

Overview

▪ 3-D Linear Static Analysis

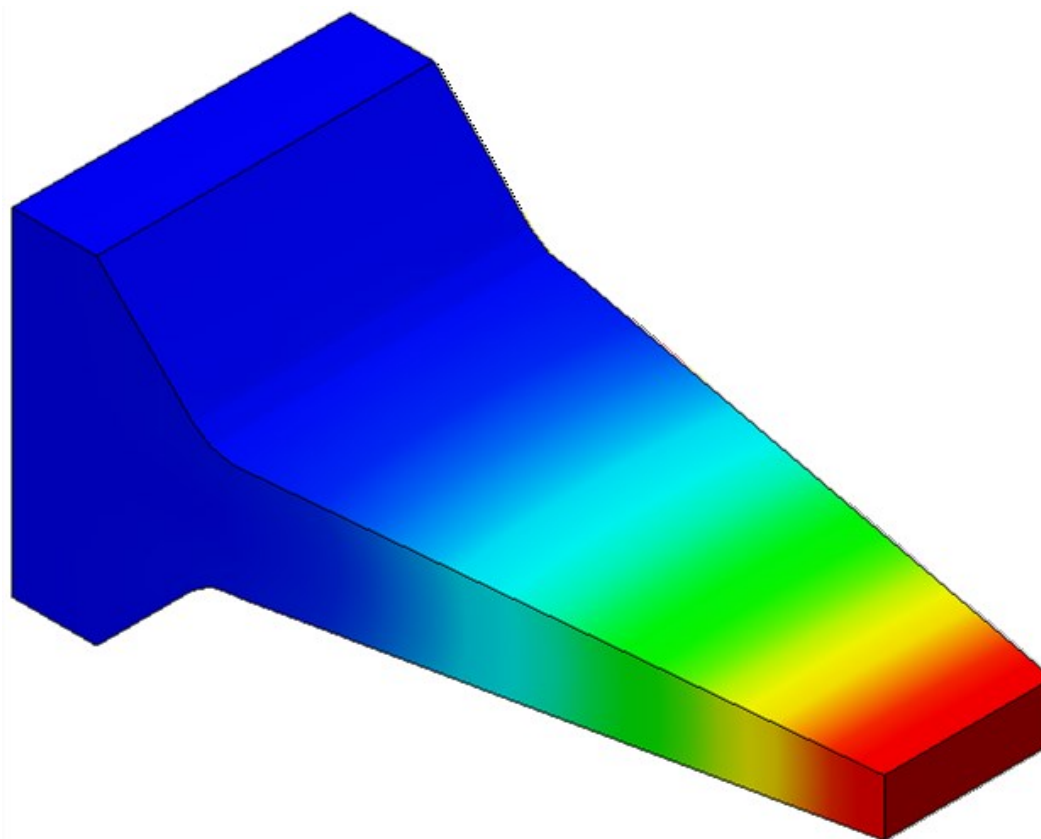
▪ Model

- Unit : N, cm
- Isotropic Elastic Material
- Solid Elements

▪ Load & Boundary Condition

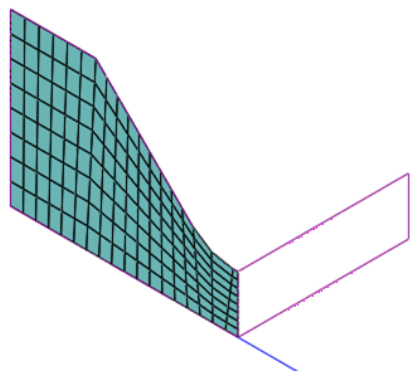
- Pressure
- Function
- Constraint

Analysis of a Solid Block

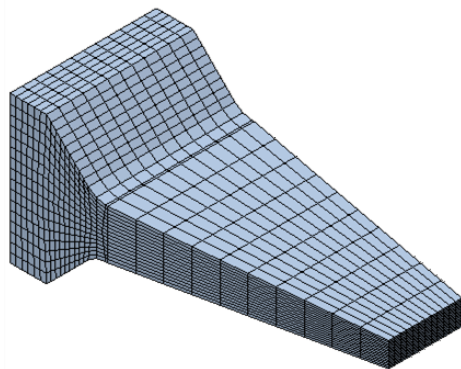


Modeling & Analysis Point

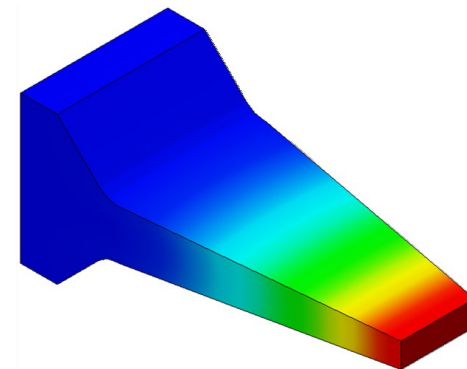
2D → 3D Modeling



Constraint & Load



Result Plot



About this Training...

This tutorial is composed of 3D solid elements generated using the sweep function for 2D concrete blocks.

The face pressure load of an element is expressed as a function.

Learning Target

How to model 2D geometry elements with *Geometry > Polyline, Fillet* function

Create K-Edge 2D Mesh of Map Mesh

How to expand from 2D Mesh to 3D Solid Mesh with Sweep function

How to apply the pressure load as a function on the face by defining a quadric equation

**Procedure**

- 1 Model Type : **[3D]**
- 2 Force(Mass) : **[N]**, Length : **[cm]**
- 3 Click **[OK]** Button



The screenshot shows the 'Analysis Setting' dialog box with the following fields and settings:


- Project Title**: [Empty text box]
- Engineer**: [Empty text box]
- Desc.**: [Empty text box]
- Model Type**: Radio buttons for 3D (selected), 2D, and Axisymmetric. A red dashed box and callout '1' highlight the 3D button.
- Gravity Direction**: Radio buttons for Y and Z (selected).
- Unit System**: Four dropdown menus. The first two are 'N' and 'cm' (highlighted with a red dashed box and callout '2'). The last two are 'J' and 'sec'. A red dashed box and callout '3' highlight the 'OK' button at the bottom.
- Initial Parameters** tab is active, showing:
 - Gravity Acceleration(g)**: 980.665 cm/sec²
 - Initial Temperature**: 0 [T]
 - Plane Strain Thickness**: 1 cm



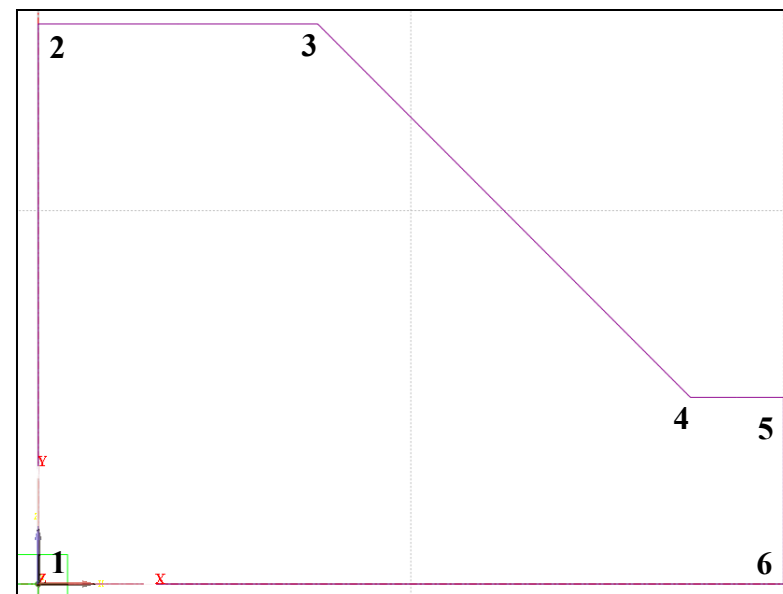
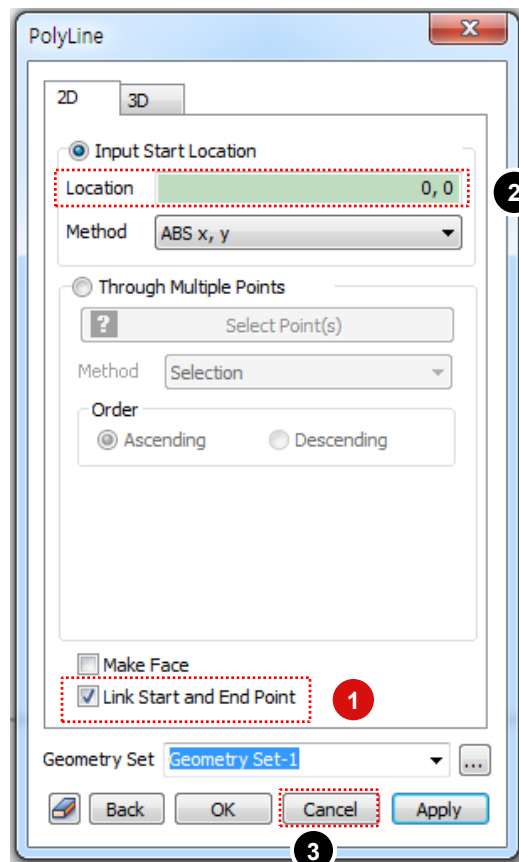
Analysis Setting Dialog is automatically activated at startup.

