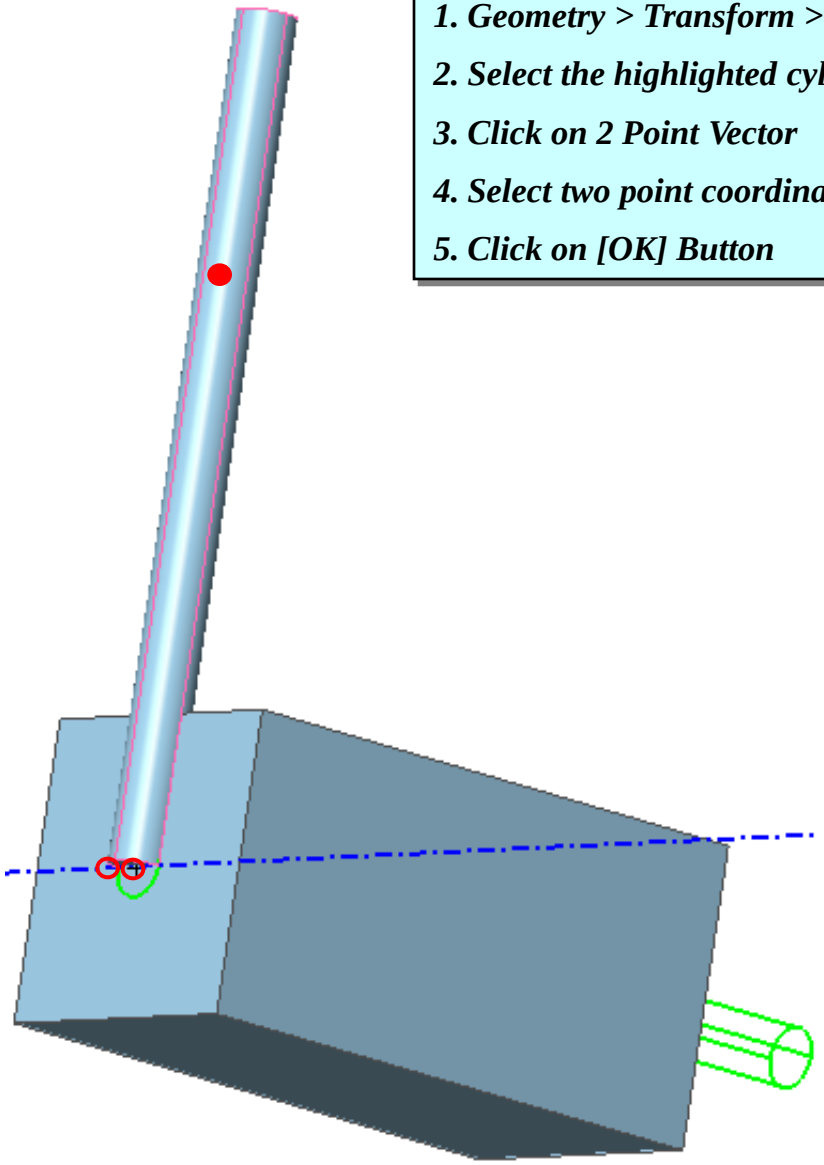




1. *Geometry > Vertex > Create ...*
2. *(x,y,z): 0, 62.5, 62.5*
3. *Click on [Apply] Button*
4. *(x,y,z): 250, 62.5, 62.5*
5. *Click on [OK] Button*
6. *Geometry > Transform > Translate ...*
7. *Select the highlighted cylinder*
8. *Click on 2 Point Vector*
9. *Select two point coordinates as shown in the figure*
10. *Click on [<] to automatically calculate the distance*
11. *Click on [OK] Button*

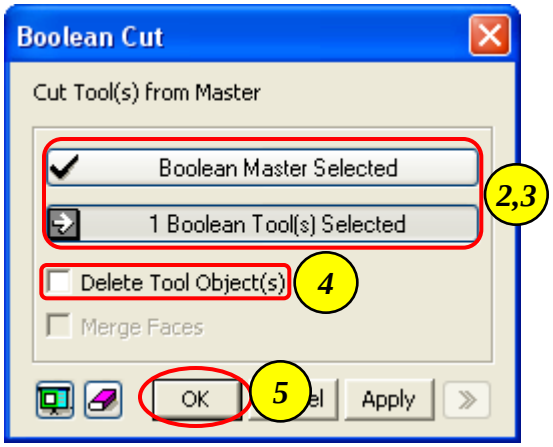
Step 5.



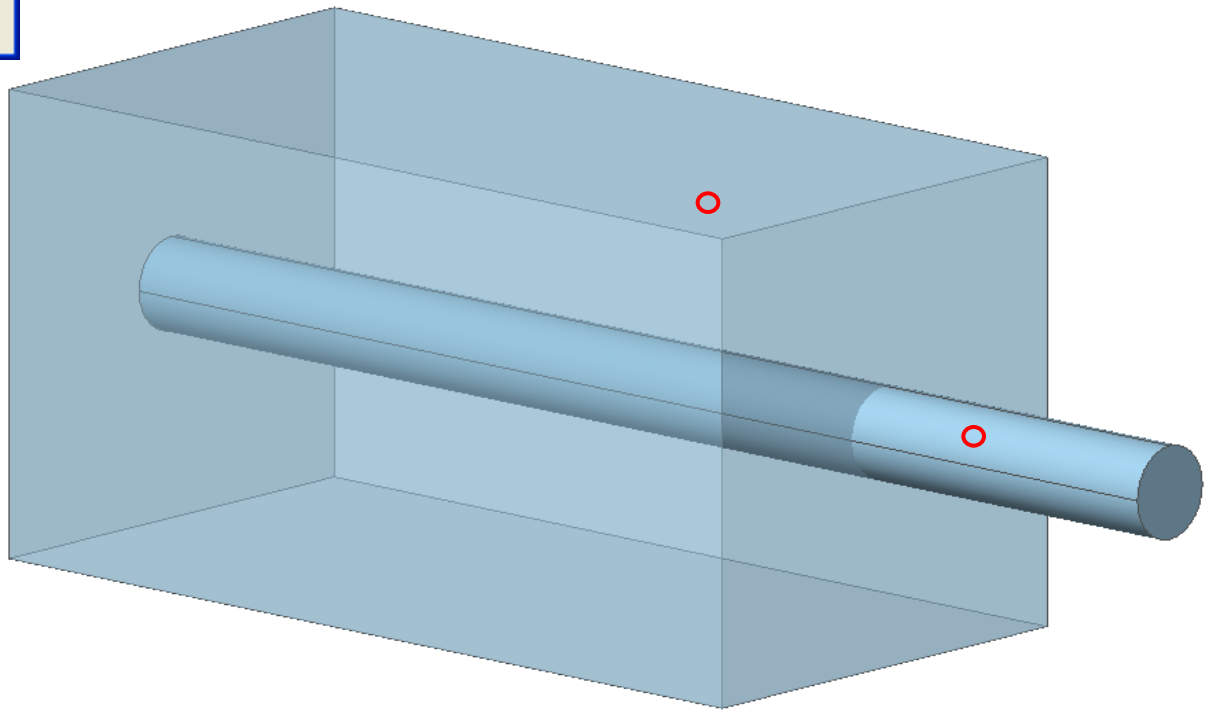


1. *Geometry > Transform > Rotate ...*
2. *Select the highlighted cylinder*
3. *Click on 2 Point Vector*
4. *Select two point coordinates as shown in the figure*
5. *Click on [OK] Button*

