

Overview

▪ 3-D Linear Static Analysis

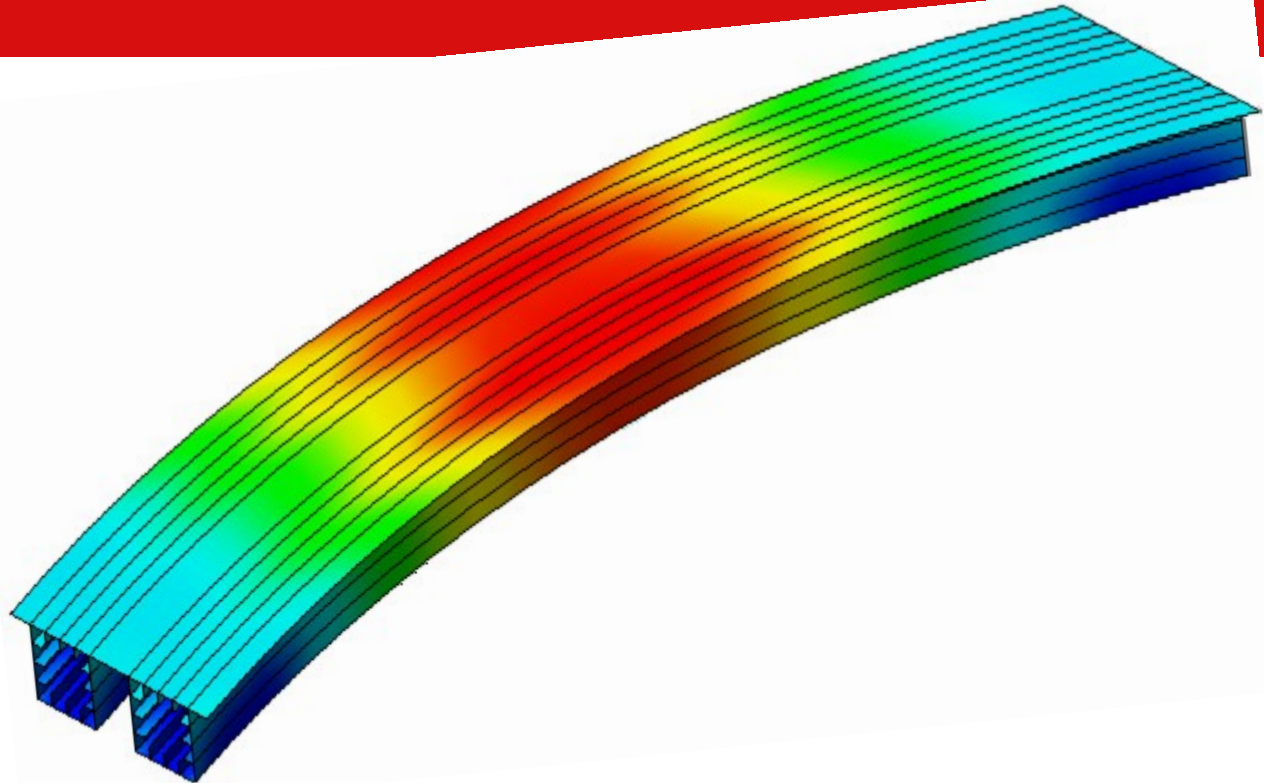
▪ Model

- Unit : N, cm
- Isotropic Elastic Material
- Plate Element

▪ Load & Boundary Condition

- Body Force
- Pressure
- Constraint

Curved Steel Box Bridge



Procedure

- 1 Analysis Type : **[3D]**
- 2 Force : **[N]**
- 3 Length : **[cm]**
- 4 Click **[OK]** Button
- 5 Click Right Mouse>
- [Move Work Plane]**
- 6 Select **[XZ Plane]**
- 7 Click **[OK]** Button



Analysis Control Dialog is automatically activated at startup.

The **Analysis Setting** dialog box is shown with the following settings and callouts:

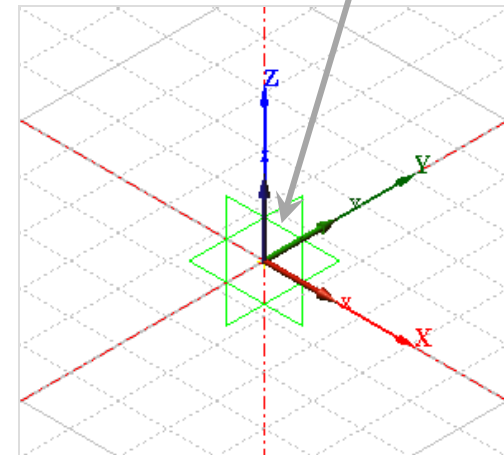
- Model Type:** 3D (Callout 1)
- Gravity Direction:** Z (Callout 2)
- Unit System:** N (Force), cm (Length), J (Energy), sec (Time) (Callouts 3 and 4 point to the unit dropdowns)
- Initial Parameters:** Gravity Acceleration (g) is 980.665 cm/sec², Initial Temperature is 0 [T], Plane Strain Thickness is 2.14748365e-11 cm.
- Buttons:** OK and Cancel buttons are at the bottom right.

