ZW3D from Entry to Master Tutorial

Solid Modeling

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ZW3D™ V2023 From Entry to Master CAD Solid Modeling

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Foreword

In this tutorial, we provide various case studies, which are from easy to difficult and combine theory with practice. We hope to improve users' 3D CAD/CAM skills and techniques with ZW3D.

The tutorial bases on our technical engineers' years of experience in the industry and ZW3D, which is the fruit of a lot of efforts and wisdom. We sincerely hope that the tutorial will do help to you, and your precious advice on it is highly welcomed.

There are three series for this tutorial: ***Primary Tutorial***, ***From Entry to Master Tutorial***, and ***Advanced Tutorial***. From easy to difficult, they offer a step-by-step learning process that can meet different user needs.

Primary Tutorial series is for users who have little or no prior 3D CAD/CAM experience. If you are green hands of 3D CAD/CAM software, or if you are a new user of ZW3D, we recommend that you get started with this tutorial. Here you can learn the basic knowledge and concepts of ZW3D, rapidly master the simple operations and workflows of ZW3D, and practice simple cases.

From Entry to Master Tutorial series is for users with basic know-how of 3D CAD/CAM software. If you have experience in 3D CAD/CAM software and want to master common functions of ZW3D, we suggest that you start with this series. Here you can dig deeper into the functions and master more operations of ZW3D.

Advanced Tutorial series is for users with practical experience in 3D CAD/CAM software. If you hope to have a comprehensive command of ZW3D and get the complicated operations done independently, you can choose to learn this series. Here you can learn to use the software more flexibly and get rich experience to increase your efficiency.

What you are learning is ***ZW3D From Entry to Master CAD Solid Modeling***, a master tutorial.

Thanks for being our user!

The ZW3D Team

Contents

Solid modeling is the most important function for designing. It can convert the concept to real 3D model. Solid modeling includes basic shape function, engineering feature, edit shape function, as well as morph function.

Key Points:

- ✧ Reference features generation
- ✧ Primitive features creation
- ✧ Create shapes from sketch
- ✧ Shapes and features editing
- ✧ Application of deformation features

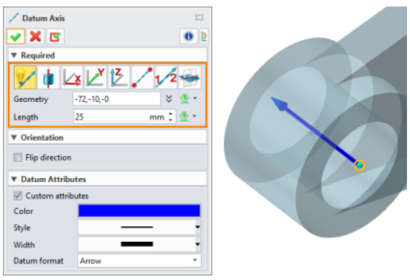
1 Reference Features

When modeling, users need to create some reference features occasionally, such as datum axis, datum plane, datum coordinate system, etc.

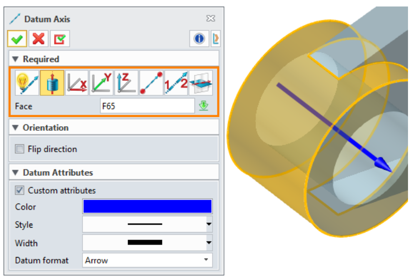
1.1 Datum Axis

Shape Ribbon Tab->Datum->Datum Axis

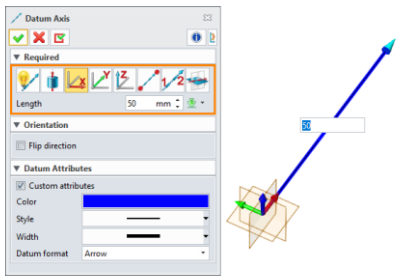
There are 6 methods for users to generate the extra datum axis in ZW3D, including generating from geometry smartly, from cylindrical face, along X/Y/Z-axis, from two points, from one point and direction, from two intersected faces. Please see the figure below.



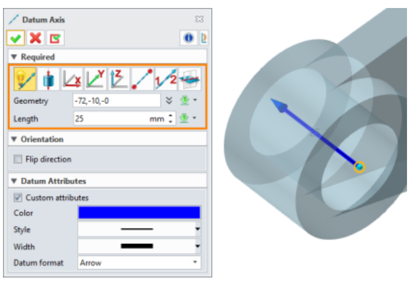
Type1 – Geometry



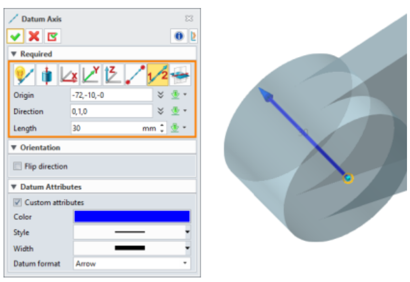
Type2 – Face Center Axis



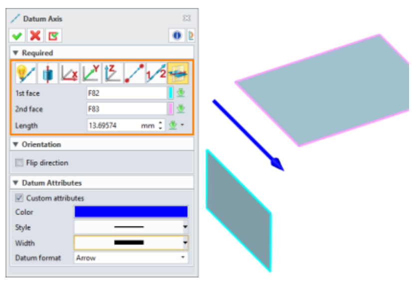
Type3 – X/Y/Z Axis



Type4 – 2 Points



Type5 – Point and Direction



Type6 – Intersected Faces

Figure 1 Datum Axis

1.2 Datum Plane

Shape Ribbon Tab->Datum->Datum Plane

ZW3D provides 6 methods for creating the extra datum plane, including smart generating, creating from three points, offsetting by XY/YZ/XZ plane, generating by the current view, generating by picking two entities, generating by moving the dynamic handle. Please refer to Figure 2.

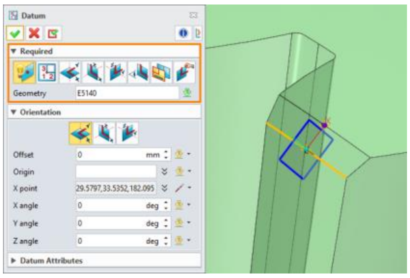
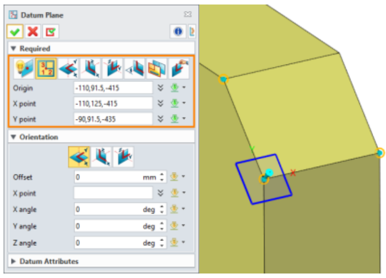
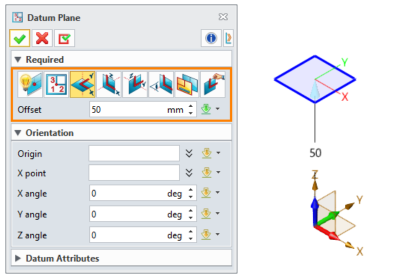
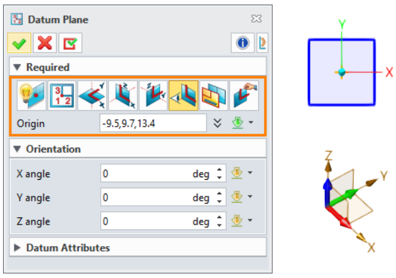
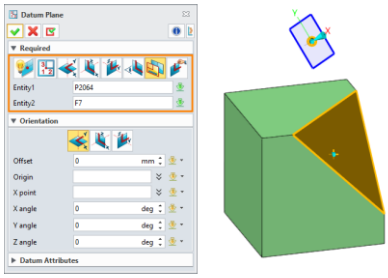
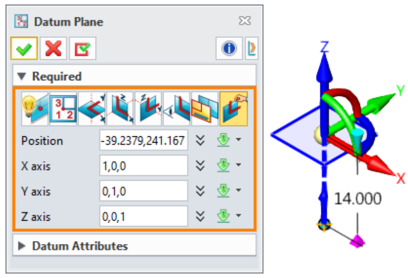
 **Type1** – Plane	 **Type2** – 3 Point Plane	 **Type3** – XY/YZ/XZ Plane
 **Type4** – View Plane	 **Type5** – 2 Entities	 **Type6** – Dynamic

Figure 2 Datum Plane

Shape Ribbon Tab->Datum->Drag Datum

Pick a datum, drag one of the eight control points on the side to modify the size of the datum.

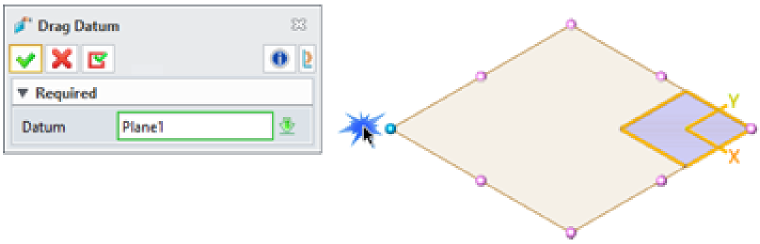


Figure 3 Drag Datum

1.3 Datum CSYS

Shape Ribbon Tab->Datum->Datum CSYS

This command let users to create datum CSYS (coordinate system) and the creation methods are the same as datum plane.

1.4 LCS

Shape Ribbon Tab->Datum->LCS

This command let users define LCS (local coordinate system). ZW3D supports the following two methods.

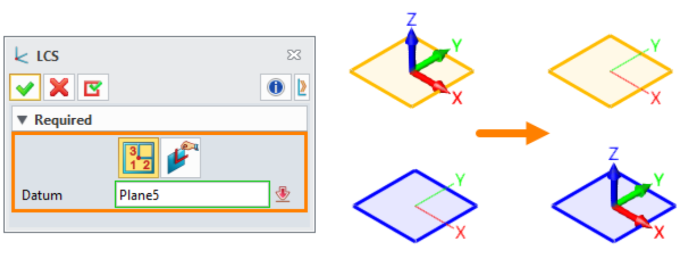
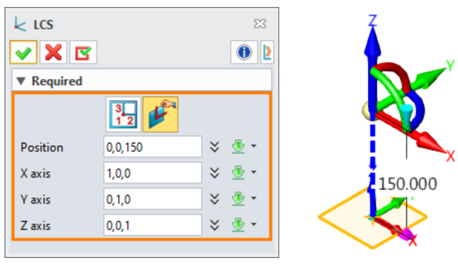
 **Type1** – Datum	 **Type2** – Dynamic
Pick a datum.	Define the LCS with dynamic handle.

Figure 4 LCS

Notes: *Only one LCS can be activated in the modeling space.*

2 Primitive Features

In the beginning, users need to create the first entity. ZW3D provides some kinds of primitive features to create the entities quickly.

2.1 Primitive Features

Shape Ribbon Tab->Basic Shape->Block/Cylinder/Cone/Sphere/Ellipsoid

Use this command to create primitive features like ***Block***, ***Cylinder***, ***Cone***, ***Sphere*** as well as ***Ellipsoid***.

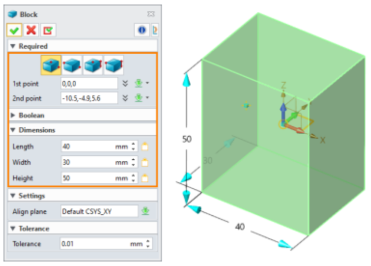
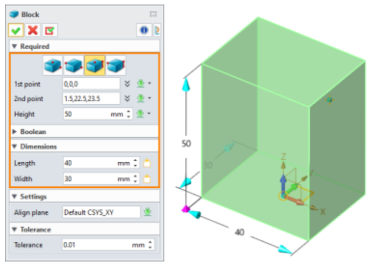
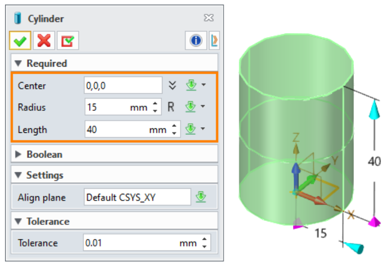
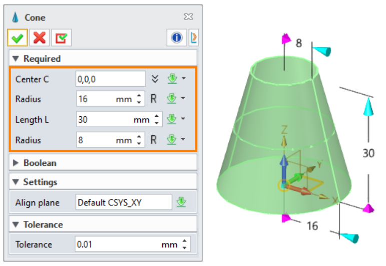
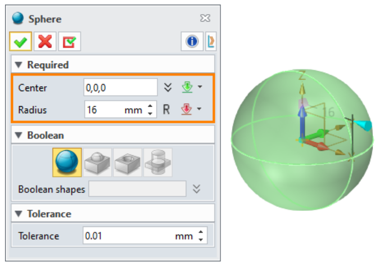
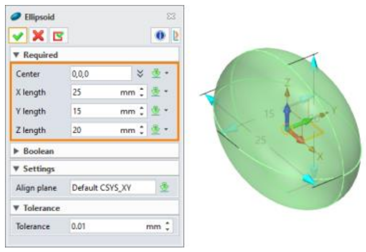
 **Block** – Center	 **Block** – Center and Height	 **Cylinder**
 **Cone**	 **Sphere**	 **Ellipsoid**

Figure 5 Primitive Features

Notes: *The primitive features are generally only used once when creating a basic at the beginning of modeling, because they are not easily controlled by parameters.*

2.2 Boolean Operations

ZW3D provides four types for Boolean Operations: ***Base***, ***Add***, ***Remove***, ***Intersect***. Users can specify the Boolean type and the Boolean shapes in the below option.

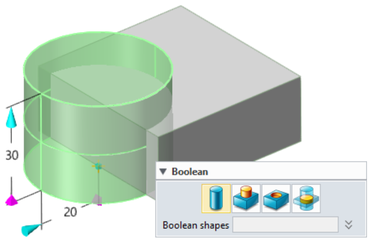


Figure 6 Boolean Operation

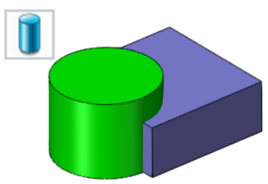
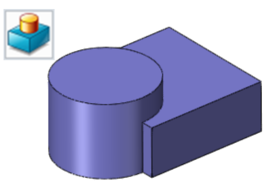
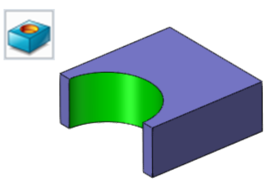
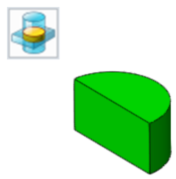
 **Base**	 **Add**	 **Remove**	 **Intersect**
Two shapes	One Shape		

Figure 7 Boolean Operation – Different Options

Notes: *The default Boolean shapes are all the visible shapes in the modeling space.*

3 Swept Features

In solid modeling, the most frequently used by users is the swept features. The sweep features are usually based on one or more sketches, sweep along a curve or rotate along an axis to create, like ***Extrude***, ***Revolve***, ***Sweep***, ***Loft*** and so on. Sketches can also be replaced by faces or curves etc.

3.1 Extrude

Shape Ribbon Tab->Basic Shape->Extrude

Use this command to create an extrude feature. Inputs include profile, extrude type, direction as well as start/end positions. What's more, users can add attributes for draft, twist, offset and endcaps. The main steps for extrude operation are listed below.

STEP 01 Create a new sketch or select the existing sketch as the ***Profile*** of the extrude feature.

STEP 02 Set the ***Extrude Type*** and set the ***Extrude Range***.

STEP 03 Define other parameters according to the requirements, such as draft angle, offset value.

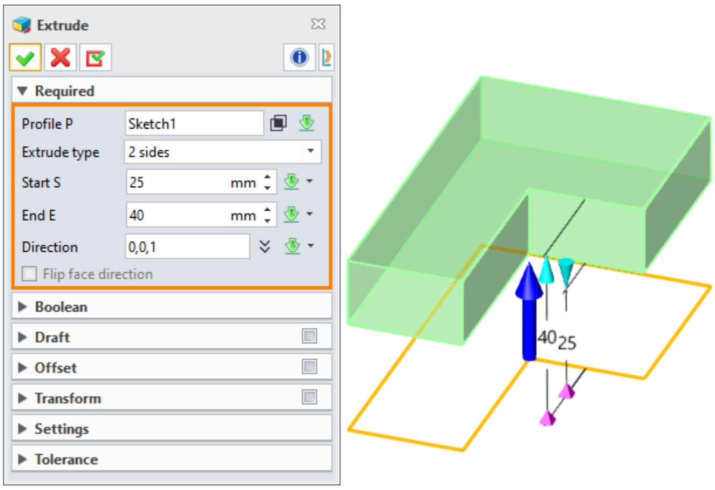
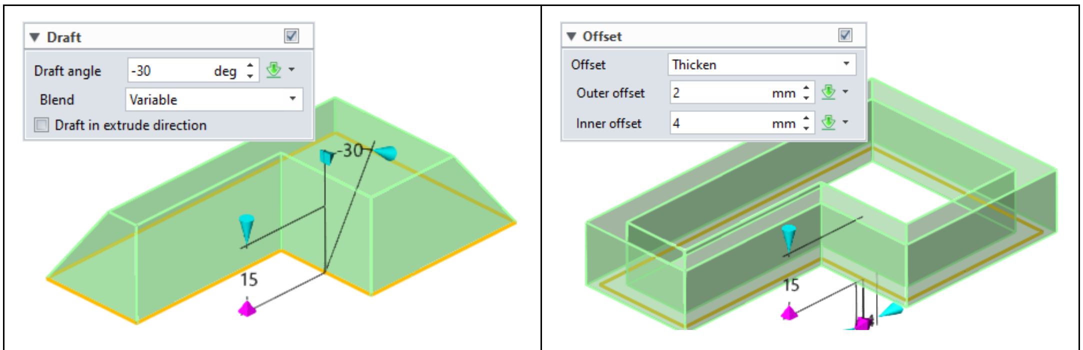


Figure 8 Extrude

- **Draft:** Enter the draft angle if need, both positive and negative values are acceptable.
- **Offset:** This option can shrink or expand the shape or surface. And it can also independently offset outward or inward by values.
- **Transform:** Create twist feature by select twist point and twist angle. Twist angle is the total angle that twisting from feature start to end. Because the resultant surface is a ruled surface, so the maximum twist is 90 degrees If you need more, please use sweep, helix, or loft feature.
- **Endcap:** There are four endcap types: both endcaps, start endcap, end endcap and no endcap. The figure below shows the ***No endcap*** option.



Draft **Offset**

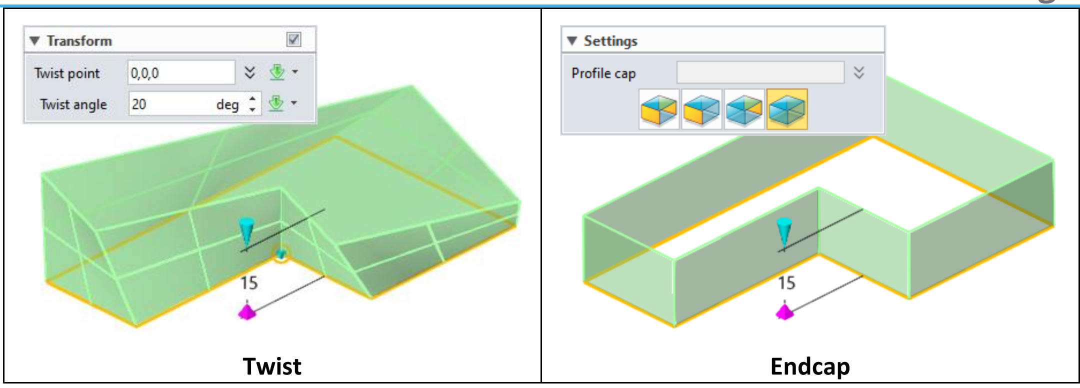


Figure 9 Extrude – Parameters

***Notes:** The **Profile Cap** option can be used to specify a "cap" if profile is closed, see the Figure 10. Or specify the boundary if the profile is open, see the Figure 11.*

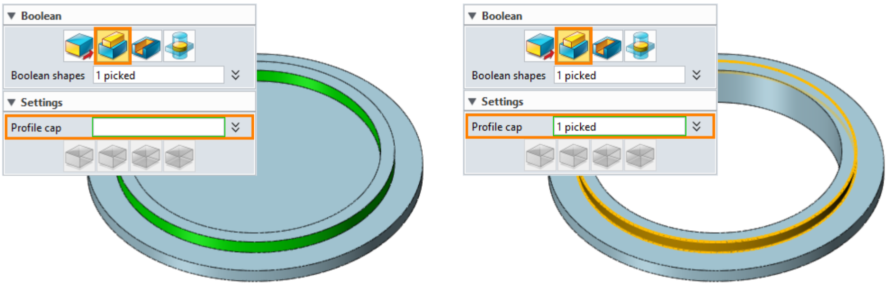


Figure 10 Extrude – Profile Cap (Closed Profile)

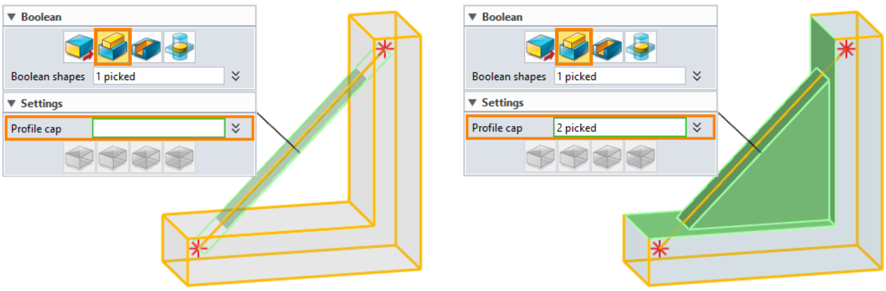


Figure 11 Extrude – Profile Cap (Open Profile)

3.2 Revolve

Shape Ribbon Tab->Basic Shape->Revolve

Use this command to create a revolve feature. The inputs include the profile, axis, revolve type as well as start/end angle position. The profile can be a sketch, wireframe or face edges.

STEP 01 Create a new sketch or select the existing sketch as the ***Profile*** of the revolve feature.

STEP 02 Define the ***Revolve Axis***, ***Revolve Type*** and set the ***Revolve Range***.

STEP 03 Define other parameters according to the requirements.

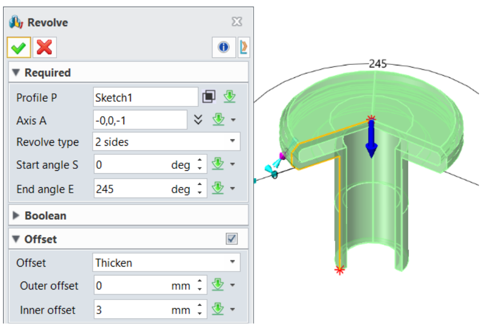


Figure 12 Revolve

***Note:** Other parameters are similar to those of **Extrude**, please refer to the **Extrude** command.*

3.3 Sweep

Shape Ribbon Tab->Basic Shape->Sweep

Use this command to create simple or variable sweep feature with a profile and a path. Both profile and path can be a sketch, wireframe or face edge.

There are 2 build-in frames during sweeping. One is ***Reference Frame*** displayed in 3D axes to indicate how the profile locates originally, another one is ***Local Frame*** on the point of path displayed in lines to show how each profile will be located along the path.

Sweep puts profiles on each point of the path by aligning the reference frame and the local frame, then connect all the profiles together.

STEP 01 Pick the blue sketch as the profile and the pink sketch as the path.

STEP 02 Choose different ***Orientation*** methods to get different results, see Figure 13.

STEP 03 If necessary, users can define the ***Transform***, refer to Figure 14.

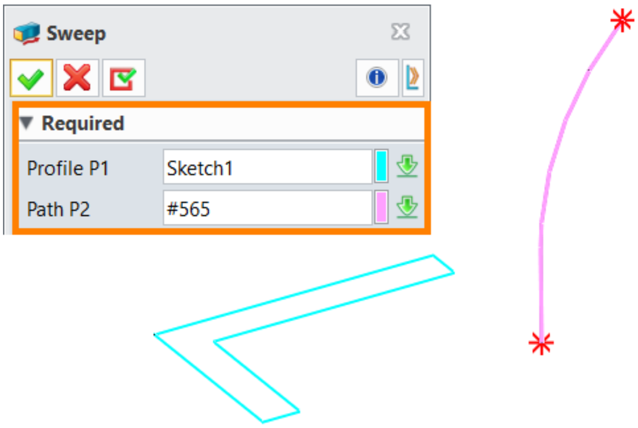


Figure 13 Sweep

- **Orientation:** The reference frame is controlled by the options as below.
  Default frame – Default frame of the profile.
  At intersection (Default) – The frame is built at the intersection of the profile plane and the sweep curve. If the intersection is not found, it is at the beginning of the path.
  At path – The frame is built at the start of the sweep path.
  Along path – The frame is built at the Profile. And the Path will be relocated basing the local frame aligning with the reference during sweeping.

