

## Overview

### ▪ 2-D Linear Static Analysis

#### ▪ Model

- Unit : tonf, m
- Isotropic Elastic Material
- Beam Elements

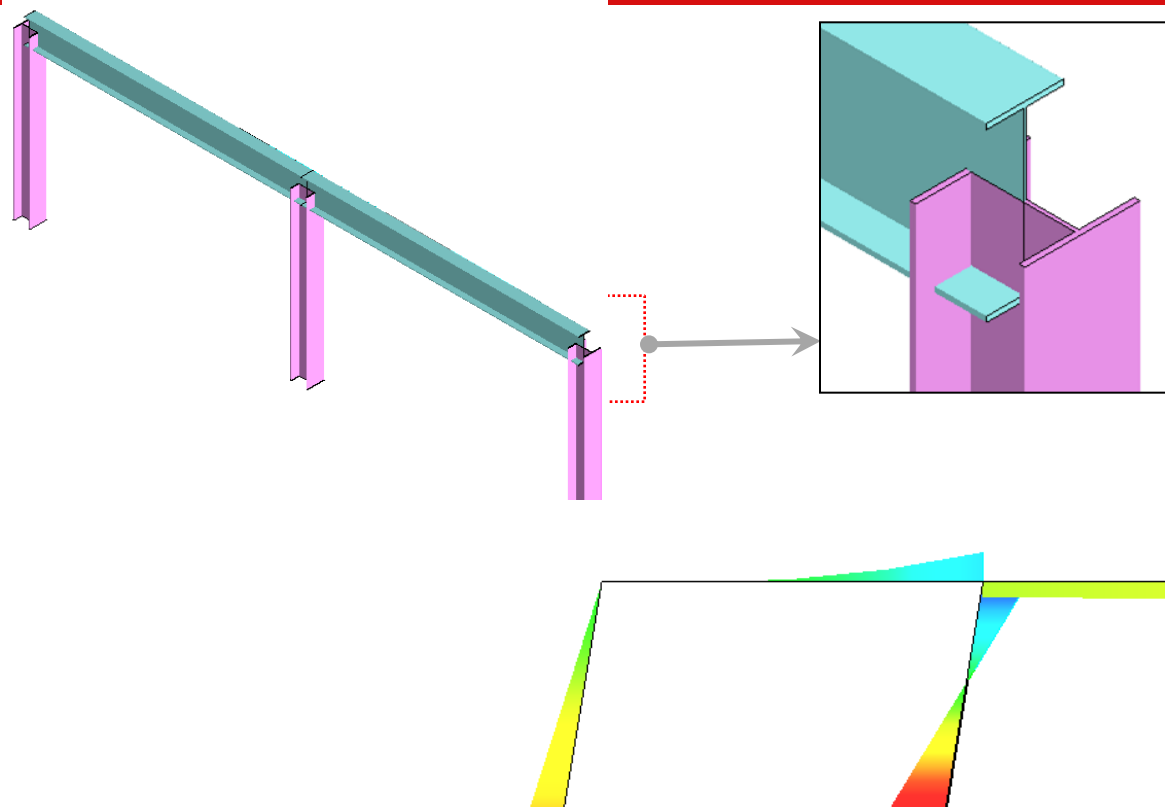
#### ▪ Load & Boundary Condition

- Beam Load
- Nodal Force
- Constraint
- Beam End Release

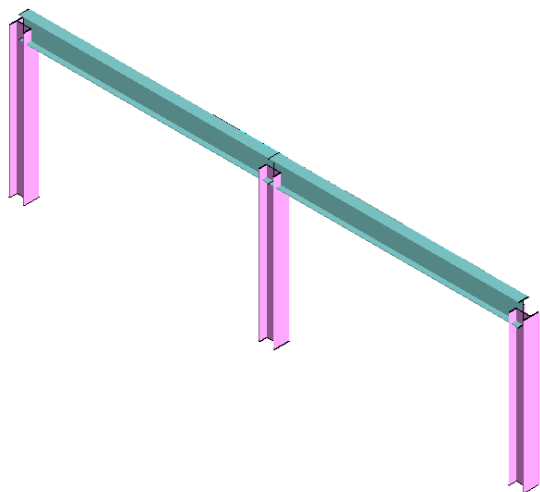
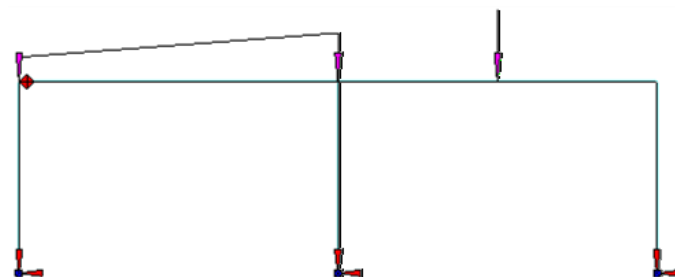
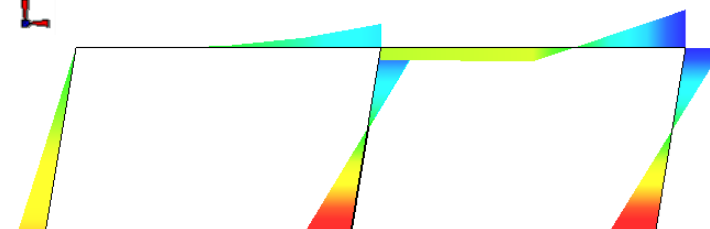
#### ▪ Result Evaluation

- Deformation
- Result Combination
- Moment Diagram

# Plane Frame



# Modeling & Analysis Point

**2D Modeling****Constraint & Load****Result Plot**

## About this Training...

This model is a 2D rahmen type model using Beam element, which is a 1D element. Columns and beam sections of rahmen are defined by H-sections and analysis is performed by defining nodal loads and distributed loads of elements. Check the result of the bending moment and save it with the Animation function.


## Learning Target


How to model 1D Mesh using *Geometry> Line, Auto Mesh> Edge* function

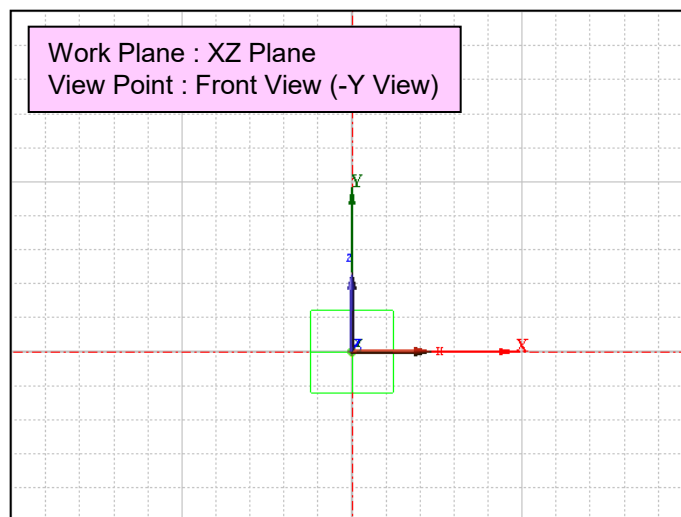
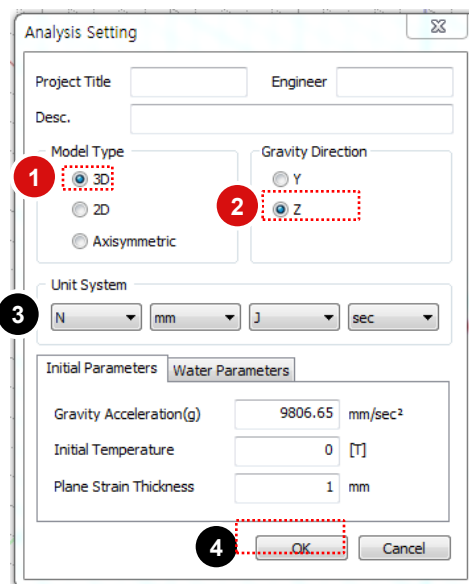
1D Property Definition, Element Beam Load Definition

