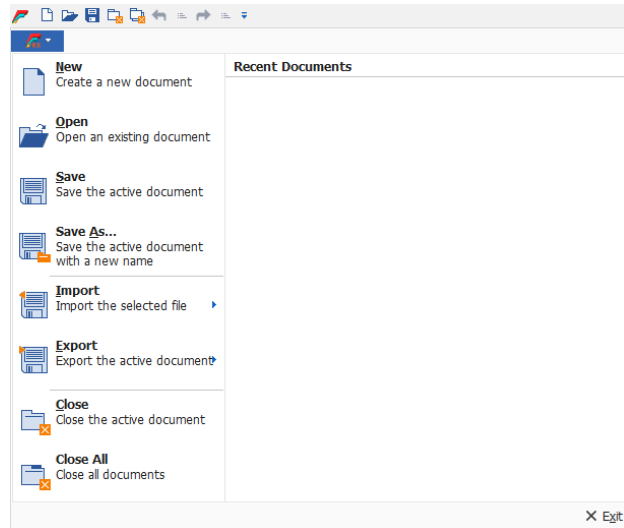




## Section 1 Start File

►Start file



### 1.1 Create File and Save

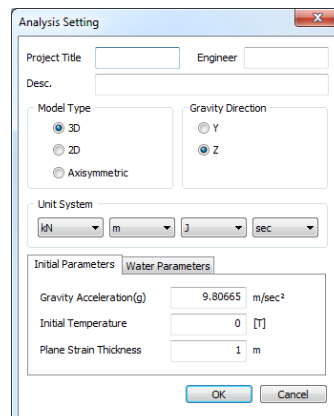
#### New

This tab creates a new file.

When creating a new file, the user can set the initial variables necessary for analysis (model type, direction and strength of gravity, system of units).

The required Model Type can be selected from 3 dimensional (3D)/ 2 dimensional (2D)/ axially symmetrical (Axisymmetric) models. For 2 dimensional models, the direction of gravity is the Y axis. For 3 dimensional models, the direction of gravity can be set as either the Y axis or the Z axis.

►Analysis condition setting



The physical quantities required for modeling, such as structure size or material property, are generally defined by specific unit systems. Because FEA NX allows conversion of force/ length/ energy/ time units any time during modeling, the user can easily change the unit system as and when needed thereby facilitating the modeling process. The unit systems supported in the FEA NX are as follows.

►Table. Unit systems supported on FEA NX

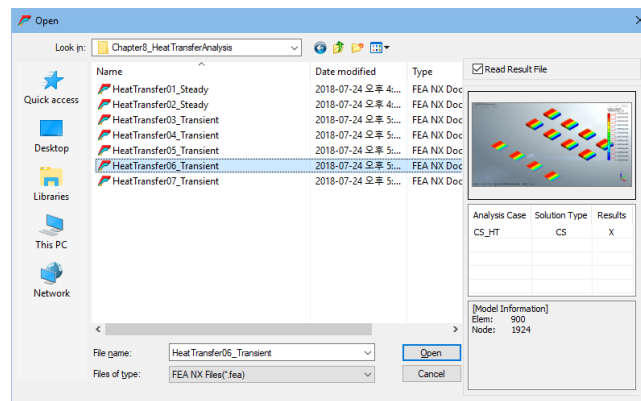
Force	Length	Energy	Time
kgf	mm	cal	sec
tonf	cm	kcal	min
N	m	J	hr
KN	in	Btu	day
lbf	ft	kJ	
Kips	Um		

### Open

This tab opens an existing saved file. The user can open both the old FEA and the new FEA NX file in FEA NX.

Depending on the file format, the user can check the image and basic information of the analysis model.

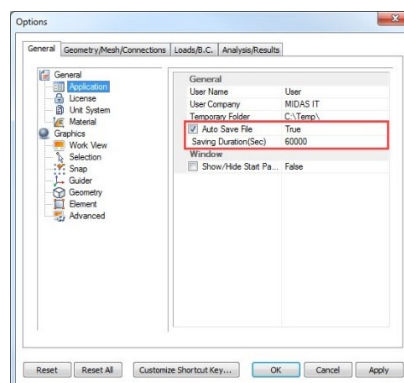
►Open file



### Save

This tab saves the current project file. Checking 'Auto Save File' and setting the time interval in the options tab automatically saves the file for every interval.

►Save file



### Save as

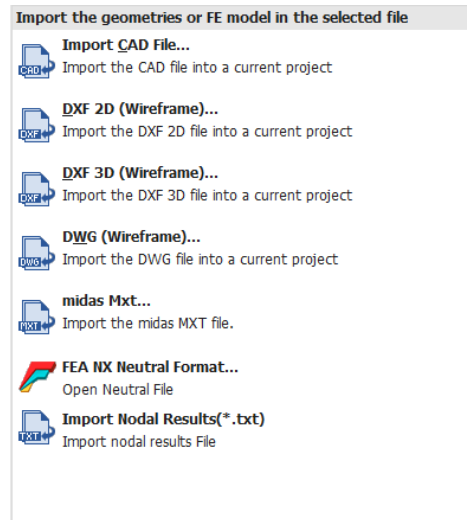
This tab saves the current project file under a new name. The user can undo/redo any operations during the modeling process, but saving the finalized model for each stage can make error correction and condition modification easier.



## 1.2 Import

► Import geometry or FE model

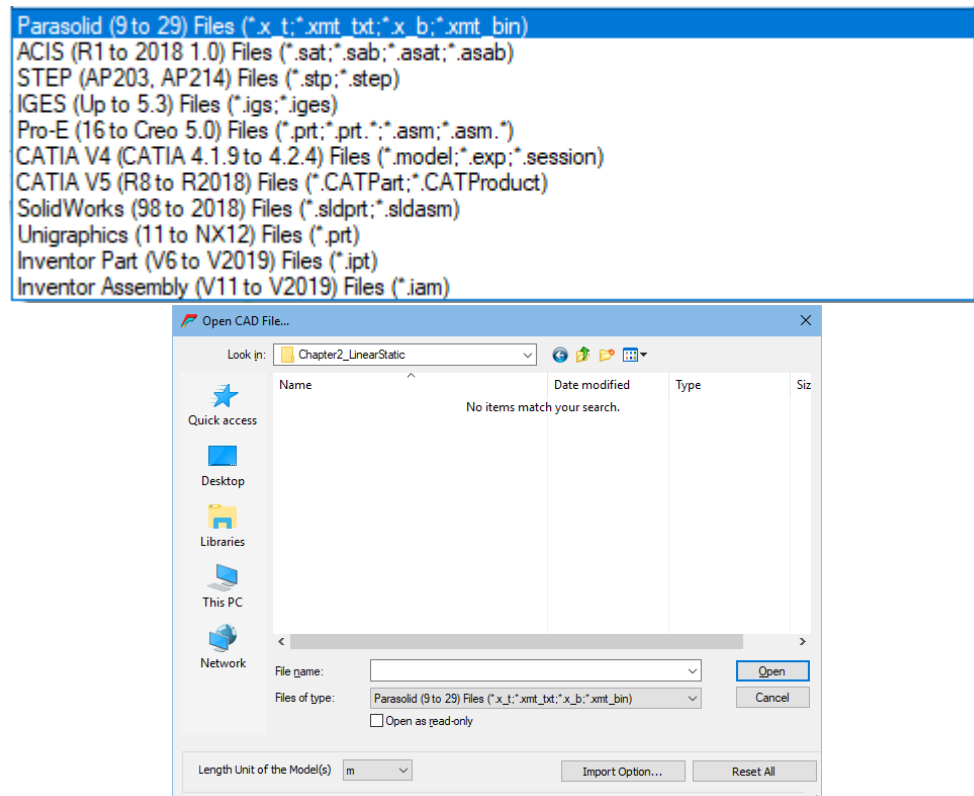
### Import CAD File



Using the Import CAD File automatically deactivates the analysis information of the current model. The user can use this function even when a file is not open and can import as many CAD geometries as they wish without limits. Parasolid (Parasolid(9 to 28) (\*.x\_t;\*.xmt\_txt;\*.x\_b;\*.xmt\_bin) file formats and optional CAD file formats as below are supported.

Because CAD geometries do not have a length unit, the user must set the length unit when importing. The set units decide the size of the CAD geometry and both the meter and feet units are supported.

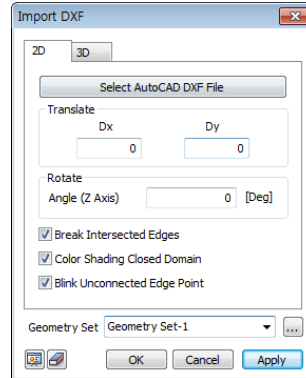
► Import CAD geometry





## DXF 2D (Wireframe)

### ►Import DXF(2D)



### Overview

This function imports a DXF file created on a higher version of AutoCAD than R13. The user can import 2D surfaces as well as wireframes (a model consisting of edges).

