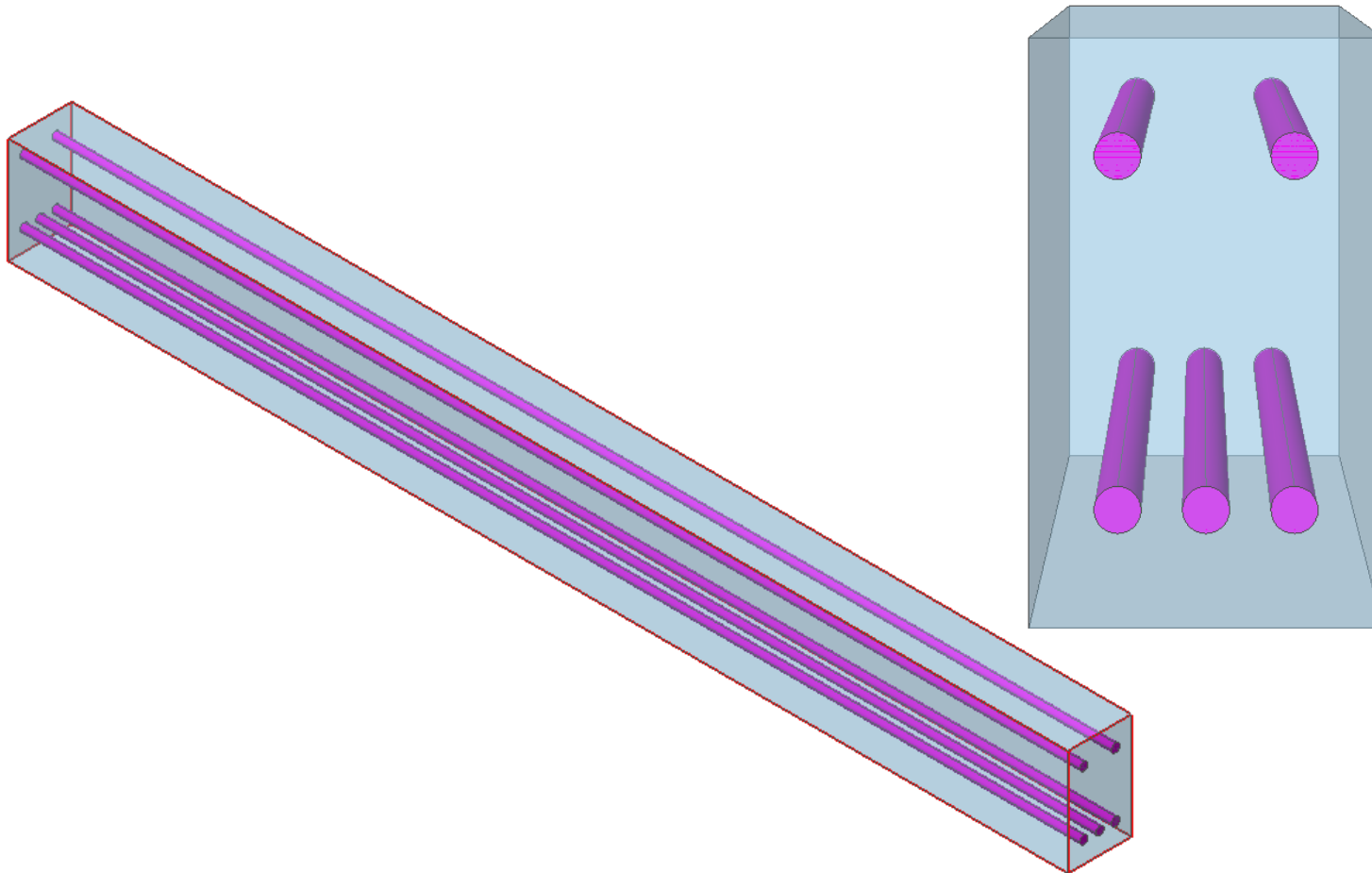


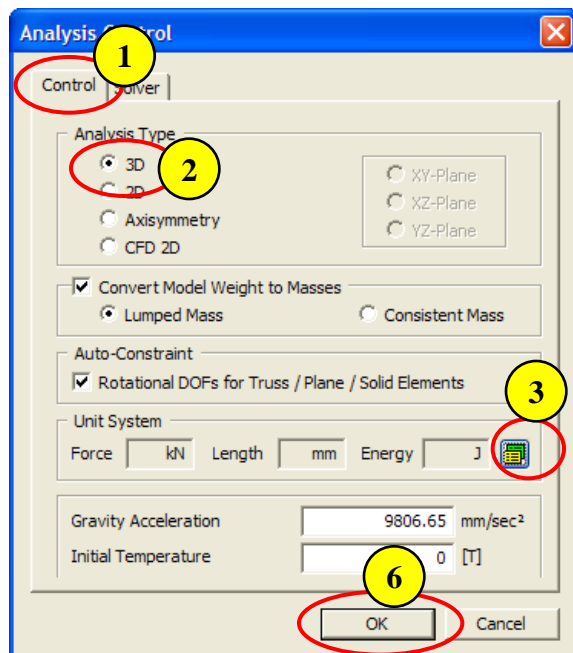
TP-1. RC Beam with Interface Elements


Overview

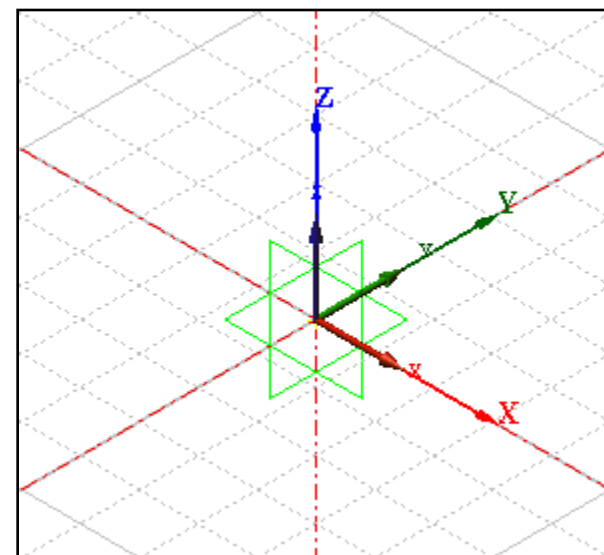
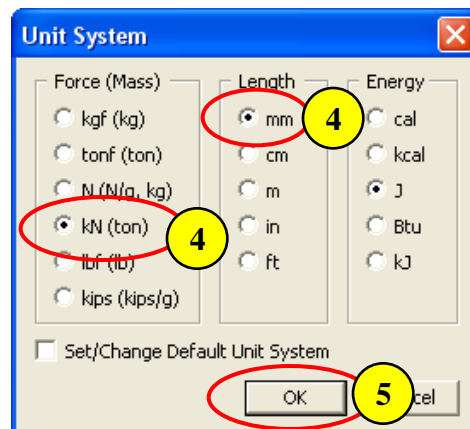
- Modeling RC beam
- Assigning Interface Elements between rebar and concrete



Step 1.