A context menu is displayed with the following options:



- Show All
- Hide All
- Show All Geometries
- Hide All Geometries
- Show All Meshes
- Hide All Meshes
- Move Work Plane** (Callout 5)
- Show/Hide Guiders
- Show All Guiders
- Hide All Guiders
- Show All Labels
- Hide All Labels

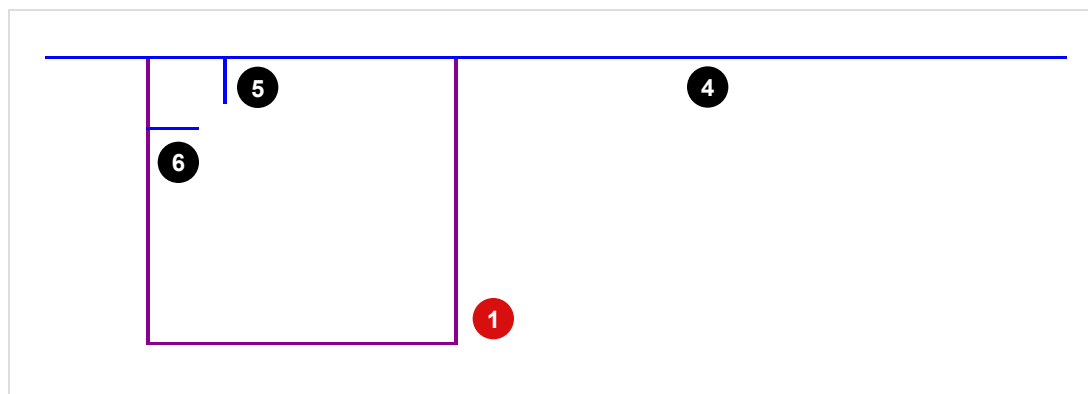
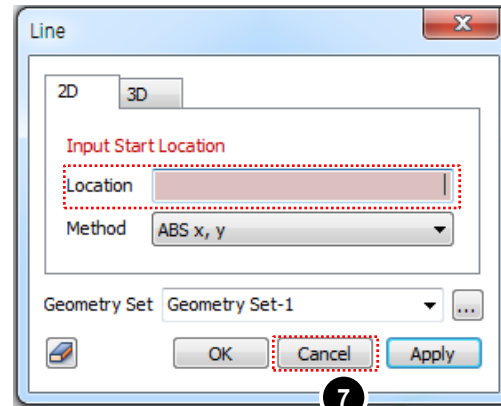
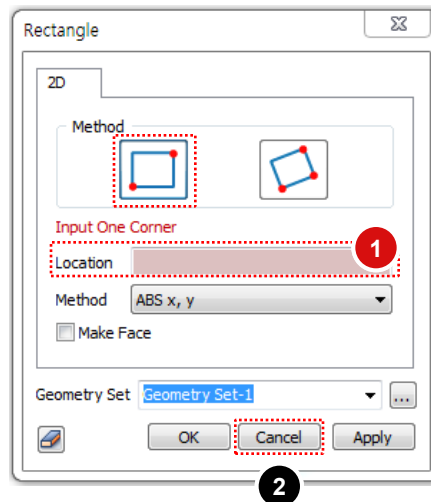
The **Move Work Plane** dialog box is shown with the following settings and callouts:


- Ref. Plane:** Three Points | Normal
- Select Plane:** (Callout 6 points to the selection button)
- Offset:** 0
- Origin:** 0, 0, 0
- Reverse Normal:** (unchecked)
- Buttons:** OK (Callout 7), Cancel, and Apply buttons are at the bottom right.




Procedure



- 1 Location : “(20, <60, -60>” 
- 2 Click [Cancel] Button 
- 3 Geometry > Point & Curve
> [Line...]
- 4 Location : “(0, <200>”
- 5 Location : “(35, <0, -10>”
- 6 Location : “(20, -15), <10>”
- 7 Click [Cancel] Button
- 8 Click [Normal View]

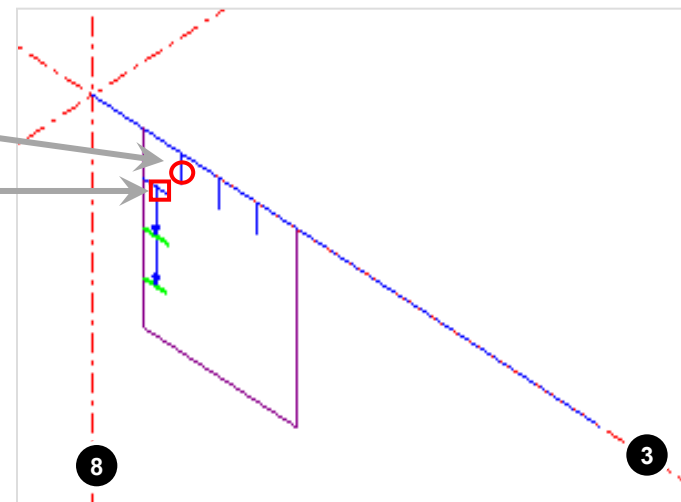
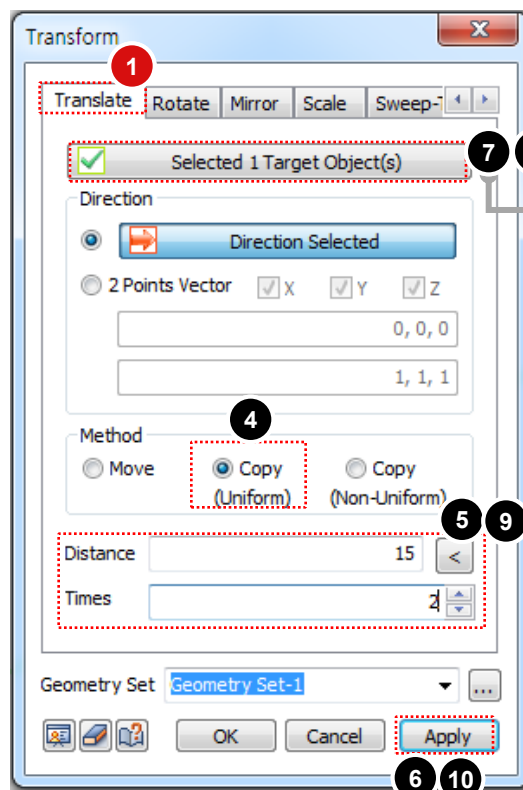


 () : “ABS x, y” , < > : “REL dx, dy”
(20) same as (20, 0)



 “Esc” as shortcut for “Cancel”.

