

SOLIDWORKS PART DESIGN COMMANDS CHEAT SHEET

13 Essential Part Design Commands Every SolidWorks User Should Know

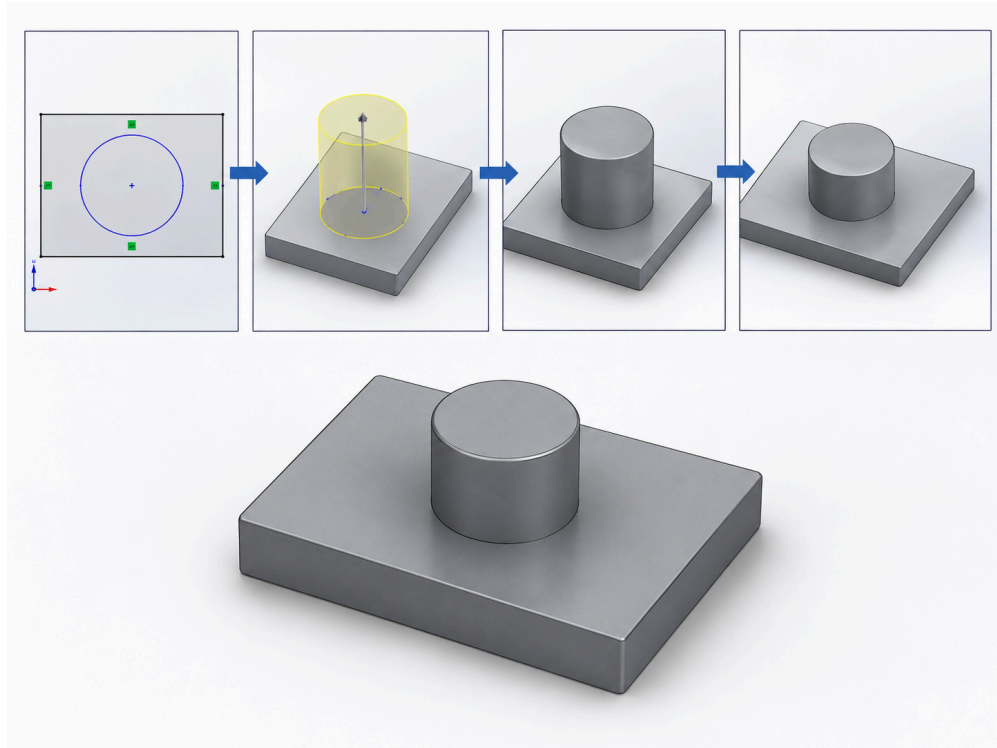
1. Extruded Boss/Base

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

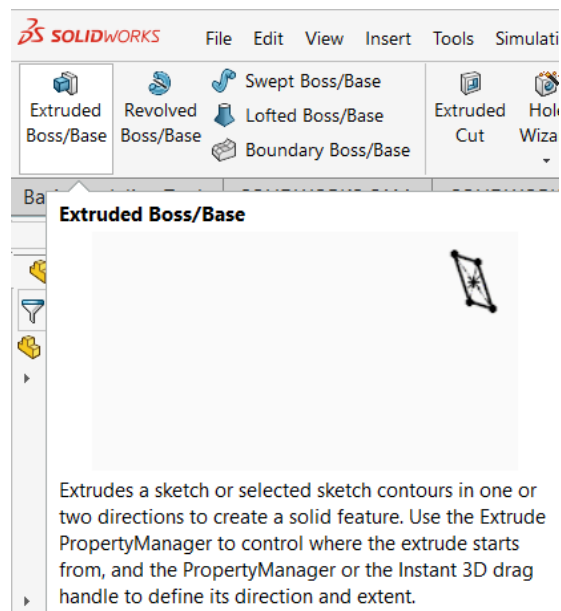
Use it for: Creating the base shape of many parts.

How it works:

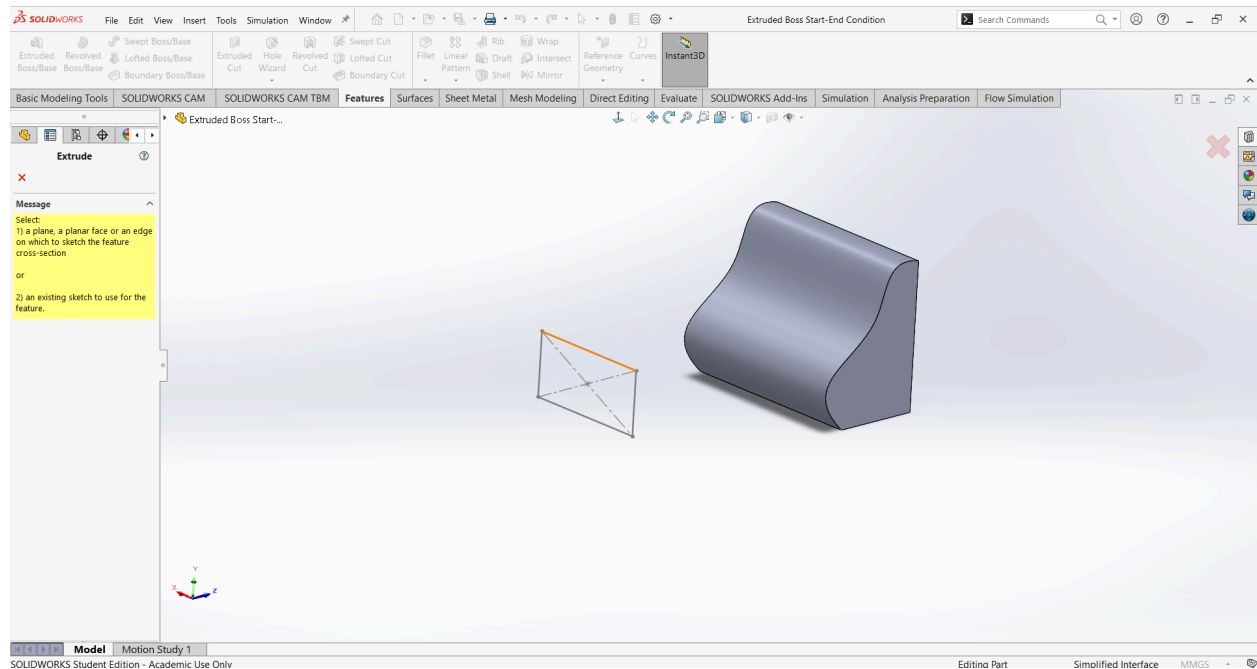
The first tool we will cover is **Extruded Boss/Base**.



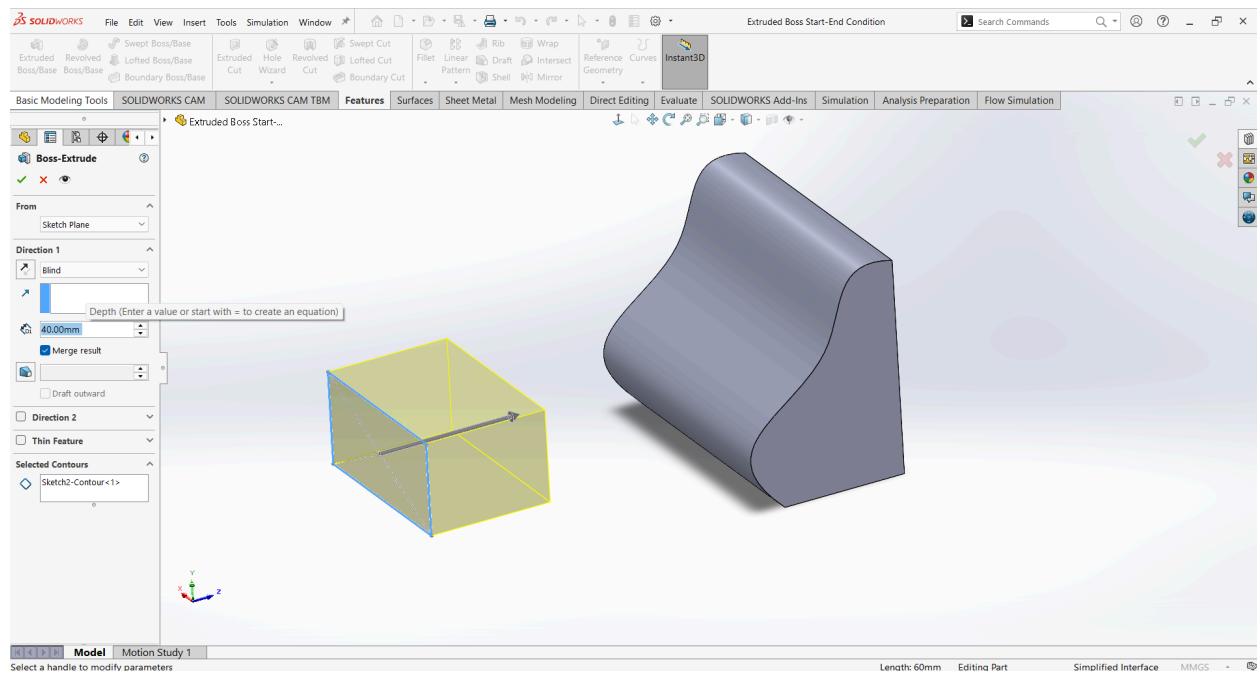
We already used this tool earlier when converting a sketch into a 3D model. Now we will explain it in more detail. Go to the **Features** tab and select **Extruded Boss/Base**.



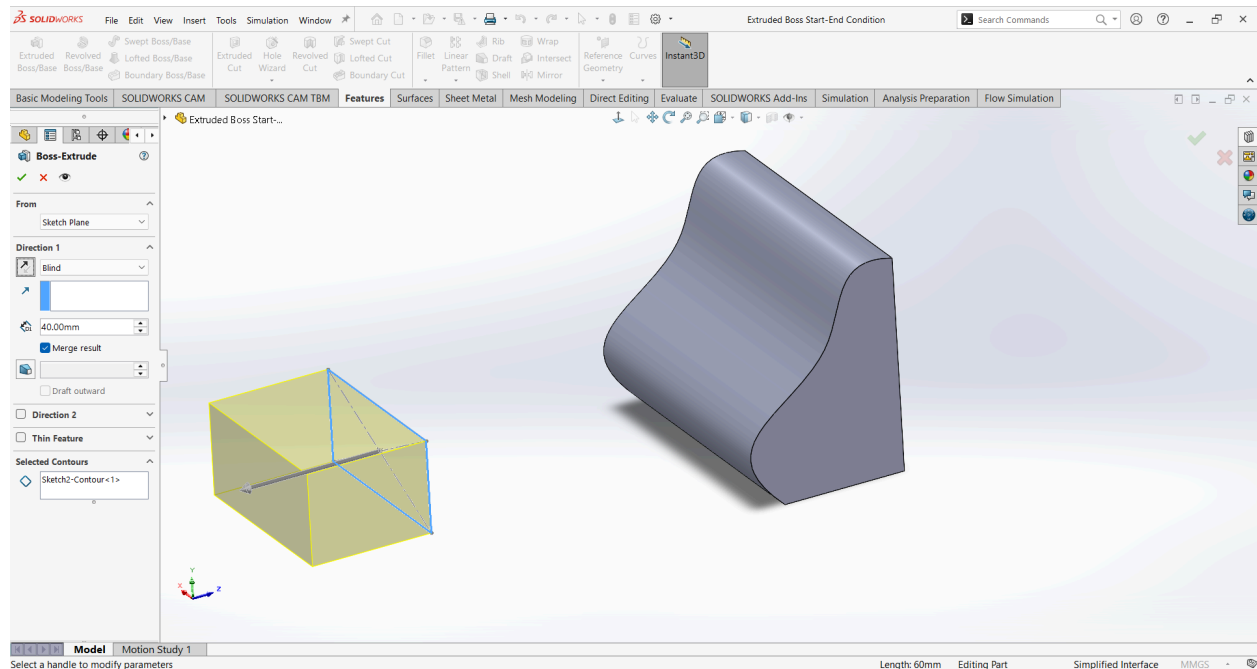
Select the rectangle, as shown.



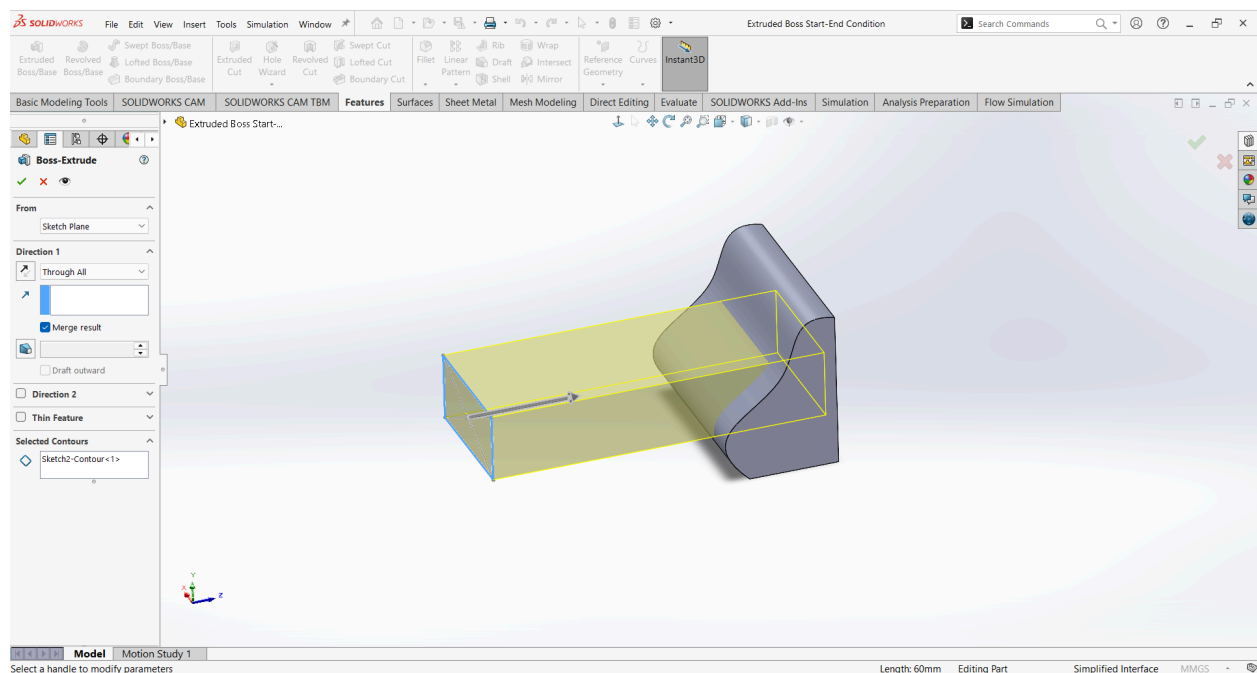
The first end condition is **Blind**. This option is used when you want to extrude the sketch to a defined depth. In this example, set the depth to **40 mm**.



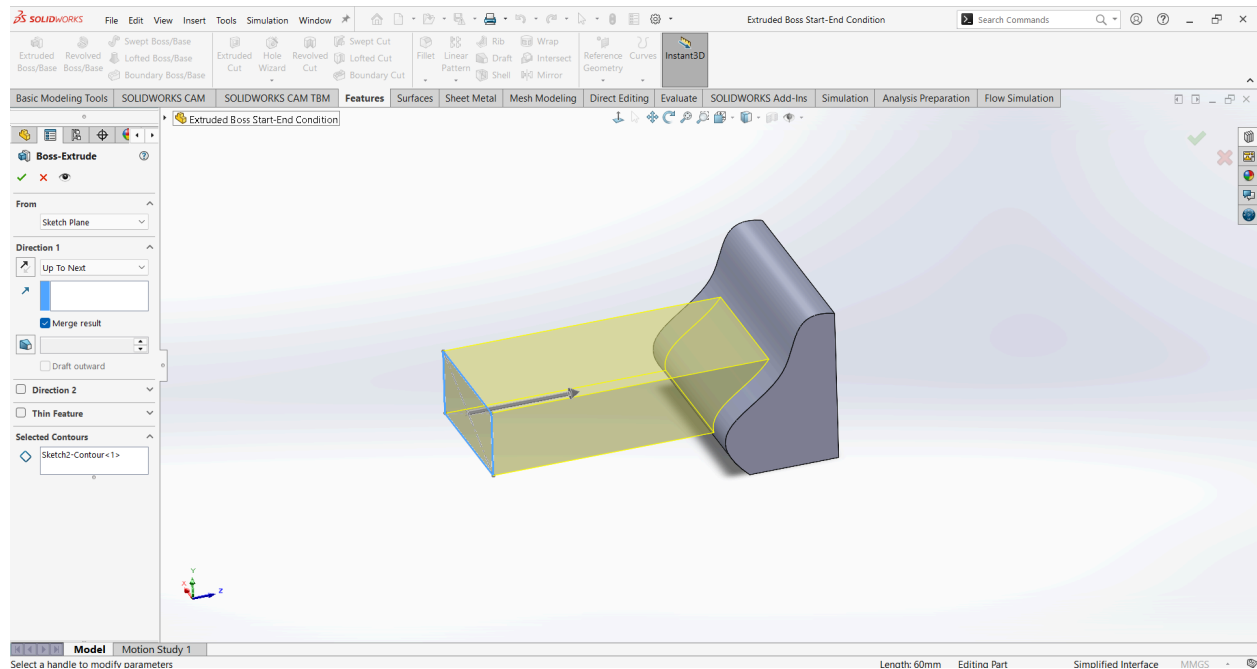
When you click the arrow next to **Blind**, you can change the direction of the extrusion.



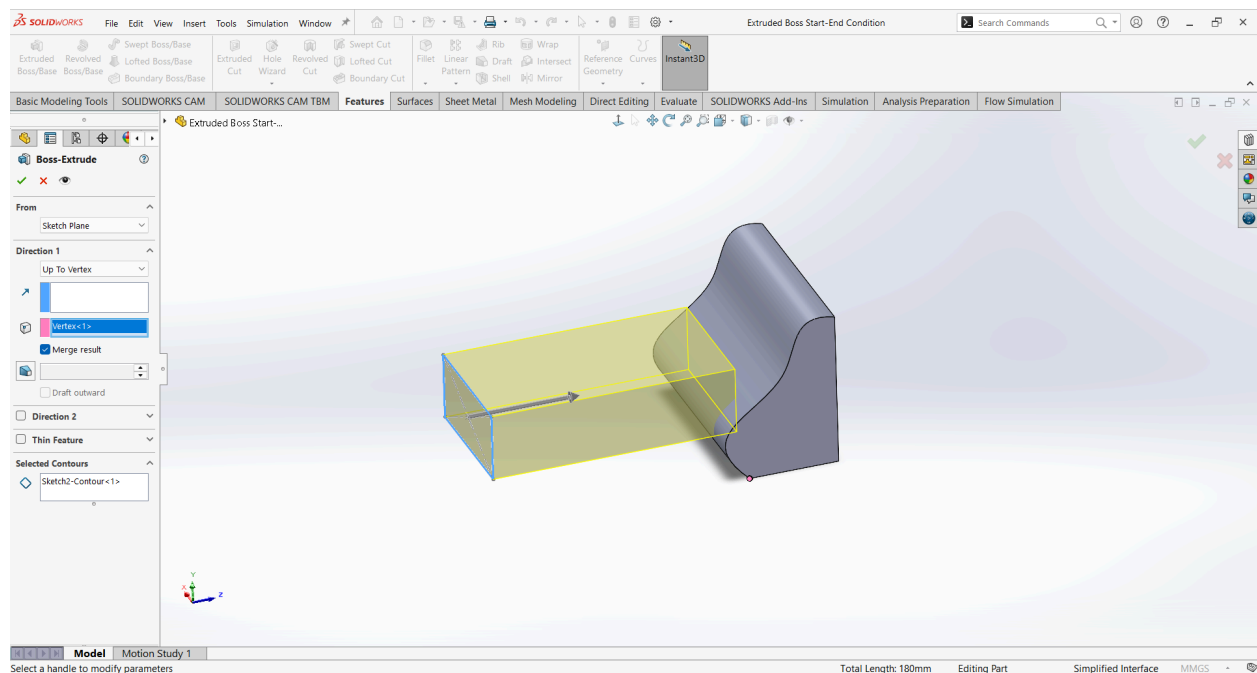
The next end condition is **Through All**. With this option, the extrusion goes through the entire model in front of the sketch.



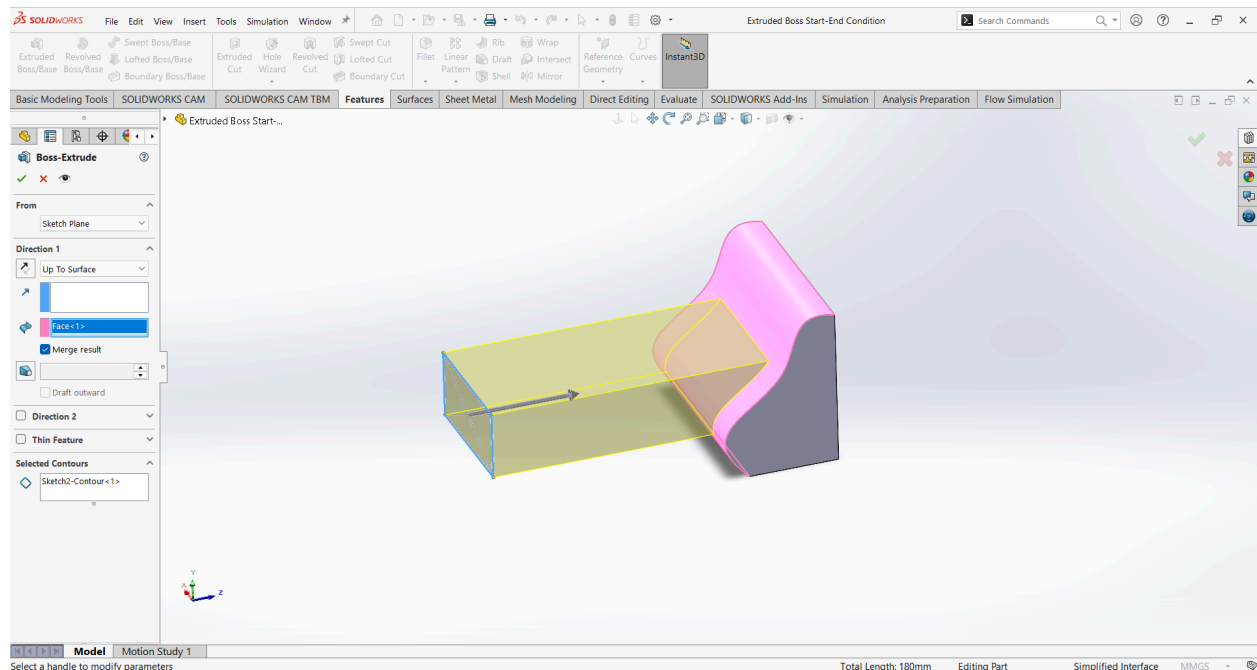
The next end condition is **Up To Next**. With this option, the extrusion extends up to the next face or body in front of the sketch.



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.

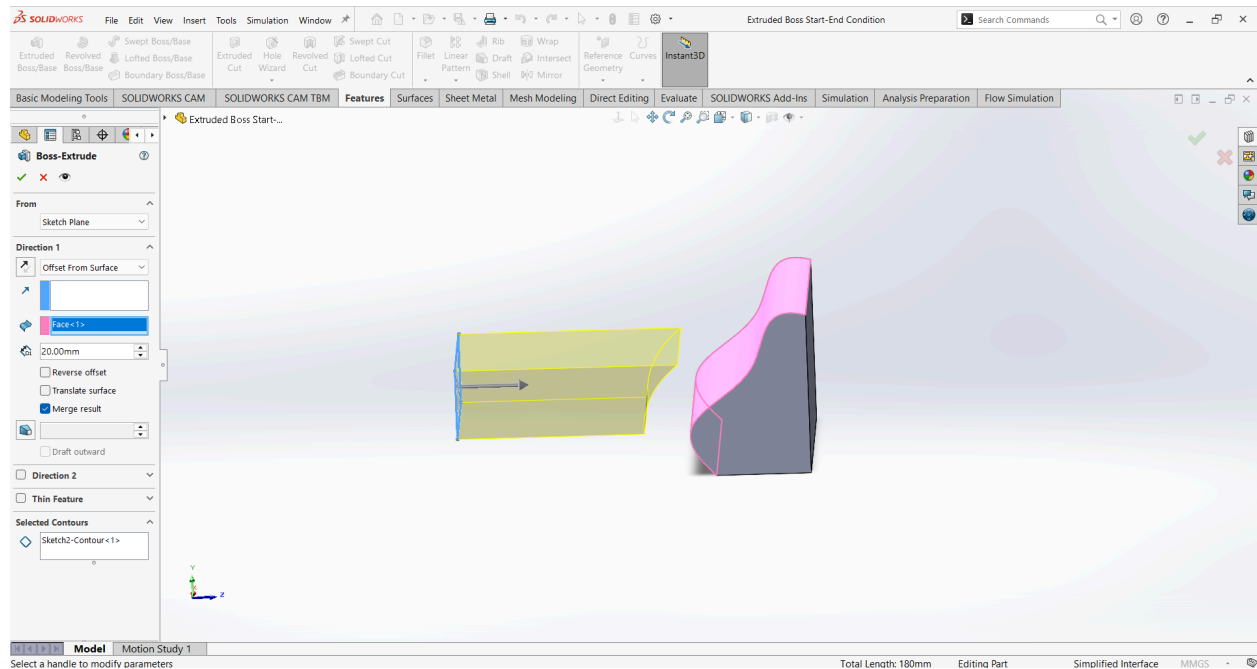


The next end condition is **Up To Surface**. With this option, you need to select a face where the extrusion will stop. First, click the selection box on the left side in the **PropertyManager**, then select a face on the model. This makes the extrusion extend up to the selected face.

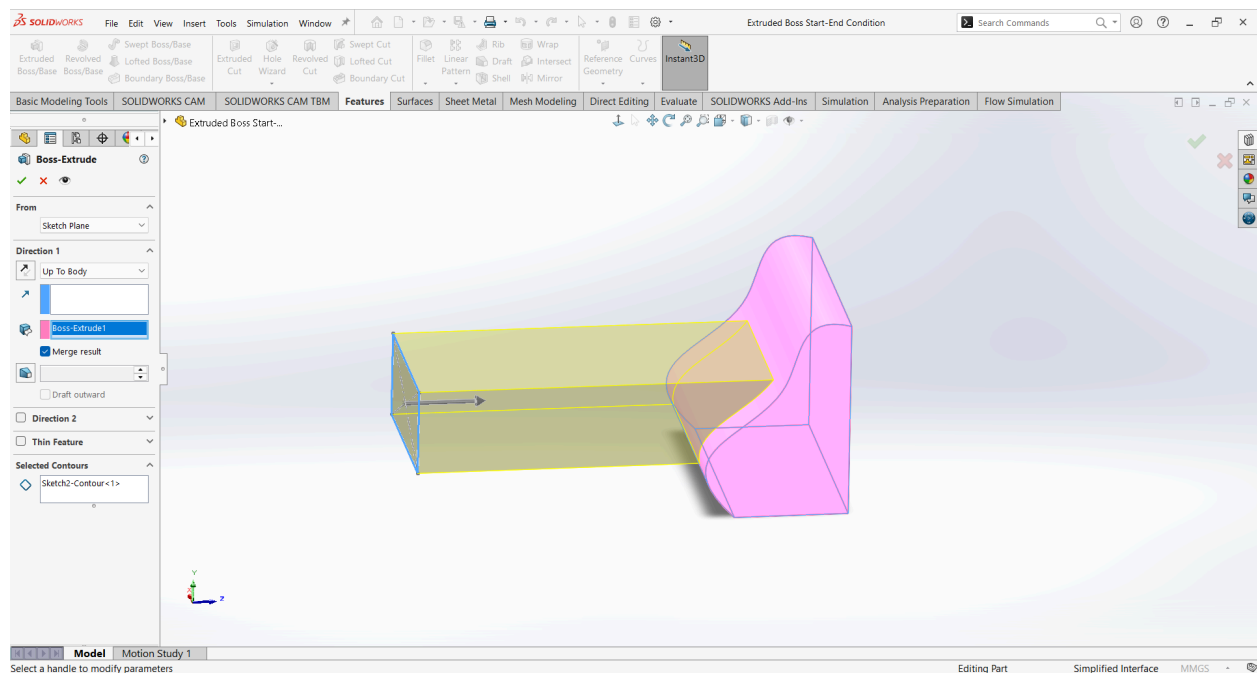


The next end condition is **Offset From Surface**.

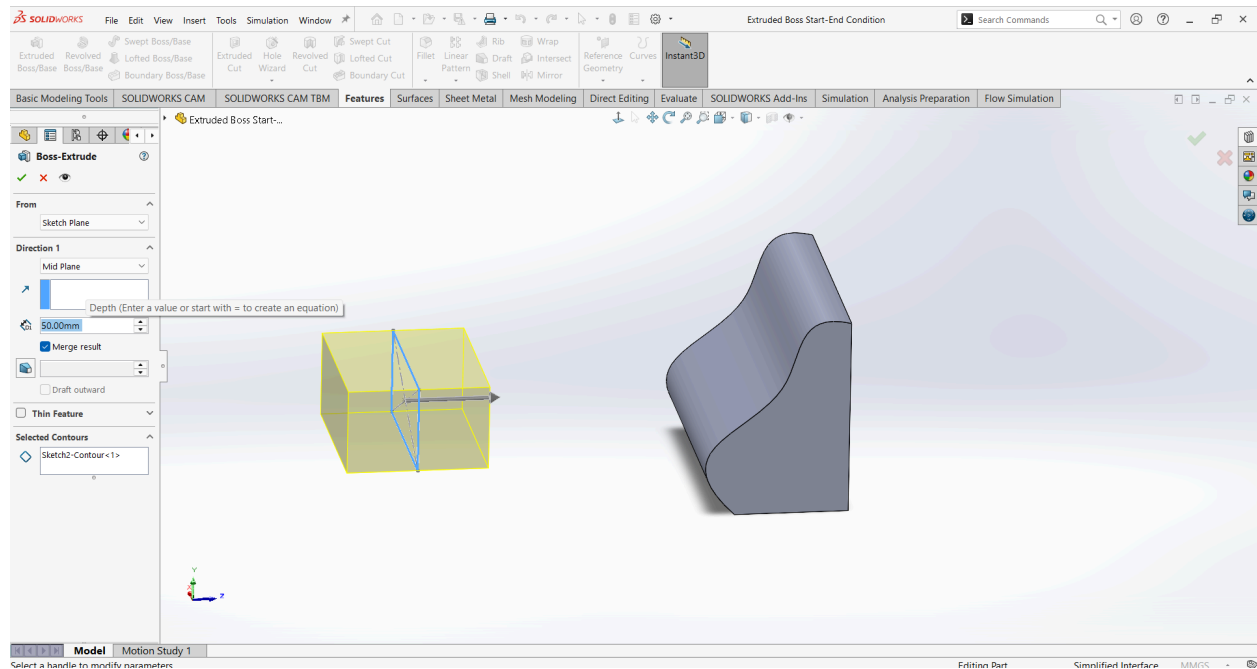
First, click the selection box on the left side in the **PropertyManager**. Then select a face on the model and define the offset distance. In this example, the offset distance is **20 mm**. This means the extrusion stops **20 mm** away from the selected face.



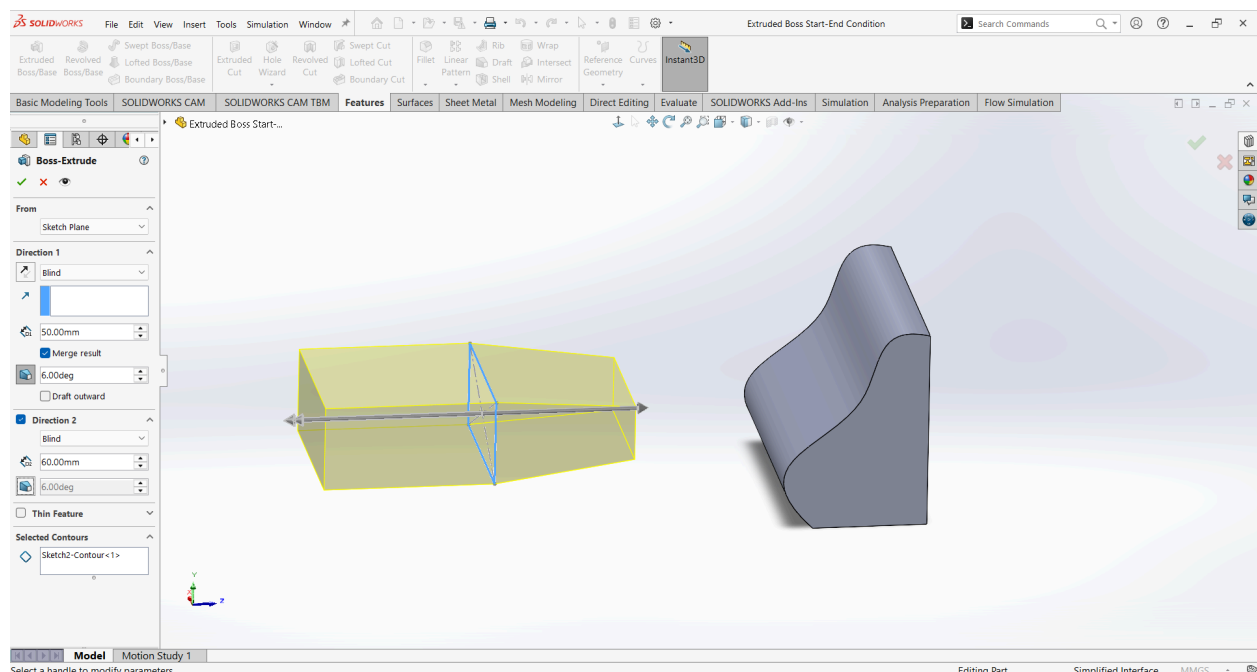
The next end condition is **Up To Body**. First, click the selection box on the left side in the **PropertyManager**. Then select the body where you want the extrusion to stop. This makes the extrusion extend up to the selected body.



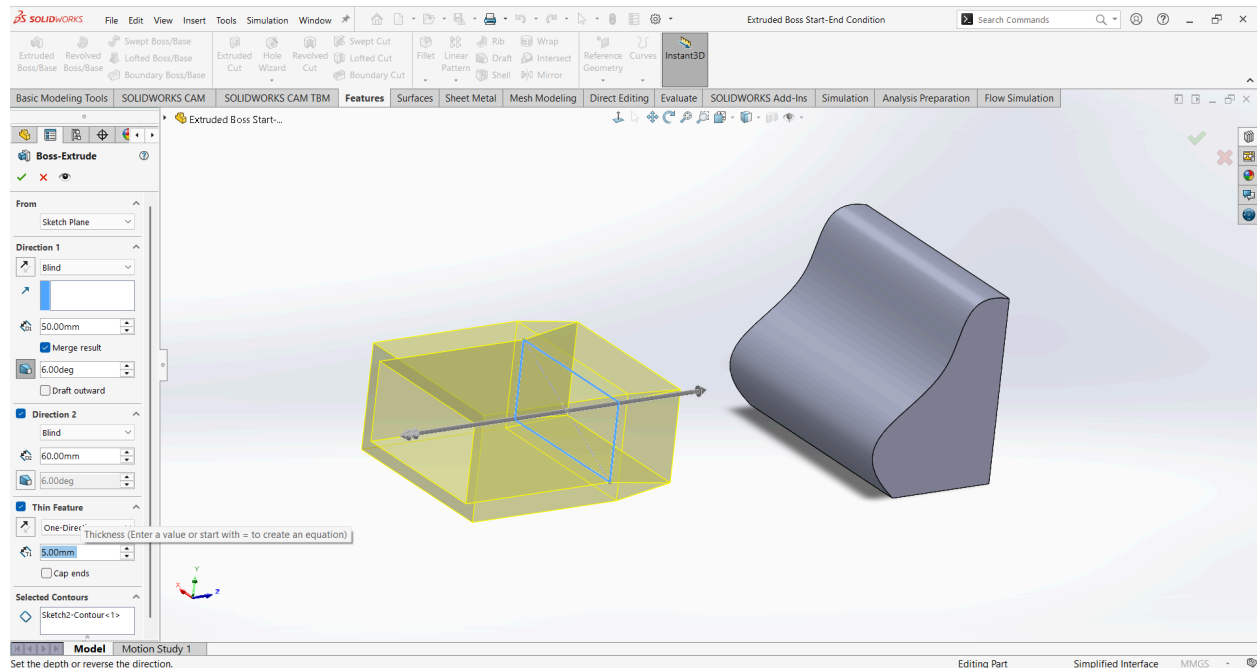
The next end condition is **Mid Plane**. With this option, the extrusion goes equally in both directions from the sketch. You also need to define the total depth of the extrusion.