How to Save and View Contour Results with the Animation Function





## Procedure

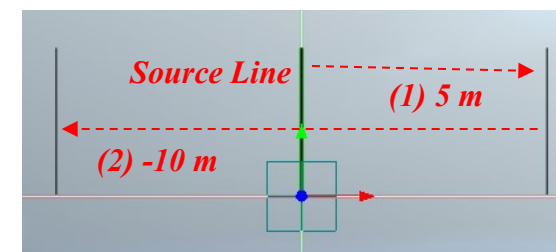
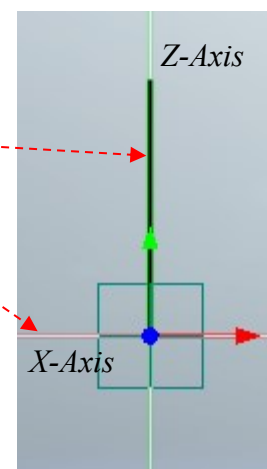
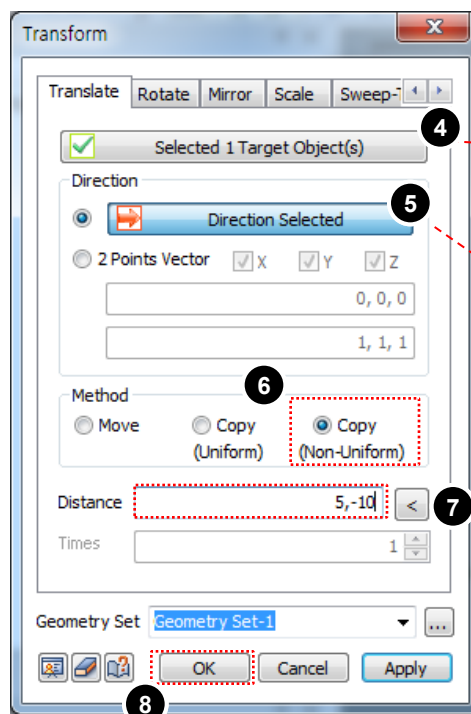
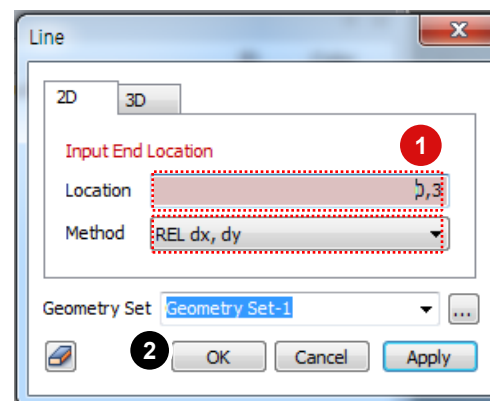
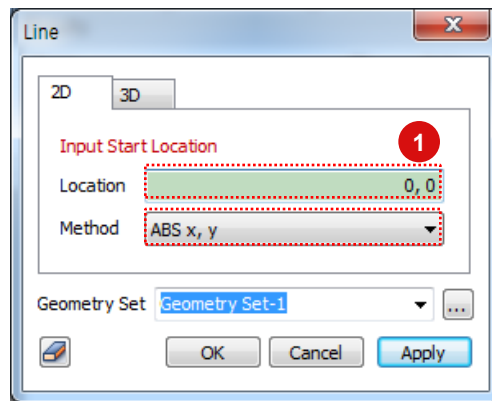
- 1 Model Type : **[3D]**
- 2 Gravity Direction : **[Z]**
- 3 Force(Mass) : **[N]**, Length : **[m]**
- 4 Click **[OK]** Button
- 5 Click Show/Hide Grid 

 Work Plane and View Point are automatically changed.



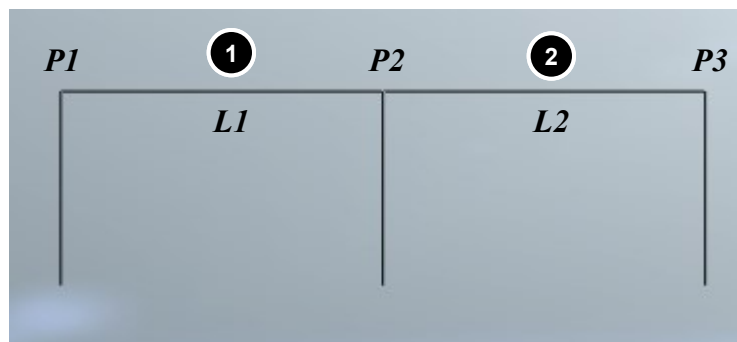
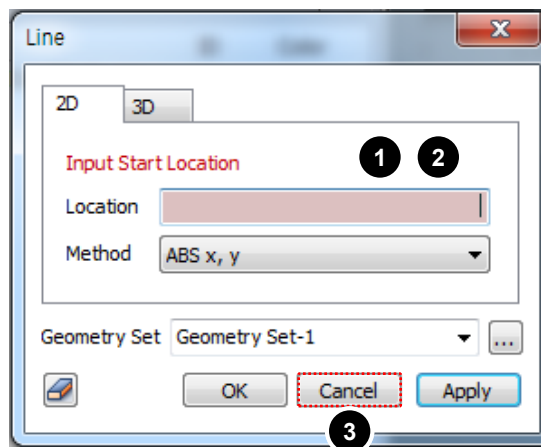
**Procedure**

- 1 Location : **(0,0) , <0, 3>** 
  - 2 Click **[OK]** Button
  - 3 Geometry > Transform > **[Translate...]**
  - 4 Select **"Line"** in Work Window
  - 5 Direction : **"X-Axis"**
  - 6 Method : **Copy (Non-Uniform)**
  - 7 Distance : **5, -10**
  - 8 Click **[OK]** Button
  - 9 Click **[Zoom All]**
  - 10 Hide Datum Axis/Plane 
  - 11 Hide WCS 
-  ( ): "ABS x, y", < >: "REL dx, dy"  
(0) same as (0, 0)




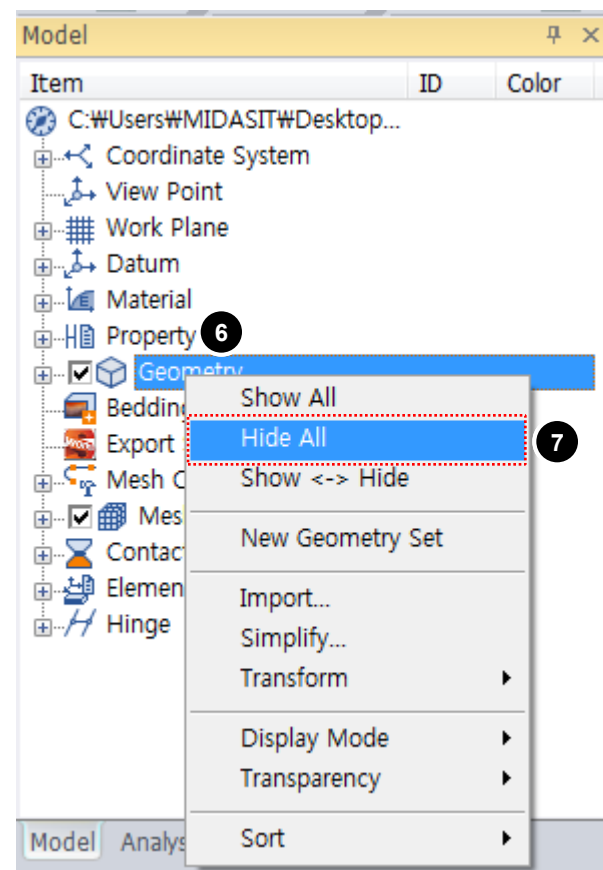
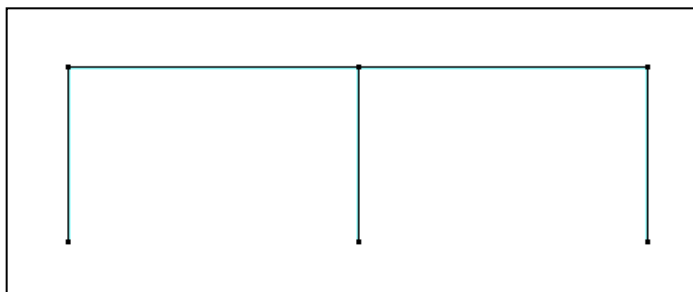
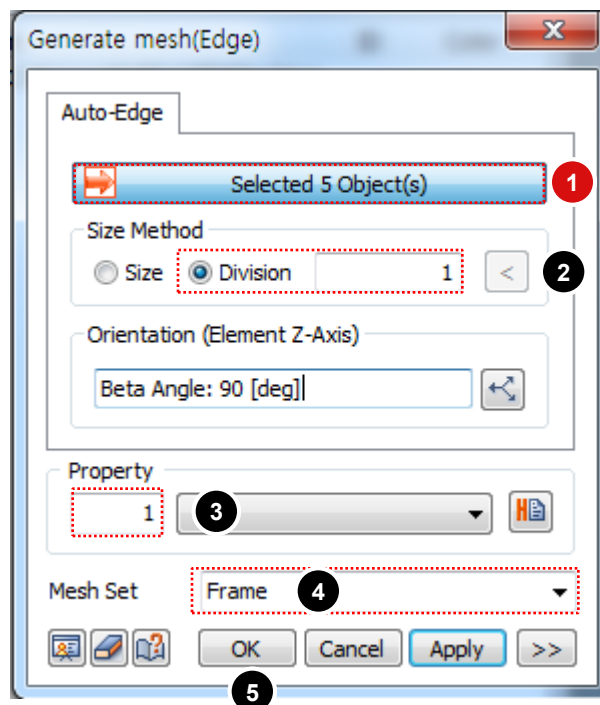
**Procedure**