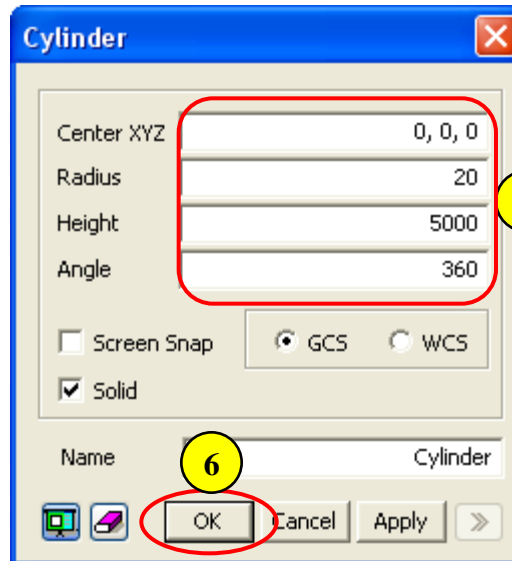
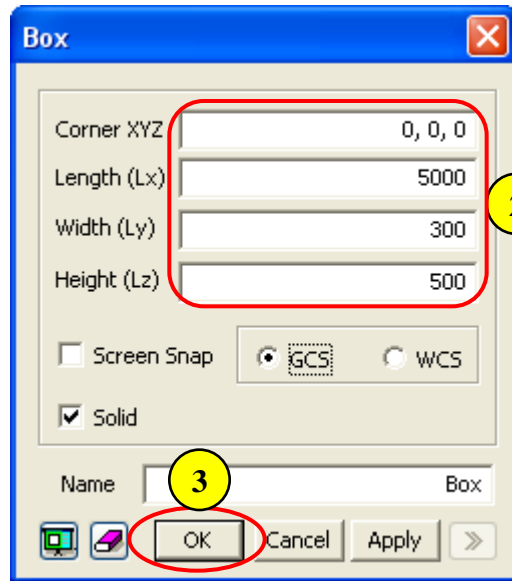


1. Analysis > Analysis Control – “Control” tab
2. Analysis Type : 3D
3. Click  Button
4. Unit : kN , mm
5. Click [OK] Button
6. Click [OK] Button

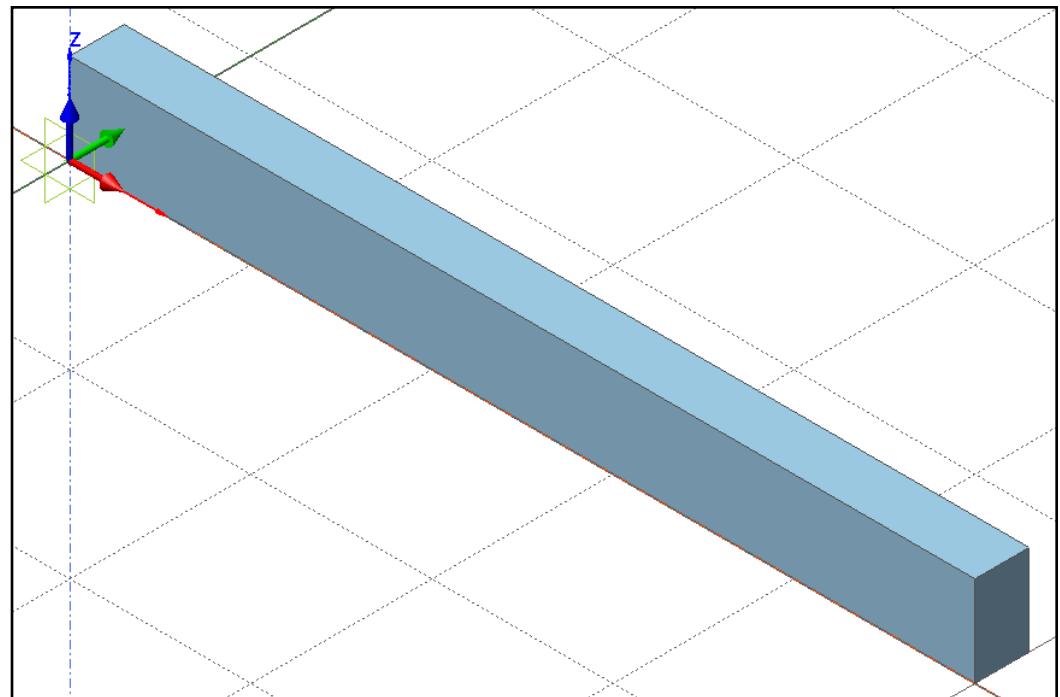


 Analysis Control Dialog is automatically activated at startup.

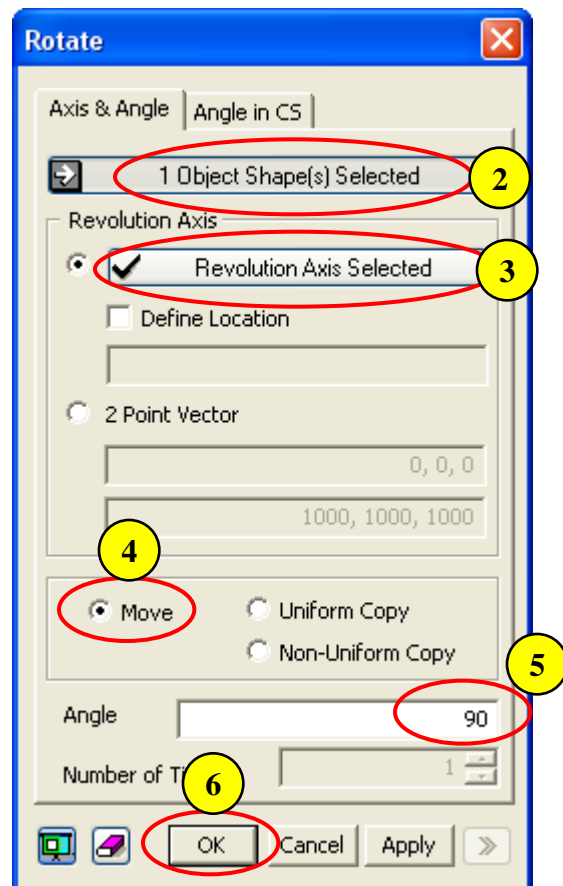
Step 2.



1. Geometry > Primitive Feature > Box...
2. Insert parameters as shown in Figure
3. Click [OK] Button
4. Geometry > Primitive Feature > Cylinder...
5. Insert parameters as shown in Figure
6. Click [OK] Button



Step 3.



