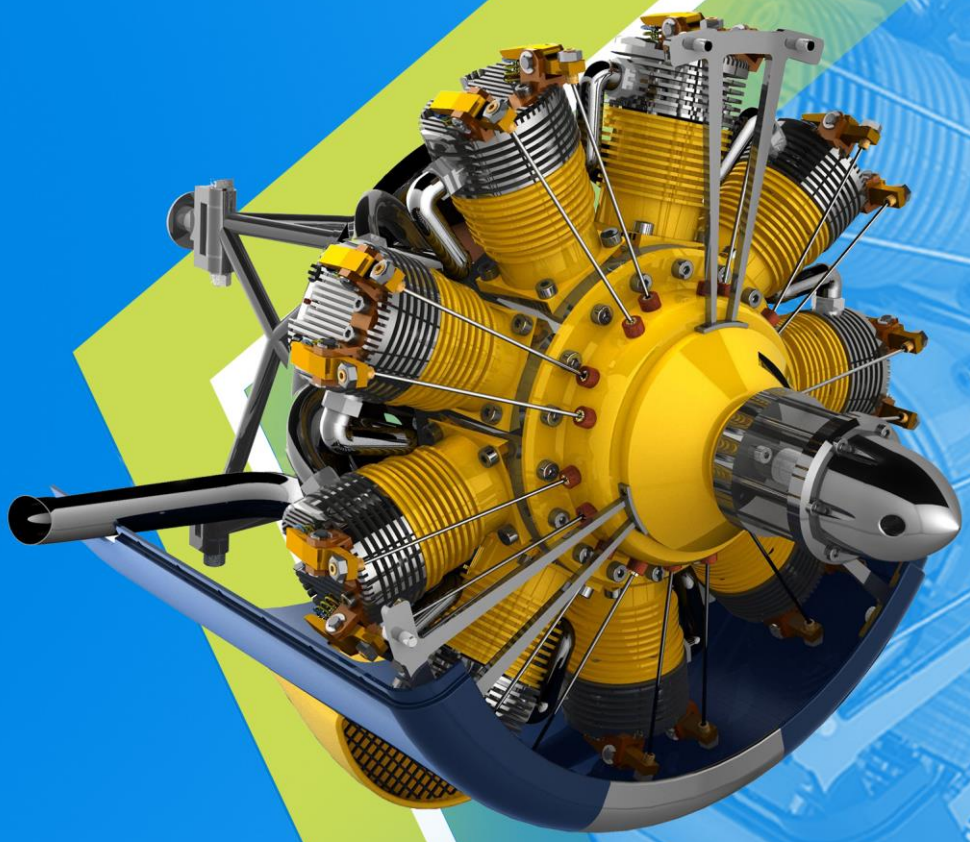
ZW3D from Entry to Master Tutorial

2D Sketch



ZW3D™ V2023 From Entry to Master CAD 2D Sketch

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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 2D Sketch***, a master tutorial.

Thanks for being our user!

The ZW3D Team

Contents

1 An Introduction to 2D Sketch

Since most models begin with the sketch, a 2D sketch is the basis for modeling. In ZW3D, the 2D sketch module includes drawing tools, editing features, constraints & dimensions tools, and checkers. Also, sketches are done in an independent environment.

Key Points:

- ✧ Draw common 2D geometries
- ✧ Edit 2D geometries
- ✧ Add well-defined constraints to the 2D sketch
- ✧ Check the 2D sketch

1.1 New 2D Sketches

In the modeling level, there are several ways to create a new sketch.

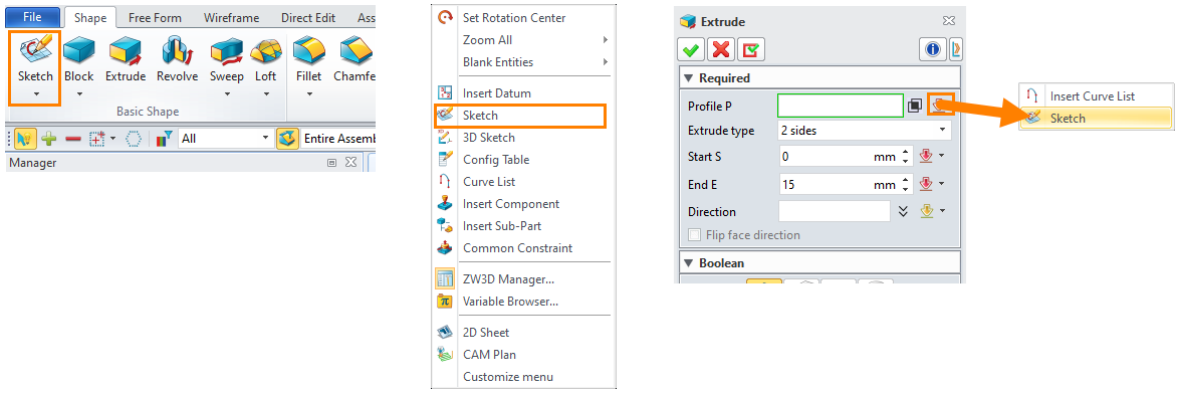


Figure 1 Insert a New 2D Sketch

Shape Ribbon Tab/Context menu->Sketch

You can also start sketching with some commands, for example, ***Extrude->Profile input field->Context menu->Sketch***

2D sketches could be datum planes, such as XY, XZ, and YZ planes.

In ZW3D, the default 2D sketch is a XY datum plane.

1.2 Pre-setting 2D Sketches

1) Turn on/off Grid Point/Line and set the Grid Spacing

You can quickly access the grid-setting options in the Document Aware Toolbar, as the below image shows.

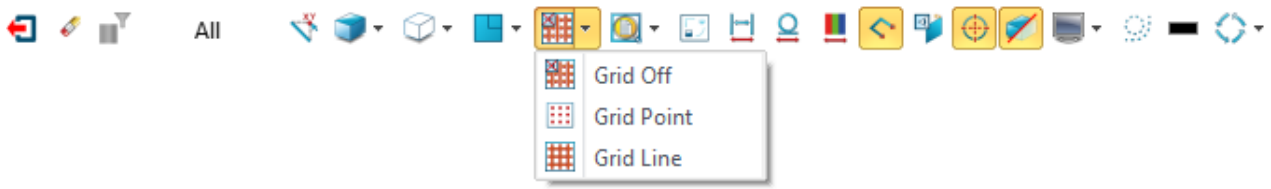


Figure 2 Turn on/off Grid

Sketch Tab->Setting->Preferences

You can set the Grid Spacing of the 2D Sketch, as shown in Figure 3.

2) Orient to sketch plane

If you want to orient a sketch plane to another point, there is a ***Plane View*** button in the ***Quick Access Tab***, as shown in Figure 4.

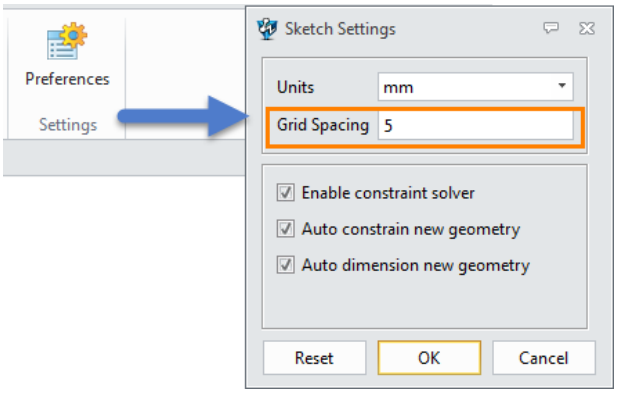


Figure 3 Set the Grid Spacing

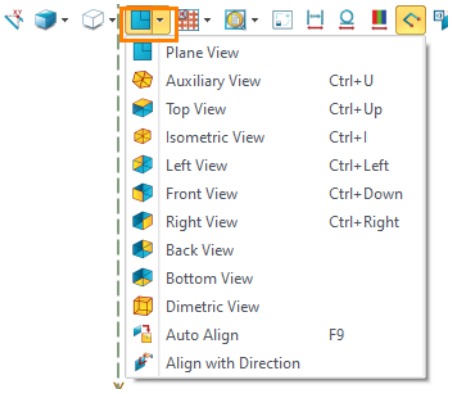


Figure 4 Change the View Angle

3) Show target/Show all

If you want to view elements that are not included in the sketch, you can click this ***Show Target*** button and all the desired objects will appear with transparency effects. Click it again, only the current sketch will be shown.

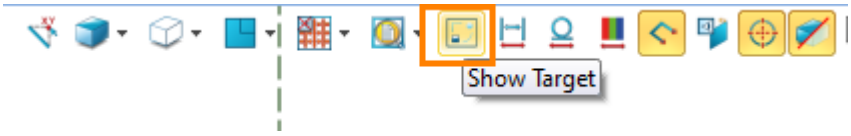


Figure 5 Show Target

4) Turn on/off dimensions and constraints

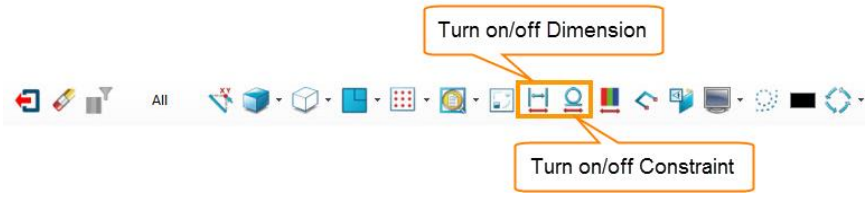


Figure 6 Turn on/off Dimensions & Constraints

5) Turn on/off constraint status color

From the ***Configuration*** dialogue box or ***Document Aware Toolbar***, you can set the constraint status color.

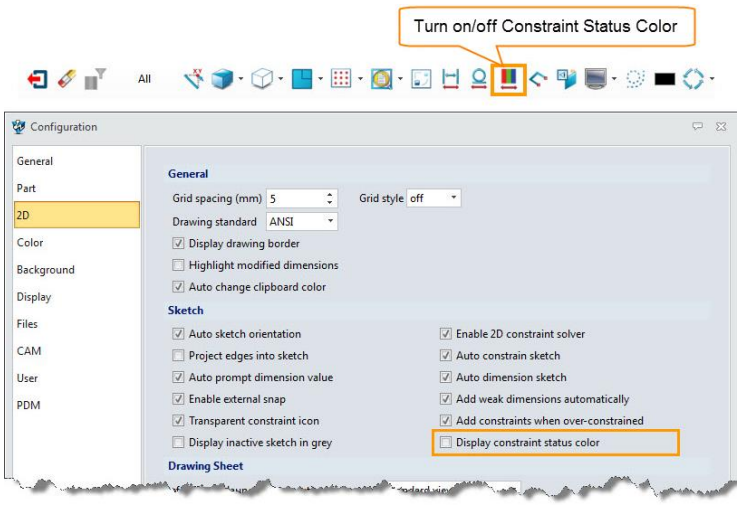


Figure 7 Turn on/off Constraint Status Color

6) Set the snap filter

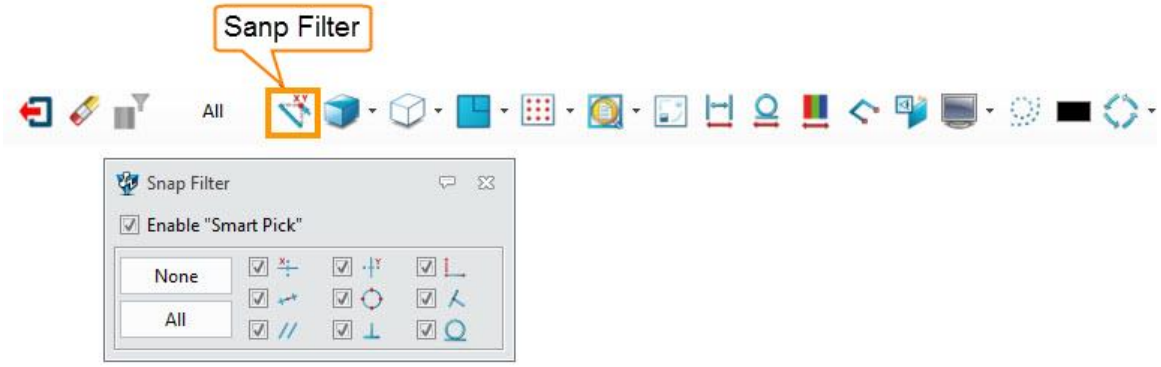


Figure 8 Set the Snap Filter

7) Turn on/off open ends

You can turn on Open Ends to help check the connectivity of your sketch. When Open Ends are turned on, there will be a small square at each open end of the line indicating that the line is not closed.

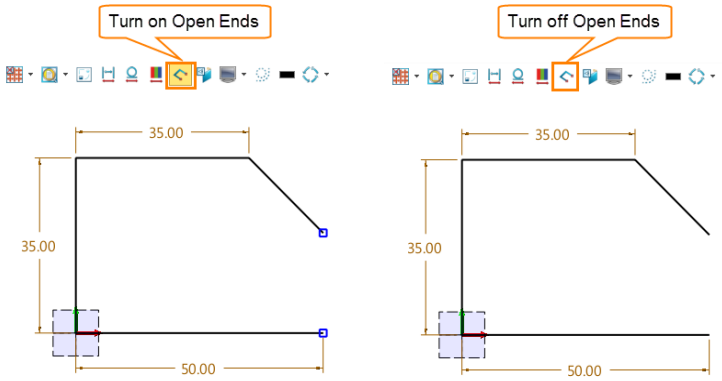


Figure 9 Turn on/off Open Ends

1.3 Exit 2D Sketches

The ***Exiting the Sketch*** icon is available in two places, as the image below shows.

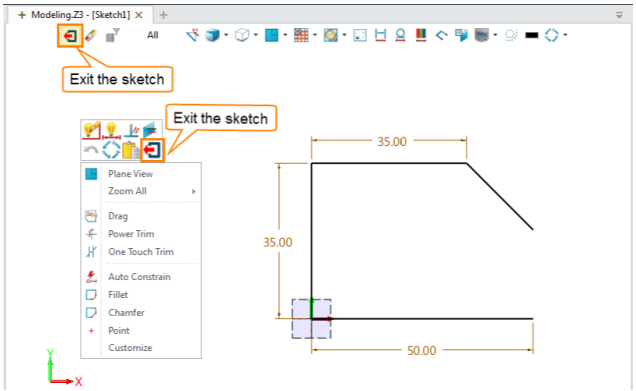


Figure 10 Exit 2D Sketch

1.4 Relocate 2D Sketches

After modifying your design, you might need to change the datum plane of the sketch or change the drawing position of the current sketch. In this case, the Relocate function can be useful.

Method 1: *Sketch ribbon Tab->Settings->Relocate*

With the Relocate dialogue box invoked, you can reassign a new datum plane for the current sketch.

Method 2:

In the history manager, you can right-click on the specific sketch, then click ***Relocate*** to invoke the Relocate dialogue box. After you exit the sketch level, you can also use this method to adjust the sketch plane in the modeling environment.

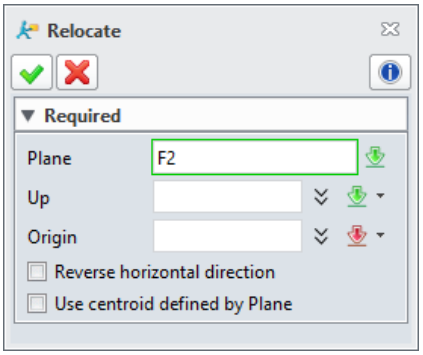


Figure 11 Relocate Dialog Box

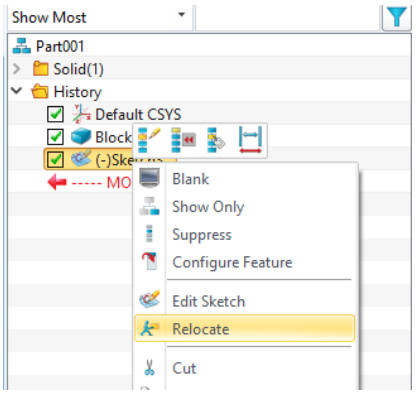


Figure 12 Relocate in History Manager

2 Drawing Common Geometries

2.1 Points

Sketch Ribbon Tab->Drawing->Points

Method 1: Click to create a point.

Method 2: Input point coordinates to create a point.

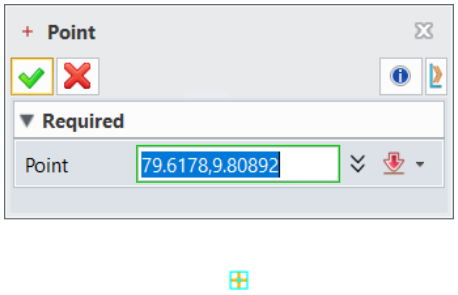


Figure 13 Create a Point with a Click

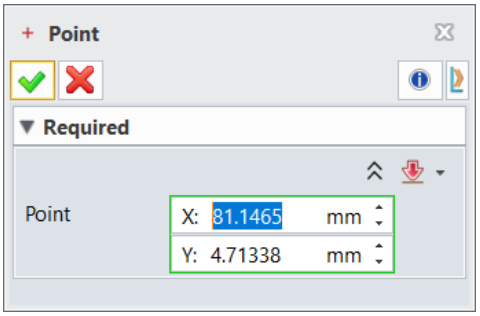


Figure 14 Create a Point with Point Coordinates

Before creating the points, you can first set the point attributes following ***Tool Ribbon Tab->Attributes->Point Attributes***.

Right-click on the points, then select ***Attributes*** to invoke the Point Attributes dialogue box and modify.

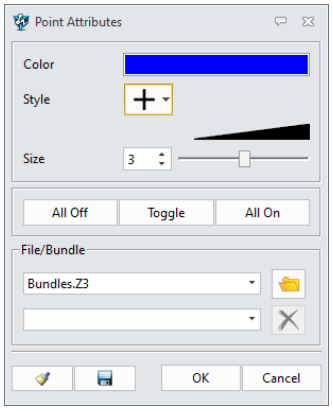


Figure 15 Set or Modify Point Attributes

Sketch Ribbon Tab->Drawing->Points on Curve

This command can help you create a variety of points on a curve in batches.

Type 1: Equally Spaced N Points

Select a curve and set the number of points, then a specified number of equally spaced points on a curve are created.

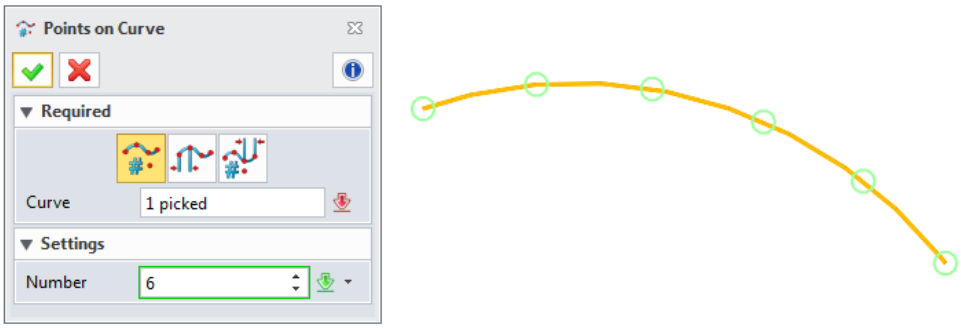


Figure 16 N Points Equally Spaced Method

Type 2: Points at Distance

Select a curve and set the value of distance, then an unspecified number of equally spaced points on a curve are created.

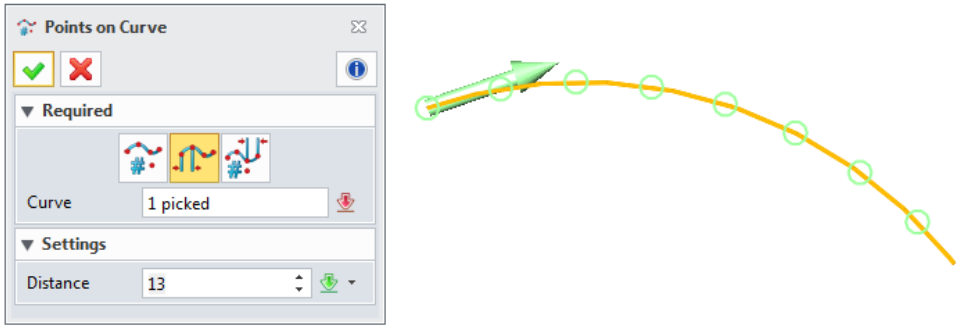
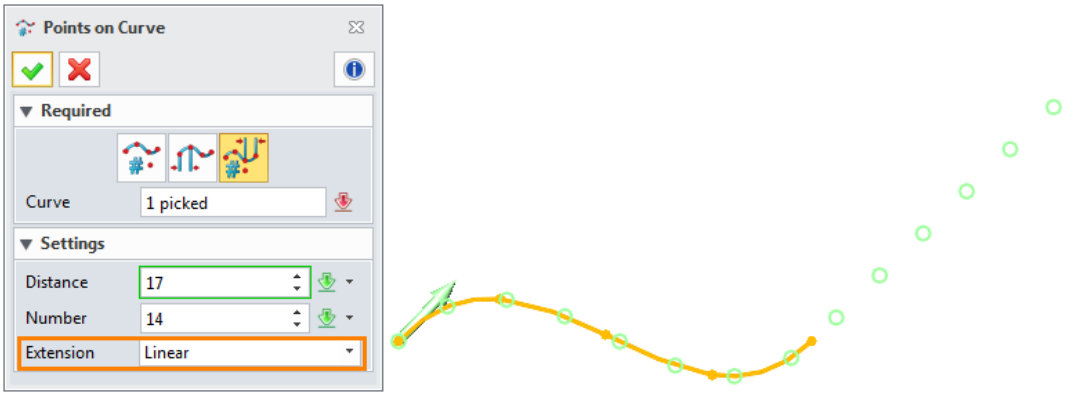


Figure 17 Points at Distance Method

Type 3: N Points at Distance

Points can be created on the selected curve according to the values of distance and the number of points.

If the arrangement of points exceeds one end of the curve, the points will be created according to the type of extension (Linear/Circular/Reflect).



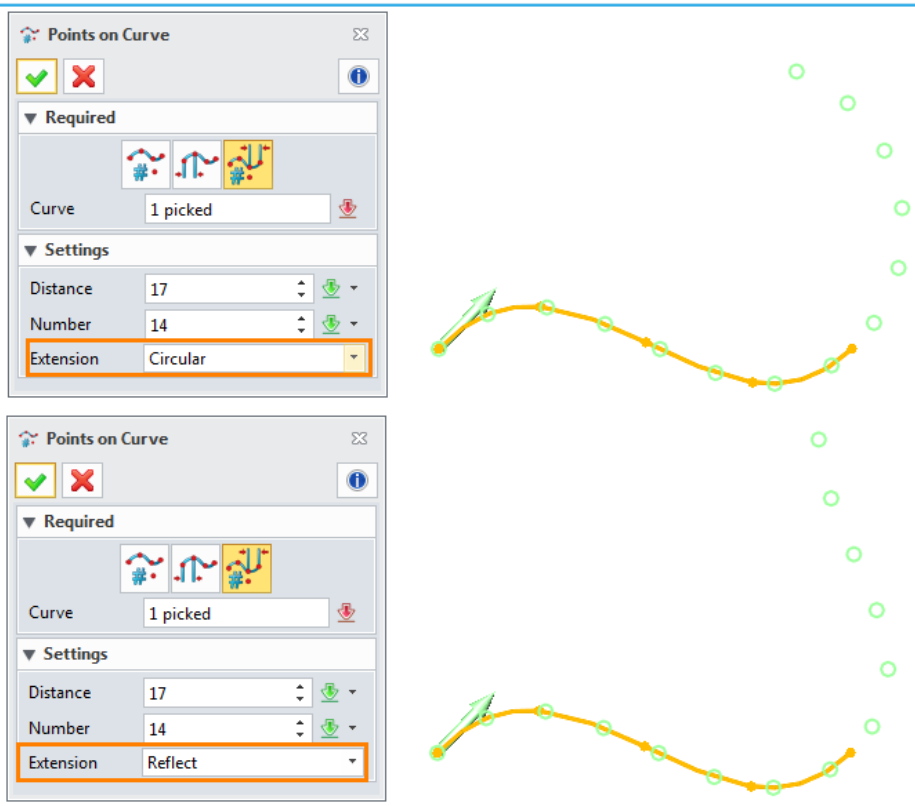


Figure 18 N Points at Distance Method

2.2 Lines and Construction Lines

2.2.1 Line

Sketch Ribbon Tab->Drawing->Line

This command can help you draw the line with two points or other reference objects.

We will focus on the “2 points” method and how to set parameters.

“Locking” the Length?

When the length is locked, you cannot drag to lengthen the line with the cursor, as shown in Figure 19.

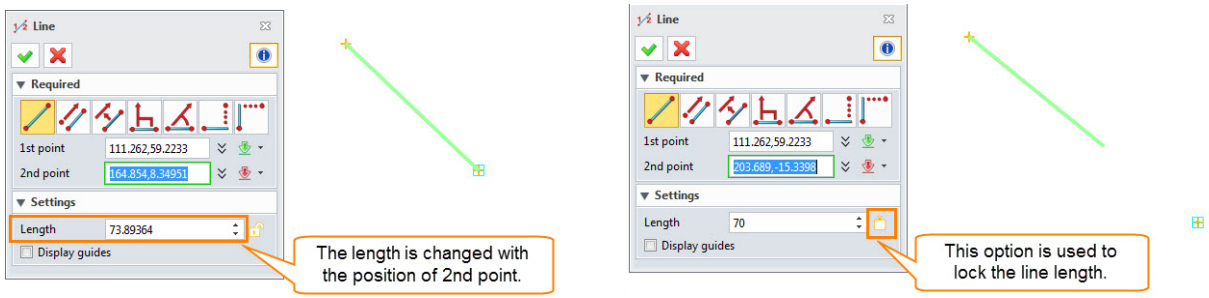


Figure 19 Create a Line

The “Display guides” option?

When this option is checked, two guidelines along the X and Y directions will appear, as shown in Figure 20.

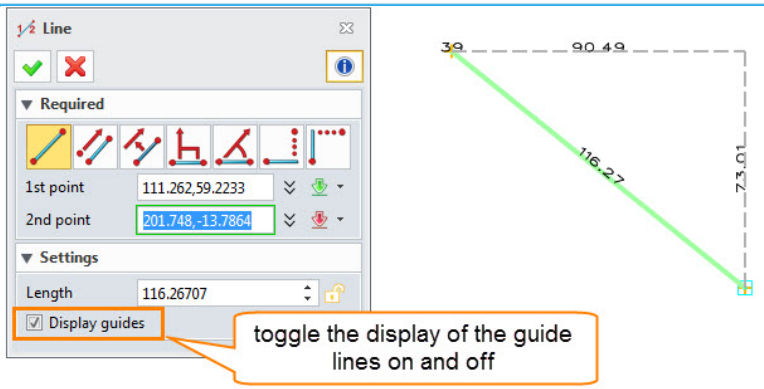


Figure 20 Create a Line with Guidelines

Create a construction line

STEP 01 Create a line.