Procedure


- 1 “Link Start and End Point” On
- 2 Location : (0) , <0, 3> , <1.5> ,
<2, -2> , <0.5> , <0, -1> 
- 3 Click [Cancel] Button 
- 4 Click “Normal View”

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

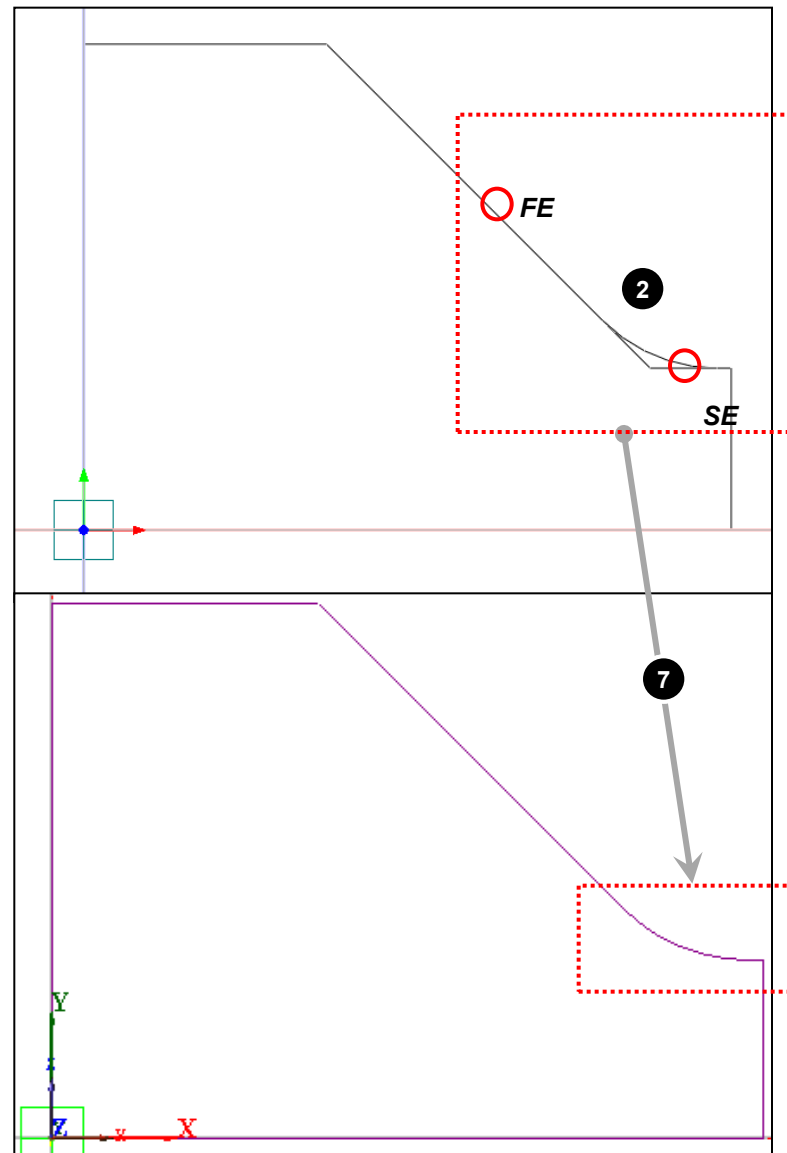
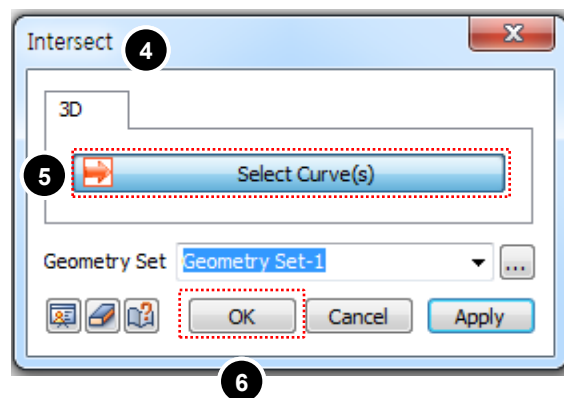
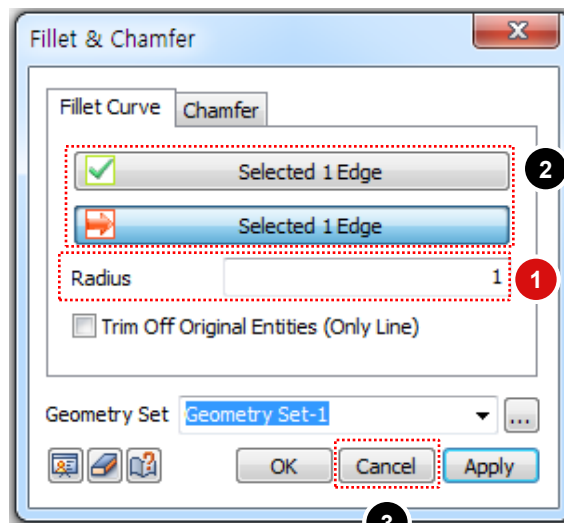
 2 [Esc] as shortcut for [Cancel].



Procedure

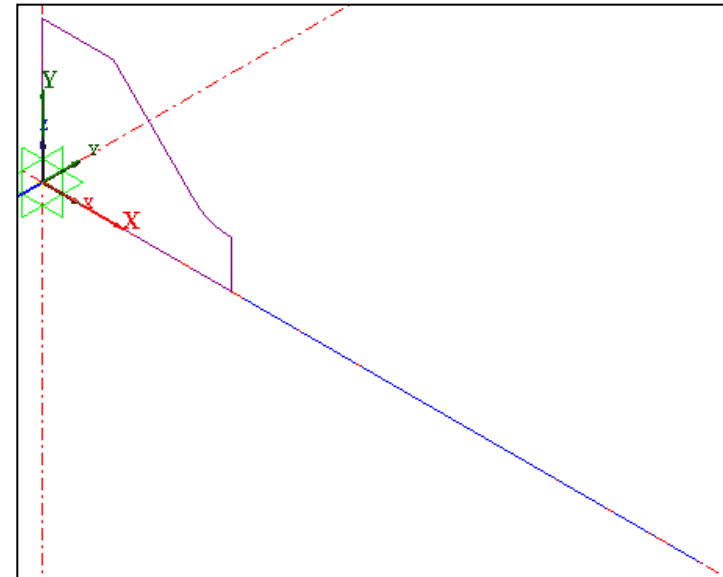
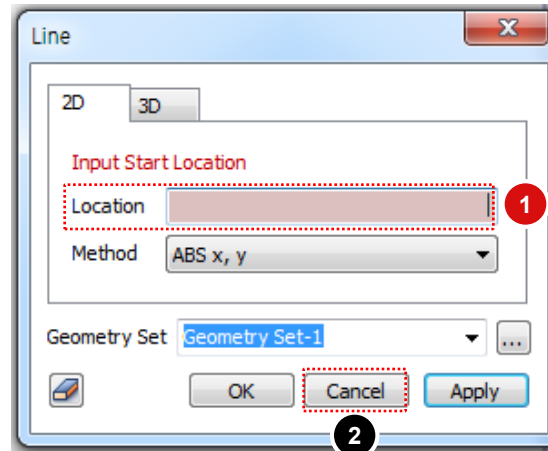
- 1 Radius : " 1 "
- 2 Select **FE** and **SE** (See Figure) 
- 3 Click **[Cancel]** Button
- 4 Geometry >
Point & Curve > **INTERSECT**
- 5 Select All
- 6 Click **[OK]** Button
- 7 **Delete** line

