GETTING STARTED WITH MASTERCAM SOLIDS

June 2017

POWERED BY MASTERCAM'S DYNAMIC MOTION™ TECHNOLOGY

Mastercam®
Be Dynamic.

GETTING STARTED WITH MASTERCAM SOLIDS

June 2017
© 2017 CNC Software, Inc. – All rights reserved.
Software: Mastercam 2018

Terms of Use

Use of this document is subject to the Mastercam End User License Agreement. The Mastercam End User License Agreement can be found at:

http://www.mastercam.com/companyinfo/legal/LicenseAgreement.aspx

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 **5**

General Tutorial Requirements 6

1 Introduction to Mastercam Solids **7**

Getting Ready to Work 8

Exercise 1: Creating the Tray Base 9

Exercise 2: Creating the Upper Bosses 15

Exercise 3: Creating the Finger Cutouts 21

Exercise 4: Creating the Marker Cutouts 25

Exercise 5: Adding the Handle 27

Exercise 6: Adding Draft and Fillets 35

Adding Draft 36

Adding Outside Fillets 38

Adding Top Fillets 39

Exercise 7: Shelling the Part 41

Other Solids Functions 43

Exercise 8: Using Solids Revolve 43

Exercise 9: Creating Solids with Sweep 46

Exercise 10: Using Boolean to Create a Hole 48

Exercise 11: Adding Chamfers 50

Exercise 12: Using Solid Impression 52

Challenge 55

2 Introduction to Model Prep 61

Getting Ready to Work 62

The Scenario 62

Exercise 1: Splitting the Feet from the Part 63

Exercise 2: Removing One Foot 70

Exercise 3: Reducing the Foot Height 73

Exercise 4: Repositioning the Feet 77

Exercise 5: Resizing the Foot Holes 83

Exercise 6: Creating the Counter bore 86

Exercise 7: Aligning the Vertical Wall 92

Exercise 8: Expanding the Wall Opening 96

Exercise 9: Increasing the Wall Height 100

Exercise 10: Adding the Fillet 103

Challenge 105

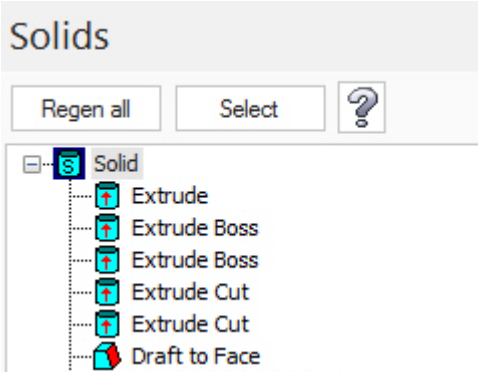
Conclusion 109

Mastercam Resources 109

Contact Us 110

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.



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.

Tutorial Goals

- Create a solid model from wireframe geometry.
- Edit a solid model from the solid history tree.
- Modify brick solid models with Mastercam's Model Prep functions.

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 2018 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 2018 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 English configuration. The tutorial provides instructions for loading the appropriate configuration file.

1

Introduction to Mastercam Solids

Mastercam Solids features the tools you need to create sophisticated solid models, using functions with dynamic preview so you can see results as soon you edit parameters. The solids history tree lists the operations that generate the solid, and you can open and edit any operation, as well as move, delete, and suppress operations.

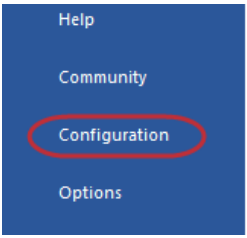
Getting Ready to Work

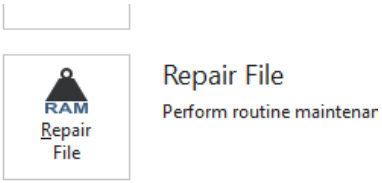
This tutorial includes the files needed to complete the exercises. You can find these files in the tutorial's `Parts` folder, which itself contains the subfolders `FlangeDemo`, `ImpressionDemo`, `MarkerTray`, `PressurePlate`, `ShippingPullTab`, and `SpindlePartial`. Place these files on your system,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

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.

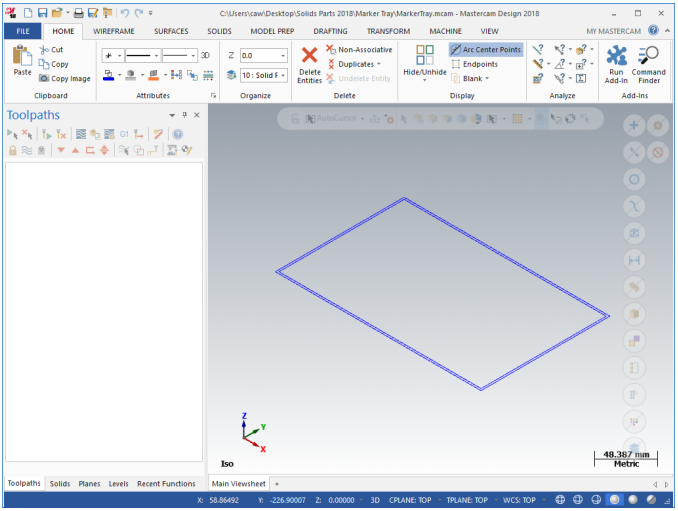
d. Click **OK**.

In this lesson, you work on the file named `MarkerTray`, which is included with this tutorial. In most cases, this is the only file you need to complete the lesson. However, a set of files named `MarkerTray01` through `MarkerTray07` are also included with this tutorial and are the results of each of the exercises. For example, `MarkerTray01` is the results of Exercise 1 and so is in the state the part should be in when starting Exercise 2 .

Exercise 1: Creating the Tray Base

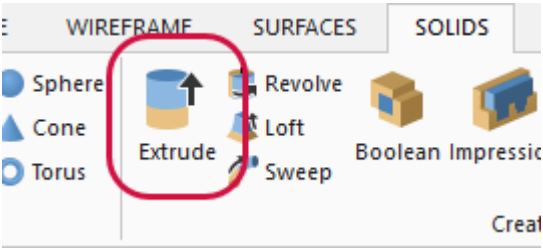
1. Open the file `MarkerTray`.

   The following picture shows the file loaded into Mastercam.



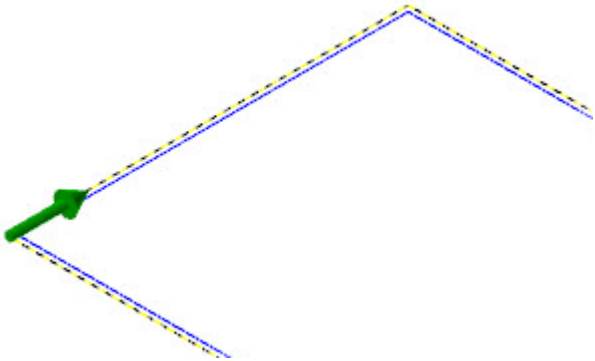
2. So you do not overwrite the original, save the file as `MarkerTray-XXX`, where *XXX* is your initials.

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



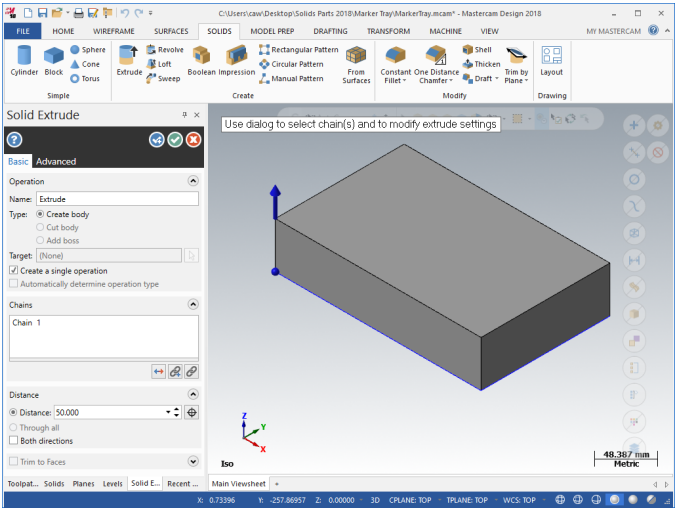
   The Chaining dialog box opens.

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

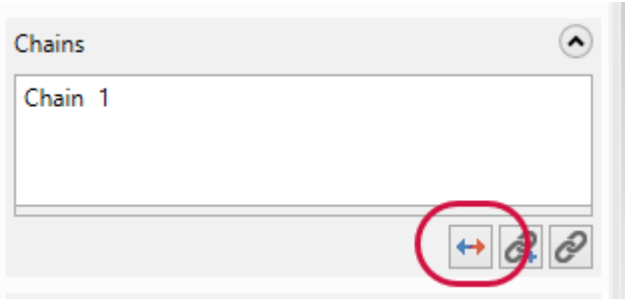


5. Click **OK** in the Chaining dialog box.

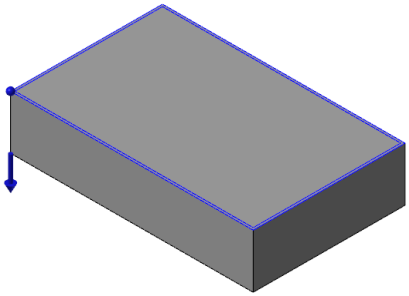
   Mastercam performs an extrusion on the selected chain and displays the Solid Extrude function panel.



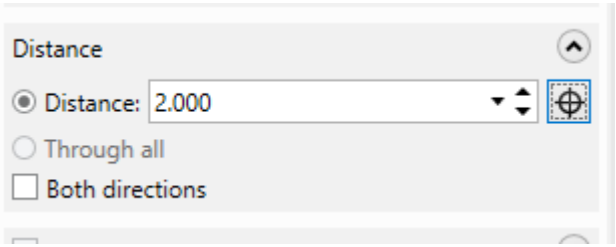
6. In the function panel, click the **Reverse All** button.



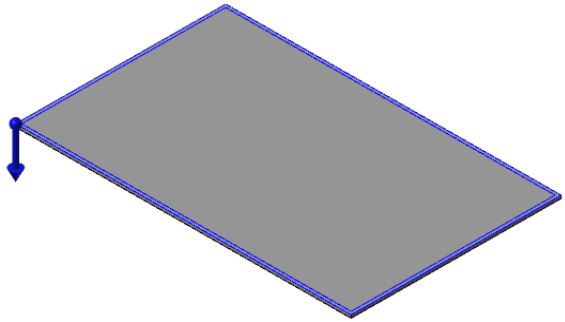
Mastercam reverses the extrude direction.



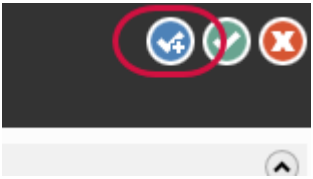
7. Set the extrude **Distance** to **2**, and press [**Tab**] to accept the new setting.



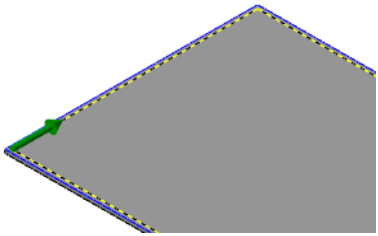
Mastercam shows a preview of the new extrusion, as shown in the picture below.



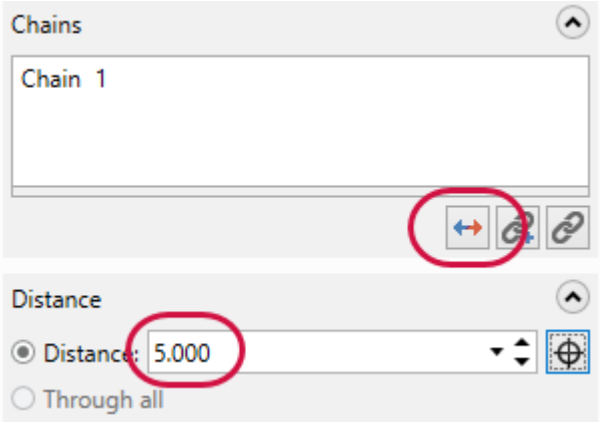
8. Click **OK and Create New Operation** to finalize the extrude operation and begin a new one.



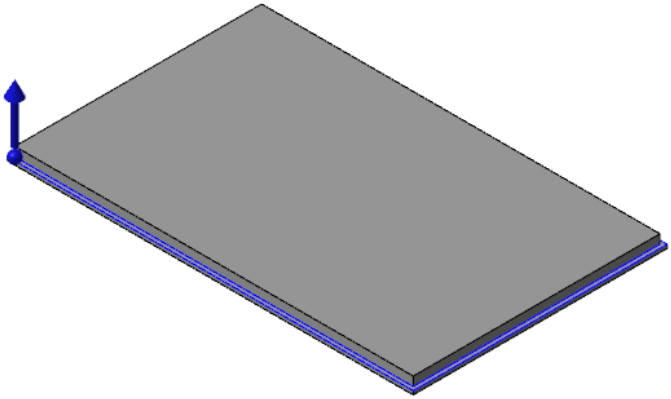
9. Chain the inside rectangle, and click **OK** in the Chaining dialog box.



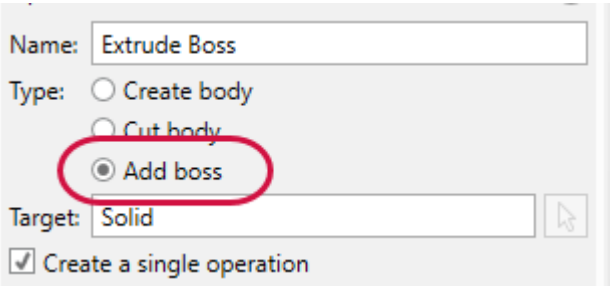
10. Reverse the extrude direction, and set the distance to **5**.



Your part should now look like the following picture.

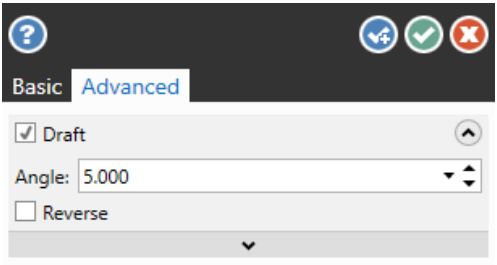


11. Set the solids operation to **Add boss**.

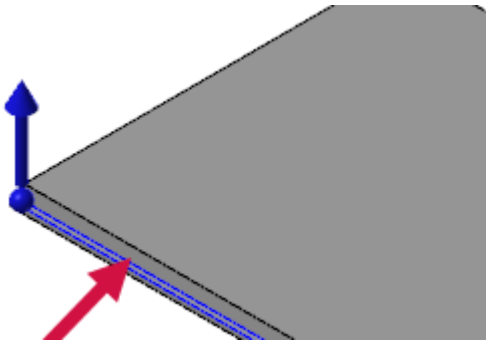


The **Add boss** option creates the extrusion as new material on the part.

12. Click the **Advanced** tab, and turn on **Draft**, leaving the angle at **5** degrees.



Mastercam updates the display 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.



13. In the function panel, click **OK** to finalize the extrude operations.

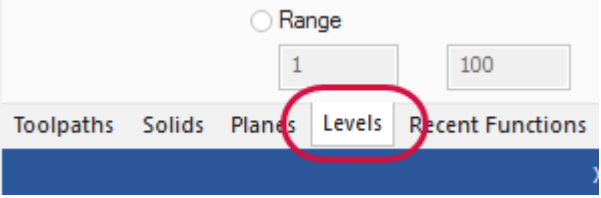
14. Save the file as `MarkerTray01-XXX`, where `XXX` is your initials.

Exercise 2: Creating the Upper Bosses

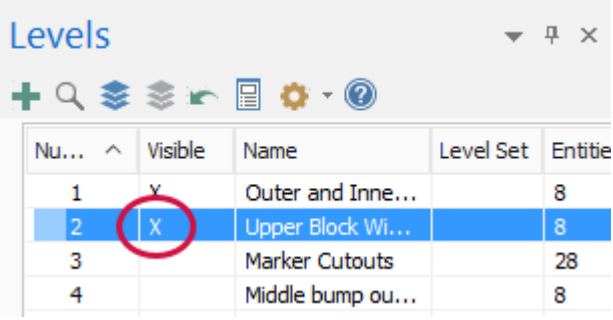
Now that you have the tray's base, you can add the bosses that form the eraser holder and the marker storage area. For this exercise, continue with the part you just saved, or load `MarkerTray01`, which accompanies this tutorial. If you choose to start with `MarkerTray01`, remember to save it under the new name `MarkerTray01-`*XXX*, where *XXX* is your initials.

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

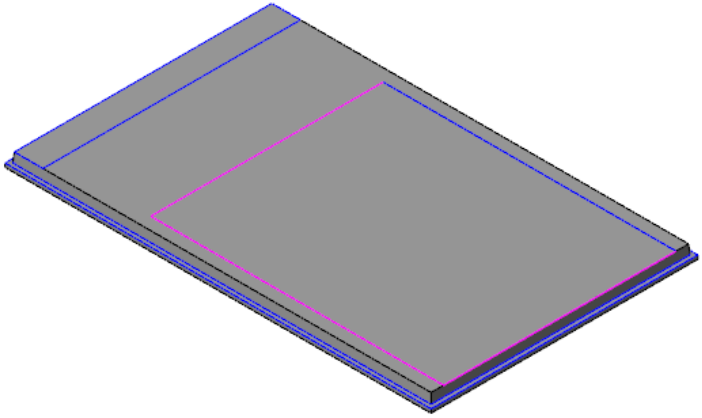
   The Levels Manager displays.



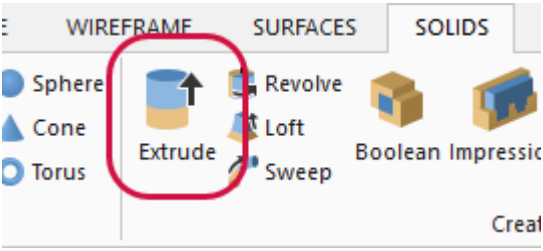
2. Turn on level 2, by clicking in its **Visible** column.



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.

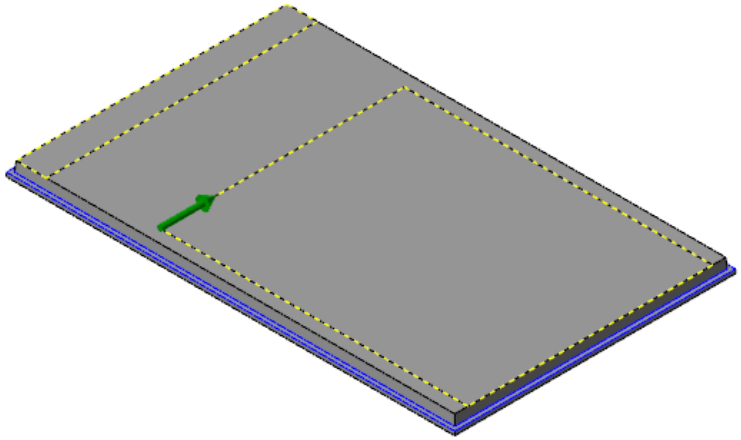


3. Choose the **Solids** tab, and then **Extrude**.

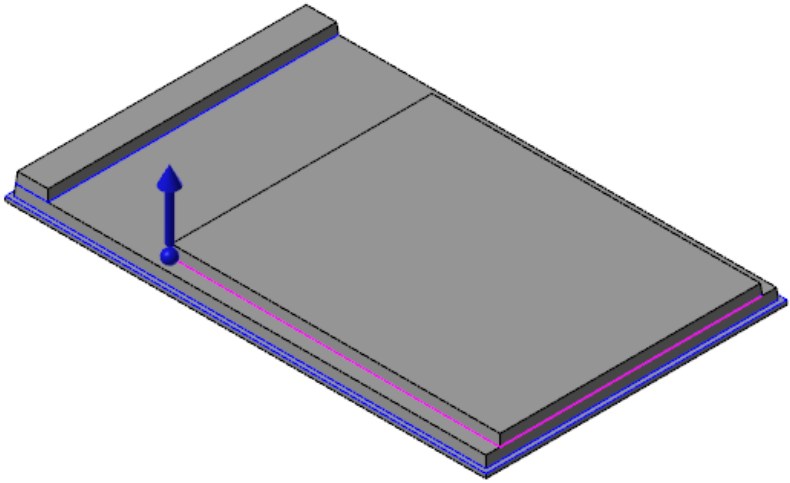


The Chaining dialog box opens.

