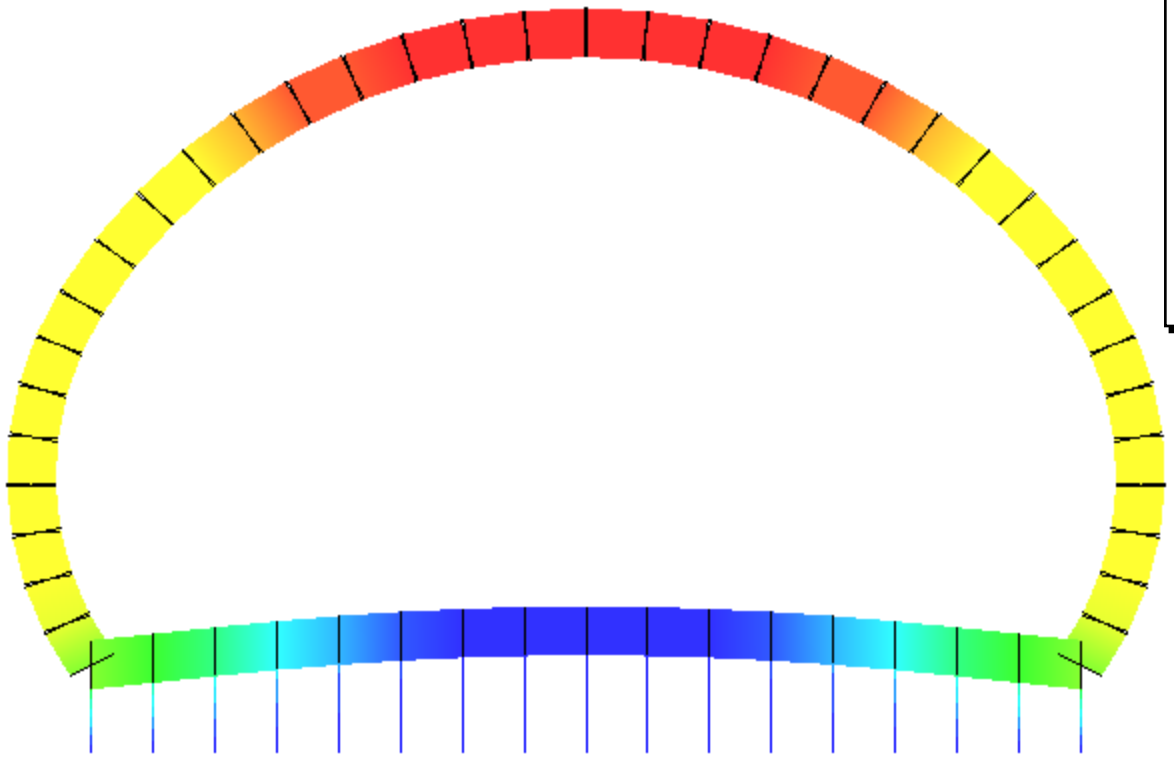
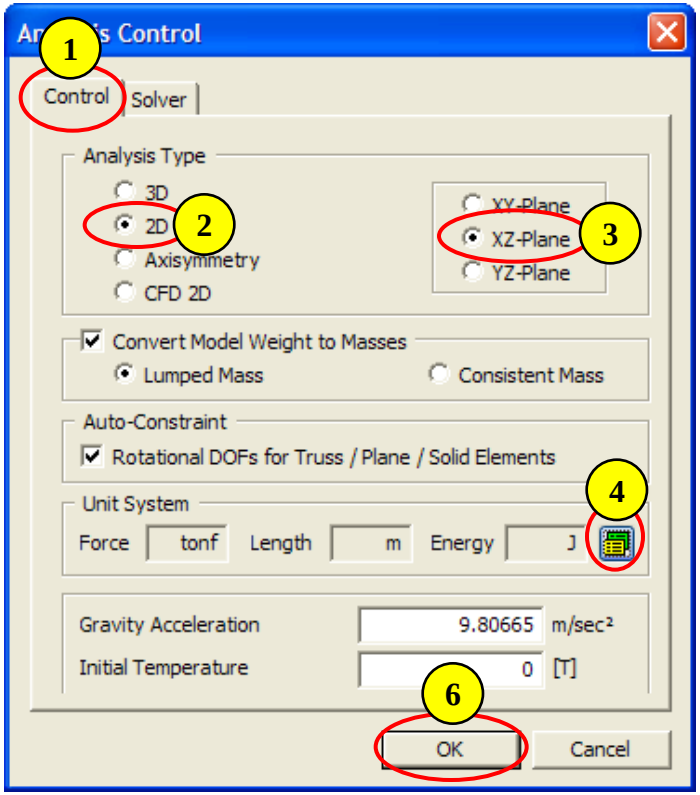
LS-14. 2D Tunnel Lining



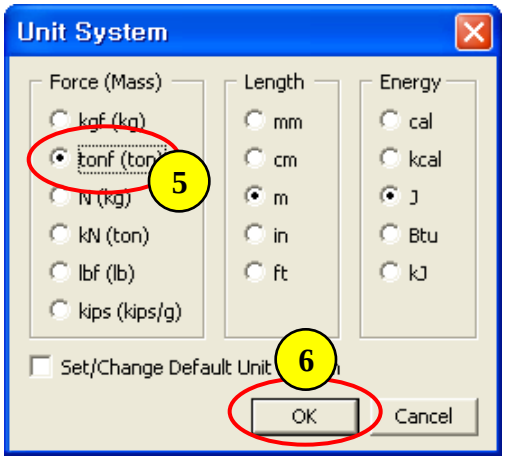
Overview

- 2-D Linear Static Analysis
- Model
  - *Unit : tonf , m*
  - *Isotropic Elastic Material*
  - *Beam Element*
  - *Surface Spring Element*
- Load & Boundary Condition
  - *Body Force*
  - *Line Beam Load*
  - *Combined Set*
  - *Constraint*

Step 1.

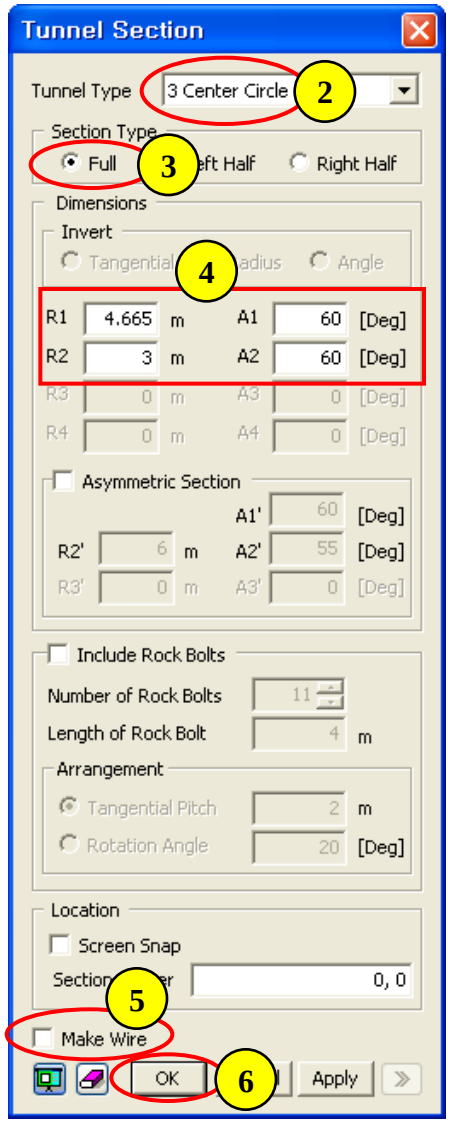


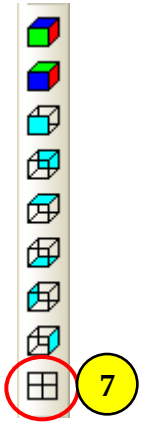
1. *Analysis > Analysis Control - "Control" tab*
2. *Select "2D"*
3. *Select "XZ-Plane"*
4. *Click*  *Button*
5. *Force : tonf*
6. *Click* **[OK]** *Button*



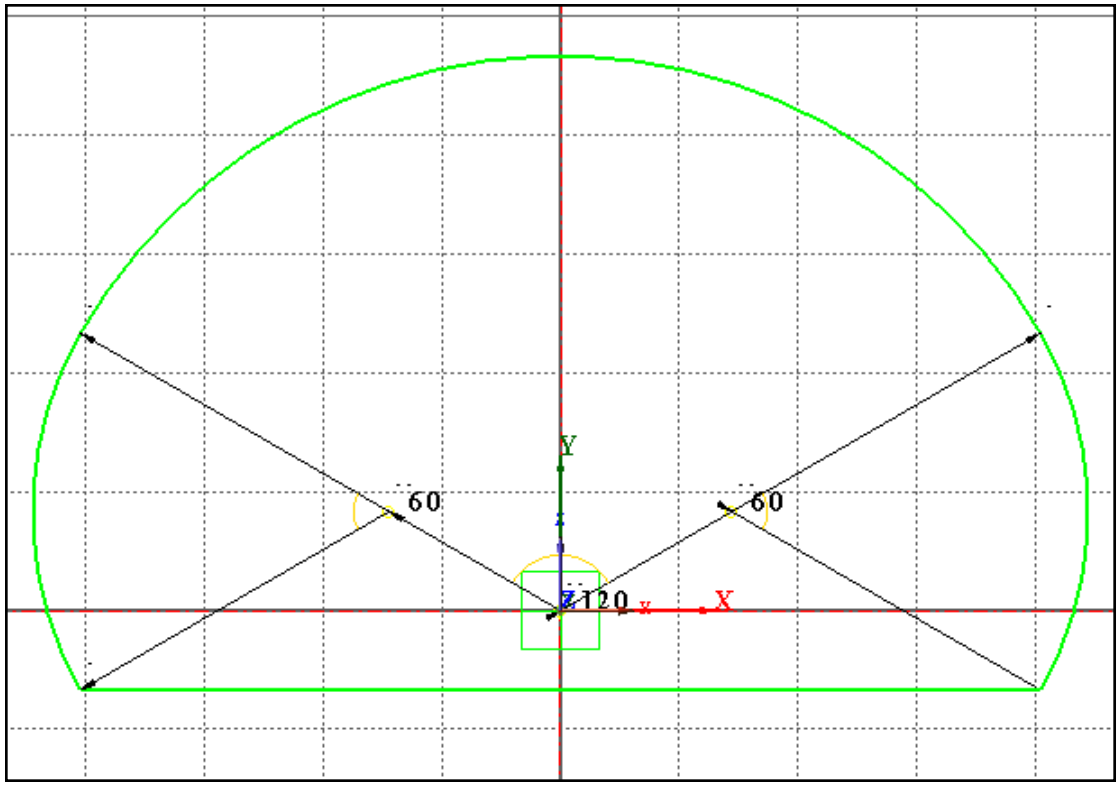
Analysis Control Dialog is automatically activated at startup.

Step 2.



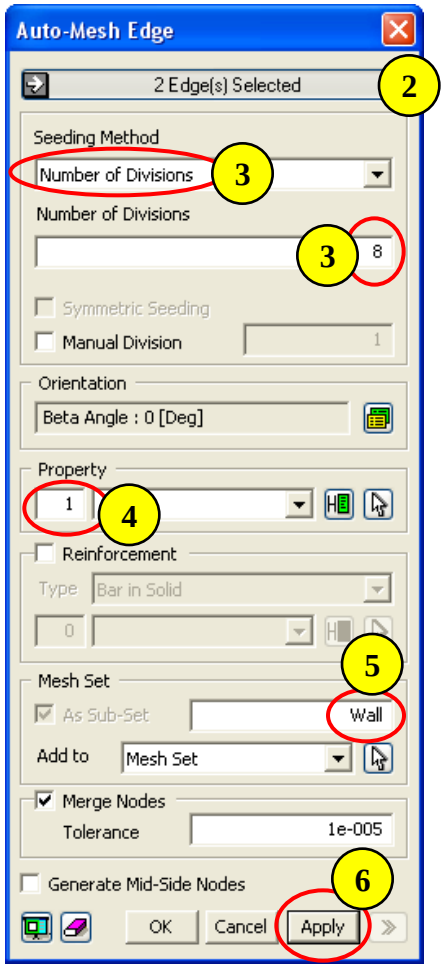


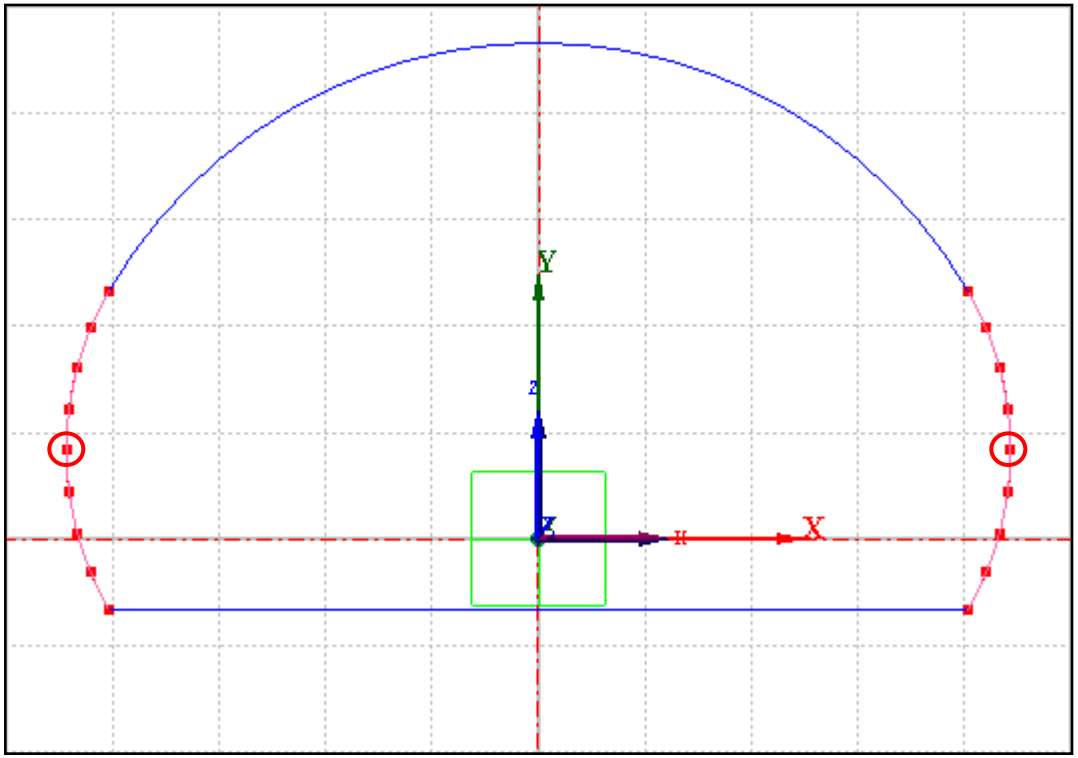
1. *Geometry > Curve > Create on WP > Tunnel (Wire)…*
2. *Select "3 Center Circle" for Tunnel Type*
3. *Section Type : Full*
4. *R1 : 4.665 , A1 : 60 , R2 : 3 , A2 : 60*
5. *Check off "Make Wire"*
6. *Click* **[OK]** *Button*
7. *Click "Normal View"*



Step 3.

