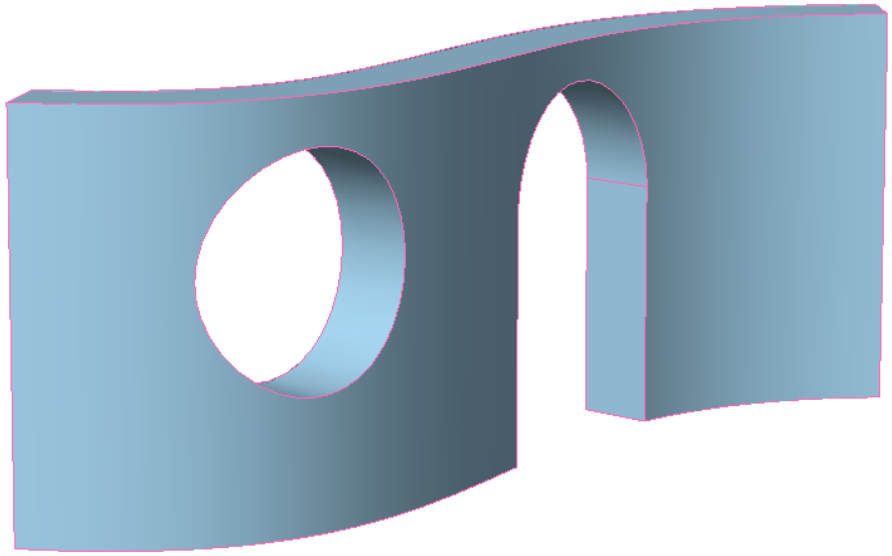
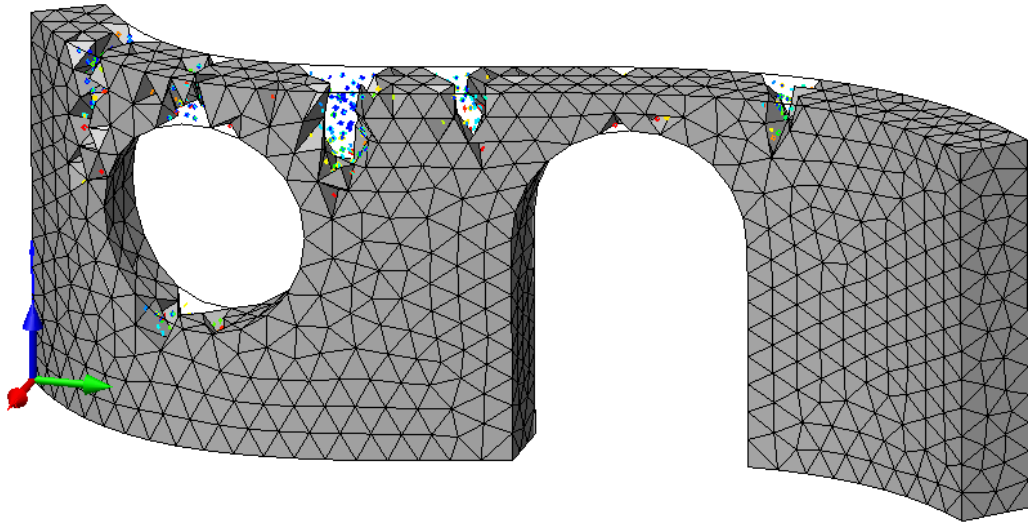
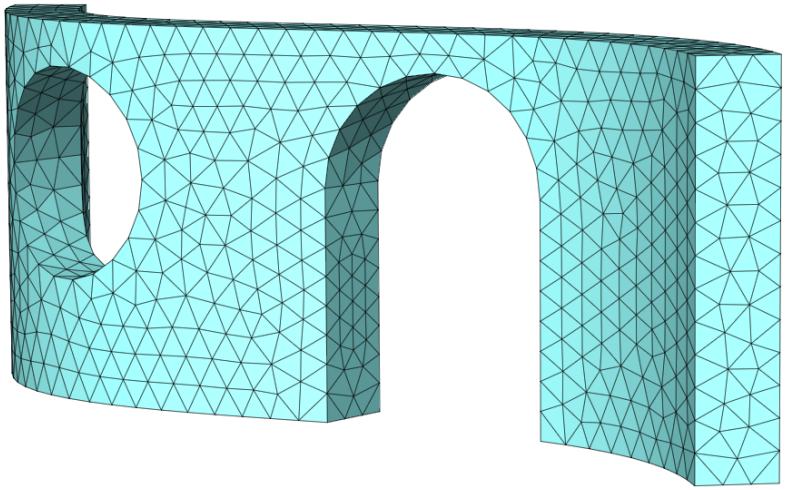
NL-5. Crack analysis for a curved wall



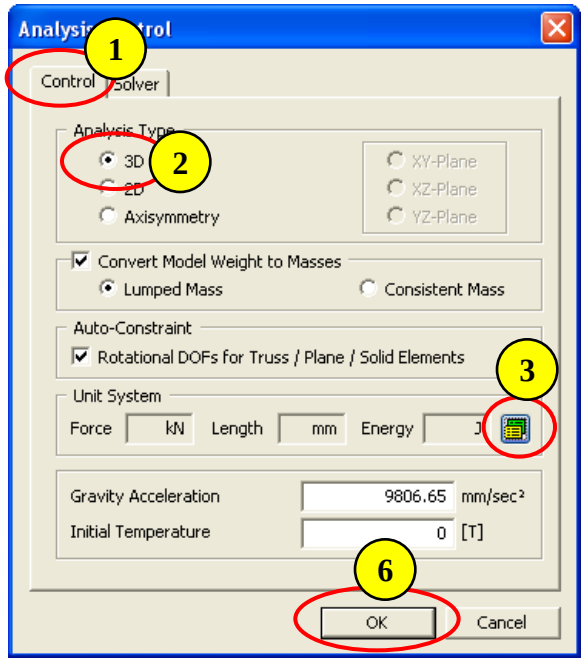
Overview

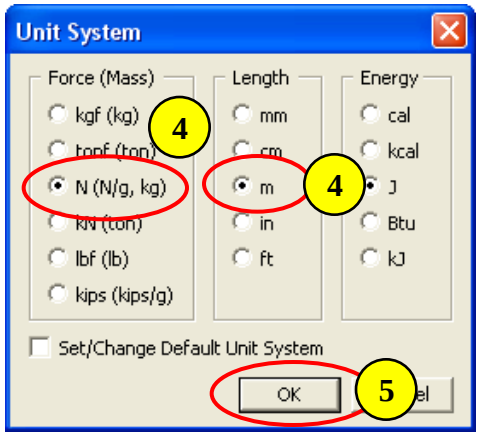
- 3D Nonlinear Analysis
- Model
  - *A curved wall with openings*
  - *TSC material model*
  - *Unit : N, m*
- Load & Boundary Condition
  - *Uniform pressure*
  - *Simply supported at bottom*
- Result Evaluation
  - *Principal Strain*
  - *Crack Pattern*



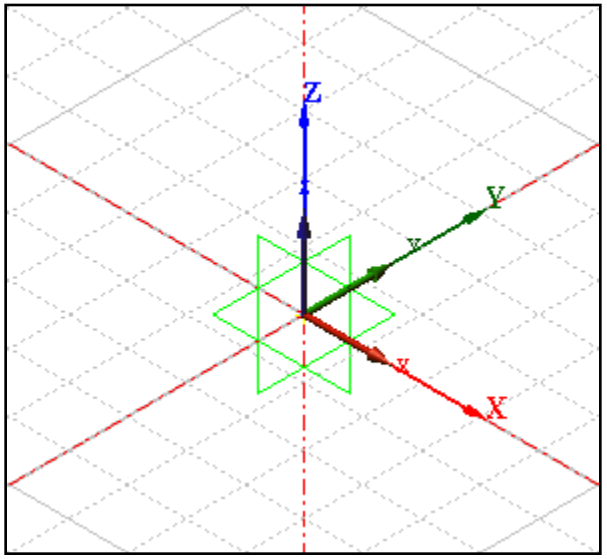


Step 1.





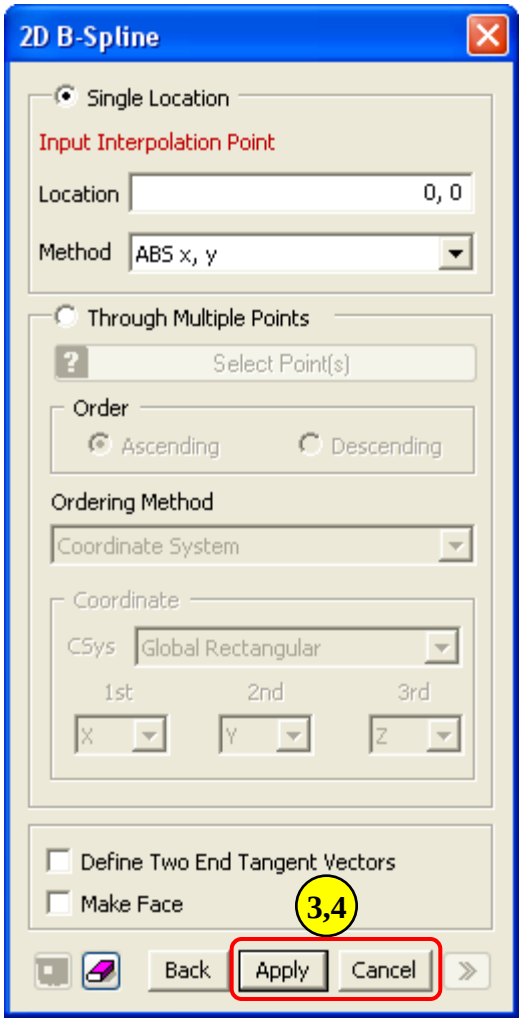
1. *Analysis > Analysis Control – "Control" tab*
2. *Analysis Type : 3D*
3. *Click* Button
4. *Unit : N , m*
5. *Click* [OK] *Button*
6. *Click* [OK] *Button*

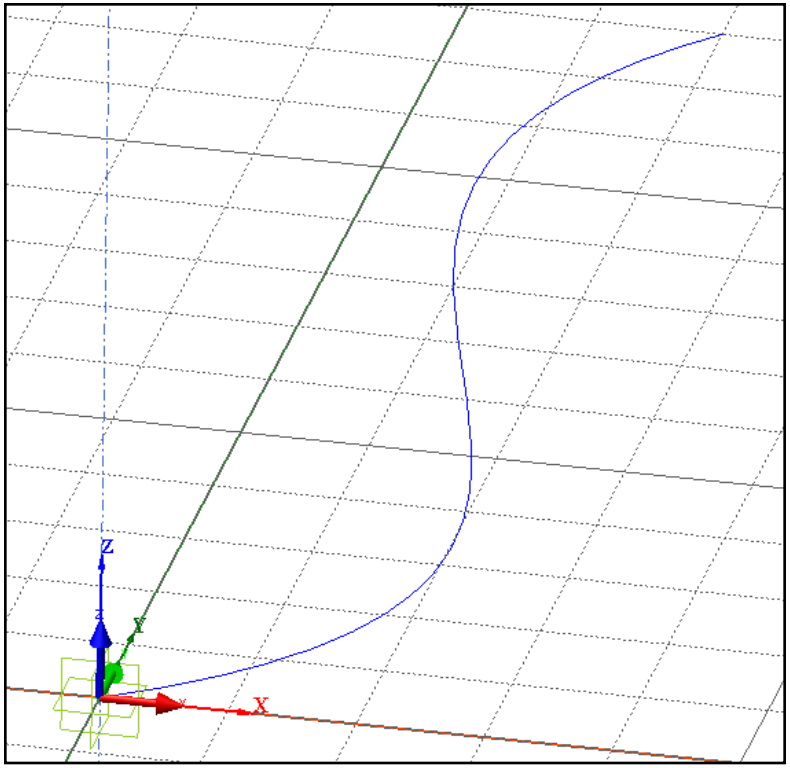


Analysis Control Dialog is automatically activated at startup.

