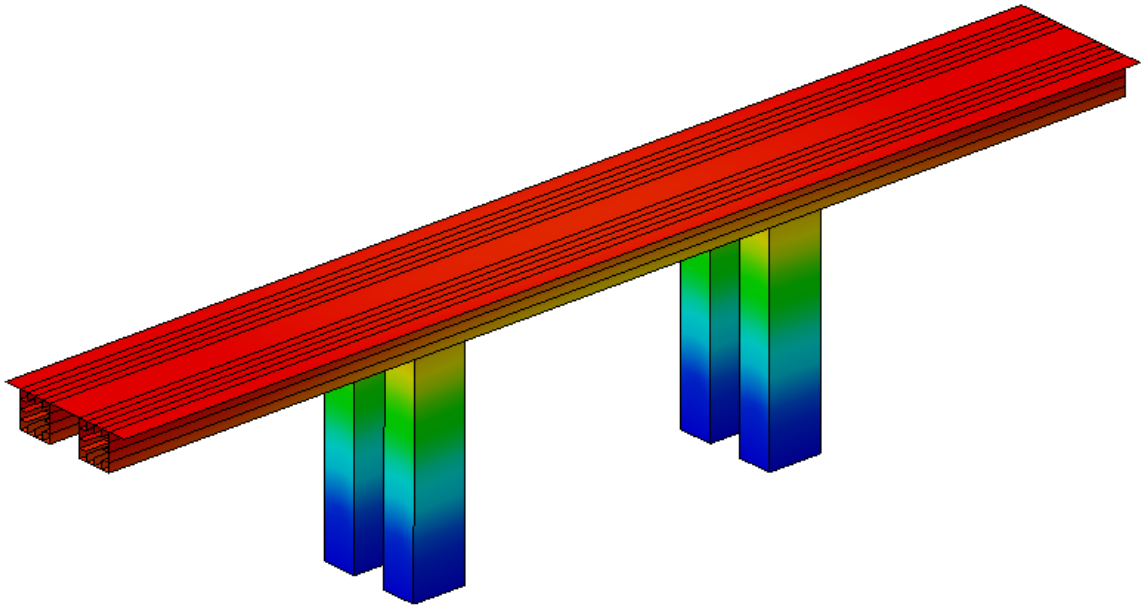
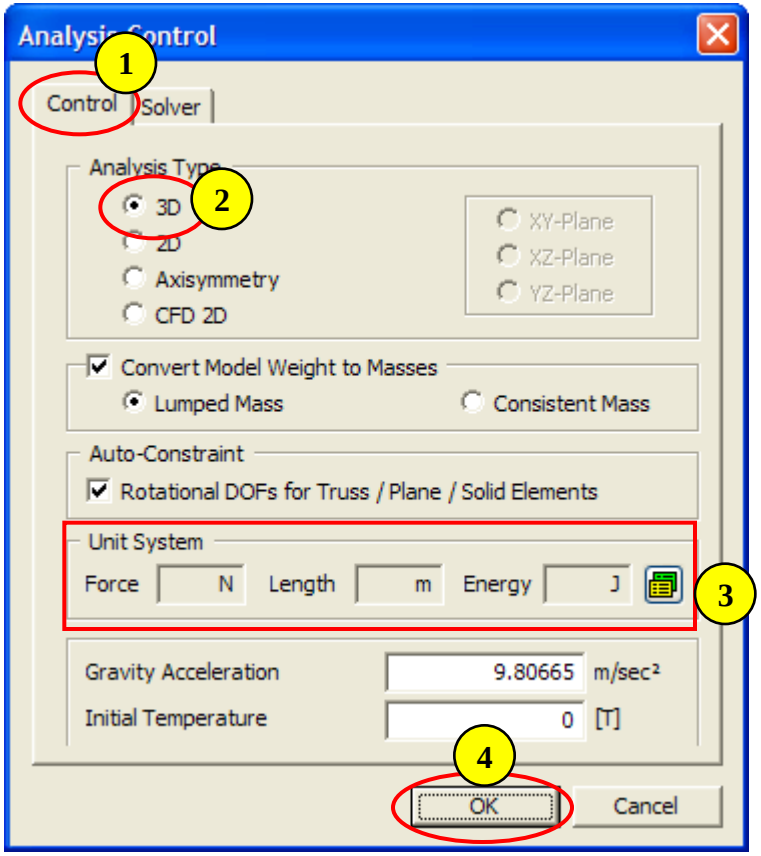
TH-1. Analysis of a Steel Box Bridge



Overview

- 3-D Time History Analysis
- Model
  - *Unit : N, m*
  - *Isotropic Elastic Material*
  - *Plate, Solid Element*
- Load & Boundary Conditions
  - *Time History Analysis*
    - *: Time Forcing Functions*
    - *: Time History Load Set*
    - *: Ground Acceleration*
  - *Constraint*
- Result Evaluation
  - *Vibration Frequency / Period*
  - *Time History Plot*

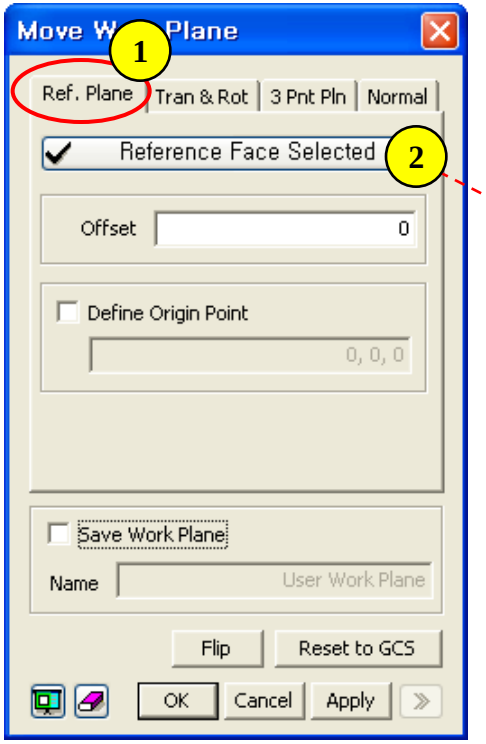
Step 1.



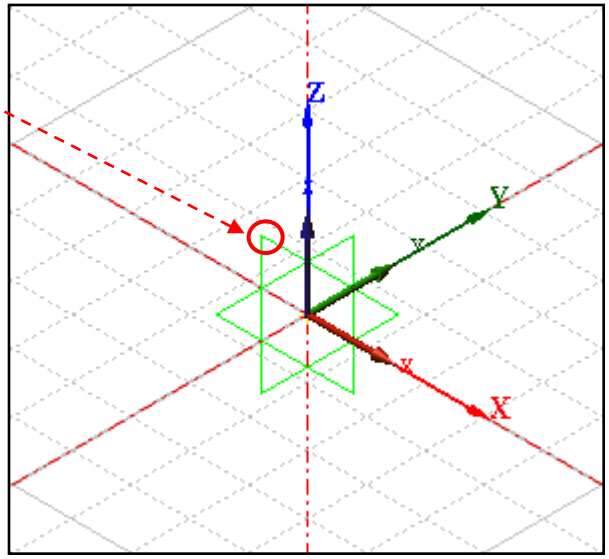
1. *Analysis > Analysis Control – "Control" tab*
2. *Analysis Type : 3D*
3. *Unit : N , m*
4. *Click* **[OK]** *Button*

Analysis Control Dialog is automatically activated at startup.

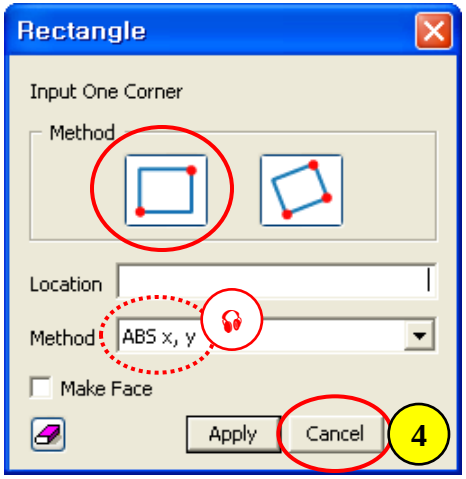
Step 2.

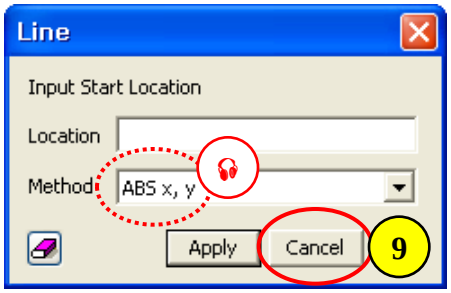


1. *Geometry > Work Plane > Move – "Ref. Plane" tab*
2. *Select "XZ Plane"*
3. *Click* **[OK]** *Button*

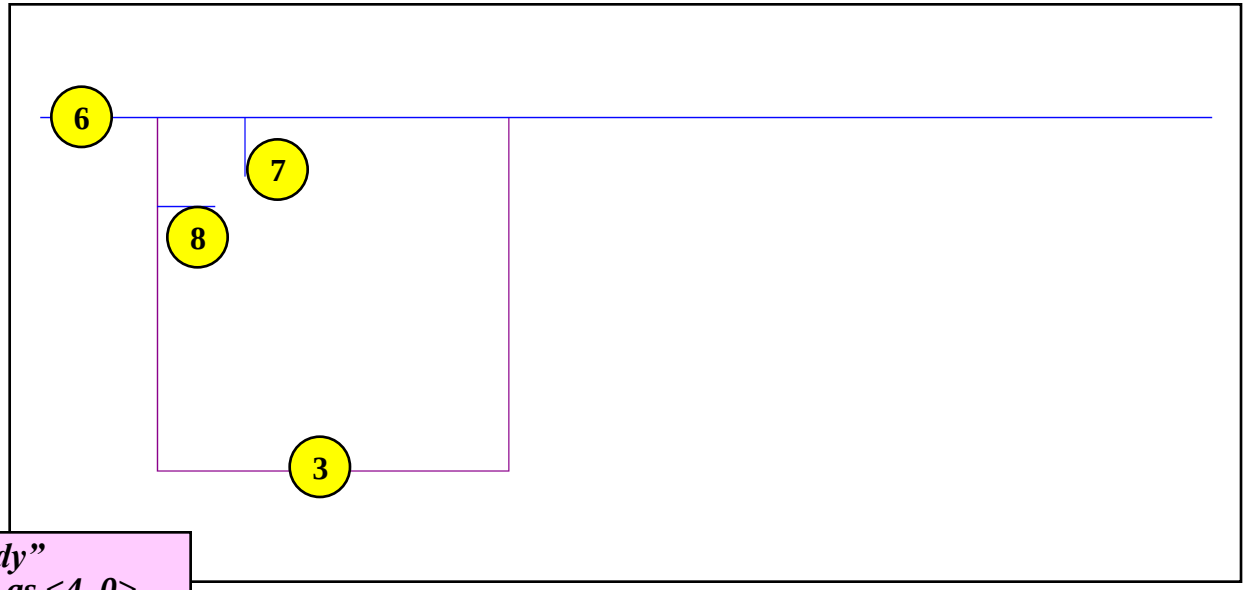


Step 3.