 FE : First Entity , SE : Second Entity



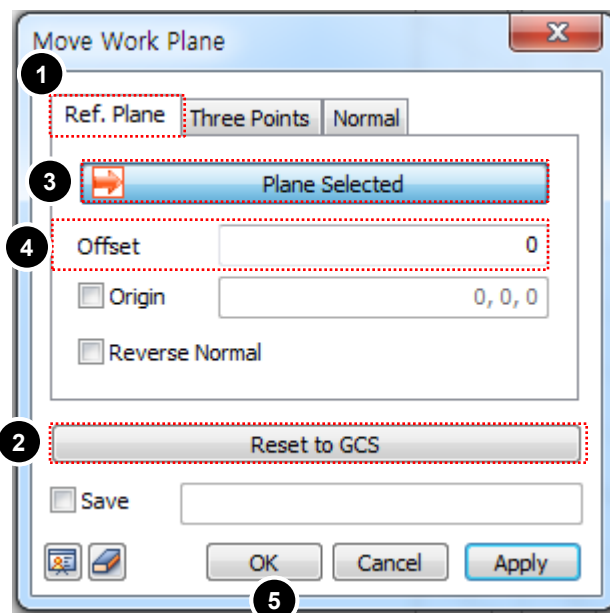
Procedure

- 1 Location : (4) , <10>
- 2 Click [**Cancel**] Button
- 3 Select : “Isometric 1 View”



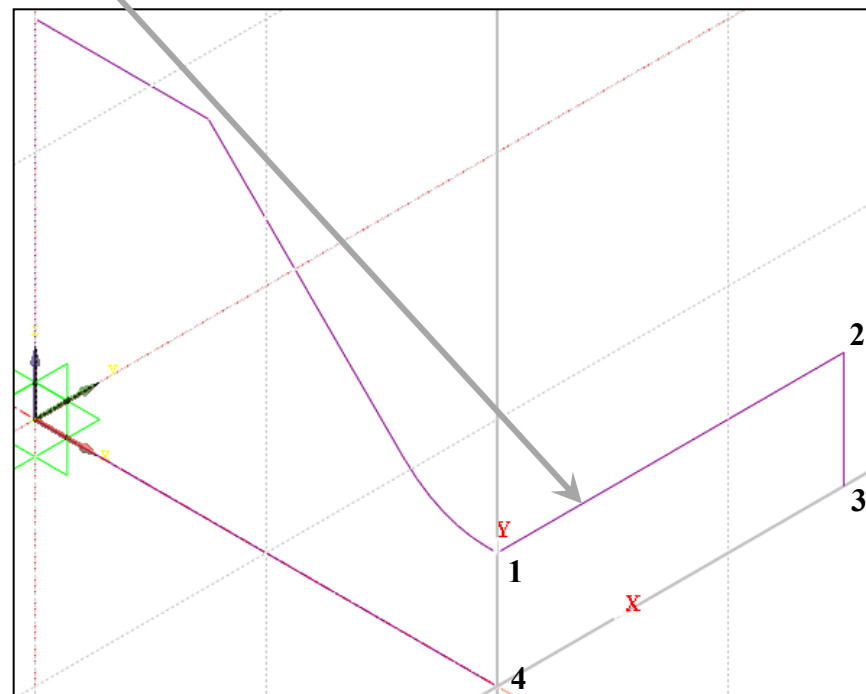
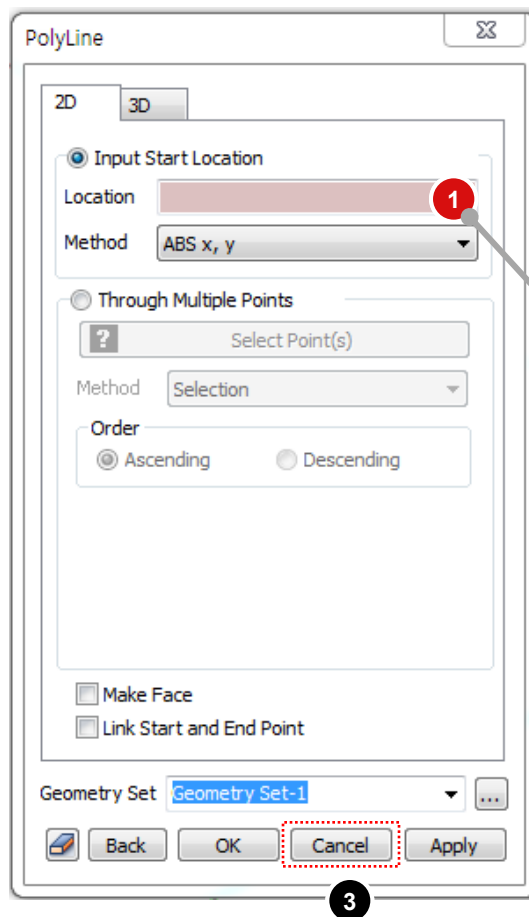
Procedure

- 1 Select : **“Ref.Plane”**
- 2 Click : **“Reset to GCS ”**
- 3 Select : **“YZ Plane”**
- 4 Offset : **“ 4 ”**
- 5 Click **[OK]** Button



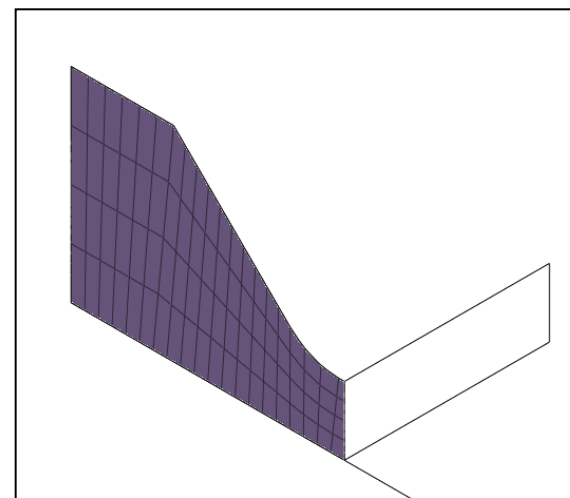
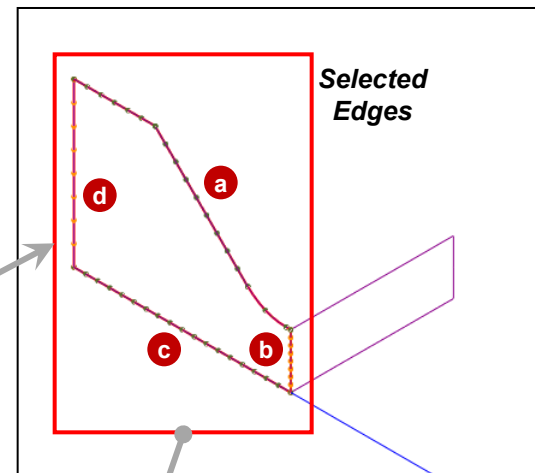
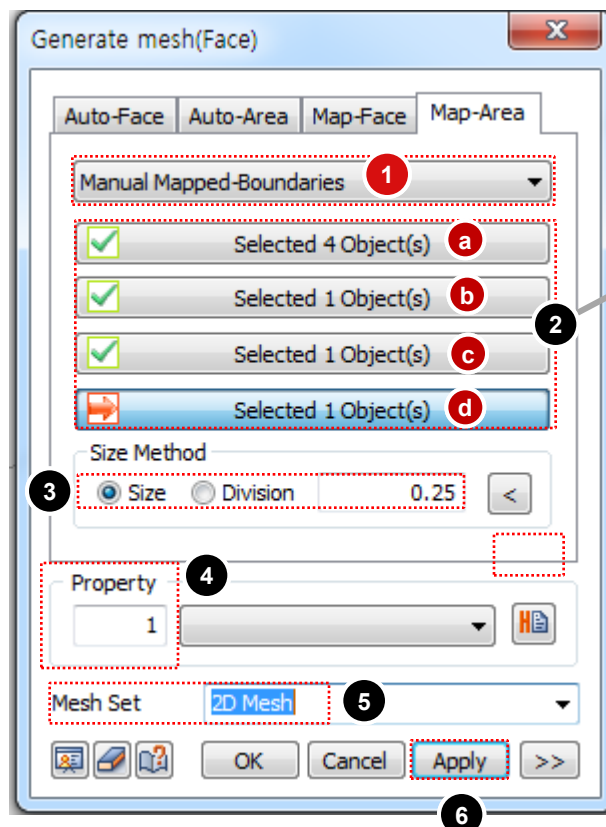
Procedure