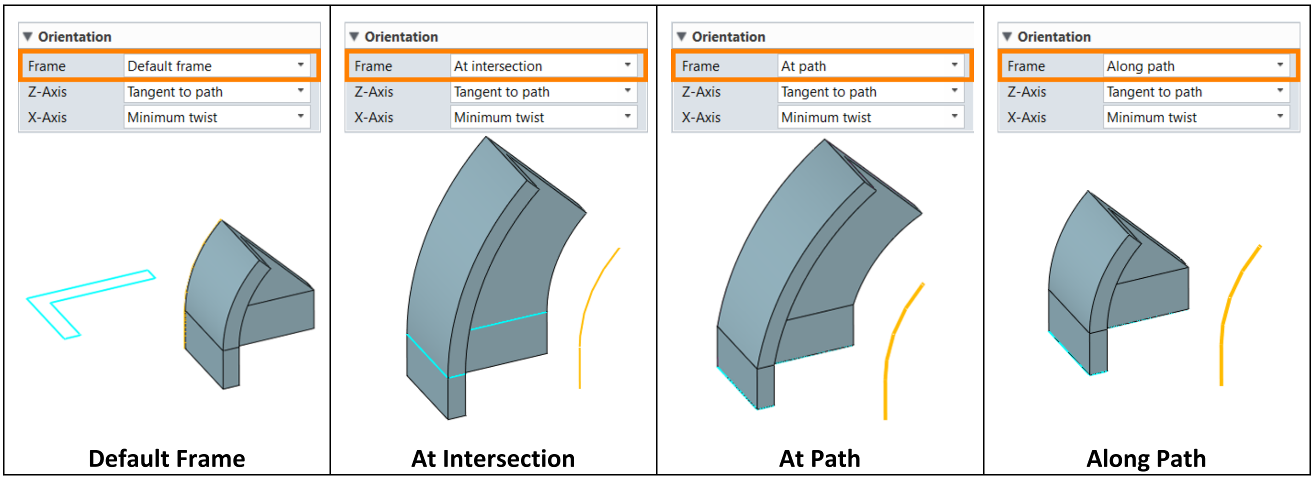


Figure 14 Sweep – Orientation

- **Transform:** Create a scale or twist sweep feature. Below figures show the results of linear or variable scale.

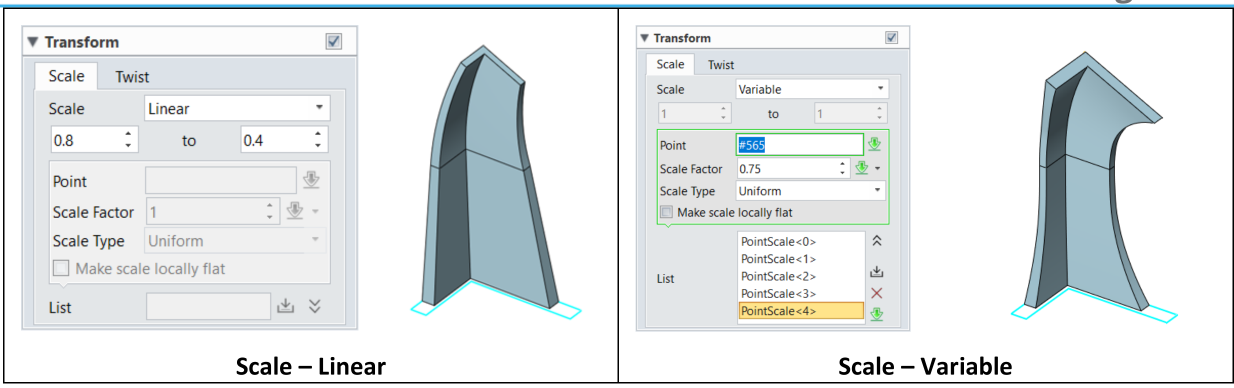


Figure 15 Sweep – Transform

Shape Ribbon Tab->Basic Shape->Spiral Sweep

Create a spiral sweep feature by revolving a closed profile about an axis and along a linear direction. This can be used to make threads or any other shape that is revolved in a linear direction such as springs and coils. Users can also set the taper attribute for this feature.

STEP 01 Draw the sketch as the profile of spiral sweep, see the trapezoidal profile as shown below.

STEP 02 Use ***Spiral Sweep*** command, pick the profile and define the Z-axis as the ***Direction***, input the ***Turns*** as **5** and ***Distance*** as **8 mm**.

STEP 03 Define other parameters like ***Boolean*** and ***Lead in/out*** as Figure 16.

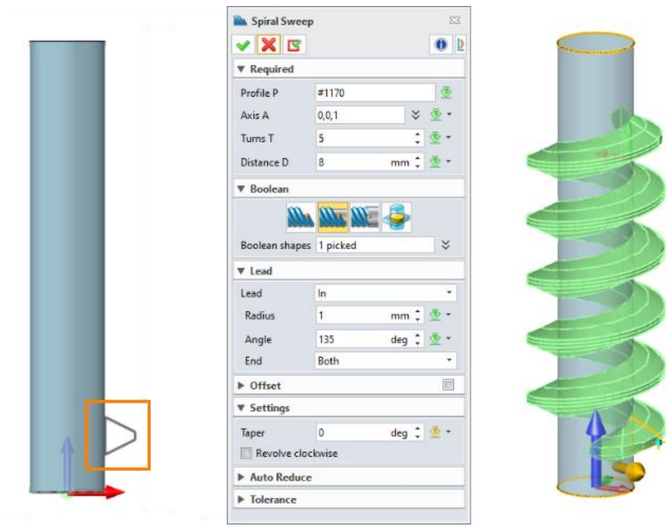
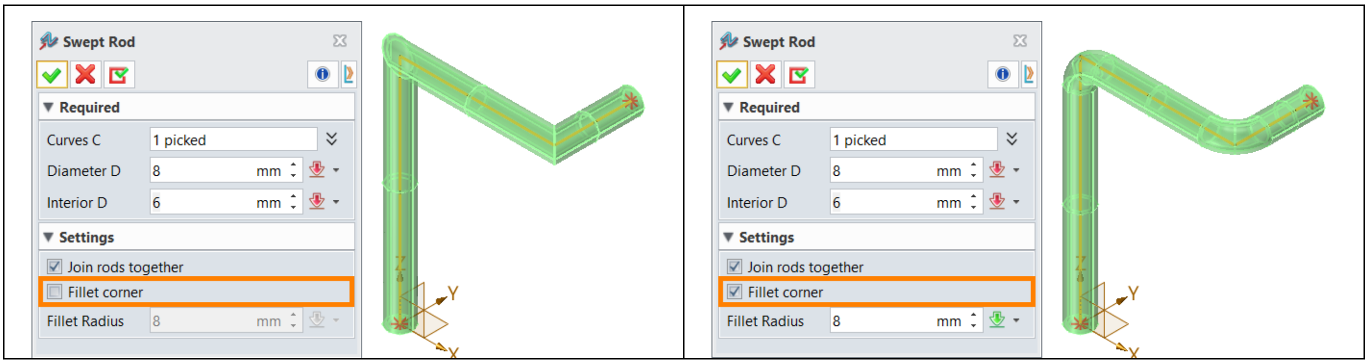


Figure 16 Spiral Sweep

Shape Ribbon Tab->Basic Shape->Swept Rod

Use this command to create a solid rod that sweeps along a network of interconnecting curves (e.g., lines, arcs, circles, or curves). The curves can be tangent or form "X", "T" or "L" shaped intersections. This command is ideal for modeling pipe, tube, or cable routes. The required inputs include the rod diameter, interior diameter, and the sweep curves. Optional inputs include the ability to join the rods together and so on.



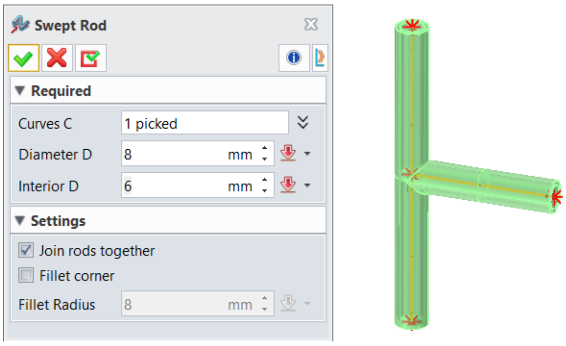
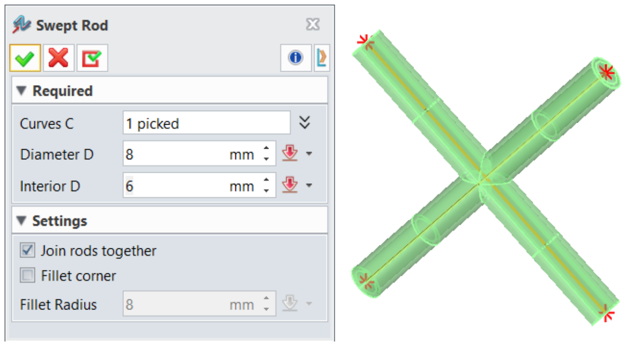
Swept Rod – L type	Swept Rod – L type with Fillets
	
Swept Rod – T type	Swept Rod – X type

Figure 17 Spiral Sweep

Shape Ribbon Tab->Basic Shape->Profile Swept Rod

Use this command to create a solid rod that sweeps along curves. Just like the ***Sweep Rod***, connected curves are valid entities for Path definition, and rods around the corner between these curves will be trimmed as mitered.

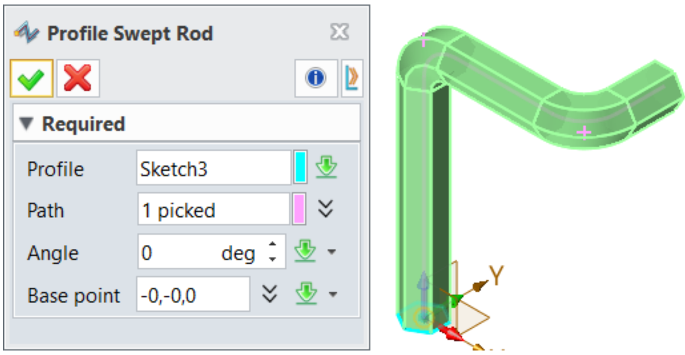


Figure 18 Profile Swept Rod

3.4 Loft

Shape Ribbon Tab->Basic Shape->Loft

Use this command to create a lofted solid or surface feature by a series of profile. The profiles can be sketches, wireframes, face edges or points. Optionally, you can define connection lines to control loft shape.

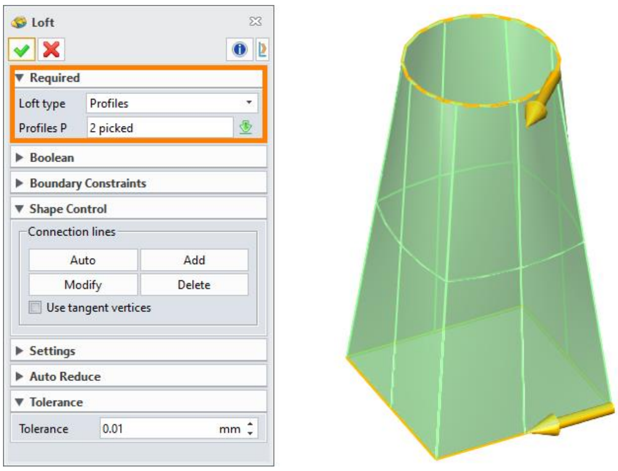


Figure 19 Loft – Profiles

- **Loft Type:** This option allows users to define start/end point and profiles to create the lofted shape. Below shows the type of ***Start point and profiles***.

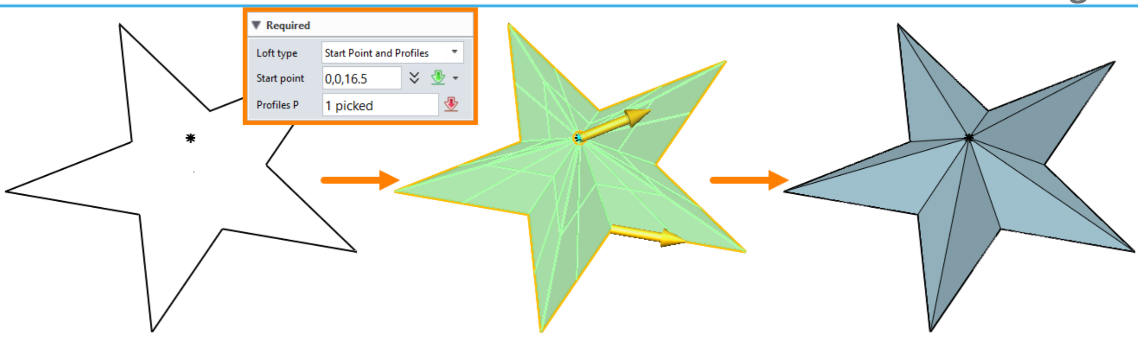


Figure 20 Loft Type – Start Point and Profiles

- **Shape Control:** This option can help you to control the shape of the lofted geometry by adjusting the connection lines.

  STEP 01 Create the connection lines by clicking ***Auto*** button.

  STEP 02 Click ***Modify*** button, then click on the red dot to move it.

  STEP 03 Connection lines can be added or removed by clicking ***Add*** or ***Delete*** button.

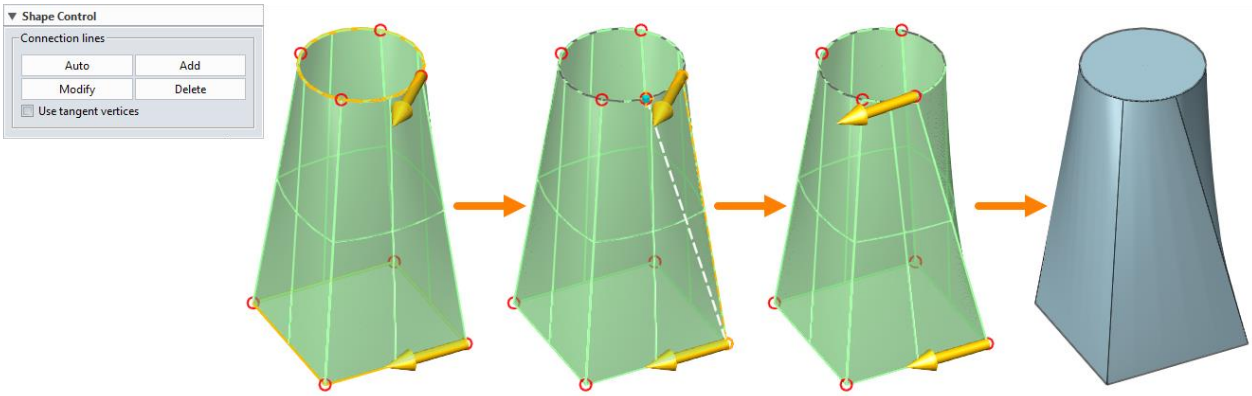


Figure 21 Shape Control in Loft Feature

Note: *If there is no other geometry existed, the boundary constrains definition is disabled.*

Shape Ribbon Tab->Basic Shape->Drive Curve Loft

Use this command to create a loft feature along the drive curve. It includes some options that are similar to the Sweep and Loft commands.

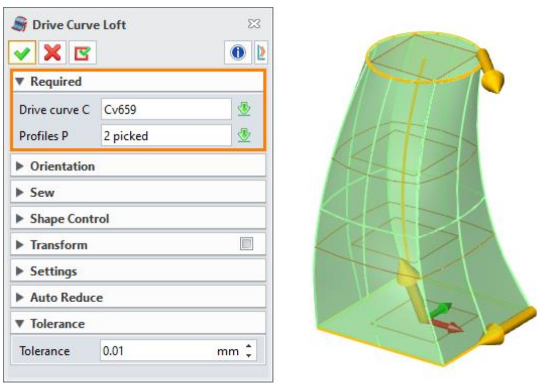


Figure 22 Drive Curve Loft

4 Engineering Features

When the main shape is completed, it usually needed to add engineering features, such as fillets, chamfers, holes, drafts, etc. This chapter will introduce engineering features one by one.

4.1 Fillet

Shape Ribbon Tab->Engineering Feature->Fillet

Use this command to create a wide variety of constant and variable fillets and corner blends. They include the amount of smoothing at corner blends, arc or conic cross-sectional fillets, variable fillet attributes, corner relief and edge constraints. The figure below shows the general fillet.

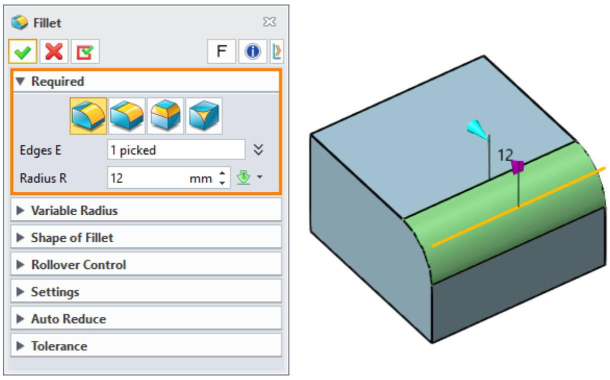


Figure 23 Fillet

And the following are three other types of fillets, ***Elliptical Fillet***, ***Loop Fillet*** and ***Vertex Fillet***.

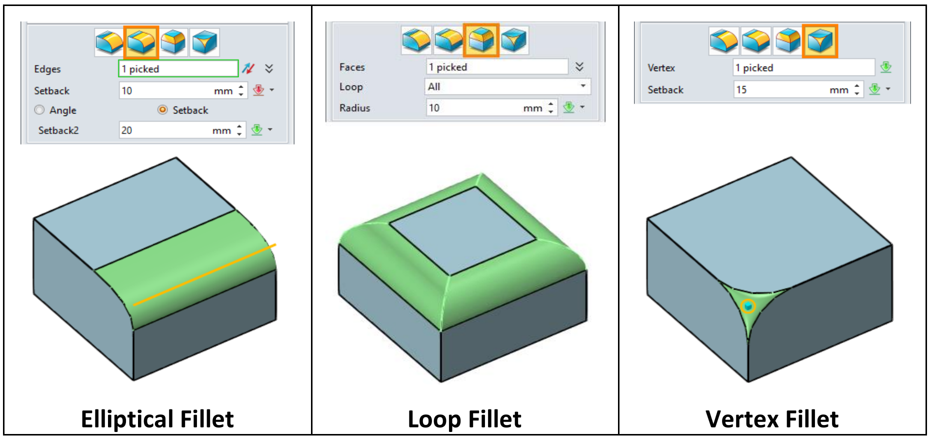


Figure 24 Fillet – Different Types

➢ **Variable Radius:** Use these options to create variable radius fillets. The variable fillet is achieved by adding variable attributes at any location you choose along a selected edge.

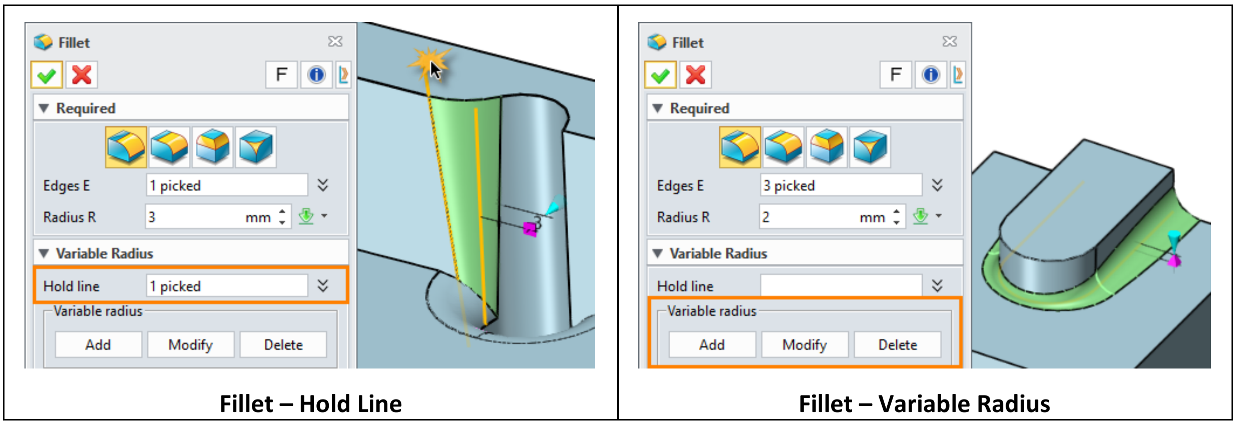


Figure 25 Fillet – Variable Radius

➢ **Rollover Control:** The following options allow users to generate fillets with different shapes.

1) **Hold fillet to edge:** The fillet will be held to the nearby edges. Figure below shows two cases.

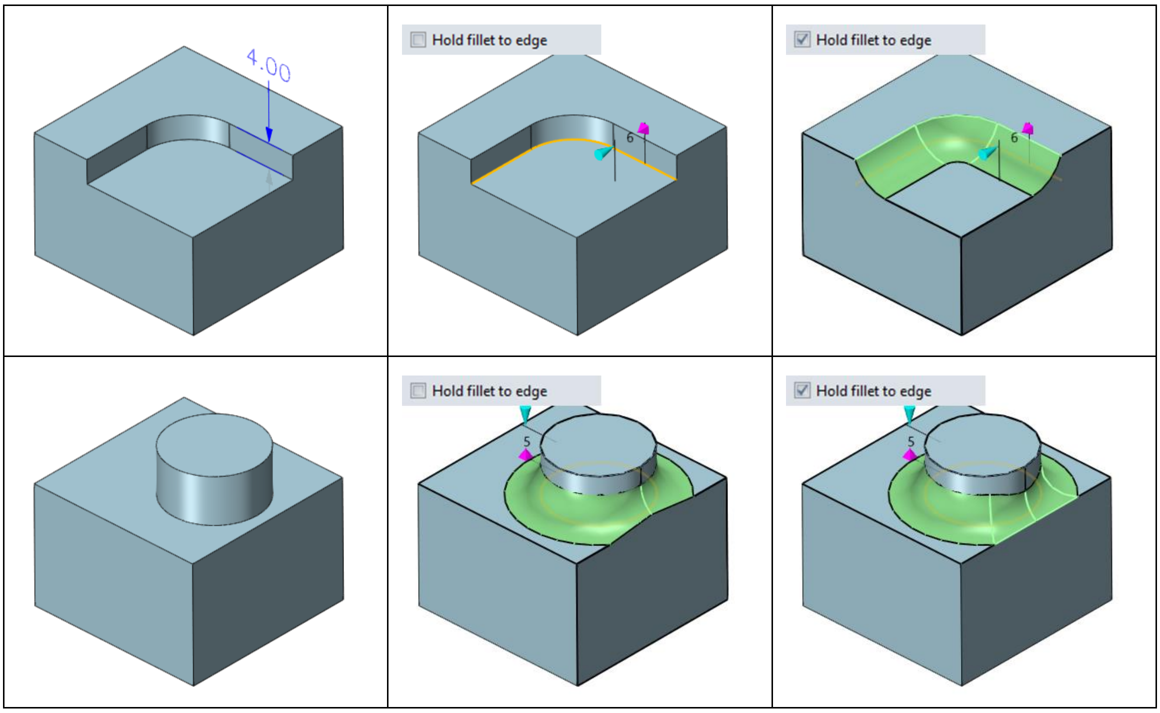


Figure 26 Fillet – Hold Fillet to Edge

2) **Search for undercuts:** Check this option to keep the original features.