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

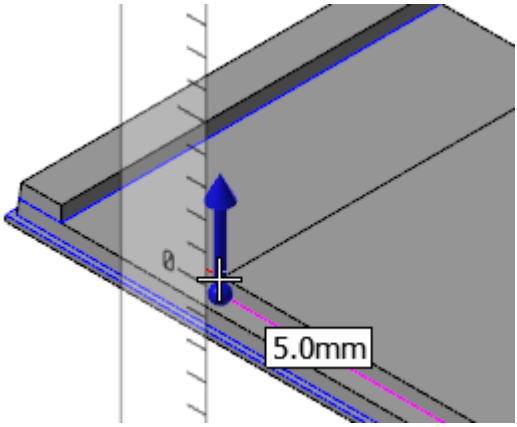


Mastercam creates the two new extrusions.



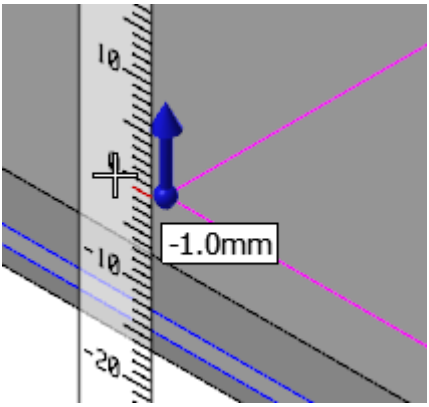
5. Click the blue arrow, and then hover your mouse over the ruler that appears.

   The ruler lets you enter linear values visually. If you prefer, you can also type values.



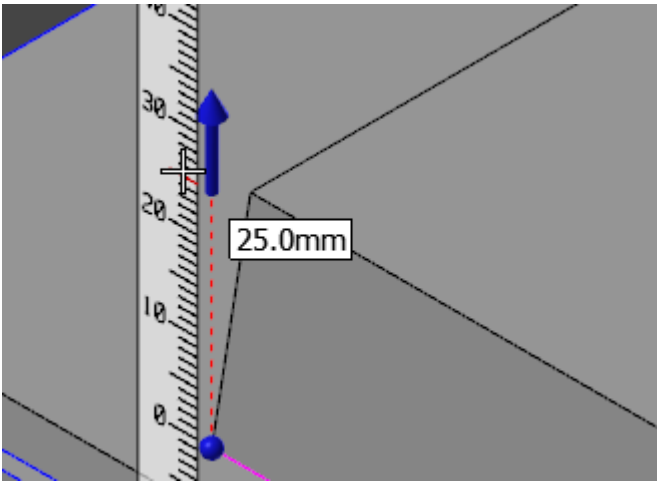
6. Using your mouse wheel, zoom in closer to the ruler.

Mastercam displays smaller increments on the ruler.

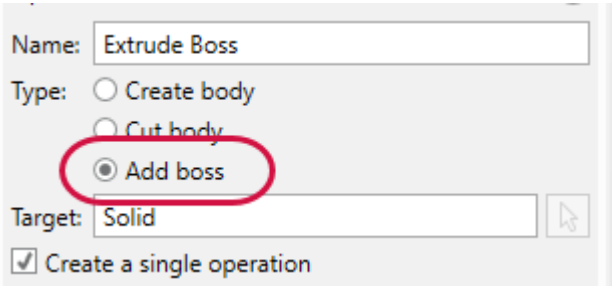


7. Move your mouse up over the ruler until it snaps to **25 mm**, and then click to set the new distance.

   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.

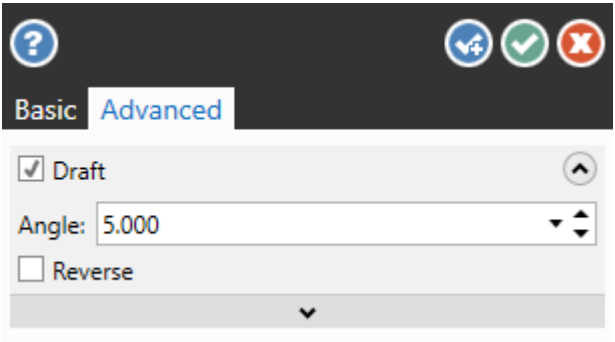


8. In the function panel, ensure that **Add boss** is selected.



9. Click the **Advanced** tab, and notice that the draft angle is still set to 5 degrees.

   Mastercam remembers your solids settings for the current session.



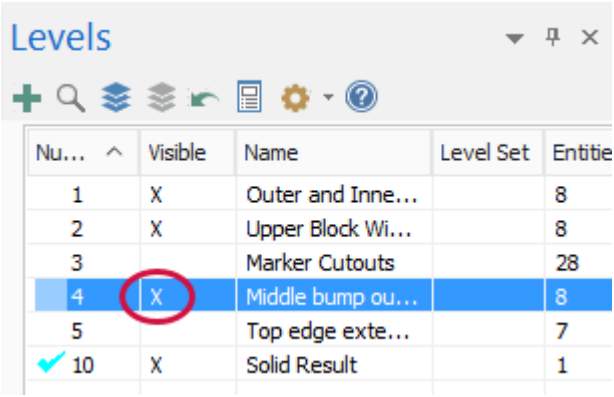
10. In the function panel, click **OK** to finalize the extrude operations.

11. Save the file as `MarkerTray02-XXX`, where `XXX` is your initials.

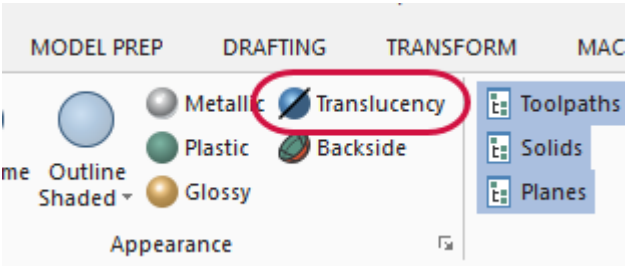
Exercise 3: Creating the Finger Cutouts

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 just saved, or load `MarkerTray02`, which accompanies this tutorial. If you choose to start with the tutorial file, remember to save it under the new name `MarkerTray02–`*XXX*, where *XXX* is your initials.

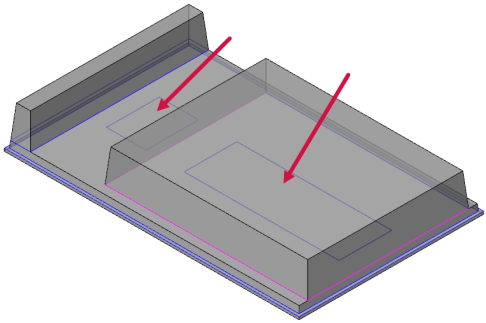
1. In the Levels Manager, turn on level 4.



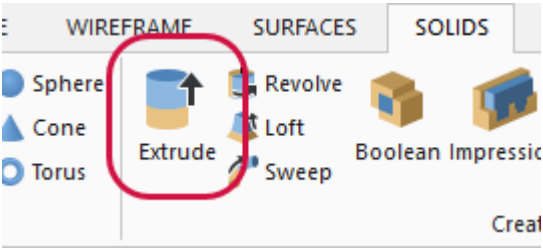
2. On Mastercam's **View** tab, click the **Translucency** button.



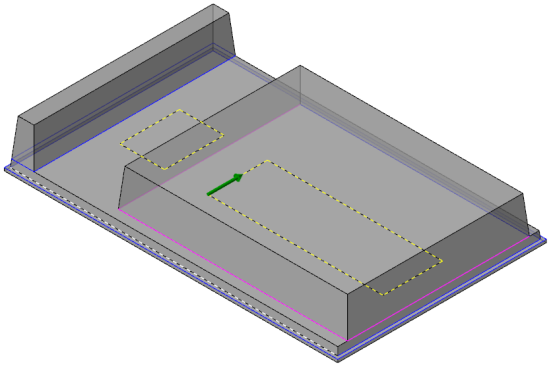
The part becomes partially transparent, allowing you to see the two new rectangles hidden under existing geometry.



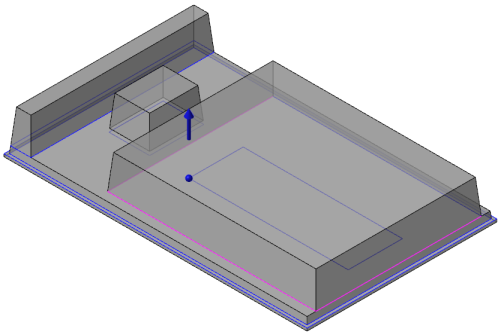
3. On Mastercam's **Solids** tab, choose **Extrude**.



4. Chain the two new rectangles, clicking **OK** in the Chaining dialog box when finished.

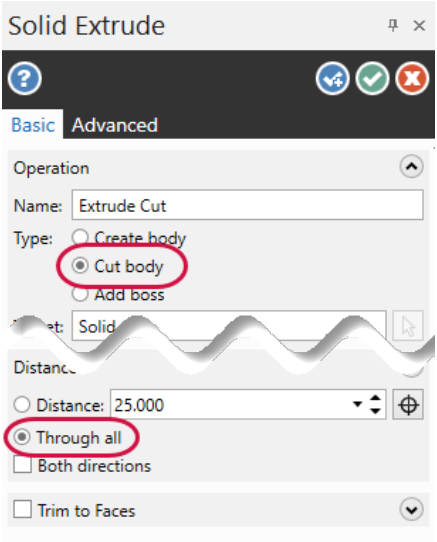


Mastercam extrudes the two chained rectangles, as shown in the picture below.

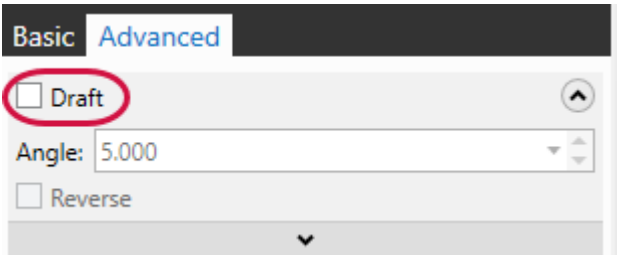


5. In the function panel, change the operation to **Cut body**, and select **Through all**.

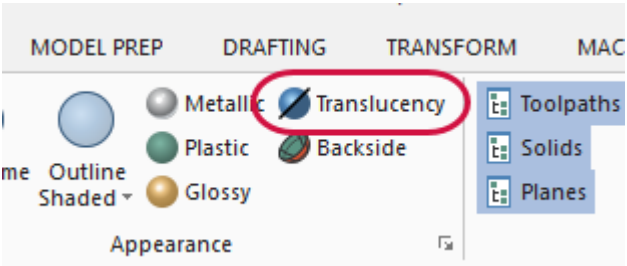
   The **Cut body** option is the opposite of **Add boss**, removing material from the part rather than adding it.



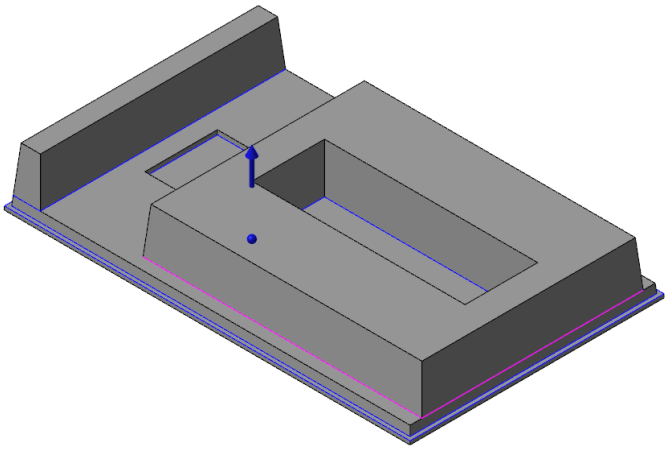
6. On the Advanced tab, turn off **Draft**.



7. Turn off translucency.



Now you can see the effect of the two new cuts more easily.

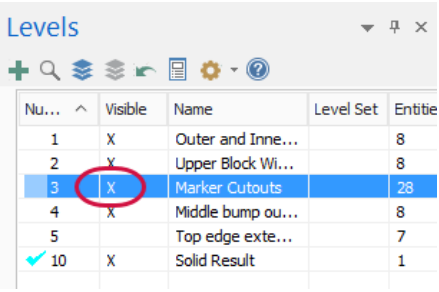


8. In the function panel, click **OK** to finalize the extrude operation.
9. Save the file as `MarkerTray03-XXX`, where *XXX* is your initials.

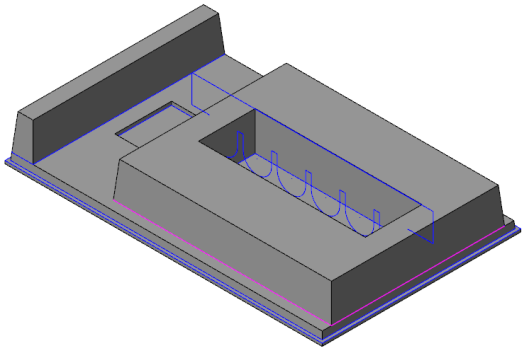
Exercise 4: Creating the Marker Cutouts

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. If you choose to start with the tutorial file, remember to save it under the new name `MarkerTray03-XXX`, where *XXX* is your initials.

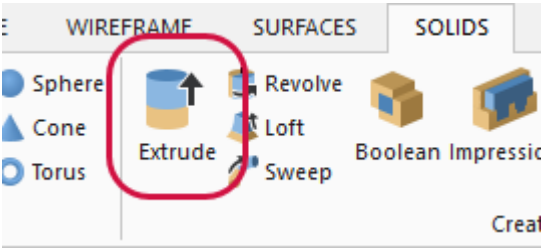
1. In the Levels Manager, turn on level 3.



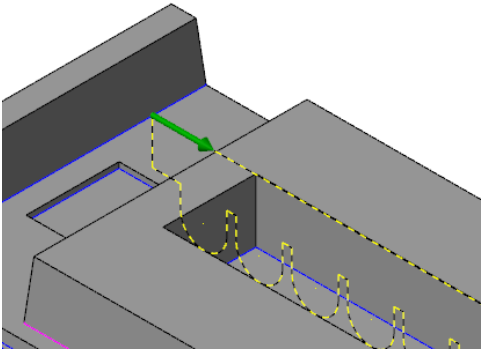
Mastercam displays the geometry created for you on that level. This geometry represents the shape of the marker cutouts.



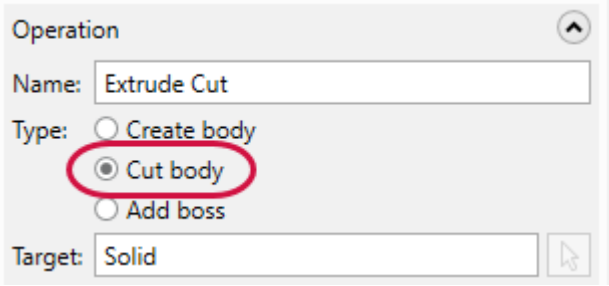
2. In Mastercam's **Solids** tab, choose **Extrude**.



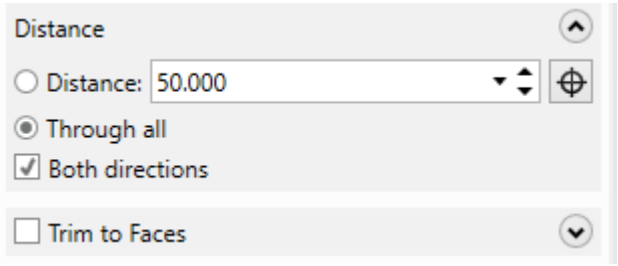
3. Chain the new geometry, clicking **OK** in the Chaining dialog box when finished.



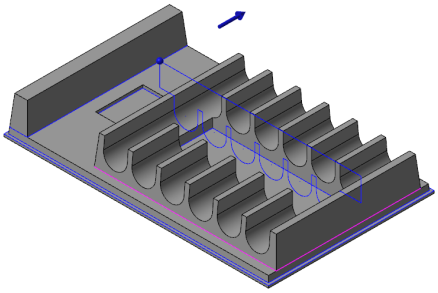
4. In the function panel, select **Cut body**.



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



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

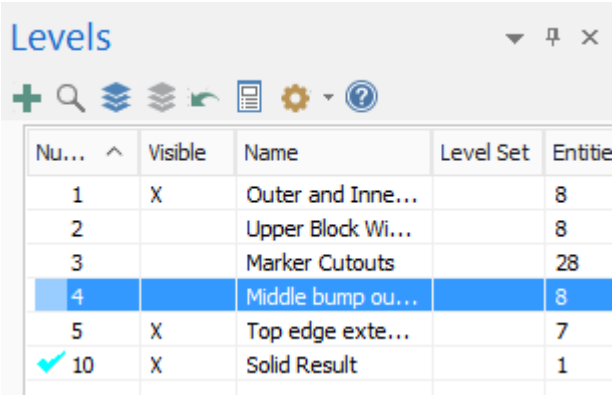


6. In the function panel, click **OK** to finalize the extrude operation.
7. Save the file as `MarkerTray04-`*XXX*, where *XXX* is your initials.

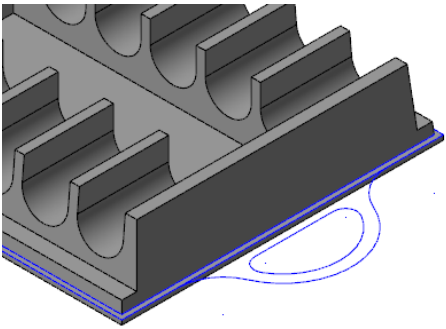
Exercise 5: Adding the Handle

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. If you choose to start with the tutorial file, remember to save it under the new name `MarkerTray04-`*XXX*, where *XXX* is your initials.

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



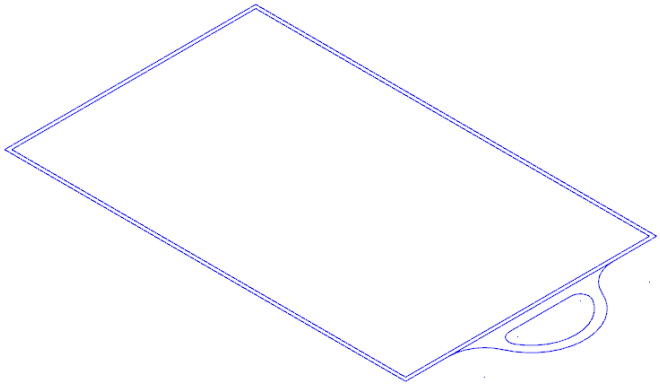
   Mastercam displays the geometry that you need to create the handle.



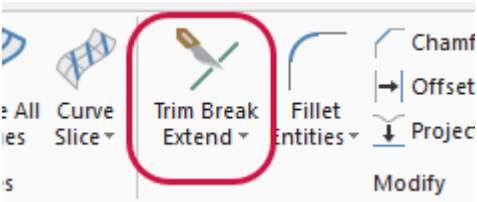
2. Select the left-hand portion of the wireframe Quick Mask button, found along the right side of the graphics window.

   Mastercam selects all wireframe geometry.

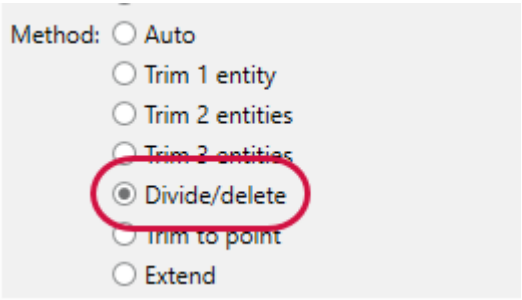
3. Press [**Alt+E**] to hide all geometry except the selected wireframe.