STEP 02 Right-click on the line and click the Toggle Type option to turn it into a construction line.

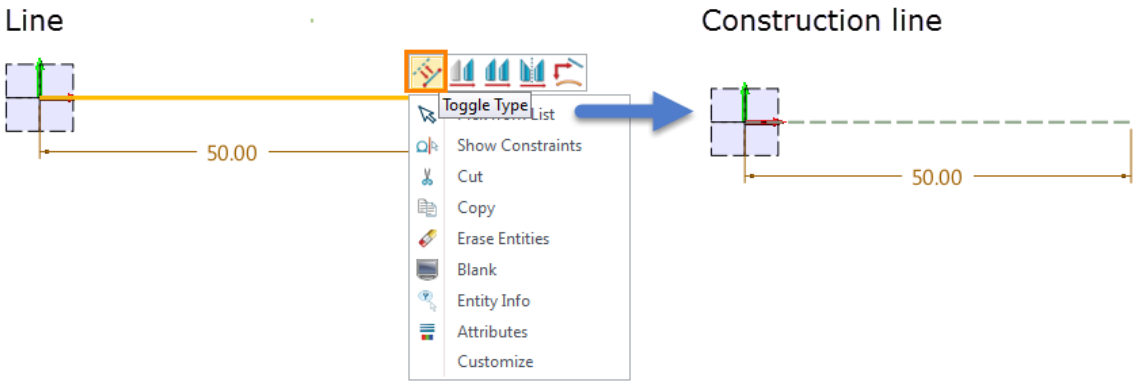


Figure 21 Toggle Line Type

Note: Click Toggle Type again and the construction line will be converted into a solid one.

2.2.2 Polyline

Sketch Ribbon Tab->Drawing->Polyline

You can create a polyline by selecting one point at a time.

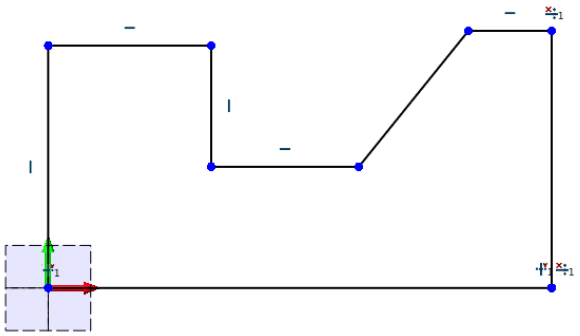


Figure 22 Polyline

2.2.3 Double line

Sketch Ribbon Tab->Drawing->Double line

This command can help you create a double polyline.

STEP 01 Set the left and right widths.

STEP 02 Pick points to draw a double line.

STEP 03 If you want to add arcs to the line, please check the ***Insert arc at the corners*** option. The radius of the arc is determined by the Left or Right width as shown in Figure 23.

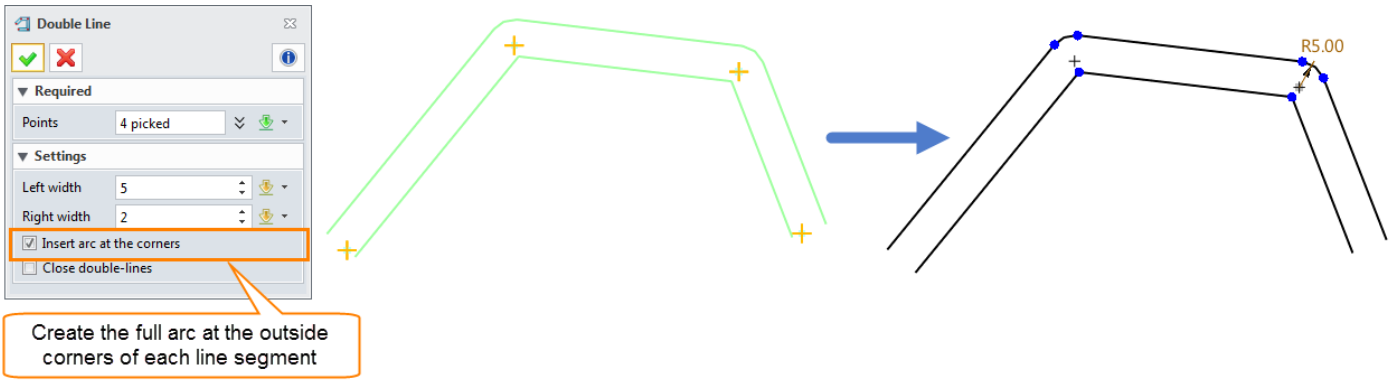


Figure 23 Double Lines with Arcs

Note: If you want to create a closed double line, please check the ***Close double-lines*** option as Figure 24 shows.

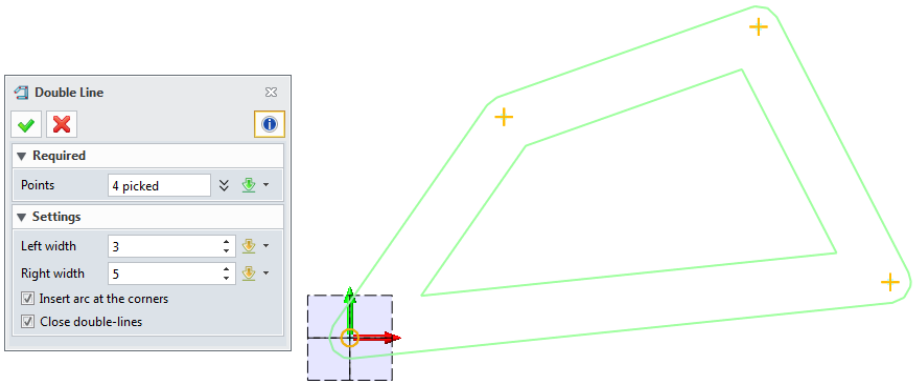


Figure 24 Closed Double Lines

2.3 Circles, Arcs and Ellipses

2.3.1 Circle

Sketch Ribbon Tab->Drawing->Circle

You can create a circle in different ways.

1) Boundary

You can draw a circle by specifying its center and a boundary point on another geometry.

Figure 25 Boundary

2) Radius

This is the default method to create a circle. You can draw a circle by specifying its center and the value of the radius/diameter.

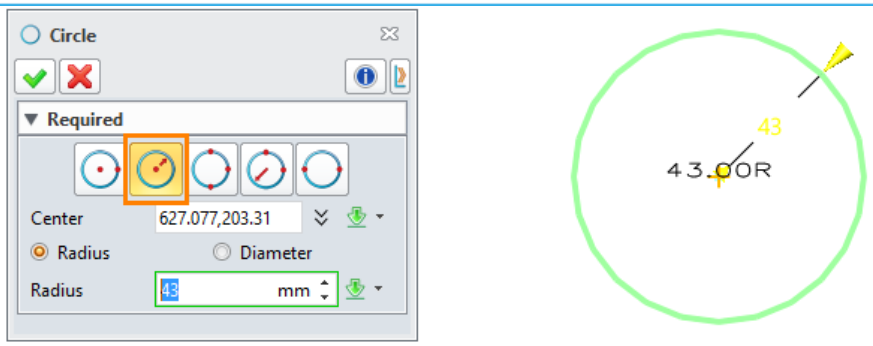


Figure 26 Radius

3) 3 points

You can draw a circle by specifying three points on the circumference. This method is useful when you want to make the circle tangent to other geometries.

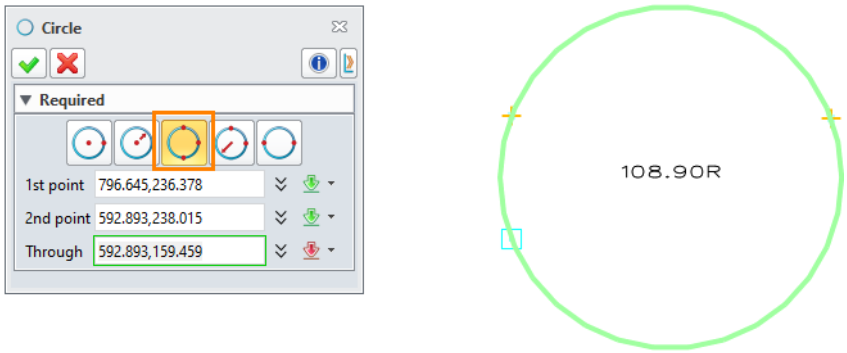


Figure 27 Three Points

4) 2 points with radius

You can draw a circle by specifying two points and the radius.

Figure 28 2 Points with Radius

5) 2 points

You can draw a circle by specifying the two endpoints of a diameter.

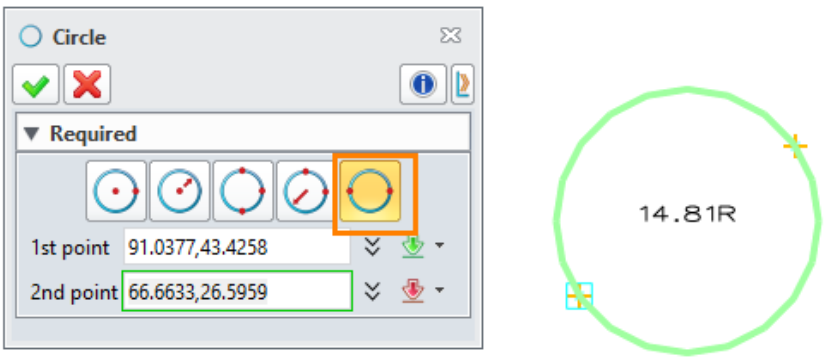


Figure 29 Two points

The “Location” option

When two boundary points are picked and the radius is specified, there could be two possible circles. So in Location input box, you can select the needed one, see Figure 30.

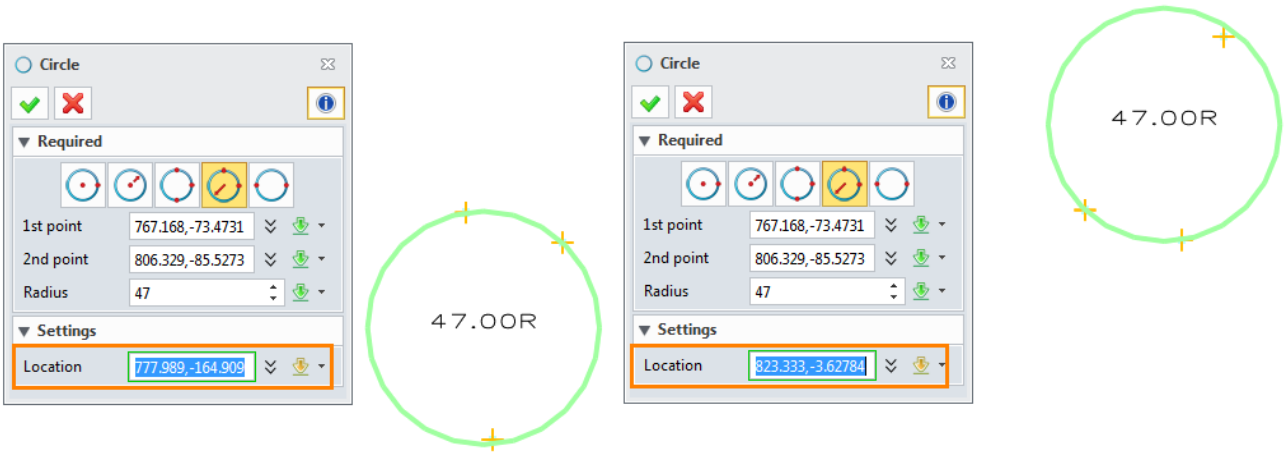


Figure 30 2Points-Radius method

2.3.2 Arc

Sketch Ribbon Tab->Drawing->Arc

STEP 01 Pick two points and determine the center.

STEP 02 If several different results are available, you could select one by checking the parameter (Clockwise/Counter) or Location parameter.

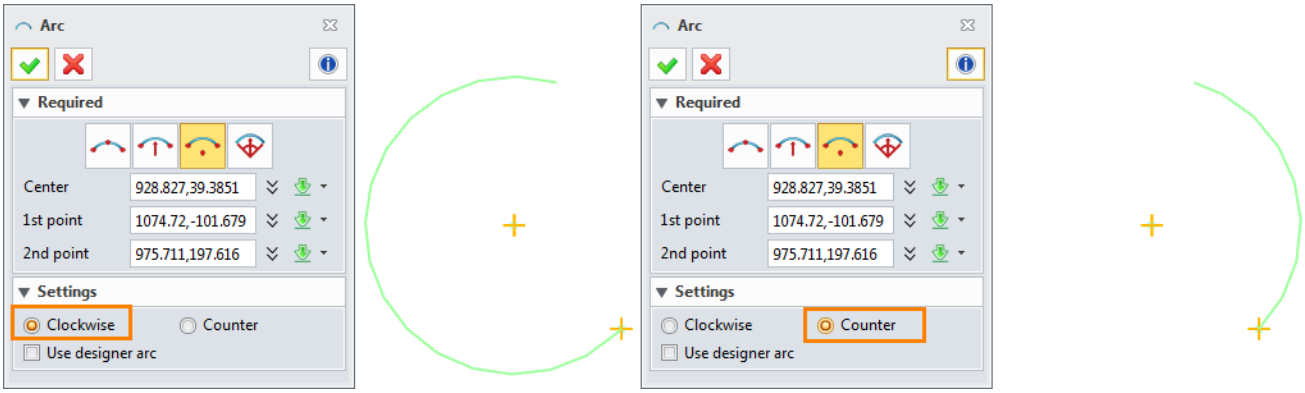


Figure 31 Arc

What does “Designer arc” mean?

Check this box to use a designer arc instead of a regular one. A designer arc is a NURB curve that matches the tangency of the arc but has zero curvature at the endpoints.

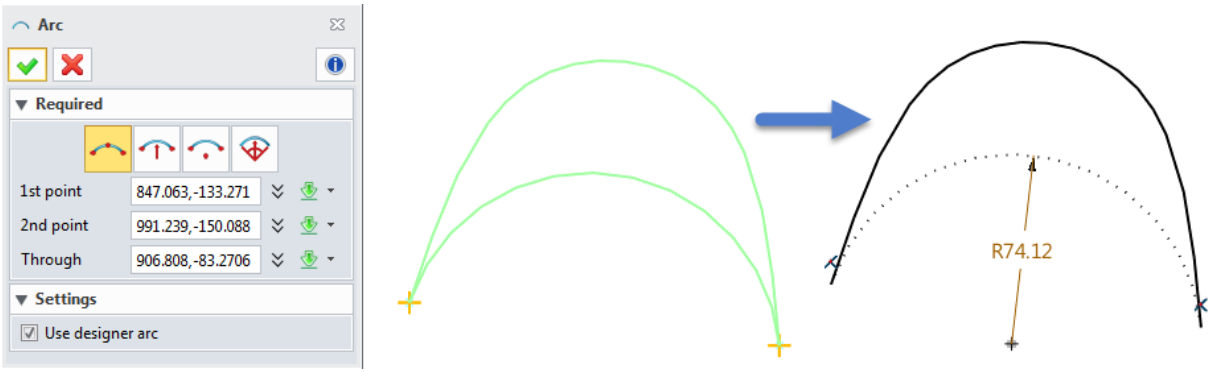


Figure 32 Designer arc

2.3.3 Ellipse

Sketch Ribbon Tab->Drawing->Ellipse

There are four ways for you to flexibly create an ellipse with points.

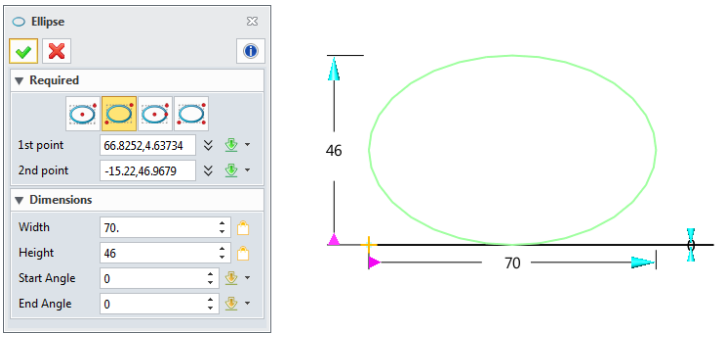


Figure 33 Creating an Ellipse

1) Center

You can draw an ellipse by specifying its center and one corner of the circumscribed rectangle.

2) Corner

You can draw an ellipse by specifying the diagonal points of a rectangle.

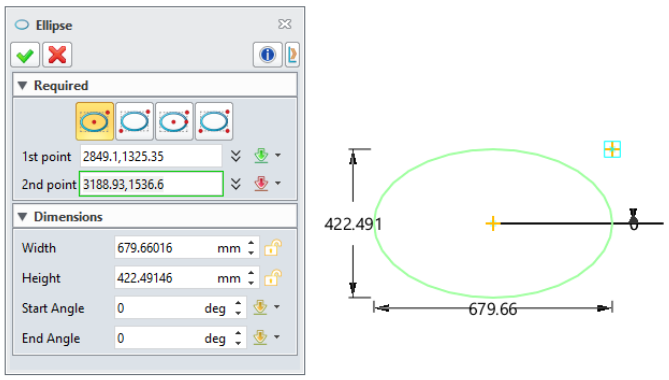


Figure 34 Center

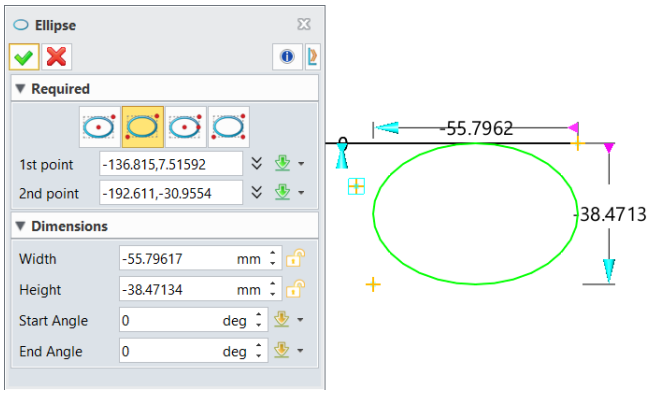


Figure 35 Corner

3) Center with a certain angle

Based on the ***Center*** method, it requires an extra parameter, the angle of the ellipse.

4) Corner with a certain angle

Based on the ***Corner*** method, it requires an extra parameter, the angle of the ellipse.

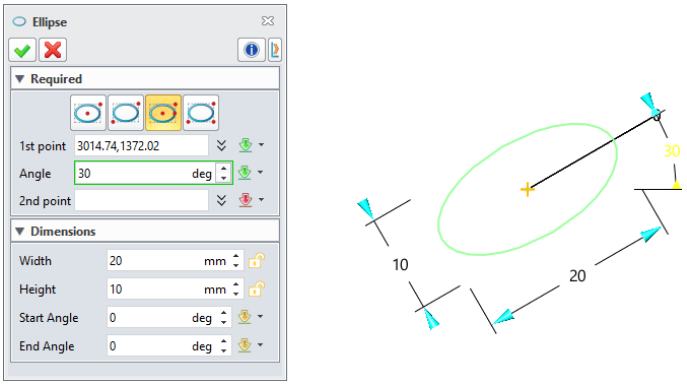


Figure 36 Center with an Angle

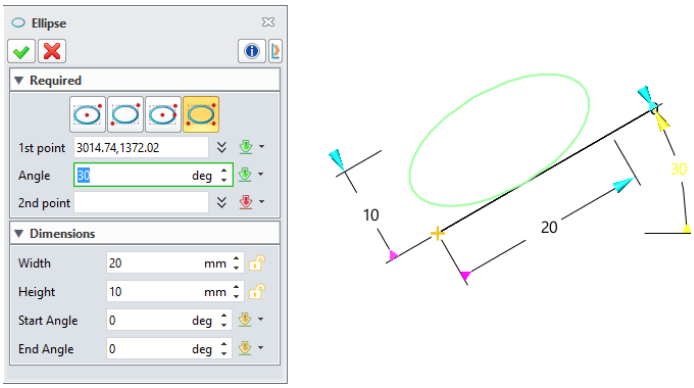


Figure 37 Corner with an Angle

2.4 Rectangles and Polygons

2.4.1 Rectangle

Sketch Ribbon Tab->Drawing->Rectangle

There are four ways to create a rectangle, which are similar to creating an ellipse (Please refer to Chapter 2.3.3). Among them, The ***Center-Corner*** and ***Corner-Corner*** methods are frequently used.

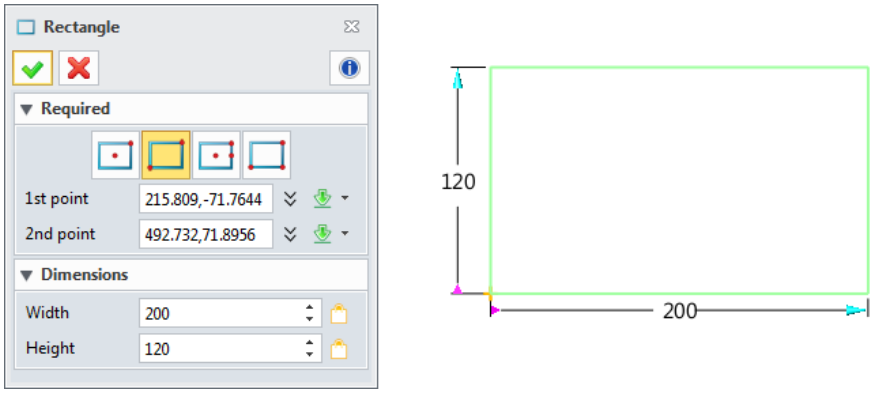


Figure 38 Rectangle

2.4.2 Polygon

Sketch Ribbon Tab->Drawing->Polygon

STEP 01 Select a method to create a polygon.

STEP 02 Set the geometric parameters, such as the center, radius and number of sides.

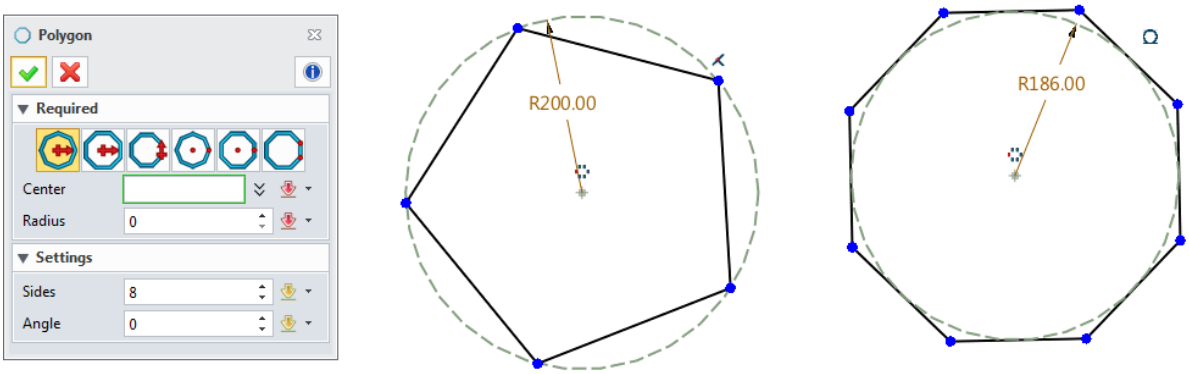


Figure 39 Polygon

When creating sketch geometries, the polygon constraint is automatically added. Nevertheless, the dimension constraints should be added manually.

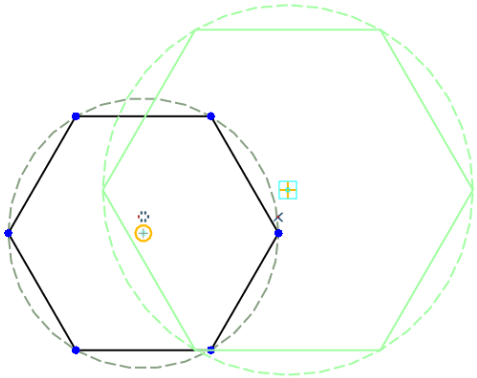


Figure 40 Drag the Polygon

1) Inscribed radius & Circumscribed radius

These two methods allow you to create a circle that is tangent to the polygon by specifying its center and radius. Different results are as the images below show.

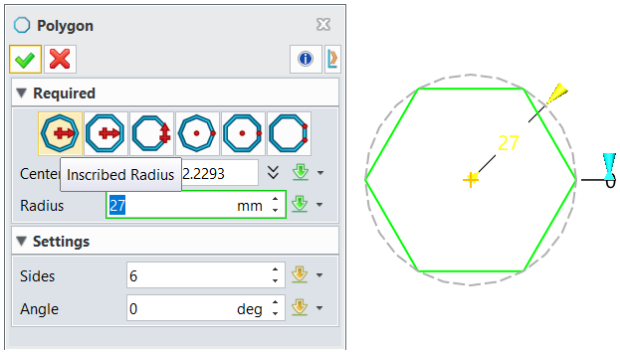


Figure 41 Inscribed Radius

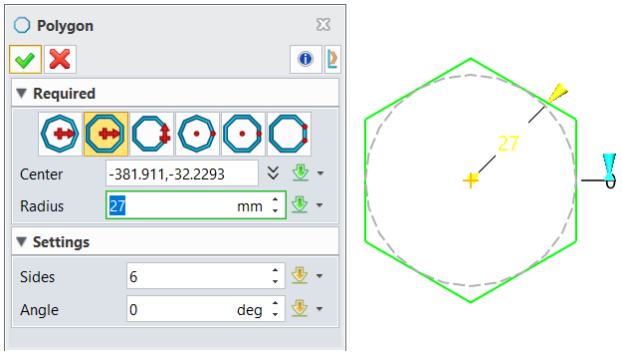


Figure 42 Circumscribed Radius

2) Inscribed boundary & Circumscribed boundary

These two methods allow you to create a circle that is tangent to the polygon by specifying its center and boundary point. Different results are as the images below show.