### Methodology

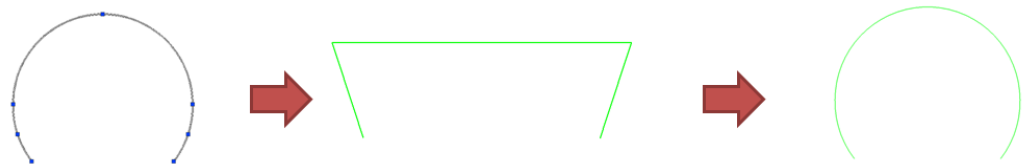
DXF files created on AutoCAD do not have tolerance that can determine the connectivity between endpoints. Although edges may seem connected to the naked eye, they maybe intersected or not connected after import. It is recommended that the user check the edges of the AutoCAD DXF file before importing.

►When the endpoints do not coincide



If a line is joined by multiple polylines, it may not be imported properly. It is recommended that these lines be exploded on AutoCAD before importing.

►Example of joined poly lines



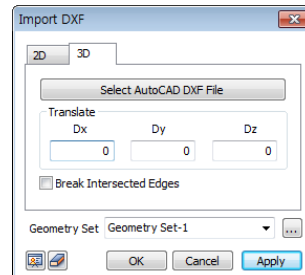
When importing a 2D DXF file, the user can simultaneously input distance from origin along x axis and y axis to move or rotate about z axis the imported file in order to place it at the desired location in the model.

Wireframes can be imported in FEA NX as a single composite line for easy management or as individual lines.

Imported intersected edges can be cut and registered using the [Break Intersected Edges] function.

## DXF 3D(Wireframe)

### ►Import DXF(3D)



### Overview

This function imports a DXF file created on a higher version of AutoCAD than R13. It imports a 3D model on to the work space. The user can call up DXF file as a wireframe (a model consisting of edges).

### Methodology

DXF files created on AutoCAD do not have tolerance that can determine the connectivity between endpoints. Although edges may seem connected to the naked eye, they maybe intersected or not connected after import. It is recommended that the user check the edges of the AutoCAD DXF file before importing

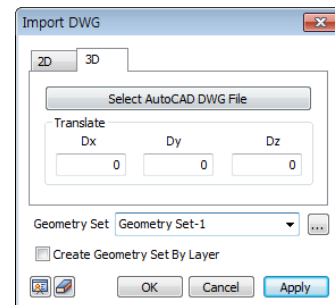
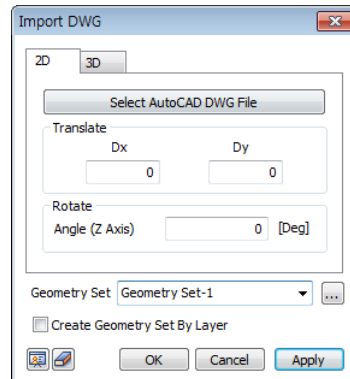
When importing a 3D DXF file, the user can simultaneously input distance from origin along x axis, y axis and z axis to place the imported file at the desired location in the model.

Wireframes can be imported on to the FEA NX as a single composite for easy management, or as individual lines.

Imported intersected edges can be cut and registered using the [Break Intersected Edges] function.

## Import DWG

- Import DWG(2D)
- Import DWG(3D)



### Overview

This function imports a DWG file created on a higher version of AutoCAD than R13. It imports a CAD model onto the 2D space or 3D space.

### Methodology

Like the Import DXF file function, DWG files created on AutoCAD do not have tolerance that can determine the connectivity between endpoints. Although edges may seem connected to the naked eye, they maybe intersected or not connected after import. It is recommended that the user check the edges of the AutoCAD DWG file before importing

When importing a 2D DWG file, the user can simultaneously input distance from origin along x axis and y axis to move or rotate about z axis the imported file in order to place it at the desired location in the model. While when importing a 3D DWG file, the user can simultaneously input distance from origin along x axis, y axis and z axis to place the imported file at the desired location in the model.

Imported edges can be registered on the geometry set for each layer using the [Create geometry set for layers].

**Import Midas Mxt file**

Model information and loading/ boundary condition/analysis data from the 'Gen, Civil, nGen' program, other Midas family programs, can be synchronized on the FEA NX. Element types, load/ boundary conditions, analysis methods not supported on the FEA NX are excluded.

The compatibility range for midas Mxt files is shown below.

FEA NX	Gen/Civil	Restrictions
Unit System	Unit System	Cable → Truss,
Project Setting	Structure Type	Wall → Plate element
Node	Node	compatible, SRC excluded
Element	Element	PSC, Composite excluded
Mesh Set	Group	
Material	Plastic Material, Material	
Property	Section, Thickness	
Supports	Supports	
Point Spring	Point Spring	
Elastic Link	Elastic Link	
Beam End Release	Beam End Release	
Plate End Release	Plate End Release	
Rigid Link	Rigid Link	
Load Set	Static Load Case	
Nodal Mass	Nodal Mass	
Self-Weight	Self-Weight	
Force, Moment	Nodal Loads	
Prescribed Displacement	Specified Displacement of Supports	
Element Beam Load	Element Beam Load	
Prestress	Pretension Load	
Pressure Load	Pressure Load	
Nodal Temperature	Nodal Temperature	
Element Temperature	Element Temperature	
Temperature Gradient	Temperature Gradient	
Response Spectrum Function	Spectrum Function	
Spectrum Load Set	Spectrum Load Case	
Time History Function	Time History Function	
Time History Load Case	Time History Load Case	
Dynamic Nodal Load	Dynamic Nodal Load	
Ground Acceleration	Ground Acceleration	
Time Varying Static Load	Time Varying Static Load	
Time History Result Function	Time History Result Function	

**Import FPN file**

This format links the model information file created on Midas family programs (FEA, GTS NX, GTS, NFX, SoilWorks, GeoXD) to the FEA NX. This file format also enables user to exchange information between different programs using text format.

**Import Nodal Results(\*txt) file**

This function allows users to import nodal results from midas Gen and Civil into FEA NX.

Displacement results will be imported as prescribed displacement (Tx, Ty, Tz).

Reaction results will be imported as nodal loads in the opposing direction (FX, FY, FZ, RX, RY, RZ).

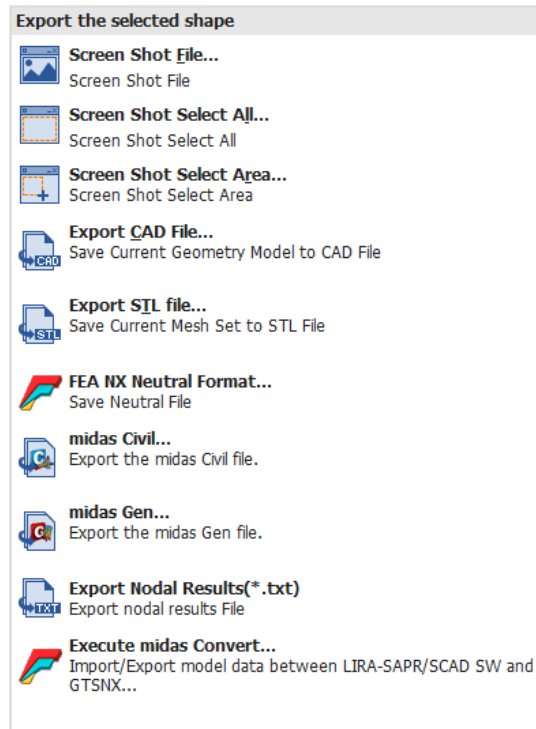


## 1.3 Export

### Overview

This function can capture the work plane or export geometric shapes and meshes.

#### ►Export work model



### Methodology

The work plane can be saved using [Screen Shot file], [Screen Shot Select All], [Screen Shot Selected Area] tabs.

The [Screen Shot file] tab only saves the work plane and the [Screen Shot Select All] tab saves the whole image. The [Screen Shot Selected Area] tab saves the area selected by the user. PNG, JPG, BMP, GIF image file formats are supported.

CAD geometric shapes and meshes on the work plane can be exported. CAD geometric shapes are exported as Parasolid files (\*.x\_t, \*.x\_b) and meshes can be exported as a STL file (\*.stl) or a neutral file (\*.fpm) supported on other Midas programs (FEA/GTSNX/GTS/NFX/SoilWorks) and non-Midas programs.