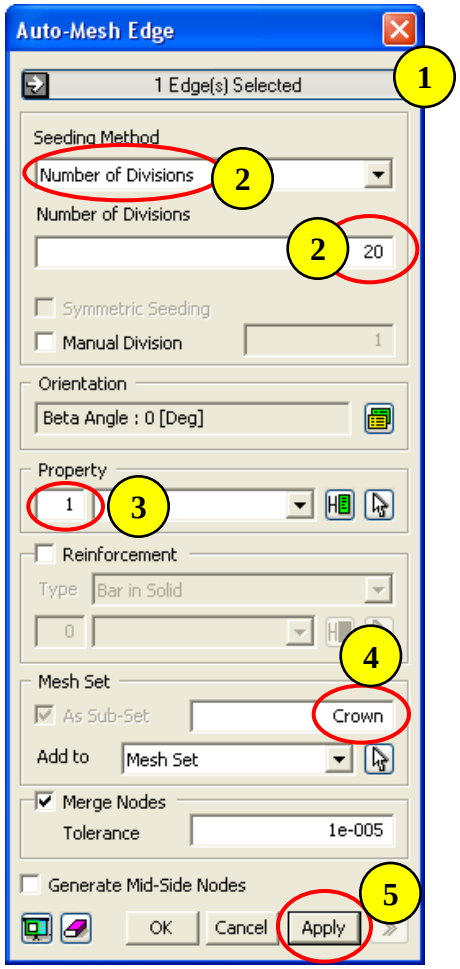
1. *Mesh > Auto Mesh > Edge …*
2. *Select 2 Edges marked by "O" (See Figure)*
3. *Seeding Method : Number of Divisions (8)*
4. *Property : 1*
5. *Mesh Set : Wall*
6. *Click* [Apply] *Button*



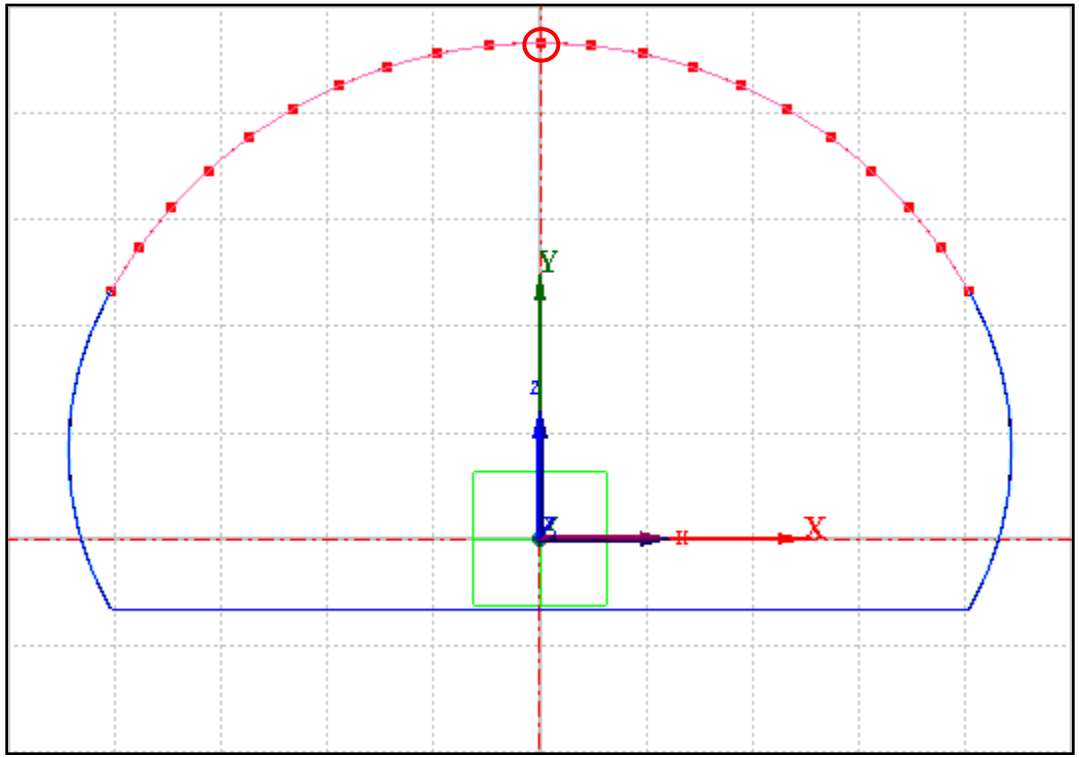


[Enter] *as shortcut for* [Apply] .

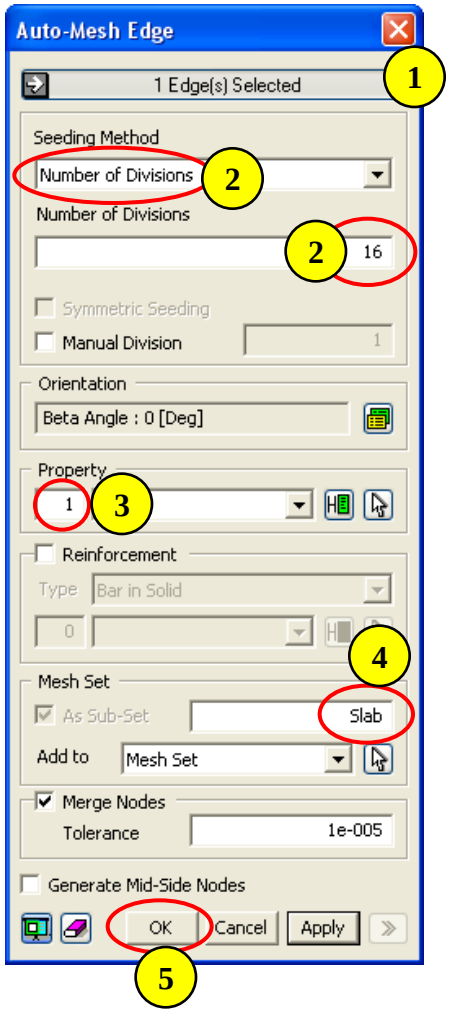
Step 4.



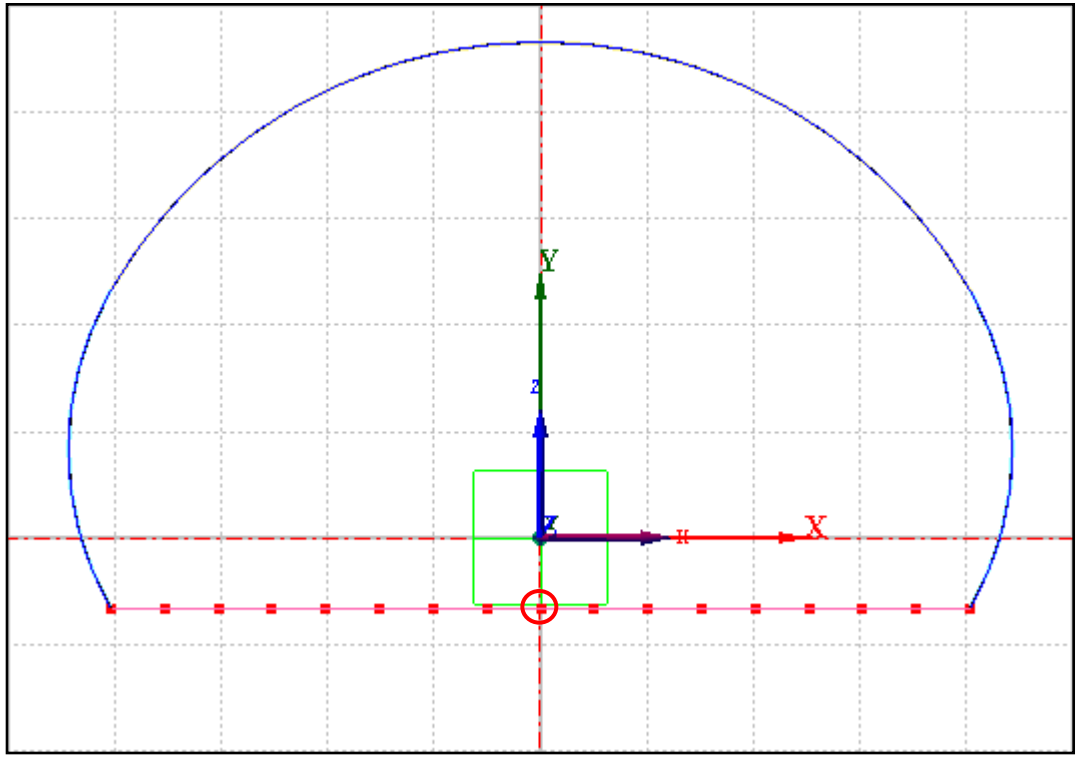
1. *Select Edge marked by "O" (See Figure)*
2. *Seeding Method : Number of Divisions (20)*
3. *Property : 1*
4. *Mesh Set : Crown*
5. *Click* **[Apply]** *Button*



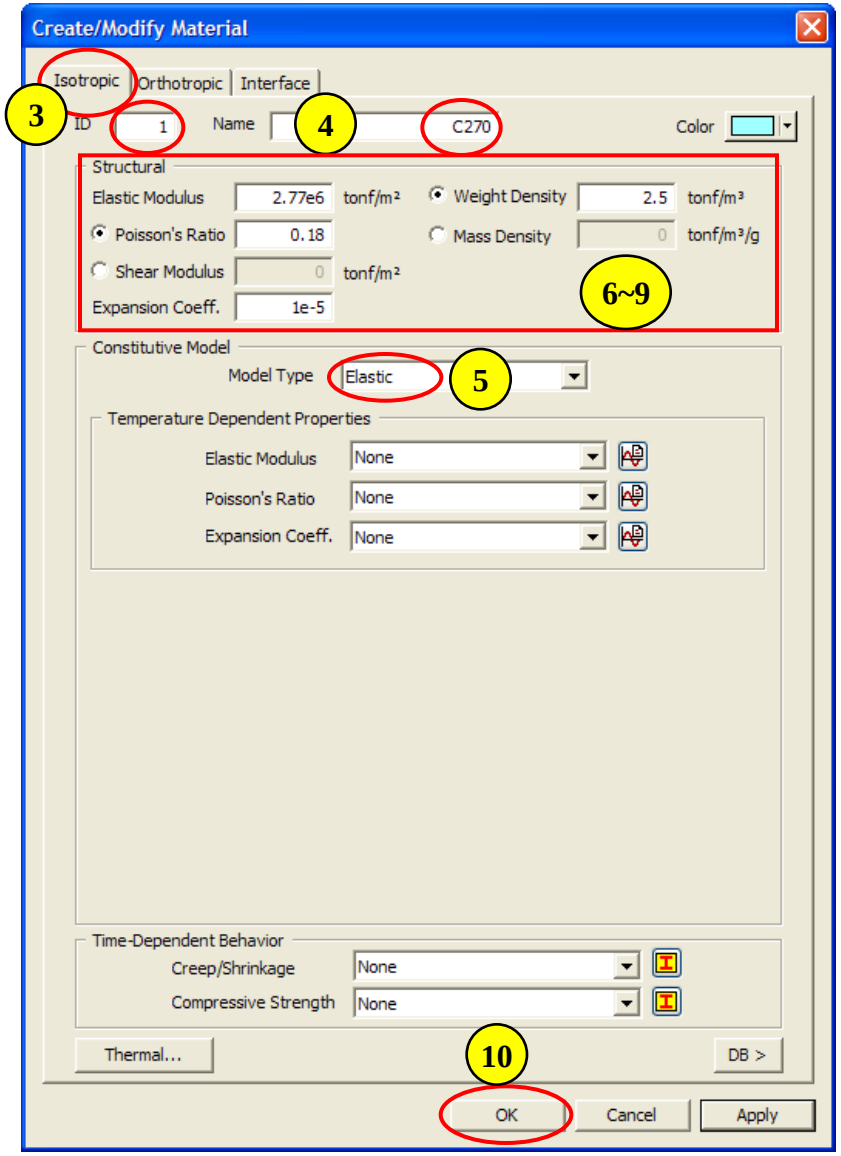
Step 5.