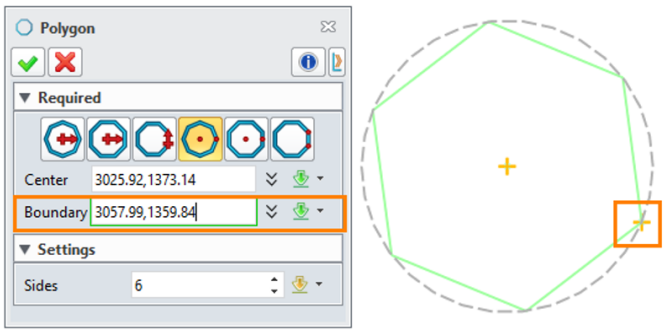


Figure 43 Inscribed Boundary

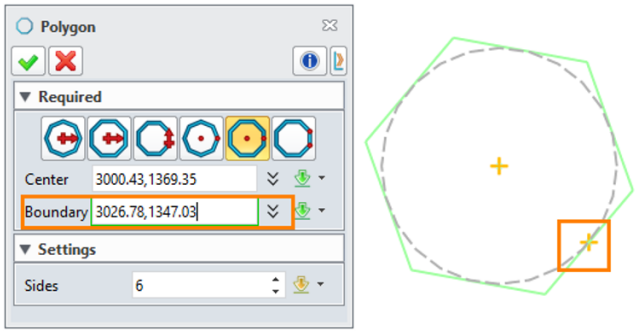


Figure 44 Circumscribed Boundary

3) Side length

You can draw a polygon by specifying a corner point and then inputting the side length.

4) Side boundary

You can draw a polygon by specifying a corner point and then a boundary point.

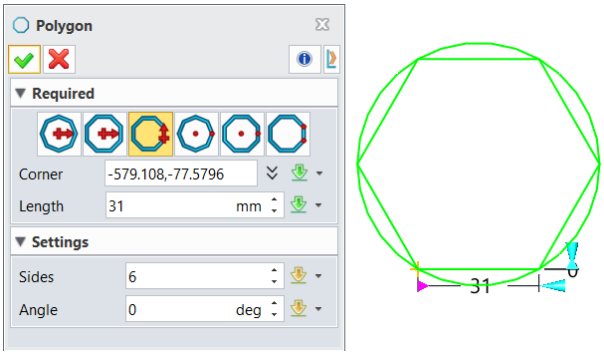


Figure 45 Side Length

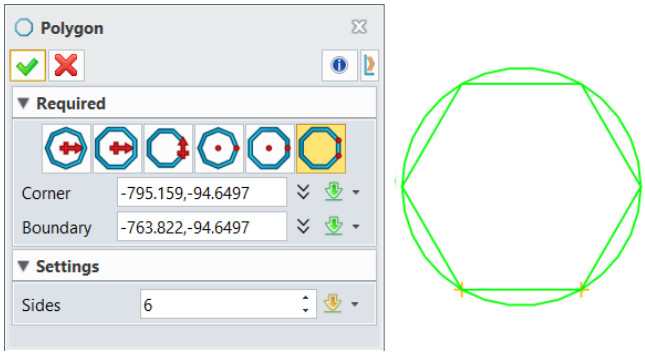


Figure 46 Side Boundary

After creating the polygon, you can drag its center to redefine its location and size.

2.5 Slots & Notches

Sketch Ribbon Tab->Drawing->Slot

This command can help you create a slot by specifying the centers of two circles and the radius.

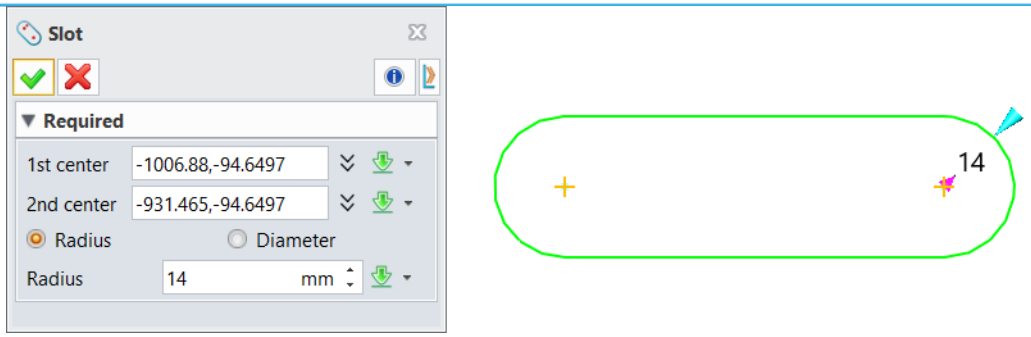


Figure 47 Slot

Sketch Ribbon Tab->Drawing->Notch

This command can help you create a notch on the curve.

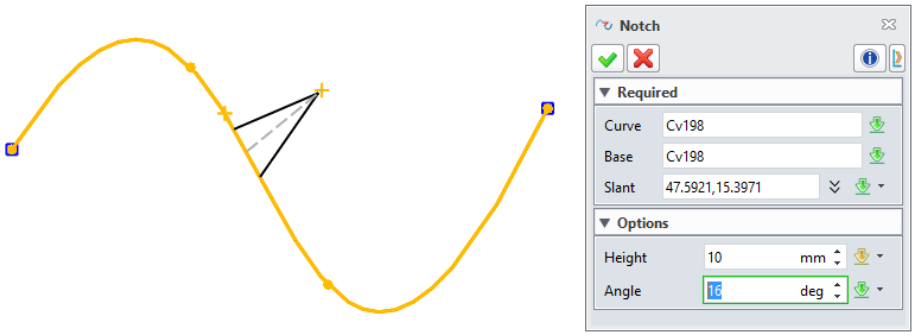


Figure 48 Notch

2.6 Splines

Sketch Ribbon Tab->Curve->Point curves

This command can help you create a spline by defining the points that it must pass through.

STEP 01 Click to pick the points.

STEP 02 Define the constraints for each point. Unfold the ***Points List***, select a point and then set the constraints according to the requirements.

As for the spline in Figure 49, the Continuity Type of its starting point and endpoint is set to be G1 (Tangent).

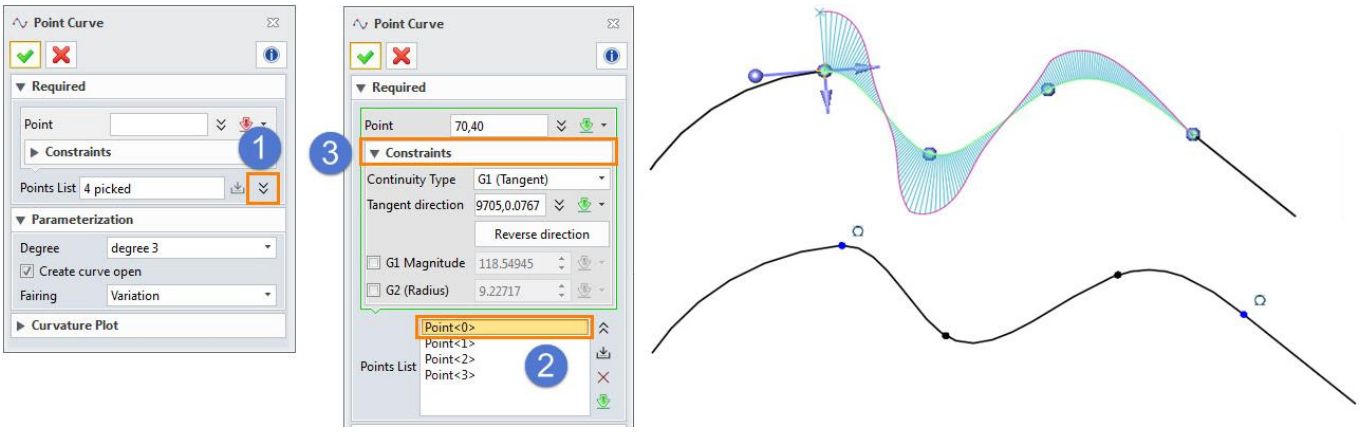


Figure 49 Creating a Spline

Note: If you want to create a closed curve, simply uncheck the ***Create curve open*** option under ***Parameterization***.

Sketch Ribbon Tab->Curve->Control Point

You can use control points to create a smooth curve.

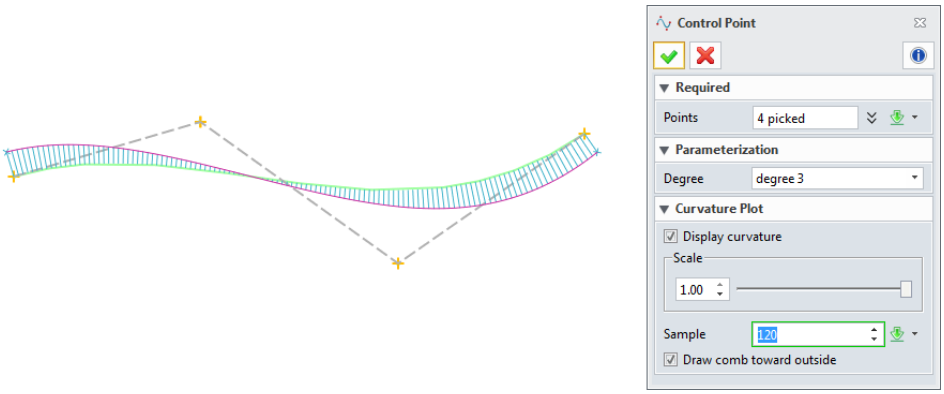


Figure 50 Creating a Curve with Control Points

Note: *As for the **Degree** option,* the degree of the curvature needs to be lower than the number of control points

For example, if you want to create a 6-degree curve, you need to specify at least 7 points. The higher the degree you choose, the more calculation needs to be done by the program. So far, the highest degree ZW3D can support is 6, and we recommend that you create curves of 3 to 5 degrees.

Sketch Ribbon Tab->Curve->3 Point Conic

You can create a conic curve by specifying 3 points.

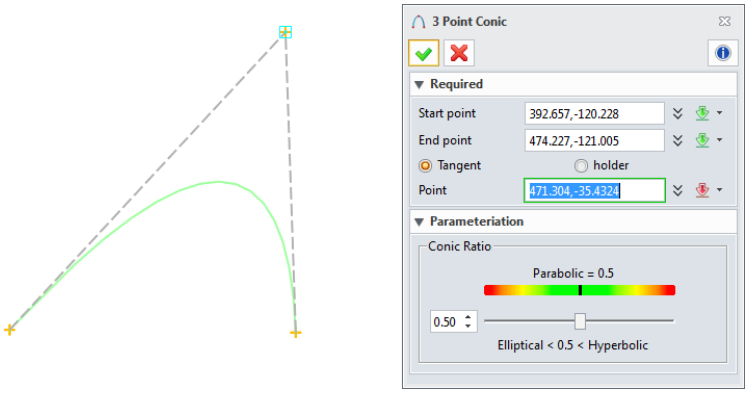


Figure 51 Creating a Conic Curve

Sketch Ribbon Tab->Curve->Point Cloud Curve

You can create an open or closed curve that passes through a cloud of points. Besides, you can specify the tangent direction of the starting point or endpoint.

Note: The curvature combs will not appear when you're creating the curve.

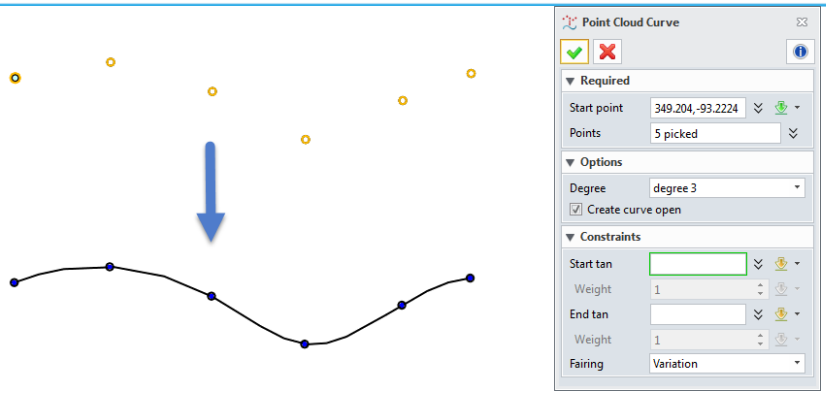


Figure 52 Point Cloud Curve

2.7 Equation Curves

Sketch Ribbon Tab->Curve->Equation

You can create a curve with an equation. Predefined curves are also provided.

STEP 01 Double-click on an equation.

STEP 02 Use the default insert point (0, 0, 0) or pick a new one to insert the equation curve.

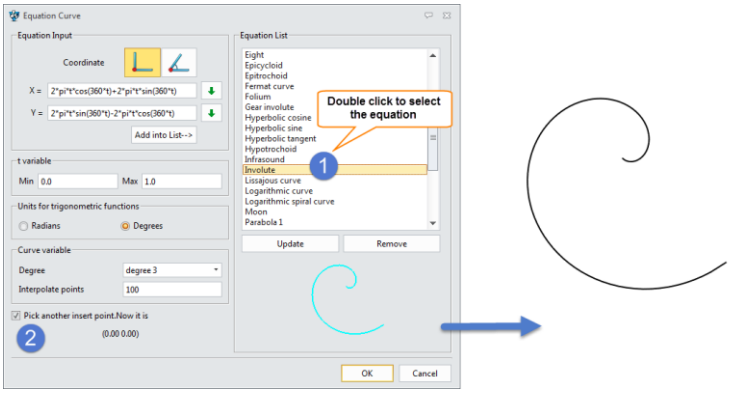


Figure 53 Insert an Equation Curve

2.8 Continuous Curves

Sketch Ribbon Tab->Drawing->Draw

You can use the Quick Draw feature to create a continuous closed or open curve without switching between commands.

During the creation of a continuous curve, two symbols (Connective & Tangent) will appear next to the selected point. They indicate the status of tangency at this point.

The default status is Connective. Clicking again will toggle the active status.

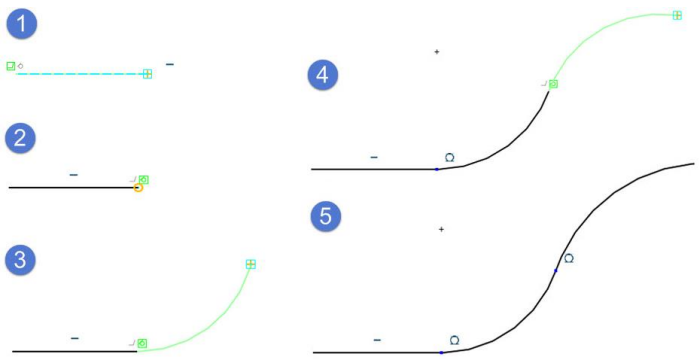


Figure 54 Quick Draw

Geometries that can be drawn with Quick Draw

1) Arc

STEP 01 When the active status of tangency is Connective, press and hold the Alt key while selecting the endpoint of the arc.

STEP 02 Pick the third point to determine the midpoint of the arc (point 4 below).

STEP 03 Middle-click when you are done drawing the arc.

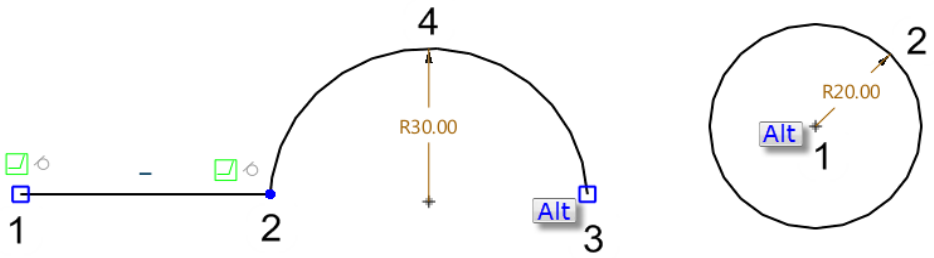


Figure 55 Creating an Arc/Circle with Quick Draw

2) Circle

STEP 01 Press and hold the Alt key while selecting the center of the circle.

STEP 02 Pick a point as the boundary of the circle.

STEP 03 Middle-click when you're done drawing the circle.

3) Curve through points

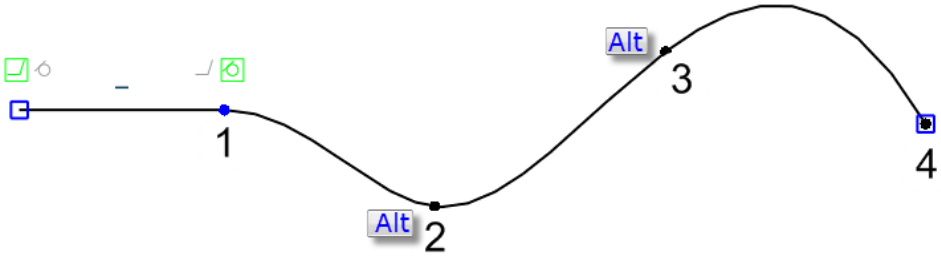


Figure 56 Creating a Curve Through Points with Quick Draw

STEP 01 Pick a point as the starting point of the spline.

STEP 02 Press and hold the Alt key while defining the second and third points.

STEP 03 Keep picking other points without pressing the Alt key.

2.9 Offset

STEP 01 Select the curves and set the value of offset distance.

STEP 02 If you find the direction incorrect in preview, check the ***Flip the direction*** option to change it. If you want to offset the curves in both directions, simply check the ***Offset in both directions*** option.

STEP 03 Check the ***Insert arcs at convex corners*** option to create the full arcs at corners. The radius of the arc is equal to that of the offset distance. If you check the ***Add fillets to hard corners*** option, fillets will be created between two adjacent curves according to the assigned value of the radius.

STEP 04 Set other parameters, for example, the offset number, according to the requirements.

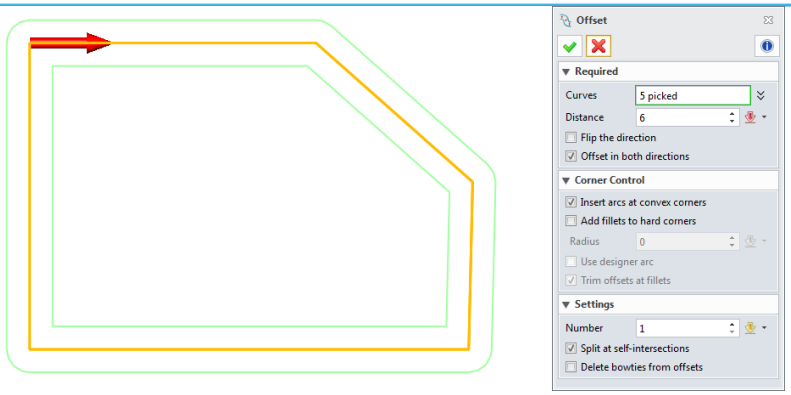


Figure 57 Offset the Curves

2.10 Blend

You can create a blend curve between two selected curves.

STEP 01 Select the first curve. The cursor's position is the starting point of the blend curve.

STEP 02 Select the second curve. The cursor's position is the starting point of the blend curve.

STEP 03 Define the type of Continuity of the starting point and endpoint. Adjust the value of Weight to get a real-time preview.

STEP 04 Check the ***Display curvature*** option to preview the blend curve.

STEP 05 Middle-click to get the result.

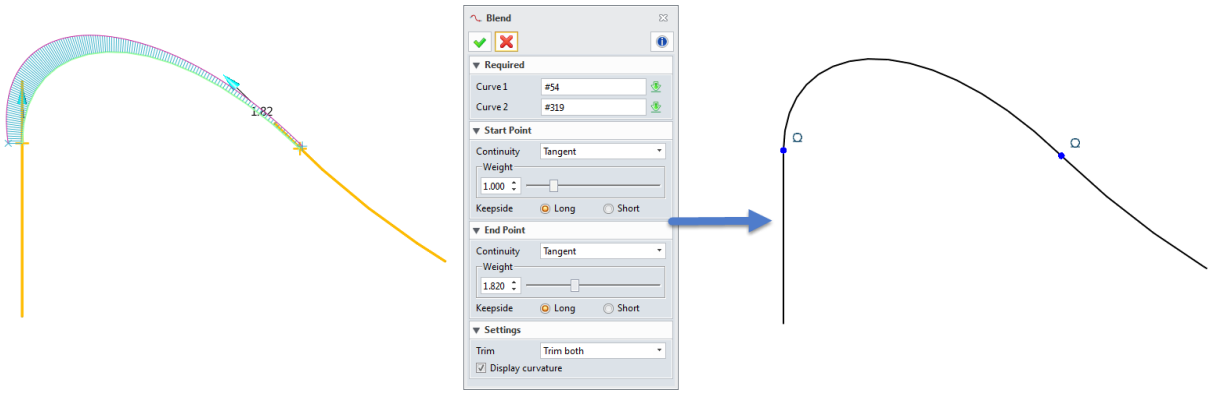


Figure 58 Blend Curve

2.11 Text

2.11.1 Ready sketch text

Sketch Ribbon Tab->Drawing->Ready Sketch Text

You can create a sketch text horizontally or along a curve and set its font, style, and size.

To edit the sketch text, simply double-click on it and proceed to modify it, its font, style, or size.

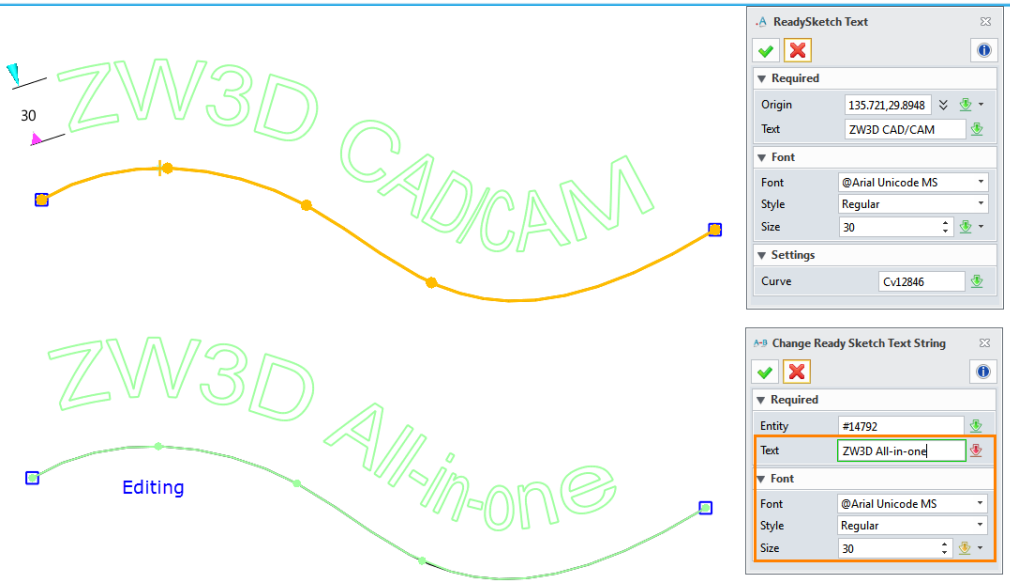


Figure 59 Create/Edit Ready Sketch Text

How to enrich your design with Ready Sketch Texts?

One way it to extrude the sketch of the Ready Sketch Text to get a solid.

The other is to wrap it around a surface, then create an inlay or embossed feature.

Figure 60 Design with Ready Sketch Text

2.11.2 Text

Sketch Ribbon Tab->Drawing->Text

This command can help you create a text feature in the sketch.

Note: It's can't be used in the modeling environment.

Figure 61 Text

2.11.3 Balloon

Sketch ribbon Tab->Drawing-> Balloon

Use this tool to create a text annotation or image balloon at the selected point.

To edit the text string, simply double-click on the text annotation.

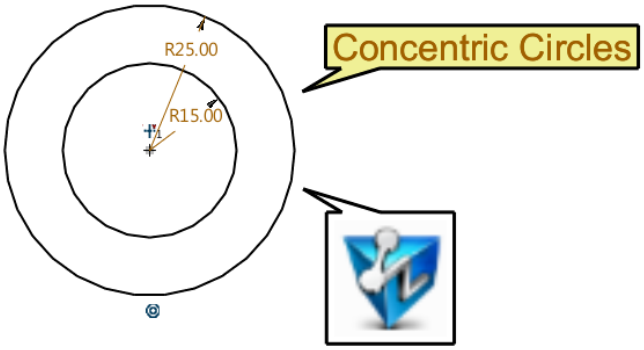


Figure 62 Balloon Annotation

2.12 Ready Sketch

Sketch Ribbon Tab->Ready Sketch

You can choose from the predefined sketches and insert one anywhere. The ready sketch geometries are well-defined. That said, you can edit the value of the dimensions to change their size.

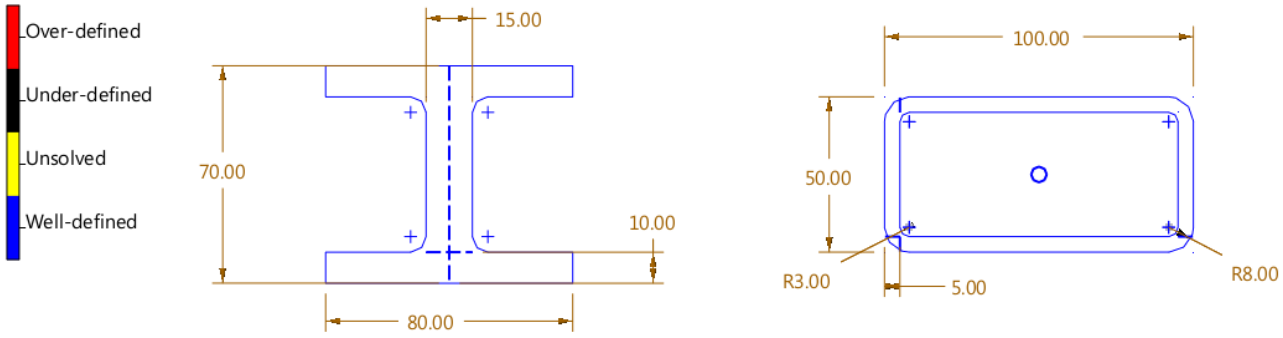


Figure 63 Ready Sketch

How to add your custom block into the Ready Sketch library?

STEP 01 Turn the sketch into a block.

STEP 02 Right-click on the block and click ***Save to Ready Sketch*** in the shortcut menu.

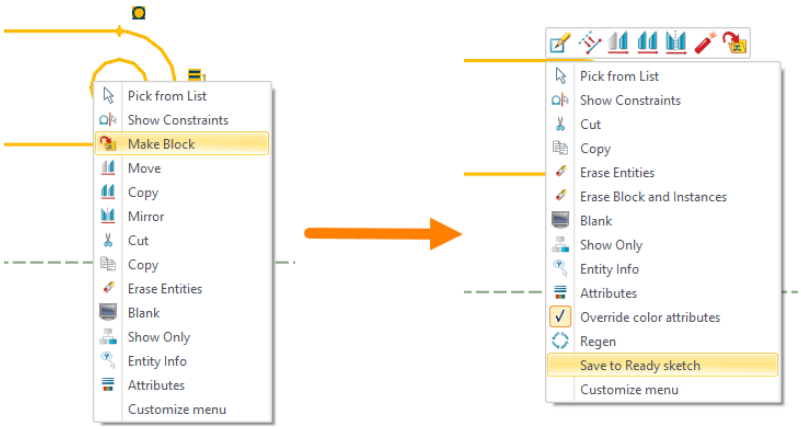


Figure 64 Customizing the Ready Sketch Library

3 Edit Curves

3.1 Fillet

Sketch Ribbon Tab->Edit Curve->Fillet

STEP 01 Select two curves and set the value of the radius.