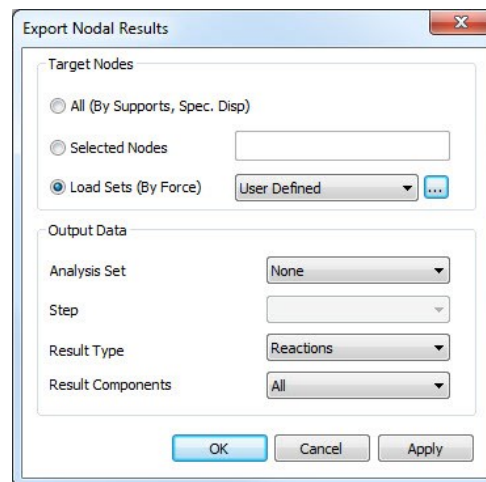
#### Export FPN file

This format links the model information file created on FEA NX to Midas family programs (FEA, GTS NX, GTS, NFX, SoilWorks, GeoXD). This file format also enables user to exchange information between different programs using text format.

#### Export Nodal Results

Export Nodal Results feature from FEA NX model to midas Gen / Civil allows for outputting the results of the reaction / displacement results calculated from the existing constraints as well as the load-displacement results of the nodes subjected to the nodal load.





#### Target Nodes

All (By Supports) – reactions and displacements will be exported at only the nodes with constraint boundary conditions defined according to the analysis set and step chosen by the user.

Selected Nodes – allows users to select specific nodes for output. Reactions will be exported at only the nodes with constraint boundary conditions defined according to the analysis set and step chosen by the user. Displacements will be exported at all selected nodes.

Load Sets (by Force) - reaction force / displacement will be exported at only the nodes where Load Set has been defined (as Force type only) according to the analysis set and step chosen by the user. User defined choice for specific load set is allowed.

In addition to the reaction force / displacement results, the "load-displacement" type was added to the output result type. The 'Load-displacement' type will simultaneously output the value of the nodal load considered in the load set and the displacement result calculated at the corresponding node after the analysis and will be loaded in Gen / Civil as Multi-Linear Point Spring (Unsymmetric) type.

The 'load' value is calculated as  $\Sigma$  (node load / load increment number) entered, and the cumulative displacement generated at the stage where the 'load' is applied is transferred to the 'displacement' data.

#### ※ Precautions

1) Gen / Civil> The Multi-Linear Point Spring (Unsymmetric) type requires a minimum of 3 data. Therefore, when outputting 'load-displacement' type in FEA NX, please perform analysis to satisfy the number of data.

2) Gen / Civil> The Multi-Linear Point Spring (Unsymmetric) type is designed to receive increasingly larger values of the load / displacement data. Therefore, if the displacement result calculated by the FEA NX is output as a decrease rather than as an increase, the Gen / Civil will not be able to recall the value.

#### Output Data

Analysis Set – select the analysis case for output.

Step – select the construction stage set for output.

Results Type – opt output reactions or displacement.



---

Results Component – select the components for output. Reactions (All, FX, FY, FZ, MX, MY, MZ).  
Displacements (All, DX, DY, DZ)

## 1.4 Close

### Overview

Close an activated file.

#### Close

This function closes the active project. If there are any changes to the model information within the project, a message will appear.

#### Close all

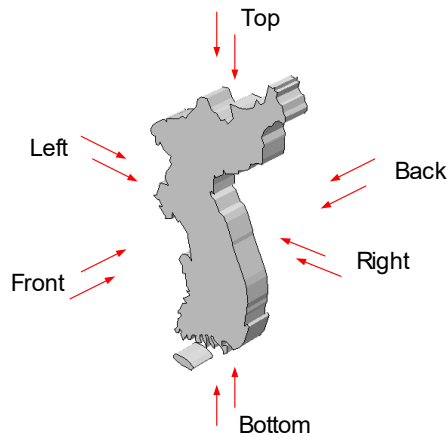
This function closes all open project files and a message will ask whether to save changes if there are any.








## Section 2 View Toolbar

















For general CAD programs, the y axis of the global coordinate system points upward. To view an imported geometric model created in CAD, the user can use the 'Two-point Isometric View' perspective to view the model in the same perspective as CAD.











### 2.1 View Toolbar

	<b>Zoom All</b> Zoom in/out to fit the whole model on the screen.
	<b>Zoomfit Grid</b> Zoom in/out to appropriately to fit both the model and the work plane grid.
	<b>Zoom Window</b> Zoom in to the designated rectangular area.
	<b>Zoom in/out</b> The user can click on the icon menu and use the left mouse button or the mouse wheel to zoom in/out. Dragging right while holding the left mouse button expands the model; dragging left shrinks the model. Scrolling upward zooms in and scrolling downward zooms out. Holding the Ctrl key allows the user to expand and shrink the model without having to select the zoom in/out icon.
	<b>Rotate</b> Click this icon menu and dragging the left mouse button on the task window rotates the model in the drag direction. For continued rotation, hold the Ctrl key and drag with the right mouse button.



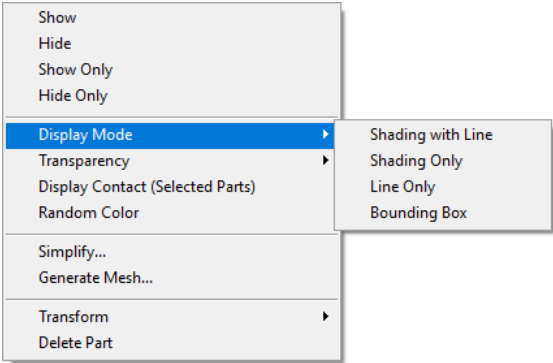
	<p><b>Rotate center</b></p> <p>Specify a reference point and rotates the model around that point. In the work plane, dragging with the middle mouse button held rotates the model around the specified reference point.</p>
	<p><b>Pan</b></p> <p>Move the model to desired location.</p> <p>In the work plane, dragging with the left mouse button held moves the model in the drag direction.</p> <p>Holding the Ctrl key and dragging with the middle mouse button held allows the user to move the model without having to 'Pan' icon.</p>
	<p><b>Normal</b></p> <p>Move the perspective in the normal direction of the working plane. (Perspective changed to a 2 dimensional view with the X-axis to the right and the Y-axis to the top of the screen.)</p>
	<p><b>Isometric 1</b></p> <p>Change the perspective to the isometric view. (Z-axis of the global coordinate system points up.)</p>
	<p><b>Isometric 2</b></p> <p>Change the perspective to the isometric view. (Y-axis of the global coordinate system points up.)</p> <p>When importing a geometric model created in CAD, use this perspective to view the model in the same perspective as CAD.</p>
	<p><b>Front</b></p> <p>Change the perspective to the front view.</p> <p>It is the Y, Z plane of the global coordinate system rotated 180 degrees about the Z-axis.</p> 
	<p><b>Back</b></p> <p>Change the perspective to the back view. It is the Y, Z plane of the global coordinate system.</p> 
	<p><b>Top</b></p> <p>Change the perspective to the top view.</p> <p>It is the X, Y plane of the global coordinate system rotated 90 degrees about the X-axis and the Y-axis.</p> 
	<p><b>Bottom</b></p> <p>Change the perspective to the bottom view.</p> <p>It is the X, Y plane of the global coordinate system rotated 90 degrees about the Y-axis and the X-axis.</p> 
	<p><b>Left</b></p> <p>Change the perspective to the left view.</p>

	<p>It is the X, Z plane of the global coordinate system rotated 180 degrees about the Z-axis.</p> 
	<p><b>Right</b></p> <p>Change the perspective to the left view. It is the X, Z plane of the global coordinate system.</p> 
	<p><b>Display mode [Geometry]</b></p> <p>Provides various view modes to display the model. <i>For more information, refer section 2.2</i></p>
	<p><b>Display mode [Mesh]</b></p> <p>Set the display modes for the created mesh object. <i>For more information, refer section 2.3</i></p>
	<p><b>Perspective view</b></p> <p>Assign perspective to the model and presents it as a three-dimensional picture.</p>
	<p><b>Geometry shadow</b></p> <p>Show a shadow of the model on the XY plane. The shading rotates along with the model.</p>
	<p><b>Bounding box</b></p> <p>Show the bounding box that circumscribes the model.</p>



## 2.2 Display Mode [Geometry]

FEA NX provides various display modes that help the user easily understand the work state and display the model as they please.  
The display mode can be called up by selecting the geometric shape and clicking the right mouse button.



### Shading + Line

Display the geometric shape using faces and outlines.