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



Procedure

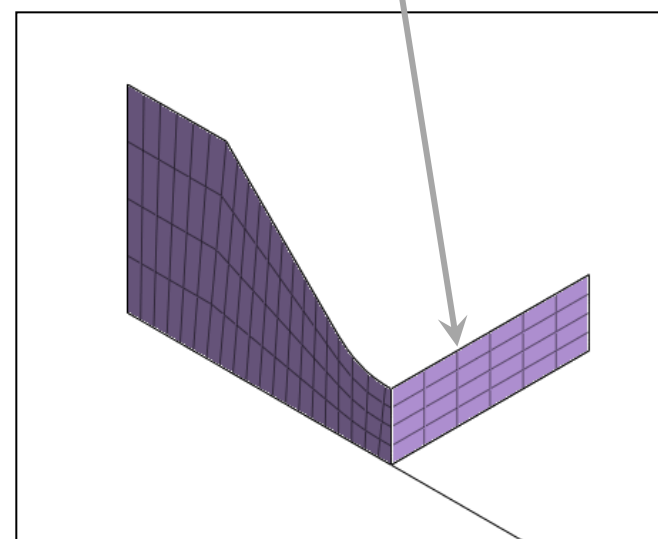
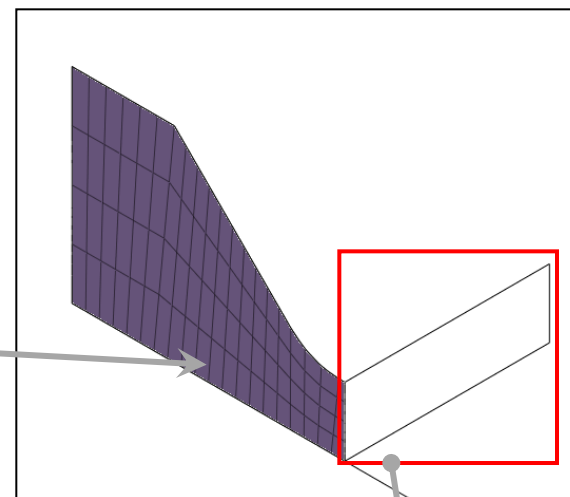
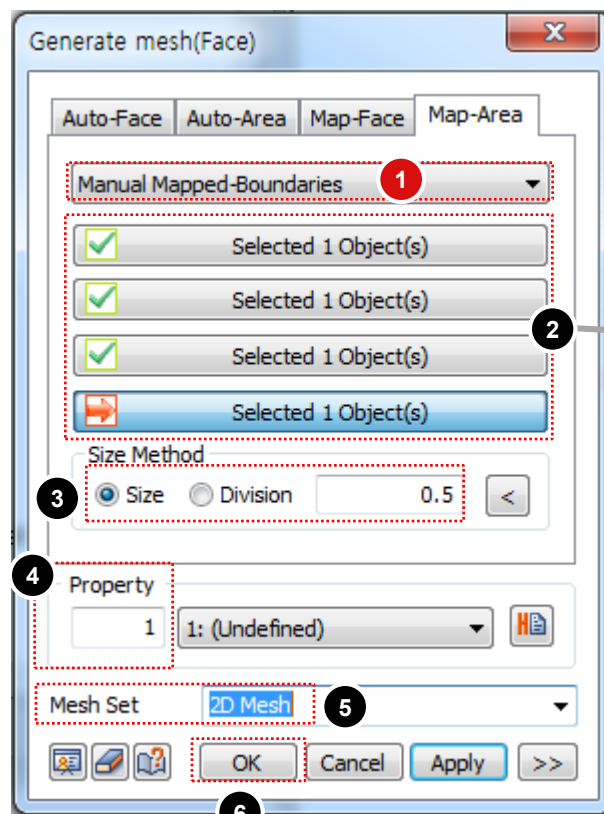
- 1 Select : **"Manual Mapped..."**
- 2 Select 7 Edge Groups (See Figure)
- 3 Mesh Size : **Element Size : 0.25**
- 4 Property : **"1"**
- 5 Mesh Set : **"2D Mesh"**
- 6 Click **[Apply]** Button



Procedure

- 1 Select : **"Manual Mapped..."**
- 2 Select each Edge (See Figure 1)
- 3 Mesh Size : **Element Size : 0.5**
- 4 Property : **"1"**
- 5 Mesh Set : **"2D Mesh"**
- 6 Click **[OK]** Button

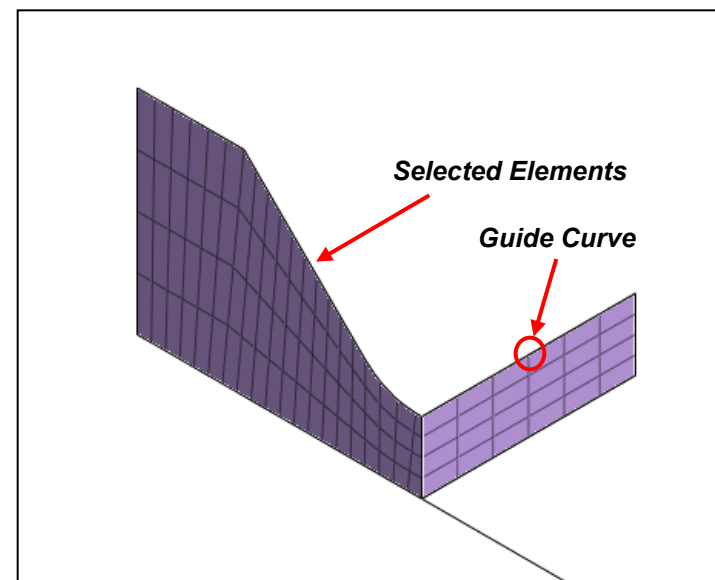
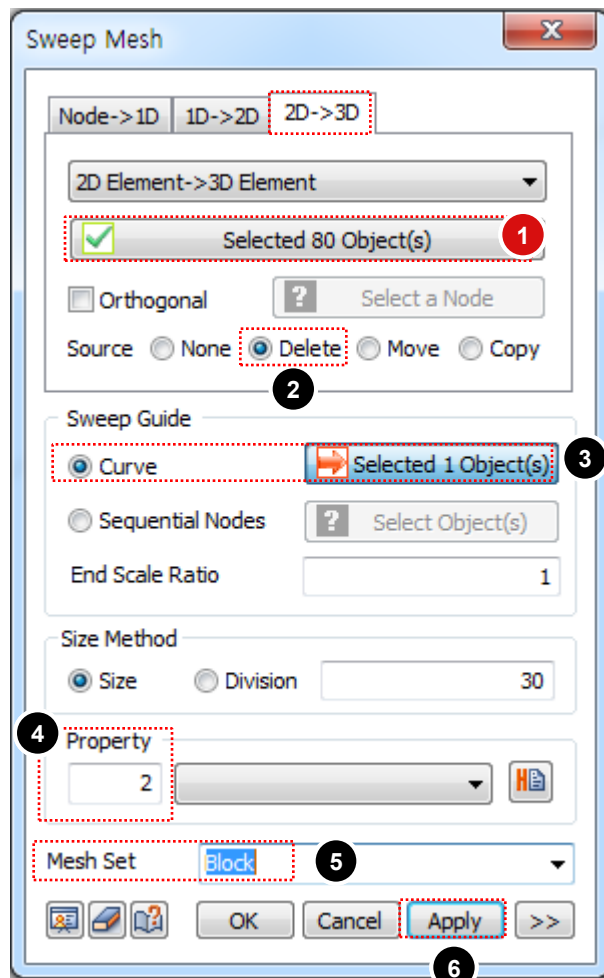
1 4 Edges should be selected