1. Geometry > Transform> Rotate...

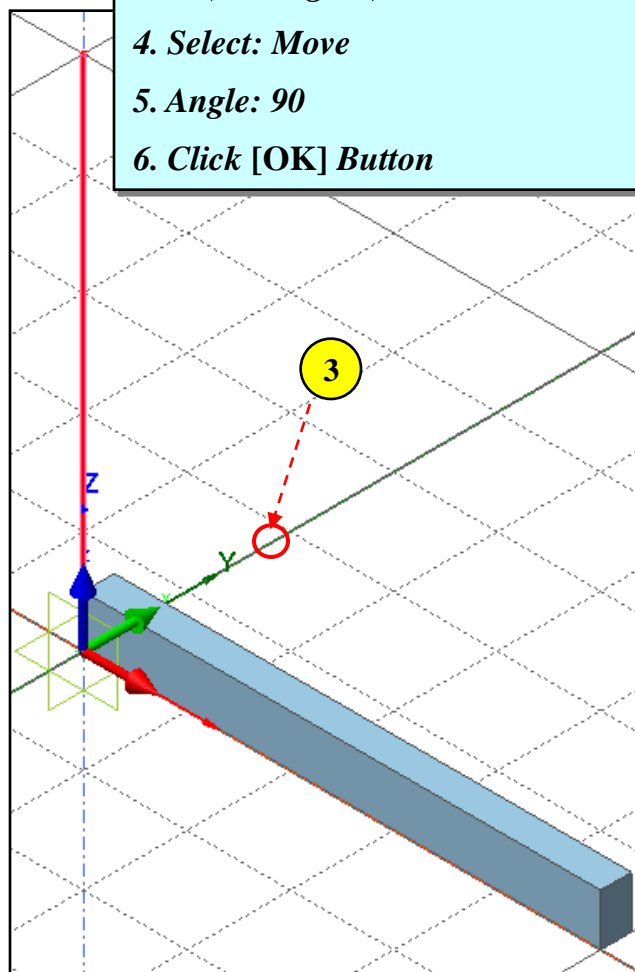
2. Select the generated Cylinder (highlighted in red) as an Object Shape

3. Select "Y - Axis" for Revolution Axis marked by "O" (See Figure)

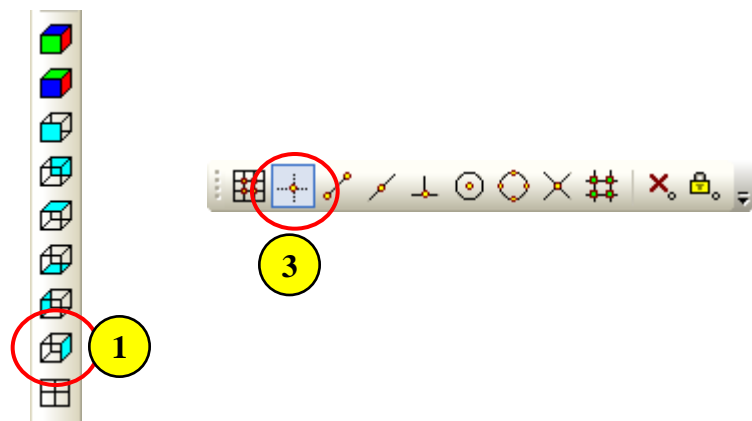
4. Select: Move

5. Angle: 90

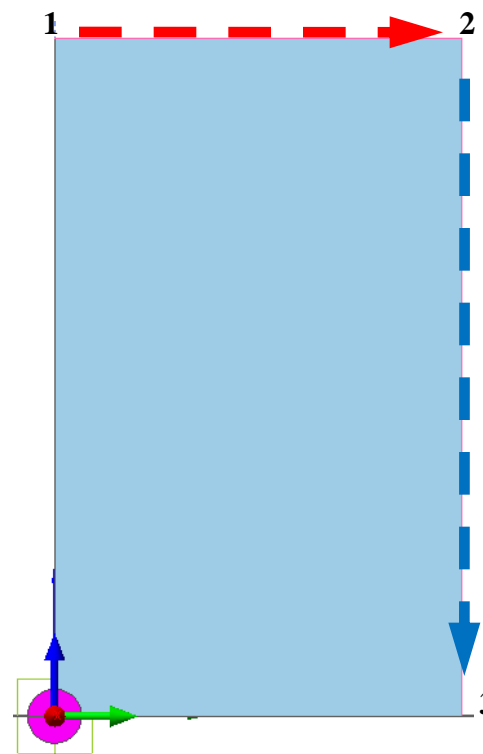
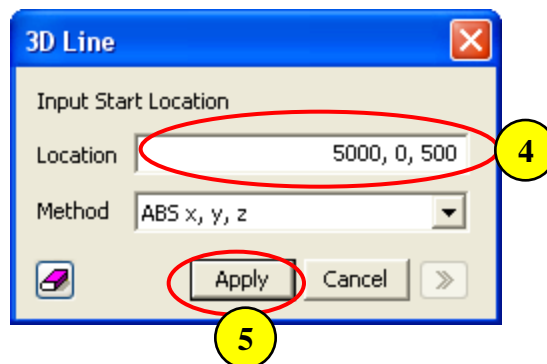
6. Click [OK] Button



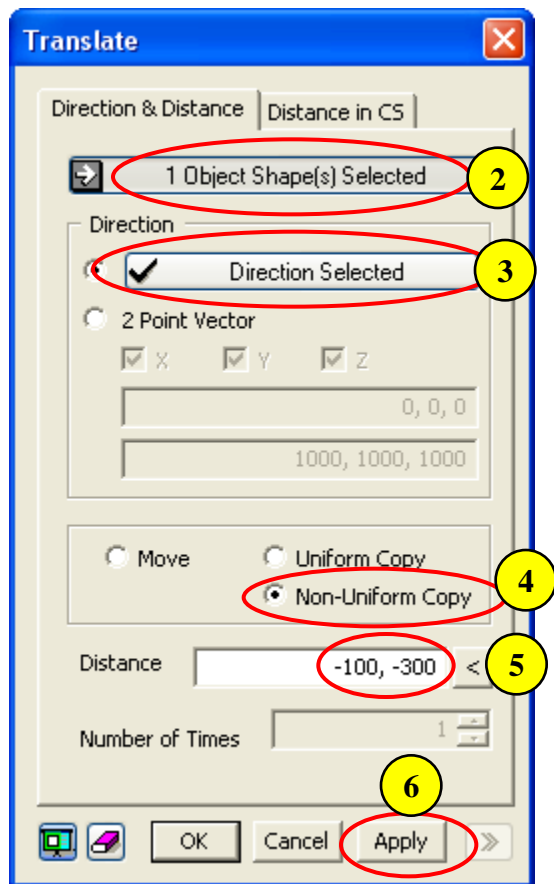
Step 4.



