



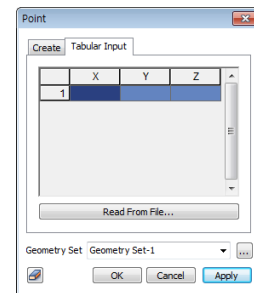
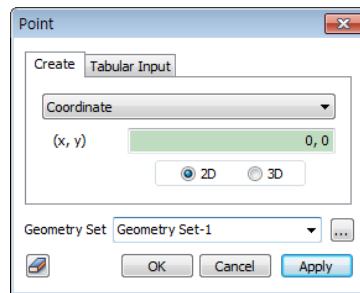
Section 1 Point and Curve

1.1 Point

Overview

Create an independent point on a 2D or 3D space.

- Create point(coordinate)
- Create point(Tabular input)



Create

There are 5 ways to create a point.

[Coordinate] : Directly input the coordinate. The user can create a point by directly clicking on the work-plane or typing in the coordinate values.

[Conic center] : Create a point at the center of an existing conic.

[Center of points] : Create a point at the center of an existing group of points.

[Curve-curve intersection] : Create a point at the intersection between two curves.

[Curve-surface intersection] : Create a point at the intersection between a curve and a face.

[Convert Node] : Create a point at the node of a created mesh.

Coordinate
Conic Center
Center of Points
Curve-Curve Intersection
Curve-Surface Intersection
Convert Node

Geometry set

Register the created point on the Geometry Set. The user can specify the name of the Geometry Set.

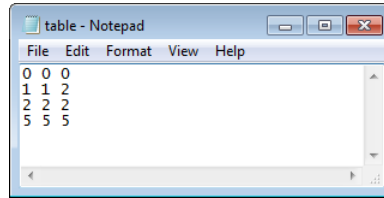
Tabular input


Directly inputs the 3D coordinate values in a table. The user can also select the import coordinates function to import a .txt file with the coordinate values. When creating a .txt file, the point coordinates are in the order of x,y,z and distinguished by a space in between.

The following example shows an input file to create points at (0,0,0), (1,1,2), (2,2,2) and (5,5,5).



► Tabular input

**Tip**

When entering the exact coordinates of a line intersection or center of a circle, it is also possible to use the center snap, intersection snap or point snap methods of the define snap ()function.

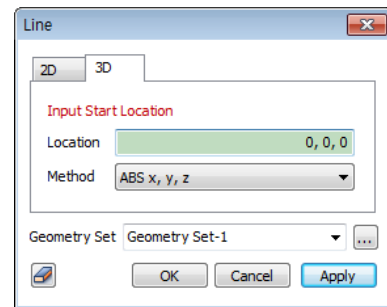
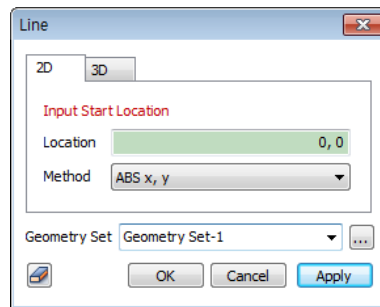
1.2 Line

► Create line – 2D

►► Create line – 3D

Overview

Create an edge type straight line on the work-plane.

**2D**

Input the coordinate of the start point and the end point to create a line. The start point input is in the absolute coordinate(x,y) form. For the end point, the user can select one of the following 3 methods:

1. [Absolute (x,y)] : Input the absolute 2D coordinate value in the work-plane.
2. [Relative (dx,dy)] : Input the relative distance from the point entered in the previous stage.
3. [Length,Angle] : Input the length and angle from the point entered in the previous stage. The angle is the rotation angle in the counter clockwise direction relative to the x axis of the work-plane.

3D

Input the coordinate of the start point and the end point to create a line. The start point input is in the absolute coordinate(x,y,z) form. For the end point, the user can select one of the following 2 methods:

1. [Absolute (x,y,z)] : Input a 3D absolute coordinate value in the work-plane.
2. [Relative (dx,dy,dz)] : Input the relative 3D distance from the point entered in the previous step.

The user can click on the work-plane or a geometry shape and directly assign the next point. The y-axis values are not entered because the work-plane is two-dimensional.

Geometry set

Register the created line on the Geometry set. The user can specify the name of the Geometry set.



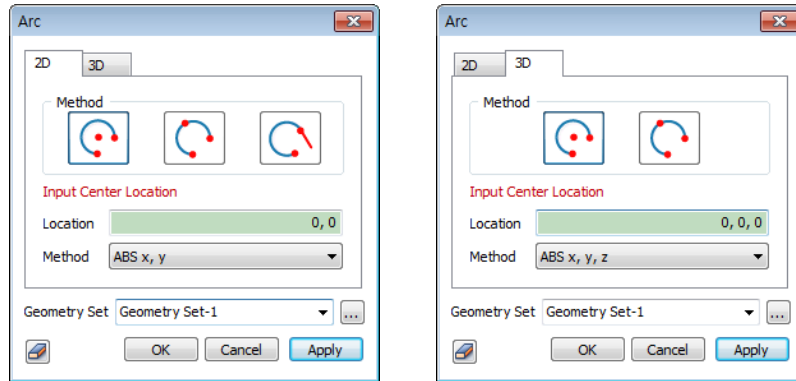
1.3

Arc

Overview

Creates an edge type arc on the work-plane

- Create arc – 2D
- Create arc – 3D



2D

Methodology

As shown below, there are 3 ways to create an arc. The points used in each method can be specified directly by clicking on the work-plane of the work screen. The arc is created in a counterclockwise direction relative to the vertical direction of the work-plane.



: Center point, Start point and End point

Creates an arc using the Center point [(Absolute x,y)], Start point [(Absolute x,y), (Radius, Start angle)] and End point [(Absolute x,y), (Included Angle), (End angle)] inputs in order.



: Start point, Secondary point and Endpoint

Creates an arc using the Start point [(Absolute x,y)], a Secondary point on arc[(Absolute x, y)] and End point [(Absolute x,y)] inputs in order.



: Edge and Curvature

The curve starts from the end of an existing edge and ends at a random point [Radius and Angle between] to draw an arc. The edge and curve are continuous, but are separate edges. The start point of the arc is set at the closest end point of the existing edge with respect to the mouse click.



3D**Methodology**

There are 2 ways to create an arc, as shown below. The points used in each method can be specified directly by clicking on the work-plane of the work screen. The arcs are created in a counterclockwise direction relative to the vertical direction of the work-plane.



: Center point, Start point and End point

Creates an arc using the center point [(Absolute x,y,z)], start point [(Absolute x,y,z)] and end point [(Absolute x,y,z)] inputs in order.



: Start point, Secondary point and Endpoint

Creates an arc using the [(Absolute x,y,z)], a secondary point on arc[(Absolute x,y,z)] and end point [(Absolute x,y,z)] inputs in order.

Geometry set

Register the created arc on the Geometry Set. The user can specify the name of the Geometry Set.

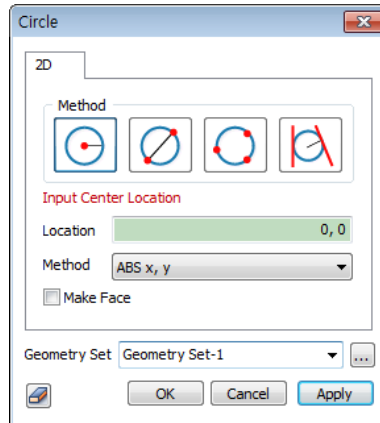


1.4 Circle

Overview

Create an edge type circle on the work-plane.

►Create circle



Methodology

There are 4 ways to draw a circle. The points used in each method can be specified directly by clicking on the work-plane of the work screen.



: Center coordinates and radius

Create a circle using center point coordinates [(Absolute x,y)] and radius.



: End points of diameter

Create a circle using one end point of the diameter [(Absolute x,y)] and the other end point [(Absolute x,y), (Relative dx,dy), (Length, Angle)].



: Three points on the circle

Create a circle using the coordinates of three arbitrary points on the circle [(Absolute x,y)].



: Within and adjacent to two edges

Create a circle that is within and adjacent to two existing edges. The user needs to input the radius and select two edges.

Make face

Creates the circular plane used to outline the circle.

Geometry set

Register the created circle on the geometry set. The user can specify the name of the geometry set.

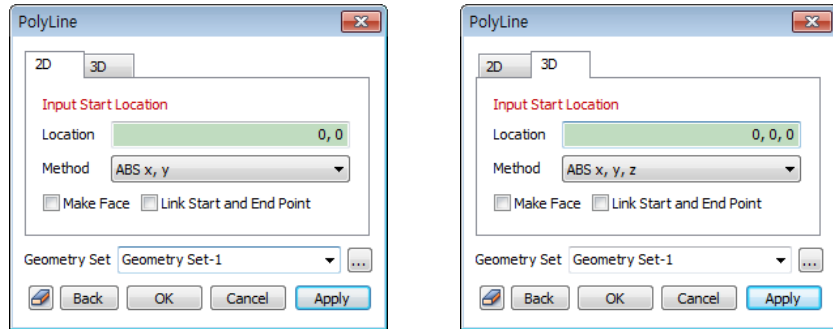


1.5 Polyline

Overview

Create an edge type polyline on the work-plane.

- Polyline – 2D
- Polyline – 3D



2D

Input the coordinates of the start point and the end point to create a polyline. The start point input is in the absolute coordinate(x,y) form. For the end point, the user can select one of the following 3 methods. The points used in each method can be specified directly by clicking on the work-plane of the work screen. The right mouse click stops the interpolation and creates the polyline.

1. [Absolute(x,y)] : Input an absolute 2D coordinate value in the work-plane.
2. [Relative(dx,dy)] : Input the relative distance from the point entered in the previous step.
3. [Length/Angle] : Input a length and angle from the point entered in the previous step. The angle is the rotation angle in the counterclockwise direction relative to the x axis of the work-plane.

3D

Input the coordinates of the start point and the end point to create a polyline. The start point input is in the absolute coordinate(x,y,z) form. For the end point, the user can select one of the following 2 methods. The user can click on the work-plane or a geometry shape and directly assign the next point. But because the work-plane is two-dimensional, the y-axis values are not entered.

1. [Absolute(x,y,z)] : Input a 3D absolute coordinate value in the work-plane.
2. [Relative(dx,dy,dz)] : Input the relative 3D distance from the point entered in the previous step.

Make Face

Create a plane that has the closed polyline as its boundary.

Link Start and End Point

Automatically connects the end point and the start point. The right mouse click on the work screen automatically connects the end point to the start point, closing the polyline.

Geometry set

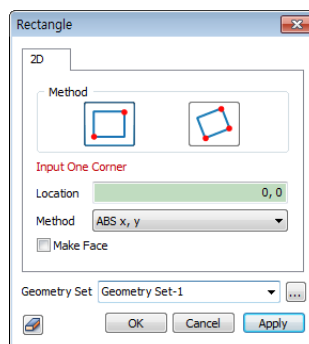
Register the generated polyline on the geometry set. The user can specify the name of the geometry set.

1.6 Rectangle

►Create rectangle

Overview

Create a wire type rectangle on the work-plane.



Methodology



: Diagonal End points

Creates a rectangle using one end point [(Absolute x,y)] and the other diagonal end point [(Absolute x,y),(Relative x,y)].



: 2 End points and length

Creates a rectangle using one End point [(Absolute x,y)], a second end point [(Absolute x, y),(Relative x,y)] and the length inputs in this order.

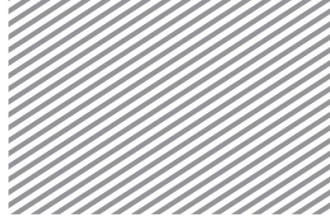
The user can directly click on the workspace of the work screen to specify points.

Make Face

Create a rectangular plane that has the rectangle as its boundary. In this case, it does not create a border of rectangular wire.

Geometry set

Register the generated rectangle on the geometry set. The user can specify the name of the geometry set.

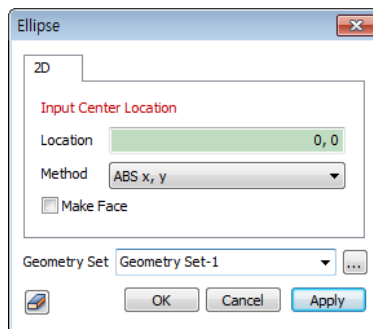


1.7 Ellipse

Overview

Create a wire type ellipse on the work-plane.

►Create ellipse



Methodology

Creates an ellipse using the center point coordinates [Absolute (x,y)] and the major and minor axis radius. The major axis input can be in the form of [Absolute (x,y)] or [Length, Angle]. When using the [Length, Angle] form, input the length and angle relative to the previous point.

The user can directly click on the workspace of the work screen to specify points.

Make Face

Create an elliptical plane that has the ellipse as its boundary line. In this case, it does not create a border of edge type wire.

Geometry set

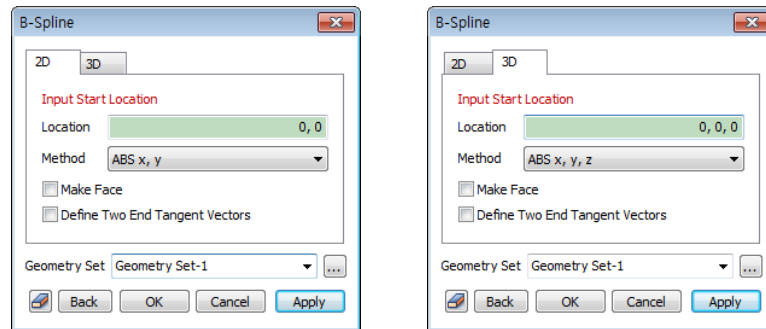
Register the generated ellipse on the geometry set. The user can specify the name of the geometry set.

1.8 B-Spline

Overview

Create an edge type B-Spline on the work-plane.

- B-Spline – 2D
- B-Spline – 3D



2D

Input the coordinates of the start point and the end point to create a B-Spline. The start point input is in the absolute coordinate(x,y) form. For the end point, the user can select one of the following 3 methods:

1. [Absolute(x,y)] : Input an absolute 2D coordinate value in the work-plane.
2. [Relative(dx,dy)] : Input the relative distance from the point entered in the previous step.
3. [Length/Angle] : Input a length and angle from the point entered in the previous step. The angle is the rotation angle in the counterclockwise direction relative to the x-axis of the work-plane.

The points can be specified directly by clicking on the work-plane of the work screen. The right mouse click stops further interpolation and creates the B-Spline.

3D

Input the coordinates of the start point and the end point to create a B-Spline. The start point input is in the absolute coordinate(x,y,z) form. For the end point, the user can select one of the following 3 methods.

- [Absolute(x,y,z)] : Input a 3D absolute coordinate value in the work-plane.
- [Relative(dx,dy,dz)] : Input the relative 3D distance from the point entered in the previous step.

The user can click on the work-plane or a geometry shape and directly assign the next point, but because the work-plane is two-dimensional, the y-axis values are not entered. The right mouse click stops further interpolation and creates the B-Spline.

Make Face

Create a plane that has the closed B-Spline as its boundary.

Define Two End Tangent Vectors

Use the tangent vectors of the start and end points to modify the shape of the overall B-Spline after it has been finished.

Geometry set

Register the created B-Spline on the geometry set. The user can specify the name of the geometry set.

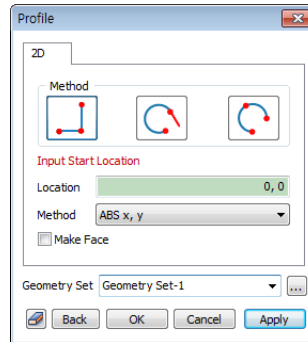


1.9 Profile

Overview

Create a wire type profile that consists of lines and arcs on the work-plane.

►Create profile



Methodology

The user can directly click on the work-plane in the work screen or input the coordinate values. When specifying the points using the mouse, the right mouse click stops further interpolation and creates the profile.



: Three point straight line profile

Creates a connected straight line using the start point coordinates [(Absolute x,y)] and continuously inputting the next point's coordinates [(Absolute x,y),(Relative dx,dy),(Length, Angle)].



: Two point arc profile

Creates an arc that is adjoined to the previous line. Because the start point of the arc is automatically assigned to the end point of the previously created line, only the end point coordinates of the arc [(Radius, Angle)] are needed. It is not possible to draw an arc connected to a line and is it only possible during the create profile process.



: Three point arc profile

Creates an arc using the coordinates of three points. When using this function during the create profile process, the end point of the previous line becomes the first point and only the coordinates for the other two points [(Absolute x,y),(Relative dx,dy)] are needed. However if this function is used first in the profile, the user must input all three points in order to create an arc.

Make Face

Create a plane that has the closed polyline as its boundary. In this case, it does not create an wire type profile.

Geometry set

Register the created profile on the Geometry set. The user can specify the name of the geometry set.

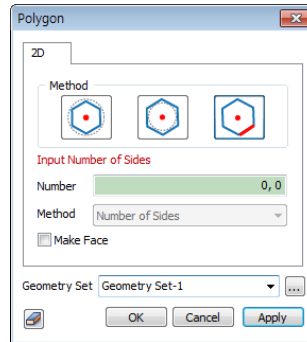


1.10 Polygon

Overview

Create a wire type polygon on the work-plane.

►Create Polygon



Methodology



: Inscribed regular polygon

Input the number of sides of the regular polygon. Input the center coordinates of the inscribed circle [(Relative x,y)] and select one of the 3 input methods for the coordinate values of the radius and direction.

(Absolute coordinates x,y)

(Relative coordinates dx,dy)

(Length, Angle)

The vertex of the polygon will be located on the second input point.