Procedure

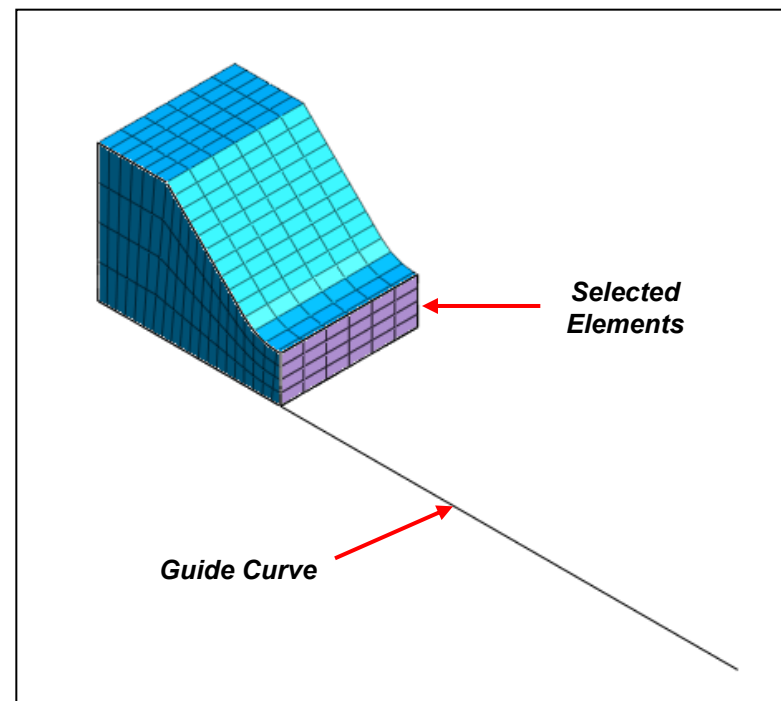
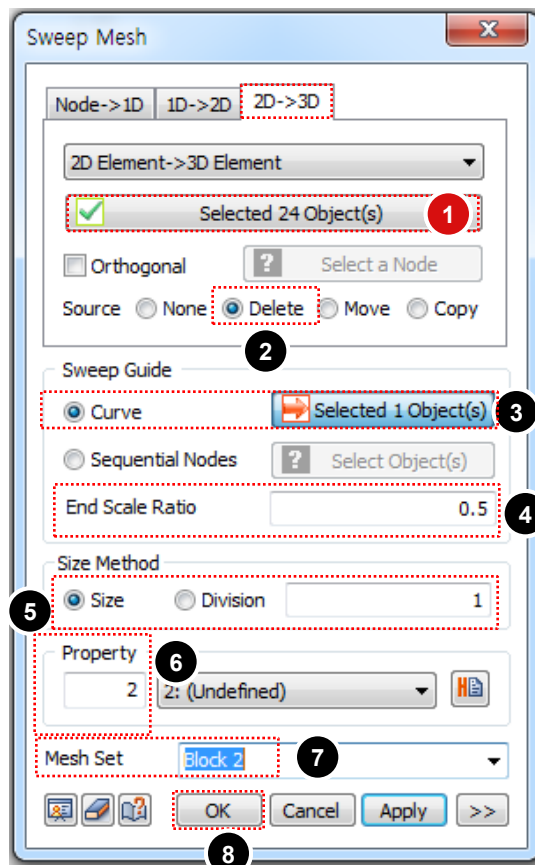
- 1 Select “**80 Elements**” (See Figure)
- 2 Source Mesh : [**Delete**]
- 3 Select “**Guide Curve**” (See Figure)
- 4 Property : “**2**”
- 5 Mesh Set : [**Block**]
- 6 Click [**Apply**] Button

 Mesh Size is automatically disabled after the selection of guide curve



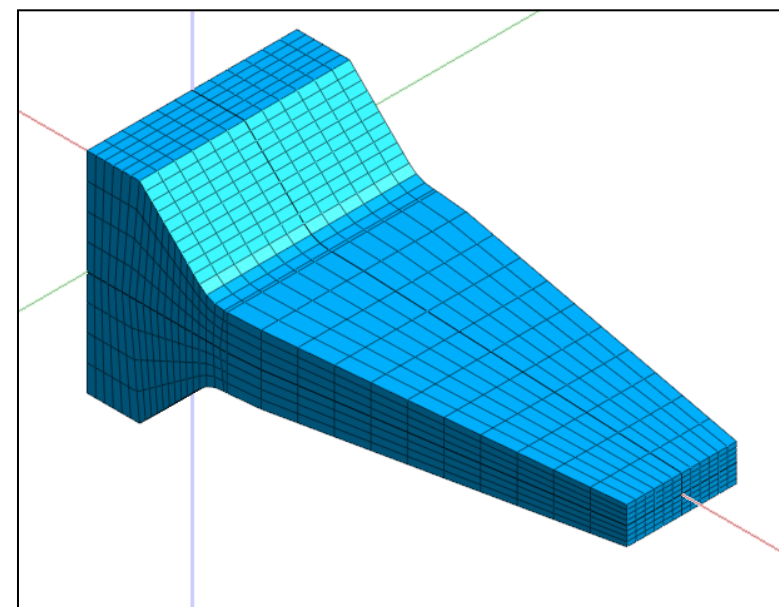
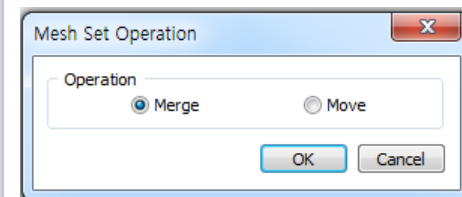
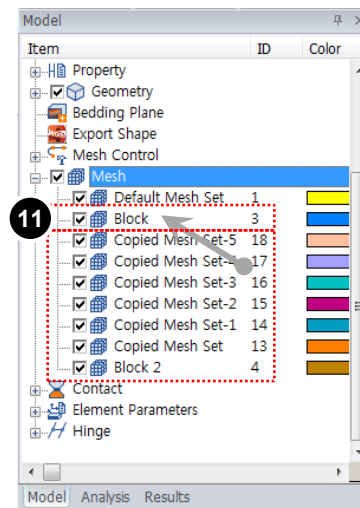
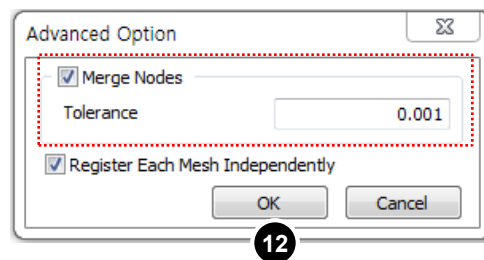
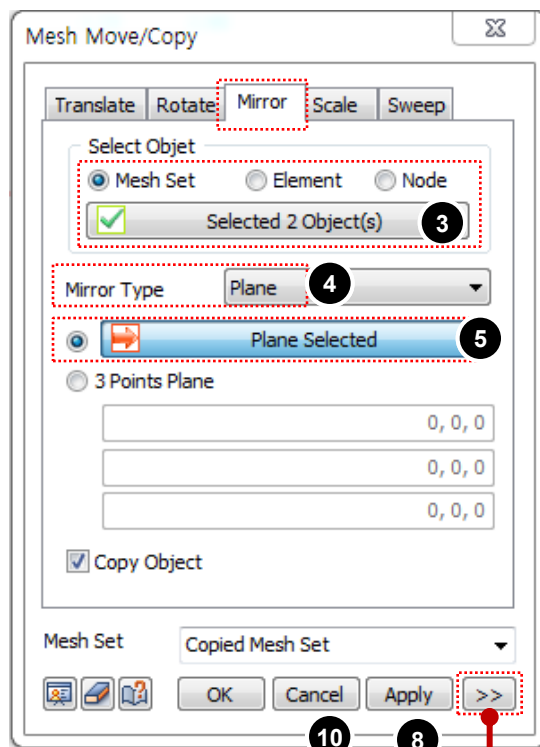
Procedure

- 1 Select “**24 Elements**” (See Figure)
- 2 Source Mesh : **[Delete]**
- 3 Select “**Guide Curve**” (See Figure)
- 4 End Scale Ratio : **0.5**
- 5 Mesh Size : **Element Size : 1**
- 6 Property : “**2**”
- 7 Mesh Set – Add to : **[Block 2]**
- 8 Click **[OK]** Button



