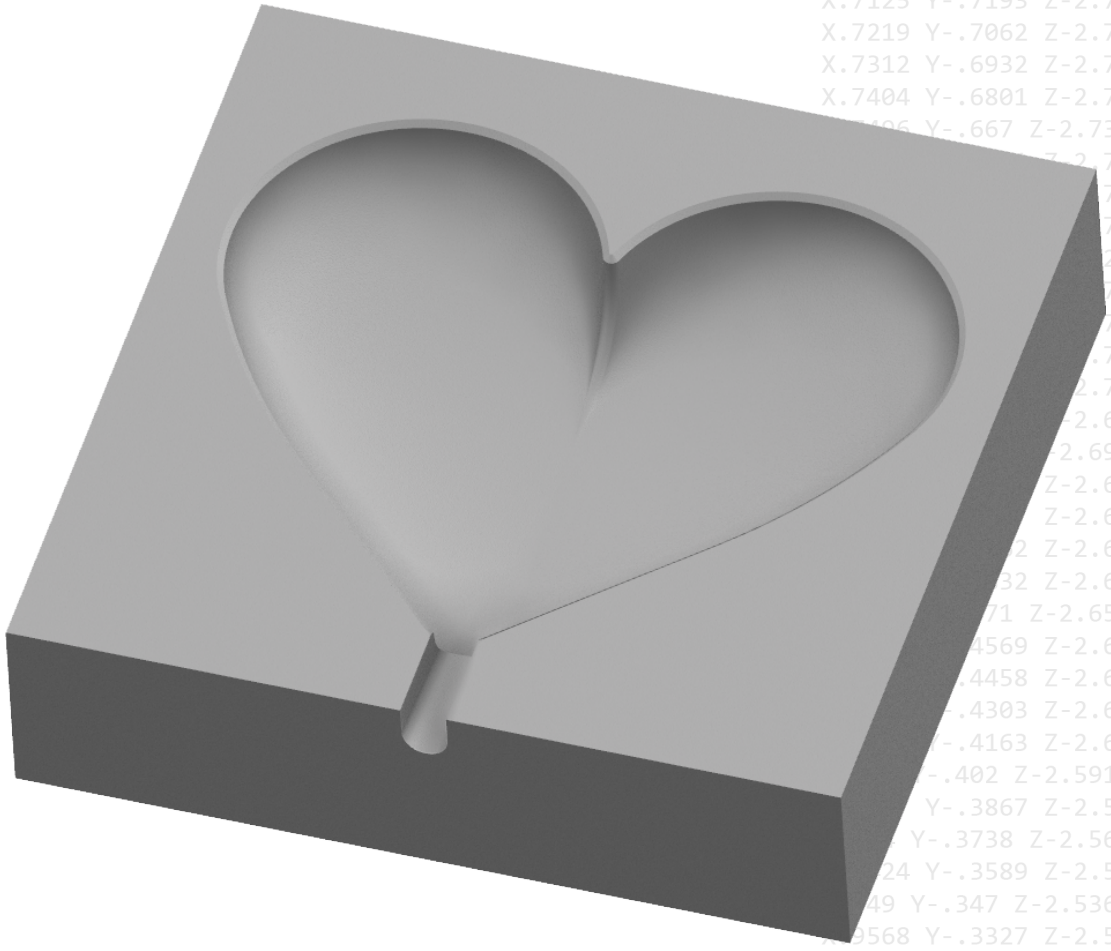
The fastest way to learn the world's most widely used CAD/CAM software.

Mastercam®

Quick Part Series

Beginner



Chocolate Mold

QUICK PART SERIES - CHOCOLATE MOLD

April 2019

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 20XX.pdf) and ReadMe file (ReadMe.PDF) – installed with each release – include the latest information about Mastercam features and enhancements.

TABLE OF CONTENTS

INTRODUCTION

Welcome to *Quick Part Series - Chocolate Mold* tutorial. In this course you will explore how to design and machine a heart-shaped candy mold. Starting with a solid heart provided with this tutorial, you will use wireframe and solid CAD tools to modify the heart to create a candy mold. 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 candy mold.

This tutorial will provide you with the basic techniques needed to create the mold, and then from there you can explore Mastercam and challenge yourself to make some other interesting products.

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 program a part using rough and finish toolpaths
- Gain knowledge on how to safely fixture the part
- Explore how to verify the toolpaths with Mastercam Simulator
- Machine the final candy mold

> **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 tutorial or your results.

General Tutorial Requirements

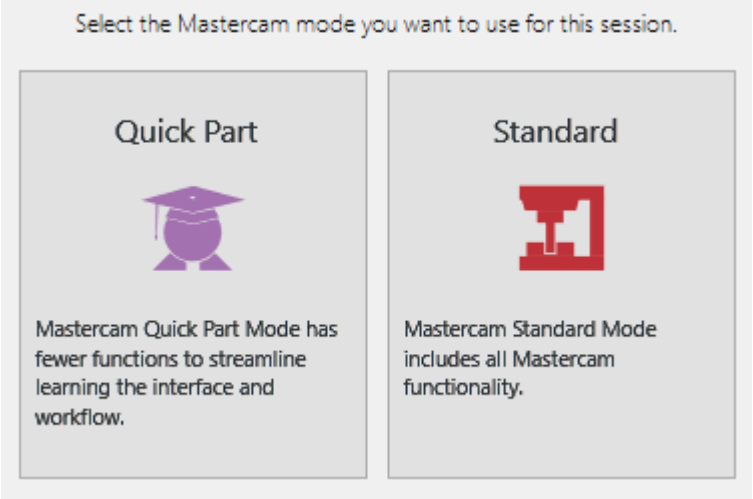
All Mastercam Project Based Learning tutorials have the following general requirements:

- You must be comfortable using the Windows® operating system.
- Each exercise in the tutorial builds on the mastery of the preceding exercise'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. If you are using an Industrial license, disregard this section.

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 complete the candy mold project using the suggested machine definition, feeds and speeds, and tooling used in this tutorial. Using different materials will affect the tooling and toolpath settings.

When calculating material requirements, consider a final mold size of 100 mm x 100 mm x 25.4 mm per student.

- Food Contact Grade Plastic UHMW Material, Polyethylene White, 1-inch thick cut into 100 mm x 100 mm x 25.4 mm pieces
- Wilton Cookie Sticks, 0.2 inch diameter x 6 inch long (5 mm diameter x 152 mm long)

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 minimum standard mill, capable of machining hard plastic, with a work envelope of at least 100 mm x 100 mm (4 inch x 4 inch), and a machining envelope that includes Z height of 100 mm (3.93 inch).
- A Machinist's Vise
- 6.0 mm Flat End Mill
- 3.0 mm Ball 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

HEART MOLD DESIGN

In this chapter you will use the design functions of Mastercam to create a heart shaped candy mold. You will start with a solid part that you will use to produce a heart-shaped impression into a block that you create using Mastercam design (CAD) tools.

Goals

- Understand levels
- Use wireframe and solid CAD tools to create geometry
- Create block for the mold

Exercise 1: Configuring Mastercam

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

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

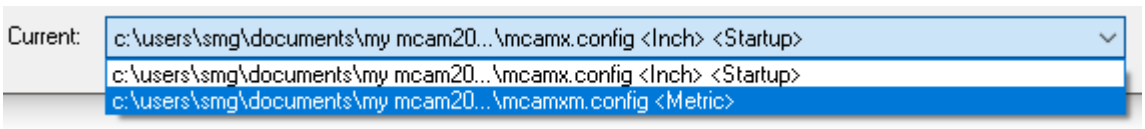
    

    OR

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

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

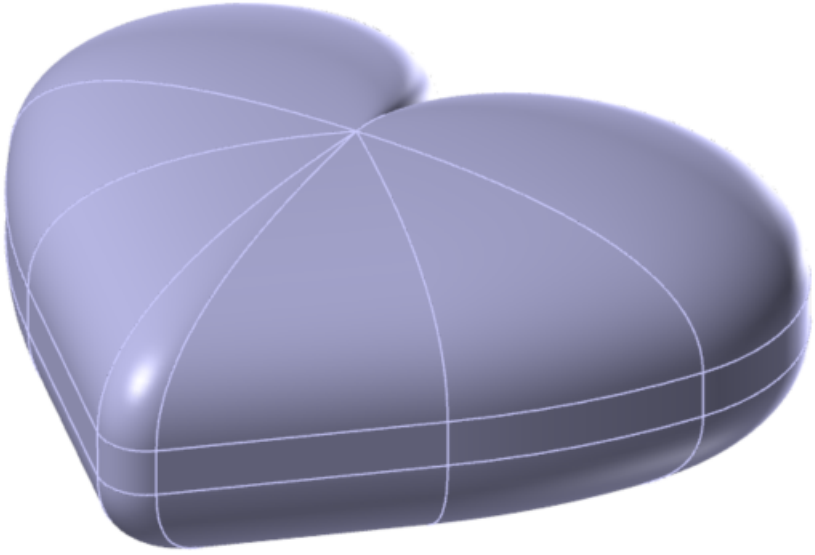
    

    d. Click **OK**.

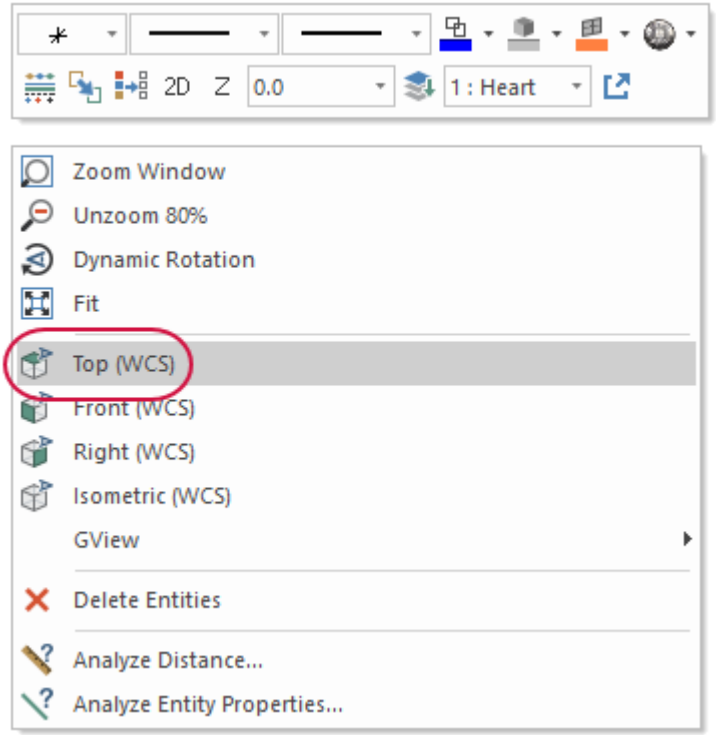
Exercise 2: Opening the File and Exploring Levels

In this exercise, you will open the `Heart mm.mcam` file provided with this tutorial and explore the Levels Manager.

1. Click **File**, **Open** and open the `Heart mm.mcam` file provided with this tutorial.

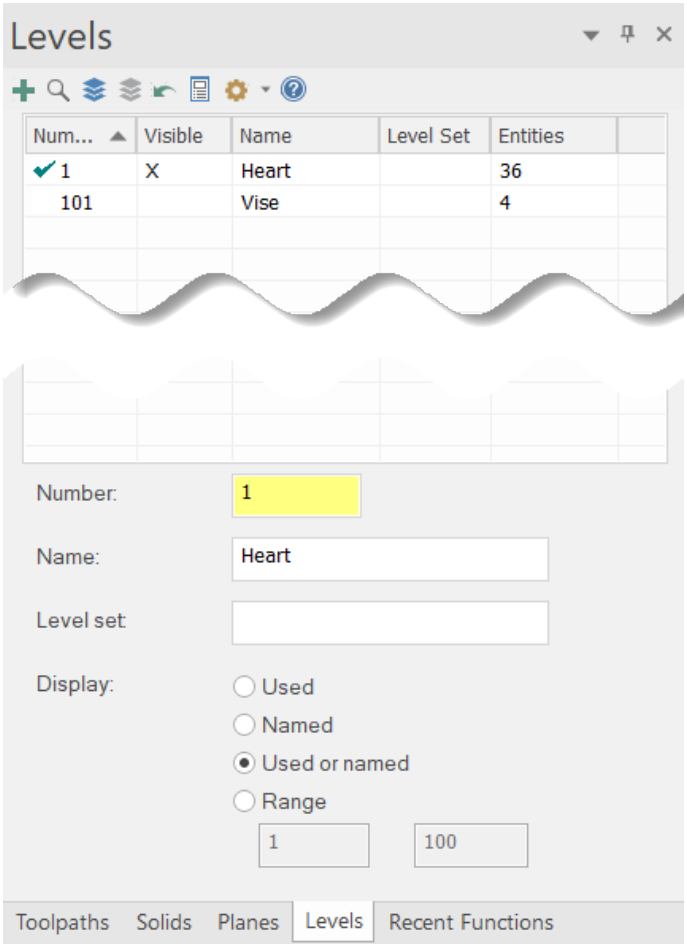


2. Save the part as `Heart mm-XXX.mcam`, where `XXX` represents your initials so that you do not overwrite the original file.
3. Right-click the heart and select **Top (WCS)**.



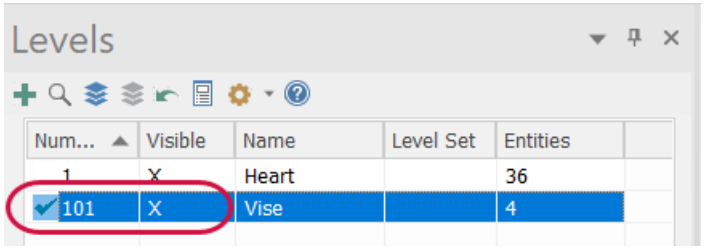
4. Right-click again and select **Fit** to bring the solid to the center of the screen.

5. Click **Levels** to open the Levels Manager. You will see that there are two levels, Level 1 which contains the heart and Level 101 which contains a sample machinist's vice.

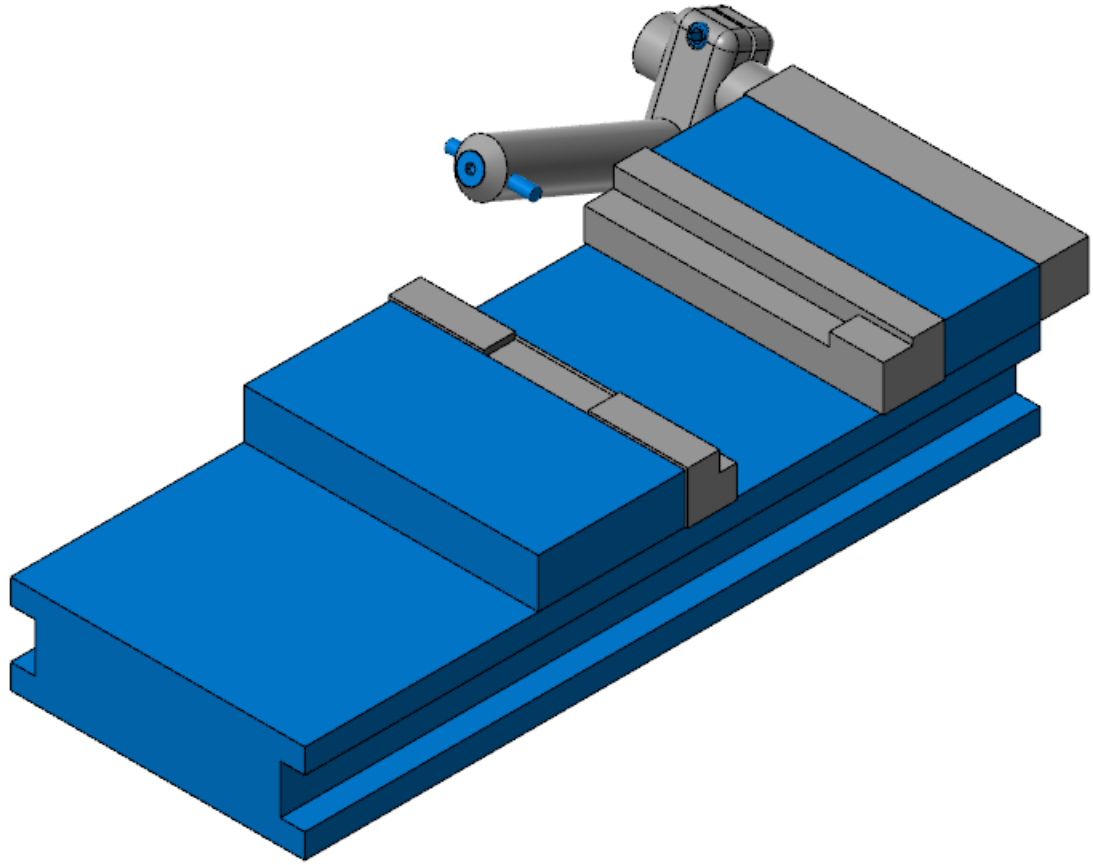


> **NOTE**
> Level 1 one is the active level as indicated by the green check in the **Number** column. The active level is always visible as indicated by the **X** in the **Visible** column.

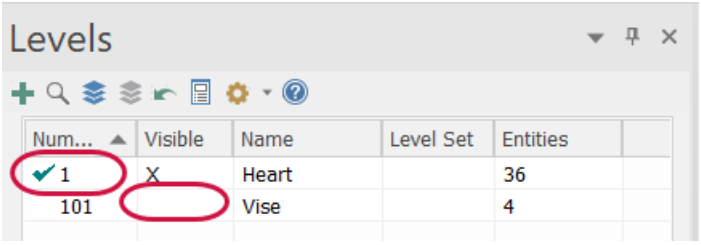
6. Click in the **Visible** column for Level 101 to make the vise show in the graphics window, and then click in the **Number** column for Level 101 to make it the active level.



7. Turn off the visibility for Level 1 by clicking in the **Visible** column to remove the **X**. Only the vise should appear in the graphics window.



8. Before beginning the next exercise, return the active level to Level 1 and hide the vise by clicking in **Visible** column for Level 101. This will ensure you do not accidentally select the vise when creating your mold geometry.



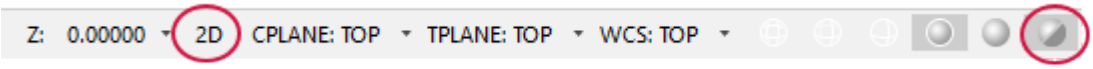
9. Save the file.