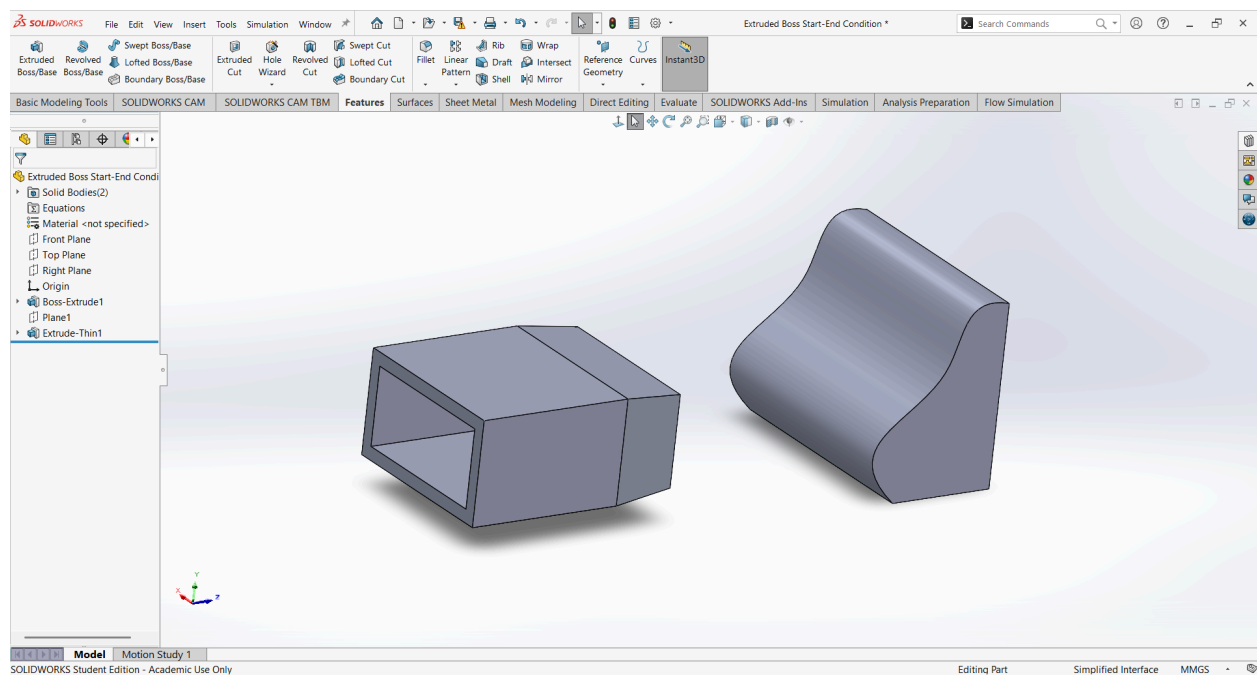
Below the depth value in the **PropertyManager**, there is a **Draft** option. This means the extrusion can be created at an angle. In this example, the draft angle is set to **6°**. There is also a **Direction 2** option. To turn it on, check the small box next to **Direction 2**. For **Direction 2**,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

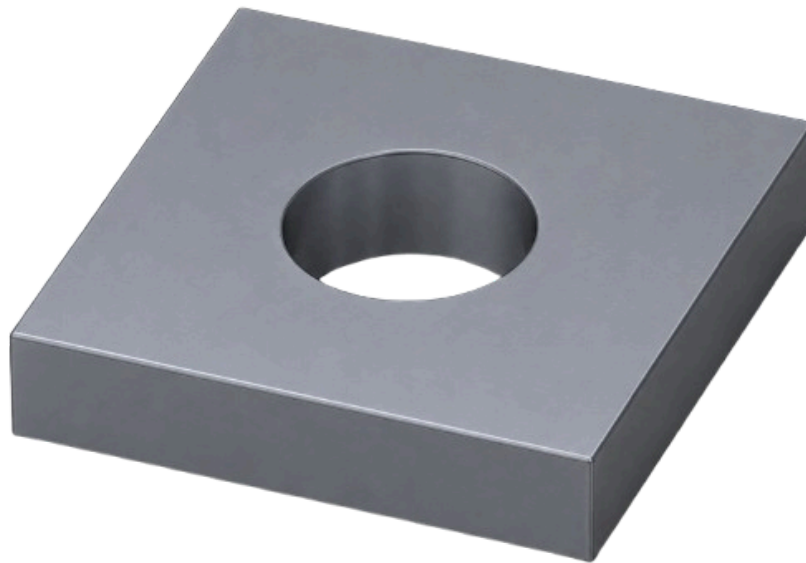
2. Extruded Cut

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

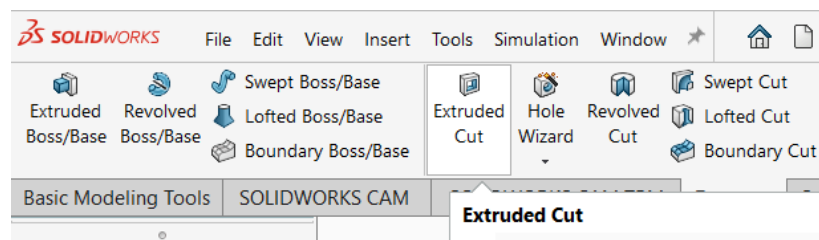
Use it for: Holes, pockets, slots, openings, and cutouts.

How it works:

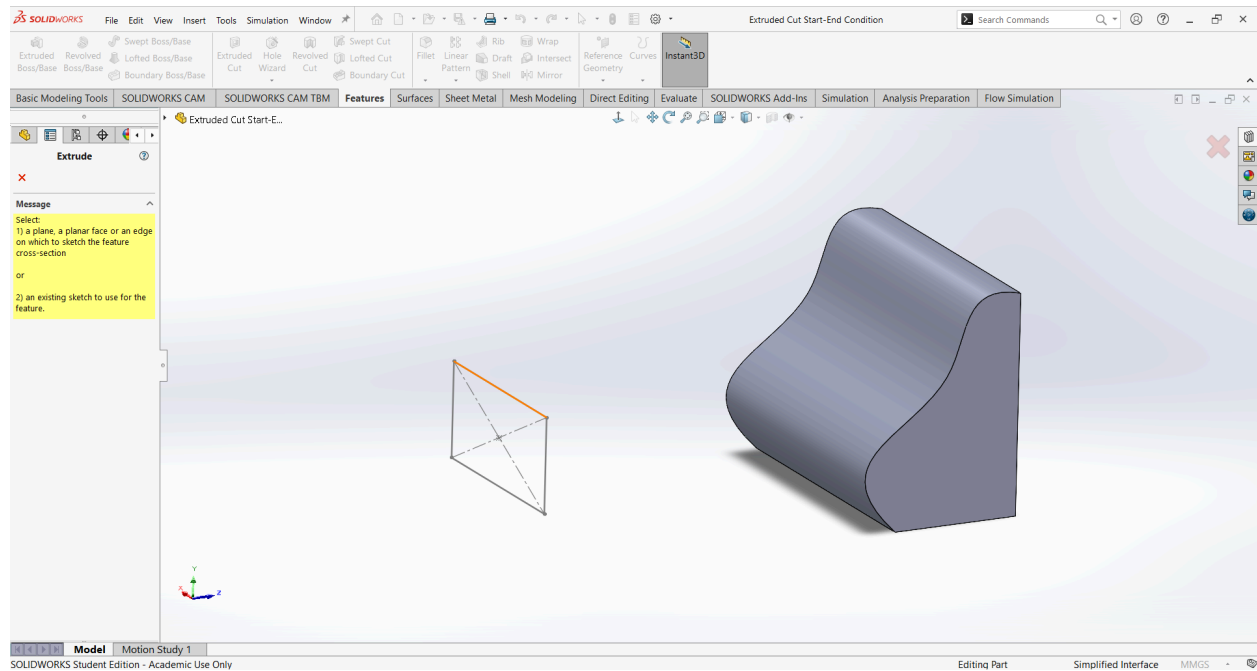
With **Extruded Boss/Base**, we add material. With **Extruded Cut**, we remove material.



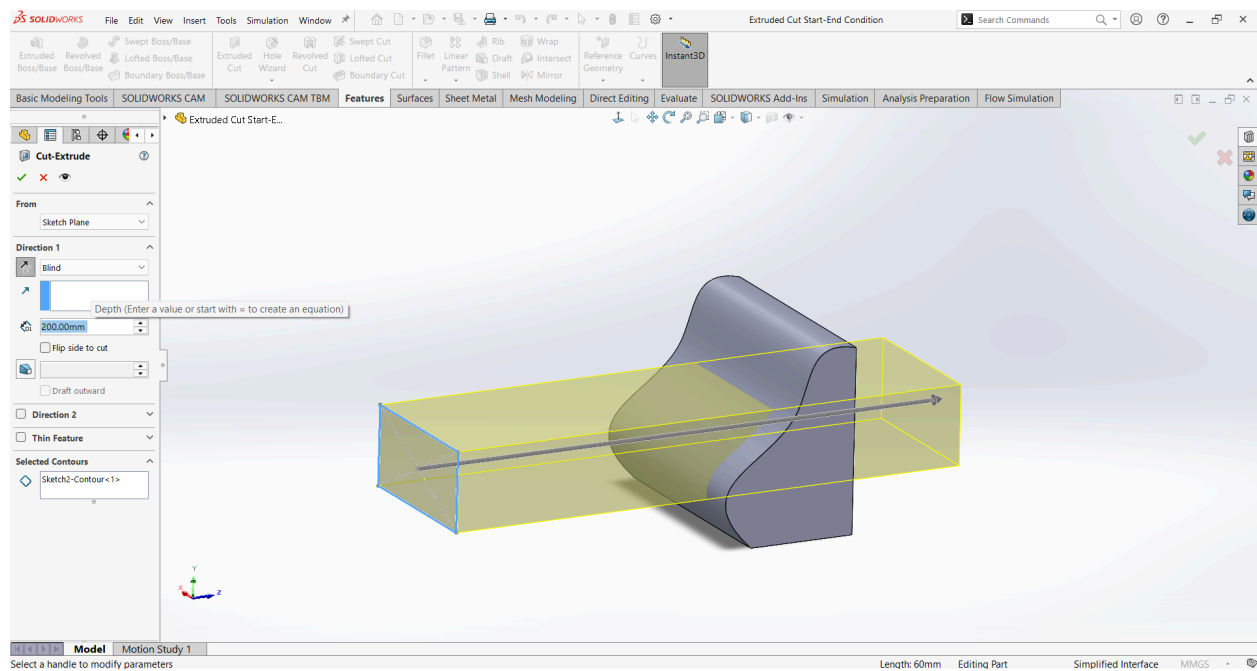
Go to the **Features** tab and select **Extruded Cut**.



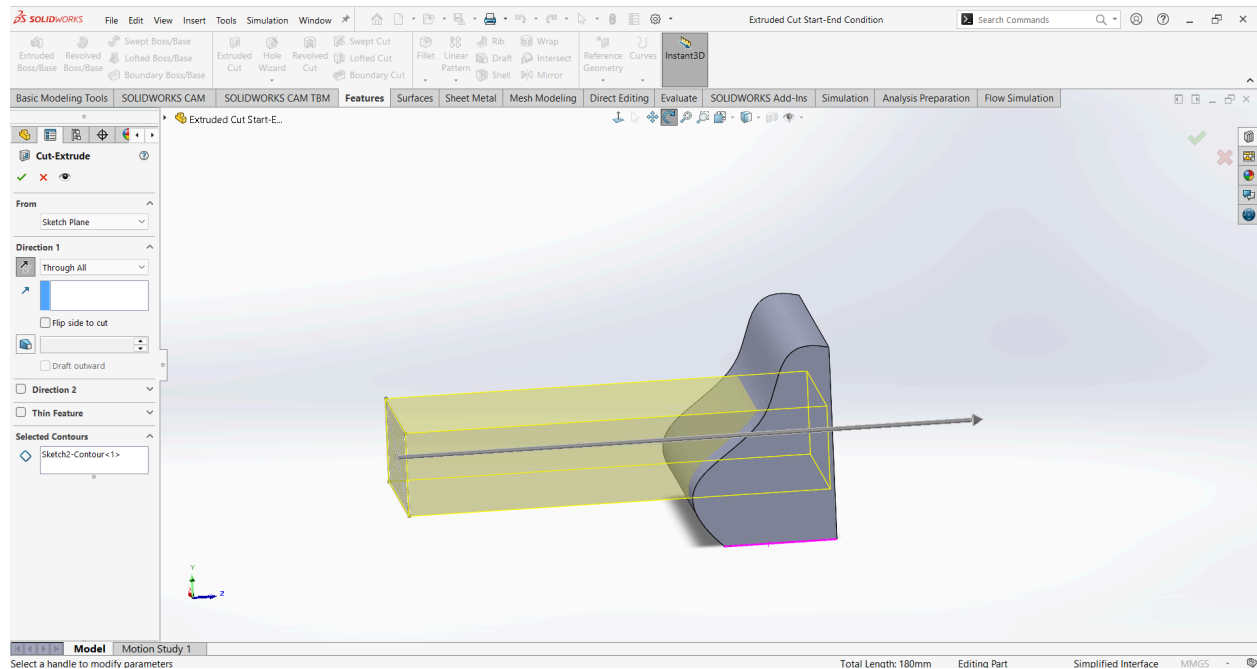
A yellow prompt appears, asking you to select a sketch. Click on the sketch.



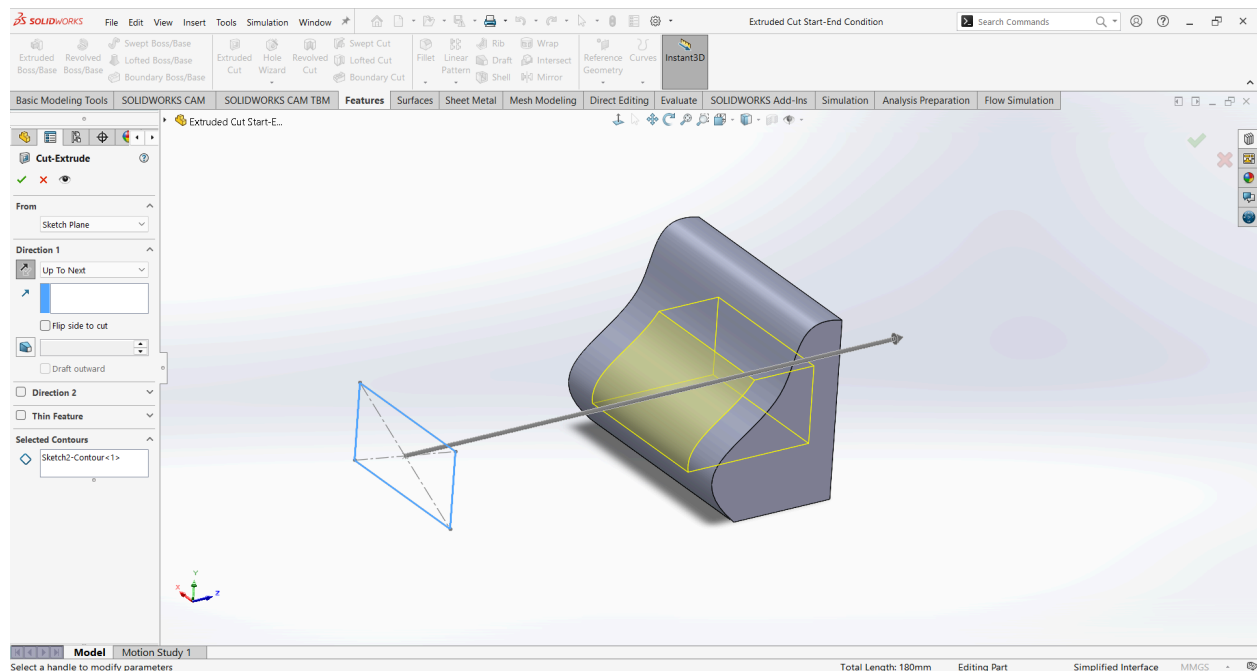
The first end condition is **Blind**. Here, we have similar options as with **Extruded Boss/Base**. With **Blind**, you define the depth of the cut. Set the cut depth to **200 mm**.



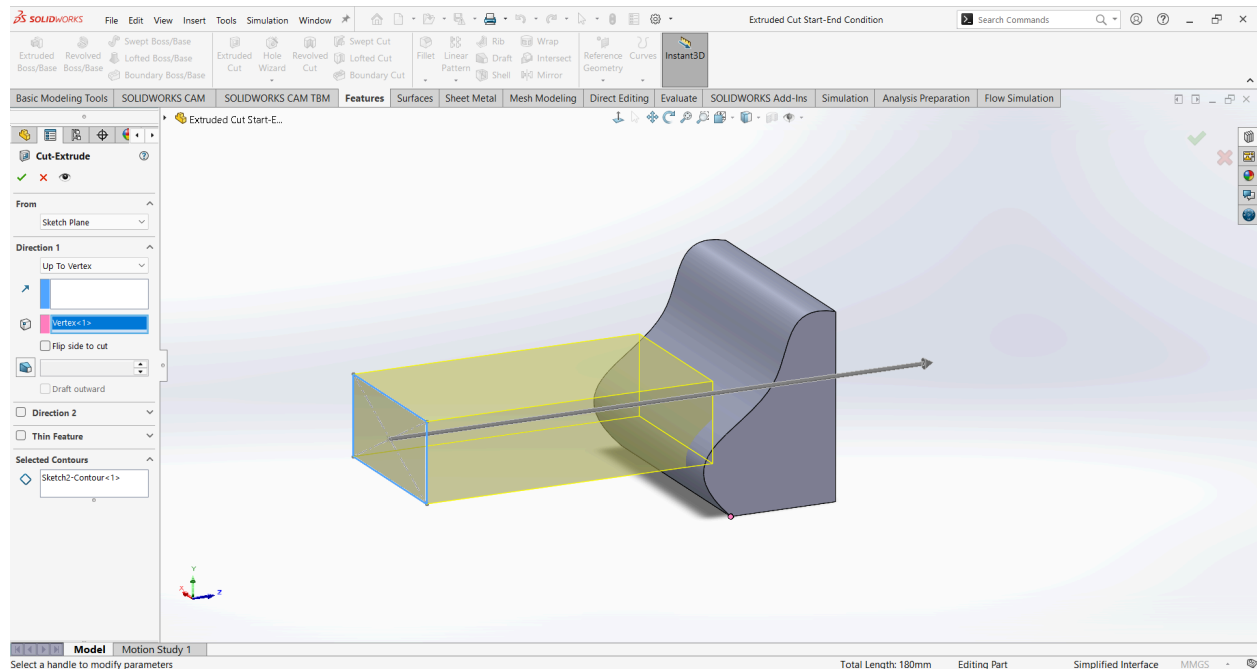
The next end condition is **Through All**. With this option, the cut goes through all the material.



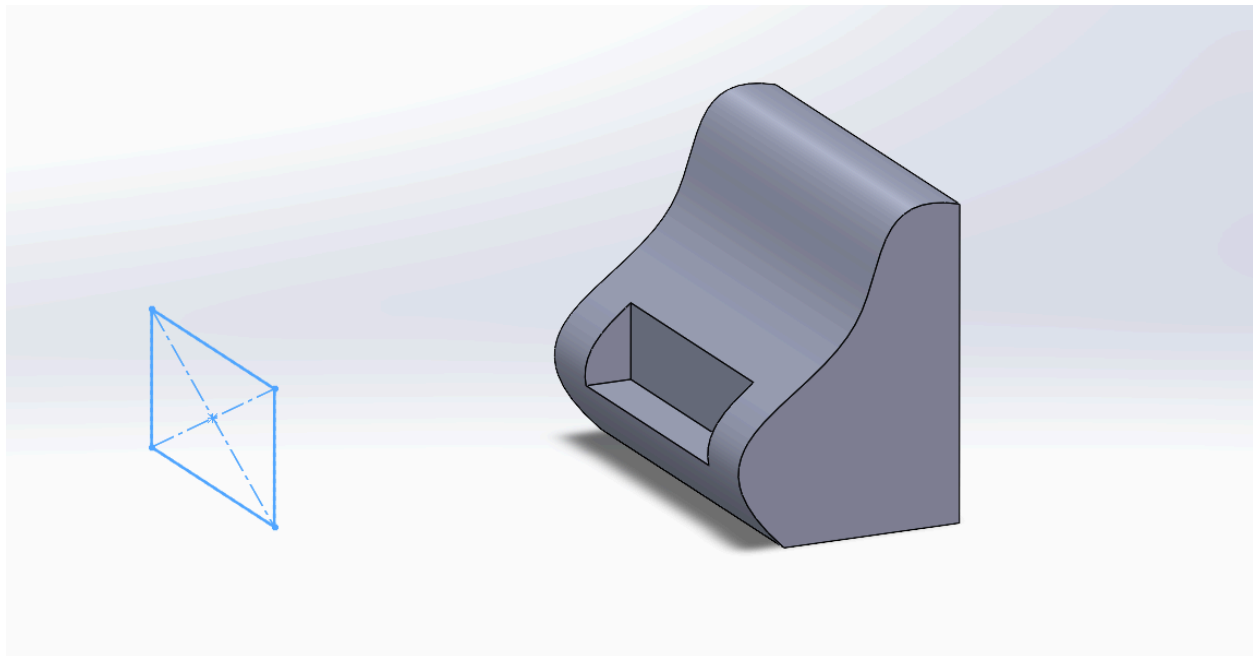
The next end condition is **Up To Next**. With this option, the cut goes up to the next face or body in front of the sketch. It also cuts through that selected geometry.



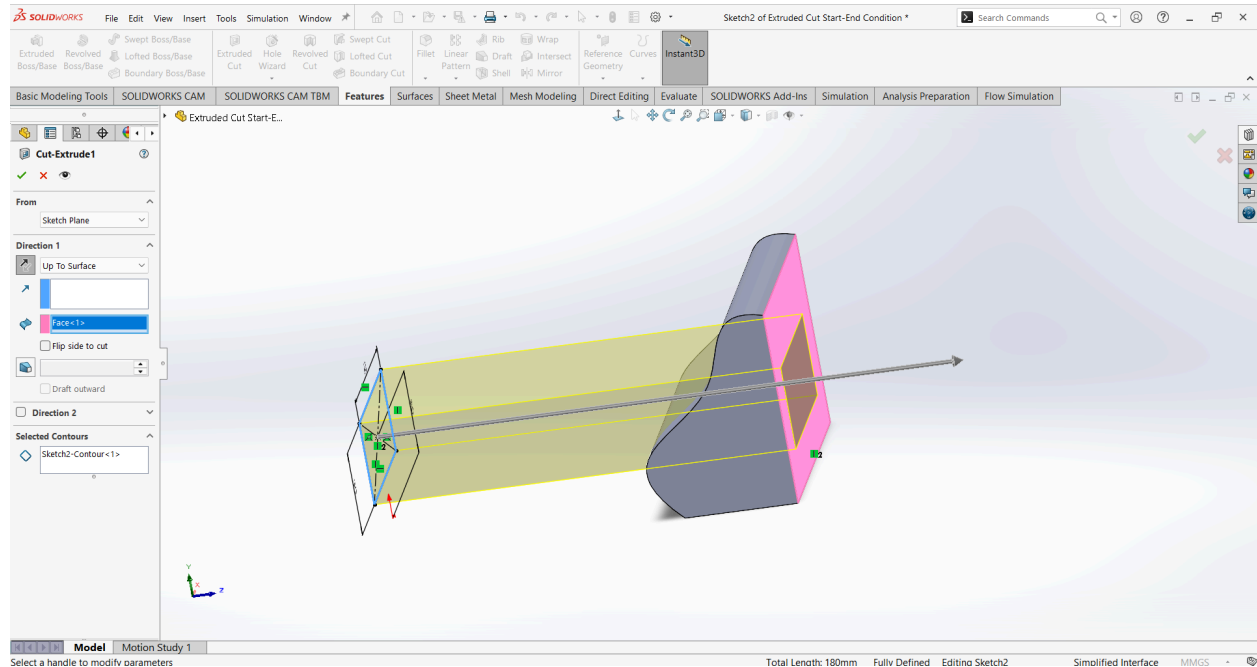
The next end condition is **Up To Vertex**. With this option, the cut goes up to the selected vertex. Click the selection box on the left side in the **PropertyManager**, then select a vertex on the model.



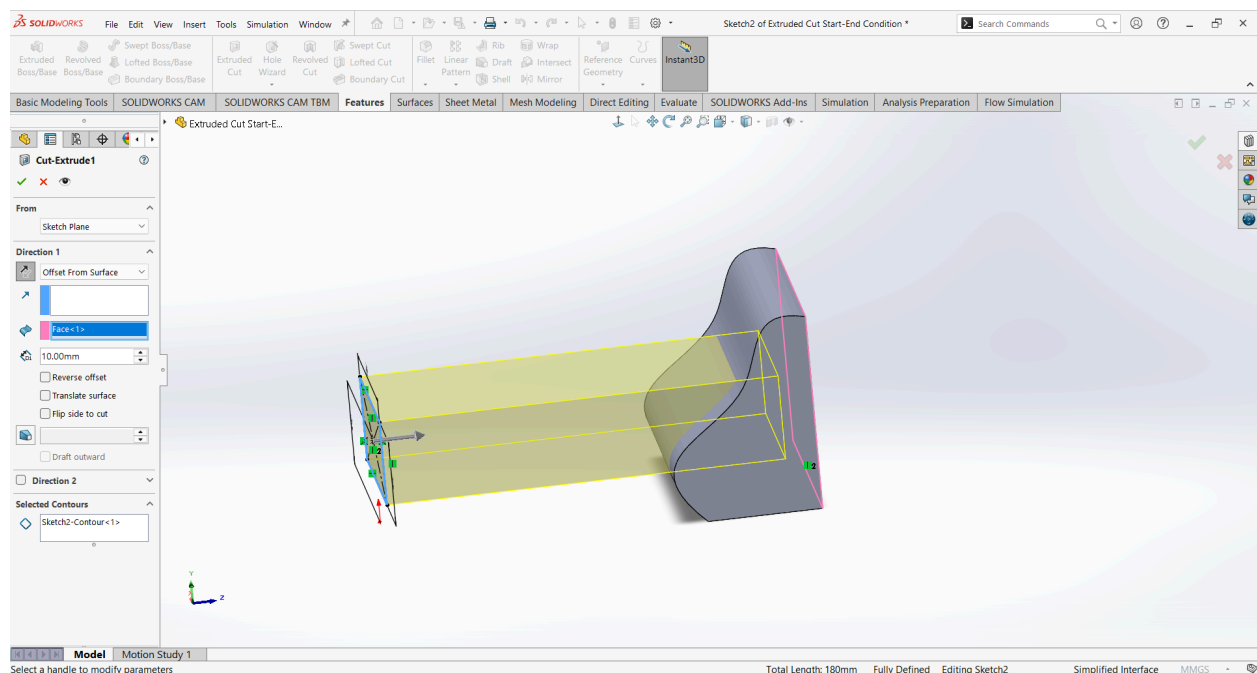
Here is the result.



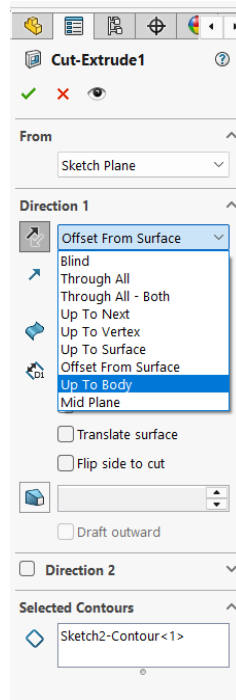
The next end condition is **Up To Surface**. With this option, the cut goes up to the surface you select. Click the selection box on the left side in the **PropertyManager**, then select a face on the model.



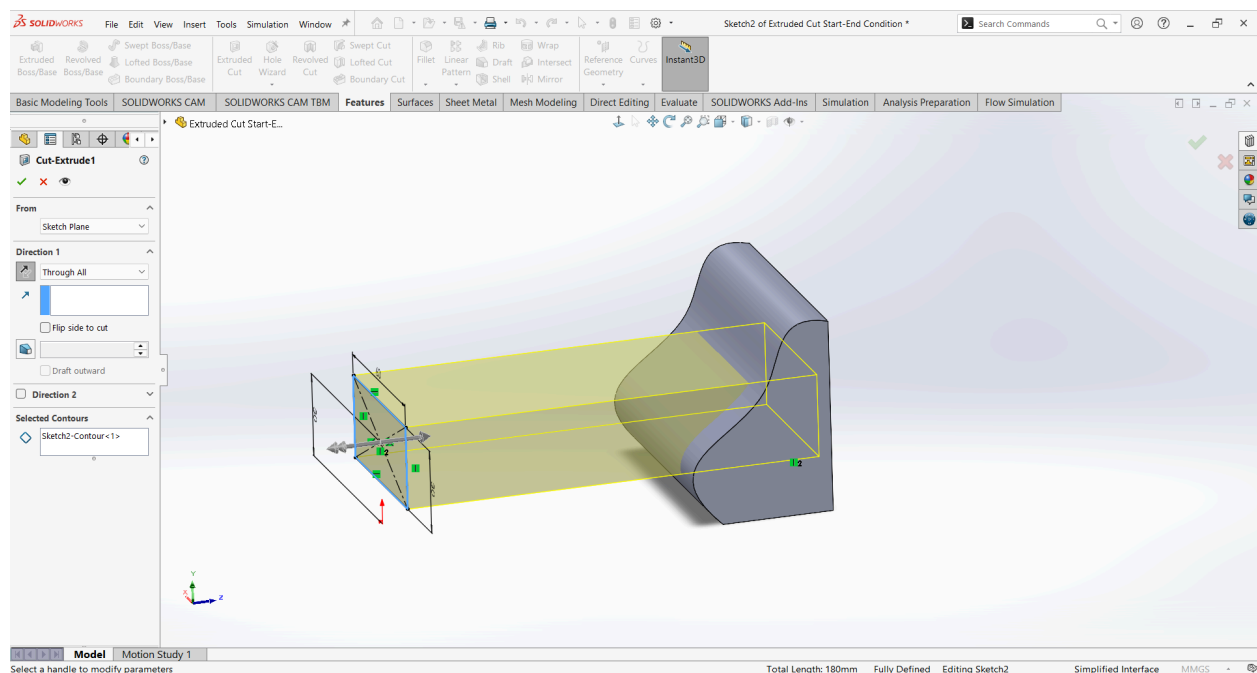
The next end condition is **Offset From Surface**. With this option, the cut stops at an offset distance from the selected surface. First, click the selection box on the left side in the **PropertyManager**. Then select a face on the model and define the offset distance.

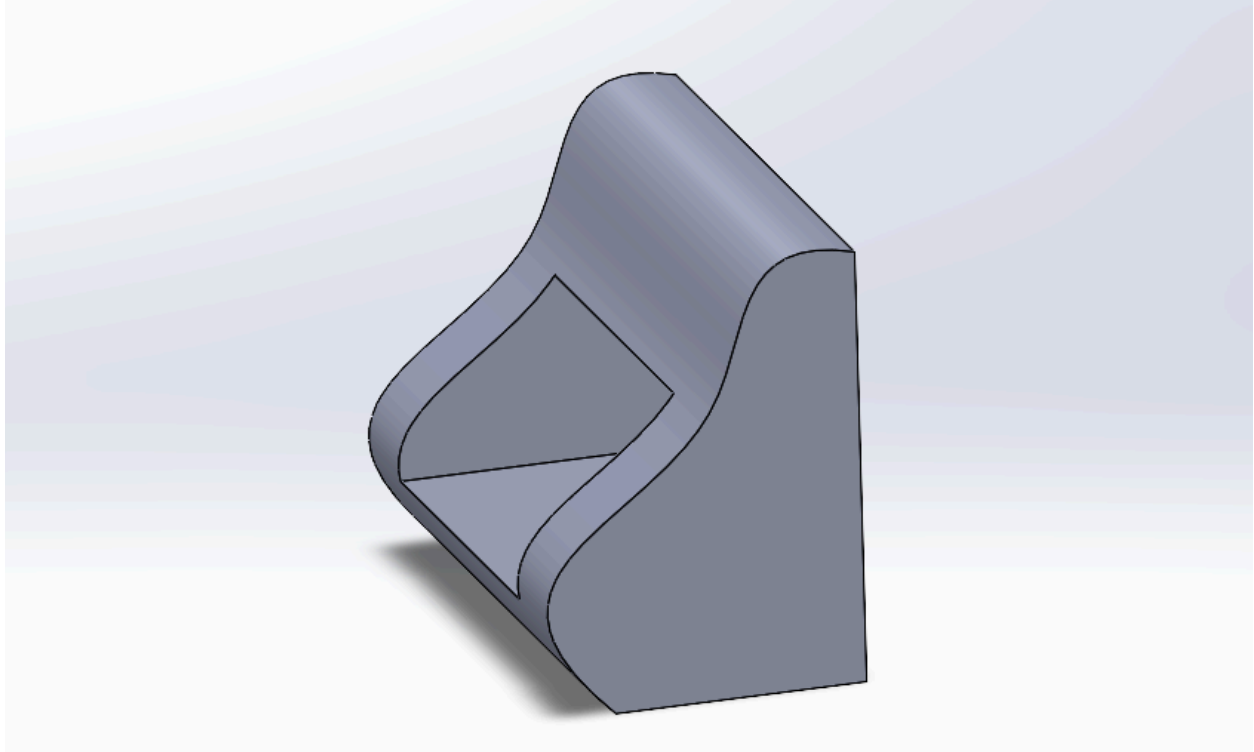


There are also **Up To Body** and **Mid Plane** options. These work the same way as they do when adding material with **Extruded Boss/Base**.



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.





3. Revolved Boss/Base

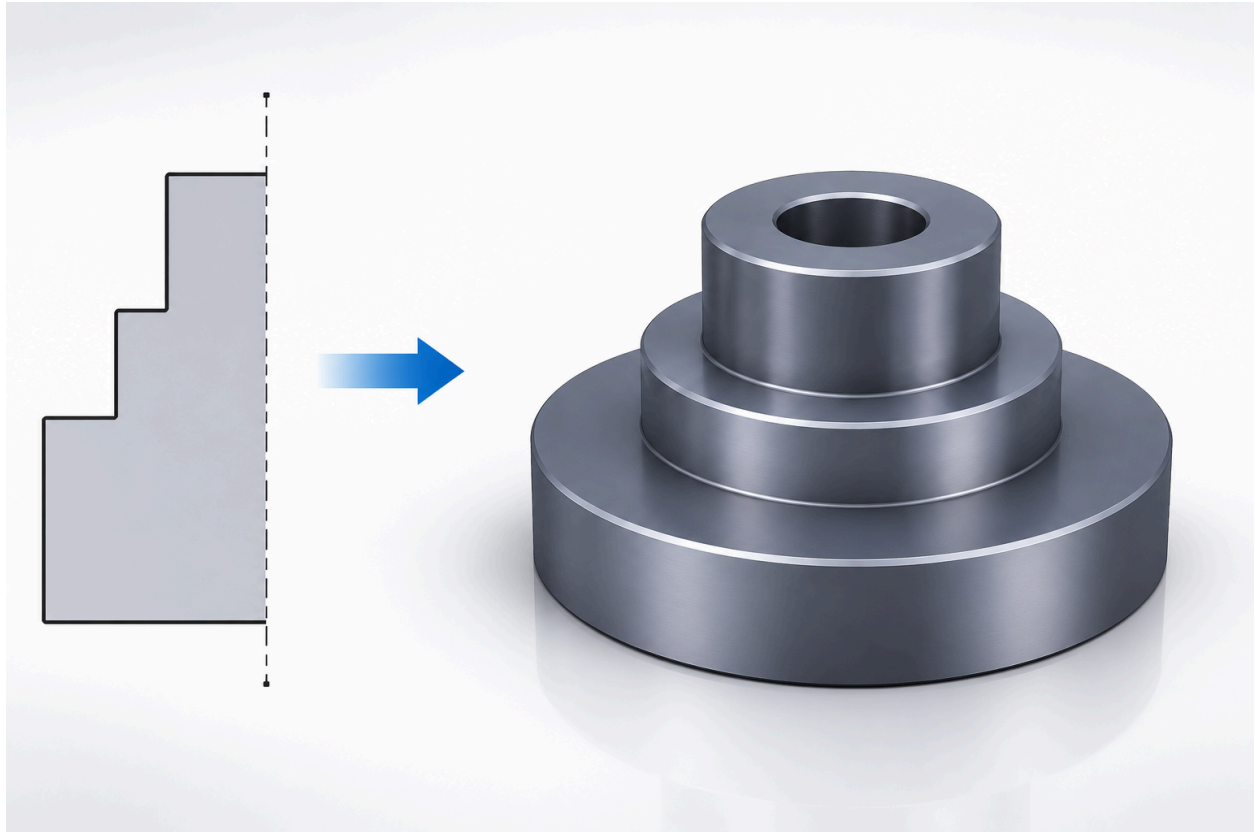
What it does: Creates a solid by revolving a sketch around an axis.

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

How it works:

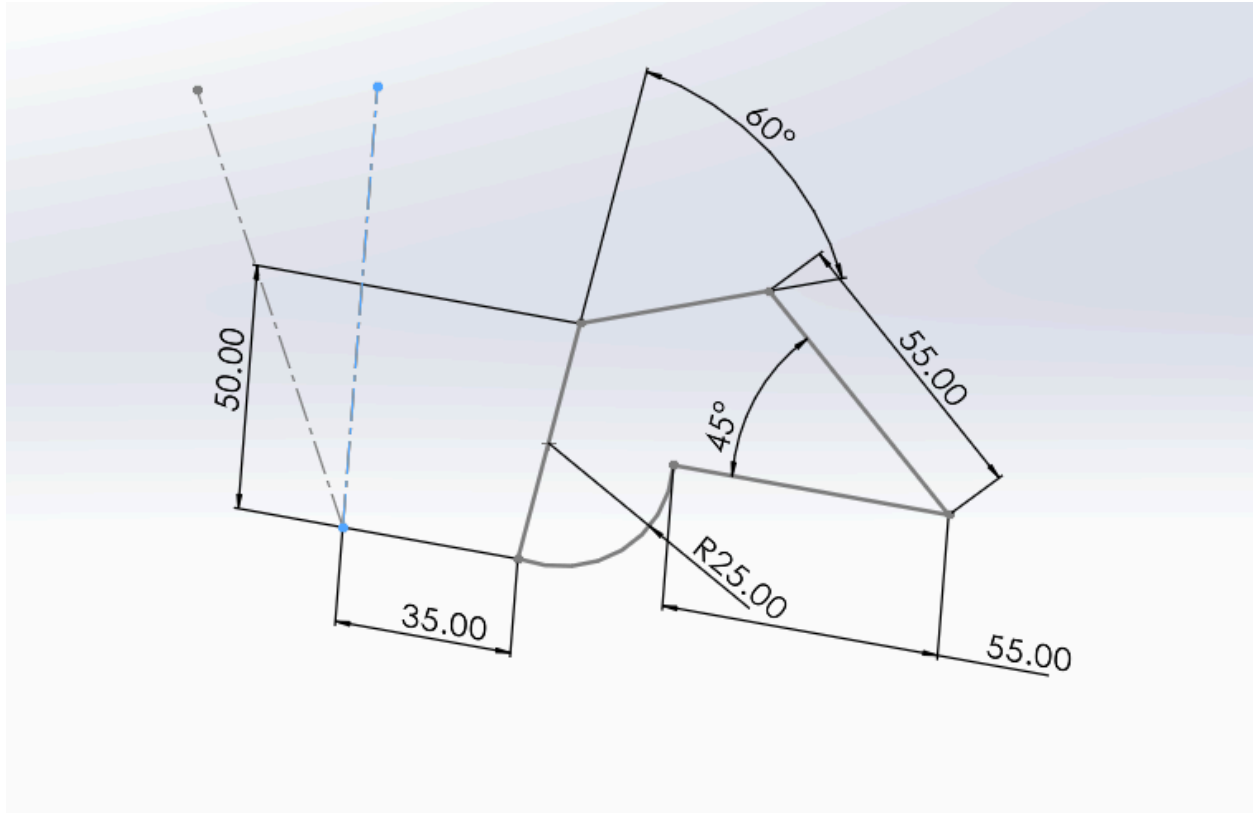
The **Revolved Boss/Base** tool is used to create a 3D feature by rotating a sketch around an axis.

