

SOLIDWORKS COMMANDS CHEAT SHEET

35 Essential Commands Every User Should Know

SOLIDWORKS has hundreds of tools and commands.

But you don't need to know all of them to become productive.

Most everyday SOLIDWORKS work is built around a relatively small group of commands that you'll use again and again.

This cheat sheet covers **50 essential SOLIDWORKS commands** across:

Sketching → Part Design → Assemblies → Drawings

Use it as a quick reference while learning, practicing, or working on your own projects.

1. SKETCHING

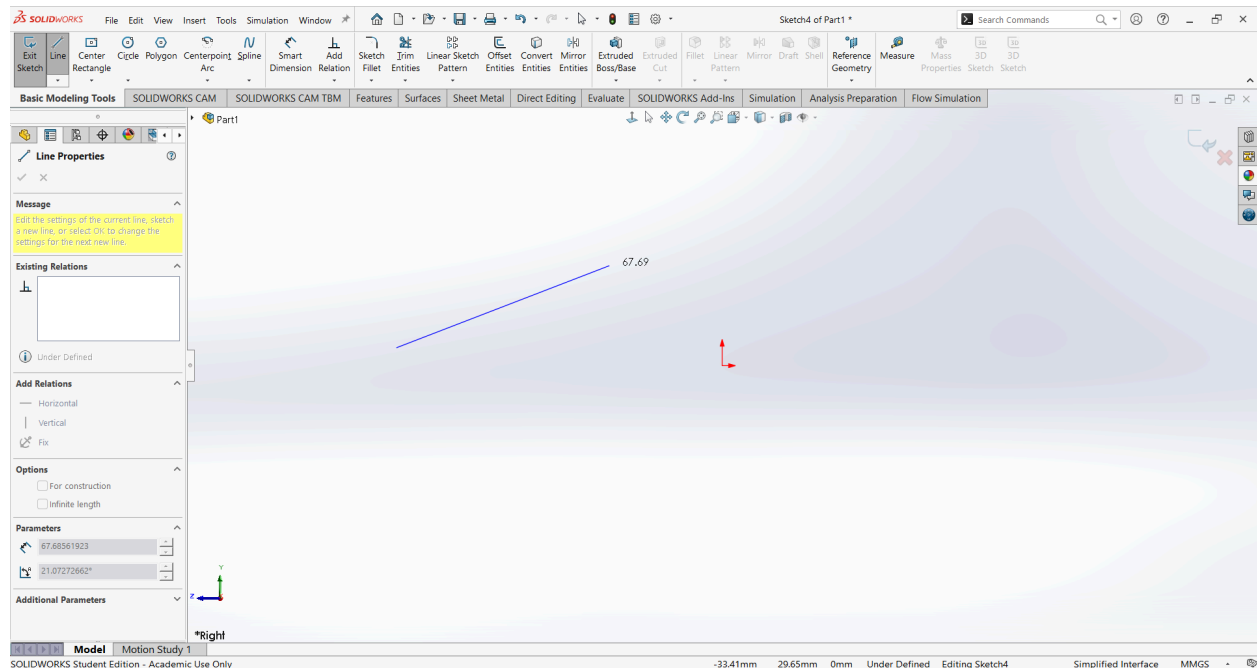
1. Line

What it does: Creates straight sketch entities.

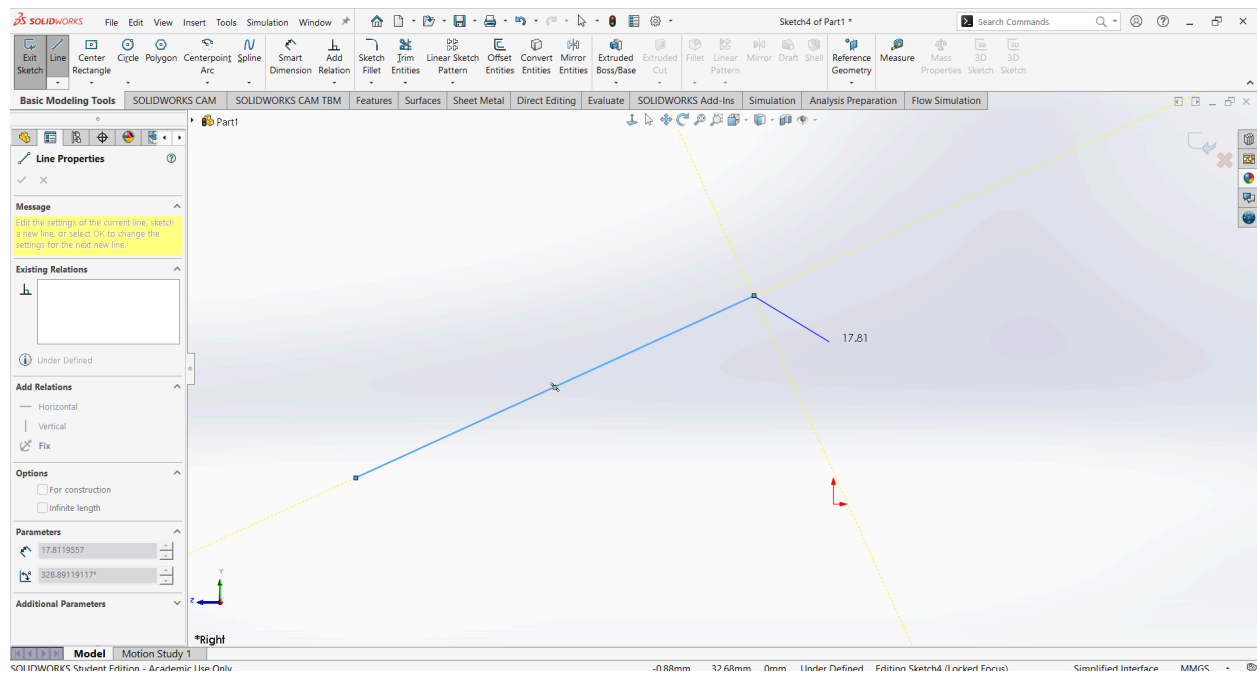
Use it for: Profiles, boundaries, construction geometry, and most basic sketches.

How it works:

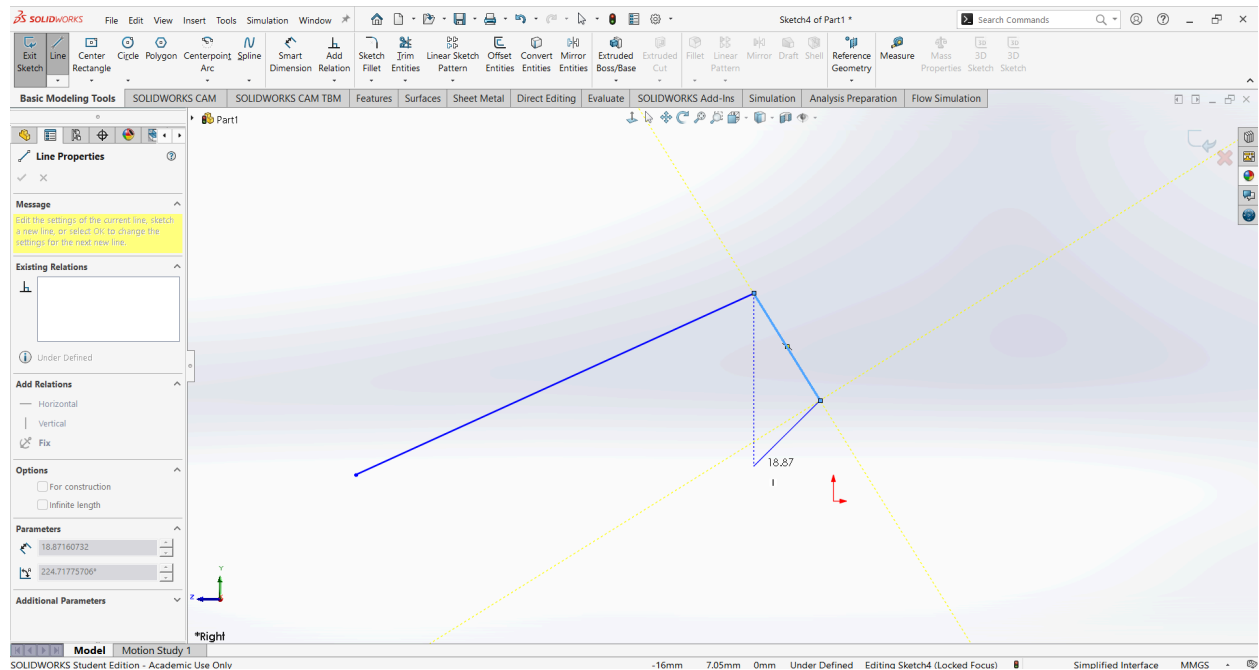
Select **Line**, then define the first point of the line by clicking anywhere in the working area.



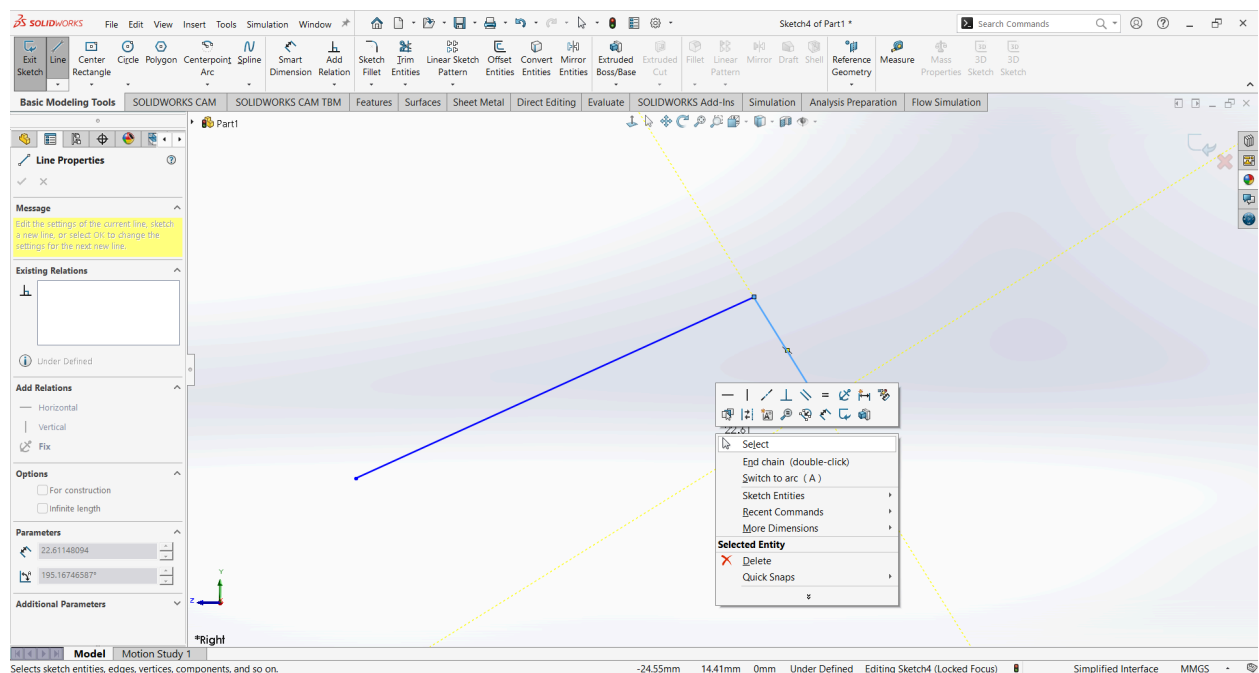
Move the mouse to define the direction and length of the line. Then click again to place the second point of the line.



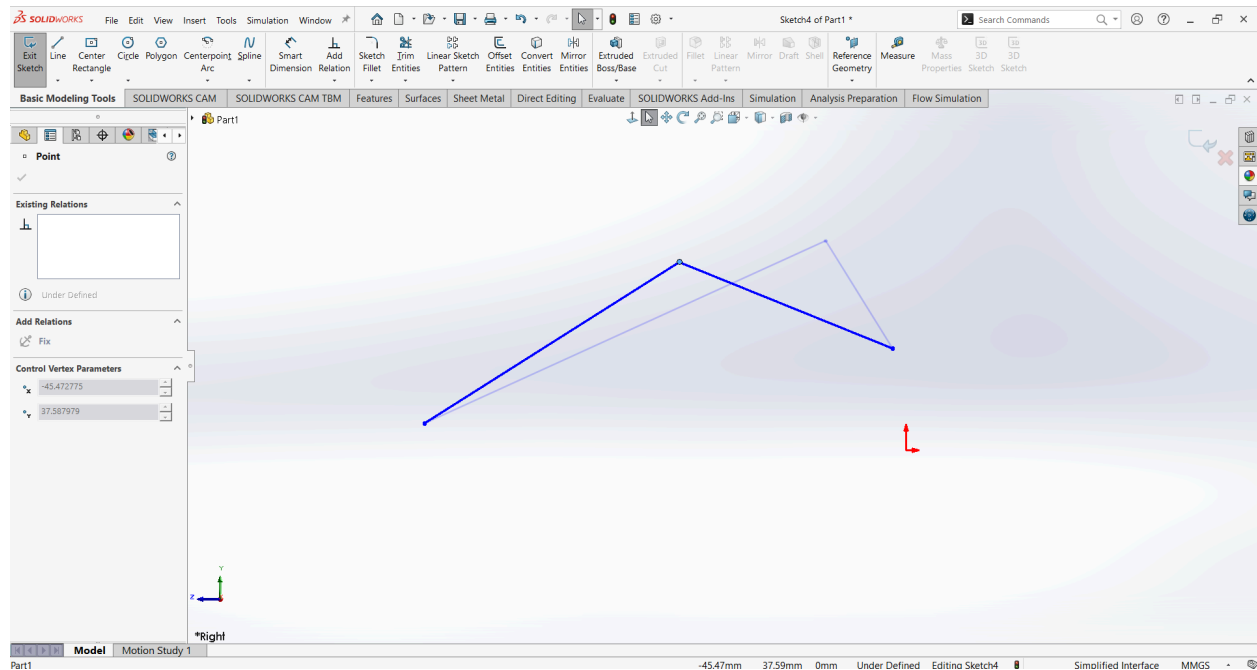
By defining additional points, you can continue creating connected lines. Each new click creates the next point and adds another line segment to the sketch.



To stop creating lines, right-click in the working area and select **Select**. This will exit the Line tool and return the cursor to selection mode.



If you click on any line point, hold the mouse button, and move the mouse, the point will move with it. This allows you to adjust the shape of the sketch by moving its points.



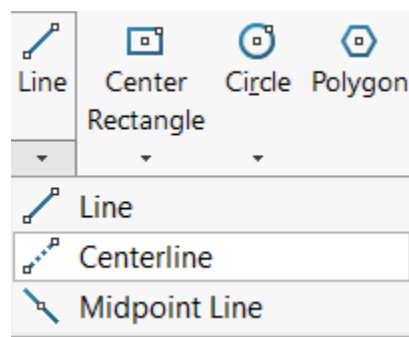
2. Centerline

What it does: Creates construction geometry that can be used as a reference.

Use it for: Symmetry, revolved features, dimensions, and positioning geometry.

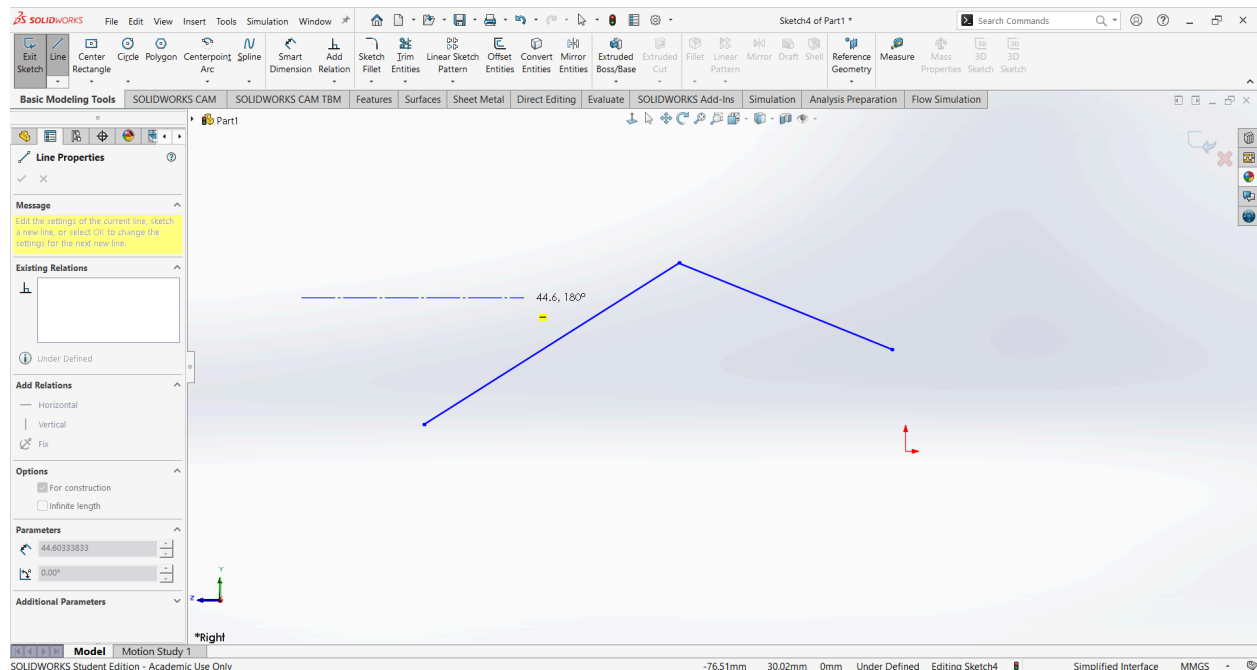
How it works:

The second line option is **Centerline**. Click **Centerline** to start creating a construction line.



Click once to define the first point of the centerline.

Then move the mouse and click again to define the second point. When the line is horizontal, a yellow horizontal relation icon will appear. This means SOLIDWORKS recognizes the line as horizontal.

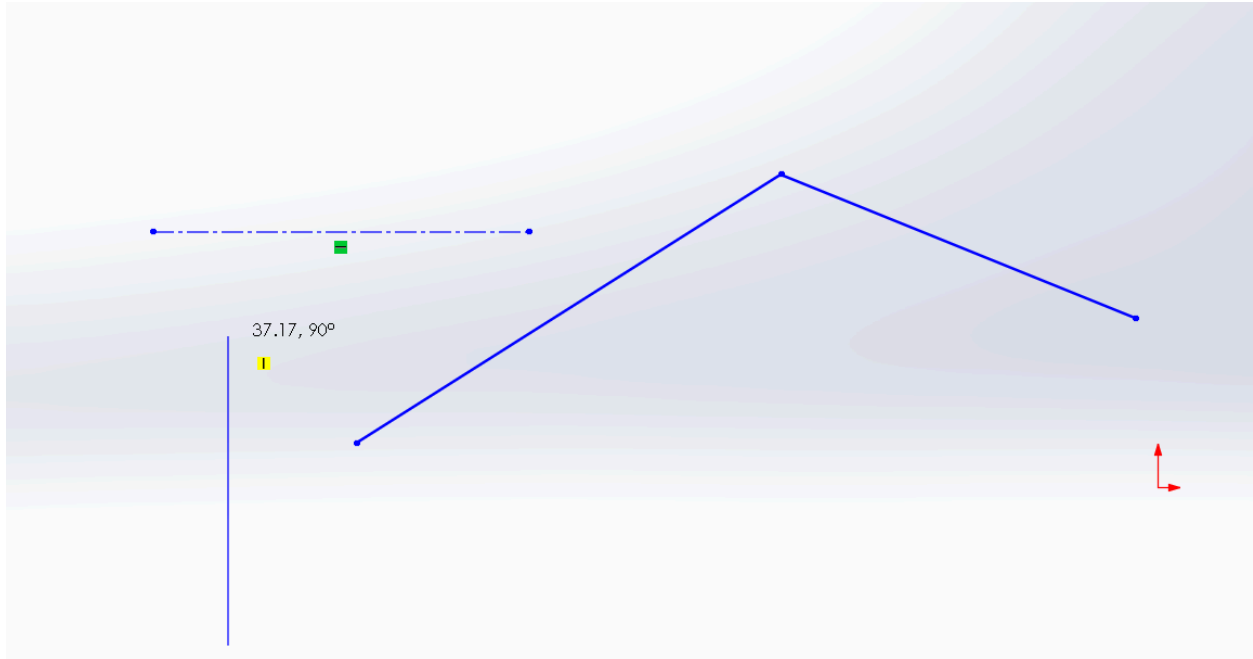


After creating the line, you will see a green horizontal relation icon. This means the line has a **Horizontal** relation applied to it. Because of this relation, you can change the length of the line, but you cannot change its angle unless you remove the horizontal relation.

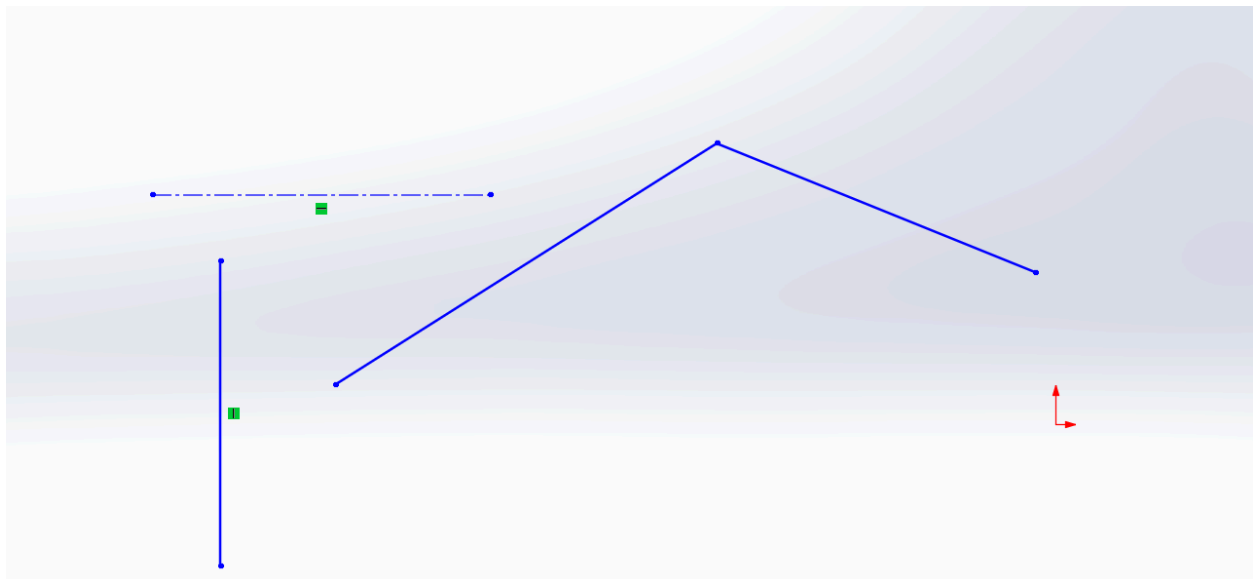


In the same way, you can create a vertical line.

When the line is vertical, a vertical relation icon will appear next to it. This means the line has a **Vertical** relation applied and will remain vertical.



After you finish creating the line, a green vertical relation icon will appear next to it. This means the line has a **Vertical** relation applied and will remain vertical unless the relation is removed.



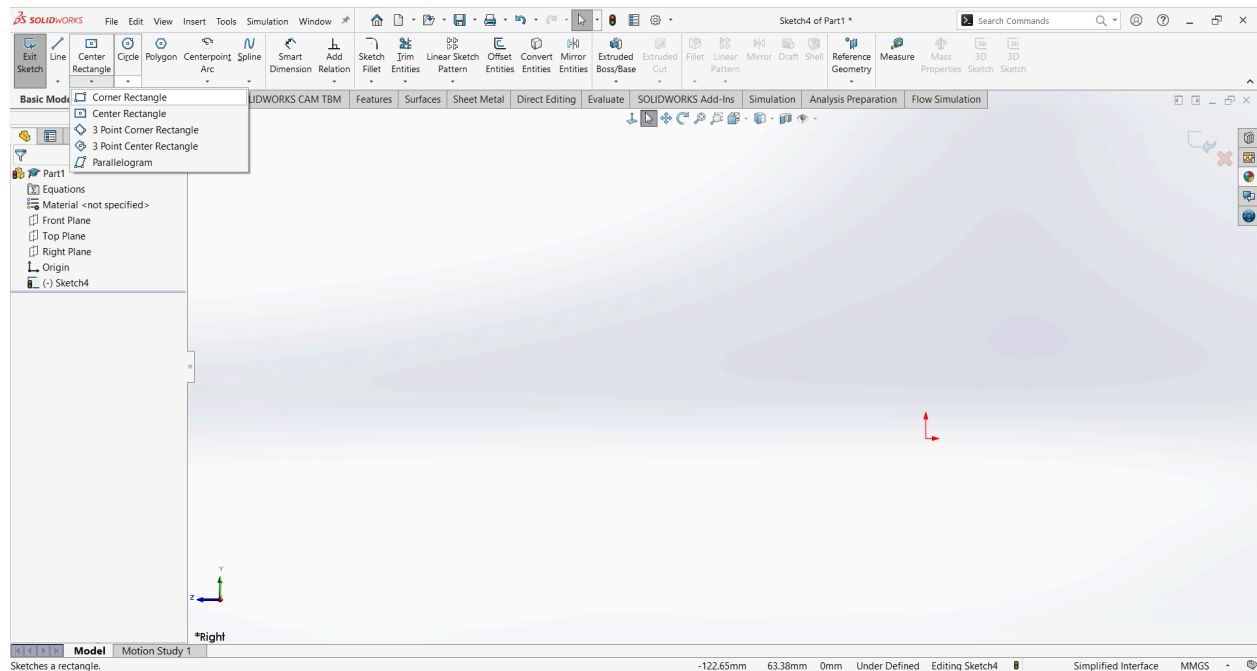
3. Rectangle

What it does: Creates rectangular sketch geometry.

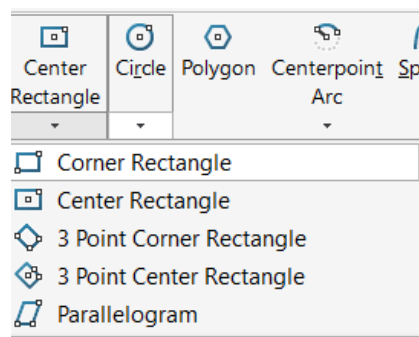
Use it for: Plates, blocks, brackets, and other rectangular profiles.

How it works:

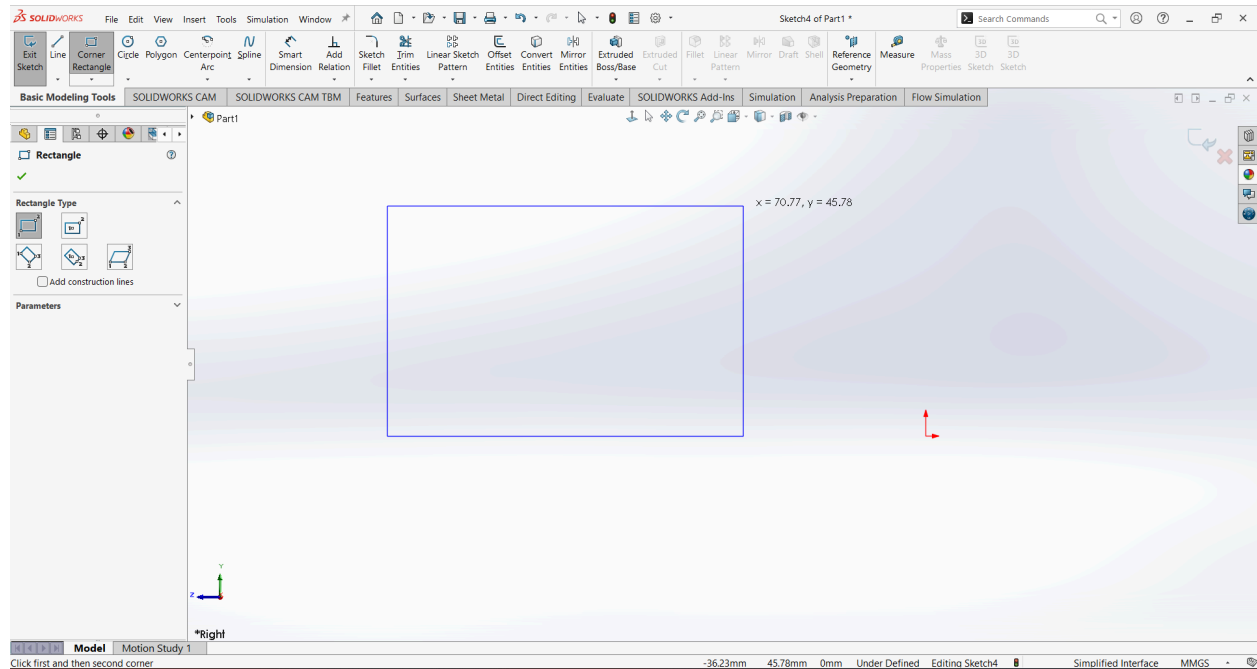
The second sketching tool is **Rectangle**. When you click the small arrow below the **Rectangle** icon, five different rectangle options will appear. Each option creates a rectangle in a slightly different way, depending on how you want to define its position and shape.



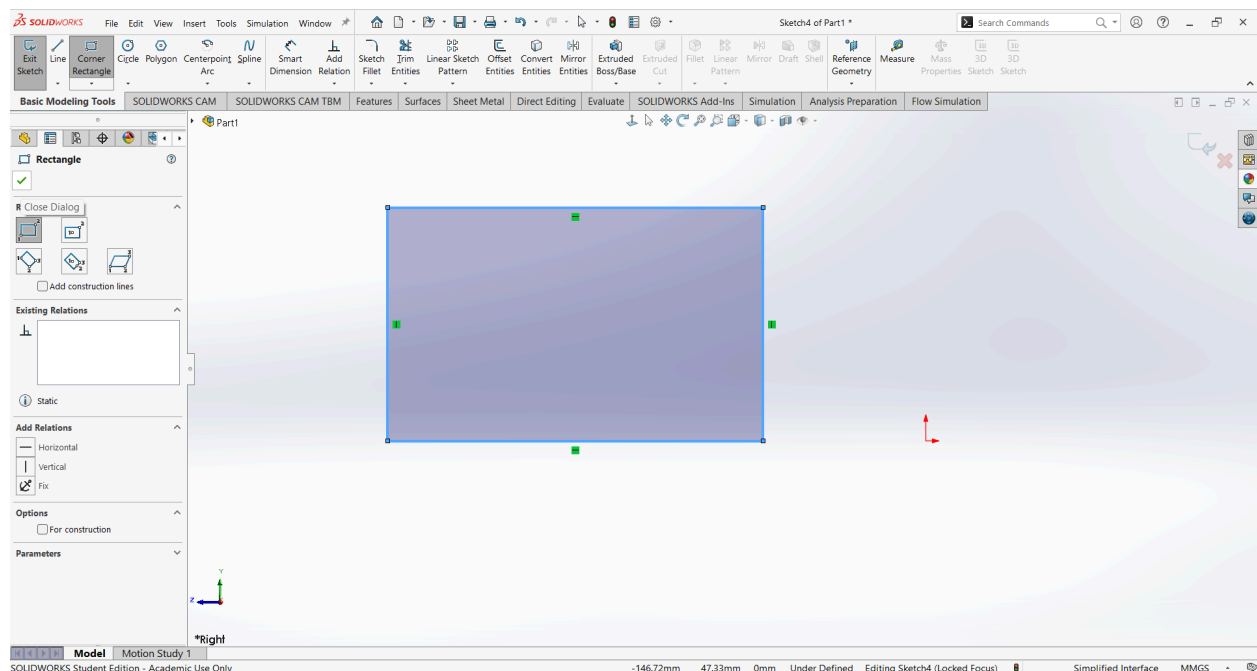
Let's start with the **Corner Rectangle**.



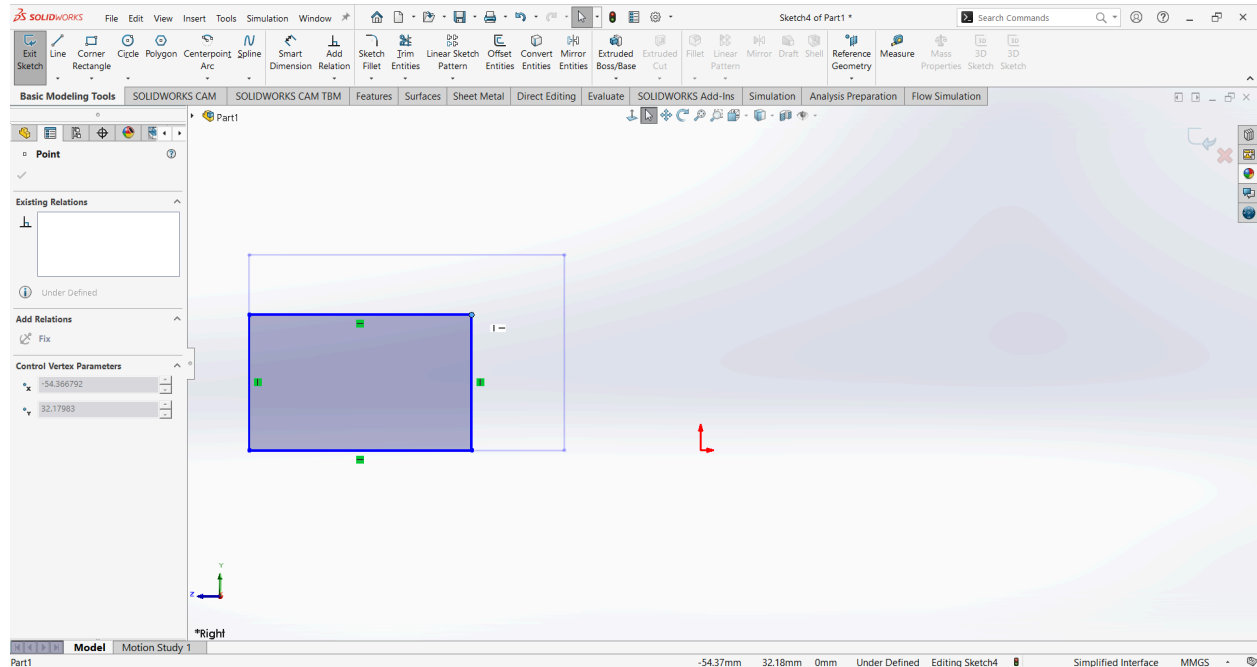
With the first click, define one corner of the rectangle. Then move the mouse to set the size of the rectangle, and click again to define the opposite corner.



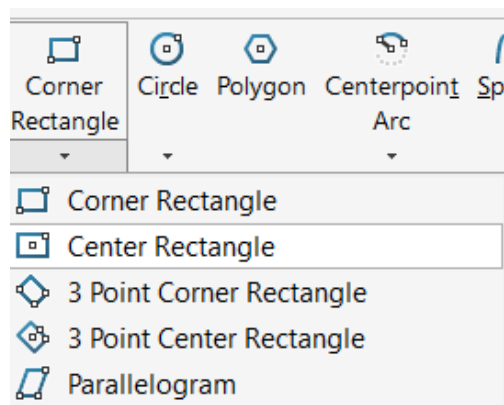
Now we have a rectangle with two vertical lines and two horizontal lines. You can see the green relation icons next to the lines. These icons show that SOLIDWORKS automatically added **Horizontal** and **Vertical** relations to keep the rectangle shape.



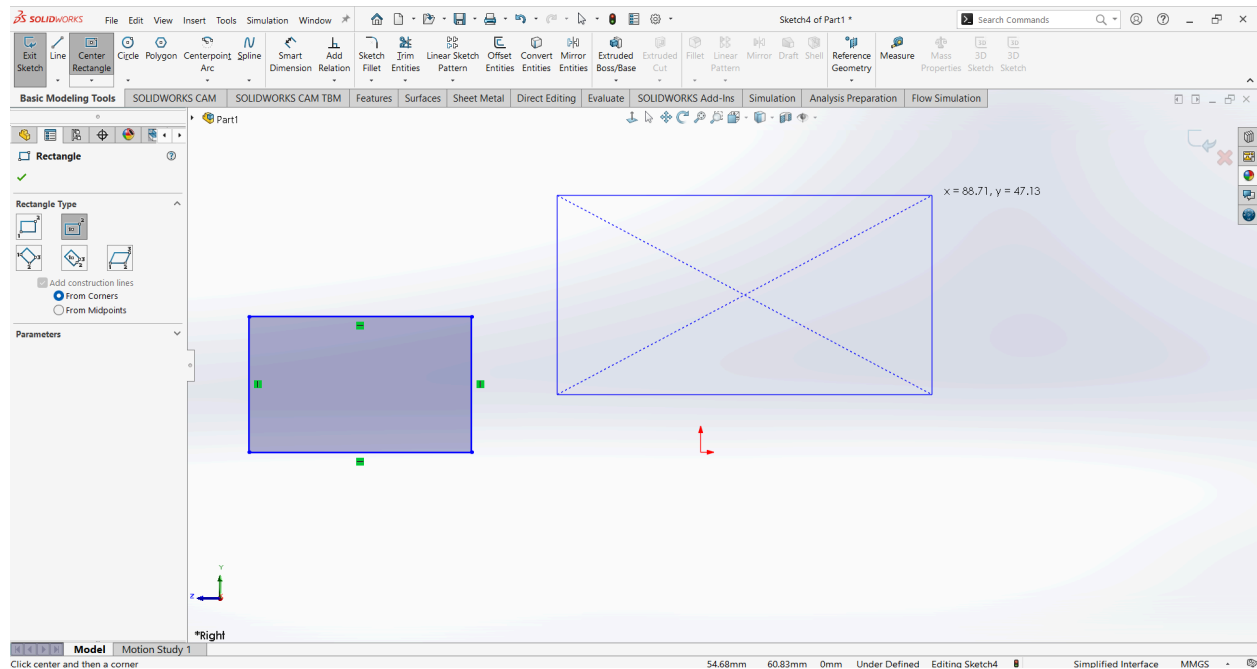
When you click on a vertex and move the mouse, you can change the shape and size of the rectangle. This allows you to adjust the rectangle before adding final dimensions.



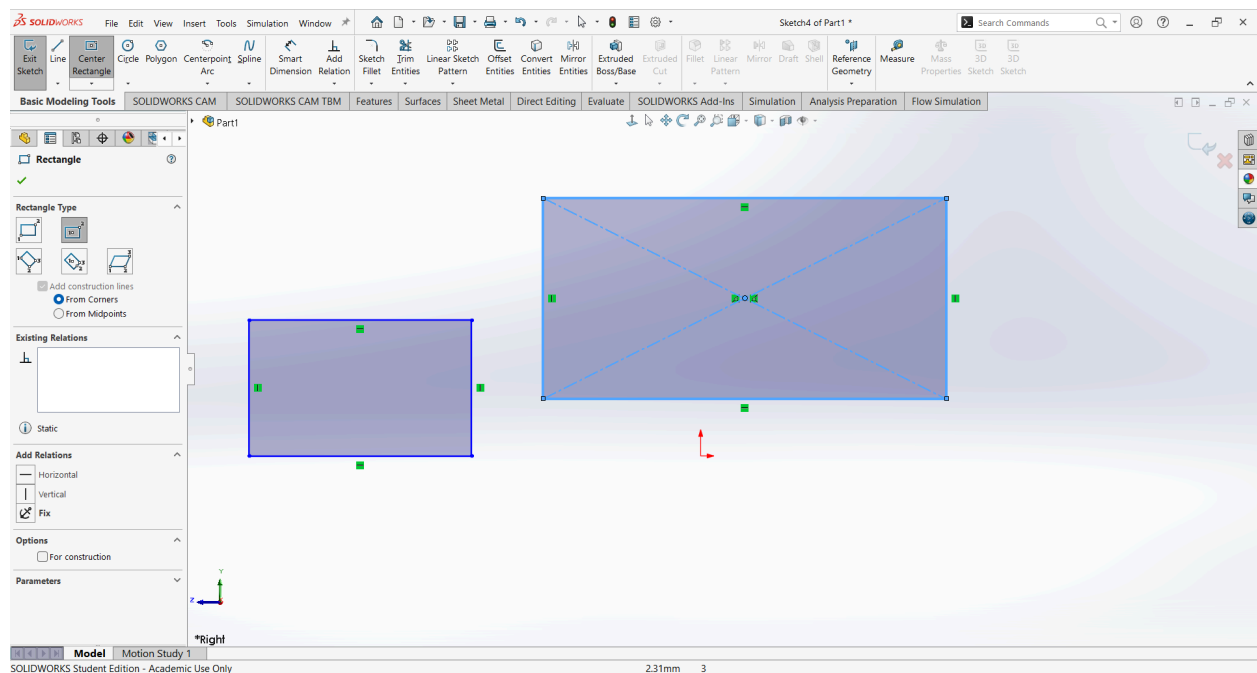
The second rectangle option is **Center Rectangle**. This tool allows you to create a rectangle from its center point instead of from one corner.



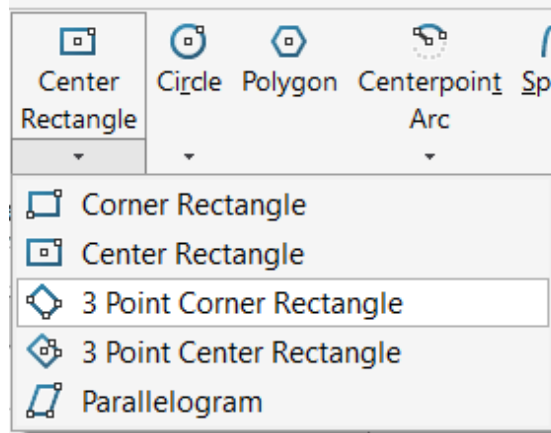
With the first click, define the center point of the rectangle. Then move the mouse to set the size of the rectangle, and click again to define one of its corners.



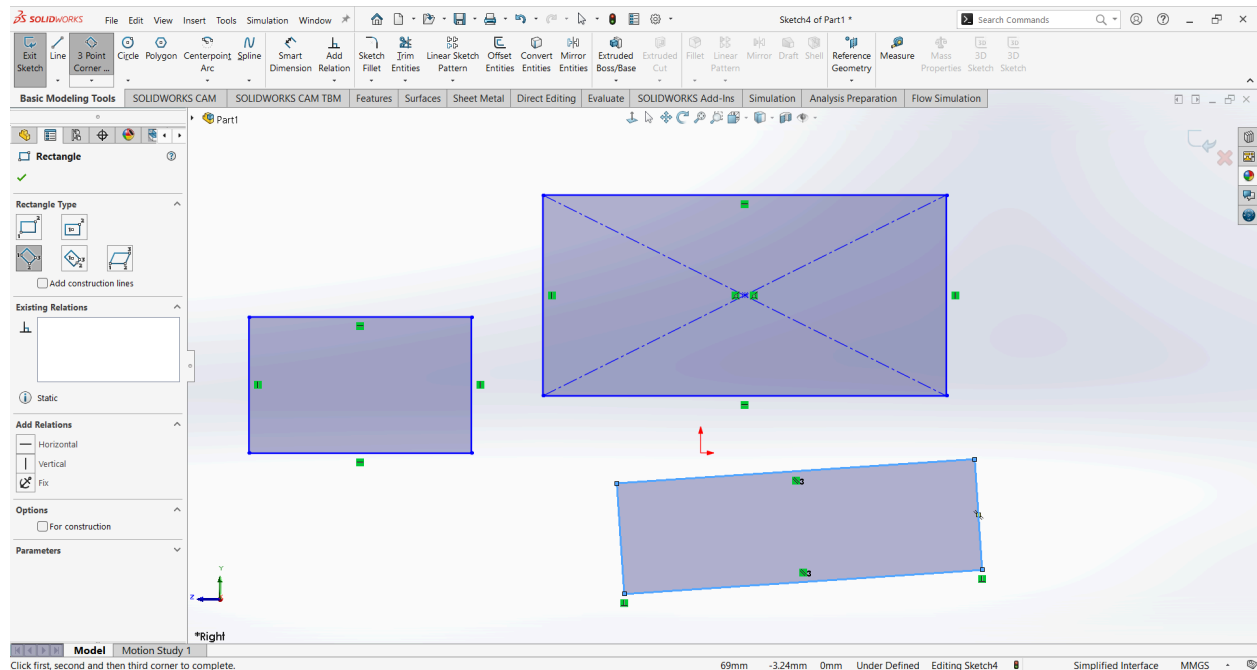
Now we have a rectangle with two construction lines connecting the opposite vertices. These construction lines cross at the center point of the rectangle and help show that the rectangle was created from the center.



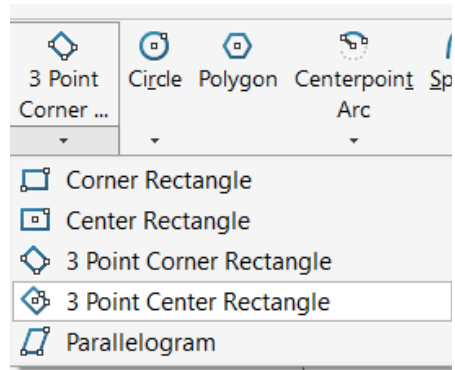
The third rectangle option is **3 Point Corner Rectangle**. This tool allows you to create a rectangle by defining three points: the first corner, the second corner, and then the width or direction of the rectangle.



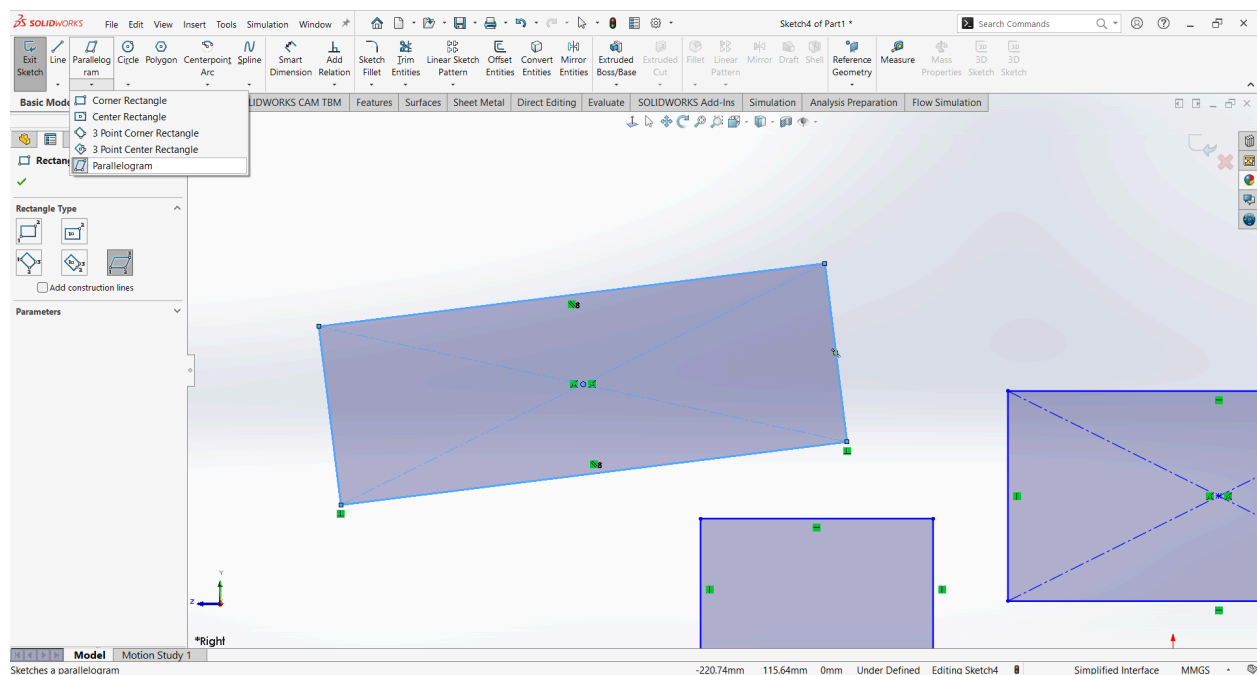
With this option, you can create a rectangle at an angle. With the first click, define the first vertex. Then move the mouse and click again to define the second vertex. After that, move the mouse further and click one more time to define the third vertex and complete the rectangle.



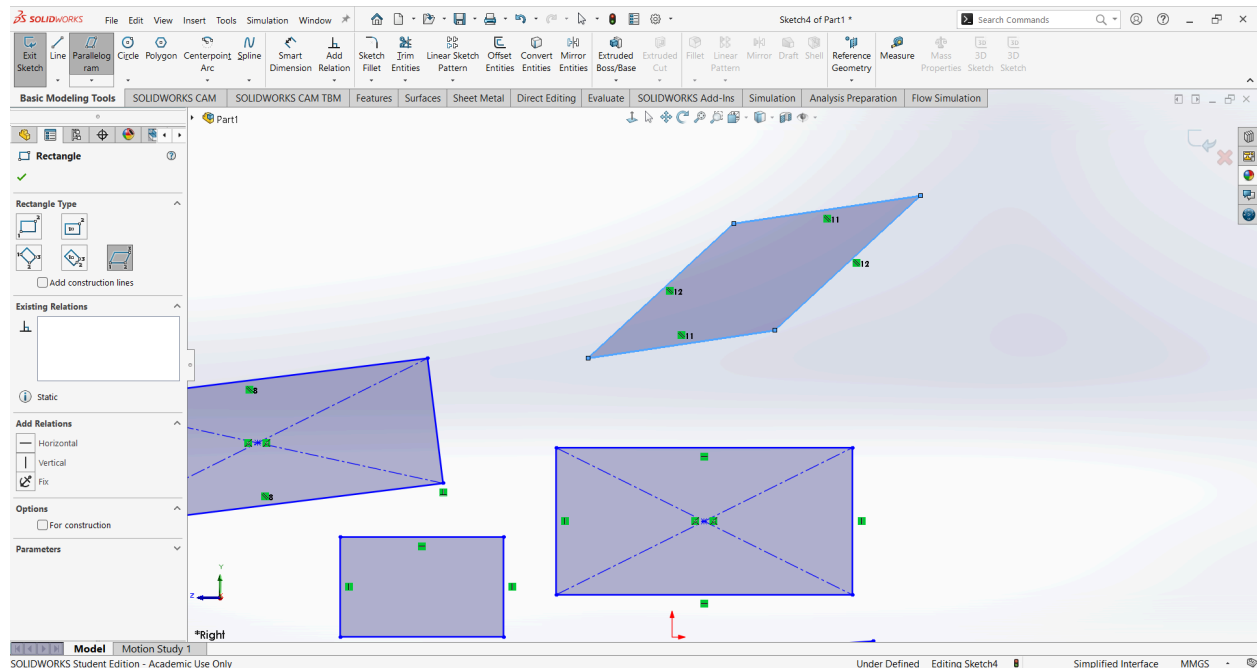
The next rectangle option is **3 Point Center Rectangle**. This tool allows you to create a rectangle from its center point, while also defining its angle and size.



With this option, the first click defines the center point of the rectangle. Then, with the next two clicks, you define two vertices in different directions. This allows you to control both the angle and size of the rectangle.



The last rectangle option is **Parallelogram**. By defining three points, we create a **Parallelogram**. The first two points define one side, and the third point defines the angle and height of the parallelogram.



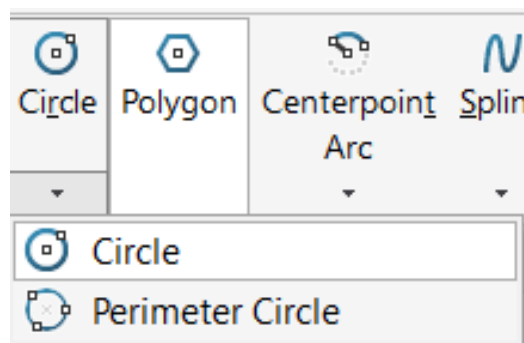
4. Circle

What it does: Creates circular sketch geometry.

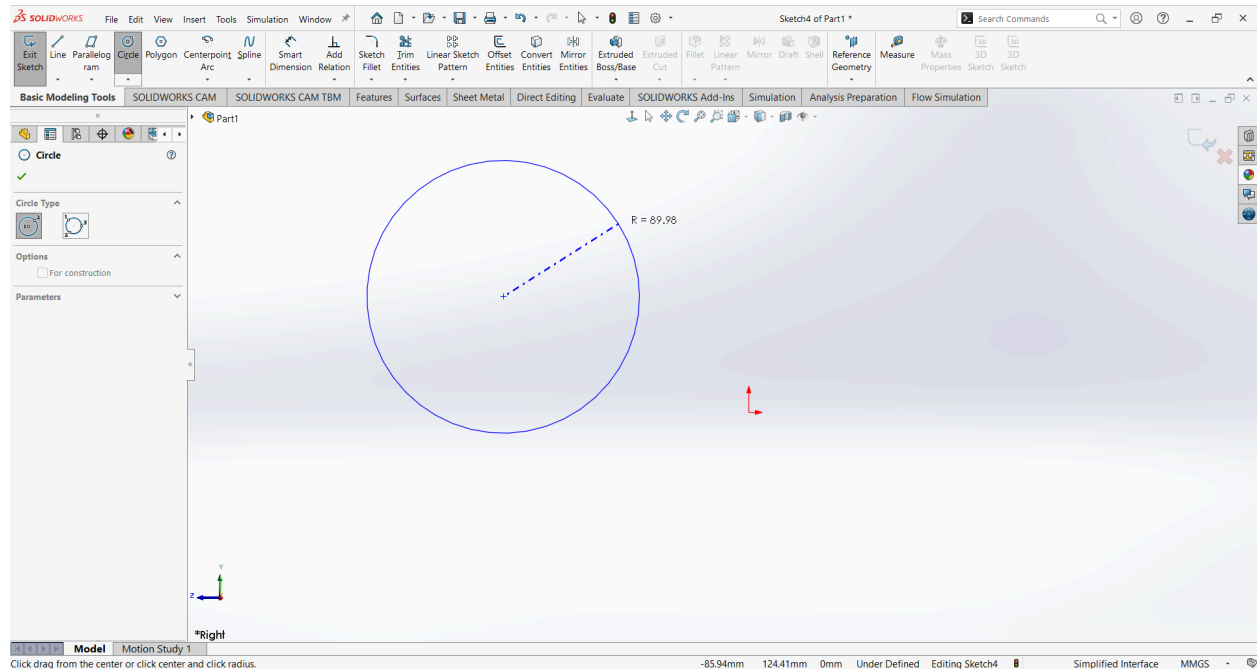
Use it for: Holes, shafts, bosses, cylinders, and revolved geometry.

How it works:

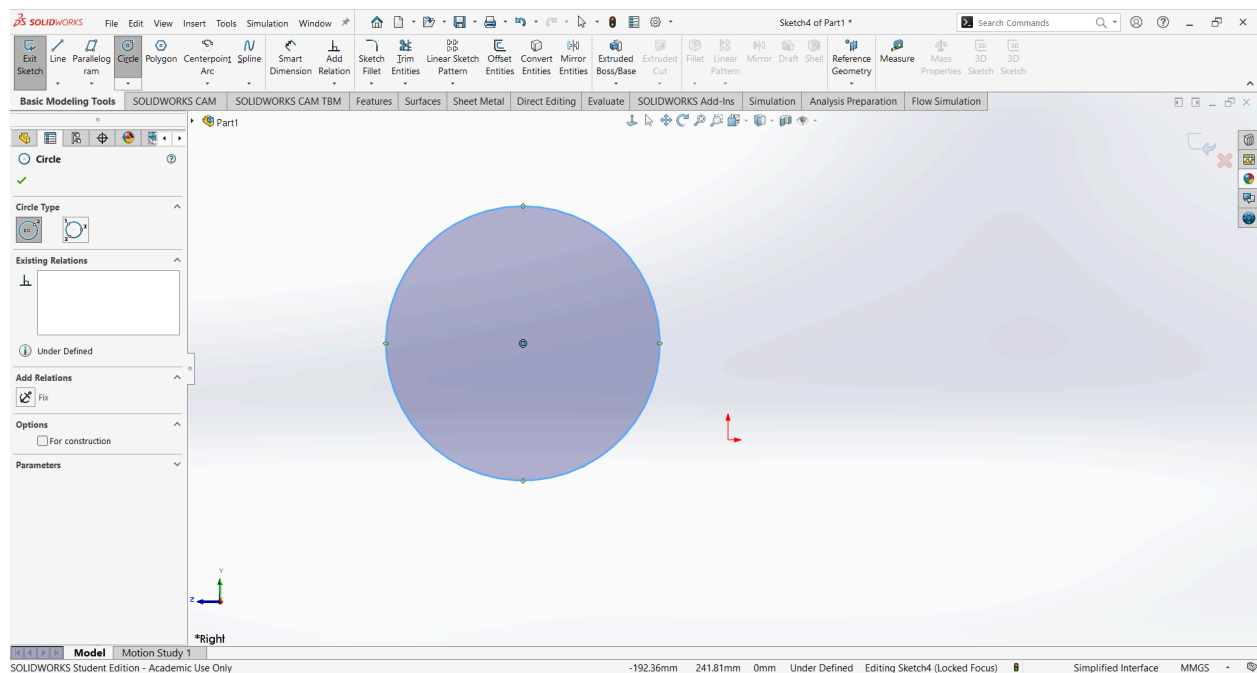
The next sketching tool is **Circle**. There are two circle options available. Click the first option to create a standard circle.



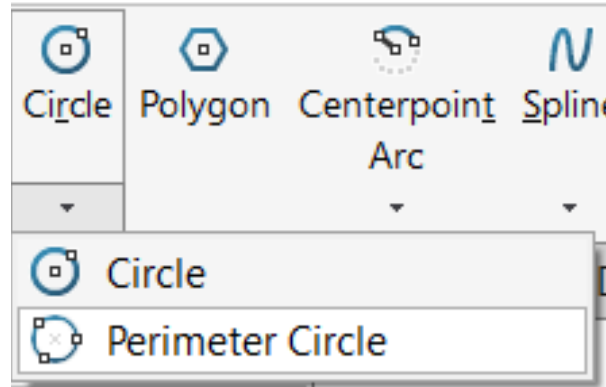
With the first click, define the center point of the circle. Then move the mouse to set the size of the circle, and click again to define the radius.



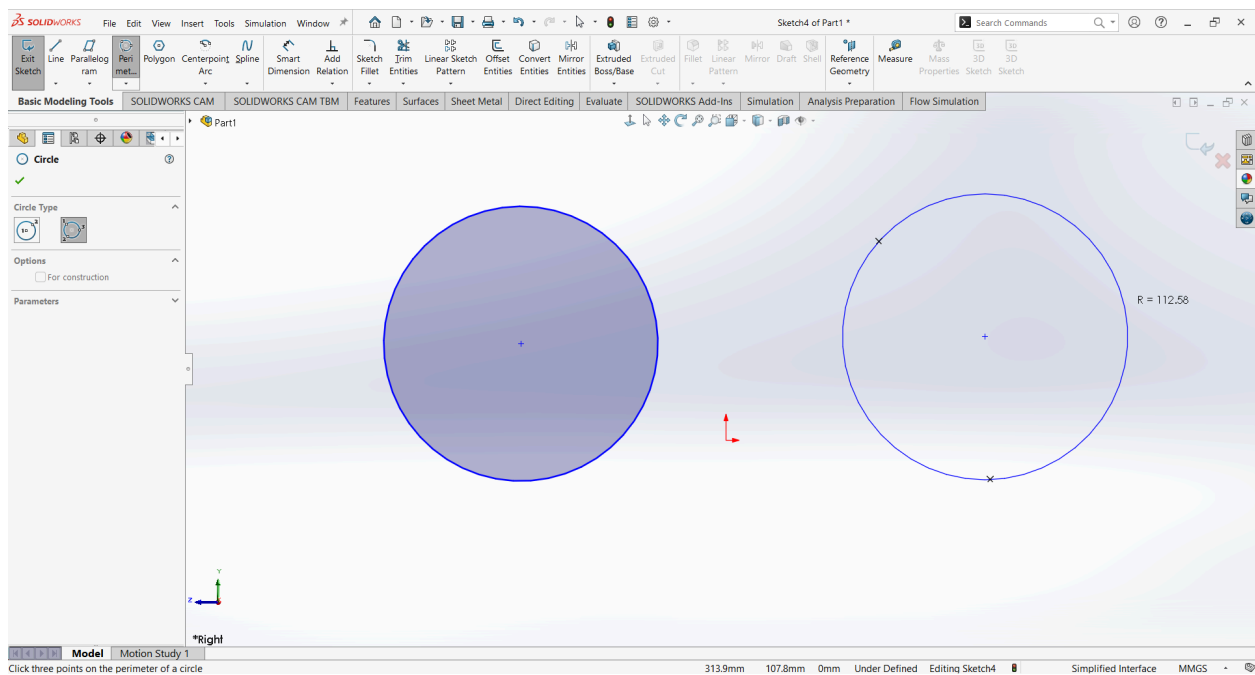
This circle is still not fixed. If you click on the circle and move the mouse, you can change its radius. If you click on the center point and move the mouse, you can change the position of the circle.



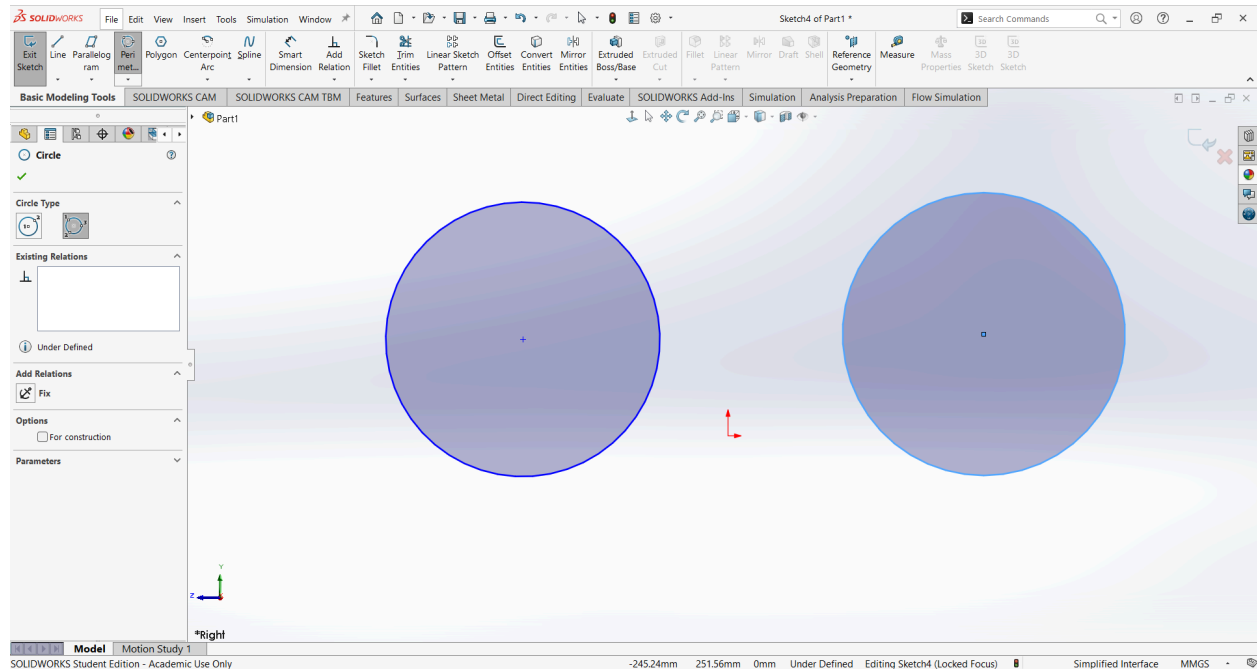
Click the second option: **Perimeter Circle**. This tool allows you to create a circle by defining points on its perimeter.



With the first click, define the first point on the perimeter of the circle. Then move the mouse and click again to define the second point. After that, move the mouse further and click one more time to define the third point on the perimeter and complete the circle.



And this is the completed circle. It was created by defining three points on its perimeter.



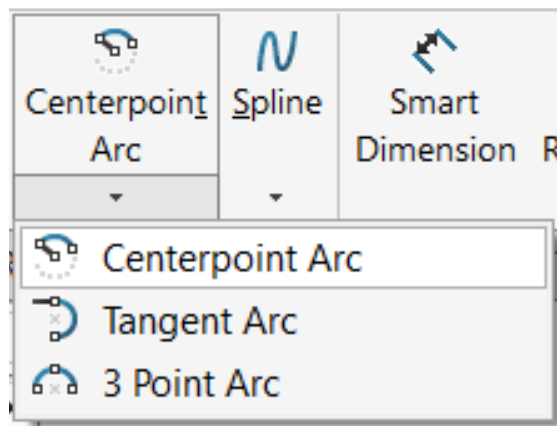
5. Arc

What it does: Creates curved sketch geometry.

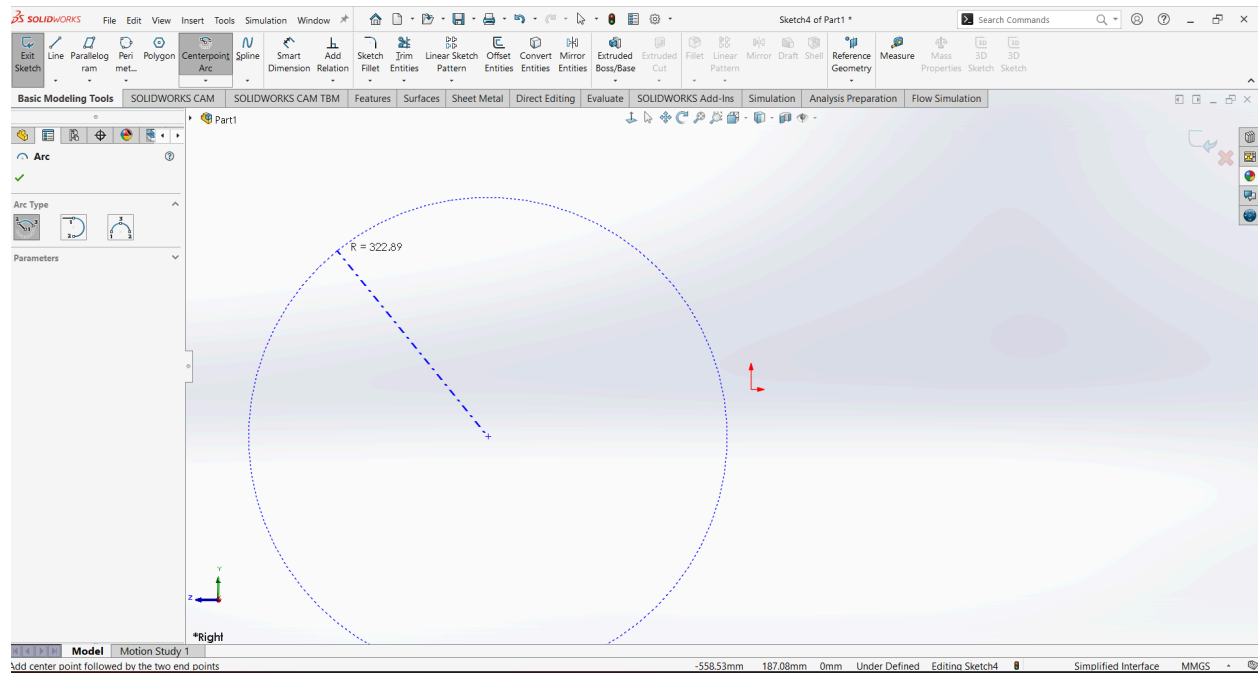
Use it for: Rounded profiles, slots, transitions, and curved features.

How it works:

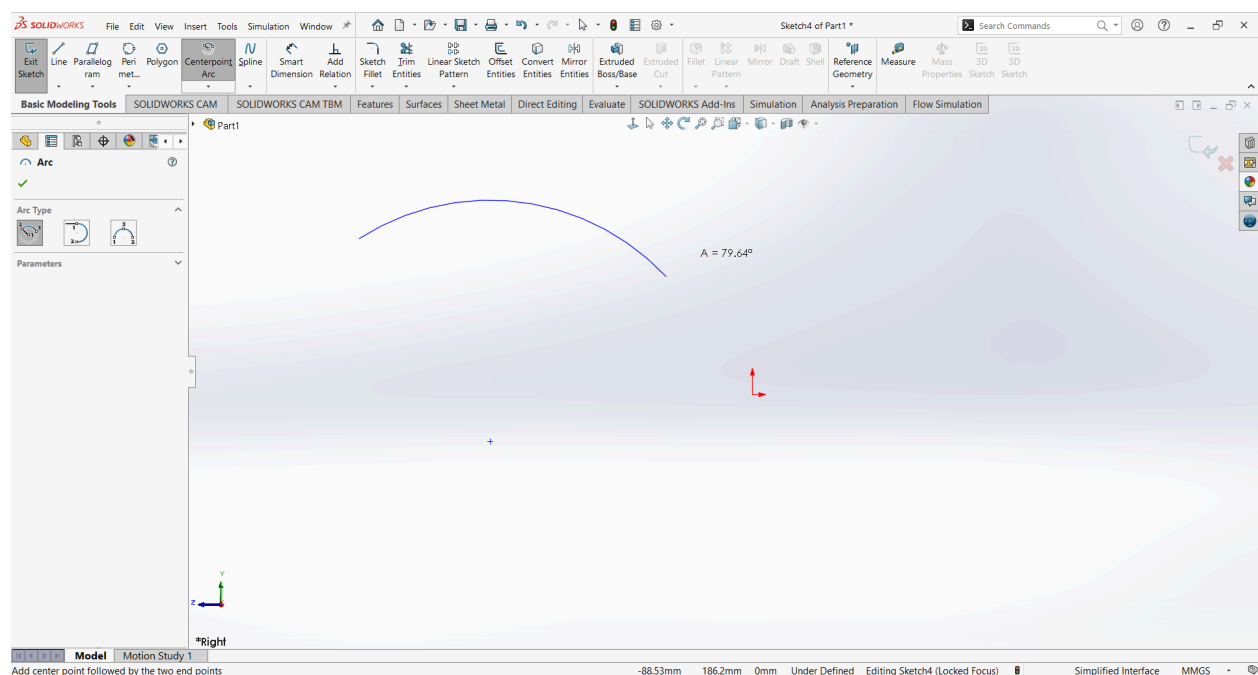
The next sketching tool is **Arc**. There are three arc options available: **Centerpoint Arc**, **Tangent Arc**, and **3 Point Arc**.



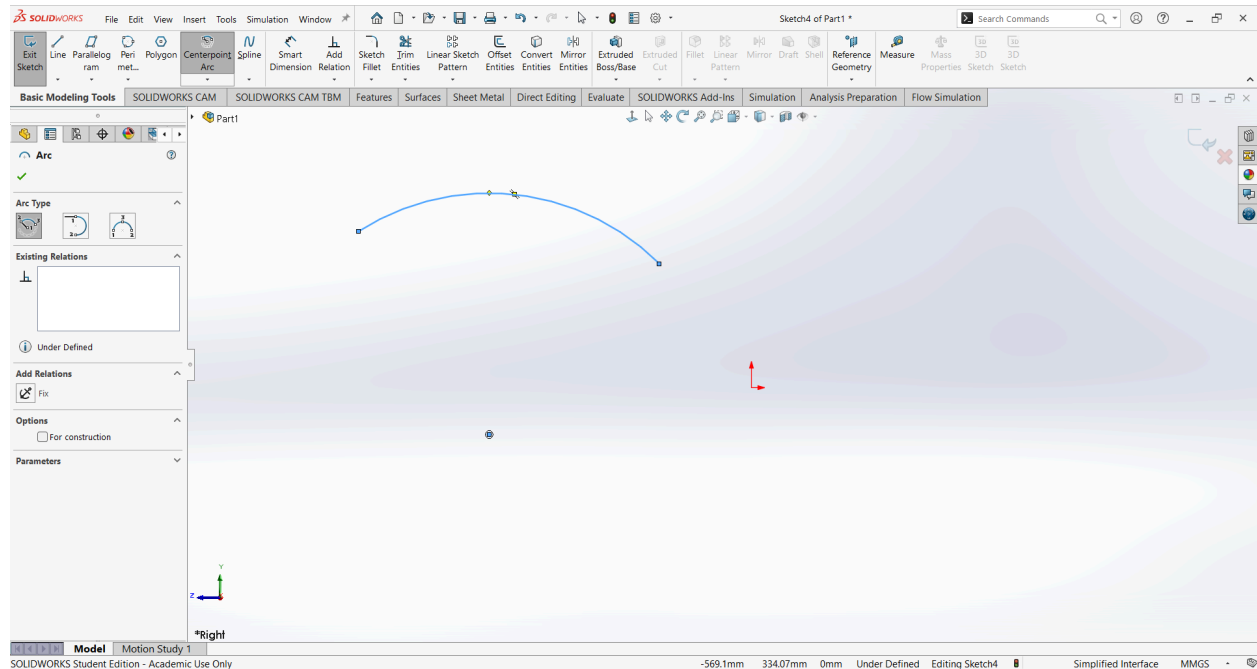
Select the first option: **Centerpoint Arc**. With the first click, define the center point of the arc. Then move the mouse and click again to define the starting radius point.



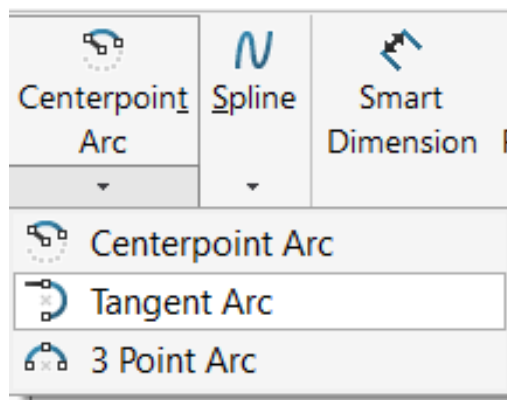
By moving the mouse further, you create the arc. With the third click, define the end radius point and complete the arc.



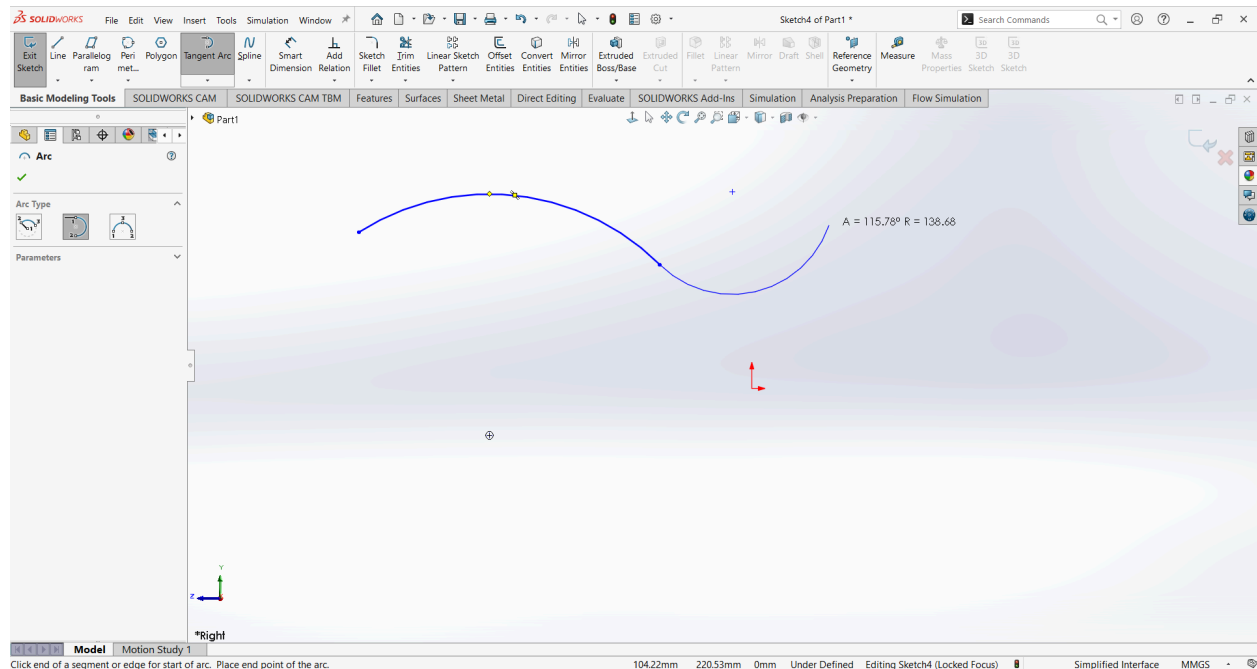
Now we have an arc that can still be moved and edited. You can change its position by moving the arc, or change its shape by clicking and dragging the vertices or the arc itself.



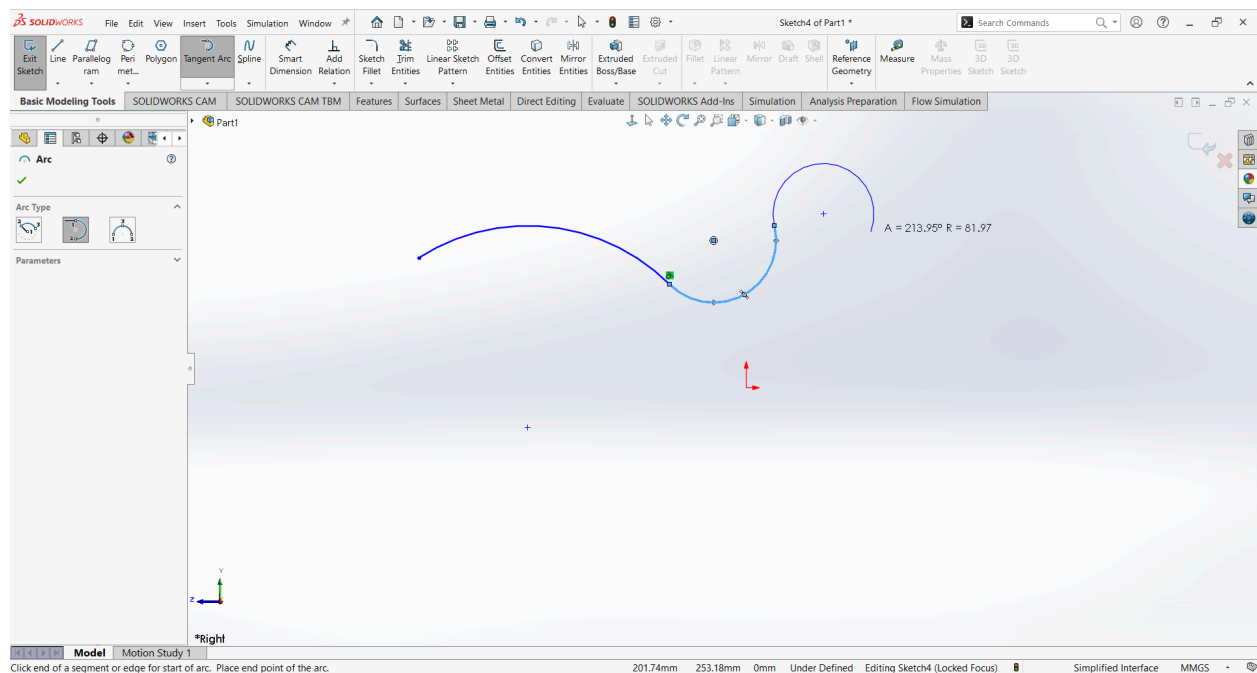
The second arc option is **Tangent Arc**. With this tool, you can create an arc that is tangent to another sketch entity, such as a line or another arc.



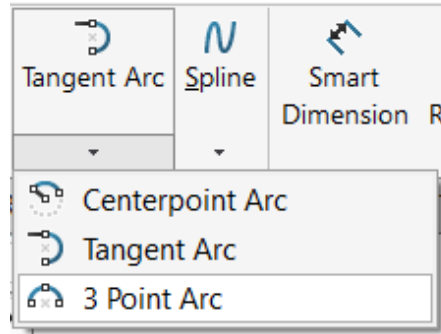
The first click is on an existing sketch entity. In this example, we select the arc that we created earlier. Then move the mouse and click again to define the endpoint of the tangent arc.



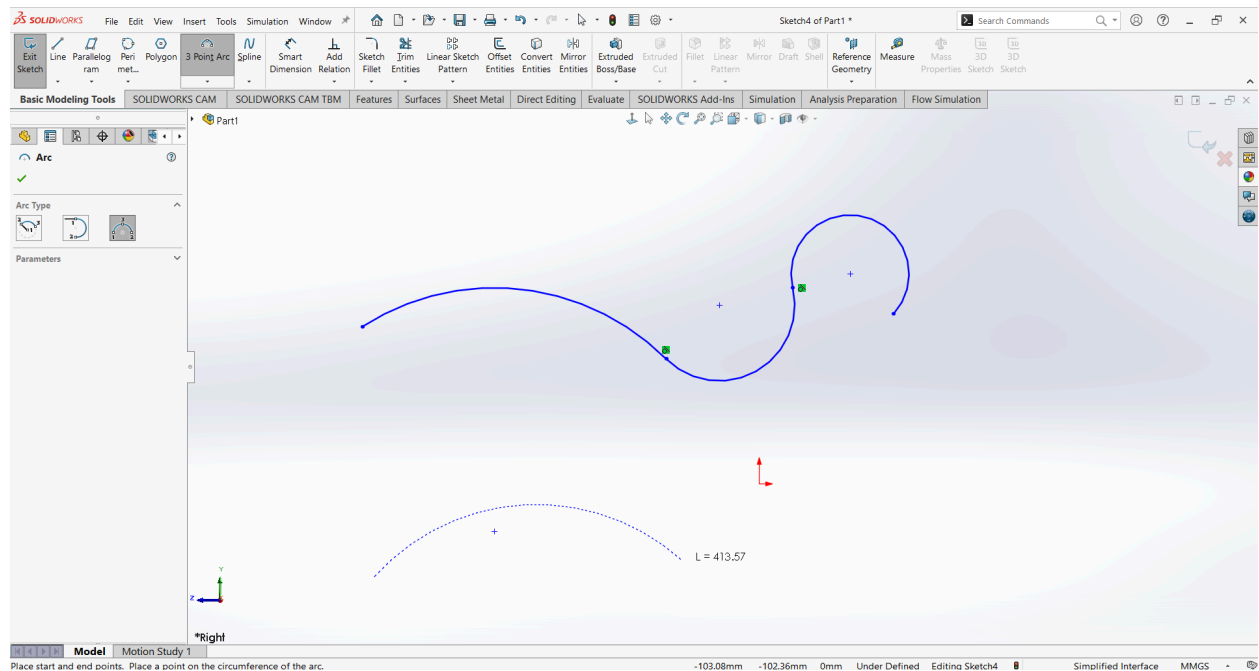
Now there is a tangent relation between these two arcs. You can see the small green relation icon where the two arcs connect. By defining the next point, you can continue creating arcs that are tangent to the previous one.



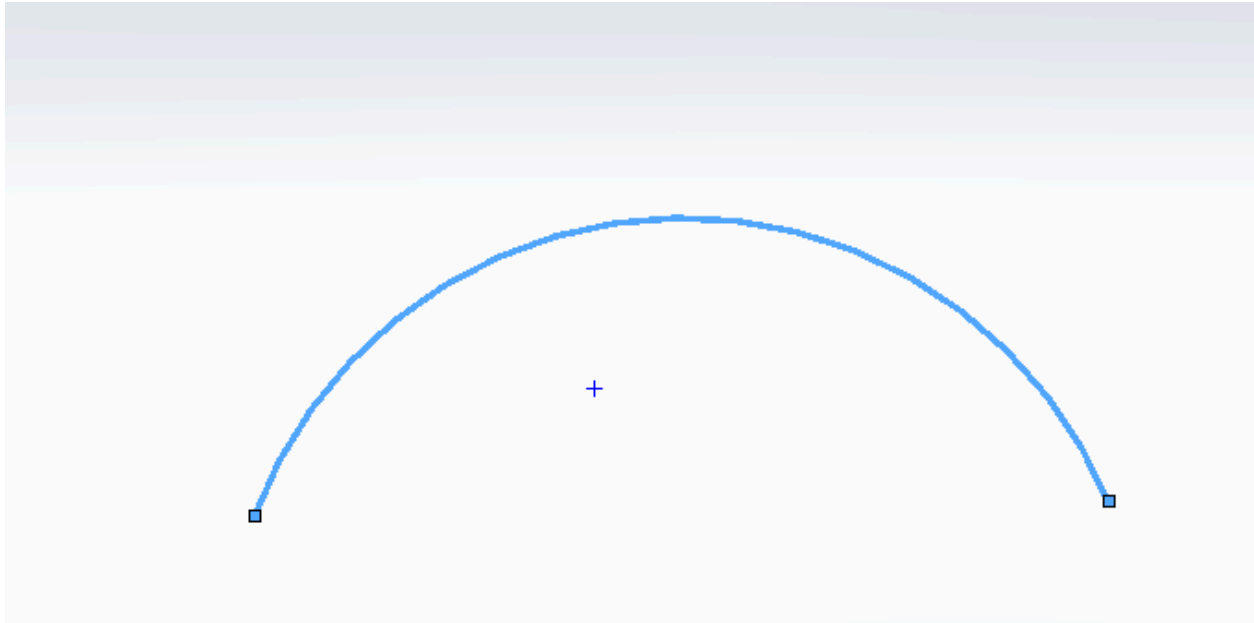
The third arc option is **3 Point Arc**. This tool allows you to create an arc by defining three points: the start point, the end point, and the point that controls the shape of the arc.



With the first click, define the start point of the arc. Then move the mouse and click again to define the end point. After that, move the mouse further and click one more time to define the shape of the arc.



Here is the completed arc. It was created by defining the start point, end point, and arc shape.



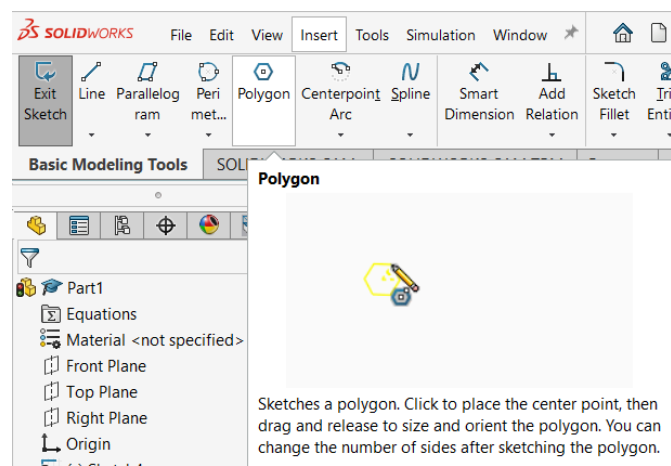
6. Polygon

What it does: Creates a multi-sided regular polygon.

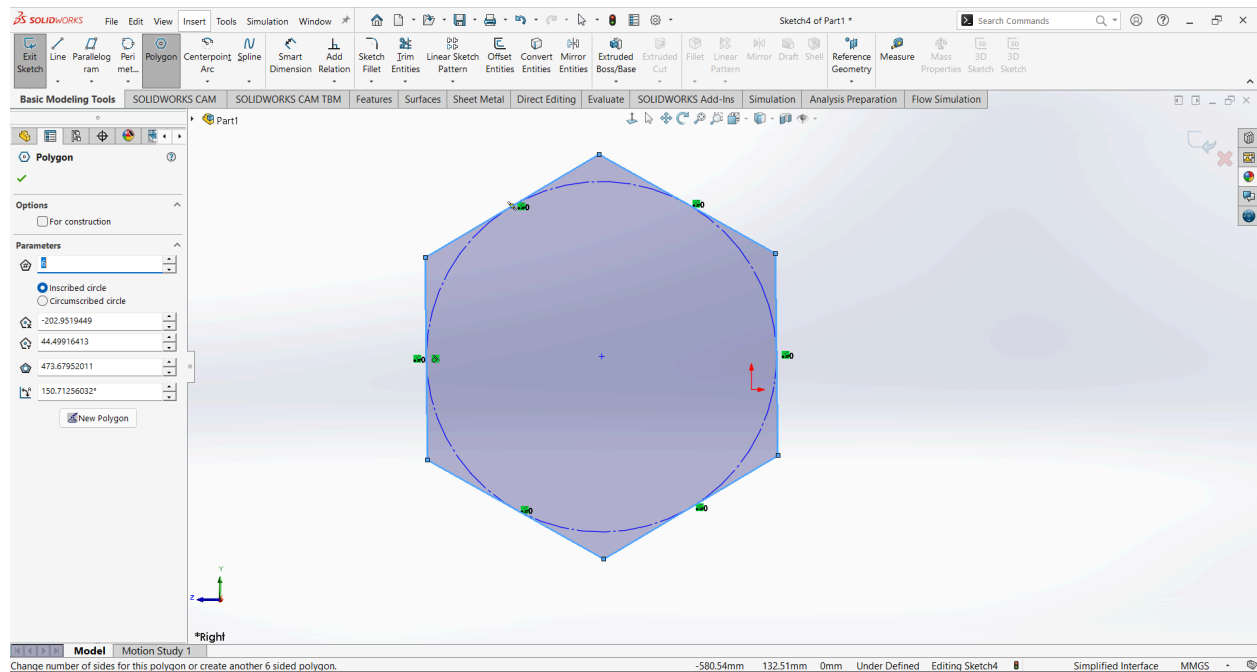
Use it for: Hexagonal profiles, bolt heads, nuts, and other regular shapes.

How it works:

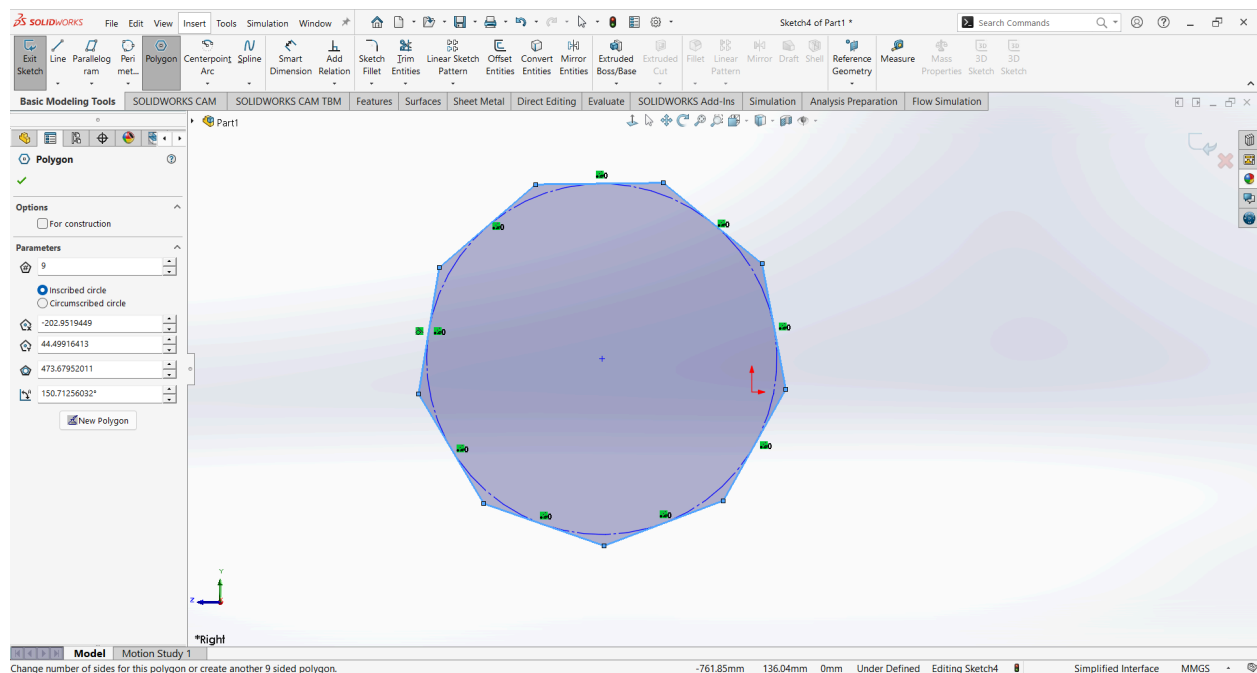
The next sketching tool is **Polygon**. This tool allows you to create shapes with multiple straight sides, such as triangles, pentagons, hexagons, and other regular polygons.



To create a polygon, first click to define the center point. Then move the mouse and click again to define one vertex of the polygon. On the left side, in the **PropertyManager**, you can also define the number of sides.



When we increase the number of sides, the polygon changes shape. The more sides we add, the closer the polygon starts to look like a circle.



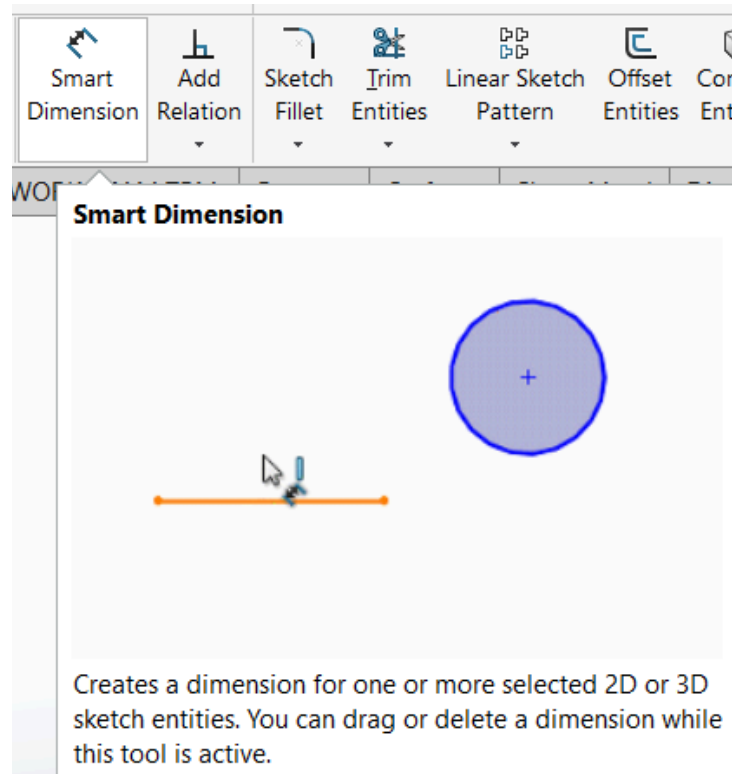
7. Smart Dimension

What it does: Adds dimensions to sketch geometry.