Exercise 3: Modifying the Solid Model

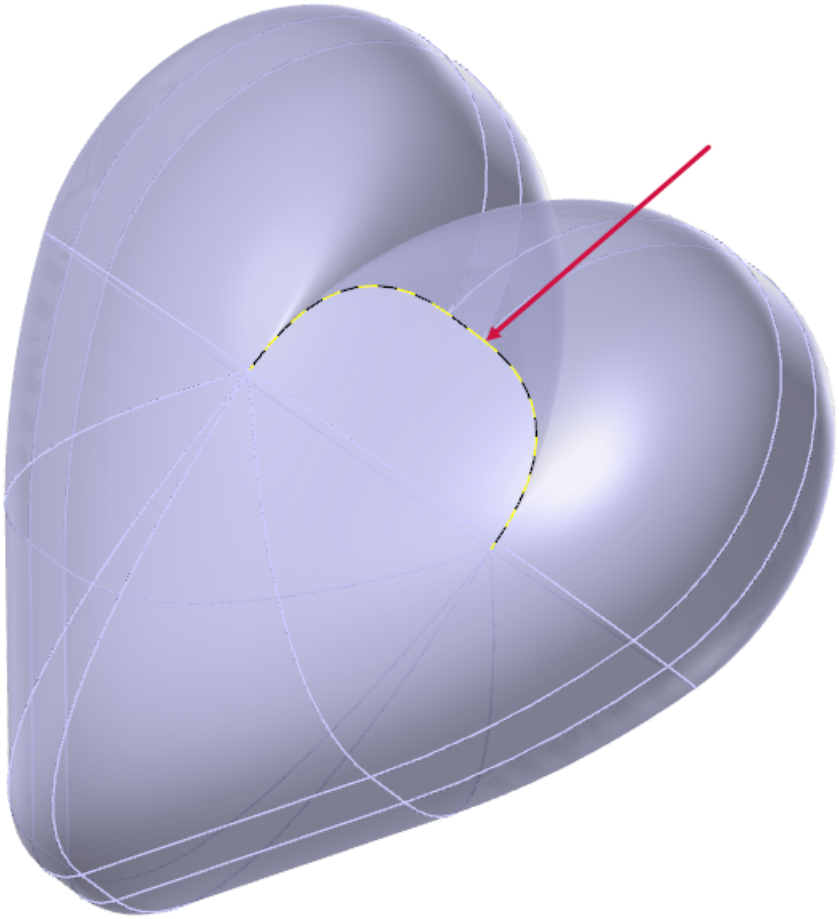
In this exercise, you modify the solid model to eliminate an interior, sharp edge where the heart joins the curves. Softening the sharp edge of the heart will ease the transition between surfaces for a better finish and will make it easier to remove the candy from the mold.

1. Your file `Heart mm-XXX.mcam` which you saved from the previous exercise should be open in Mastercam.

2. Activate the **Translucency** toggle and set the **2D/3D** toggle on the Status bar to **2D**. Setting your construction mode to 2D will place any geometry you create parallel to the current construction plane (**Top (WCS)**) at the current system Z depth.

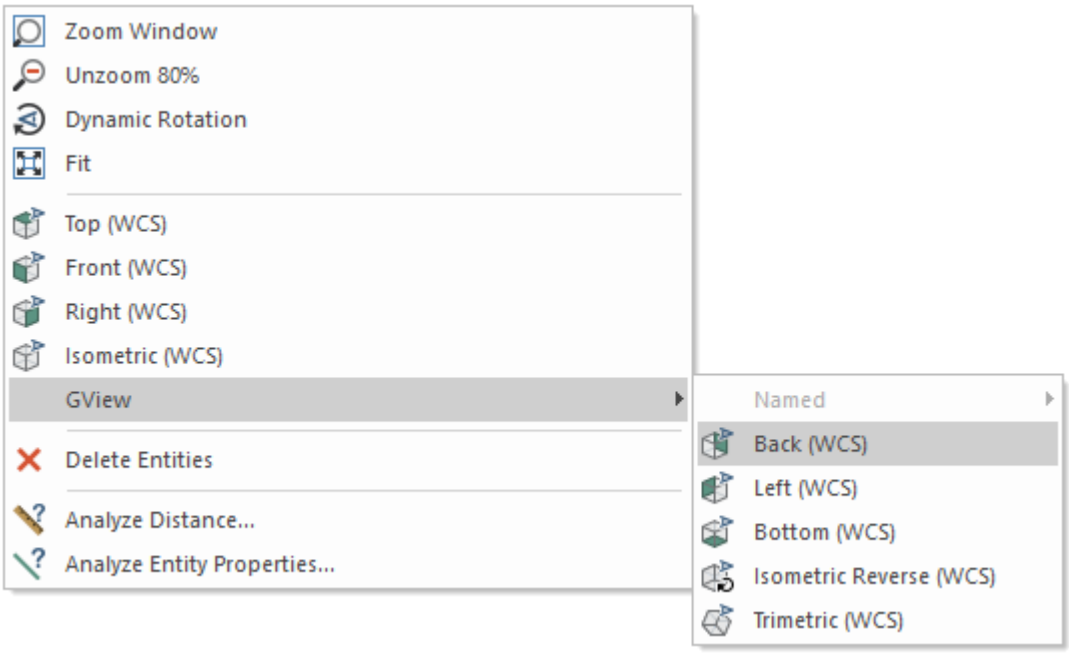


3. Hold down the middle mouse button and rotate the heart until you can see the internal, sharp edge as shown in the following image.

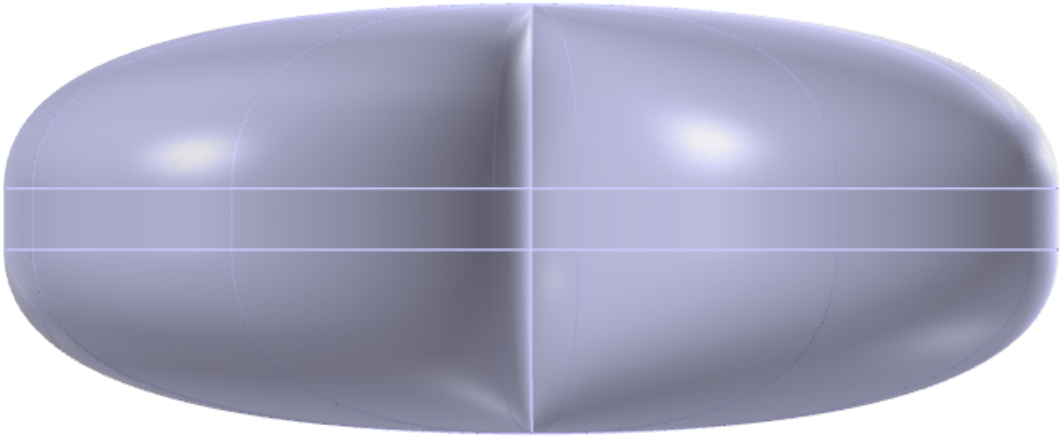


You will use Mastercam's CAD functions to soften this edge.

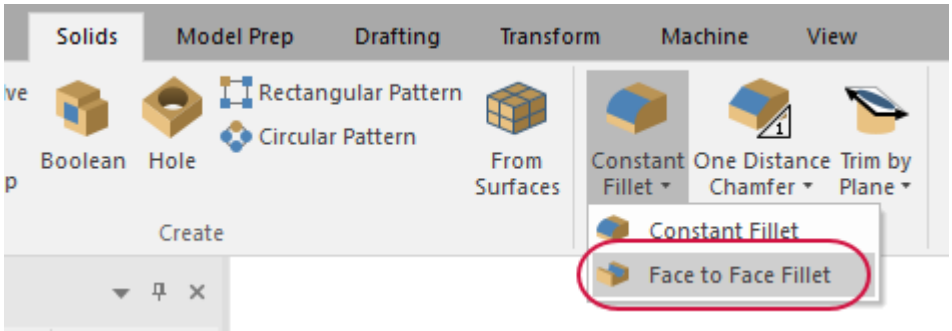
4. Right-click in the graphics window and select **GView**, **Back (WCS)**.



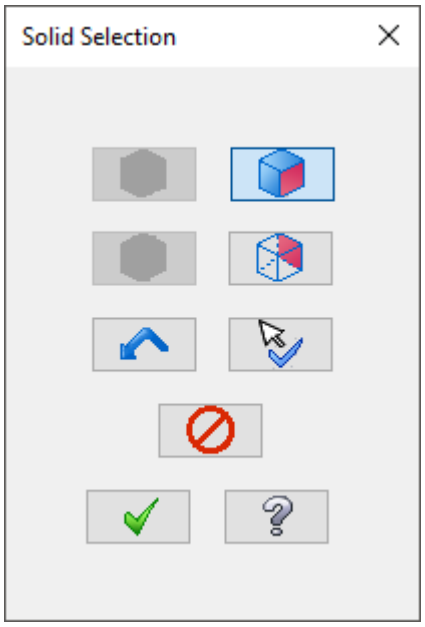
The model rotates to show the back view.



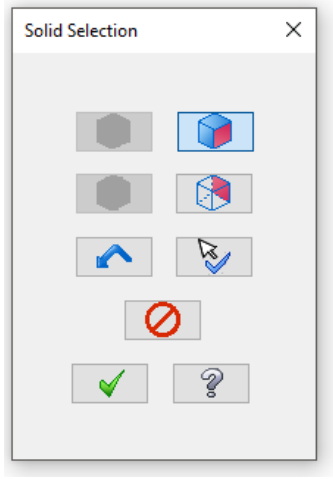
5. On the **Solids** tab, click **Face to Face Fillet**.

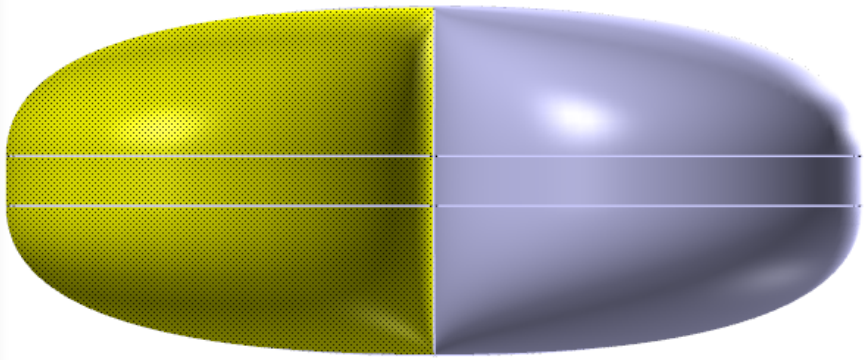


The **Solid Selection** dialog box displays and Mastercam prompts you to select one or more entities

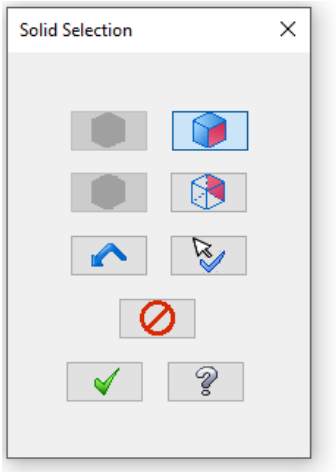


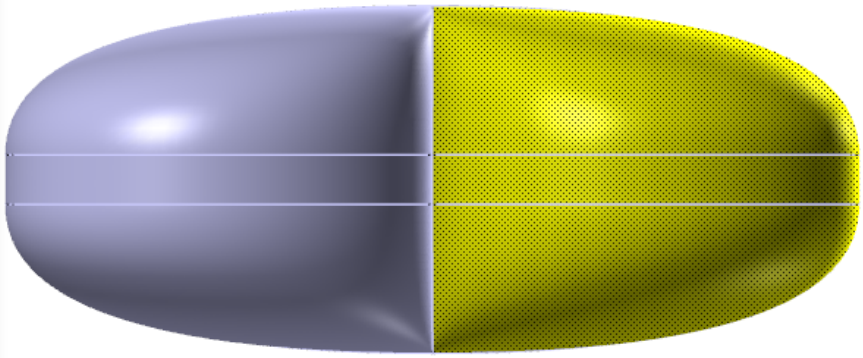
6. Select the three faces on the left side of the model and click **OK** in the **Solid Selection** dialog box.





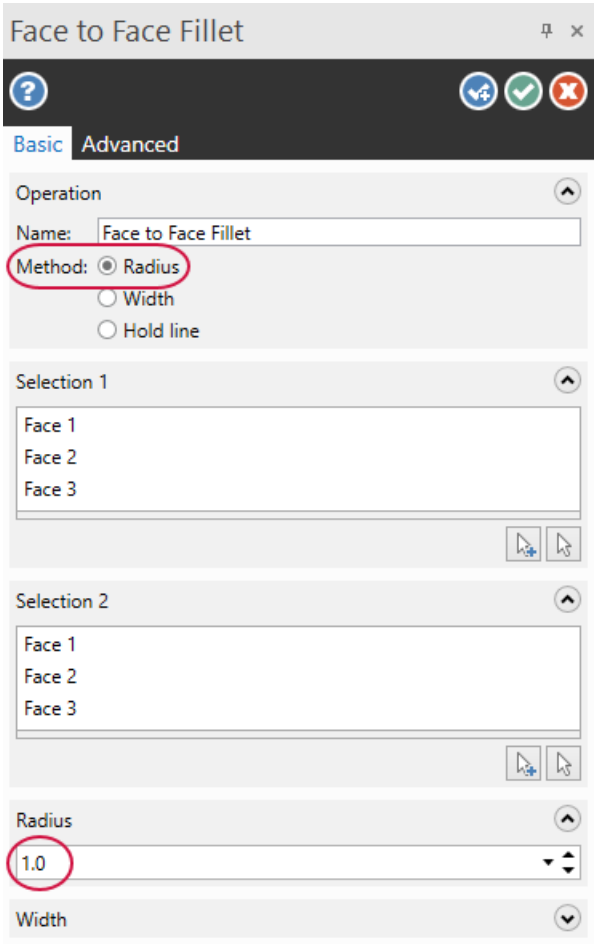
7. Select the three faces on the right side of the model and click **OK**.



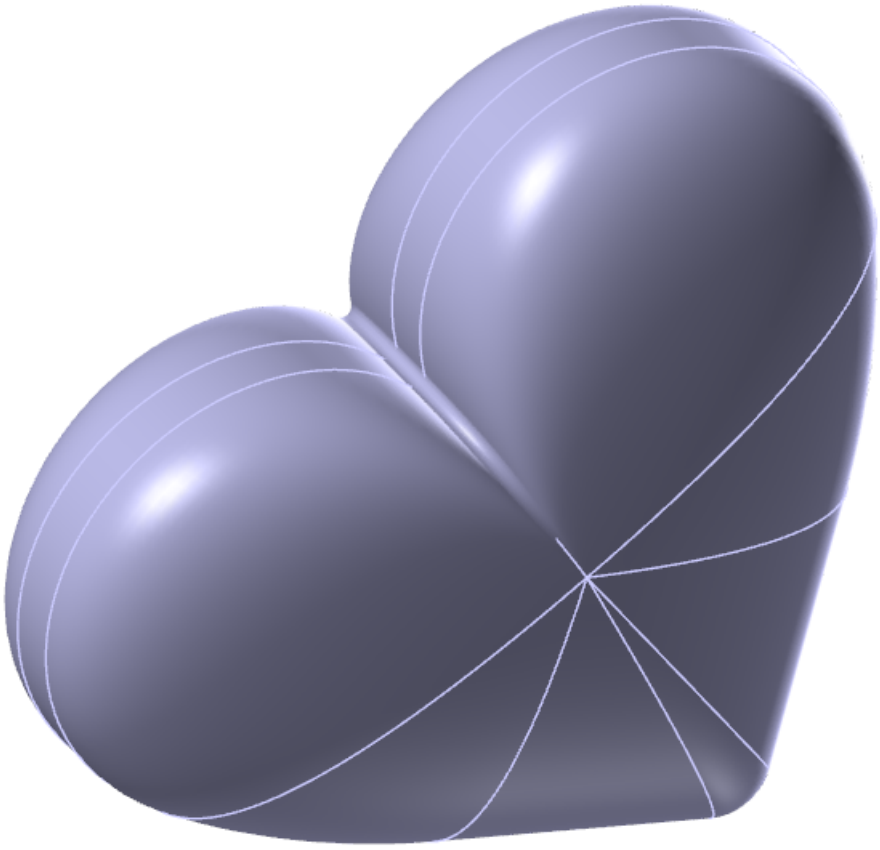


The **Face to Face Fillet** function panel displays.

8. Choose the **Radius** as the **Method** and enter **1.0** in the **Radius** field. A preview of the results shows in the graphics window.



9. Click **OK** (green checkmark). Rotate the solid to see the results.



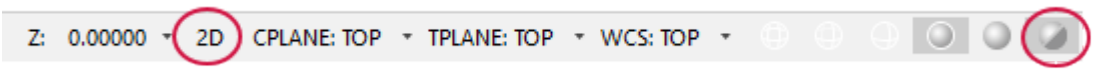
> **NOTE**
> Use the **Translucency** and **Shaded** toggles on the **Status Bar** to change your view.

10. Save your file.

Exercise 4: Creating Geometry for the Mold

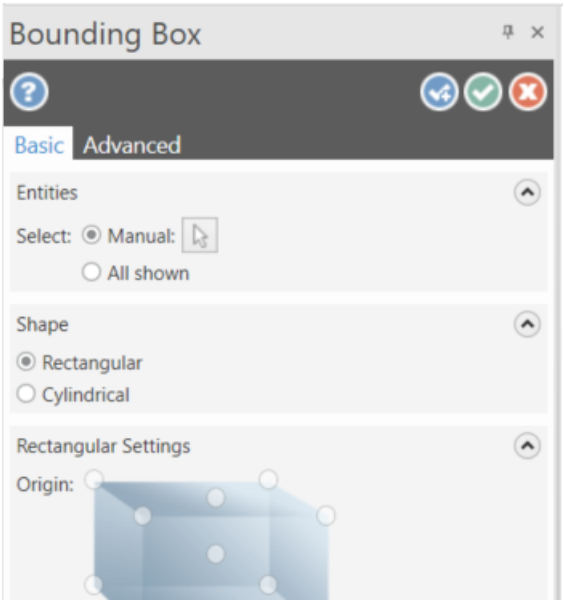
In this exercise, you will find the center point of the solid heart using the Bounding Box function, and create the wireframe geometry that will become the mold.

1. Your file `Heart mm-XXX.mcam` which you saved from the previous exercise should be open in Mastercam.
2. Ensure that the **2D/3D** toggle on the **Status Bar** is set to **2D**. Setting your construction mode to 2D will place any geometry you create parallel to the current construction plane (**Top (WCS)** at the current system Z depth.