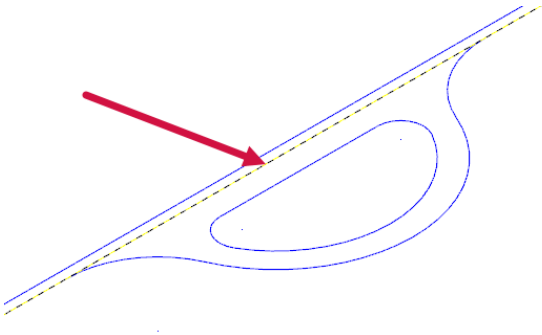
4. In Mastercam's **Wireframe** tab, select **Trim Break Extend**.



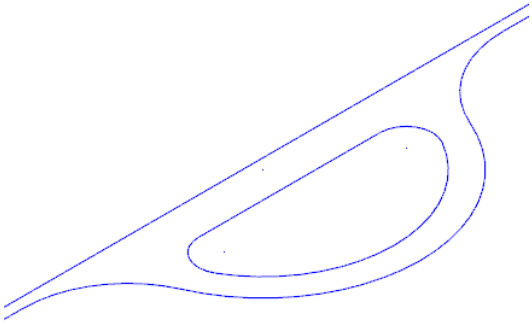
5. In the Trim Break Extend function panel, select **Divide/delete**.



6. Click the line indicated in the picture below.



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

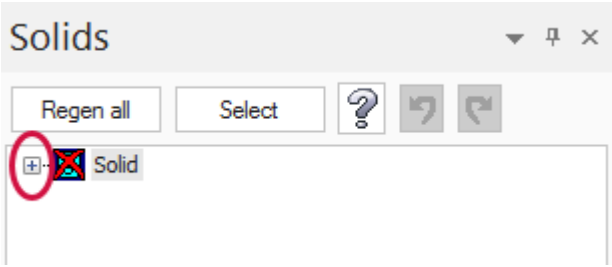


7. Click the function panel's **OK** button.
8. At the bottom of the function panel, click the **Solids** tab.

   Mastercam displays the Solids Manager.

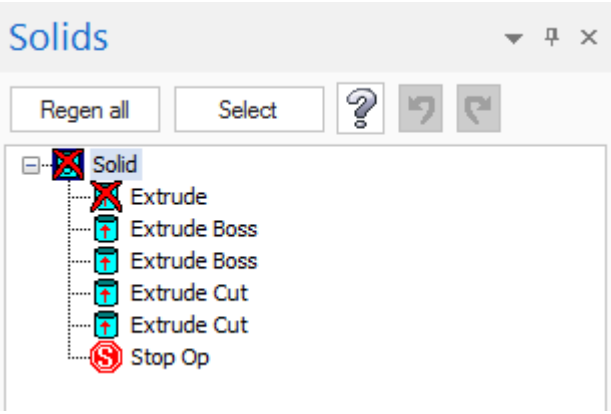
9. Click the plus sign (+) next to the item in the tree.

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

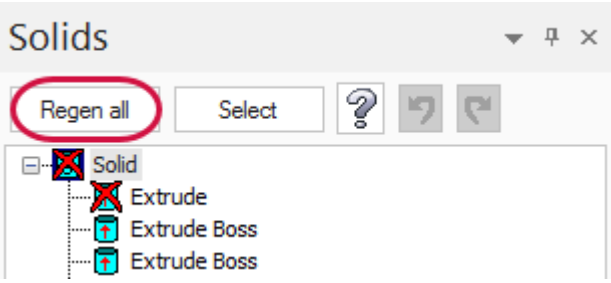


   Notice that the first extrude operation is marked dirty.

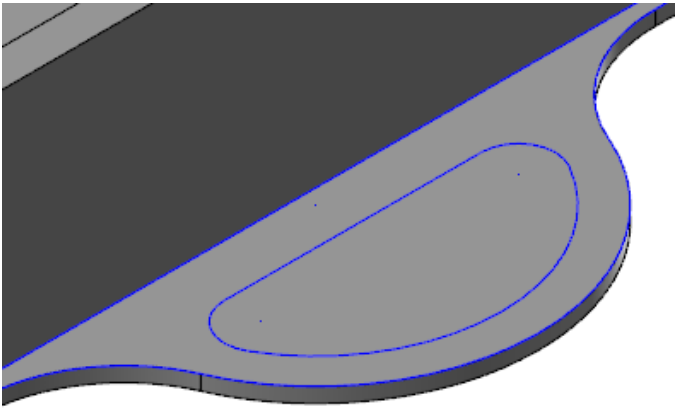
   By adding the handle, you changed the geometry that makes up the tray's base, which means that the extrude operation must be regenerated.



10. Press [**Alt+E**] to turn the full geometry back on, and then click **Regen all** to regenerate the solids operations.

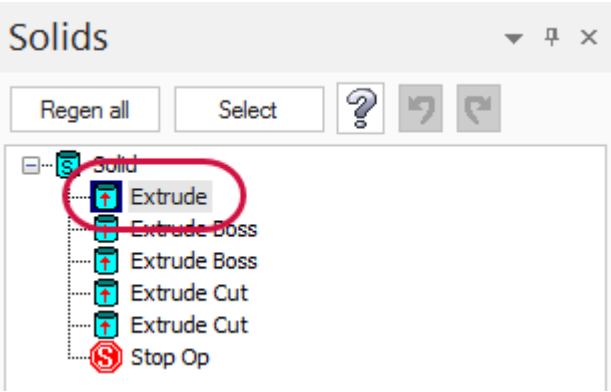


The handle is now a full part of the tray's base.



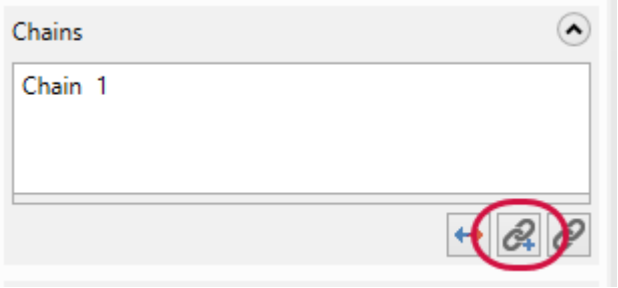
11. Double-click the first extrude in the tree to reopen the operation.

    Mastercam opens the Solid Extrude function panel so you can make changes to the operation.

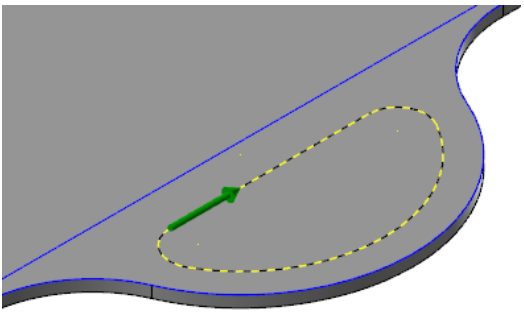


12. In the function panel, click the **Add Chains** button.

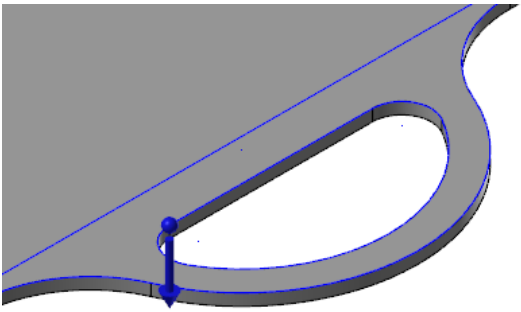
    Mastercam opens the Chaining dialog box.



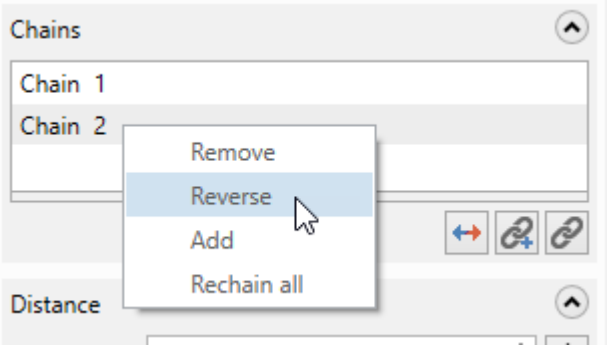
13. Chain the inside of the handle, and click **OK**.



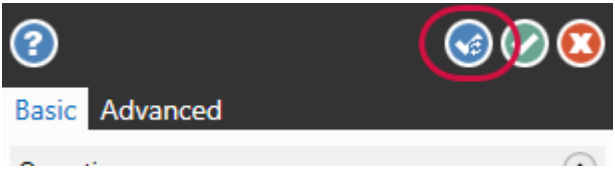
Mastercam adds the chain to the extrude operation, cutting the handle opening.



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



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



OK and Regenerate saves you from going to the solids tree to regenerate.

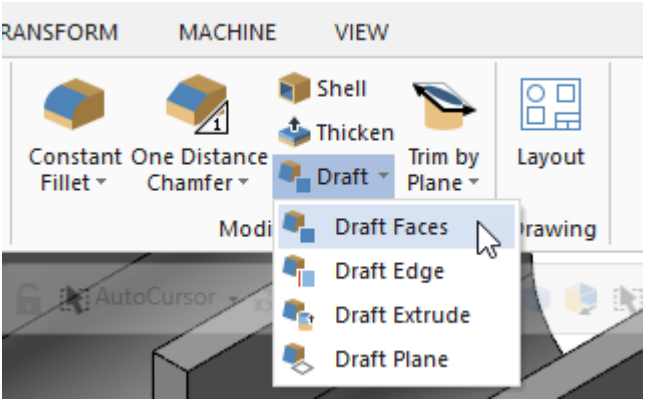
16. Save the file as `MarkerTray05-XXX`, where `XXX` is your initials.

Exercise 6: Adding Draft and Fillets

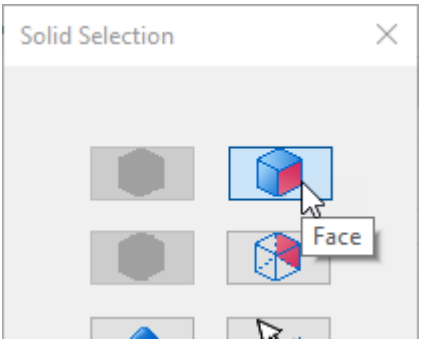
In this exercise, you add draft to a face and fillets to some edges. Continue with the part you just saved, or load `MarkerTray05`, which accompanies this tutorial. If you choose to start with the tutorial file, remember to save it under the new name `MarkerTray05-`*`XXX`*, where *XXX* is your initials.

Adding Draft

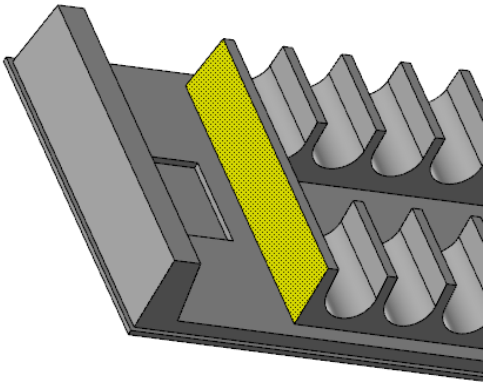
1. In Levels Manager, turn off all levels but 10.
2. In Mastercam's **Solids** tab, choose **Draft, Draft Faces**.



3. In the **Solid Selection** dialog box, ensure that the **Face** button is selected.

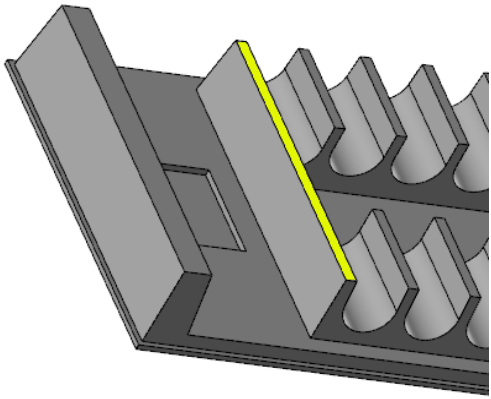


4. Select the face indicated in the picture below, and click **OK** in the Solid Selection dialog box.



5. Select the top face of the same wall (as indicated in the picture below) for the reference face.

   Mastercam creates the draft relative to the reference face.

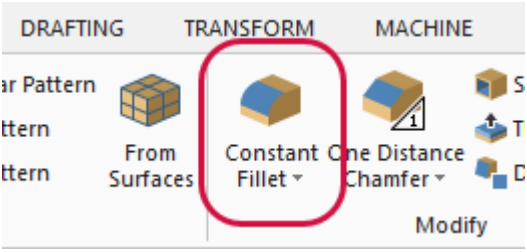


6. Change the draft angle to 10.

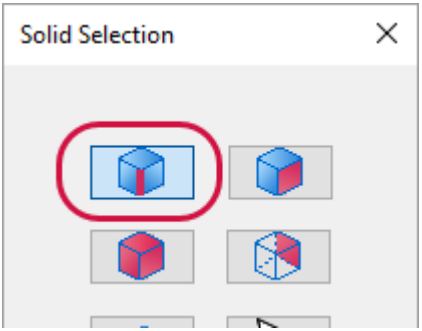
7. Click **OK** to accept the changes.

Adding Outside Fillets

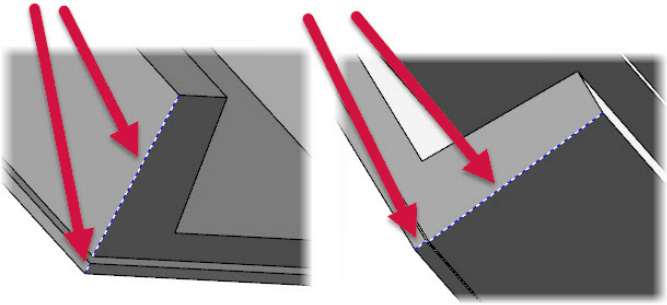
1. In Mastercam's Solids tab, choose **Constant Fillet**.



2. Turn off all filters except edges.

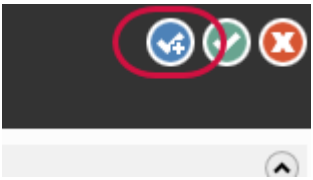


3. Select all 4 of the tray's left outside vertical edges.



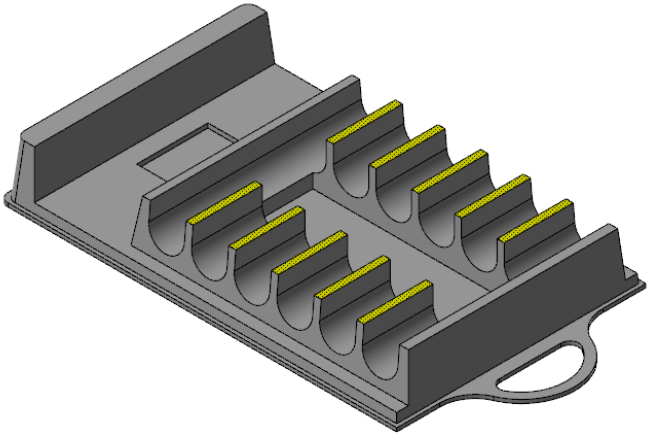
4. Click **OK** in the Solid Selection dialog box, and then set the fillet radius to **5**.

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

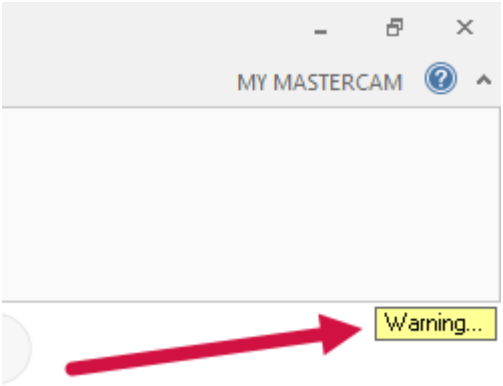


Adding Top Fillets

1. Turn the face selection filter back on, turn off edge selection, select the ten faces shown below, and click **OK** in the Solid Selection dialog box.



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.

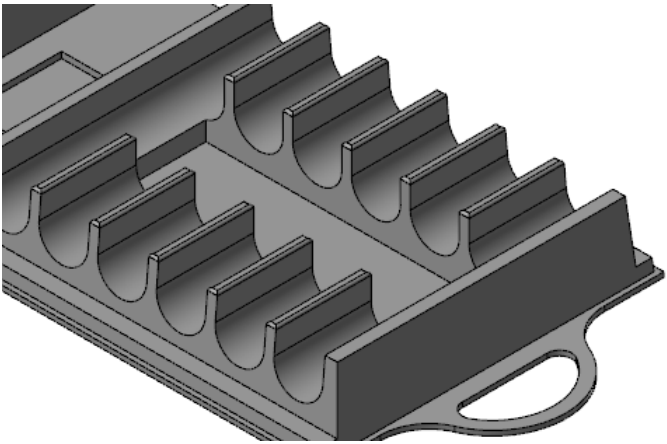


2. Click the warning to open the entire message.

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

3. Reduce the fillet's radius to 1.

The warning closes, and you get the result shown in the following picture.



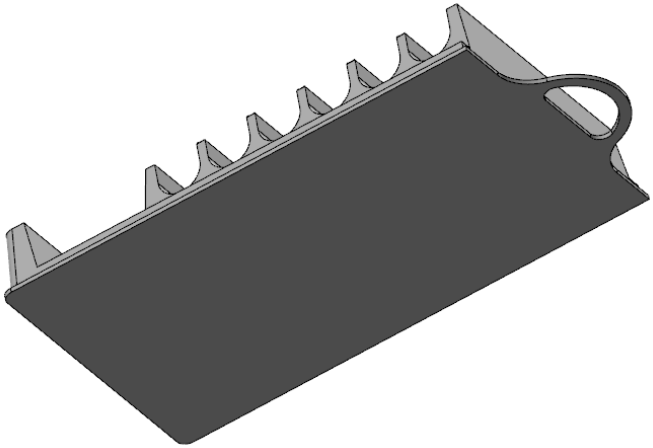
4. Click **OK** to finalize the operation.
5. Save the file as `MarkerTray06-`*XXX*, where *XXX* is your initials.

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.

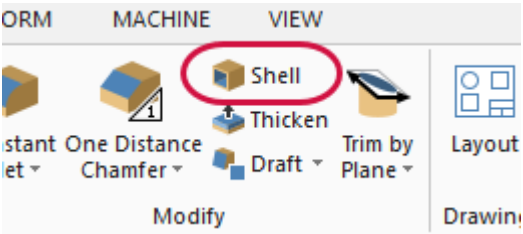
Exercise 7: Shelling the Part

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. If you choose to start with the tutorial file, remember to save it under the new name `MarkerTray06-`*XXX*, where *XXX* is your initials.

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

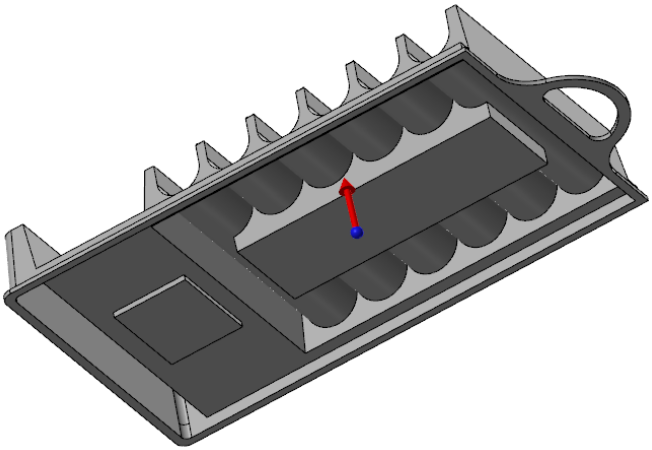


2. From Mastercam's Solids tab, choose **Shell**.



   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 the one for faces.



4. Change the shell thickness for **Direction 1** to 3.
5. Click **OK** to finalize the operation.
6. Save the file as `MarkerTray07-XXX`, where *XXX* is your initials.

   This completes the MarkerTray part.