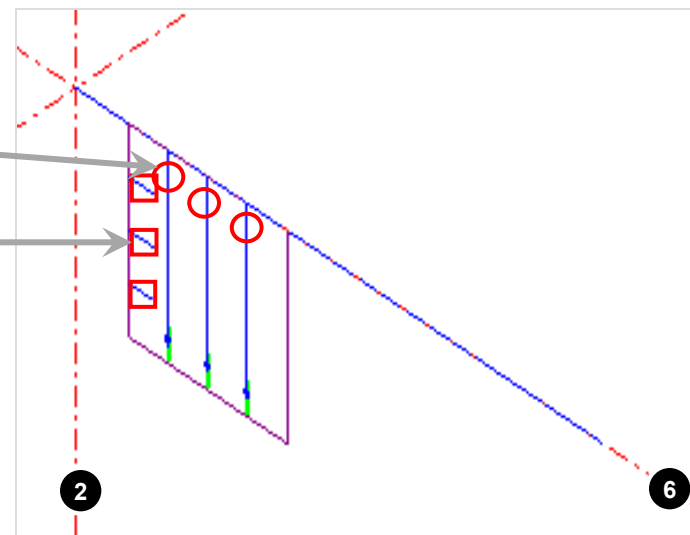
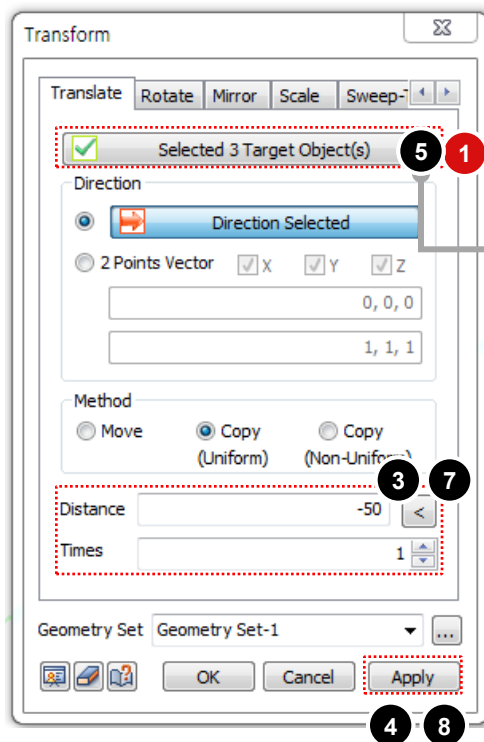
Procedure

- 1 [Translate] tab
- 2 Select Edge marked by []
(See Figure)
- 3 Direction : [X-Axis]
- 4 Check on [Copy(Uniform)]
- 5 Distance : “15”, Number of Times : “2”
- 6 Click [Apply] Button
- 7 Select Edge marked by []
(See Figure)
- 8 Direction : [Z-Axis]
- 9 Distance : “-15”, Number of Times : “2”
- 10 Click [Apply] Button



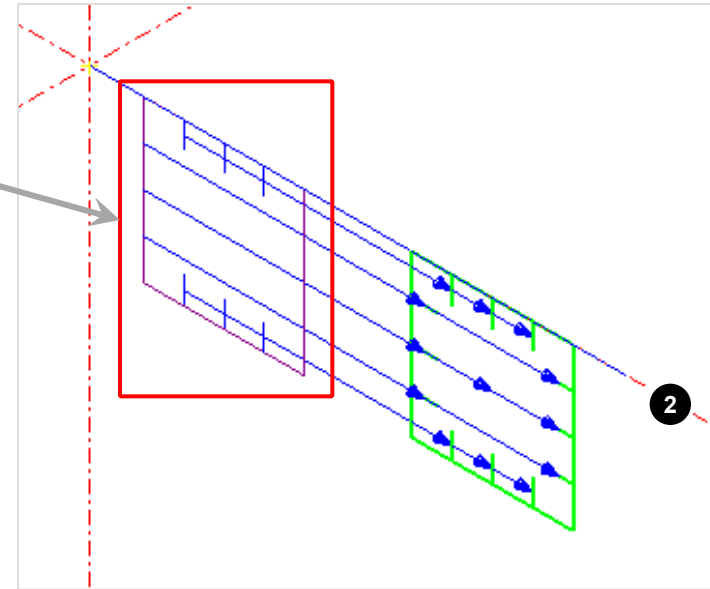
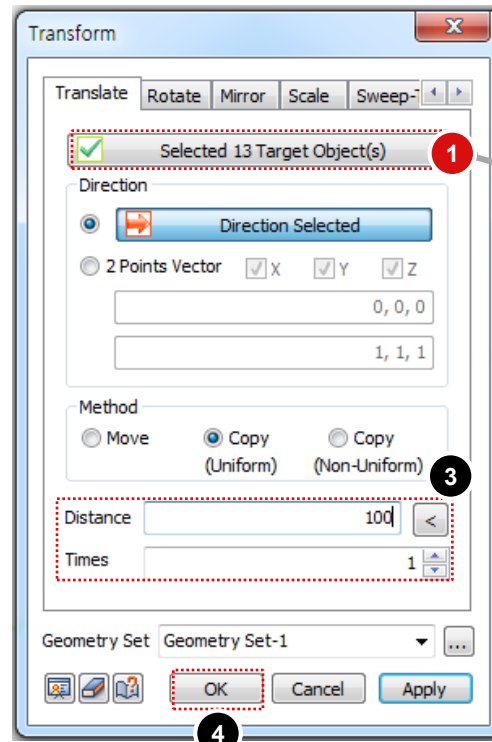
Procedure

- 1 Select Edges marked by []
(See Figure)
- 2 Direction : [Z-Axis]
- 3 Distance : “-50”, Number of Times : “1”
- 4 Click [Apply] Button
- 5 Select Edges marked by []
(See Figure)
- 6 Direction : [X-Axis]
- 7 Distance : “50”, Number of Times : “1”
- 8 Click [Apply] Button