: Circumscribed regular polygon

Input the number of sides of the regular polygon. Input the center coordinates of the inscribed circle [(Relative x,y)] and select one of the 3 input methods for the coordinate values of the radius and direction.

(Absolute coordinates x,y)

(Relative coordinates dx,dy)

(Length, Angle)

A tangent point will be located on the second input point.



: Edge length, Center point and angle between X axis

Input the number of sides of the regular polygon. Input the center point coordinates of the polygon [Absolute (x,y)]. Input the angle between a vertex and the x axis to create a polygon.

Make Face

Create a plane that has the closed polyline as its boundary. In this case, it does not create a wire type polygon.

Geometry set

Register the generated polygon on the geometry set. The user can specify the name of the geometry set.



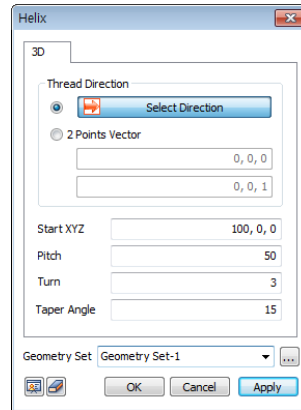
1.11

Helix

►Create Helix

Overview

Create a wire type helix on the work-plane.



Direction

Determine the helix direction by selecting the reference axis direction or a two-point vector.

[Select direction] : Determine the direction vector that is to be used as the reference axis of the helix. The user can select the datum axis, datum plane, plane or edge.

[2 points vector] : Determine the direction vector that is to be the reference axis of the helix by entering the coordinates of its start and end points. The user can also directly click on the work screen to specify the Start and End points.

Other inputs

Create a spiral by entering the Start XYZ, Pitch, Turn and Taper angle.

[Start XYZ] : Input the 3D absolute coordinates for the start point of the helix. The user can also directly click on the work screen to specify the point.

[Pitch] : Defined as the height increase in the axis direction per one rotation period.

[turn] : The number of revolutions in the helix.

[Taper angle] : The angle between the reference axis and side slope of the helix.

Some errors can occur when creating a helix; so the input value and the angle between the reference axis and side slope may not match exactly.

Geometry set

Register the generated helix on the geometry set. The user can specify the name of the geometry set.



1.12 Tunnel

Overview

Create a wire type tunnel section on the work-plane.

►Create Tunnel section

Tunnel Section

Tunnel Type: 3 Center Circle

Section Type: ☒ Full ☐ Left Half ☐ Right Half

Dimensions

Invert: ☐ Tangential ☒ Radius ☐ Angle

R1: 6.5 m A1: 60 [deg]
R2: 6 m A2: 55 [deg]
R3: 0 m A3: 0 [deg]
R4: 0 m A4: 0 [deg]

Asymmetric Section

A1': 60 [deg]
R2': 6 m A2': 55 [deg]
R3': 0 m A3': 0 [deg]

Include Rock Bolts

Number of Rock Bolts: 11
Length of Rock Bolts: 4 m

Arrangement: ☒ Tangential Pitch ☐ Rotation Angle

Tangential Pitch: 2 m
Rotation Angle: 20 [deg]

Location

Screen Snap: ☐
Section Center: 0, 0

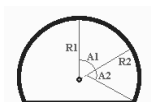
☒ Make wire

Geometry Set: Geometry Set-1

OK Cancel Apply

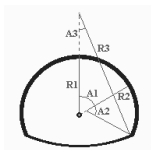
Tunnel type

There are 4 types of tunnels: [3 Center Circle], [3 Center Circle + Invert], [5 Center Circle] and [5 Center Circle + Invert]. There are 3 types of sections: [Full], [Left Half], [Right Half].



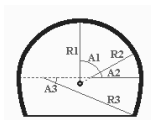
: 3 Center Circle

Create a tunnel using 3 arcs, all with different center points and diameters.



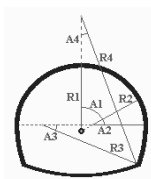
: 3 Center Circle + Invert

Create a tunnel using 3 different arcs and an invert.



: 5 Center Circle

Create a tunnel using 5 arcs, all with different center points and diameters.



: 5 Center Circle + Invert

Create a tunnel using different 5 arcs and an invert.

Size



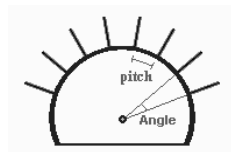
Input the Radius (R1~R4) and Angle (A1~A4) of the tunnel section. If the section is not horizontally symmetrical, check the axis symmetry section to create a non-symmetric tunnel shape. In this case, the Inputs A1'~A3', R2'~R3' represent the shape on the left half section.

Invert

This function is only active when the tunnel type is [3 center circle + invert] or [5 center circle+invert]. The invert can be drawn in an arc form when the bottom of the tunnel is not flat. The invert shape can be created with one of the following 3 methods:

1. [Tangential] : Creates an invert shape based on the Radius (R) and Angle (A) input and the 3 center circle (or 5 center circle) information.
2. [Radius] : Creates an invert shape based on the 3 center circle (or 5 center circle) information and the appropriate Angle (A) calculated by the Radius (R) input.
3. [Angle] : Creates an invert shape based on the 3 center circle (or 5 center circle) information and the appropriate Radius (R) calculated by the Angle (A) input.

Rock bolt



If the tunnel section contains rock bolts, check [Include Rock Bolts] to place the edges. Input the number of rock bolts and their length. The user can align the bolts by using two methods: [Tangential Pitch] or [Rotation Angle].

Location

Input the center point location of the created tunnel section.

The user can input the center point coordinates on the work-plane by either checking the [Screen Snap] and using the mouse, or directly entering the coordinates.

Make wire

Create the tunnel section shape using a single wire. If the user does not check the option, it creates the tunnel section shape using multiple edges.

Geometry set

Register the created tunnel section on the Geometry set. The user can specify the name of the Geometry set.

1.13

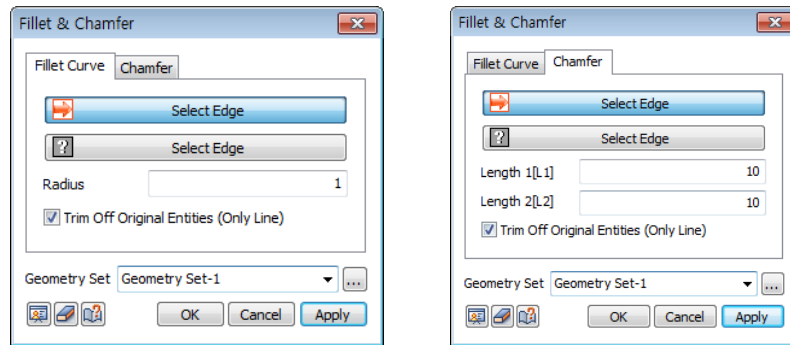
Fillet Curve & Chamfer

►Fillet Curve

►►Chamfer

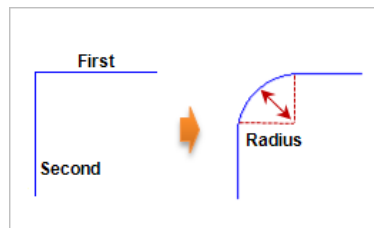
Overview

Applies fillet and chamfer to the intersection between two edges. This function is only applicable when the edges exist on the work-plane.



Fillet

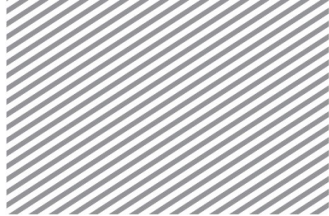
Methodology



Select the two edges that need to be filleted and input the radius. The applied fillet curves become a single wire type.

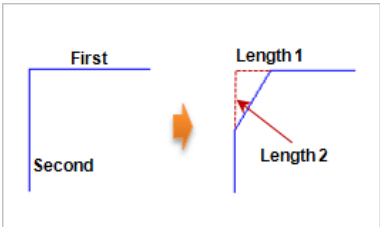
Trim Off Original Entities (Only Lines)

Remove the exterior edge protruding from the filleted area. If the selected edge is an arc or circle, it is not removed.



Chamfer

Methodology



Select the two edges that need to be trimmed and input the radius. The applied line trim becomes a single wire type.

Trim off Original Entities (Only Line)

Remove the exterior edge protruding from the trimmed area.

Geometry set

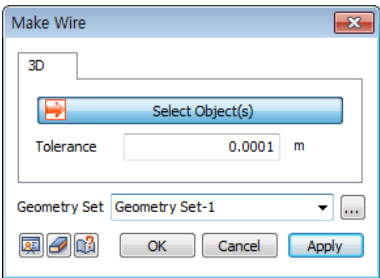
Register the filleted or trimmed edge on the Geometry set. The user can specify the name of the Geometry set.

1.14
Make Wire

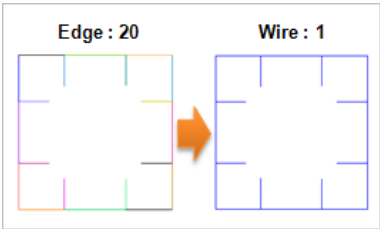
►Make wire

Overview

Create a single wire from selected edges on the work-plane.



Methodology



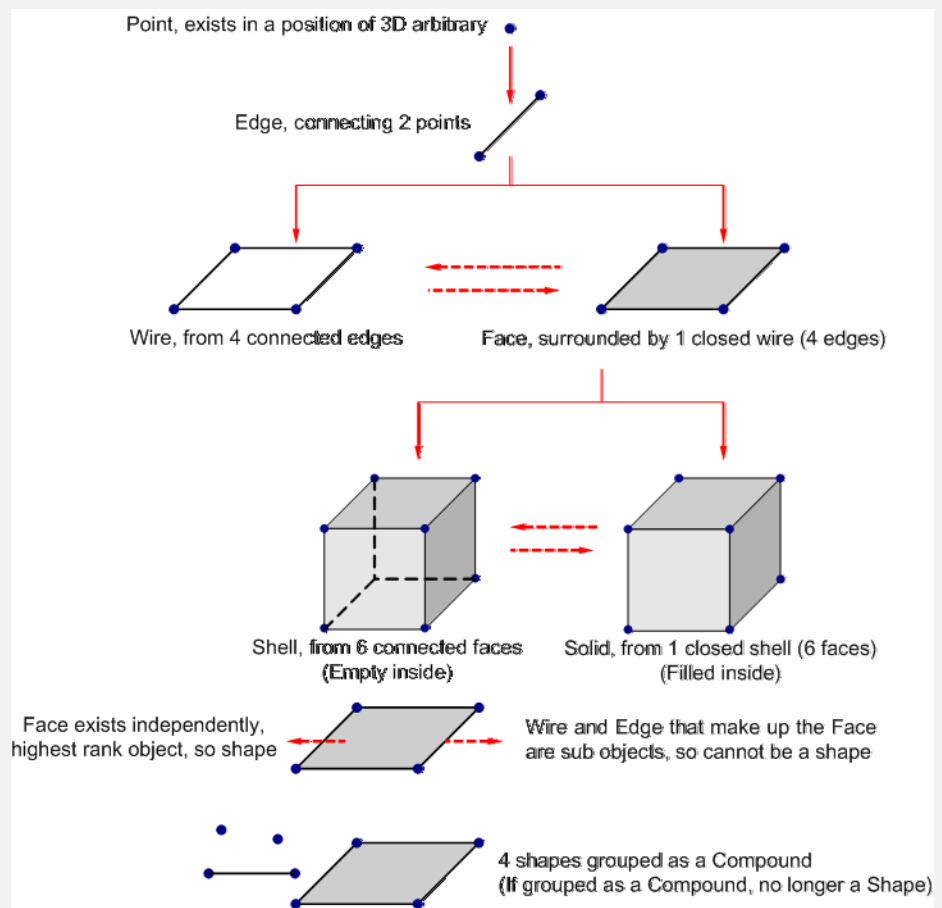
Create a single wire from edges within the Tolerance.
[Tolerance] is the allowed error criterion that determines whether the edges are connected. For example when the distance between the edges is 2.0e-006, although the value is not zero, it can be made into a single wire by setting the error as 0.0001 (distance larger than the value between edges).
After the operation is performed, a single wire remains and the selected edges are deleted.

Geometry set

Register the wire on the Geometry set. The user can specify the name of the Geometry set.

Tip

A wire is a concept of collective edges and has these edges as its sub-shape. Because the wire is a set of edges, the user can use commands such as **Extend** to create shells, a set of faces created by individual edges. For these reasons, it is not appropriate to use the **create wire** function when creating a face by extruding multiple edges. The geometry shape hierarchy is as follows.





1.15

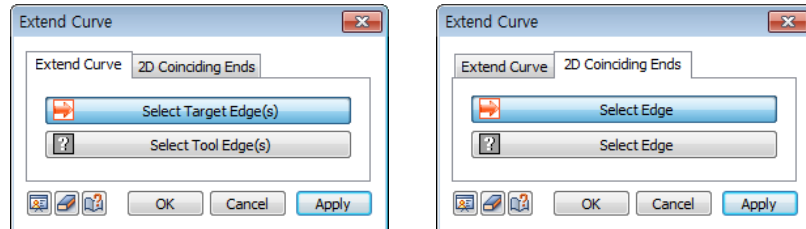
Extend Curve

►Extend Curve

►►2D Coinciding Ends

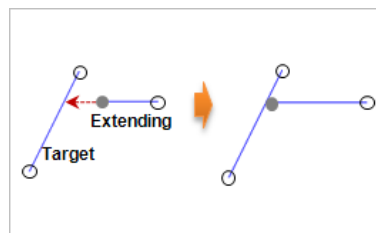
Overview

Extend the length of existing edges or trims edges to match the end points on the work-plane.



Extend Curve

Methodology

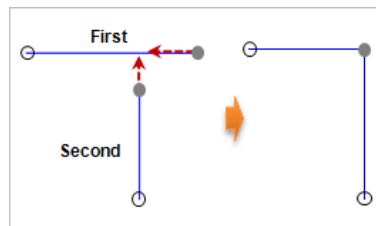


Select the reference edge to which the target edge will extend.

The function does not work for edges that would not meet even if they were extended. The extend function is unavailable for B-Spline type edges.

2D Coinciding Ends

Methodology



Select end points that need to be matched. After the operation, the selected edges are removed and only the extended edges remain.

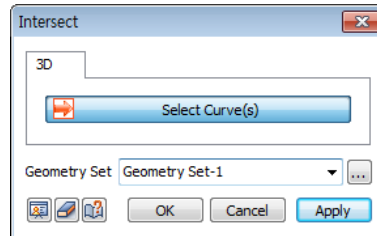
When one of the edges is long enough and only the short edge needs to be extended, the short edge is extended to and the long edge is cut until the intersection point.

1.16 Intersect

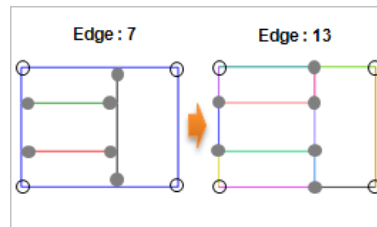
►Intersect

Overview

Cut the intersecting edges at the intersection point.



Methodology



Select the edges to cut at the intersection.

After the operation, the selected edges are removed and only the cut edges remain.

Geometry set

Register the generated edges on the Geometry set. The user can specify the name of the Geometry set.

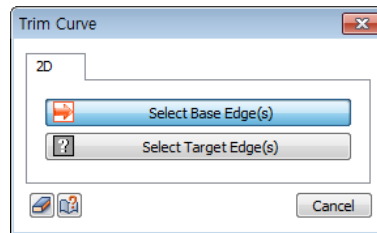


1.17 Trim Curve

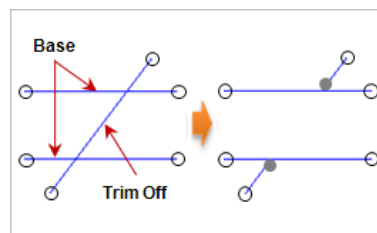
Overview

Cut the intersecting edges based on the intersection point. This function is only applicable when the edges exist on the work-plane. After the operation, the selected edges are removed and only the cut edges remain.

►Cut line



Methodology



Select the reference edge for the cut and then select the target edge. The user can select multiple reference edges. The edges will remain unchanged after the operation.

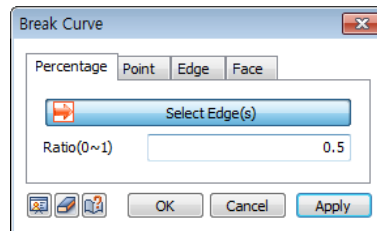
1.18

Break Curve

►Break curve-Percentage

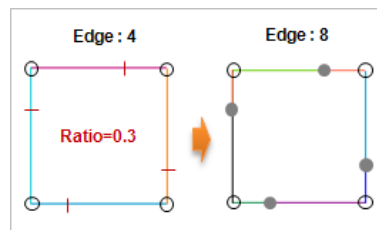
Overview

Divide an edge in the 3D space.



Percentage

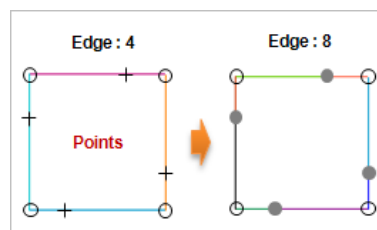
Methodology



Divide the edge using a ratio. Multiple edges can be selected and divided. The original length is viewed as 1 and the ratio is set at a value between 0 and 1.

