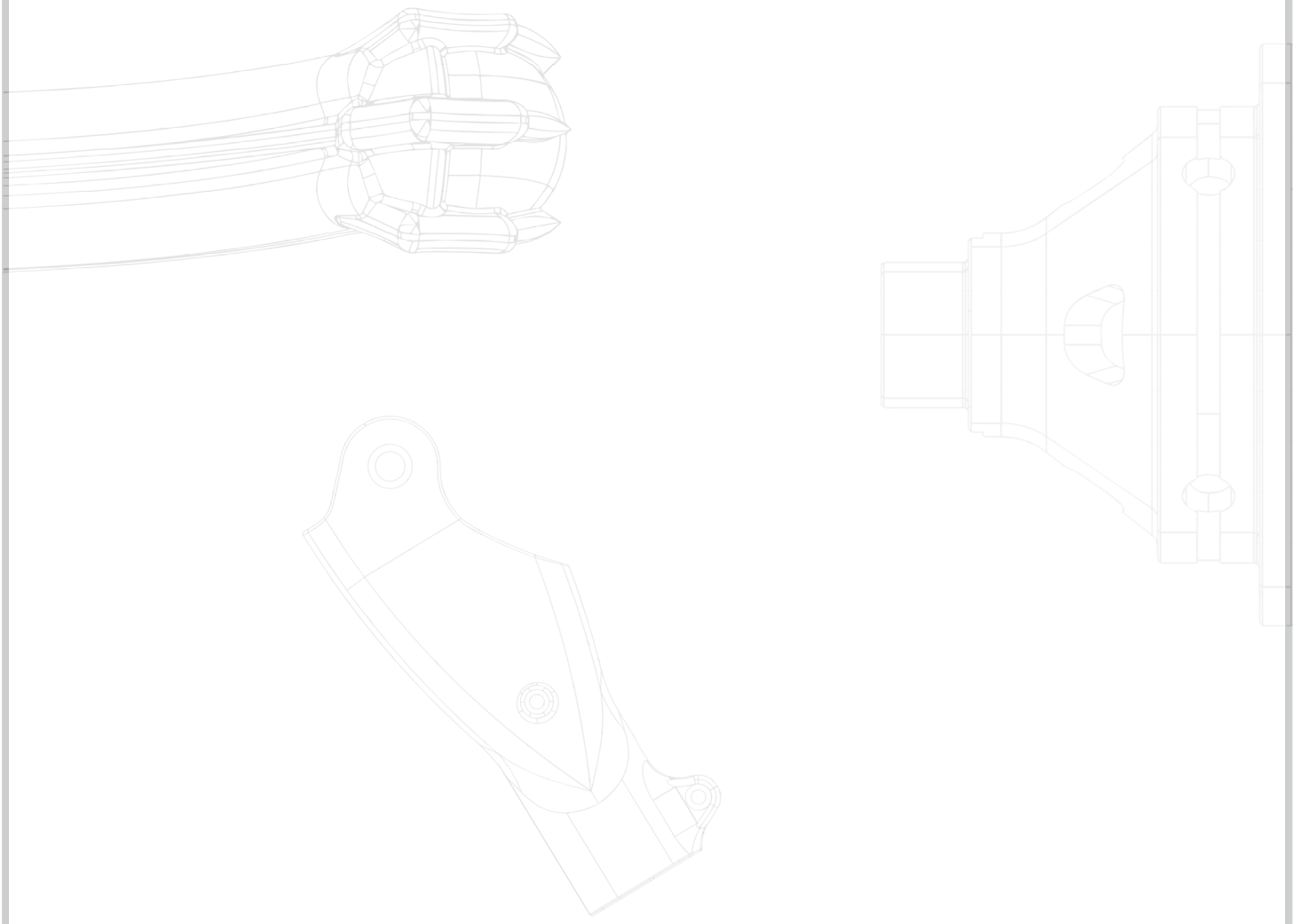
MASTERCAM SOLIDS TUTORIAL

June 2018



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Mastercam

MASTERCAM SOLIDS TUTORIAL

June 2018

Software: Mastercam 2019

Terms of Use

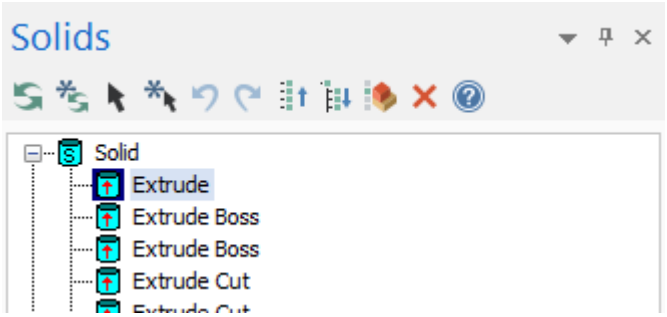
Be sure you have the latest information!

Information might have changed or been added since this document was published. The latest version of the document is installed with Mastercam or can be obtained from your local Reseller. A ReadMe file (ReadMe.PDF) – installed with each release – includes the latest information about Mastercam features and enhancements.

TABLE OF CONTENTS

INTRODUCTION

Mastercam Solids features functions for creating and editing solid models, with support for both history-based and brick solids (no history tree). When you create a solid model in Mastercam, all operations appear in the solids history tree, where you can edit the solid model in many ways. This information is stored in the Solids Manager.



Mastercam's Model Prep functions let you modify solid models lacking a history tree. Mastercam can identify individual geometry elements and even entire features. Using Model Prep functions such as Push/Pull, Move, and Split Solid Faces, you can make changes to a solid, despite having no access to the model's original wireframe.

Goals

- Create a solid model from wireframe geometry.
- Edit a solid model from the solid history tree.
- Create holes using the Hole function.
- Modify brick solid models with Model Prep functions.

Preparing the Files

This tutorial includes the files needed to complete the exercises. You can find these files in the tutorial's `Parts` folder, which itself contains the subfolders `FlangeDemo`, `ImpressionDemo`, `MarkerTray`, `PressurePlate`, `ShippingPullTab`, `SolidsHole`, and `SpindlePartial`. Place these files on your system, but keep an unmodified set. In preparation for the following exercises, set up Mastercam as described here.

Each folder includes a set files marked with numbers and are the results of each exercise. For example, `MarkerTray01` is the result of Exercise 1 and is in the state the part should be in when starting Exercise 2.

The only exception is the file used for "Using the Solid Hole Function" on page 53. You will be reusing the same part to create different holes.

WARNING: Screen colors in the tutorial pictures were modified to enhance image quality; they may not match your Mastercam settings or the tutorial results. These color differences do not affect the lesson or your results.

Estimated time to complete this tutorial: 4 hours

General Tutorial Requirements

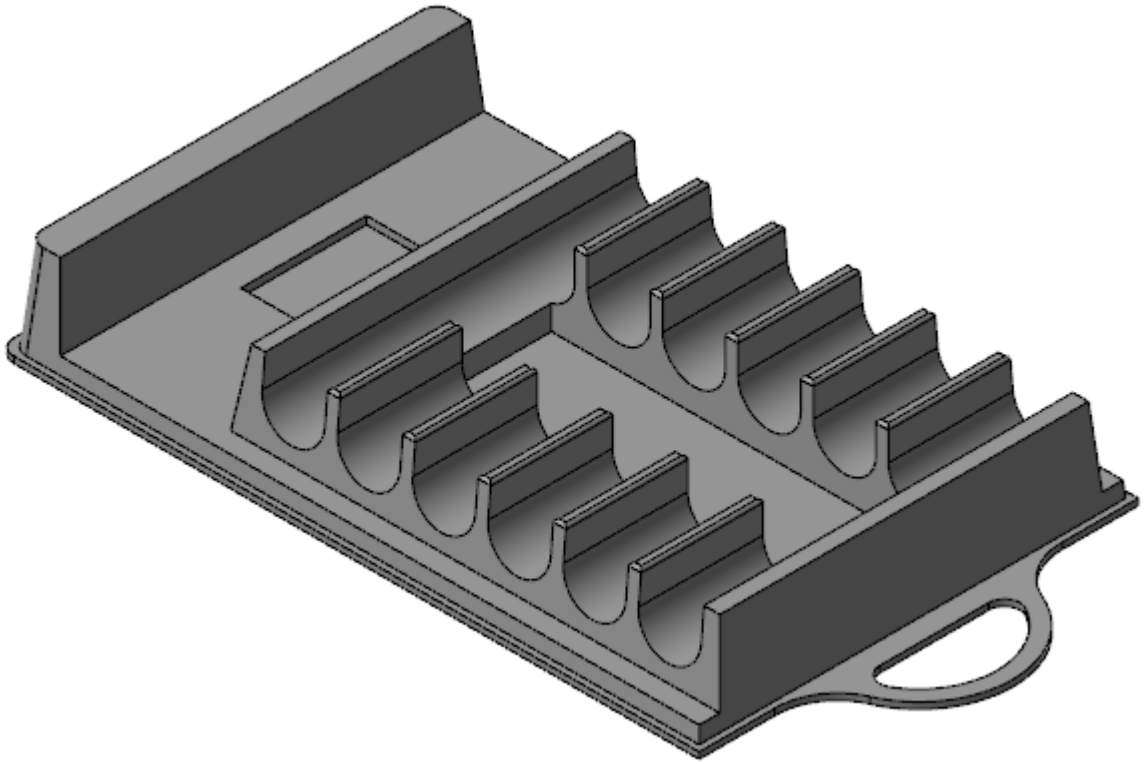
All Mastercam 2019 tutorials have the following general requirements:

- You must be comfortable using the Windows® operation system.
- The tutorials cannot be used with Mastercam Demo/Home Learning Edition. The Demo/HLE file format (`emcam`) is different from Mastercam (`mcam`), and basic Mastercam functions, such as file conversions and posting, are unavailable.
- Each lesson in the tutorial builds on the mastery of the preceding lesson's skills. We recommend that you complete them in order.
- Additional files may accompany a tutorial. Unless the tutorial provides specific instructions on where to place these files, store them in a folder that can be access from the Mastercam 2019 workstation, either with the tutorial or in any location that you prefer.
- You will need an internet connection to view videos that are referenced in the tutorials. All videos can be found on our YouTube channel:
  www.youtube.com/user/MastercamTechDocs
- All Mastercam tutorials require you to configure Mastercam to work in a default Metric or Inch configuration. The tutorial provides instructions for loading the appropriate configuration file.

CHAPTER 1

USING THE EXTRUDE, DRAFT, FILLET, AND SHELL FUNCTIONS

Mastercam Solids features tools you need to create solid models, using functions with dynamic preview so you can see the results as soon as you edit parameters. In this chapter, you will create a marker tray using the Extrude, Draft, Fillet, and Shell functions.



Goals

- Learn to use the Extrude, Draft, Fillet, and Shell functions.
- Create a marker tray part from start to finish.

Exercise 1: Creating the Tray Base

1. Start Mastercam using your preferred method:
   a. Double-click Mastercam's desktop icon.

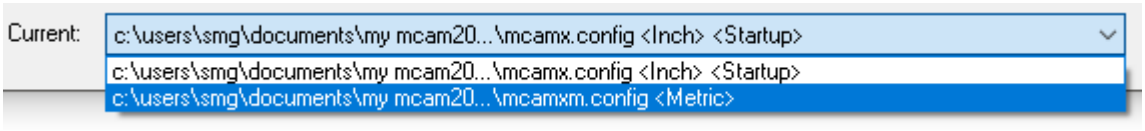
   

   OR

   b. Launch Mastercam from the Windows Start menu.

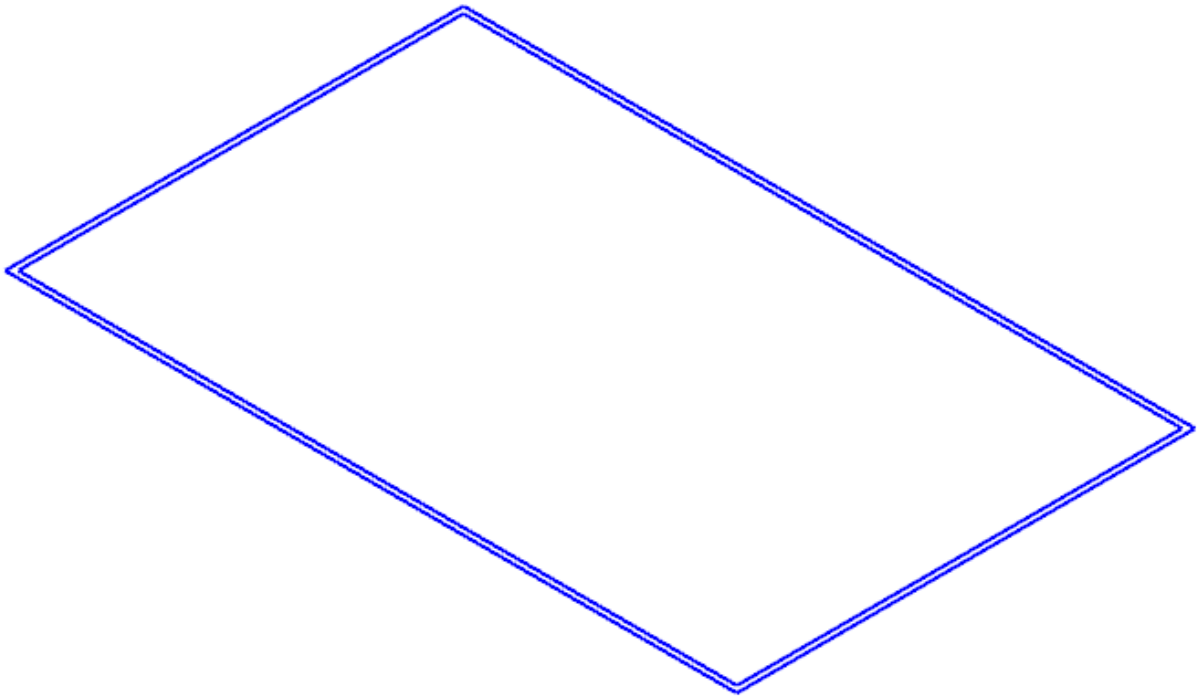
2. Select the default metric configuration file:
    a. Click the **File** tab.
    b. Choose **Configuration** from Mastercam's Backstage View to open the **System Configuration** dialog box.

    c. Choose `...\mcamxm.config <Metric>` from the **Current** drop-down list.

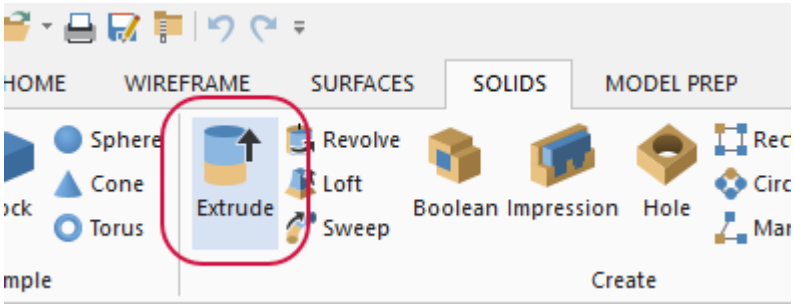


    d. Click **OK**.

1. Open the file `MarkerTray`, provided with this tutorial.

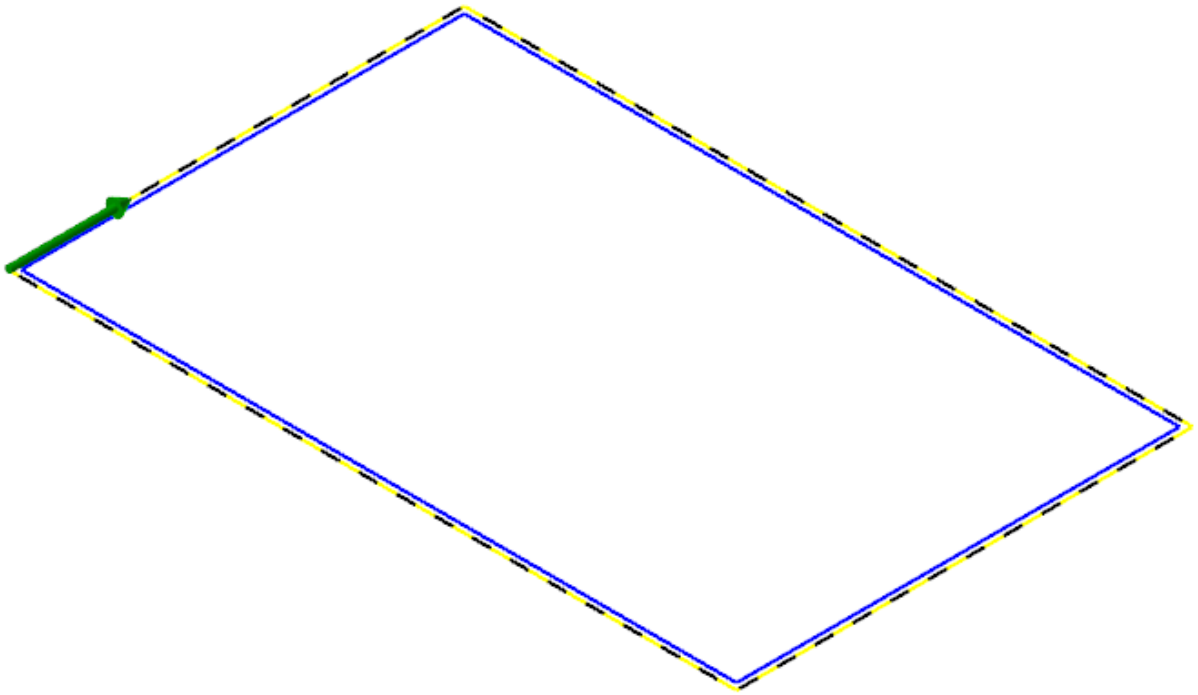


2. Select the **Solids** tab, and then **Extrude**.



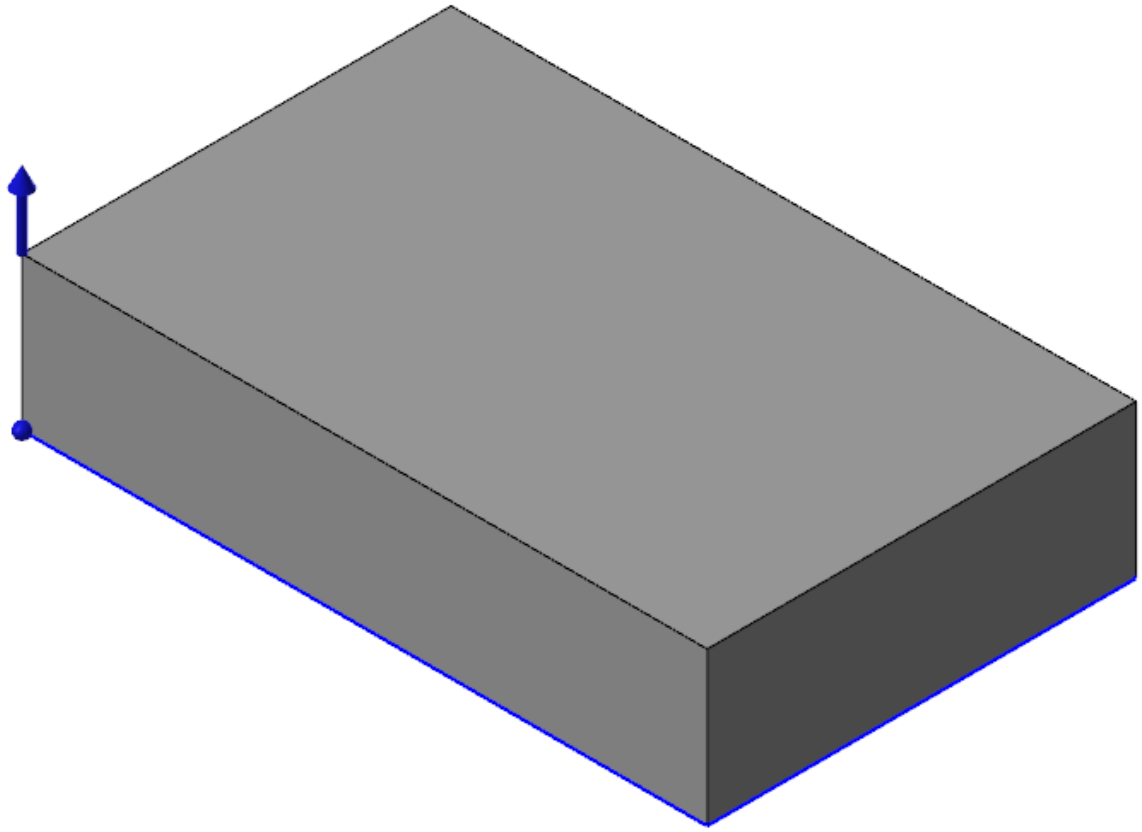
The **Chaining** dialog box opens.

3. Chain the outer rectangle, as shown in the picture below.

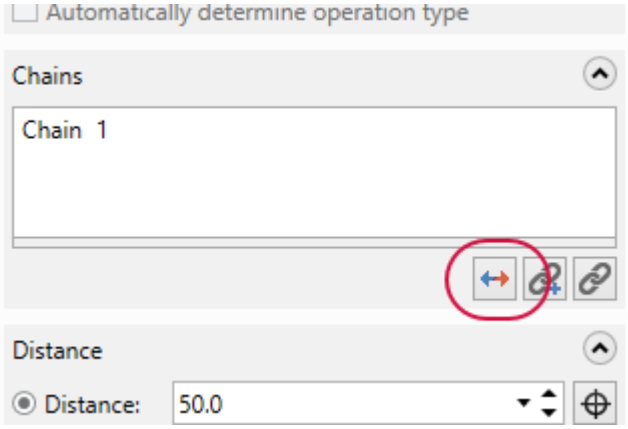


4. Click **OK** in the **Chaining** dialog box.

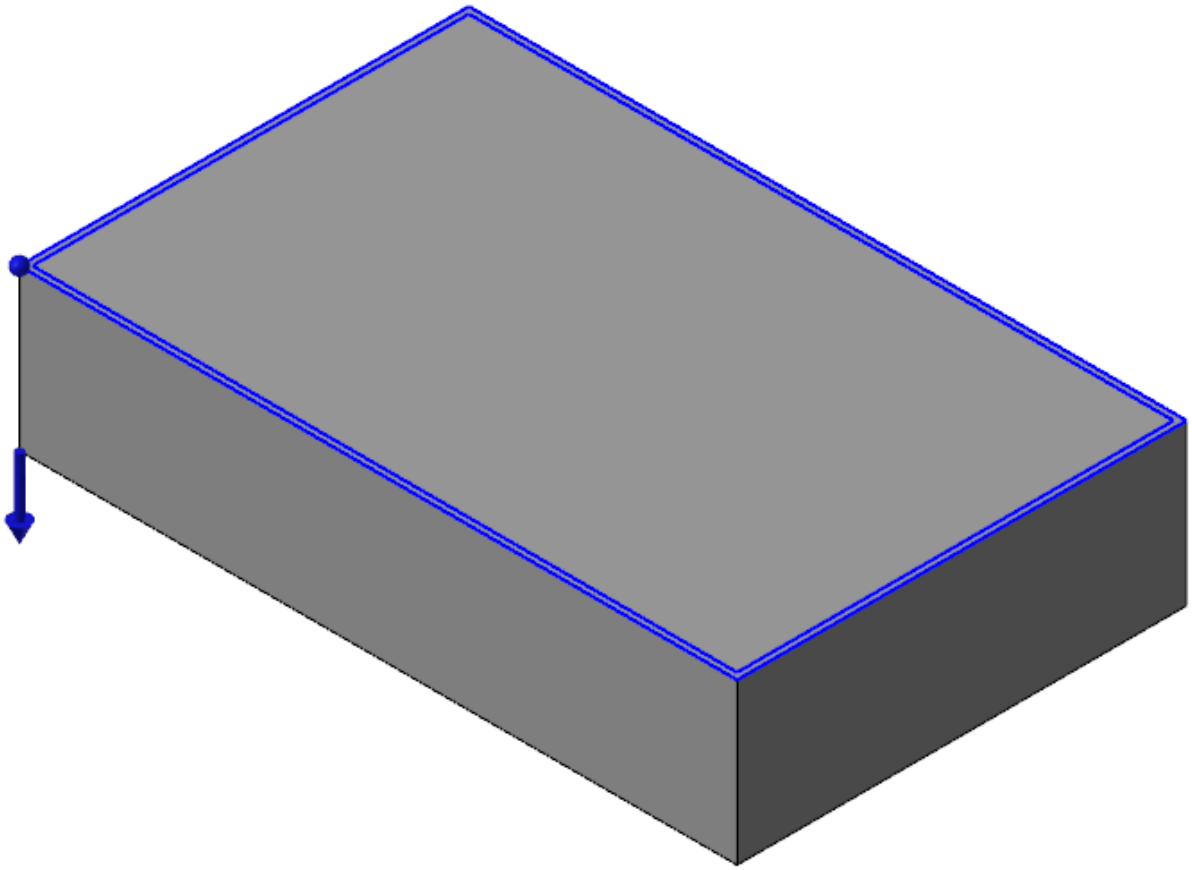
   Mastercam performs an extrusion on the selected chain and displays the **Solid Extrude** function panel.



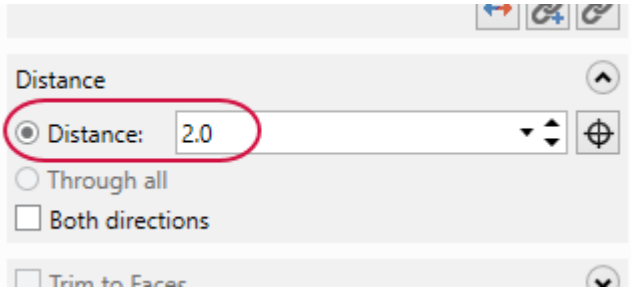
5. In the function panel, click **Reverse All**.



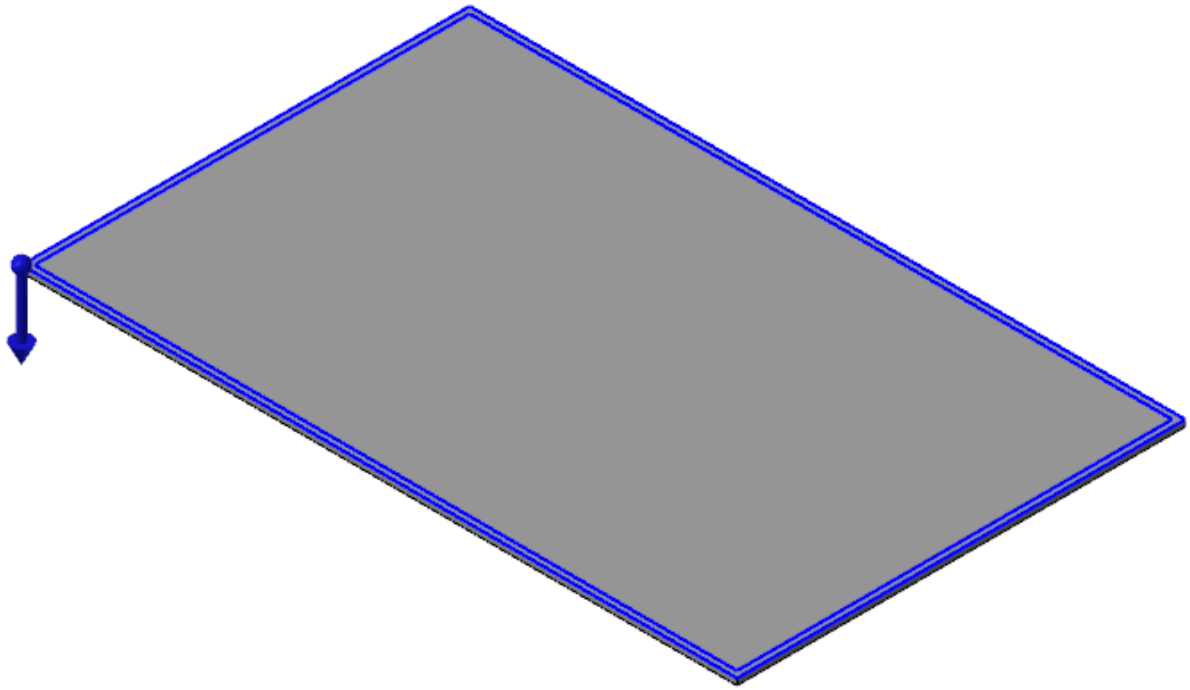
Mastercam reverses the extrude direction.



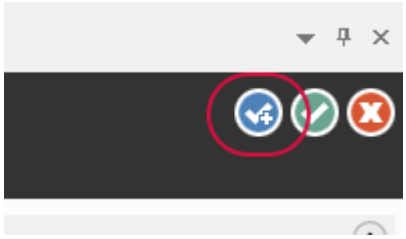
6. Set the extrude **Distance** to **2**, and press [**Tab**] or [**Enter**] to accept the new setting.



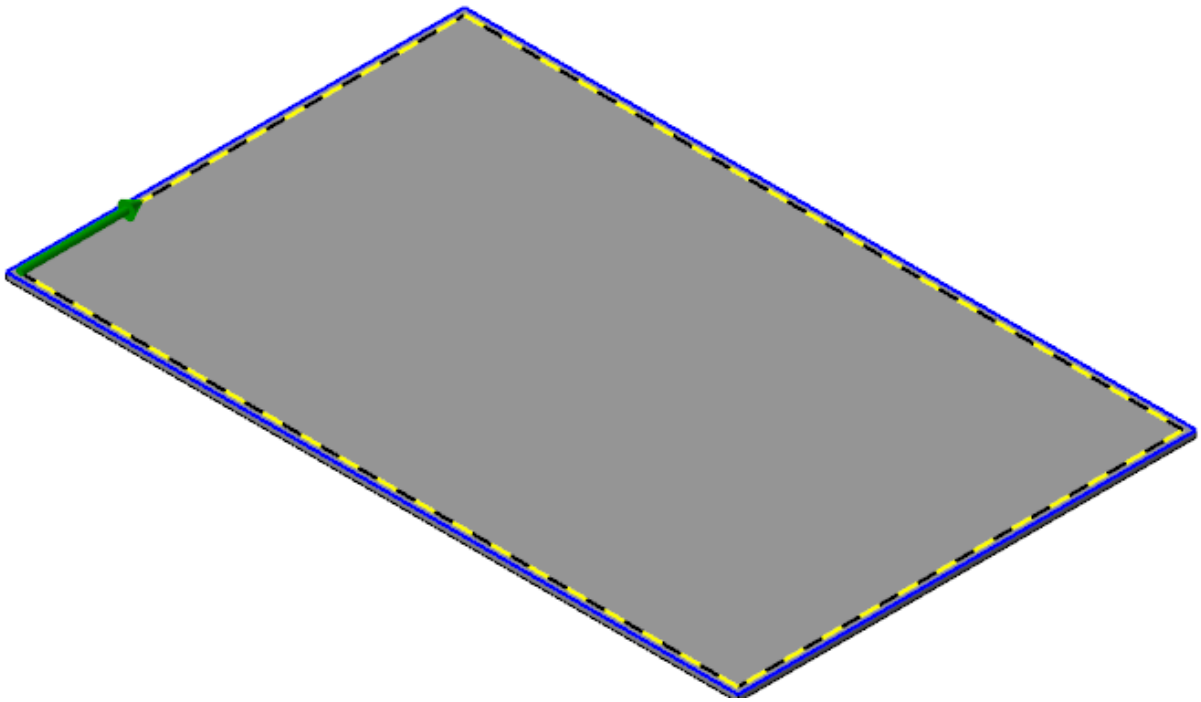
The part displays as shown below:



7. Click **OK and Create New Operation** to finalize the extrude operation and begin a new one.

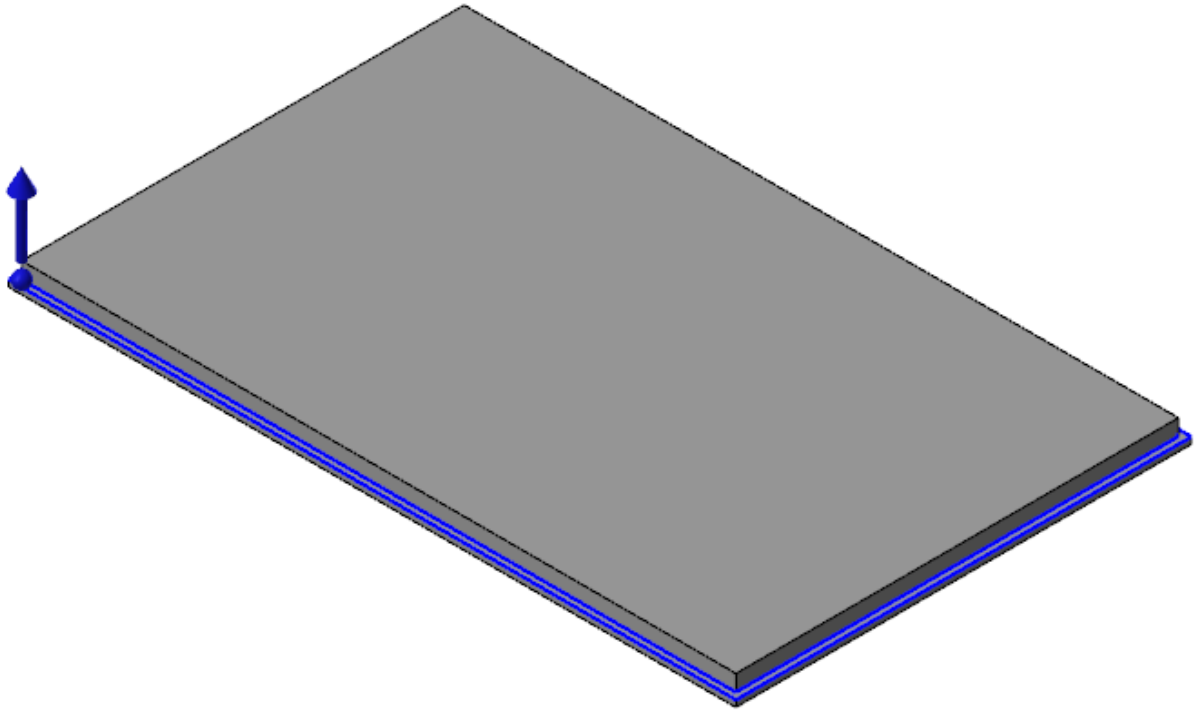


8. Chain the inside rectangle, and click **OK** in the **Chaining** dialog box.

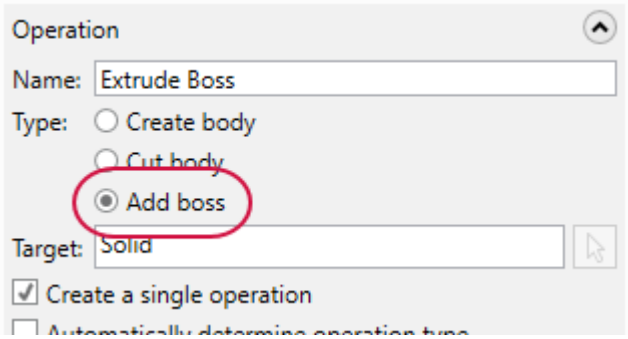


9. Reverse the extrude direction, and set the distance to **5**.

10. Your part should now look like the following picture.

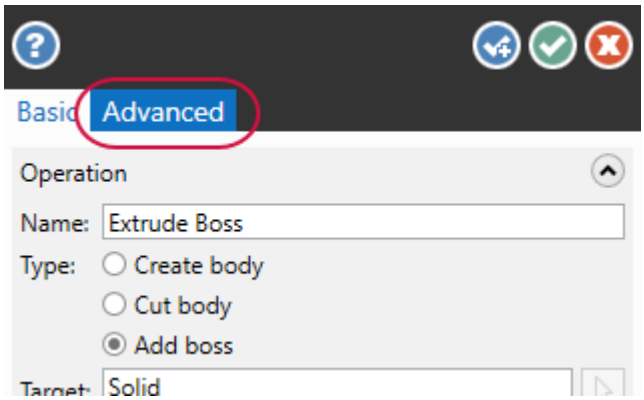


11. Set the **Type** to **Add boss**.

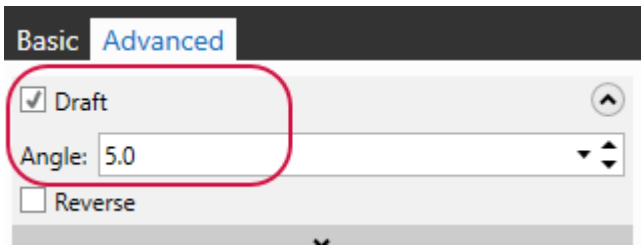


The **Add boss** option creates the extrusion as new material on the part.

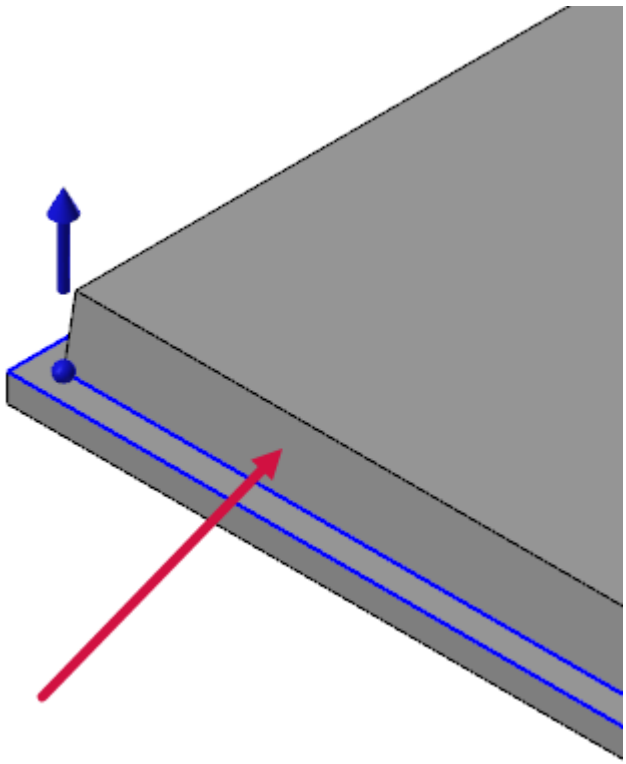
12. Select the **Advanced** tab.



13. Select **Draft** and leave **Angle** at **5**.



14. The graphics window updates to show the draft on the extrusion. If you have difficulty seeing the draft on the part, toggle the **Draft** option several times, watching how the part's geometry changes.



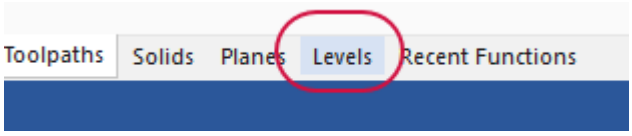
15. In the **Solid Extrude** function panel, click **OK** to create the operation.

16. Save the file as `MarkerTray01-XXX`, where `XXX` is your initials.

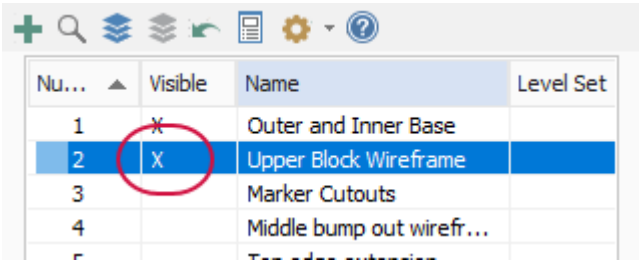
Exercise 2: Creating the Upper Bosses

Now that you have the tray's base, you can add the bosses that form the eraser holder and the marker storage area. For this exercise, continue with the part you created in the previous exercise, or load `MarkerTray01`, which accompanies this tutorial.

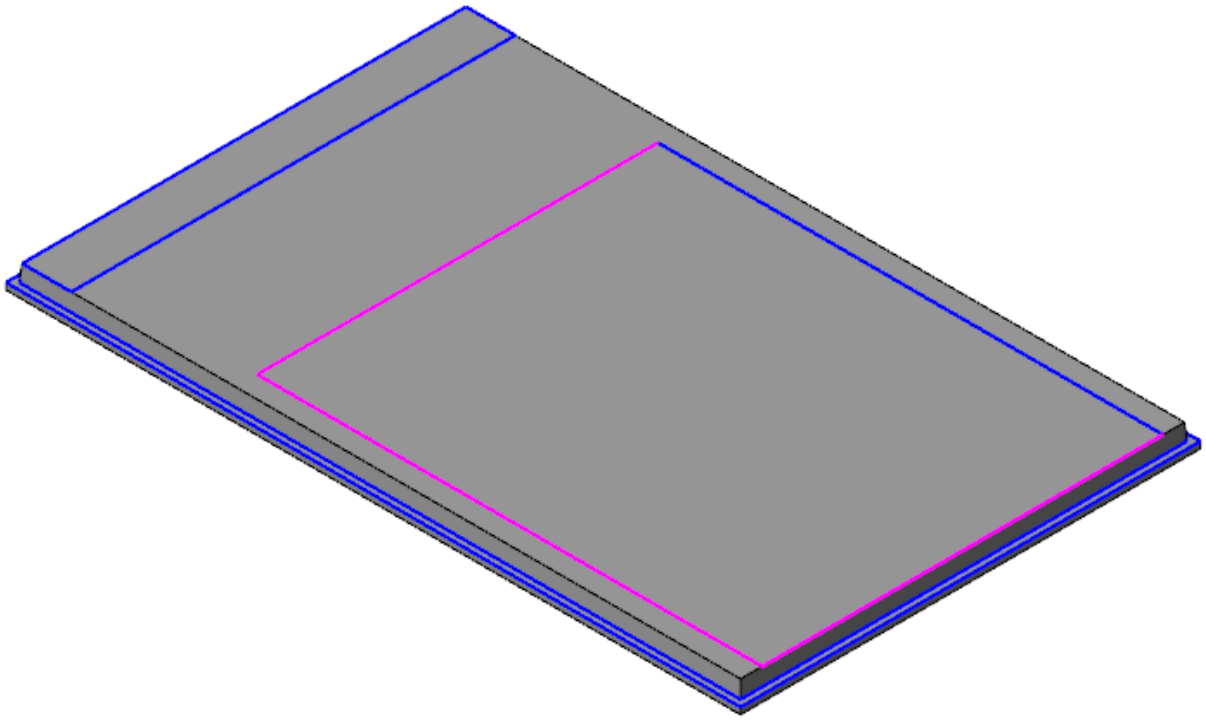
1. Click the **Levels** tab in the lower-left of Mastercam's window.



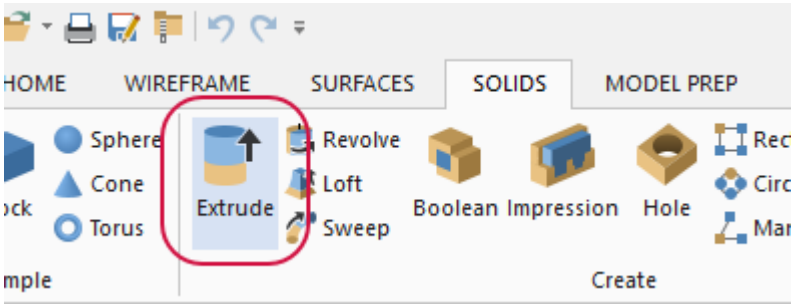
2. Turn on the **Upper Block Wireframe** level, by clicking in its **Visible** column.



The part displays new geometry, as shown in the picture below. This geometry was previously created for you and provides the basis of the new bosses.

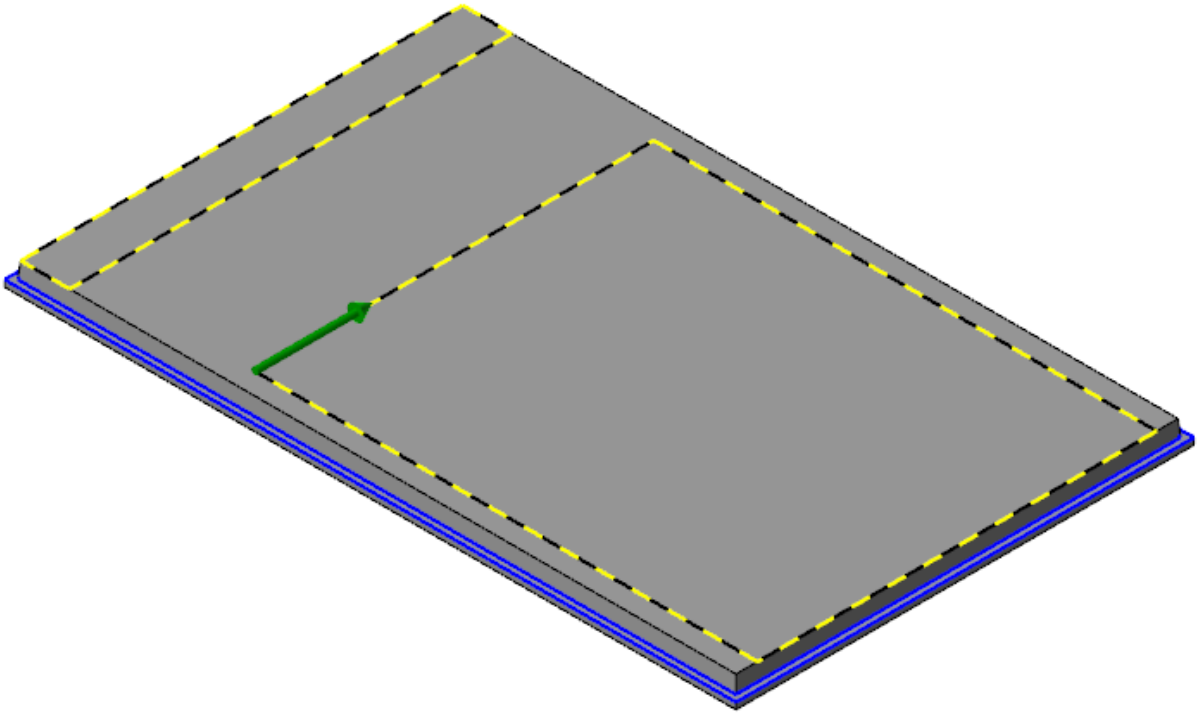


3. Select the **Solids** tab, and then **Extrude**.

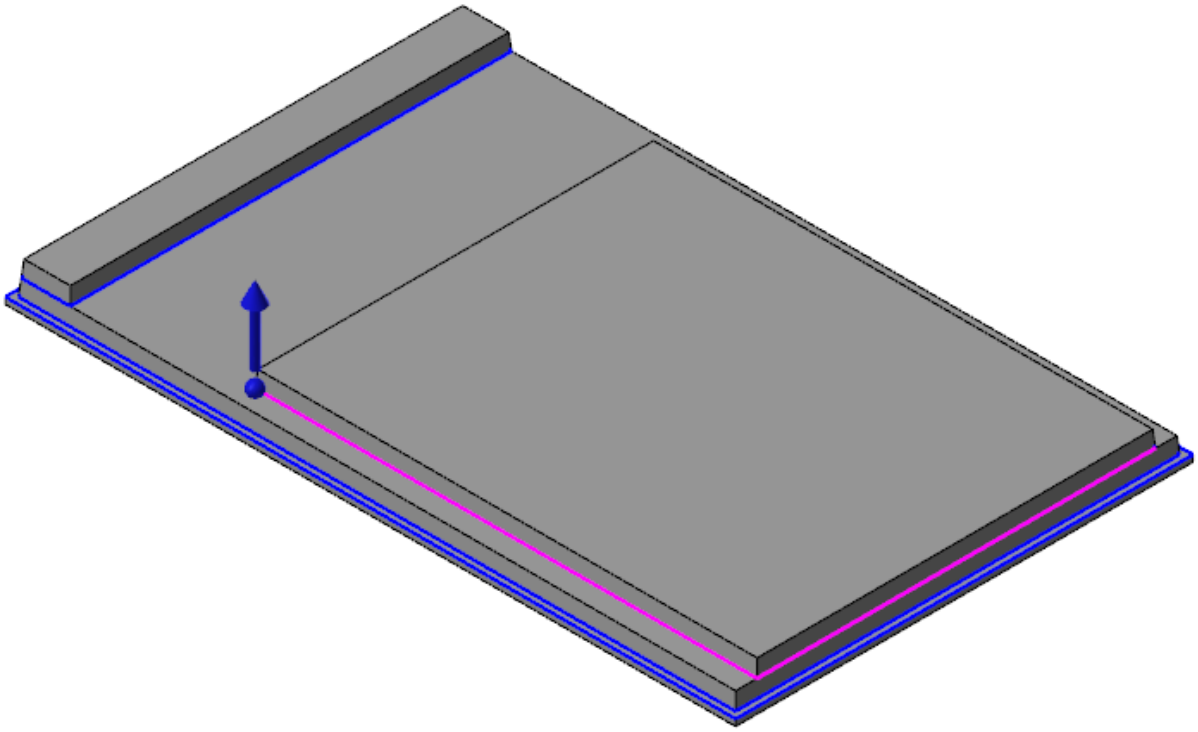


The **Chaining** dialog box opens.

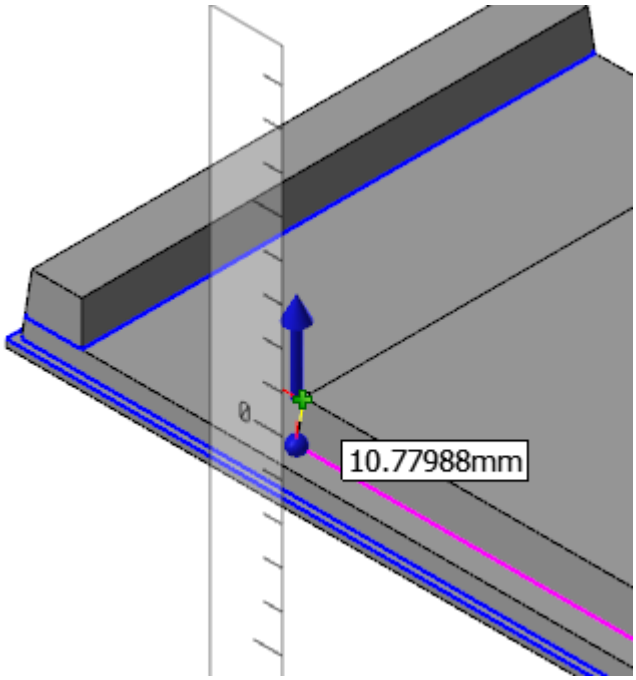
4. Chain the two new rectangles, and click **OK** in the **Chaining** dialog box. The selection order does not matter.