- 1 Draw Line “**L1**” by Clicking P1 & P2
- 2 Draw Line “**L2**” by Clicking P2 & P3
- 3 Click [**Cancel**] Button



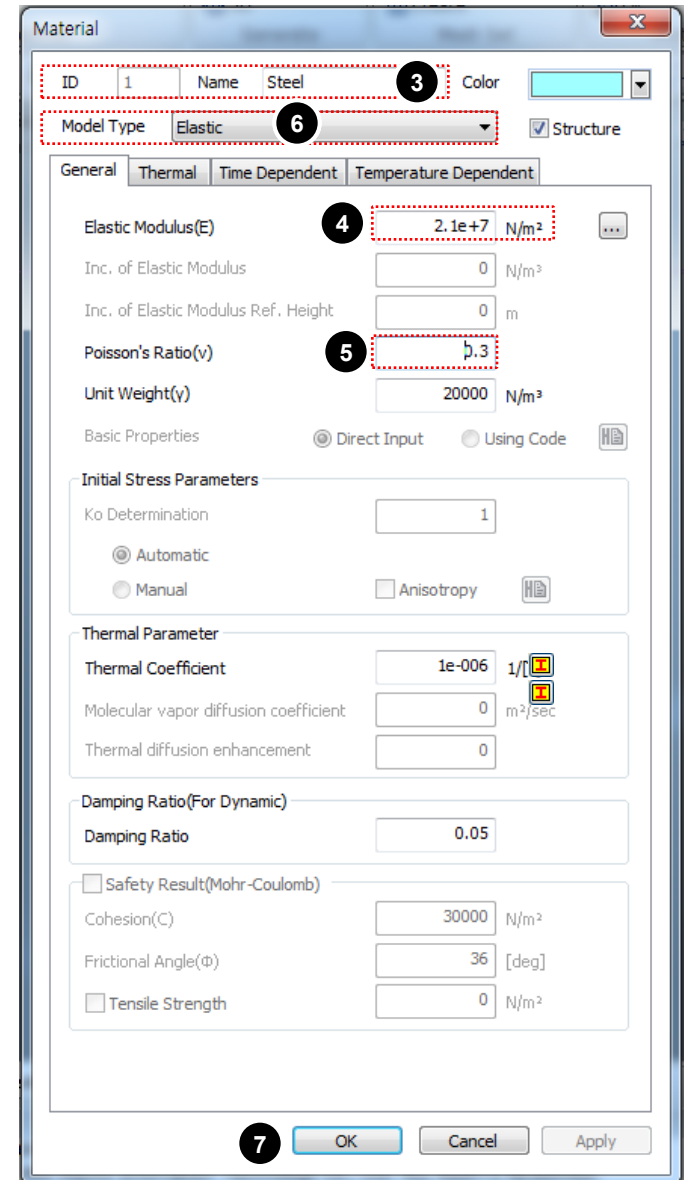
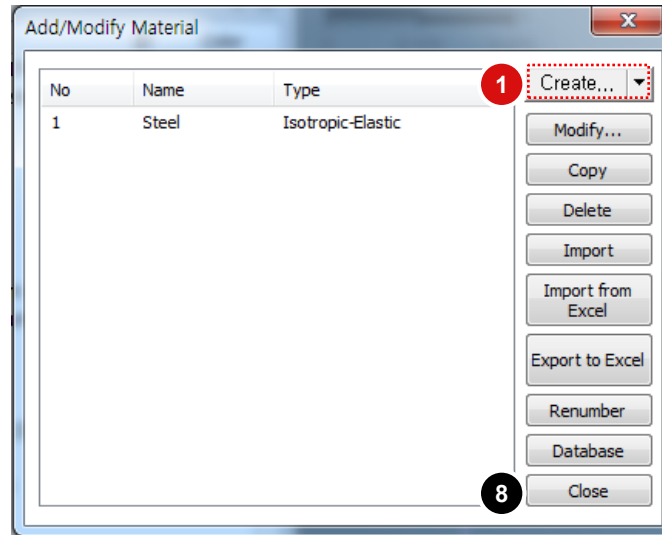
## Procedure

- 1 Select [  ] Displayed
- 2 Size Method : “ **Division** ”  
Number of Divisions : “ **1** ”
- 3 Property : “ **1** ”
- 4 Mesh Set : [ **Frame** ]
- 5 Click [ **OK** ] Button
- 6 Model Tree : [ **Geometry** ]
- 7 Click Right Mouse Button and Select [ **Hide All** ]



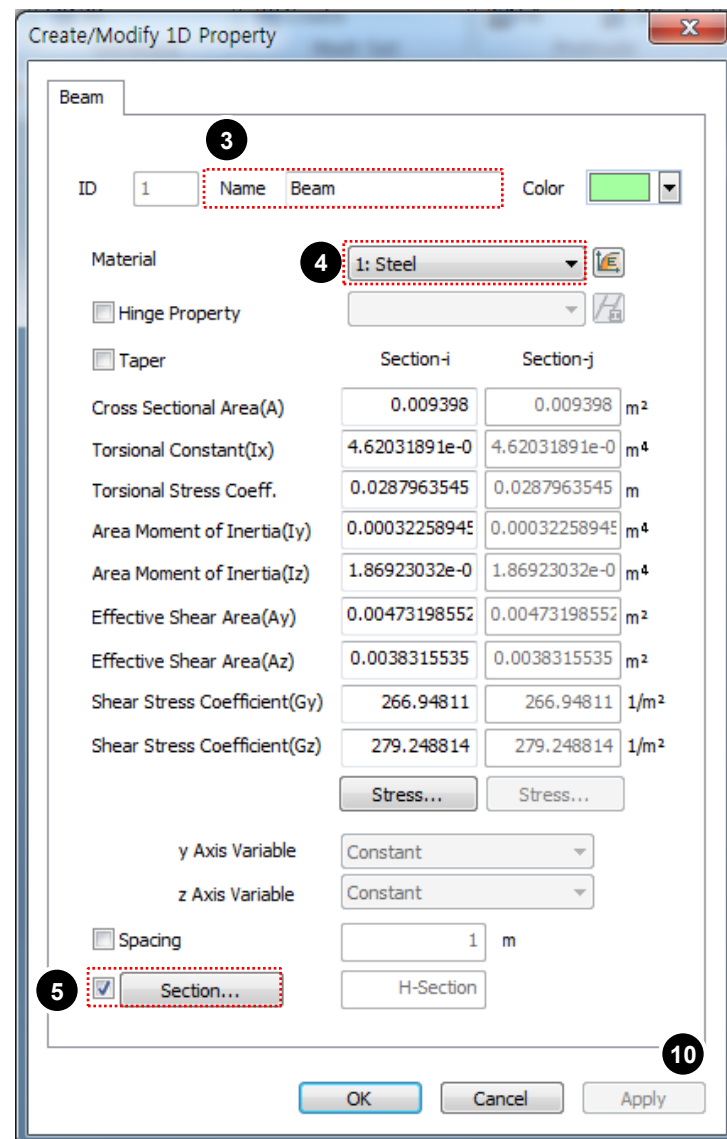
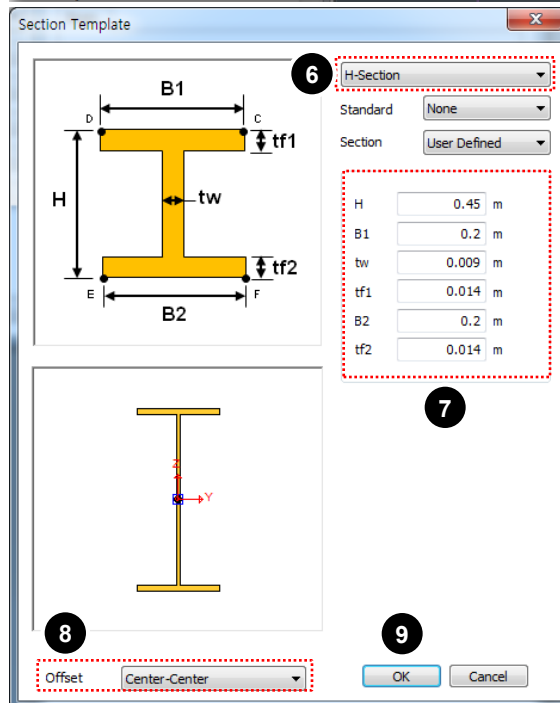
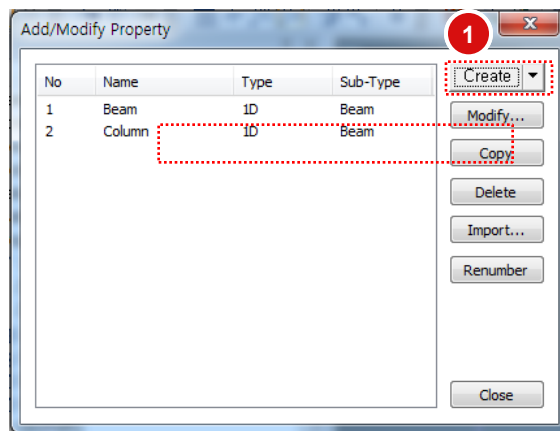
**Procedure**