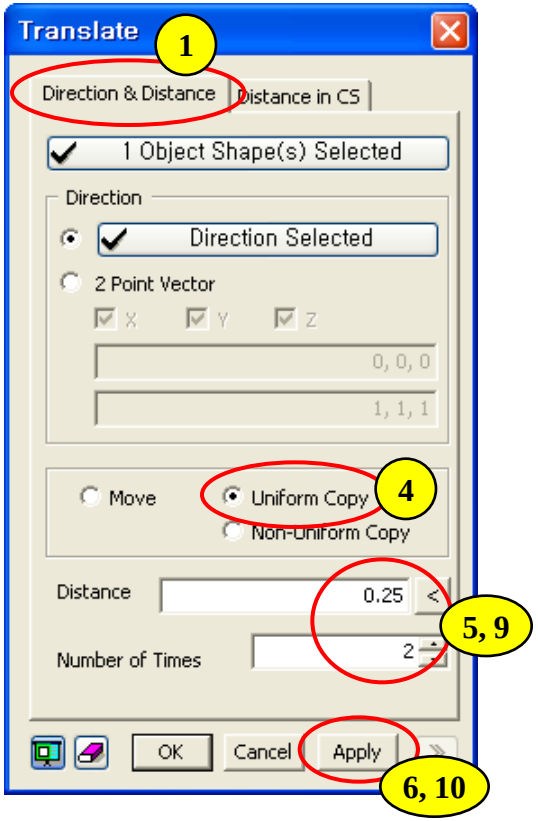
1. *Click "Normal View"*
2. *Geometry > Curve > Create on WP > Rectangle (Wire) …*
3. *Location : (0.5) , <1, -1>*
4. *Click* **[Cancel]** *Button*
5. *Geometry > Curve > Create on WP > Line …*
6. *Location : (0) , <4>*
7. *Location : (0.75) , <0, -0.2>*
8. *Location : (0.5, -0.25) , <0.2>*
9. *Click* **[Cancel]** *Button*

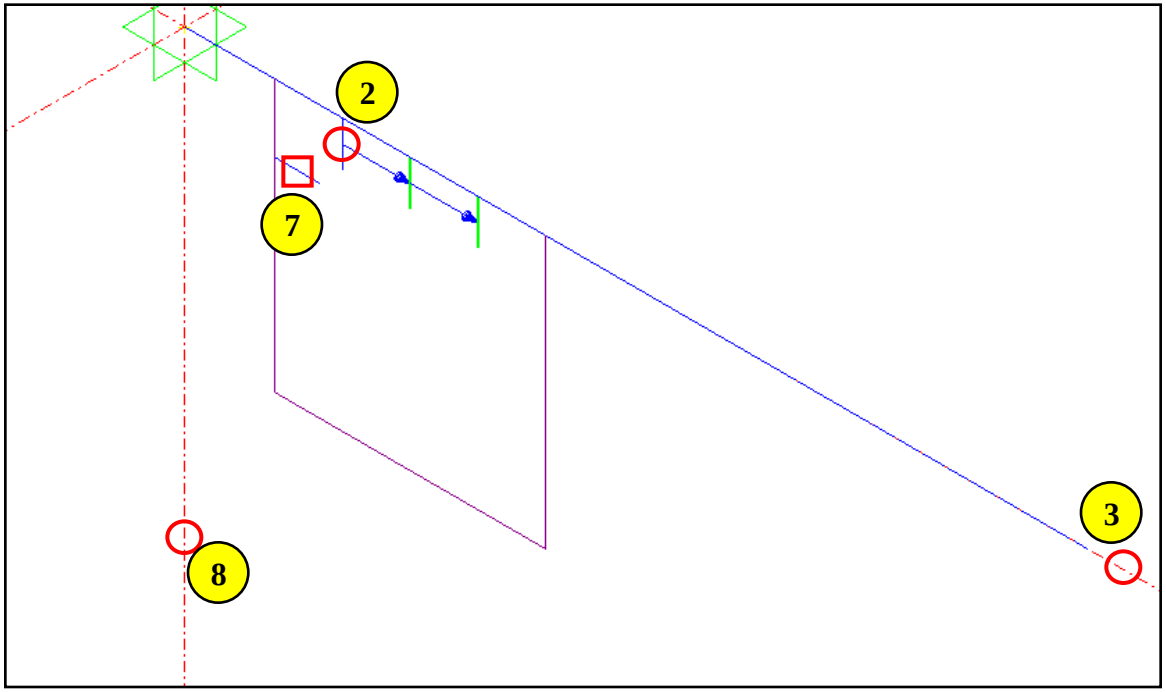


- *() : "ABS x, y" , <> : "REL dx, dy"*
  (0.5) same as (0.5, 0), <4> same as <4, 0>
- **[Esc]** *as shortcut for* **[Cancel].**

Step 4.

1. *Geometry > Transform > Translate – "Direction & Distance" tab*
2. *Select Edge marked by "O" (See Figure)*
3. *Direction : X-Axis*
4. *Check on "Uniform Copy"*
5. *Distance (0.25) , Number of Times (2)*
6. *Click* **[Apply]** *Button*
7. *Select Edge marked by "□" (See Figure)*
8. *Direction : Z-Axis*
9. *Distance (-0.25) , Number of Times (2)*
10. *Click* **[Apply]** *Button*

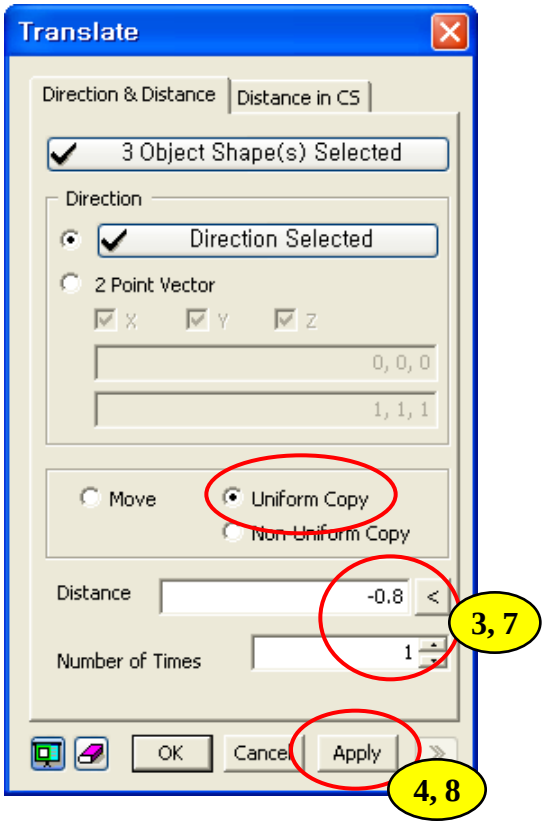


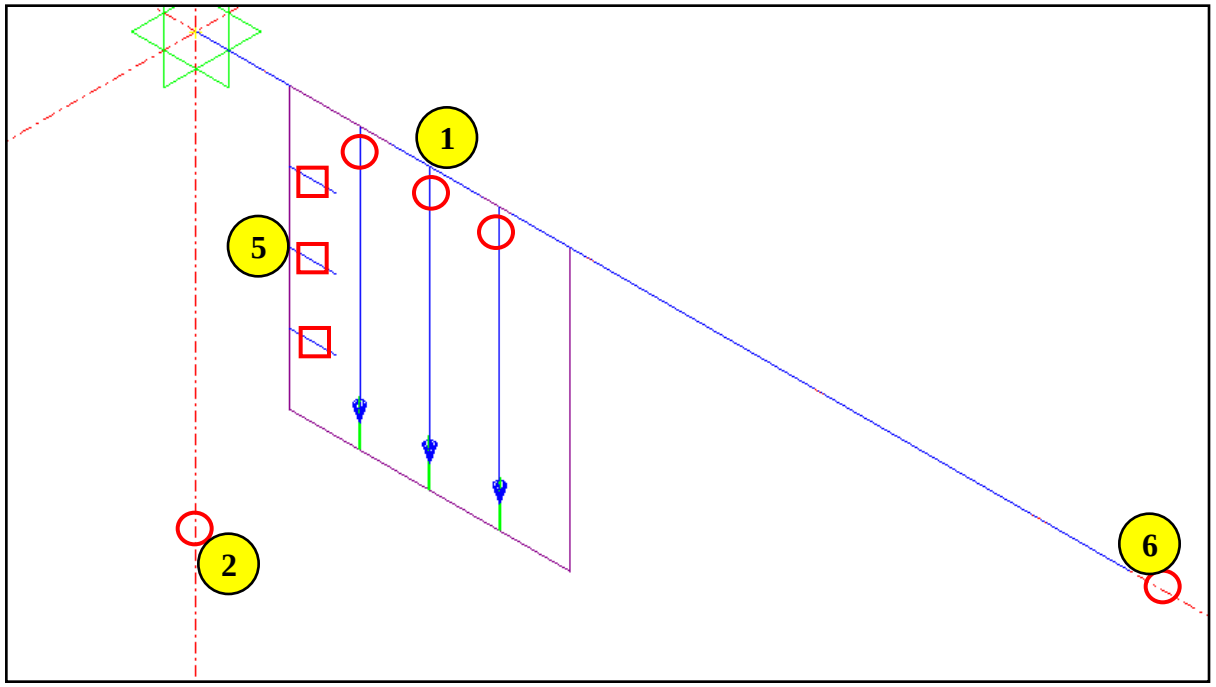


[Enter] *as shortcut for* **[Apply].**

Step 5.

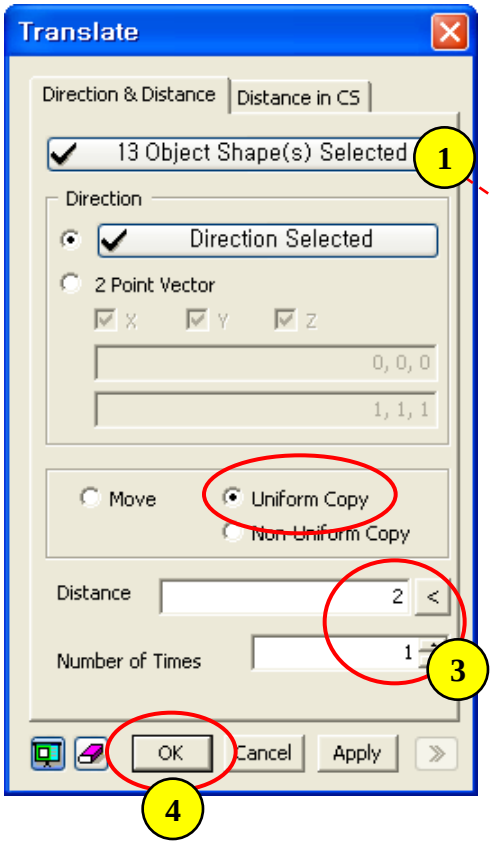
1. *Select Edges marked by "O" (See Figure)*
2. *Direction : Z-Axis*
3. *Distance (-0.8) , Number of Times (1)*
4. *Click* **[Apply]** *Button*
5. *Select Edges marked by "□" (See Figure)*
6. *Direction : X-Axis*
7. *Distance (0.8) , Number of Times (1)*
8. *Click* **[Apply]** *Button*

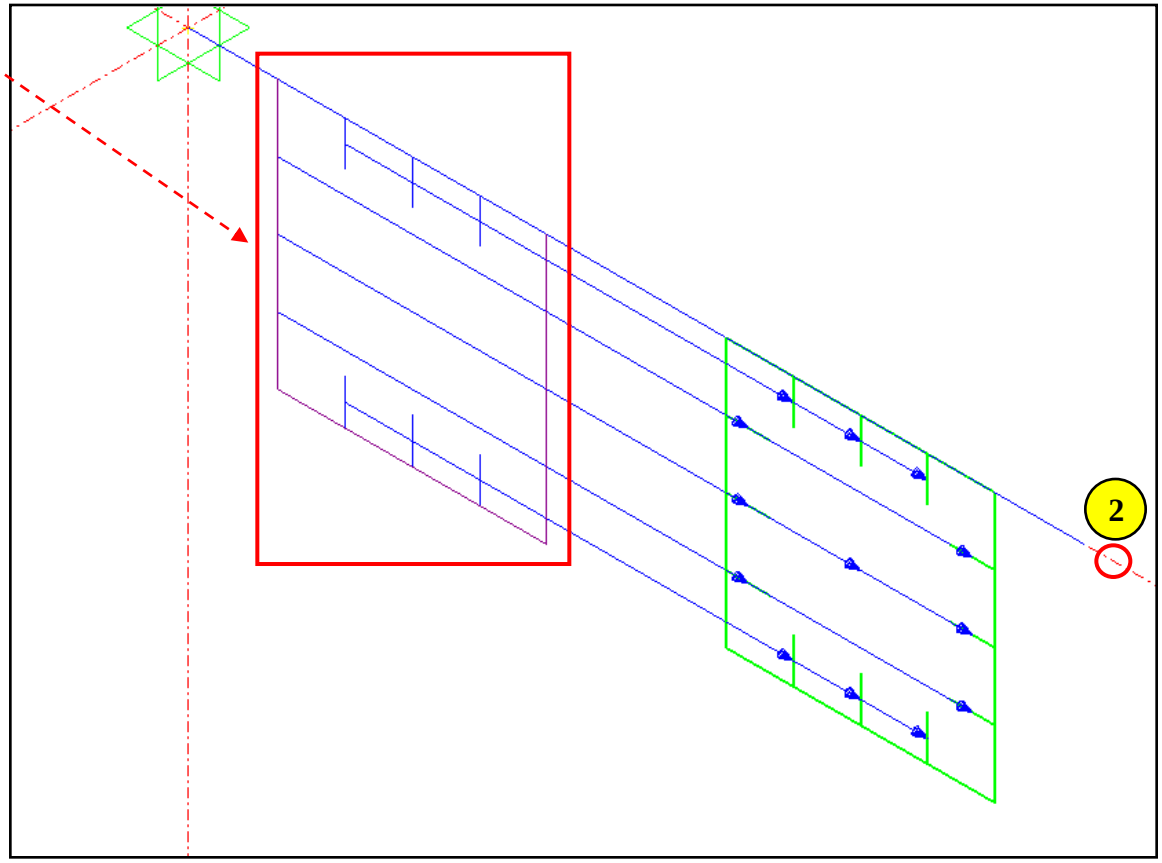




Step 6.