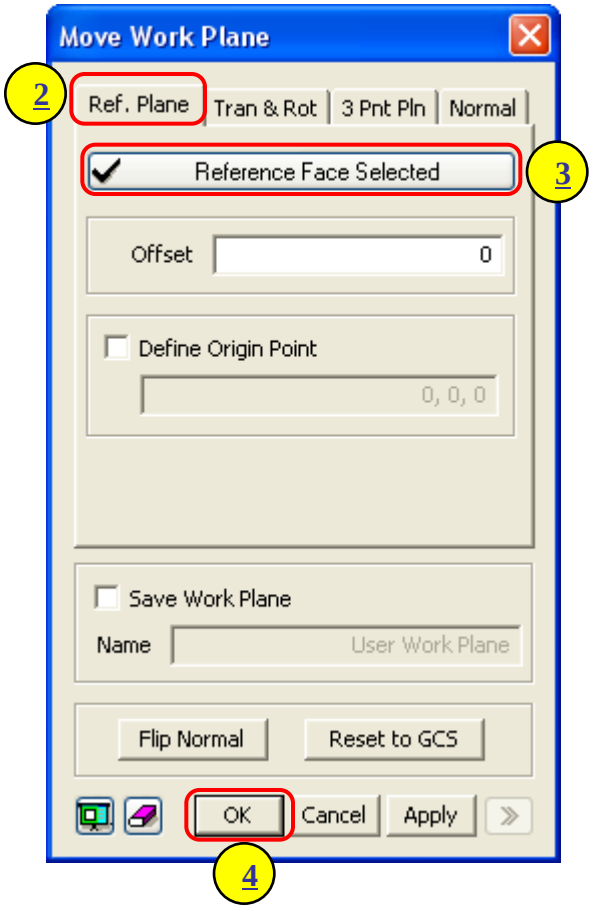
Step 2.

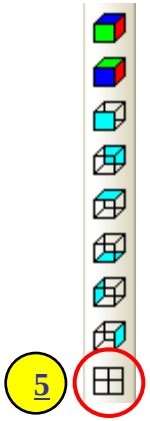
1. *Geometry > Curve > Create on WP > B-Spline…*
2. *Draw an arbitrary B-Spline as shown in the picture*
3. *Click* **[Apply]** *Button*
4. *Click* **[Cancel]** *Button*



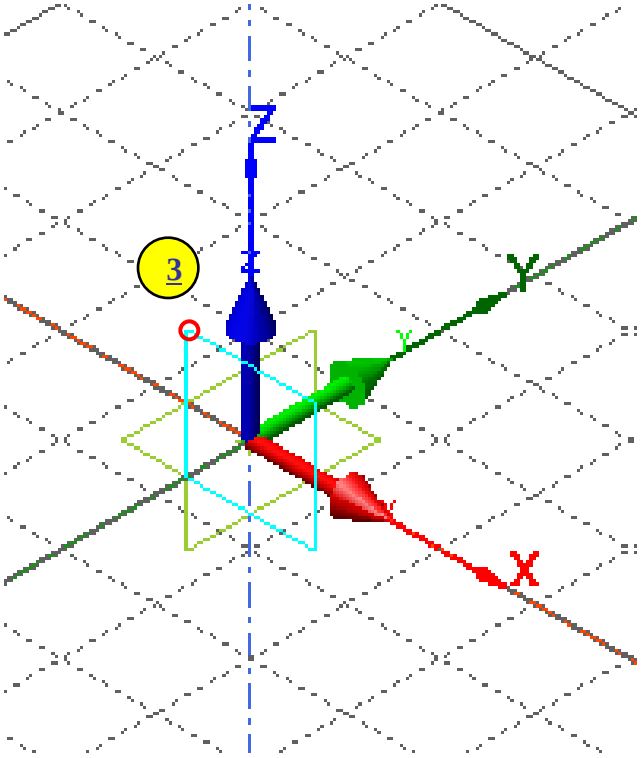


Step 3.



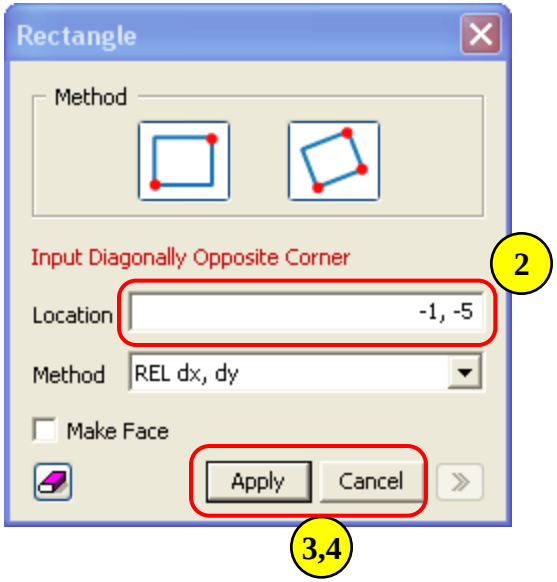


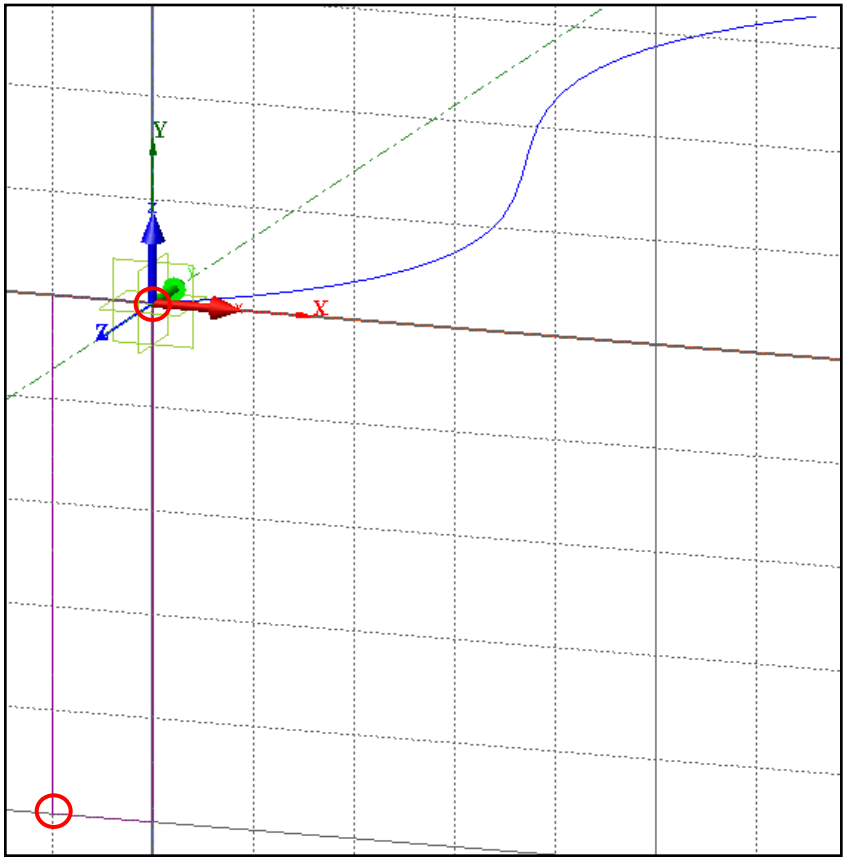
1. Geometry > Work Plane > Move…
2. Select the “Ref. Plane” tab
3. Select “XZ” plane as reference face
4. Click on [OK] Button
5. Toggle on “Normal” view



Step 4.

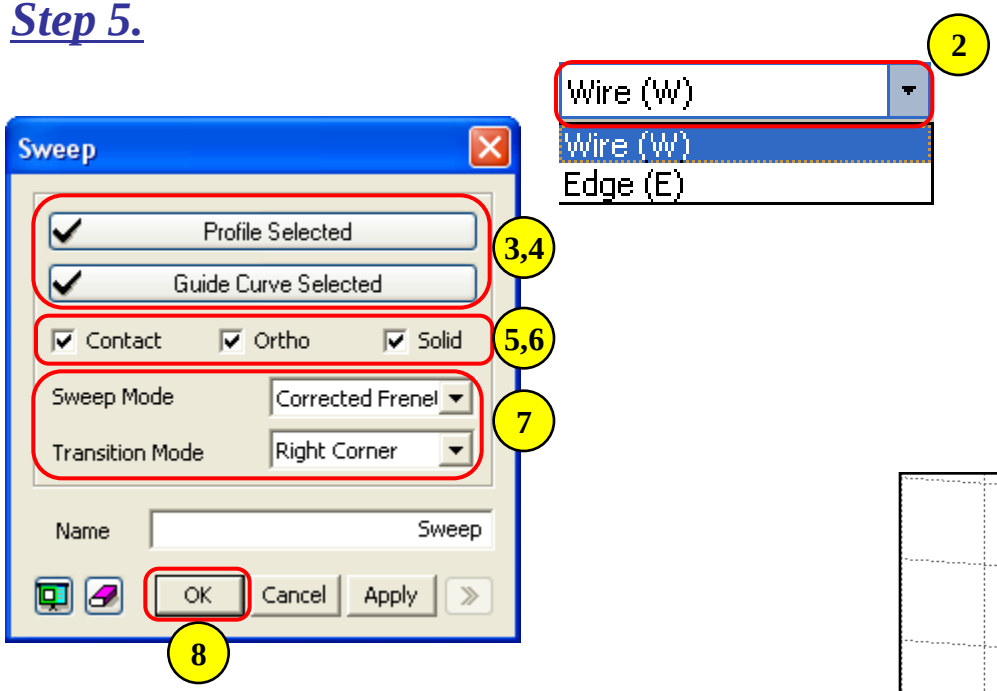
1. *Geometry > Curve > Create on WP > Rectangle (Wire)…*
2. *Location: (0,0), <-1,-5>*
3. *Click* **[Apply]** *Button*
4. *Click* **[Cancel]** *Button*