Point

Methodology

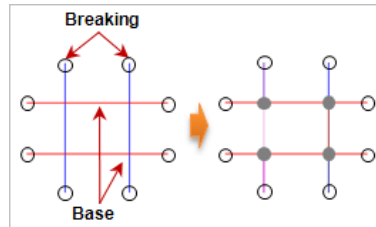


Divide the edge using points. If there is no geometry shape to divide, the user can specify the object on the work-plane using the snap option or directly enter the coordinate values. The selected point is projected onto the edge at the shortest distance and that point is the reference point for the division.



Edge

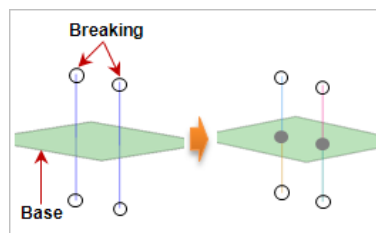
Methodology



Divide edges using another edge. Select the reference edge and divide the target edge. The selected edge is projected onto the edge at the shortest distance and that point is the reference point for the division.

Face

Methodology



Divide the edge using a face. The selected face is projected onto the edge at the shortest distance and that point is the reference point for the division.

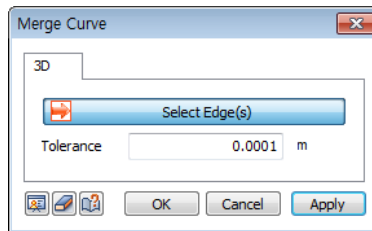


1.19 Merge Curve

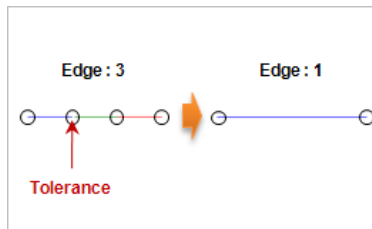
Overview

Select edges that exist on the work-plane and connect the end points using appropriate extend or trim operations.

►Merge curve



Methodology



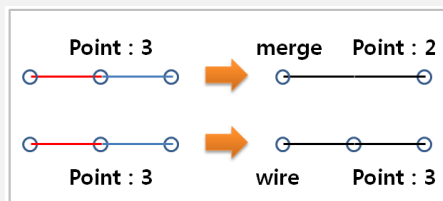
Create a single edge from edges within the error limit. The error is the allowed error criterion that determines whether the edges are connected. For example when the distance between the edges is $2.0e-006$, although the value is not zero, it can be made into a single edge by setting the error as 0.0001 (distance larger than the value between edges).

After the operation, the selected edges are removed and only a single joined edge remains.

Tip

* Join line and wire

Join line and wire are both ways to join multiple edges into a single line. The middle points used to create the edges are removed but for join line, but not for wire. So if the small points can be ignored, it is best to use the join line function. If not, it is best to use the create wire function.



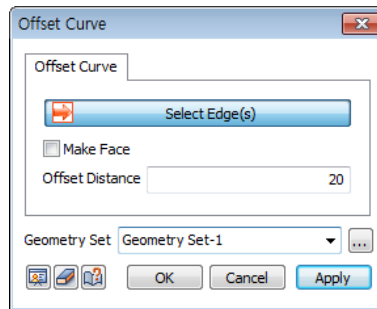


1.20 Offset Curve

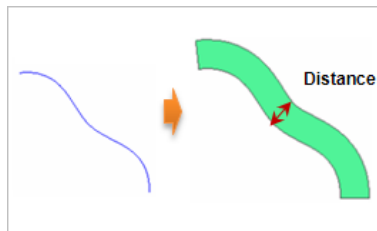
Overview

Create a new line by offsetting an existing edge by a certain distance. The offset is created only when the edges exist on the work-plane.

►Offset curve



Methodology



Select the edge and line to offset on the work-plane. Circles or arcs can also be selected. Because the offset is always in the normal direction of the target edge, there is no need to set the direction. The user only needs to input the distance to automatically offset. Checking the [Make Face] option creates a plane that contains the offset edges.

Geometry set

Register the generated edges on the Geometry set. The user can specify the name of the Geometry set.

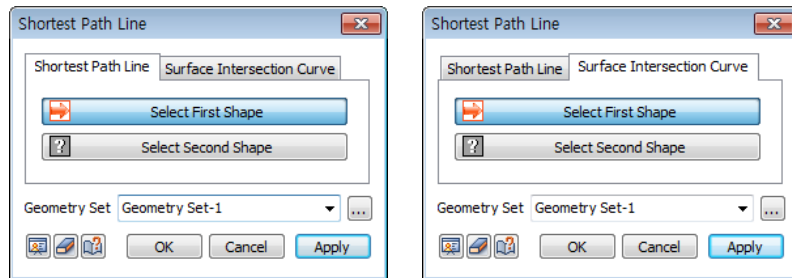


1.21 Shortest Path Line

- Shortest path line
- Surface intersection curve

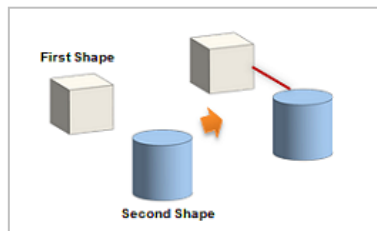
Overview

Create a shortest distance line between two shapes or an intersection curve between two intersecting shapes.



Shortest path line

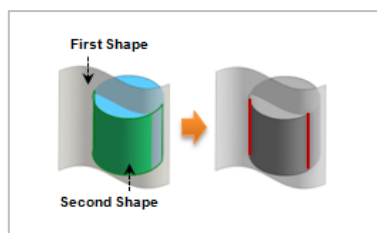
Methodology



Create the shortest possible line between two shapes by selecting them in order.

Surface Intersection Curve

Methodology



Selecting two shapes create an edge type line at the intersection. If the intersection is a single edge, a single edge is created but if there are more intersections, a compound shape consisting of edges is created.

Geometry set

Register the generated edges on the Geometry set. The user can specify the name of the Geometry set.



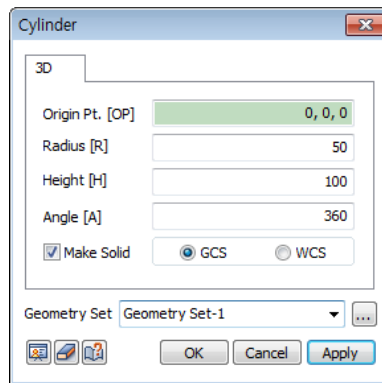
Section 2 Surface and Solid

2.1 Cylinder

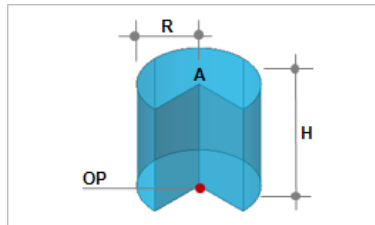
Overview

Create a shell or solid type cylinder.

►Create cylinder



Methodology



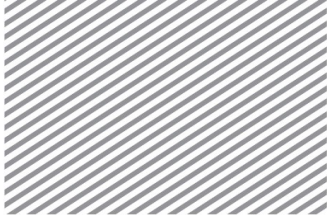
Create a cylinder from the Origin Point(OP), Radius(R), Height(H) and Angle(A) input.

[Origin Point (OP)]

The Origin point consists of the Center point coordinates of the bottom face of the cylinder. The user can directly specify the center point of the work-plane or geometry shape by selecting the points with the mouse. Because the workspace is 2-dimensional, the y axis value is not entered.

[Angle (A)]

The angle is the rotation angle of the circle between the top and bottom face of the cylinder. If the user enters 360, it creates a generic cylinder with circular top and bottom faces.



Make Solid

Check this option to create a solid type cylinder with a volume. Un-checking the option will create a shell type cylinder.



For GCS (Global coordinate system) and WCS (Work-plane coordinate system), please refer to General information-Manage modeling toolset.

- GCS
Enter the center coordinates with reference to the Global coordinate system. In this case, the center coordinates are input in 3D space.
- WCS
Enter the center coordinates with reference to the Work-plane coordinate system. In this case, the center coordinates are input in 2D space.

Geometry set

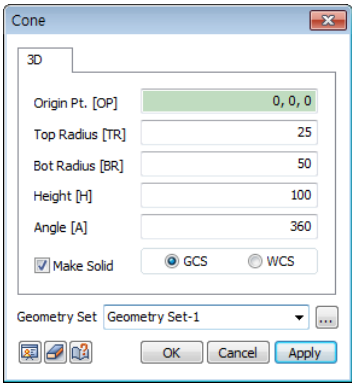
Register the created cylinder on the Geometry set. The user can specify the name of the Geometry set.

**2.2
Cone**

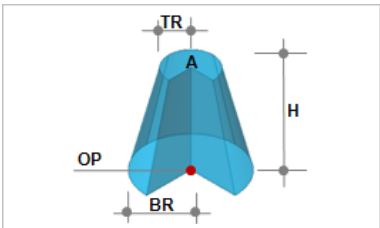
►Create cone

Overview

Create a shell or solid type cone.



Methodology



Create a cone from the Origin point (OP), Top Radius (TR), Bottom Radius (BR), Height (H) and Angle (A) input.

[Origin Point (OP)]



The center point consists of the Center point coordinates at the bottom face of the cone. The user can directly specify the center point of the work-plane or geometry shape by selecting with the mouse. Because the workspace is 2-dimensional, the y-axis value is not entered.

[Angle (A)]

The angle is the rotation angle of the circle between the top and bottom face of the cone. If the user enters 360, it creates a generic cone with circular top and bottom faces.

Make Solid



For GCS (Global coordinate system) and WCS (Work-plane coordinate system), please refer to General information-Manage modeling toolset.

Check this option to create a solid type cone with a volume. Un-checking the option will create a shell type cone.

- GCS

Enter the center coordinates with reference to the Global coordinate system. In this case, the center coordinates are input in 3D space.

- WCS

Enter the center coordinates with reference to the Work-plane coordinate system. In this case, the center coordinates are input in 2D space.

Geometry set

Register the created cone on the Geometry set. The user can specify the name of the Geometry set.

Tip

- When creating a solid or 2D mesh on a cone, the user can observe mesh deformation at the vertex. This phenomenon happens near the pole, which is created near the revolution axis of the rotated body.
- Mesh deformation can also happen during the general modeling process when a vertex is created on a curved face. This phenomenon may negatively effect on the analysis results. This problem cannot be solved on the geometry shape and it is best to use move node or merge node functions after creating the mesh for any modifications.



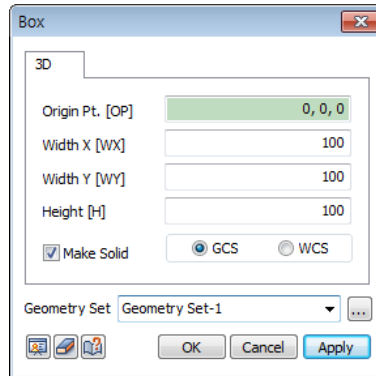
2.3

Box

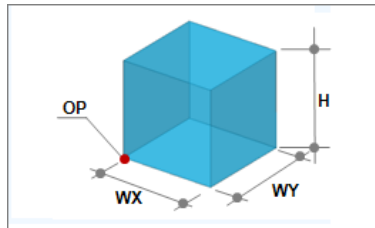
Overview

Create a shell or solid type box.

►Create box



Methodology.



Create a box from the Origin Point(OP), Width X(WX), Width Y(WY) and Height(H).

The Origin Point(OP) consists of the corner point coordinates of the bottom face of the box.

The user can directly specify the center point of the work-plane or geometry shape by selecting the points with the mouse. Because the workspace is 2-dimensional, the y-axis value is not entered.

Make Solid

Check this option to create a solid type box with a volume. Un-checking the option will create a shell type box.



For GCS (Global coordinate system) and WCS (Work-plane coordinate system), please refer to General information-Manage modeling toolset.

• GCS

Enter the center coordinates with reference to the Global coordinate system. In this case, the center coordinates are input in 3D space.

• WCS

Enter the center coordinates with reference to the Work-plane coordinate system. In this case, the center coordinates are input in 2D space.

Geometry set

Register the created box on the Geometry set. The user can specify the name of the Geometry set.

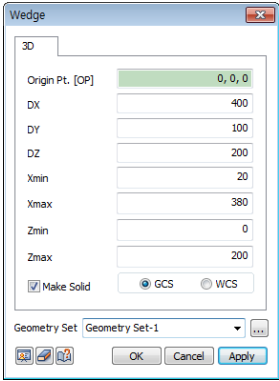


2.4 Wedge

Overview

Create a shell or solid type, wedge-shaped atypical hexahedral.

►Create wedge



Methodology

Create a wedge shape from the Origin Point (OP), Length (DX, DY, DZ) and Shape width (Xmin, Xmax, Zmin, Zmax).

The Origin Point (OP) is the corner point coordinates of the bottom face of the hexahedral. The user can directly specify the center point of the work-plane or geometry shape with the mouse. Because the workspace is 2-dimensional, the y-axis value is not entered.

DX, DY, DZ represent the bottom face x direction length, height and bottom face width of the hexahedral respectively.

X min, X max is the relative distance of the Start and End point coordinates of the top face in the x-axis.

Z min, Z max is the relative distance of the Start and End point coordinates of the top face in the z-axis.

Tip

The hexahedral is created as follows. If the Corner coordinates are (X,Y,Z), The bottom face is the rectangle that starts at (X,Y,Z) and ends at (DZ,DY,DX). The top face is the rectangle that starts at (X+X min,Y,Z+Z min) and ends at (X+X max,Y,Z+Z max). The hexahedral is set based on the two rectangles.

Make Solid

Check this option to create a solid type hexahedral with a volume. Un-checking the option will create a shell type hexahedral.

• GCS

Enter the center coordinates with reference to the Global coordinate system. In this case, the center coordinates are input in 3D space.

• WCS

Enter the center coordinates with reference to the Work-plane coordinate system. In this case, the center coordinates are input in 2D space.

Geometry set

Register the created hexahedral on the Geometry set. The user can specify the name of the Geometry set.



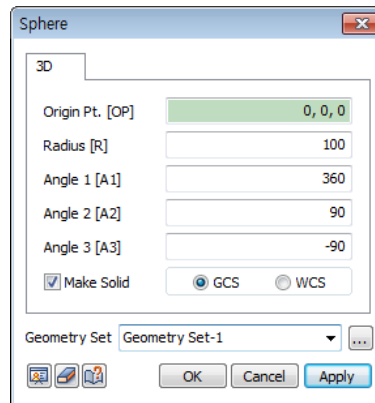
For GCS (Global coordinate system) and WCS (Work-plane coordinate system), please refer to General information-Manage modeling toolset.

2.5 Sphere

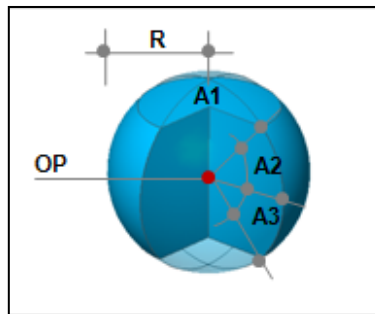
Overview

Create a shell or solid type sphere.

►Create sphere



Methodology



Create a sphere from the Origin point (OP) and Angles (A1, A2, A3).

Origin point (OP)

The center point coordinates of the sphere. The user can directly specify the center point of the work-plane or geometry shape by selecting the points with the mouse. Because the workspace is 2-dimensional, the y-axis value is not entered.

Angle (A1, A2, A3)

The angle input to create a sphere. The values represent:

- A1: Rotation angle of the sphere's vertical plane
- A2: Start angle of the sphere's vertical plane ($0 < A2 \leq 90$)
- A3: End angle of the sphere's vertical plane ($-90 \leq A3 \leq 0$)



Make Solid

Check this option to create a solid type sphere with a volume. Un-checking the option will create a shell type sphere.

- GCS

Enter the center coordinates with reference to the Global coordinate system. In this case, the center coordinates are input in 3D space.

- WCS

Enter the center coordinates with reference to the Work-plane coordinate system. In this case, the center coordinates are input in 2D space.

Geometry set

Register the created sphere on the Geometry set. The user can specify the name of the Geometry set.

Tip

- When creating a solid or 2D mesh on a sphere, the user can observe mesh deformation at the vertex. This phenomenon happens near the pole, which is created near the revolution axis of the rotated body.
- Mesh deformation can also happen during the general modeling process when a vertex is created on a curved face. This phenomenon may have a negative effect on the analysis results. This problem cannot be solved on the geometry shape and it is best to use move node or merge node functions after creating the mesh for any modifications.



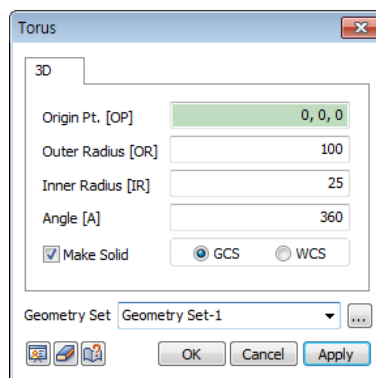
For GCS (Global coordinate system) and WCS (Work-plane coordinate system), please refer to General information-Manage modeling toolset.

2.6 Torus

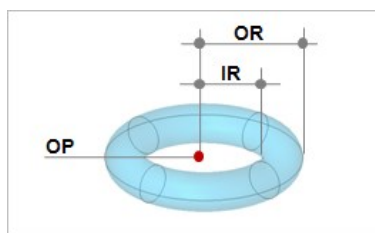
Overview

Create a donut shaped rotated body. Using a closed section creates a shell or solid type torus. Using an open section creates a rotated body with a blocked center. The sectional shape is a circle or an arc.