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

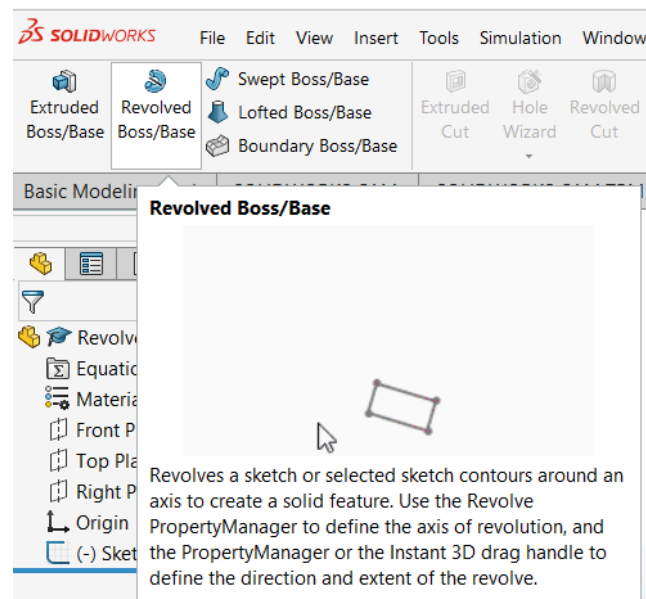


This is useful for creating round or symmetrical parts, such as shafts, wheels, pulleys, knobs, bottles, cylinders, and other rotational shapes.

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

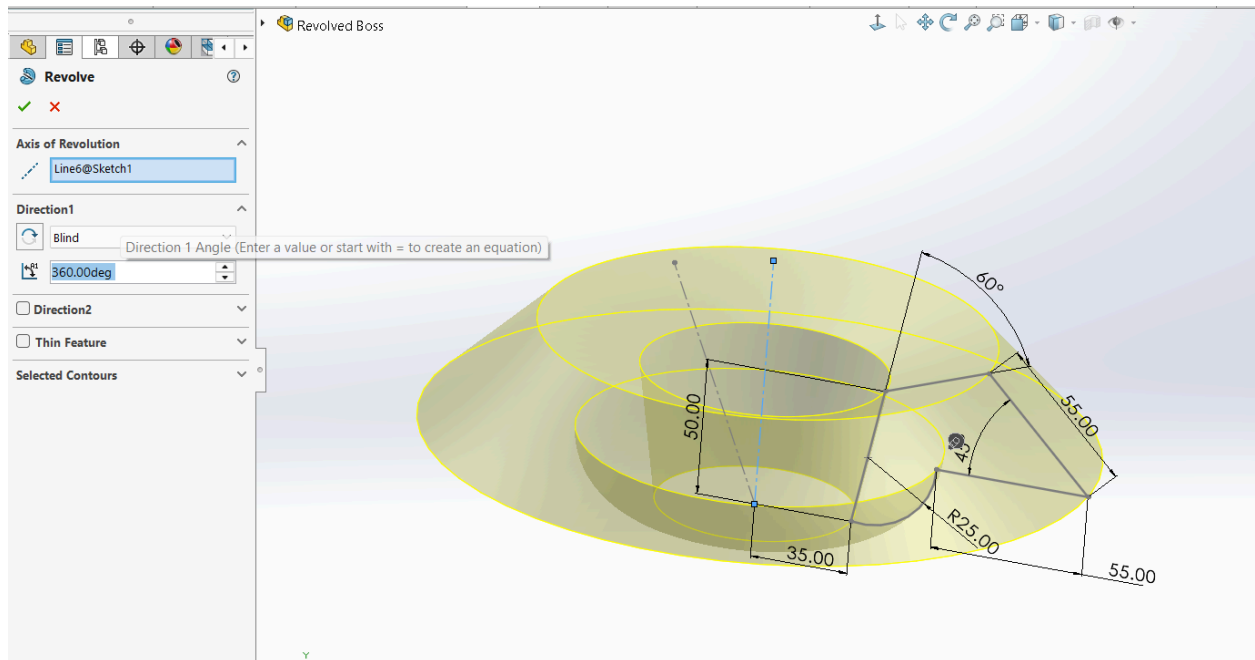


To access **Revolved Boss/Base**, go to the **Features** tab. Find the **Revolved Boss/Base** tool in the toolbar. Click **Revolved Boss/Base** to open the tool settings.

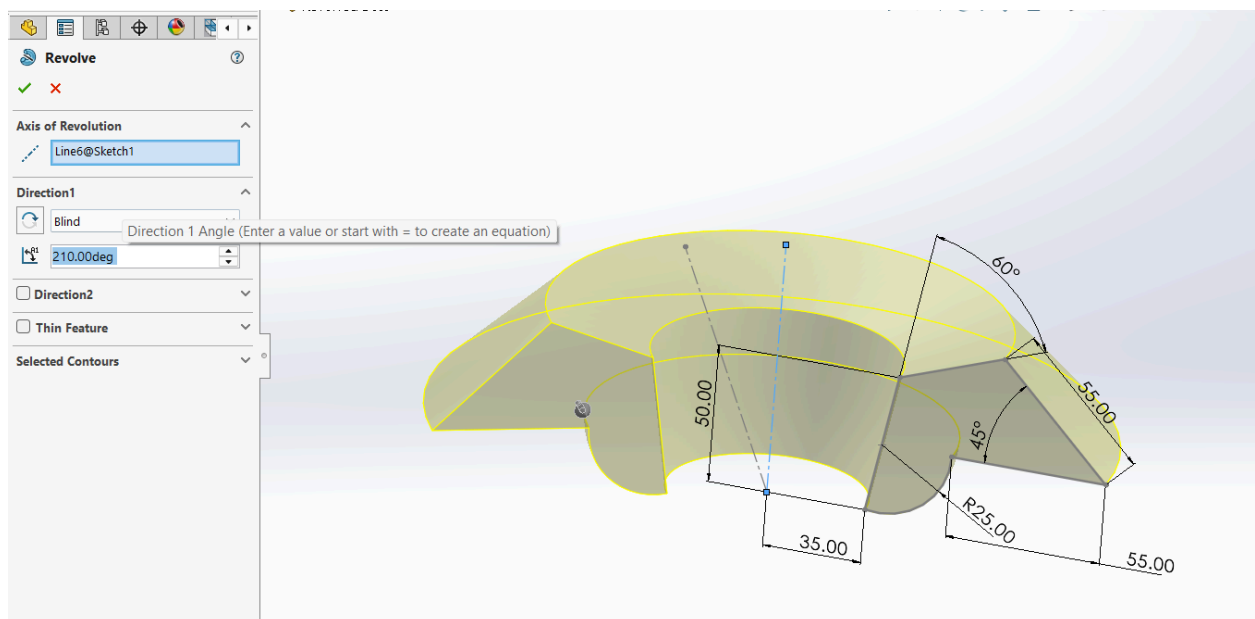


The **PropertyManager** will appear on the left side. Here, you can select the sketch profile, choose the axis of revolution, and define the rotation angle for the revolved feature.

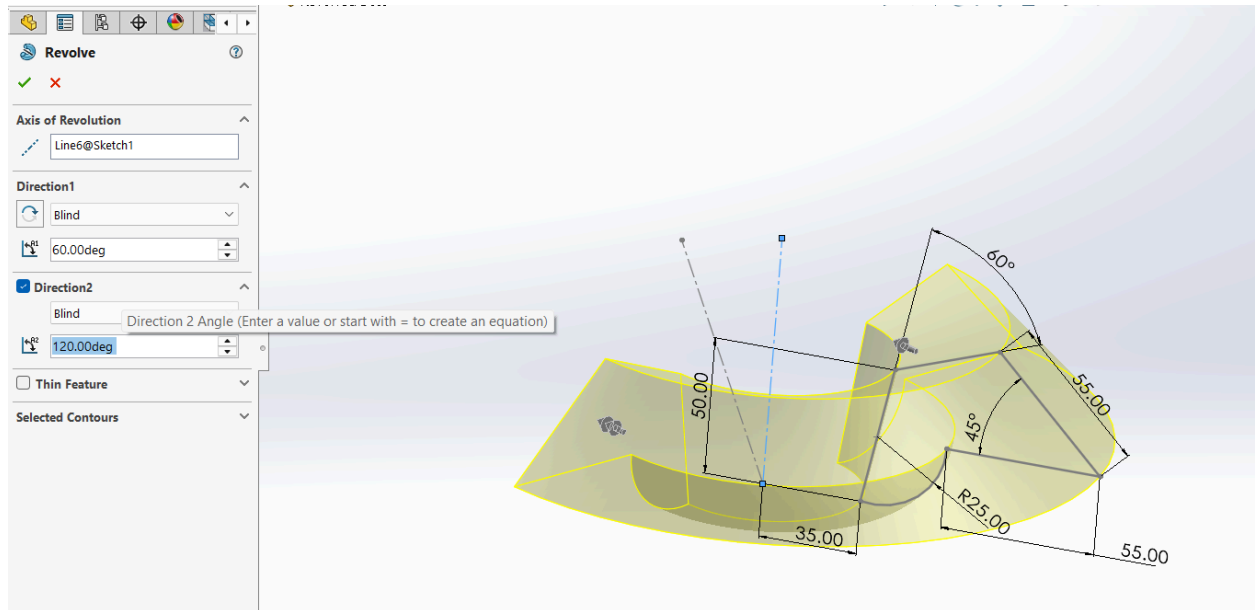
The **Axis of Revolution** is the axis we selected earlier. In the preview, you can see that the sketch rotates **360°** around this axis.



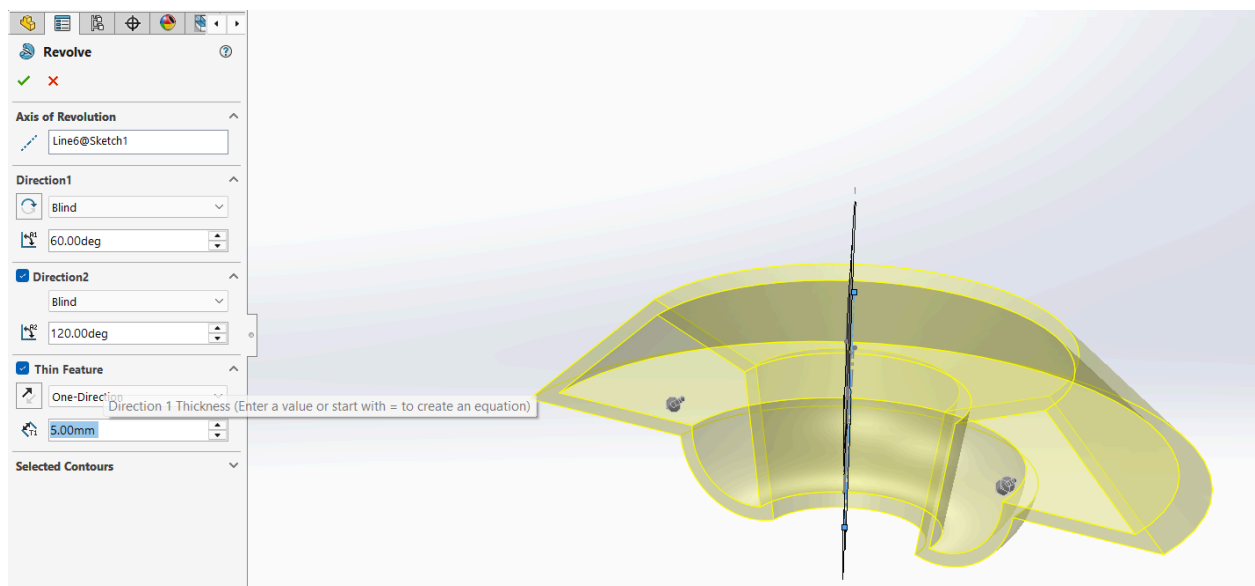
If you do not want a full **360°** revolution, you can change the angle value. For example, set the angle to **210°**. Now the sketch rotates only **210°** around the selected axis, creating this result.



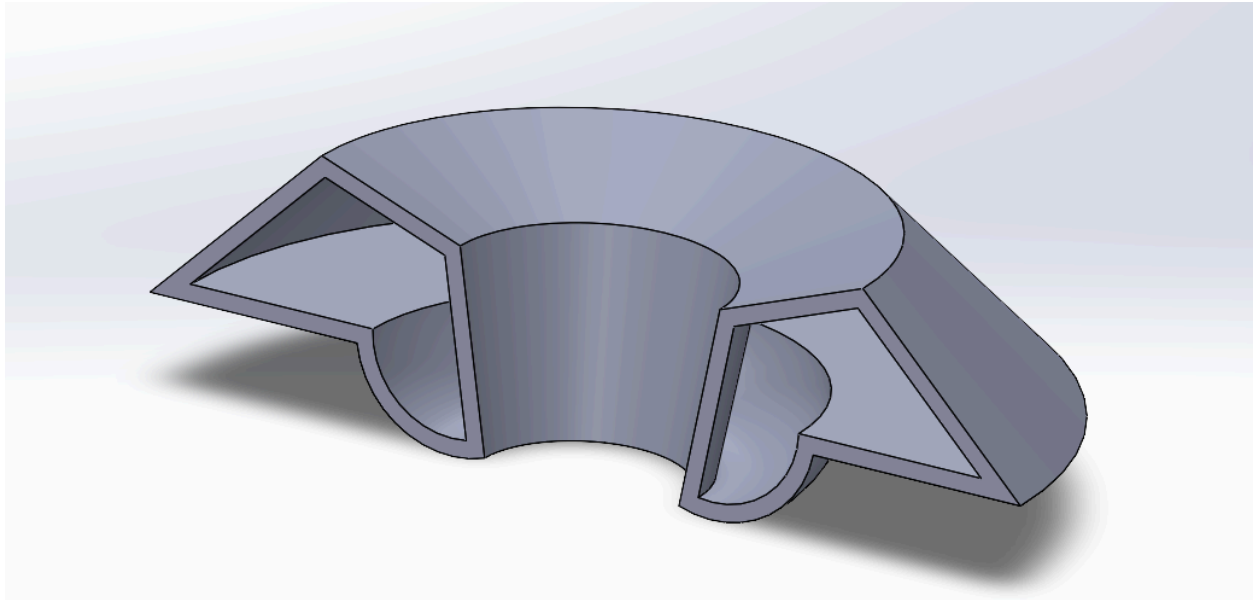
You can also turn on **Direction 2**. For **Direction 1**, set the angle to **60°**. For **Direction 2**, set the angle to **120°**. This is the final result.



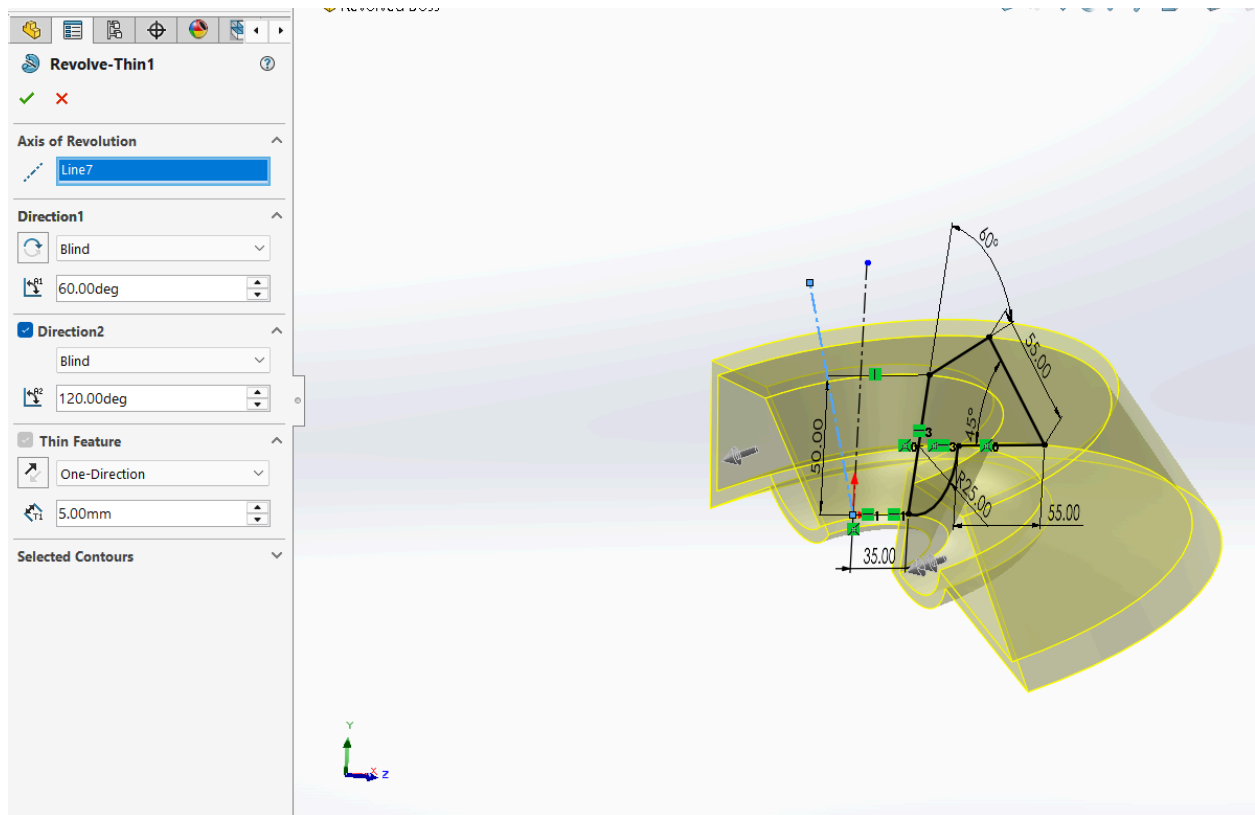
If you turn on the **Thin Feature** option, you can define a wall thickness for the revolved feature. In this example, set the thickness to **5 mm**. Now the feature creates a hollow body instead of a solid body.

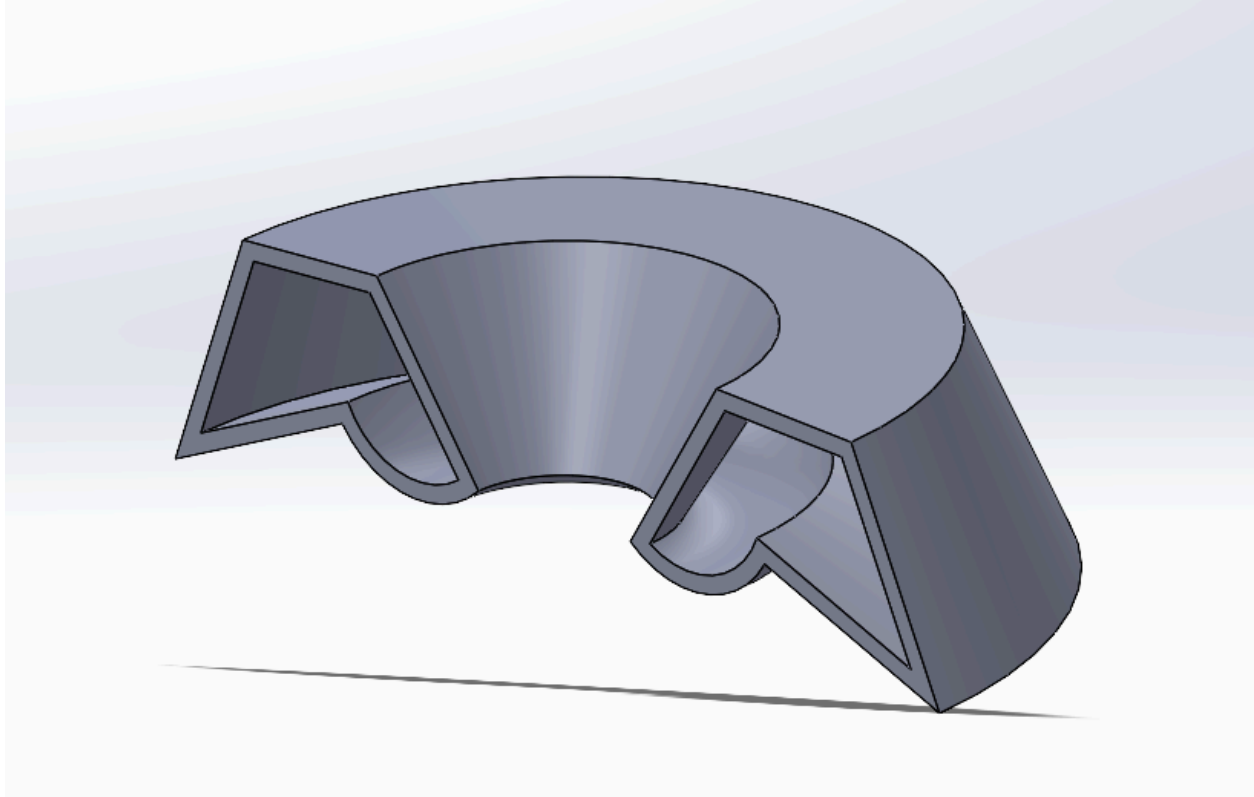


Click **OK**. This is the final result.



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.





4. Sweep

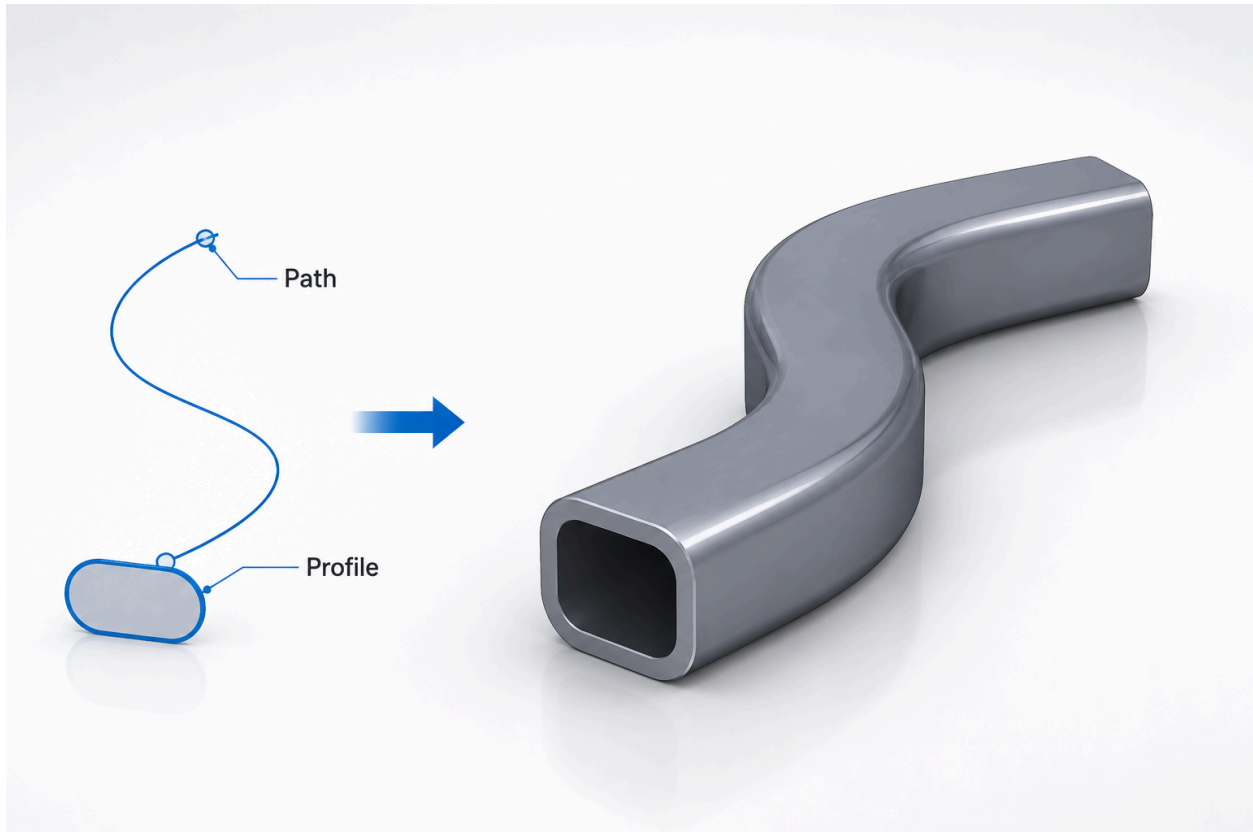
What it does: Creates geometry by moving a profile along a path.

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

How it works:

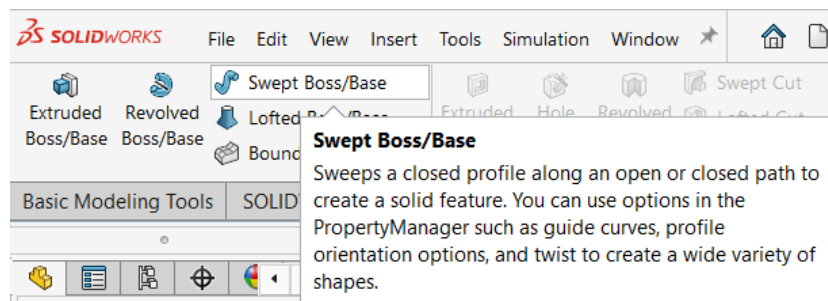
If you want to extrude a sketch profile along a curve or path, you can use the **Swept Boss/Base** tool.

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.

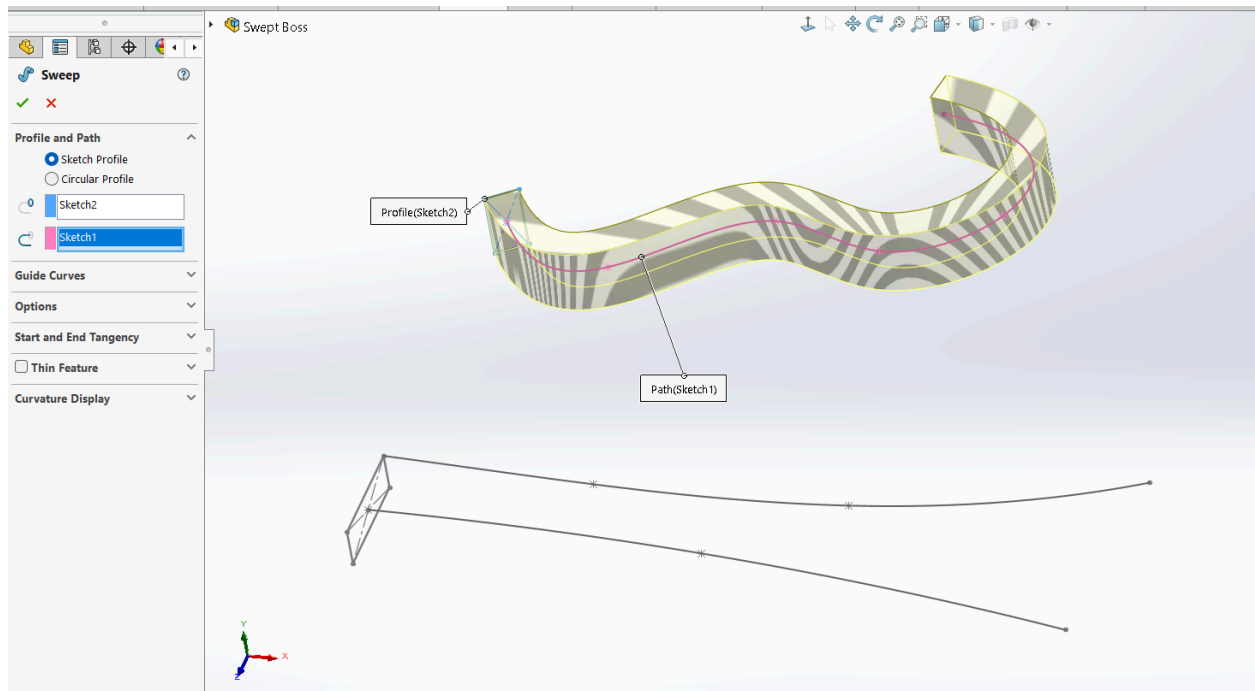


To create a sweep, you need two main elements: a **profile sketch** and a **path sketch**.

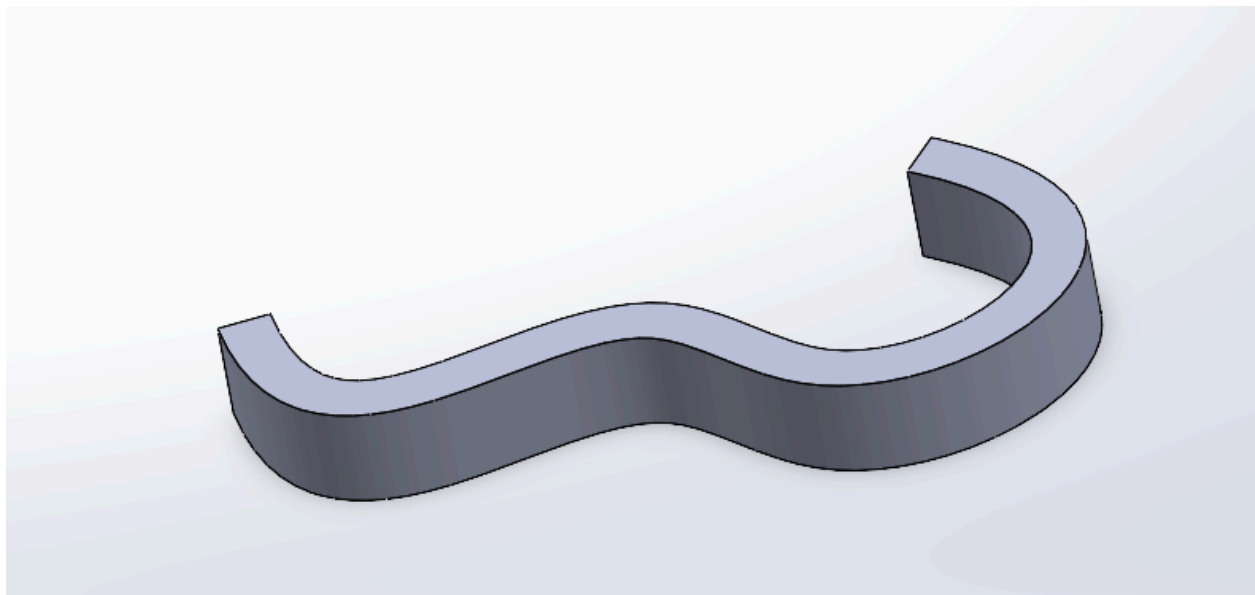
Go to the **Features** tab and select **Swept Boss/Base**.



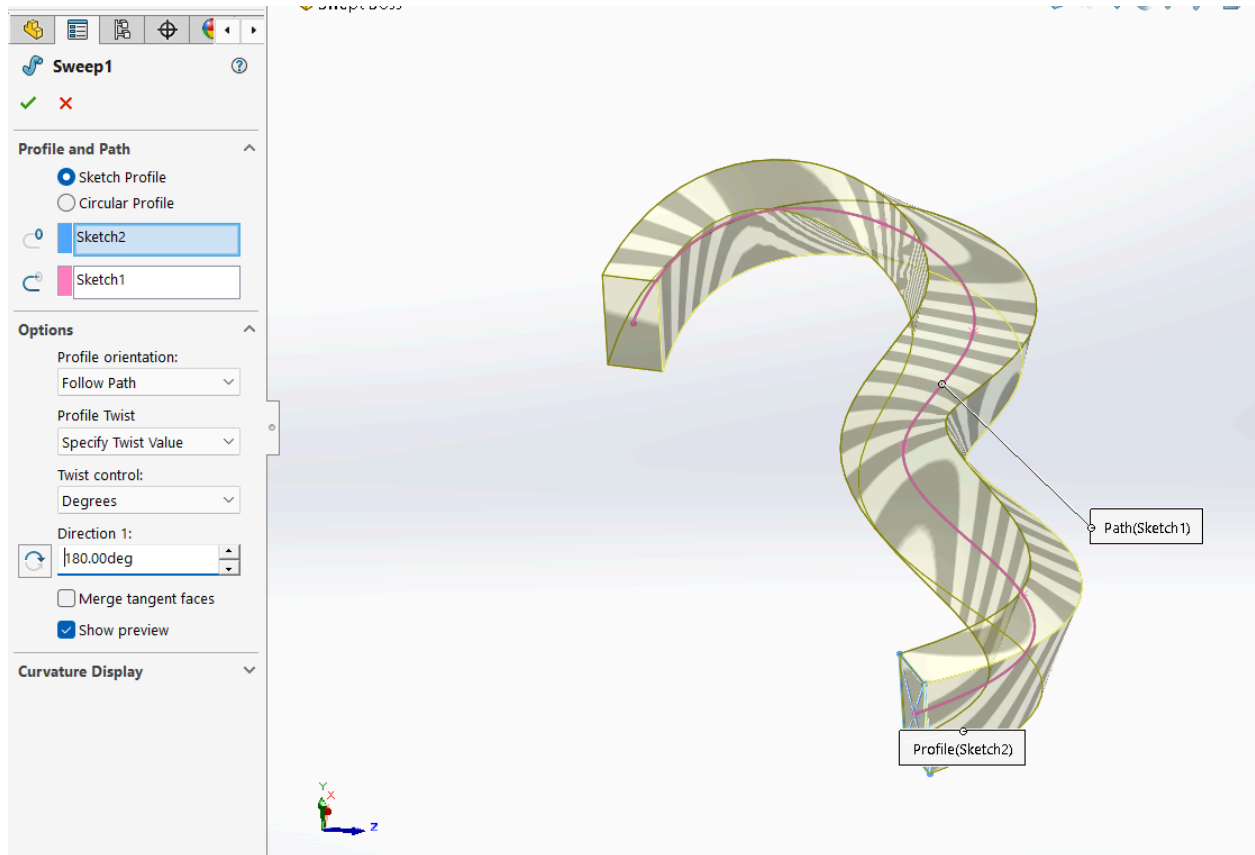
The sweep properties appear on the left side in the **Property Manager**. Select the first option, **Sketch Profile**. In the first selection box, select the profile sketch. In the second selection box, select the curve or path. You can see the sweep preview before confirming the feature.



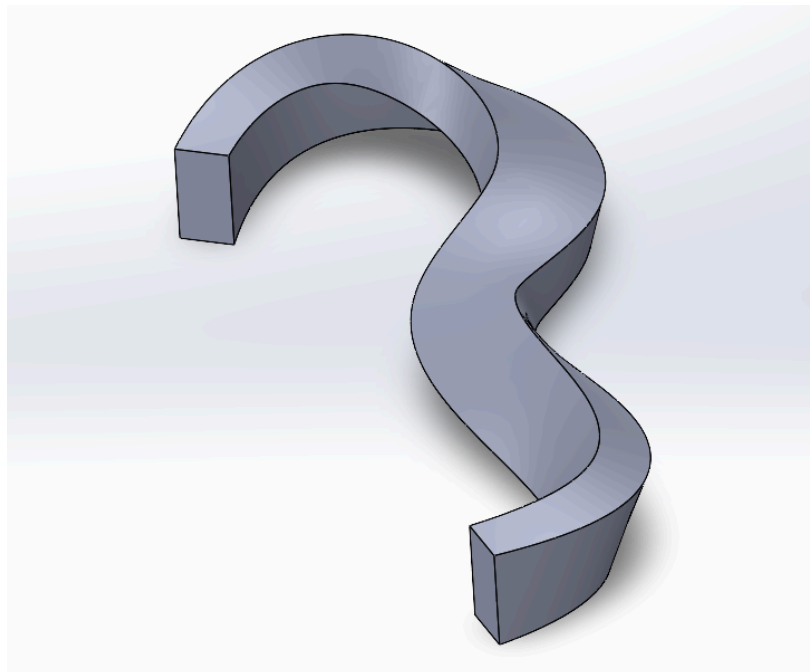
Click **OK**. This is the final result.



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.