STEP 02 Middle-click or click the ***OK*** button to finish filleting.

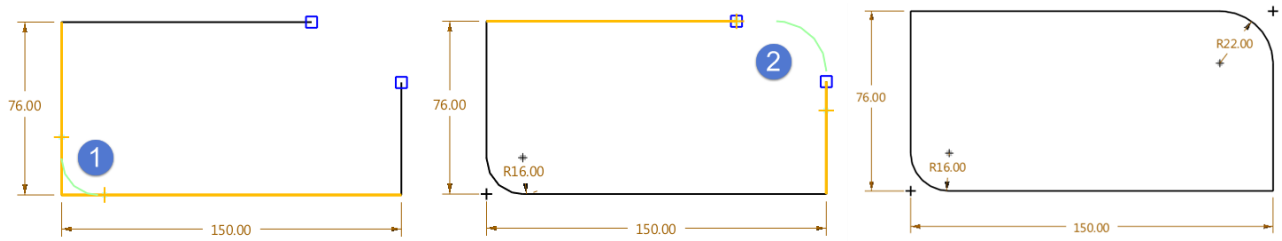


Figure 65 Fillet

Sketch Ribbon Tab->Edit Curve->Fillet Chain

STEP 01 Select the chain of curves.

STEP 02 Define the value of the radius.

STEP 03 Middle-click or click the ***OK*** button to finish filleting.

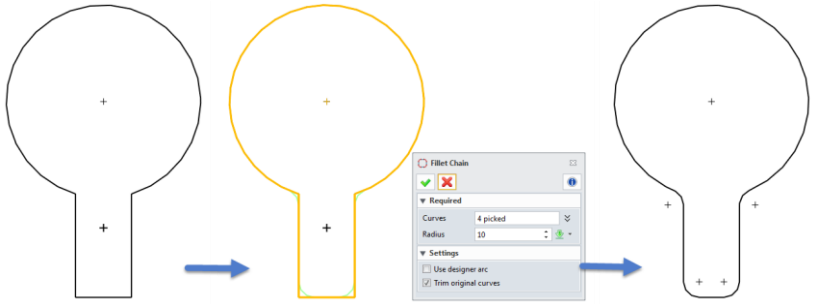


Figure 66 Fillet Chain

3.2 Chamfer

Sketch Ribbon Tab->Edit Curve->Chamfer

There are three ways to chamfer a sketch. The following example will be done with the 2 Setback Values method.

STEP 01 Pick two curves.

STEP 02 Set the values of two setbacks.

STEP 03 Determine the types of ***Trim*** and ***Extension*** according to the requirements (You can refer to Figure 18) and click ***OK*** to finish the operation.

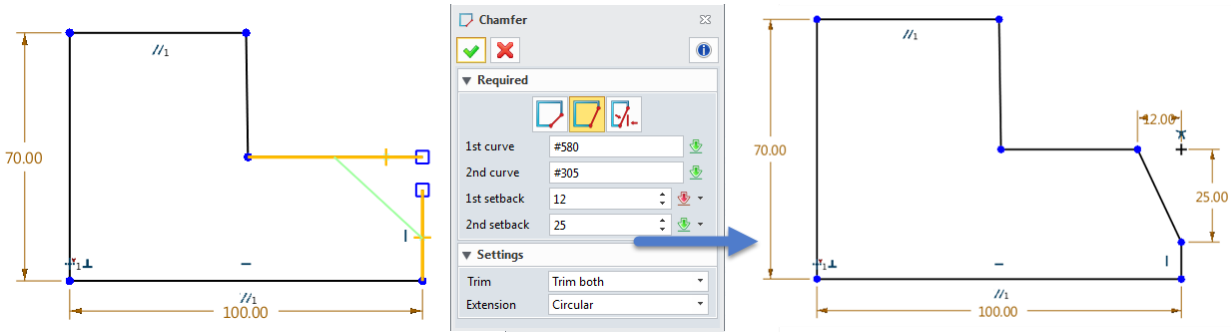


Figure 67 Chamfer

Sketch Ribbon Tab->Edit Curve->Chamfer Chain

STEP 01 Select the chain of curves.

STEP 02 Set the value of the setback.

STEP 03 Middle-click and the chamfers between each group of adjacent curves will be created.

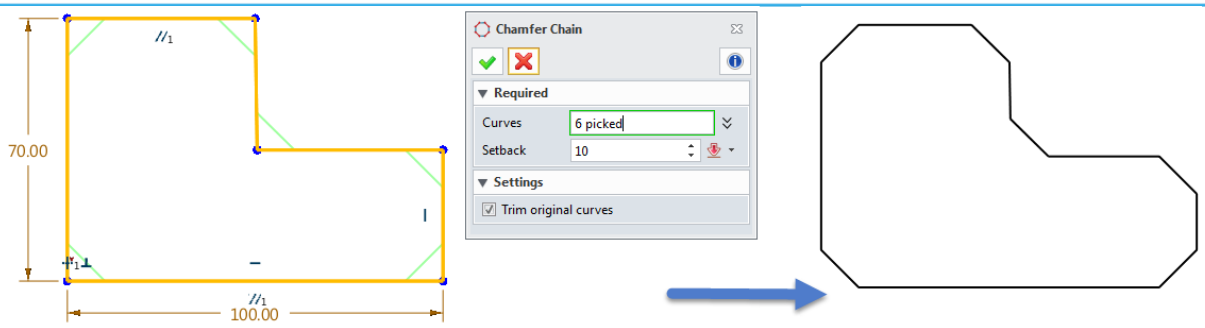


Figure 68 Chamfer Chain

3.3 Trim

Sketch Ribbon Tab->Edit Curve->Power Trim

When the cursor is over an entity, press and hold down the left mouse button to trim it.

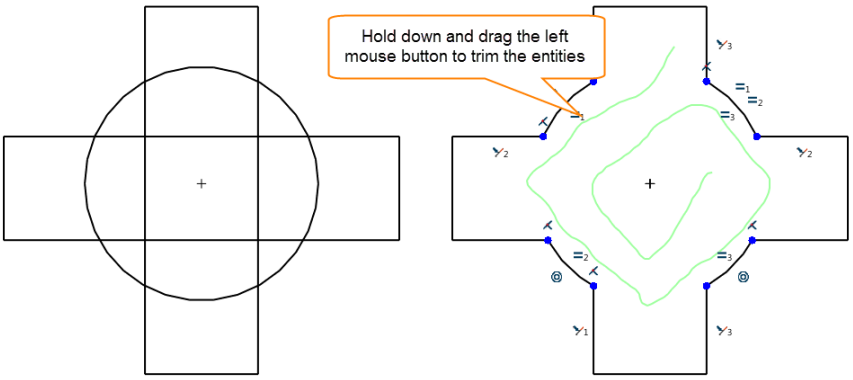


Figure 69 Power Trim

Note: You can't trim a closed curve.

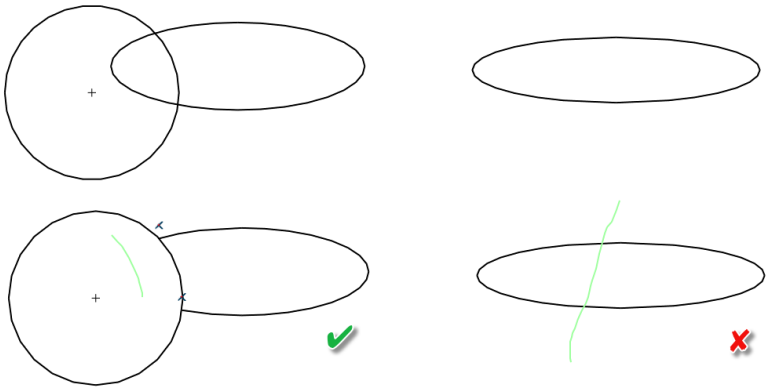


Figure 70 The Availability of Power Trim

Sketch Ribbon Tab->Edit Curve->One Touch Trim

You can trim the selected curve with the nearest intersecting or non-intersecting curves as the boundaries.

STEP 01 Select the curve.

STEP 02 Click ***OK*** to finish trimming.

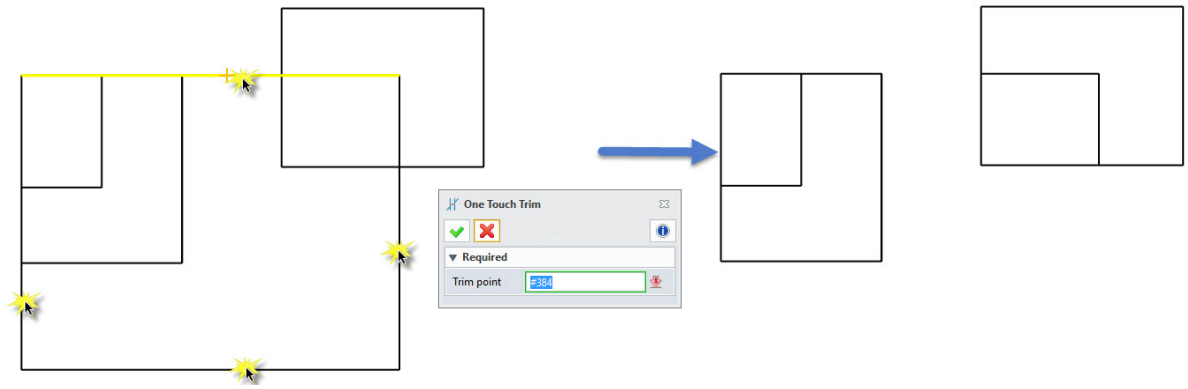


Figure 71 One-Touch Trim

Sketch Ribbon Tab->Edit Curve->Trim/Extend

STEP 01 Select the curves that you are trimming or extending.

STEP 02 Select the destination curve.

STEP 03 Middle-click to finish. If the selected curves and the destination one is intersecting, the selected curves will be trimmed. Otherwise, they will be extended.

Note: You can refer to Figure 18 for the types of Extension.

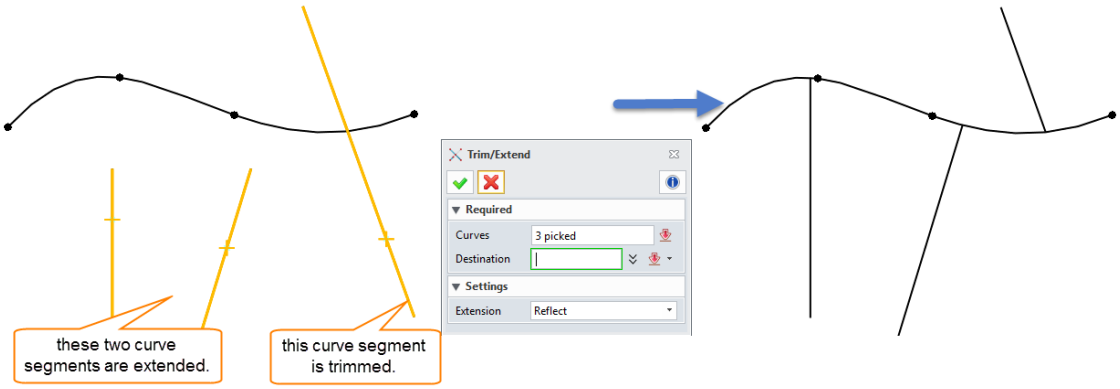


Figure 72 Trim or Extend

Sketch Ribbon Tab->Edit Curve->Trim/Spilt Curve

STEP 01 Select the curves as the boundary curves (Blue ones in the image below).

STEP 02 Pick the curves and set the Trim mode (Keep/Delete/Break).

STEP 03 Middle-click to finish.

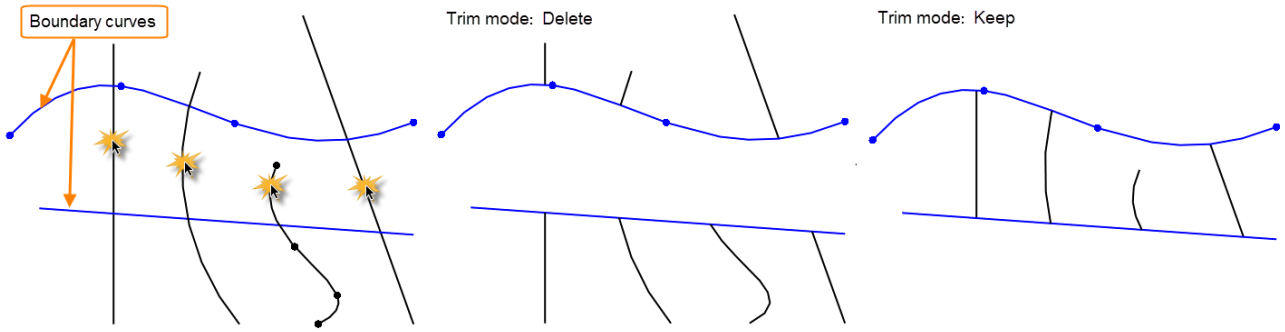


Figure 73 Trim/Split Curve

Sketch Ribbon Tab->Edit Curve->Trim/Split at Points

You can trim or split the curve at selected break points.

STEP 01 Select the curve.

STEP 02 Pick the breakpoints.

STEP 03 Select the segments to keep them or middle click to split the curve.

Sketch Ribbon Tab->Edit Curve->Trim/Extend to Corner

You can create a corner by trimming two curves that intersect or extending them if they are not parallel.

STEP 01 Select the first curve to keep one side.

STEP 02 Select the second curve to keep one side.

STEP 03 Click ***OK*** to finish.

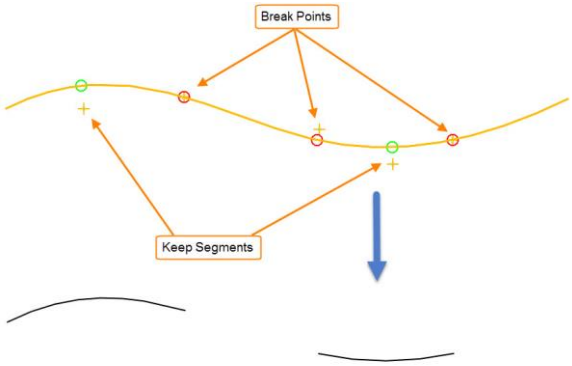


Figure 74 Trim/Split at Points

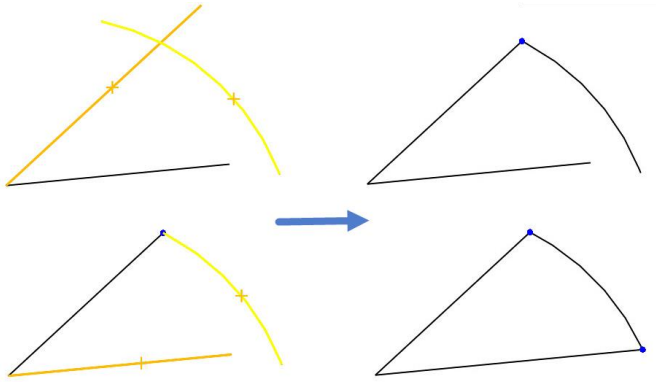


Figure 75 Trim/Extend to Corner

Sketch Ribbon Tab->Edit Curve->Delete Bowties

When the offset distance of curves with fillets is greater than the fillet's radius, bowties will be automatically created and require manual deletion.

STEP 01 Select the bowties.

STEP 02 Middle-click to finish deleting.

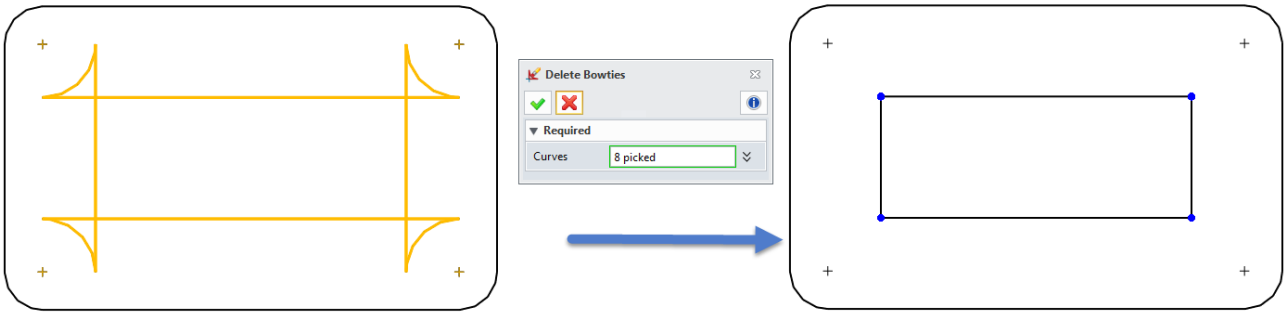


Figure 76 Delete Bowties

Sketch Ribbon Tab->Edit Curve->Spilt at Intersections

You can split the curves at self-intersections or curve-curve intersections.

STEP 01 Select the bowties.

STEP 02 Check the ***Settings*** options according to your actual geometries, then middle-click to finish splitting the curves at the intersections.

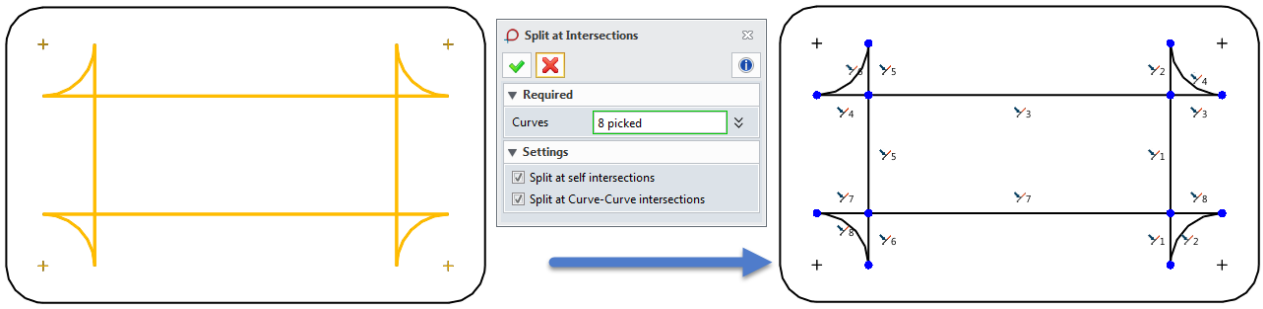


Figure 77 Split at Intersections

3.4 Trace Profile

Sketch Ribbon Tab->Edit Curve->Trace Profile

Most of the geometries in the sketch are references, only a few geometries are useful for future modeling. ***Trace Profile*** is designed to help you specify which parts of the sketch are for modeling.

1) Specify the profile

When the Trace Profile dialog box pops up, you can click the parts that are for modeling and they will turn blue.

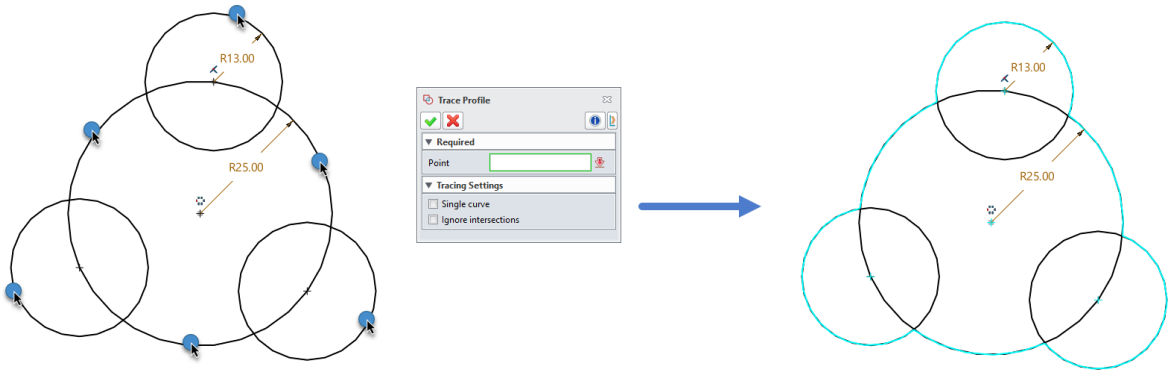


Figure 78 Specify the Profile

Then, return to the CAD module and execute other modeling commands like Extrude. And you will find that the base of your model is the traced part.

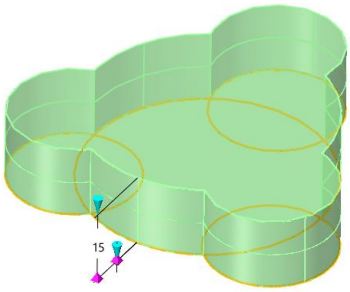


Figure 79 Trace Profile as the Base of Modeling

2) Delete trace profile

Sketch Ribbon Tab->Edit Curve->Delete Trace Profile

After clicking this command, you can select the trace profile you want to delete.

4 Edit Splines

Sketch Ribbon Tab->Edit Curve->Modify

You can modify the position, tangency, and radius of curves at any point on the curve. Let's take a through the point curve as an example and see how to modify it.

1) Add the point

Pick any point on the curve, except the existing ones. They will automatically become new points that the curve will pass through.

2) Delete the point

Pick any existing point, then right-click and select ***Delete Point*** in the context menu to delete the selected point.

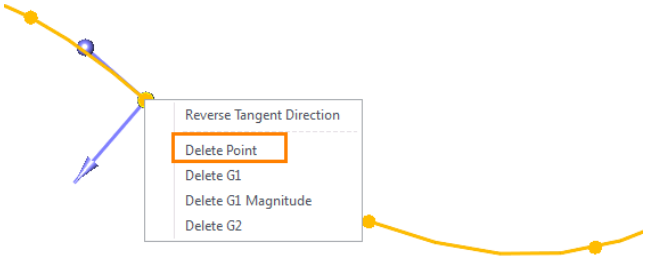


Figure 80 Delete the Point

3) Modify the tangency of any point

Pick a point, and the control handle which controls the tangency, magnitude, and radius of curvature will appear. To modify the tangency, you can directly drag the control handles or enter values.

If you check the ***Display inflection points*** and ***Display minimum radius*** options, the inflection points and minus radius will appear as shown in the image below.

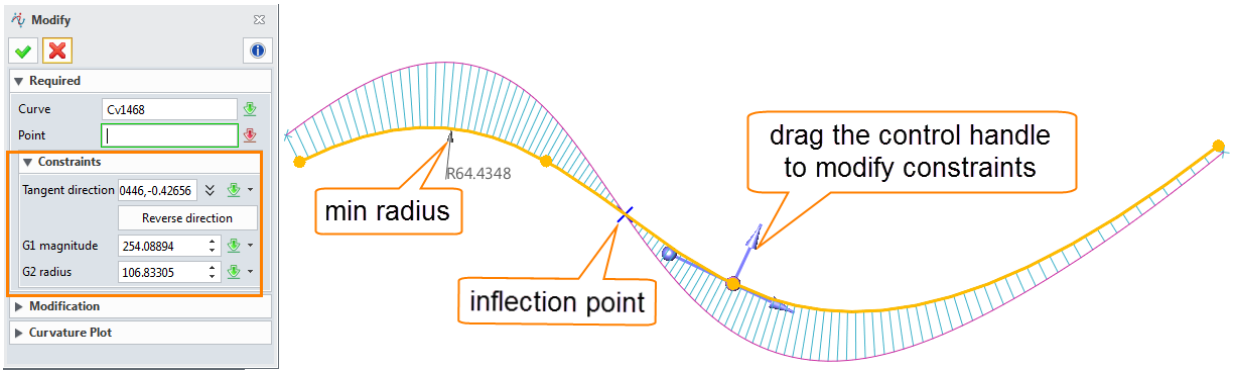


Figure 81 Modify the Point

4) Modify the curve with control points

Select the curve, and then click the ***Convert to Control Point Curve*** button.

After that, the selected curve becomes one with control points. Then you can modify the control points to shape the curve.

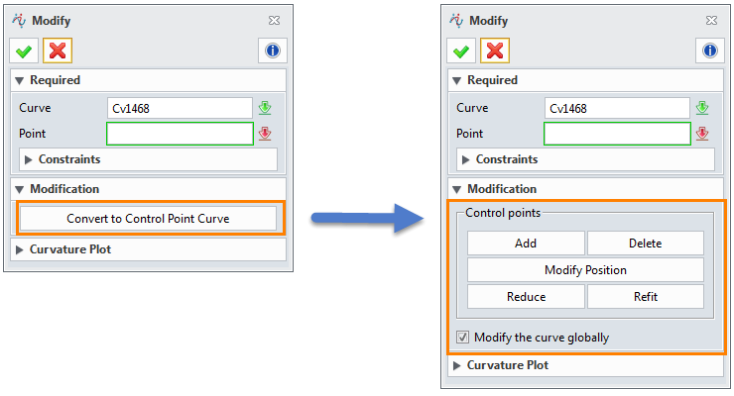


Figure 82 Convert to Control Point Curve

5 Import References

5.1 Reference

Sketch Ribbon Tab->Reference->Reference

When you are drawing some geometries, especially the structure of a surface, you might need to refer to some external geometries of the model. With the ***Reference*** command, you can choose how you'd like the reference geometries to be in the Reference dialogue box.

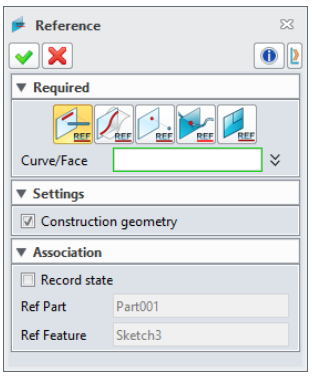


Figure 83 Reference

1) Curve/Face

In this mode, you can choose a curve or a face from the external model, and a curve projected from the external curve/face will be automatically created on the sketch plane.

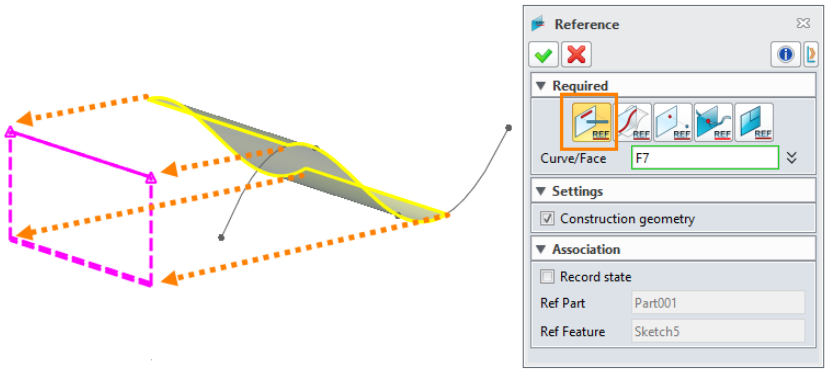


Figure 84 Projected References on the Sketch Plane

2) Intersection Curve

In this mode, you can choose a face which is intersected with the current sketch datum as reference. Then, a curve indicating the intersecting part will be generated, as shown in Figure 85.