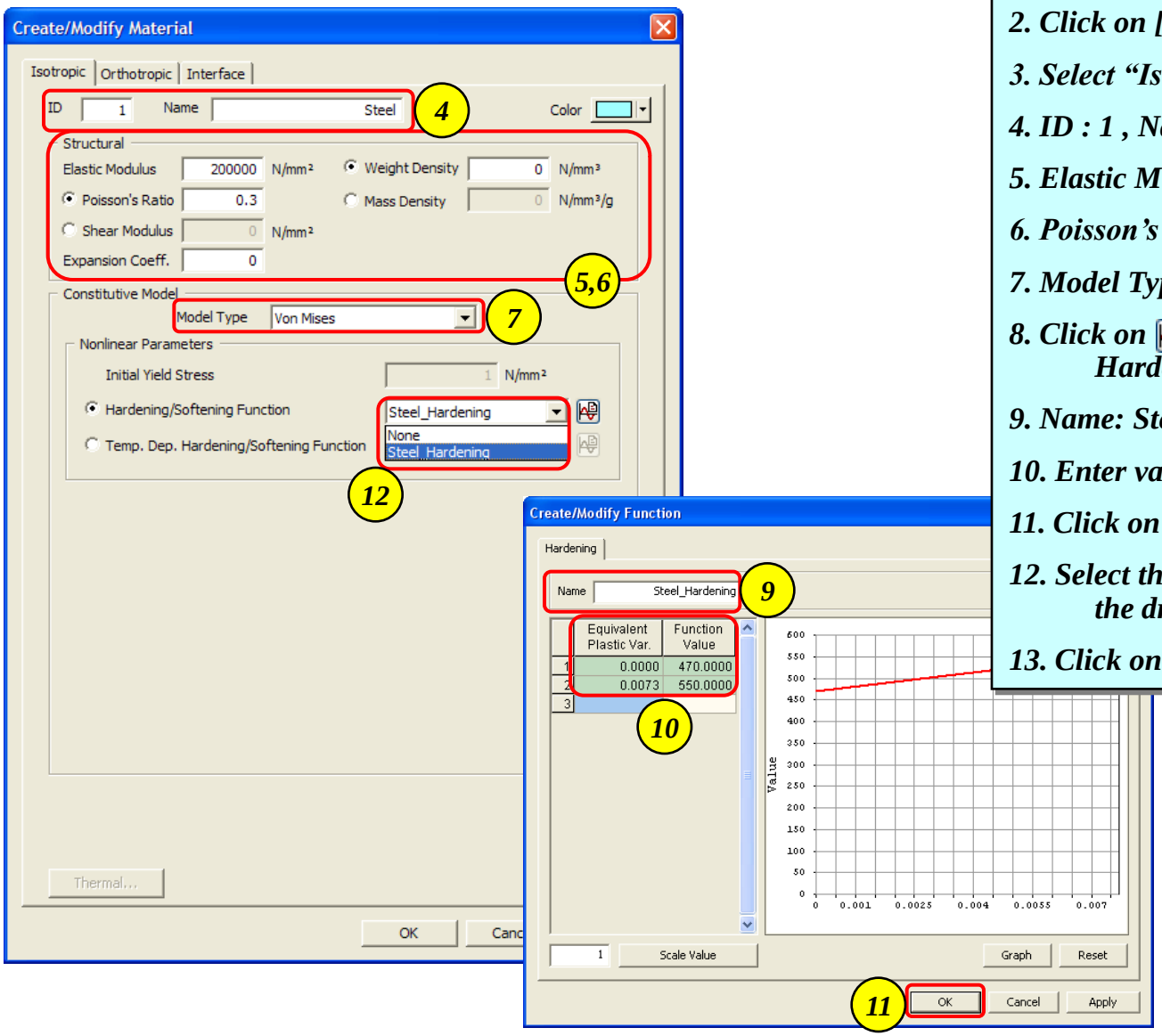
Step 6.



1. *Geometry > Boolean Operation > Cut ...*
2. *Select the Box for Boolean Master*
3. *Select the cylinder for Boolean Tool*
4. *Tick off Delete Tool Object(s)*
5. *Click on [OK] Button*



Step 7.