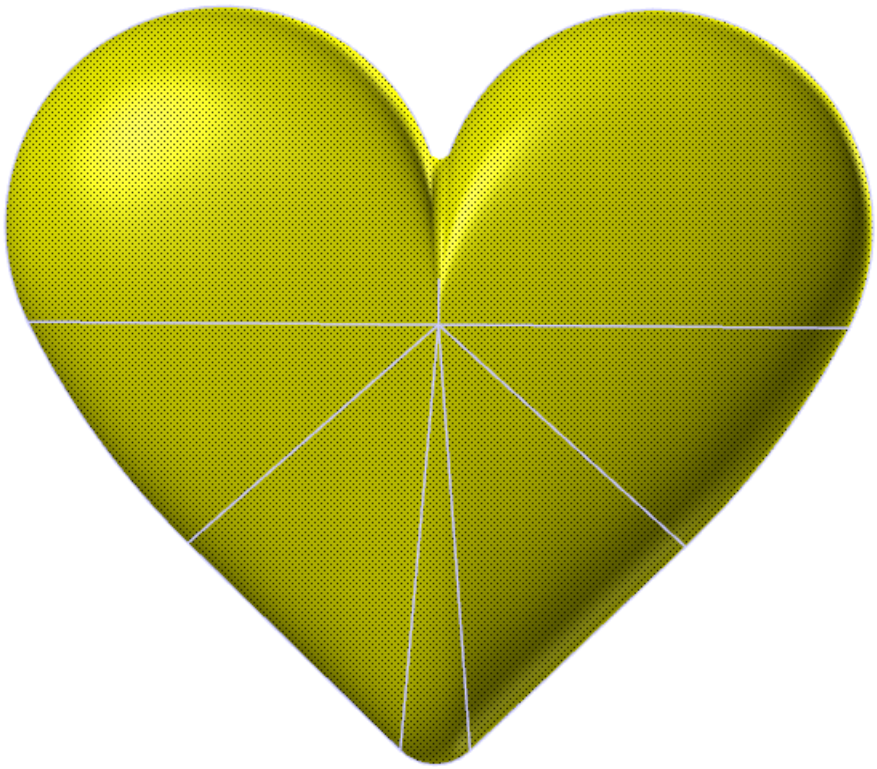


3. Click **Translucency** on the **Status Bar** to make the solid heart translucent.

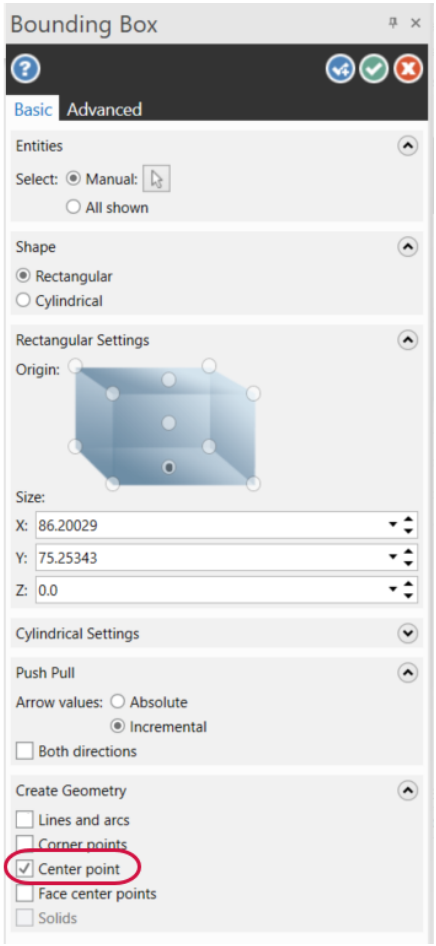
4. On the **Wireframe** tab, select **Bounding Box** to open the function panel.



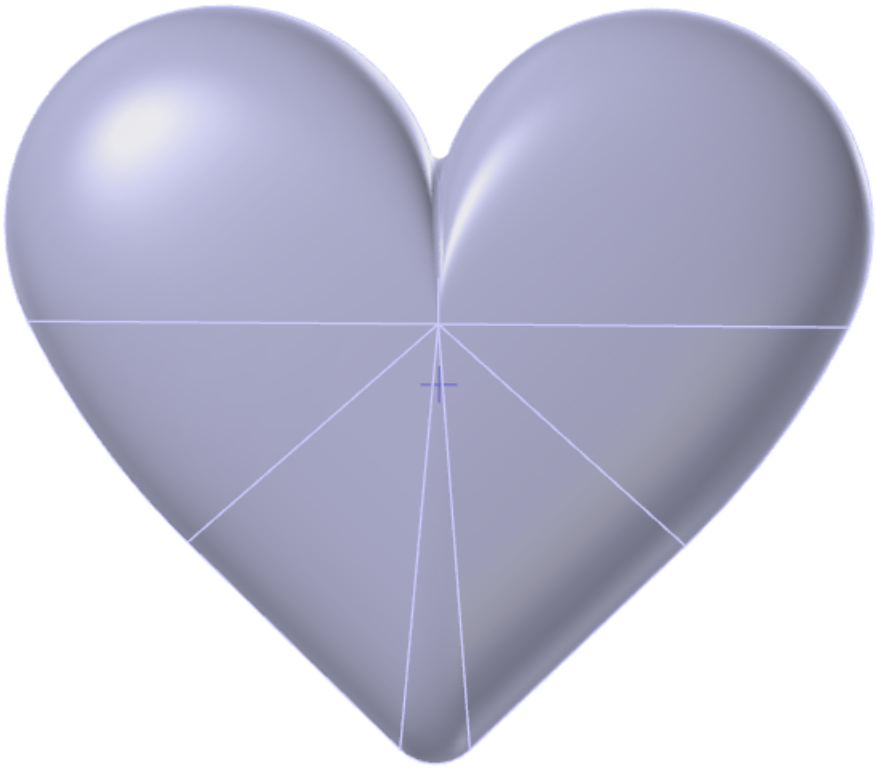
5. Select the heart in the graphics window and click **End Selection**.



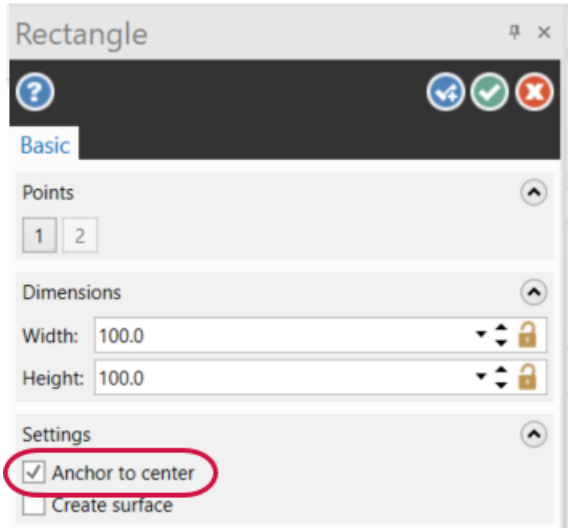
6. In the **Create Geometry** section of the **Bounding Box** function panel, select **Center Point**. This will locate the exact center point of the solid heart.



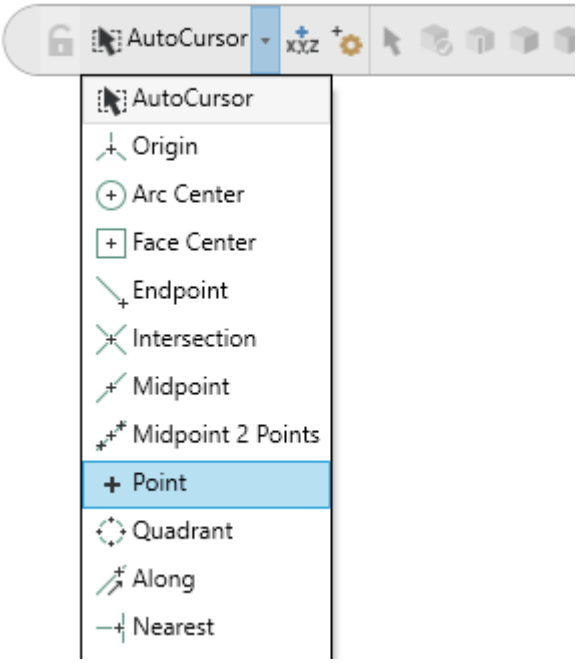
7. Click **OK** in the **Bounding Box** function panel to accept. You will use the bounding box center point in the next step.



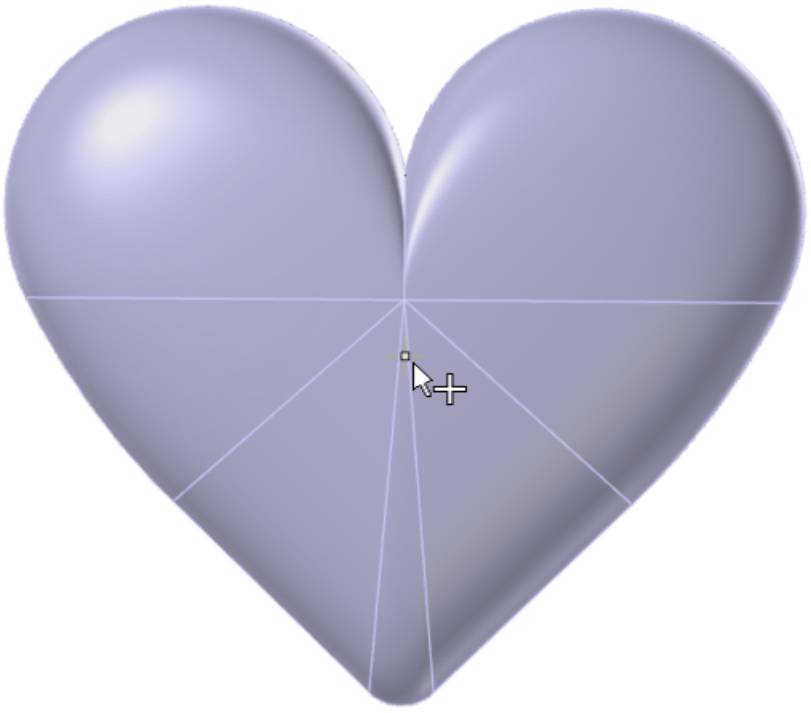
8. From the **Wireframe** tab, click **Rectangle** to open the **Rectangle** function panel.
9. Select **Anchor to center**. This will center the rectangle you create around the solid heart.



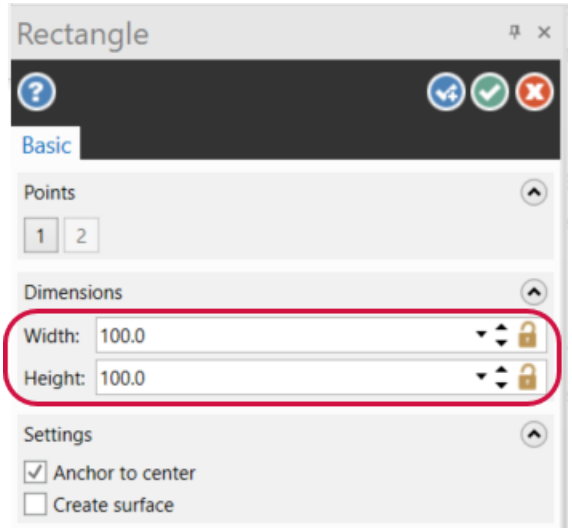
10. On the Selection Bar, choose **AutoCursor**, **Point**.



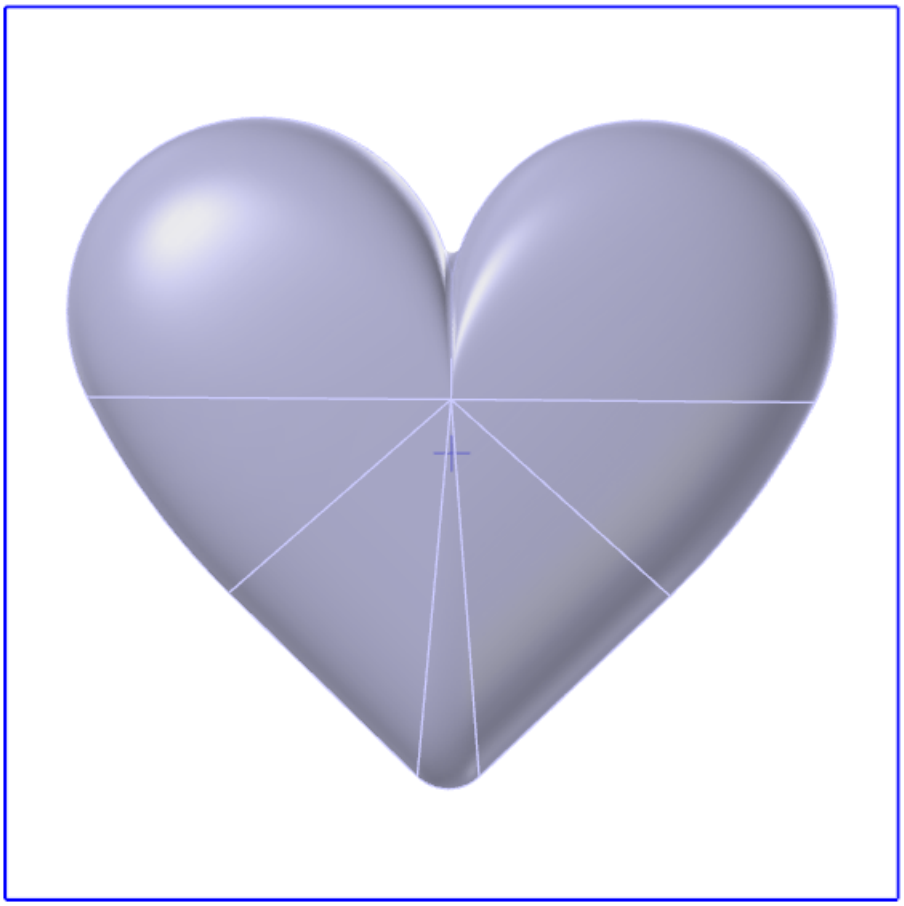
11. Click on the point at the center of the solid heart. This will anchor the rectangle to that point.



12. Enter **100.0** in the **Width** and **Height** fields.



13. Click **OK** to create the rectangle that will become the block for the mold

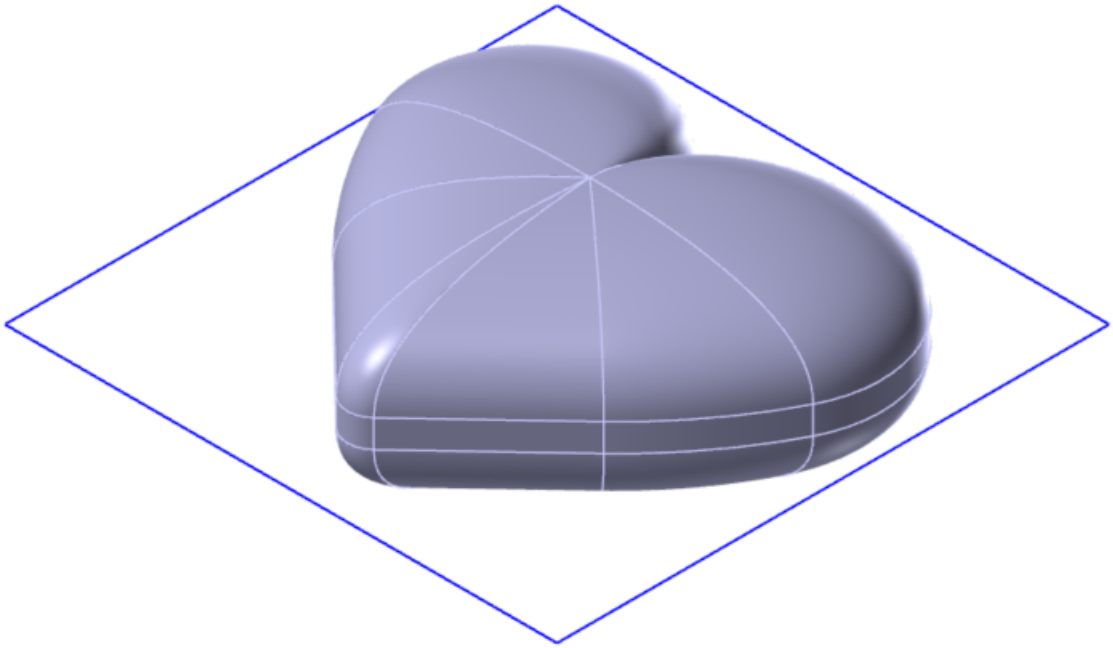


14. Save your file.

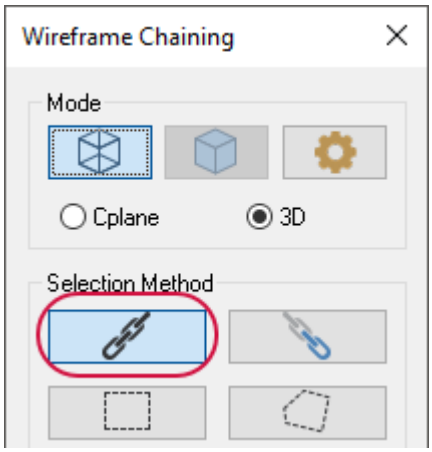
Exercise 5: Creating the Block for the Mold

In this exercise you will create a block for the candy mold by extruding the rectangle you created in the previous exercise.

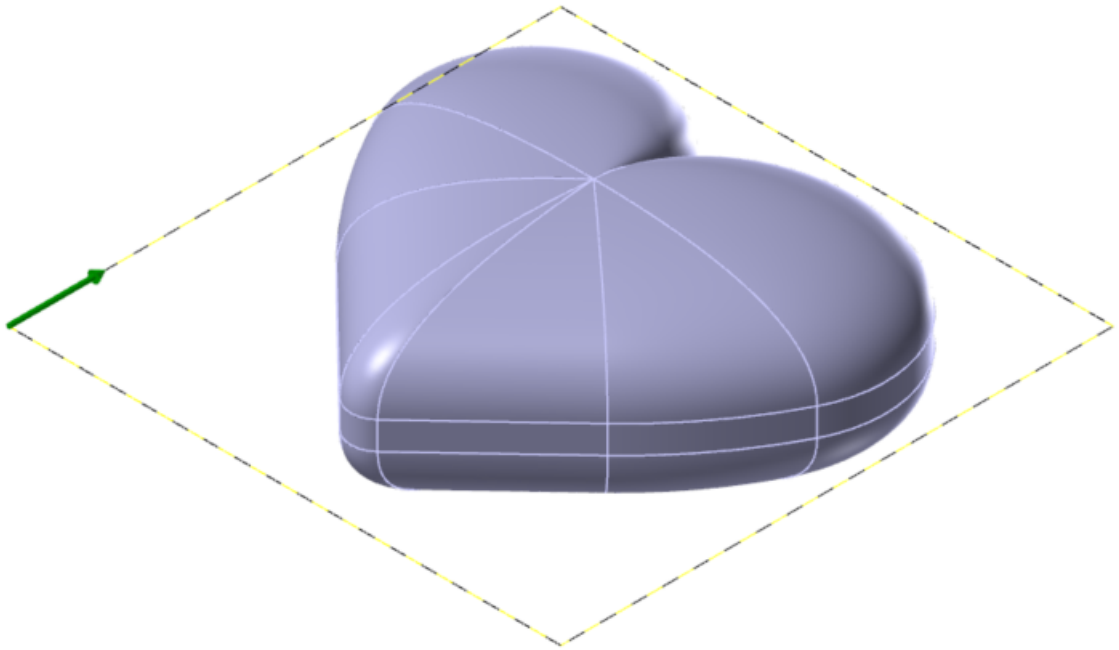
1. Your file `Heart mm-XXX.mcam` which you saved from the previous exercise should be open in Mastercam.
2. Right-click in the graphics window and select the **Isometric (WCS)** view.