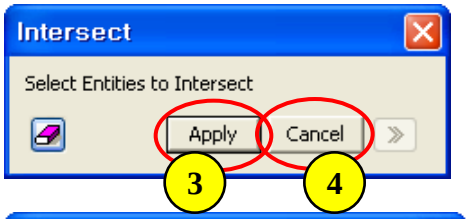
1. *Select Edges (See Figure)*
2. *Direction : X-Axis*
3. *Distance (2) , Number of Times (1)*
4. *Click* **[OK]** *Button*

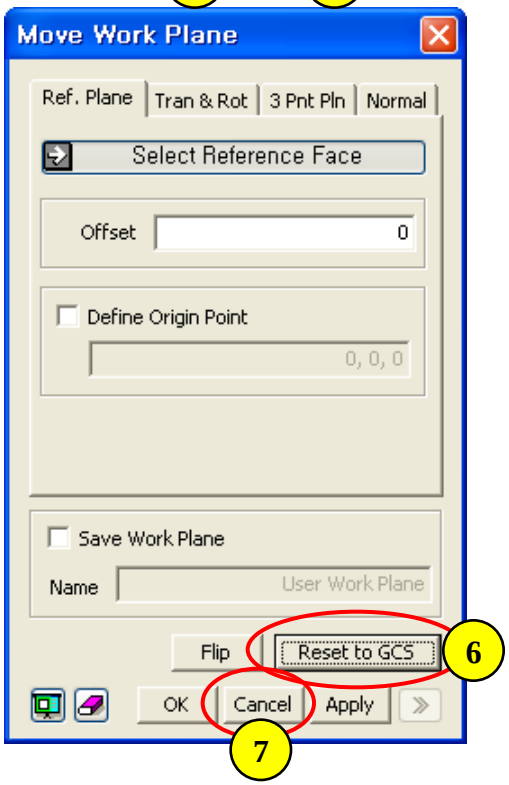


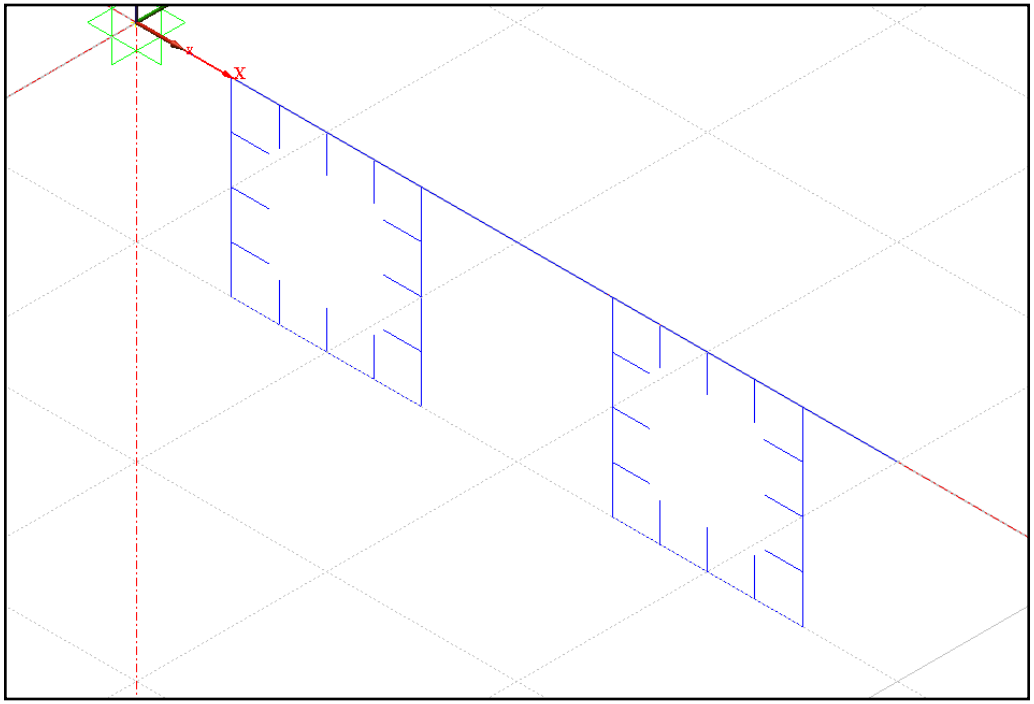


Step 7.

1. *Geometry > Curve > Intersect …*
2. *Select* *"Displayed"*
3. *Click* **[Apply]** *Button*
4. *Click* **[Cancel]** *Button*
5. *Geometry > Work Plane > Move …*
6. *Click* **[Reset to GCS]** *Button*
7. *Click* **[Cancel]** *Button*



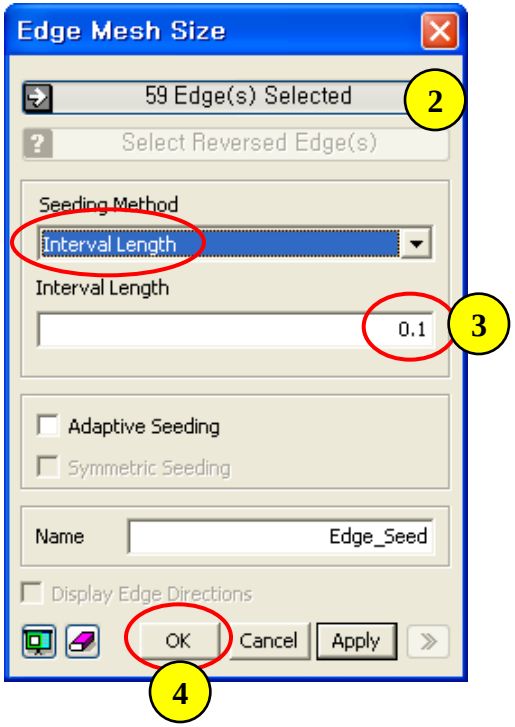


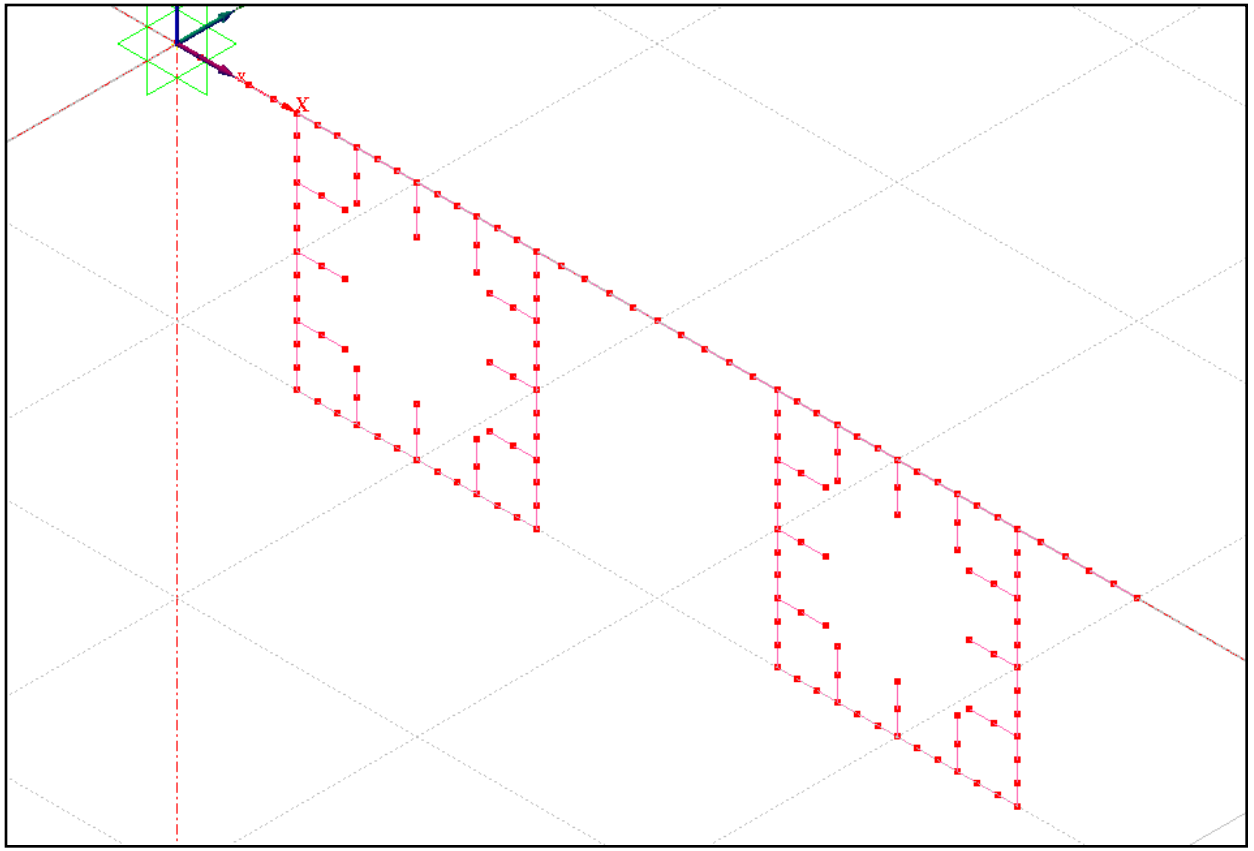


"Ctrl+A" as shortcut for "Select Displayed".

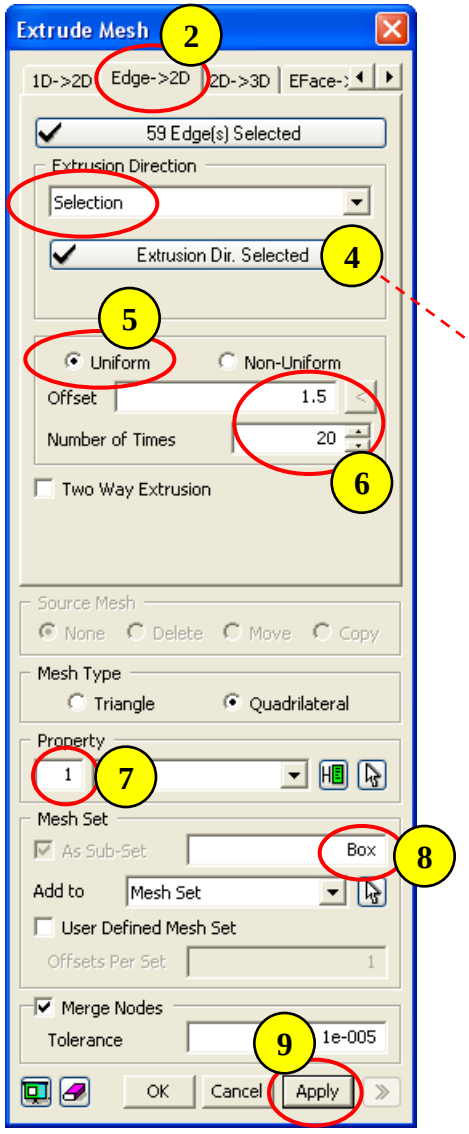
Step 8.