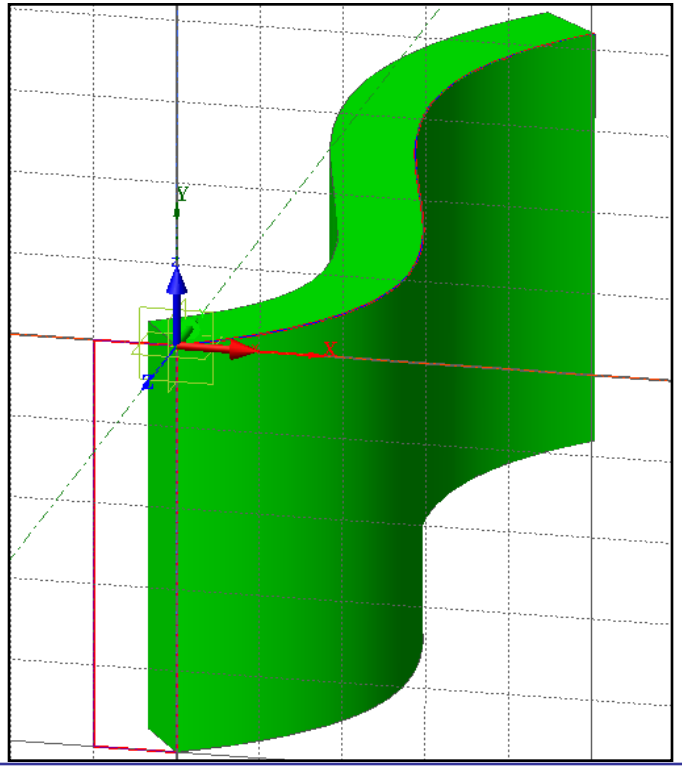


- *() : "ABS x, y" , <> : "REL dx, dy"*
  (0) same as (0, 0)
- *[Esc] as shortcut for [Cancel]*

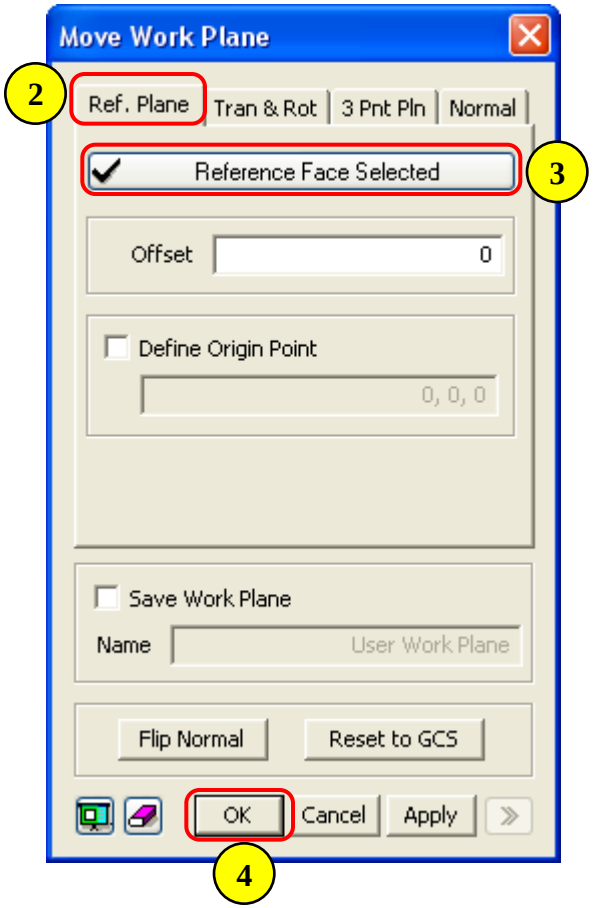
Step 5.

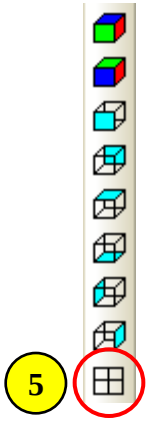


1. *Geometry > Generator Feature > Sweep…*
2. *Change the selecting filter to "Wire (W)"*
3. *Select the rectangle as Profile*
4. *Select the B-SPline as guide curve*
5. *Check on Solid, Ortho, Contact*
6. *Sweep Mode: Corrected Frenet*
7. *Transition Mode: Right Corner*
8. *Click* **[OK]** *Button*

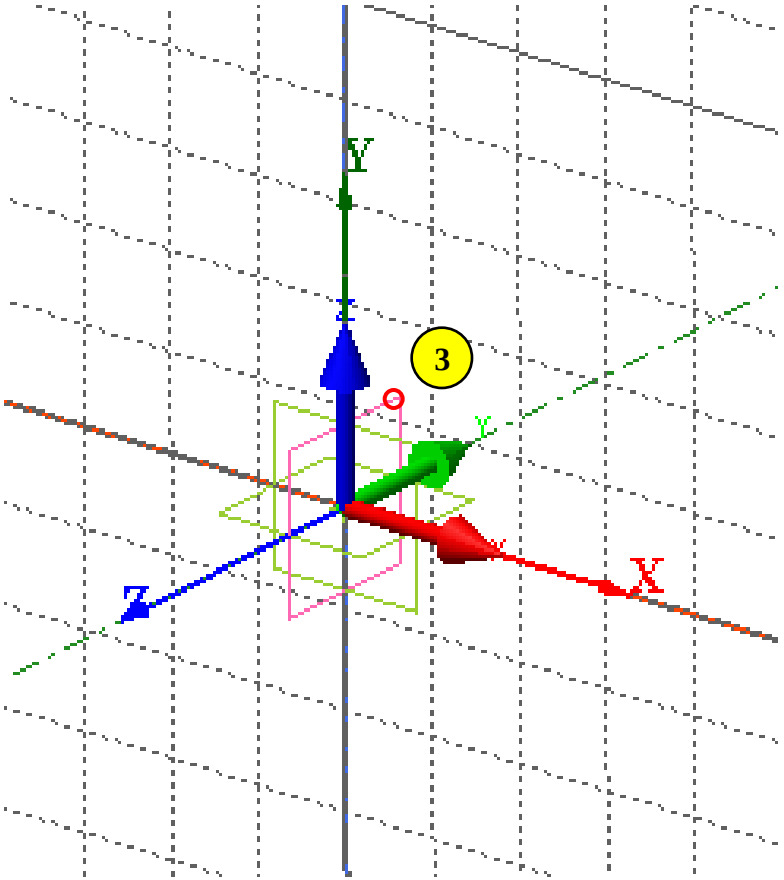


Step 6.

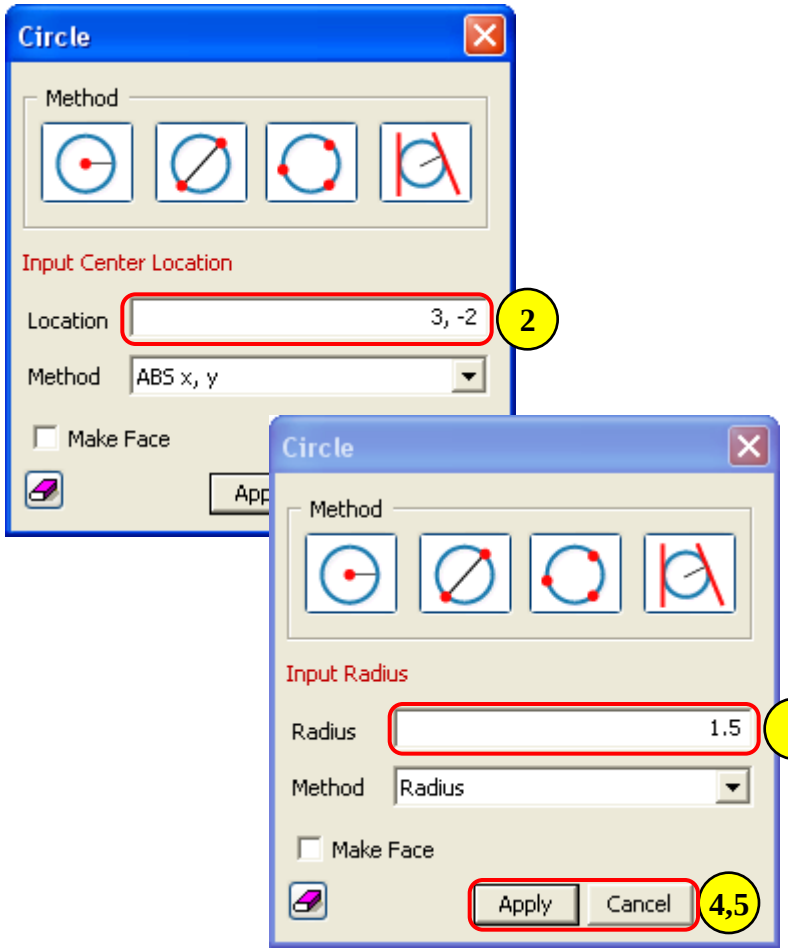




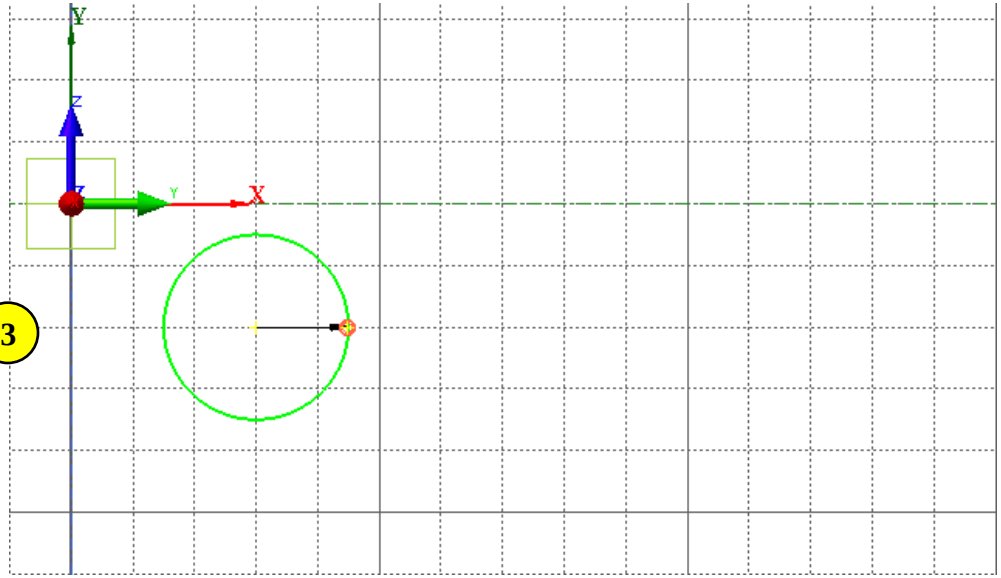
1. *Geometry > Work Plane > Move…*
2. *Select the "Ref. Plane" tab*
3. *Select "YZ" plane as reference face*
4. *Click on [OK] Button*
5. *Toggle on "Normal" view*



Step 7.