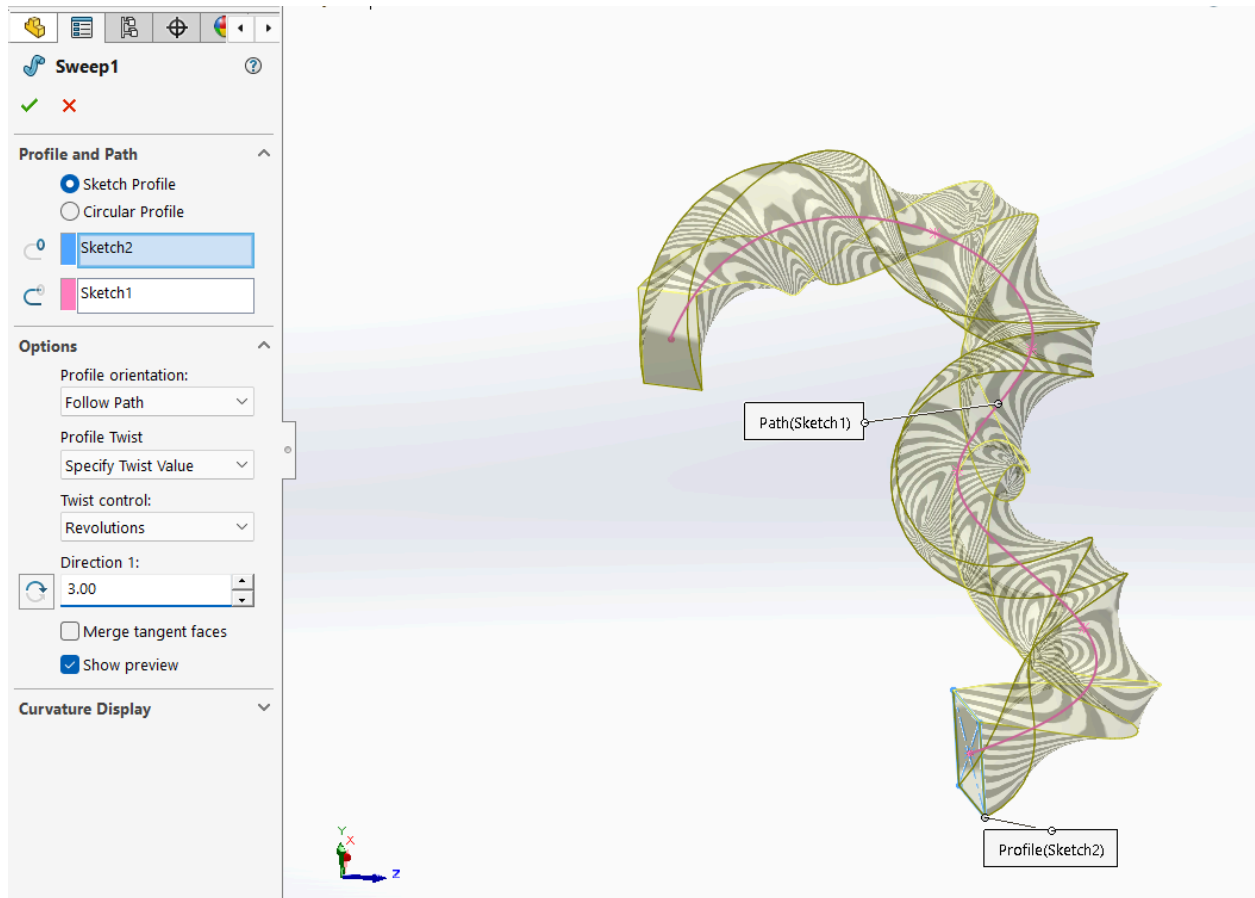


Click **OK**. This is the final result.

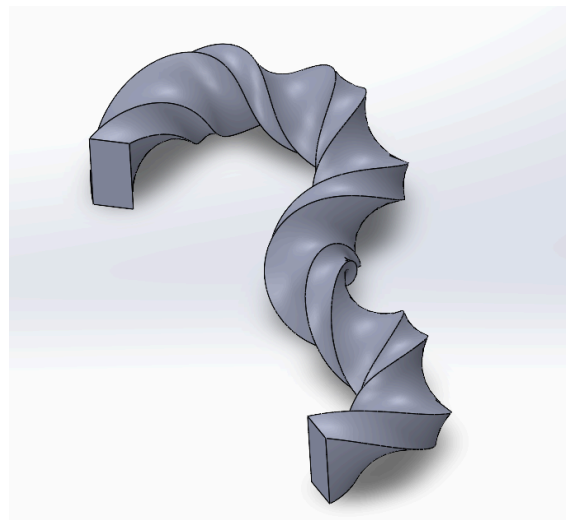


Go back and edit the same **Swept Boss/Base** feature.

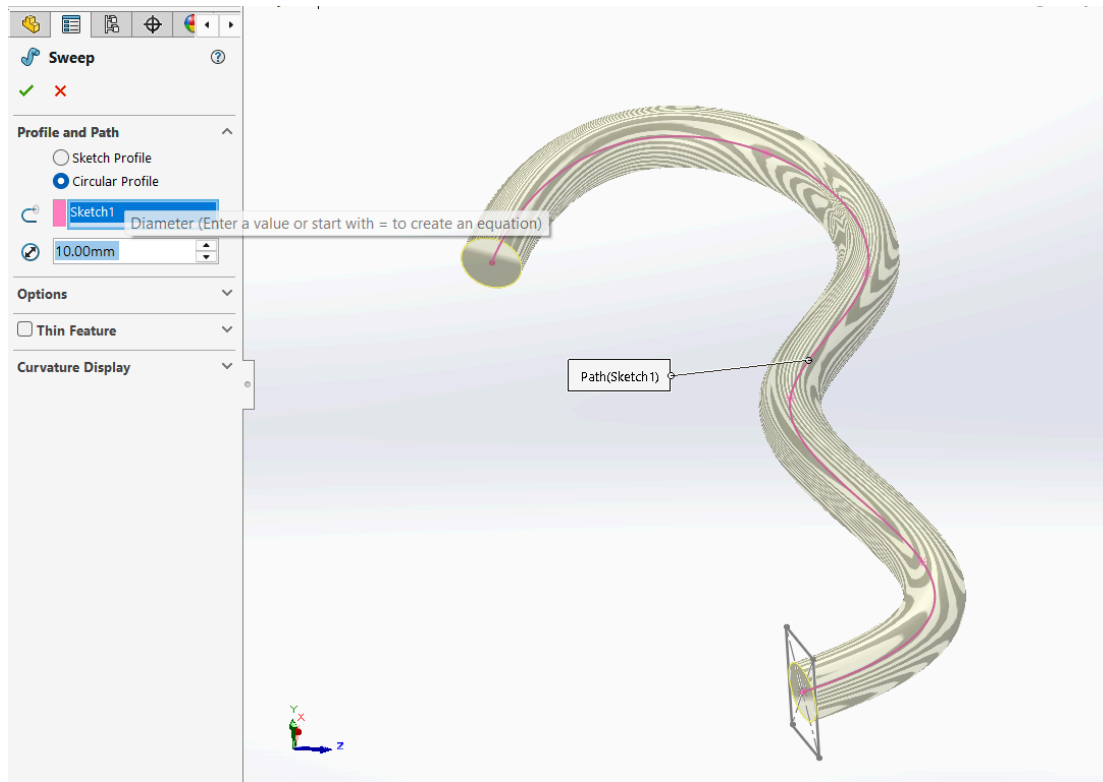
Instead of **Degrees**, select **Revolutions** for the twist control. For **Direction 1**, enter **3**. This means the profile will make **3 full revolutions** as it follows the curve. This is the preview.



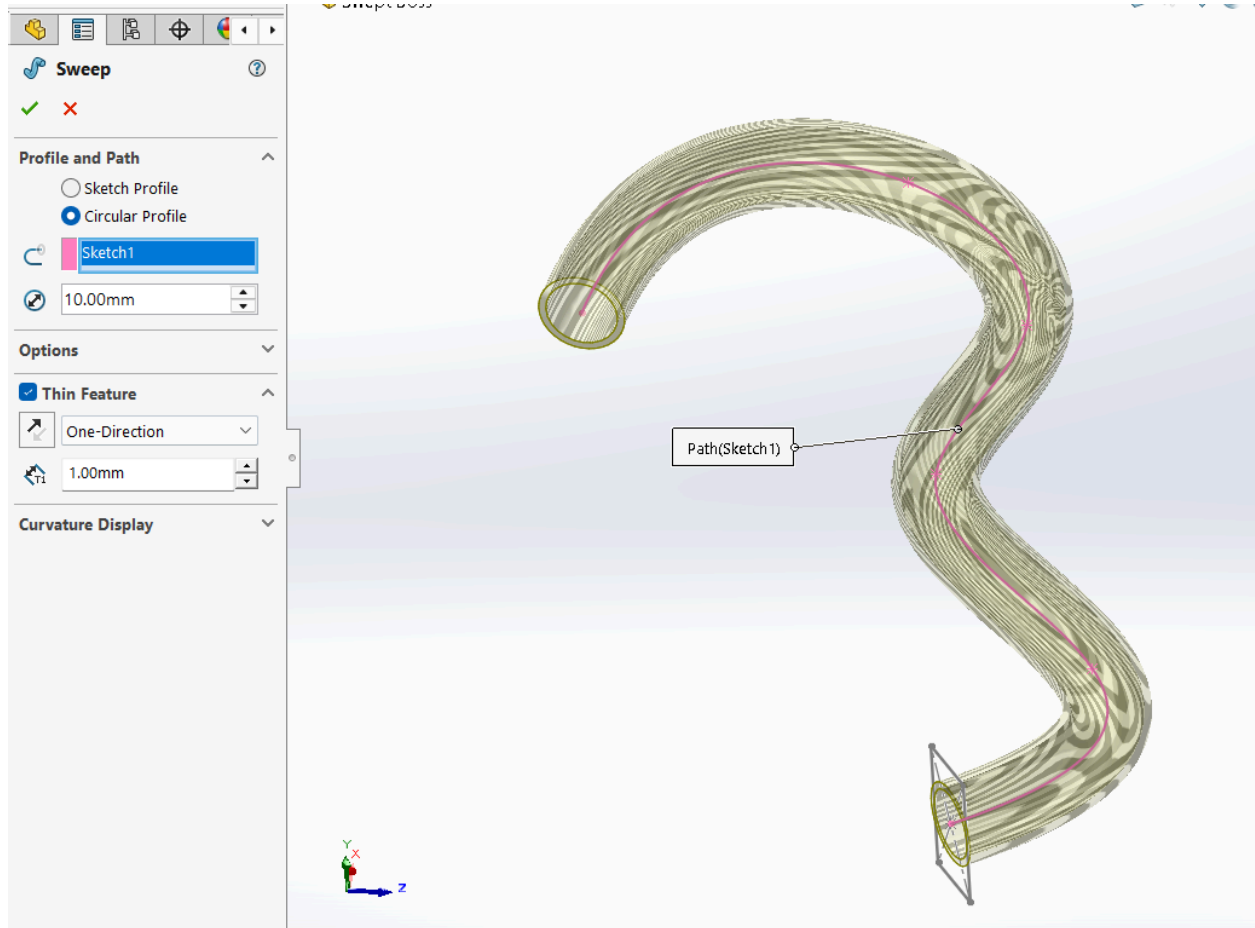
Click **OK**. This is the final result.



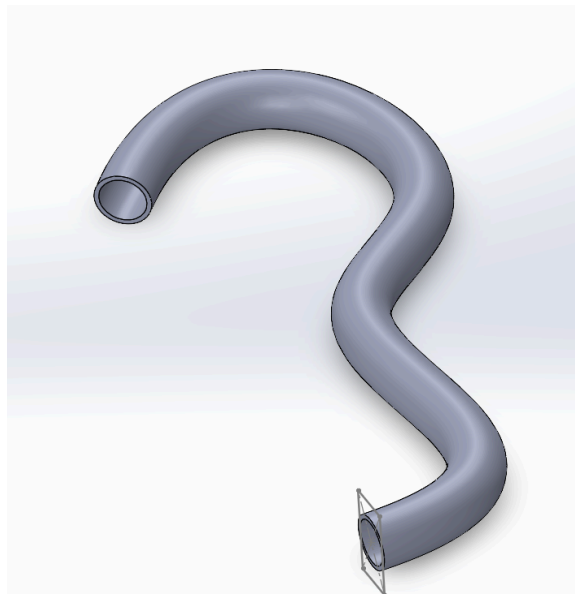
Now go back and edit the same **Swept Boss/Base** feature. Select the second option, **Circular Profile**. With this option, SOLIDWORKS creates a circular profile automatically, so you do not need to select a separate profile sketch. The circle is swept along the selected curve or path. Define the diameter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



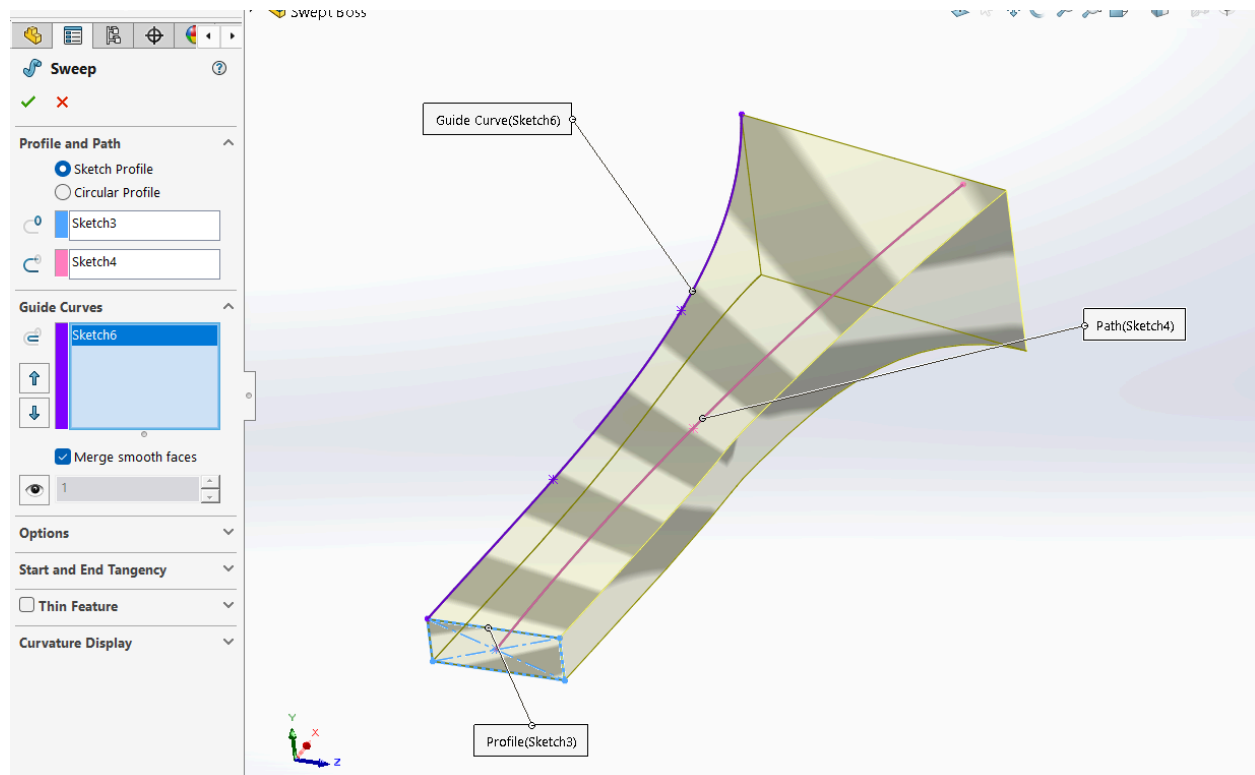
You can also use the **Thin Feature** option. Turn on **Thin Feature** and set the thickness to **1 mm**. Then click **OK**.



Now we have created a hollow curved tube.

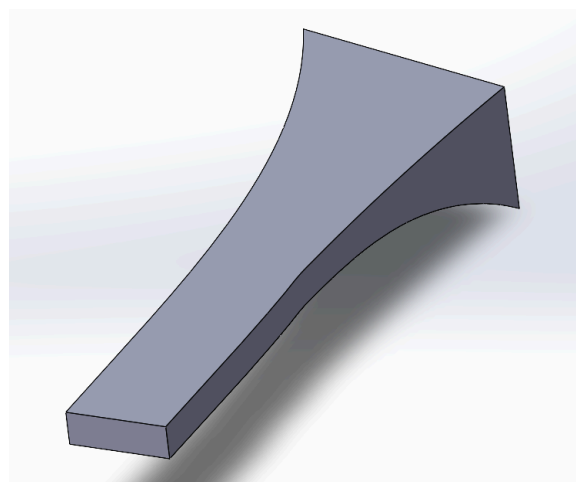


In the second example, we will also use **Guide Curves**. Select the profile sketch and the path curve as before. Then open the **Guide Curves** selection box and select the purple line. The guide curve helps control the shape of the sweep as the profile follows the path.



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.



5. Loft

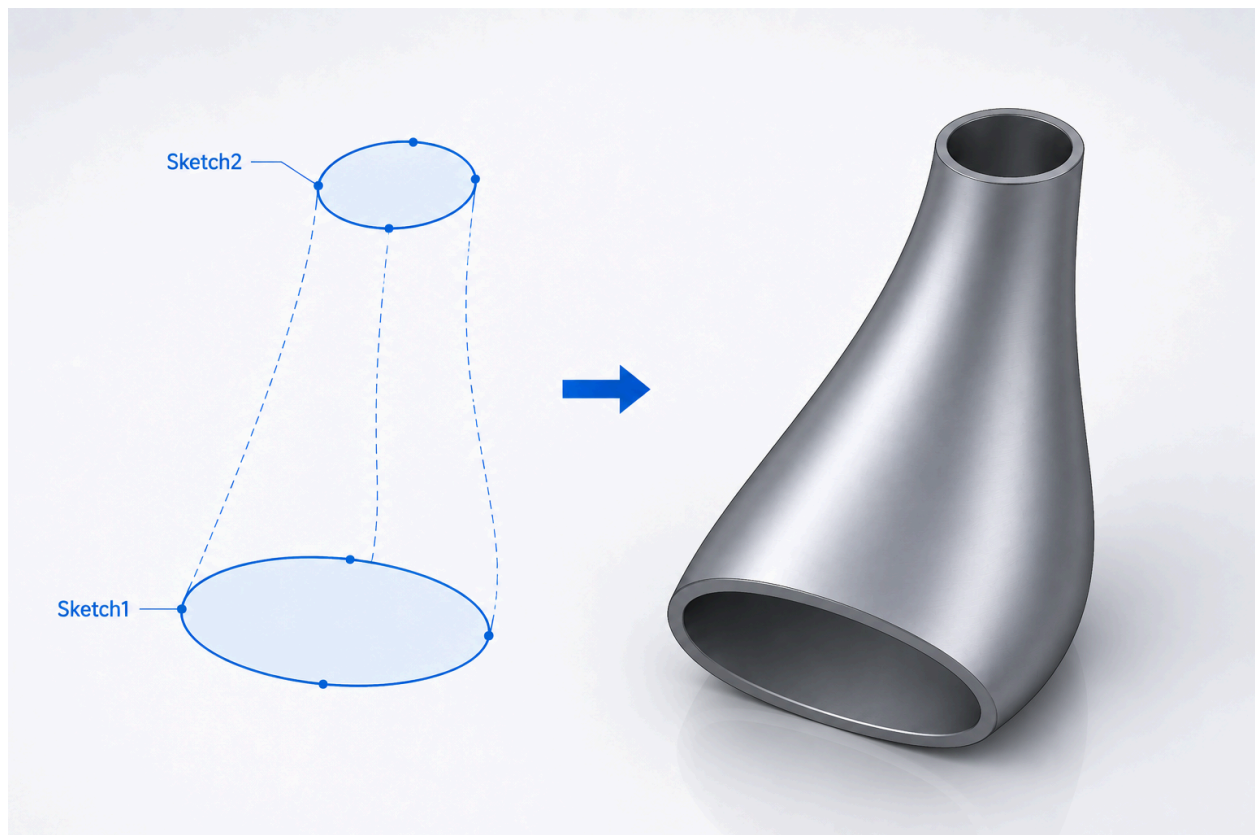
What it does: Creates a transition between two or more profiles.

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

How it works:

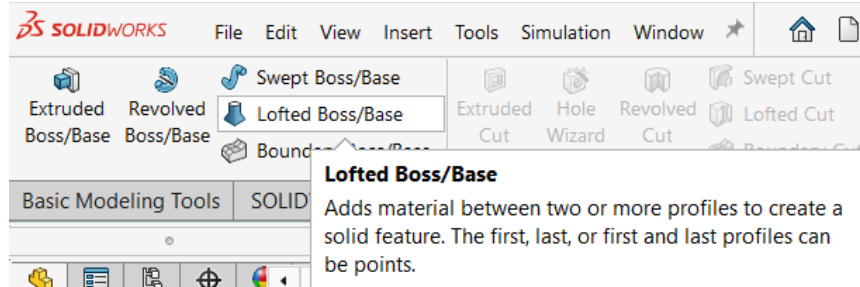
To connect two or more sketch profiles into one solid body, we use the **Lofted Boss/Base** tool.

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.



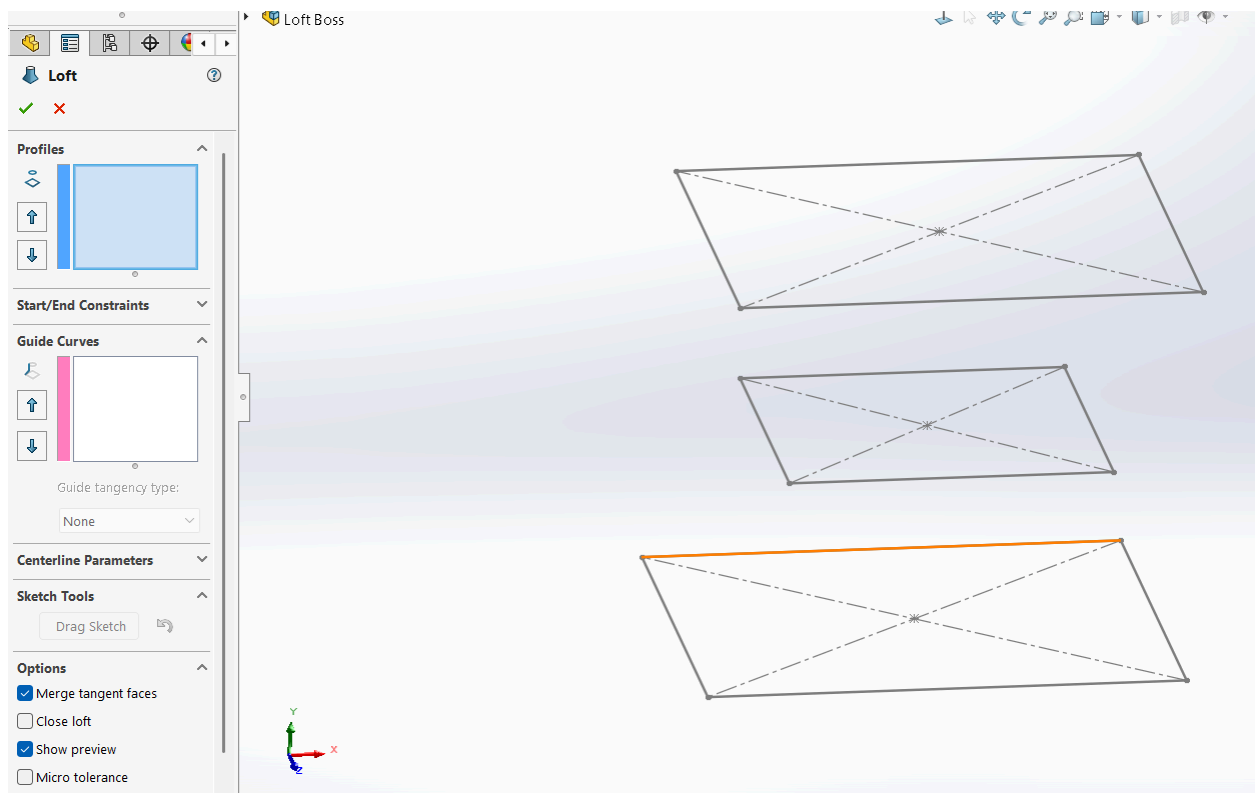
The sketches are usually created on different parallel planes.

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

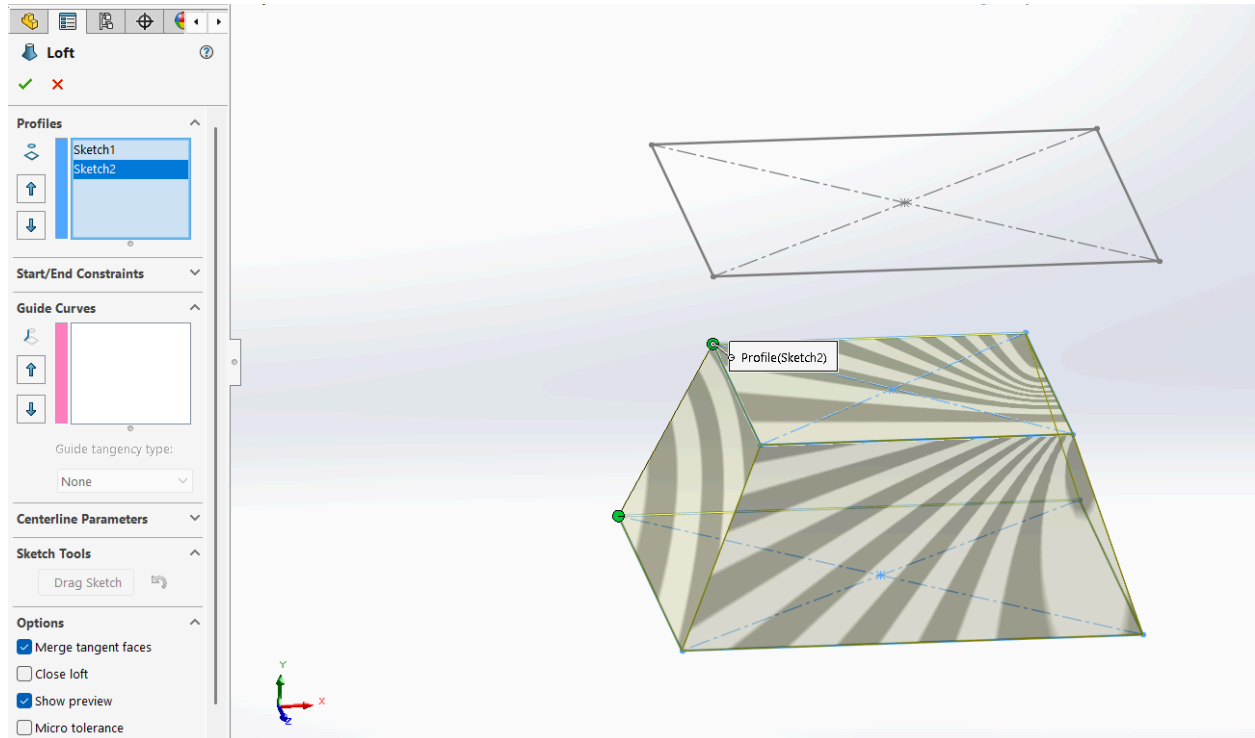


The loft properties appear on the left side in the **PropertyManager**.

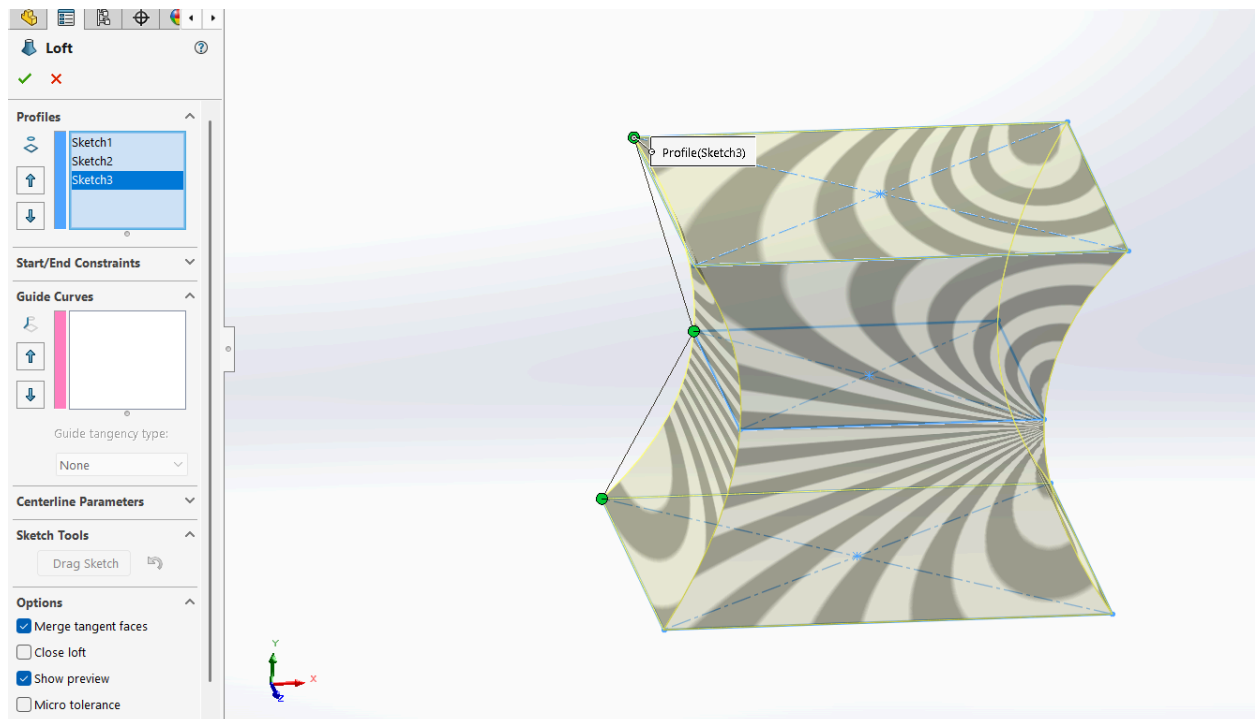
In the **Profiles** section, select the sketch profiles you want to connect. First, select the first profile.



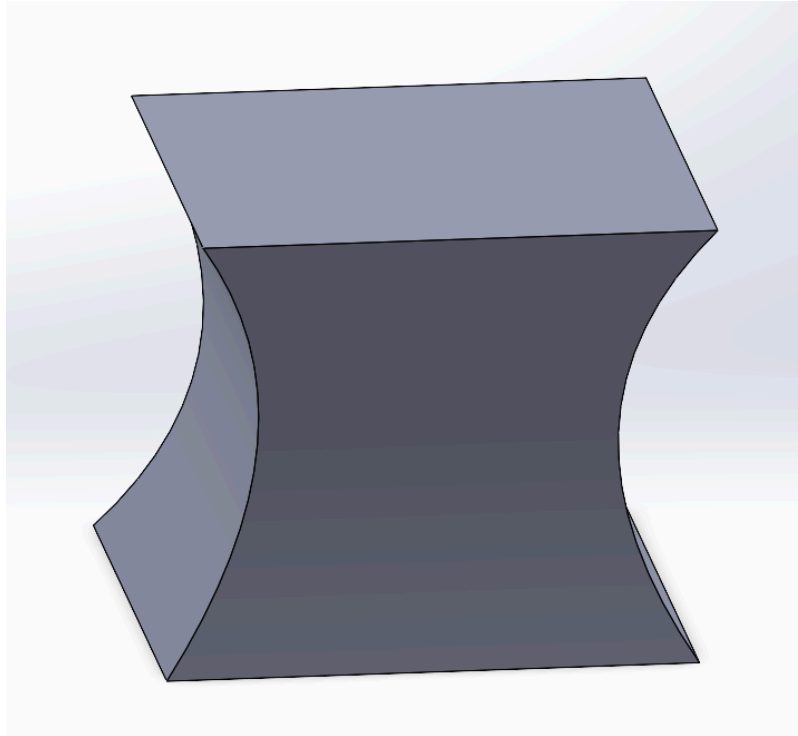
Now select the second profile. Make sure you select both profiles from the same side. If the profiles are selected from opposite sides, the loft preview can become twisted.



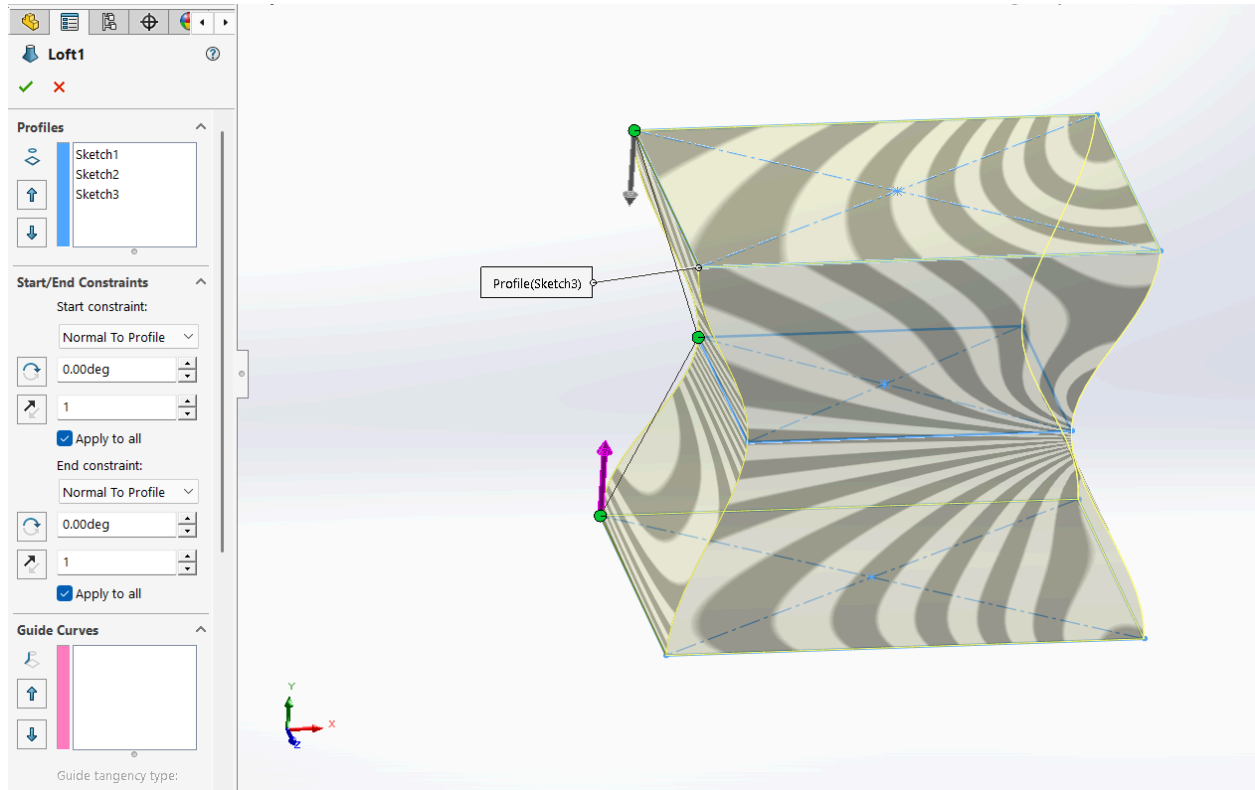
Now select the third profile. After selecting all three profiles, SOLIDWORKS creates a 3D body by blending the profiles together.



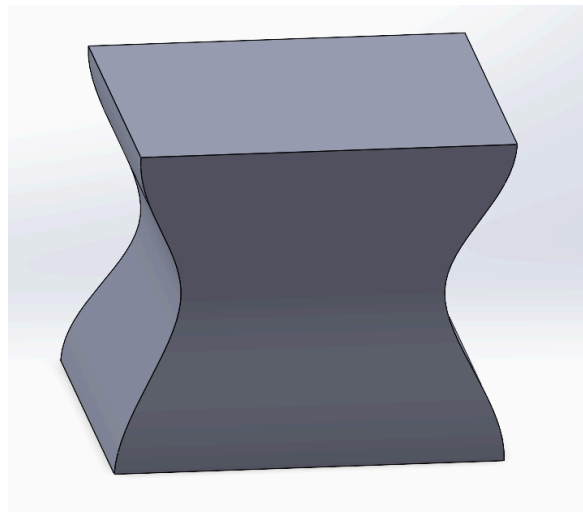
Click **OK**. This is the final result.