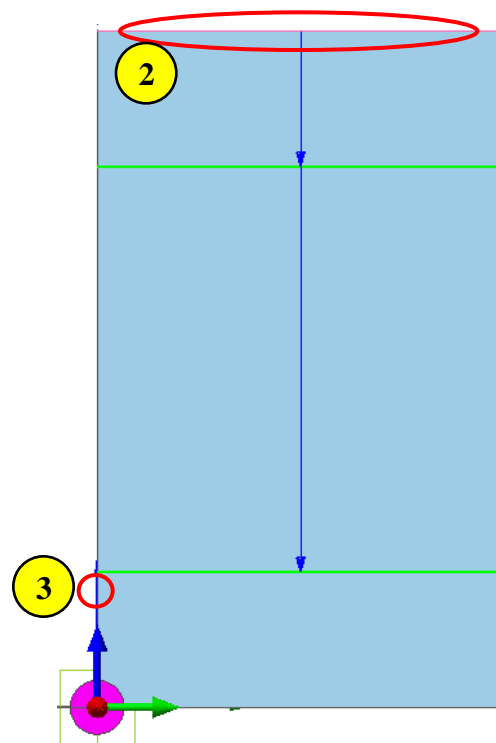
1. Click on “Right View”
2. Geometry > Curve > Create 3D> Line...
3. Select : Vertex Snap
4. Location : Node-1 to Node-2 and Node-2 to Node-3
5. Click [Apply] Button



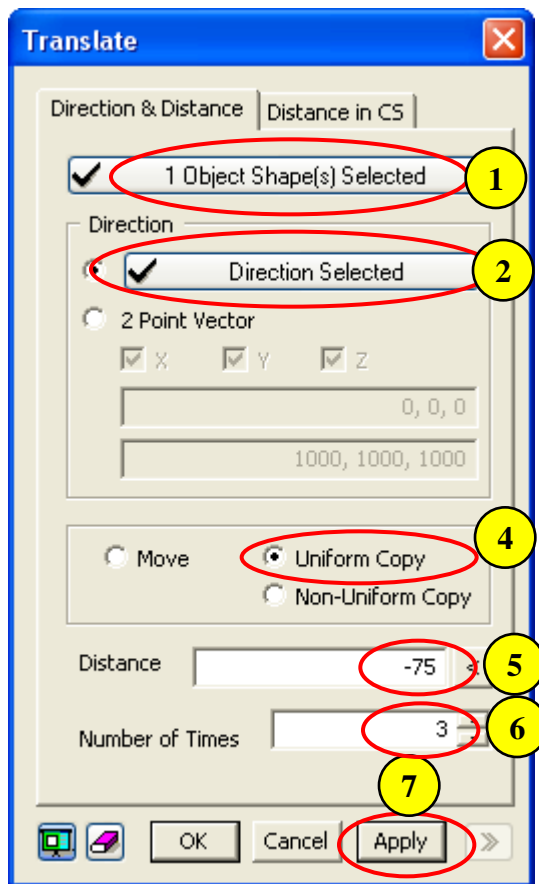
Step 5.



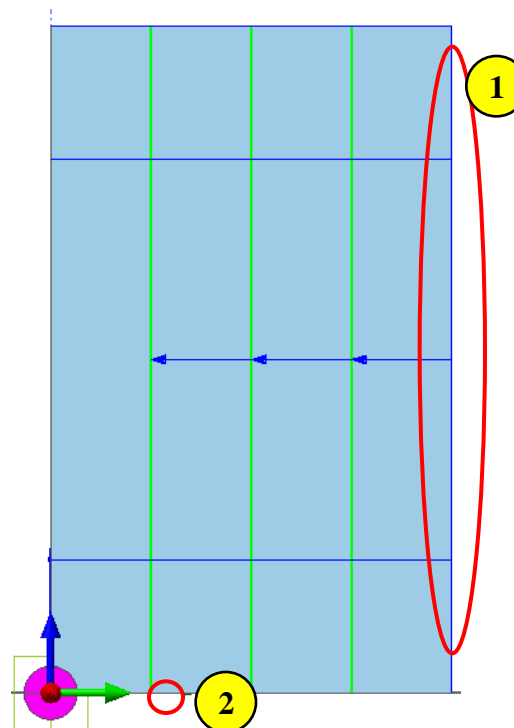
1. Geometry > Transform> Translate...
2. Select Top edge as shown in Figure
3. Select "Z - Axis" for Direction marked by "O" (See Figure)
4. Select: Non-Uniform Copy
5. Distance: "-100, -300"
6. Click [Apply] Button



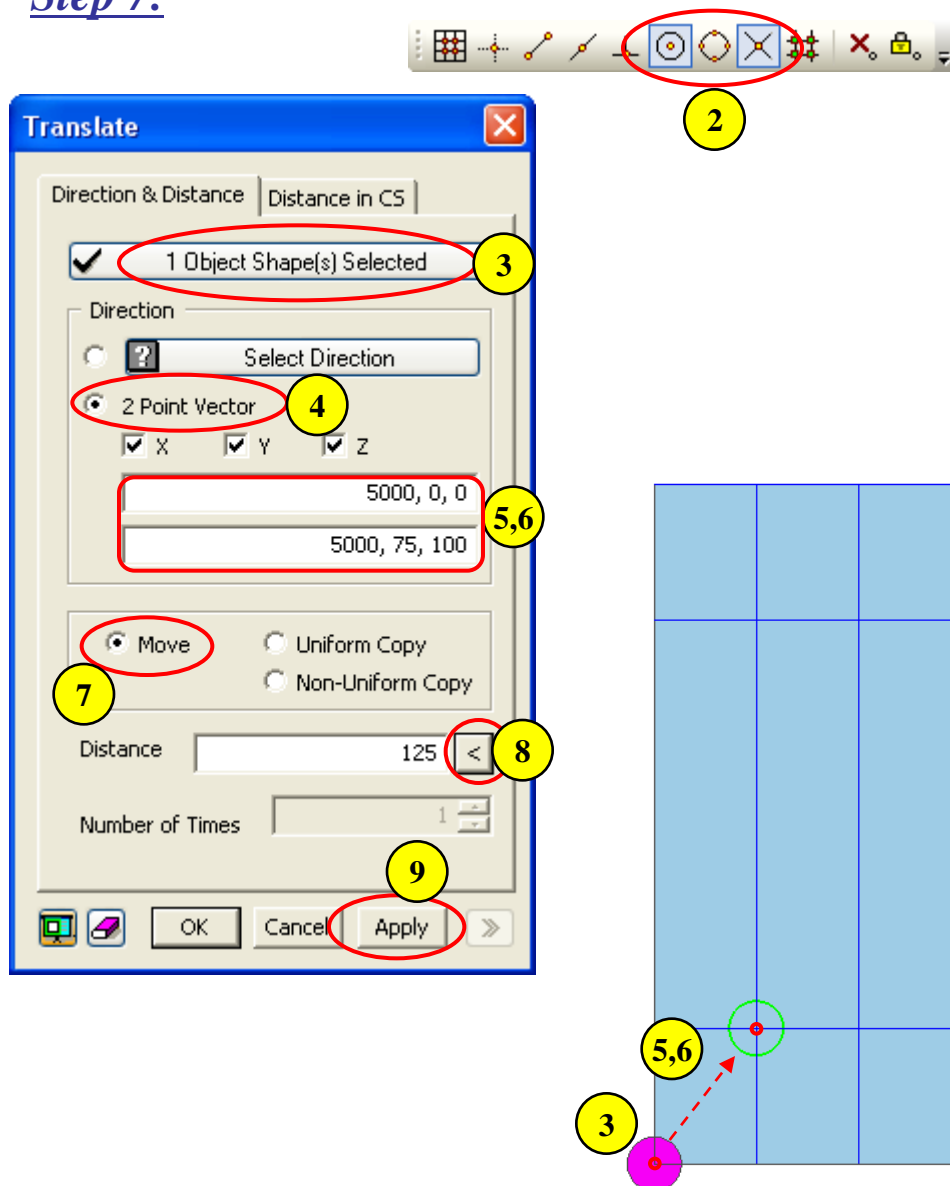
Step 6.