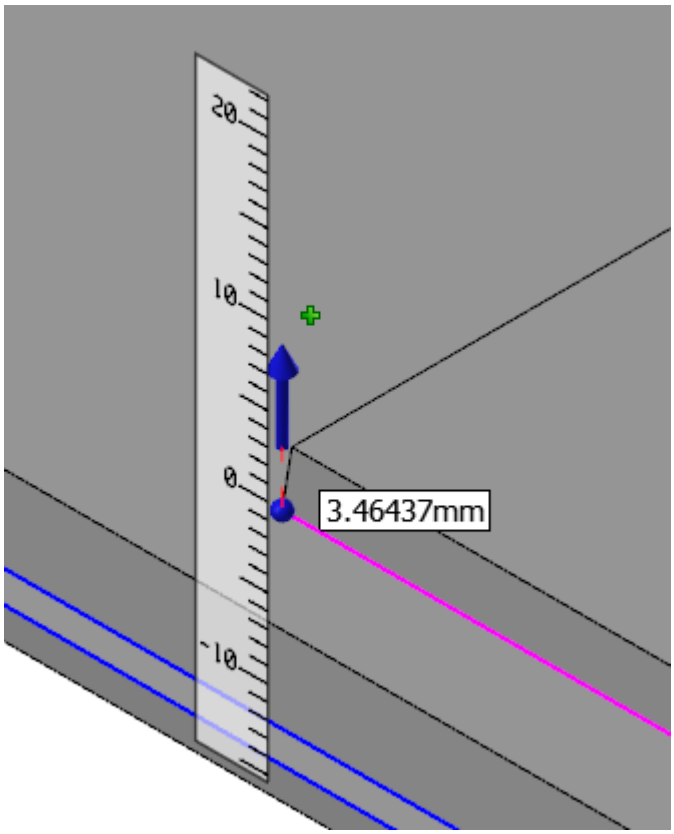
5. Mastercam creates the two new extrusions.



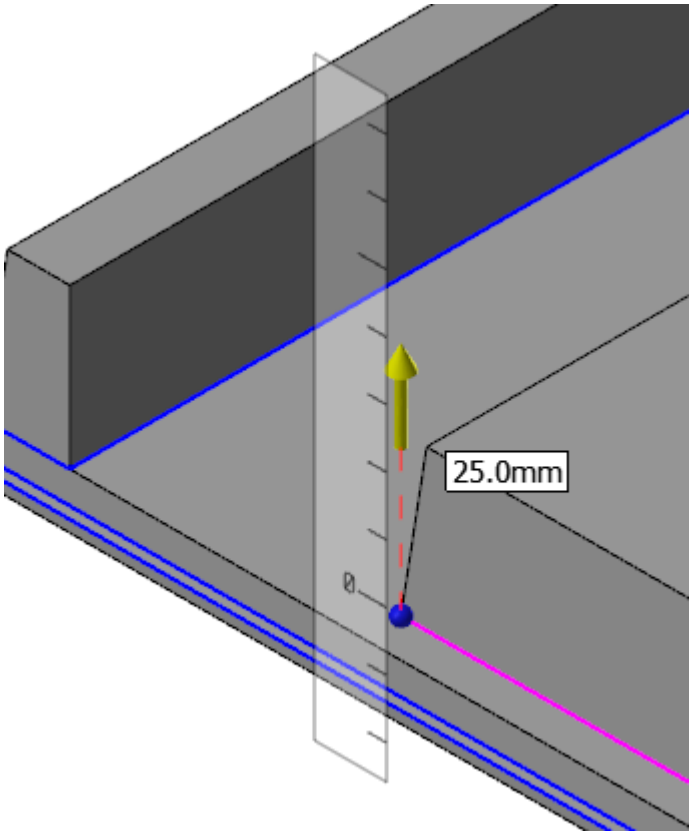
6. Click the blue arrow, and then hover your mouse. A ruler displays, that lets you set the extrude distance.



7. Using your mouse wheel, zoom in closer to the ruler. Mastercam displays smaller increments on the ruler.

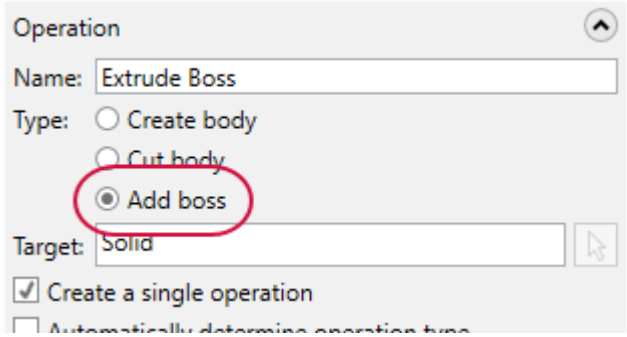


8. Move your mouse up over the ruler until it snaps to **25 mm**, and then click to set the new distance. If you prefer, you can also type values.

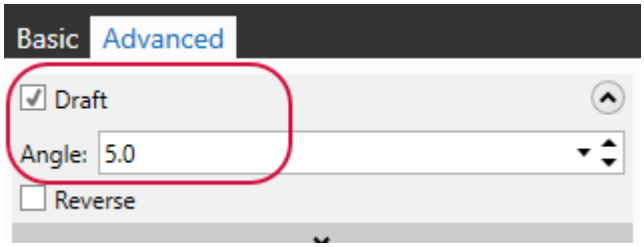


When you move your mouse over the ruler, the selected value snaps to the ruler's increments. When you move the mouse outside the ruler, no snapping is in effect.

9. In the **Solid Extrude** function panel, ensure that **Add boss** is selected.



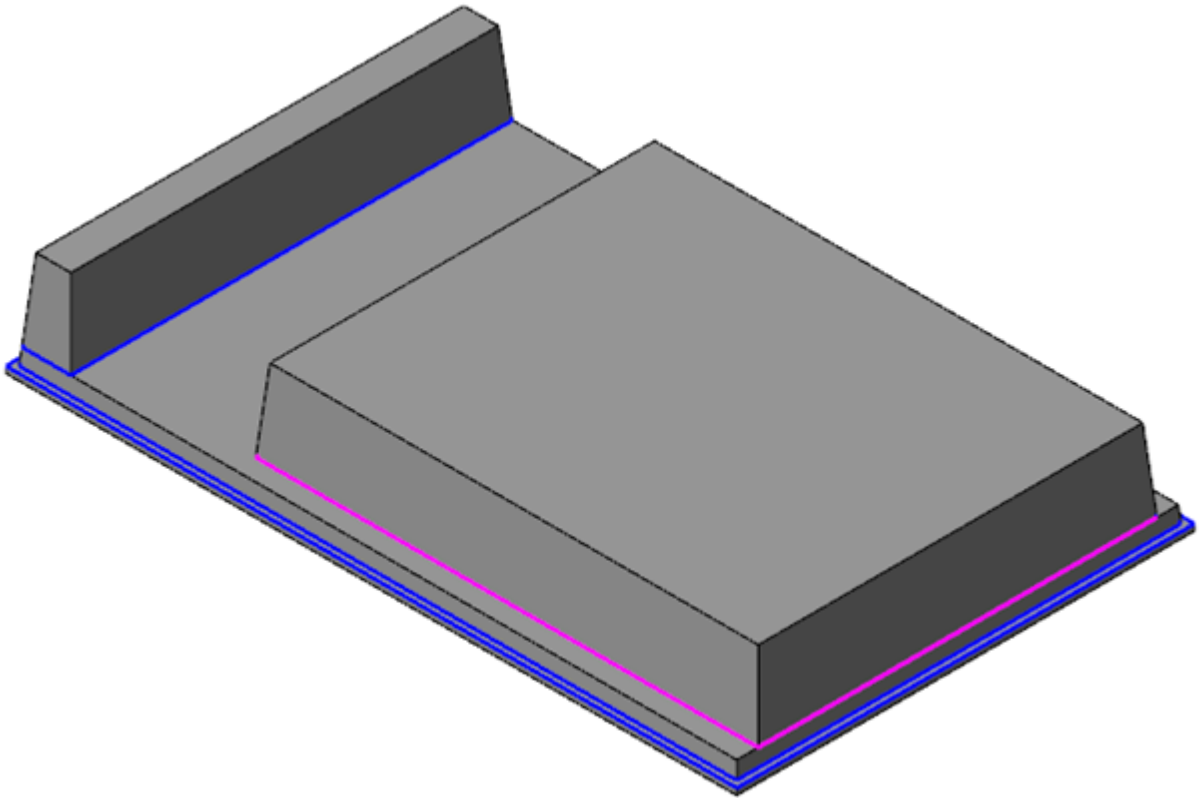
10. Click the **Advanced** tab, and notice that **Angle** is still set to **5**.



Mastercam remembers your settings for the current session.

11. In the function panel, click **OK** to create the operation.

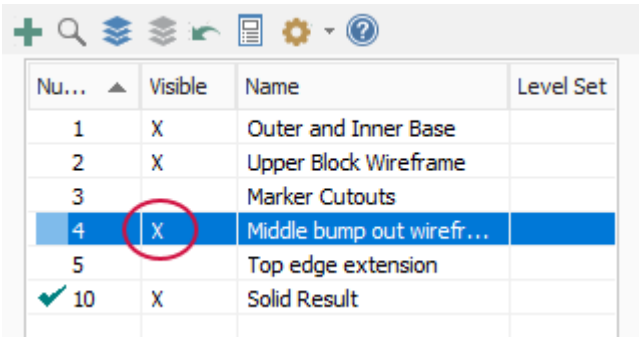
12. Save the file as `MarkerTray02-XXX`.



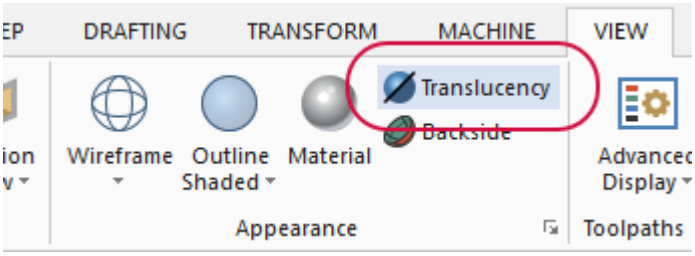
Exercise 3: Creating the Finger Cutouts

Next, you create finger cutouts in the new extrusions. These cutouts make it easier for the tray's user to get their fingers under the eraser or the markers. For this exercise, continue with the part you created in the previous exercise, or load `MarkerTray02`, which accompanies this tutorial.

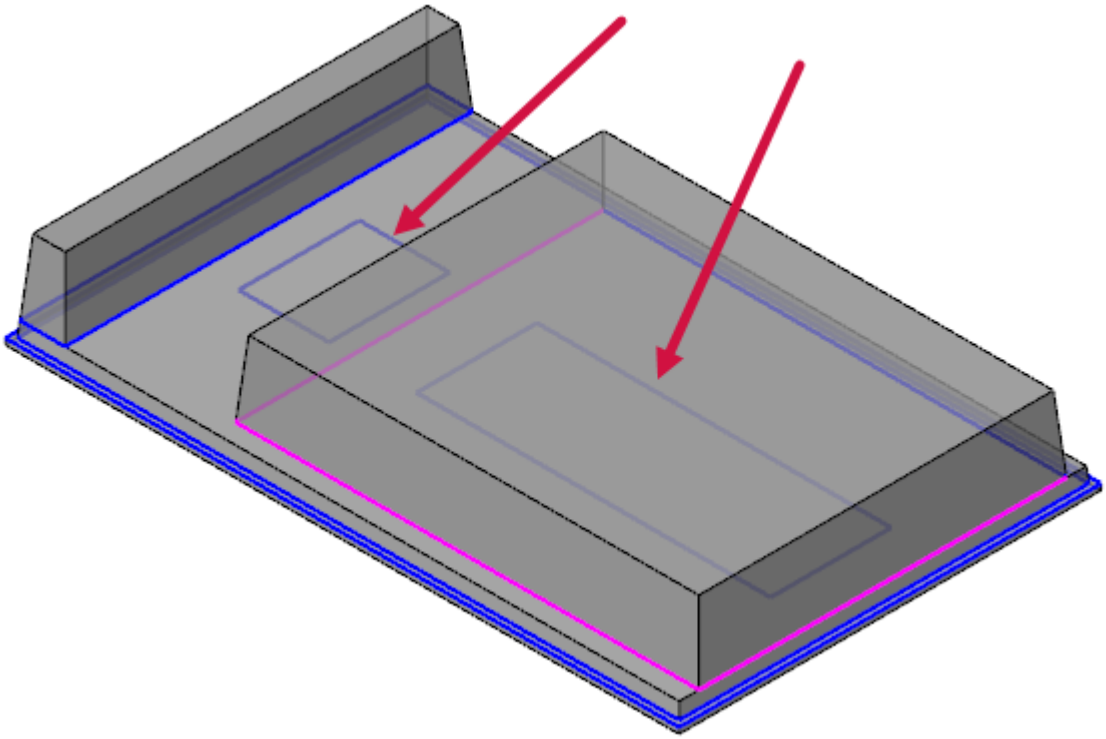
1. In the Levels Manager, turn on level 4.



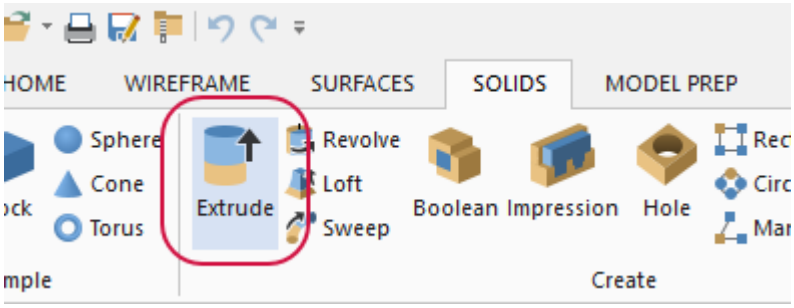
2. Select **Translucency** on the **View** tab.



3. The part becomes partially transparent, allowing you to see the two rectangles from level 4 that are hidden under existing geometry.

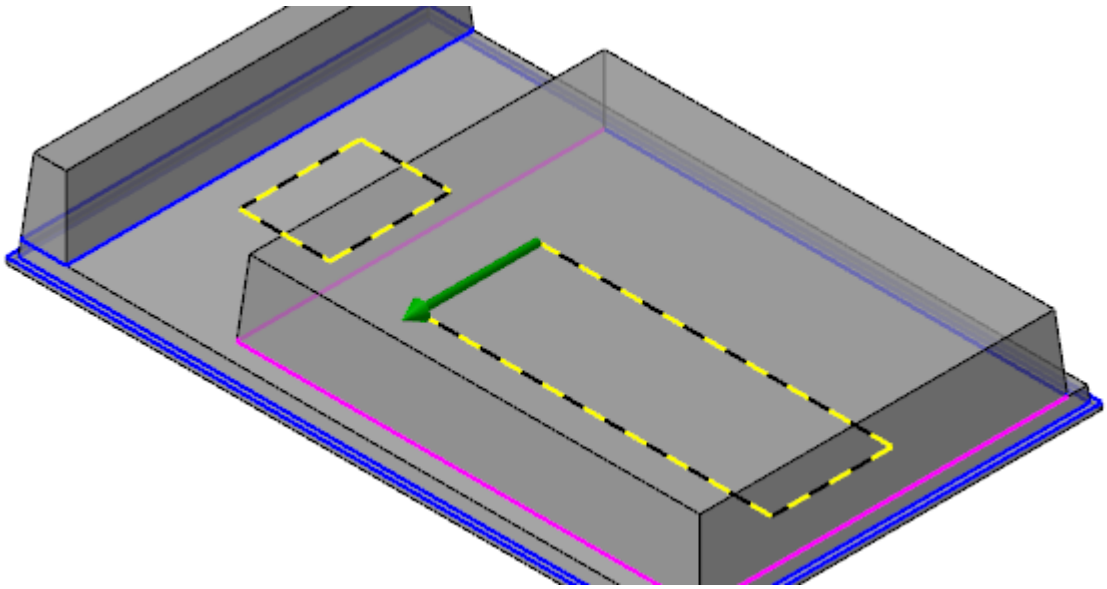


4. Select **Extrude** on the **Solids** tab.

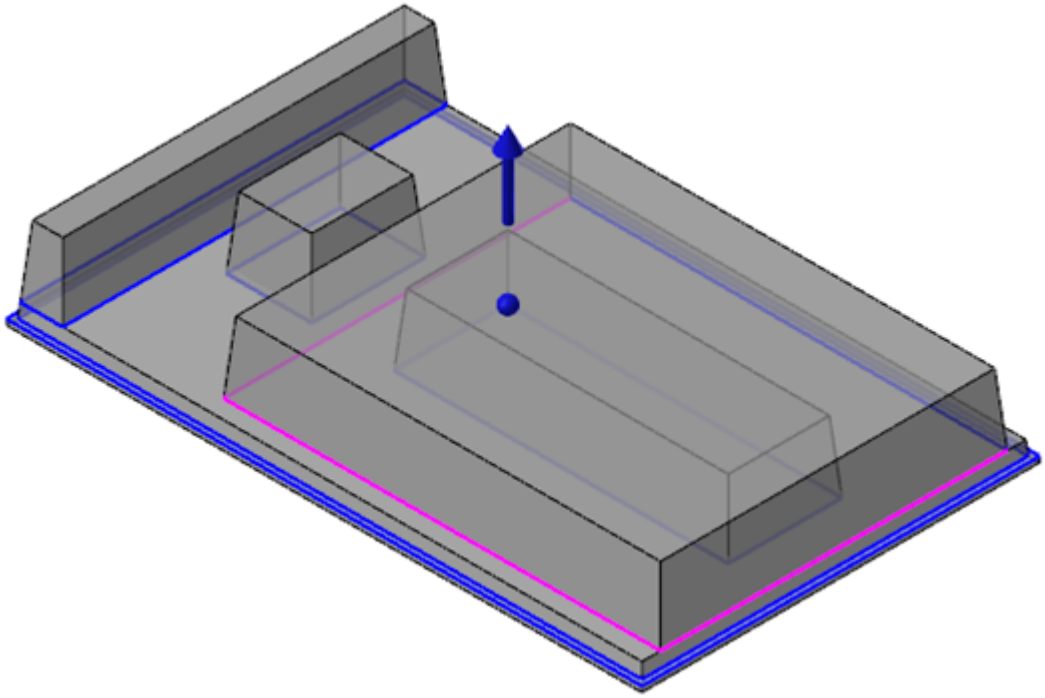


The **Chaining** dialog box opens.

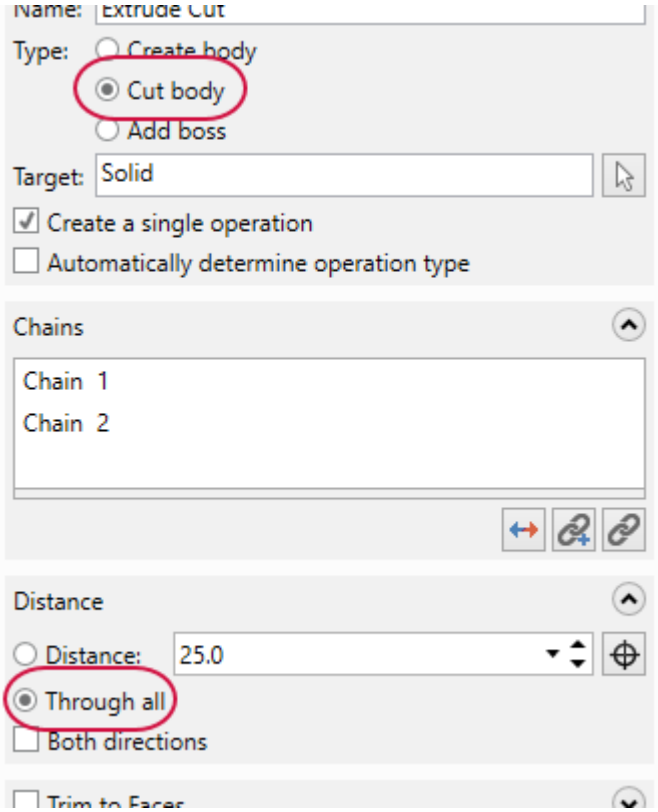
5. Chain the two new rectangles and click **OK** in the Chaining dialog box.



6. The two chained rectangles are extruded, as shown in the picture below.

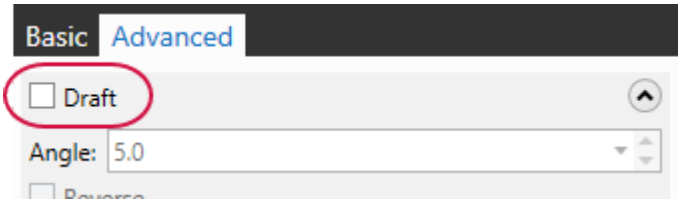


7. In the **Solid Extrude** function panel, change the operation to **Cut body**, and select **Through all**.

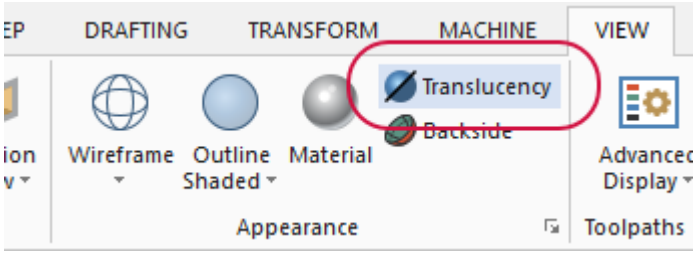


The **Cut body** option is the opposite of **Add boss**, removing material from the part rather than adding it.

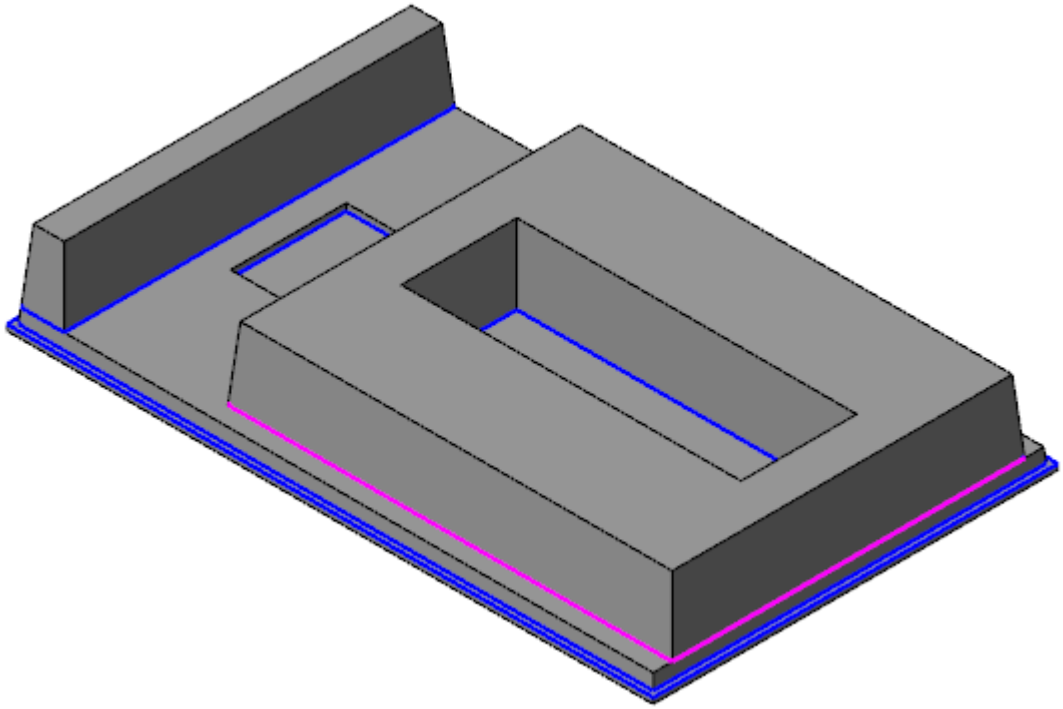
8. On the **Advanced** tab, turn off **Draft**.



9. Turn off **Translucency**.



10. Click **OK** to create the operation.
11. Now you can see the effect of the two new cuts more easily.

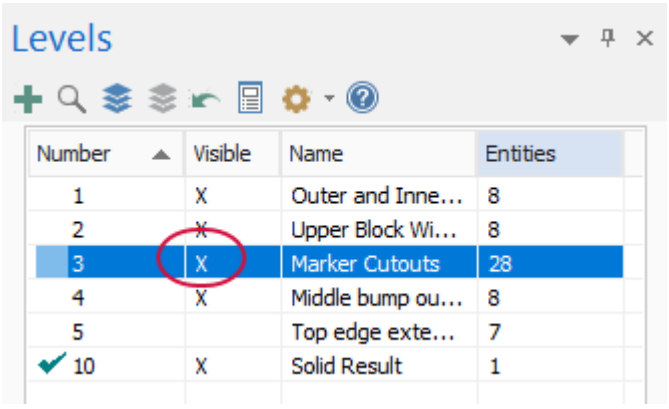


12. Save the file as `MarkerTray03-XXX`.

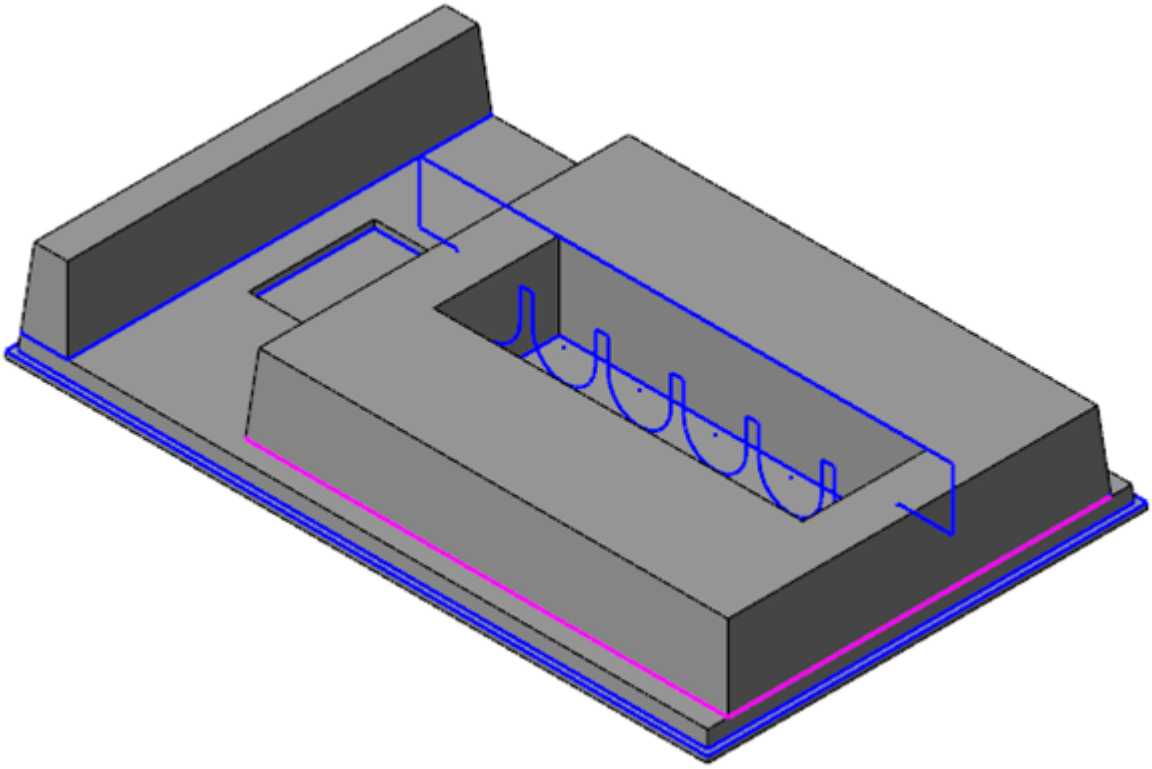
Exercise 4: Creating the Marker Cutouts

Now you create the cutouts for each individual marker. For this exercise, continue with the part you just saved, or load `MarkerTray03`, which accompanies this tutorial.

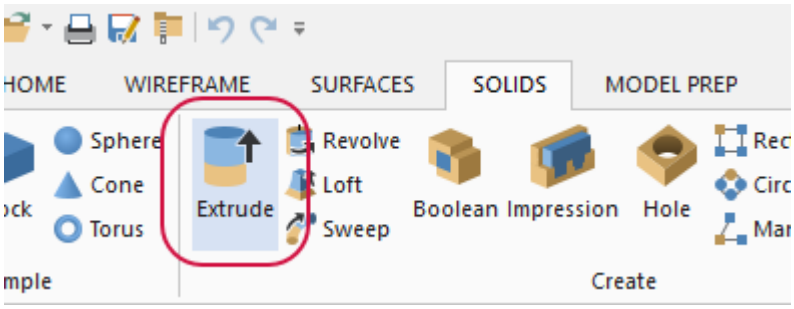
1. In the Levels Manager, turn on level 3.



This geometry represents the shape of the marker cutouts.

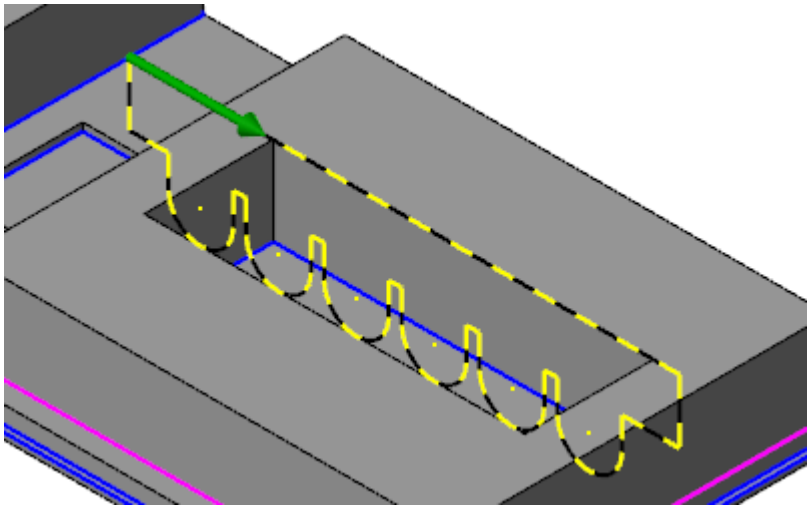


2. Select **Extrude** on the **Solids** tab.



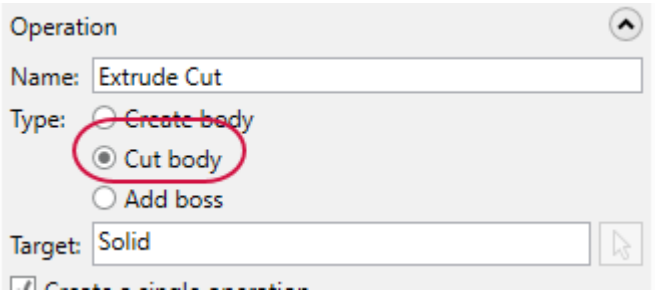
The **Chaining** dialog box opens.

3. Chain the new geometry, and click **OK** in the **Chaining** dialog box.

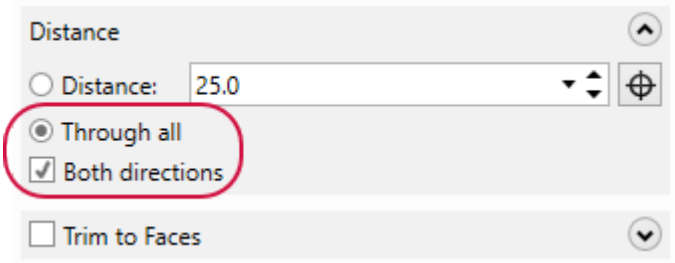


The **Solid Extrude** function panel opens.

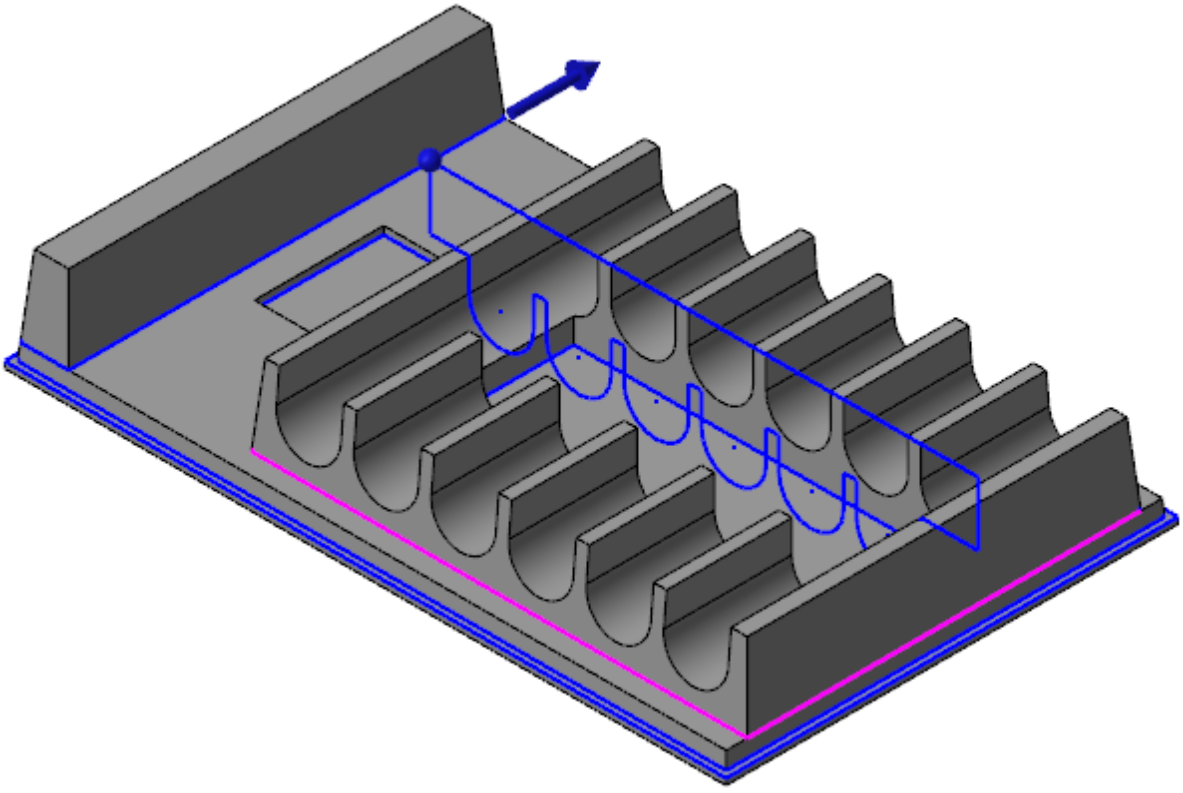
4. Select **Cut body**.



5. In the **Distance** group, select **Through all** and **Both directions**.



6. Extrude cuts the shape specified by the chained geometry through both sides of the boss, giving the results shown below.

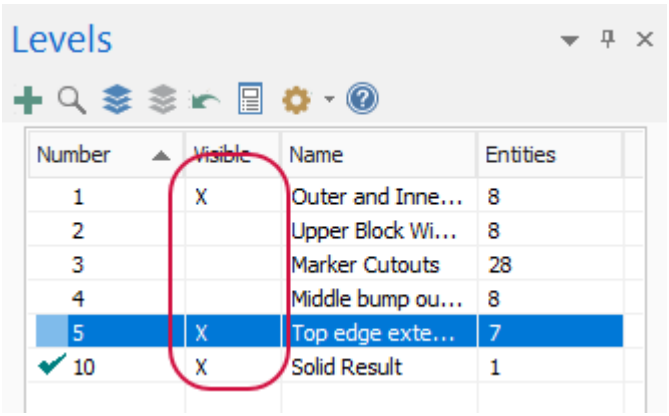


7. Select **OK** to create the operation.
8. Save the file as `MarkerTray04-XXX`.

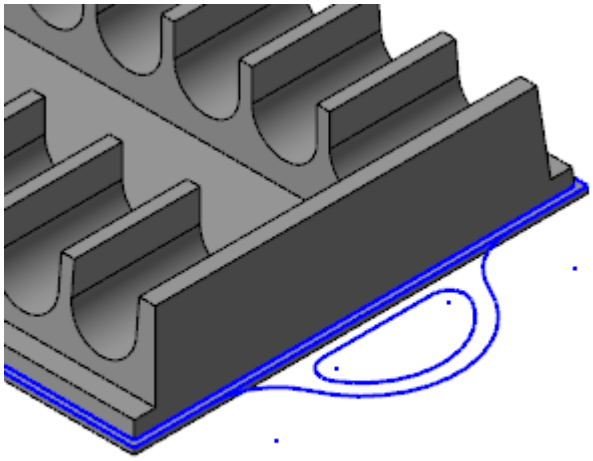
Exercise 5: Adding the Handle

Because Mastercam Solids maintains a history tree of all operations used to create a solid model, you can easily edit the model to correct errors or add new features. To demonstrate this, you now add a handle to the marker tray using the solids tree. For this exercise, continue with the part you just saved, or load `MarkerTray04`, which accompanies this tutorial.

1. In the Levels Manager, turn on level 5, and turn off levels 2, 3, and 4.



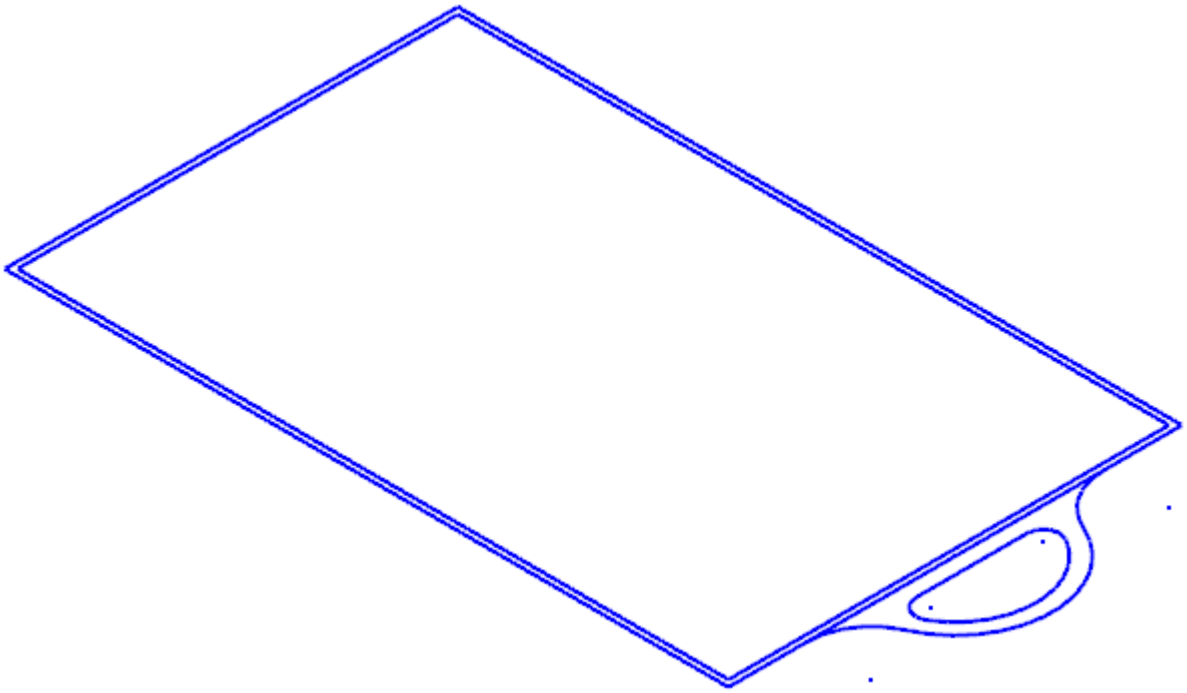
2. Mastercam displays the geometry that you need to create the handle.



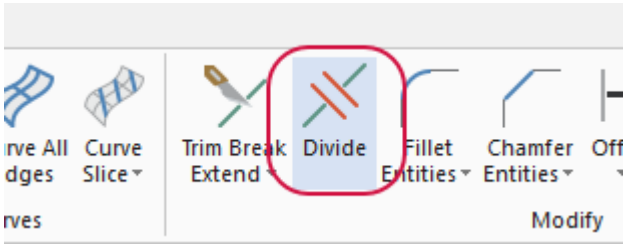
3. Select the left-hand portion of the **Wireframe Quick Mask** button, found along the right side of the graphics window.

Mastercam selects all wireframe geometry.

4. Press [**Alt+E**] to hide all geometry except the selected wireframe.



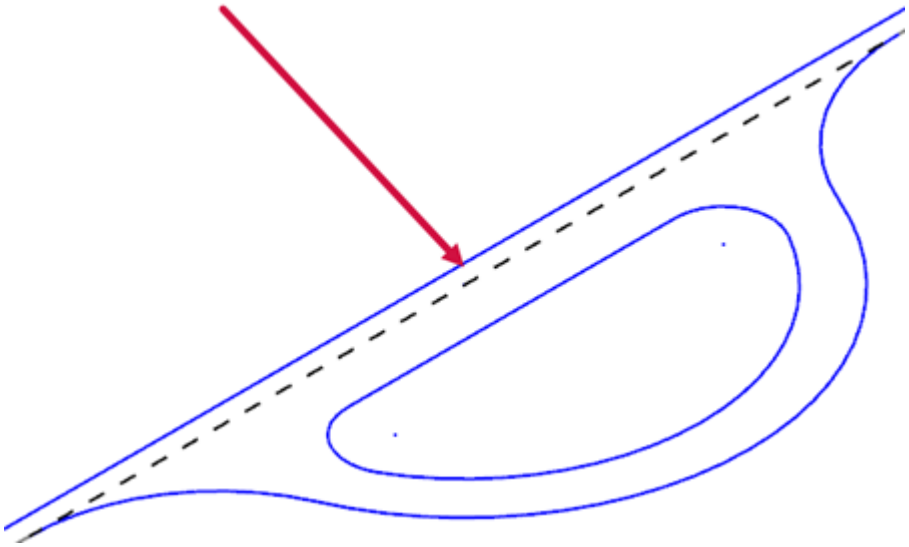
5. Select **Divide** on the **Wireframe** tab.



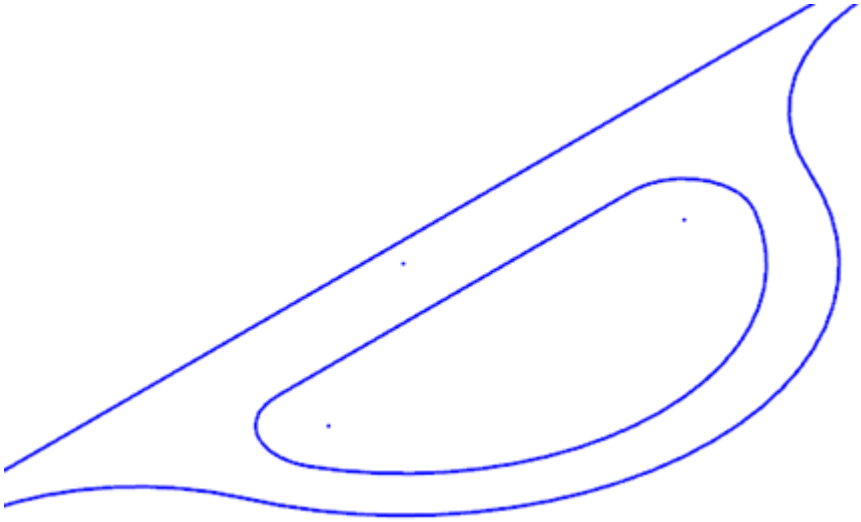
The **Divide** function panel opens.

6. Ensure that **Trim** is selected.

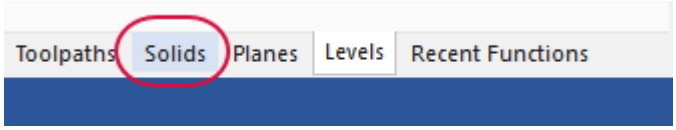
7. Click the line indicated in the picture below.



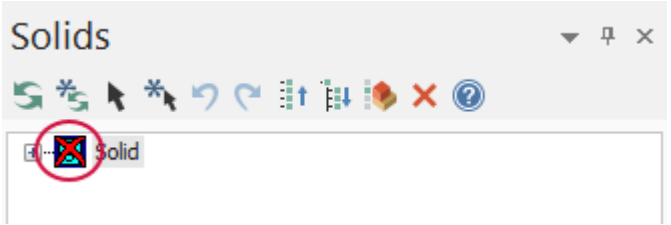
8. Mastercam trims the line, leaving the handle merged with the main part of the tray.



9. Click **OK** in the **Divide** function panel.
10. Select the **Solids** tab to open the Solids Manager.

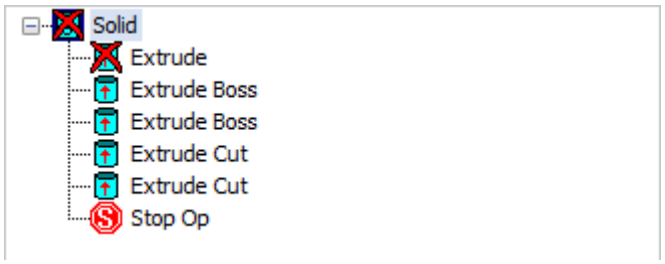


11. Click the plus sign (+) next to the item in the tree.



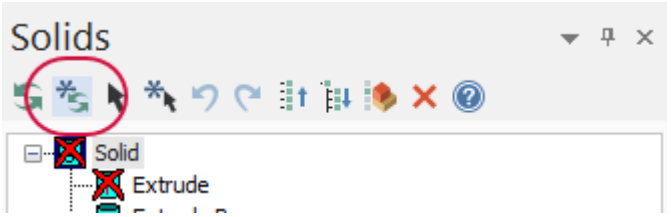
Mastercam opens the list to show the operations you created to generate the solid.

12. Notice that the first extrude operation is marked dirty.

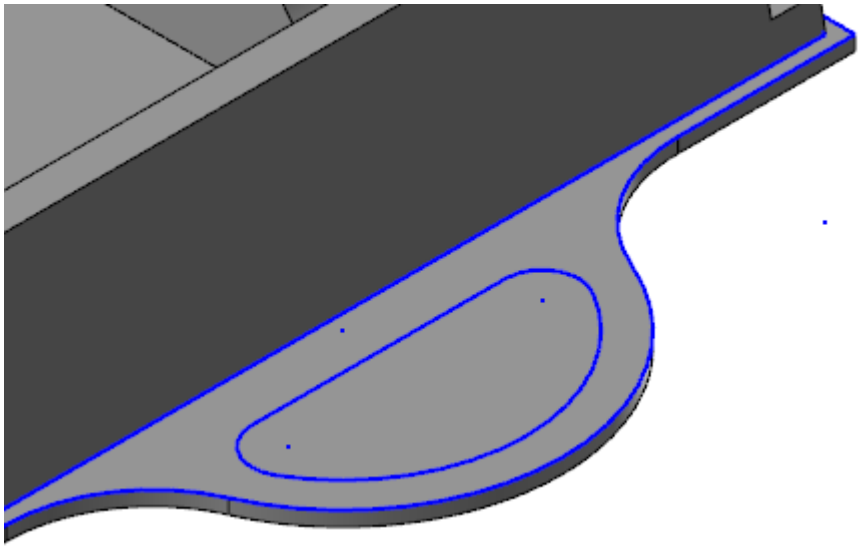


By adding the handle, you changed the geometry that makes up the tray's base, which means that the extrude operation must be regenerated.

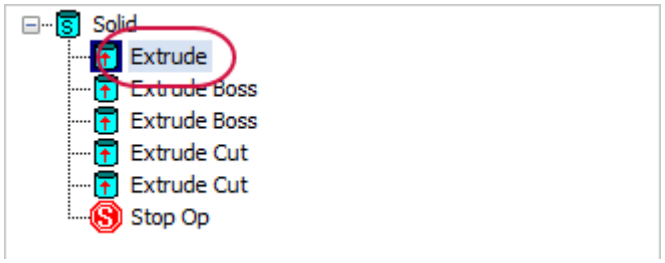
13. Press [**Alt+E**] to show the hidden entities.
14. Select **Regen** to regenerate the solid operation.



15. The handle is now a full part of the tray's base.

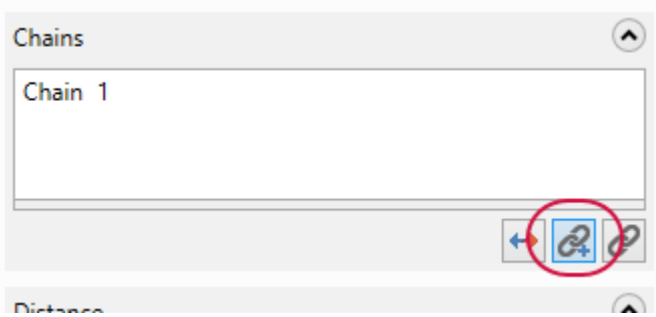


16. Double-click **Extrude** in the Solids Manager.



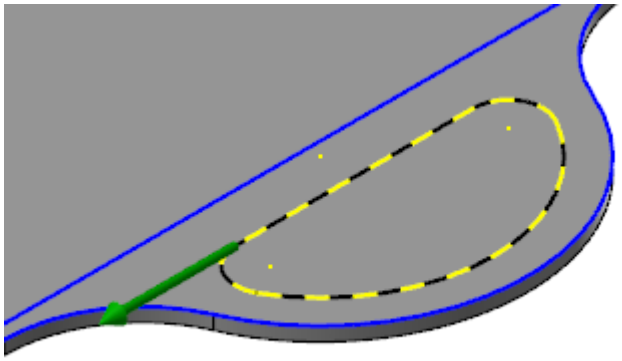
The **Solid Extrude** function panel opens, so you can make changes to the operation.

17. Select **Add Chains**.

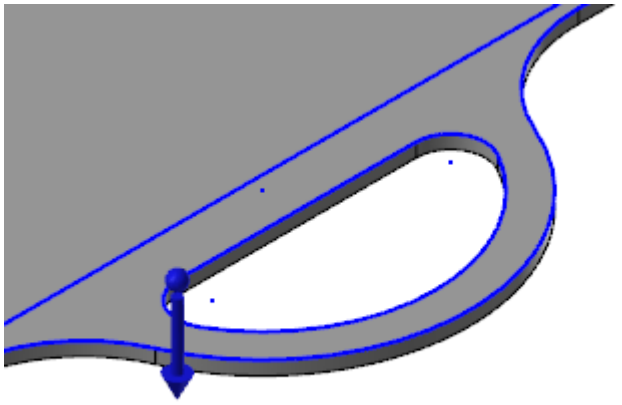


The **Chaining** dialog box opens.

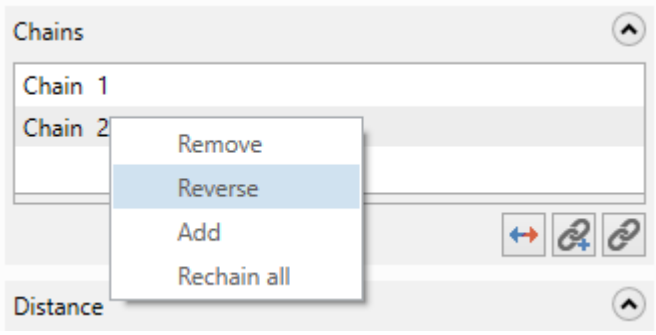
18. Chain the inside of the handle, and click **OK**.



19. The chain is then added to the operation, cutting the handle opening.



20. If the chaining arrow is not pointing down, right-click **Chain 2**, and click **Reverse**.



21. Click **OK and Regenerate** to both finalize and regenerate the operation.

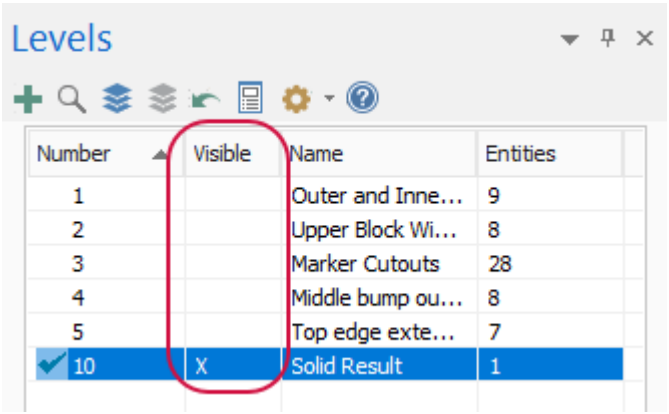
OK and Regenerate saves you from going to the Solids Manager to regenerate.