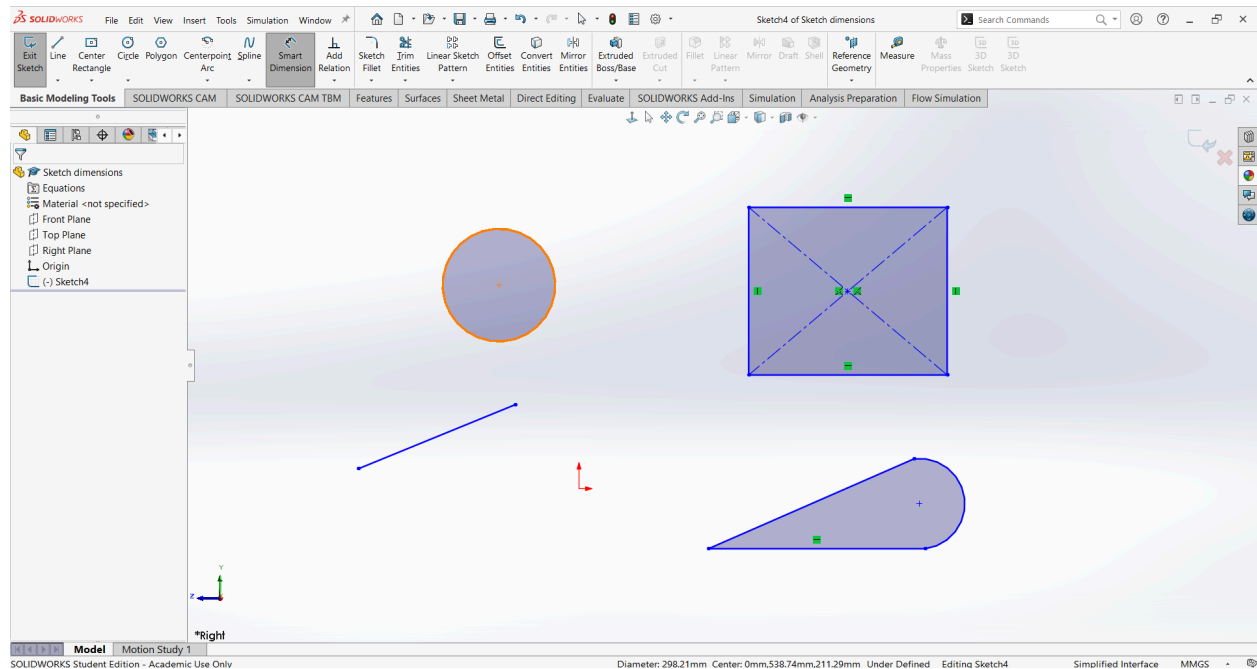
Use it for: Controlling lengths, distances, diameters, radii, and angles.

How it works:

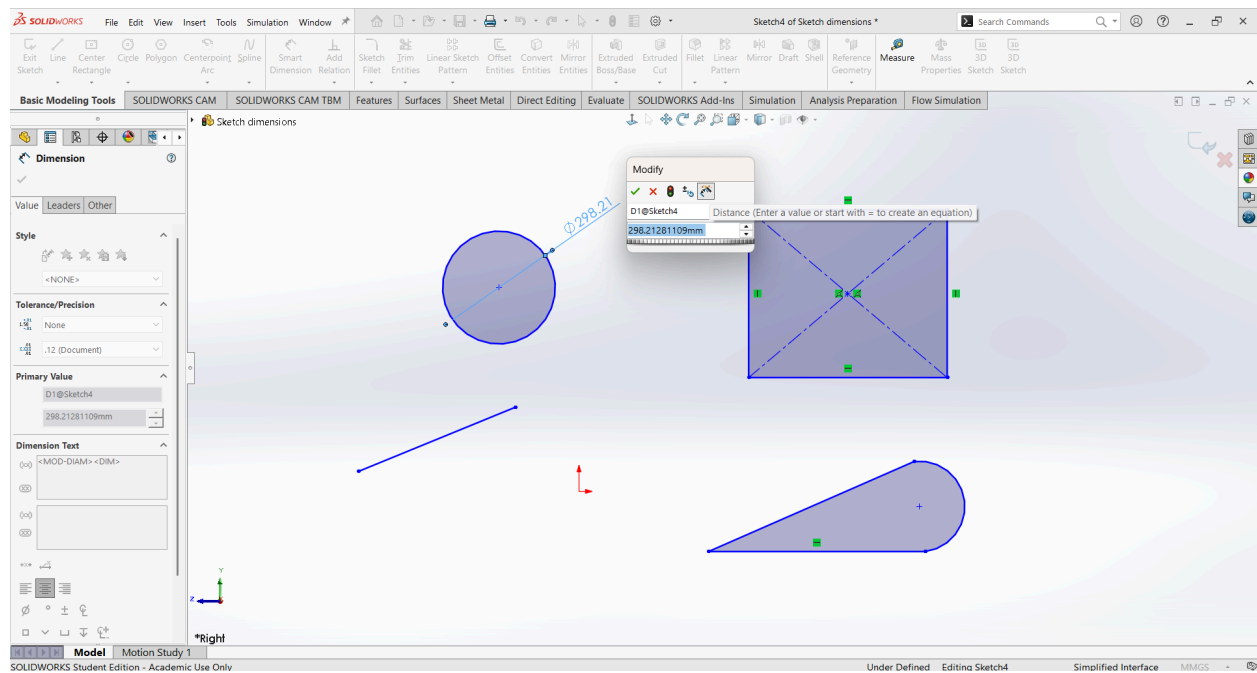
After defining sketch relations, the next step is to define dimensions. To add dimensions, we use the **Smart Dimension** tool.



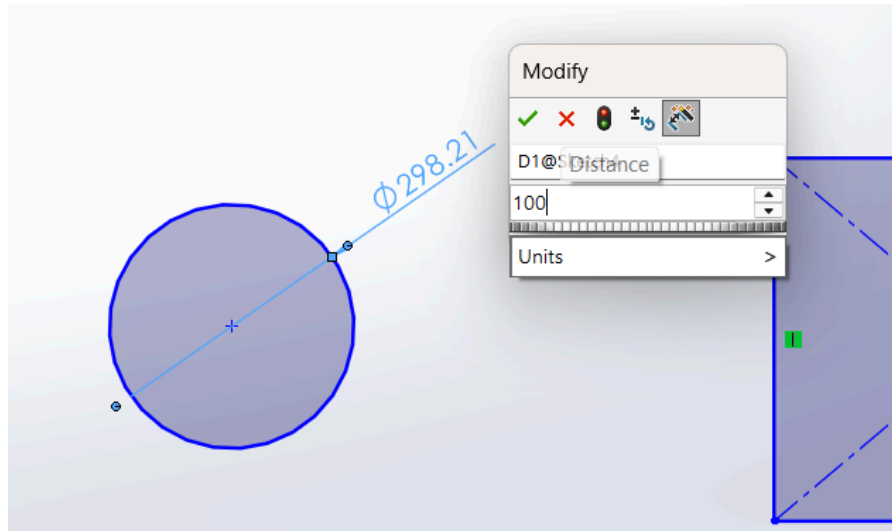
Click the **Smart Dimension** tool. Then click on the circle to select it. Move the mouse to place the dimension, and click again to confirm the dimension position.



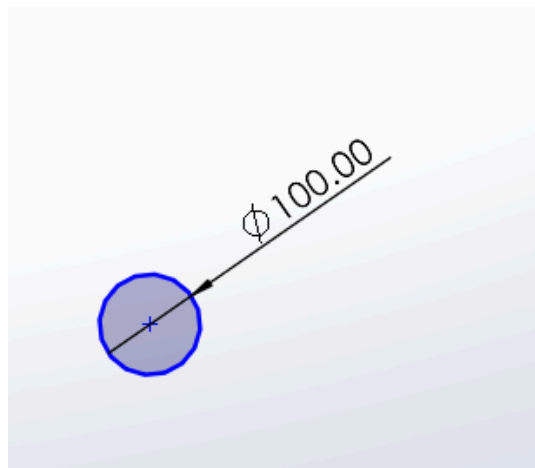
The diameter value will appear automatically. A small dialog box will also appear, where you can enter or change the diameter value.



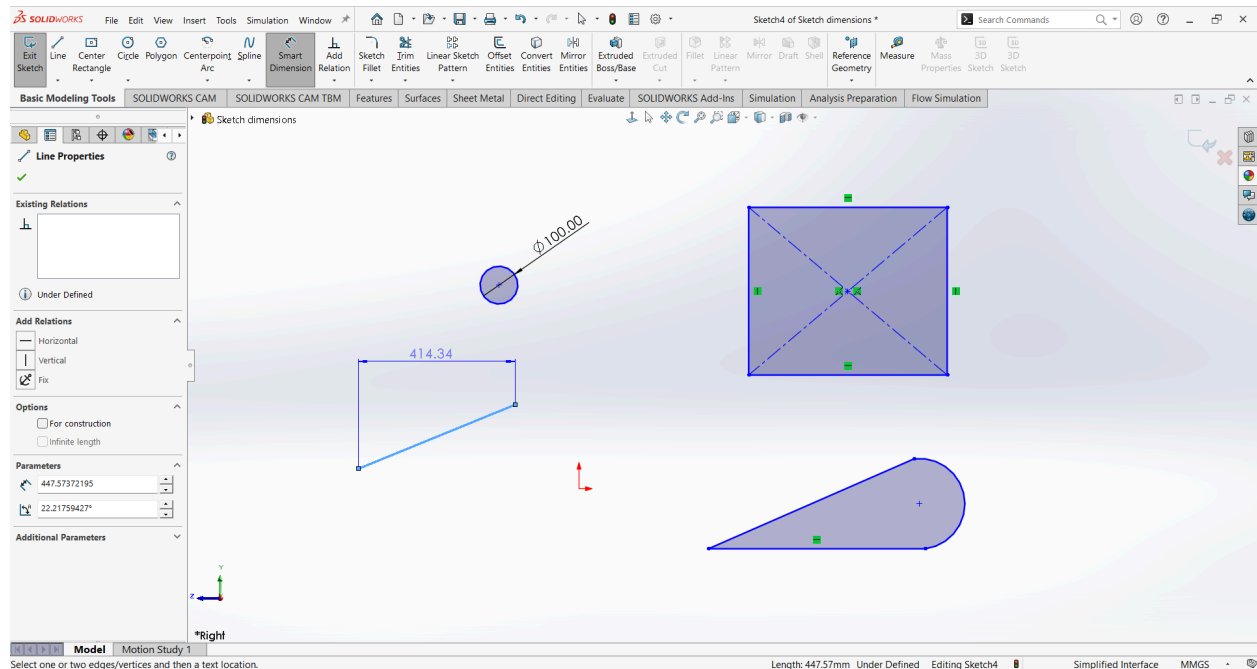
Type **100** as the diameter value and click **OK**.



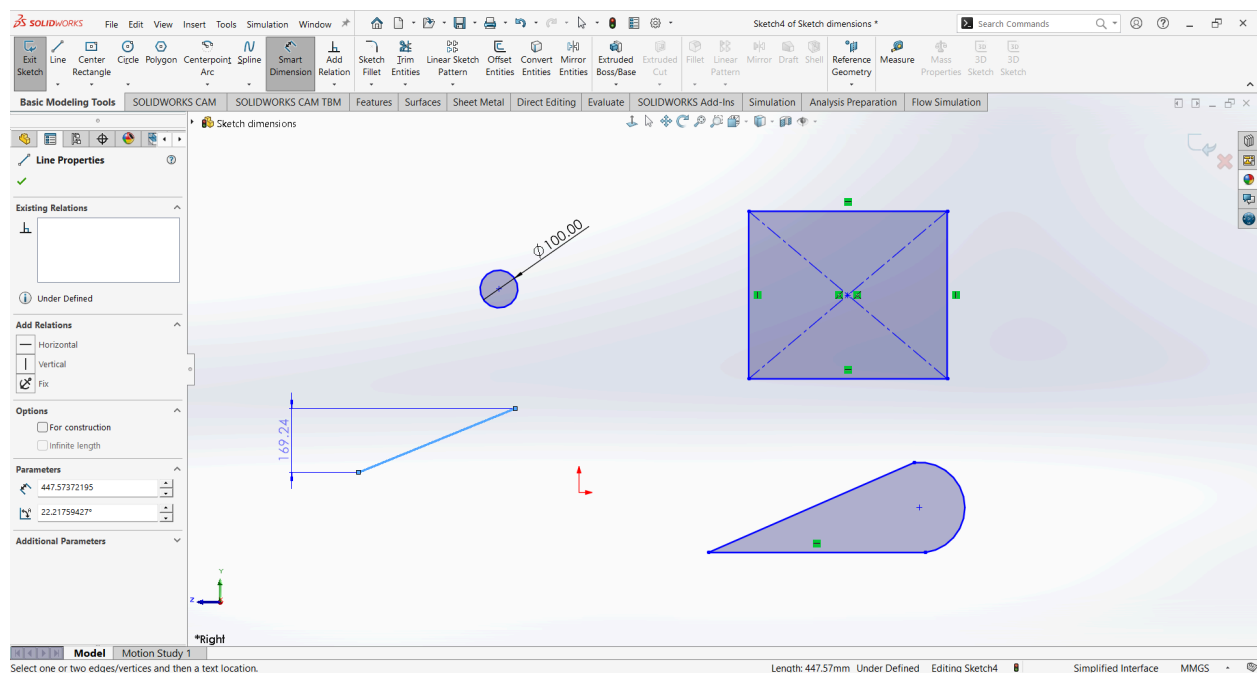
Now the diameter of the circle is **100**.



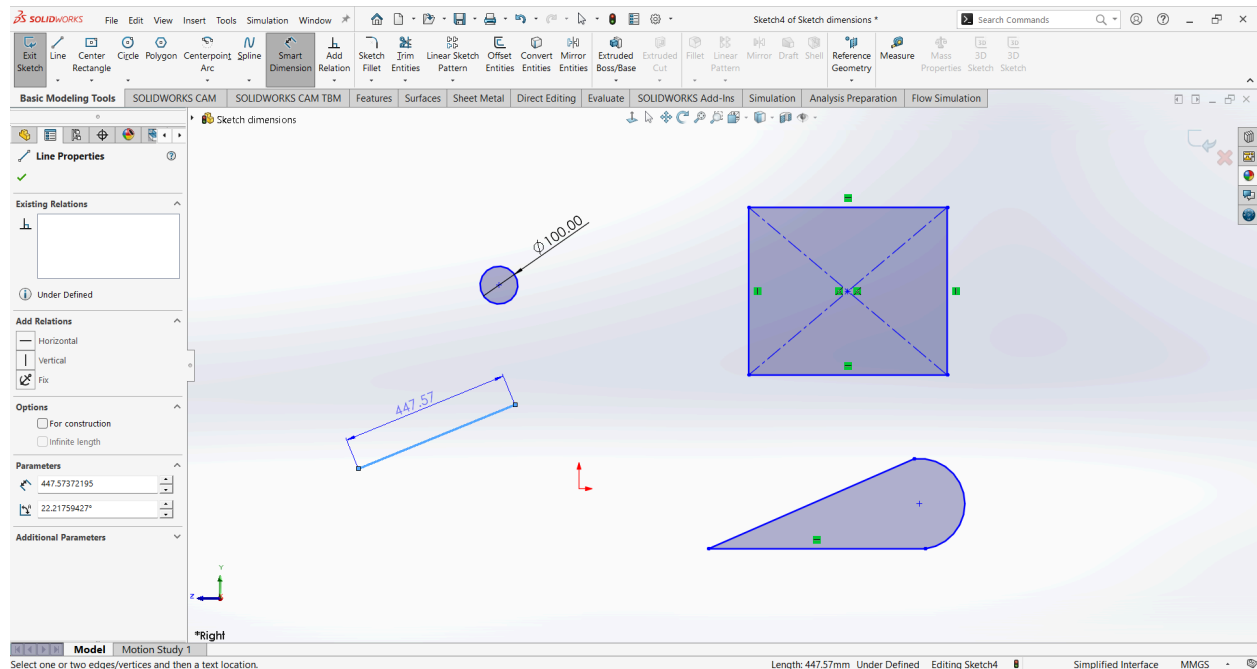
Select the **Smart Dimension** tool. Then select one endpoint of the line and the second endpoint of the line. When you move the mouse upward, SOLIDWORKS creates a horizontal dimension for the line.



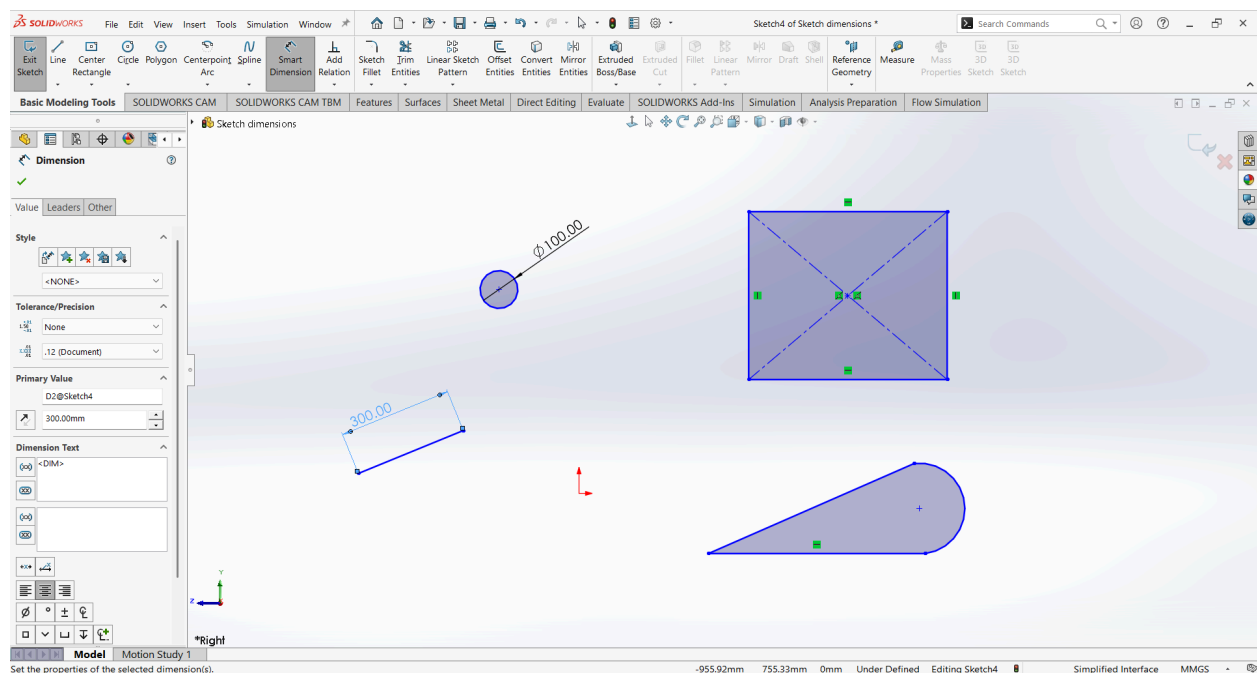
When you move the mouse to the left side, SOLIDWORKS creates a vertical dimension for the line.



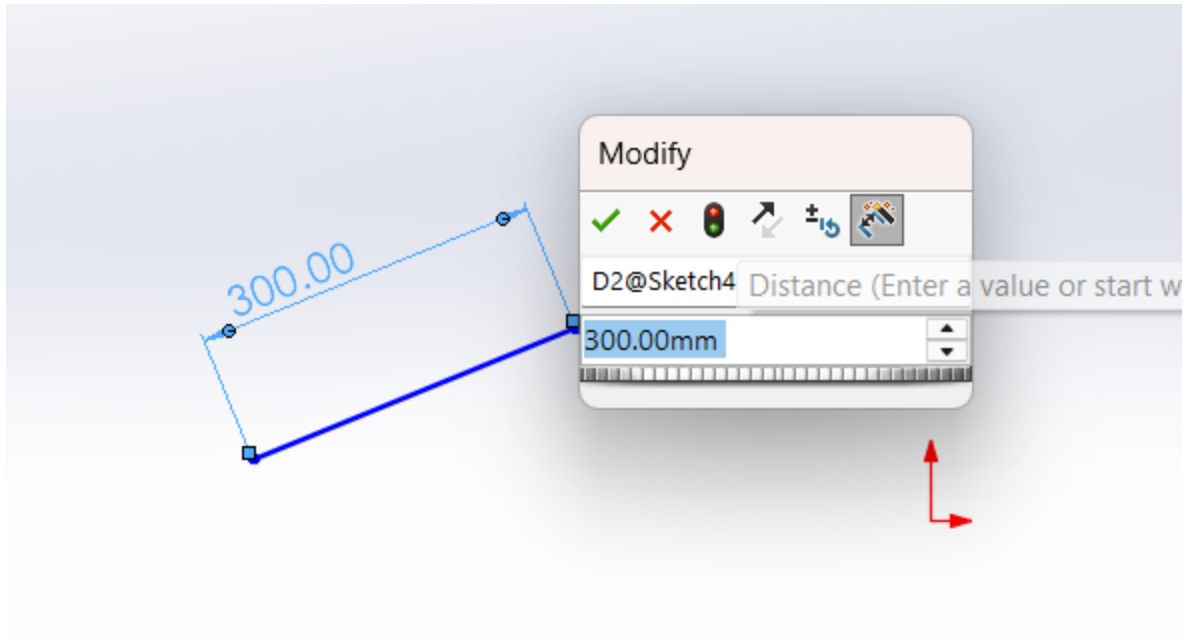
When you move the mouse perpendicular to the line, SOLIDWORKS creates a dimension for the true length of the line.



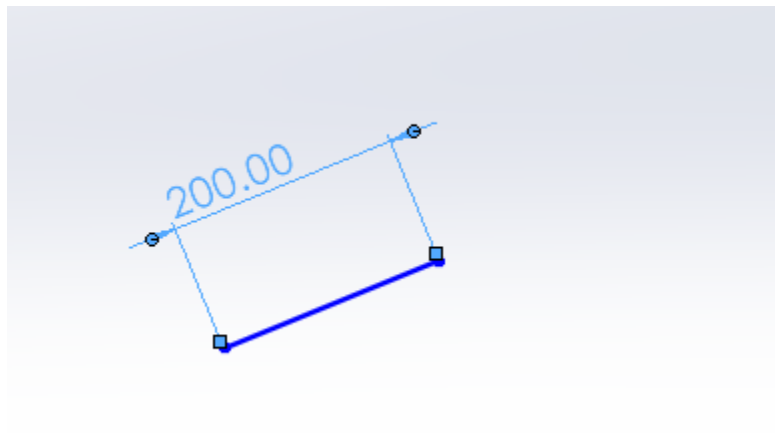
Once you choose the type of dimension you want, click again to place and confirm the dimension.



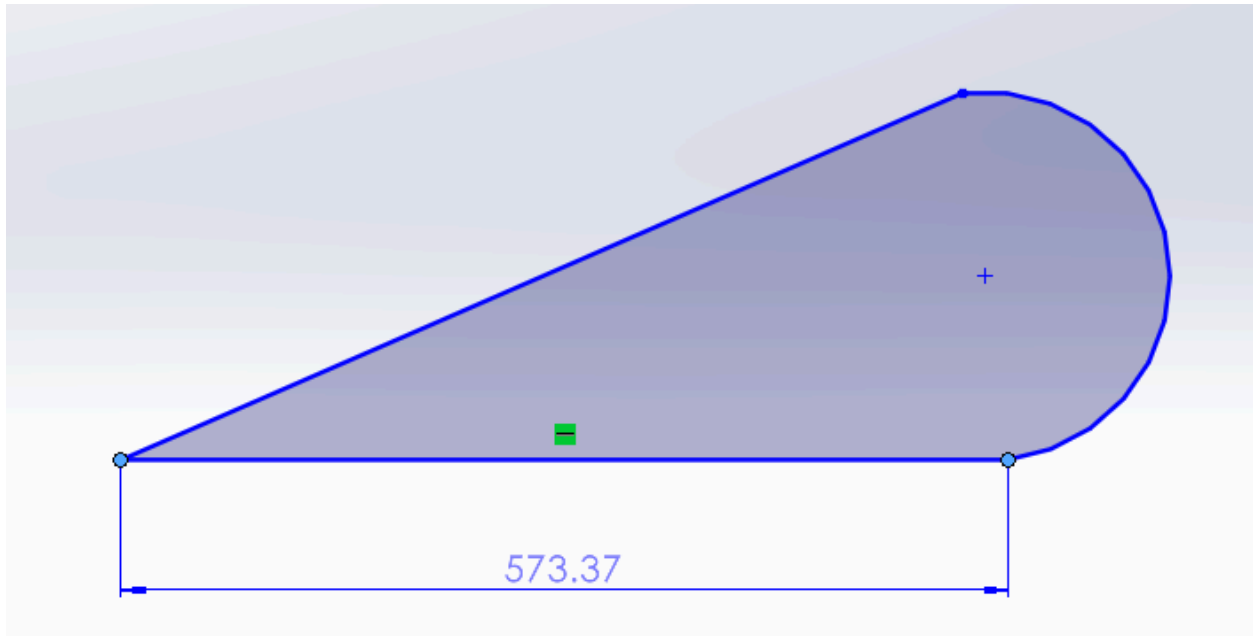
To change the dimension value again, double-click on the value **300**. Then type **200** and click **OK**.



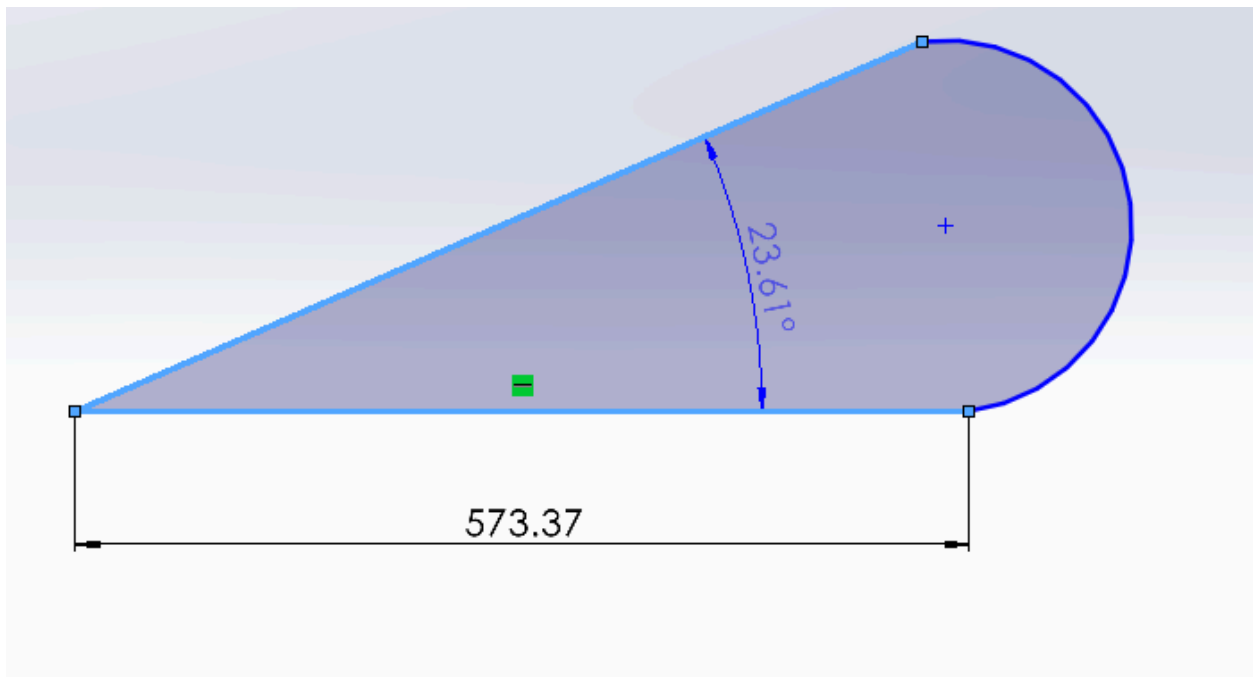
Now the dimension value is changed to **200**.

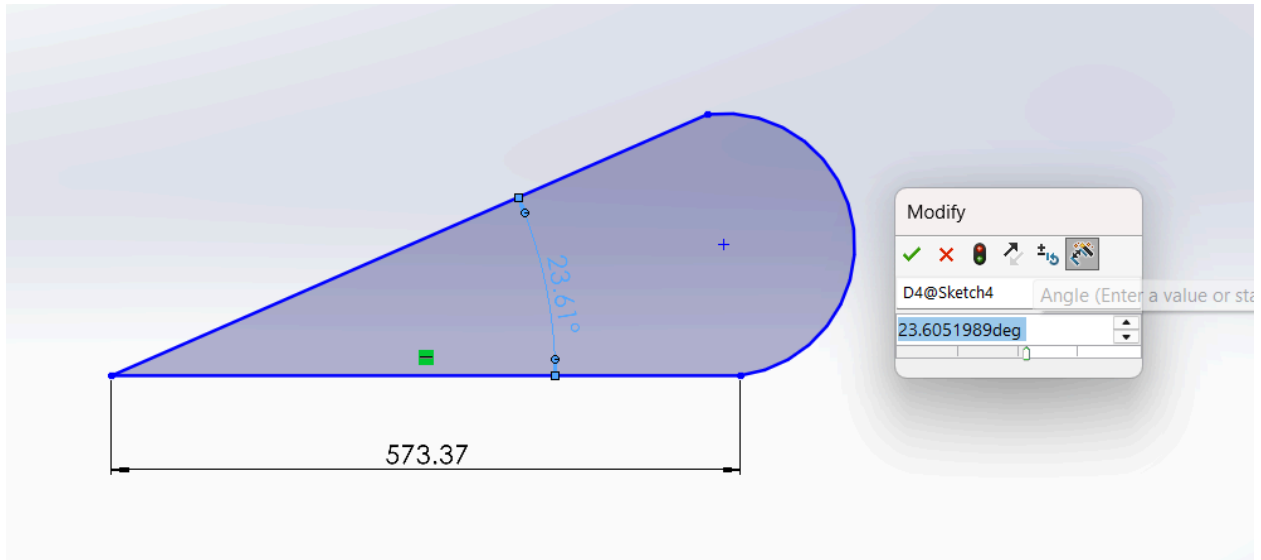


Create the following sketch using two lines and one arc. Add a dimension to the horizontal line by selecting the two points, as shown.

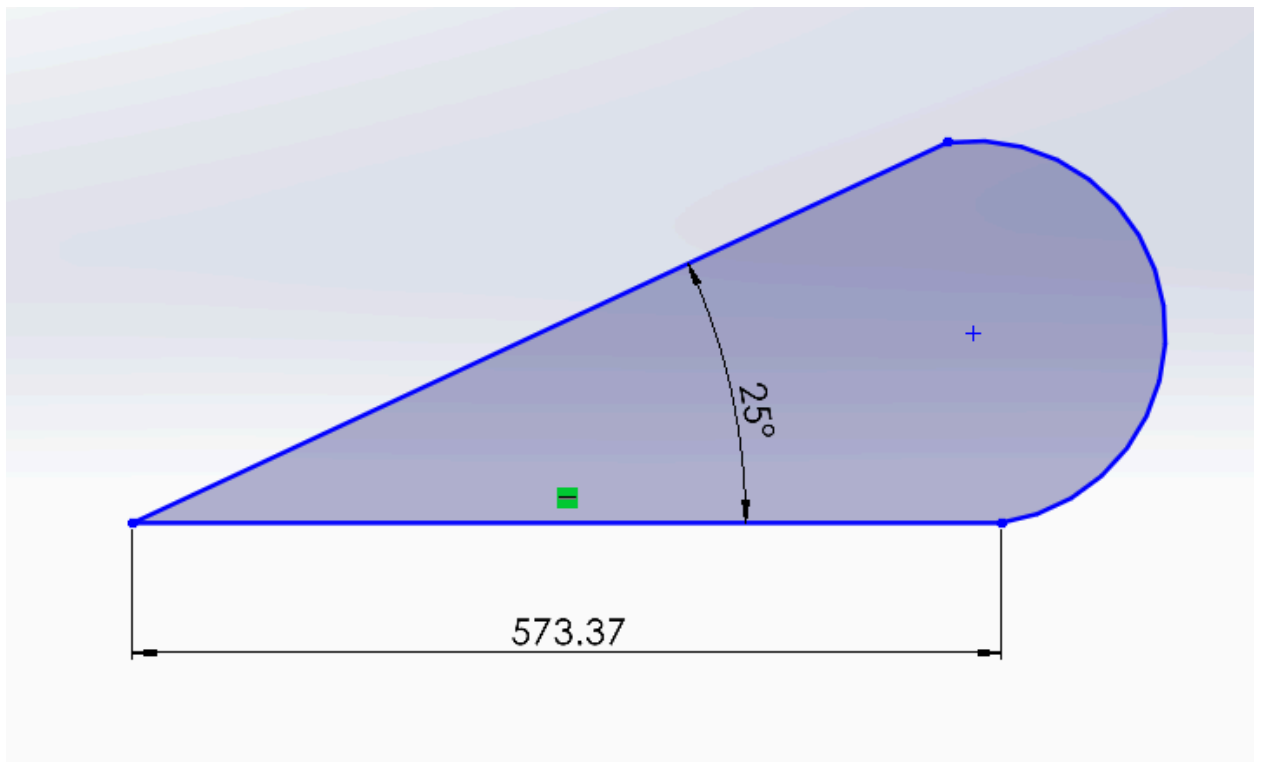


Select the **Smart Dimension** tool. Then select the first line and the second line. Move the mouse to place the angular dimension, and click again to confirm it.

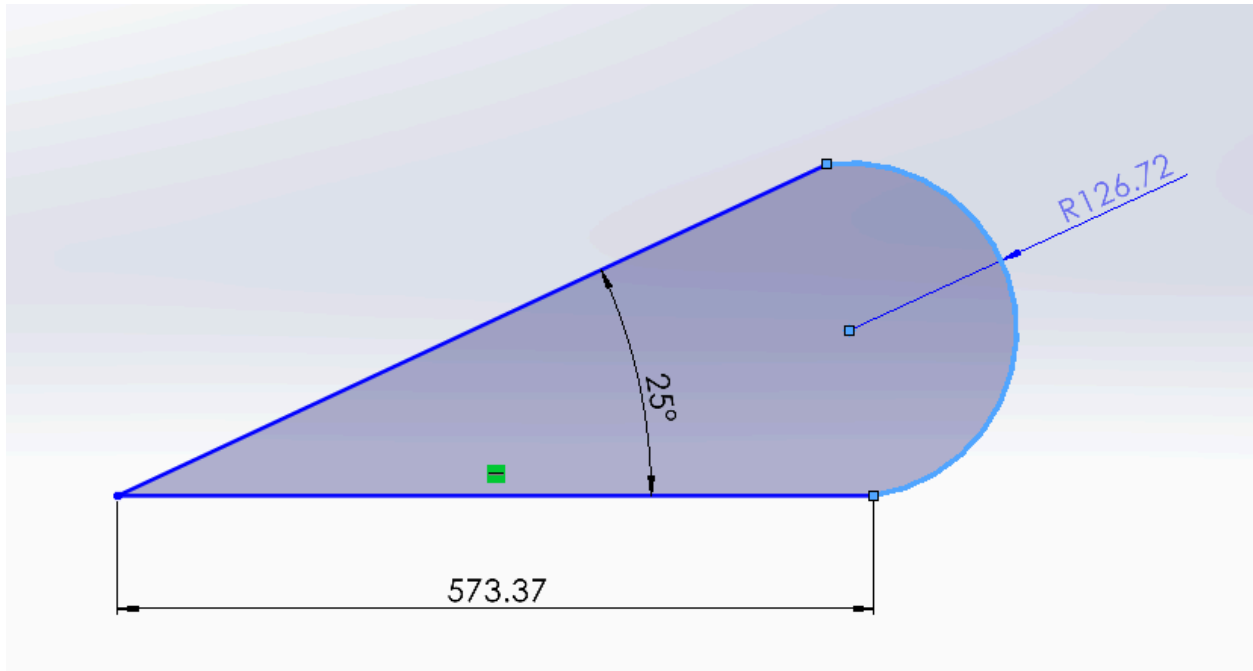




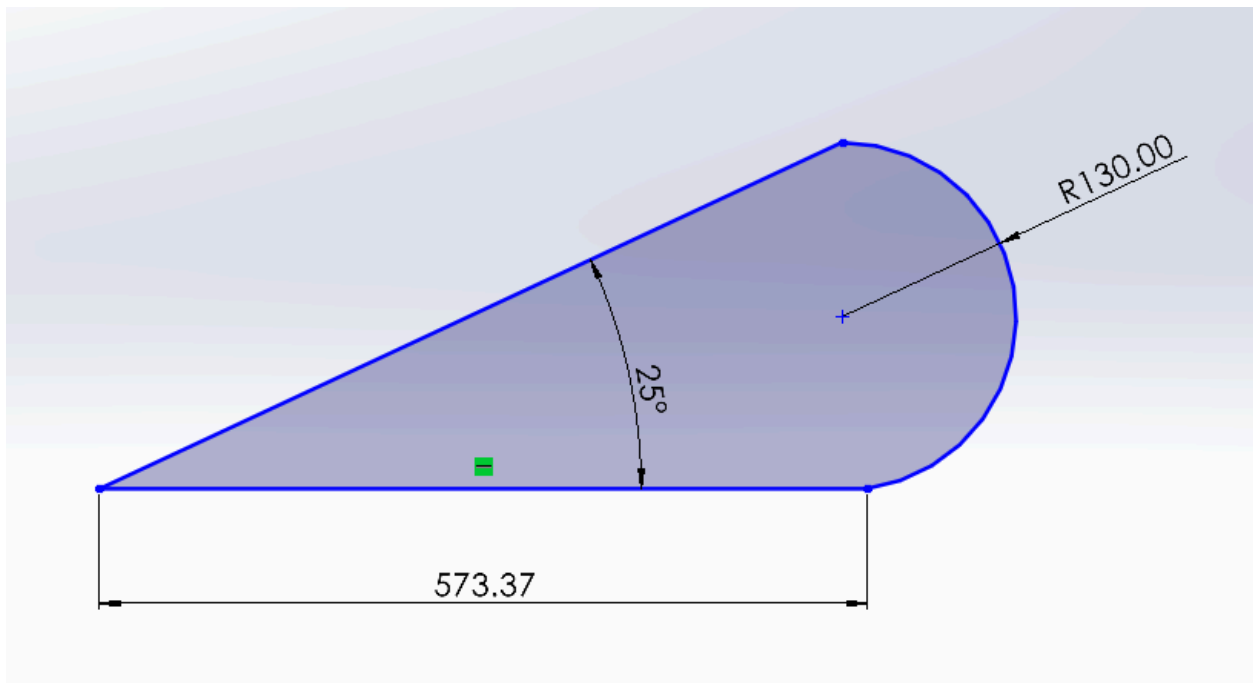
Double-click on the angle value. Then type **25°** and click **OK**.



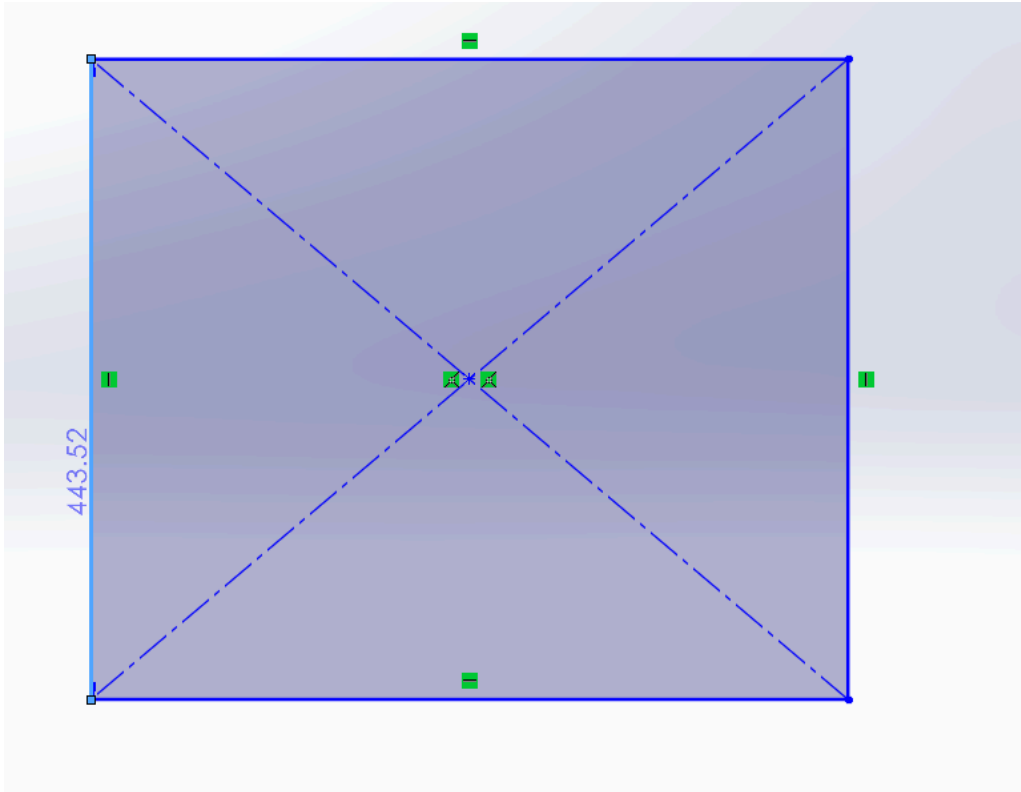
Select the **Smart Dimension** tool. Then click on the arc.



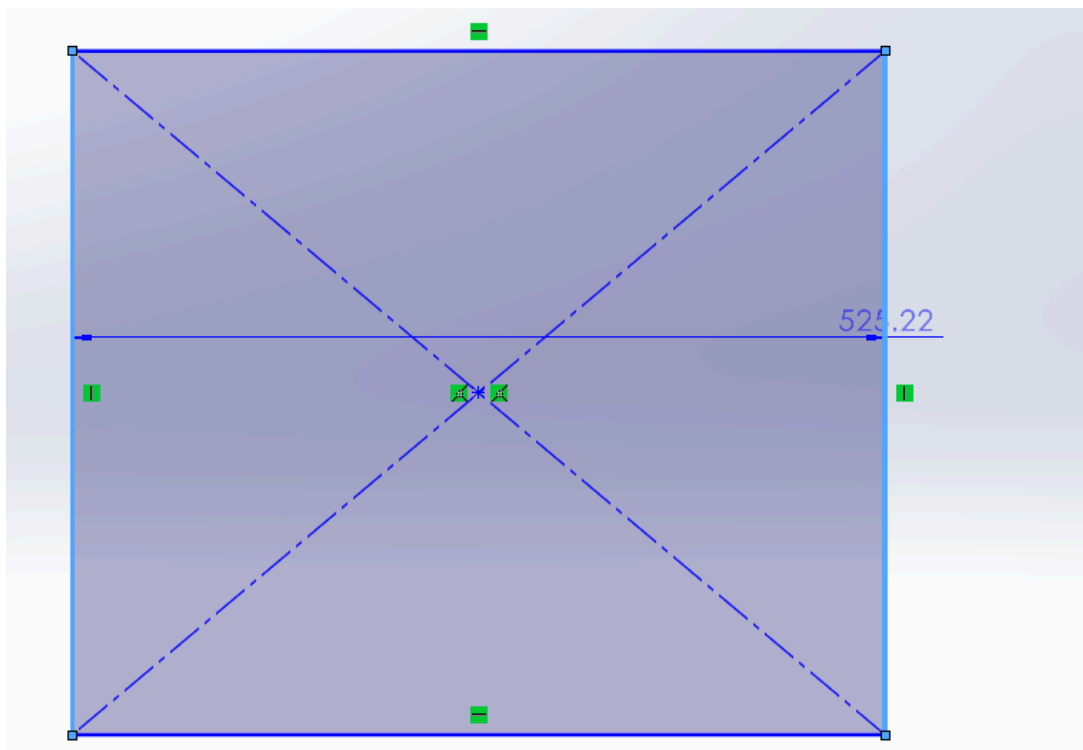
Set the radius value to **130 mm** and click **OK**.



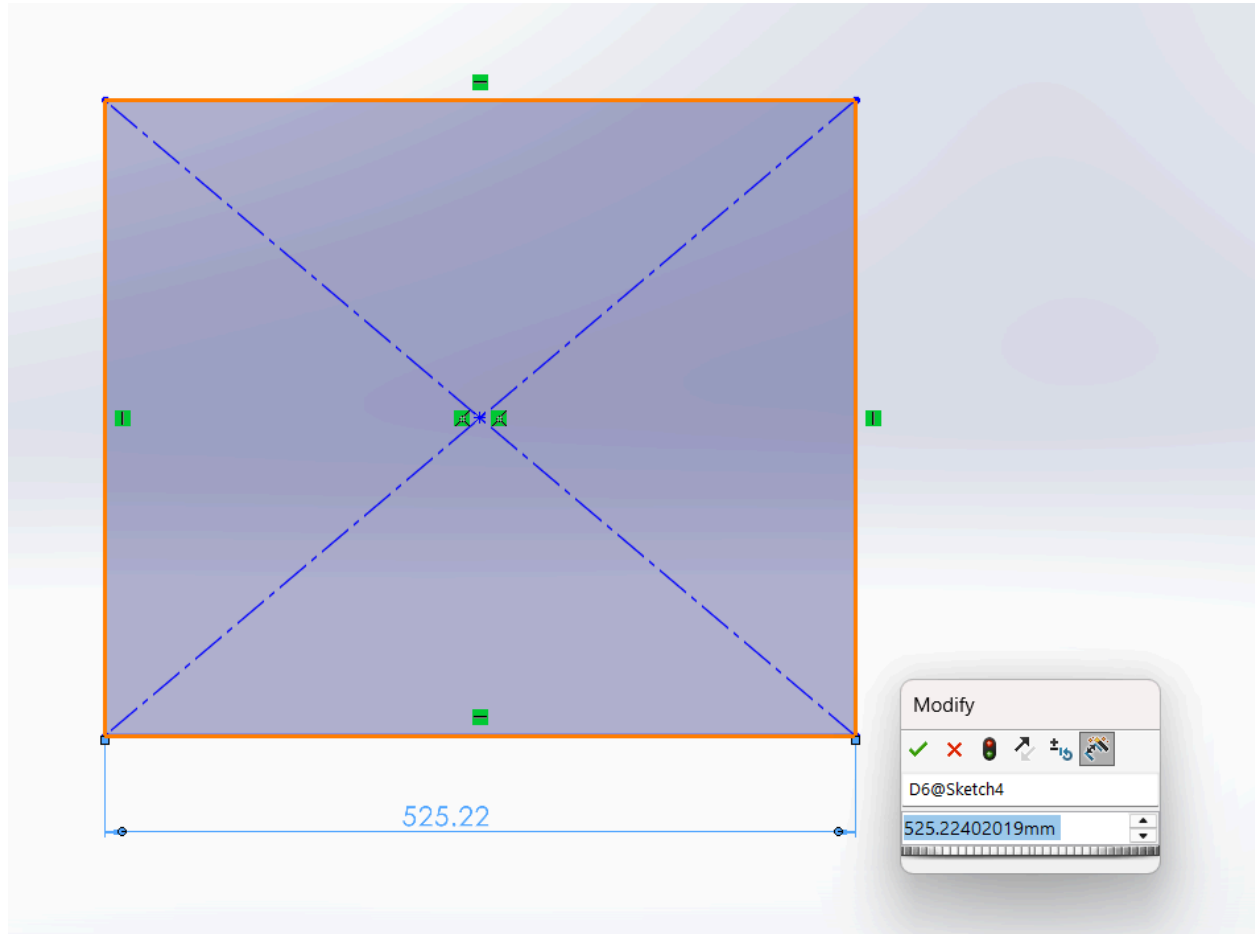
Draw a **Center Rectangle**. Then select the **Smart Dimension** tool and click on the left vertical line.



Now select the right vertical line.

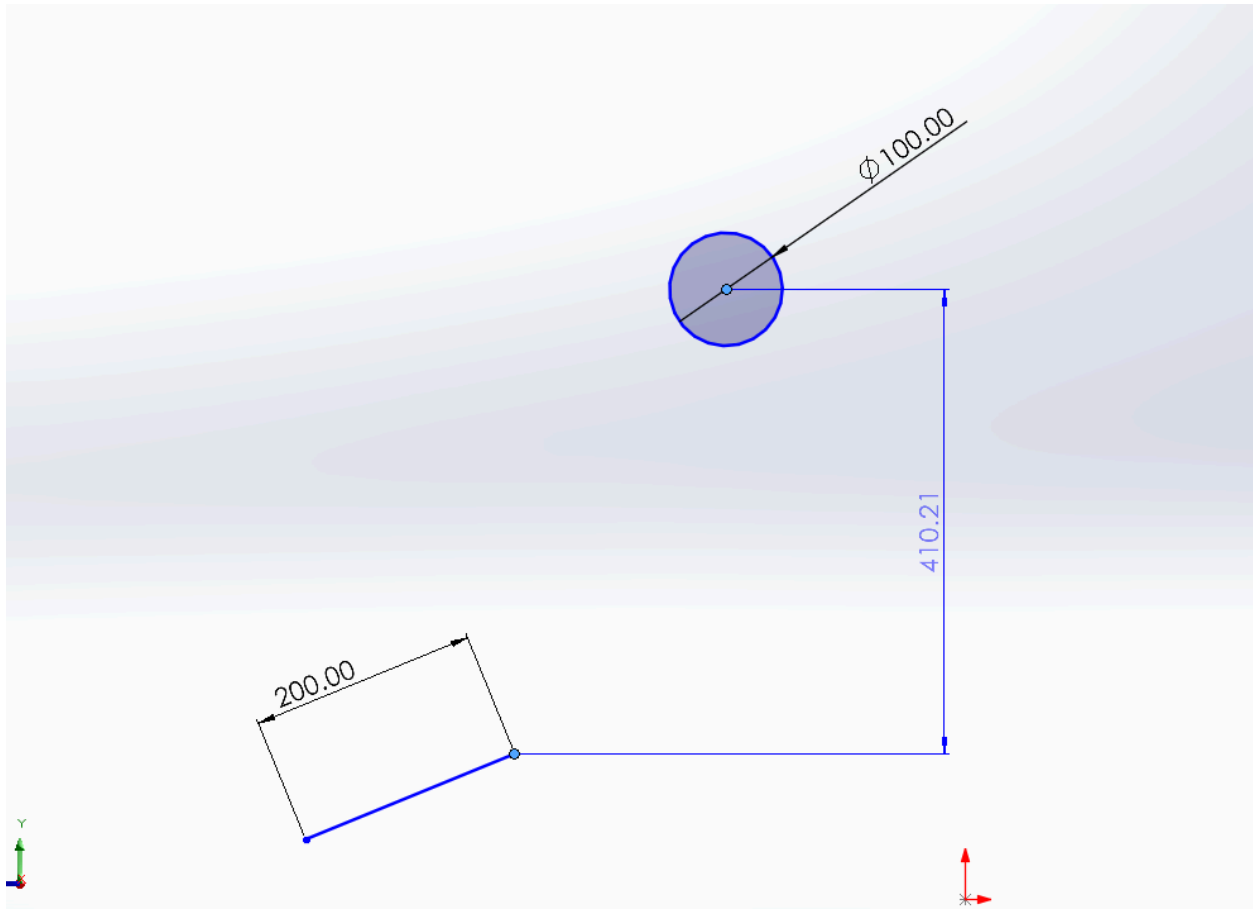


Drag the dimension below the rectangle and click to place it.



When defining the dimension of a line, there are several ways to select it. You can select the line itself, select two points on the line, or select two vertical lines at the start and end of the line.

To define the distance between two different sketch entities, such as a circle and a line, select the **Smart Dimension** tool. Then click the center point of the circle and select a point on the line.



These are the main ways to define dimensions in a sketch. Whether you need to define a length, angle, radius, or diameter, you can use the same tool: **Smart Dimension**.

8. Sketch Relations

What it does: Defines geometric relationships between sketch entities.

Common relations include:

- Coincident
- Horizontal
- Vertical

- Parallel
- Perpendicular
- Tangent
- Concentric
- Equal

How it works:

To fully define a sketch, we first need to define relations between sketch entities. Common sketch relations include:

Parallel — when two lines are parallel.

Perpendicular — when two lines are perpendicular.

Coincident — when two sketch entities share the same point.

Tangent — when two sketch entities touch with a smooth tangent connection.

Vertical — when a line is vertical.

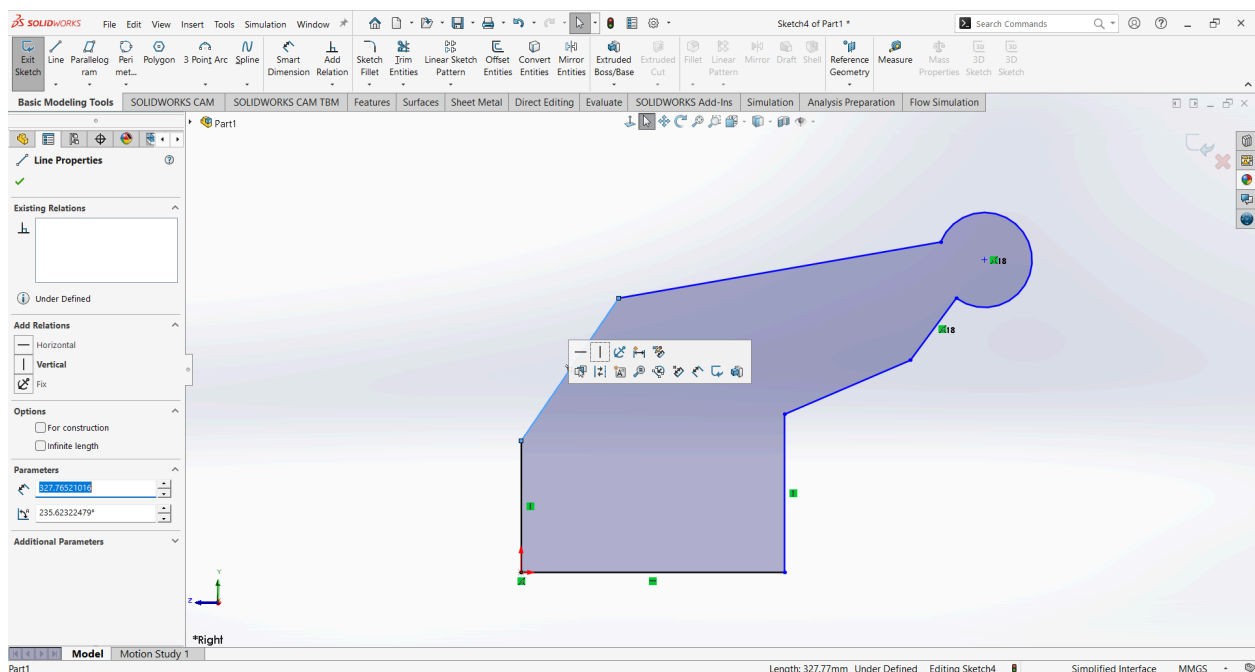
Horizontal — when a line is horizontal.

Concentric — when two arcs or circles have the same center point.

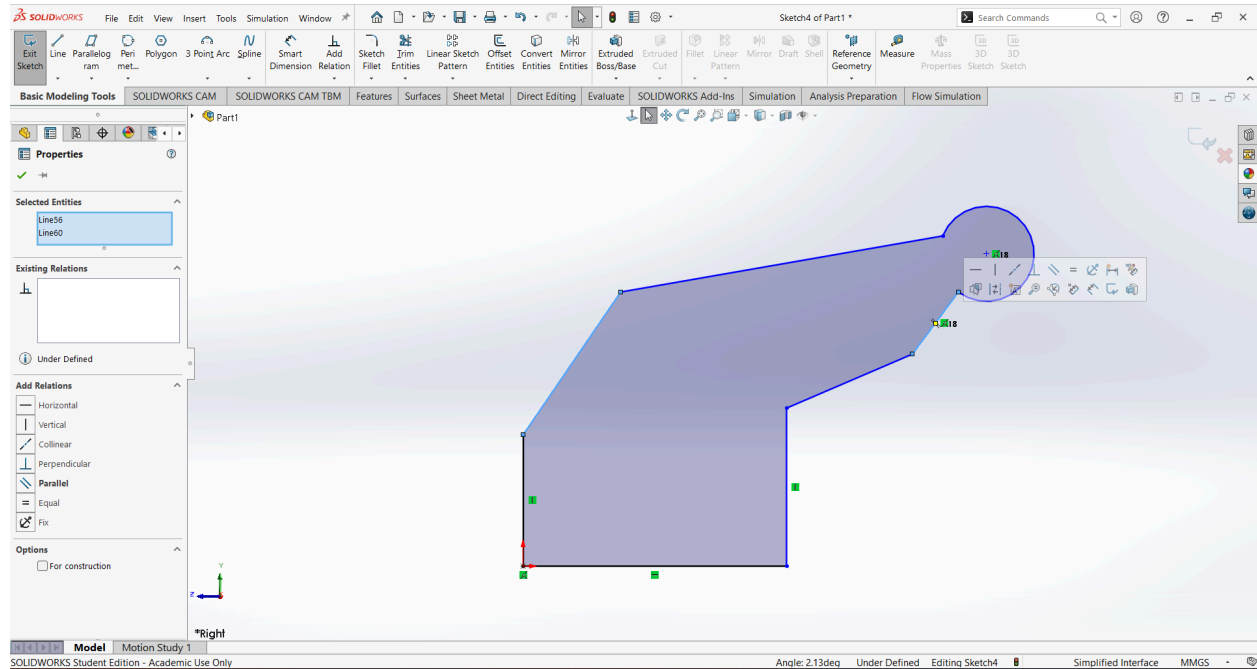
Coradial — when two arcs or circles have the same radius.

Collinear — when two lines lie on the same straight line.

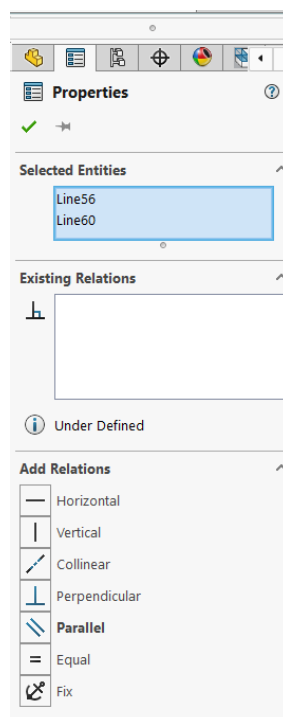
If you click on a line, the **Add Relation** options appear on the left side in the PropertyManager. Depending on the selected entity, different relation options will be available. In this example, we can make the line **Vertical** or **Horizontal**.



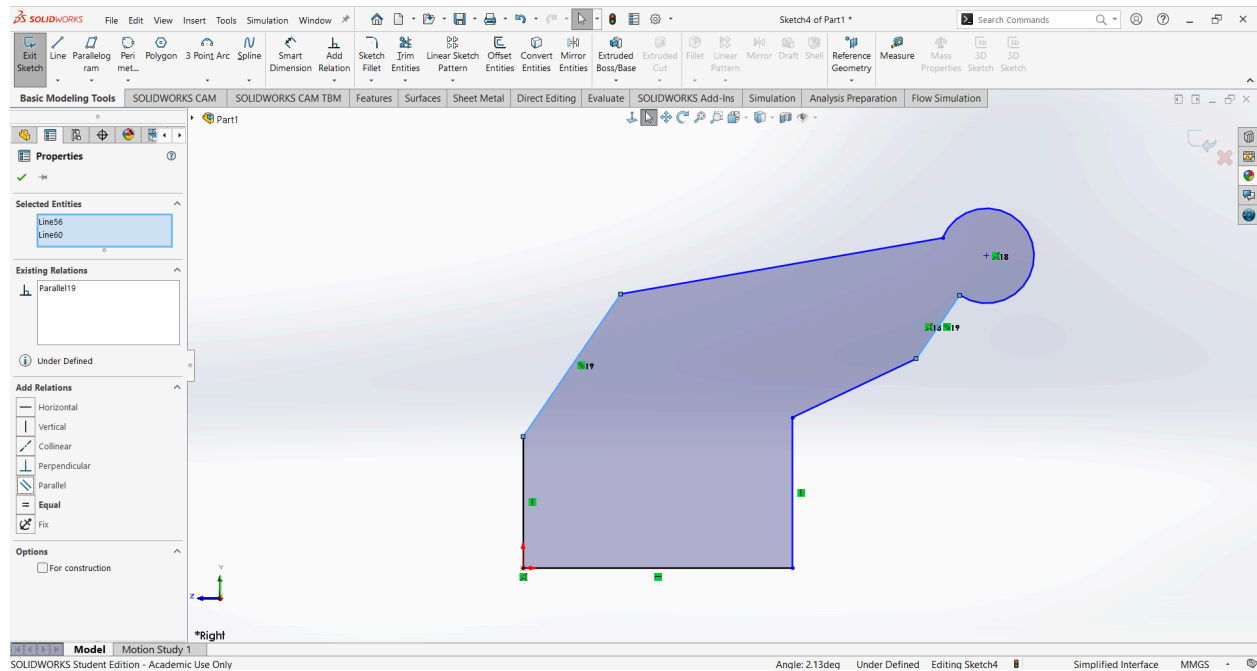
When you click on one line, hold **CTRL**, and then click on the second line, more relation options appear on the left side in the **PropertyManager**.



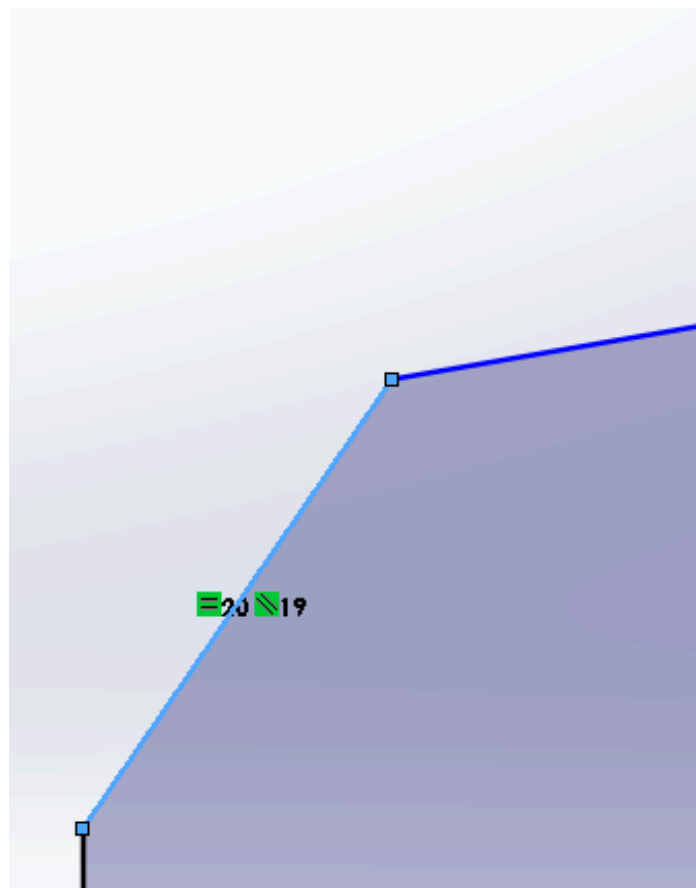
Here you can see the available sketch relations. These relations can be added depending on which sketch entities are selected.



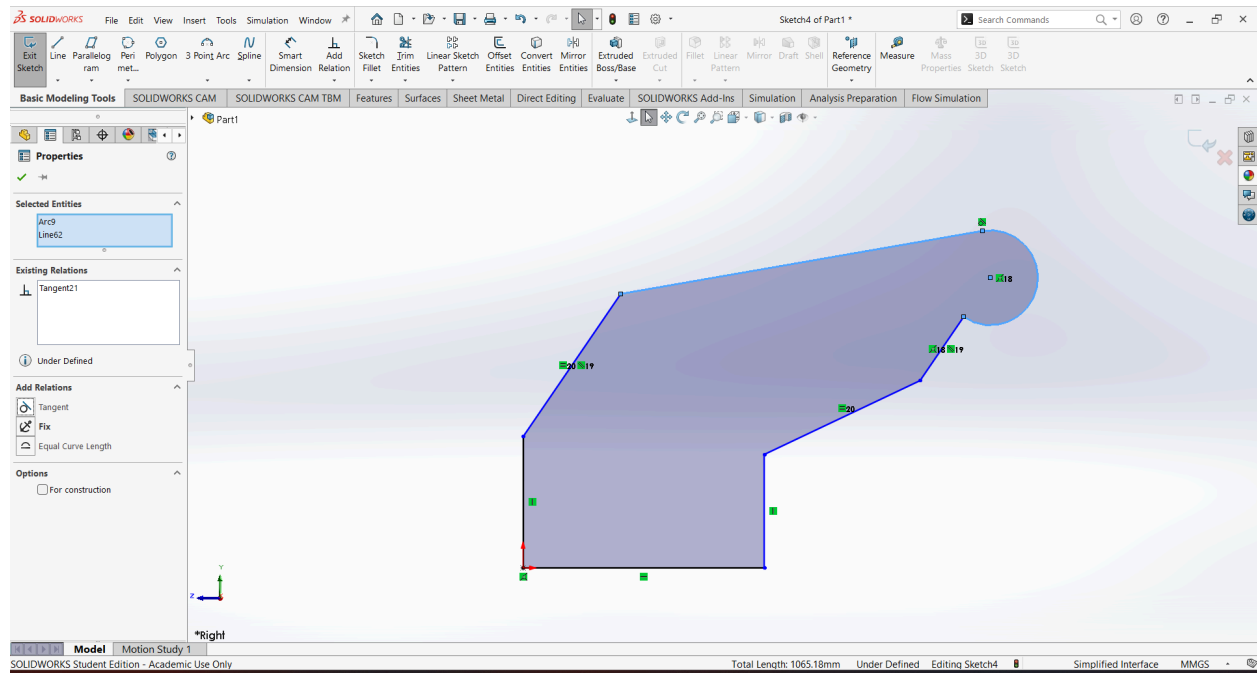
We will choose the **Parallel** relation. This will make the selected lines parallel to each other.



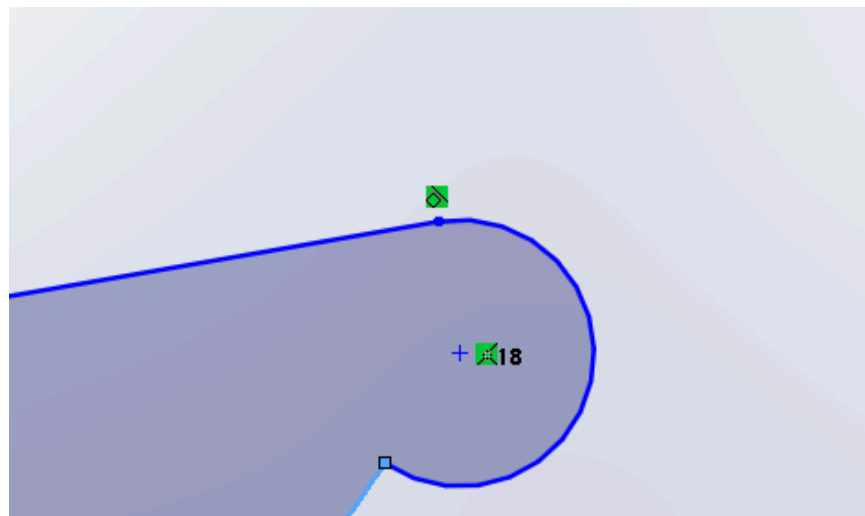
This means that the two selected lines are now parallel. You can also see the small green relation icon next to the lines, which shows that the **Parallel** relation has been applied.



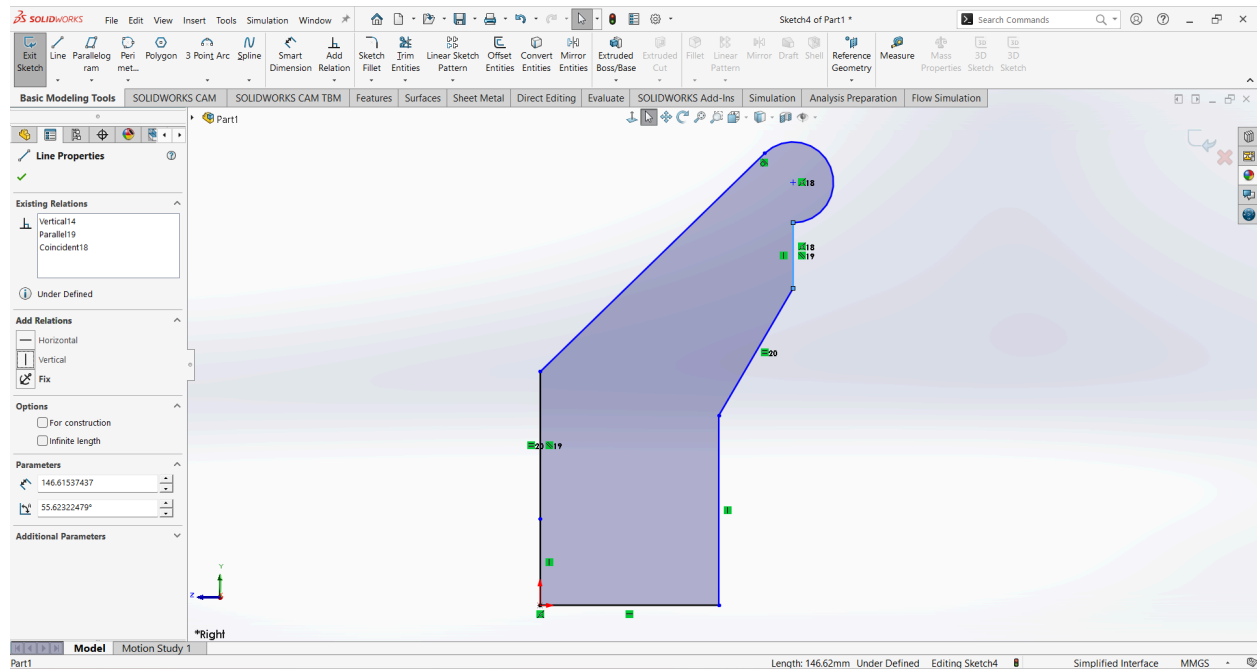
Select the upper line, hold **CTRL**, and then select the arc. On the left side, in the **PropertyManager**, choose the **Tangent** relation and click **OK**.



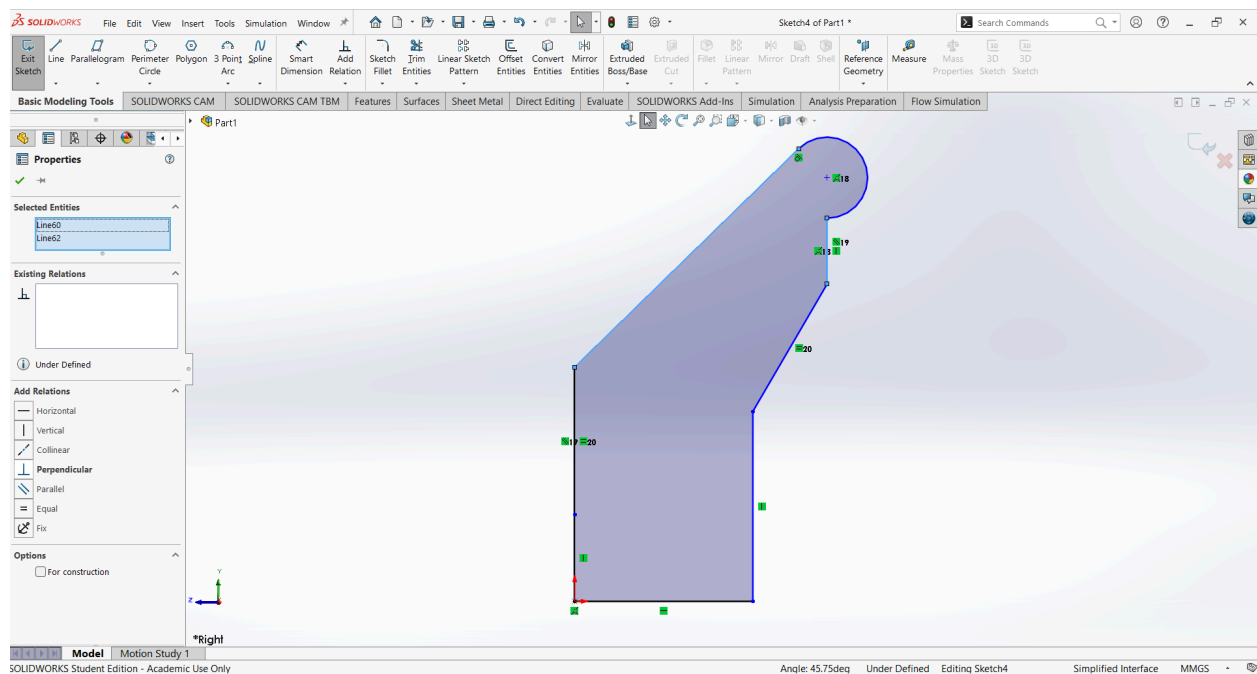
Now we have a tangent relation between the line and the arc. You can see the small green tangent relation icon, which shows that the **Tangent** relation has been applied.



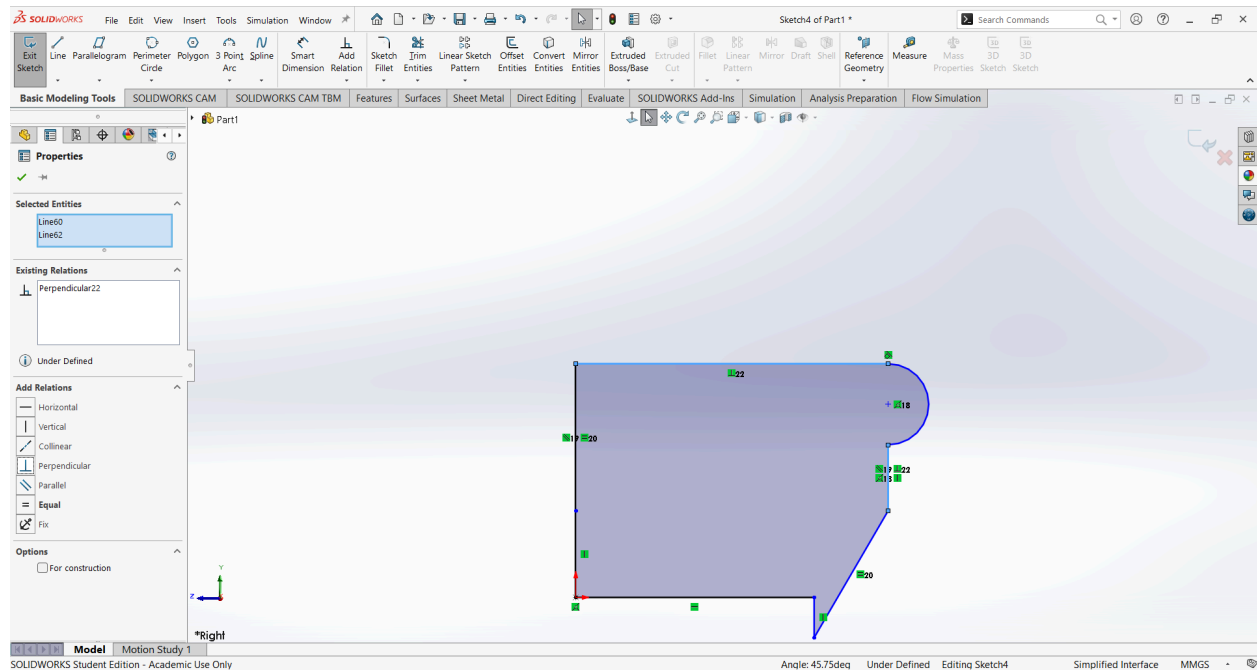
Select the line below the arc. Then add the **Vertical** relation to make the line vertical.



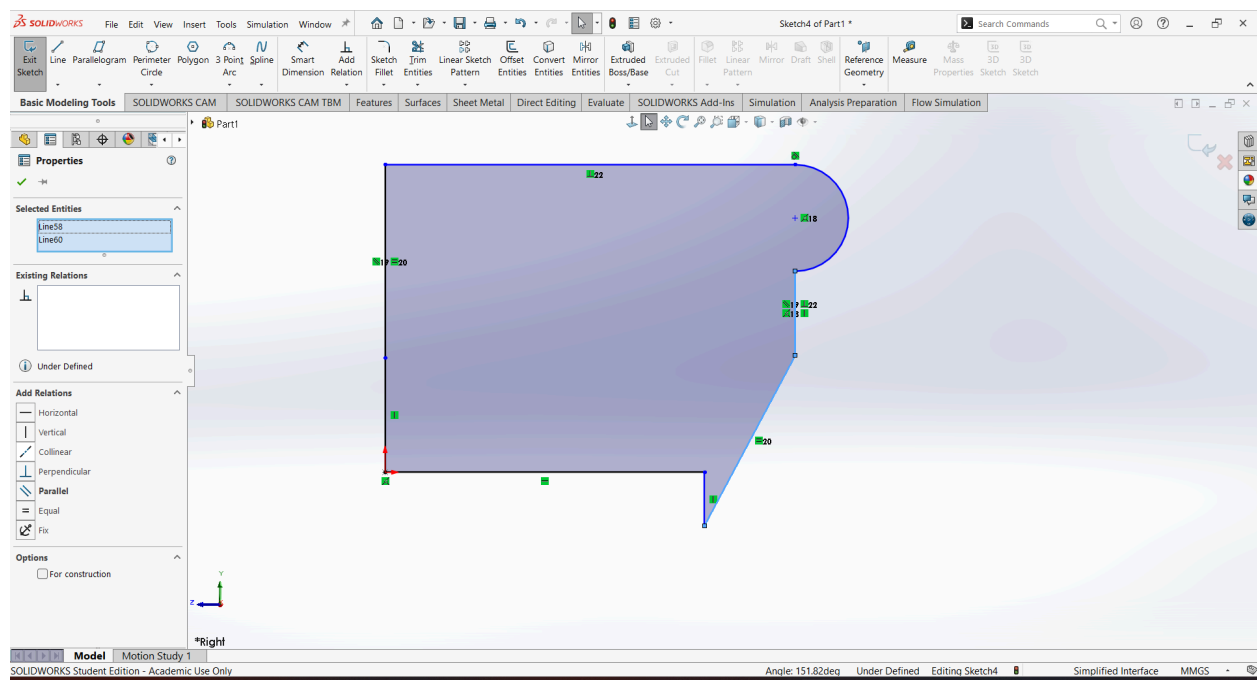
Now select the two following lines while holding **CTRL**.



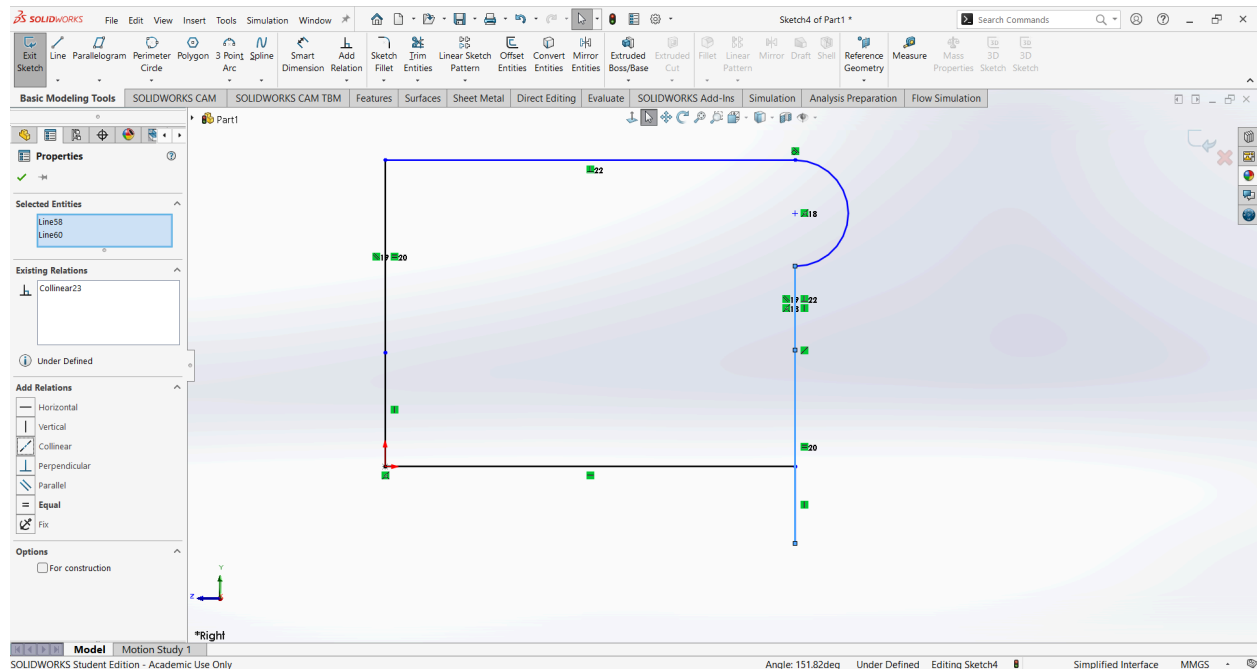
Choose the **Perpendicular** relation. Now these two lines are perpendicular to each other.



Now select the two lines below the arc while holding **CTRL**.

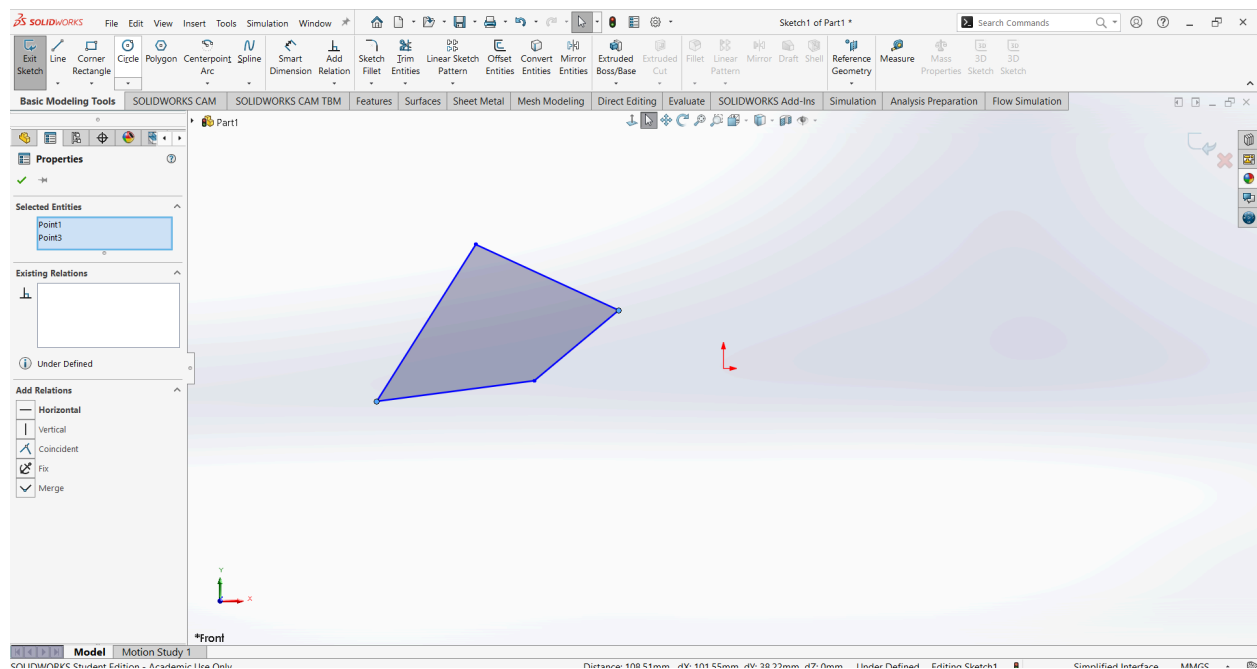


Choose the **Collinear** relation. This means that the selected lines are placed on the same straight line.

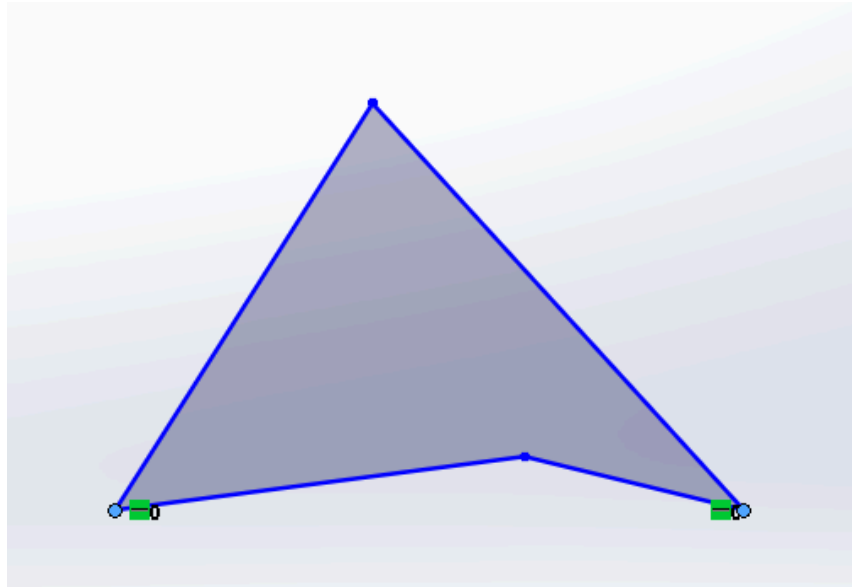


You can also select two points while holding **CTRL** and apply either a **Vertical** or **Horizontal** relation. This means the two selected points will be aligned on the same vertical or horizontal line.

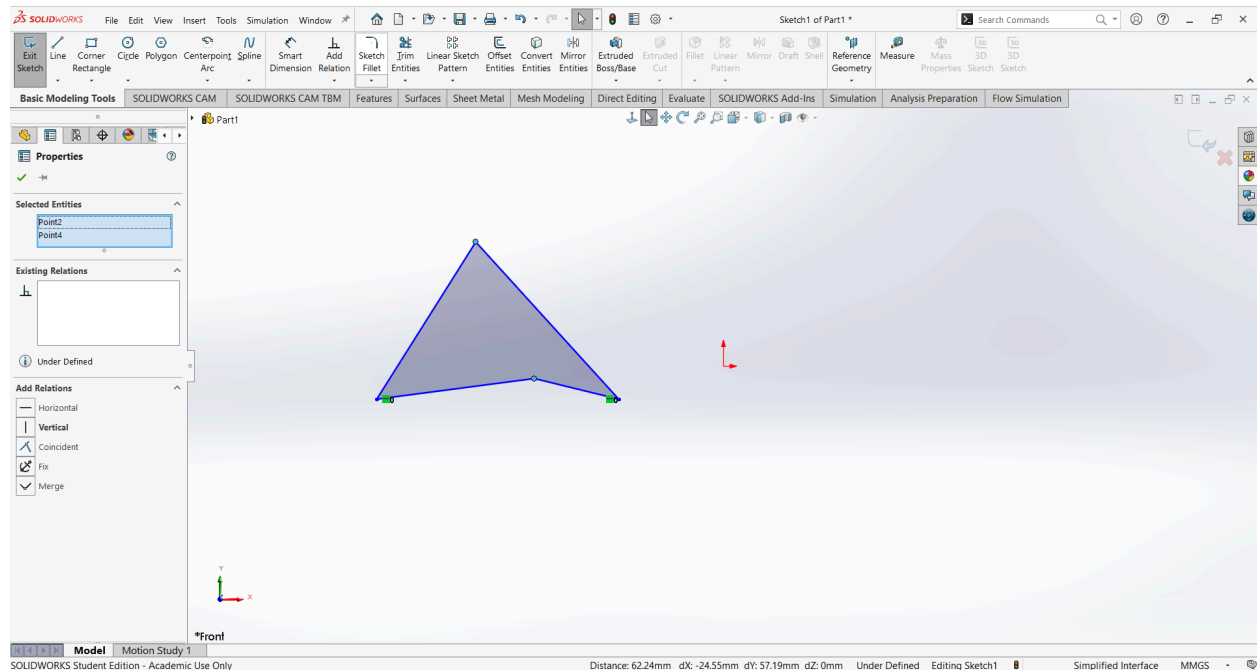
Select the first point. Then hold **Ctrl** on the keyboard and select the second point. Make sure both points stay selected at the same time.



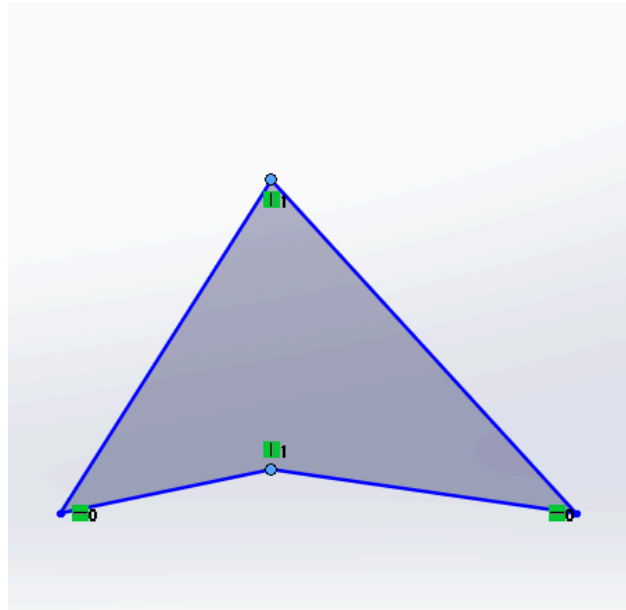
In the **PropertyManager** on the left side, select **Horizontal**. Now the two points are horizontally aligned. A green **Horizontal** relation symbol appears next to them.



Select the first line. Then hold **Ctrl** on the keyboard and select the second line. Make sure both lines stay selected at the same time.



In the **PropertyManager**, select the **Vertical** relation. Now the two lines are vertically aligned. A green **Vertical** relation symbol appears next to them.



Now both lines are vertical. You can see the small green **Vertical** relation symbol next to them.

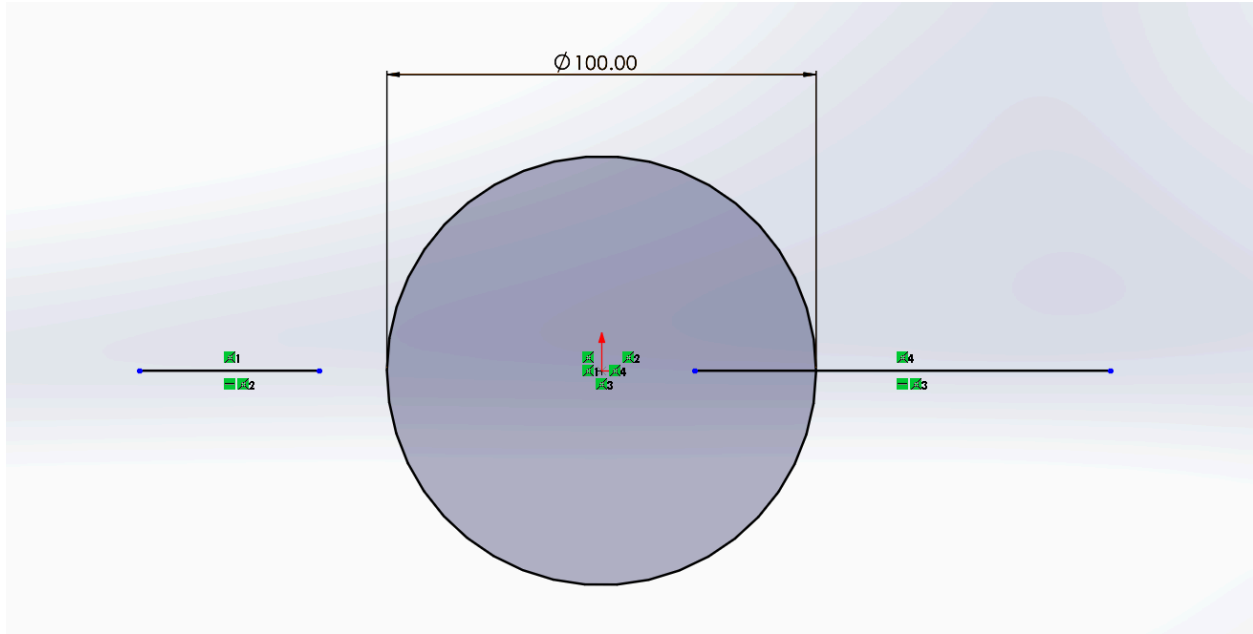
9. Trim Entities

What it does: Removes unwanted portions of sketch geometry.

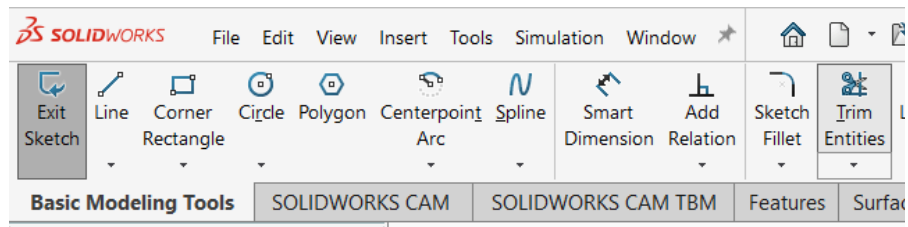
Use it for: Cleaning intersecting geometry and creating final profiles.