►Create torus



Methodology



Create a torus from the Origin Point (OP), Outer Radius (OR), Inner Radius (IR), Height (H) and Angle (A) input.

[Origin point (OP)]

The center point coordinates of the torus. The user can directly specify the center point of the work-plane or geometry shape by clicking. Because the workspace is 2-dimensional, the y axis value is not entered.

[Outer radius (OR) and Inner radius (IR)]

The outer circumference radius and inner circumference radius respectively. $0 < IR < OR$ needs to be satisfied.

[Angle (A)]

The angle is the start angle of the sectional shape (circle or arc) of the rotated body.

Make Solid

Check this option to create a solid type torus with a volume. Un-checking the option will create a shell type torus.



GCS (Global coordinate system) and WCS (Work-plane coordinate system), please refer to General information-Manage modeling toolset.

- GCS
Enter the center coordinates with reference to the Global coordinate system. In this case, the center coordinates are input in 3D space.
- WCS
Enter the center coordinates with reference to the Work-plane coordinate system. In this case, the center coordinates are input in 2D space.

Geometry set

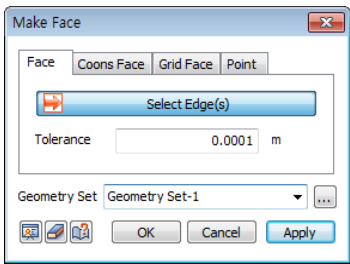
Register the created torus on the Geometry set. The user can specify the name of the Geometry set.

2.7 Make Face

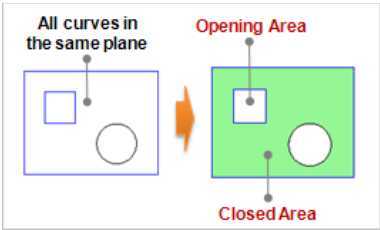
►Make Face

Overview

Create a plane from multiple closed edges. The edges are the outline of the plane.



Face



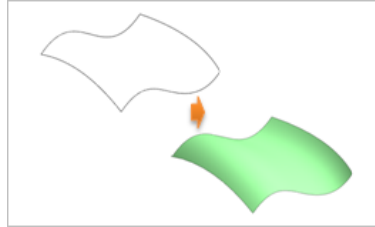
Create a face by selecting the edges that form its outline. There is no limit to the numbers of edges the user can select, but there should be no overlapping outlines in order to create a normal plane.

[Tolerance]

The criterion that determines whether an edge is connected. For example when the distance between the edges is 2.0×10^{-6} , although the value is not zero, it is determined to be connected with a tolerance of 0.0001 (distance larger than the value between edges).

Tip

If there is an outline inside an area formed by another outline, it is still possible to create a plane that considers both the exterior and interior outlines. However, if the exterior and interior outlines are not on the same plane, the interior face is projected onto the exterior face.

Coons face

Create a 3D curved face using 2 or 4 outline edges.

When using 4 edges, the selected outline needs form a closed area to generate a normal curved face.

When using 2 edges, the curved face is created by the two outlines and the closest distance between them. When creating a Coons face using 2 edges, the face is formed by connecting the selected edges in a straight line. Because the face is created by connecting the Start points and End points of the two edges, it creates a twisted face if the edge directions do not match. In this case, use the reverse direction option to connect the Start and End points of each edge to create a normal face.

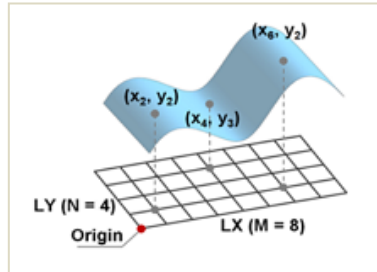
Tip

A Coons face is a curved face that has small errors and a fast production rate. However, using more than 4 outlines to create the face is inconvenient. When using 4 edges to create a Coons face, use the join line function to reduce the number of outlines to at most 4.

The face is not created if the shape is not shaded in the preview window after the edges have been selected. A face that cannot be shaded is one that has a problem in its configuration and has an effect on future modeling processes. In this case, it is recommended that the edges be modified or recreated or rejoined before creating the Coons face again. If there is no appropriate solution, it is safe to remodel the abnormal face using the shape modification function in the modification toolset.



Grid Face



Set an imaginary grid on the XY plane of the GCS and input the height information for each grid point to create a compound shape-curved face.

M(X direction number), N(Y direction number) represent the number of grid lines in each axis direction.

Origin X and Origin Y are the X, Y coordinates of the grid start point. LX and LY are the length in the X and Y direction of the plane.

The height information can be directly entered into a table dialog box or input from a .txt file.

Point



Create a curved face in the space that contains all selected points.

The points can be selected on the screen or their coordinates directly entered into a table dialog box.

When using a table to input the coordinates, the [Make Points] becomes active. Check this option to create the input point and un-check this option to create the face only. Check [Make Points As Compound] to create points that are grouped as a compound shape.

Geometry set

Register the created face on the Geometry set. The user can specify the name of the Geometry set.

2.8

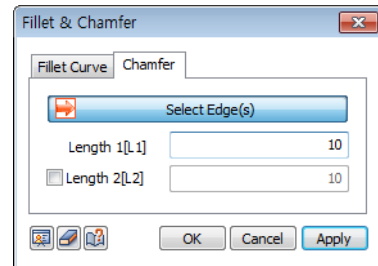
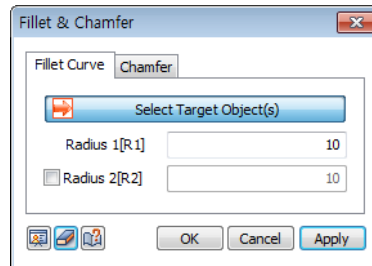
Overview

Fillet and chamfer

Create a fillet or chamfer shell on a shell or solid type edge.

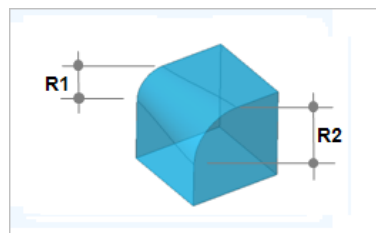
► Fillet and Chamfer (Fillet curve)

►► Fillet and Chamfer (Chamfer)




Fillet Curve

Methodology



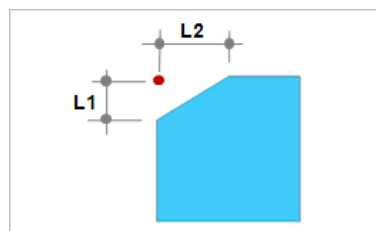
Select the edge to fillet. Only a sub edge of the solid or shell can be selected.

Input the fillet radius (R1) at the Start point and the fillet radius (R2) at the End point of the edge to be filled.

The edge direction is not shown, but the user can check the preview button () to preview the shape.


Chamfer

Methodology



Select the edge to chamfer. Only a sub edge of the solid or shell can be selected.

Input the length (L1) at the Start point and the length (L2) at the End point of the edge to be chamfered.

The edge direction is not shown, but the user can check the preview button () to preview the shape.

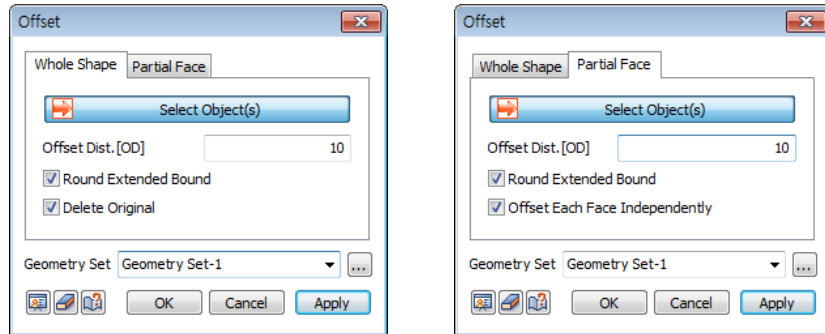


2.9 Offset

Overview

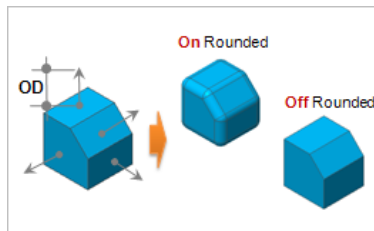
Create a shell by offsetting the whole or parts of a solid or shell.

- Whole shape
- Partial face



Whole shape

Methodology



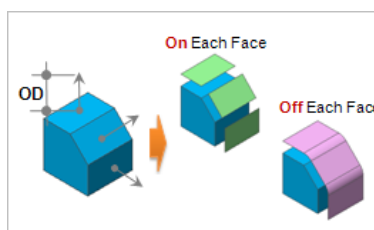
Offset all the shapes (Solid, Shell, Face) of the selected object by the same distance (Offset Distance (OD)).

Check the [Round Extended Bound] to connect any disconnected offset faces of the object using curved faces. The offset distance (OD) is the radius used for the fillet.

Check the [Delete original] to remove all target objects used in the offset process, leaving only the final shape.

Partial Face

Methodology



Offset parts of the selected object.

Check the [Offset Each Face independently] to offset each face separately. Un-checking this option and selecting multiple connected faces creates a shell type offset face.

Tip

When offsetting only some shells of a compound shape from the select face tab, the faces diverge when

offset.

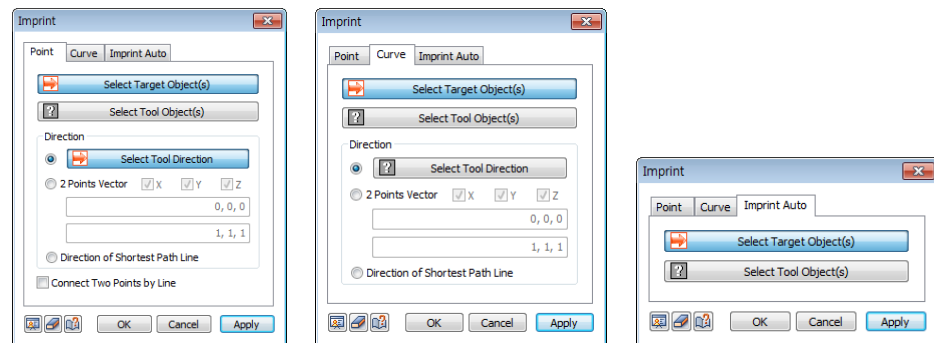
Entering a positive offset distance, the offset is performed in the normal direction to the face. So when offsetting individual faces separately, it is good to use the preview function for each face to check whether it has been offset properly. If the shell has been created by joining or connecting faces, the program adjusts the different normal directions into one direction to ease the operation.

2.10 Imprint

- Imprint – Point
- Imprint – Curve

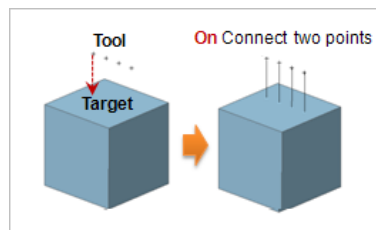
Overview

Create a curve or point on a specified face based on the projected shape.



Imprint Point

Methodology



Select the imprint face and point. The imprint direction can be set using the following 2 methods.

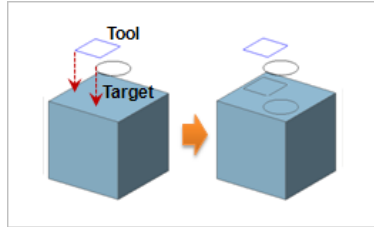
[Select Direction] : Determine the direction vector for imprint. The user can select the datum axis, datum plane, face or edge.

[2 point vector] : Determine the direction vector that is to be the reference axis of the imprint by entering the coordinates of its Start and End points. The user can also directly click on the work screen to specify the Start and End points.

Check the [Connect two points by Line] option to draw a line connecting the origin point and the projected point.

Imprint Curve

Methodology



Select the imprint face and point. The imprint direction can be set using the following 3 methods:

1. [Select Direction] : Determine the direction vector for imprint. The user can select the datum axis, datum plane, face or edge.
2. [2 point vector] : Determine the direction vector that is to be the reference axis of the imprint by entering the coordinates of its Start and End points. The user can also directly click on the work screen to specify the Start and End points.
3. [Direction of the Shortest Path Line] : Project in the shortest distance direction between the tool shape and the target shape.

Imprint Auto

Methodology

Select the imprint faces, lines and points to automatically perform imprint.

If the line passes through multiple solids, using the automatic imprint function divides the line for every solid and the intersections are automatically imprinted.

Geometry set

Register the created lines or points on the Geometry set. The user can specify the name of the Geometry set.

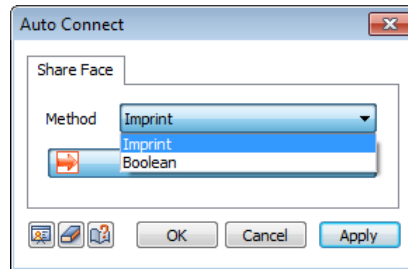
2.11

Auto Connect

►Automatically creates shared face

Overview

Automatically creates a shared face between objects.



Methodology

Select objects to automatically correct a shared face between them. This function facilitates the crossover operation between solids and one of the following operations is automatically performed.

- ✓ Imprint : In case of solid and solid in contact
- ✓ Boolean : Solid within a solid(embed) or partially included - Intersection set

Tip

For the set intersection function, the solid shape changes with the selection order. However, in the case of automatic connection, the program determines and creates the shared face so that the selection order does not change the shape. Hence, it is useful to use the Geometry > Boolean > Solid > Cut function if the user wants to include sections that are not in contact.



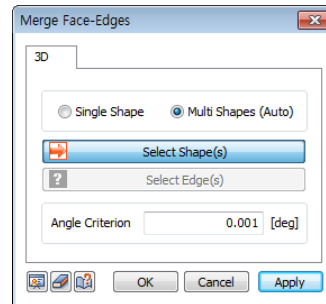
2.12

Merge Face-Edge

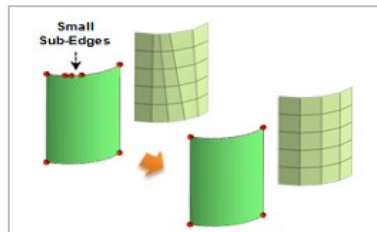
Overview

Merge the outline of a selected face into one edge while keeping the face.

►Merge face-edge



Methodology



Perform the merge face-edge operation on a single or multiple shapes. For a single shape, the merge operation is only available for a single face; for multiple shapes, the operation is automatically available for all edges included in the selection criteria.

The merge operation is performed only when the angle between the selected edges are within the criteria. The merging edges need to be connected to other edges for the operation.

Tip

The merge face-edge operation does not work on edges existing on a node (an edge of length 0). These edges are not common but can happen during the modeling process. In this case, the join edge function is impossible and needs to be modified using Create mesh and manual operations such as Move node.

If the merging edge is composed of 2 short edges on one side and one long edge on the other side in the suturing process, the long edge is divided to fit in with the 2 short edges, such that all the shared edges are in the same phase for the fuse operation. Hence, when performing the merge face-edge operation after the fuse process, the shell edge can be formed too short by mistake if the adjacent edge is short. To prevent this problem, it is possible to perform the fuse process before the merge face-edge operation. For large shell models, the merge face-edge operation may be slow and hence it is efficient to perform partial fuse and merge face-edge operations at the same time.

The Start point of the object is shown as a red circle and the End point a blue circle.

2.13 Bedding Plane

Overview

A Wizard that can be used to create multiple planes on the 3D space.

► Bedding plane

	Plane Name	Depth (m)
1	Bedding Plane-1	10.2400
2	Bedding Plane-2	0.2350
3	Bedding Plane-3	10.7800
4	Bedding Plane-4	-3.4500
5		

Methodology

Define the plane name and specify the name and position of the irradiation hole. Each irradiation hole can be added by clicking the Add button, and the depth for each plane is directly input. The depth is input with reference to the GCS and is cumulated under the ground surface. For example, inputs of -10m ground surface, 30m soil plane and 60m weathered rock indicate that the soil layer is located 40m below the ground surface and the weathered rock layer 70m below the ground surface.

The position and number of irradiation holes need to be larger than 1 and 3 respectively to create a bedding plane and hence, 3 or more columnar sections need to be input when creating a surface using multiple columnar sections.

The separation distance creates bedding planes by separating the columnar sections by a certain distance.

Import

The information of bedding planes can be defined with an Excel file in the 'Bedding Plane Wizard'. The information of several boreholes can be imported at one time. The bedding planes will be created by correction of borehole depth based on the 'Plane Name'.



	A	B	C	D	E	F	G
1	[m]						
2		Borehole Name	BB-1	BB-2	BB-3	BB-4	BB-5
3		Location	10,-5,-10	30,35,55	-20,-35,27.5	-50,57.23,10.87	-97.61,0,65.18
4							
5	No.	Plane Name	Depth				
6	1	Bedding Plane-1	-3.5	5.235	2.35	7.2	10.24
7	2	Bedding Plane-2	-5.25	2.983	7.63	-10.3	0.235
8	3	Bedding Plane-3	-10.37	8.23	2.55	12.5	10.78
9	4	Bedding Plane-4	3.75	0	-8.52	7.5	-3.45
10							

Geometry set

The created bedding planes and columnar sections are registered on the Geometry set, under the model works tree, as a face and line shape respectively. The user can specify the name of the Geometry set.