1. *Mesh > Size Control > Along Edge …*
2. *Select* “*Displayed*”
3. *Seeding Method - Interval Length : 0.1*
4. *Click* **[OK]** *Button*

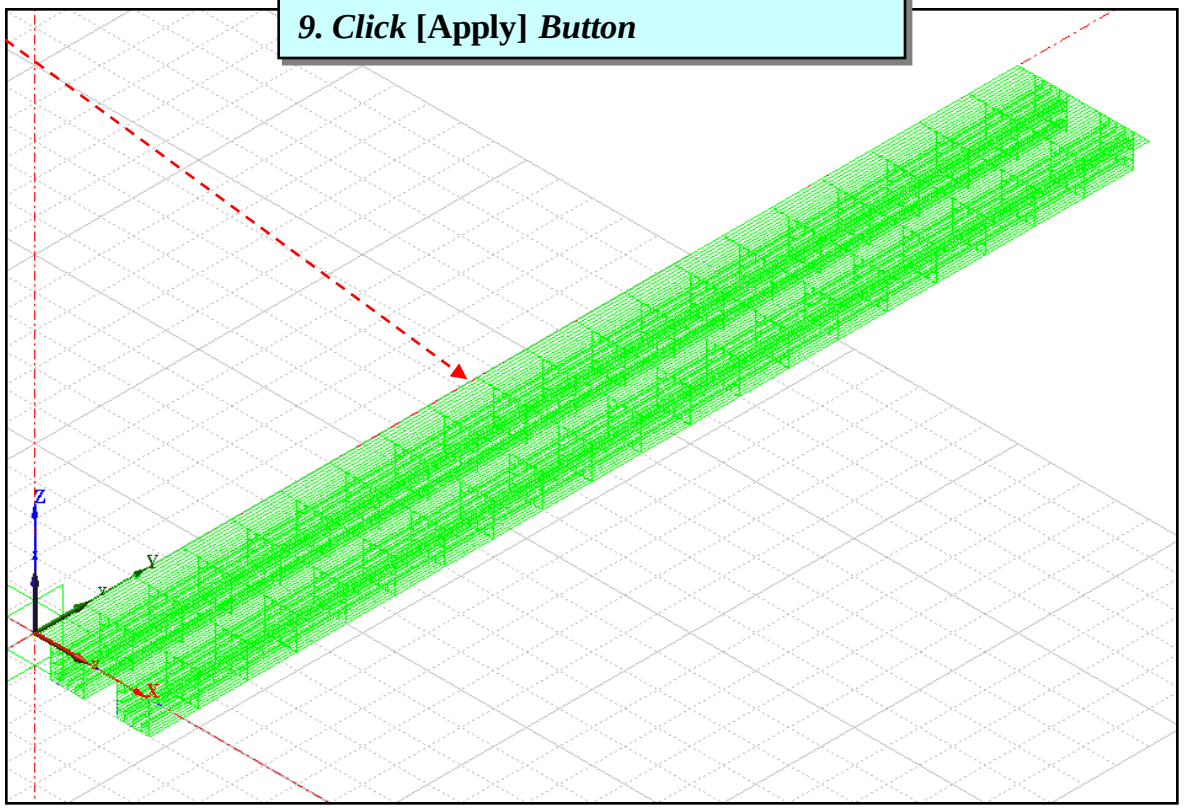




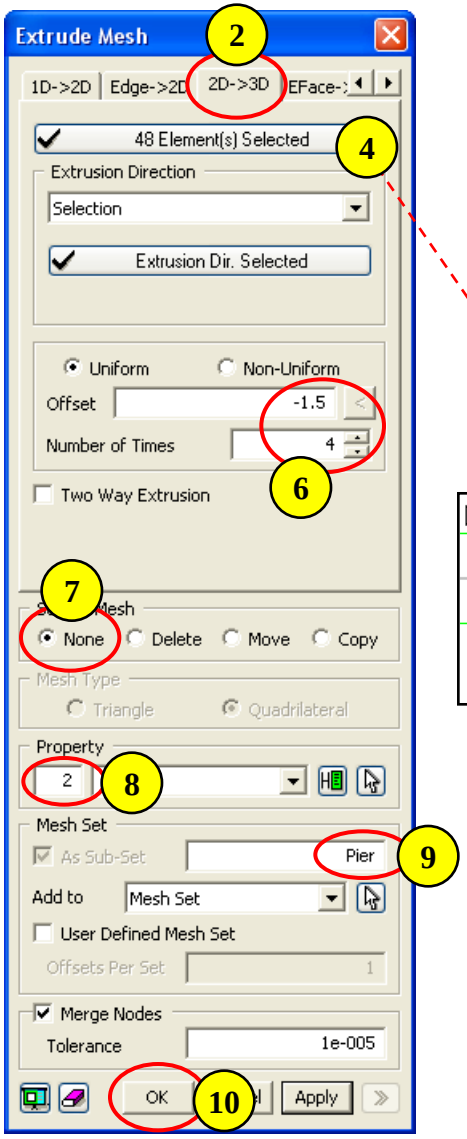
Step 9.

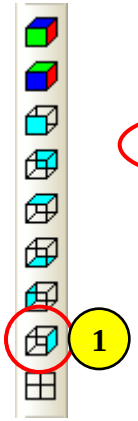


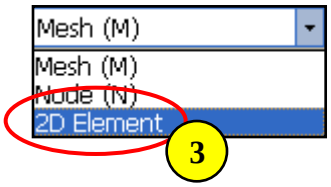
1. *Mesh > Protrude Mesh > Extrude ...*
2. *Select "Edge->2D" Tab*
3. *Select "Displayed"*
4. *Extrusion Direction : Y-Axis*
5. *Check on "Uniform"*
6. *Offset : 1.5 , Number of Times : 20*
7. *Property (1)*
8. *Mesh Set : Box*
9. *Click* **[Apply]** *Button*



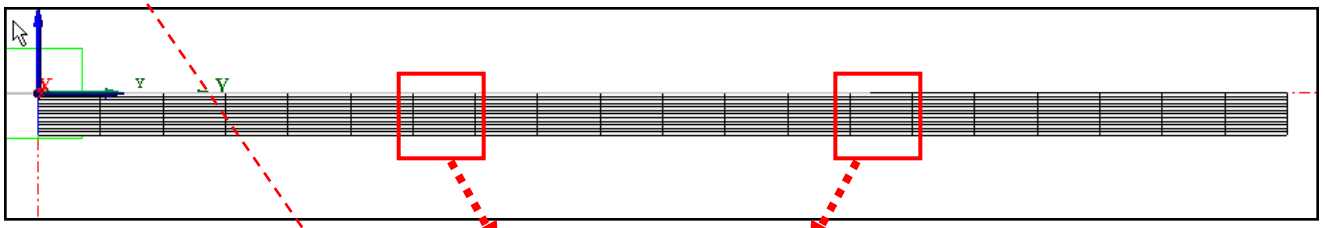
Step 10.

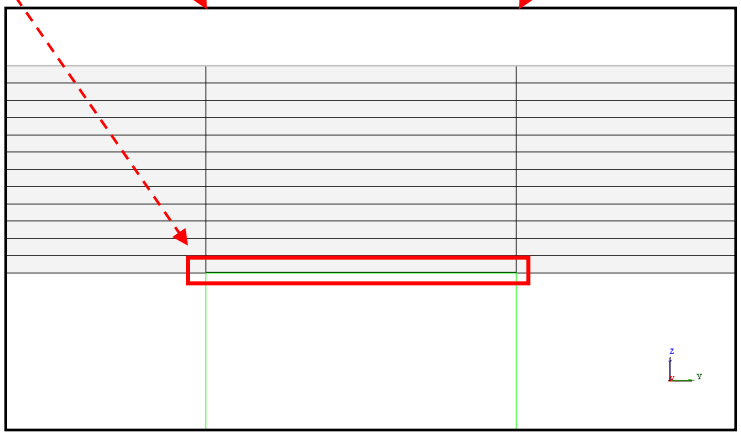




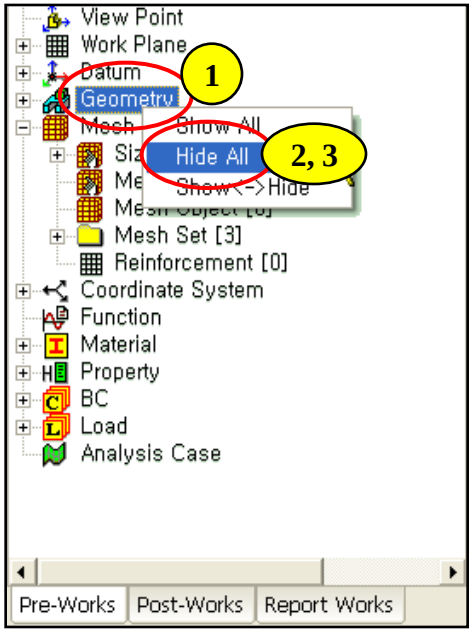


1. *Click “Right View”*
2. *Select “2D->3D” Tab*
3. *Change Selection Filter to “2D Element”*
4. *Select 48 Elements (See Figure)*
5. *Extrusion Direction : Z-Axis*
6. *Offset (-1.5) , Number of Times (4)*
7. *Source Mesh : None*
8. *Property (2)*
9. *Mesh Set : Pier*
10. *Click* **[OK]** *Button*



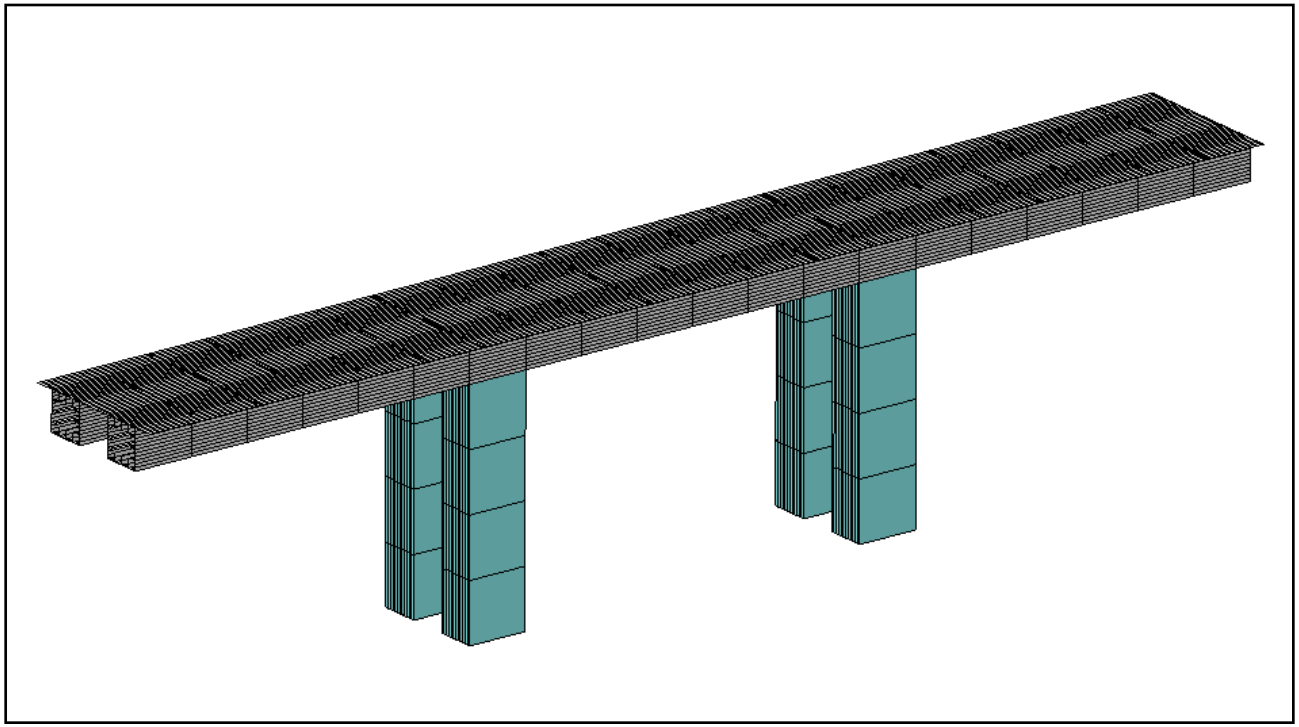


Step 11.