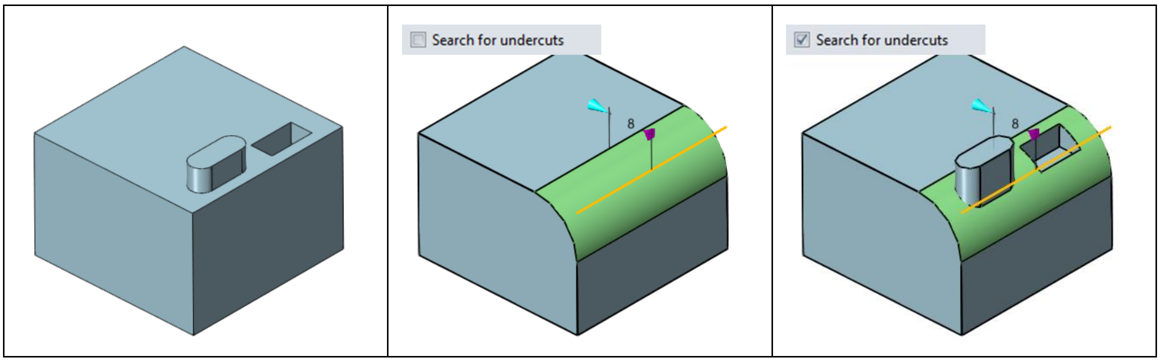


Figure 27 Fillet – Search for Undercuts

3) **Mitered corners:** Check this option to get a mitered solution at corners of uniform convexity.

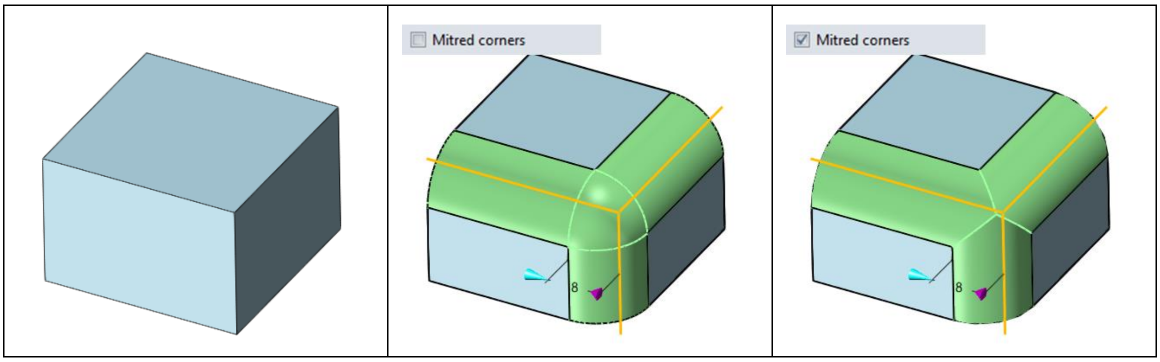


Figure 28 Fillet – Mitered Corners

4) **Trace corners:** Check this option, the rails of the fillet and corner patch faces are traced onto the support. This will produce more aesthetically pleasing results.

5) **Blend corners:** Check this option to create a smoother fillet by FEM-based surface fitting method.

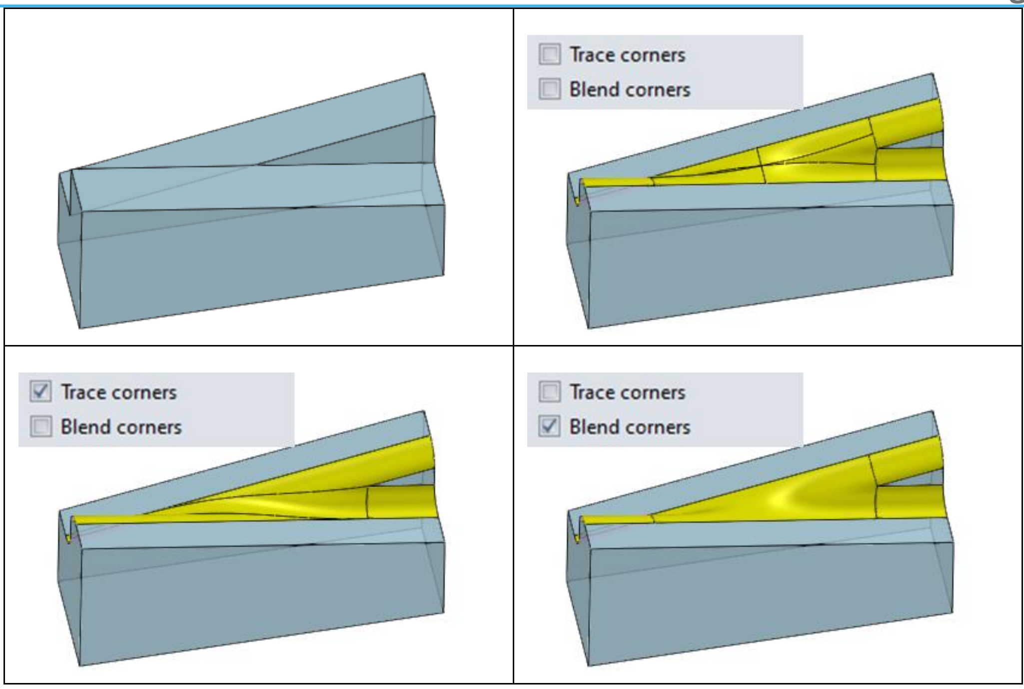


Figure 29 Fillet – Trace Corners and Blend Corners

4.2 Chamfer

Shape Ribbon Tab->Engineering Feature->Chamfer

ZW3D provides 3 types of chamfer including general chamfer, asymmetric chamfer and vertex chamfer, see the figures below. Most of the parameters are similar to fillet feature. So please refer to the fillet feature to understand the variable chamfer and rollover control options.

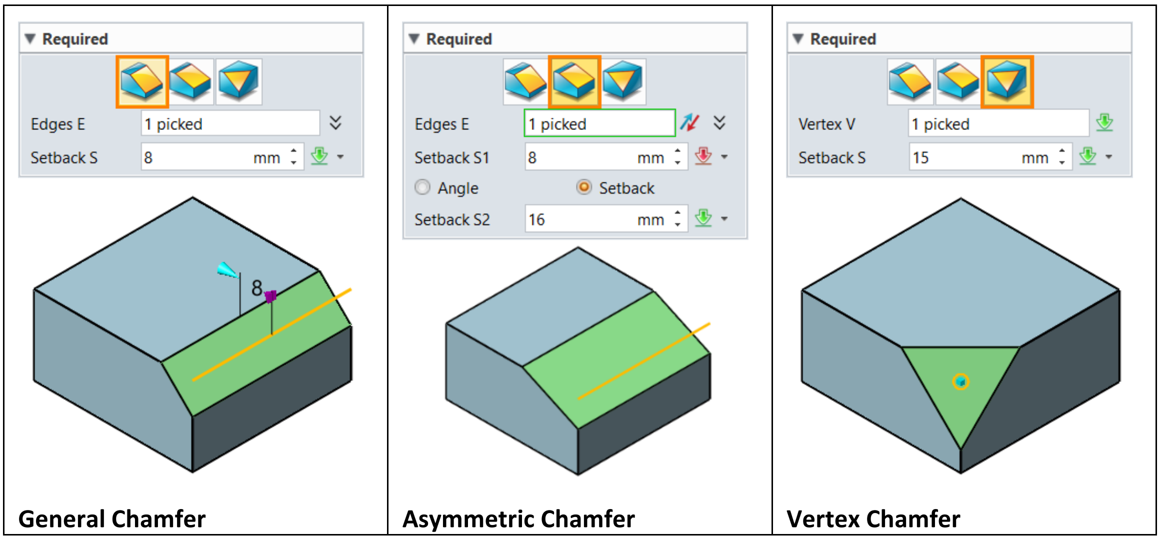
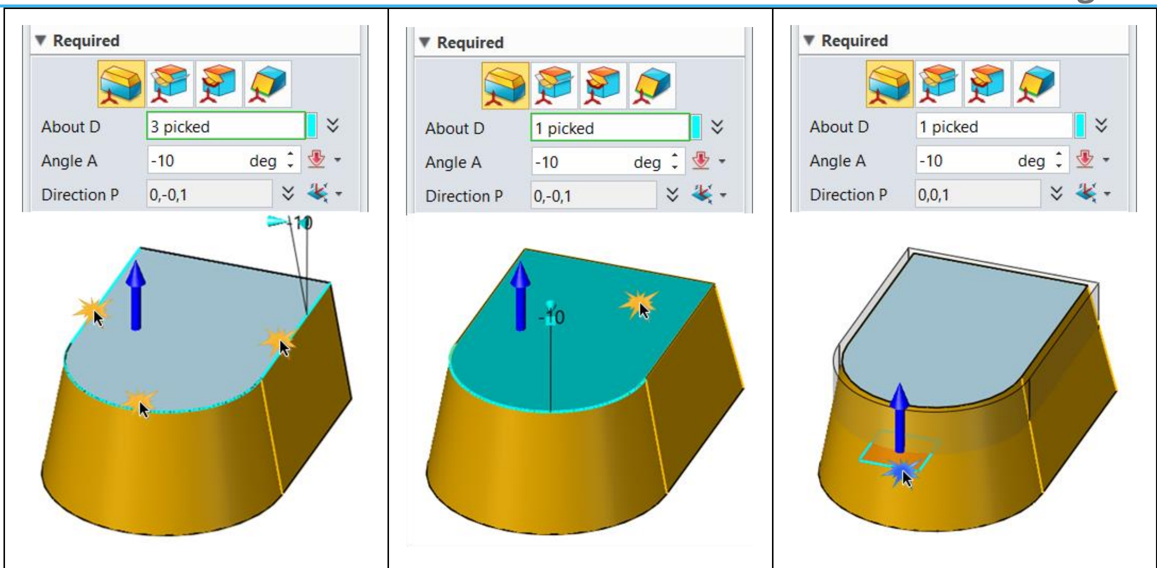


Figure 30 Chamfer

4.3 Draft

Shape Ribbon Tab->Engineering Feature->Draft

Use this command to create a draft feature on selected entities. The entities can be edges, faces or datum. Draft features mainly use in mold design, which make the casting parts ejected from the mold smoothly.



Draft – Pick Edges Draft – Pick Face Draft – Pick Datum

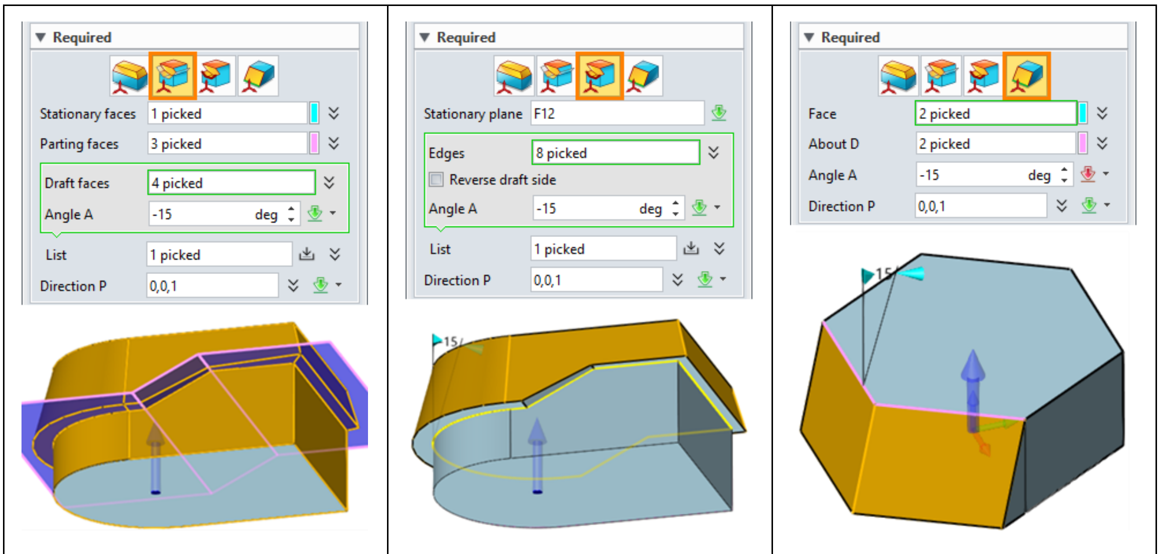
Figure 31 Draft

In addition, ZW3D also provides the following different options, picking the parting faces or parting lines for single-sided draft, and face draft, please see the figures below. And take the left figure as an example.

STEP 01 Pick the top face as the ***Stationary Faces*** and pick the bule surfaces as the ***Parting Faces***.

STEP 02 Pick the side faces as the ***Draft Faces*** and define the ***Draft Angle***, add to the list.

STEP 03 Define **Z-axis** as the ***Draft direction*** and confirm.



Draft – Parting Face Draft – Parting Edge Draft – Face Draft

Figure 32 Draft – Different Options

- **Variable Draft:** Use this option to specify the draft angles for particular faces.

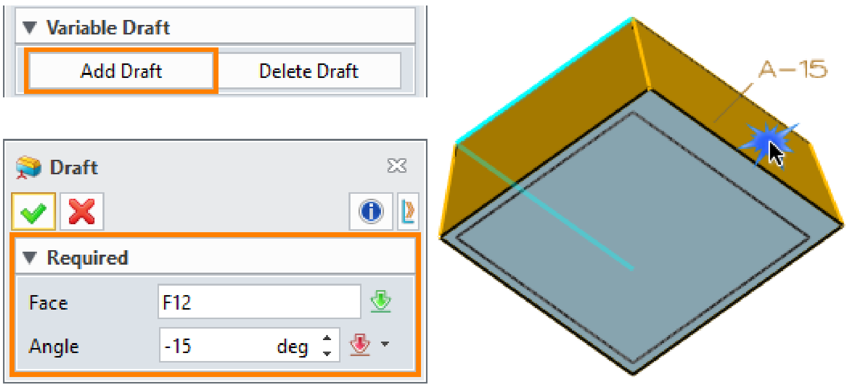


Figure 33 Variable Draft

- **Setting:** The following options allow users to generate drafts with different shapes.

1) **Extension:** This option controls the path of the draft face.

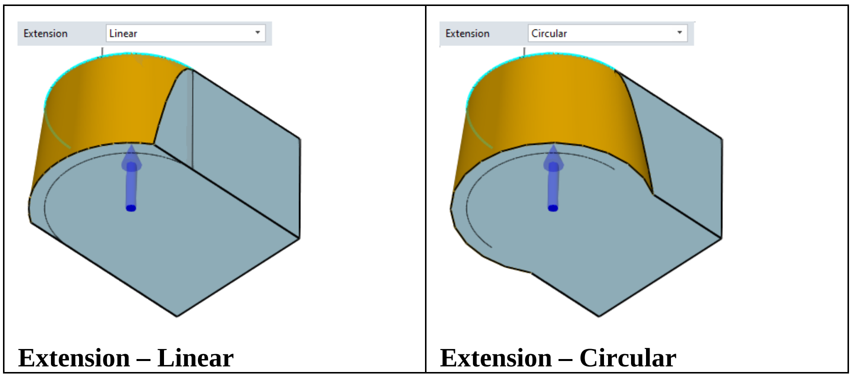


Figure 34 Draft – Extension

2) **Draft faces:** Specify faces for drafting.

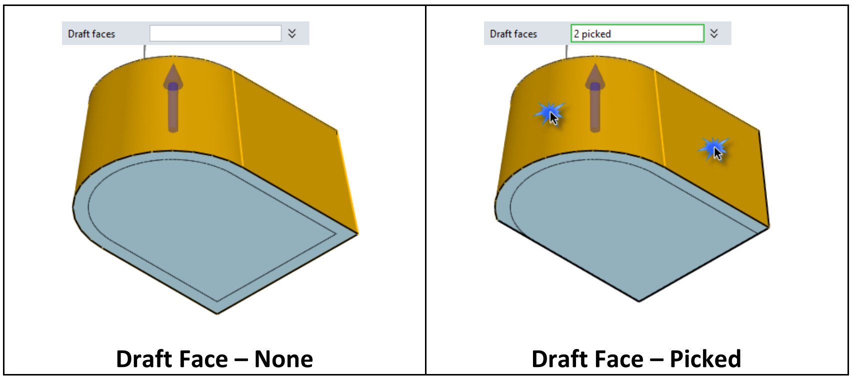


Figure 35 Draft – Draft Face

3) **Recompute fillets:** When this option is checked, all the fillets will be removed before the draft operation. And they will be added after the operation.

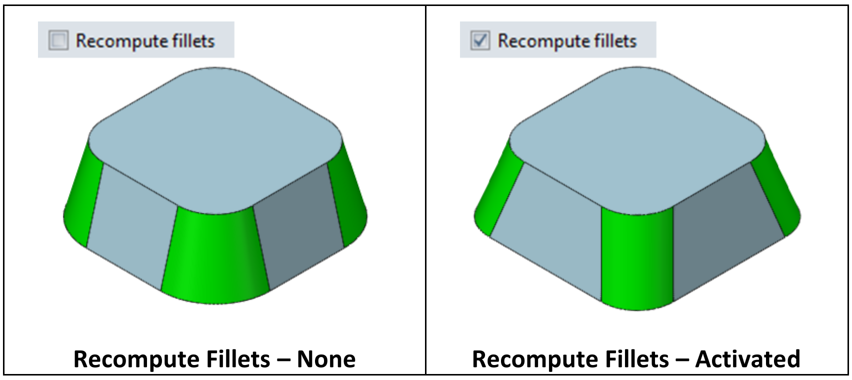


Figure 36 Draft – Recompute Fillets

4) **Side S:** Use this option to specify which side needs to add draft.

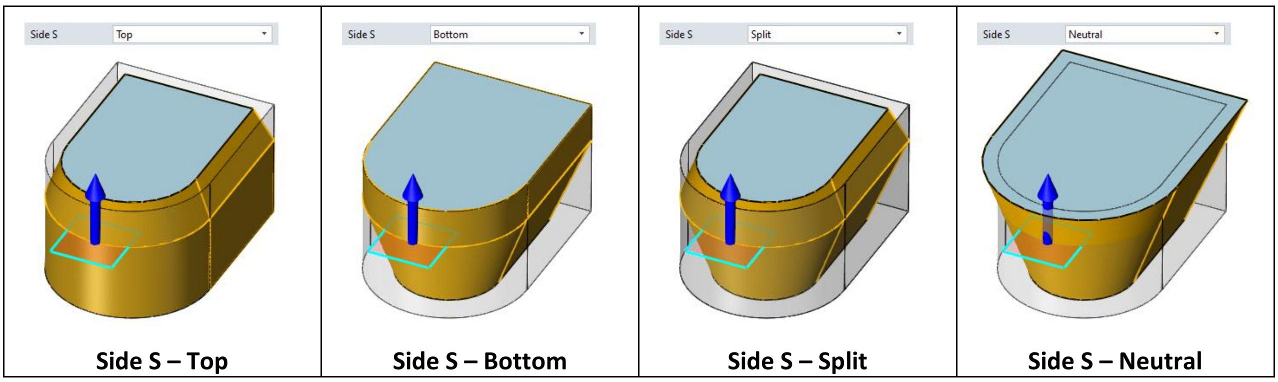


Figure 37 Draft – Draft Side

Shape Ribbon Tab->Engineering Feature->Asymmetrical Draft

Use this command to create an asymmetrical draft feature and it is usually used for mold design. The objects that can be selected are edges, faces, datums, etc. The following is a figure for ***Asymmetrical Draft*** by picking parting lines.

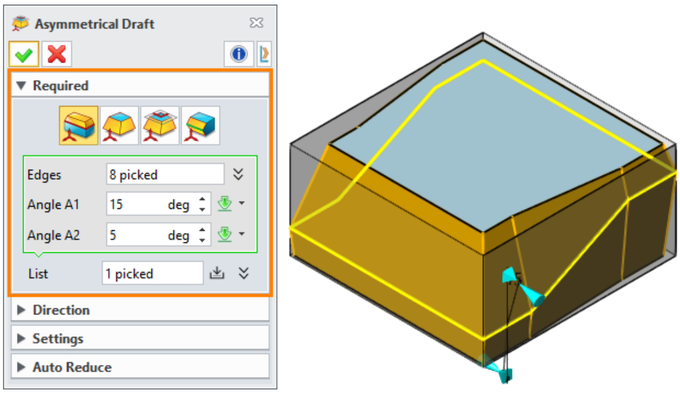


Figure 38 Asymmetrical Draft

4.4 Hole

Shape Ribbon Tab->Engineering Feature->Hole

This command can be used to create three types of hole, including general hole, clearance hole and thread hole. And there are five hole shapes including simple, tapered, counter-bore, counter-sink and spot-face, the following figure shows different hole shapes.

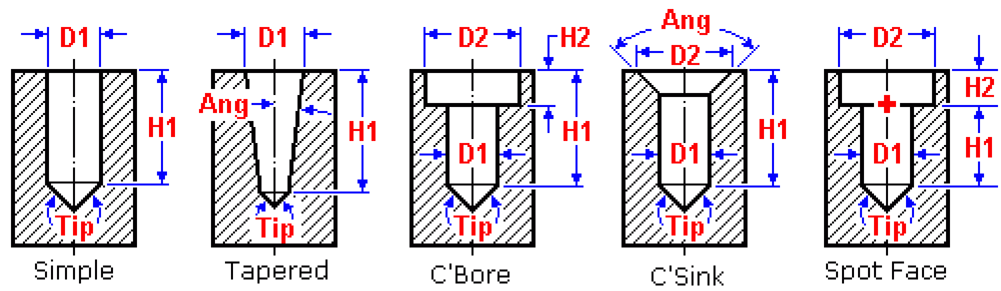


Figure 39 Hole Type

Take a counter-bore thread hole for example.

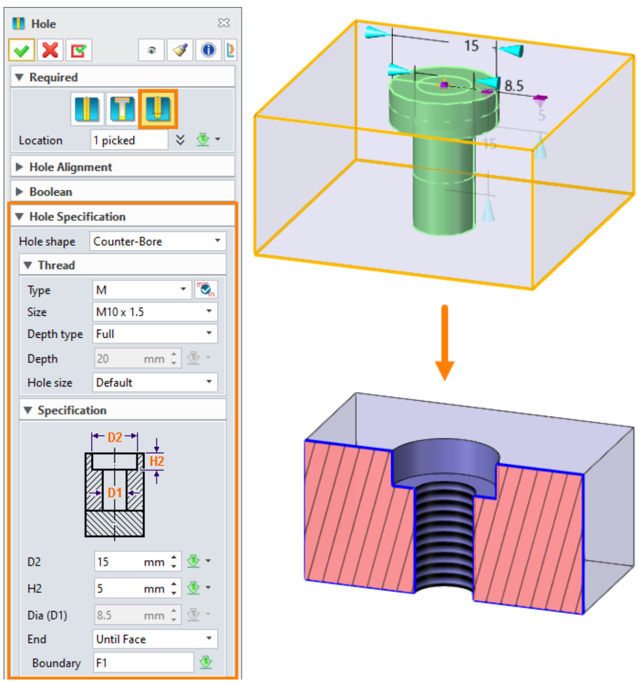


Figure 40 Hole – Counter-Bore Thread Hole

Here are more parameters of hole specification.

➢ **Metric and inch thread:** The thread specifications can be switched to metric or imperial.

