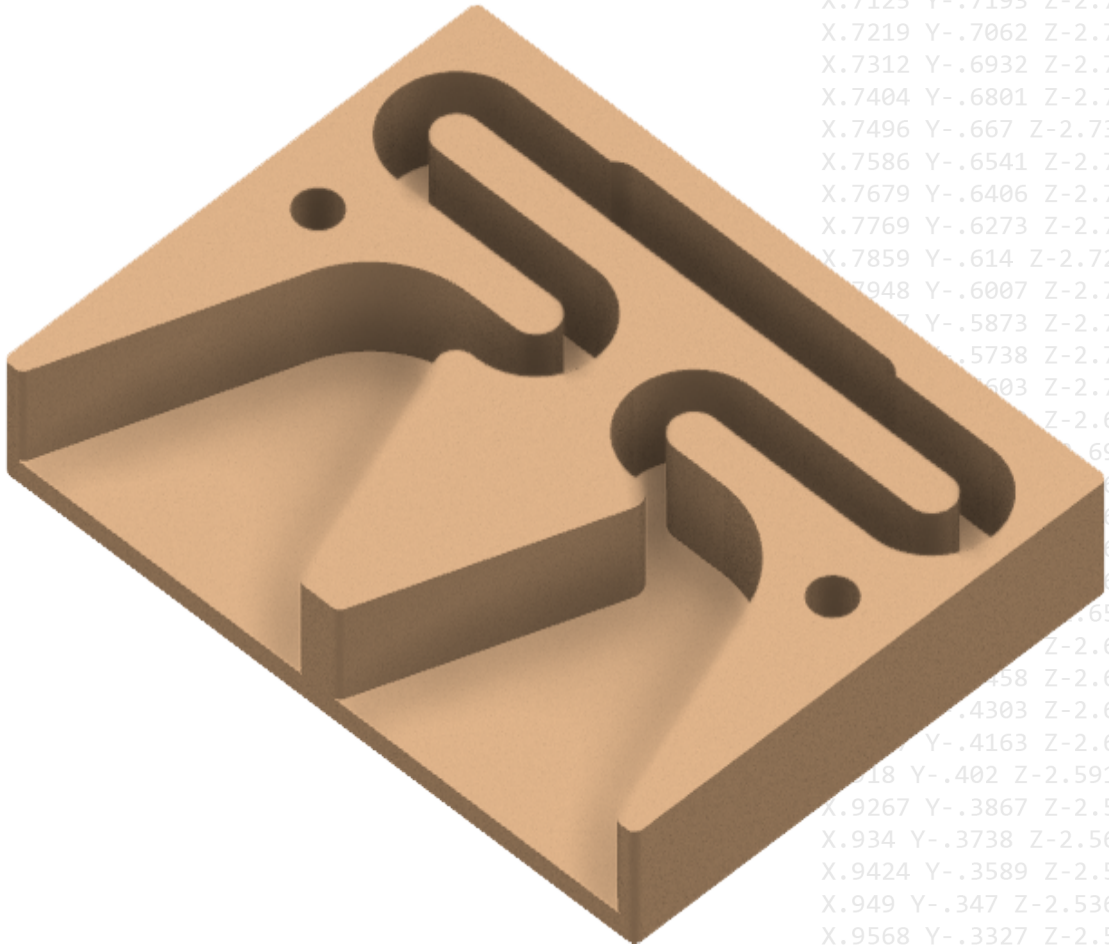
The fastest way to learn the world's most widely used CAD/CAM software.

Mastercam®

Quick Part Series

Beginner



Acoustic Amplifier

QUICK PART SERIES - ACOUSTIC AMPLIFIER

August 2019
© 2019 CNC Software, Inc. – All rights reserved.
Software: Mastercam 2020

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 available online or can be obtained from your local Reseller.

A What's New in Mastercam (What's New in Mastercam 2020.pdf) and ReadMe file (ReadMe.PDF) – installed with each release – include the latest information about Mastercam features and enhancements.

TABLE OF CONTENTS

Introduction **7**

Tutorial Goals 7

General Tutorial Requirements 7

Starting Mastercam Quick Part 8

Mastercam Part Files 8

Material Requirements 8

Machine and Tool Requirements 9

Accessing the Home Learning Edition 9

Creating the Acoustic Amplifier Geometry **11**

Exercise 1: Configuring Mastercam 11

Exercise 2: Creating the Base Shape 12

Exercise 3: Creating the Sound Channel Geometry 16

Exercise 4: Trimming the Sound Channel Geometry 21

Exercise 5: Offsetting the Sound Channel Geometry 26

Exercise 6: Creating the Amplifier Channel Geometry 29

Exercise 7: Mirroring the Geometry 36

Exercise 8: Extruding the Geometry 42

Exercise 9: Creating the Index Holes 46

Exercise 10: Creating the Phone Slot Geometry 53

Exercise 11: Extruding the Phone Slot Geometry 57

Exercise 12: Adding Fillets to Part Edges 59

Show Us What You Learned! 62

Programming the Index Holes **63**

Exercise 1: Setting Up the Stock 63

Exercise 2: Creating the Circle Mill Toolpath 66

Exercise 3: Verifying the Circle Mill Toolpath 73

Show Us What You Learned! 76

Programming the Amplifier Channel **77**

Exercise 1: Creating the Dynamic Mill Toolpath 77
Exercise 2: Verifying the Dynamic Mill and Circle Mill Toolpaths 84
Show Us What You Learned! 87
Programming the Cell Phone Opening 89
Exercise 1: Creating the Dynamic Mill Toolpath 89
Exercise 2: Creating the Contour Toolpath 93
Exercise 3: Creating the Section View 99
Exercise 4: Backplotting the Toolpaths 104
Show Us What You Learned! 109
Programming the Tabs 111
Exercise 1: Creating the Support Geometry 111
Exercise 2: Creating the Tab Points 118
Exercise 3: Creating the Contour Toolpath 120
Exercise 4: Verifying All the Toolpaths 126
Show Us What You Learned! 131
Duplicating the Model and Editing the Toolpaths 133
Exercise 1: Copying and Modifying Toolpath Groups 133
Exercise 2: Translating the Solid Model 138
Exercise 3: Re-chaining the Toolpaths 141
Exercise 4: Verifying All Toolpaths 152
Show Us What You Learned! 155
Fixturing the Stock 157
Fixture options 157
Show Us What You Learned! 158
Post Processing 159
Exercise 1: Post Processing the Operations 159
Show Us What You Learned! 162
Challenge: Self-Guided Project 163
Suggestion 1: Charging Station 163

Suggestion 2: Charging Station with Viewing Mount 164
Design Tips 164
Conclusion **167**

INTRODUCTION

Welcome to the *Quick Part Series - Acoustic Amplifier* tutorial. In this course you will explore how to design and machine an acoustic amplifier that you can use to enjoy music from your smartphone. Starting with the instructions provided with this tutorial, you will use wireframe and solid CAD tools to create the geometry for the model. Using a variety of rough and finish toolpaths, you will program and verify the model to ensure it can be cut successfully. Finally, you will post out your toolpaths to generate NC code and cut the part.

This tutorial will provide you with the basic techniques needed to create the acoustic amplifier. From there, you can explore Mastercam and challenge yourself to make some other interesting projects.

Tutorial Goals

- Understand basic wireframe and solid model design tools and how they can be used to create a complex solid model
- Learn how to mirror geometry to create a symmetrical design
- Learn how to program a part for a router using standard and dynamic toolpaths
- Identify safe hold-down options for router work, such as tabs and woodscrews, or a vacuum table
- Verify operations

> **NOTE**
> 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.

General Tutorial Requirements

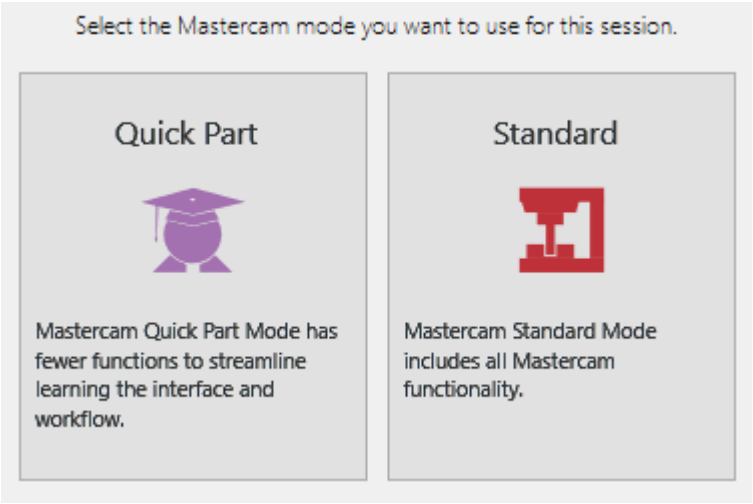
All Mastercam Project Based Learning tutorials have the following general requirements:

- You must be comfortable using the Windows® operating system.
- 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 accessed from the Mastercam 2020 workstation, either with the tutorial or in any location that you prefer.
- All Mastercam tutorials require you to configure Mastercam to work in a default Metric or Inch configuration. The tutorial provides instructions for loading the appropriate configuration file.

Starting Mastercam Quick Part

> **NOTE**
> The *Quick Part Series* tutorials assume you are using an Educational license and the Mastercam Quick Part interface. You do not need to install the Quick Part interface to complete this tutorial. The Quick Part interface is only available to Educational licenses.

The Quick Part interface is a simplified version of the Mastercam interface, hiding some of the more complex functions. Upon starting Mastercam, the following prompt displays:



Select **Quick Part**. To return to the standard Mastercam interface, you need to close and re-open Mastercam. Then select **Standard**.

Mastercam Part Files

The provided part files are in .mcam format. This allows both Industrial and Educational license users to open the part files. Files saved with an Educational license have an .emcam file extension. This does not affect the completion of the tutorial.

Material Requirements

The following materials are recommended to machine the part we use in this tutorial with the suggested machine definition, feeds and speeds, and tooling. Using different materials will affect the tooling and toolpath settings.

- Solid wood, 20 inches x 8 inches x 1.25 inches
- 2 Wood dowels, 3 inches x 0.5 inch (diameter)

Machine and Tool Requirements

> **WARNING**
> This tutorial is completed using a specific machine and material. The settings, tooling, feeds, and speeds recommended in this tutorial are based on the use of this equipment, as well as the material to be machined. Your results may differ, based on the machine, its age and prior utilization, and tooling used, as well as the material selected for machining.

This tutorial recommends the following:

- A standard router, preferably with a vacuum table. The cutting surface should be large enough to accommodate the workpiece and any required fixtures.

  If you do not use a vacuum table, you will need to program your toolpaths to accommodate for any hardware, such as clamps or screws, that you use to hold down your stock.
- 3/8 in. Flat End Mill

Accessing the Home Learning Edition

The Mastercam Home Learning Edition (HLE) is a free version of Mastercam that lets you practice design and programming skills without having a full Mastercam license. The HLE software lets you save your Mastercam files, but it will not post code.

The HLE is available for free at the following location: https://signup.mastercam.com/demo-hle.

CHAPTER 1

CREATING THE ACOUSTIC AMPLIFIER GEOMETRY

In this chapter you will use the design (CAD) functions of Mastercam to create the acoustic amplifier. You will use wireframe, transform, and solid functions to produce the overall acoustic amplifier design.

Goals

- Understand Z depth and construction planes.
- Explore wireframe, solid, and transform functions to create geometry.
- Use Trim to Entities and Divide functions to modify geometry.

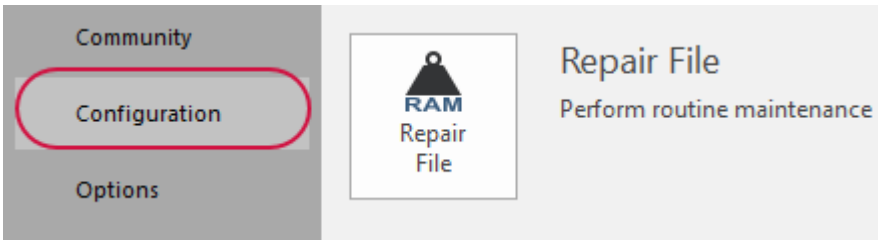
Exercise 1: Configuring Mastercam

In this exercise, you open Mastercam and set your system configuration to be inch.

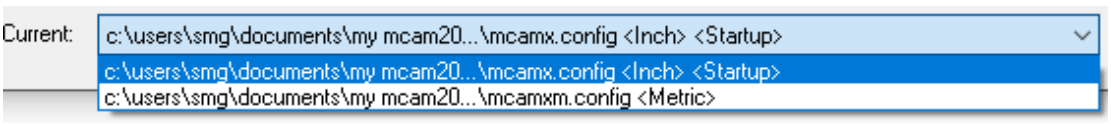
1. Start Mastercam using your preferred method:
    a. Double-click Mastercam's desktop icon.

    OR

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

    c. Choose ...\mcamx.config <Inch> from the **Current** drop-down list.

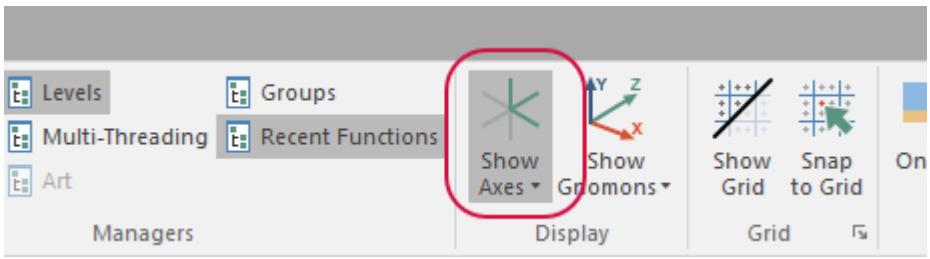
    

    d. Click **OK**.

Exercise 2: Creating the Base Shape

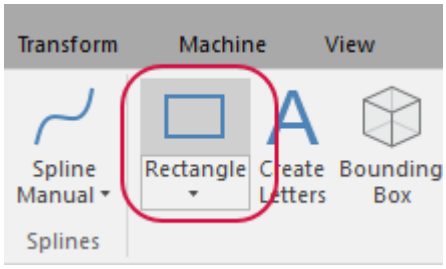
In this exercise you will create the base rectangular shape of the acoustic amplifier.

1. Select **Show Axes** on the **View** tab.



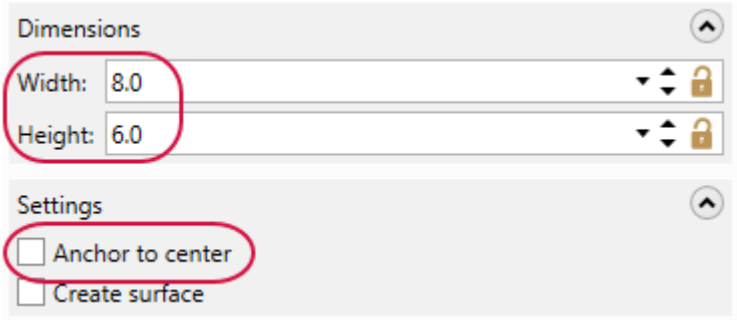
> **NOTE**
> You can also press **[F9]** to display the axes.

2. Select **Rectangle** on the **Wireframe** tab.

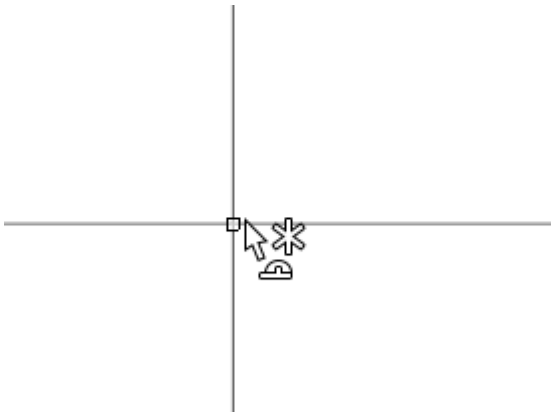


The **Rectangle** function panel displays.

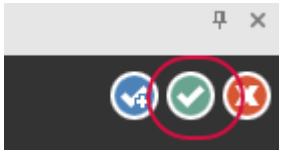
3. Set the following parameters:
   a. Enter **8.0** for **Width**.
   b. Enter **6.0** for **Height**.
   c. Ensure that **Anchor to center** is deselected.



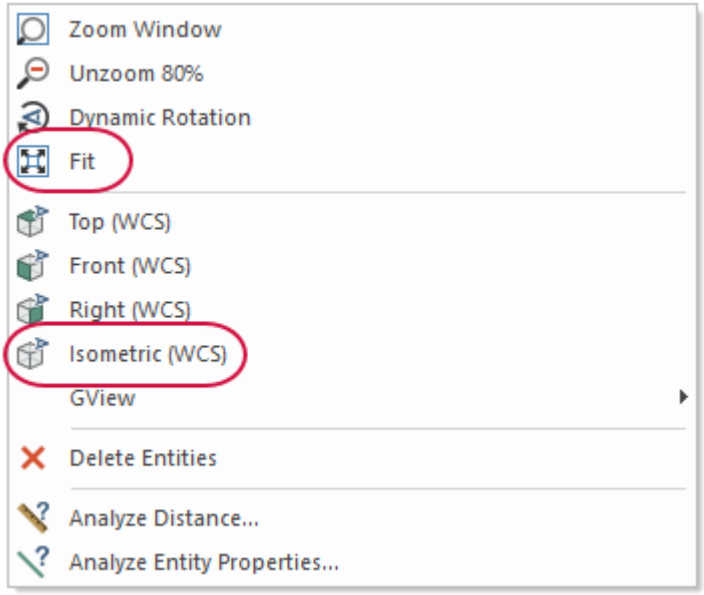
4. Select the origin in the graphics window to place the rectangle.



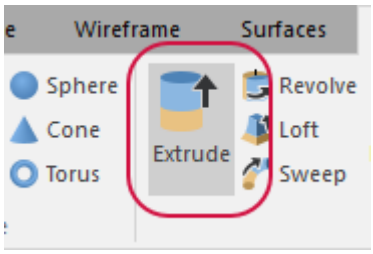
5. Click **OK** in the **Rectangle** function panel.



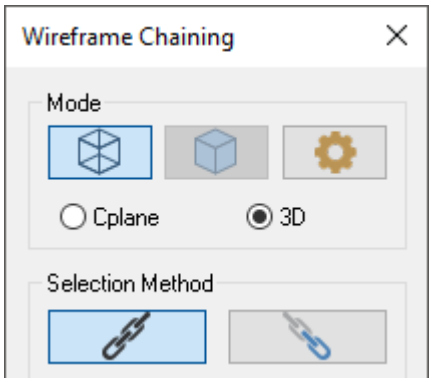
6. Right-click in the graphics window and select **Fit**. Right-click again and select **Isometric (WCS)**.



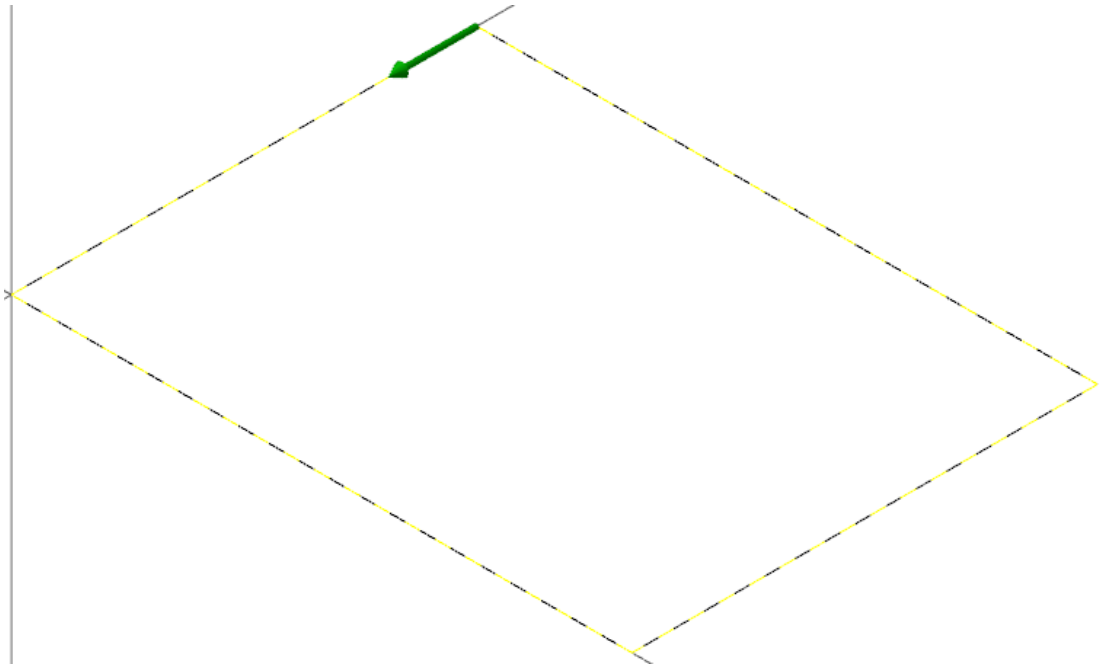
7. On the **Solids** tab, select **Extrude**.



The **Chaining** dialog box displays.



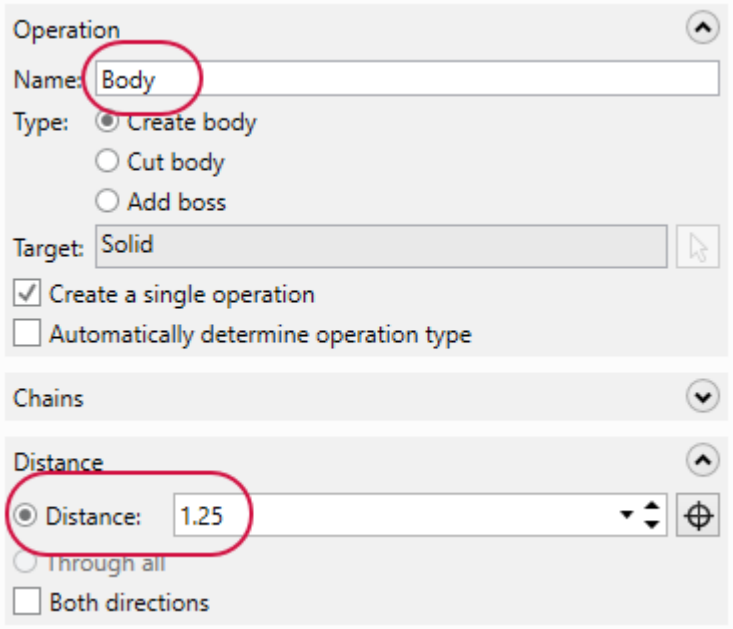
8. Select the rectangle.



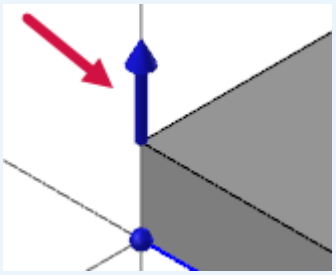
9. Click **OK** in the **Chaining** dialog box.

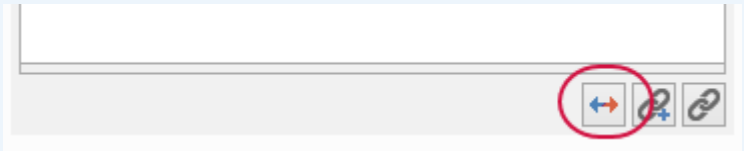
   The **Solid Extrude** function panel displays.

10. Set the following parameters:
    a. Enter **Body** for **Name**.
    b. Enter **1.25** for **Distance**. This is the thickness of the part.



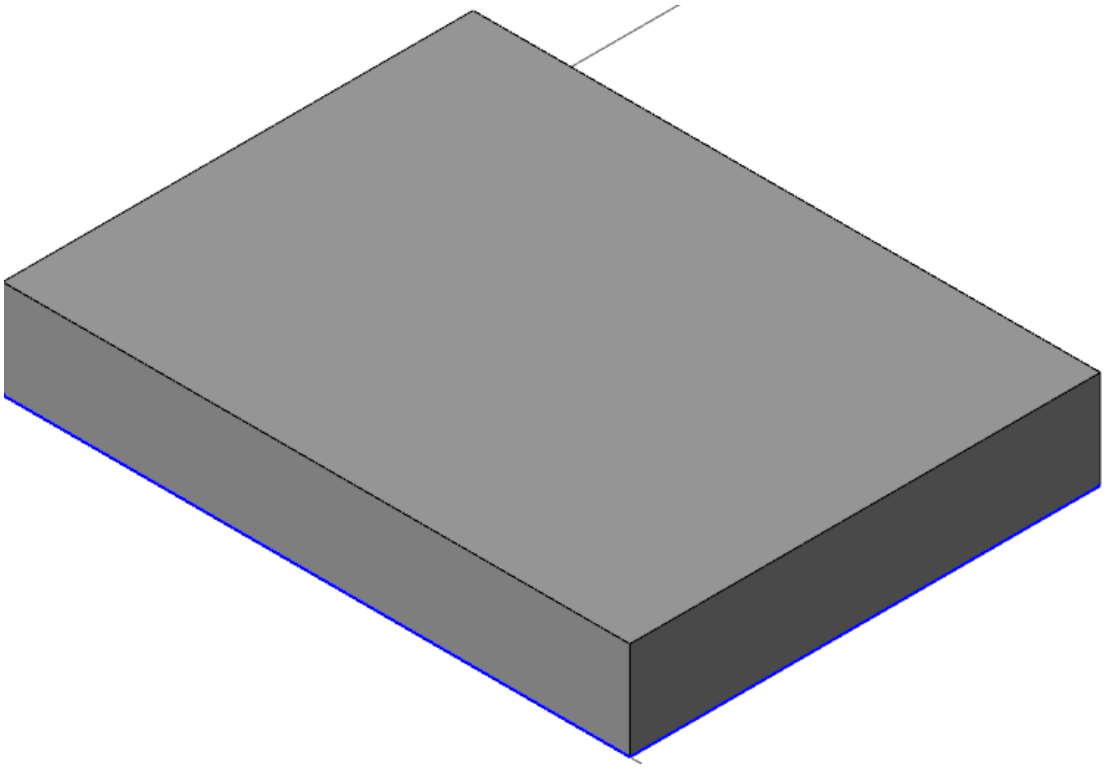
NOTE
If the rectangle is not extruding up from the base, as shown below, select **Reverse All** to reverse the direction of the extrusion.





11. Click **OK** in the function panel.

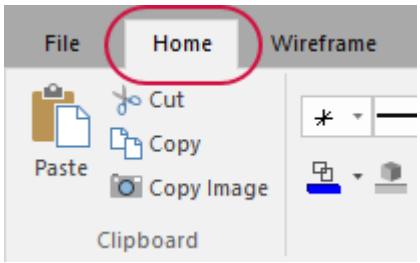
12. Save your file as Amplifier-XXX, where XXX represents your initials.



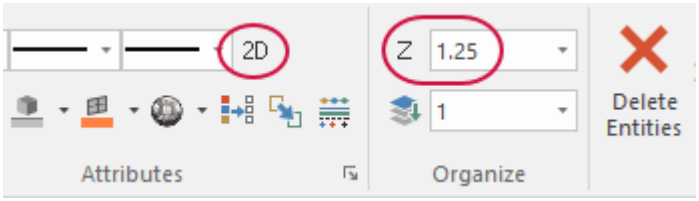
Exercise 3: Creating the Sound Channel Geometry

In this exercise you will create the sound channel geometry using wireframe geometry. You will create the line geometry first, then use trimming functions to create the base shape of the channel. In the following exercise, you will use the trimming functions to create the base shape of the channel.

1. Your file Amplifier-XXX which you saved from the previous exercise should be open in Mastercam.
2. Select the **Home** tab.

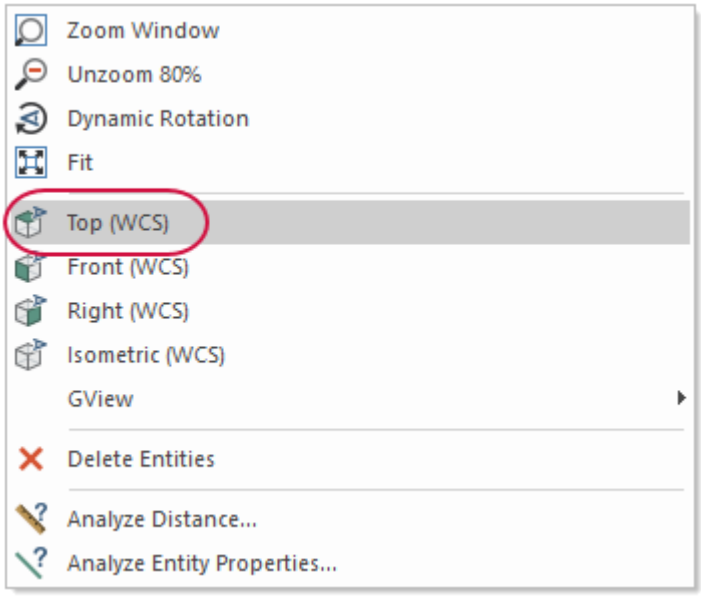


3. Set the following:
    a. Select **3D** in the **Attributes** group. The icon changes to **2D**.
    b. Enter **1.25** for **Set Z Depth**.

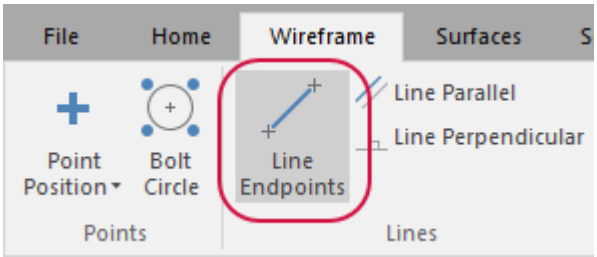


> **NOTE**
> The **Z Depth** and **2D/3D Construction plane** value can also be changed on the **Status Bar**.

4. Right-click in the graphics window and select **Top (WCS)**.

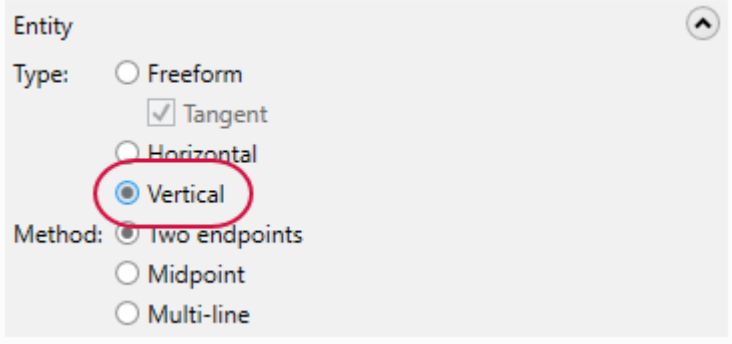


5. Select **Line Endpoints** on the **Wireframe** tab.

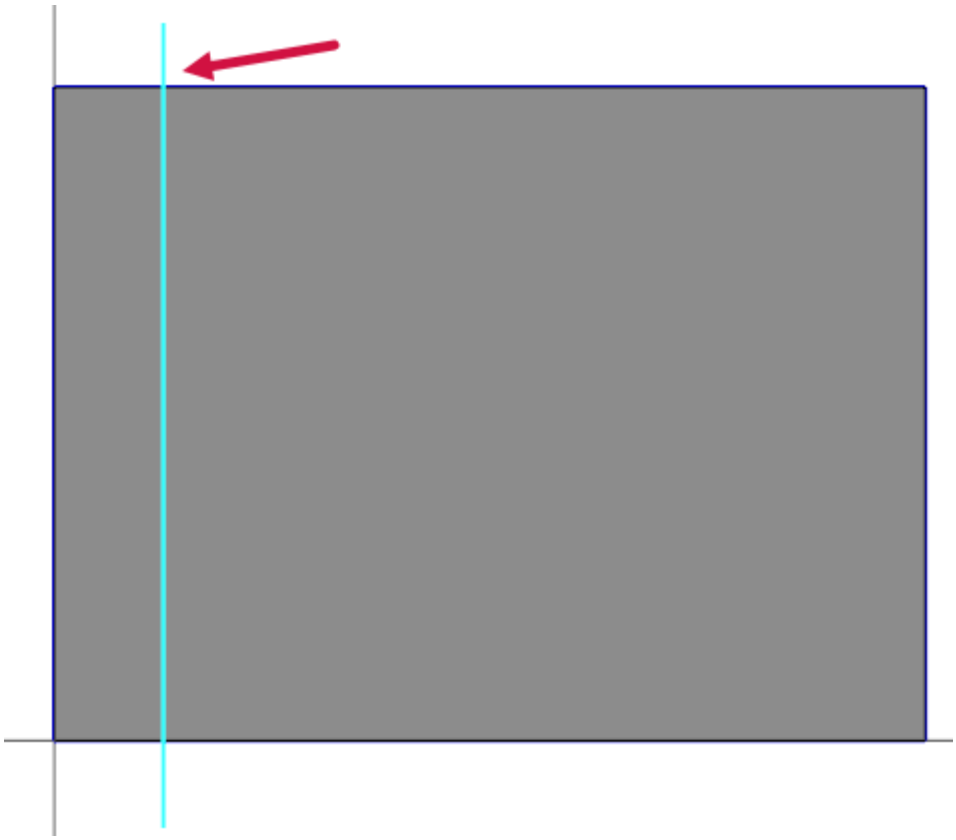


The **Line Endpoints** function panel displays.

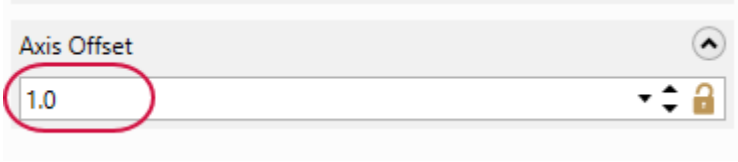
6. Set **Type** to **Vertical**.



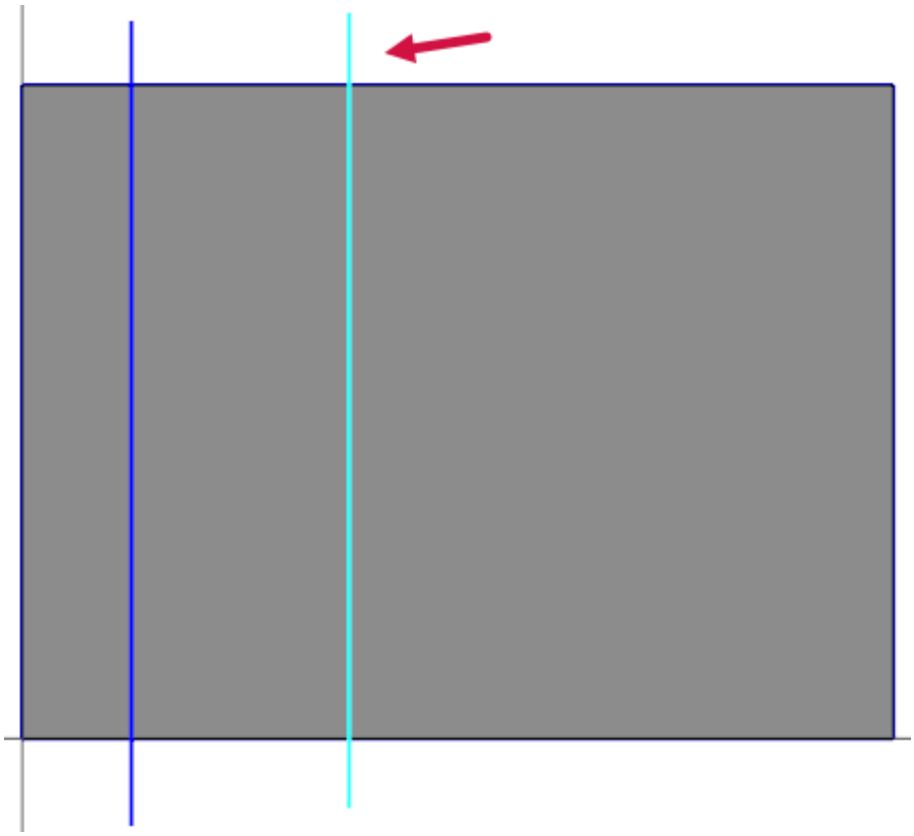
7. Click in the graphics window above the solid rectangle and drag down to create a vertical line that passes over the part.



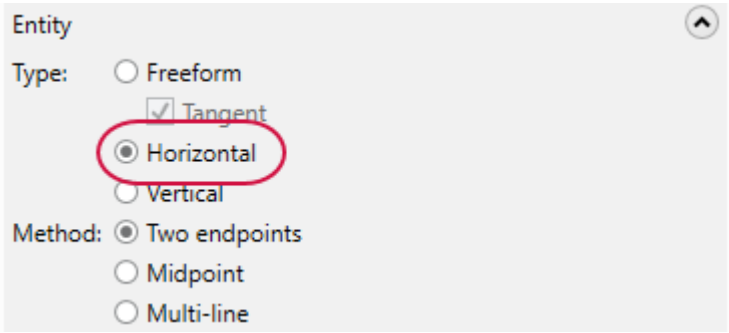
8. Enter **1.0** for **Axis Offset**. Press [**Enter**].



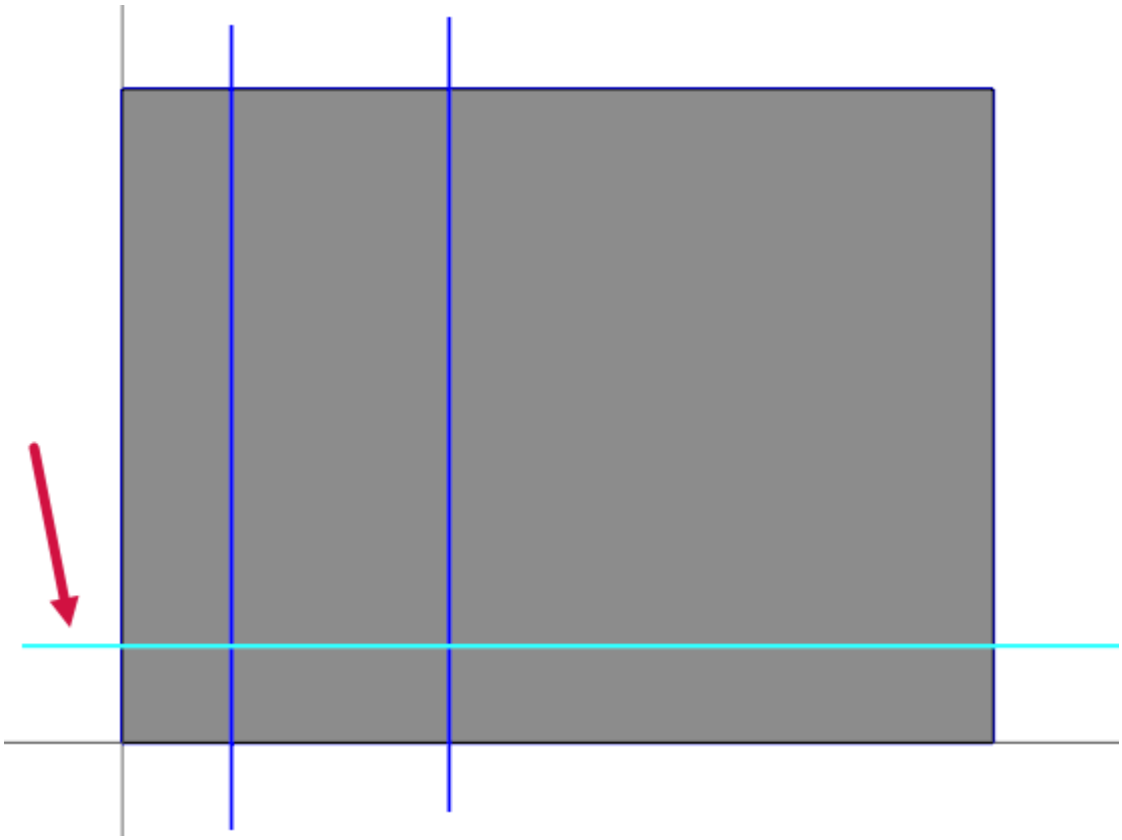
9. Click in the graphics window and create another vertical line with an **Axis Offset** of **3.0**. Press [**Enter**].



10. Set **Type** to **Horizontal**.



11. Click in the graphics window to the left of the solid rectangle and create a horizontal line that passes over the part.

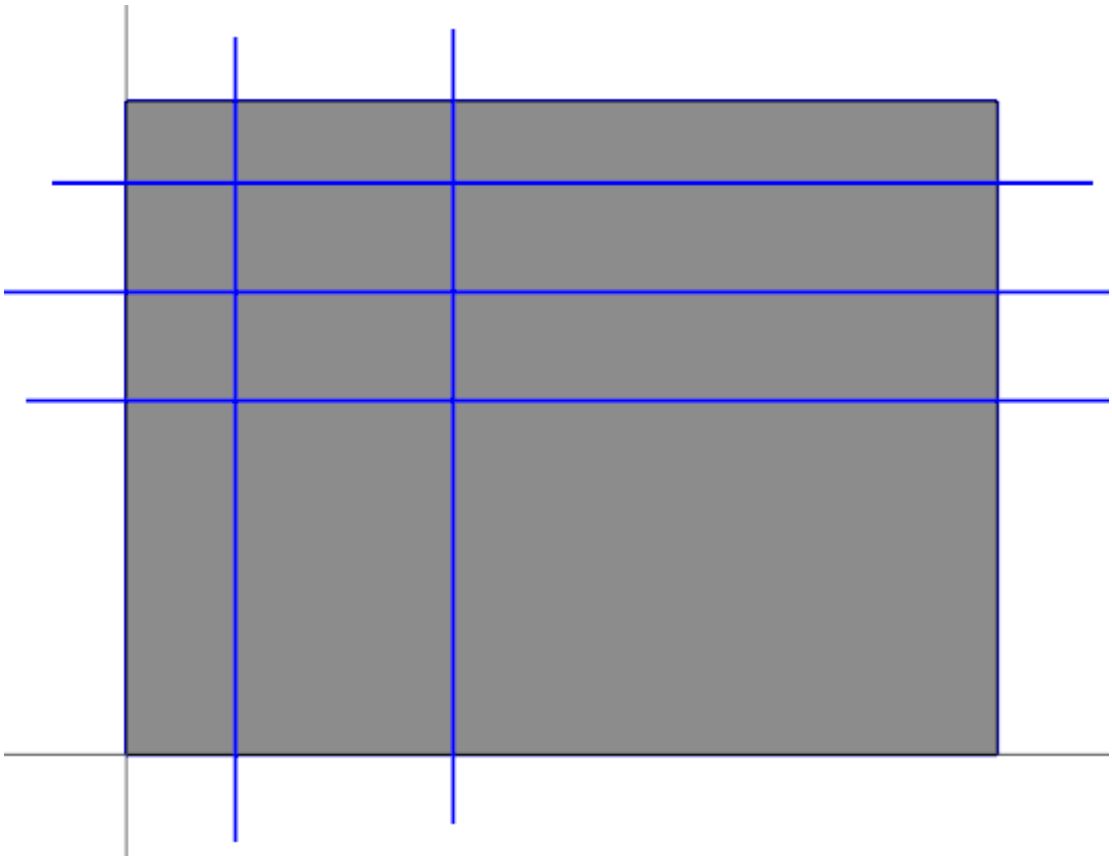


The line can be in any location over the rectangle. It will be moved in the next step.

12. Enter **3.25** for **Axis Offset**. Press [**Enter**].

13. Create two more horizontal lines. One with an **Axis Offset** of **4.25** and then another with **5.25**.

    There should now be a total of two vertical lines and three horizontal lines.

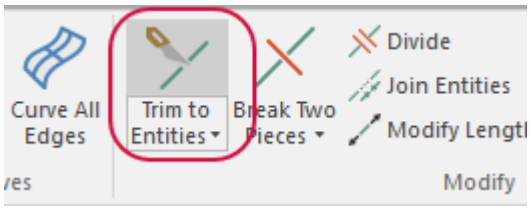


14. Click **OK** in the **Line Endpoints** function panel. You will trim the lines in the next exercise.
15. Save your file.

Exercise 4: Trimming the Sound Channel Geometry

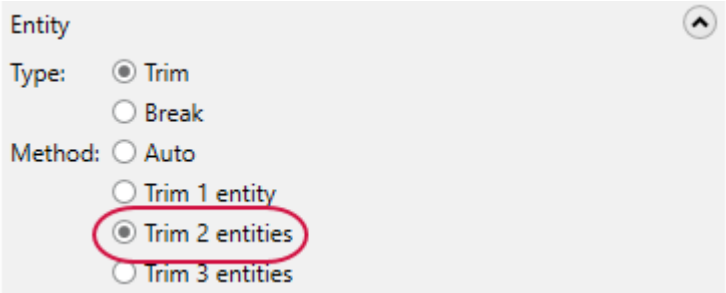
In this exercise you will trim and add fillets the geometry created in the previous exercise to create a smoother channel shape.

1. Your file Amplifier-XXX which you saved from the previous exercise should be open in Mastercam.
2. Select **Trim to Entities** on the **Wireframe** tab.

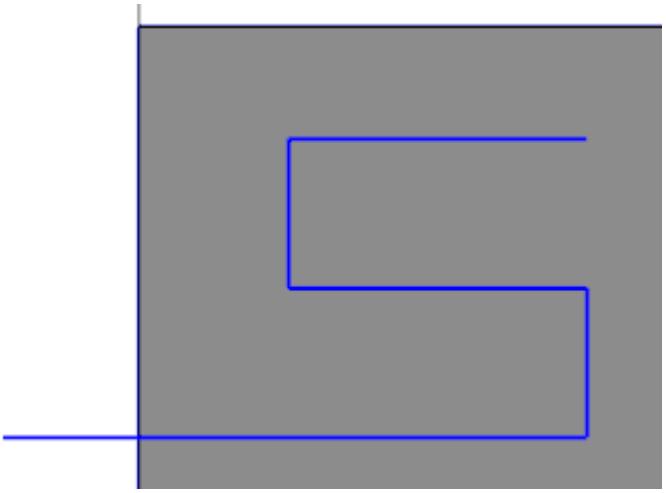


    The **Trim to Entities** function panel displays.

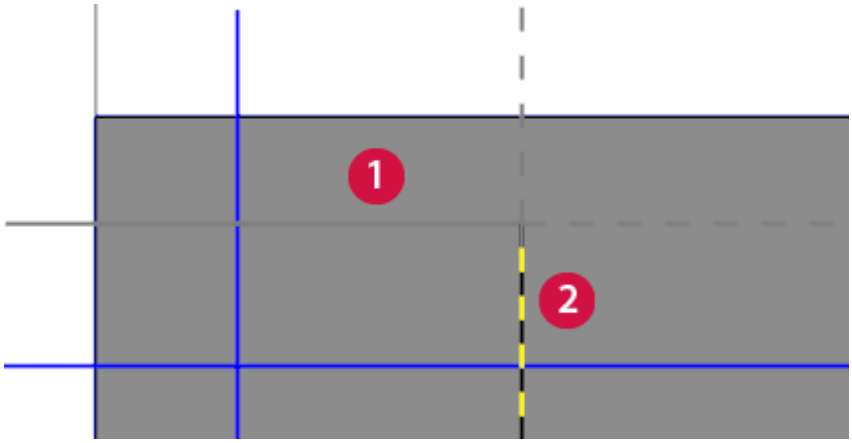
3. Set **Method** to **Trim 2 entities**.



Line entities will be trimmed to achieve the following shape:

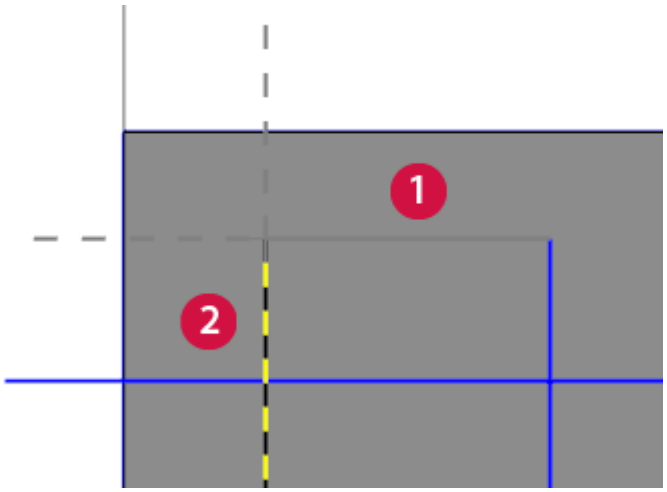


4. Select the lines in the order shown in the following image for the first trim.

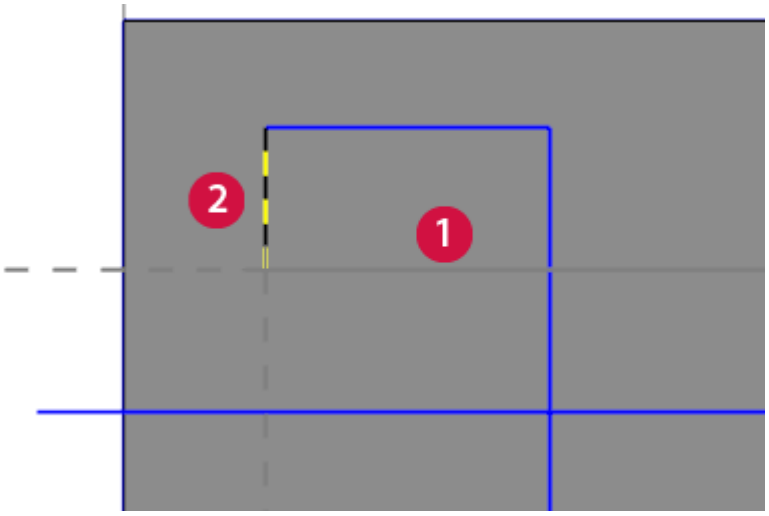


Mastercam's **Trim to Entities** function trims, breaks, and extends entities selected in the graphics window. Auto-preview displays the results of the selected function as you move the mouse over the final entity selection. A solid line represents what will be created. A dashed line represents what will be removed.

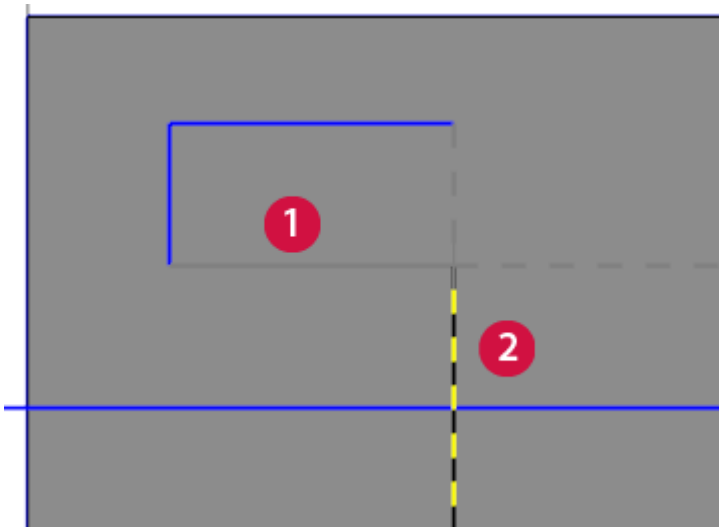
5. Select the lines in the order shown in the following image for the second trim.



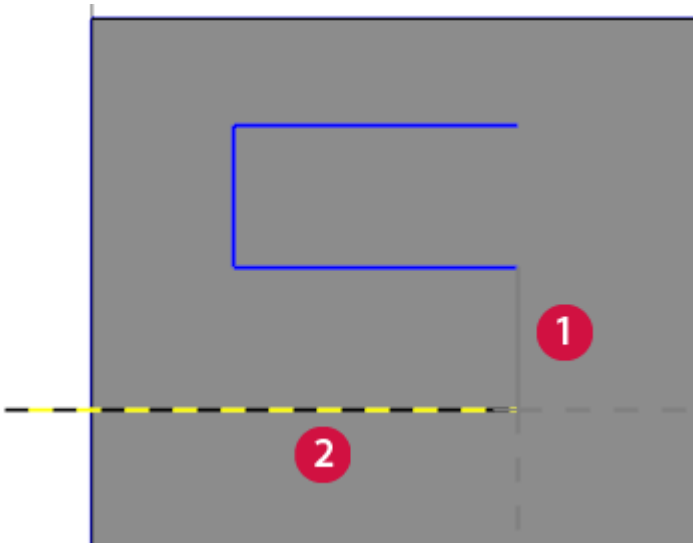
6. Select the lines in the order shown in the following image for the third trim.



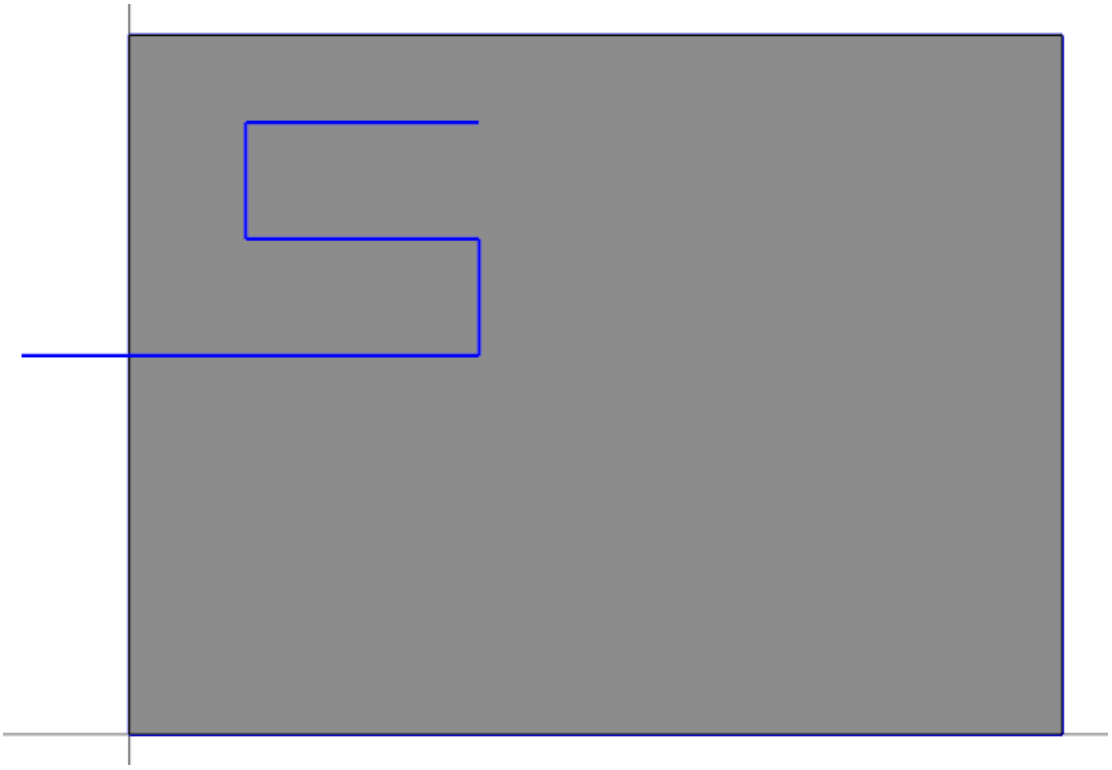
7. Select the lines in the order shown in the following image for the fourth trim.



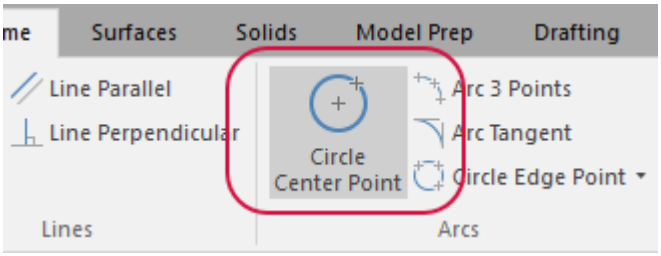
8. Select the lines in the order shown in the following image for the fifth trim.



9. Click **OK** in the **Trim to Entities** function panel. Your part should match the image below.

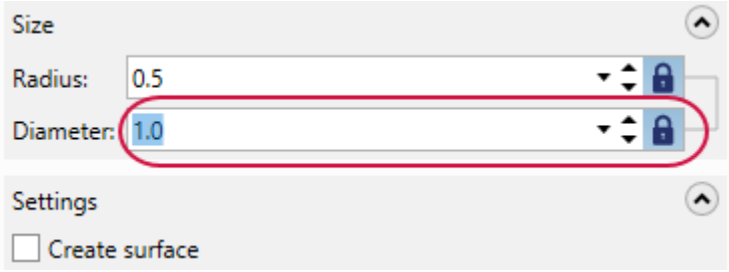


10. Select **Circle Center Point** on the **Wireframe** tab.

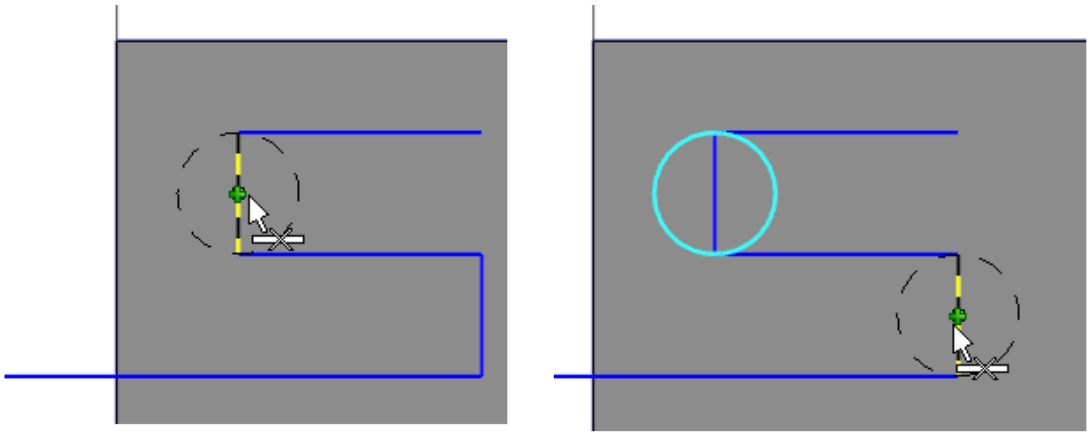


The **Circle Center Point** function panel displays.