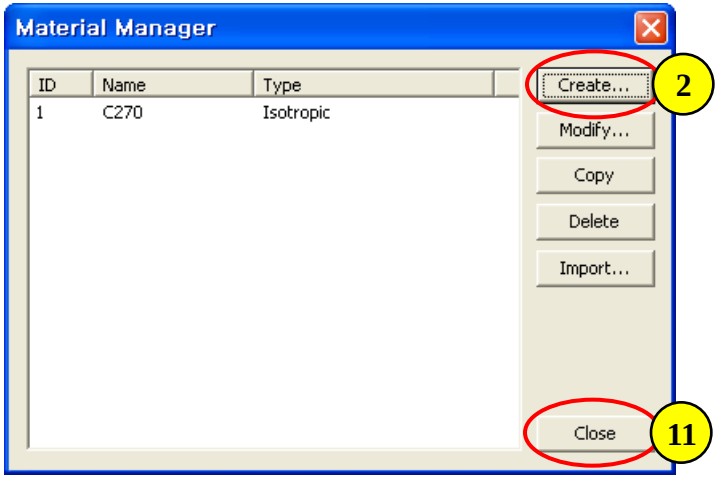
1. *Select Edge marked by "O" (See Figure)*
2. *Seeding Method : Number of Divisions (16)*
3. *Property : 1*
4. *Mesh Set : Slab*
5. *Click* **[OK]** *Button*



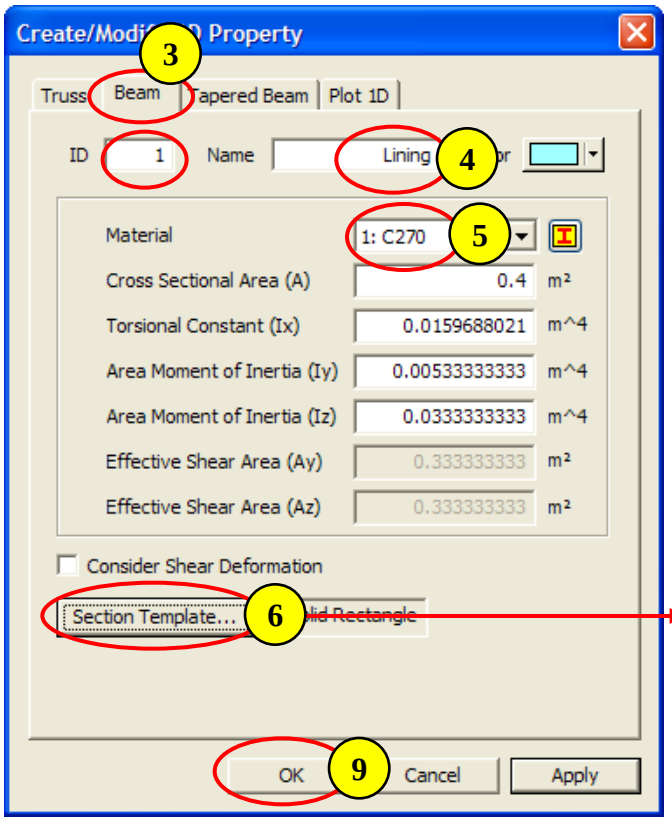
Step 6.

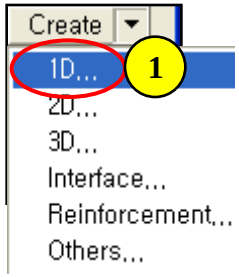


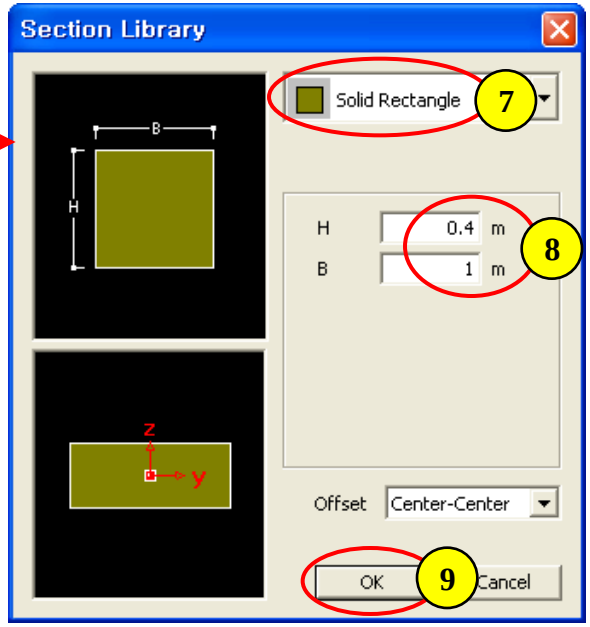
1. *Analysis > Material …*
2. *Click* [Create] *Button*
3. *Select "Isotropic" tab*
4. *ID (1) , Name : C270*
5. *Model Type : Elastic*
6. *Elastic Modulus : 2.77e6 tonf/m²*
7. *Poisson's Ratio : 0.18*
8. *Expansion Coeff. : 1e-5*
9. *Weight Density : 2.5 tonf/m³*
10. *Click* [OK] *Button*
11. *Click* [Close] *Button*



Step 7.

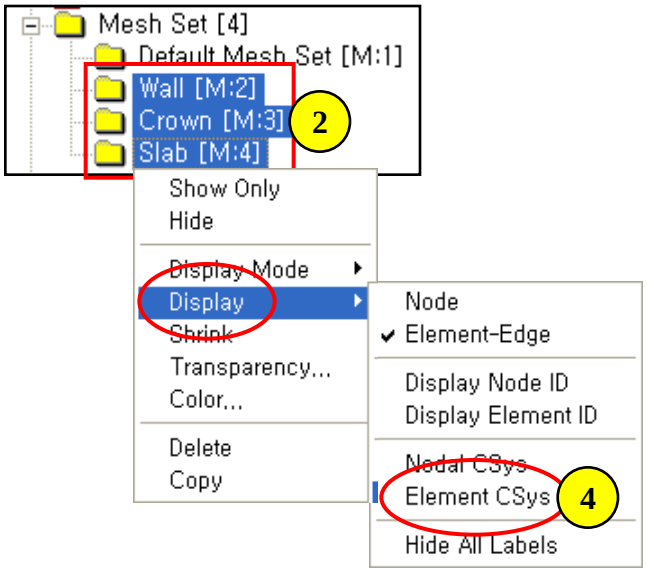




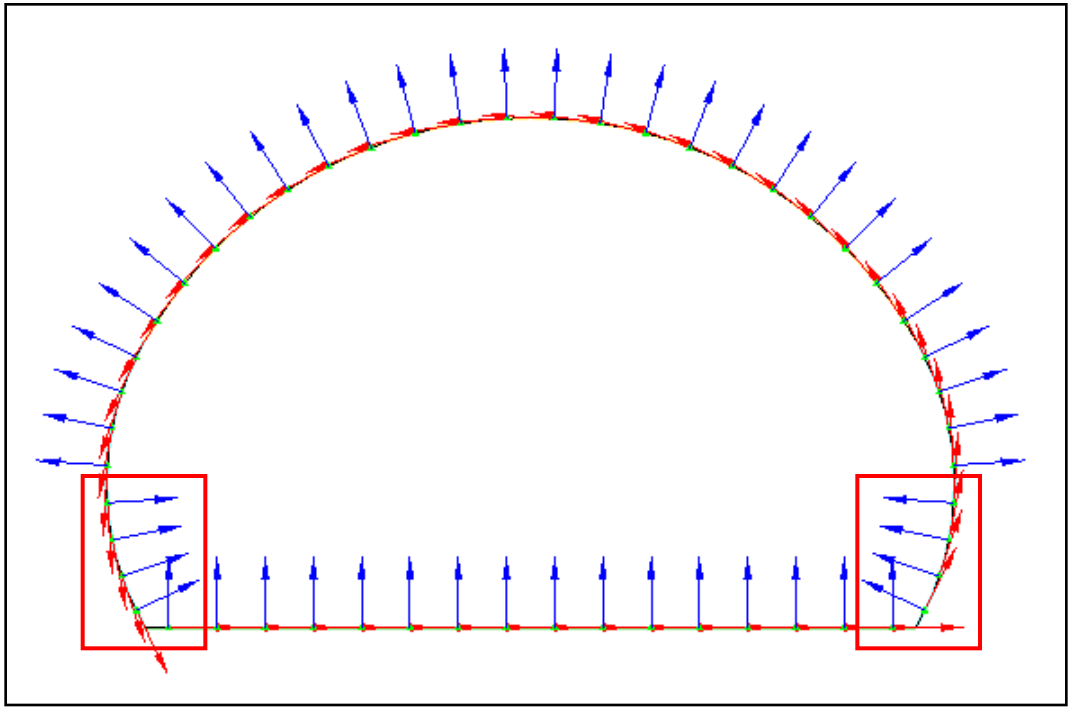


1. *Analysis > Property ...*
2. *Create 1D ...*
3. *Select "Beam" Tab*
4. *ID : 1 , Name : Lining*
5. *Select "1: C270" for Material*
6. *Click* **[Section Template...]** *Button*
7. *Select "Solid Rectangle"*
8. *H : 0.4 m , B : 1 m*
9. *Click* **[OK]** *Button*
10. *Click* **[Close]** *Button*

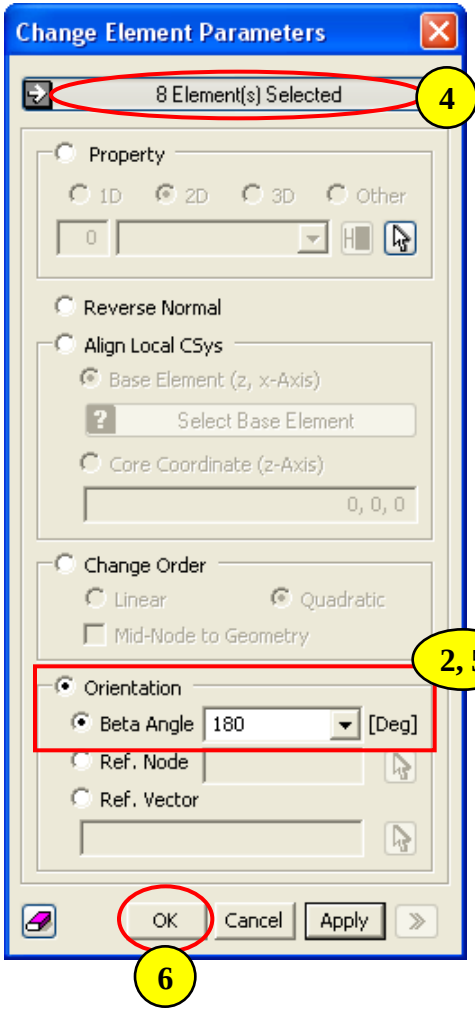
Step 8.

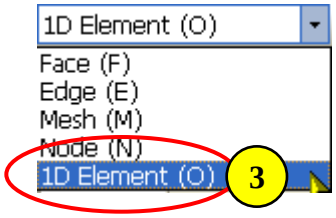


1. *Works Tree > Mesh > Mesh Set ...*
2. *Select "Wall" , "Crown" , "Slab"*
3. *Click Right Mouse Button*
4. *Select "Display > Element CSys"*

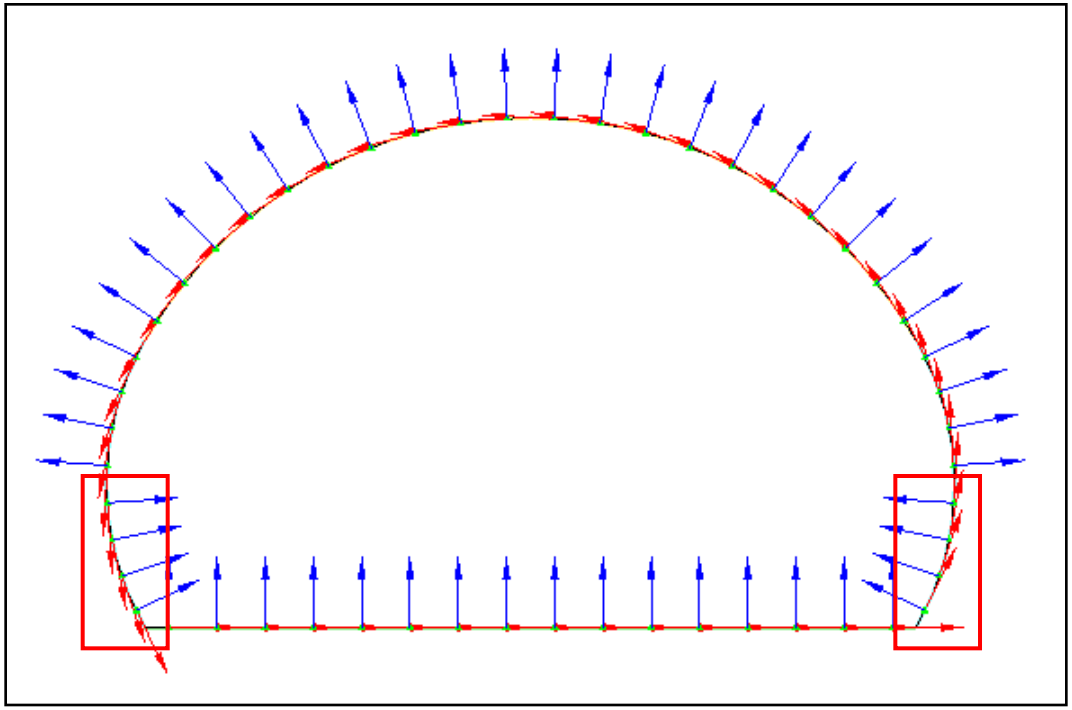


Step 9.