22. Save the file as `MarkerTray05-XXX`.

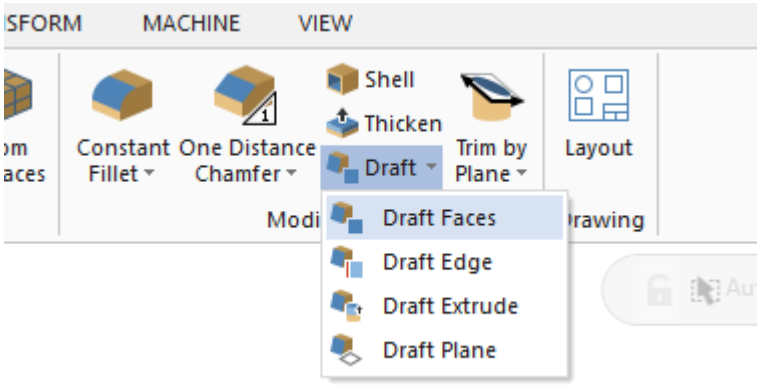
Exercise 6: Adding Draft and Fillets

In this exercise, you add draft to a face and fillets to some edges. Continue with the part you just saved, or load `MarkerTray05`, which accompanies this tutorial.

1. In Levels Manager, turn off all levels but 10.



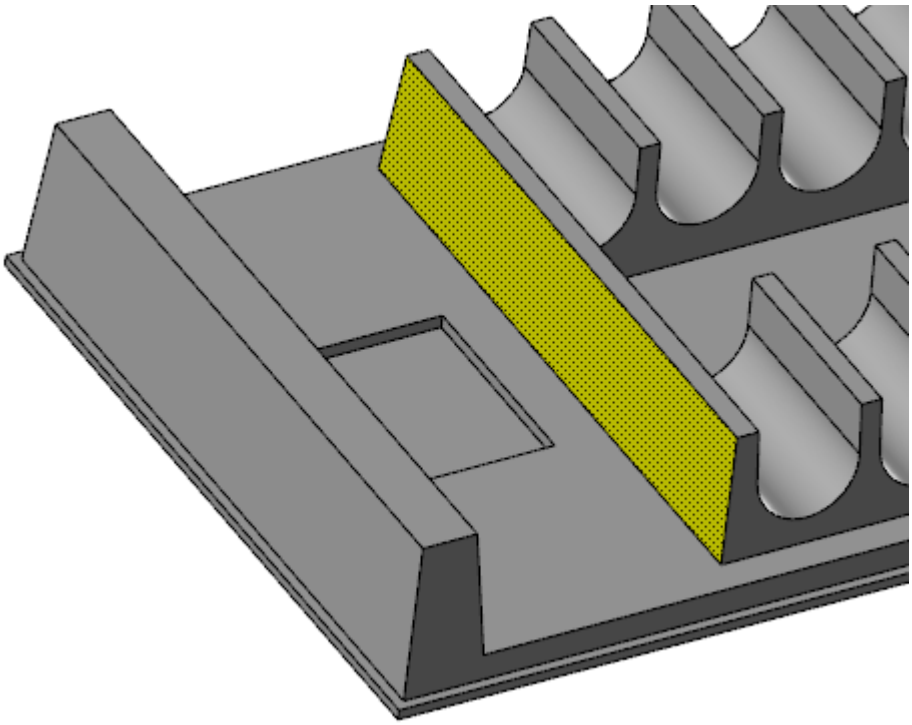
2. Select **Draft**, **Draft Faces** on the **Solids** tab.



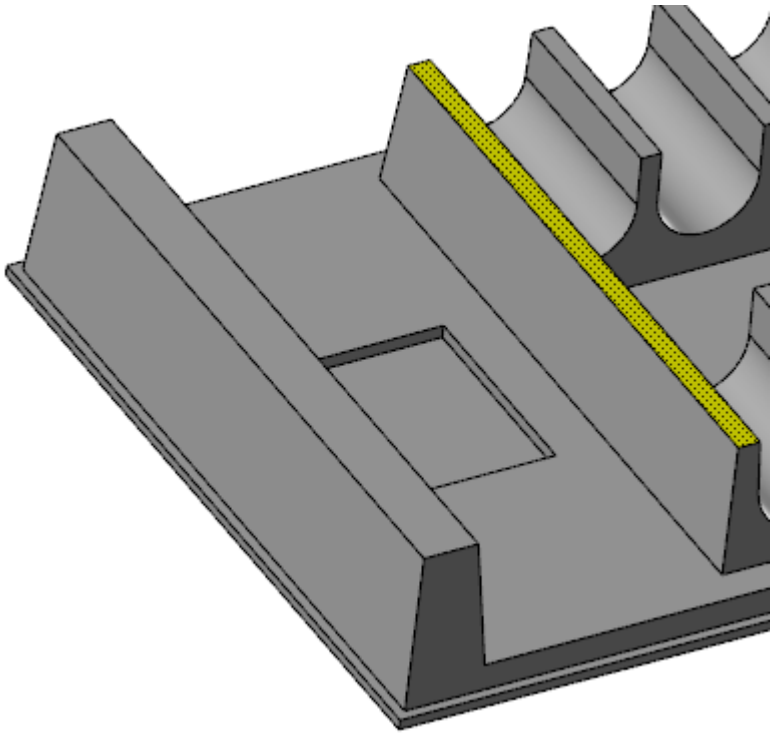
The **Solid Selection** dialog box opens.

3. Ensure that the **Face** filter is selected.

4. Select the face indicated in the picture below, and click **OK** in the **Solid Selection** dialog box.

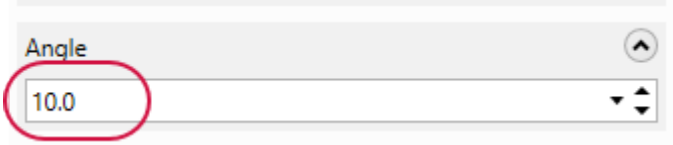


5. Select the top face of the same wall (as indicated in the picture below) for the reference face. The draft is created relative to the reference face.



The **Draft to Face** function panel opens.

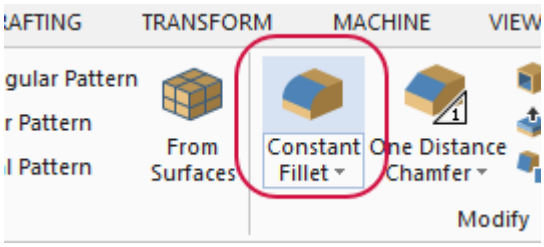
6. Set the **Angle** to **10**.



7. Click **OK** in the function panel to create the draft.

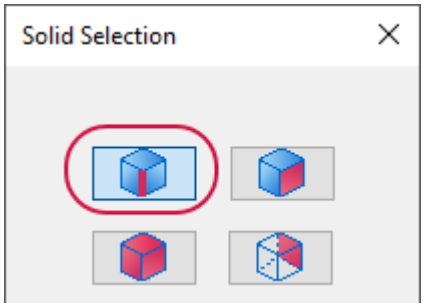
Next, you will create the outside fillets.

1. Select **Constant Fillet** on the **Solids** tab.

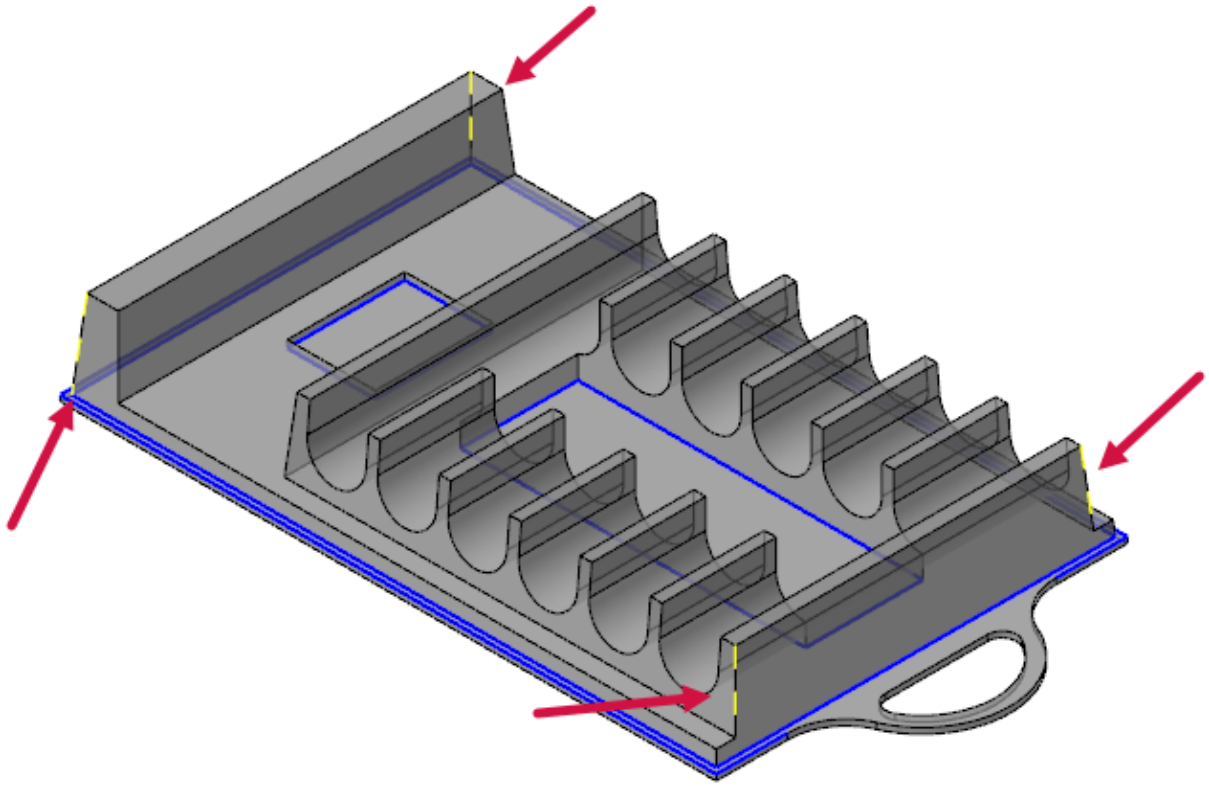


   The **Solid Selection** dialog box opens.

2. Turn off all selection filters except **Edge**.



3. Select all four of the tray's left outside vertical edges.



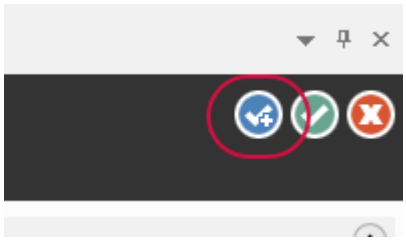
   Note: **Translucency** has been turned on for the picture above in order to show you the edges to be selected.

4. Click **OK** in the **Solid Selection** dialog box.

   The **Constant Radius Fillet** function panel opens.

5. Set **Radius** to **5**.

6. Select **OK and Create New Operation**.

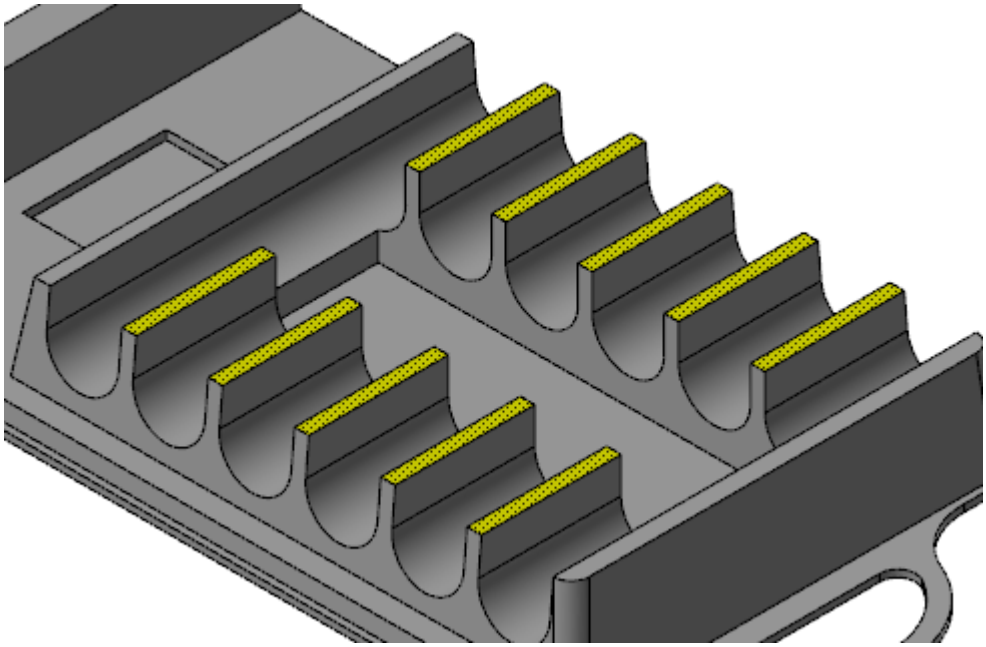


   The function panel closes, and the **Solid Selection** dialog box opens.

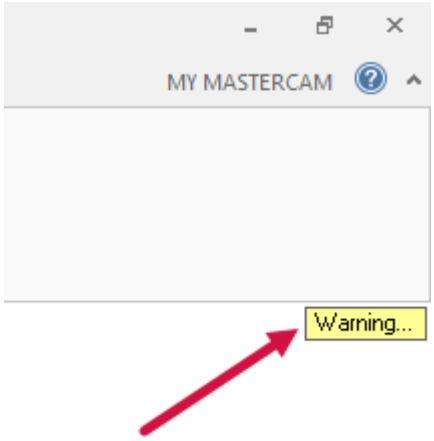
You will now add the top fillets.

1. Turn the **Face** filter back on and turn off **Edge** selection.

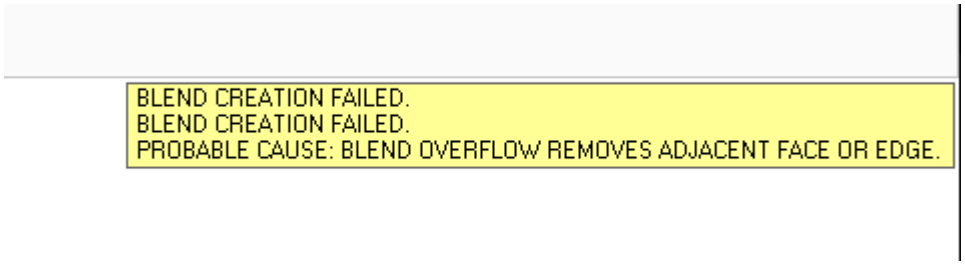
2. Select the ten faces shown below, and click **OK** in the **Solid Selection** dialog box.



3. Notice the yellow warning that appears in the upper-right corner of the graphics area. This indicates that there is a problem with your current settings.

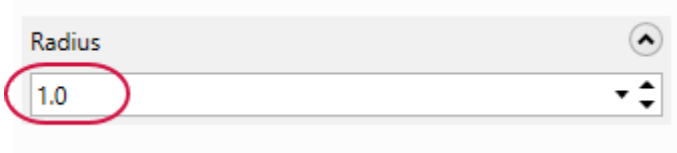


4. Click the warning to open the entire message.

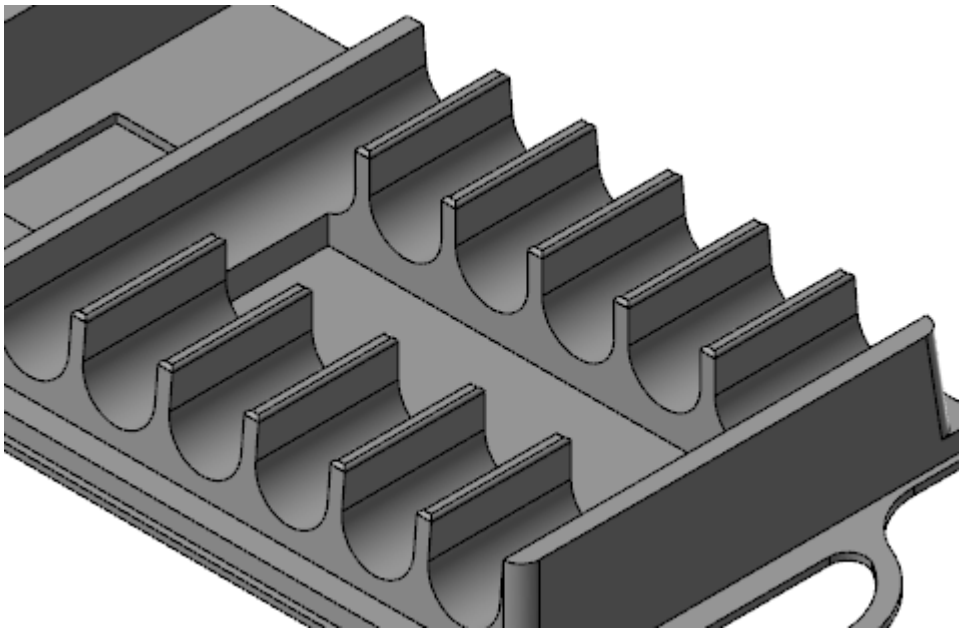


This warning displayed because the fillet's radius is too large.

5. Set the **Radius** to **1**.



6. The warning no longer displays and you get the result show in the following picture.



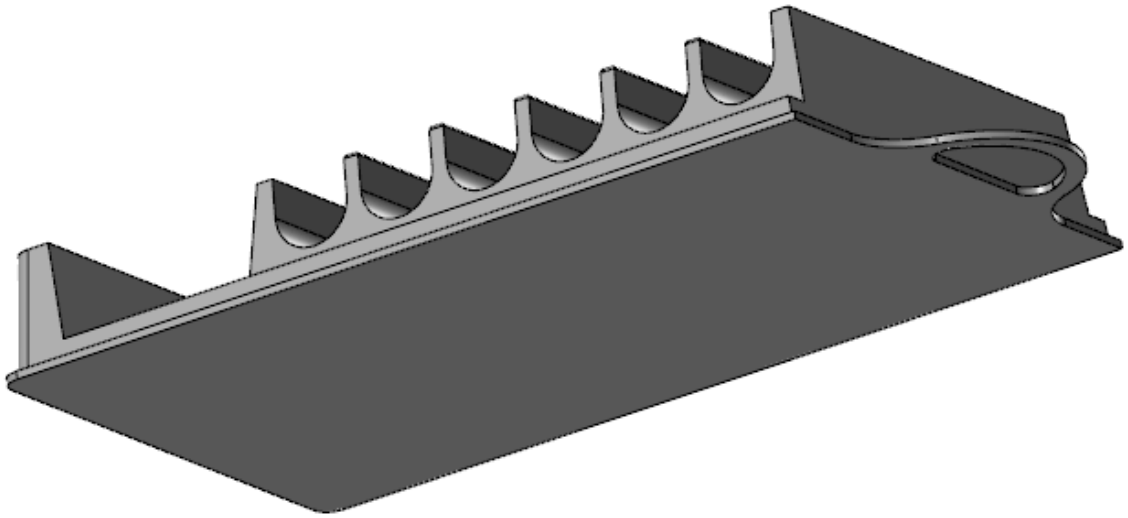
7. Click **OK** to create the fillets.
8. Save the file as `MarkerTray06-XXX`.

Note: The marker tray part has many other edges that should be filleted. However, to avoid a lot of repetition, this tutorial leaves those steps out.

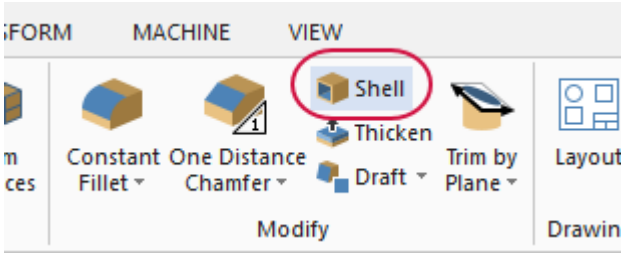
Exercise 7: Shelling the Part

To complete the part, you now use **Shell** to remove extra interior material. For this exercise, continue with the part you just saved, or load `MarkerTray06`, which accompanies this tutorial.

1. Rotate the view so you can see the bottom of the part.

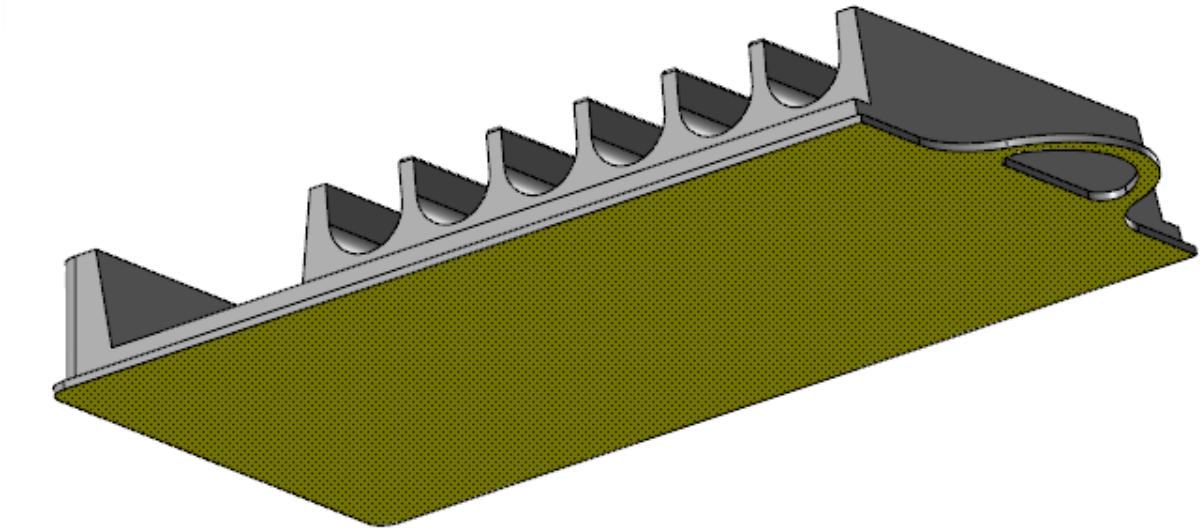


2. Select **Shell** on the **Solids** tab.



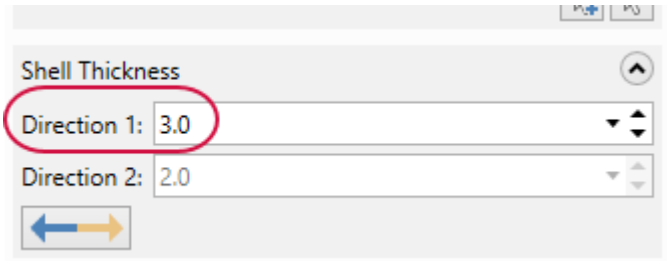
The **Solid Selection** dialog box opens.

3. Select the part's bottom face, and click **OK** in the **Solid Selection** dialog box.

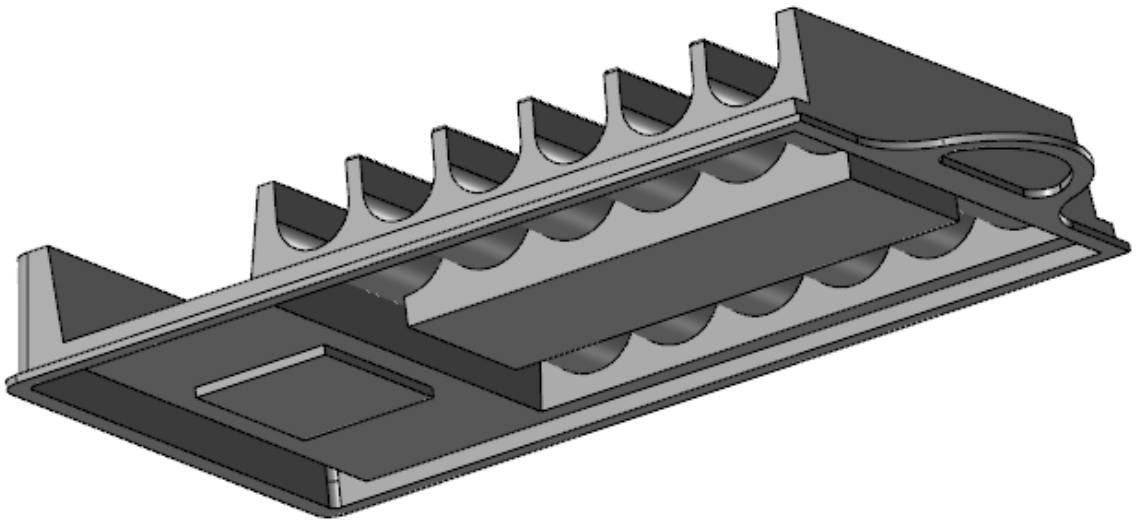


If you have trouble selecting the part's bottom face, turn off all filters in the **Solid Selection** dialog box except for **Face**.

4. Set the **Shell Thickness** for **Direction 1** to **3**.



5. Click **OK** to create the operation.
6. The part should now look like the picture below.



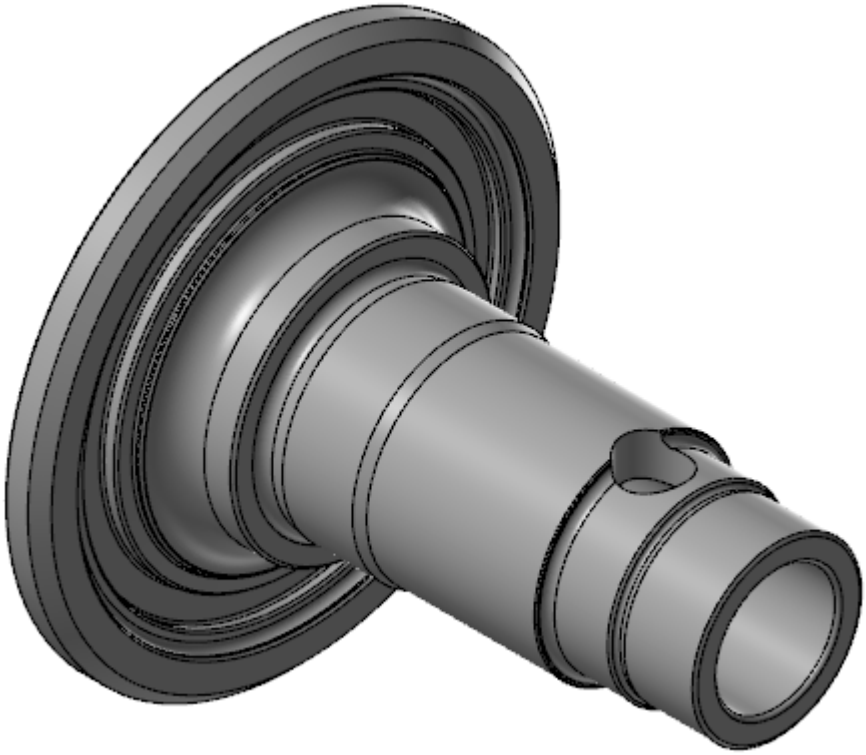
7. Save the file as `MarkerTray07-XXX`.

This completes the `MarkerTray` part.

CHAPTER 2

USING THE REVOLVE, SWEEP, AND BOOLEAN FUNCTIONS

In this chapter, you will be using the Revolve, Sweep, and Boolean functions to create a spindle. These functions also have a dynamic preview so you can see the results as soon as you edit parameters.

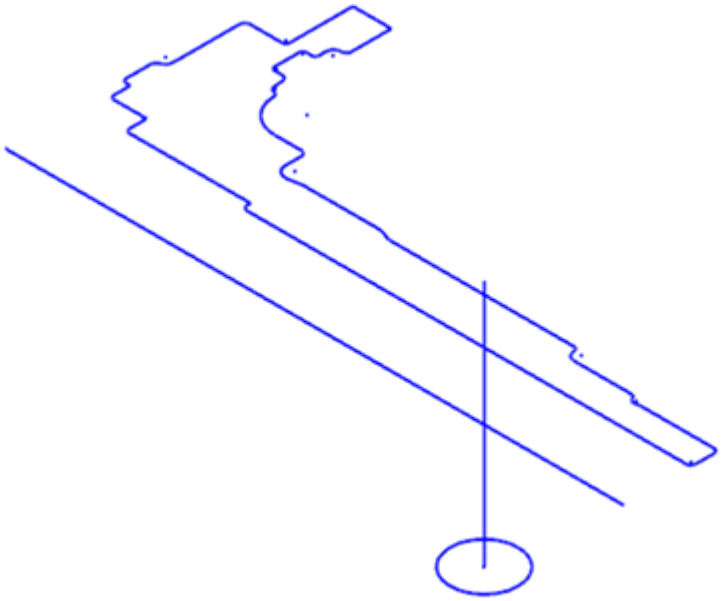


Goals

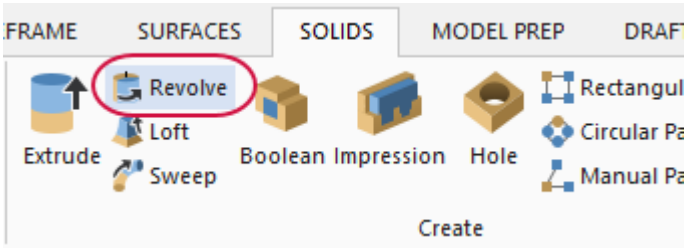
- Learn to use the Revolve, Sweep, and Boolean functions.
- Create a spindle from wireframe.

Exercise 1: Revolving the Spindle Wireframe

1. Open the file `SpindlePartial`, which is included with this tutorial.

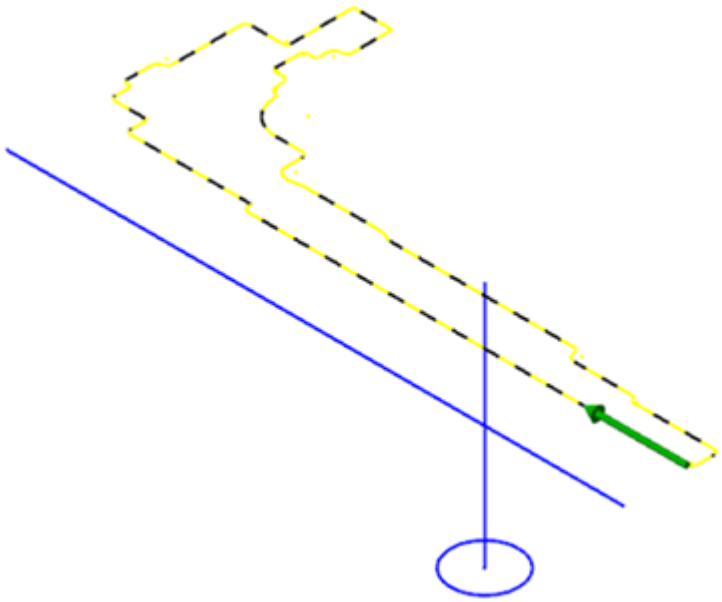


2. So you do not overwrite the original, save the file as `SpindlePartial-XXX`, where *XXX* is your initials.
3. Select **Revolve** on the **Solids** tab.

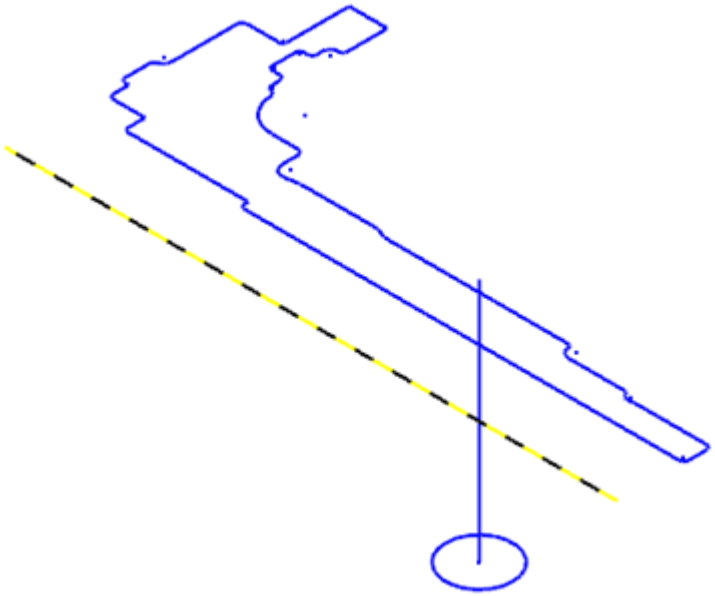


The **Chaining** dialog box opens.

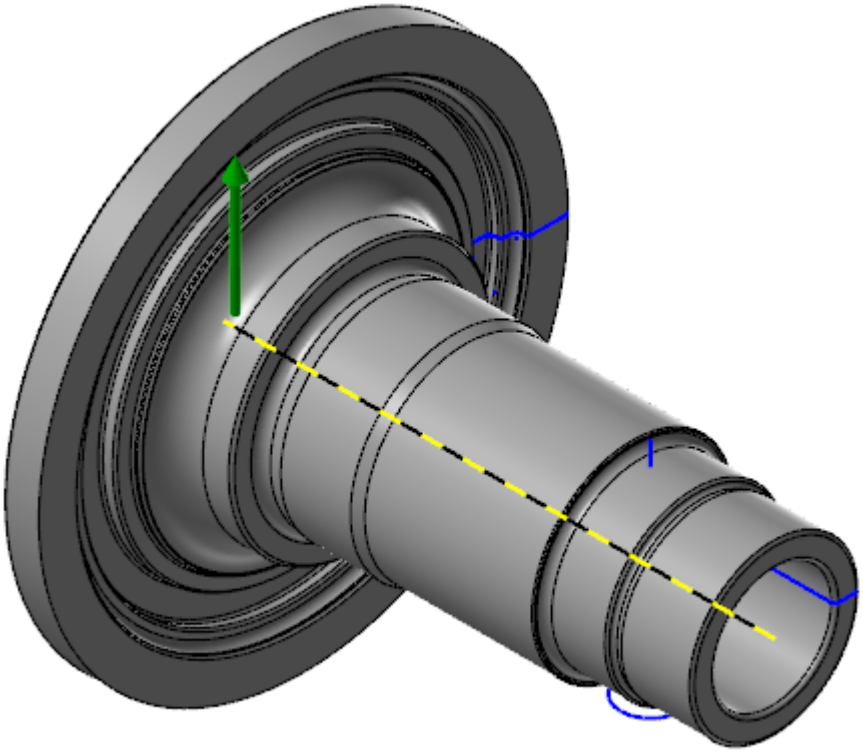
4. Chain the geometry as shown below, and click **OK** in the **Chaining** dialog box. This is the chain that is going to be revolved.



5. Select the line indicated below. This line is your axis of rotation.



6. The first selected chain is the revolved around the axis of rotation chain.

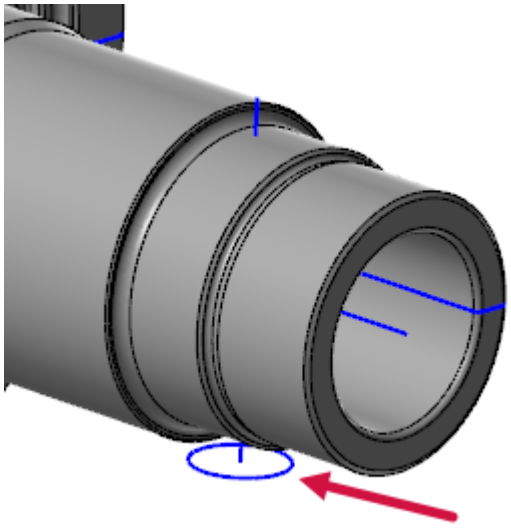


7. Click **OK** to create the operation. There does not need to be any parameter changes.
8. Save the file as `SpindlePartial01-XXX`, *XXX* are your initials.

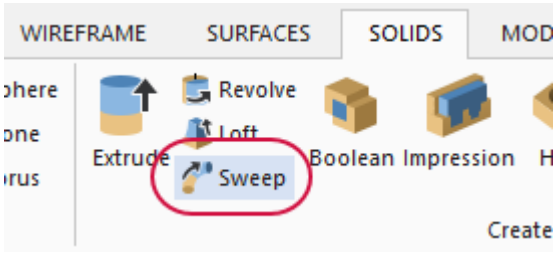
Exercise 2: Sweeping Wireframe

Next, you will create a Sweep operation to begin creating a hole in the spindle. For this exercise, continue with the part you created in the previous exercise, or load `SpindlePartial01`, which accompanies this tutorial.

1. Rotate the view so you can see the circle indicated.

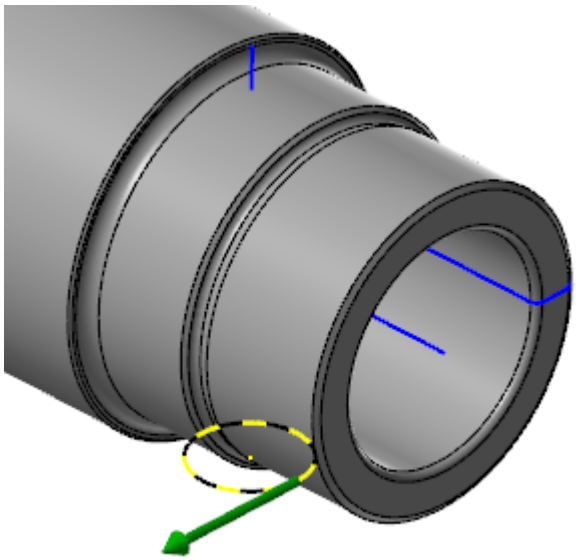


2. Select **Sweep** from the **Solids** tab.

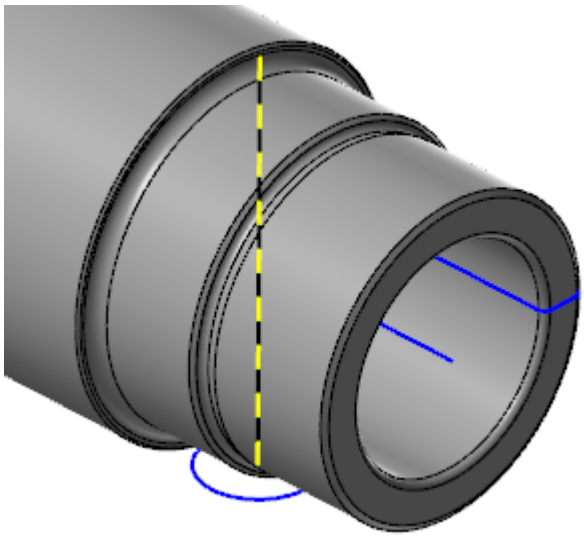


The **Chaining** dialog opens.

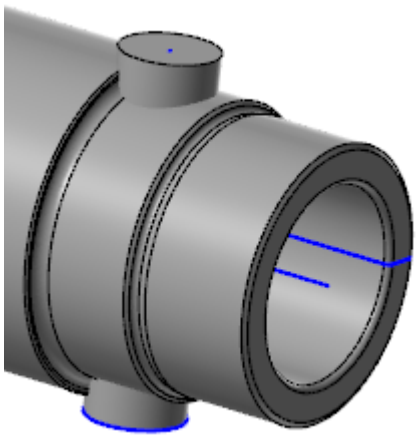
3. Chain the circle (chaining direction does not matter), and click **OK** in the **Chaining** dialog box. This is the chain to be sweeped.



4. Chain the line indicated in the picture below. This line is the sweep's along curve.



5. Mastercam sweeps the circle along the line forming the new solid shown in the picture below.

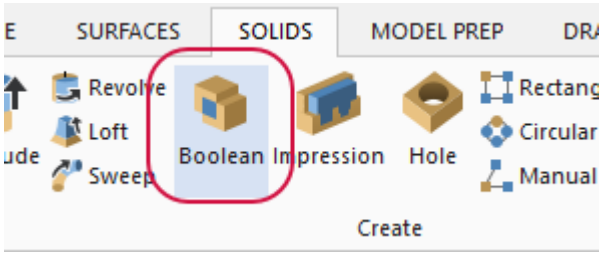


6. Click **OK** to finalize the operation.
7. Save the file as `SpindlePartial02-XXX`.

Exercise 3: Creating a Hole

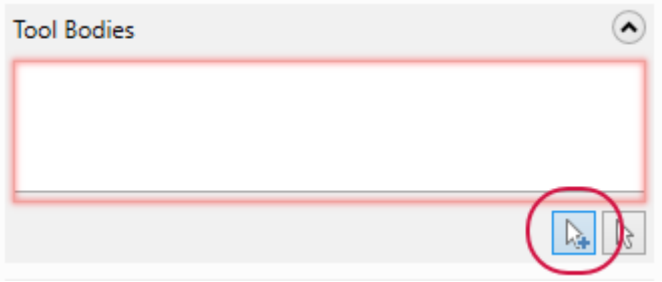
Next, you will create a hole using a Boolean operation. For this exercise, continue with the part you created in the previous exercise, or load `SpindlePartial02`, which accompanies this tutorial.

1. Select **Boolean** from the **Solids** tab.



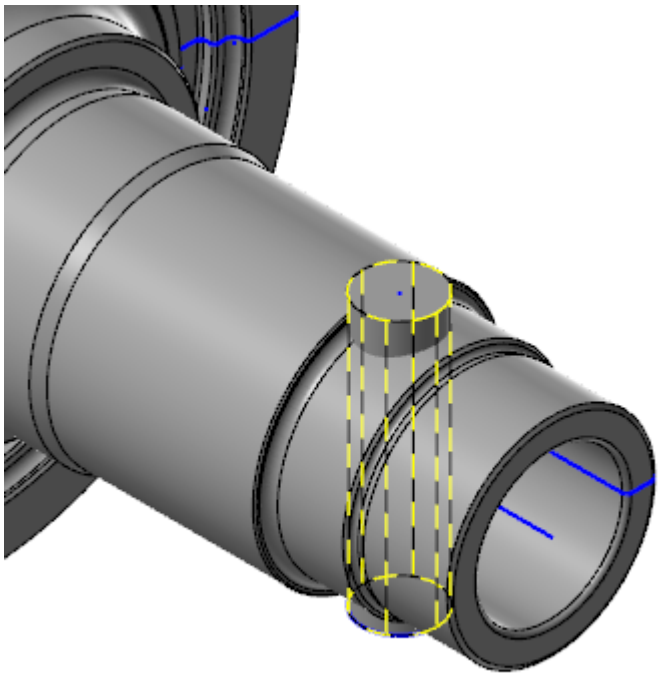
The **Boolean** function panel opens.

2. Click **Add Selection** under **Tool Bodies** to select the tool body.

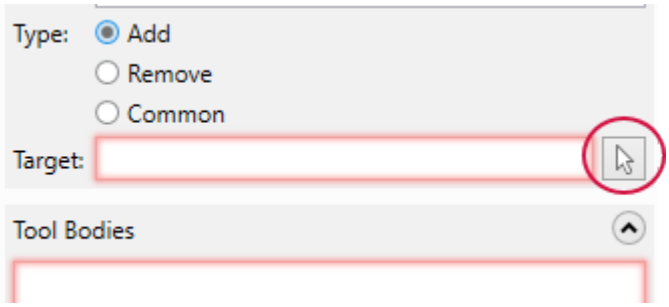


The **Solid Selection** dialog box opens.

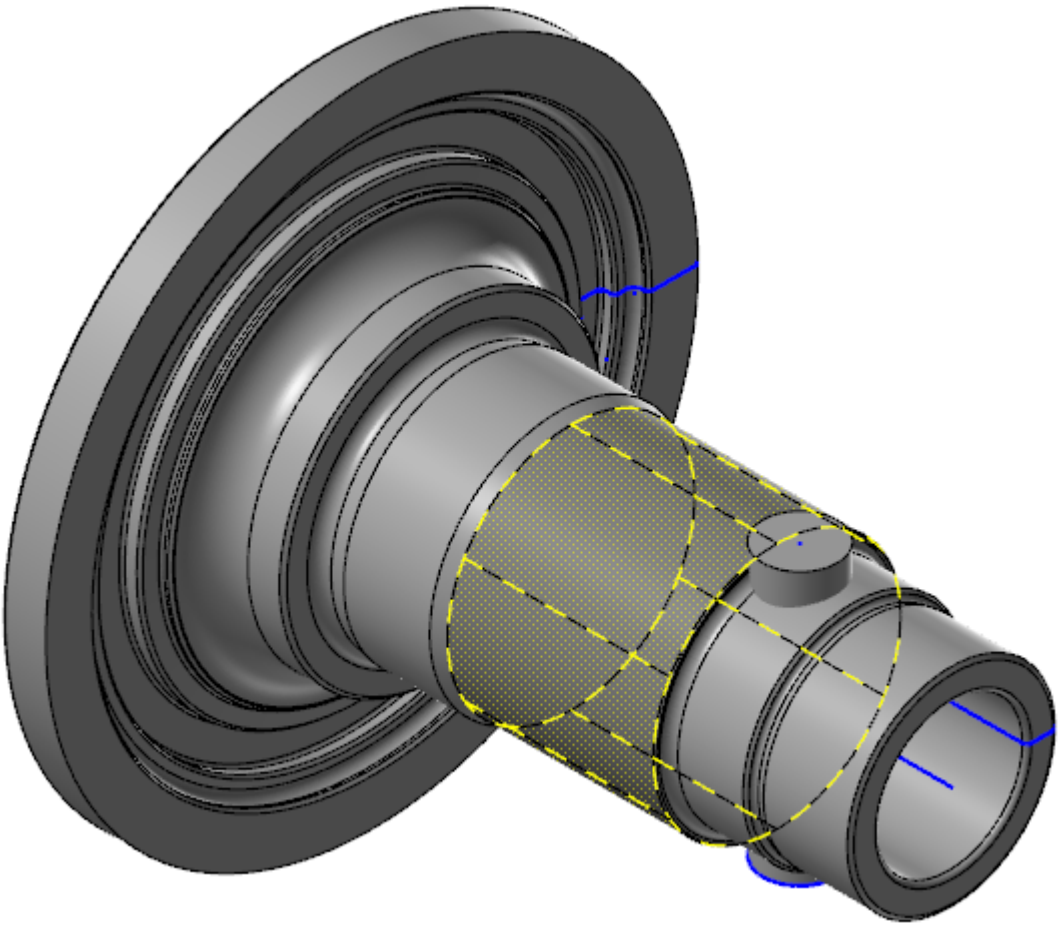
3. Turn off all filters except **Body**.
4. Select the sweeped solid.



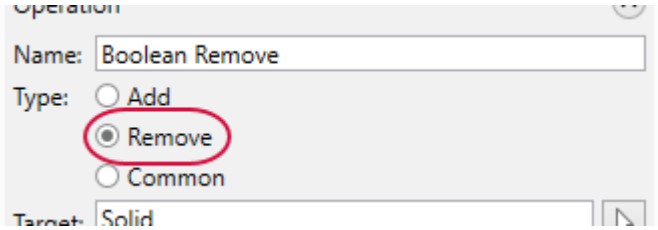
5. Click **OK** in the **Solid Selection** dialog box.
6. Click **Select Target Body** to select the main body of the part.



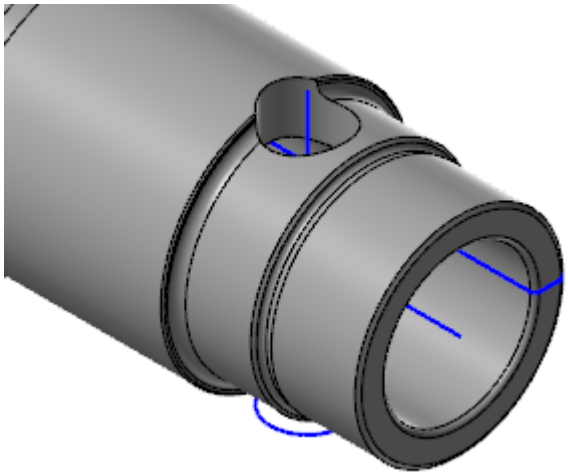
7. Select the main body of the part.



8. Change the **Type** to **Remove**.



9. Boolean removes all material from the selected area.



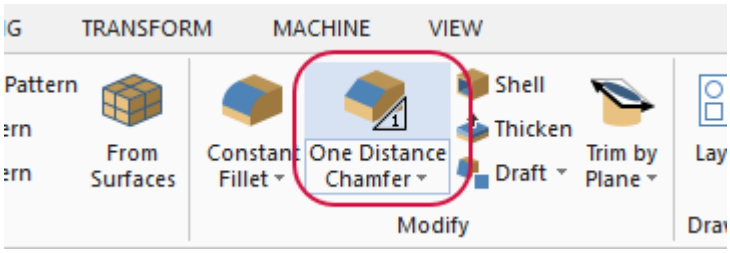
10. Click **OK** to finalize the operation.

11. Save the file as `SpindlePartial03-XXX`.

Exercise 4: Adding Chamfers

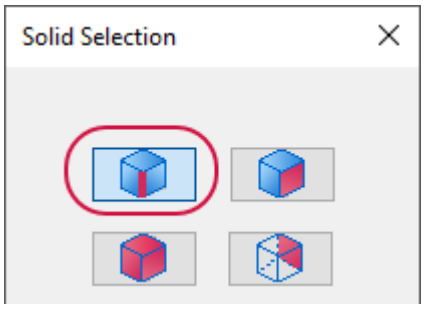
Next, you will create a One Distance Chamfer operation to add chamfers. For this exercise, continue with the part you created in the previous exercise, or load `SpindlePartial03`, which accompanies this tutorial.

1. Select **One Distance Chamfer** from the **Solids** tab.

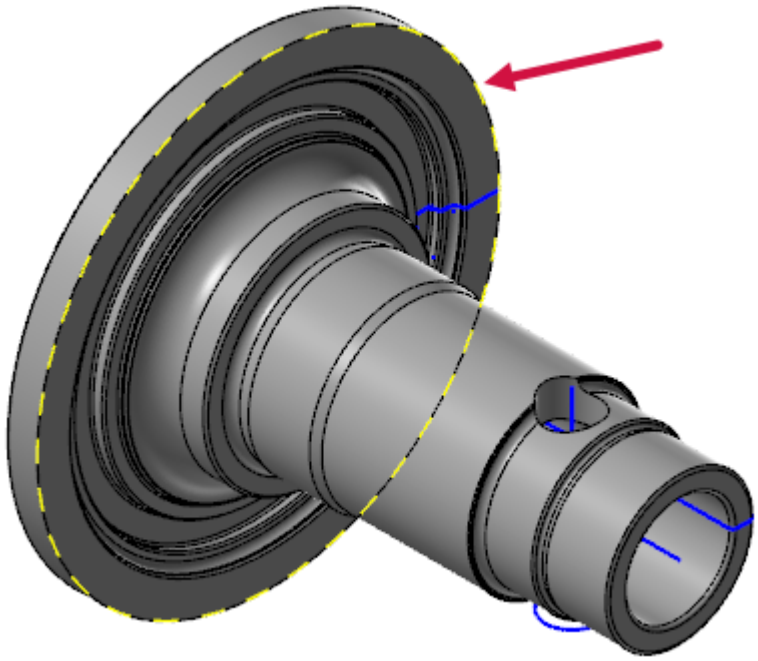


   The Solid Selection dialog box opens.