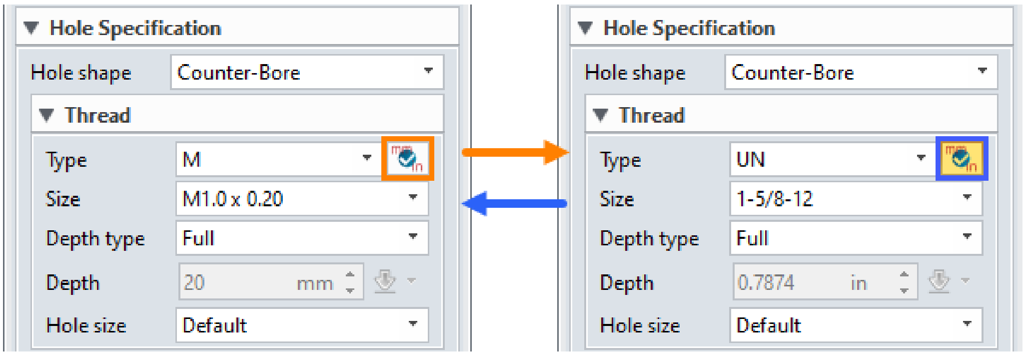


Figure 41 Hole – Switch Units

➢ **Set Chamfer:** Use this option to set chamfer of the hole.

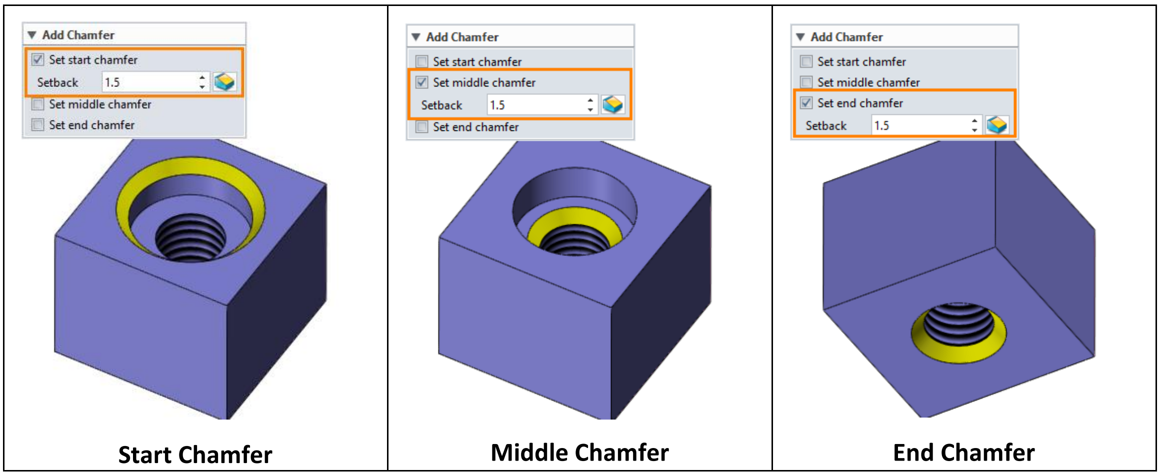


Figure 42 Hole – Set Chamfer

➢ **Hole Tolerance:** This option allows users to define the hole tolerance by value or tolerance table. This attribute can be shown in the ***Hole Table*** in 2D drafting.

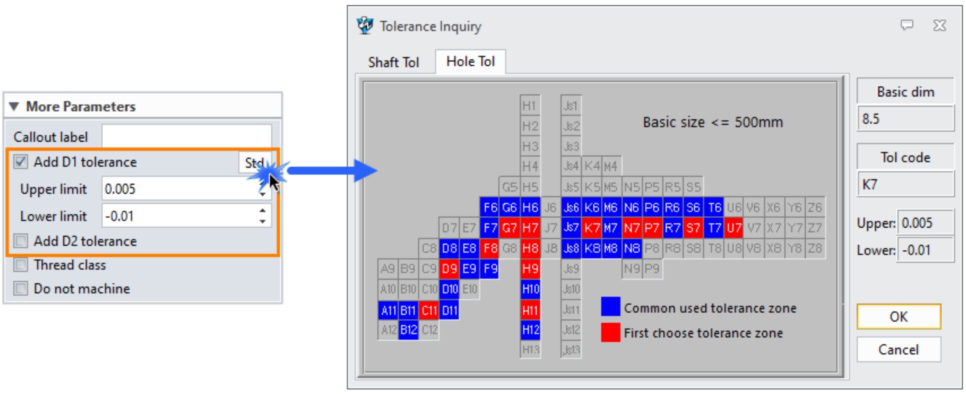


Figure 43 Hole Tolerance

➢ **Thread Class:** Define the thread class, such as 1B, 2B and 3B.

➢ **Do not machine:** With this attribute, holes will be ignored in ZW3D CAM.

4.5 Rib

Shape Ribbon Tab->Engineering Feature->Rib

Use this command to create a rib feature by an open profile sketch. The required inputs include the profile, rib thickness, draft angle, boundary faces and the draft plane. The boundary faces can limit or expand the extent of the rib feature.

STEP 01 Create a new sketch or select the existing sketch as the ***Profile*** of the rib feature.

STEP 02 Define others parameters as below.

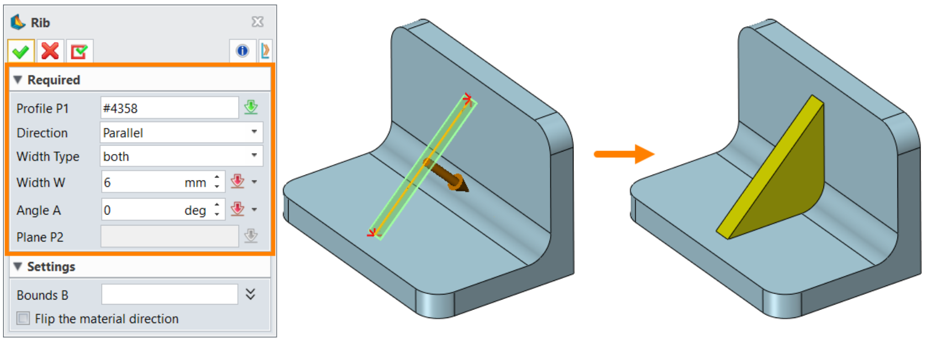


Figure 44 Rib

Shape Ribbon Tab->Engineering Feature->Rib Network

Use this command to create a network of interconnected ribs. The command supports multiple profiles to define the rib paths. Each profile can define rib sections of varying widths or you can use a single profile for constant rib widths.

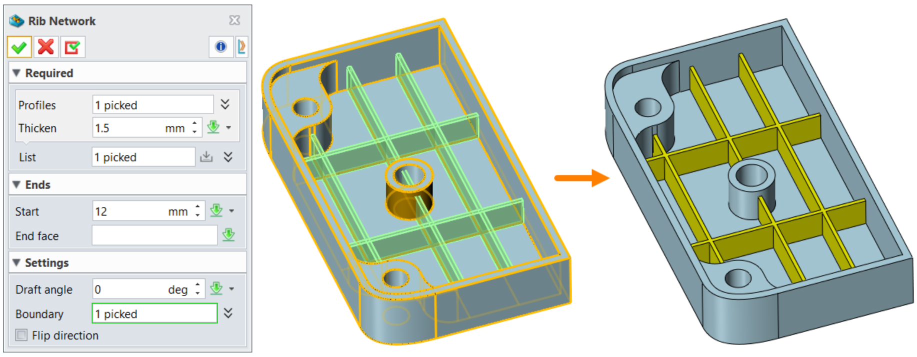


Figure 45 Rib Network

4.6 Thread

Shape Ribbon Tab->Engineering Feature->Thread

Use this command to create a real thread shape feature by revolving a closed profile around a cylindrical face and along its axis. This command can be used to make thread features or any other shape that is revolved in a linear direction.

STEP 01 Pick the face which needs to add the threaded feature.

STEP 02 Create a sketch as the thread profile, see the triangle in Figure 46.

STEP 03 Define the ***Turns*** and ***Pitch*** for the thread feature.

STEP 04 Set other parameters like ***Boolean*** and ***Lead in/out***.

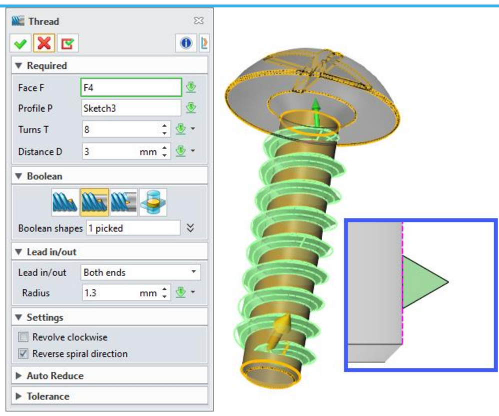


Figure 46 Thread

Shape Ribbon Tab->Engineering Feature->Flag Ext Thread

This command can be used to create texture external thread. Its advantage is to save memory and improve modeling efficiency. All of these attributes can be read by 2D drawings and CAM processing.

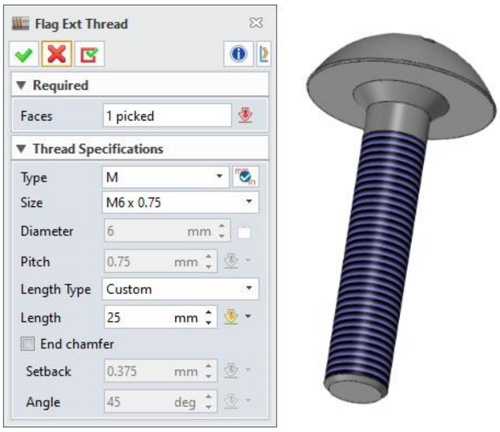


Figure 47 Flag Ext Thread

4.7 Lip

Shape Ribbon Tab->Engineering Feature->Lip

Use this command to create a constant lip feature along selected edges based on two offset distances. In this command, after picking an edge, users need to define the first side of the offset value.

STEP 01 Pick an edge then click the first side of the offset value; the following example is the vertical face.

STEP 02 Pick another edge and the corresponding first side face.

STEP 03 Set the offset values for both directions.

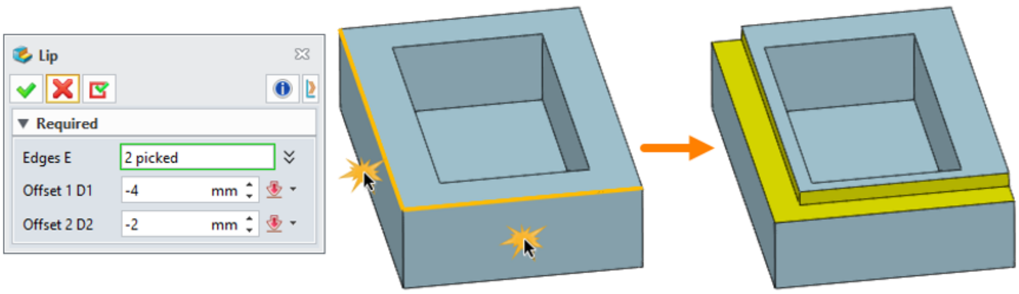
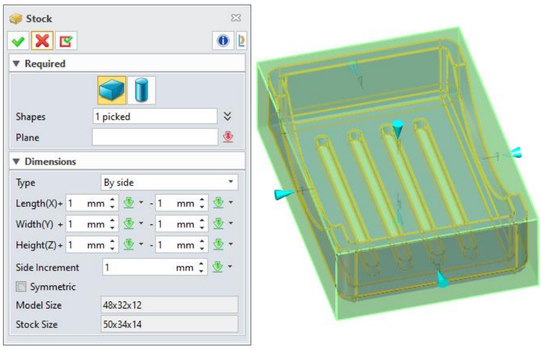


Figure 48 Lip

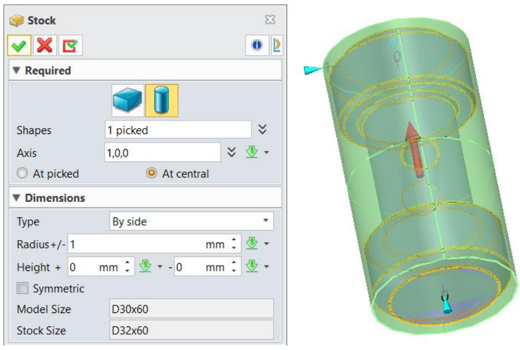
4.8 Stock

Shape Ribbon Tab->Engineering Feature->Stock

Use this command to generate a bounding box or cylinder for the picked objects. The objects can be shapes or surfaces. In this command, users can see the minimum envelope value, and set the offset value for the stock.



Stock – Block



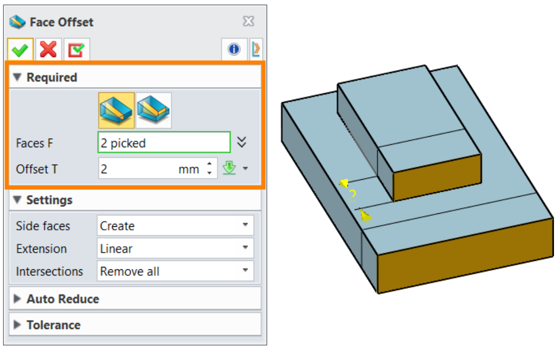
Stock – Cylinder

5 Editing Features

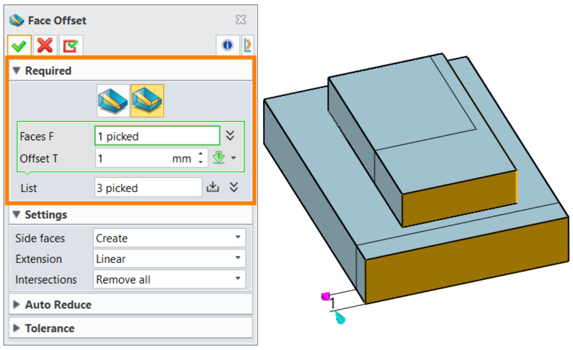
5.1 Offset

Shape Ribbon Tab->Edit Shape->Face Offset

Use this command to offset one or more faces of a shape. The offset can be equidistant or non-equidistant.



Face Offset – Constant



Face Offset – Variable

Shape Ribbon Tab->Edit Shape->Vol Offset

Use this command to offset the whole shape.

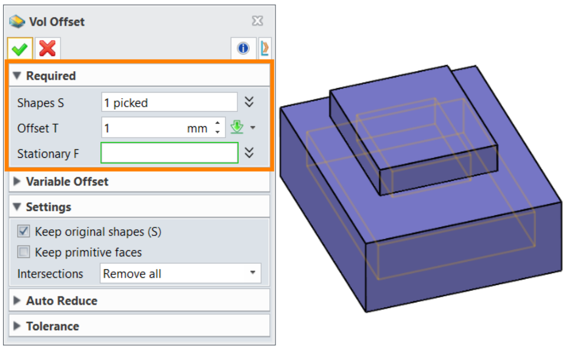


Figure 49 Vol Offset

5.2 Shell

Shape Ribbon Tab->Edit Shape->Shell

Use shell command to create an shell feature for a shape. As shown in the figures below, the original model is solid. After using the shell command, it will become equidistant hollow. Users can also define the open faces for the shell shape.

STEP 01 Pick the shape that need to be shelled.

STEP 02 Set the ***Thickness*** of the shell feature. A positive value means an outward offset, and a negative value means an inward offset.

STEP 03 And users can set the ***Opening***. The figure at the bottom left shows that the opening is not set, and the bottom right shows to set the two ends faces as the open.

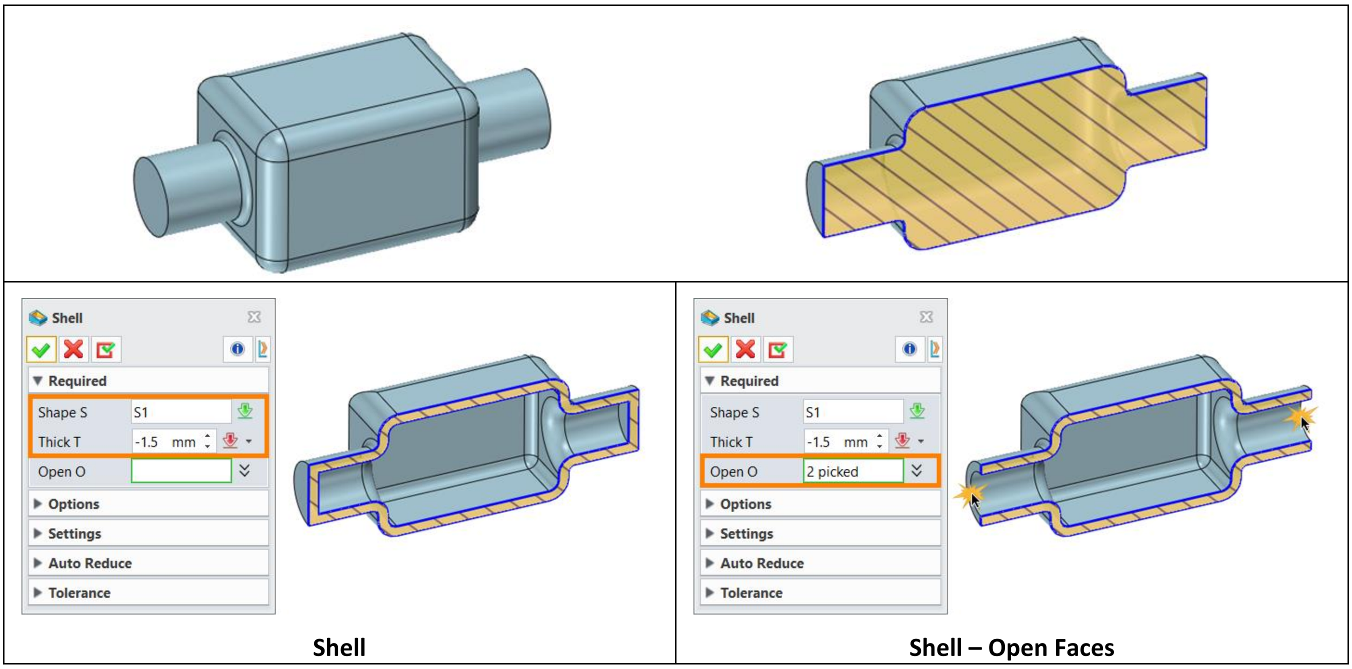


Figure 50 Shell

5.3 Thicken

Shape Ribbon Tab->Edit Shape->Thicken

Use this command to thicken a sheet or surfaces. Here is a sample of thickening a sheet.

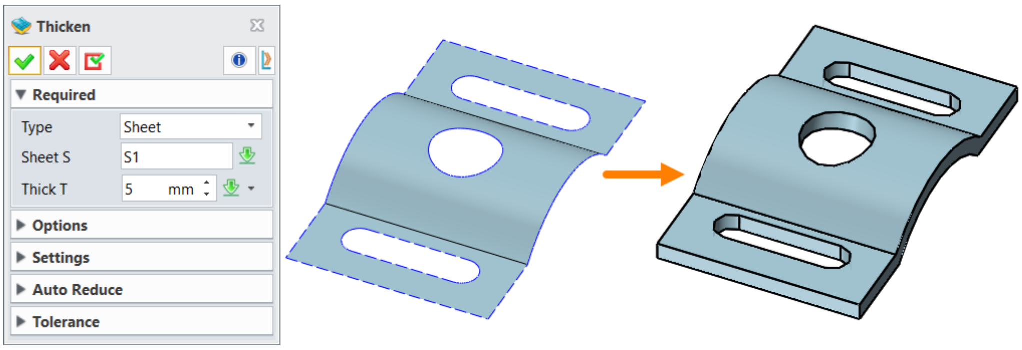


Figure 51 Vol Offset

5.4 Boolean Operation (Add & Remove & Intersect Shape)

Shape Ribbon Tab->Edit Shape->Add Shape/Remove Shape/Intersect Shape

This command can be used to do Boolean operations with shapes and surfaces. There are three types, add, remove, and intersect. The following figures are the Boolean operations between two shapes, the base is a block, and the operator (added/removed/intersected) is a cylinder.

STEP 01 Pick the block as the ***Base***.

STEP 02 Pick the cylinder as the ***Operator (Added/Removed/Intersected)***.

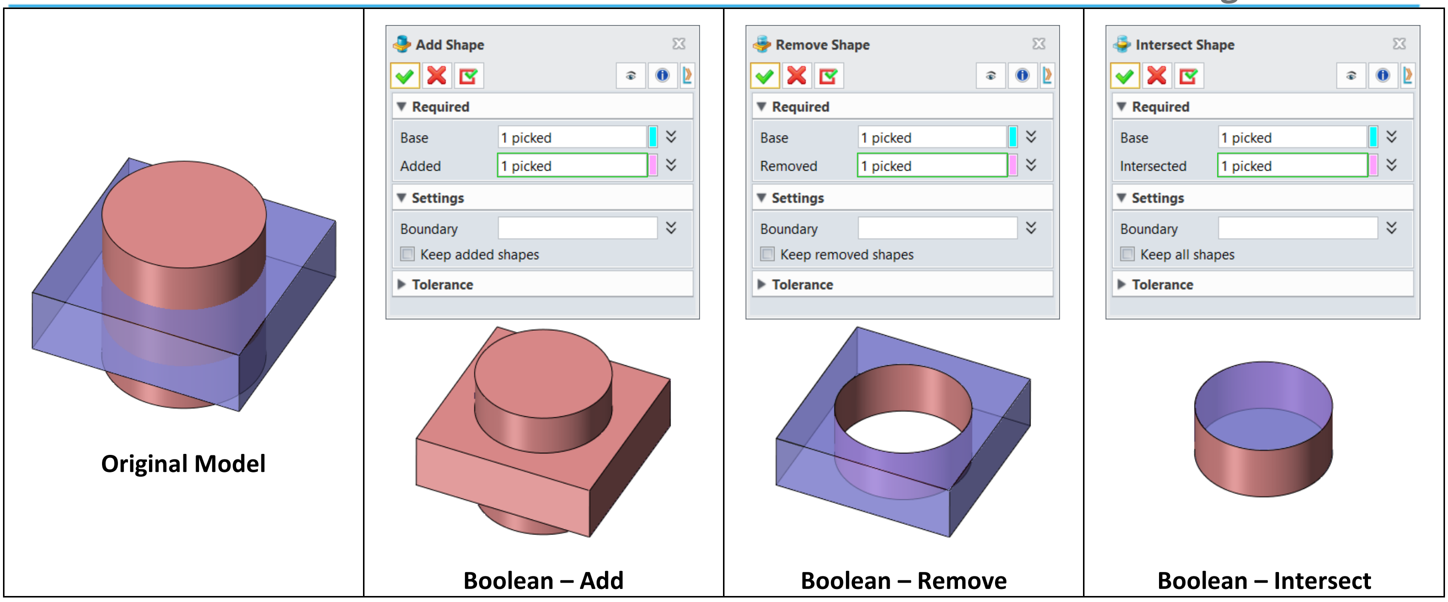


Figure 52 Boolean Operations

In addition, Boolean operations can also be done with solid and surface. The following figure is an example, the base is a cylinder, the added operator is a surface. The Boolean result is a surface.

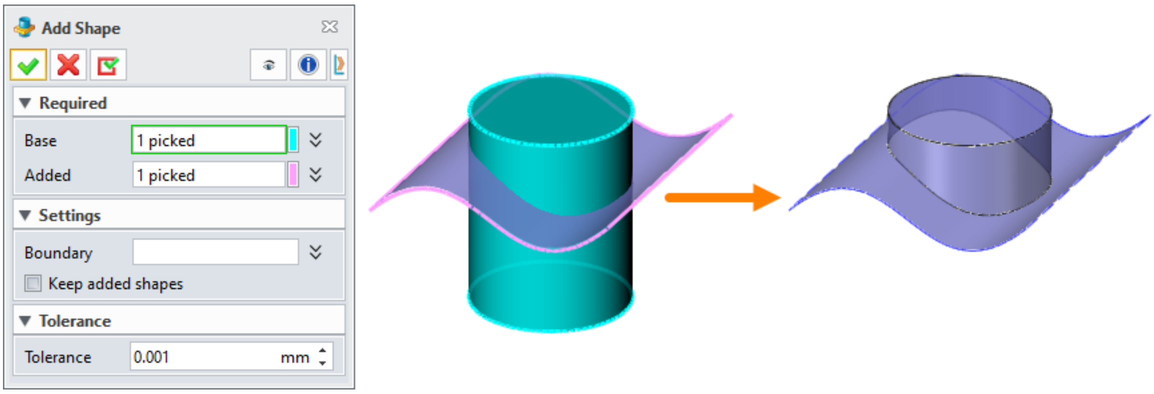


Figure 53 Boolean Operation – Solid with Surface

5.5 Trim & Divide

Shape Ribbon Tab->Edit Shape->Divide/Trim

Use this command to divide or trim entities. In this command, the base can be solid or surface, the operator can be solid, surface or datum. The following figures are the example of divide and trim command.