Procedure


- 1 Select **[13 Edges]** (See Figure)
- 2 Direction : **[X-Axis]**
- 3 Distance : **"100"** , Number of Times : **"1"**
- 4 Click **[OK]** Button

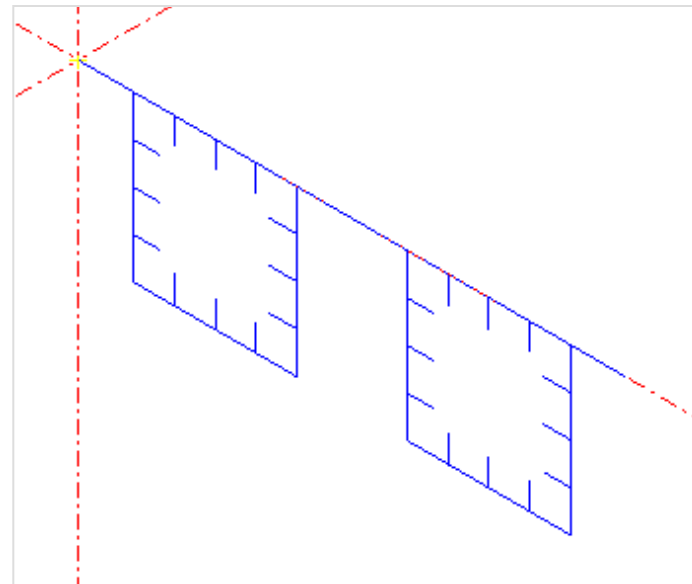
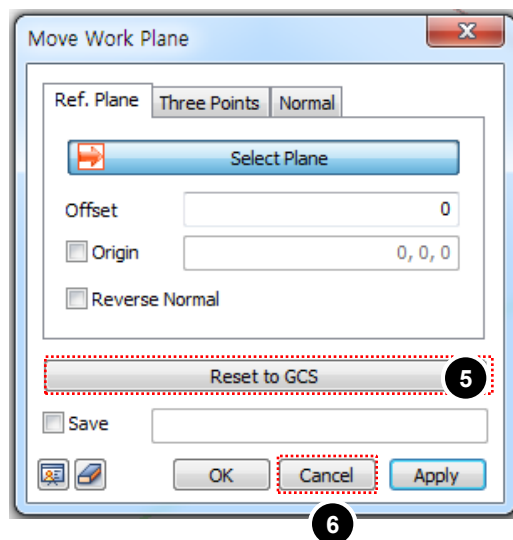
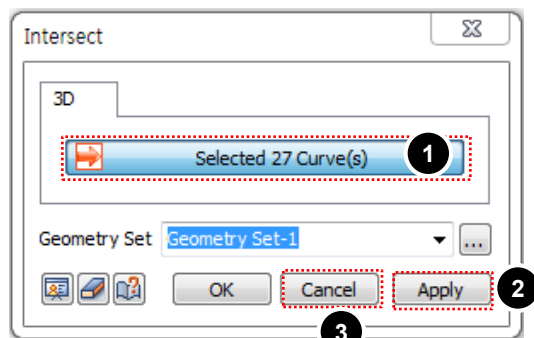


Procedure

- 1 Select Displayed 
- 2 Click **[Apply]** Button 
- 3 Click **[Cancel]** Button
- 4 Click Right Mouse >
[Move Work Plane]
- 5 Click **[Reset to GCS]** Button
- 6 Click **[Cancel]** Button

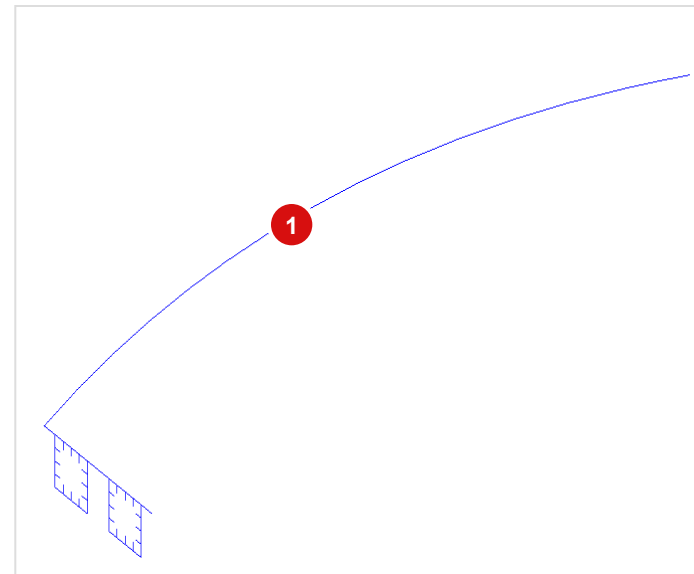
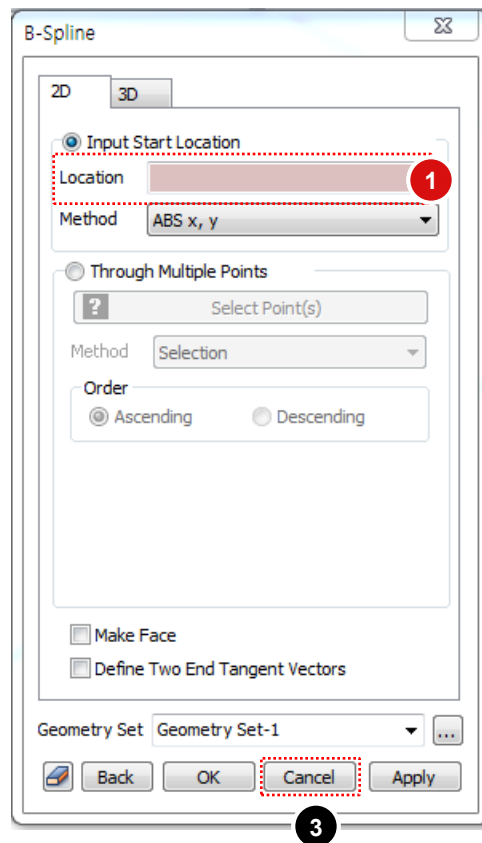
 1 “Ctrl+A” as shortcut for “Select Displayed All”.