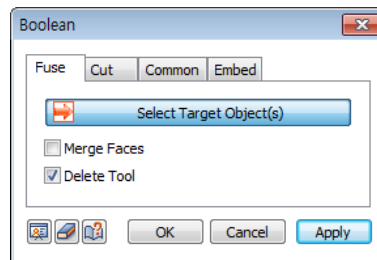
Section 3 Boolean

3.1 Solid

Overview

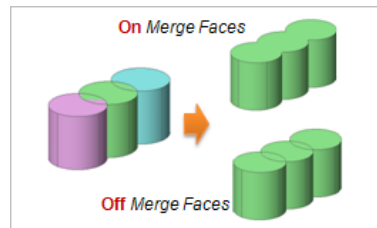
Perform boolean operations on selected shapes.

► Boolean



Fuse

Methodology



Merge shapes using the [Fuse] function. It is mostly applied to solids and solids and the outer boundary lines of each shape remains. However, if the [Merge faces] option is on, the objects are defined as a single face and the outlines are automatically removed.

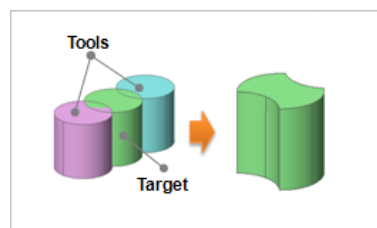
Tip

For general cases, the [Fuse] function is applied to solids.

- Fuse of line(edge, wire) and line(edge, wire) : All lines are cut at the intersection points and form a single wire. If the lines are not connected, the operation forms a compound shape.
- Fuse of face(face, shell) and face(face, shell) : Does the same function as Geometry > Boolean > Surface > Sew

Cut

Methodology



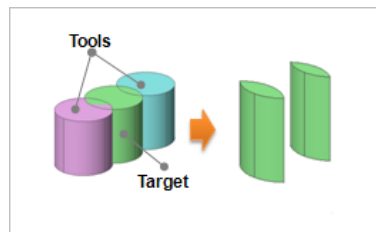


Remove the overlap between the target shape and tool shapes using a [Cut] function. It is mostly applied to solids and solids.

Tip

For general cases, the [Boolean > Cut] function is applied to solids.

- Cut of master line (edge, wire) and tool shape (face, shell) : Divide the edge at its intersection with the shape, creating wires.
- Cut of master shape (face, shell) and tool shape (face, shell) : The overlapping shape is removed from the master shape.
- Cut of master shape (edge, wire) and tool shape (solid) : The intersection with the solid is removed from the shape.

Common**Methodology**

Remove everything but the overlap between the target shape and tool shapes from the [Common] function. It is mostly applied to solids and solids. If the [Common] is applied to two faces, it may create an inappropriate shape.

Tip

For general cases, the [Boolean > Common] function is applied to solids.

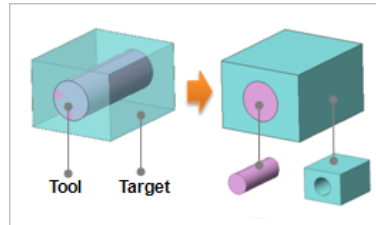
- Common of line (edge, wire) and shape (face, shell) : Divide the edge at its intersection with the shape, creating wires.
- Common of shape (face, shell) and shape (face, shell) : Only the overlap between the shapes remain as a shell.
- Common of shape (edge, wire) and shape (solid) : The overlap between the shape and solid remain as a shell.

For all cases, the results are the same



Embed

Methodology



Embed the target shape on the tool shape. It is often used when modeling a solid element that has another solid element with a different material embedded in it. The [Common] function is done between the target shape and the tool shape and the results are embedded into the target shape.

Delete Tool

Remove the tool shape after each operation.

Tip

1. Using this function on Points or Compounds can create an incorrect shape and it is not recommended.
2. [Fuse], [Cut] and [Common] functions are greatly affected by shape error. The shape error determines whether a shape is intersecting an adjacent shape and, as the compound modeling process continues, the errors that may cumulate and cause problems. If the [Fuse], [Cut] and [Common] set operations do not function properly, use the Boolean > Surface > Sew function to correct for errors and then re-perform the operation.

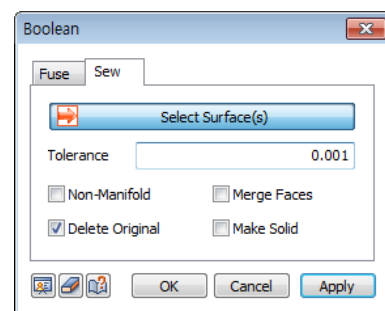
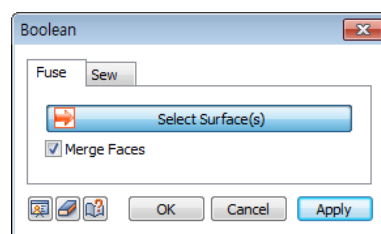
3.2 Surface

Overview

Select independent faces to create a single shell.

► Boolean – Fuse

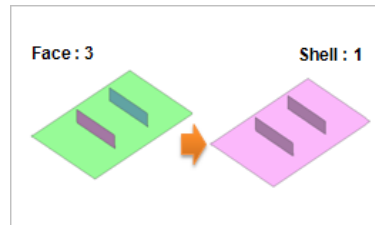
►► Boolean – Sew





Fuse

Methodology



Merge the faces (shell, face) that overlap, pass through or have no fuse operation into a single shell using the [Fuse]. Check the [Merge faces] option to merge outlines to create a face when possible. The outlines are deleted afterwards.

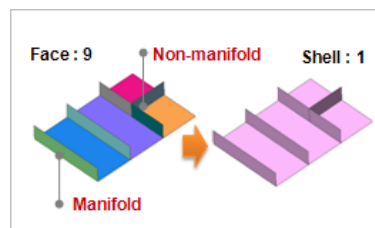
Tip

It is possible to create a shell using the Fuse function for adjoined faces, but this method may take too long and using many Fuse commands makes the shape unstable. It is recommended to use the Sew function for modeling. However, for cases where two shapes are not adjacent and do not overlap nor pass through each other, the Sew operation is not possible. In such case the shell needs to be created using the Fuse function.

Fuse is a simple union set operation that cannot be used when two faces are apart. When two faces pass through each other, the faces are divided with reference to the intersection line and each face becomes a sub-shape of the shell. Hence, if two faces pass through slightly, a very long and thin face maybe included in the shell after the Fuse operation. It is best to use the delete sub-shape menu and delete faces from the shell that are below a certain size.

Sew

Methodology




Combine independent faces (shell, face) into one shell. This function is used when the faces (shell, face) meet at the boundary edge with no overlap.

[Tolerance]

The allowable limit used in the Sew operation. If the gap between the face outlines is less than the tolerance, it creates a single shell without a free edge.

Checking the [Non-Manifold] option sews non-manifold faces that have three or more faces meeting at a single edge. Uncheck the option to not fuse Non-manifold faces.

Checking the [Make Solid] option automatically changes a perfectly closed shell into a solid.

Press the preview button [] to see the results of the Sew operation on the selected shape (shell, face). The areas displayed in dark red lines are the B-Spline sections that have not undergone the sew operation because they are not within the allowable limit. In this case, gradually increasing the limit and previewing the results repeatedly can prevent B-Splines from being displayed. Pressing the confirm or apply button with the B-Spline included creates a shell with a B-Spline included.

Be aware that a very large tolerance can create an abnormal shape.

<When fuse does not occur even with tolerance adjustment>

- Check to see if the face overlaps at that position.
- If a face is not sewed, sew the other faces first and then reconstruct the edges of the un-sewed face for another sew operation.
- It is possible to perform the sew operation without removing a local B-Spline. In this case, a seed is assigned to the B-Spline to create a mesh and the fuse node operation is performed in the subsequent mesh creating process.
- Modify the faces such that adjacent faces meet at the edges with the same length using Geometry > Surface&Solid > Merge Face-Edge. If a mesh is created with the same mesh size without fuse, the positions of the nodes are very similar. This process combines all the shared nodes of the created mesh into one using the merge node function.

Some faces are not shaded after the sew operation. This happens when there are problems in a face's configuration that create erroneous relationships with adjacent faces in the sew process, resulting in an incorrect shell. These errors can be corrected by modifying the incorrect face and then performing the sew operation. If there are problems after the suturing, the incorrect face can be removed from the shell by using the sub-shape remove function.



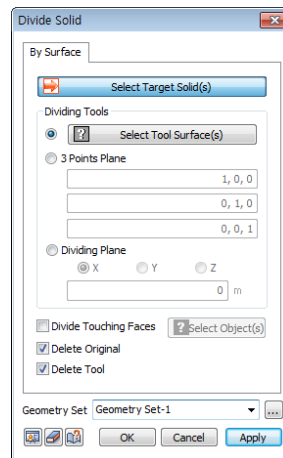
Section 4 Divide

4.1 Solid

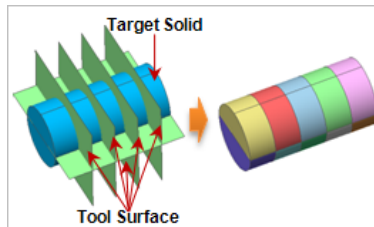
►Solid – Divide into faces

Overview

Divide a solid into faces.



Methodology



Select the target solid and divide it using the tool faces.

The division tool face can be set in 3 ways.

[Select tool surface] : Select the target surfaces directly for division. If the tool shape consists of multiple faces, it is recommended to group the faces into a single shell to obtain accurate results.

[3 points vector] : Divide the solid using an infinite face defined by specifying three points. The user can also click directly on the work screen to input the coordinates of the three points.

[Dividing plane] : Divides the solid using an infinite face created with reference to the Global Coordinate System (GCS).

Divide Touching Faces

When multiple solids are adjacent with faces in contact, dividing parts of the solid can also divide the adjacent solid and their contact face. This allows the sharing of face nodes when creating a solid.

Delete Original, Delete Tool

Check this option to delete the original target shape and tool shape after the division operation.

Geometry set

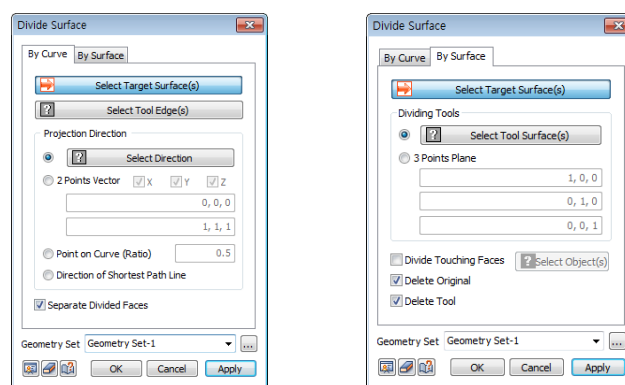
Register the created solid on the Geometry set. The user can specify the name of the Geometry set.

4.2 Surface

- Surface – By curve
- Surface – By surface

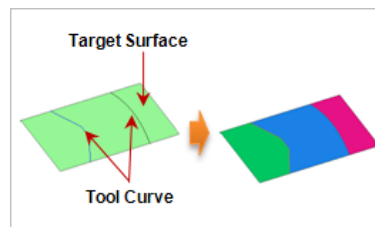
Overview

Divide multiple faces using arbitrary lines or faces.



By Curves

Methodology



Select the target surface (shell, face) and the tool curves for division. If the tool shape consists of multiple edges, it is recommended to merge the edges into a single edge instead of selecting multiple edges. If the tool curves do not exist in the same plane as the target surface, the tool edge is projected onto the target surface and then divided. In this case, the projection direction needs to be specified. The direction can be set in the following 4 ways:

1. [Projection Direction] : Determine the projection direction vector of the tool edge. The user can select the datum axis, datum plane, plane or edge.
2. [2 point vector] : Determine the direction vector by entering the coordinates of its Start and End points. The user can also directly click on the work screen to specify the Start and End points.
3. [Point on Curve] : Project an arbitrary point on the tool edge in the shortest distance direction to the target surface. The arbitrary point can be specified by a ratio of 0 to 1 from the Start point to the End point of the tool edge.
4. [Direction of Shortest Path Line] : Project in the shortest distance direction from the tool edge to the target shape.

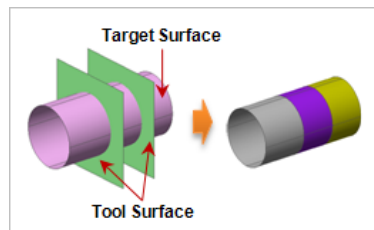


Separate divided faces

Check this option to separate the shell into face units.

By Surface

Methodology



Select the target surface (shell, face) and the tool faces for division. The tool faces can be directly selected or an infinite tool face can be created by defining the coordinates of three points.

Divide Touching faces

When multiple solids are adjacent with faces in contact, dividing parts of the solid can also divide the adjacent solid and their contact face. This allows the sharing of face nodes when creating a solid.

Delete Original, Delete Tool

Check this option to delete the original target shape and tool shape after the division operation.

Geometry set

Register the created face on the Geometry set. The user can specify the name of the Geometry set.

Tip

When working with an imported geometry data file, the shape may not divide well. The methods for dividing a face with a line are as follows:

- When the tool line cuts across the target shape completely, it has a similar effect as the imprint operation.
- Create a temporary solid by extruding the target surface using an arbitrary value and create a temporary face by extruding the tool curve. Dividing the created solid by using the created face leaves only the desired face. Using the sub-shape extend command deletes the temporary solid and the face.



Section 5 Protrude

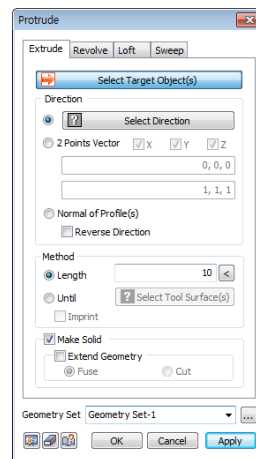
5.1 Extrude

Overview

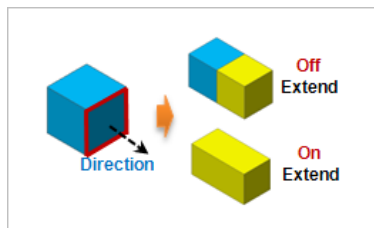
Create a solid, face and line by extruding a geometry shape (face, line, point) in a straight direction. It is possible to create a solid using a face, a face using a line, and a line using a point.

For special cases, the solid option can be set to create a solid using a closed wire or edge.

► Extrude



Methodology



Input the geometry shape (face, line, point) for the Extrude operation and enter the extend direction and length. The extend direction can be set using the following 3 methods:

1. [Select Direction] : Determine the extend direction vector of the selected section. The user can select the datum axis, datum plane, plane or edge.
2. [2 point vector] : Determine the extend direction vector by entering the coordinates of its Start and End points. The user can also directly click on the work screen to specify the Start and End points.
3. [Normal of profile] : Extend the section in the normal direction to the plane when the direction can be specified. If there are multiple sections, the shape is extended for each normal direction. If the extend section is a curved face or a straight line, the normal direction cannot be defined and it cannot be extended. Check the [Reverse Direction] option to reverse the extend direction.

[Length]

If the extend direction has a finite length (straight line edge or two-point vector), click the right "<" button to automatically enter the extend length.



Make Solid

Used on solids created by a closed line. Using an open edge or wire does not cause errors, but be aware that it may create an incorrect shape..

[Fuse]

Use the Fuse operation between solids to expand the shape during extend process.

[Cut]

Use the Cut operation between solids to divide the shape during the extend process.

Geometry set

Register the geometry shape created from the extend line command on the Geometry set. The user can specify the name of the Geometry set.



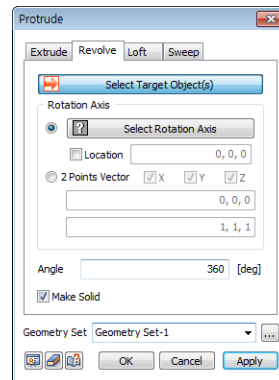
5.2 Revolve

Overview

Create a solid, face and line by rotating the geometry shape (face, line, point). It is possible to create a solid using a face, to create a face using a line, and to create a line using a point.

For special cases, the solid option can be set to create a solid using a closed wire or edge.

►Revolve



Methodology



Select the geometry shape (face, line, point) and input the revolution axis and angle.
The revolution axis can be set using the following two methods:

1. [Rotation Axis] : Determine the revolution axis for rotation for the selected section. The user can select the datum axis, datum plane, plane or edge. Check [Position] to specify the Start point of the revolution axis directly. Entering the position moves the revolution axis to that value.
2. [2 points vector] : Determine the extend direction vector by entering the coordinates of its Start and End points. The user can also directly click on the work screen to specify the Start and End points.

Make Solid

Used on solids created by a closed line. Using an open edge or wire does not cause errors, but be aware that it may create an incorrect shape.

Geometry set

Register the geometry shape created from the rotation extend line command on the Geometry set. The user can specify the name of the Geometry set.

Tip

When creating a solid or 2D mesh on a rotated body, the user can observe mesh deformation at the

vertex. This happens near the pole, which is created near the revolution axis of the rotated body. This problem cannot be solved on the geometry shape and it is best to use move node or merge node functions after creating the mesh for any modifications.

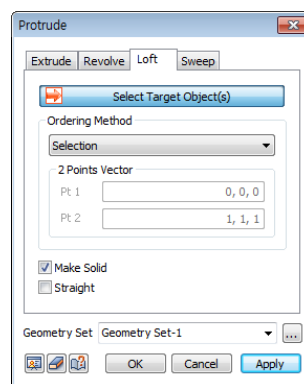
5.3 Loft

Overview

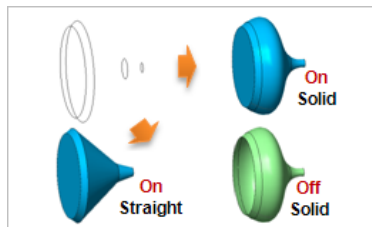
Create a shell or solid that connects the selected sections in order of selection. The line type that connects the objects can be set as a B-Spline or a straight line.