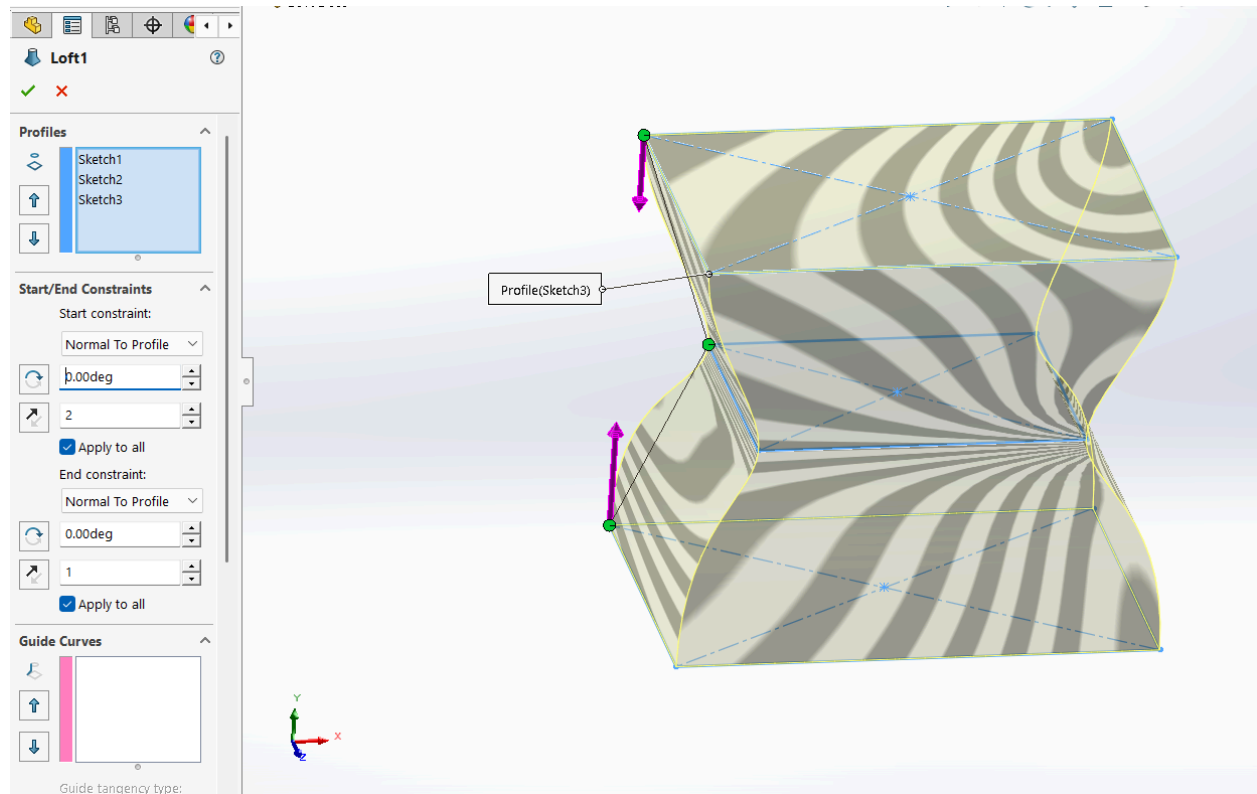
Now go back and edit the **Lofted Boss/Base** feature. Open **Start/End Constraints**. For **Start Constraint**, select **Normal To Profile** and set the value to **1**. For **End Constraint**, also select **Normal To Profile** and set the value to **1**. Now you can see that the loft starts and ends 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.



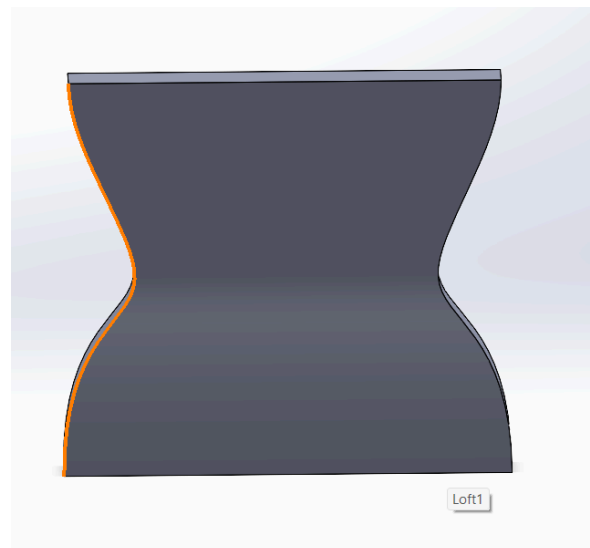
Click **OK**. This is the final result.



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.



6. Fillet

What it does: Rounds selected model edges.

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

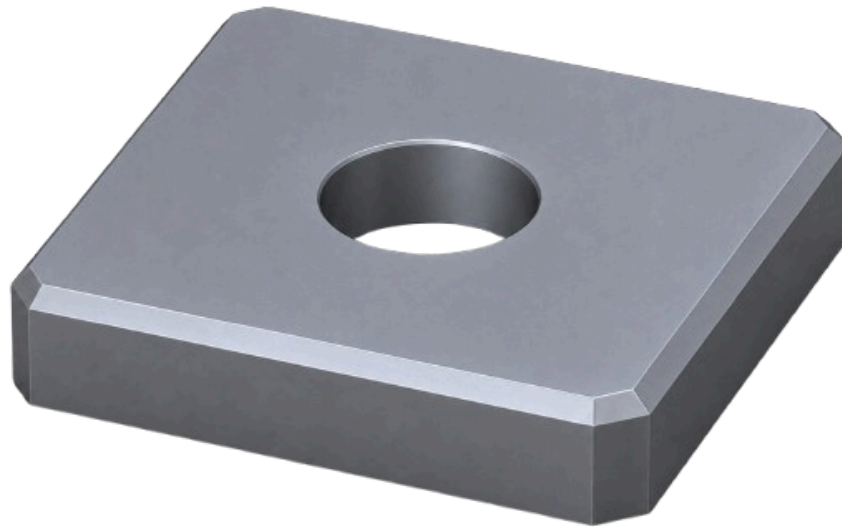
How it works:

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

Fillet creates a rounded edge by applying a radius.

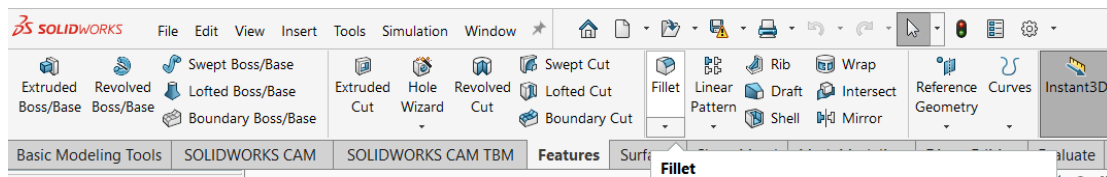


Chamfer creates a straight angled edge by applying a distance and angle, or two distance values.

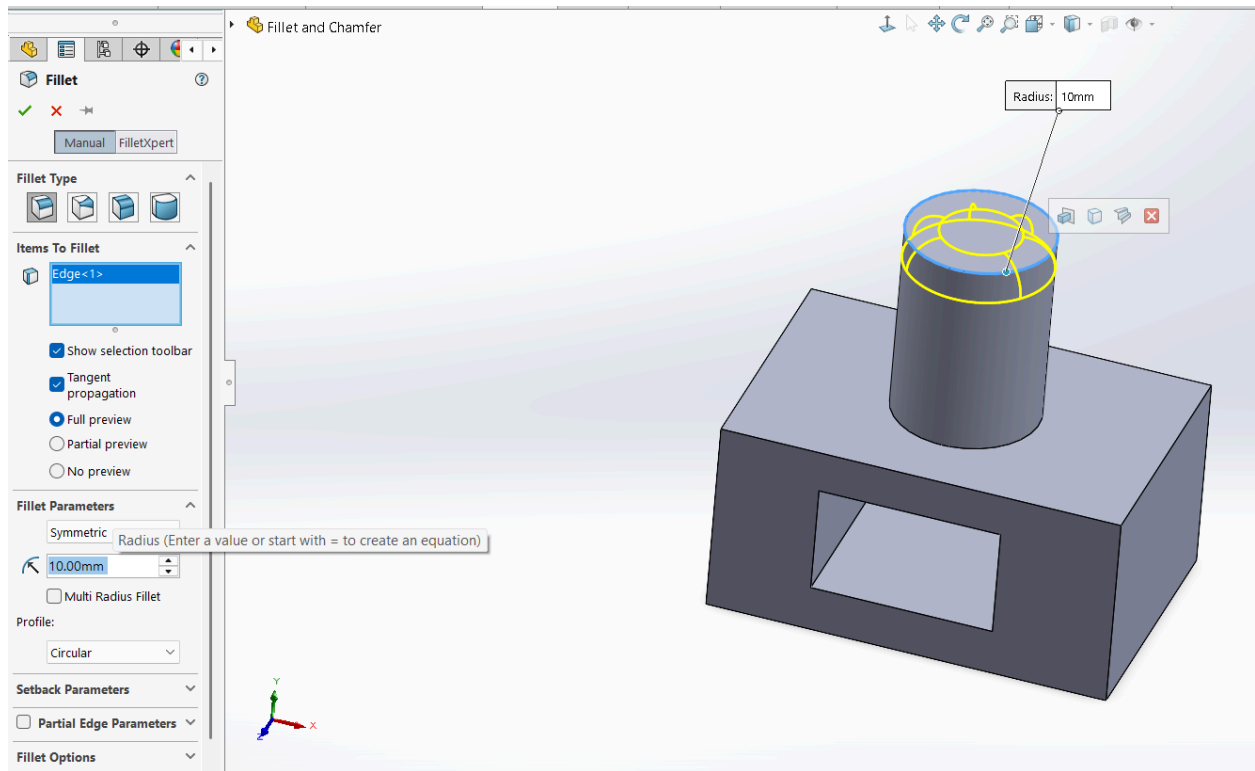


Both tools are useful for making parts look more realistic, removing sharp corners, and preparing models for manufacturing.

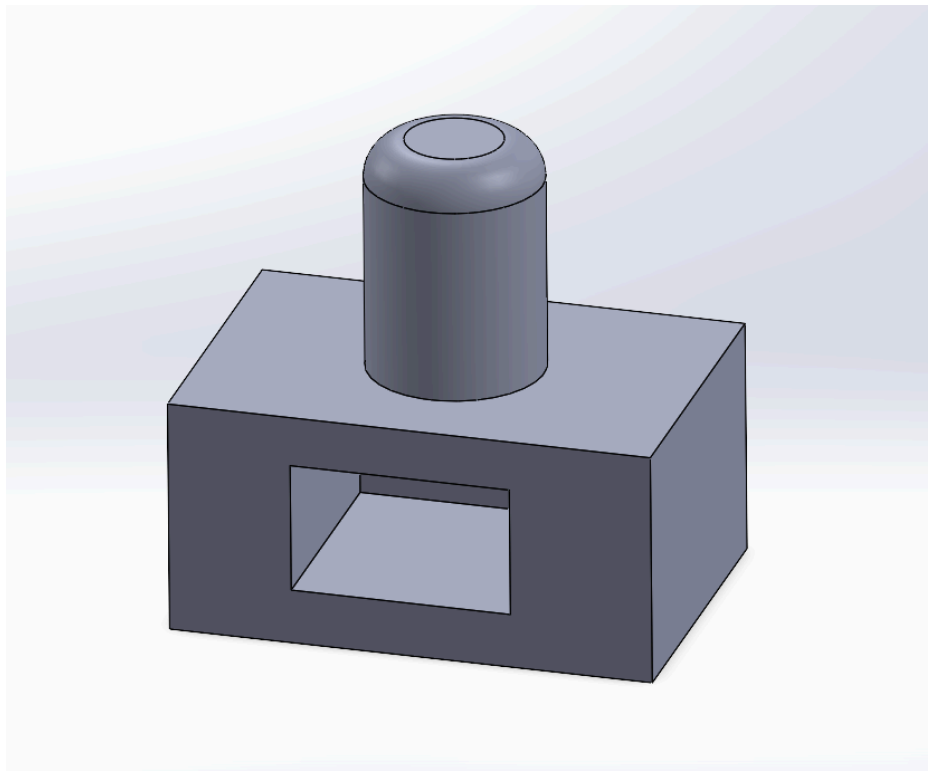
Let's start with **Fillet**. Go to the **Features** tab and select **Fillet**.



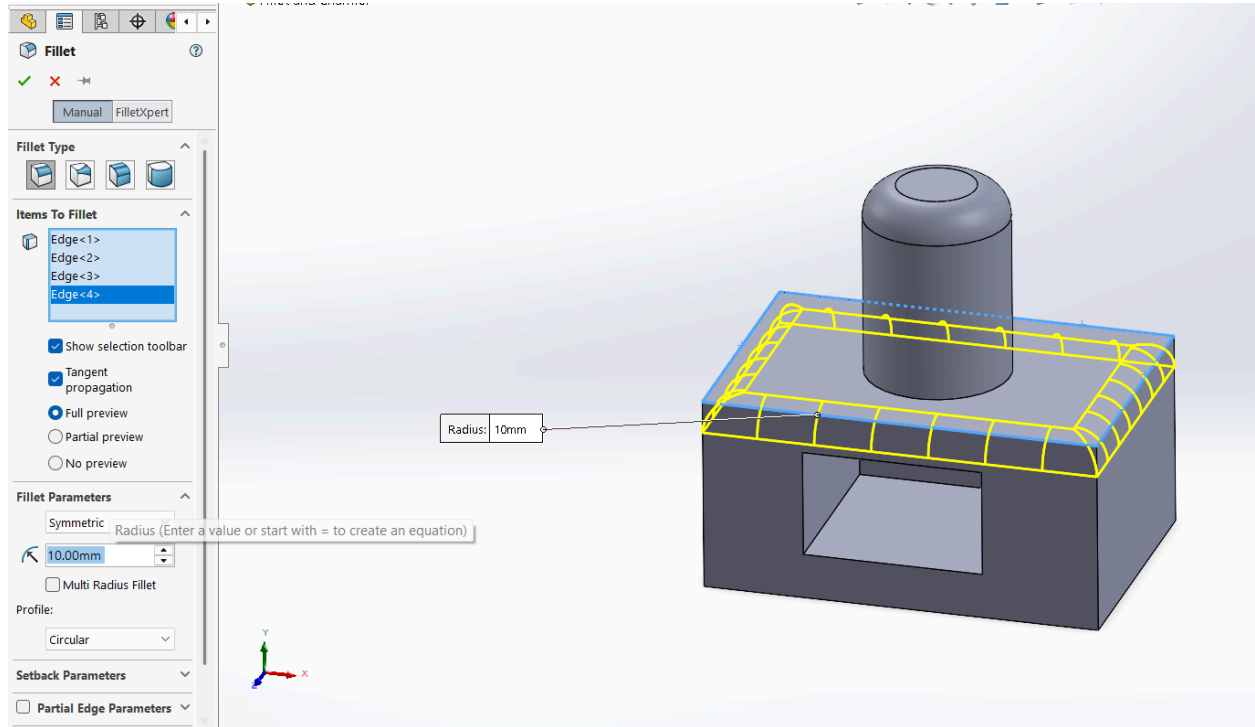
The fillet properties appear on the left side in the **PropertyManager**. Here, you can define the fillet radius. Leave the radius set to **10 mm**, then select the edge shown below. You can see the fillet preview before confirming the feature.



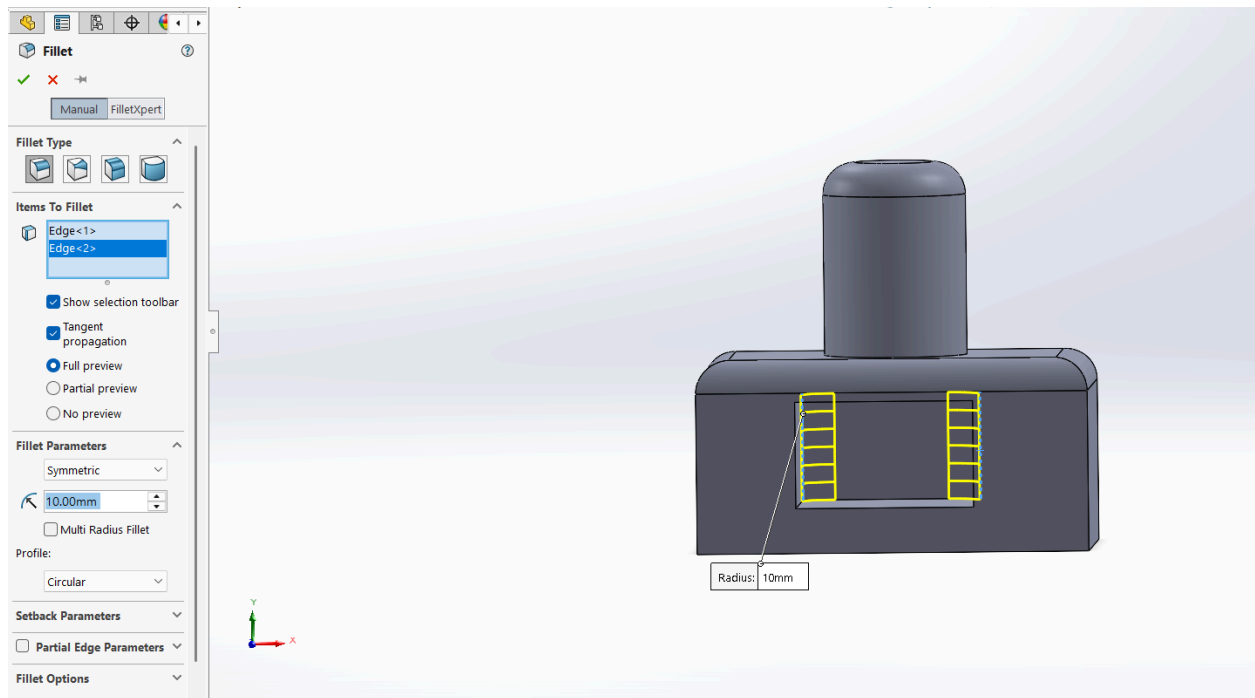
Here is the result with a rounded edge using a **10 mm** fillet radius.

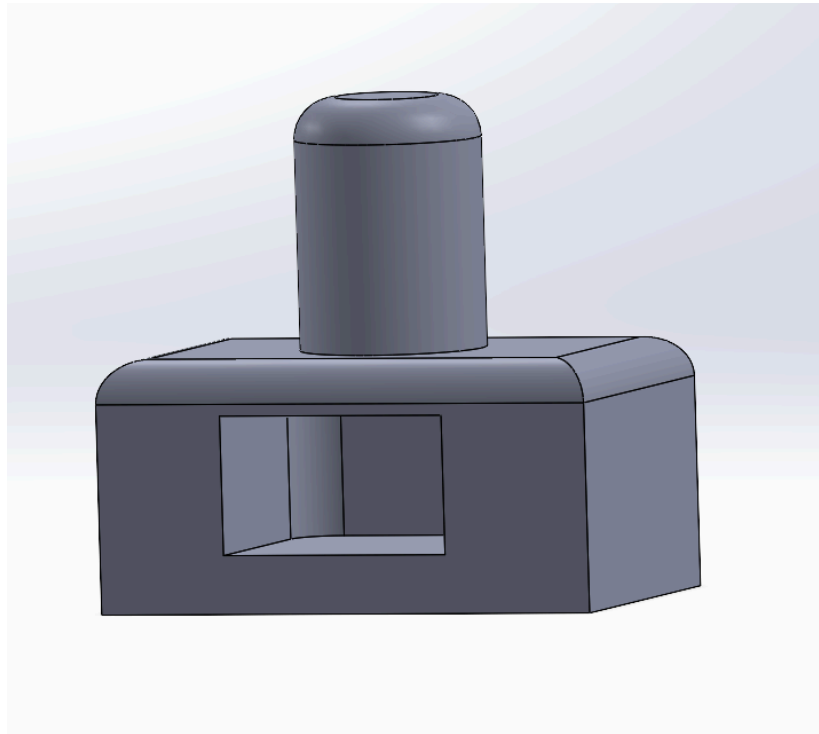


Select **Fillet** again, then select the edge shown below.



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





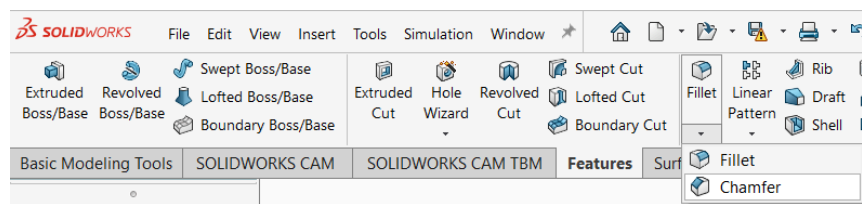
7. Chamfer

What it does: Creates an angled or beveled edge.

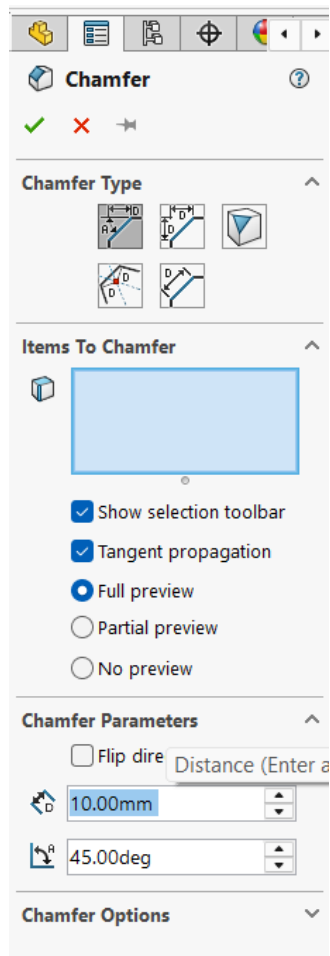
Use it for: Edge preparation, assembly clearance, and manufacturing details.

How it works:

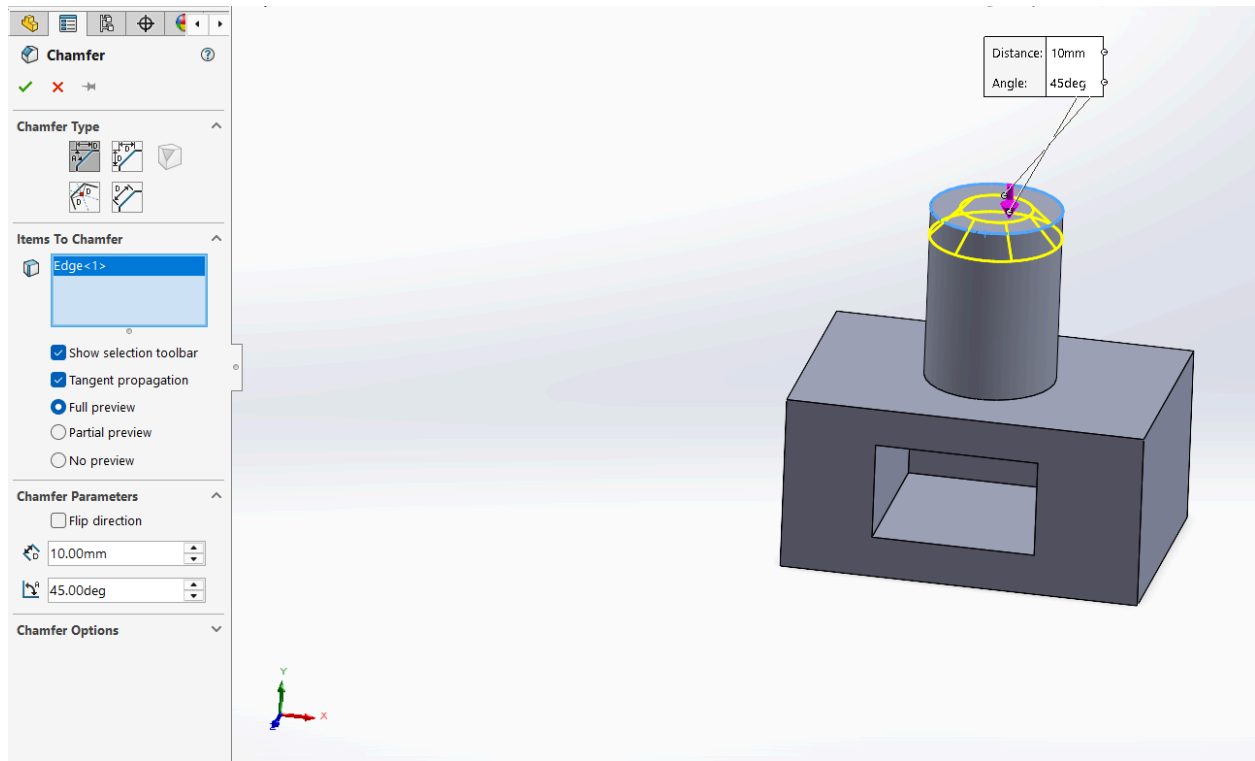
The second tool is **Chamfer**. To access **Chamfer**, click the small arrow below **Fillet**, then select **Chamfer**.



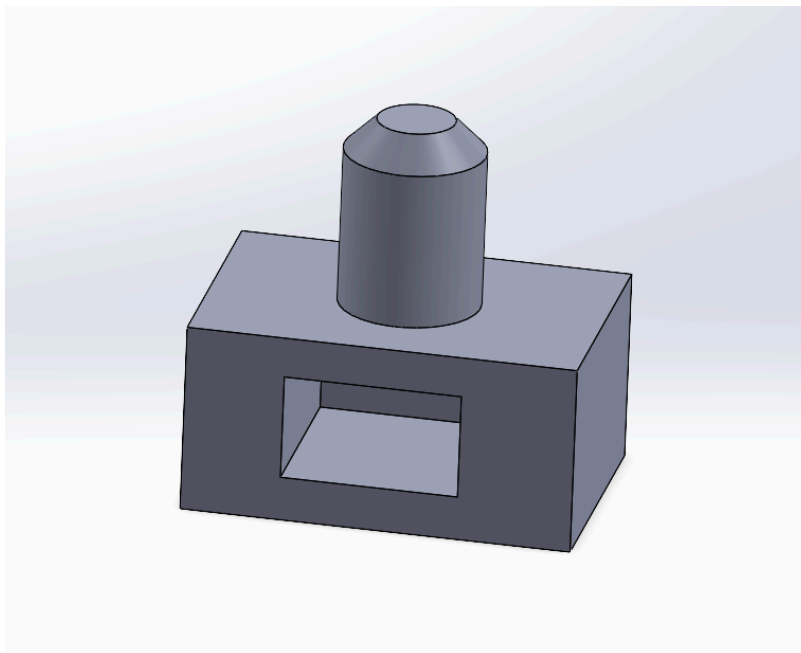
With **Chamfer**, we create a straight cut on the selected edge. To define the chamfer, you need to set the distance and angle. In this example, leave the distance set to **10 mm** and the angle set to **45°**.



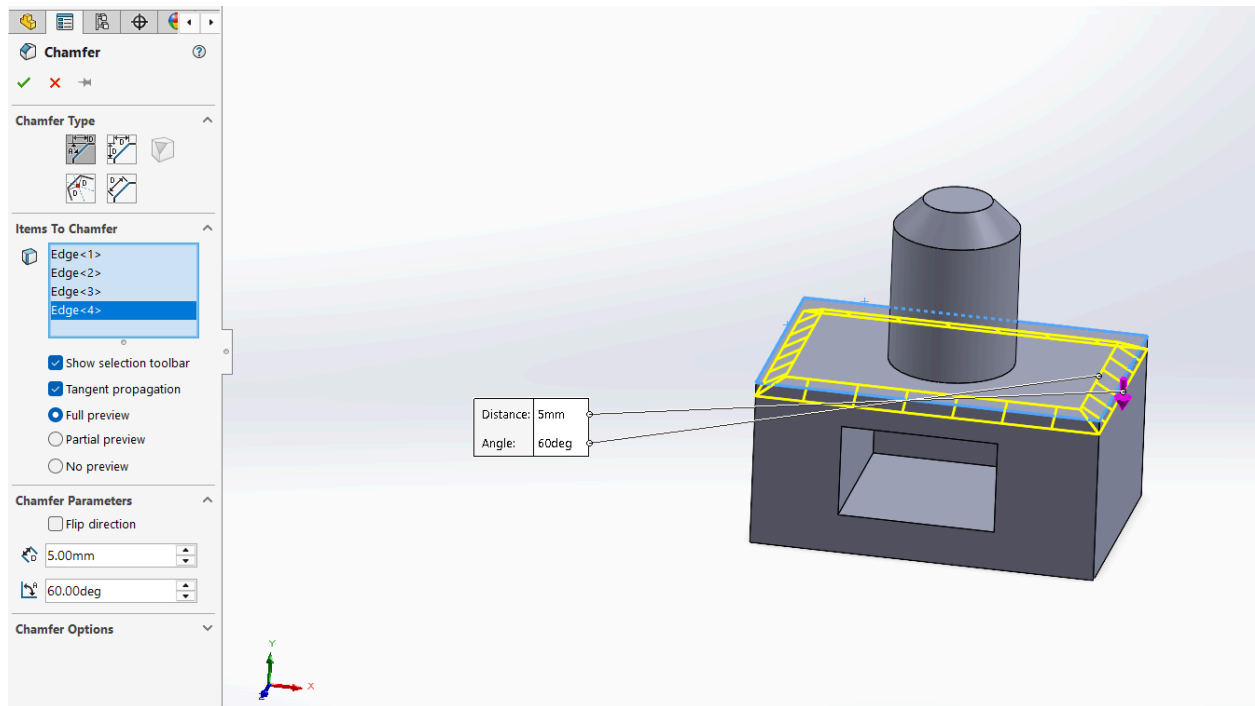
Select the edge shown below and check the chamfer preview.



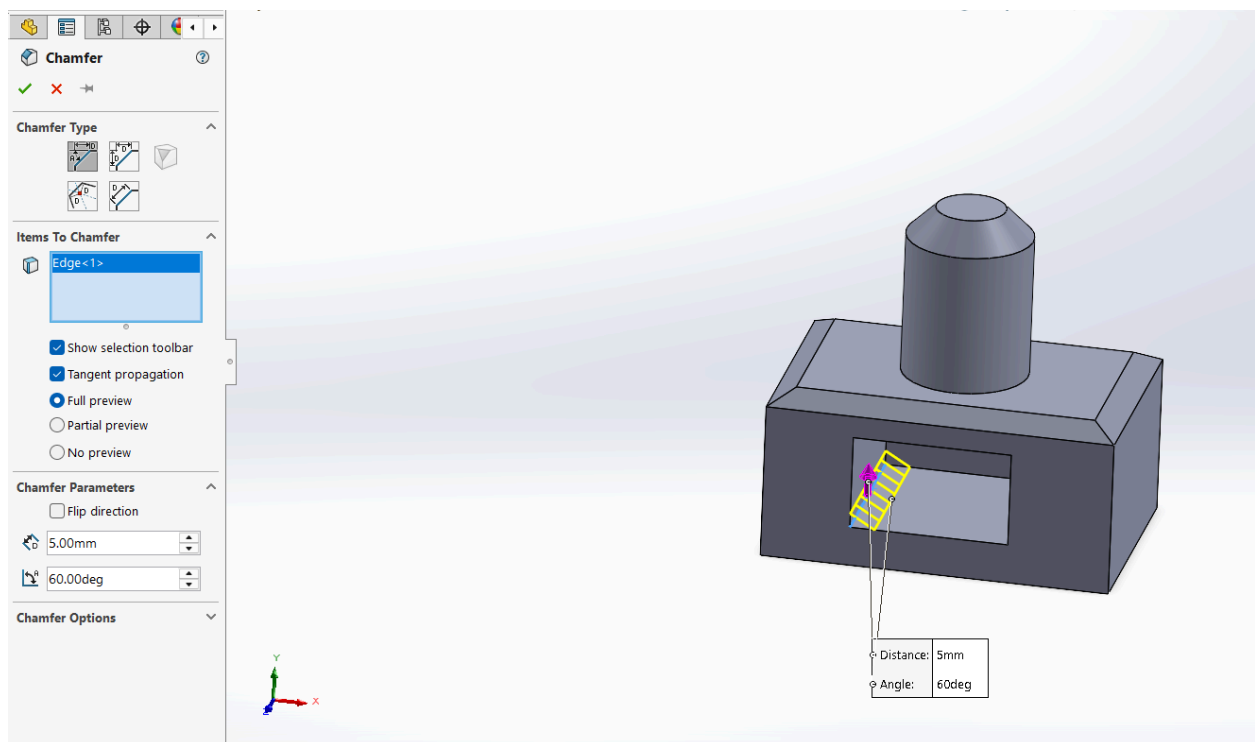
This is the final result.

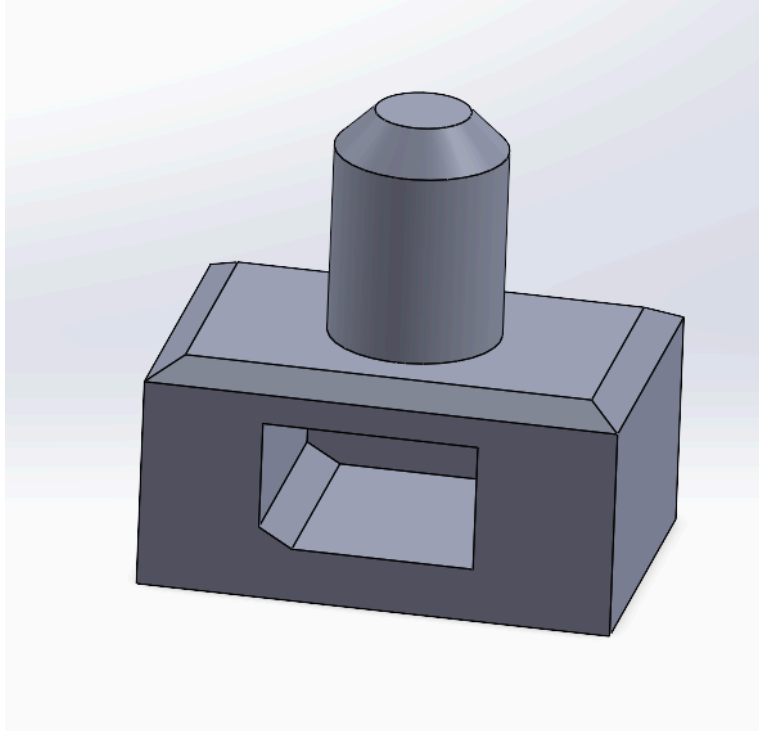


Select **Chamfer** again. Change the distance to **5 mm** and the angle to **60°**. Then select the edge shown below.



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





8. 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:

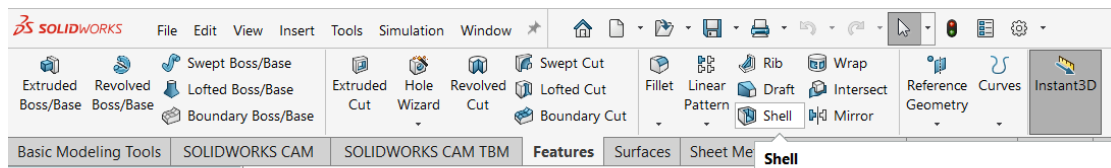
To create a hollow part, we use the **Shell** tool.

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.

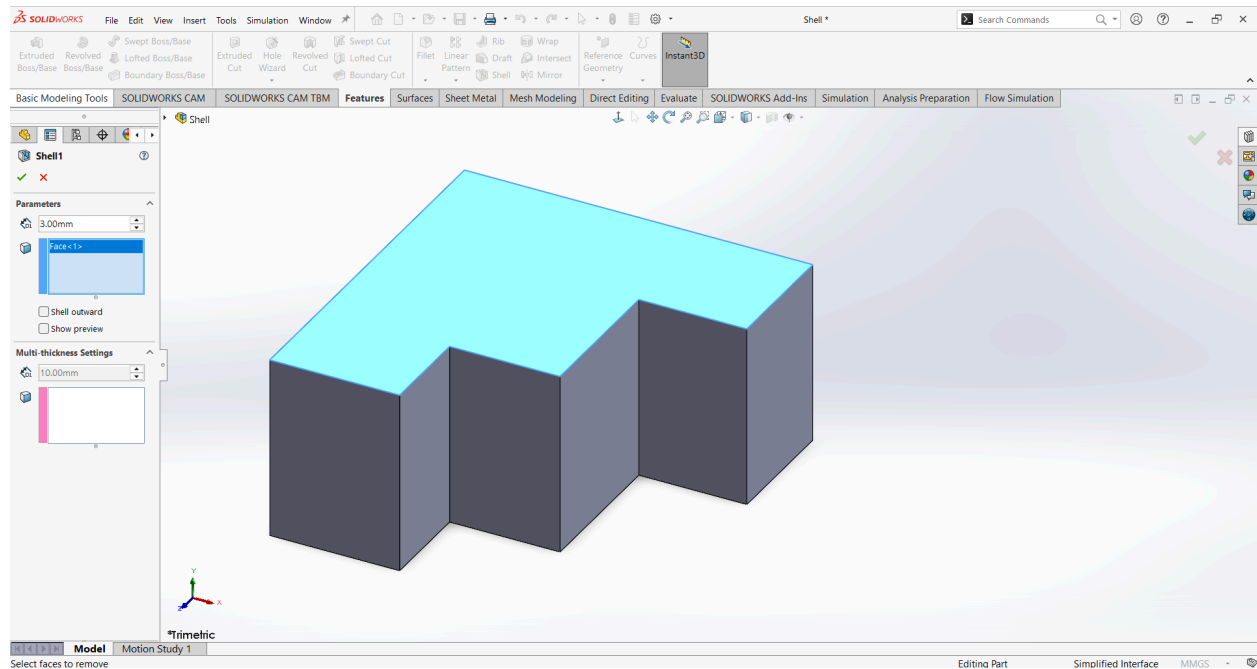


With **Shell**, you can select a face to remove, and SOLIDWORKS will hollow out the model from that opening. You can also define the wall thickness to control how thick the remaining material should be.