 2 “Enter” as shortcut for “Apply”



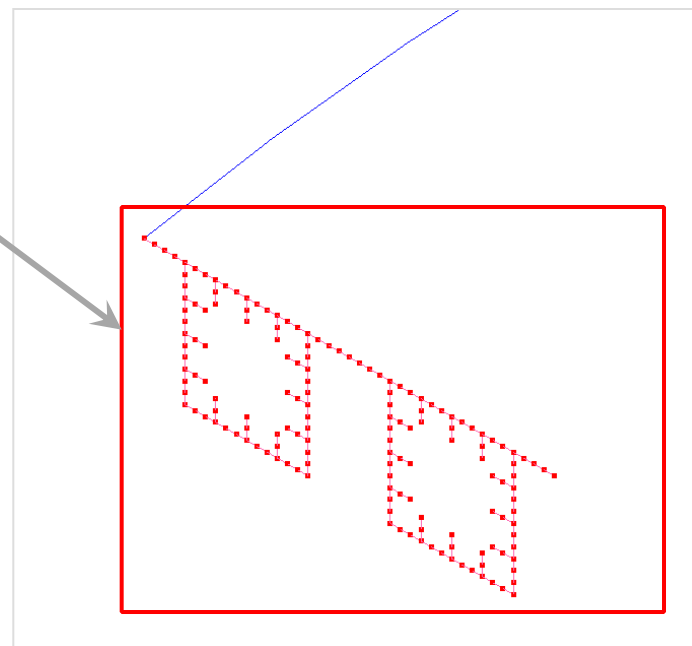
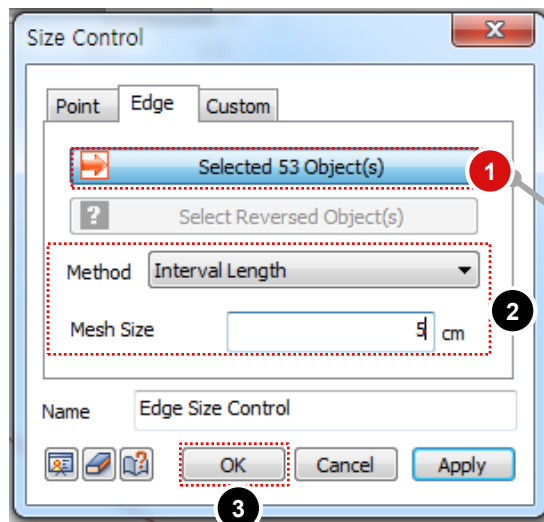
Procedure

- 1 Location : “(0), <0, 500>,<200, 500>”
- 2 Click Right Mouse Button on the
Work Window
- 3 Click **[Cancel]** Button



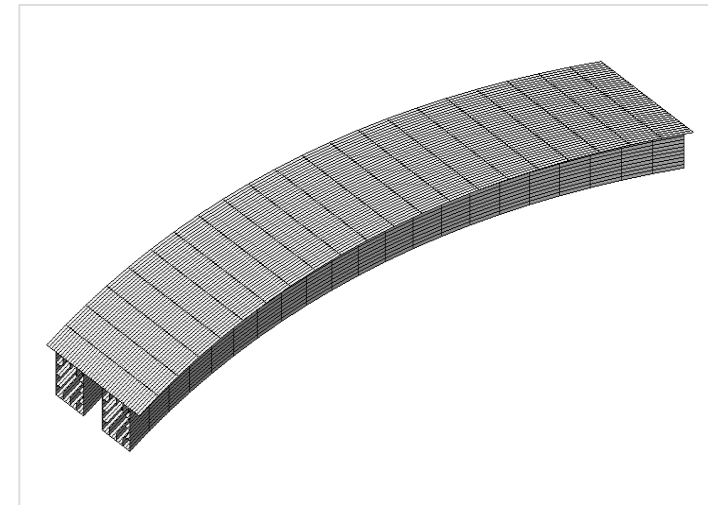
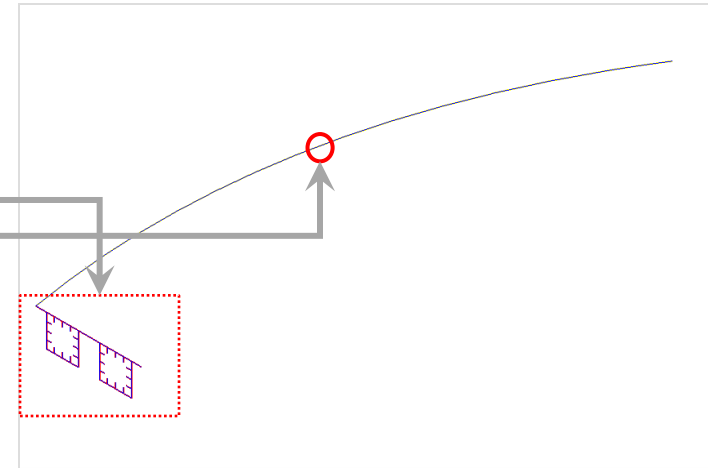
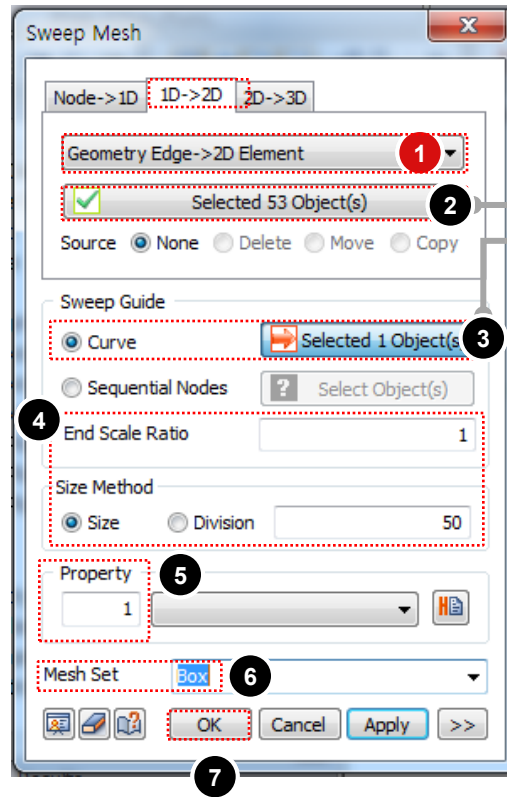
Procedure