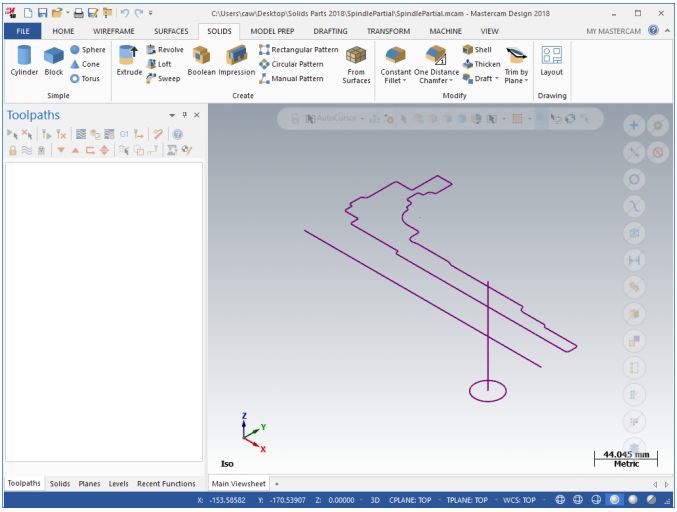
Other Solids Functions

In this section, you try other Mastercam Solids functions, including Revolve, Sweep, and Boolean.

Exercise 8: Using Solids Revolve

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

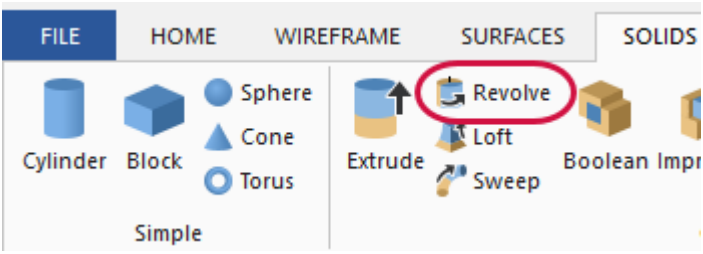
   After loading the part, Mastercam looks like the following picture.



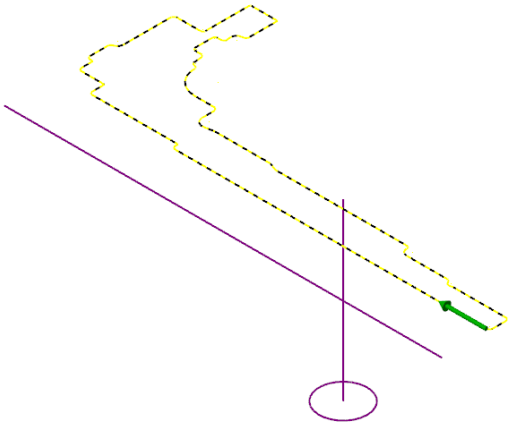
2. So you do not overwrite the original, save the file as `SpindlePartial-`*XXX*, where *XXX* is your initials.

3. From Mastercam's Solids tab, choose **Revolve**.

   The Chaining dialog box opens.



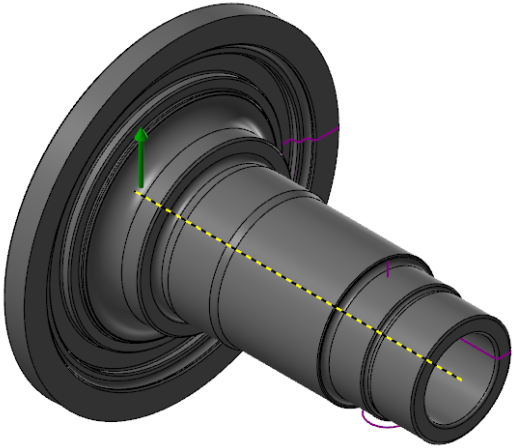
4. Chain the geometry as shown below, and click **OK** in the Chaining dialog box.



In the next steps, you revolve this geometry to create the main part.

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

   Mastercam revolves the geometry around the line you clicked.

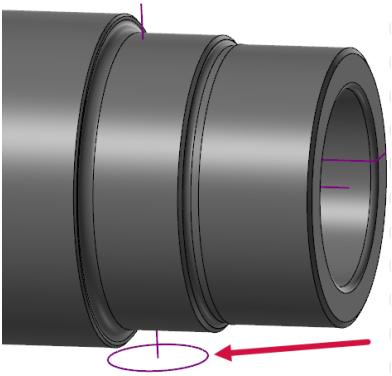


6. Click **OK** to finalize the operation.

7. Save the file as `SpindlePartial01-XXX`, where *XXX* is your initials.

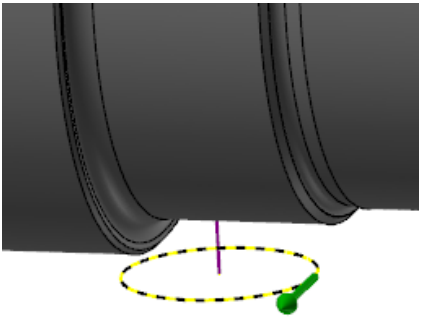
Exercise 9: Creating Solids with Sweep

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

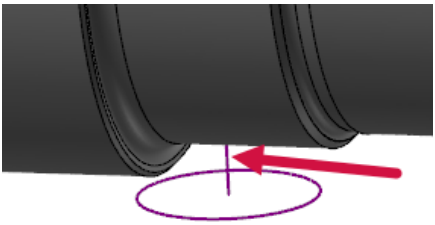


2. From Mastercam's Solids tab, choose **Sweep**.

3. Chain the circle (chaining direction does not matter), and click **OK** in the Chaining dialog box.



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

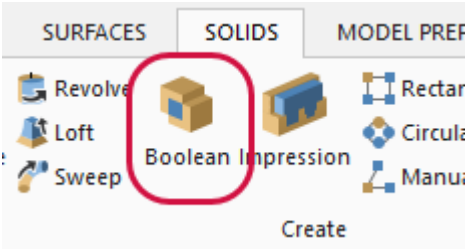


Mastercam sweeps the circle along the line forming the new solid shown in the picture below.

5. Click **OK** to finalize the operation.
6. Save the file as `SpindlePartial02-XXX`, where *XXX* is your initials.

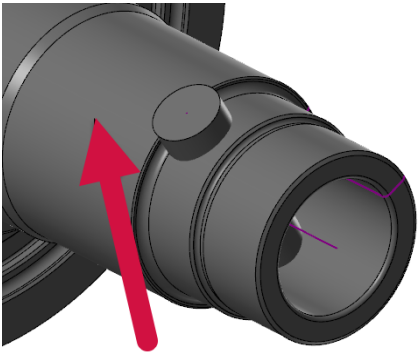
Exercise 10: Using Boolean to Create a Hole

1. From Mastercam's Solids tab, choose **Boolean**.



2. Click the main body of the part. This is the Boolean function's target body.

   The Solid Selection dialog box opens.

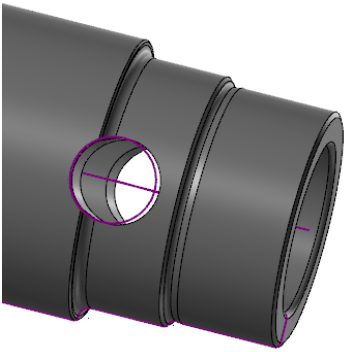


3. Click the solid body you just created, and click **OK** in the Solid Selection dialog box.

   Mastercam sets the selected feature as the tool body, and the Boolean function panel opens.

4. Change the operation type to **Remove**.

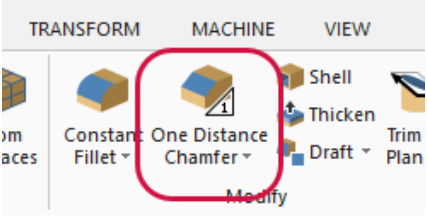
   Mastercam removes all material from the selected area.



5. Click **OK** to finalize the operation.
6. Save the file as `SpindlePartial03-`*XXX*, where *XXX* is your initials.

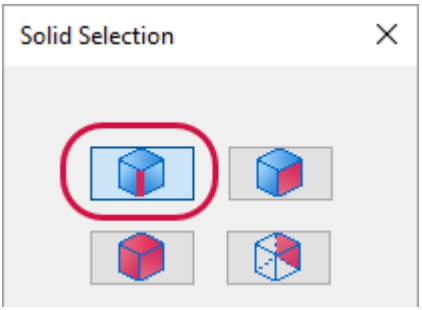
Exercise 11: Adding Chamfers

1. From Mastercam's **Solids** tab, choose **One Distance Chamfer**.

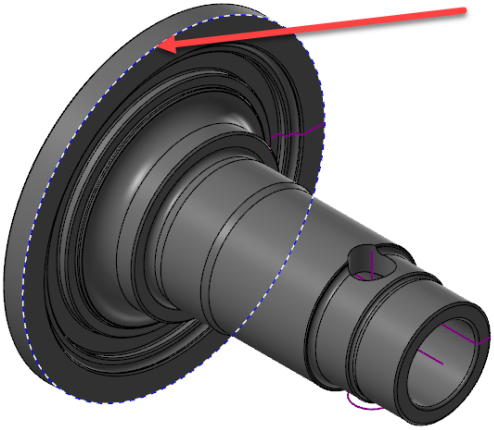


2. In the **Solid Selection** dialog box, turn off all filters except edges.

   Choosing the appropriate filter makes selection easier.

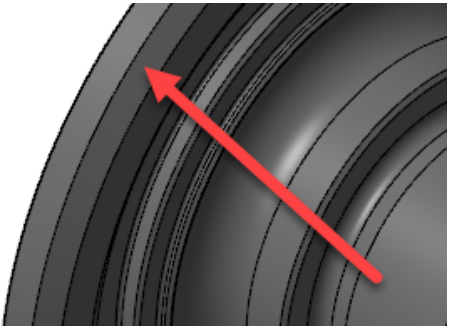


3. Select the edge indicated in the picture below, and then click **OK** in the Solid Selection dialog box.



4. Change the chamfer's distance to 5.

Mastercam creates the specified chamfer.



5. Click **OK** to finalize the operation.
6. Save the file as `SpindlePartial04-XXX`, where *XXX* is your initials.

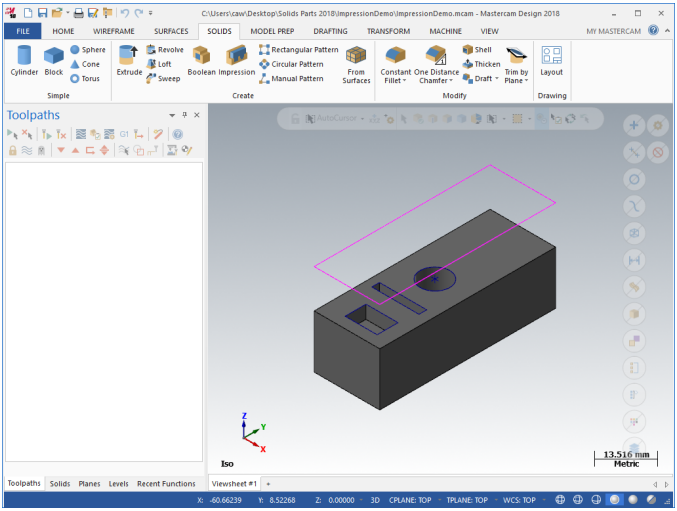
You have now completed the spindle part.

Exercise 12: Using Solid Impression

Use the Solid Impression function to create a new solid body that is a negative impression of a selected closed solid body. To create the impression, you select a planar wireframe chain that the function 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 to a selected solid face.

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

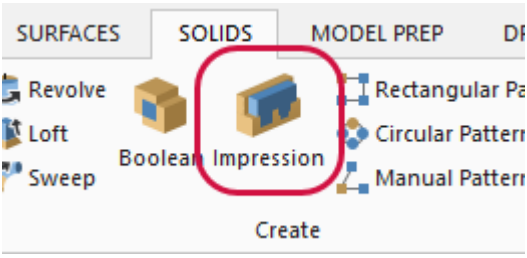
   After loading the part, Mastercam looks like the following picture.



2. So you do not overwrite the original, save the file as `ImpressionDemo-`*XXX*, where *XXX* is your initials.

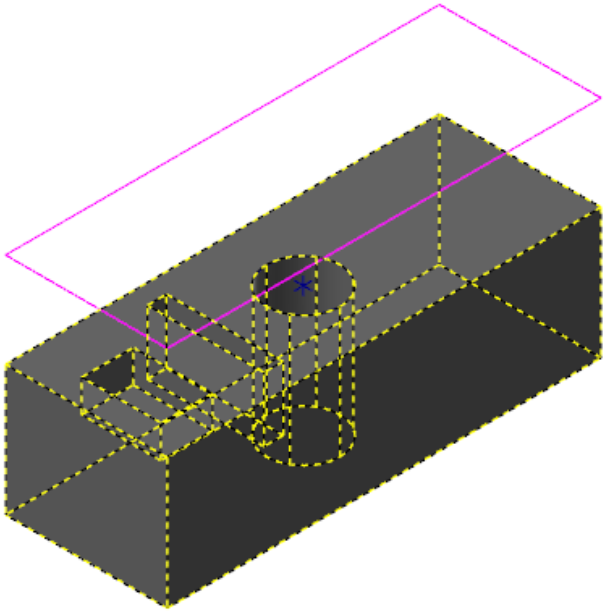
3. In the Solids tab, click **Impression**.

   The Chaining dialog box opens.

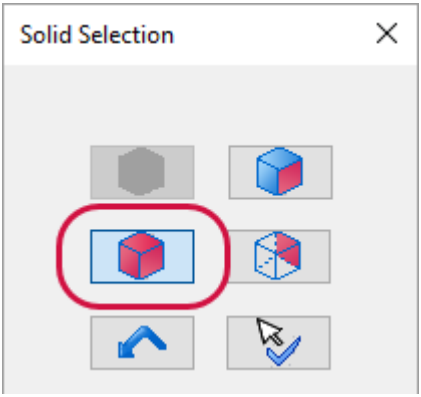


4. Chain the wireframe rectangle that is positioned above the solid.

   The Solid Selection dialog box opens.

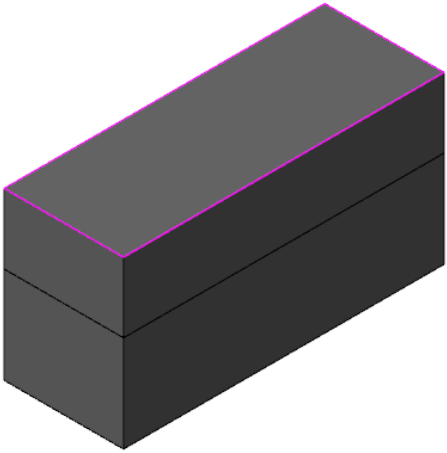


5. Turn off all of the masks except for body.

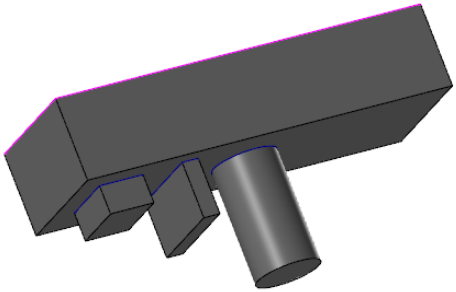


6. In the graphics window, click the solid body, and then click **OK** in the Solid Selection dialog box.

   Mastercam creates a new solid body above the original geometry.



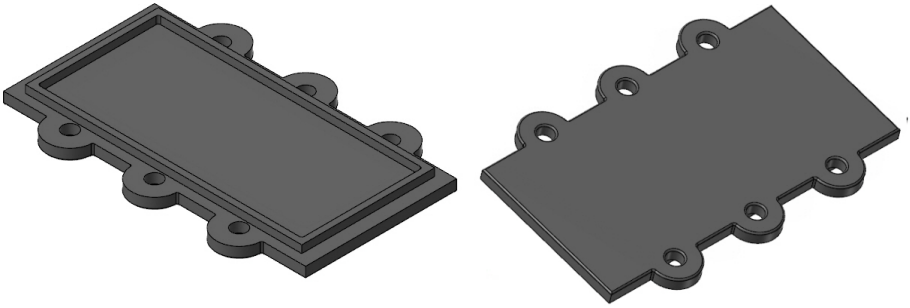
7. Delete the original solid body and rotate the new one to see the solid impression.



You have now completed this part of the tutorial. Test your knowledge with the following challenge exercises.

Challenge

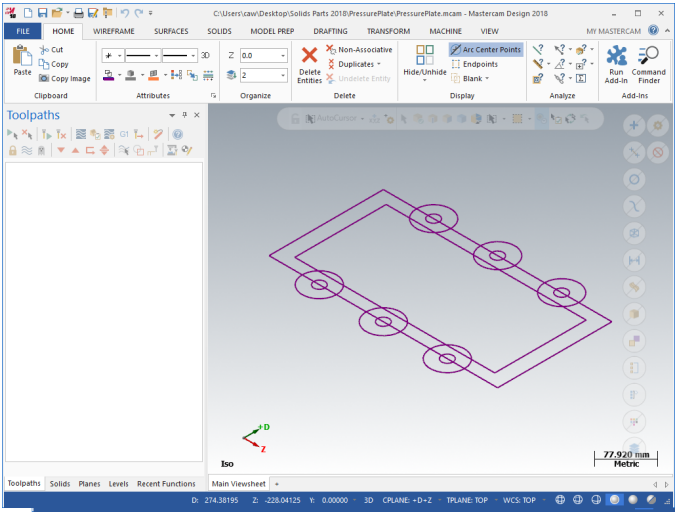
In this section, you test your Solids knowledge and skills by applying Solids functions to supplied geometry to create a pressure plate part. If you have trouble with any step, refer to the videos. The following pictures show two views of the part you will create.



Note: In most steps, the left picture is a *before* screen shot and the right is an *after*.

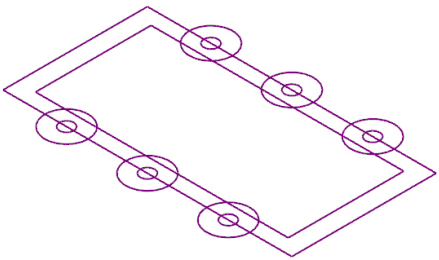
1. Load the file `PressurePlate`, included with this tutorial.

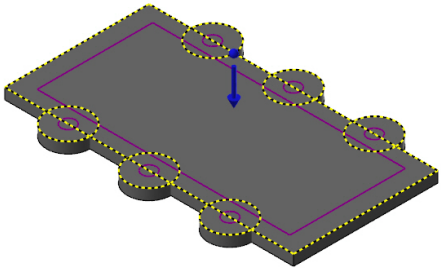
   After loading the part, Mastercam should look like the following picture.



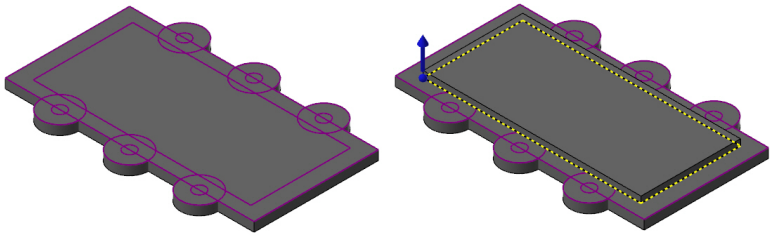
2. So you do not overwrite the original, save the file as `PressurePlate-`*XXX*, where *XXX* is your initials.

3. Extrude the outside rectangle and the six large circles downward by **15** mm.



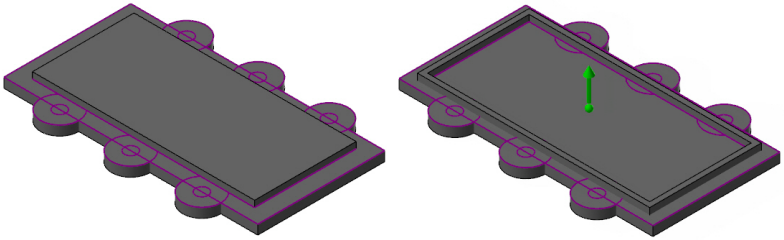


4. Extrude the inner rectangle upward by **10** mm.



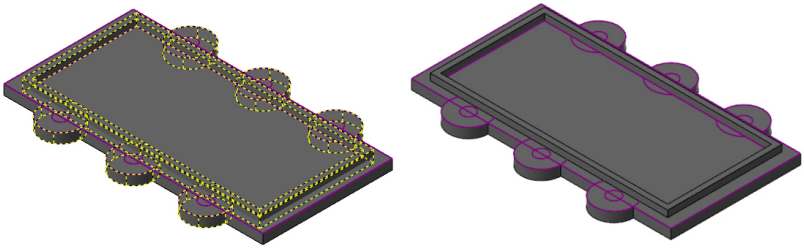
Click here to watch a video of Steps 3 and 4.