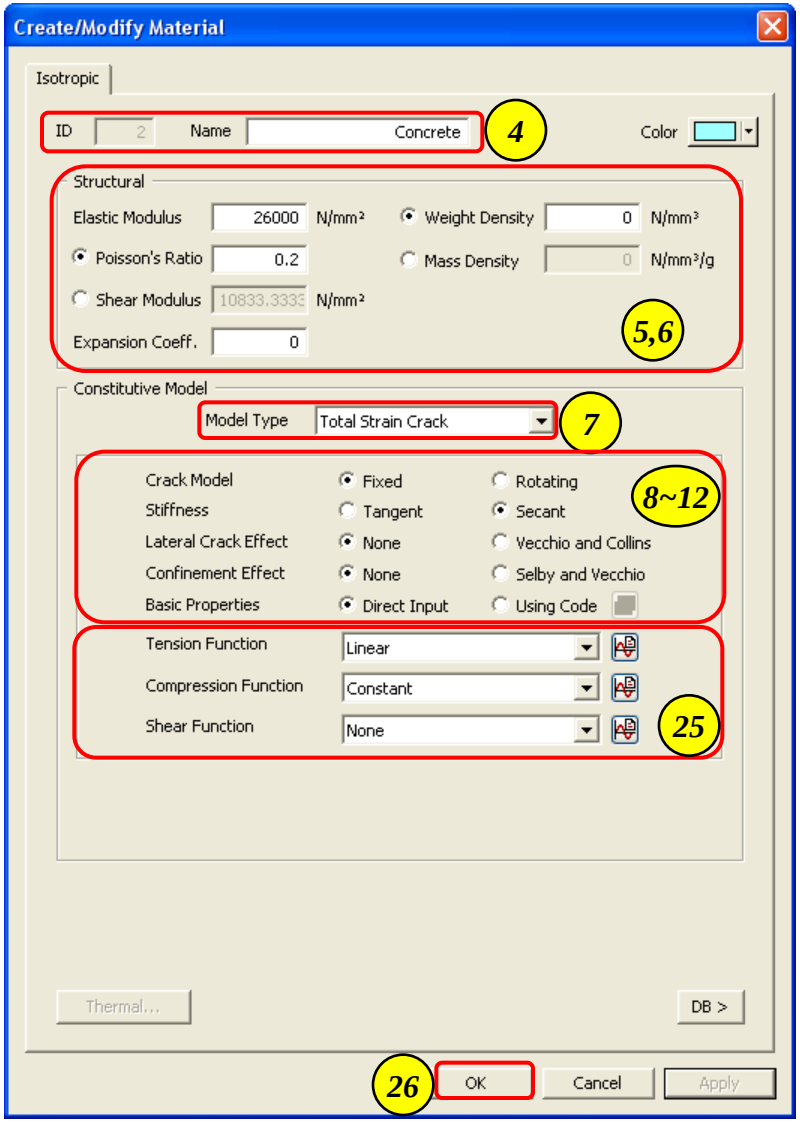


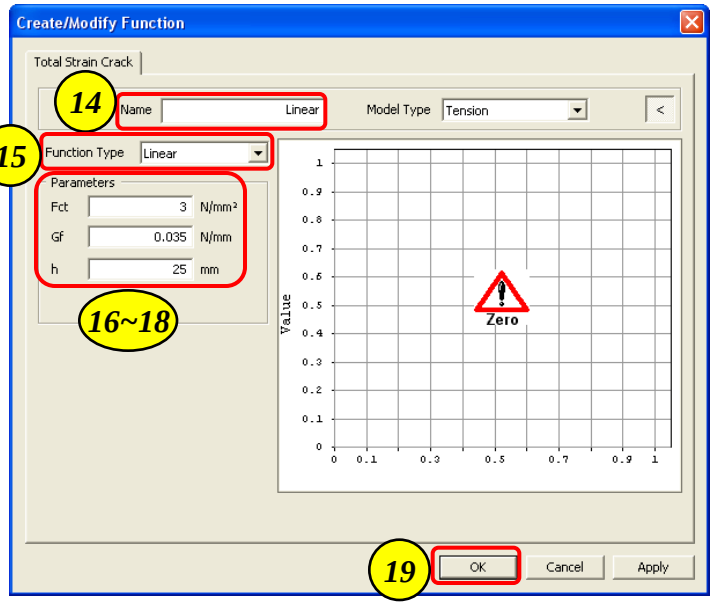
1. *Analysis > Material …*
2. *Click on [Create] Button*
3. *Select "Isotropic" tab*
4. *ID : 1 , Name : Steel*
5. *Elastic Modulus : 200000 N/mm²*
6. *Poisson's Ratio : 0.3*
7. *Model Type : Von Mises*
8. *Click on to define Hardening/Softening Function*
9. *Name: Steel_Hardening*
10. *Enter values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

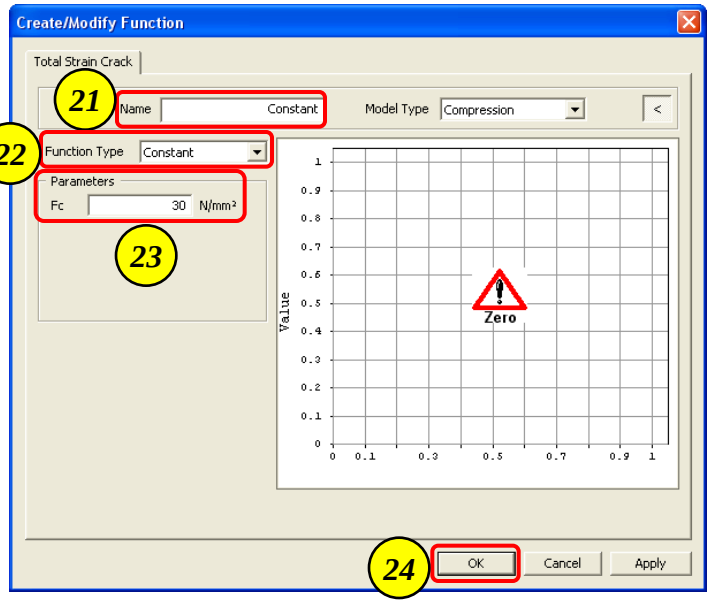
Step 8.

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

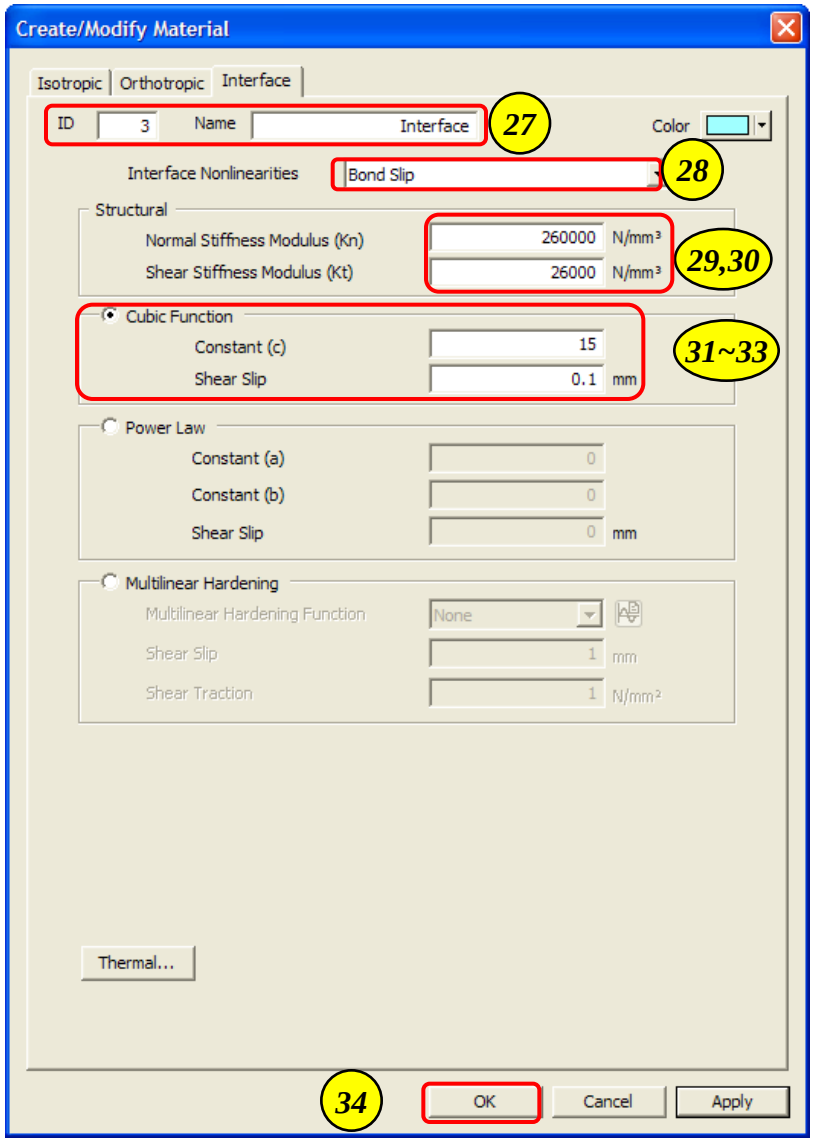
Step 9.