3) Point

Like ***Curve/Face***, this mode requires you to select a point. Then, a reference point will be generated on the sketch datum.

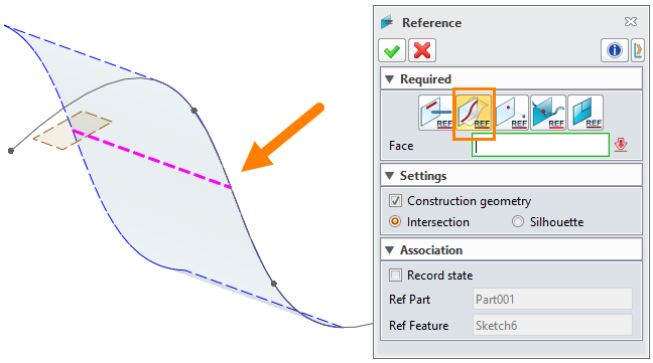


Figure 85 The Reference Curve

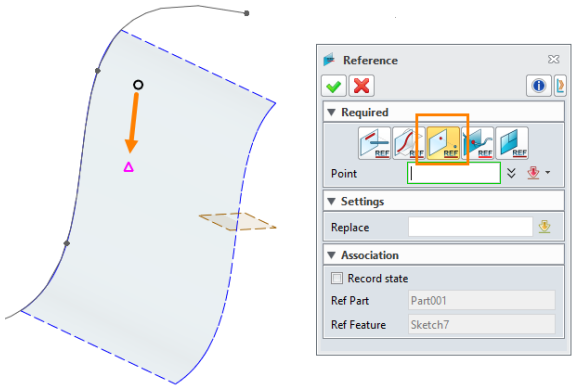


Figure 86 The Projected Reference Point

4) Intersection point

Like ***Intersection Curve***, you can select a curve which is intersected with the current sketch datum as reference in this mode. Then, the intersection point will become a reference point.

5) Reference datum

By selecting an external datum, a projected reference curve will be created.

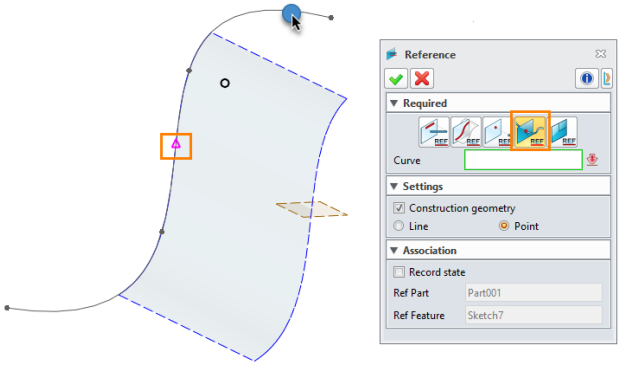


Figure 87 The Intersection Point as Reference

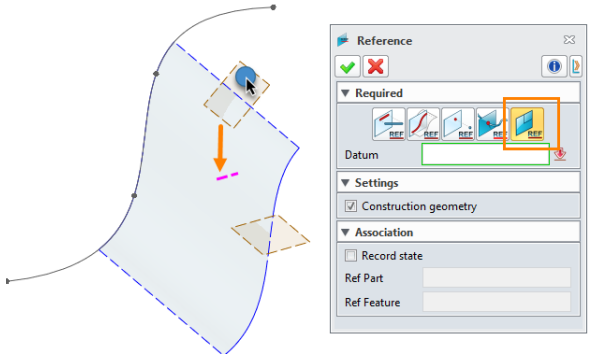


Figure 88 The Projected Reference Curve

5.2 Image

Sketch Ribbon Tab->Reference->Image

This command can help you refer to hand-drawn sketches when modeling.

STEP 01 Specify the path of the image file.

STEP 02 Determine the insertion position of the image. You can change the size of the image by dragging its opposite corner or specifying the width and length in the Image dialogue box.

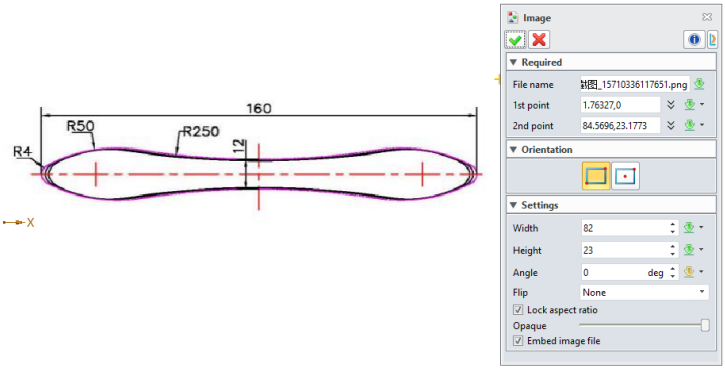


Figure 89 Inserting Reference Images

6 Edit the Sketch

6.1 Pattern

Sketch Ribbon Tab->Basic Editing->Pattern

In the 2D sketch level, you can find linear and circular patterns.

1) Linear Pattern

STEP 01 Select an entity.

Note: The selected entity will be highlighted in orange.

STEP 02 Specify the direction and parameters (such as Number and Pitch Distance) of the pattern.

STEP 03 Determine the second direction and parameters of the pattern according to the requirements.

STEP 04 Select the unnecessary instances and toggle them to be off. Then, those entities will turn red.

2) Circular Pattern

STEP 01 Select an entity.

STEP 02 Specify the center.

STEP 03 Set the parameters of the pattern, such as Pitch Angle and Span Angle.

STEP 04 Select the unnecessary instances and toggle them to be off.

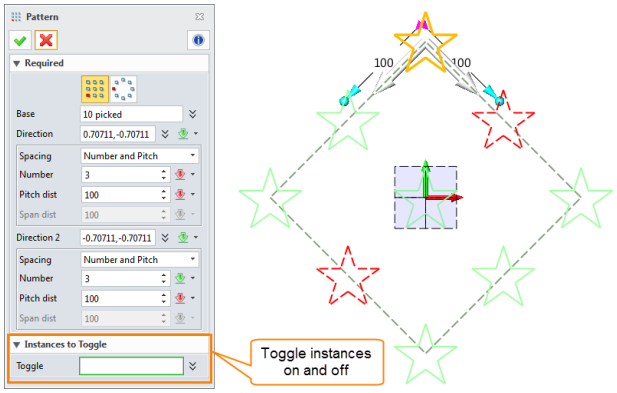


Figure 90 Linear Pattern

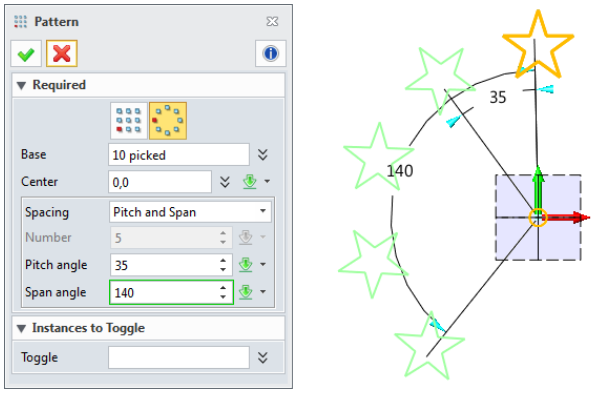


Figure 91 Circular Pattern

6.2 Move/Copy/Rotate

Sketch Ribbon Tab->Basic Editing->Move

STEP 01 Select the entity that you are moving.

STEP 02 Determine a reference point and a destination point.

STEP 03 Specify the direction of movement, angle and scale according to the requirements.

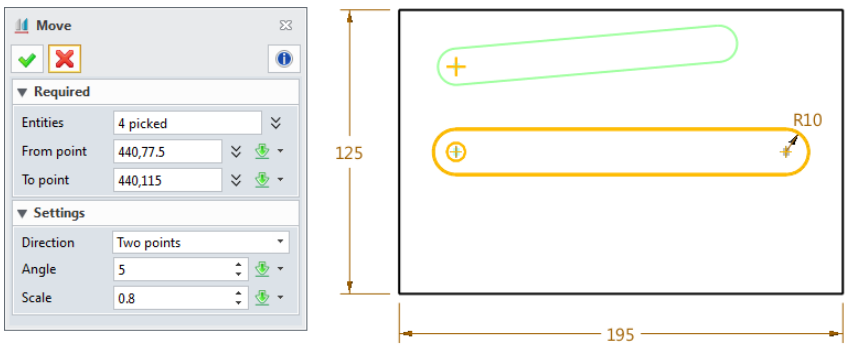


Figure 92 Move the Entities

Sketch Ribbon Tab->Basic Editing->Copy

In addition to all the ***Move*** parameters, you can specify the number of copies.

STEP 01 Select the entity that you are copying.

STEP 02 Determine a reference point and a destination point.

STEP 03 Specify the parameters.

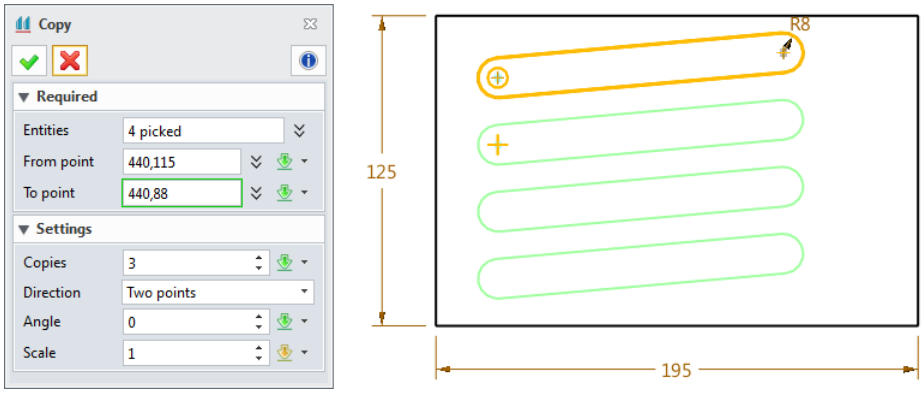


Figure 93 Copy the Entities

Sketch Ribbon Tab->Basic Editing->Rotate

You can directly rotate the entity or rotate it while making copies.

STEP 01 Select the entity and define the base point.

STEP 02 Set the angle of rotation by specifying a value (when the ***Angle*** option is checked) or the rotating points (when the ***Points*** option is checked).

STEP 03 Select a rotating method, ***Move*** (Figure 94) or ***Copy*** (Figure 95).

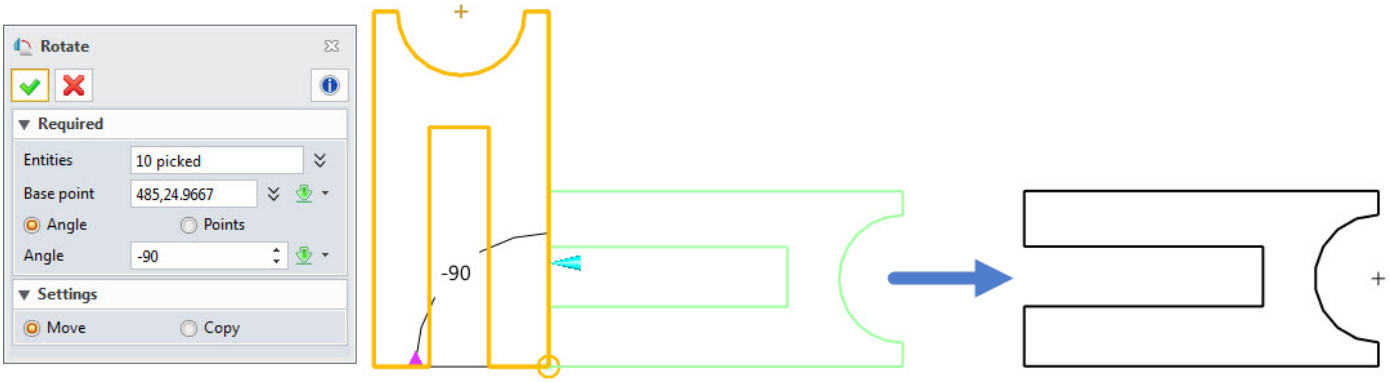


Figure 94 Rotate—Move the Entities

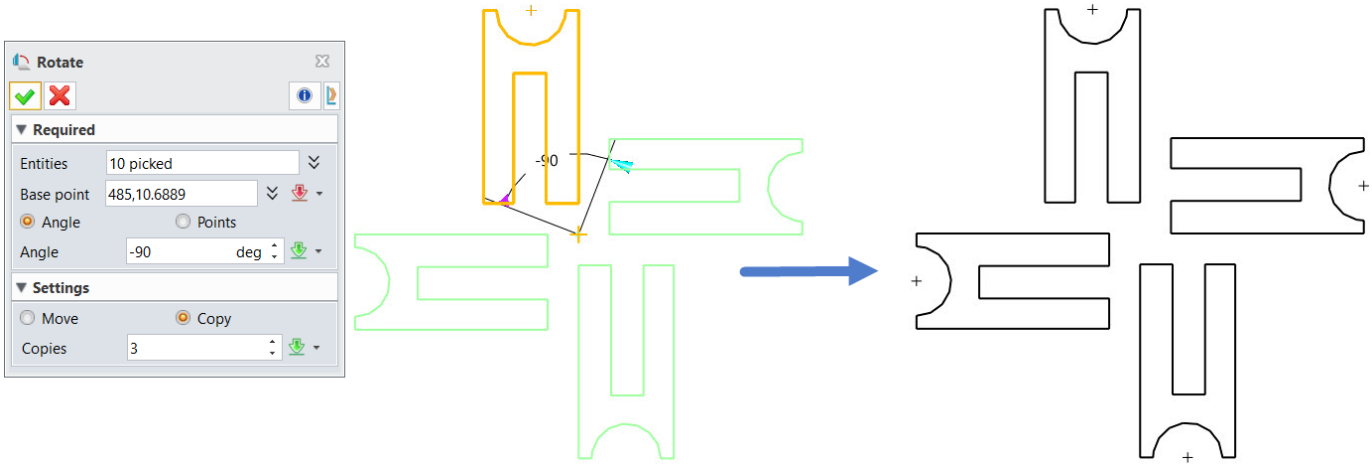


Figure 95 Rotate—Copy the Entities

6.3 Mirror

Sketch Ribbon Tab->Basic Editing->Mirror

STEP 01 Select some entities.

STEP 02 Determine the line of symmetry (a geometric or construction line).

STEP 03 Check the ***Keep original entities*** option to keep the original entities if necessary.

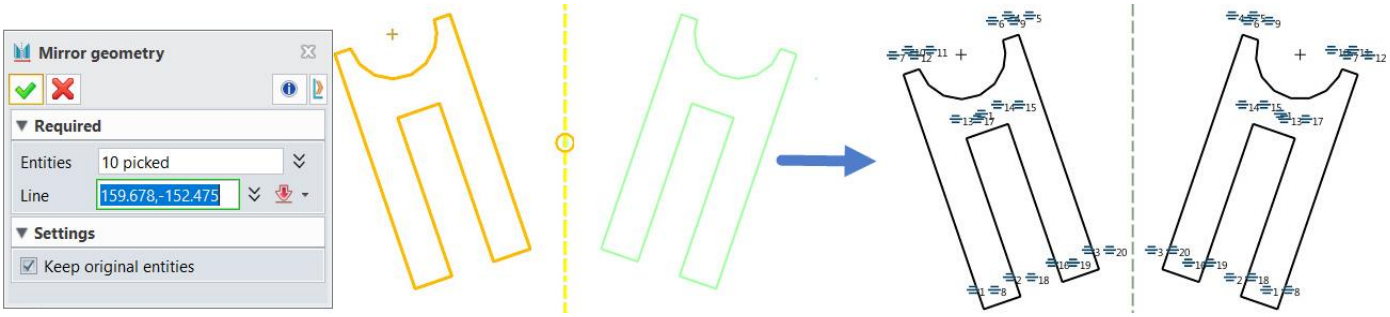


Figure 96 Mirror the Entities

Notes: 1) The mirror constraints are automatically created.

2) When you change the size of the original entities, the mirrored entities are auto updated.

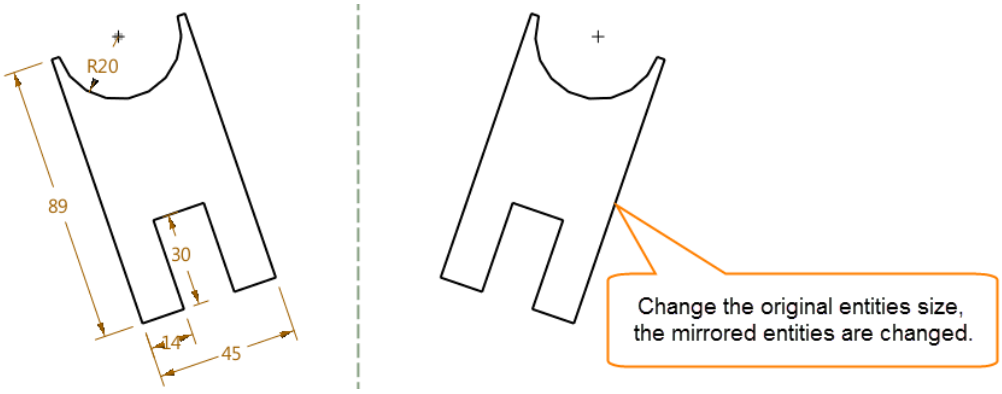


Figure 97 Auto-updated Mirrored Entities

6.4 Scale/Stretch/Drag

Sketch Ribbon Tab->Basic Editing->Scale

STEP 01 Select a scale type, ***Factor*** or ***Point***.

STEP 02 Select the entities and define the base point.

STEP 03 Select the scale method, ***Uniform*** or ***Non-uniform***. Then, define the related parameters.

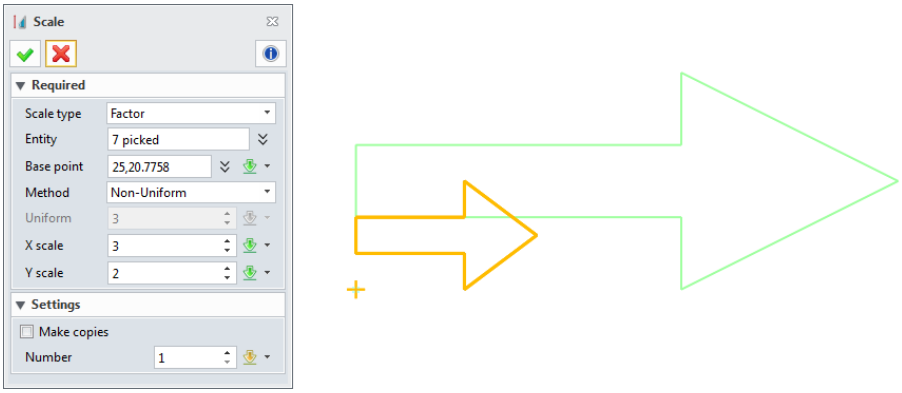


Figure 98 Factor—Non-uniform Scale

Note: If the scale type is ***Point***, the value of scale will be automatically calculated according to the information of the point.

(Value of Scale = distance between the ***To point*** and the ***Base point***/distance between the ***From point*** and the ***Base point***).

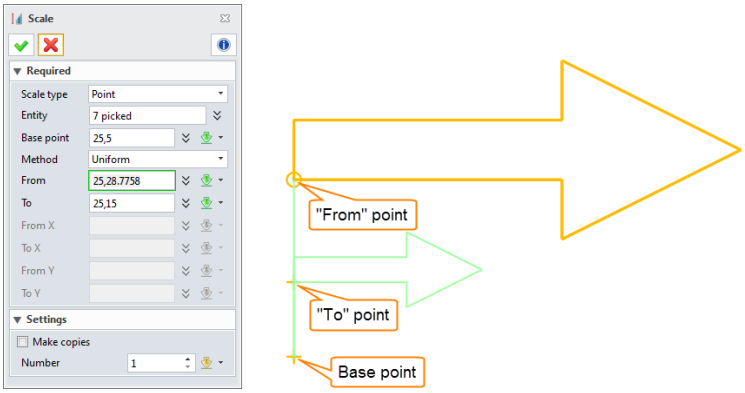


Figure 99 Point—Uniform Scale

Sketch Ribbon Tab->Basic Editing->Stretch

You can stretch both well-defined and under-defined geometries.

STEP 01 Select the necessary points within a rectangular. As Figure 100 shows, the selected points will be marked in green circles.

STEP 02 Determine the ***From point*** and the ***To point***.

STEP 03 Set the direction: Two points method.

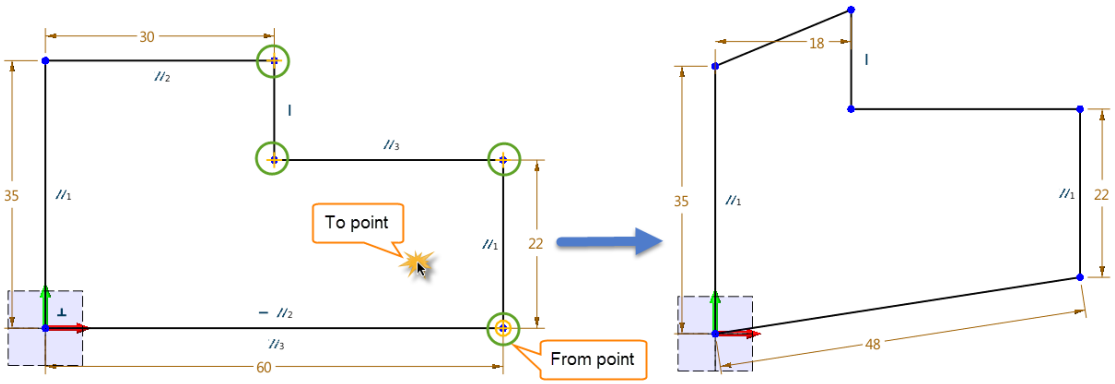


Figure 100 Stretch the Points

Sketch Ribbon Tab->Basic Editing->Drag

STEP 01 Select the entity and define the ***From point***.

STEP 02 Define the ***To point***.

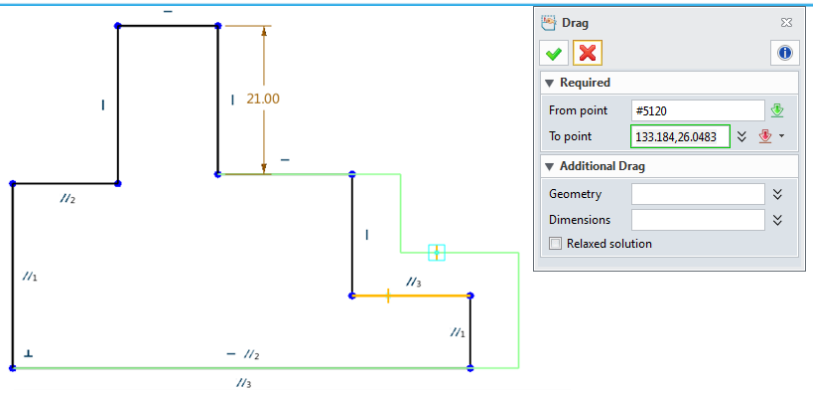


Figure 101 Simple Drag

The additional Drag parameters?

If you want to drag more geometries at once, you can select them in the ***Geometry*** box, as shown in the left image of Figure 102.

If you want to drag some dimensions together, you can define them in the ***Dimensions*** box, as shown in the right image of Figure 102.

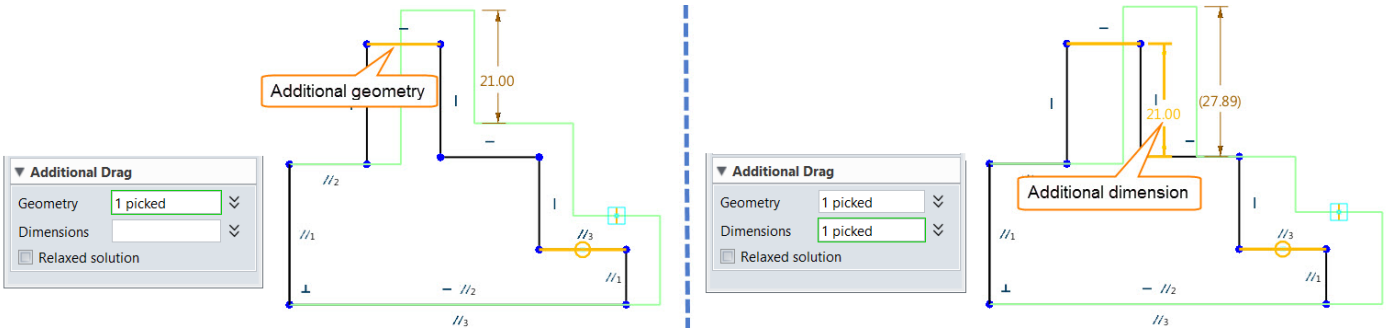


Figure 102 Additional Drag

If you want to drag the entire closed geometry, simply check the ***Relaxed solution*** option.

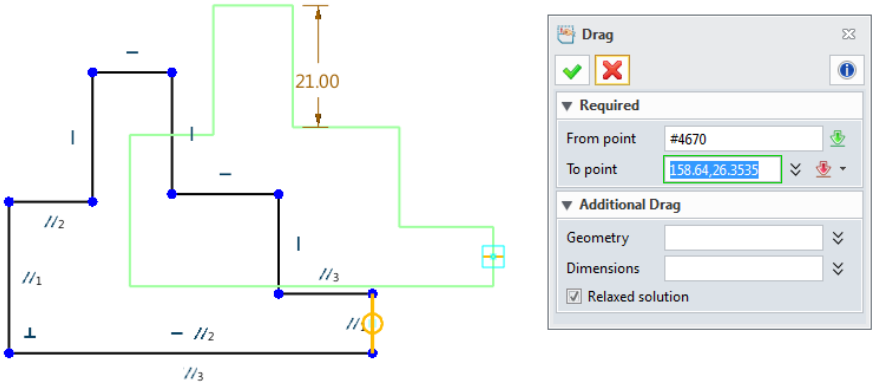


Figure 103 Drag-Relaxed solution

7 Constraints

7.1 Setting Constraint Status

1) Customize the constraint color

Tool Ribbon Tab->Attributes->Constraint Color Settings

Click the button of color and choose a color from the Standard dialogue box.