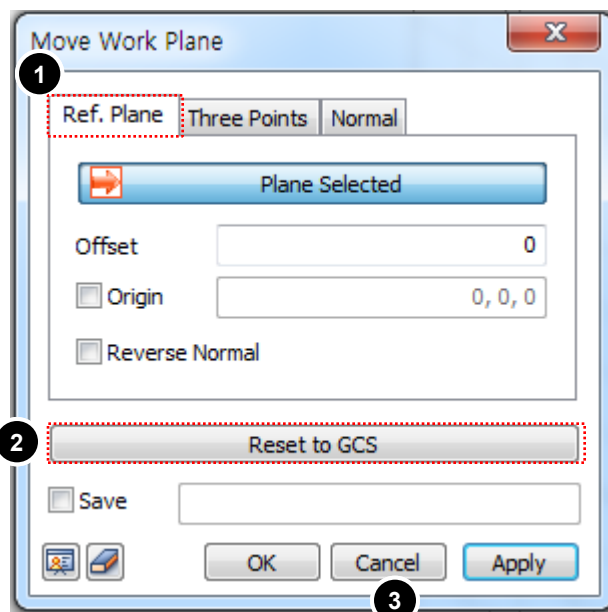
Procedure

- 1 Click Right Mouse Button and Select "Hide All"
- 2 Mesh > Transform > **Mirror...**
- 3 Select All "Displayed"
- 4 Mirror Type : "Plane"
- 5 Mirror Plane : "XZ-Plane"
- 6 Advanced Option > [Merge Nodes]
- 7 Click [OK] Button
- 8 Click [Apply] Button
- 9 Repeat "Step 3rd~6th for XY-Plane"
- 10 Click [Cancel] Button
- 11 Model Tree > Mesh > Drag [Merge Nodes] to Block
- 12 Click [OK] Button




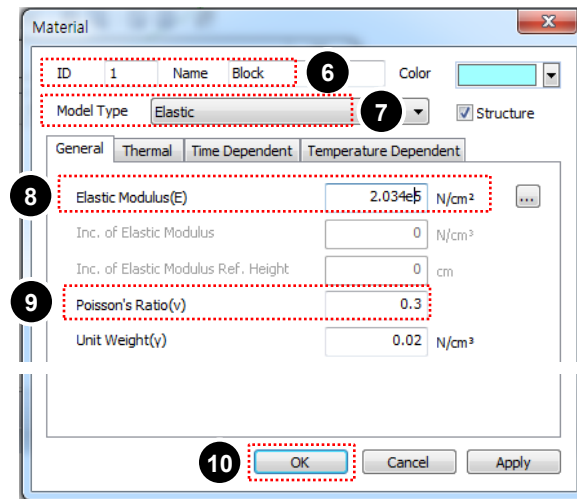
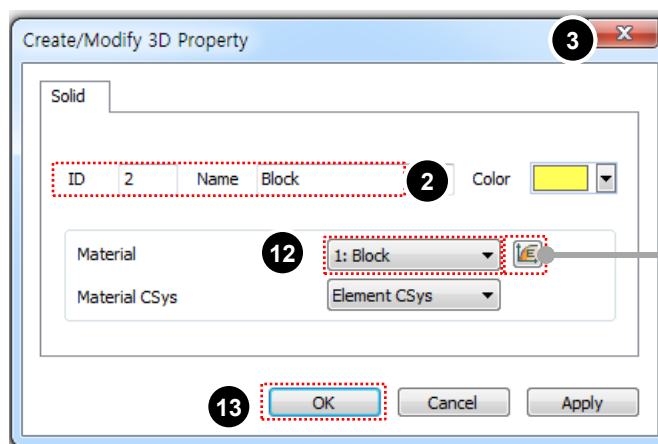
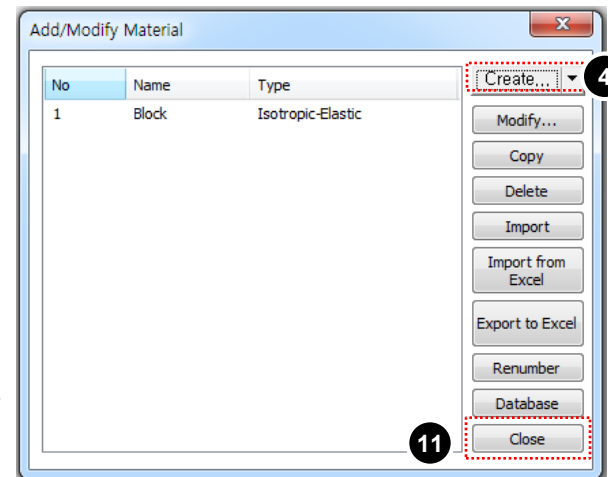
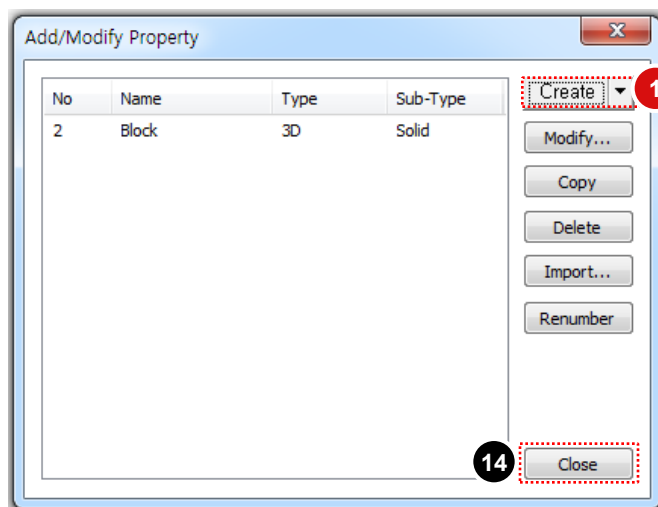
Procedure

- 1 Select : **“Ref.Plane”**
- 2 Click : **“ Reset to GCS ”**
- 3 Click **[Cancel]** Button





Procedure

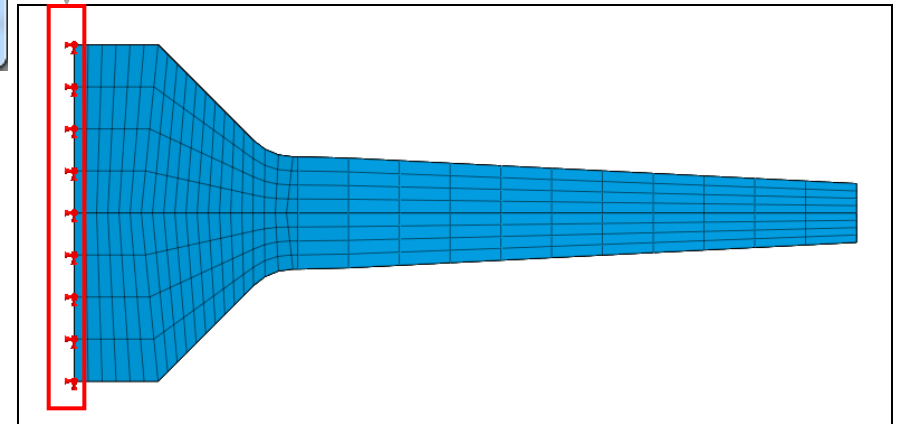
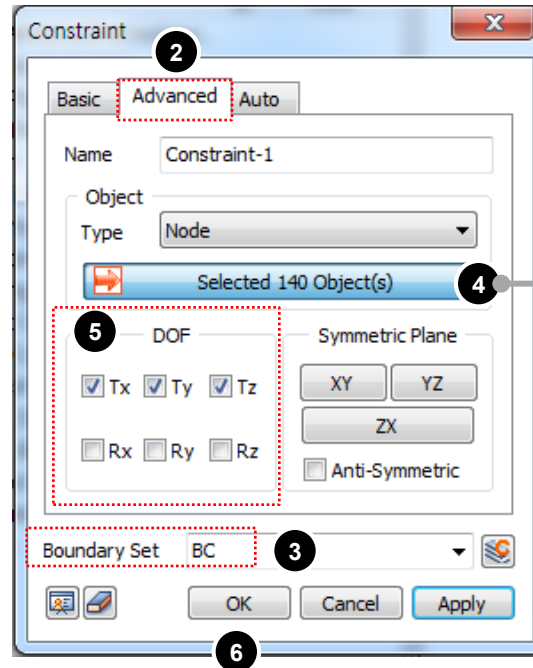
- 1 Create [3D]
- 2 ID : "2", Name : "Block"
- 3 Click  Button (Material)
- 4 Click [Create...] Button
- 5 Section : "Isotropic" tab
- 6 ID : "1", Name : "Block"
- 7 Model Type : "Elastic"
- 8 Elastic Modulus : $2.034e5 \text{ N/cm}^2$
- 9 Poisson's Ratio : "0.3"
- 10 Click [OK] Button
- 11 Click [Close] Button
- 12 Select : "1: Block" for Material
- 13 Click [OK] Button
- 14 Click [Close] Button