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



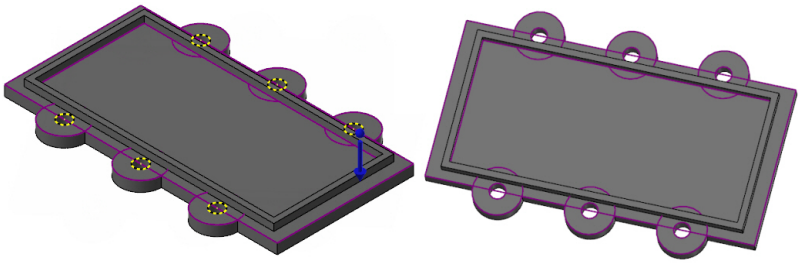
Click here to watch a video of Step 5.

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



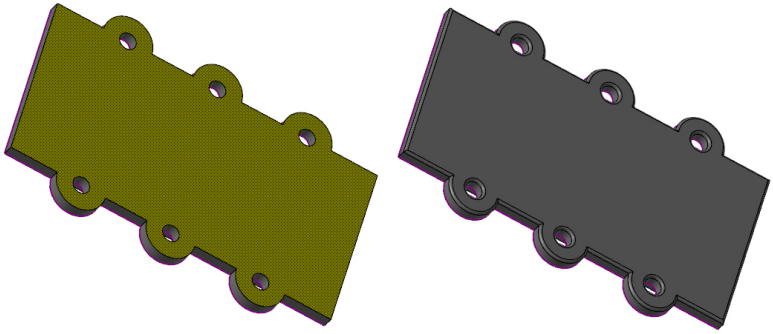
Click here to see a video of Step 6.

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



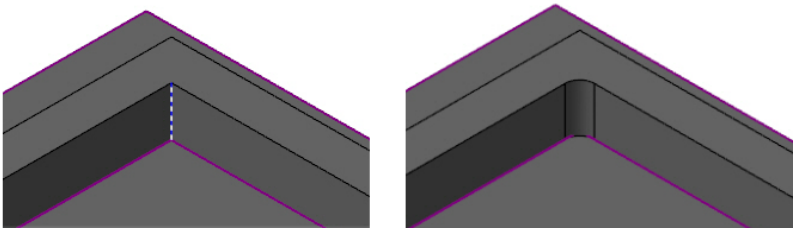
Click here to watch a video of Step 7.

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



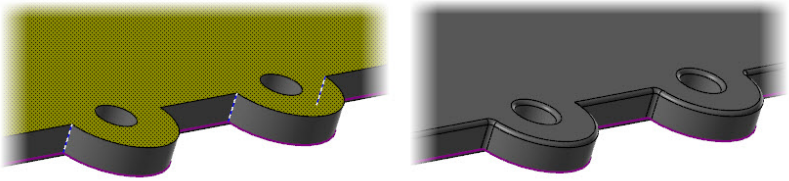
Click here to watch a video of Step 8.

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



Click here to watch a video of Step 9.

10. Use the solids tree to change the **3** mm bottom fillet to **2** mm and to add **2** mm fillets to the 12 vertical edges of the rounded tabs.



Click here to watch a video of Step 10.

2

Introduction to Model Prep

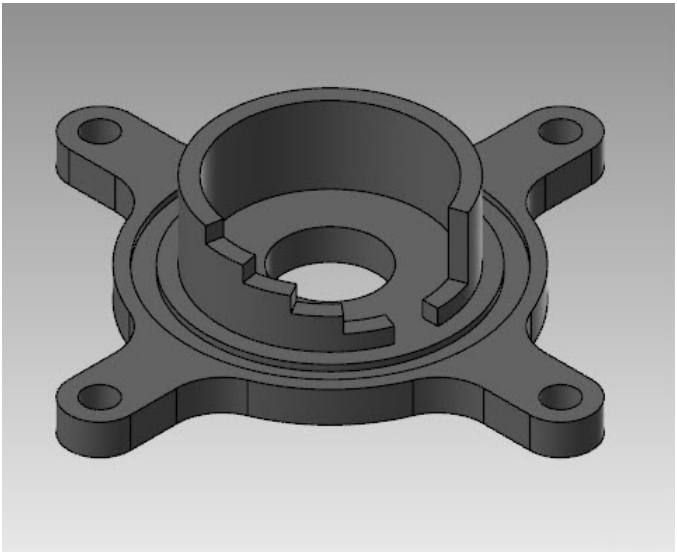
Model Prep provides functions, such as Push/Pull, Move, and Split Solid Faces, that let you edit solids with no history tree. When you use Model Prep functions, Mastercam recognizes edges, faces, and even complete features, giving you the power to make changes without the operations or wireframe used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

Getting Ready to Work

In this lesson, you work on the file named `FlangeDemo`, which is included with this tutorial. This is the only file you need to complete the lesson. However, a set of files named `FlangeDemo01` through `FlangeDemo10` are also included with this tutorial and are the results of each of the exercises. For example, `FlangeDemo01` is the results of Exercise 1 and so can be used when starting Exercise 2 . If you have problems with an exercise, you can load the appropriate file and continue with the tutorial.

The Scenario

This lesson is based on the part file shown in the following picture.



In the scenario, the part file requires changes based on new specifications. The required changes are as follows:

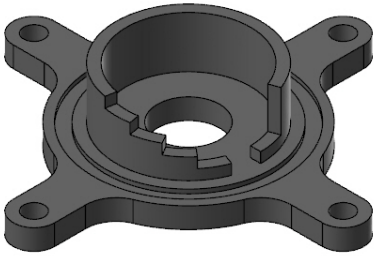
- Remove one of the feet.
- Reduce the foot height by 5 mm.
- Reposition the three remaining feet to be 120 degrees apart.
- Increase the foot hole diameter to 12 mm.
- Reduce the part's center hole diameter to 20 mm and add a 30 mm counter bore, 2 mm deep.
- Align the opening on the vertical wall with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

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`.

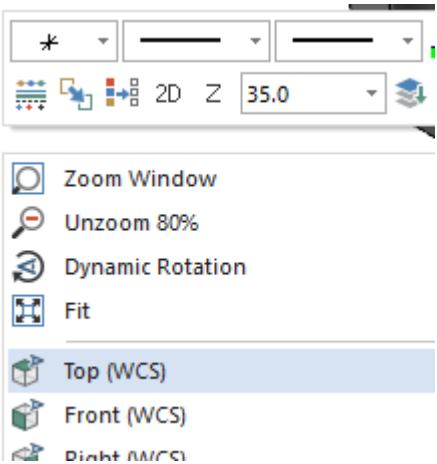
   The part should look like the picture below.



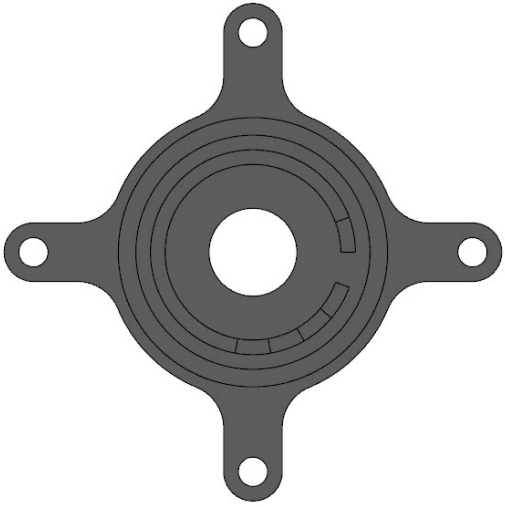
2. Save the file as `FlangeDemo-`*XXX*, where *XXX* is your initials.

   Saving the file under a new name ensures that you do not accidentally over-write the original.

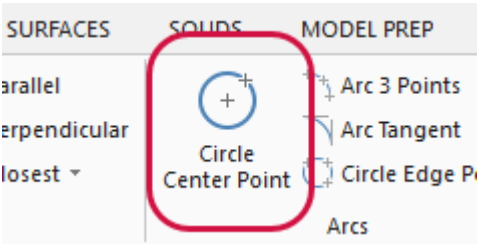
3. Right-click in the graphics area, and choose **Top (WCS)** from the context menu.



Mastercam displays the part in the top view.

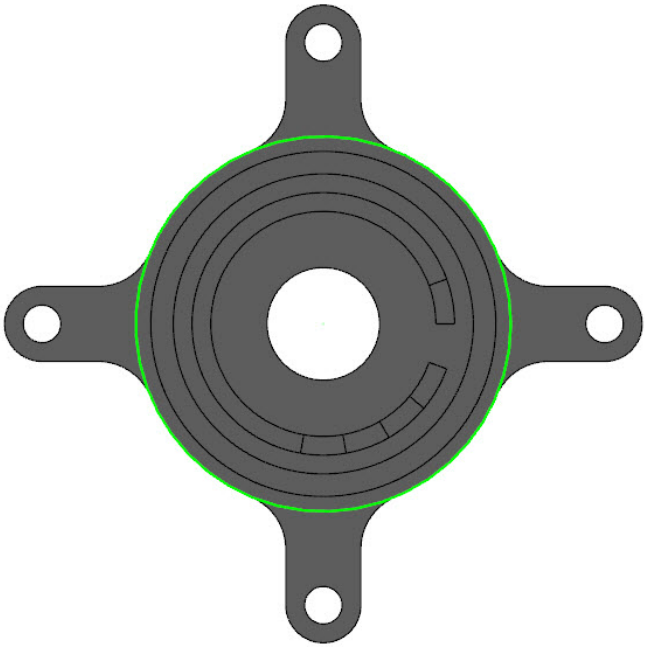


4. In Mastercam's Wireframe tab, choose **Circle Center Point**.



5. Draw a circle centered on the part, with a diameter (100 mm) 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.

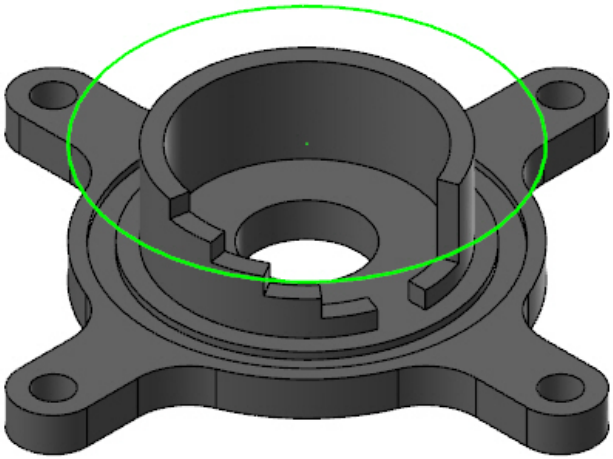


6. From Mastercam's **Model Prep** tab, choose **Split Solid Faces**.

This function splits single faces into multiple faces that you can manipulate separately.

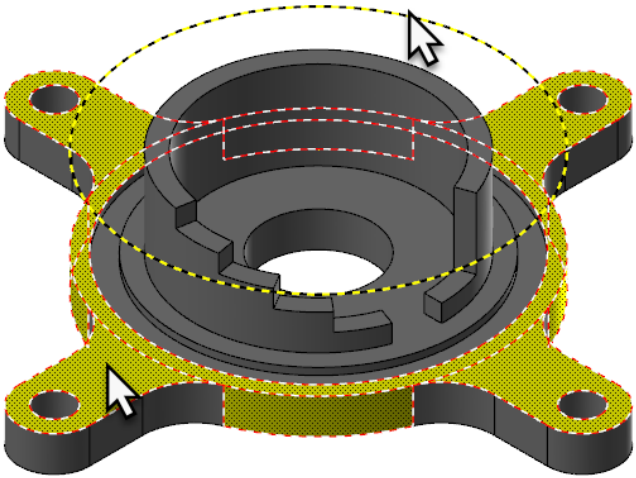
7. Right-click in the graphics area, and choose **Isometric (WCS)**.

   Mastercam displays the part in isometric view.

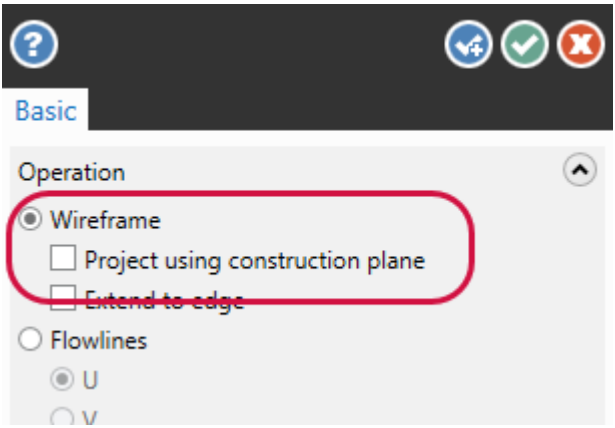


8. Click the circle and one of the feet.

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

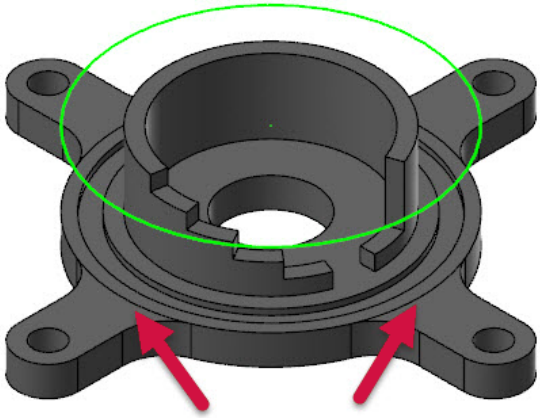


9. In the Split Solid Face function panel, make sure **Wireframe** is selected and **Project using construction plane** is not selected.

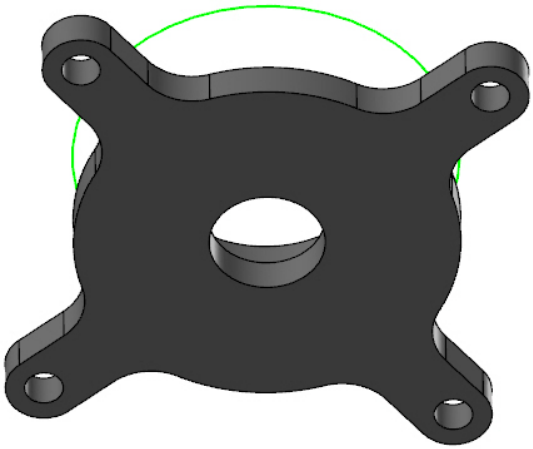


10. Click **OK and Create New Operation** to finalize your choices and begin a new Split Solid Face operation.

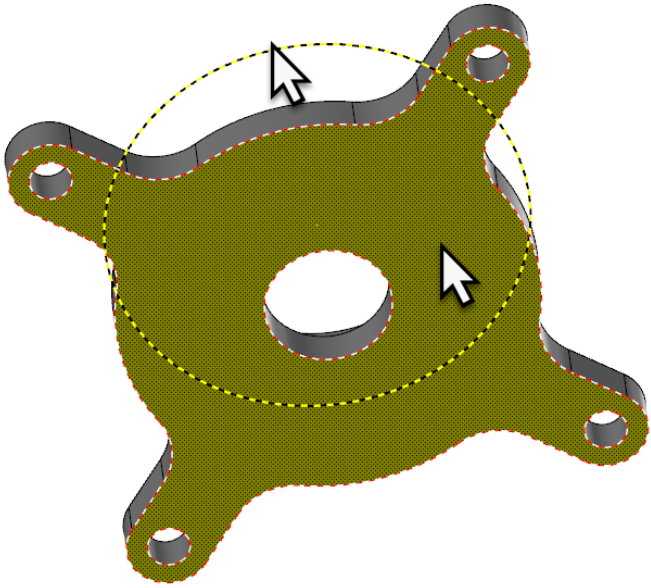
Mastercam splits the top face of the feet from the rest of the part. (The arrows in the picture indicate the split on the front feet.)



11. Hold down your mouse wheel and move the mouse to flip the part as shown in the picture.

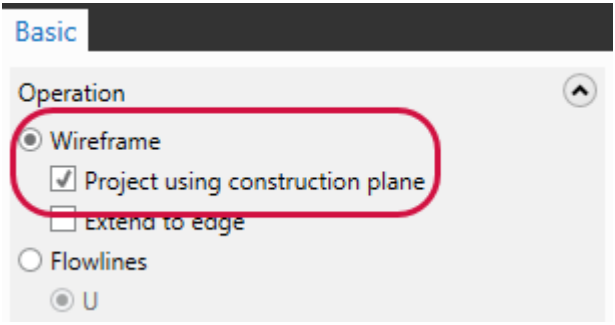


12. Select the bottom of the part and the circle.



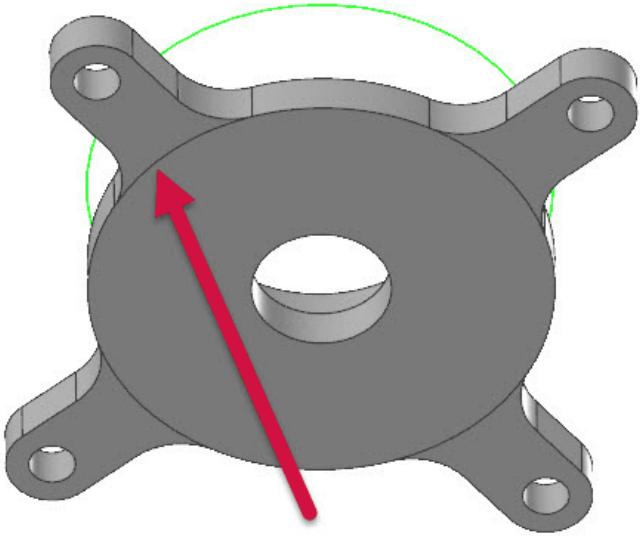
13. In the Split Solid Face function panel, ensure **Wireframe** and **Project using construction plane** are selected.

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



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

Mastercam splits the face.



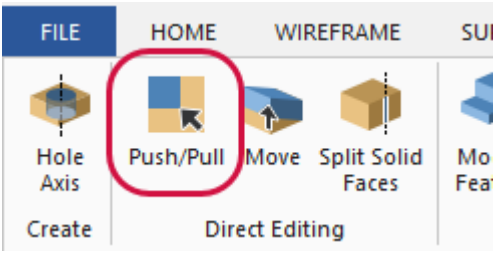
15. Delete the circle, and save the file as `FlangeDemo01-`*XXX*, where *XXX* is your initials.

Exercise 2: Removing One Foot

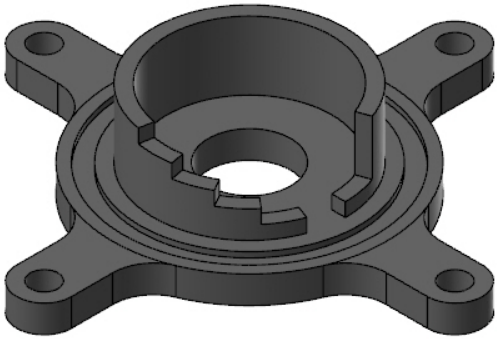
With the feet split from the rest of the part, you can remove one of them, leaving three as required by the modification instructions. For this exercise, continue with the part you just saved, or load `FlangeDemo01`, which accompanies this tutorial. If you choose to start with the tutorial file, remember to save it under the new name `FlangeDemo01-`*XXX*, where *XXX* is your initials.

1. From Mastercam's Model Prep tab, choose **Push/Pull**.

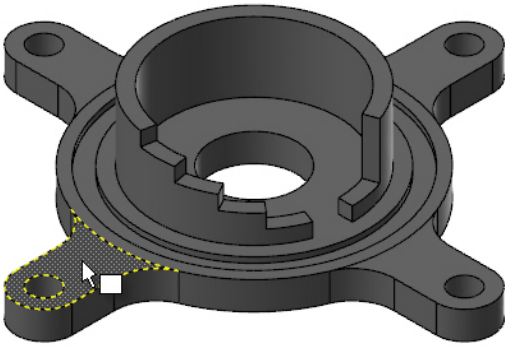
   The Push/Pull function extends bosses or produces cuts based on selected faces or features. You can also use Push/Pull to create fillets from edges or to remove fillets.