Note: If unchecked, the constraint will not appear in the working area even when the ***Display color*** option is turned on.

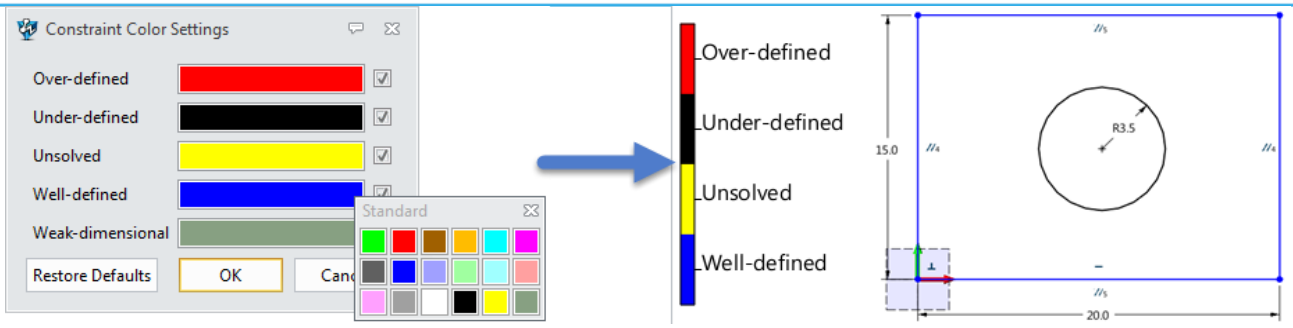


Figure 104 Customizing Constraint Colors

2) Turn on/off constraint status color

Please refer to Figure 7.

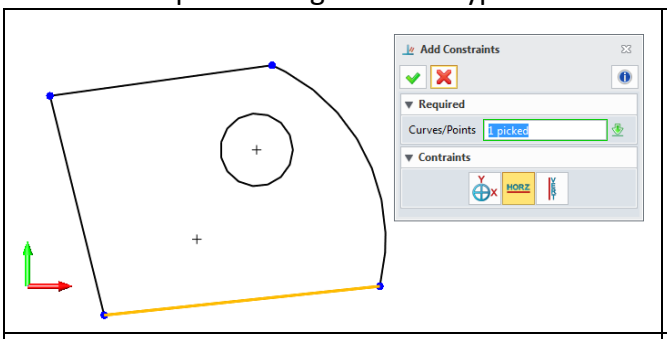
7.2 Add Constraints

Constraint Ribbon Tab->Constraint->Add constraints

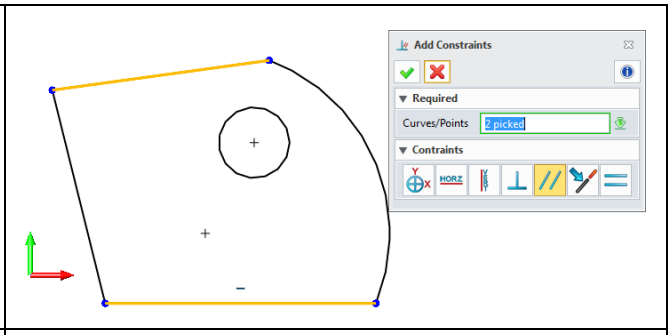
STEP 01 Select the entities (points or curves).

STEP 02 According to the selected entities, the system shows the available constraint types. Select the needed type to finish the constraining.

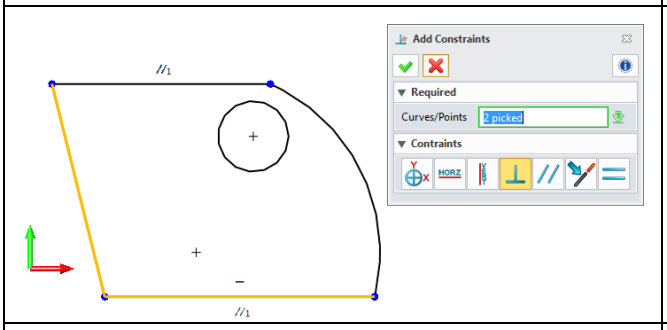
An example of using different types of constraints is illustrated below.



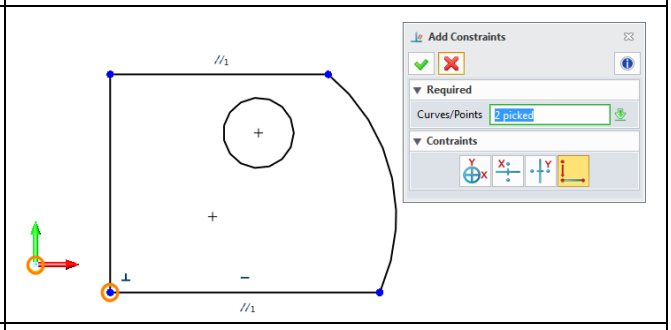
1. Horizontal constraint



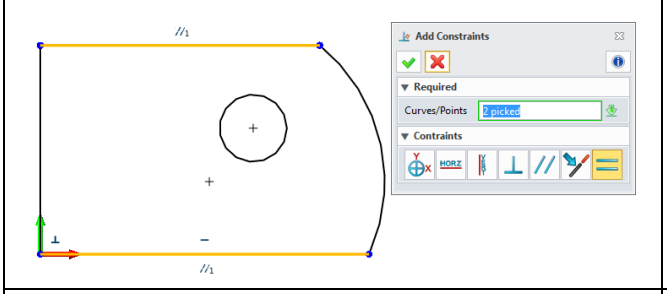
2. Parallel constraint



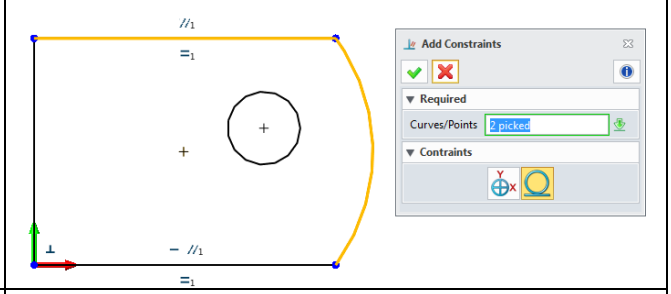
3. Perpendicular constraint



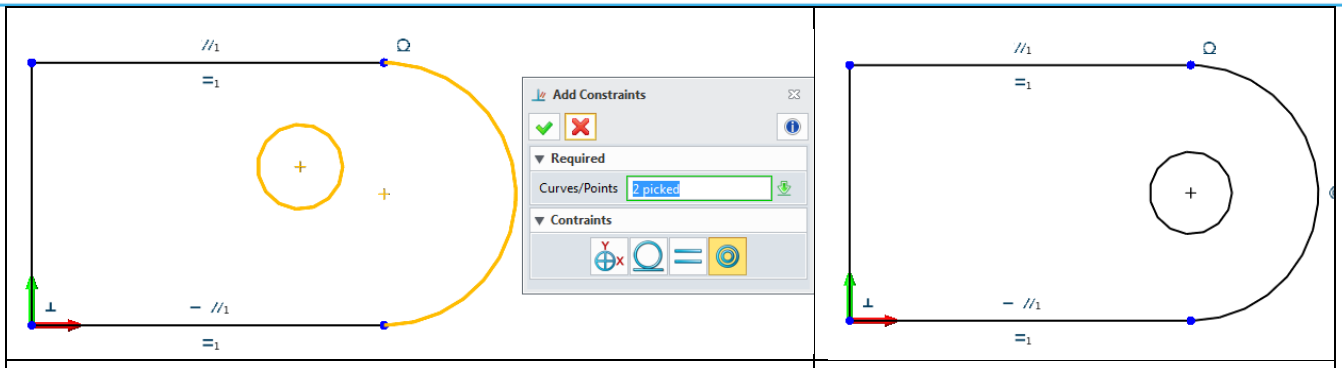
4. Points coincident constraint



5. Equal length constraint



6. Tangent constraint



7. Concentric constraint	8. Result

Figure 105 Adding Constraints

After adding the constraints, we should add dimensions to well define it. The ***Quick Dimension*** command is very helpful in this case.

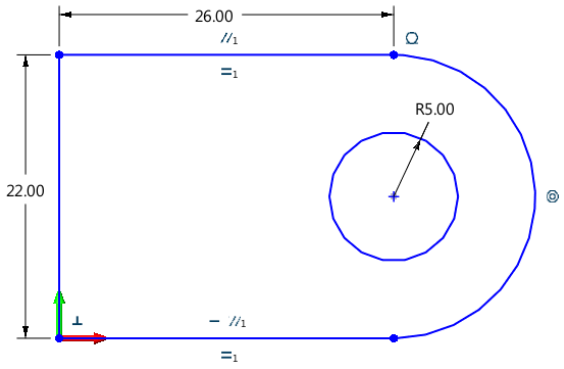
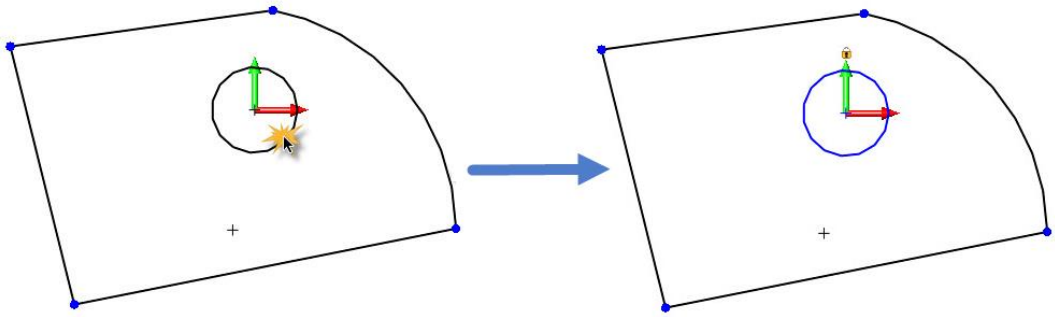
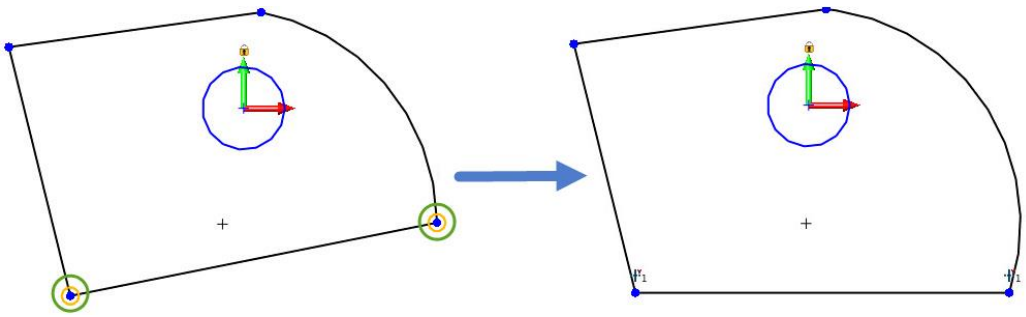
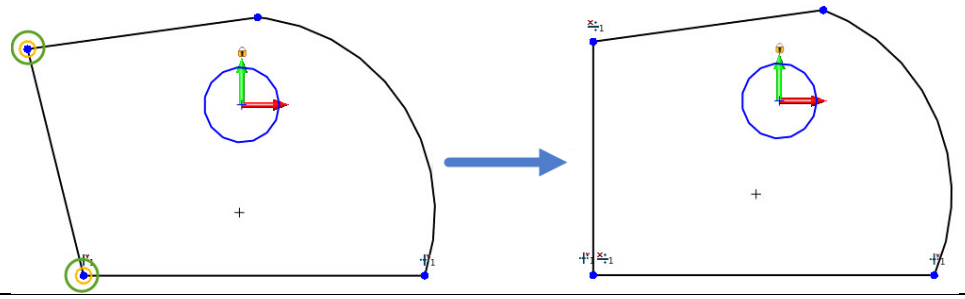
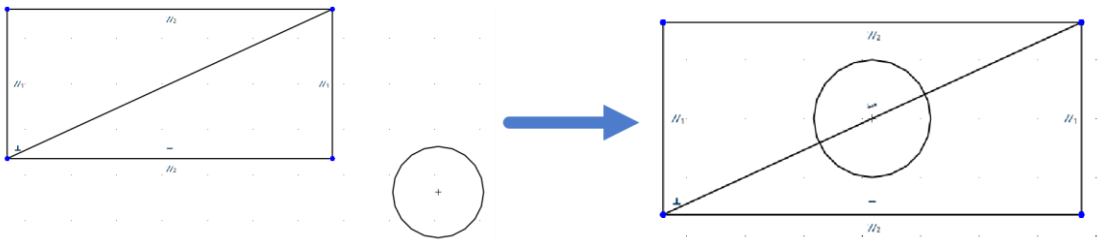
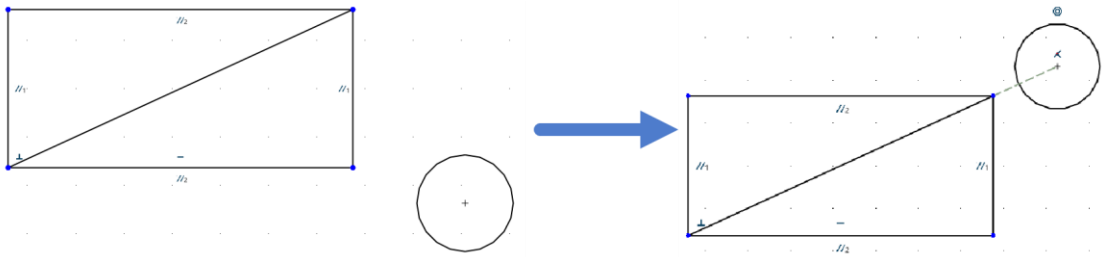
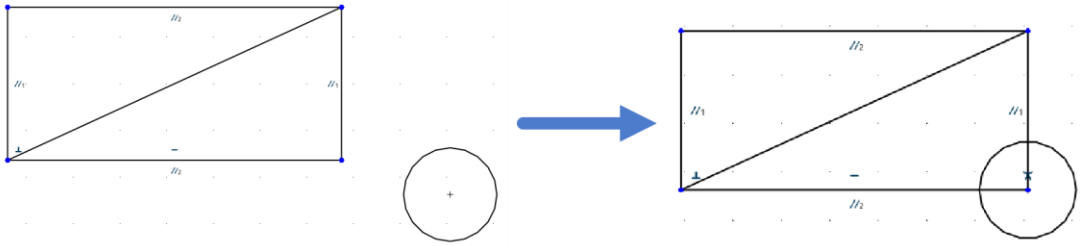
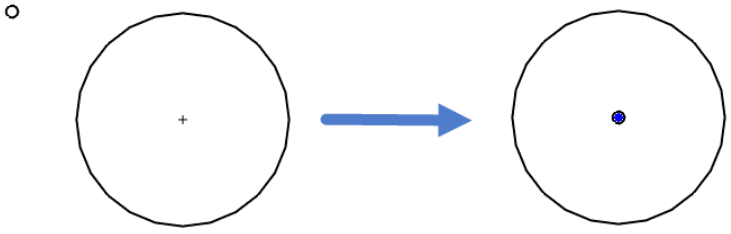
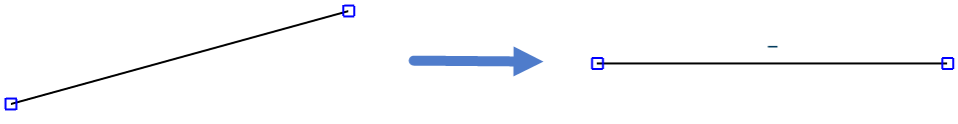


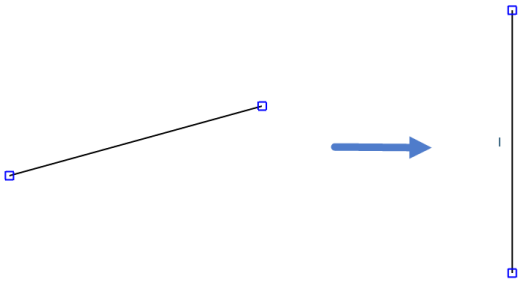
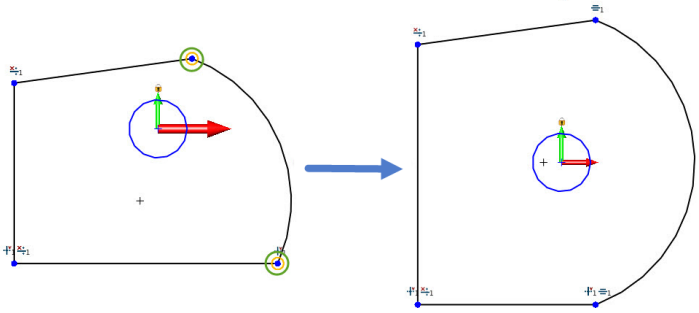
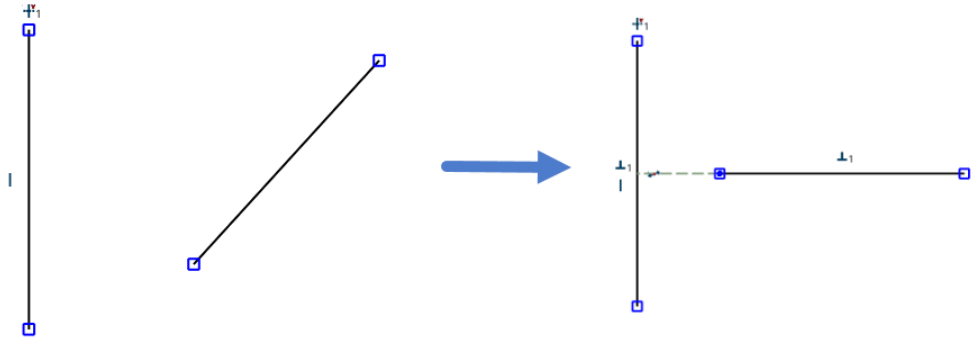
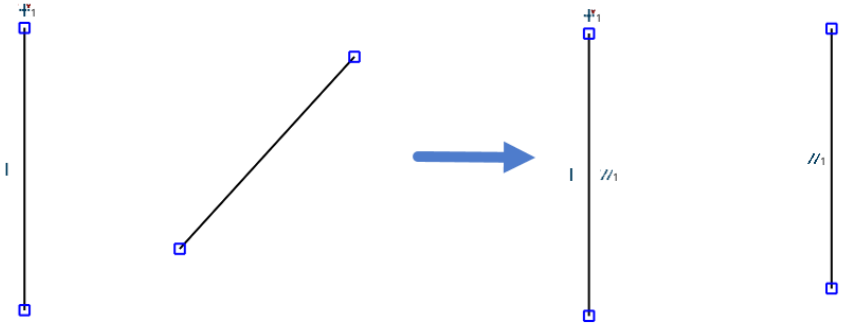
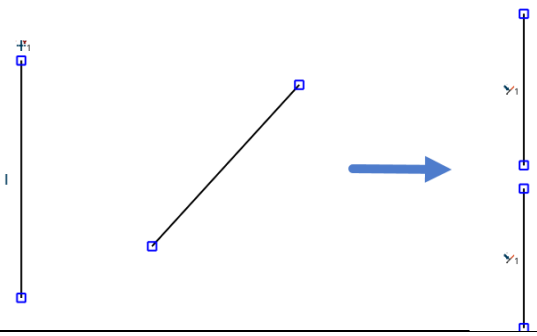
Figure 106 Adding Dimensions

Constraint Ribbon Tab->Constraint->Fix…

Besides the regular ***Add constraints*** command, there are other ways to add constraints. The common procedures are to select a type of constraint first, then pick the objects to be constrained.

Fix	Anchor points or entities 
Points Horizontal	Align two points horizontally 

Points Vertical	Align two points vertically 
Point to Midpoint	Locate a point in the middle of two selected points 
Point to Line/ Curve	Locate a point on the selected curve 
Point to Intersection	Locate a point on the intersection point of two selected curves 
Point Coincident	Locate a point on another point 
HORZ Horizontal	Make a line horizontal 

Vertical	Make a line vertical 
Symmetrical	Mirror two points according to the line of symmetry 
Perpendicular	Make two lines perpendicular to each other 
Parallel	Make two lines parallel to each other 
Collinear	Make two lines lie on the same line 

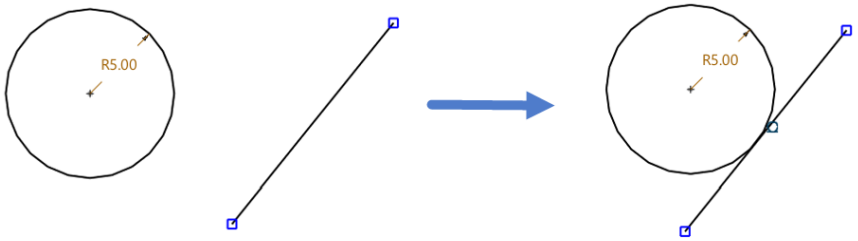
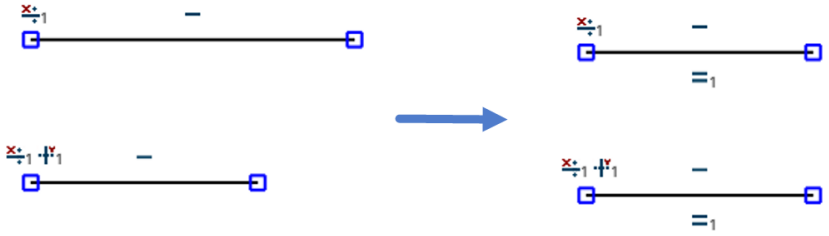
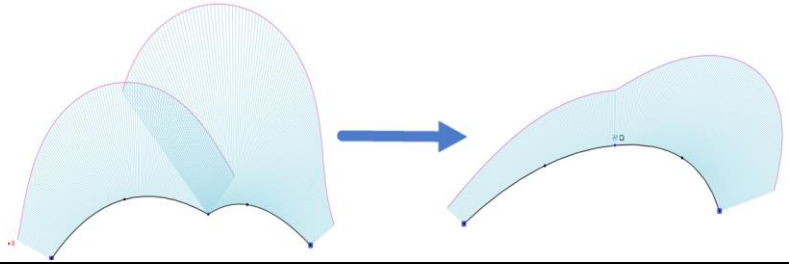
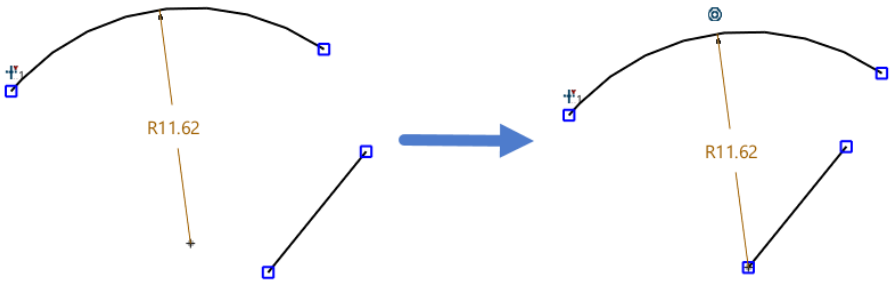
Tangent	Make one curve tangent to another 
Equal Length	Make two curves equally long 
Equal Curvature	Make the curvature of two connected curves equal 
Point to Center	Make a point the center of an arc or circle 

Figure 107 Constraints

Constraint Ribbon Tab->Constraint->Auto Constraints

This command analyzes the selected sketch geometries and automatically adds constraints and dimensions to them.

STEP 01 Select the base point. In the example below, the base point is the datum.

STEP 02 Select the entities that lack constraints or dimensions.

STEP 03 Define the rules of constraints and dimensions, then click ***OK*** to finish.

As the green circles in Figure 108 show, one constraint (Equal Length) and two dimensions are automatically added.

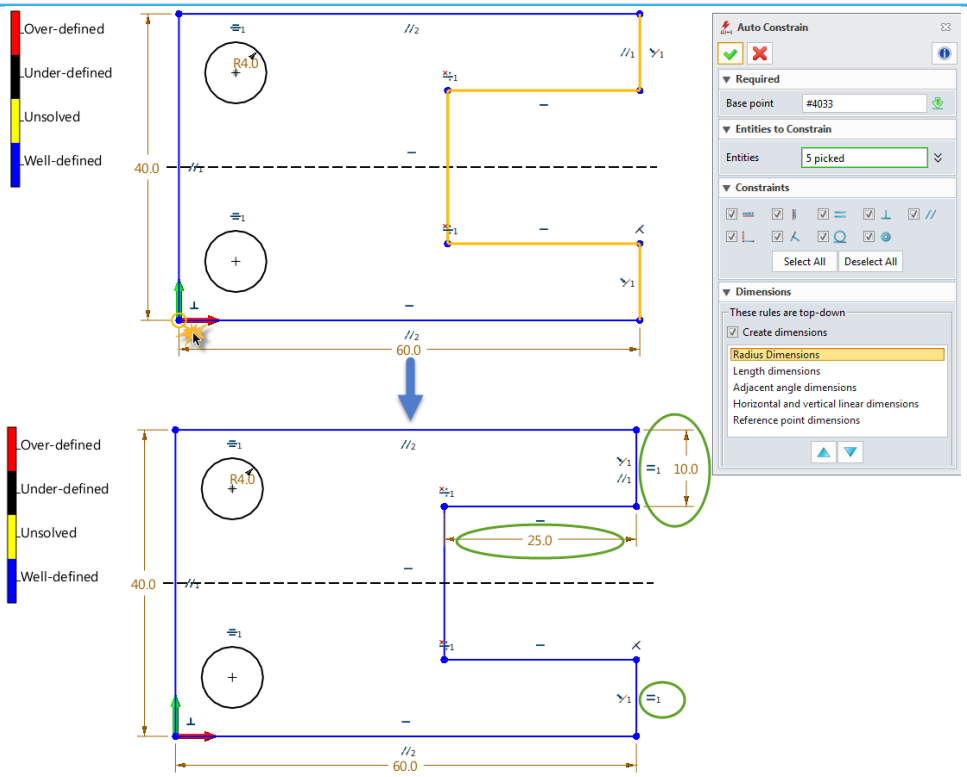


Figure 108 Auto Constraints

7.3 Inquire Constraints and Constraint Status

Constraint Ribbon Tab->Constraint->Show Constraints

You can check all the constraints of the selected entities with this command.

Click the ***Delete*** button to delete the selected constraints or click the ***Delete All*** button to delete all the constraints.