- 1 Click **[Create...]** Button
- 2 Select **[Isotropic]**
- 3 ID : **"1"** , Name : **"Steel"**
- 4 Elastic Modulus : **"2.1e+7"** N/m<sup>2</sup>
- 5 Poisson's Ratio : **"0.3"**
- 6 Model Type : **[Elastic]**
- 7 Click **[OK]** Button
- 8 Click **[Close]** Button



**Procedure**

- 1 Create [1D]
- 2 Select [Beam] tab
- 3 ID : "1", Name : "Beam"
- 4 Material : "1: Steel"
- 5 Click [Section Template...] Button
- 6 Section : "H-Section"
- 7 H "0.45", B1 "0.2", tw "0.009",  
tf1 "0.014", B2 "0.2", tf2 "0.014"
- 8 Offset : [Center-Center]
- 9 Click [OK] Button
- 10 Click [Apply] Button





**Procedure**

- 1 ID : "2" , Name : "Column"
- 2 Material : "1: Steel"
- 3 Click [Section Template...] Button
- 4 Section : "H-Section"
- 5 H "0.3", B1 "0.3", tw "0.01",  
tf1 "0.015", B2 "0.3", tf2 "0.015"
- 6 Offset : [Center-Center]
- 7 Click [OK] Button
- 8 Click [OK] Button
- 9 Click [Close] Button

Create/Modify 1D Property

Beam

1 ID 2 Name Column Color  

Material 2 1: Steel  

☐ Hinge Property  

☐ Taper  

	Section-i	Section-j
Cross Sectional Area(A)	0.0117	0.0117 m <sup>2</sup>
Torsional Constant(Ix)	7.57305502e-0	7.57305502e-0 m <sup>4</sup>
Torsional Stress Coeff.	0.0270780987	0.0270780987 m
Area Moment of Inertia(Iy)	0.0001993275	0.0001993275 m <sup>4</sup>
Area Moment of Inertia(Iz)	6.75225e-005	6.75225e-005 m <sup>4</sup>
Effective Shear Area(Ay)	0.00756353098	0.00756353098 m <sup>2</sup>
Effective Shear Area(Az)	0.00267420725	0.00267420725 m <sup>2</sup>
Shear Stress Coefficient(Gy)	166.426006	166.426006 1/m <sup>2</sup>
Shear Stress Coefficient(Gz)	367.42296	367.42296 1/m <sup>2</sup>

Stress... Stress...

y Axis Variable Constant  

z Axis Variable Constant  

☐ Spacing 1 m

3 ☒ Section... H-Section

8 OK Cancel Apply

Section Template

4 H-Section

Standard None

Section User Defined

H 0.3 m

B1 0.3 m

tw 0.01 m

tf1 0.015 m

B2 0.3 m

tf2 0.015 m

6 Offset Center-Center

7 OK Cancel

Add/Modify Property

No	Name	Type	Sub-Type
1	Beam	1D	Beam
2	Column	1D	Beam

Create  

Modify...  

Copy  

Delete  

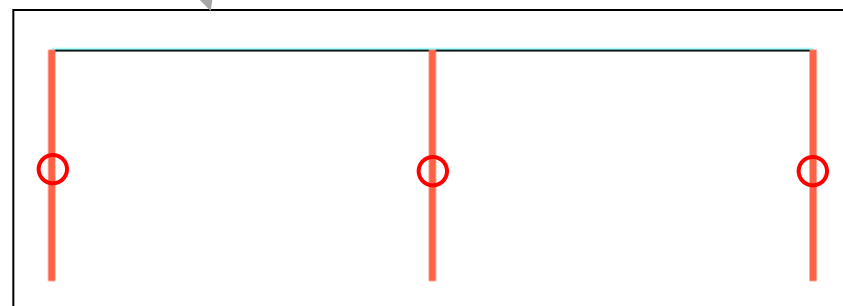
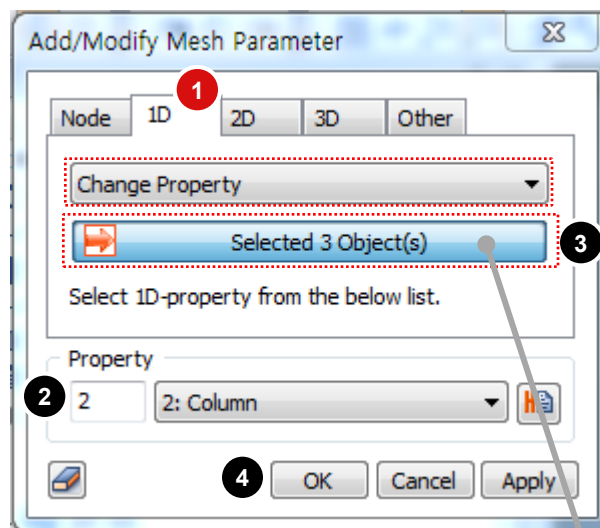
Import...  

Renumber  

9 Close

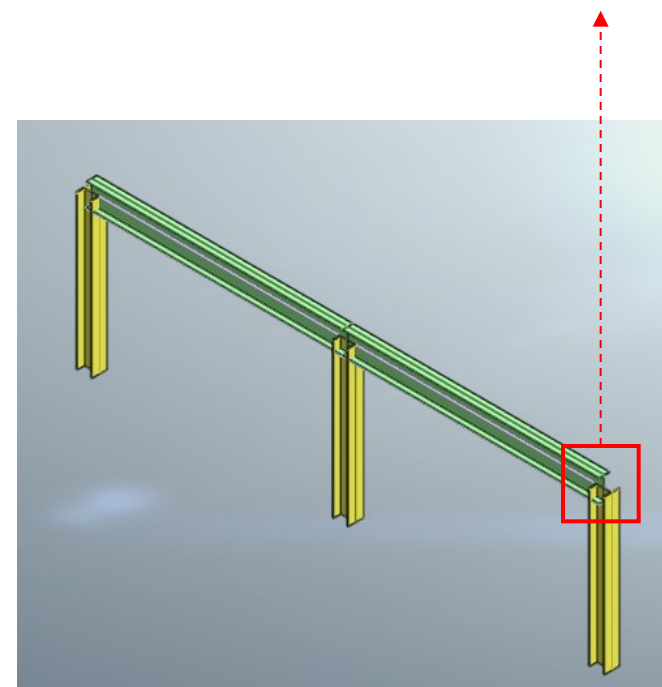
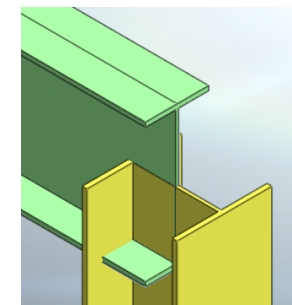
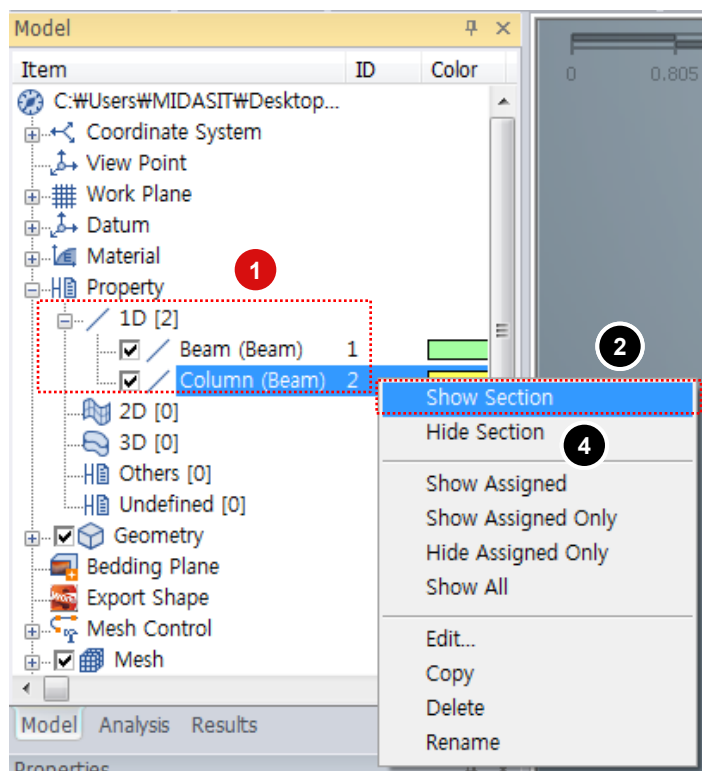
**Procedure**

- 1 Select : **"Change Property"** , **"1D"**
- 2 Property : **"2: Column"**
- 3 Select 3 Vertical Elements Marked by **"○"** (See Figure)
- 4 Click **[OK]** Button