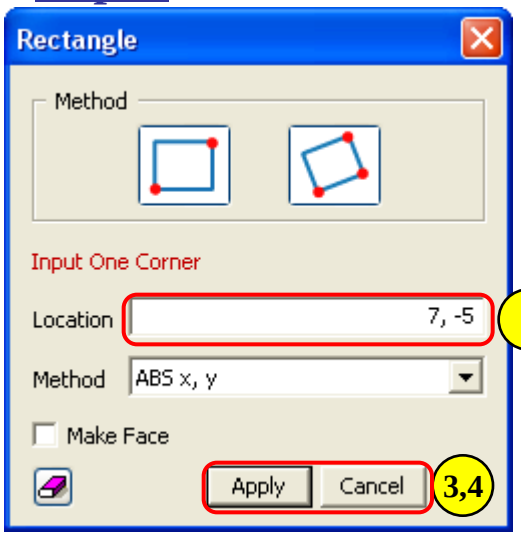


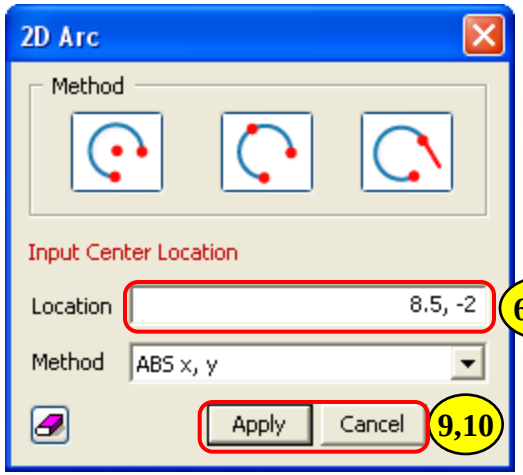
1. *Geometry > Curve > Create on WP > Circle …*
2. *Location: (3,-2)*
3. *Radius: 1.5*
4. *Click* **[Apply]** *Button*
5. *Click* **[Cancel]** *Button*



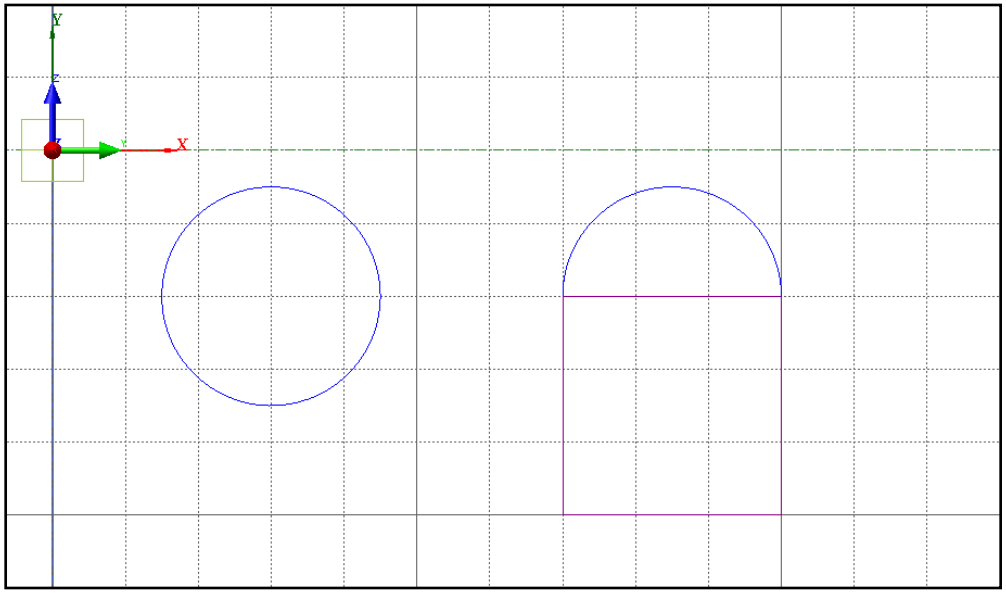
- *() : "ABS x, y" , <> : "REL dx, dy"*
  (0) same as (0, 0)
- *[Esc] as shortcut for [Cancel]*

Step 8.



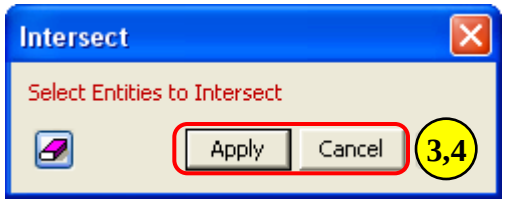


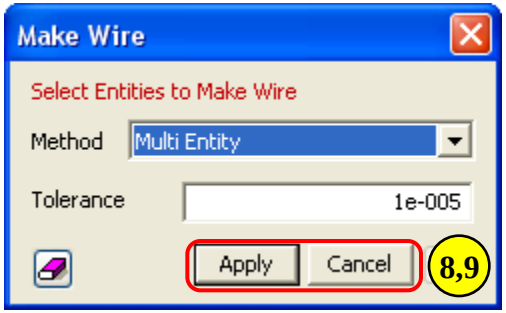
1. *Geometry > Curve > Create on WP > Rectangle (Wire) …*
2. *Location: (7,-5), <3,3>*
3. *Click* **[Apply]** *Button*
4. *Click* **[Cancel]** *Button*
5. *Geometry > Curve > Create on WP > Arc …*
6. *Location (Center): (8.5, -2)*
7. *Start Location: (10,-2)*
8. *End Location: (7,-2)*
9. *Click* **[Apply]** *Button*
10. *Click* **[Cancel]** *Button*

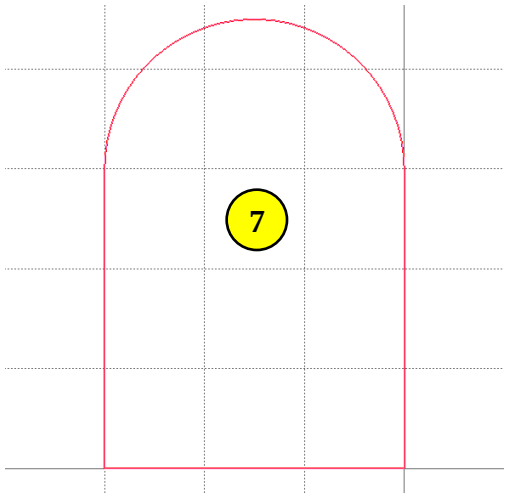


- *() : "ABS x, y" , <> : "REL dx, dy"*
  (0) same as (0, 0)
- *[Esc] as shortcut for [Cancel]*

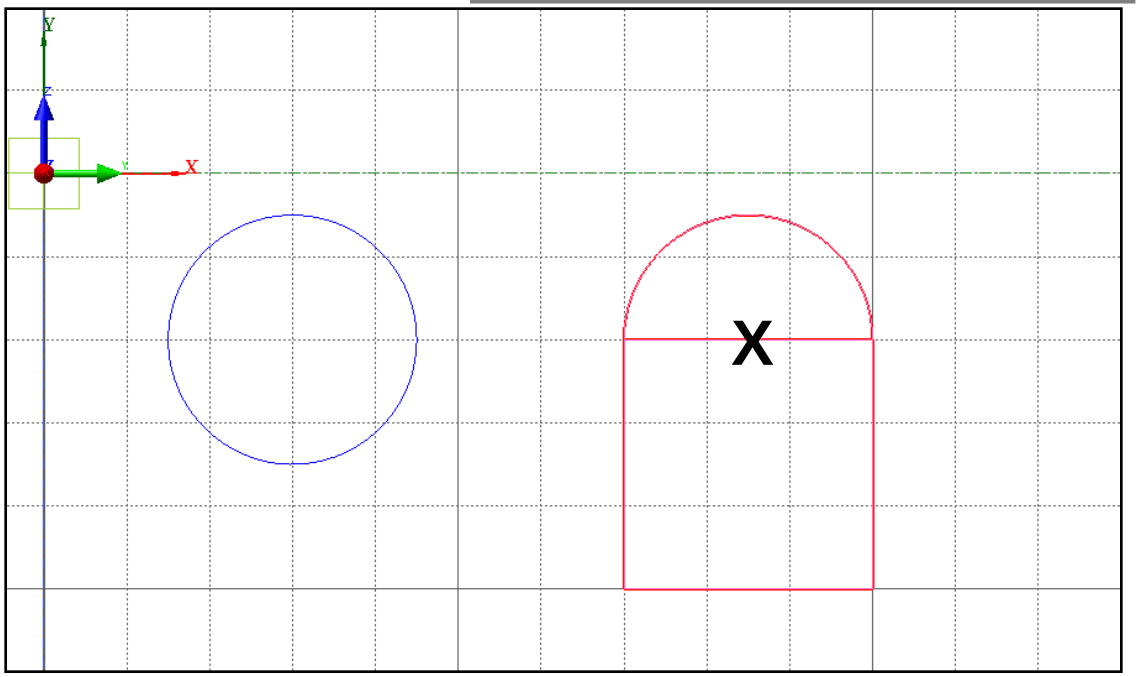
Step 9.



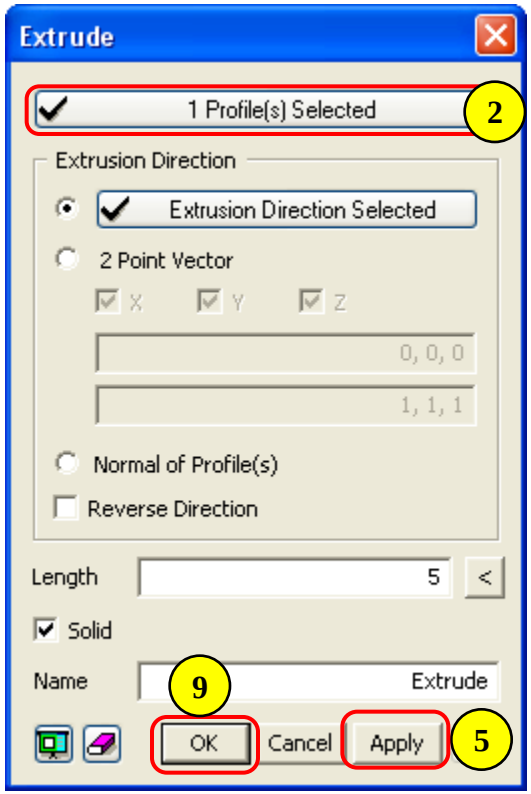


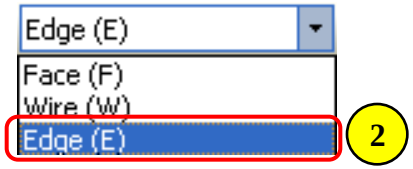


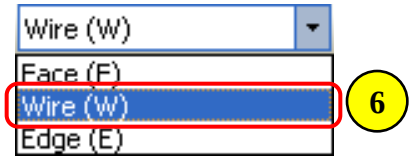
1. *Geometry > Curve > Intersect …*
2. *Select the rectangle and arc*
3. *Click* **[Apply]** *Button*
4. *Click* **[Cancel]** *Button*
5. *Select the highlighted edge and delete it*
6. *Geometry > Curve > Make Wire …*
7. *Select the edges as shown in the picture*
8. *Click* **[Apply]** *Button*
9. *Click* **[Cancel]** *Button*