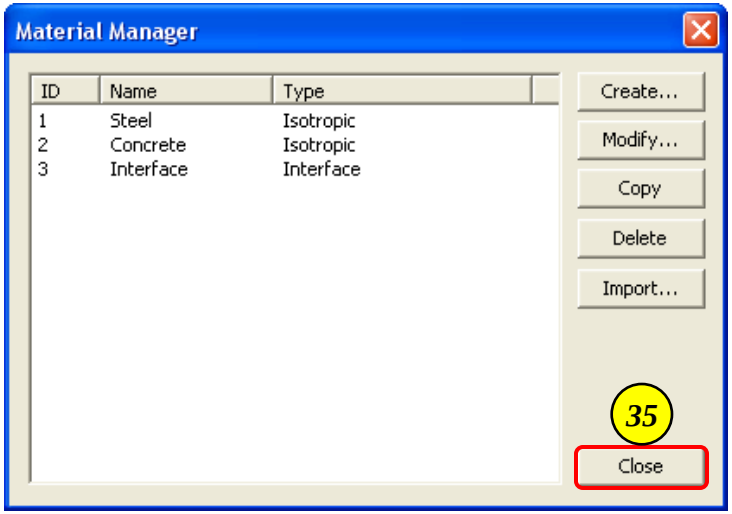

Create/Modify Material
Isotropic
ID 2
Name Concrete
4
Color
Structural
Elastic Modulus 26000 N/mm²
Weight Density 0 N/mm³
Poisson's Ratio 0.2
Mass Density 0 N/mm³/g
Shear Modulus 10833.3333 N/mm²
Expansion Coeff. 0
5,6
Constitutive Model
Model Type Total Strain Crack
7
Crack Model Fixed Rotating
Stiffness 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

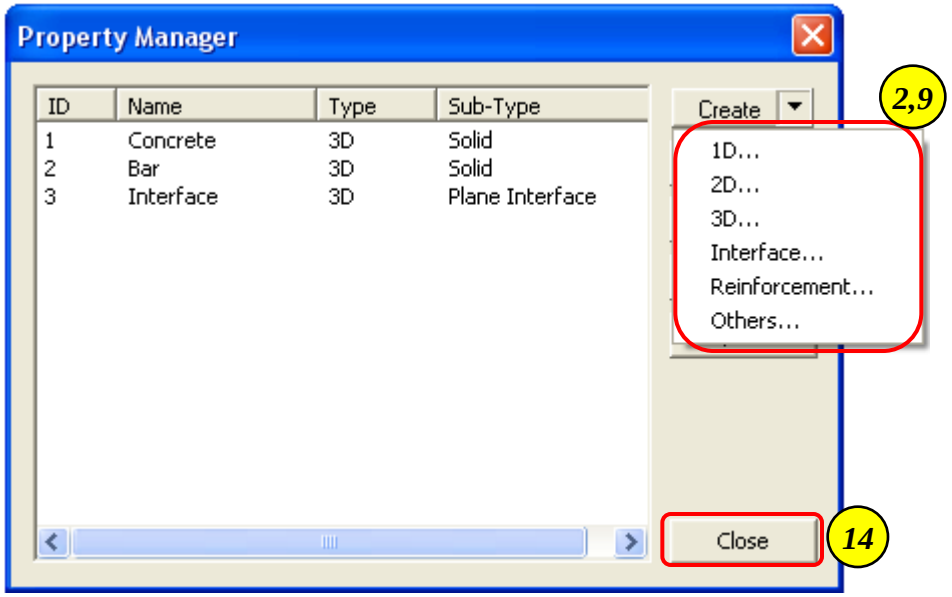

Create/Modify Function
Total Strain Crack
21
Name Constant
Model Type Compression
22
Function Type Constant
Parameters
Fc 30 N/mm²
23
Value
Zero
24
OK
Cancel
Apply

Step 10.


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
mm
Shear Traction
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

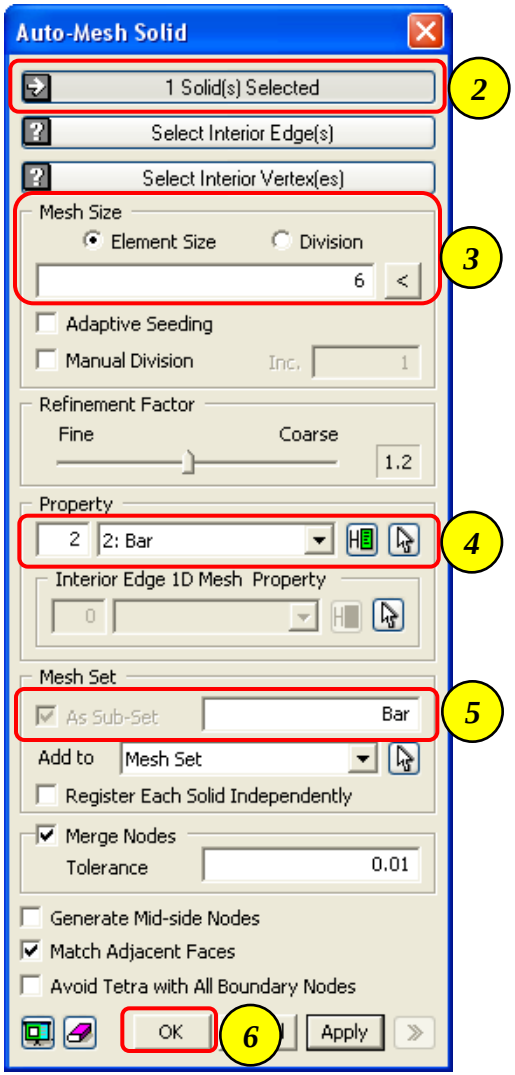
Step 11.