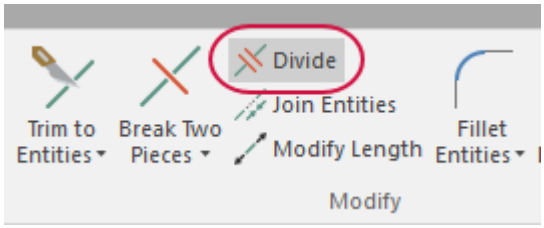
11. Enter **1.0** for **Diameter** and click the **Lock** icon next to **Diameter**.



12. Select the midpoint of the two lines shown below.

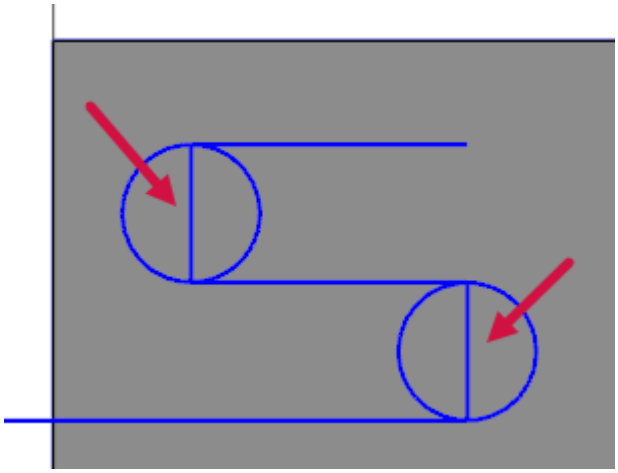


13. Click **OK** in the **Circle Center Point** function panel.
14. Select **Divide** on the **Wireframe** tab.

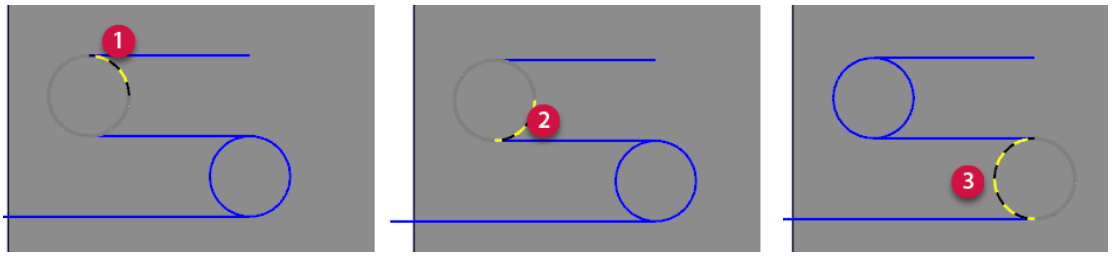


The **Divide** function panel displays.

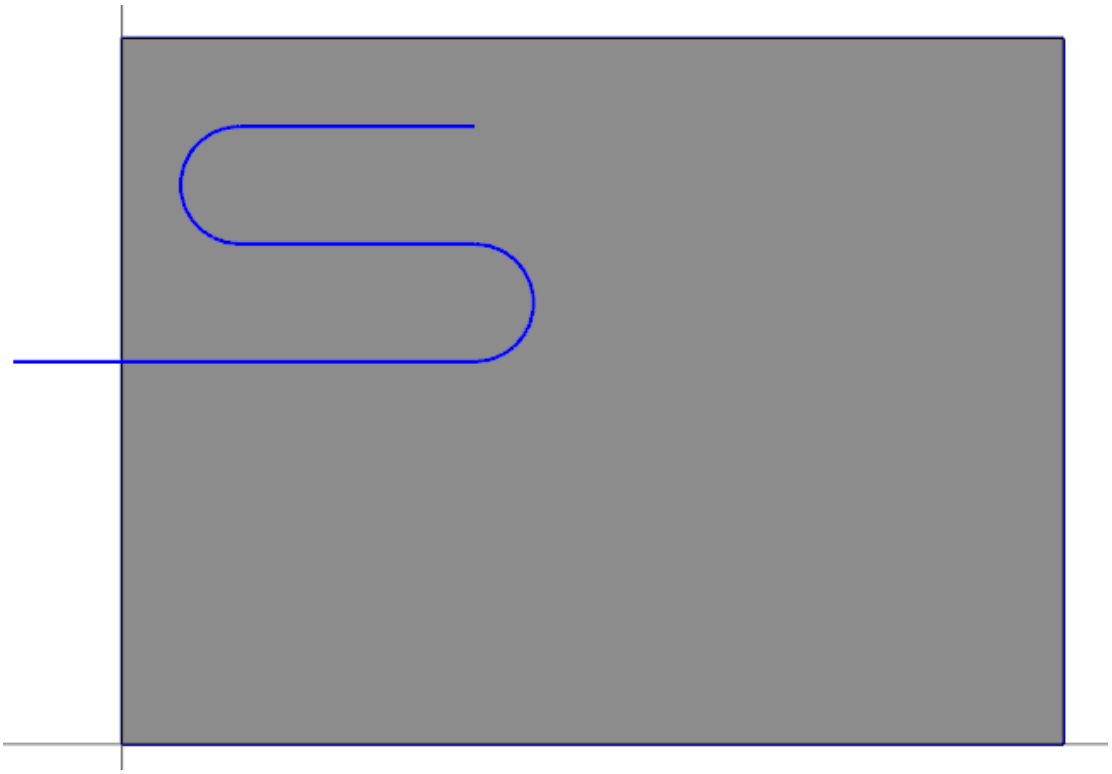
15. Select the two center lines of the created arcs.



16. Select the following arcs.



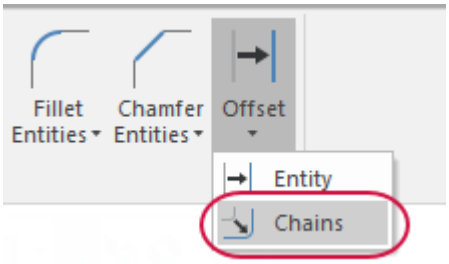
17. Click **OK** in the **Divide** function panel.
18. Save your file.



Exercise 5: Offsetting the Sound Channel Geometry

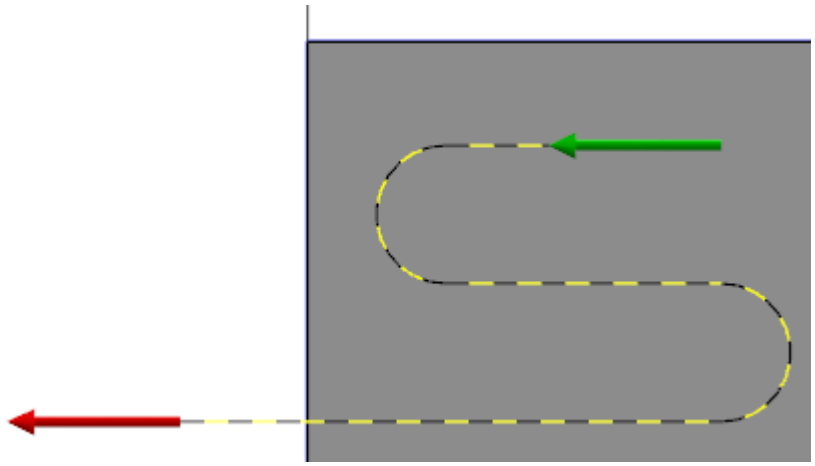
In this exercise, you will offset the geometry created from the previous exercises.

1. Your file Amplifier-XXX which you saved from the previous exercise should be open in Mastercam.
2. Select **Offset**, **Chains** from the **Wireframe** tab.

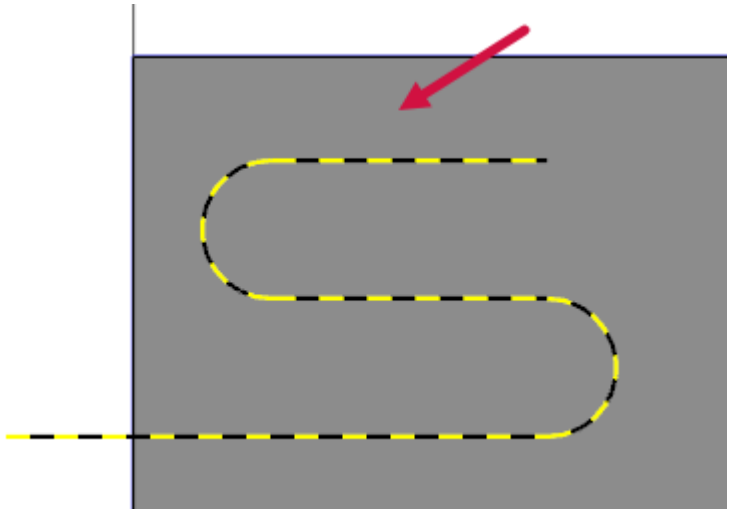


The **Chaining** dialog box displays.

3. Select the line geometry as shown below.

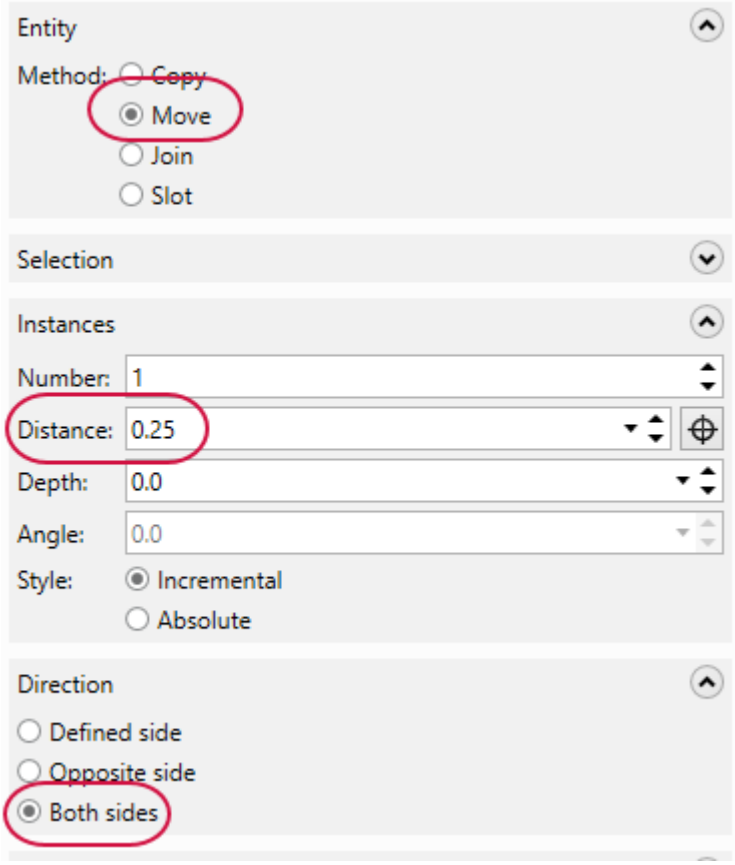


4. Click **OK** in the **Chaining** dialog box.
5. Click in the graphics window above the top line to indicate the offset direction.



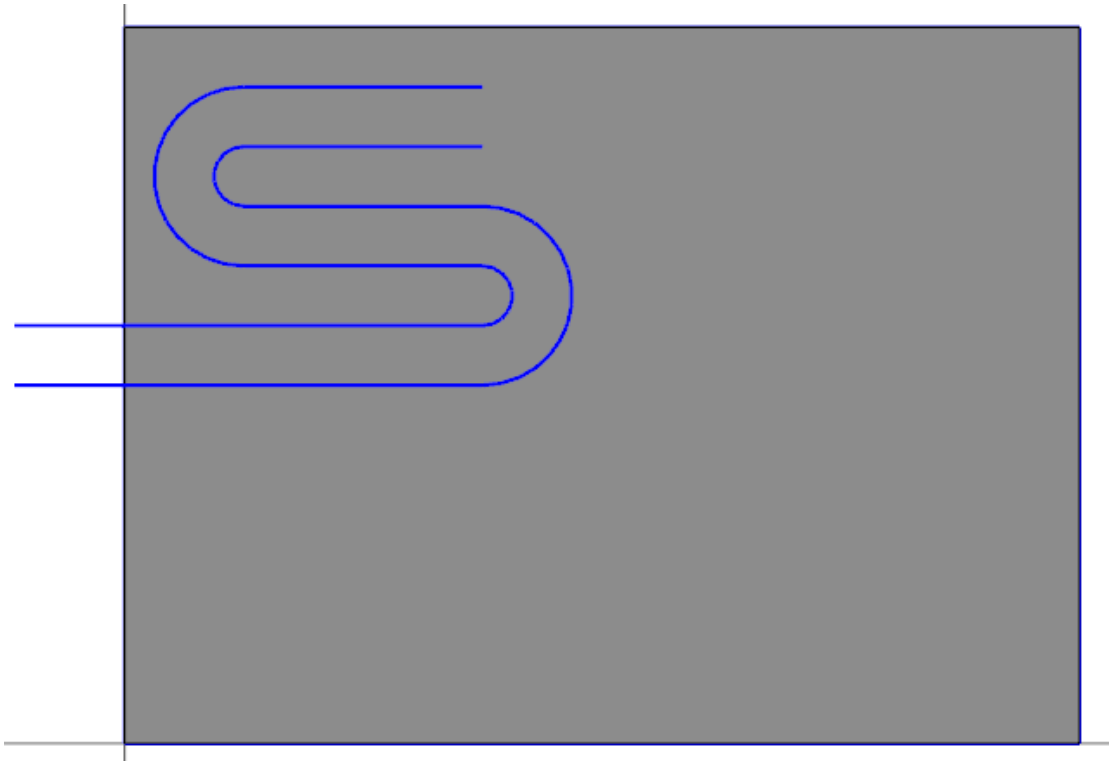
The **Offset Chains** function panel is now active.

6. Set the following parameters:
    a. Set **Method** to **Move**.
    b. Enter **0.25** for **Distance**.
    c. Set **Direction** to **Both sides**.



7. Click **OK** in the **Offset Chains** function panel.
8. Right-click in the graphics window and select **Clear Colors**.

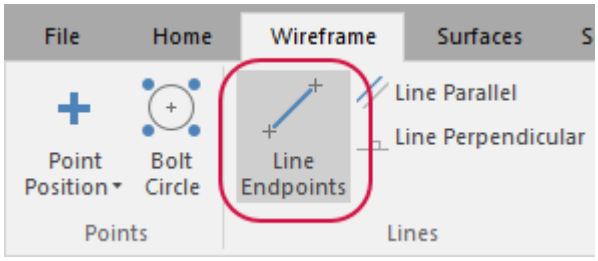
9. Save your file.



Exercise 6: Creating the Amplifier Channel Geometry

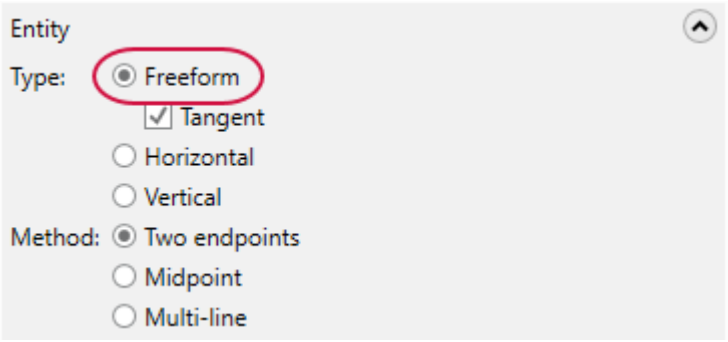
In this exercise, you will create the amplifier channel geometry that feeds from the sound channel geometry made from the previous exercises.

1. Your file Amplifier-XXX which you saved from the previous exercise should be open in Mastercam..
2. Select **Line Endpoints** on the **Wireframe** tab.

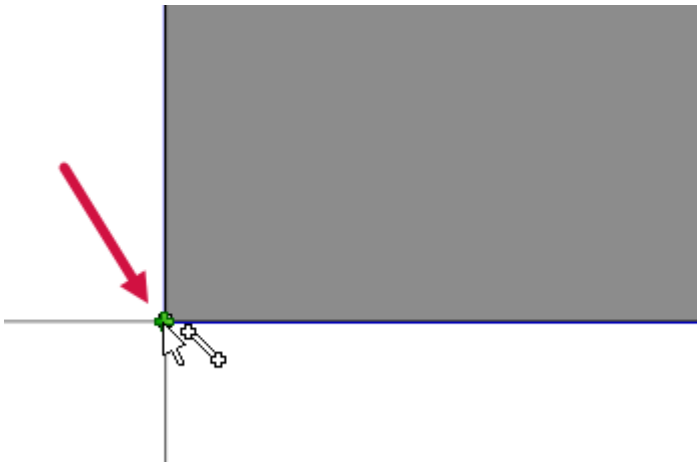


The **Line Endpoints** function panel displays.

3. Set **Type** to **Freeform**.

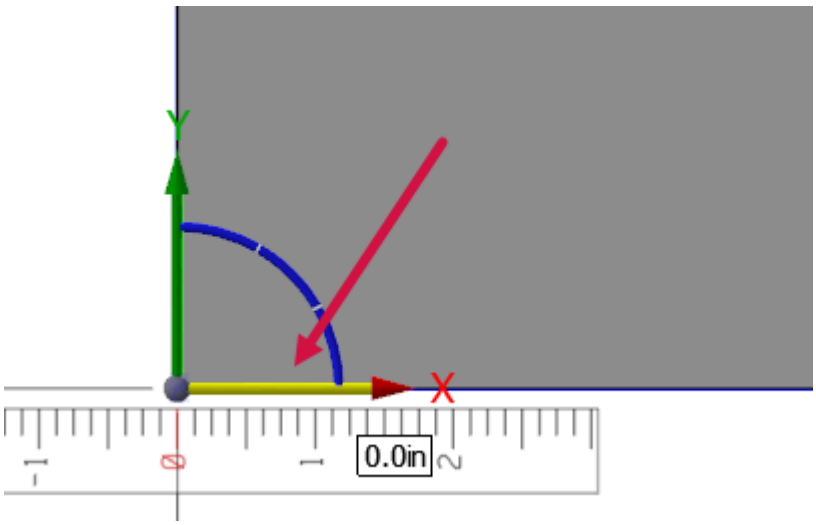


4. Hold down the [**Shift**] key and select the origin in the graphics window.

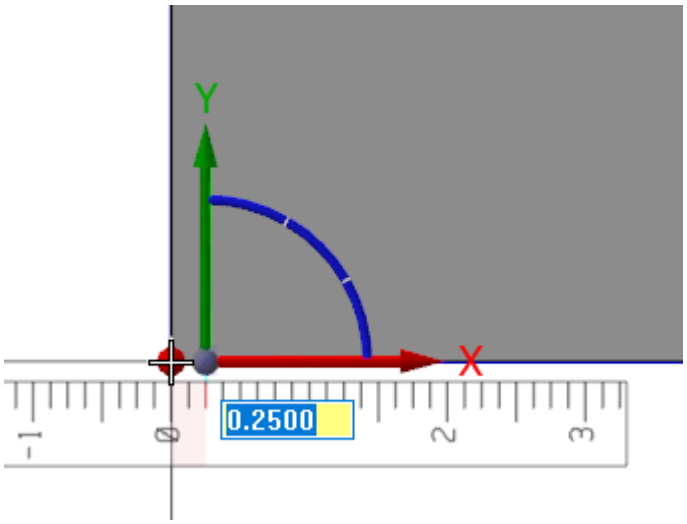


The Dynamic gnomon displays.

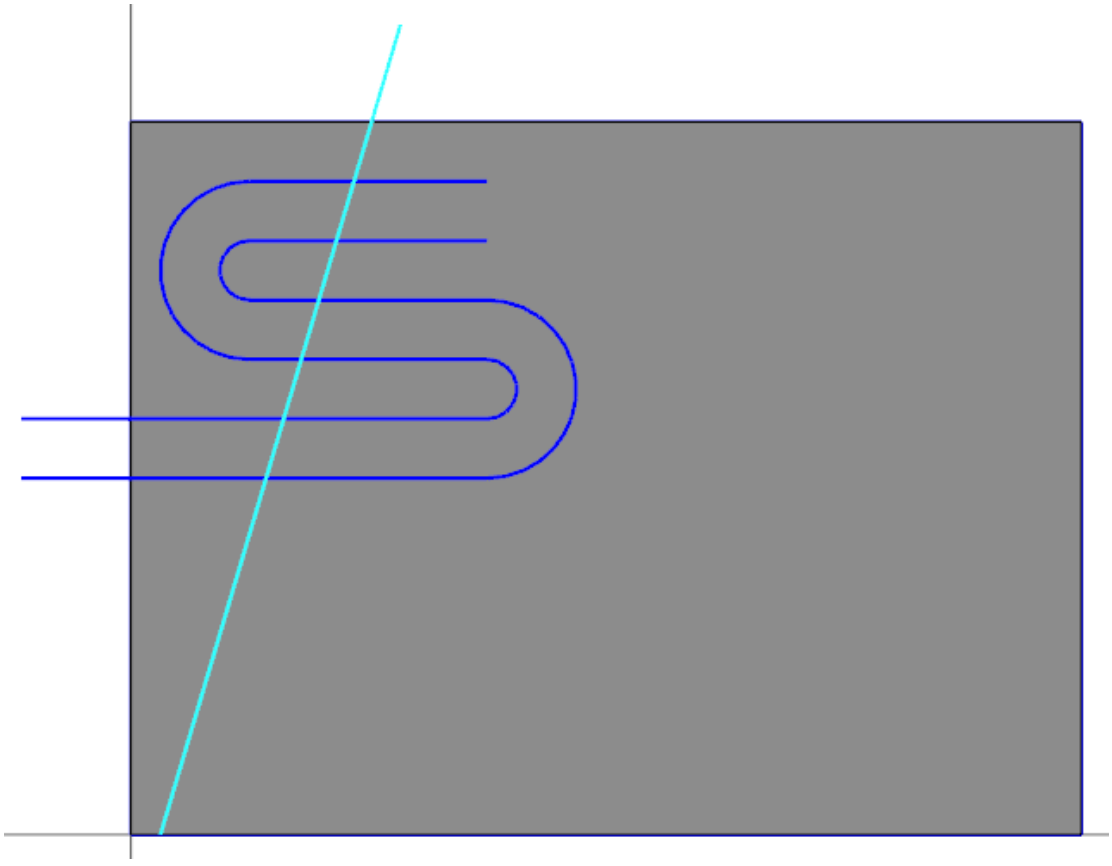
Select the X axis arrow.



5. Enter **0.25** and press [**Enter**].

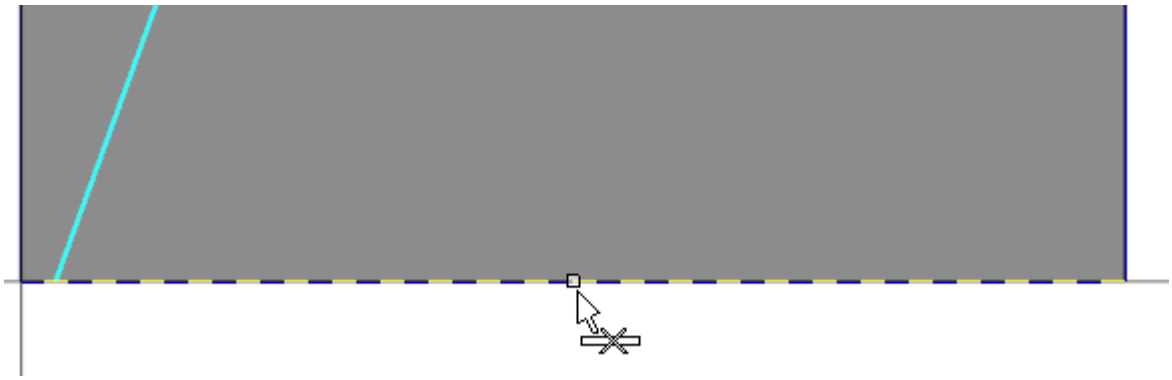


6. Press [**Enter**] twice.
7. Click in the graphics window above the solid rectangle to create the line.

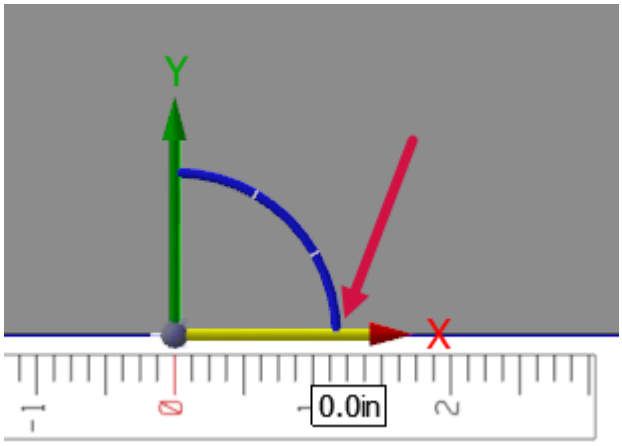


8. Enter **70.0** for **Angle**.

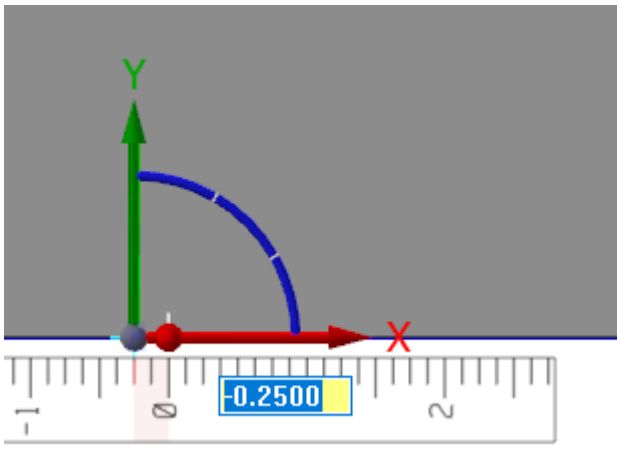
9. Hold down the [**Shift**] key and select the midpoint of the bottom line in the graphics window.



10. Select the X axis arrow.

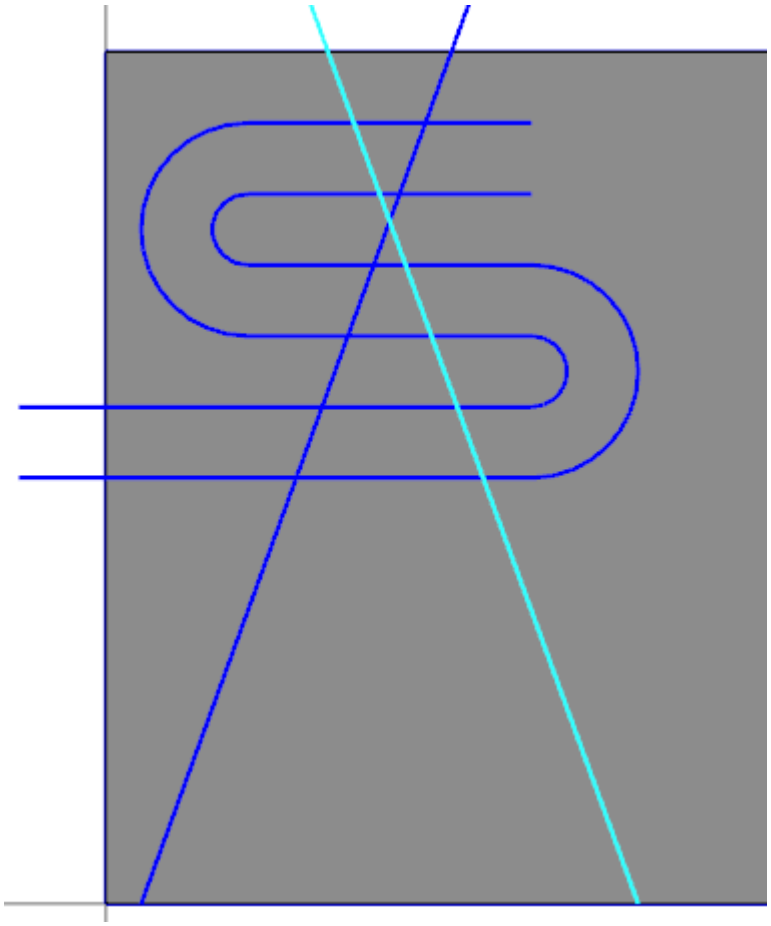


11. Enter **-0.25** and press [**Enter**].



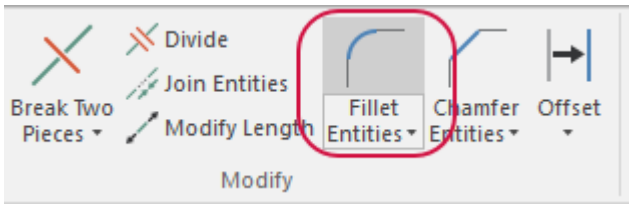
12. Press [**Enter**] twice.

13. Click in the graphics window above the solid rectangle to create the line.



14. Enter **110.0** for **Angle**.

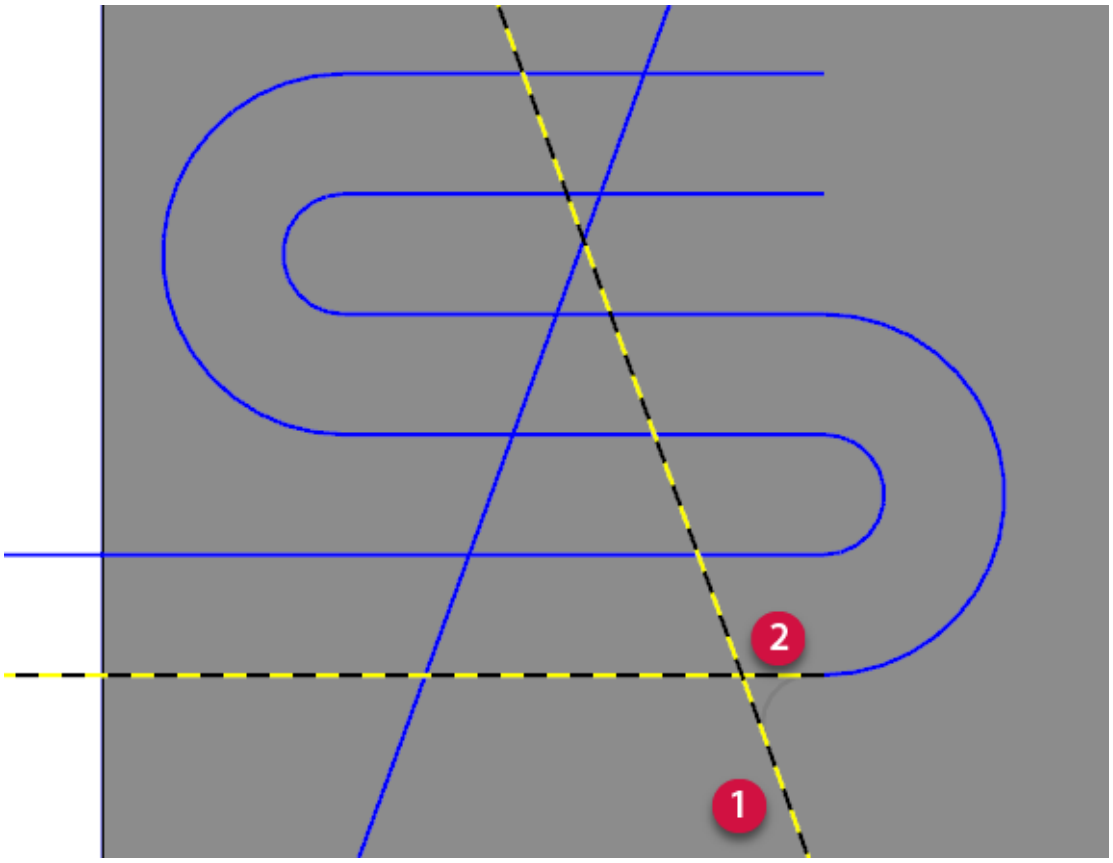
15. Click **OK** in the **Line Endpoints** function panel.
16. Select **Fillet Entities** on the **Wireframe** tab.



The **Fillet Entities** function panel displays.

17. Enter **0.2** for **Radius**.

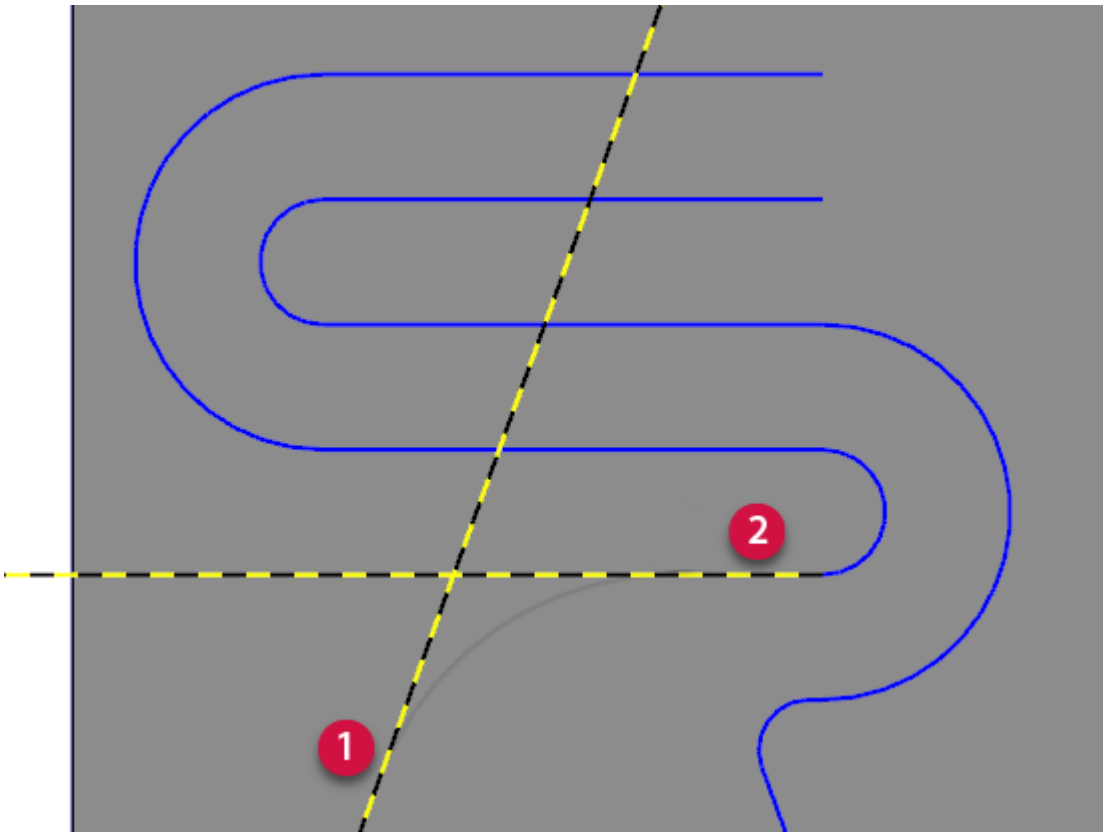
18. Select the lines in the order shown in the following image.



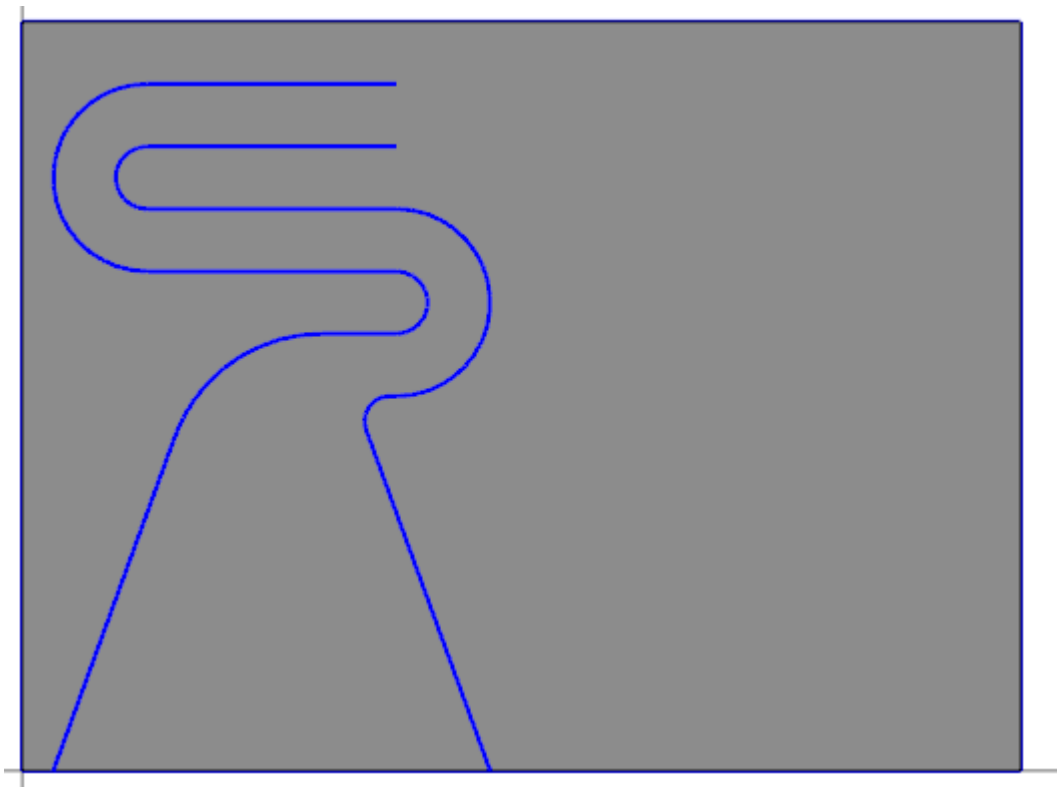
19. Click **OK and Create New Operation** in the **Fillet Entities** function panel.

20. Enter **1.25** for **Radius**.

21. Select the lines in the order shown in the following image.



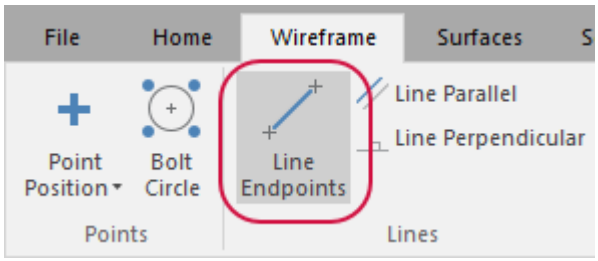
22. Click **OK** in the **Fillet Entities** function panel.
23. Save your file.



Exercise 7: Mirroring the Geometry

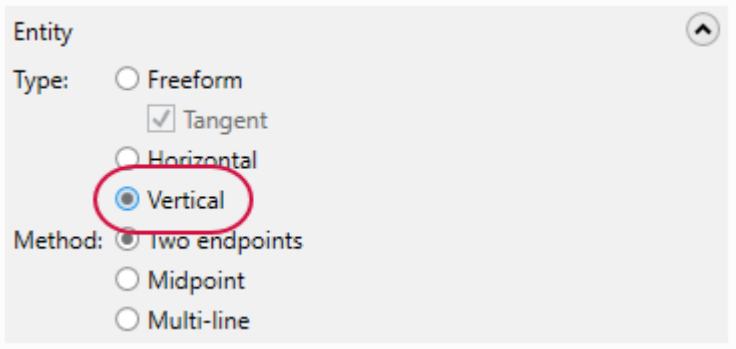
In this exercise, you will use the Mirror function to mirror the created geometry.

1. Your file Amplifier-XXX which you saved from the previous exercise should be open in Mastercam.
2. Select **Line Endpoints** on the **Wireframe** tab.

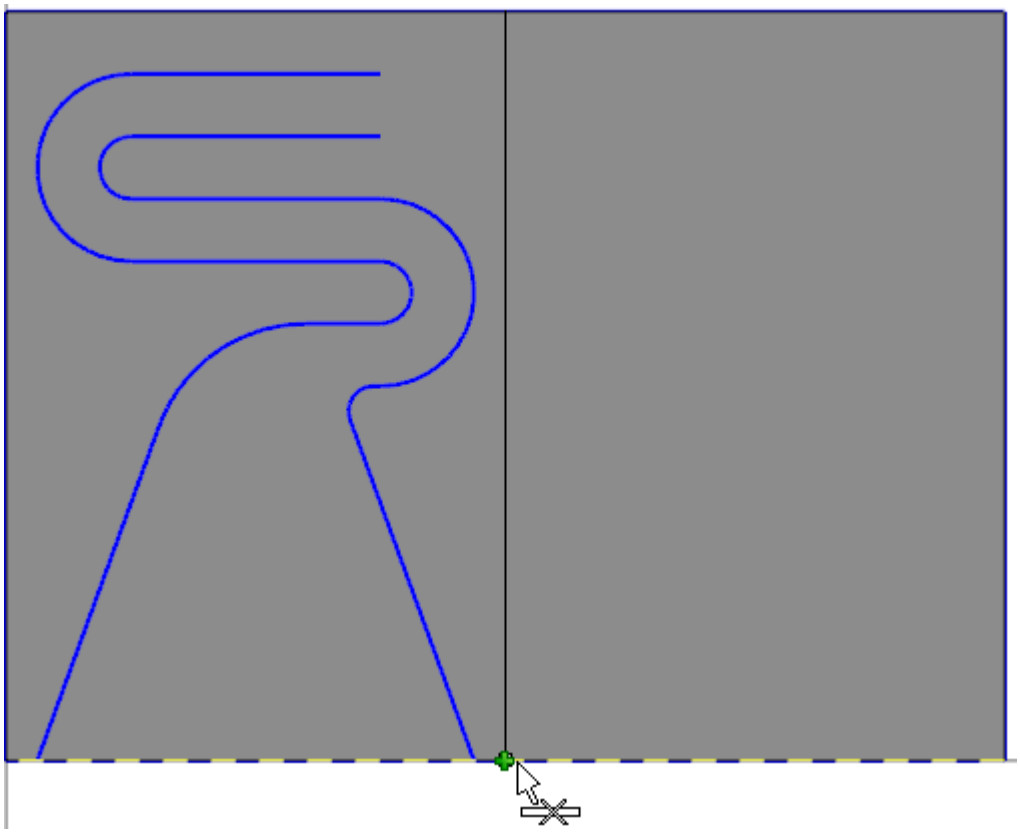


   The **Line Endpoints** function panel displays.

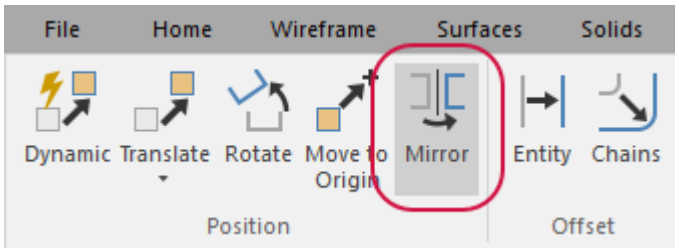
3. Set **Type** to **Vertical**.



4. Select the midpoint of the top line and select the midpoint of the bottom line to create a line across the part.

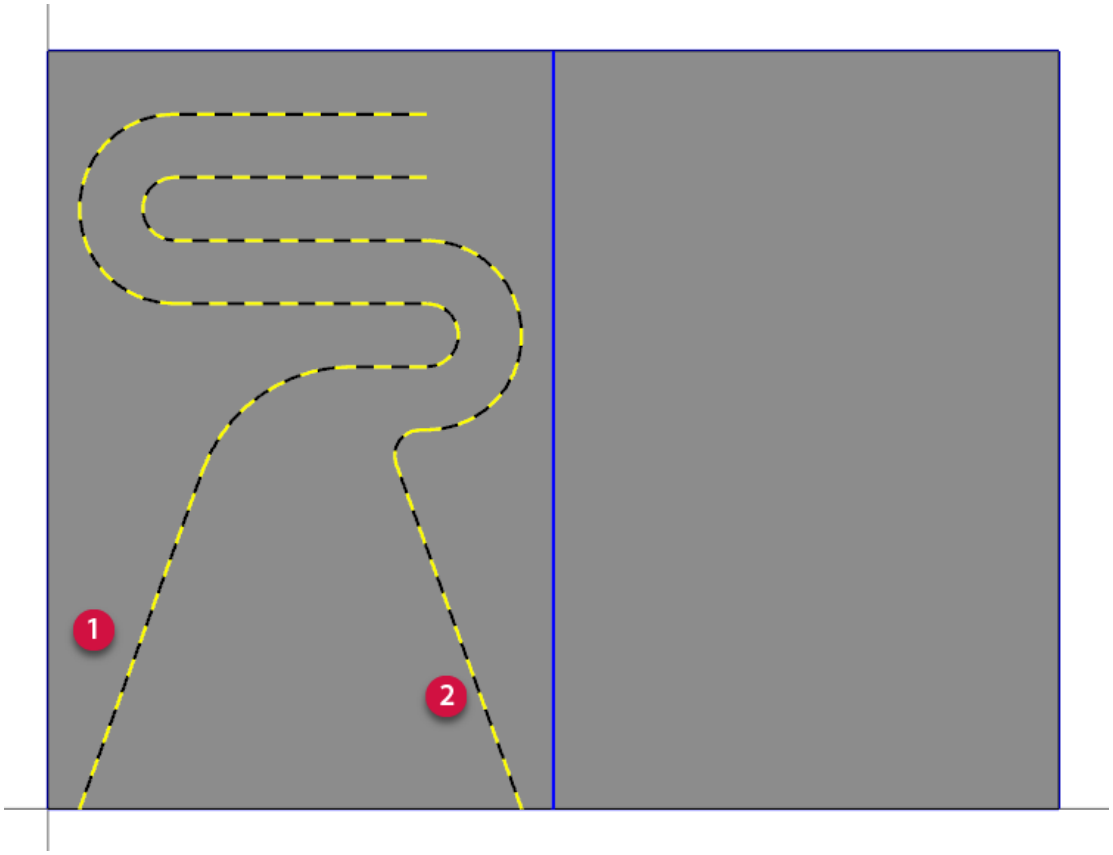


5. Click **OK** in the **Line Endpoints** function panel.
6. Select **Mirror** on the **Transform** tab.



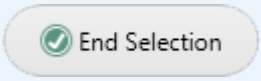
The **Mirror** function panel displays.

7. Hold down the [**Shift**] key and select the two lines you have created.



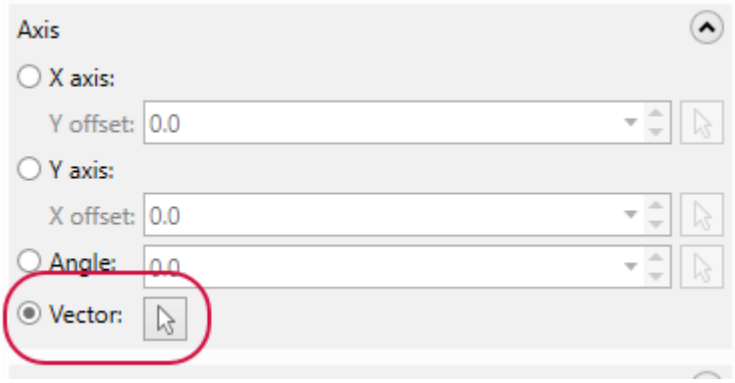
8. Press [**Enter**].

   The **Mirror** function panel is now active.

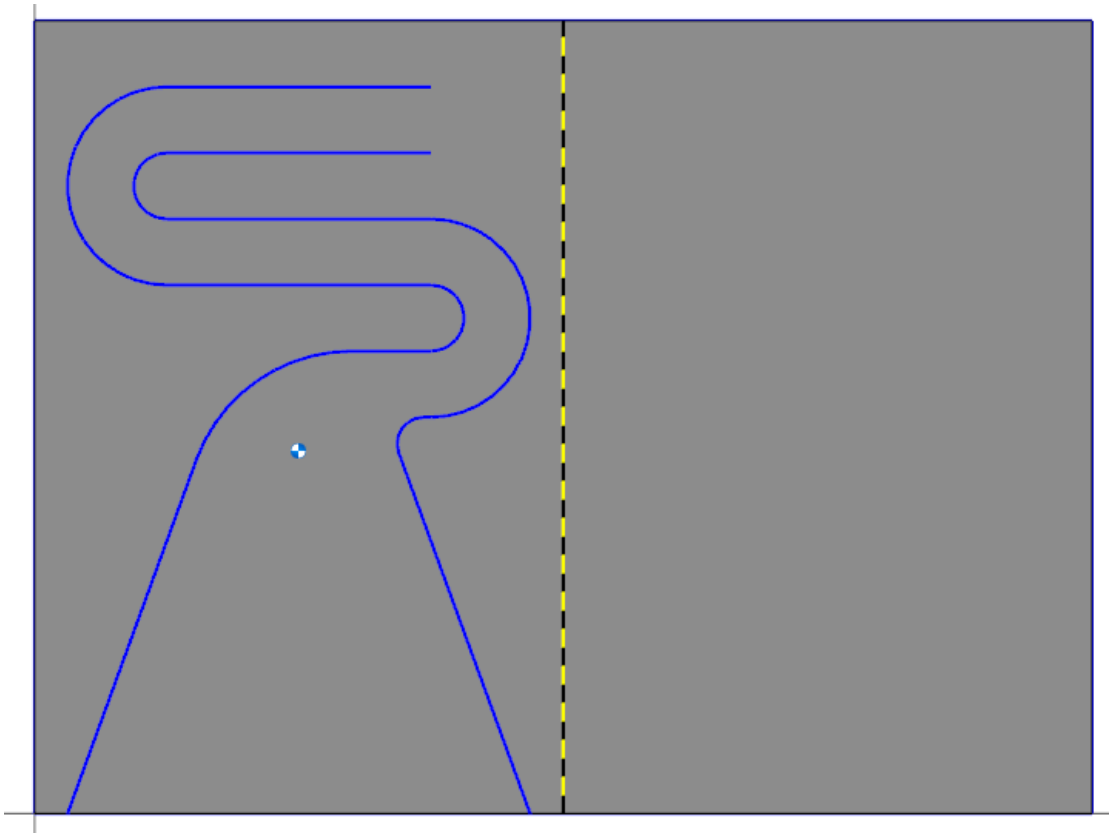
> **NOTE**
> You can also select **End Selection** to finish selecting geometry instead of pressing [**Enter**].
>
> 
>

> **NOTE**
> When you first enter in the **Mirror** function, the entities will mirror about the X axis by default. It will be corrected in the next steps.

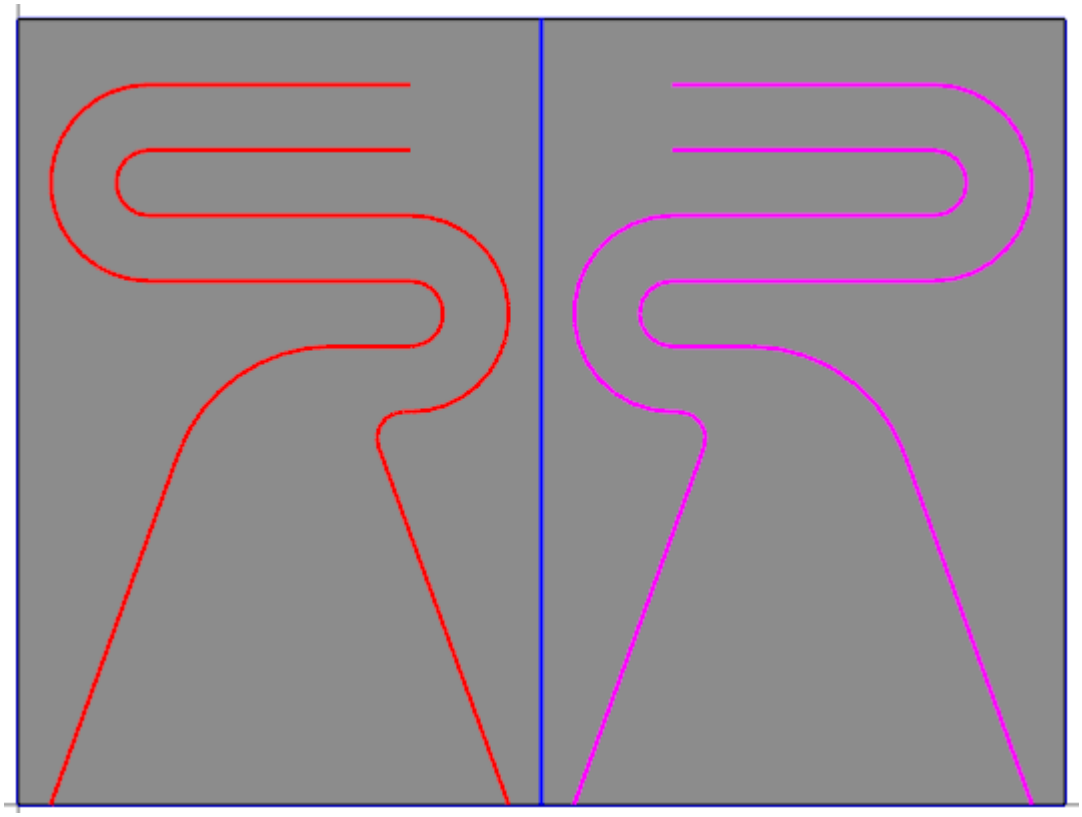
9. Set **Axis** to **Vector** and click **Select Vector**.



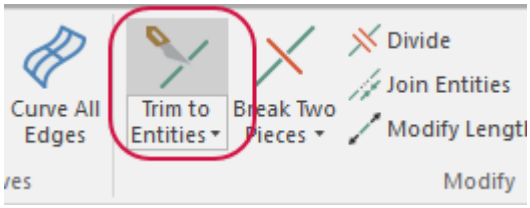
10. Select the vertical line as the mirror axis.



11. Click **OK** in the **Mirror** function panel.

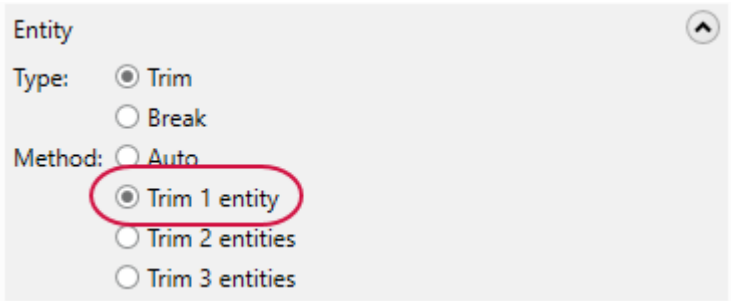


12. Select **Trim to Entities** on the **Wireframe** tab.

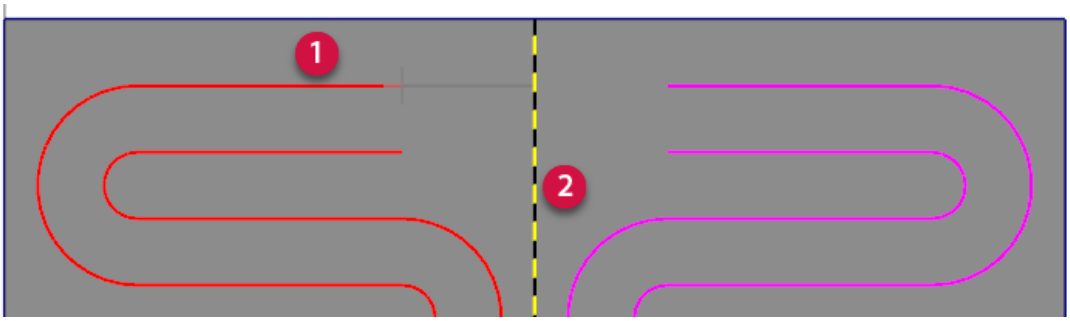


The **Trim to Entities** function panel displays.

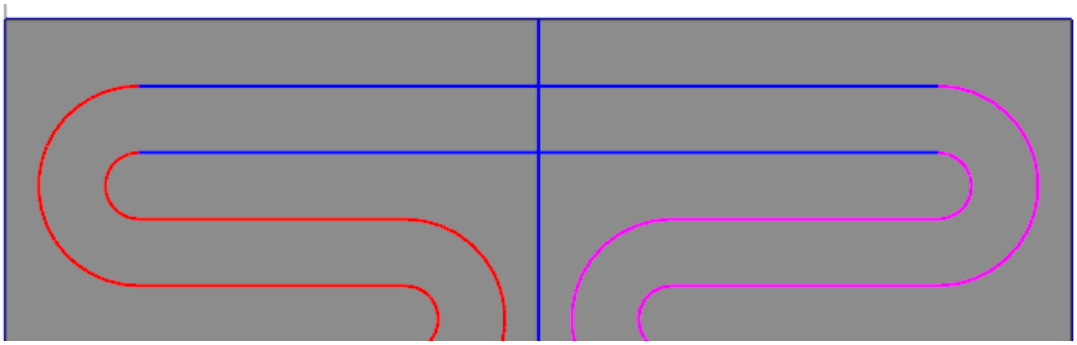
13. Set **Method** to **Trim 1 entity**.



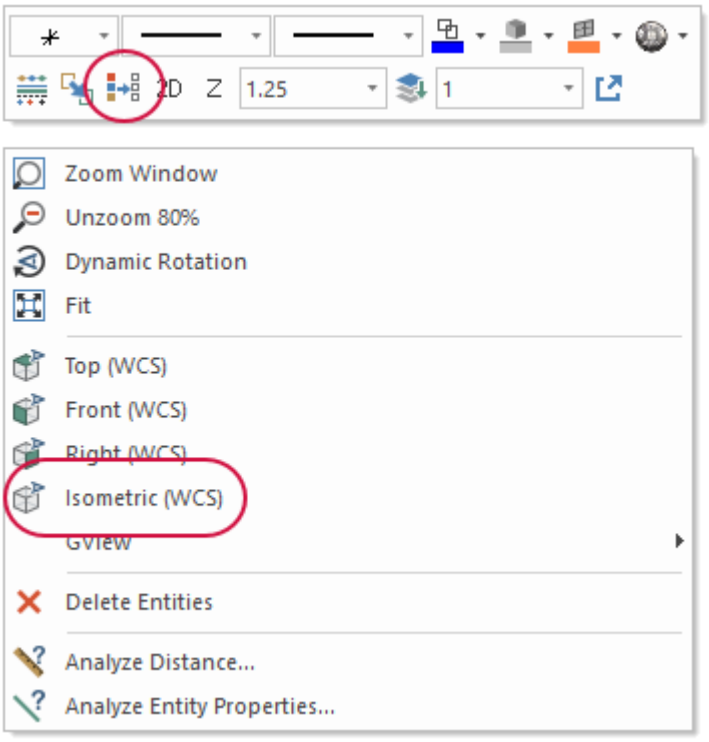
14. Select the top line and then the vertical axis line as shown in the following image.