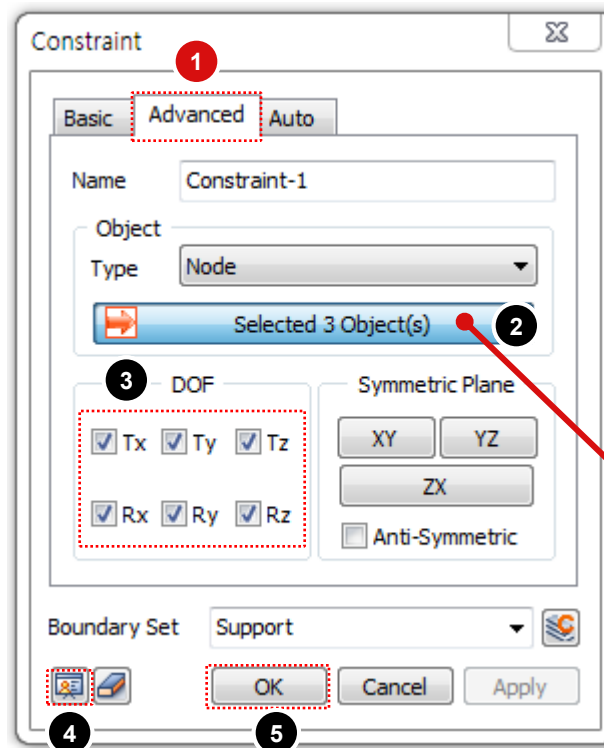
**Procedure**

- 1 Select : **"Beam" & "Column"**
- 2 Click Right Mouse Button and Select :  
**"Show Section"**
- 3 Click **"Isometric 1"**
- 4 Select **"Hide Section"**



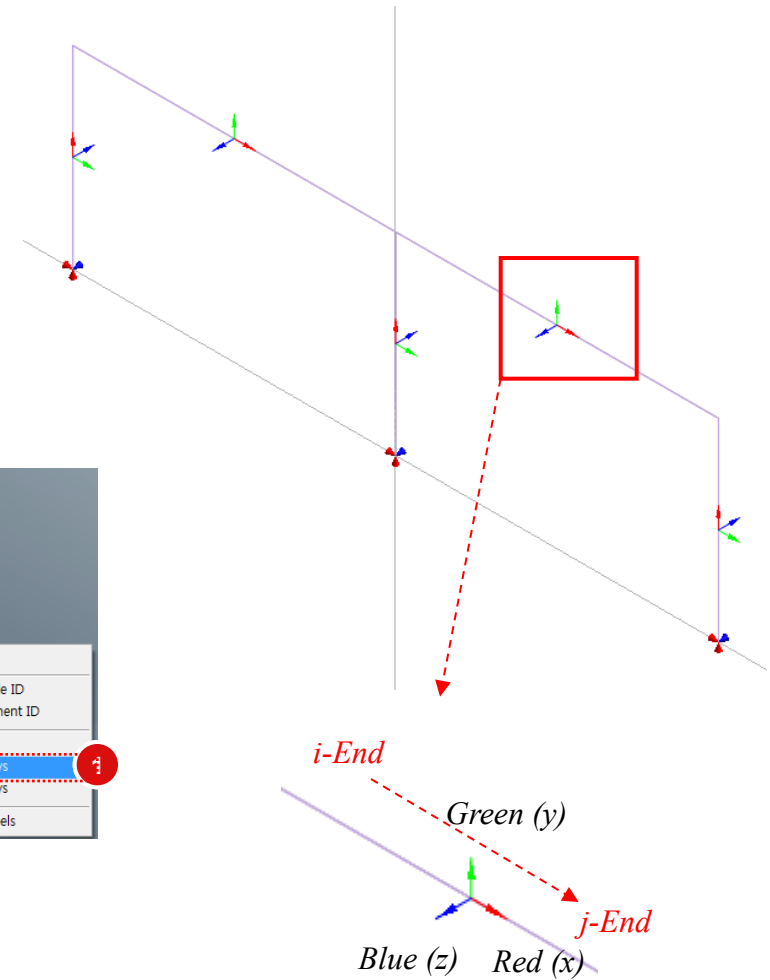
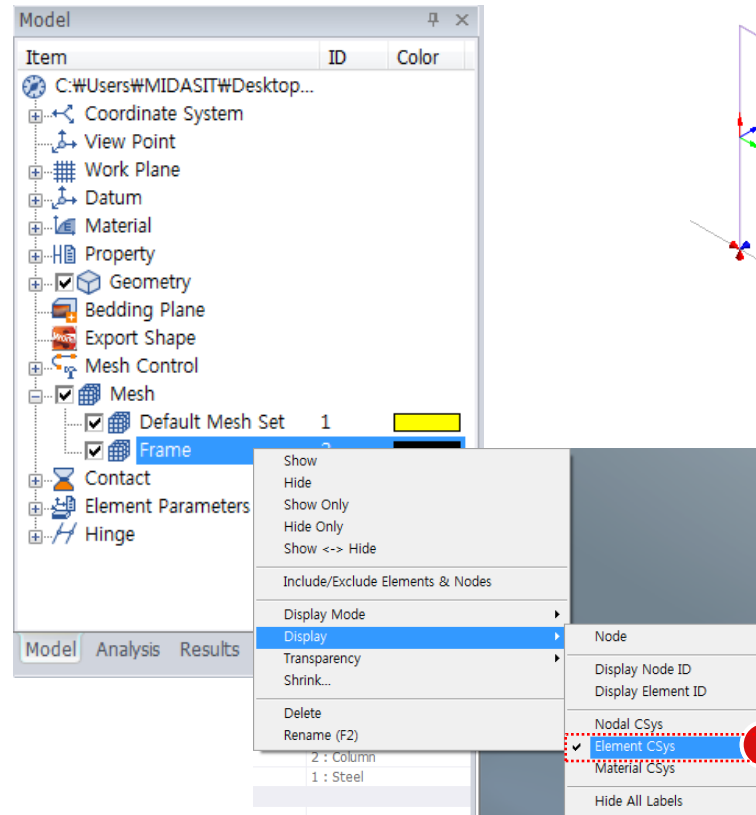
**Procedure**

- 1 [Advanced] Tab
- 2 Select 3 Nodes Marked by  
“O” (See Figure)
- 3 Check on “Tx”, “Tz” & “Ry”
- 4 Boundary Set : [Support]
- 5 Click [Preview] Button
- 6 Click [OK] Button



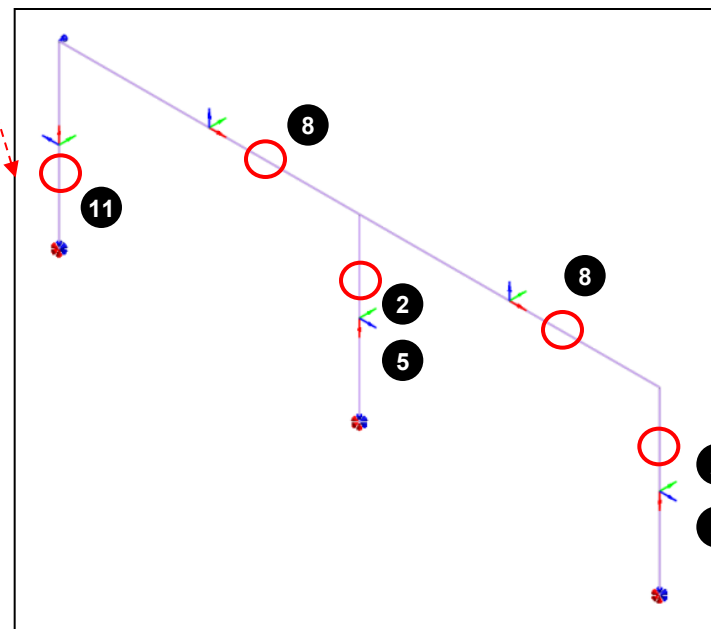
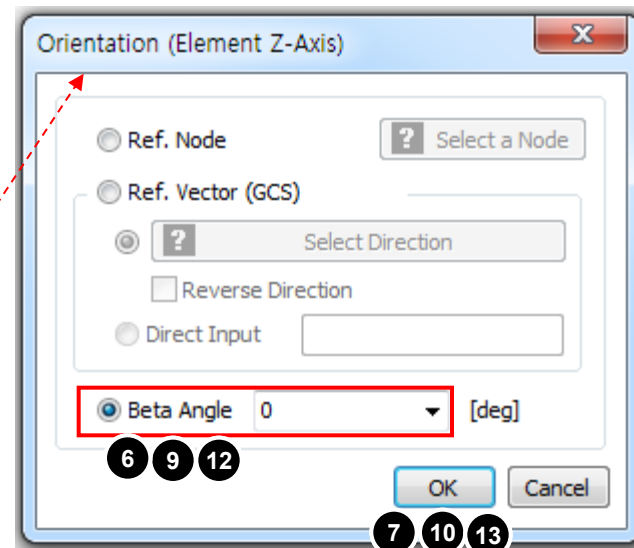
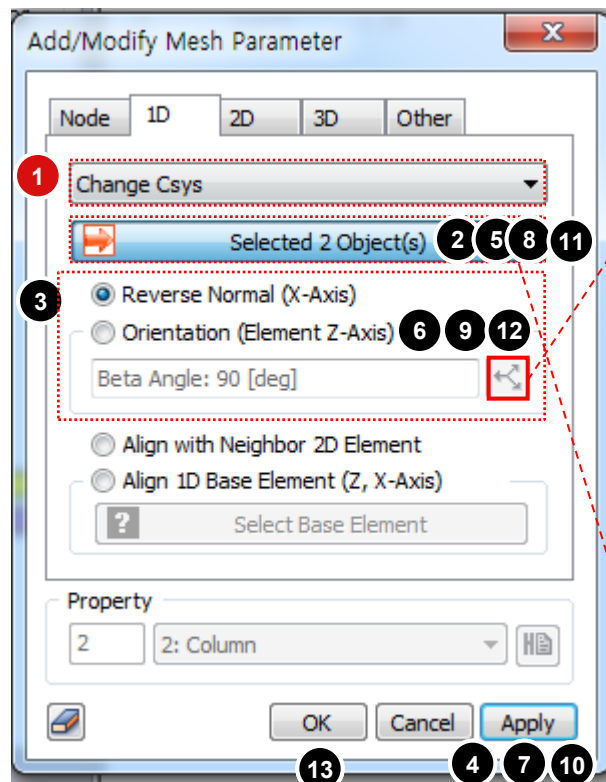
**Procedure**