1. Select Right edge as shown in Figure
2. Select “Y - Axis” for Direction marked by “○” (See Figure)
4. Select: Uniform Copy
5. Distance: -75
6. Number of Times : 3
7. Click [Apply] Button

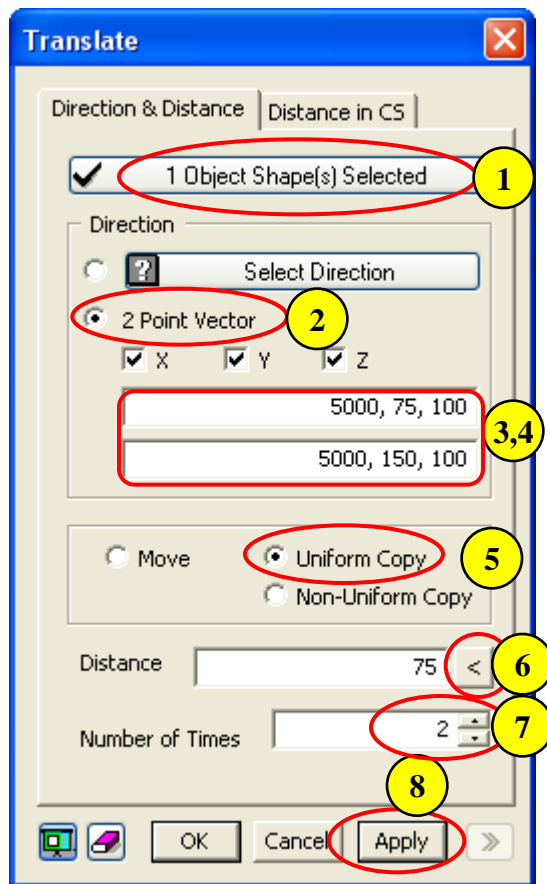



Step 7.

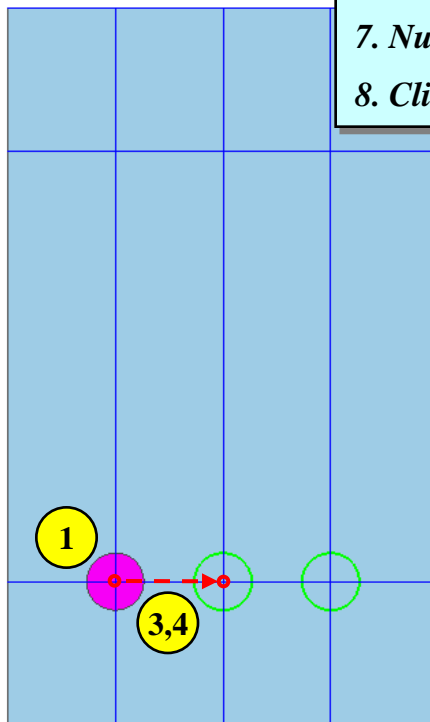


1. Geometry > Transform> Translate...
2. Select: Center and Intersection Snap buttons
3. Select the cylinder at the bottom left corner of the model as shown in Figure
4. Select "2 Point Vector" in "Direction" selection box
5. Click in the first box and select the center of cylinder
6. Click in the second box and select the marked intersection
7. Select: Move
8. Click on  to automatically calculate the distance between two assigned points
9. Click [Apply] Button

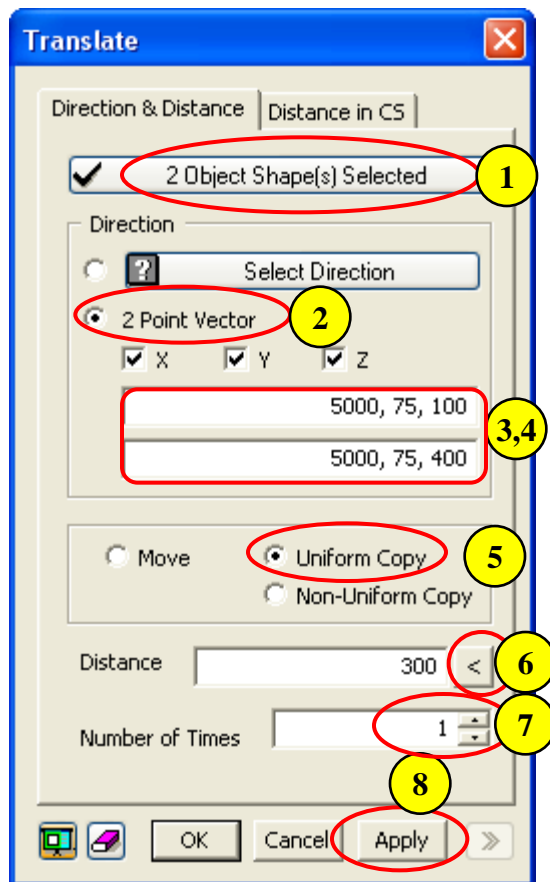
Step 8.




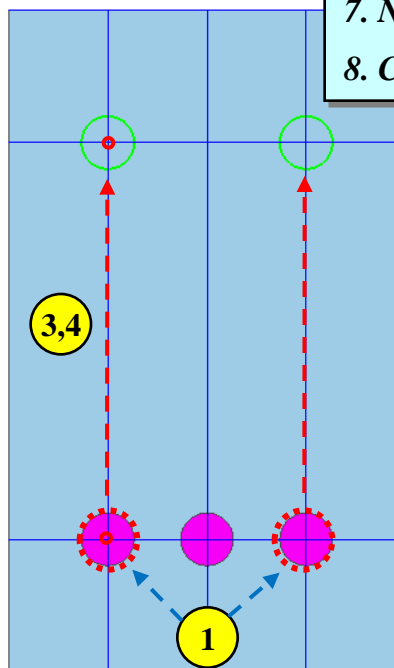
1. Select the cylinder as shown in Figure
2. Select "2 Point Vector" in "Direction" selection box
3. Click in the first box and select the center of cylinder
4. Click in the second box and select the marked intersection
5. Select: Uniform Copy
6. Click on  to automatically calculate the distance between two assigned points
7. Number of Times: 2
8. Click [Apply] Button



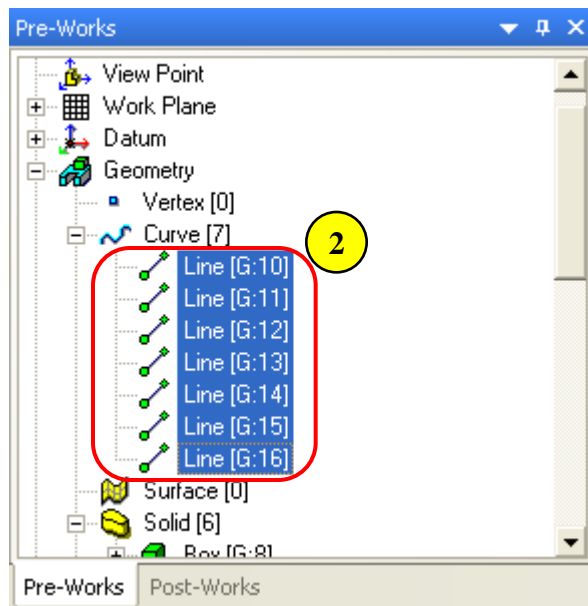
Step 9.