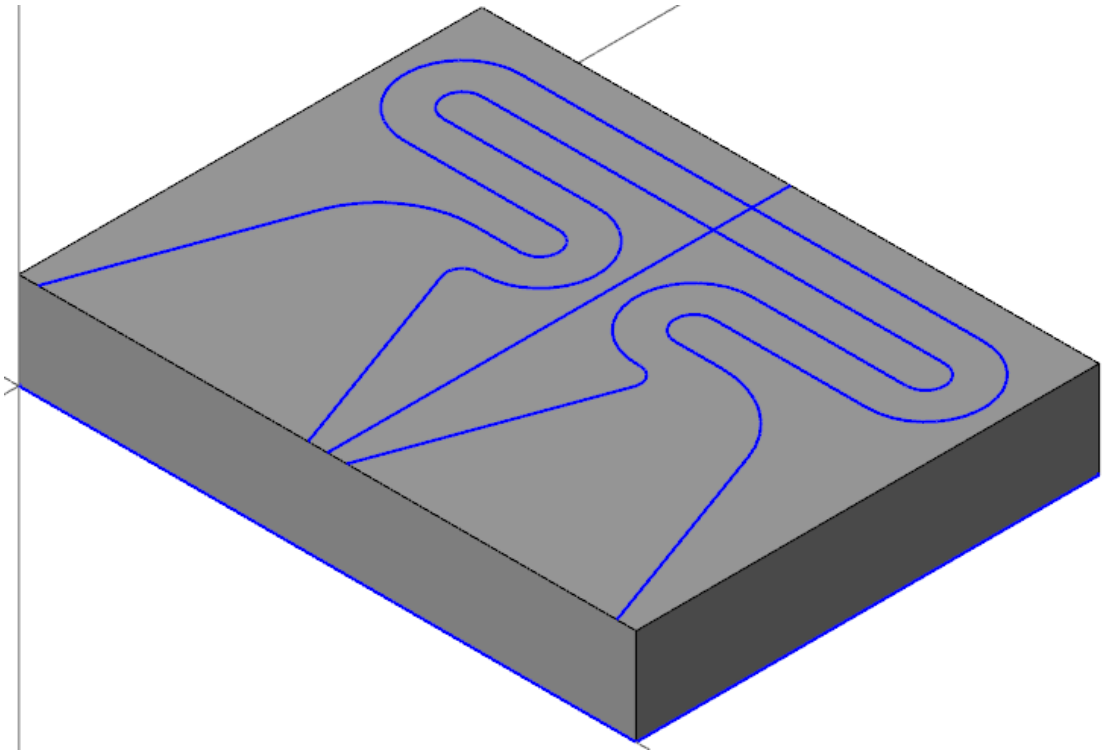
15. Repeat this process for the other three lines.



16. Click **OK** in the **Trim to Entities** function panel.
17. Right-click in the graphics window and select **Clear Colors**. Then, select **Isometric (WCS)**.



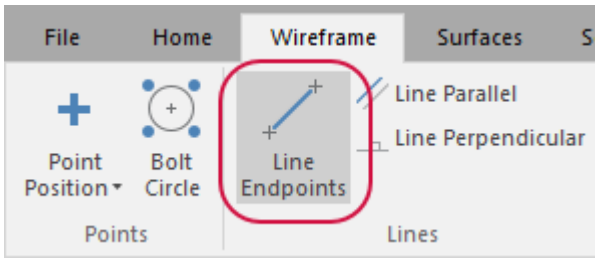
18. Save your file.



Exercise 8: Extruding the Geometry

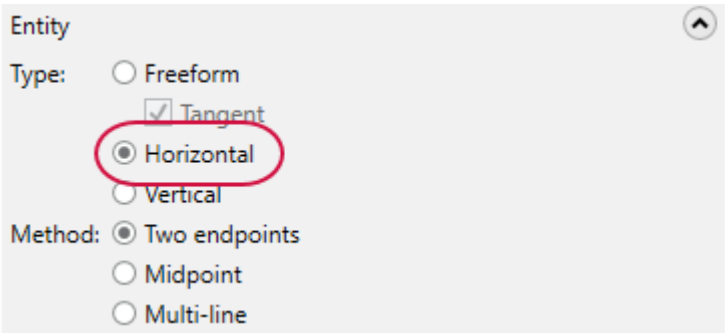
In this exercise, you will extrude the geometry that was created and mirrored in the previous exercises.

1. Your file Amplifier-XXX which you saved from the previous exercise should be open in Mastercam.
2. Select **Line Endpoints** on the **Wireframe** tab.

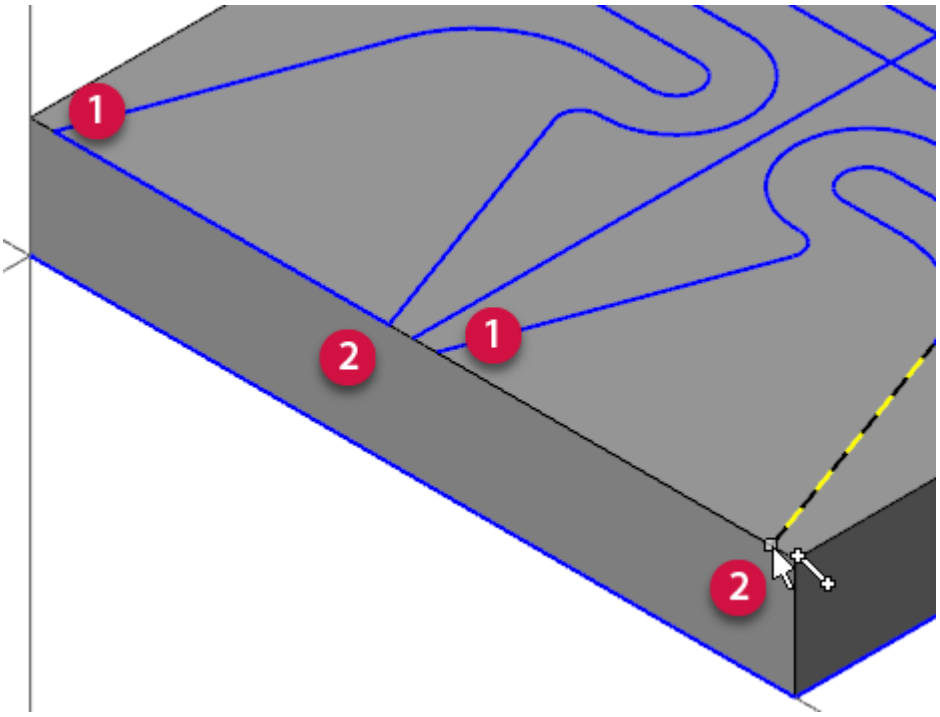


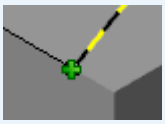
The **Line Endpoints** function panel displays.

3. Set **Type** to **Horizontal**.

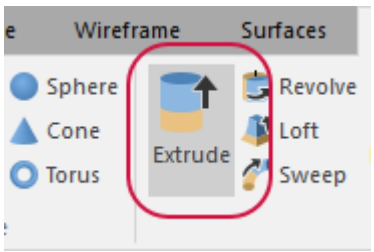


4. Select the following endpoints to close the open pockets.



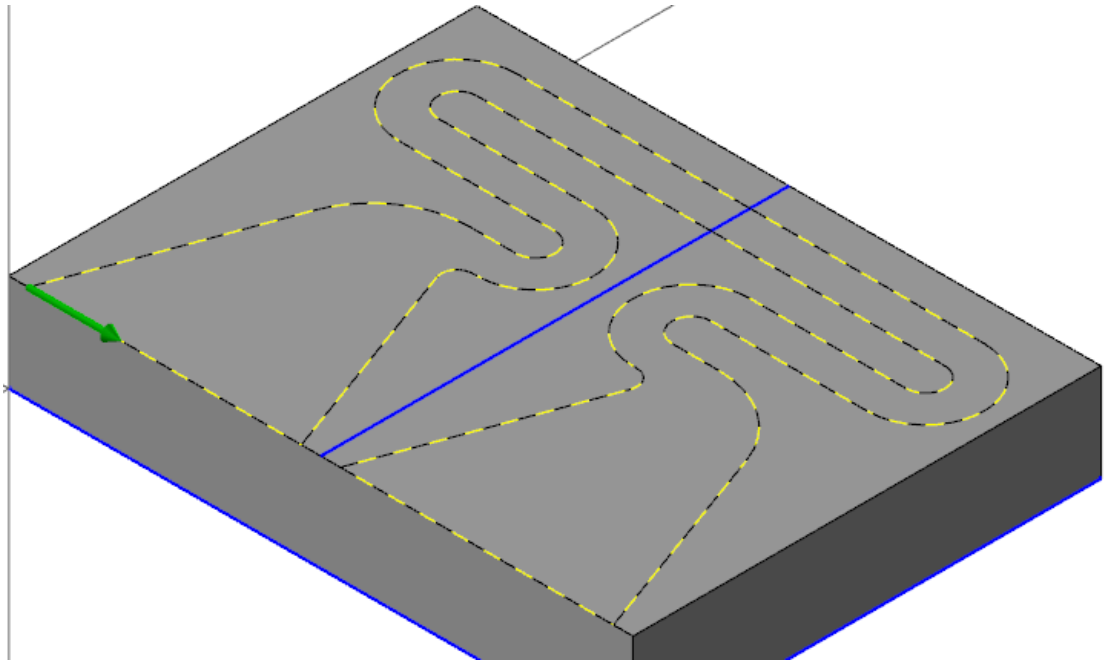
> **NOTE**
> Ensure that you are selecting the endpoint of the lines. Wait for the green endpoint icon before clicking.
>
> 
>
> You can also press [**Alt+S**] to show the part as wireframe to view the lines more clearly.

5. Click **OK** in the **Line Endpoints** function panel.
6. Select **Extrude** on the **Solids** tab.



The **Chaining** dialog box displays.

7. Select the chain as shown below.



> **NOTE**
> Ensure that you see a green arrow on the chain you selected. If you see a red arrow, then the chain is not closed and you will not be able to extrude the chain.

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

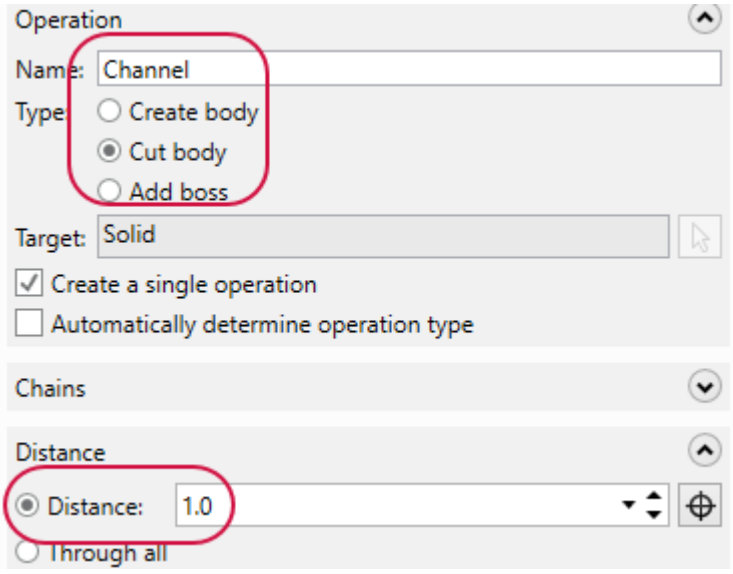
   The **Solid Extrude** function panel displays.

9. Set the following parameters:

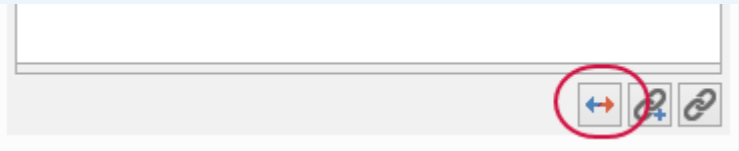
   a. Enter **Channel** for **Name**.

   b. Set **Type** to **Cut body**.

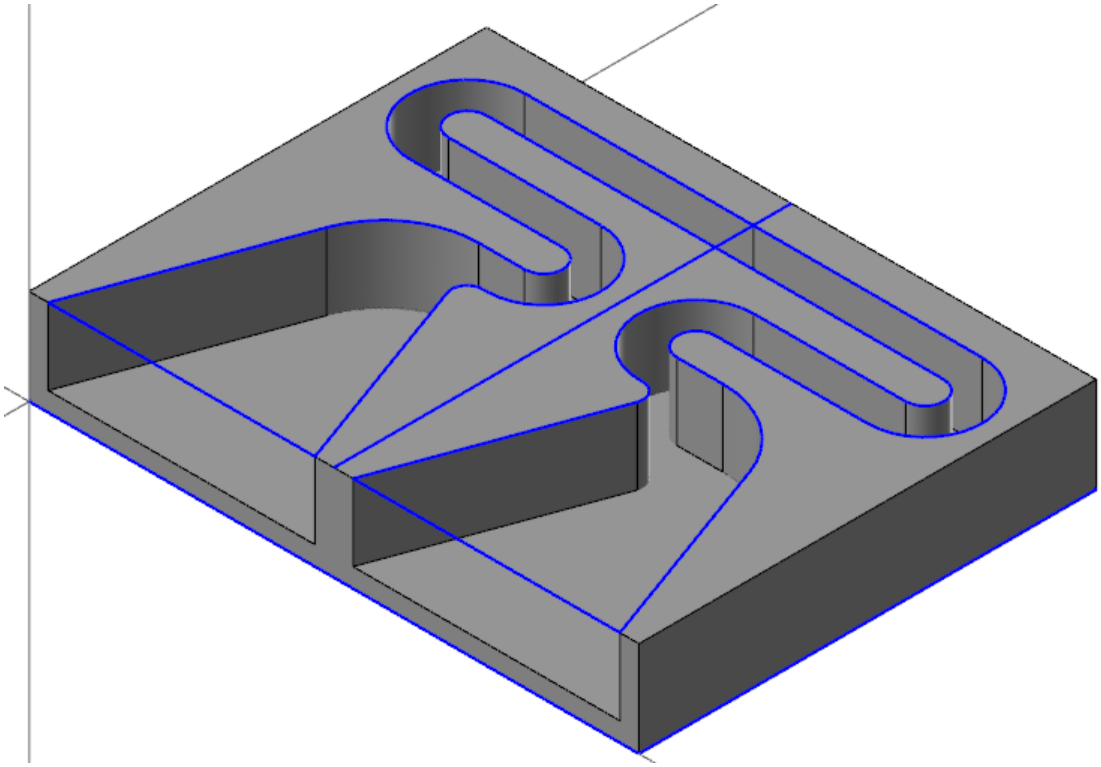
   c. Enter **1.0** for **Distance**.



NOTE
If the channel is not extruding down to cut the part select **Reverse All** to reverse the direction of the extrusion chain.



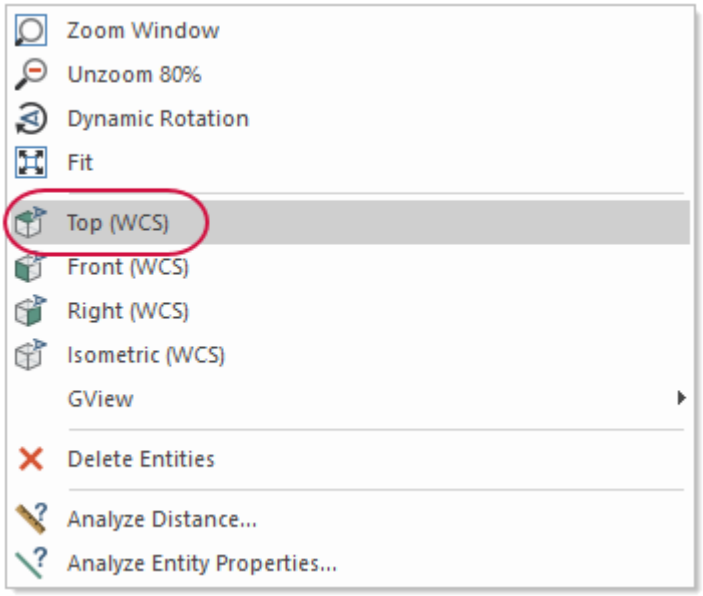
10. Click **OK** in the **Solid Extrude** function panel.
11. Save your file.



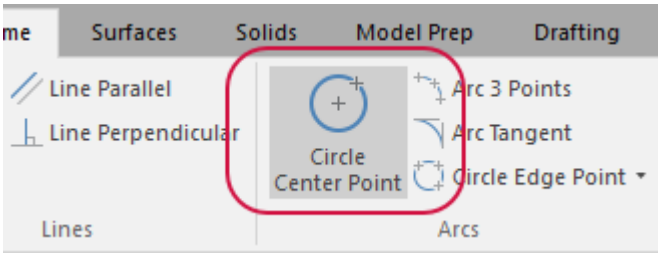
Exercise 9: Creating the Index Holes

In this exercise, you will create the index holes used to combine the top and bottom pieces of the acoustic amplifier.

1. Your file Amplifier-XXX which you saved from the previous exercise should be open in Mastercam.
2. Right-click in the graphics window and select **Top (WCS)**.



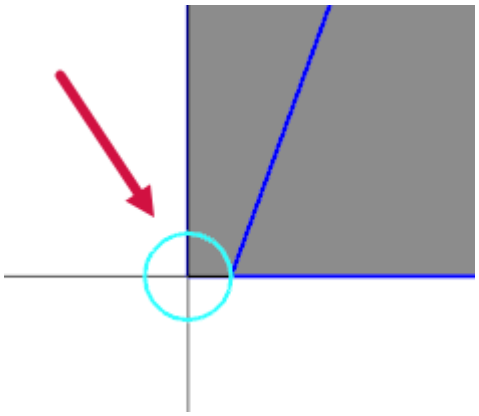
3. Select **Circle Center Point** on the **Wireframe** tab.



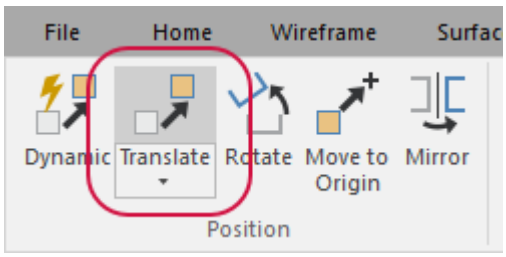
The **Circle Center Point** function panel displays.

4. Enter **0.504** for **Diameter**.

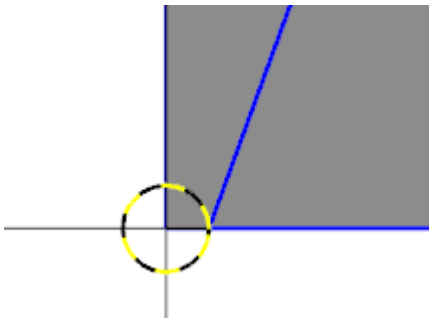
5. Create the arc at the center of the origin.



6. Click **OK** in the **Circle Center Point** function panel.
7. Select **Translate** on the **Transform** tab.

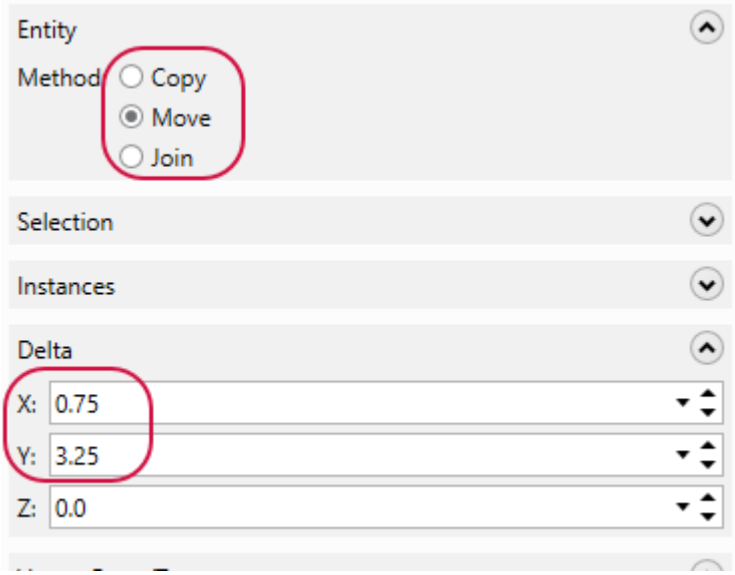


8. Select the circle and press [**Enter**].

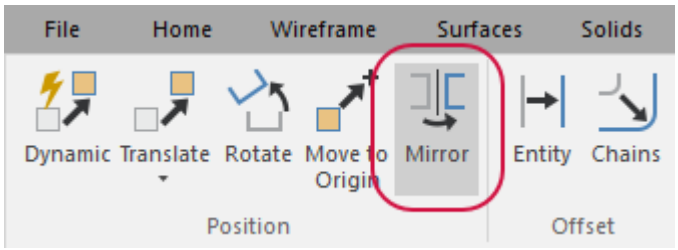


The **Translate** function panel becomes active.

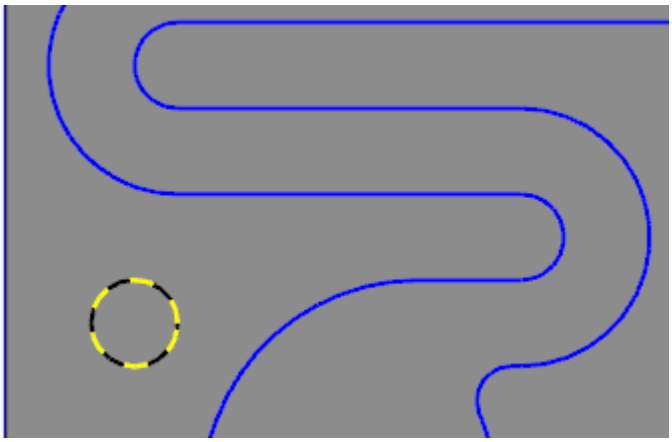
9. Set the following parameters:
    a. Set **Method** to **Move**.
    b. Enter **0.75** for **X**.
    c. Enter **3.25** for **Y**.



10. Click **OK** in the **Translate** function panel.
11. Select **Mirror** on the **Transform** tab.

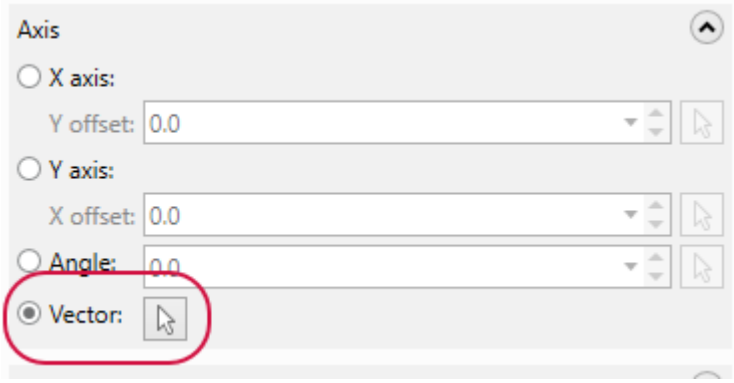


12. Select the circle and press [**Enter**].

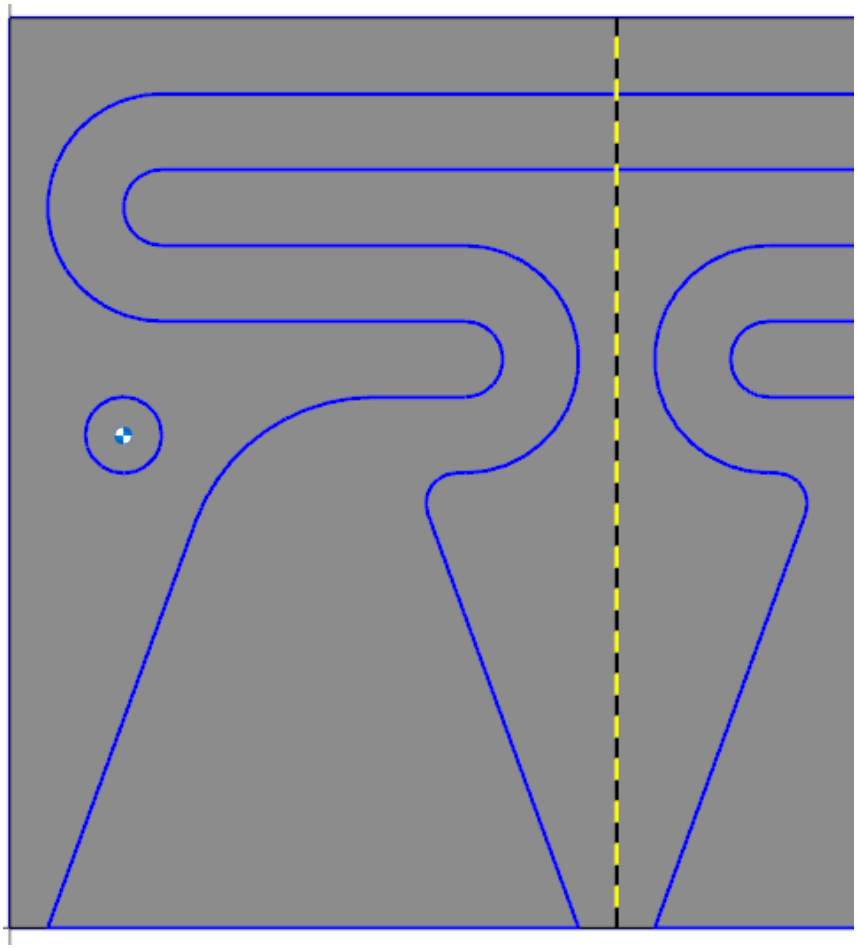


The **Mirror** function panel is now active.

13. Ensure that **Axis** is set to **Vector**. Click **Select Vector** to return to the graphics window.

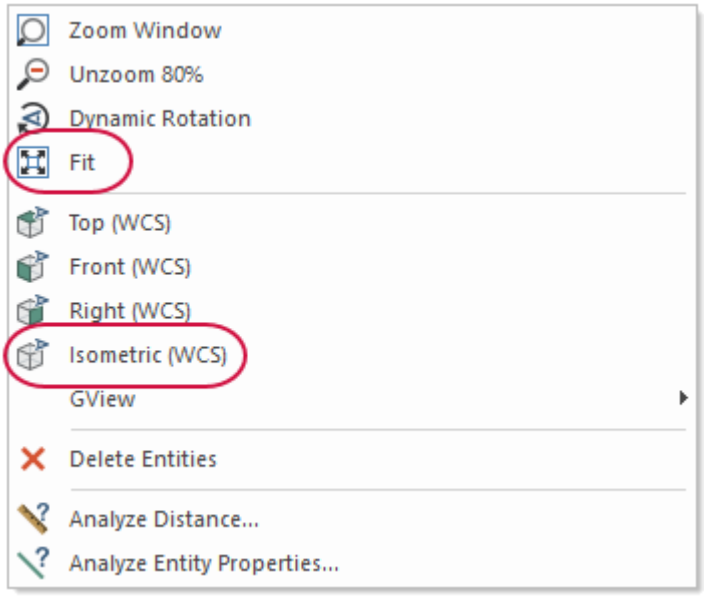


14. Select the vertical line shown below.

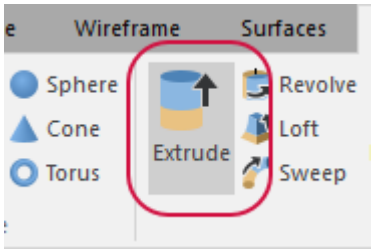


15. Click **OK** in the **Mirror** function panel.

16. Right-click in the graphics window and select **Fit**. Then select **Isometric (WCS)**.

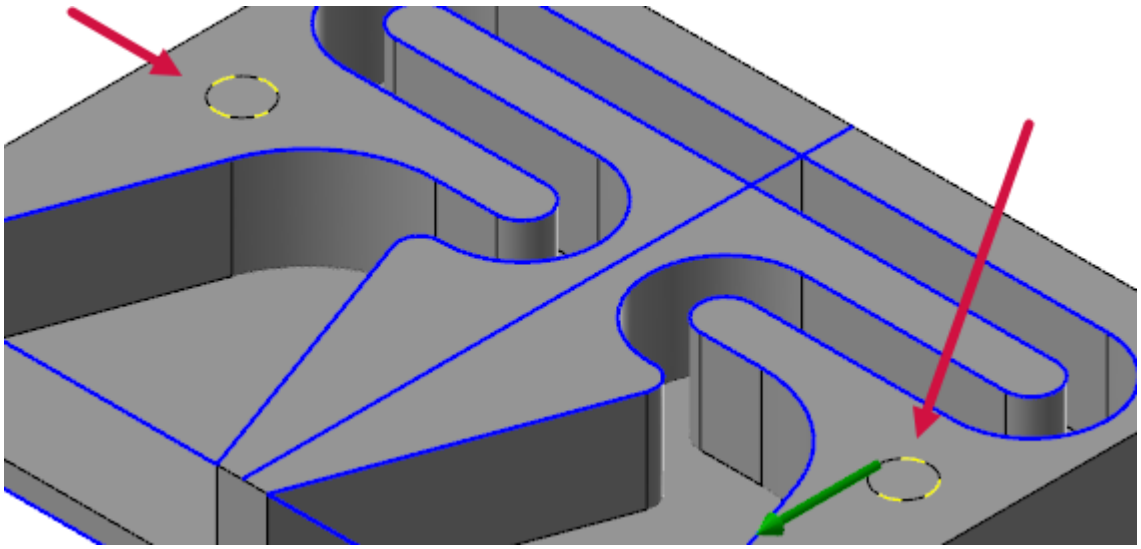


17. Select **Extrude** on the **Solids** tab.



The **Chaining** dialog box displays.

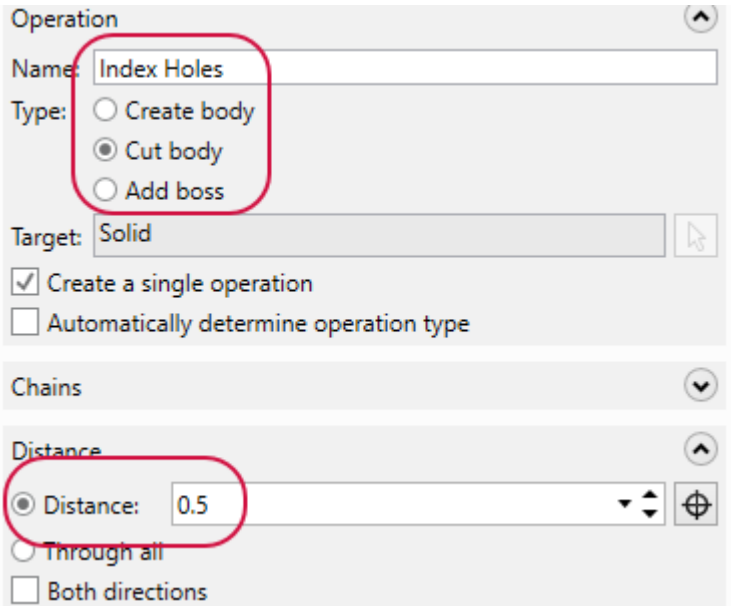
18. Select the two wireframe circles you just created.



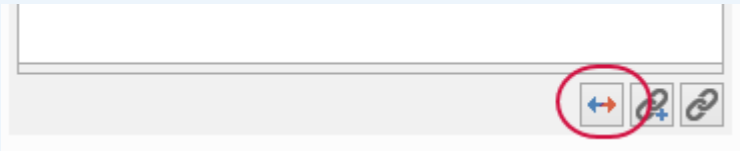
19. Click **OK** in the Chaining dialog box.

The **Solid Extrude** function panel displays.

20. Set the following parameters:
    a. Enter **Index Holes** for **Name**.
    b. Ensure that **Type** is set to **Cut Body**.
    c. Enter **0.5** for **Distance**.

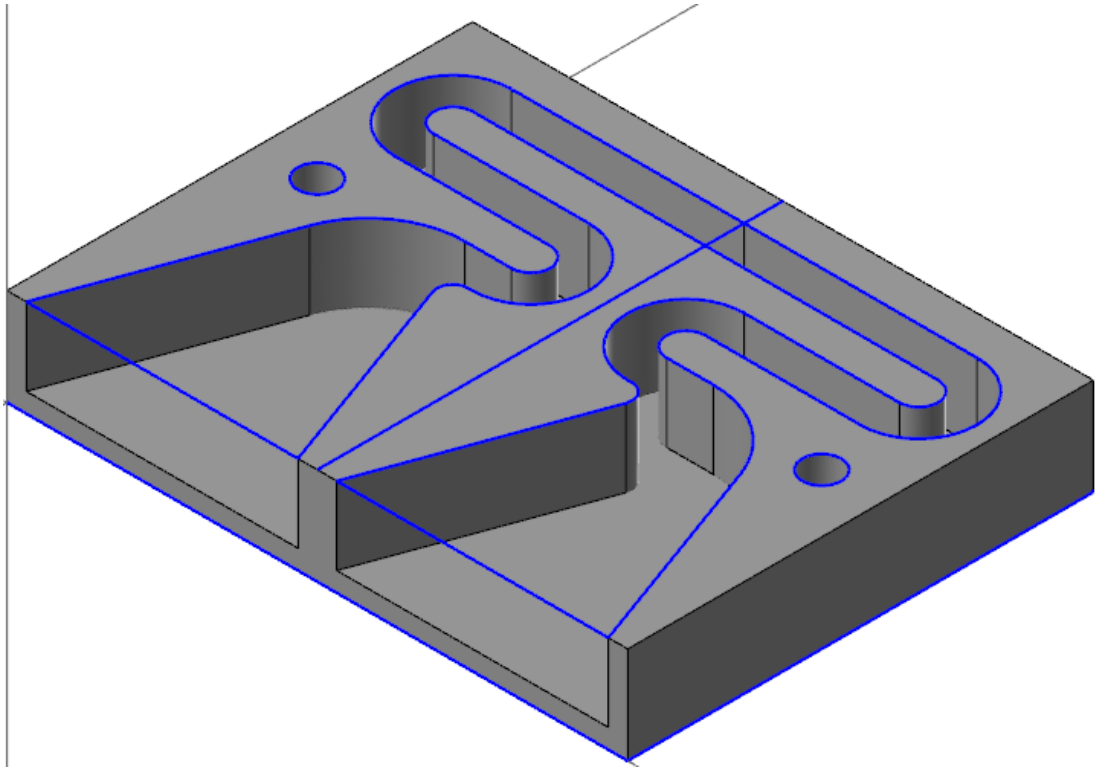


> **NOTE**
> If the index holes are not extruding down to cut the part select **Reverse All** to reverse the direction of the extrusion chains.



21. Click **OK** in the **Solid Extrude** function panel.
22. Right-click in the graphics window and select **Clear Colors**.

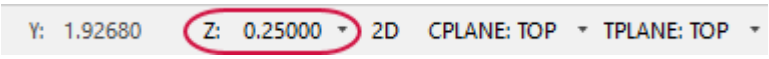
23. Save your file.



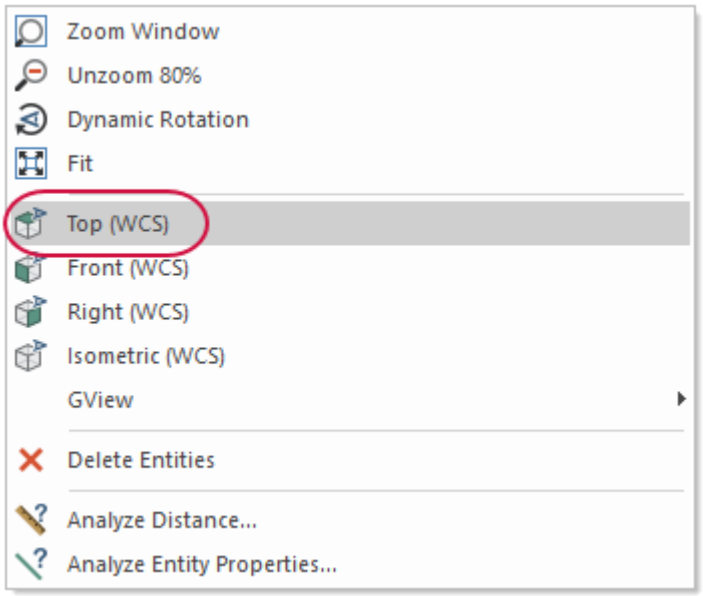
Exercise 10: Creating the Phone Slot Geometry

In this exercise, you will create the geometry necessary for the phone slot. For this exercise, the dimensions given are for a Samsung Galaxy® 9 phone.

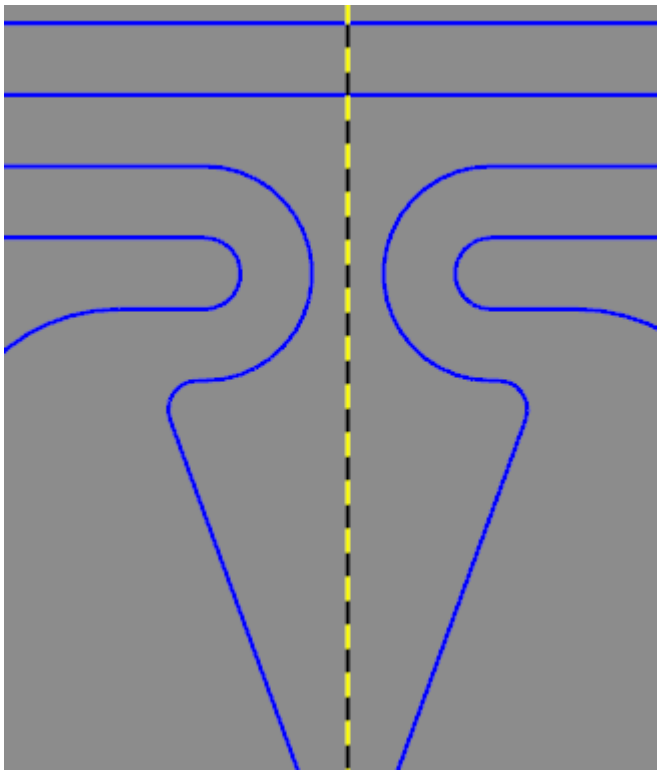
1. Your file Amplifier-XXX which you saved from the previous exercise should be open in Mastercam.
2. In the **Status Bar** at the bottom of the Mastercam window, click the value next to **Z** and enter **0.25**. Press [**Enter**] to apply the value.



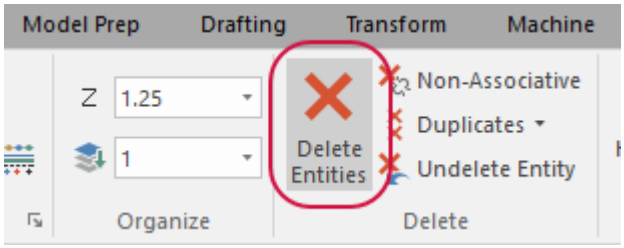
3. Right-click in the graphics window and select **Top (WCS)**.



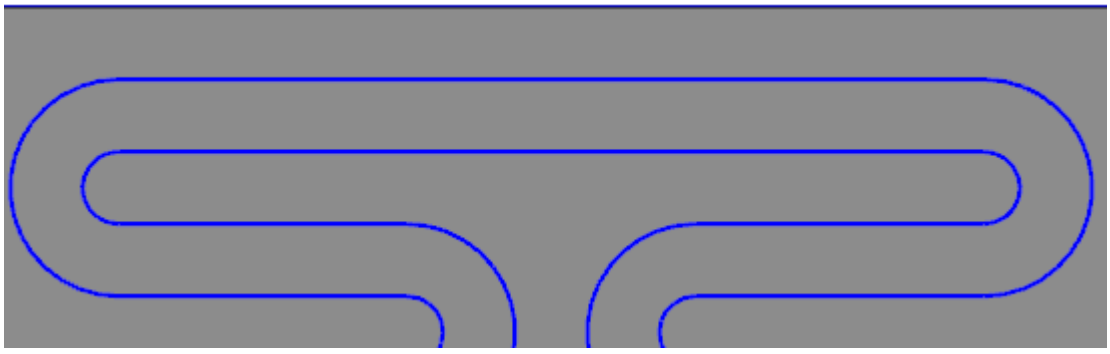
4. Select the vertical line used as the vector line.



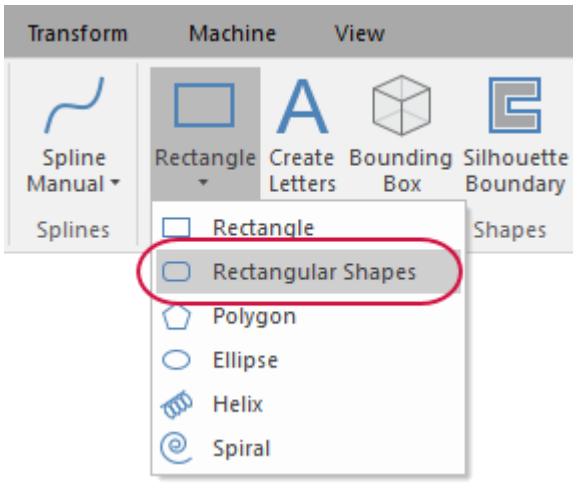
5. Select **Delete Entities** on the **Home** tab or press the [**Delete**] key to remove the selected line from the part file.



The part displays as shown below.

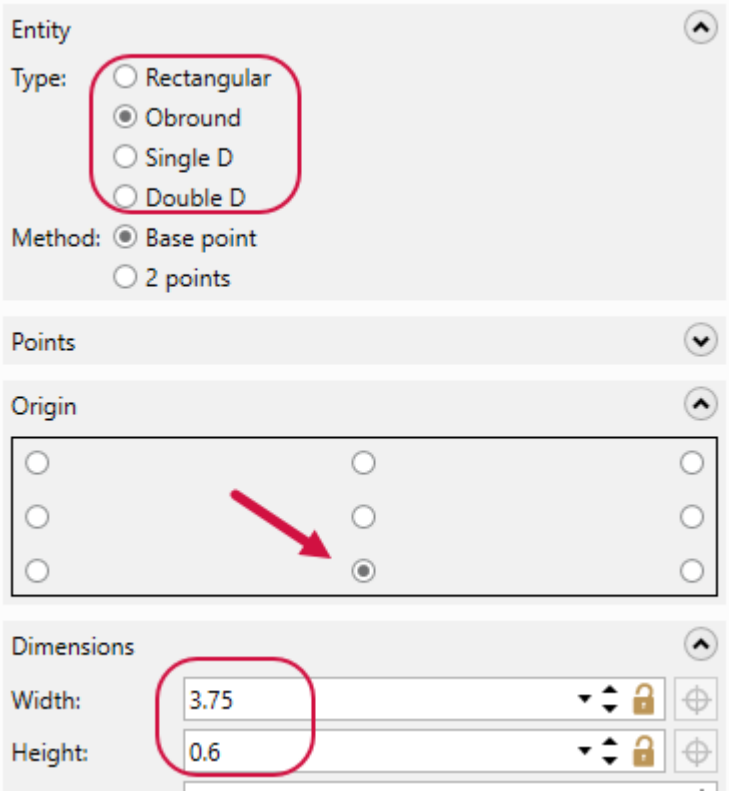


6. Select **Rectangle**, **Rectangular Shapes** on the **Wireframe** tab.

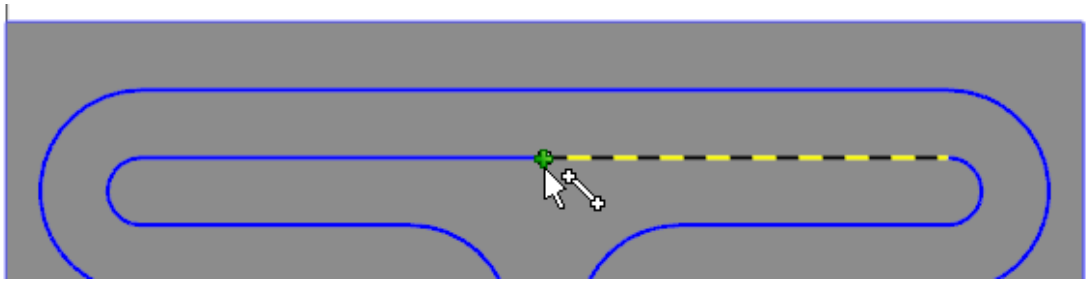


The **Rectangular Shapes** function panel displays.

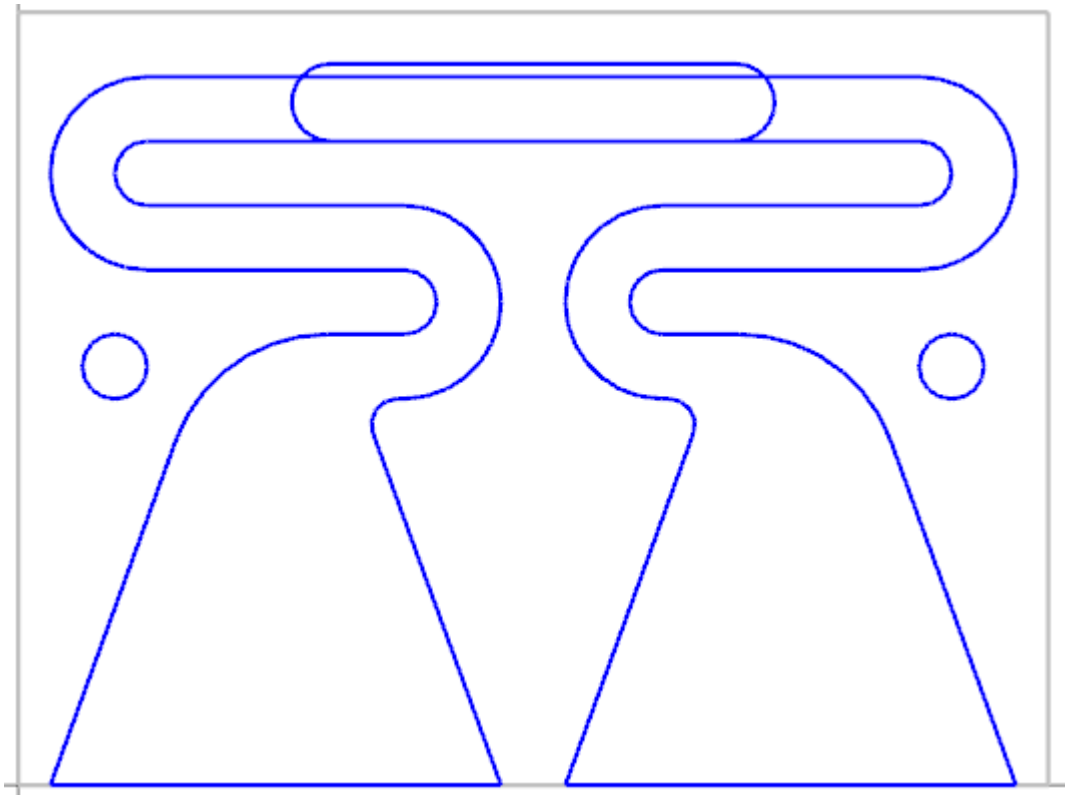
7. Enter the following parameters:
    a. Set **Type** to **Obround**.
    b. Set **Origin** to the bottom middle point.
    c. Enter **3.75** for **Width**.
    d. Enter **0.6** for **Height**.



8. Select the endpoint of the line segment as shown below.



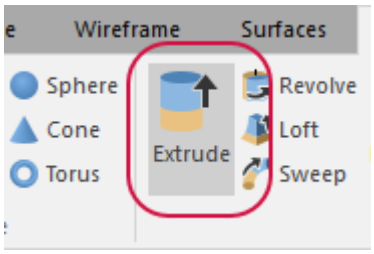
9. Click **OK** in the **Rectangular Shapes** function panel.
10. Press [**Alt+S**] to display the part only as wireframe.
11. Save your file.



Exercise 11: Extruding the Phone Slot Geometry

In this exercise, you will extrude the phone slot geometry created in the previous exercises.

1. Your file Amplifier-XXX which you saved from the previous exercise should be open in Mastercam.
2. Select **Extrude** on the **Solids** tab.



   The **Chaining** dialog box displays.

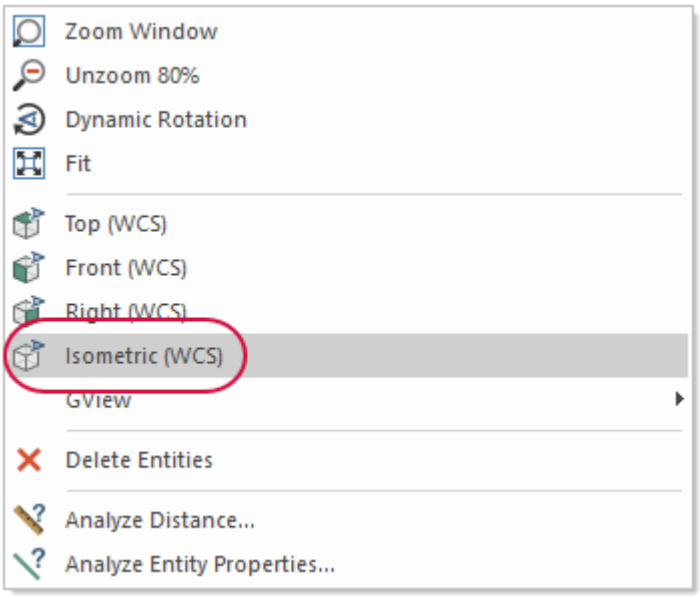
3. Select the obround shape you just created as shown below.

> **NOTE**
> It may be easier to select the chain from the top line. Additionally, ensure that you see a green arrow on the chain you selected. If you see a red arrow, then the chain is not closed and you will not be able to extrude the chain.

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

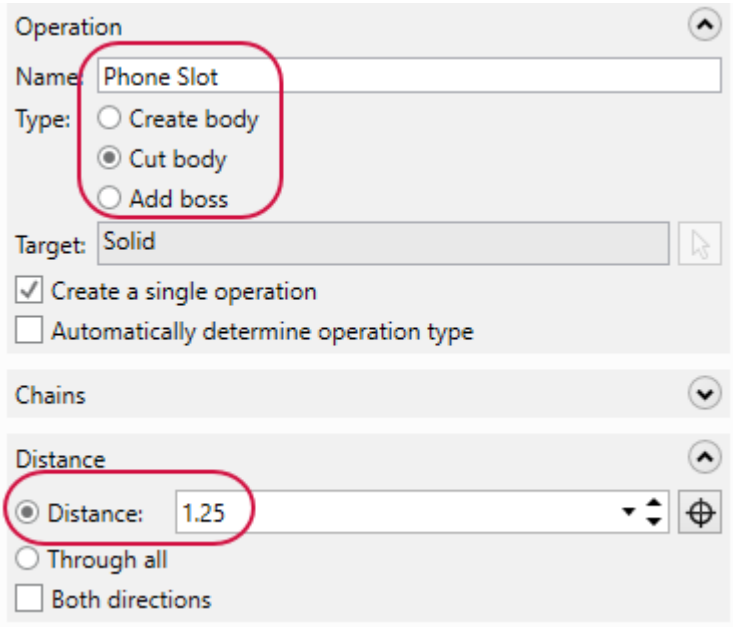
   The **Solid Extrude** function panel displays.

5. Right-click in the graphics window and select **Isometric (WCS)**.

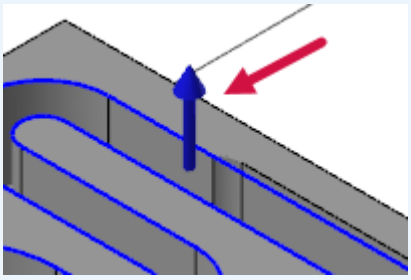


6. Press [**Alt+S**] to shade the part.

7. Enter the following parameters:
    a. Enter **Phone Slot** for **Name**.
    b. Ensure that **Method** is set to **Cut body**.
    c. Set **Distance** to **1.25**.

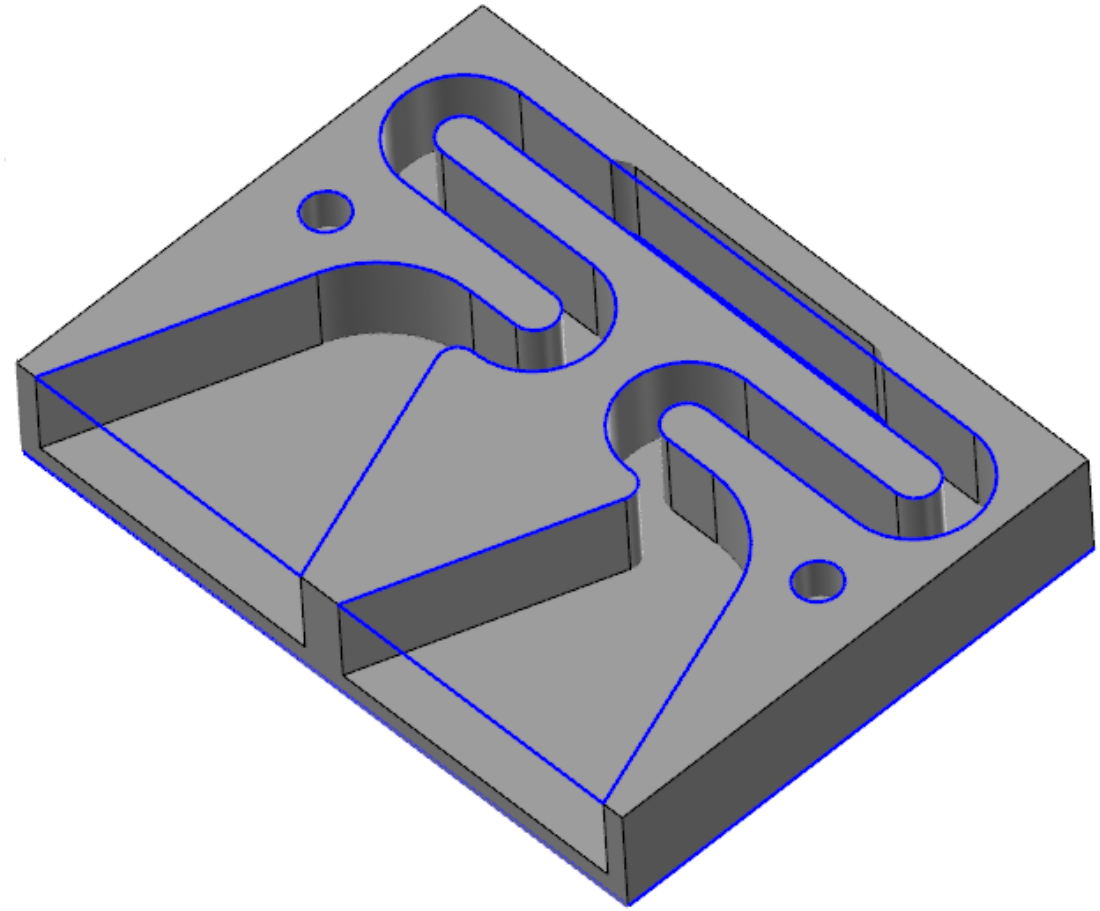


NOTE
Ensure that the direction chain is pointing up. Select **Reverse All** if it is not.



8. Click **OK** in the **Solid Extrude** function panel.

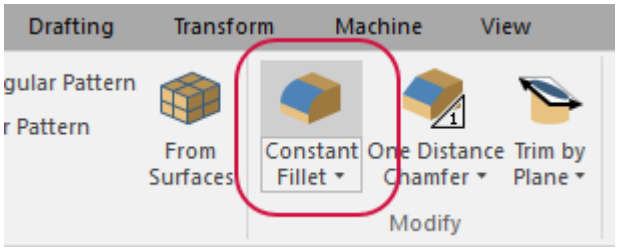
9. Save your file.



Exercise 12: Adding Fillets to Part Edges

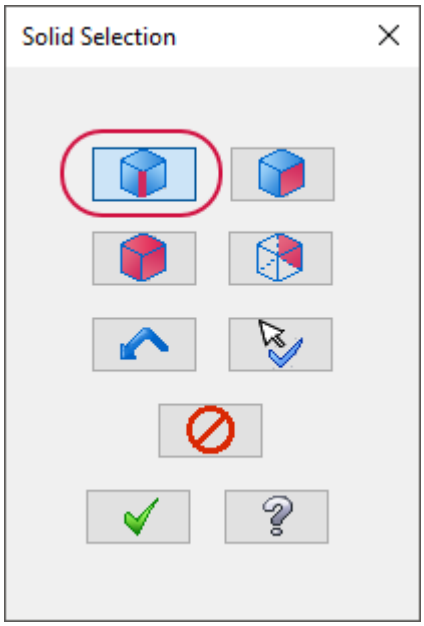
In this exercise, you will add fillets to the outside edges of the part.

1. Your file Amplifier-XXX which you saved from the previous exercise should be open in Mastercam.
2. Select **Constant Fillet** from the **Solids** tab.



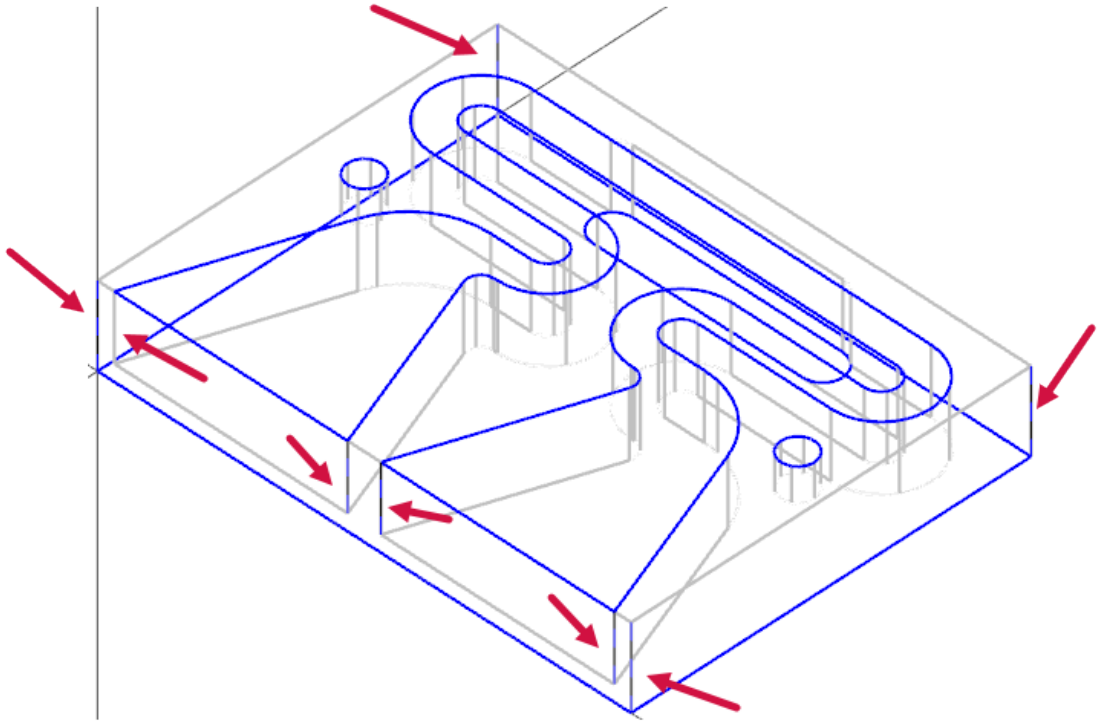
The **Solid Selection** dialog box displays.