Procedure

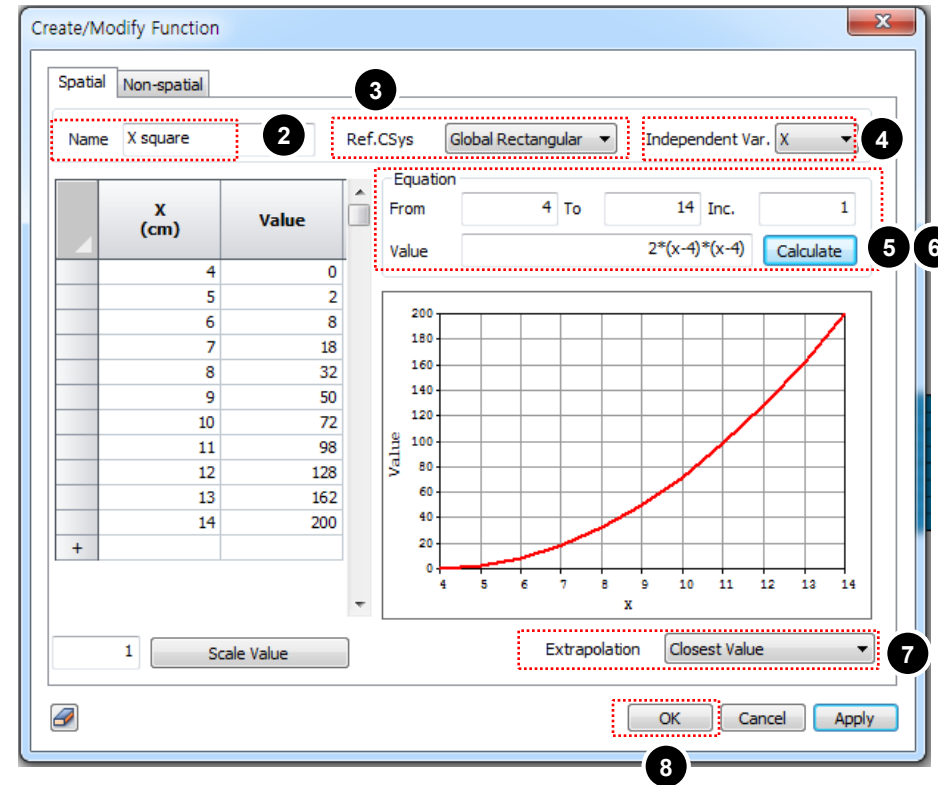
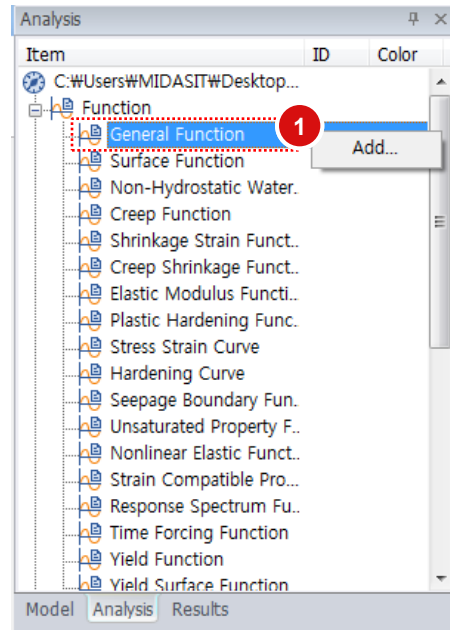
- 1 Click **"Front View"**
- 2 **[Advanced]** tab ...
- 3 BC Set : **[BC]**
- 4 Select 221 Nodes Marked by
"□" (Right Side, See Figure) 
- 5 DOF : **Tx, Ty, Tz**
- 6 Click **[OK]** Button

 Drag mouse to select element edges
(Window Selection)




Procedure

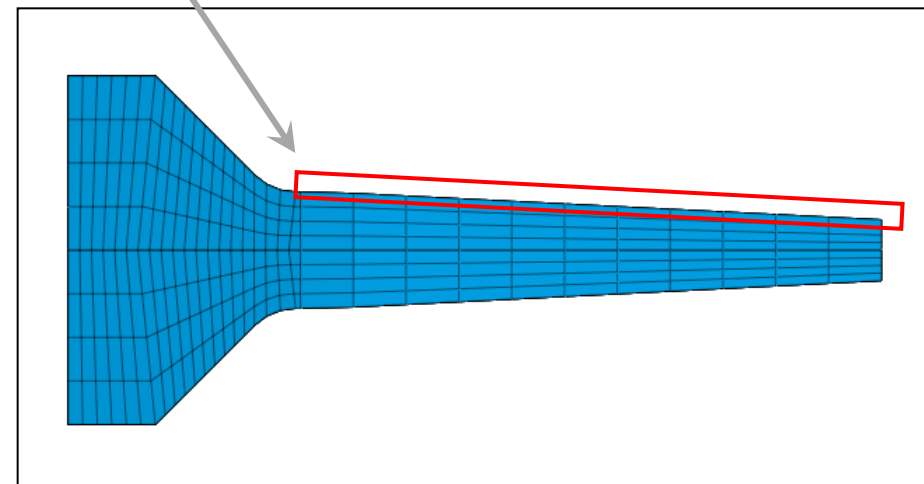
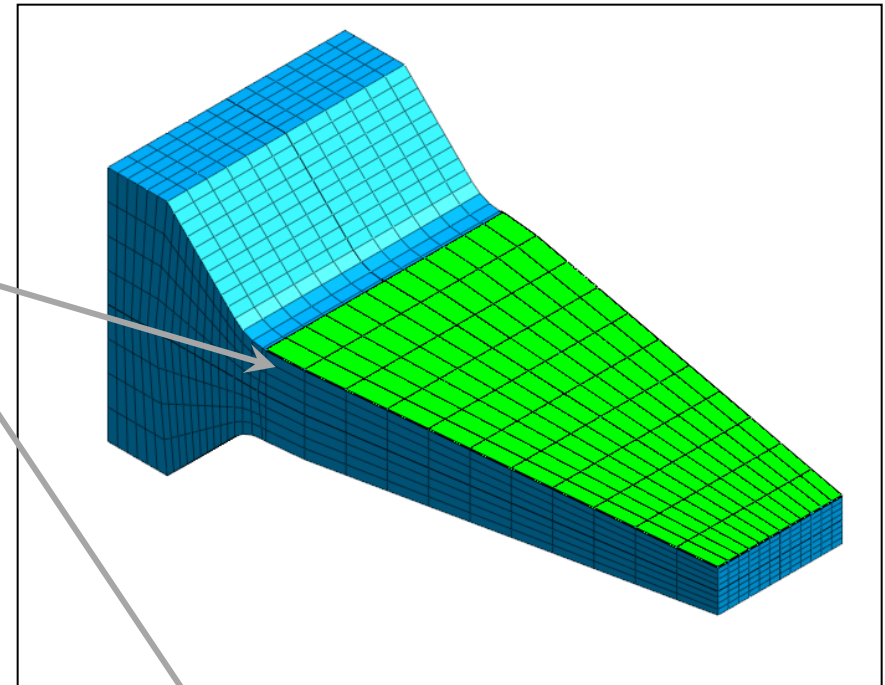
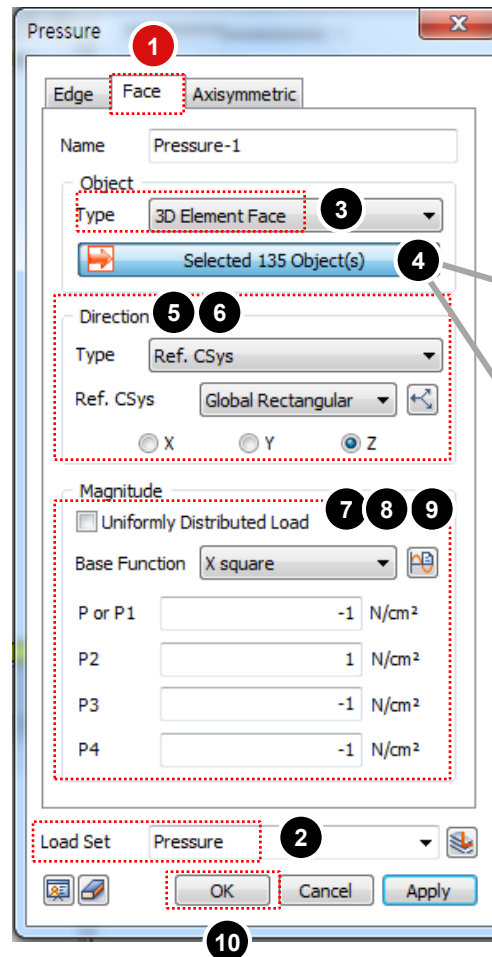
- 1 Click Right Mouse > **Add**
- 2 Name : **"X square"**
- 3 Ref. CSys : **Global Rectangular**
- 4 Independent Var. : **"X"**
- 5 From : **"4"**, To : **"14"**, Inc : **"1"**,
Value : **"2*(X-4)*(X-4)"**
- 6 Click **[Calculate]** Button
- 7 Extrapolation : **"Closest Value"**
- 8 Click **[OK]** Button