- 1 Select **[53 Edges]** (See Figure)
- 2 Seeding Method - **[Interval Length]** : “5”
- 3 Click **[OK]** Button



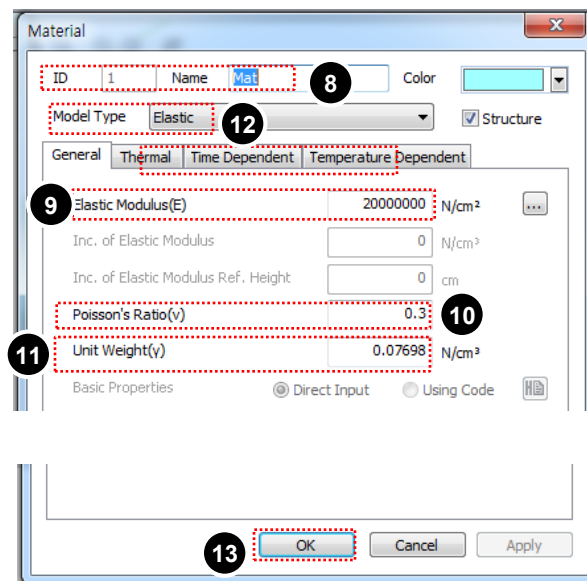
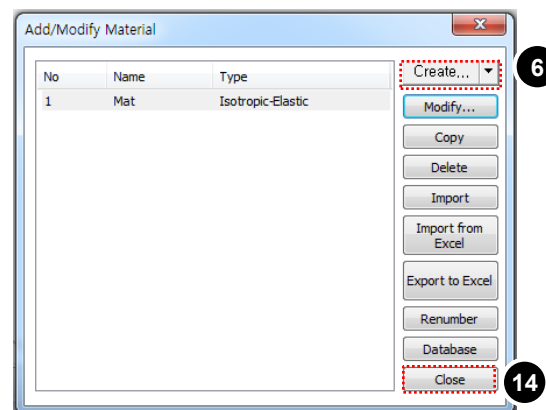
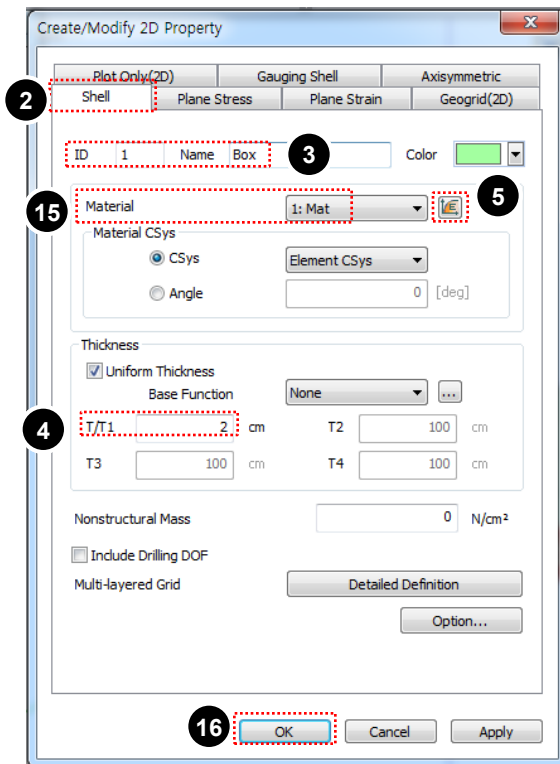
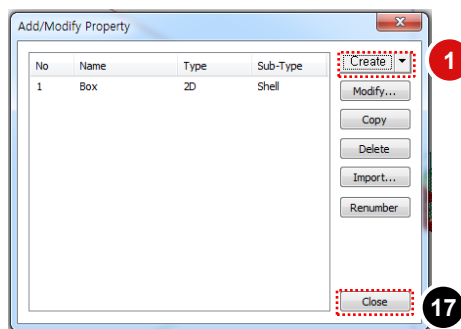
Procedure

- 1 Select **[Geometry Edge→2D]** Tab
- 2 Select **[53 Edges]** (See Figure)
- 3 Select Guide Curve marked by [**O**]
(See Figure)
- 4 Mesh Size - Element Size :
"50", Scale Factor : "1"
- 5 **[Property]** : "1"
- 6 Mesh Set : **[Box]**
- 7 Click **[OK]** Button