3. Ensure that the **Edge** button is selected. Click any other highlighted button to clear that selection filter.



   Only the **Edge** button will be highlighted.

4. Press [**Alt+S**] to display the part only as wireframe.
5. Select the following edges.

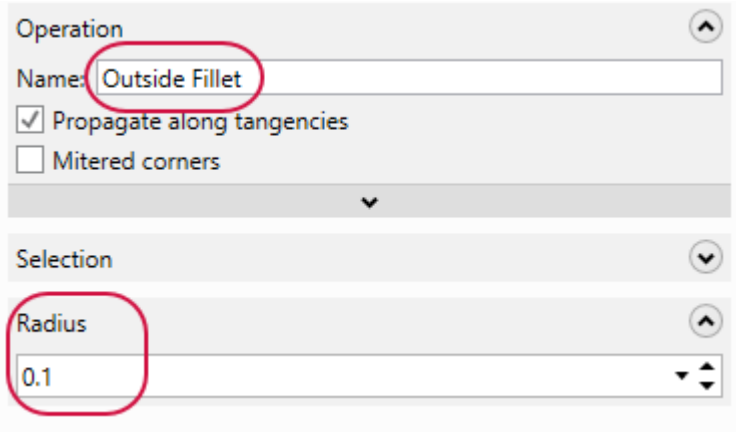


6. Click **OK** in the **Solid Selection** dialog box.

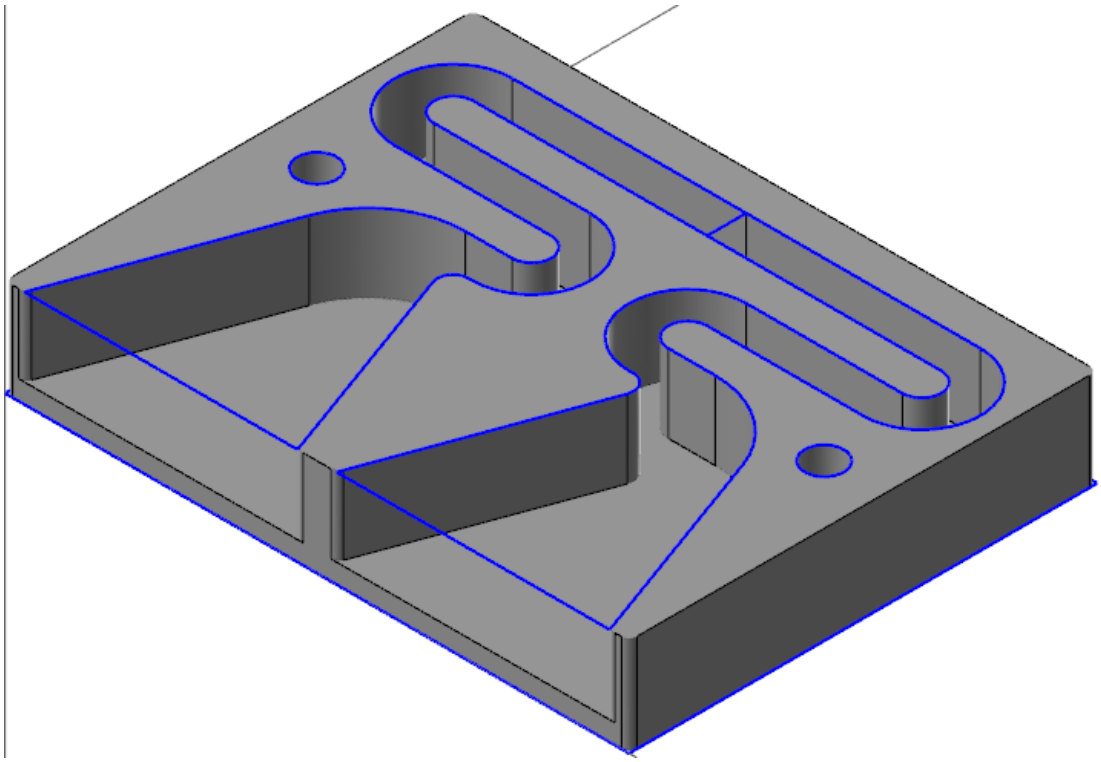
   The **Constant Radius Fillet** function panel displays.

7. Press [**Alt+S**] to shade the part.

8. Set the following parameters:
    a. Enter **Outside Fillet** for **Name**.
    b. Enter **0.1** for **Radius**.



9. Click **OK** in the **Constant Radius Fillet** function panel.
10. Save your file.



SHOW US WHAT YOU LEARNED!

Can you answer these questions?

1. When using the Mirror function, which of the following is *not* used as a mirror axis?
    a. Single point
    b. X axis
    c. Angle
    d. Vector
2. When creating horizontal lines using the Line Endpoints function, you can create a line with a specific angle.
    a. True
    b. False
3. When chaining geometry to extrude as a solid, the chained geometry must be flat and closed.
    a. True
    b. False
4. The Z Depth setting sets the depth of the current construction plane.
    a. True
    b. False
5. You can only create fillets with wireframe.
    a. True
    b. False
6. Which of the following is a type of shape you can create with the Rectangular Shapes function?
    a. Ellipse
    b. Obround
    c. Rhombus
    d. Polygon
7. What are the differences between the Trim to Entities and the Divide functions?

CHAPTER 2

PROGRAMMING THE INDEX HOLES

In this chapter, you set up the stock to be machined. Then, you create the Circle Mill toolpath that will be used to mill the two index holes.

Goals

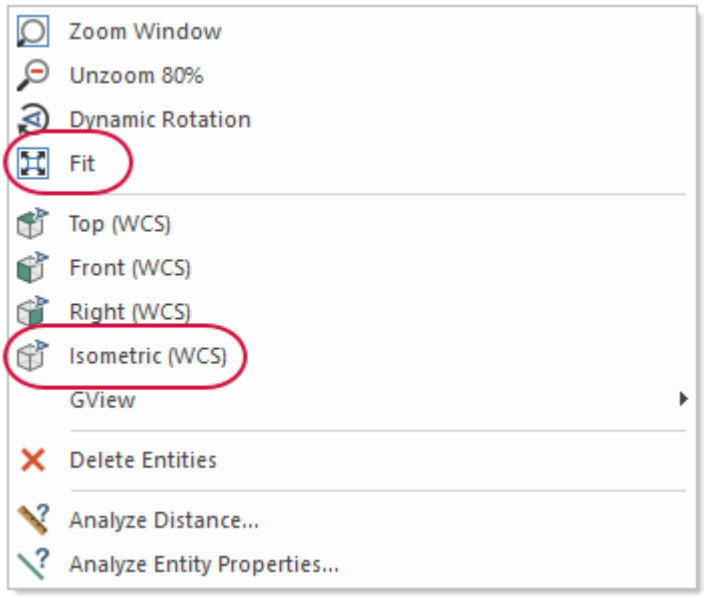
- Set up the stock.
- Program a Circle Mill toolpath.
- Explore tool libraries including filter by tool type.
- Experiment with toolpath verification using Mastercam Simulator.

> **CAUTION**
> When programming toolpaths, only change the values of the indicated parameters. Leave unmentioned values at their default values.

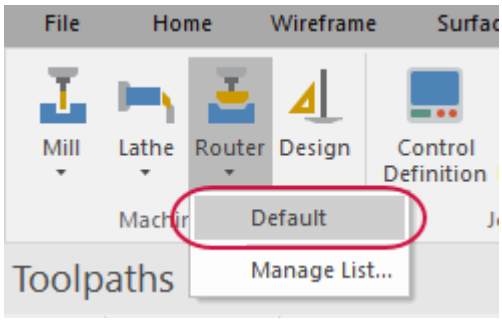
Exercise 1: Setting Up the Stock

In this exercise, you choose a machine definition and set up your stock.

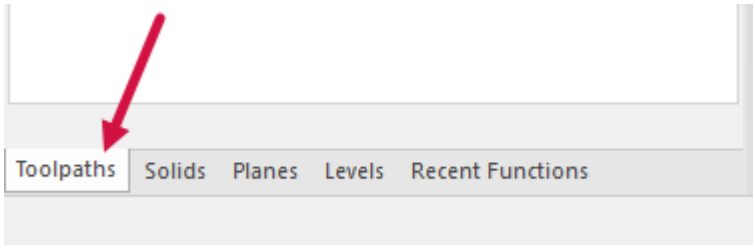
1. Your file Amplifier-XXX which you saved from the previous exercise should be open in Mastercam.
2. If necessary, right-click in the graphics window and select **Fit**. Then, select **Isometric (WCS)**.



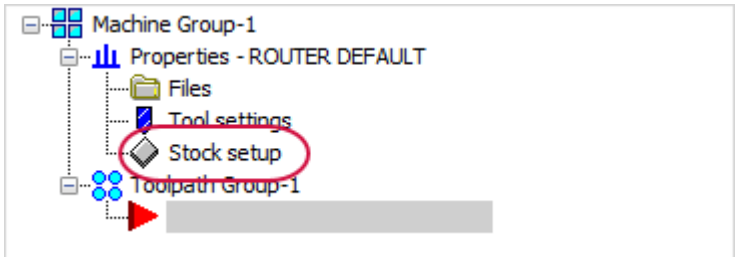
3. Select **Router**, **Default** on the **Machine** tab.



4. Select the **Toolpaths** tab to bring the Toolpaths Manager to the forefront.



5. Expand the **Properties** group and select **Stock setup**. Note that the **Machine Group-1** is the **Router Default**.



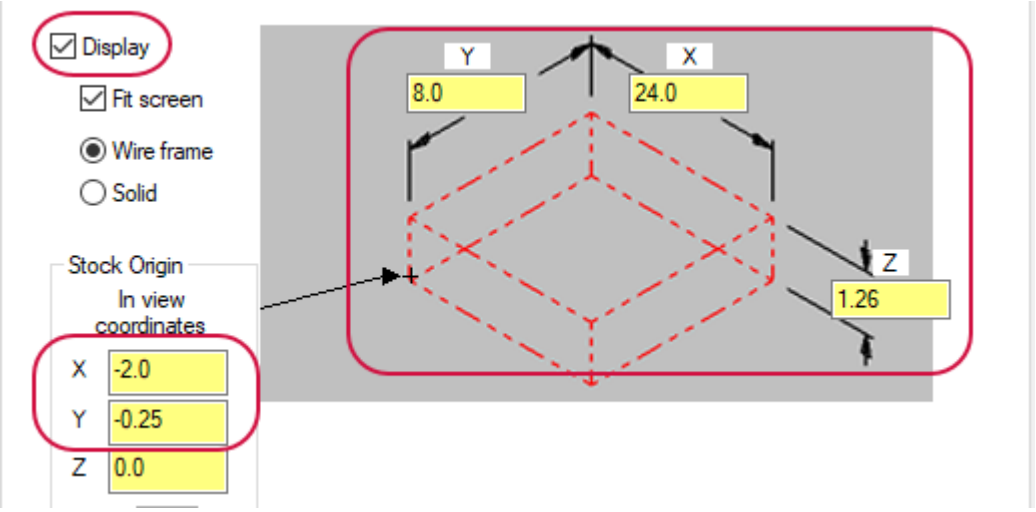
The **Machine Group Properties** dialog box displays.

6. Set the following parameters:

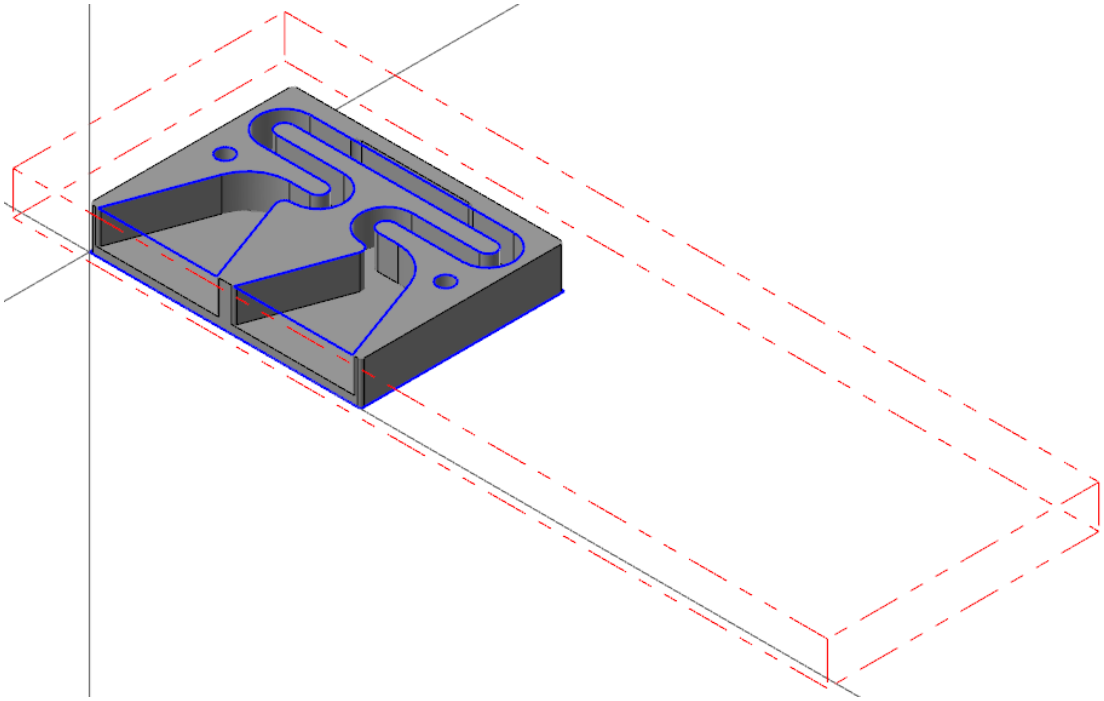
> **CAUTION**
> These values may vary depending on your machine and material. Make sure to measure your materials and modify these values, as necessary!

   a. Select **Display**.
   b. Enter **8.0** for **Y**.
   c. Enter **24.0** for **X**.
   d. Enter **1.26** for **Z**.
   e. In Stock Origin, enter **-2.0** for **X** and **-0.25 for Y**.

f. Set the stock to the bottom left origin.



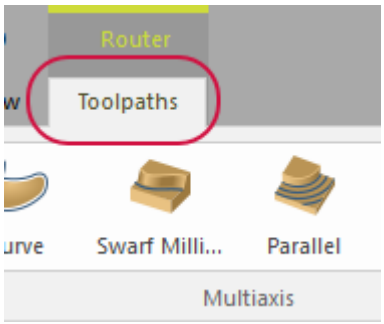
7. Click **OK** in the **Machine Group Properties** dialog box.
8. Save your file.



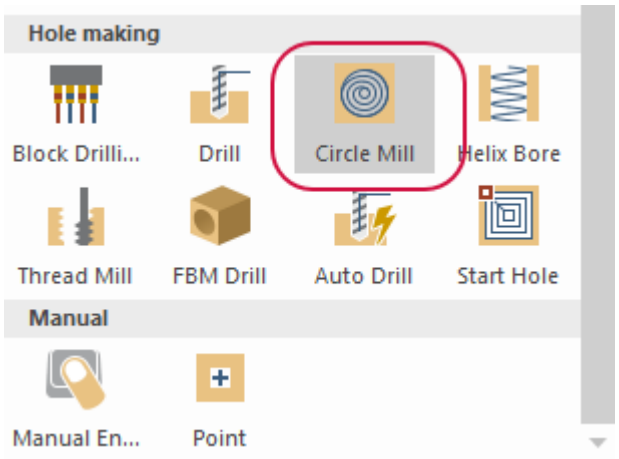
Exercise 2: Creating the Circle Mill Toolpath

In this exercise, you will create the Circle Mill toolpath that will machine the two index holes created in the previous chapter.

1. Your file Amplifier-XXX which you saved from the previous exercise should be open in Mastercam.
2. Select the **Router Toolpaths** contextual tab.

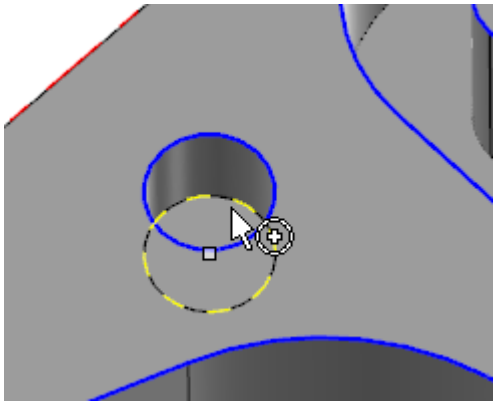


3. Select **Circle Mill** from the **2D** gallery drop-down.



The **Toolpath Hole Definition** function panel displays.

4. Select the bottom arc of the two index holes.

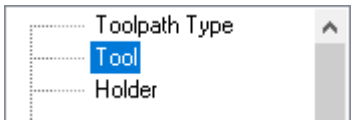


You may need to zoom in and rotate the part to select the correct geometry.

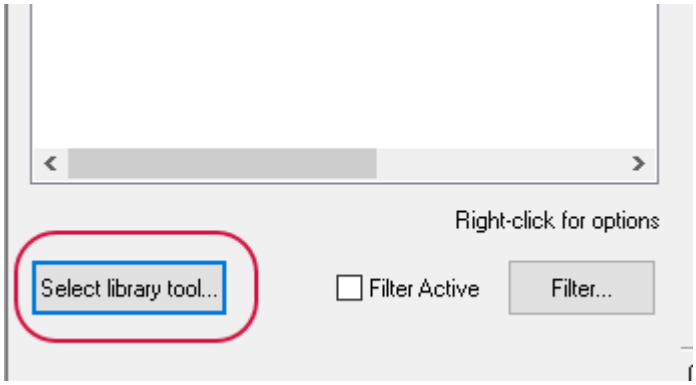
5. Click **OK** in the **Toolpath Hole Definition** function panel.

The **2D Toolpaths - Circle Mill** dialog box displays.

6. Select the **Tool** page.

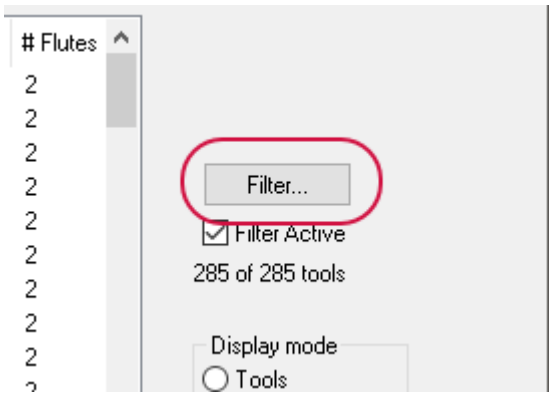


7. Click **Select library tool**.



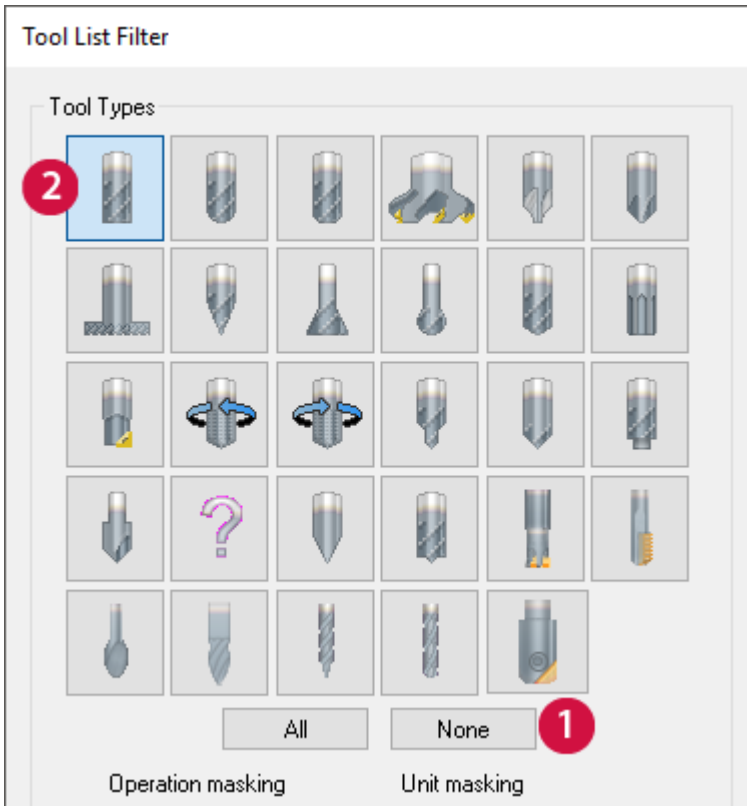
The **Tool Selection** dialog box displays.

8. Select **Filter**.

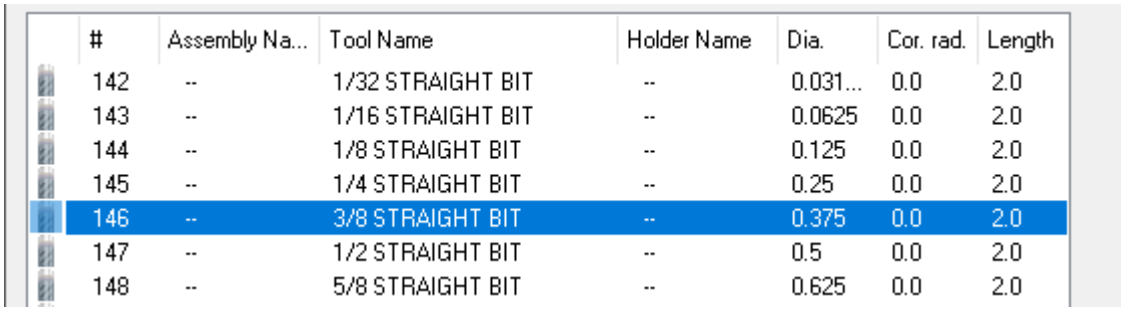


The **Tool List Filter** dialog box displays.

9. Select **None** to remove all tool filters, and then select **Endmill1 Flat**.

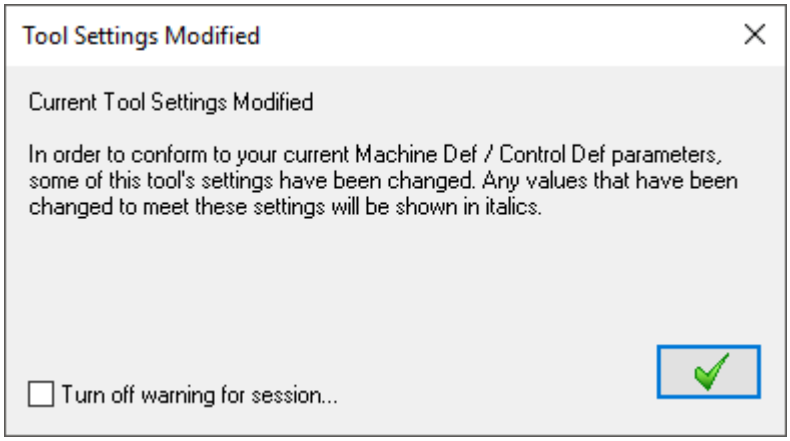


10. Click **OK** in the **Tool List Filter** dialog box.
11. In the **Tool Selection** dialog box, select **3/8 STRAIGHT BIT** and click **OK** to return to the **Tool** page.

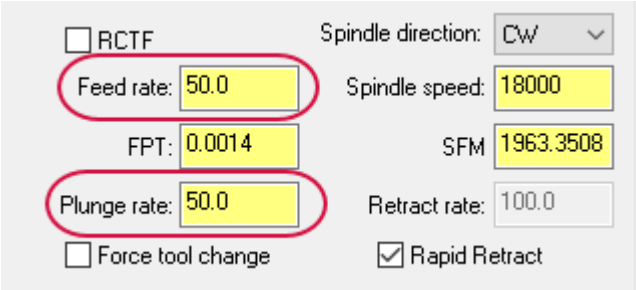


#	Assembly Na...	Tool Name	Holder Name	Dia.	Cor. rad.	Length
142	--	1/32 STRAIGHT BIT	--	0.031...	0.0	2.0
143	--	1/16 STRAIGHT BIT	--	0.0625	0.0	2.0
144	--	1/8 STRAIGHT BIT	--	0.125	0.0	2.0
145	--	1/4 STRAIGHT BIT	--	0.25	0.0	2.0
146	--	3/8 STRAIGHT BIT	--	0.375	0.0	2.0
147	--	1/2 STRAIGHT BIT	--	0.5	0.0	2.0
148	--	5/8 STRAIGHT BIT	--	0.625	0.0	2.0

12. You will receive a tool settings modified warning. This message informs you that the tool values have been modified to fit your current machine and control definitions. Click **OK** to accept the changes.

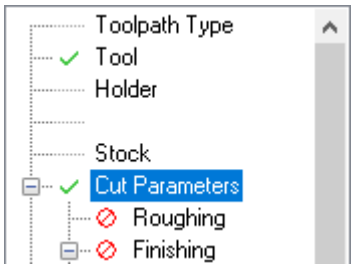


13. In the **Tool** page, enter **50.0** for **Feed rate** and **Plunge rate**.



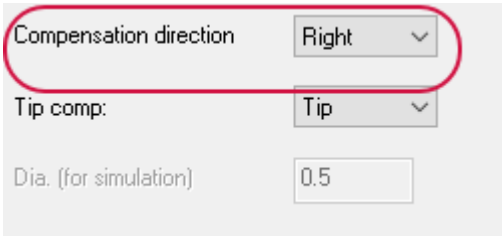
> **CAUTION**
> These values may vary depending on your machine and material.

14. Select the **Cut Parameters** page.

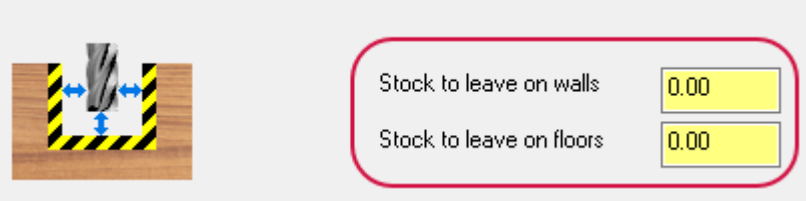


15. Set the following parameters:

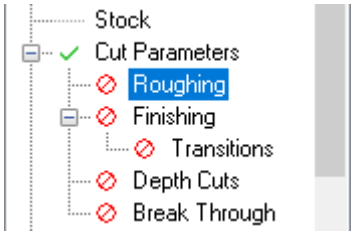
    a. Set **Compensation direction** to **Right**.



b. Enter **0.00** for **Stock to leave on walls** and **Stock to leave on floors**.



16. Select the **Roughing** page.



17. Set the following parameters:

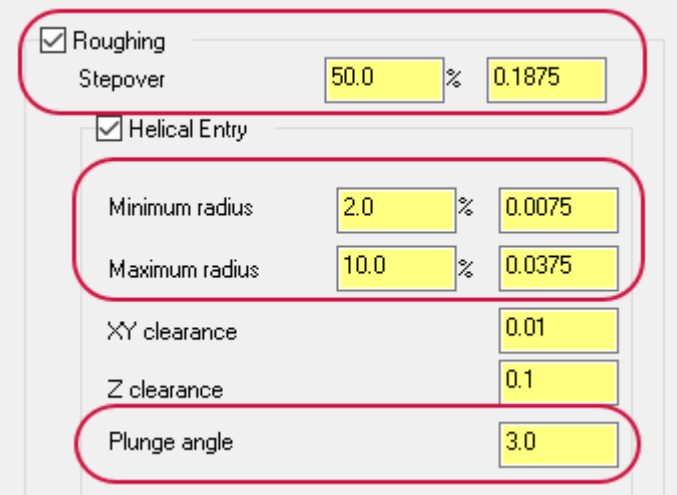
    a. Select **Roughing**.

    b. Enter **50.0**% for **Stepover**.

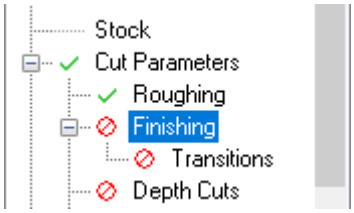
    c. Enter **2.0**% for **Minimum radius**.

    d. Enter **10.0**% for **Maximum radius**.

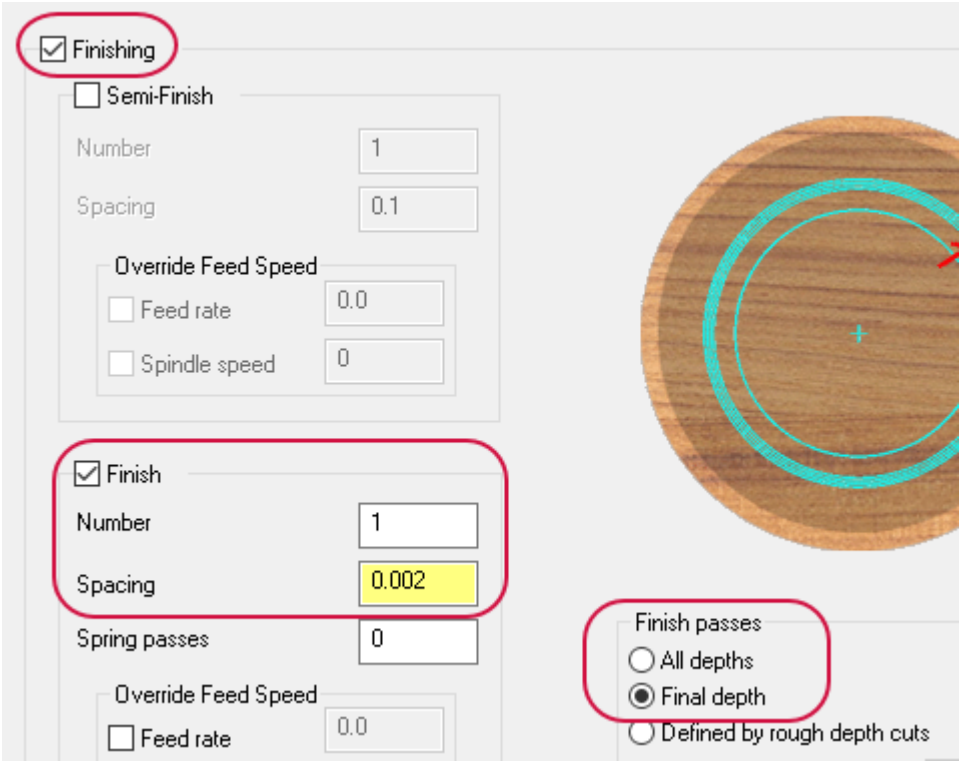
    e. Enter **3.0** for **Plunge angle**.



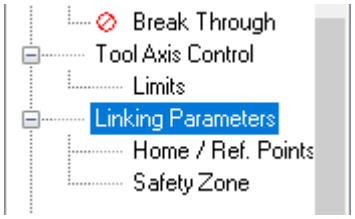
18. Select the **Finishing** page.



19. Set the following parameters:
    a. Select **Finishing**.
    b. Ensure that **Finish** is selected.
    c. Ensure that **1.0** is entered for **Number**.
    d. Enter **0.002** for **Spacing**.
    e. Set **Finish passes** to **Final depth**.

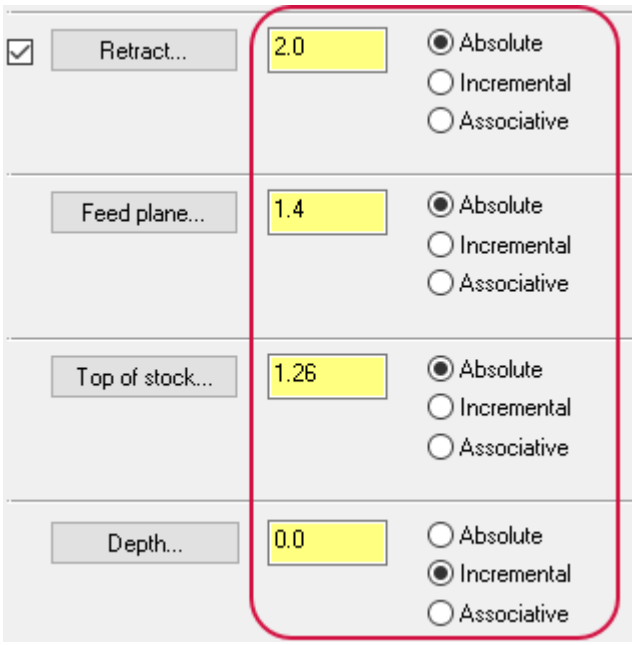


20. Select the **Linking Parameters** page.

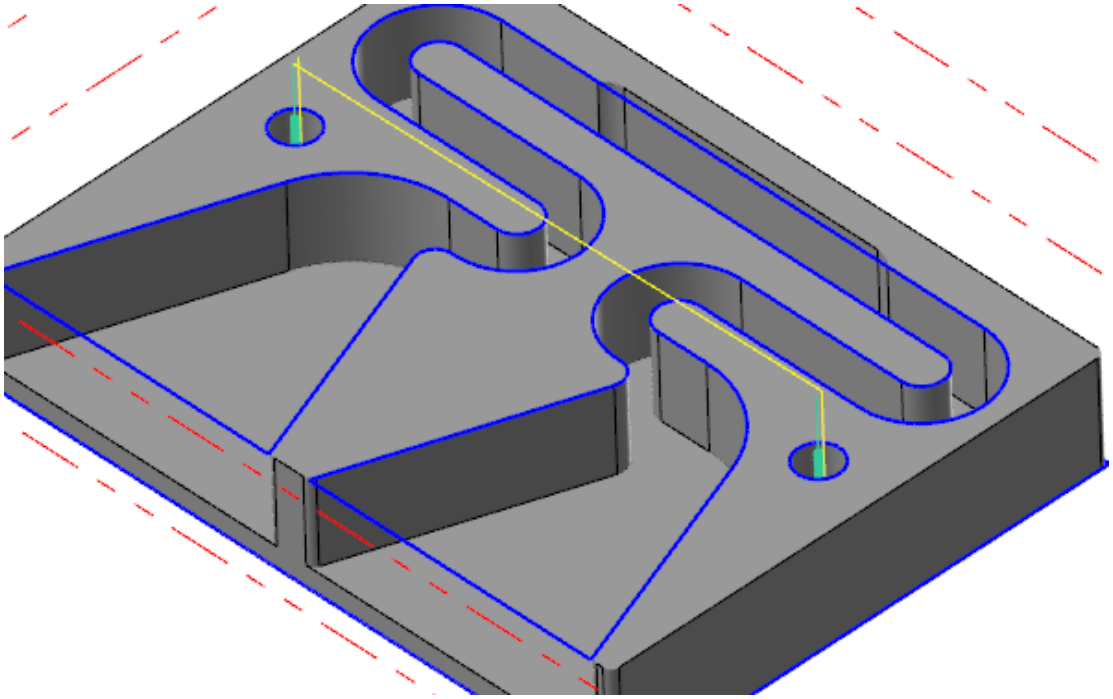


21. Set the following parameters:
    a. Set **Retract**, **Feed plane**, and **Top of stock** to **Absolute**.
    b. Enter **2.0** for **Retract**.
    c. Enter **1.4** for **Feed plane**.
    d. Enter **1.26** for **Top of stock**.

e. Ensure that **Depth** is set to **Incremental** and has a value of **0.0**.



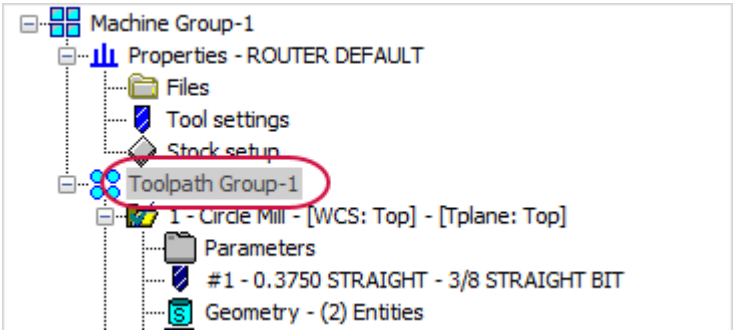
22. Click **OK** in the **2D Toolpaths - Circle Mill** dialog box.
23. Save your file.



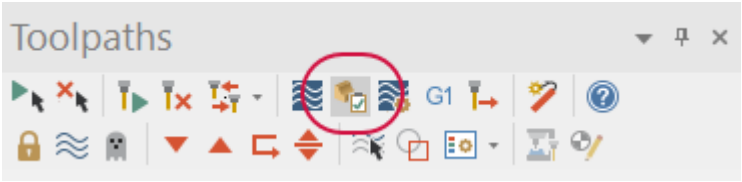
Exercise 3: Verifying the Circle Mill Toolpath

In this exercise, you use Mastercam Simulator to view and verify the Circle Mill toolpath.

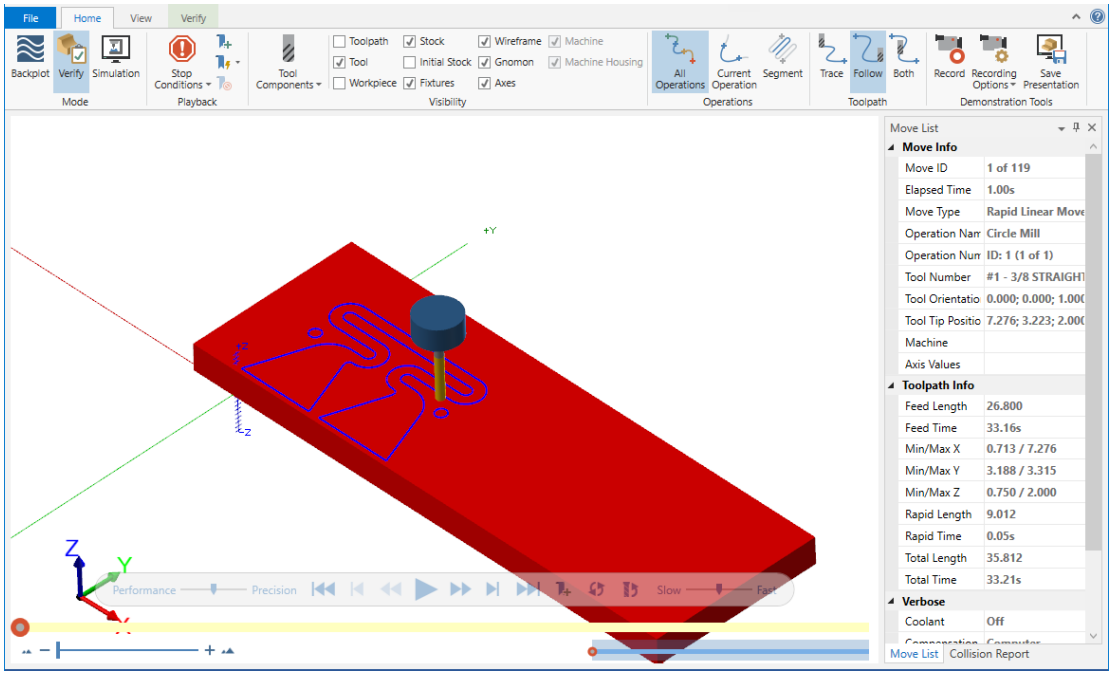
1. Your file Amplifier-XXX which you saved from the previous exercise should be open in Mastercam.
2. In the Toolpaths Manager, select **Toolpath Group-1**.



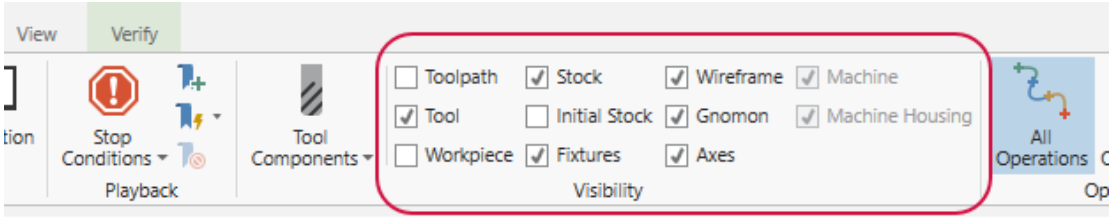
3. Select **Verify selected operations** from the Toolpaths Manager toolbar.



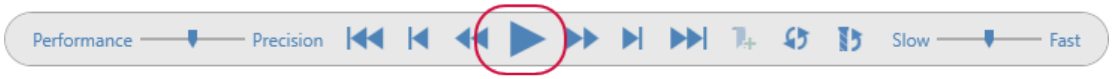
Mastercam Simulator displays.



4. Select the **Visibility** checkboxes on the **Home** tab to experiment with what displays in the graphics window.

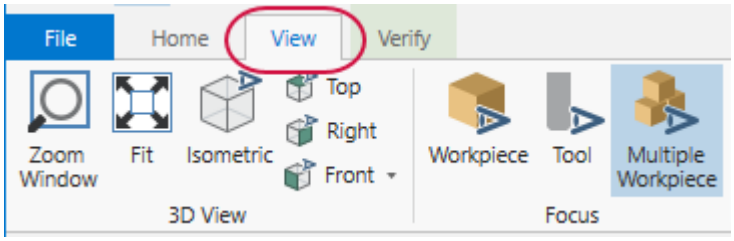


5. Use the playback bar to run the verification.

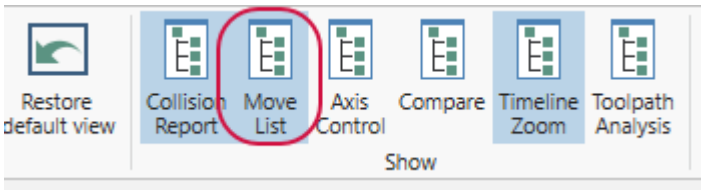


The verification shows the milling of the two index holes.

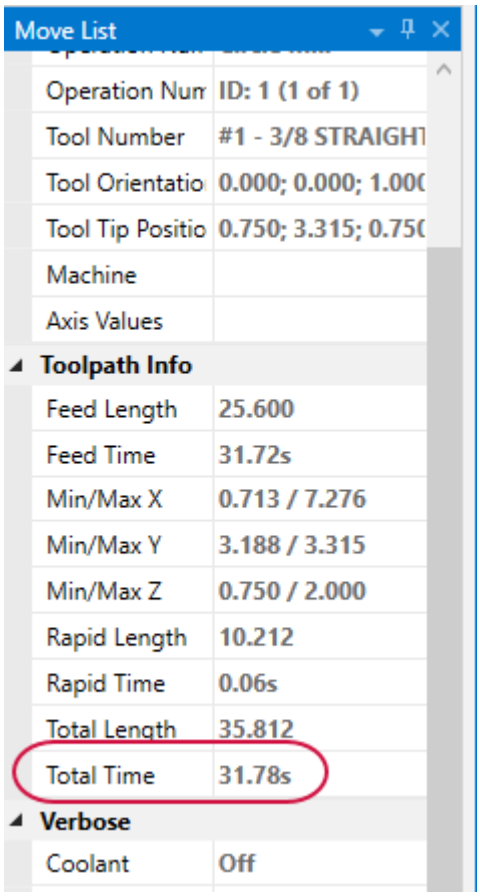
6. Run the simulation a few times and experiment with the **Visibility** options to see the results.
7. Select the **View** tab.



8. Ensure that the **Move List** button is highlighted. If not, click the button to enable the Move List pane.



9. Review the Move List pane to display information about the section of the current toolpath in verification, such as **Total Time**.



The total machining time of the Circle Mill toolpath is short.

> **NOTE**
> Your **Total Time** may vary.

10. Exit Mastercam Simulator once you have finished viewing the verification.
11. Save your file.

SHOW US WHAT YOU LEARNED!

Can you answer these questions?

1. The Stock origin can be located at a different location other than the machine origin.
    a. True
    b. False
2. You must enter the size of the stock manually.
    a. True
    b. False
3. You can display the stock in the graphics window to view the orientation and the location of the part relative to stock.
    a. True
    b. False
4. Verify gives you the chance to view tool motion with material removal.
    a. True
    b. False
5. In Mastercam Simulator, you can limit the objects you see on-screen, such as toolpaths, tools, stock, or fixtures.
    a. True
    b. False
6. Mastercam Simulator reports how long it will take to machine a toolpath.
    a. True
    b. False
7. When machining this part, why would you choose a Circle Mill toolpath instead of a Drill toolpath to machine the index holes?
8. What are the benefits of using simulation to review toolpaths?

CHAPTER 3

PROGRAMMING THE AMPLIFIER CHANNEL

In this chapter, you create the Dynamic Mill toolpath that will machine the amplifier channel created in Chapter 1. You then use Mastercam Simulator to verify the created toolpath.

Goals

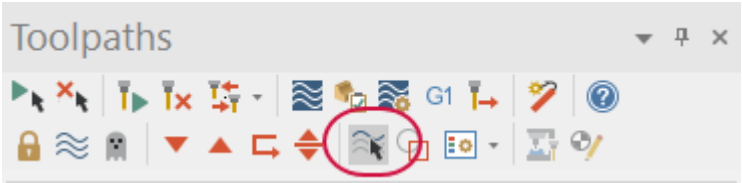
- Program a Dynamic Mill toolpath.
- Explore the Preview chain function.

> **CAUTION**
> When programming toolpaths, only change the values of the indicated parameters. Leave unmentioned values at their default values.

Exercise 1: Creating the Dynamic Mill Toolpath

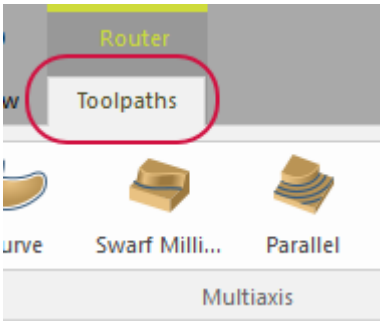
In this exercise, you will create the Dynamic Mill toolpath.

1. Your file Amplifier-XXX which you saved from the previous exercise should be open in Mastercam.
2. Click **Only display selected toolpaths** in the Toolpaths Manager.

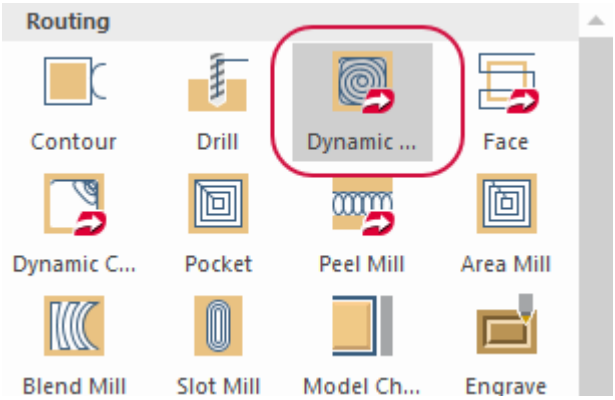


By selecting this button, only toolpaths that you have selected in the Toolpaths Manager will display in the graphics window.

3. Select the **Router Toolpaths** contextual tab.

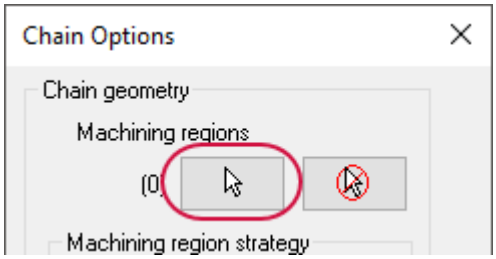


4. Select **Dynamic Mill** from the **2D** gallery drop-down.



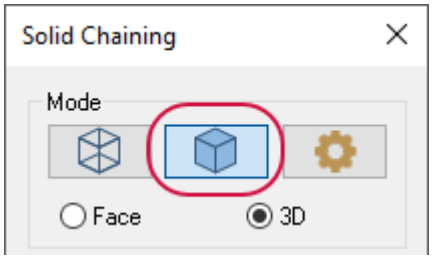
The **Chain Options** dialog box displays.

5. Click **Select machining chains** to return to the graphics window to select the machining regions.

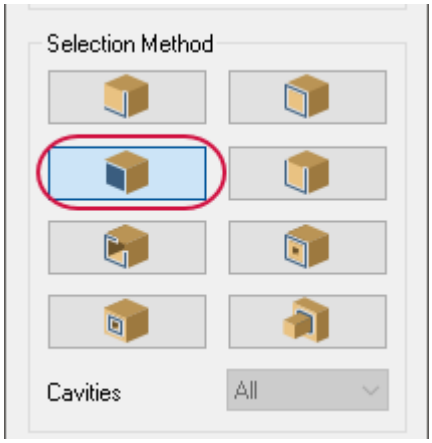


The **Chaining** dialog box displays.

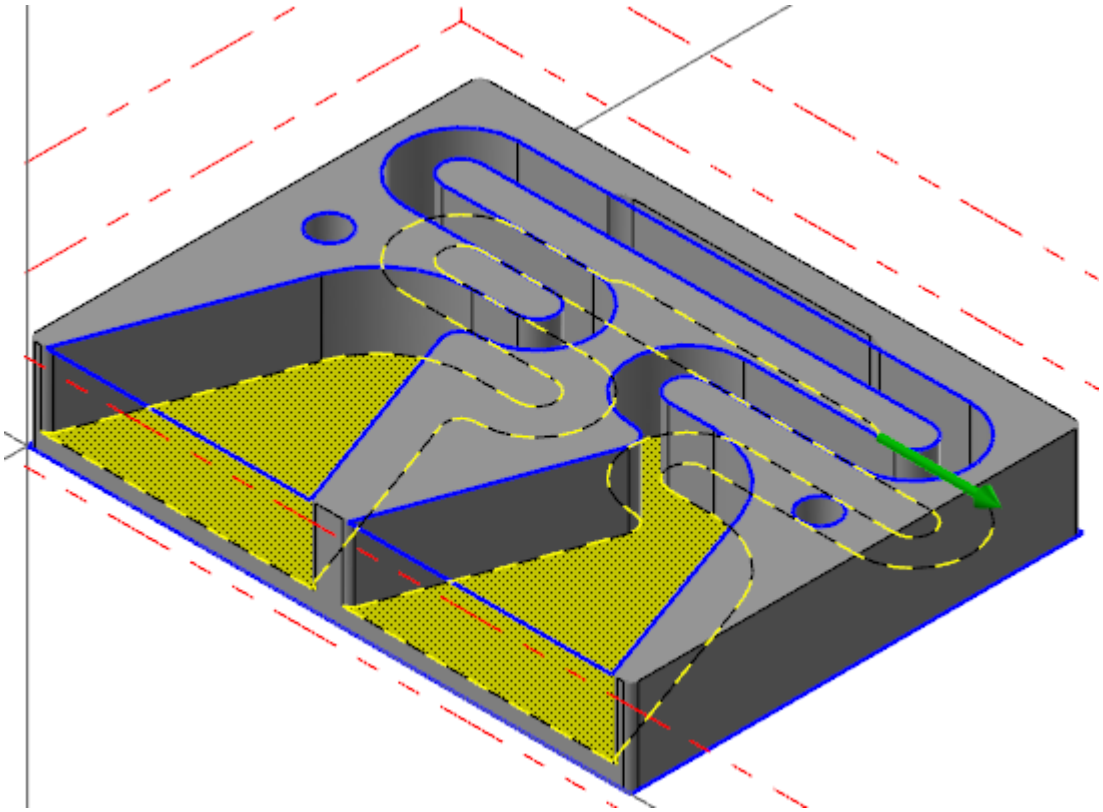
6. Select **Solids** to set the chaining mode to solids.



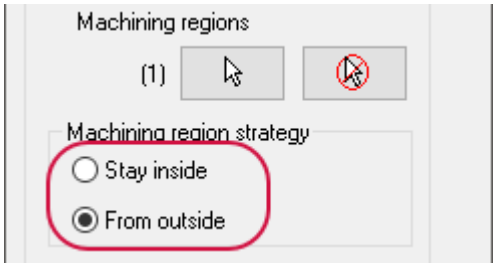
7. Ensure that the **Selection Method** is set only to **Face**. Click any selection method button to remove that selection filter.



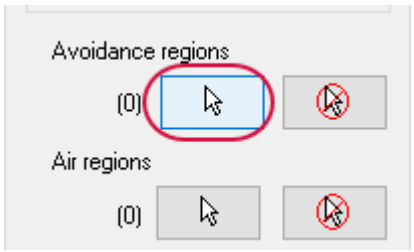
8. Select the bottom solid face of the channel as shown below.



9. Click **OK** in the **Chaining** dialog box to return to the **Chain Options** dialog box.
10. In the **Chain Options** dialog box, set **Machining region strategy** to **From outside**.

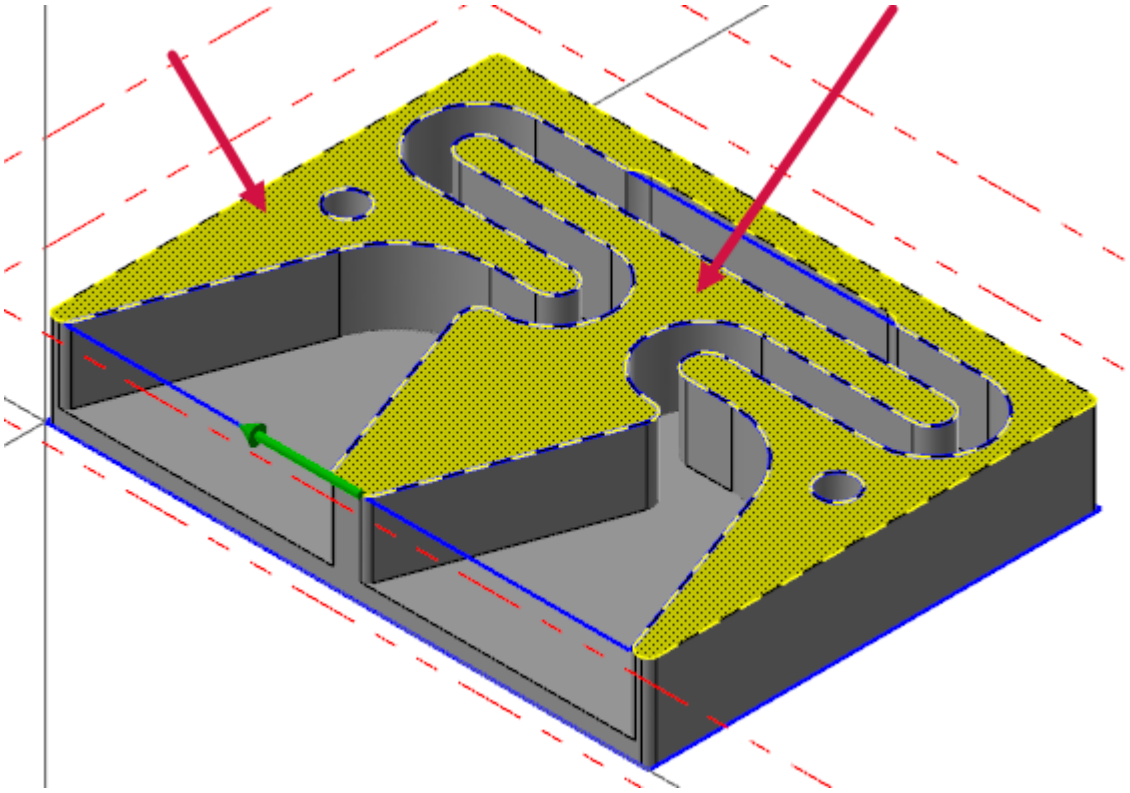


11. Click **Select avoidance chains** to return to the graphics window to select the avoidance regions.



The **Chaining** dialog box displays.

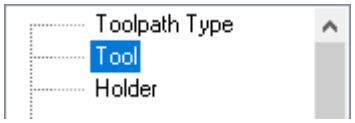
12. Select the two top solid faces as shown below.



13. Click **OK** in the **Chaining** dialog box and then click **OK** again in the **Chain Options** dialog box.

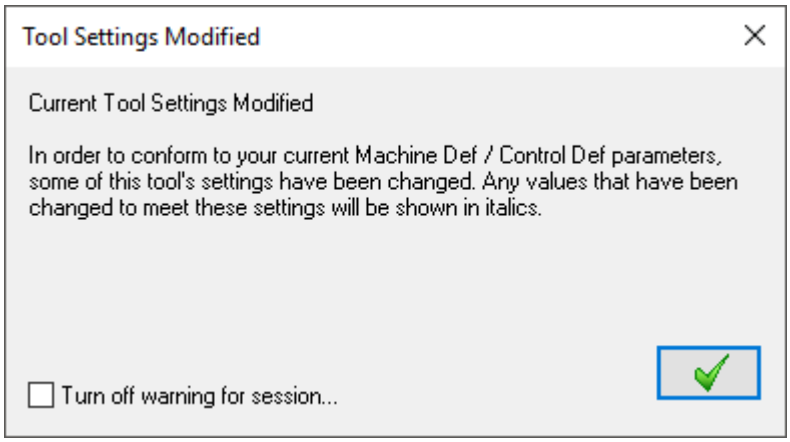
    The **2D High Speed Toolpaths - Dynamic Mill** dialog box displays.

14. Select the **Tool** page.

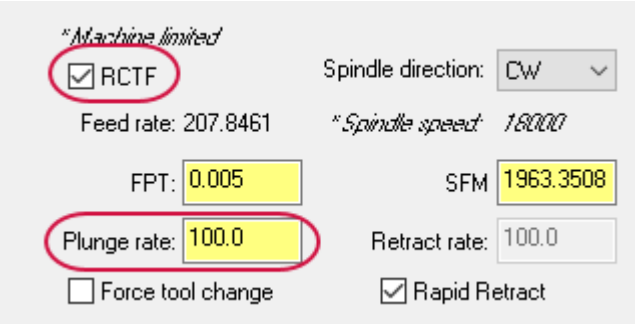


15. Select the **3/8 STRAIGHT BIT** tool from the tool list.

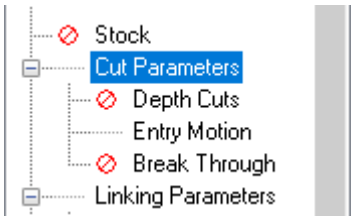
16. You will receive a tool settings modified warning. This message informs you that the tool values have been modified to fit your current machine and control definitions. Click **OK** to accept the changes.