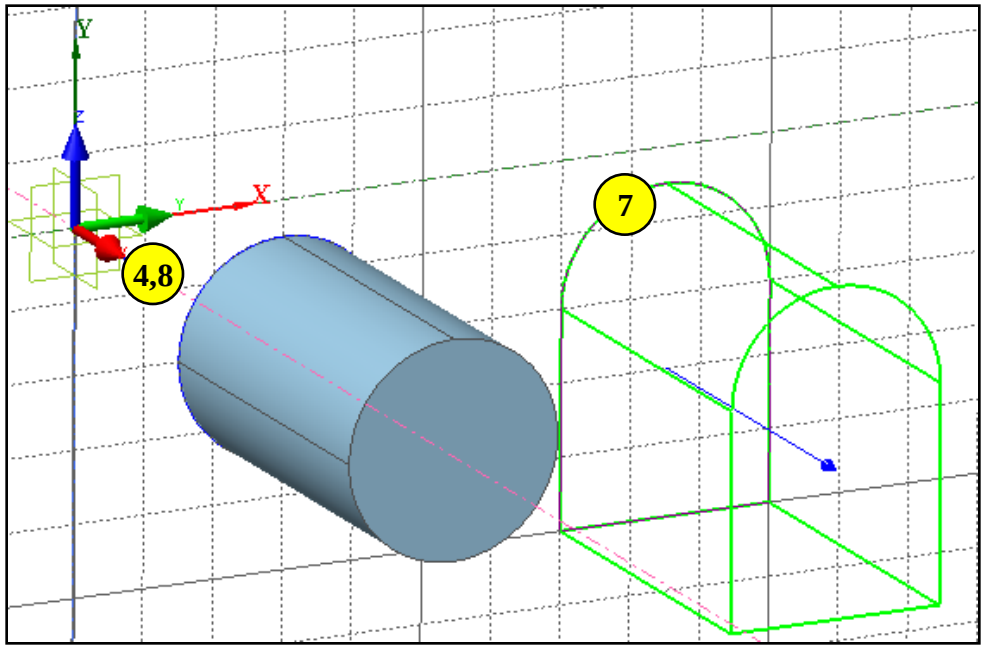
Step 10.





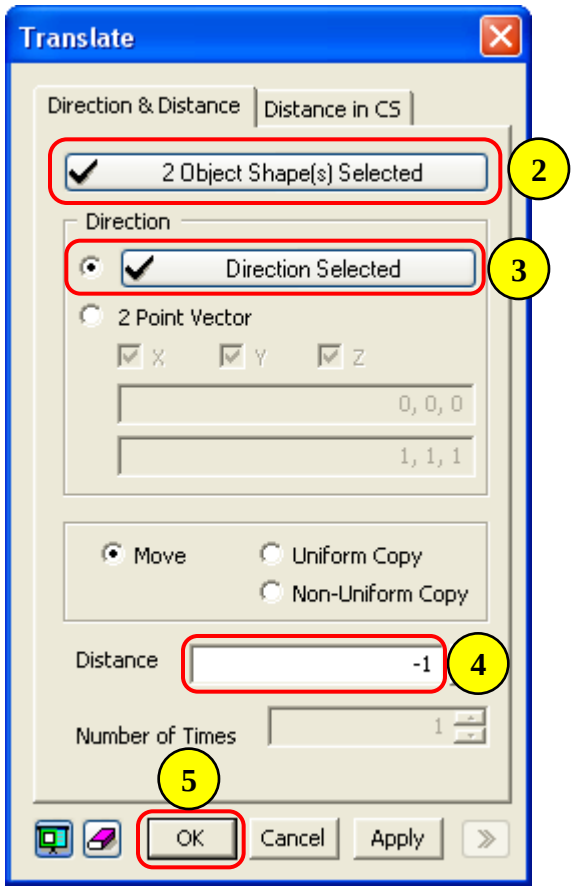


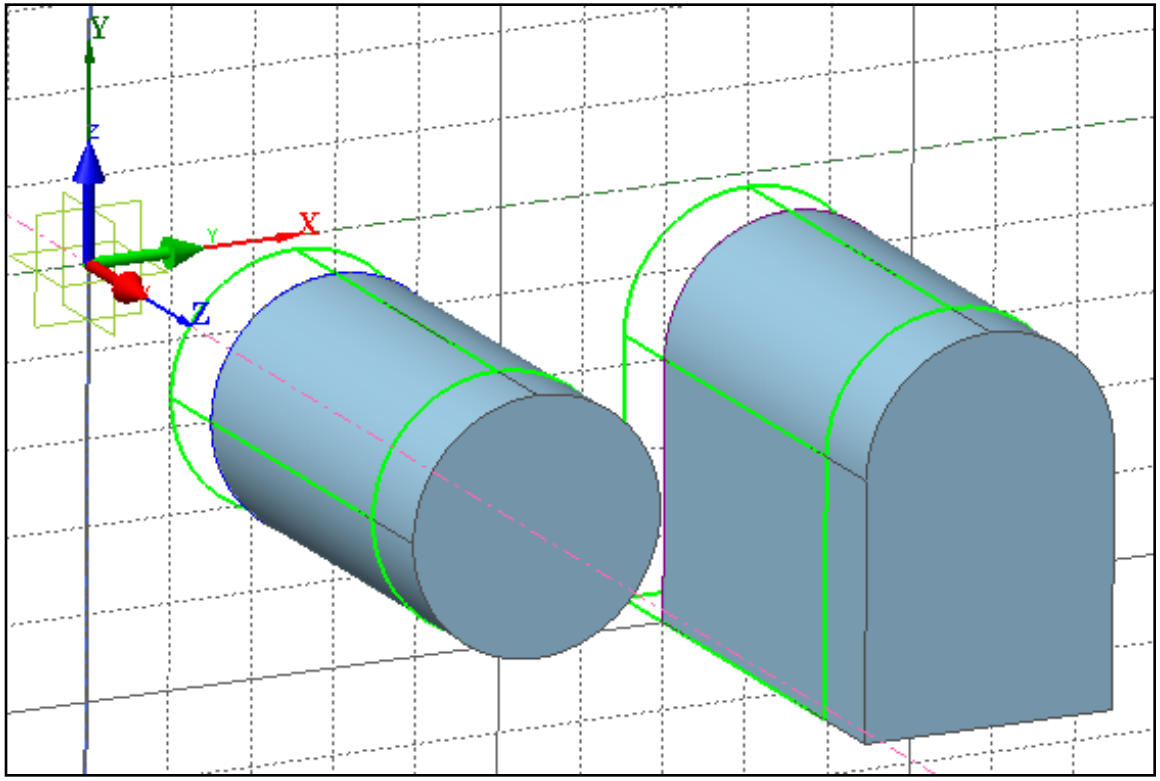
1. *Geometry > Generator Feature > Extrude…*
2. *Change the selecting filter to "Edge (E)"*
3. *Select the circle*
4. *Extrusion Direction: Axis-X*
5. *Click* **[Apply]** *Button*
6. *Change the selecting filter to "Wire (W)"*
7. *Select the wire as shown in the picture*
8. *Extrusion Direction: Axis-X*
9. *Click* **[OK]** *Button*



Step 11.

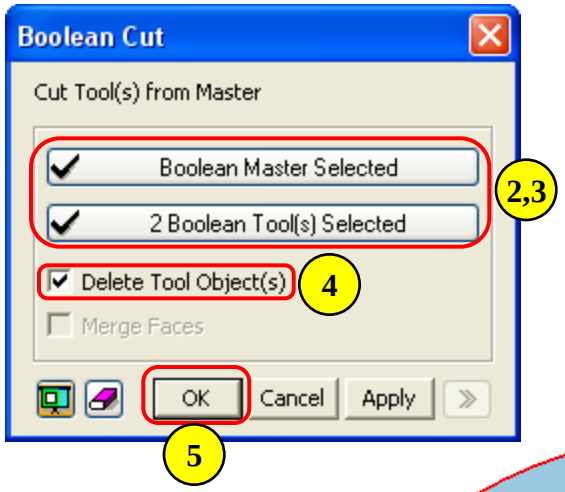
1. *Geometry > Transform > Translate …*
2. *Select two extruded solids*
3. *Direction: Axis-X*
4. *Distance: -1*
5. *Click* **[OK]** *Button*

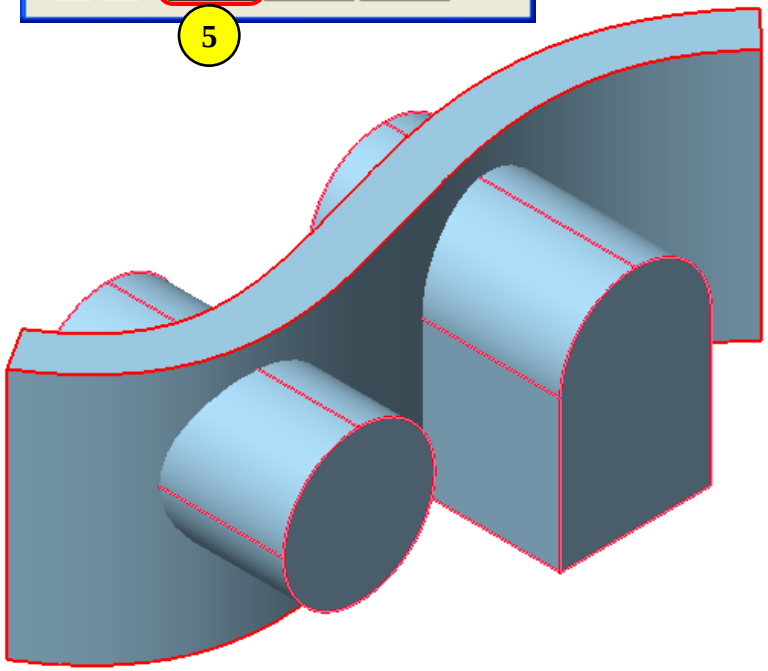


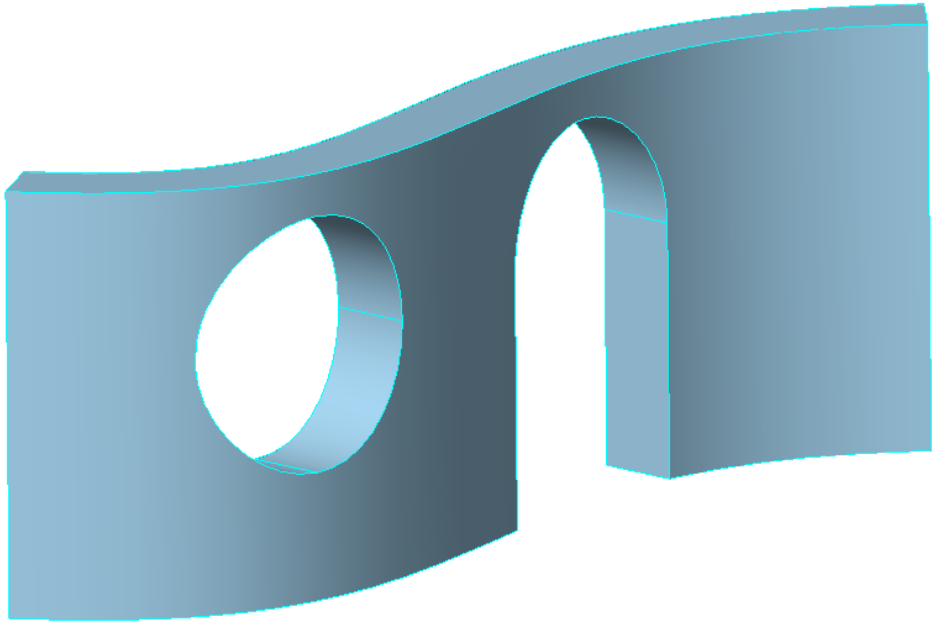


Step 12.