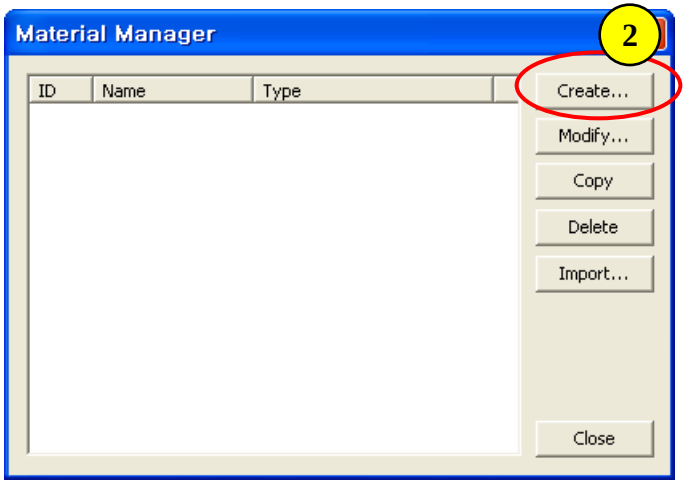


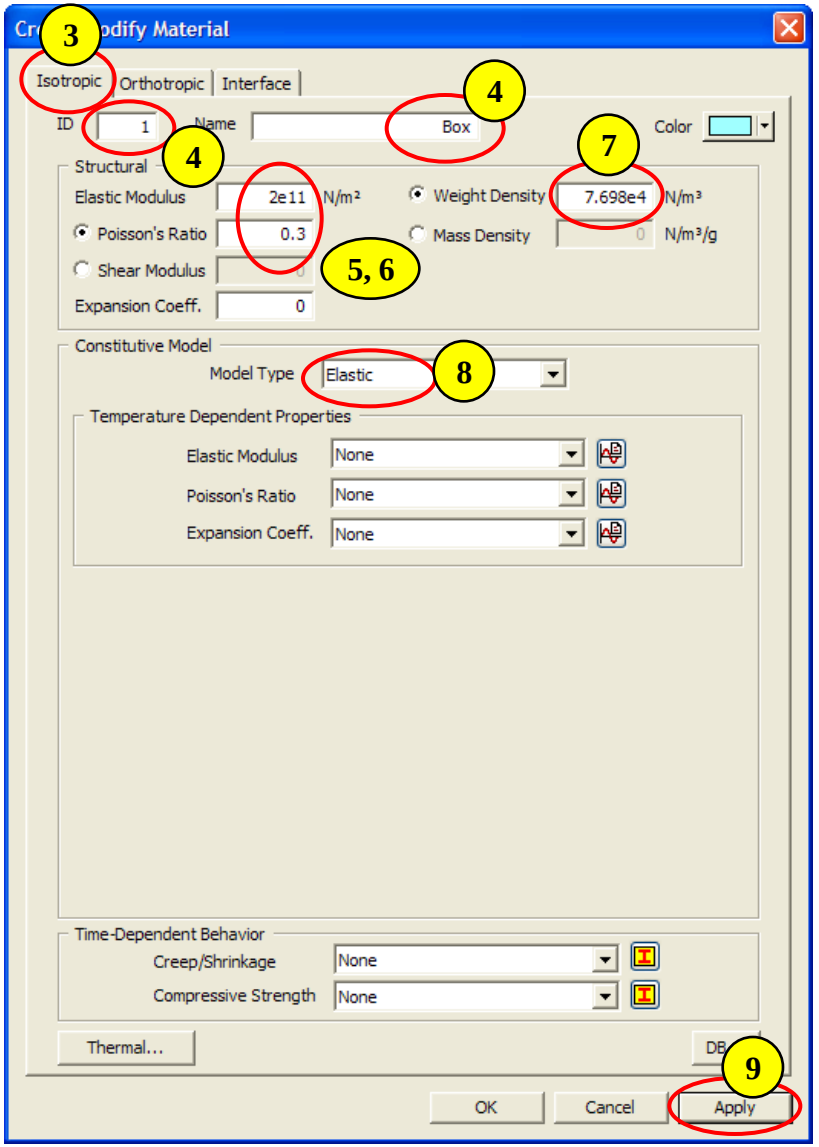
1. Pre-Works Tree : Geometry ...

2. Click Right Mouse Button and Select "Hide All"



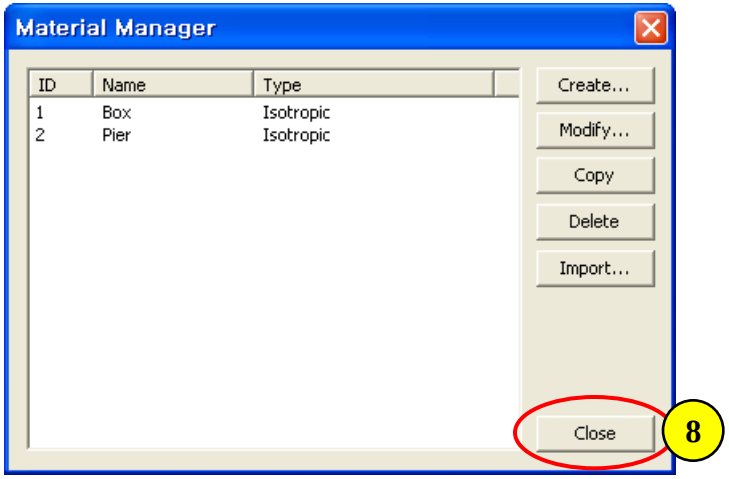
Step 12.

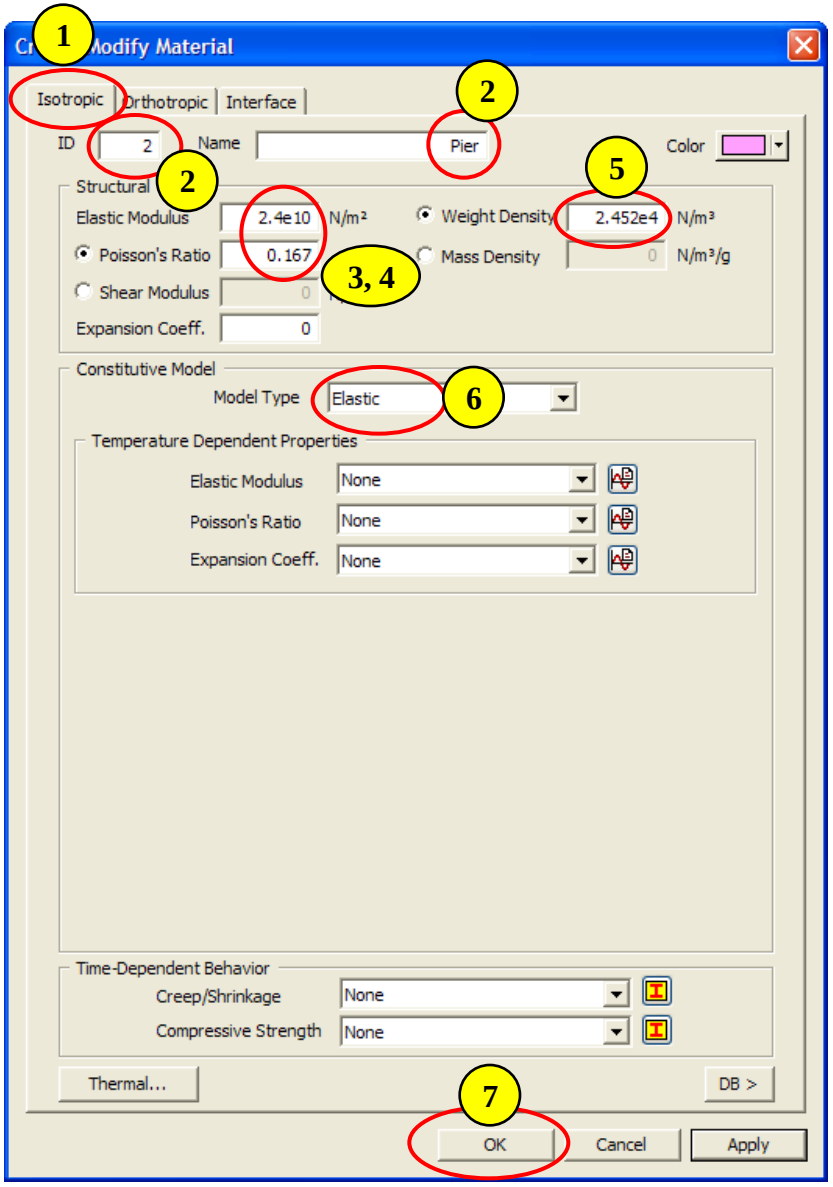




1. *Analysis > Material …*
2. *Click* [Create] *Button*
3. *Select "Isotropic" tab*
4. *ID: 1 , Name : Box*
5. *Elastic Modulus : 2e11 N/m²*
6. *Poisson's Ratio : 0.3*
7. *Weight Density : 7.698e4 N/m³*
8. *Model Type : Elastic*
9. *Click* [Apply] *Button*

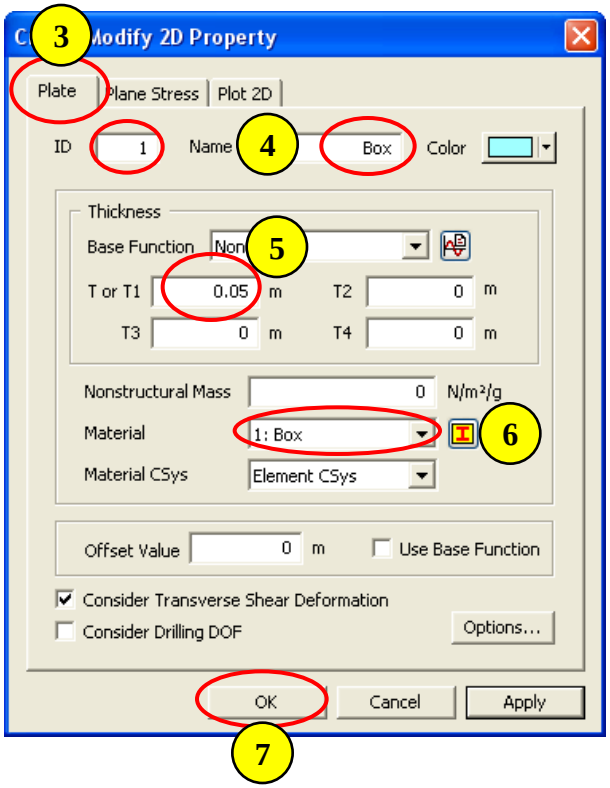
Step 13.

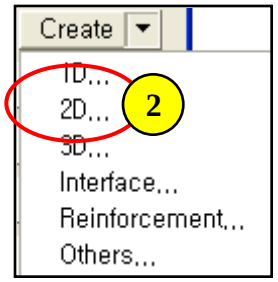




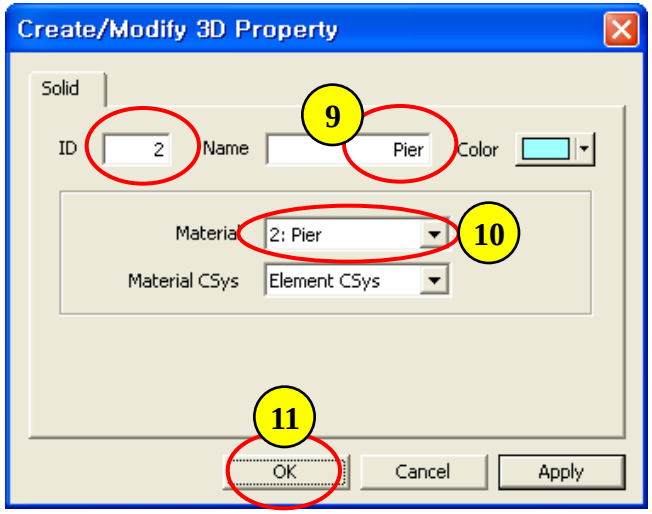
1. *Select "Isotropic" tab*
2. *ID : 2 , Name : Pier*
3. *Elastic Modulus : 2.4e10 N/m²*
4. *Poisson's Ratio : 0.167*
5. *Weight Density : 2.452e4 N/m³*
6. *Model Type : Elastic*
7. *Click* **[OK]** *Button*
8. *Click* **[Close]** *Button*

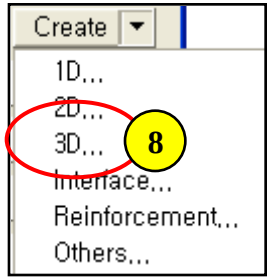
Step 14.