STEP 01 Pick the cylinder as the ***Base***.

STEP 02 Pick the surface as the ***Cutting/Trimming***.

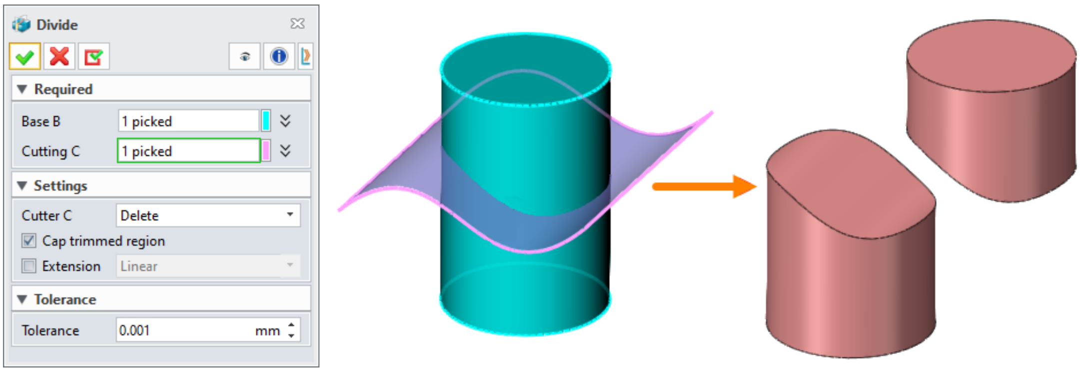


Figure 54 Divide

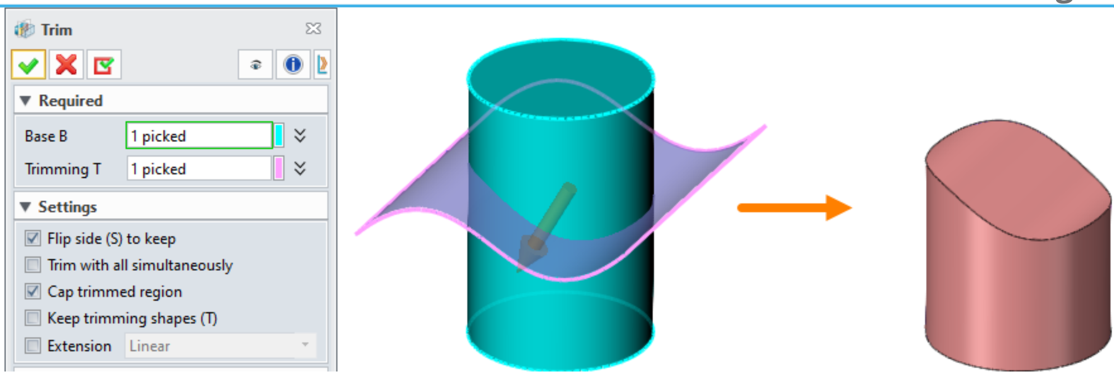


Figure 55 Trim

5.6 Simplify

Shape Ribbon Tab->Edit Shape->Simplify

This command can be used to simplify some existed features from a model. The feature can be a hole, fillet, chamfer, pocket, draft, etc. This command is usually used in direct edit modeling, mold and electrode design and so on. Here are some cases of simplify command.

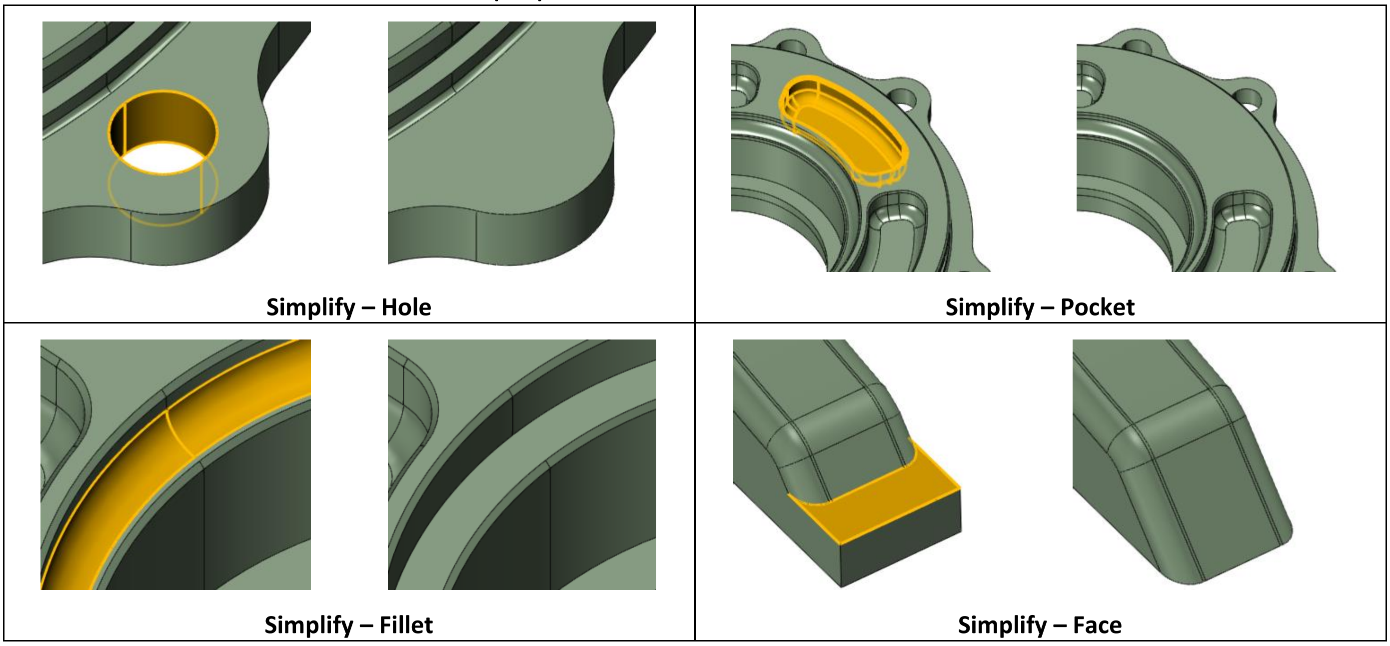


Figure 56 Simplify

5.7 Replace

Shape Ribbon Tab->Edit Shape->Replace

Use this command to replace a surface by an existing surface. Both the original surface and the replaced surface can be flat or curved surface.

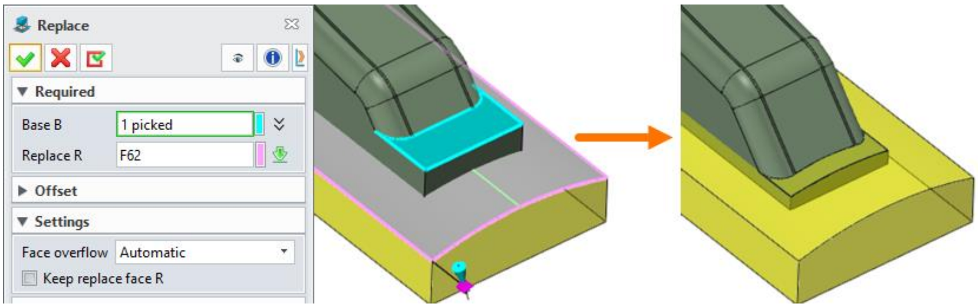


Figure 57 Replace

6 Pattern Features

In many cases, users need to reuse the geometries or features. Therefore, ZW3D provides a series of pattern features, including pattern, mirror, move, copy, scale, etc.

6.1 Pattern Geometry

Shape Ribbon Tab->Basic Editing->Pattern Geometry

Use this command to pattern geometries, such as shapes, faces, curves, points, datums, etc. The following is a figure of the basic pattern feature, linear pattern. In this command, users need to define the base pattern geometries, one or two pattern directions, pattern number and spacing.

STEP 01 Pick the small block as the ***Base***.

STEP 02 Define the Z-axis as the main direction, set ***Number*** as 2, and ***Spacing*** as **20 mm**.

STEP 03 Define the X-axis as the second direction, set ***Number*** as 8, and ***Spacing*** as **10 mm**.

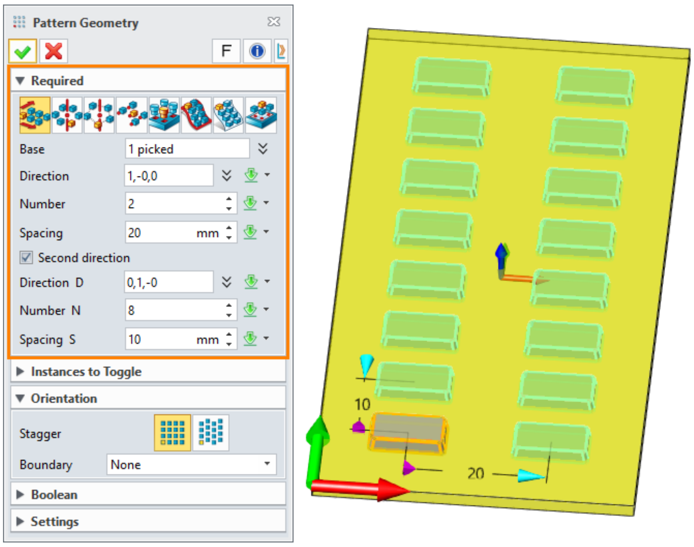
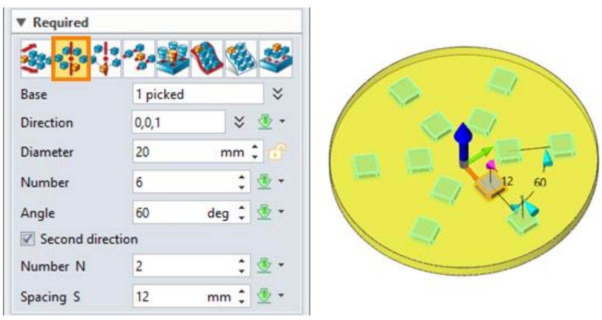
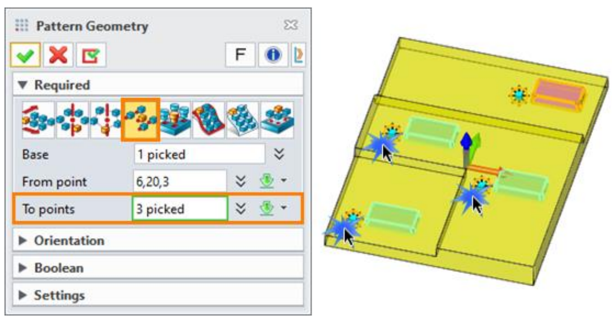


Figure 58 Pattern Geometry – Linear

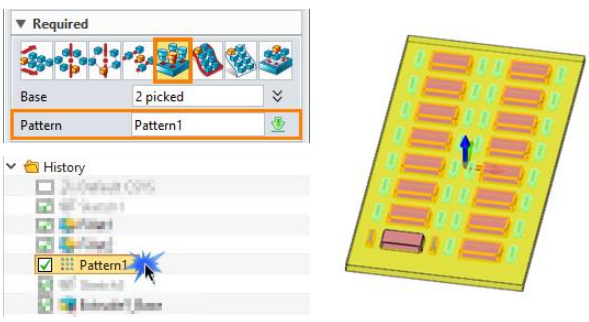
And, eight different types of pattern geometry are provided, including linear, circular, polygon, point to point, at pattern, at curves, at face, and fill pattern. The following are some results.



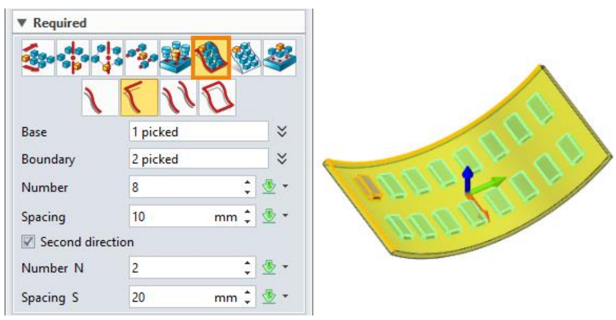
Pattern Geometry – Circular



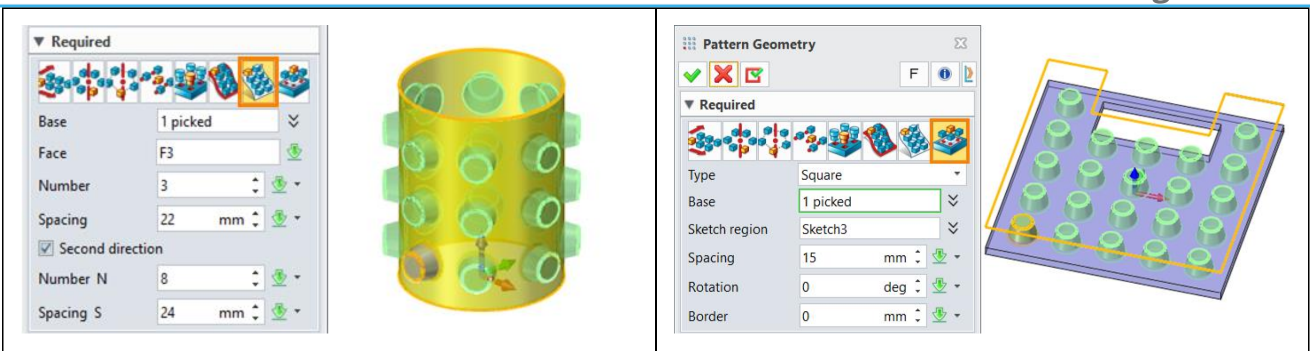
Pattern Geometry – Point to Point



Pattern Geometry – At Pattern



Pattern Geometry – At Curves



Pattern Geometry – At Face **Pattern Geometry – Fill**

Figure 59 Pattern Geometry – Different types

And, the following is about more parameters introduction in pattern command.

- **Derived Pattern:** Users can derive either ***Number*** or ***Spacing*** value of the pattern feature. There are three options.
  None – Input the ***Number*** and ***Spacing*** of the pattern feature without deriving, see Figure 60.
  Spacing – Input the ***Number*** of the pattern feature, then the spacing (like distance, angle) will be derived automatically, see Figure 61.
  Number – Input the ***Spacing*** value of the pattern feature, then the pattern number will be derived automatically, see Figure 62.

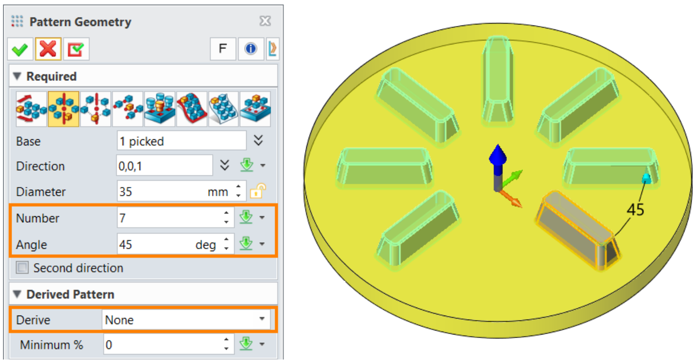


Figure 60 Derived Pattern – None

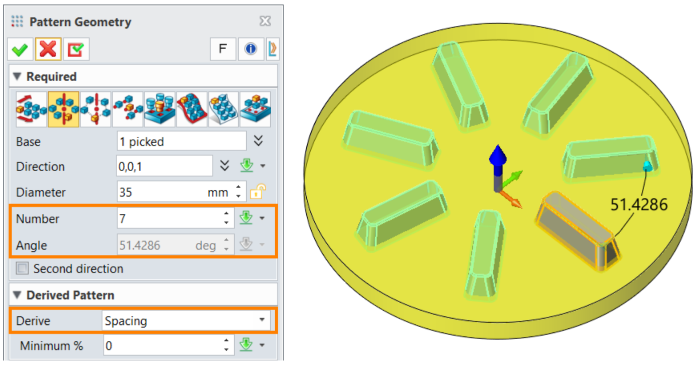


Figure 61 Pattern Geometry – Linear

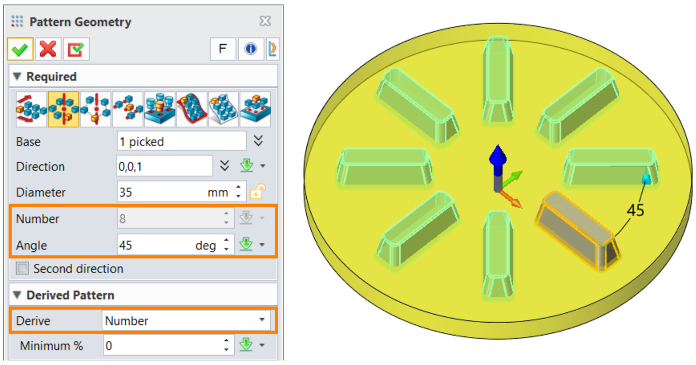


Figure 62 Pattern Geometry – Linear

- **Instances to Toggle:** Use this option to pick the pattern entities that need to be excluded. In the preview, the picked entities will be displayed in red and removed from the result. Pick the red entities to keep it back to the result.

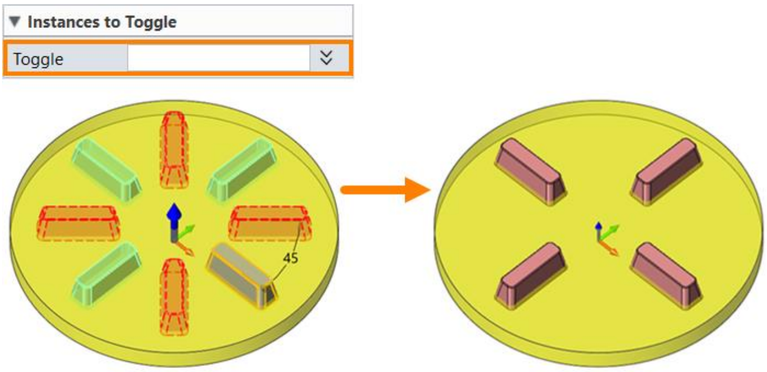


Figure 63 Pattern Geometry – Instances to Toggle

- **Alignment:** The alignment has two options, align with base and align with pattern.

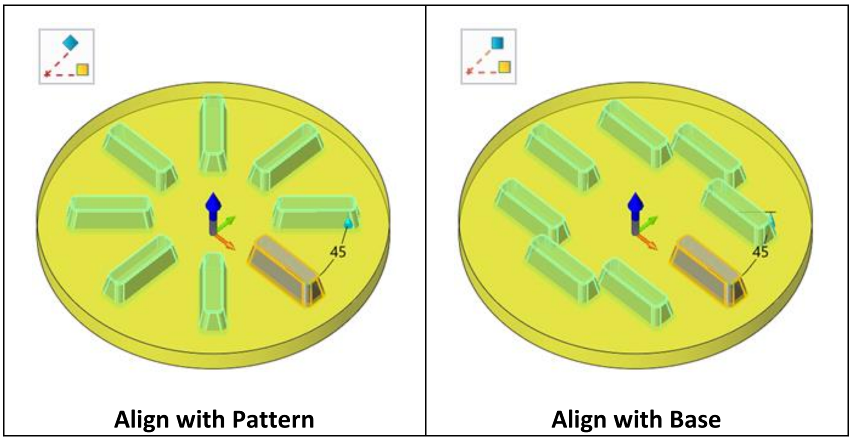


Figure 64 Pattern Geometry – Alignment

- **Stagger:** Activating this option will stagger the distance of the rows in the second direction.

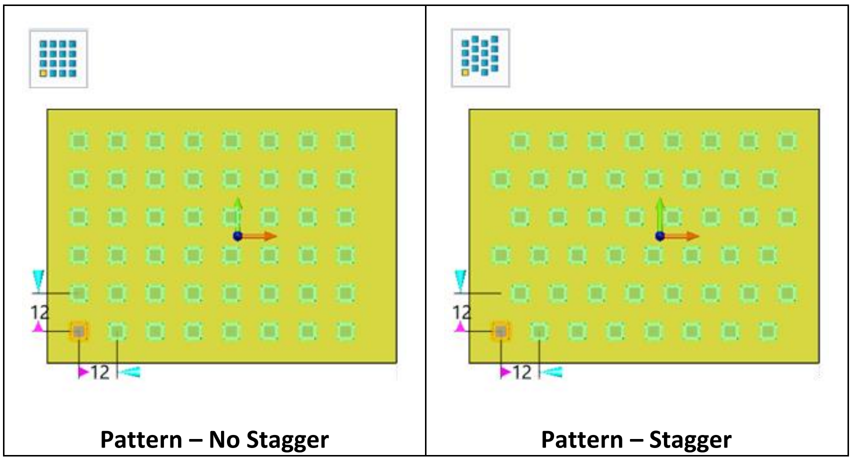


Figure 65 Pattern Geometry – Stagger

- **Associative Copy:** Check this option, the pattern entities will keep the association with the original entity, and the pattern feature can be redefined. But if this option is not checked, the pattern entities will be created as static geometries. They are independent and the pattern feature cannot be redefined again.

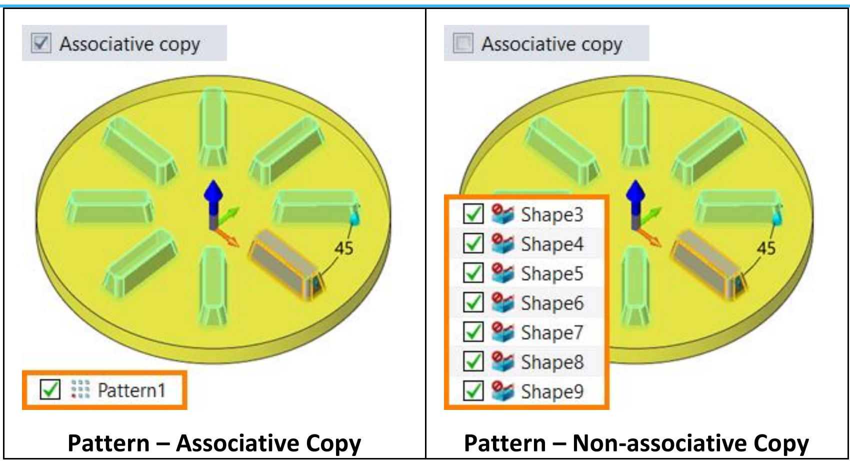


Figure 66 Pattern Geometry – Associative Copy

6.2 Pattern Feature

Shape Ribbon Tab->Basic Editing->Pattern Feature

Use this command to pattern features directly on the modeling history tree. This command is similar to ***Pattern Geometry***, it also provides multiple pattern types. Here is an example of patterning ribs and holes.

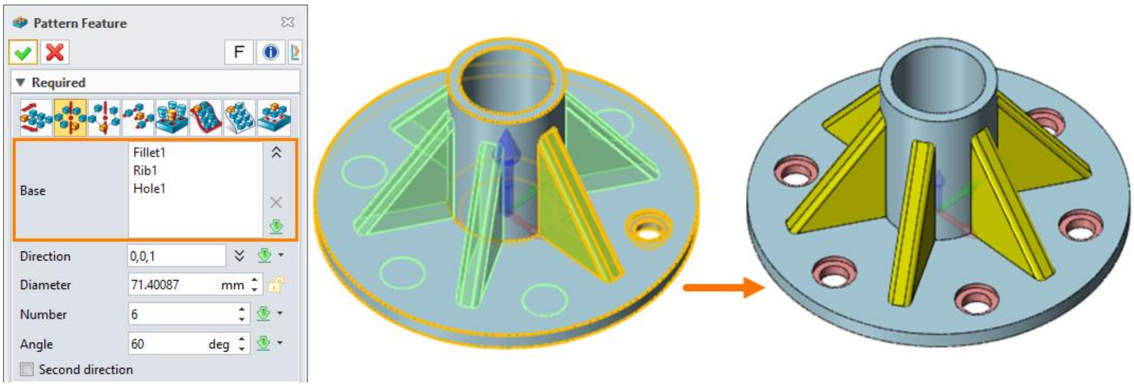


Figure 67 Pattern Feature

- **Variable Pattern:** This option allows users to add the variables for the picked parameters of the pattern features. There are two different types: parameter list is used to set the constants increment, and parameter table is used to set scattered values. Figure 68 is a pattern of hole feature. The holes are patterned along with the path with a diameter increment of 1 mm.