- 1 Click Right Mouse Button and Select :  
“Display > Element CSys”



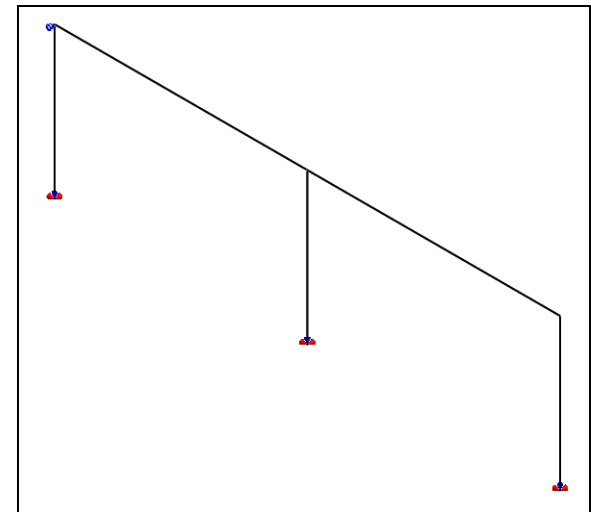
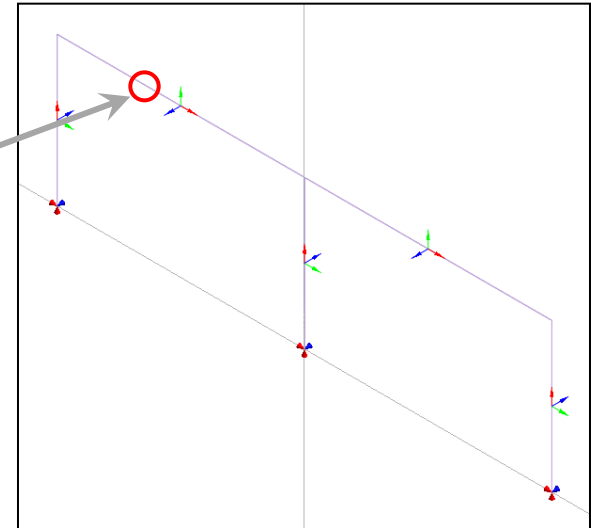
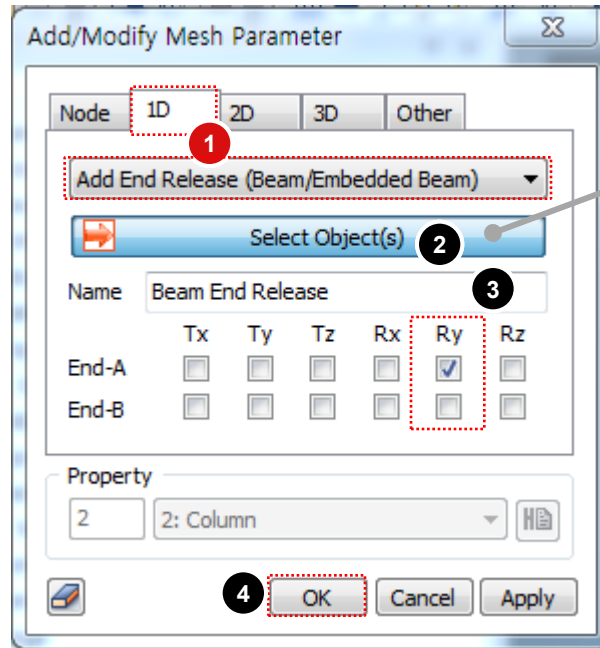
**Procedure**

- 1 Click **[Change Csys]**
- 2 Select 2 Elements Marked by "O" (See Figure)
- 3 Click **[Reverse Normal]**
- 4 Click **[Apply]**
- 5 Select 2 Elements Marked by "O" (See Figure)
- 6 Click **[Orientation]** > Beta Angle:0
- 7 Click **[OK]** > Click **[Apply]**
- 8 Select 2 Elements Marked by "O" (See Figure)
- 9 Click **[Orientation]**> Beta Angle:90
- 10 Click **[OK]** > Click **[Apply]**
- 11 Select 1 Element Marked by "O" (See Figure)
- 12 Click **[Orientation]**>BetaAngle:180
- 13 Click **[OK]** > Click **[OK]**



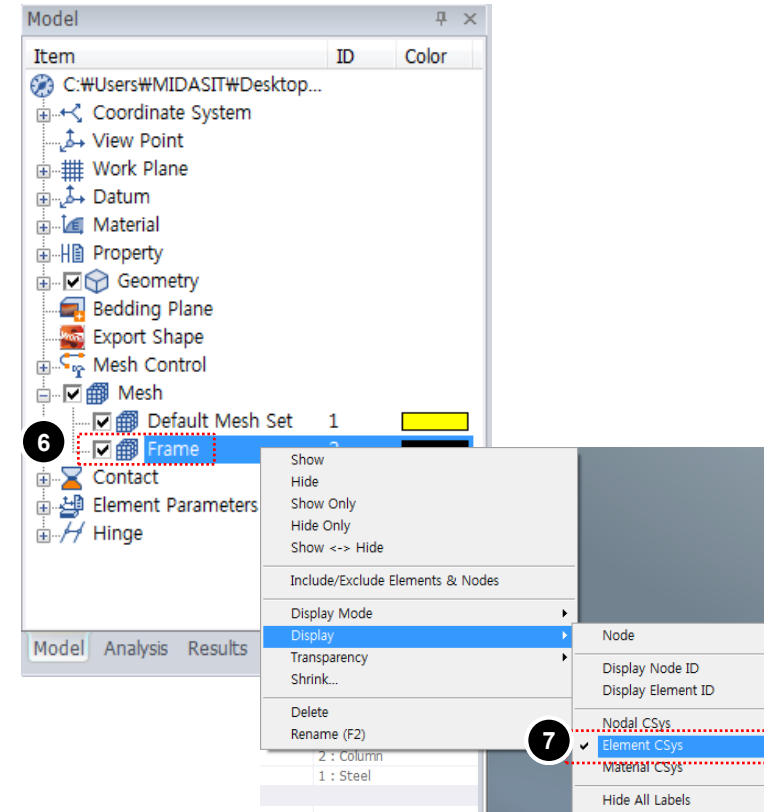
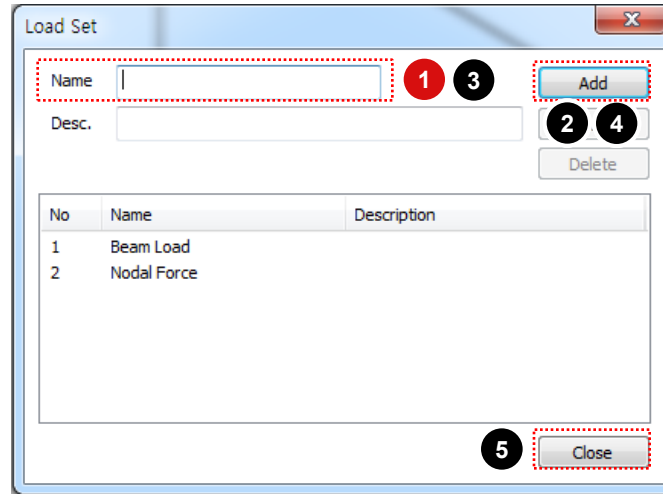
**Procedure**

- 1 [1D] Tab, Select **Add End Release**
- 2 Select 1 Element Marked by  
“O” (See Figure)
- 3 “End-A, Ry ”
- 4 Click [OK] Button