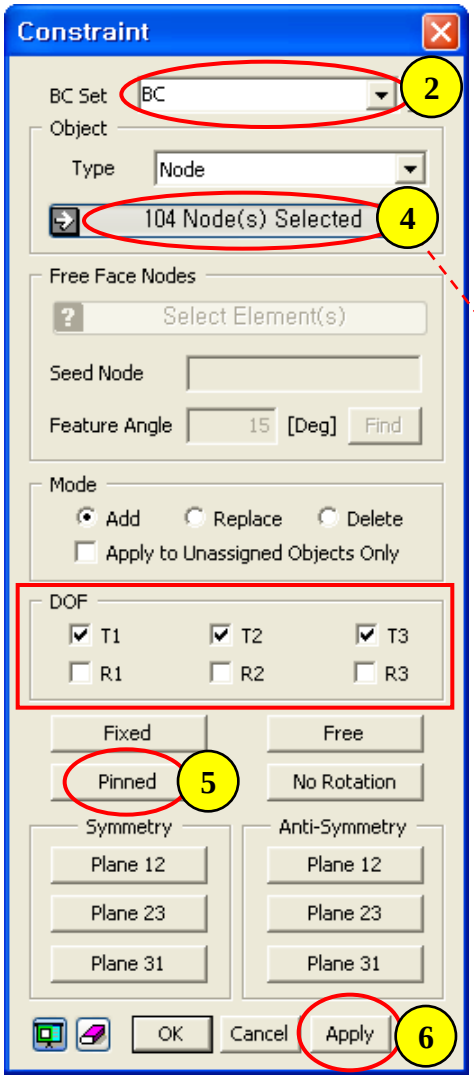


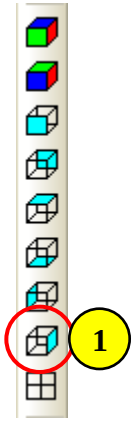
1. *Analysis > Property …*
2. *Create 2D …*
3. *Select "Plate" Tab*
4. *ID : 1 , Name : Box*
5. *Thickness : 0.05m*
6. *Select "1: Box" for Material*
7. *Click* **[OK]** *Button*
8. *Create 3D ...*
9. *ID : 2 , Name : Pier*
10. *Select "2: Pier" for Material*
11. *Click* **[OK]** *Button*
12. *Click* **[Close]** *Button*



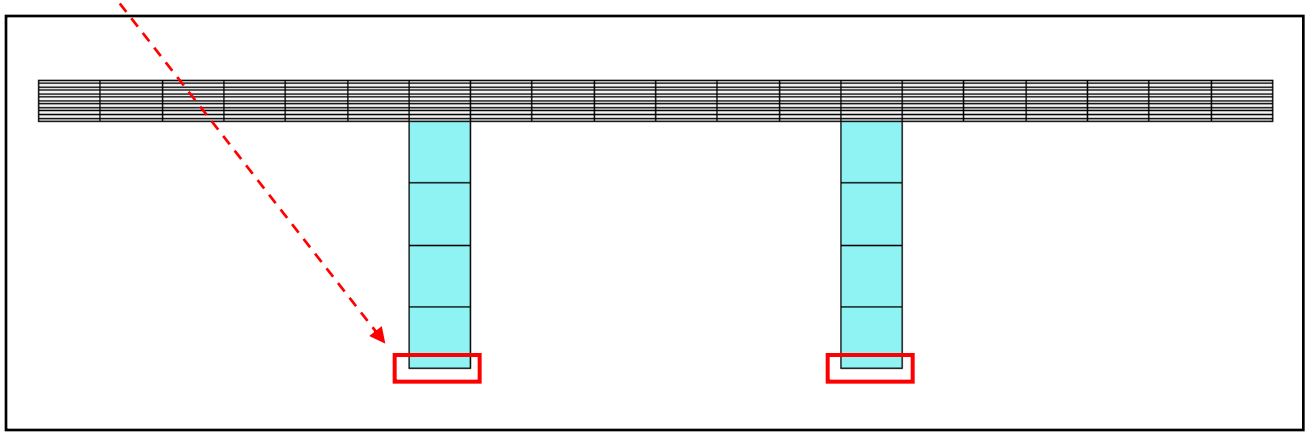


Step 15.

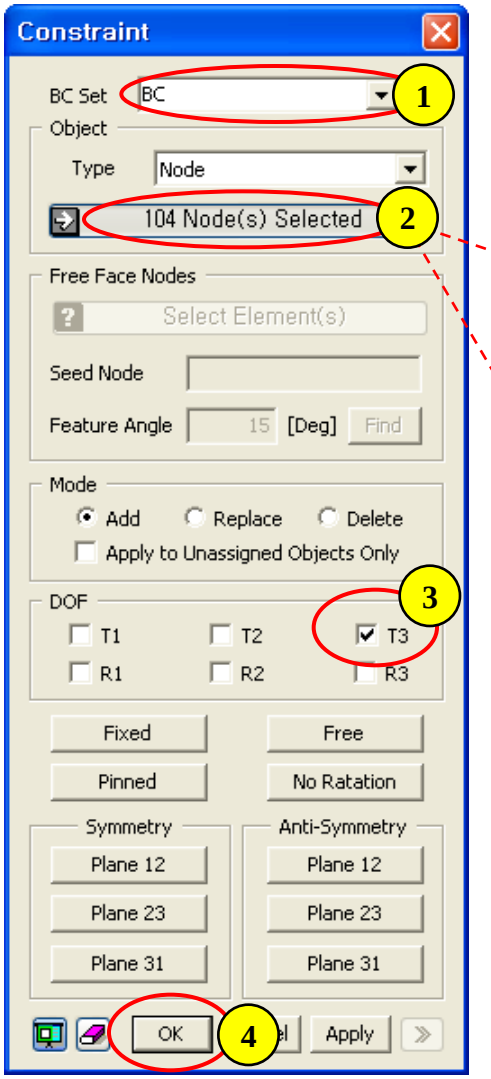




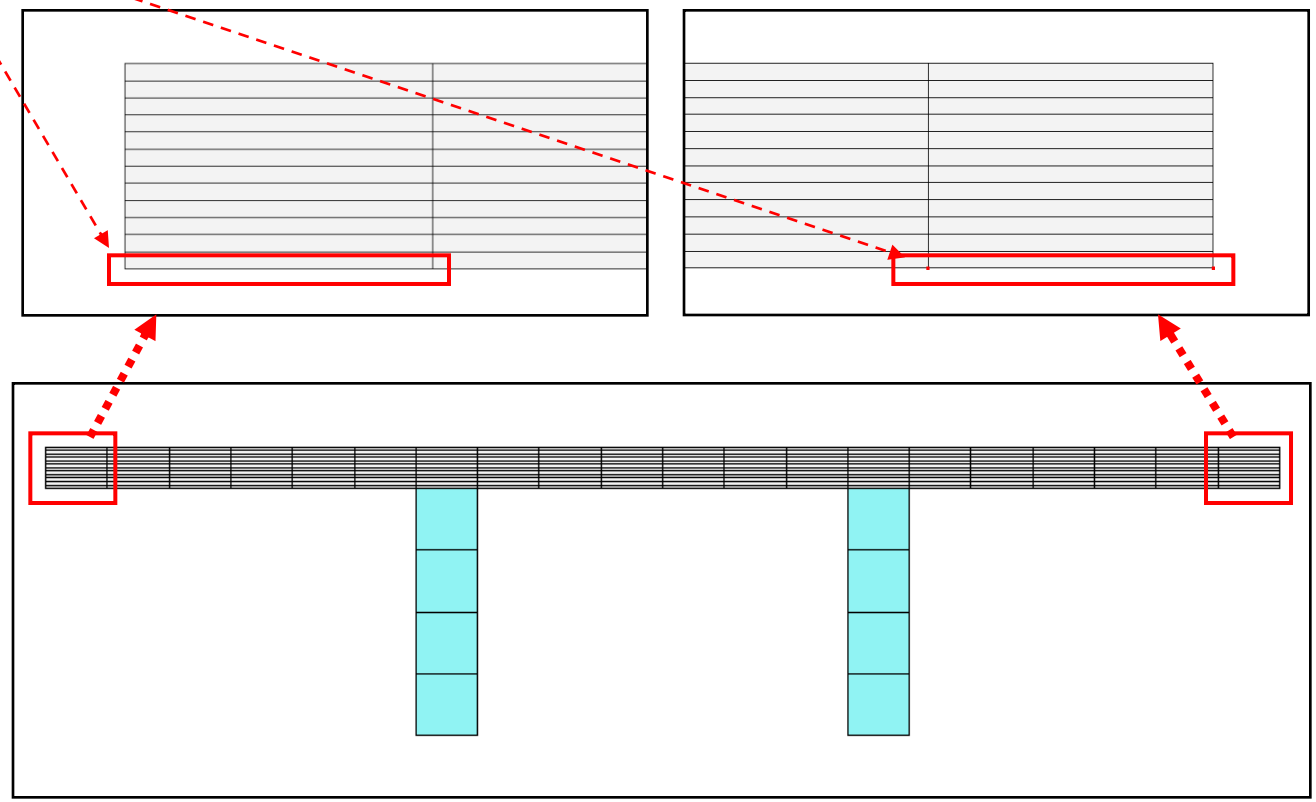
1. *Analysis > BC > Constraint …*
2. *BC Set : BC*
3. *Click "Right View"*
4. *Select 104 Nodes (See Figure)*
5. *Click* **[Pinned]** *Button*
6. *Click* **[Apply]** *Button*



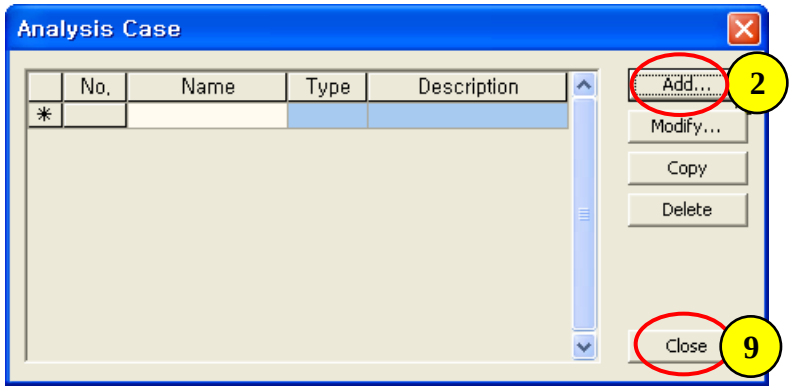
Step 16.