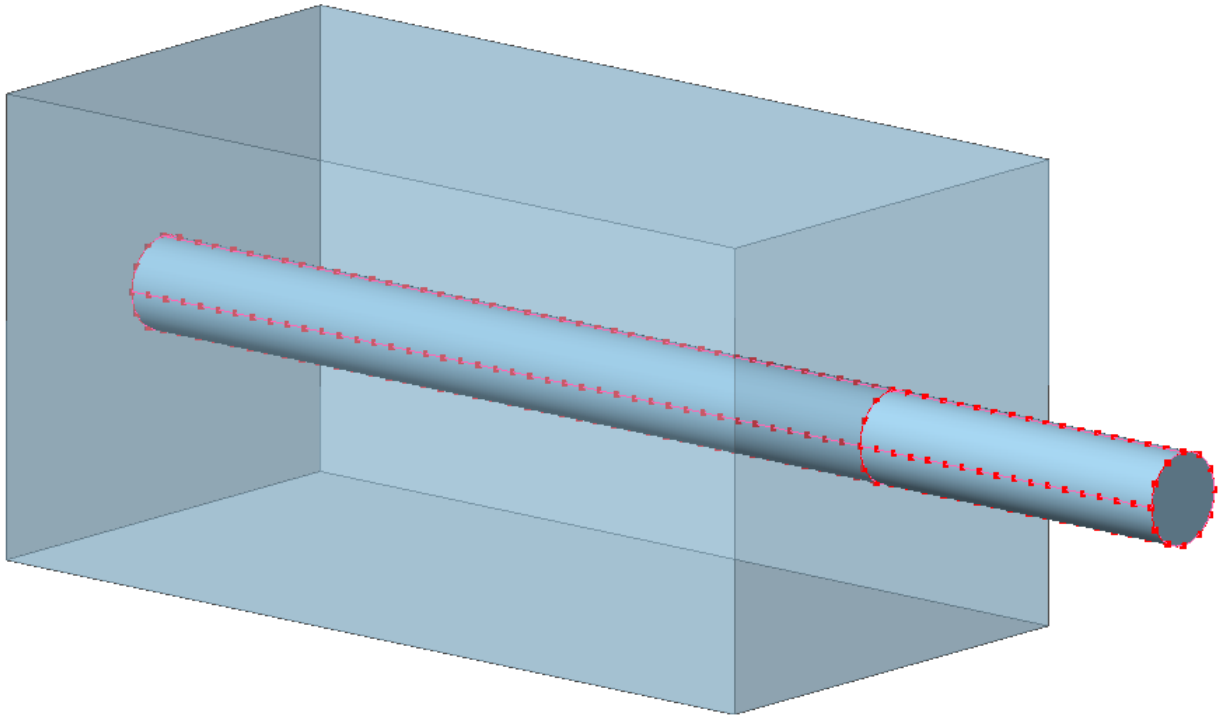


1. Analysis > Property...
2. Create 3D…
3. ID : 1 , Name : Concrete
4. Material : (2: Concrete)
5. Click on [Apply] Button
6. ID : 2 , Name : Bar
7. Material : (1: Steel)
8. Click on [Apply] Button
9. Create Interface…
10. ID : 4 , Name : Interface
11. Type: Plane
12. Material : (3: Interface)
13. Click on [OK] Button
14. Click on [Close] Button

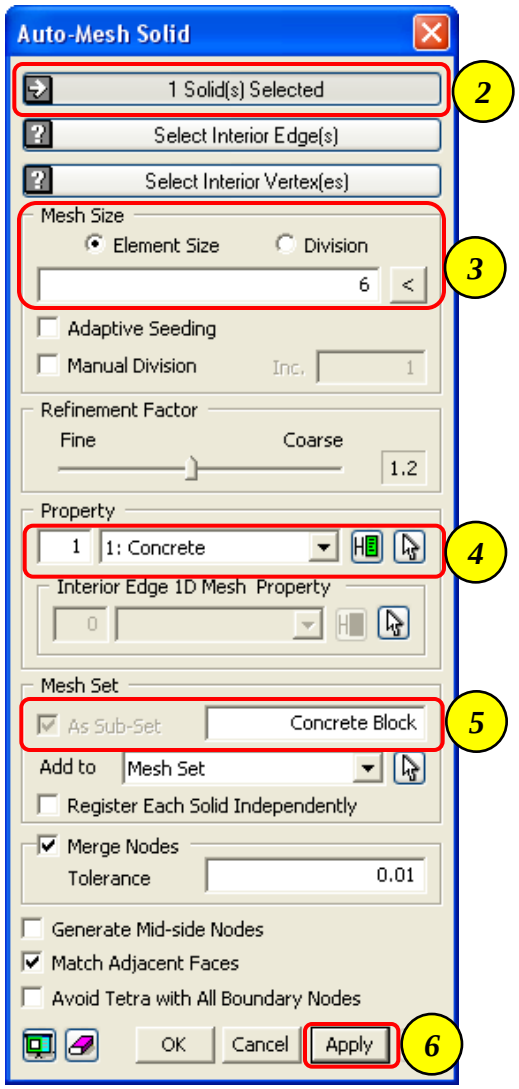
Step 12.

1. Mesh > Auto Mesh > Solid ...
2. Select the cylinder
3. Element Size: 6
4. Property: 2 : Bar
5. Mesh Set: Bar
6. Click on [OK] Button

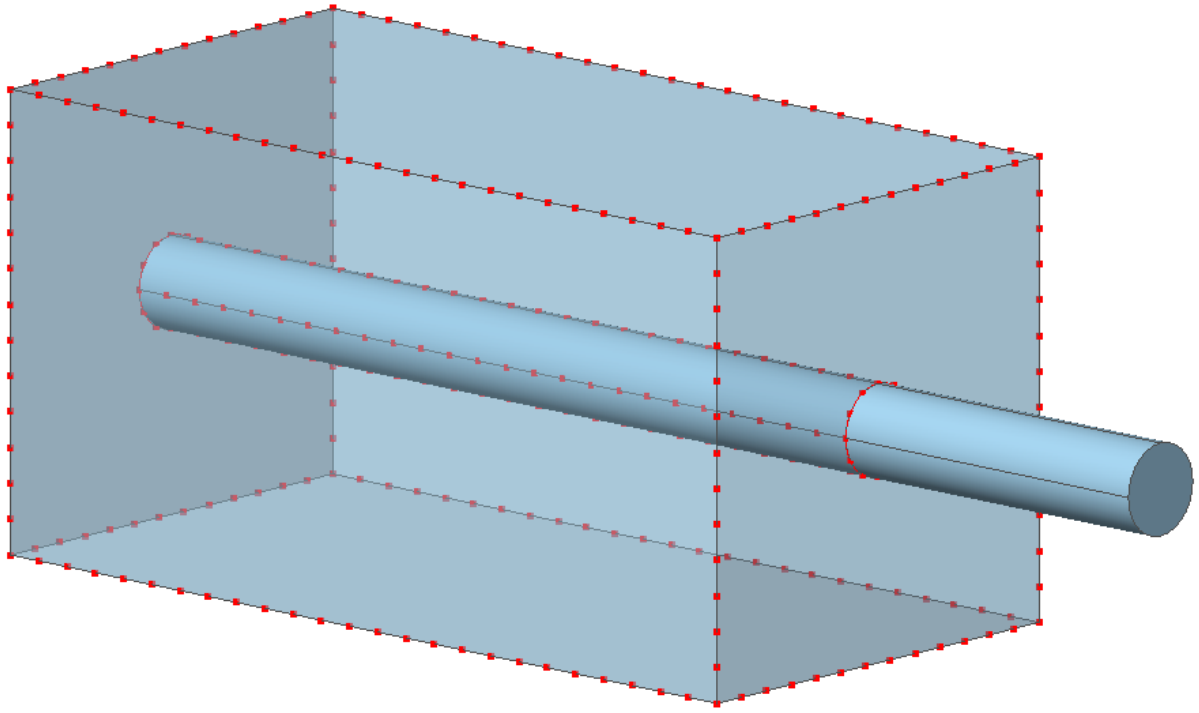




Step 13.