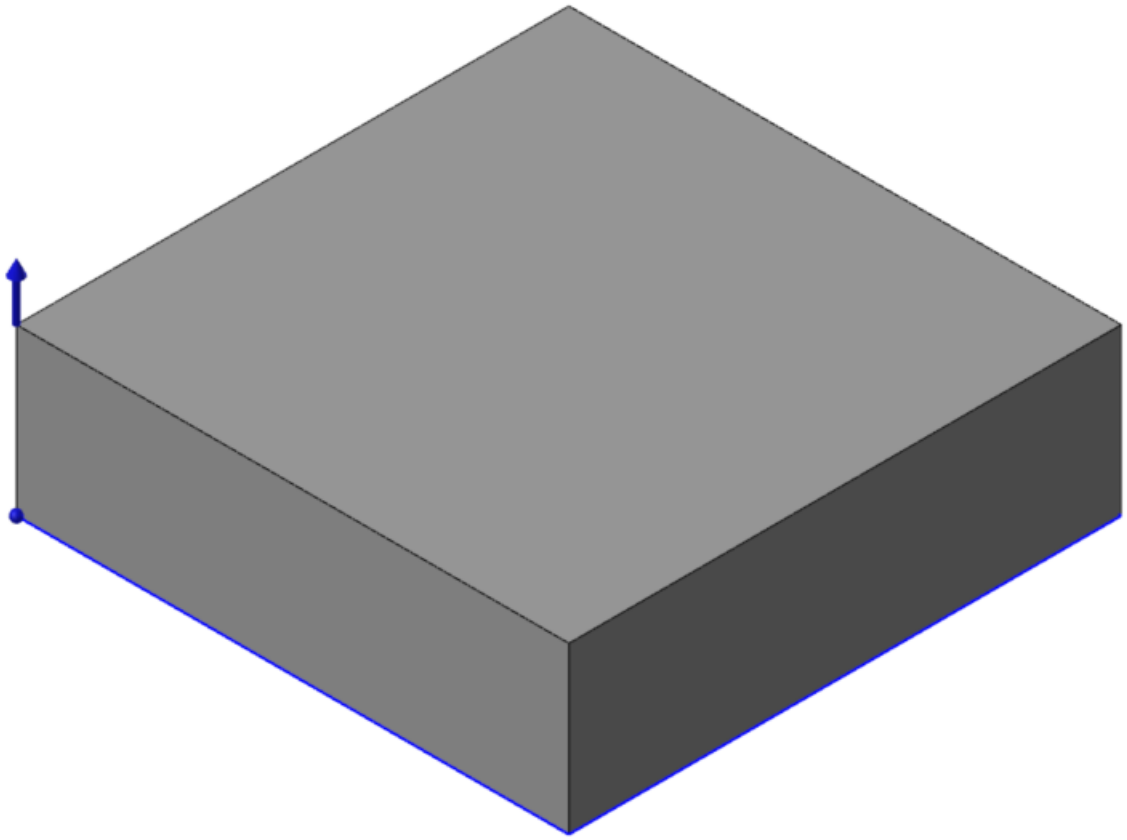
3. Click **Extrude** on the **Solids** tab to open the **Wireframe Chaining** dialog box.



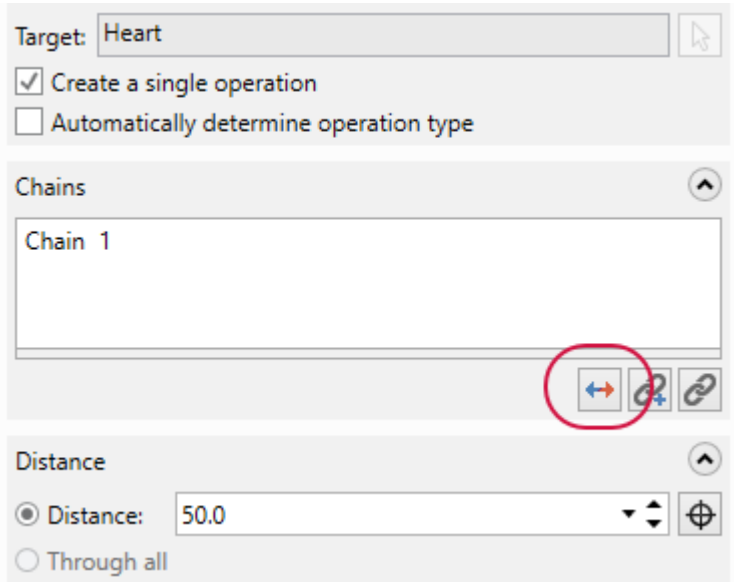
4. Ensure that the **Selection Method** is set to **Chains** and then select the rectangle.

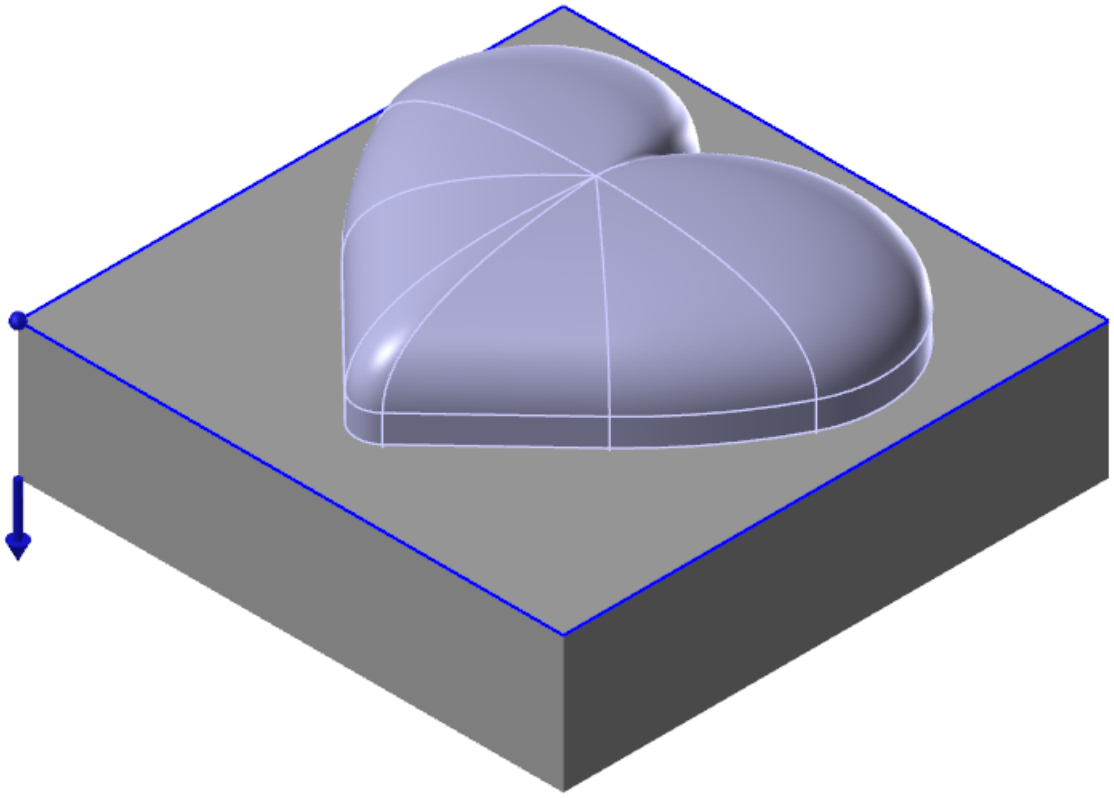


5. Click **OK** in **Wireframe Chaining** dialog box and the **Solid Extrude** function panel opens. The solid may be extruded in a positive direction as shown in the following image.

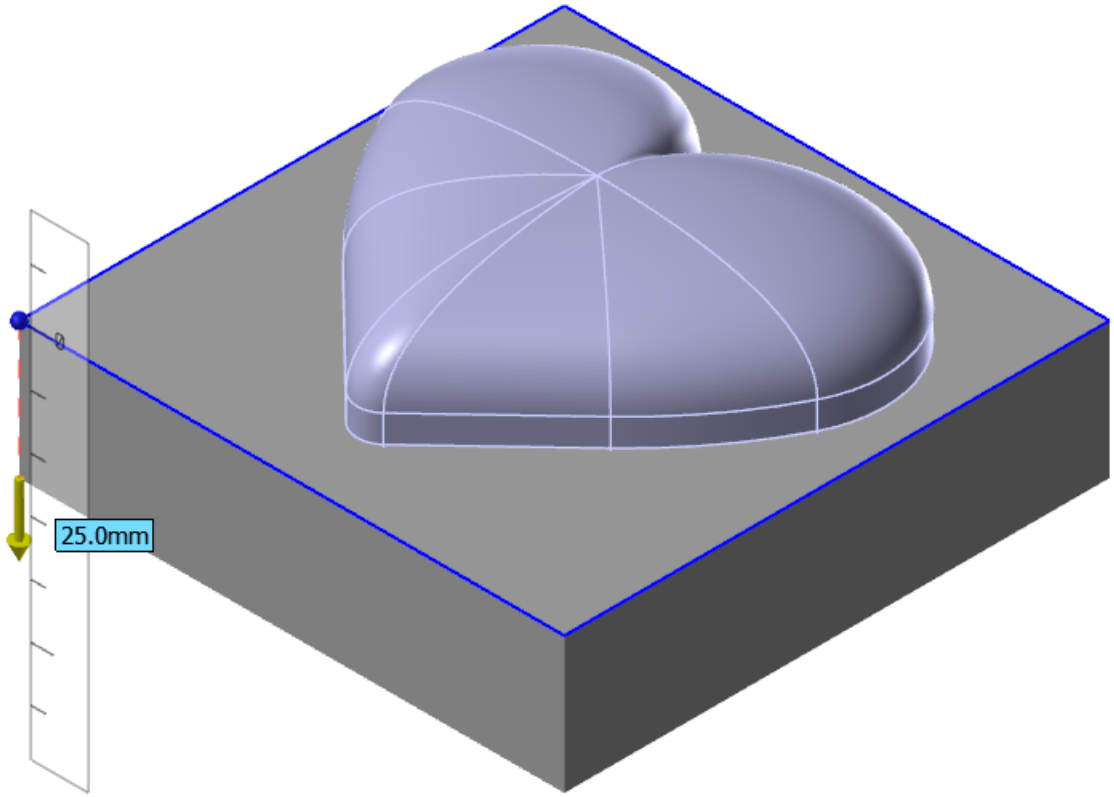


6. Click **Reverse** in the **Solid Extrude** function panel to change the direction to a negative direction.



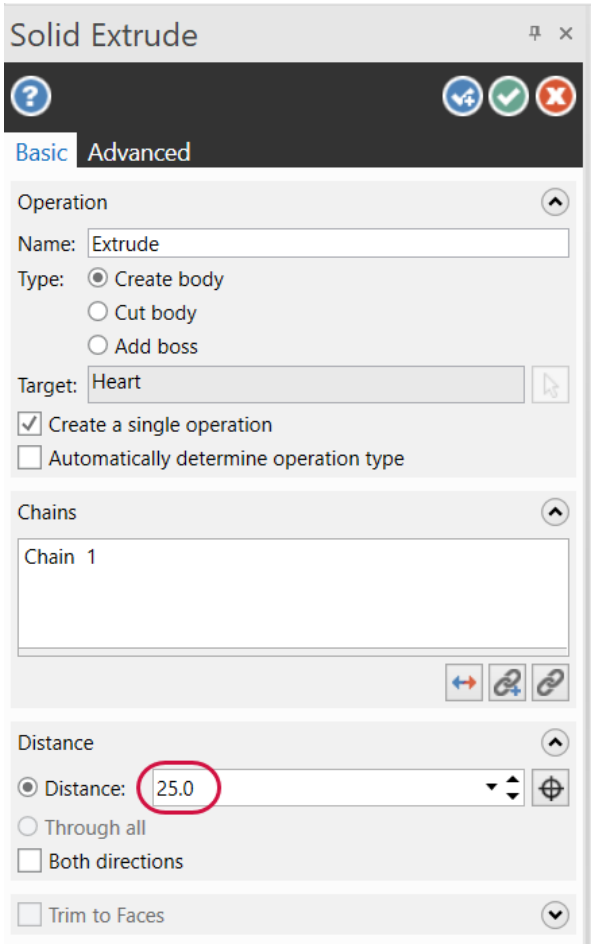


7. Hover over the arrow until the ruler displays and then click and drag until the distance is **25.0mm**.



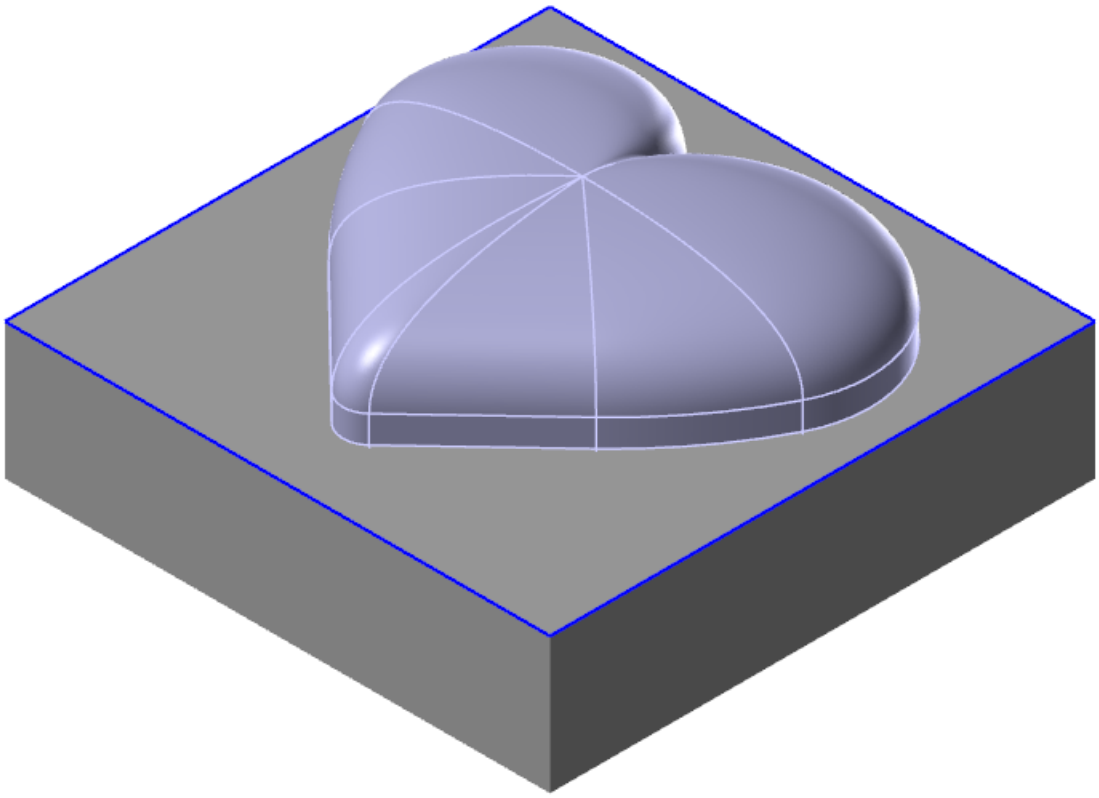
> **NOTE**
> You can type **25.0mm** in on-screen field and press [**Enter**] to accept the value.

8. Alternately you can enter **25.0** in the **Distance** field of the **Solid Extrude** function panel.



9. Verify that the **Type** is set to **Create body**.

10. Click **OK** to create the block.

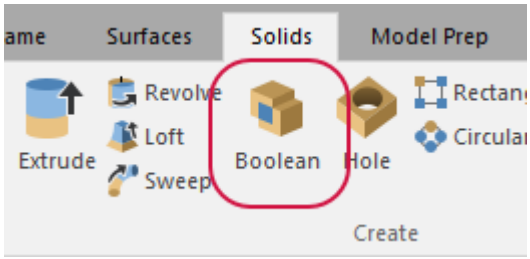


11. Save your file.

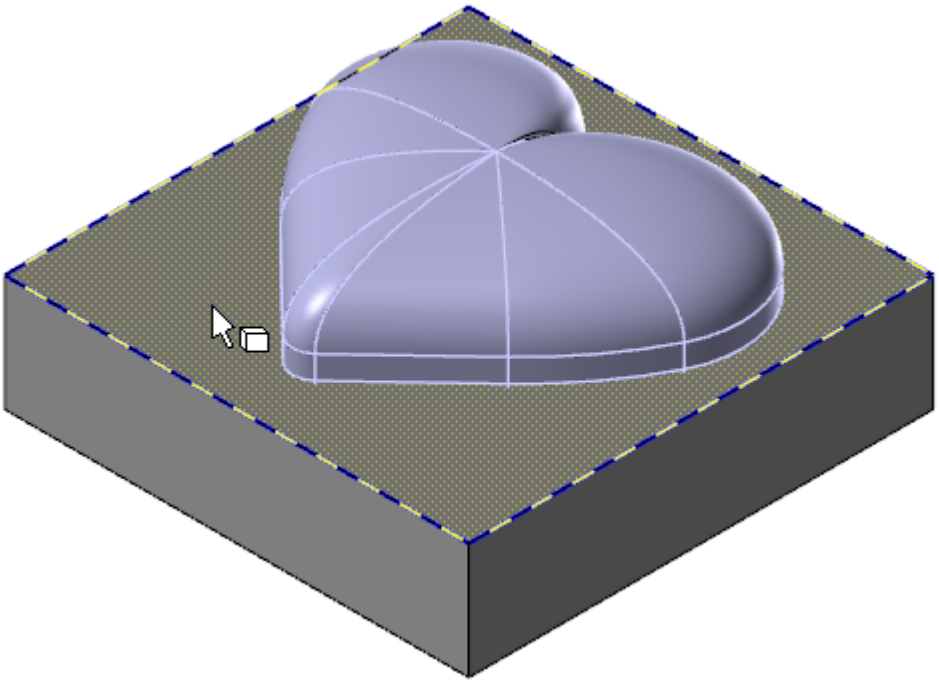
Exercise 6: Using Boolean to Create the Mold Impression

In this exercise you will use the Boolean function to transform the solid heart into the impression that will be your candy mold. When using Boolean, you will identify the target body, which is the block and the tool body, which is the solid heart.