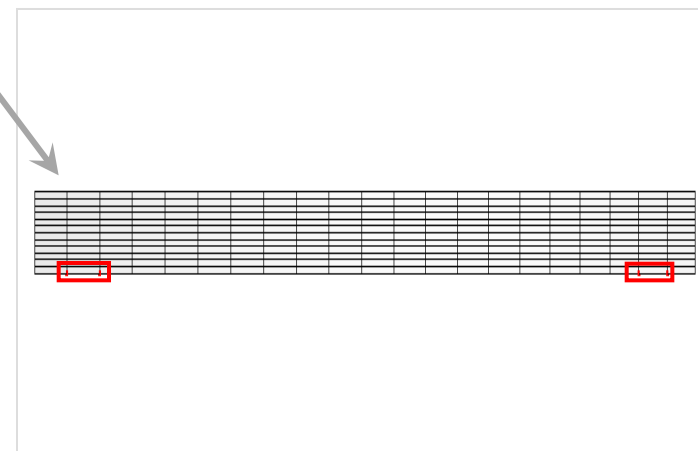
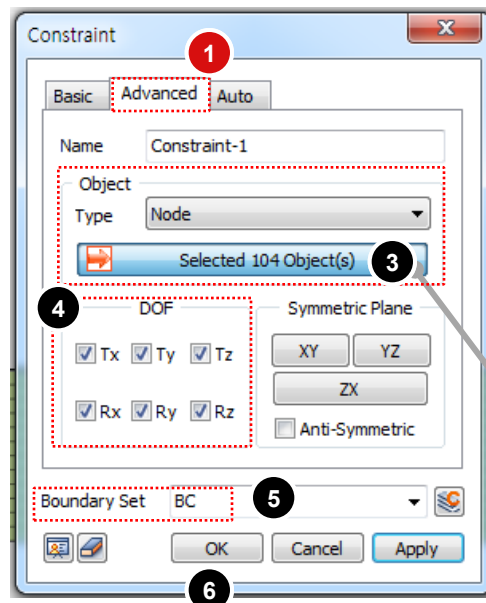
Procedure

- 1 Create [2D...]
- 2 Select [Shell] tab
- 3 ID : "1", Name : "Box"
- 4 T or T1 : "2" cm
- 5 Click [Material] Button
- 6 Click [Create] Button
- 7 Select [Isotropic] tab
- 8 ID : "1", Name : "Mat"
- 9 Elastic Modulus : $2e7$ N/cm²
- 10 Poisson's Ratio : "0.3"
- 11 Unit Weight : $7.698e-2$ N/cm³
- 12 Model Type : [Elastic]
- 13 Click [OK] Button
- 14 Click [Close] Button
- 15 Select "1: Mat" for Material
- 16 Click [OK] Button
- 17 Click [Close] Button




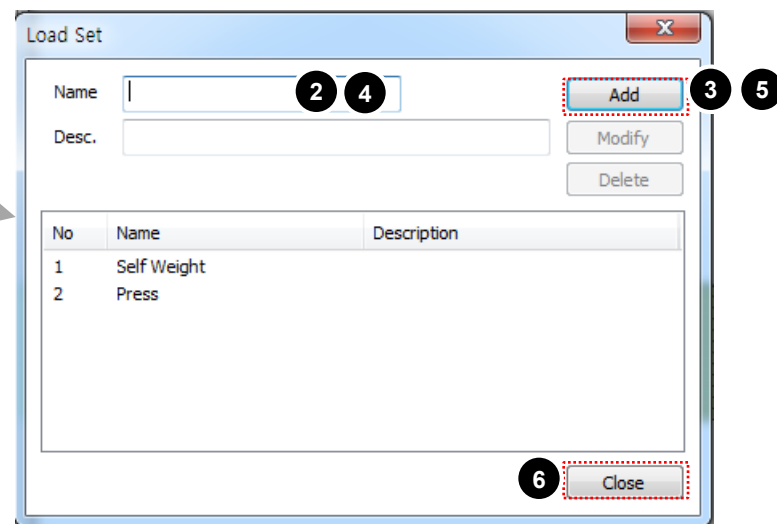
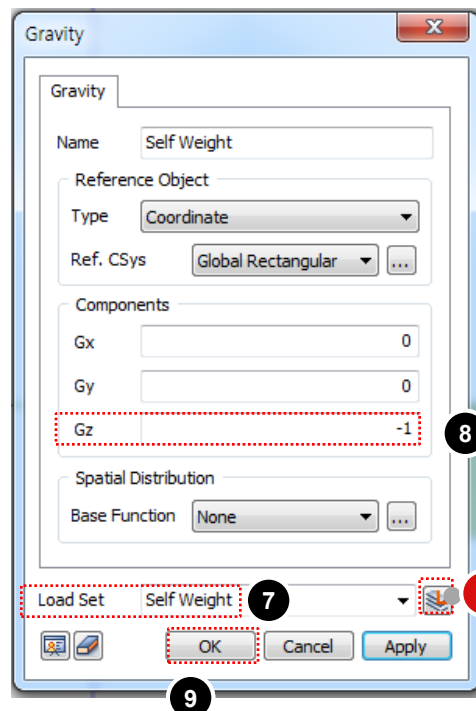
Procedure

- 1 [Advanced] Tab
- 2 Click [Right View]
- 3 Select [104 Nodes] (See Figure)
- 4 DOF : Tx, Ty, Tz, Rx, Ry, Rz
- 5 Boundary Set : BC
- 6 Click [OK] Button