**Procedure**

- 1 Name : **"Beam Load"**
- 2 Click **[Add]** Button
- 3 Name : **"Nodal Force"**
- 4 Click **[Add]** Button
- 5 Click **[Close]** Button
- 6 Model Trees :  
**"Mesh – Frame"**
- 7 Click Right Mouse Button and Select :  
**"Display > Element CSys"**





**Procedure**

- 1 [Element Beam Load]
- 2 Select 1 Element Marked by "O" (See Figure)
- 3 Type : "Force – Distributed"
- 4 Direction : "Global Z"
- 5 Value : Fraction :  
"x1(0), w1(-1), x2(1), w2(-2)"
- 6 Click [Apply] Button
- 7 Select 1 Element Marked by "O" (See Figure)
- 8 Type : "Force – Concentrated"
- 9 Direction : "Global Z"
- 10 Value : Fraction : "x1(0.5), w1(-10)"
- 11 Load Set : [Beam Load]
- 12 Click [OK] Button

Beam Load

Line Beam Load | Element Beam Load 1

Name: Element Beam Load-1

Object: 1D Element

Selected 1 Object(s)

Force ☒ Moment ☐

Distributed ☒ Concentrated ☐

Direction: Global Z

Projection: ☐ Yes ☒ No

Value: Fraction ☒ Length ☐

x1: 0 x2: 1

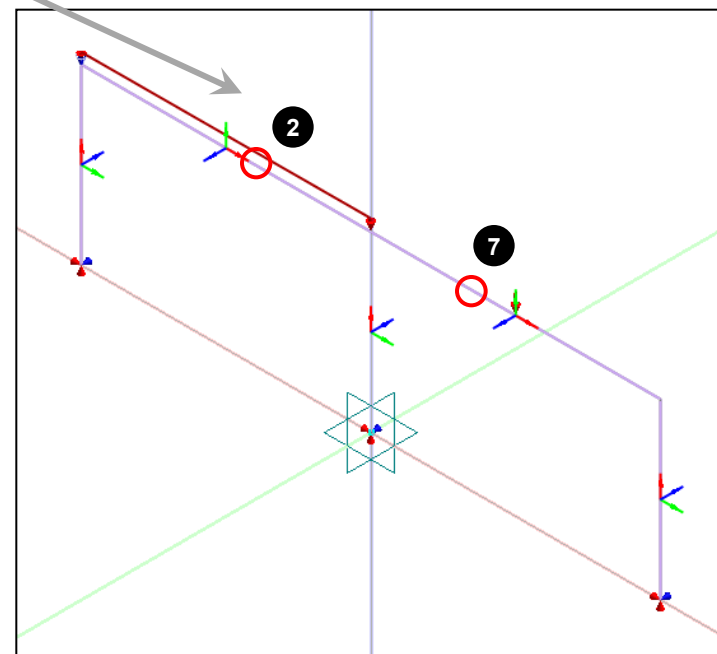
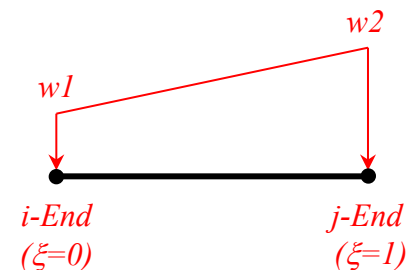
w1: -1 N/m

w2: -2 N/m

Base Function: None

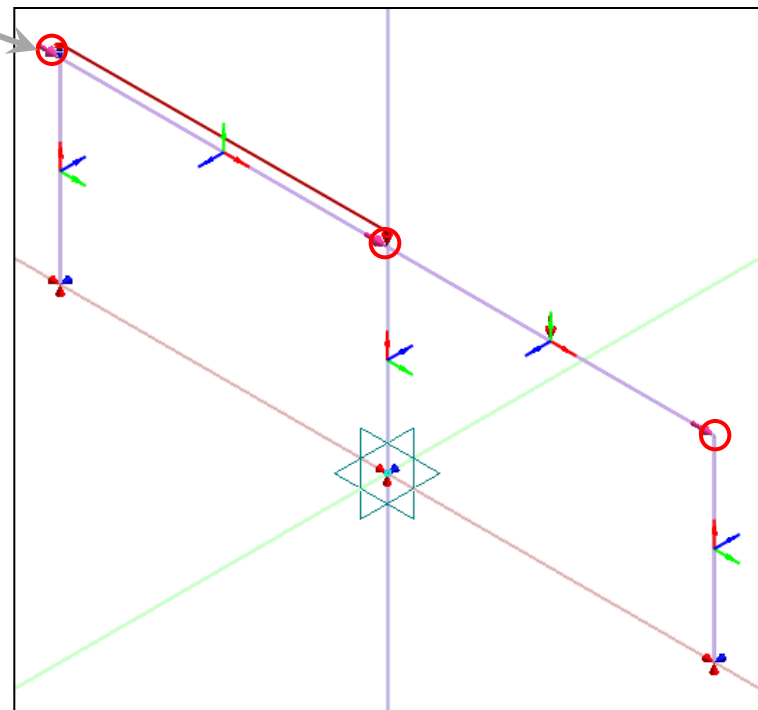
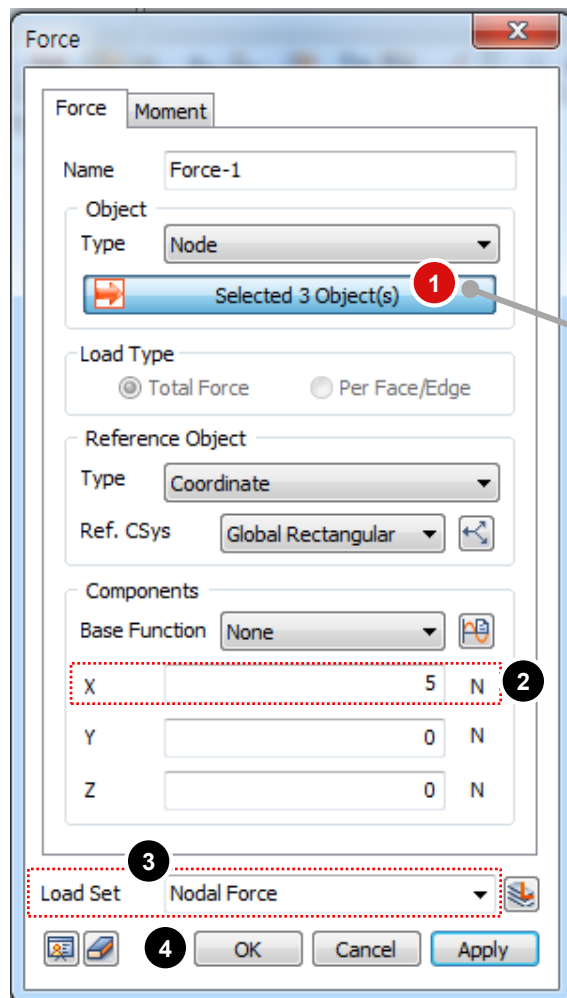
Load Set: Beam Load

OK Cancel Apply



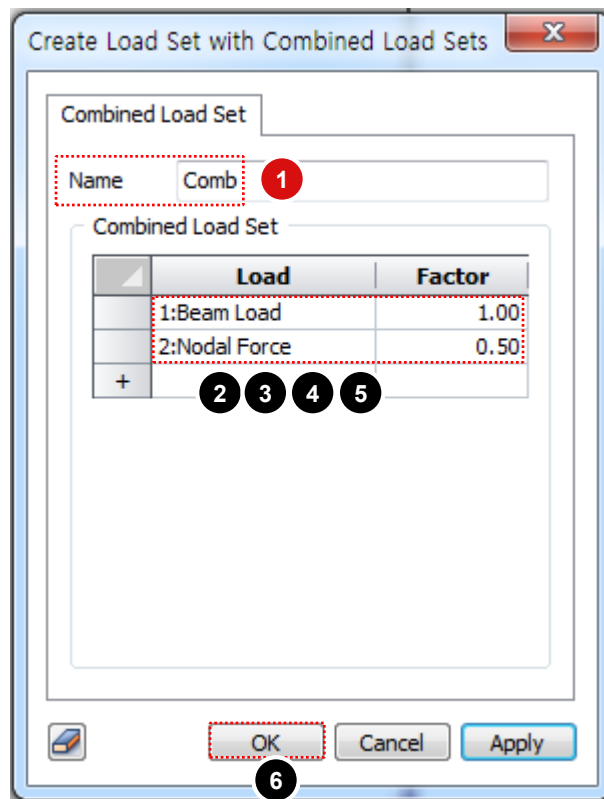
**Procedure**

- 1 Select 3 Nodes Marked by "O" (See Figure)
- 2 X : "5"
- 3 Load Set : [Nodal Force]
- 4 Click [OK] Button