2. Turn off all filters except **Edge**.



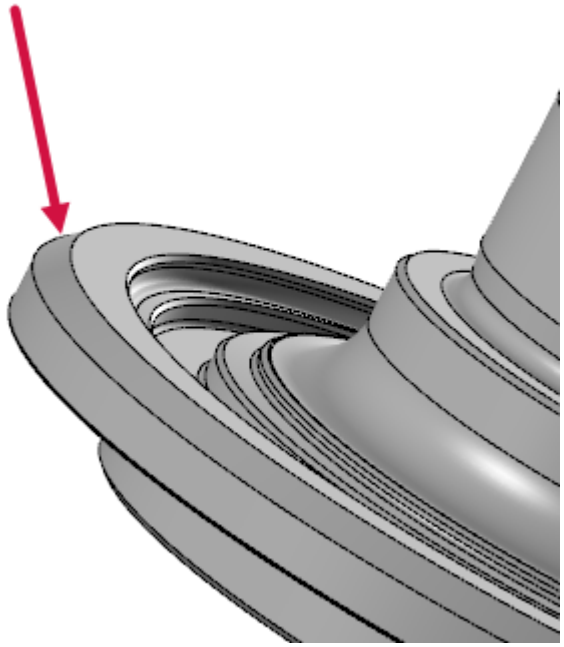
3. Select the edge indicated in the picture below, and then click **OK** in the **Solid Selection** dialog box.



   The **One Distance Chamfer** function panel opens.

4. Set **Distance** to **5**.

5. Click **OK** to finalize the operation.



6. Save the file as `SpindlePartial04-XXX`.

   You have now completed the spindle part.

CHAPTER 3

USING THE SOLID IMPRESSION FUNCTION

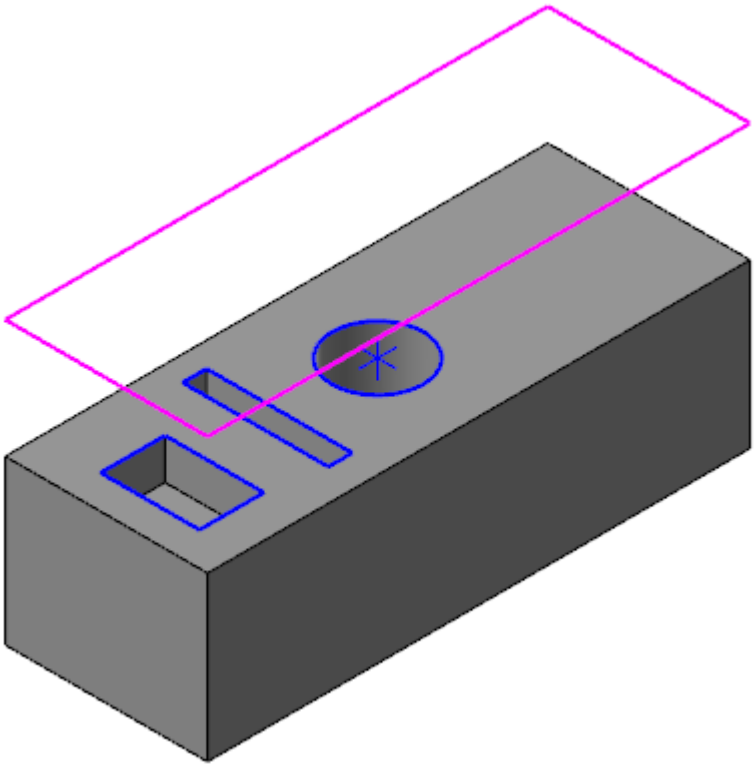
Mastercam's Solid Impression allows you to create a new solid body that is a negative impression of a selected closed body. In this chapter, you will create a solid impression by selecting a wireframe chain which then projects to the solid body. The depth of the projection is from the wireframe chain to the bottommost face of the selected solid or solid face.

Goals

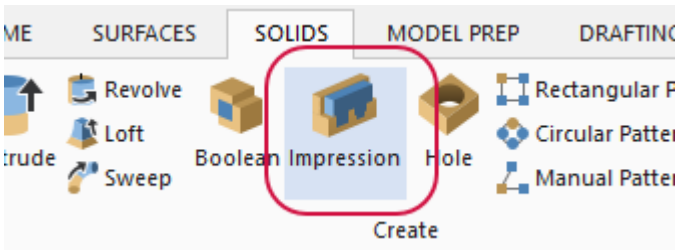
- Create an impression from a solid.

Exercise 1: Creating an Impression

1. Open the file `ImpressionDemo`, which is included with this tutorial.

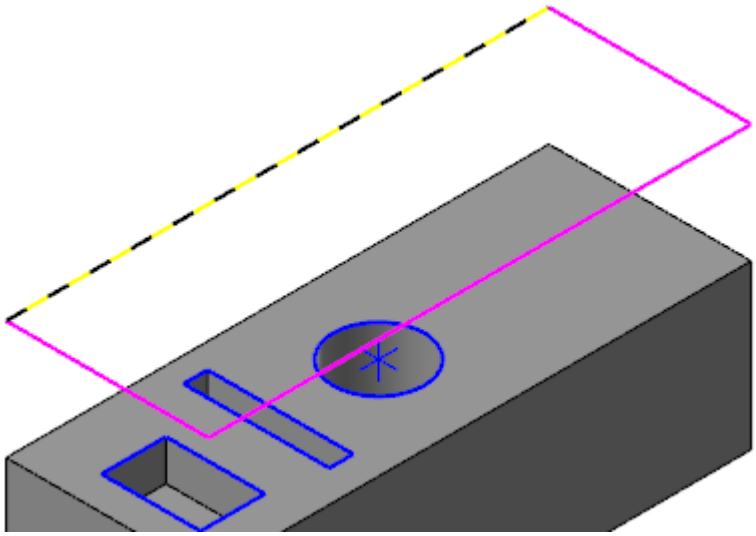


2. So you do not overwrite the original, save the file as `ImpressionDemo-XXX`, where *XXX* is your initials.
3. Select **Impression** on the **Solids** tab.



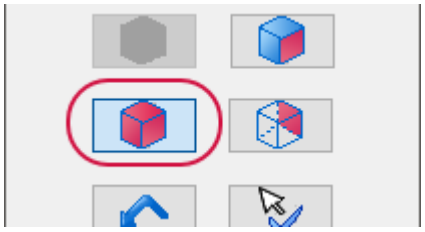
The **Chaining** dialog box opens.

4. Chain the wireframe rectangle that is positioned above the solid.

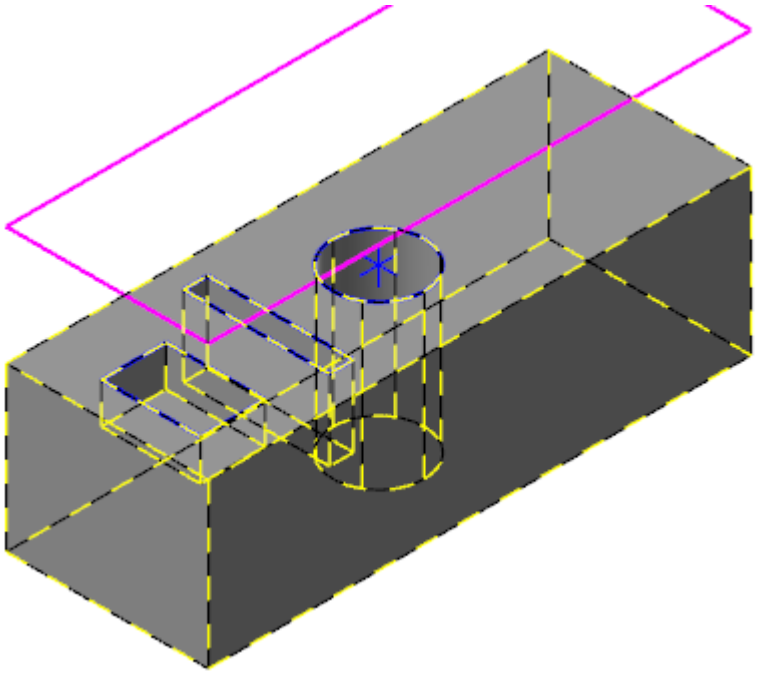


The **Solid Selection** dialog box opens.

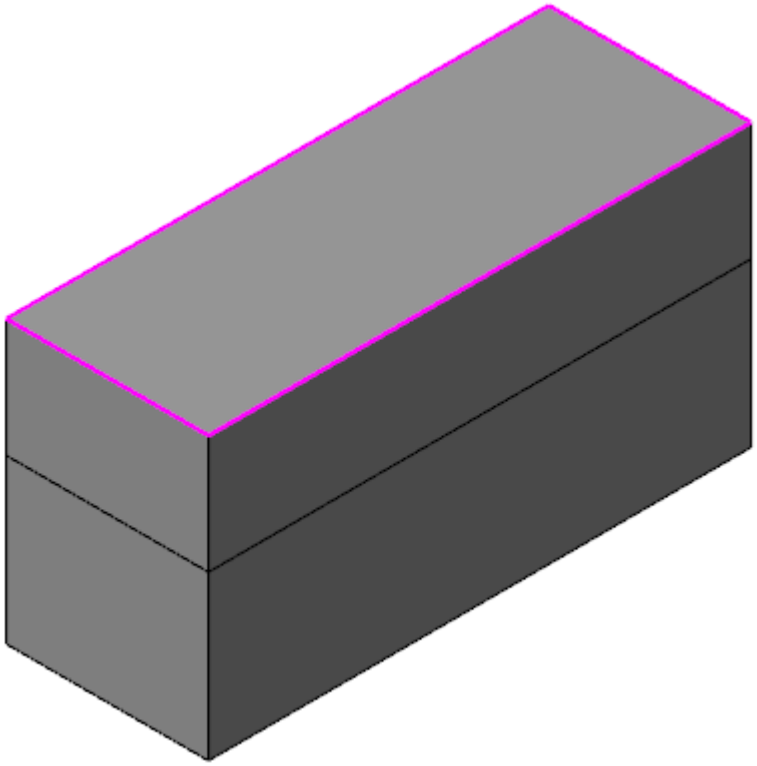
5. Turn off all of the filters except for **Body**.



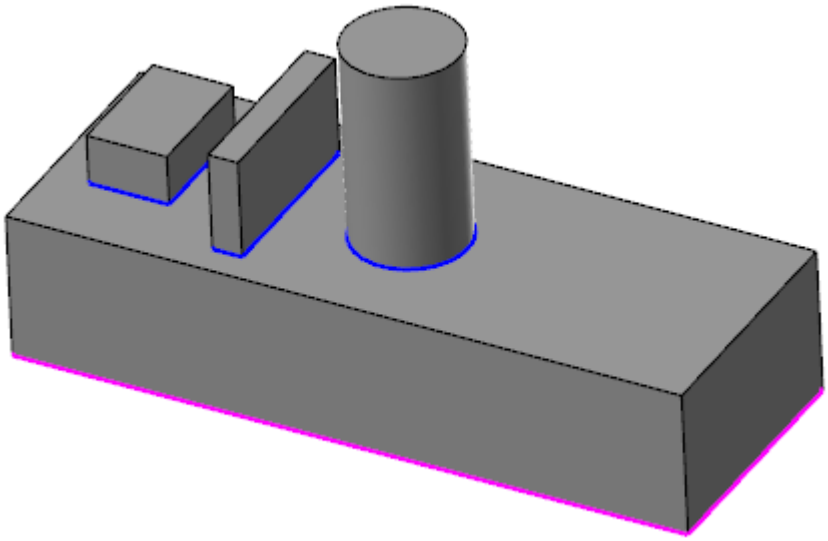
6. Select the solid body, and then click **OK** in the **Solid Selection** dialog box.



7. **Solid Impression** creates a new solid body above the original geometry.



8. Delete the original solid body and rotate the new one to see the solid impression.



9. Save your file.

   You have now completed the mold part.

CHAPTER 4

USING THE SOLID HOLE FUNCTION

The Hole function automates the creation and editing of holes in solids, saving you from creating circles and extruding them. You select the hole positions, dimensions, and type. In this chapter, you will create holes, experimenting with creating holes with planes, and how to create and use presets.

Hole saves time by not requiring you to create wireframe geometry and multiple extrude operations for simple holes. It allows you to correctly represent the bottom of a hole as it will be machined, and not just as a flat bottom. You create multiple holes in one operation, and can utilize a library of standard and user-created sizes to save time.

In this chapter, you will be reusing the `SolidsHole` part file to create holes of different sizes and types.

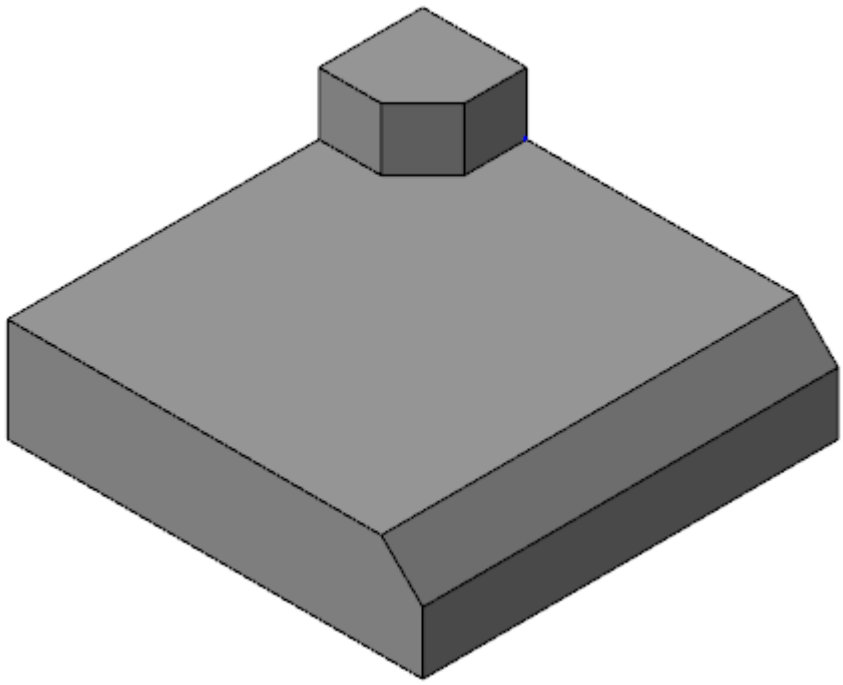
Goals

- Create a basic hole.
- Create holes on a vector, construction plane, and solid face.
- Create a preset.
- Create holes using the preset.

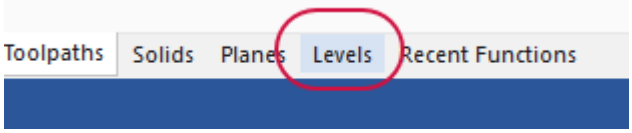
Exercise 1: Creating Holes

In this exercise, you will create a basic hole with the `SolidsHole` part file. This part file will be reused for the next few exercises.

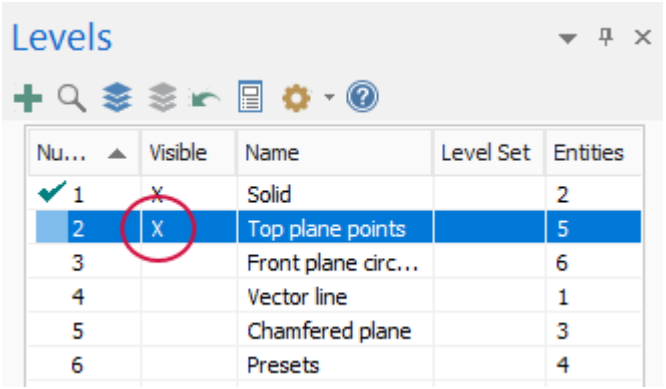
1. Load the file `SolidHole`.



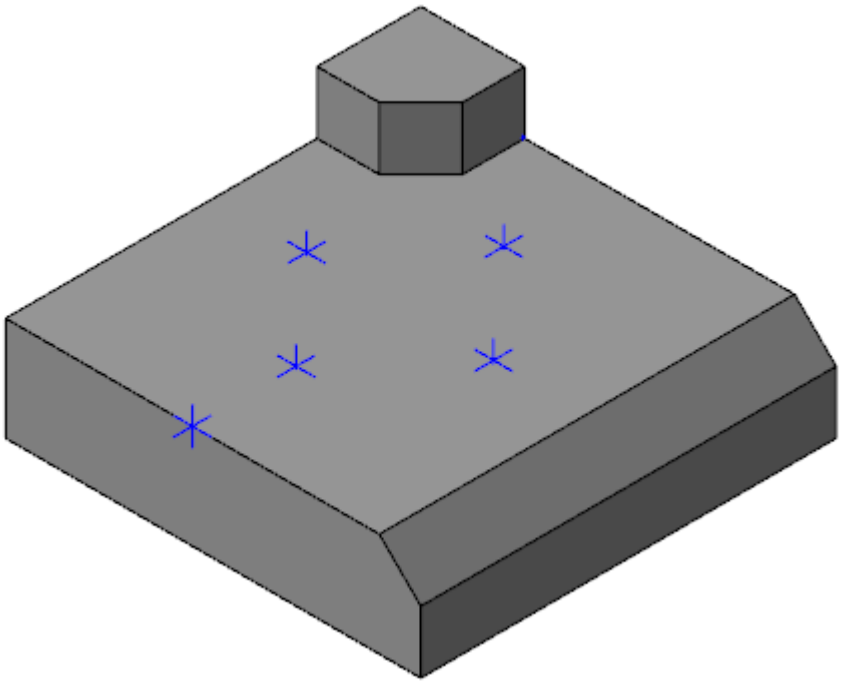
2. Save the file under the new name `SolidHole01-XXX`, where *XXX* is your initials.
3. Select the **Levels** tab to open the Levels Manager.



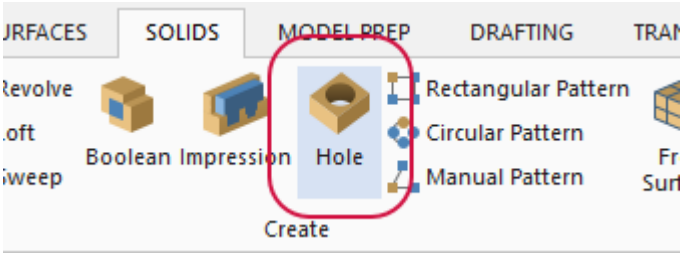
4. Select in the **Visible** column for level 2 to display it.



5. There are now five points displayed that you will use to create holes.

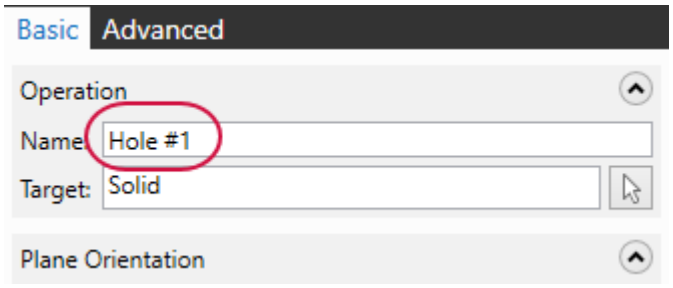


6. On the **Solids** tab, select **Hole**.



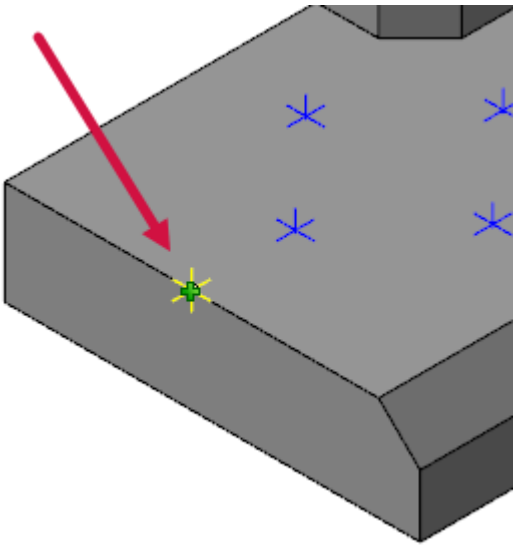
The **Hole** function panel opens.

7. Enter **Hole #1** as the **Name** of the operation.

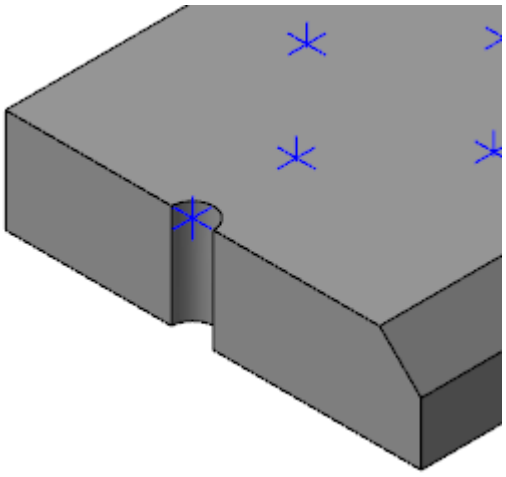


8. Select **Add Position** to select a hole position.

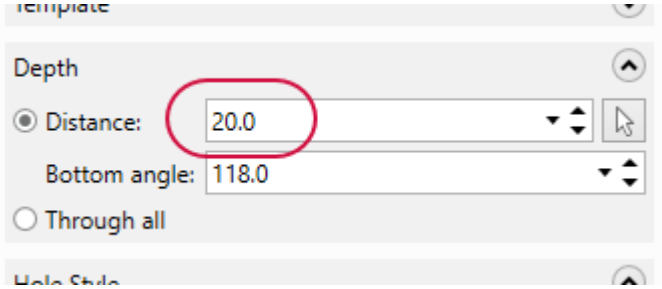
9. Select the point shown in the picture below.



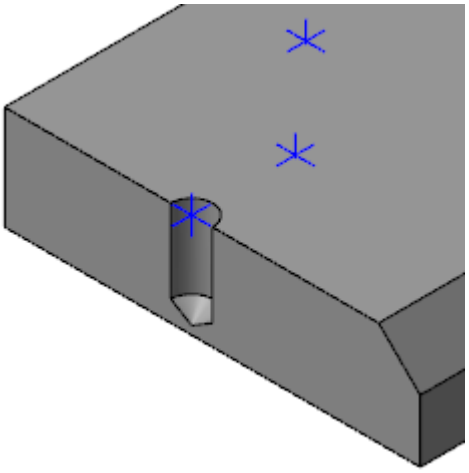
10. Press [**Enter**]. You will now have a hole that cuts into the side of the part.



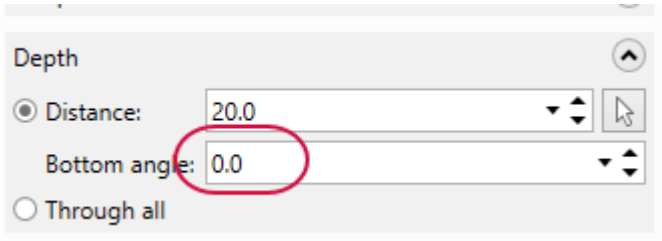
11. Set **Distance** under **Depth** to **20**.



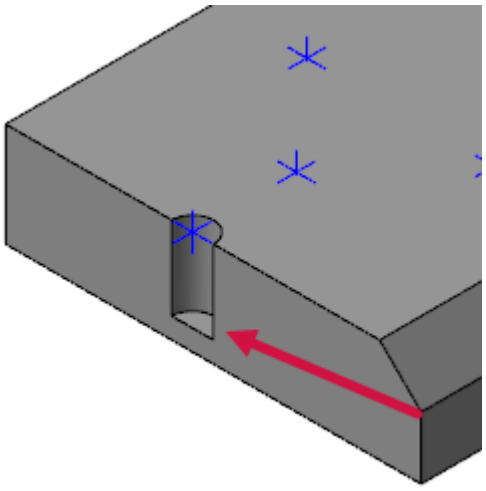
12. This sets the depth of the created hole.



13. Set the **Bottom angle** to **0**.

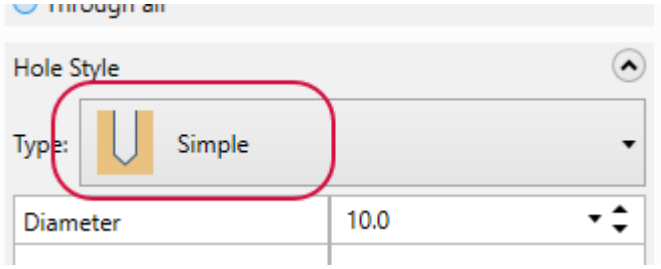


14. This removes the angle from the bottom of the hole, leaving it flat.



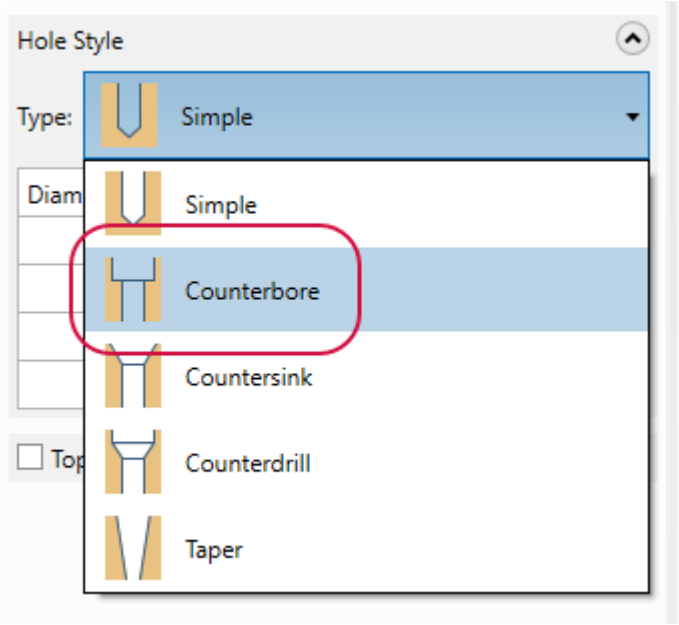
15. Set **Bottom angle** back to **118** to remove the flat bottom.

16. Notice that the **Type** is currently set to **Simple**.

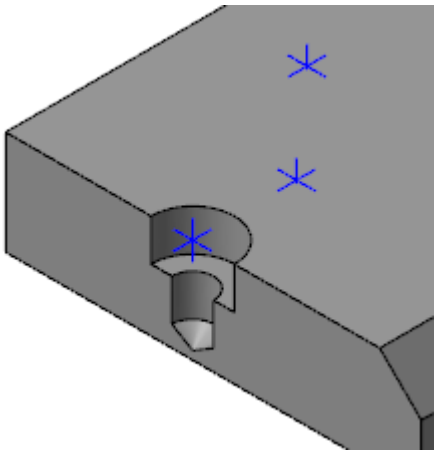


This sets the style of hole you are creating. You can create a counterbore, countersink, counterdrill, or a taper hole.

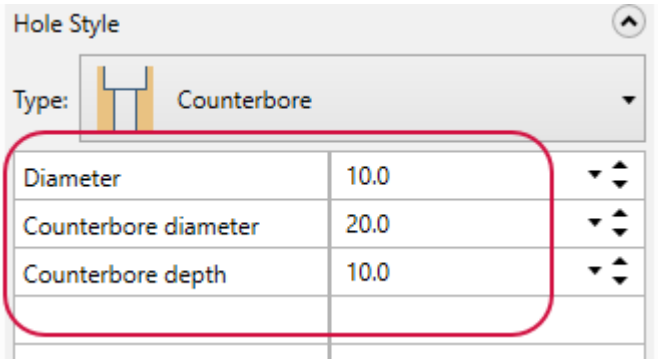
17. Change **Type** to **Counterbore**.



18. The hole is now a counterbore type.

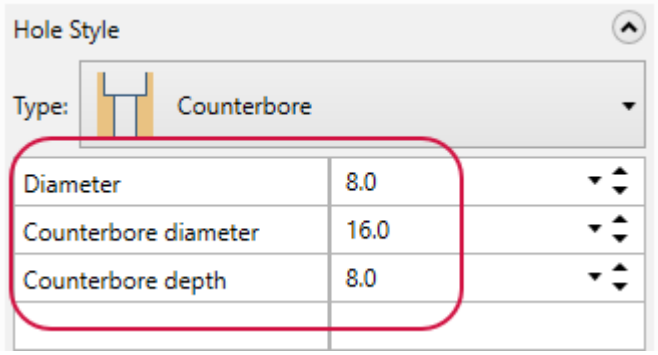


19. The **Hole Style** grid automatically updates to include information that is important for a counterbore hole type.

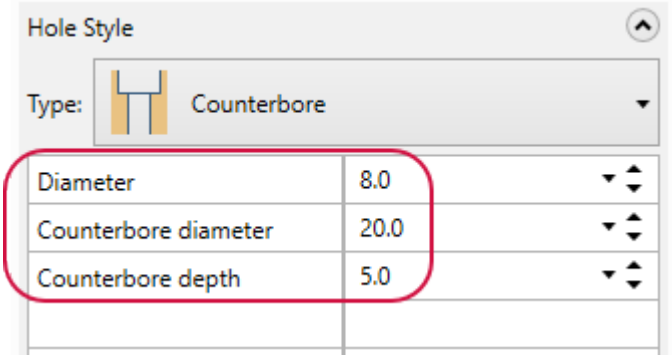


The **Diameter** stays the same, but the **Counterbore diameter** is twice that amount. The **Counterbore depth** is also the same as the **Diameter**.

20. Change **Diameter** to **8**.
21. The **Counterbore diameter** and **Counterbore depth** values automatically update. **Counterbore diameter** is now **16**, and **Counterbore depth** is now **8**.

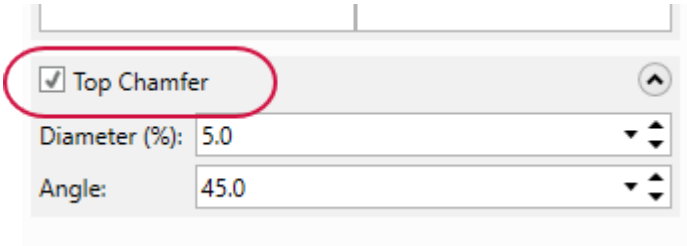


22. Change **Counterbore diameter** to **20** and **Counterbore depth** to **5**.



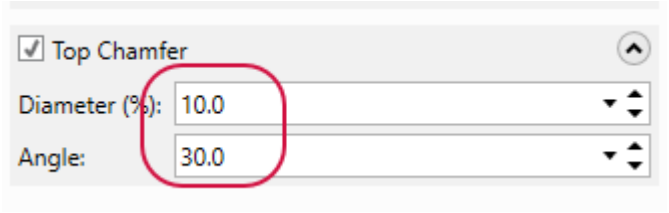
You will notice that the other values do not change. Changing the **Diameter** will change the counterbore values, but changing the counterbore values does not change the **Diameter**.

23. Select **Top Chamfer**.

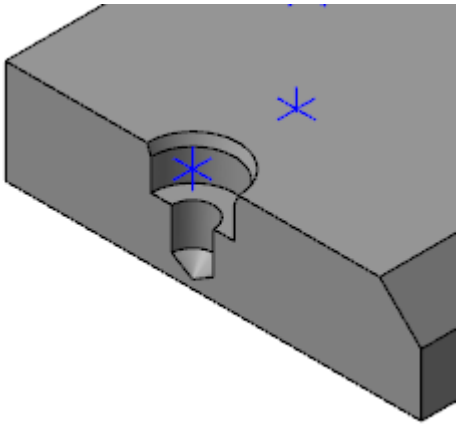


This option creates a chamfer at the top of the created holes.

24. Set **Diameter (%)** to **10.0** and **Angle** to **30**.



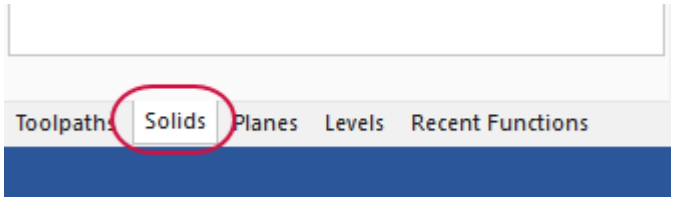
25. The created hole should match the picture below:



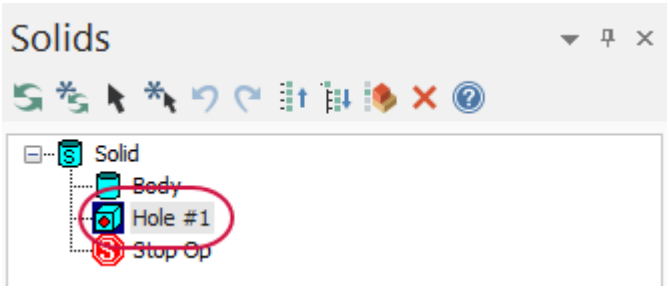
26. Click **OK** to create the hole.

Next, you will modify the hole operation to create more holes.

1. Select the **Solids** tab, if the Solids Manager is not currently open.



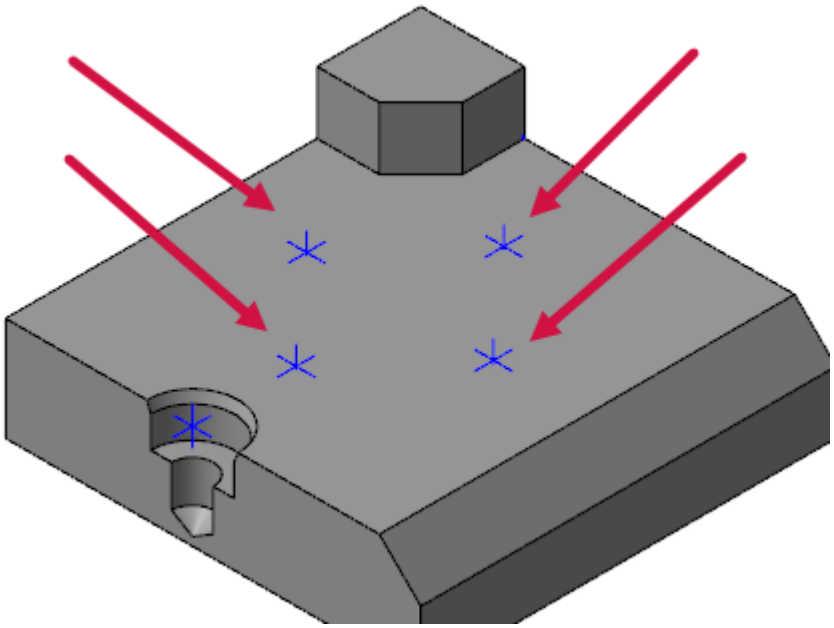
2. Double-click on Hole #1.



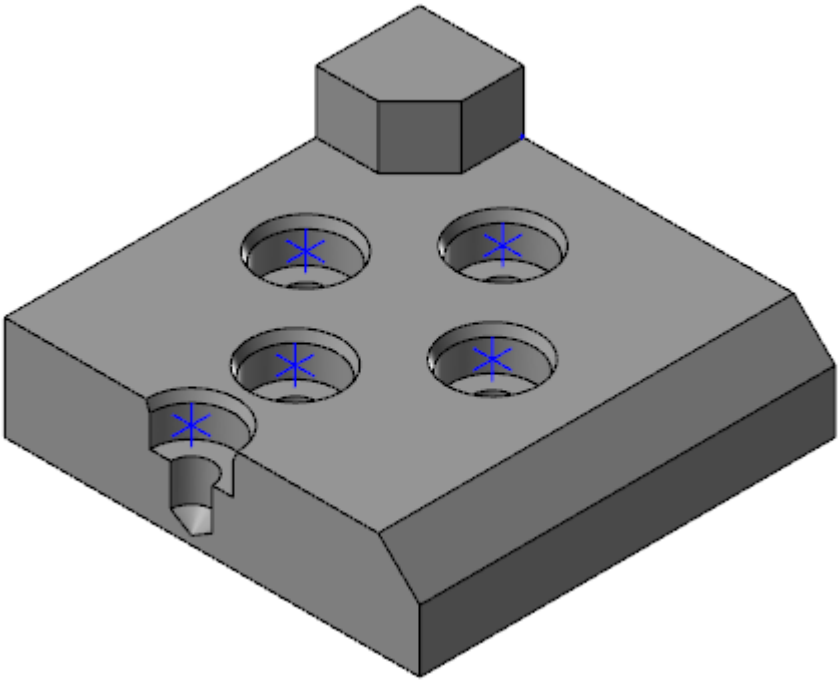
The **Hole** function panel opens.

3. Select **Add Position**.

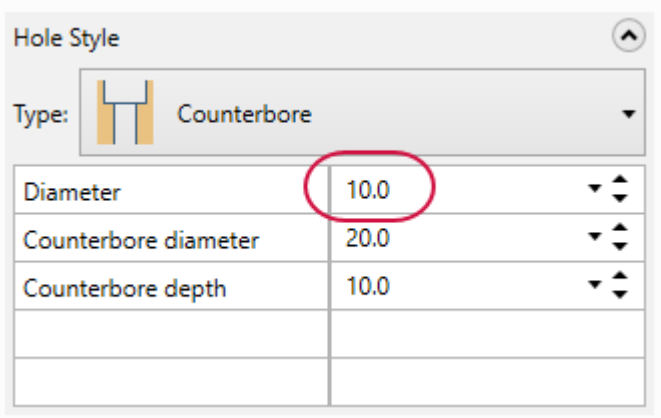
4. Select the four points shown below, and press [**Enter**].



5. There are now a total of five holes being created in one operation.

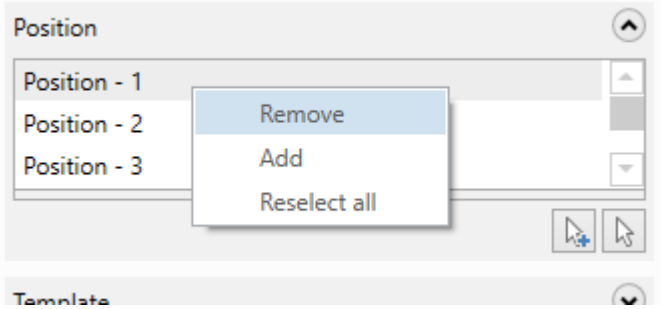


6. Change **Diameter** to **10**.

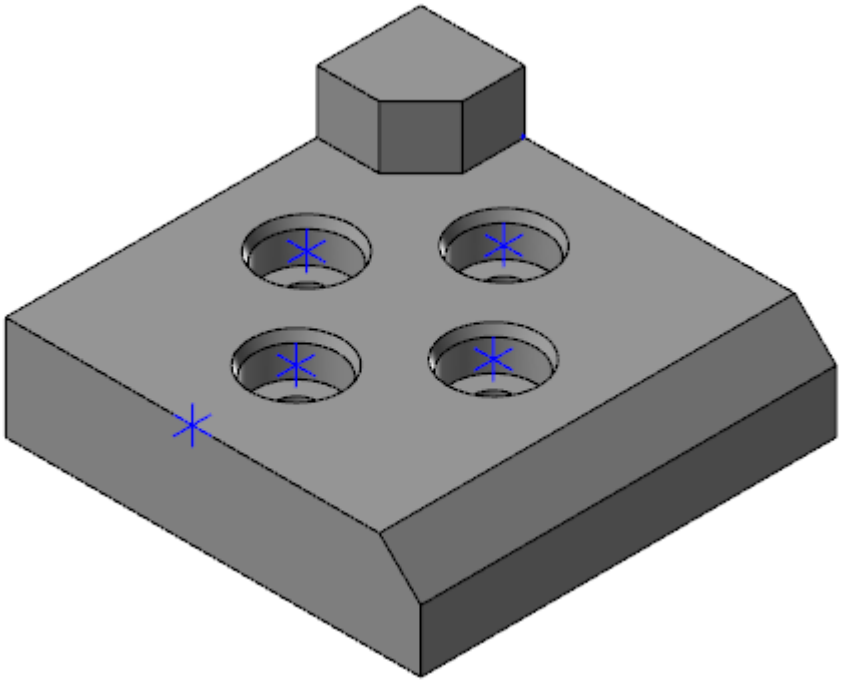


All five holes now have the same diameter.

7. Right-click on **Position - 1** in the **Position** list, and select **Remove**.



8. This removes the point located on the edge of the solid.



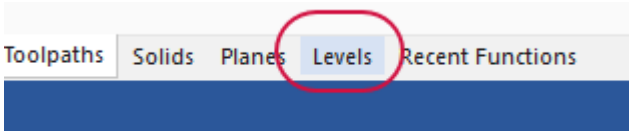
9. Select **OK and Regenerate** on the **Hole** function panel to accept these changes and regenerate the operation.

10. Save your part.

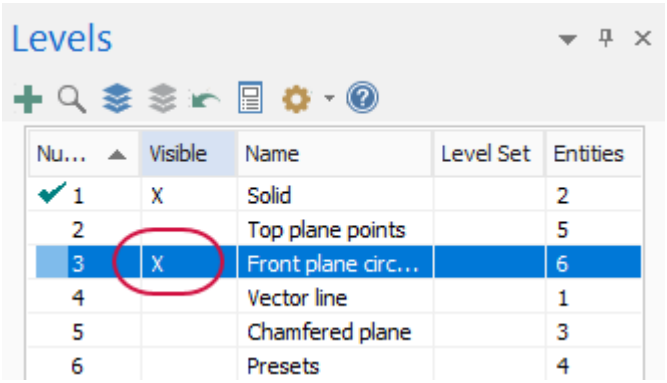
Exercise 2: Creating Holes With a CPlane

In this exercise, you will use the same part file (`SolidsHole`) to create holes using a construction plane (CPlane). It is recommended that you do not use your part file from the previous exercise so that you can fully see the holes you are creating.

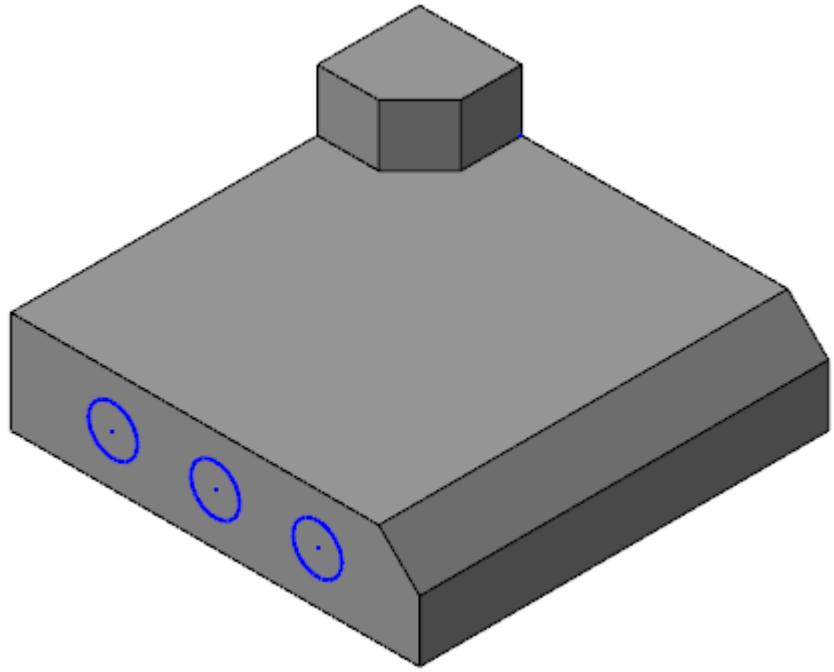
1. Open the file `SolidsHole`.
2. Save the file under the new name `SolidHole02-XXX`.
3. Select the **Levels** tab.



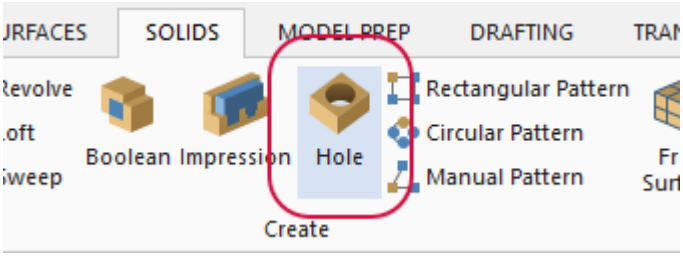
4. Select in the **Visible** column for level 3 to display it.



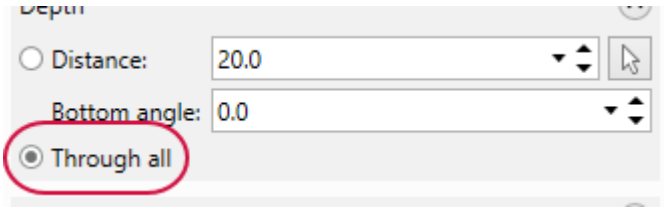
5. Three wireframe circles display on the front face of the solid and the other point geometry is hidden.



6. Select **Hole** on the **Solids** tab.

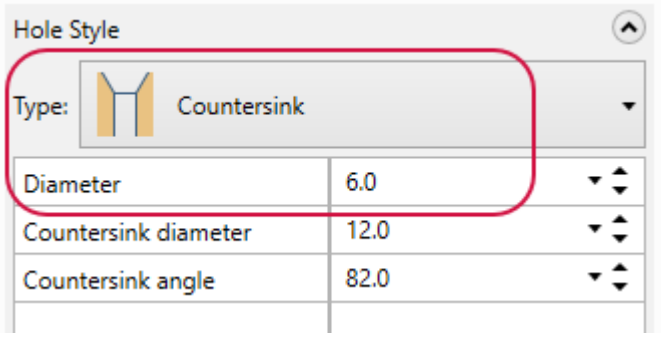


7. Set **Depth** to **Through all**.

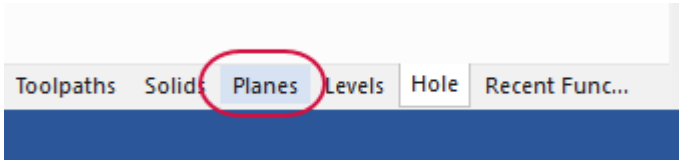


This will extend the holes entirely through the target body.

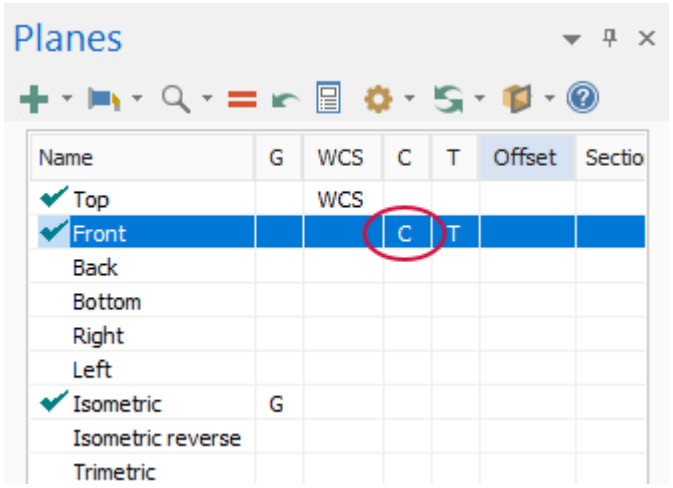
8. Set **Type** to **Countersink** and **Diameter** to **6**.



9. Select the **Planes** tab to open the Planes Manager.

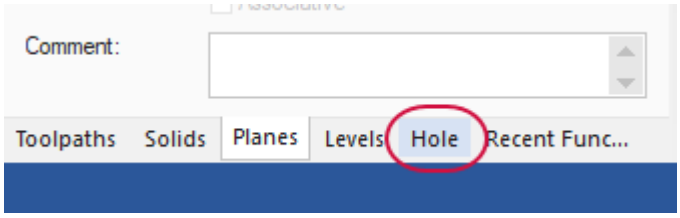


10. Select the **C** column for **Front**.

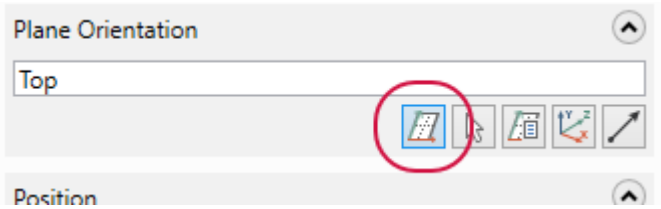


This sets the construction plane to the Front plane.

11. Select the **Hole** tab to return to the function panel.



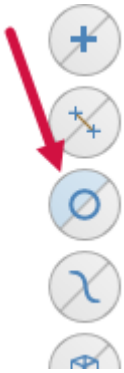
12. Select **CPlane**.



This sets the plane orientation to the current CPlane, which is now Front. **Plane Orientation** displays the current plane or vector location the holes will be placed on.

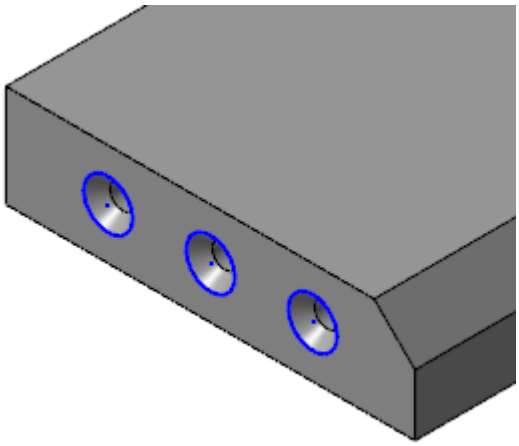
13. Select **Add Position**.

14. Click the left side of the **Arc Quick Mask**.



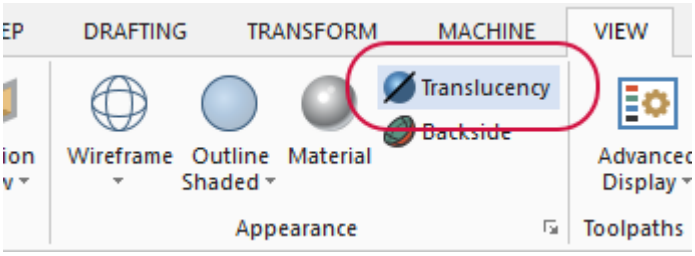
This selects all the arc entities currently displayed on screen.

15. Press [**Enter**].

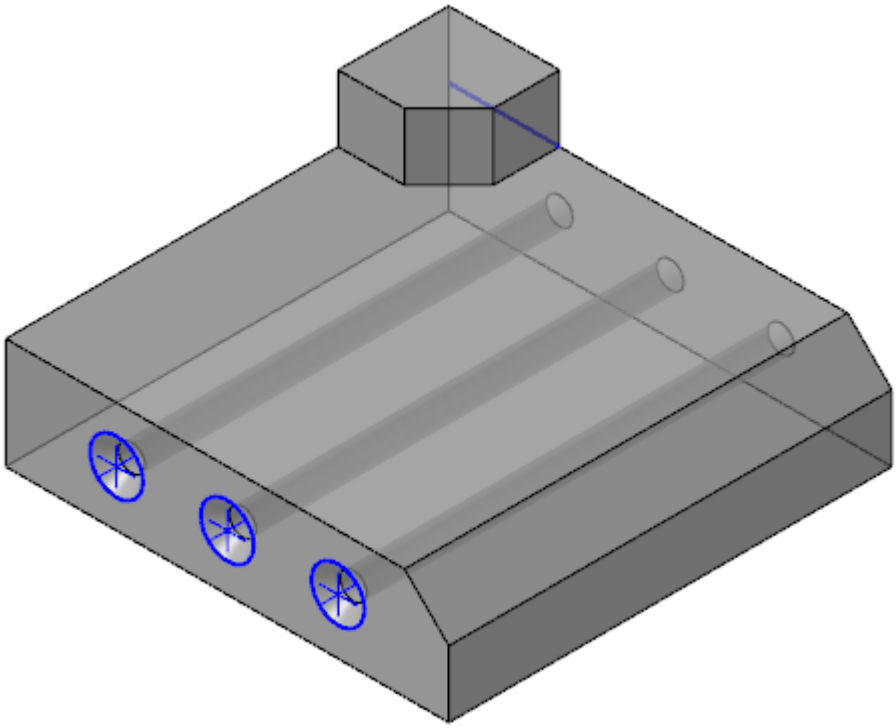


The points on the front face of the solid have been selected.