### Shading

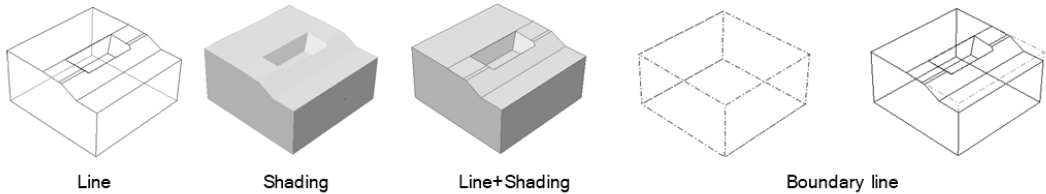
Display the geometric shape using only faces.

### Line

Display the geometric shape using only outlines.

### Bounding box

Display the geometry object by its bounding box than its actual shape.



### Tip

When displaying the geometric shape using the [Shading] or [Shading and Line] method, the user can specify the shading transparency to display the interior of the object.  
The transparency can be specified by selecting the geometric shape and clicking the right mouse button > Contents menu > Transparency.  
Complex geometric shapes can be effectively represented by applying different display modes to individual geometric shapes.

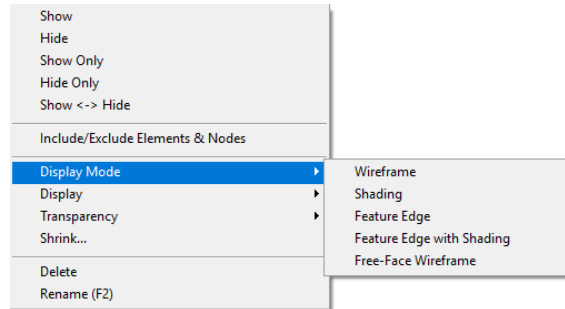


## 2.3

### Display Mode [Mesh]

The units for mesh display are mesh sets.

The assignment of display modes for meshes is the same that for geometric shapes; the mesh set is selected first and the display mode sub-menu is brought up from the pop-up menu.



Just like for geometric shapes, various display modes can be assigned to individual mesh sets. Color coding is especially useful when sorting and checking model information such as the object characteristics, materials or coordinate system.

Various mesh set display modes are listed below.

#### Wireframe

Display the mesh set as a wire frame, showing only edges and no faces.

#### Shading

Display the mesh set using faces and edges of all elements.

#### Feature Edge

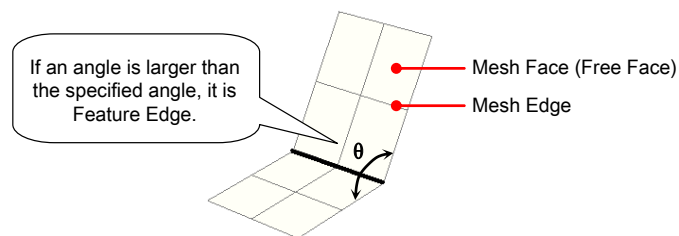
Display the mesh set using its feature edges. A feature edge is an element edge whose angle (created by the two free faces making up that edge) is larger than the specified angle.

#### Feature Edge with Shading

Display the mesh set using faces (without the sides) of elements and feature edges.

#### Free-Face Wireframe

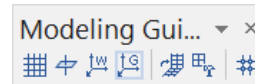
Display the free faces of the mesh set as wire frames. All internal elements are hidden.













## Section 3

## Modeling Guide Toolbar



## 3.1

Modeling Guide  
Toolbar

	<b>Show/Hide Grid</b> Show or hide the grid on the screen.
	<b>Show/Hide Datum Axis/Plane</b> Show or hide the datum axis/plane on the screen.
	<b>Show/Hide WCS</b> Show or hide the WCS on the screen. The WCS (Work-plane Coordinate System) is the coordinate system of the work-plane. Work-plane is user-defined plane to facilitate modeling
	<b>Show/Hide GCS</b> Show or hide the GCS on the screen. The GCS (Global Coordinate System) is the fixed global coordinate system. <i>For more information, refer section 3.2</i>
	<b>Move Work Plane</b> Move the work-plane to the desired location. This can be done by [Reference plane], [3 point plane] or [Normal direction] methods. <i>For more information, refer section 3.3</i>
	<b>Define Grid</b> The grid is always located on the XY plane of the work-plane to ease the modeling process. When modeling with the grid, the grid snap (  ) function can be used to specify the desired location and easily estimate the approximate model or element sizes. The grid setting can be set according to the convenience of the user and the dimensions of the model. <i>For more information, refer section 3.4</i>
	<b>Define Snap</b> The user can specify the location of the point using various snap options shown in the table below. <i>For more information, refer section 3.5</i>



## 3.2 Show/Hide GCS

The user can show or hide the GCS.

The GCS (Global Coordinate System) is the fixed global coordinate system.

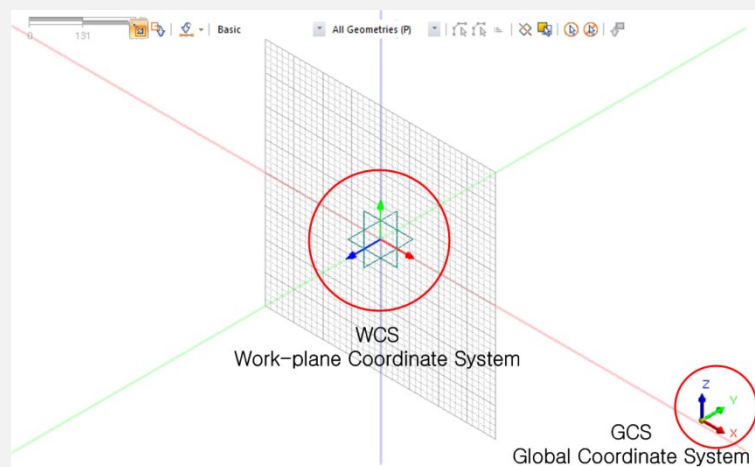
### Tip

There are two basic coordinate systems used in the GTX NX; The Global Coordinate System (GCS) and the Work-plane Coordinate System (WCS).

The GCS (Global Coordinate System) is the fixed global coordinate system expressed in the bottom right-hand corner using red (X-axis), green (Y-axis) and blue (Z-axis) arrows.

The WCS (Work-plane Coordinate System) is the coordinate system that moves with the work-plane and can be found at the center of the work screen. Because the work-plane is used for entering the 2-dimensional coordinates of a shape, the WCS changes along with the workplace. The absolute 3-dimensional coordinates are necessary to create a shape in space, but in most cases the only relative coordinates such as the model length are given. In this case, modeling can be done easily by moving the work-plane to an appropriate position and then entering the 2-dimensional coordinates (the XY plane on the WCS).

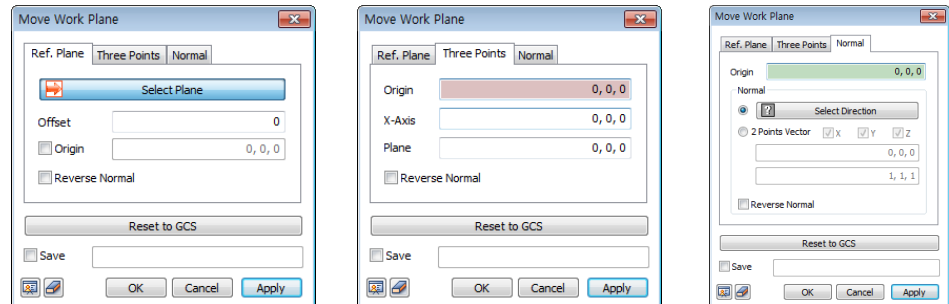
Please note that when extruding the geometric shape, the load/boundary conditions and extrude direction follows the GCS.



< Global Coordinate System(GCS) and Work-plane Coordinate System(WCS)>

### 3.3 Move Work Plane

#### Move work plane



This function moves the current work-plane to the desired location. This can be done by [Reference plane], [3 point plane] or [Normal direction] methods.

[Reference plane]: This function moves the grid to a plane parallel to the reference plane. Clicking the [Normal] button after moving the grid helps the user work more easily. This function is convenient when working on a plane different from the specified work-plane.

FEA NX provides 7 basic work-planes: XY(0,0,1), XZ(0,-1,0), XZ(0,1,0), YX(0,0,1), YZ(1,0,0), ZX(0,1,0), ZY(1,0,0).

If the workspace is a certain distance from the plane, the user can specify the grid origin using the [Offset] function.

[3 point plane]: This function moves the work-plane by selecting 3 points. The work-plane moves to the plane defined by the selected points, with the vector created by the first and second points the X-axis and the vector created by the first and third points the Y-axis.

[Normal direction]: This function moves the work-plane by selecting a vector and a reference point. The reference point is defined as the origin and the normal direction to the vector is defined as the vertical axis.

#### Reverse Normal

Reverse the vertical direction (the Z-axis of the WCS) of the plane.