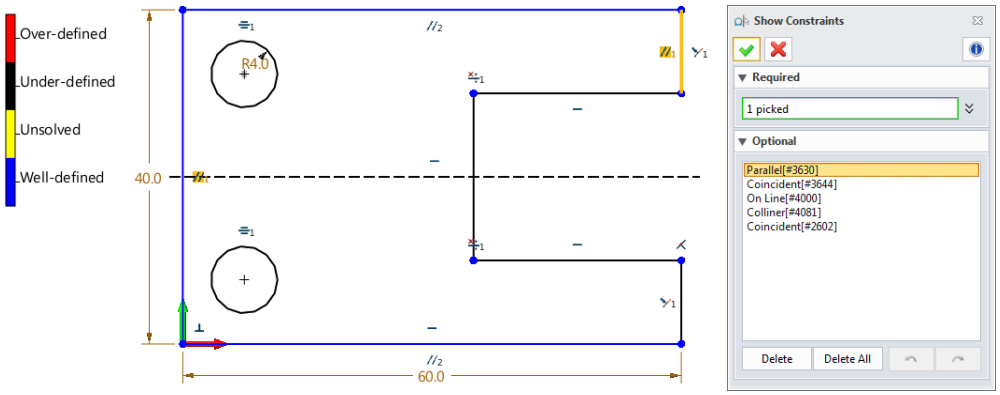


Figure 109 Show Constraints

Constraint Ribbon Tab->Constraint->Constraint Status

This function can help you check the constraint status of the current or specific sketch geometries.

Once the Show constraint status dialogue box is invoked, the general information about constraint status is displayed in the list as a summary.

Also, you can use these  buttons to check the detailed information about constraint status one by one.

If you want to directly check the constraint status of a certain geometry, simply click the  button.

To delete the selected entity, simply click the ***Delete*** button.

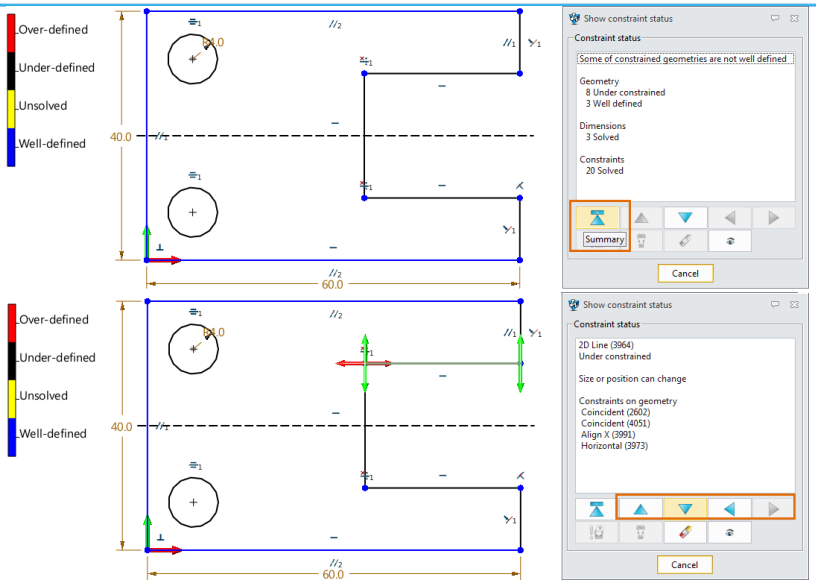


Figure 110 Inquire Constraint Status

8 Dimensions

8.1 Set Dimension Attributes

Tools Ribbon Tab->Attribute->Dimension

Before adding dimensions, please remember to set their attributes: ***Color*** and ***Decimals***.

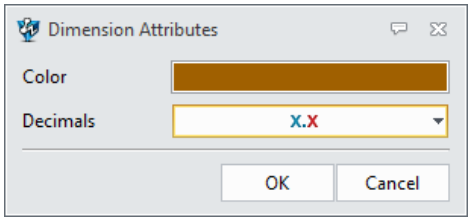


Figure 111 Dimension Attributes

8.2 Quick Dimension

This feature can help you quickly create a dimension when you pick an entity (line/circle/arc) or dimension points. The available types of dimensions are dependent on the selected entity.

Quick Dimension Mode			
	Automatic		Horizontal
	Vertical		Aligned
	2 line angular dimension		Offset
	Horizontal angular dimension		Vertical angular dimension
	Radial		Diameter
	Arc angular		

Figure 112 Quick Dimension Modes

8.3 Add Linear Dimensions

Method 1: Constraint Ribbon Tab->Dimension->Quick Dimension

STEP 01 Select the geometry, by picking a line or two points.

STEP 02 Locate the dimension text.

STEP 03 Enter the value of the new dimension and click ***OK***.

Note: The default Quick Dimension mode is ***Automatic***. You can always choose another linear dimension mode to suit your needs.

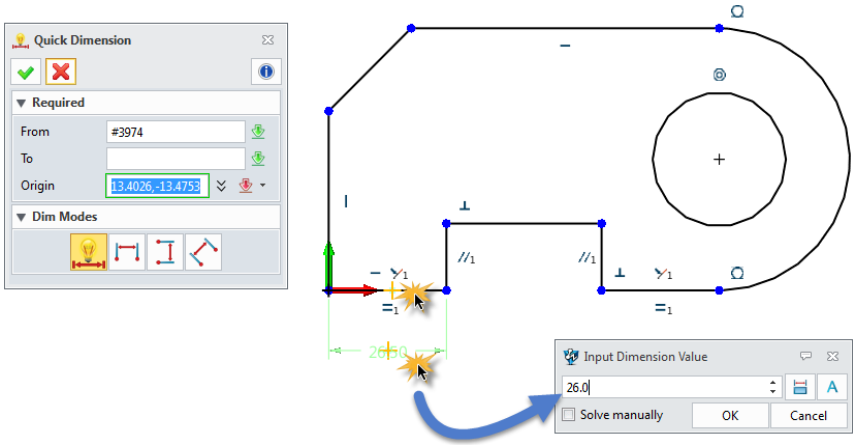


Figure 113 Quick Dimension—Picking a Line

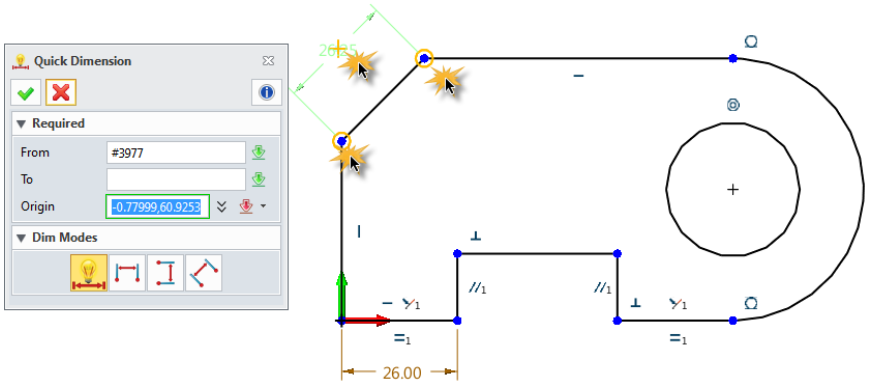


Figure 114 Quick Dimension—Picking Two Points

Method 2: Constraint Ribbon Tab->Dimension->Linear

STEP 01 Select one type of linear dimension, for example, Vertical.

STEP 02 Select two points.

STEP 03 Locate the dimension text.

STEP 04 Enter the value of the new dimension and click ***OK***.

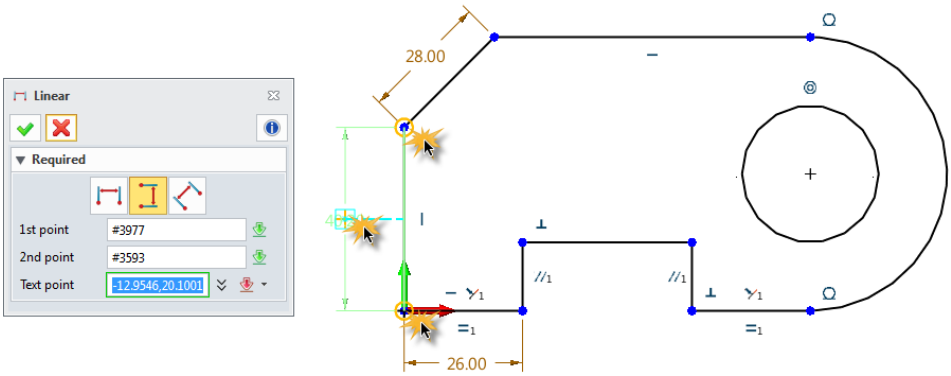


Figure 115 Adding Linear Dimensions

8.4 Add Linear Offset Dimensions

Constraint Ribbon Tab->Dimension->Linear Offset

You can create a linear offset dimension between two parallel lines or a linear projected distance dimension between a point and a line with this command.

STEP 01 Select a type of linear offset: ***Offset***/***Projected Distance***.

STEP 02 Select two parallel lines, or one line and one point.

STEP 03 Locate the dimension text.

STEP 04 Enter the value of the new dimension and click ***OK***.

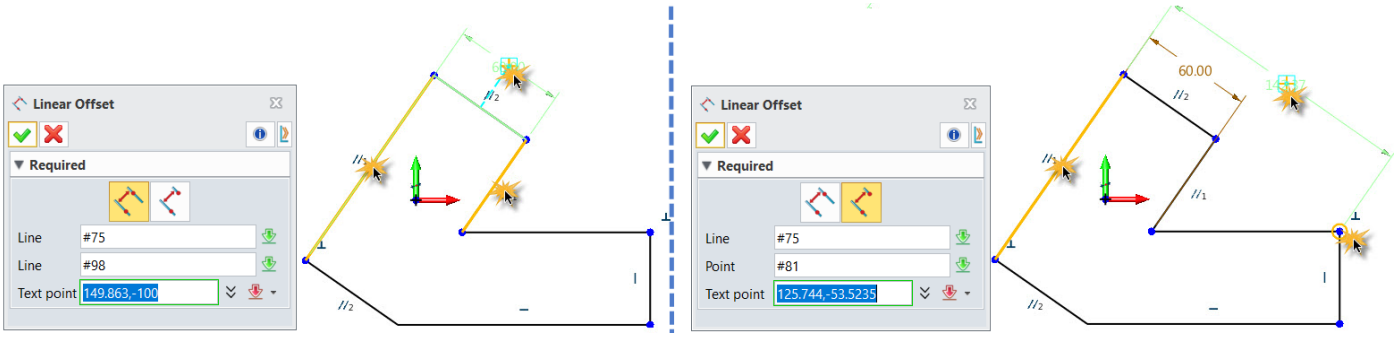


Figure 116 Linear Offset Dimension

8.5 Angular Dimension

Method 1: Constraint Ribbon Tab->Dimension->Quick Dimension

If you pick two unparallel lines, the default Quick Dimension mode will be ***Angular***.

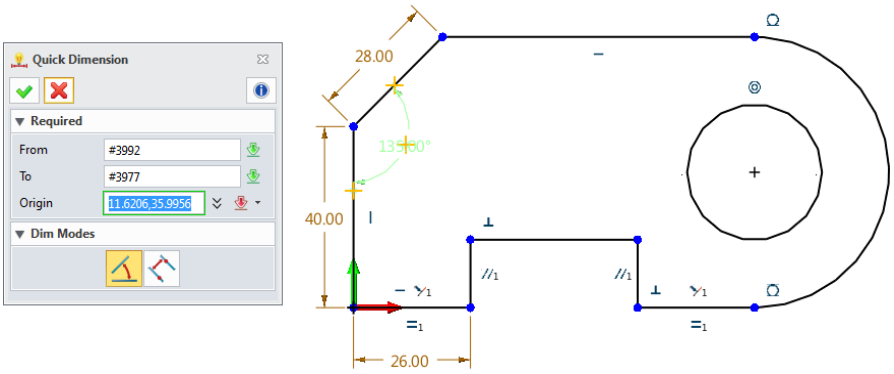


Figure 117 Quick Dimension—Picking Two Unparallel Lines

Method 2: Constraint Ribbon Tab->Dimension->Angular

STEP 01 Select one type of angular dimension (Two curves/Horizontal/Vertical/Arc).

STEP 02 Pick the necessary entity following the instructions displayed in the lower left corner.

STEP 03 Locate the dimension text.

STEP 04 Enter the value of the new dimension and click ***OK***.

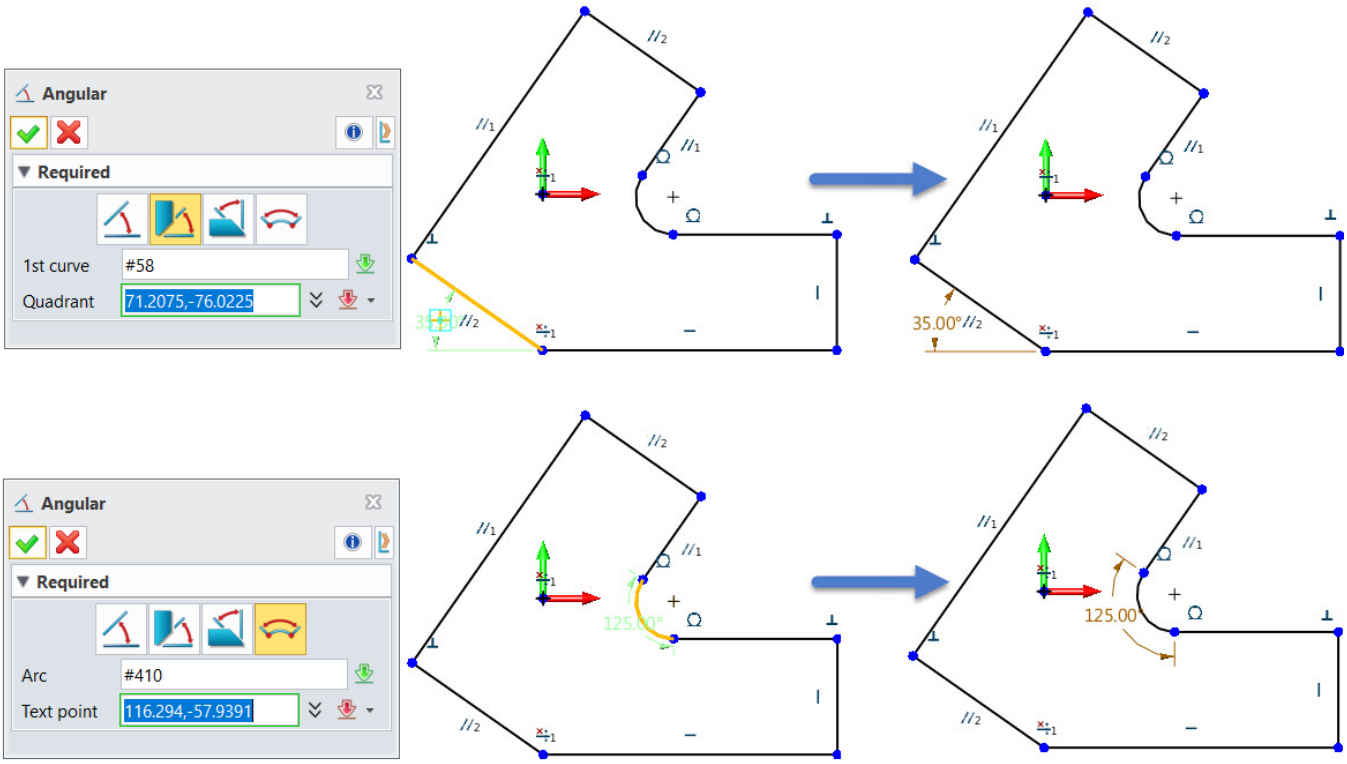


Figure 118 Angular Dimensions

8.6 Add Radial/Diametric Dimensions

Method 1: Constraint Ribbon Tab->Dimension->Quick Dimension

STEP 01 Pick a circle or arc.

STEP 02 Select a dimension mode.

Note: If a circle is picked, these two  modes are available. If an arc is picked, these three  modes are available.

STEP 03 Locate the dimension text.

STEP 04 Enter the value of the new dimension and click ***OK***.

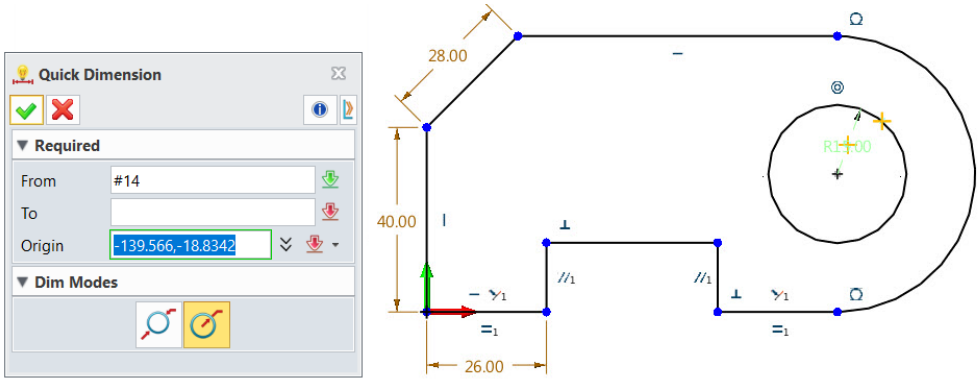


Figure 119 Quick Dimension—Picking a Circle/Arc

Method 2: Constraint Ribbon Tab->Dimension->Radial/Diametric

Select an arc or circle to create a radial or diametric dimension.

8.7 Add Arc Length Dimensions

Method 1: Constraint Ribbon Tab->Dimension->Quick Dimension

Select an arc, and then select the ***Arc Length*** mode to create the arc length dimension.

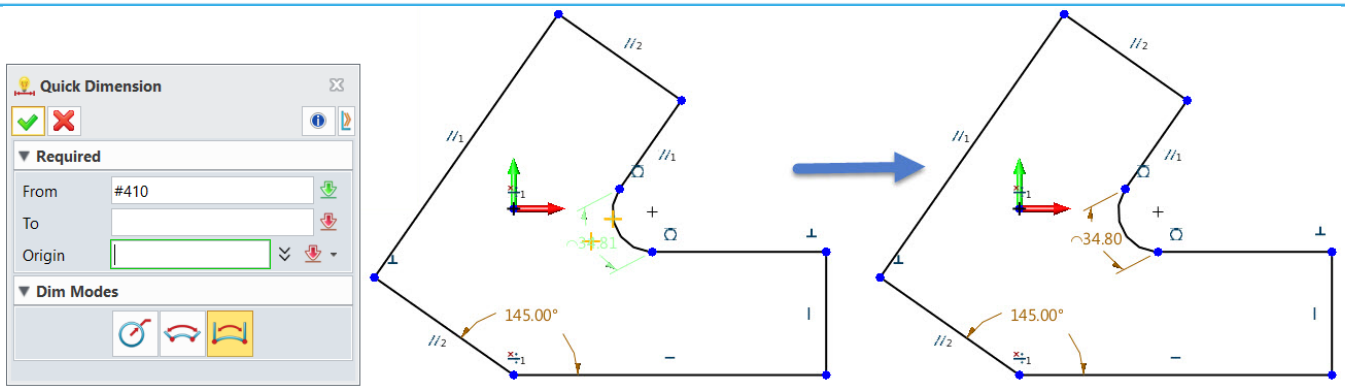


Figure 120 Quick Dimension—Picking an Arc

Method 2: Constraint Ribbon Tab->Dimension->Arc Length

Select an arc and locate the dimension text to create the arc length dimension.

8.8 Modify Dimension Values

The ***Modify Value*** command is to help you change the value of the dimension and redefine the sketch geometry. There are three ways to access this function.

Method 1: Constraint Ribbon Tab->Dimension->Modify Value

STEP 01 Select the dimension.

STEP 02 Enter a new dimension value and click ***OK***.

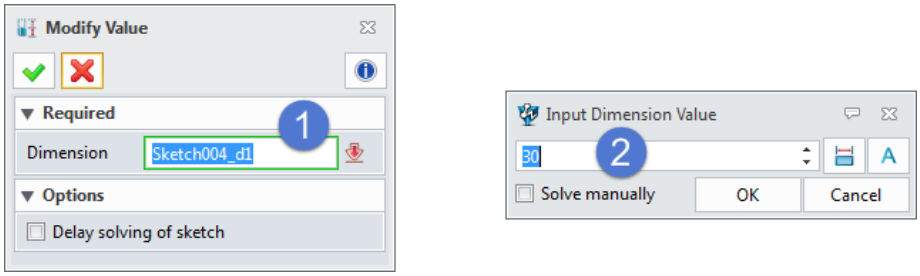


Figure 121 Modify Dimension Values—Method 1

Method 2: Right-click on the dimension, then click the Modify Value icon.

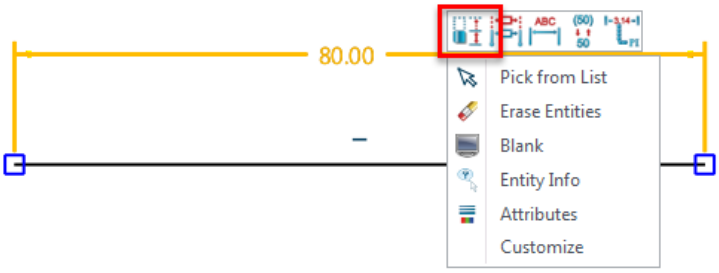


Figure 122 Modify Dimension Values—Method 2

Method 3: Double-click on the dimension to modify its value.

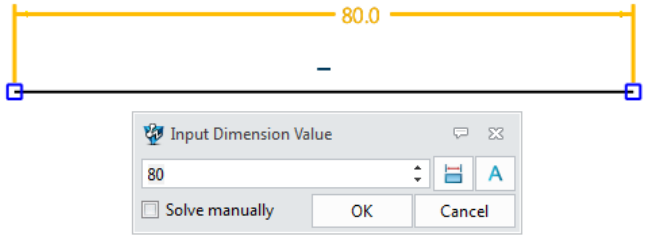


Figure 123 Modify Dimension Values—Method 3

Delayed updates of dimensions?

When modifying the dimension value, if the ***Delay solving of sketch*** or ***Solve manually*** option is checked, the sketch geometry will not be updated immediately. Also, the modified dimension value will appear in square brackets [ ].

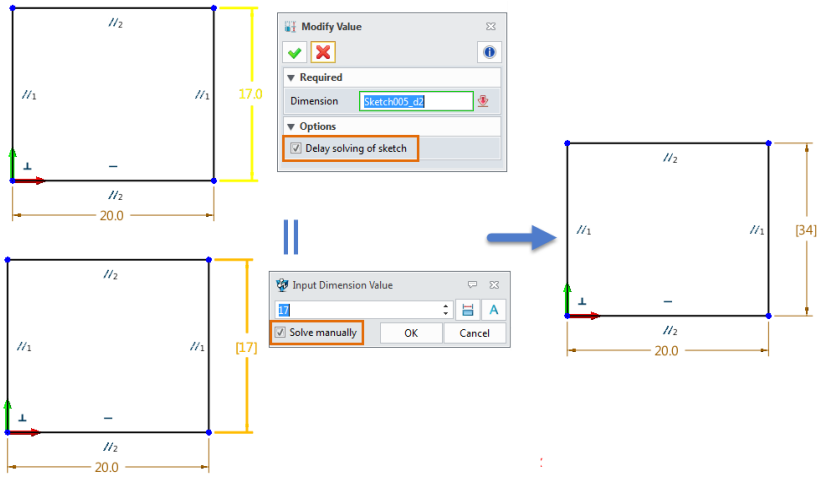


Figure 124 Delayed Updates of Dimension

The sketch will be updated only when a further ***Solve Current Sketch (Automatically/Manually)*** command is executed.

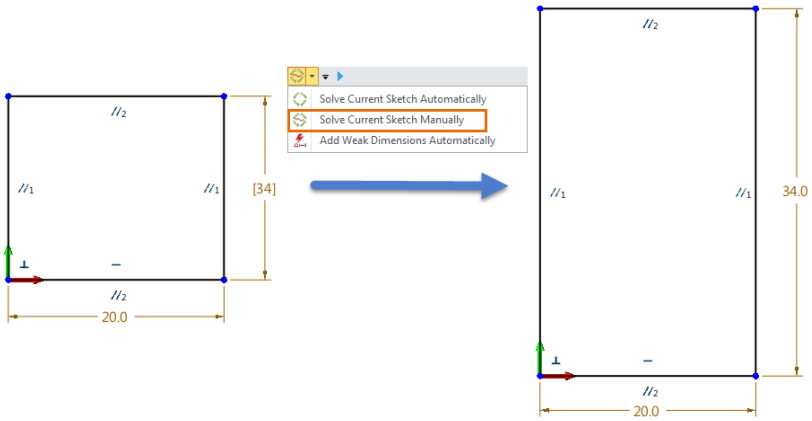


Figure 125 Update the Sketch Manually

Note: If you need to modify many dimensions values in a sketch, it is recommended that you delay updates of dimensions so that the sketch geometries are less likely to become distorted.

9 Check the Sketch

9.1 Curve Connectivity

Inquire Ribbon Tab->Constraint->Curve Connectivity

By default, the information about the curve connectivity of the visible geometries will appear in the Output window. You can also pick certain geometries and check their curve connectivity.

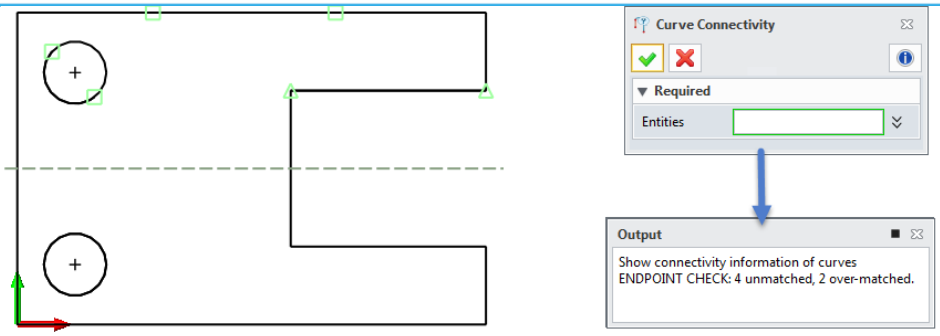


Figure 126 Curve Connectivity

9.2 Check Overlaps

Inquire Ribbon Tab->Sketch Doctor->Overlap

STEP 01 Click this command and all the overlaps between visible geometries will appear in the Overlap Inquiry dialogue box.

STEP 02 Pick a geometry from the list and it will be highlighted in the working area.

STEP 03 Click the ***Delete*** button to delete the unneeded geometry.

STEP 04 Click the ***Refresh*** button to inquire the overlapping geometries again.