16. On the **View** tab, select **Translucency**.



17. You can see the points going all the way through the part, because **Through all** has been selected.

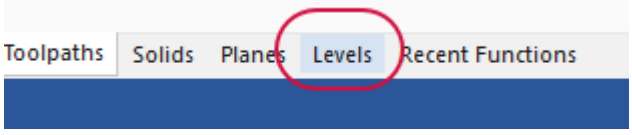


18. Select **Translucency** again to turn it off.
19. Click **OK** in the **Hole** function panel to create the holes.
20. Save your part.

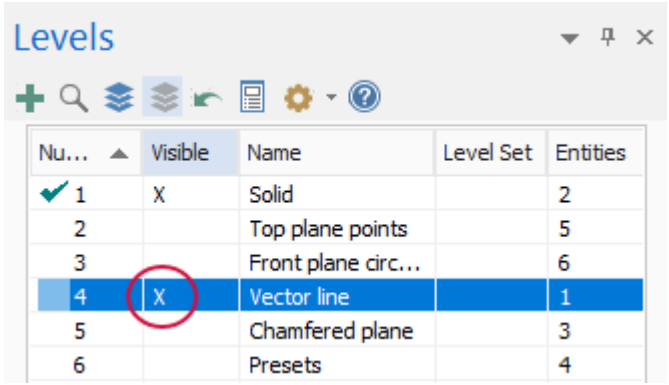
Exercise 3: Creating Holes With a Vector

In this exercise you will create a hole placed on a vector.

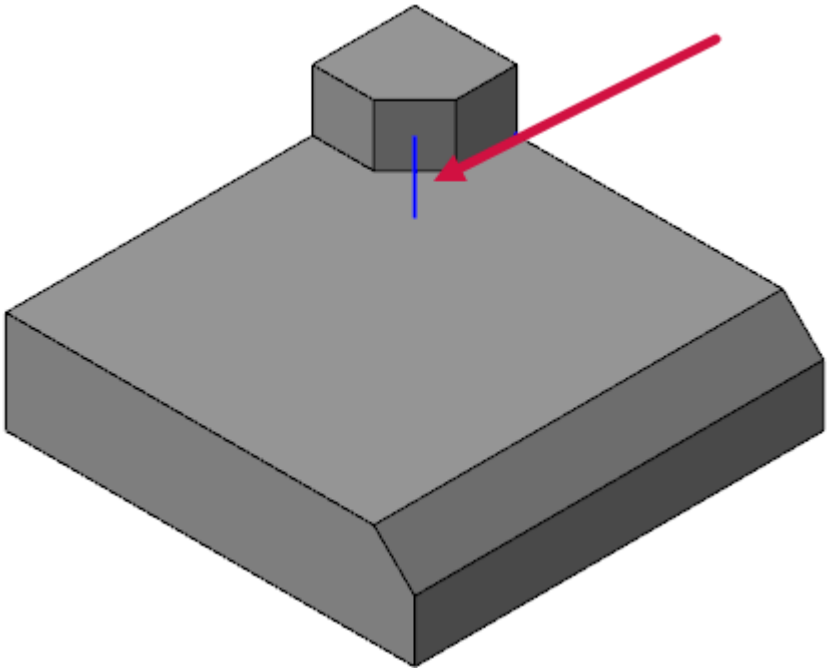
1. Load the file `SolidHole`.
2. Save the file under the new name `SolidHole03-XXX`.
3. Select the **Levels** tab to open the Levels Manager.



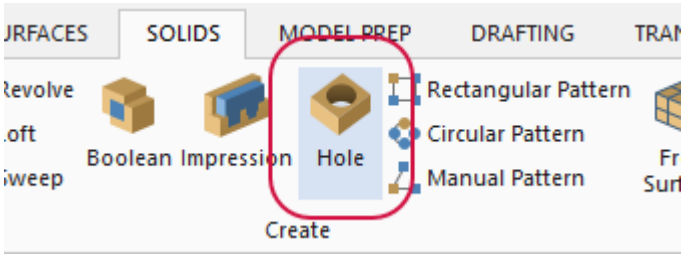
4. Select in the **Visible** column for level 4 to display it.



5. A line is now displayed next to the boss.

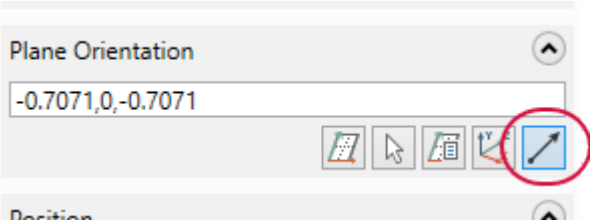


6. Select **Hole** on the **Solids** tab.



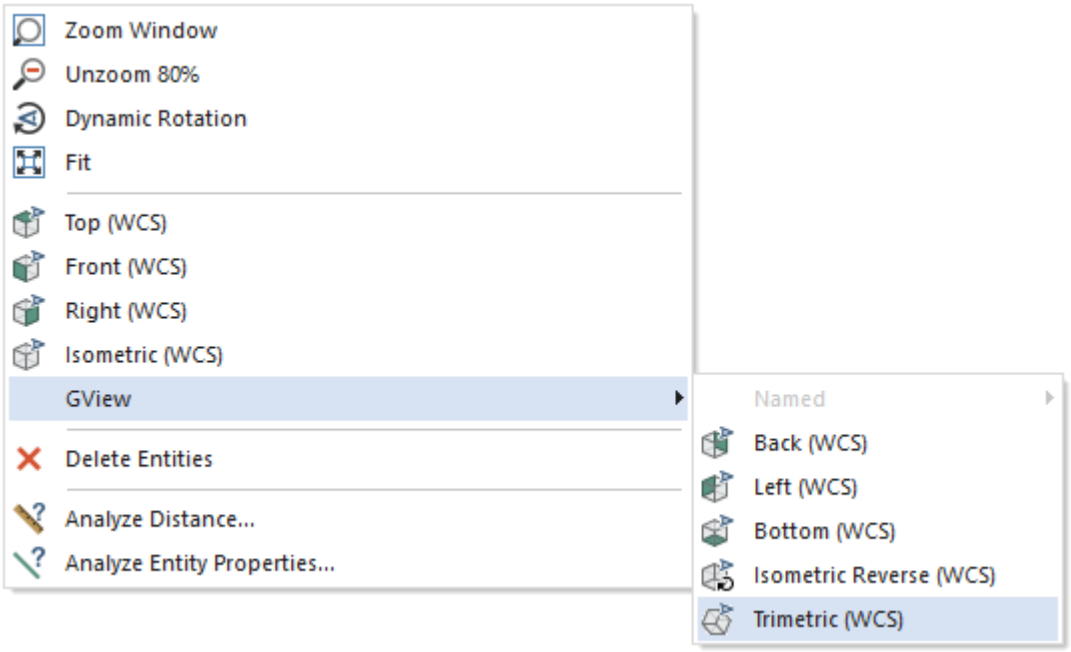
The **Hole** function panel opens.

7. Click **Select Vector** under **Plane Orientation**.

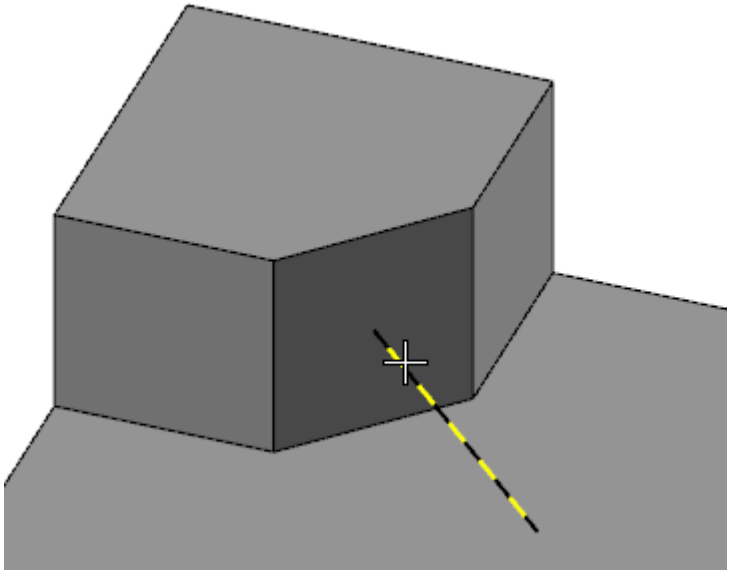


This has you select a line, edge, or two points to specify the hole direction.

8. Right-click in the graphics window and select **GView**, **Trimetric (WCS)** to change the view.



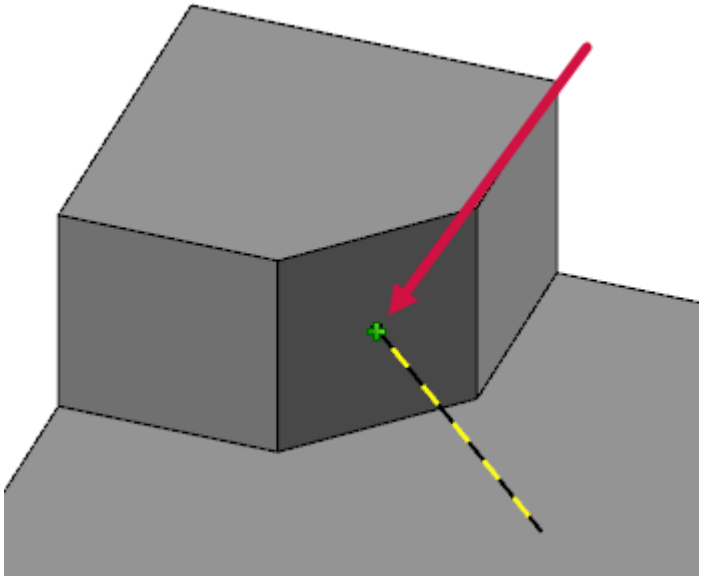
9. Select the line shown below:



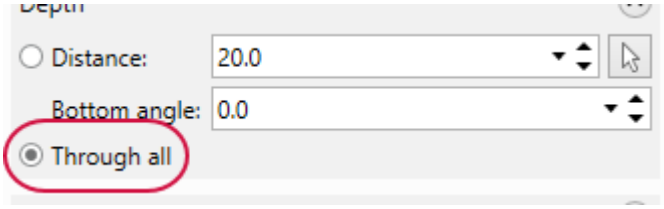
Note: Make sure that your cursor matches the image above. If you do accidentally select the endpoint of the line, ensure that you select the endpoint on the opposite side to complete the vector selection.

10. Select **Add Position**.

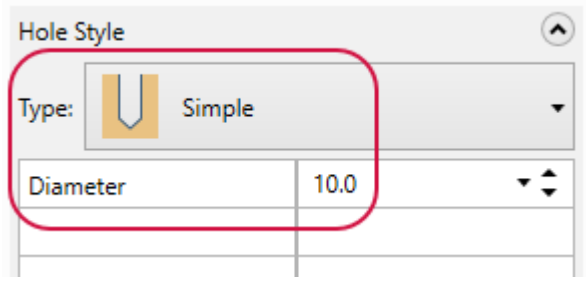
11. Select the endpoint of the line is touching the face of the boss and press [**Enter**].



12. Set **Depth** to **Through all**.

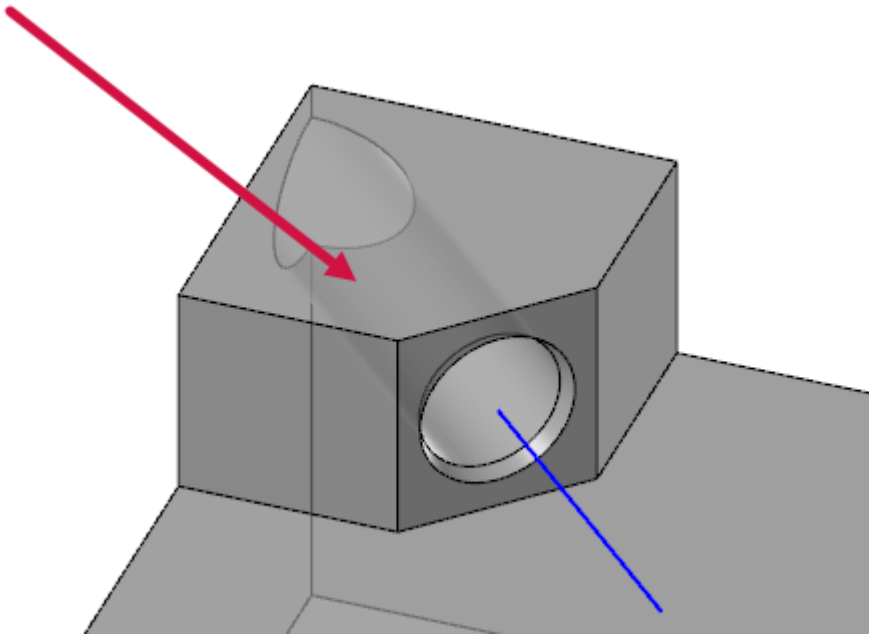


13. Set the **Type** to **Simple**, and enter a **Diameter** of **10**.



14. Click **OK** to create the operation.

15. You have now created a hole by using a vector line, as shown below:

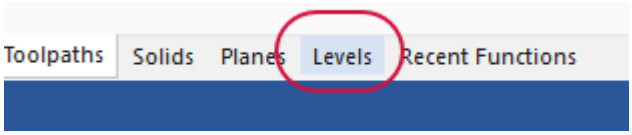


16. Save your file.

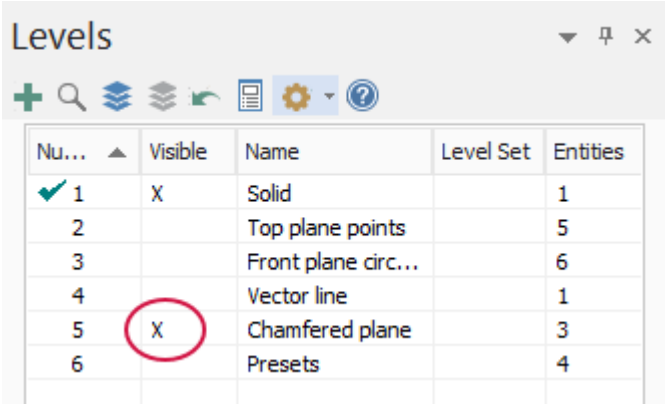
Exercise 4: Creating Holes With a Solid Face

In this exercise, you will be creating holes on the chamfered face of the part file.

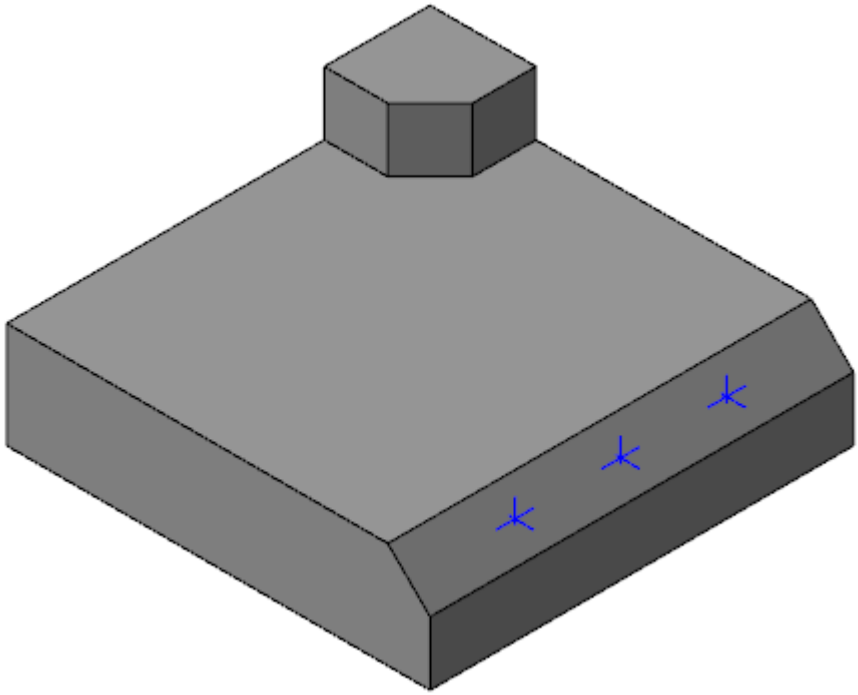
1. Load the file `SolidHole`.
2. Save the file under the new name `SolidHole04-XXX`.
3. Select the **Levels** tab to open the Levels Manager.



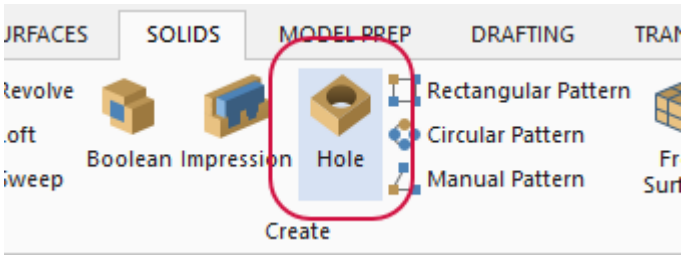
4. Select in the **Visible** column for level 5 to display it.



5. Points are now displaying on the chamfered face of the part.

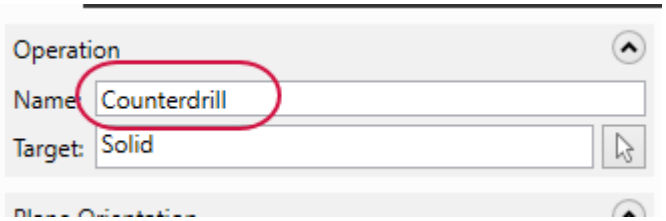


6. Select **Hole** on the **Solids** tab.

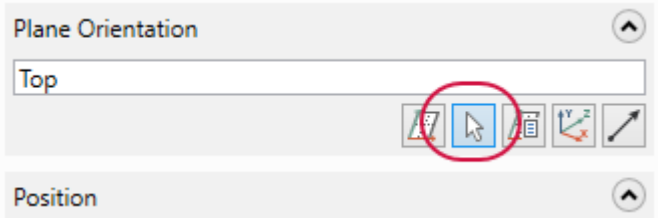


The **Hole** function panel opens.

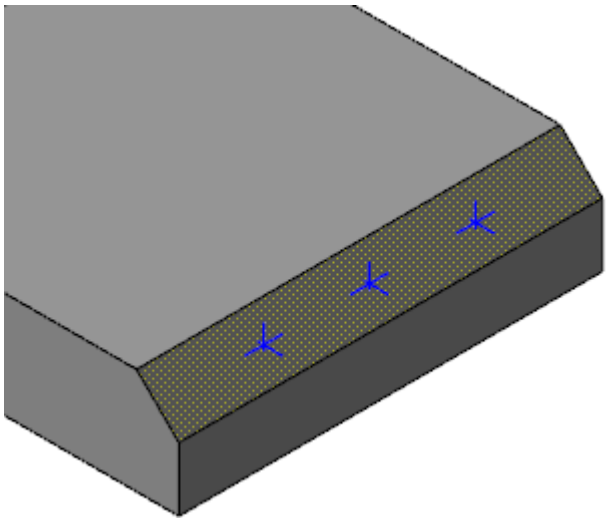
7. Set the **Name** of the operation to **Counterdrill**.



8. Click **Select Face** under **Plane Orientation**.

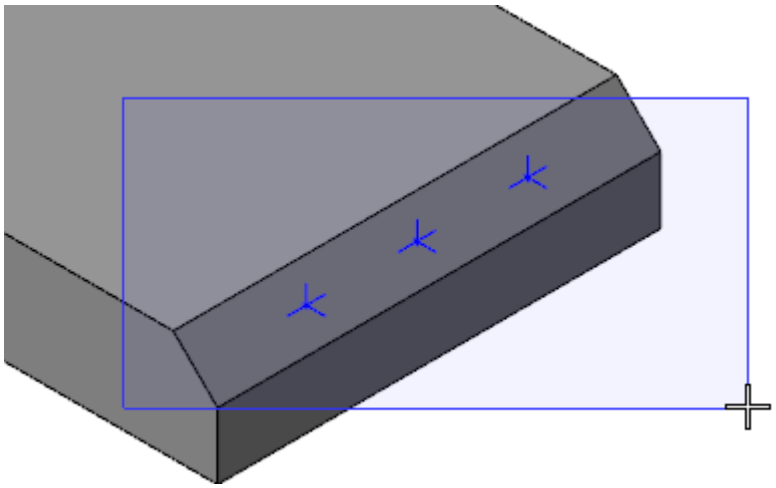


9. Select the chamfered face shown below:

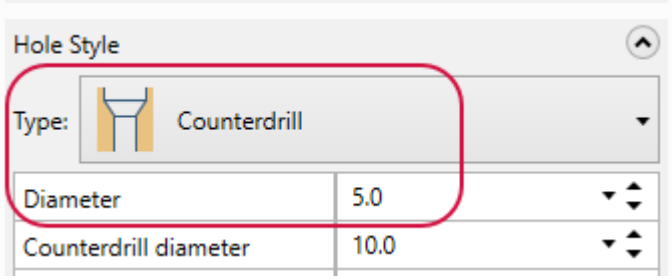


10. Select **Add Position**.

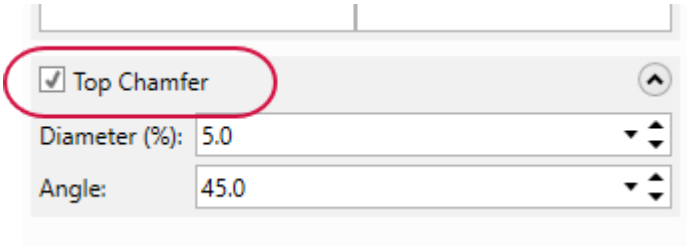
11. Window select the points and press [**Enter**].



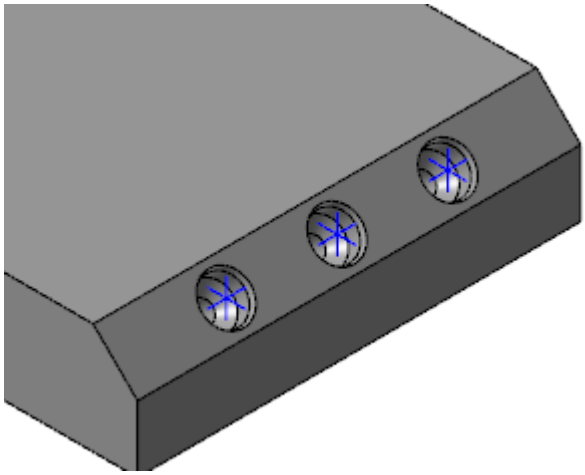
12. Set the **Type** to **Counterdrill** and set **Diameter** to **5**.



13. Select **Top Chamfer**.



14. Click **OK** to create the operation.
15. You now have three holes on the chamfered solid face.

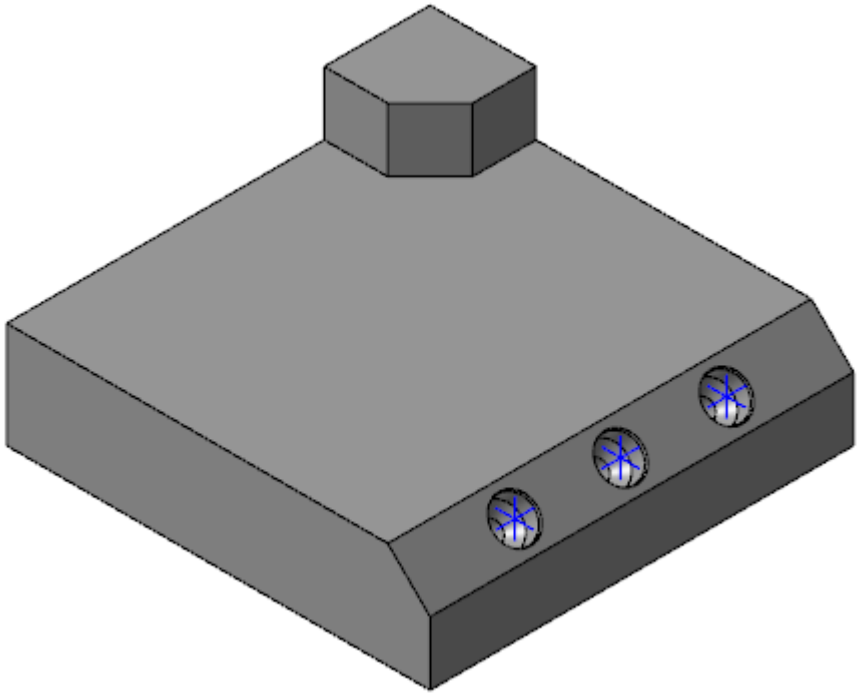


16. Save your part.

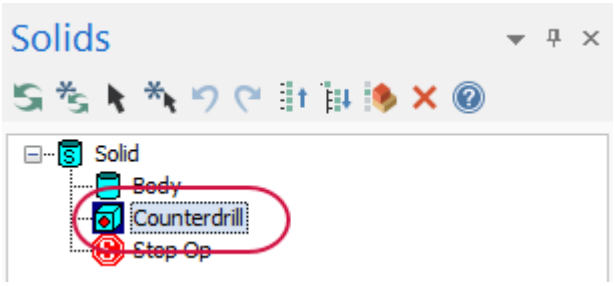
Exercise 5: Working with Presets

In this exercise, you will be creating hole presets and then using them to create subsequent holes. Presets allow you to save the parameters of holes that are not in the provided library to use in future part creation. In this case, it will be the parameters from the Counterdrill operation created in the previous exercise. For this exercise, continue with the part you created in the previous exercise, or load `SolidHole04`, which accompanies this tutorial.

1. Open the file, `SolidsHole-4`, if it is not already open.



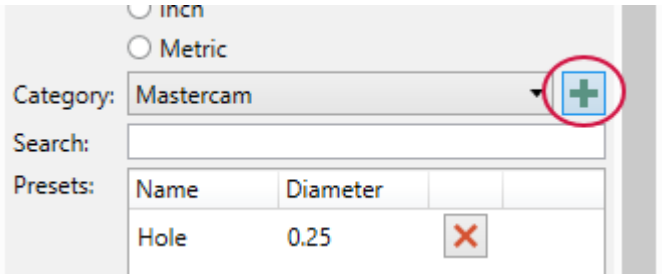
2. In the Solids Manager, double-click on the **Counterdrill** operation.



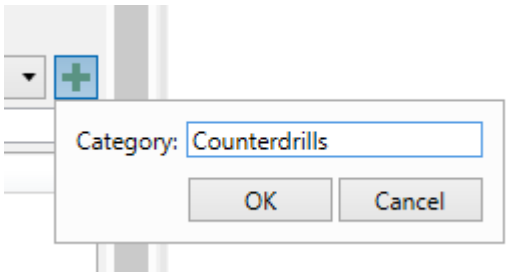
The **Hole** function panel displays.

3. Select the arrow next to **Template** to expand the group.

4. Click the plus next to **Category**.

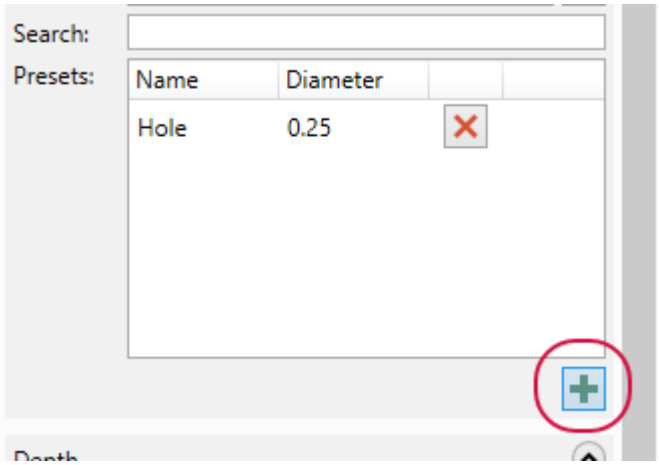


5. Enter **Counterdrills** as the category name, and click **OK**.

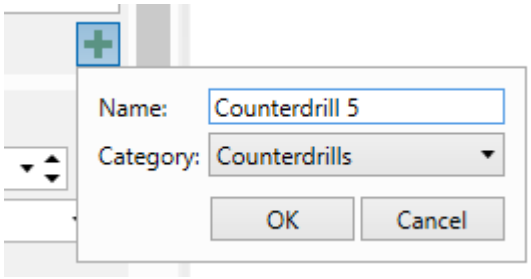


You have now created a new template category.

6. Select **Add Preset**.



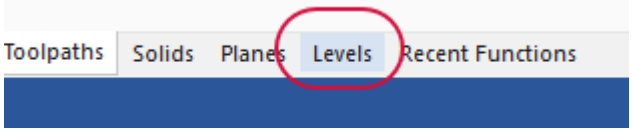
7. Enter **Counterdrill 5** for the **Name**, and set the **Category** to **Counterdrills**.



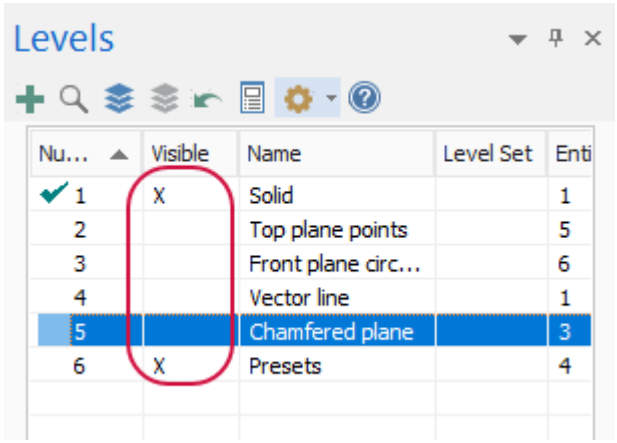
8. Click **OK**. You have now added the counterdrill hole to your **Counterdrills** category template.

9. Click **OK** in the **Hole** function panel.

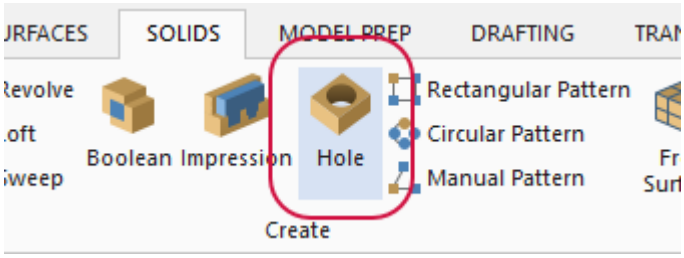
10. Select the **Levels** tab to open the Levels Manager.



11. Turn on level 6, and turn off all other levels except for level 1.

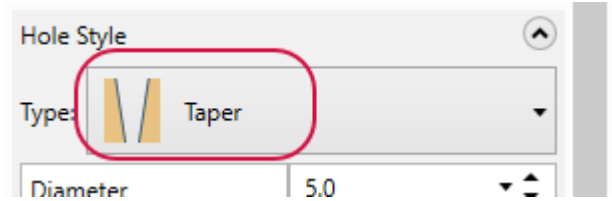


12. Select **Hole** on the **Solids** tab.

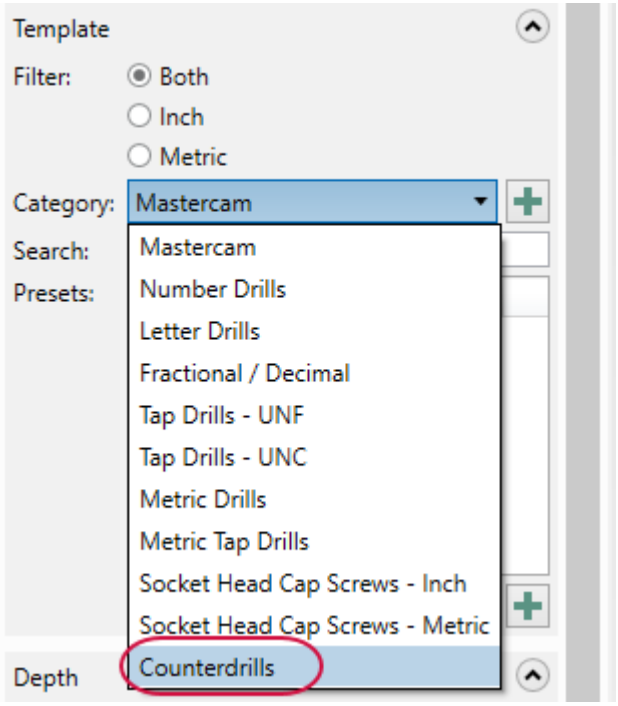


The **Hole** function panel opens.

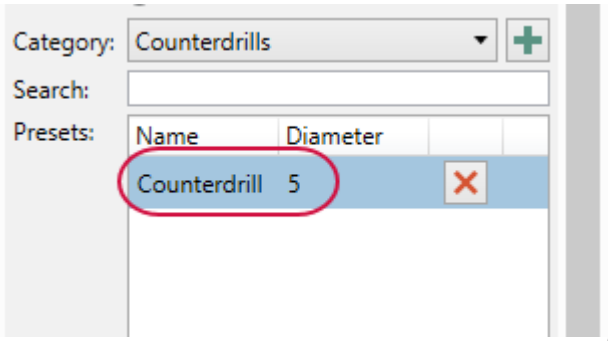
13. Change the **Type** to **Taper**.



14. Set the **Category** to **Counterdrills**.

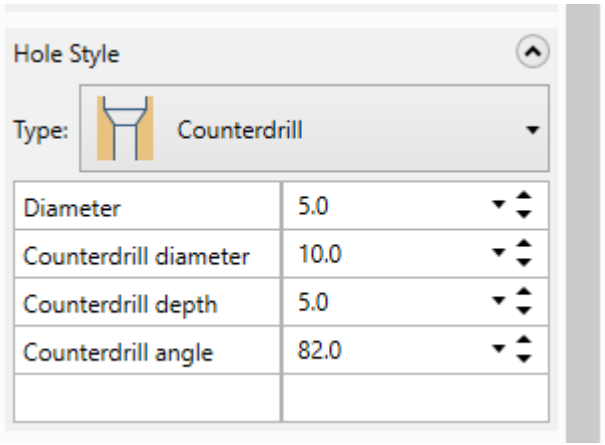


15. Select the preset you saved earlier.

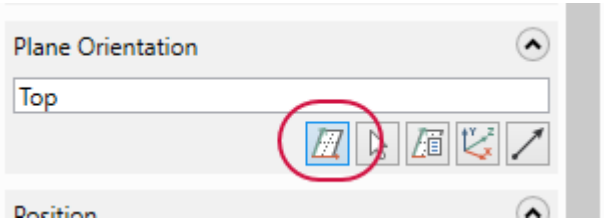


;

16. The parameters are then automatically updated to the selected preset. You will notice that the Type has been changed to Counterdrill, and the parameters match from the previous exercise.

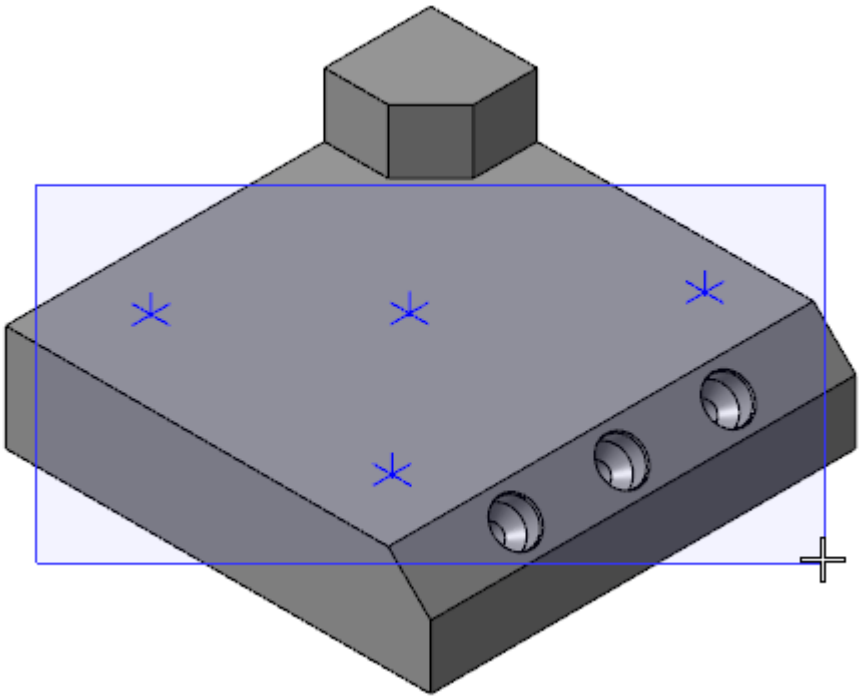


17. Select **CPlane** to set the **Plane Orientation** to **Top**.

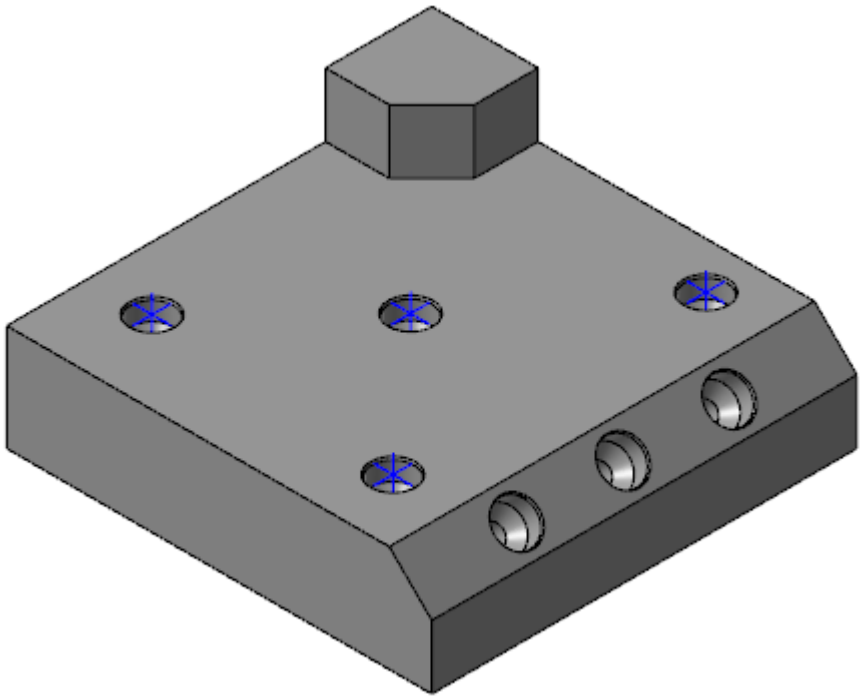


18. Select **Add Position**.

19. Window select the points on the top of the face and press [**Enter**].



20. You will now have points placed at the top of the face, with parameters saved in the presets.

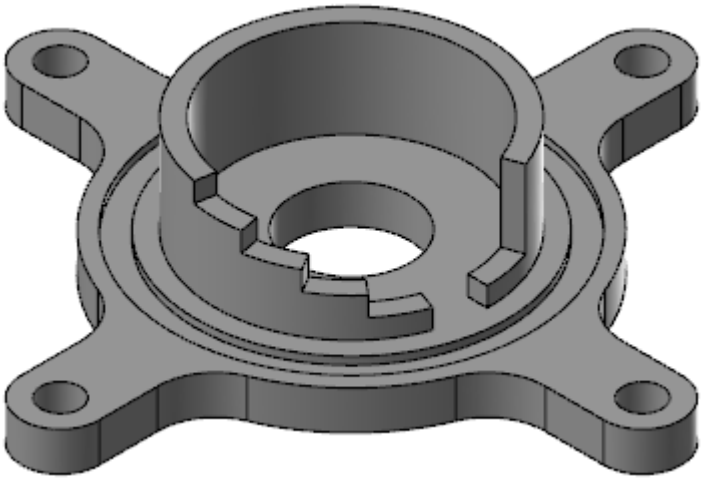


21. Save your part.

CHAPTER 5

MODIFYING A PART WITH MODEL PREP

Model Prep provides functions, such as **Push/Pull**, **Move**, and **Split Solid Faces**, that let you edit solids with no history tree. When you use Model Prep functions, Mastercam recognizes edges, faces, and even complete features, giving you the power to make changes without the operations or wireframe used to create the solid.



In this chapter, you will be modifying the above part based on new specifications that have to be made. The required changes are as follows:

- Remove one of the feet.
- Reduce the foot height by 5 mm.
- Reposition the three remaining feet to be 120 degrees apart.
- Increase the foot hole diameter to 12 mm.
- Reduce the part's center hole diameter to 20 mm and add a 30 mm counter bore, 2 mm deep.
- Align the opening on the vertical wall with one of the feet.
- Increase the opening width from 20 degrees to 45 degrees.
- Increase the vertical wall height from 35 mm to 50 mm.
- Add a 10 mm fillet to the vertical wall's inside bottom edge.

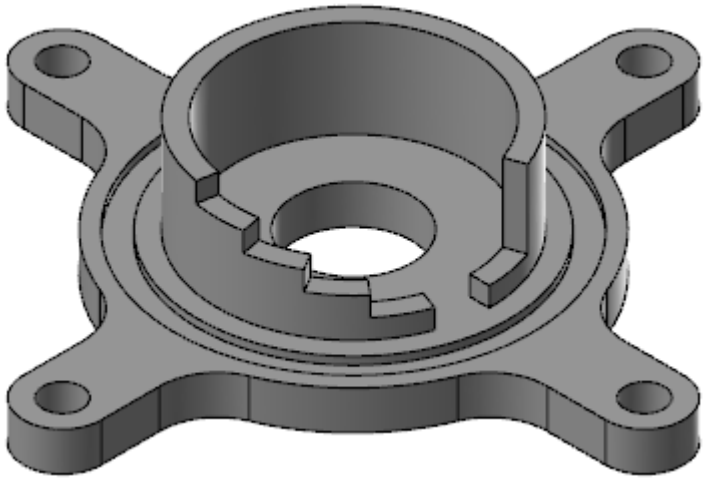
Goals

- Modify a part using various Model Prep functions

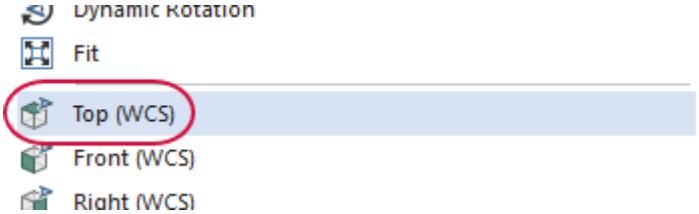
Exercise 1: Splitting the Feet from the Part

To manipulate the part's feet as needed for the edits, you must first separate them from the rest of the part.

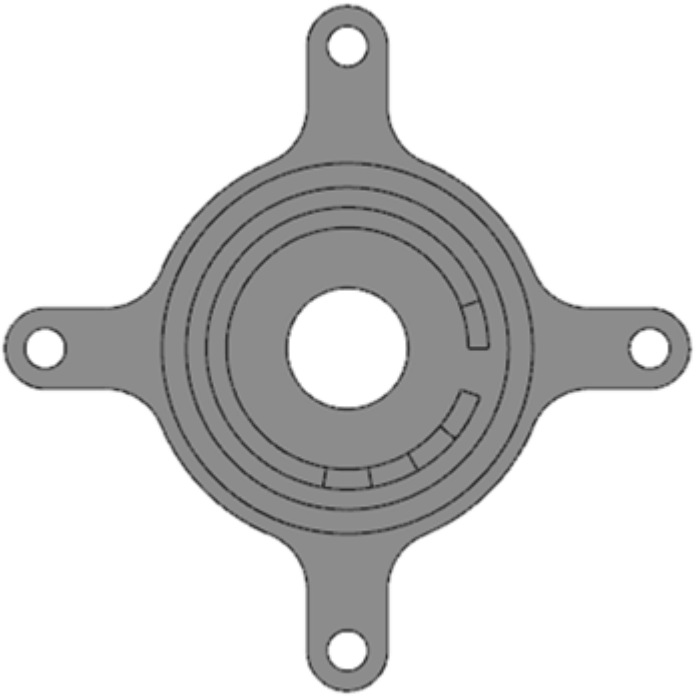
1. Load the file `FlangeDemo`.



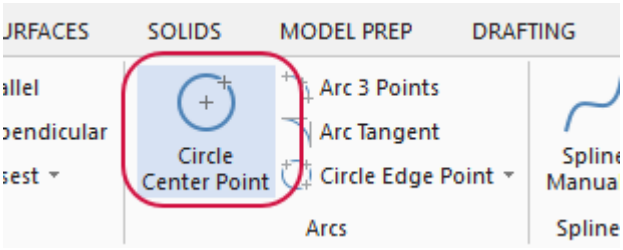
2. Save the file as `FlangeDemo-XXX`, where *XXX* is your initials. This prevents you from saving over the original file.
3. Right-click in the graphics area, and select **Top (WCS)**.



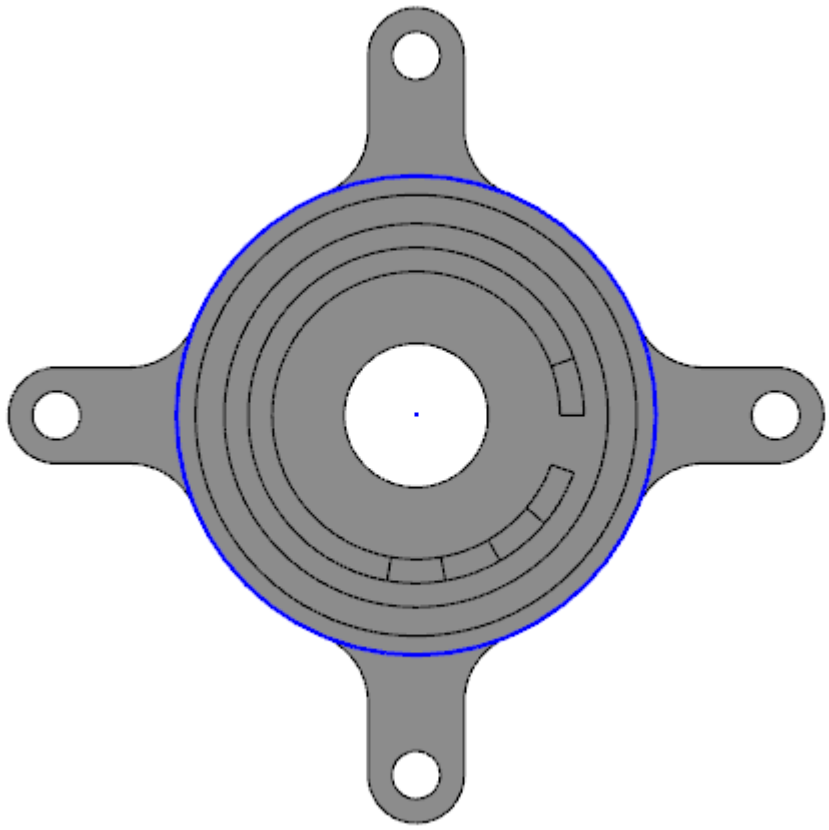
4. Mastercam displays the part in the top view.



5. Select **Circle Center Point** on the **Wireframe** tab.

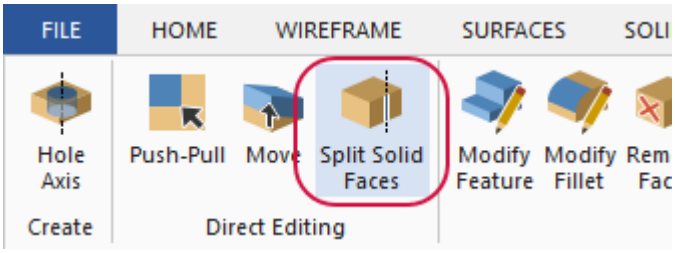


6. Draw a circle centered on the part, with a diameter (**100**) that matches the outermost circular edge of the part.



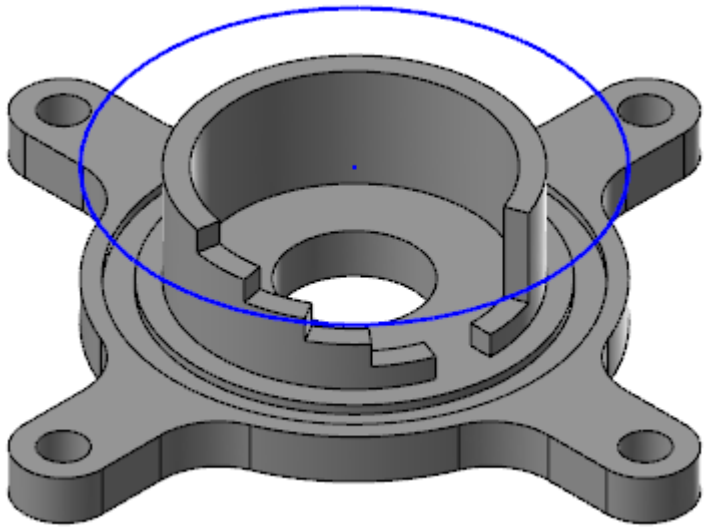
You will use this circle as helper geometry to separate the feet from the main body of the part.

7. Select **Split Solid Faces** on the **Model Prep** tab. This function splits single faces into multiple faces that you can manipulate separately.

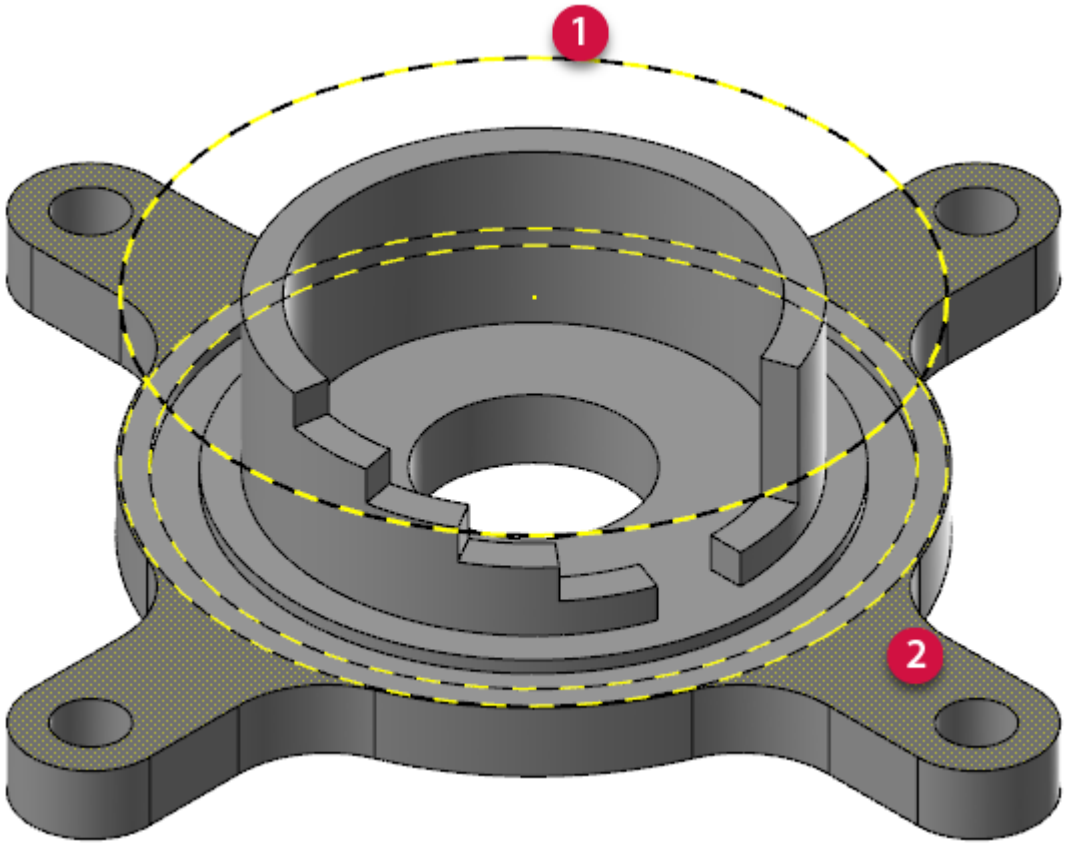


The **Split Solid Face** function panel opens.