How it works:

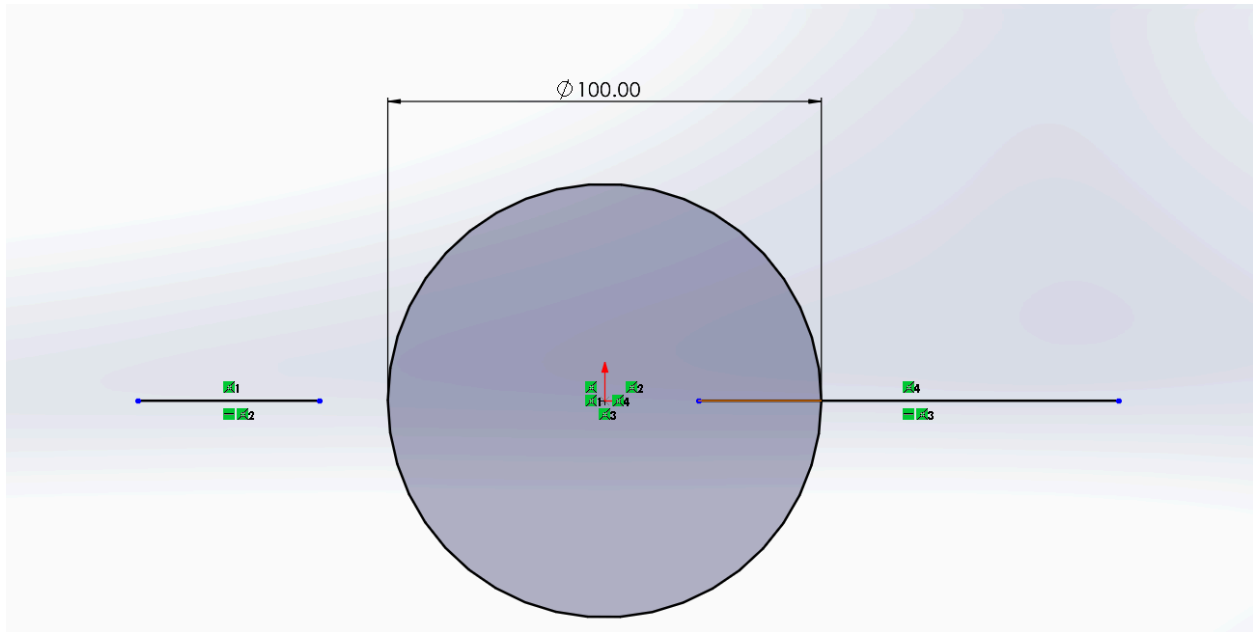
To extend or trim a line, we use the **Trim Entities** or **Extend Entities** tool.



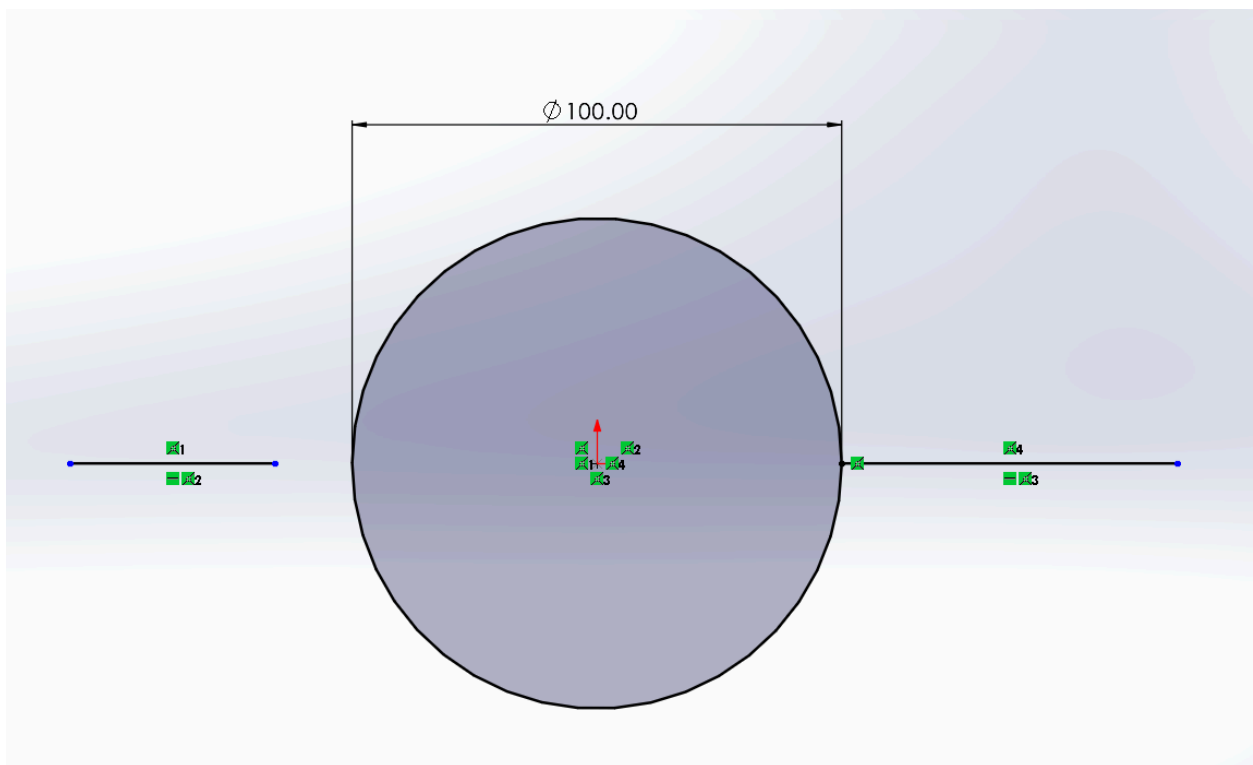
On the right side, there is a line that we want to trim.



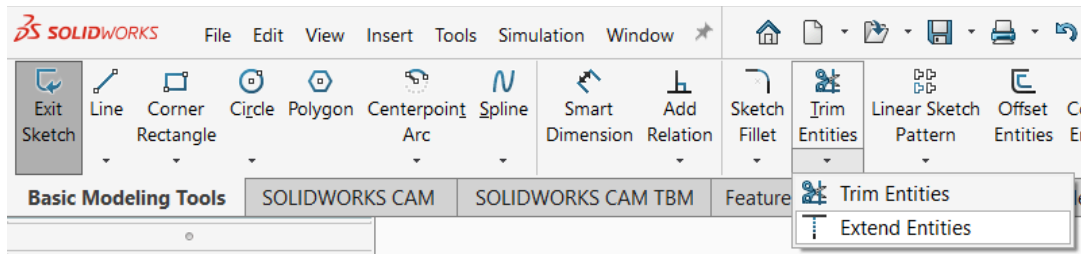
Select the **Trim Entities** tool, then click the part of the line on the left side of the circle.



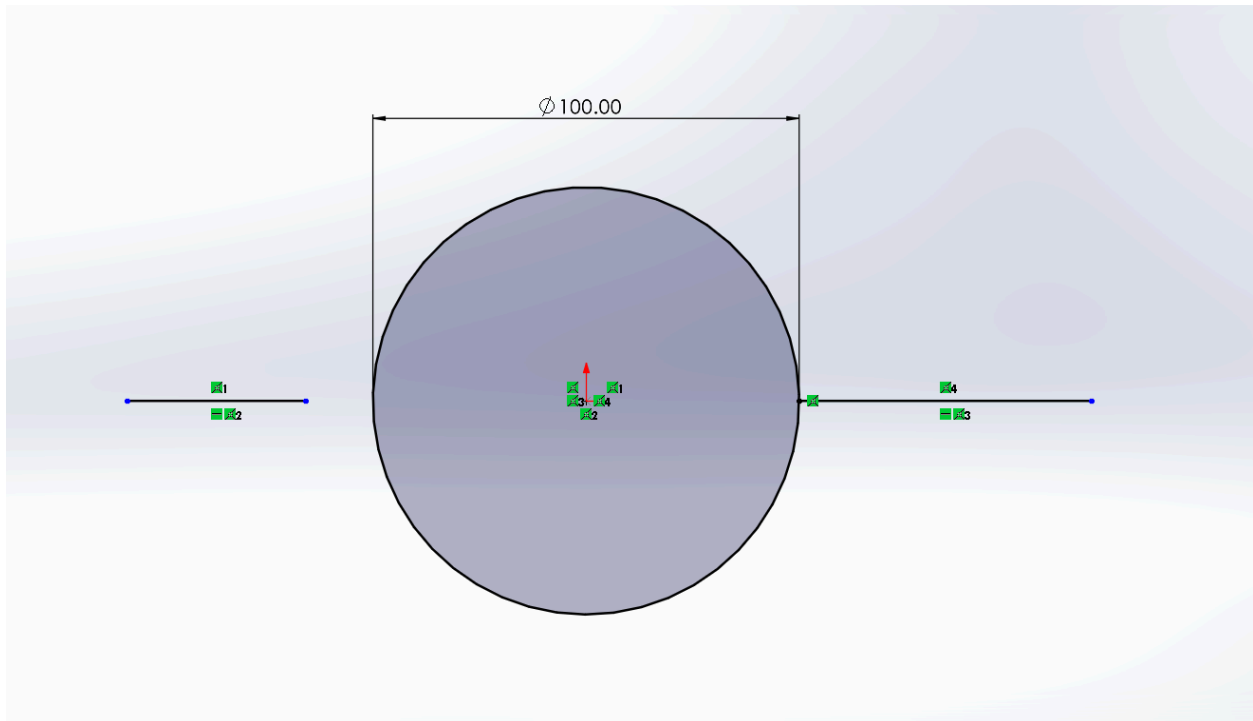
This trims the line up to the circle.



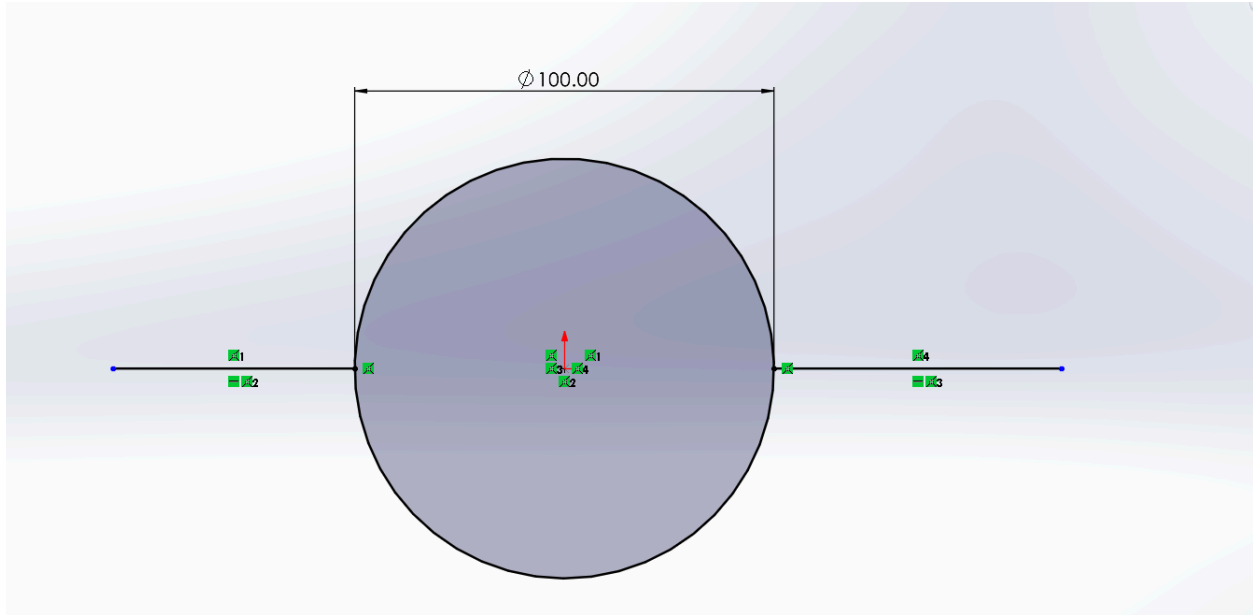
On the left side, there is a line that we want to extend to the circle. Click the small arrow below the **Trim Entities** tool, then select **Extend Entities**.



Click the line on the left side of the circle.



Now this line has been extended to the circle.



The **Trim Entities** and **Extend Entities** tools can be used with different sketch entities, not only lines. You can also use them with circles, arcs, splines, and other sketch geometry.

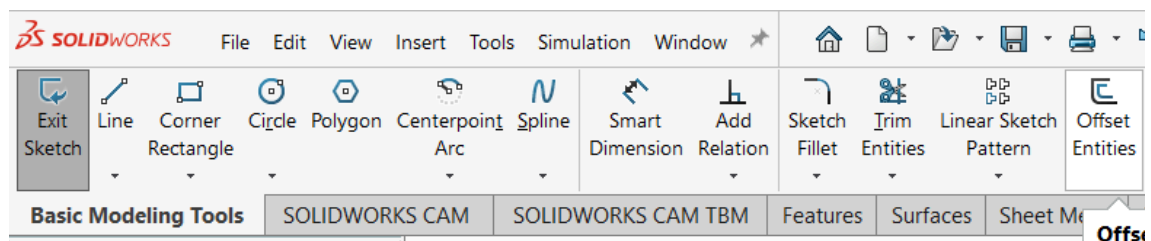
10. Offset Entities

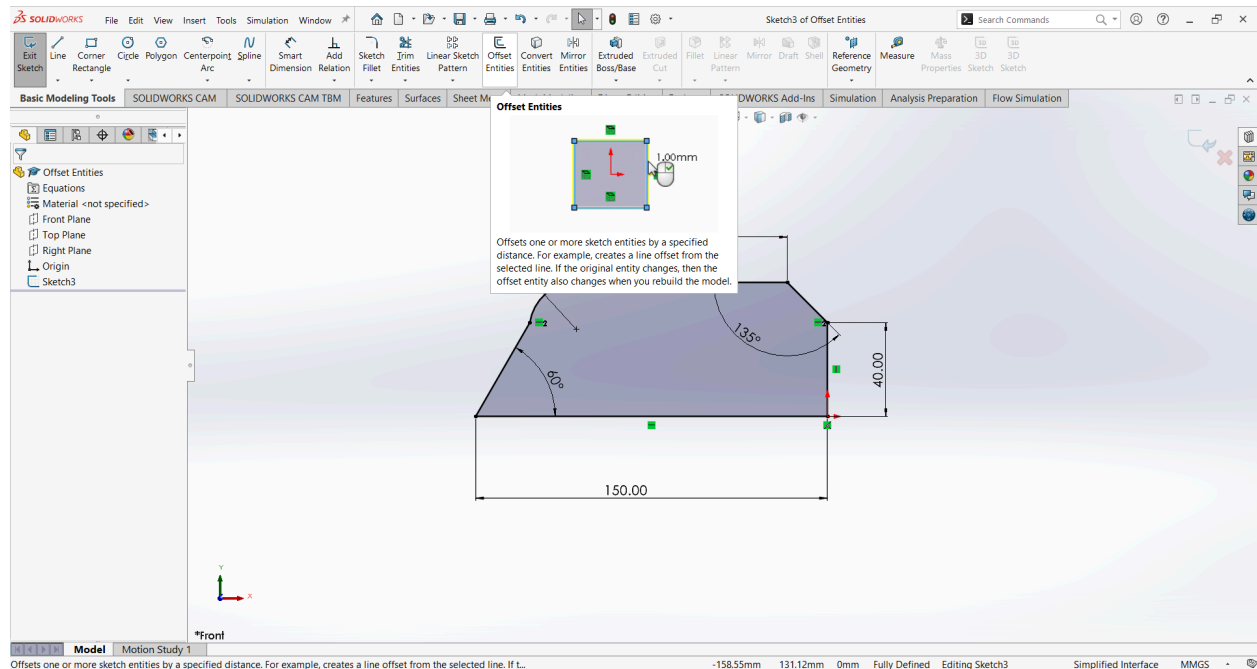
What it does: Creates offset copies of existing sketch geometry.

Use it for: Wall thicknesses, borders, grooves, and parallel profiles.

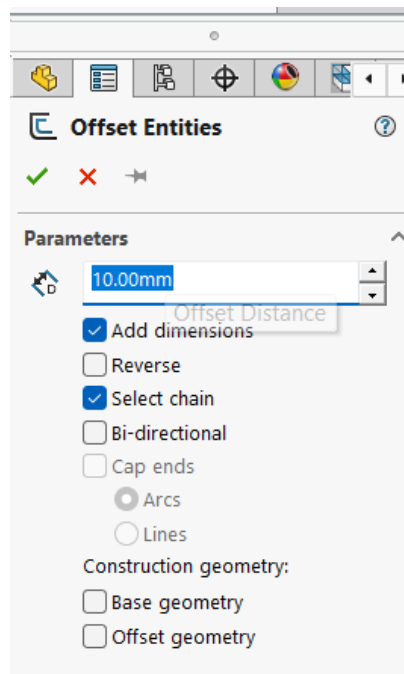
How it works:

When you have a sketch and want to create the same sketch with an offset distance, either bigger or smaller, use the **Offset Entities** tool. Select **Offset Entities**.

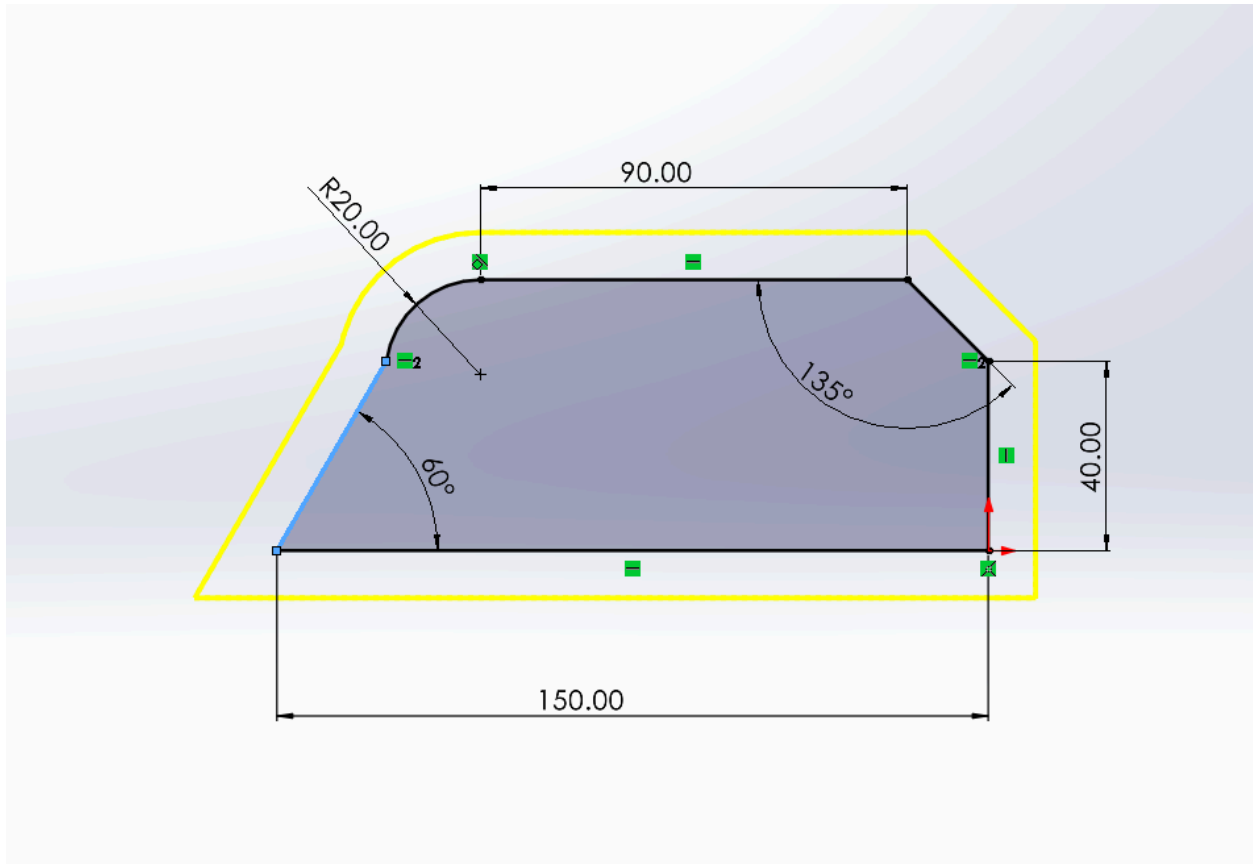




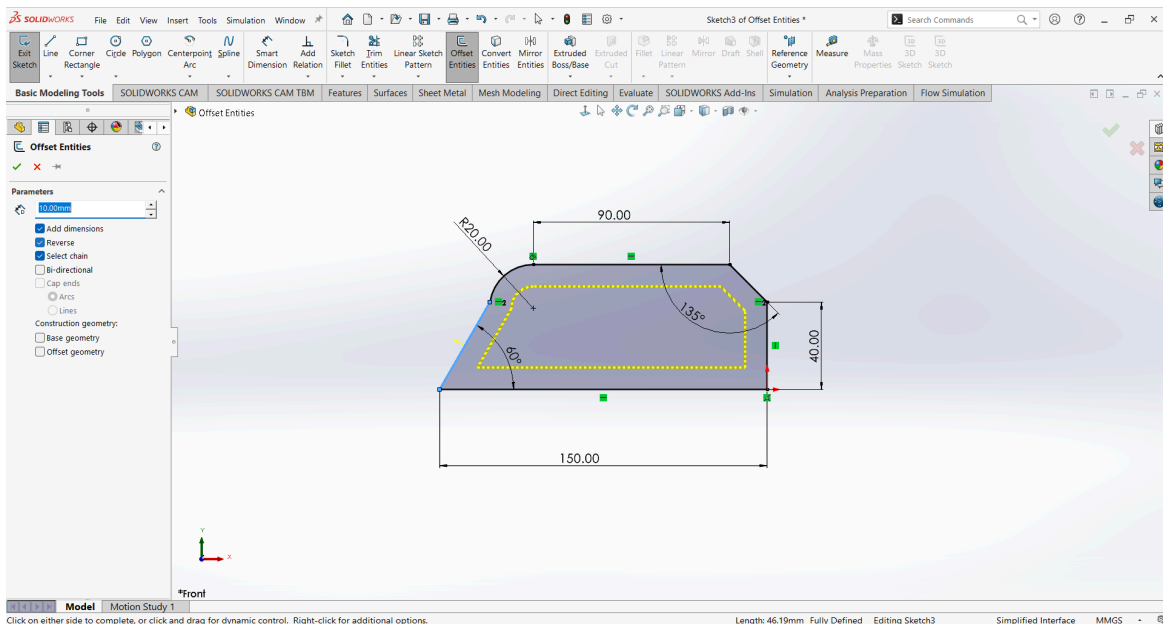
On the left side, in the **PropertyManager**, you can define the offset distance value.

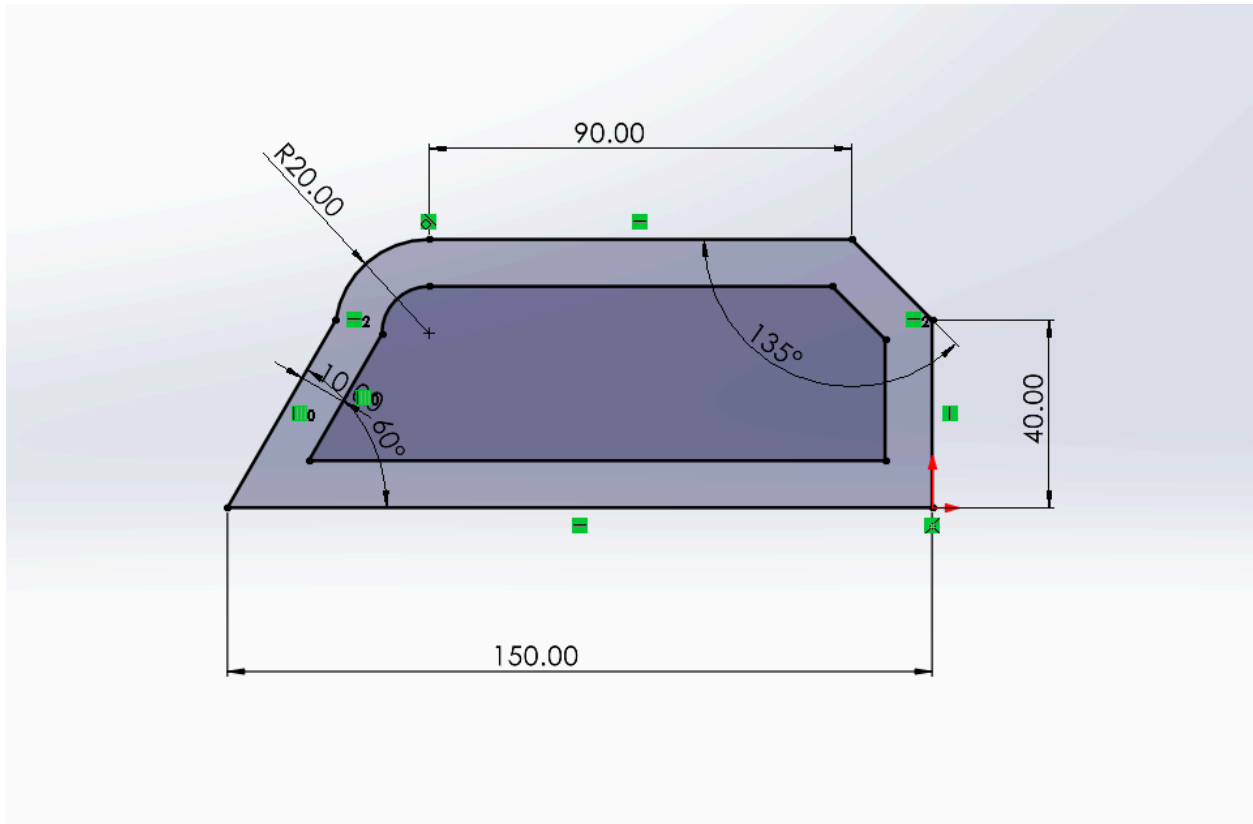


You can choose to create the offset to the outside of the sketch.



Or, on the left side in the **PropertyManager**, click **Reverse** to create the offset toward the inside of the sketch.





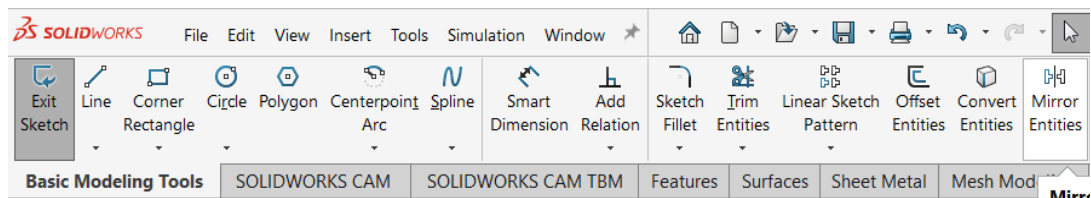
11. Mirror Entities

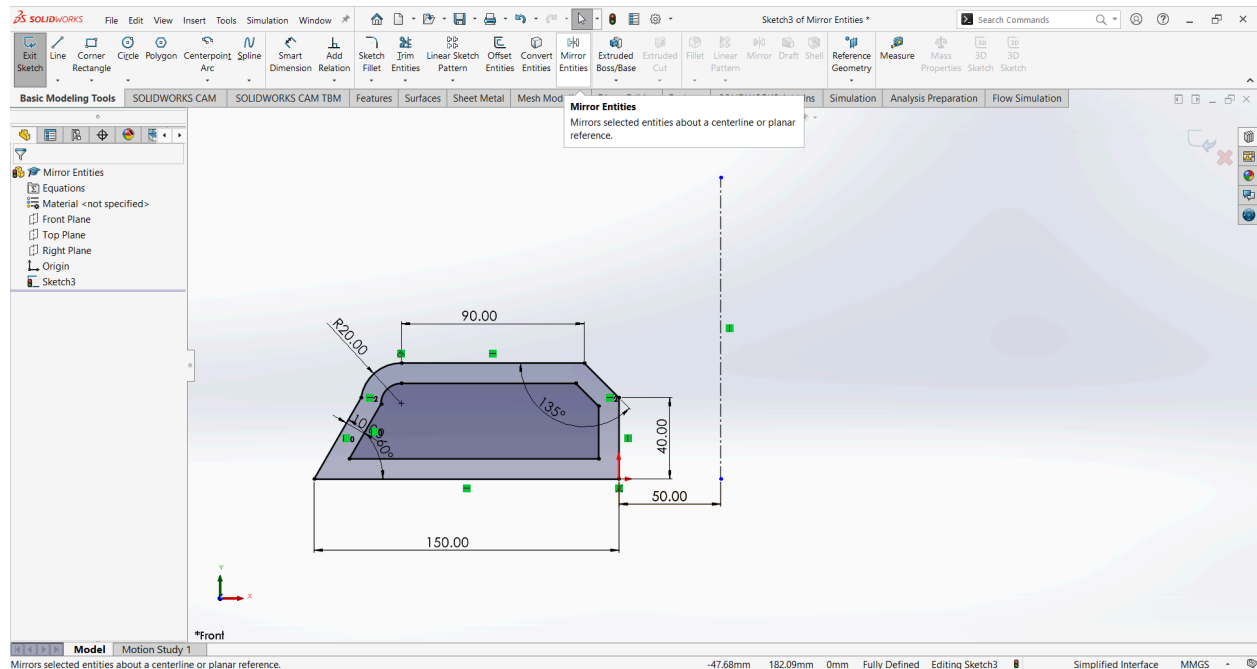
What it does: Mirrors sketch geometry around a centerline.

Use it for: Symmetrical sketches.

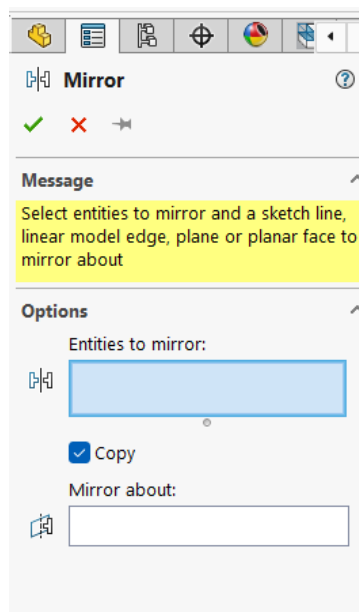
How it works:

When you have a sketch that you want to mirror about a specific line, use the **Mirror Entities** tool. In the image below, there is a sketch and a vertical centerline on the right side. We want to mirror the sketch about this centerline. Select **Mirror Entities**.

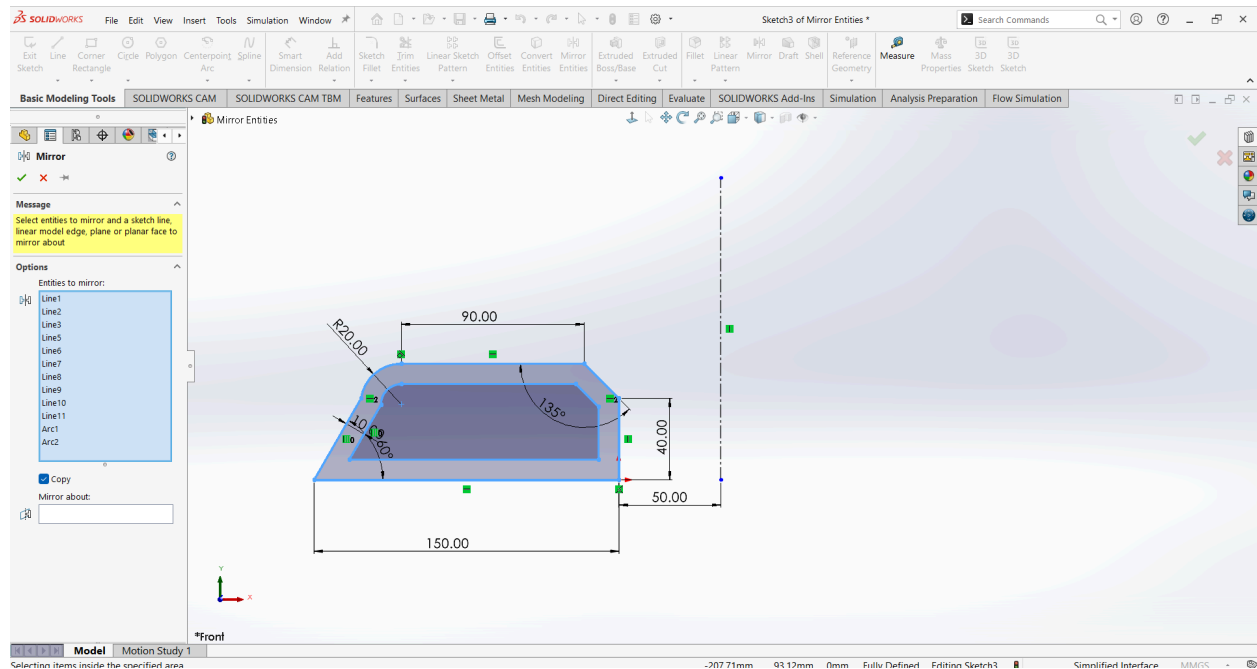




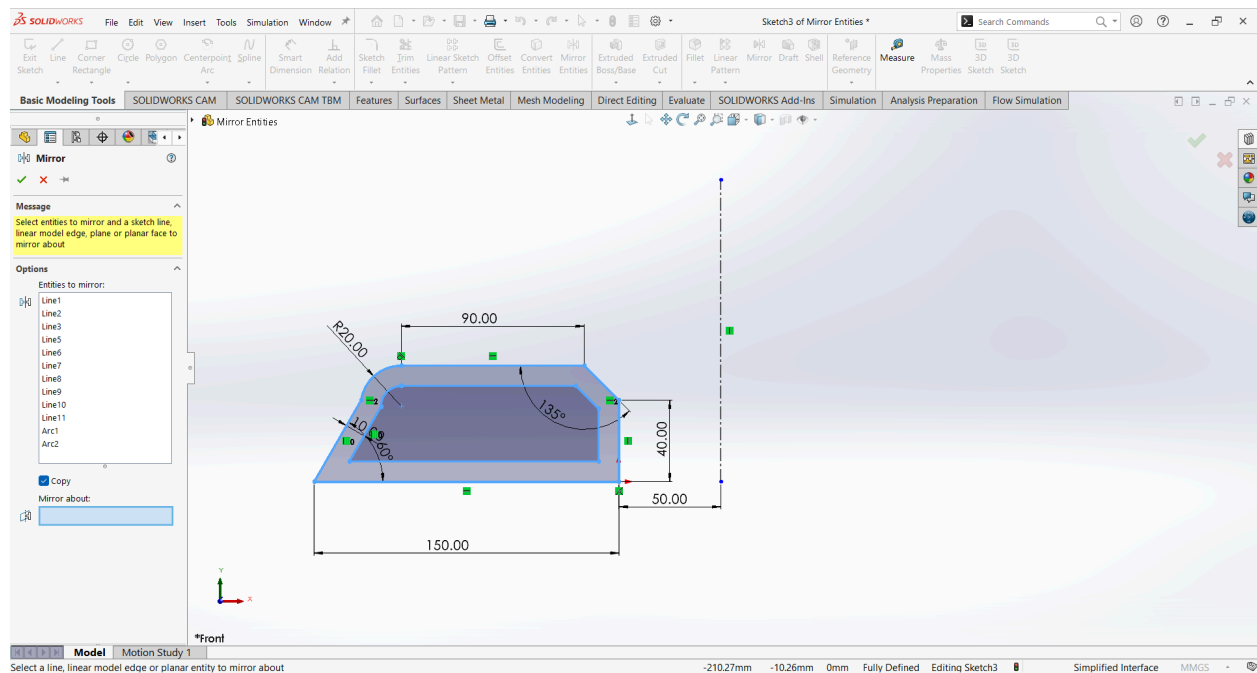
On the left side, two selection boxes appear in the **PropertyManager**. The first box is blue, which means it is active. In this box, select the entities you want to mirror. In this example, select the whole sketch.



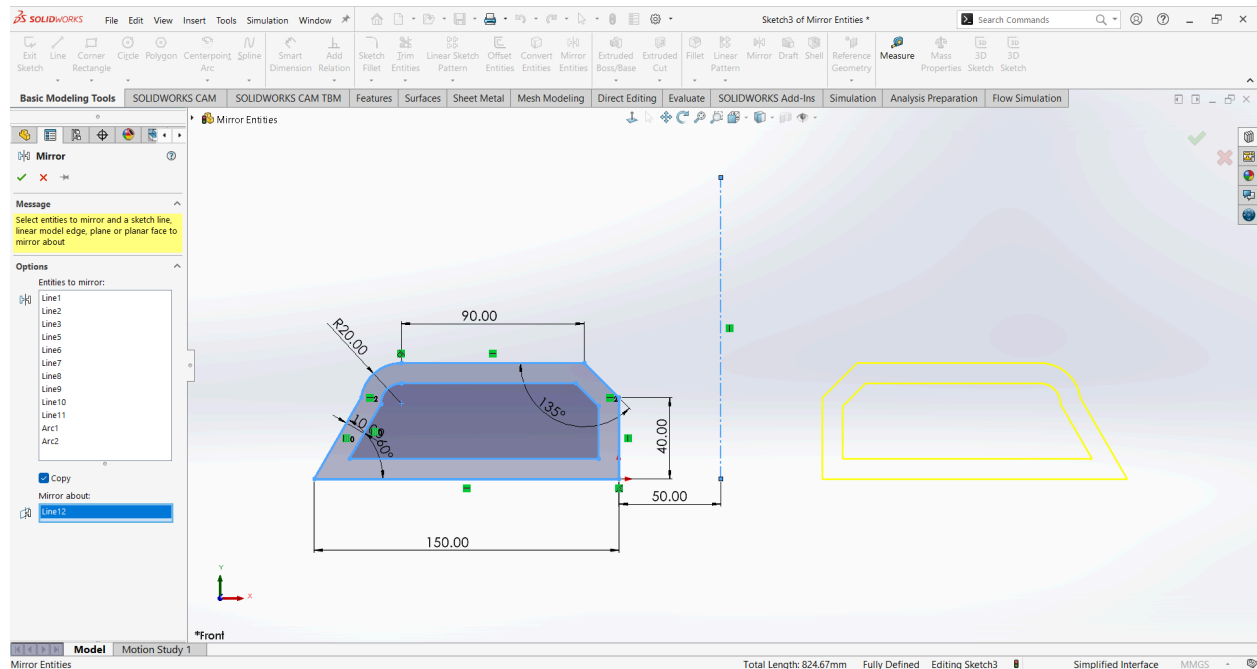
You can select each sketch entity one by one, or you can use box selection. To do this, click above the sketch on the left side, hold the mouse button, drag below the sketch, and release the mouse button. Make sure all sketch entities are selected.



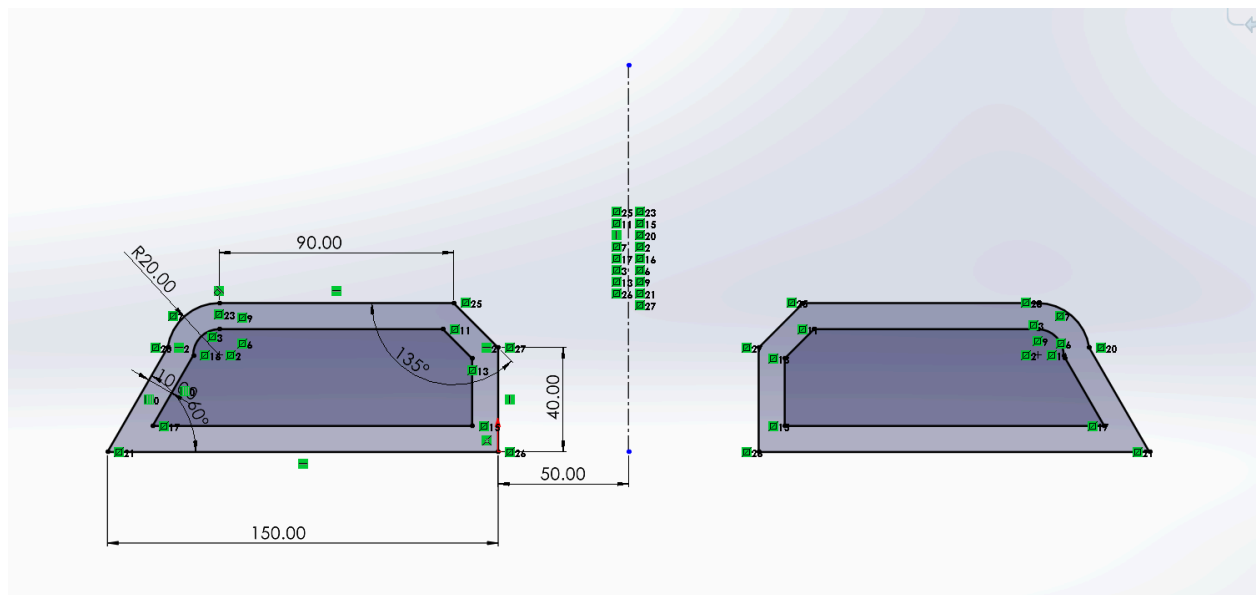
Then click the second selection box to make it active. In the second box, define the line you want to mirror about. Select the vertical centerline.



On the right side, you can see the mirrored sketch in yellow. This is only a preview of the mirrored geometry.



To finish the action, click **OK** on the left side in the **PropertyManager**. Now we have two sketches: the original sketch and the mirrored sketch.



12. Linear Sketch Pattern

What it does: Repeats sketch entities in one or two linear directions.

Use it for: Repeated holes, slots, or other sketch geometry.

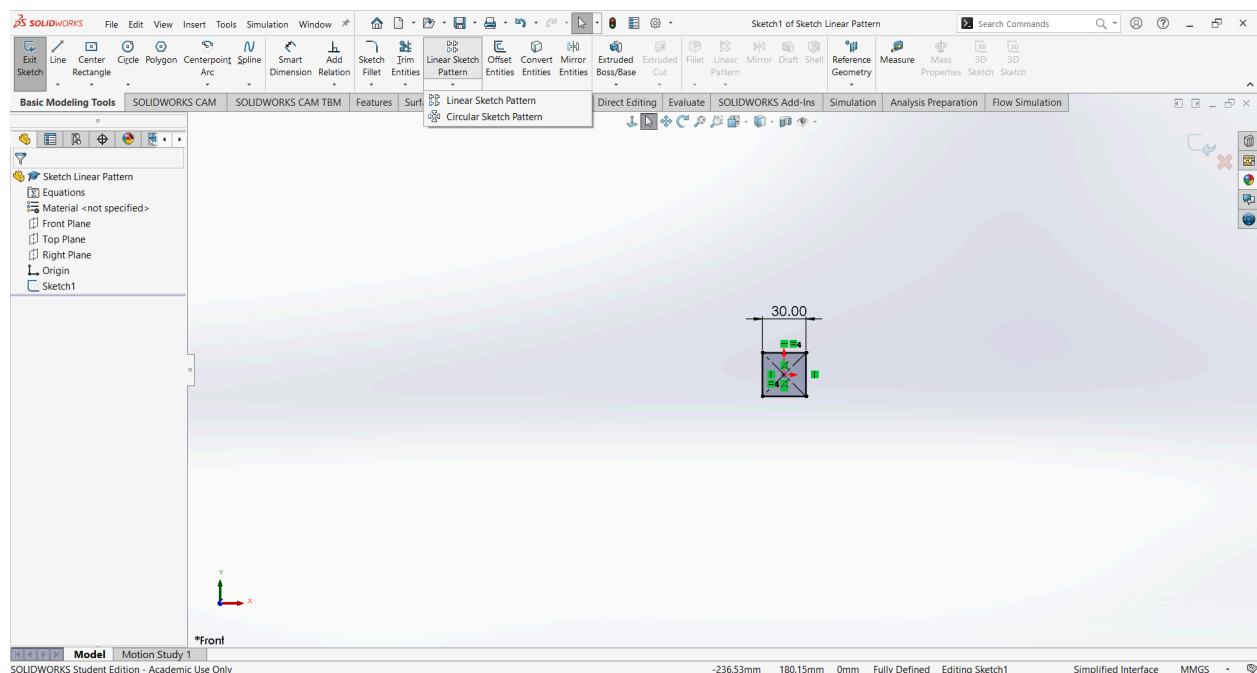
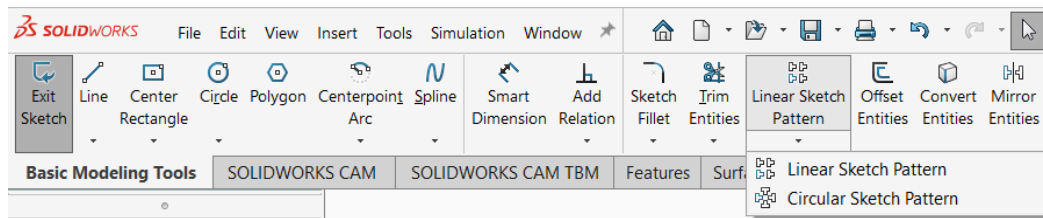
How it works:

Linear Sketch Pattern is used to copy sketch entities in a straight direction.

Instead of drawing the same shape multiple times manually, you can create one sketch entity and repeat it with a defined spacing and number of instances.

Linear Sketch Pattern is useful for creating repeated holes, slots, lines, circles, rectangles, or other sketch entities inside a sketch.

To access **Sketch Linear Pattern**, go to the **Basic Modeling Tools** tab. Find the **Linear Sketch Pattern** tool. This tool allows you to copy selected sketch entities in one or two straight directions.



On the left side, the **PropertyManager** appears. Here, you can define the main settings for the **Linear Sketch Pattern**. You can choose the pattern direction, set the spacing distance, define the number of instances, and select the sketch entities you want to pattern.

The image shows the 'Linear Pattern' PropertyManager dialog box. At the top, there is a toolbar with icons for selection, list, pattern, center, color, and navigation. Below the toolbar, the title 'Linear Pattern' is displayed with a green checkmark and a red X icon. The dialog is divided into three main sections: 'Direction 1', 'Direction 2', and 'Entities to Pattern'. 'Direction 1' is expanded and shows settings for the X-axis, including a spacing of 10.00mm, a count of 2, and a display instance count checkbox. 'Direction 2' is collapsed and shows settings for the Y-axis, including a spacing of 10.00mm, a count of 1, and a display instance count checkbox. The 'Entities to Pattern' section is also collapsed and shows a selection box for choosing entities to pattern.

Linear Pattern

✓ ✗

Direction 1

X-axis

10.00mm

☐ Dimension X spacing

2

☒ Display instance count

0.00deg

☒ Fix X-axis direction

Direction 2

Y-axis

10.00mm

☐ Dimension Y spacing

1

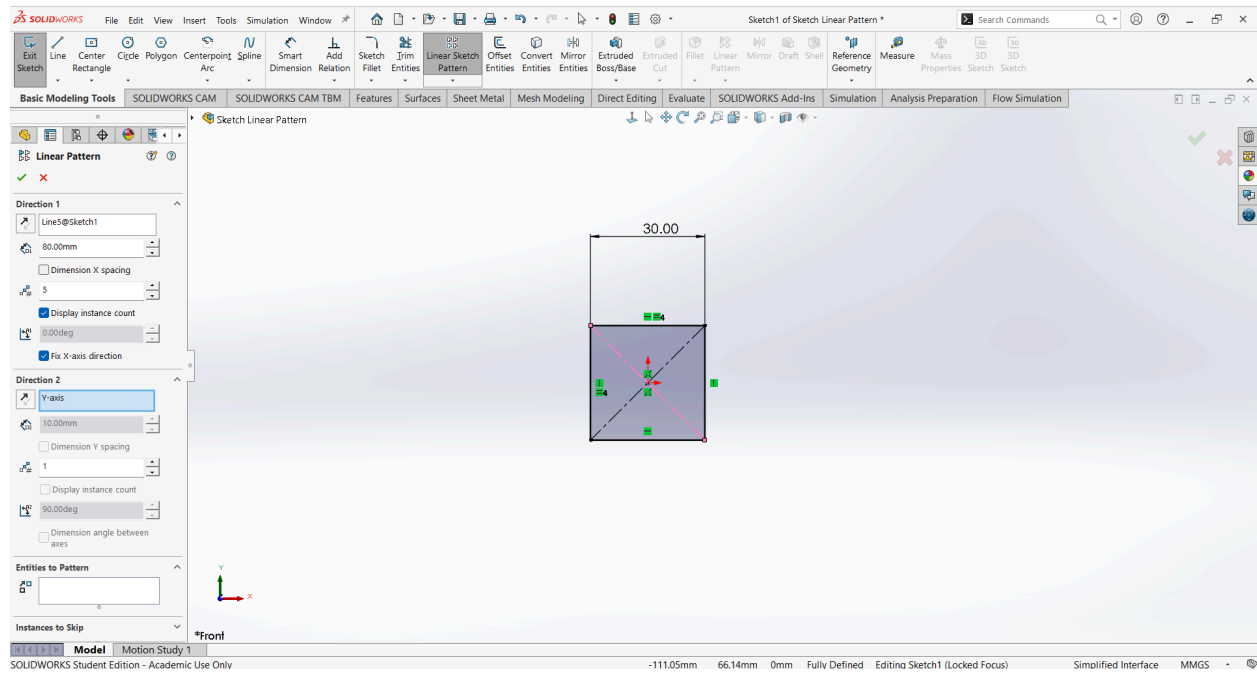
☐ Display instance count

90.00deg

☐ Dimension angle between axes

Entities to Pattern

For **Direction 1**, select the centerline, as shown. Set the spacing distance to **80 mm**. Set the number of instances to **5**.



For **Direction 2**, select the second centerline, as shown. Set the spacing distance to **70 mm**. Set the number of instances to **4**.

Linear Pattern

✓ ✗

Direction 1

Line5@Sketch1

80.00mm

☐ Dimension X spacing

5

☒ Display instance count

0.00deg

☒ Fix X-axis direction

Direction 2

Line6@Sketch1

70.00mm

☐ Dimension Y spacing

4

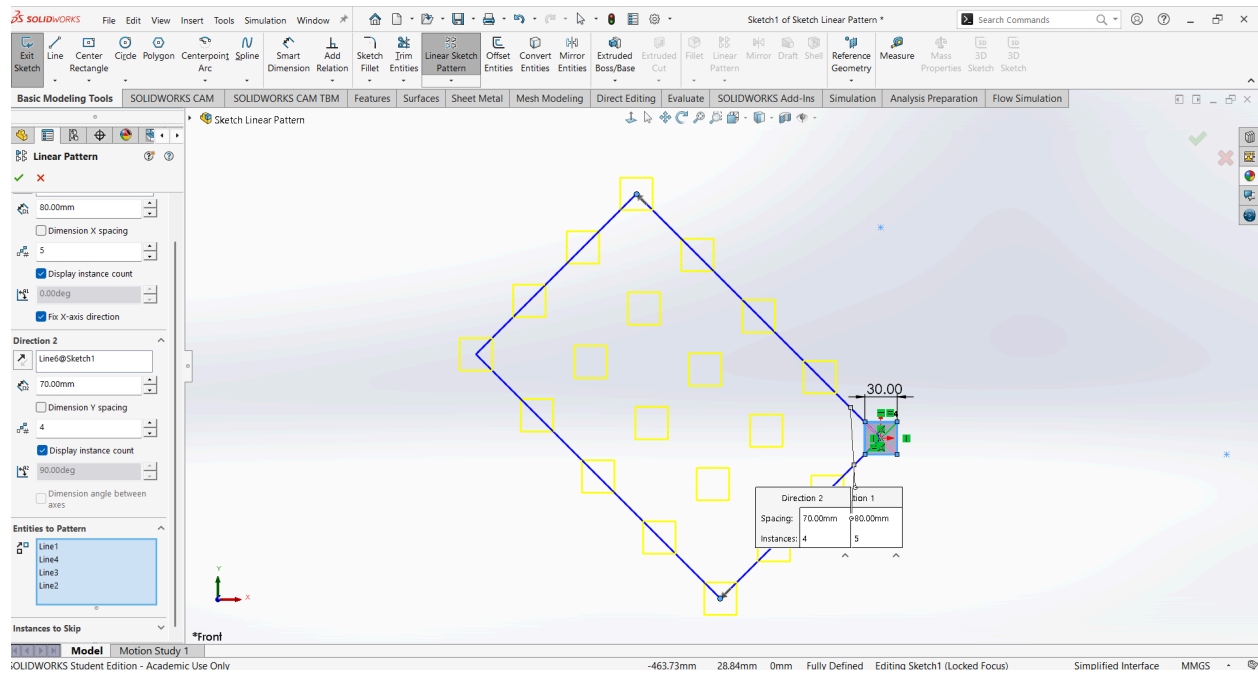
☒ Display instance count

90.00deg

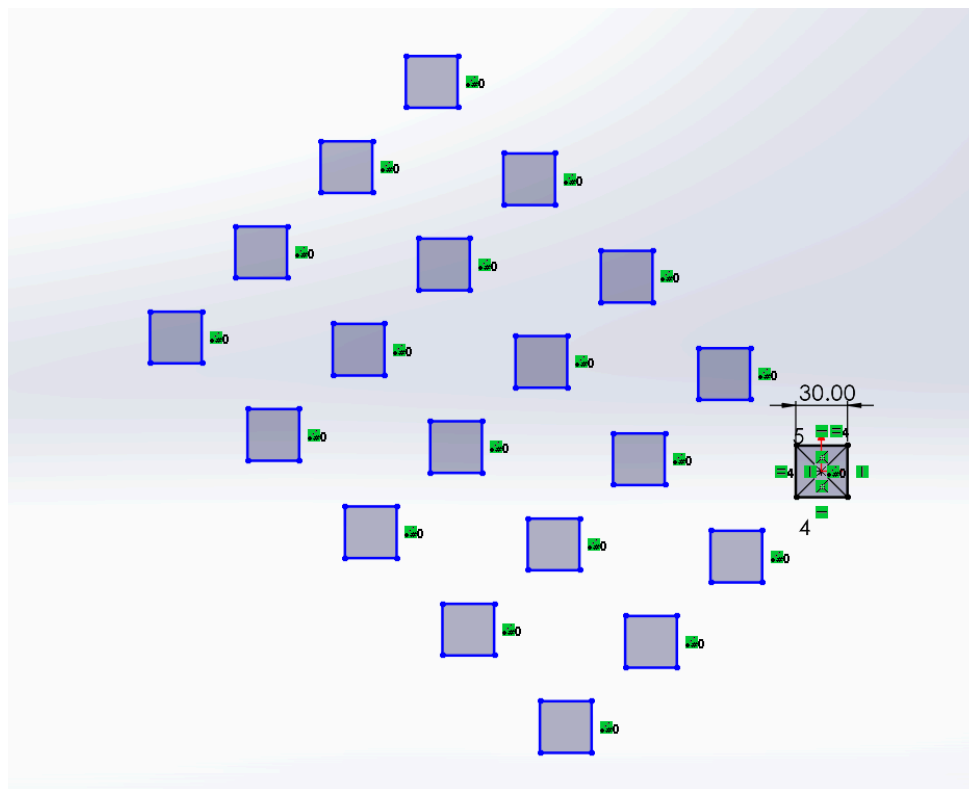
☐ Dimension angle between axes

Entities to Pattern

For **Entities to Pattern**, select the entire rectangle. Make sure all four lines of the rectangle are selected.



Click **OK** by selecting the green check mark. This is the final result.



13. Circular Sketch Pattern

What it does: Repeats sketch entities around a center point.

Use it for: Bolt patterns, circular openings, and radial geometry.

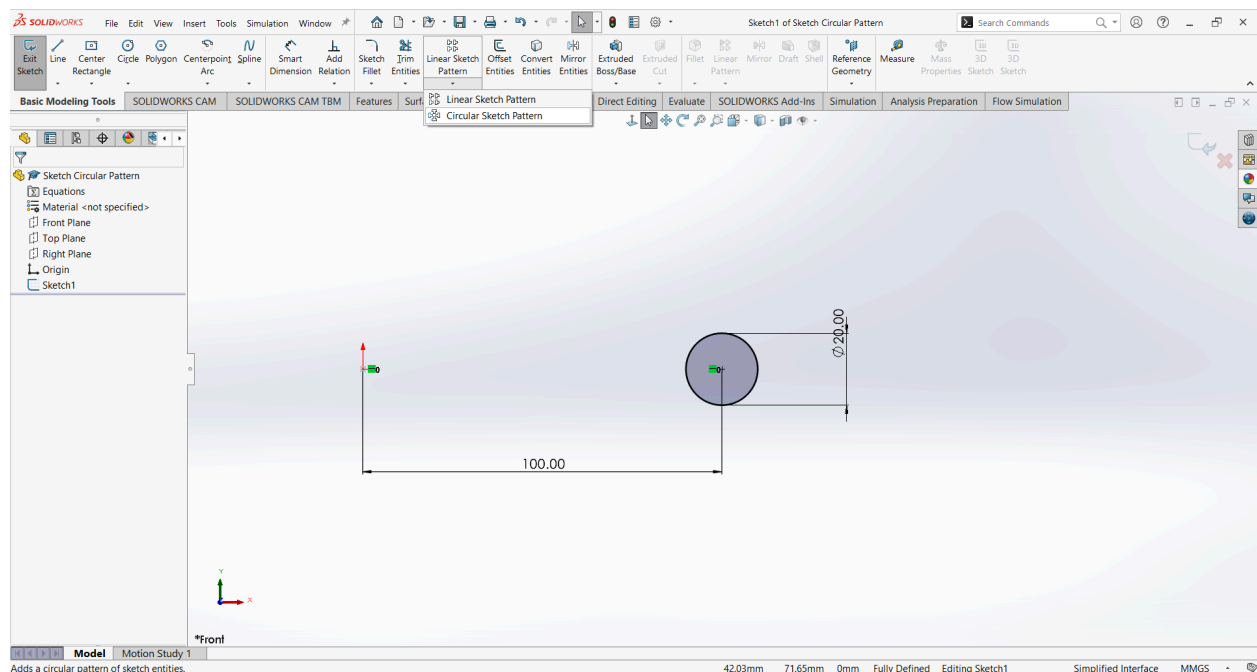
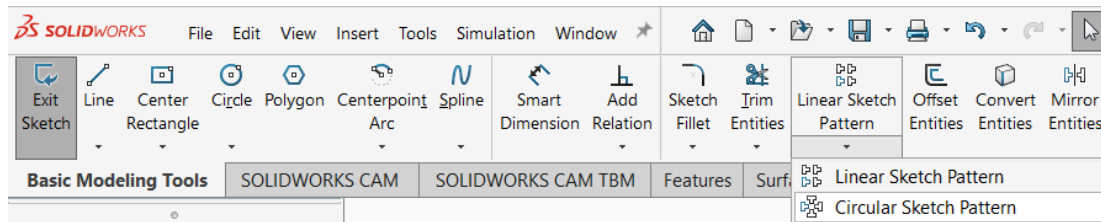
How it works:

Circular Sketch Pattern is used to copy sketch entities around a center point or circular reference.

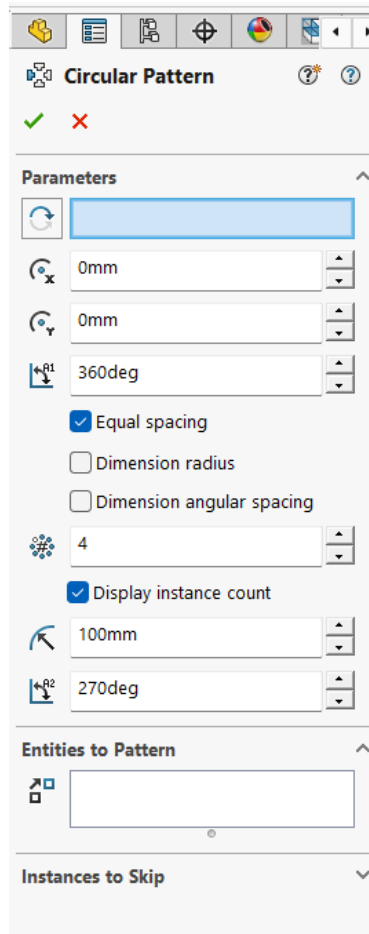
Instead of drawing the same shape multiple times manually, you can create one sketch entity and repeat it in a circular direction.

Circular Sketch Pattern is useful for creating repeated holes, circles, slots, lines, or other sketch entities around a center point.

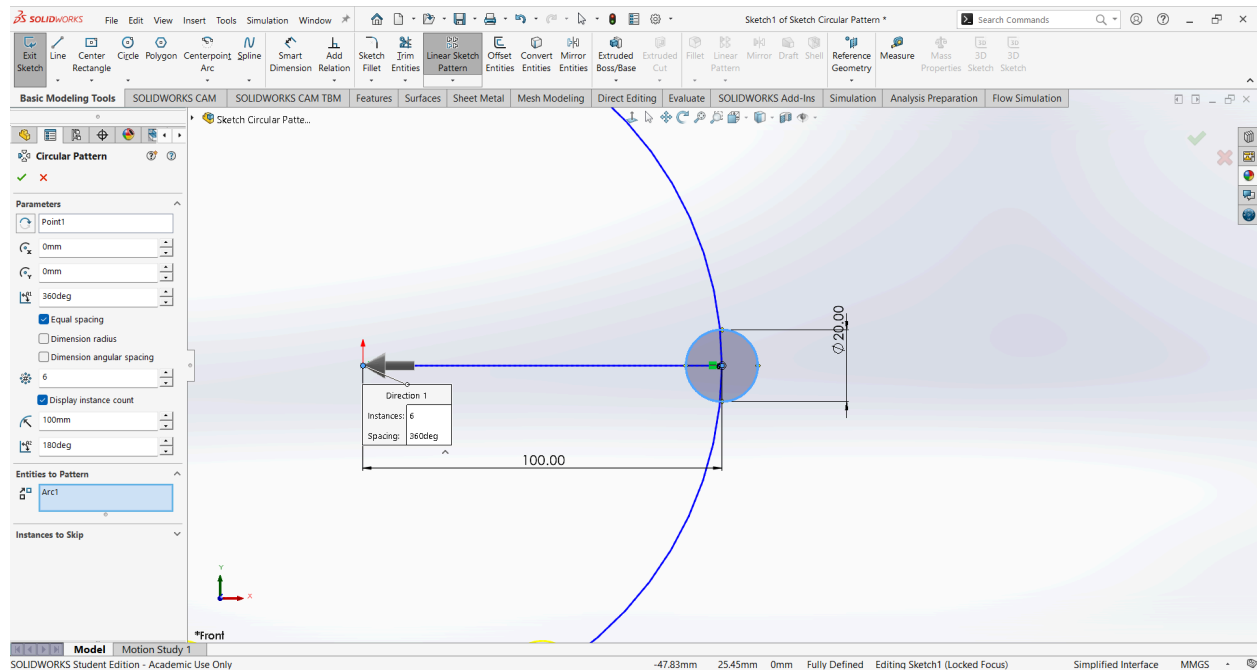
To access the Circular **Sketch Pattern**, go to the **Basic Modeling Tools** tab. Find the **Linear Sketch Pattern** tool. Click the small arrow below it to open the drop-down menu. From the list of available tools, select **Circular Sketch Pattern**.



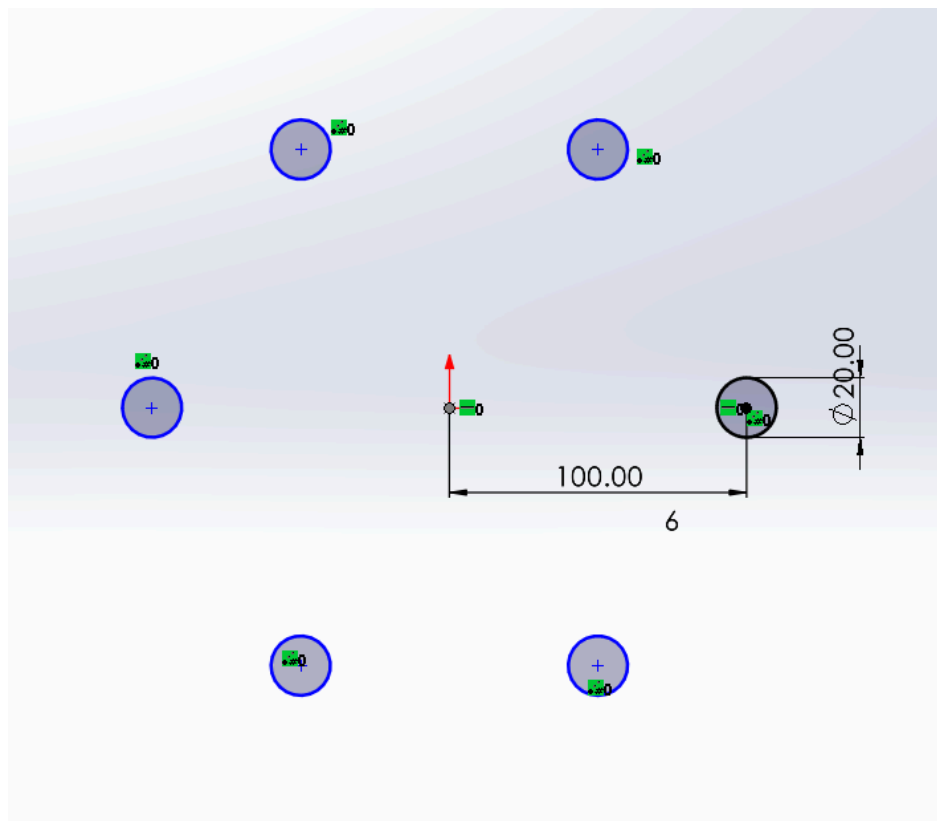
On the left side, the **PropertyManager** appears. Here, you can define the main settings for the **Circular Sketch Pattern**. You can choose the center point around which the pattern will rotate. You can also define the number of instances, the spacing angle, and the sketch entities you want to pattern.



For **Center Point**, select the origin point. Set the number of instances to **6**. Set the spacing angle to **360°**. For **Entities to Pattern**, select the circle.



Click **OK** by selecting the green check mark. This is the final result.



2. PART DESIGN

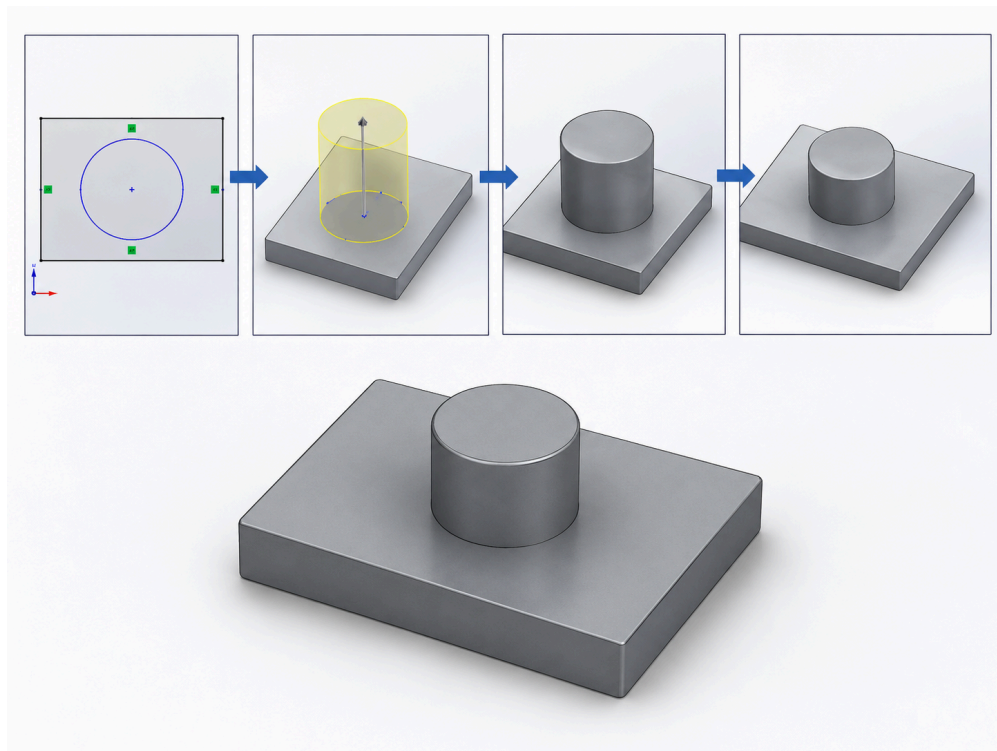
14. Extruded Boss/Base

What it does: Turns a 2D sketch into a 3D solid by extruding it.

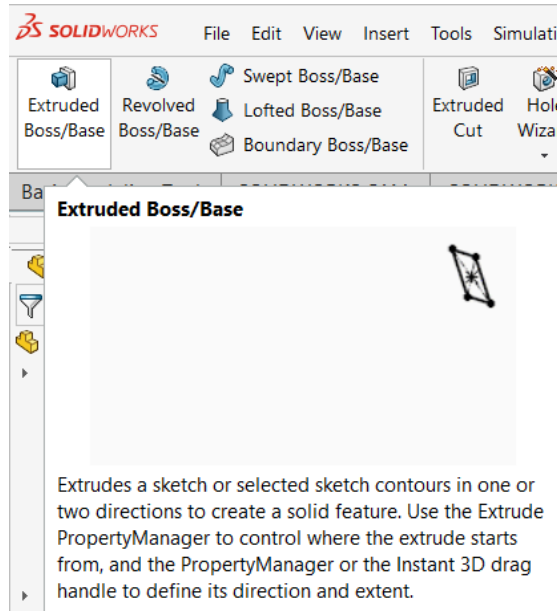
Use it for: Creating the base shape of many parts.

How it works:

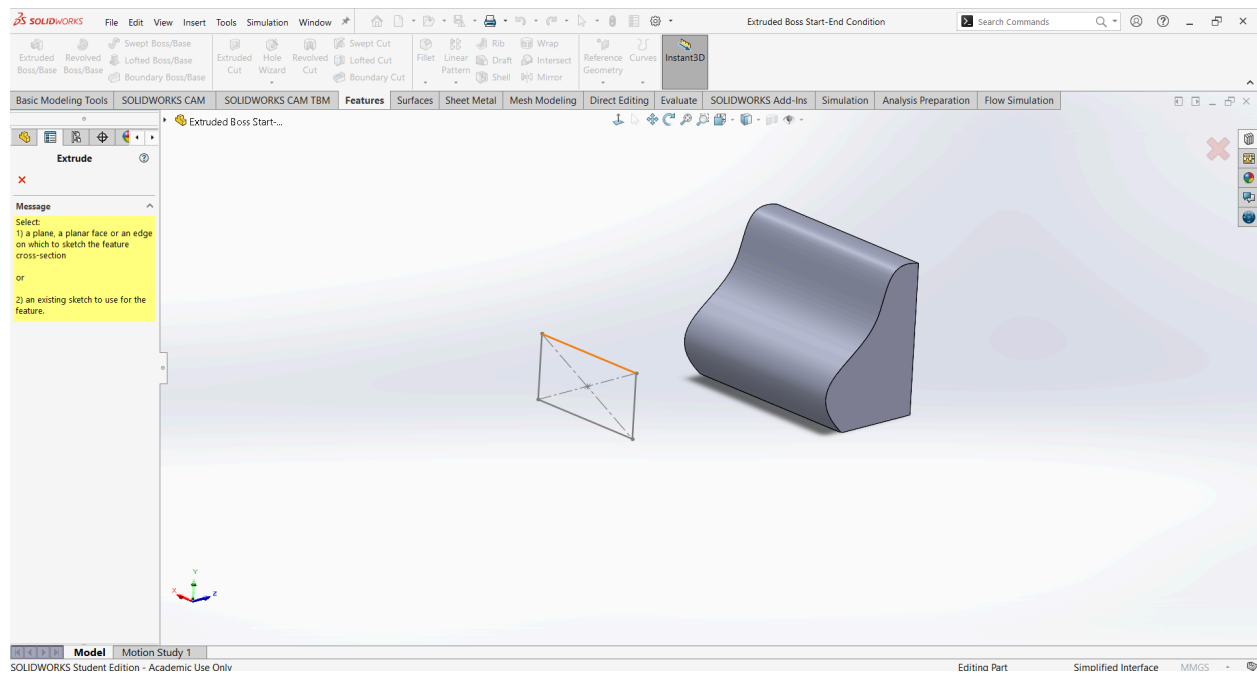
The first tool we will cover is **Extruded Boss/Base**.



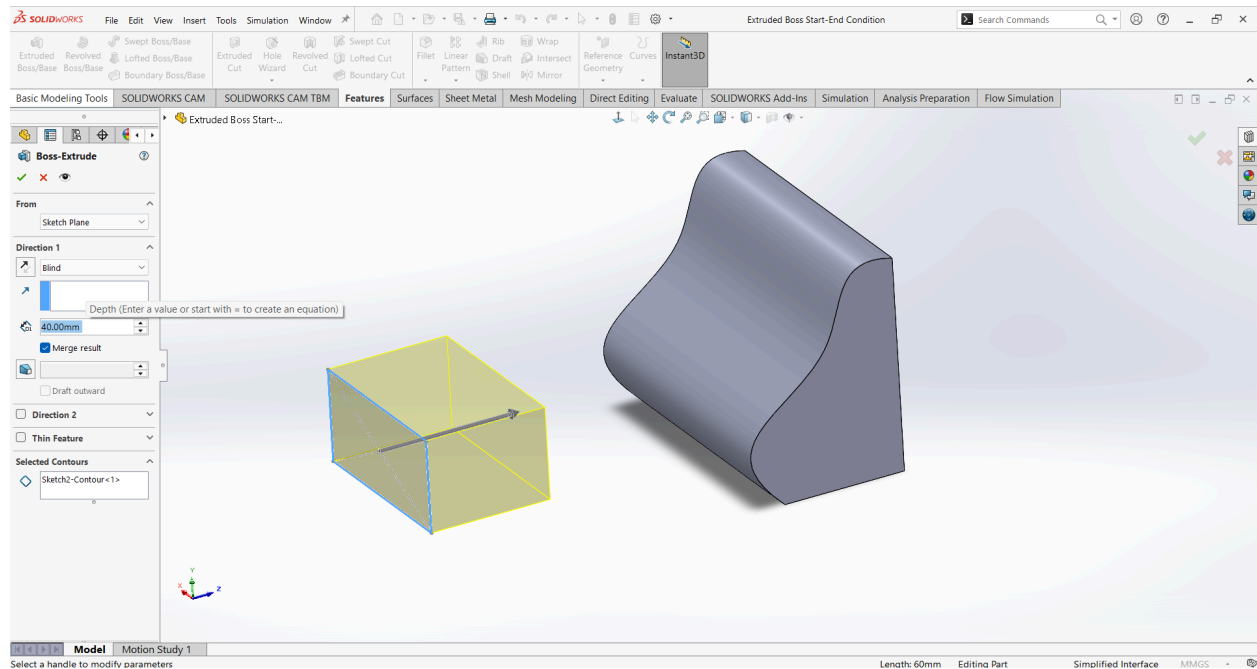
We already used this tool earlier when converting a sketch into a 3D model. Now we will explain it in more detail. Go to the **Features** tab and select **Extruded Boss/Base**.



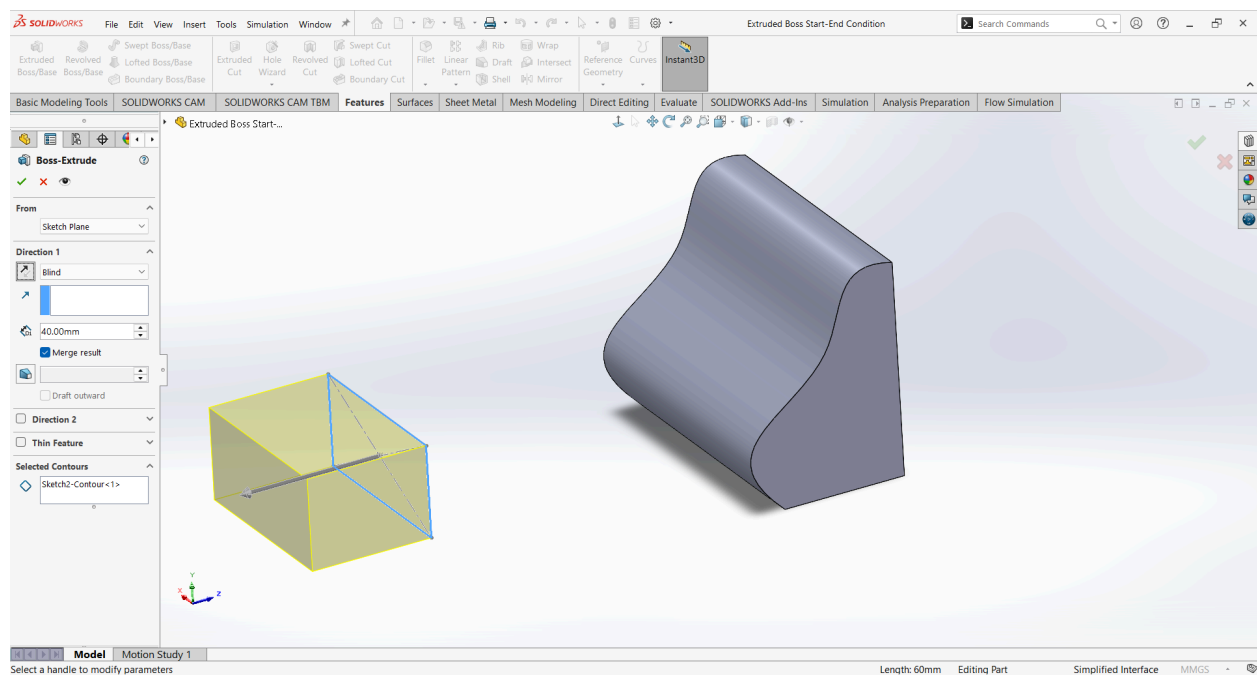
Select the rectangle, as shown.



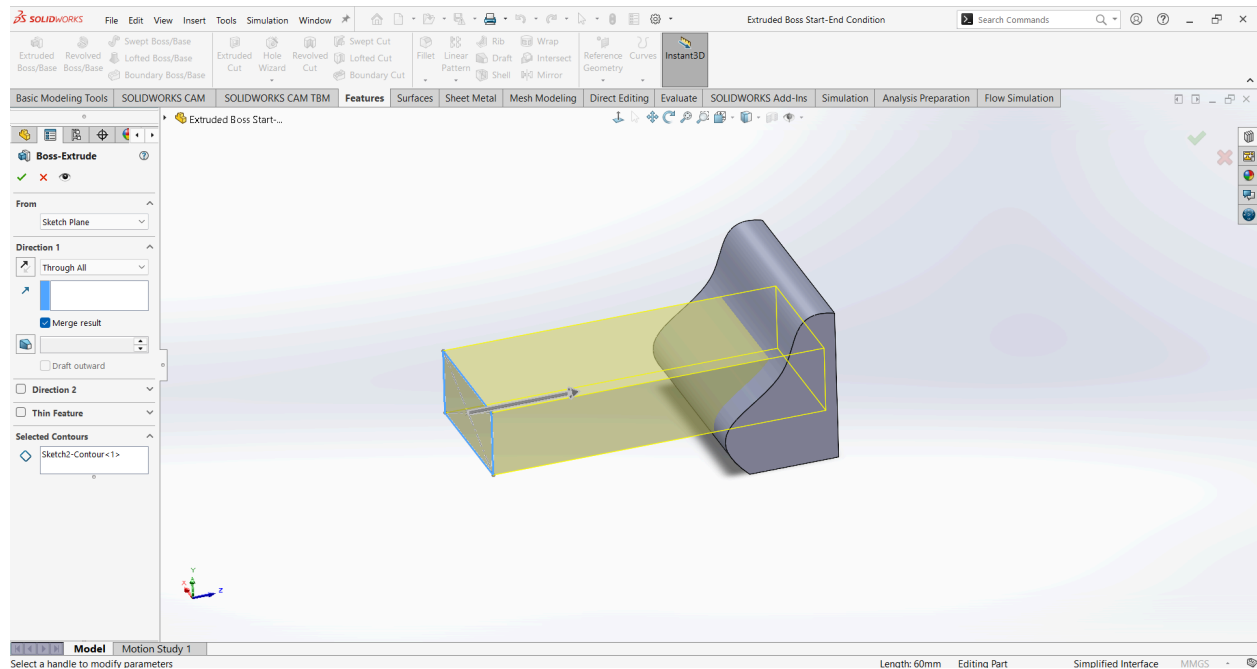
The first end condition is **Blind**. This option is used when you want to extrude the sketch to a defined depth. In this example, set the depth to **40 mm**.



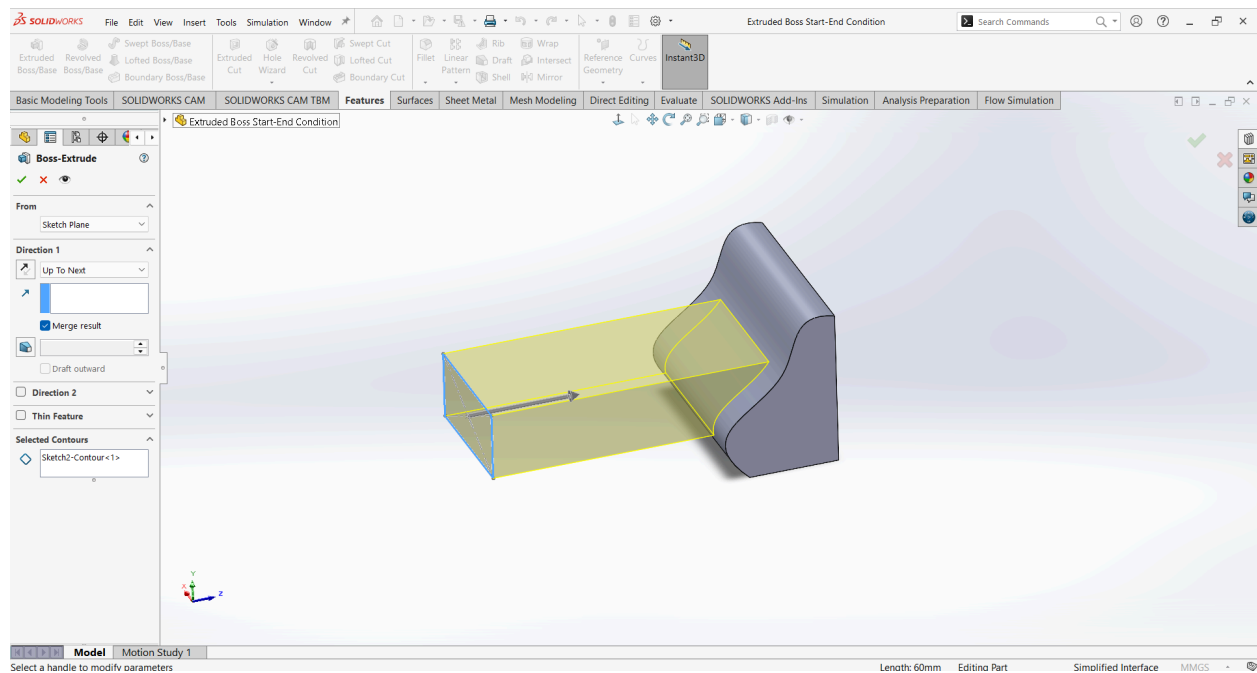
When you click the arrow next to **Blind**, you can change the direction of the extrusion.



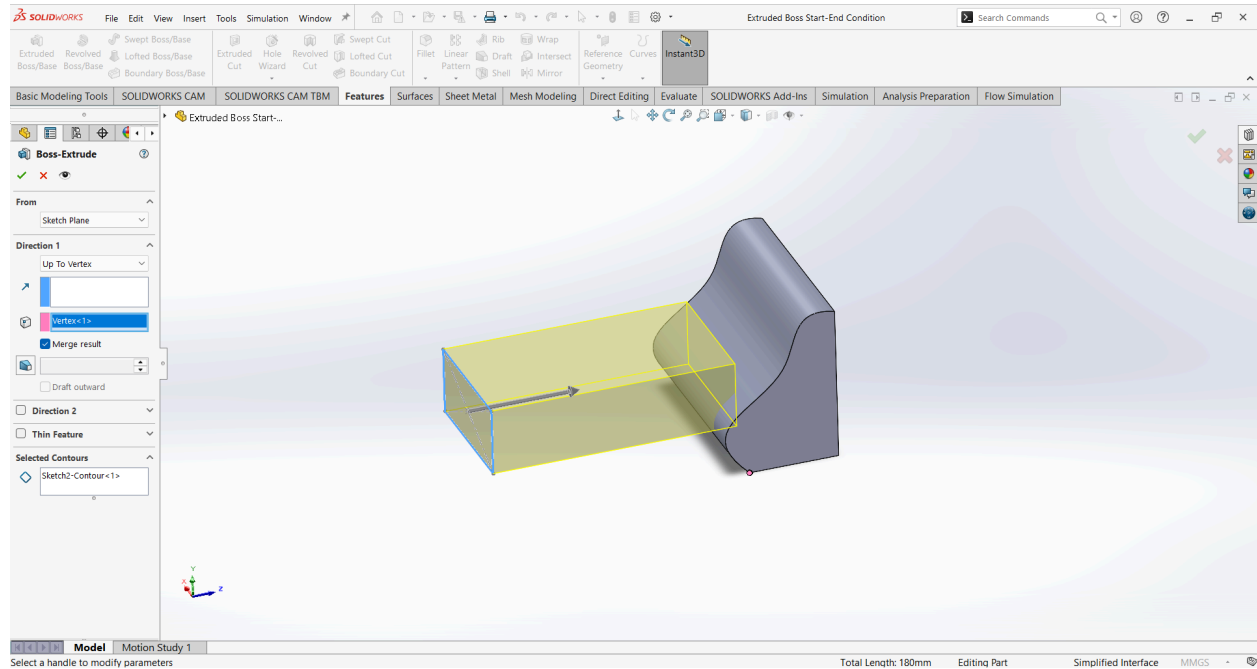
The next end condition is **Through All**. With this option, the extrusion goes through the entire model in front of the sketch.



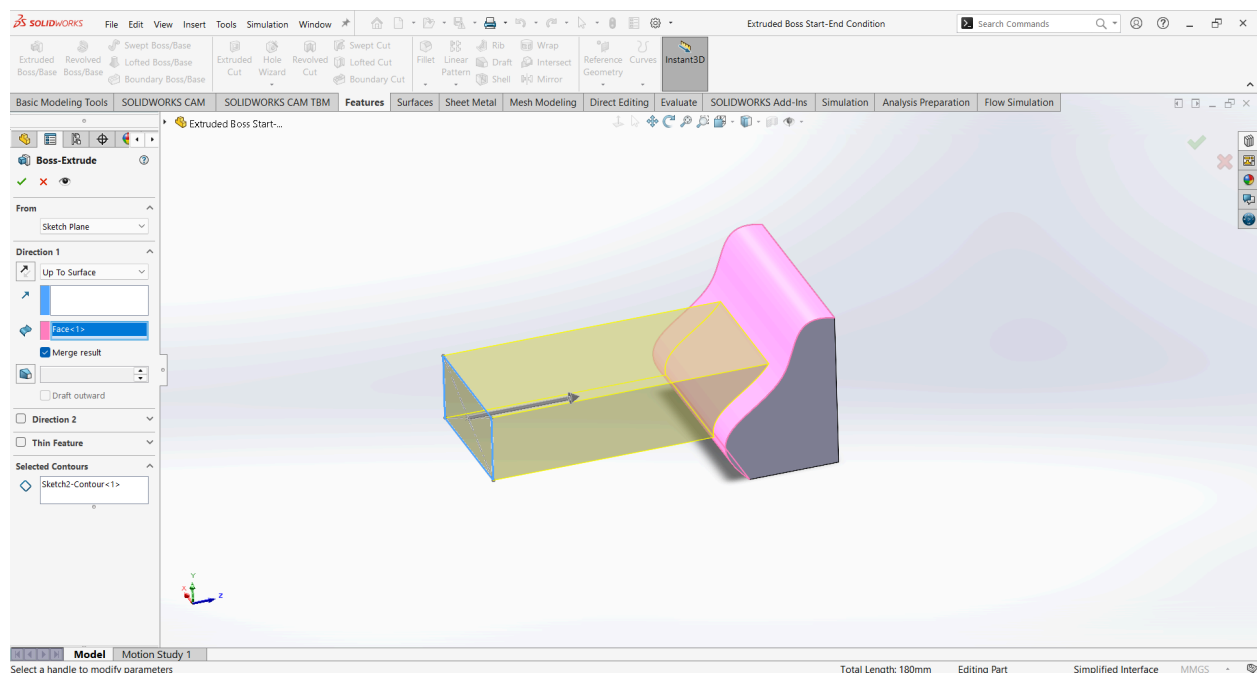
The next end condition is **Up To Next**. With this option, the extrusion extends up to the next face or body in front of the sketch.



The next end condition is **Up To Vertex**. With this option, you need to select a vertex that defines where the extrusion will stop. First, click the selection box on the left side in the **PropertyManager**, then select a vertex on the model.

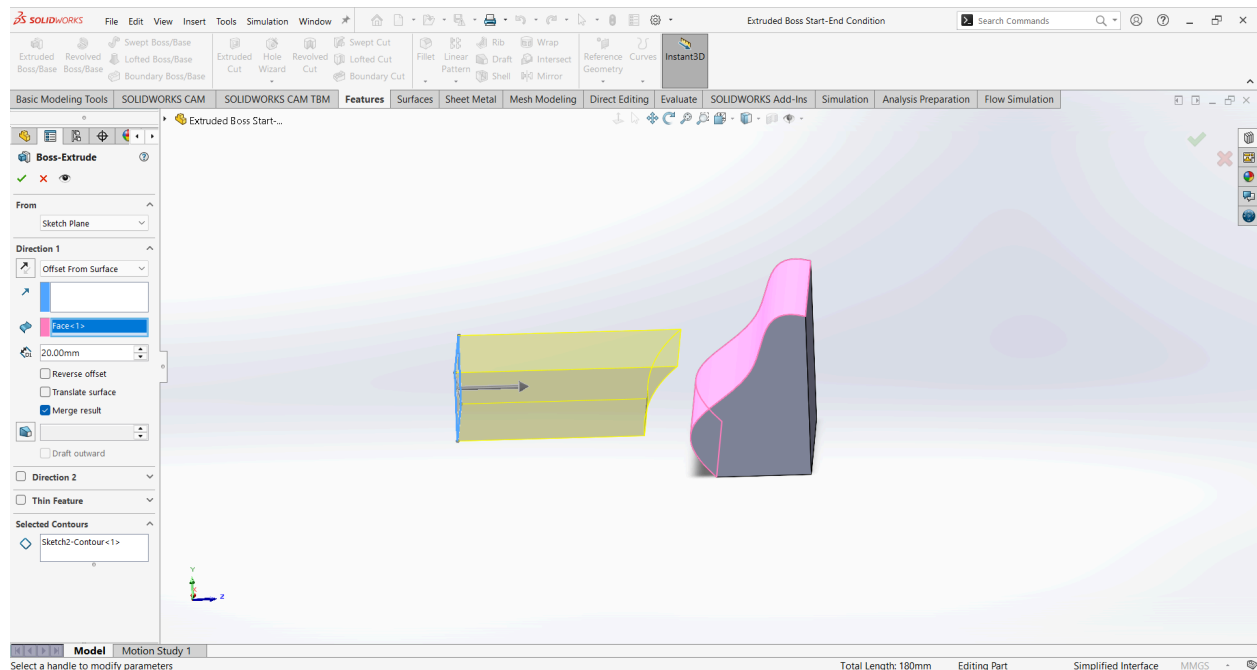


The next end condition is **Up To Surface**. With this option, you need to select a face where the extrusion will stop. First, click the selection box on the left side in the **PropertyManager**, then select a face on the model. This makes the extrusion extend up to the selected face.

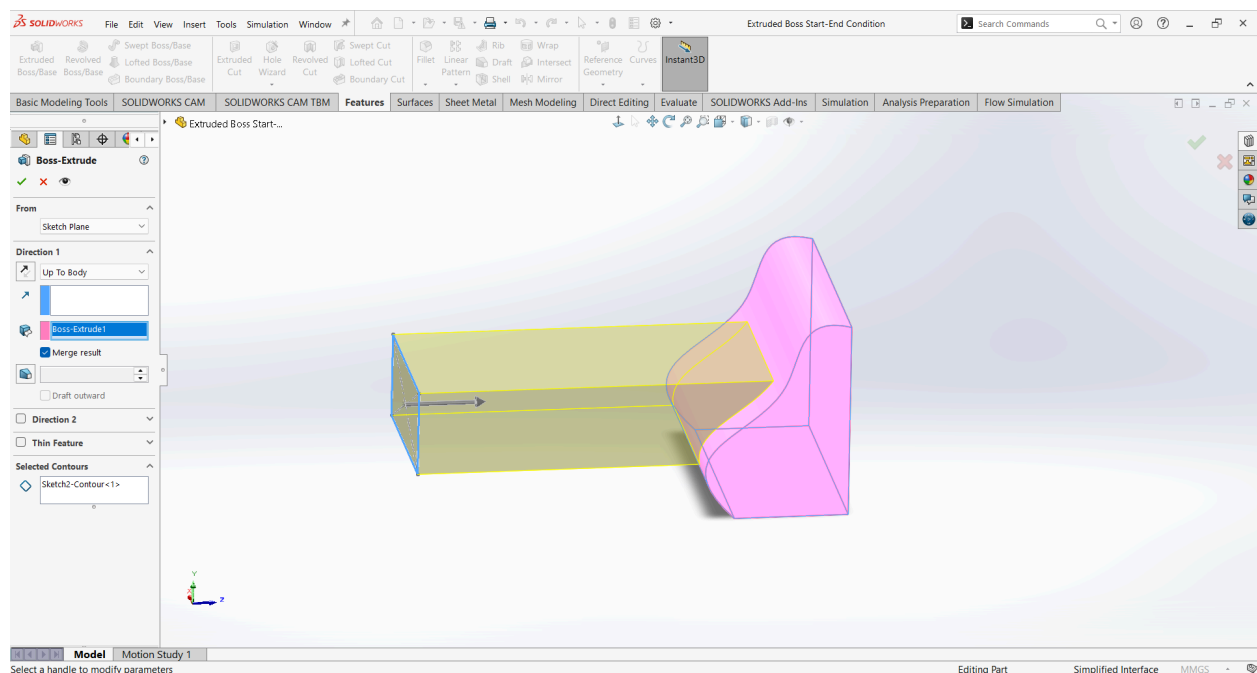


The next end condition is **Offset From Surface**.