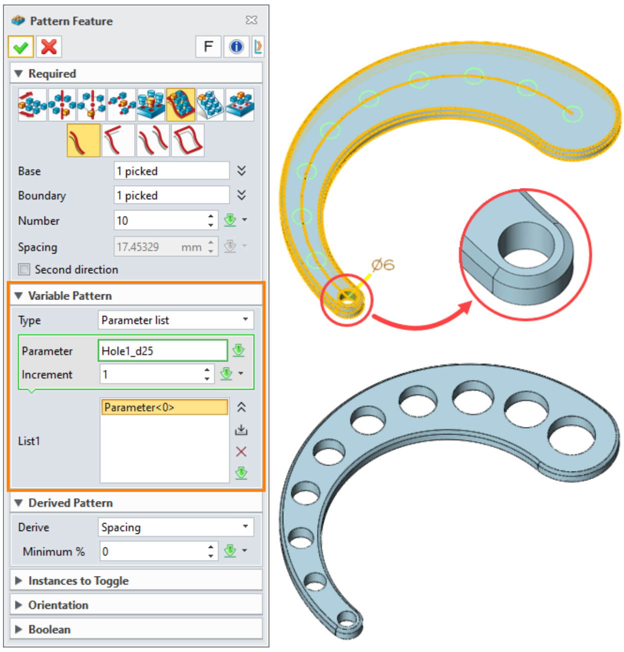


Figure 68 Pattern Feature

6.3 Mirror Geometry & Mirror Feature

Shape Ribbon Tab->Basic Editing->Mirror Geometry/Mirror Feature

The usage of these two commands is the same as ***Pattern***, one is for mirroring geometries, the other is for mirroring features.

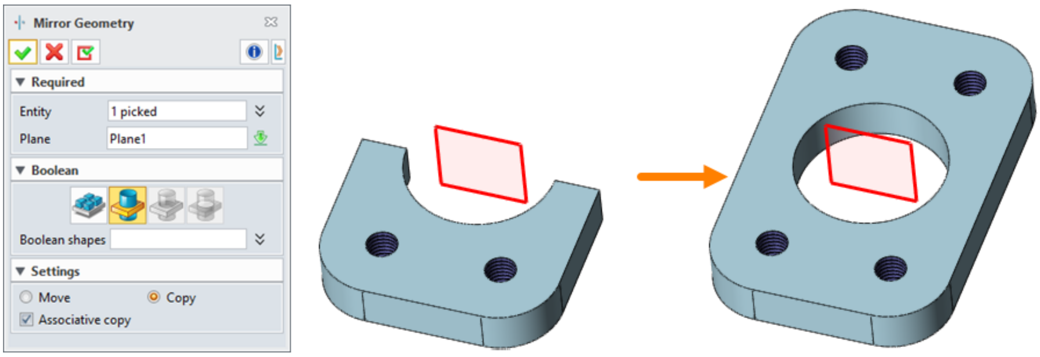


Figure 69 Mirror Geometry

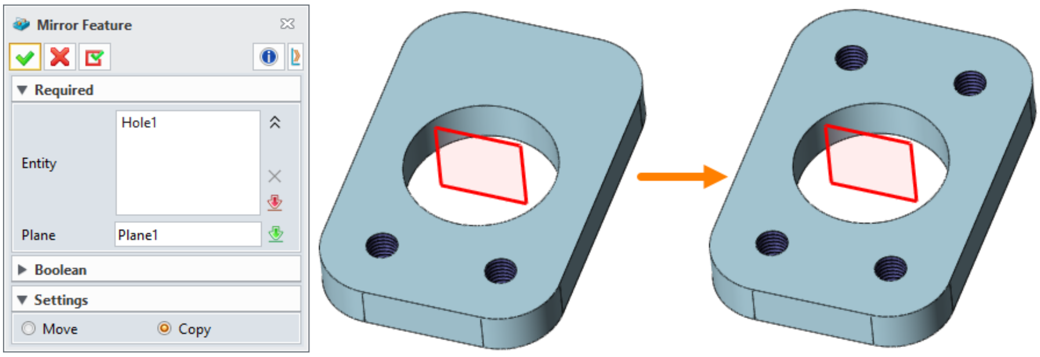
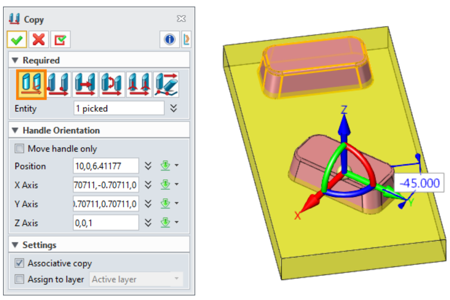


Figure 70 Mirror Feature

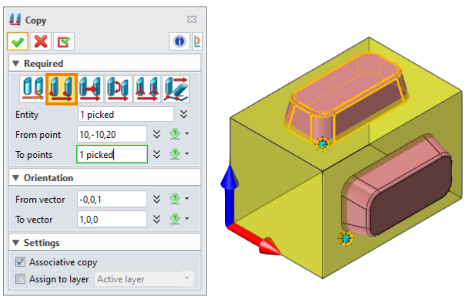
6.4 Move/Copy

Shape Ribbon Tab->Basic Editing->Move/Copy

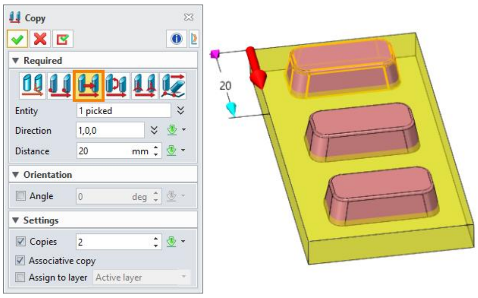
Use this command to ***Move*** or ***Copy*** entities. ZW3D provides six different methods, which are dynamic, from point to point, along with a direction, rotate around an axis, align with frame, along with a path.



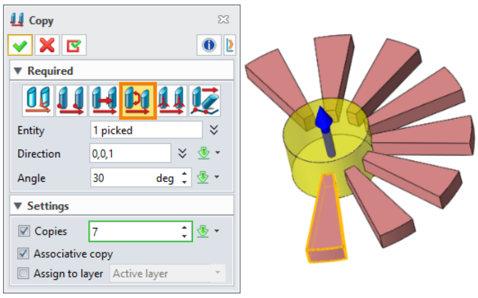
Copy – Dynamic Handle



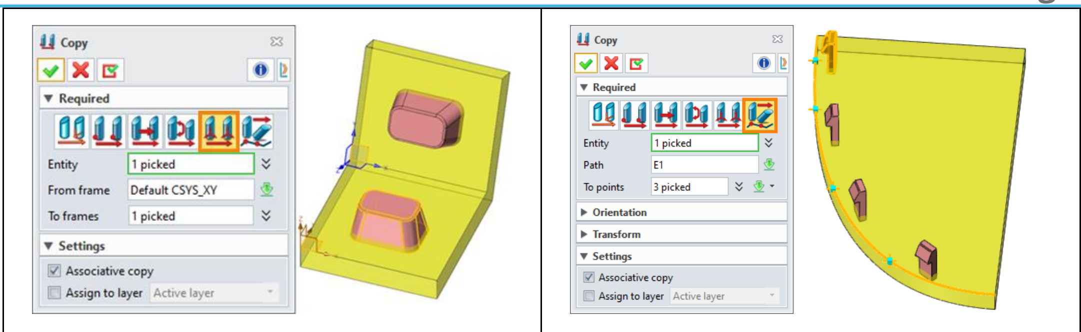
Copy – From Point to Point



Copy – Along with a Direction



Copy – Rotate around an Axis



Copy – From Frame to Frame **Copy – Along Path**

Figure 71 Copy

6.5 Scale

Shape Ribbon Tab->Basic Editing->Scale

Use this command to scale the entities with a uniform or non-uniform proportion.

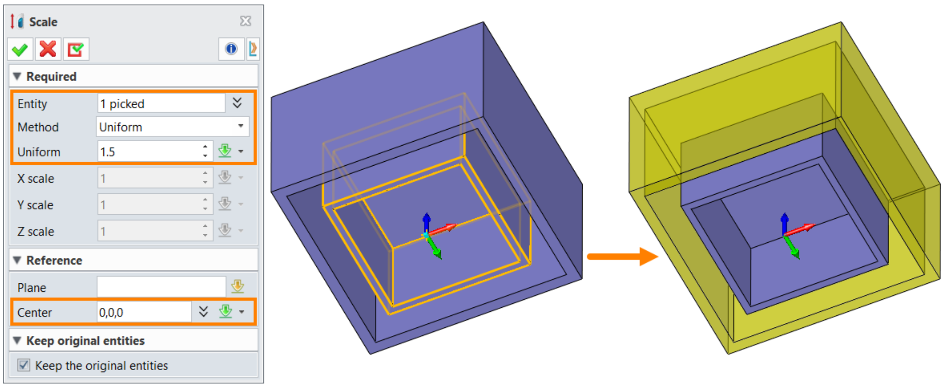


Figure 72 Scale

7 Morph Features

ZW3D provides some deformation features, like bend, morph, etc. Using these features, users can quickly change the shape of the entity. Of course, this kind of command is usually used in models that are not strict on parameters.

7.1 Bend

Shape Ribbon Tab->Morph->Cylindrical Bend

Use this command to bend shape with a cylinder. There are two methods to define the shape, bending radius or bending angle. The figure below shows the bending radius method.

STEP 01 Pick the rack as ***Shape*** and pick the bottom face as the ***Datum***.

STEP 02 Use ***Radius*** method to bend and set the ***Radius*** value as **50 mm**.

STEP 03 Set other parameters as below to get the result.

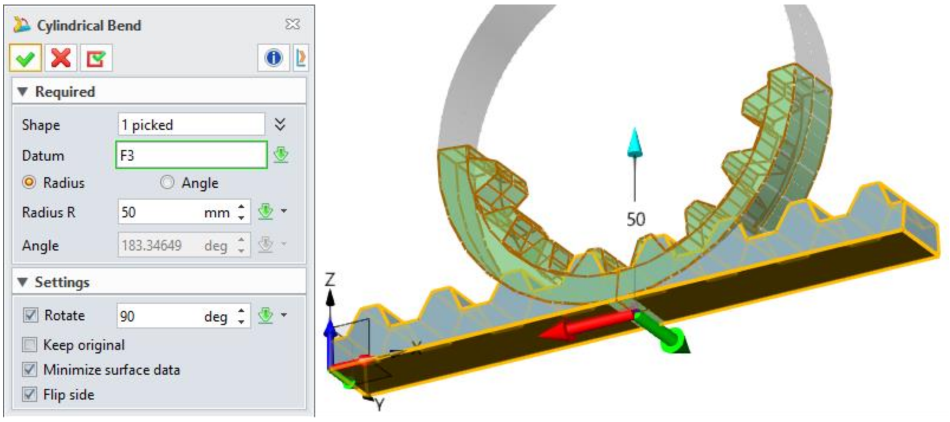


Figure 73 Cylindrical Bend

Shape Ribbon Tab->Morph->Toroidal Bend

Use this command to bend shape with a torus, sphere or ellipsoid. Toroidal bend is widely used for ring, bracelet and bottle design.

STEP 01 Pick the shape and pick the bottom face as the ***Datum***.

STEP 02 Set the parameters as below.

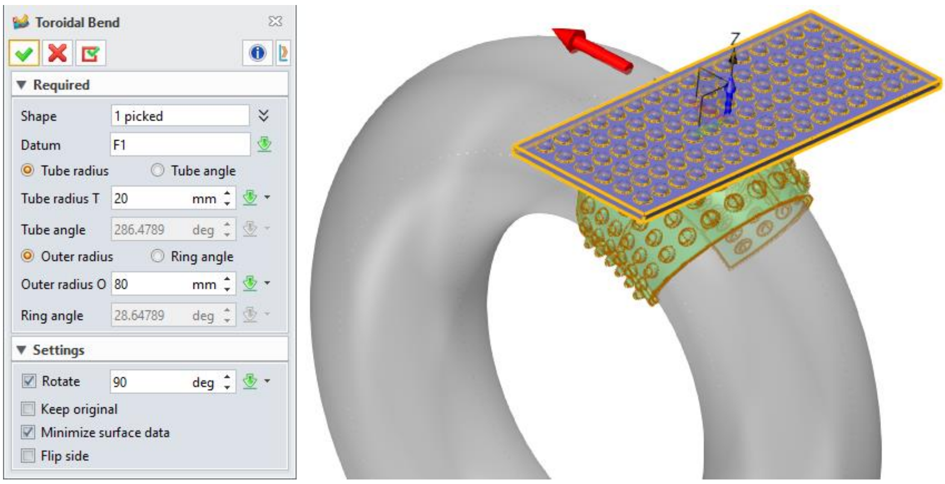


Figure 74 Toroidal Bend

Shape Ribbon Tab->Morph->Twist

This command is also named as Sprial Bend. It is used to twist a shape along specified axis. Twist feature is widely used for gear, cutter and drill design.

STEP 01 Pick the block as ***Shape*** and pick the square face as the ***Datum***.

STEP 02 Define the ***Twist Range*** as **-50 mm** and the ***Twist Angle*** as 360 deg.

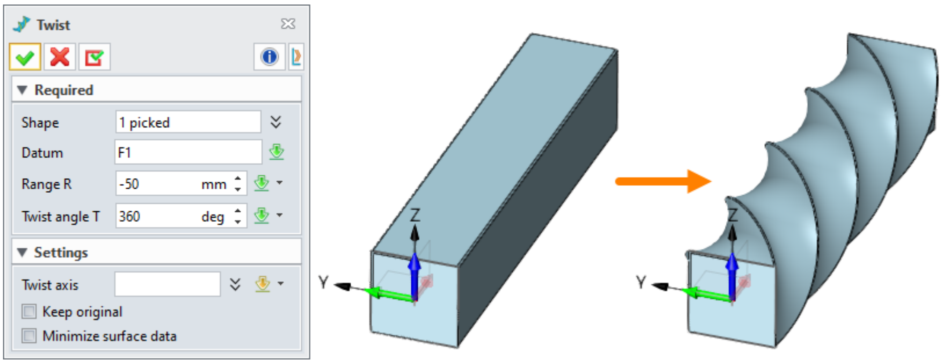


Figure 75 Twist

Shape Ribbon Tab->Morph->Taper

Use this command to taper a shape and make it smaller in the specified side. This command is similar to Draft command. It is a substitute for draft function in some cases.

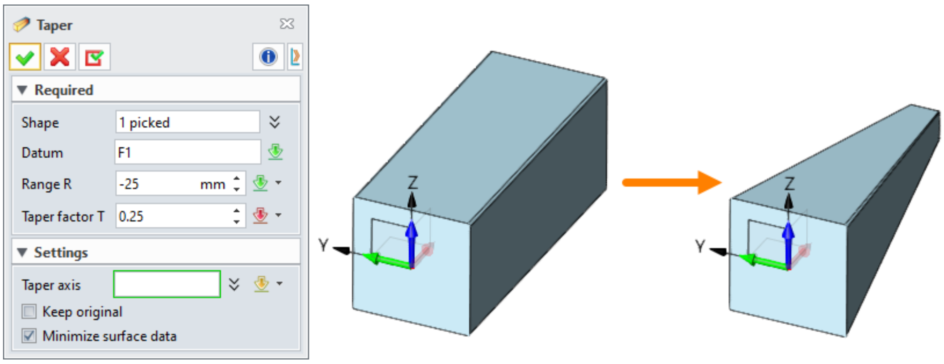


Figure 76 Taper

Shape Ribbon Tab->Morph->Stretch

Use this command to stretch shape along X, Y and Z direction in specified range. Stretch command is difference with Scale command. The scaling factor and the stretch effect of each point is different.

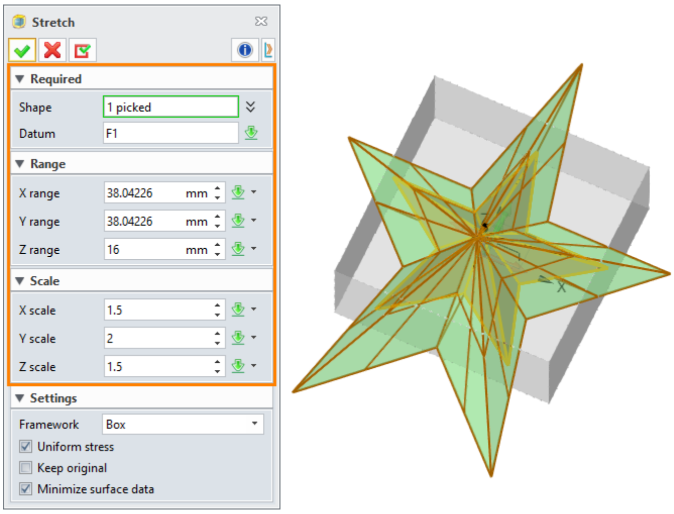


Figure 77 Stretch

7.2 Morph

Shape Ribbon Tab->Morph->Morph with Point

Use this command to transform a shape by warping face geometry. The modification is not limited to a single face, but work across edges to maintain the integrity of the solid shape. The transformation is performed by grabbing a point on a face and dragging it in different ways. There are six ways that you can move the point on a face. The figure below shows the translate along direction type.

STEP 01 Pick the spoon shape as ***Geometry***.

STEP 02 Define one or more points as the morph reference. The figure below is a point.

STEP 03 Define the Z-axis as ***Direction*** and the ***Morph Distance*** is **25 mm**.

STEP 04 Set the ***Influence*** as **75 mm** and others as below.

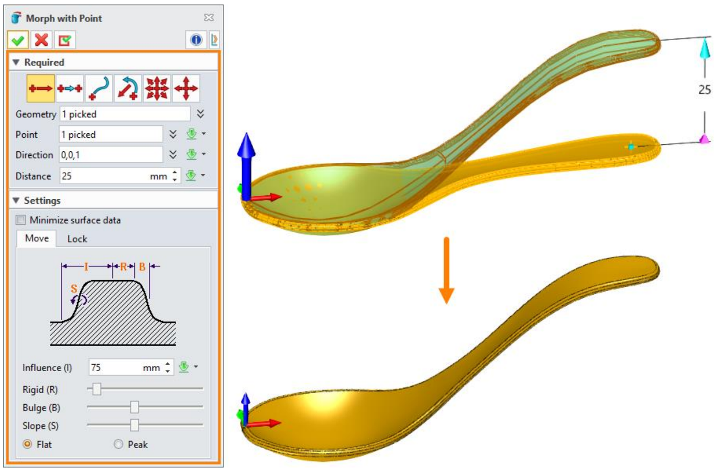


Figure 78 Morph with Point

- **Influence:** Specify the radius of the influence scope.
- **Rigid:** Specify radius for rigid motion. For example, R=100 means 100% of the Radius of Influence.
- **Bulge:** Specify bulge factor for transition region.
- **Slope:** Specify slope for transition region.
- **Flat/Peak:** This determines whether the surfaces at the new point location are rounded (flat) or sharp (peak). This option is useful when dragging a point on the corner or edge of a part.

Shape Ribbon Tab->Morph->Morph with Curve

This command is similar to the ***Morph Shape at Point*** command. It also morphs a shape by warping face geometry. This command allows users to grab a curve on a model or a datum plane that cuts through a model. The surfaces near the curve or plane are dragged just as surfaces near a point.

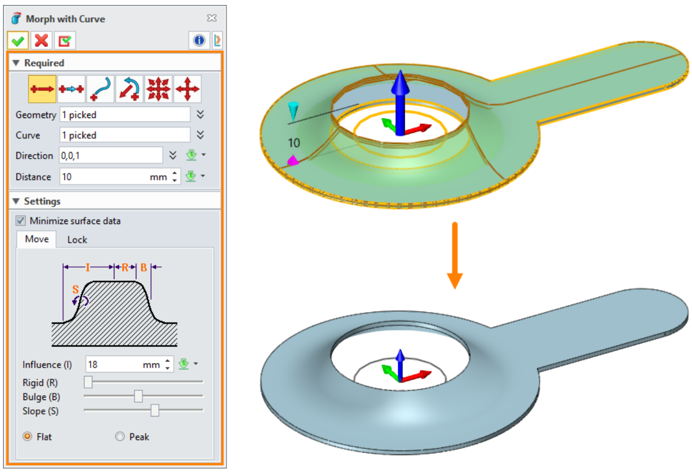


Figure 79 Morph with Curve

Shape Ribbon Tab->Morph->Morph by Offset

This command is similar to the ***Morph Shape at Curve*** command. It also morphs a shape by warping face geometry. This command allows users to grab a curve on a shape and offset it rather than moving it.

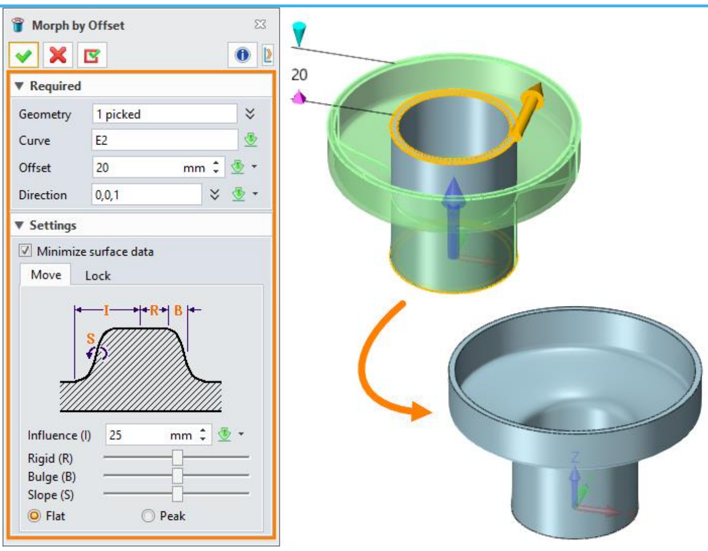


Figure 80 Morph by Offset

Shape Ribbon Tab->Morph->Morph to Curve

This command is similar to the ***Morph Shape at Point*** and ***Morph Shape at Curve*** commands. This command allows users to grab a curve on a model and morph it to a destination curve. The surfaces near the curve are modified.

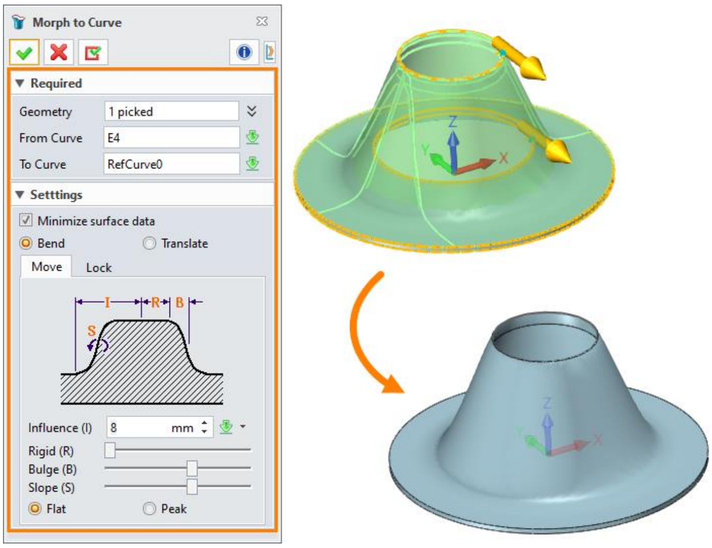


Figure 81 Morph to Curve

8 Case for Solid Modeling

In this chapter, you will learn how to use ZW3D for solid modeling through two cases.