1. Click **Boolean** on the **Solids** tab. Mastercam prompts you to select the target body.

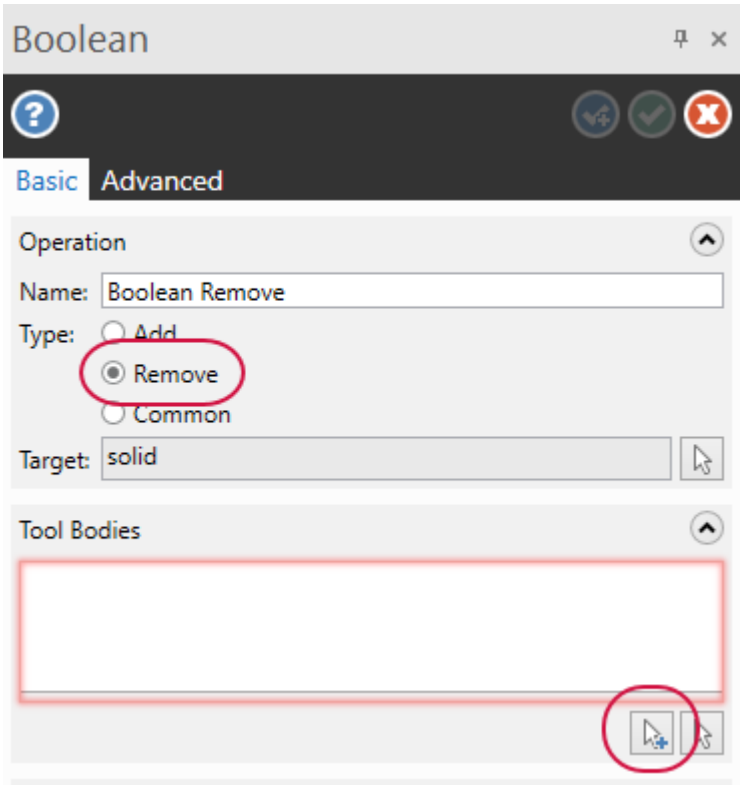


2. Select the top surface of the block to be the target body.

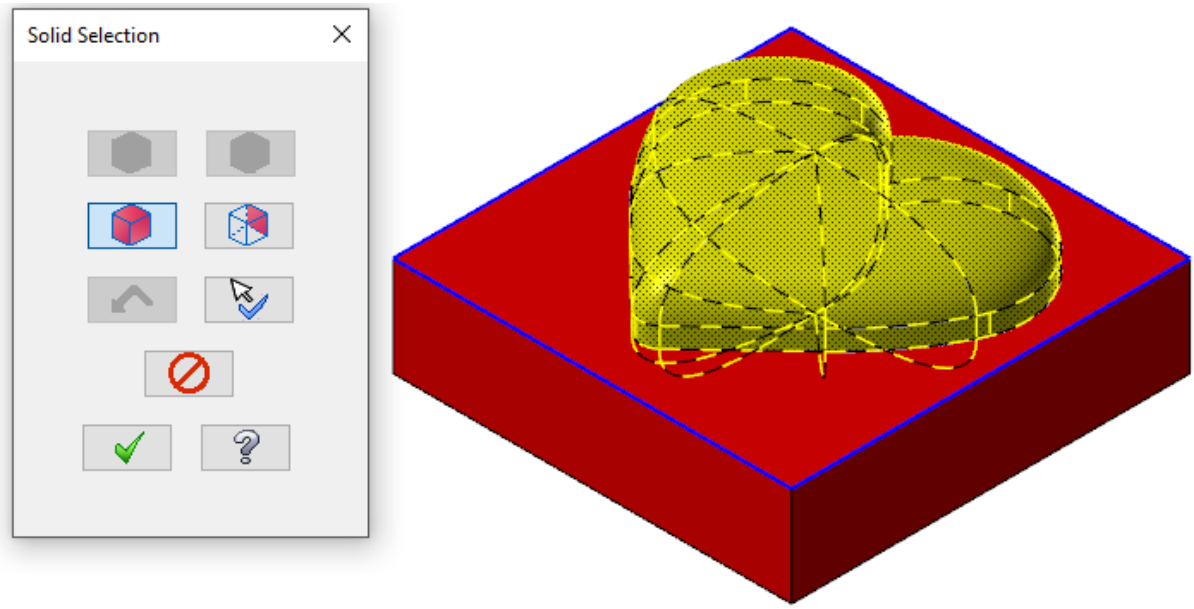


The **Boolean** function panel opens and Mastercam prompts you to select the tool bodies

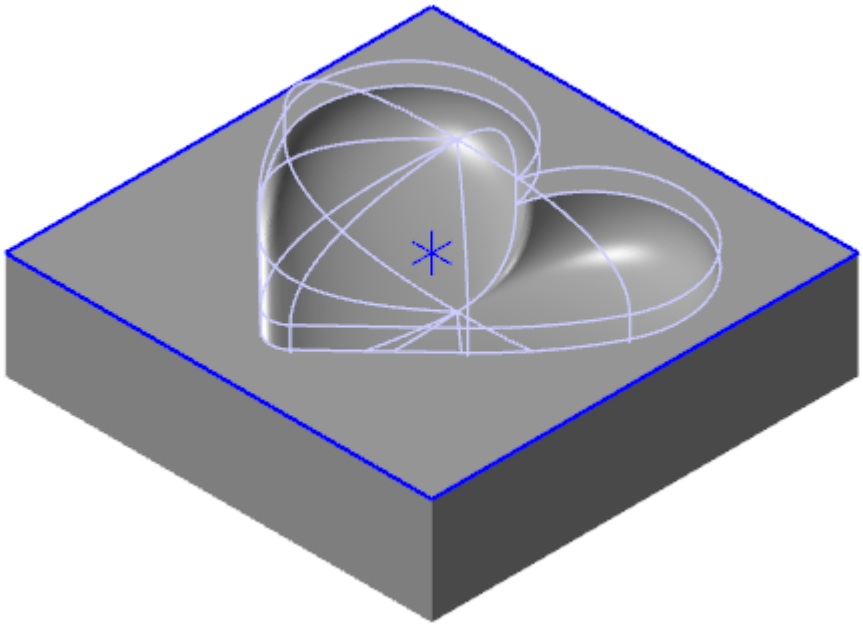
3. In the **Boolean** function panel, select **Remove** and then click **Add Selection**. By choosing **Remove**, the solid heart will be taken away from the block leaving an impression that will become the candy mold.



4. Ensure that the **Body** selection filter is chosen in the **Solid Selection** dialog box, and then select the solid heart



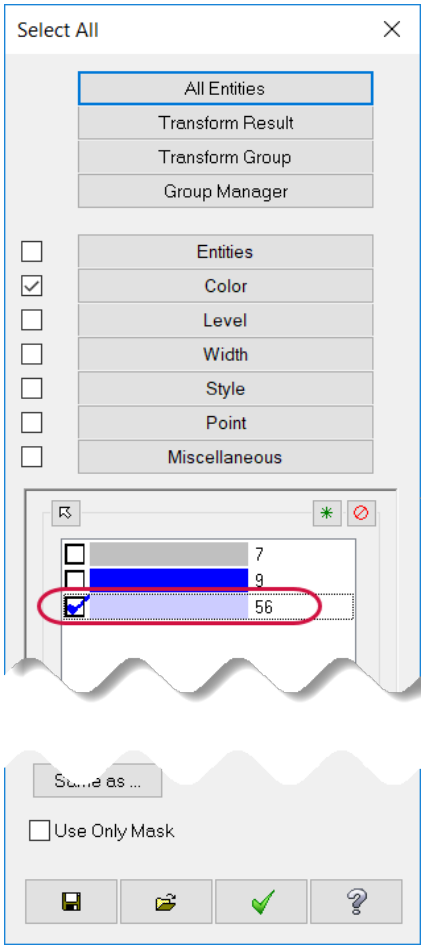
5. Click **OK** in the **Boolean** function panel and the heart is removed from the block.



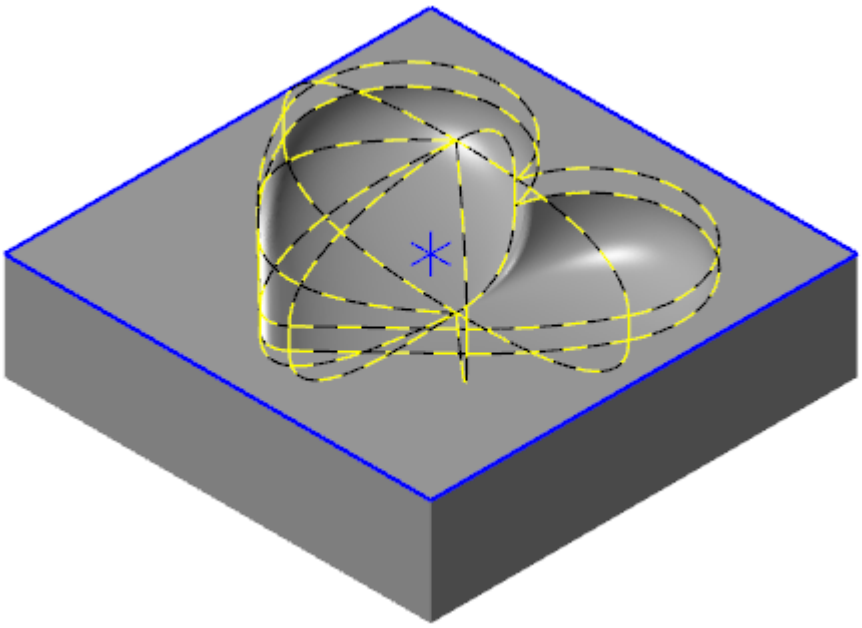
6. Click the **Select all Entities by Color** Quick Mask.

Quick Masks help you to quickly select entities in the graphics window. Many Quick Masks have two functions, Select all entities or Select only entities, (depending on whether you click the right or left side of the button).

7. In the **Select All** dialog box, ensure that **All entities** is chosen and then select the check box for the lavender color

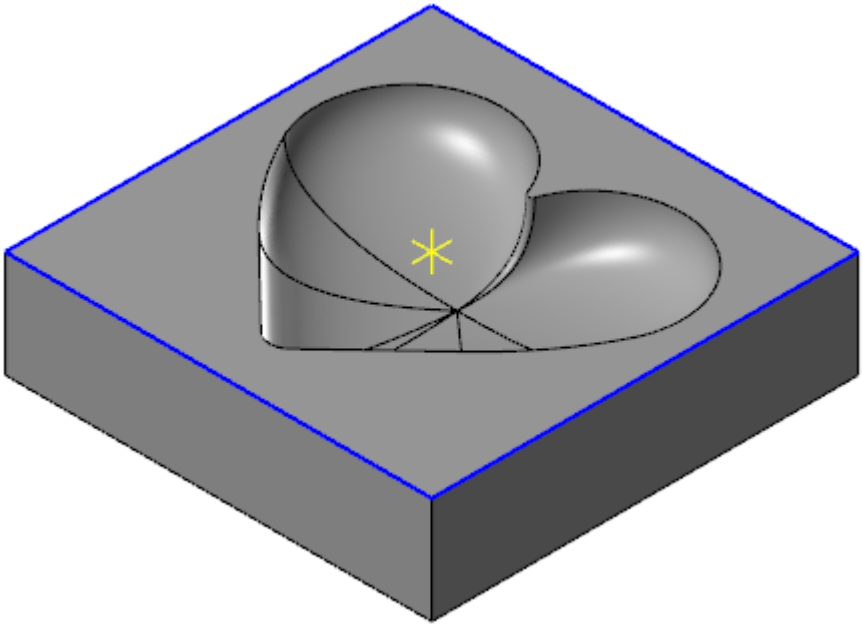


8. Click **OK** and the wireframe heart geometry is selected in the graphics window.



9. Press [**Delete**] on your keyboard.
10. Click the **Select all point entities Quick Mask**.

This selects the point that was created from the Bounding Box.

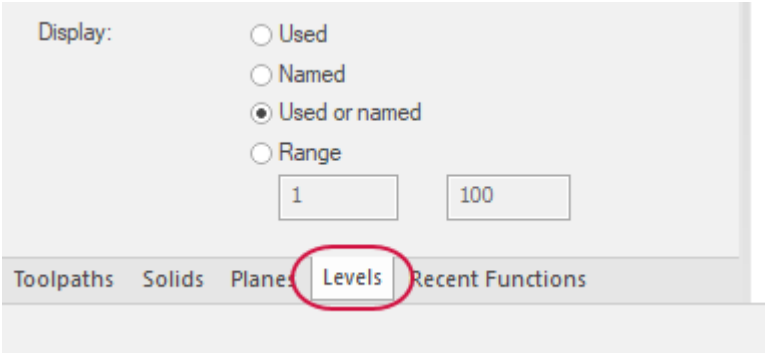


11. Press [**Delete**] on your keyboard to remove the point.
12. Save your file.

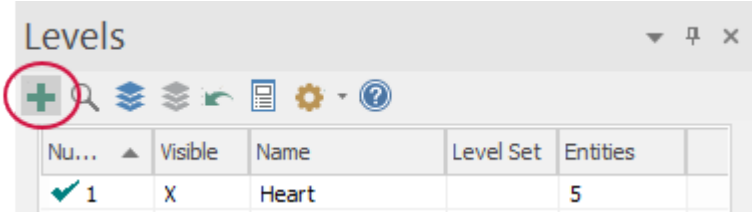
Exercise 7: Adding a Containment Boundary

A containment boundary is used to limit the areas of the part that can be machined. In this exercise, you will add a wireframe containment boundary to the heart shape. When programming the toolpaths you will set the containment boundary strategy to the **Stay Inside**. This allows the tool to machine the area inside the geometry and not pass through the boundary.

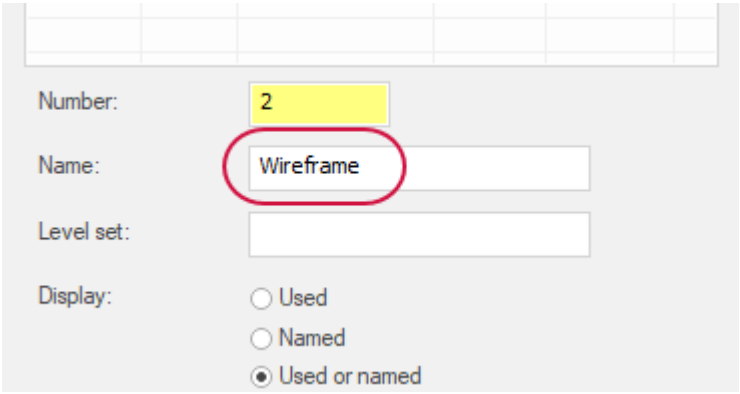
1. Your file, `Heart mm-XXX.mcam`, which you saved from the previous exercise should be open in Mastercam.
2. Click the **Levels** tab to bring the Levels Manager to the forefront.



3. Click **Add a new level**, (green plus sign icon).

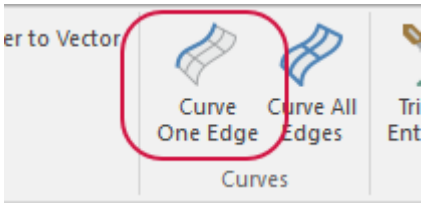


4. Click **Add a new level**, (green plus sign icon). In the Levels Manager enter **Wireframe** in the **Name** field of the new level.

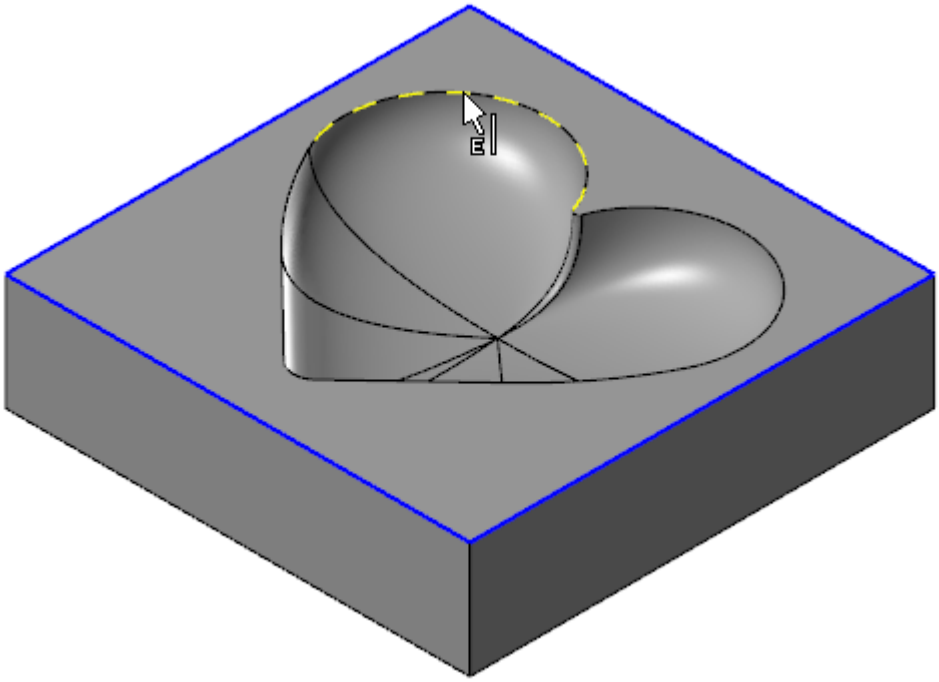


Notice that level 2 is the active level. The new geometry you will create in this exercise will reside on this level. Ensure that the Vise level (101) is not visible. The Visible column should be empty. If there is an X in the column click it to remove the X and hide the level in the graphics window.

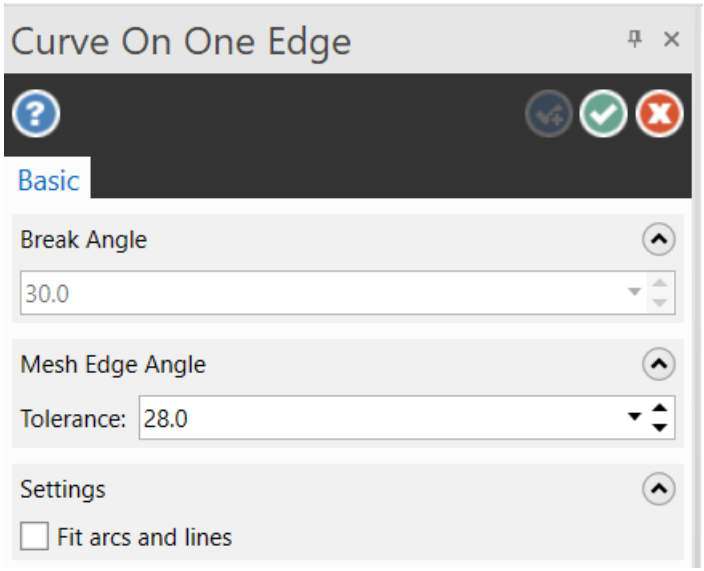
5. On the **Wireframe** tab, click **Curve One Edge**.