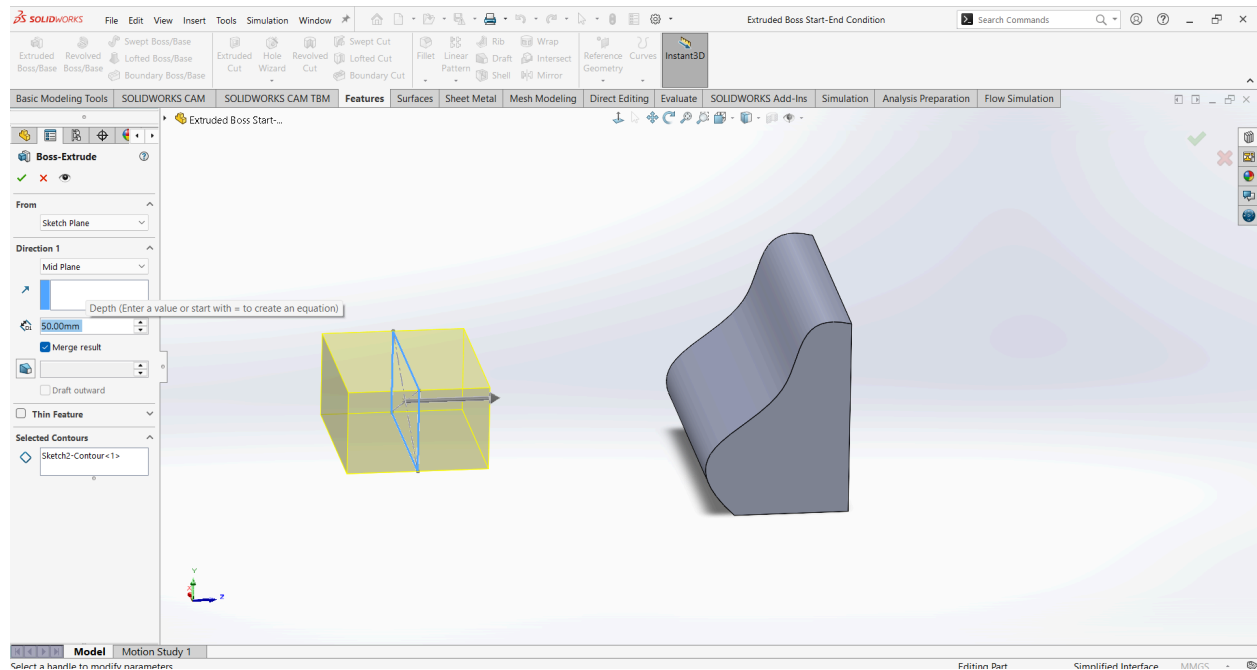
First, click the selection box on the left side in the **PropertyManager**. Then select a face on the model and define the offset distance. In this example, the offset distance is **20 mm**. This means the extrusion stops **20 mm** away from the selected face.



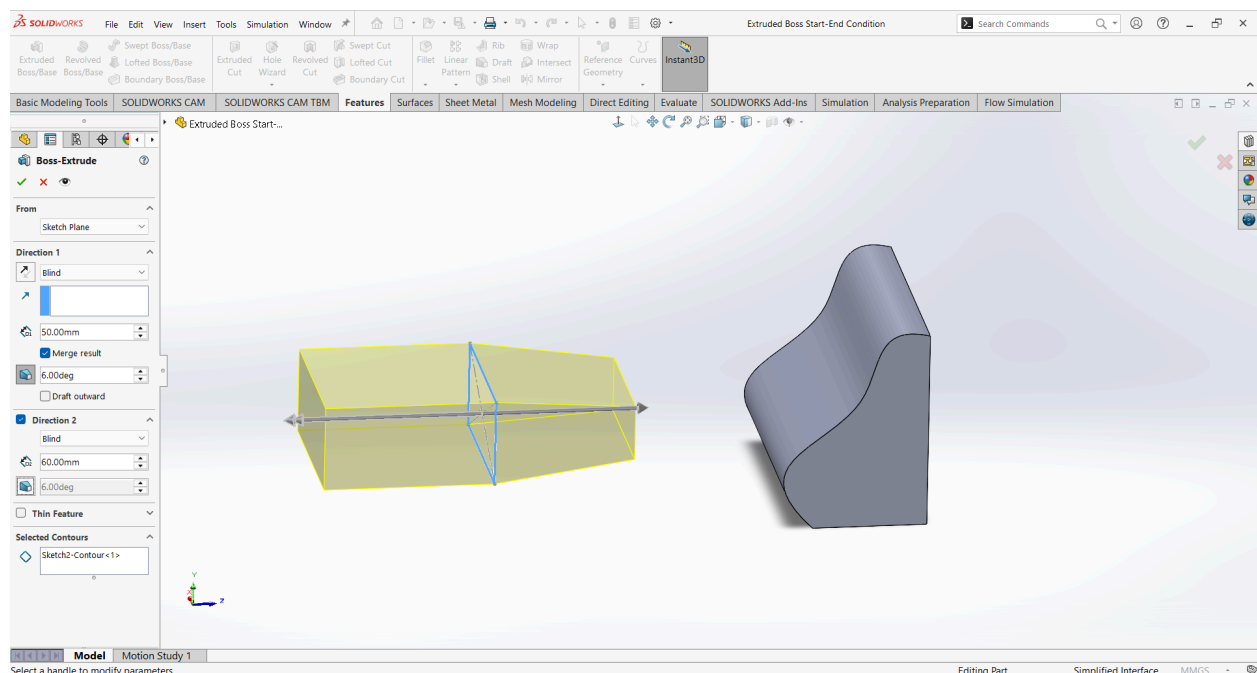
The next end condition is **Up To Body**. First, click the selection box on the left side in the **PropertyManager**. Then select the body where you want the extrusion to stop. This makes the extrusion extend up to the selected body.



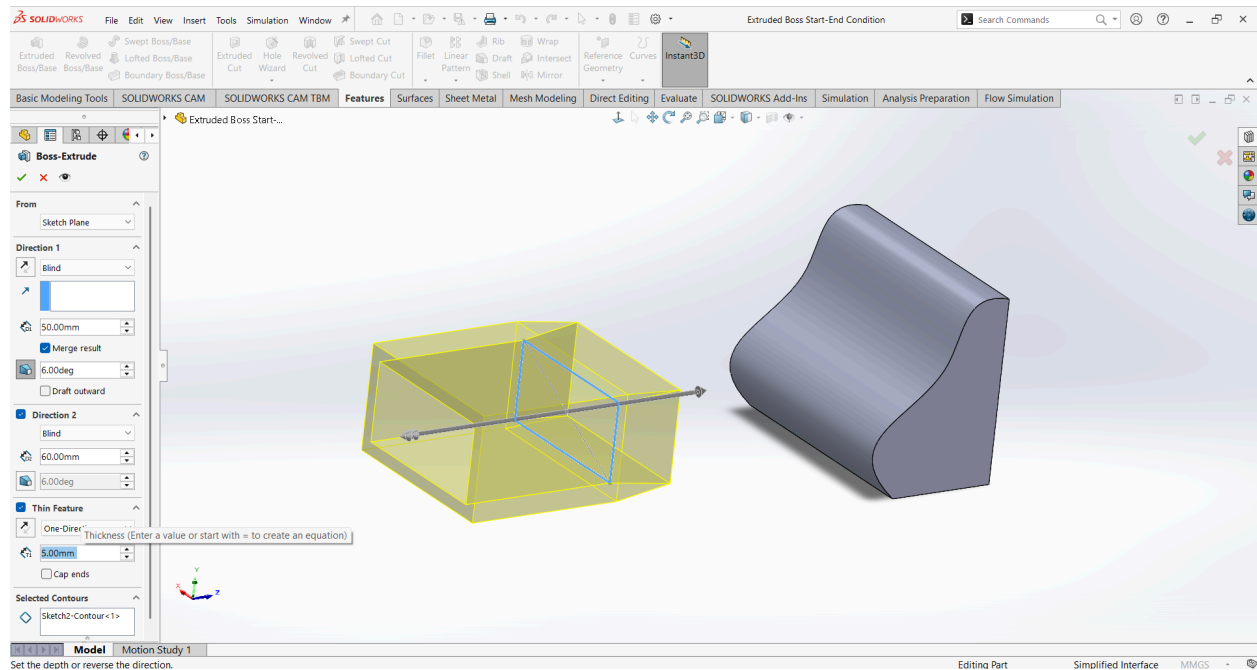
The next end condition is **Mid Plane**. With this option, the extrusion goes equally in both directions from the sketch. You also need to define the total depth of the extrusion.



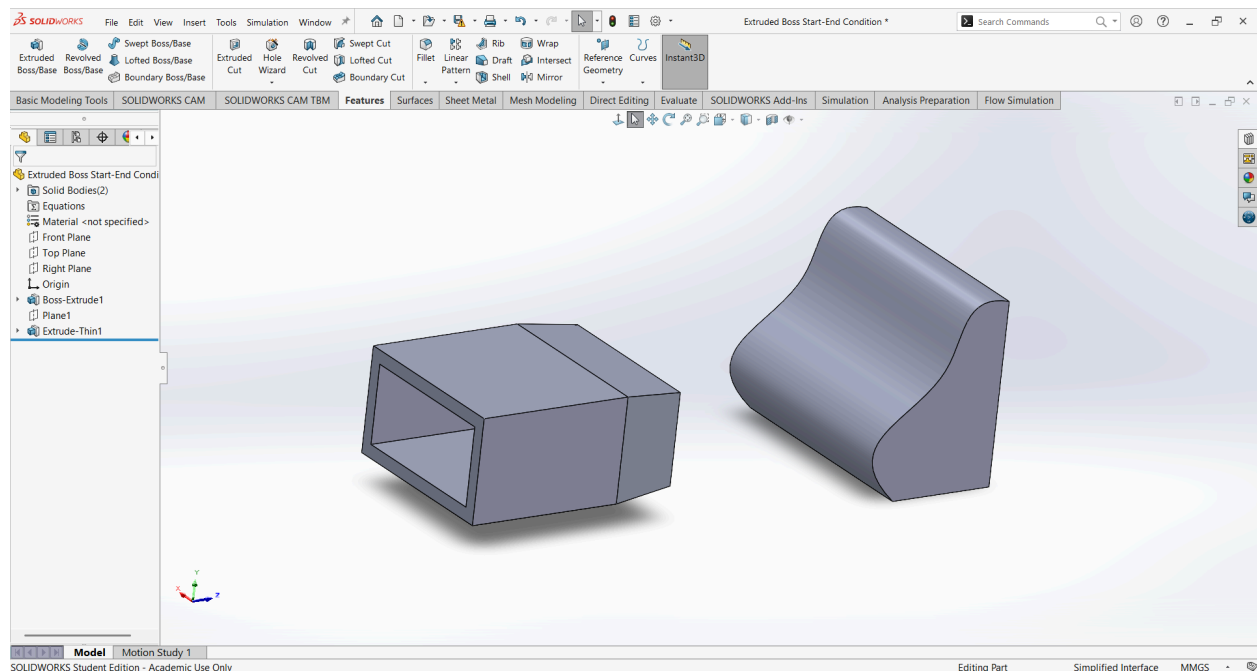
Below the depth value in the **PropertyManager**, there is a **Draft** option. This means the extrusion can be created at an angle. In this example, the draft angle is set to **6°**. There is also a **Direction 2** option. To turn it on, check the small box next to **Direction 2**. For **Direction 2**, you can define a different depth. In this example, set the depth to **60 mm** and leave the draft option turned off.



There is also a **Thin Feature** option. This option allows you to create an extrusion with a thin wall, using a thickness value that you define. In this example, the wall thickness is set to **1 mm**.



Click **OK**.



These are the main options for extruding a sketch.

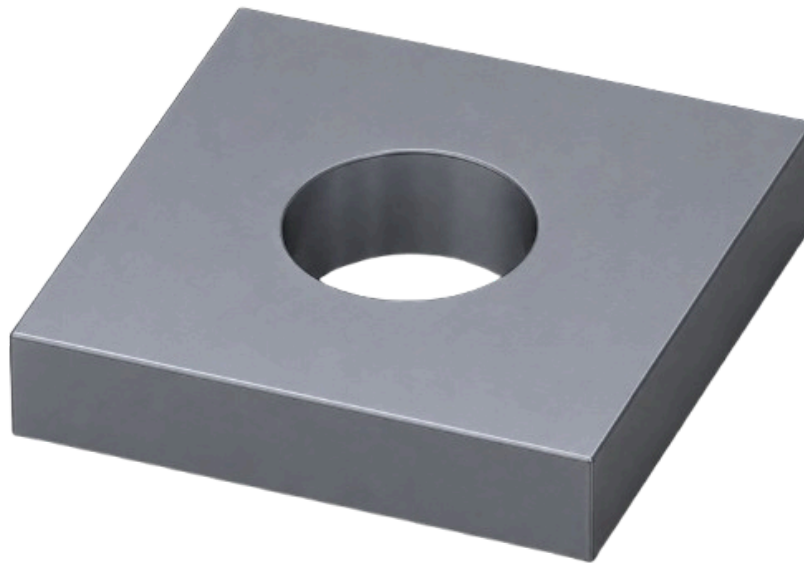
15. Extruded Cut

What it does: Removes material using an extruded sketch profile.

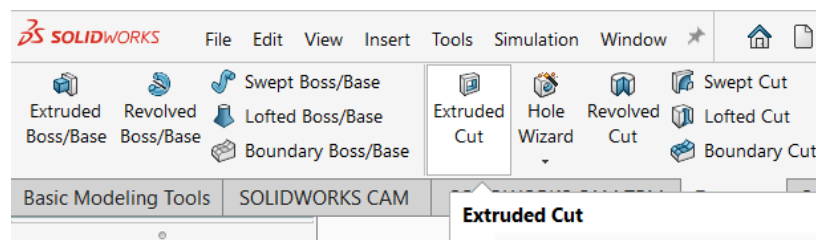
Use it for: Holes, pockets, slots, openings, and cutouts.

How it works:

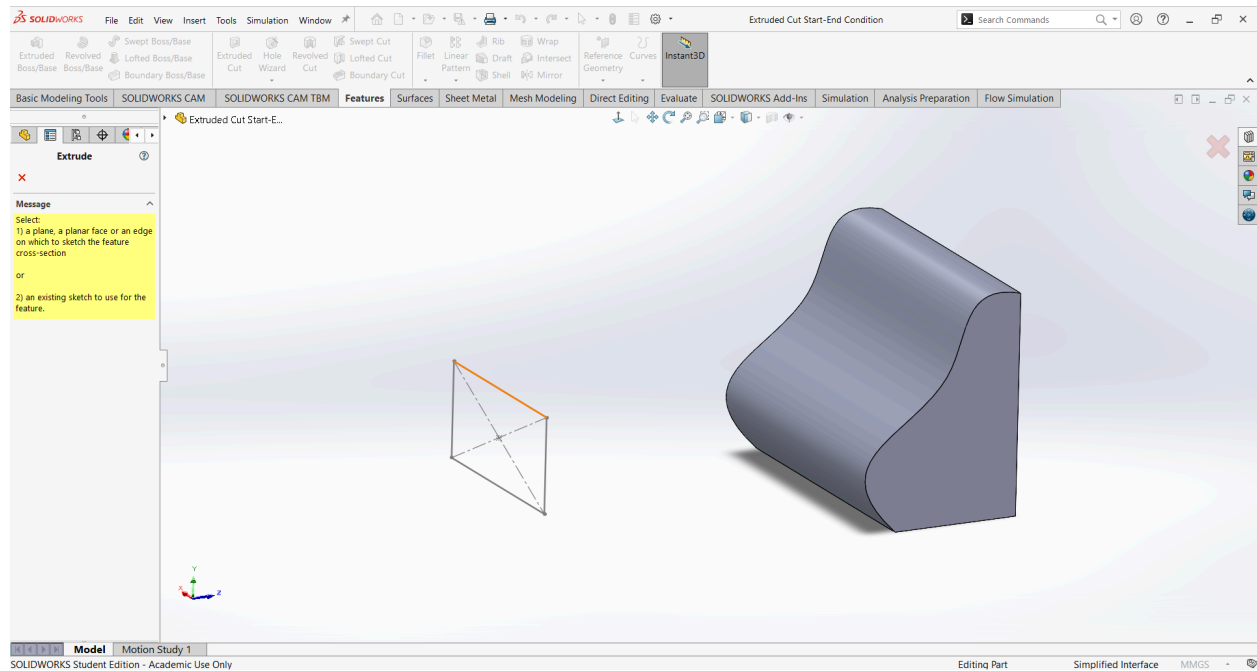
With **Extruded Boss/Base**, we add material. With **Extruded Cut**, we remove material.



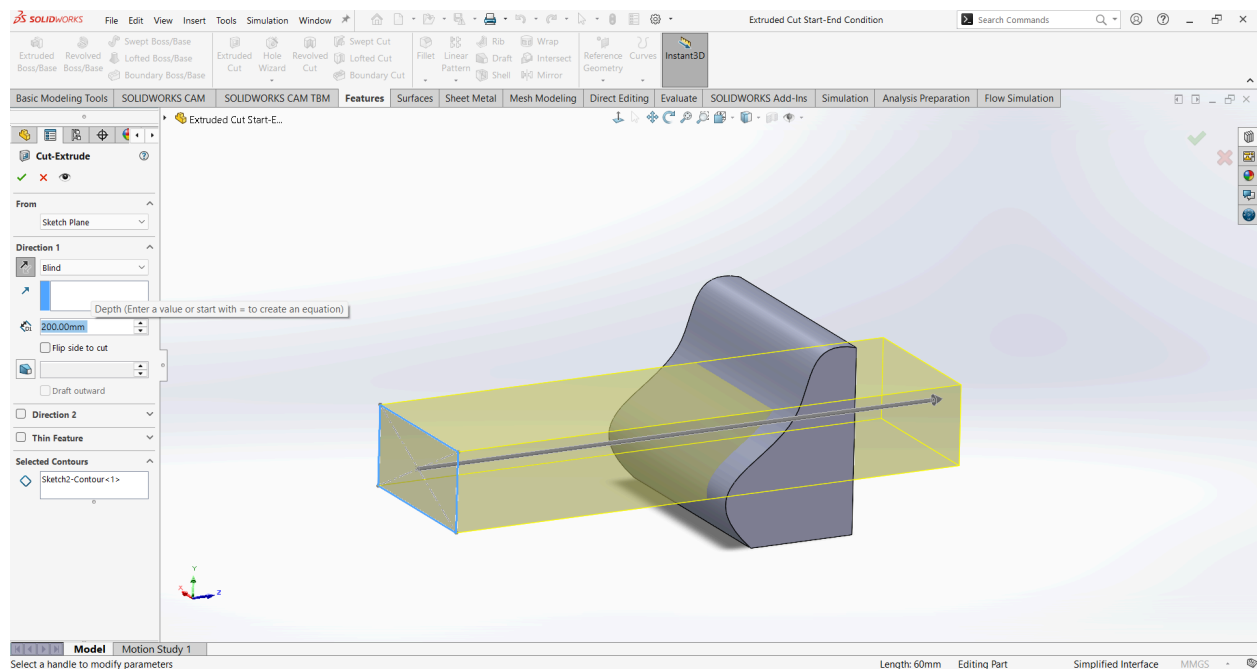
Go to the **Features** tab and select **Extruded Cut**.



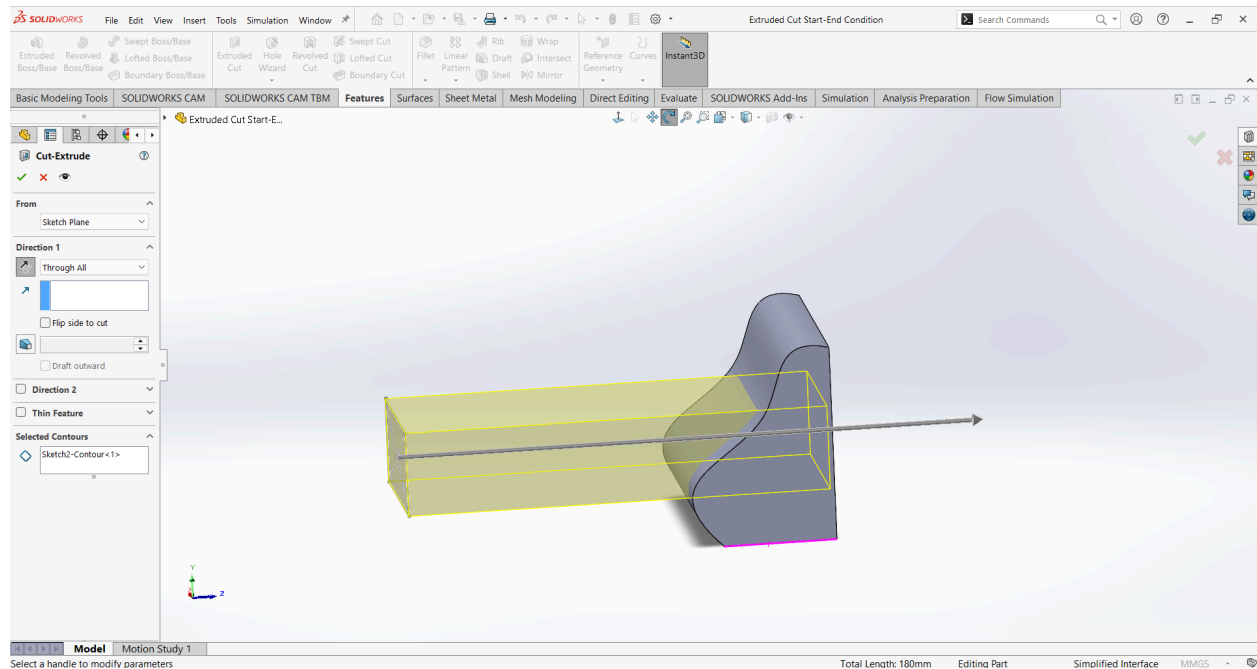
A yellow prompt appears, asking you to select a sketch. Click on the sketch.



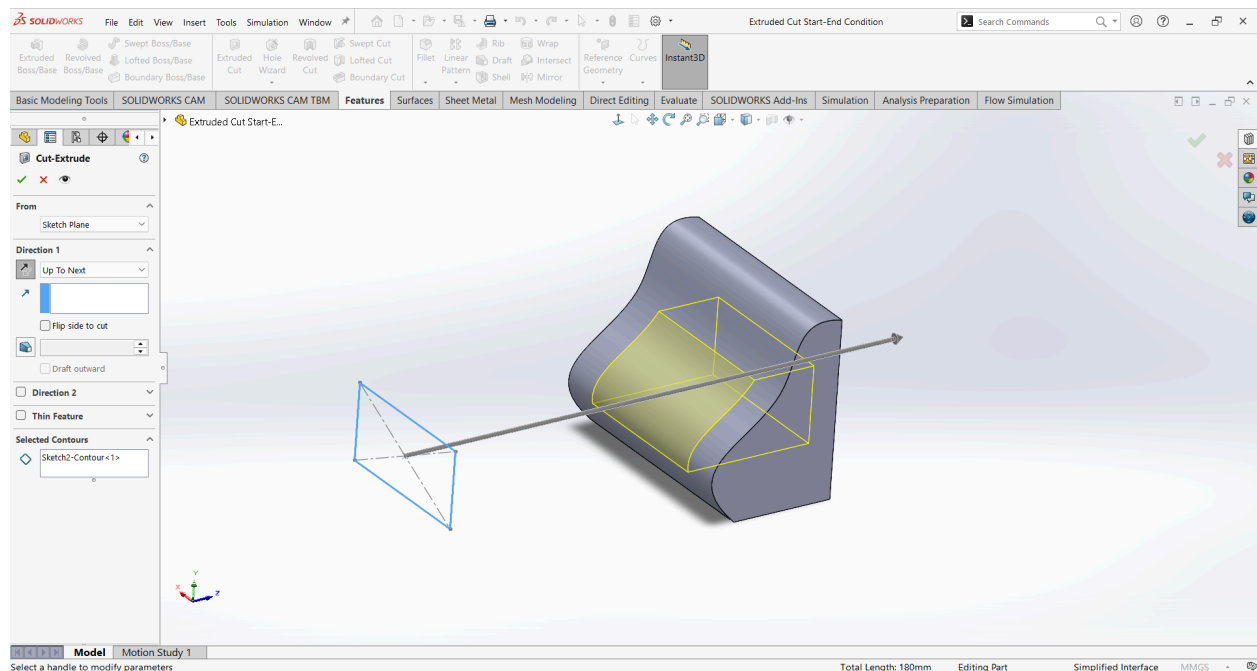
The first end condition is **Blind**. Here, we have similar options as with **Extruded Boss/Base**. With **Blind**, you define the depth of the cut. Set the cut depth to **200 mm**.



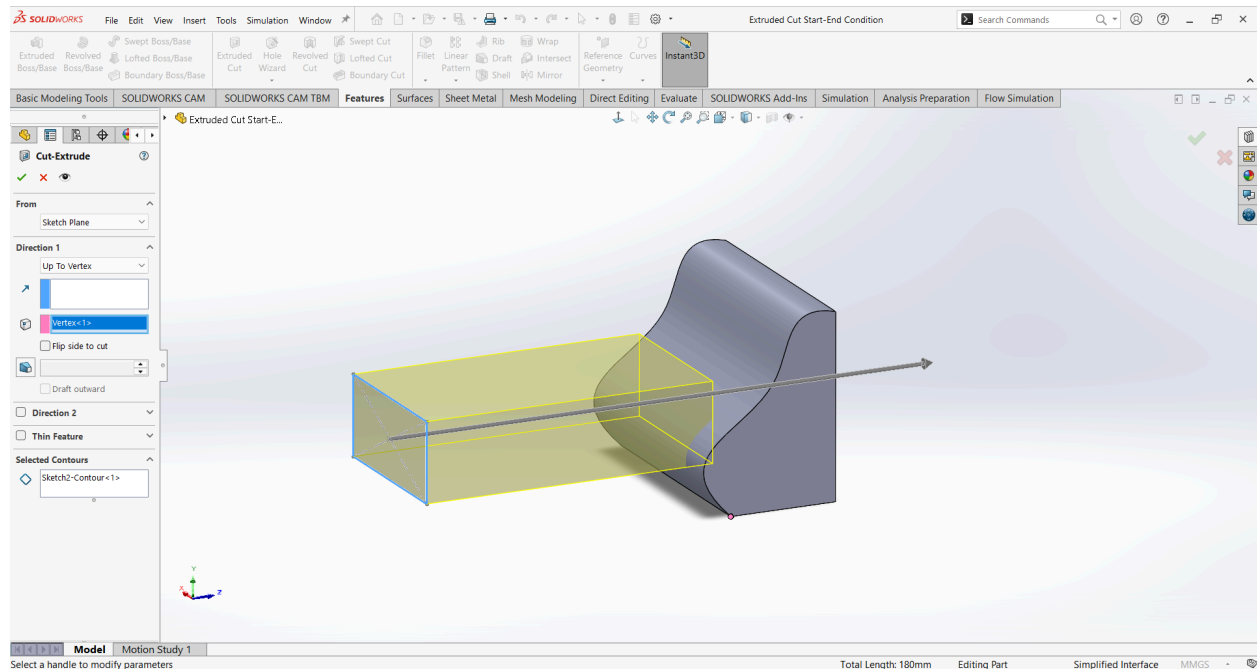
The next end condition is **Through All**. With this option, the cut goes through all the material.



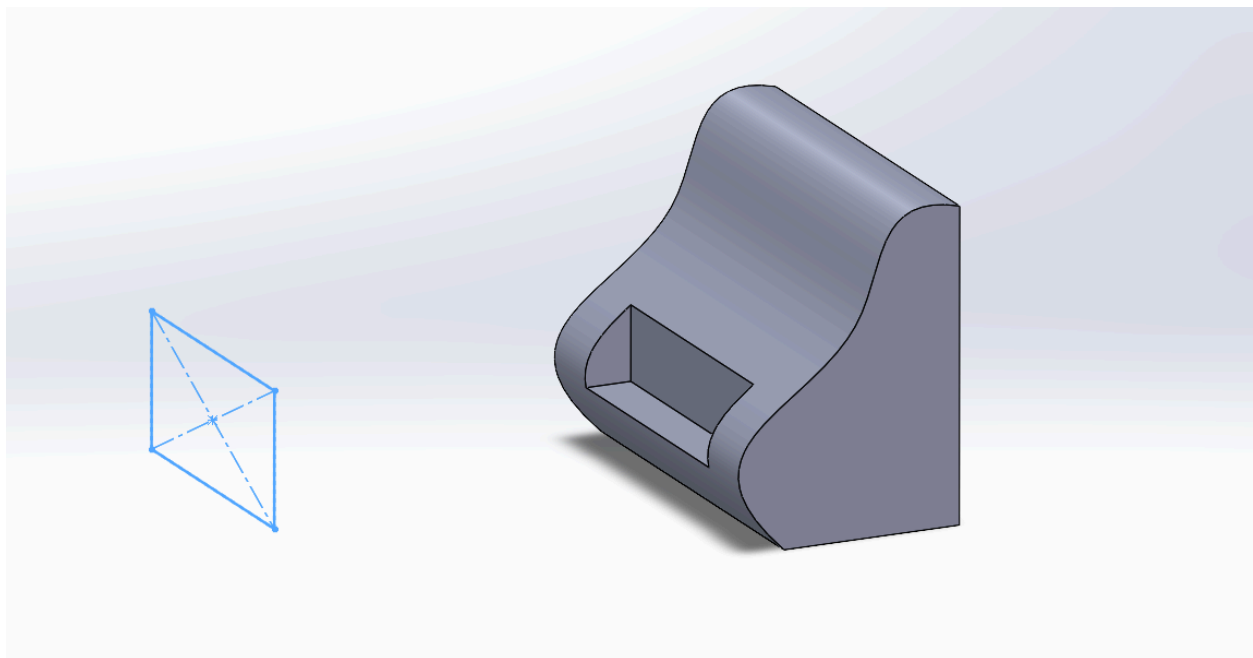
The next end condition is **Up To Next**. With this option, the cut goes up to the next face or body in front of the sketch. It also cuts through that selected geometry.



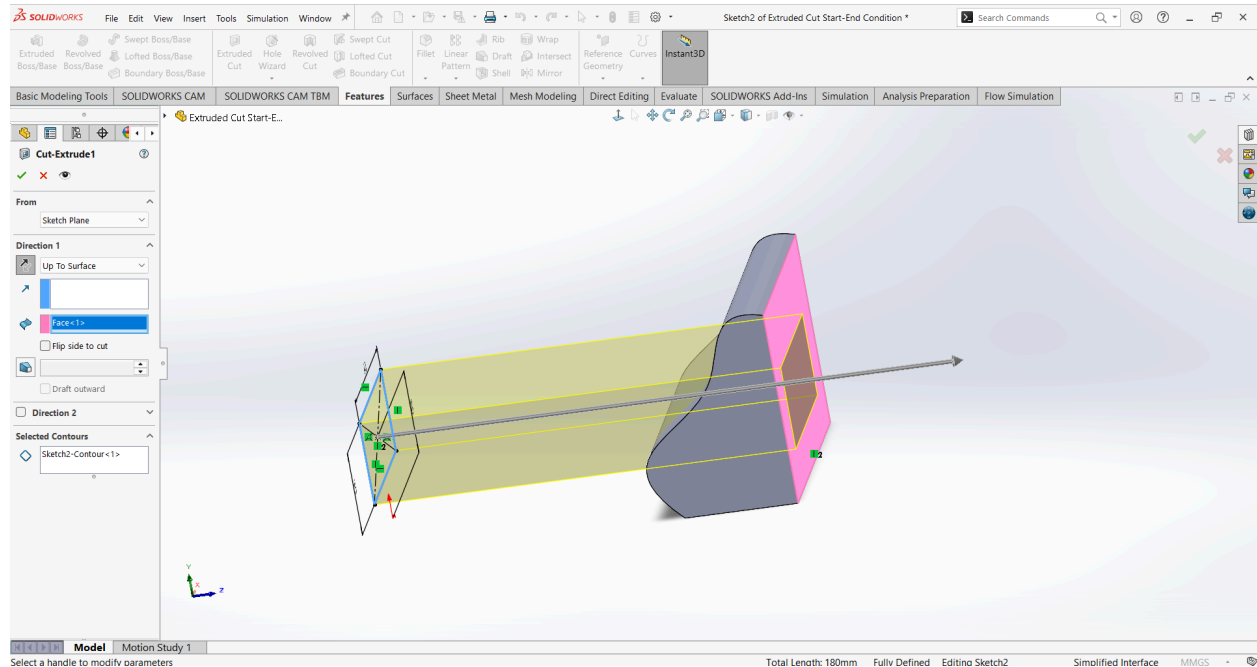
The next end condition is **Up To Vertex**. With this option, the cut goes up to the selected vertex. Click the selection box on the left side in the **PropertyManager**, then select a vertex on the model.



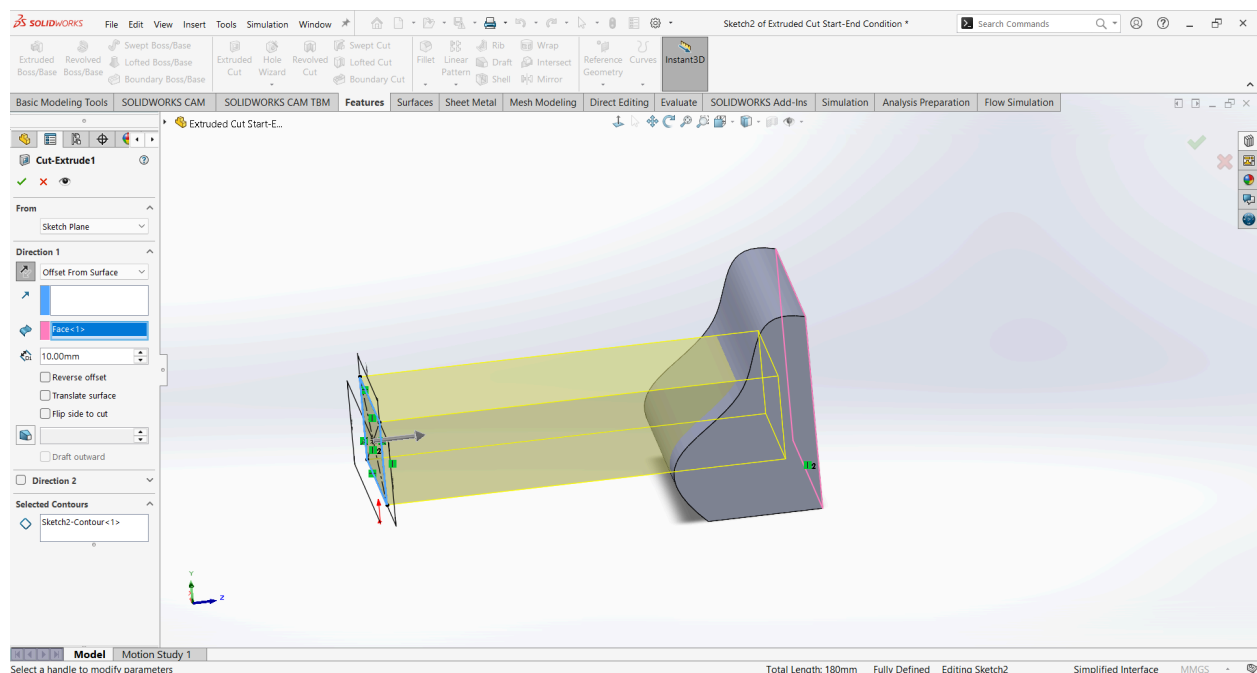
Here is the result.



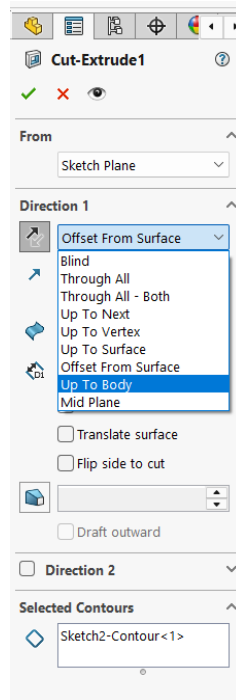
The next end condition is **Up To Surface**. With this option, the cut goes up to the surface you select. Click the selection box on the left side in the **PropertyManager**, then select a face on the model.



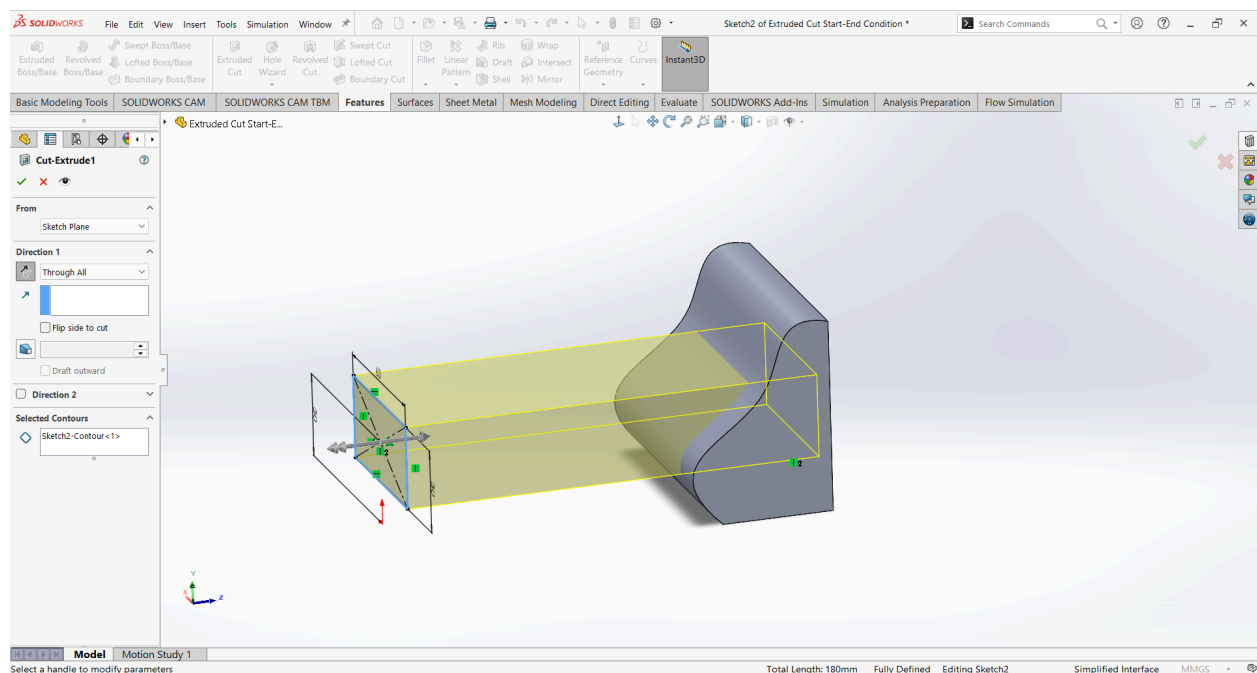
The next end condition is **Offset From Surface**. With this option, the cut stops at an offset distance from the selected surface. First, click the selection box on the left side in the **PropertyManager**. Then select a face on the model and define the offset distance.

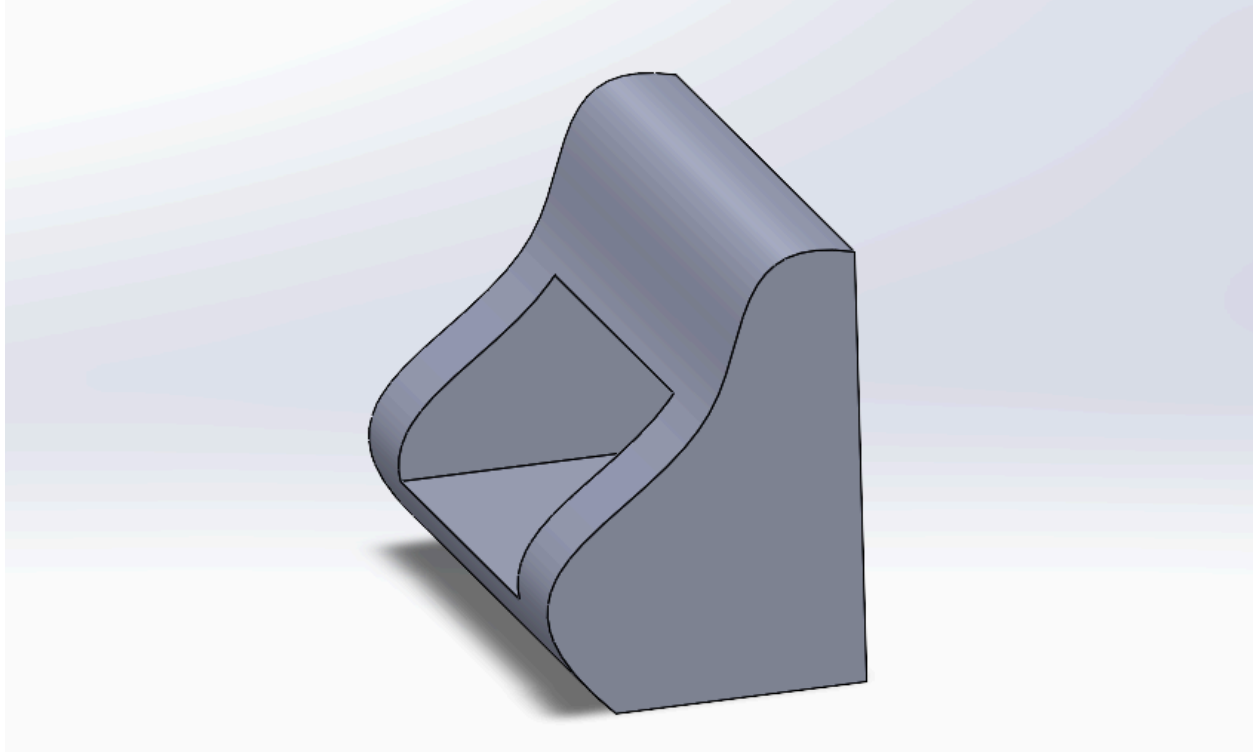


There are also **Up To Body** and **Mid Plane** options. These work the same way as they do when adding material with **Extruded Boss/Base**.



You can also change the cut direction by clicking the arrow on the left side in the **PropertyManager**. In this example, there is no material on the opposite side to cut.





16. Revolved Boss/Base

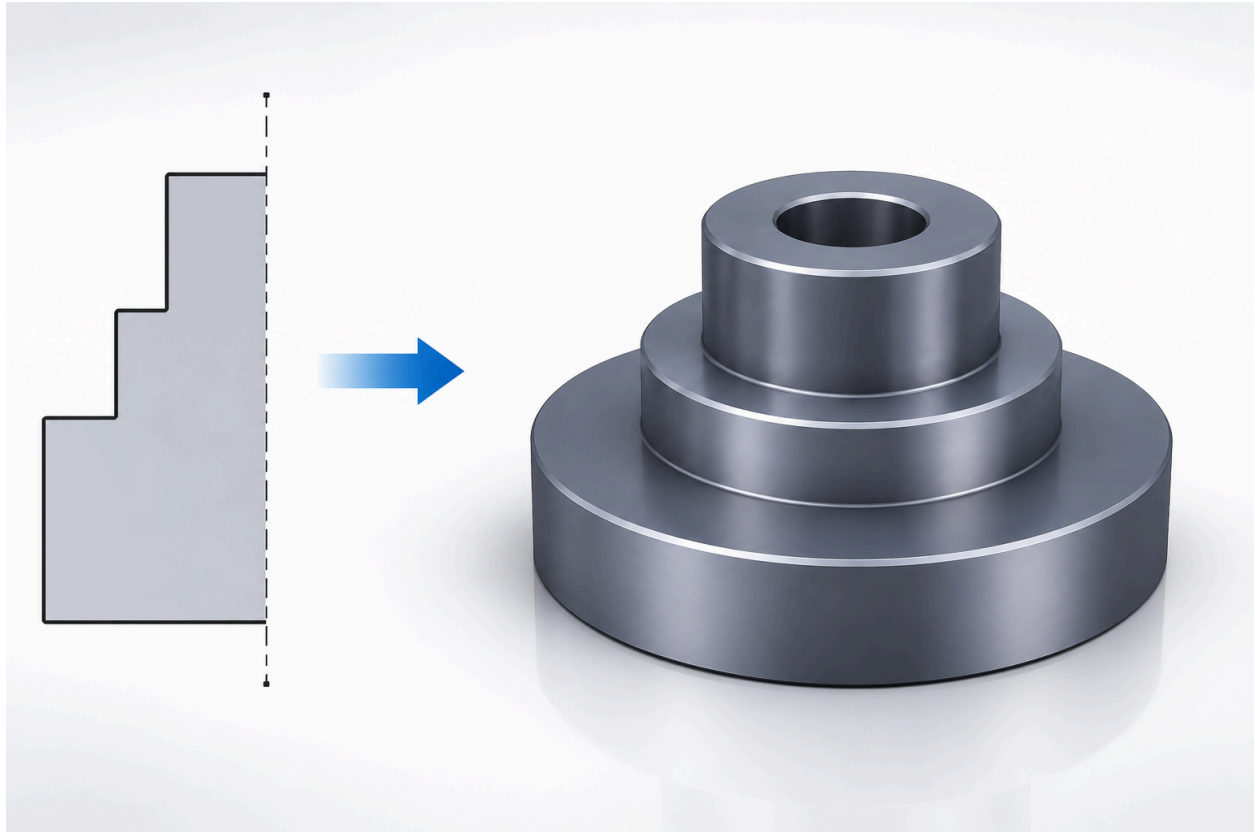
What it does: Creates a solid by revolving a sketch around an axis.

Use it for: Shafts, pulleys, wheels, handles, and rotational parts.

How it works:

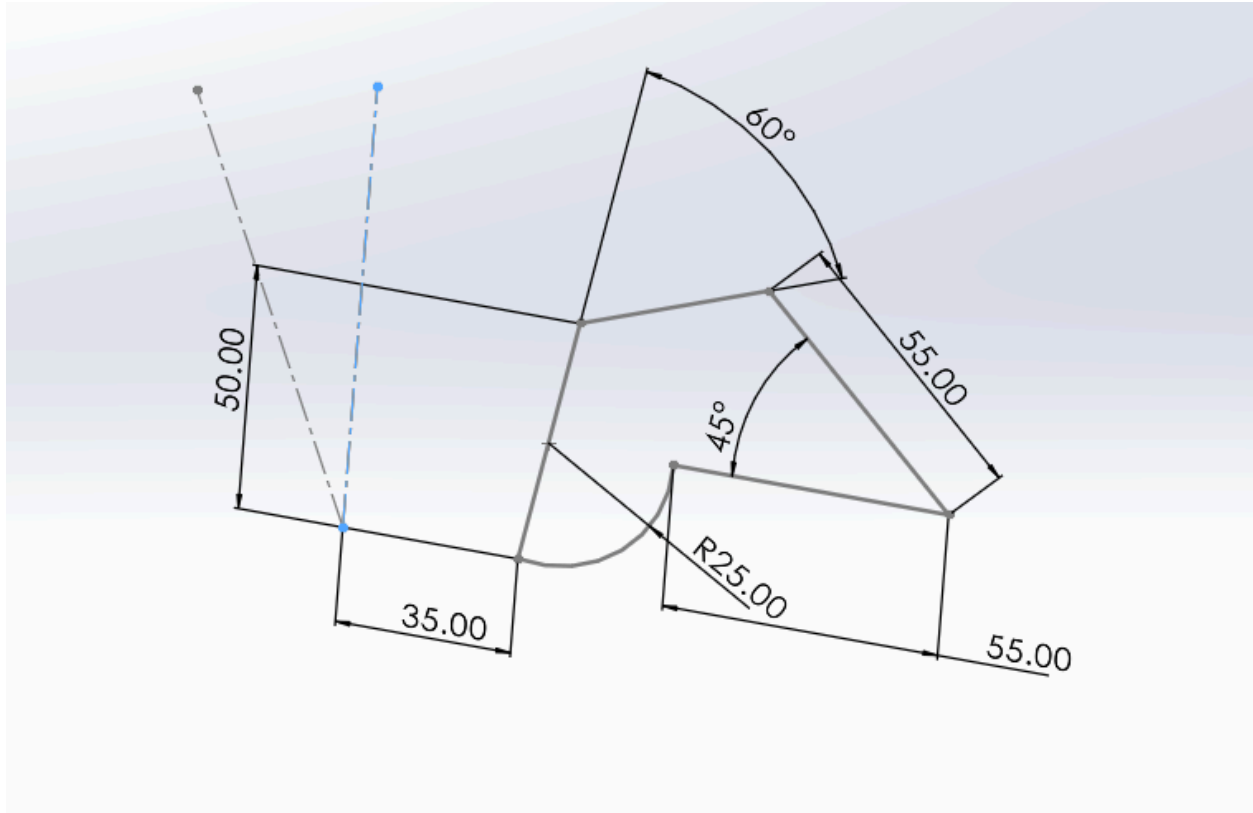
The **Revolved Boss/Base** tool is used to create a 3D feature by rotating a sketch around an axis.

Instead of extruding a sketch in a straight direction, this tool revolves the sketch around a centerline or selected edge.

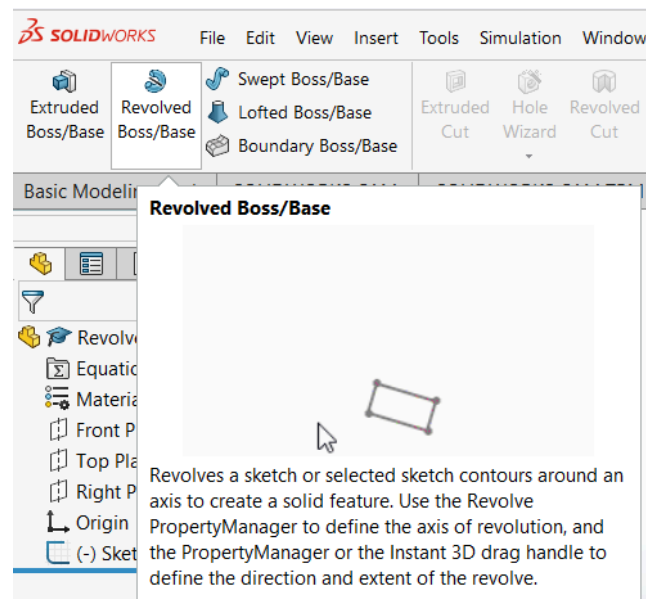


This is useful for creating round or symmetrical parts, such as shafts, wheels, pulleys, knobs, bottles, cylinders, and other rotational shapes.

First, select the axis of revolution, shown in blue.

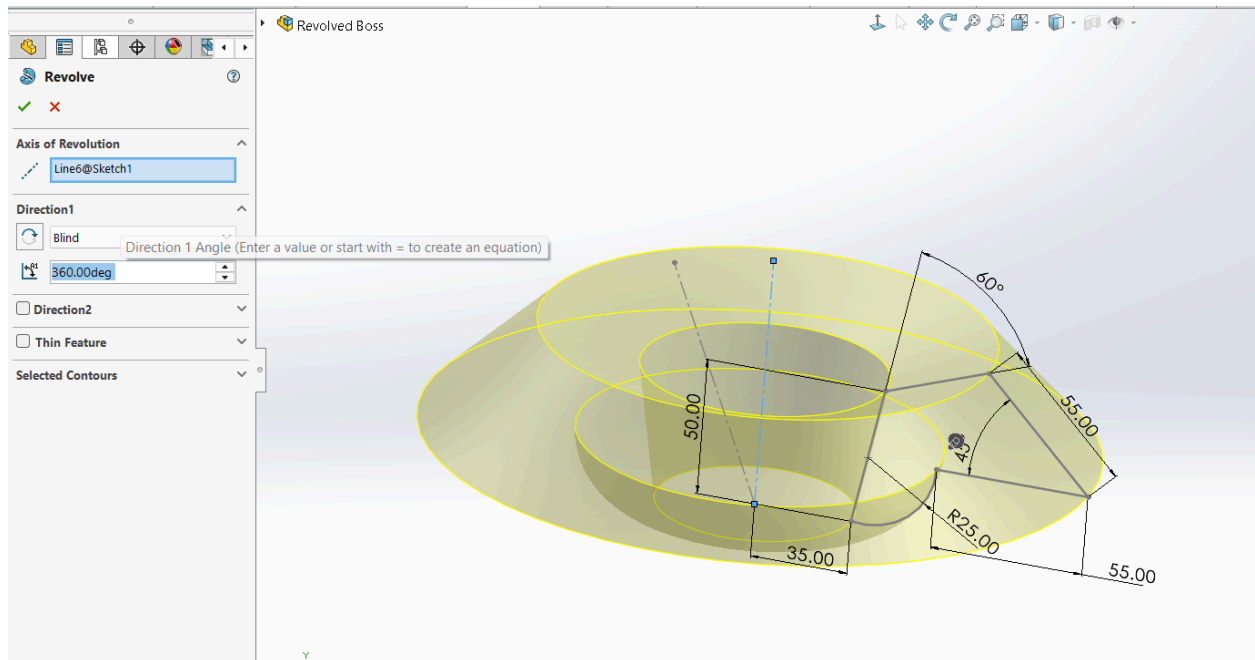


To access **Revolved Boss/Base**, go to the **Features** tab. Find the **Revolved Boss/Base** tool in the toolbar. Click **Revolved Boss/Base** to open the tool settings.

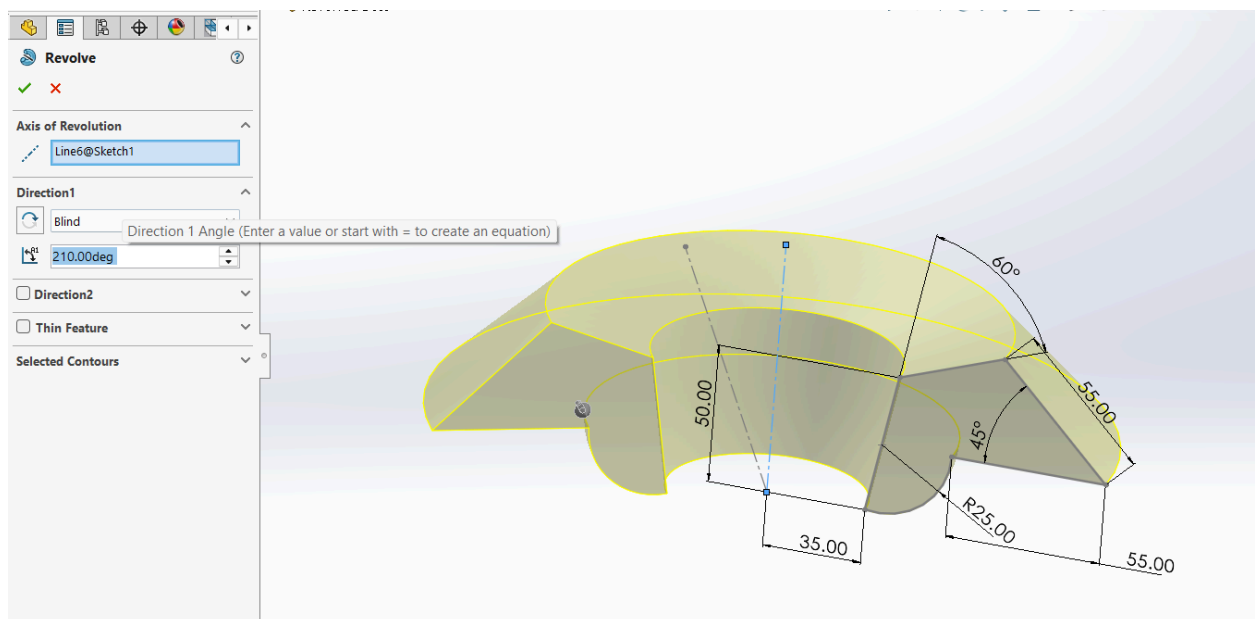


The **PropertyManager** will appear on the left side. Here, you can select the sketch profile, choose the axis of revolution, and define the rotation angle for the revolved feature.

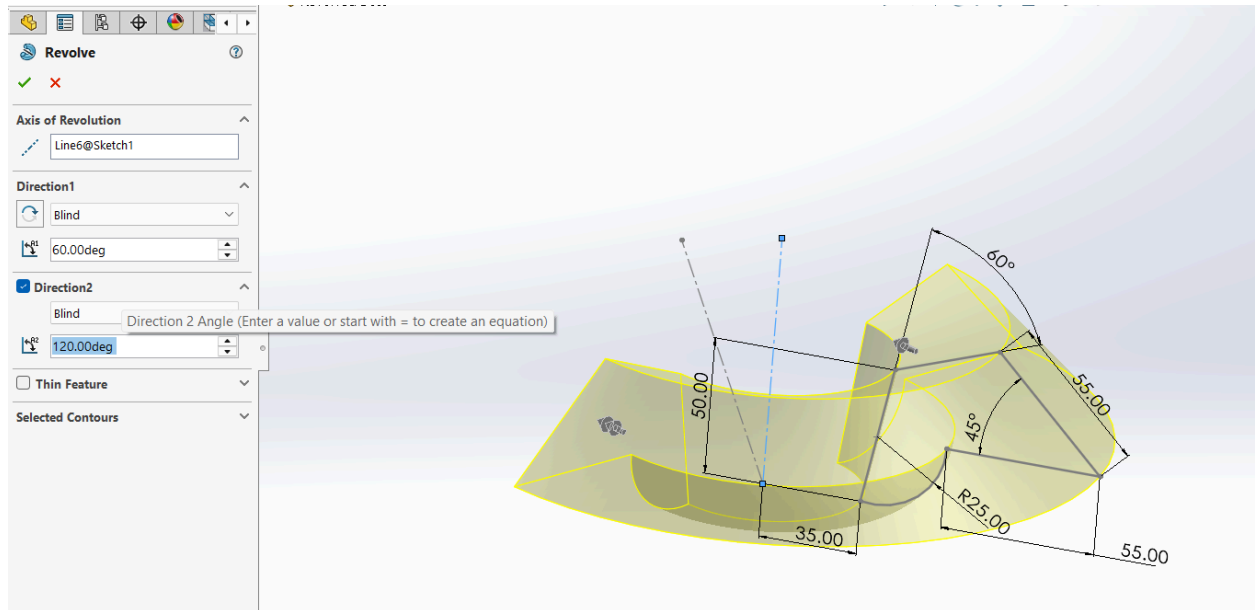
The **Axis of Revolution** is the axis we selected earlier. In the preview, you can see that the sketch rotates **360°** around this axis.



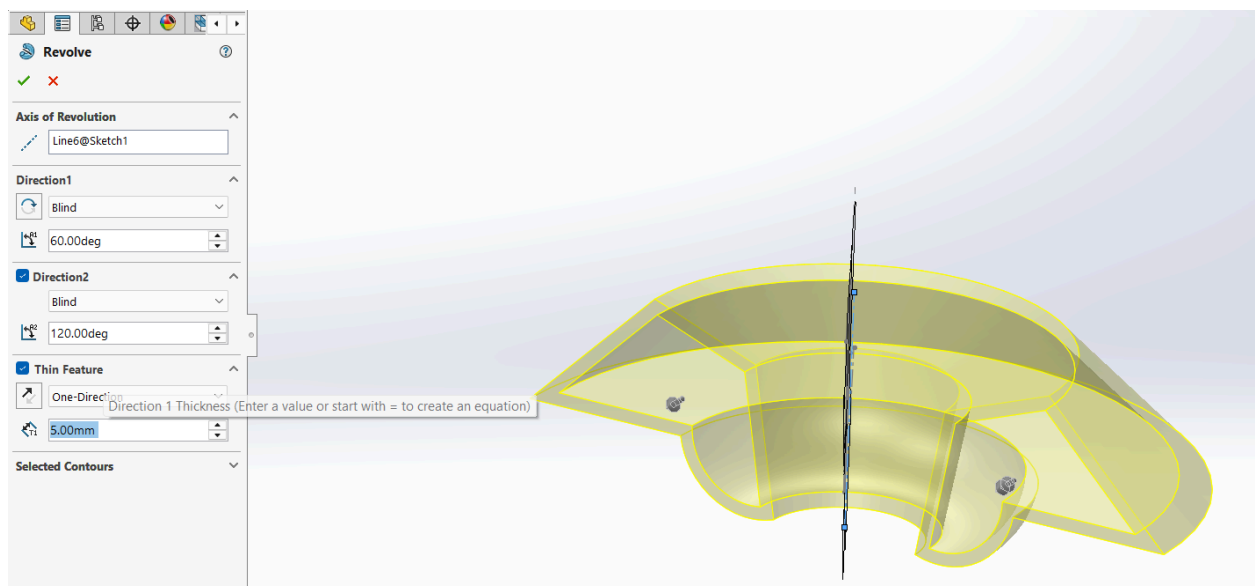
If you do not want a full **360°** revolution, you can change the angle value. For example, set the angle to **210°**. Now the sketch rotates only **210°** around the selected axis, creating this result.



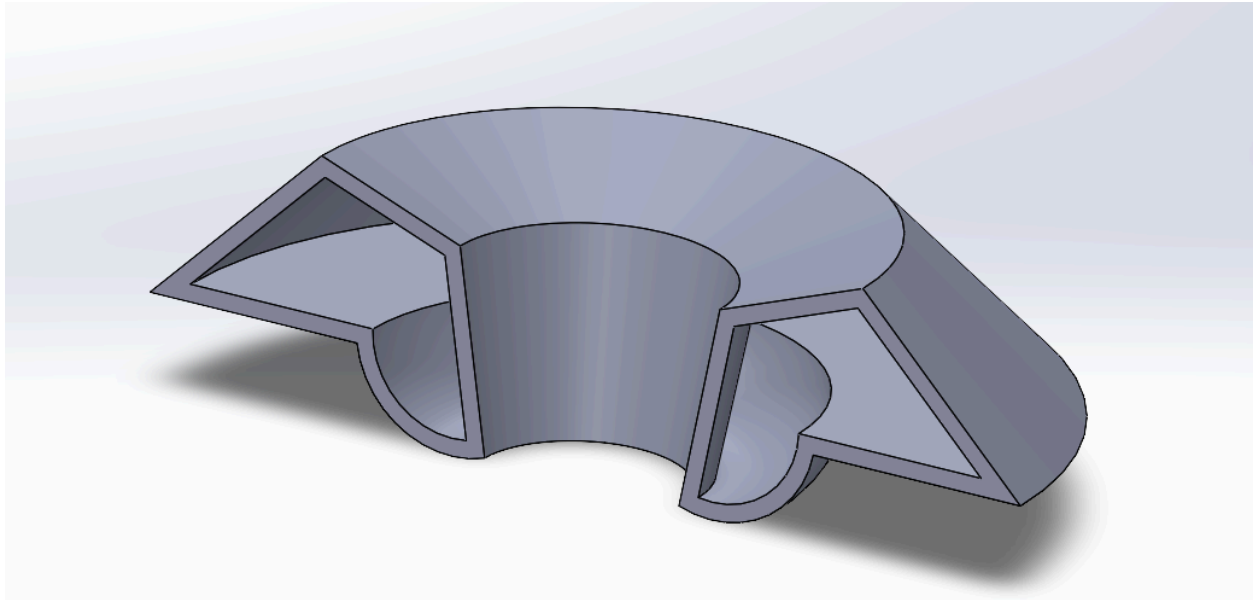
You can also turn on **Direction 2**. For **Direction 1**, set the angle to **60°**. For **Direction 2**, set the angle to **120°**. This is the final result.



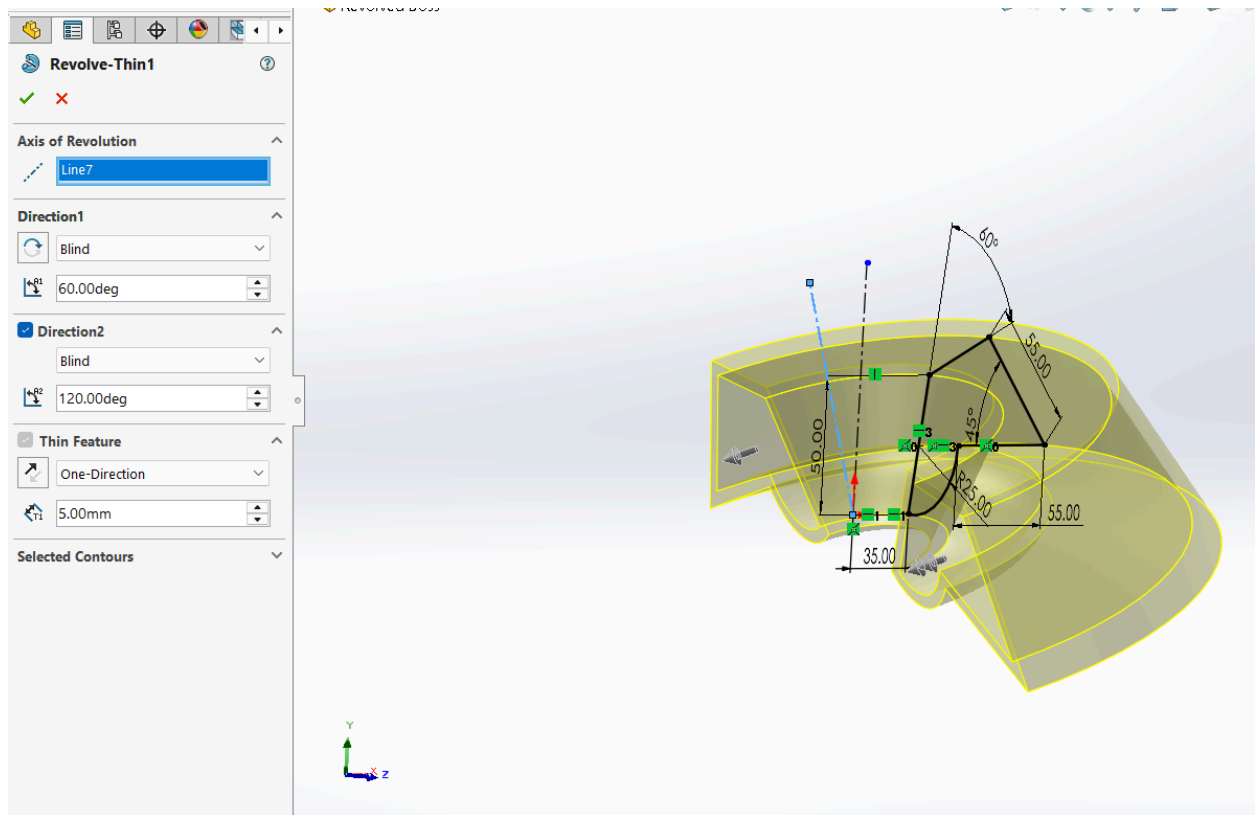
If you turn on the **Thin Feature** option, you can define a wall thickness for the revolved feature. In this example, set the thickness to **5 mm**. Now the feature creates a hollow body instead of a solid body.

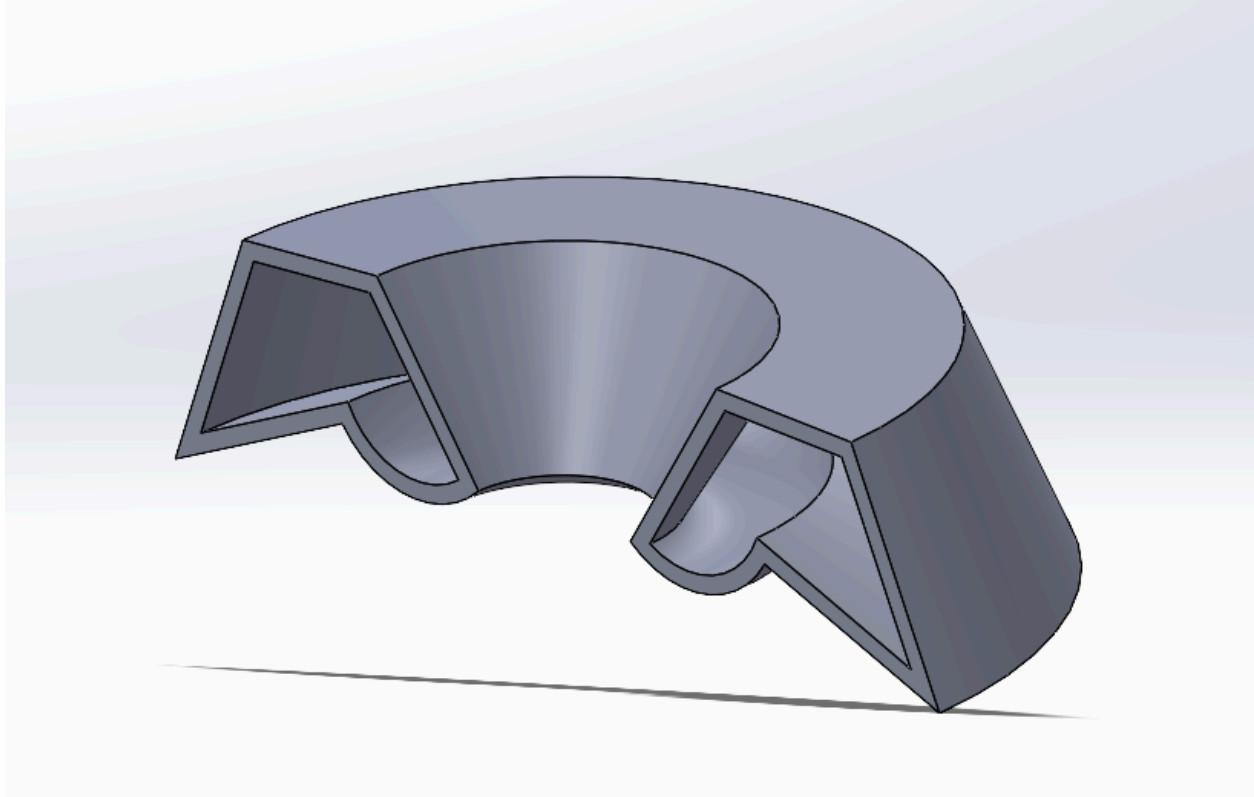


Click **OK**. This is the final result.



Now go back and edit the **Revolved Boss/Base** feature we created before. Change the **Axis of Revolution** and select the second axis. Now the same sketch rotates around a different axis, which slightly changes the shape of the model.





17. Sweep

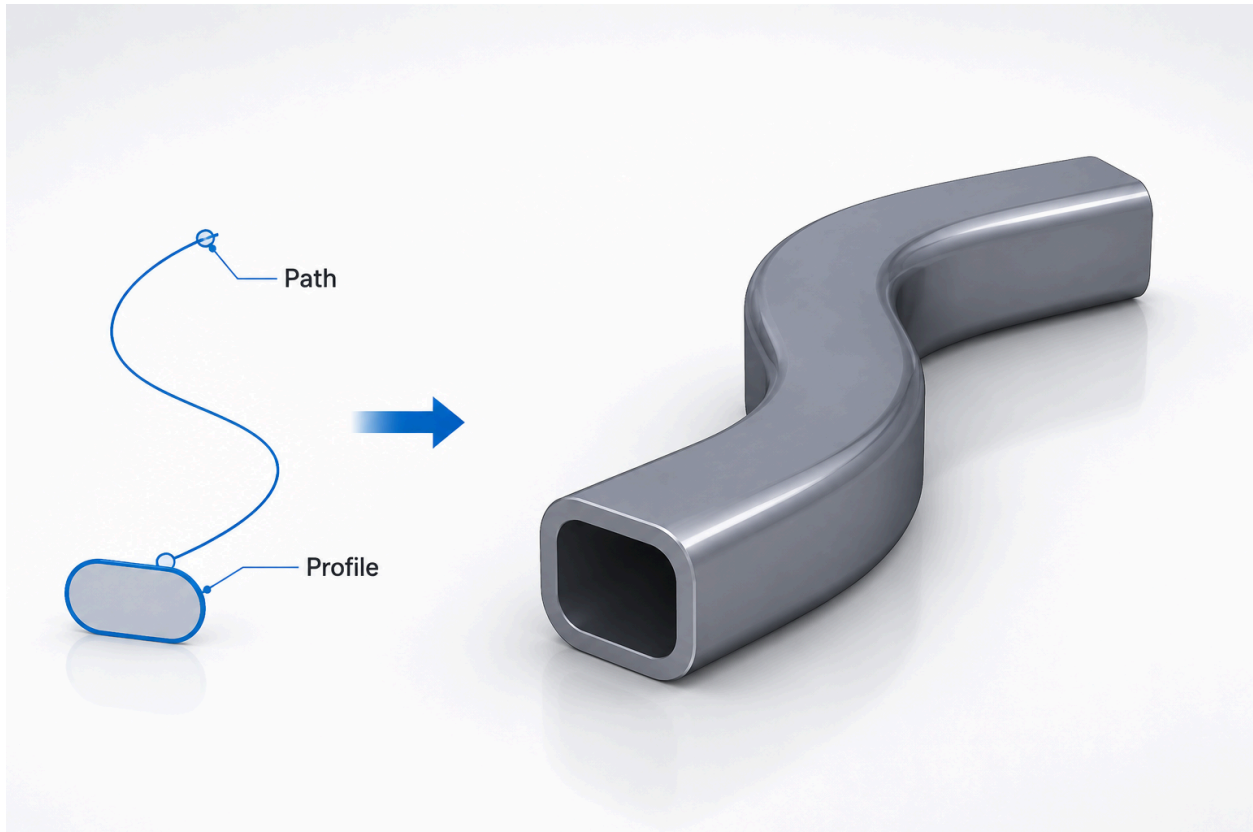
What it does: Creates geometry by moving a profile along a path.

Use it for: Pipes, tubes, handles, wires, and curved features.

How it works:

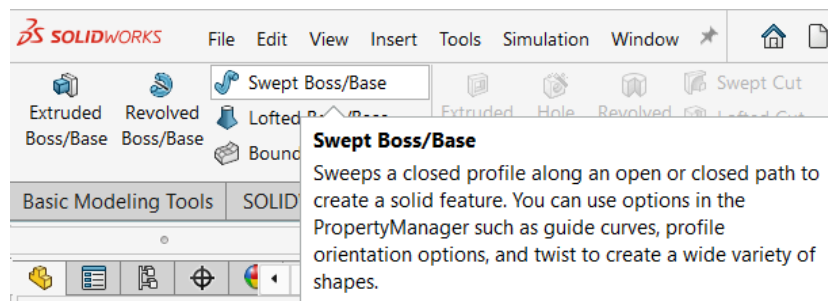
If you want to extrude a sketch profile along a curve or path, you can use the **Swept Boss/Base** tool.

The **Swept Boss/Base** tool creates a solid body by moving a sketch profile along a selected path. This is useful for creating shapes such as pipes, tubes, handles, wires, frames, or curved features.

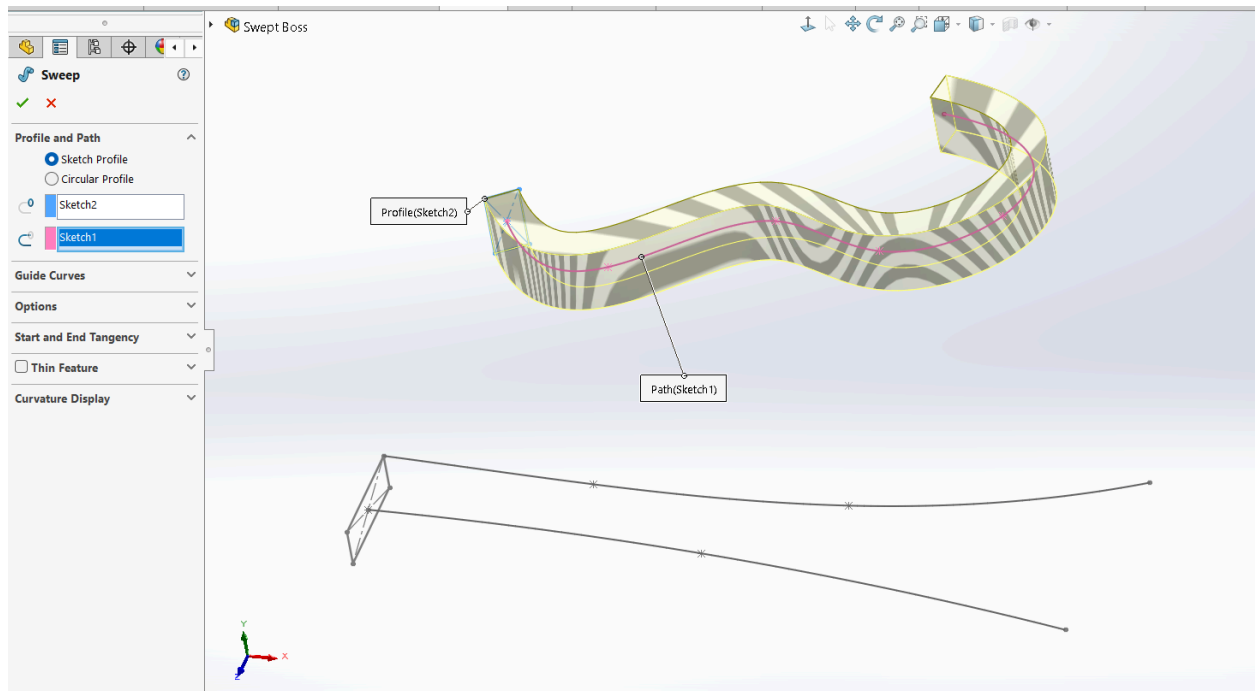


To create a sweep, you need two main elements: a **profile sketch** and a **path sketch**.

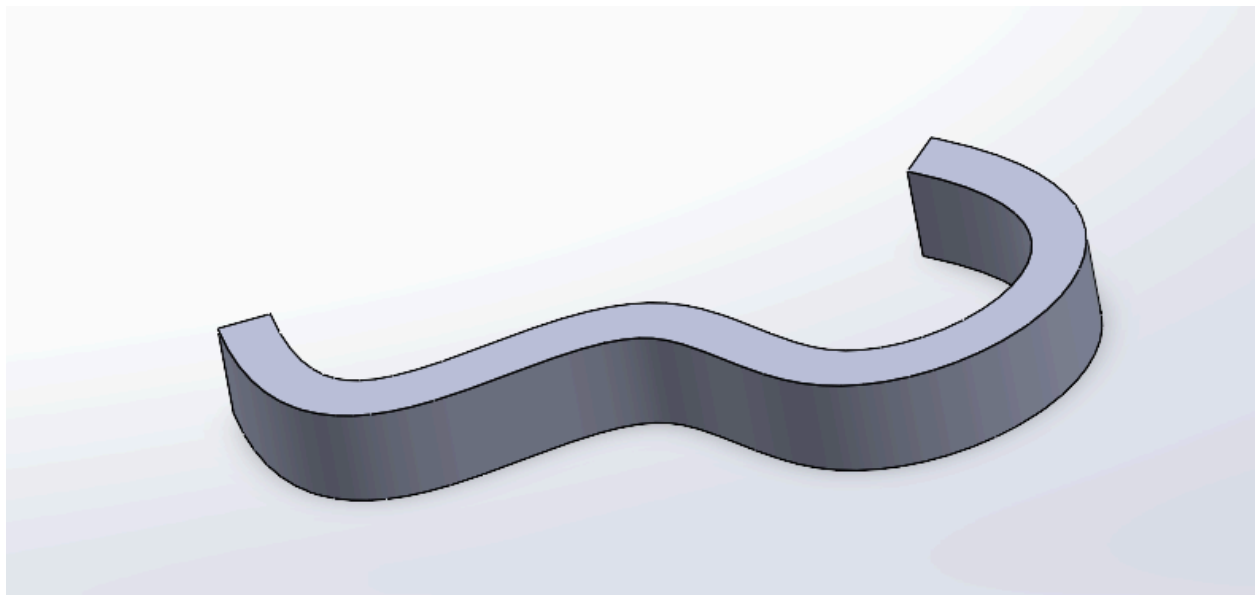
Go to the **Features** tab and select **Swept Boss/Base**.



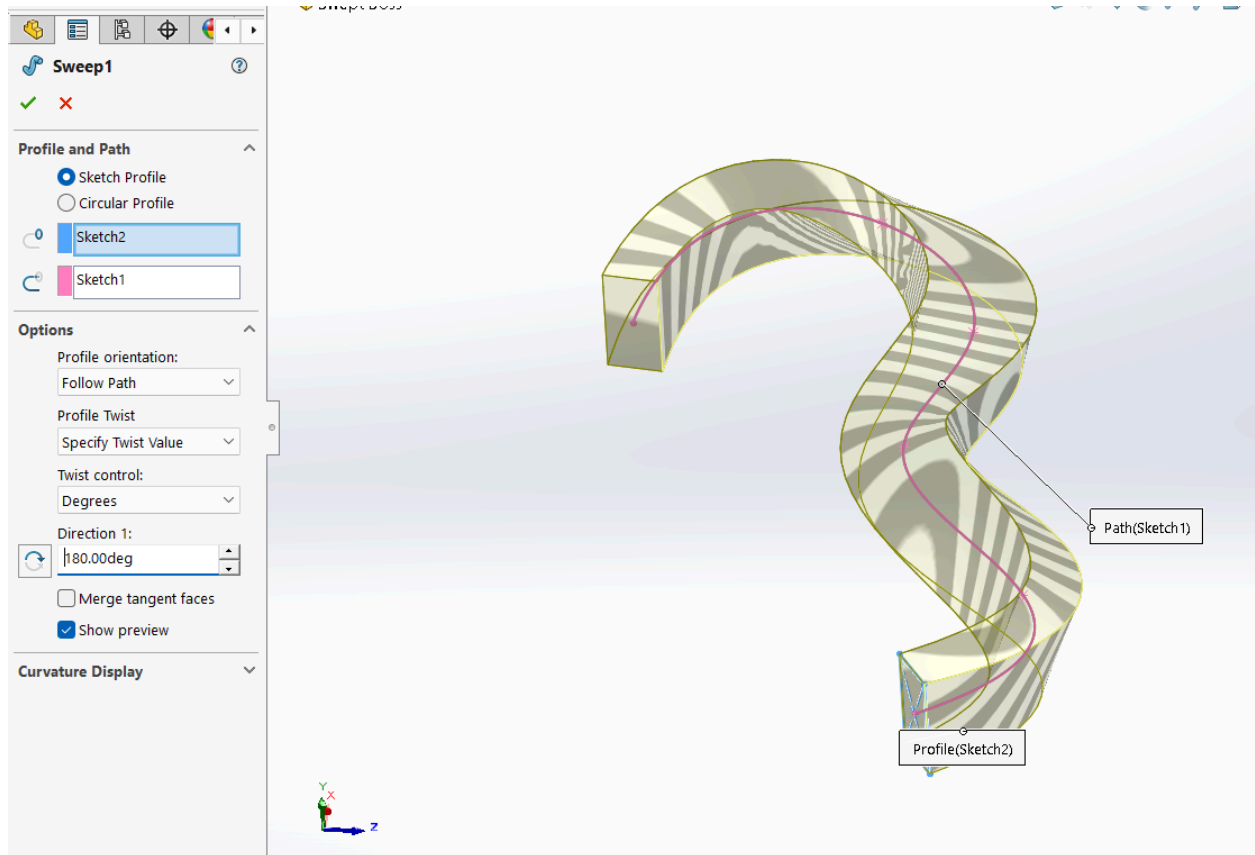
The sweep properties appear on the left side in the **Property Manager**. Select the first option, **Sketch Profile**. In the first selection box, select the profile sketch. In the second selection box, select the curve or path. You can see the sweep preview before confirming the feature.



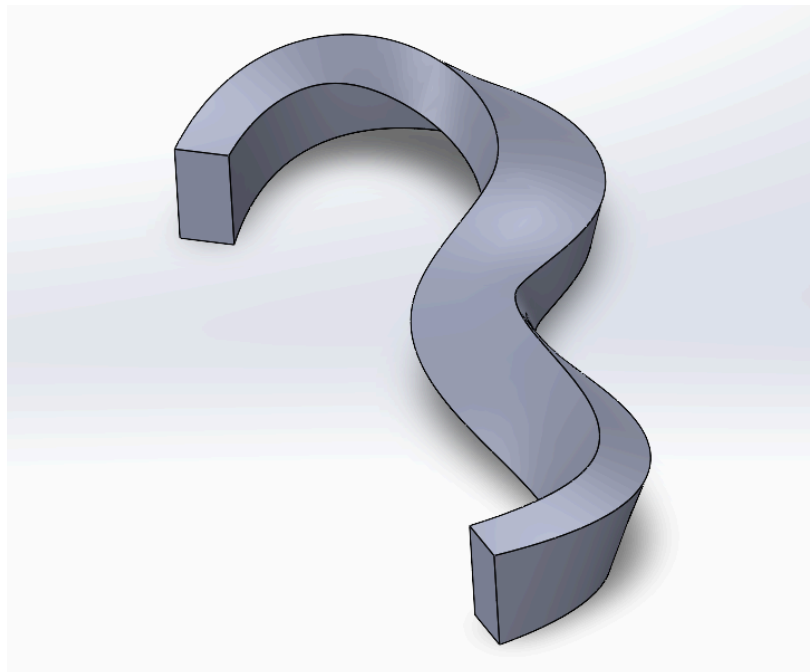
Click **OK**. This is the final result.



Go back and edit the same **Swept Boss/Base** feature. In **Options**, change **Profile Twist** to **Specify Twist Value**. For **Twist Control**, select **Degrees**. Set the twist angle to **180°** and select **Direction 1**. Now you can see that the profile twists as it follows the curve.

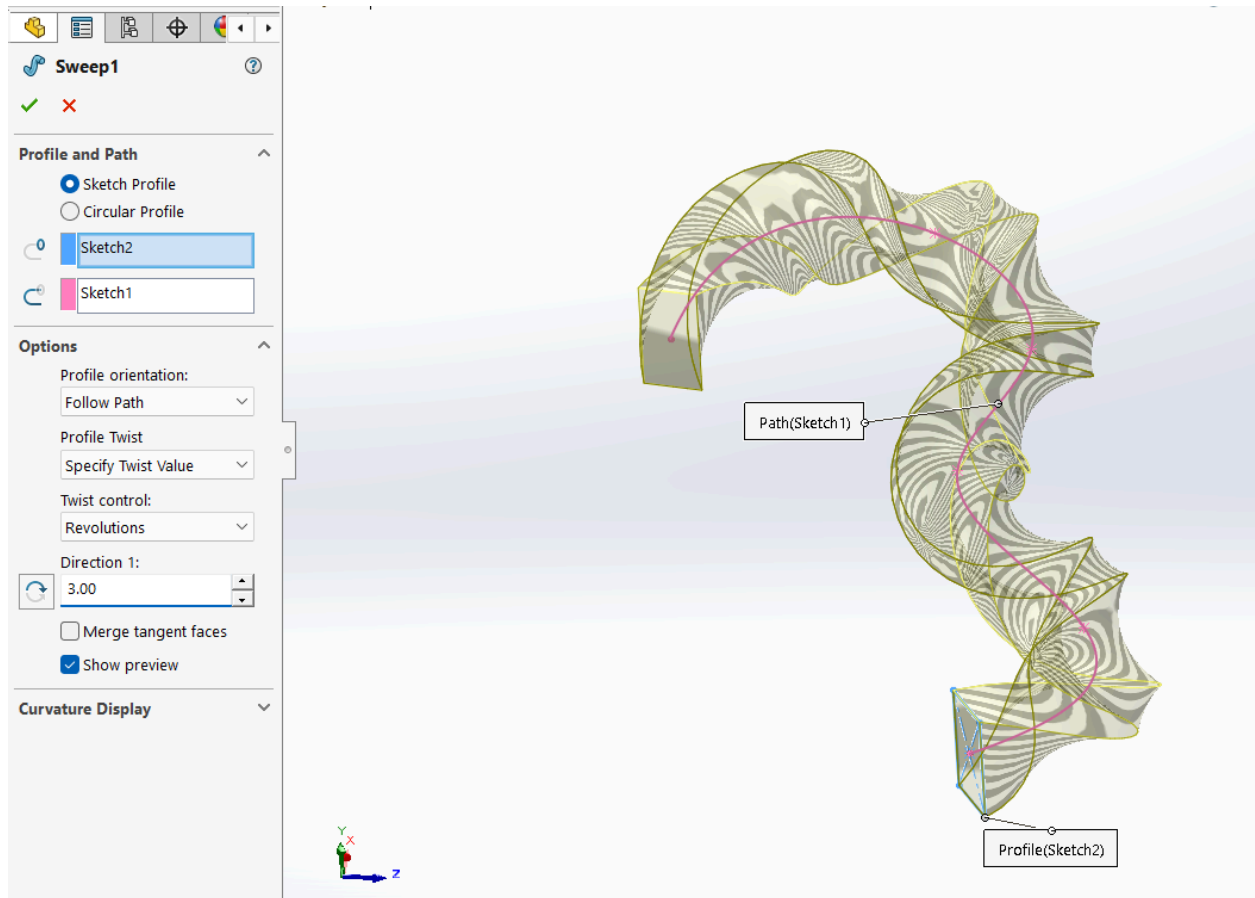


Click **OK**. This is the final result.

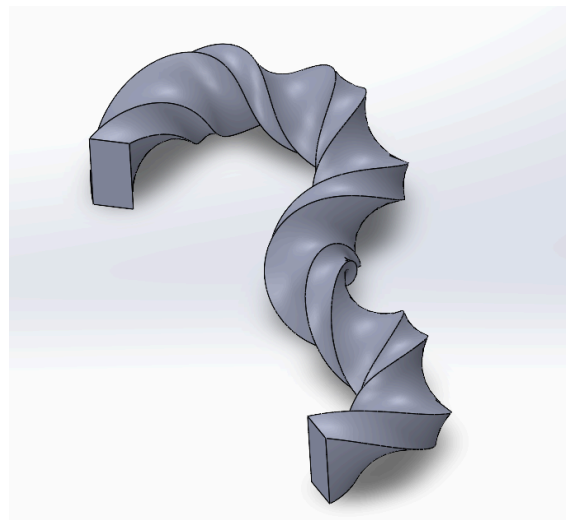


Go back and edit the same **Swept Boss/Base** feature.

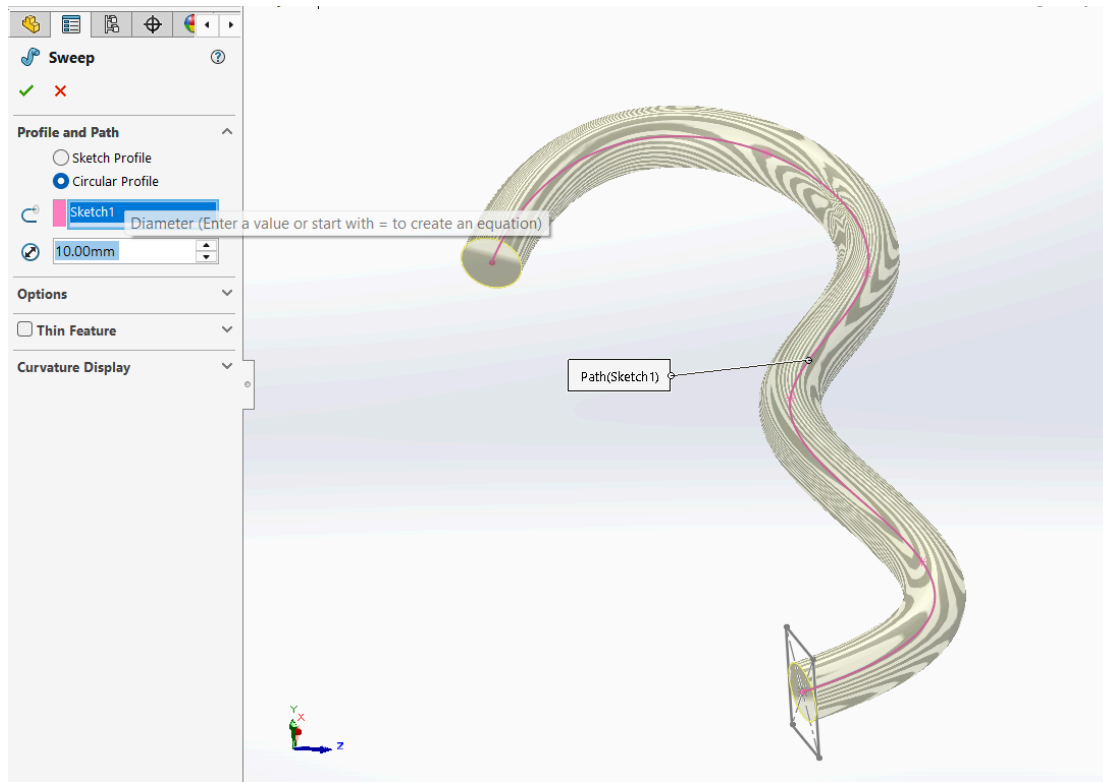
Instead of **Degrees**, select **Revolutions** for the twist control. For **Direction 1**, enter **3**. This means the profile will make **3 full revolutions** as it follows the curve. This is the preview.