8. Right-click in the graphics window, and select **Isometric (WCS)** to rotate the part view.

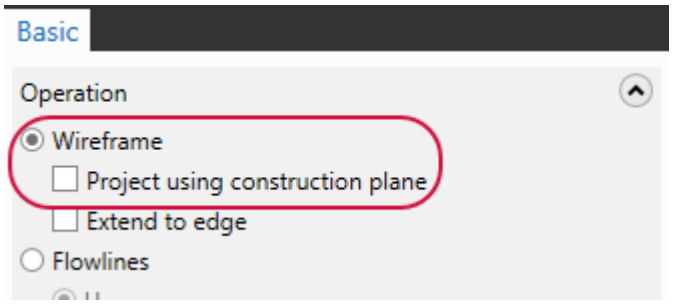


9. Click the circle and one of the feet.

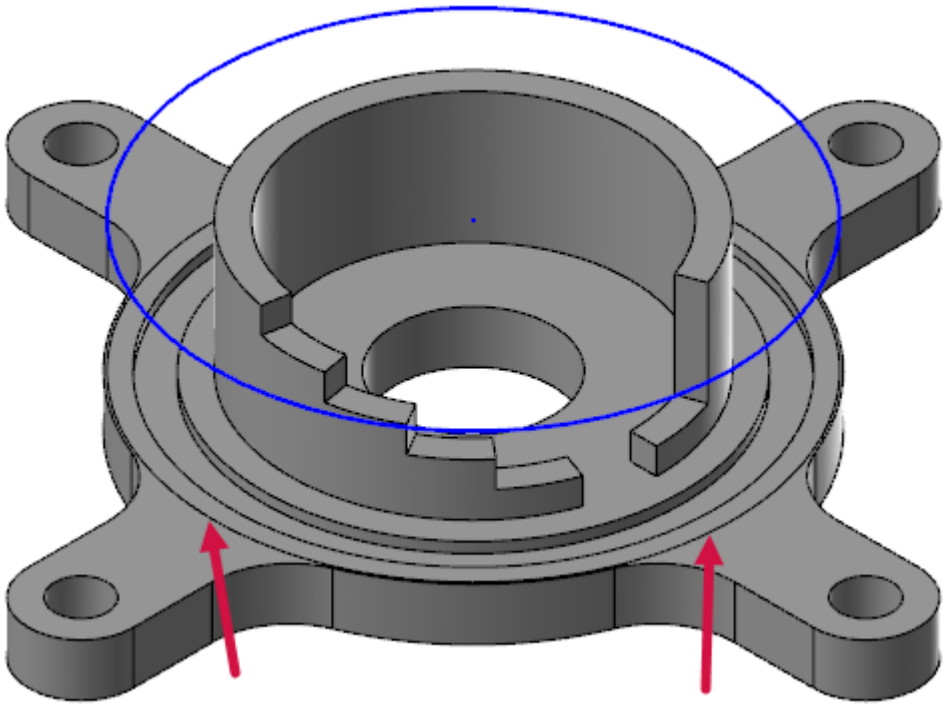


The circle projects down onto the selected face, indicating where the face will be split.

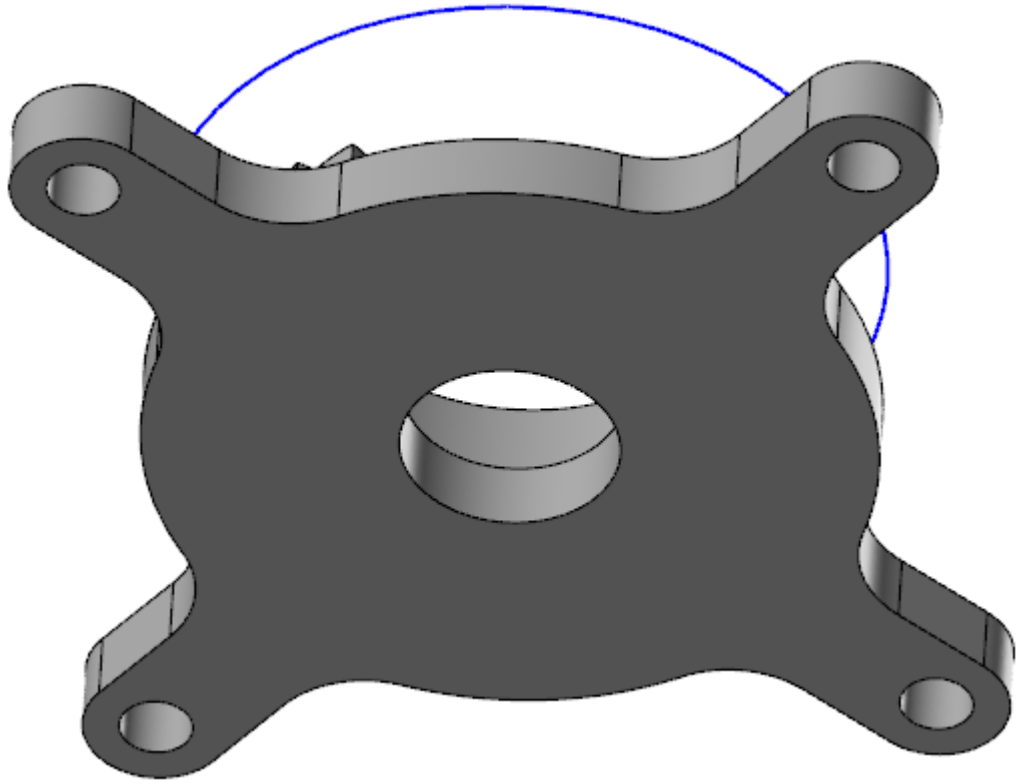
10. Ensure that **Wireframe** is selected and **Project using construction plane** is not selected.



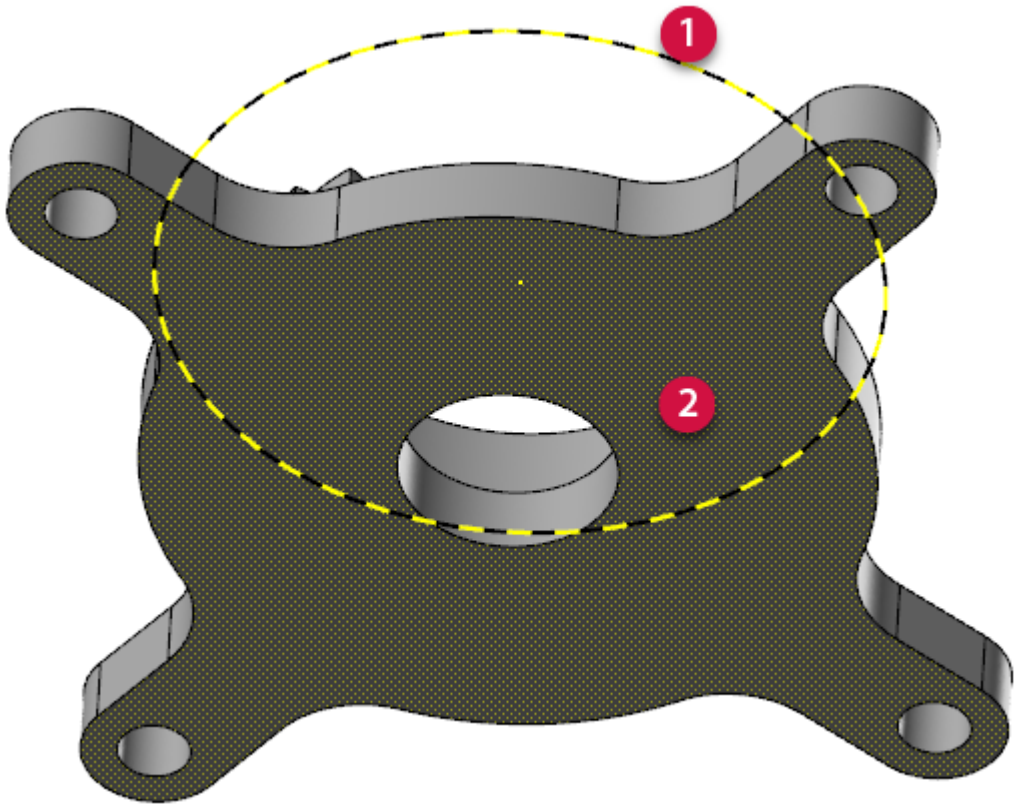
11. Click **OK and Create New Operation** to finalize your choices and begin a new **Split Solid Face** operation.
12. The top face of the feet is split from the rest of the part. The arrows in the picture indicate the split on the front feet.



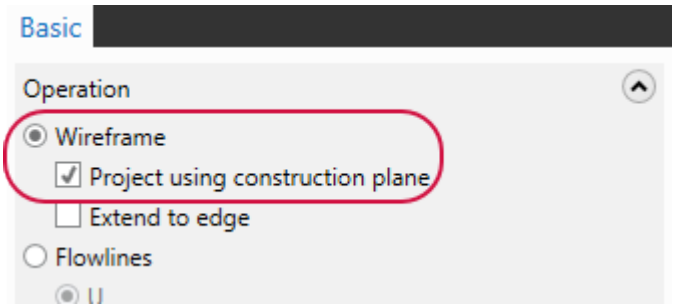
13. Hold down your mouse wheel and move the mouse to flip the part as shown below.



14. Select the bottom of the part and the circle.

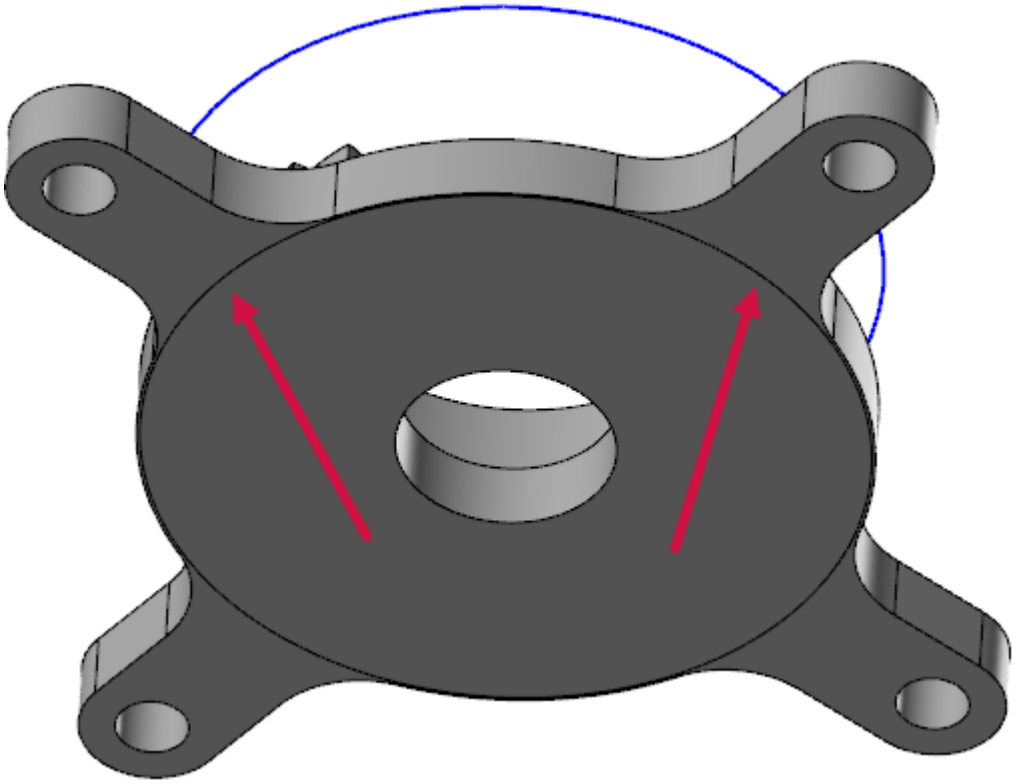


15. Select **Project using construction plane** on the **Split Solid Face** function panel.



The circle projects onto the face based on the construction plane, so you can split the face from the bottom.

16. Click **OK** to create the operation. The back face of the part is also split.

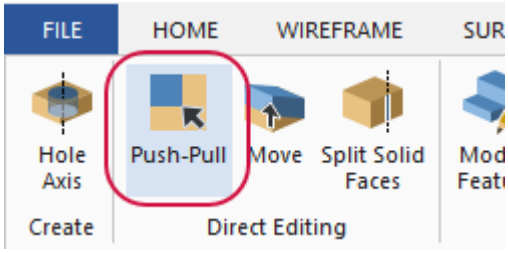


17. Delete the circle, and save the file.

Exercise 2: Removing One Foot

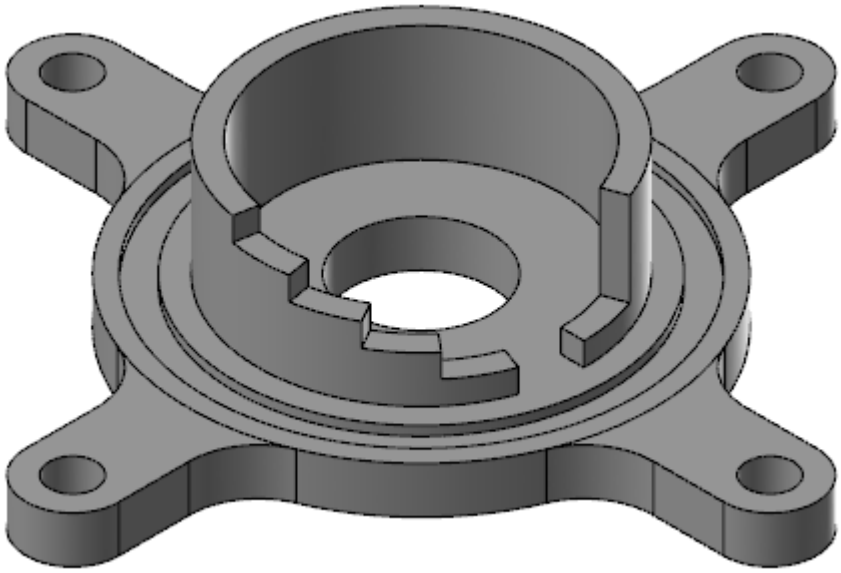
With the feet split from the rest of the part, you can remove one of them, leaving three as required by the modification instructions. For this exercise, continue with the part you just saved, or load `FlangeDemo01`, which accompanies this tutorial.

1. Select **Push-Pull** from the **Model Prep** tab.

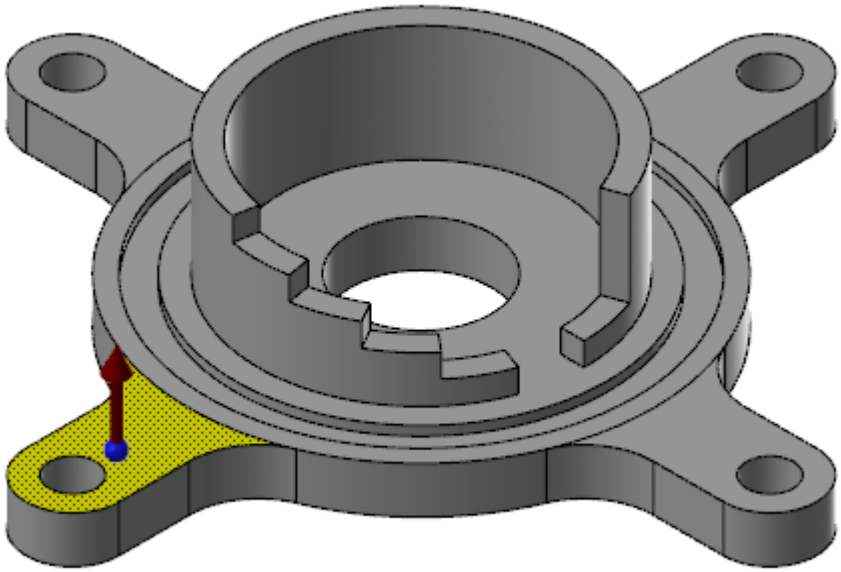


The **Push-Pull** function panel opens. Push-Pull extends bosses or produces cuts based on selected faces or features. You can also use Push-Pull to create fillets from edges or to remove fillets.

2. Place the part in Isometric view.

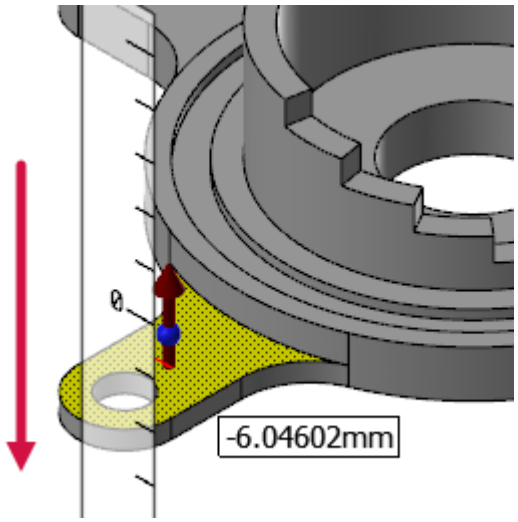


3. Click the lower-left foot, as shown.

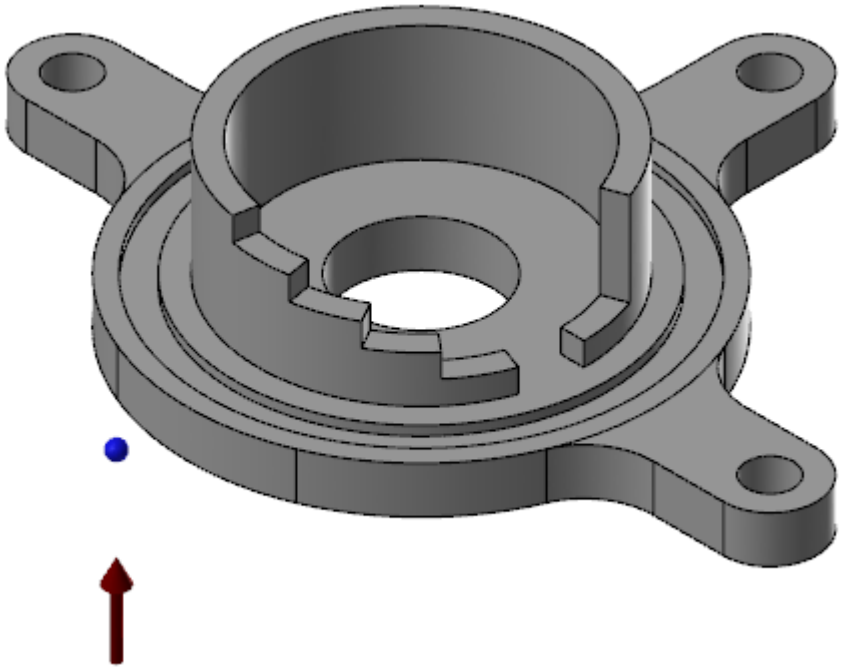


This is the foot you will remove from the part. The foot is highlighted and a red single-axis arrow control is placed on its face.

4. Click the red arrow, drag downward to remove the foot, and click again to set the change.



5. Your part should look like the picture below.

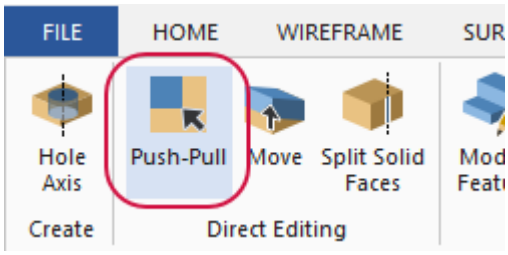


6. Click **OK** to finalize the operation.
7. Save the file as `FlangeDemo02-XXX`.

Exercise 3: Reducing the Foot Height

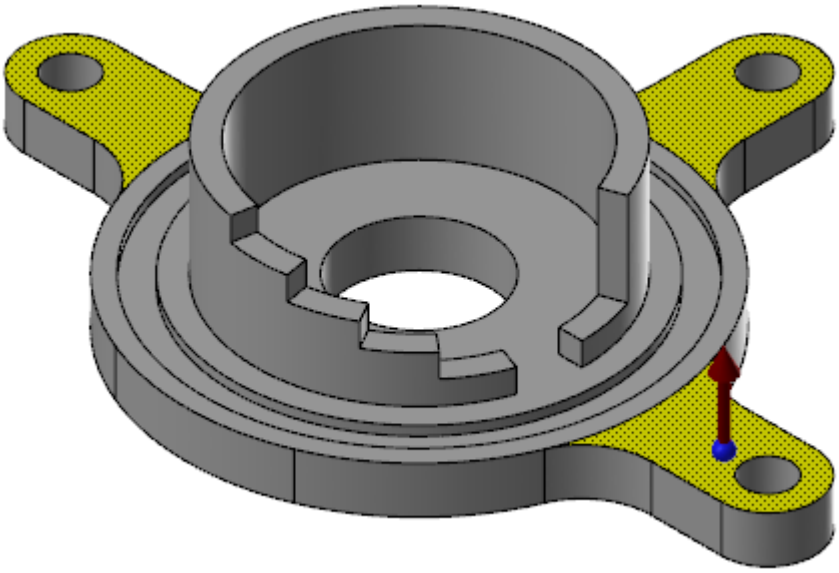
Now that you have the required three feet, you will reduce the thickness of the remaining feet, and center them on the part's outside rim. For this exercise, continue with the part you just saved, or load `FlangeDemo02`, which accompanies this tutorial.

1. Select **Push-Pull** on the **Model Prep** tab.

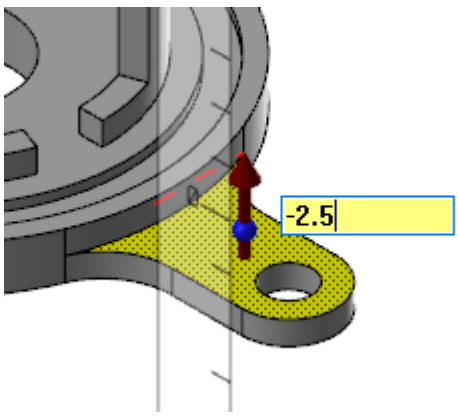


   The **Push-Pull** function panel opens.

2. Click the top face of each foot, and press [**Enter**].

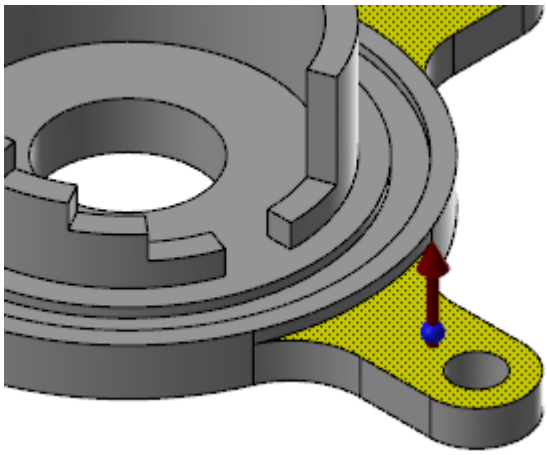


3. Click the red arrow, and enter **-2.5**.



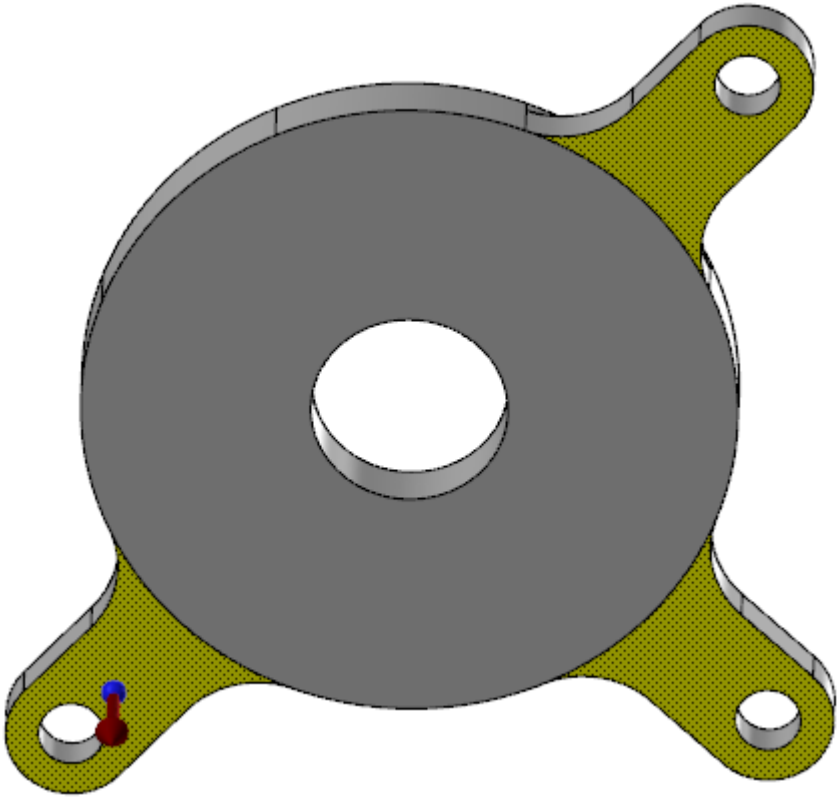
4. Press [**Enter**] twice.

   Push-Pull reduces the foot height by 2.5 mm.

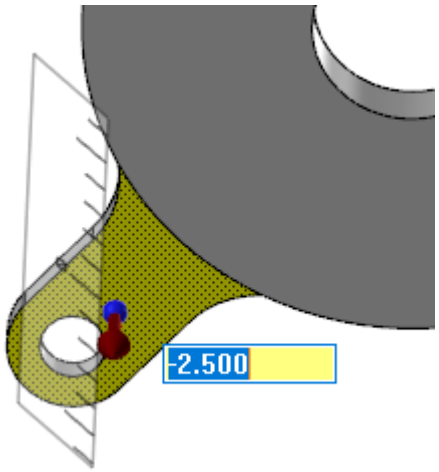


Note: The first time you press [**Enter**], Mastercam shows the result and gives you a chance to type another value. The second time, Mastercam accepts the entry, and closes the entry field.

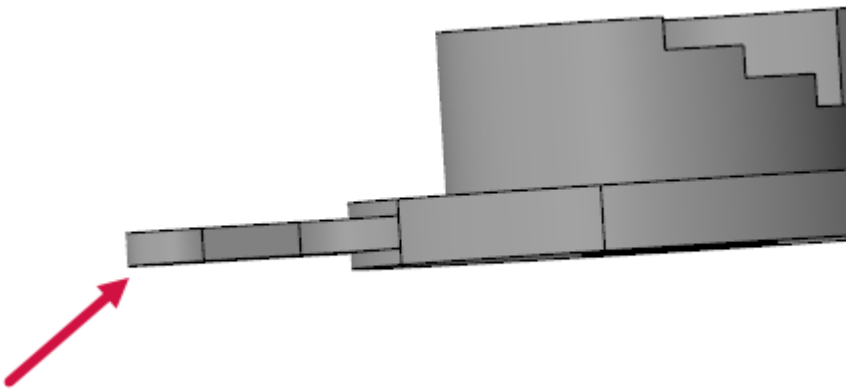
5. Click **OK and Create New Operation** to finalize your choices and begin a new **Push-Pull** operation.
6. Rotate the view to show the bottom of the part, and select the bottom faces of the three feet.



7. Click the red arrow, and enter **-2.5** into the box that appears.



8. Press [**Enter**] twice.
9. Click **OK** to finalize the operation.
10. Push-Pull reduces the foot height by 2.5 mm, leaving the feet centered on the rim, as you can see by rotating the part to view the edge.

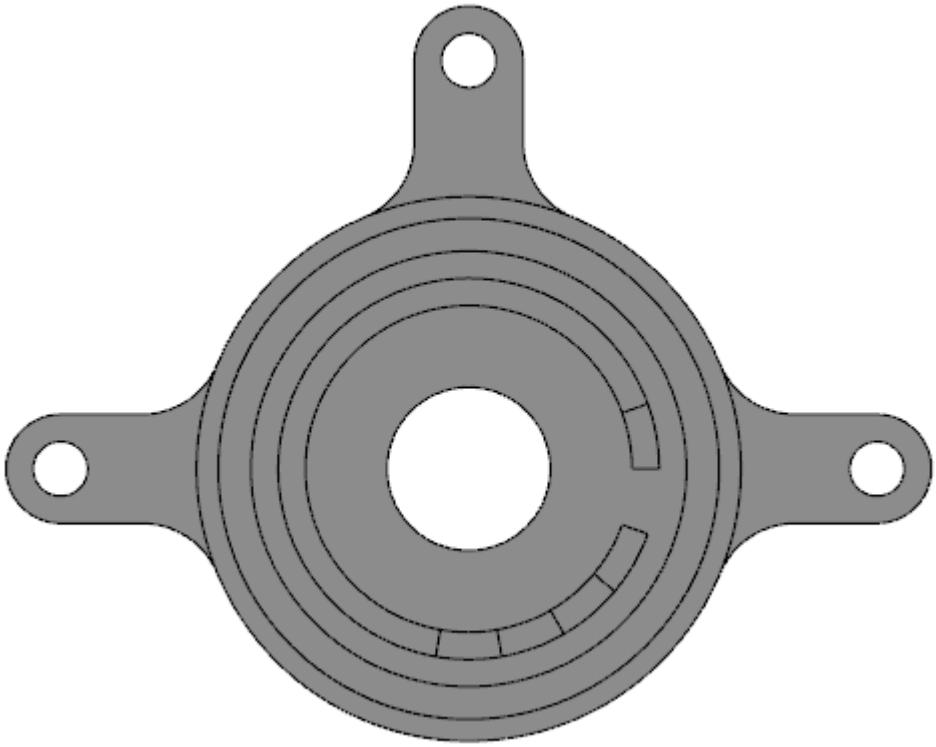


11. Save the file as `FlangeDemo03-XXX`.

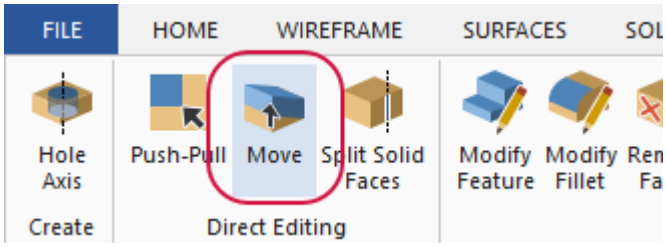
Exercise 4: Repositioning the Feet

The next step is to reposition the feet so they are equidistant from each other. For this exercise, continue with the part you just saved, or load `FlangeDemo03`, which accompanies this tutorial.

1. Place the part into top view.

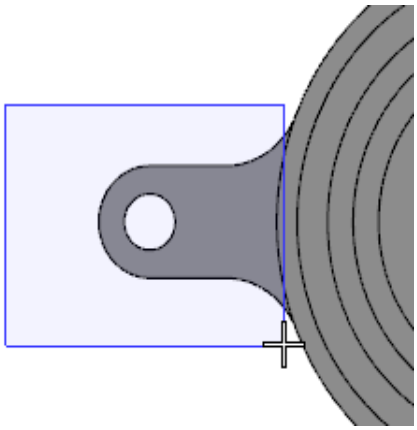


2. Select **Move** on the **Model Prep** tab.

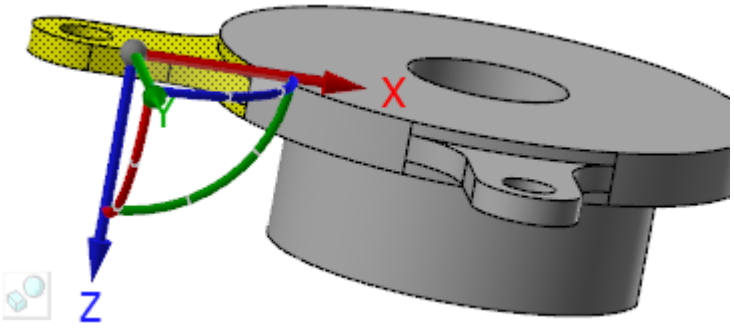


The **Move** function panel opens. The Move function moves (and sometimes resizes), rotates, or copies solid features or faces.

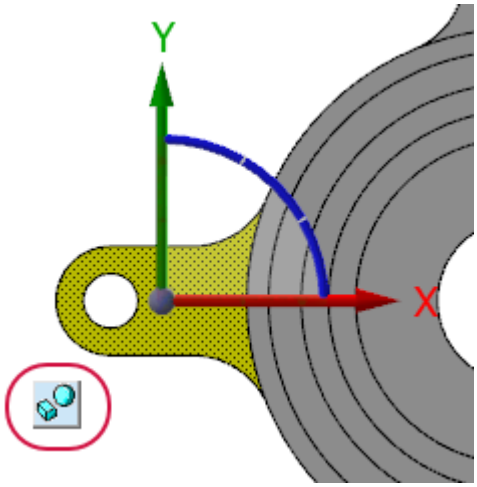
3. Use window selection to select all of the faces of the left-hand foot.



4. Rotate the view and check that you did not miss any faces.

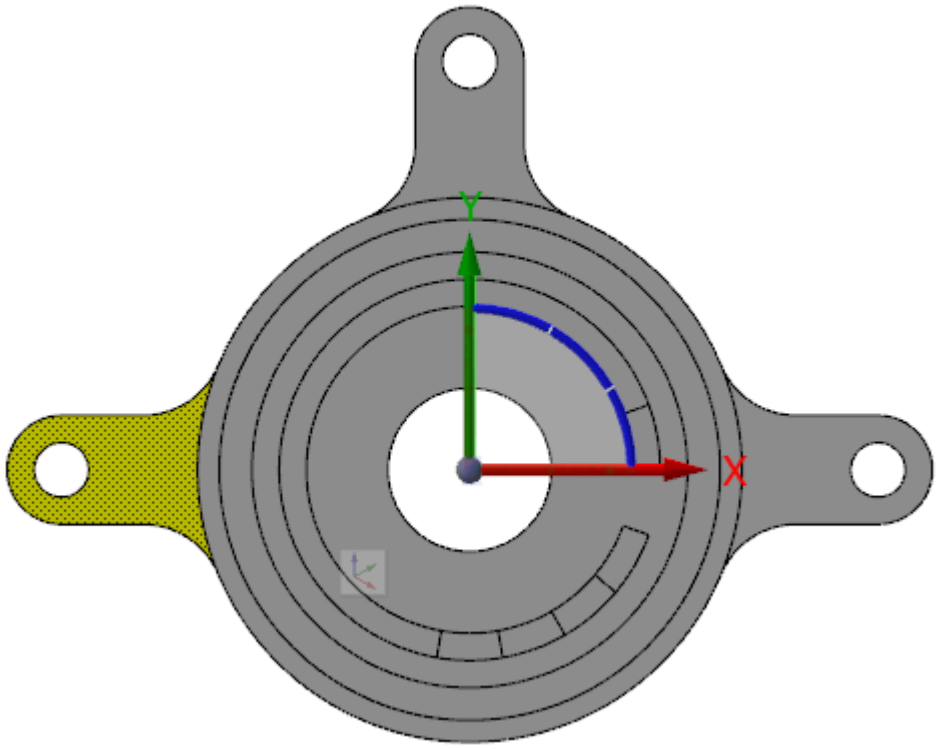


5. Return to top view after you have ensured that all of the faces have been selected.
6. Click the gnomon's button to change from geometry manipulation mode to gnomon manipulation mode.



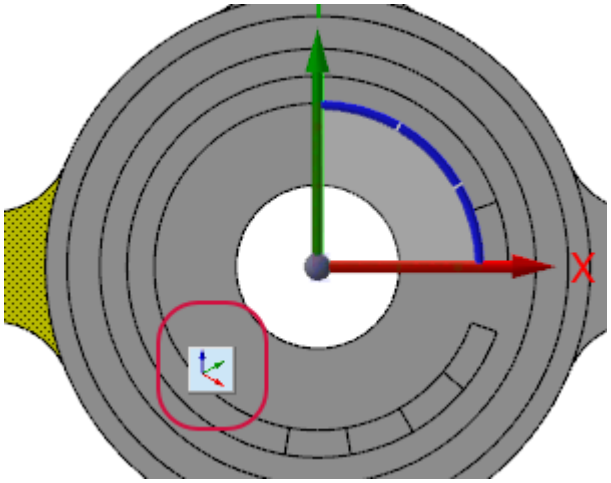
The gnomon's current location is the center of rotation for **Move**. The gnomon manipulation mode lets you specify a new center of rotation.

7. Click the ball at the gnomon's origin, and place the gnomon at the center of the part.



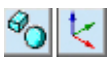
The part's center point is now the center of rotation.

8. Click the gnomon's button again, to change from gnomon manipulation mode back to geometry manipulation mode.

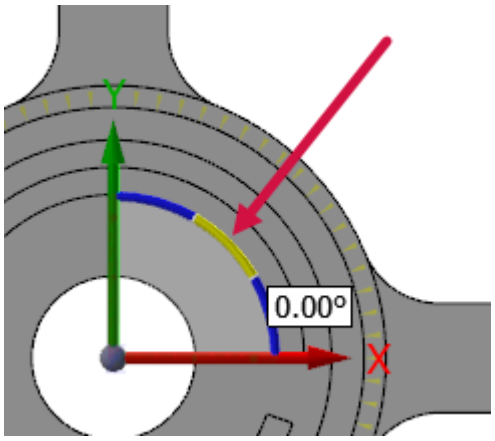


You can now use the gnomon as a tool for rotating the selected foot into a new position.

Note: The image shown on the gnomon's button indicates the gnomon's mode. In the pictures below, the left icon indicates geometry manipulation mode, and the right icon indicates gnomon manipulation mode.

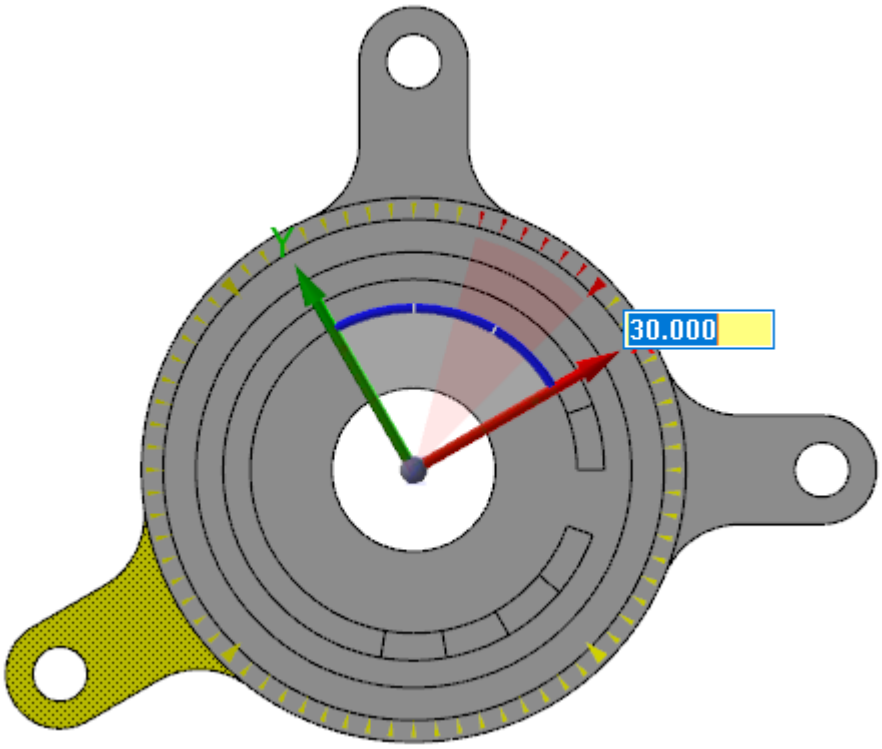


9. On the gnomon, click the center segment of the blue control.



This segment of the control activates 2D rotation around the Z axis.

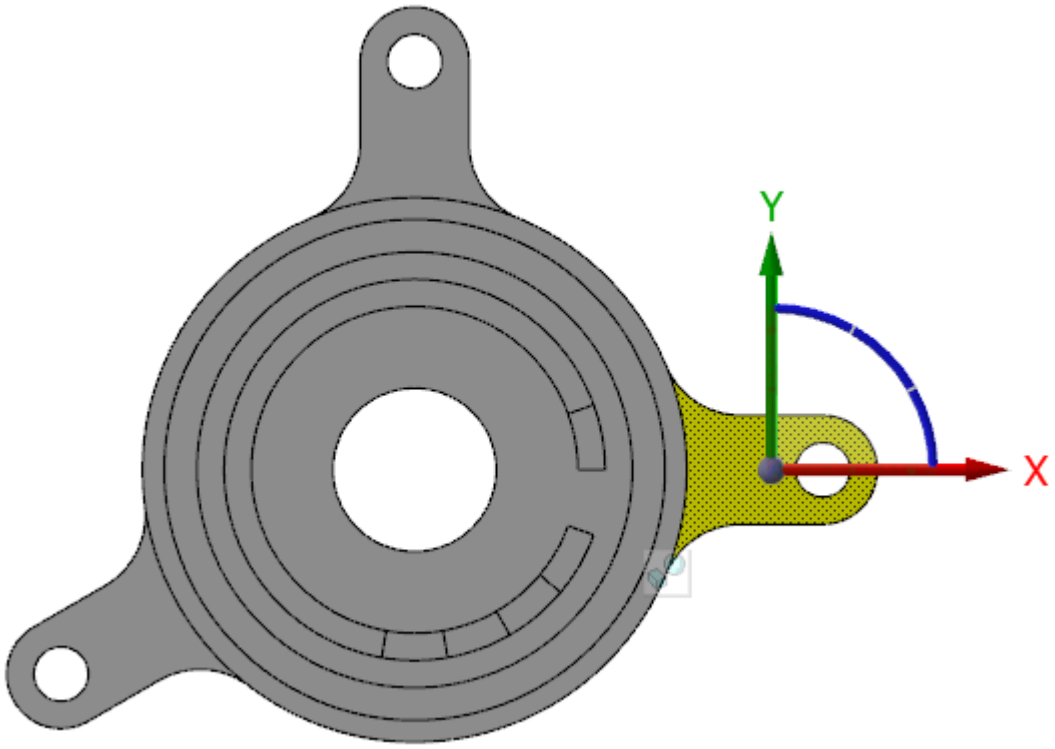
10. Type **30**, and press [**Enter**].



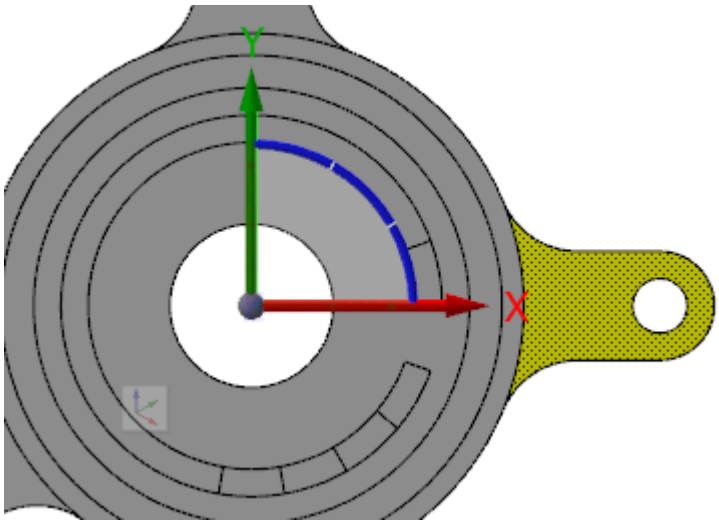
The selected foot rotates with the mouse movement, as shown in the picture below.

11. Click **OK and Create New Operation** to finalize your changes and begin a new **Move** operation.

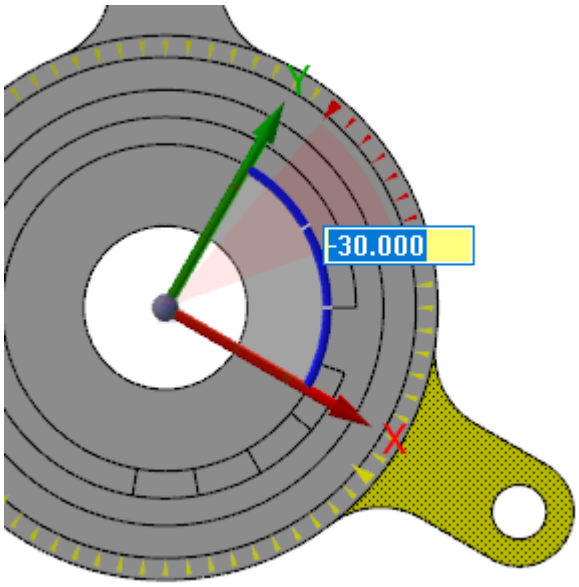
12. Window select the right-hand foot, as shown.



13. Switch to gnomon manipulation mode, and move the gnomon to the center of the part.

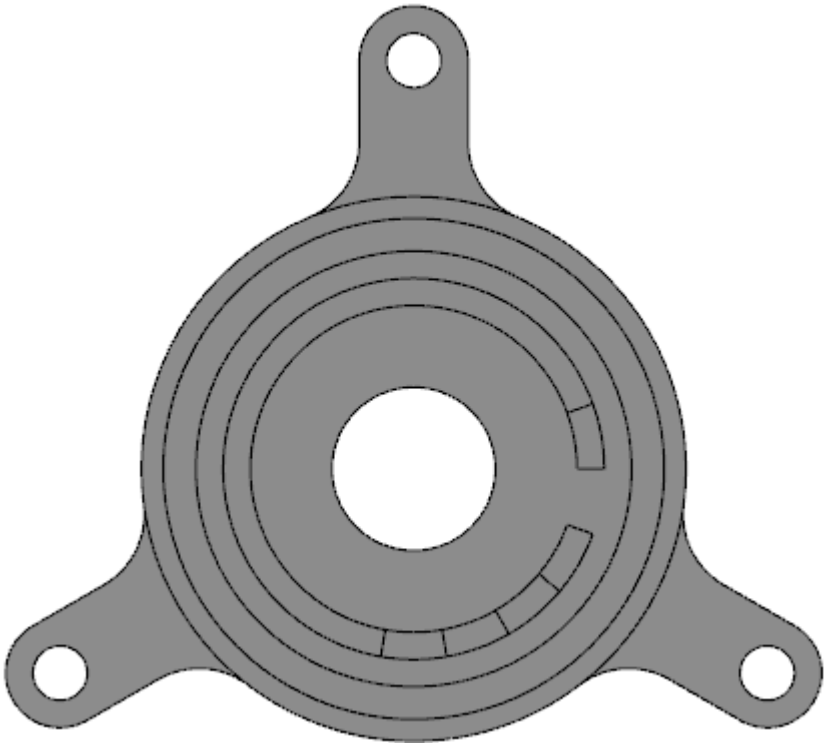


14. Switch back to geometry manipulation mode, and rotate the gnomon 30 degrees clockwise (-30 degrees).



Move rotates the foot into its correct position.

15. Click **OK** in the **Move** function panel.
16. Your part should now look like the picture below.

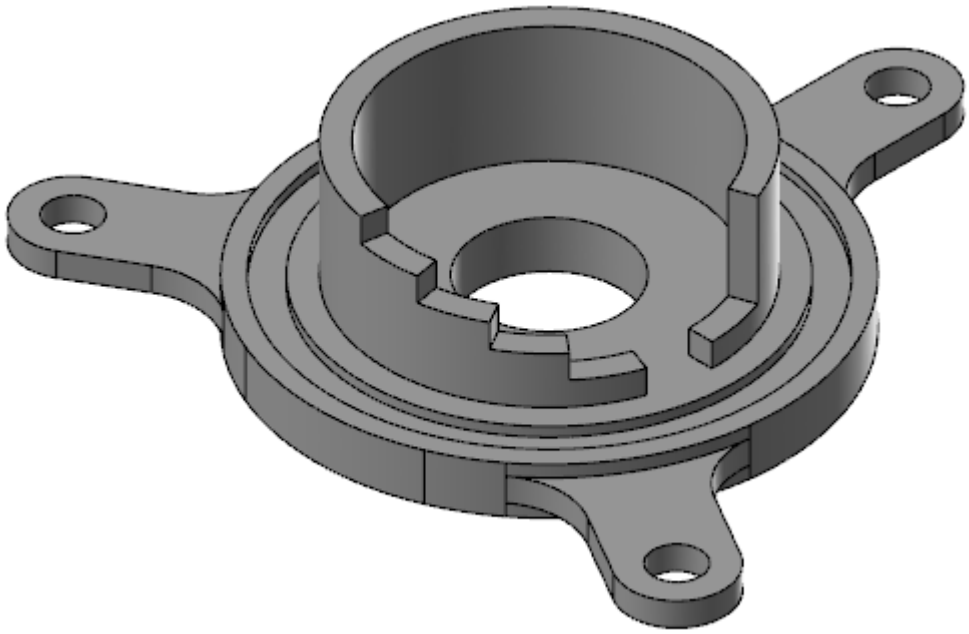


17. Save the file as `FlangeDemo04-XXX`.

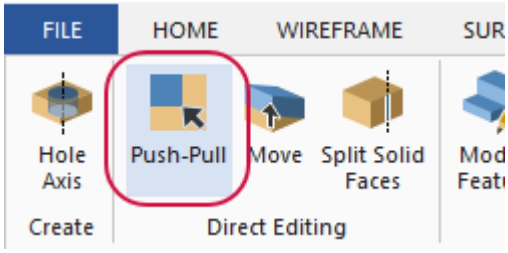
Exercise 5: Resizing the Foot Holes

Next, you will re-size the hole in each foot, increasing them from 10 mm to 12 mm. For this exercise, continue with the part you just saved, or load `FlangeDemo04`, which accompanies this tutorial.