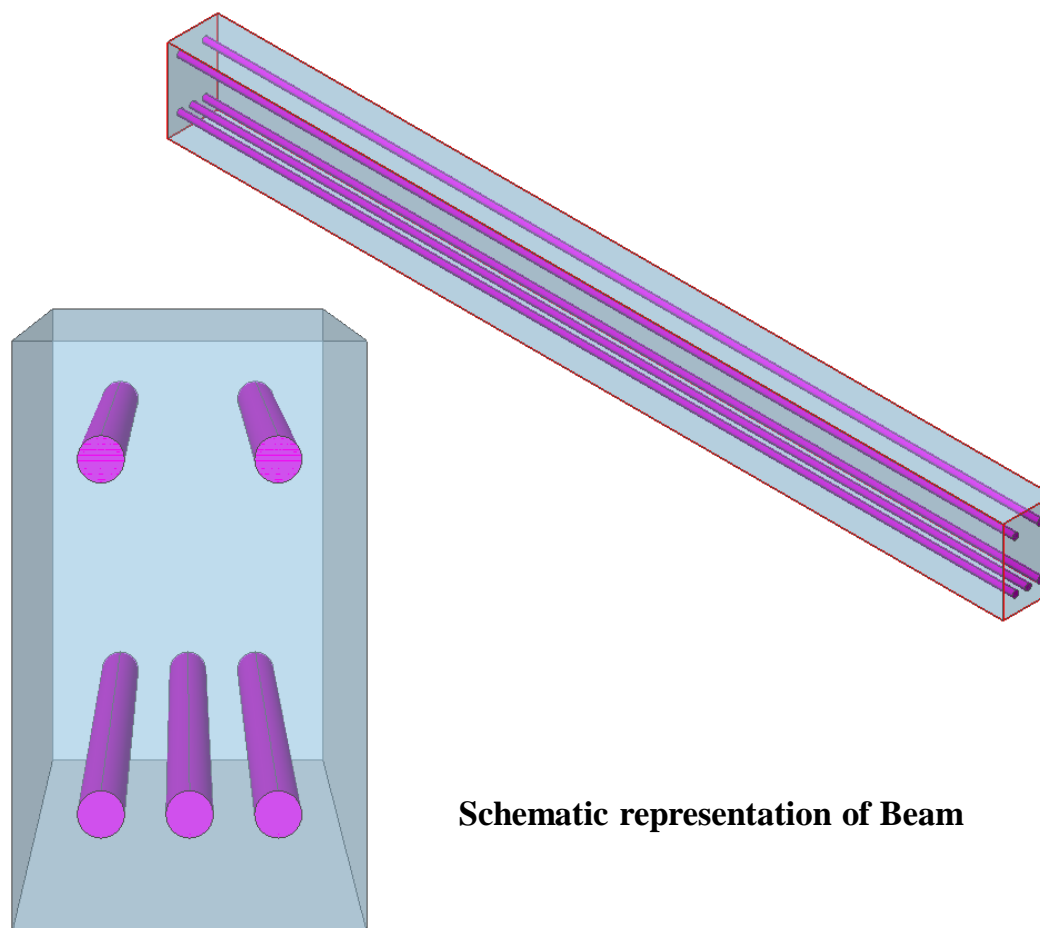
1. Select the highlighted cylinders as shown in Figure
2. Select "2 Point Vector" in "Direction" selection box
3. Click in the first box and select the center of cylinder
4. Click in the second box and select the marked intersection
5. Select: Uniform Copy
6. Click on  to automatically calculate the distance between two assigned points
7. Number of Times: 2
8. Click [Apply] Button



Step 10.

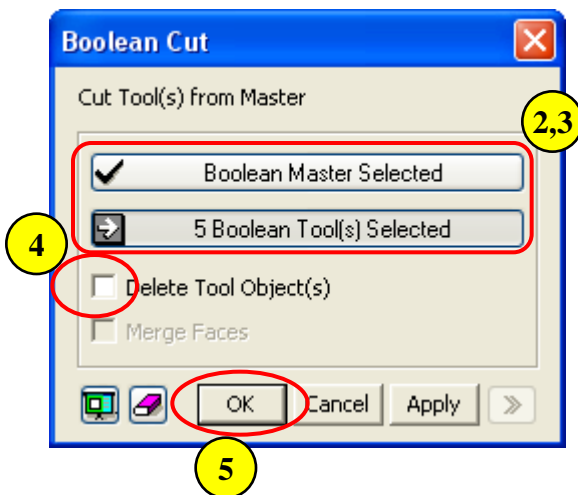


1. From Works-Tree select all lines under Curve category
2. Delete all the selected Lines

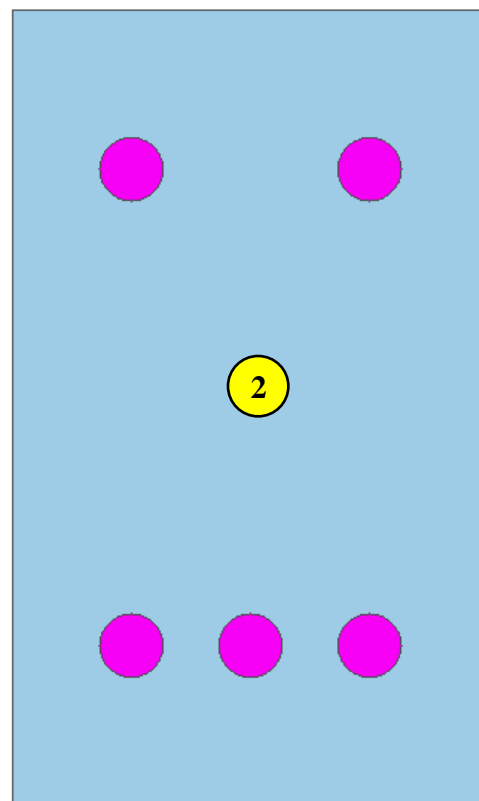


Schematic representation of Beam

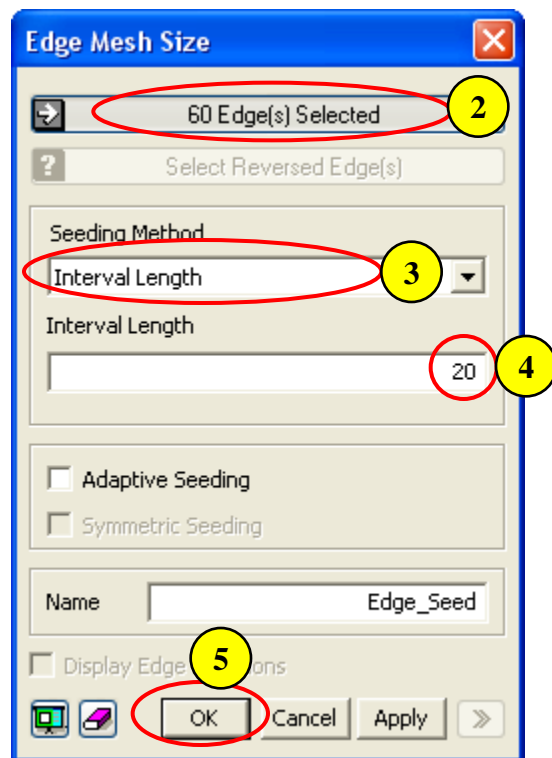
Step 11.