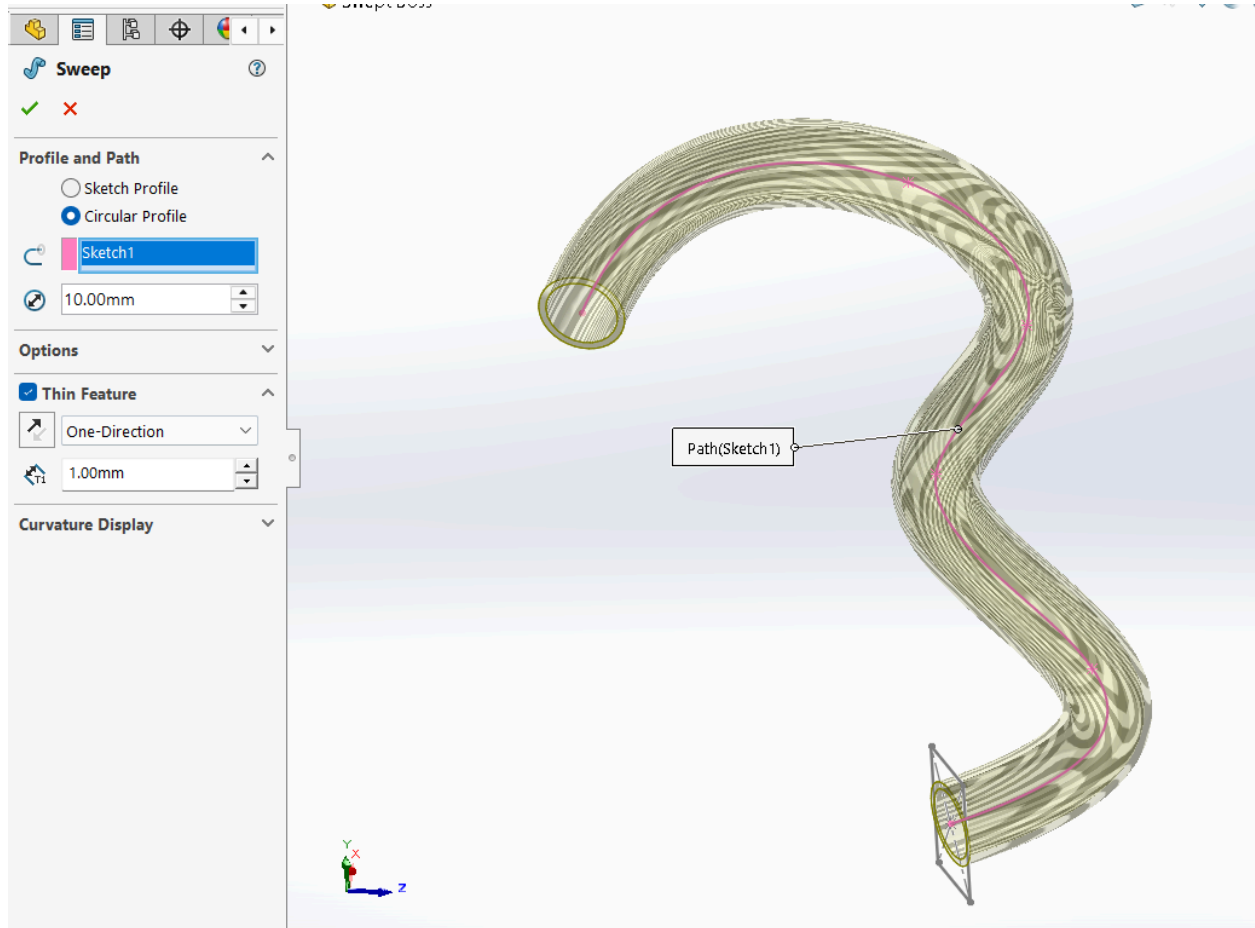
Click **OK**. This is the final result.



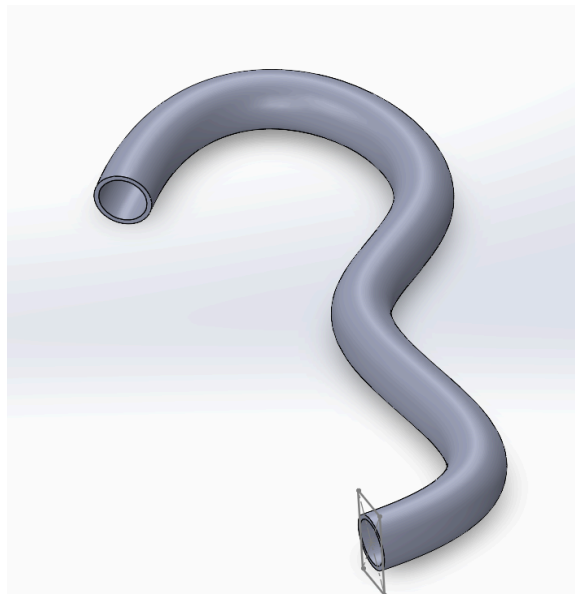
Now go back and edit the same **Swept Boss/Base** feature. Select the second option, **Circular Profile**. With this option, SOLIDWORKS creates a circular profile automatically, so you do not need to select a separate profile sketch. The circle is swept along the selected curve or path. Define the diameter of the circle. In this example, set the diameter to **10 mm**.



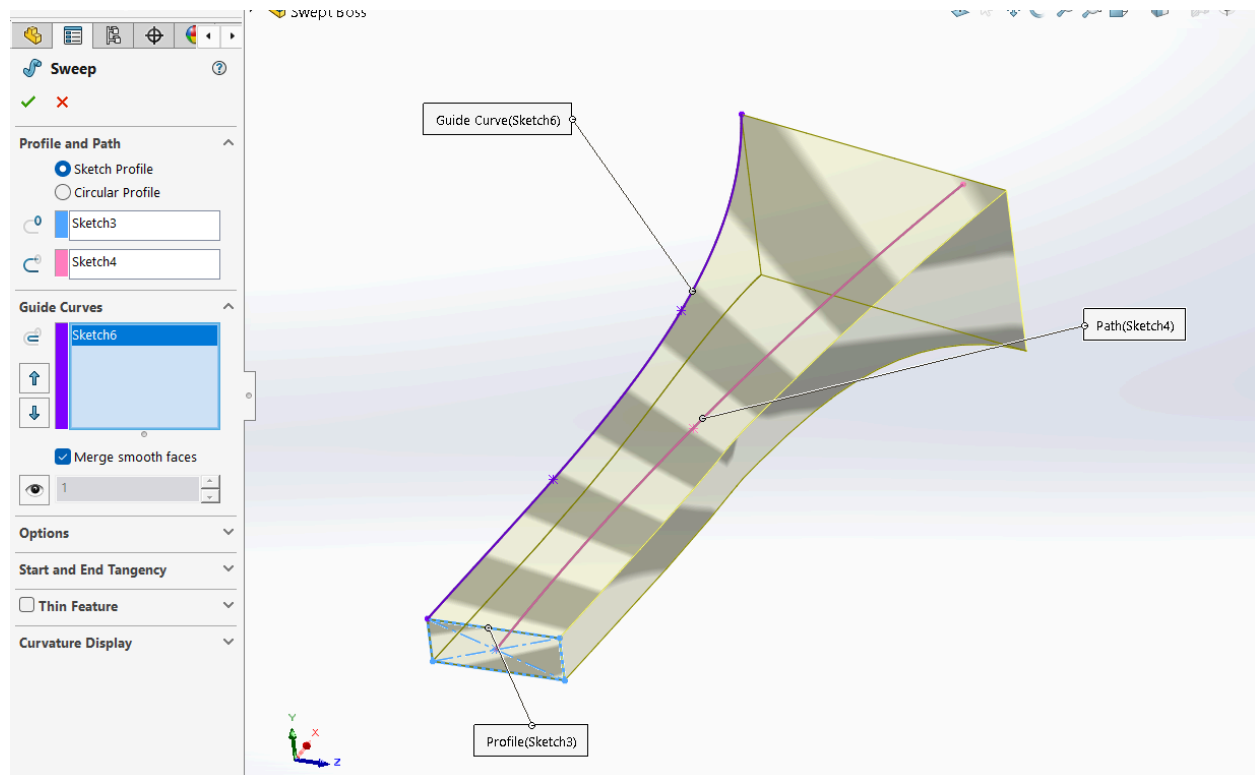
You can also use the **Thin Feature** option. Turn on **Thin Feature** and set the thickness to **1 mm**. Then click **OK**.



Now we have created a hollow curved tube.

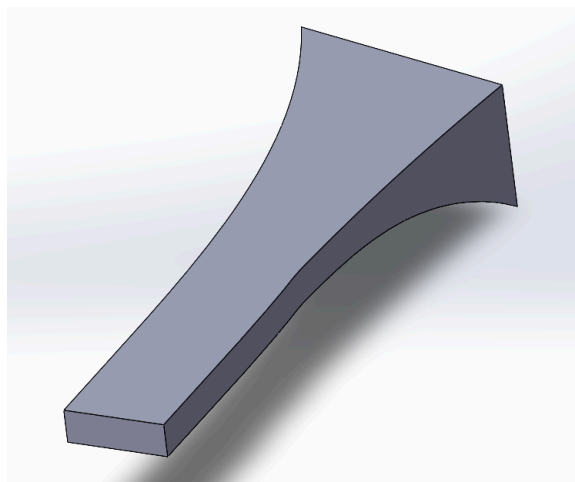


In the second example, we will also use **Guide Curves**. Select the profile sketch and the path curve as before. Then open the **Guide Curves** selection box and select the purple line. The guide curve helps control the shape of the sweep as the profile follows the path.



Click **OK**. This is the final result. The **Guide Curve** affects the shape of the model. It controls how the profile changes as it moves along the path.

Without guide curves, the cross-section stays constant along the path. With guide curves, the cross-section can change and follow the guide curve, creating a more controlled and complex shape.



18. Loft

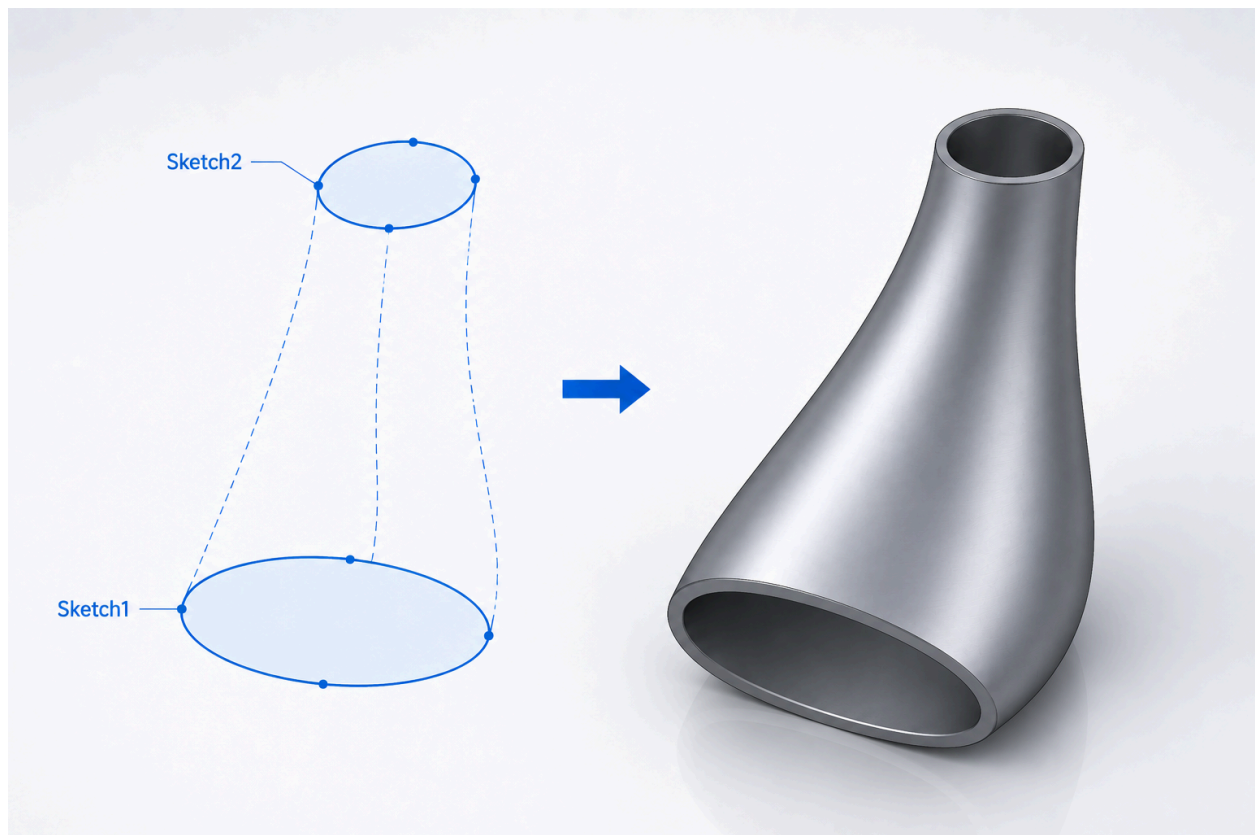
What it does: Creates a transition between two or more profiles.

Use it for: Shapes that change size or geometry from one section to another.

How it works:

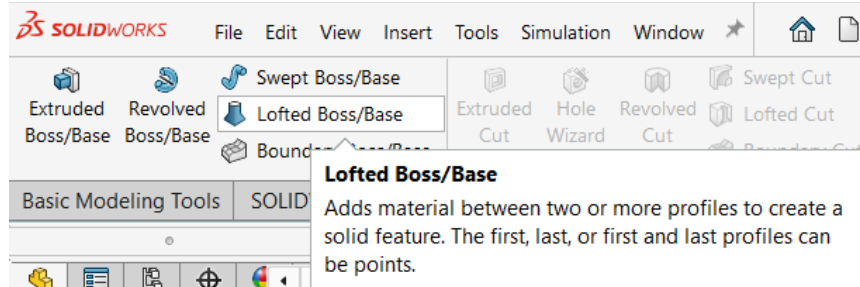
To connect two or more sketch profiles into one solid body, we use the **Lofted Boss/Base** tool.

The **Lofted Boss/Base** tool creates a solid shape by blending between multiple sketches. This is useful when you want to create a model that changes shape from one profile to another, such as from a circle to a rectangle, or between different-sized profiles.



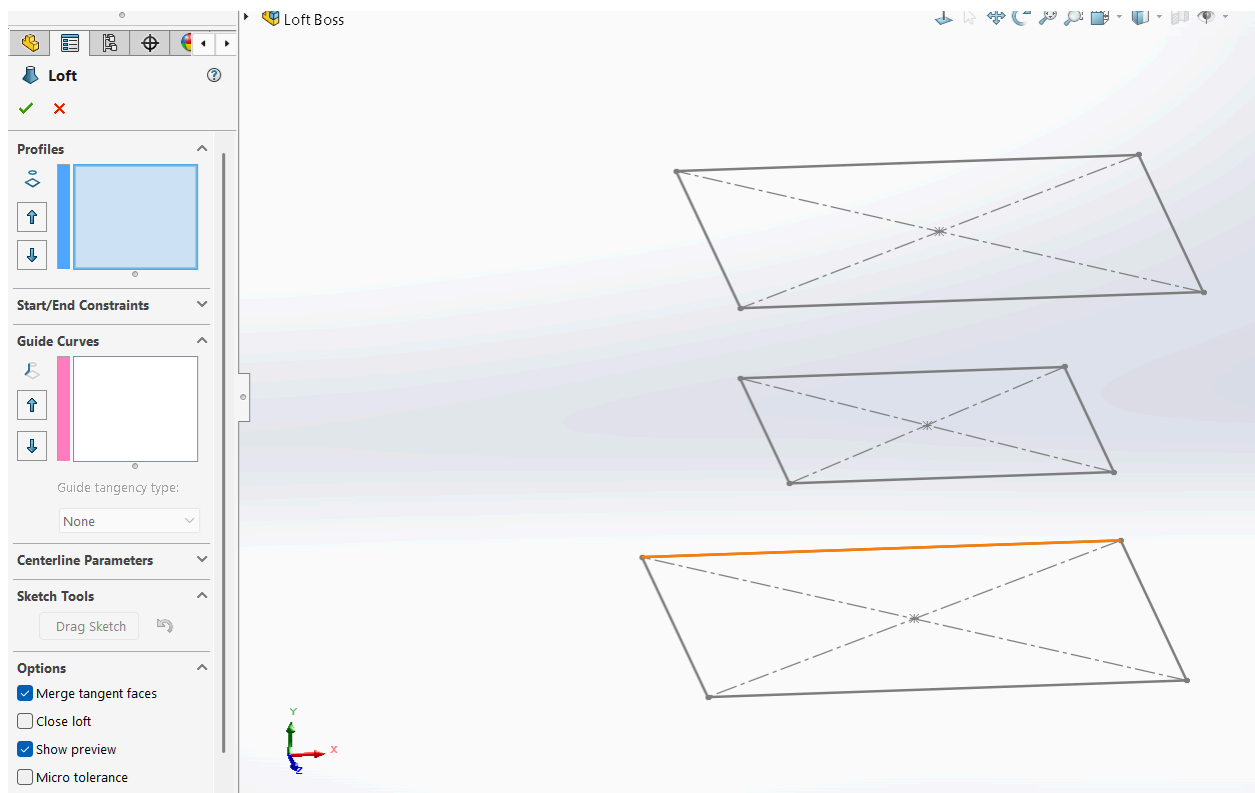
The sketches are usually created on different parallel planes.

Go to the **Features** tab and select **Lofted Boss/Base**.

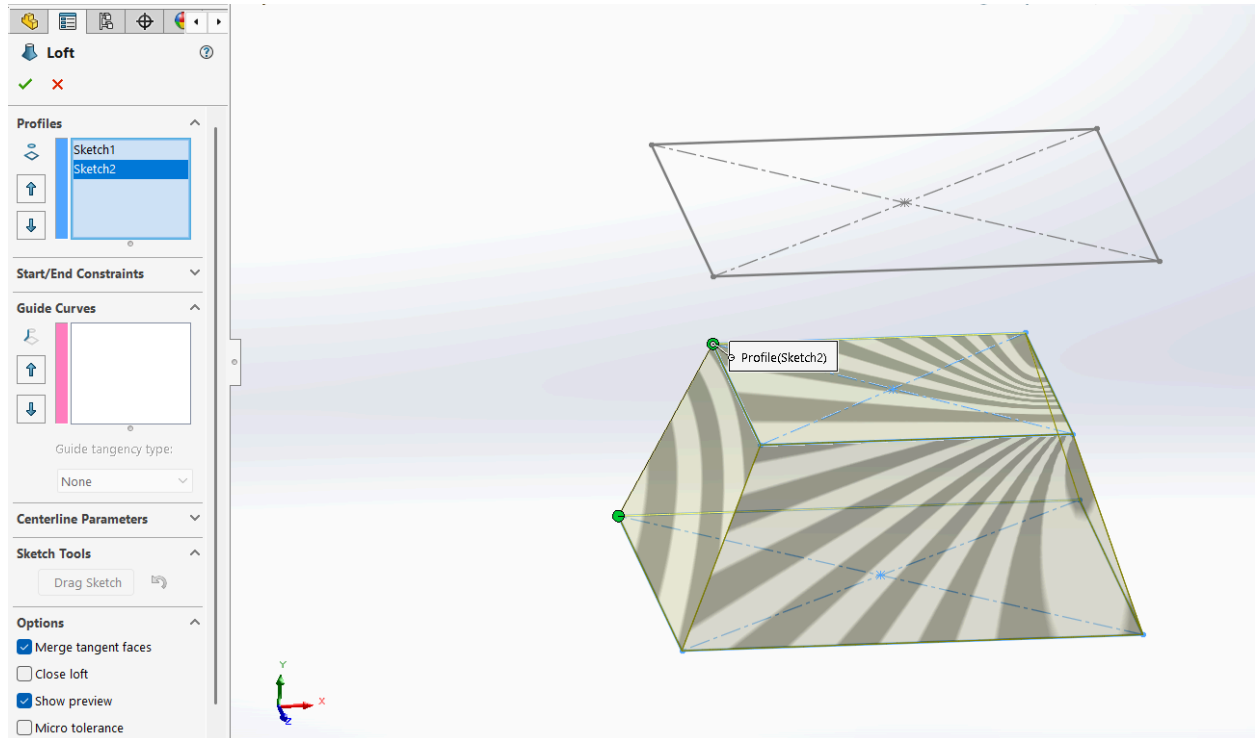


The loft properties appear on the left side in the **PropertyManager**.

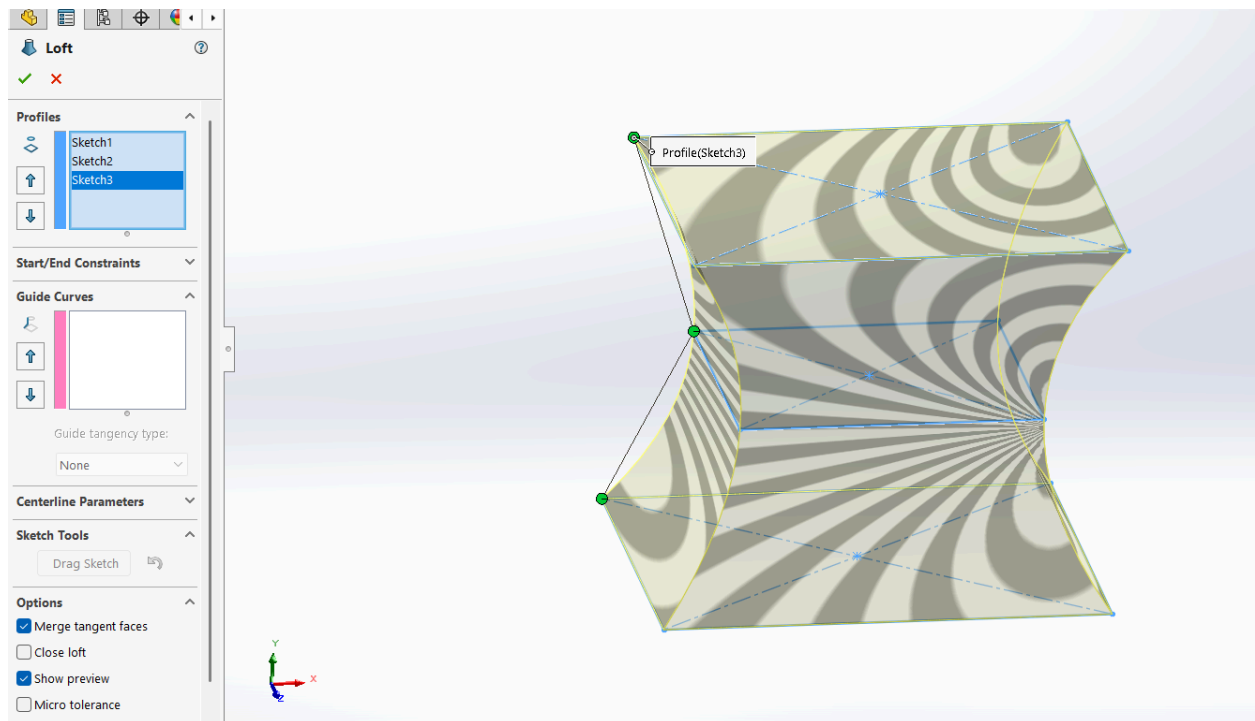
In the **Profiles** section, select the sketch profiles you want to connect. First, select the first profile.



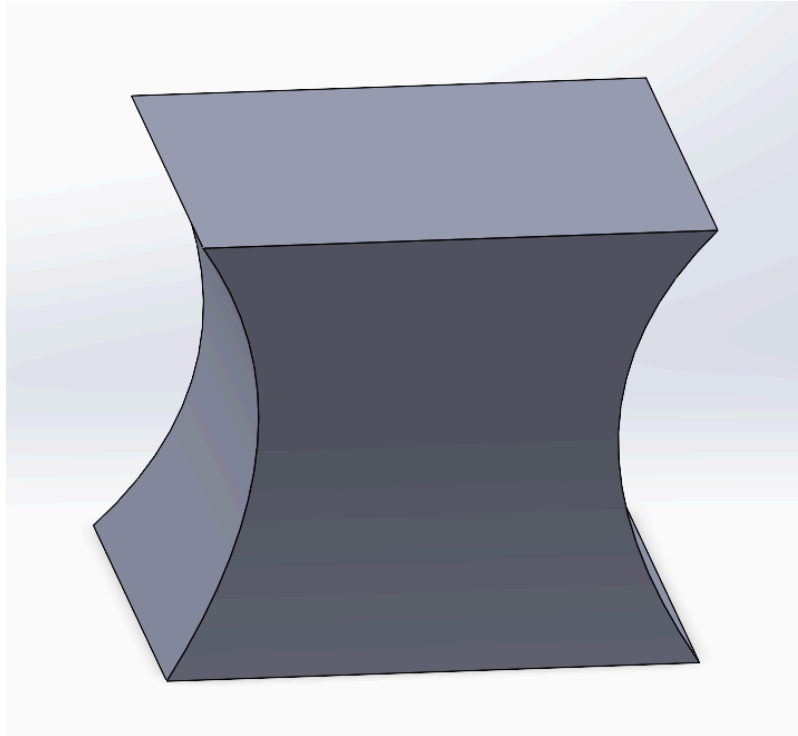
Now select the second profile. Make sure you select both profiles from the same side. If the profiles are selected from opposite sides, the loft preview can become twisted.



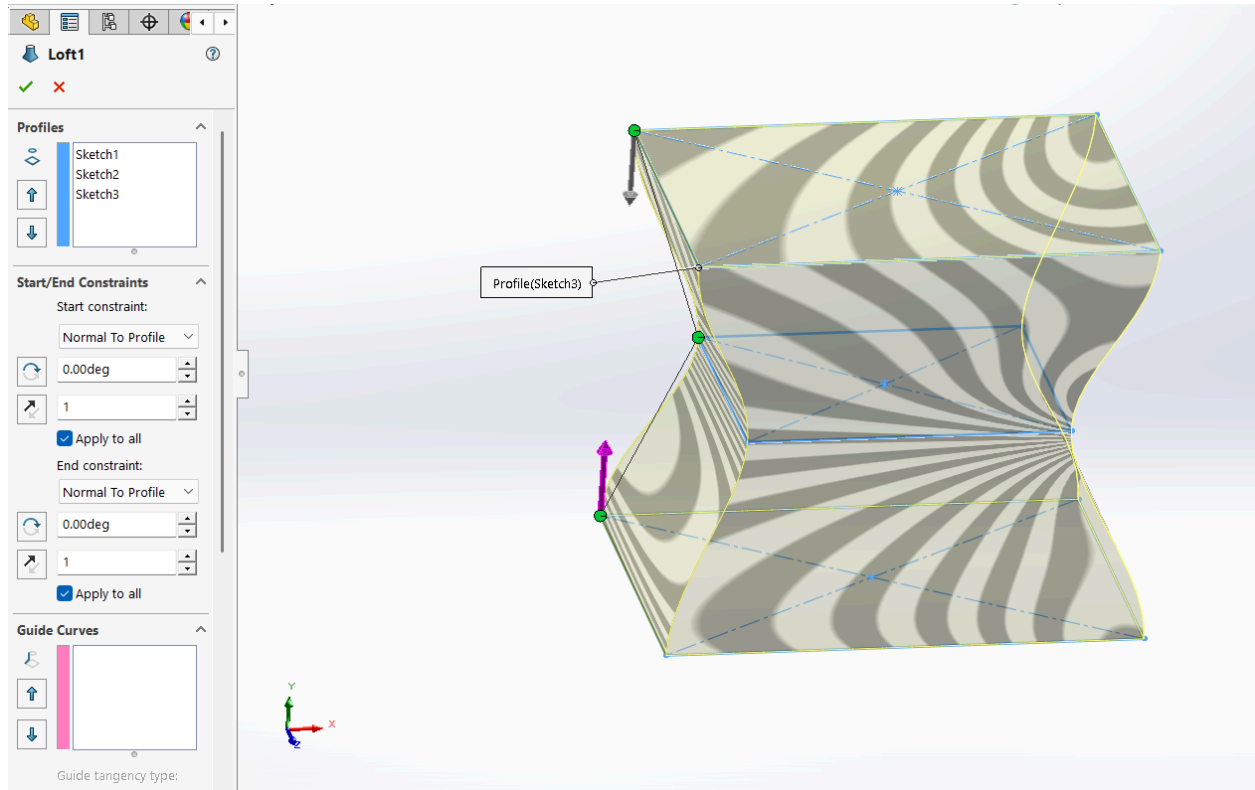
Now select the third profile. After selecting all three profiles, SOLIDWORKS creates a 3D body by blending the profiles together.



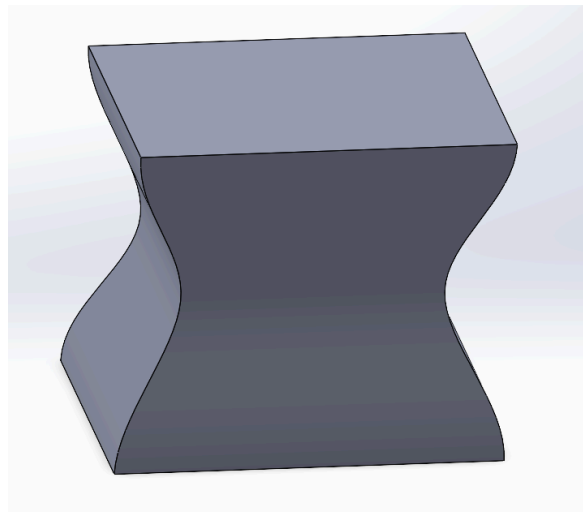
Click **OK**. This is the final result.



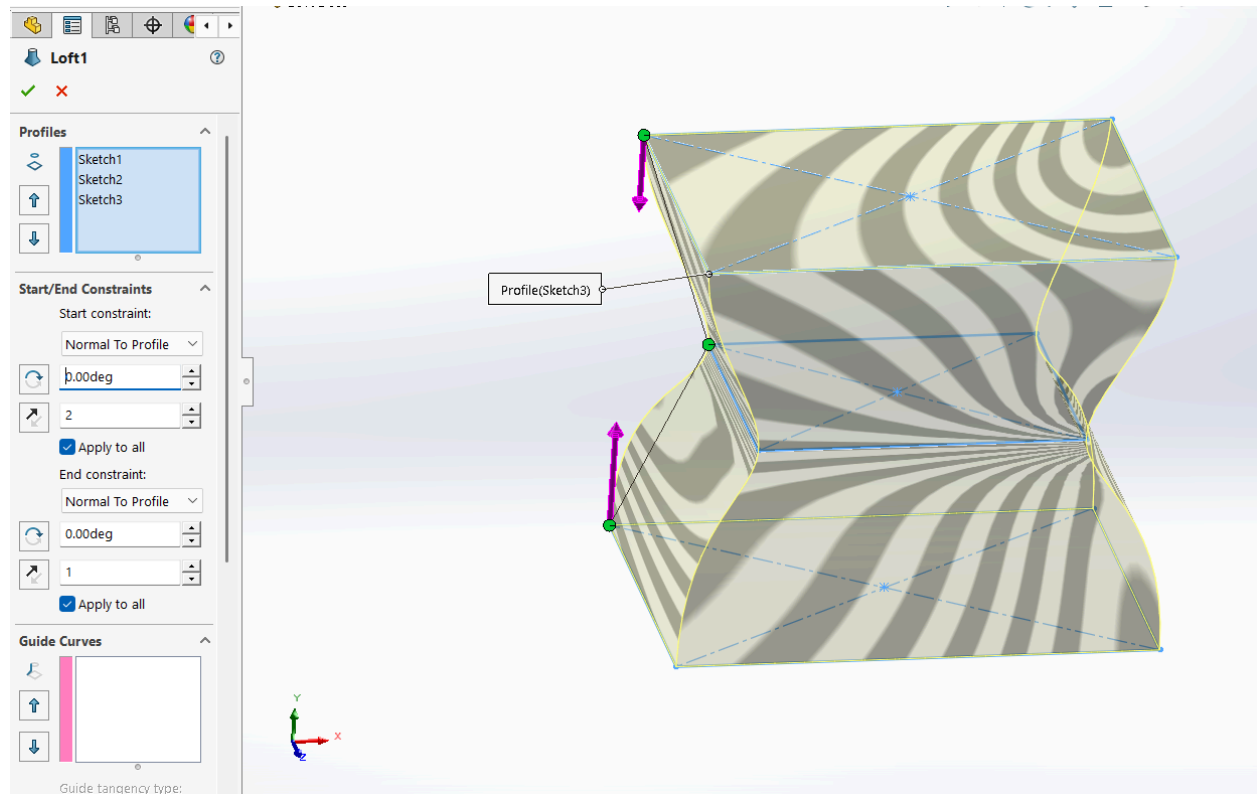
Now go back and edit the **Lofted Boss/Base** feature. Open **Start/End Constraints**. For **Start Constraint**, select **Normal To Profile** and set the value to **1**. For **End Constraint**, also select **Normal To Profile** and set the value to **1**. Now you can see that the loft starts and ends normal to the sketch profiles. If you increase the value above **1**, the normal influence becomes stronger, which changes the shape of the loft near the start and end profiles.



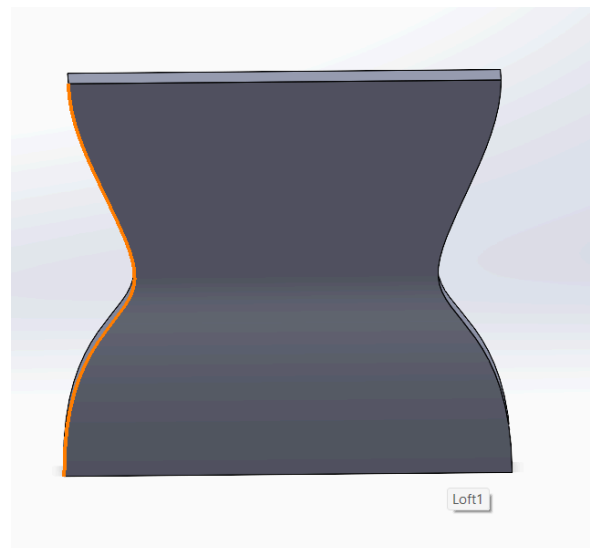
Click **OK**. This is the final result.



For example, change the **Start Constraint** value to **2**. Now you can see the difference between the start and end of the loft. The start side has a stronger normal influence, which changes the shape of the model near the first profile.



Click **OK**. This is the final result.



19. Fillet

What it does: Rounds selected model edges.

Use it for: Removing sharp edges, improving manufacturability, and creating smooth transitions.

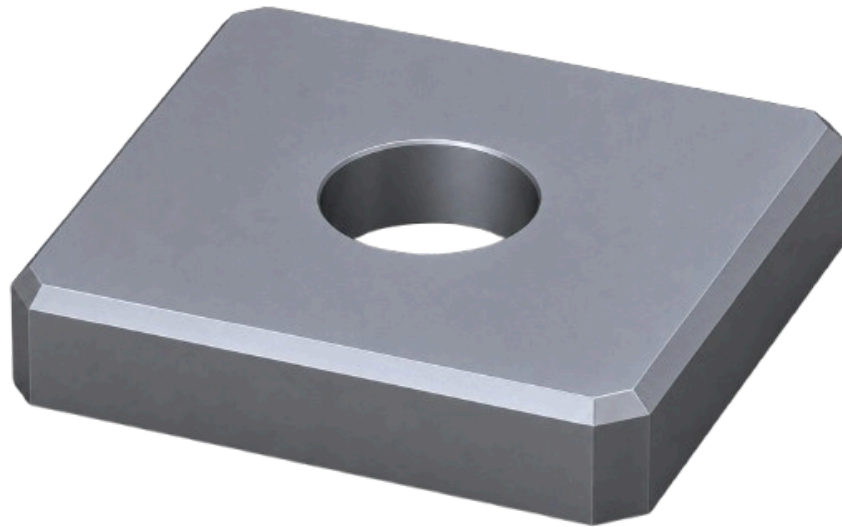
How it works:

These tools are used to modify sharp edges on a 3D model.

Fillet creates a rounded edge by applying a radius.

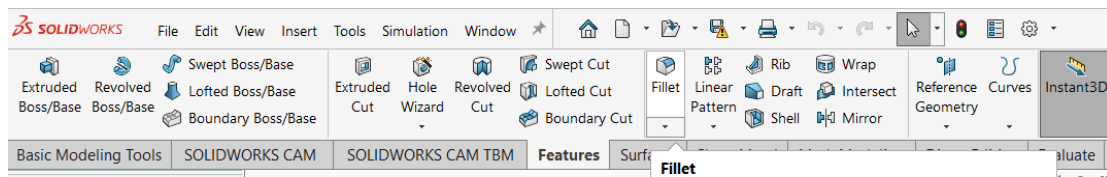


Chamfer creates a straight angled edge by applying a distance and angle, or two distance values.

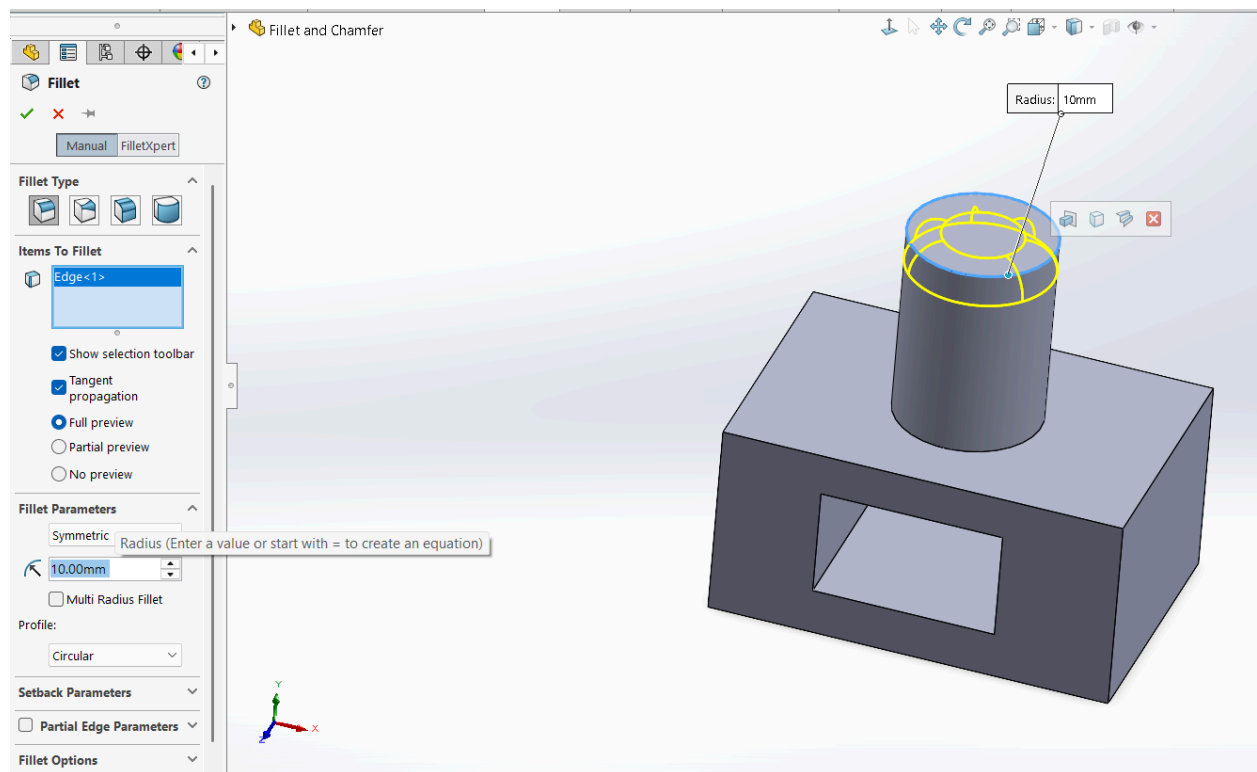


Both tools are useful for making parts look more realistic, removing sharp corners, and preparing models for manufacturing.

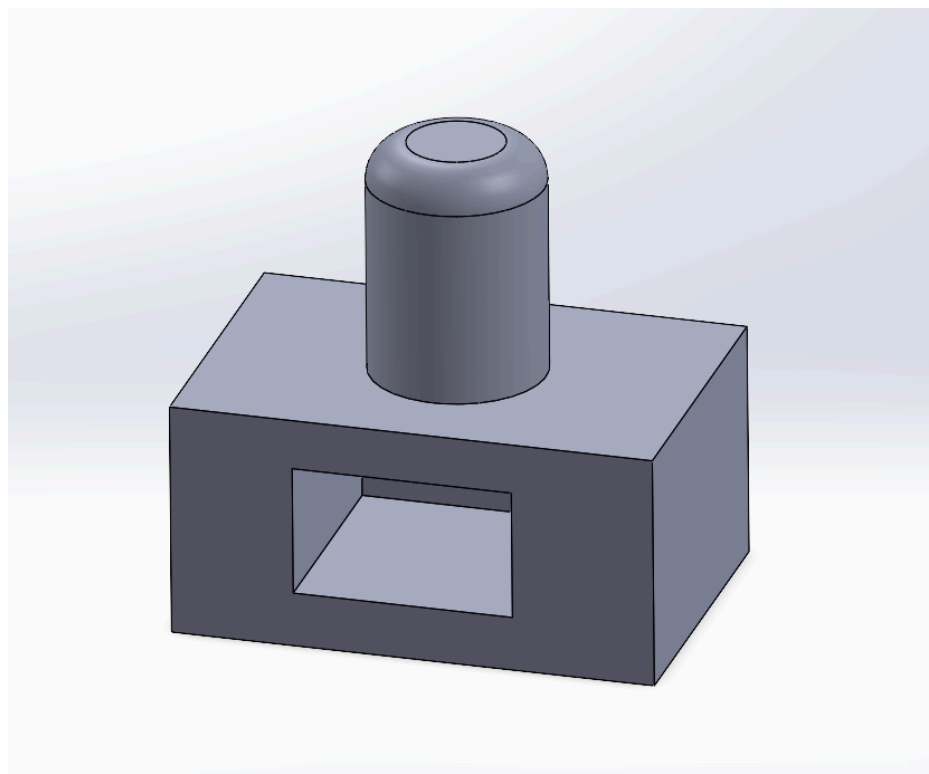
Let's start with **Fillet**. Go to the **Features** tab and select **Fillet**.



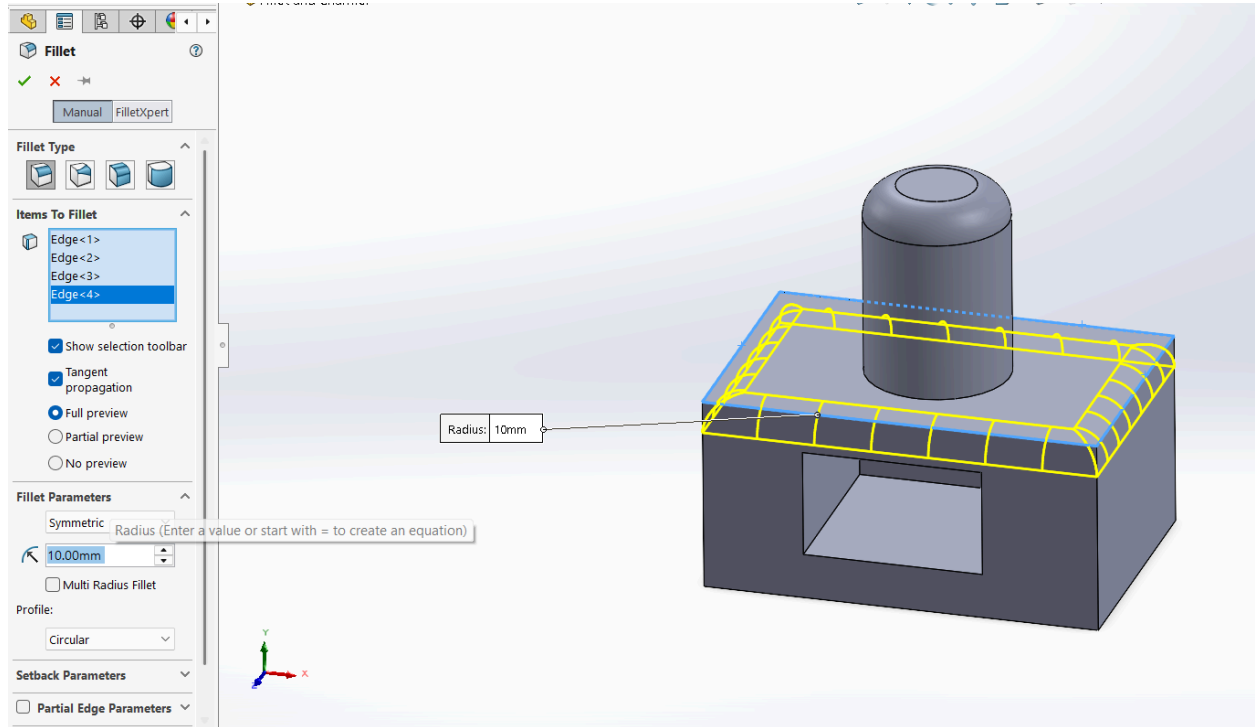
The fillet properties appear on the left side in the **PropertyManager**. Here, you can define the fillet radius. Leave the radius set to **10 mm**, then select the edge shown below. You can see the fillet preview before confirming the feature.



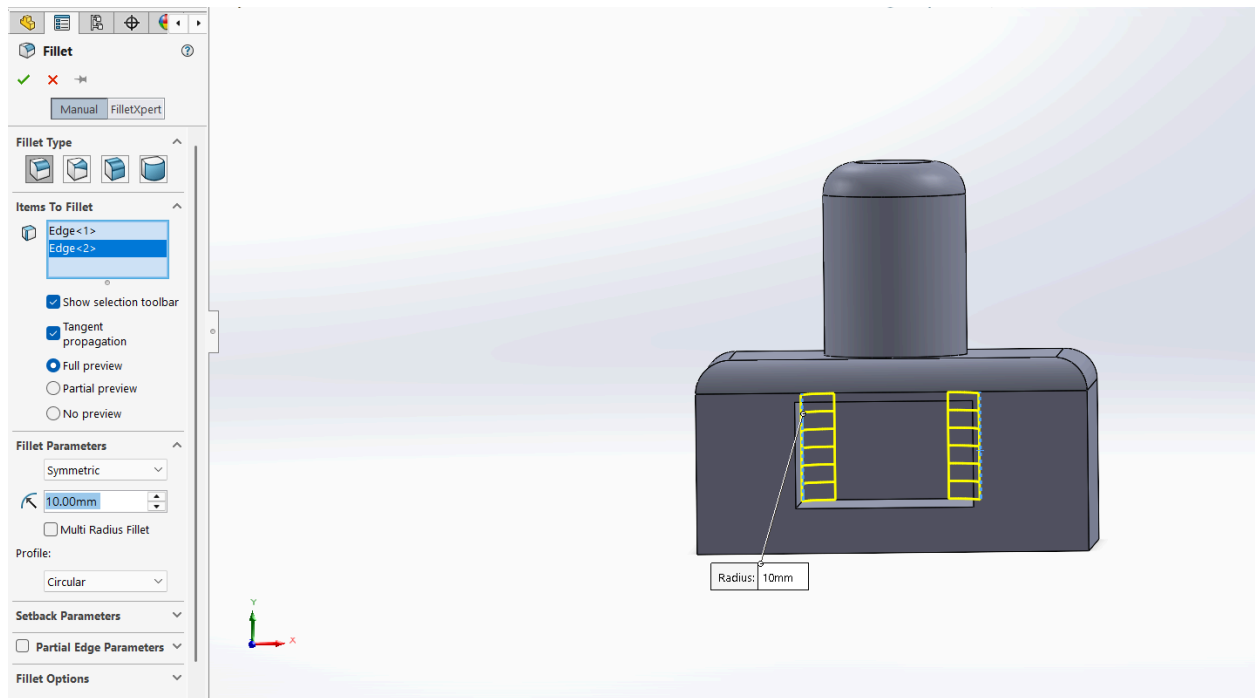
Here is the result with a rounded edge using a **10 mm** fillet radius.

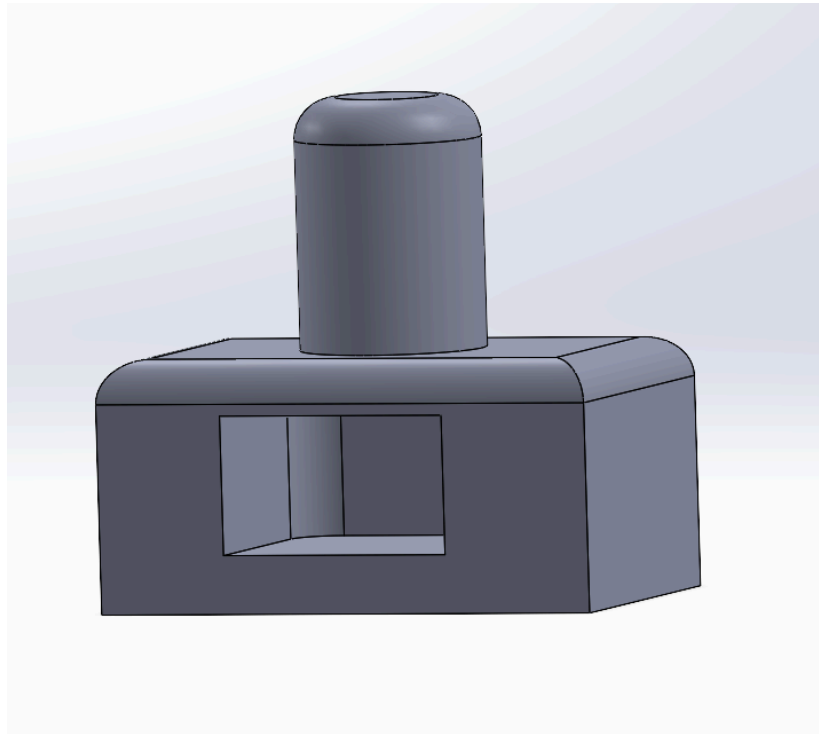


Select **Fillet** again, then select the edge shown below.



You can also apply a **Fillet** to the inner edges. Select **Fillet**, then select the inner edges shown below.





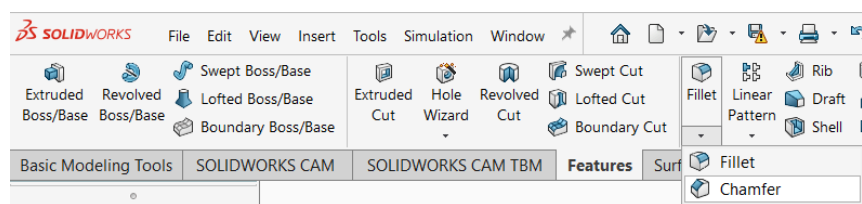
20. Chamfer

What it does: Creates an angled or beveled edge.

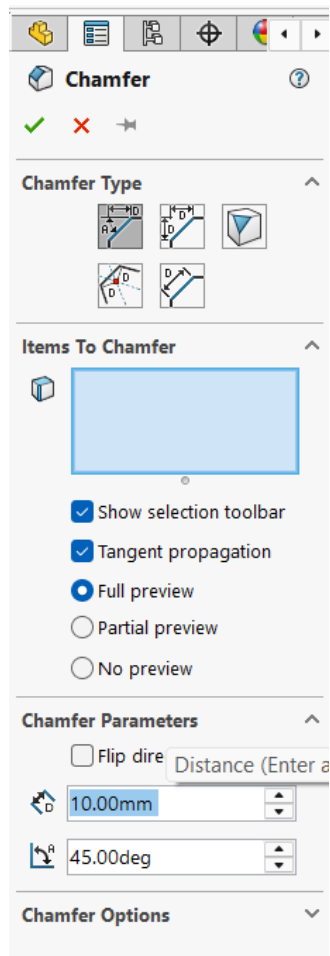
Use it for: Edge preparation, assembly clearance, and manufacturing details.

How it works:

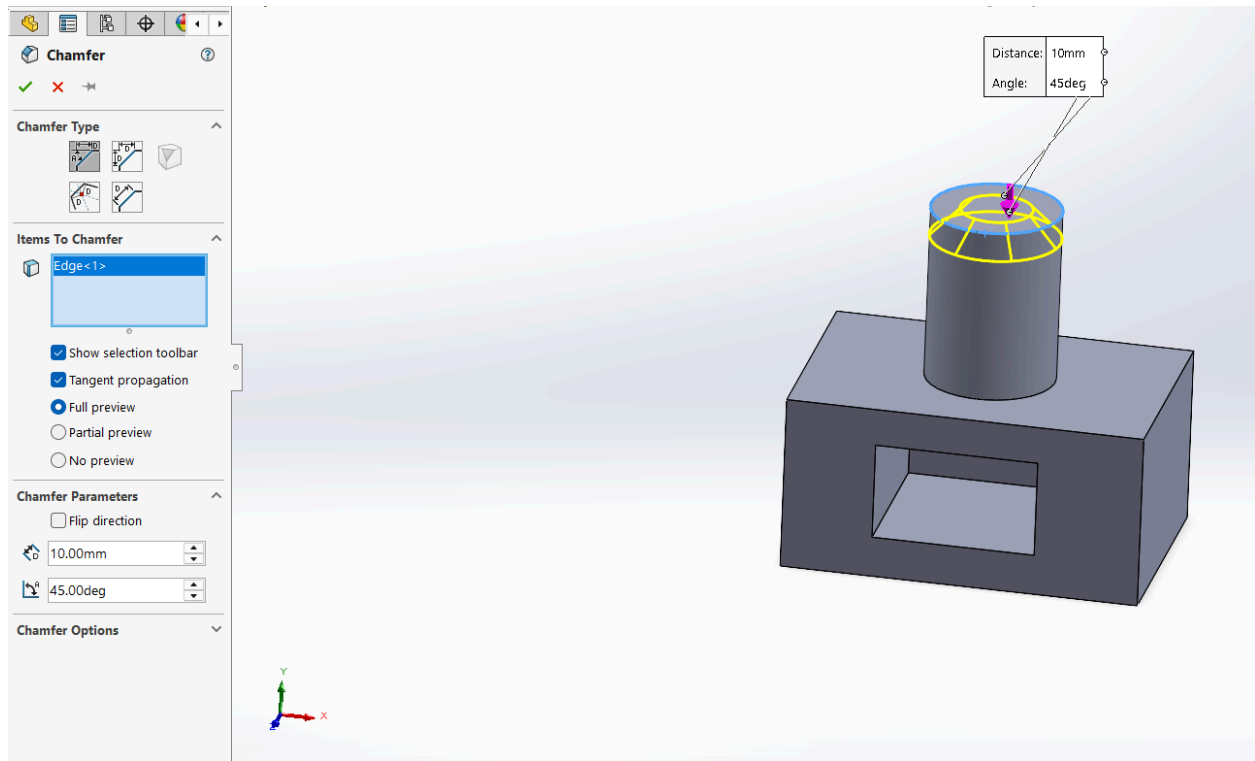
The second tool is **Chamfer**. To access **Chamfer**, click the small arrow below **Fillet**, then select **Chamfer**.



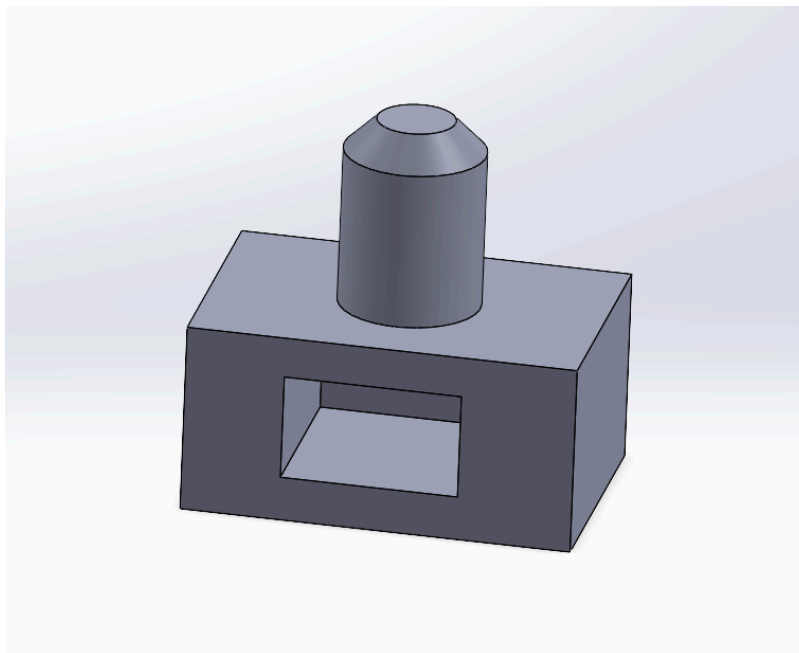
With **Chamfer**, we create a straight cut on the selected edge. To define the chamfer, you need to set the distance and angle. In this example, leave the distance set to **10 mm** and the angle set to **45°**.



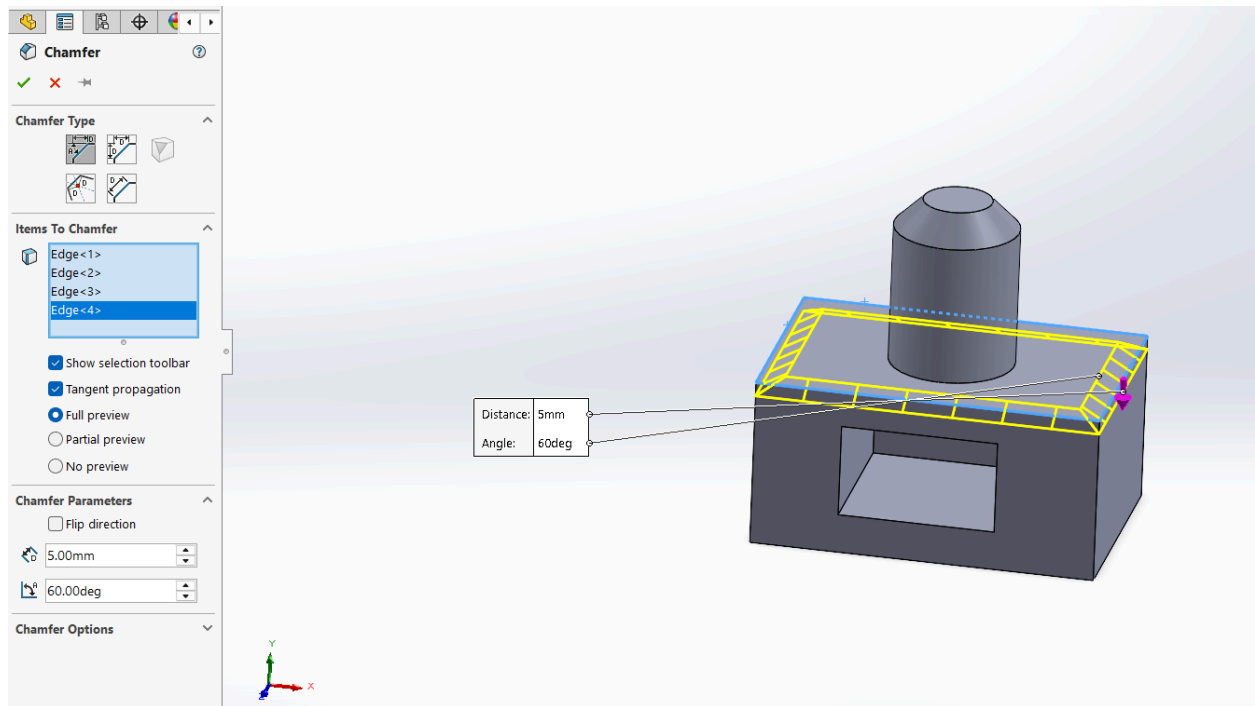
Select the edge shown below and check the chamfer preview.



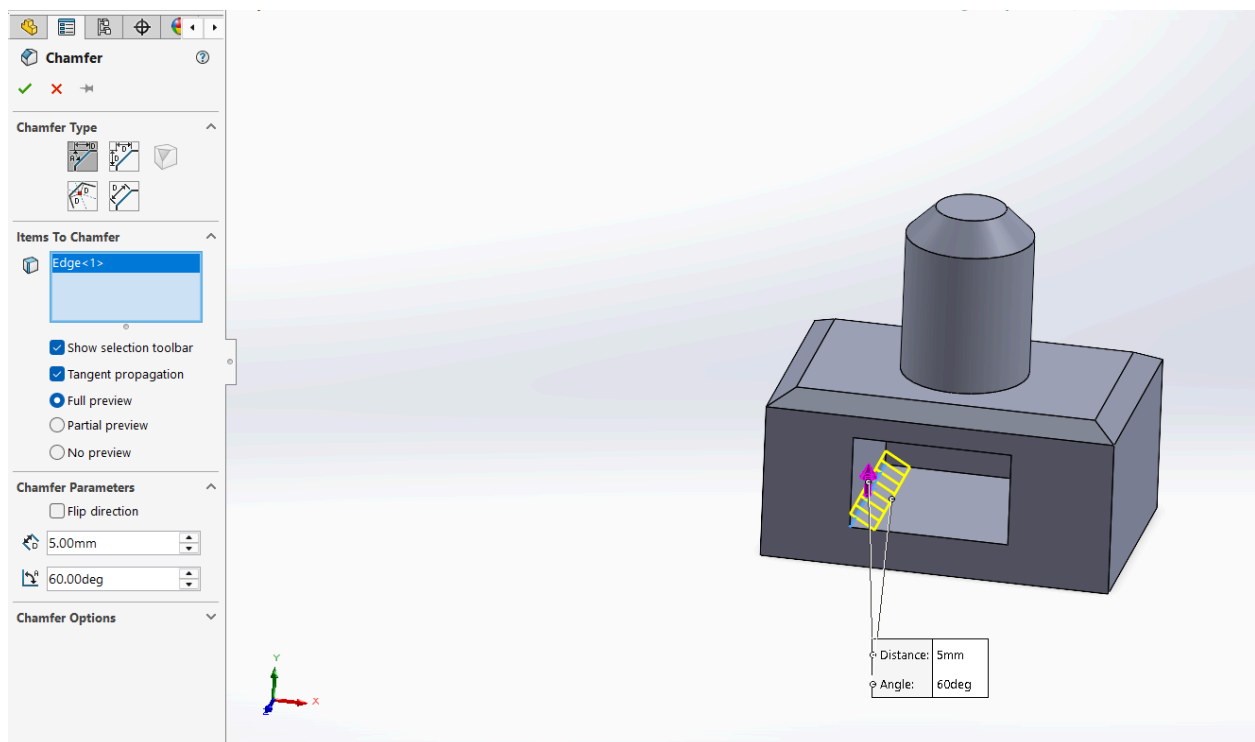
This is the final result.

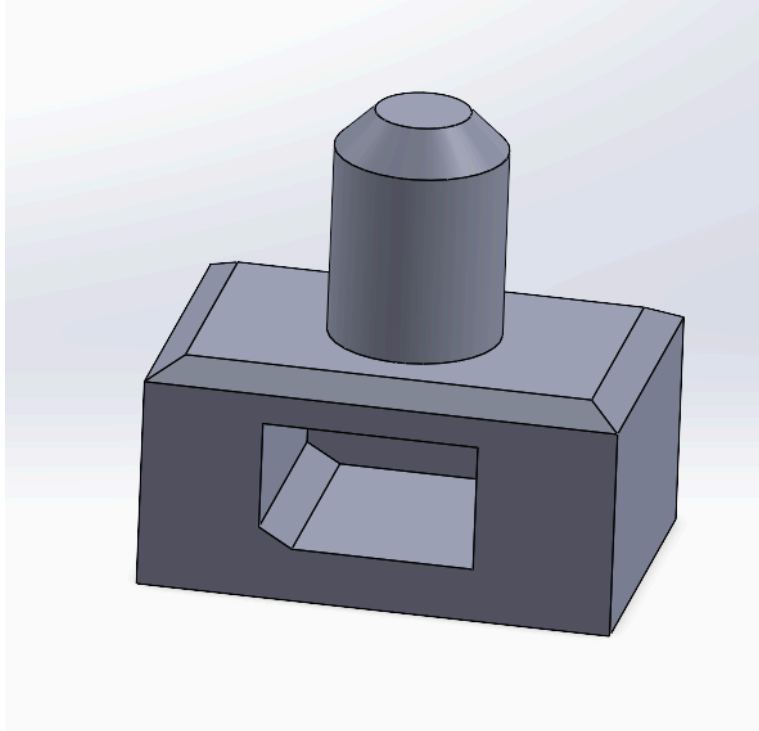


Select **Chamfer** again. Change the distance to **5 mm** and the angle to **60°**. Then select the edge shown below.



You can also apply a **Chamfer** to inner edges, as shown below.





21. Shell

What it does: Hollows out a solid model while maintaining a specified wall thickness.

Use it for: Housings, covers, containers, and molded parts.

How it works:

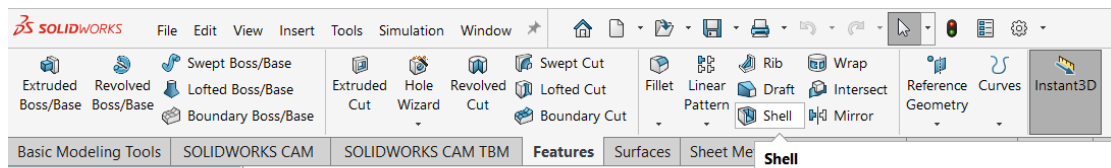
To create a hollow part, we use the **Shell** tool.

The **Shell** tool removes material from the inside of a solid body and leaves a defined wall thickness. This is useful when you want to create parts such as plastic housings, covers, containers, casings, boxes, or any model that needs to be hollow instead of completely solid.

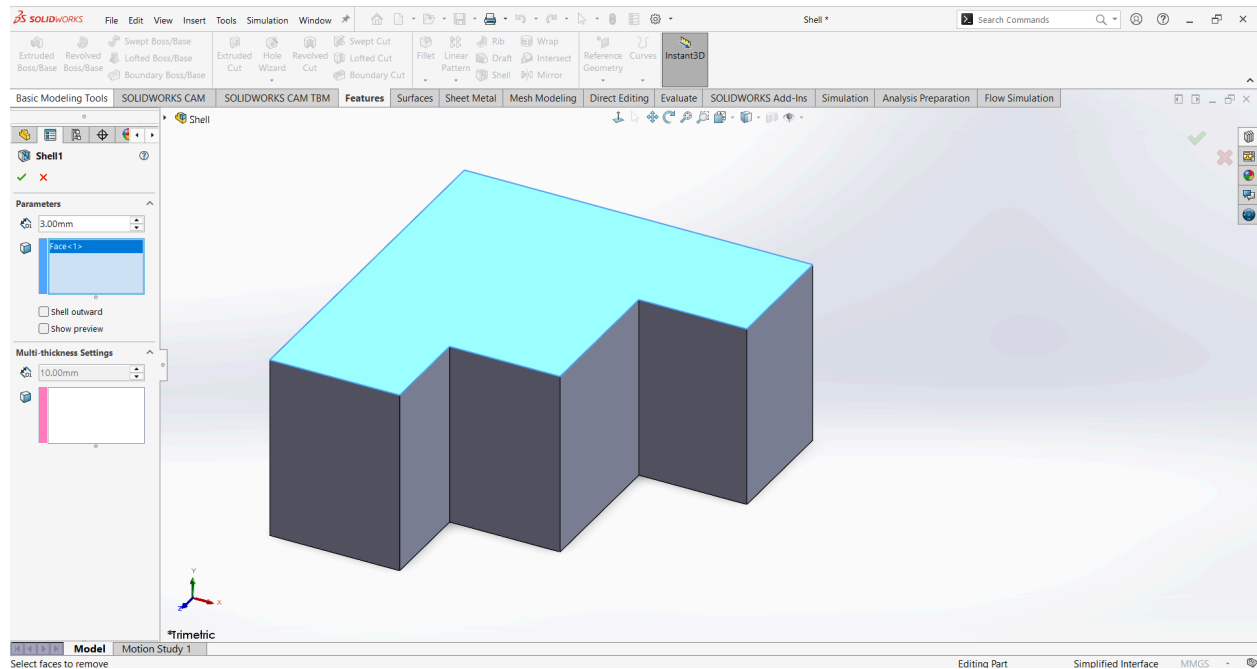


With **Shell**, you can select a face to remove, and SOLIDWORKS will hollow out the model from that opening. You can also define the wall thickness to control how thick the remaining material should be.

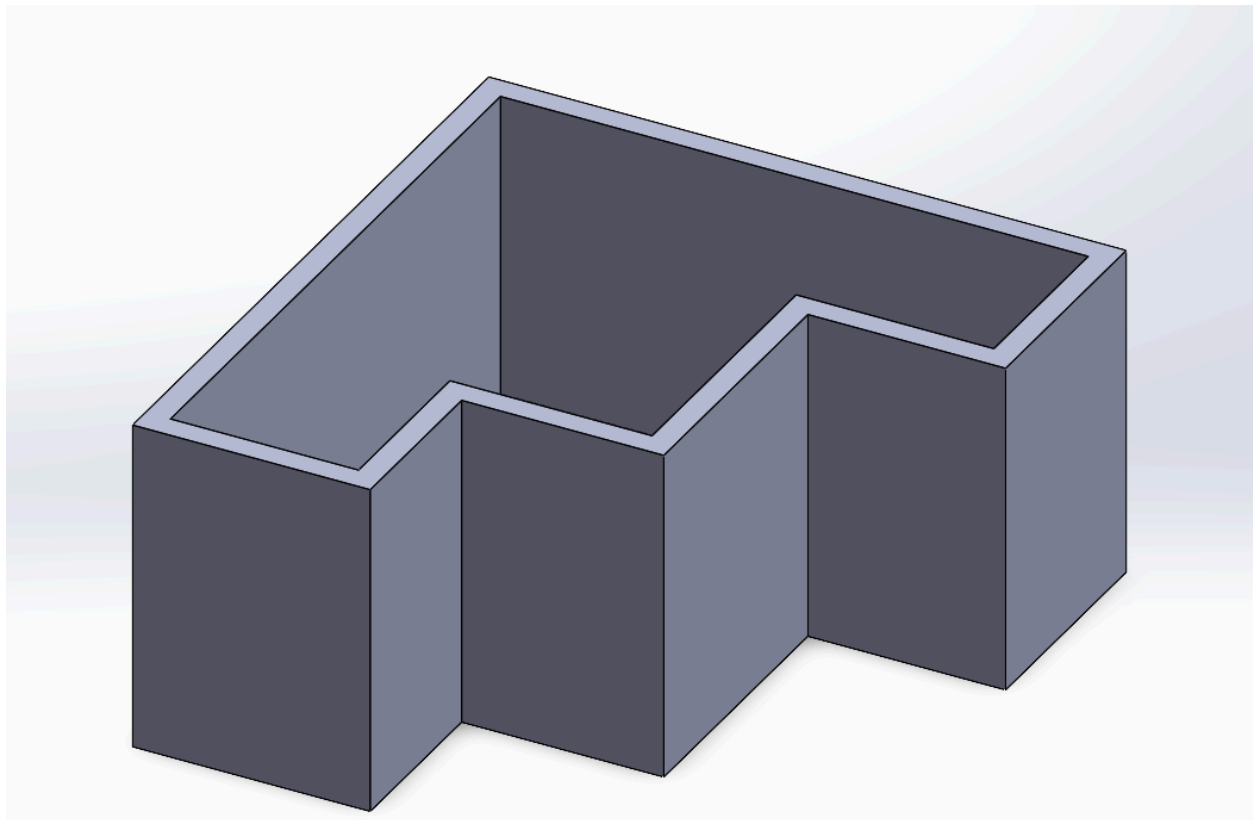
Go to the **Features** tab and select **Shell**.



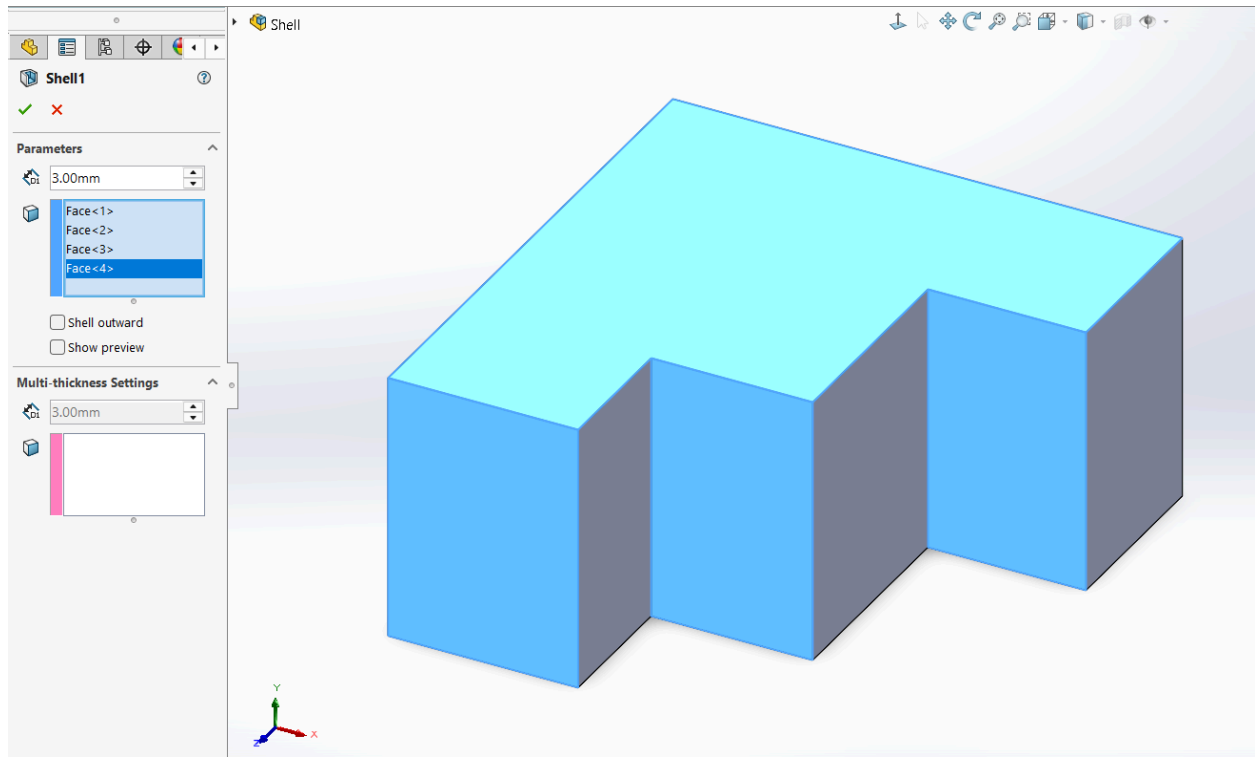
The shell properties appear on the left side in the **PropertyManager**. First, define the wall thickness. Then select the face you want to remove. Click the selection box in the **PropertyManager**, then select the upper face of the model.



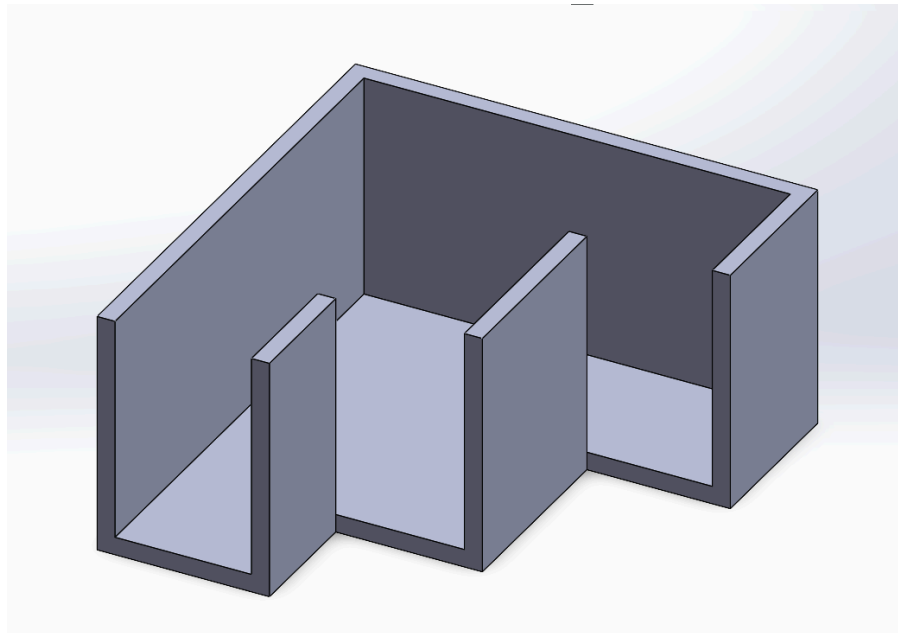
This is the final result.



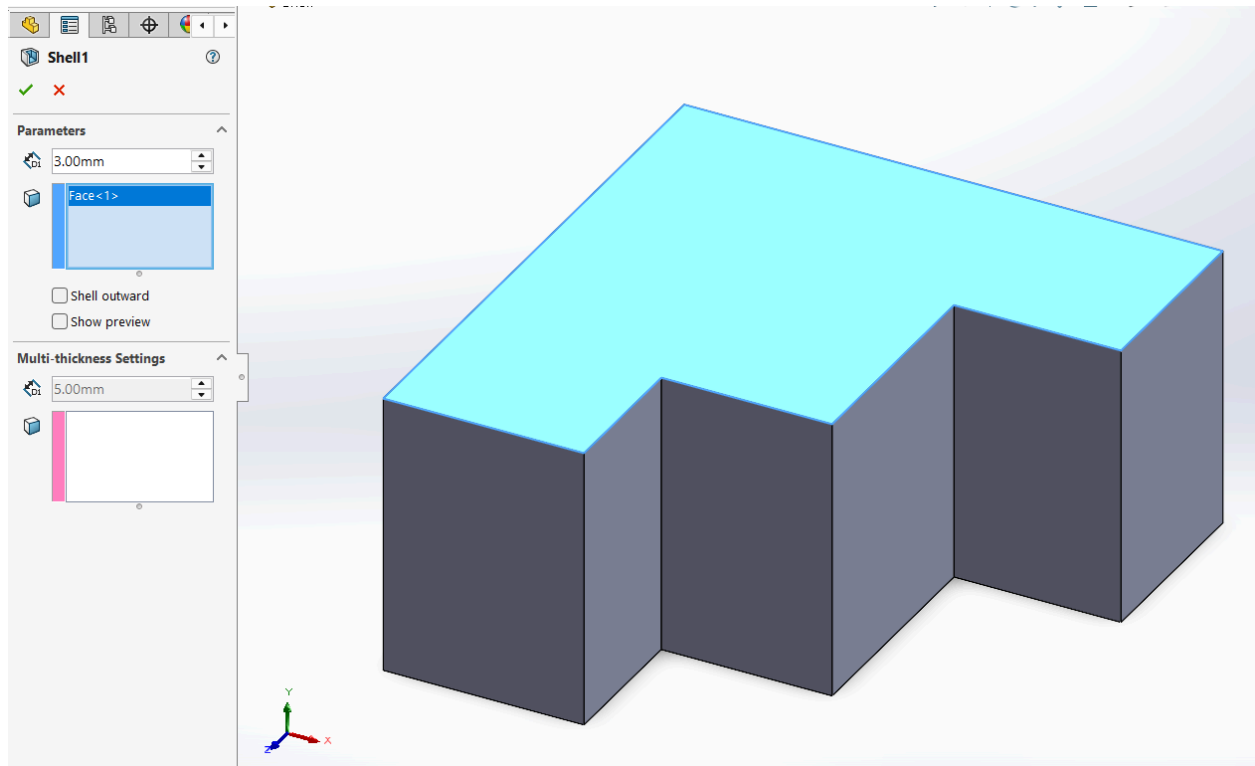
You can also remove multiple faces using the **Shell** tool. Click the selection box in the **PropertyManager**, then select multiple faces on the model.



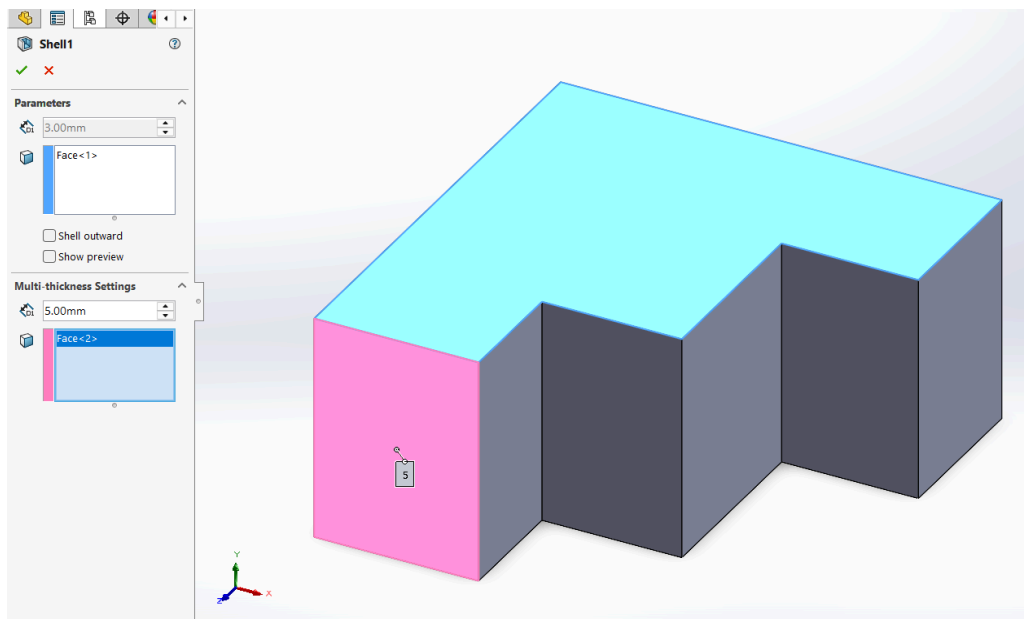
This is the result.



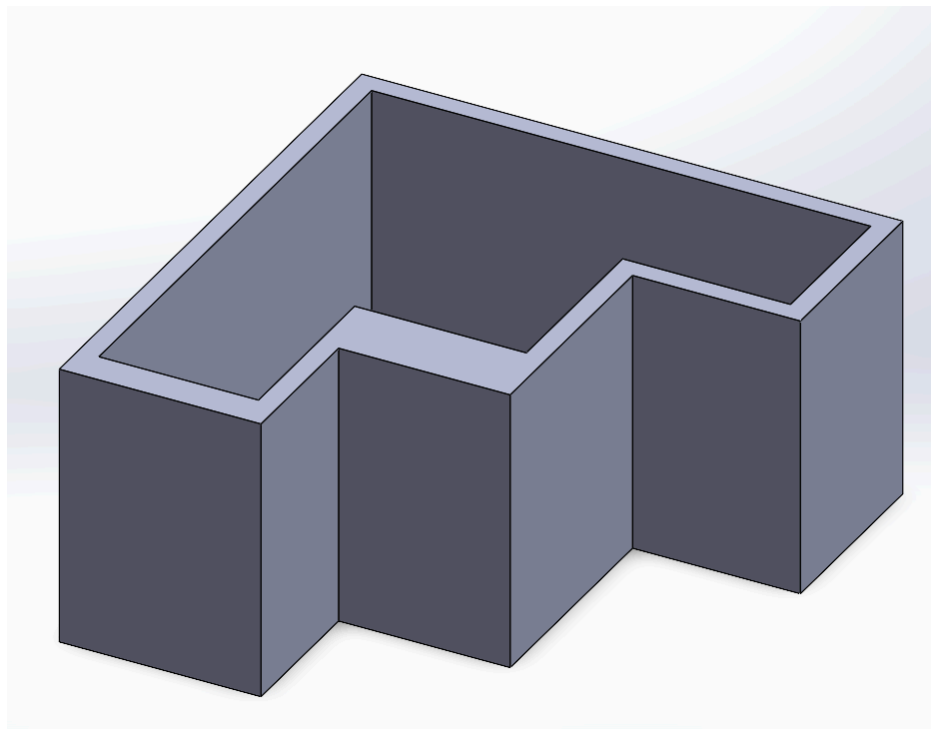
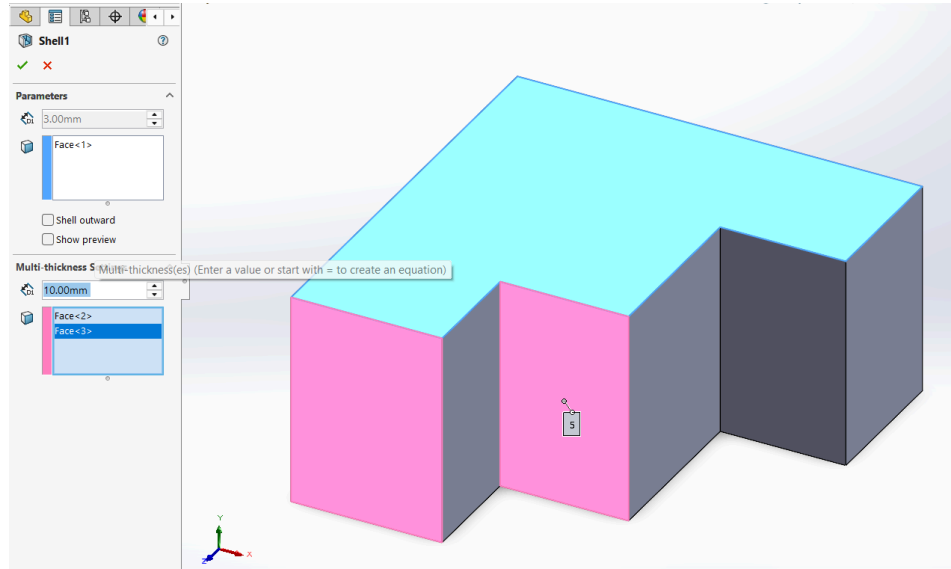
You can also apply different wall thicknesses to different faces. First, define the main shell thickness and select the face you want to remove.



Then, in the second selection box, select the face where you want to apply a different thickness. Above that selection box, define the thickness value for that face. In this example, the pink face has a wall thickness of **5 mm**.



Then select the second face and set its thickness to **10 mm**. Click **OK**.



Now you can see that we have different wall thicknesses on these two faces. One face has a thickness of **5 mm**, and the other face has a thickness of **10 mm**. The rest of the shell uses the main thickness of **3 mm**.

22. Rib

What it does: Creates thin reinforcing geometry from sketch entities.

Use it for: Structural reinforcement and strengthening plastic or cast components.

How it works:

The **Rib** tool is used to add thin support features to a 3D model.

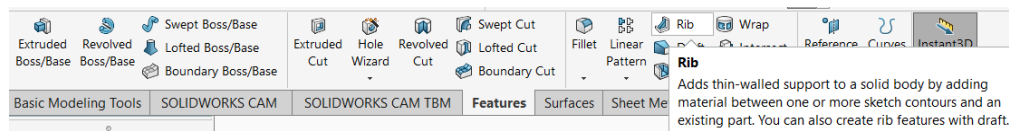


Ribs are commonly used to strengthen parts without adding too much material.

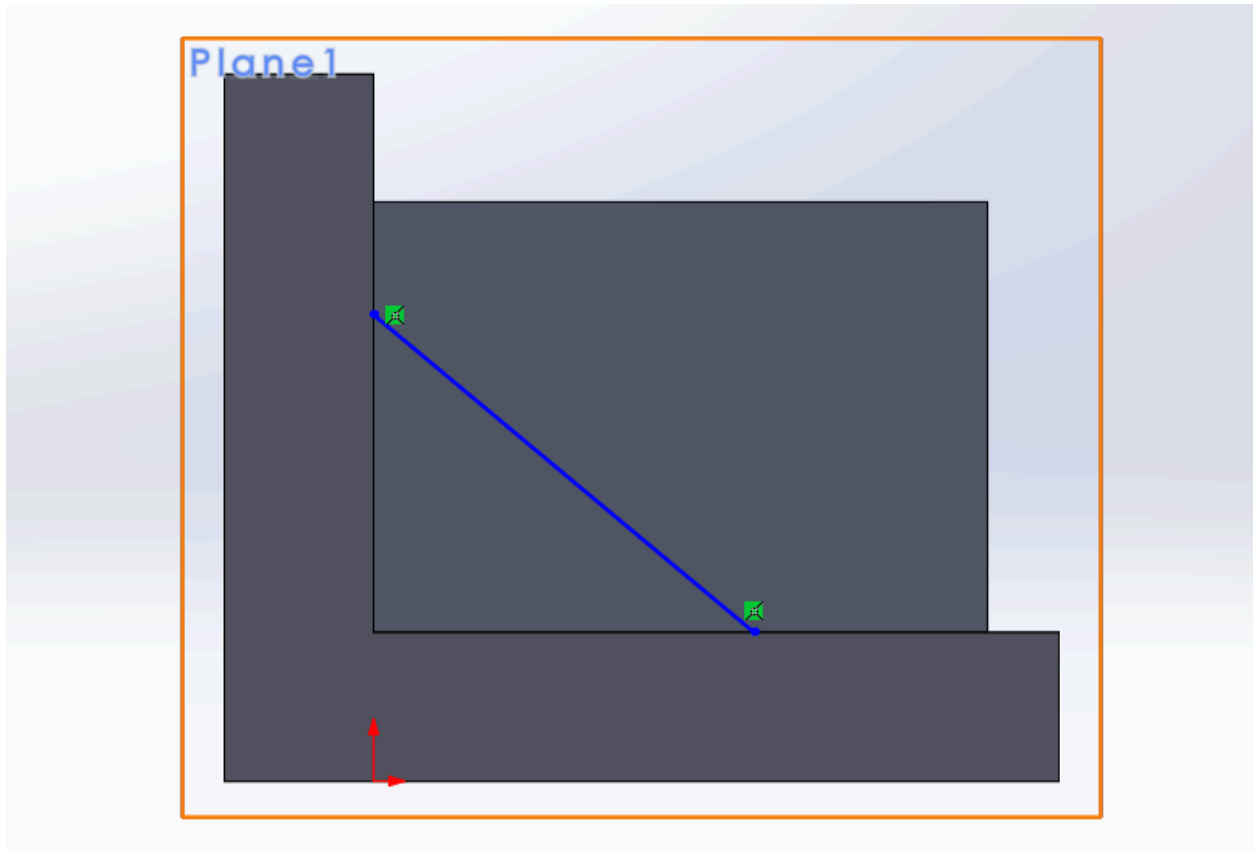
Instead of creating a full solid feature manually, you can create a simple sketch line, and SOLIDWORKS will turn it into a rib with a defined thickness and direction.

This tool is useful for plastic parts, brackets, housings, supports, and other models that need extra strength.

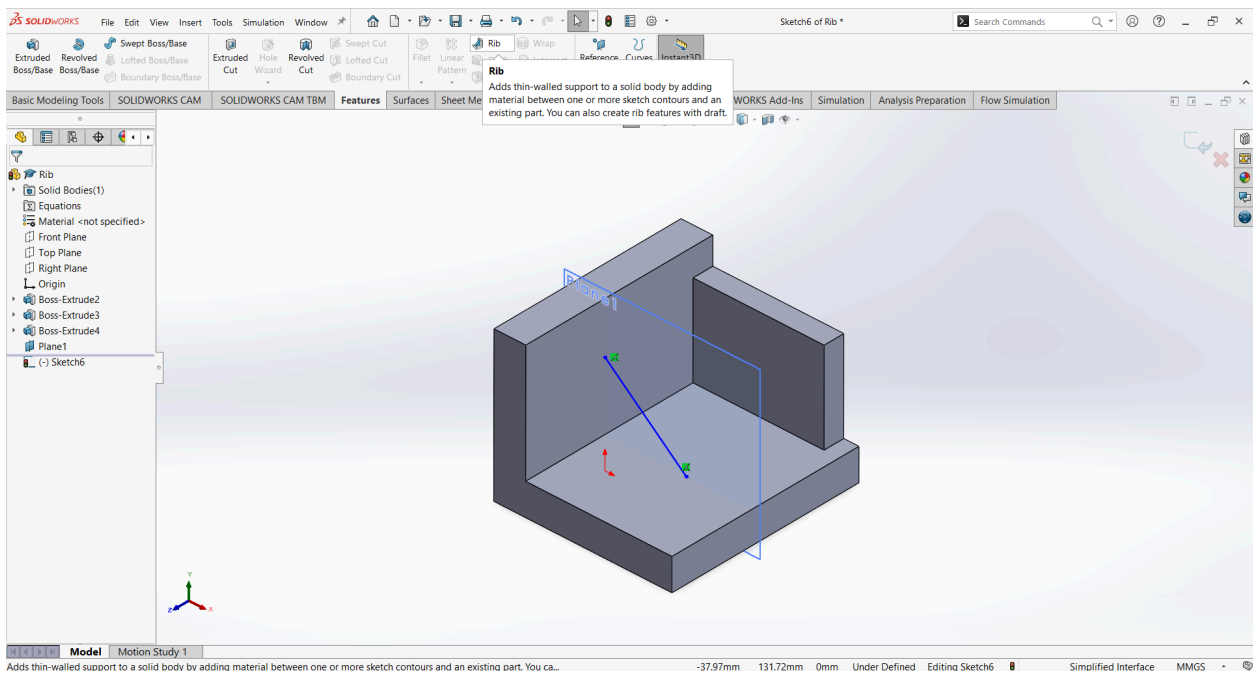
Go to the **Features** tab and select **Rib**.