A solid can be created by selecting a closed wire or edge on the solid option.

► Loft



Methodology



Select shapes to perform loft. The selection method depends on the selection order. The order can be largely divided into [Creation], [Selection] and [Vector].

[Creation] : Select section shapes in order of connection to create a shape. The selection shapes are selected individually with the mouse.

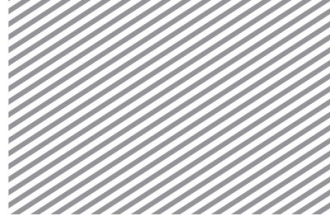
[Selection] : Select section profiles in order of connection to create a shape. The section profiles are selected individually with the mouse.

[Vector] : Select multiple sections that are connected depending on the organized vector direction of the object shape. 2 point vectors can be used to define the coordinates of the direction vectors at Pt1 and Pt2. The reference point and order of the profile is applied in the same way as organizing a section profile using the coordinate system.

Make Solid, Straight

Check the [Make Solid] option to create a solid using a closed line.

Using this operation on an open edge or wire does not cause errors, but be aware that it may create an incorrect shape.



Sections with the [Straight] option checked on will connect each selected section using straight lines, not curved.

Geometry set

Register the geometry shape created from the loft extend command on the Geometry set. The user can specify the name of the Geometry set.

Tip

A solid cannot be created, even when the solid option is on and the wires and edges are closed, if the planes cannot be created. In this case, it is recommended to create the planes individually and use the Sew operation (Main menu > Geometry > Boolean > Surface > Sew).

Tip

Sometimes, a twisted geometry can form during the extend loft operation. This is because the wires or edges selected as the section profile are not in the same direction.



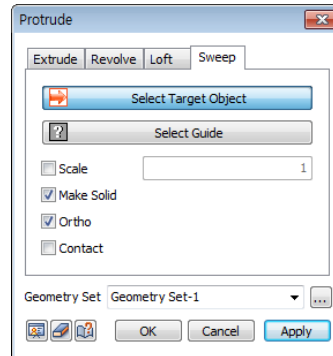
5.4 Sweep

►Sweep

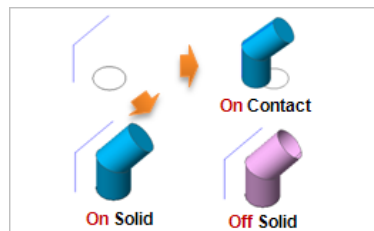
Overview

Create a face, shell or solid by extruding the selected section along a guide curve.

Working with a wire as a section profile creates a shell. Working with an edge as a section profile creates a face.



Methodology



Select the Object to perform sweep and extend along the guide curve.

[Scale] Check to input the scale size and extend with the end shape scaled to that factor.

[Make Solid] Check to extend the face to create a solid.

[Ortho] Check to extend the selected section in the direction perpendicular to the guide curve.

[Contact] Check to move the section to the start point of the guide curve and then sweep extend.

Geometry set

Register the geometry shape created from the sweep extend command on the Geometry set. The user can specify the name of the Geometry set.

Tip

The object is best created when the guide curve exists on the center position of the sweep profile.

A solid cannot be created, even when the solid option is used and the wires and edges are closed, if the planes cannot be created. In this case, it is recommended to create the planes individually and then perform the fuse process to create a solid.

Use the preview button to check the shape. If the shape is not appropriate, use the sweep mod or bend mod functions to create an appropriate shape.

Sometimes complexities in geometry will prevent the desired result from being generated through the sweep functions. In this case, use the Sweep pattern (Main menu > Geometry > Transform > Sweep-Translate) to copy the object profile using the guide curve and use the Loft extend (Main menu > Geometry > Protrude > Loft) to obtain the same result.



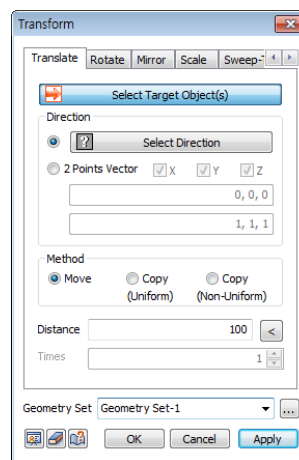
Section 6 Transform

6.1 Translate

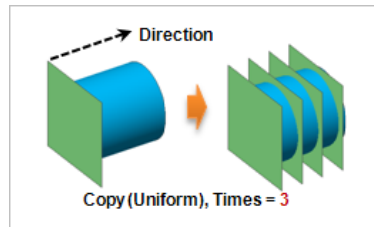
Overview

Move the object by a certain distance. This command can be applied regardless of the type of shape. The [Copy] options can be used to leave the original object and only move the copied object.

►Translate



Methodology



Select the shape to move and specify the direction. The direction can be defined in 2 ways:

1. [Select Direction] : Determine the direction vector for move. The user can select the datum axis, datum plane, face or edge.
2. [2 point vector] : Determine the direction vector to be the reference axis of the move by entering the coordinates of its Start and End points. The user can also directly click on the work screen to specify the Start and End points.

The target object can be moved/copied using [Move], [Copy (Uniform)] and [Copy (Non-Uniform)].

[Move] : Move target shape by desired distance.

[Copy (Uniform)] : The repeat option is activated and copies the target object repeatedly by the entered number.

[Copy (Non-Uniform)] : List the copy distance by a space or comma (,) and when repeating, enter number @ distance(Example : 2,3,4,4,4 or 2 3 3@4)

Distance



If the direction has a finite length (straight line edge or two-point vector), click the right "<" button to automatically enter the length.

Geometry set

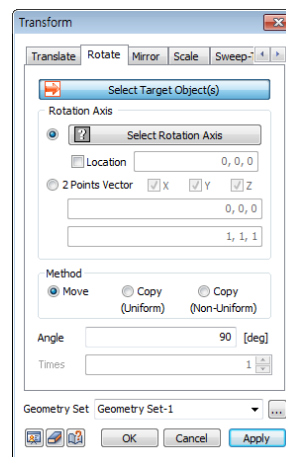
Register the geometry shape created from the move command on the Geometry set. The user can specify the name of the Geometry set.

6.2 Rotate

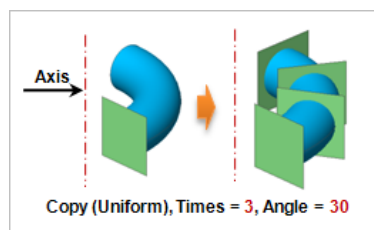
Overview

Move or copy the object by rotation. This command can be applied regardless of the type of shape and the [Copy] option can be used to leave the original object and only move the copied object.

►Rotation move



Methodology



Select the shape to rotate and specify the revolution axis. The revolution axis can be defined in 2 ways:

1. Check [Location] to specify the Start point of the revolution axis directly. Entering the position moves the revolution axis to that value.
2. [2 point vector] : Determine the direction vector for rotation extend by entering the coordinates of its Start and End points. The user can also directly click on the work screen to specify the Start and End points. The target object can be moved/copied using [Move], [Copy (Uniform)] and [Copy (Non-Uniform)].

[Move]

Enter the angle directly to rotate.

[Copy (Uniform)]

The repeat option is activated and rotation copies the target object repeatedly by the entered number.

[Copy (Non-Uniform)]



List the rotation angle by a space or comma (,) and when repeating, enter number @ distance (Example : 10,20,25,25,25 or 10 20 3@25).

Geometry set

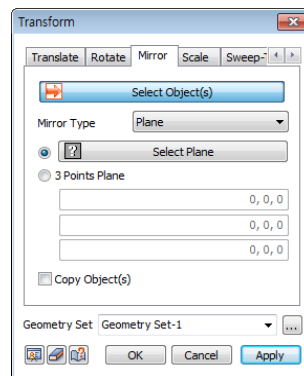
Register the geometry shape created from the rotate command on the Geometry set. The user can specify the name of the Geometry set.

6.3 Mirror

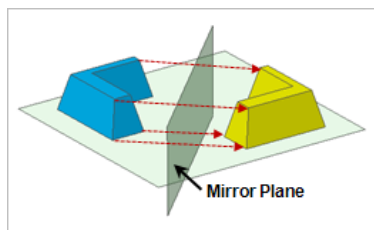
Overview

Move the target object. This command can be applied regardless of the type of shape and the [Copy] option can be used to leave the original object and only move the copied object.

►Mirror



Methodology



Select the shape to mirror and specify the mirror type.
The mirror type can be defined by [Point], [Axis] and [Plane].

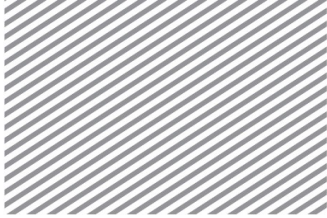
[Point] type : Mirror the selected point with reference to a point. The mirrored point can be selected directly by clicking on the work screen or checking the coordinate system option and entering the coordinates.

[Axis] type : Mirror the selected axis with reference to an axis. The mirror axis can be selected directly by clicking on the work screen or checking the Two-point vector option and entering the Start and End point coordinates of the direction vector.

[Plane] type : Mirror the selected plane with reference to a plane. The mirror plane can be selected directly by clicking on the work screen or checking the Three-point plane option and entering the three coordinates to define an infinite plane.

Geometry set

Register the geometry shape created from the mirror command on the Geometry set. The user can specify the name of the Geometry set.

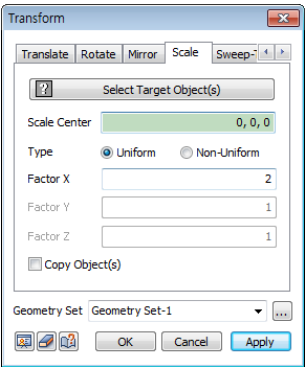


6.4 Scale

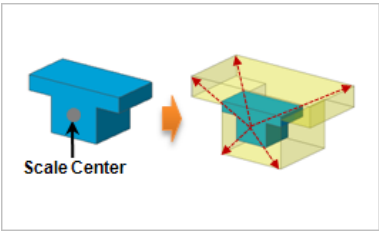
Overview

Scale the target object with reference to a point. This command can be applied regardless of the type of shape and the [Copy] option can be used to leave the original object and only move the copied object.

►Scale



Methodology



Select the shape to scale and specify the reference point coordinates to scale up or down.
There are 2 scale methods; [Uniform] and [Non-Uniform]
[Uniform] : All axis's are scaled by the same amount.
[Non-Uniform] : Each axis direction (GCS standard) can be scaled by a different factor.

Geometry set

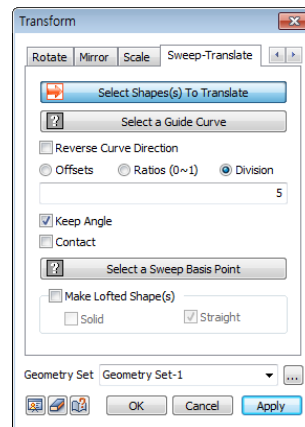
Register the geometry shape created from the scale command on the Geometry set. The user can specify the name of the Geometry set.

6.5 Sweep-Translate

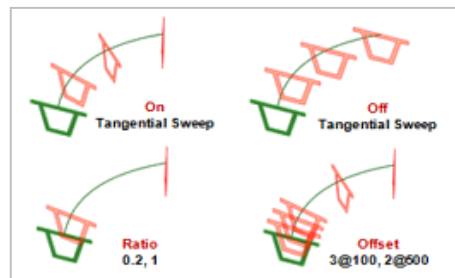
Overview

Copy the target object along a guide curve. This command can be applied regardless of the type of shape.

►Sweep-Translate



Methodology



Select the shape to move and specify the guide curve (edge or wire).
Check [Reverse Curve Direction] to move in the opposite direction of the guide curve.

There are 3 move methods: [Offsets], [Ratio (0~1)], [Division].

1. [Offsets] : Specify the offset interval and input the move distance. If the entered move distance is longer than the guide curve, the shape does not move past the end of the guide curve.
Enter a space or comma to enter multiple distances to copy and move multiple shapes.

2. [Ratio (0~1)] : Specify the guide curve ratio and input the move distance. The ratio at the Start point of the guide curve is 0 and the ratio at the End point of the guide curve is 1. Enter multiple ratios to copy and move multiple shapes.

3. [Division] : Specify the number of guide curve divisions and input the move distance. Copy and move the equally divided edge to the division position.

Check the [Keep angle] option to apply the curves of the guide curve during sweep pattern move and check the [Contact] option to move the guide curve of the target object to the Start point.

Select a Sweep Basis point

The reference point used when applying a vertical, line contact option on a plane. The reference point must be a particular point on the shape.

Make Lofted Shape



It is convenient to use this option instead of the loft extend option when creating a shape by interpolating a section along a guide curve with discontinuous curvature. The solid can be made in line form. The resulting shape is created and the original shape is not copied.

Geometry set

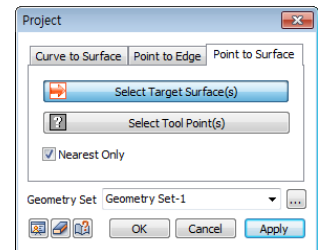
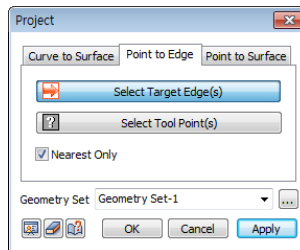
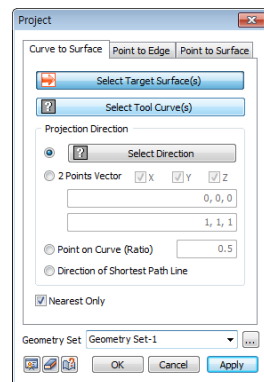
Register the geometry shape created from the sweep pattern command on the Geometry set. The user can specify the name of the Geometry set.

6.6 Project

- Project – Curve to Surface
- Project – Point to edge
- Project – Point to surface

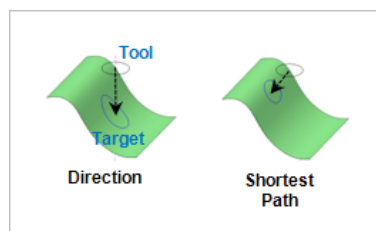
Overview

Project the selected shape. The user can project [Curve to Surface], [Point to Edge] or [Point on Surface].



Curve to Surface

Methodology



Select the target surface and tool line for projection.

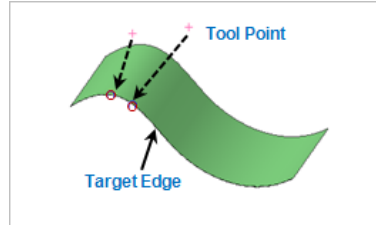
The projection direction can be set using the following 4 methods:

1. [Select direction] : Determine the projection direction vector. The user can select the datum axis, datum plane, plane or edge.
2. [2 point vector] : Determine the direction vector of the projection axis by entering the coordinates of its Start and End points. The user can also directly click on the work screen to specify the Start and End points.
3. [Point on Curve (Ratio)] : Select a point on the line. Project the shortest distance direction from the selected point to target surface.
4. [Direction of Shortest path Line] : Project in the shortest distance direction from the tool edge to the target shape.

Point to Edge



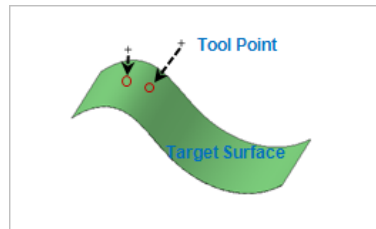
Methodology



Select the target edge and tool point for projection. The selected point is projected in the vertical direction.

Point to Surface

Methodology



Select the target surface (shell or face) and tool point for projection. The selected point is projected in the vertical direction.

Nearest Only

Projection is performed infinitely in the selected direction and if the target shape is a curved plane or shell, it can meet the target shape more than twice. Check this option to select only the projected shape closest to the tool shape.

Geometry set

Register the created line or point on the Geometry set. The user can specify the name of the Geometry set.

Tip

There is no function to create a straight line and project a point at the same time. Imprinting the point and connecting it to a line can be a similar function.

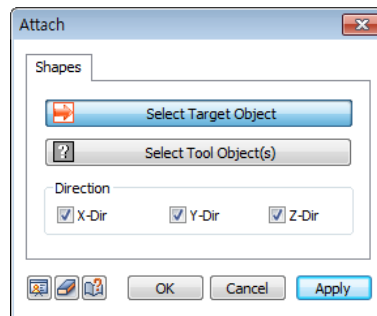
6.7 Attach

Overview

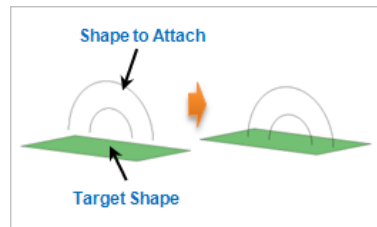
Attach the target shape and tool shape. Here, the selected shape moves in the direction with shortest distance between it and the shape to which it is being attached.

If there are more than two tool shapes, the attaching shape moves to the closest tool shape.

►Attach



Methodology



Select the target shape and the shape to attach. All target shapes can be selected if they are independent shapes.

The attach direction can be selected from [X-Dir], [Y-Dir] or [Z-Dir].