8.1 Case1

In this case, more basic features will be introduced, like datum, extrude, Booleans, fillet, hole, etc.

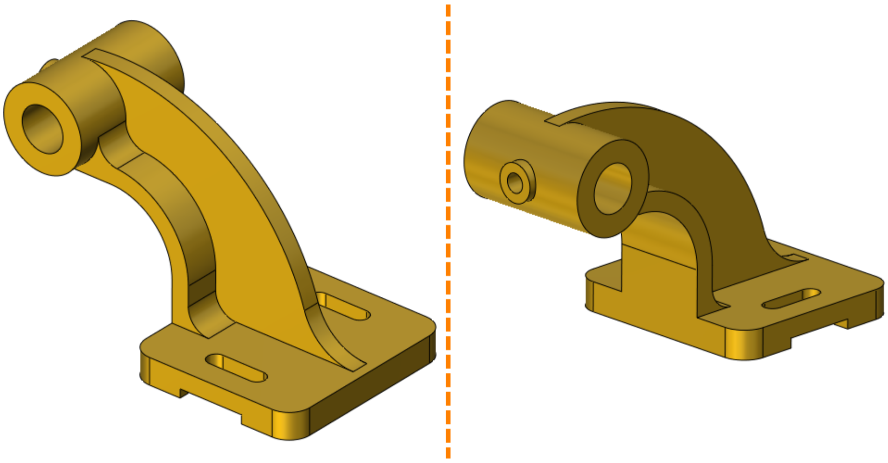


Figure 82 Case1 – Support

1) Create the base shape

STEP 01 Create sketch1 on the **XY** plane, draw a rectangle by **80 mm × 90 mm**.

STEP 02 Use ***Extrude*** command, pick the sketch1, set the ***Extrude Type*** as **Symmetrical**, the ***Extrude Value*** on each side is **6 mm**, define the ***Direction*** as **Z-axis**, as Figure 83.

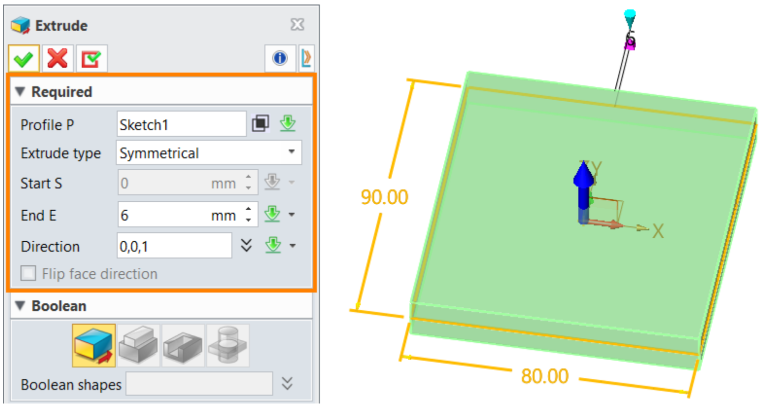


Figure 83 Case1 – Extrude Base

STEP 03 Create sketch2 on the **XZ** plane, draw a rectangle as below.

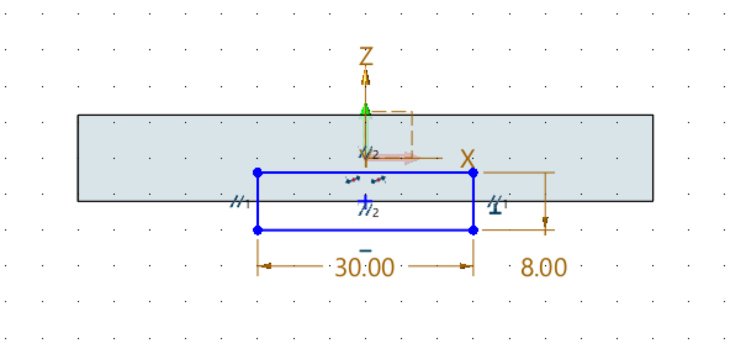


Figure 84 Case1 – Sketch2

STEP 04 Extrude the sketch2, keep the ***Extrude Type*** as **Symmetrical**, check the drop-down list and select **Through All** option. Pick the **Remove** Boolean option.

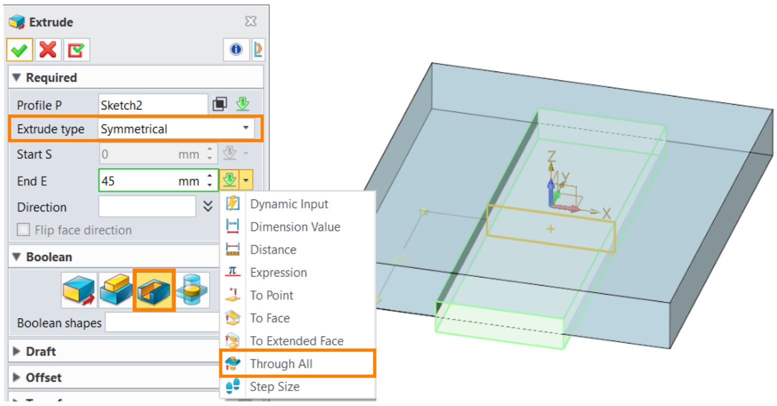


Figure 85 Case1 – Extrude Cut

2) Create datum

STEP 05 Create the Datum1 (blue color) by using ***Datum*** command, pick ***XY Plane*** option and set the offset value as **74 mm**.

STEP 06 Use the same method to create Datum2 (red color) by offset the **YZ** plane and the offset value is **-95 mm**, see the Figure 86.

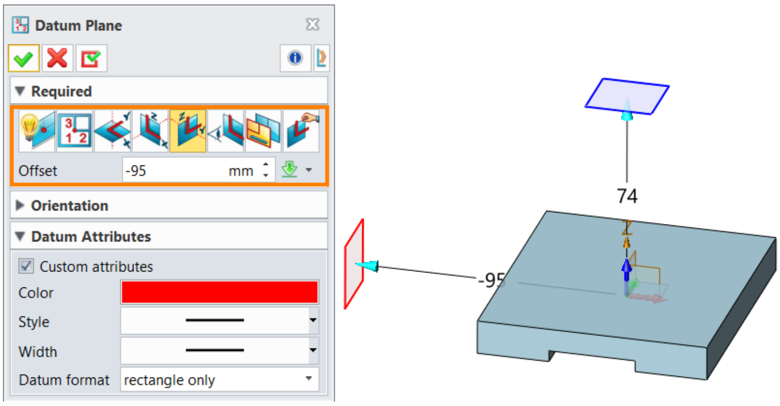


Figure 86 Case1 – Create datum

3) Create the second shape

STEP 07 Create sketch3 on the **XZ** plane, then refer Datum1 and Datum2, draw a circle aligns with those two references which radius is **19 mm**.

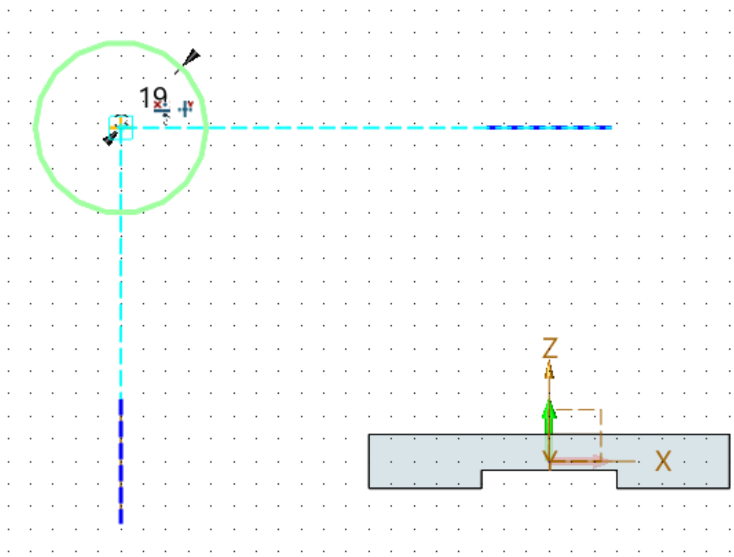


Figure 87 Case1 – Sketch3

STEP 08 Extrude the sketch3 with **Symmetrical** type, the value is **30 mm** and the ***Boolean Type*** is **Base**.

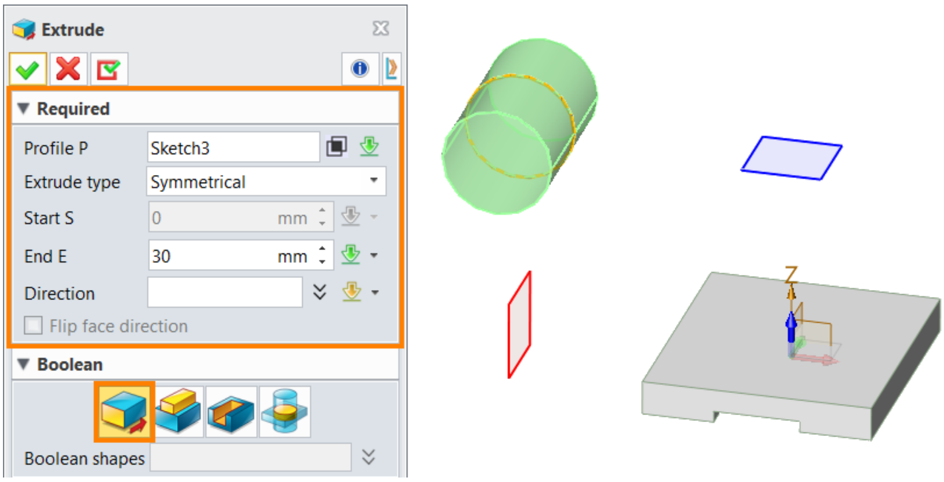


Figure 88 Case1 – Extrude the Sketch3

STEP 09 Create sketch4 on the Datum2, refer the Datum1 then draw a **8 mm** circle at the middle point.

STEP 10 Use ***Extrude*** command for sketch4, define the ***Extrude Type*** as **2 Sides**, set the ***Start*** value is **0 mm** and the ***End*** value is **-22 mm**, the ***Boolean Type*** is **Add**, see as below.

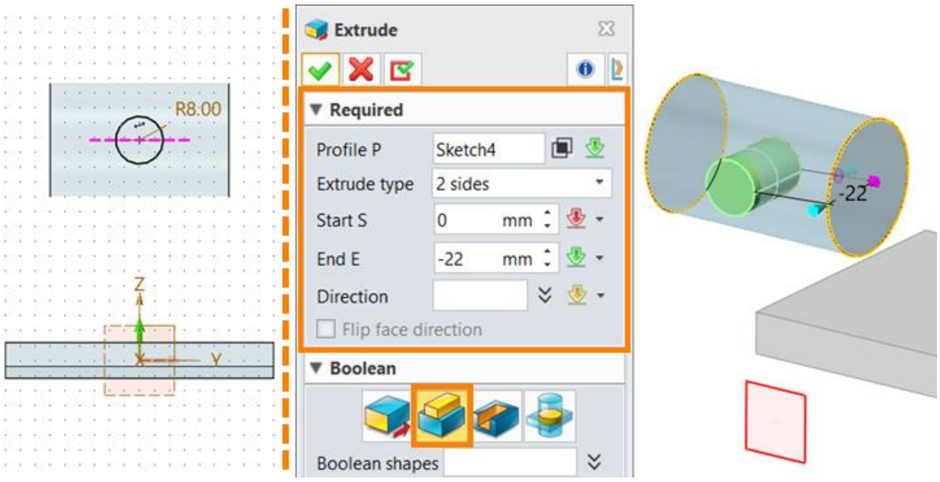


Figure 89 Case1 – Extrude the Sketch4

4) Create the combination

STEP 11 Draw sketch5 on the **XZ** plane and as Figure 90.

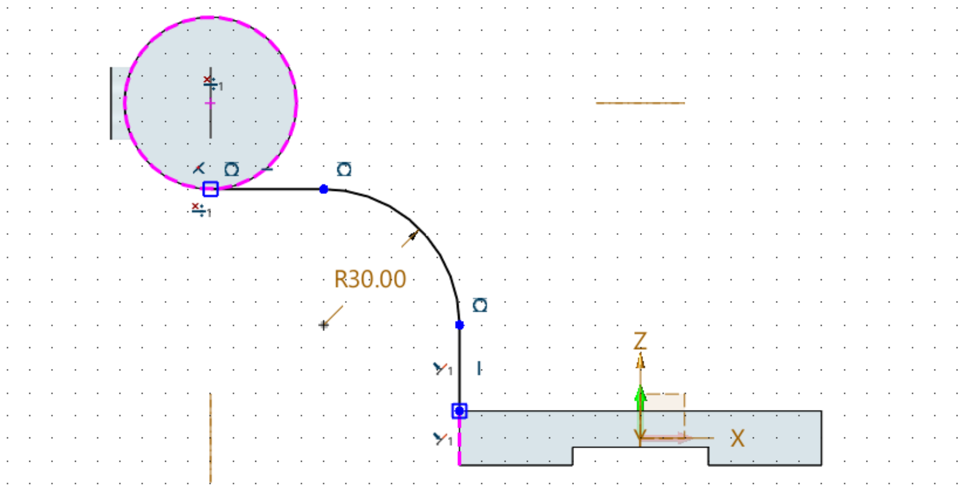


Figure 90 Case1 – Sketch5

STEP 12 Extrude the sketch5 with an offset value, the ***Boolean Type*** is **Add** and pick the shape1 and shape2 in the modeling space. The detailed parameters as shown below.

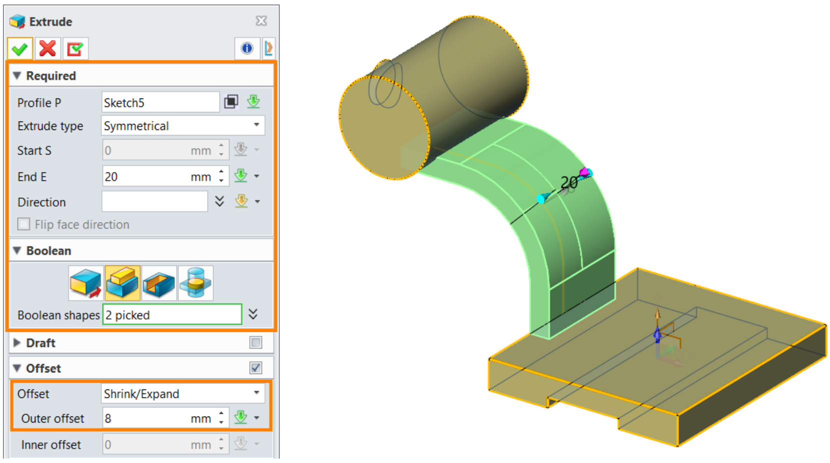


Figure 91 Case1 – Extrude the Sketch5

STEP 13 Create sketch6, draw an arc as Figure 92.

STEP 14 Extrude the sketch6, the parameters refer to Figure 92, pick the highlight surface as the profile cap.

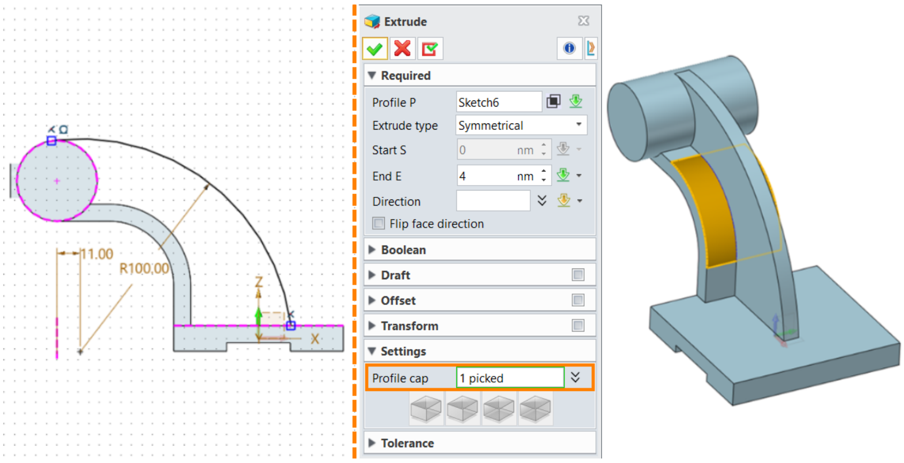


Figure 92 Case1 – Extrude the Sketch6

5) Create the hole

STEP 15 Use ***Hole*** feature to generate a general through-hole with **20 mm** diameter, set the ***End*** option as **Thru All**, see the left of Figure 96.

STEP 16 Use ***Hole*** feature again to generate a general hole with **8 mm** diameter, set the ***End*** option as **Until Face** and pick the Hole1 as the boundary, see the right of Figure 96.

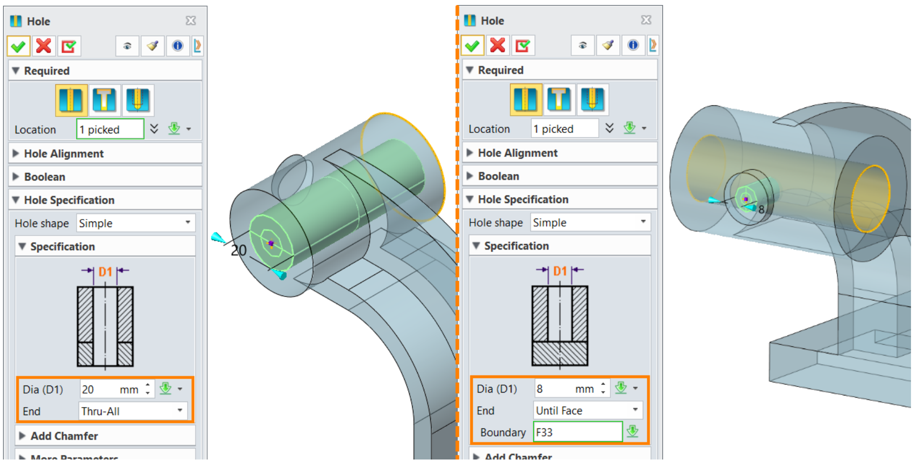


Figure 93 Case1 – Hole

6) Create the slot

STEP 17 Create sketch7 on the **XY** plane as below.

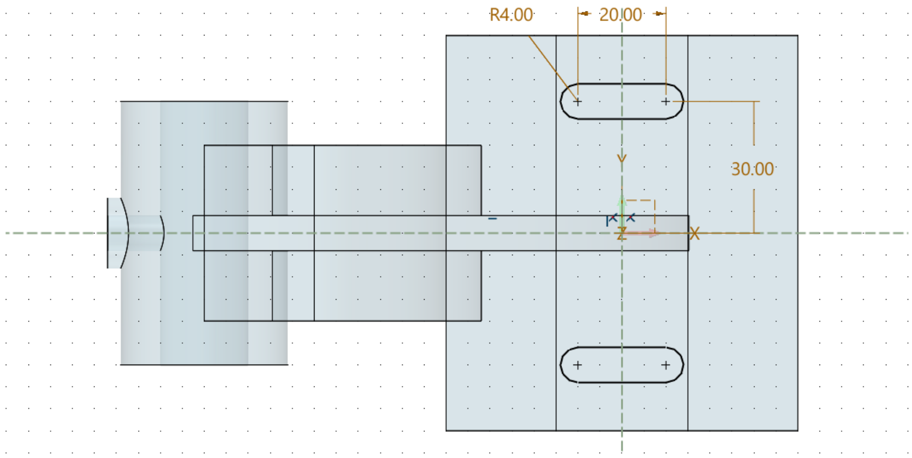


Figure 94 Case1 – Sketch7

STEP 18 Use ***Extrude*** command for sketch7 to cut the slots.

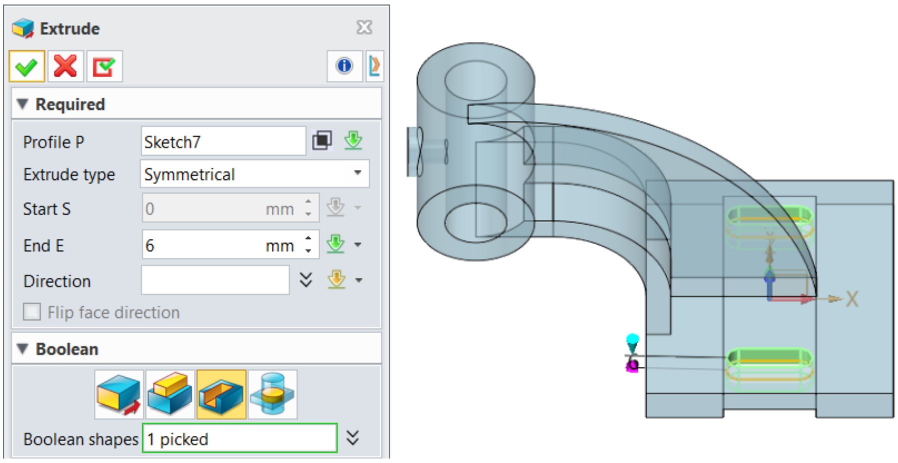


Figure 95 Case1 – Extrude Cut the Sketch7

7) Fillet

STEP 19 Use ***Fillet*** command, pick edges and add the corresponding fillets as below.

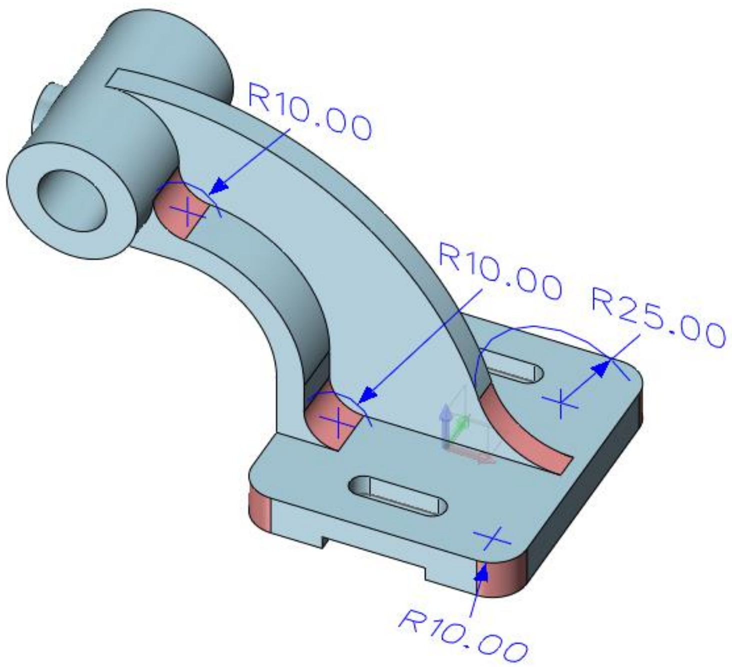


Figure 96 Case1 – Fillets

8.2 Case2

In the second case, more features such as revolve, pattern as well as parametric modeling will be introduced.