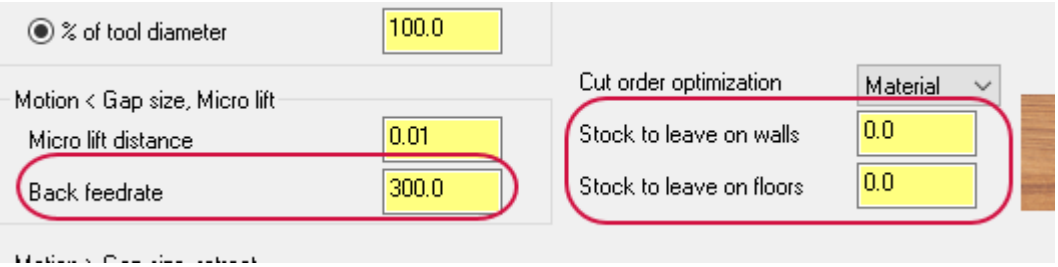
17. Select the following parameters on the **Tool** page:
    a. Select **RCTF**.
    b. Enter **100.00** for **Plunge rate**.



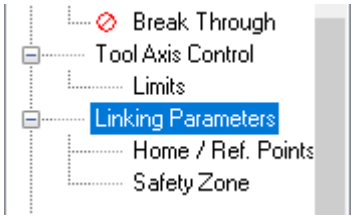
18. Select the **Cut Parameters** page.



19. Set the following parameters:
    a. Enter **300.00** for **Back feedrate**.
    b. Enter **0.0** for **Stock to leave on walls** and **Stock to leave on floors**.

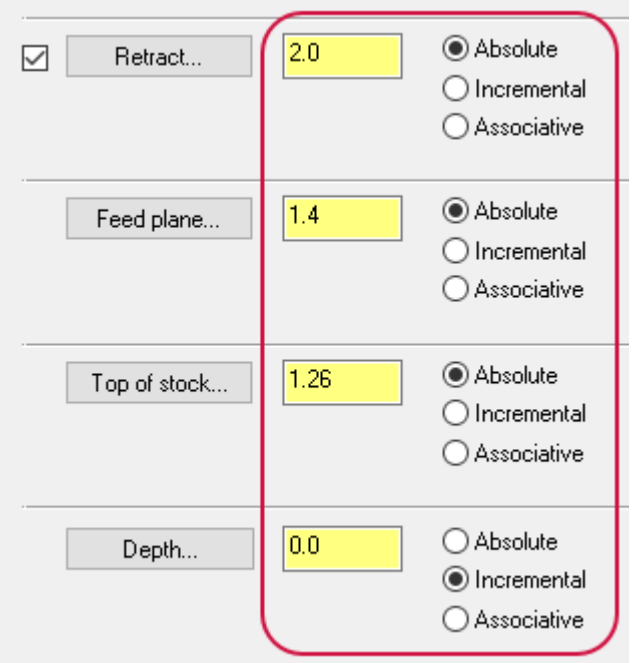


20. Select the **Linking Parameters** page.

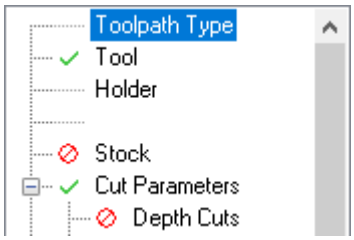


21. Set the following parameters:
    a. Set **Retract**, **Feed plane**, and **Top of stock** to **Absolute**.
    b. Enter **2.0** for **Retract**.
    c. Enter **1.4** for **Feed plane**.
    d. Enter **1.26** for **Top of stock**.

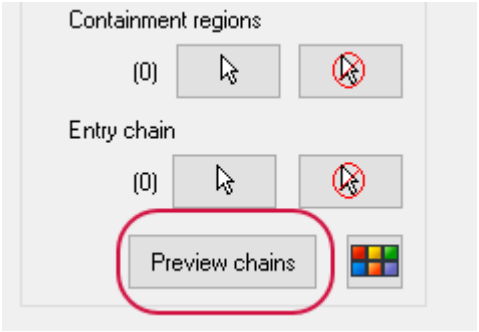
e. Ensure that **Depth** is set to **0.0** and **Incremental**.



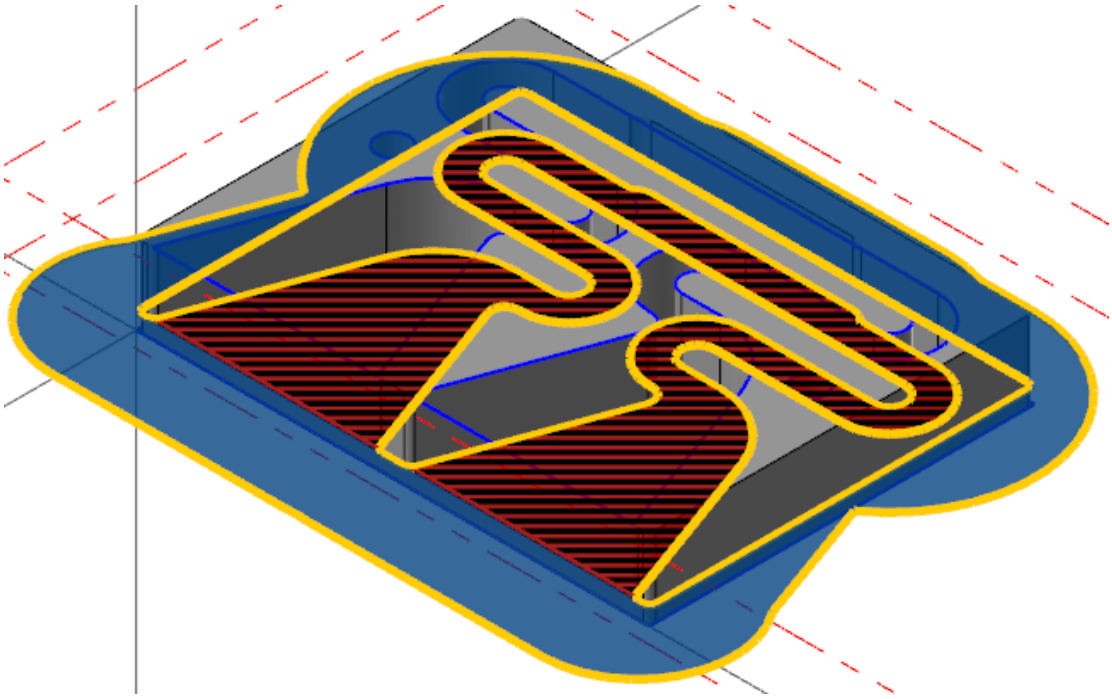
22. Select the **Toolpath Type** page.



23. Select the **Preview chains** button.

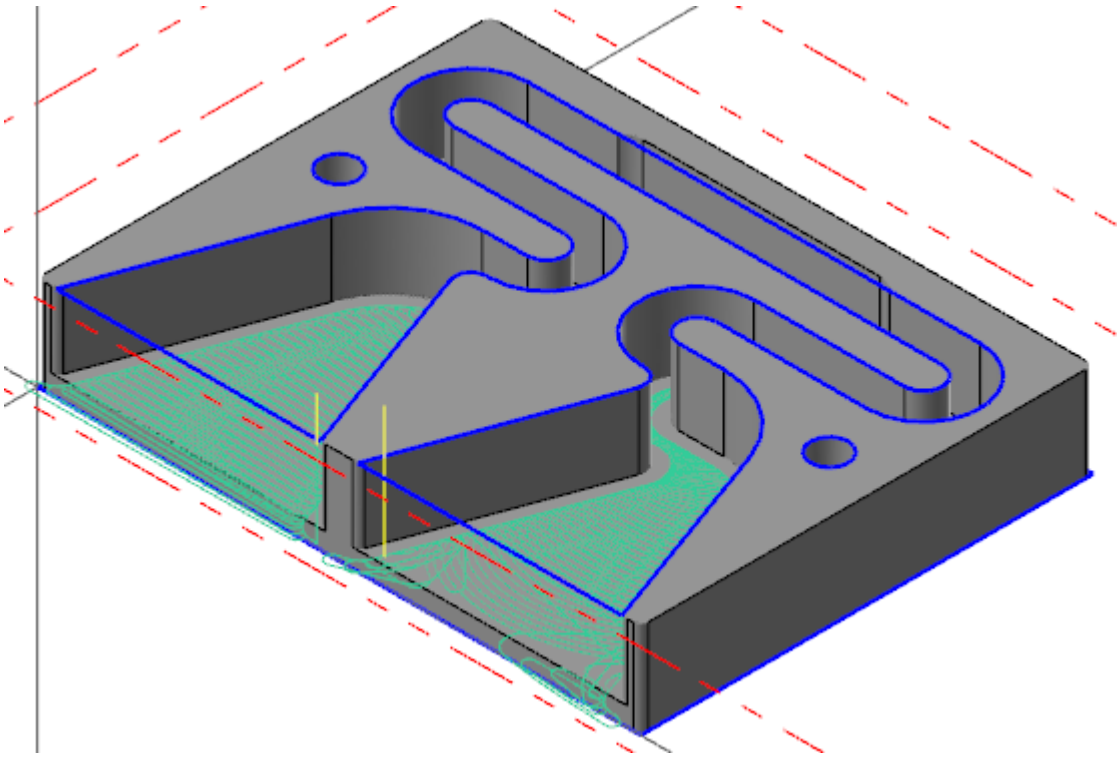


24. Your part displays as shown below in the graphics window.



This allows you to ensure that you have selected the correct chain geometry and you are machining the correct areas.

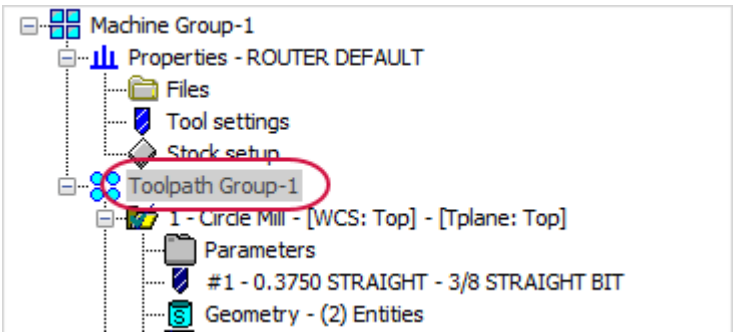
25. Click **OK** in the **2D High Speed Toolpaths - Dynamic Mill** dialog box.
26. Rotate the part to view the toolpath. You can also press [**Alt+S**] to display the part as wireframe.
27. Save your file.



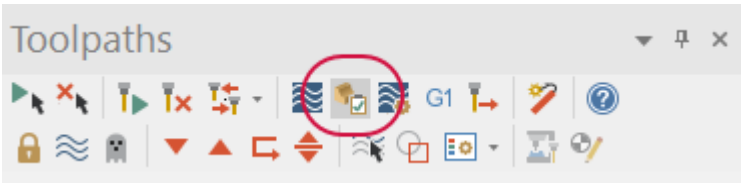
Exercise 2: Verifying the Dynamic Mill and Circle Mill Toolpaths

In this exercise, you use Mastercam Simulator to view and verify the Circle Mill and Dynamic Mill toolpaths.

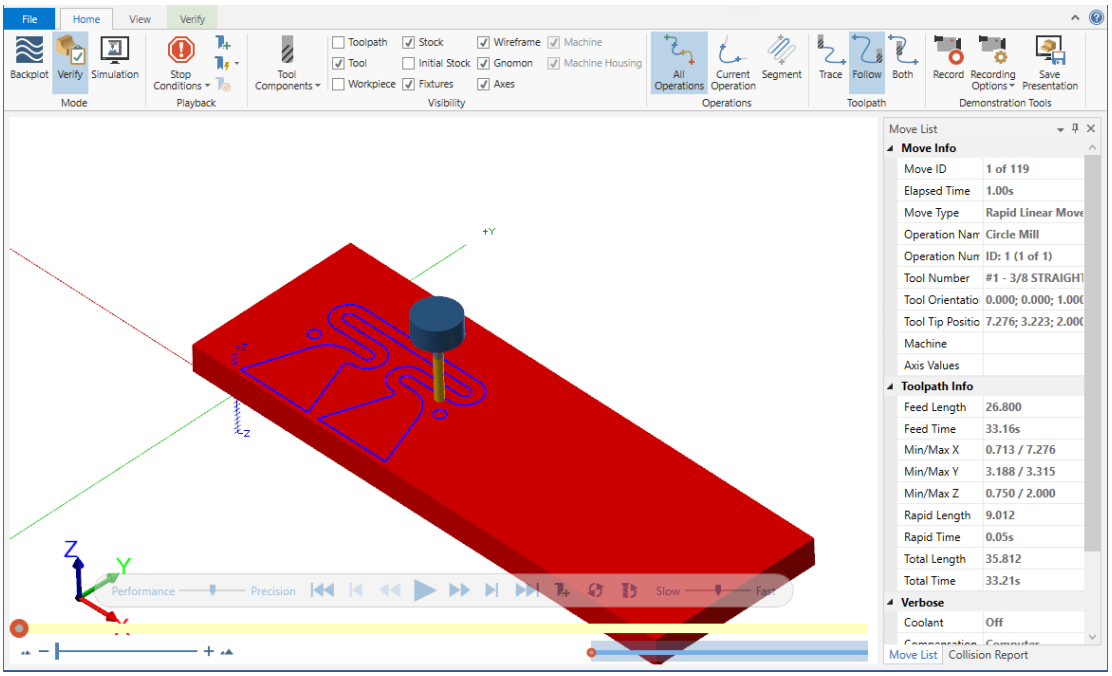
1. Your file Amplifier-XXX which you saved from the previous exercise should be open in Mastercam.
2. In the Toolpaths Manager, select **Toolpath Group-1**.



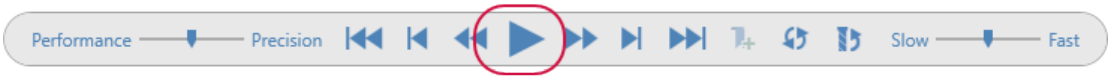
3. Select **Verify selected operations** from the Toolpaths Manager toolbar.



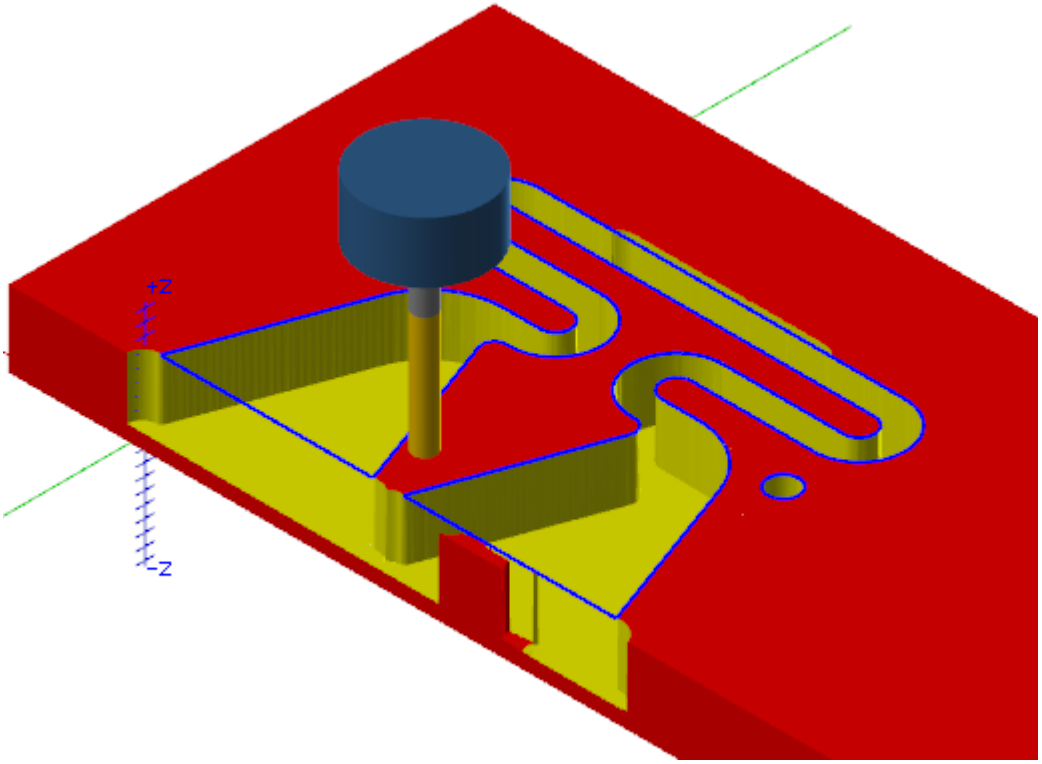
Mastercam Simulator displays.



4. Use the playback bar to run the verification.



The verification shows the machining of the channel geometry and the drilling of the index holes.



NOTE

You may see extra material at the entrance of one channel of the amplifier geometry. This is because of the extra stock we added to the Y axis when we created the stock origin. entered in the **Stock Origin** of the **Stock Setup** tab. We will address this extra material in a later chapter.

Rotate the part or select **Top view** to see that the extra material is outside the machining region.

5. Exit Mastercam Simulator once you have finished viewing the verification.
6. Save your file.

SHOW US WHAT YOU LEARNED!

Can you answer these questions?

1. You must chain toolpath geometry for Dynamic Mill one entity at a time.
    a. True
    b. False
2. What are the differences between the two machining strategies, **From outside** and **Stay inside**?
3. When creating a Dynamic Mill toolpath, which of the following **Chain geometry** types is *not* considered?
    a. Machining regions
    b. Check regions
    c. Containment regions
    d. Air regions
4. Absolute values for linking parameters (such as **Clearance** and **Retract**) are always measured from the origin (0,0,0) of the current WCS.
    a. True
    b. False
5. What are the benefits of using **Preview chains** in the *Dynamic Mill* toolpath?

CHAPTER 4

PROGRAMMING THE CELL PHONE OPENING

In this chapter, you create Dynamic Mill and Contour toolpaths to machine the cell phone opening created in Chapter 1. You then use Classic Backplot to simulate the toolpath, as well as use Section Views to help view the simulation.

Goals

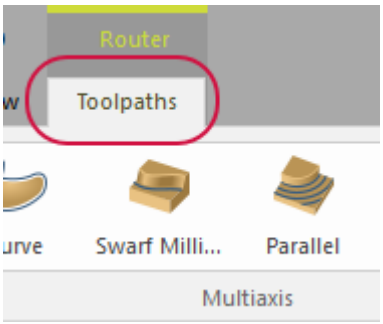
- Program a Dynamic Mill and 2D Contour toolpath.
- Explore Classic Backplot.
- Explore using section views to cut away the part to see interior geometry and features.

> **CAUTION**
> When programming toolpaths, only change the values of the indicated parameters. Leave unmentioned values at their default values.

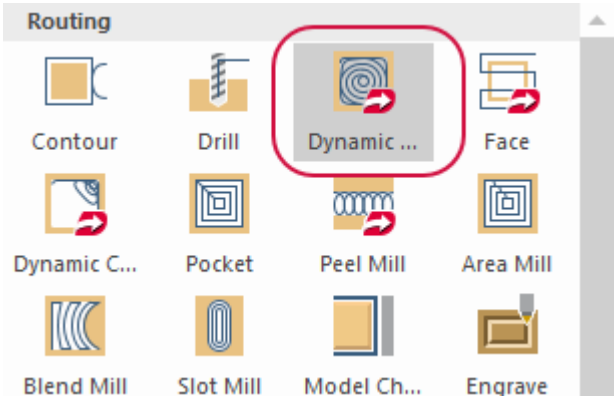
Exercise 1: Creating the Dynamic Mill Toolpath

In this exercise, you will create the Dynamic Mill toolpath.

1. Your file Amplifier-XXX which you saved from the previous exercise should be open in Mastercam.
2. Select the **Router Toolpaths** contextual tab.

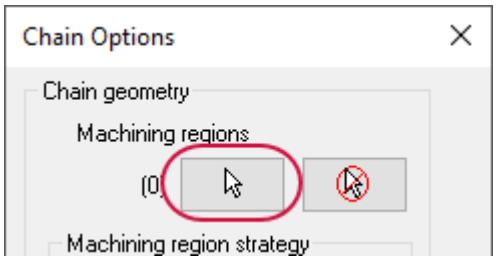


3. Select **Dynamic Mill** from the **2D** gallery drop-down.



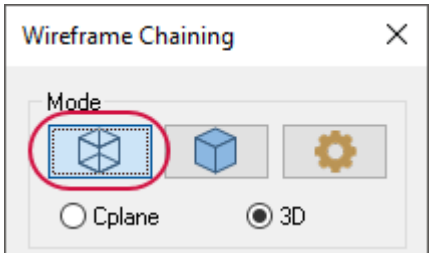
The **Chain Options** dialog box displays.

4. Click **Select machining chains** to return to the graphics window to select the machining regions.

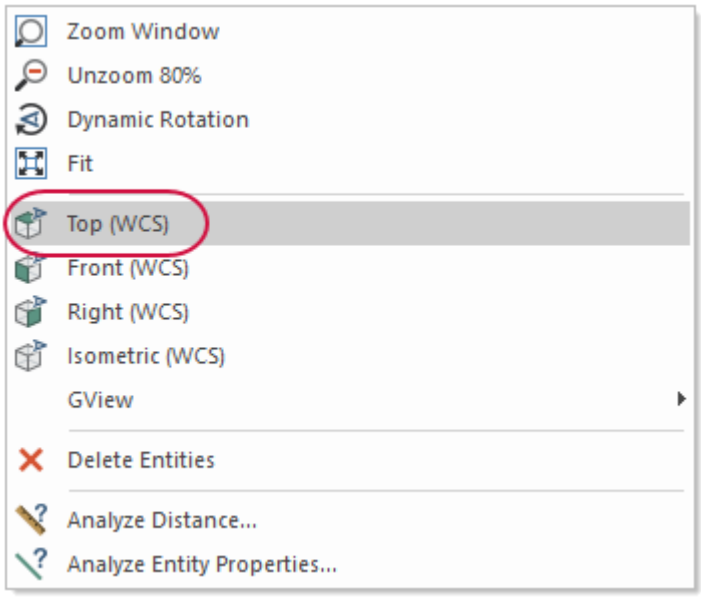


The **Chaining** dialog box displays.

5. Select **Wireframe** to set the chaining mode to wireframe.



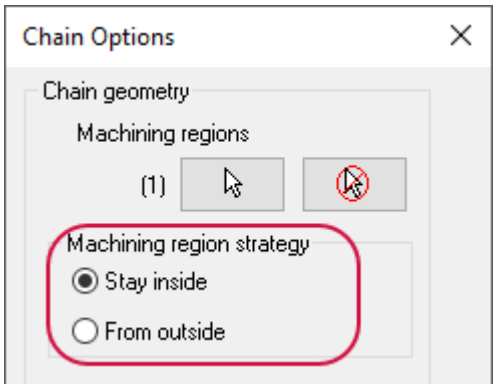
6. Right-click in the graphics window and select **Top (WCS)**.



7. Select the phone slot geometry as shown below.

8. Click **OK** in the **Chaining** dialog box to return to the **Chain Options** dialog box.

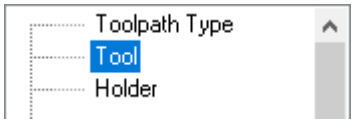
9. Set **Machining region strategy** to **Stay inside**.



10. Click **OK** in the **Chain Options** dialog box.

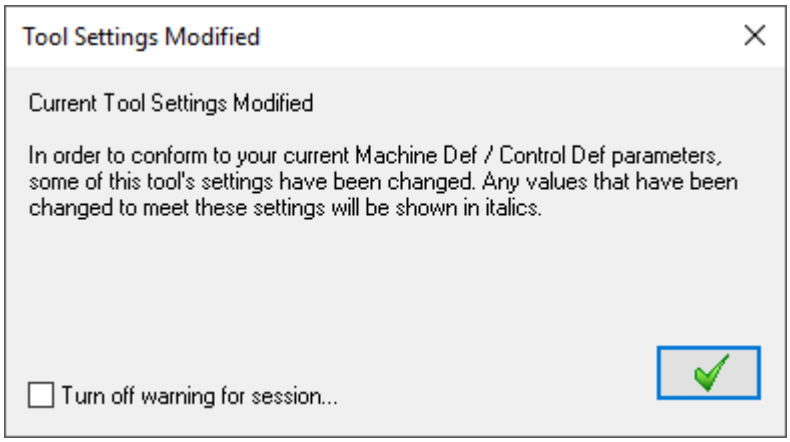
    The **2D High Speed Toolpath - Dynamic Mill** dialog box displays.

11. Select the **Tool** page.

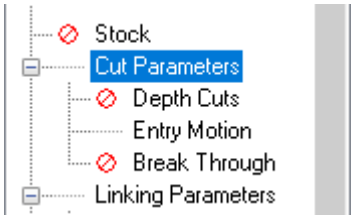


12. Select the **3/8 STRAIGHT BIT** tool from the tool list.

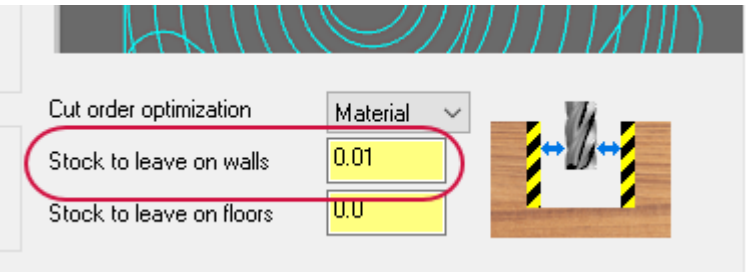
13. You will receive a tool settings modified warning. This message informs you that the tool values have been modified to fit your current machine and control definitions. Click **OK** to accept the changes.



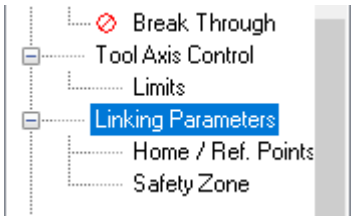
14. Select the **Cut Parameters** page.



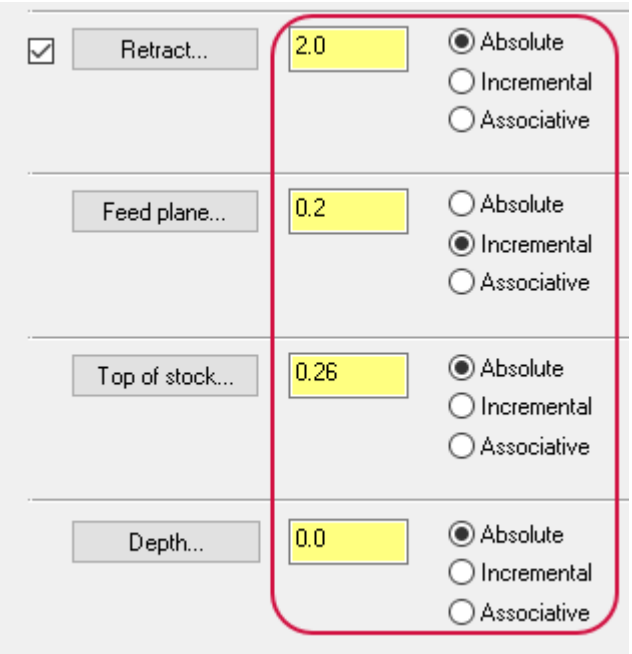
15. Enter **0.01** for **Stock to leave on walls**.



16. Select the **Linking Parameters** page.

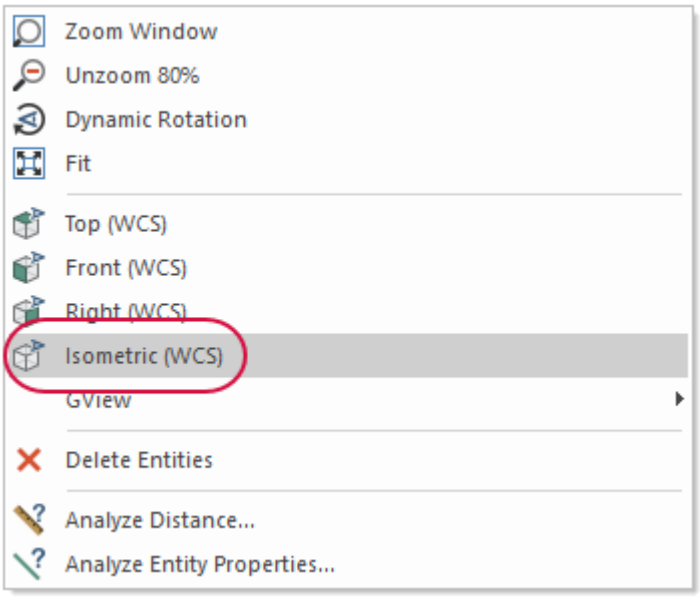


17. Enter the following parameters:
    a. Ensure that **Retract** is set to **2.0** and **Absolute**.
    b. Enter **0.2** for **Feed plane** and set to **Incremental**.
    c. Enter **0.26** for **Top of stock** and set to **Absolute**.
    d. Ensure that **Depth** is set to **0.0** and set to **Absolute**.

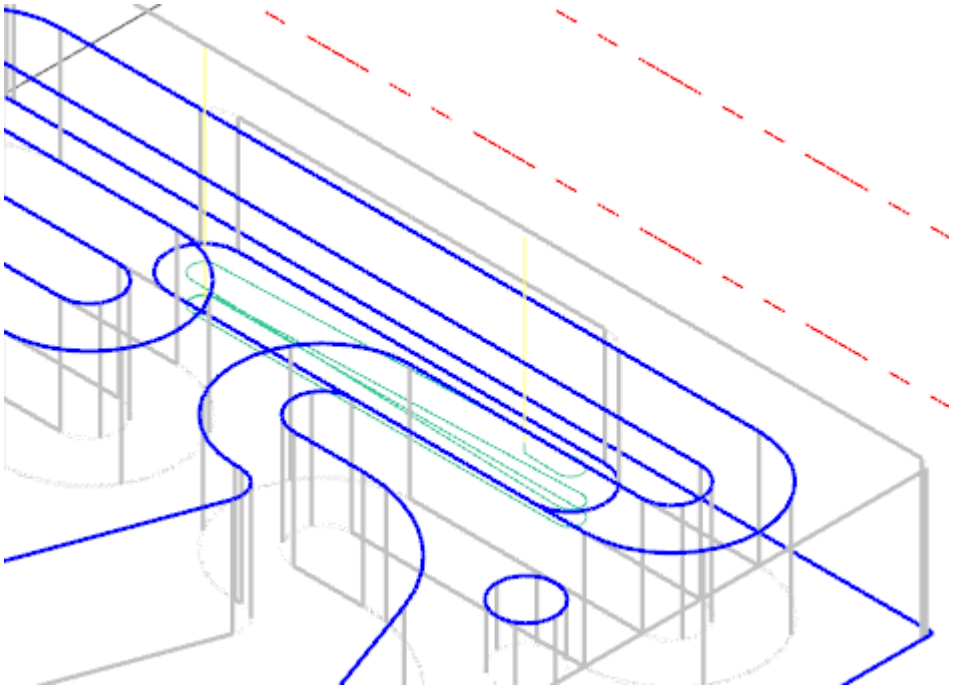


18. Click **OK** in the **2D High Speed Toolpath - Dynamic Mill** dialog box.

19. Right-click in the graphics window and select **Isometric (WCS)**.



20. Press [**Alt+S**] to view the part as wireframe.



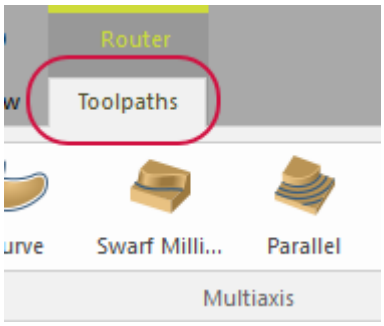
This allows you to view the toolpath you have just created.

21. After you have viewed your toolpath, press [**Alt+S**] again to shade the part.
22. Save your file.

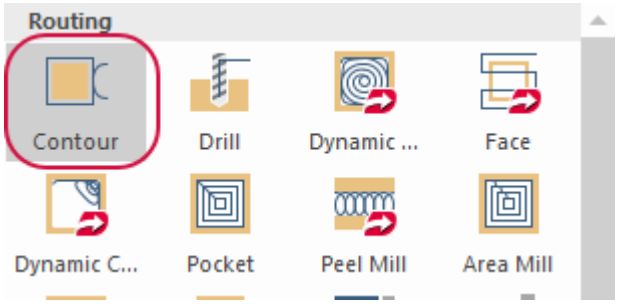
Exercise 2: Creating the Contour Toolpath

In this exercise, you will create a Contour (Ramp) toolpath.

1. Your file Amplifier-XXX which you saved from the previous exercise should be open in Mastercam.
2. Select the **Router Toolpaths** contextual tab.

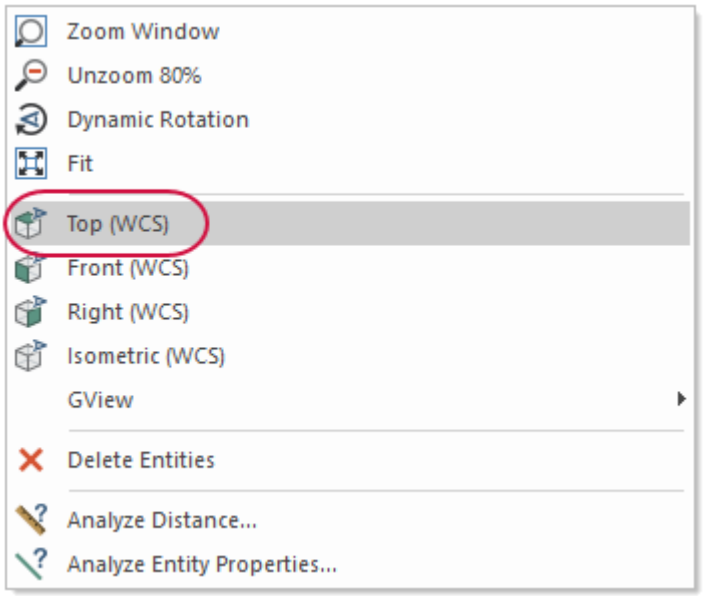


3. Select **Contour** from the **2D** gallery drop-down.

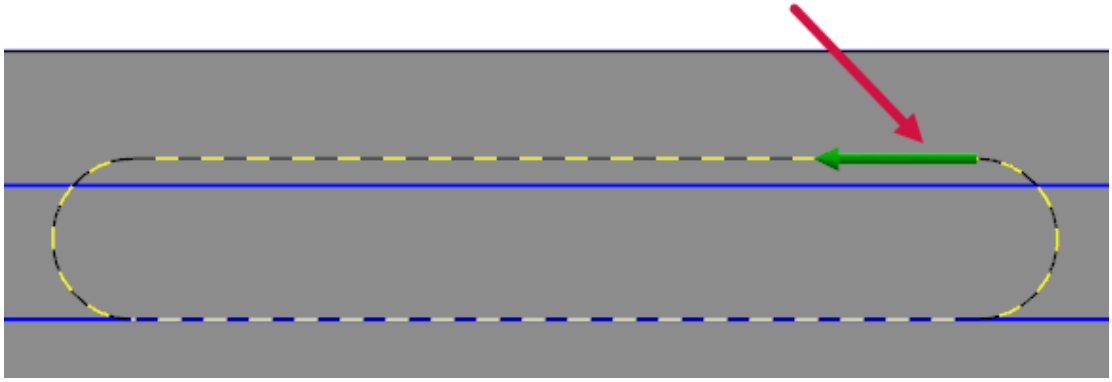


The **Chaining** dialog box displays.

4. Right-click in the graphics window and select **Top (WCS)**.



5. Select the phone slot geometry as shown below. It may be helpful to unshade the part.

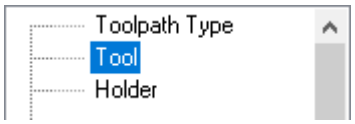


Ensure that your chain direction (the green arrow) and the start point matches the chain shown previously. If not, use the **Reverse** button on the **Chaining** dialog box to change the direction of the chain.

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

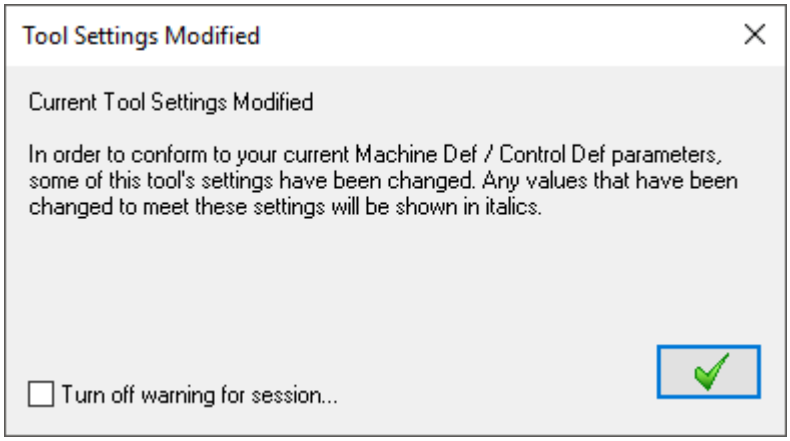
   The **2D Toolpaths - Contour** dialog box displays.

7. Select the **Tool** page.

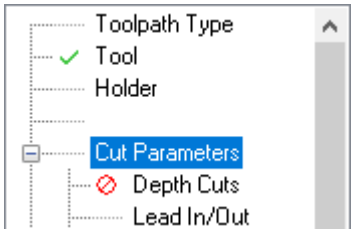


8. Select the **3/8 STRAIGHT BIT** tool from the tool list.

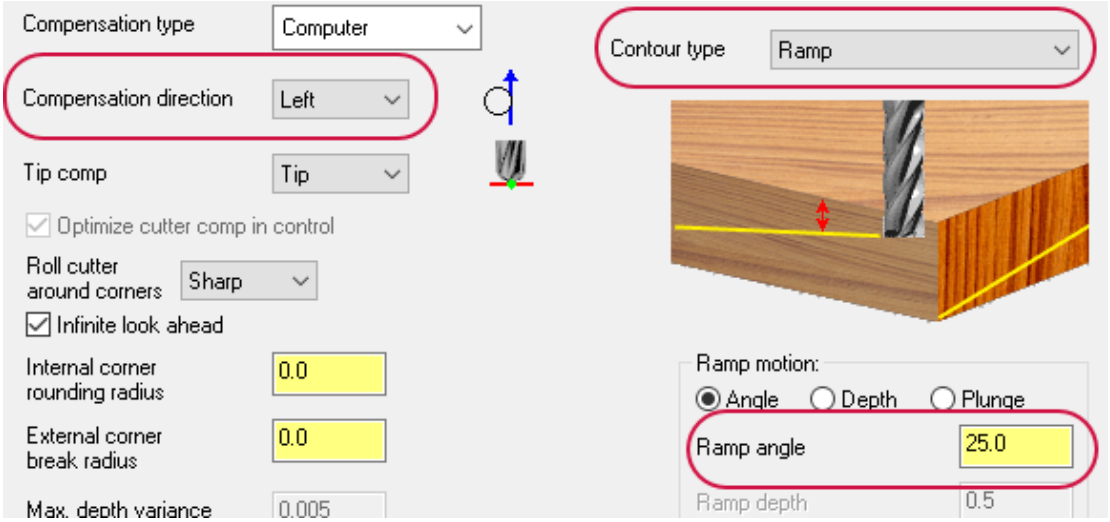
9. You will receive a tool settings modified warning. This message informs you that the tool values have been modified to fit your current machine and control definitions. Click **OK** to accept the changes.



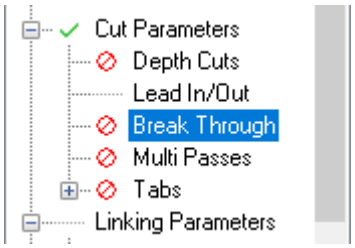
10. Select the **Cut Parameters** page.



11. Enter the following parameters:
    a. Ensure that **Compensation direction** is set to **Left**.
    b. Set **Contour type** to **Ramp**.
    c. Enter **25.0** for **Ramp angle**.

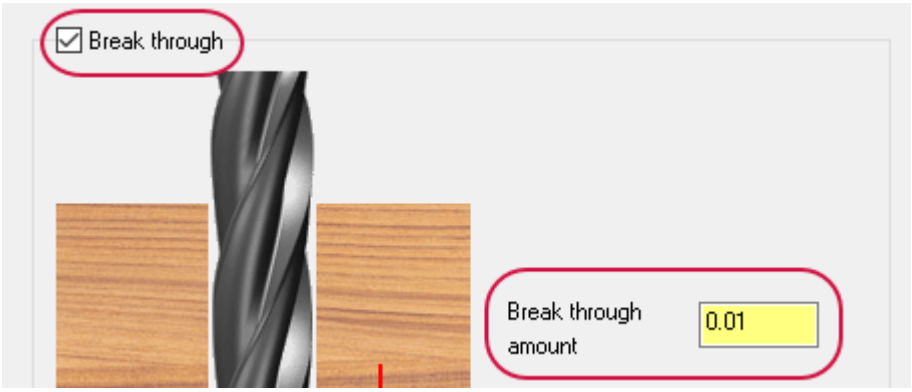


12. Select the **Break Through** page.

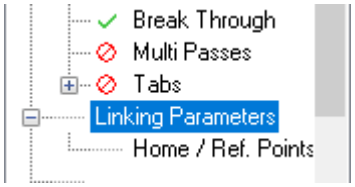


> **WARNING**
> Enter a break through amount only if your set-up includes a spoilboard. You risk damaging your equipment (tool damage, cutting into your table, etc.) if you create a break through without a spoilboard!

13. Enter the following parameters:
    a. Select **Break through**.
    b. Enter **0.01** for **Break through amount**.

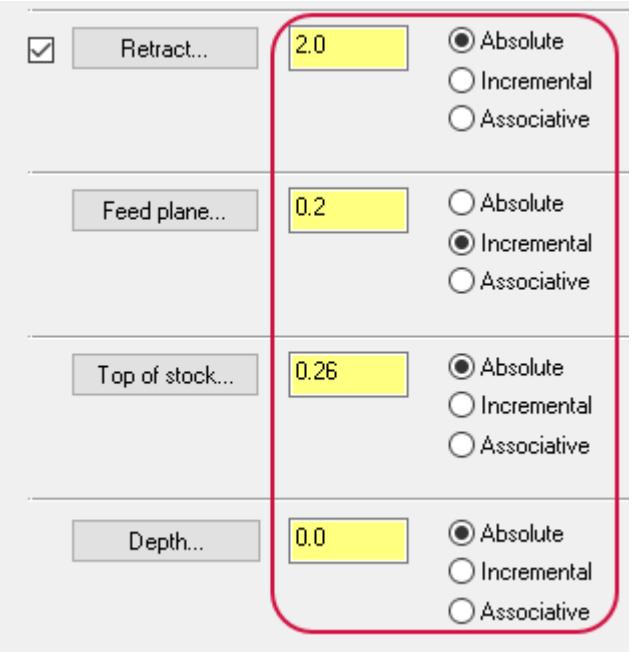


14. Select the **Linking Parameters** page.

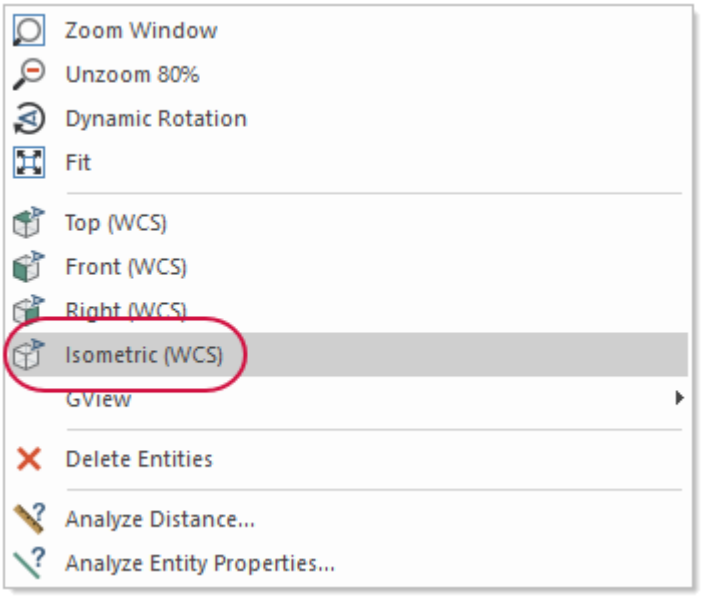


15. Enter the following parameters:
    a. Enter **2.0** for **Retract** and set to **Absolute**.
    b. Enter **0.2** for **Feed plane** and set to **Incremental**.
    c. Enter **0.26** for **Top of stock** and set to **Absolute**.

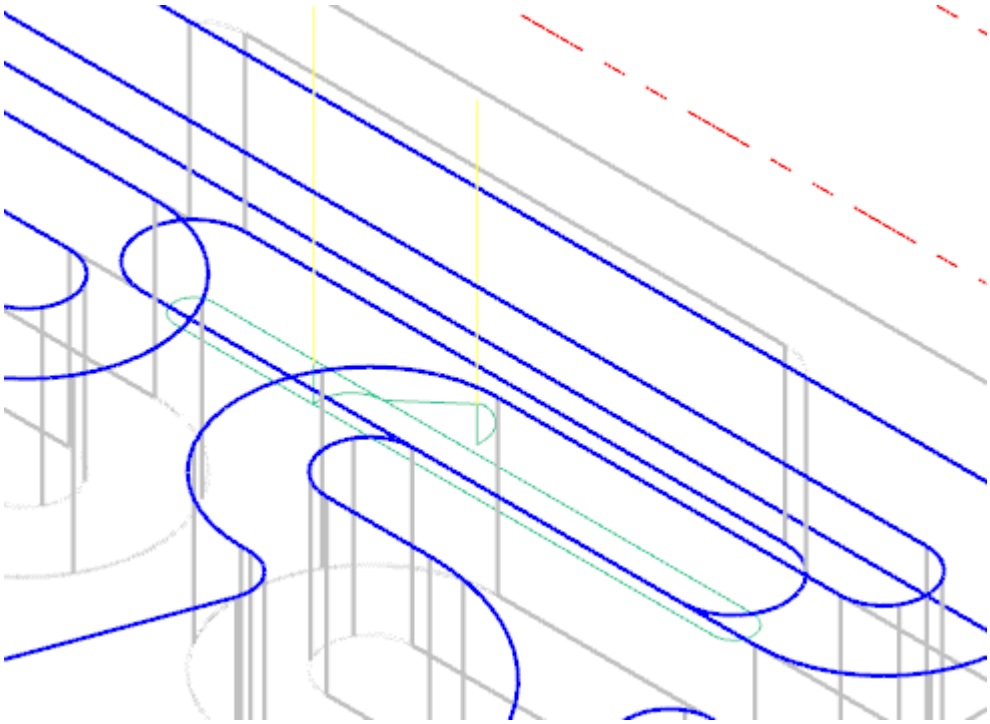
d. Ensure that **Depth** is set to **0.0** and **Absolute**.



16. Click **OK**.
17. Right-click in the graphics window and select **Isometric (WCS)**.



18. Press [**Alt+S**] to view the part as wireframe.

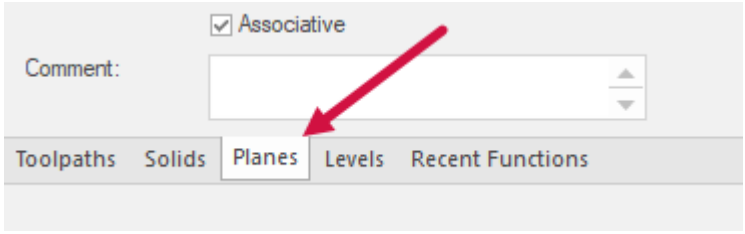


19. After you have viewed your toolpath, press [**Alt+S**] again to shade the part.
20. Save your file.

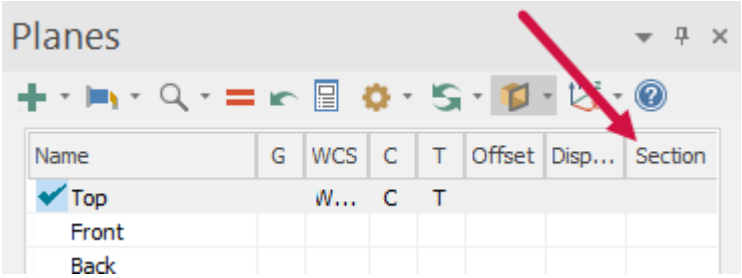
Exercise 3: Creating the Section View

In this exercise, you will create a section view that you use during simulation to help get a better view of the machining.

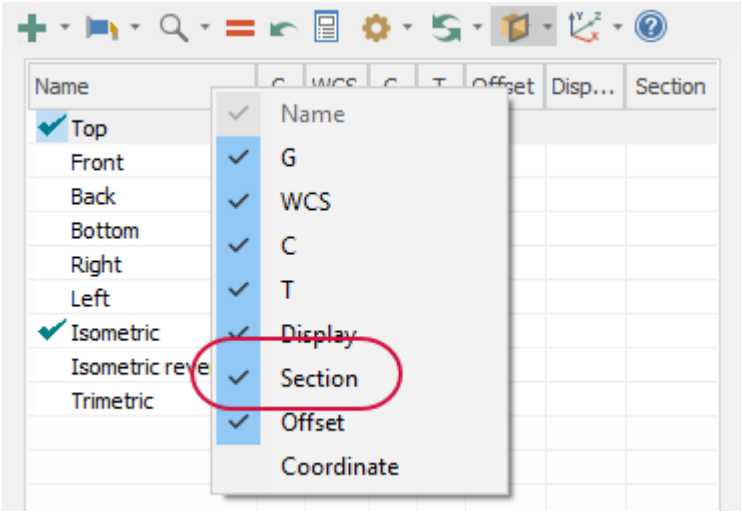
1. Your file Amplifier-XXX which you saved from the previous exercise should be open in Mastercam.
2. Select the **Planes** tab to bring the Planes Manager to the forefront.



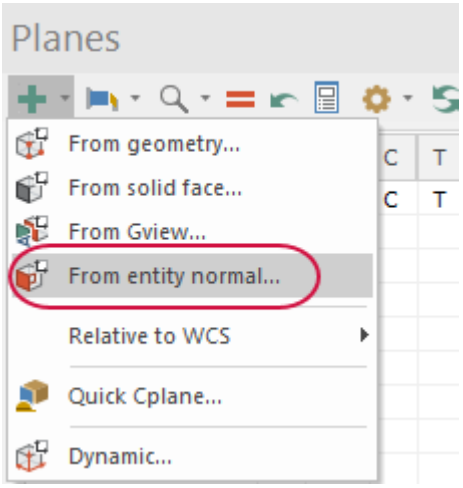
3. Ensure that the **Section** column is displayed.



If is it not, right-click on the columns and select **Section**.

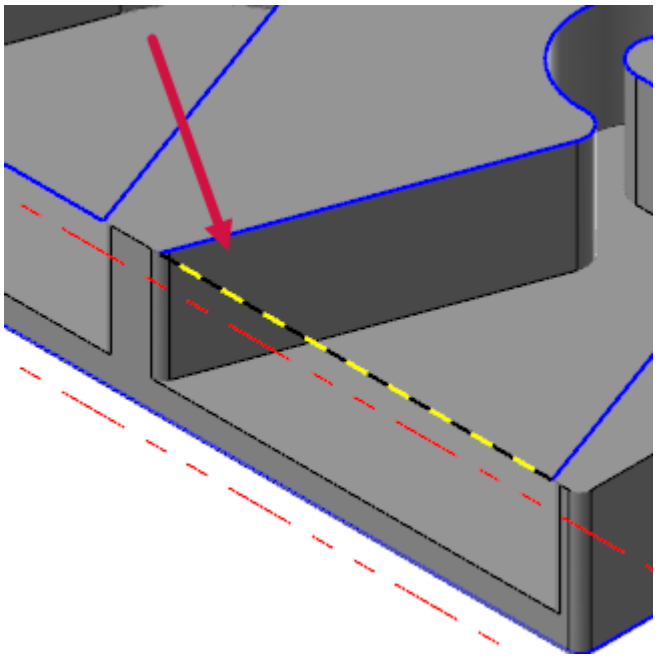


4. Select **Create new plane**, **From entity normal**.

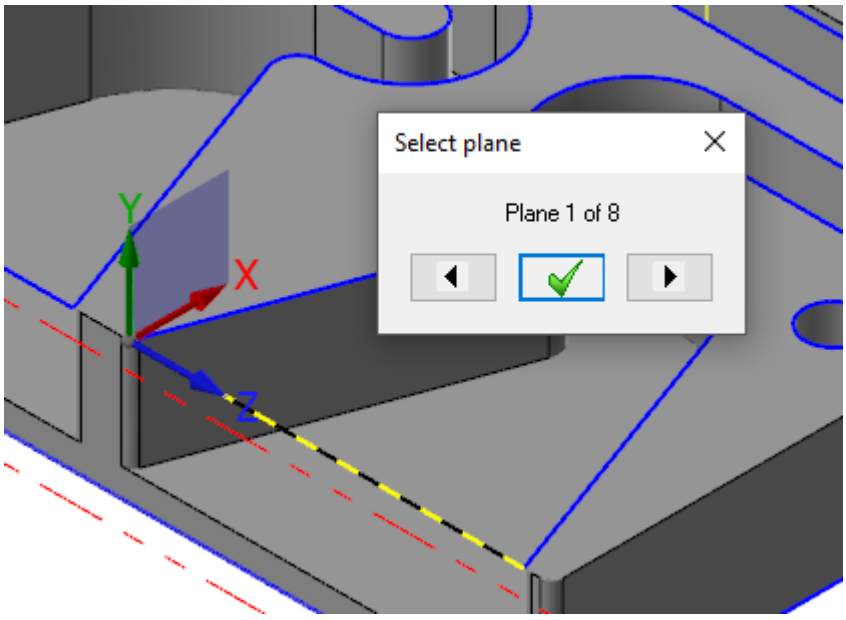


This new plane will be the basis for the section view.

5. Select the line entity as shown below.



6. Use the **Select plane** dialog box buttons to find the plane that is shown below.

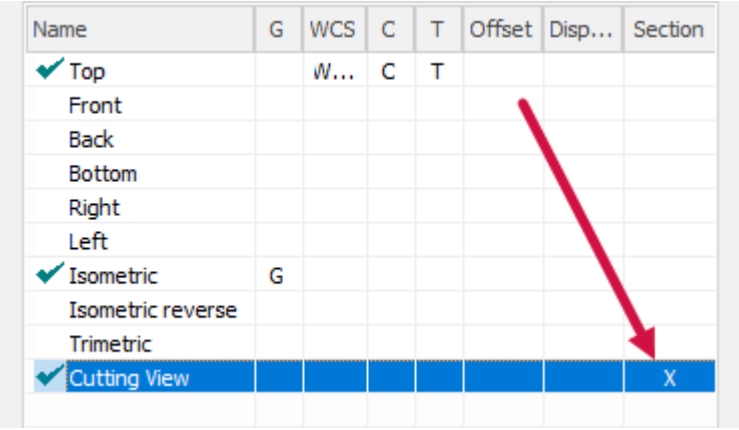


7. Click **OK**.

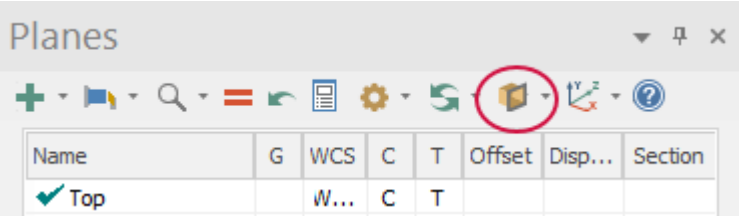
   The **New Plane** function panel displays.

8. Enter **Cutting View** for **Name** and click **OK**.

9. In the Planes Manager, click in the **Section** column for the **Cutting View** plane you just created.

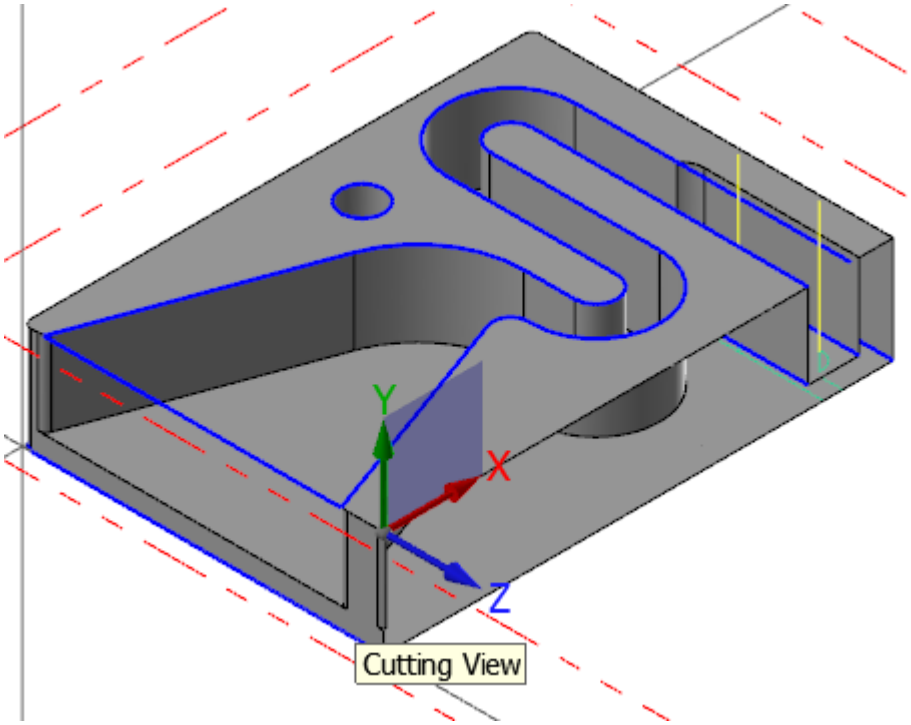


10. Then, click **Section View** on the Planes Manger toolbar.

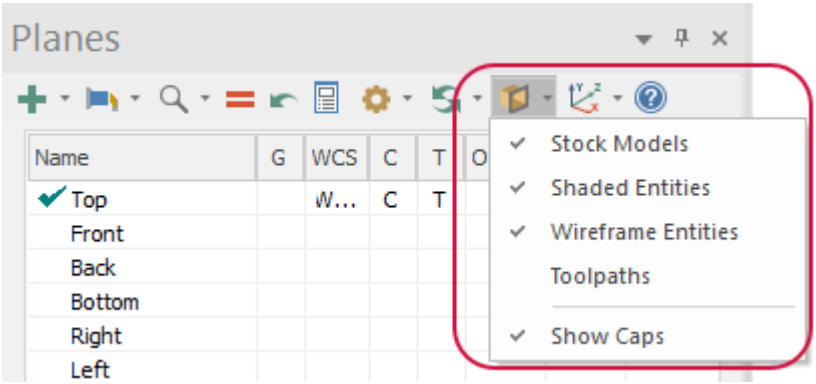


> **NOTE**
> The **Section View** button and drop-down is also on the **View** tab.