1. Change the part's view to isometric.

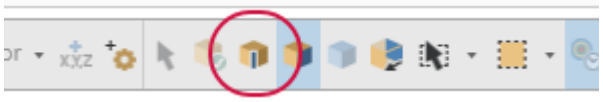


2. Select **Push-Pull** on the **Model Prep** tab.



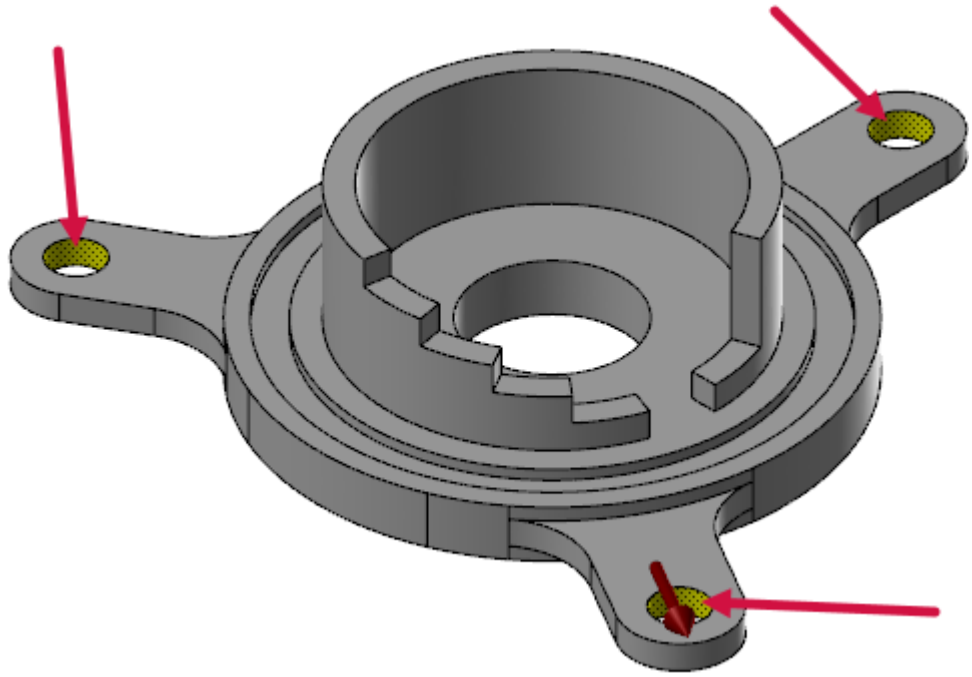
The **Push-Pull** function panel opens.

3. On the Selection Bar, turn off the **Edge Selection** filter. Only **Face Selection** should be on.

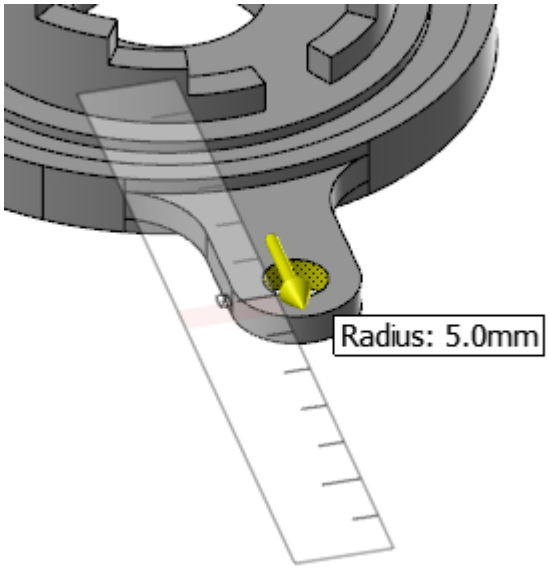


You can now more easily select the holes without accidentally selecting the hole edges.

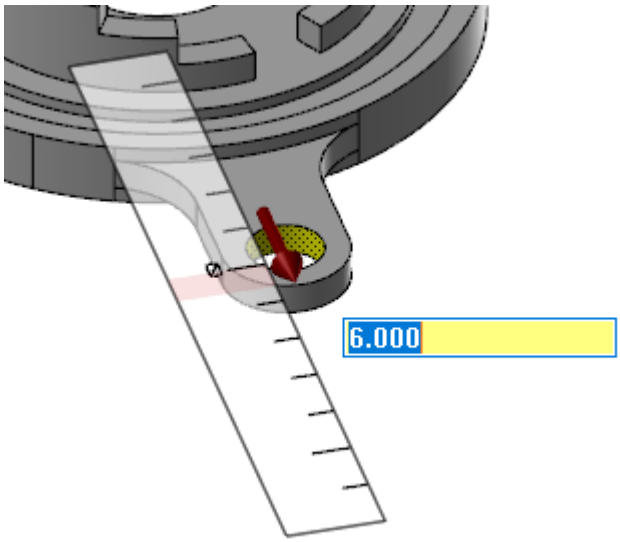
4. Select the three foot holes shown below.



5. Click the red arrow to display the ruler control.

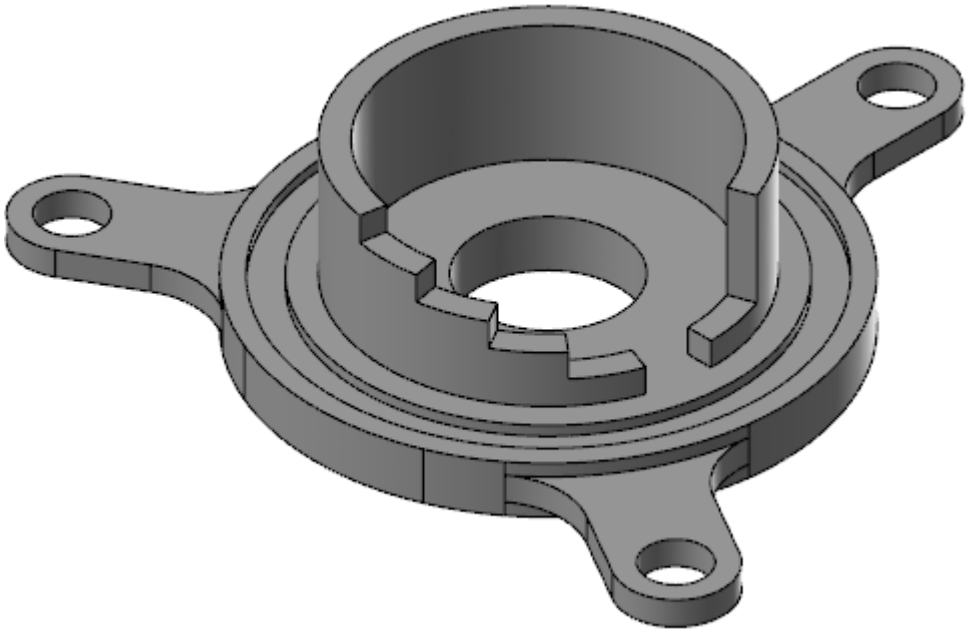


6. Type **6.0** and press [**Enter**].



Note: When you move your mouse pointer outside of the ruler, you can select any value. When you move your mouse pointer over the ruler, the displayed value snaps to the ruler increments.

7. Press [**Enter**] again to finalize the changes.
8. Click **OK** in the function panel.
9. The holes are now 12 mm in diameter, as shown in the picture below.

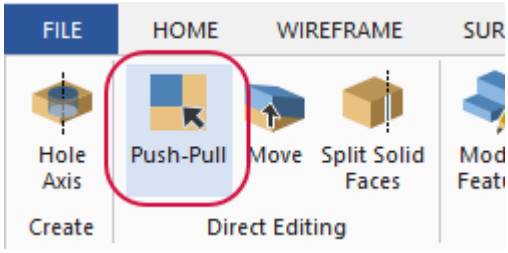


10. Save the file as `FlangeDemo05-XXX`.

Exercise 6: Creating the Counterbore

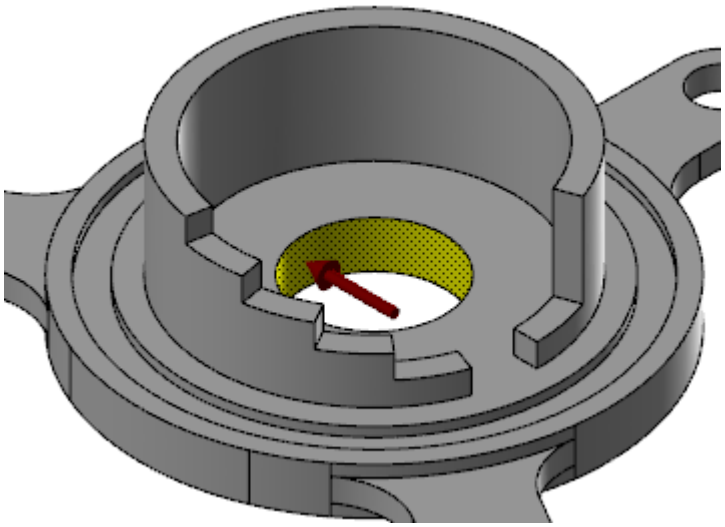
The specifications state that your part's center hole be reduced to 20 mm. You must also add a counter bore. For this exercise, continue with the part you just saved, or load `FlangeDemo05`, which accompanies this tutorial.

1. Select **Push-Pull**.

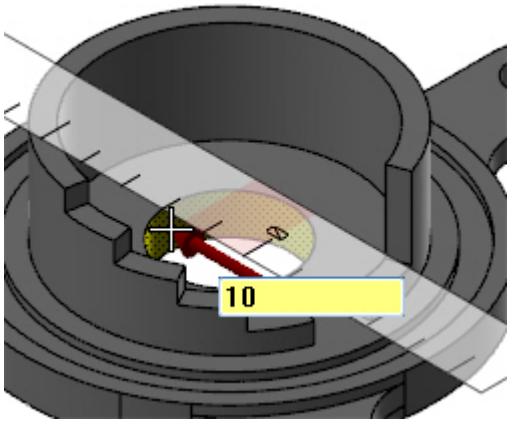


   The **Push-Pull** function panel opens.

2. Select the center hole.



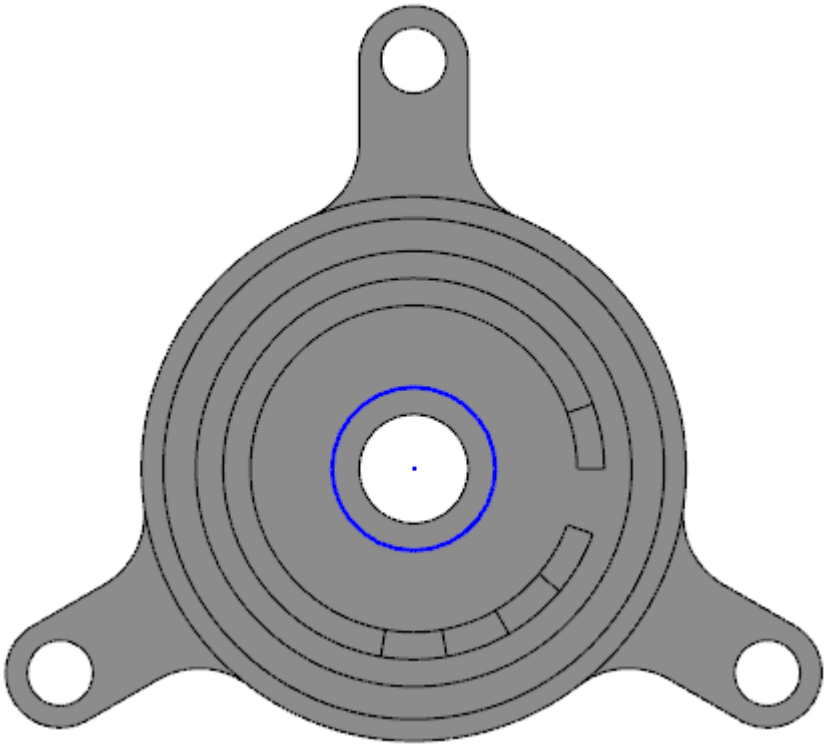
3. Click the red arrow, type **10**, and press [**Enter**] twice.



   Push-Pull changes the hole's diameter to 20 mm.

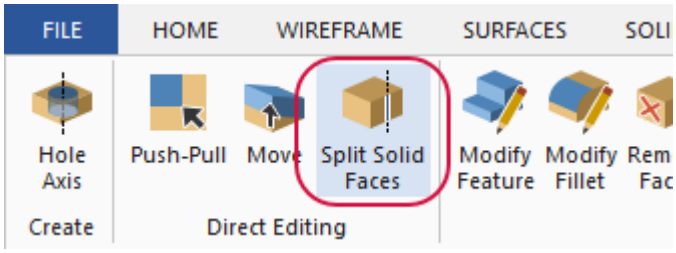
4. Click **OK** in the **Push-Pull** function panel.
5. Set the part to **Top (WCS)** view.

6. Create a **30** mm diameter circle at the center of the part by using **Circle Center Point**.



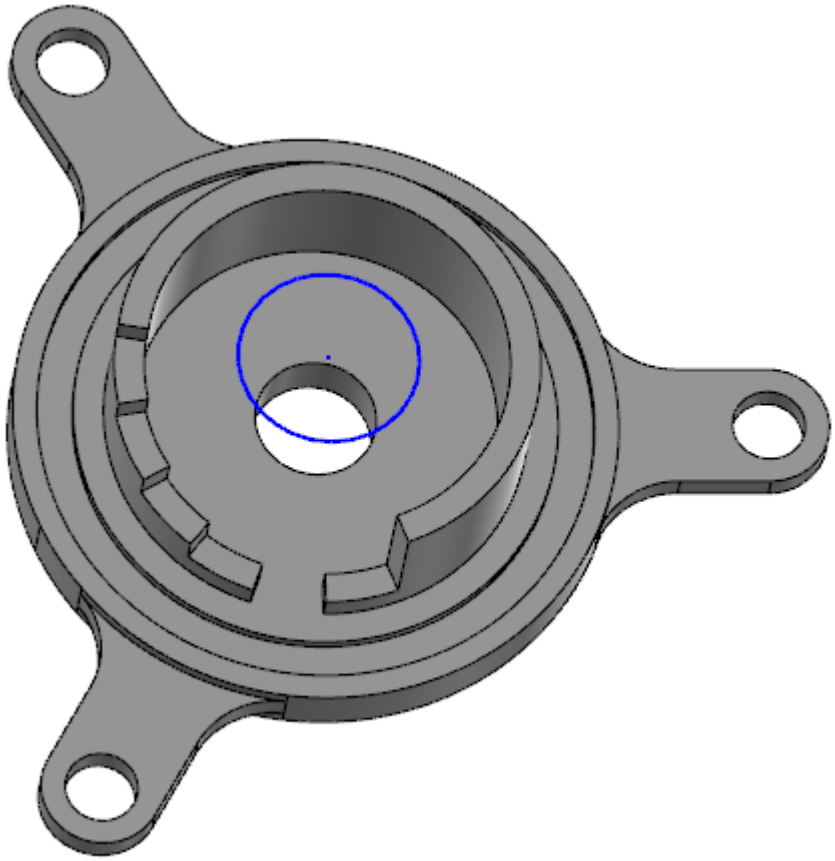
You will use this circle to split the part's inner face at the diameter needed for the counter bore.

7. Select **Split Solid Faces** from the **Model Prep** tab.



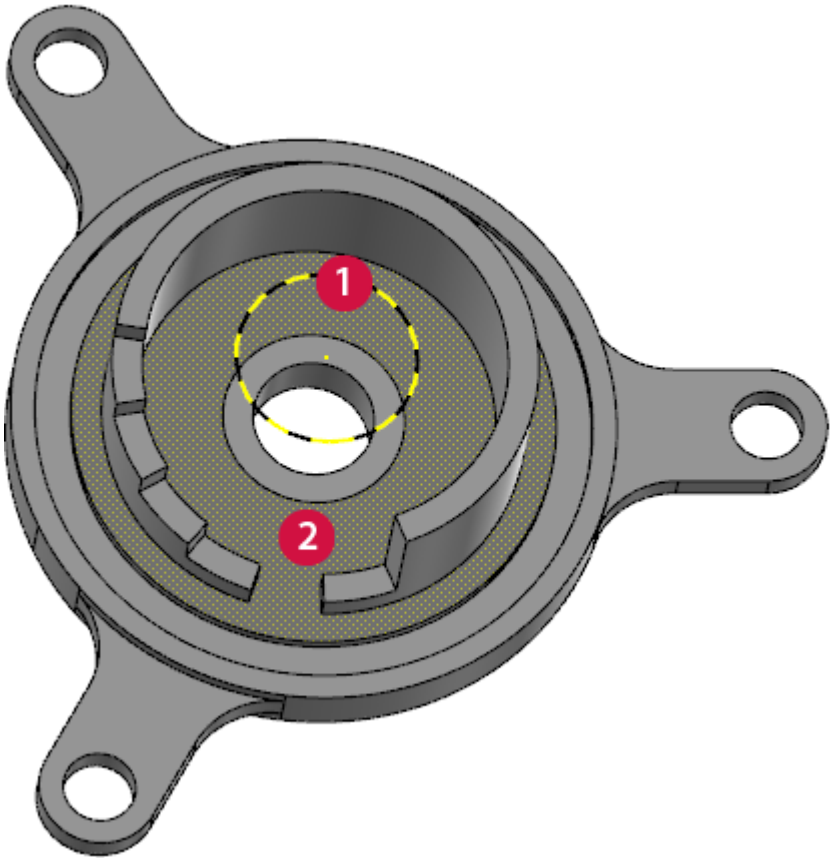
The **Split Solid Face** function panel opens.

8. Rotate the part's view until it looks similar to the picture.

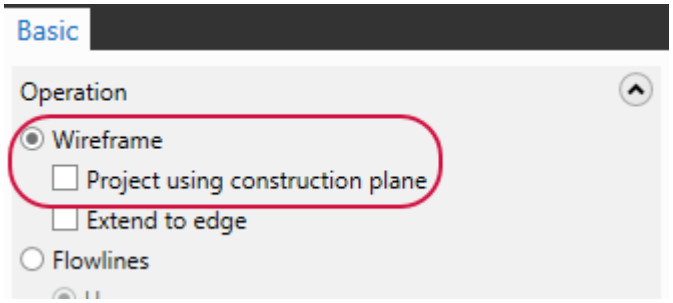


If the circle is laying completely on top of the solid, you would have to turn off the face filter in general selection to pick the circle. When the circle is positioned over empty space it can be selected without changing the filter.

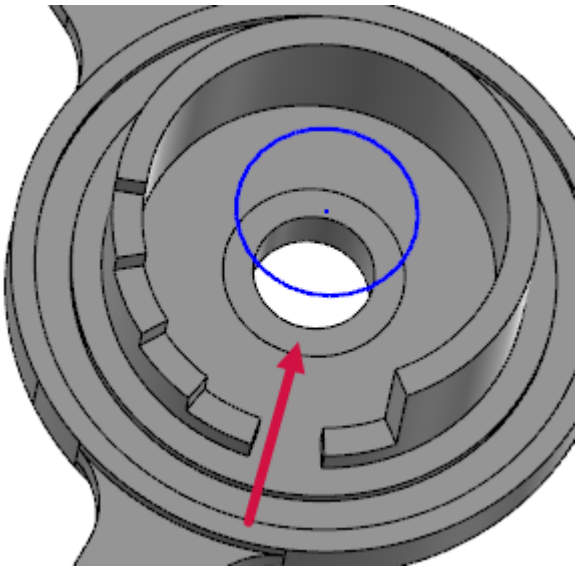
9. Click the circle and then the part's inner face, as shown below.



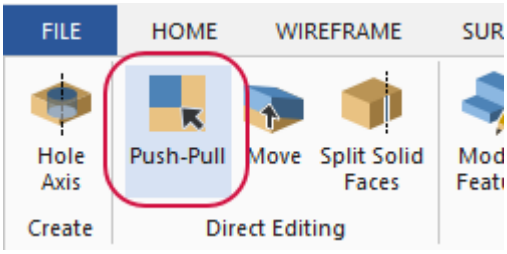
10. Ensure that **Wireframe** is selected, and **Project using construction plane** is deselected.



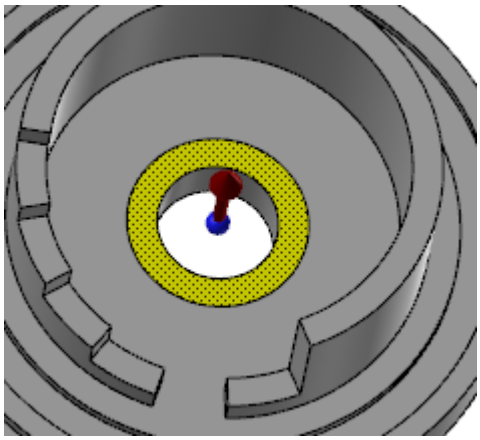
11. Click **OK** in the function panel to split the solid face.



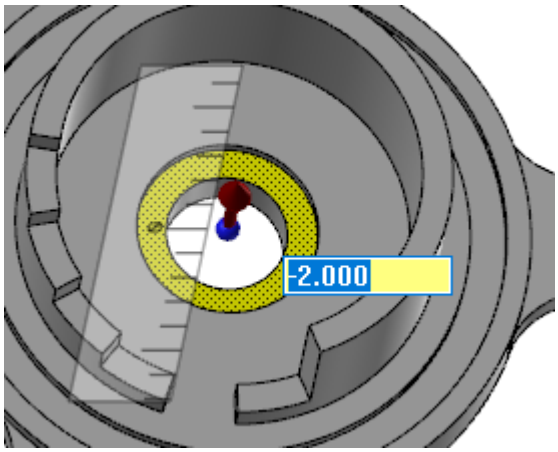
12. Delete the wireframe circle, as it is no longer needed.
13. Select **Push-Pull**.



14. Click the portion of the newly split face shown in the picture.



15. Click the red arrow, type **-2**, and then press **[Enter]** twice.



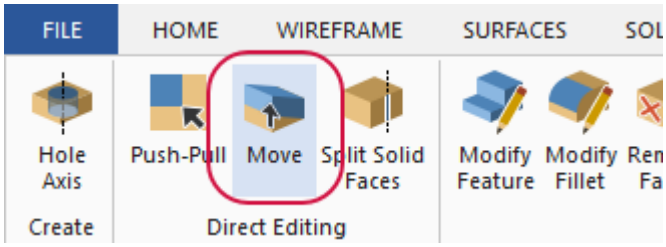
Mastercam creates the counter bore.

16. In the function panel, click **OK**.
17. Save the file as `FlangeDemo06-XXX`.

Exercise 7: Aligning the Vertical Wall

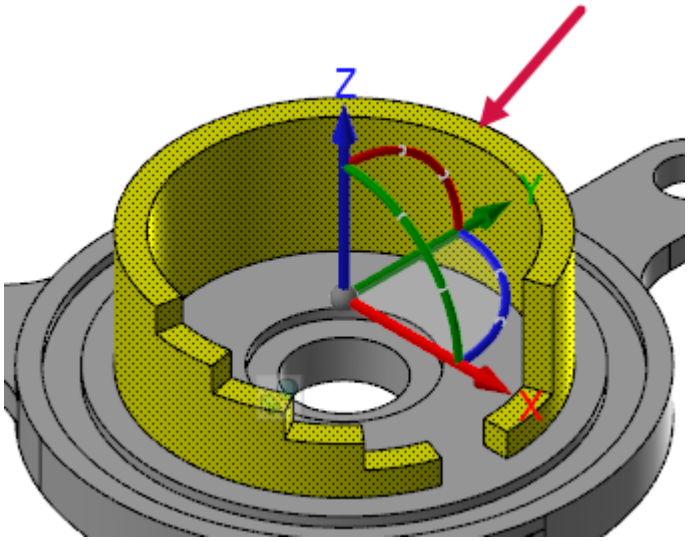
Next you will rotate the vertical wall so the opening aligns with one of the feet. For this exercise, continue with the part you just saved, or load `FlangeDemo06`, which accompanies this tutorial.

1. Select **Move** from the **Model Prep** tab.

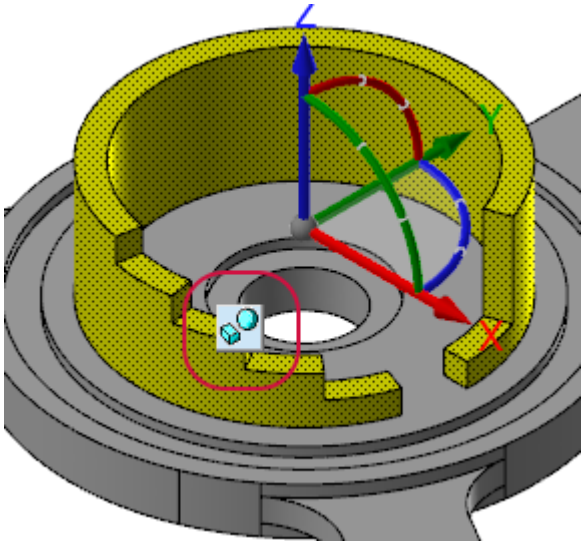


The **Move** function panel displays.

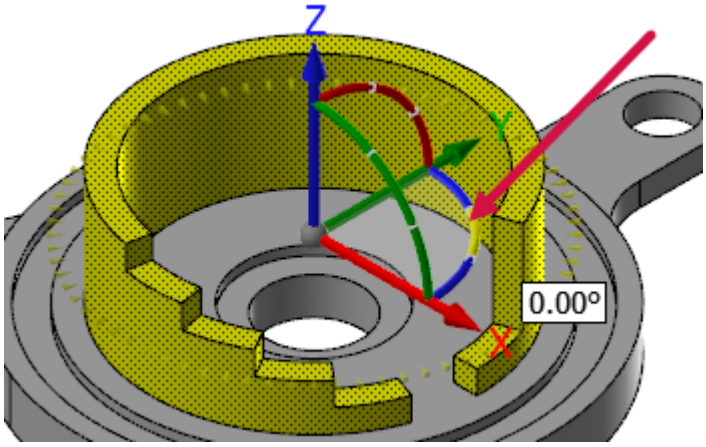
2. Double-click the top face of the vertical wall, as shown. The entire feature is selected.



3. Click the gnomon button to switch to gnomon manipulation mode.

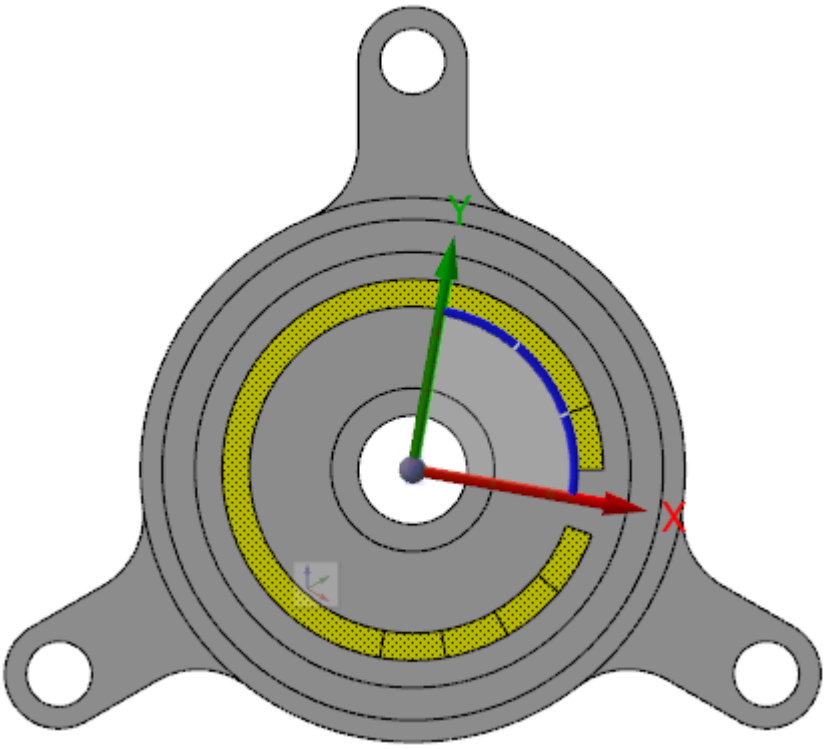


4. Click the center segment of the gnomon's blue arc.



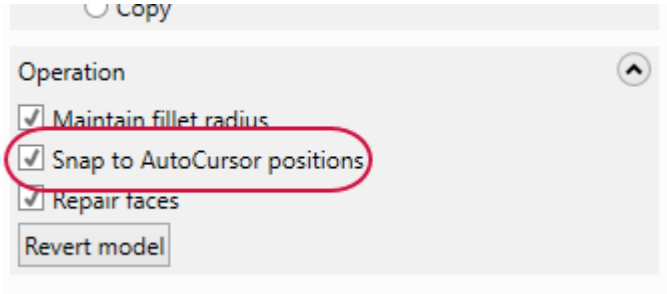
You have now set the gnomon for 2D rotation around the Z axis.

5. Type **-10**, and press [**Enter**] twice.



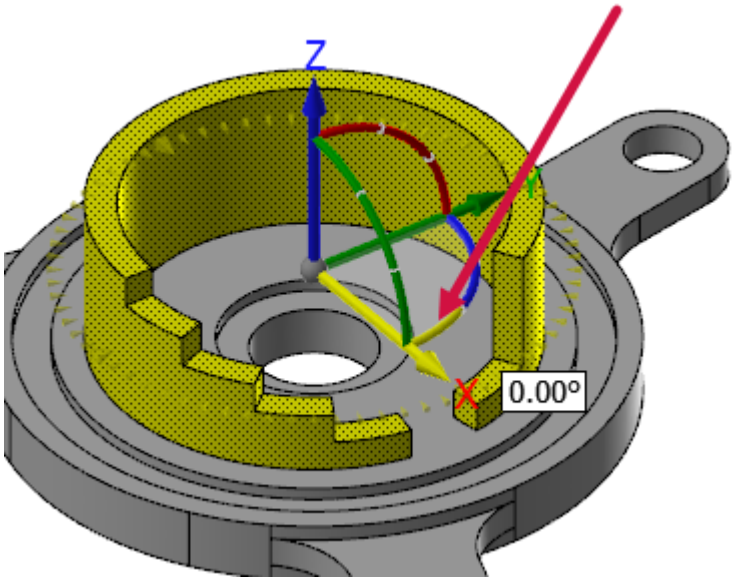
The gnomon's X axis rotates to center on the opening in the vertical wall, as shown (in top view).

6. Click the gnomon's button to change back to geometry manipulation mode.
7. Ensure that the **Snap to AutoCursor positions** option is selected in the function panel.

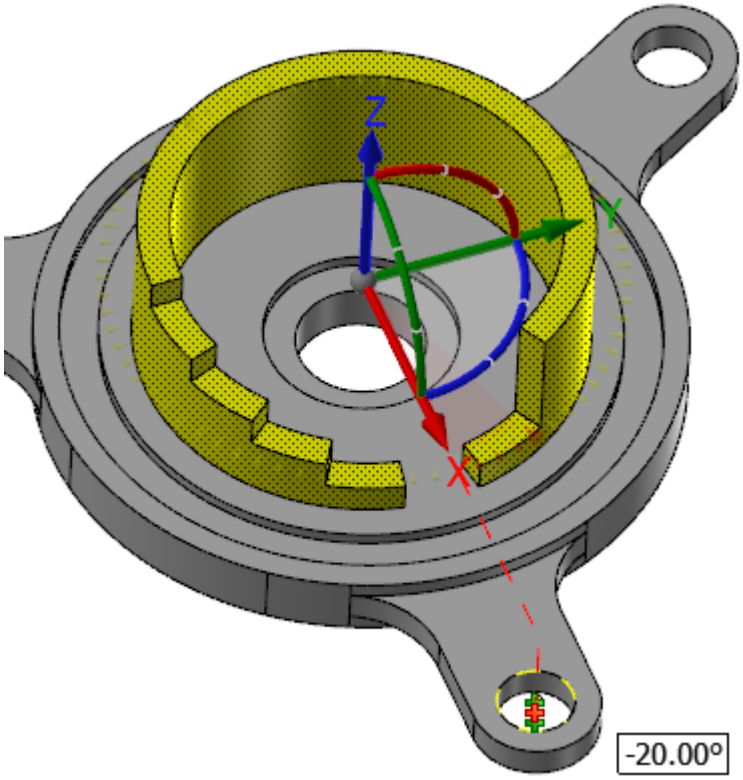


You are now in 3D rotation mode.

8. Click the right segment of the gnomon's blue arc.



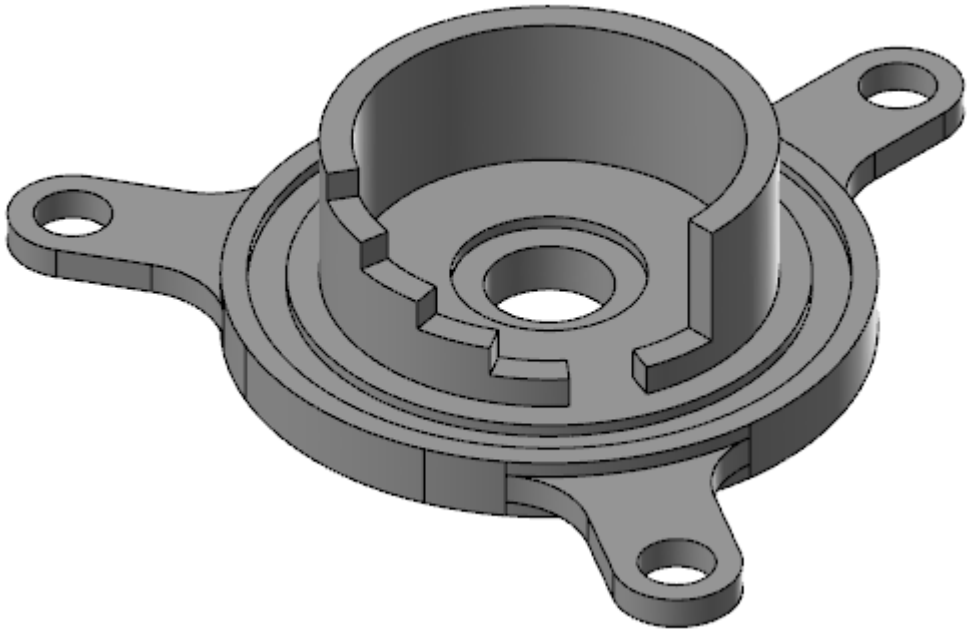
9. Move your mouse cursor to the center of the hole in the lower foot, as shown.



Because you have **Snap to AutoCursor positions** on, you can easily snap the mouse to the hole's exact center, which should be an angle of **-20** degrees.

10. Click to set the angle, and then click **OK** in the function panel.

11. The opening in the wall is now aligned with the hole in the foot, as shown.

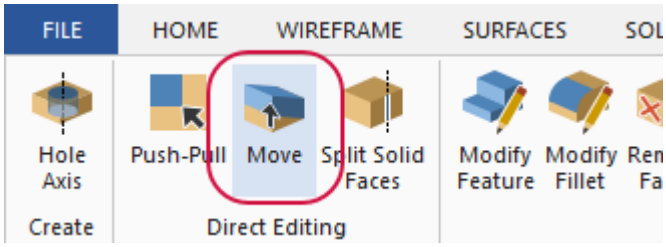


12. Save the file as `FlangeDemo07-XXX`.

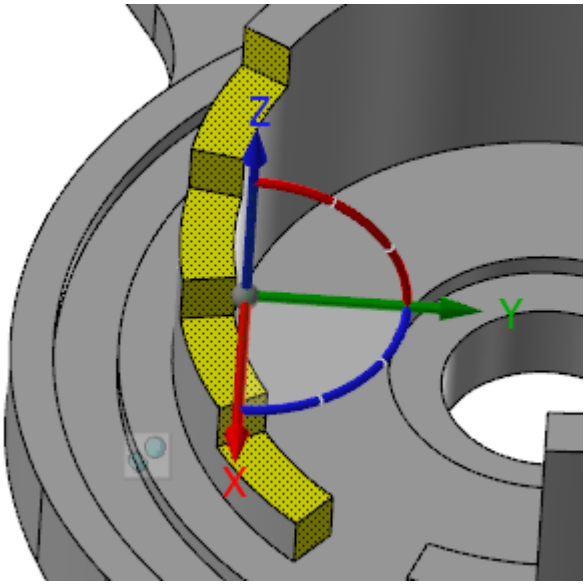
Exercise 8: Expanding the Wall Opening

According to the part's new specifications, you must expand the opening in the wall from 20 degrees to 45 degrees. For this exercise, continue with the part you just saved, or load `FlangeDemo07`, which accompanies this tutorial.

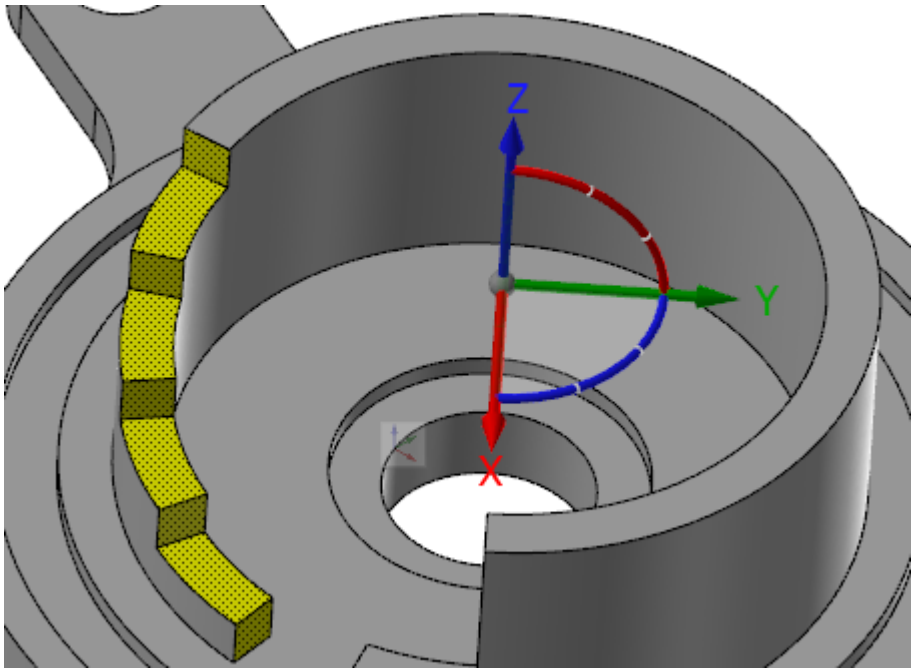
1. Select **Move** on the **Model Prep** tab.



2. Select all nine faces of the step-like portion of the wall.

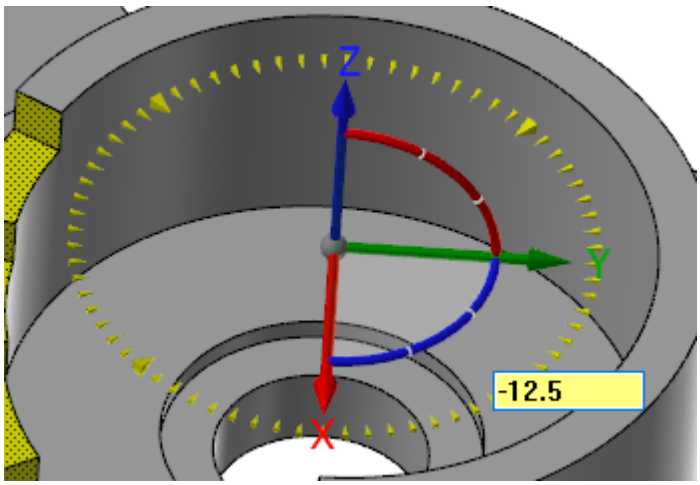


3. Click the gnomon's button to select gnomon manipulation mode, and place the gnomon at the center of the part.



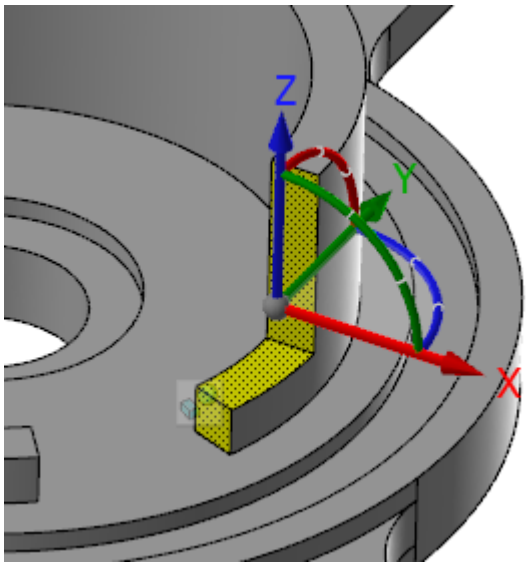
The gnomon is now set to the center of rotation.

4. Switch back to geometry manipulation mode, and rotate the geometry **-12.5** degrees.

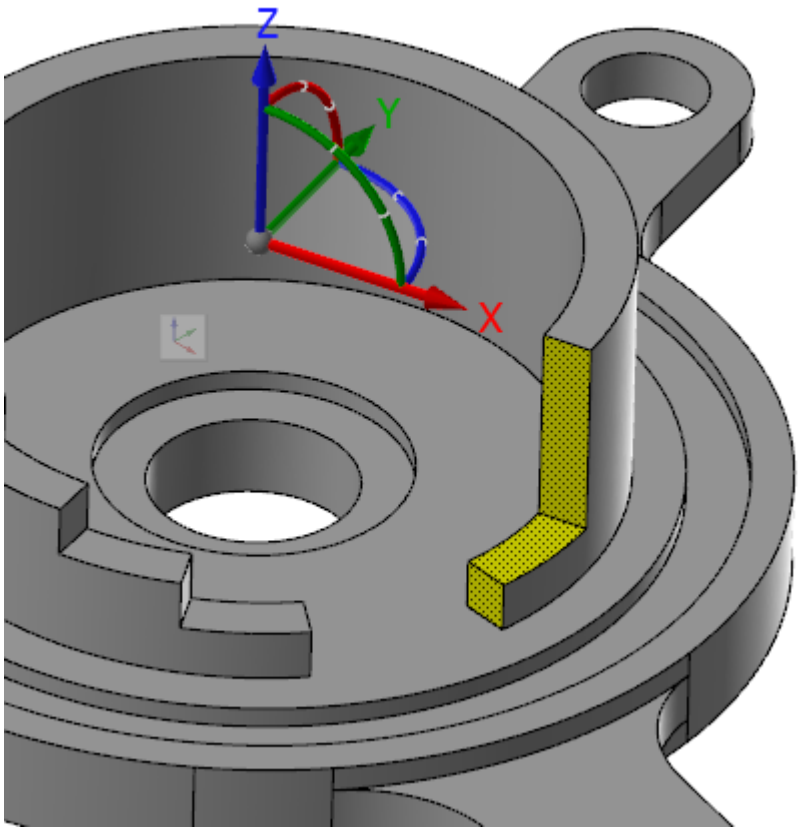


5. In the function panel, click **OK and Create New Operation**.

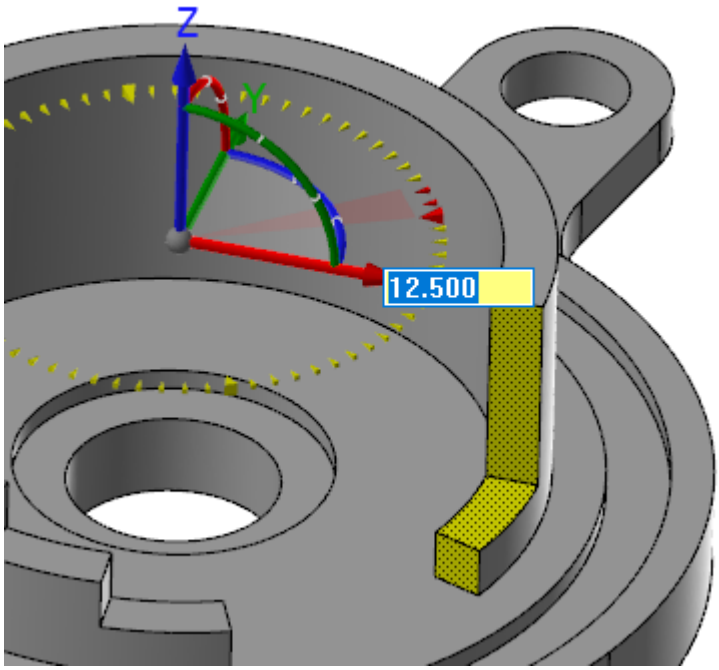
6. Select all three faces of the opposite side of the wall opening.



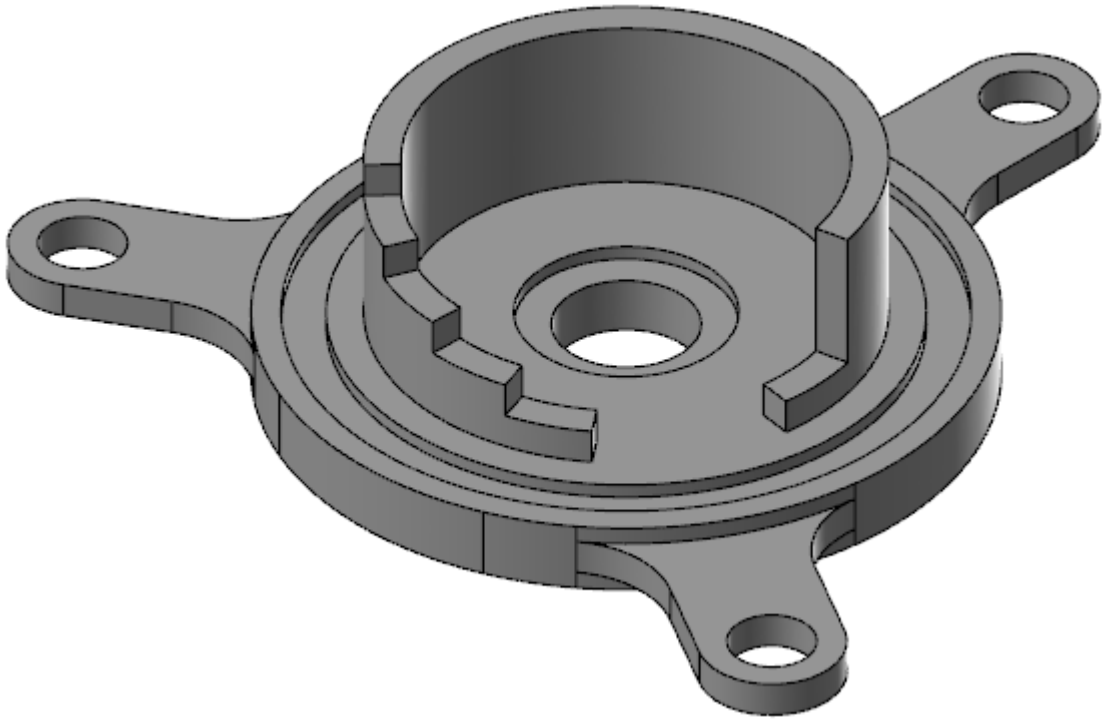
7. Click the gnomon's button to select gnomon manipulation mode, and place the gnomon at the center of the part.



8. Switch back to geometry manipulation mode, rotate the selected geometry **12.5** degrees.



9. Click **OK** in the **Move** function panel.



By rotating each side of the opening by 12.5 degrees, you added 25 degrees to the opening, while maintaining its centering on the foot's hole.

10. Save the file as `FlangeDemo08-XXX`.

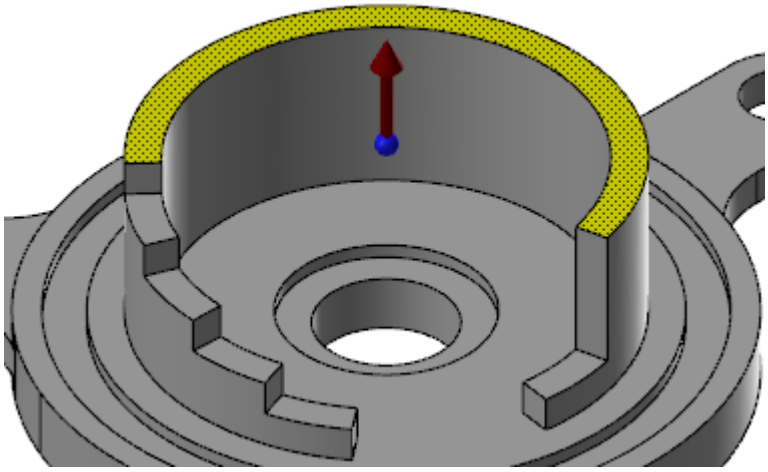
Exercise 9: Increasing the Wall Height

In this exercise, you increase the height of the vertical wall. Continue with the part you just saved, or load `FlangeDemo08`, which accompanies this tutorial.

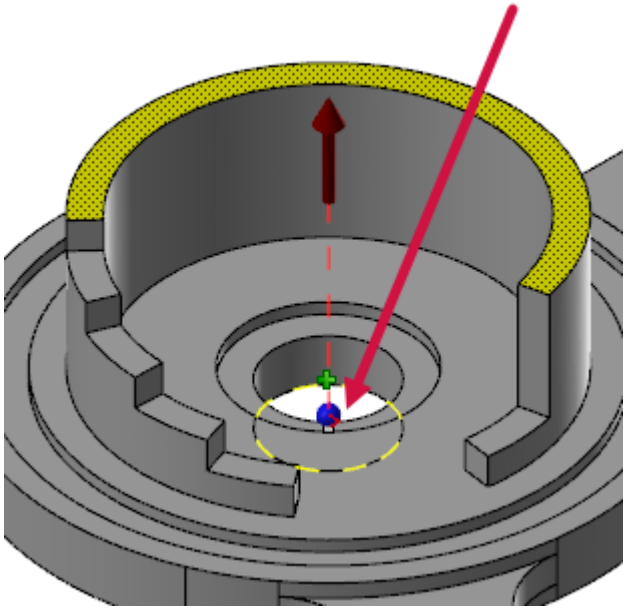
1. Select **Push-Pull** on the **Model Prep** tab.

   The **Push-Pull** function panel opens.

2. Select the face shown below.

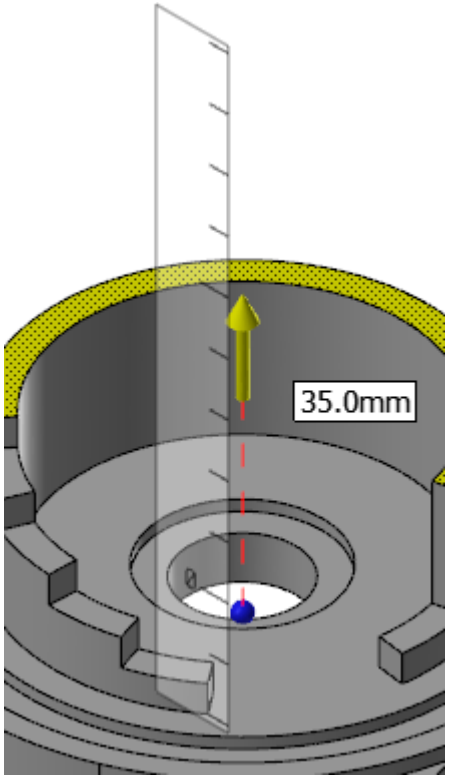


3. Drag the ball on the bottom of the arrow to the bottom of the part, as shown in the picture below.

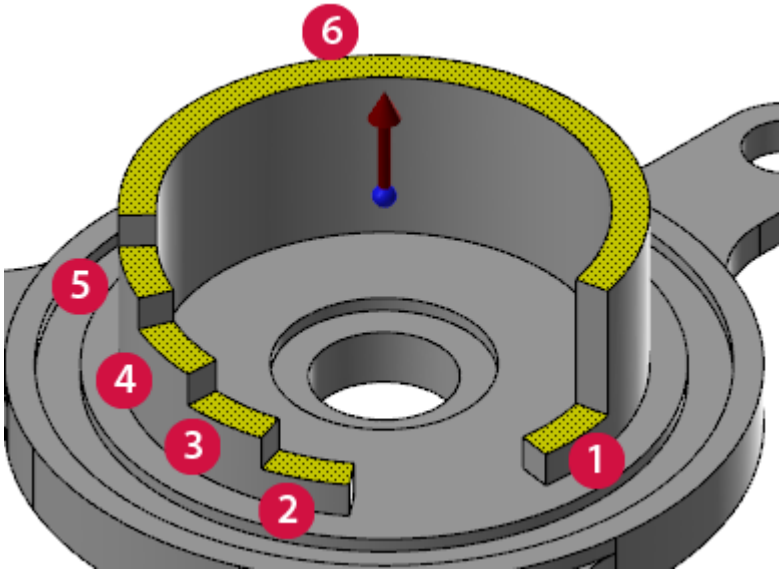


   The reference-point ball marks the zero point from which the control's setting is measured.