1. *Geometry > Boolean Operation > Cut …*
2. *Select the curved wall as Boolean Master*
3. *Select two extruded solids as Boolean Tools*
4. *Check on Delete Tool Object(s)*
5. *Click* **[OK]** *Button*



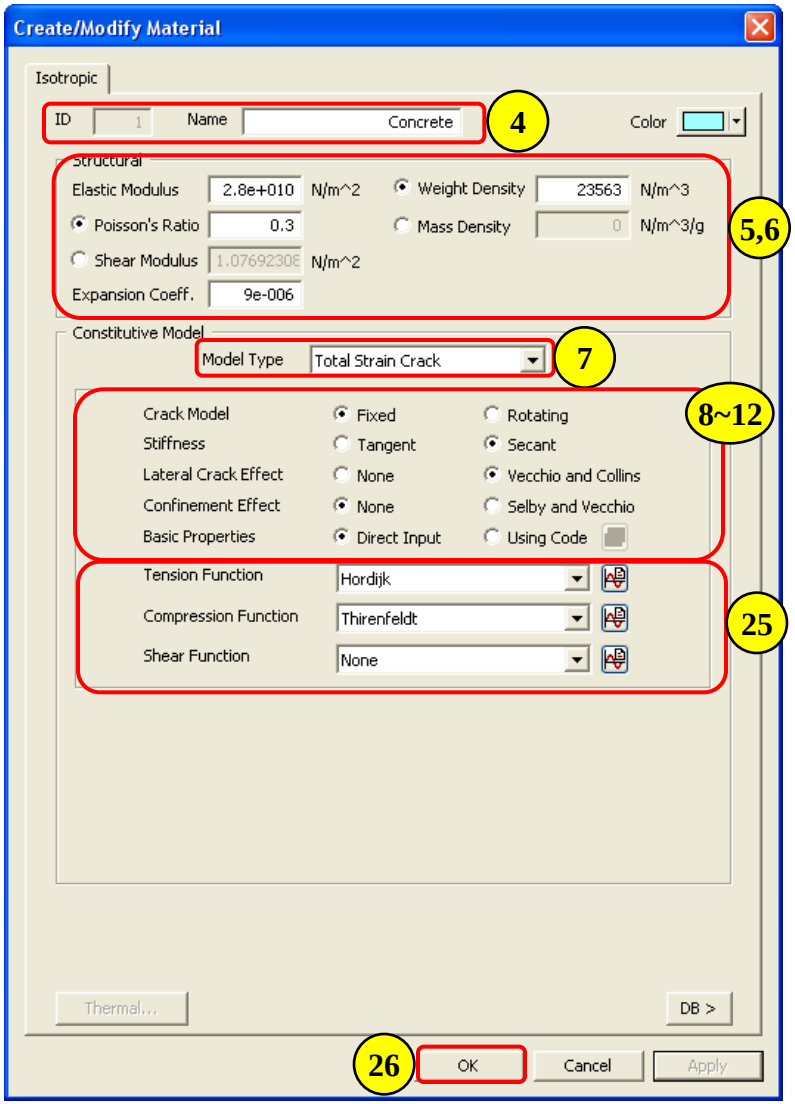


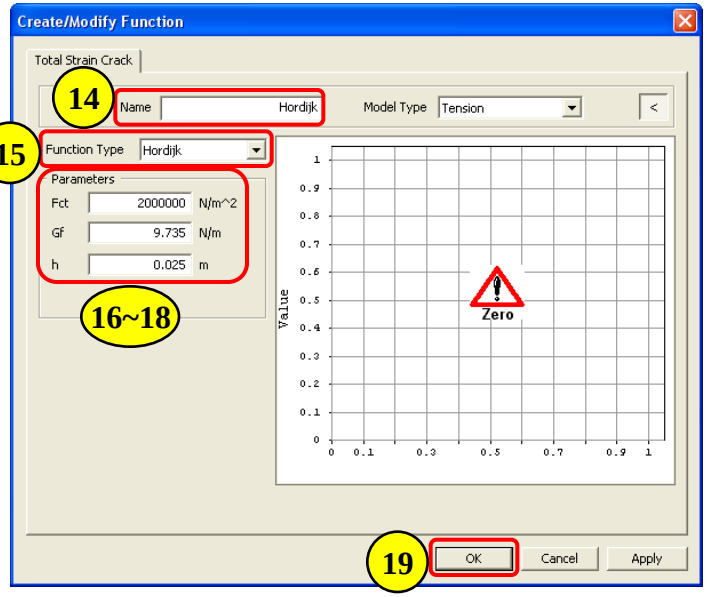


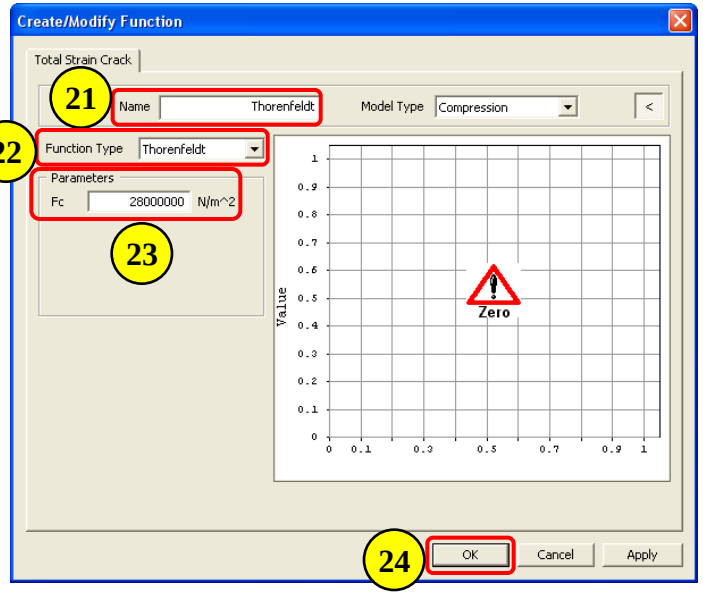
Step 13.

1. Analysis > Material …
2. Click [Create] Button
3. Select “Isotropic” tab
4. ID : 1 , Name : Concrete
5. Elastic Modulus : 2.8e+010 N/m^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 = 2000000 N/m^2
17. Gf = 9.735 N/m
18. h = 0.025 m
19. Click on [OK] Button
20. Click on to define Compression Function
21. Name: Thorenfeldt
22. Function Type: Thorenfeldt
23. Fc = 28000000 N/m^2
24. Click on [OK] Button
25. Select the tension, compression, shear functions from the drop lists
26. Click on [OK] Button

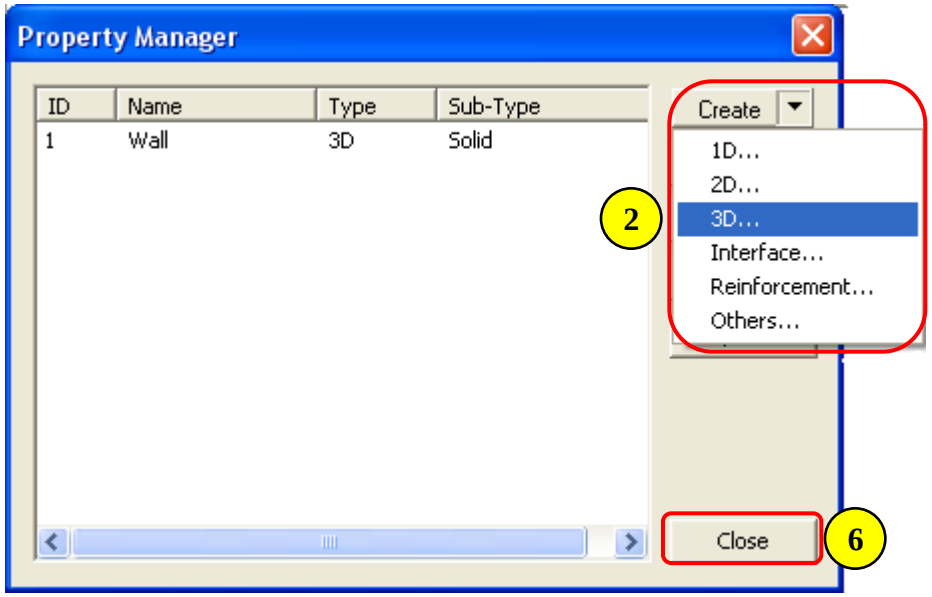
Step 14.


Create/Modify Material
Isotropic
ID 1
Name Concrete
4
Color
Structural
Elastic Modulus 2.8e+010 N/m^2
Weight Density 23563 N/m^3
Poisson's Ratio 0.3
Mass Density 0 N/m^3/g
Shear Modulus 1.07692308 N/m^2
Expansion Coeff. 9e-006
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 Thirenfeldt
Shear Function None
25
Thermal...
DB >
26
OK
Cancel
Apply

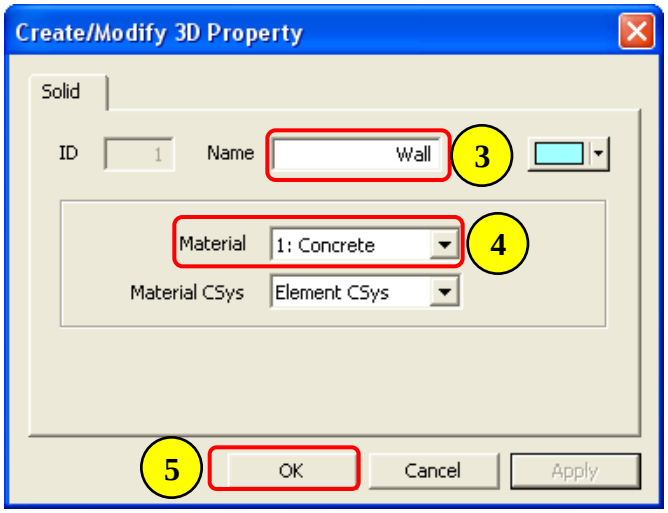

Create/Modify Function
Total Strain Crack
14
Name Hordijk
Model Type Tension
15
Function Type Hordijk
Parameters
Fct 2000000 N/m^2
Gf 9.735 N/m
h 0.025 m
16~18
Value
Zero
19
OK
Cancel
Apply