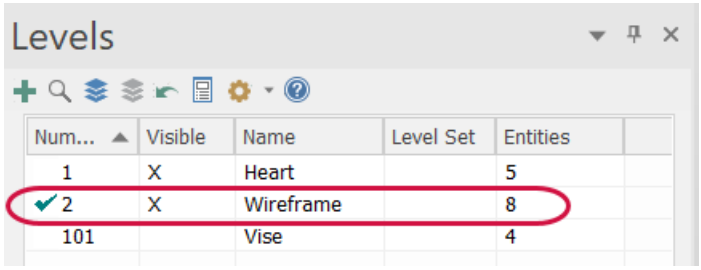
6. Press the [**Shift**] key and select the edge of the heart. This method selects all tangent edges.



7. Click **OK** in the **Curve On One Edge** function panel.



The new wireframe entities appear on level 2 in the Levels Manager.

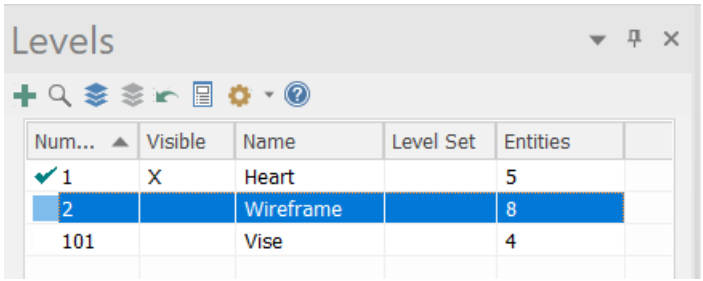


8. Save your file.

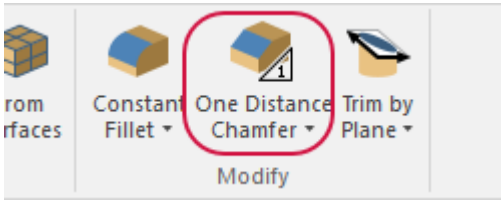
Exercise 8: Adding a Chamfer to the Mold

In this exercise, you will use one of Mastercam's chamfer functions to add a bevel to the edge of the heart impression. Adding a chamfer to the mold edge will make it easier to remove the candy from the mold.

1. Your file, `Heart mm-XXX.mcam`, which you saved from the previous exercise should be open in Mastercam.
2. In the Levels Manager, click level **1**, Heart to make it the active level (designated by a green check), and clear the visibility for level **2**, **Wireframe**.

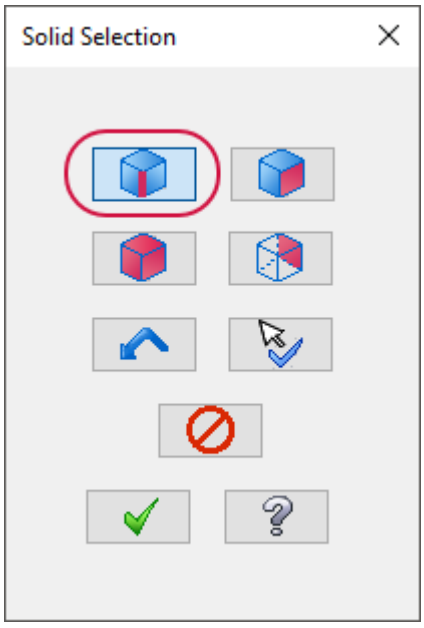


3. On the **Solids** tab, click **One Distance Chamfer**.

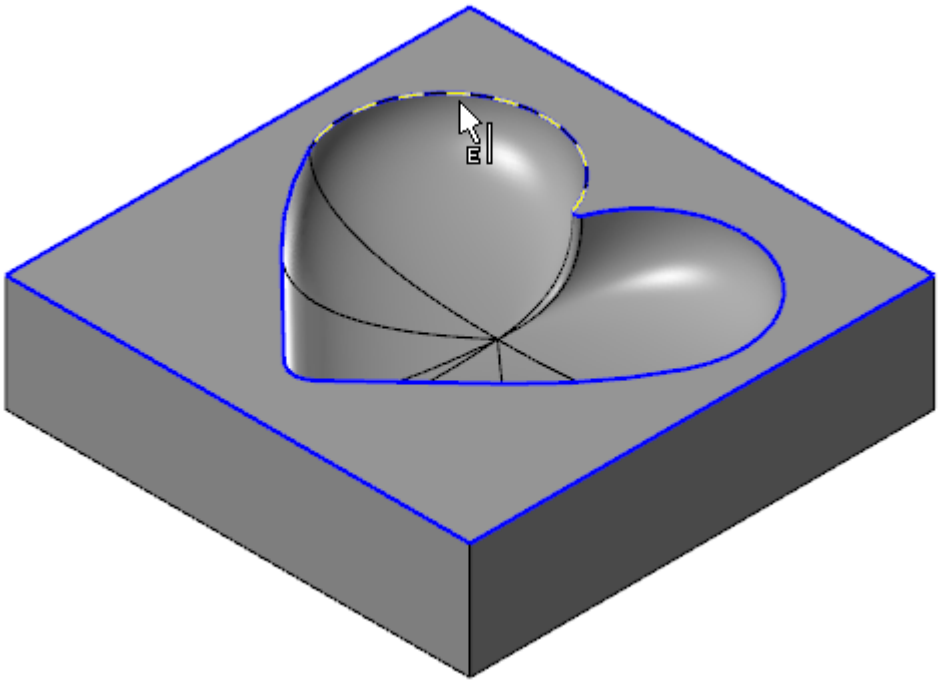


The **Solid Selection** dialog box displays and Mastercam prompts you to select the entities to chamfer.

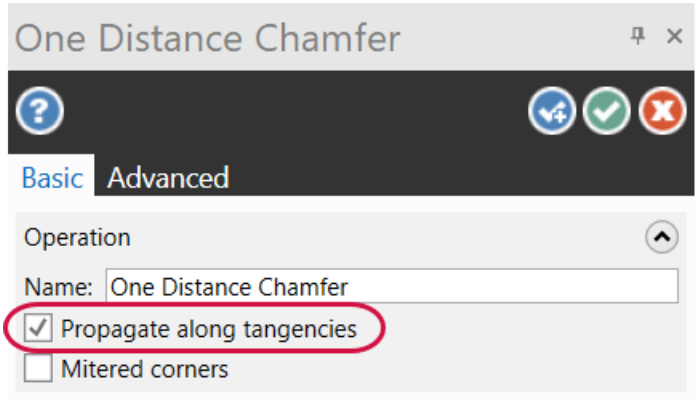
4. Click **Edges** to set the selection filter to only select edges. Deselect **Face**, **Body**, or **From back** if they are selected.



5. Select one segment of the heart's edge and click **OK** in the **Solid Selection** dialog box.

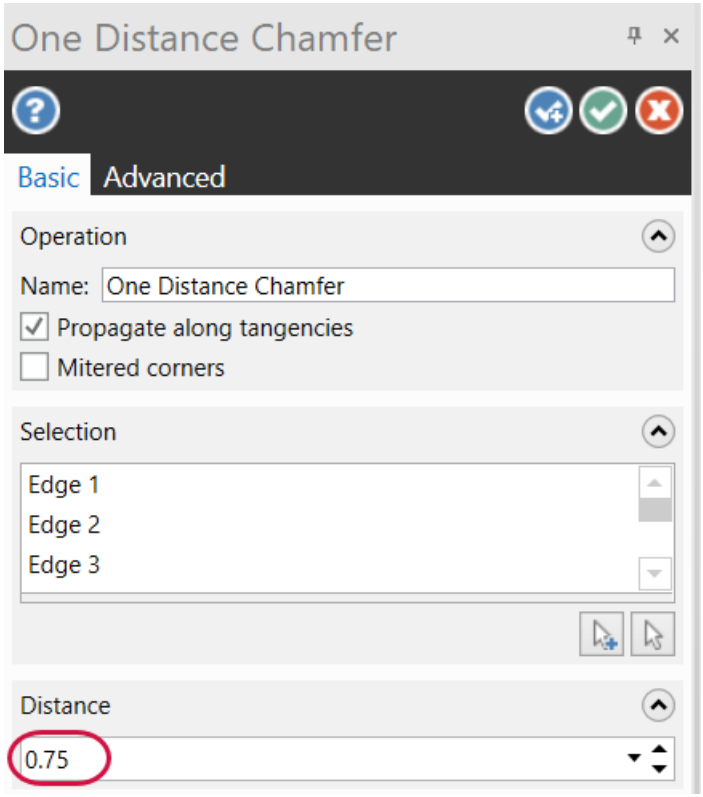


6. In the **One Distance Chamfer** function panel, select **Propagate along tangencies** to chamfer all edges that are tangent to the one you selected.

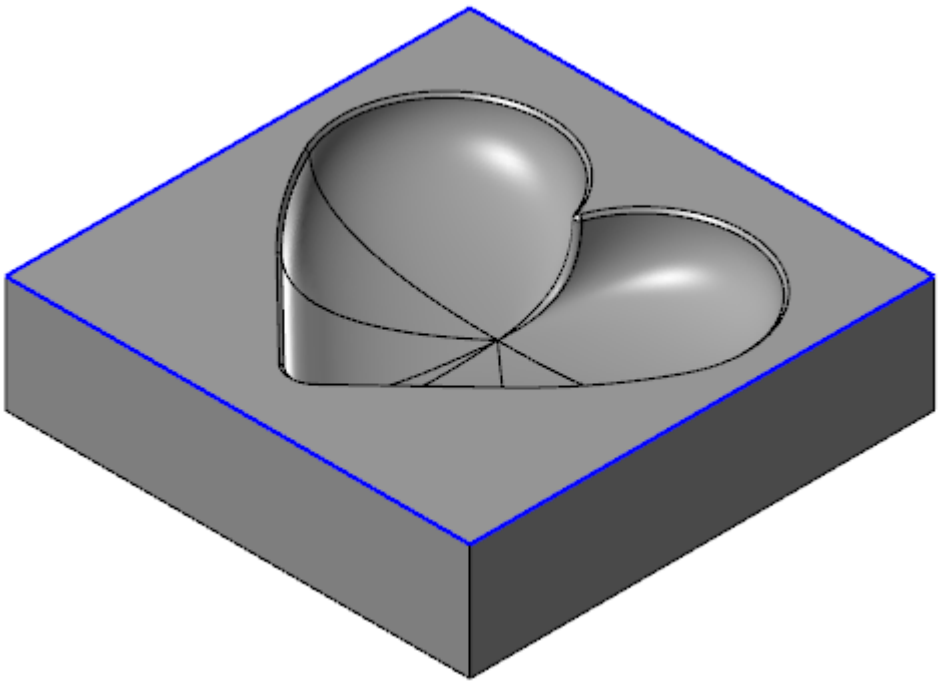


> **NOTE**
> You can also hold down the [**Shift**] key when selecting the edge to chamfer and all tangent edges will be selected automatically.

7. Enter a value of **0.75** in the **Distance** field and then click **OK.**



The heart has chamfers on all edges. Hold down the middle mouse button and rotate your part to see the all of chamfered edges.



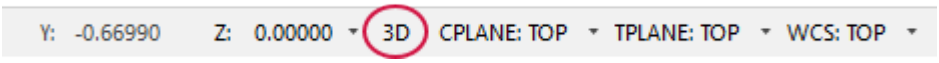
8. Save your file.

Exercise 9: Adding Stick Slot Geometry

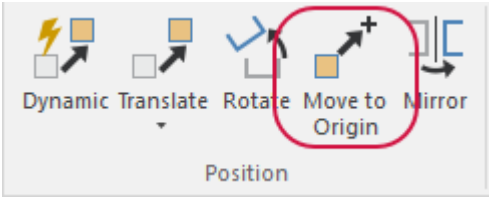
In this exercise, you will create the geometry required to machine a slot in the mold to hold a candy stick. The slot will be angled at five degrees to ensure that the stick is firmly embedded in the candy and seals the opening to prevent leakage of the liquid chocolate.

1. Your file, `Heart mm-XXX.mcam`, which you saved from the previous exercise should be open in Mastercam.
2. On the **View** tab, click **Show Axes** and **Show Gnomons** to activate them. Use the function's drop downs to choose which axes and gnomons display in the graphics window.

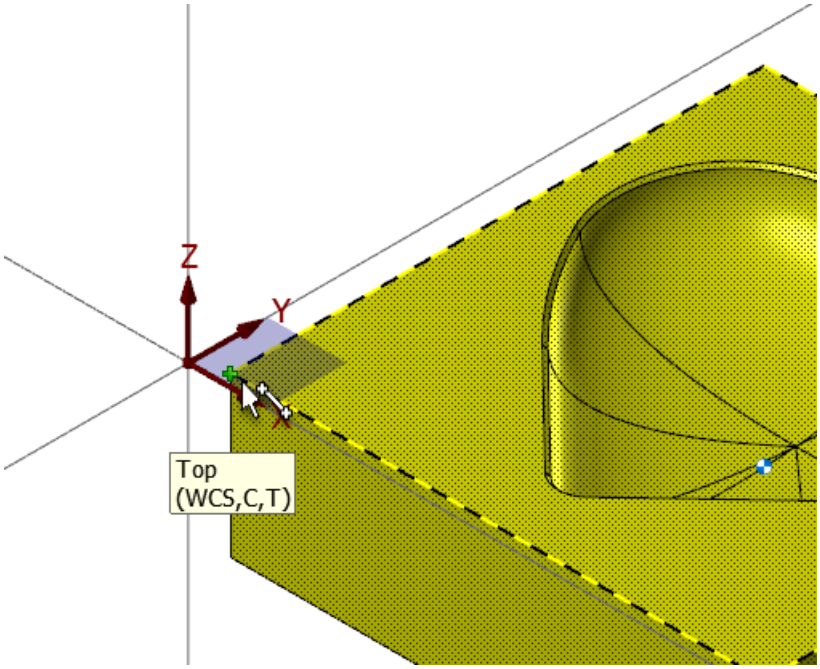
3. Set the **2D/3D** toggle on the **Status Bar** to **3D**.



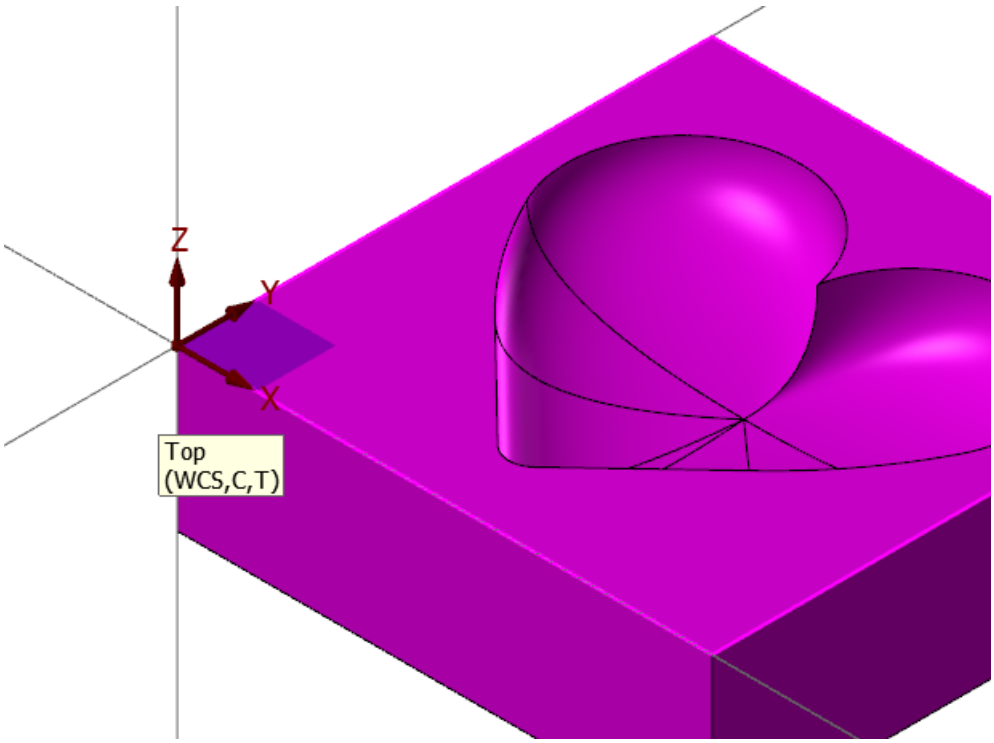
4. Right-click in the graphics window and set your view to **Isometric (WCS)**.
5. Right-click and choose **Fit** to center your part in the graphics window.
6. On the **Transform** tab, click **Move to Origin**.



7. Click the lower left corner of the mold as shown in the following image.

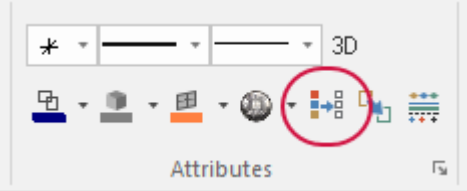


The mold moves to the origin.



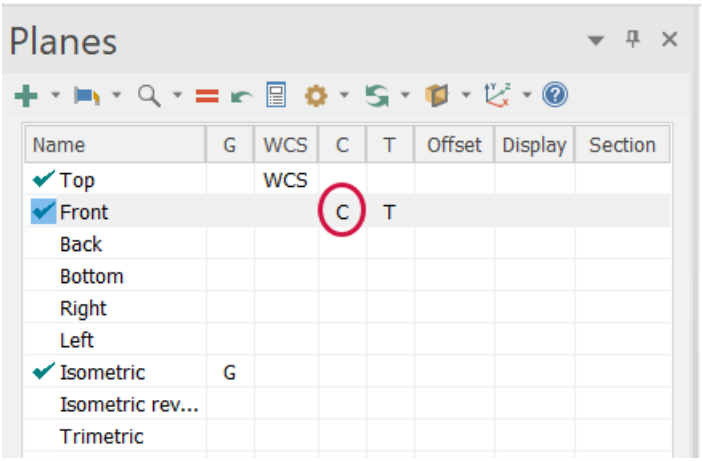
The mold color is now purple which is the results color for a transformation.

8. Click **Clear Colors** on the **View** tab to return all entities their default colors.

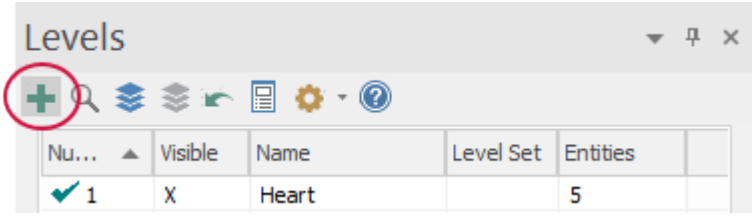


9. Select **Show Axes** and **Show Gnomons** to hide them in the graphics window.
10. Set that the **2D/3D** toggle on the **Status Bar** is set to **2D**. Setting your construction mode to 2D will place any geometry you create parallel to the current construction plane (**Top (WCS)**) at the current system Z depth.