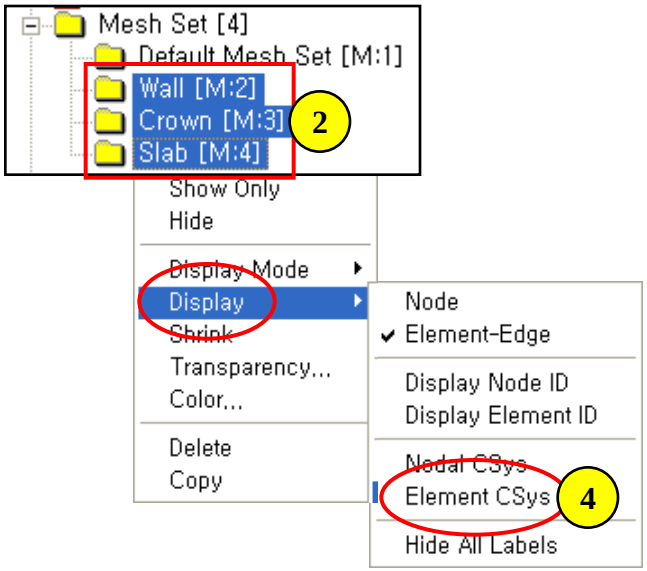




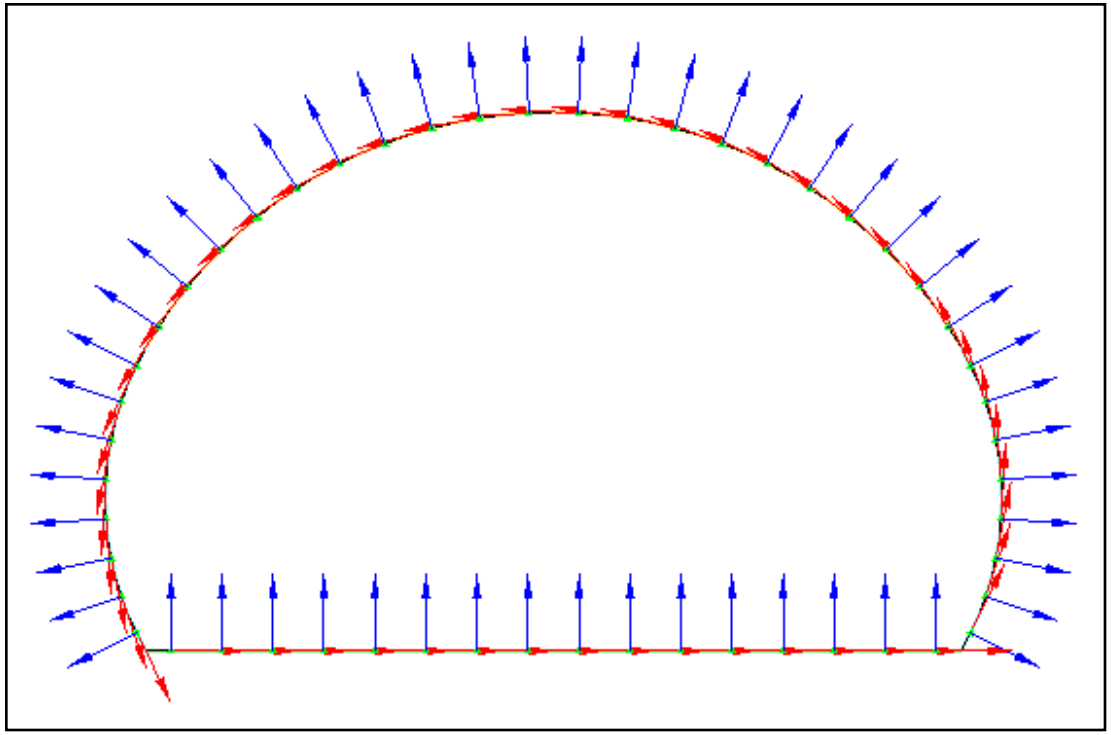
1. *Mesh > Element > Change Parameter ...*
2. *Check on "Orientation"*
3. *Change Selection Filter to "1D Element (O)"*
4. *Select 8 Elements marked in box (See Figure)*
5. *Select "180" for Beta Angle*
6. *Click* **[OK]** *Button*



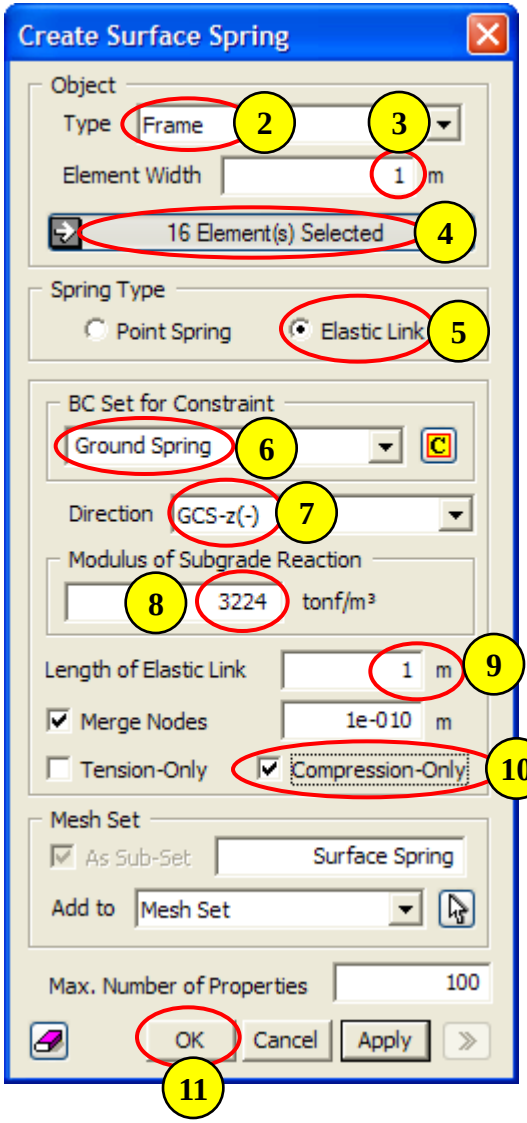
Step 10.



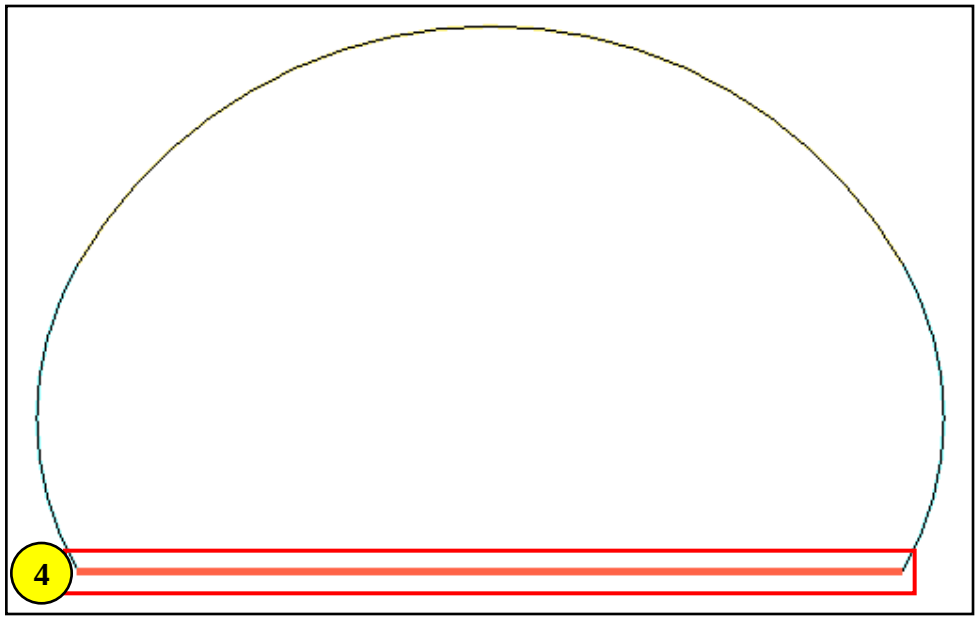
1. *Works Tree > Mesh > Mesh Set ...*
2. *Select "Wall" , "Crown" , "Slab"*
3. *Click Right Mouse Button*
4. *Select "Display > Element Csys" to hide Element CSys*



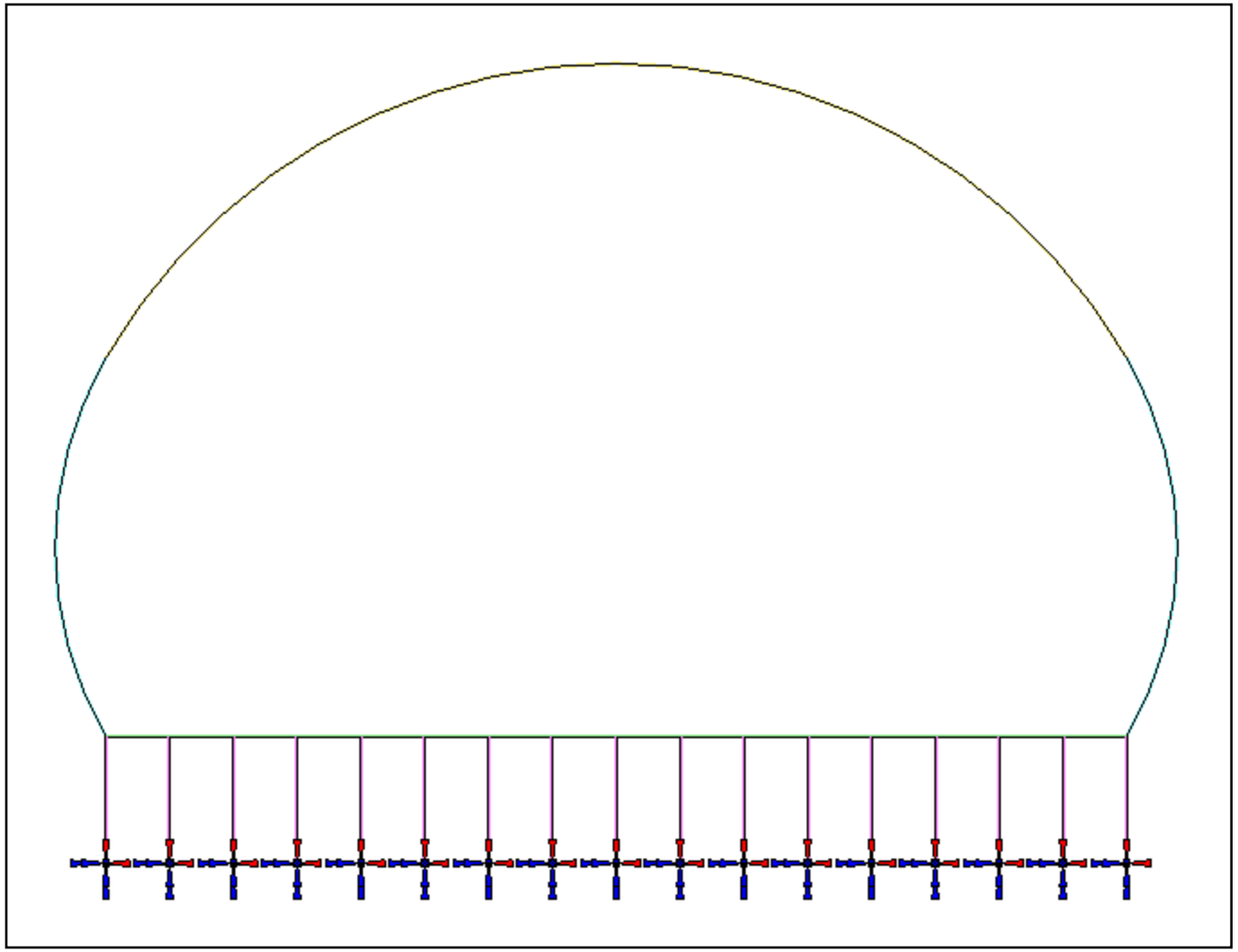
Step 11-1.