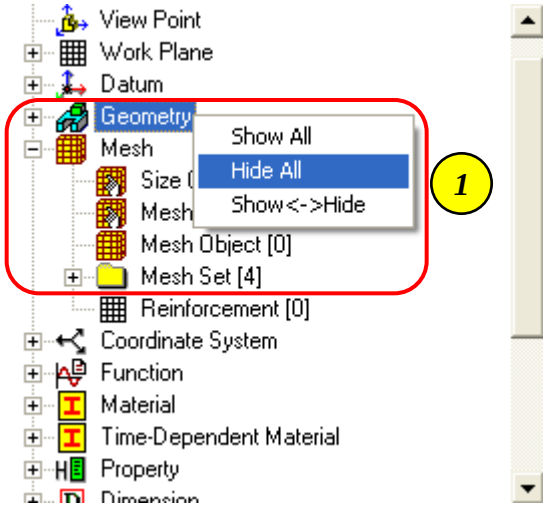


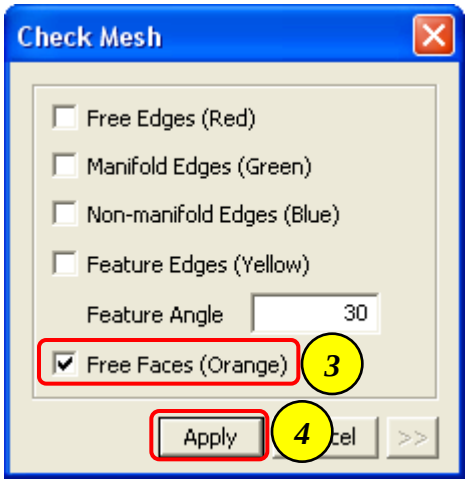
1. *Mesh > Auto Mesh > Solid ...*
2. *Select the concrete Box*
3. *Element Size: 6*
4. *Property: 1 : Concrete*
5. *Mesh Set: Concrete Block*
6. *Click on [Apply] Button*

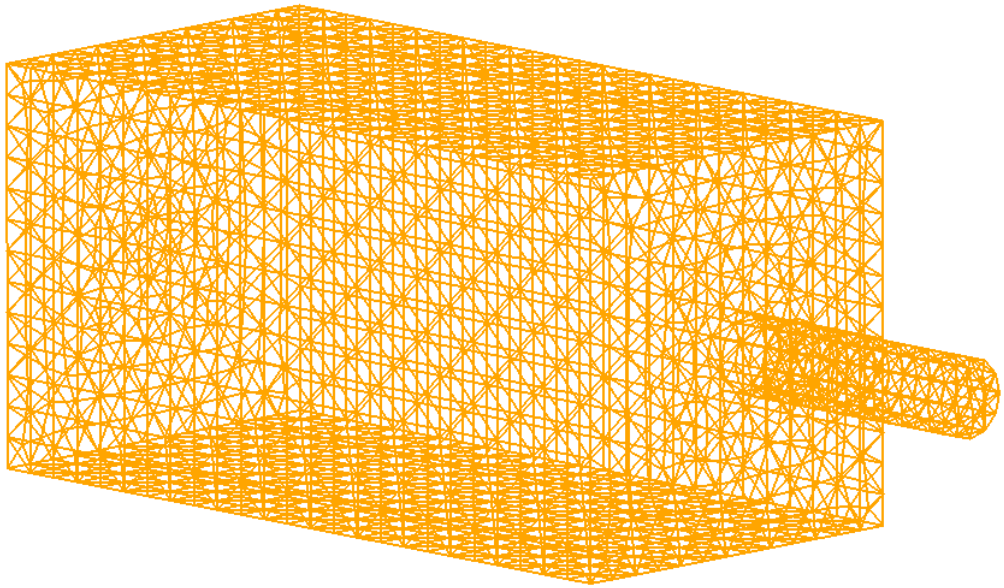


Step 14.

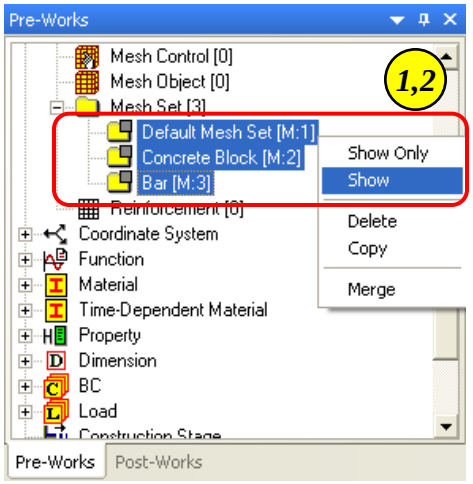
1. From Pre-works tree select Hide for Geometry and Mesh
2. Mesh > Check Mesh …
3. Tick on Free Faces
4. Click on [Apply] Button

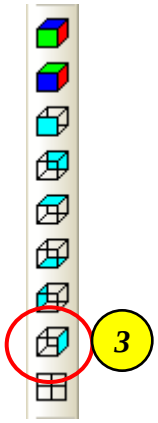




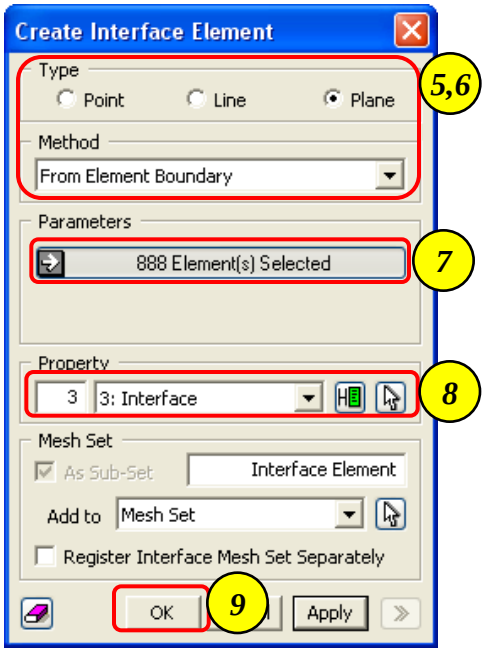


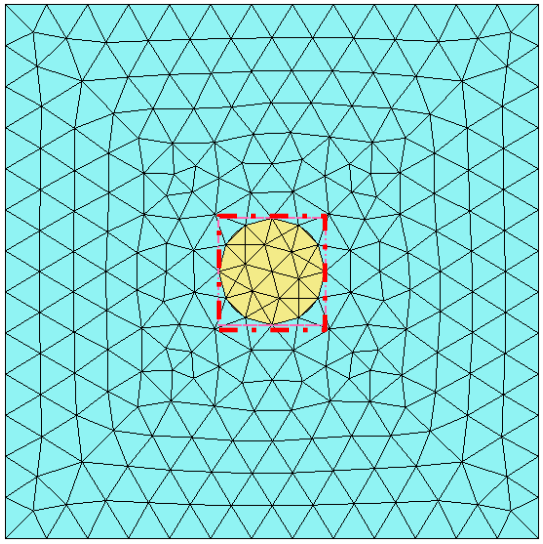
Step 15.