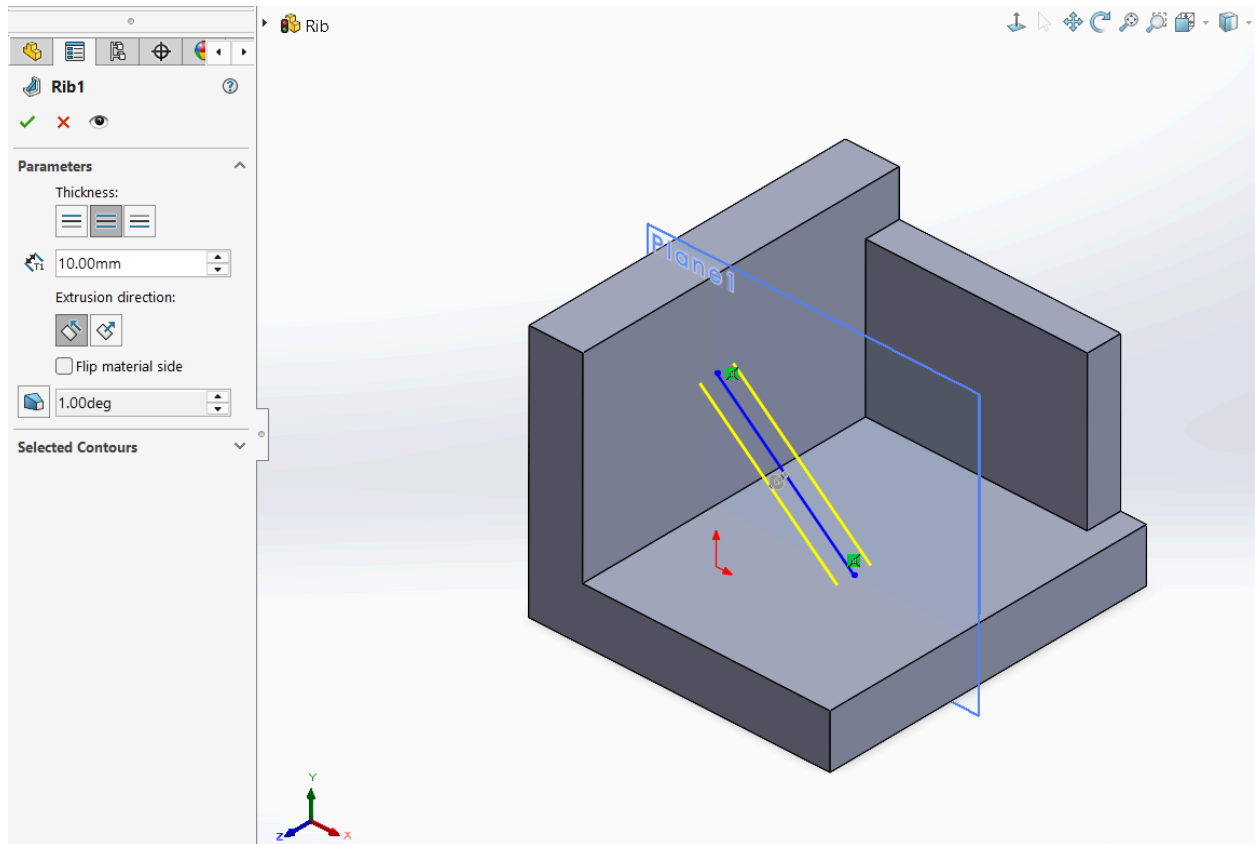
First, we need to create a line on the plane. This line will be used to create the rib. One line is enough. Create a line that connects the two perpendicular edges, as shown below.



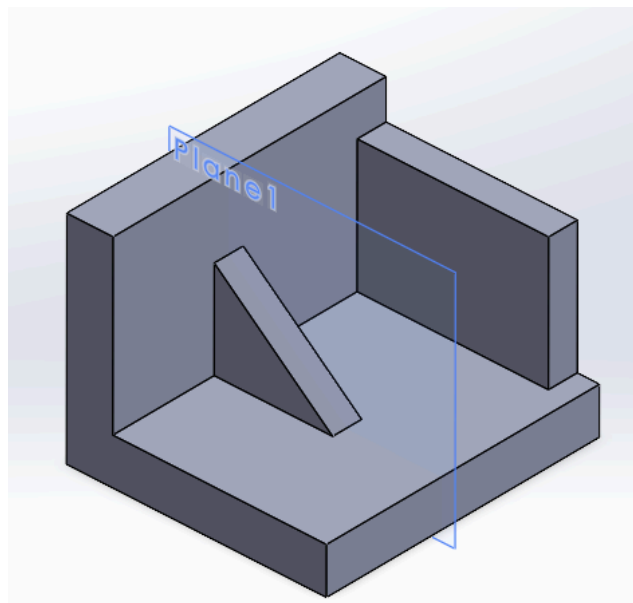
Now select the **Rib** tool.



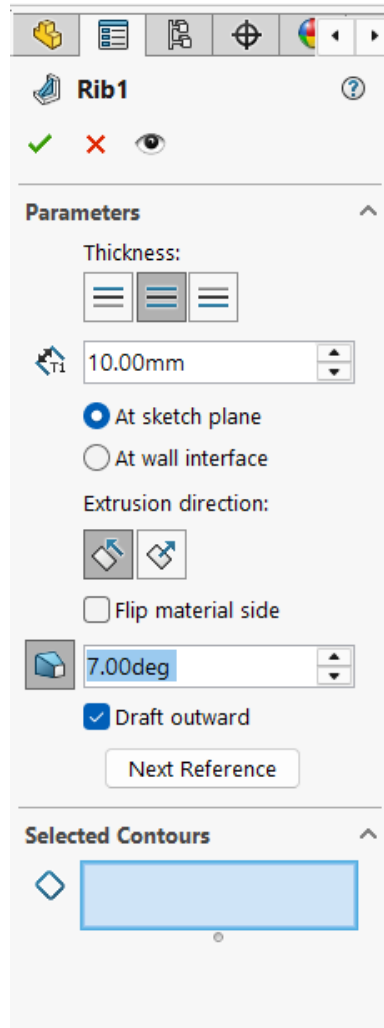
The rib properties appear on the left side in the **PropertyManager**. The thickness is set to **10 mm** by default. Click **OK**.



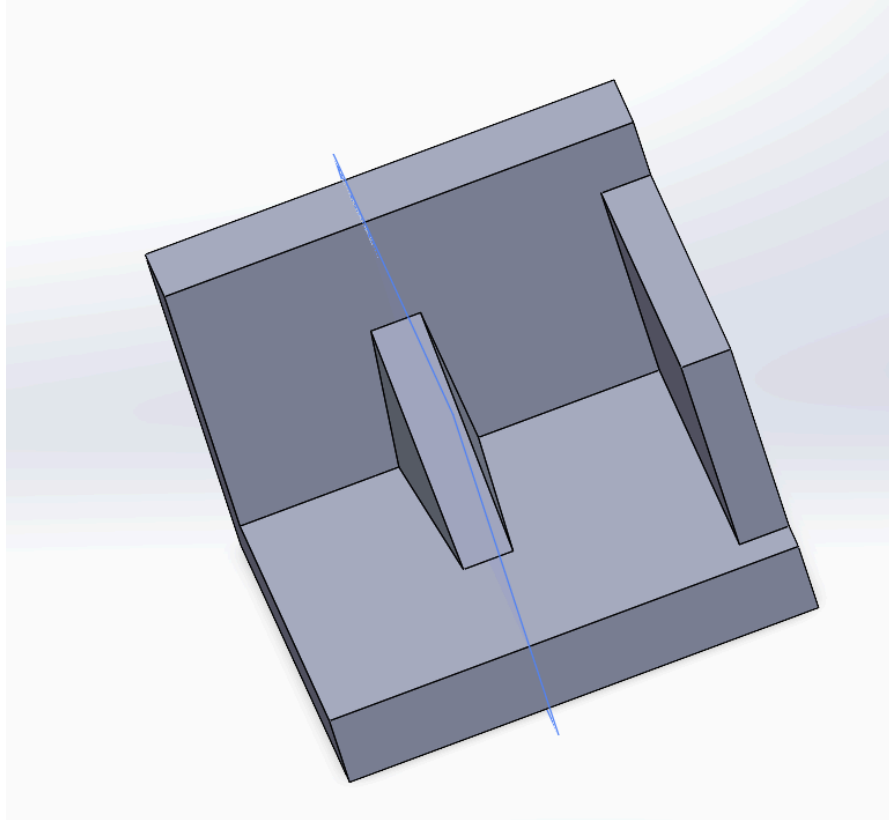
This is final result.



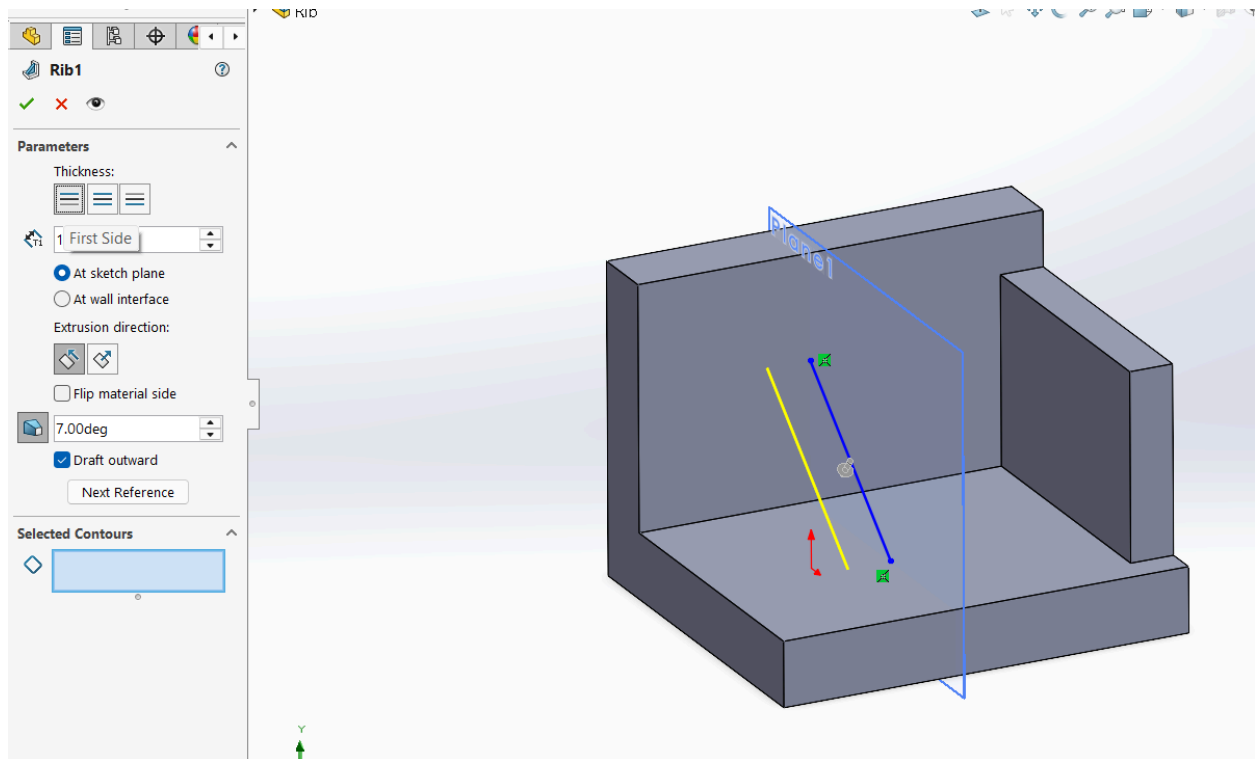
Now go back to the **PropertyManager**. Turn on **Draft** and set the draft angle to **7°**.



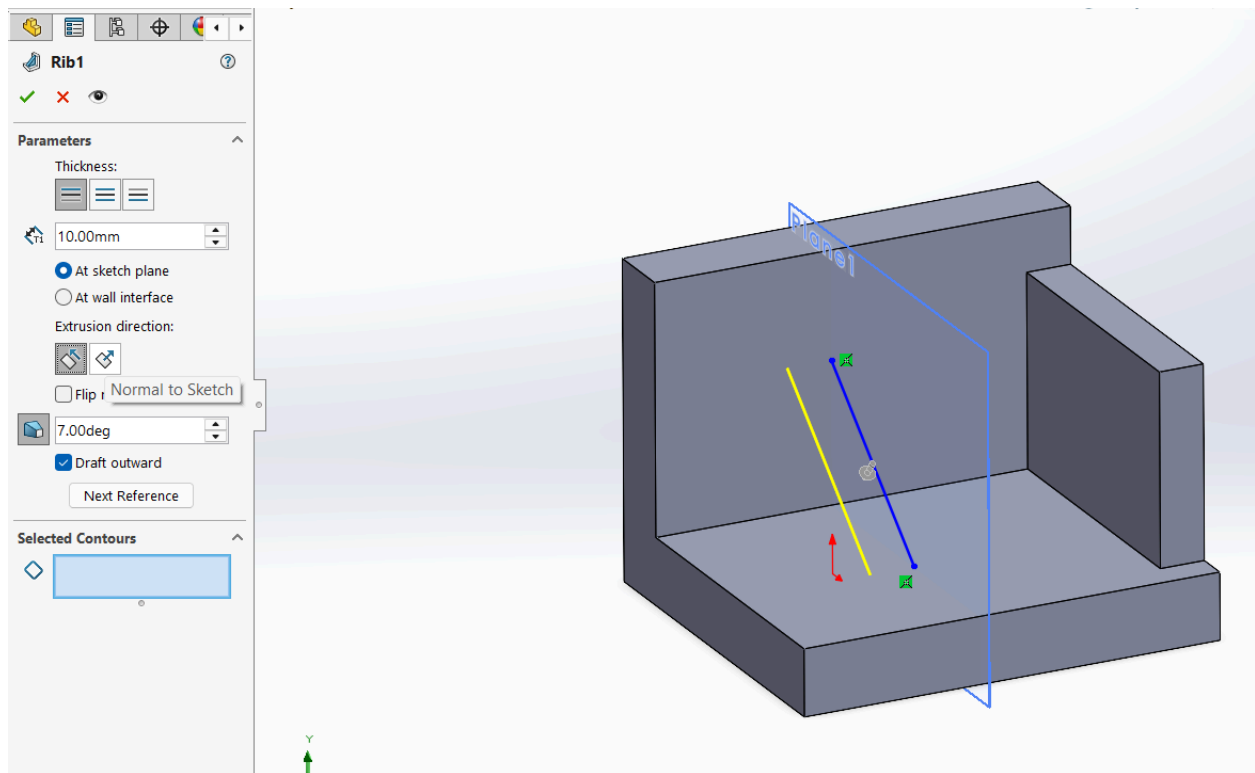
Now we have a rib with a draft angle of **7°**.



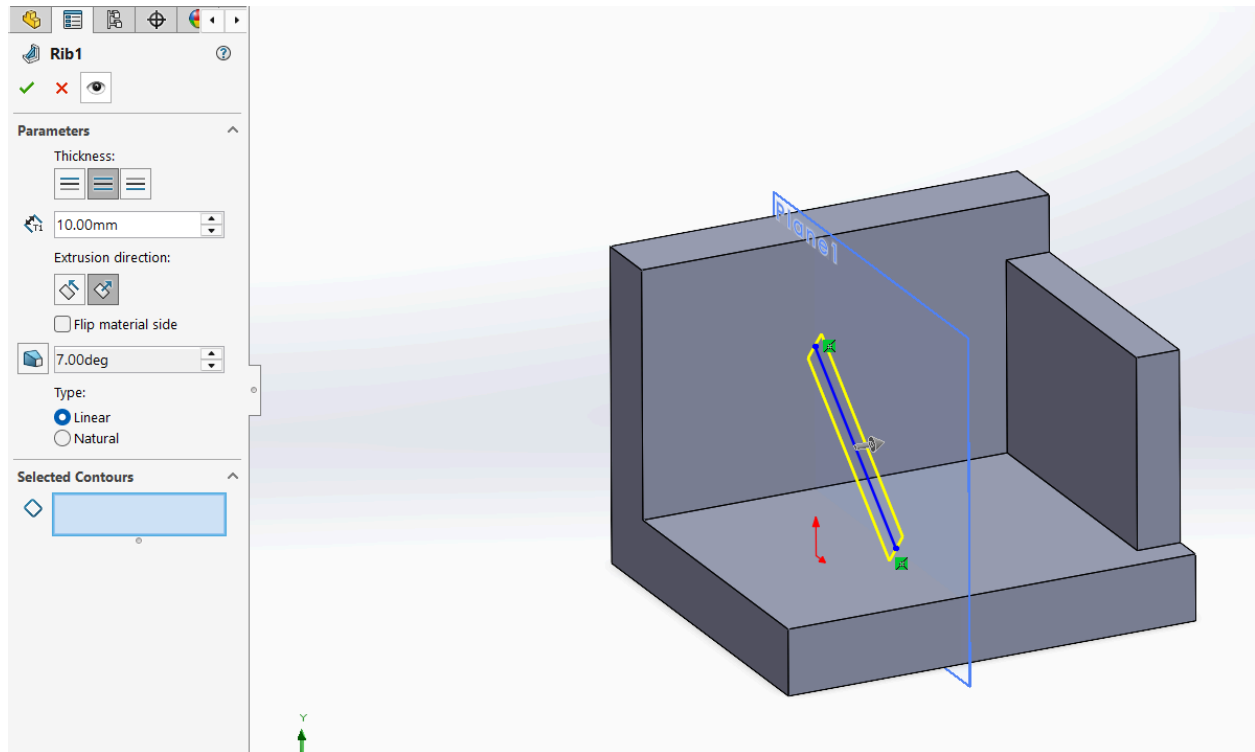
You can also choose the rib direction. The rib can be created on one side, on the opposite side, or from the middle of the sketch line.



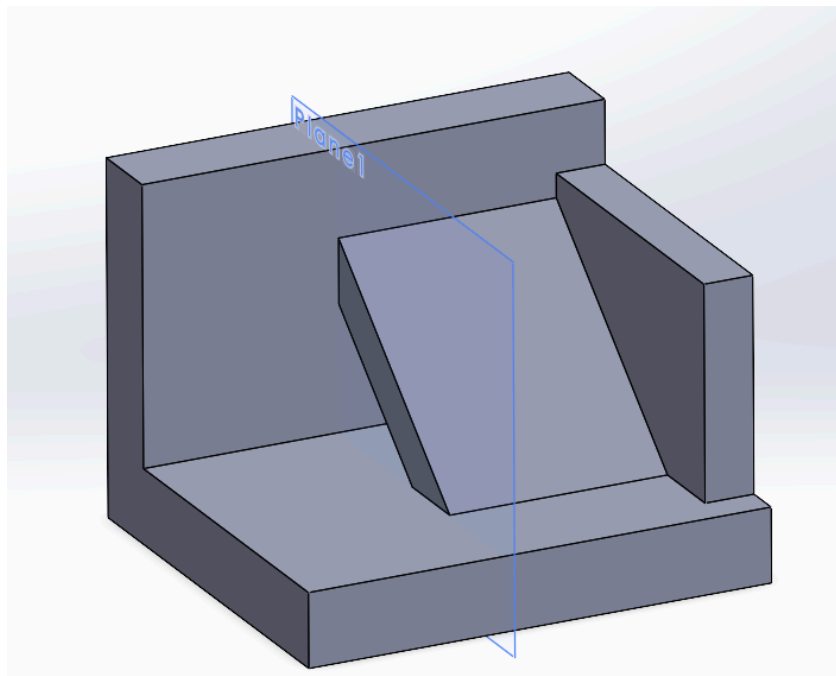
You can also use the **Normal to Sketch** option.



Select the **Middle** option, then click **OK**.



This is the final result.



23. Hole Wizard

What it does: Creates standardized engineering holes.

Use it for: Counterbores, countersinks, tapped holes, clearance holes, and other standard hole types.

How it works:

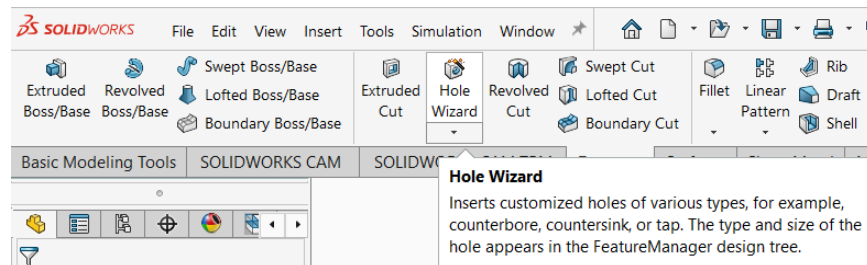
Hole Wizard is used to create standard holes quickly and accurately. Instead of sketching and cutting holes manually, you can choose from predefined hole types, sizes, standards, and end conditions.

This tool is commonly used for creating simple holes, counterbore holes, countersink holes, tapped holes, and other standard mechanical hole features.

With **Hole Wizard**, you can also define the exact position of the hole on a face using sketch points and dimensions.

The goal of this section is to understand how to create professional and standardized holes in SOLIDWORKS more efficiently.

Go to the **Features** tab. In the **Features** toolbar, find and select the **Hole Wizard** tool. After selecting **Hole Wizard**, the **PropertyManager** will open on the left side. Here, you can define the type of hole you want to create, including the hole standard, hole type, size, end condition, and position. This tool allows you to create accurate standard holes without manually sketching and cutting each hole.



In the **Hole Wizard PropertyManager**, you can define the main settings for the hole.

Here, you can choose the **hole type**, **standard**, **hole size**, **end condition**, and **tolerance**.

The **hole type** controls what kind of hole you want to create, such as a simple hole, counterbore, countersink, or tapped hole.

The **standard** allows you to select a predefined standard, such as **ANSI Metric**, **ISO**, or another available standard.

The **hole size** controls the diameter or thread size of the hole.

The **end condition** defines how deep the hole will go into the part.

The **tolerance** section allows you to add manufacturing tolerance information if needed.

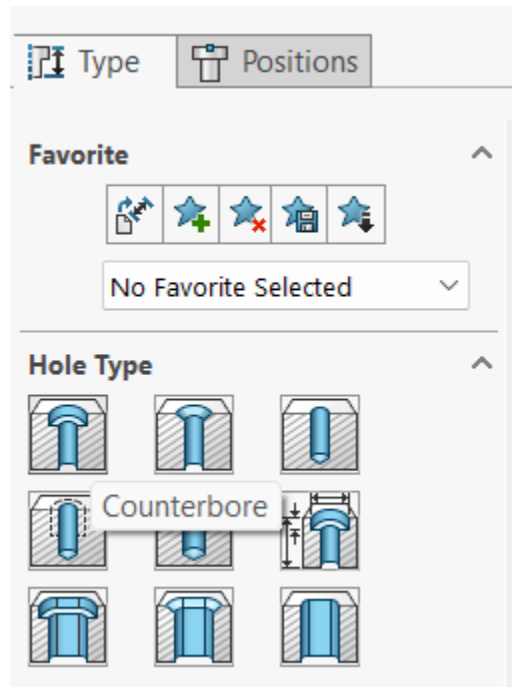
These settings help you create accurate and standardized holes directly inside SOLIDWORKS.

The screenshot shows the 'Hole Specification' dialog box in SOLIDWORKS. The dialog is divided into several sections:

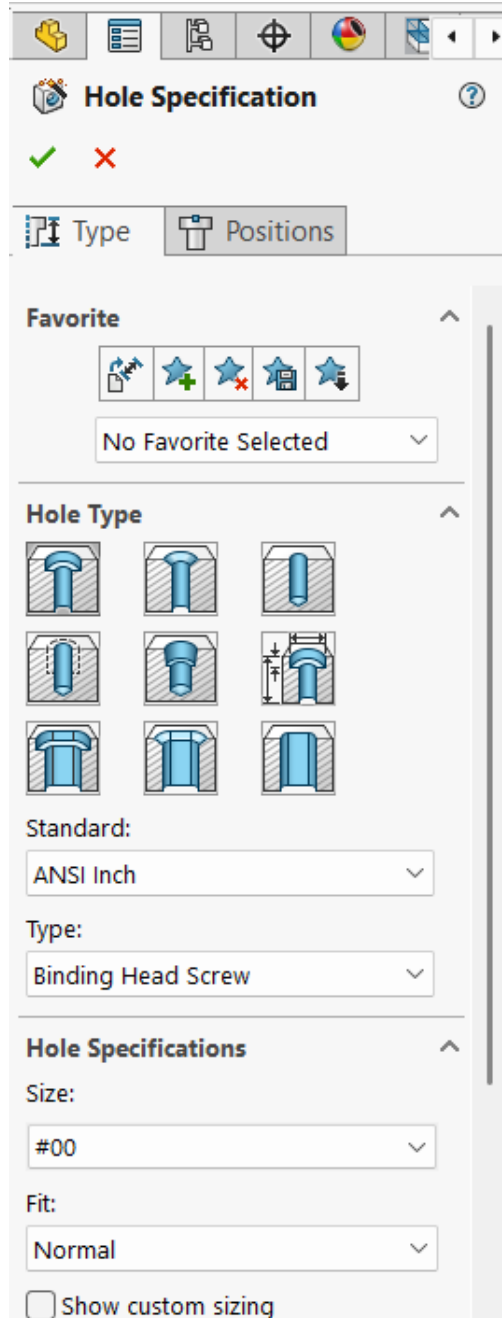
- Favorite:** Includes icons for adding, removing, and favoriting holes, and a dropdown menu showing 'No Favorite Selected'.
- Hole Type:** A grid of icons representing different hole types (e.g., standard hole, counterbore, countersink). Below the grid is a 'Standard:' dropdown set to 'ANSI Metric' and a 'Type:' dropdown set to 'Hex Bolt - ANSI B18.2.3.5M'.
- Hole Specifications:** Includes a 'Size:' dropdown set to 'M5', a 'Fit:' dropdown set to 'Normal', and a checkbox for 'Show custom sizing' which is currently unchecked.
- End Condition:** Includes a dropdown set to 'Blind', a depth input field set to '7.00mm', and two icons for selecting the end condition (a hole icon and a part icon).
- Options:** Includes three checkboxes: 'Head clearance' (unchecked), 'Near side countersink' (unchecked), and 'Under head countersink' (unchecked).
- Tolerance/Precision:** Includes a 'Callout value' dropdown set to 'Diameter (5.5)', a tolerance dropdown set to 'None' (with a small table showing tolerance values: +.01, 1.50, -.01), and a precision dropdown set to '.12 (Document)' (with a small table showing precision values: .01, .005, .01).

For this example, select **Counterbore** as the hole type. A **Counterbore** hole creates a straight hole with a larger cylindrical recess at the top. This type of hole is commonly used when you want the head of a bolt or screw to sit below the surface of the part. After selecting

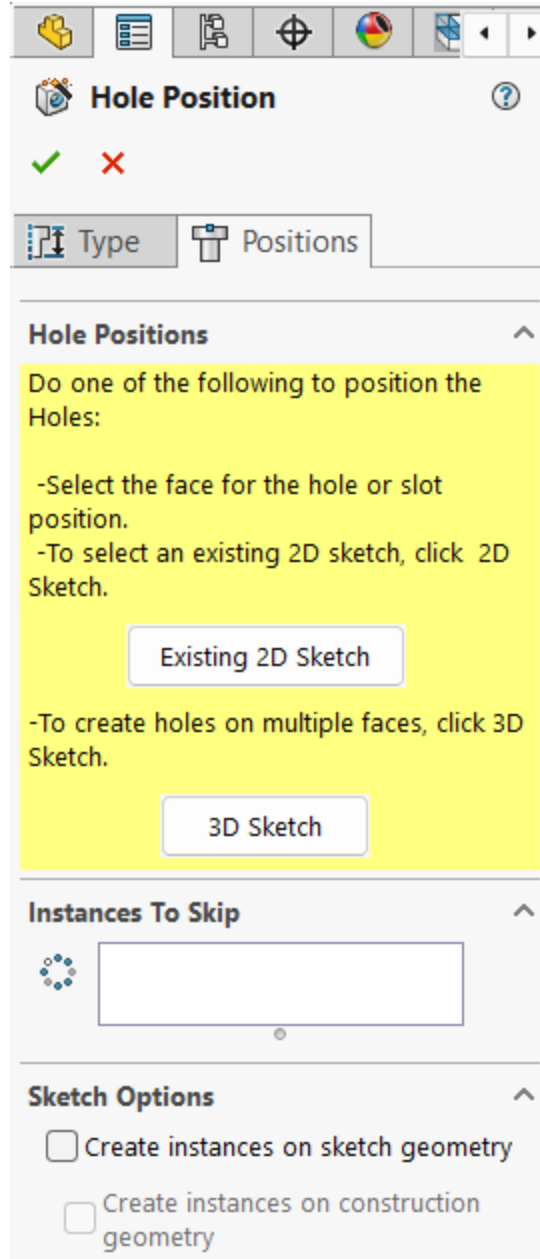
Counterbore, the available settings in the **PropertyManager** will update based on this hole type.



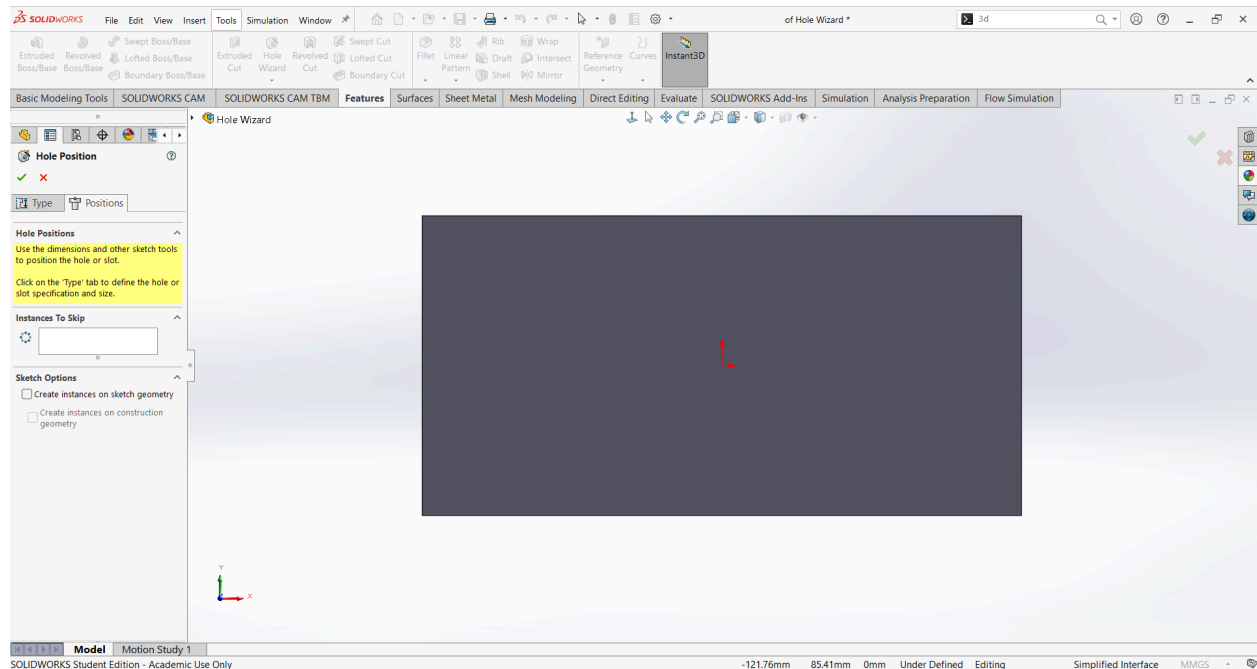
Set the **Standard** to **ANSI Inch**. Then set the **End Condition** to **Through All**. The **ANSI Inch** standard defines the available counterbore hole sizes based on inch units. The **Through All** end condition means the hole will cut completely through the part. Make sure both settings are selected correctly before continuing.



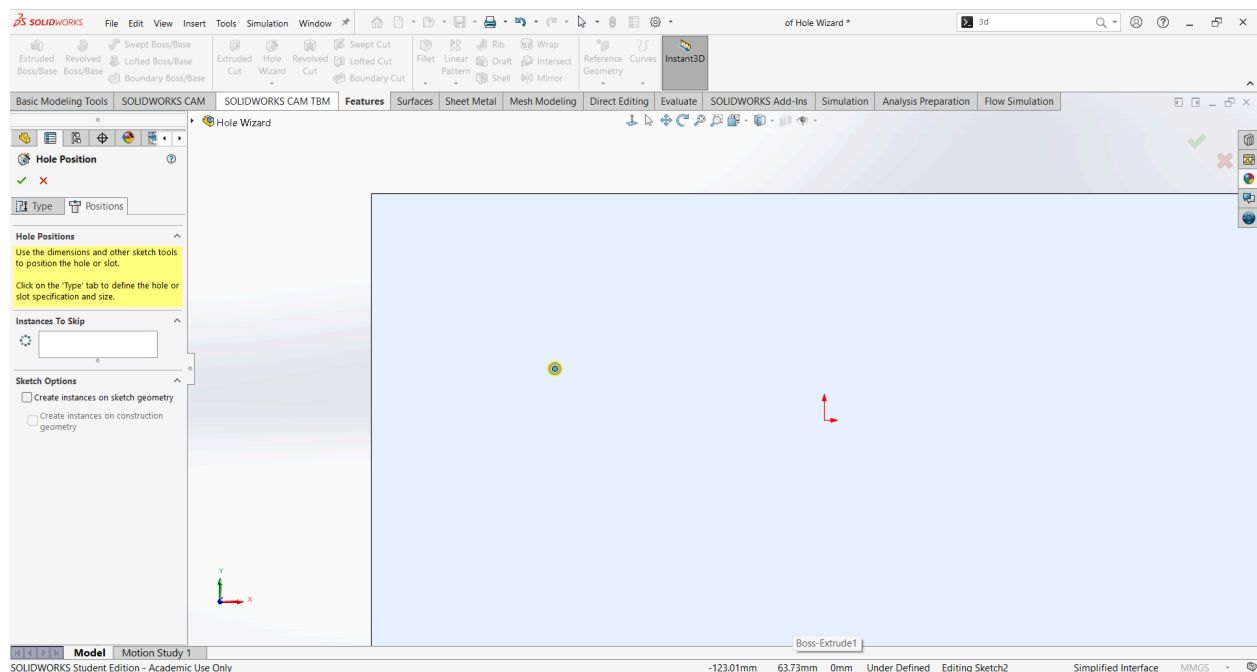
Now we need to define the hole position. In the **Hole Wizard PropertyManager**, select the second tab called **Positions**. This tab allows you to place the hole on the model. After selecting **Positions**, choose the face where you want to place the hole. Then click on the face to place a point. This point defines the center location of the hole. You can later use dimensions and relations to position the hole accurately.



Now select the face where you want to position the hole. For this example, select the **front face** of the model. After selecting the front face, SOLIDWORKS will allow you to place a point on that face. This point represents the center position of the hole. Click on the front face to place the hole point. You can then use **Smart Dimension** and sketch relations to fully define the exact position of the hole.

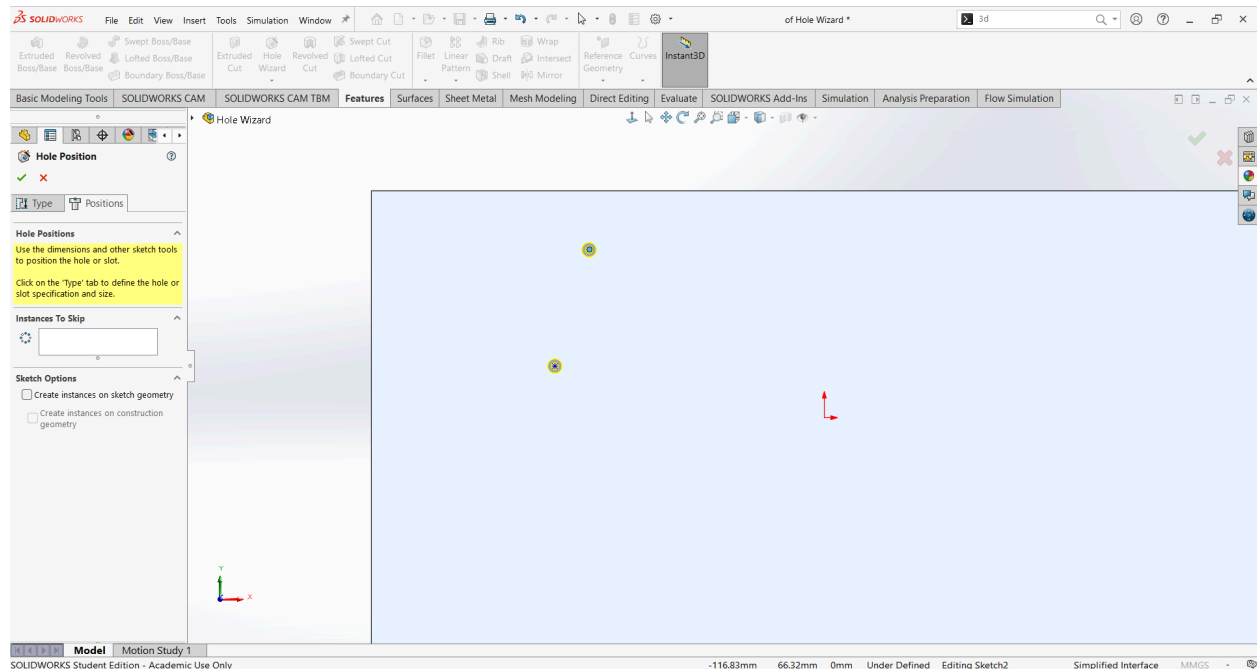


Now place the hole point on the selected face. Move your mouse over the front face. Then click once on the face to place the point. This point represents the center location of the hole. At this stage, the point is placed freely on the face. Next, you can use **Smart Dimension** or sketch relations to position the hole accurately.

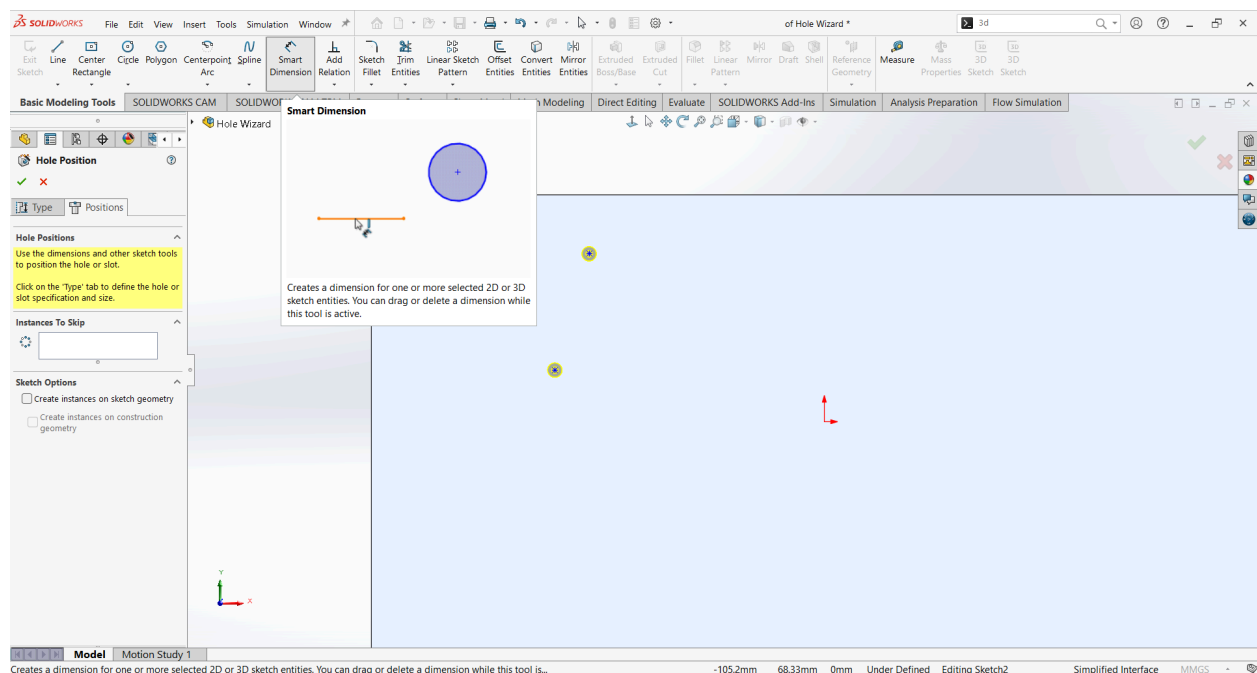


At this stage, you can add multiple hole points on the selected face. Each point will create a separate hole using the same **Hole Wizard** settings. For this example, place a second point on

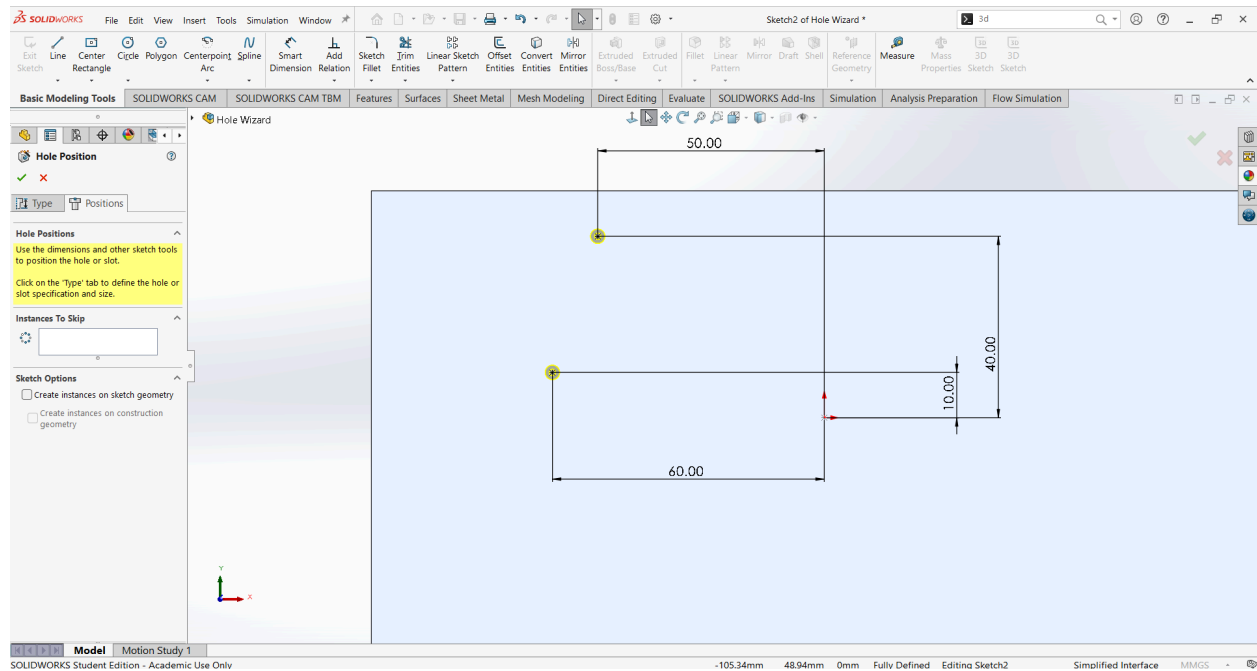
the face. Click once more on the face to create the second hole point. Now you should have two hole points on the front face. Next, we will define their positions using dimensions or relations.



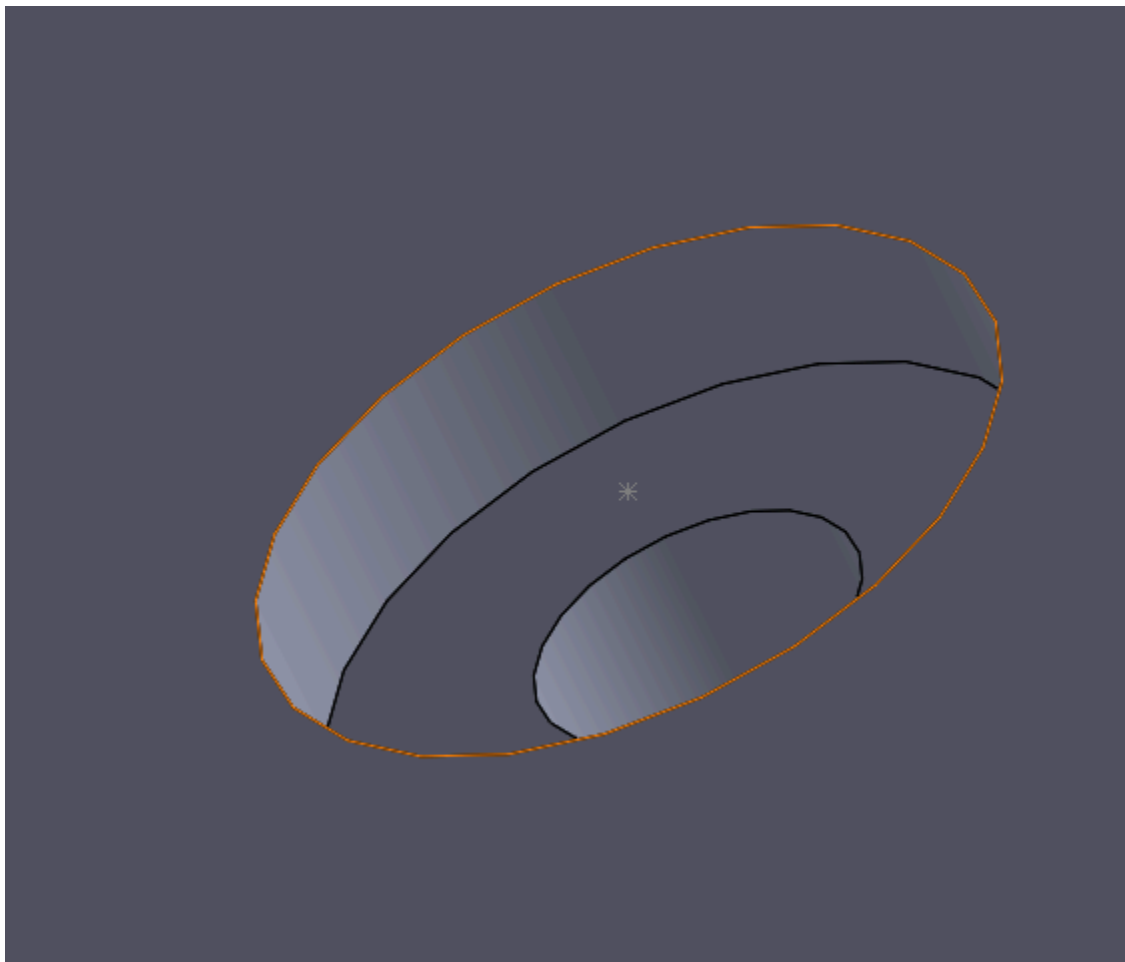
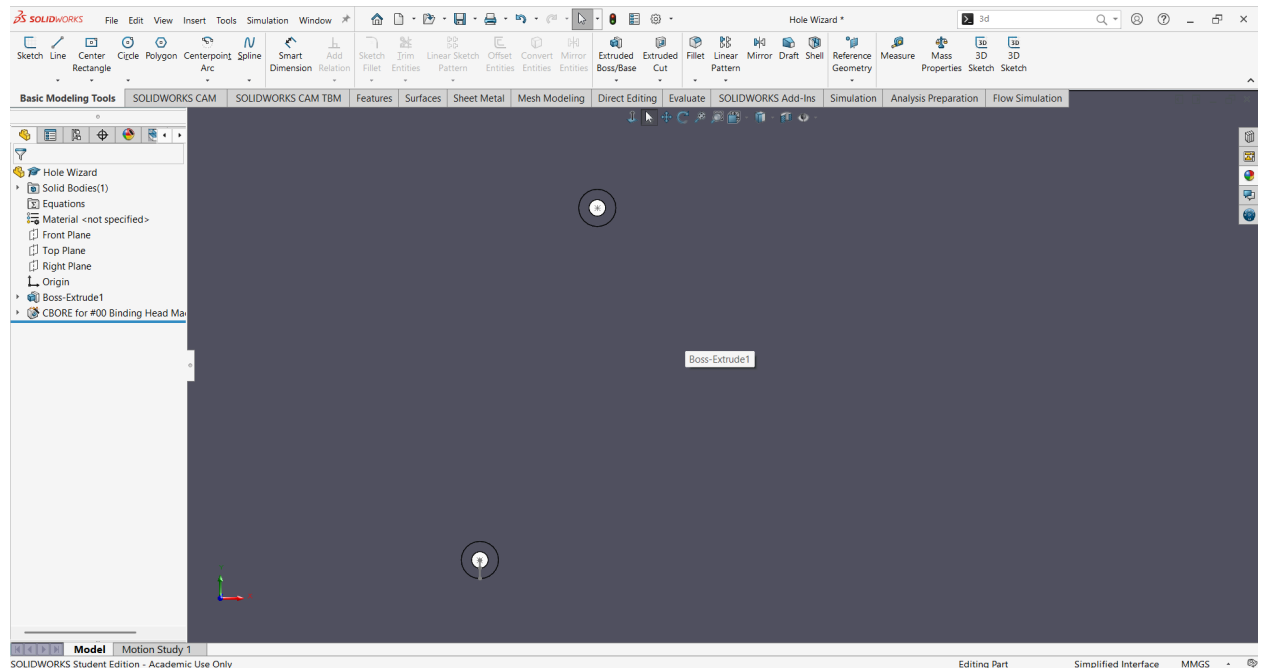
Before clicking **OK**, we can define the exact position of the hole points using **Smart Dimension**. Go to the **Basic Modeling Tools** tab. Select **Smart Dimension**. Now select the hole point and the reference edge or reference line you want to dimension from. Then enter the required dimension value. Repeat this process for the other direction if needed. This allows you to fully define the location of each hole point before creating the final **Hole Wizard** feature.



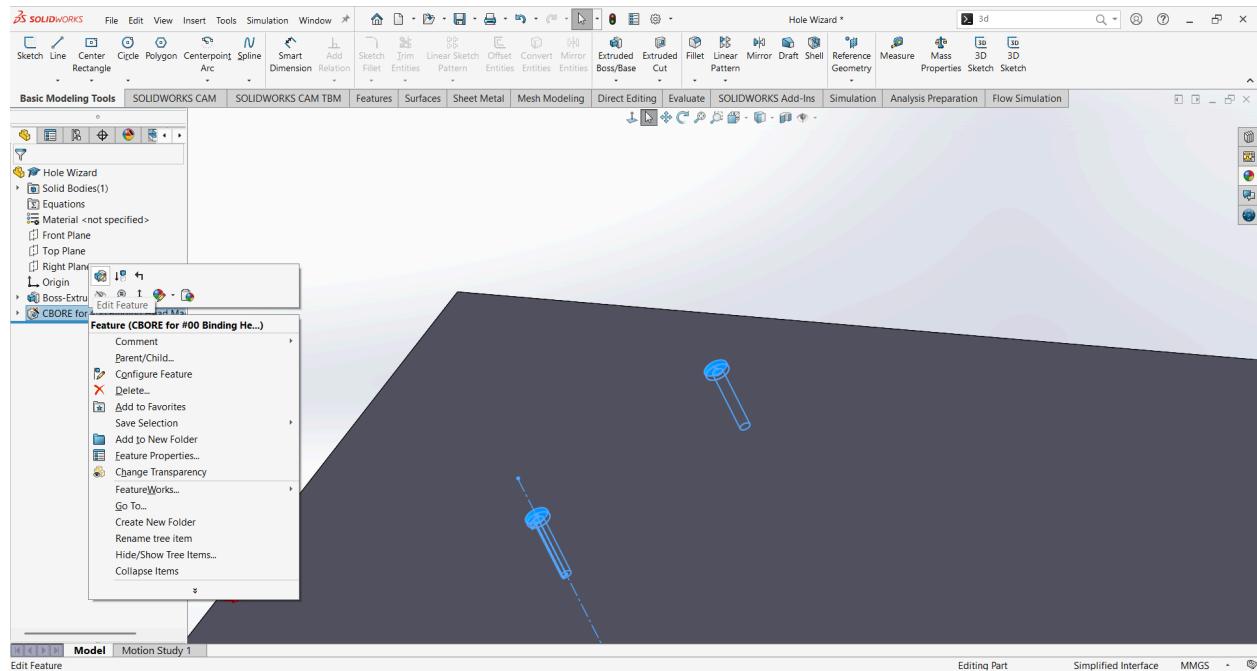
Apply the dimensions shown. Use **Smart Dimension** to define the exact position of each hole point. Select the hole point and the reference edge, then enter the required dimension value. Repeat the process for the remaining dimensions, as shown. Make sure both hole points are fully defined before continuing. Once the hole positions are correct, click **OK** by selecting the green check mark.



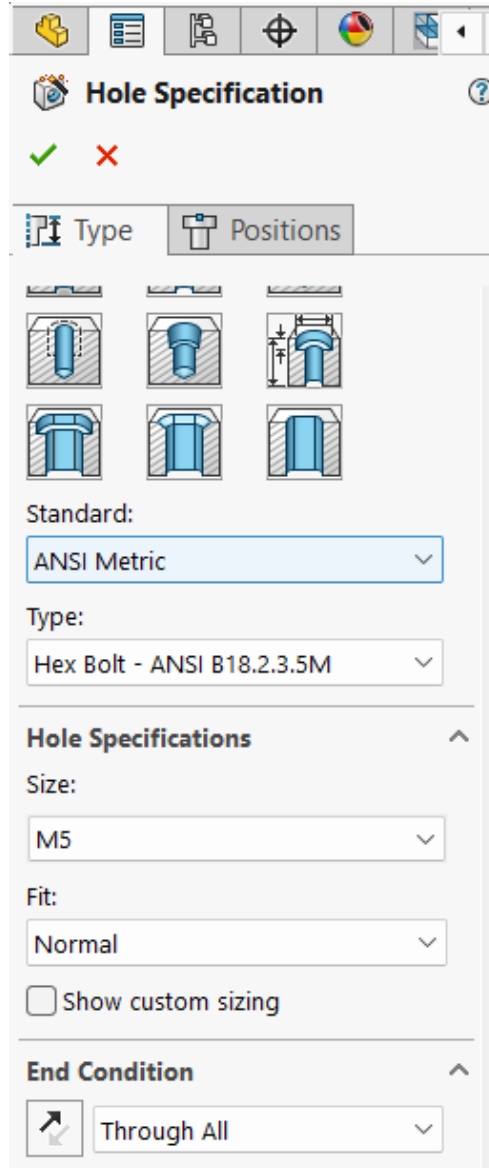
Click **OK** by selecting the green check mark. The **Hole Wizard** feature is now created. You can see the holes on the selected face of the model. Because we placed two hole points, SOLIDWORKS created two counterbore holes using the same settings. You can also find the new **Hole Wizard** feature in the **FeatureManager Design Tree**.



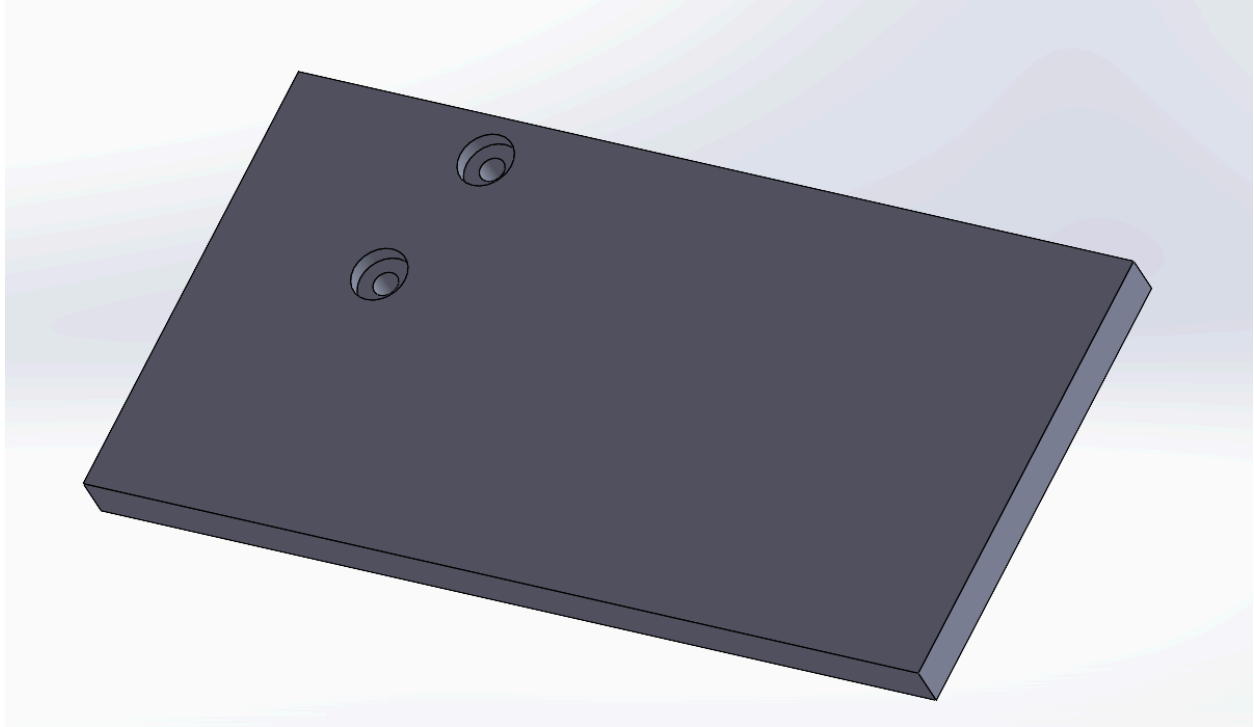
If you want to change the hole settings later, go to the **FeatureManager Design Tree**. Find the **Hole Wizard** feature. Right-click the **Hole Wizard** feature and select **Edit Feature**. This will reopen the **Hole Wizard PropertyManager**. Here, you can change the hole type, standard, size, end condition, tolerance, or other parameters. After making the changes, click **OK** by selecting the green check mark. The holes will update automatically based on the new settings.



Now change the hole standard and size. In the **Hole Wizard PropertyManager**, set the **Standard** to **ANSI Metric**. Then select **M5** as the hole size. This changes the hole from an inch-based standard to a metric standard. After selecting **ANSI Metric M5**, check the preview to make sure the hole updates correctly. Click **OK** by selecting the green check mark.



Here are the updated holes. The holes have now been changed to **ANSI Metric M5**. You can see that the **Hole Wizard** feature updated automatically based on the new standard and size. This is one of the main benefits of using **Hole Wizard** because you can quickly edit the hole parameters without recreating the holes from the beginning.



24. Linear Pattern

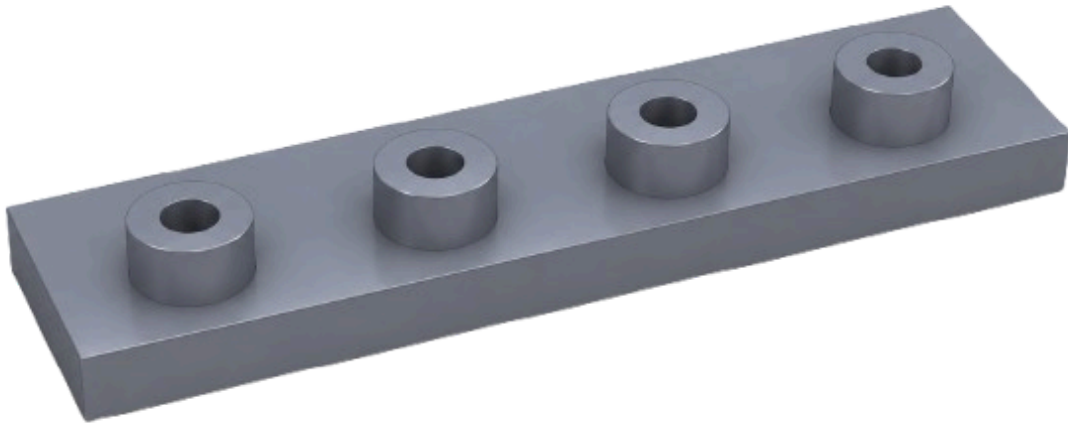
What it does: Repeats features in one or two linear directions.

Use it for: Repeated holes, ribs, bosses, cuts, and other features.

How it works:

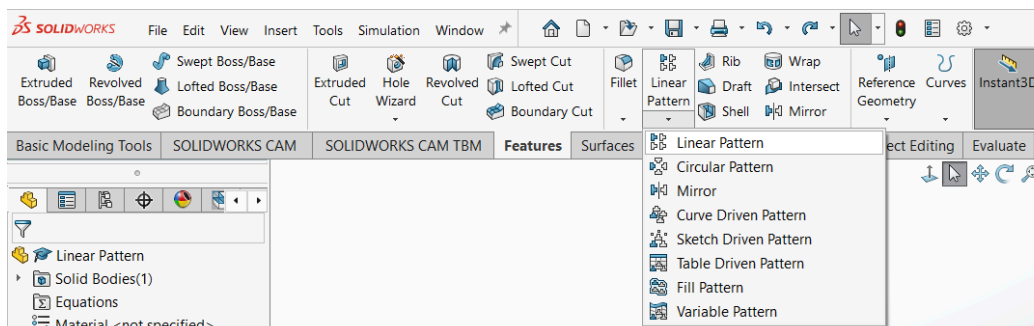
The **Linear Pattern** tool is used to copy an existing feature in one or two straight directions.

Instead of creating the same feature multiple times manually, you can create one feature and repeat it with a defined spacing and number of instances.

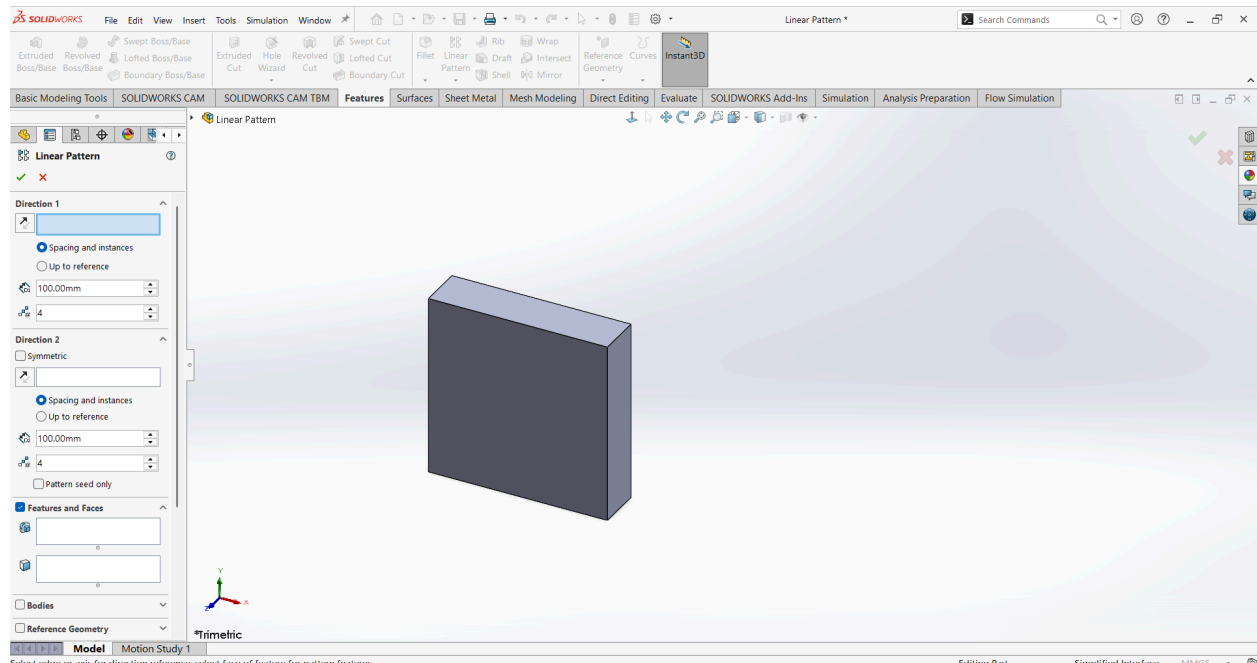


This tool is useful for repeated holes, cuts, bosses, ribs, slots, and other features that follow a straight pattern.

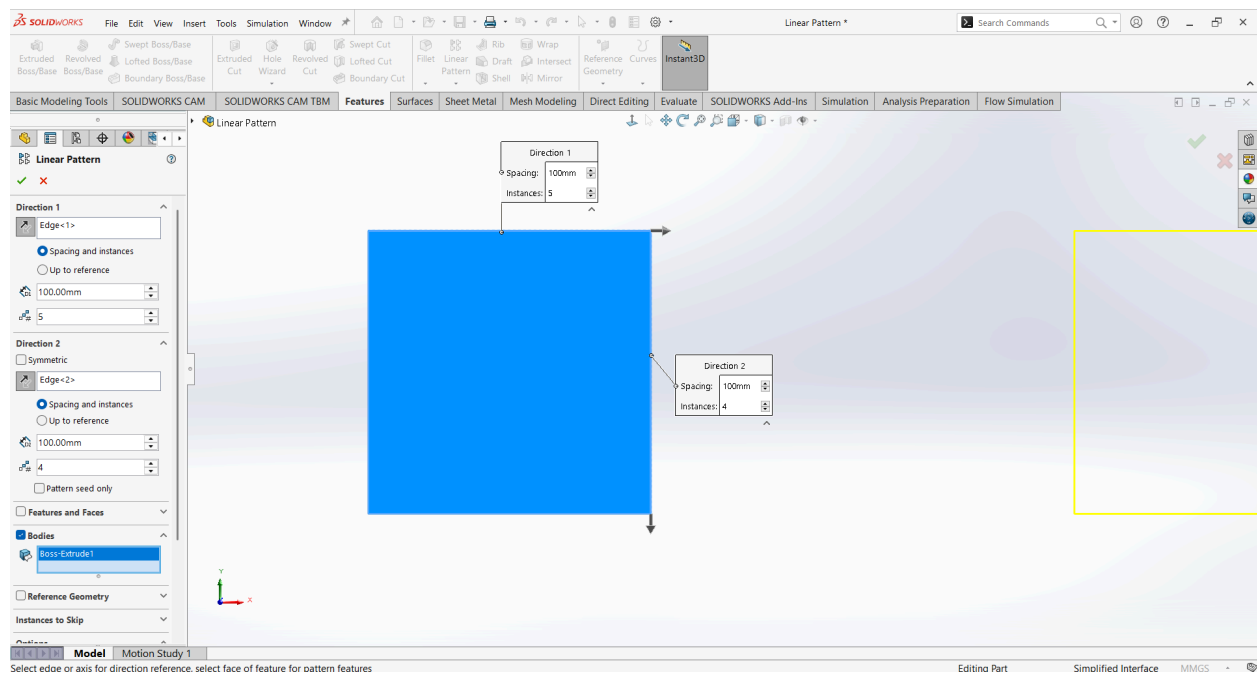
To access **Linear Pattern**, go to the **Features** tab. Find the **Linear Pattern** tool in the toolbar. Click **Linear Pattern** to open the tool settings.



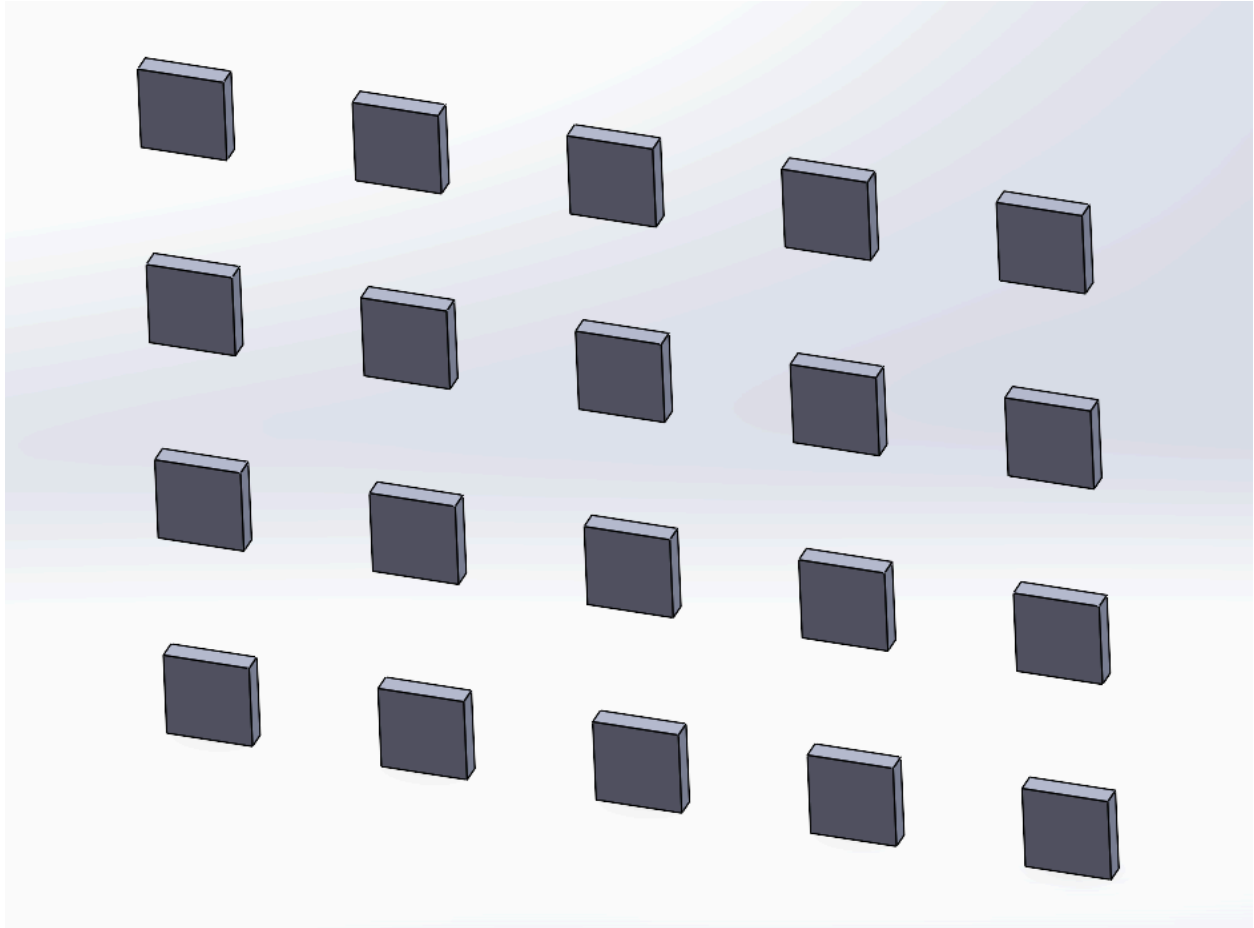
The **PropertyManager** will appear on the left side, where you can define the pattern direction, spacing, number of instances, and the feature you want to pattern.



For **Direction 1**, select the horizontal edge. Set the spacing distance to **100 mm**. Set the number of instances to **5**. For **Direction 2**, select the vertical edge. Set the spacing distance to **100 mm**. Set the number of instances to **4**. Then, under **Bodies to Pattern**, select the body you want to pattern.



Click **OK** by selecting the green check mark. This is the final result.



25. Circular Pattern

What it does: Repeats features around an axis.

Use it for: Bolt circles, fan blades, holes, and radial features.

How it works:

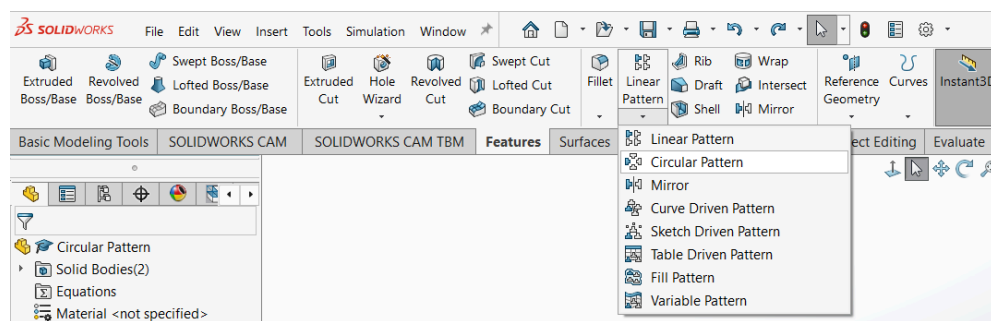
The **Circular Pattern** tool is used to copy an existing feature, body, or face around a circular direction.

Instead of creating the same feature multiple times manually, you can create one feature and repeat it around an axis, circular edge, or cylindrical face.

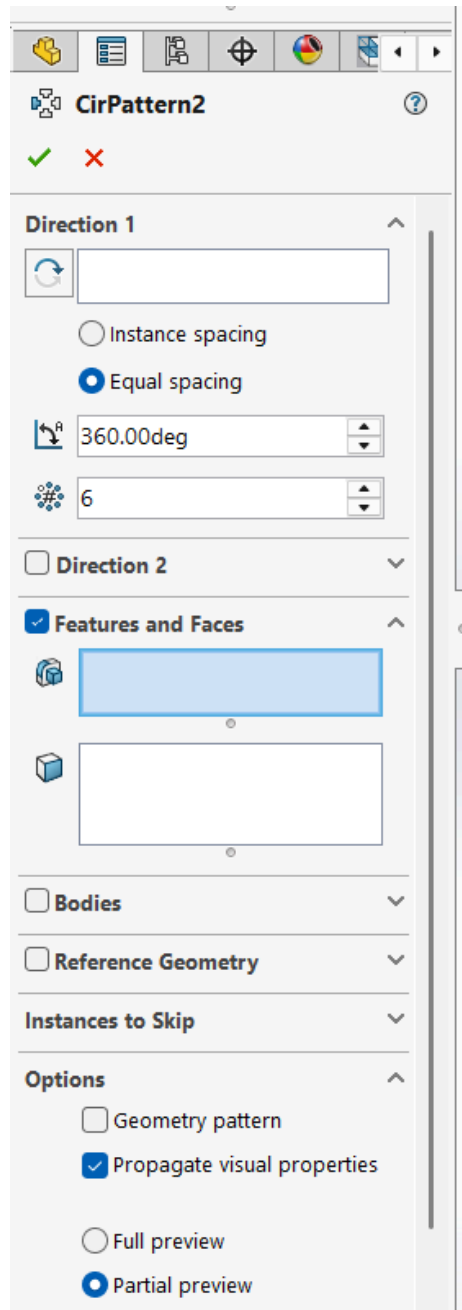


This tool is useful for repeated holes, cuts, bosses, ribs, slots, and other features arranged in a circular pattern.

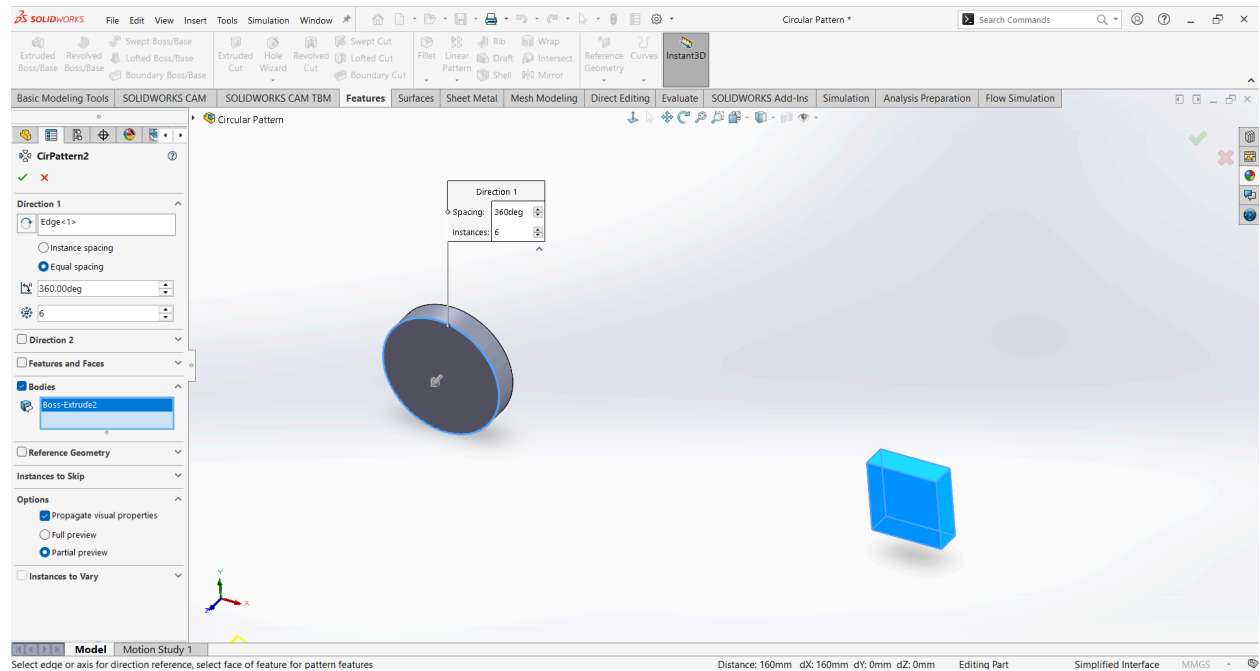
To access **Circular Pattern**, go to the **Features** tab. Find the **Linear Pattern** tool in the toolbar. Click the small arrow below **Linear Pattern** to open the drop-down menu. From the list of available pattern tools, select **Circular Pattern**.



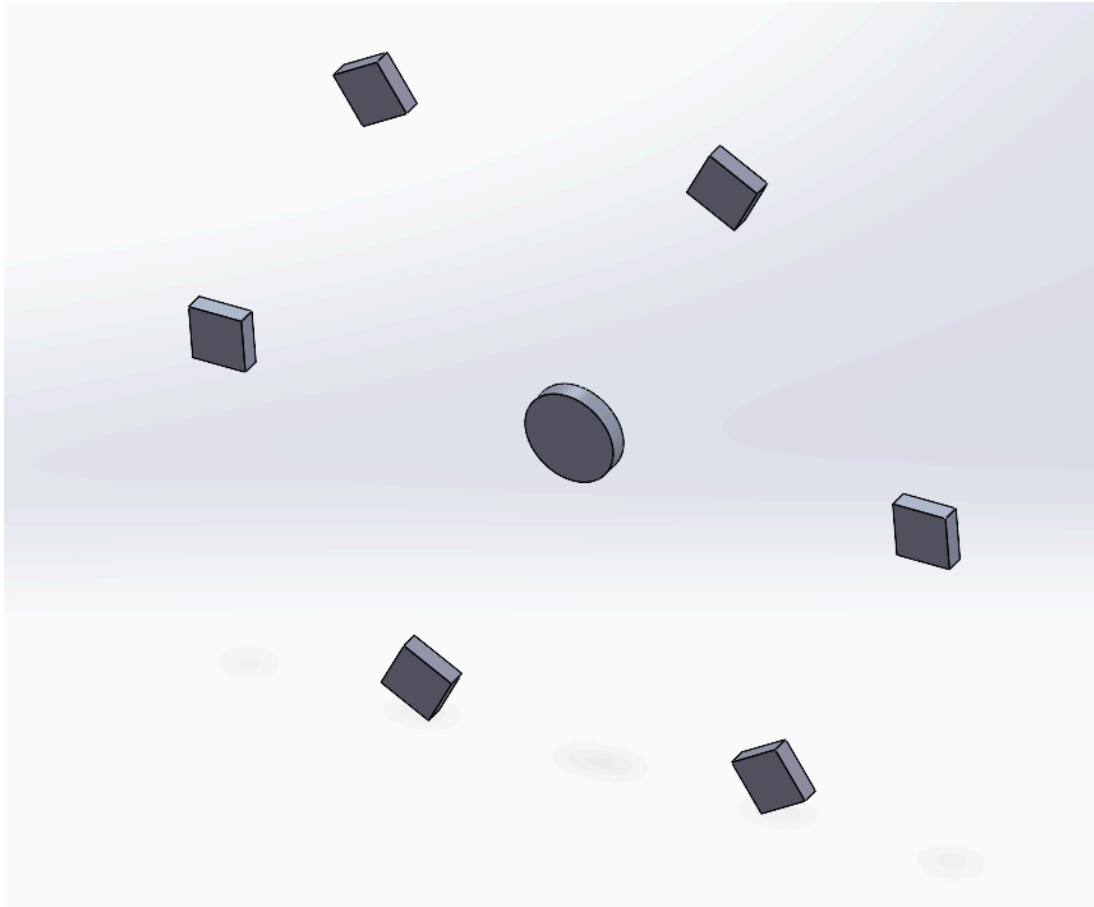
The **PropertyManager** will appear on the left side, where you can define the pattern direction, number of instances, spacing angle, and the feature or body you want to pattern.



For **Direction**, select the circular edge on the cylinder. Set the spacing angle to **360°**. Set the number of instances to **6**. Then, under **Bodies to Pattern**, select the rectangular body on the right side.



Click **OK** by selecting the green check mark. This is the final result.



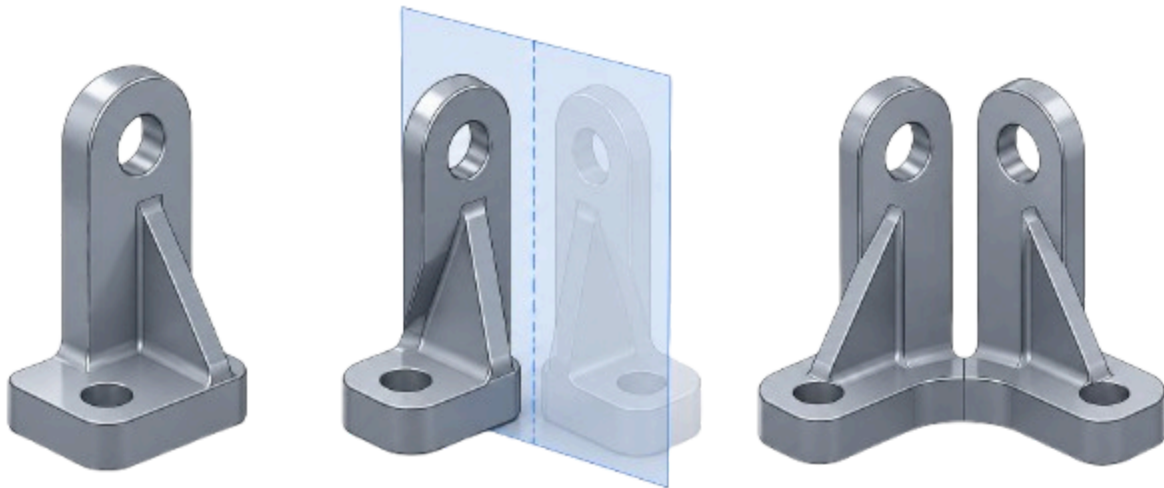
26. Mirror

What it does: Creates a mirrored copy of features or bodies.

Use it for: Symmetrical components.

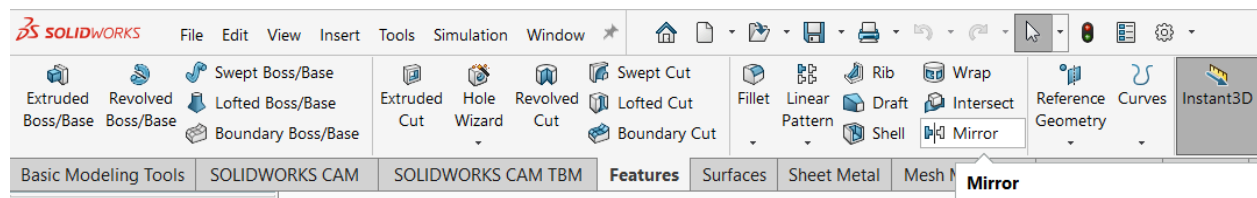
How it works:

If we have a model, body, or feature that we want to copy symmetrically to the other side, we can use the **Mirror** tool.

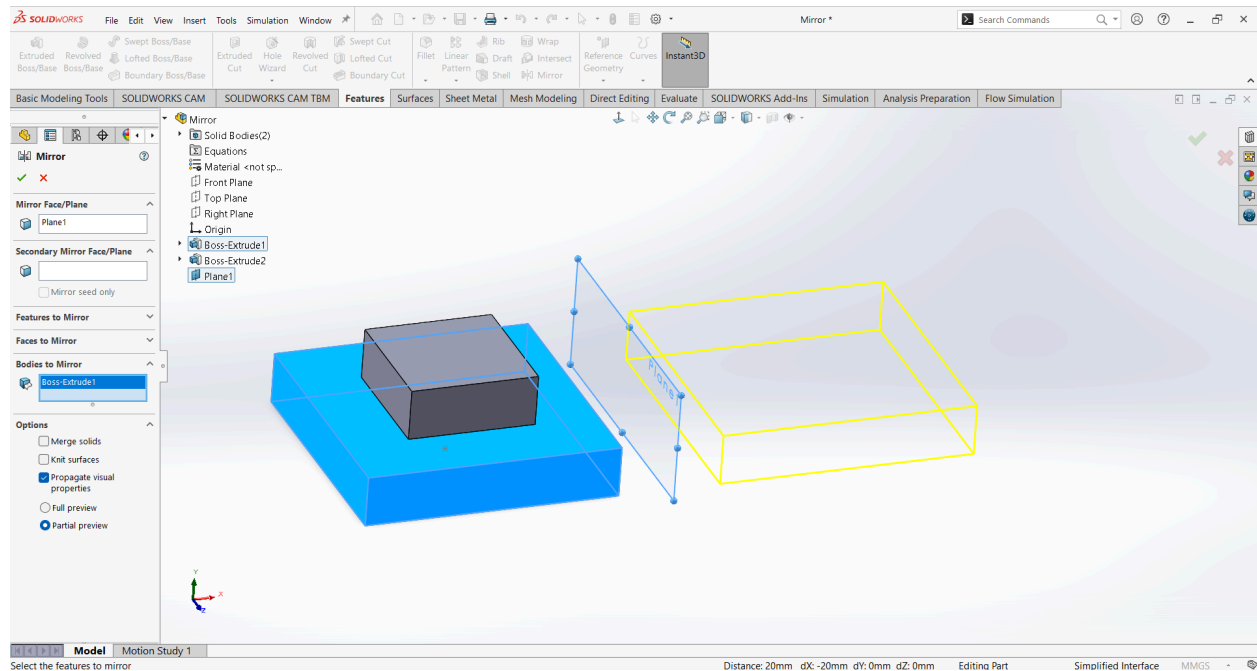


The **Mirror** tool creates a mirrored copy of selected features, faces, bodies, or components using a plane or flat face as the mirror reference.

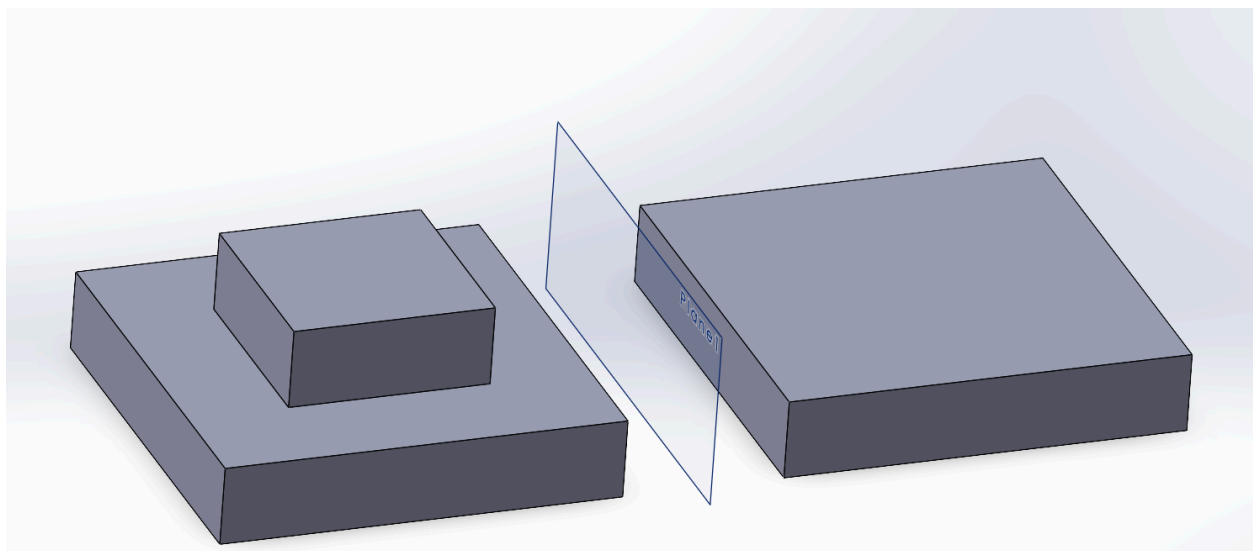
Go to the **Features** tab and select **Mirror**.



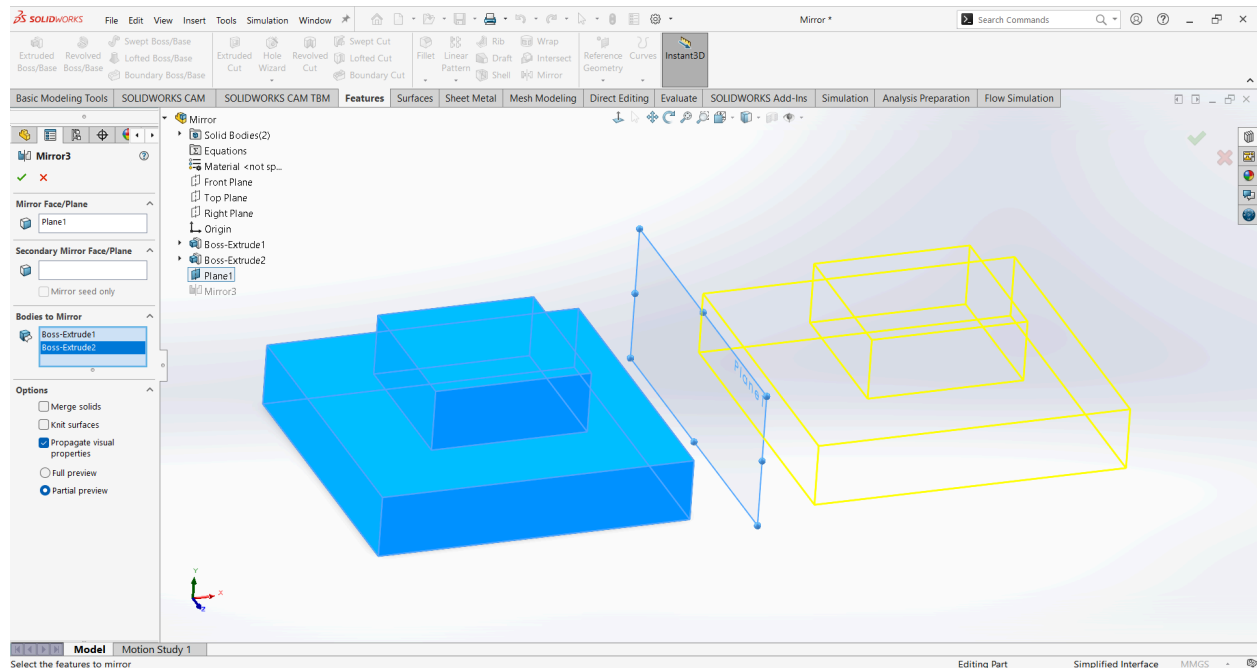
The mirror properties appear on the left side in the **PropertyManager**. In the first selection box, select the mirror reference. Select **Plane1** as the mirror plane. Then go to **Bodies to Mirror** and select the lower body. You can see the mirror preview before confirming the feature.



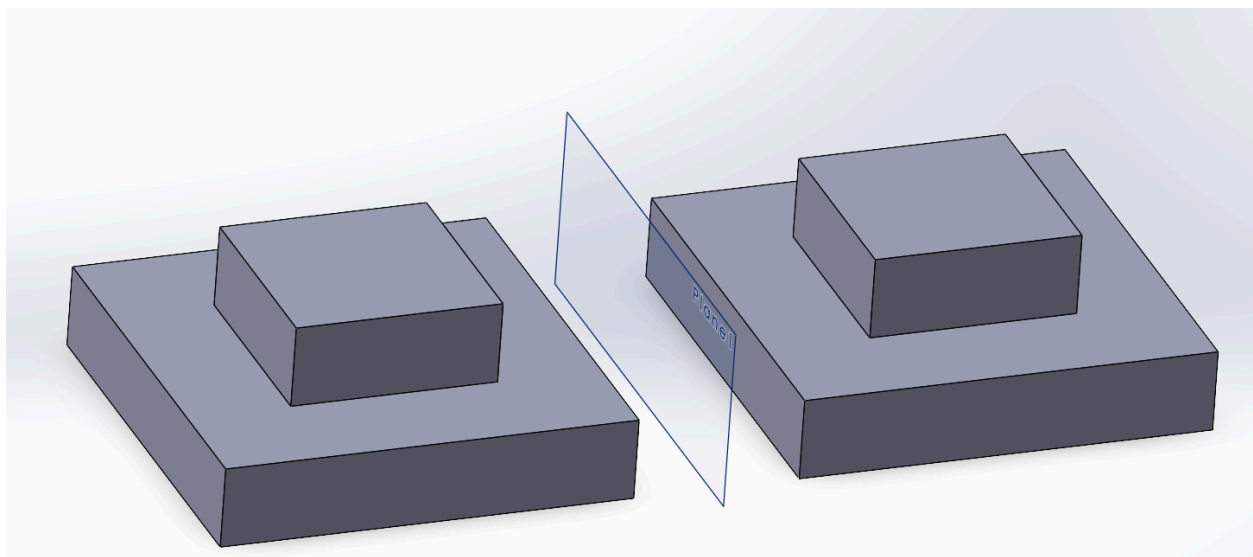
Click **OK**. This is the final result.



Now go back and edit the **Mirror** feature we created before. Add another body to the **Bodies to Mirror** selection box. Now you can see that both bodies will be mirrored.



Click **OK**. This is the final result.



3. ASSEMBLIES

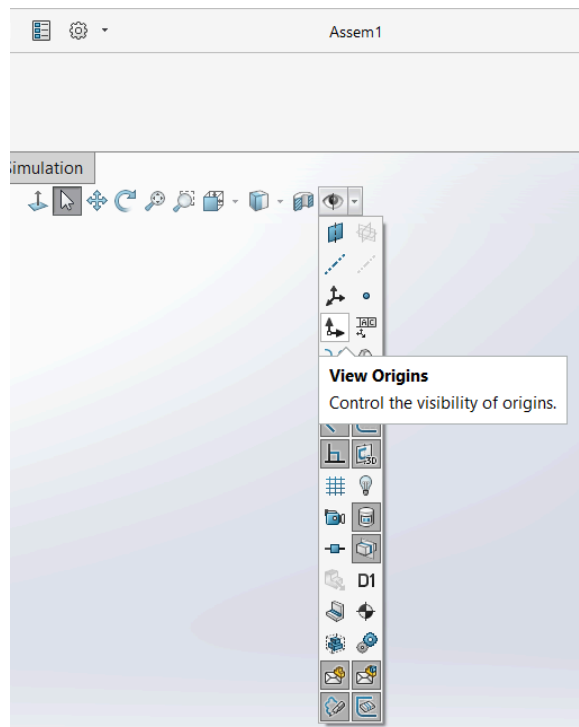
27. Insert Components

What it does: Adds parts or subassemblies to an assembly.