Procedure

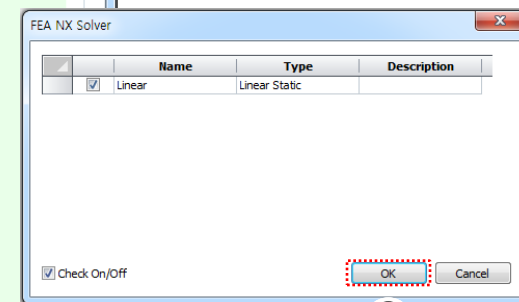
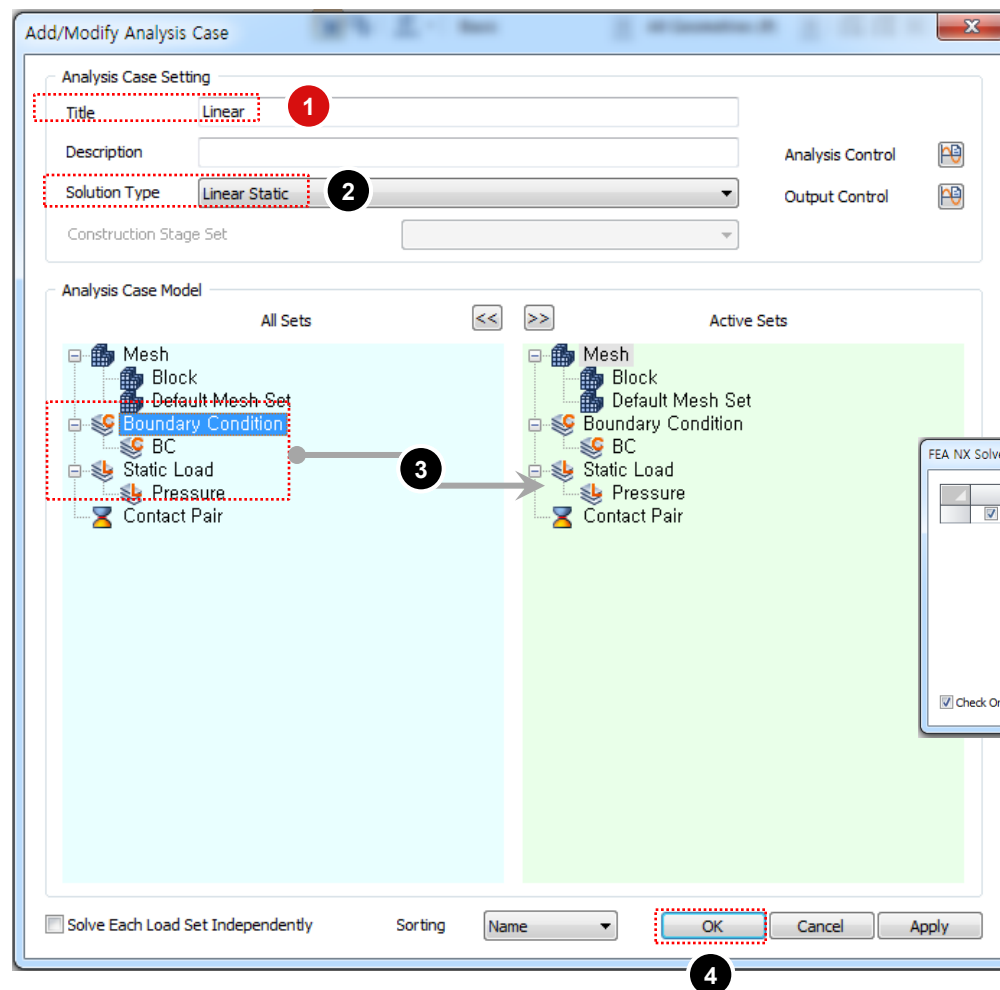
- 1 [Face] tab
- 2 Load Set : **"Pressure"**
- 3 Object Type : **"3D Element Face"**
- 4 Select 120 Element Faces Marked by **"□"** (See Figure) 
- 5 Direction : **"Ref. Csys-Z"**
- 6 Ref Csys : **"Global Rectangular"**
- 7 Check off : **"Uniform"**
- 8 P ~ P4 : **"-1"**
- 9 Select Base Function : **"X square"**
- 10 Click [OK] Button

 Use  "Polyline" Selection.



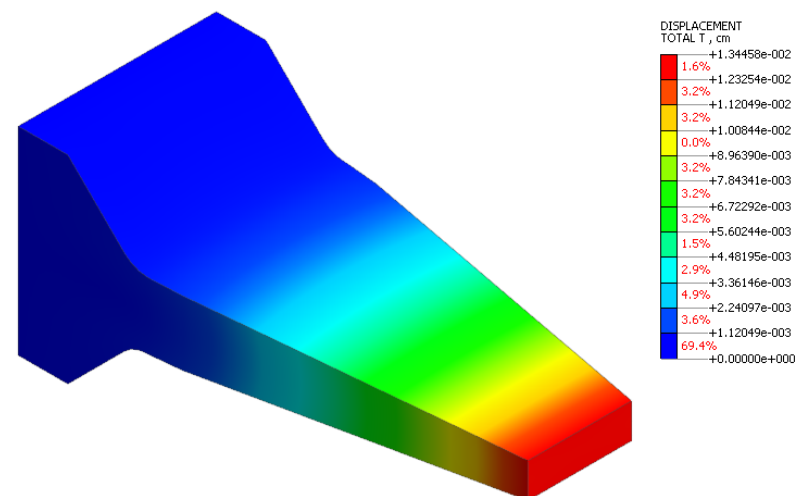
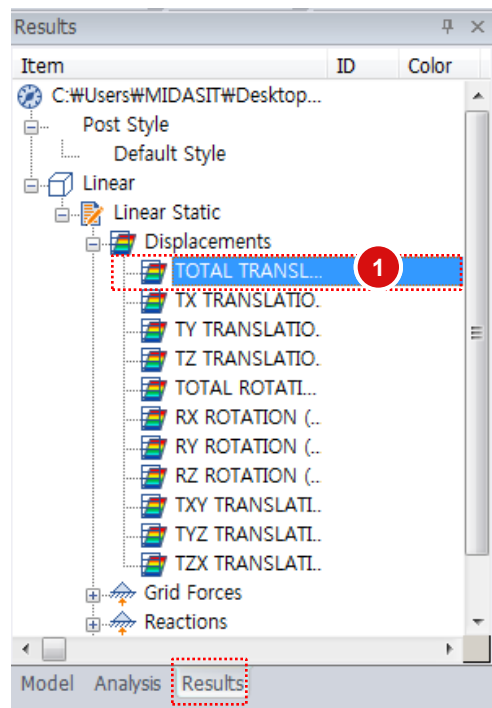
Procedure

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



Procedure

- 1 Double Click [TOTAL TRANSL...]



Procedure**1 [S-VON MISES]**