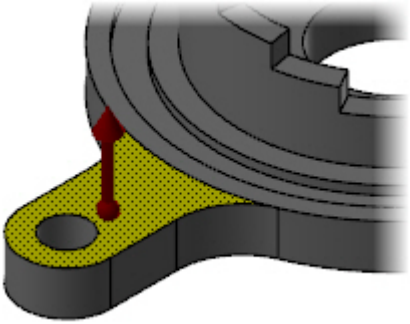
2. Place the part into isometric view.



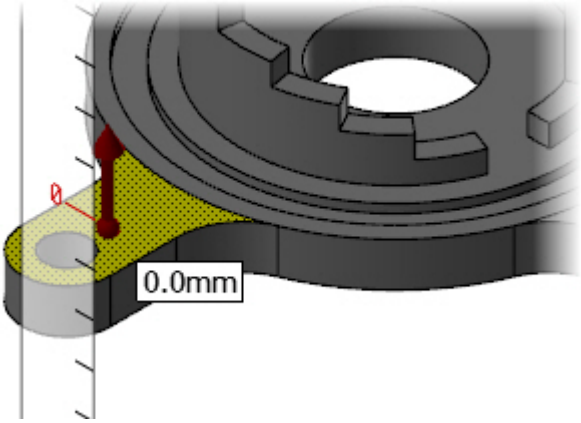
3. Click the lower-left foot, as shown.



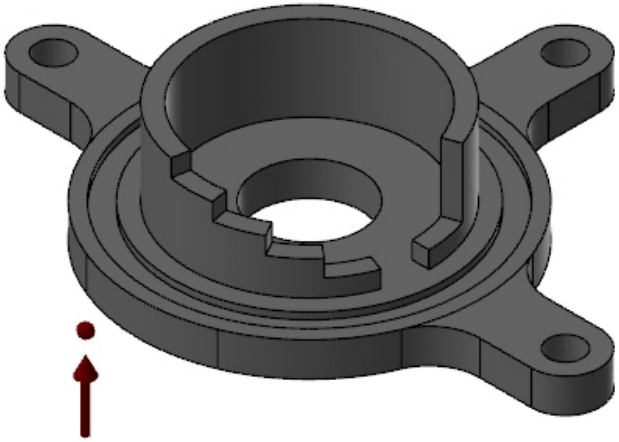
This is the foot you will remove from the part. Mastercam highlights the foot and places a red single-axis arrow control on its face.



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



Your part should look like the picture below.



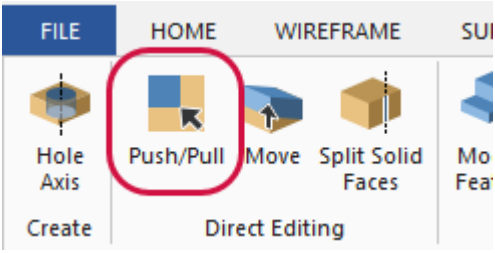
5. Click **OK** to finalize the operation.

6. Save the file as `FlangeDemo02-`*XXX*, where *XXX* is your initials.

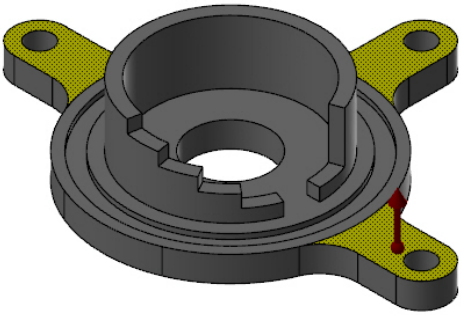
Exercise 3: Reducing the Foot Height

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. If you choose to start with the tutorial file, remember to save it under the new name `FlangeDemo02-`*XXX*, where *XXX* is your initials.

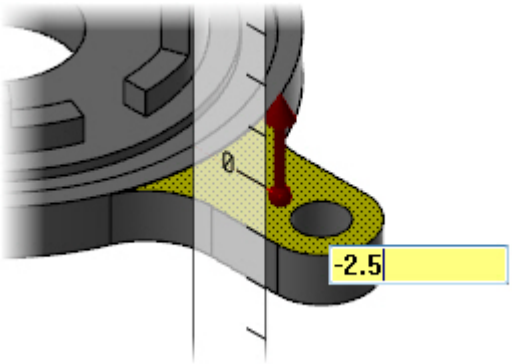
1. From Mastercam's Model Prep tab, choose **Push/Pull**.



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



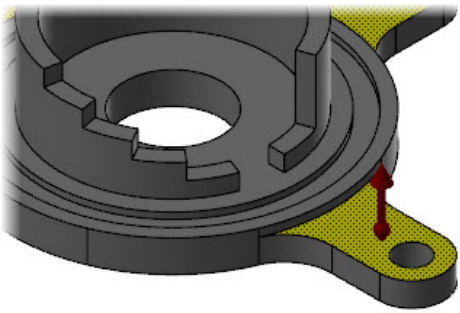
3. Click the red arrow, and type **-2.5**.



With the Push/Pull function, you can type values or, as in the previous exercise, drag the arrow with your mouse.

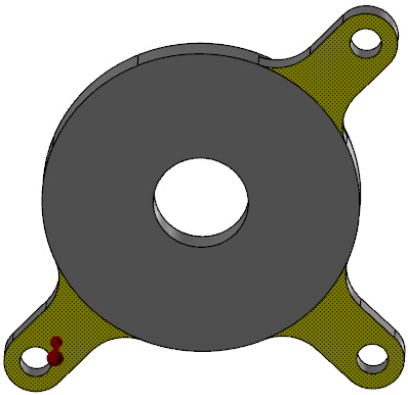
4. Press [**Enter**] twice.

Mastercam reduces the foot height by 2.5 mm.

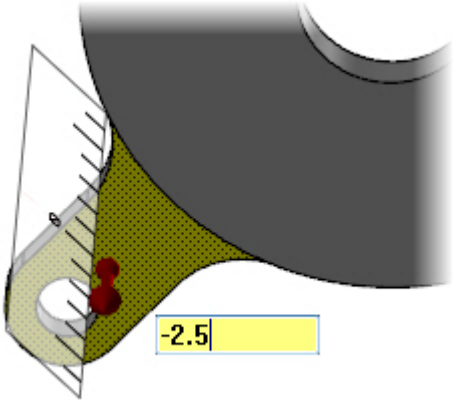


Note: The first time you press [**Enter**], Mastercam shows the result and gives you a chance to type another value. The second time, Mastercam accepts the entry, and closes the entry field.

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 type **-2.5** into the box that appears.



8. Press [**Enter**] twice.

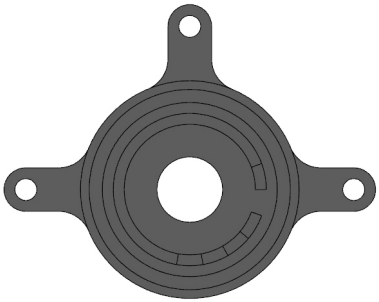
   Mastercam reduces the foot height by 2.5 mm, leaving the feet centered on the rim, as you can see by rotating the part to view the edge.

9. Click **OK** to finalize the operation.
10. Save the file as `FlangeDemo03-`*XXX*, where *XXX* is your initials.

Exercise 4: Repositioning the Feet

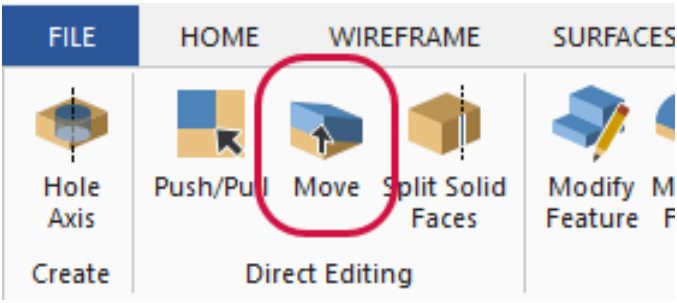
The next step is to reposition the feet so they are equidistant from each other. For this exercise, continue with the part you just saved, or load `FlangeDemo03`, which accompanies this tutorial. If you choose to start with the tutorial file, remember to save it under the new name `FlangeDemo03-`*XXX*, where *XXX* is your initials.

1. Place the part into top view.

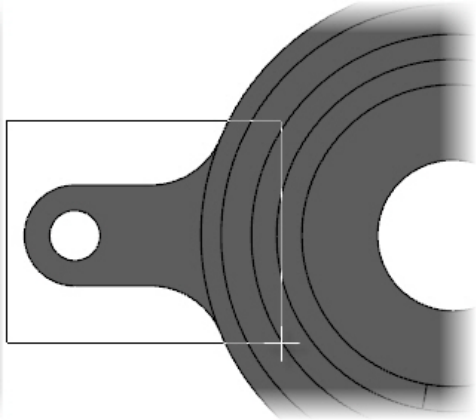


2. From the **Model Prep** tab, choose **Move**.

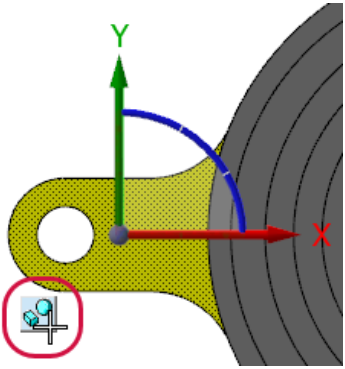
   The Move function moves (and sometimes resizes), rotates, or copies solid features or faces.



3. Use window selection to select all of the faces of the left-hand foot. Rotate the view and check that you did not miss any faces.



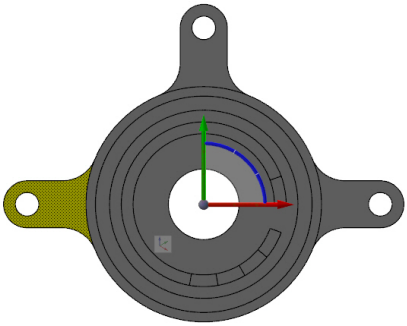
4. Click the gnomon's button to change from geometry manipulation mode to gnomon manipulation mode.



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

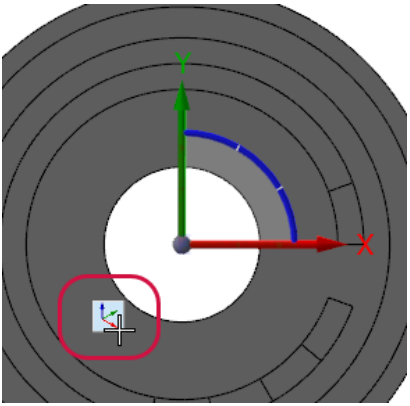
5. Click the ball at the gnomon's origin, and place the gnomon at the center of the part.

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

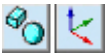


6. Click the gnomon's button again, to change from gnomon manipulation mode back to geometry manipulation mode.

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

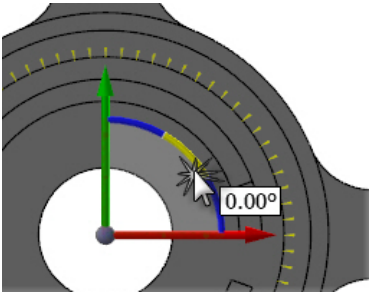


   Note: The image shown on the gnomon's button indicates the gnomon's mode. In the pictures below, the left icon indicates geometry manipulation mode, and the right icon indicates gnomon manipulation mode.



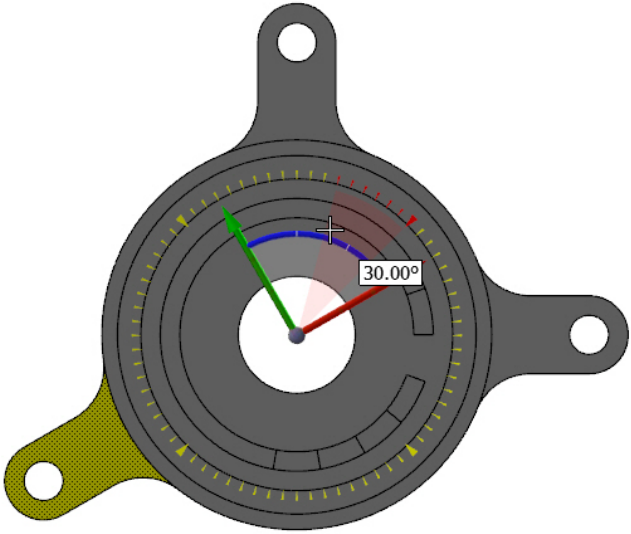
7. On the gnomon, click the center segment of the blue control.

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



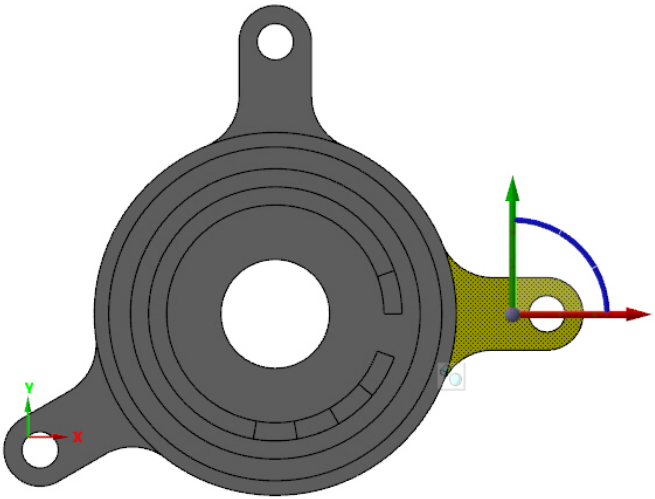
8. Move your mouse, rotating the gnomon 30 degrees counter-clockwise, and then click to accept the rotation.

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

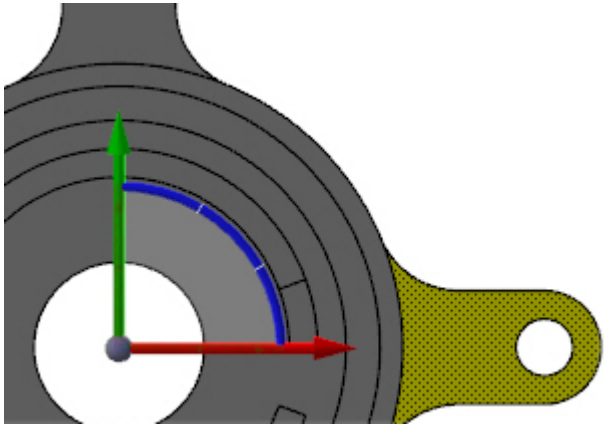


9. In the Move function panel, click **OK and Create New Operation** to finalize your changes and begin a new Move operation.

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

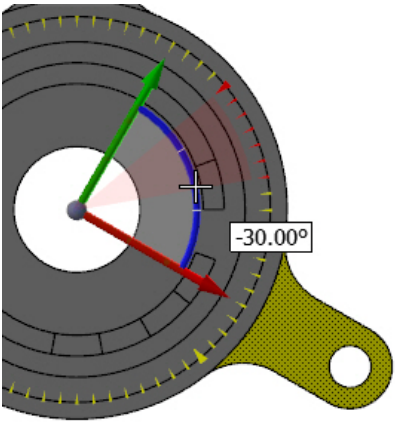


11. Switch to gnomon manipulation mode, and move the gnomon to the center of the part.



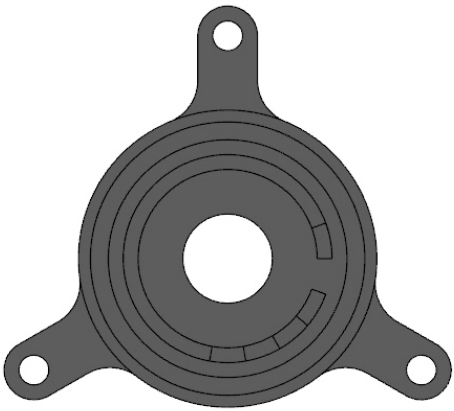
12. Switch back to geometry manipulation mode, and rotate the gnomon 30 degrees clockwise (-30 degrees).

    Mastercam rotates the foot into its correct position.



13. In the function panel, click **OK** to finalize the rotation.

    Your part should now look like the picture below.

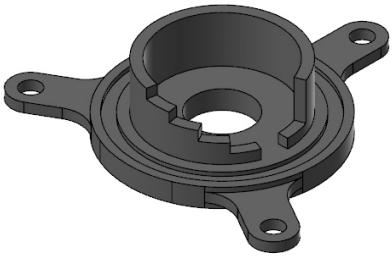


14. Save the file as `FlangeDemo04-XXX`, where *XXX* is your initials.

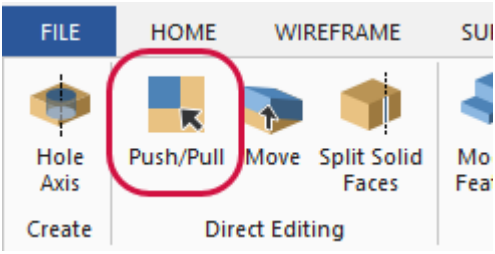
Exercise 5: Resizing the Foot Holes

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. If you choose to start with the tutorial file, remember to save it under the new name `FlangeDemo04-XXX`, where *XXX* is your initials.

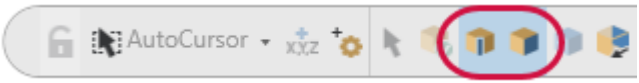
1. Change the part's view to isometric.



2. From the **Model Prep** tab, choose **Push/Pull**.

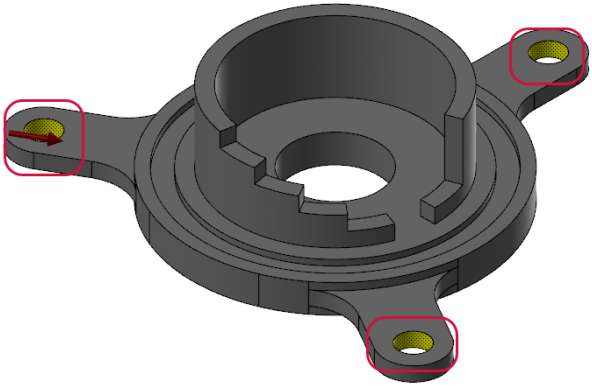


3. In Mastercam's Selection Bar, turn off the **Edge Selection** filter. Only **Face Selection** should be on.

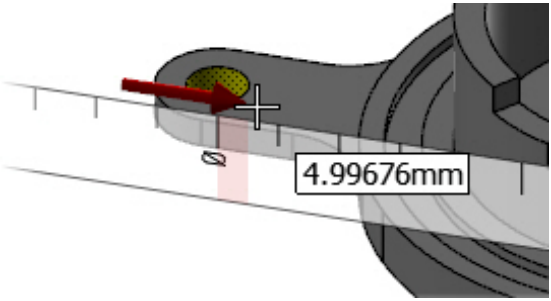


You can now more easily select the holes without accidentally selecting the hole edges.

4. As shown in the picture below, click all three foot holes to select them as geometry for the Push/Pull function.

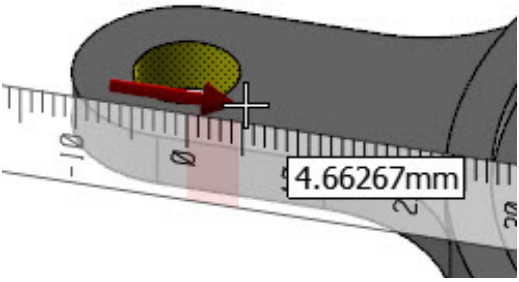


5. Click the red arrow to display the ruler control.

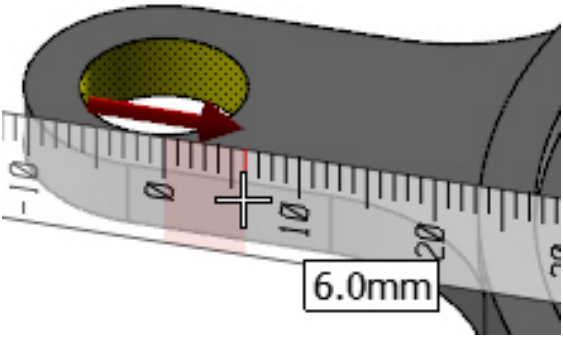


6. Use your mouse wheel to zoom in on the part until the ruler shows finer increments.

   The closer you zoom, the smaller the ruler's increments.