1. *Mesh > Element > Create Surface Spring ...*
2. *Object Type : Frame*
3. *Element Width : 1 m*
4. *Select 16 Elements (See Figure)*
5. *Select "Elastic Link" for Spring Type*
6. *BC Set for Constraint : Ground Spring*
7. *Direction : GCS-z(-)*
8. *Modulus of Subgrade Reaction : 3224 tonf/m³*
9. *Length of Elastic Link : 1 m*
10. *Check on "Compression–only"*
11. *Click* **[OK]** *Button*

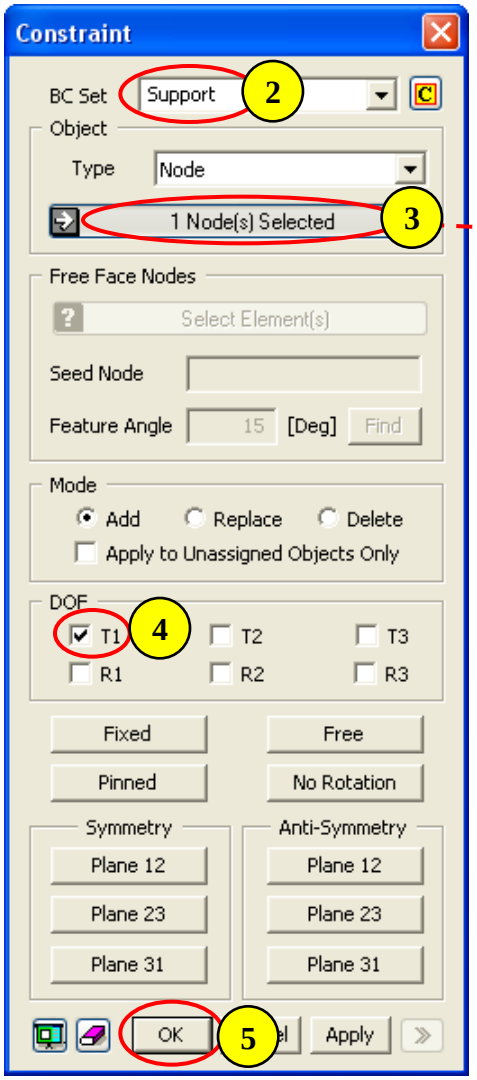


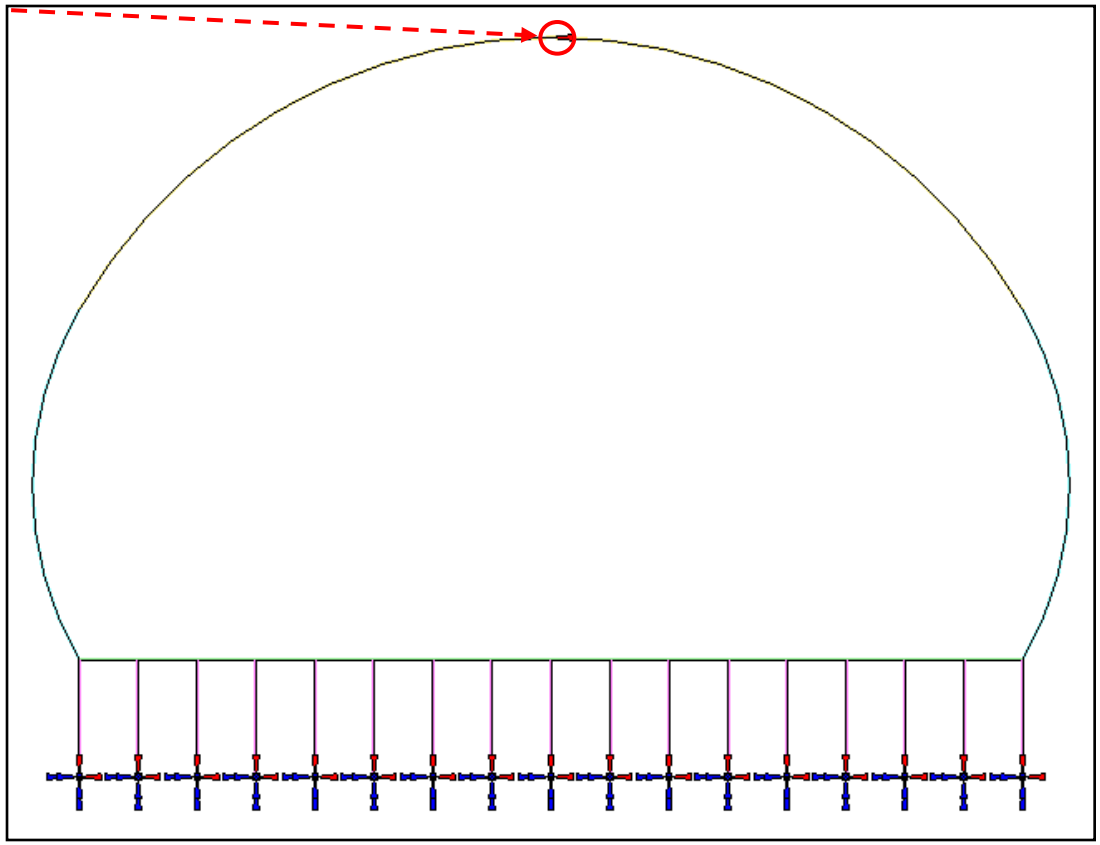
Step 11-2.



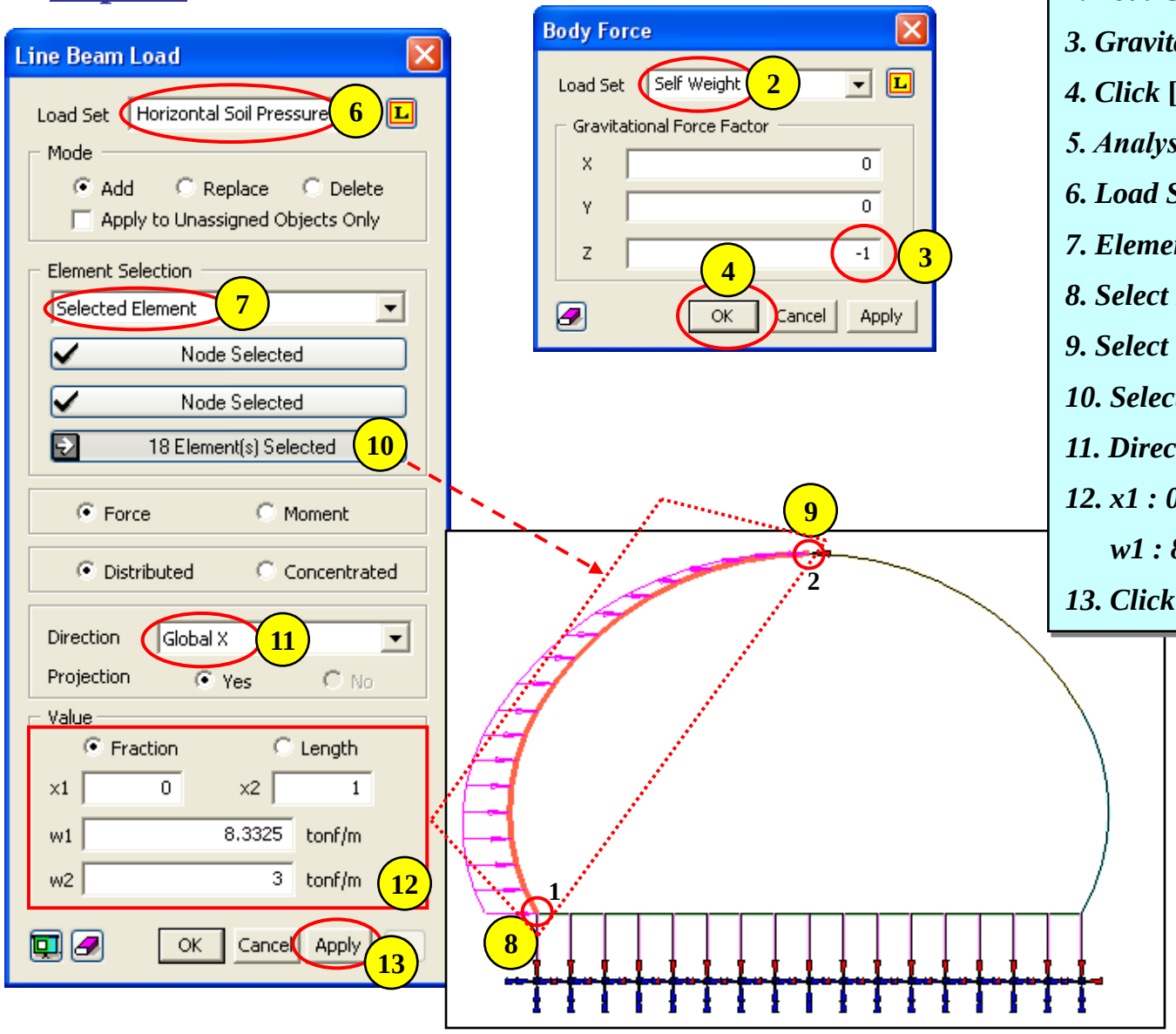
Step 12.

1. *Analysis > BC > Constraint ...*
2. *BC Set : Support*
3. *Select Node marked by "O" (See Figure)*
4. *Check on "T1"*
5. *Click* **[OK]** *Button*



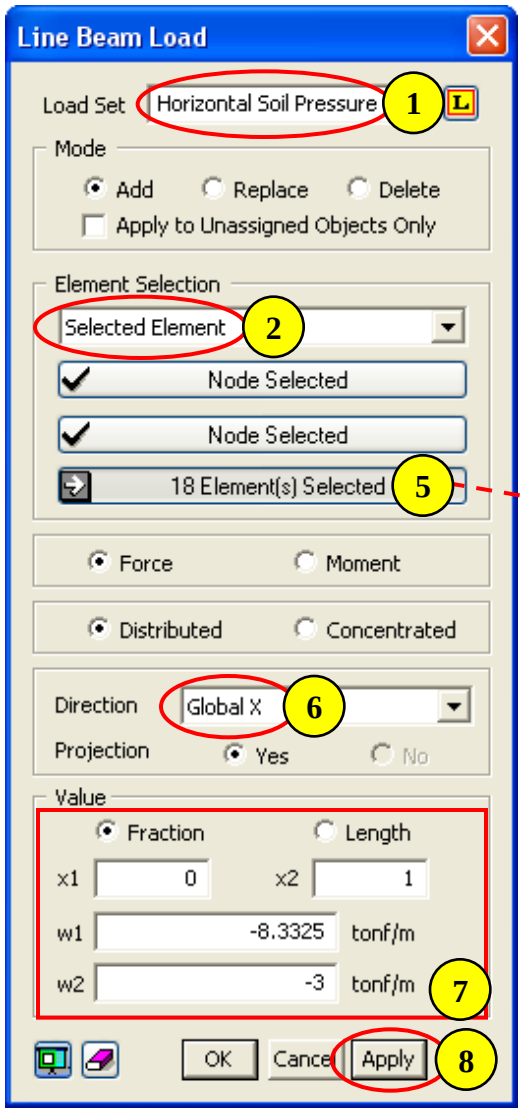


Step 13.

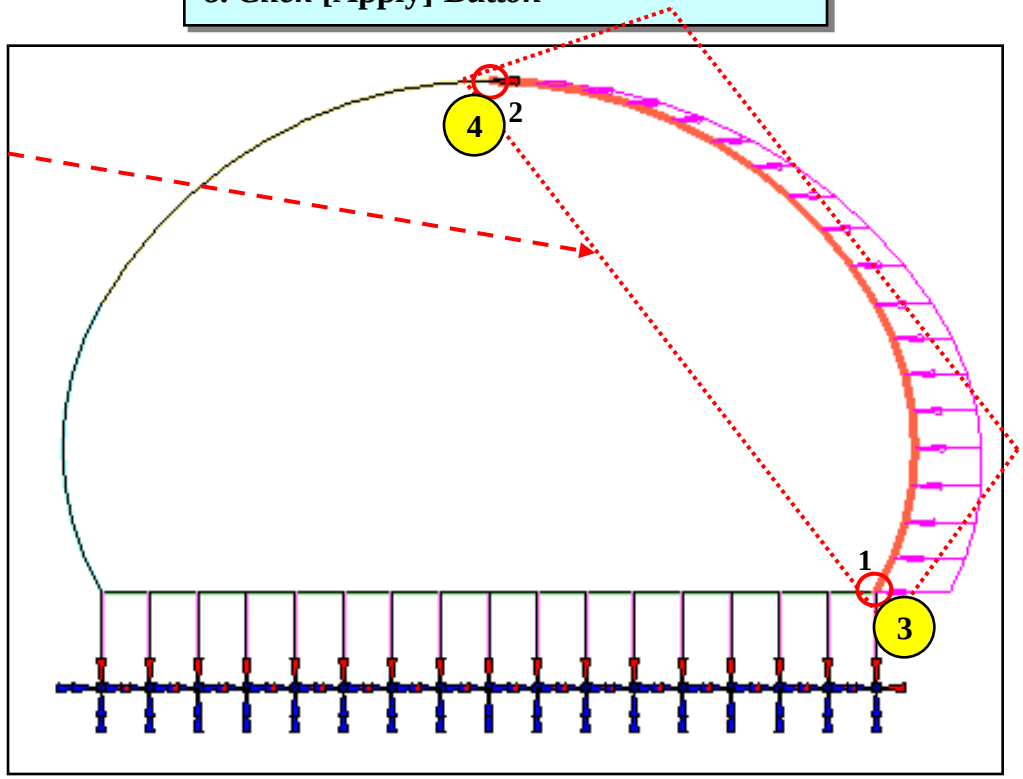


1. *Analysis > Load > Body Force ...*
2. *Load Set : Self Weight*
3. *Gravitational Force Factor : Z (-1)*
4. *Click* **[OK]** *Button*
5. *Analysis > Load > Line Beam Load …*
6. *Load Set : Horizontal Soil Pressure*
7. *Element Selection : Selected Element*
8. *Select First Node (See Figure)*
9. *Select Second Node (See Figure)*
10. *Select 18 Elements (See Figure)*
11. *Direction : Global X*
12. *x1 : 0 , x2 : 1*
    w1 : 8.3325 , w2 : 3
13. *Click* **[Apply]** *Button*

Step 14.