4. Place the mouse pointer over the arrow. You see that the part's height is 35 mm as measured from the reference-point ball you placed at the bottom of the part.

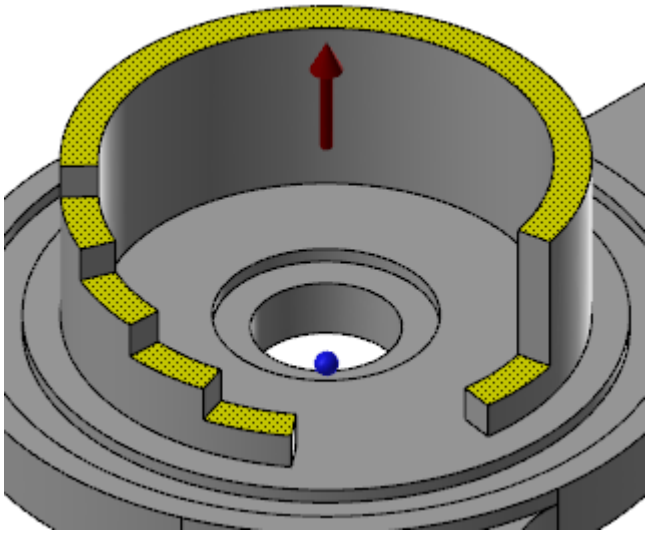


5. Select all of the wall's six top faces, *selecting the largest one last*, which places the arrow in the correct orientation.



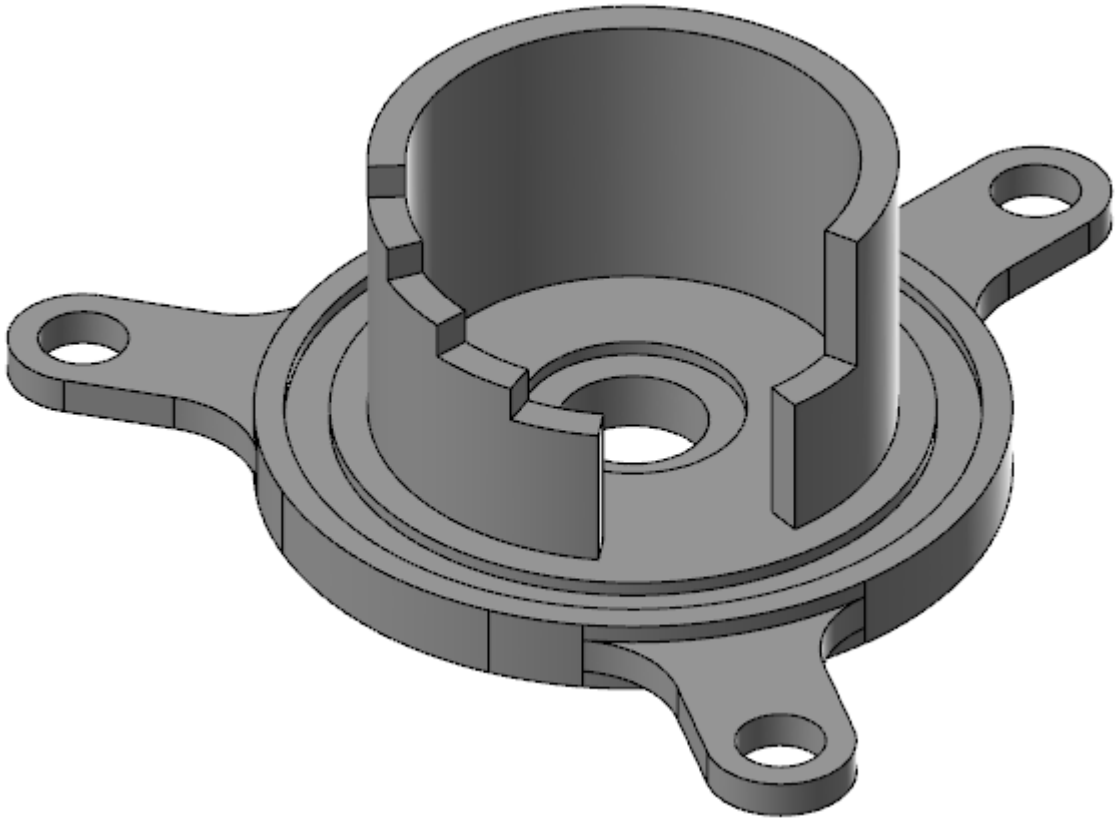
These are the faces that you will move upward to increase the wall's height.

6. Drag the ball back to the bottom.



You can change the reference point at any time. Just click the ball and move it as needed, including snapping it to other geometry.

7. Click the red arrow, type **50** followed by [**Enter**] twice.
8. Click **OK** in the function panel.
9. **Push-Pull** changes the wall's height to **50** mm, measured from the bottom of the part.



10. Save the file as `FlangeDemo09-XXX`.

Exercise 10: Adding the Fillet

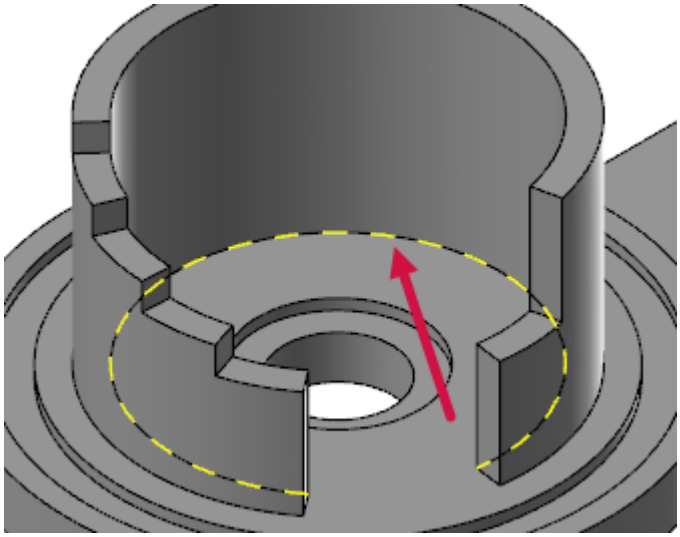
In this exercise, you add a fillet to the part. Continue with the part you just saved, or load `FlangeDemo09`, which accompanies this tutorial.

1. Select **Push-Pull** from the **Model Prep** tab.

   The **Push-Pull** function panel opens.

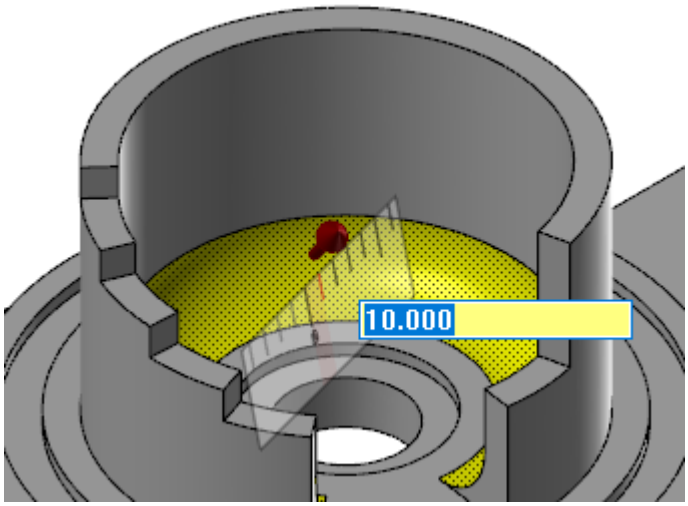
2. In the Selection Bar, turn off **Face Selection**, and turn on **Edge Selection**.

3. Select the wall's inside edge, as shown in the image below.

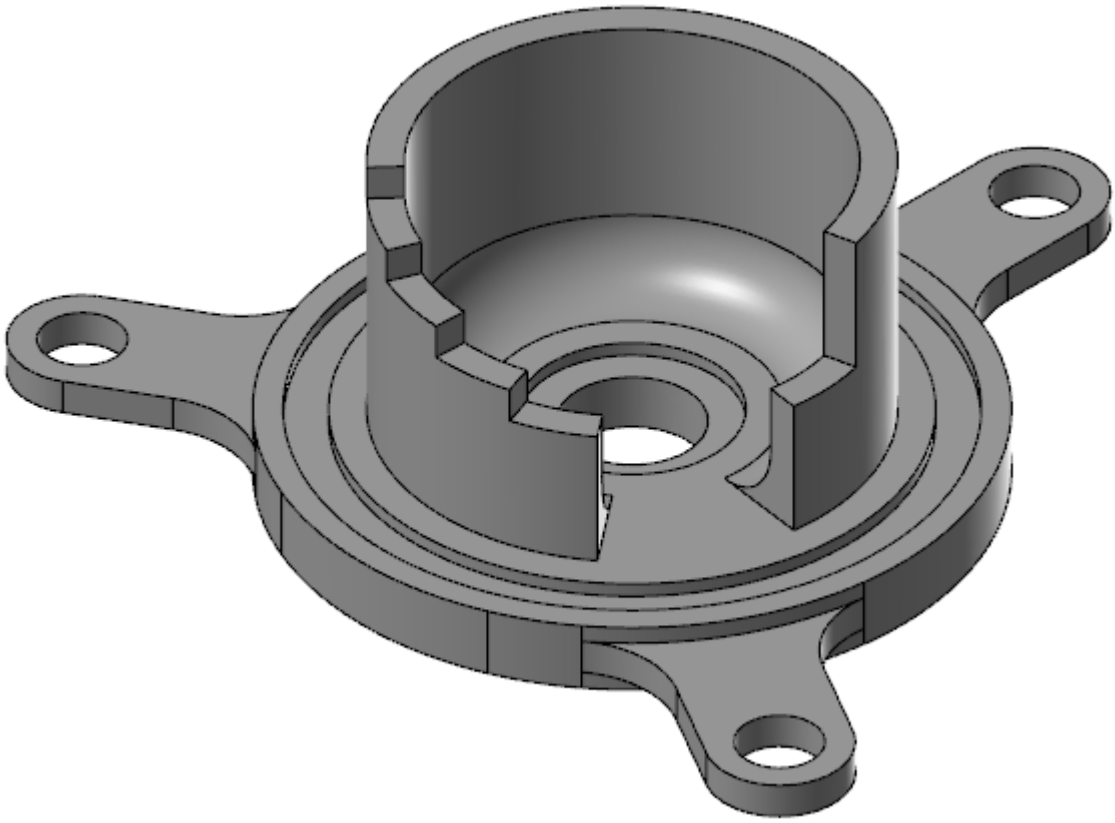


   This is where you will create the fillet.

4. Click the red arrow, type **10.0**, and press [**Enter**] twice.



5. In the function panel, click **OK** to finalize the change.



6. Save the file as `FlangeDemo10-XXX`.

You have now finished modifying the flange part to meet the new specifications.

CHAPTER 6

CHALLENGE PARTS

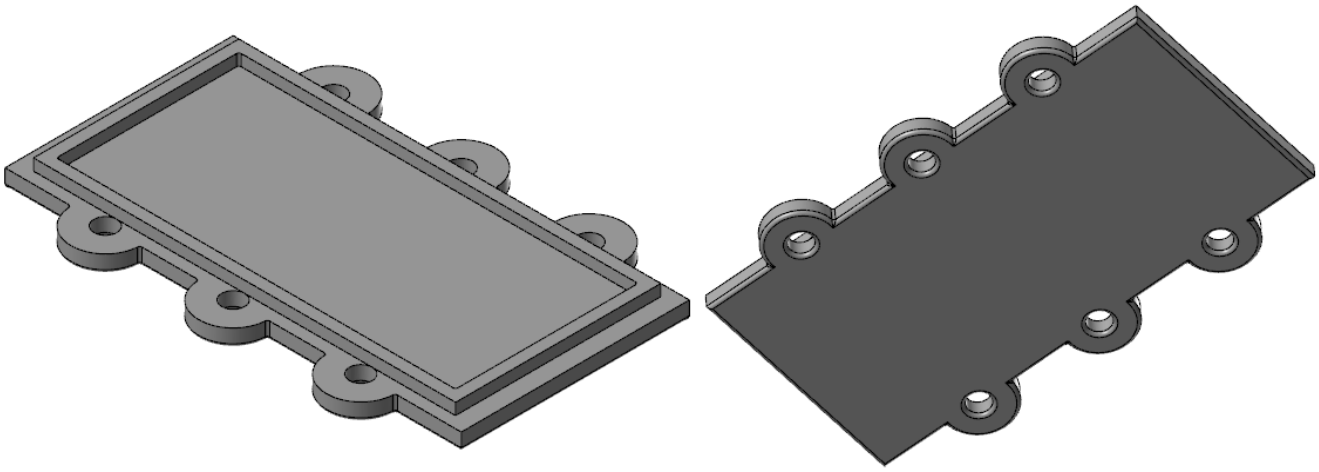
The following two exercises are challenge exercises meant to test your skills and knowledge for both Mastercam Solids and Model Prep functions. Follow the directions provided to re-create the part file shown. If you have trouble with any step, refer to the accompanying videos.

Goals

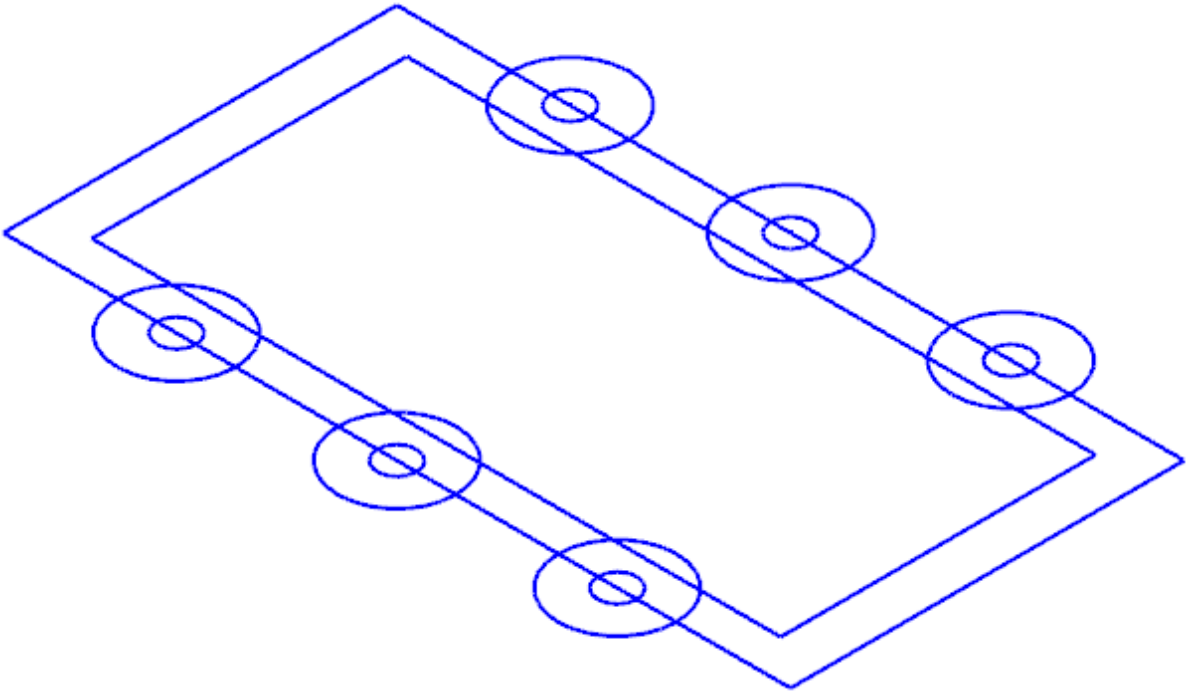
- Create a pressure plate part.
- Modify the `ShippingPullTab.mcam` part file.

Exercise 1: Creating a Solid Part

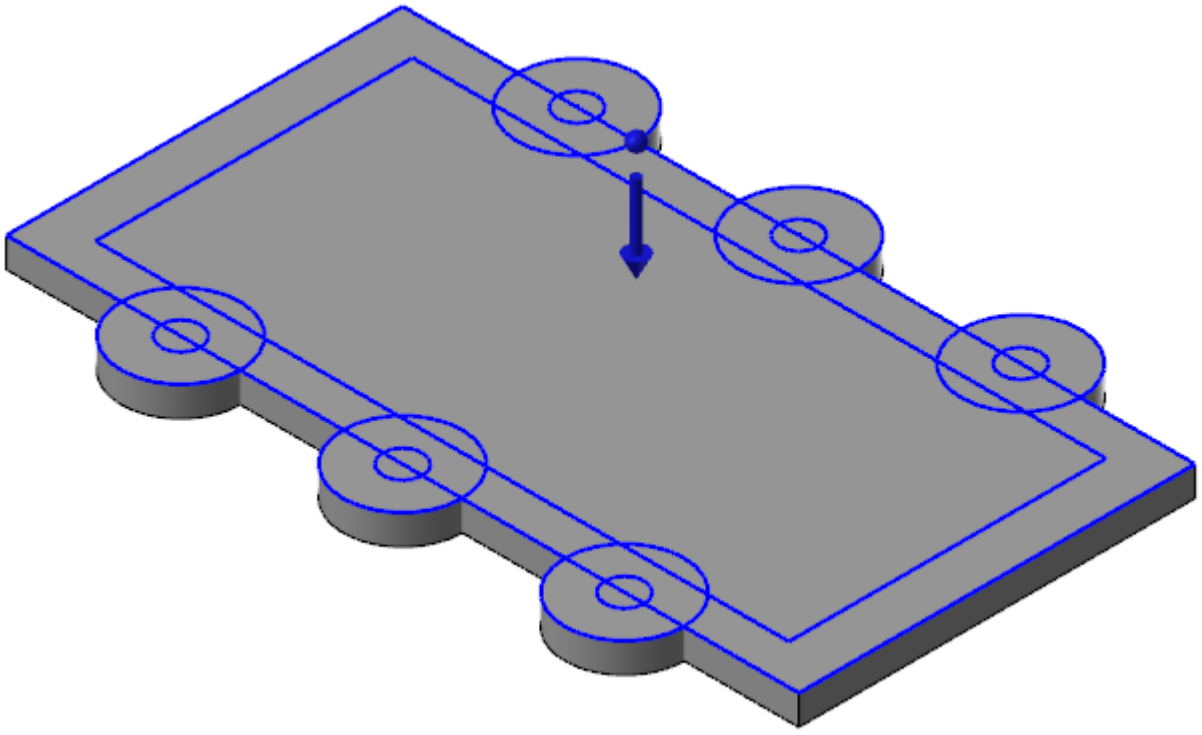
In this exercise, you test your solids knowledge and skills by applying solids functions to supplied geometry to create a pressure plate part. If you have trouble with any step, refer to the videos. The following pictures show two views of the part you will create.



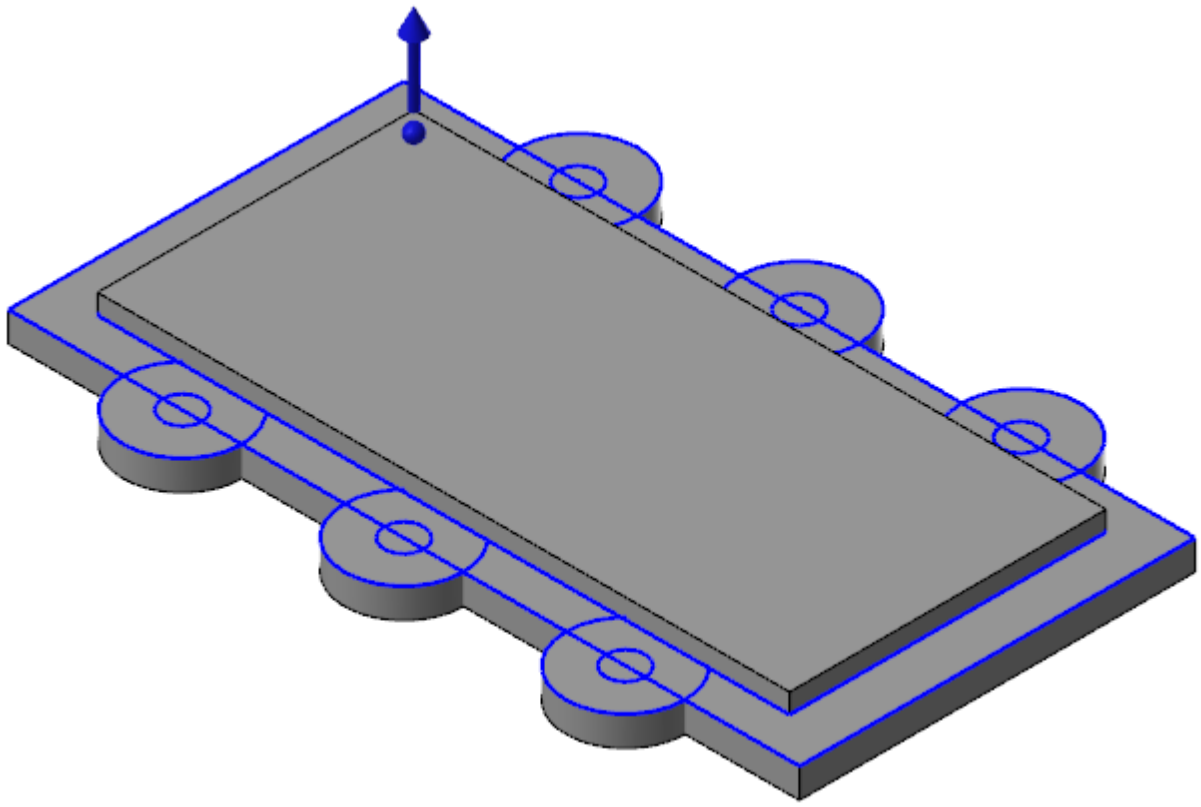
1. Load the file `PressurePlate`, included with this tutorial.



2. So you do not overwrite the original, save the file as `PressurePlate-`*XXX*, where *XXX* is your initials.
3. Extrude the outside rectangle and the six large circles downward by **15** mm.

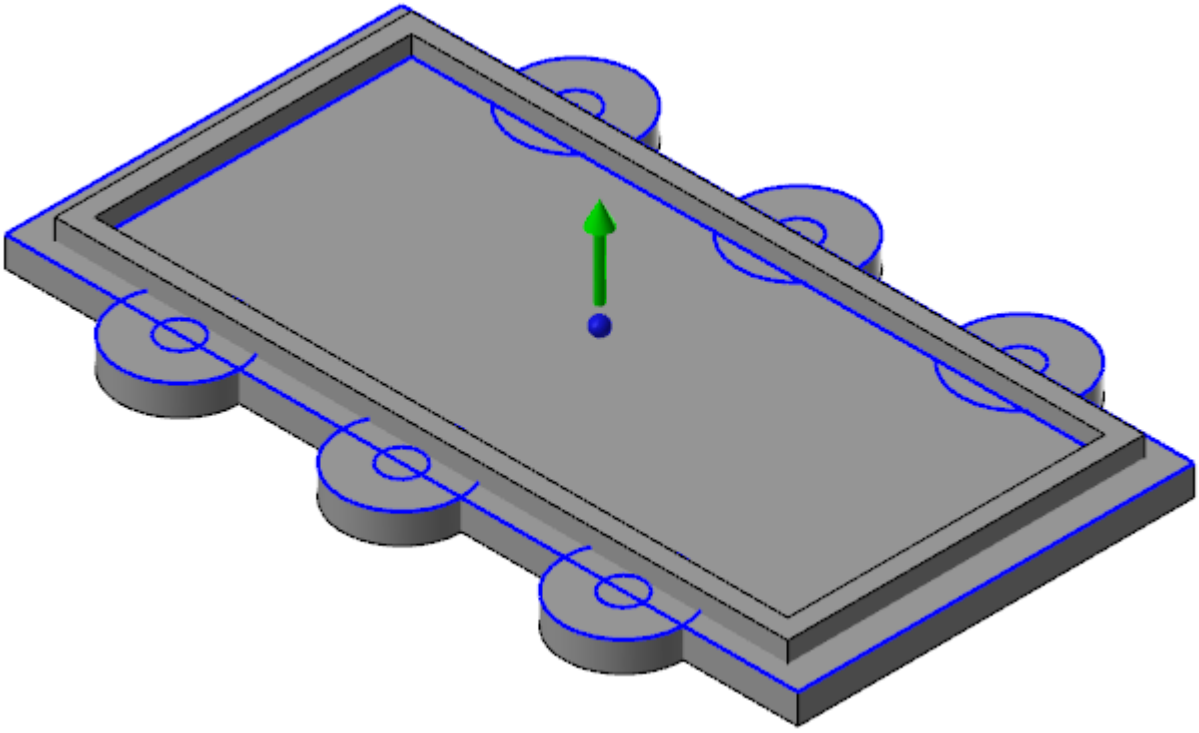


4. Extrude the inner rectangle upward by **10** mm.



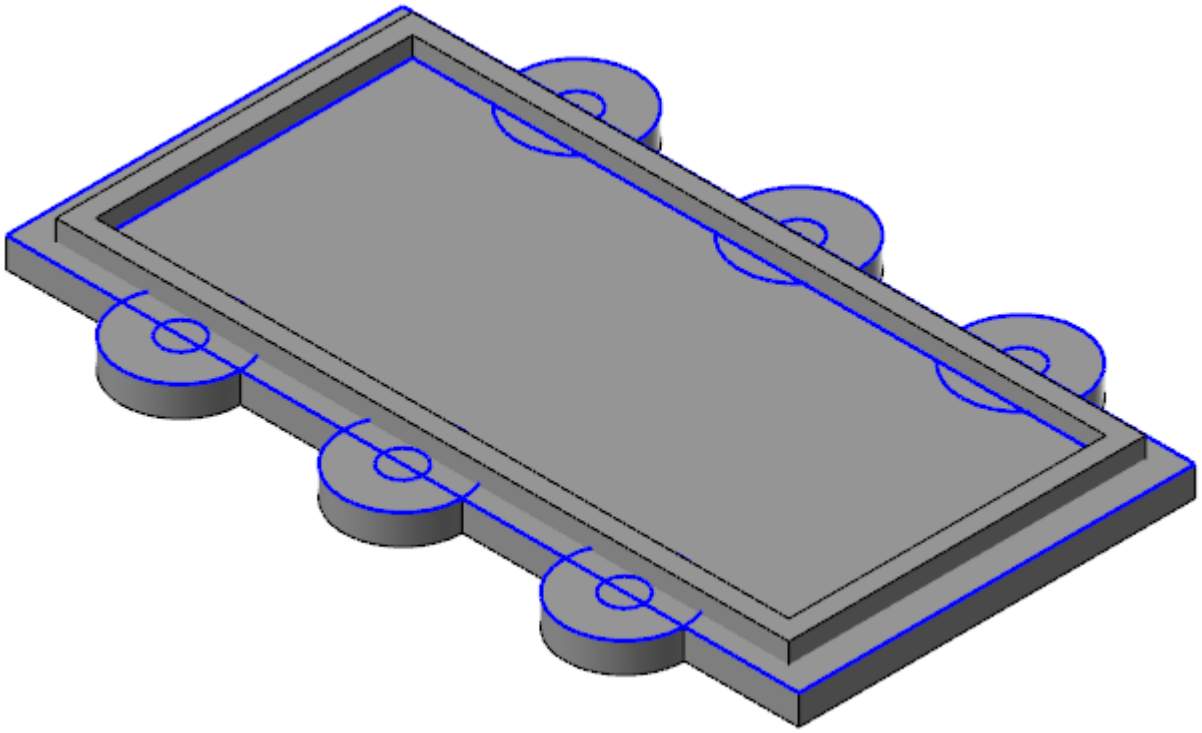
View the YouTube video of steps 3 and 4: https://www.youtube.com/watch?v=6bQJWKrJnx4

5. Use **Shell** to cut out the center rectangle to a thickness of **10** mm. Make sure to select only the face.



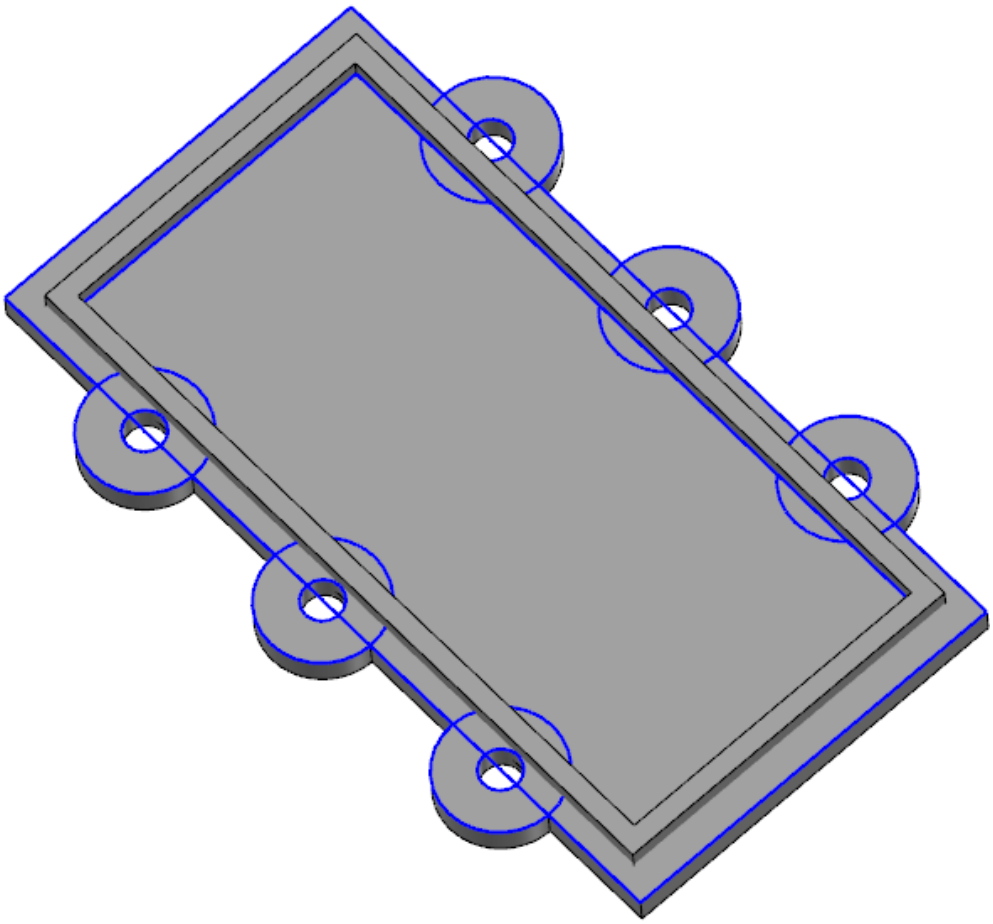
View the YouTube video of step 5: https://www.youtube.com/watch?v=wAOlX8kNuhQ

6. Use **Boolean** to join all features into a single solid.



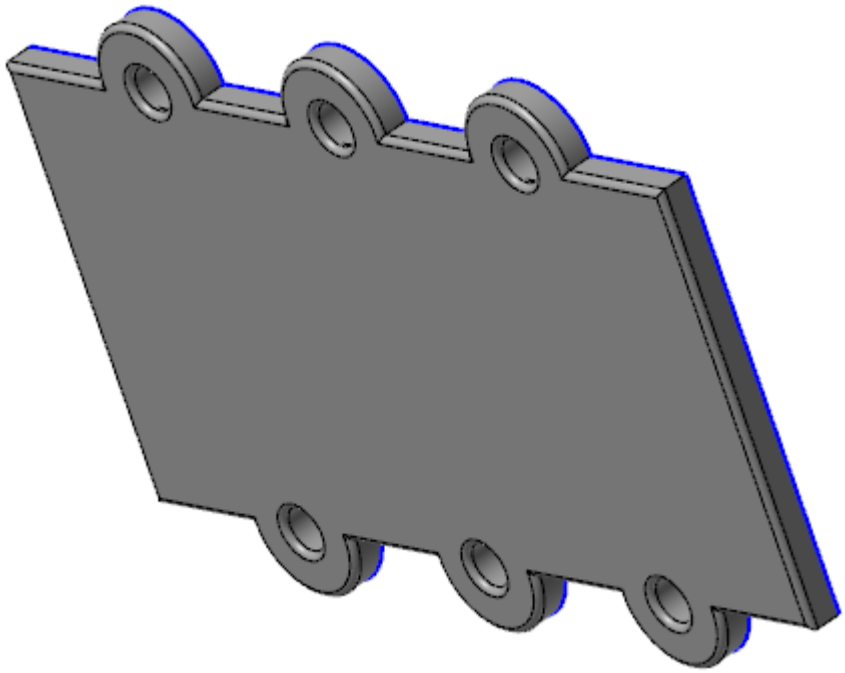
View the YouTube video of step 6: https://www.youtube.com/watch?v=pm2FhRXJOfU

7. Use **Extrude** to create holes from the six smaller circles.



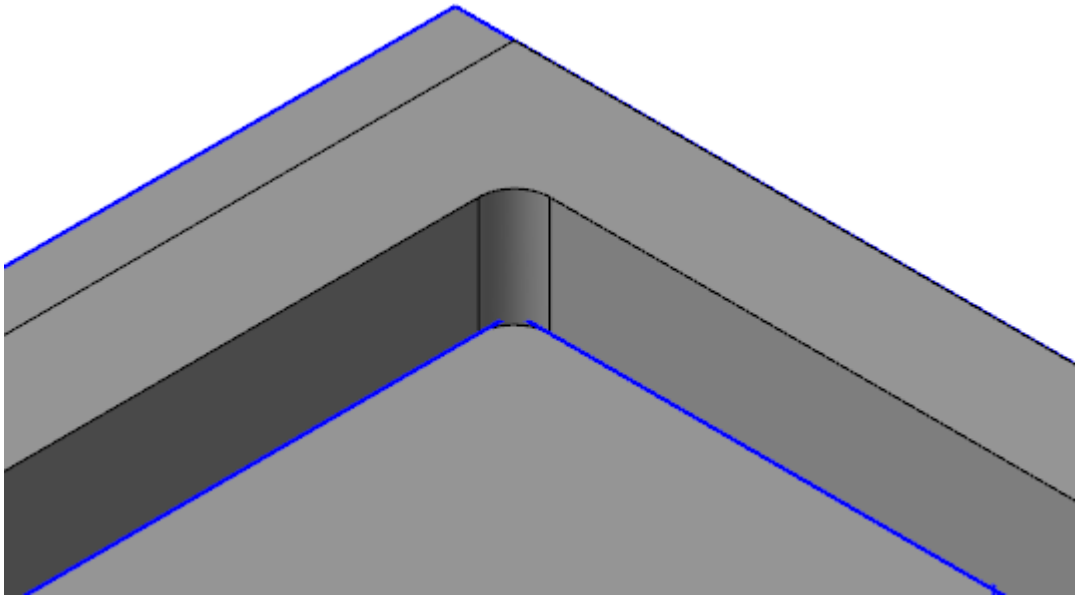
View the YouTube video of step 7: https://www.youtube.com/watch?v=7yTyt8MBWs8

8. Add a **3** mm fillet to the part's bottom face.



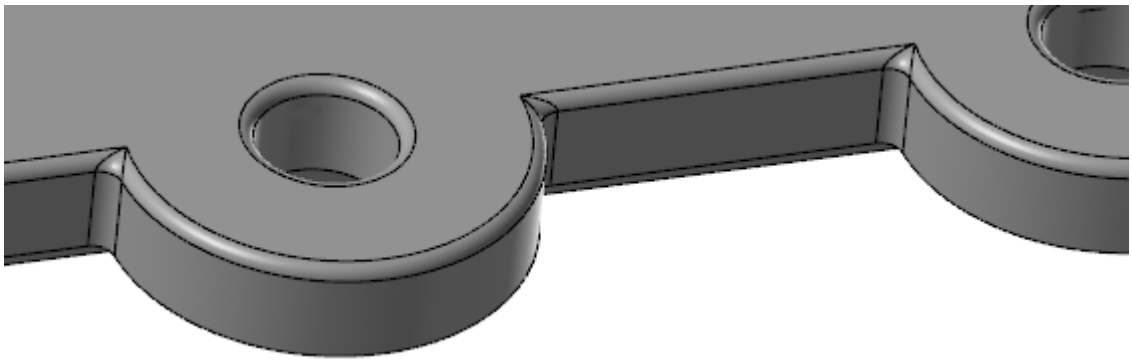
View the YouTube video of step 8: https://www.youtube.com/watch?v=-YFKTxtfsHY

9. Add **3** mm fillets to the inner corners.



View the YouTube video of step 8: https://www.youtube.com/watch?v=s6APxMpHMrU

10. Use the Solids Manager to change the **3** mm bottom fillet to **2** mm and to add **2** mm fillets to the twelve vertical edges of the rounded tabs.



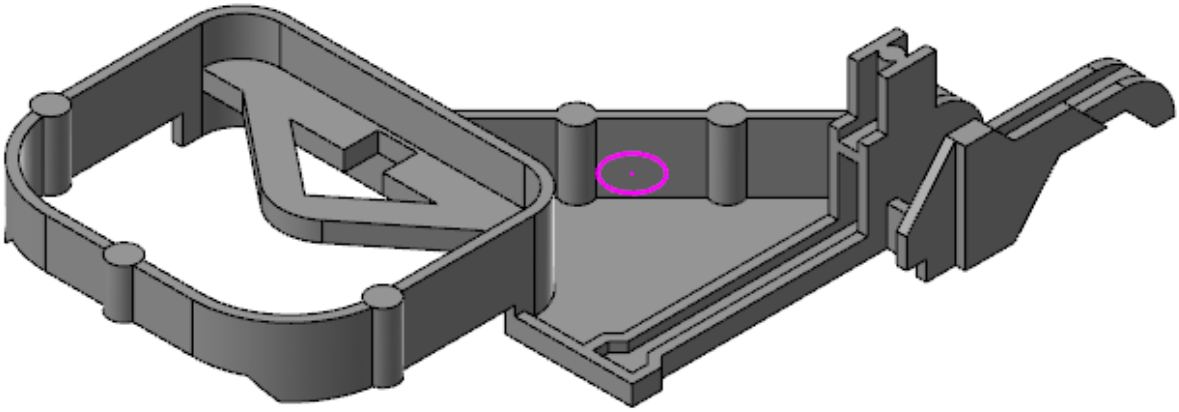
View the YouTube video of step 8: https://www.youtube.com/watch?v=ckr3TezdC2s

Exercise 2: Modifying a Part

In this exercise, you test your Model Prep knowledge and skills as you make changes to a solid part. Try to complete the edits on your own. If you have trouble, refer to the linked videos.

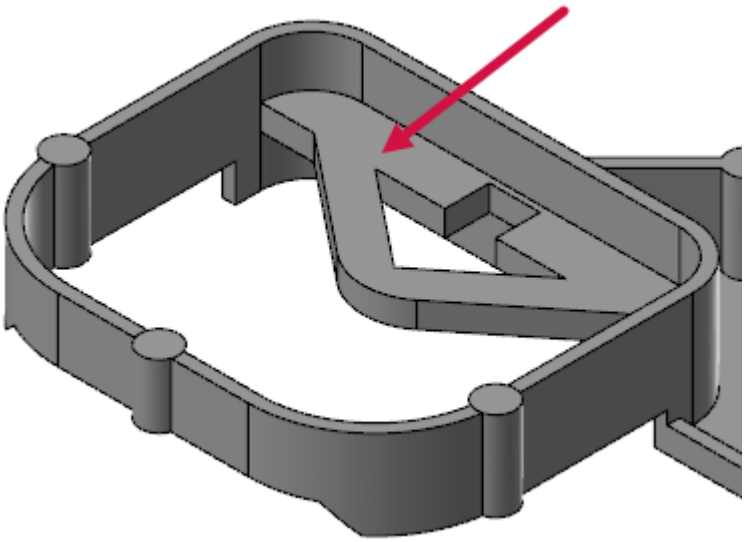
In this challenge, you work on the file named `ShippingPullTab`, included with this tutorial. In most cases, this is the only file you need to complete the challenge. However, a set of files named `ShippingPullTab03` through `ShippingPullTab07` are also included with this tutorial and are the results of each of the challenge steps. For example, `ShippingPullTab03` is the result of Step 3.

1. Load the file `ShippingPullTab`, included with this tutorial.



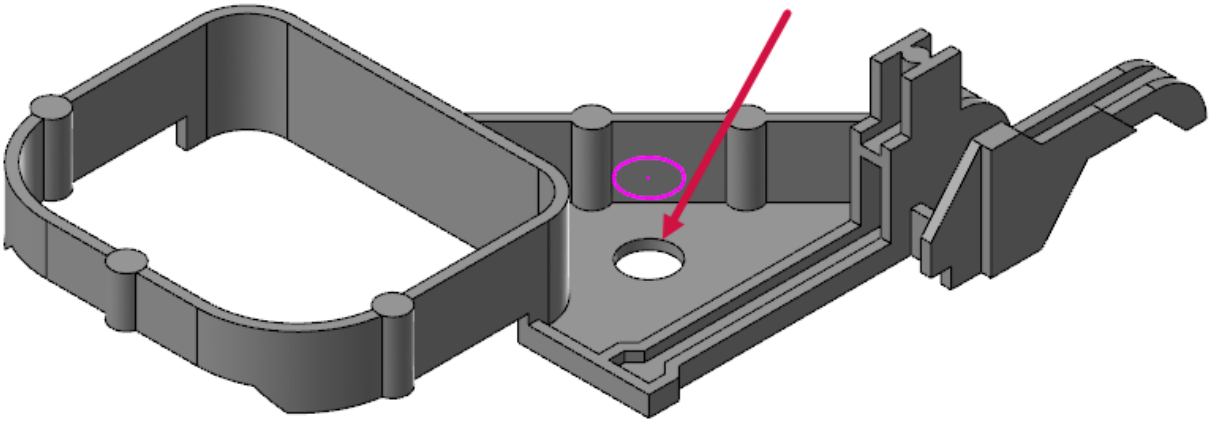
2. To avoid overwriting the original, save the file as `ShippingPullTab-XXX`.

3. Remove the flange shown below.



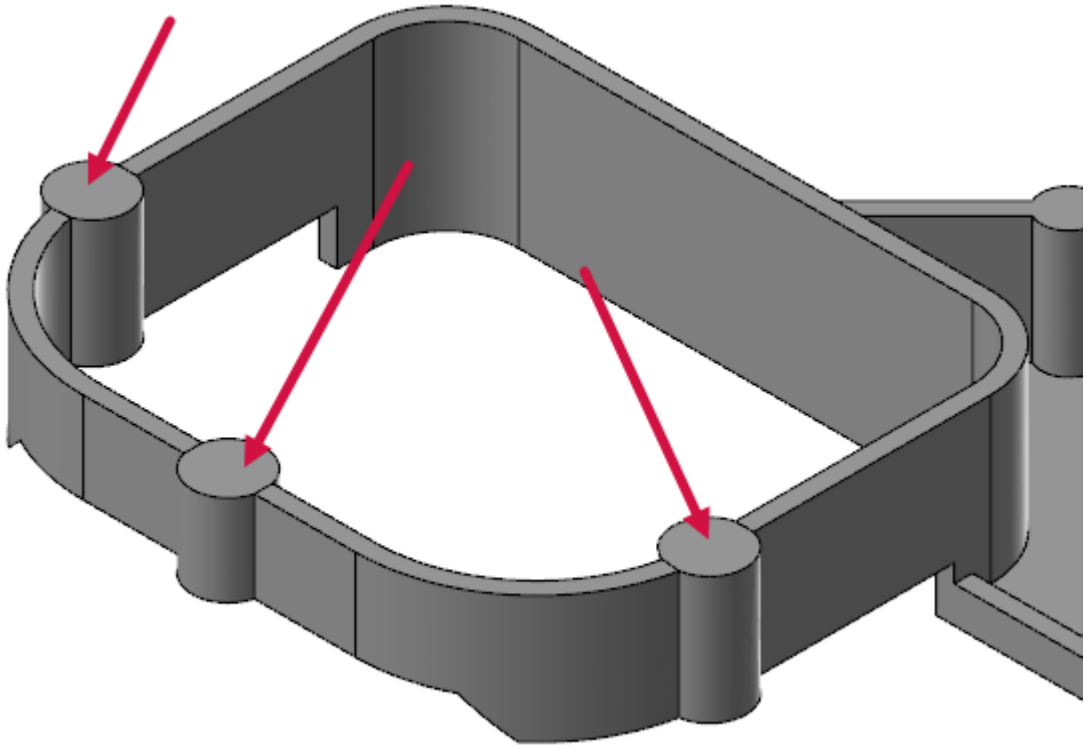
View the YouTube video of step 3: https://www.youtube.com/watch?v=Fdi6JQ-PiM4

4. Add a **5** mm circular cutout to the part, as shown in the following picture.



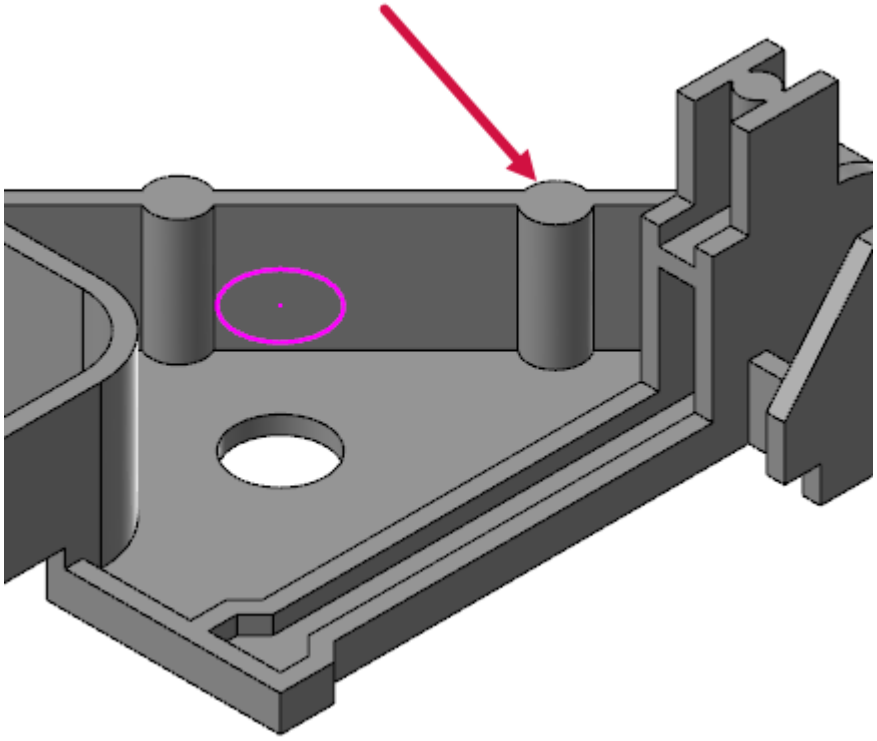
View the YouTube video of step 4: https://www.youtube.com/watch?v=jVCK38tbRKs

5. Increase the diameter of the indicated features to **2** mm.



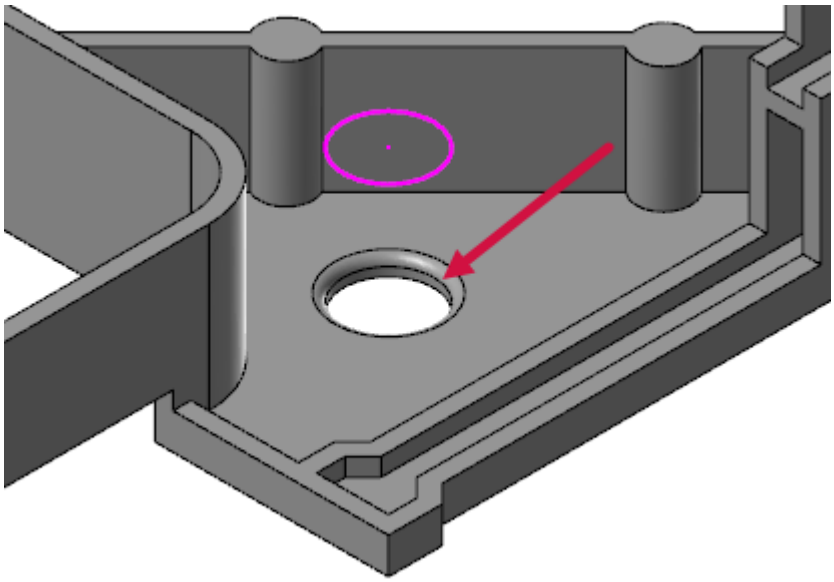
View the YouTube video of step 5: https://www.youtube.com/watch?v=cm3AjUn12Uk

6. Move the indicated feature **4** mm to the right, as shown below.



View the YouTube video of step 6: https://www.youtube.com/watch?v=cCLOuaZSkaw

7. Add a **0.5** mm fillet to the **5** mm hole, as shown below.



View the YouTube video of step 7: https://www.youtube.com/watch?v=lsph6jEHbOY

CONCLUSION

Congratulations! You have completed the *Mastercam Solids Tutorial*! Now that you have mastered the skills in this tutorial, explore Mastercam's other features and functions.

You may be interested in other tutorials that we offer. Mastercam tutorials are being constantly developed, and we will add more as we complete them. Visit our website, or select **Help**, **Tutorials** from the **File** tab.

Mastercam Resources

Enhance your Mastercam experience by using the following resources:

- *Mastercam Documentation*—Mastercam installs a number of helpful documents for your version of software in the Documentation folder of your Mastercam 2019 installation.
- *Mastercam Help*—Access Mastercam Help by selecting **Help**, **Contents** from Mastercam's **File** tab or by pressing [**Alt+H**] on your keyboard.
- *Mastercam Reseller*—Your local Mastercam Reseller can help with most questions about Mastercam.
- *Technical Support*—Our Technical Support department (+1 860-875-5006 or support@mastercam.com) is open Monday through Friday from 8:00 a.m. to 5:30 p.m. USA Eastern Standard Time.
- *Mastercam Tutorials*—We offer a series of tutorials to help registered users become familiar with basic Mastercam features and functions. Visit our website, or select **Help**, **Tutorials** from Mastercam's **File** tab to see the latest publications.
- *Mastercam University*—Mastercam University, an affordable online learning platform, gives you 24/7 access to Mastercam training materials. Take advantage of more than 180 videos to master skills at your own pace and help prepare for Mastercam Certification. For more information on Mastercam University, please contact your Authorized Mastercam Reseller, visit www.mastercamu.com, or email training@mastercam.com.
- *Online Communities*—You can find a wealth of information at www.mastercam.com. For tech tips and the latest Mastercam news, follow us on Facebook (www.facebook.com/mastercam), Twitter (www.twitter.com/mastercam), or Google+ (plus.google.com/+mastercam). Visit our YouTube channel to see Mastercam in action (www.youtube.com/user/MastercamCadCam)! Registered users can search for information or ask questions on the Mastercam Web forum, forum.mastercam.com, or use the knowledgebase at kb.mastercam.com.

Contact Us

For questions about this or other Mastercam documentation, contact the Technical Documentation department by email at techdocs@mastercam.com.

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