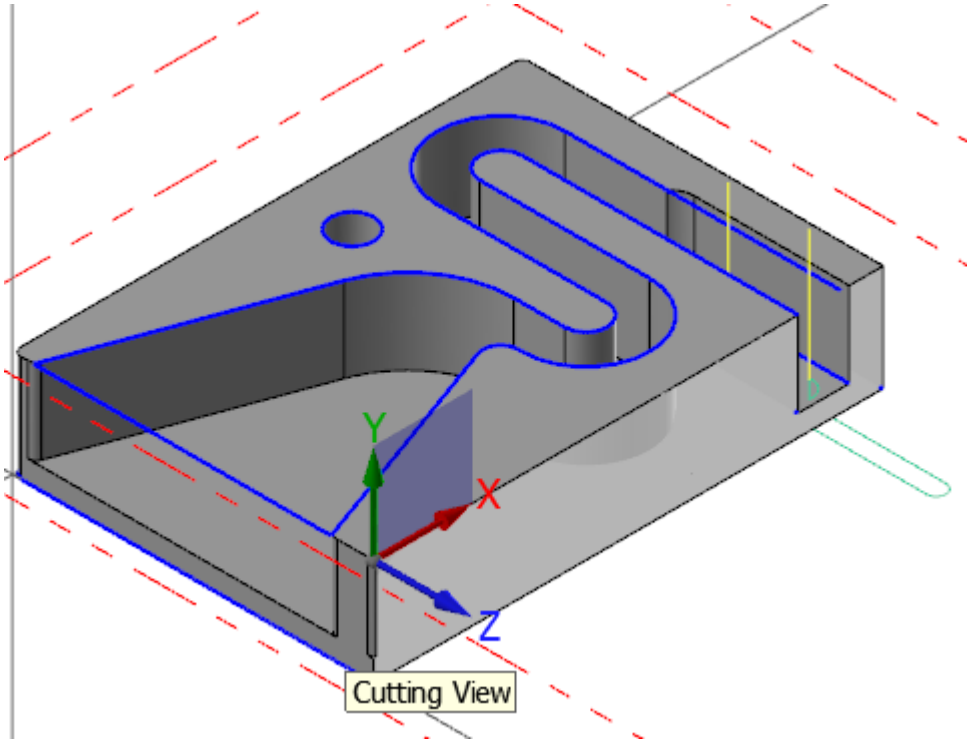
11. The part then displays as shown below.



12. In the **Section View** drop-down menu, deselect **Toolpaths** and select **Show Caps**.



By deselecting **Toolpaths**, the toolpaths are not cut by the section view. **Show Caps** fills in sections of the part with a solid cap.

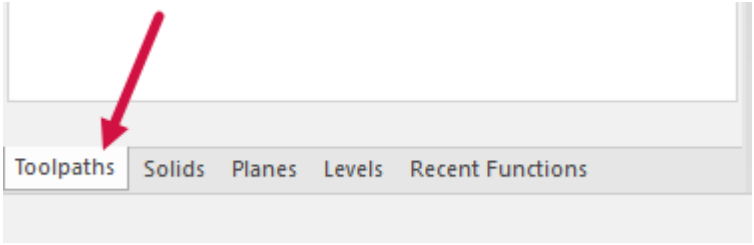


13. Save your file.

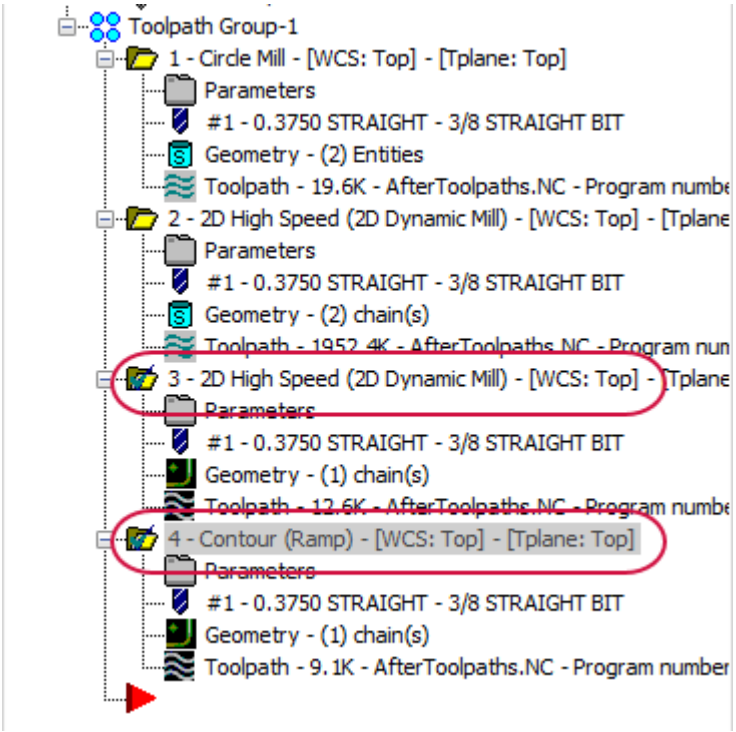
Exercise 4: Backplotting the Toolpaths

In this exercise, you will use Classic Backplot to simulate the Dynamic Mill and Contour toolpaths created in the previous exercises and use the section view to easily view the motion.

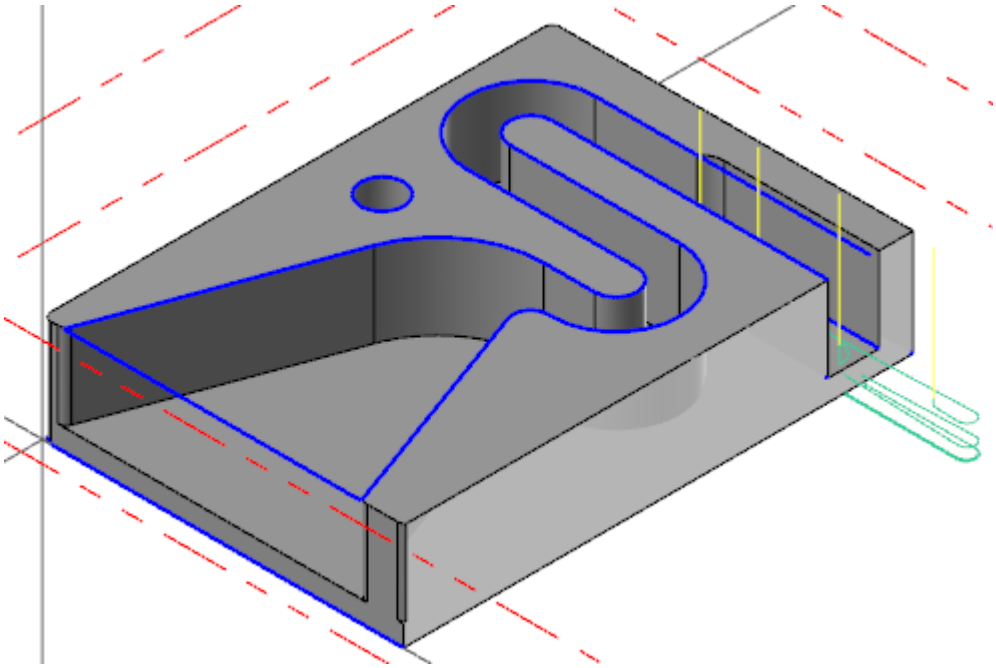
1. Your file Amplifier-XXX which you saved from the previous exercise should be open in Mastercam.
2. Select the **Toolpaths** tab to bring the Toolpaths Manager to the forefront.



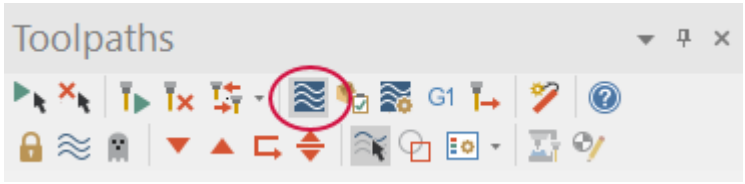
3. In the Toolpaths Manager, hold down [**Ctlr**] and select **3 - 2D High Speed (2D Dynamic Mill)** and **4 - Contour (Ramp)**.



4. Both toolpaths are displayed in the graphics window and you can see them clearly due to the section view created in the previous exercise.

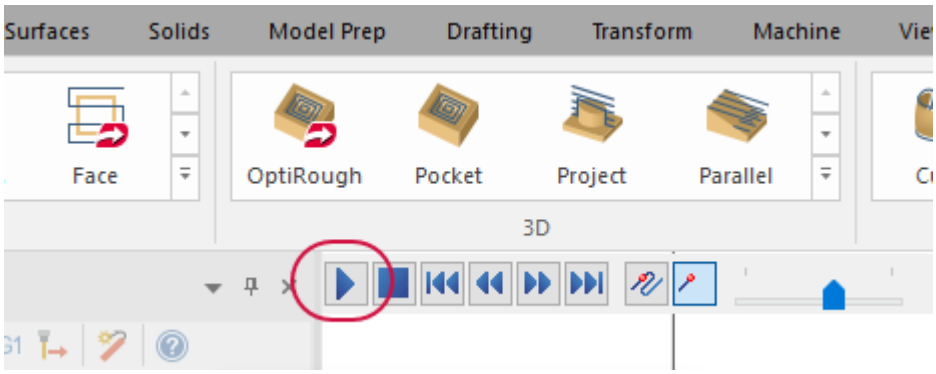


5. Select **Backplot selected operations** from the Toolpaths Manager toolbar.

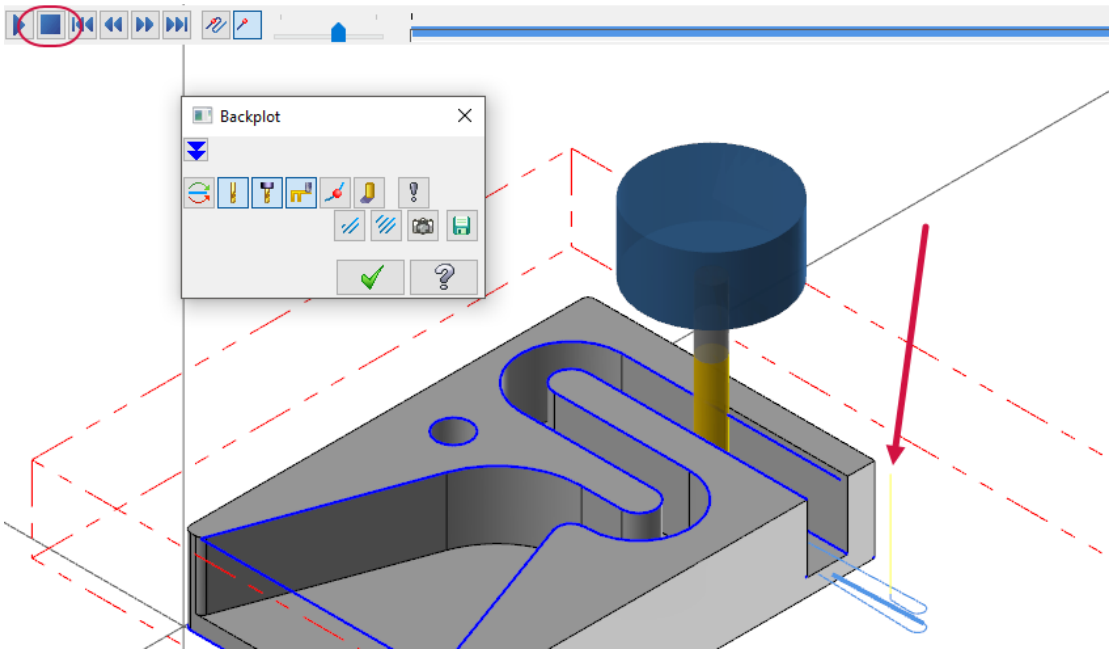


The **Backplot** dialog box and playback bar display.

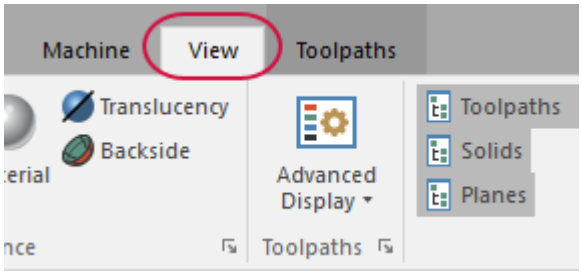
6. Press **Play** in the playback bar to begin the simulation.



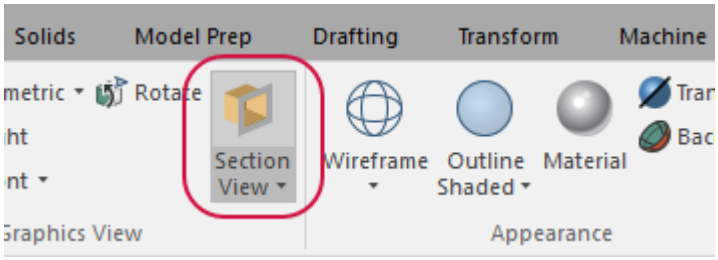
7. Press **Stop** about halfway through the simulation.



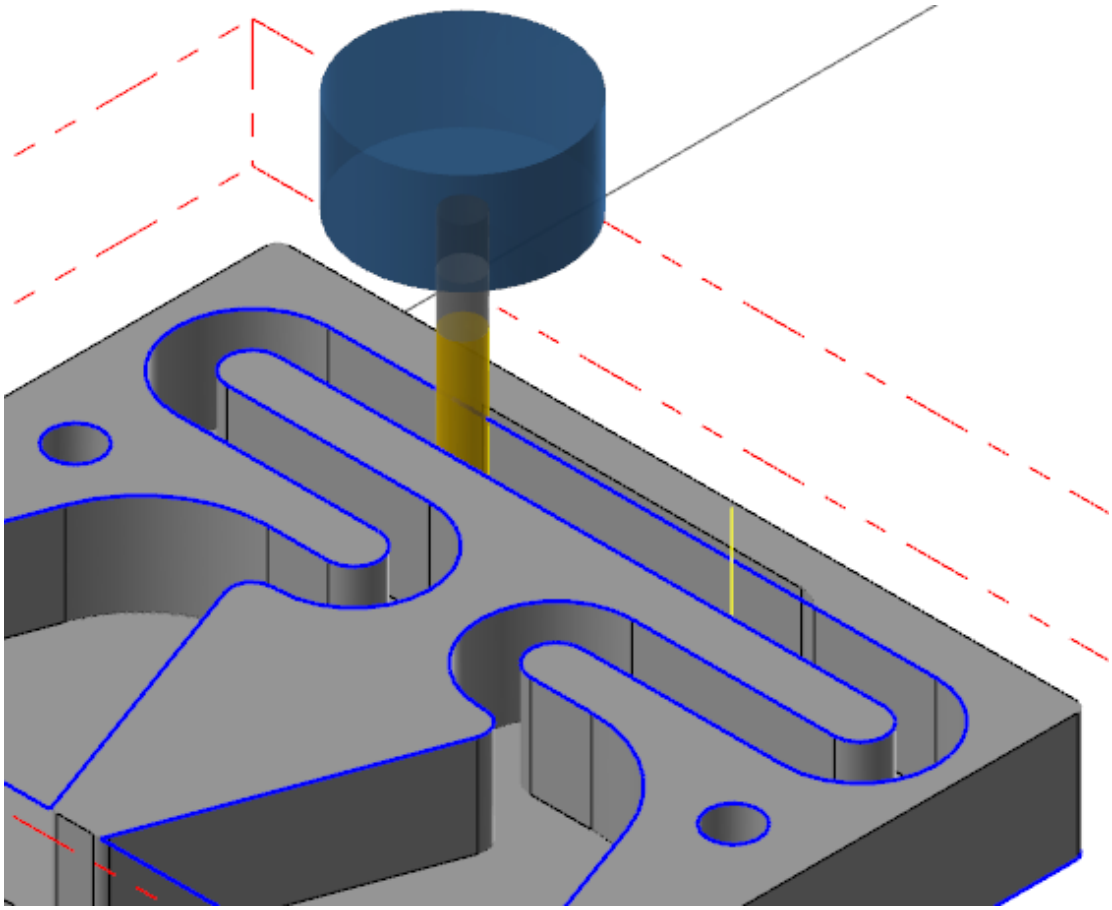
8. Select the **View** tab.



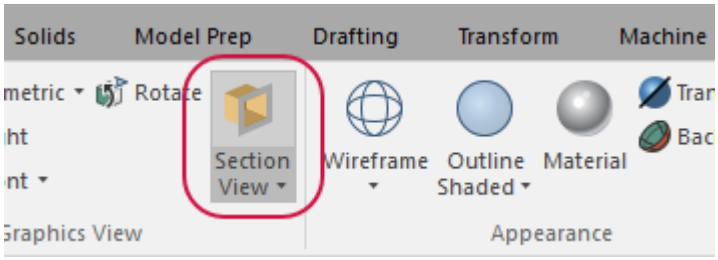
9. Select **Section View** to toggle section views off.



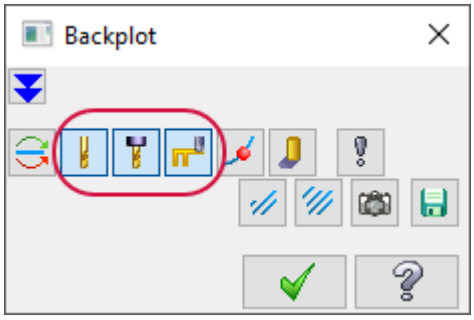
10. You can no longer see the toolpath simulation clearly in the graphics window. By creating and using section views, you are easily able to see the interior of a part.



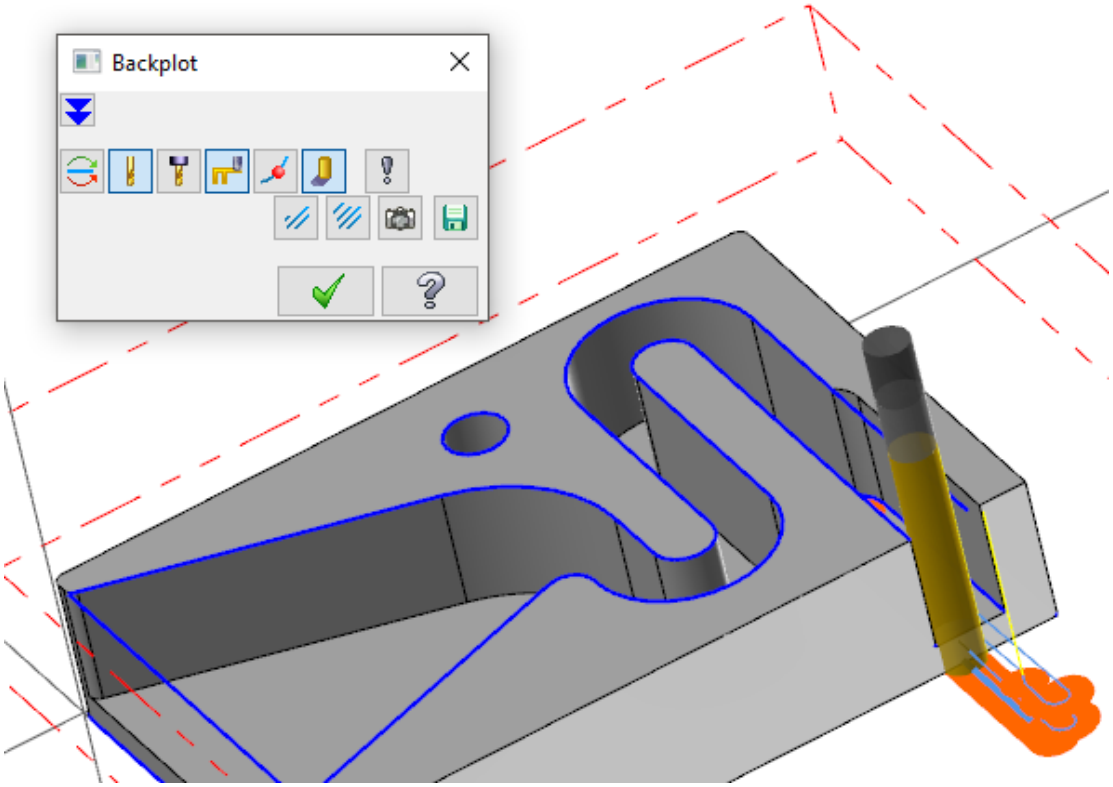
11. Select **Section View** to toggle section views on.



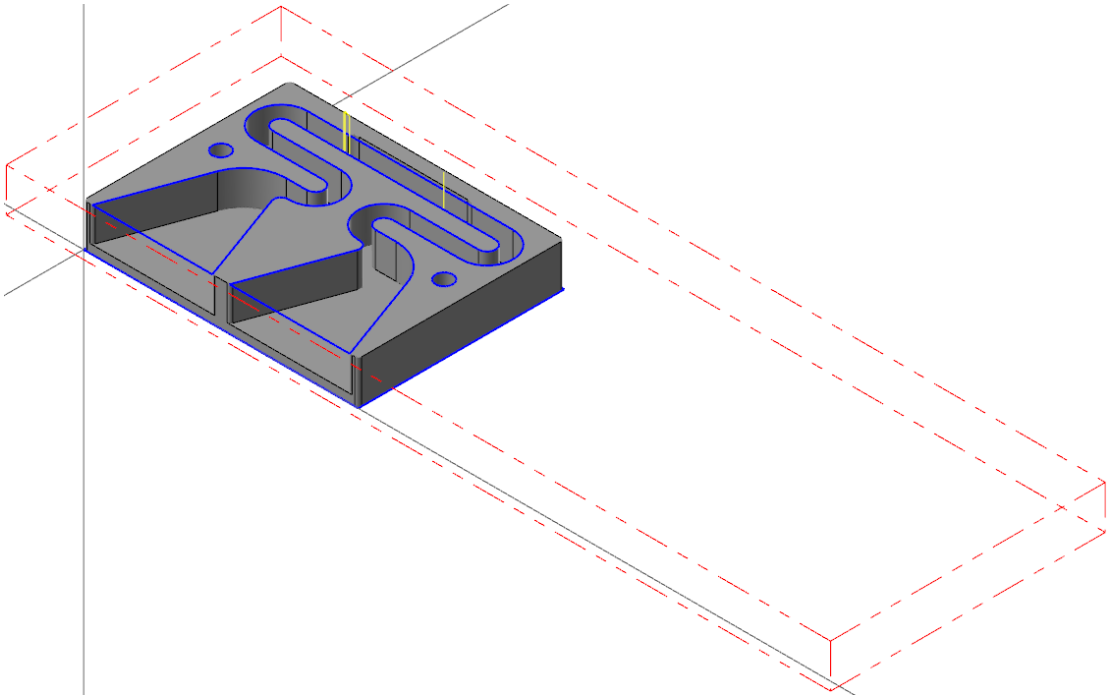
12. Return to the **Backplot** dialog box. This dialog box includes several visibility options, similar to the options within Mastercam Simulator.



13. Experiment with the options and continue running the simulation. You can also rotate the part in the graphics window view the simulation.



14. Once you are satisfied with the simulation, click **OK** in the **Backplot** dialog box.
15. On the **View** tab, select **Section View** again to toggle section views off.
16. Return your part to Isometric view if necessary.
17. Save your part.



SHOW US WHAT YOU LEARNED!

Can you answer these questions?

1. Which of the following is *not* a **Contour type** you can select for a 2D Contour toolpath?
    a. 2D
    b. Ramp
    c. 3D
    d. Dynamic
2. What are the benefits of using the Section View function?
3. You can control visibility options (such as tool visibility) when using Classic Backplot.
    a. True
    b. False
4. What is the purpose of the **Break through** option for 2D Contour toolpaths?
5. Which of the following is *not* an entity you can turn off sectioning for?
    a. Stock Models
    b. Toolpaths
    c. Wireframe Entities
    d. Unshaded entities

CHAPTER 5

PROGRAMMING THE TABS

In this chapter, you use the **Curve All Edges** function to create wireframe to support the filleted edges of the part. Then, you create tab points and a Contour toolpath to machine those points as tabs.

Tabs are uncut areas of a toolpath that assist in holding down a part. While the tool is cutting the contour, it retracts slightly at the tab location to leave a small amount of stock, then plunges back to the normal cutting depth. Tabs are especially useful when you are cutting out small parts, multiple parts, or nested parts.

Refer to the "Using Tabs" Help topic in Mastercam for more information.

Goals

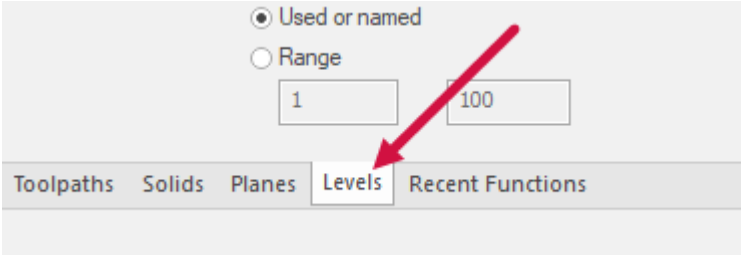
- Understanding the creation and use of tabs.
- Explore and use the **Curve All Edges** function.
- Explore the Levels Manager.
- Program a Contour toolpath.

> **CAUTION**
> When programming toolpaths, only change the values of the indicated parameters. Leave unmentioned values at their default values.

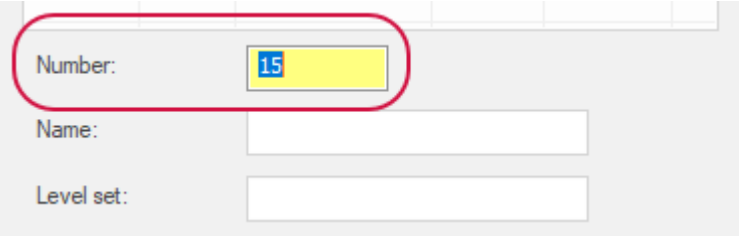
Exercise 1: Creating the Support Geometry

In this exercise, you will create tab geometry based on the solid model. Later in this chapter, you will use this geometry to program a Contour toolpath.

1. Your file Amplifier-XXX which you saved from the previous exercise should be open in Mastercam.
2. Select the **Levels** tab to bring the Levels Manager to the forefront.

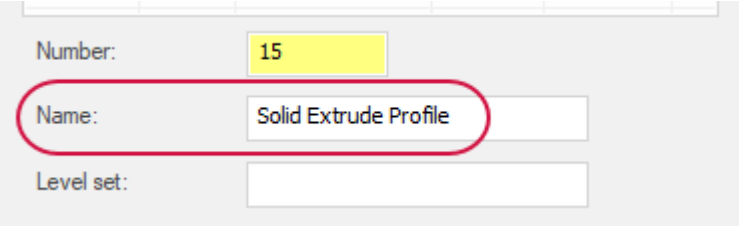


3. In the **Number** parameter of the Levels Manager, enter **15** and press [**Enter**].

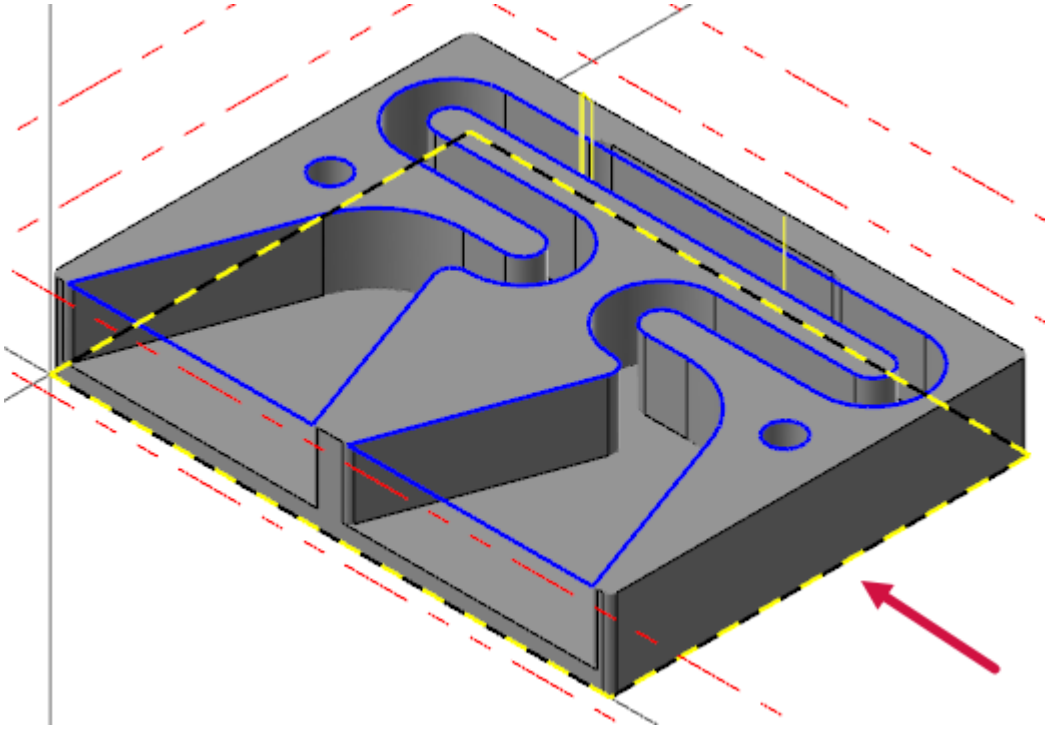


There is now a new level in the Levels Manager. By default, the new level becomes your main level.

4. Enter **Solid Extrude Profile** for **Name**.



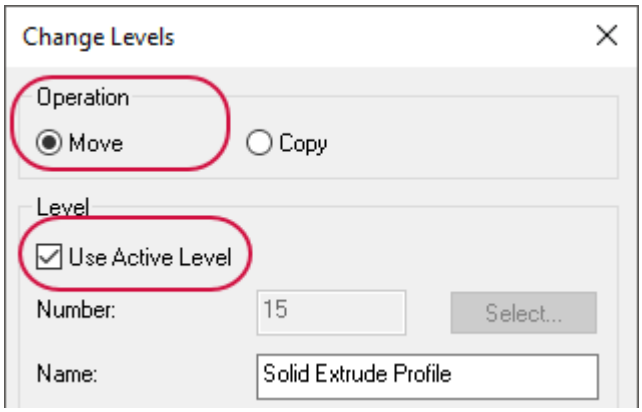
5. In the graphics window, hold [**Shift**] and select the rectangle wireframe as shown below.



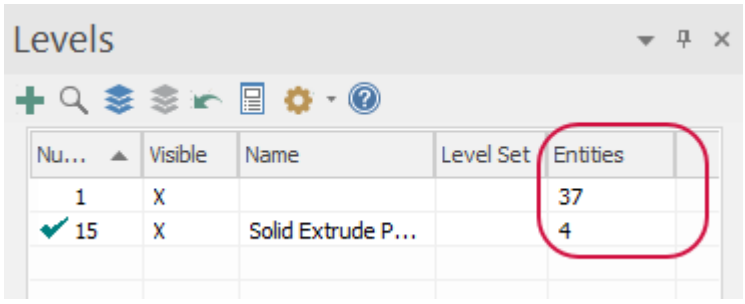
6. Right-click in the graphics window and select **Change level** from the mini-toolbar.

The **Change Levels** dialog box displays.

7. Ensure that **Operation** is set to **Move** and **Use Active Level** is selected.

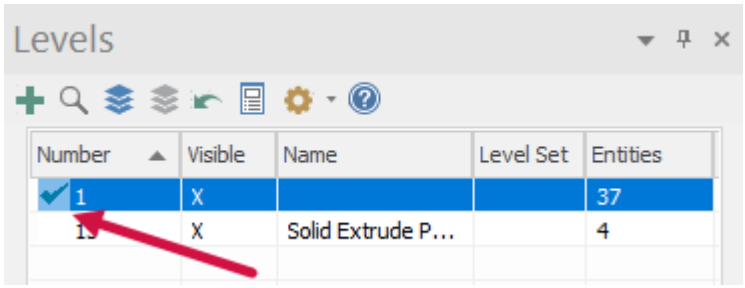


8. Click **OK**.

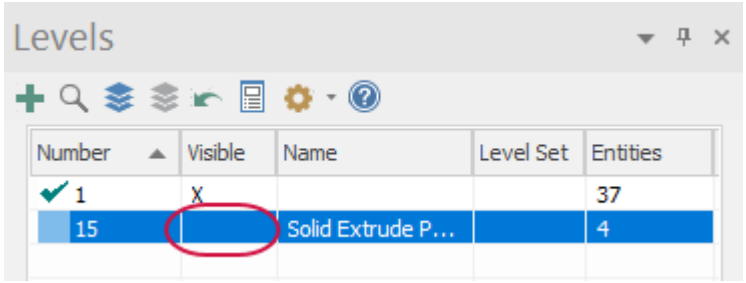


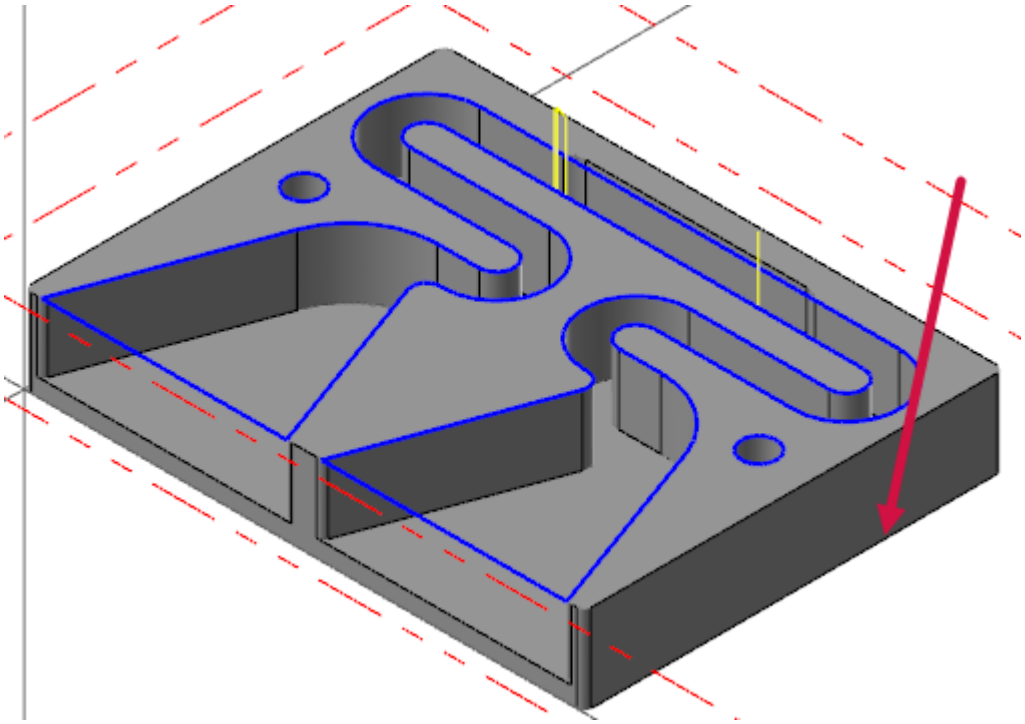
The wireframe entities have moved from Level 1 to Level 15.

9. In the Levels Manager, click in the **Number** column for Level 1 to set it as the main level.

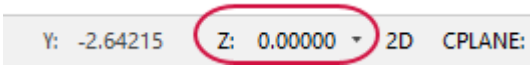


10. Click in the **Visible** column for Level 15 to clear the **X** and hide the entities from the graphics window.

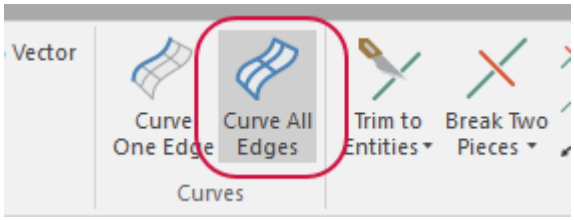




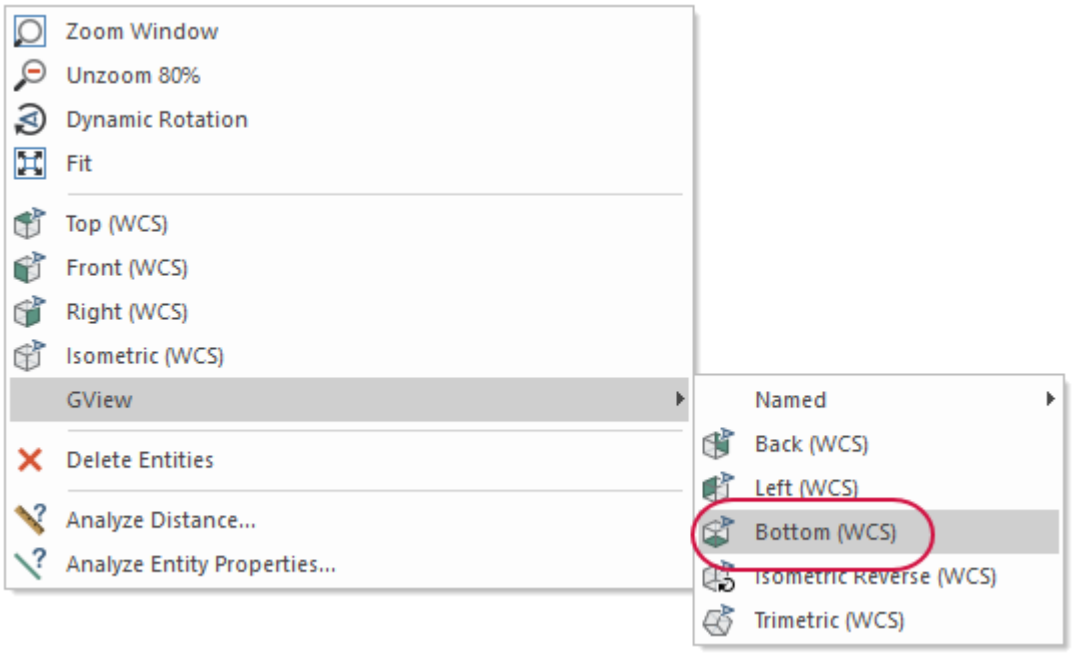
11. In the **Status Bar**, click the value next to **Z** and enter **0.0**. Press [**Enter**] to apply the value.



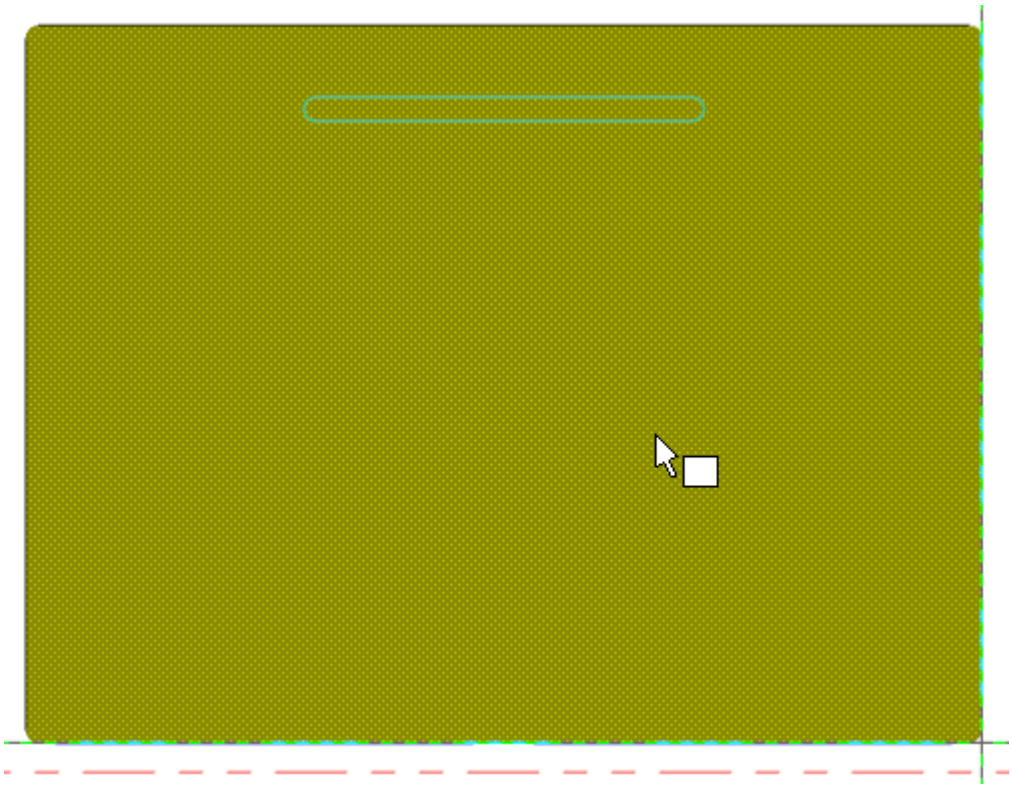
12. Select **Curve All Edges** on the **Wireframe** tab.



13. Right-click in the graphics window and select **Gview**, **Bottom (WCS)**.



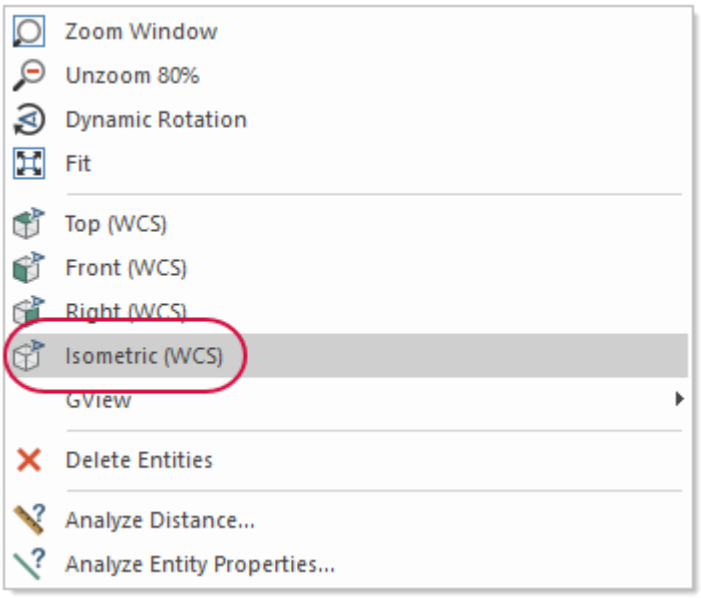
14. Select the bottom face of the solid body and press [**Enter**].



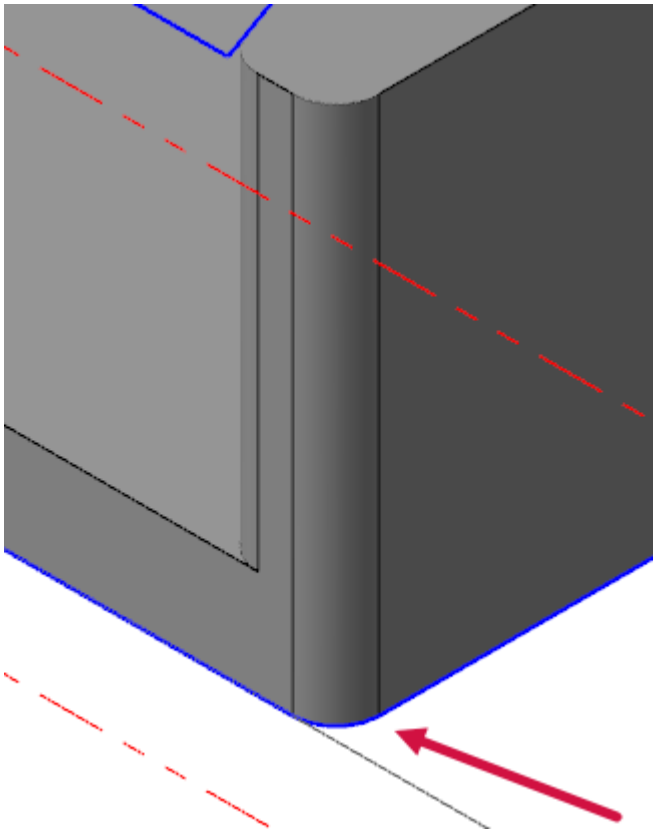
The **Curve On All Edges** function panel is now active.

15. Click **OK** in the **Curve On All Edges** function panel.

16. Right-click in the graphics window and select **Isometric (WCS)**.

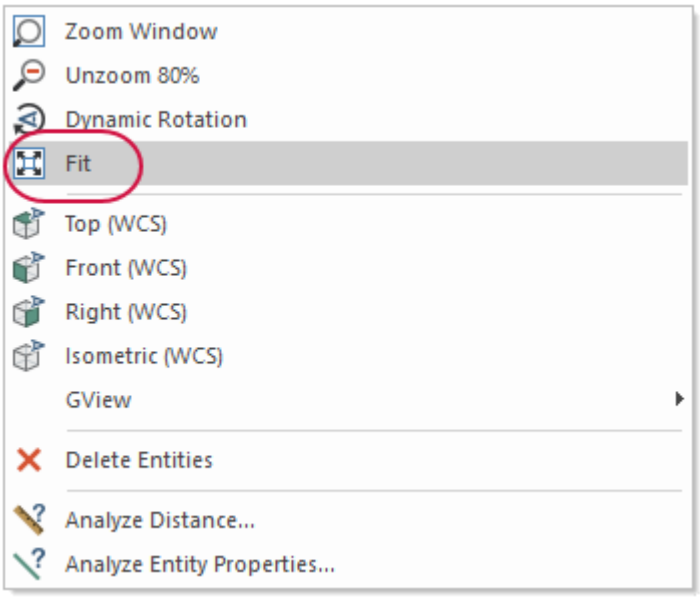


17. Use the middle mouse wheel to zoom into a corner of the solid part.

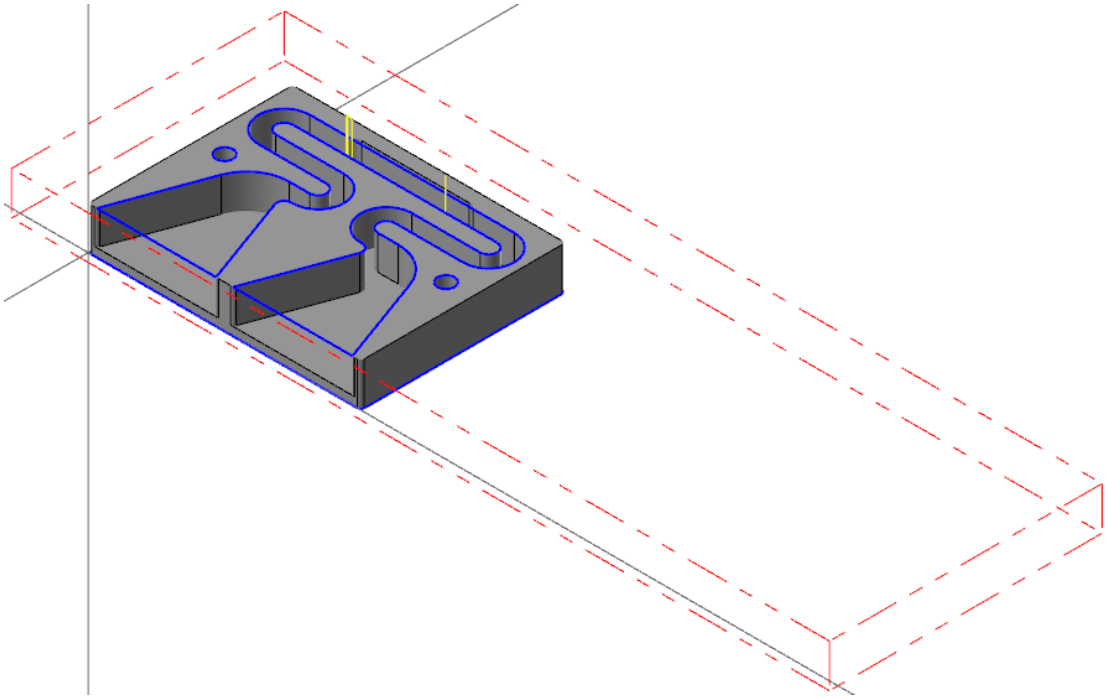


Curve On All Edges creates wireframe geometry based on the edges of the solid face, including the fillets.

18. Right-click in the graphics window and select **Fit**.



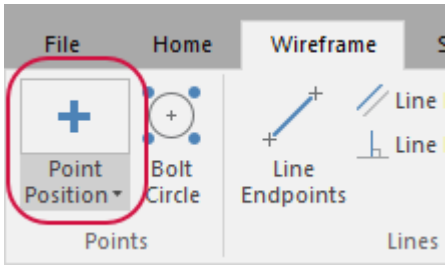
19. Save your part.



Exercise 2: Creating the Tab Points

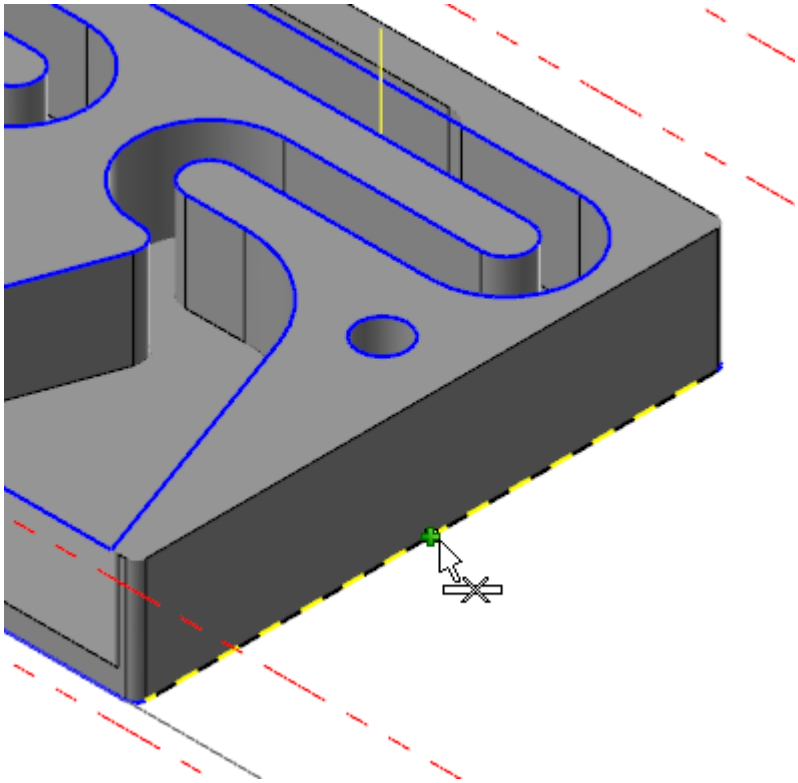
In this exercise, you will create points on the wireframe geometry created in the previous exercise. The points will be used by a Contour toolpath to automatically create tabs. The tabs will be used to help secure the part to the table during machining.

1. Your file Amplifier-XXX which you saved from the previous exercise should be open in Mastercam.
2. Select **Point Position** on the **Wireframe** tab.



   The **Point Position** function panel displays.

3. Select the midpoint of the three wireframe lines created in the previous exercise. You do not need to create a point on the front of the part.

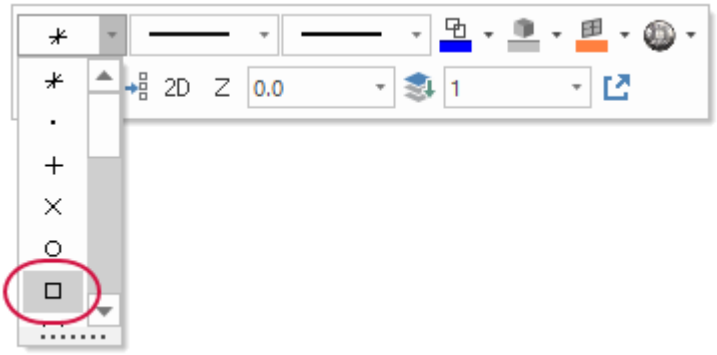


   You will need to rotate the part or unshade the part to select the midpoint of the three lines.

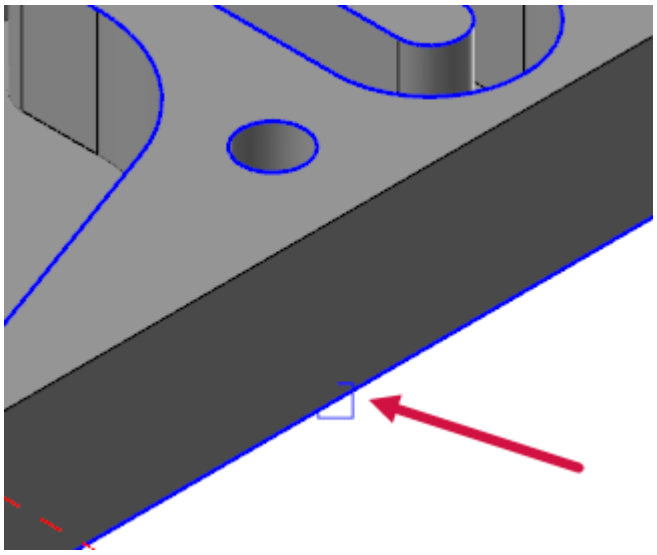
4. Click **OK** in the **Point Position** function panel.
5. If necessary, return your part to Isometric view.

6. Click the **Select all point entities** Quick Mask. All point entities are then selected in the graphics window.

7. Right-click in the graphics window and select the square point style from the **Point Style** drop-down.



The point entities now display as squares.

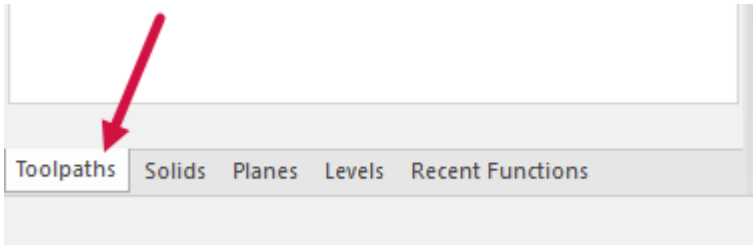


8. Save your file.

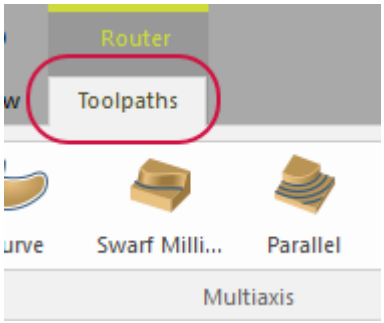
Exercise 3: Creating the Contour Toolpath

In this exercise, you will create a Contour toolpath that will machine the outside walls of the part. The Contour toolpath will automatically create tabs based on the square points created in the previous exercise.

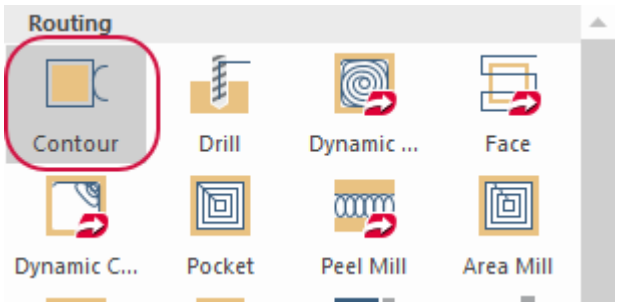
1. Your file Amplifier-XXX which you saved from the previous exercise should be open in Mastercam.
2. Select the **Toolpaths** tab to bring the Toolpaths Manager to the forefront.



3. Select the **Router Toolpaths** contextual tab.

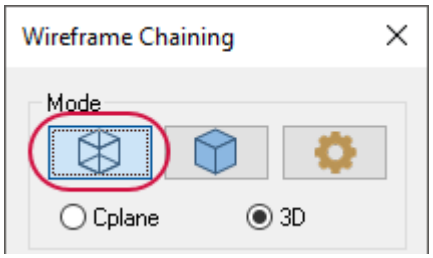


4. Select **Contour** from the **2D** gallery drop-down.

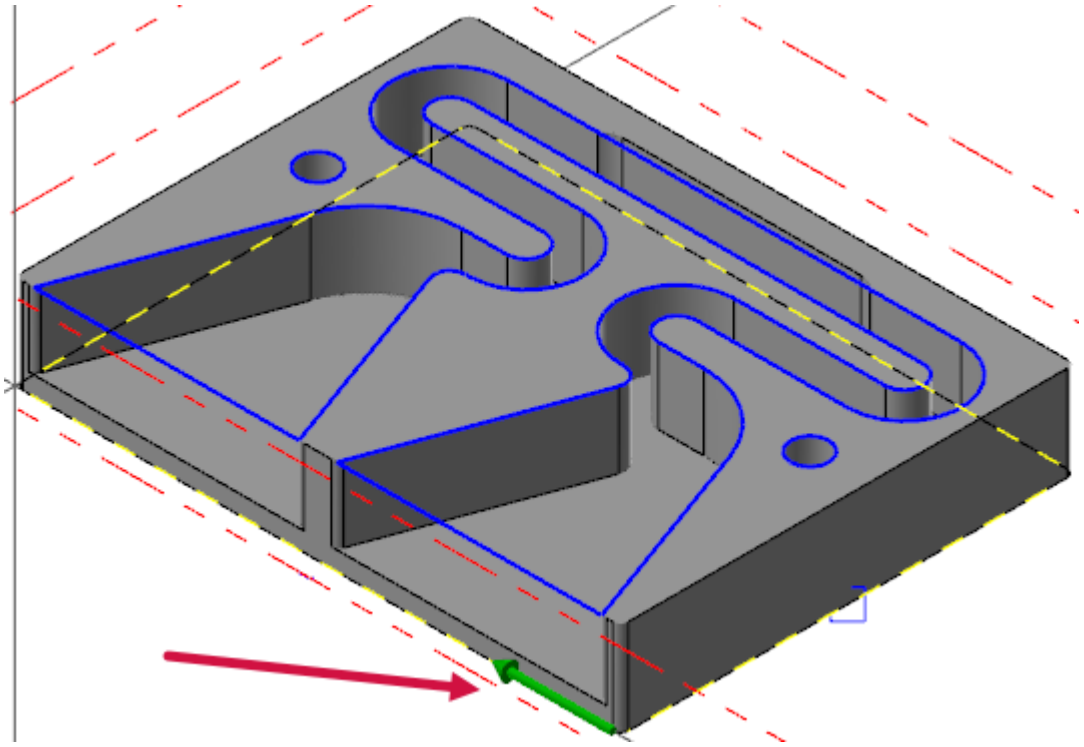


The **Chaining** dialog box displays.