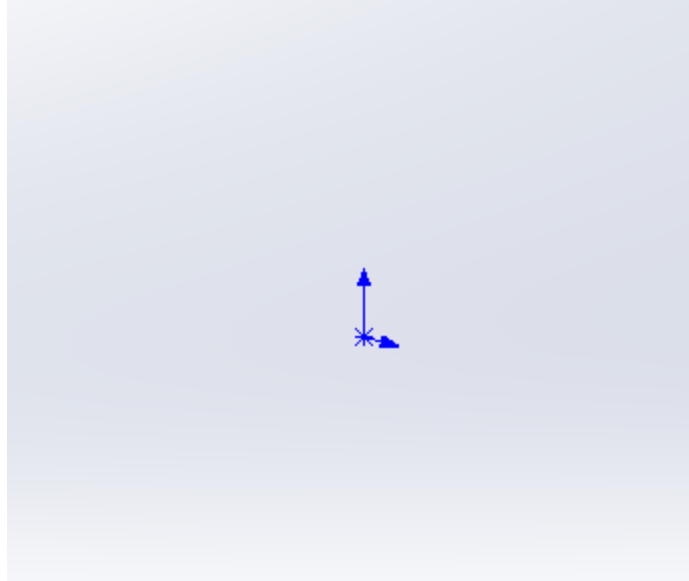
Quick Tip: The first component is commonly used as the foundation for positioning the rest of the assembly.

How it works:

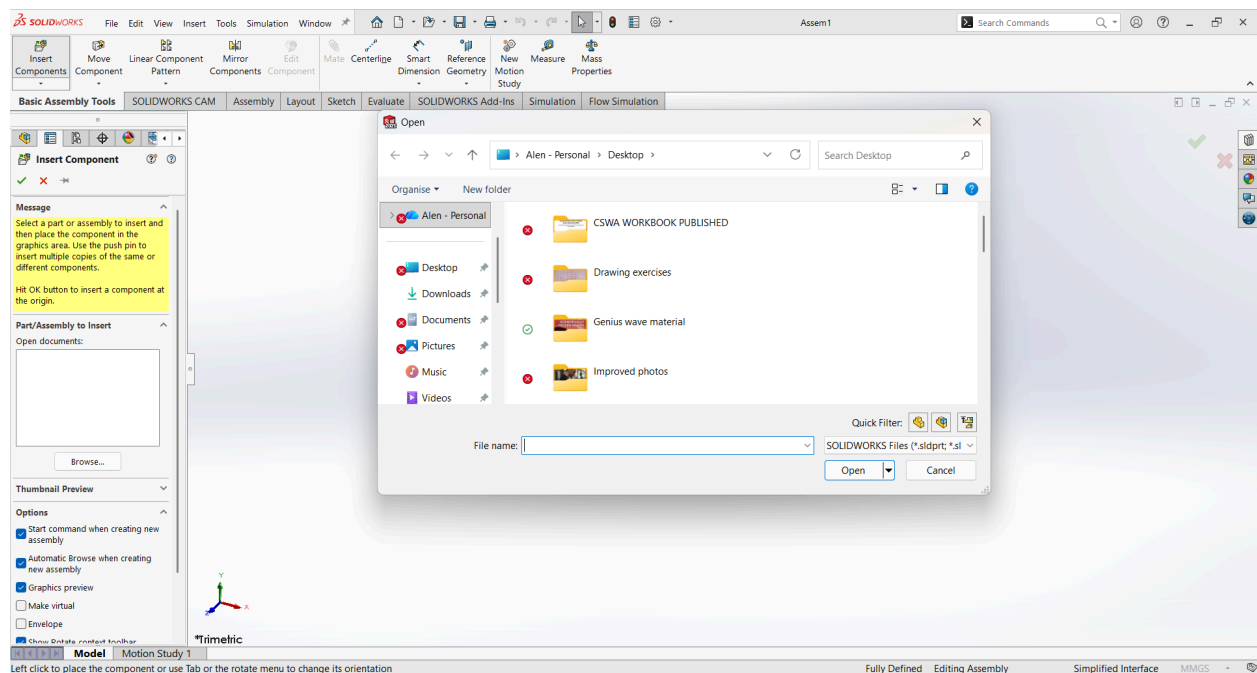
Now we will insert components into the assembly. Before we do that, let's make the assembly origin visible. Go to the **Heads-up View Toolbar**, click the small arrow next to the eye icon, then select **View Origins**.



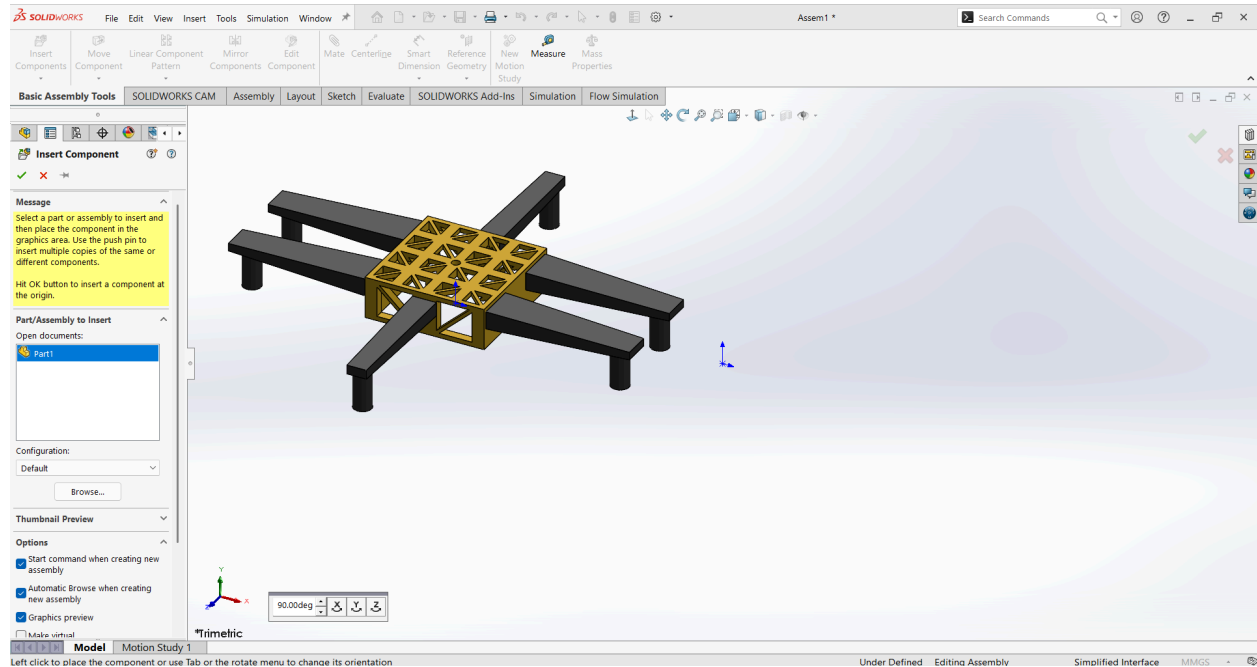
Now the assembly origin is visible.



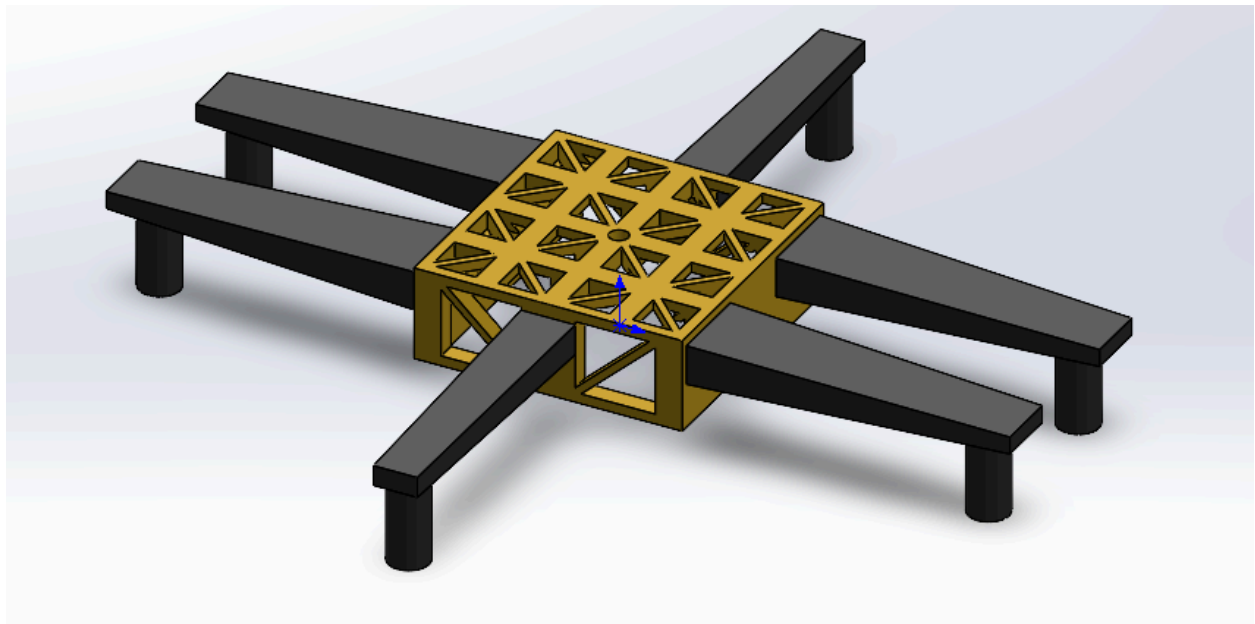
Go to the **Basic Assembly Tools** tab and select the first tool, **Insert Components**. A dialog box opens automatically on the left side in the **PropertyManager**. Another dialog box also opens, allowing you to select the components you want to insert into the assembly.



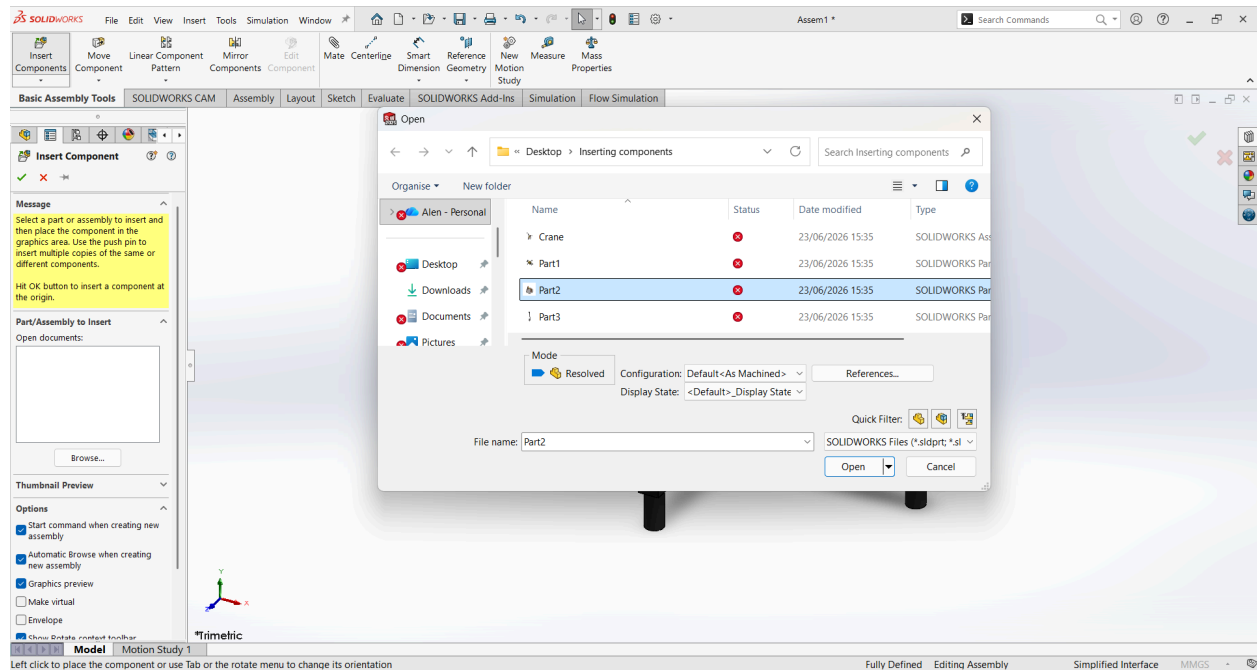
Find the component file on your computer. Select the component, then move your mouse into the SOLIDWORKS graphics area. You will see the component preview appear in the graphics area. As you move the mouse, the component moves with it.



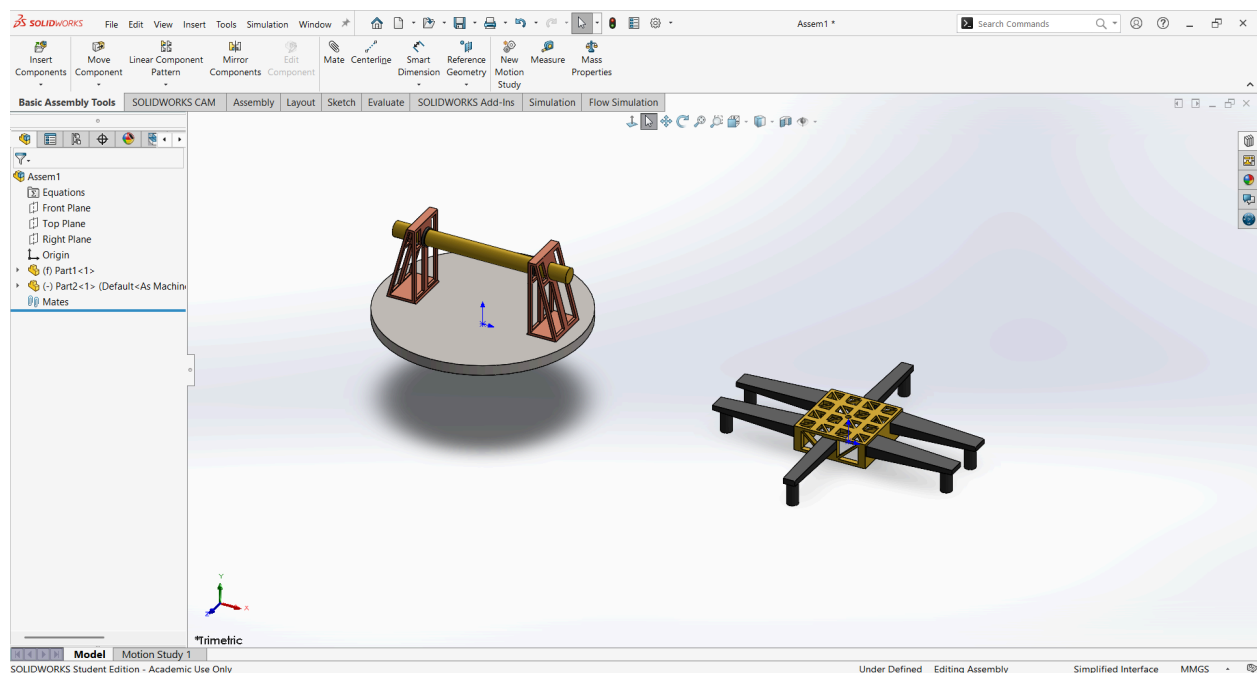
Move the mouse to the assembly origin. The component automatically snaps to the origin. Click there to place the component. Now the component origin and the assembly origin are coincident. The component is also fixed, which means you cannot move it freely in the assembly.



Let's insert the second component. Go to the **Basic Assembly Tools** tab and select **Insert Components**. When the dialog box opens, select the second component.



The new component appears on the screen. Click anywhere in the graphics area to place the component.



28. Mates

What it does: Defines relationships between assembly components.

Common mates include:

- Coincident
- Concentric
- Parallel
- Perpendicular
- Distance
- Angle

How it works:

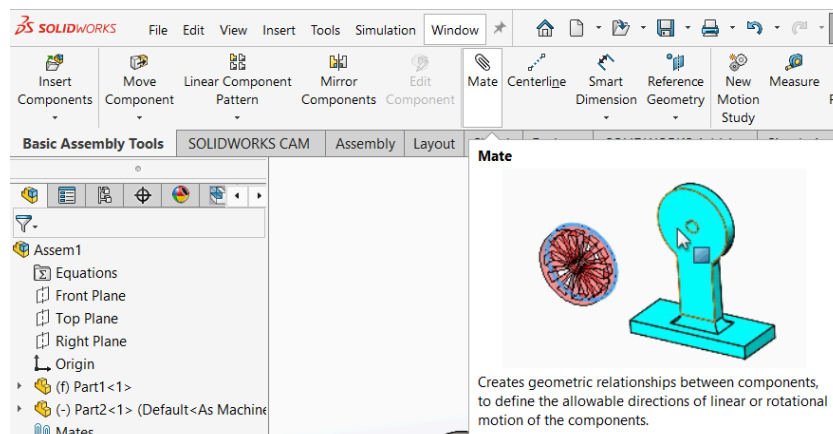
Now we will explain **Standard Mates**.

In an assembly, **mates** are used to define the relationship between components. They control how parts are positioned, aligned, and allowed to move relative to each other.

For example, mates can make two faces touch, align two holes, keep two parts parallel, or allow one component to rotate around another.

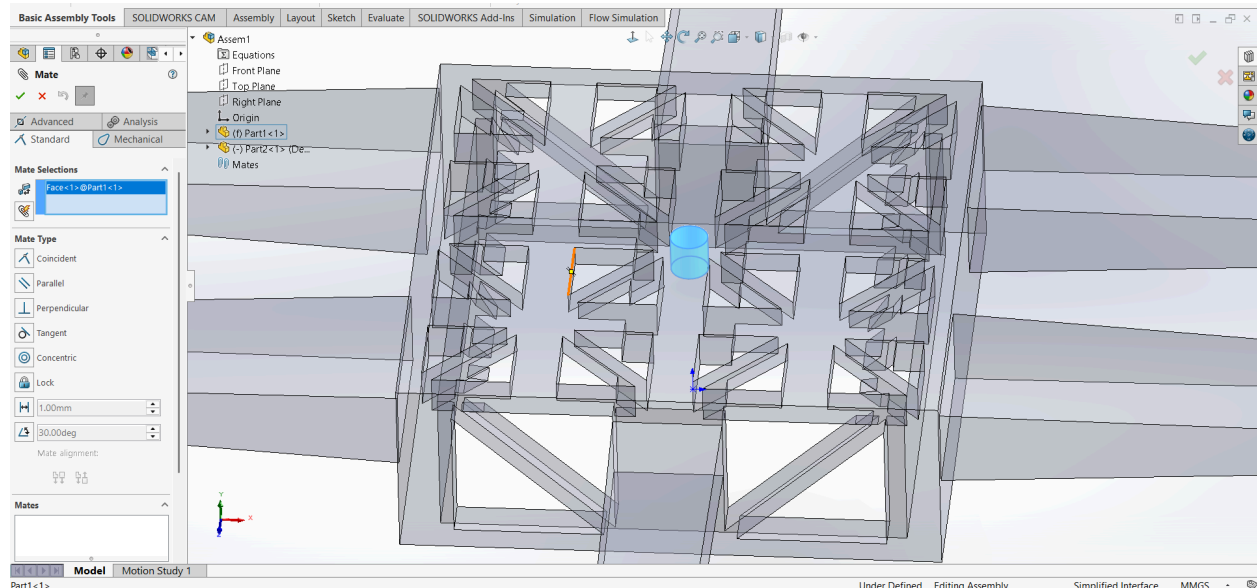
Mates are very important because they turn separate parts into a functional assembly that behaves more like a real mechanical system.

Go to the **Basic Assembly Tools** tab and select **Mate**.

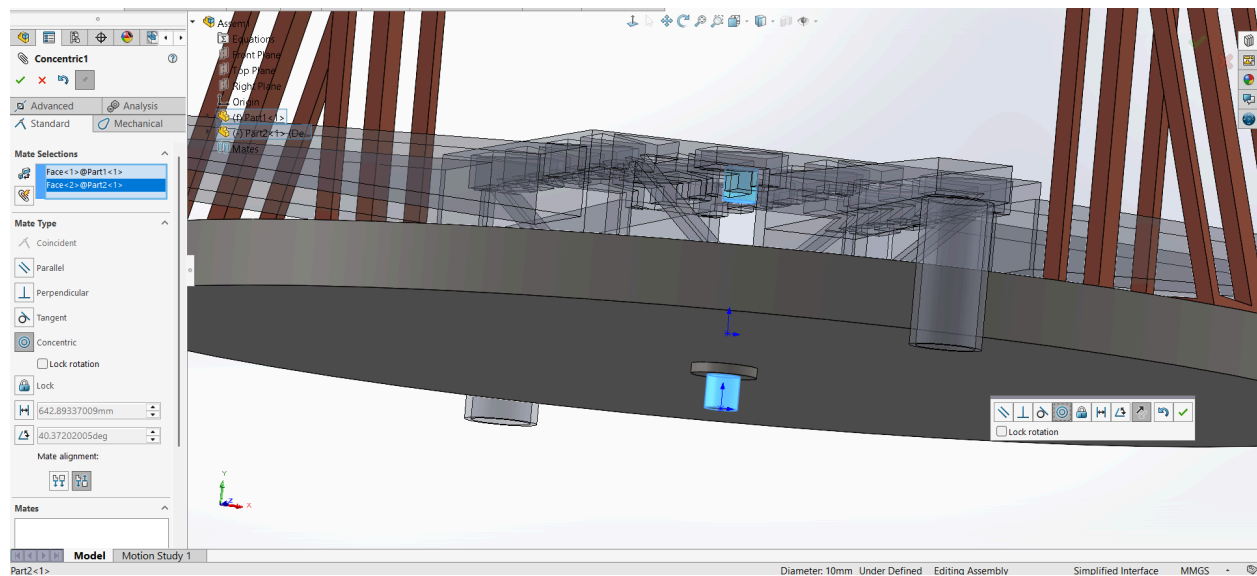


A new dialog box appears on the left side in the **PropertyManager**. Here, you can see the available **Standard Mates**, such as **Coincident**, **Parallel**, **Perpendicular**, **Tangent**, **Concentric**, **Distance**, and **Angle**. In the selection box, we need to select the faces, edges, or other geometry between which we want to define the mate relationship.

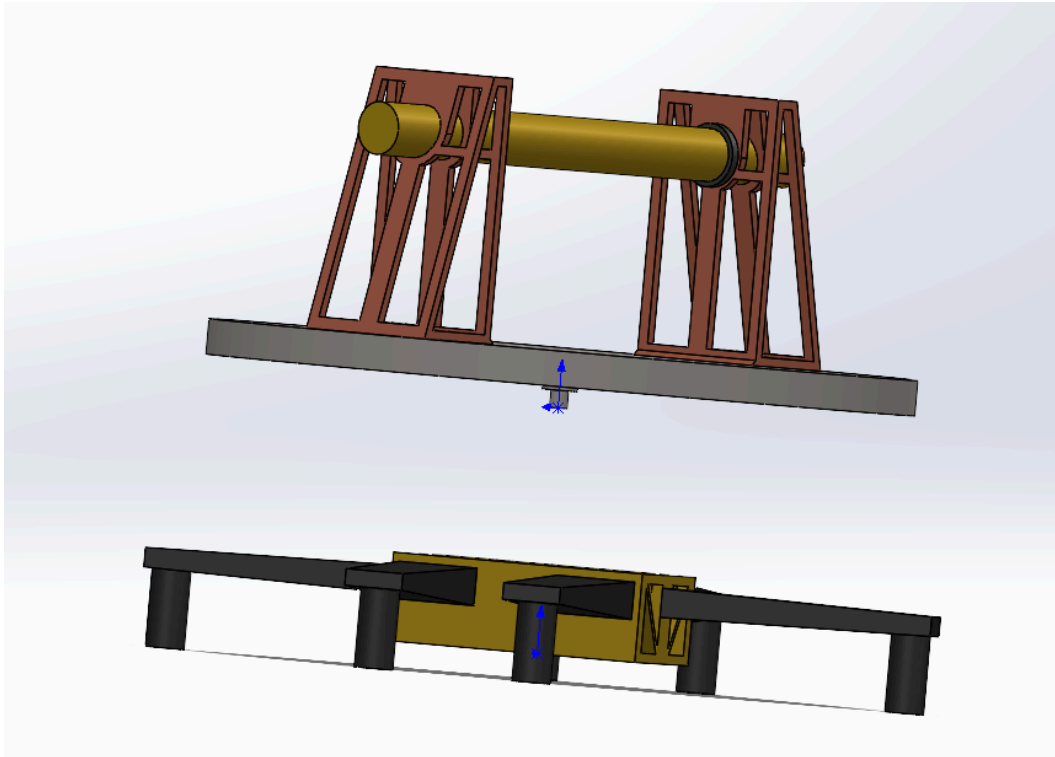
First, select the hole on **Component 1**.



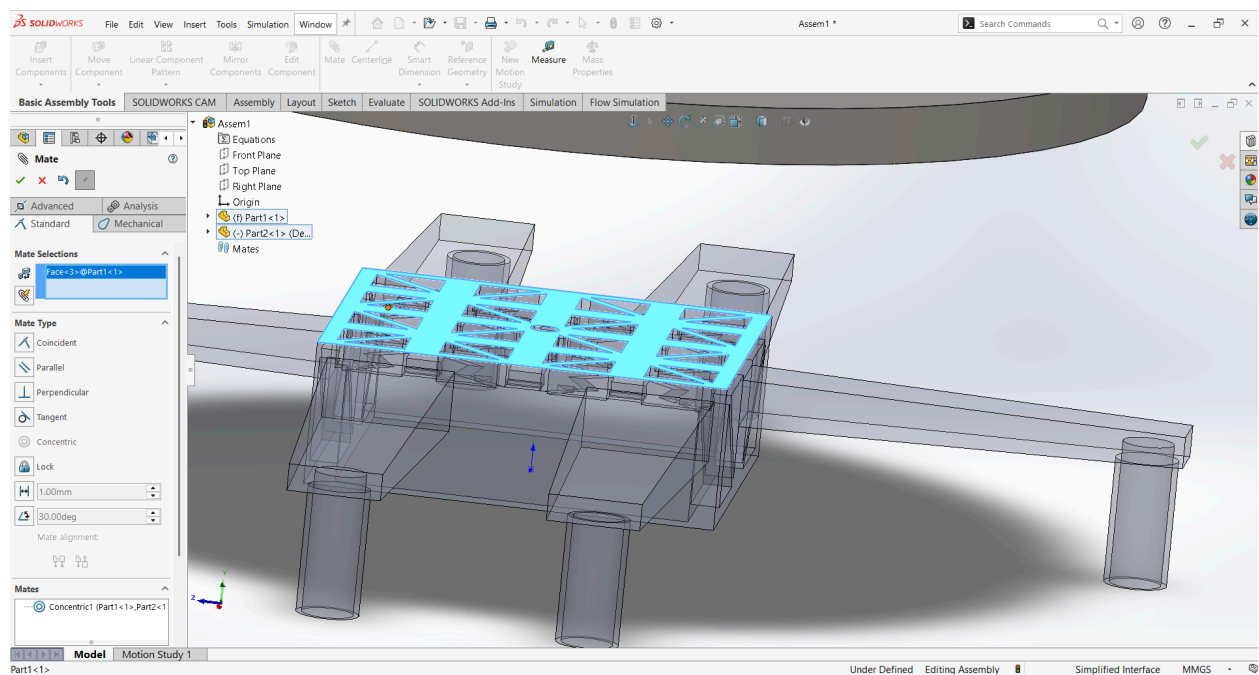
Then select the small pin at the bottom of **Component 2**. SOLIDWORKS automatically suggests the mate type **Concentric**. This is the connection type we want, because the pin should align with the hole. Click **OK**.



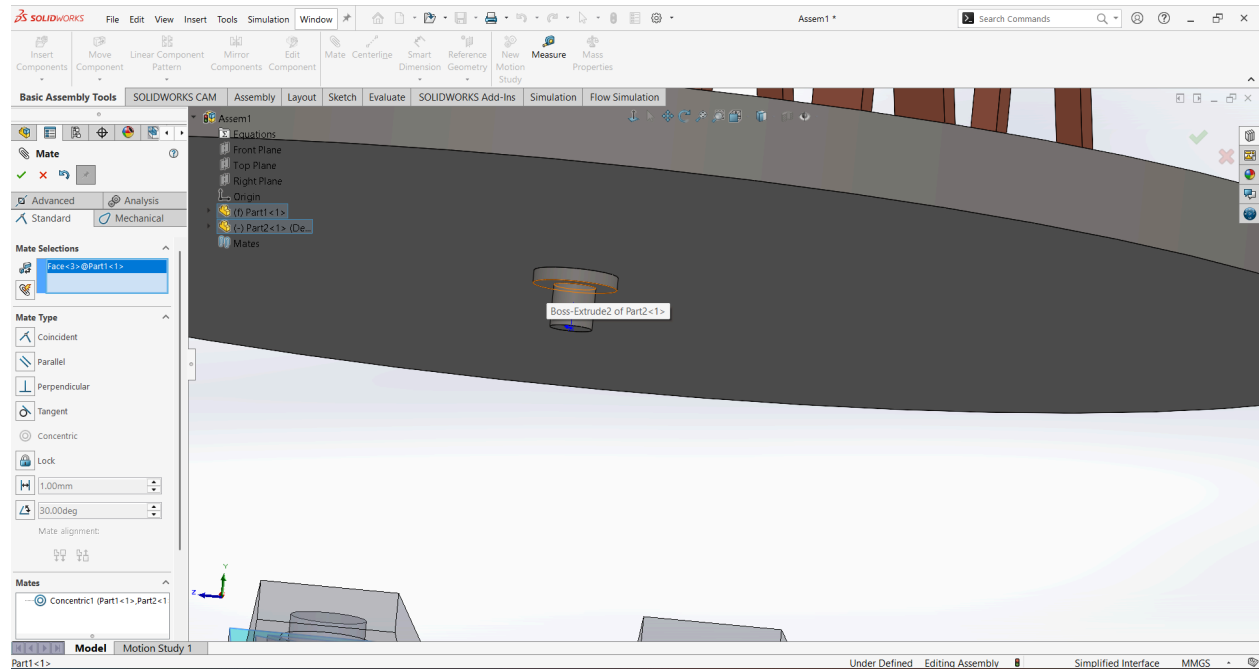
Now we have created a **Concentric** mate. Component 2 is aligned with the hole, but it can still move up and down along the shared axis.



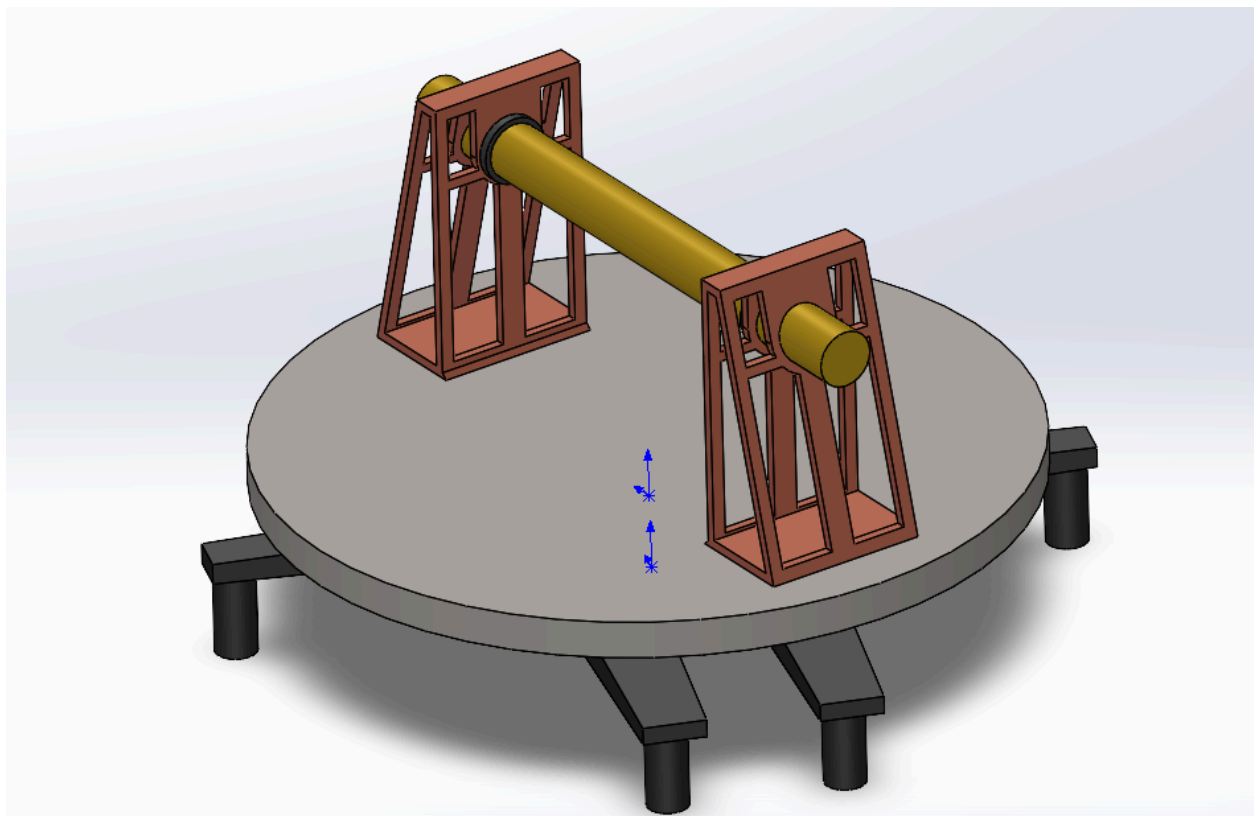
Now we need to prevent the up-and-down movement. Go to **Mate** again and select the upper face on **Component 1**.



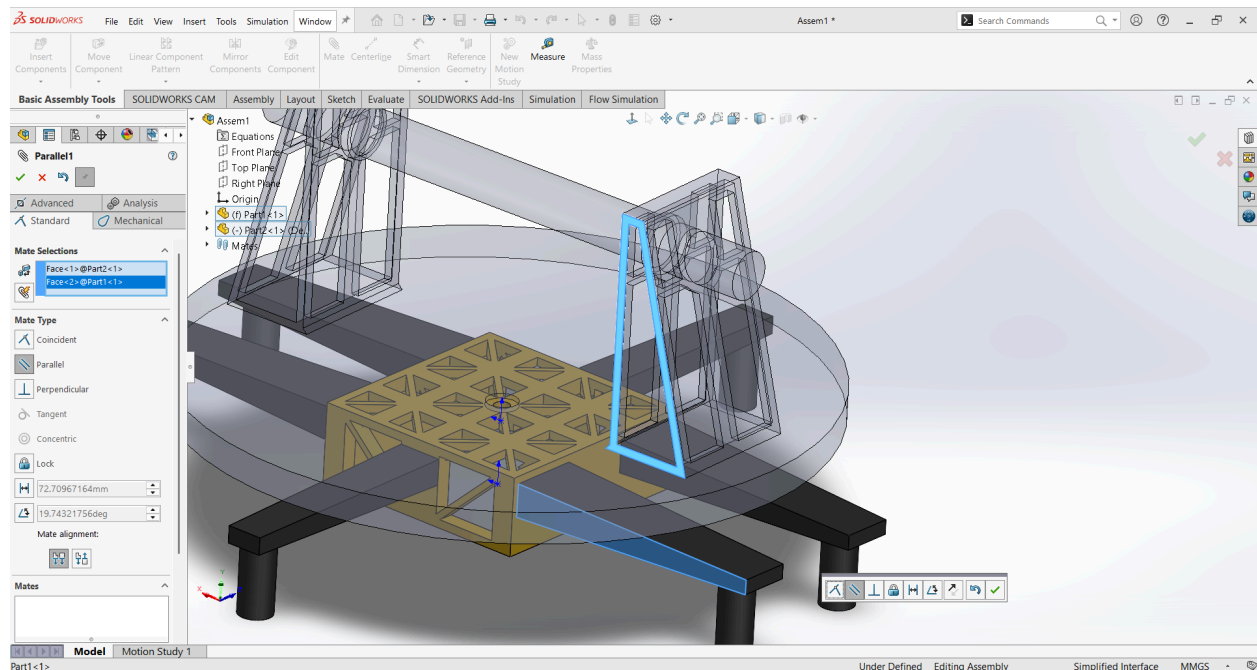
Now select the face on the smaller cylinder of **Component 2**. Click **OK**.



Now **Component 2** can only rotate around the shared axis.

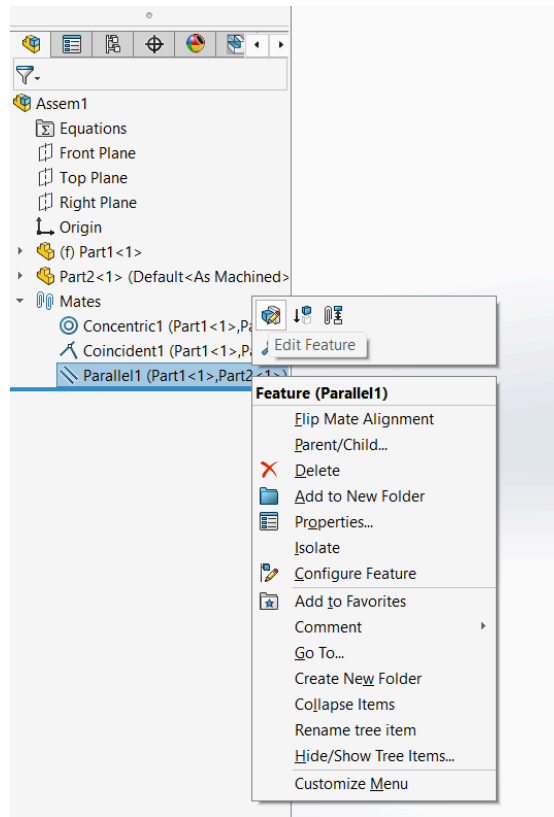


If we want to prevent rotation and fully restrict the movement of **Component 2**, we need to add another mate. Go to **Mate** and select the faces on **Component 1** and **Component 2**, as shown. Select the mate type **Parallel**. Click **OK**.

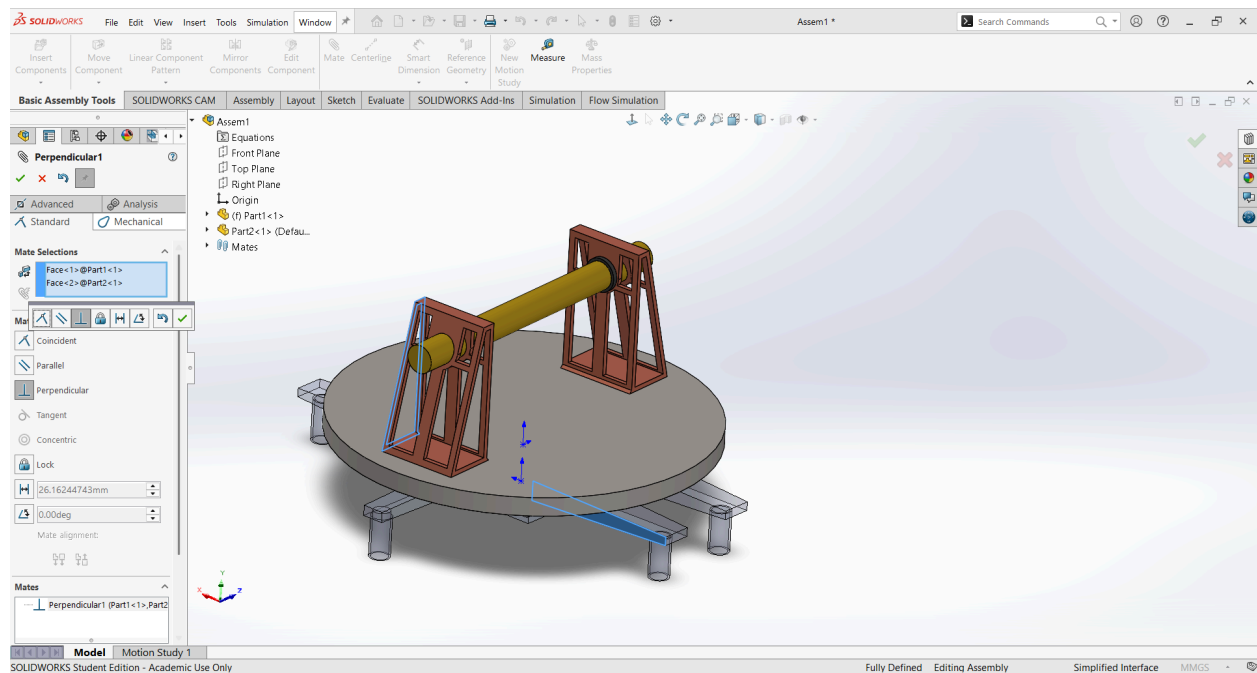


Now these two faces are parallel. This prevents **Component 2** from rotating, so the component is fully constrained.

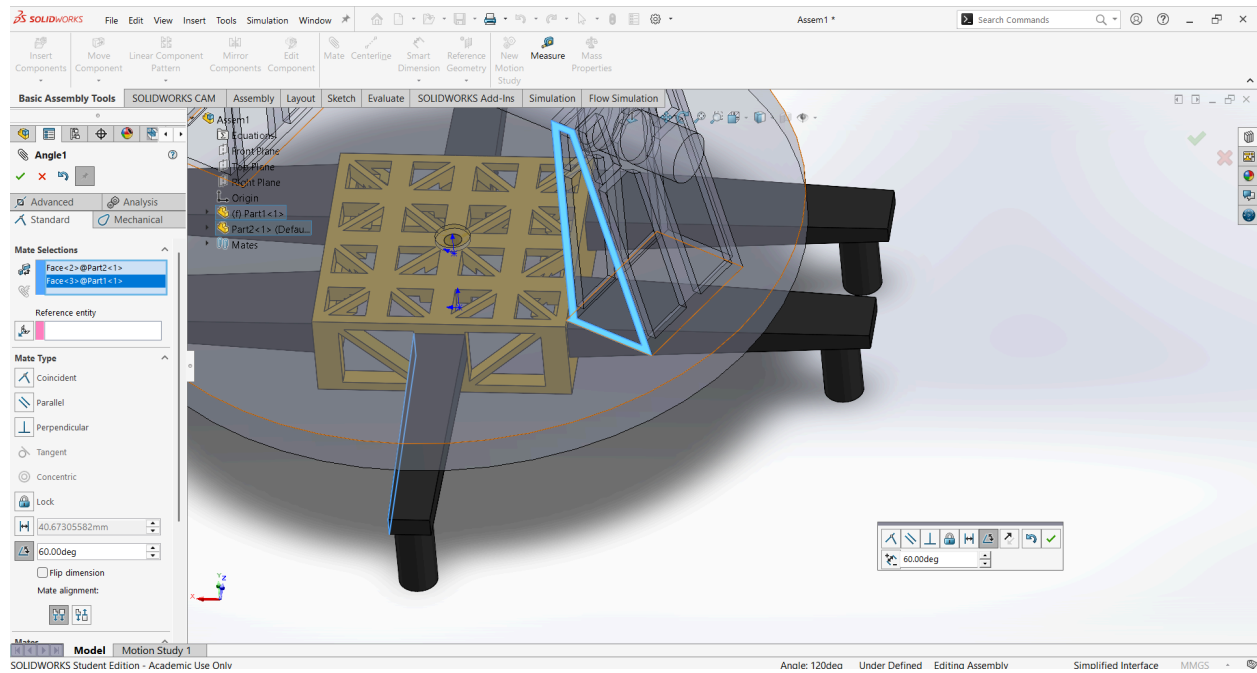
In the **FeatureManager Design Tree**, there is a folder called **Mates**. This folder contains all the mates we have created in the assembly. Go to the last mate, **Parallel**, right-click it, and select **Edit Feature**.



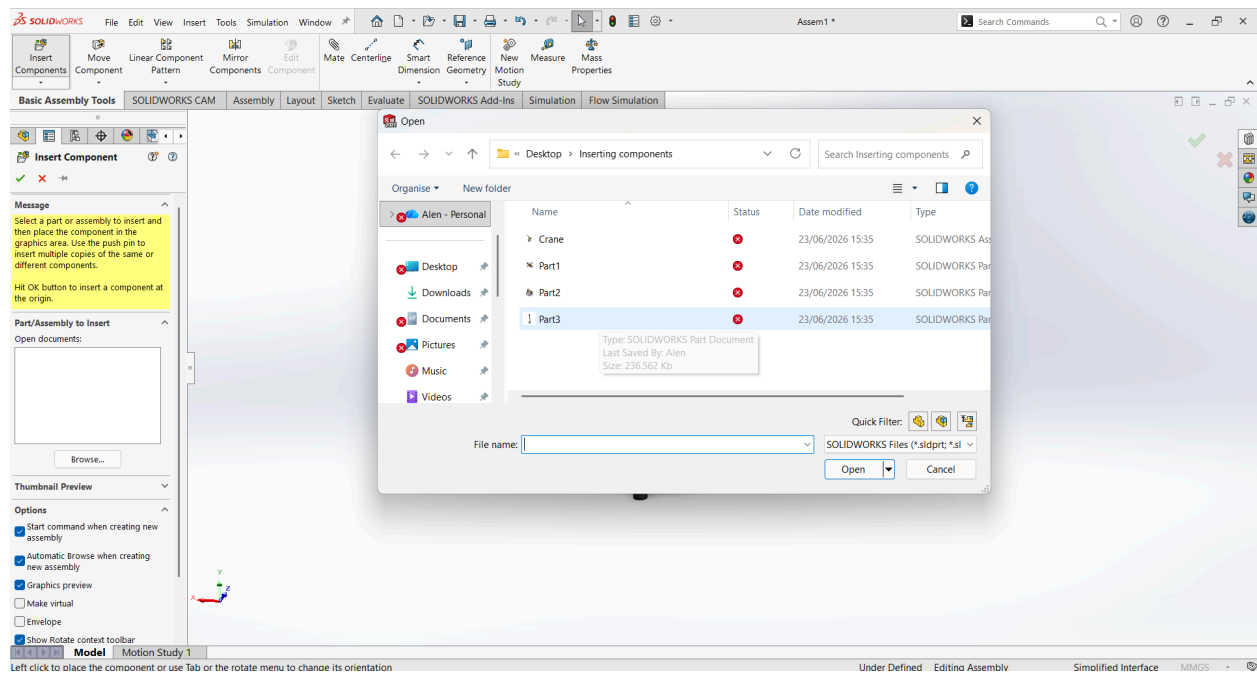
Now change the mate type. Select **Perpendicular**. Now these two faces have a **Perpendicular** relationship, which means they are positioned at a **90° angle** to each other.



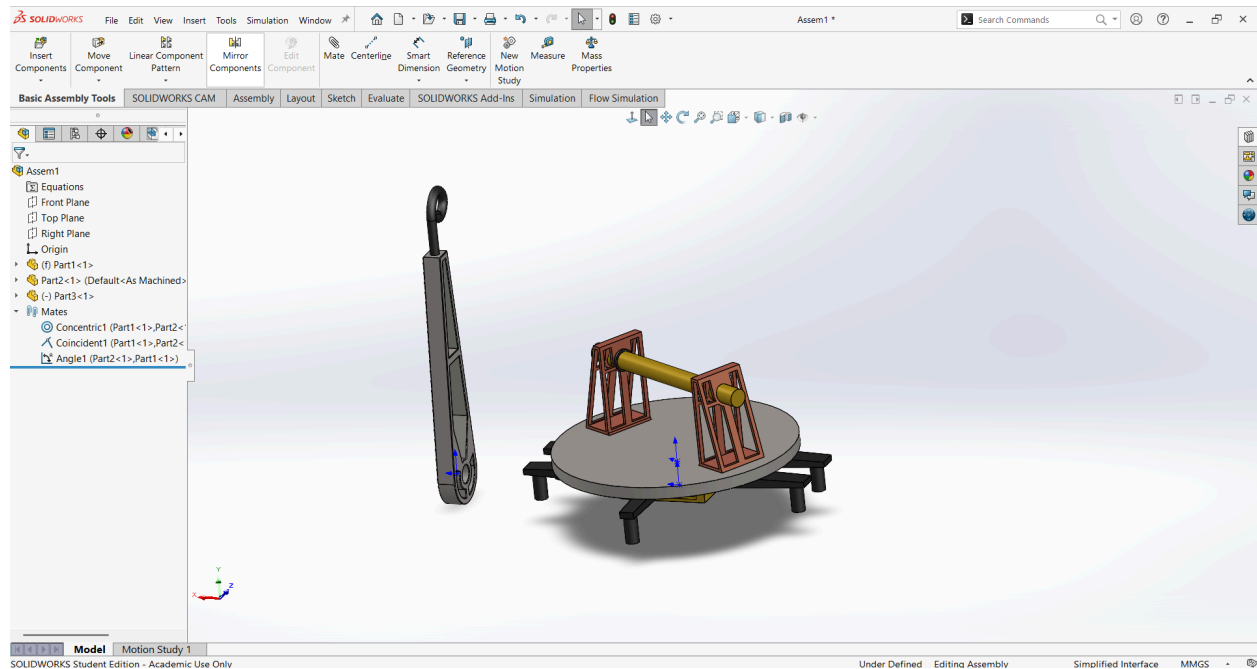
Now select **Angle** from the **Mate Type** options. Set the angle value to **60°**. Now there is a **60° angle** between these two faces.



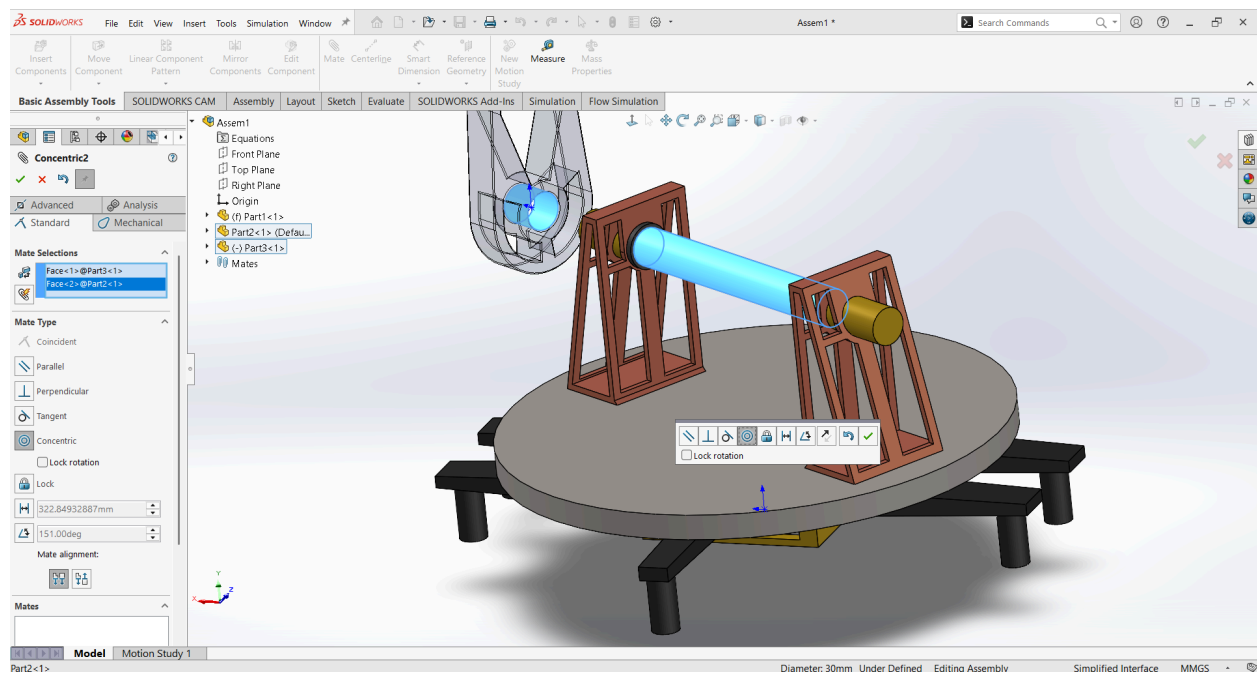
Click **OK**. Now let's insert the third component. Go to **Insert Components** and select **Component 3**.



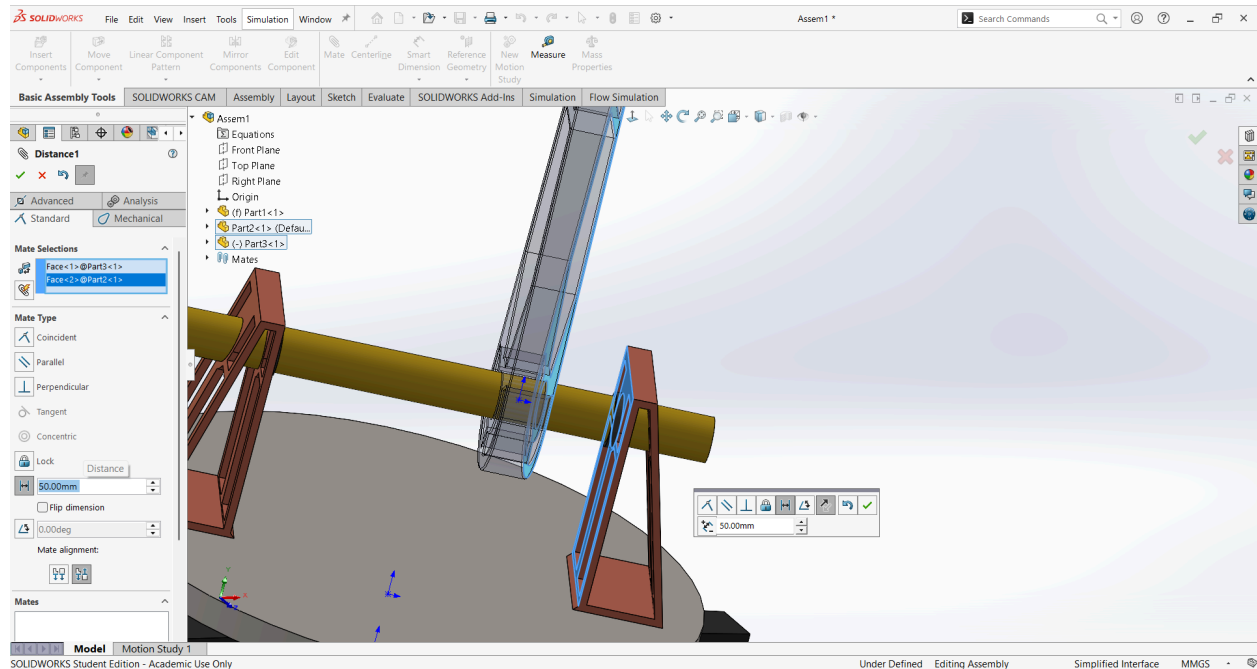
Place it anywhere in the graphics area.



Now we need to connect **Component 3**. Go to **Mate**. Select the shaft on **Component 2**, then select the hole on **Component 3**. SOLIDWORKS automatically suggests a **Concentric** mate. Click **OK**.

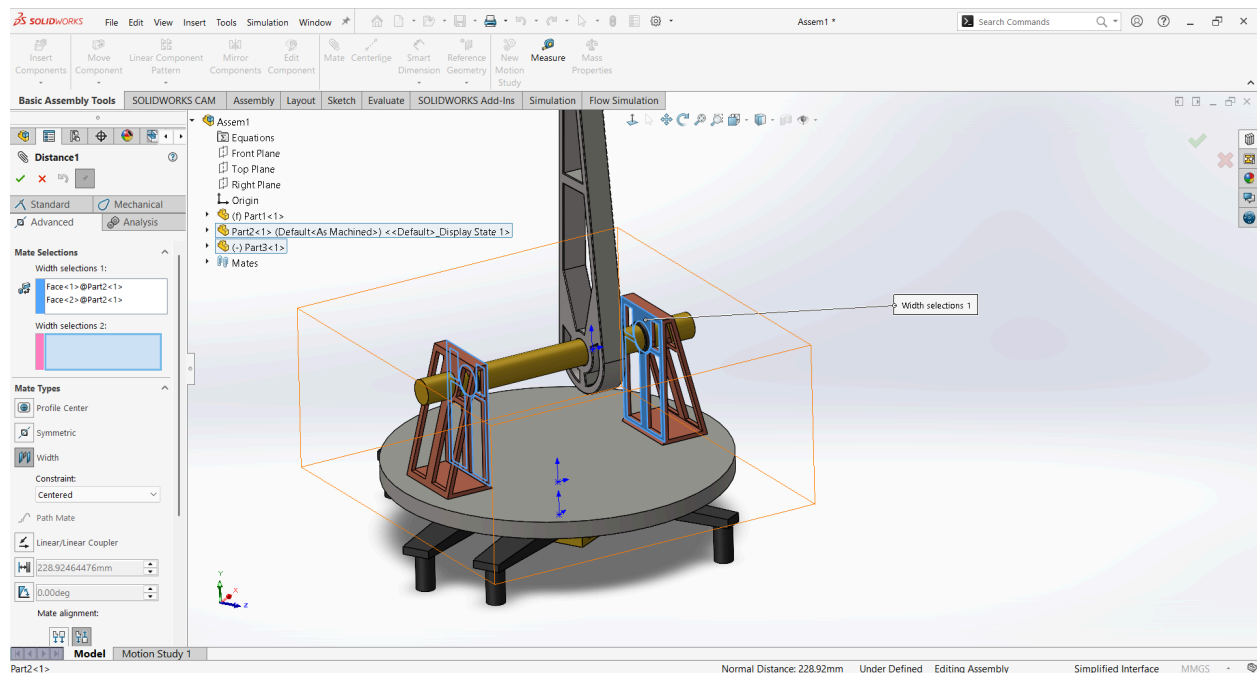


Now go to **Mate** again. Select the face on **Component 2**, then select the face on **Component 3**. Select the **Distance** mate type and set the distance to **50 mm**. Now there is a **50 mm** distance between these two faces.

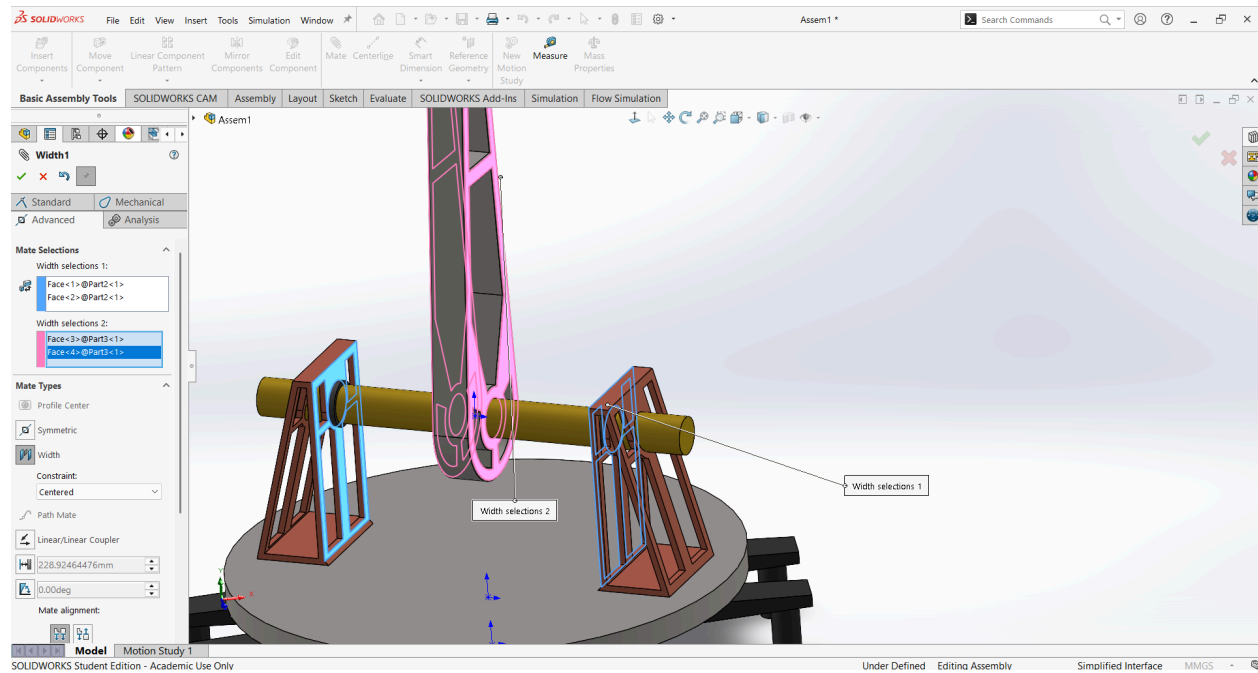


Delete previous Angle mate.

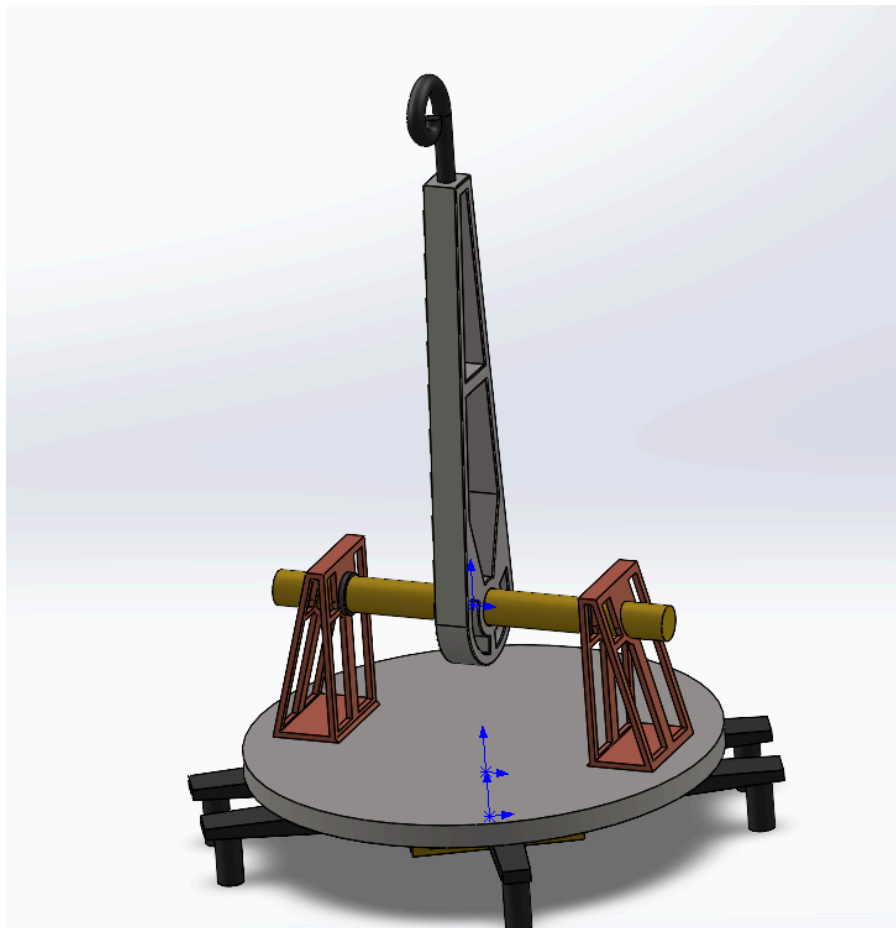
Now let's show one **Advanced Mate**. Go to **Advanced Mates** and select **Width**. In the first selection box, select the two faces of **Component 2**, shown in blue.



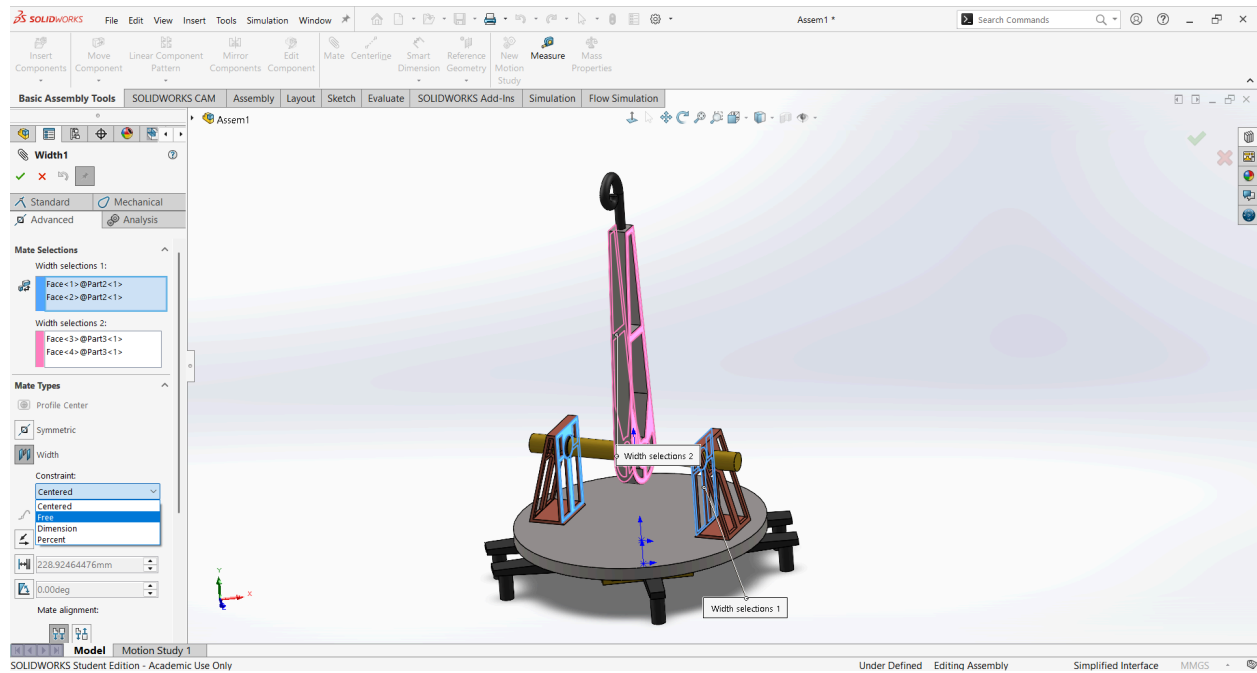
In the second selection box, select the two faces of the other component, shown in pink.



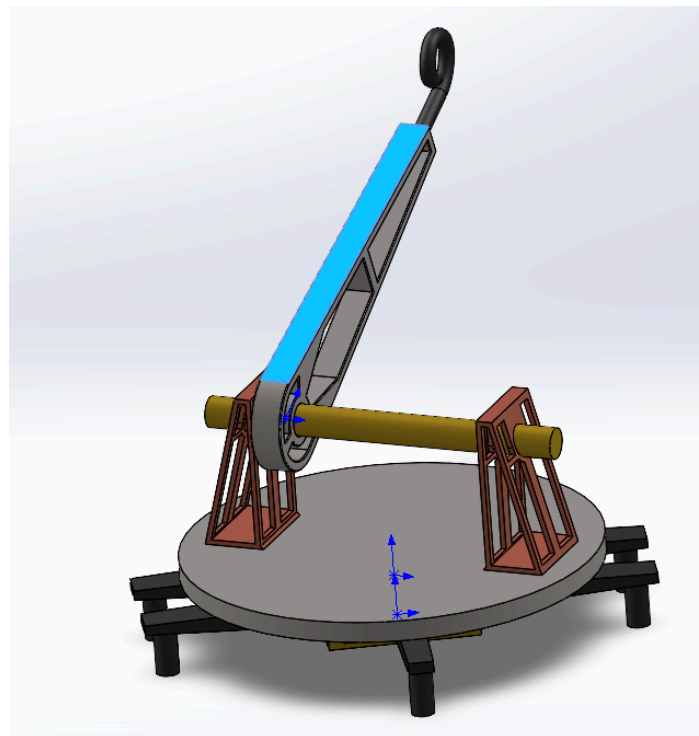
Click **OK**. Now the part is positioned exactly in the middle of the shaft.



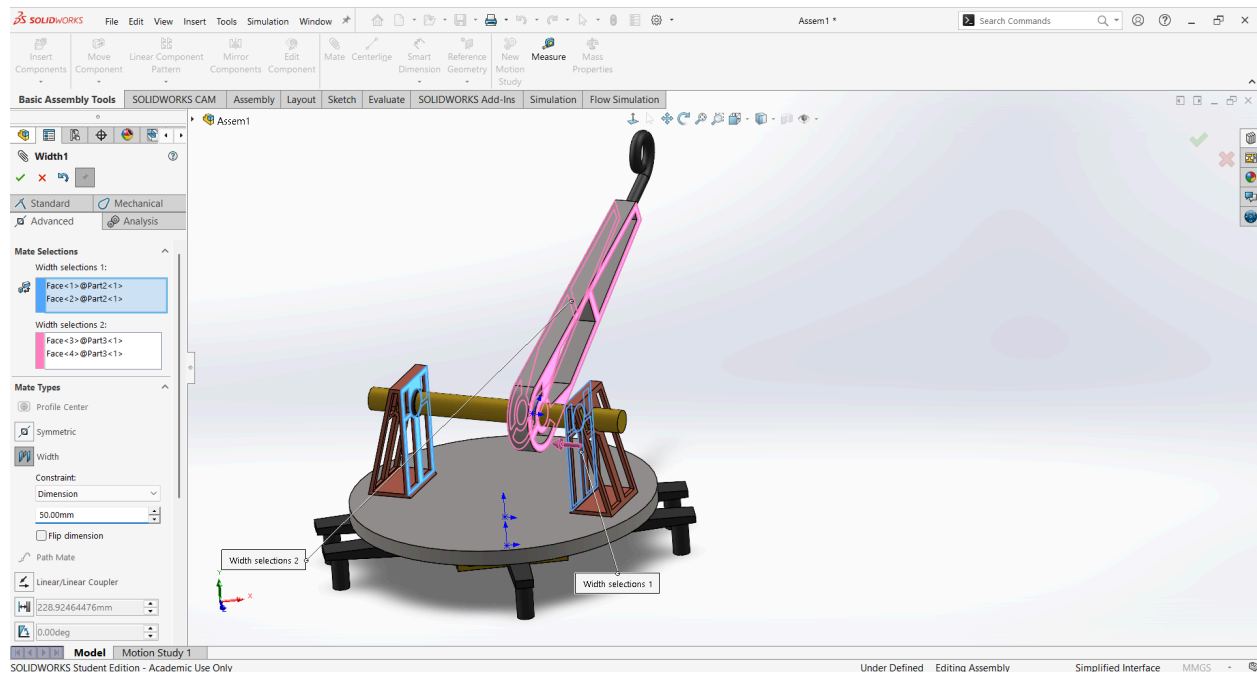
Now go back and edit the **Width** mate we created. Change the **Constraint Type**. Right now, the constraint type is set to **Centered**. Click on it and change it to **Free**.



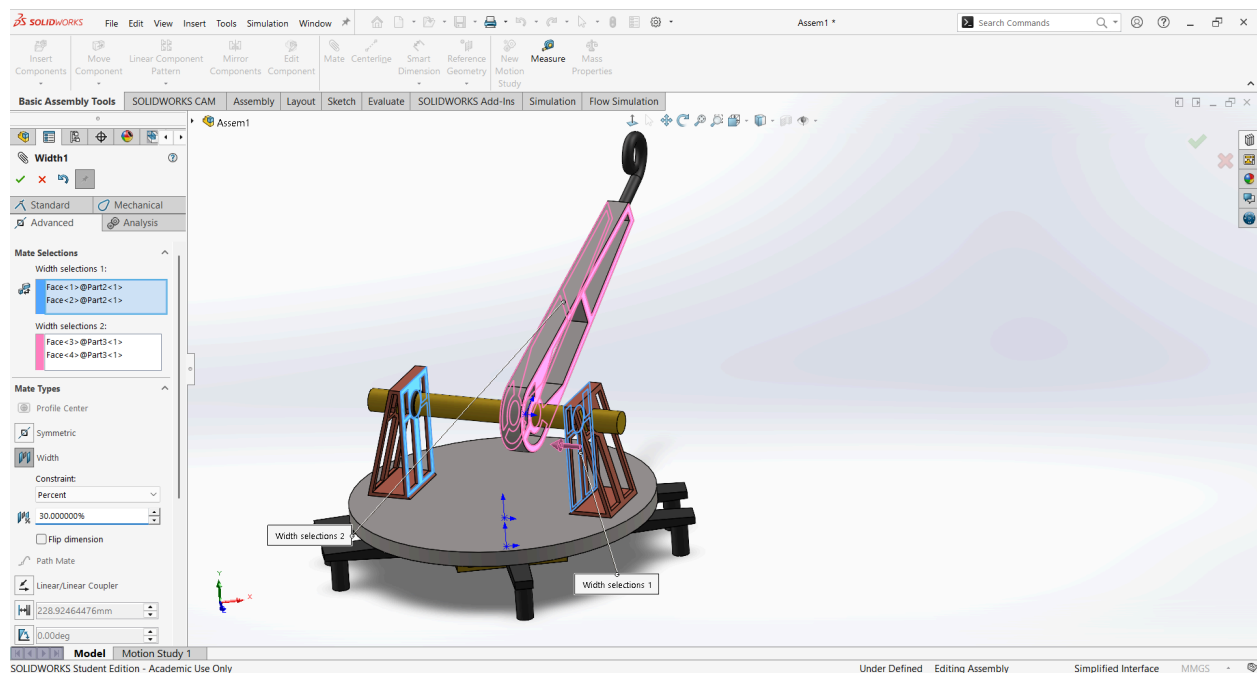
Now we can move **Component 3** freely along the shaft of **Component 2**.



Let's change the **Constraint Type** again. This time, set it to **Dimension** and enter a value of **50 mm**. Now **Component 2** and **Component 3** are positioned with a **50 mm** distance between them.

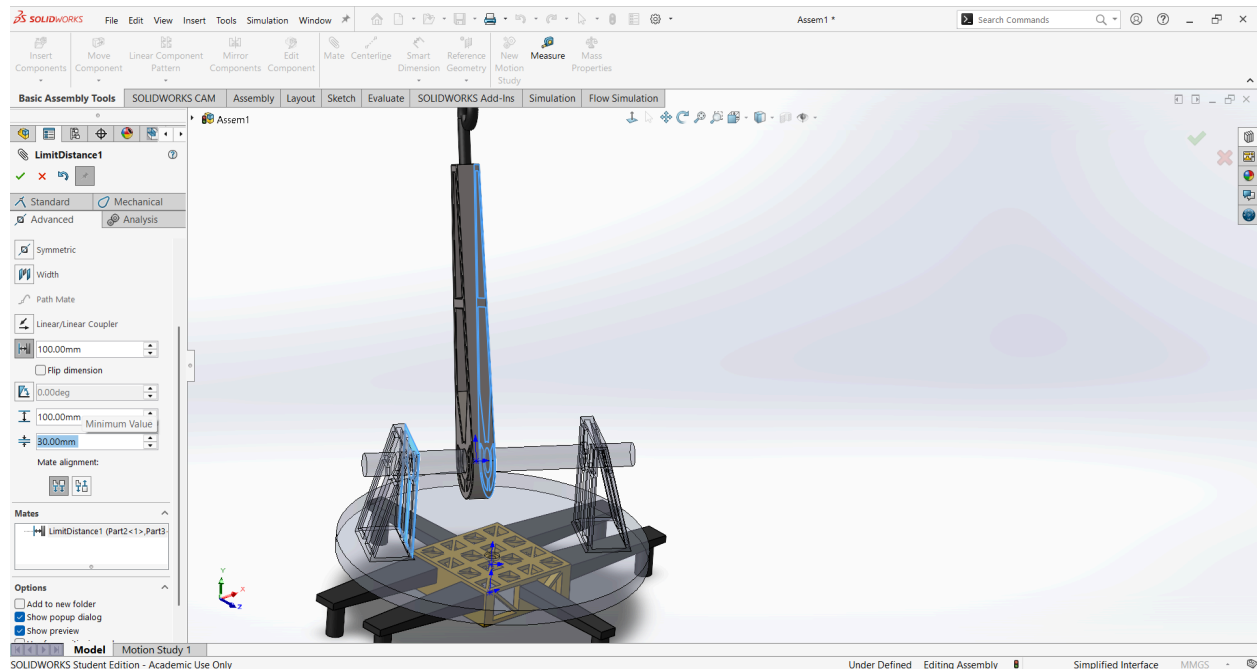


Now change the **Constraint Type** to **Percent**. With this option, you can define the position of **Component 3** along the shaft as a percentage. Set the value to **30%**.

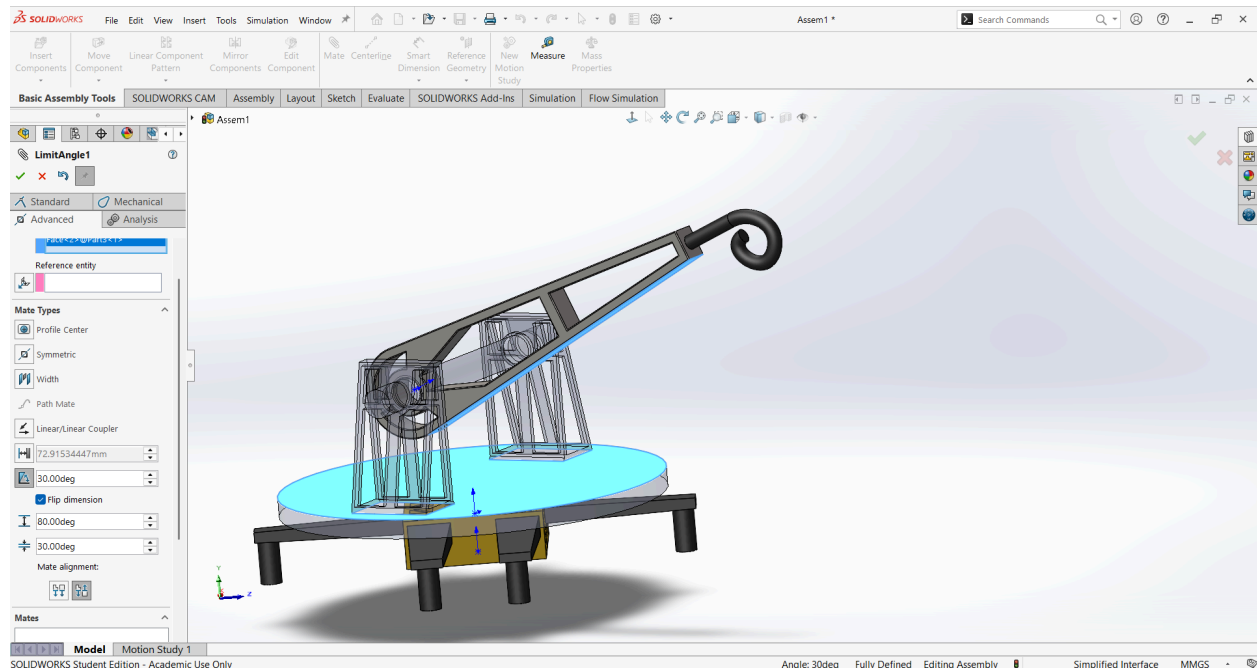


Click **OK**.

Now let's show the second **Advanced Mate** type: **Distance**. With this mate, we can define a minimum and maximum distance, creating a movement range. Select the two faces on **Component 2** and **Component 3**. Set the **Maximum Distance** to **100 mm** and the **Minimum Distance** to **30 mm**. Now **Component 3** can move only within this distance range.



We can do the same thing with an **Angle** mate. Select the faces on **Component 2** and **Component 3**, as shown. Define the **Minimum Angle** as **30°** and the **Maximum Angle** as **80°**. Now the component can rotate only within this angle range. If needed, change **Mate Alignment** or use **Flip Dimension** to position the components correctly.



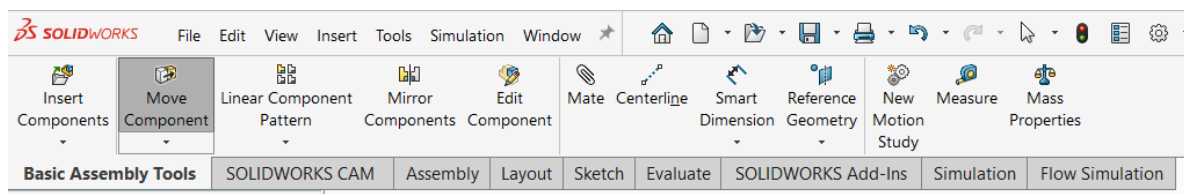
29. Move Component

What it does: Moves components inside an assembly.

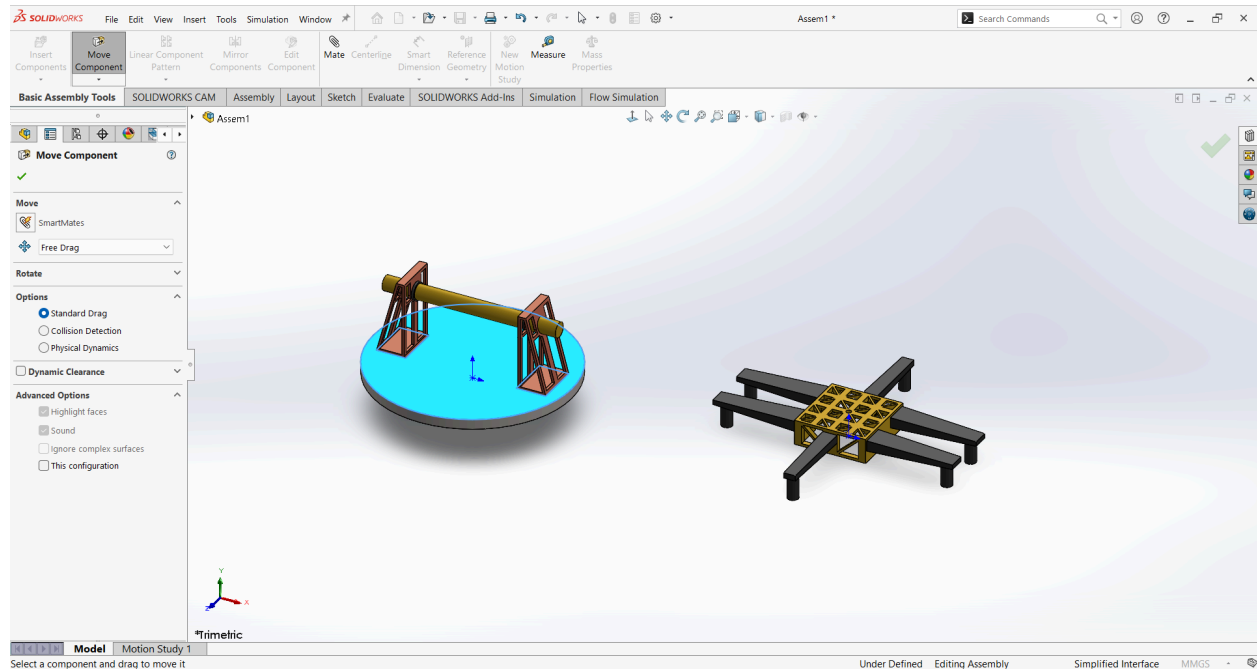
Use it for: Positioning parts and testing remaining degrees of freedom.

How it works:

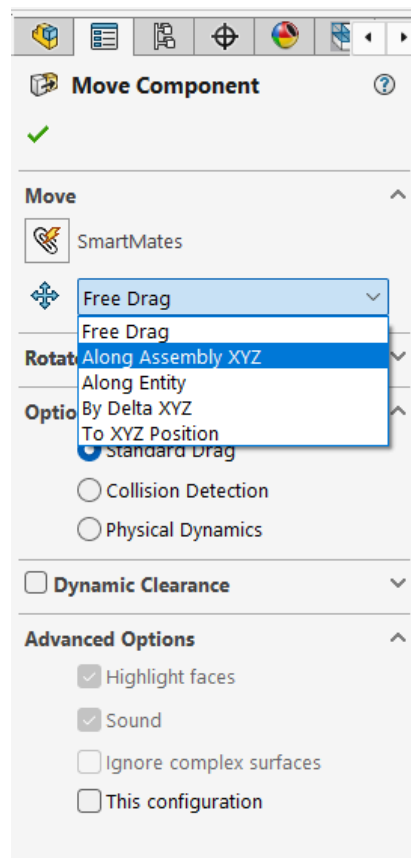
The second component is not fixed, so we can move it freely. Go to the **Basic Assembly Tools** tab and select **Move Component**.



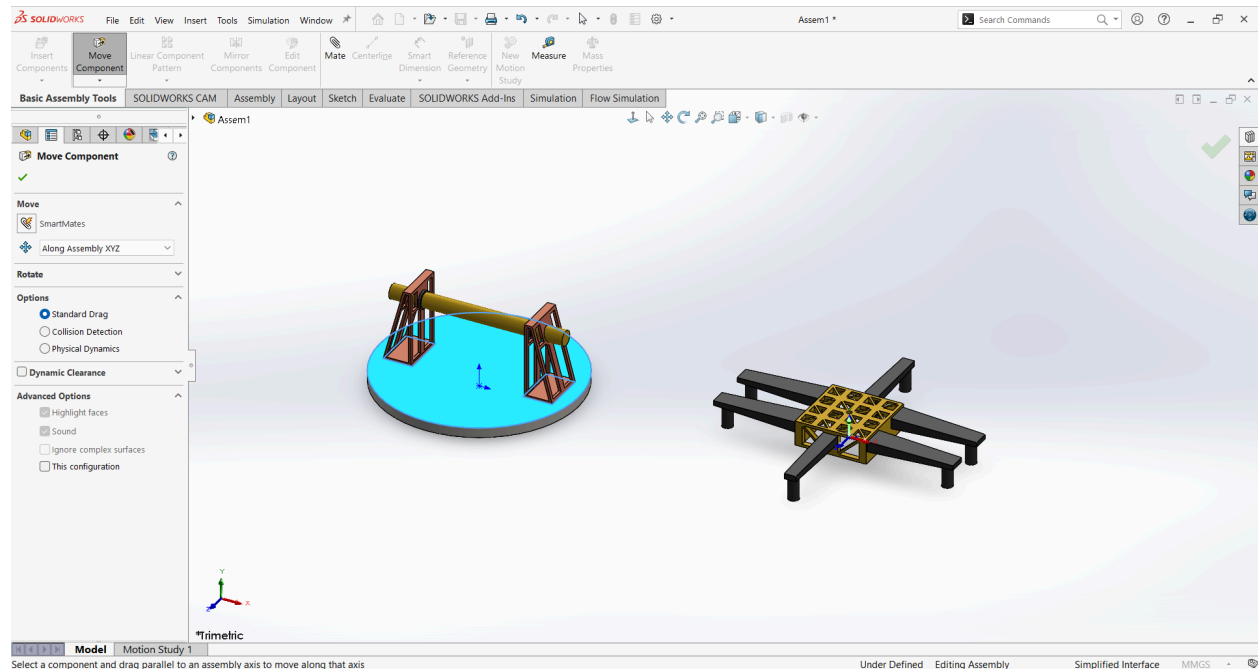
Property Manager appears. By default, the **Move** option is set to **Free Drag**. Now, if you click and drag the component with the mouse, you can move it freely in the graphics area.



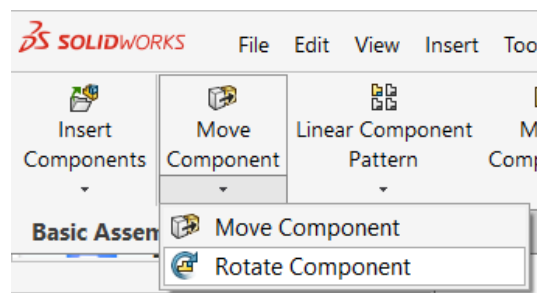
There are also a few other options under **Move**. Select **Along Assembly XYZ**.



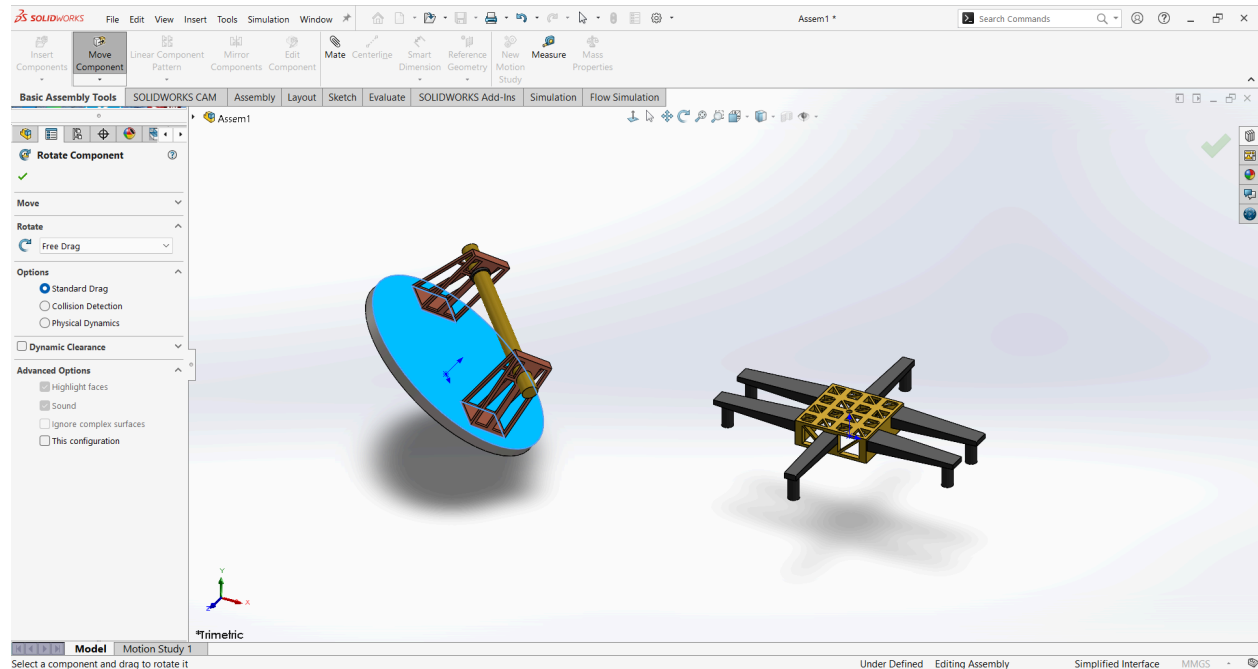
Now you can move the component along the assembly **X**, **Y**, and **Z** directions.



Go to the **Move Component**.



Click the small arrow below the tool, then select **Rotate Component**. Now you can rotate the component using the mouse.



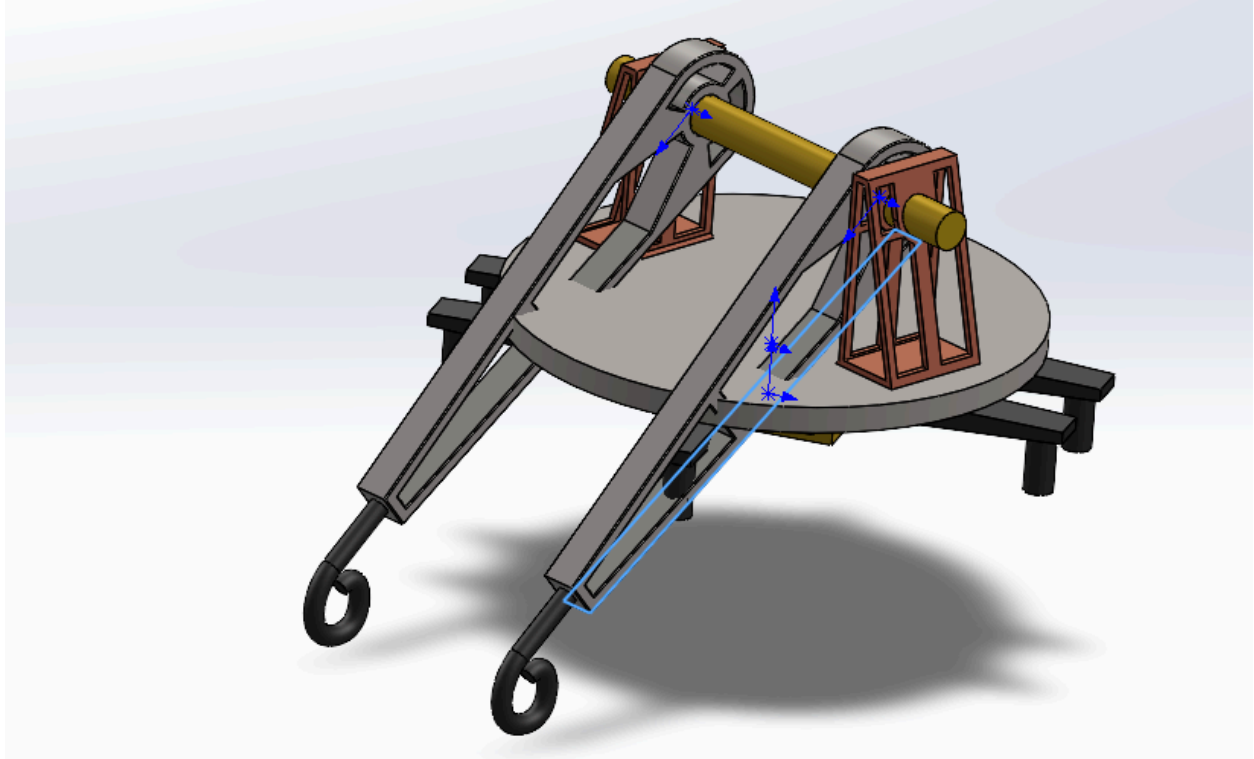
30. Collision Detection

What it does: Checks whether assembly components physically overlap.