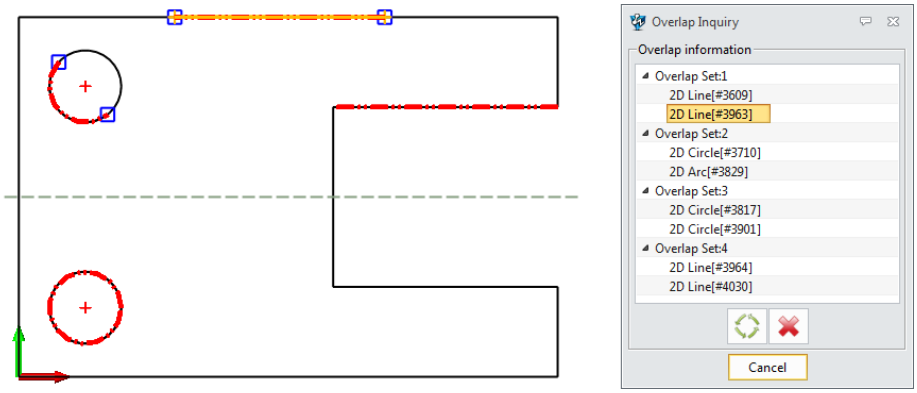


Figure 127 Overlap Inquiry

Note: During the analysis, blank geometries will not be considered.

9.3 Curvature Plot

Constraint Ribbon Tab->Constraint->Curvature Plot

This command allows you to inquire the curvature plots, which tell you the curvature of curves at a set of points.

STEP 01 Select the curve, then the curvature plots will appear.

STEP 02 If you want to check the inflection points and peak points, simply check the corresponding ***Show*** options.

STEP 03 If you check ***Show minimum radius***, the value of the min radius will appear.

STEP 04 Check ***Create points*** and click ***OK*** to create new points at the inflection/peak points.

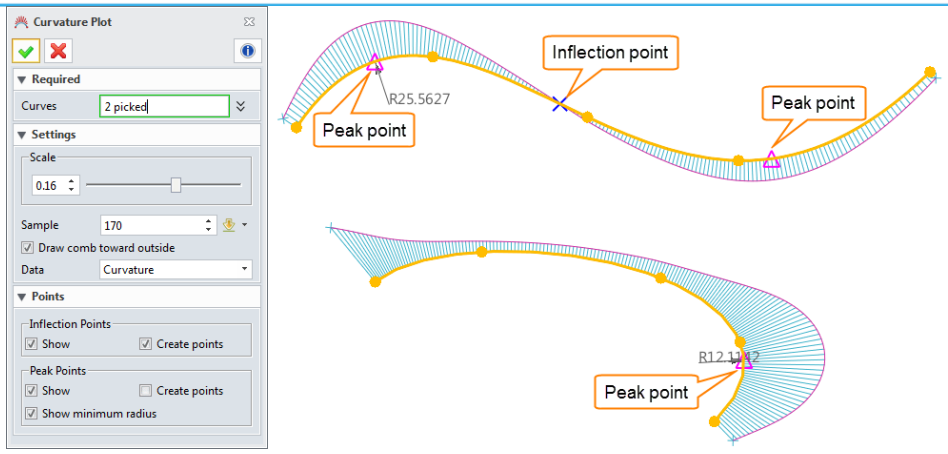


Figure 128 Curvature Plots

10 Cases—2D Sketch

10.1 General Process

STEP 01 Draw some necessary construction geometries and define them with constraints or dimensions.

STEP 02 Draw the outline of the sketch with drawing tools like ***Line***, ***Circle***, and ***Arc***.

Note: Some dimensions and constraints will be automatically created based on the default settings.

STEP 03 Add the necessary constraints, such as ***Tangent*** and ***Equal Length***.

STEP 04 Add dimensions to the sketch or modify their value.

Note: The ***Delay dimension update*** function is available for modification of dimensions. Please remember to click the icon again to update the sketch.

10.2 Case 1

STEP 01 Set the dimension attributes of the sketch in the Dimension Attributes dialogue box.

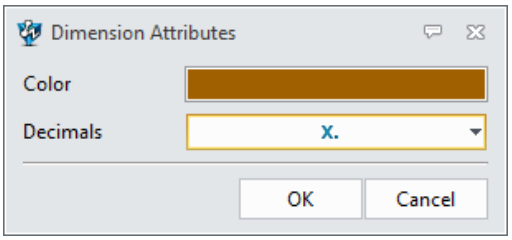


Figure 129 Set Dimension Attributes

STEP 02 Draw a horizontal line that passes through the datum with ***Line*** and convert it into a construction line.

STEP 03 Create a vertical construction line and set the distance between it and the datum as 150 mm.

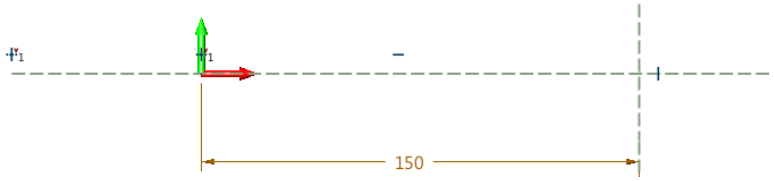


Figure 130 Construction Geometries

STEP 04 Draw two circles whose centers are the datum. Their radii are 25 mm and 45 mm respectively.

Draw another two circles whose centers are the intersection point of the horizontal and vertical construction lines. Their radii are 7.5 mm and 20 mm respectively.

STEP 05 Add constraints to the centers and the intersection point with the ***Point to Intersection*** feature. Turn on ***Constraint status color***, and then check the sketch. The result is as Figure 131 shows.

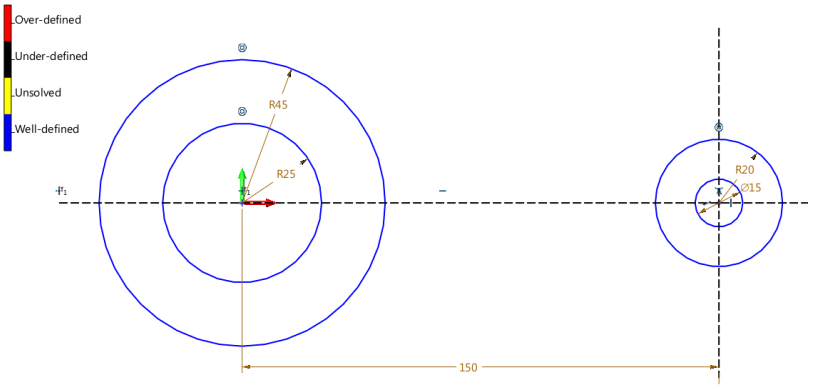


Figure 131 Drawing the Base Circles

STEP 06 Draw two horizontal lines that are tangent to the circle with a radius of 20 mm. Then, trim the sketch with ***One Touch Trim***. The result should be as Figure 132 shows.

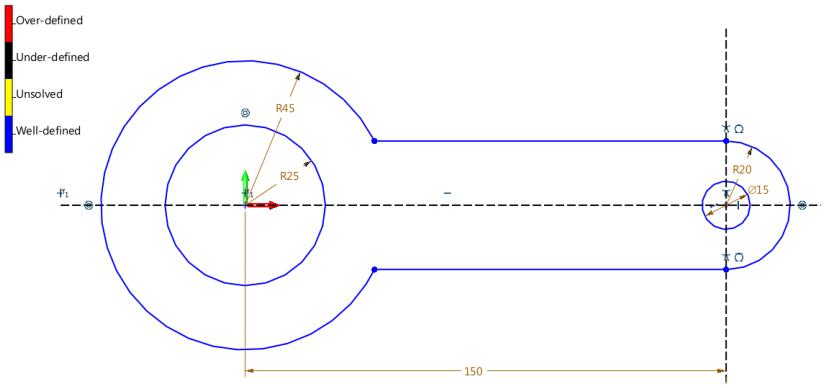


Figure 132 Add Two Horizontal Lines

STEP 07 Fillet the horizontal line and the circle whose radius is 45 mm at a value of 30 mm.

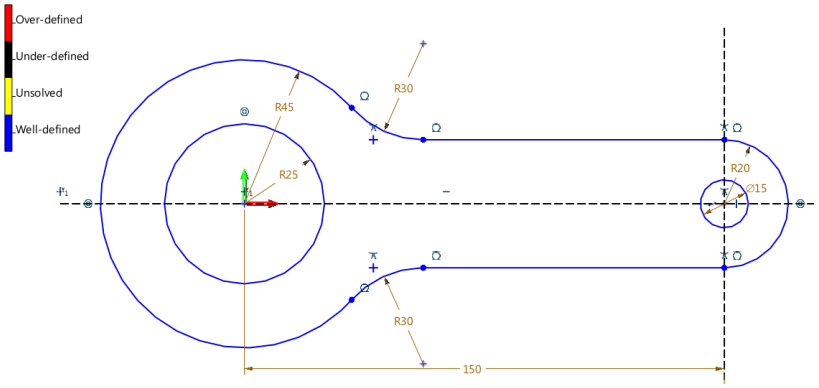


Figure 133 Filleting the Sketch

STEP 08 Create a line that is 30 degrees from the horizontal line, as shown in Figure 134. Then, add the angular dimension to it and convert it into a construction line.

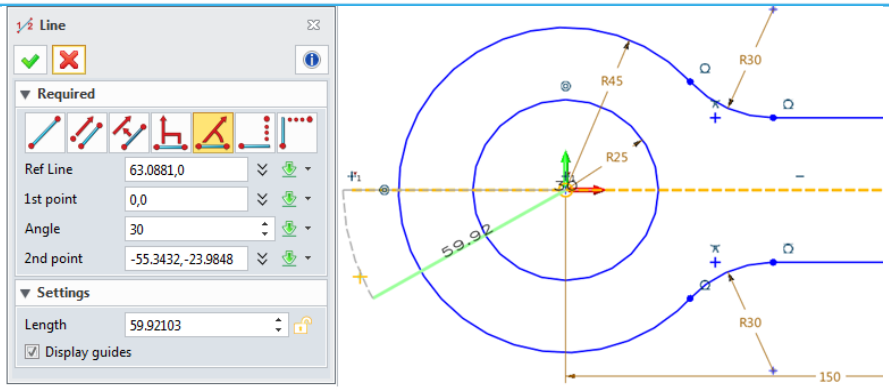


Figure 134 Creating a Line

STEP 09 Create two lines parallel to the line generated in Step 8 with ***Offset***. Then, add parallel constraints and offset dimensions to them, as shown in Figure 135.

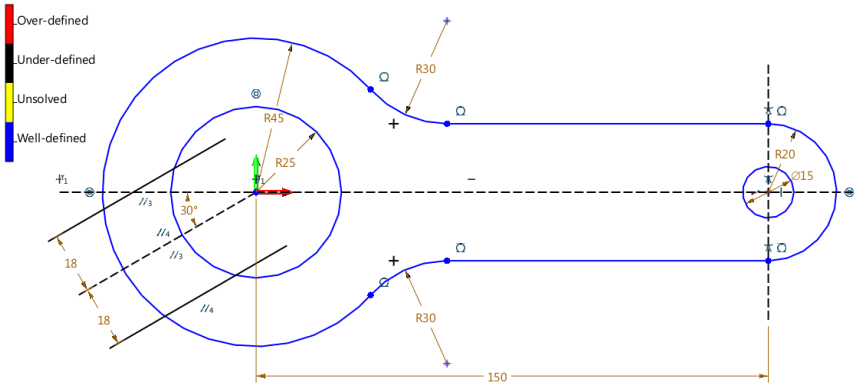


Figure 135 Creating Two Parallel Lines

STEP 10 Trim or delete the unneeded curves.

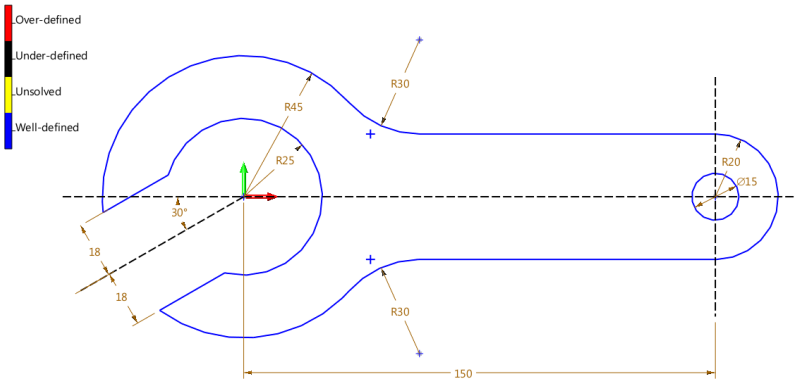


Figure 136 Trimming the Sketch

STEP 11 Fillet the tips of the sketch. The well-defined result is as Figure 137 shows.

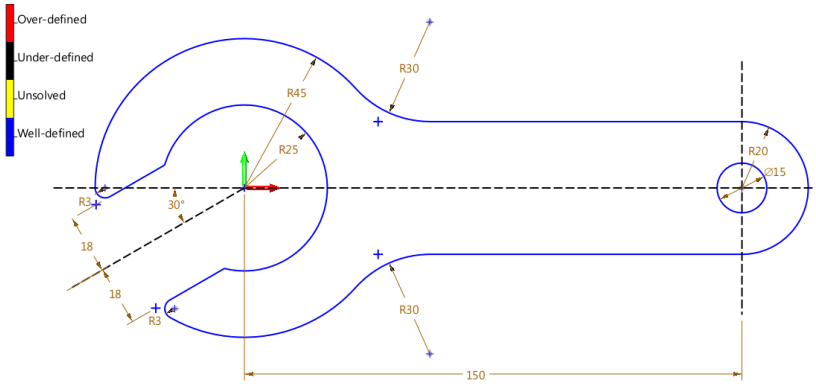


Figure 137 Final Sketch

10.3 Case 2

STEP 01 Draw three construction lines, as shown in Figure 138.

STEP 02 Draw a rectangle with a width of 80 mm and a height of 50mm as well as a circle whose center is the datum and radius is 15 mm.

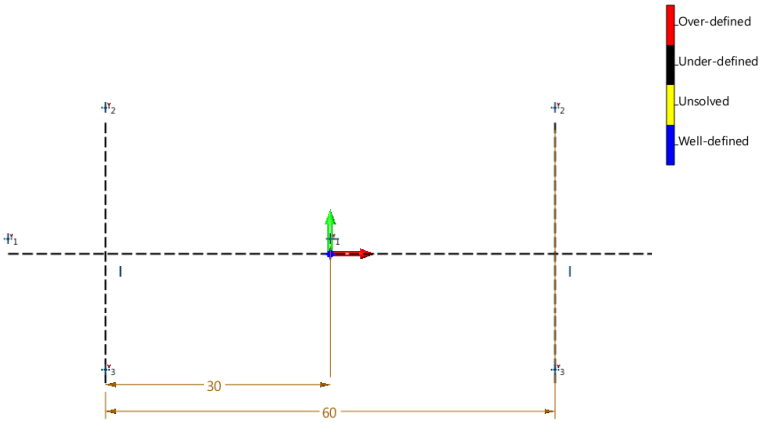


Figure 138 Construction Lines

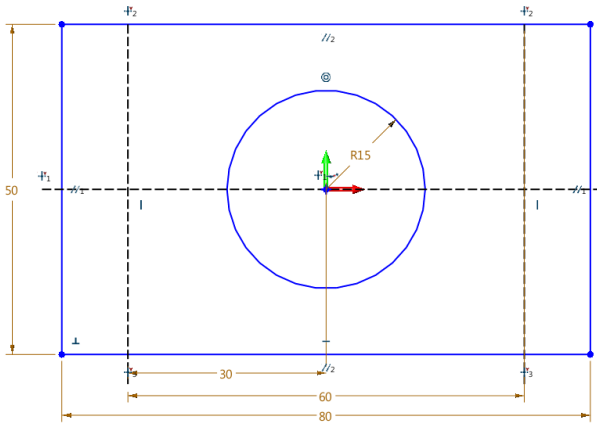


Figure 139 The Base Geometries

STEP 03 Fillet the four corners of the rectangle with the ***Fillet Chain*** feature at a value of 10 mm.

STEP 04 Create a circle with a radius of 5 mm, then ***Mirror*** twice to get the below result.

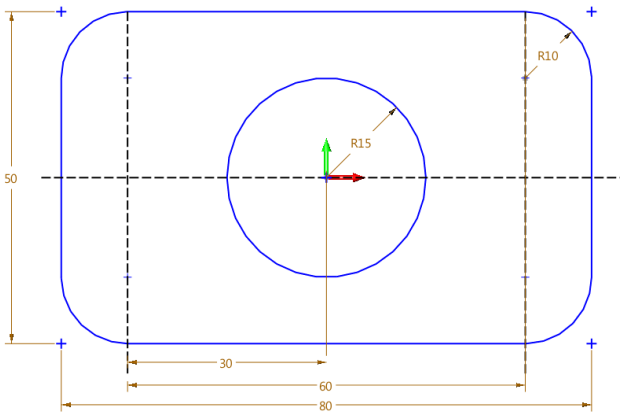


Figure 140 Fillet

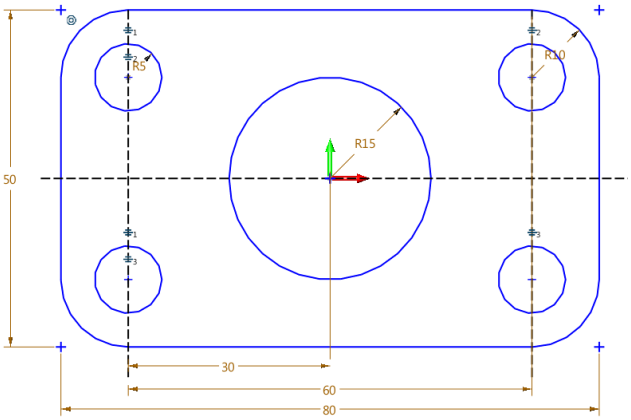


Figure 141 Mirror

STEP 05 Create a slot with ***Slot***. The intersection points of construction lines are the centers of the arcs with a radius of 5 mm.

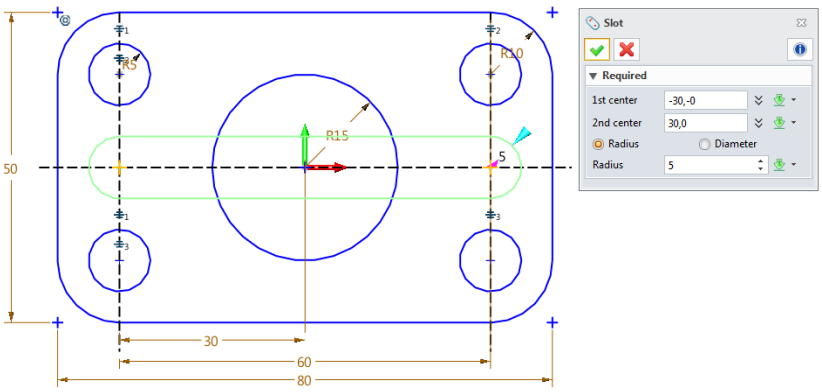


Figure 142 Slot

Note: You need to add the ***Points to Intersection*** constraints manually.

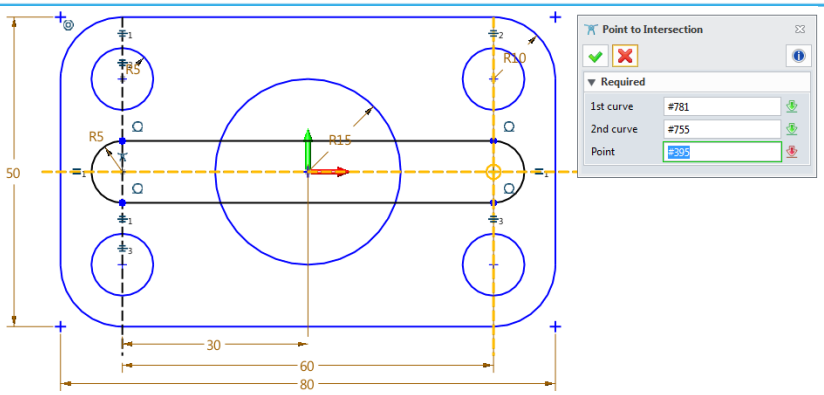


Figure 143 Manually Add Constraints

STEP 06 Create a rectangle whose center is the datum and width is 10 mm. And set the dimensions (5 mm and 10 mm) as shown in Figure 144. To get a clear sketch, you can turn off the previously added constraints and blank some dimensions.

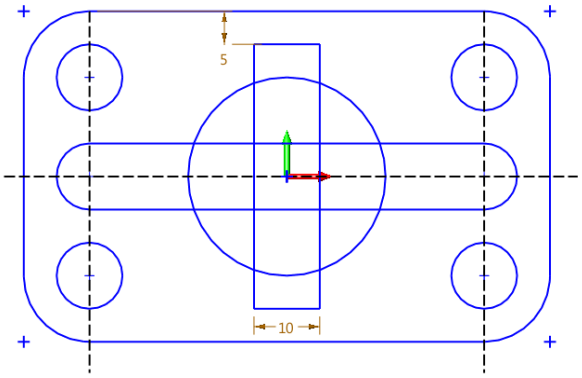


Figure 144 Rectangle

STEP 07 Trim the unneeded geometries with ***Power Trim*** to get the final well-defined result.

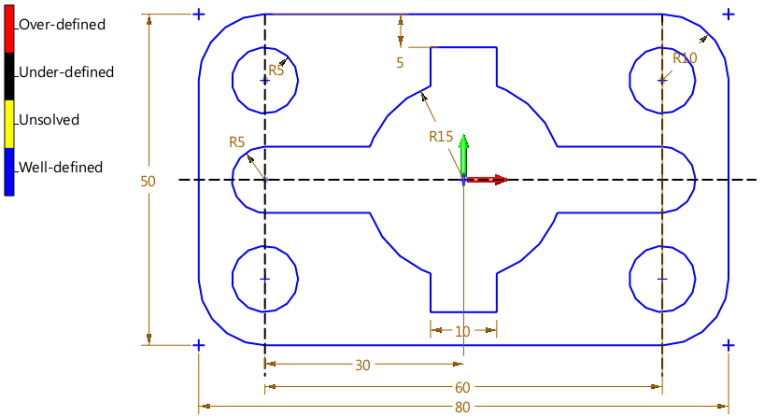
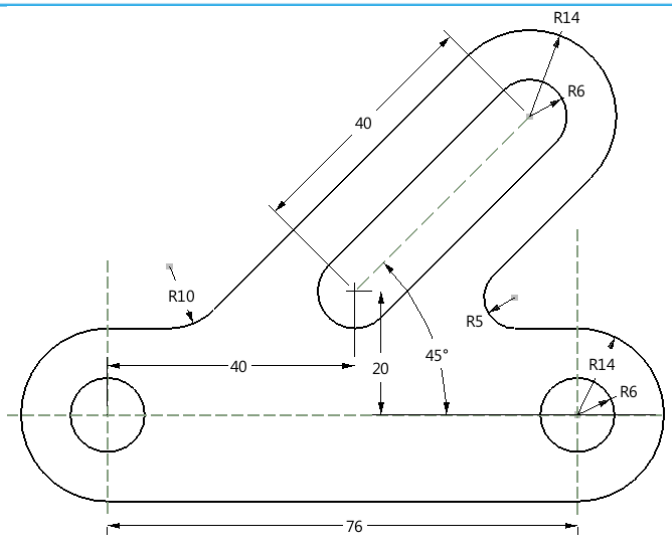


Figure 145 Final Sketch

11 Exercises

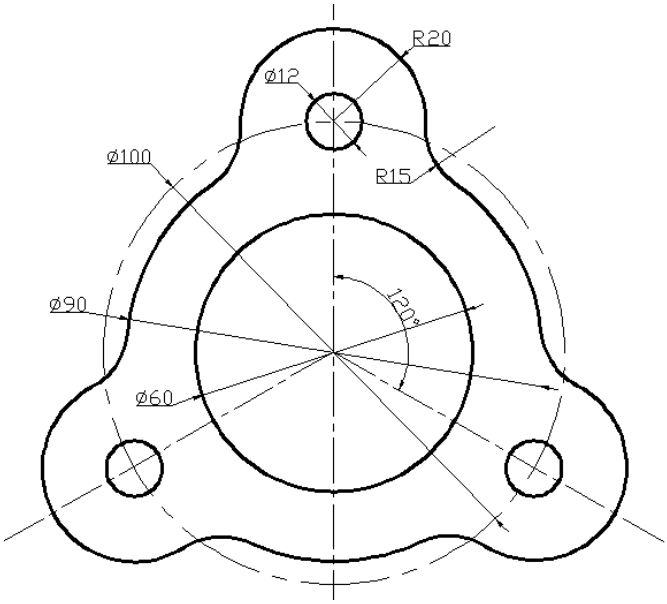
Exercise 1:

Note: Recommended commands: ***Slot*** and ***Fillet***.



Exercise 2:

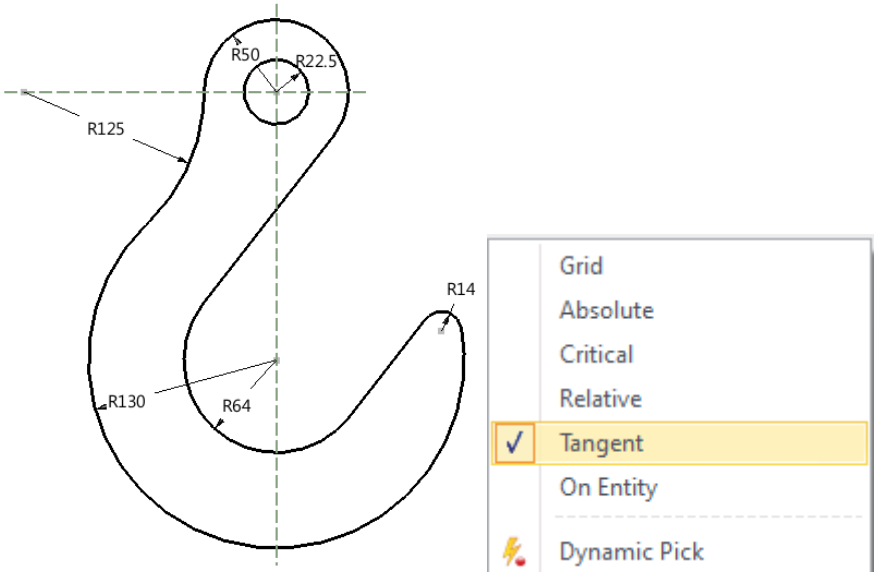
Note: Recommended commands: ***Circle, Fillet, Pattern*** and ***Power Trim***.



Exercise 3:

Notes: 1. Recommended commands: ***Line, Circle*** and ***Arc***.

2. When drawing the arc, set the ***Snap*** method as ***Tangent***.



Exercise 4:

Note: Figure out a way to quickly draw this sketch.