5. If necessary, select **Wireframe** to set the chaining mode to wireframe.



6. Select the wireframe rectangle as shown below. Ensure you are selecting in the area pointed out below.

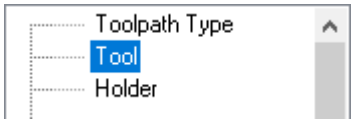


Ensure that your chain direction (the green arrow) matches the chain direction above. If not, use the **Reverse** button on the **Chaining** dialog box to change the direction of the chain.

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

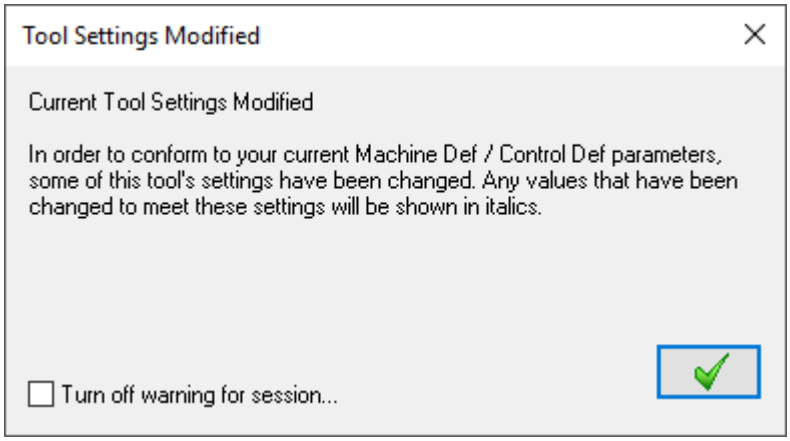
   The **2D Toolpaths - Contour** dialog box displays.

8. Select the **Tool** page.

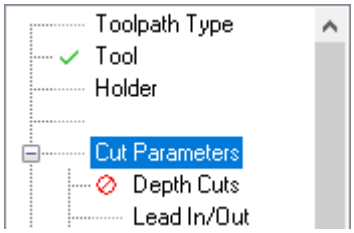


9. Select the **3/8 STRAIGHT BIT** tool from the tool list.

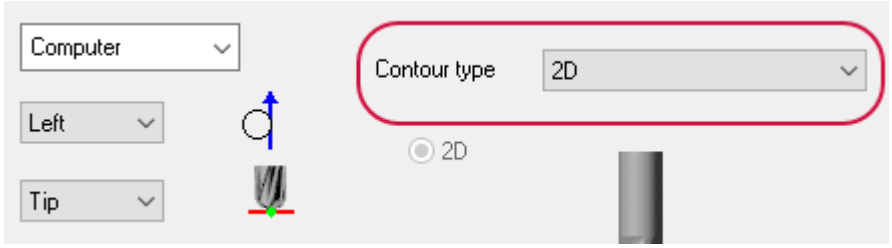
10. You will receive a tool settings modified warning. This message informs you that the tool values have been modified to fit your current machine and control definitions. Click **OK** to accept the changes.



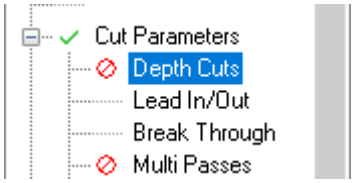
11. Select the **Cut Parameters** page.



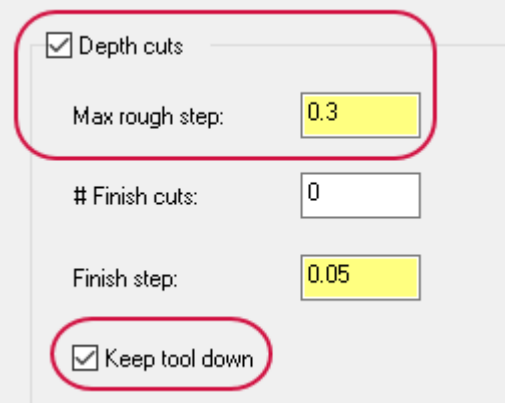
12. Set **Contour type** to **2D**.



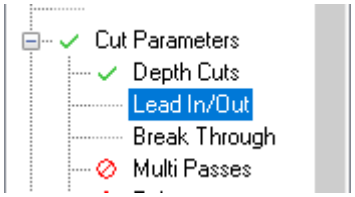
13. Select the **Depth Cuts** page.



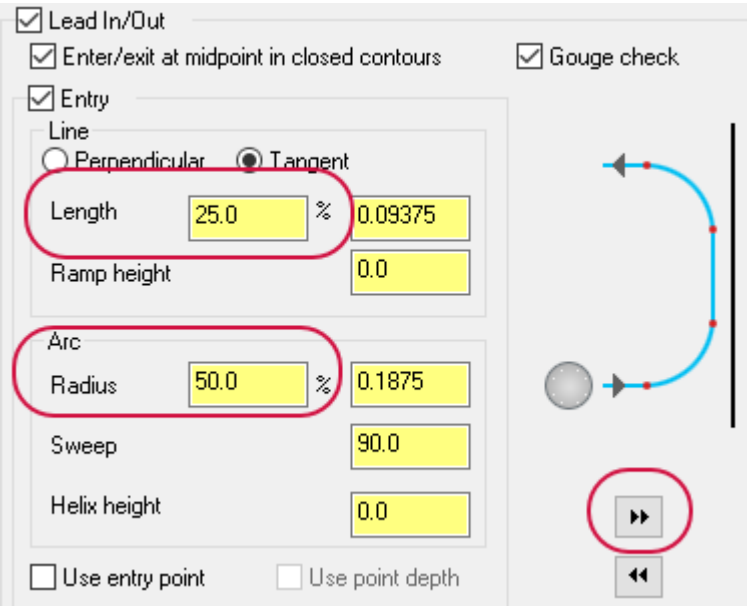
14. Enter the following parameters:
    a. Select **Depth cuts**.
    b. Enter **0.3** for **Maximum rough step**.
    c. Select **Keep tool down**.



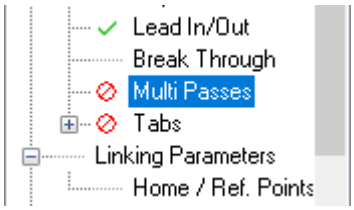
15. Select the **Lead in/Out** page.



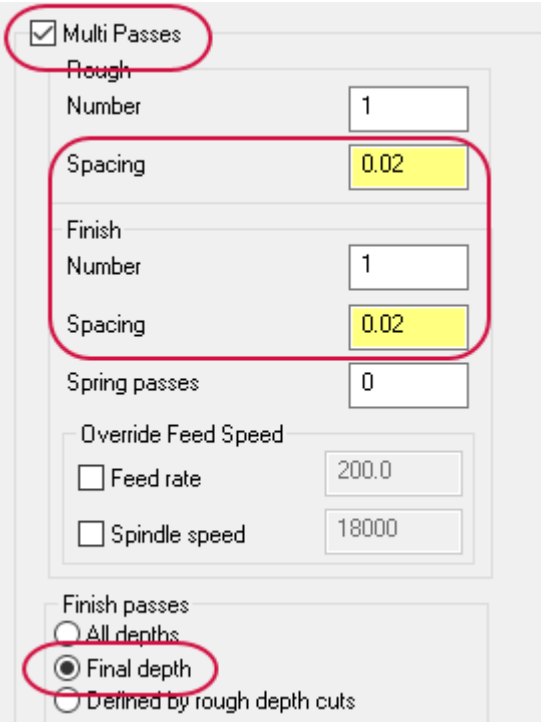
16. Enter the following parameters:
    a. Enter **25.0** for **Length**.
    b. Enter **50.0** for **Radius**.
    c. Click the **Copy** button to copy these values from the **Entry** section to the **Exit** section.



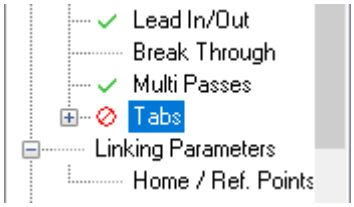
17. Select the **Multi Passes** page.



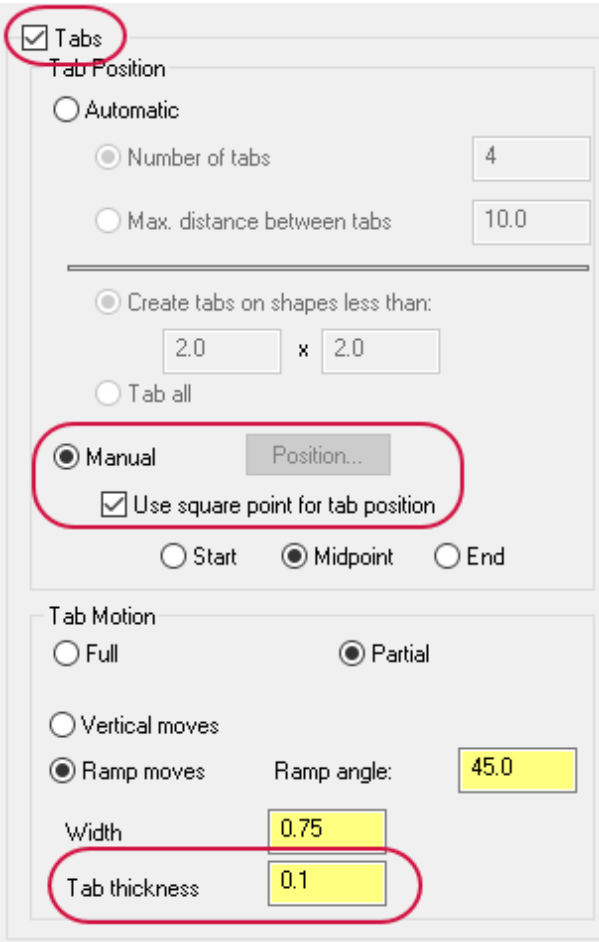
18. Enter the following parameters:
    a. Select **Multi Passes**.
    b. Enter **0.02** for **Spacing** in both the **Rough** and **Finish** group.
    c. Enter **1.0** for **Number** under **Finish**.
    d. Set **Finish passes** to **Final depth**.



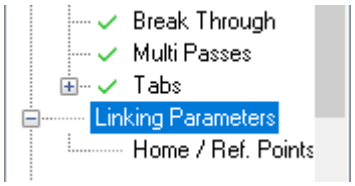
19. Select the **Tabs** page.



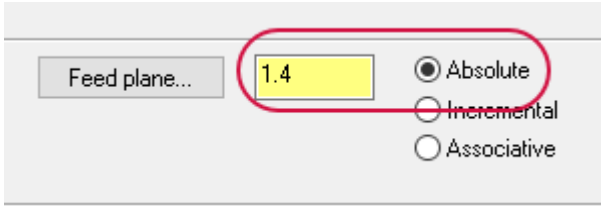
20. Enter the following parameters:
    a. Select **Tabs**.
    b. Ensure **Tab Position** is set to **Manual**.
    c. Select **Use square point for tab position**.
    d. Enter **0.1** for **Tab thickness**.



21. Select the **Linking Parameters** page.

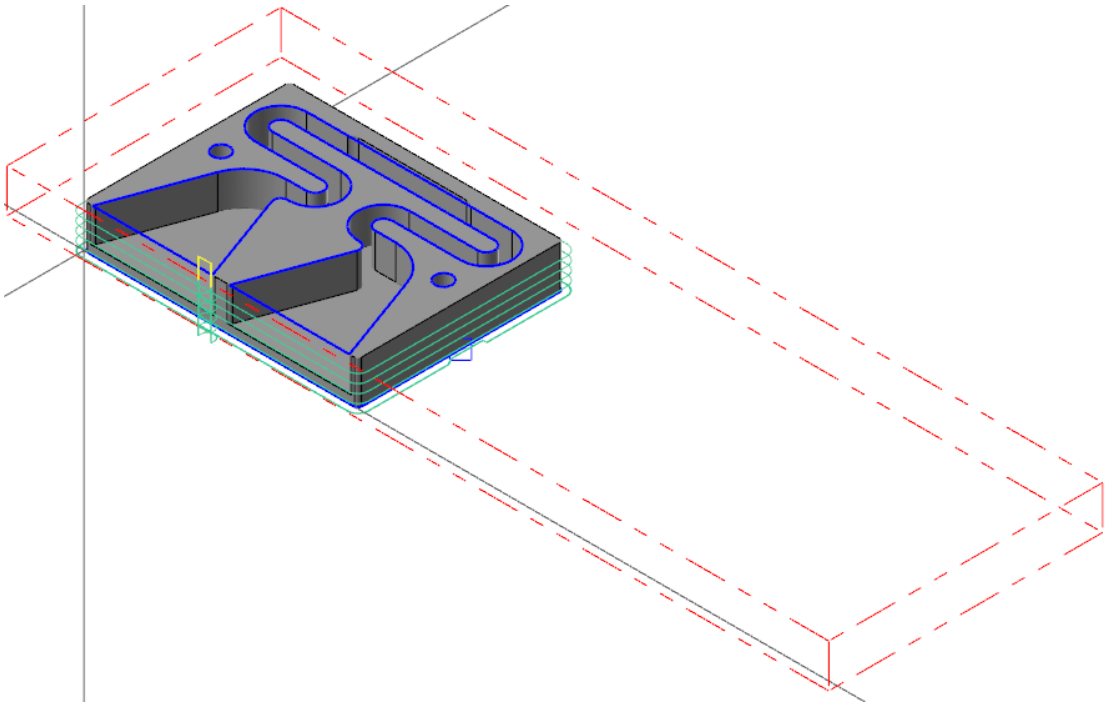


22. Enter **1.4** for **Feed plane** and set to **Absolute**.



23. Click **OK** in the **2D Toolpaths - Contour** dialog box.

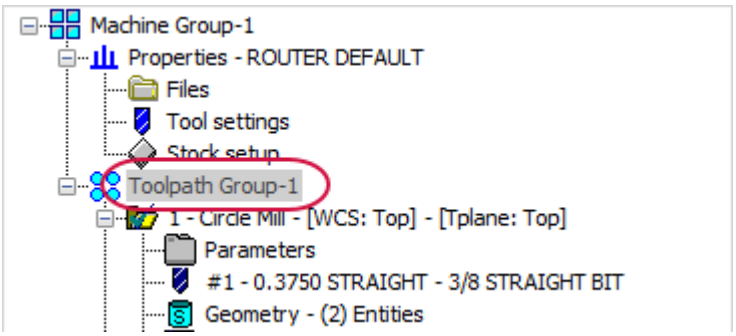
24. Save your file.



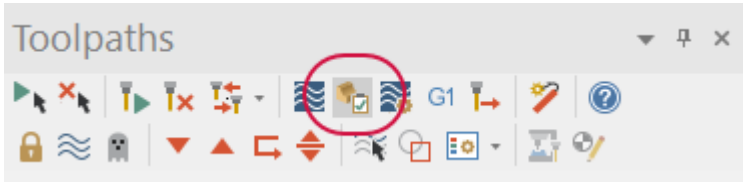
Exercise 4: Verifying All the Toolpaths

In this exercise, you use Mastercam Simulator to view and verify all of the created toolpaths so far.

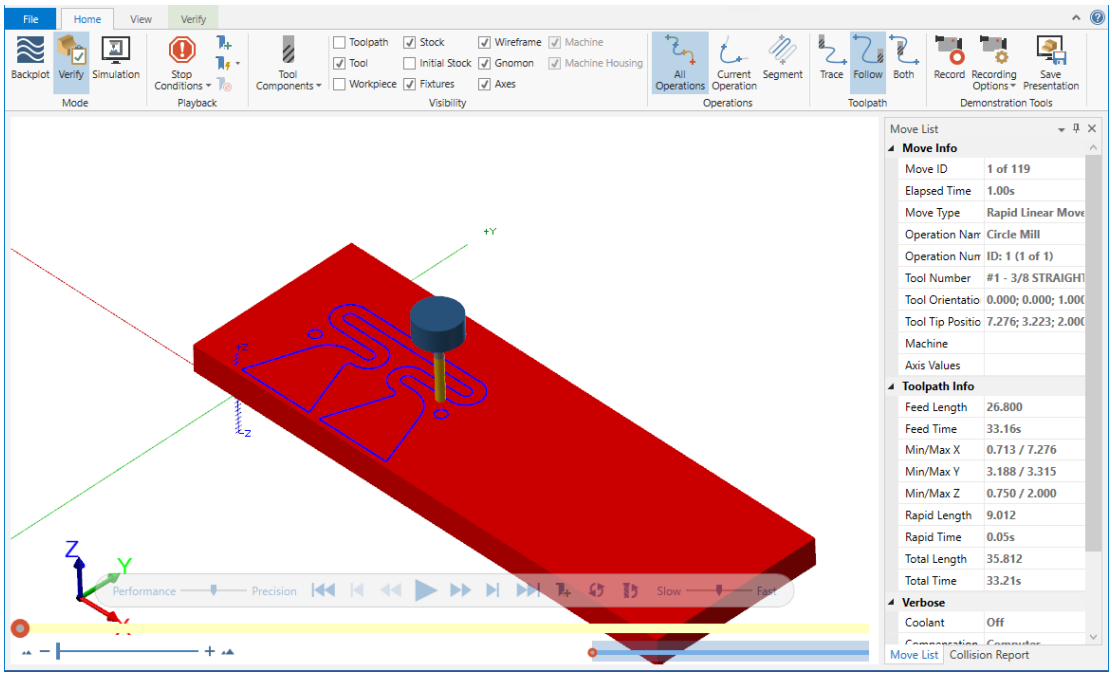
1. Your file Amplifier-XXX which you saved from the previous exercise should be open in Mastercam.
2. In the Toolpaths Manager, select **Toolpath Group - 1**.



3. Select **Verify selected operations** from the Toolpaths Manager toolbar.

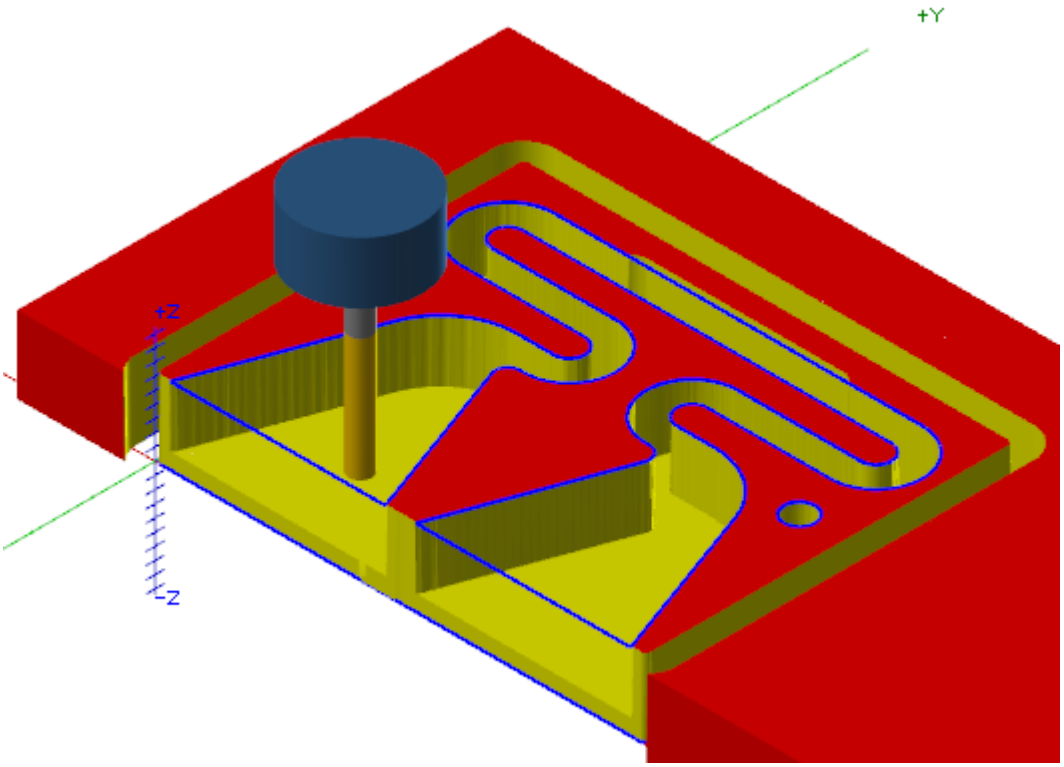


Mastercam Simulator displays.

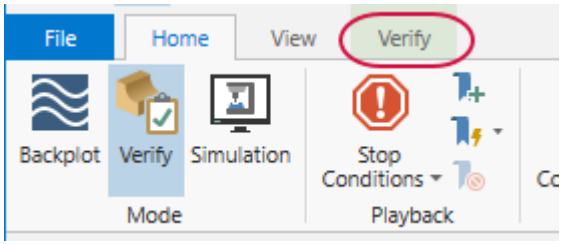


4. Use the playback bar to run the verification.

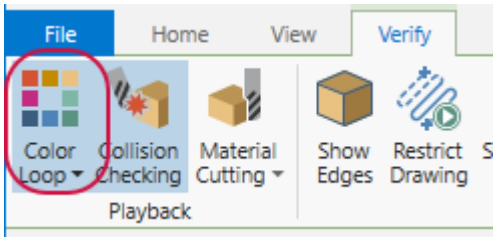
The verification shows the machining of the entire part.



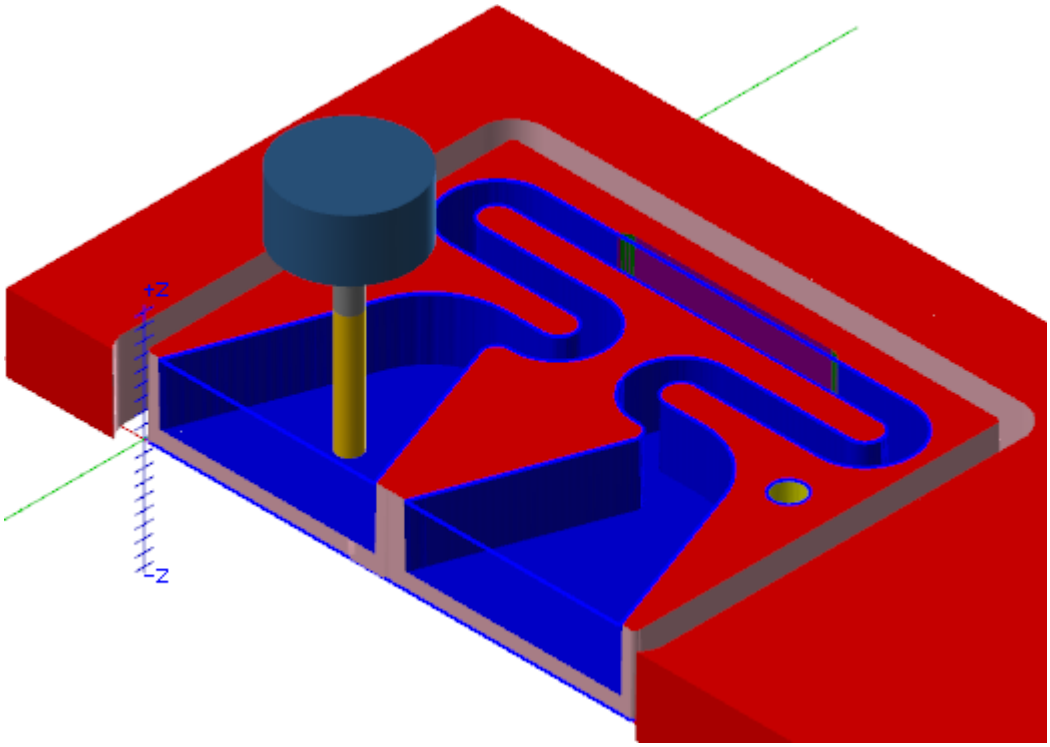
5. Select the **Verify** contextual tab in Mastercam Simulator.



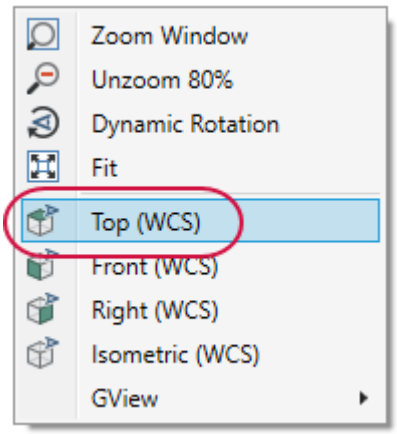
6. Select **Color Loop**.



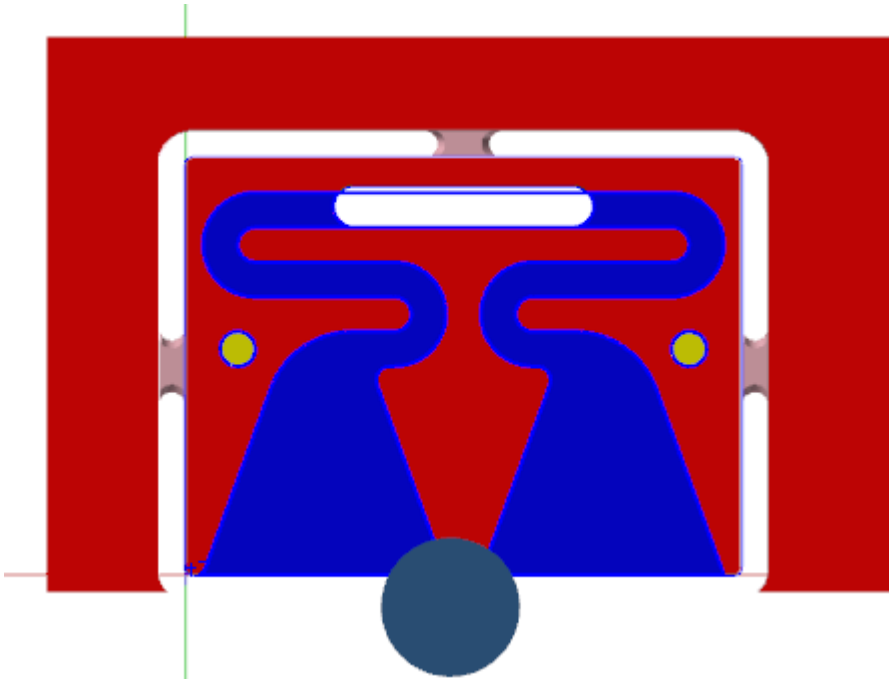
The part then changes colors in the Mastercam Simulator graphics window. **Color Loop** changes the color of the cut stock by operation or tool.



7. Right-click in Mastercam Simulator's graphics window and select **Top (WCS)**.



You can now see the tabs created by the Contour toolpath made in the previous exercise.



8. Exit Mastercam Simulator once you have finished viewing the verification.
9. Save your file.

SHOW US WHAT YOU LEARNED!

Can you answer these questions?

1. Tabs are uncut areas of a toolpath that assist in holding down a part.
    a. True
    b. False
2. Depth cuts are used to divide the total depth into separate Y axis cuts to avoid plunging the tool directly into material.
    a. True
    b. False
3. The Color Loop function in Mastercam Simulator changes the color of the cut stock only by tool type.
    a. True
    b. False
4. Which of the following is a Quick Mask selection type?
    a. Points
    b. Lines
    c. Solids
    d. Arcs
    e. All
    f. None
5. What is the differences between multi passes and depth cuts?

CHAPTER 6

DUPLICATING THE MODEL AND EDITING THE TOOLPATHS

In this chapter, you use the **Transform** function to create a second solid model to serve as the bottom part of the acoustic amplifier. You then use various functions in the Toolpaths Manager to duplicate the toolpaths and re-chain the geometry.

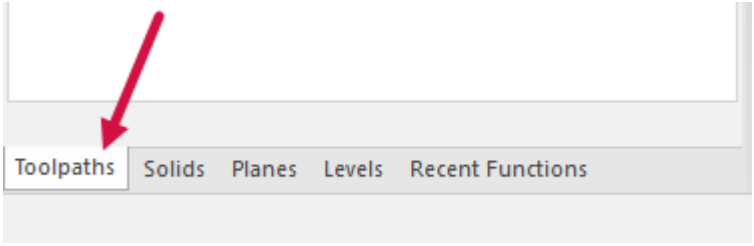
Goals

- Explore various functions of the Toolpaths Manager.
- Use the **Transform** function to copy and move the solid model.

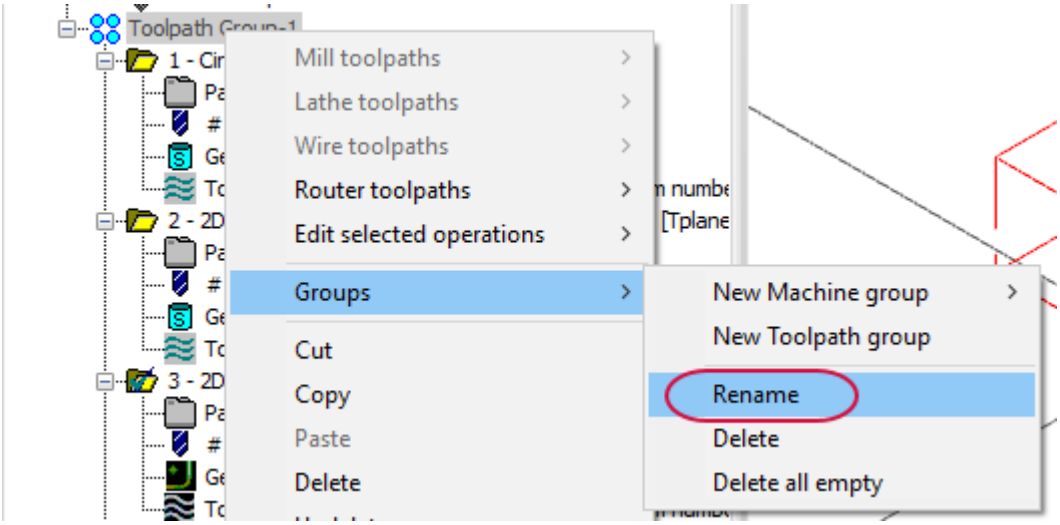
Exercise 1: Copying and Modifying Toolpath Groups

In this exercise, you will duplicate the toolpath group in the Toolpaths Manager. You will also remove toolpaths that will not be needed to machine the second solid model.

1. Your file Amplifier-XXX which you saved from the previous exercise should be open in Mastercam.
2. If necessary, select the **Toolpaths** tab to bring the Toolpaths Manager to the forefront.

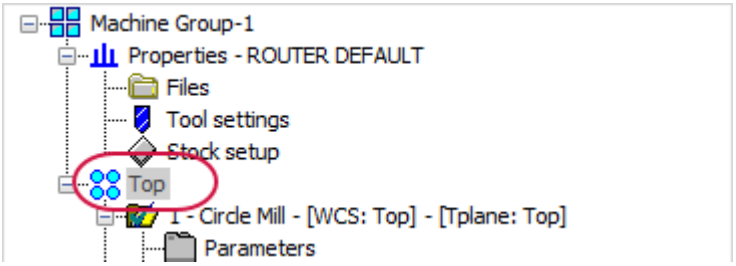


3. Right-click **Toolpath Group-1** and select **Groups**, **Rename** from the menu.

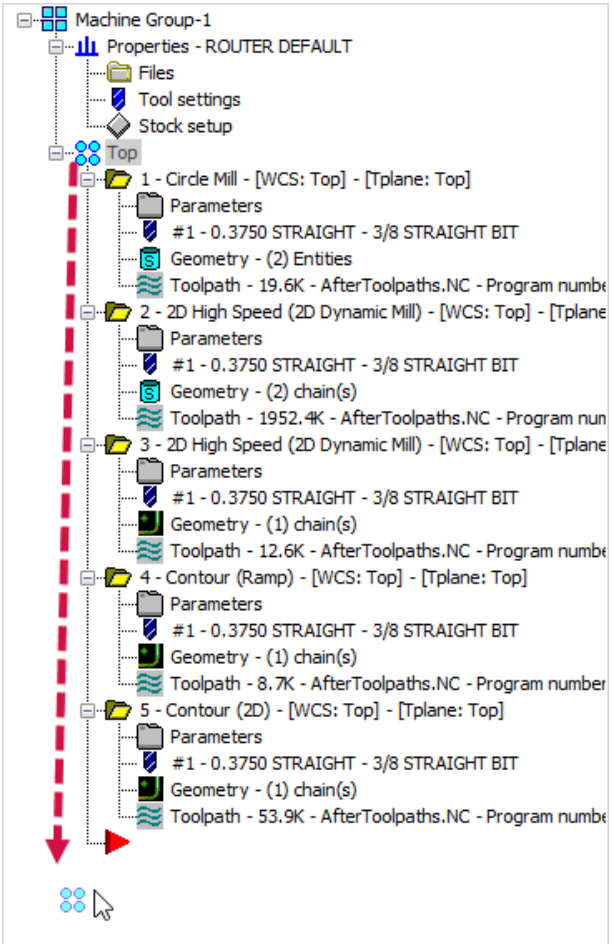


NOTE
You can also triple left-click **Toolpath Group-1** to change the name.

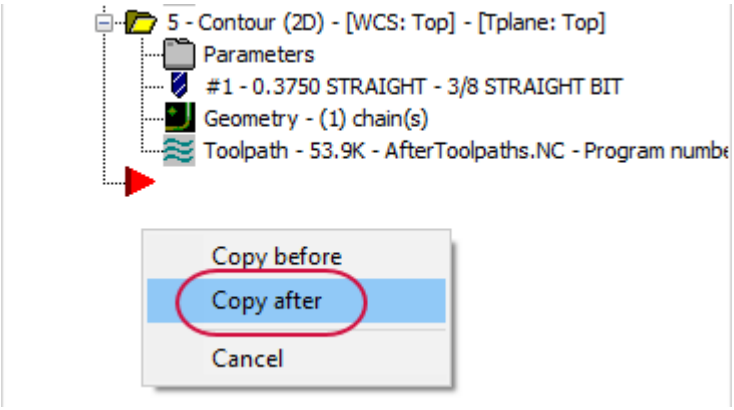
4. Type **Top** and press [**Enter**].



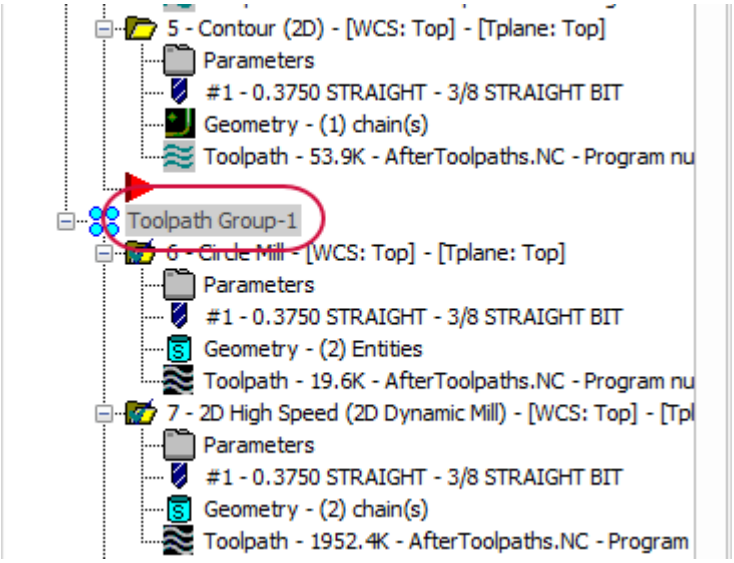
5. Right-click the **Top** toolpath group and drag your mouse to the bottom of the Toolpaths Manager.



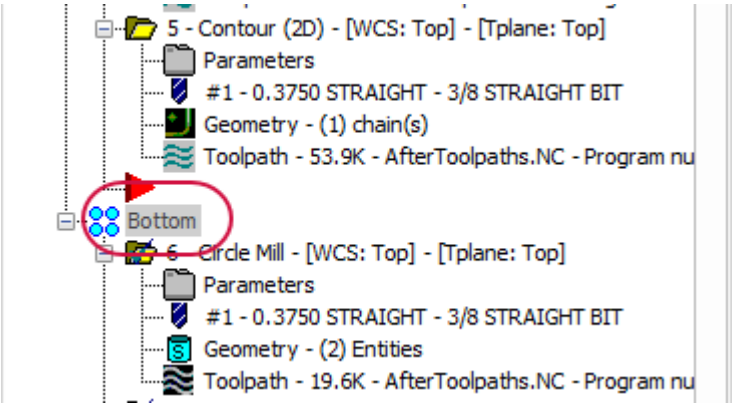
6. Release the mouse and select **Copy after** from the pop-up menu.



There will now be a new **Toolpath Group-1** displayed in the Toolpaths Manager.

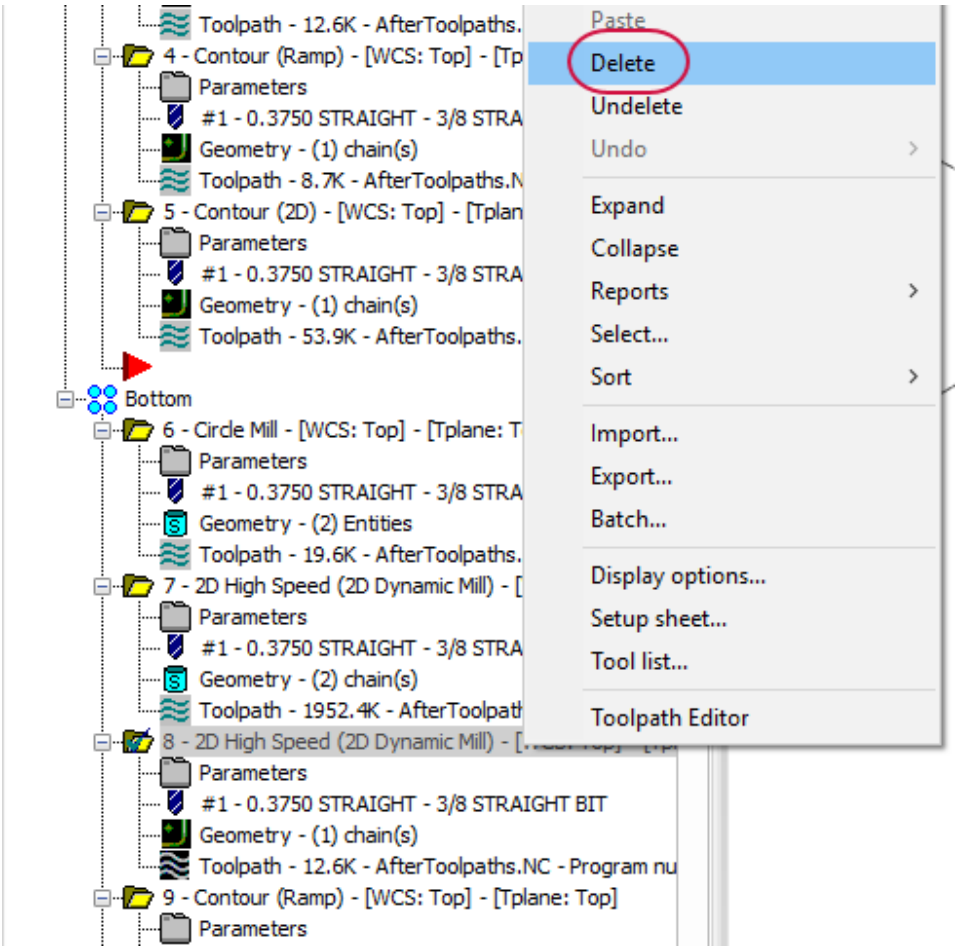


7. Rename **Toolpath Group-1** to **Bottom**.

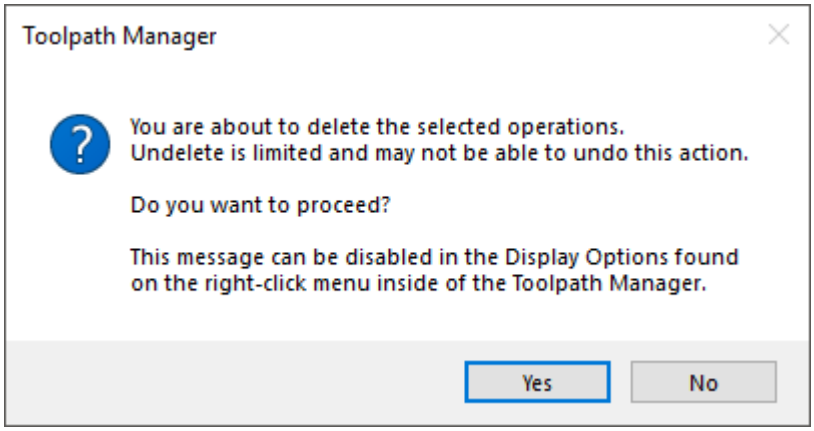


8. Select **8 - 2D High Speed (2D Dynamic Mill)** in the Toolpaths Manager.

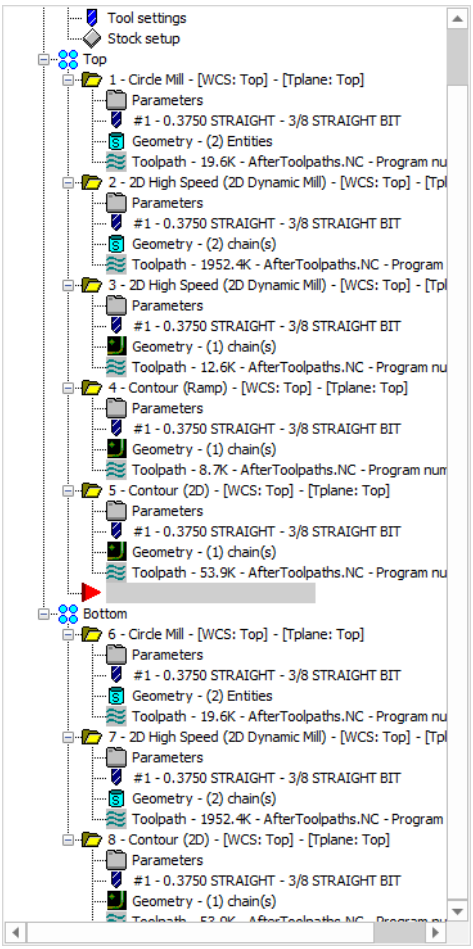
9. Right-click **8 - 2D High Speed (2D Dynamic Mill)** and select **Delete** from the menu. This toolpath is used to machine the phone slot.



10. A warning dialog box displays. Click **Yes** to proceed with the deletion.



11. Select and delete **8 - Contour (Ramp)** toolpath. The Toolpaths Manager should match the image shown below.

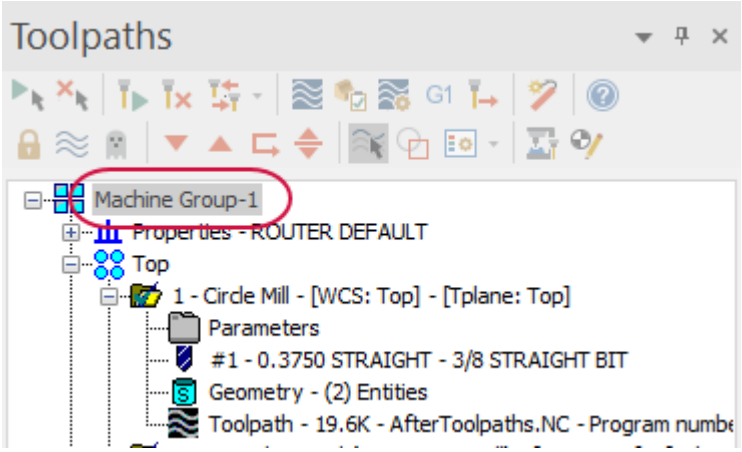


12. Save your file.

Exercise 2: Translating the Solid Model

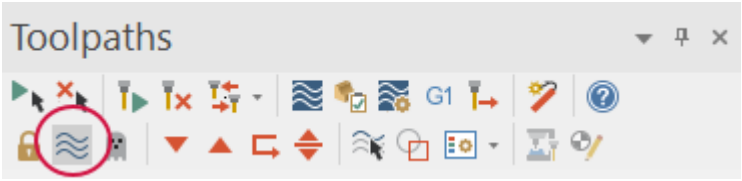
In this exercise, you will use **Translate** to duplicate the solid model and move it to a specific location.

1. Your file Amplifier-XXX which you saved from the previous exercise should be open in Mastercam.
2. In the Toolpaths Manager, select **Machine Group - 1**.

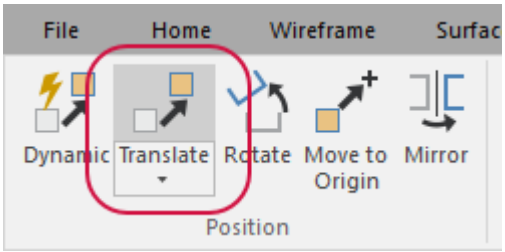


This selects both the **Top** and **Bottom** toolpath groups.

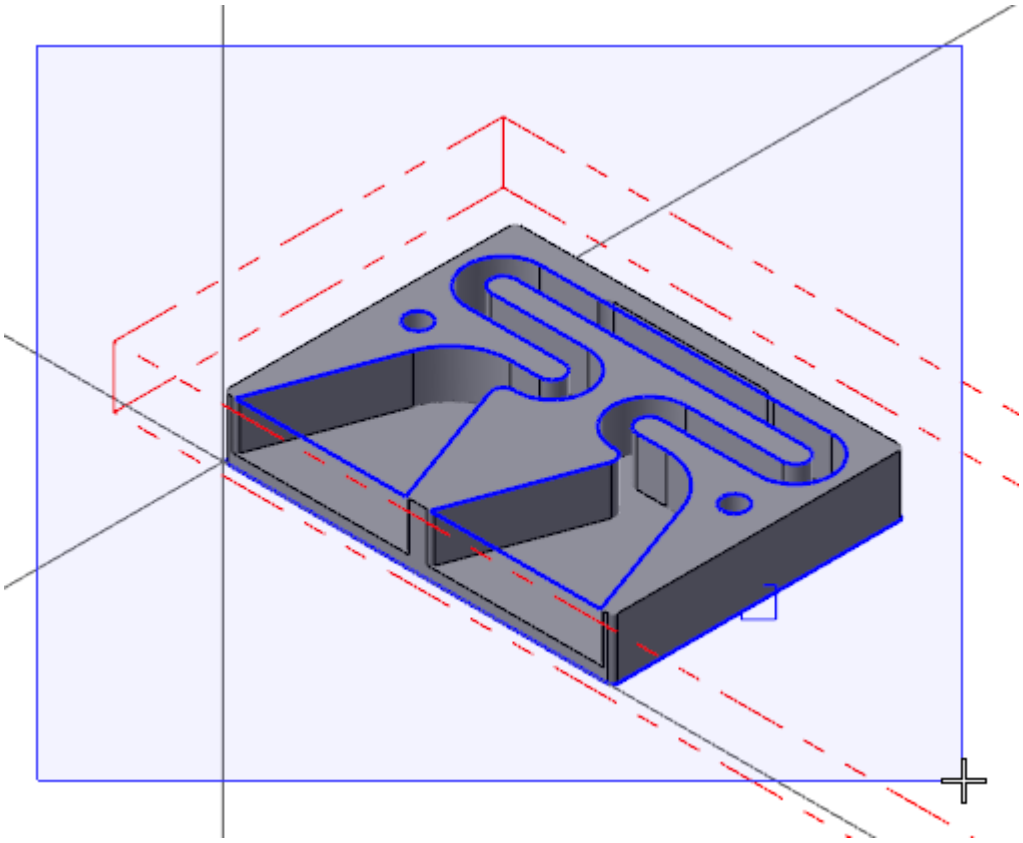
3. Select **Toggle display on selected toolpaths operations** to hide the toolpaths in the graphics window.



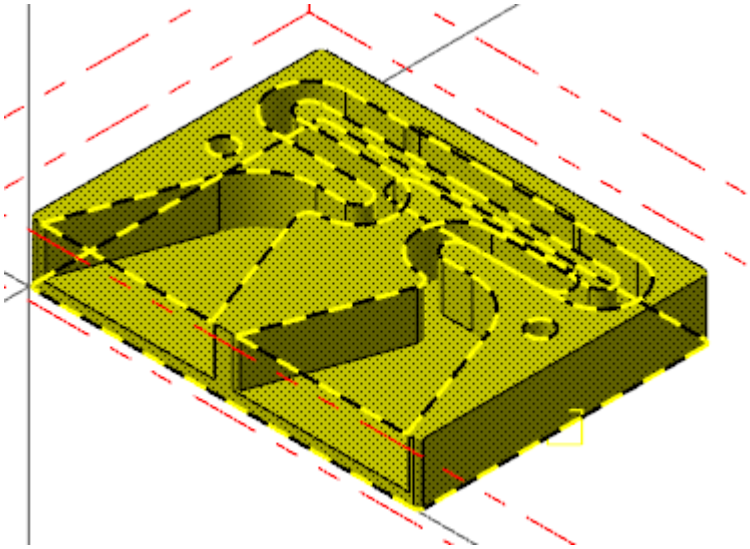
4. Select **Translate** on the **Transform** tab.



5. Left-click in the graphics window and drag your mouse over the entire solid model and wireframe.



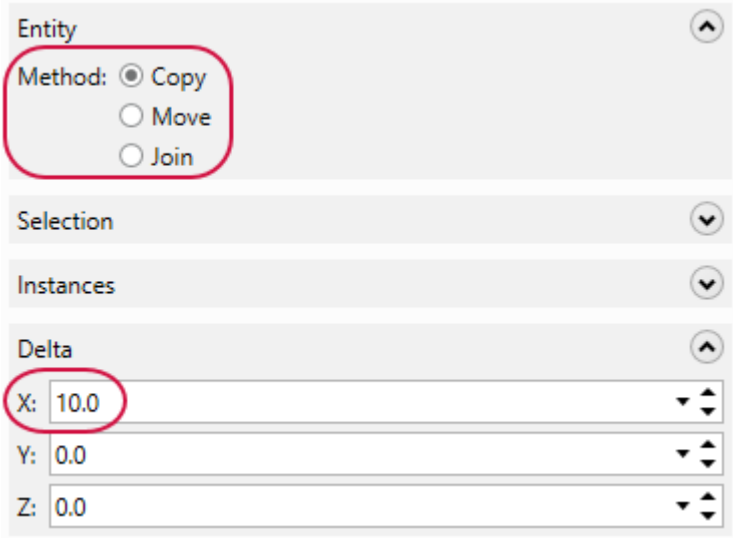
6. Release the mouse to select the geometry.



7. Press [**Enter**].

   The **Translate** function panel is now active.

8. Ensure that **Method** is set to **Copy** and enter **10.0** for **Delta X**.

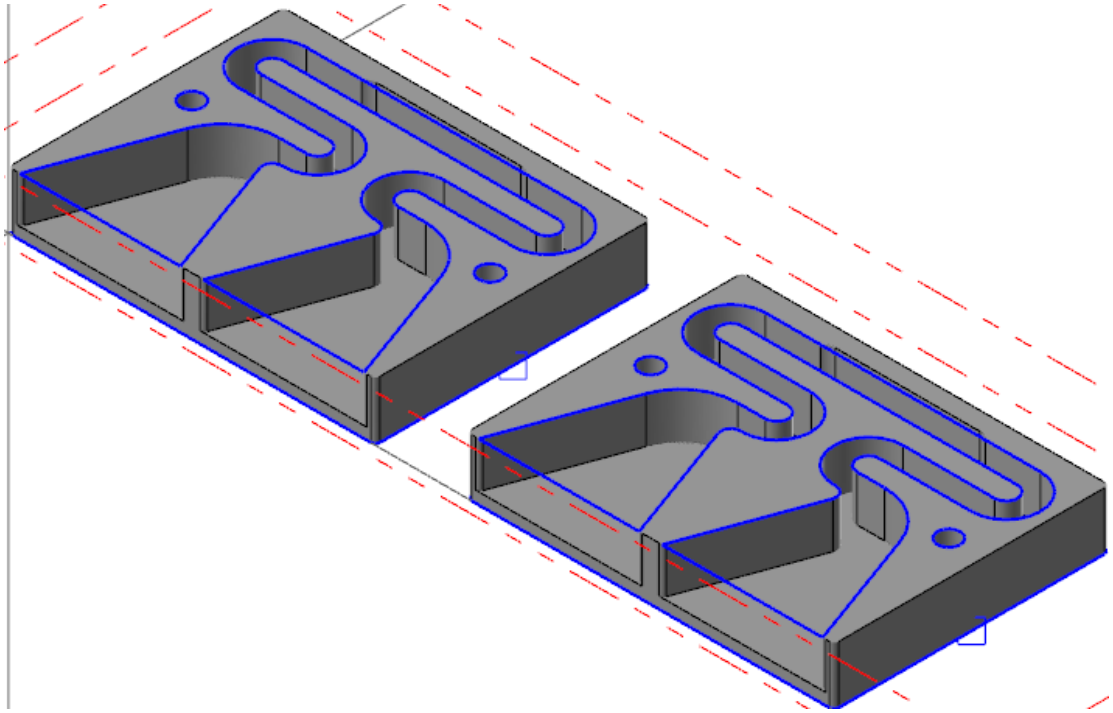


9. Click **OK** in the **Translate** function panel.

   The part is copied and pasted in the location you entered.

10. Right-click in the graphics window and select **Clear Colors**.

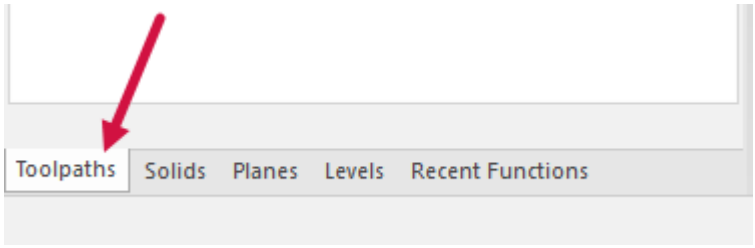
11. Save your file.



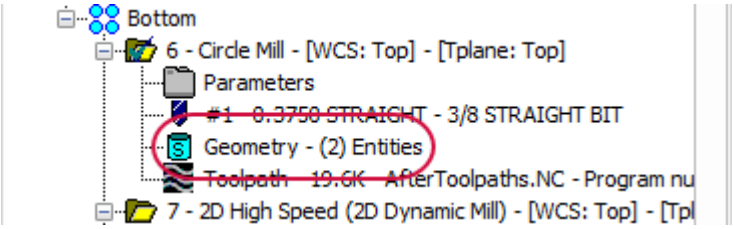
Exercise 3: Re-chaining the Toolpaths

In this exercise, you will re-chain the toolpaths in the **Bottom** toolpath group from the original part to the translated part. The copied toolpaths are still referencing geometry from the original solid model. The toolpaths need to be updated to use the geometry on the copied solid model instead.

1. Your file Amplifier-XXX which you saved from the previous exercise should be open in Mastercam.
2. If necessary, select the **Toolpaths** tab to bring the Toolpaths Manager to the forefront.

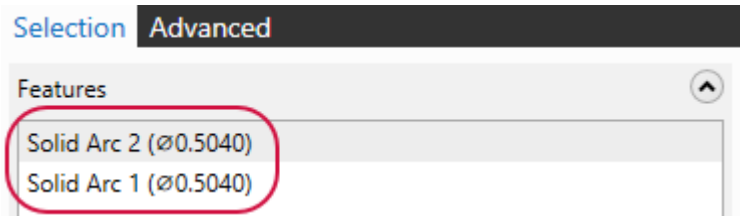


3. In the **Bottom** toolpath group, select **Geometry** under **6 - Circle Mill** in the Toolpaths Manager.

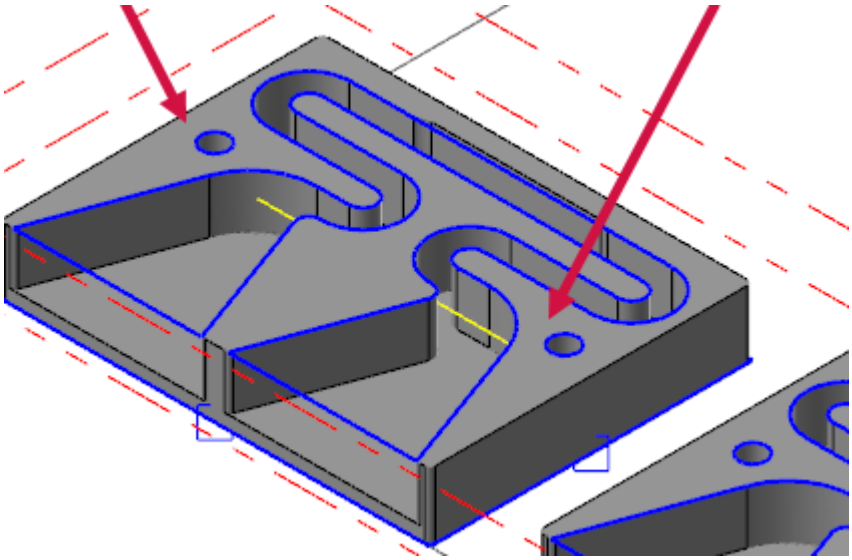


The **Toolpath Hole Definition** function panel displays.