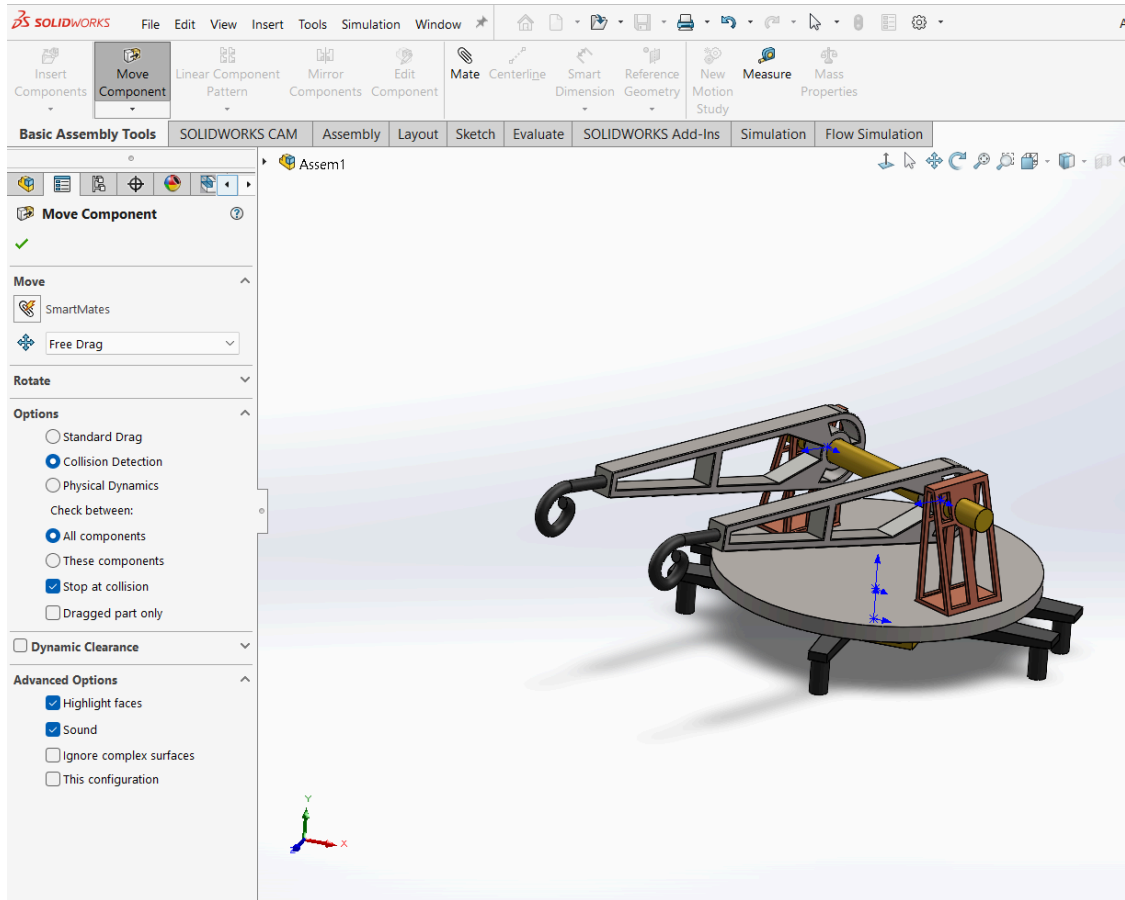
Use it for: Finding design problems before manufacturing or assembly.

How it works:

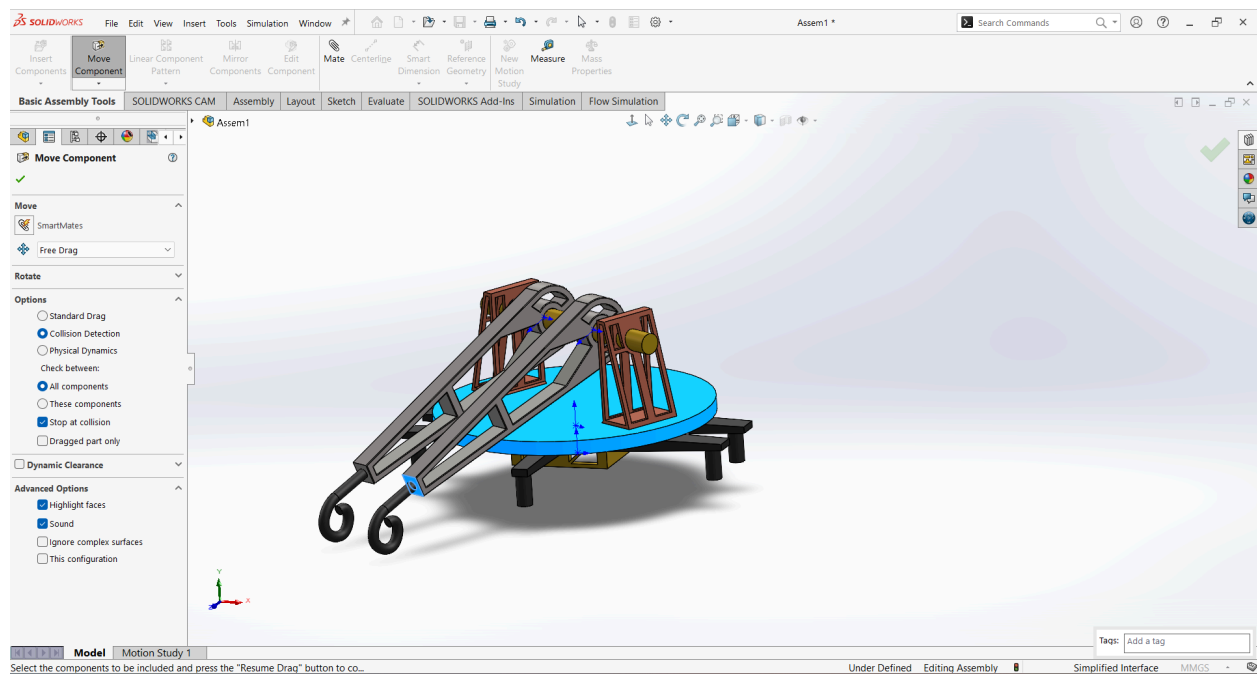
Now you can see that when you move **Component 3**, it goes through the material of **Component 1**. We want to be able to detect the collision.



Go to the **Move Component** tool. Under **Options**, select **Collision Detection**, and also make sure **Stop at collision** is checked.



Now if you rotate component 3 you can see that it will hit component 1.



4. DRAWINGS

31. Orthographic View

What it does: Shows a 3D model from different 2D directions, such as Front, Top, and Side.

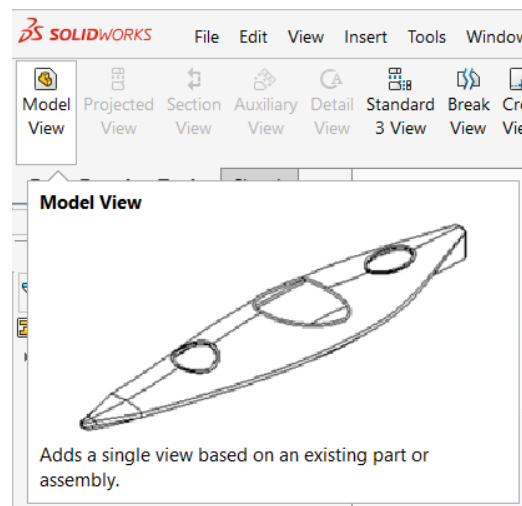
Use it for: Clearly communicating the shape, size, and features of a part in an engineering drawing.

How it works:

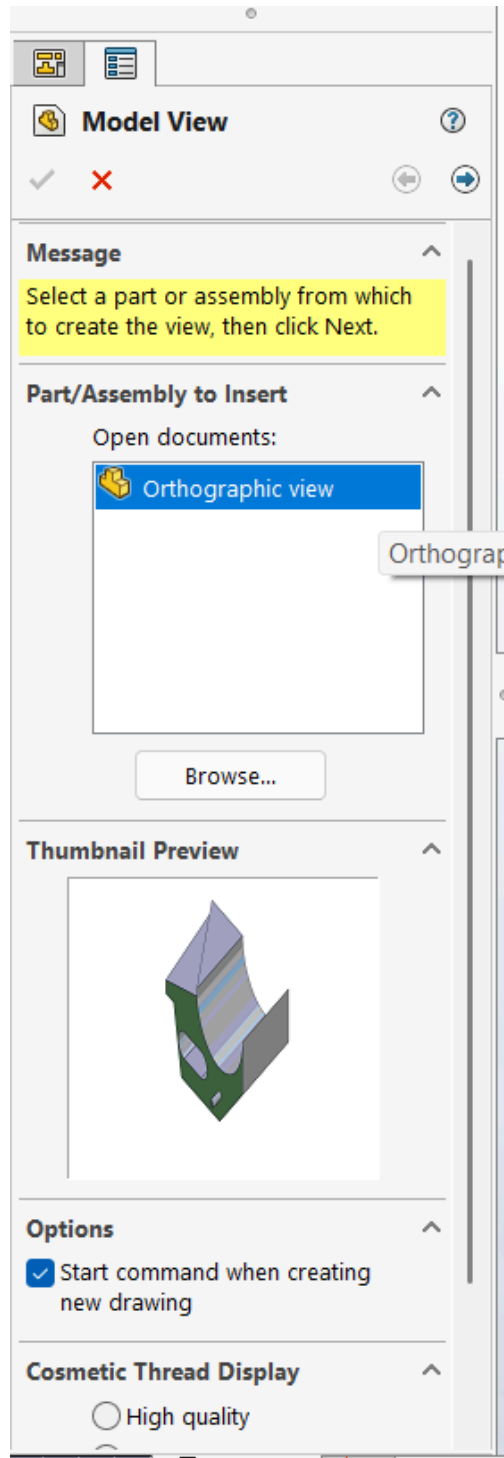
Orthographic views are 2D drawing views created from a 3D model, such as the **Front**, **Top**, **Right**, and **Isometric** views.

These views help clearly show the shape and size of the part from different directions.

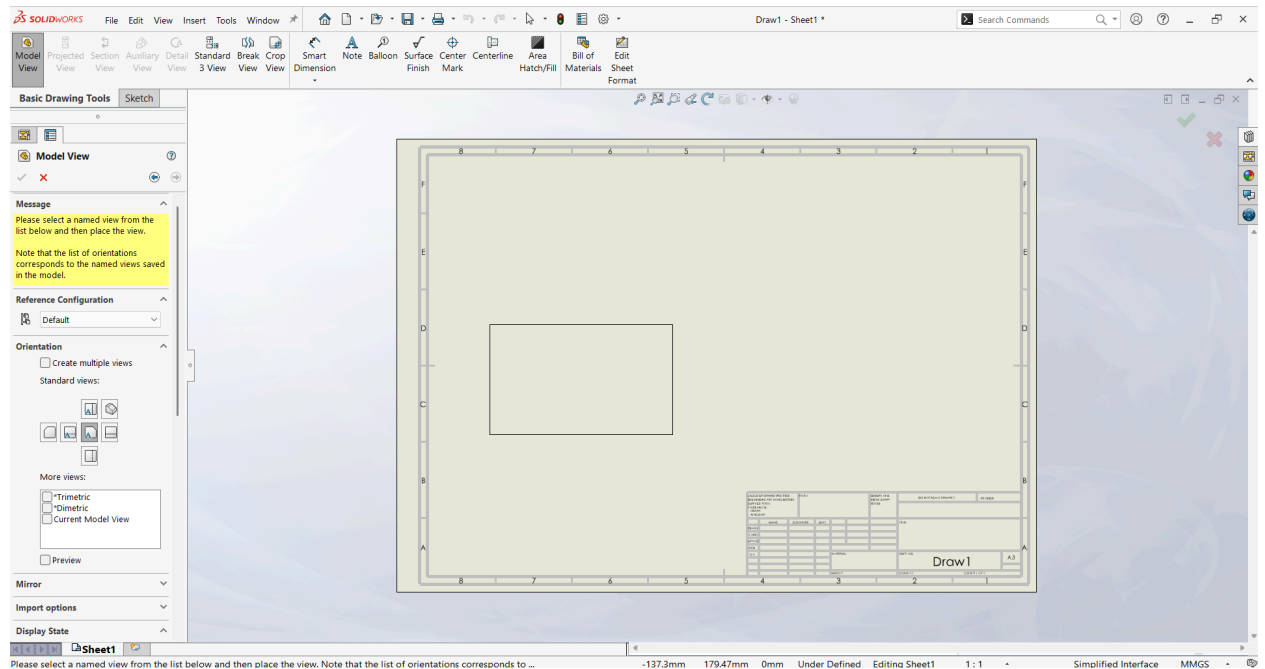
To begin, click the **Model View** tool.

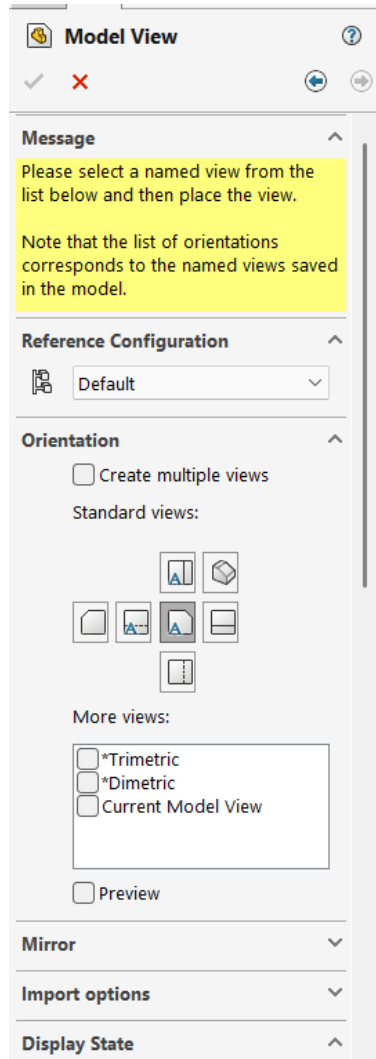


Double-click the part or assembly you want to use. If you do not see it in the list, click **Browse** and select the file manually.

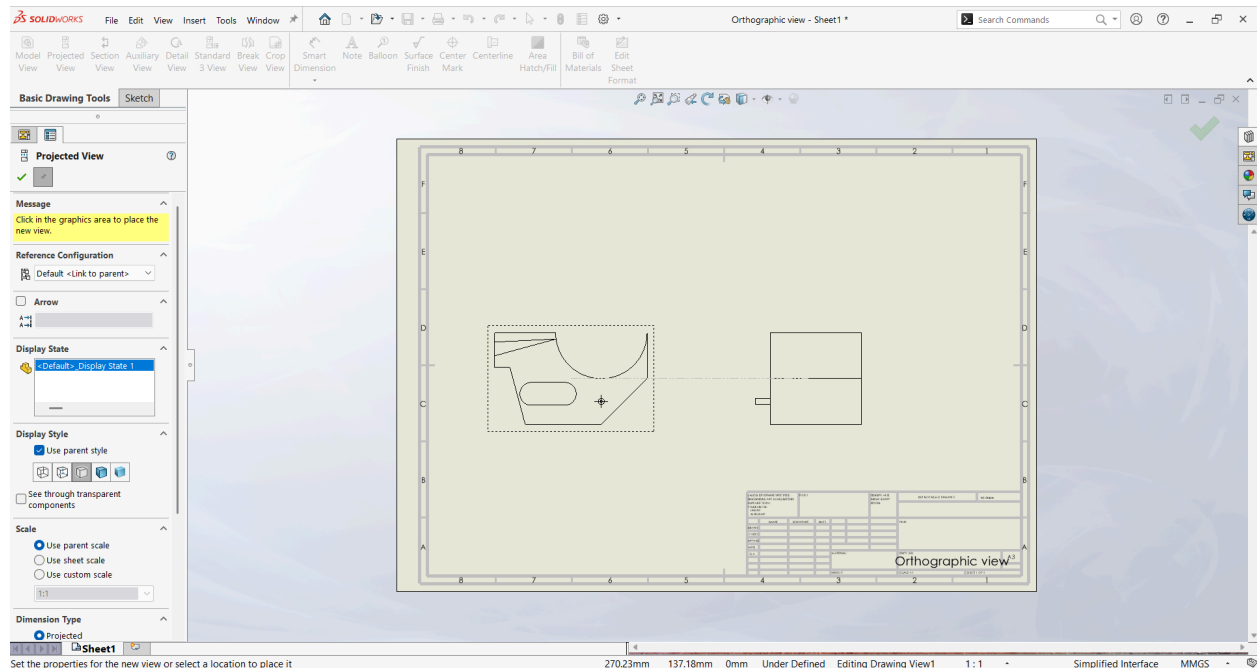


Move the mouse over the drawing sheet. On the left side, in the **PropertyManager**, you can choose which view you want to place under **Orientation**.

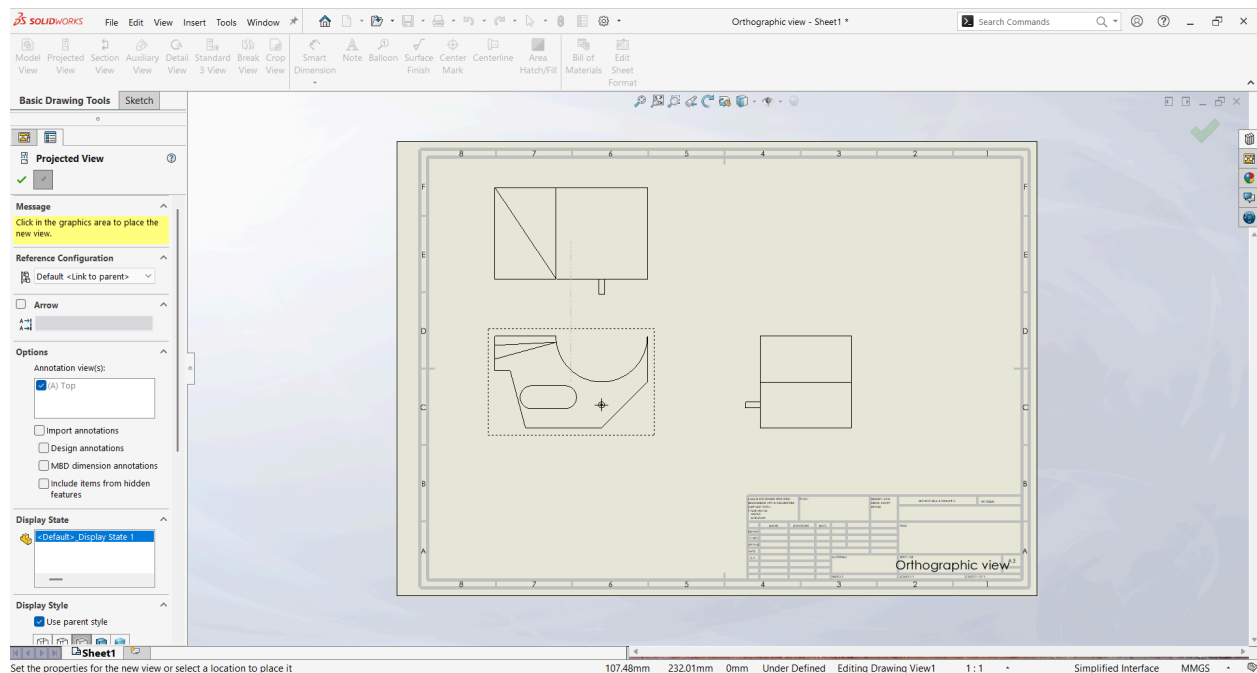




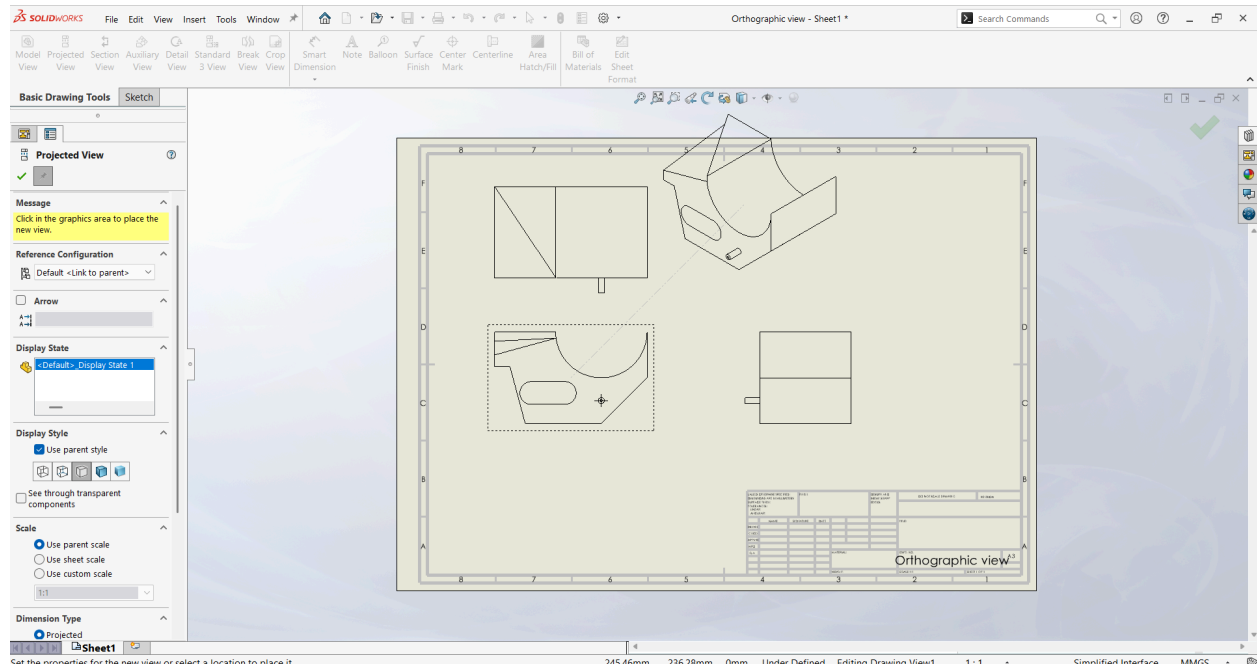
Click on the drawing sheet to place the first view. If you move the mouse to the left, SOLIDWORKS automatically creates another projected view. Click again to place the second view.



When you move the mouse upward, SOLIDWORKS automatically creates another projected view. Click again to place this view on the drawing sheet.

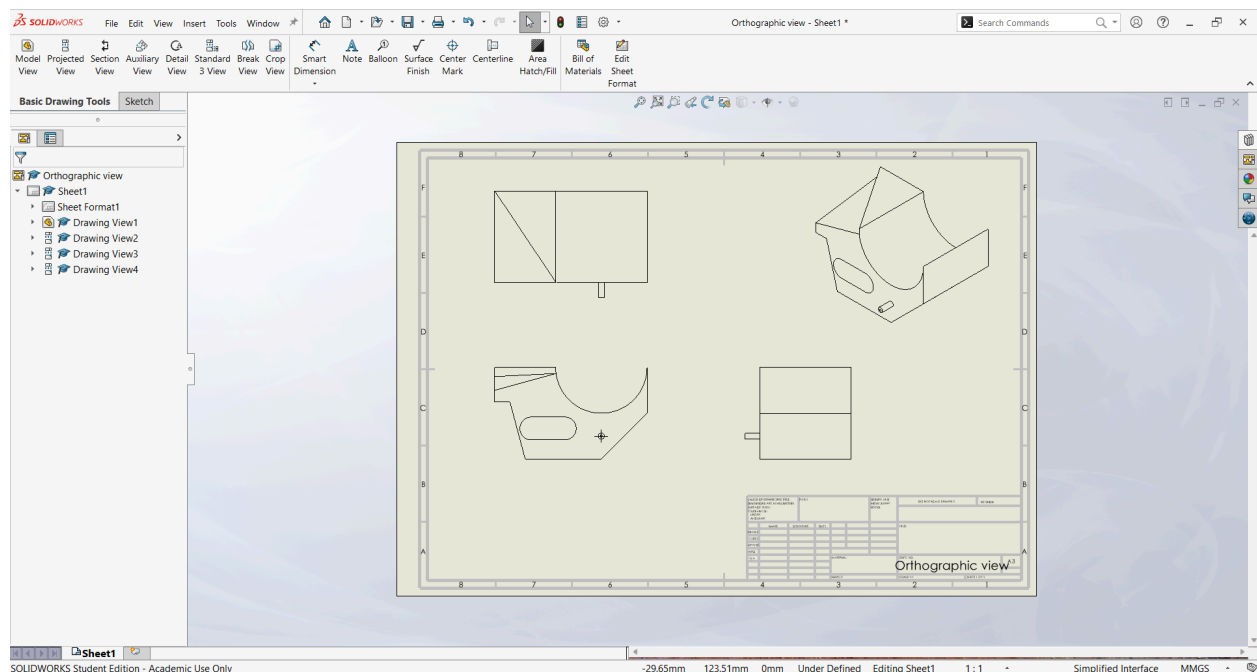


When you move the mouse diagonally to the side, SOLIDWORKS automatically creates an **Isometric View**. Click again to place the view on the drawing sheet.



To finish with view creation just press Esc.

To move a drawing view, click and hold the view, then drag it with the mouse. You can position the view anywhere you want on the drawing sheet.



32. Projected View

What it does: Shows a 3D model from different 2D directions, such as Front, Top, and Side.

Use it for: Clearly communicating the shape, size, and features of a part in an engineering drawing.

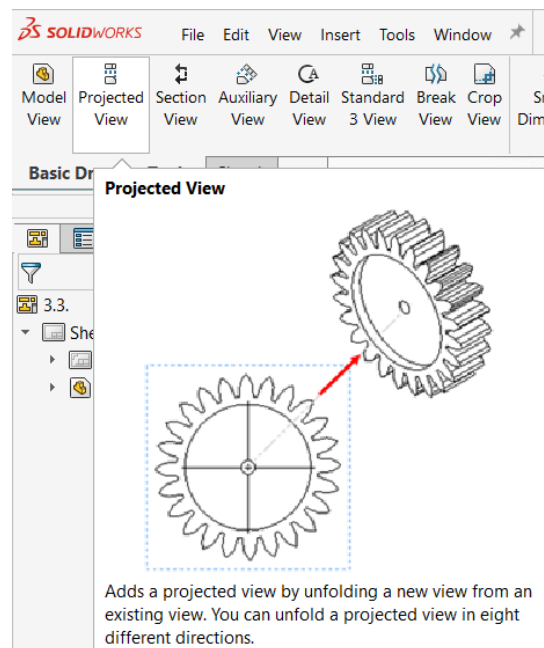
How it works:

A **Projected View** is a drawing view created from an existing view.

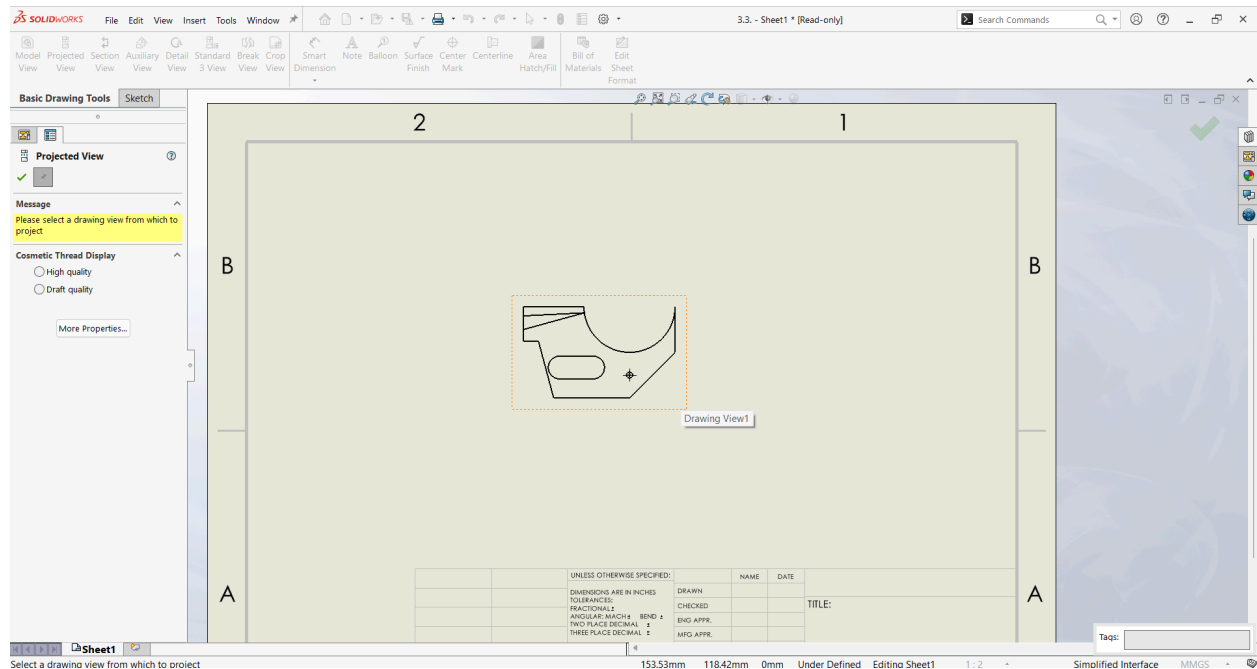
It shows the same part or assembly from another direction, such as the top, right, left, or bottom side.

Projected views are useful because they help show the complete shape of the model in a clear 2D drawing.

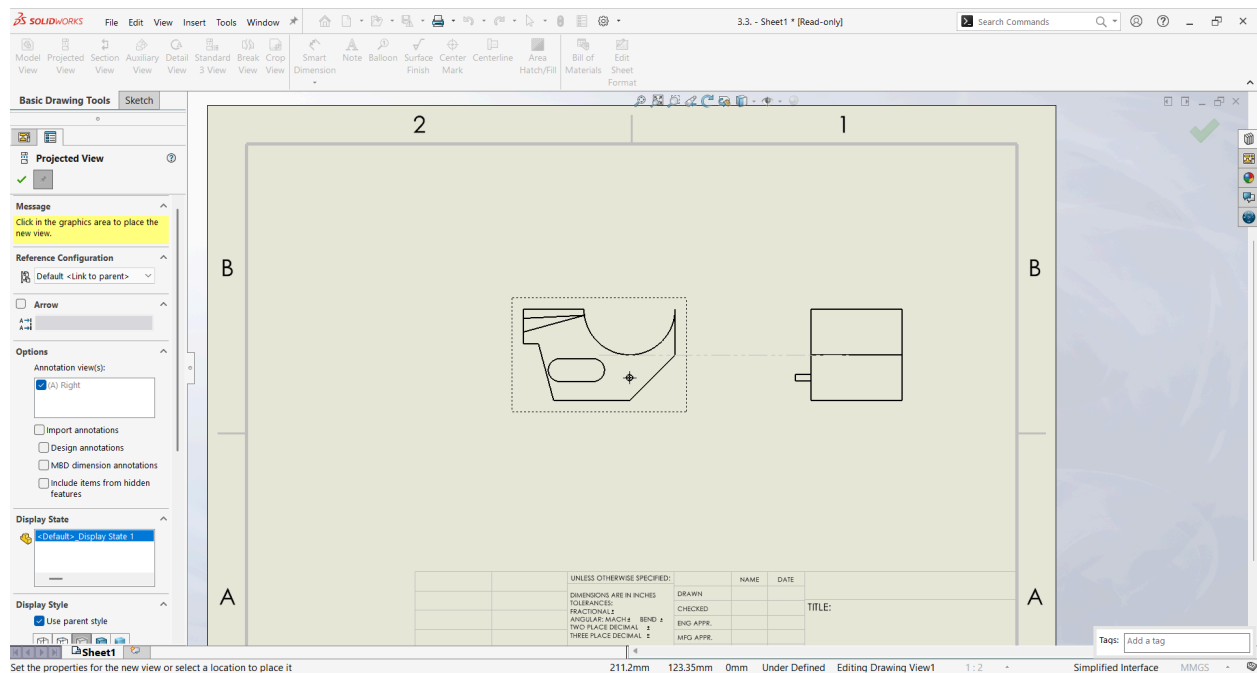
To create a **Projected View**, go to the **Basic Drawing Tools** tab and select **Projected View**.



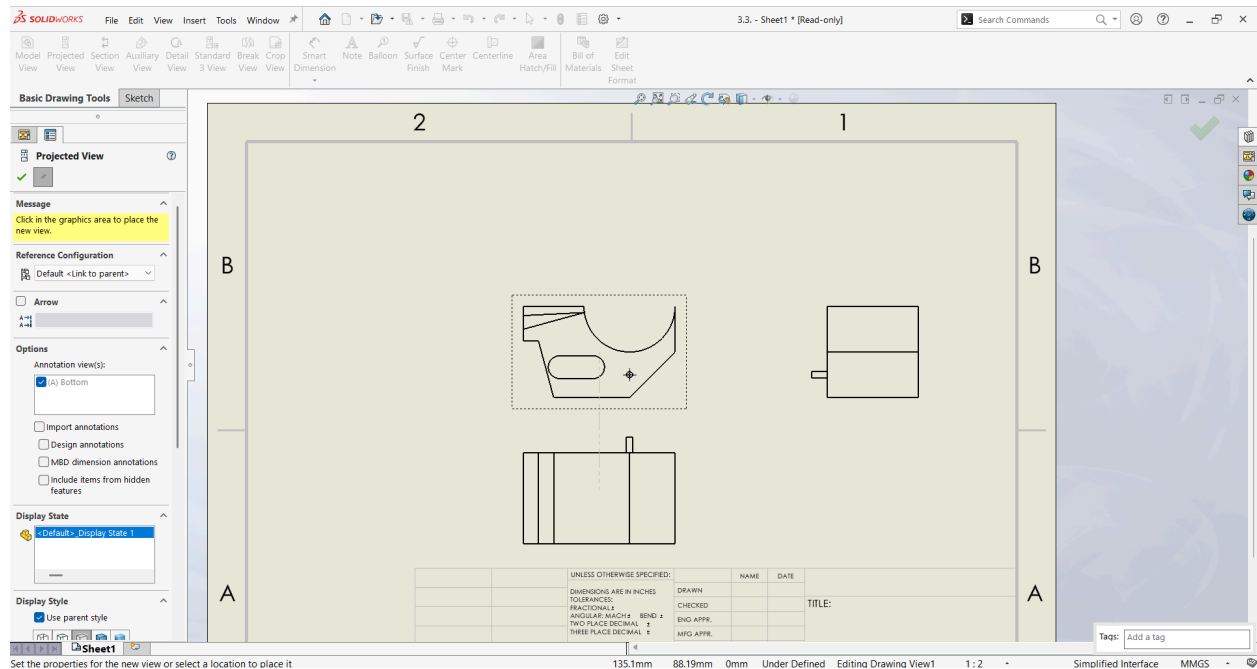
Click on the existing drawing view.



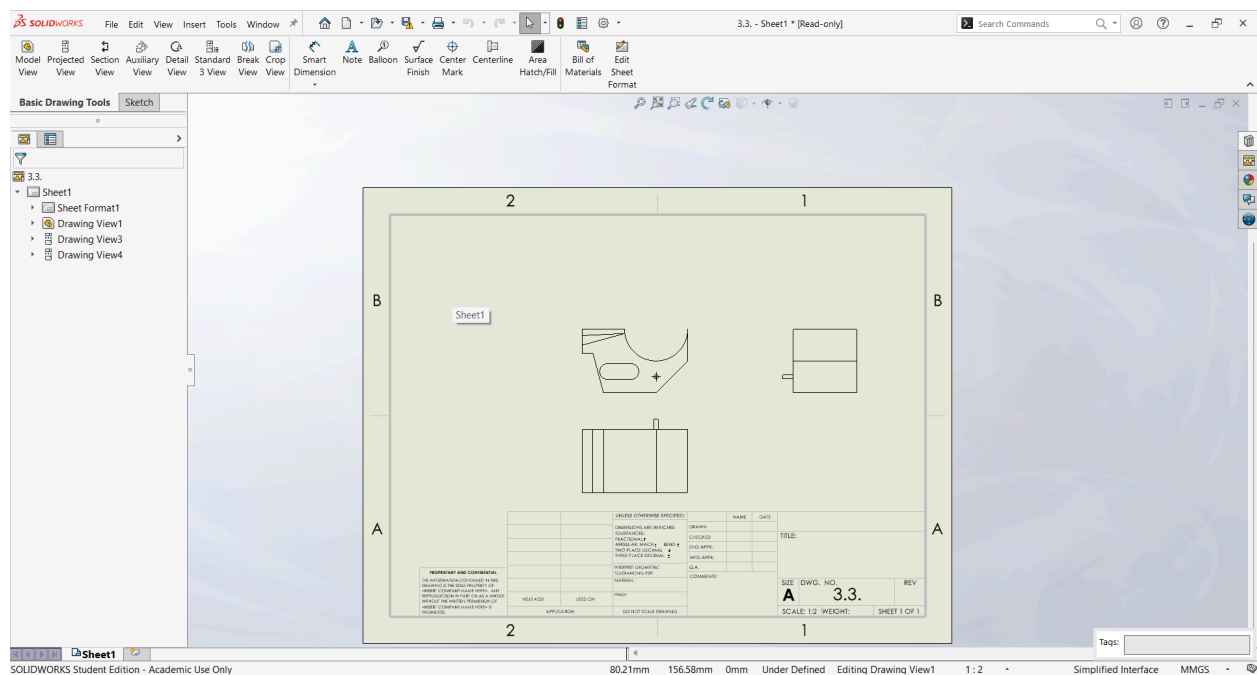
Move the mouse to the right side of the existing view. Then click on the drawing sheet to place the projected view.



Move the mouse downward and click to place the second **Projected View**.



When you are finished, click **OK**.



33. Section View

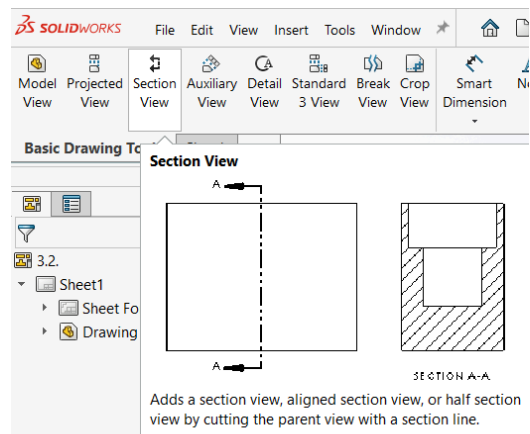
What it does: Cuts through the model virtually to reveal internal geometry.

Use it for: Showing internal features that are difficult to communicate with standard views.

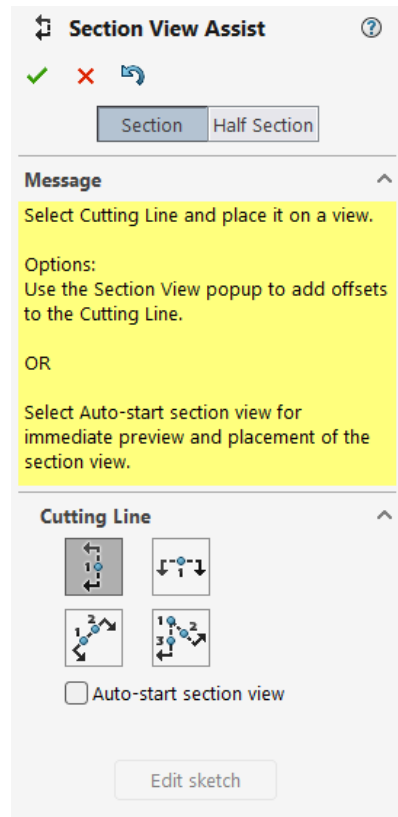
How it works:

A **Section View** is a drawing view that shows the inside of a part or assembly. It is created by cutting through the model with a section line. This allows us to see internal features that are not clearly visible from the outside, such as holes, slots, pockets, or hidden geometry.

To create a **Section View**, go to the **Basic Drawing Tools** tab and select **Section View**.

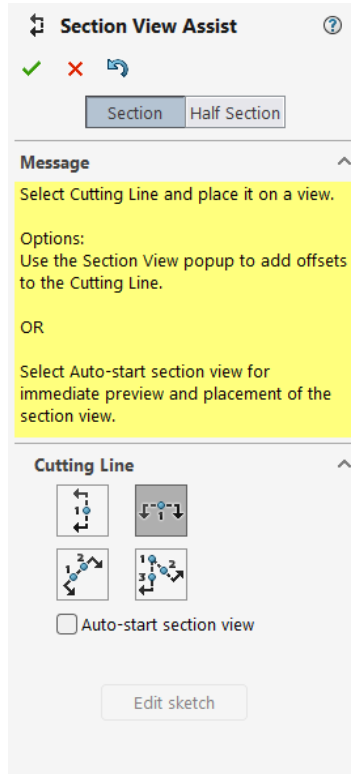


First, choose the cutting line direction. In this example, select a **Vertical** cutting line.

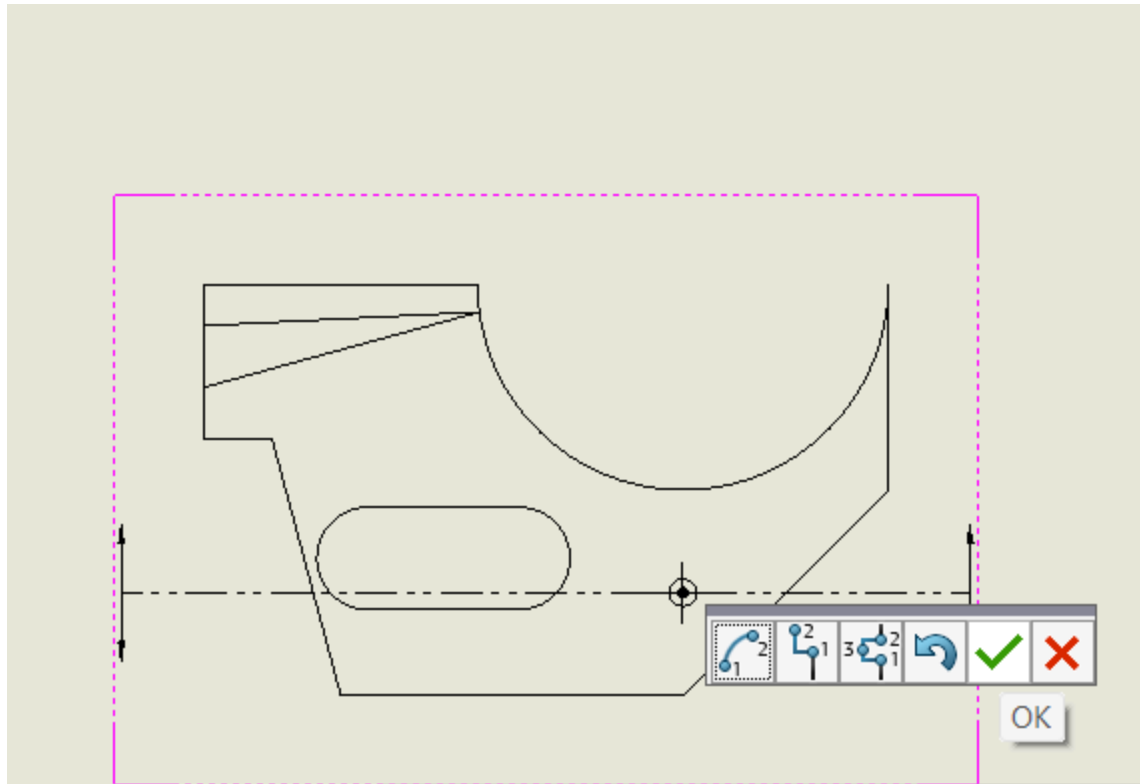


Click on the drawing view to place the vertical cutting line, as shown.

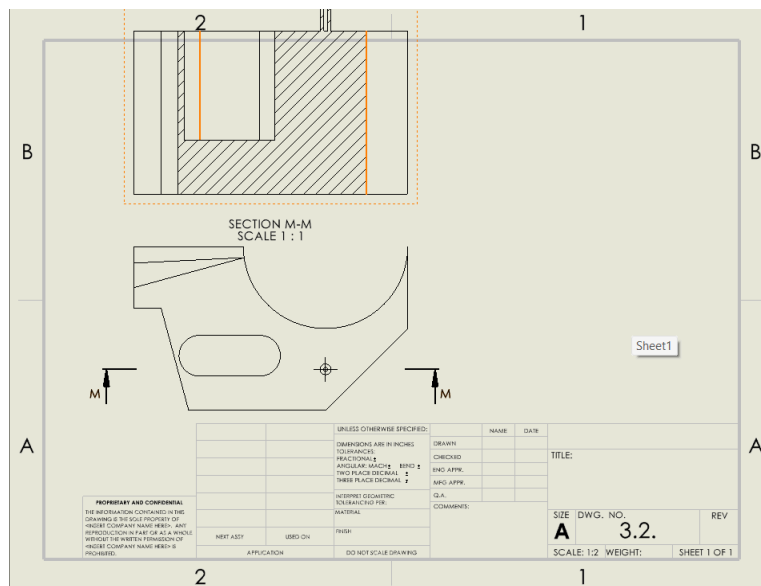
Select the **Section View** tool again. This time, choose a **Horizontal** cutting line.



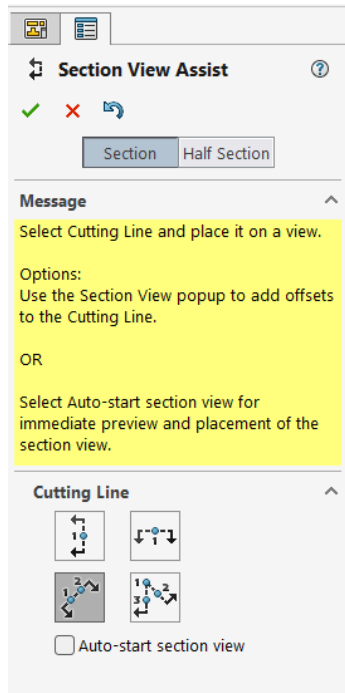
Position the cutting line as shown and click on it.



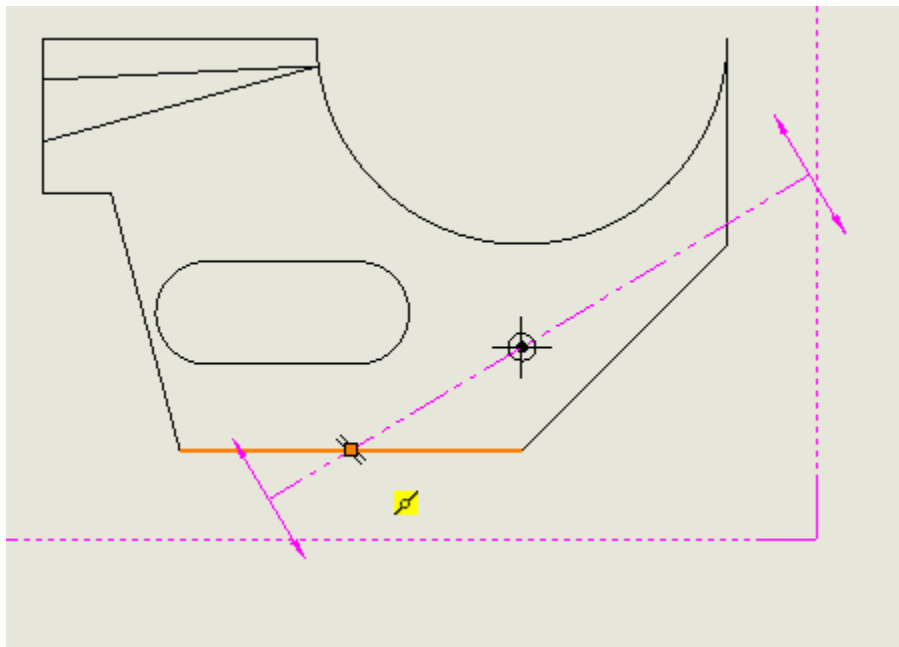
Click on the green check mark to confirm it. Move the mouse upward to create the **Section View**. Click on the drawing sheet to place the view.



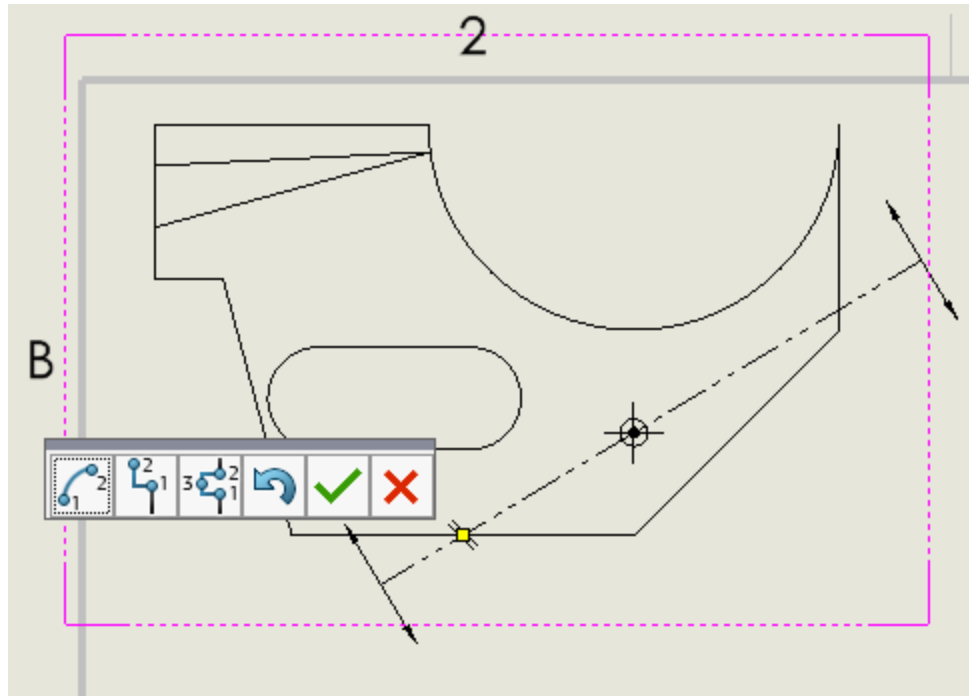
Go to the **Section View** tool again. This time, choose an **Aligned** cutting line. Move the mouse over the drawing view to position the cutting line.



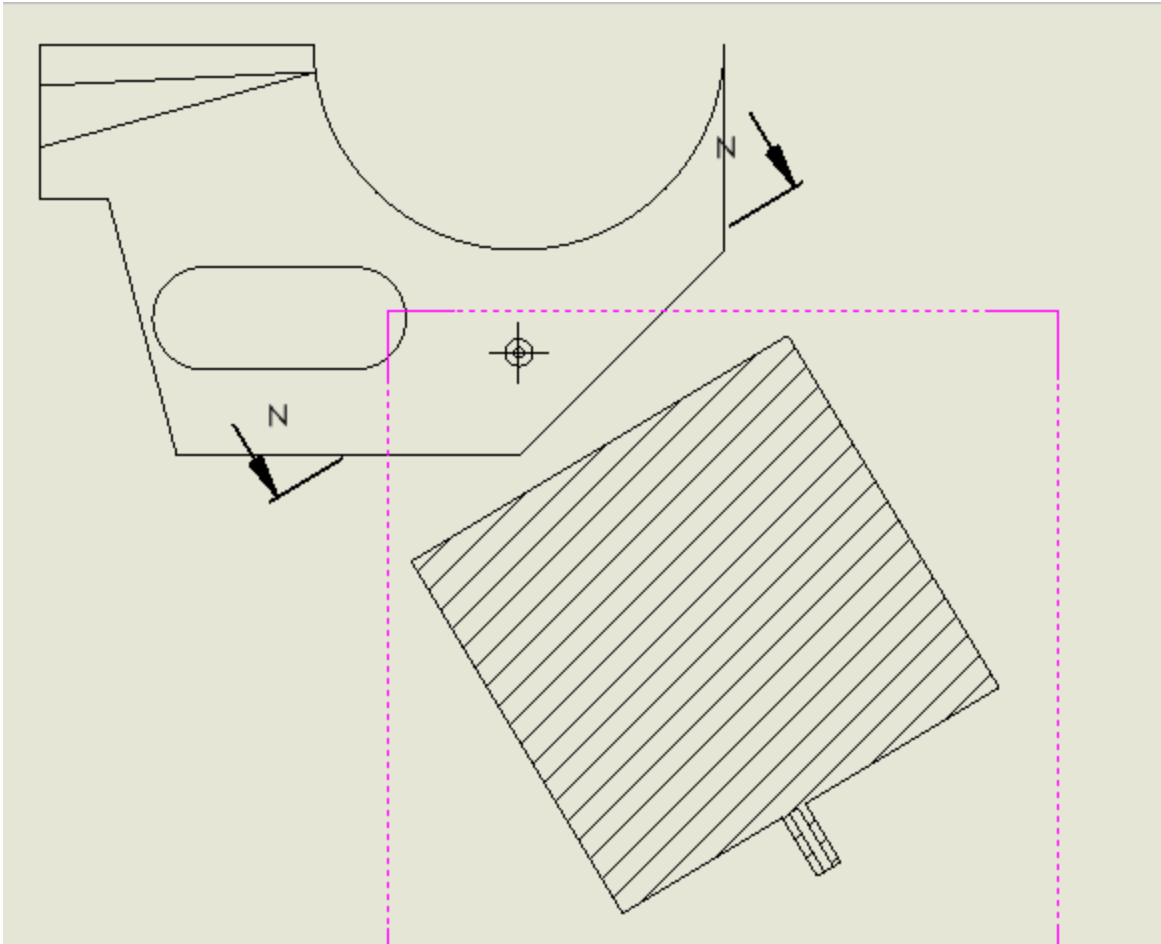
To position the cutting line, you need to select two points. Click to select the first point in the center of the small circle.



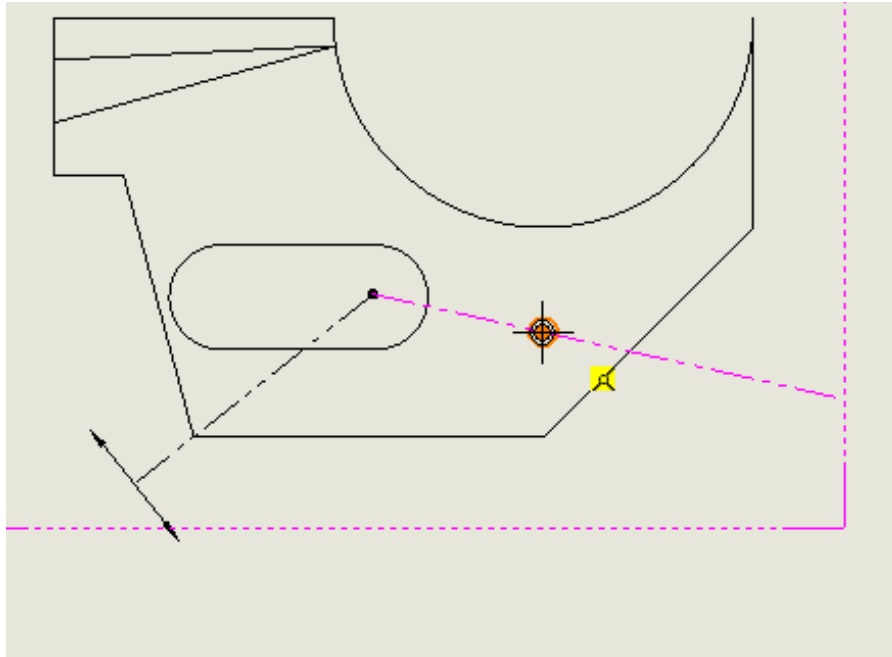
Click again to select the second point on the horizontal line.



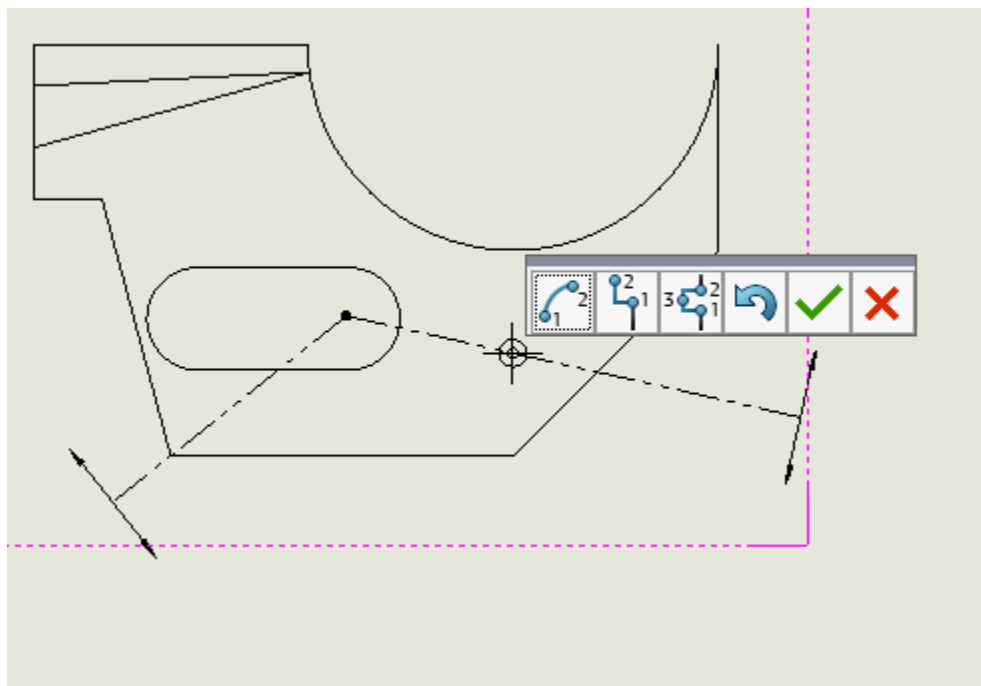
Click on the green check mark to confirm it. Move the mouse to create the section view. Click on the drawing sheet to place the view.



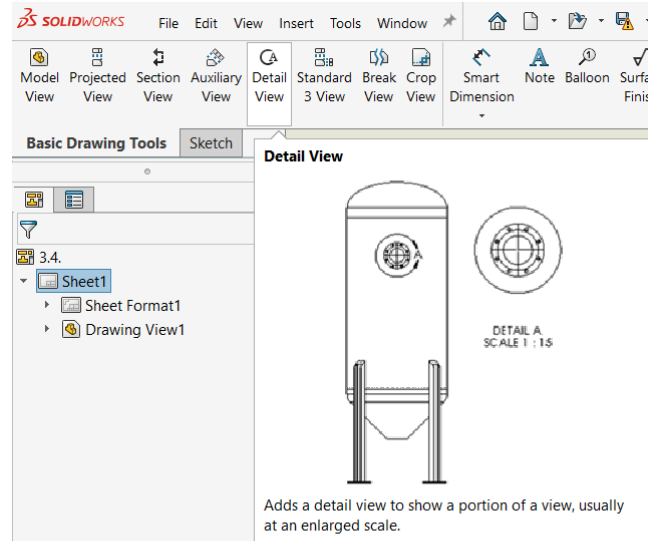
The last option is **Offset Cutting Line**. Go to the drawing view and define three points. First, select the center point.



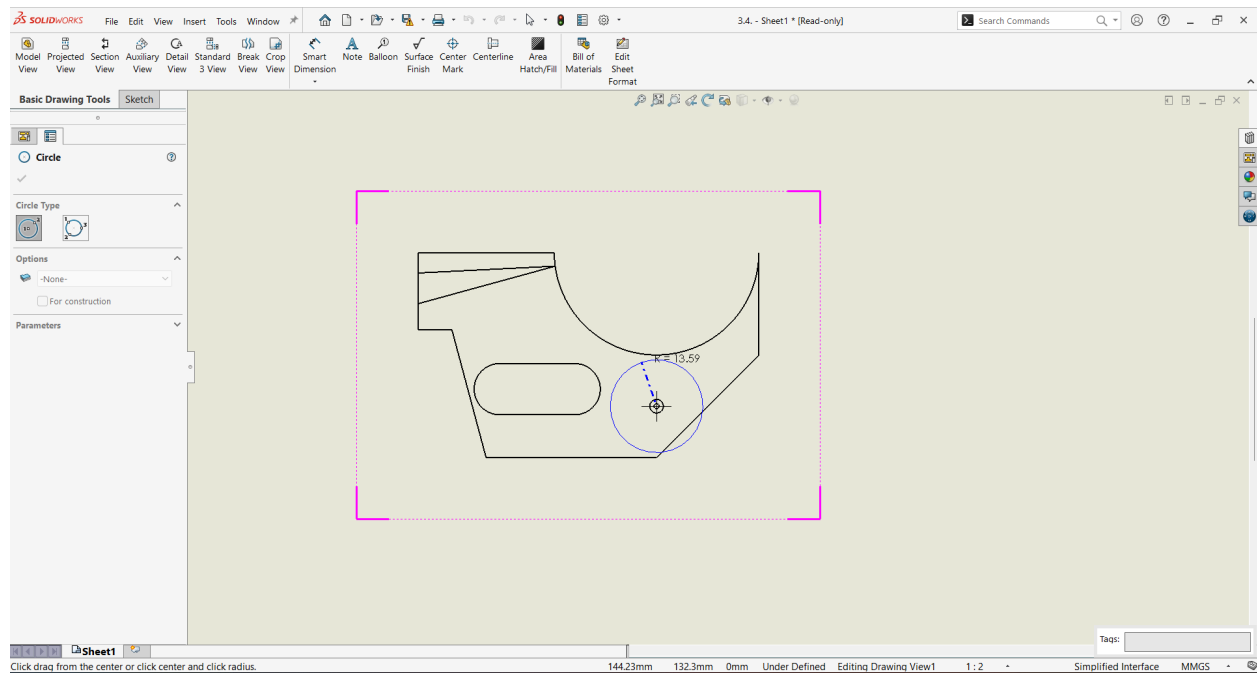
The remaining two points define the angle of the cutting line.



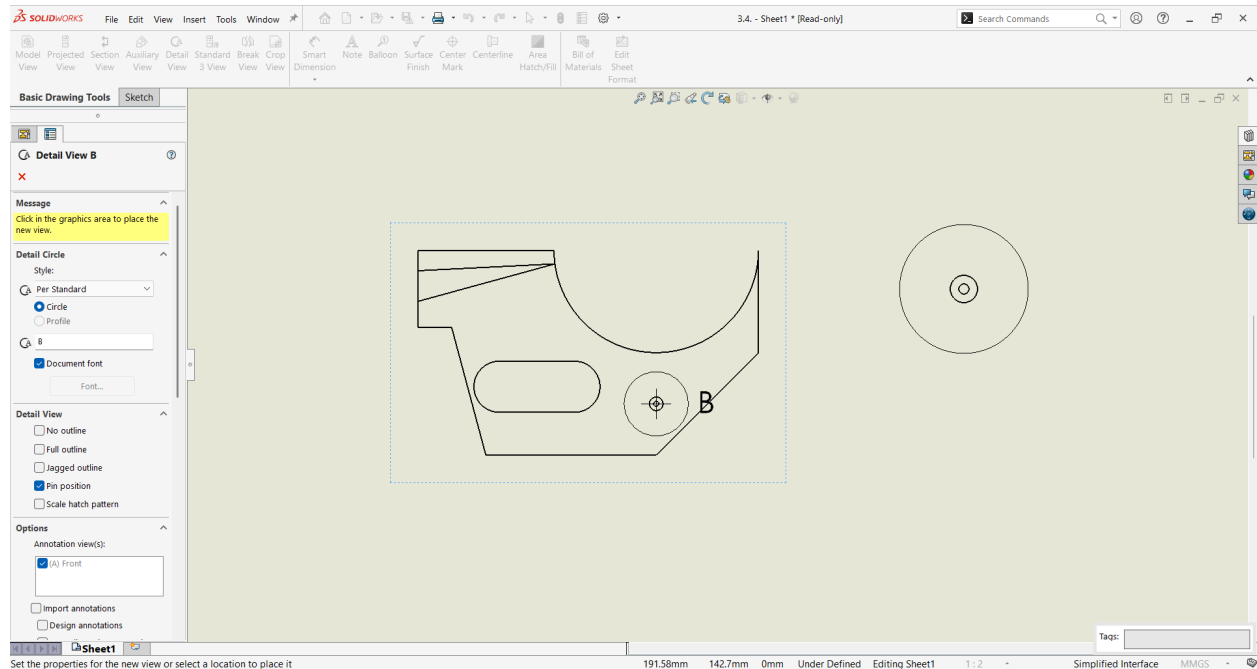
Click on the green check mark to confirm it. Move the mouse to drag out the view. Click on the drawing sheet to place it.



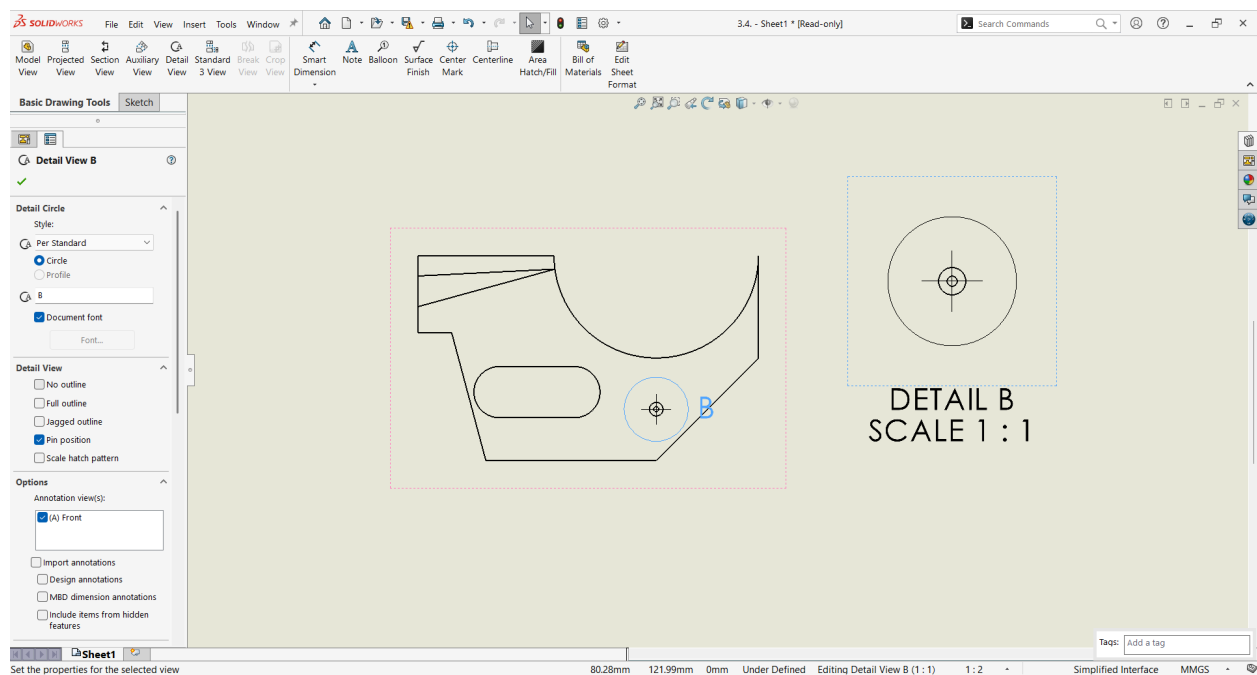
The next step is to draw a circle around the area you want to enlarge. Make sure the detail you want to show is completely inside the circle.



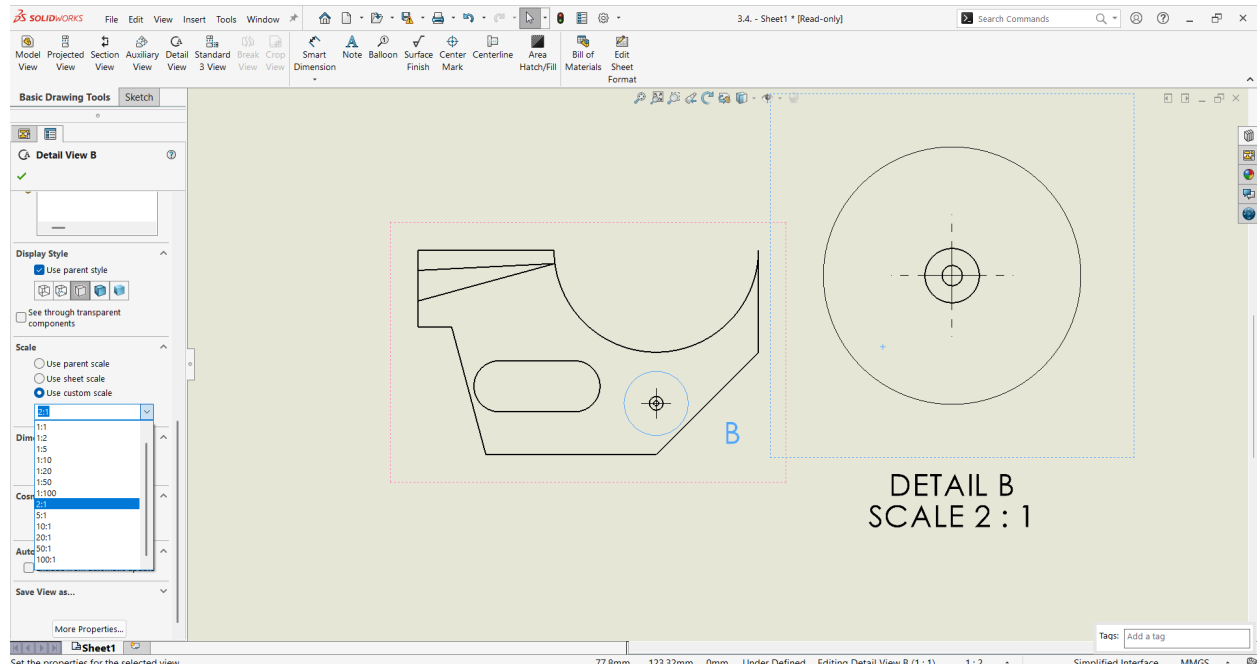
Move the mouse to the right side of the original view. Click on the drawing sheet to place the **Detail View**.



The view properties appear on the left side in the **PropertyManager**. Here, you can change options such as the view name and scale.



Set the scale to **2:1**.



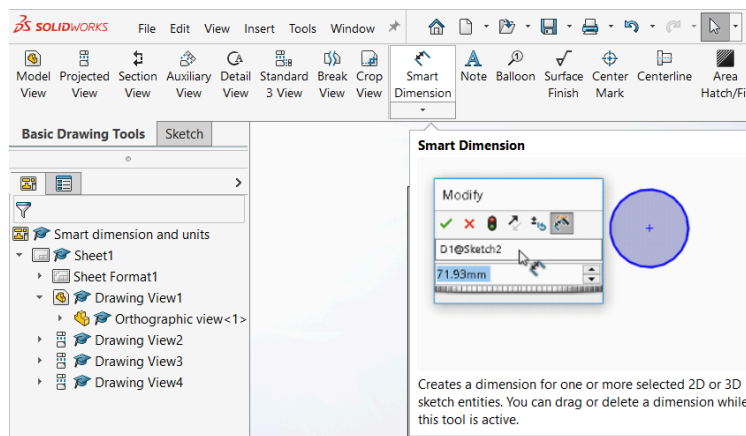
35. Smart Dimension

What it does: Adds dimensions to drawing geometry.

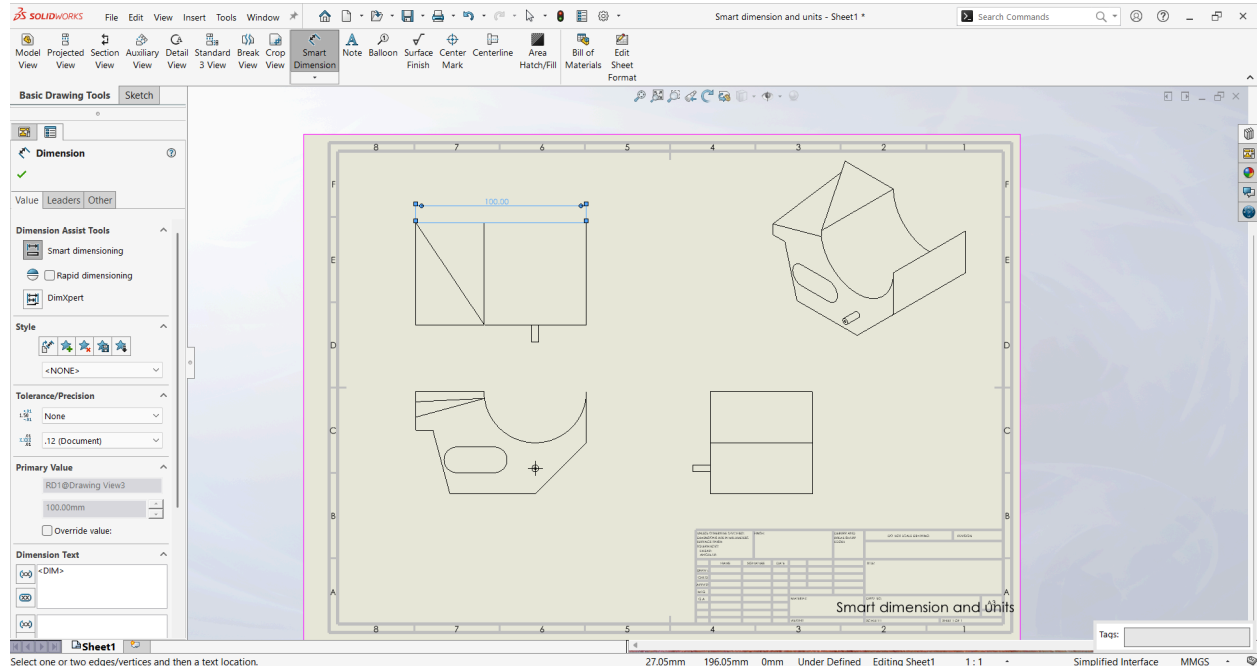
Use it for: Communicating sizes and locations required to manufacture or inspect the component.

How it works:

To add dimensions to a drawing view, we use the **Smart Dimension** tool. This works the same way as when adding dimensions in a sketch.

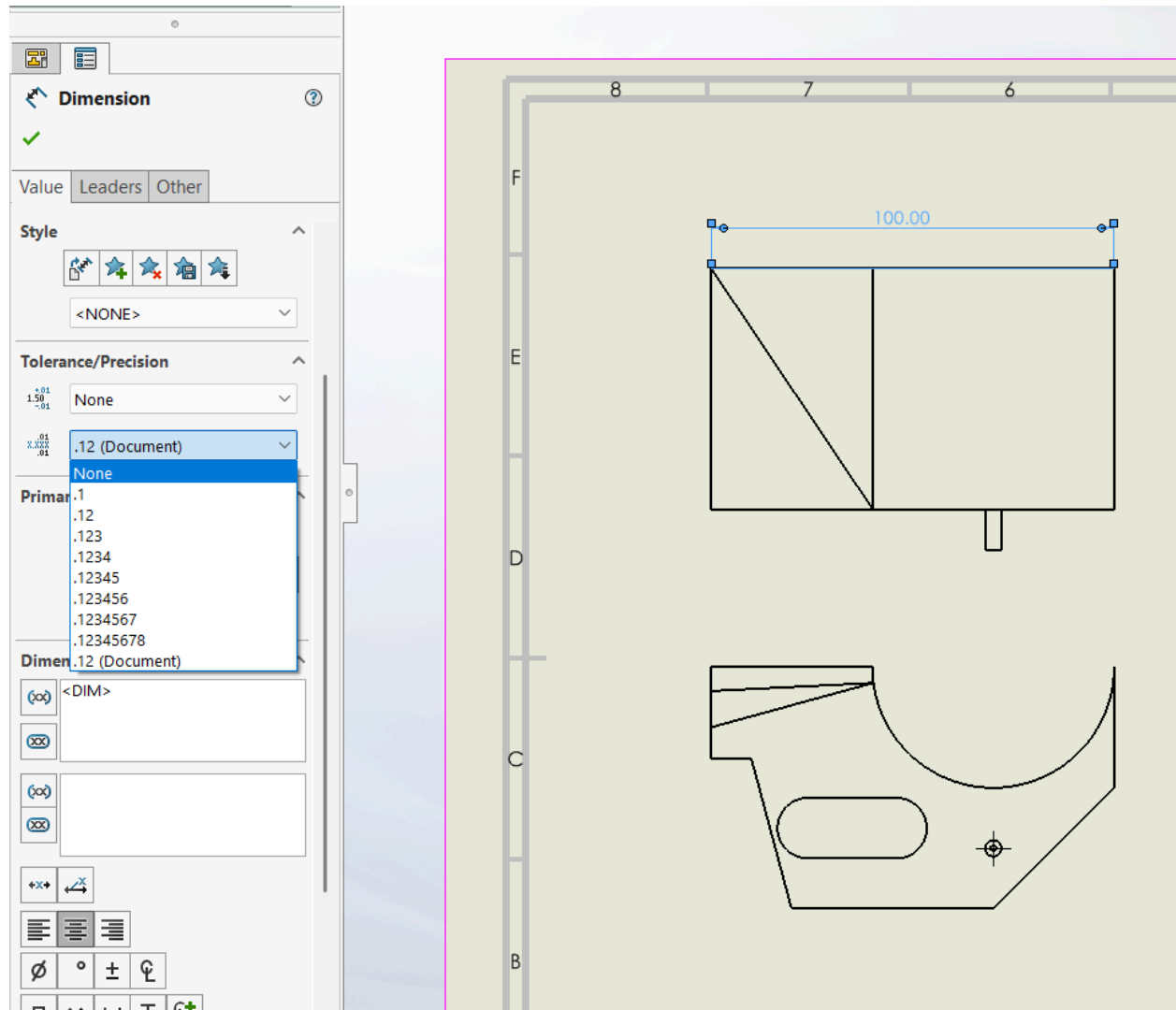


Go to the **Basic Drawing Tools** tab and select **Smart Dimension**.

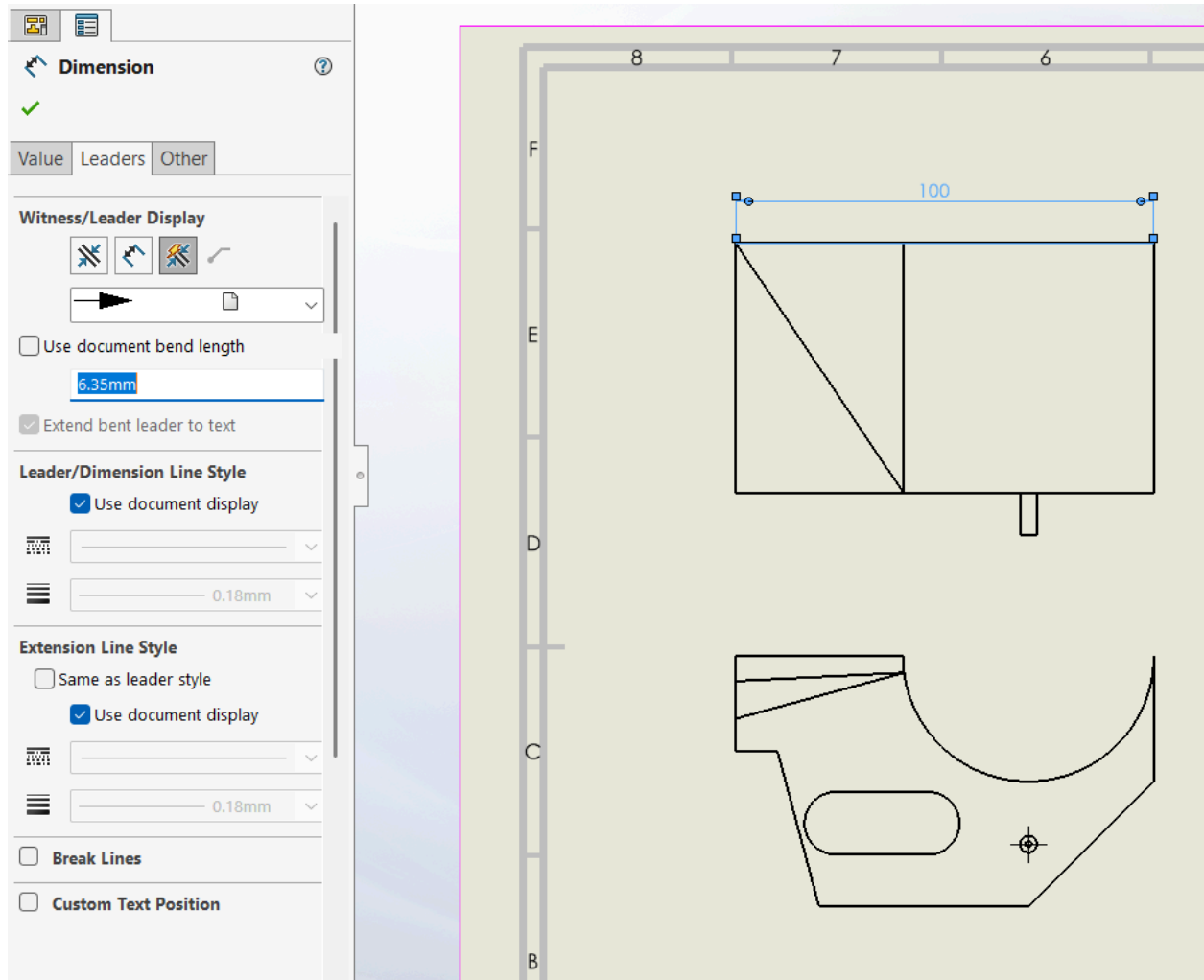


Adding dimensions works the same way as in sketching. If you want to add the length of a line, you can select the line directly. You can also select the two endpoints of the line, or select two vertical lines that represent the start and end of the distance.

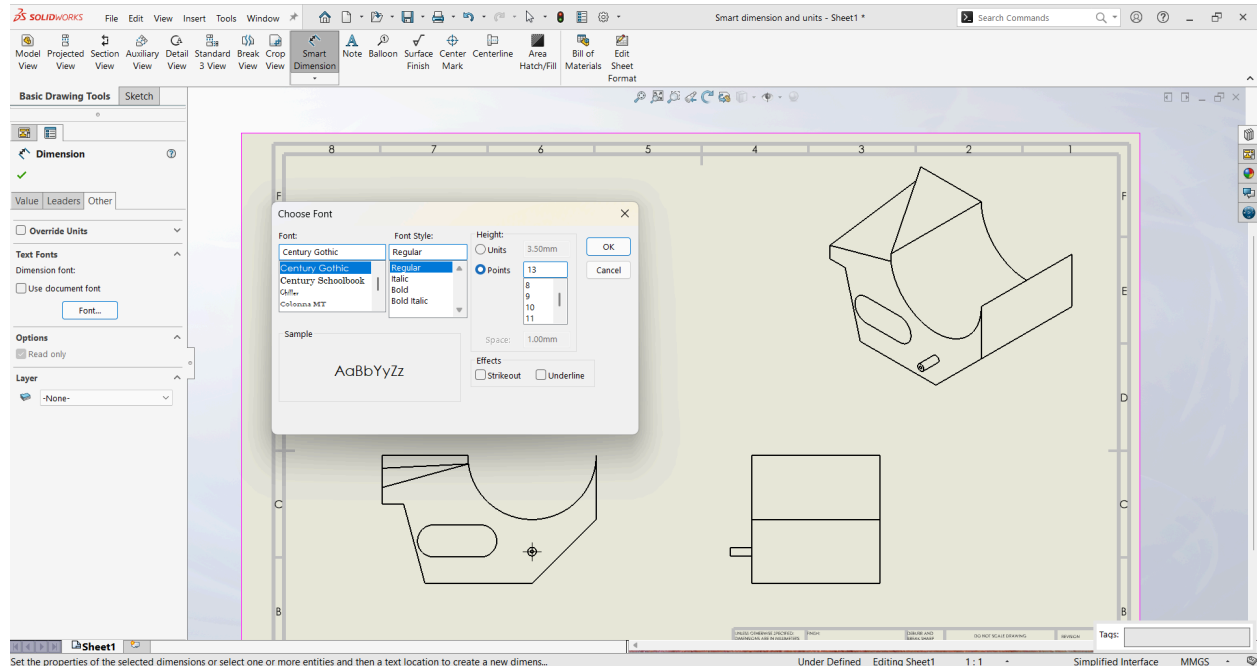
To change the number of decimal places, click on the dimension. Then go to the **Tolerance/Precision** and select how many decimal places you want to display.



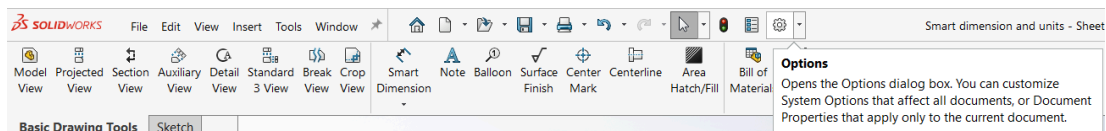
To change the size of the leader, click on the dimension. Then go to the **PropertyManager** on the left side and select the **Leaders** tab. Under **Witness/Leader Display**, define the leader length.



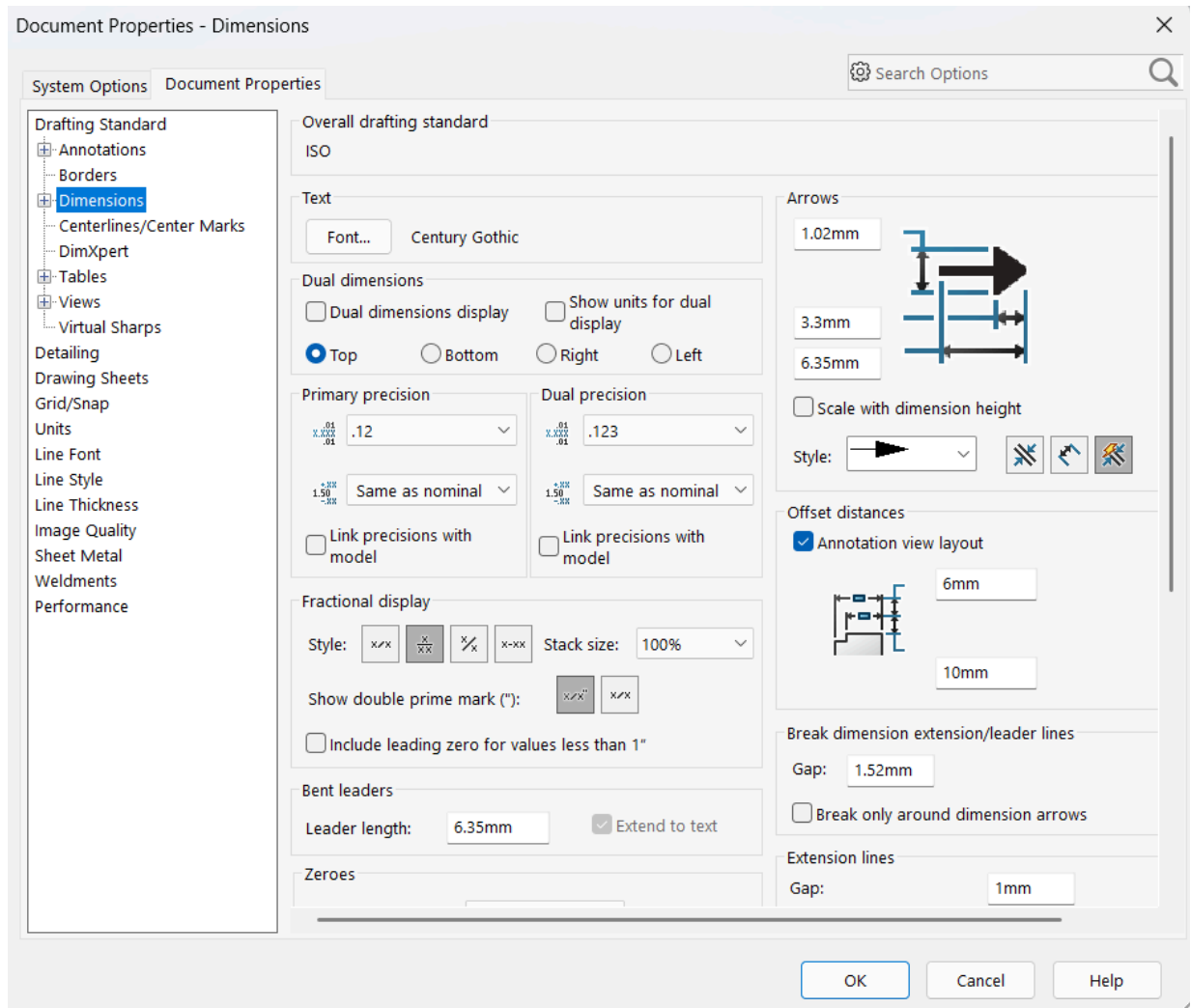
To change the font, go to the **Other** tab in the dimension properties. Then select **Font**. A new dialog box opens. Under **Height**, choose **Points** and define the font size.



To change the properties for the entire drawing sheet, go to **Options**.

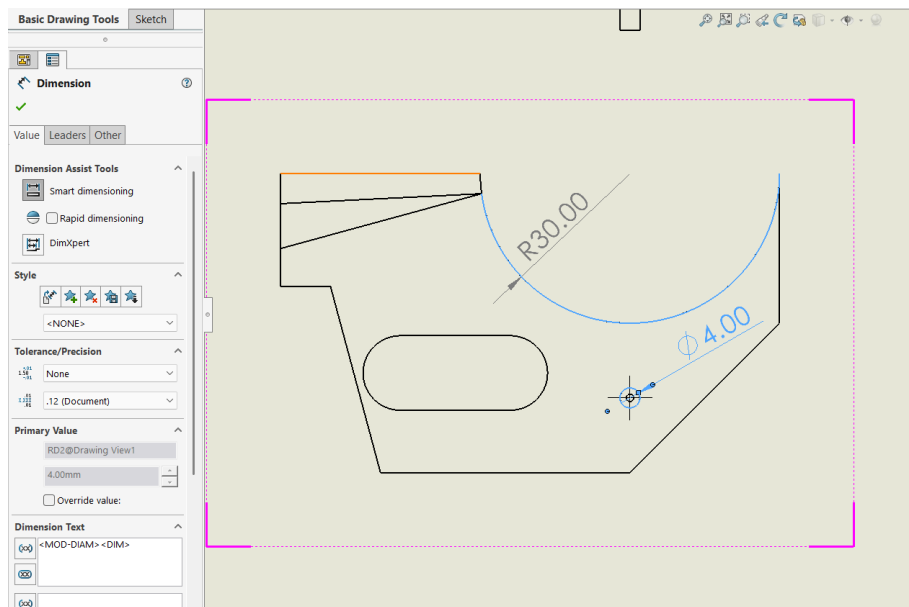
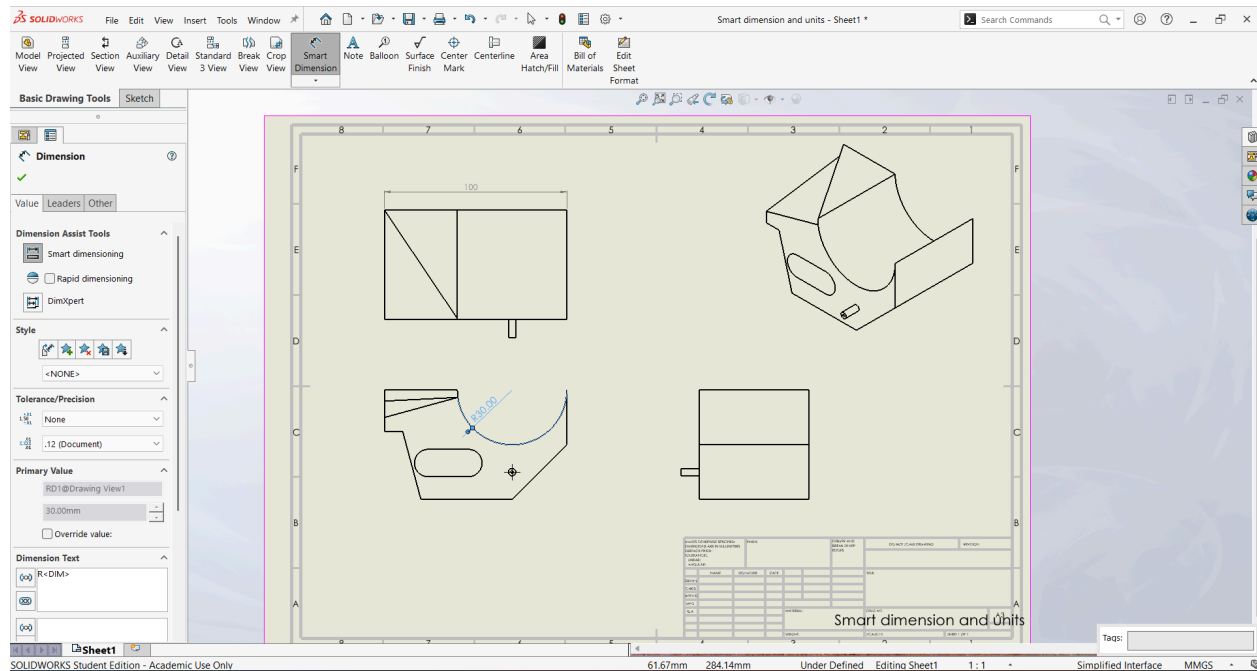


A new dialog box opens. Select the **Document Properties** tab, then select **Dimensions**. Here, you can change the font of the dimension text and adjust the arrow size for the entire drawing sheet.



We can also add a radius dimension using the **Smart Dimension** tool.

Select **Smart Dimension**, then click on the arc.



Want to go beyond individual commands and learn how to actually use **SOLIDWORKS** from beginner to advanced level?

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- ✓ **Engineering Drawings** – Turn your 3D models into professional technical drawings

Whether you're completely new to SOLIDWORKS or want to take your existing skills to the next level, the course gives you a **structured path from the fundamentals to advanced modeling**.

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