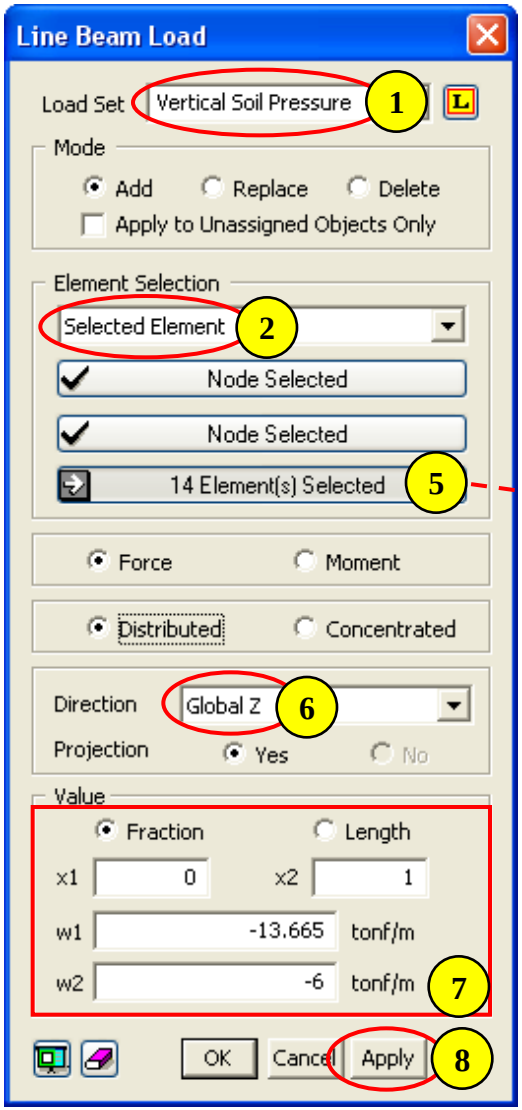


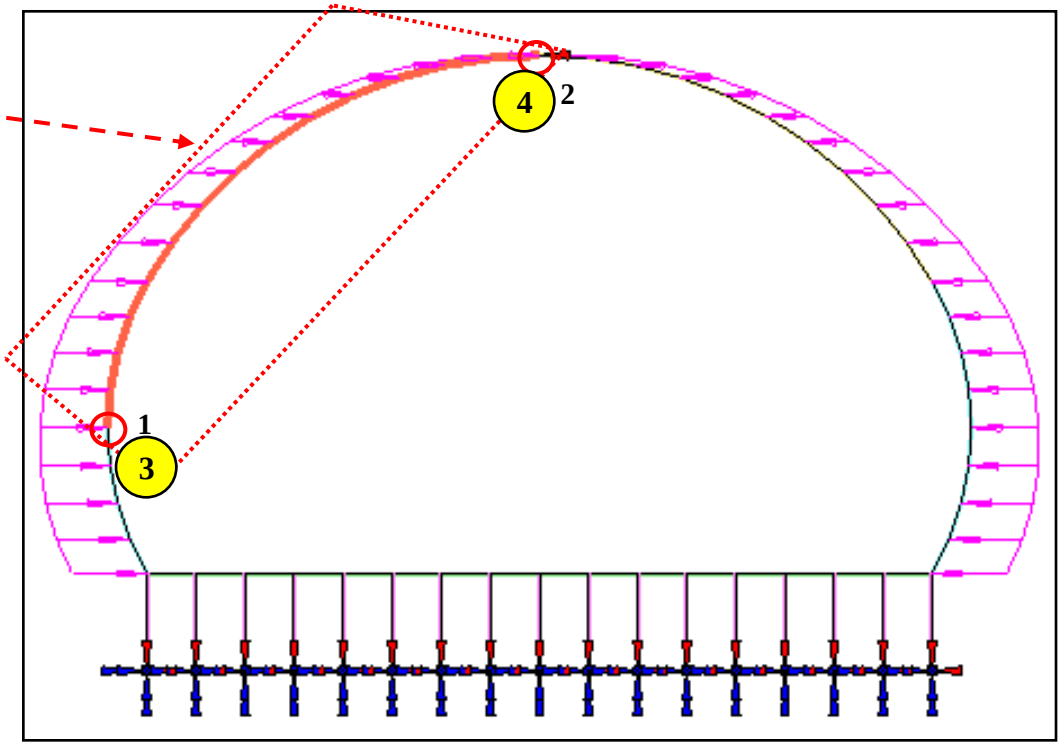
1. *Load Set : Horizontal Soil Pressure*
2. *Element Selection : Selected Element*
3. *Select First Node (See Figure)*
4. *Select Second Node (See Figure)*
5. *Select 18 Elements (See Figure)*
6. *Direction : Global X*
7. *x1 : 0 , x2 : 1*
   w1 : -8.3325 , w2 : -3
8. *Click* **[Apply]** *Button*



Step 15.

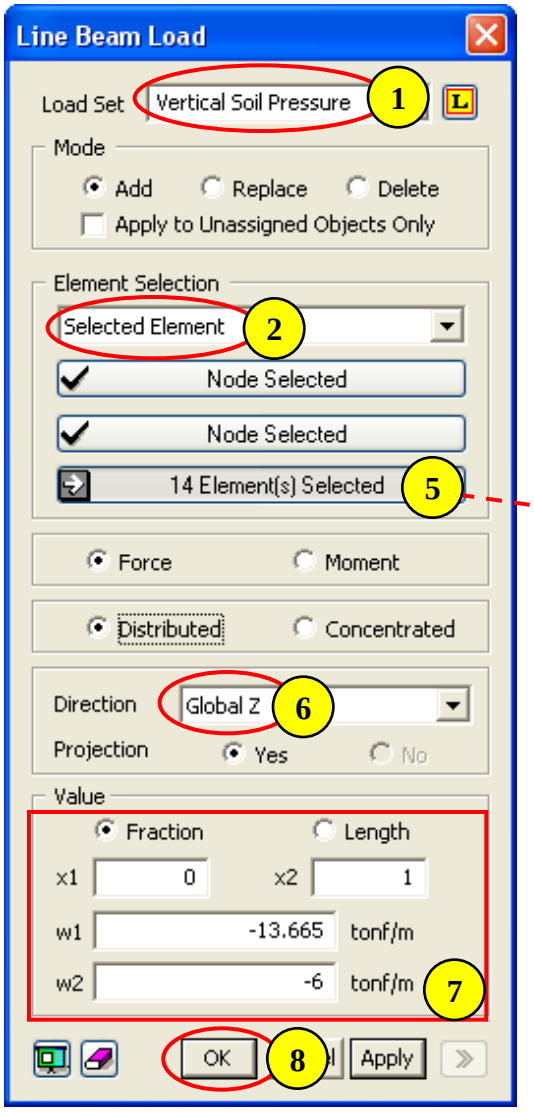
1. *Load Set : Vertical Soil Pressure*
2. *Element Selection : Selected Element*
3. *Select First Node (See Figure)*
4. *Select Second Node (See Figure)*
5. *Select 14 Elements (See Figure)*
6. *Direction : Global Z*
7. *x1 : 0 , x2 : 1*
   w1 : -13.665 , w2 : -6
8. *Click* [Apply] *Button*

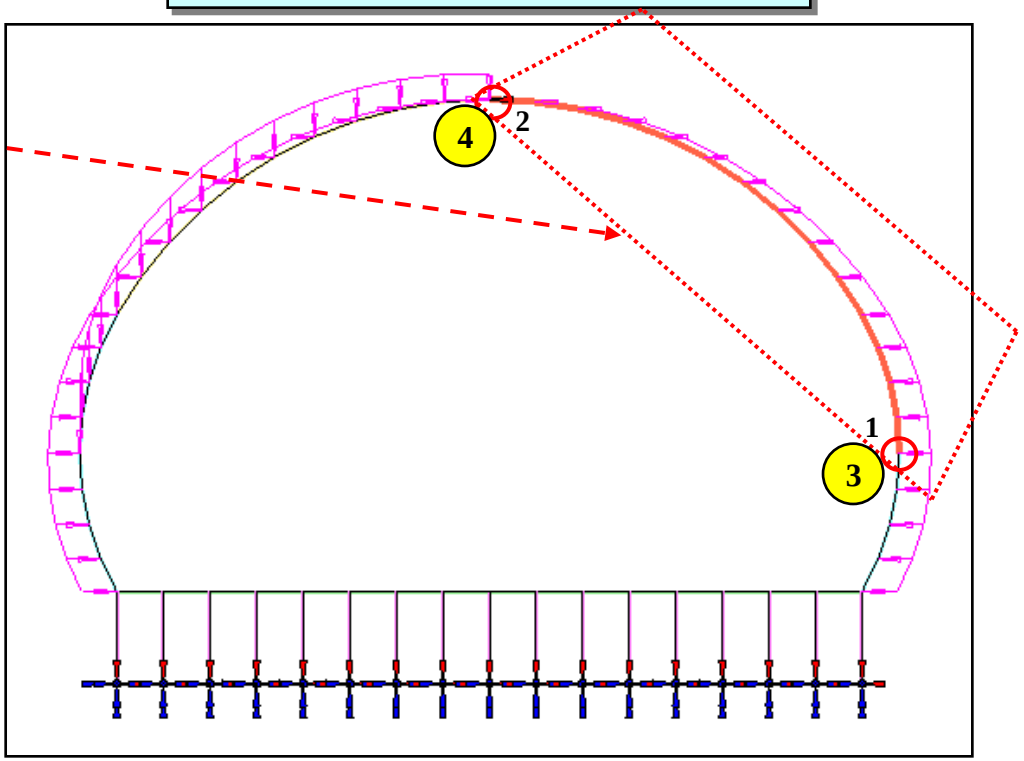




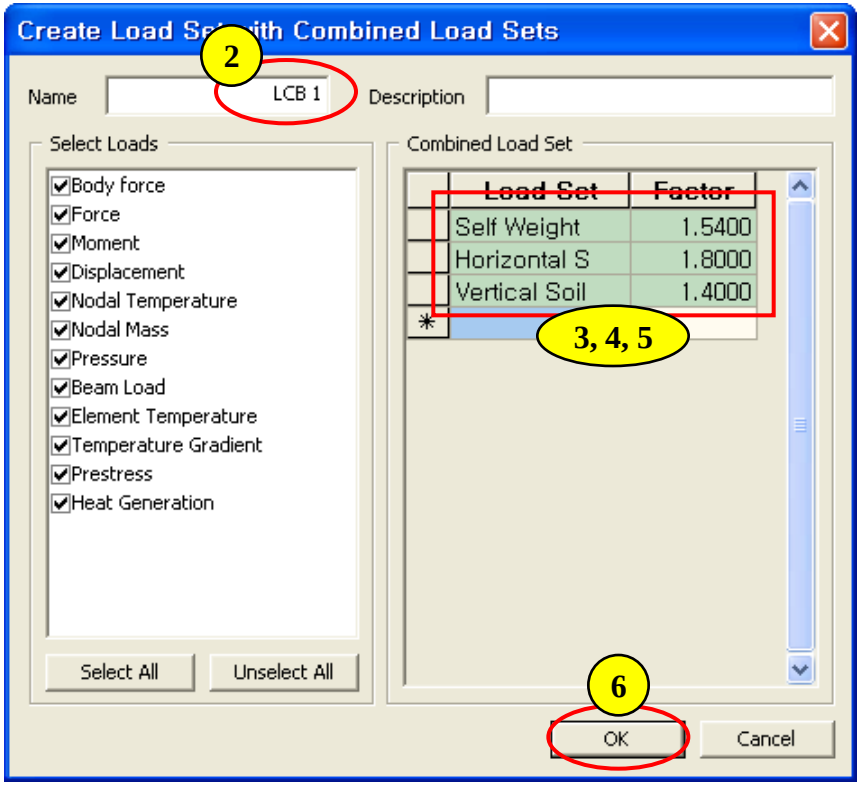
Step 16.

1. Load Set : Vertical Soil Pressure
2. Element Selection : Selected Element
3. Select First Node (See Figure)
4. Select Second Node (See Figure)
5. Select 14 Elements (See Figure)
6. Direction : Global Z
7. x1 : 0 , x2 : 1
   w1 : -13.665 , w2 : -6
8. Click [OK] Button



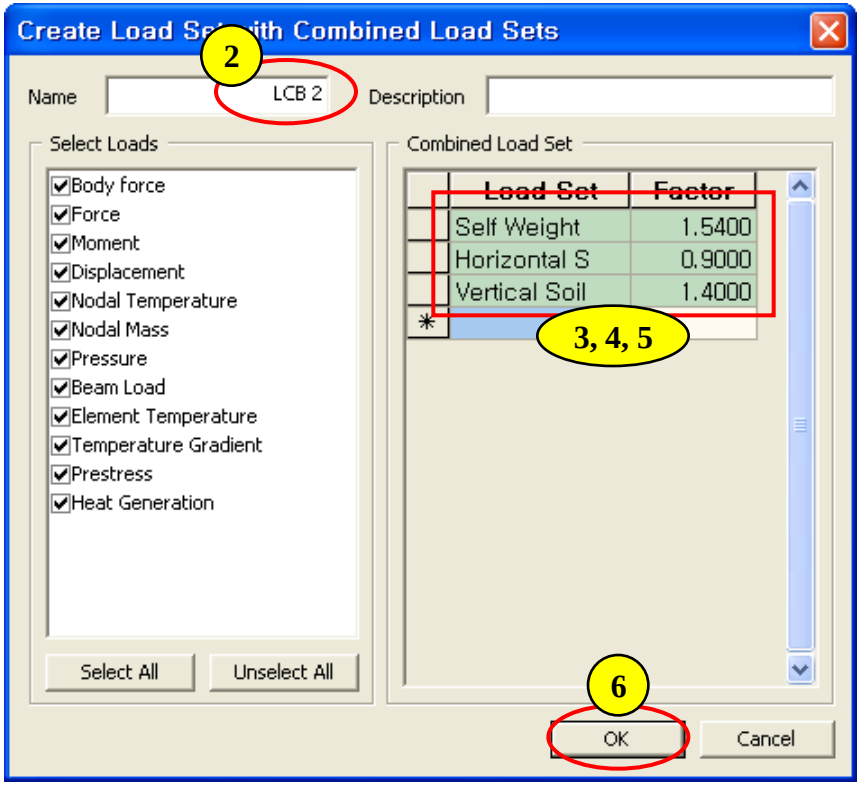


Step 17.