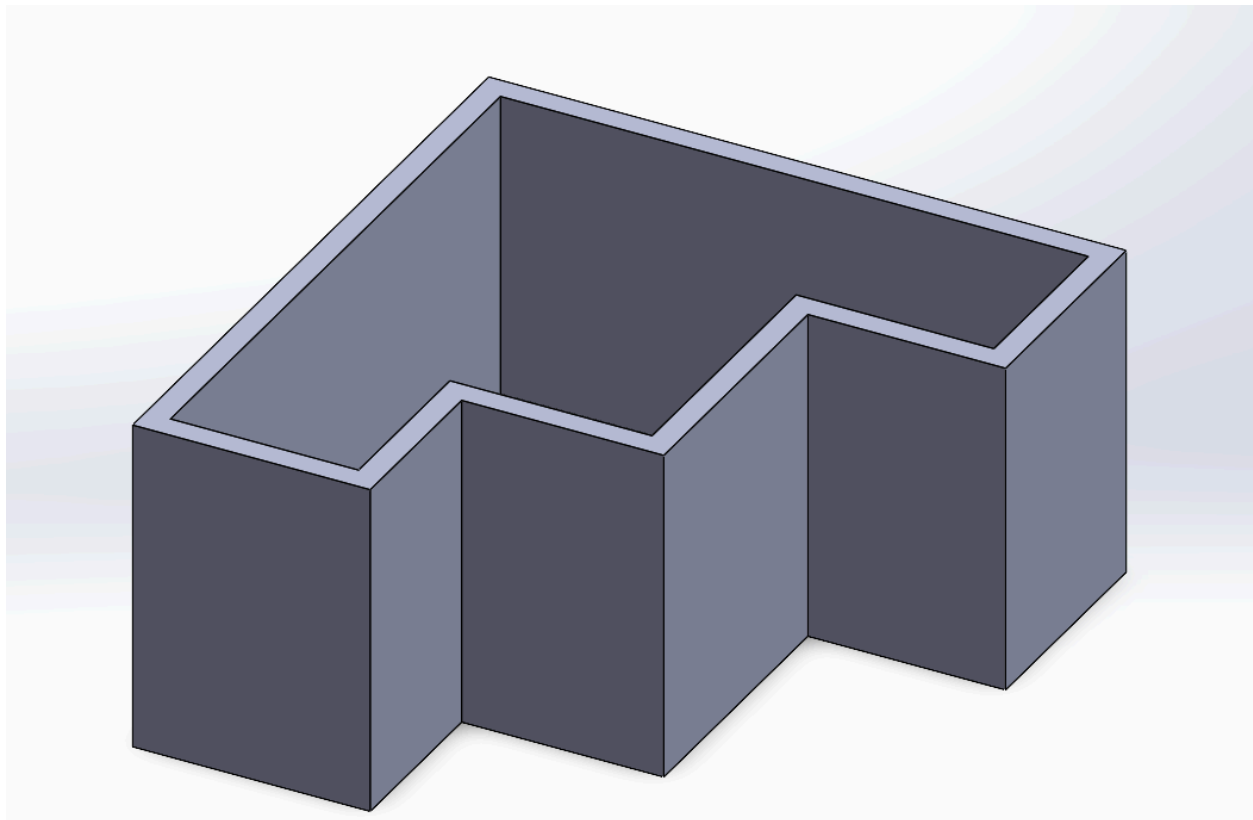
Go to the **Features** tab and select **Shell**.



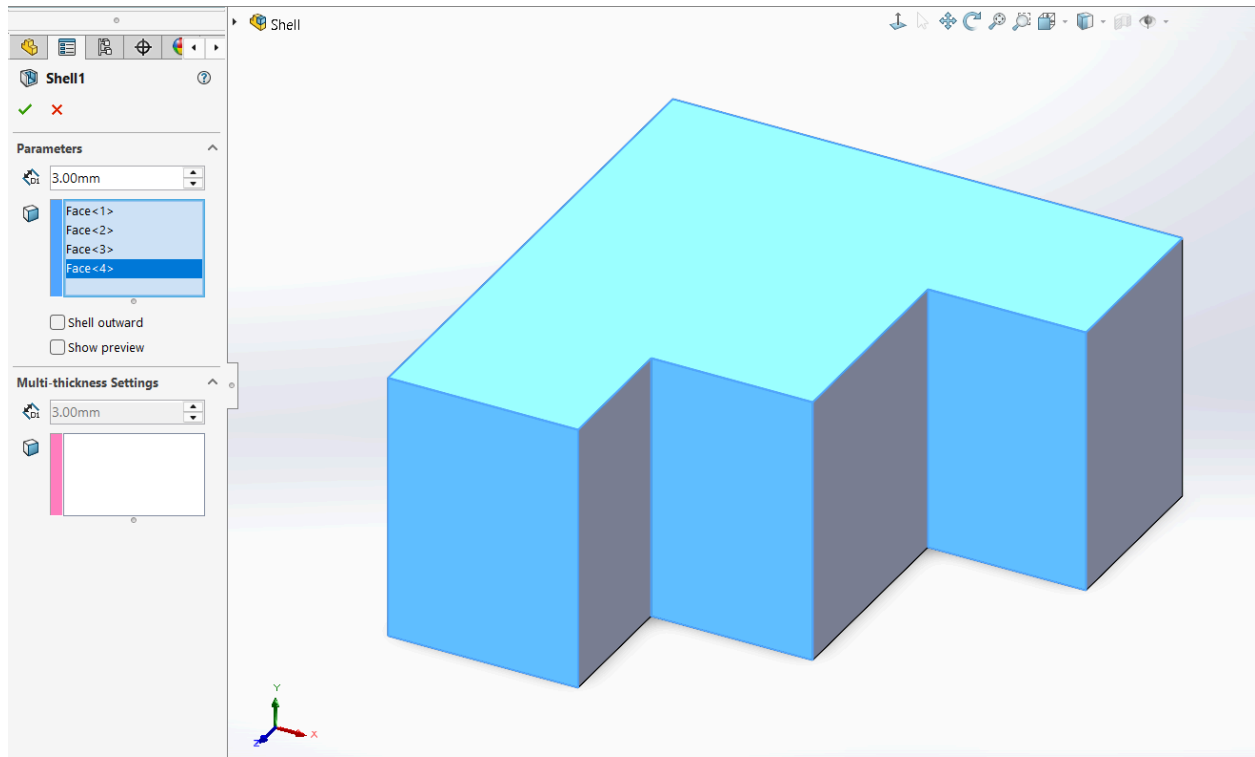
The shell properties appear on the left side in the **PropertyManager**. First, define the wall thickness. Then select the face you want to remove. Click the selection box in the **PropertyManager**, then select the upper face of the model.



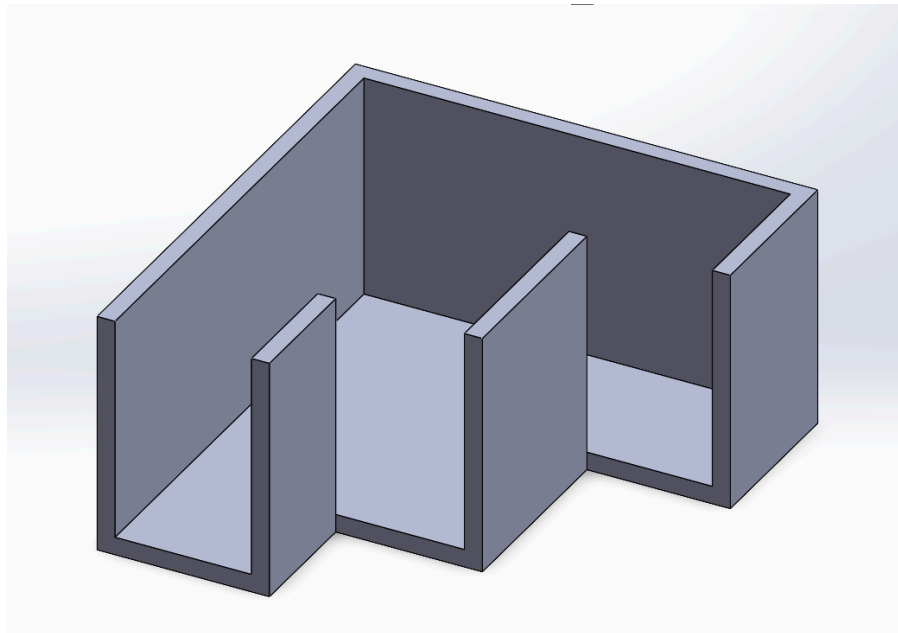
This is the final result.



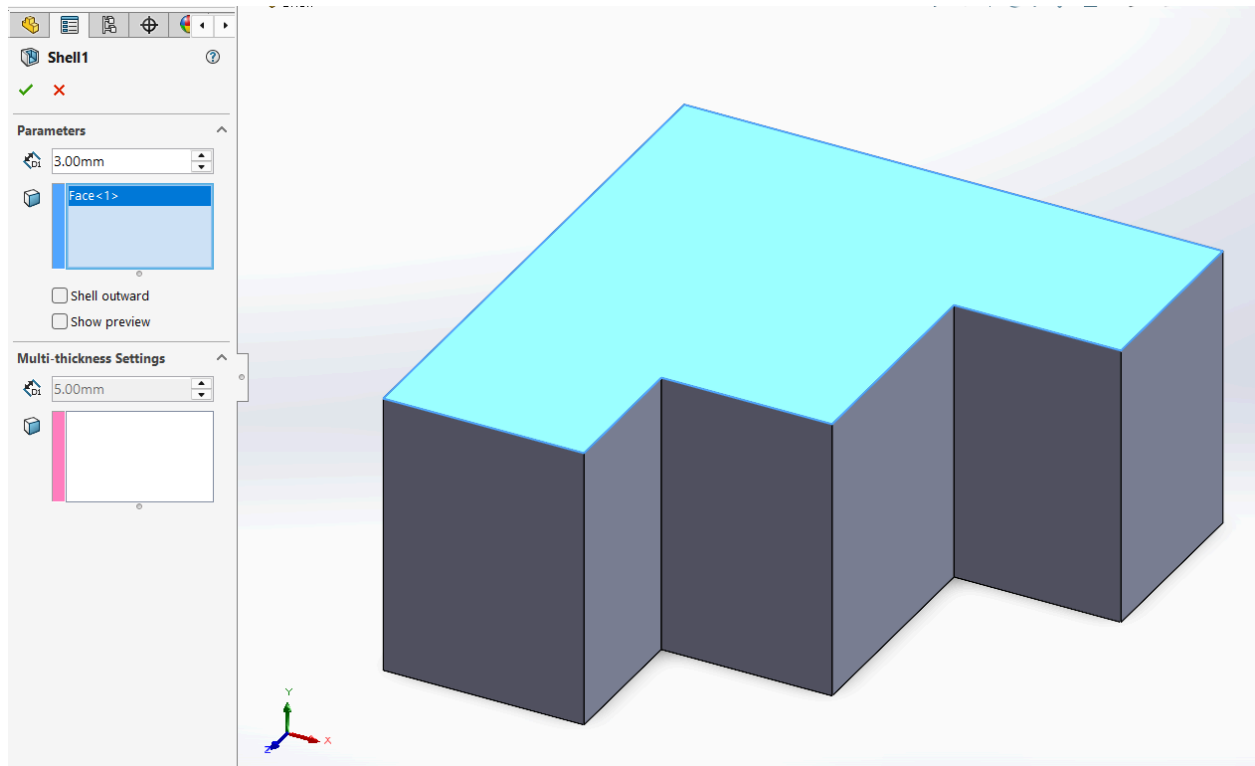
You can also remove multiple faces using the **Shell** tool. Click the selection box in the **PropertyManager**, then select multiple faces on the model.



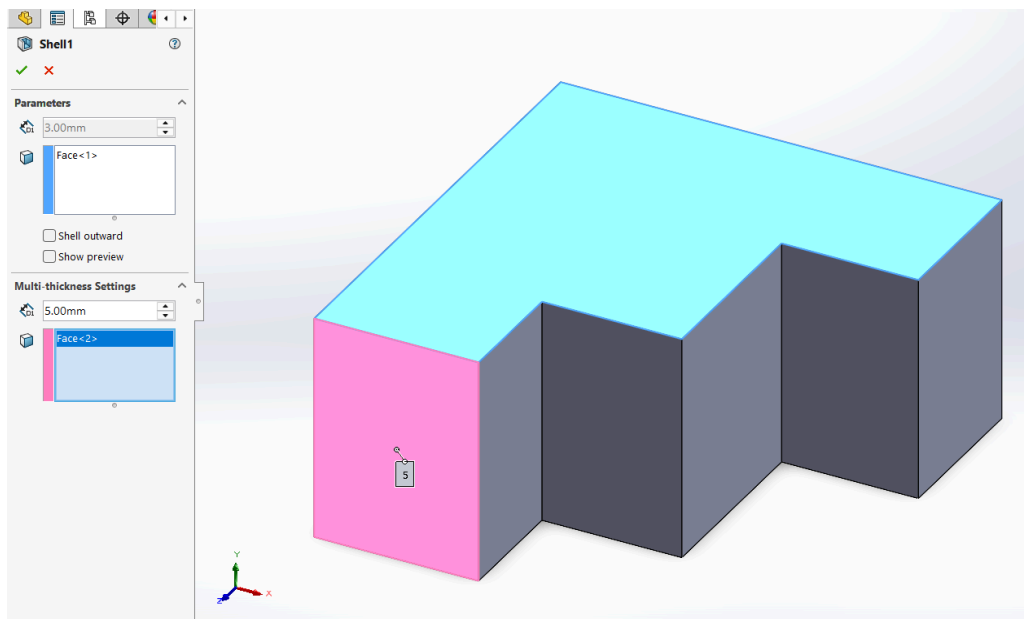
This is the result.



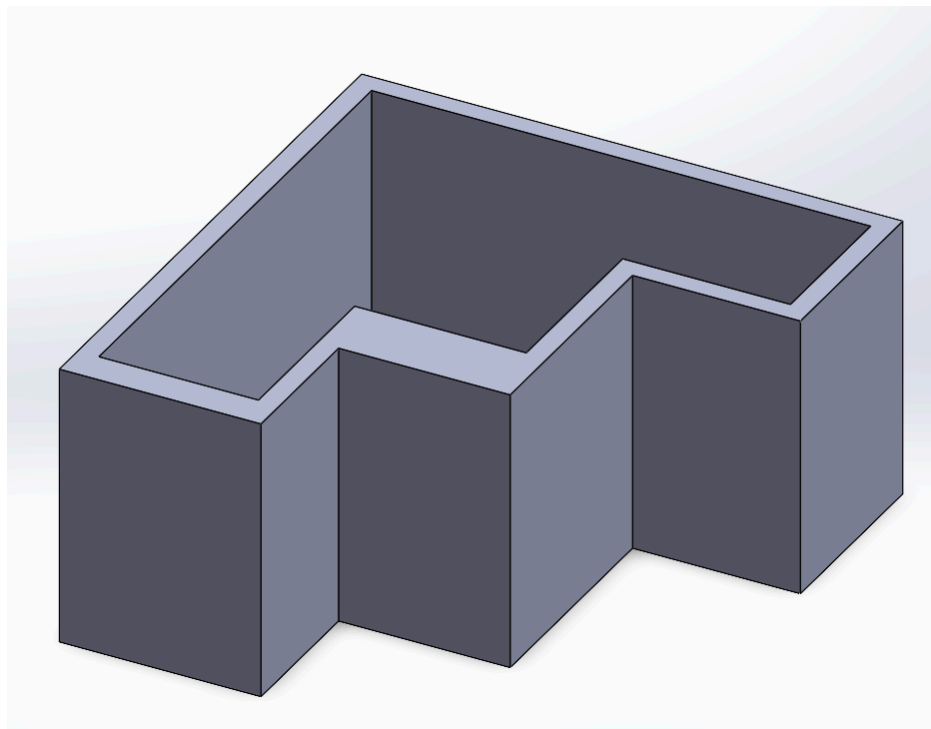
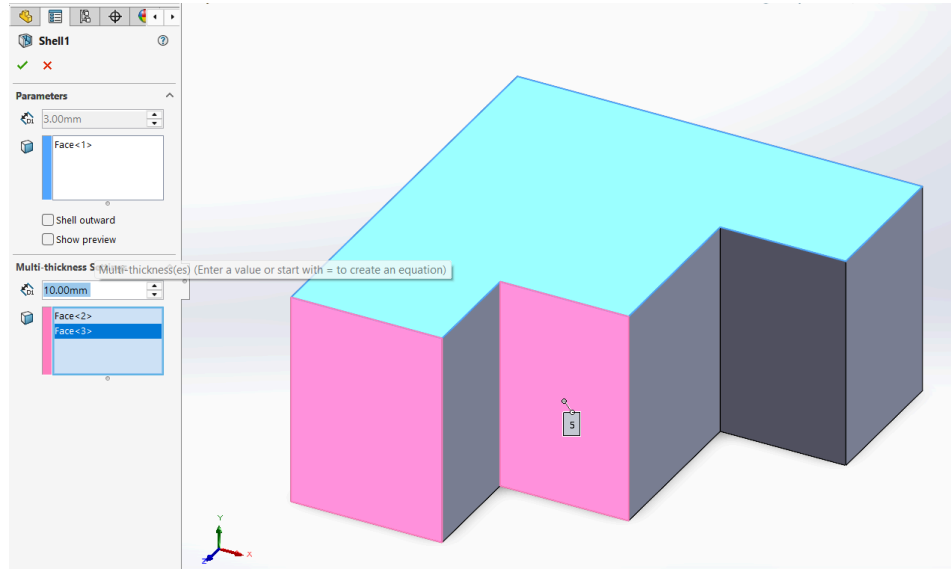
You can also apply different wall thicknesses to different faces. First, define the main shell thickness and select the face you want to remove.



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**.

9. 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.

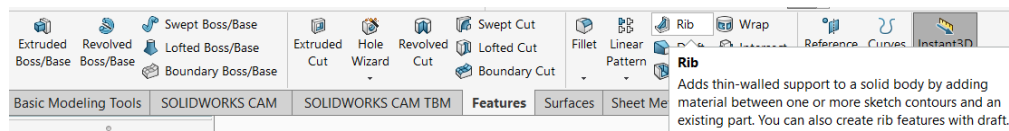


Ribs are commonly used to strengthen parts without adding too much material.

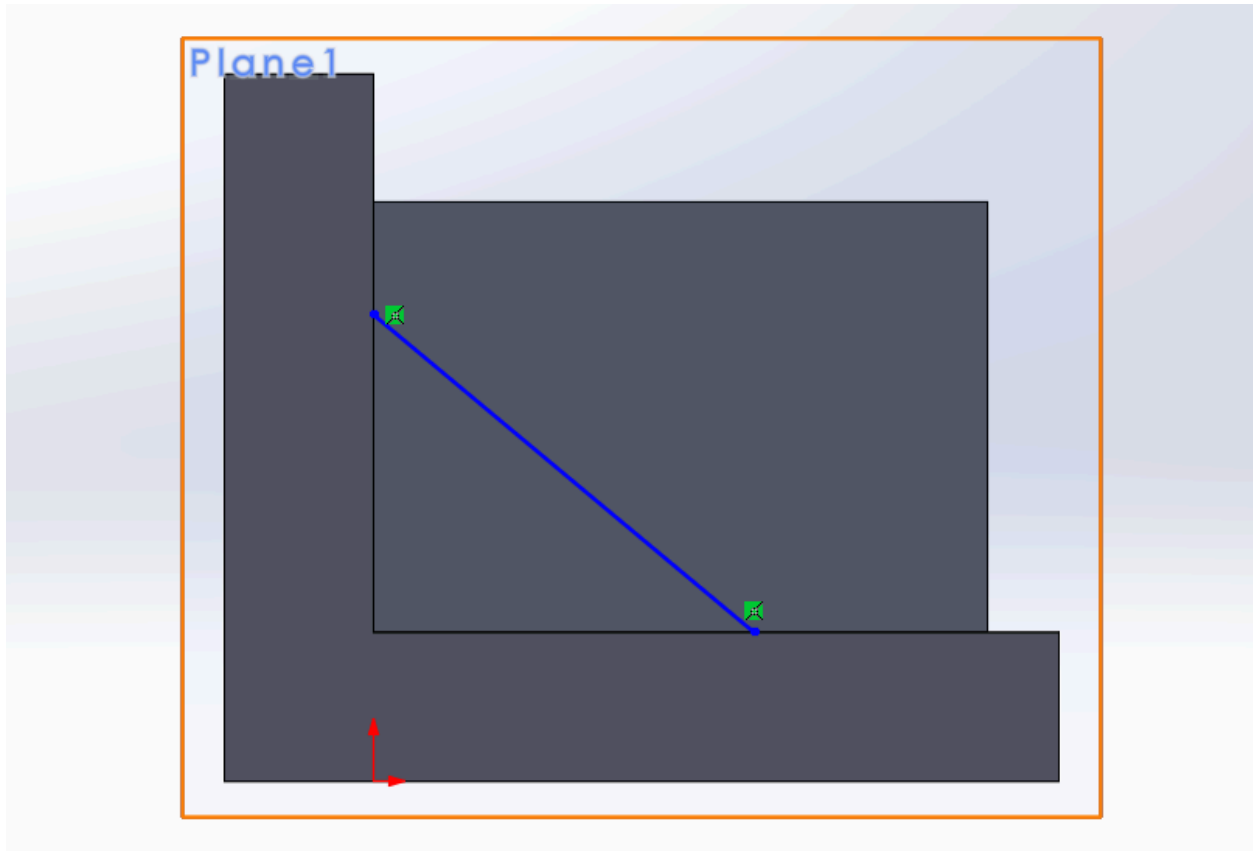
Instead of creating a full solid feature manually, you can create a simple sketch line, and SOLIDWORKS will turn it into a rib with a defined thickness and direction.

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

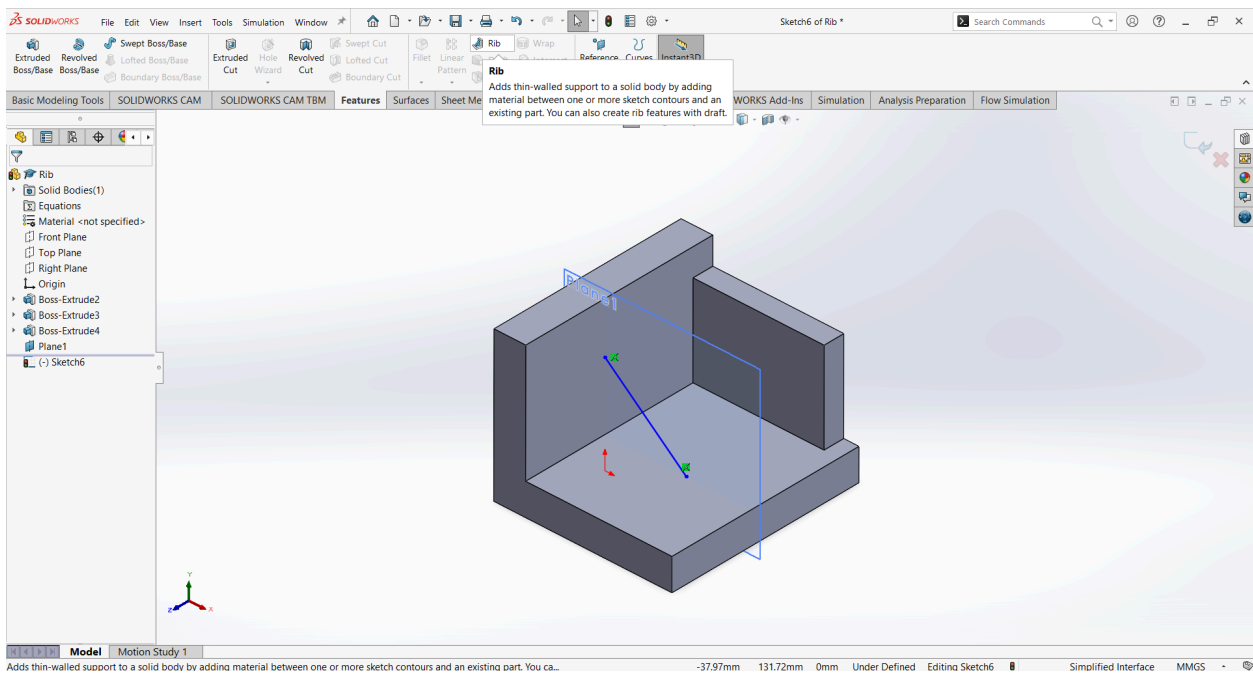
Go to the **Features** tab and select **Rib**.



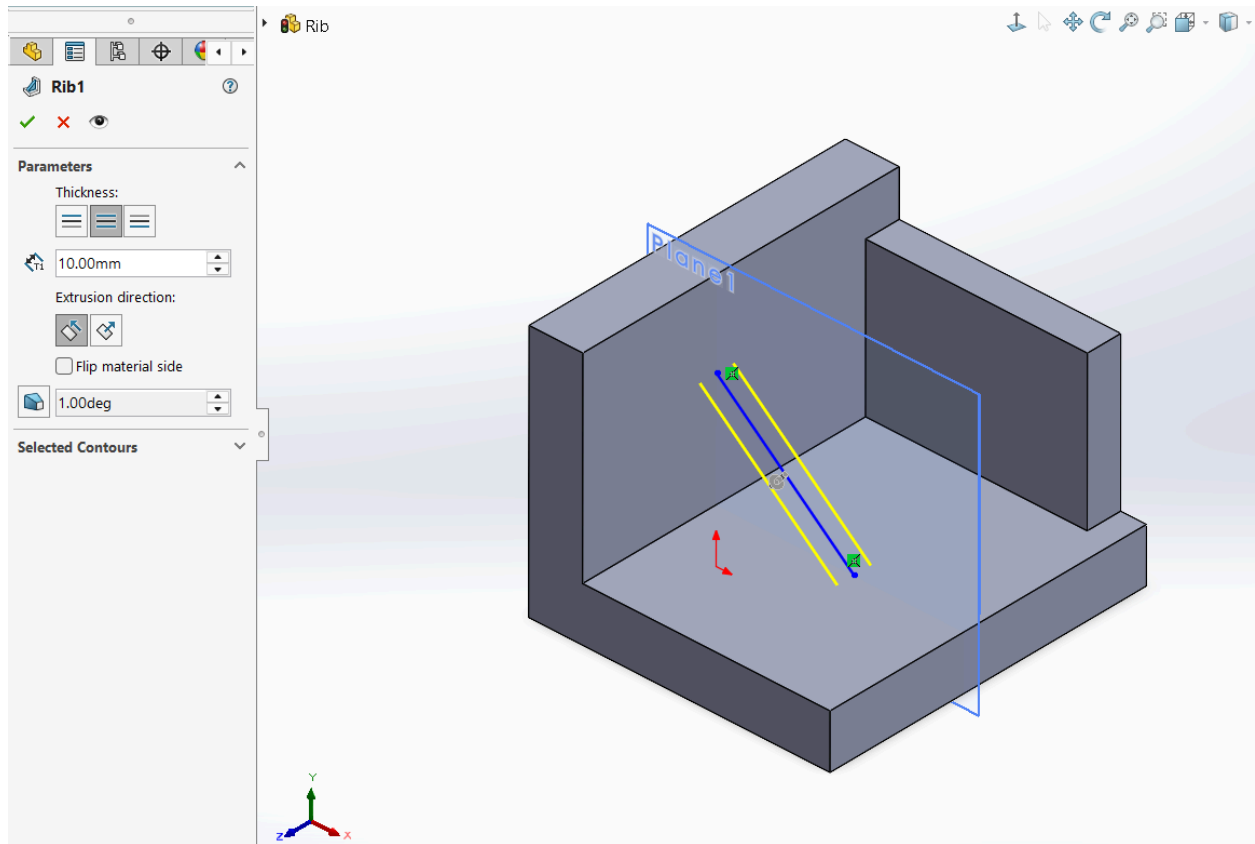
First, we need to create a line on the plane. This line will be used to create the rib. One line is enough. Create a line that connects the two perpendicular edges, as shown below.



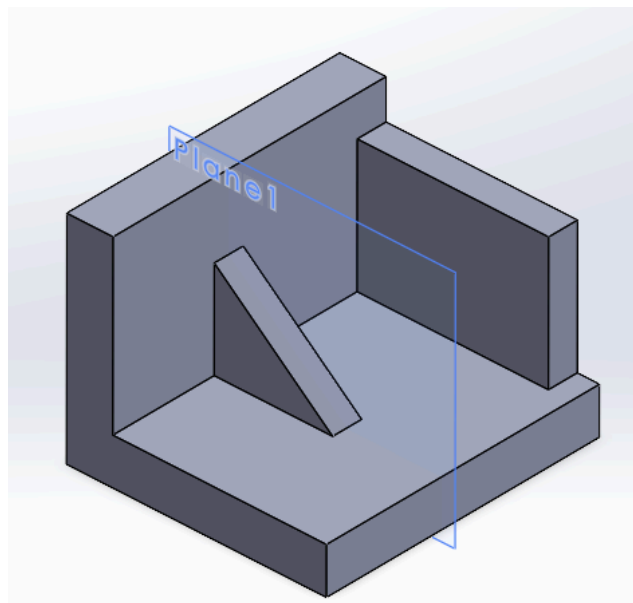
Now select the **Rib** tool.



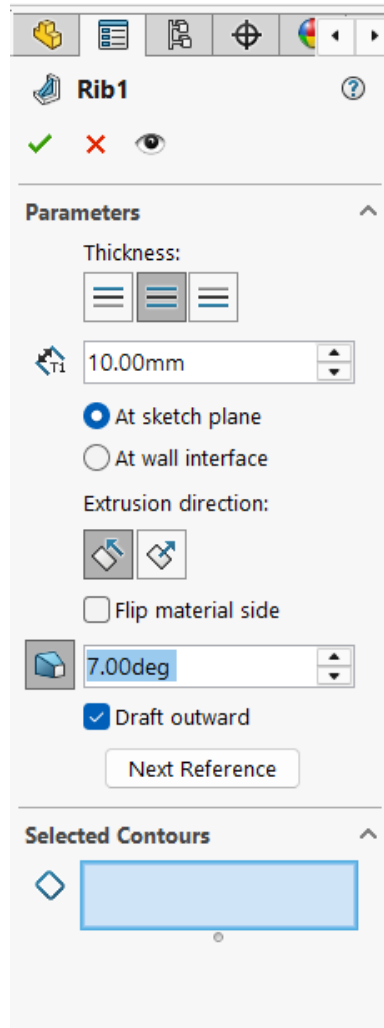
The rib properties appear on the left side in the **PropertyManager**. The thickness is set to **10 mm** by default. Click **OK**.



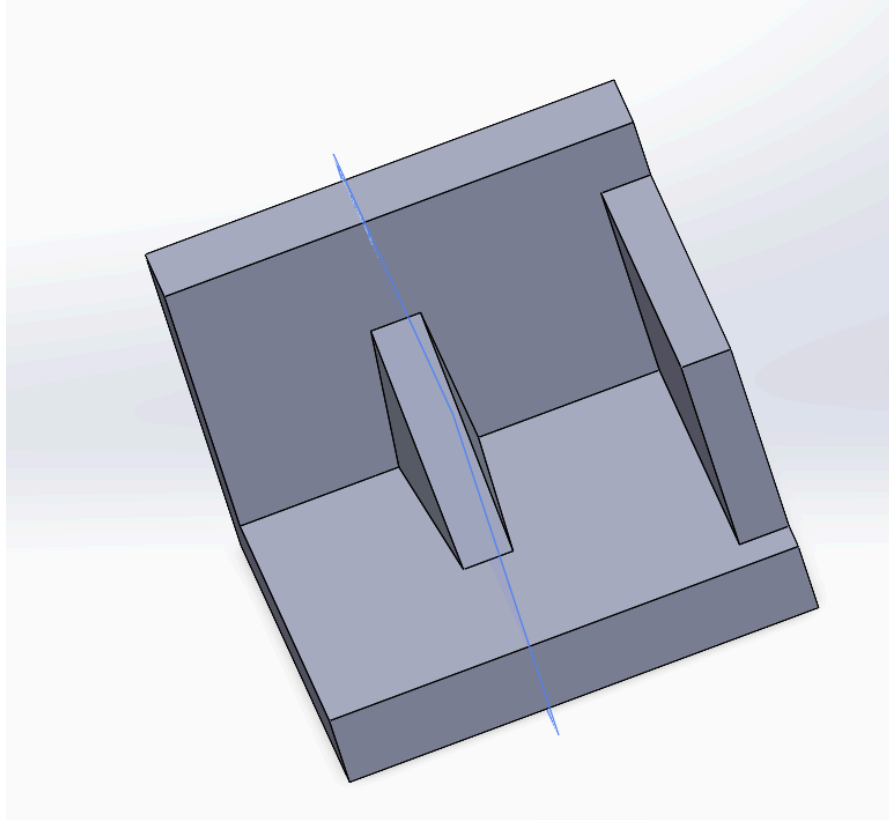
This is final result.



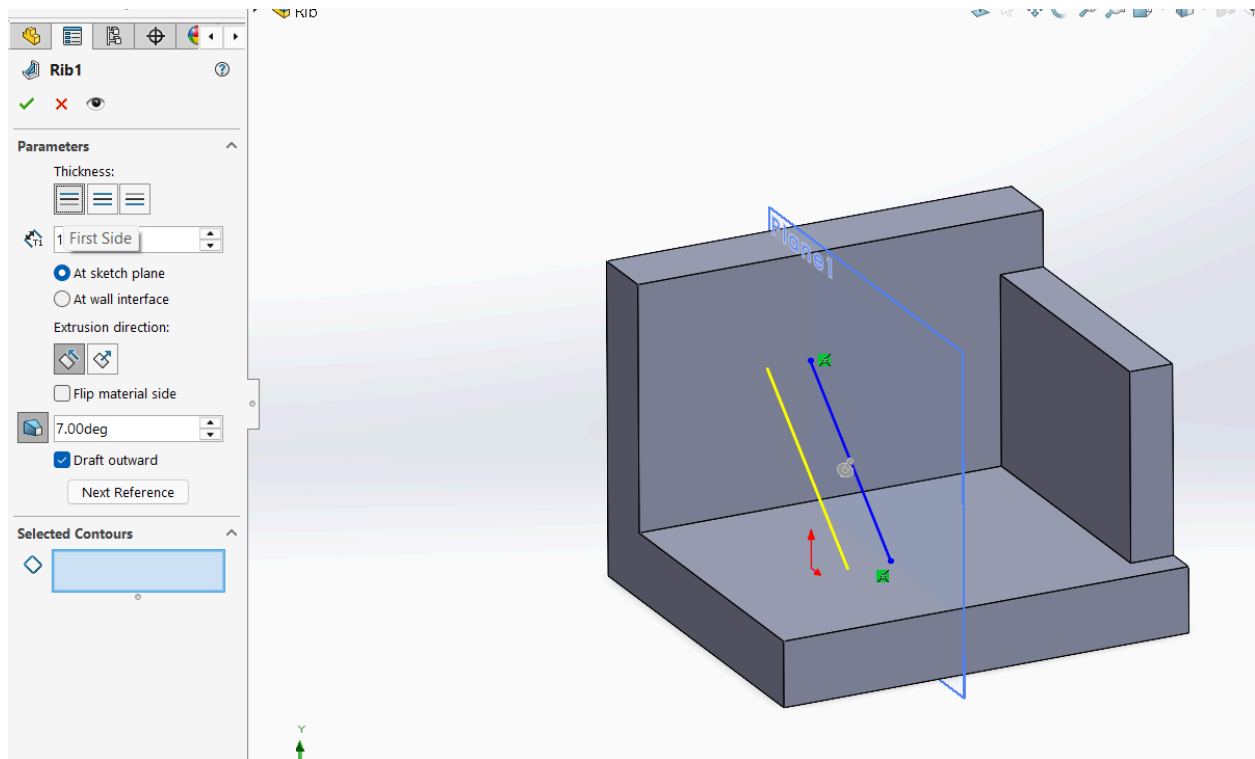
Now go back to the **PropertyManager**. Turn on **Draft** and set the draft angle to **7°**.



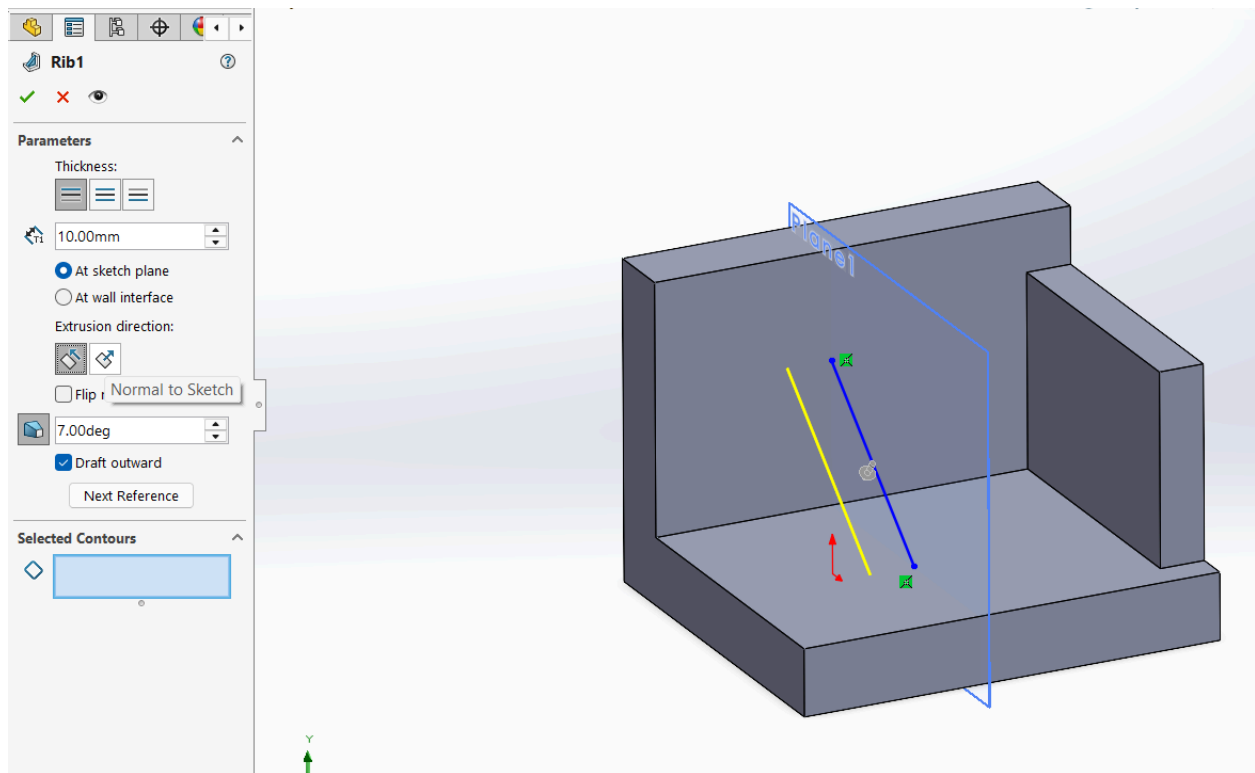
Now we have a rib with a draft angle of **7°**.