#### Reset to GCS

Return to the initial grid position.

#### Save


Check the [Save] button and enter a name to register the workplace under Work tree > Work-plane.

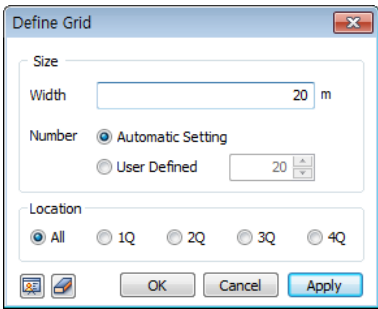


### 3.4

#### Define Grid

#### Define grid

The grid is always located on the XY plane of the work-plane to ease the modeling process. When modeling with the grid, the grid snap (  ) function can be used to specify the desired location and easily estimate the approximate model or element sizes.



### 3.5

#### Define Snap

The grid setting can be set according to the convenience of the user and the dimensions of the model.

<b>Grid snap</b>	Positions the mouse snap on a grid point in the work-plane.
<b>Point snap</b>	Positions the mouse snap on a point.
<b>End snap</b>	Positions the mouse snap on the closest endpoint of an edge.
<b>Middle snap</b>	Positions the mouse snap on the midpoint of an edge.
<b>Perpendicular snap</b>	Positions the mouse snap on the perpendicular point of an edge.
<b>Center snap</b>	Positions the mouse snap on the center point of a circle/arc.
<b>Quadrant snap</b>	Positions the mouse snap on the four circle/arc quadrant points.
<b>Intersection snap</b>	Positions the mouse snap on the intersection point between two edges.
<b>Tangent snap</b>	Positions the mouse snap on the tangent point.
<b>Arbitrary snap</b>	Designates an arbitrary snap.
<b>Node</b>	Positions the mouse snap on a node.

Tip

When using a snap related to a particular edge, such as end snap or middle snap, the user needs to position the mouse above the target edge of the snap, rather than on the position of the target (end, middle).

## Section 4 Select Toolbar

The select tools are essential for the overall modeling process and are used to specify the target for various functions. When working with large and complex models, selection of geometry or model components can be made very simple by using appropriate select tool. FEA NX provides the user with a variety of effective selection methods for efficient work progress.




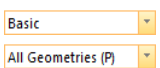








All select methods in the FEA NX are provided in the select toolbar.



When selecting an object in the work-plane using the mouse, the boundary of the detected object is displayed in green and the user can check in advance whether the target object is correctly detected. The selected target is then highlighted in green.

## 4.1

### Select Toolbar

	<b>Pick/Rectangle Select</b> Sets the mode to select the specified object
	<b>Unselect Mode</b> Sets the mode to unselect the specified object
	<b>Select Screen</b> Selects/Unselects individual objects or objects within an area set by dragging the mouse. There are four ways to select an area; rectangular, ellipse, polygon and polyline. <i>For more information, refer section 4.2</i>
	<b>Selection Filter</b> Primarily filters the selected objects by their specified types. <i>For more information, refer section 4.3</i>
	<b>Select All Neighboring Edges or Surfaces</b> Automatically selects adjacent elements (faces/lines) after selecting a face or line.
	<b>Select Neighboring Edges or Surfaces with Feature Angle</b> Automatically selects adjacent elements (faces/lines) within the feature angle after selecting a face or line.
	<b>Limit Feature Angle</b> Limits the feature angle for selecting adjacent lines or faces.
	<b>Intersect</b> Includes objects within the borderlines of an area created by a rectangle, circle or polygon.
	<b>Front Selection Only</b> Selects only the objects at the front in the current view. It is used when the object consists of multiple layers and the user wants to select one face.
	<b>Select All</b> Selects all objects in the current work space. In the view all perspective, all objects in the current work space are selected.
	<b>Unselect All</b> Unselects all objects in the current work space. In the view all perspective, all objects in the current work space are unselected.
	<b>Select Node/Element by ID</b> Selects or unselects objects by entering the ID in the node, element select mode.



## 4.2 Select Screen

This function selects/unselects individual objects or objects within an area set by dragging the mouse. There are four ways to select an area; rectangle, circle, polygon and polyline.

### Rectangle (Pick/Window)

Select/unselect all current selected objects.

Individual objects or objects within an area set by dragging the mouse can be selected or unselected.


The 'Pick' and 'Window' are distinguished only by mouse click or mouse drag.

#### [Pick Select]


Select/unselect objects by individual clicks.

In the select mode, an already selected object can be unselected by selecting pick again.

#### [Window Select]

Select/unselect objects within a rectangular area created from a diagonal movement by mouse drag. 

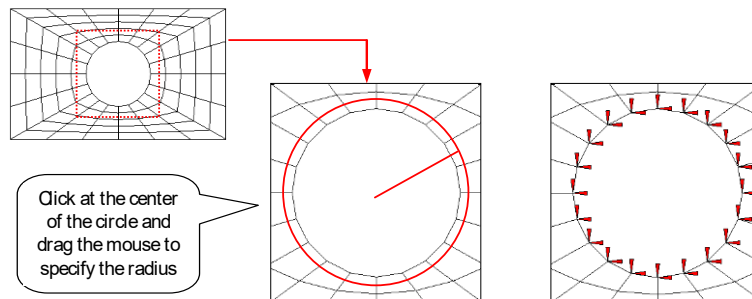
The Esc key cancels the definition of the rectangle during mouse drag.

 The direction of the mouse drag during Window Select is not important. To include/exclude objects is determined only by the Include Intersected option.

### Circle

Select/unselect objects within a circular area.

The area is defined by a left mouse click at the center of the circle and a mouse drag that determines the size of the radius. The Esc key cancels the definition of the circle during mouse drag.



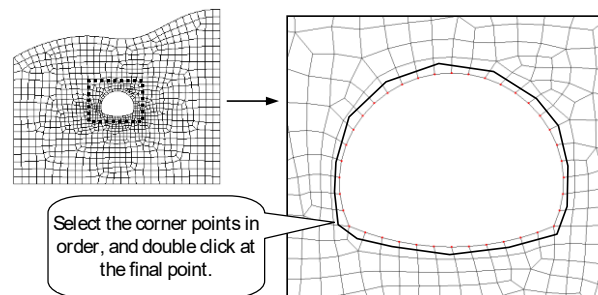
Example of specifying constraints of nodes around the hole by using Circle Selection.

### Polygon

Selects/unselects objects within a polygon.

The polygon is defined by clicking consecutive points and double-clicking the left mouse button at the final point. The area is defined by the connected start and end points.

Clicking the right mouse button cancels the definition of the polygon.

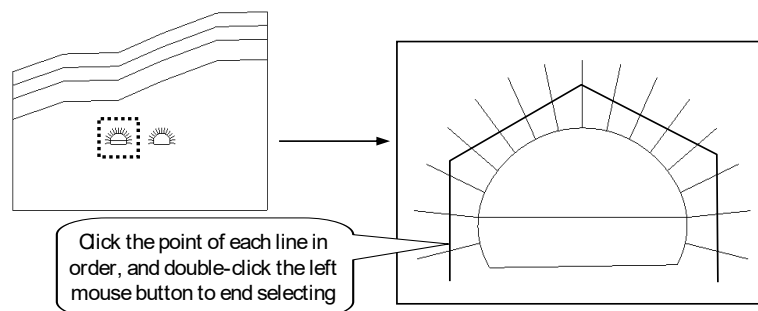


Example of selecting nodes around the tunnel by using Polygon Selection.

### Polyline

Selects/unselects objects intersected by a polyline.

To define multi-line, the user can specify the polyline by clicking consecutive points and double-clicking the left mouse button at the final point. Clicking the right mouse button cancels the definition of the multi-line.



Example of selecting lines representing rock bolt by using Polyline Selection



## 4.3 Selection Filter

🔊 The datum is the reference point, axis, or plane for modeling.

🔊 Because the selection filter composition is different for every command, it is recommended that the user check the select filter composition when using a command for the first time.

►Table. The selection difference between the Main select filter and the Sub select filter

The Selection filter primarily filters the selected objects by their specified types.

The Selection filter is composed of an object type predetermined by the current work status and the summoned command types. The user can specify a particular object type in this configuration.

The Selection filter-related work status is basically classified by the command execution viability.

### Main selection filter

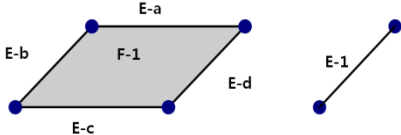
The Main selection filter has the most basic default composition.

Only geometric elements (sub-objects not selected), mesh sets and datum 📍 registered on the shape set can be selected in the Main selection filter.