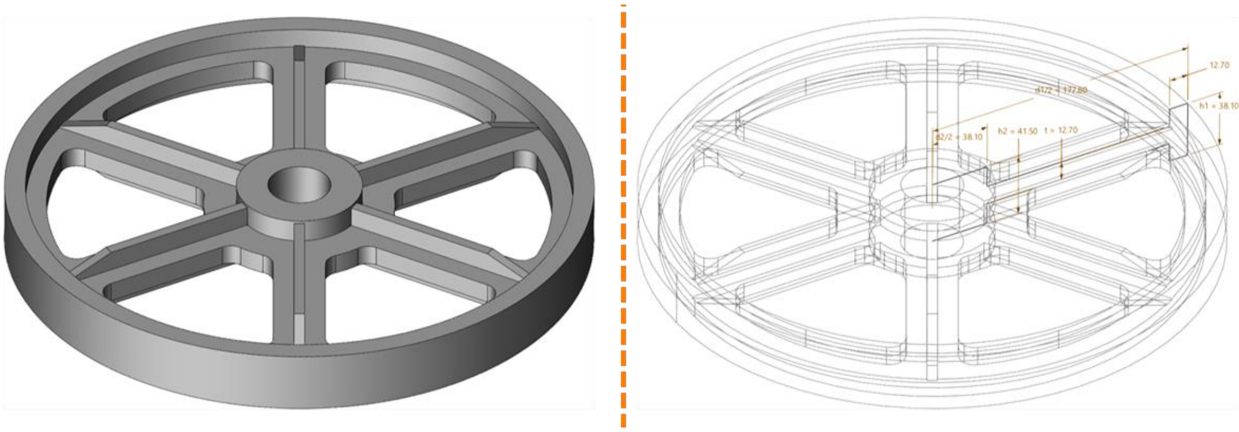


Figure 97 Case2 – Crank Pulley

1) Define the variables

STEP 01 Use ***Equation Manager*** command, define the variables as Figure 98.

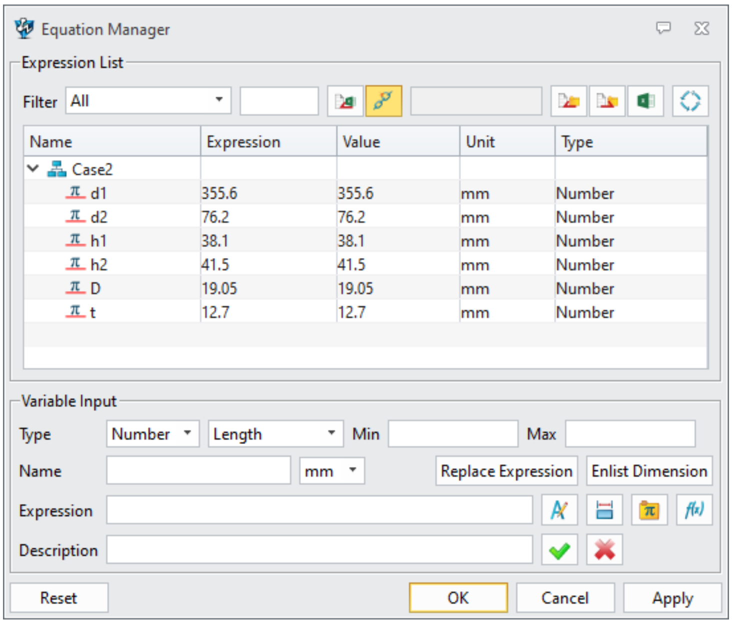


Figure 98 Case2 – Equation Manager

2) Create the base

STEP 02 Create sketch1 on the **XZ** plane as below. Most of the dimensions need to define links to variables.

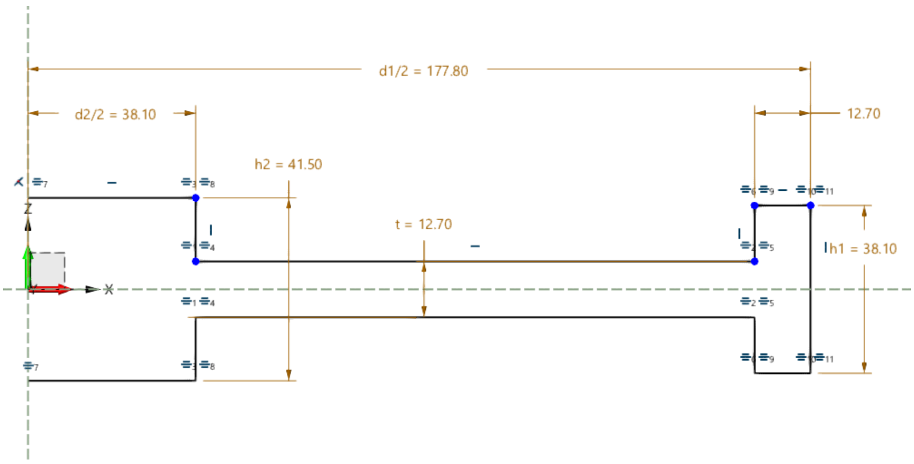


Figure 99 Case2 – Sketch1

STEP 03 Use ***Revolve*** command to revolve the sketch1 as below, the ***Rotation Axis*** is **Z-axis**.

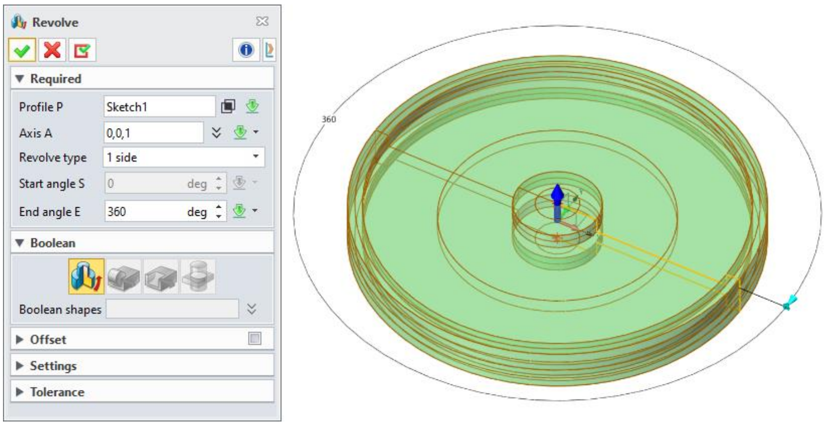


Figure 100 Case2 – Revolve

3) Create the ribs

STEP 04 Create sketch2 on the **XZ** plane as below.

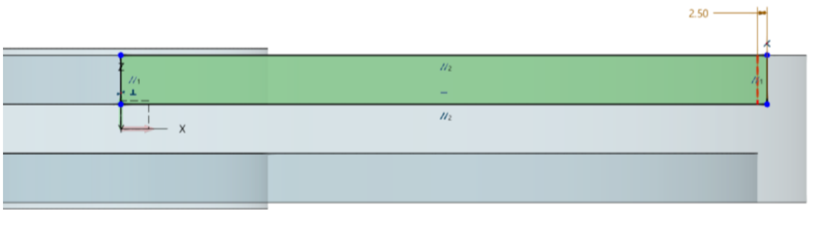


Figure 101 Case2 – Sketch2

STEP 05 Extrude sketch2 with **Symmetrical** type, the single ***Side*** value is **3.2 mm**.

STEP 06 Add a ***Draft*** feature on the outer side as below.

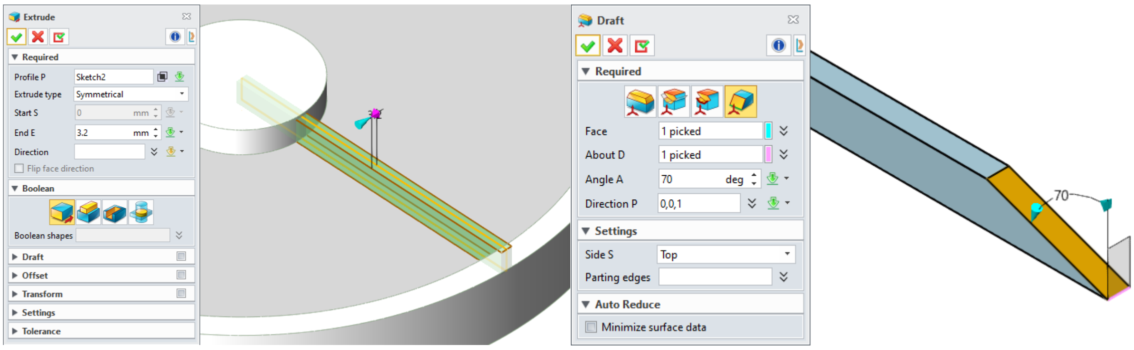


Figure 102 Case2 – Create the Rib

STEP 07 Use ***Mirror Geometry*** command to mirror the rib to the downside by picking the **XY** plane.

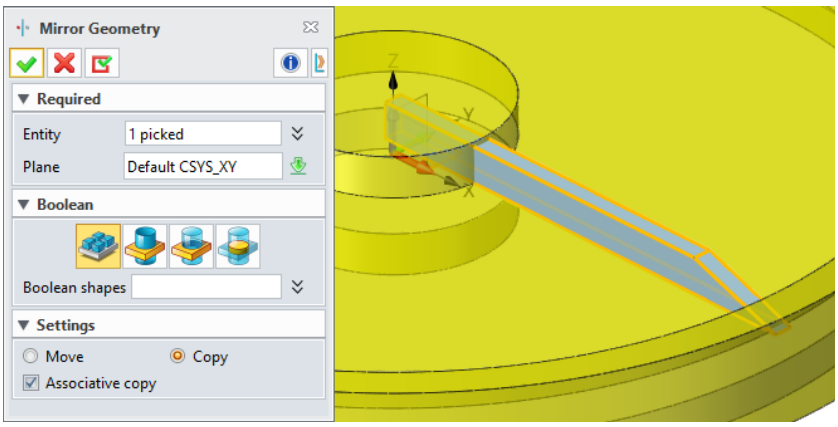


Figure 103 Case2 – Mirror Geometry

STEP 08 Use ***Pattern Geometry*** command, pick the ***Circular Pattern*** option, pick the two ribs as the ***Base*** then input the parameters as below.

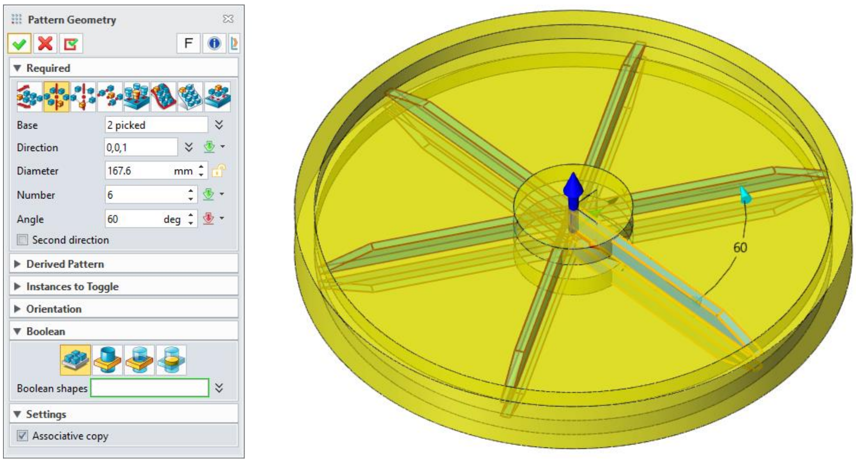


Figure 104 Case2 – Pattern Geometry

STEP 09 Use ***Add Shape*** command to combine the base and the ribs.

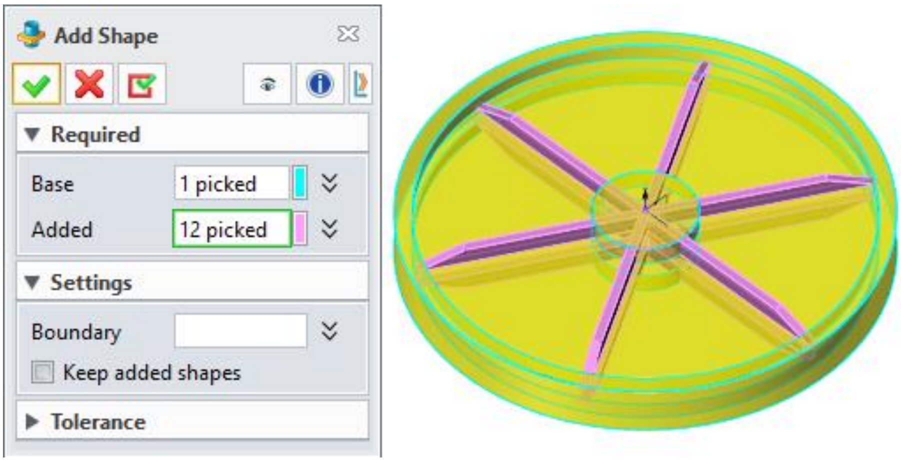


Figure 105 Case2 – Combine the Base and the Ribs

4) Create the fan-shape holes

STEP 10 Draw sketch3 on the top face of the middle cylinder. Refer to a sector area (the red hidden lines) then offset to generate a smaller fan-shaped profile.

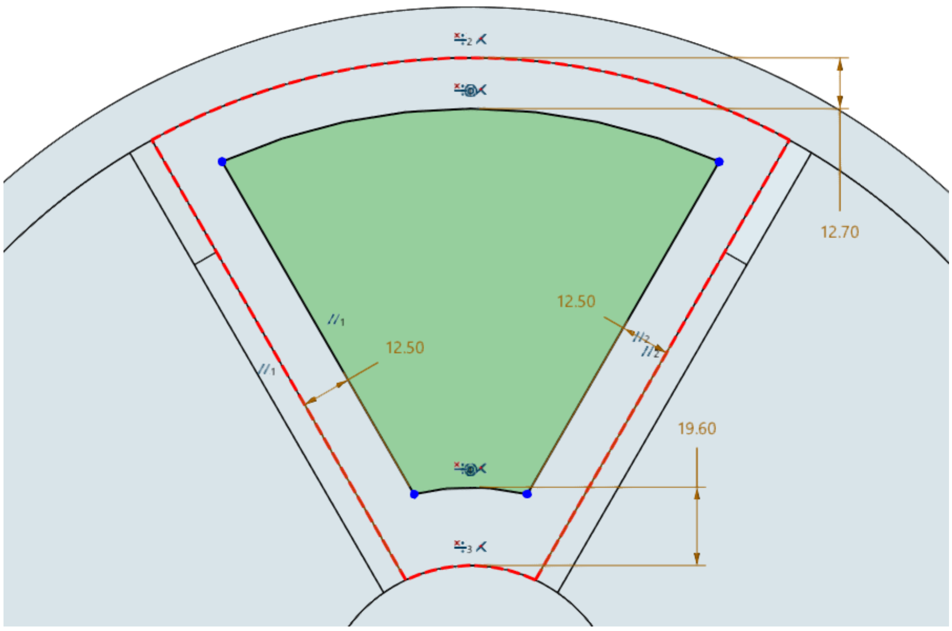


Figure 106 Case2 – Sketch3

STEP 11 Extrude to cut the base, set the ***End*** value as the variable **h2**. This ensures that it is always a penetration hole.

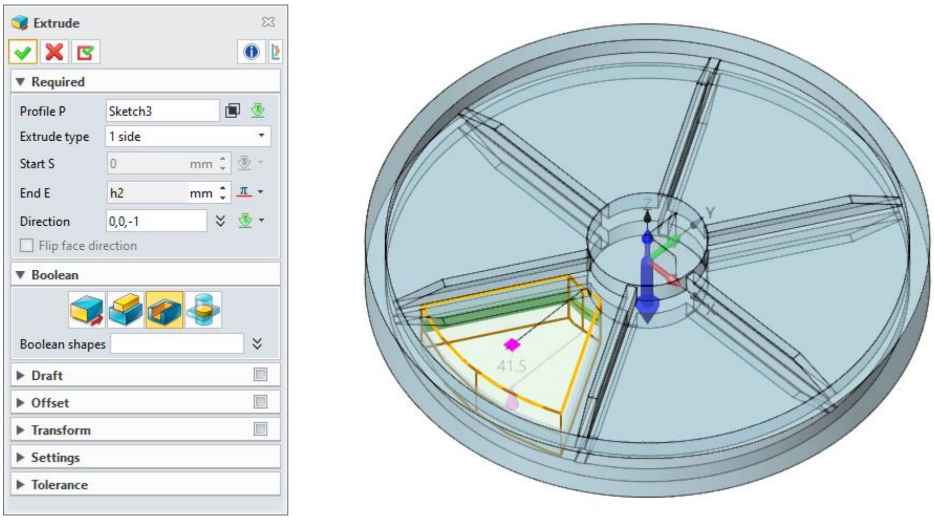


Figure 107 Case2 – Extrude Cut the Fan-shapes Hole

STEP 12 Add fillets as below.

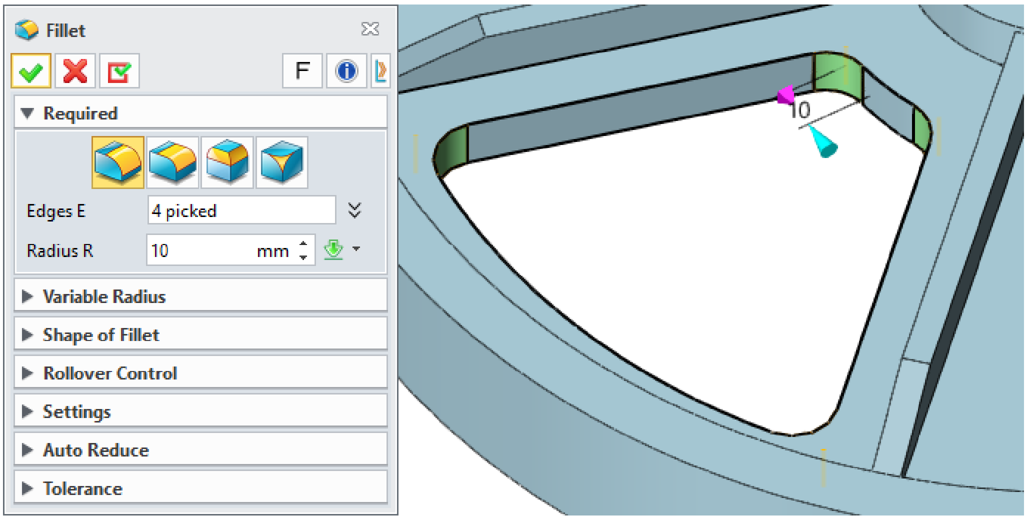


Figure 108 Case2 – Fillets

STEP 13 Use ***Pattern Feature*** command to pattern the fan-shape hole, the features can be picked in the Modeling History Tree, or the corresponding surfaces in the modeling space.

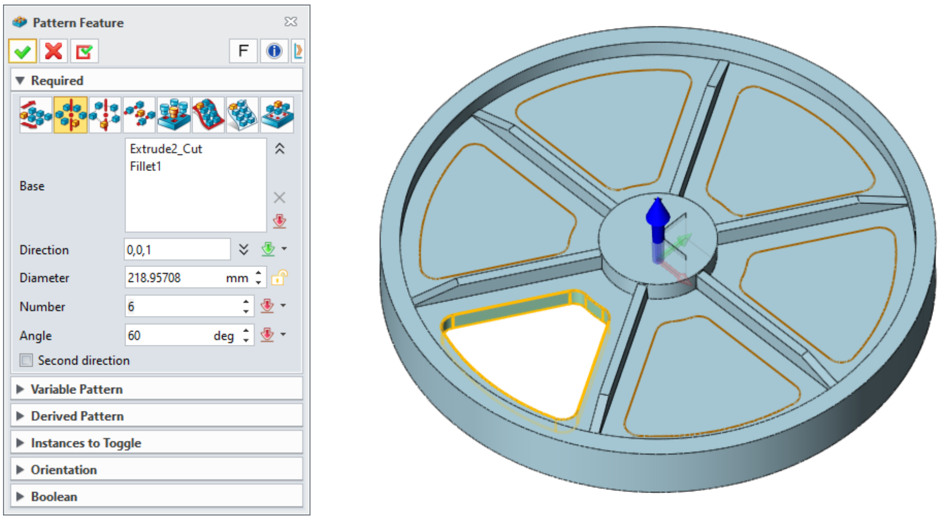


Figure 109 Case2 – Pattern Feature

5) Create the hole

STEP 14 Use ***Hole*** command to create the center hole, set the diameter as the Variable **D**.

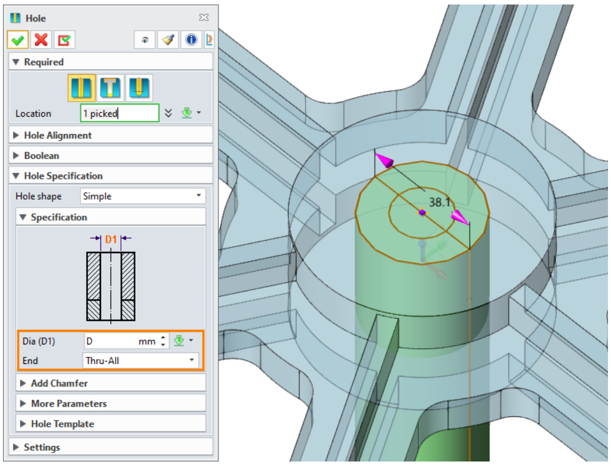


Figure 110 Case2 – Create the Hole

STEP 15 At this point, the modeling process is over.

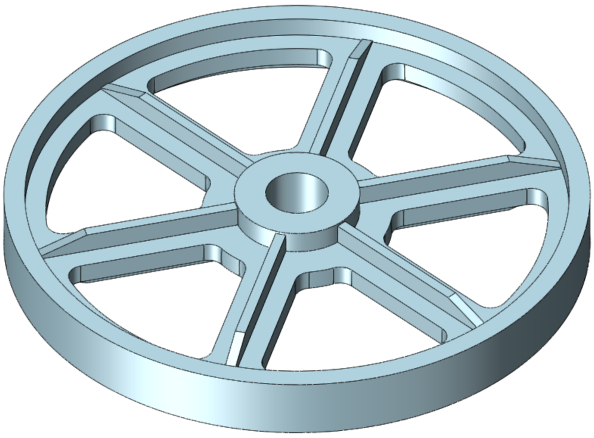


Figure 111 Case2 – Finished

6) Modify the variables

STEP 16 Normally, users can modify the variables by double-click on the variables directly, or in the ***Equation Manager***. After the modification, regen the history, the model will update automatically.

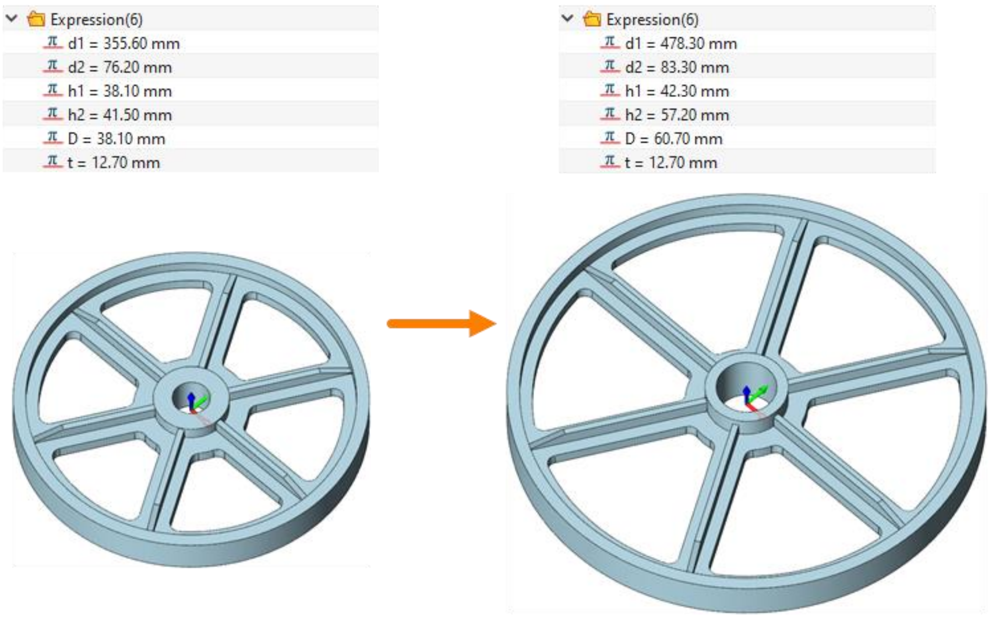


Figure 112 Case2 – Modify the Variables