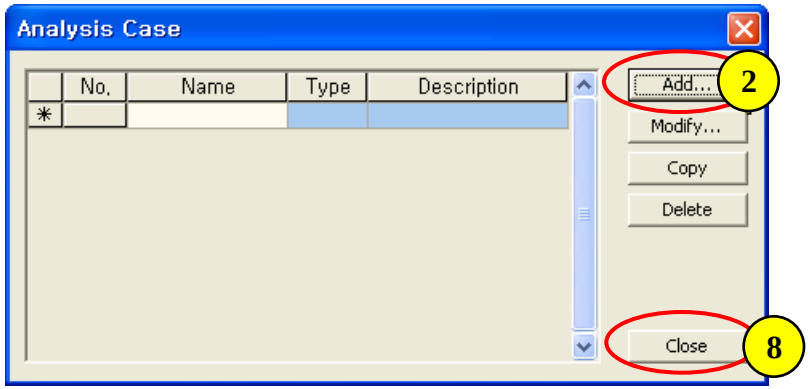
1. *Analysis > Load > Combined Set …*
2. *Name : LCB 1*
3. *Select "Self Weight" and Factor (1.54)*
4. *Select "Horizontal Soil Pressure" and Factor (1.8)*
5. *Select "Vertical Soil Pressure" and Factor (1.4)*
6. *Click* **[OK]** *Button*

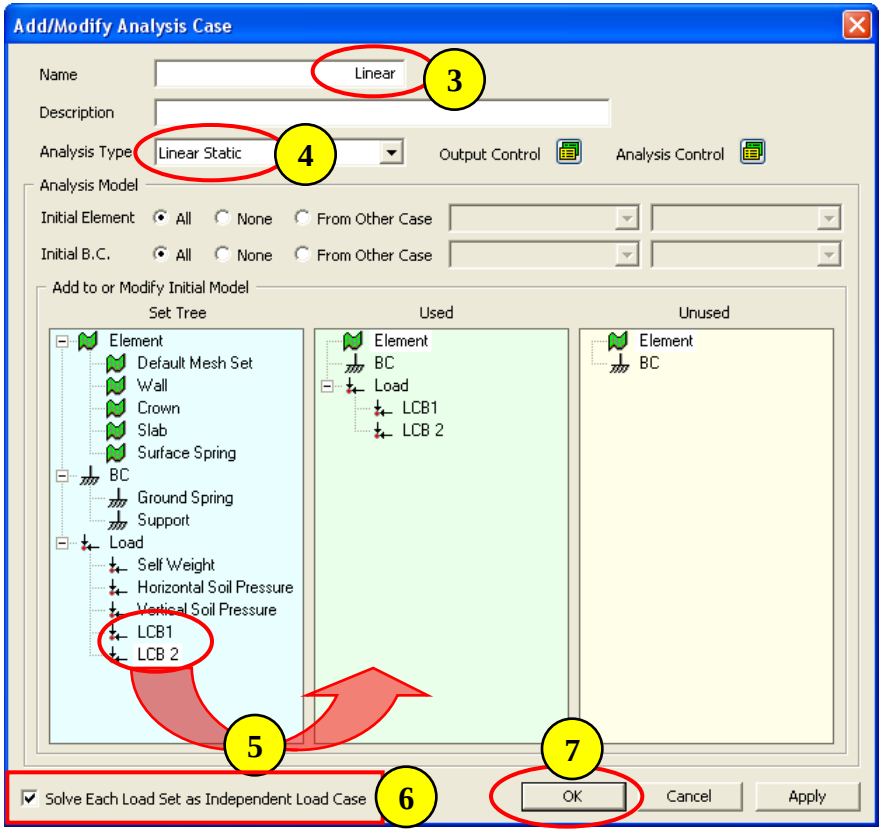
Step 18.



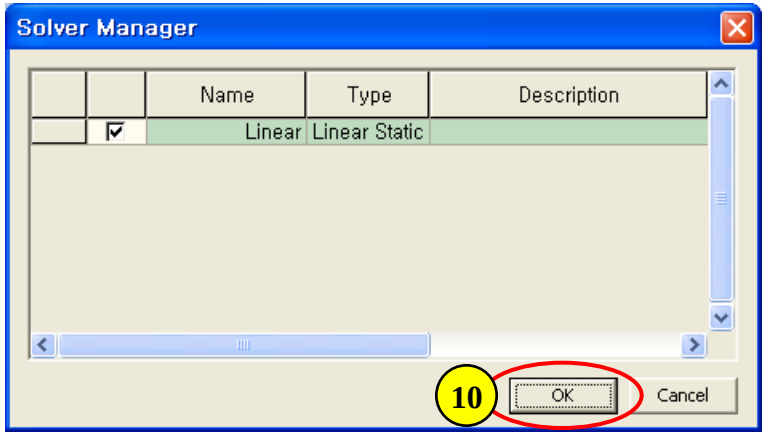
1. *Analysis > Load > Combined Set …*
2. *Name : LCB 2*
3. *Select "Self Weight" and Factor (1.54)*
4. *Select "Horizontal Soil Pressure" and Factor (0.9)*
5. *Select "Vertical Soil Pressure" and Factor (1.4)*
6. *Click* **[OK]** *Button*

Step 19.



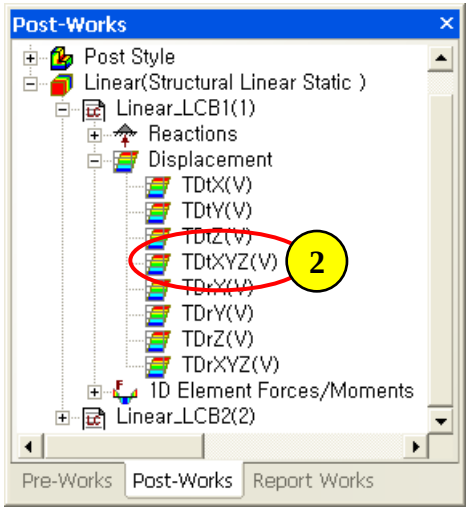


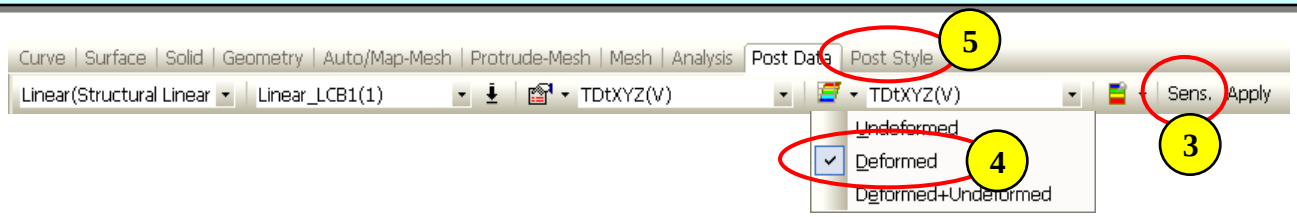
1. *Analysis > Analysis Case …*
2. *Click* **[Add]** *Button*
3. *Name : Linear*
4. *Analysis Type : Linear Static*
5. *Drag & Drop "LCB 1" and "LCB 2" to "Used" Window*
6. *Check on "Solve Each Load Set as Independent Load Case"*
7. *Click* **[OK]** *Button*
8. *Click* **[Close]** *Button*
9. *Analysis > Solve ...*
10. *Click* **[OK]** *Button*

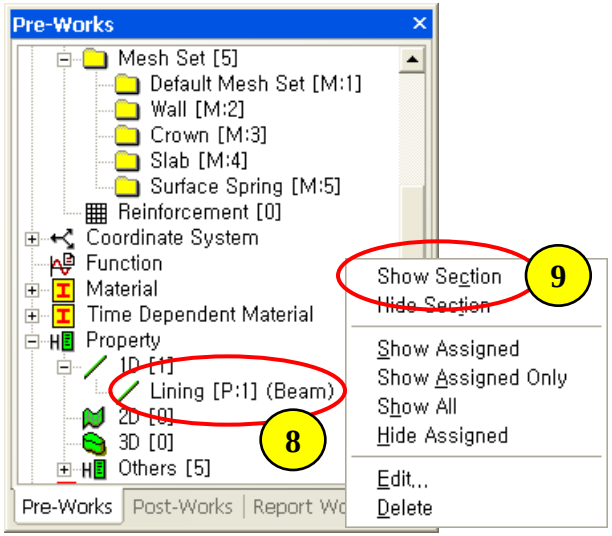


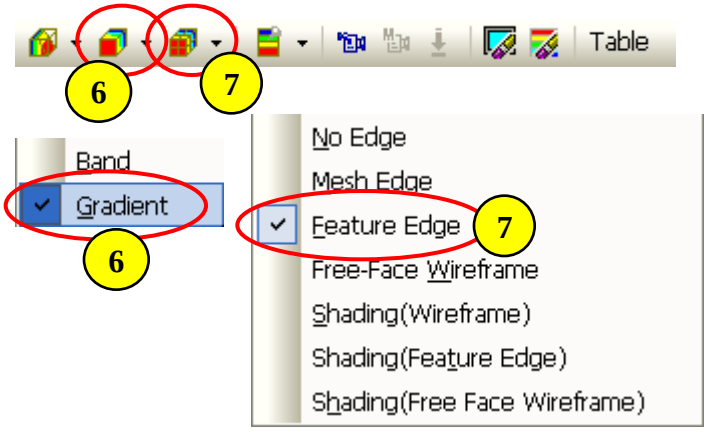
Step 20-1.

1. *Post-Works Tree : Linear(Structural Linear Static) > Linear_LCB1(1) > Displacement...*
2. *Double Click "TDtXYZ(V)"*
3. *Click "Sens." Button*
4. *Select "Deformed" for Mesh Shape (See Figure)*
5. *Click "Post Style" Toolbar*
6. *Select "Gradient" for Contour Type*
7. *Select "Feature Edge" for Edge Type*
8. *Pre-Works Tree : Property > 1D > Lining [P:1] (Beam)...*
9. *Click Right Mouse Button and Select "Show Section"*