### Sub selection filter

The Sub selection filter is restructured into types based on command execution viability.

Geometric elements registered in the shape set can be selected, as well as the sub-objects of the shape (point, edge, face etc.), loadings, boundaries and individual joints and elements. 📍

		Shape	Type
	Shape	F-1	Face (Four Subdivision Edges)
		E-1	Edge
Shape	Selection filter	Selectable objects	
Face	Main selection filter	1 (F-1)	
	Sub selection filter	1 (F-1)	
Line	Main selection filter	1 (E-1)	
	Sub selection filter	5 (E-1, E-a, E-b, E-c, E-d)	



The Selection filter composition is as follows.

► Table. Selectable elements in the Main selection filter

Selection filter	Selectable objects
<b>Data axis (X)</b>	Select the data axis. ► Used to specify the direction (Translate/Extrude/Project), rotation axis etc.
<b>Data plane (U)</b>	Select the data plane. ► Used to specify the work/reflect/divide/mirror plane etc.
<b>Geometry related</b>	
<b>All geometric shape (P)</b>	Select the "Shape", the highest ranking geometry.
<b>Solid (D)</b>	Select a solid.
<b>Shell (J)</b>	Select a shell.
<b>Face (A)</b>	Select a face.
<b>Wire (W)</b>	Select a wire.
<b>Edge (G)</b>	Select an edge.
<b>Point (B)</b>	Select a point.
<b>Complex object (C)</b>	Select a complex object.
<b>Mesh set / joint / element related</b>	
<b>Mesh set (M)</b>	Select a mesh set ► Used to select a mesh set itself or the joints and elements in the mesh set.

► Table. Selectable elements in the Sub selection filter

<b>Geometry related</b>	
<b>Face (F)</b>	Select a face, a lower ranking geometry of the "Shape".
<b>Line (E)</b>	Select a line.
<b>Point (V)</b>	Select a point.
<b>Load/Boundary (L)</b>	Select the load/boundary conditions applied on the geometry.
<b>Mesh set / joint / element related</b>	
<b>Node (N)</b>	Select a node.
<b>Element (T)</b>	Select an element.

## Tip

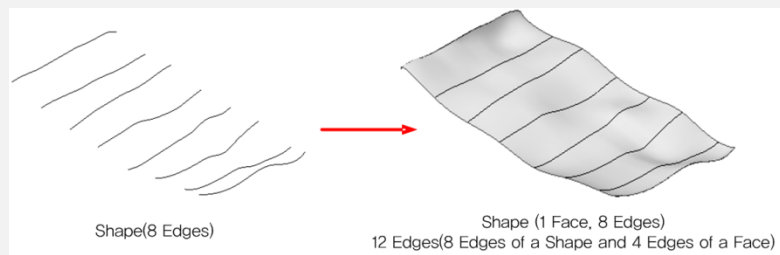
## \* Difference in selection between the Main selection filter and the Sub selection filter

The user may be unfamiliar to the difference in selected objects between the Main selection filter and the Sub selection filter, where only the highest ranking geometry "Shape" is selected for the Main selection filter, but this function is useful during modeling.

The following example will explore the convenience of using the Main selection filter.

The figure below shows a Face, interpolated by Edges created from the cross sections of a random 3-dimensional curved face.

To hide (or remove) the eight Edges that are no longer useful after the face has been created, let us consider case when only the eight Edges are selected.



- 1) When the selection of all geometry is allowed in the Main selection filter (not just "Shape")  
When using the [Rectangle selection] with the selection filter set at 'all geometric shape', nine "Shapes" (one Face, eight Edges) are selected.
- 2) When using the [Rectangle selection] with the Main selection filter set at <Edge>  
A total of twelve Edges are selected, four of which are the lower rank Edges of the Face and cannot be hidden individually.
- 3) When allowing only the "shape" selection in the Sub selection filter  
When using the [Rectangle selection] with the selection filter set at <Edge>, the eight Edges can be easily selected and can be hidden directly.

Thus, the Main selection filter and Sub selection filter are separated to provide convenient selection during actual work.

Manual selection is possible, but because the "Shape" Edges and the lower rank Edges of the Face exist in the same position, it is not easy to manually select individual edges.

In practical use, the user can take advantage of the work tree to make this work much easier.



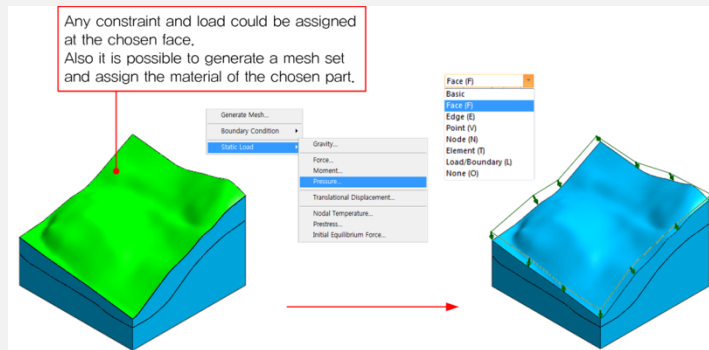
Tip

✱ Selection filter use

- Command execution mode using the Sub selection filter

🔊 To select nodes or elements based on geometric shapes, the object must be shown on the screen.

The direct constraints and loading conditions can be defined on the work screen using Sub selection filters. 🗨 Also, the parts can be specified as selection elements, making it possible to assign materials or mesh sets. The Sub selection filter selects the sub-objects of a shape (point, edge, face etc.), as well as joints and elements. Application of the Sub selection filter on the work screen helps the user easily define properties using geometric features without a separate command execution mode process.

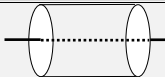
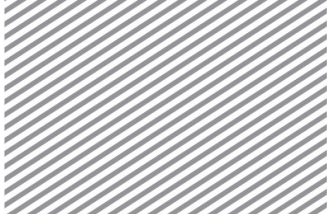


- Specifying the direction and axis of rotation

Setting the direction for Translate, Extrude or Project, as well as the axis of rotation for Rotate and Revolve, is often needed during the modeling process. In this case, the direction and axis of rotation can be specified by various methods using the Datum and the geometric features of the shape.

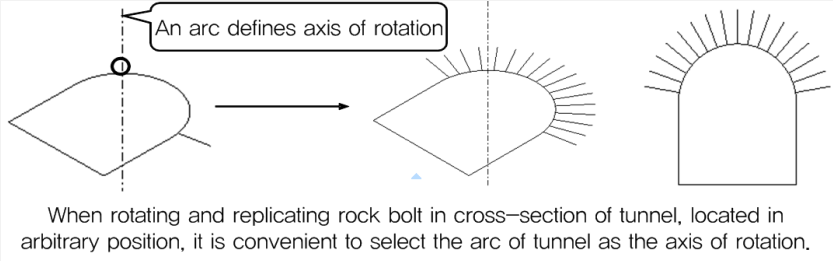
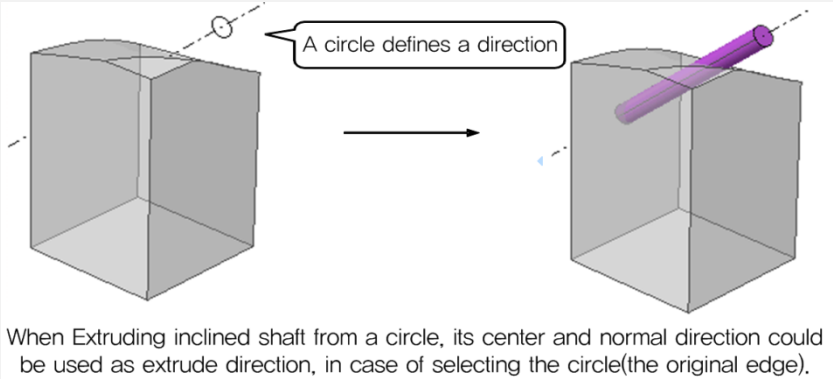
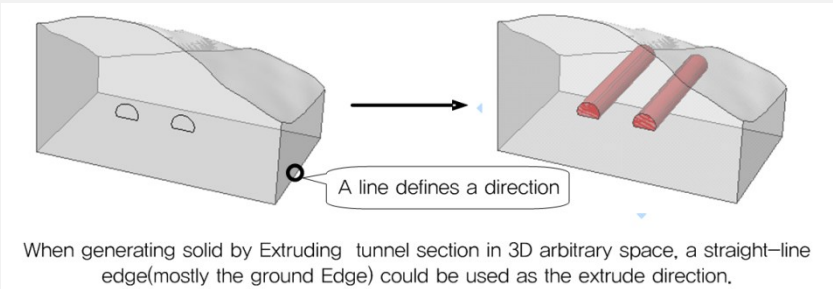
The following is a summary of the methods used to specify the rotation axis direction.