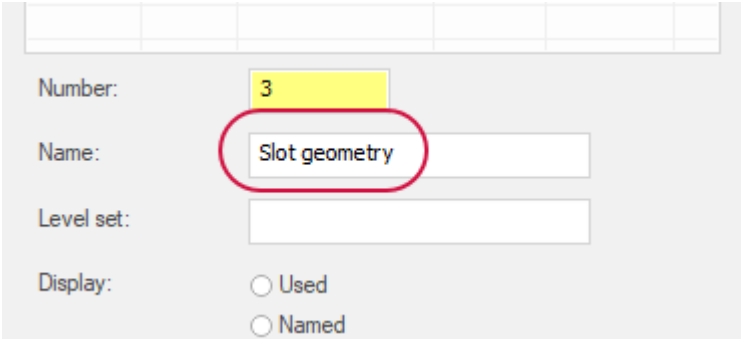
11. In the Planes Manager, click in the **C** column to set your construction plane (Cplane) to **Front**. The construction plane is the plane on which you will create the geometry.



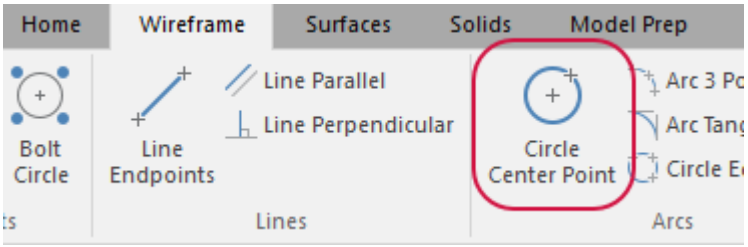
12. In the Levels Manager, click **Add a new level**.



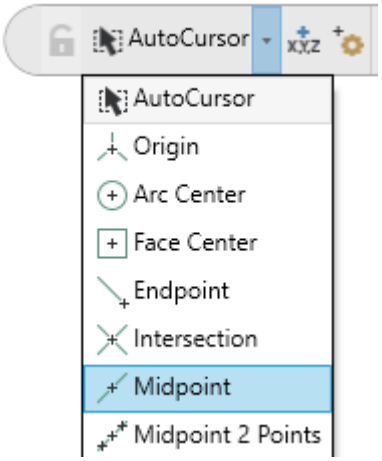
13. Enter **Slot geometry** in the **Name** field of the new level.



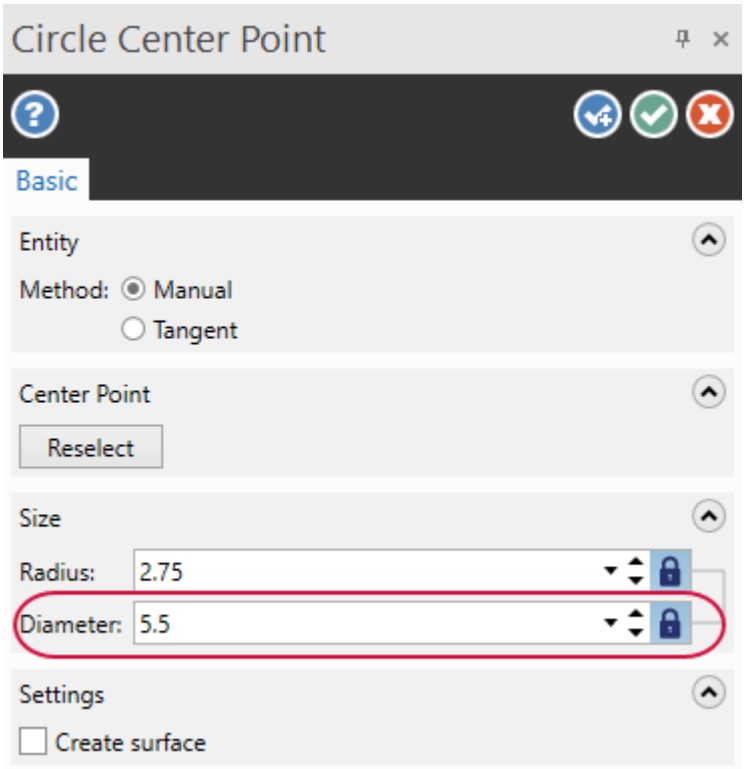
14. Right-click in the graphics window and set your view to **Front (WCS)**. Choose **Fit** to center your part in the graphics window.
15. On the **Wireframe** tab, click **Circle Center Point**.



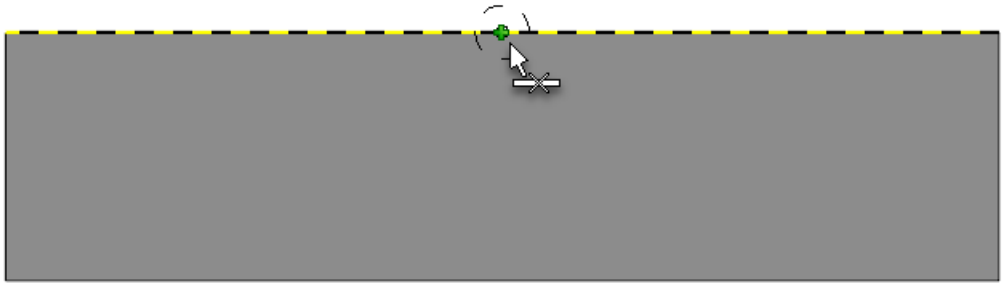
16. Select **AutoCursor**, **Midpoint** to set a snap to position.



17. In the **Circle Center Point** function panel, enter **5.5** in the **Diameter** field and then click the **Lock** icon to maintain that diameter.



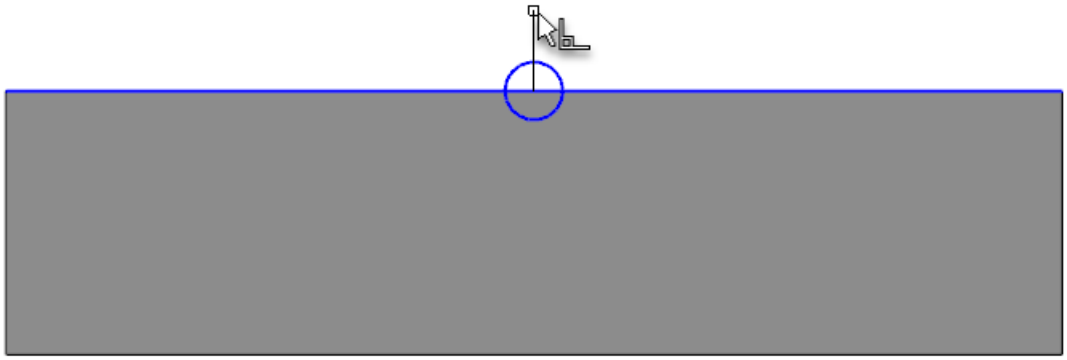
18. Hover over the top edge of the mold until the center point displays and click to set the location of the circle.



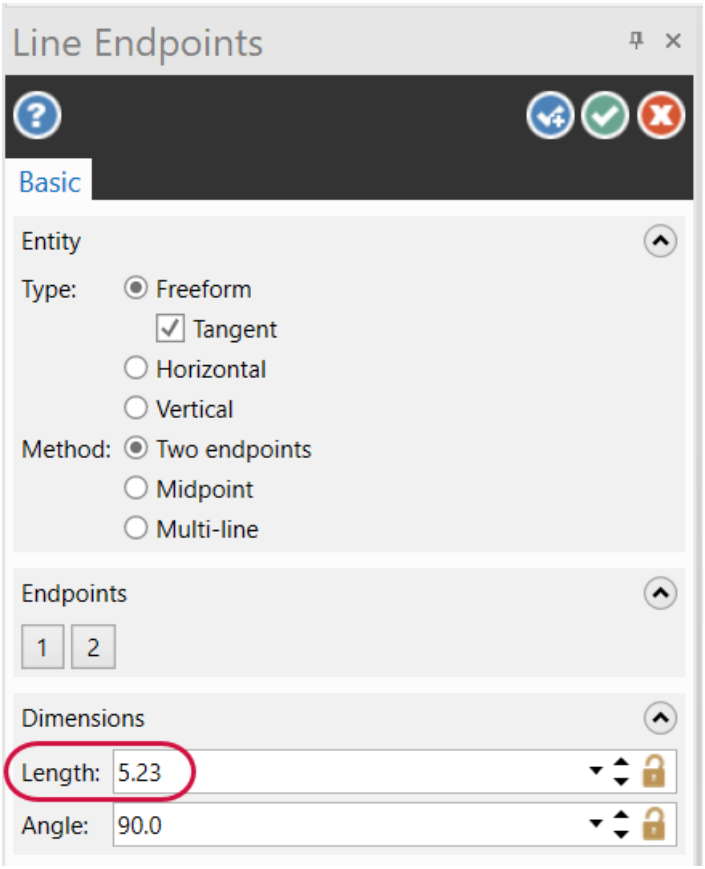
19. In the **Circle Center Point** function panel, unlock the diameter and then click **OK**.

> **NOTE**
> The function panel will maintain settings from session to session, unlocking the diameter before closing is a good practice.

20. On the **Wireframe** tab, click **Line Endpoints**.
21. Click the center point of the circle and drag up approximately 5 mm. Click to set the second endpoint.

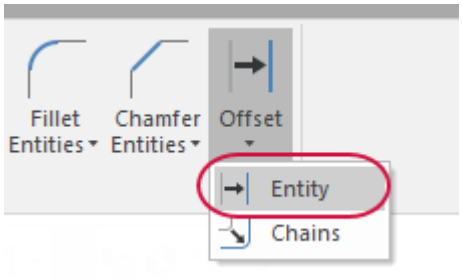


22. In **Line Endpoint** function panel, enter a **Length** of **5.23** and an **Angle** of **90**.

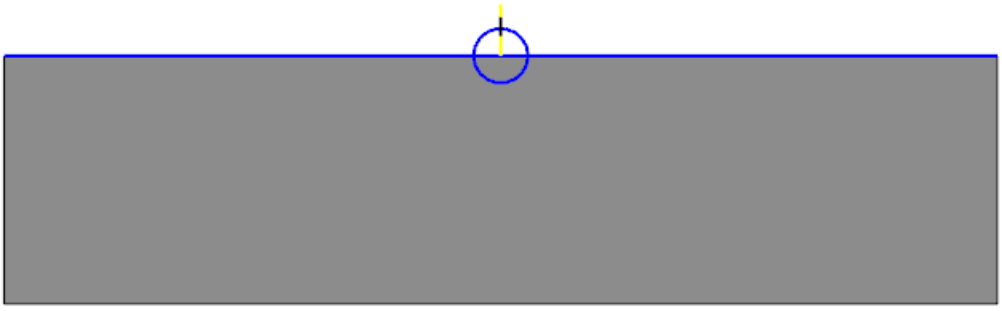


23. Click **OK** to accept the values.

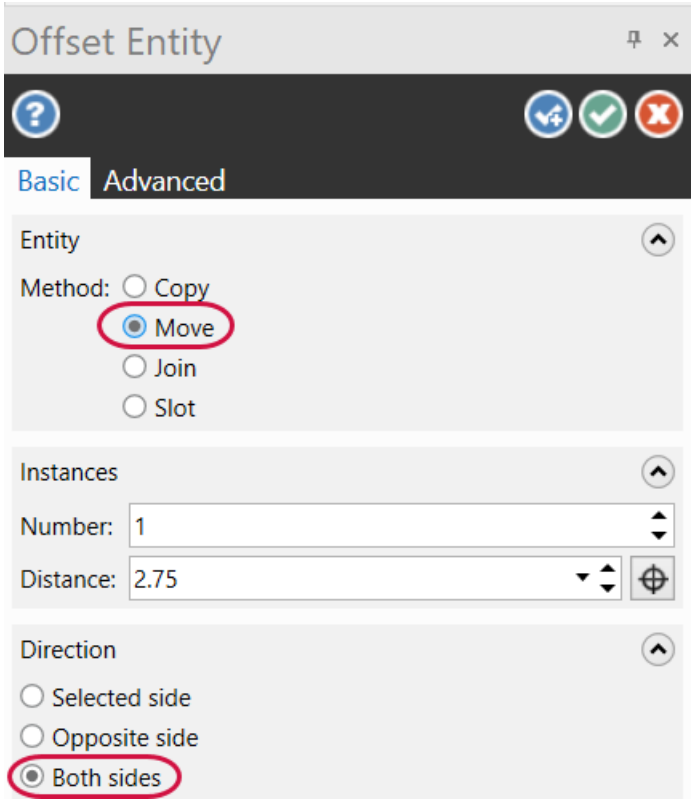
24. On the **Wireframe** tab, click **Offset Entity**.



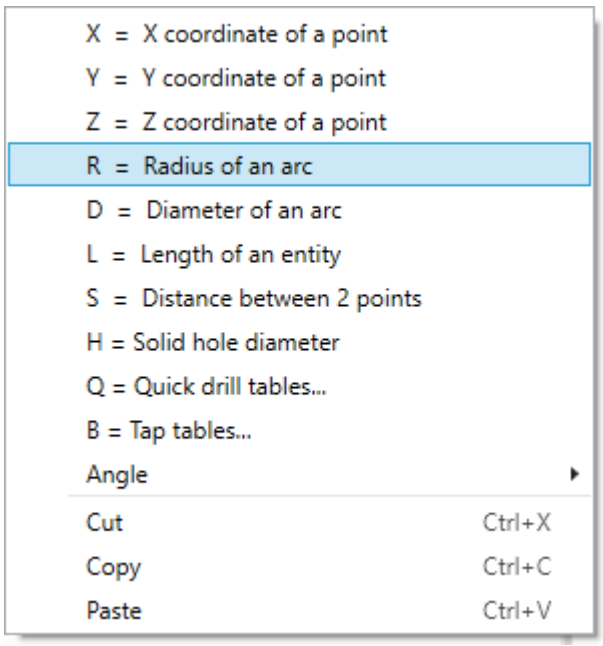
25. Select the line you just drew and then click to the left in the graphics window to indicate the direction to offset.



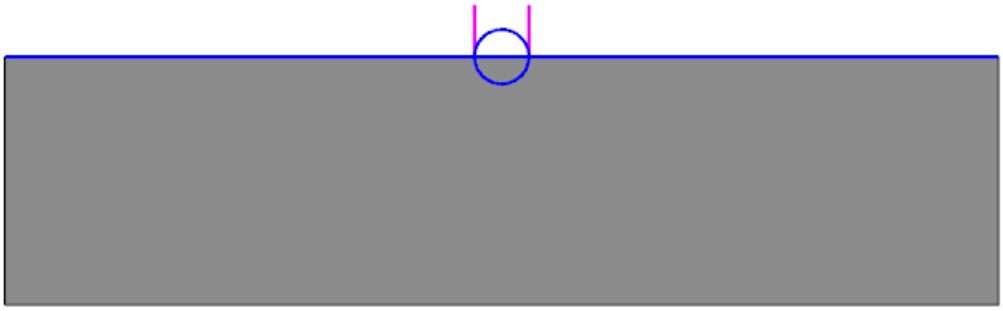
26. On the **Offset Entity** function panel, choose **Move** and **Both sides** for the **Direction**.



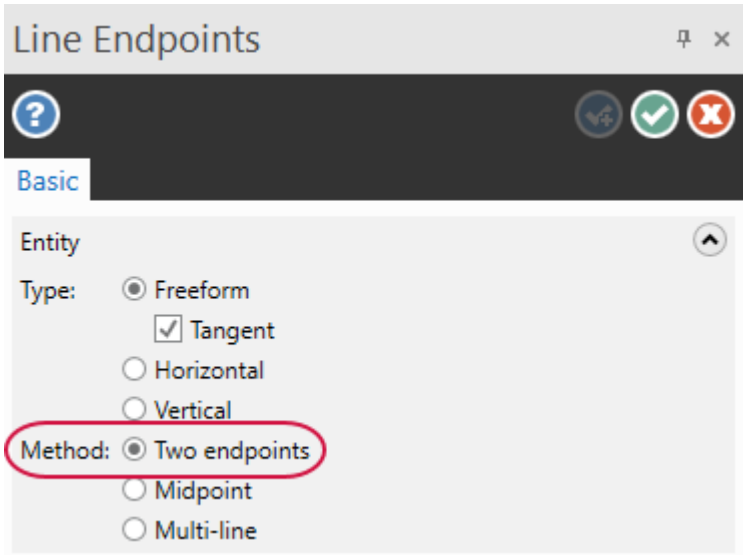
27. In the **Distance** field, right-click and select **R = Radius of an arc**.



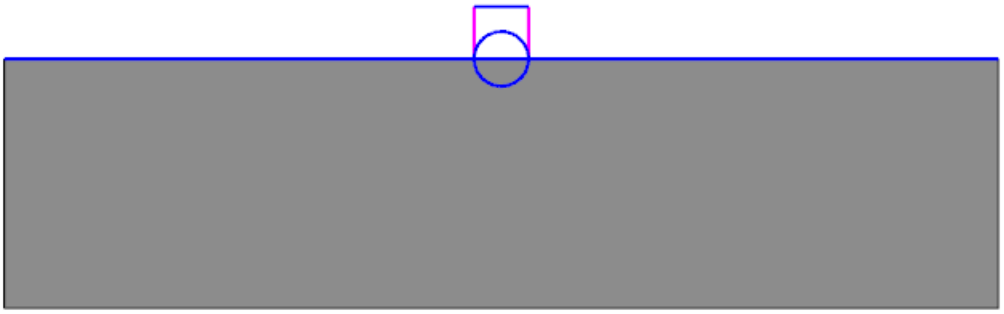
28. Select the arc. The lines move to be tangent to the arc.



29. Click **OK** in the **Offset Entity** function panel.
30. On the **Wireframe** tab, click **Line Endpoints**. Ensure that the **Method** is **Two endpoints**.



31. Draw a line between the two endpoints of the lines and click **OK**.

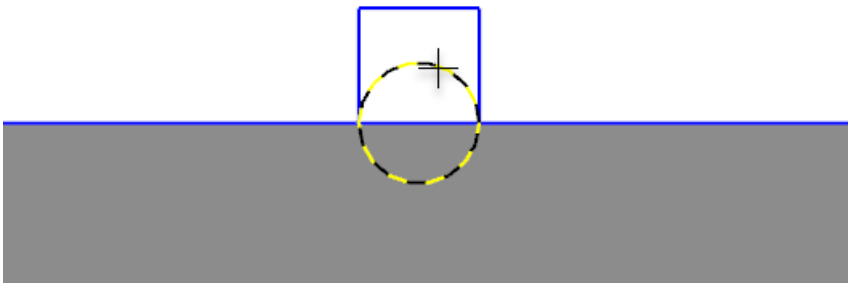


32. Right-click in the graphics window and click **Clear Colors** to return the purple results color to the default color.