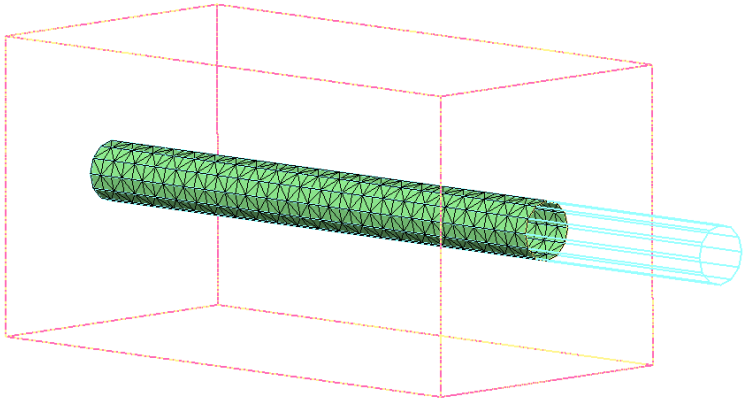




1. *From Pre-works tree select Hide for Geometry and Mesh*
2. *Select the highlighted mesh sets and select Show*
3. *Click on "Right View"*
4. *Mesh > Element > Create Interface Element …*
5. *Type: Plane*
6. *Method: From Element Boundary*
7. *Select the highlighted mesh set*
8. *Property: 3: Interface*
9. *Click on [OK] Button*

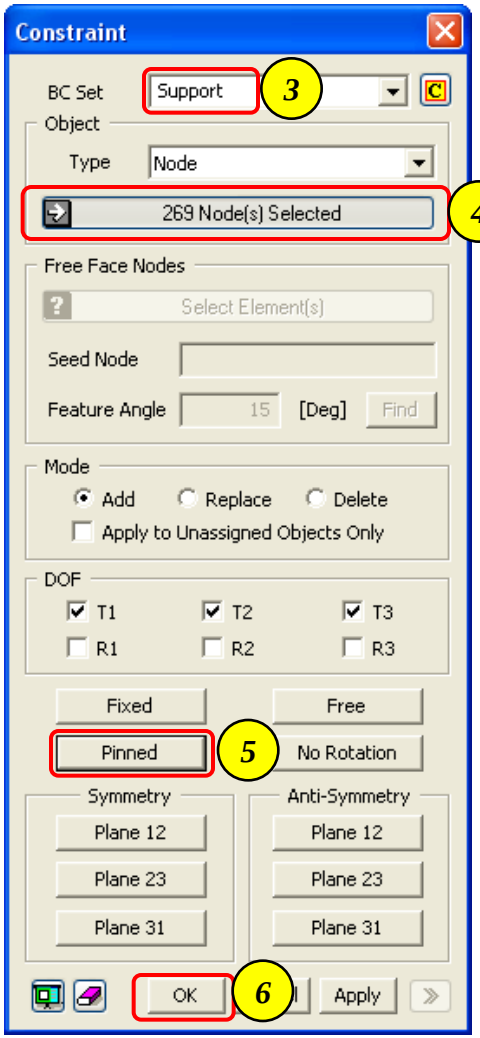


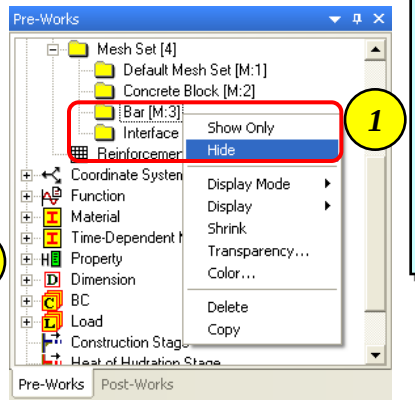


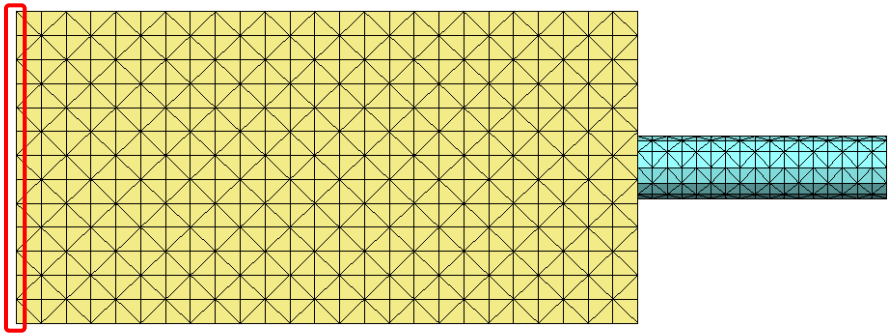


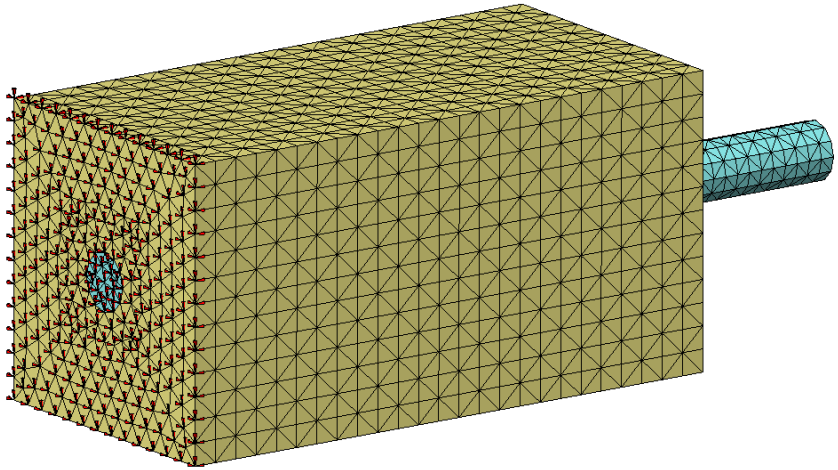
Step 16.

1. Hide Bar mesh set from work tree
2. Analysis > BC > Constraint …
3. BC Set : Support
4. Select the highlighted nodes as shown in the figure
5. Click on “Pinned”
6. Click [OK] Button