Available object	Used geometric features	Selection filter
Datum axis ①	Direction of Datum axis	Datum axis
	"Normal" direction to the Datum Plane	
Datum plane		Datum plane
Edge (Straight line)	Direction of straight line	Edge
	"Normal" direction to the plane where the circle/arc and its center point exist	
Edge (Circle or arc)		Edge
Plane surface	"Normal" direction to the plane	Face
Rotated body face	Central axis of the rotated body (cylinder etc.)	Face



2 point vector	User specified 2 point vector	N/A
Normal line to the profile ②	"Normal" direction to the Profile, whose "plane" definition allows Extrude etc.	N/A

- ① The selection filter for specifying direction, axis of rotation is fundamentally set as <Datum axis>.
- ② This is only restrictively supported for some functions such as Extrude.



The use of such selection methods that utilizes the geometric features of objects allows the convenient specification of direction and axis of rotation, even for very complex models.

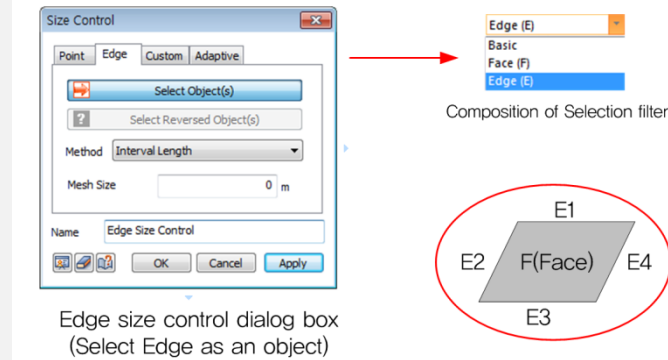
- Selection using the concept of hierarchy for geometric shapes

When selecting lower rank objects using the concept of hierarchy, selecting a higher rank object provides the selection functions for all lower rank objects (sub-objects that correspond to the type in selection filter) within it.

In the figure below, the lines need to be selected individually when the Sub selection filter is set as Edge (E). However, setting the Sub selection filter as a Face (F) and selecting the face selects all 4



lower rank lines.



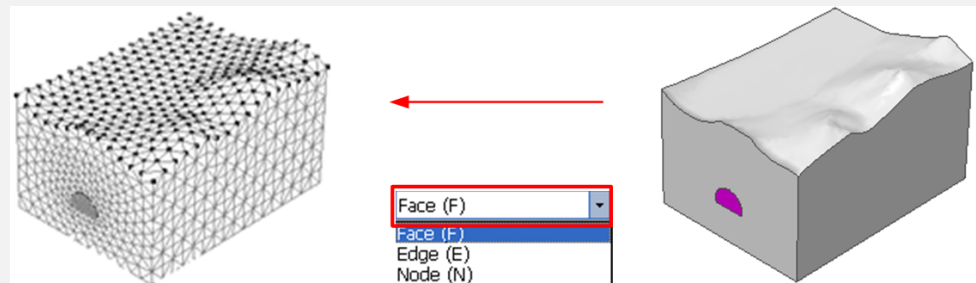
In other words, when selecting all the lower rank objects in a particular object, it is convenient to select the higher rank object first and then set the [Sub selection filter] to that higher rank object type.

- Node, Element Selection

Similar to the concept of hierarchy, when selecting nodes or elements, setting the selection filter to Mesh Set (M) and selecting the Mesh Set selects all the nodes and elements within the mesh set.

If the mesh is created on a geometric model, the user can choose the nodes and elements based on the geometric shapes (line, face) that form the mesh. 🗨

🗨 To select nodes or elements based on geometric shapes, the object must be shown on the screen.

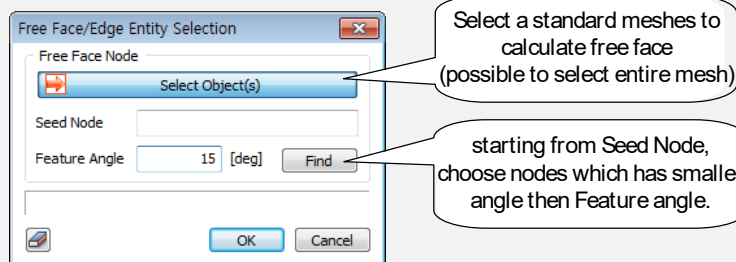


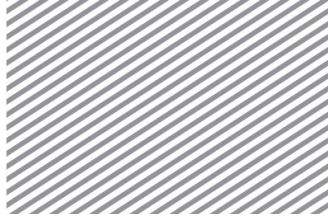
- Free face / free end-node selection function using the feature angle for nodes and elements

The loading, boundary condition dialog box provides additional functions that consider individual loading and boundary conditions for easier selection.

The most useful selection functions are for the adjacent Free Face of a 3D element and adjacent Free End of a 2D element. 🗨

🗨 Because this selection method is not related to the geometric shape, it is more convenient for meshes created by hand than those created using geometric shapes.



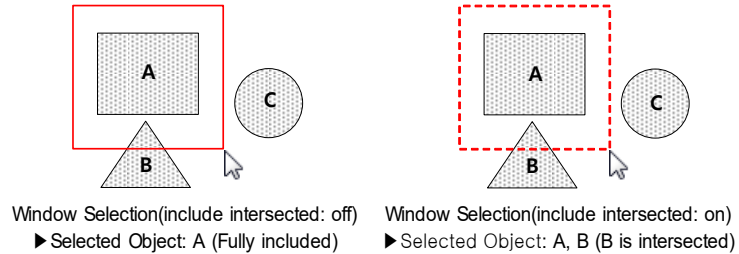


## 4.4

### Intersect

🔔 The borderlines of the area are displayed in dotted lines for the Intersection selection function. Pressing and releasing the Ctrl key when drawing the rectangle, circle or polygon switches to the Intersection selection option.

This function selects objects that are within the borderlines of an area created by a rectangle, circle or polygon. 📍



## 4.5

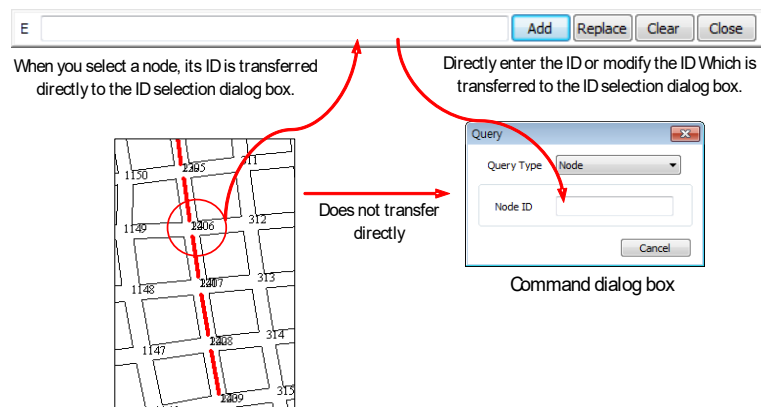
### Select Node/Element by ID

🔔 The ID selection dialog box acts as a middleman between the Selection manager and the Command dialog box.

This function selects/unselects objects by entering the ID in the node, element select mode.

This selection method can be coupled with other selection methods. The node or element select mode is significant when the selection filter is set as "Node (N)" or "Element (T)" and can only be used in the node, element control functions (functions that require direct selection of the node/element). When ID selection is called up, a dialog box appears on the upper right corner of the work screen and the selection process is as follows.

- ① Select individual nodes and elements by directly entering their ID in to the ID selection dialog box.
- ② Selecting nodes and elements using other methods (Pick/Window, Circle etc.) in the work-plane transfers the node/element ID in to the ID selection dialog box. The user can check the transferred ID and edit to select the desired nodes and elements. 📍 It is useful to use the ID selection function when selecting some of the overlapping nodes or elements.



Example of selecting specific node by using ID Selection, at where nodes are overlapped

The ID selection dialog box buttons are as follows.

🔔 When selecting new nodes, elements from the work-plane after deleting all content in the ID field, the existing selection is still shown in the ID selection dialog box.

<b>Add</b>	Adds a new node, element ID to the selection.
<b>Change</b>	Releases all of the existing selection and only selects the nodes, elements with new ID. (Select substitution)
<b>Delete</b>	Deletes all content in the ID field. 📍
<b>Close</b>	Closes the ID selection dialog box.