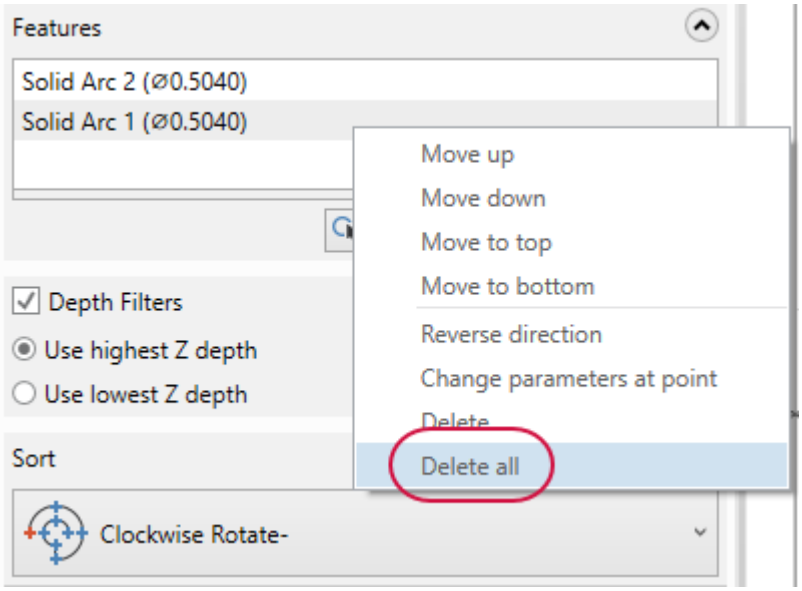
4. Select a solid arc in the **Features** list.



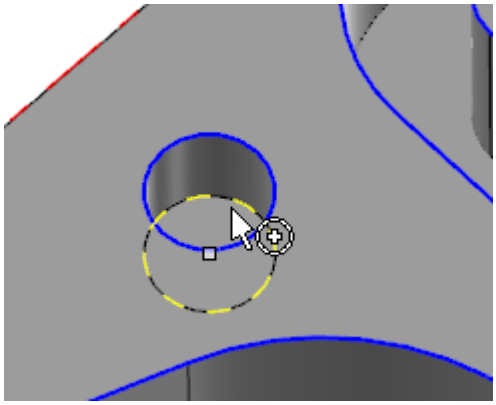
You can see that the toolpath is referencing geometry from the original solid model.



5. Right-click in the **Features** list and select **Delete all** from the menu.



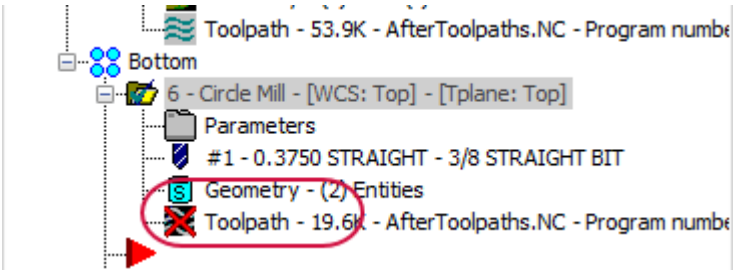
6. Select the bottom arc of the two index holes on the new solid model that was created in the previous exercise.



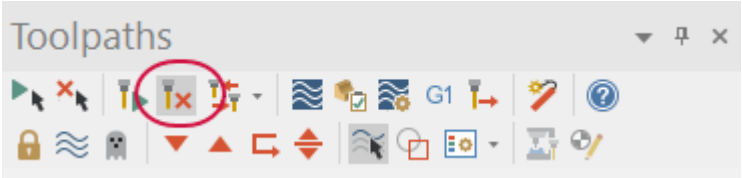
You may need to zoom in and rotate the part to select the correct geometry.

7. Click **OK** in the **Toolpath Hole Definition** function panel.

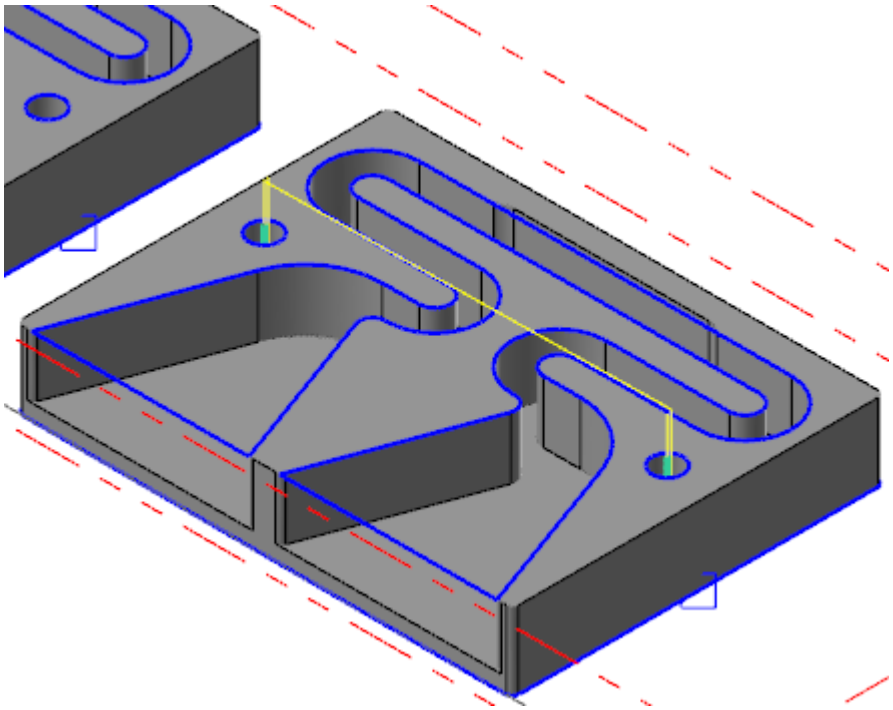
The Circle Mill toolpath is now marked dirty in the Toolpaths Manager.



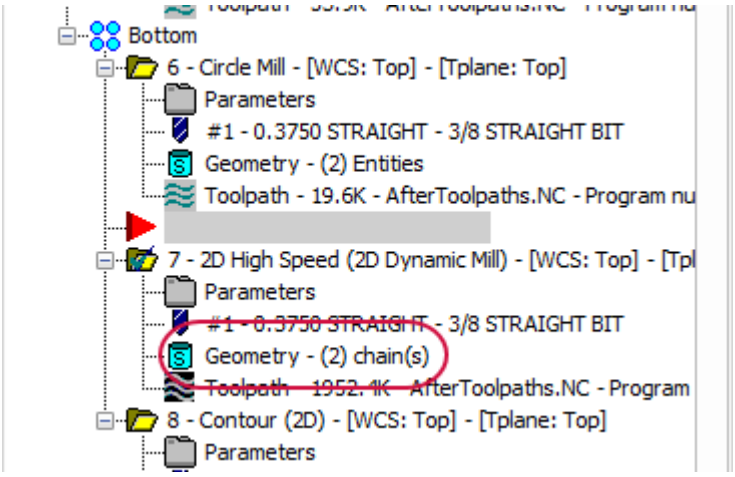
8. Select **Regenerate all dirty toolpaths** in the Toolpaths Manager toolbar.



The Circle Mill toolpath regenerates and displays as shown below on the translated solid model.

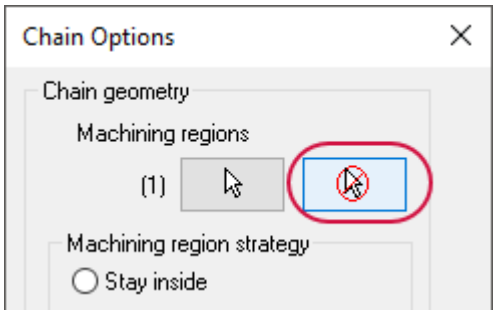


9. Select **Geometry** under **7 - 2D High Speed (2D Dynamic Mill)** in the Toolpaths Manager.



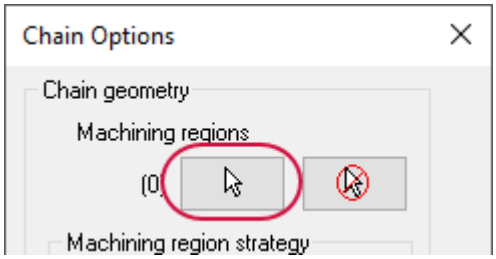
The **Chain Options** dialog box displays.

10. Click **Remove selected machining chains** under **Machining regions**.



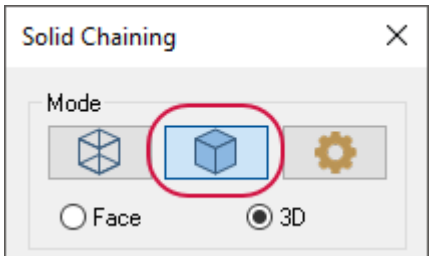
The selected machining region is removed.

11. Click **Select machining chains** to return to the graphics window to select the machining regions.

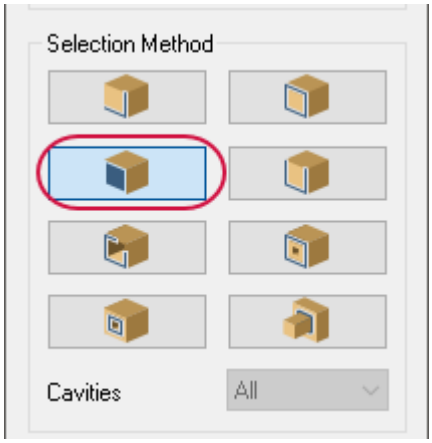


The **Chaining** dialog box displays.

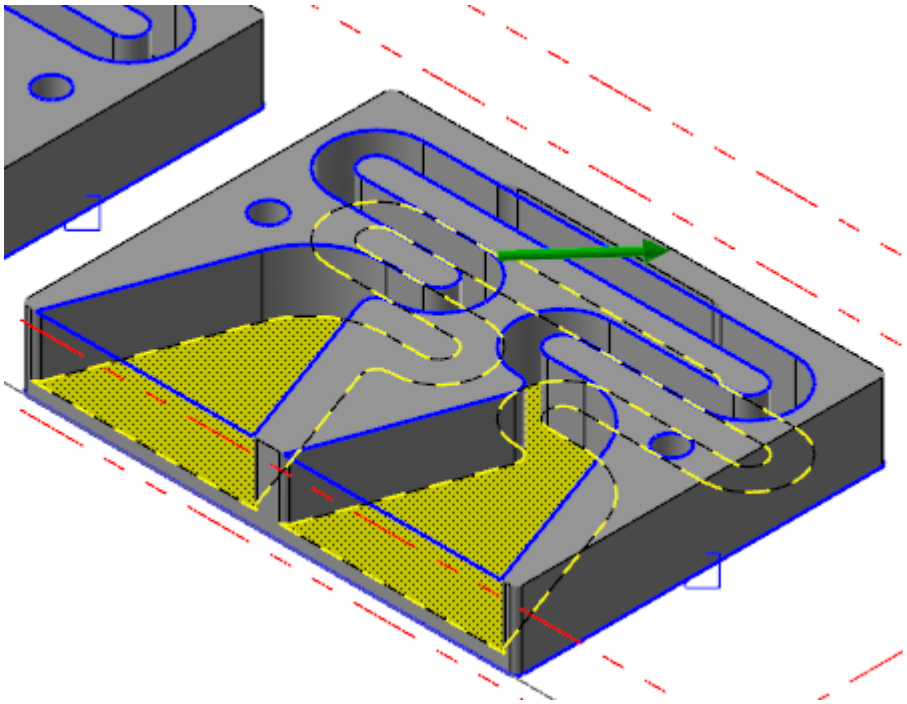
12. If necessary, select **Solids** to set the chaining mode to solids.



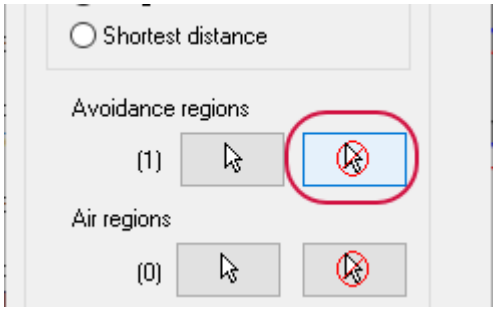
13. Ensure that the **Selection Method** is set only to **Face**. Click any active selection method button to remove that selection filter.



14. Select the bottom solid face of the channel on the translated solid model as shown below.

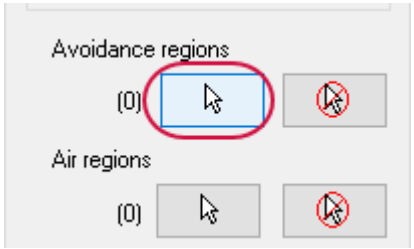


15. Click **OK** in the **Chaining** dialog box to return to the **Chain Options** dialog box.
16. Click **Remove selected avoidance chains** under **Avoidance regions**.



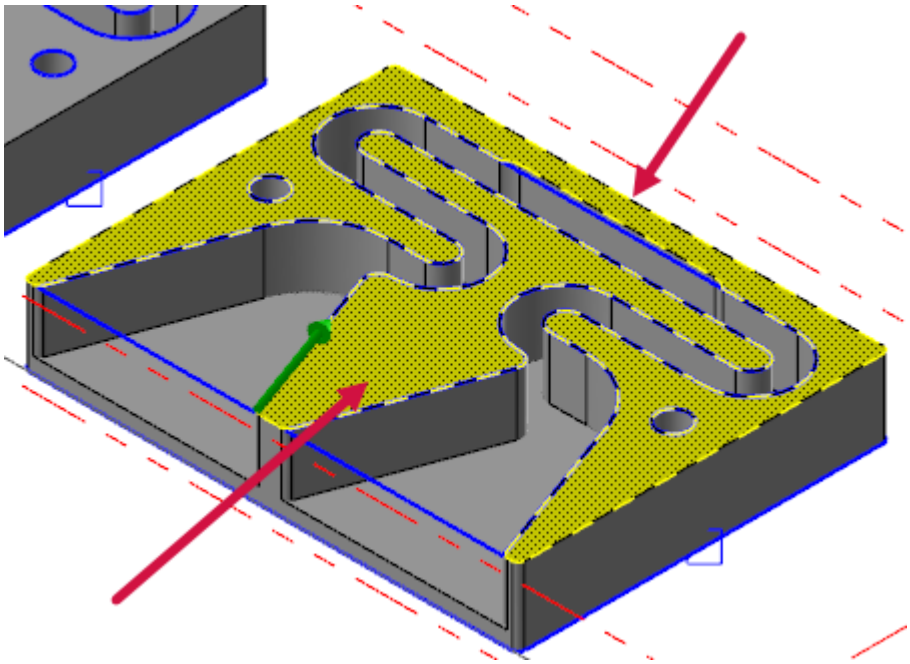
The selected machining region is removed.

17. Click **Select avoidance chains** to return to the graphics window to select the avoidance regions.

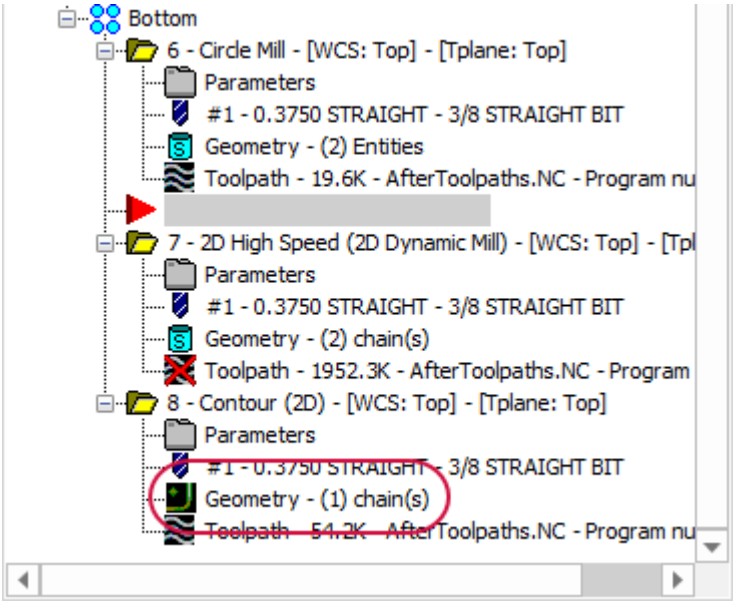


The **Chaining** dialog box displays.

18. Select the two top solid faces on the translated solid model as shown below.

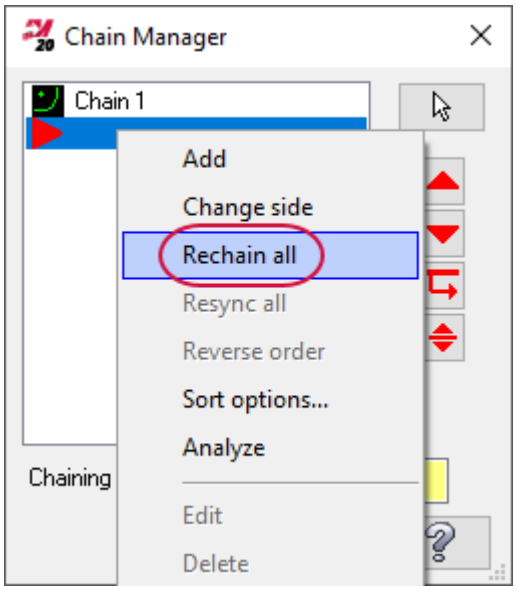


19. Click **OK** in the **Chaining** dialog box, and then click **OK** again in the **Chain Options** dialog box.
20. Select **Geometry** under **8 - Contour (2D)** in the Toolpaths Manager.



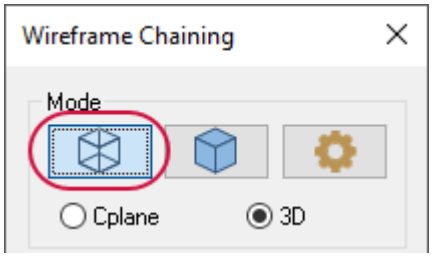
The **Chain Manager** dialog box displays.

21. Right-click in the **Chain** list and select **Rechain all** from the menu.

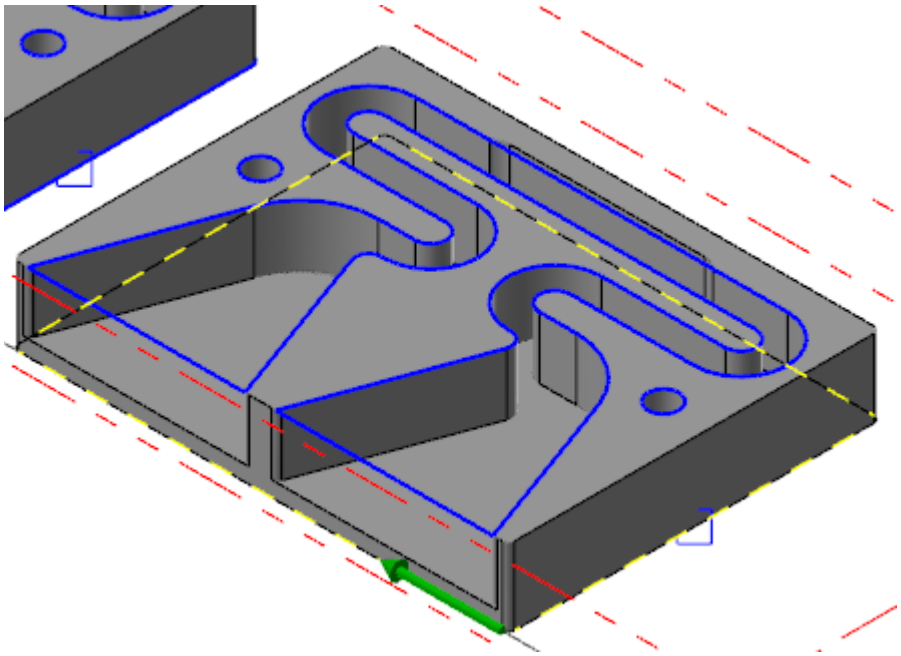


The **Chaining** dialog box displays.

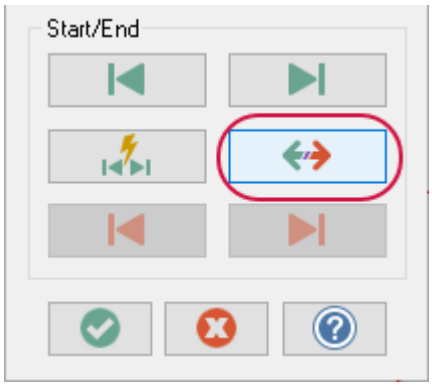
22. Select **Wireframe** to set the chaining mode to wireframe.



23. Select the wireframe rectangle as shown below.

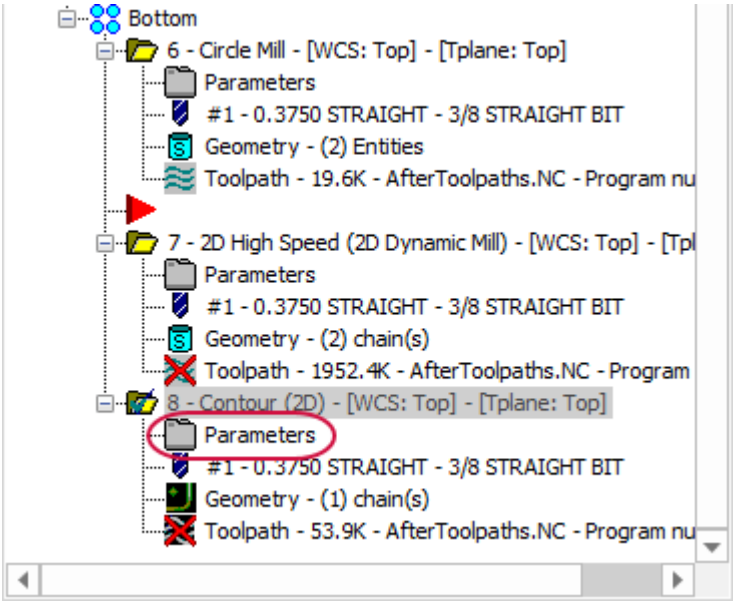


Ensure that your chain direction (the green arrow) matches the chain direction above. If not, use the **Reverse** button on the **Chaining** dialog box to change the direction of the chain.



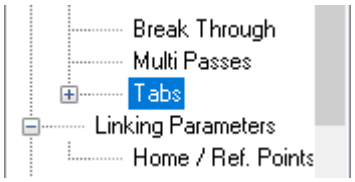
24. Click **OK** in the **Chaining** dialog box. Then, click **OK** in the **Chain Manager** dialog box.

25. Select **Parameters** under **8 - Contour (2D)** in the Toolpaths Manager.

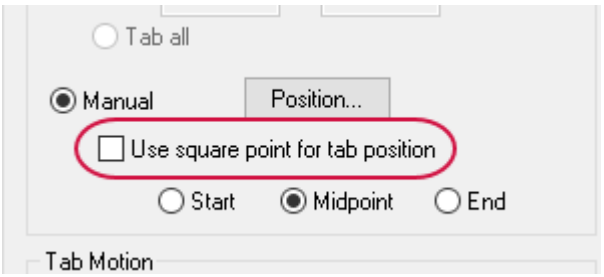


The **2D Toolpaths - Contour** dialog box displays.

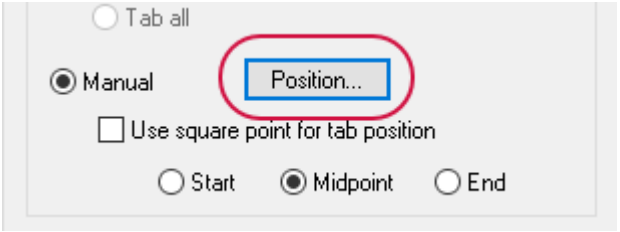
26. Select the **Tabs** page.



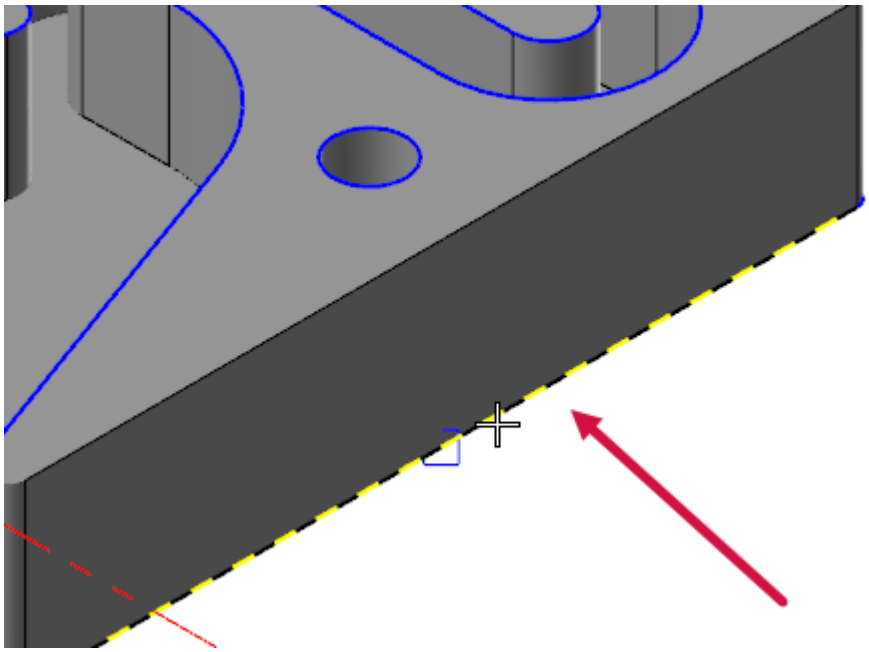
27. Deselect **Use square point for tab position**.



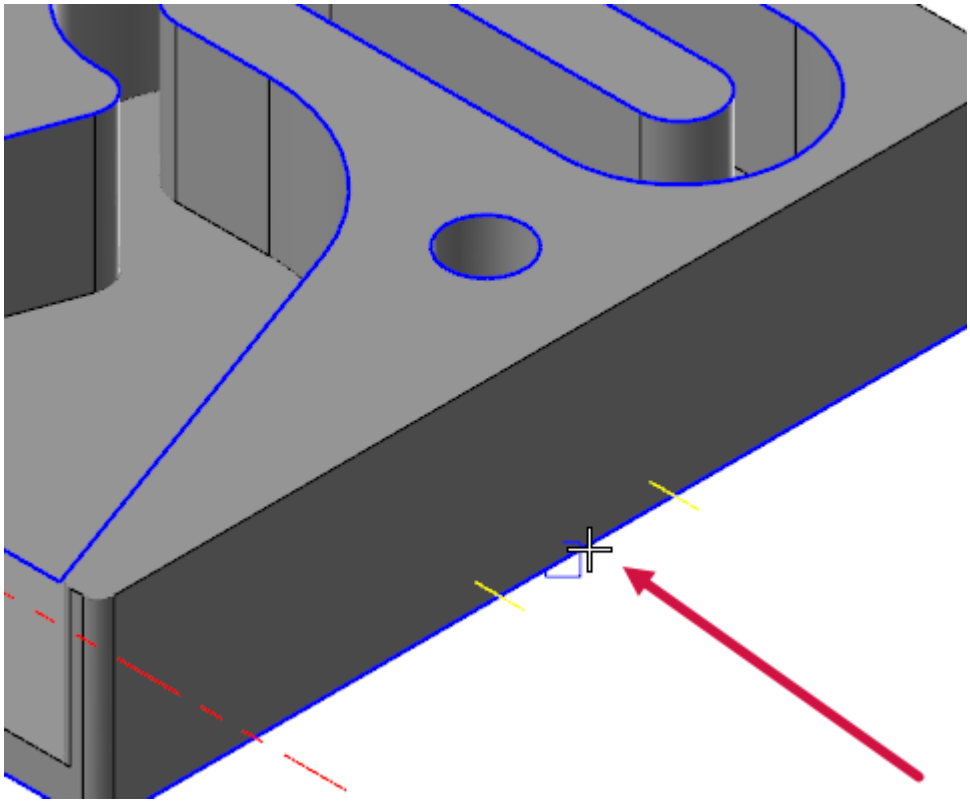
28. Select **Position** to return to the graphics window to select the tab positions.



29. Select the wireframe on the translated part as shown below.



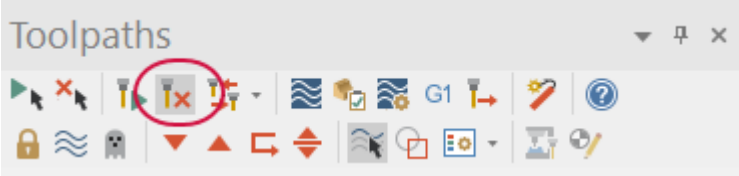
30. Place the tab by left-clicking at the midpoint of the line.



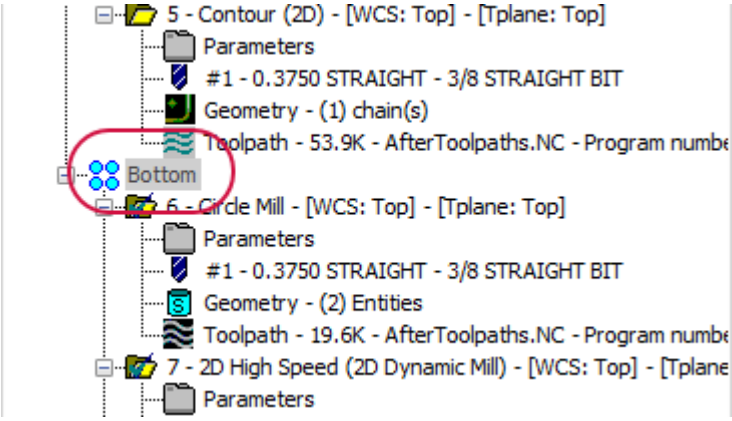
> **NOTE**
> When positioning the tabs, they will not "snap" to the square point created in the previous chapter. You need to manually place the tab.

31. Repeat the previous two steps with the other wireframe lines around the translated part. You will have three tabs when finished.

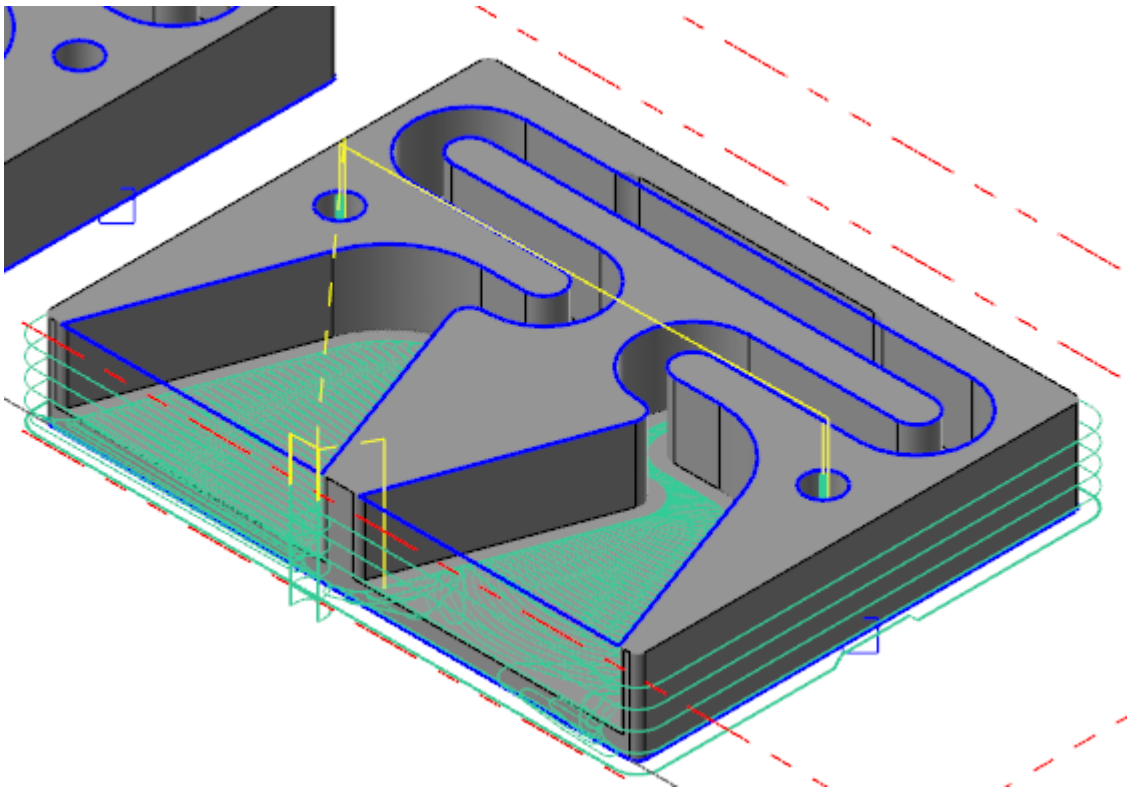
32. Press [**Enter**] after you have placed the tabs to return to the **2D Toolpaths - Contour** dialog box.
33. Click **OK** in the **2D Toolpaths - Contour** dialog box.
34. Select **Regenerate all dirty toolpaths** in the Toolpaths Manager toolbar.



35. In the Toolpaths Manager, select the **Bottom** toolpath group.



The part displays with the updated toolpaths.

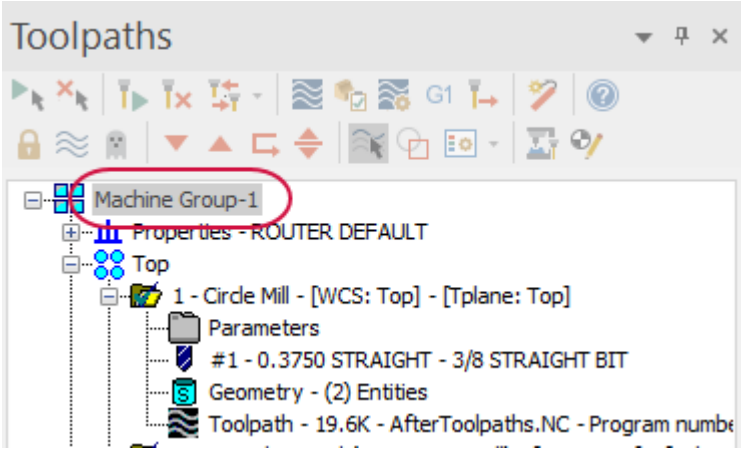


36. Save your file.

Exercise 4: Verifying All Toolpaths

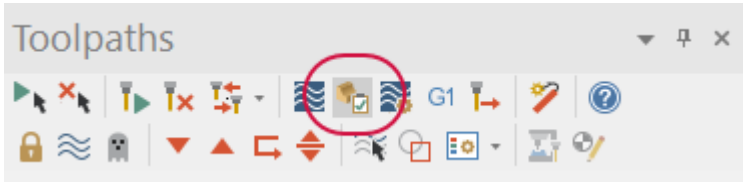
In this exercise, you will verify all of the toolpaths created in previous exercises and chapters.

1. Your file Amplifier-XXX which you saved from the previous exercise should be open in Mastercam.
2. In the Toolpaths Manager, select **Machine Group - 1**.

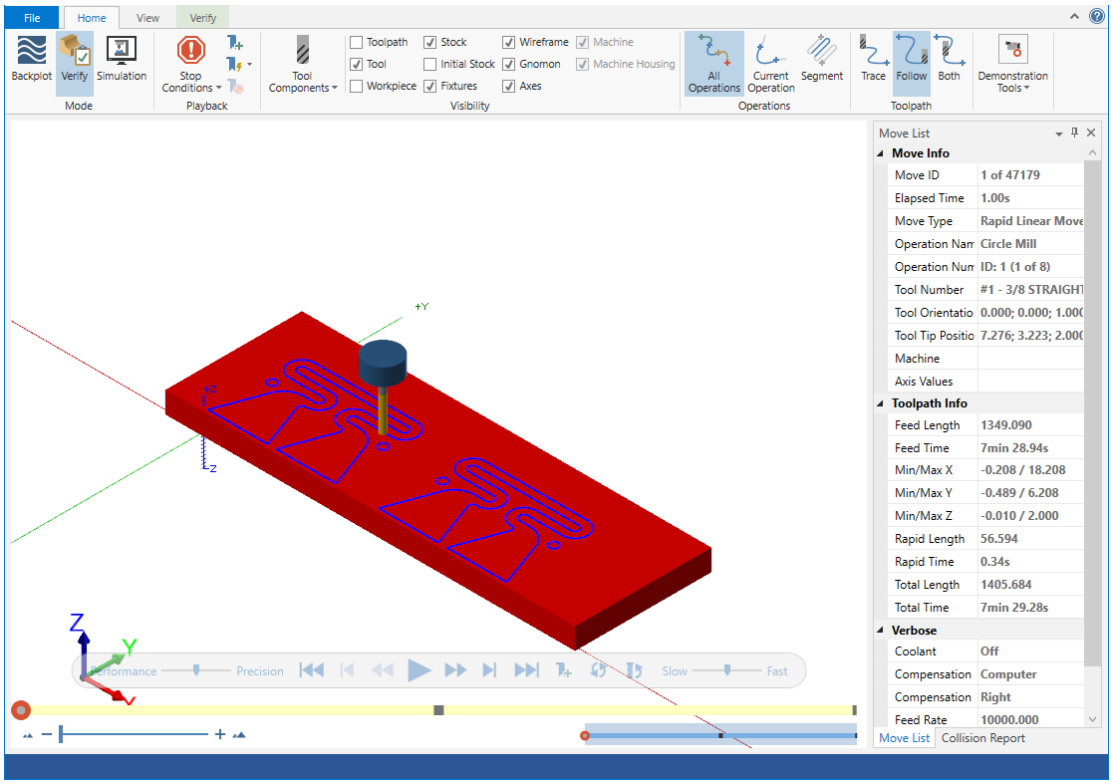


This selects both the **Top** and **Bottom** toolpath groups.

3. Select **Verify selected operations** from the Toolpaths Manager toolbar.

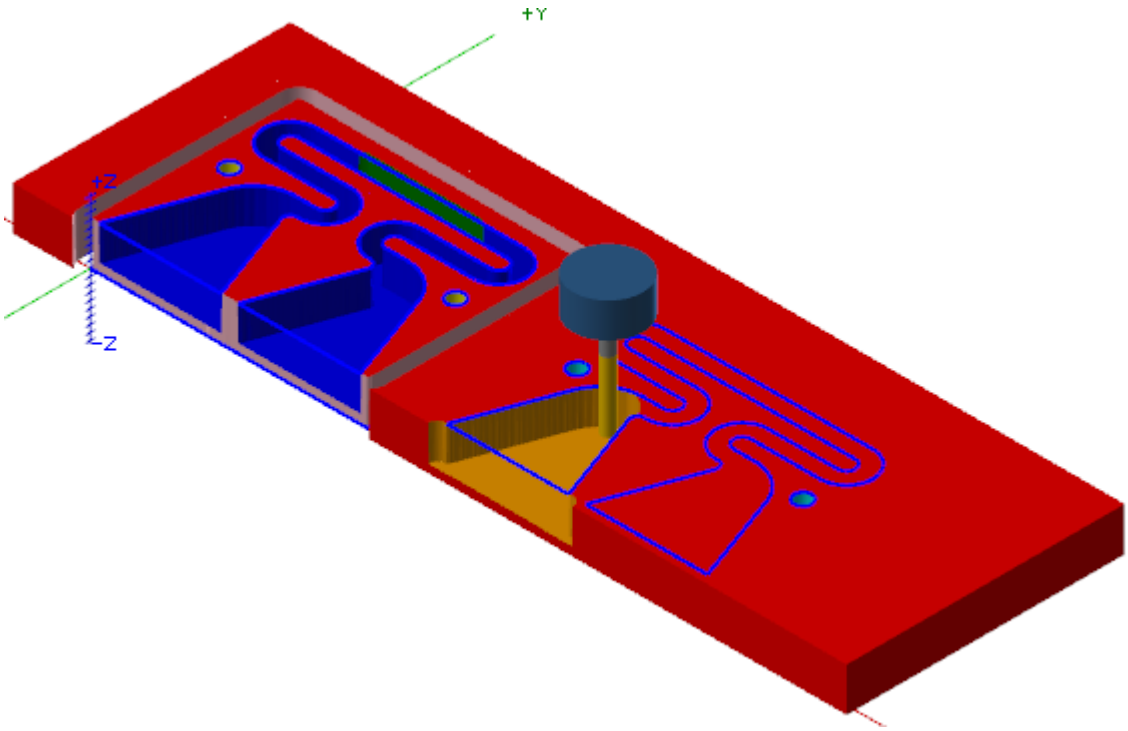


Mastercam Simulator displays.

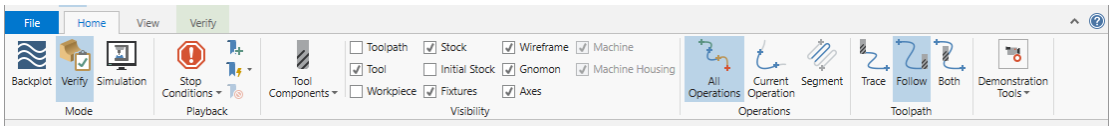


4. Use the playback bar to run the verification.

The verification shows the machining of both parts.



5. Experiment with the available options on the **Home**, **View**, and **Verify** tabs of Mastercam Simulator.



6. Exit Mastercam Simulator once you have finished viewing the verification.
7. Save your file.

SHOW US WHAT YOU LEARNED!

Can you answer these questions?

1. If a toolpath needs to be edited after creation, it must be deleted and re-created.
    a. True
    b. False
2. The Translate function will only move, copy, or join wireframe geometry.
    a. True
    b. False
3. What is the benefit of having multiple toolpath groups in one machine group?
4. Describe a method you can employ to reuse a toolpath.

CHAPTER 7

FIXTURING THE STOCK

Selection of the proper fixture is important to successful machining. The choice of fixture involves consideration of the size of the piece you are machining, and the ability to machine around the exterior and the interior of the piece successfully, while avoiding the fixture itself.

For this tutorial, we recommend that you use a vacuum table. Additionally, the tabs created in the previous chapters will help to hold the part in place. However, if your machining environment does not include a vacuum table, this chapter will also briefly discuss other means of holding down parts when machining using a router.

Goals

- Explore and learn about different ways to hold down the stock.

Fixture options

- **Vacuum table.** A vacuum table is a system that uses air pressure to hold the stock and workpieces to the router, keeping them still during machining. (recommended)

  A vacuum table will allow full access to all sides of your part, except the side that is in contact with the table's surface. However, the security of the part is relative to the amount of surface area that is in contact with the vacuum. In other words, the smaller the part, the more difficult it is for a vacuum table to hold it in place.

- **Screwing down the stock.** You can use screws to hold down the stock. This option requires a spoilboard. The location of the screws need to be considered when machining.

  A spoilboard is a disposable work surface mounted atop the router’s permanent table.

- **Double-sided tape**. You can use double-sided tape to keep the stock connected to the router. If you choose this option, we recommend that you use double-sided carpet tape.

- **Low profile clamps**. You can use low profile clamps to hold down the stock. Low profile clamps that take up very little space, which is ideal for a router. The location of the clamps need to be considered when machining.

SHOW US WHAT YOU LEARNED!

Can you answer these questions?

1. What should be considered if you are using a vacuum table to hold a very small part?
2. When preparing a part to be machined with a router, what is the main benefit of using a vacuum table over hold-down clamps?

CHAPTER 8

POST PROCESSING

Post processing, or posting, refers to the process by which the toolpaths in your part file are converted to a format that can be understood by your machine tool's control (for example, G-codes). A program called a post processor, or post, reads your part file and writes the appropriate NC code. Generally, every machine tool or control requires its own post processor, customized to produce code formatted to meet its exact requirements.

In this chapter you will explore post processing using the default machine chosen for this project. If you use a different machine, it will require a different post.

Goals

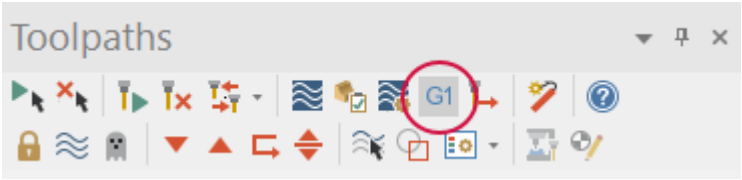
- Understand post processing concepts
- Explore Code Expert and the NC code

Exercise 1: Post Processing the Operations

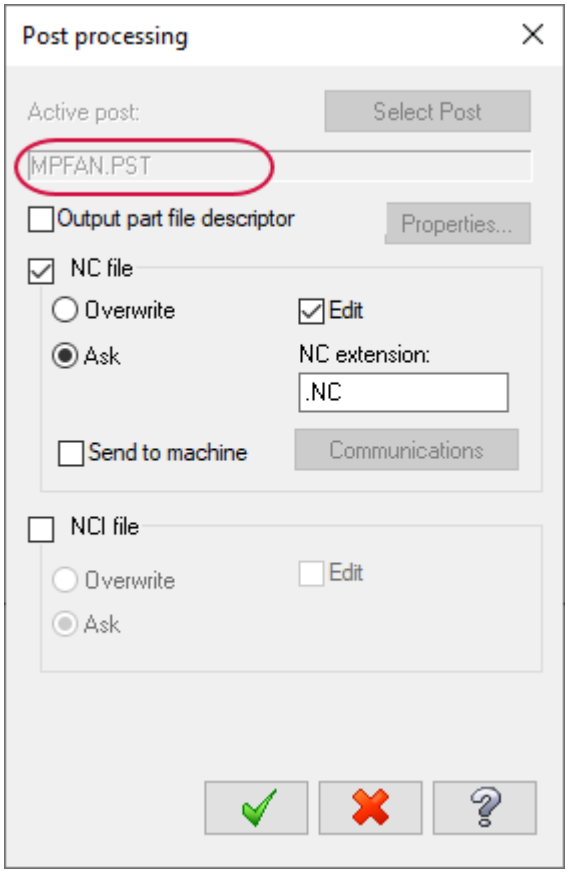
In this exercise, you will send the completed file to post processor. The post processor creates a machine-readable NC program from the Mastercam file.

1. Your file, Amplifier-XXX, which you saved from the previous lesson should be open in Mastercam.
2. Verify that all of the toolpaths are selected in Toolpaths Manager.

3. Click **G1** to post the toolpaths for the selected operations. The post processor creates a machine-readable NC program from the Mastercam file.

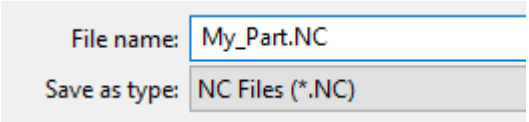


The **Post processing** dialog box displays. The name of the post processor for the machine you have chosen displays as the **Active post** field in read-only mode.

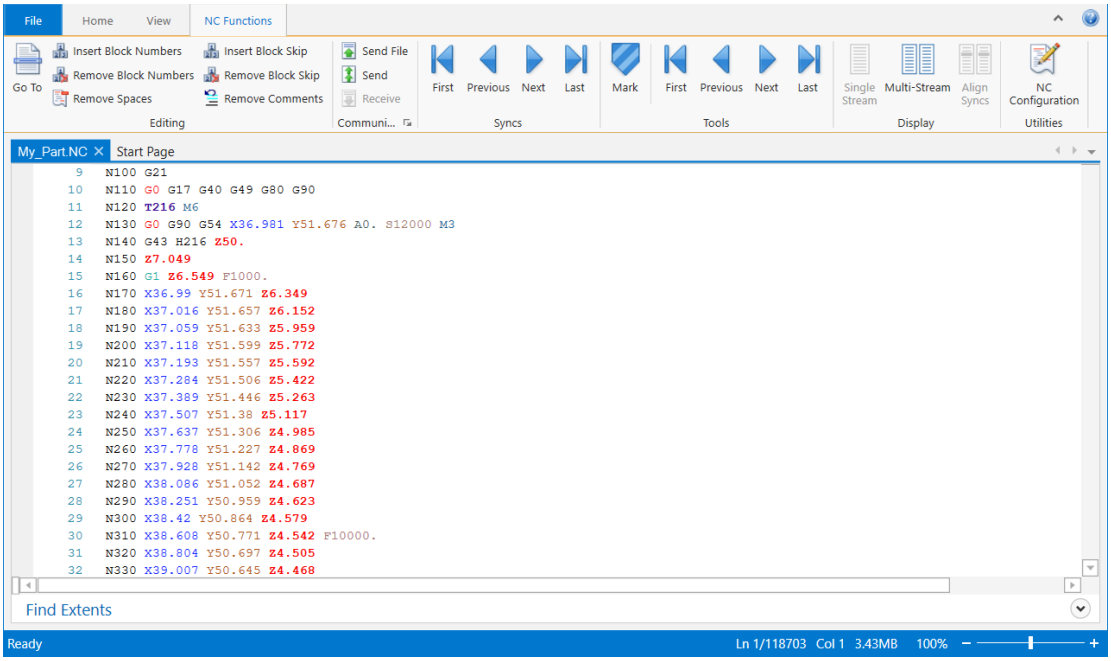


4. Click **OK**. The **Save As** dialog box opens.

5. Name your NC file and click **Save**.



The file opens in Mastercam Code Expert.



6. Use the Code Expert controls to explore the NC code.

> **CAUTION**
> **Customizing the post file is an advanced task that should only be attempted by knowledgeable users.** Programs created with incorrect posts can cause your machine tool to crash or behave in unpredictable ways.

> **NOTE**
> Click the help icon in the upper corner of Mastercam Code Expert to learn more about the program

SHOW US WHAT YOU LEARNED!

Can you answer these questions?

1. The post processor creates a machine-readable NC program from the Mastercam file.
    a. True
    b. False
2. You must post all of the operations in the Toolpaths Manager.
    a. True
    b. False
3. Posted NC programs consist mainly of X, Y, and Z values that represent point locations in the CNC machine.
    a. True
    b. False
4. Describe how you transfer the information in your Mastercam file to your machine?

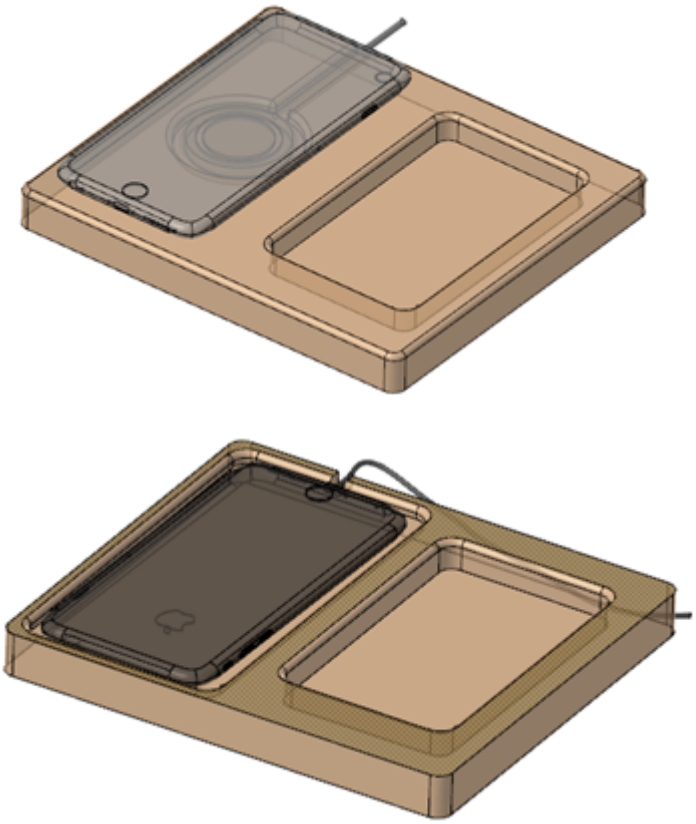
CHAPTER 9

CHALLENGE: SELF-GUIDED PROJECT

Now that you have a basic understanding of designing and programming in Mastercam, it is time to design and customize a part of your own. For this self-guided project, you will design a phone charging station in Mastercam.

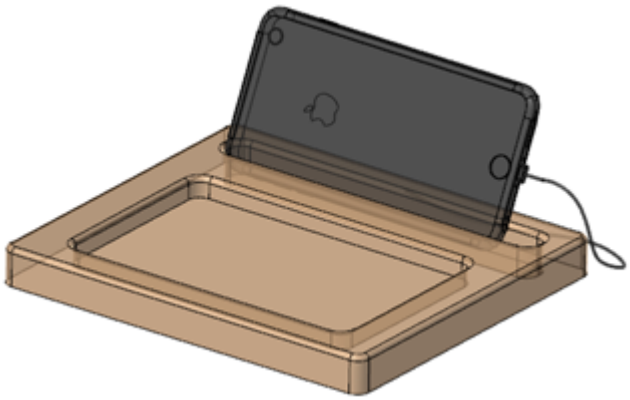
Suggestion 1: Charging Station

- Design your phone charging station to include a recess for a wireless charging pad, a slot for the charging cable and an area to hold money or keys.
- Consider the dimensions of your phone, and the location of the charging connection on the phone.



Suggestion 2: Charging Station with Viewing Mount

- Design your charging station with a slot to hold the phone to watch movies or videos.
- Consider the dimensions of your phone, and the depth of the slot needed to hold the phone at a good viewing angle.



Successful completion of this self-guided project will demonstrate the following skills in Mastercam:

- Using Mastercam's wireframe and solid model design tools to create a machinable, solid model.
- Programming a part using a variety of 2D Router toolpaths.
- Securing stock to the machine.
- Confirming toolpath operations with Mastercam Simulator (Verify).

As part of this project, you will be expected to prepare a brief presentation. The presentation should be no longer than five minutes and clearly outline the steps that you have taken to complete the project, including which ideas you explored, what new technology you learned to facilitate the process, problems that you experienced and how you overcame them, and how you improved the project from inception through to the finished product.

This self-guided project will be evaluated against the following categories:

- Creativity – the process and ability to explore and express multiple ideas in a unique way.
- Initiative – the ability to overcome projects independently and with a positive attitude.
- Iteration – the ability to develop the product with progressive improvement over time.
- Continued Learning – the ability to attempt new techniques as part of the project.
- Presentation – the ability to articulate and explain the process and project.

Design Tips

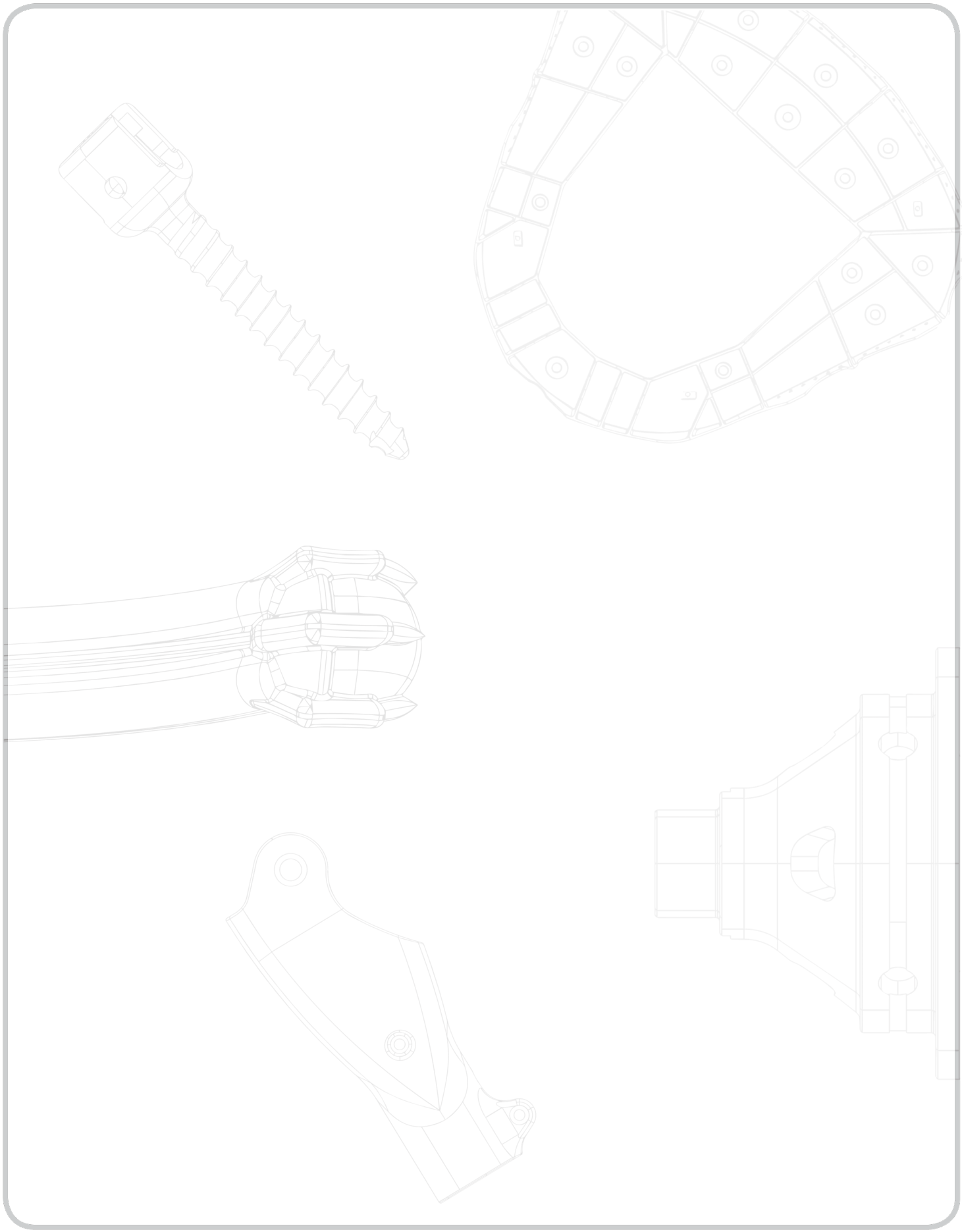
- Prove out your design using a scrap piece of wood. You can approximate feeds and speeds if the scrap piece you use is the same material as your finished product.
- Incorporate the use of tabs into your design to aid in proper fixturing of your workpiece.
- If you have access to a 3D printer, test out your design for form, fit, and function.

- If your design requires a recess that will hold your phone, begin by machining a slightly undersized recess, and gradually increase the size of the opening. Test the opening with your phone while the part is on the machine to determine the best, final size.
- Using a tool with the same radius as the internal corner can cause burn marks. Choose a tool with a slightly smaller radius to allow the tool to machine an internal corner without causing burn marks.
- Use a hand router to round corners for finishing.
- Add felt pads as feet to protect the furniture.

CONCLUSION

Congratulations! You have completed the *Quick Part Series - Acoustic Amplifier*! 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.



CNC Software, Inc.
671 Old Post Road
Tolland, CT 06084

www.mastercam.com

Mastercam® is a registered trademark of CNC Software, Inc. ©1983-2019. All rights reserved.
All other trademarks are property of their respective owners.