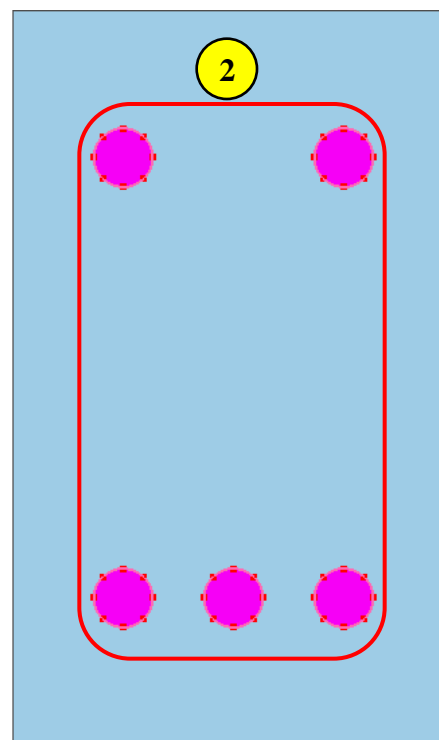
1. *Geometry > Boolean Operation > Cut...*
2. *Select the main body as shown in Figure*
3. *Select 5 cylinders for Boolean Tools*
4. *Unselect: Delete Tool Object(s)*
5. *Click [OK] Button*



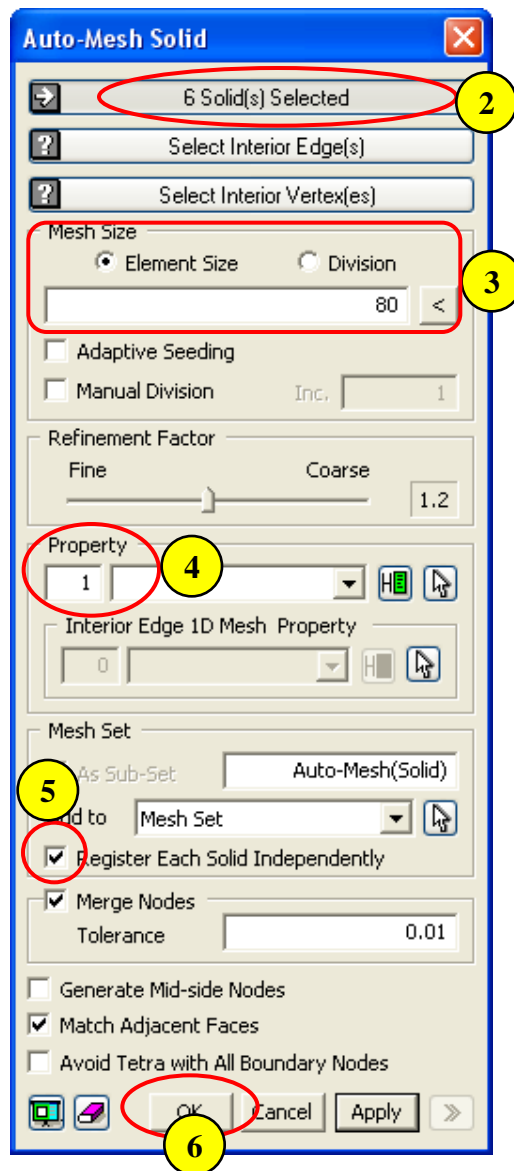
Step 12.



1. Mesh > Size Control > Along Edge...
2. Select all edges of cylinders
3. Seeding Method: Interval Length
4. Interval Length: 20
5. Click [OK] Button



Step 13.