## Section 5 Advanced View Control Toolbar

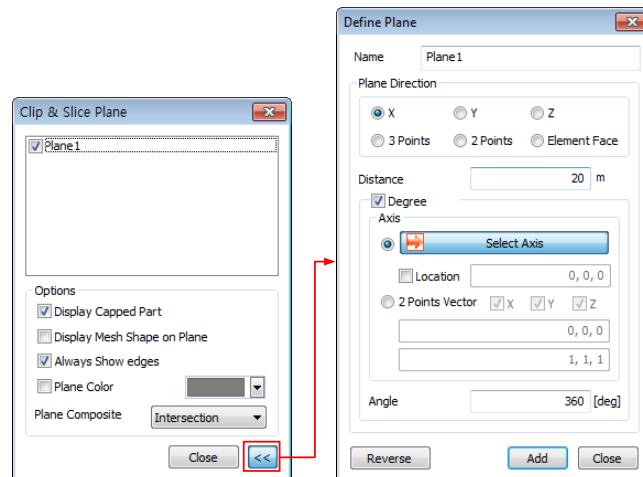
►Advanced view control toolbar



### 5.1 Clipping Plane

Geometric shapes can be cut by a specified plane direction and the section can be viewed. Necessary sections can be saved by pressing the [Add] button. Clicking the icon shows the clipped section. This function can be used to understand the interior shape of the geometry during pre-processing and is also useful in displaying particular cross sections during post-processing.

►Cutting plane & Clipping



#### Plane Direction

[X]: Cuts the shape in the plane normal to the X-axis. It is possible to edit the distance and move the plane.  
[Y]: Cuts the shape in the plane normal to the Y-axis. It is possible to edit the distance and move the plane.  
[Z]: Cuts the shape in the plane normal to the Z-axis. It is possible to edit the distance and move the plane..  
[3 point definition]: Defines 3 points in the work-plane to form a cutting plane. The shape is cut in this plane.  
[2 point definition]: Defines 2 points in the work-plane to form a cutting plane. The shape is cut in this plane.  
[Element face]: Cuts the shape in the plane containing the element face.

#### Distance

Display the distance between the plane and the origin.

#### Degree

Cross-section can be defined by Angle from selected axis.

#### Reverse

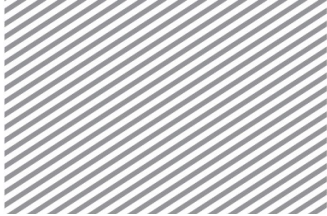
Show the section by changing the normal direction.

#### Display Capped Part

Uncheck to display the section only.

#### Display Mesh Shape on Plane





Check to display the mesh formed on the section.

**Always Show Edges**

Check to view the edges that make up the geometric shape.

**Plane Color**

Specify the color of the section.

**Plane Composite**

[Union]

Displays all sections and shapes on the screen.

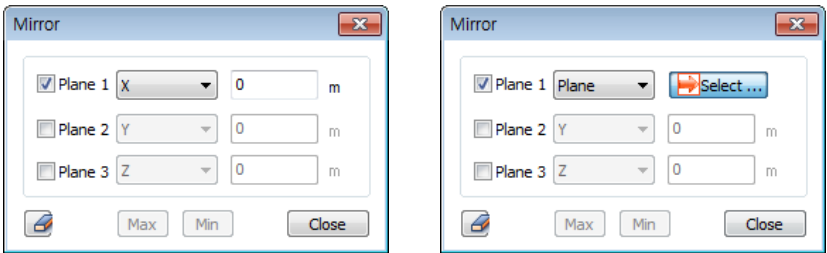
[Intersection]

Displays the intersection between sections and shapes on the screen.

**5.2  
Mirror Plane**

Displays the symmetrical model centered on a specified plane or planes normal to the X, Y, Z axis. The distance between the original model and the symmetrical model can be adjusted.

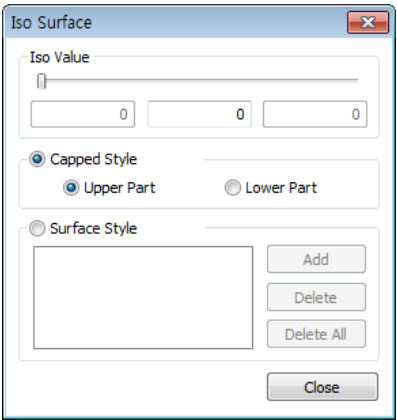
- ▶ View symmetric model (Axis)
- ▶▶ View symmetric model (Plane)



When configuring planes 1, 2, 3 individually, clicking the [Max] button displays the furthest symmetrical model with respect to the original model, while clicking the [Min] button displays the closest symmetrical model with respect to the original model.

**5.3  
Iso Value Surface**

- ▶Set equipotential surface

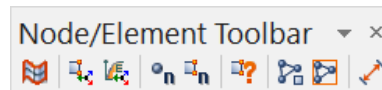











Sets a criterion for the resulting value and displays an area inside/outside of a specified value range. Checking the 'Surface Style' displays an area with the same result value as a face.



## Section 6 Node/Element Toolbar

►Node/Element Toolbar

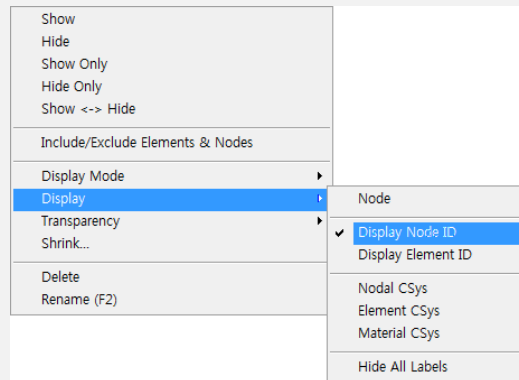


	<b>Show/Hide Element Free Edge</b> Shows/hides the B-Spline. A B-Spline is where the node is not shared between 2D elements.
	<b>Show/Hide Element Coordinate System</b> Shows/hides the element coordinate system. Because the element coordinate system is determined by the order and direction of the element, a unified coordinate system is necessary for a consistent load output. The element coordinate system can be modified at Mesh > Element > Parameter > Change CSys
	<b>Show/Hide Material Coordinate System</b> Shows/hides the material coordinate system. The user can also view the material coordinate system by selecting the mesh at Work tree > Mesh and then right mouse click > Display > Material CSys.
	<b>Show/Hide Node ID</b> Shows/hides the node ID. This function is useful when checking the results at a particular node.
	<b>Show/Hide Element ID</b> Shows/hides the element ID. This function is useful when checking the results at a particular element.
	<b>Query Node/Element</b> Checks the information of a specified node/element. The user can input the particular node/element number or click the node/element on the screen. The information is displayed in the output window.
	<b>Active Element</b> Shows/hides desired meshes that make up the total model. This function is effective when working with large complex models because it shows partial models. Selecting [All] shows all elements and the [Previous] button undoes the enabling/disabling of the element.
	<b>Active All Elements</b> Returns to the original state (view all elements) after Activating partial elements.
	<b>Measure Distance</b> Measure the distance or angle between two points. This function plays the same role as Geometry > Tools > Measure.



Tip

The user can set the display of nodes/elements of the model by selecting the mesh from Work tree > Mesh and then right mouse click > display to check the desired option.





## Section 7 Run Analysis Toolbar

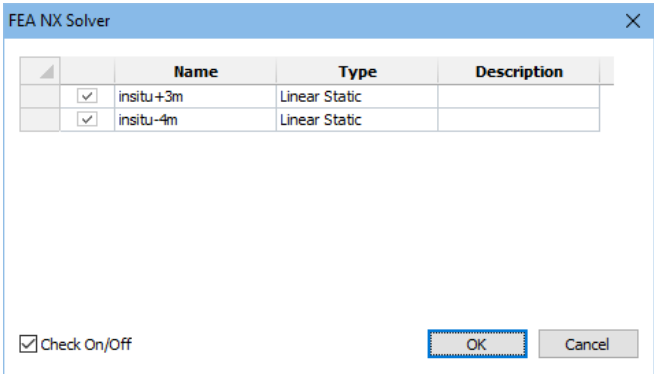
►Run Analysis Toolbar



### Perform

Run the analysis. The check button can be used to perform partial analysis of the case.

►Analysis execution manager



### Pre-mode

Switch to the pre-processing mode. <Unlock Icon>  
Information of geometry, mesh, loading and boundary conditions etc. are displayed in real time in a tree structure form similar to that of Microsoft Windows Explorer. A pop-up menu provides various control functions for object select and modification, show/hide etc.

### Post-mode

Switch to the post-processing mode. <Lock Icon>  
After the analysis has successfully finished, the program automatically loads the analysis results and the verifiable results are displayed in a tree structure form. The user can perform various functions such as graph output, table output, save post-processing graphic state etc.  
When performing additional work on geometric shapes or meshes, the user needs to switch back to the [Pre-processing mode] and reanalyze after the work has finished.