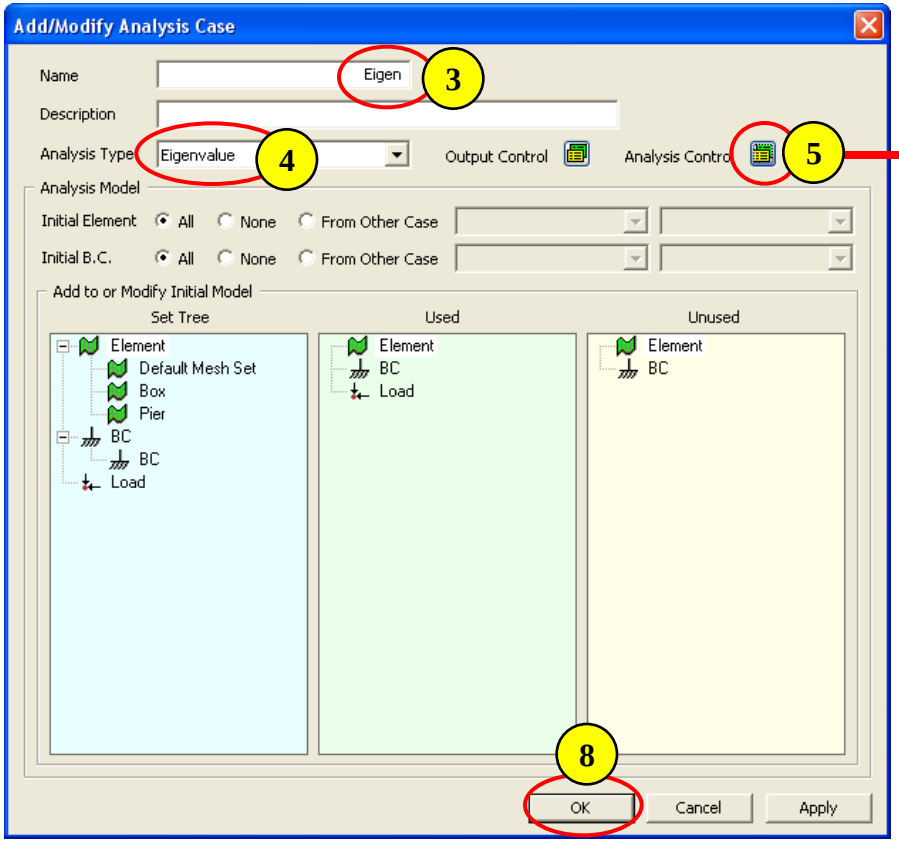


1. *BC Set : BC*
2. *Select 104 Nodes (See Figure)*
3. *Check on "T3"*
4. *Click* **[OK]** *Button*

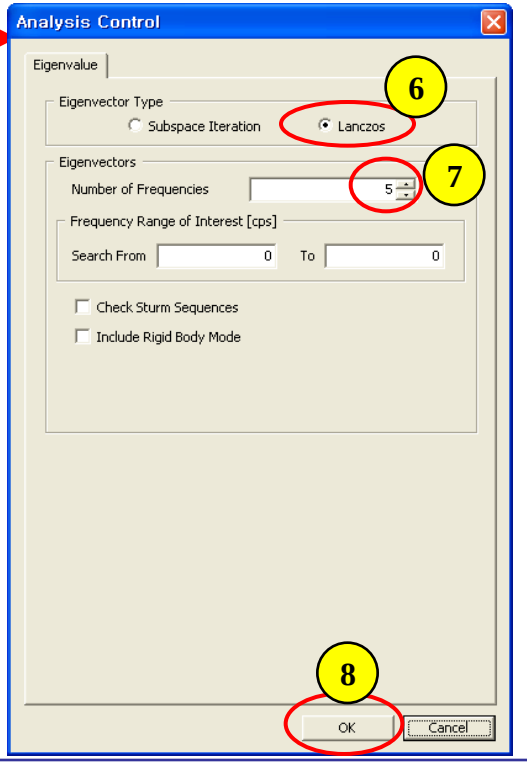


Step 17.

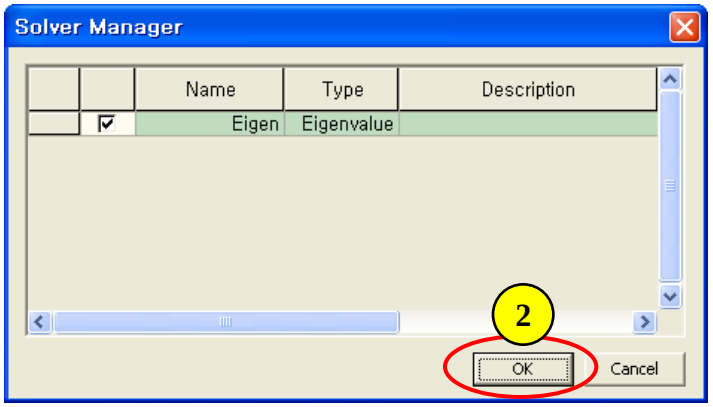


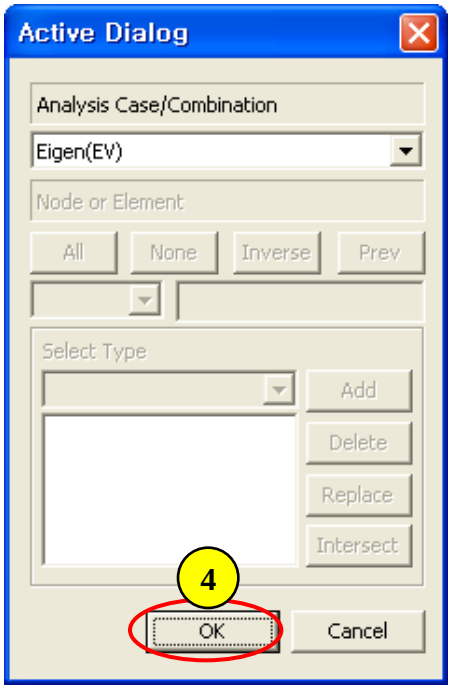


1. *Analysis > Analysis Case …*
2. *Click* **[Add]** *Button*
3. *Name : Eigen*
4. *Analysis Type : Eigenvalue*
5. *Click*  *Button (Analysis Control)*
6. *Eigenvector Type : Lanczos*
7. *Number of Frequencies (5)*
8. *Click* **[OK]** *Button*
9. *Click* **[Close]** *Button*



Step 18.





1. *Analysis > Solve …*
2. *Click* **[OK]** *Button*
3. *Post > Vibration Frequency / Period …*
4. *Click* **[OK]** *Button*
5. *Check Period of Mode 1 and 2*

EIGENVALUE ANALYSIS

Mode No	Frequency w (rad/sec)	Frequency f (cycle/sec)	Period T (sec)	Tolerance
1	27.027153	4.301505	0.232477	0.000000
2	32.459450	5.166082	0.193570	0.000000
3	65.532776	10.429865	0.095879	0.000000
4	119.039970	18.945799	0.052782	0.000000
5	147.761108	23.516911	0.042523	0.000000

MODAL PARTICIPATION MASSES(%) PRINTOUT

Mode No	TRAN-X		TRAN-Y		TRAN-Z		ROTN-X		ROTN-Y		ROTN-Z	
	MASS(%)	SUM(%)	MASS(%)	SUM(%)	MASS(%)	SUM(%)	MASS(%)	SUM(%)	MASS(%)	SUM(%)	MASS(%)	SUM(%)
1	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00
2	86.16	86.16	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00
3	0.00	86.16	88.73	88.73	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00
4	0.64	86.80	0.00	88.73	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00
5	0.00	86.80	0.00	88.73	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00

Mode No	TRAN-X		TRAN-Y		TRAN-Z		ROTN-X		ROTN-Y		ROTN-Z	
	MASS	SUM	MASS	SUM	MASS	SUM	MASS	SUM	MASS	SUM	MASS	SUM
1	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00
2	218004.2	218004.2	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00
3	0.00	0.00	224520.6	224520.6	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00
4	1627.90	219632.1	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00	0.00
5	0.00	0.00	0.00	0.00	3.07	3.07	0.00	0.00	0.00	0.00	0.00	0.00

MODAL PARTICIPATION FACTOR PRINTOUT

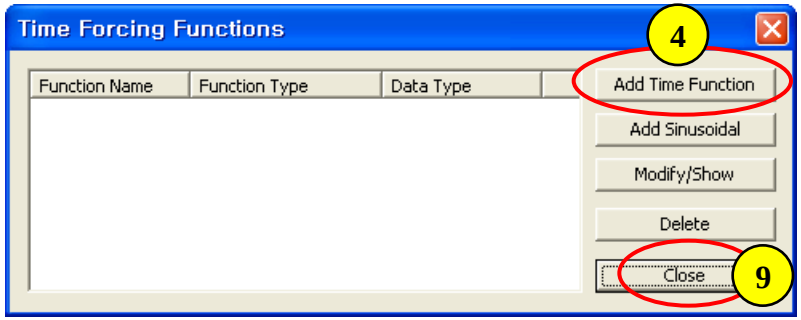
Mode No	TRAN-X	TRAN-Y	TRAN-Z	ROTN-X	ROTN-Y	ROTN-Z
	Value	Value	Value	Value	Value	Value
	0.00	-0.00	0.00	0.00	0.00	0.00
	466.91	0.00	0.00	0.00	0.00	0.00
	-0.00	473.84	0.00	0.00	0.00	0.00
	-40.35	0.00	0.00	0.00	0.00	0.00
	0.00	0.00	1.75	0.00	0.00	0.00

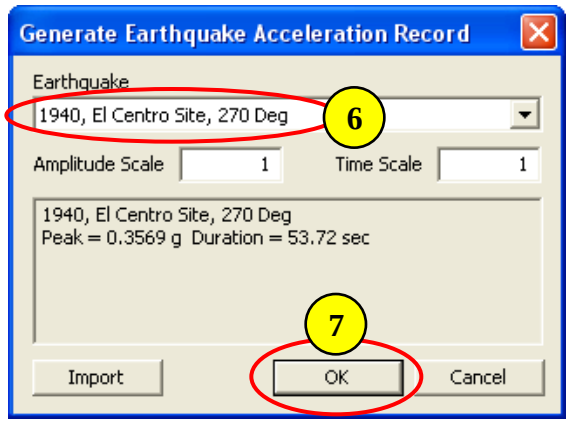
MODAL DIRECTION FACTOR PRINTOUT

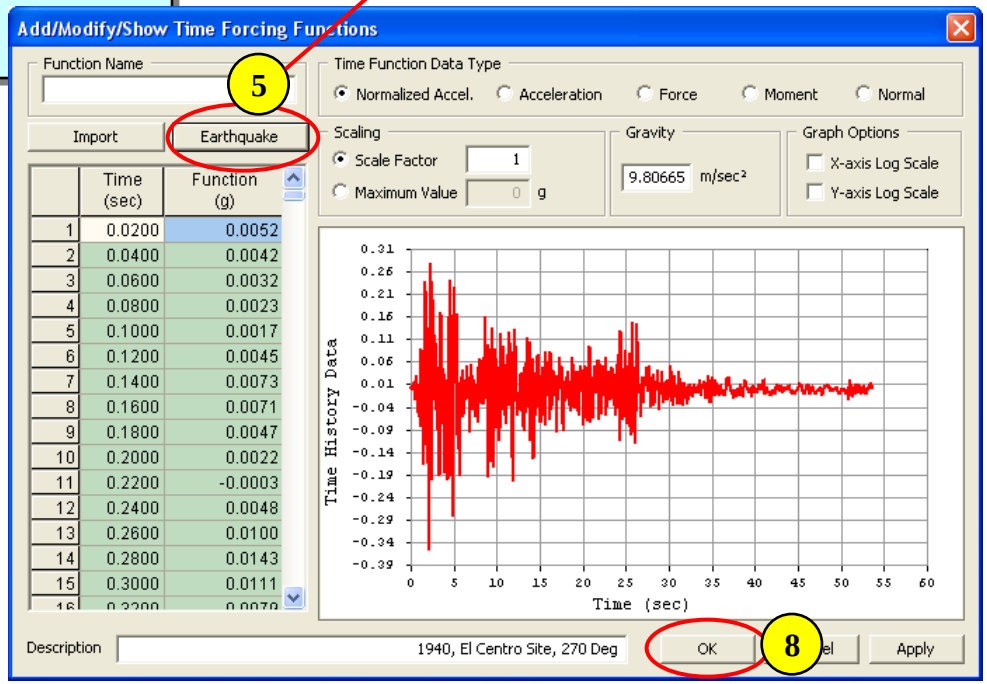
Mode No	TRAN-X	TRAN-Y	TRAN-Z	ROTN-X	ROTN-Y	ROTN-Z
	Value	Value	Value	Value	Value	Value
1	98.13	1.75	0.12	0.00	0.00	0.00
2	99.58	0.01	0.41	0.00	0.00	0.00
3	0.00	98.95	1.05	0.00	0.00	0.00
4	95.60	3.85	0.54	0.00	0.00	0.00
5	98.16	0.01	1.83	0.00	0.00	0.00

Step 19.