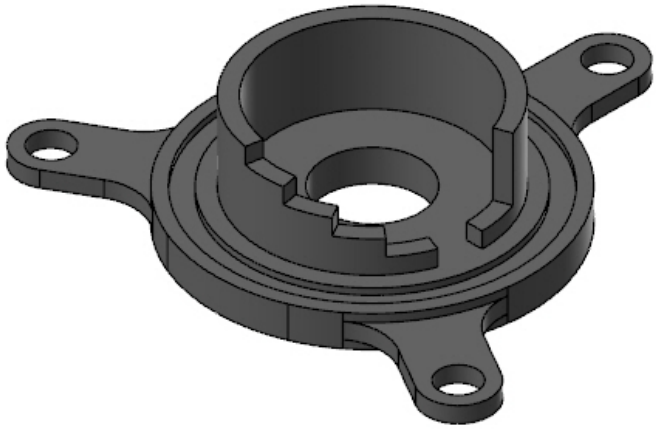
7. Move your mouse pointer over the ruler, and click when the hole's radius equals **6** mm.



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.

8. In the function panel, click **OK** to finalize your change and end the function.

   The holes are now 12 mm in diameter, as shown in the picture below.

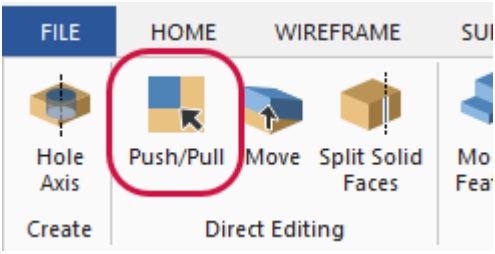


9. Save the file as `FlangeDemo05-`*XXX*, where *XXX* is your initials.

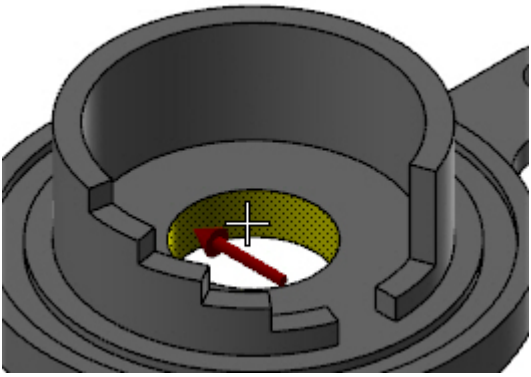
Exercise 6: Creating the Counter bore

The specifications state that your part's center hole be reduced to 20 mm. You must also add a counter bore. You can use Model Prep to make these changes. For this exercise, continue with the part you just saved, or load `FlangeDemo05`, which accompanies this tutorial. If you choose to start with the tutorial file, remember to save it under the new name `FlangeDemo05-`*XXX*, where *XXX* is your initials.

1. From the Model Prep tab, choose **Push/Pull**.

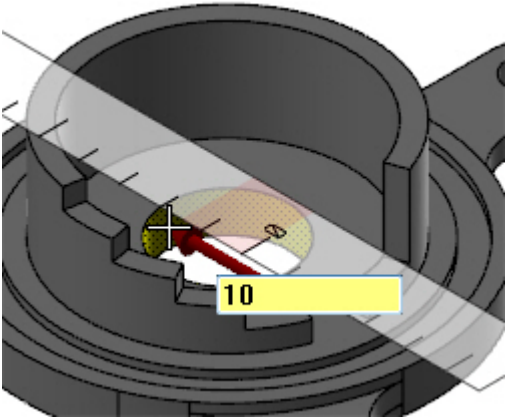


2. Click the center hole to select it as geometry for the Push/Pull function.



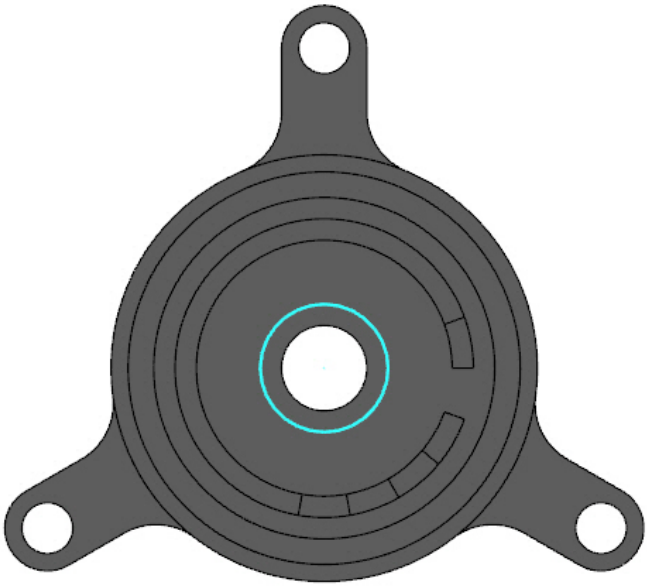
3. Click the red arrow, type **10**, and press [**Enter**] twice.

   Mastercam changes the hole's diameter to 20 mm.

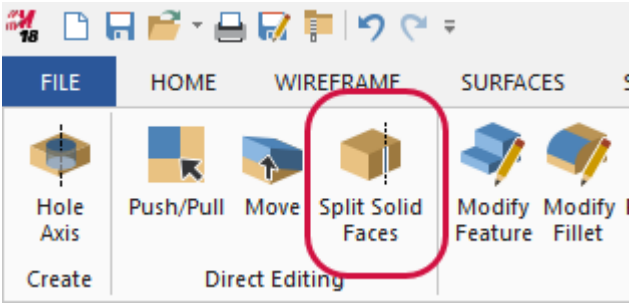


4. In the function panel, click **OK** to finalize the change.
5. In Top view, create a **30** mm diameter circle at the center of the part. (Use **Circle Center Point**.)

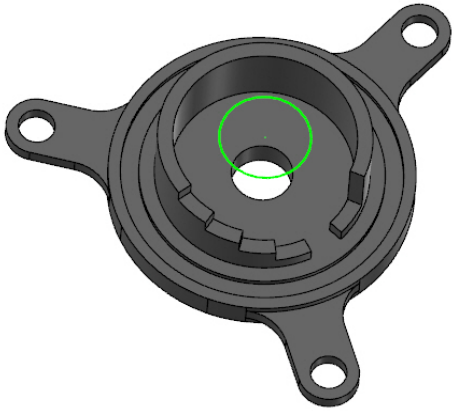
You will use this circle to split the part's inner face at the diameter needed for the counter bore.



6. From the Model Prep tab, choose **Split Solid Faces**.

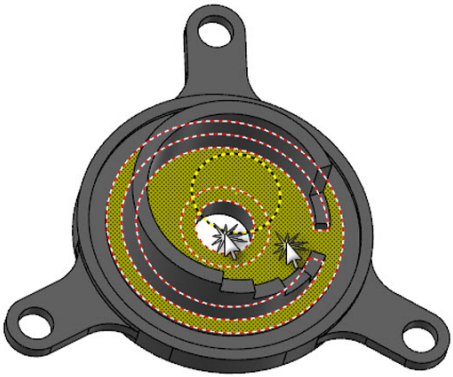


7. Rotate the part's view until it looks similar to the picture.



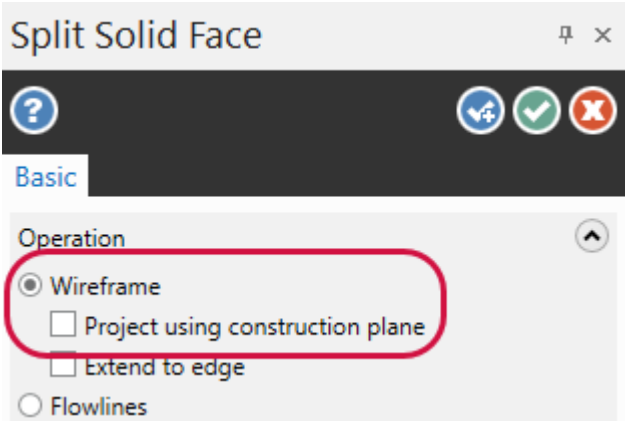
If the circle is laying completely on top of the solid, you would have to turn off the face filter in general selection to pick the circle. When the circle is positioned over empty space it can be selected without changing the filter.

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

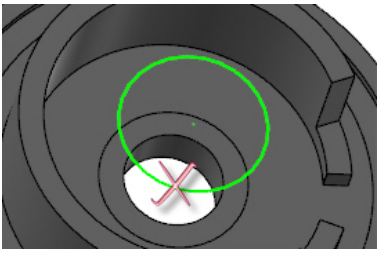


9. In the function panel, make sure that **Wireframe** is selected, and **Project using construction plane** is deselected. Then click **OK**.

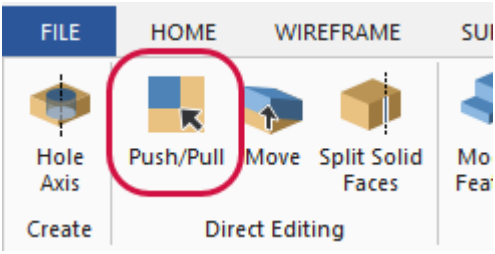
Mastercam splits the face.



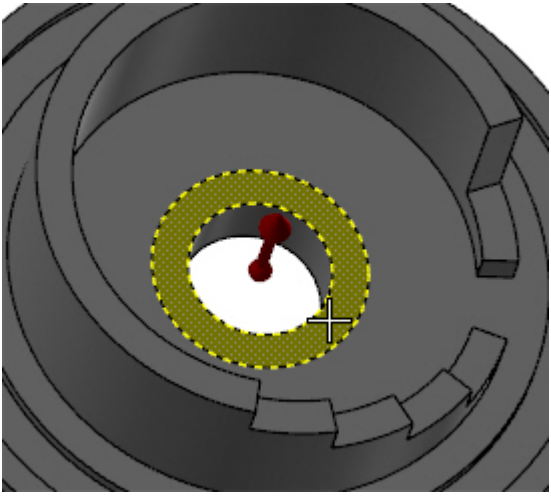
10. Delete the circle, as it is no longer needed.



11. From the Model Prep tab, choose **Push/Pull**.

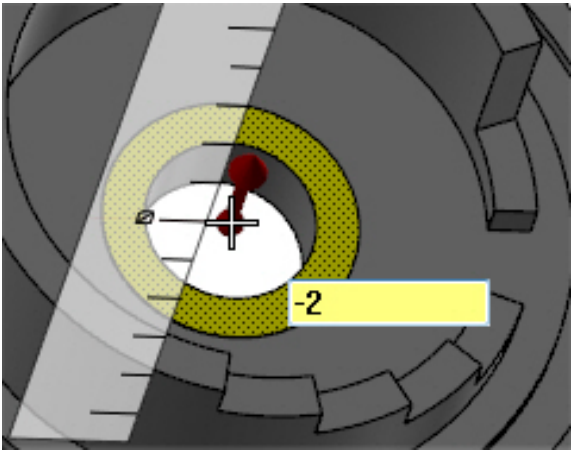


12. Click the portion of the newly split face shown in the picture.



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

    Mastercam creates the counter bore.



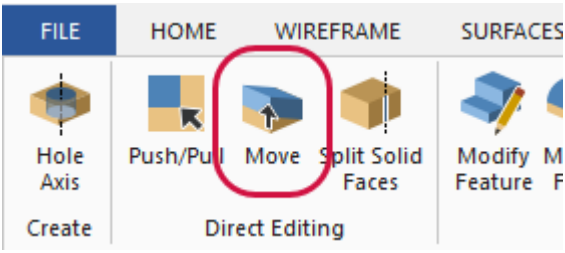
14. In the function panel, click **OK**.

15. Save the file as `FlangeDemo06-`*XXX*, where *XXX* is your initials.

Exercise 7: Aligning the Vertical Wall

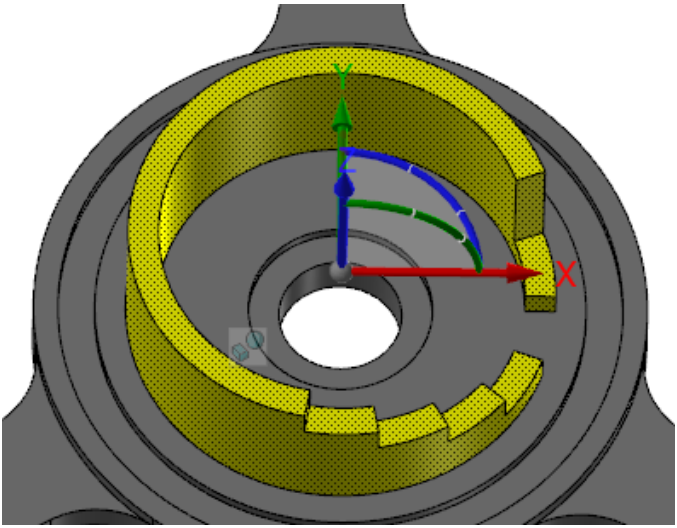
Next you will rotate the vertical wall so the opening aligns with one of the feet. For this exercise, continue with the part you just saved, or load `FlangeDemo06`, which accompanies this tutorial. If you choose to start with the tutorial file, remember to save it under the new name `FlangeDemo06-`*XXX*, where *XXX* is your initials.

1. From the Model Prep tab, choose **Move**.



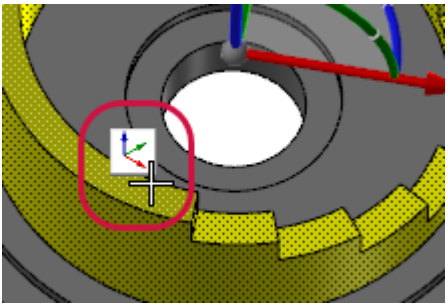
2. Double-click the top face of the vertical wall, as shown.

   Mastercam selects the entire feature.

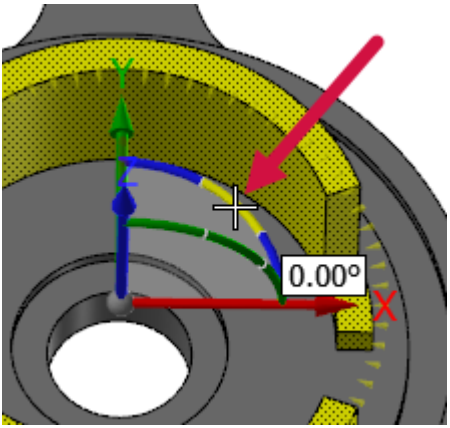


Note: A single click picks a face, and a double-click picks a feature. Hold down [**Shift**] while selecting to pick tangent faces.

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



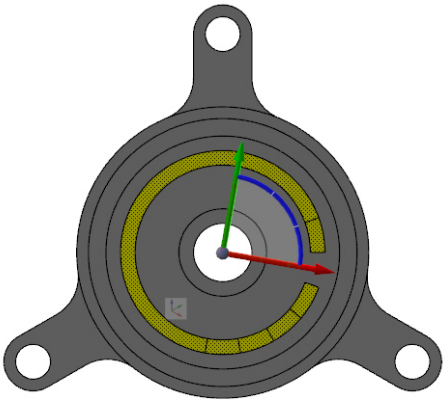
4. Click the center segment of the gnomon's blue arc.



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

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

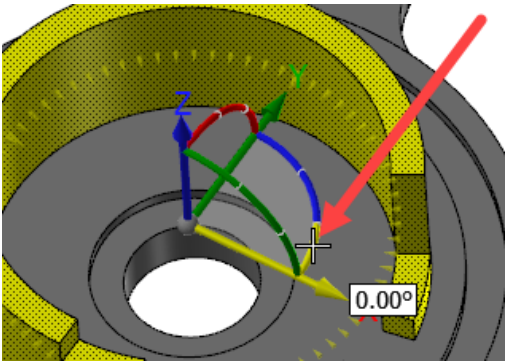
The gnomon's X axis rotates to center on the opening in the vertical wall, as shown (in top view).



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

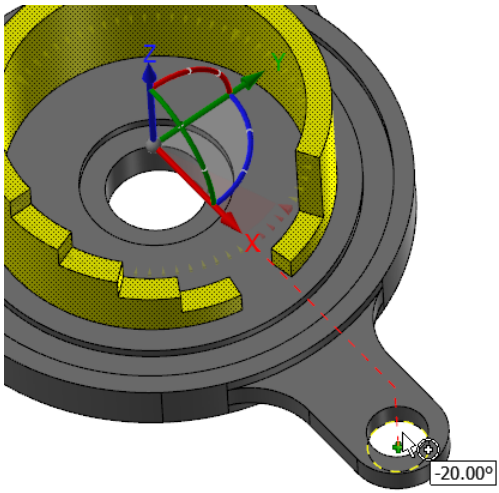
   You are now in 3D rotation mode.

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



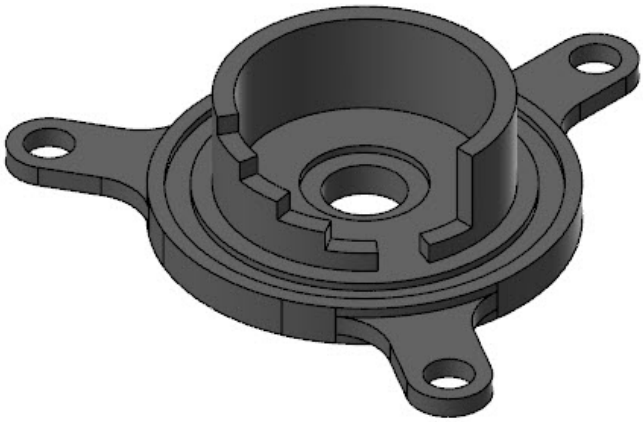
8. Move your mouse cursor to the center of the hole in the lower foot, as shown.

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.



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

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

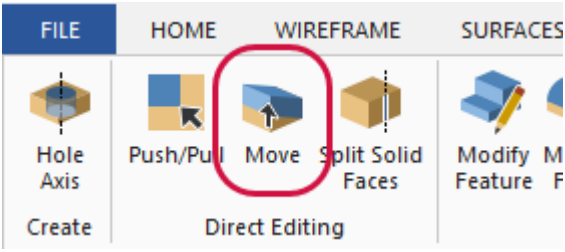


10. Save the file as `FlangeDemo07–XXX`, where *XXX* is your initials.

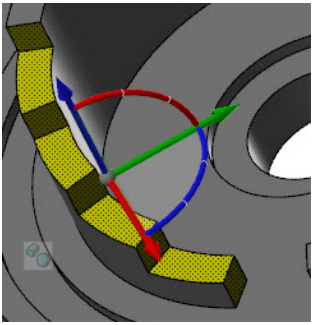
Exercise 8: Expanding the Wall Opening

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. If you choose to start with the tutorial file, remember to save it under the new name `FlangeDemo07–XXX`, where *XXX* is your initials.

1. From the Model Prep tab, choose **Move**.

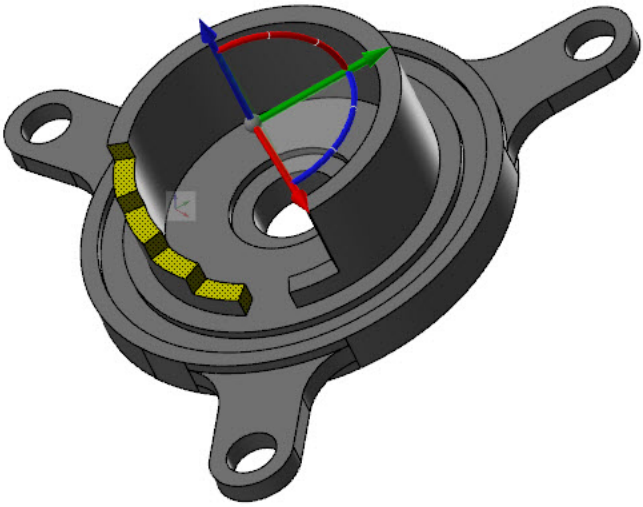


2. Select all nine faces of the step-like portion of the wall.

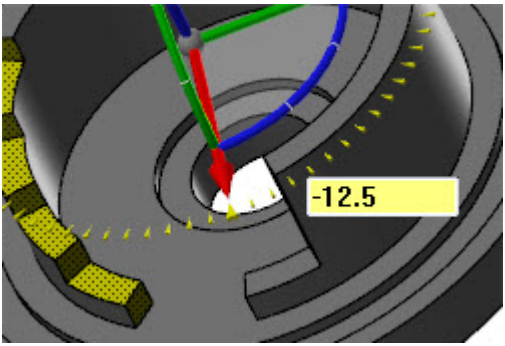


3. Click the gnomon's button to select gnomon manipulation mode, and place the gnomon at the center of the part.

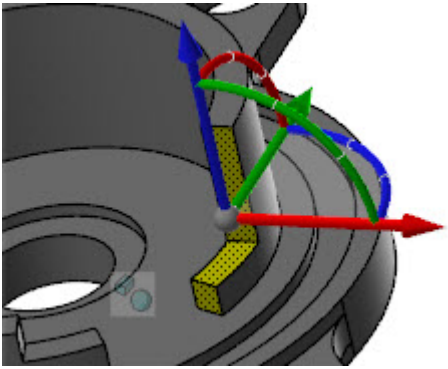
   The gnomon is now set to the center of rotation.