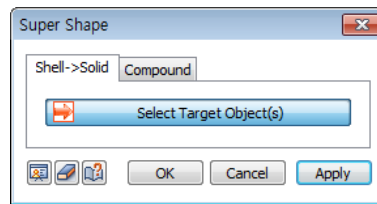
Section 7 Sub Shape

7.1 Super Shape

► Super Shape

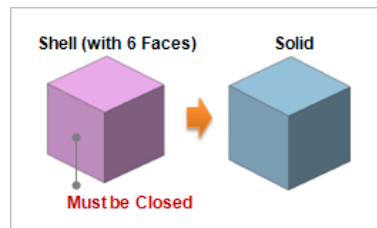
Overview

Create a solid or compound shape using selected target shapes.



Shell → Solid

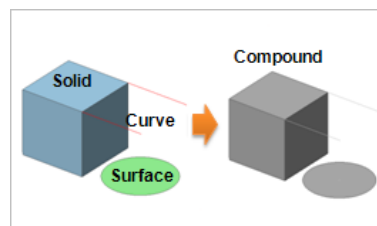
Methodology



Selecting closed shells or faces and running this option automatically fills the interior and creates a solid.

Compound

Methodology



A compound is a group of shapes that cannot be defined as a single shape (point, line, face, solid). Multiple shapes can be selected to form a compound.

Compounds are created during the modeling process when a shape is completely divided into two or more shapes. For example, when the 4 side faces of a cube are removed, the resulting compound is formed by two completely separate top and bottom faces.

A compound can also be created from grouping un-connectable shapes. For example, performing a crossover operation on two separated solids can result in a compound.

Hence, if a compound is created, acknowledge it as a set of un-connectable shapes.

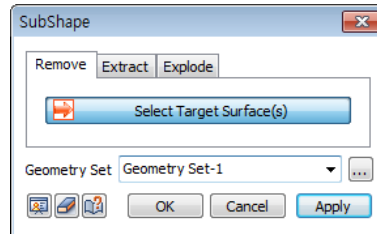


7.2 Remove

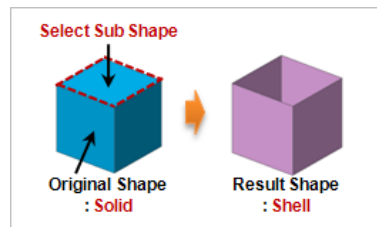
Overview

Remove the desired outer face of a selected shell or solid.

►Sub Shape - Remove



Methodology



Select the faces to delete and press the [OK] or [Apply] button to delete the selected faces. For solids, deleting a face creates a shell. For faces, deleting a face leaves the outline wires.

Geometry set

Register the created shape on the Geometry set. The user can specify the name of the Geometry set.



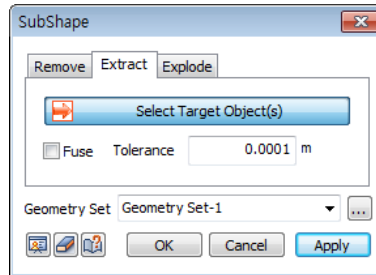
7.3

Extract

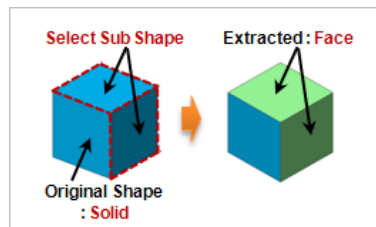
Overview

Extract a geometry shape (face, line, point) of the selected shape to create an independent shape.

► Sub Shape – Extract



Methodology



Select the geometry shape (face, line, point) to extract. A face can be extracted from a face, a line from a face and a point from a line. If the extracted shapes are within the error range, check the [Merge] option to merge the shapes into a single shape. For example, an object that is extracted with two adjacent faces, such that the error between the outlines is small, can create a single shell without any free edges.

Geometry set

Register the created shape on the Geometry set. The user can specify the name of the Geometry set.

Tip

Disassembling a face can be used when an edge of an existing face is needed during the modeling process. In this case, the face is deleted but using the extract command leaves the face intact and extracts the independent edge.

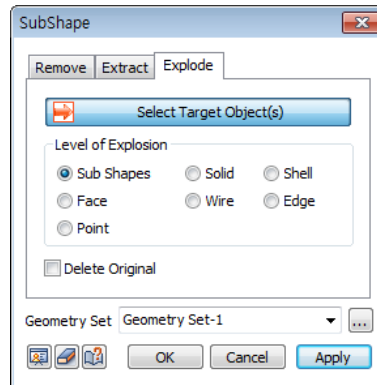


7.4 Explode

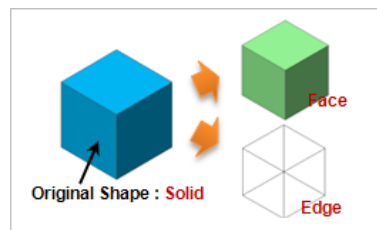
Overview

Disassemble a selected shape into its sub-shapes.

►Sub Shape – Explode



Methodology



Select the shape and set the disassemble level to disassemble into its sub-shapes.

The disassemble level can be selected from [Sub Shapes], [Solid], [Shell], [Face], [Wire], [Edge] and [Point].

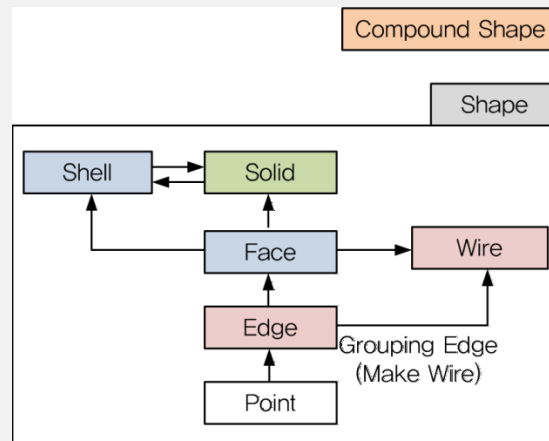
Disassemble is applied to target shapes that have a higher rank than the disassemble level. In other words, if the shape is solid, it can be divided into shapes that have a lower rank than a solid such as [Shell], [Face], [Wire], [Edge] or [Point].

[Sub Shapes] is the option that disassembles the shape by the geometry rank just below it.

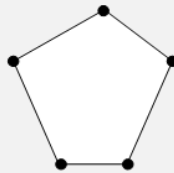
Compound shapes are a set of simple shapes and applying [Sub Shapes] disassembles each shape in the compound individually.

Tip

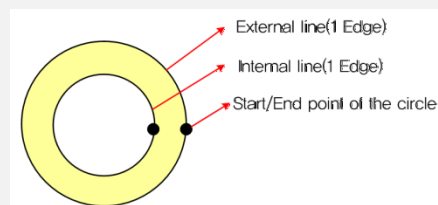
The geometry ranking is as follows: [Solid] > [Shell] > [Face] > [Wire] > [Edge] > [Point].



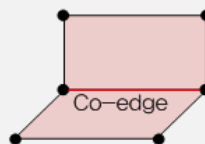
For example, the sub-shapes of a shape consisting of faces and solids are as follows.
The properties of the selected shape can be verified from Tools > Show/hide > Properties.



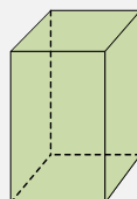
Shape		Face
Sub-shape	Wire	1
	Edge	5
	Point	5



Shape		Face
Sub-shape	Wire	2
	Edge	2
	Point	2



Shape		Face
Sub-shape	Face	2
	Wire	2
	Edge	7
	Point	6



Shape		Face
Sub-shape	Shell	1
	Face	6
	Wire	6
	Edge	12
	Point	8



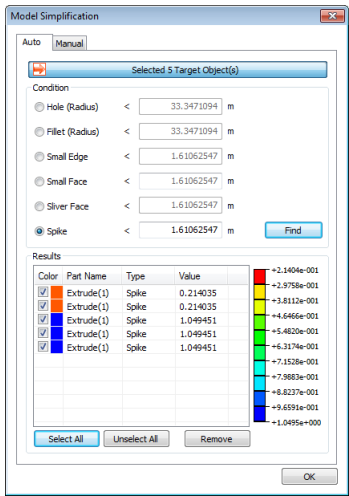
Section 8 Remove

8.1 Face/Edge

Overview

Find and delete faces/Edges that are smaller in value (area for face, length for edge) than the input value.

►Model Simplification >
Auto



Methodology

Select the geometry shape to check and input the micro-line/micro-face threshold values. Press the find button to automatically find faces or lines with areas or lengths less than the threshold value. The user can check and delete the selected shapes.

When using an imported geometry shape data file, very small faces (especially, long faces with small widths) can also be imported, which leads to a reduction of work efficiency and the creation of incorrect shapes. The best way to solve this problem is to erase the problematic faces and restructure the shape using adjacent faces. However, if the model is too large and the problematic faces cannot be checked individually by the naked eye, it is efficient to use this option to adjust the allowance and automatically delete small faces.

When using many edges to create a face, some faces may not form. This problem can be caused by very short edges that overlap. In this case, the function can be used to find and delete such micro-edges when creating a face.



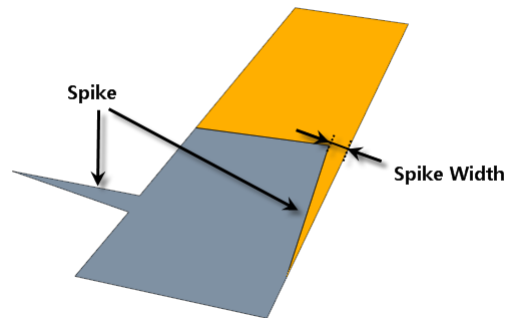
Condition : Find and remove selected entities automatically

- ✓ Hole (Radius) : Input radius of hole
- ✓ Fillet (Radius) : Input radius of fillet
- ✓ Small Edge : Input the length of edge
- ✓ Small face : Input the length for the longest edge of face
- ✓ Silver Face : Input the width of strip
- ✓ Spike : Input the width of spike

Results : All entities which meet criteria are listed in the dialog

- ✓ Selected entities will be highlighted in the model view
- ✓ Double click on the selected entity to fit zoom to window

►Spike Width





8.2 Remove Manual

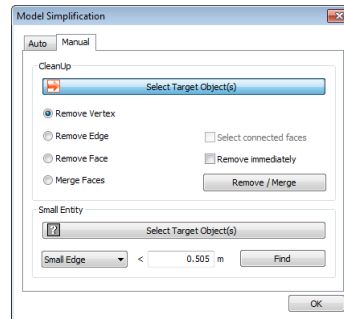
Overview

Infinitesimal lines or faces are main sources of error when creating an element. Unintended infinitesimal faces can be created during repeated divisions or shared surface creation during complex geometry modeling.

Elements are created by linear interpolation of the input size with reference to the division point of the geometry shape. Here, the infinitesimal lines or faces, which cannot create an element, need to be modified by combining with adjacent shapes, deleting or regenerating. This function searches for infinitesimal lines or faces with reference to the input size and deletes or merges them.

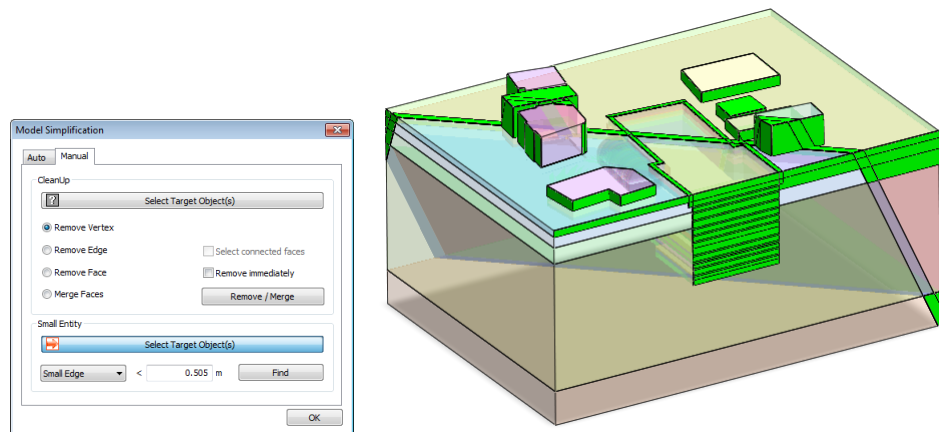
8.1 Face/Edge can only be performed on created faces and lines, and Manual delete searches and modifies faces/lines that are sub-shapes of a solid.

►Model Simplification >
Manual



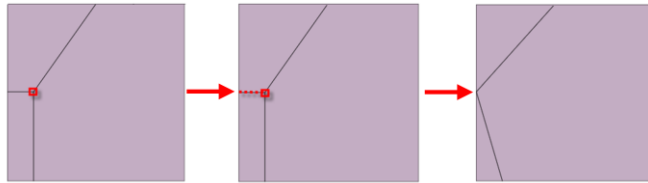
Methodology

Select infinitesimal line or face shapes manually to delete them from the specified geometry, or automatically figure out the total number and positions of the infinitesimal faces on the solid, with reference to the will-be created element size by entering the infinitesimal area directly. Input the infinitesimal length and press the Find button to highlight faces that are smaller than the input value, as shown in the figure below.

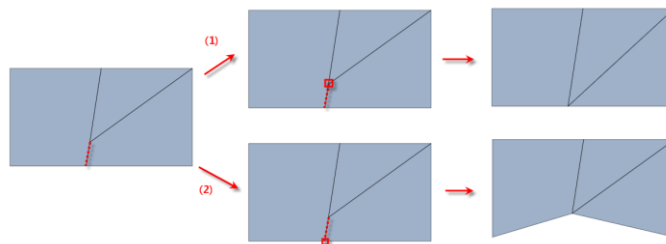


It can also delete the sub-shapes of the solid and stretch the adjacent shapes to fit automatically. For example, deleting a small hole on the solid creates a solid that automatically fills the deleted hole. Selecting a particular face (sub-shape) of a solid cannot maintain the solid shape and so, it is not automatically deleted. In this case, the specified solid needs to be decomposed into faces.

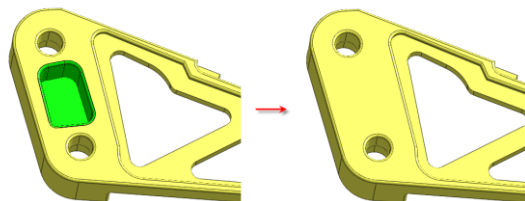
- ✓ Remove Vertex : Remove the shortest edge of ones which are connected to the selected vertex



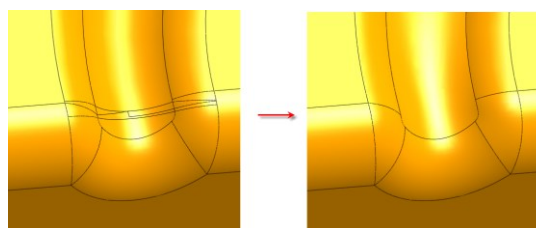
- ✓ Remove Edge : Remove selected edge and corresponding vertex at the same time



- ✓ Remove Face : Remove selected face and merge



- ✓ Merge Faces : Merge faces with removing selected edge





8.3

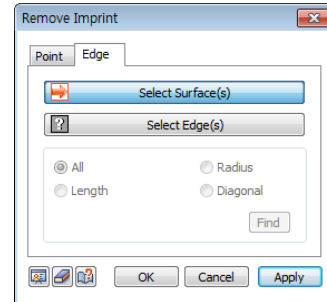
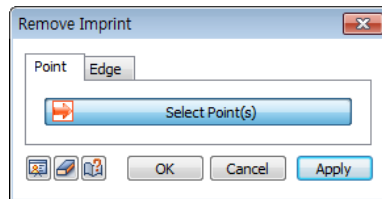
Imprinted Object

Overview

Delete edges or points that exist in an interior object inside the selected geometry shape.

►Imprinted Object – Point

►►Imprinted Object- Edge



Point

Select points created by imprint and press the apply button to automatically erase them.

Edge

Select the surface that contains the interior edges and then select the target edges. Select one of the following methods to automatically find and select interior edges..

[All] : Automatically select all selectable interior edges.

[Radius] : Input the radius and select all interior edges within that radius. It can be linked with the arc shaped interior edges.

[Length] : Input the edge length and select all interior edges that are shorter than the specified value.

[Diagonal] : Input the diagonal length of the boundary box and select interior edges that have a shorter diagonal length than the specified value.



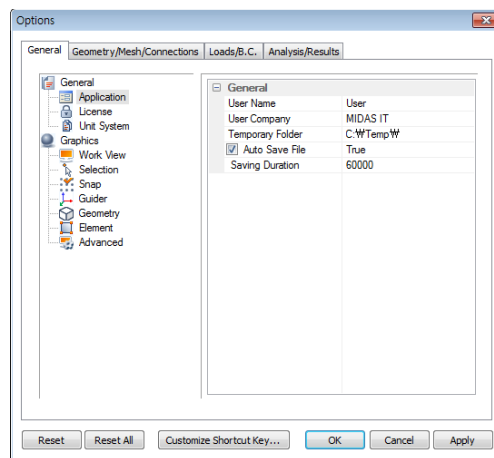
Section 9 Tools

9.1 Options

Overview

Set the general environment of the program.

The sections are General, Geometry/Mesh/Connections, Loads/Boundary Condition, Analysis/Results.



General

[General]

- Application : Specify the user name, company name, temporary file folder, file save interval, etc.
- License : Authorize the program license. The Stand Alone (USB Hard lock) method and the Web Authorization method are supported.
- Unit system : Specify the force/length/time unit system used for analysis. Use the Unit converter at the bottom right to convert the units before or after analysis.