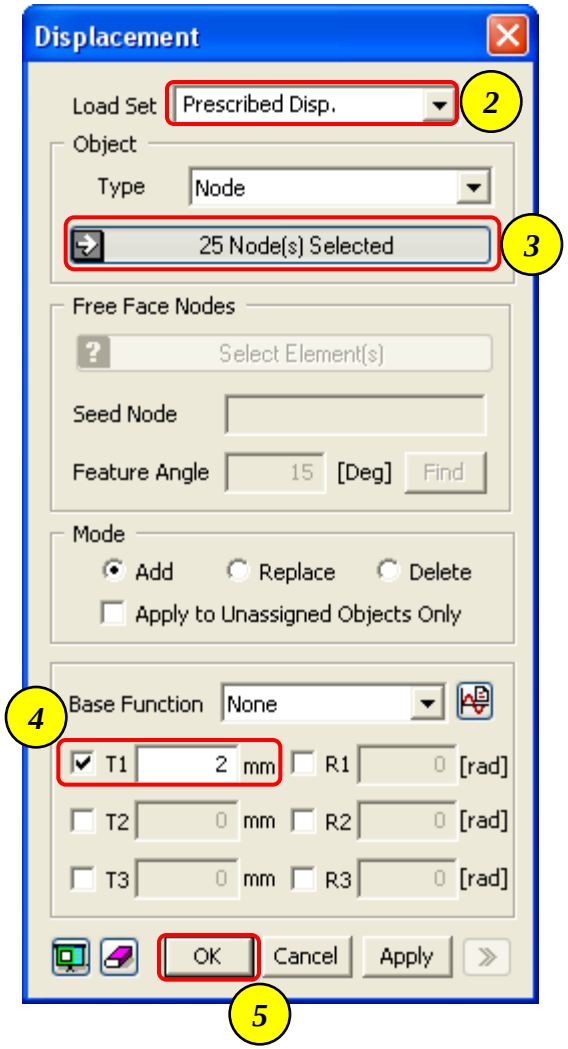




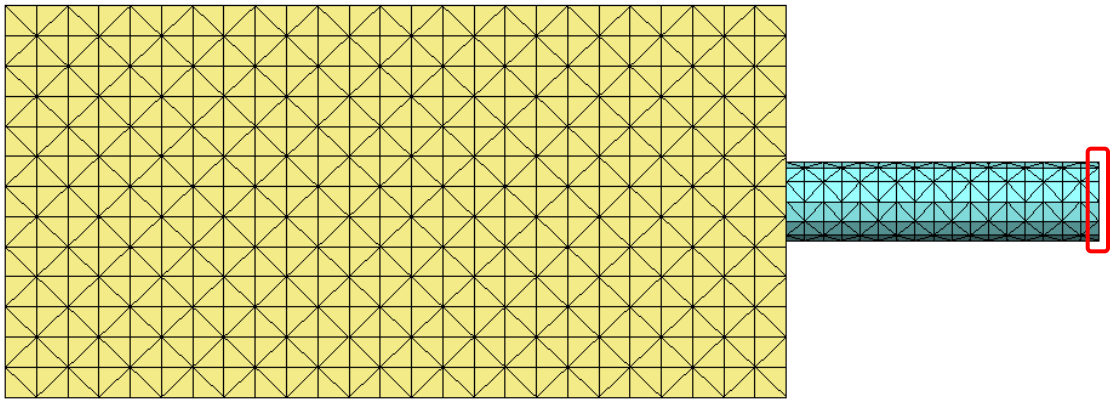




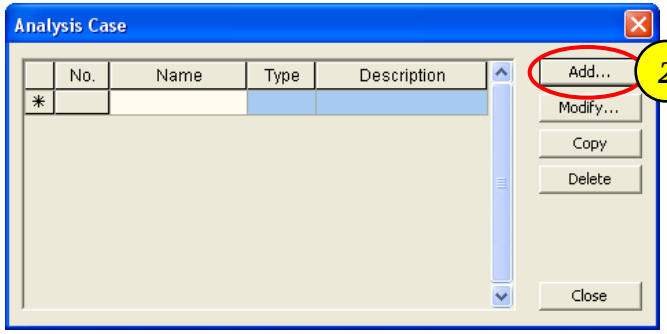
Step 17.



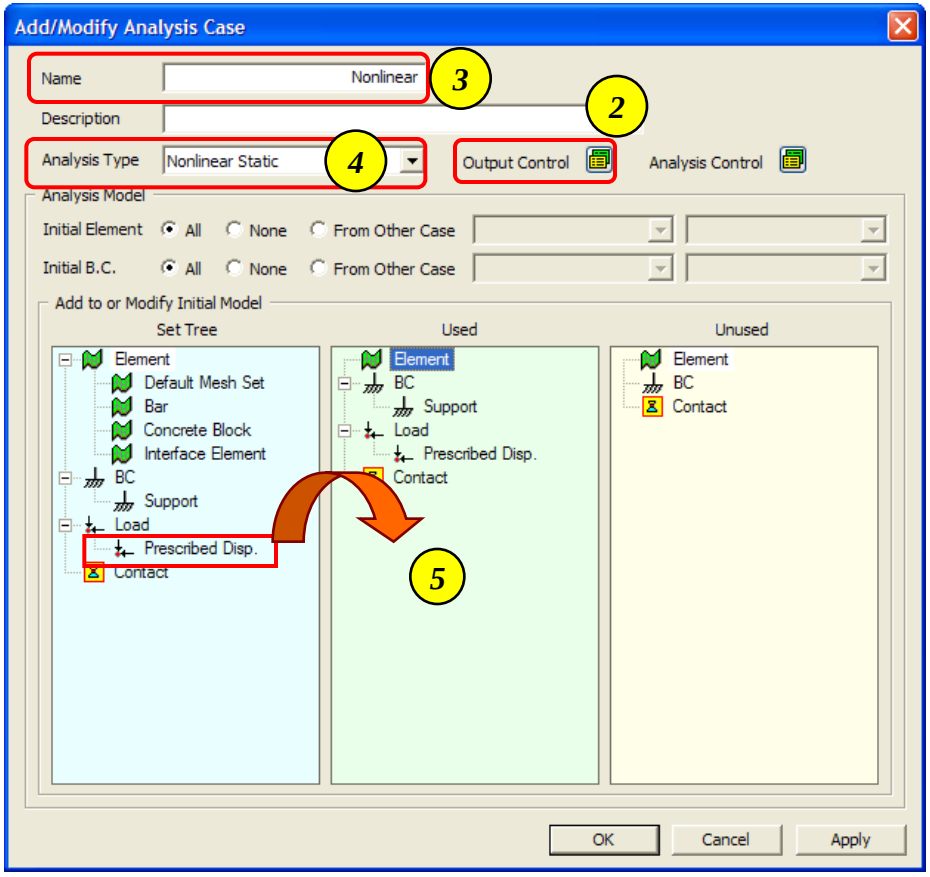
1. *Analysis > Load > Displacement…*
2. *Load Set : Prescribed Disp.*
3. *Select the nodes as shown in the figure*
4. *T1 = 2 mm*
5. *Click [OK] Button*

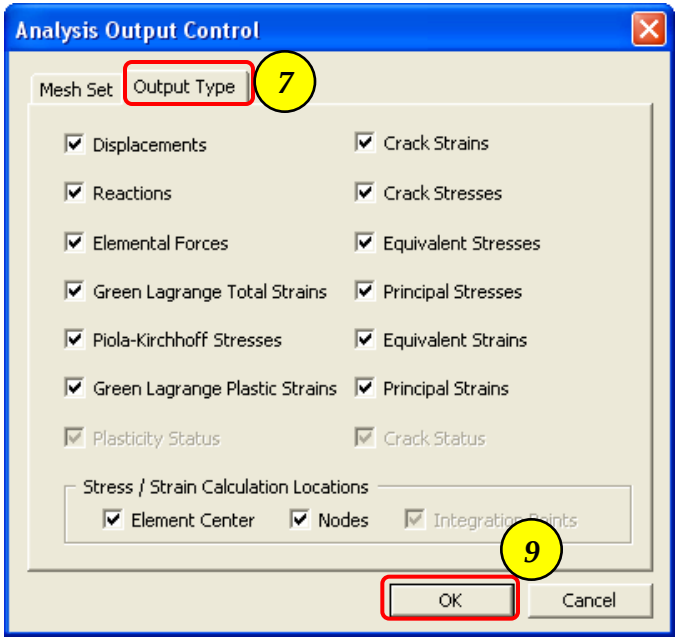


Step 18.



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 19.

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