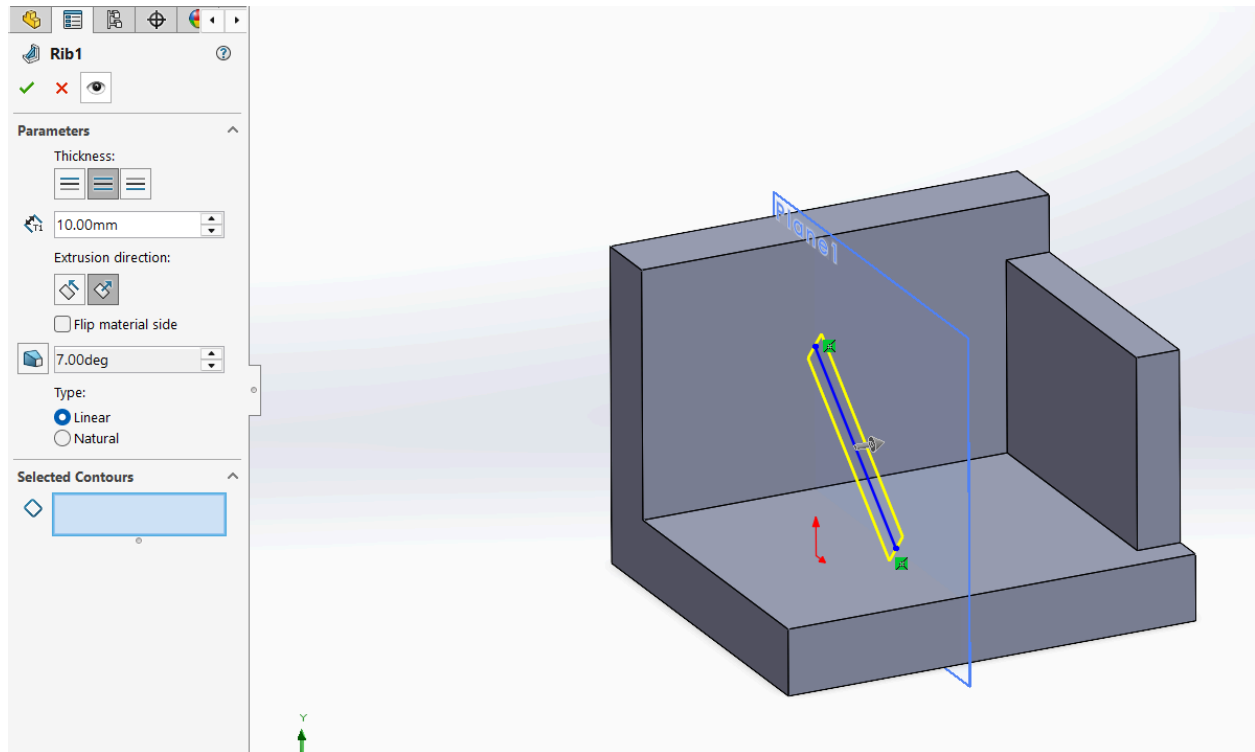
You can also choose the rib direction. The rib can be created on one side, on the opposite side, or from the middle of the sketch line.



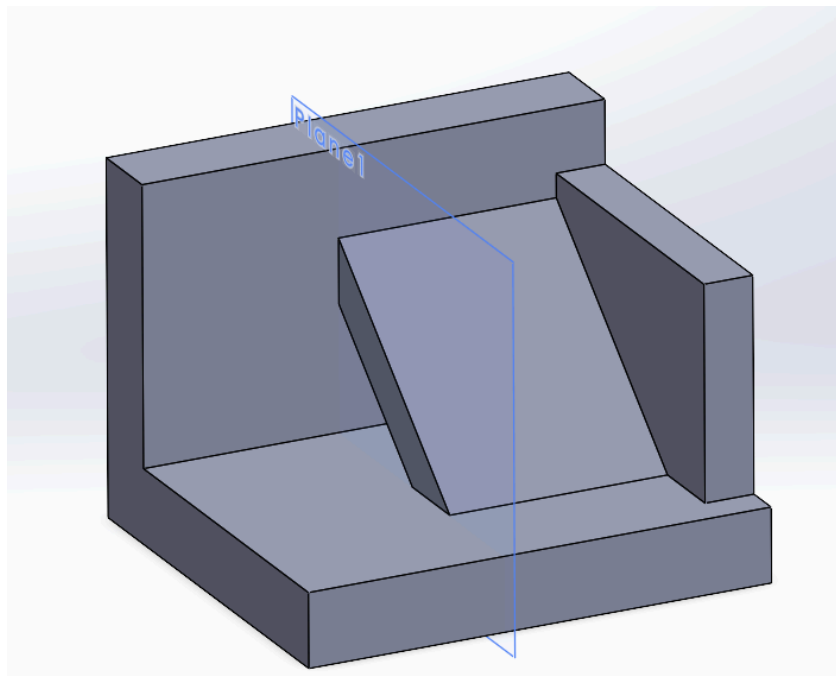
You can also use the **Normal to Sketch** option.



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



This is the final result.



10. 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:

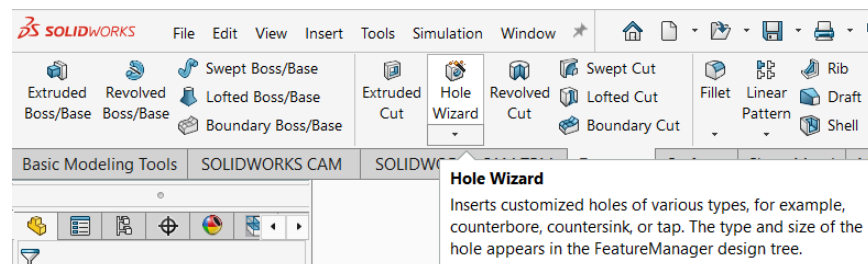
Hole Wizard is used to create standard holes quickly and accurately. Instead of sketching and cutting holes manually, you can choose from predefined hole types, sizes, standards, and end conditions.

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.

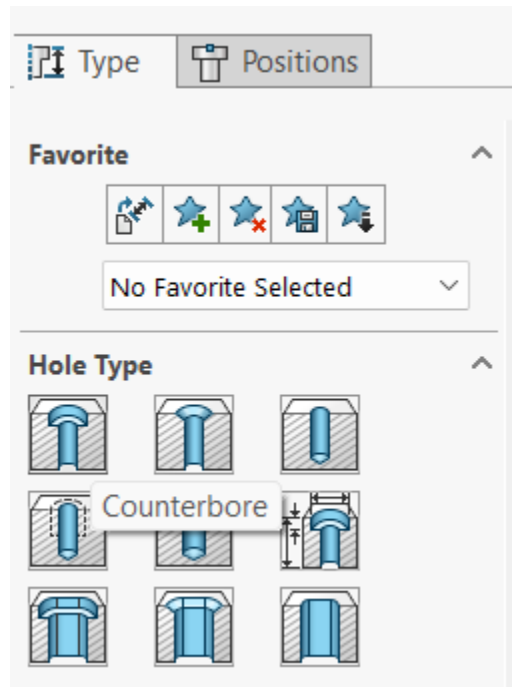
These settings help you create accurate and standardized holes directly inside SOLIDWORKS.

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

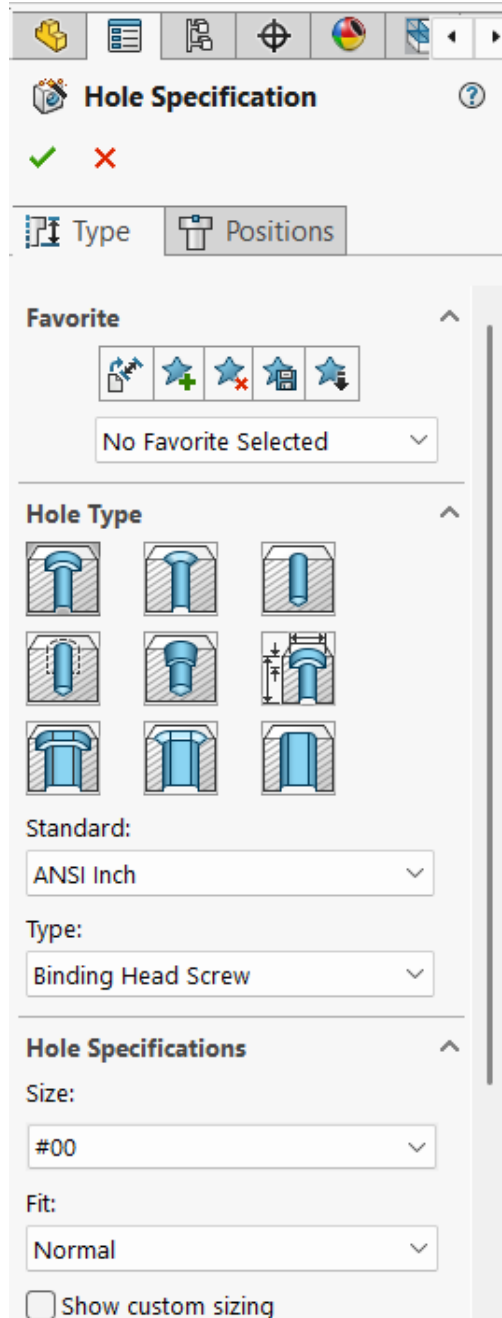
- Favorite:** Contains icons for favorite settings and a dropdown menu showing 'No Favorite Selected'.
- Hole Type:** Displays nine icons representing different hole types. The 'Counterbore' type is selected, which is characterized by a larger cylindrical recess at the top of the hole.
- Standard:** A dropdown menu set to 'ANSI Metric'.
- Type:** A dropdown menu set to 'Hex Bolt - ANSI B18.2.3.5M'.
- Hole Specifications:**
 - Size:** A dropdown menu set to 'M5'.
 - Fit:** A dropdown menu set to 'Normal'.
 - ☐ Show custom sizing
- End Condition:**
 - ☒ Blind
 - Depth:** A text box containing '7.00mm'.
 - Icons for 'Through All' and 'Through Selected' hole types.
- Options:**
 - ☐ Head clearance
 - ☐ Near side countersink
 - ☐ Under head countersink
- Tolerance/Precision:**
 - Callout value:** A dropdown menu set to 'Diameter (5.5)'.
 - Upper tolerance:** A dropdown menu set to 'None'.
 - Lower tolerance:** A dropdown menu set to '.12 (Document)'.

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

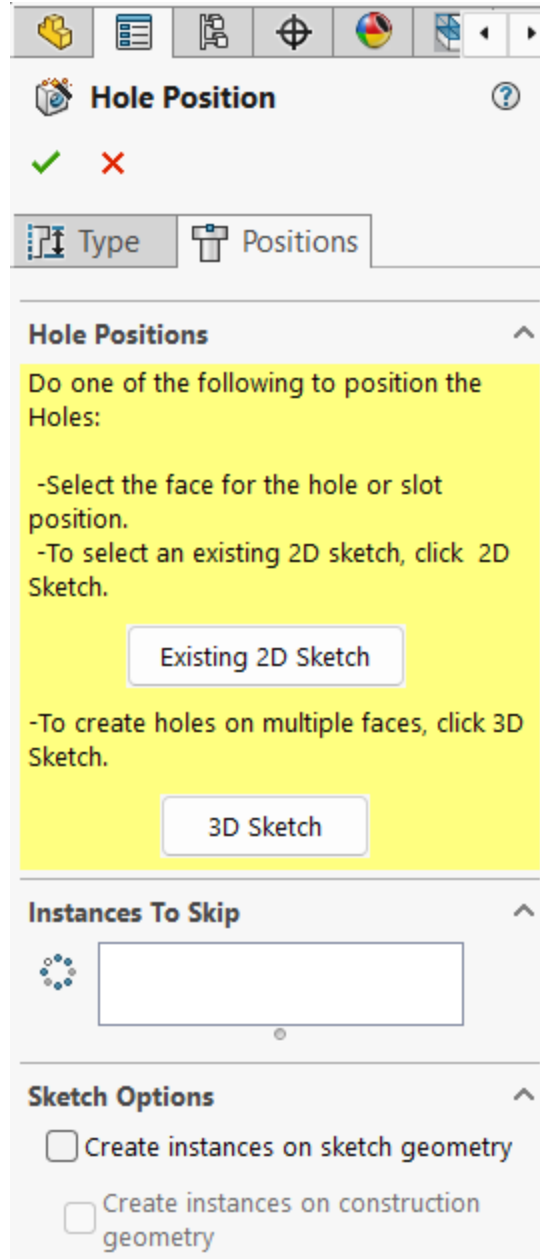
Counterbore, the available settings in the **PropertyManager** will update based on this hole type.



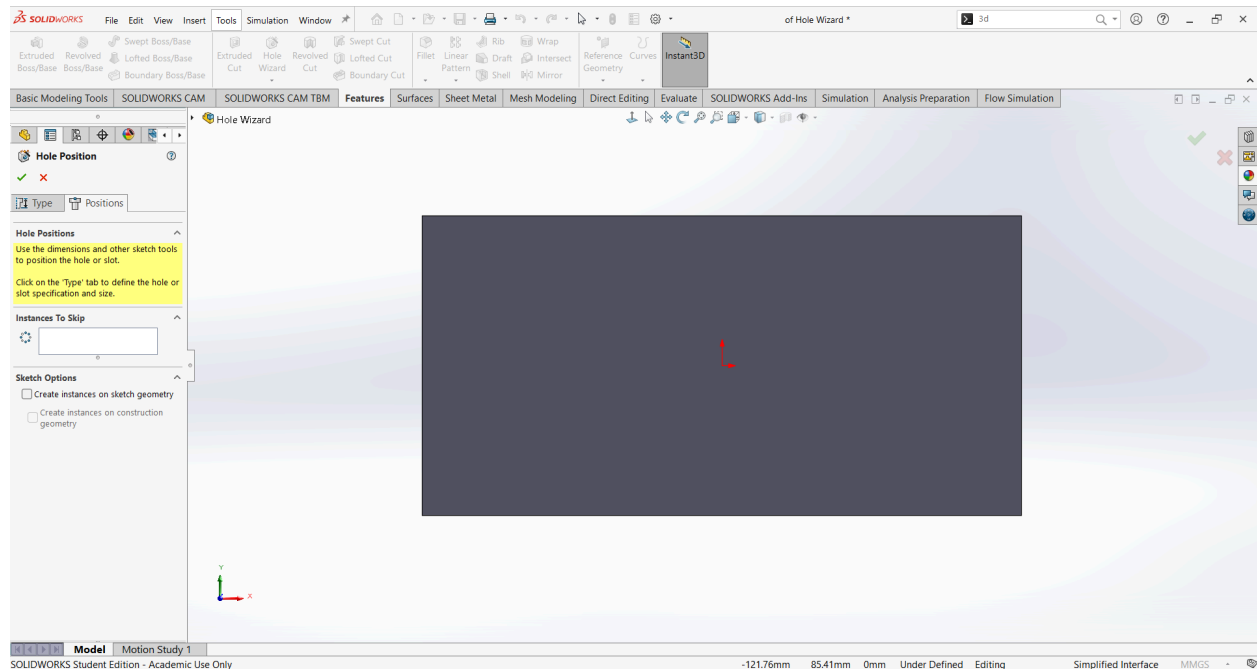
Set the **Standard** to **ANSI Inch**. Then set the **End Condition** to **Through All**. The **ANSI Inch** standard defines the available counterbore hole sizes based on inch units. The **Through All** end condition means the hole will cut completely through the part. Make sure both settings are selected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



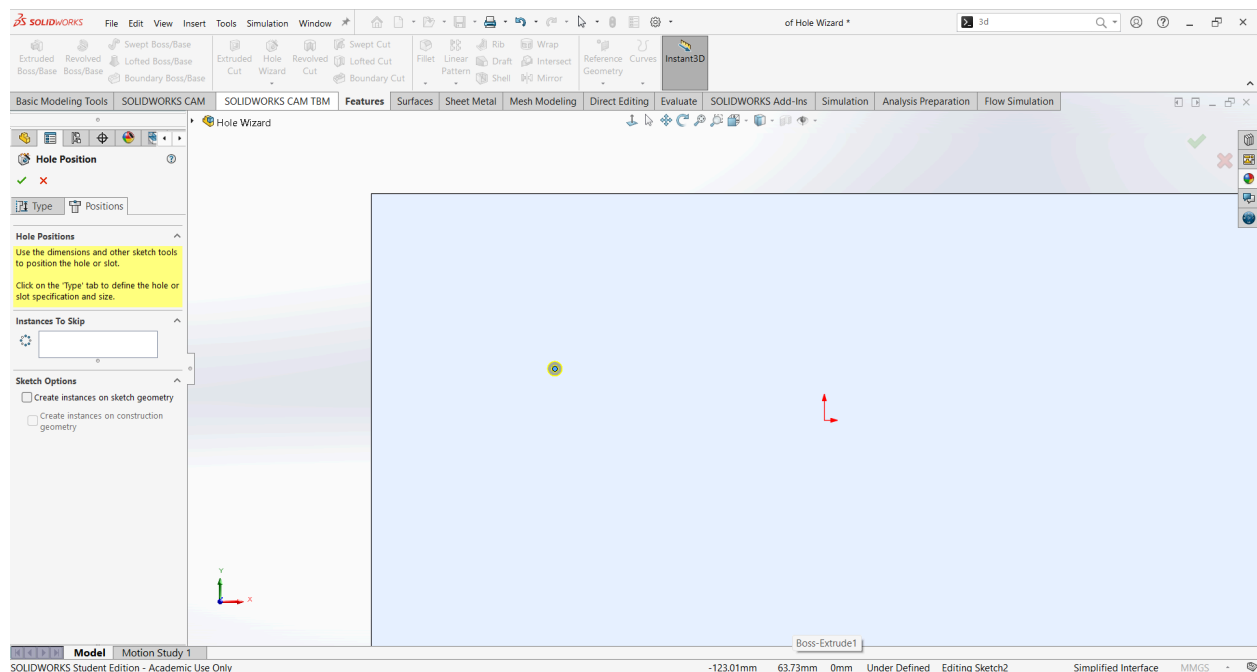
Now we need to define the hole position. In the **Hole Wizard PropertyManager**, select the second tab called **Positions**. This tab allows you to place the hole on the model. After selecting **Positions**, choose the face where you want to place the hole. Then click on the face to place a point. This 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.

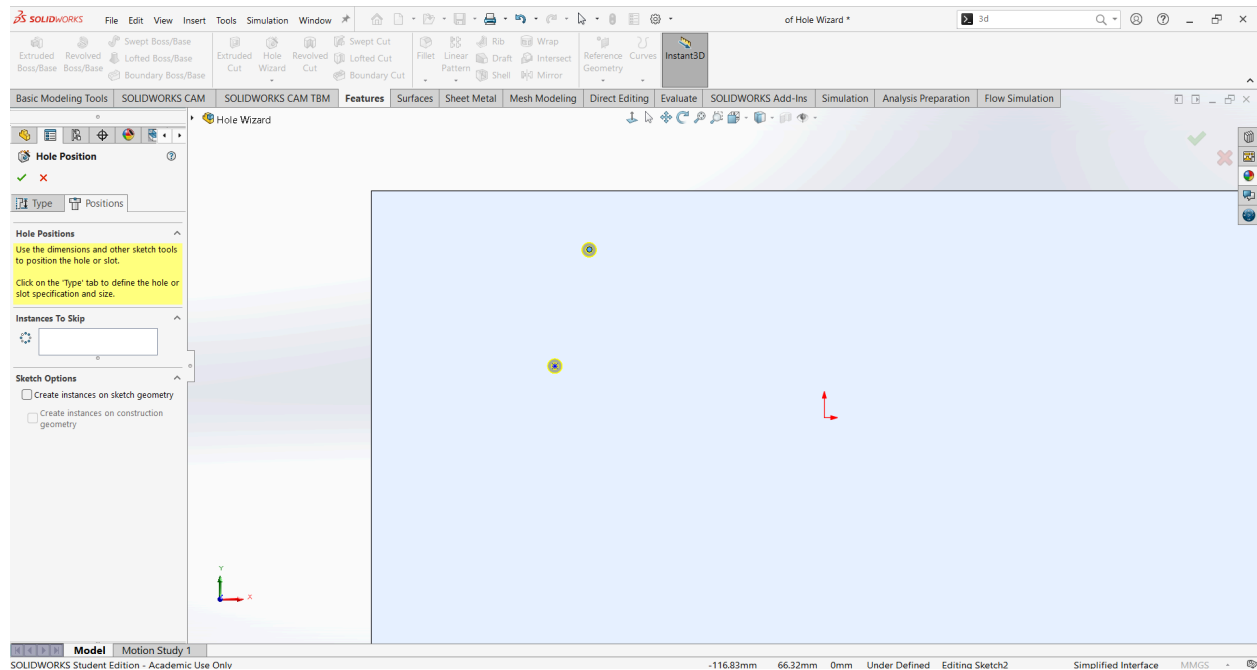


Now place the hole point on the selected face. Move your mouse over the front face. Then click once on the face to place the point. This point represents the center location of the hole. At this stage, the point is placed freely on the face. Next, you can use **Smart Dimension** or sketch relations to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

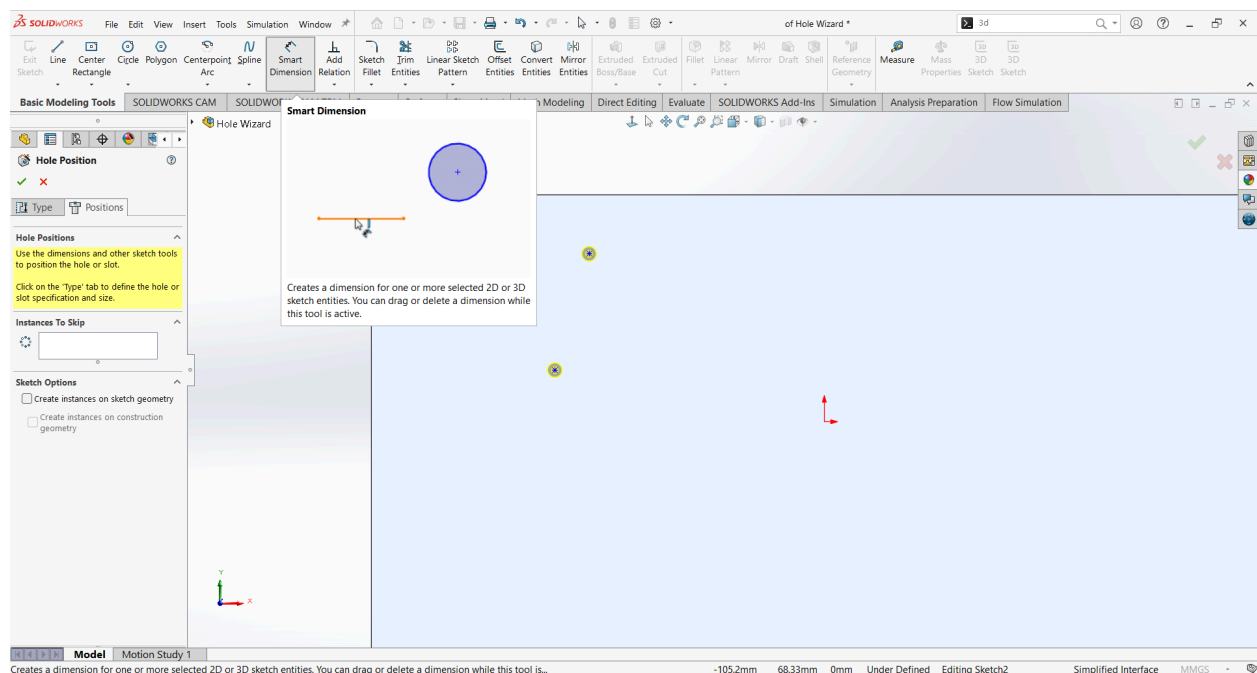


At this stage, you can add multiple hole points on the selected face. Each point will create a separate hole using the same **Hole Wizard** settings. For this example, place a second point on

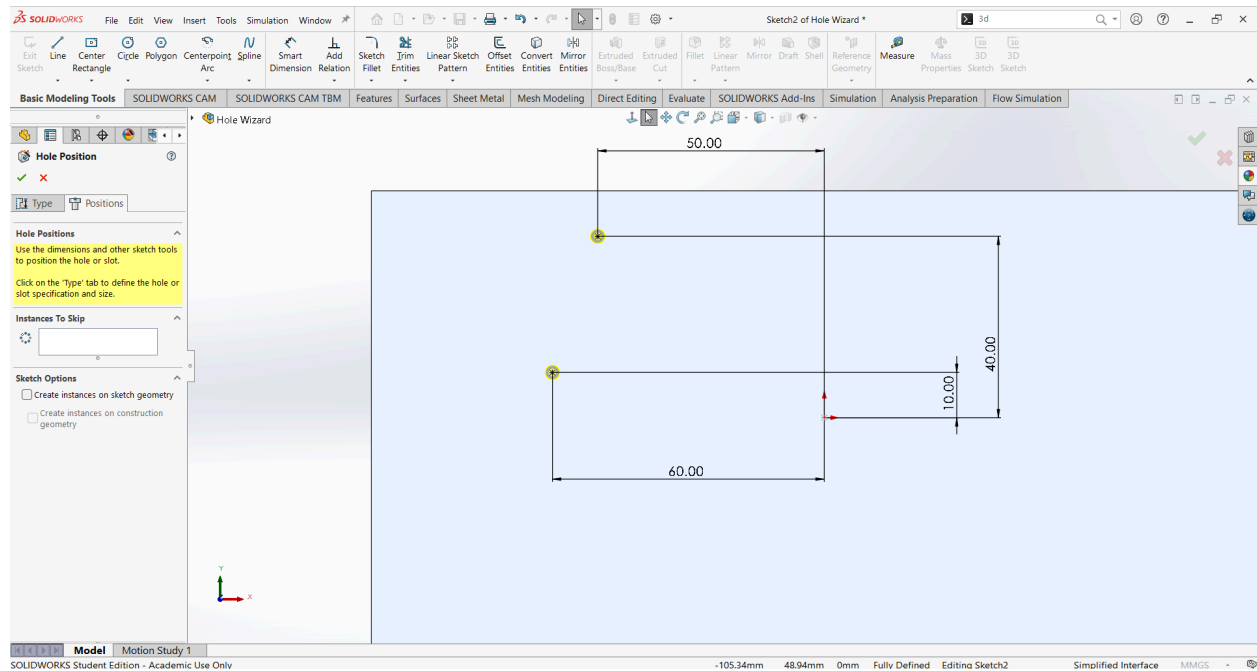
the face. Click once more on the face to create the second hole point. Now you should have two hole points on the front face. Next, we will define their positions using dimensions or relations.



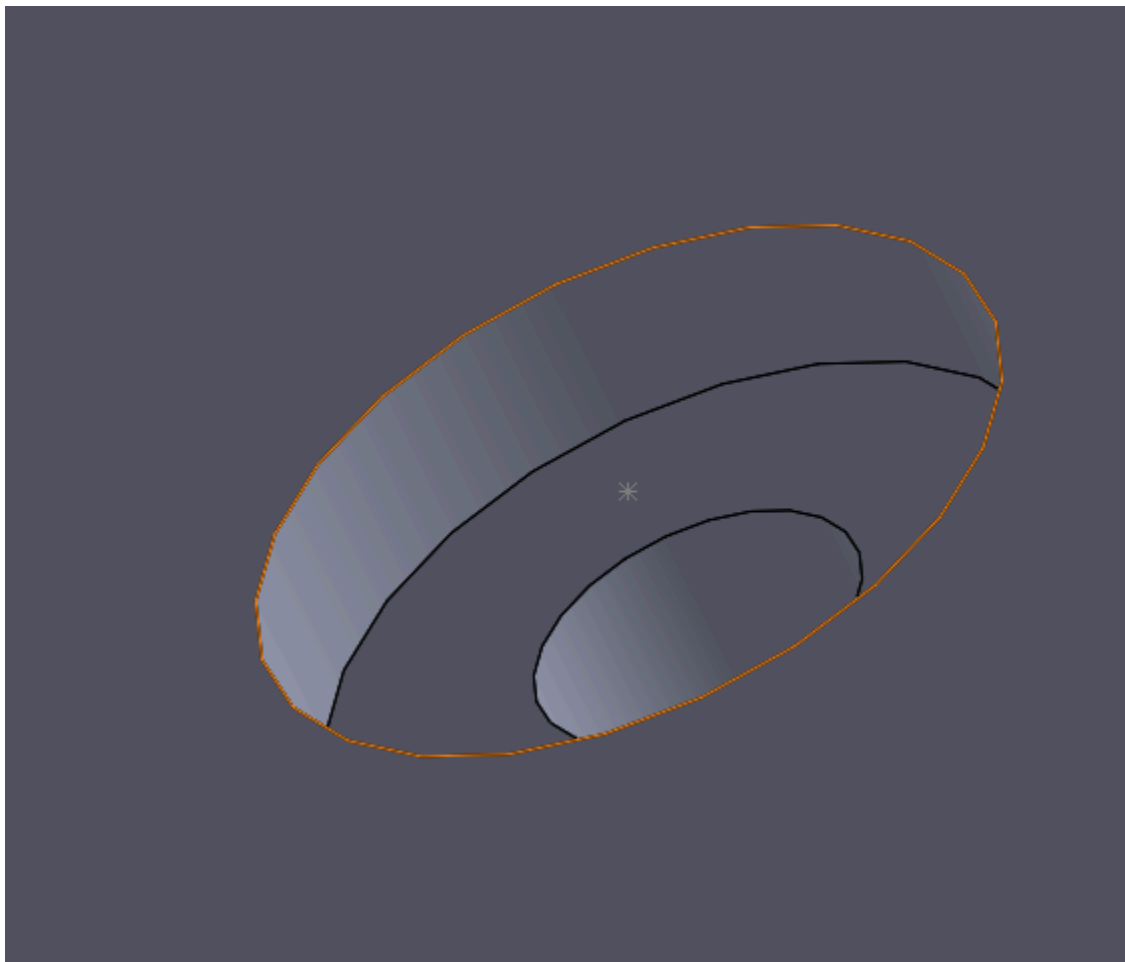
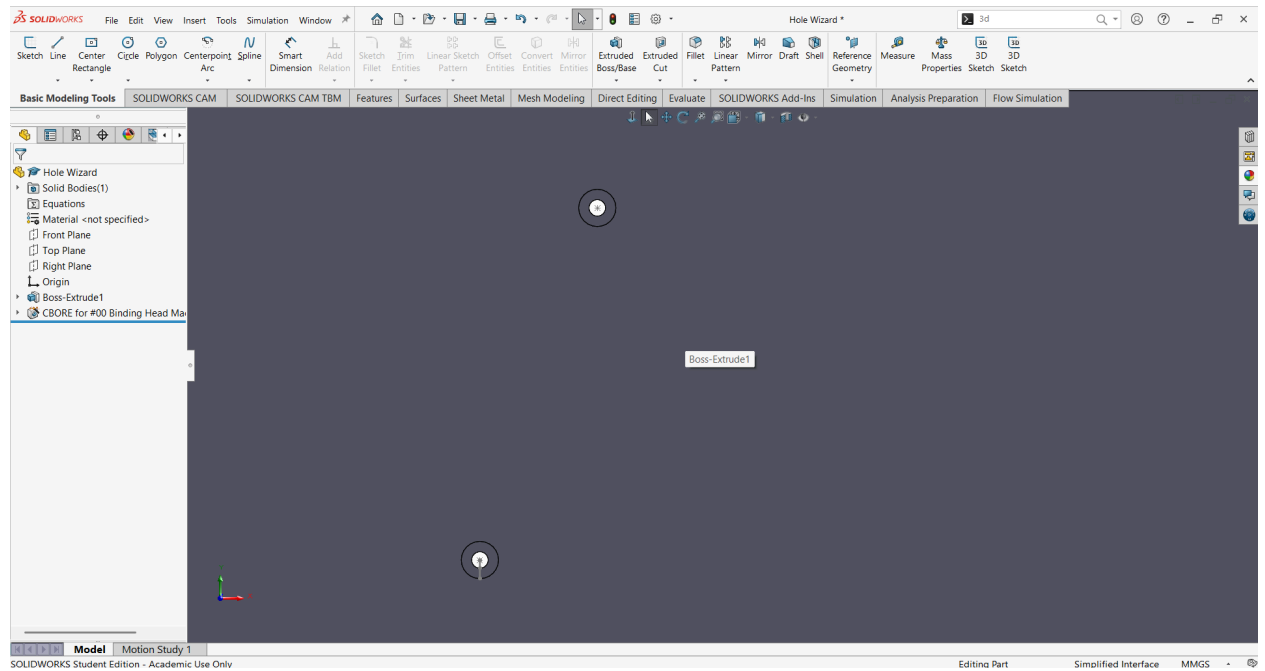
Before clicking **OK**, we can define the exact position of the hole points using **Smart Dimension**. Go to the **Basic Modeling Tools** tab. Select **Smart Dimension**. Now select the hole point and the reference edge or reference line you want to dimension from. Then enter the required dimension value. 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.



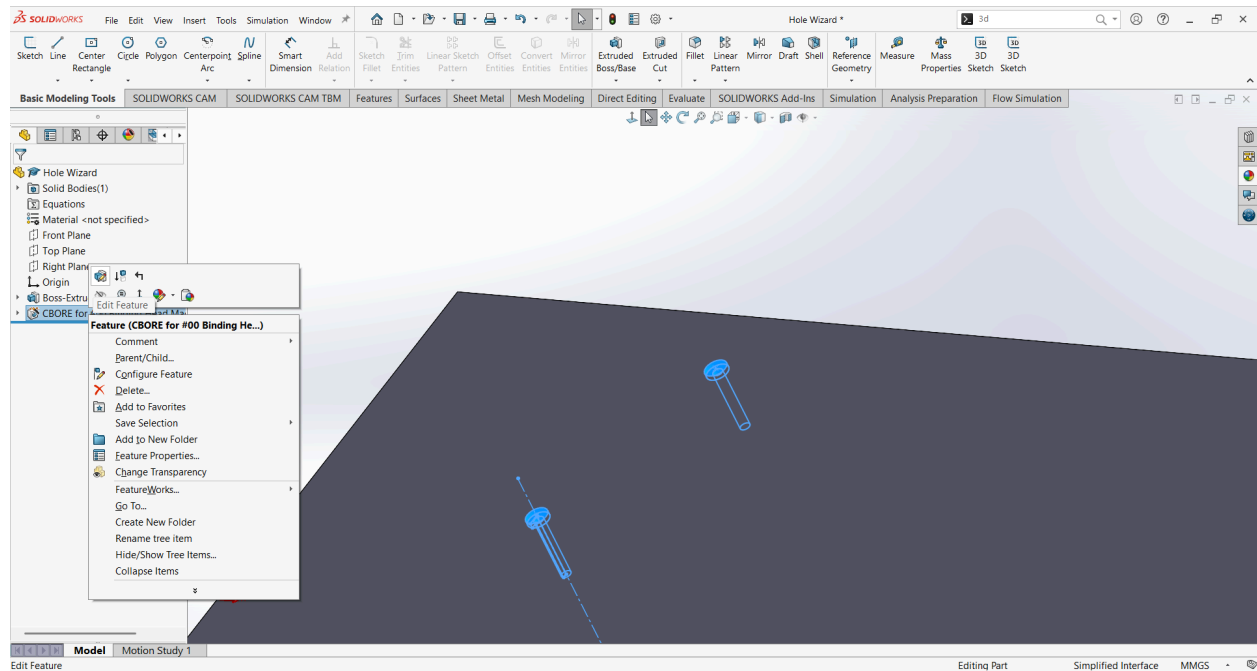
Apply the dimensions shown. Use **Smart Dimension** to define the exact position of each hole point. Select the hole point and the reference edge, then enter the required dimension value. Repeat the process for the remaining dimensions, as shown. Make sure both hole points are fully defined before continuing.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



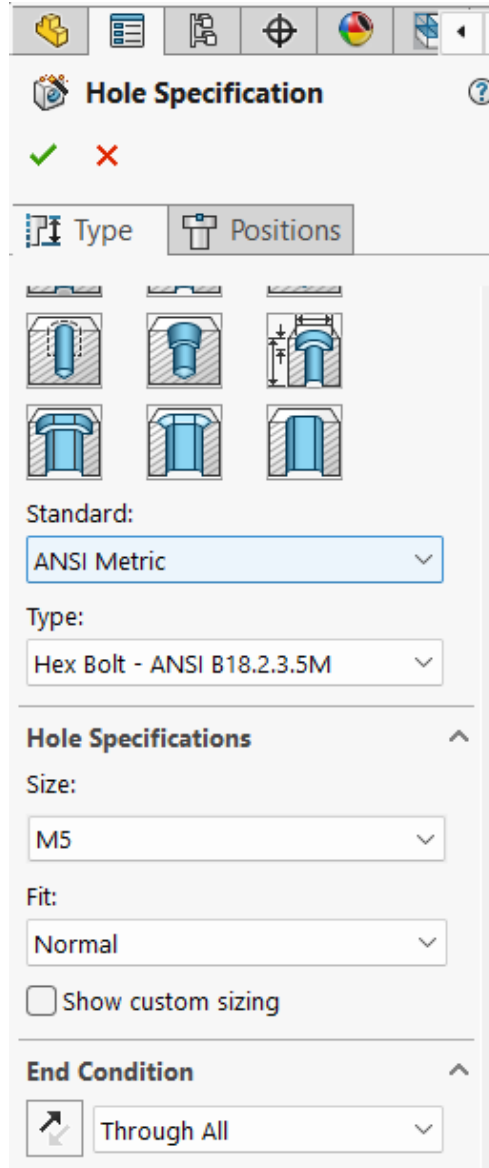
Click **OK** by selecting the green check mark. The **Hole Wizard** feature is now created. You can see the holes on the selected face of the model. Because we placed two hole points, SOLIDWORKS created two counterbore holes using the same settings. You can also find the new **Hole Wizard** feature in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



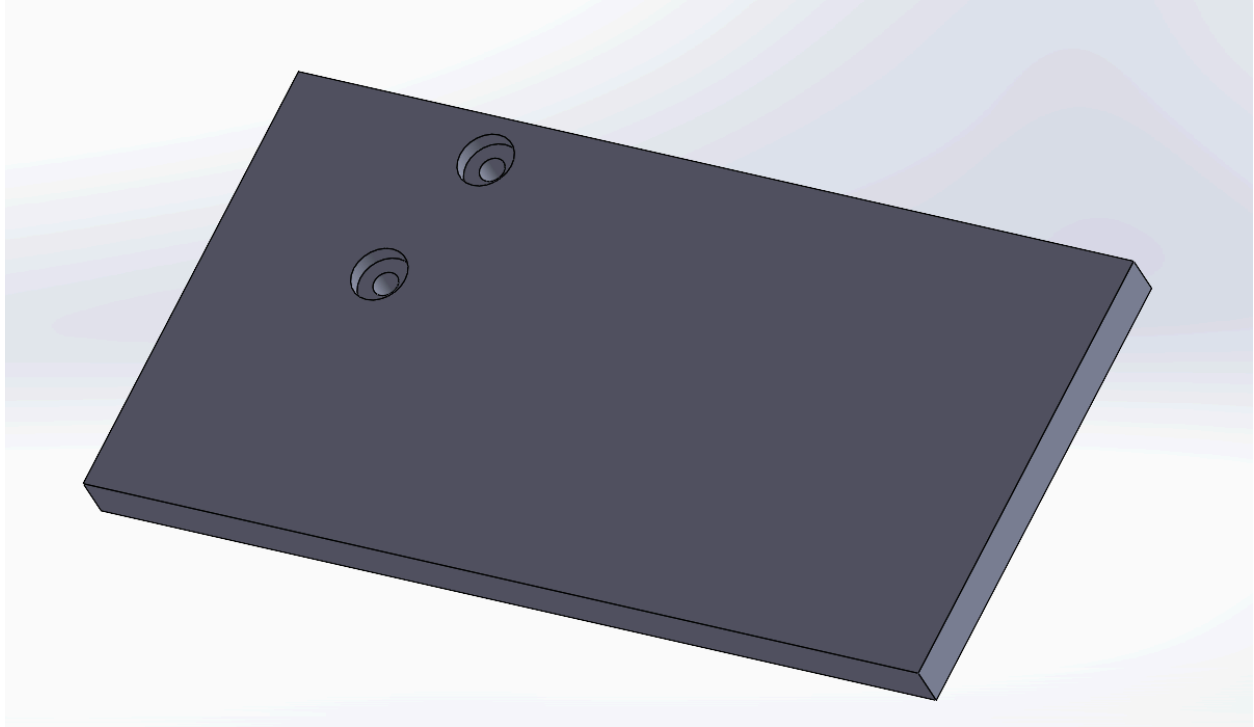
If you want to change the hole settings later, go to the **FeatureManager Design Tree**. Find the **Hole Wizard** feature. Right-click the **Hole Wizard** feature and select **Edit Feature**. This will reopen the **Hole Wizard PropertyManager**. Here, you can change the hole type, standard, size, end 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.