Create/Modify Function
Total Strain Crack
21
Name Thorenfeldt
Model Type Compression
22
Function Type Thorenfeldt
Parameters
Fc 28000000 N/m^2
23
Value
Zero
24
OK
Cancel
Apply

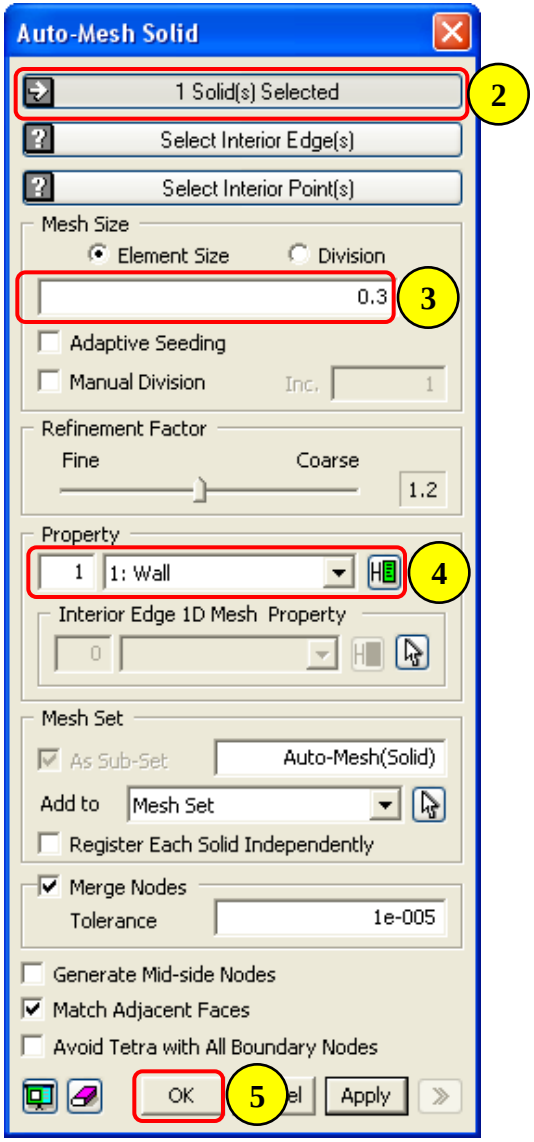
Step 15.



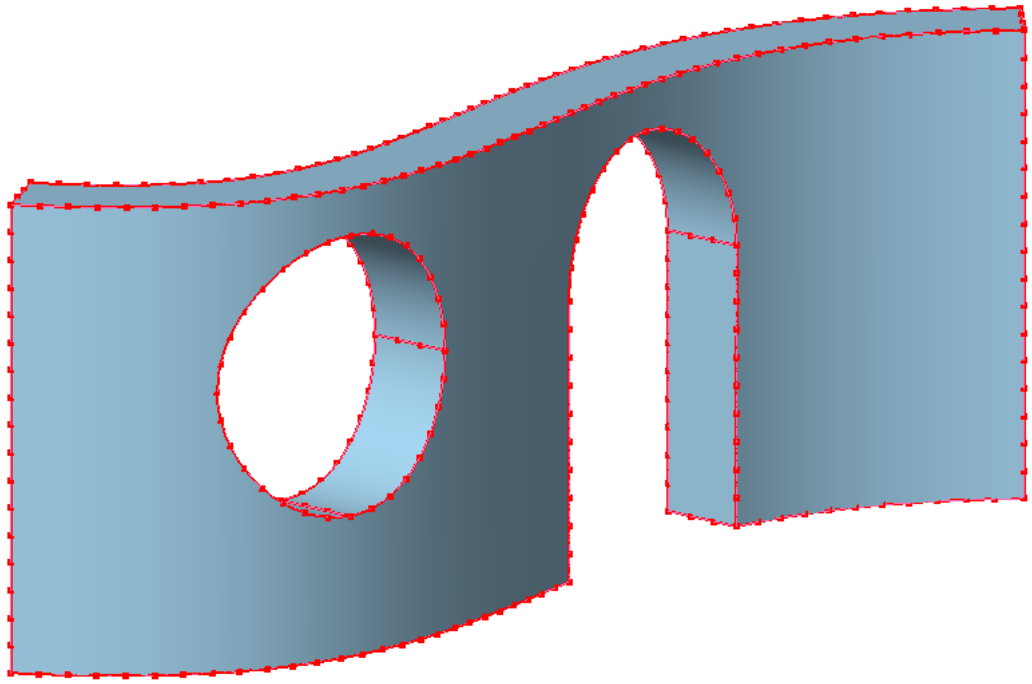
1. *Analysis > Property …*
2. *Create > 3D …*
3. *Name: Wall*
4. *Material: 1: Concrete*
5. *Click* **[OK]** *Button*
6. *Click* **[Close]** *Button*



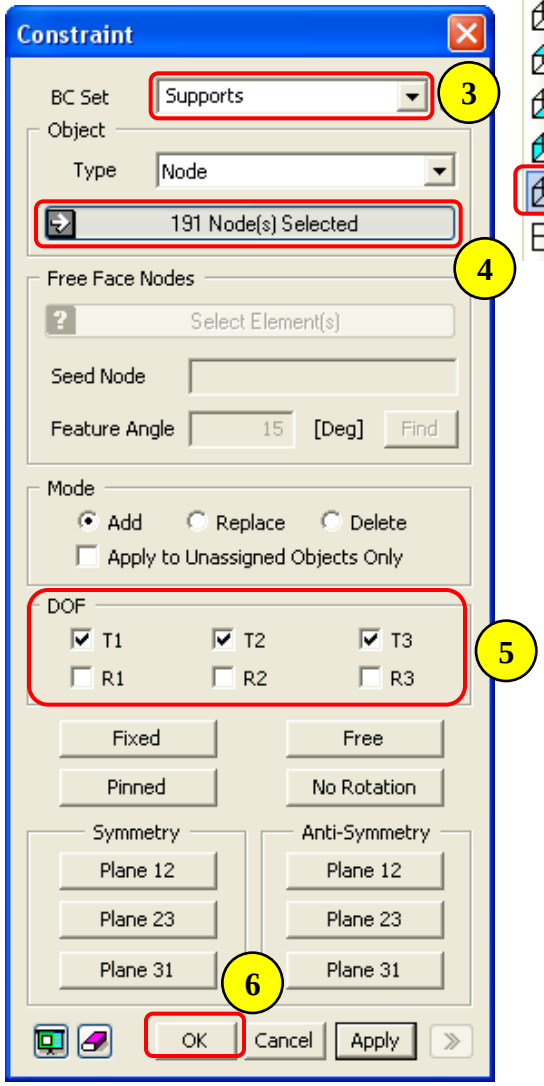
Step 16.



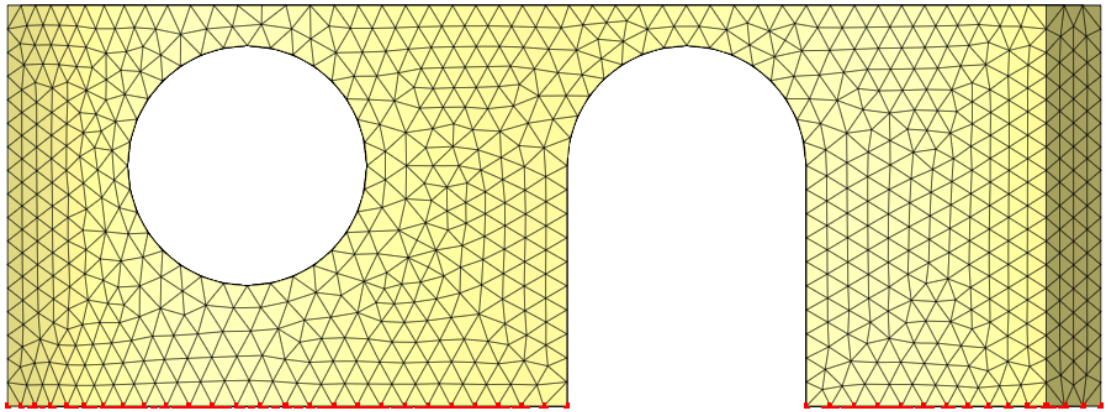
1. *Mesh > Auto Mesh > Solid …*
2. *Select the wall*
3. *Element Size: 0.3*
4. *Property: 1: Wall*
5. *Click* **[OK]** *Button*

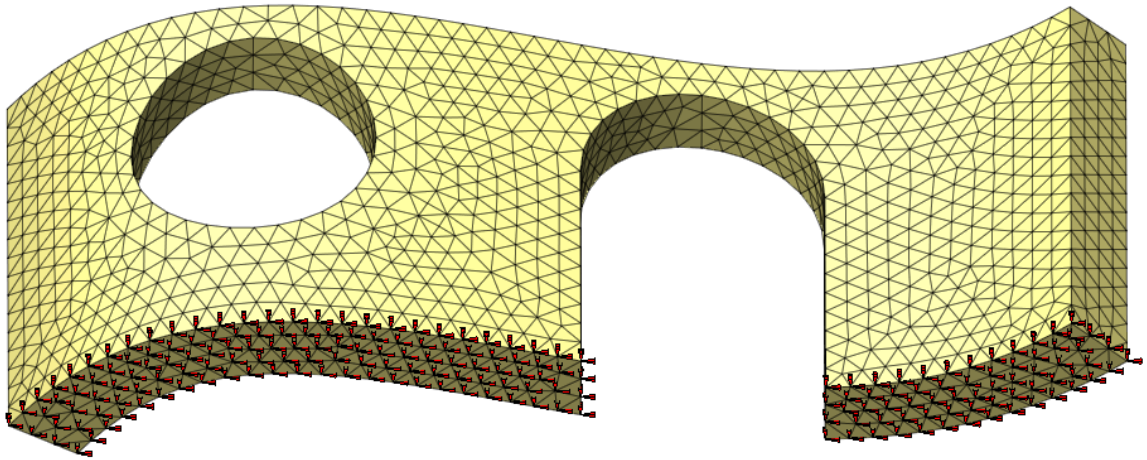


Step 17.