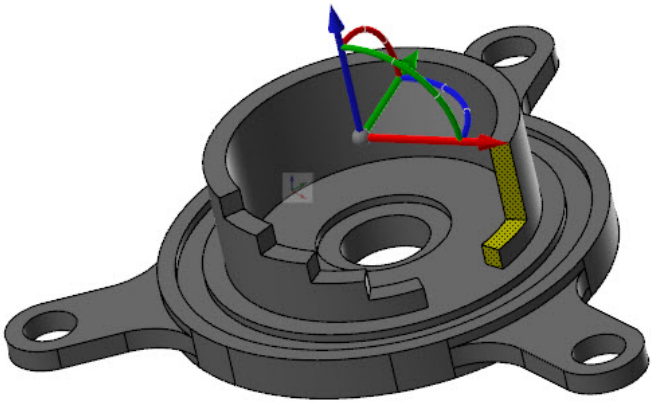
4. Switch back to geometry manipulation mode, and rotate the geometry **-12.5** degrees.



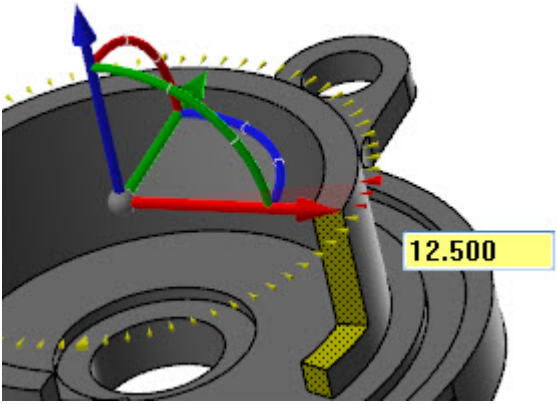
5. In the function panel, click **OK and Create New Operation**.
6. Select all three faces of the opposite side of the wall opening.



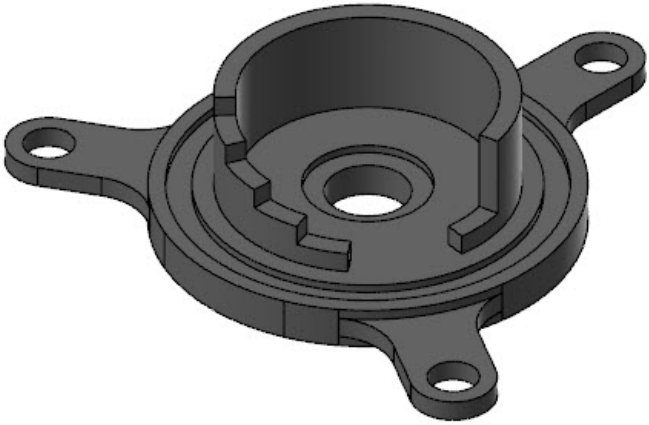
7. Click the gnomon's button to select gnomon manipulation mode, and place the gnomon at the center of the part.



8. Switch back to geometry manipulation mode, rotate the selected geometry **12.5** degrees, and click **OK** in the 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. Your part should now look like the picture below.

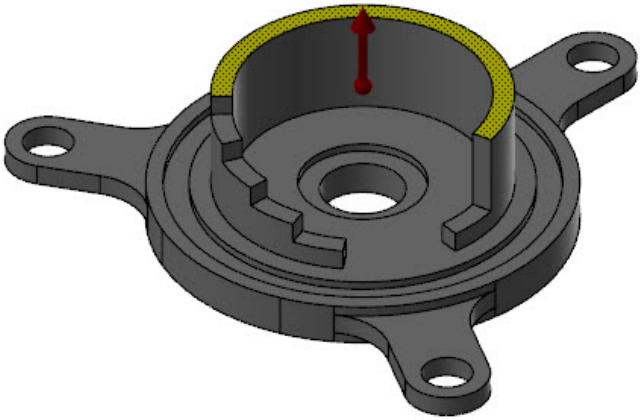


9. Save the file as `FlangeDemo08-XXX`, where *XXX* is your initials.

Exercise 9: Increasing the Wall Height

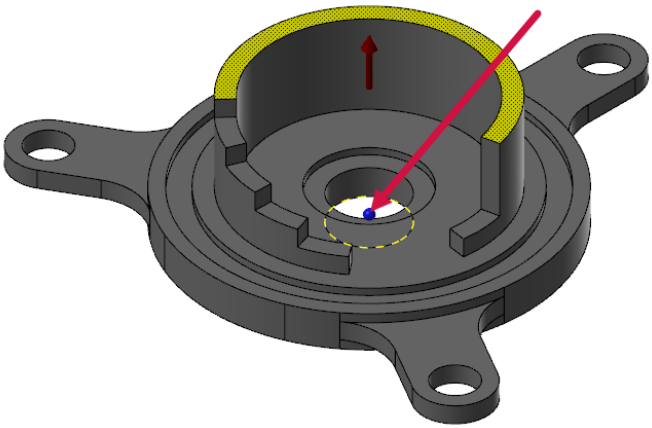
In this exercise, you increase the height of the vertical wall. Continue with the part you just saved, or load `FlangeDemo08`, which accompanies this tutorial. If you choose to start with the tutorial file, remember to save it under the new name `FlangeDemo08-`*`XXX`*, where *XXX* is your initials.

1. Choose **Model Prep, Push/Pull**, and select the top face of the wall.



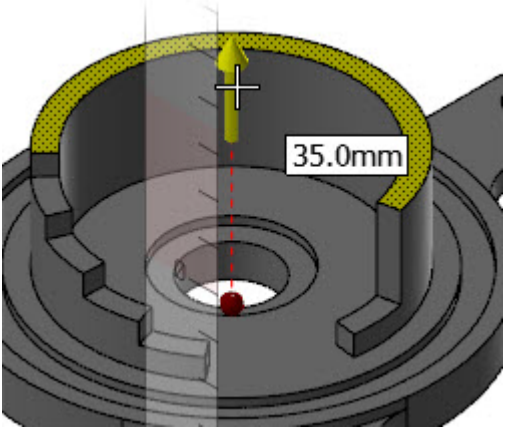
2. Drag the ball on the bottom of the arrow to the bottom of the part, as shown in the picture below.

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



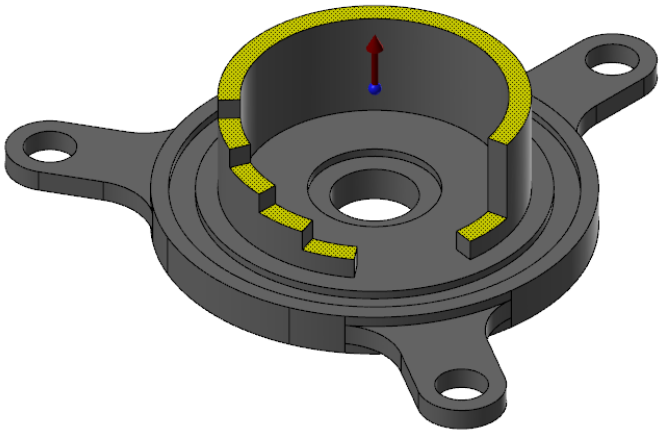
3. Place the mouse pointer over the arrow.

   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.

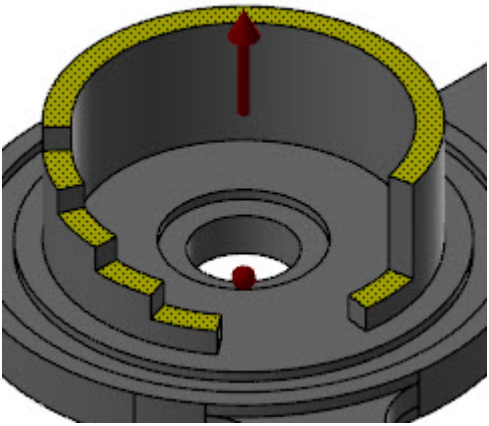


4. Select all of the wall's six top faces, *selecting the largest one last*, which places the arrow in the correct orientation.

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



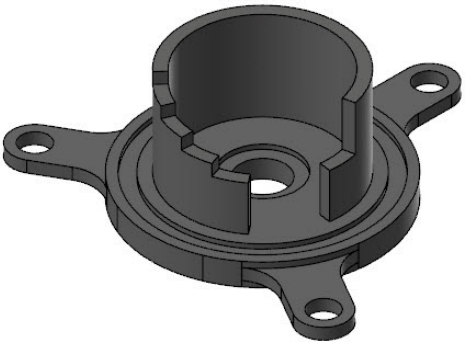
5. Drag the ball back to the bottom.



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

6. Click the red arrow, type **50** followed by [**Enter**] twice, and click **OK** in the function panel.

Mastercam changes the wall's height to 50 mm, measured from the bottom of the part.

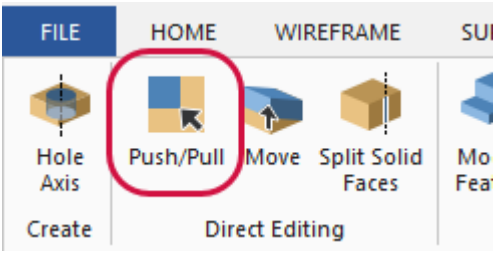


7. Save the file as `FlangeDemo09-`*XXX*, where *XXX* is your initials.

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. If you choose to start with the tutorial file, remember to save it under the new name `FlangeDemo09-`*XXX*, where *XXX* is your initials.

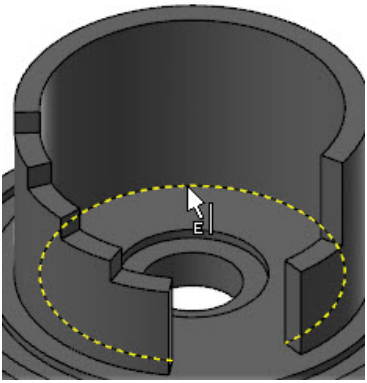
1. Choose **Model Prep**, **Push/Pull**.



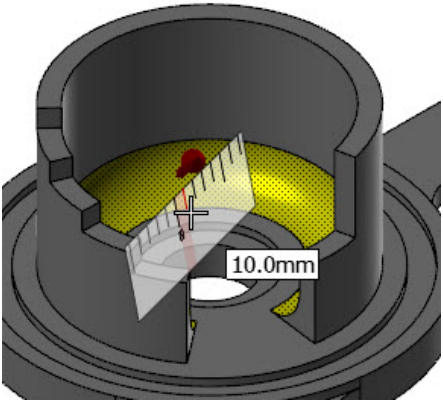
2. In Mastercam's Selection Bar, turn off the face selection filter, and turn on the edge selection filter.

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

   This is where you will create the fillet.

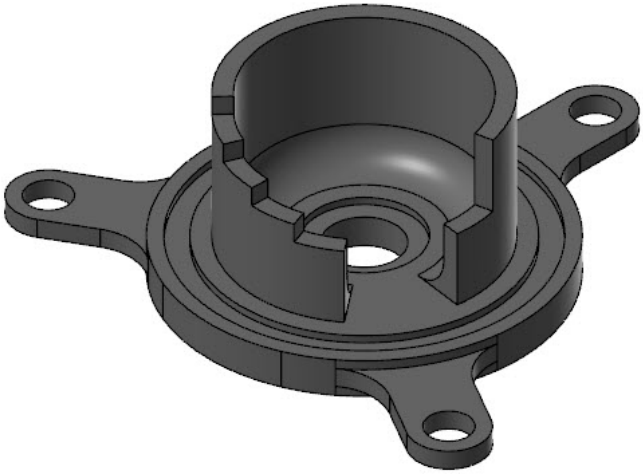


4. Click the red arrow, drag to create a 10 mm fillet, and click to accept the change.



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

   Your finished part should look like the picture below.



6. Save the file as `FlangeDemo10-XXX`, where *XXX* is your initials.

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

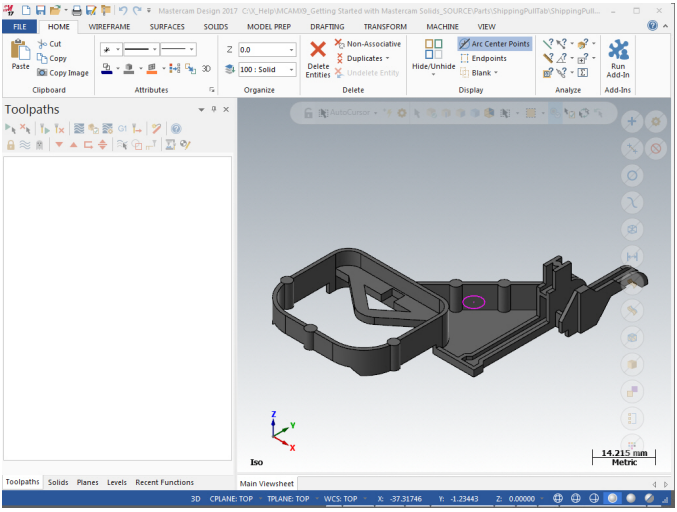
Challenge

In this section, 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.

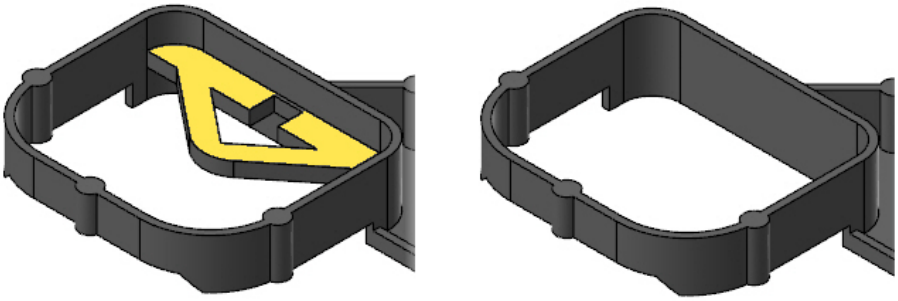
Note: In most steps, the left picture is a *before* screen shot and the right is an *after*.

1. Load the file `ShippingPullTab`, included with this tutorial. The picture below shows the part as it appears in Mastercam.



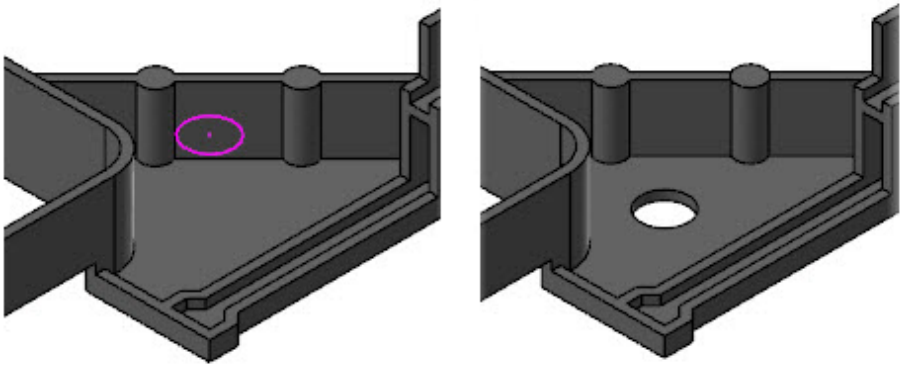
2. To avoid overwriting the original, save the file as `ShippingPullTab-`*XXX*, where *XXX* is your initials.

3. Remove the flange shown below.



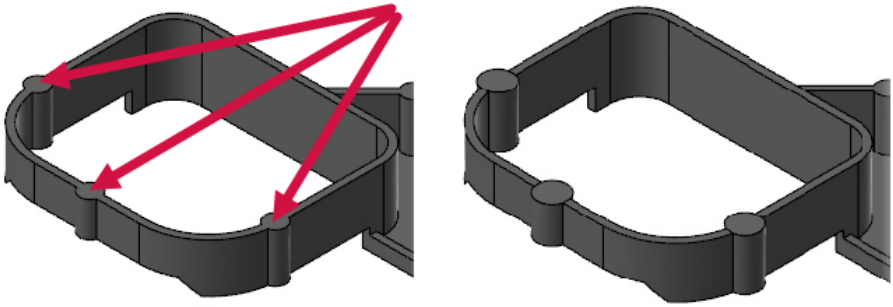
Click here to watch a video of Step 3.

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



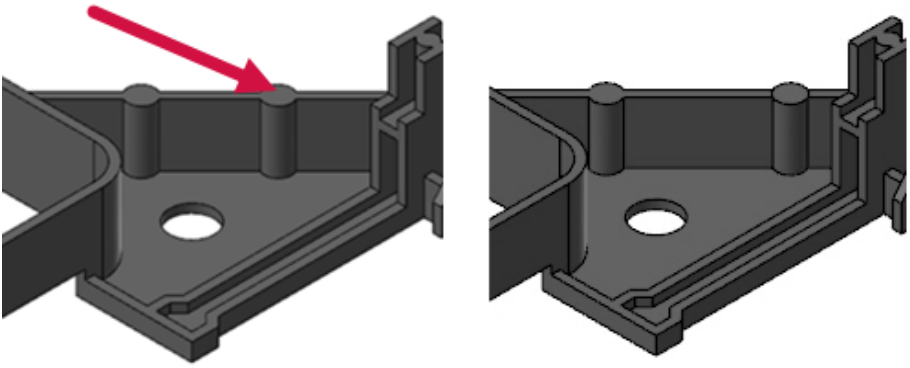
Click here to watch a video of Step 4.

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



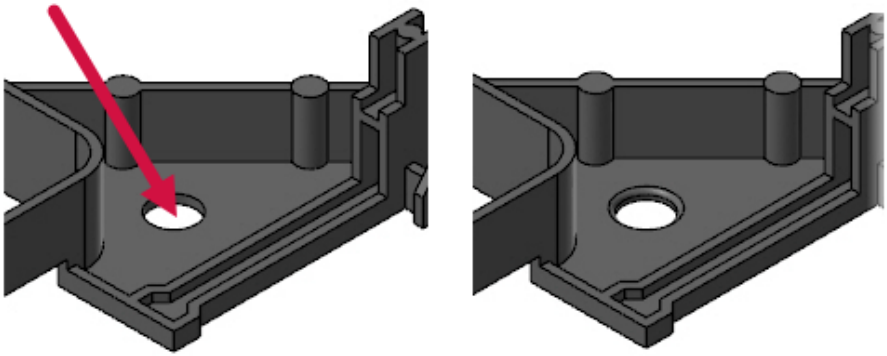
Click here to watch a video of Step 5.

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



Click here to watch a video of Step 6.

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



Click here to watch a video of Step 7.

Conclusion

Congratulations! You have completed the *Getting Started with 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 2018 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 (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.

ATTENTION! UPDATES MAY BE AVAILABLE.
PLEASE REFER TO MASTERCAM.COM/SUPPORT
FOR THE LATEST DOWNLOADS.

CNC Software, Inc.
671 Old Post Road
Tolland, CT 06084 USA
www.mastercam.com

Mastercam
Be Dynamic.

Mastercam® is a registered trademark of CNC Software, Inc. © 1983-2017. All rights reserved.
All other trademarks are property of their respective owners.