1. *Select "Analysis" Toolbar*
2. *Click "Pre-Mode"*
3. *Analysis > Time History Analysis > Time Forcing Functions …*
4. *Click* [Add Time Function] *Button*
5. *Click* [Earthquake] *Button*
6. *Select "1940, El Centro Site, 270 Deg" for Earthquake*
7. *Click* [OK] *Button*
8. *Click* [OK] *Button*
9. *Click* [Close] *Button*

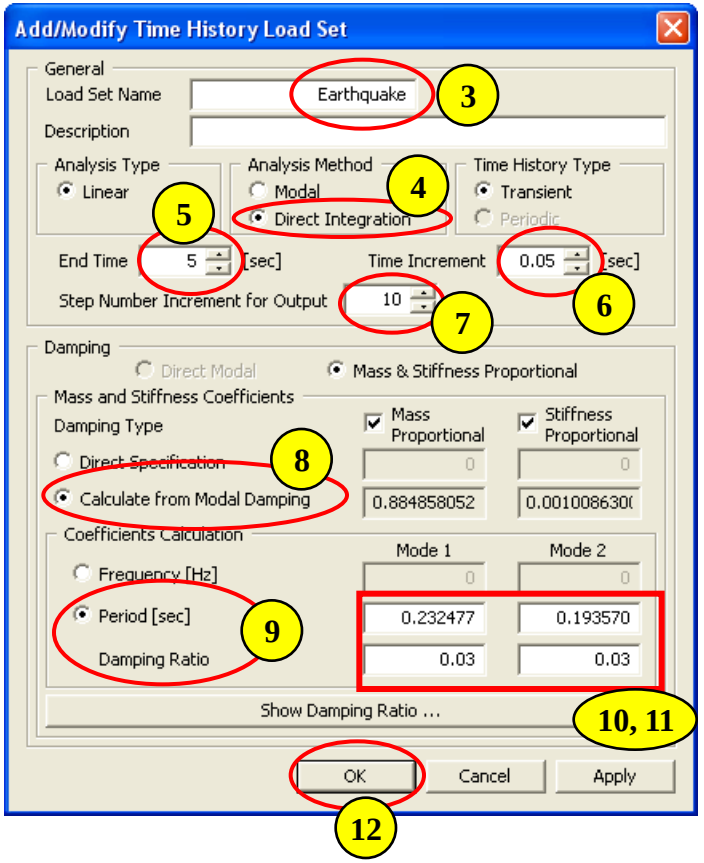


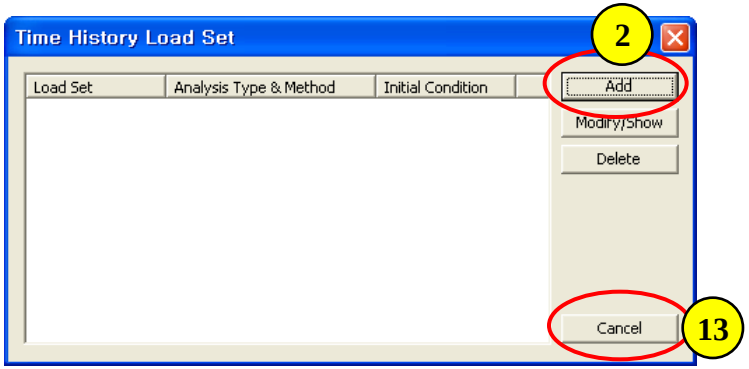




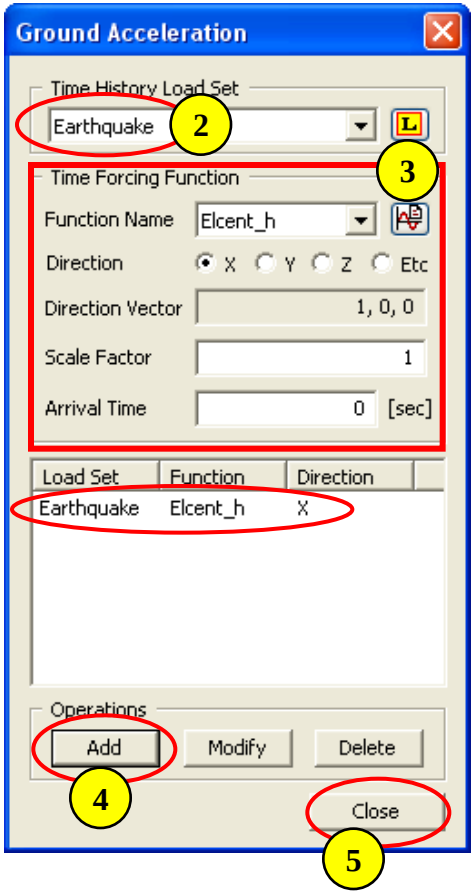
Step 20.

1. *Analysis > Time History Analysis > Time History Load Set …*
2. *Click* [Add] *Button*
3. *Load Set Name : Earthquake*
4. *Select "Direct Integration" for Analysis Method*
5. *End Time : 5 sec*
6. *Time Increment : 0.05 sec*
7. *Step Number Increment for Output : 10*
8. *Check on "Calculate from Modal Damping"*
9. *Check on "Period [sec]" for Coefficients Calculation"*
10. *Input Period of Mode 1 and 2 from Eigenvalue Result (See Step 18.)*
11. *Damping Ratio : 0.03*
12. *Click* [OK] *Button*
13. *Click* [Close] *Button*



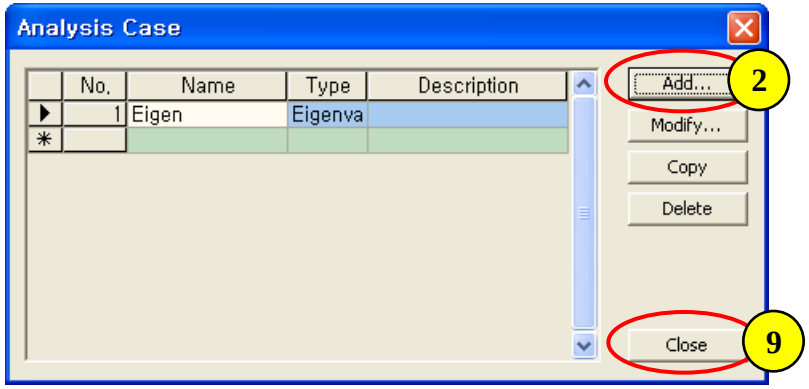


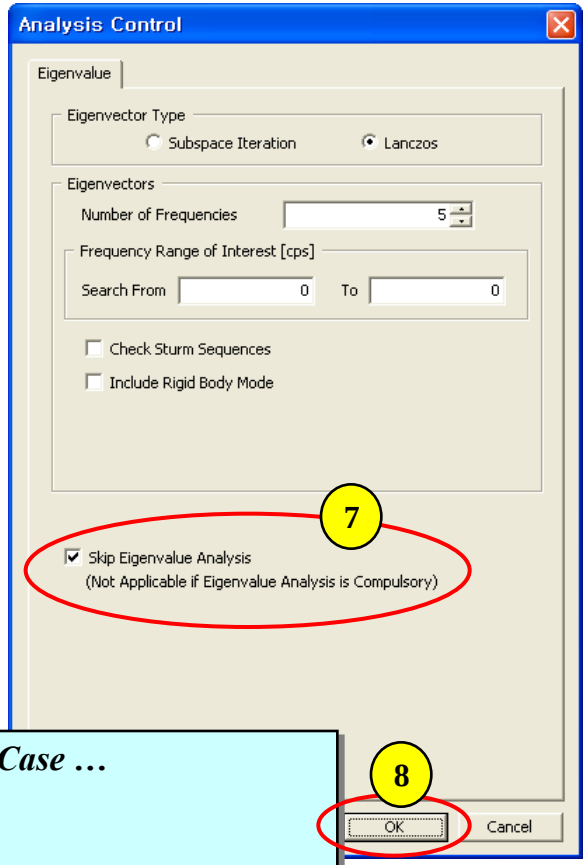
Step 21.

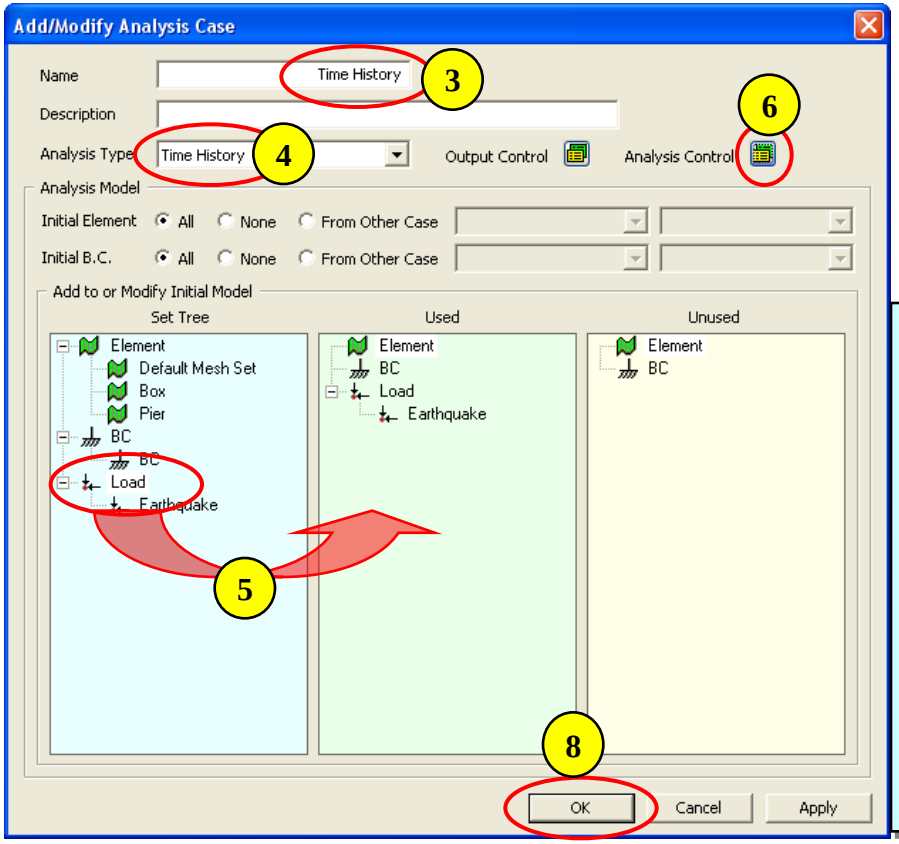


1. *Analysis > Time History Analysis > Ground Acceleration …*
2. *Select "Earthquake" for Time History Load Set*
3. *Select "Elcent_h" for Function Name*
4. *Click* **[Add]** *Button*
5. *Click* **[Close]** *Button*

Step 22.

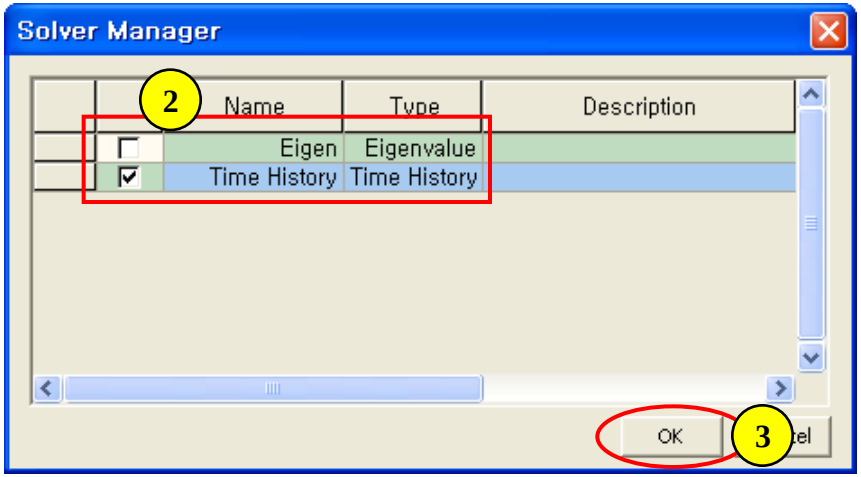






1. *Analysis > Analysis Case …*
2. *Click "Add" Button*
3. *Name : Time History*
4. *Analysis Type : Time History*
5. *Drag & Drop "Load" to "Used" Window*
6. *Click* **[...]** *Button of Analysis Control*
7. *Check on "Skip Eigenvalue Analysis"*
8. *Click* **[OK]** *Button*
9. *Click* **[Close]** *Button*

Step 23.



1. *Analysis > Solve …*
2. *Check off "Eigen"*
3. *Click* **[OK]** *Button*

Step 24.

1. *Post > Time History Analysis > Time History Plot …*
2. *Check off "Sensitive"*
3. *Select "0.5" for Time Step*
4. *Select "DXYZ" for Result*
5. *Click* **[Apply]** *Button*
6. *Repeat Step3~6 for other Time Step*