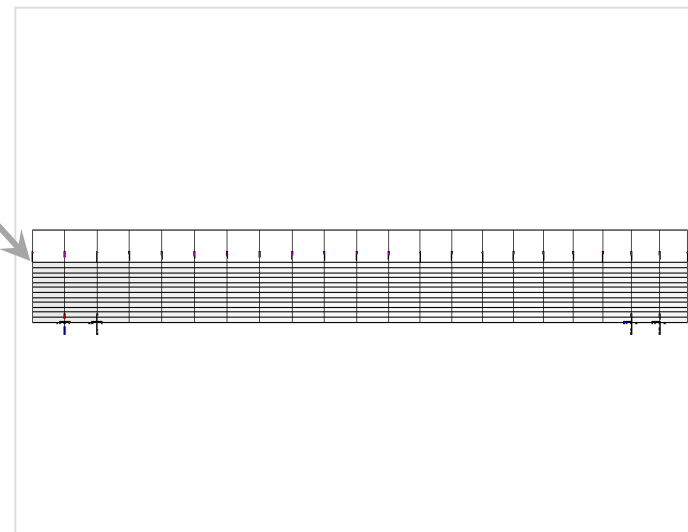
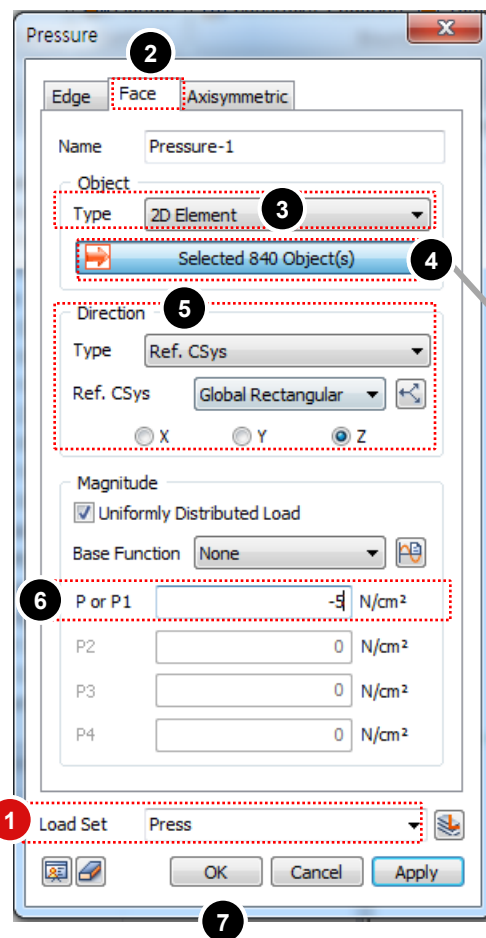
Procedure

- 1 Click []Button
- 2 Name : **"Self Weight"**
- 3 Click [Add] Button
- 4 Name : **"Press"**
- 5 Click [Add] Button
- 6 Click [Close] Button
- 7 Select [Self Weight] for Load Set
- 8 [Z]: **"-1"**
- 9 Click [OK] Button



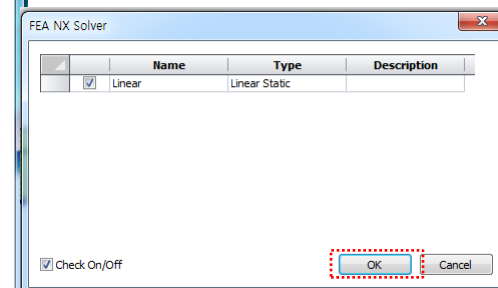
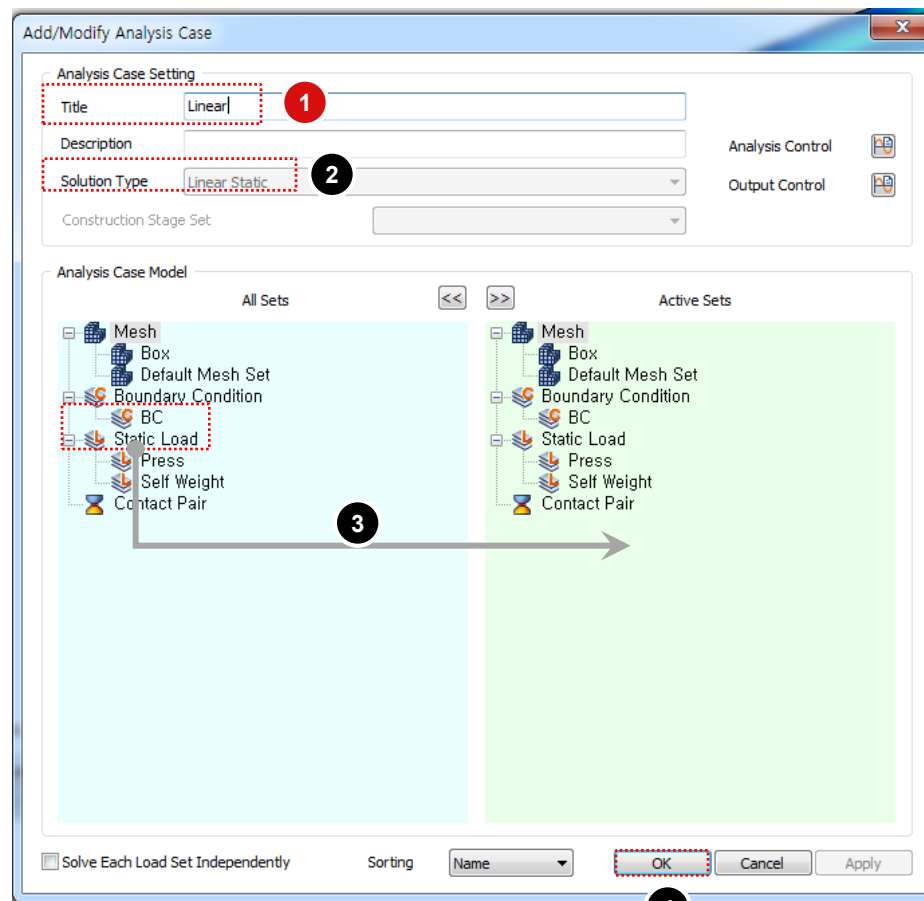
Procedure

- 1 Select **[Press]** for Load Set
- 2 Type : **[Face]**
- 3 Object Type : **[2D Element]**
- 4 Select **[840 Elements]** (See Figure)
- 5 Direction : **[Ref. CSys-Axis]**
- 6 P or P1 : **"-5" N/cm²**
- 7 Click **[OK]** Button



Procedure

- 1 Title : **"Linear"**
- 2 Solution Type : **[Linear Static]**
- 3 Drag & Drop **[Load]** to **[Active]** Window
- 4 Click **[OK]** Button
- 5 File > **[Save...]** (Box Girder.feb)
- 6 Analysis > **[Solve...]**
- 7 Click **[OK]** Button



Procedure

1 Linear > Linear Static > Displacements

2 TOTAL TRANSLATION

3 Shell Element Stresses >

S-VON MISES TOP