[Graphic]

- Work View : Set the options for the screen.
 - The mouse operation can be used to fit the utilized 3D CAD wheel operation.
 - The Smoothing Surface Rendering method regulates the edge line tessellation of a cylinder shaped curved surface. The available levels are from 1 to 5, with a higher quality and smoother surface as the level gets higher.
 - The Shape of the Dynamic View is the option for the displayed modeling shape on the work window during view rotation.
- Selection : Specify the options for the model part.
- Guider : Specify the options for the screen guider.
- Geometry shape : Specify the color for each geometry shape type.
- Element : Specify the color for each element type.
- Advanced : Adjust the advanced options for the graphic setting. It determines whether to show the shadow and controls its shade. The available levels are from 1 to 5, with darker information shown as the level gets higher.



Geometry/Mesh/Connect**[Geometry]**

- Common : Adjust the options used on all geometry shapes.
- Import : Set the import option.

[Mesh Set]

- Common : Adjust the basic options used on the mesh feature.
- Size Control : Adjust the color of the symbol shown when the seed function is used.

Load/Boundary Condition

[Coordinate system] : Specify the color of the coordinate system symbol. Symbol 1, Symbol 2, and Symbol 3 represent the X axis, Y axis, and Z axis respectively.

[Mesh] : Specify the size of the node and element number.

[Static load] : Specify the size and color of the static load symbol.

[Dynamic load] : Specify the size and color of the dynamic load symbol.

[Boundary condition] : Specify the color of the boundary condition symbol.

Analysis/Results**[Analysis]**

- Number of processors : Specify the number of CPUs used for analysis. For a dual core CPU, input 2 processors and for a quad core CPU, input 4 processors to increase the analysis speed.
- Element Formulation : Specify the formula applied to the element.
In terms of speed, a faster solution can be obtained in order of Reduced (Efficiency) > Standard (Stability) > Hybrid (Accuracy). In terms of accuracy, a more accurate solution can be obtained in order of Hybrid (Accuracy) > Reduced (Efficiency) > Standard (Stability).
- Equation Solver (Structural) : Specify the method for solving a finite element simultaneous equation. If the setting is automatic, the program automatically determines one of the following methods: Multifrontal, Dense matrix or AMG(Algebraic Multi-Grid).
- 2D Element Setting (Structure) : The [Unique Shell Normal Generation] function judges two adjacent shell elements to have two different normal directions when the value between the normal direction vectors are larger than the input value. Because the element size is relatively larger than the curvature, if this value is increased for a rough curved mesh, a smooth contour that considers the curvature of the geometry shape can be calculated.
The [Consider Drilling DOF(degree of freedom)] option calculates the stiffness of the in-plane deformation by considering the rotation about the out-plane axis (drilling degree of freedom).



[Result]

- **General** : Input an analysis result that is extremely small and can be considered as 0. The default value is set as $1e-12$, and results lower than this value is considered as 0.
- **Contour** : Determine the various settings for contour representation of analysis results.
- **Vector** : Display results that are represented with a (V) using vectors. Here, specify how to represent the vectors.
- **Deform** : Specify the basic settings for checking the deformed shape of analysis results.
- **No result entity** : Specify how to represent the no result entities when displaying analysis results.
- **Diagram** : Determine the basic settings for diagrams.
- **Graph** : Select whether to show the graph.
- **Animation** : Specify the location in which the animation image types and files are saved.
- **Legend** : Specify the background color and number of result bands displayed on the screen.

Customize Shortcut Key

The user can call up frequently used commands by defining a shortcut key.



9.2

Terrain Geometry Maker

Overview

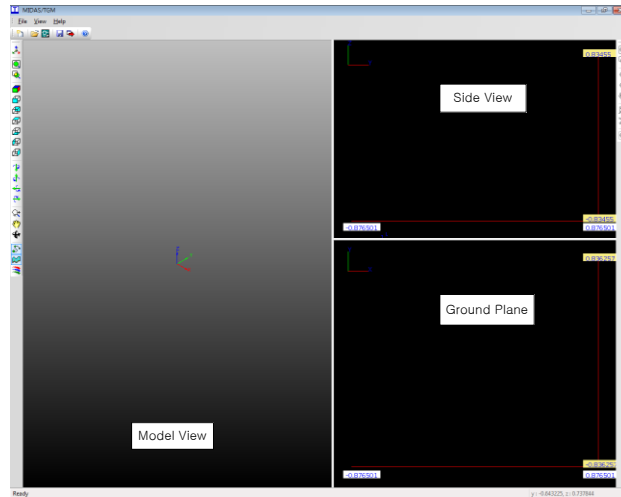
The Terrain Geometry Maker (TGM) function can be used to model a terrain geometry using a CAD DXF file. When using FEA NX, first use the CAD DXF file to set the bounds of the analysis area and save it as a file (*.tms format) that can be used in the FEA NX. Import this file to FEA NX to create the terrain geometry.

Methodology

Step 1. Tools > Terrain Geometry Maker > Terrain Geometry Maker

Use the CAD DXF file to set the bounds of the analysis area and save it as a file (*.tms format) that can be used in FEA NX. Activating this function creates a new window called MIDAS/TGM.

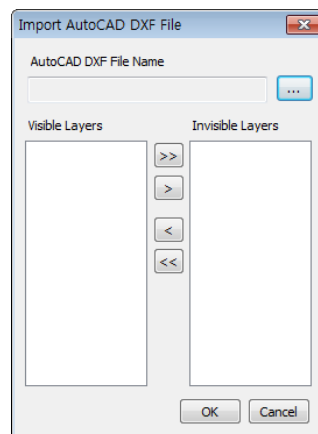
► MIDAS/TGM



Step 2 : File > Import DXF

Import the AutoCAD DXF file.

► Import AutoCAD DXF file



[Visible layers]


The layers that are needed for the terrain geometry out of all the layers in the AutoCAD DXF files. The unused layers are selected and moved to the invisible layer.

[Invisible layers]

The layers in the AutoCAD DXF file that are not used for the terrain geometry.

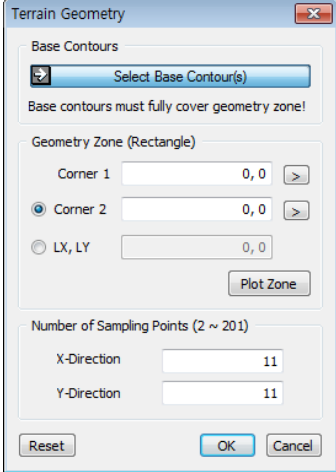
Step 3 : Set analysis boundary



Move the mouse to the area to be modeled and click the Terrain geometry information icon () on the right toolbox to set the analysis boundary.

Step 4 : Set section zone

►Terrain geometry information



[Base Contours]

Select the minimum zone that contains the analysis boundary by dragging on the XY plane.

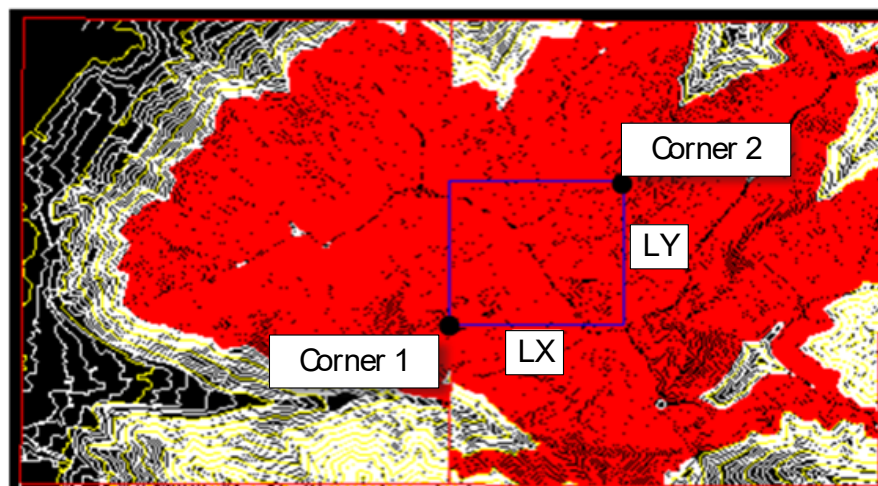
[Geometry Zone (Rectangle)]

Specify the rectangular analysis boundary that is within the base contour zone. Corner 1 and Corner 2 are the diagonal corners that specify the rectangular boundary. LX, LY specify the X axis and Y axis lengths of the specified boundary. Click the Display area to view the specified boundary on the plane.

[Number of Sampling Points]

Specify the number of sampling points in the selected region with respect to the X direction and Y direction.

► Example of Terrain geometry information zone

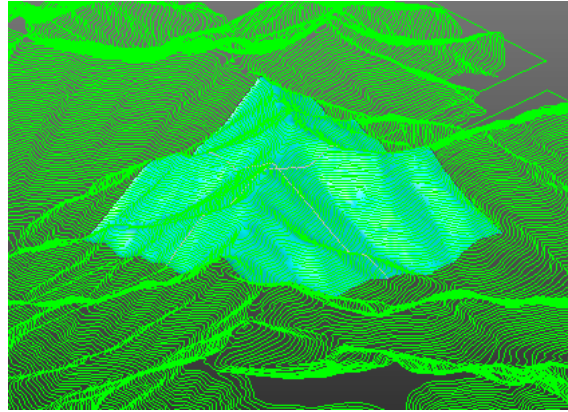


Step 5 : Check geometry face

Check the geometry face on the View model window.



► Example of Terrain geometry information zone



Step 6 : File > Export surface

Save the generated terrain face as a *.tms file that can be used in the FEA NX.

Step 7 : Main menu : Tools > Terrain Geometry Maker > Import TMS file

Import a generated *.tms file using the MIDAS/TGM function.

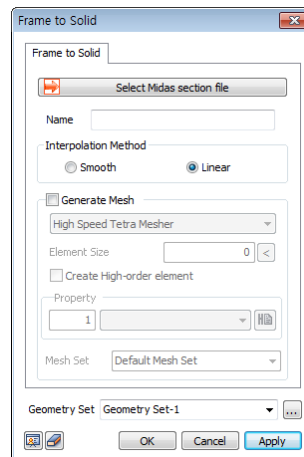
9.3

Frame to Solid

Overview

Convert frame data created on midas Civil or midas Gen into solid data.

►Frame to Solid



Methodology

Select a frame data (*.mcs) file created on midas Civil or midas Gen. Defining the Linear interpolation (straight line) or Smoothstep (smooth interpolation), automatically calls up a solid element. This depends on the multidimensional curved lines of the selected frame element's end section, Checking mesh can create a mesh on the solid during conversion.

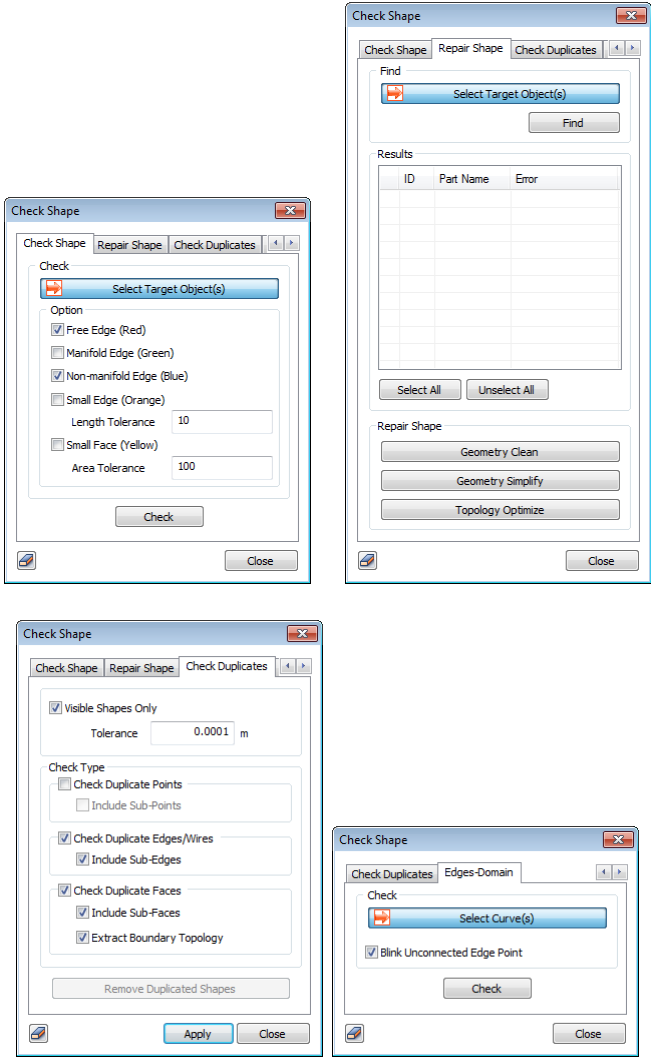


9.4 Check Geometry

Overview

Check the detailed information of a selected shape.

- Check Shape
- Repair Shape



- Check Duplicates
- Edges Domain

Check shape

Methodology

Select the geometry shape to be checked and select the option to view the information on the screen. The geometric information is also shown on the output window.

[Free Edge (Red)]

The object outlines are indicated in red.

[Manifold Edge (Green)]

The edges between 2 meeting faces are indicated in green.

[Non-Manifold Edge (Blue)]

The edges between 3 meeting faces are indicated in blue.



[Small Edge (Orange)]

The edges shorter than the input length are indicated in orange.

[Small Face (Yellow)]

The faces smaller than the input area are indicated in yellow.

Repair shape

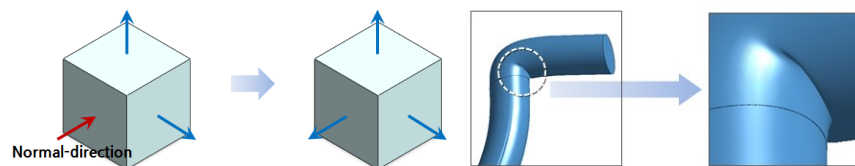
Methodology

Find and modify geometric errors automatically for atypical and inaccurate shape. The user can perform the function when have failed to create mesh for selected geometry. Recommended to be performed "Geometry Clean → Geometry Simplify → Topology Optimize" continuously.

✓ Geometry Clean : Modify invalid shape

► Abnormal topology

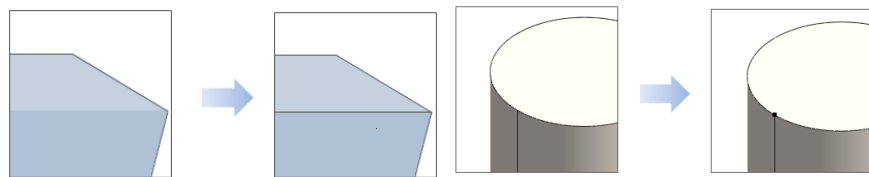
► Tangled shape



✓ Geometry Simplify ; Modify irregular shape

- B-Spline surface ; Plane, Cylinder, Sphere, Cone and Torus
- B-Spline curve ; Line, Circle and Ellipse
- Irregular shape : Normalized and primitive shape

► Omission of Edge or Vertex



✓ Topology Optimize ; Performance improvement in creating mesh

- Simplify geometry and delete duplicated edge / surface automatically
- Delete unnecessary edge or vertex automatically

Check Duplicates

Methodology

Check for repeated shapes in the same position.

[Visible Shape Only]

Repeated shapes are represented by visible shapes on the screen. The check is performed regardless of the Show/hide status of the target.

[Check Duplicate Points]

Check for duplicated points. The function only checks for independent points. Overlapping points are indicated in yellow. Check the [Include Sub-Points] to perform the check on sub-shape points (ex: the corner points of a box).

[Check Duplicate Edges/Wires]

Check repeated edges or wires. The function only checks for independent edges and wires. Overlapping edges/wires are indicated in green. Check the [Include Sub-Edges] to perform the check on sub-shape edges (ex: the edges of a rectangle).

[Check Duplicate faces]

Check duplicated faces. The function only checks for independent faces. Overlapping faces are indicated in orange. Check the [Include Sub-Faces] to perform the check on sub-shape edges (ex: a face of a box). Use the [Extract Boundary Topology] to only check faces that are exactly overlapping. An overlapping face has the same number of points on the outline, the same coordinates and the same edge direction.

[Remove Duplicated Shape]

Remove repeated independent shapes and only leave a single shape. However, the sub-shapes are not deleted.

Tip

Check for shared faces after creating a 3D geometry object. Creating a mesh on an unshared solid created a free face makes obtaining accurate analysis results difficult. Such shared faces can be checked using Check Shape > Check Duplicates. To view the object information, change the view mode of the solid to linear view.

When the shared face of adjacent shapes are not shown, use Boolean > Solid to create a shared face.

Edges-Domain

Methodology

Check the connection of lines. When importing geometry shapes from CAD, this option can be used to check for areas that are not connected.

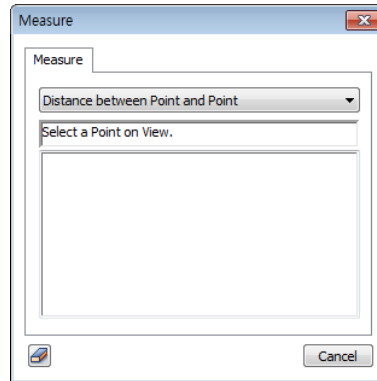


9.5 Measure

Overview

Measure the shortest distance or angle between shapes using various methods.

►Measure



Methodology

Measure the distance or angle by selecting a shape's point, line, face etc. on the screen.

The distance can be measured using [Distance between point and point], [Distance between point and Edge], [Distance between point and face], [Distance between edge and edge], [Distance between edge and face], [Distance between face and face]. The angle can be measured using [Angle between three points].

Tip

Checking the needed snaps with Define snap () makes it easier to select points on the screen.