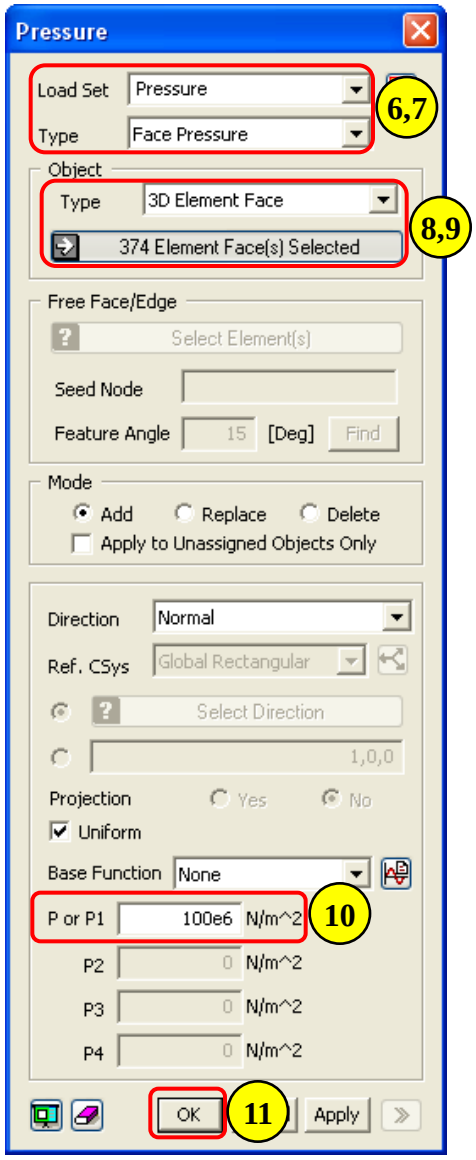


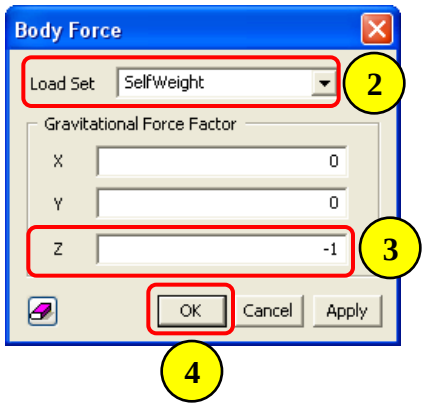
1. *Change view point to "Right"*
2. *Analysis > BC > Constraint …*
3. *BC Set: Supports*
4. *Select the highlighted nodes as shown in the picture*
5. *DOF: T1, T2, T3*
6. *Click* **[OK]** *Button*



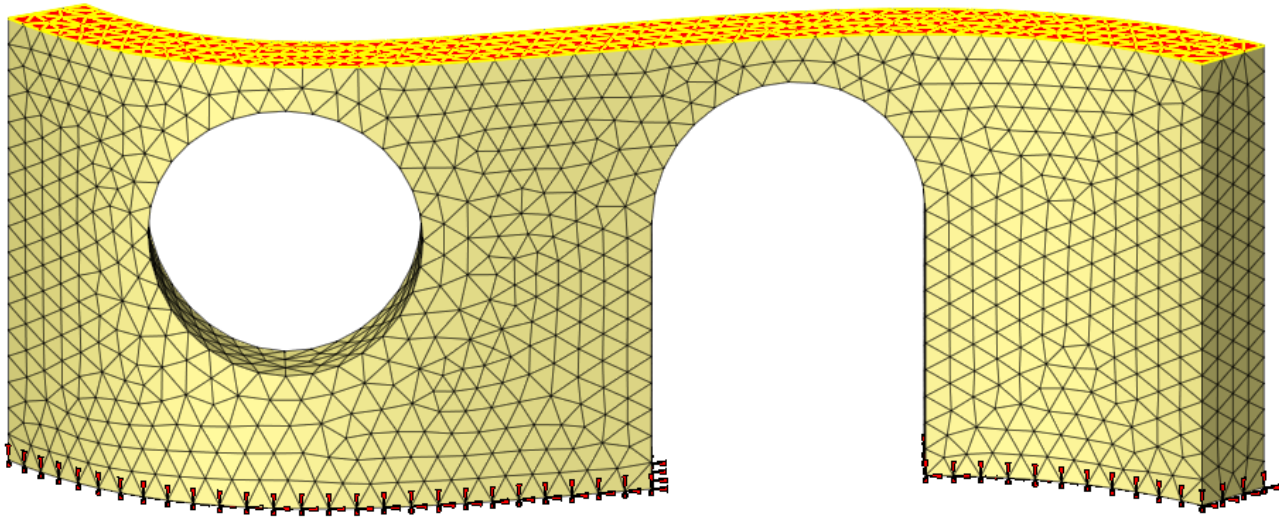


Step 18.

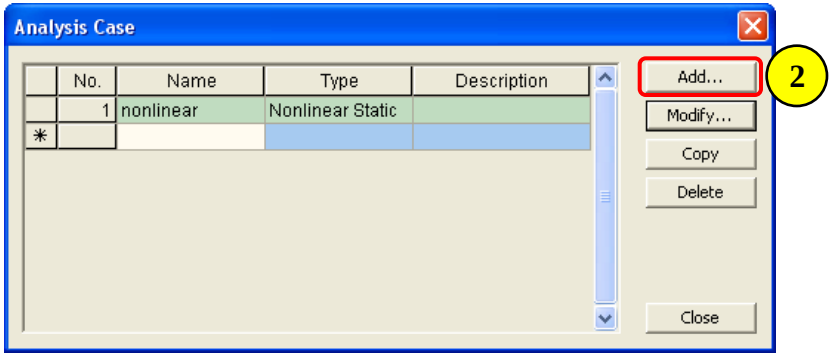


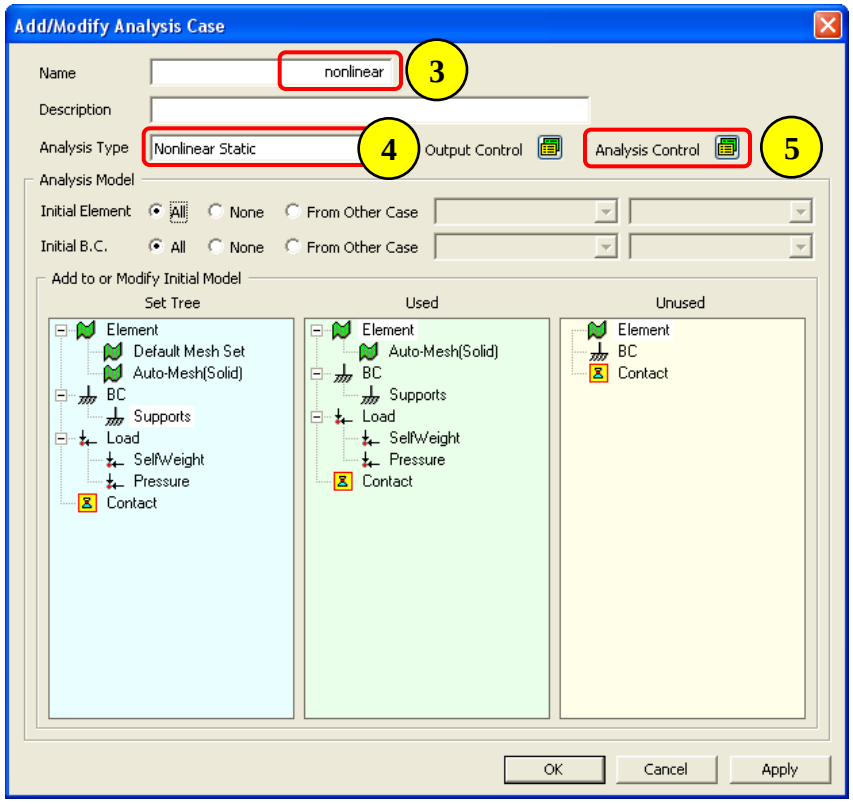


1. *Analysis > Load > Body Force …*
2. *Load Set: Self Weight*
3. *Gravitational Force Factor: Z=-1*
4. *Click* **[OK]** *Button*
5. *Analysis > Load > Pressure …*
6. *Load Set: Pressure*
7. *Type: Face Pressure*
8. *Object Type: 3D Element Face*
9. *Select the highlighted faces*
10. *P1: 100e6 N/m²*
11. *Click* **[OK]** *Button*



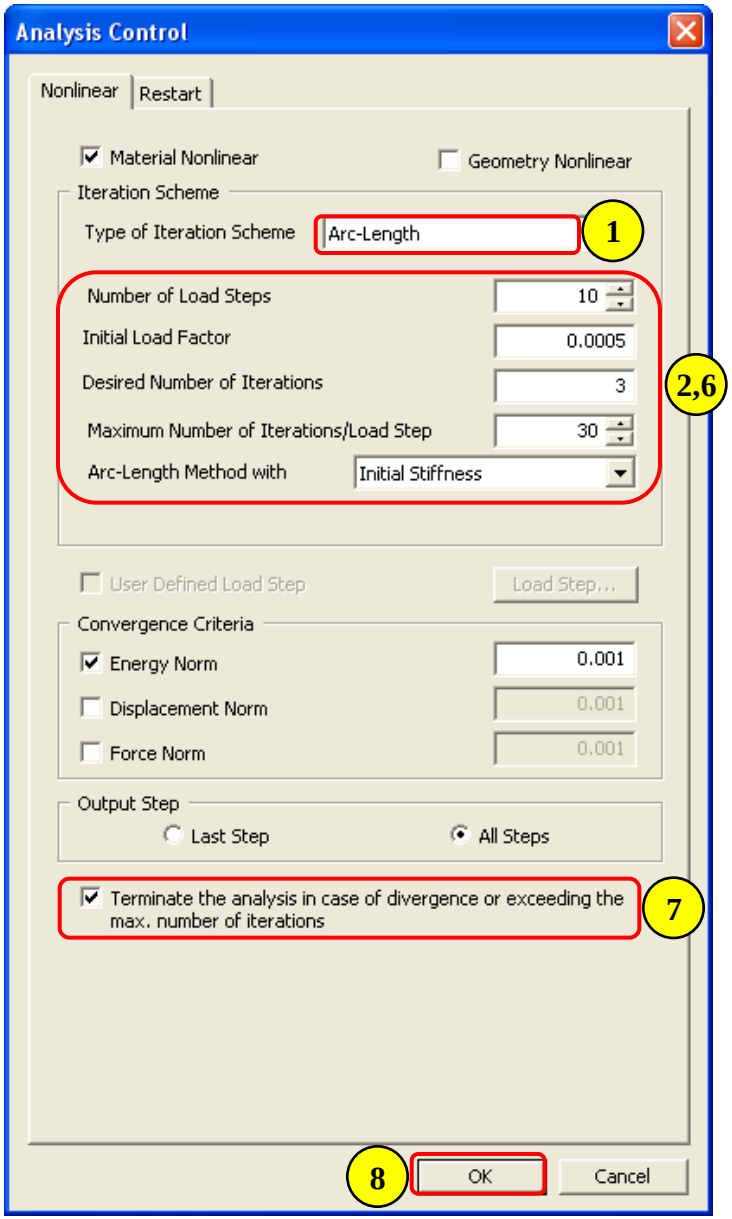
Step 19.





1. *Analysis > Analysis Case …*
2. *Click on Add*
3. *Name: nonlinear*
4. *Analysis Type: Nonlinear Static*
5. *Click on Analysis Control*

Step 20.



1. *Type of Iteration Scheme: Arc-Length*
2. *Number of Load Steps: 10*
3. *Initial Load Factor: 0.0005*
4. *Desired Number of Iterations: 3*
5. *Maximum Number of Iterations: 30*
6. *Arc-Length Method with: Initial Stiffness*
7. *Check on "Terminate the analysis in case of divergence or exceeding the max. number of iterations"*
8. *Click* **[OK]** *Button*