11. Linear Pattern

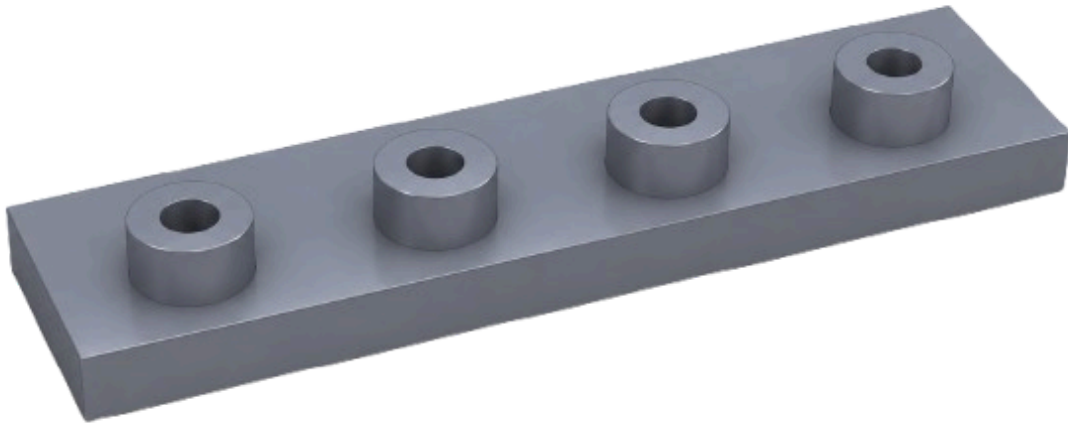
What it does: Repeats features in one or two linear directions.

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

How it works:

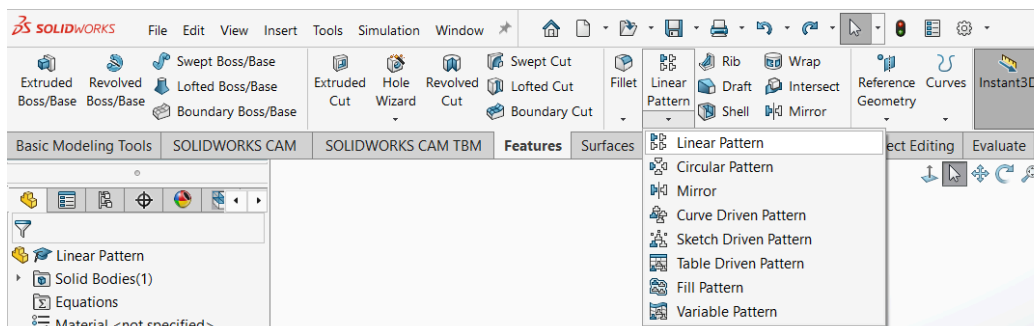
The **Linear Pattern** tool is used to copy an existing feature in one or two straight directions.

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.

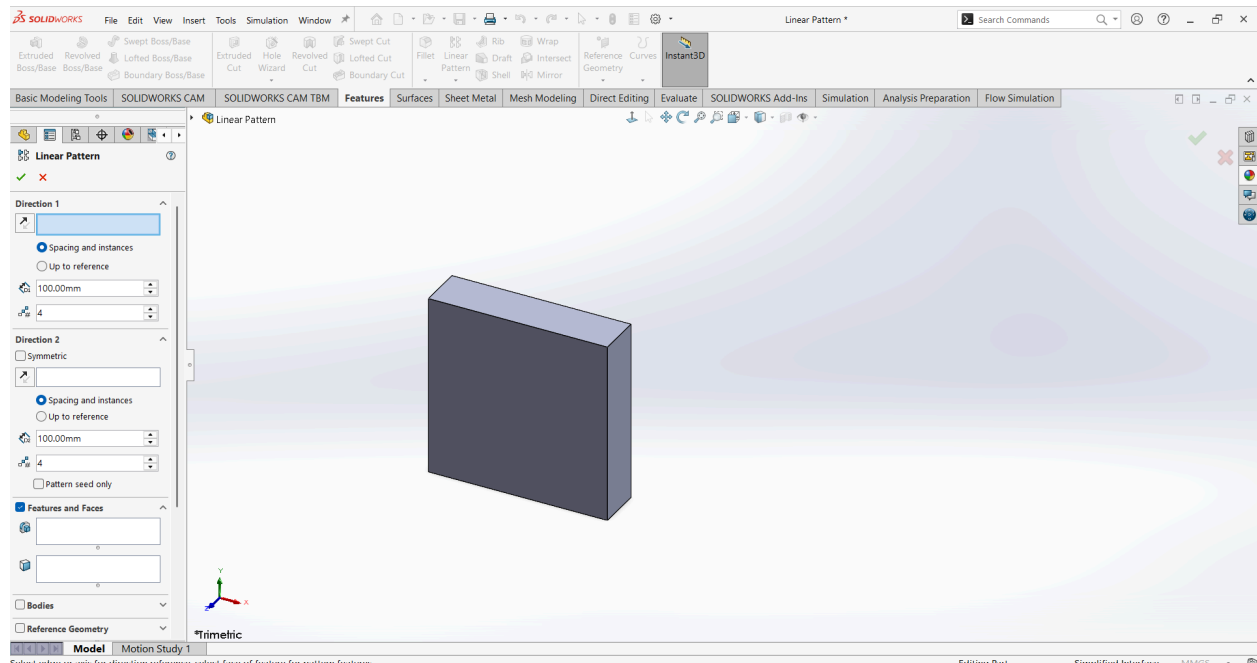


This tool is useful for repeated holes, cuts, bosses, ribs, slots, and other features that follow a straight pattern.

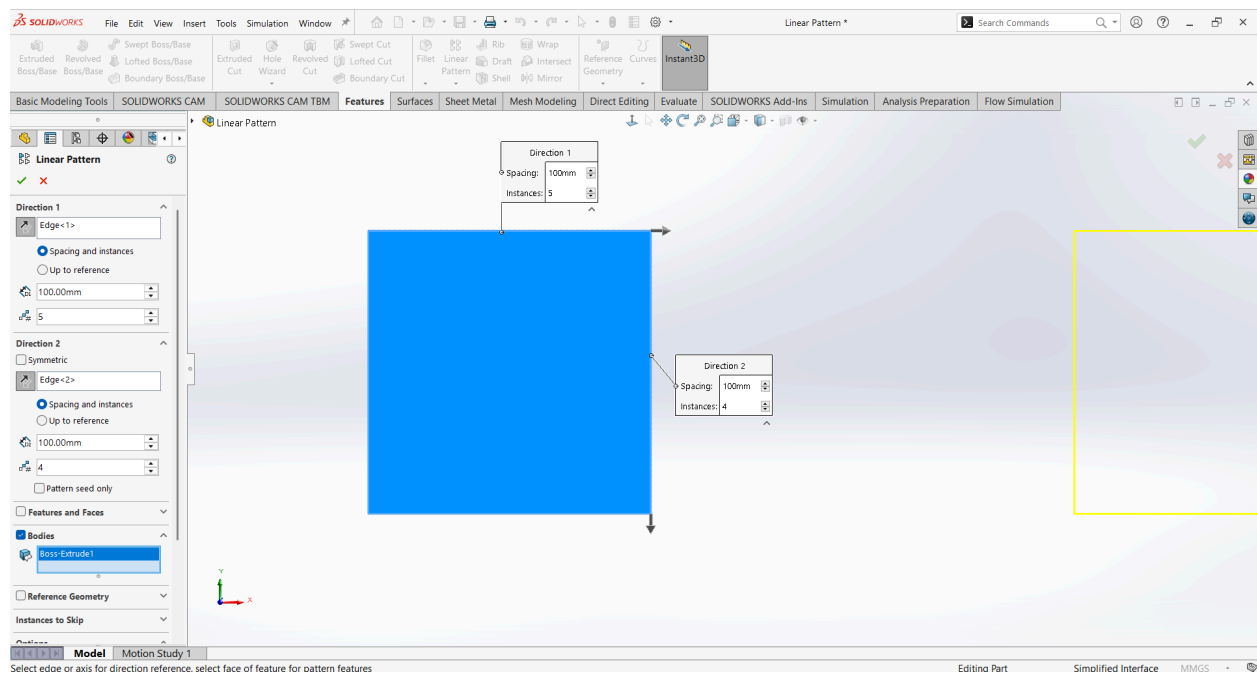
To access **Linear Pattern**, go to the **Features** tab. Find the **Linear Pattern** tool in the toolbar. Click **Linear Pattern** to open the tool settings.



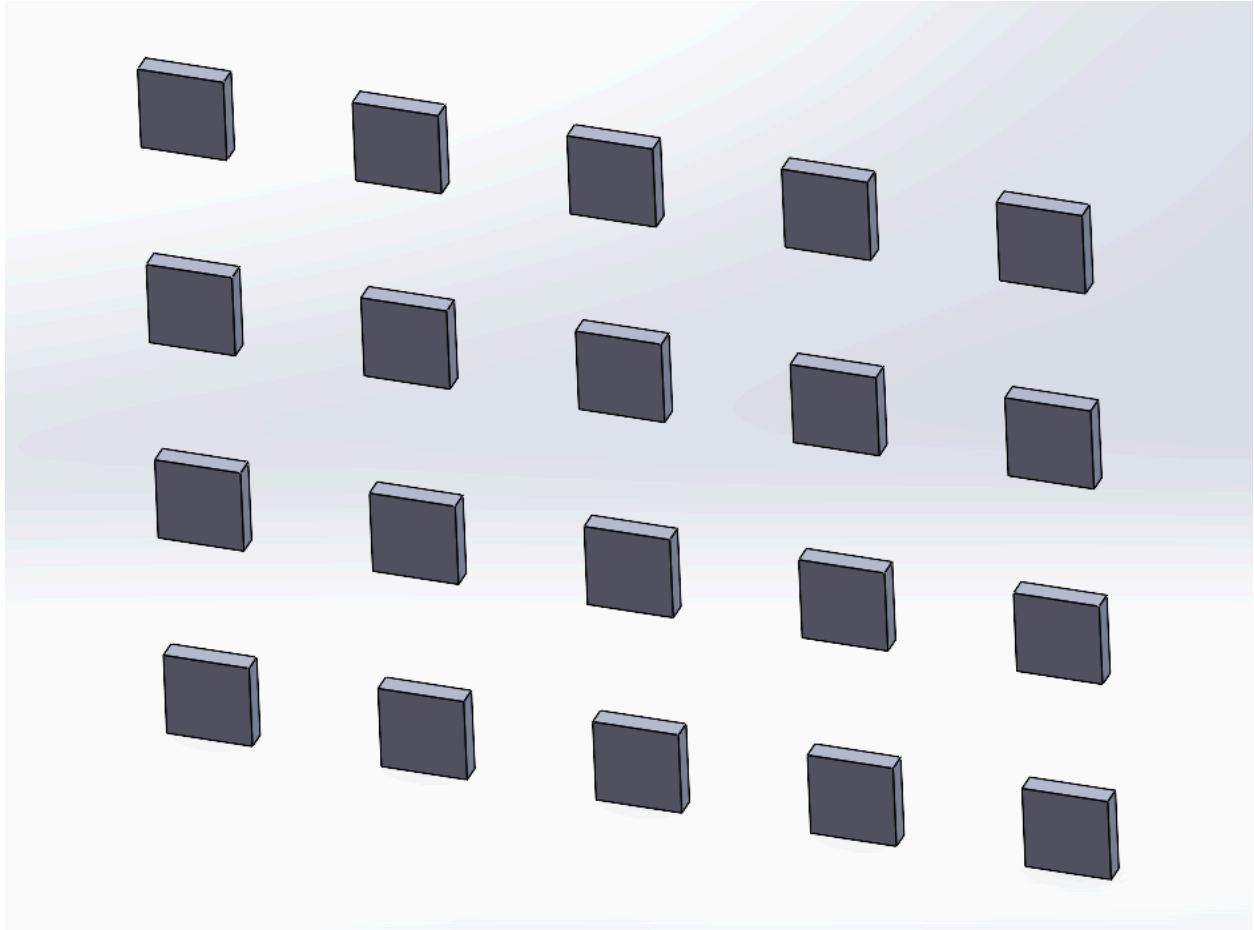
The **PropertyManager** will appear on the left side, where you can define the pattern direction, spacing, number of instances, and the feature you want to pattern.



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.



12. Circular Pattern

What it does: Repeats features around an axis.

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

How it works:

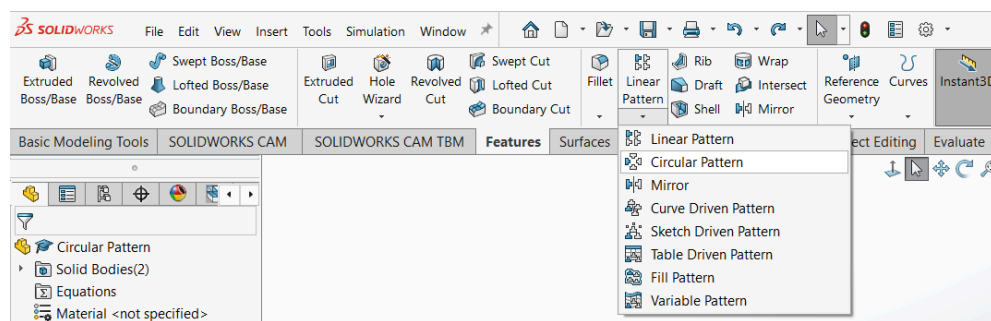
The **Circular Pattern** tool is used to copy an existing feature, body, or face around a circular direction.

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.

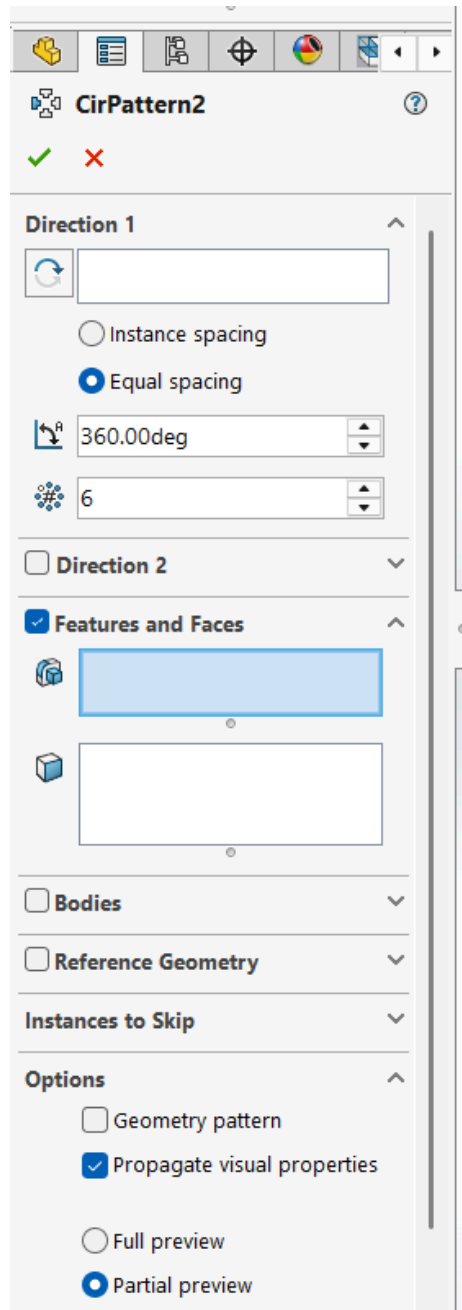


This tool is useful for repeated holes, cuts, bosses, ribs, slots, and other features arranged in a circular pattern.

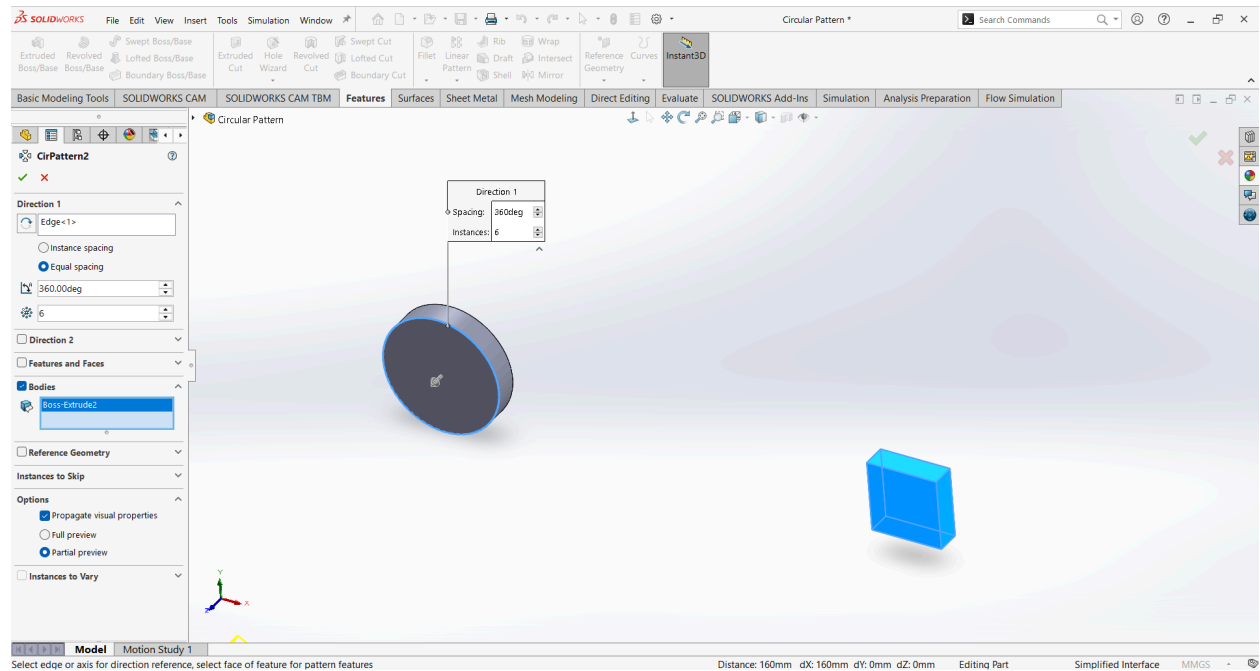
To access **Circular Pattern**, go to the **Features** tab. Find the **Linear Pattern** tool in the toolbar. Click the small arrow below **Linear Pattern** to open the drop-down menu. From the list of available pattern tools, select **Circular Pattern**.



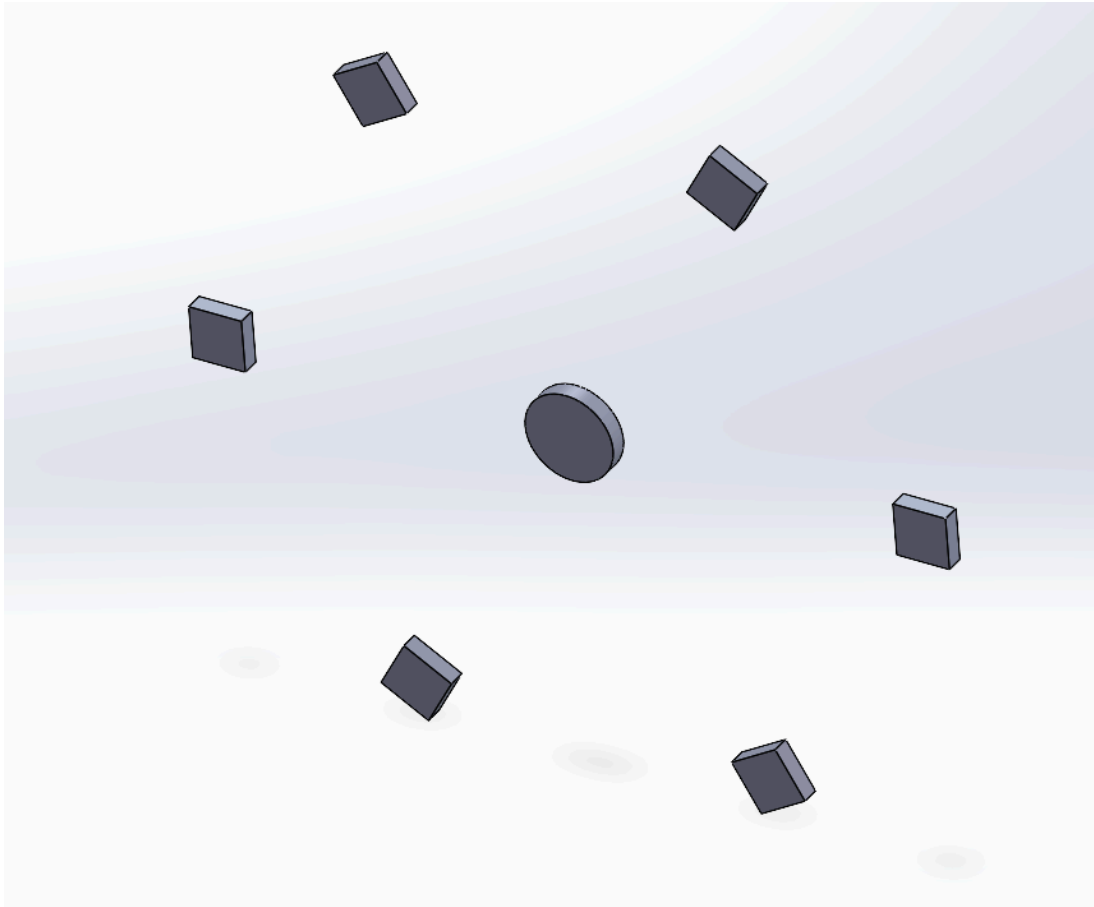
The **PropertyManager** will appear on the left side, where you can define the pattern direction, number of instances, spacing angle, and the feature or body you want to pattern.



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.



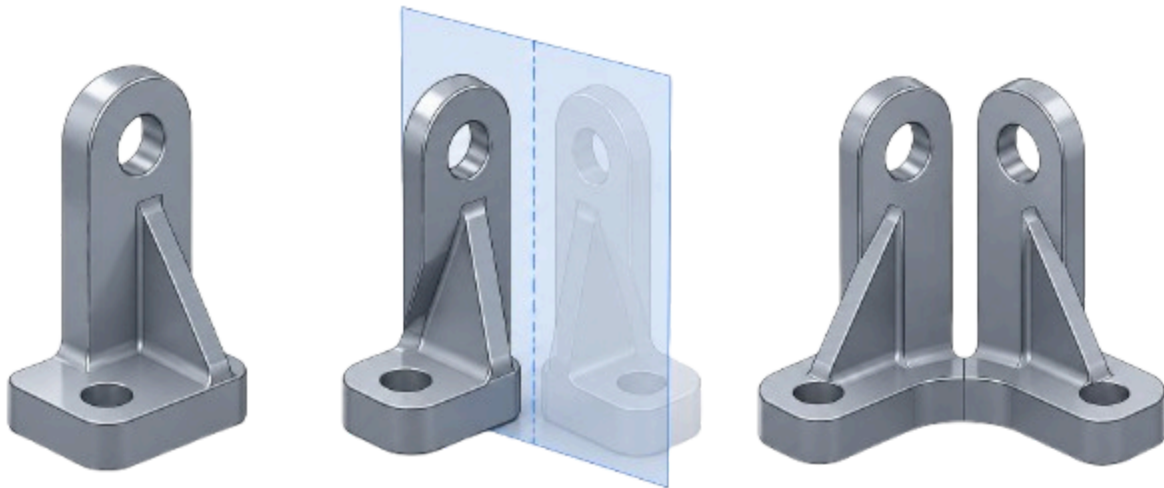
13. Mirror

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

Use it for: Symmetrical components.

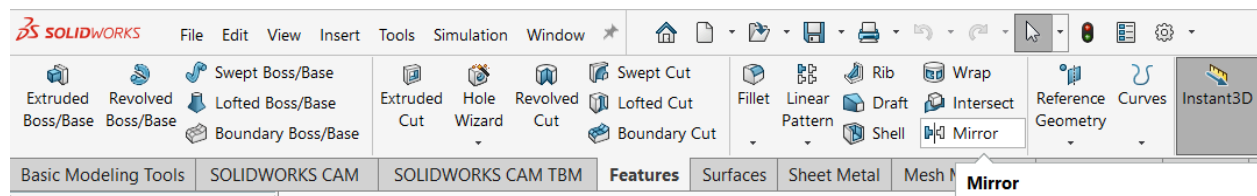
How it works:

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

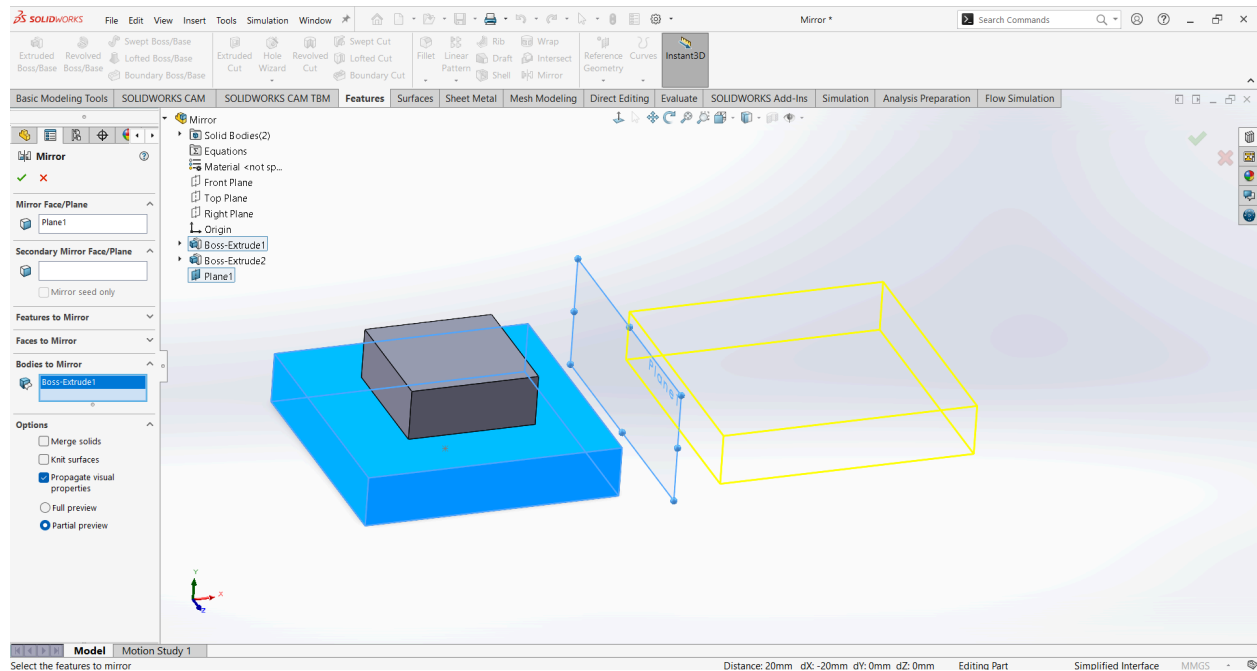


The **Mirror** tool creates a mirrored copy of selected features, faces, bodies, or components using a plane or flat face as the mirror reference.

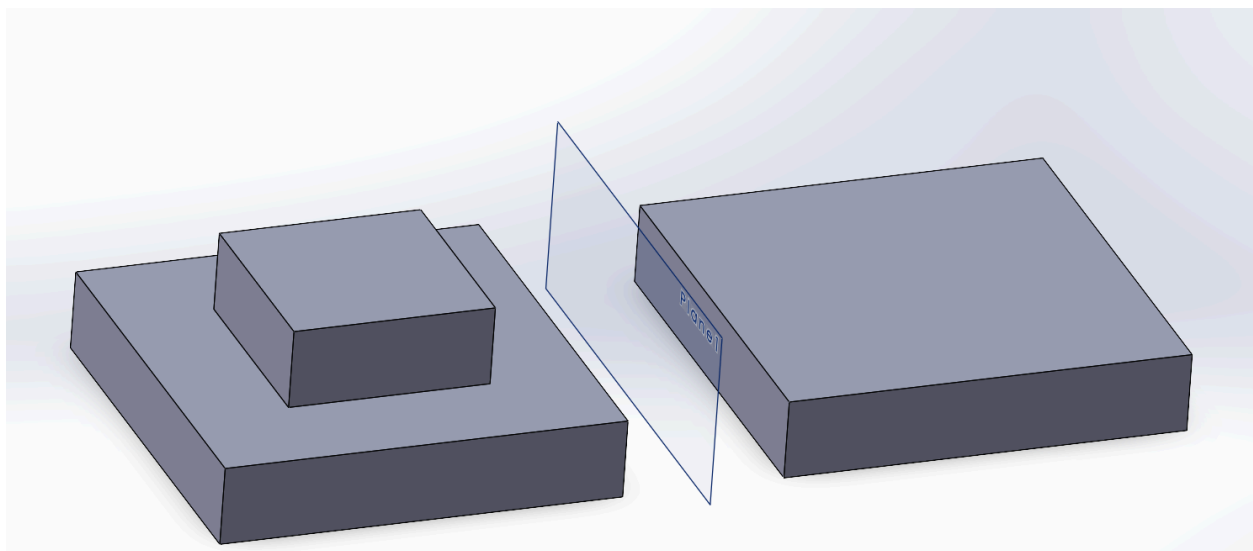
Go to the **Features** tab and select **Mirror**.



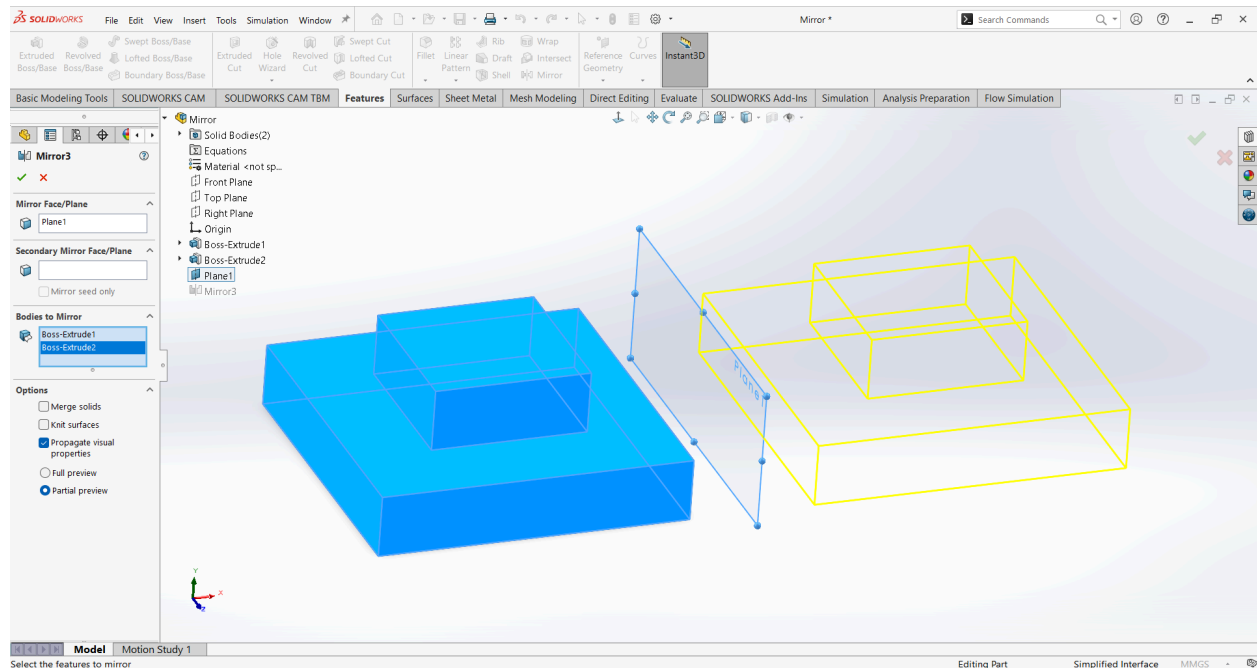
The mirror properties appear on the left side in the **PropertyManager**. In the first selection box, select the mirror reference. Select **Plane1** as the mirror plane. Then go to **Bodies to Mirror** and select the lower body. You can see the mirror preview before confirming the feature.



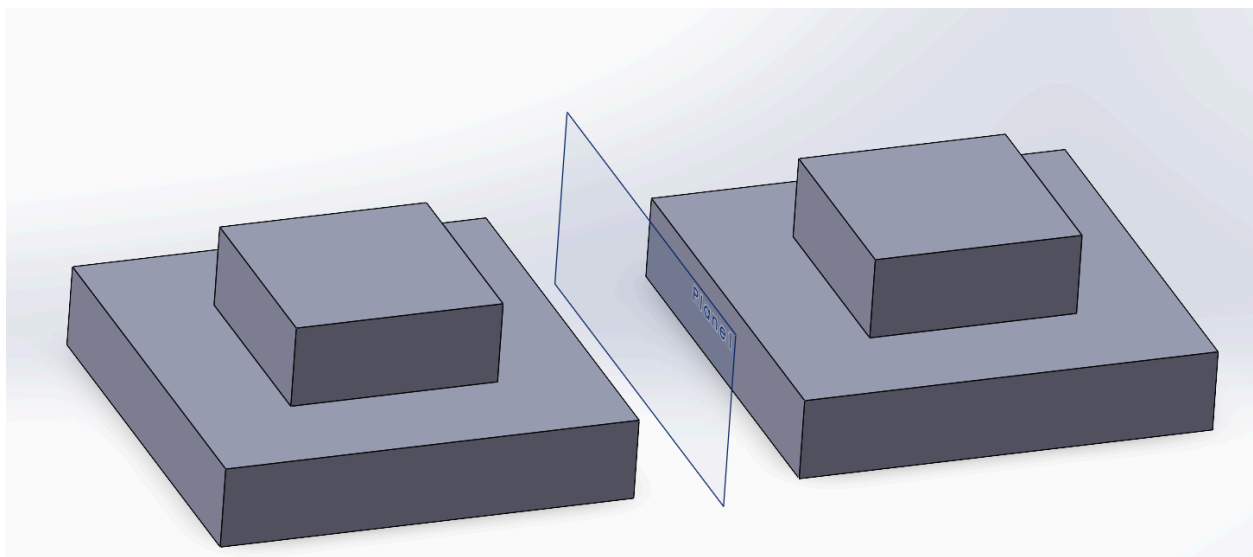
Click **OK**. This is the final result.



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.



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