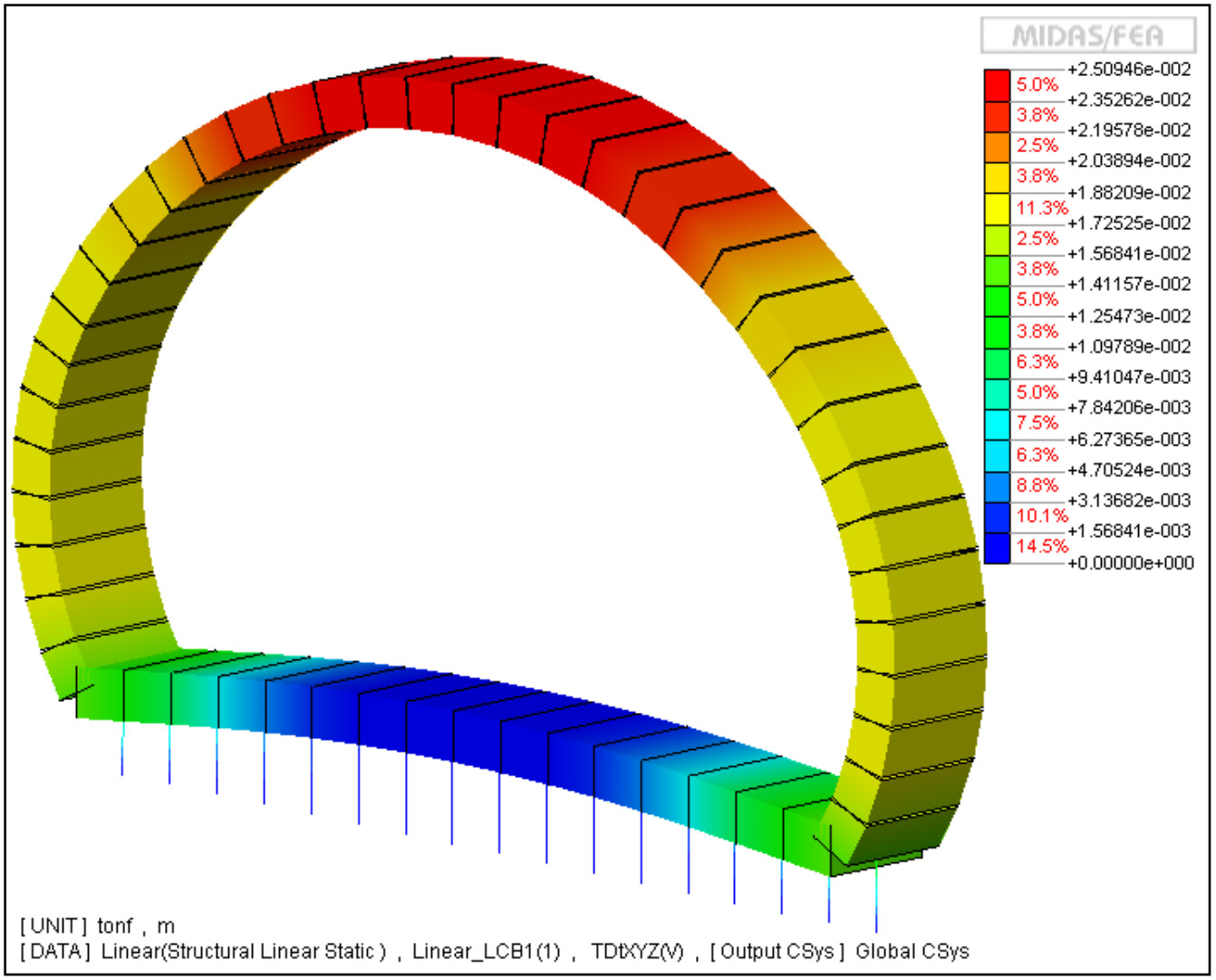




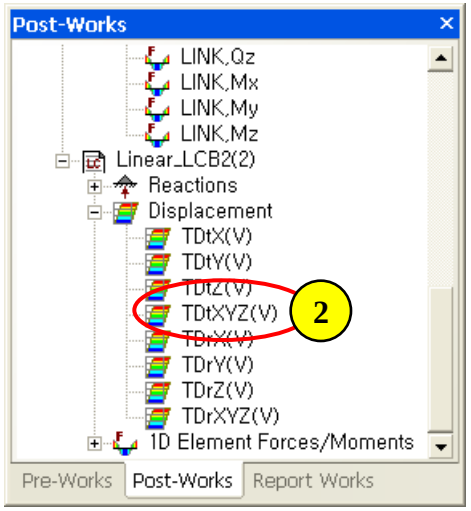




Step 20-2.

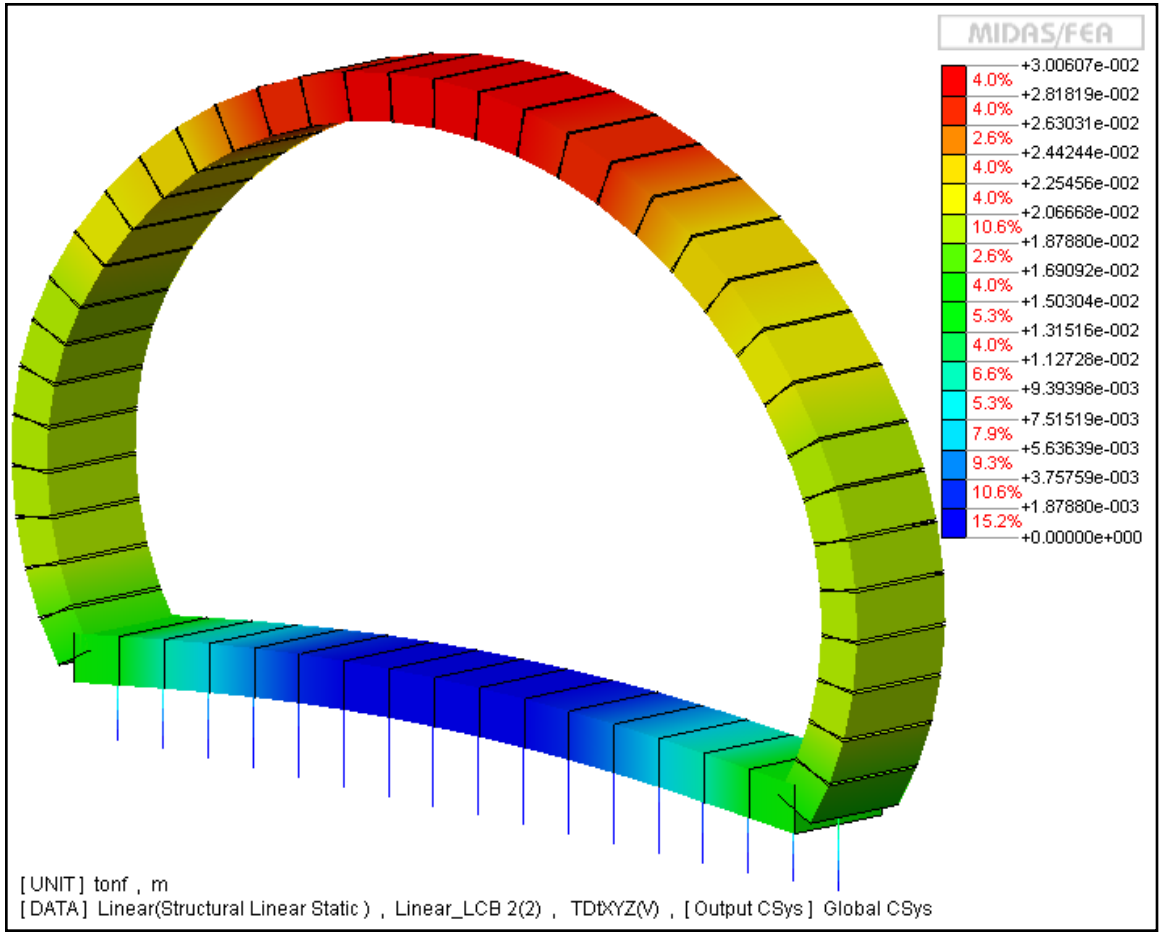

MIDAS/FEA
5.0% +2.50946e-002
3.8% +2.35262e-002
2.5% +2.19578e-002
3.8% +2.03894e-002
11.3% +1.88209e-002
2.5% +1.72525e-002
3.8% +1.56841e-002
5.0% +1.41157e-002
3.8% +1.25473e-002
6.3% +1.09789e-002
5.0% +9.41047e-003
7.5% +7.84206e-003
6.3% +6.27365e-003
8.8% +4.70524e-003
10.1% +3.13682e-003
14.5% +1.56841e-003
+0.00000e+000
[UNIT] tonf , m
[DATA] Linear(Structural Linear Static) , Linear_LCB1(1) , TDtXYZ(V) , [Output CSys] Global CSys

Step 21.



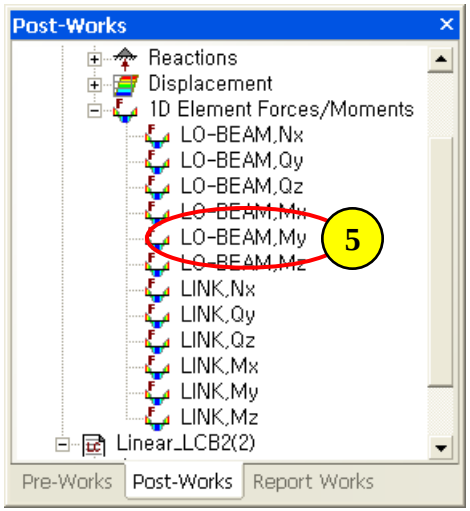
1. Post-Works Tree : Linear(Structural Linear Static) > Linear_LCB2(2) > Displacement...

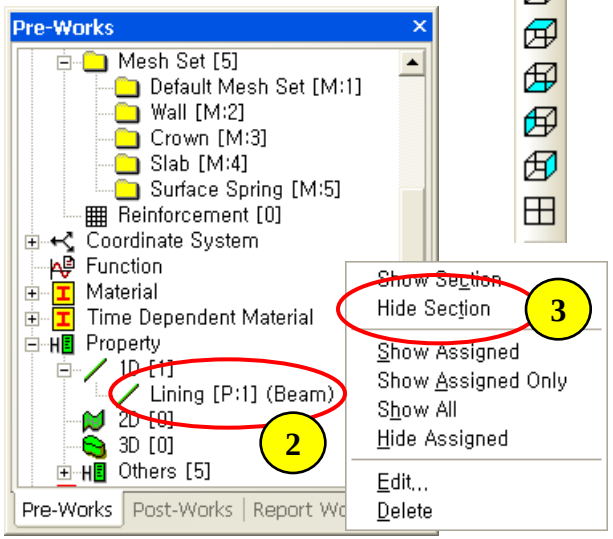
2. Double Click "TDtXYZ(V)"

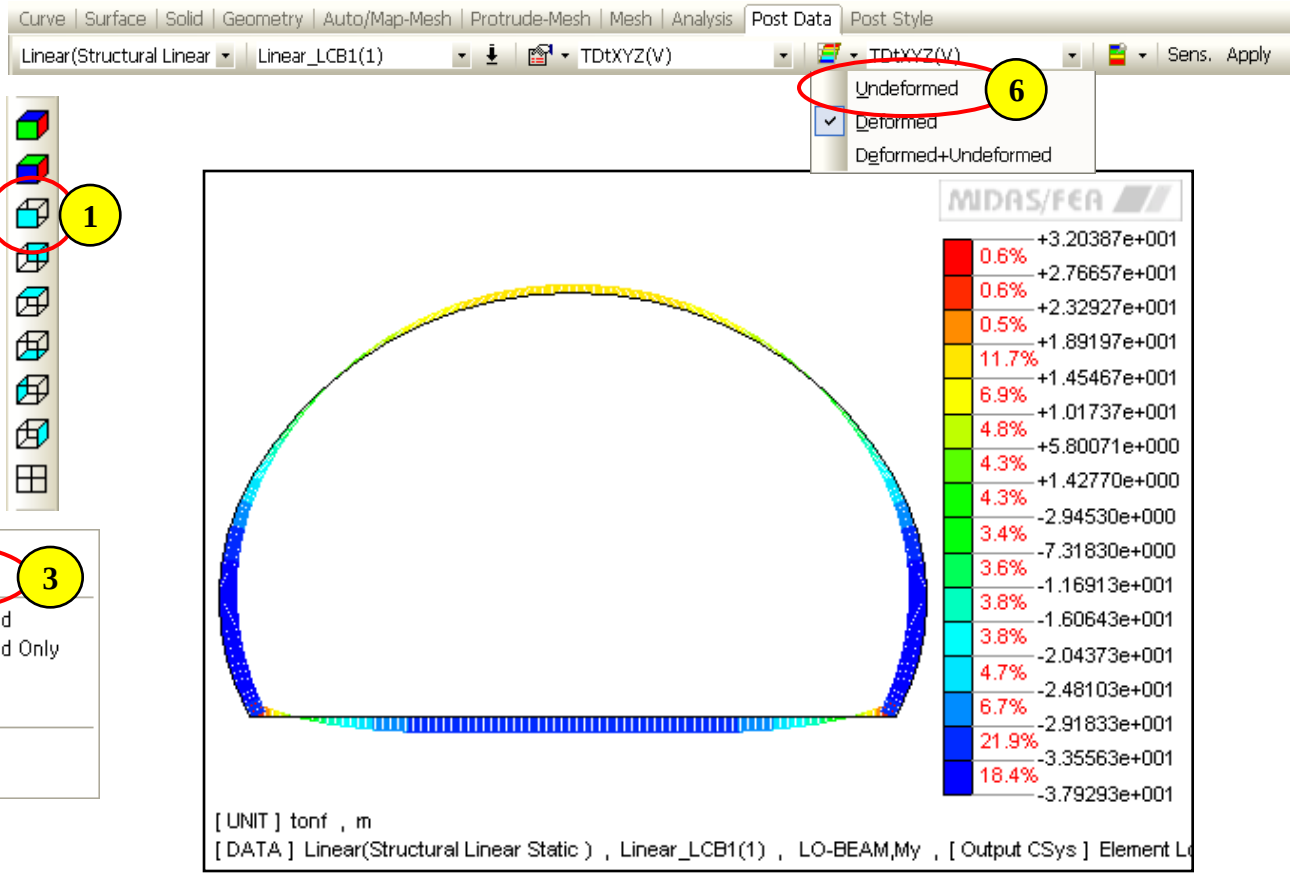


Step 22.

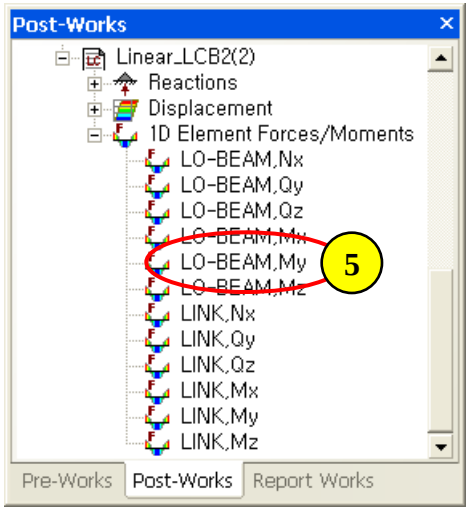
1. Click “Front View”
2. Pre-Works Tree : Property > 1D > Lining [P:1] (Beam)...
3. Click Right Mouse Button and Select “Hide Section”
4. Post-Works Tree : Linear(Structural Linear Static) > Linear_LCB1(1) > 1D Element Forces/Moments...
5. Double Click “LO-BEAM,My”
6. Select “Undeformed” for Mesh Shape (See Figure)







Step 23.



1. *Post-Works Tree : Linear(Structural Linear Static) > Linear_LCB2(2) > 1D Element Forces/Moments...*
2. *Double Click "LO-BEAM,My"*