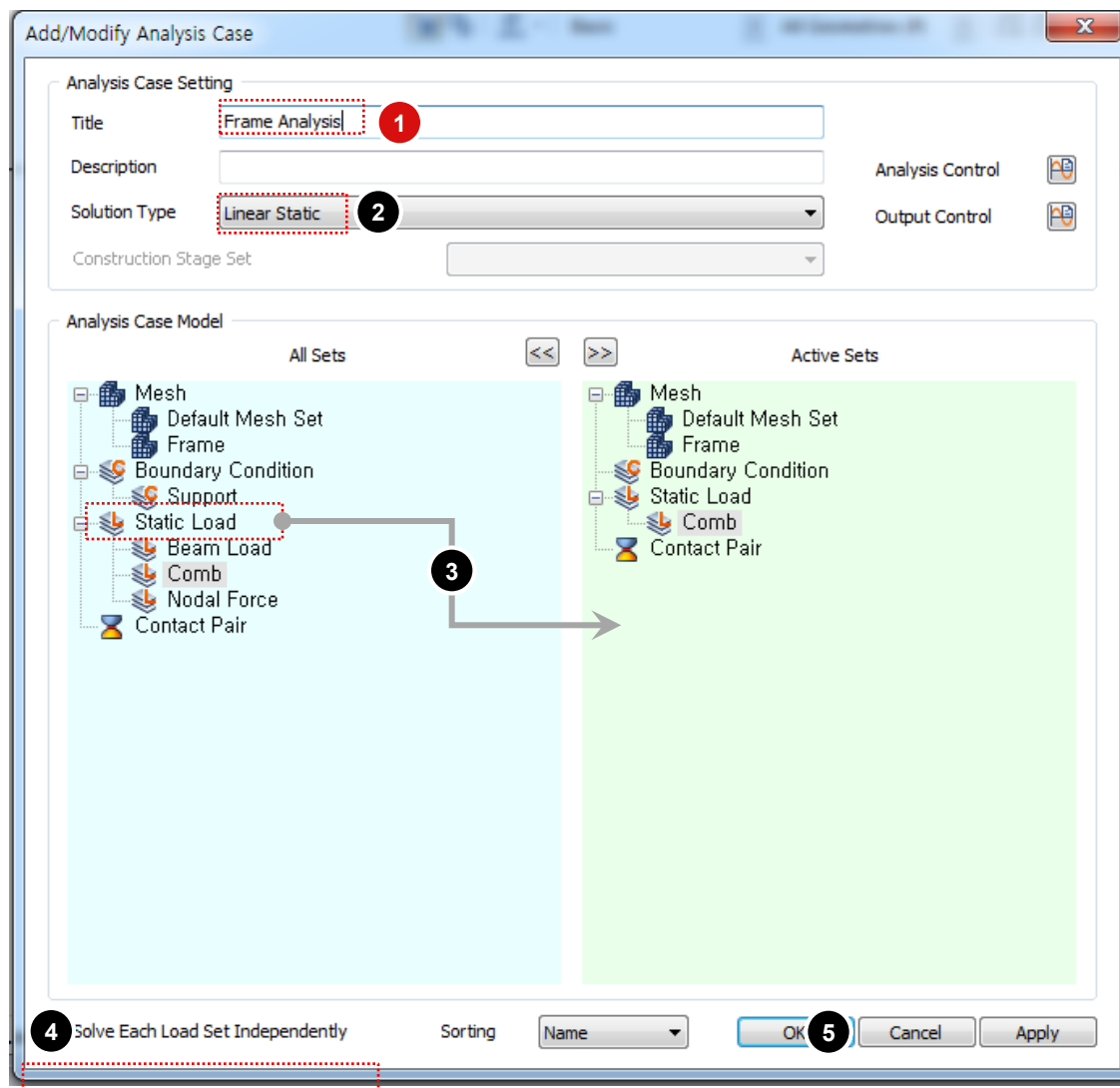
**Procedure**

- 1 Name : **"Comb"**
- 2 Load Set : **[Beam Load]**
- 3 Factor : **"1.0"**
- 4 Load Set : **[Nodal Force]**
- 5 Factor : **"0.5"**
- 6 Click **[OK]** Button



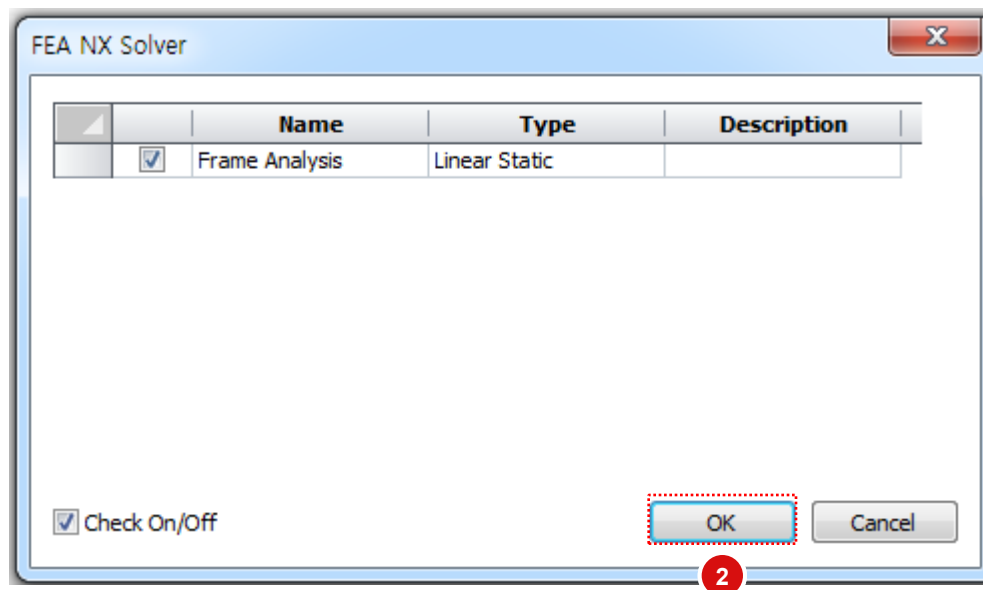
**Procedure**

- 1 Title : **"Frame Analysis"**
- 2 Solution Type : **[Linear Static]**
- 3 Drag & Drop **[Static Load]** to  
**[ActiveSets]** Window
- 4 Check on **"Solve Each Load Set Independent "**
- 5 Click **[OK]** Button
- 6 File > **Save...** : **"plane Frame.fea"**
- 7 Analysis > **Perform**



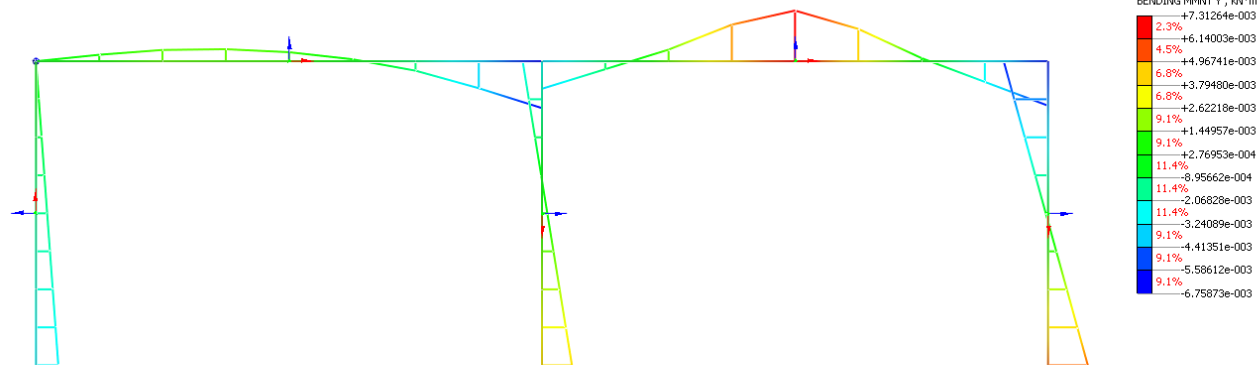
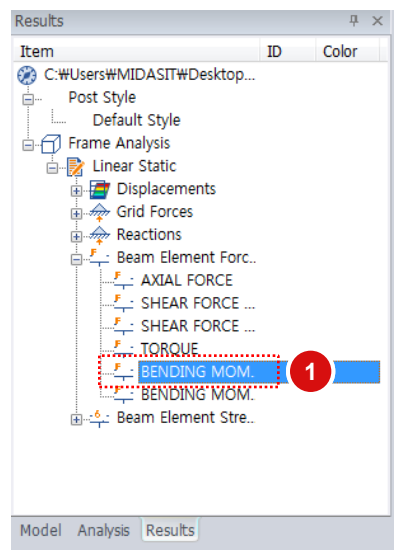
**Procedure**

- 1 Perform click
- 2 OK click



**Procedure**

- 1 Double Click **[BENDING MOM.]**



## Click “Animation Recording”

**Procedure**

- 1 Click **[Play]** Button
- 2 Click **[Save File]** and Save Animation as **[AVI]** format file