1. Mesh > Auto Mesh > Solid...

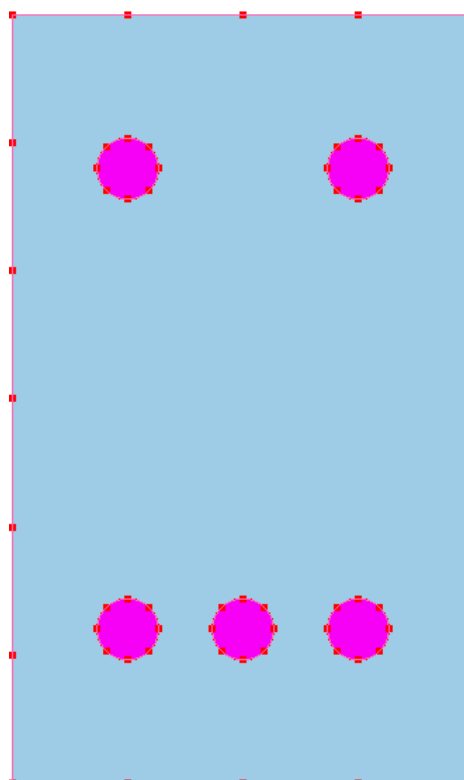
2. Select: All

3. Mesh Size: Element Size (80)

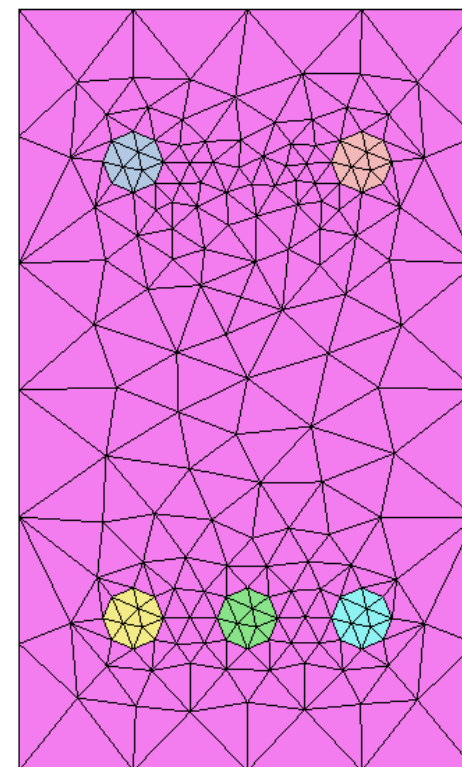
4. Property : 1

5. Select: Register Each Solid Independently

6. Click [OK] Button

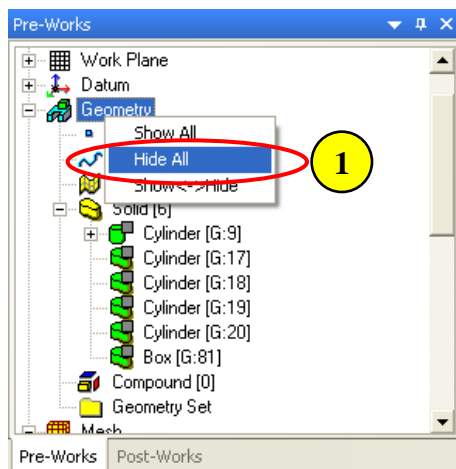



Model with assigned seeds

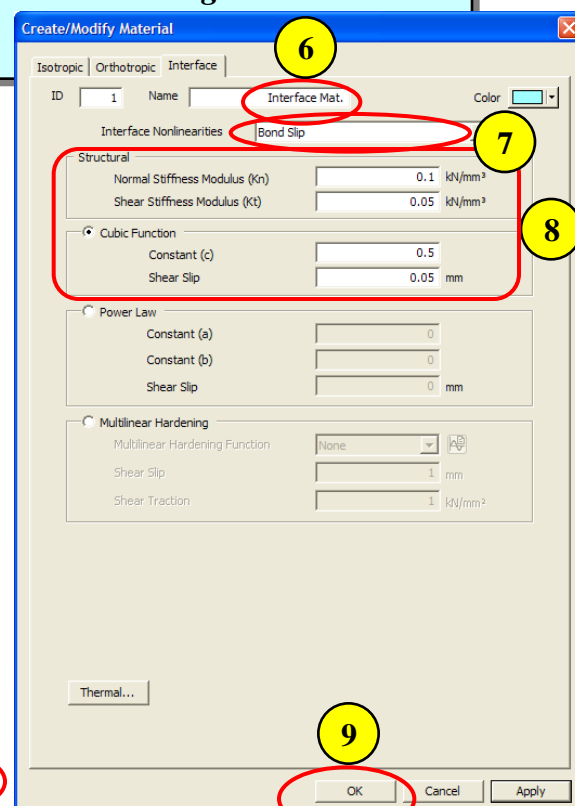
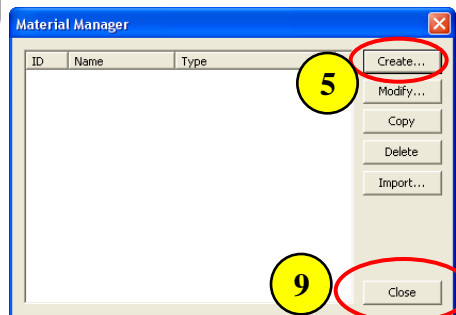
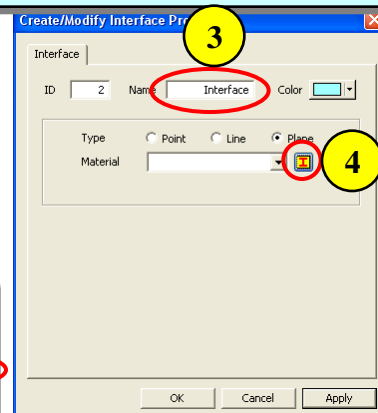
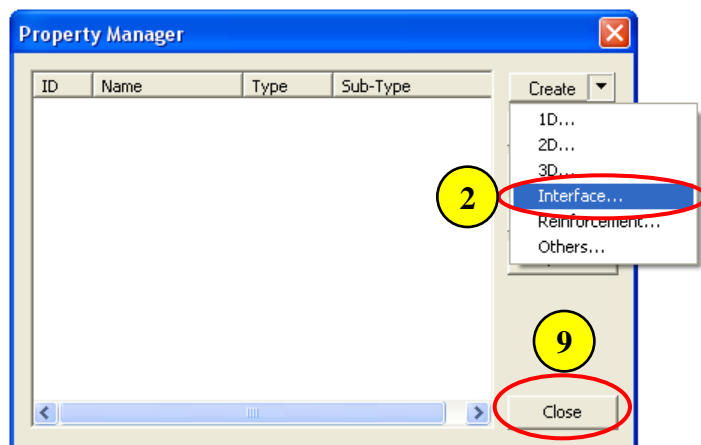


Generated mesh

Step 14.

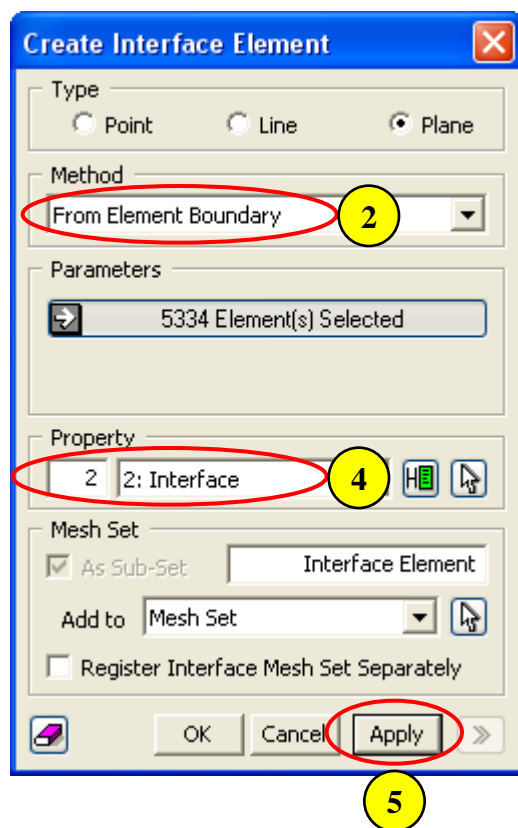


1. From Works-Tree right click on “Geometry” and select “Hide All”
2. Analysis > Property... > Create > Interface...
3. Name: Interface
4. Click on  to define associated material model
5. Click on “Create” and select Interface Tab
6. Name: Interface Mat.
7. Interface Nonlinearities: Bond Slip
8. Insert corresponding parameters as shown in Figure*
9. Click [OK] and [Cancel] Buttons



* The assigned parameters are just for example and there is no specific reference for them.

Step 15.



1. Mesh > Element > Create Interface Element...

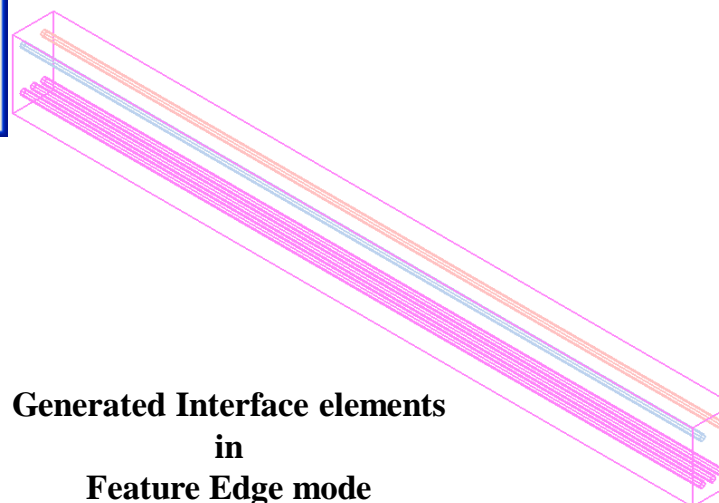
2. Method: From Element Boundary

3. Parameters: Select each cylinder separately (see Figure)

4. Property: 2:Interface

5. Click [Apply] Button

6. Repeat the same procedure to create other interface elements



Generated Interface elements
in
Feature Edge mode