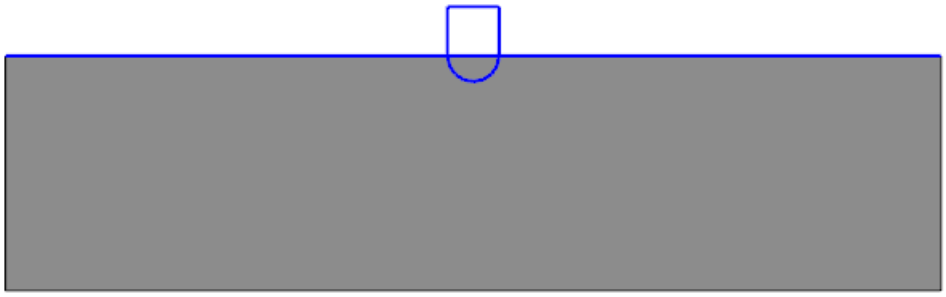
33. On the **Wireframe** tab, click **Divide**.

34. In the **Divide** function panel, choose **Trim**.

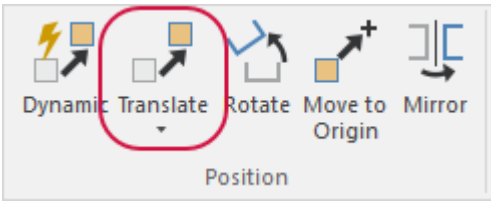
35. Click the top of the arc to remove that section.



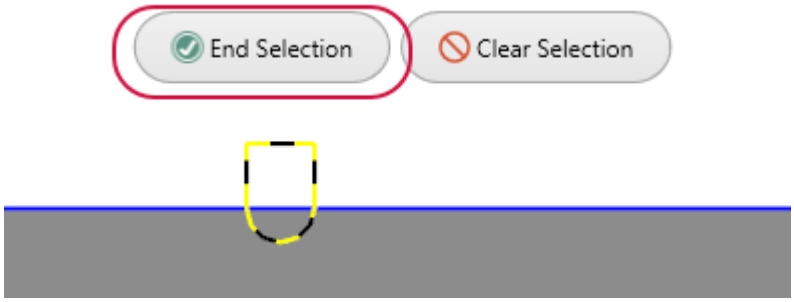
36. Click **OK** in the **Divide** function panel. The geometry should look like the following image.



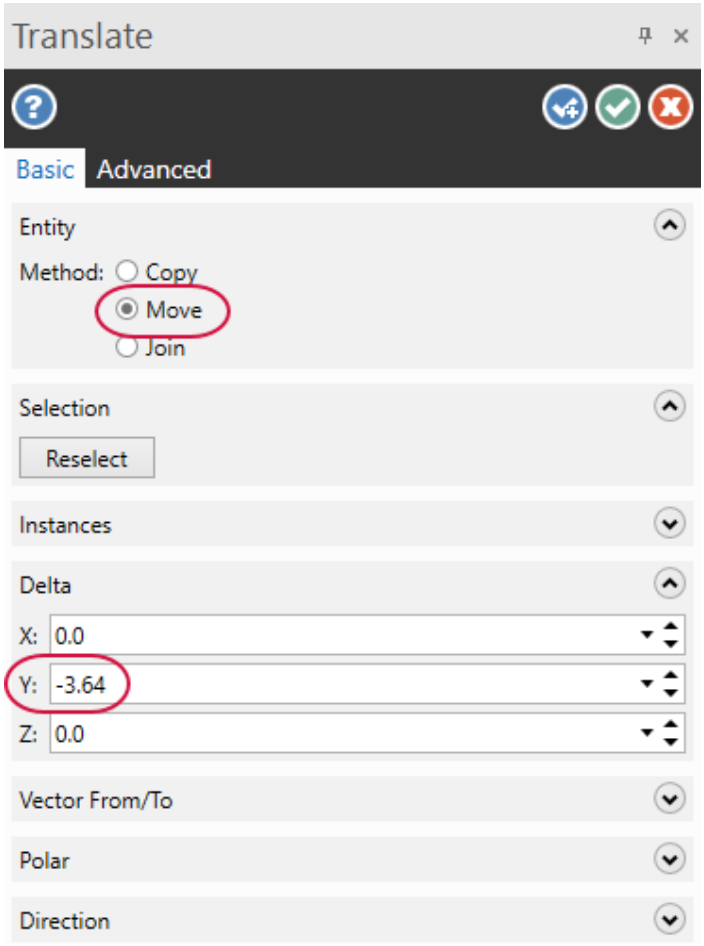
37. On the **Transform** tab, click **Translate**. You will be using the Translate function to move the geometry into the proper position.



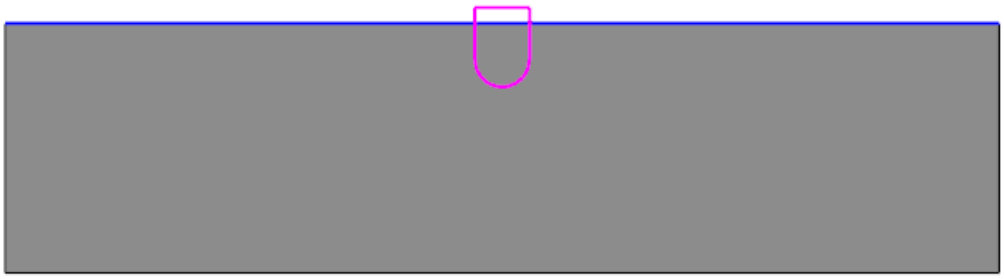
38. Select the geometry you created and then click **End Selection**.



39. In the **Translate** function panel select **Move** as the **Method** and enter **-3.64** in the **Y** axis field. Entering a negative value places the circle below the Y axis.



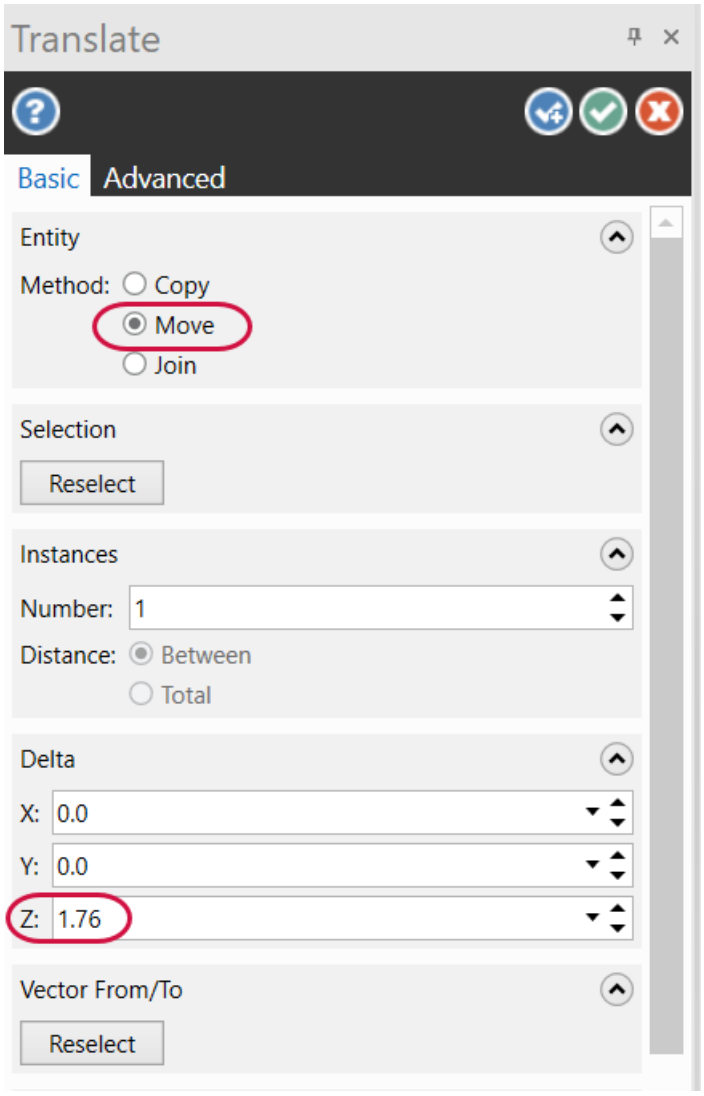
40. Click **OK** to accept the values.



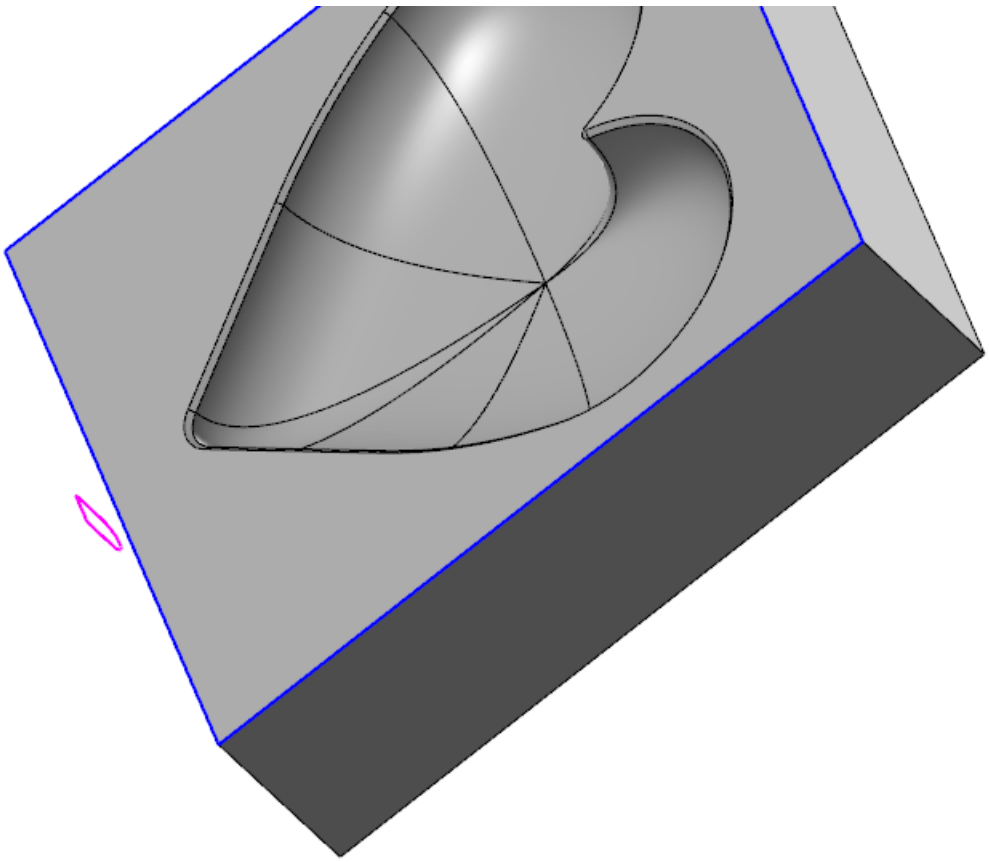
41. On the **Transform** tab, click **Translate**.

42. Click the **Quick Mask**, **Select all results entities** to quickly select the geometry you just moved.

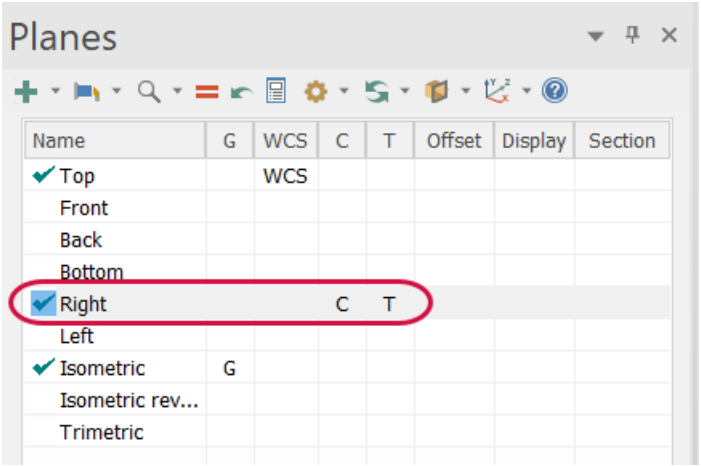
43. In the **Translate** function panel, enter **1.76** in the **Z** field to move the geometry forward.



44. Click **OK**. Rotate your part to view the geometry.

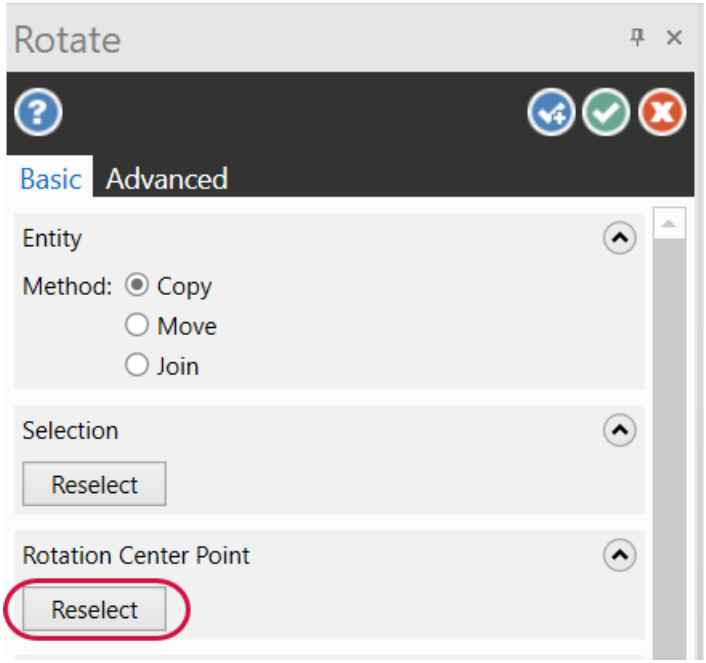


45. In the Planes Manager, set the construction plane to **Right**.

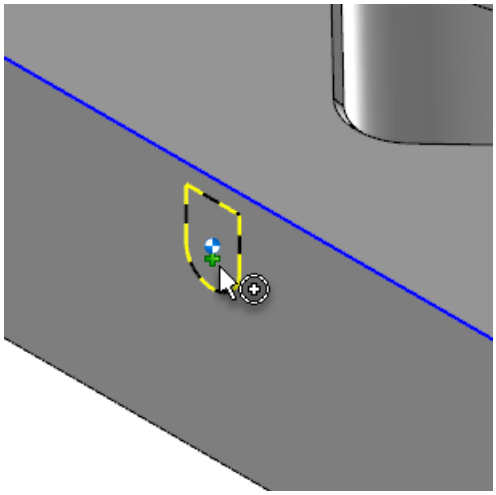


46. On the **Transform** tab, click **Rotate**.
47. Select the geometry by clicking the entities or use the **Quick Mask**, **Select all results entities**, and then click **End Selection**.

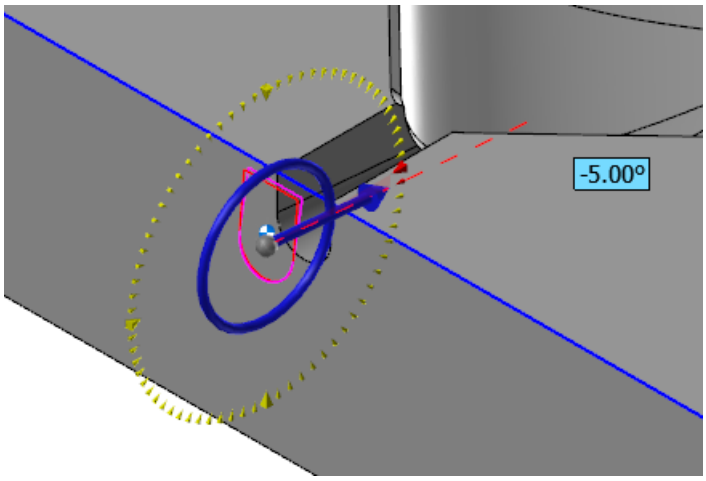
48. In the **Rotate** function panel, click **Rotation Center Point Reselect.**



49. In the graphics window, click the center point of the original arc which appears as a green AutoCursor point, (not the blue and white rotation point).

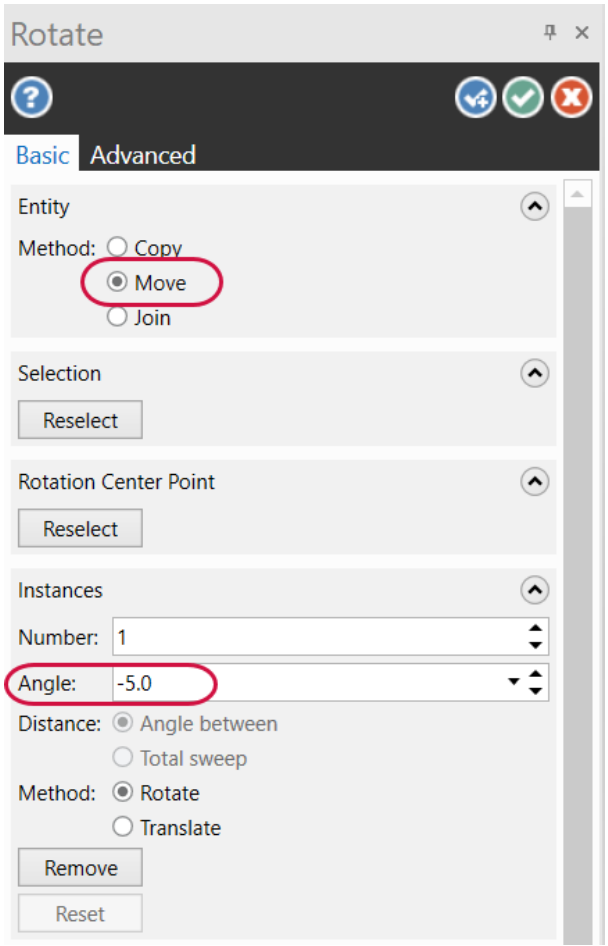


50. Hover over the Dynamic Gnomon until it highlights and then click and rotate it to **-5.00** degrees, or enter the value in the on-screen field and press [**Enter**].



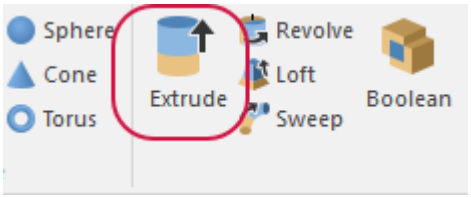
> **NOTE**
> Alternately, you can also enter **-5.00** in the **Angle** field of the **Rotate** function panel.

51. Choose **Move** in the **Rotate** function panel and click **OK**.



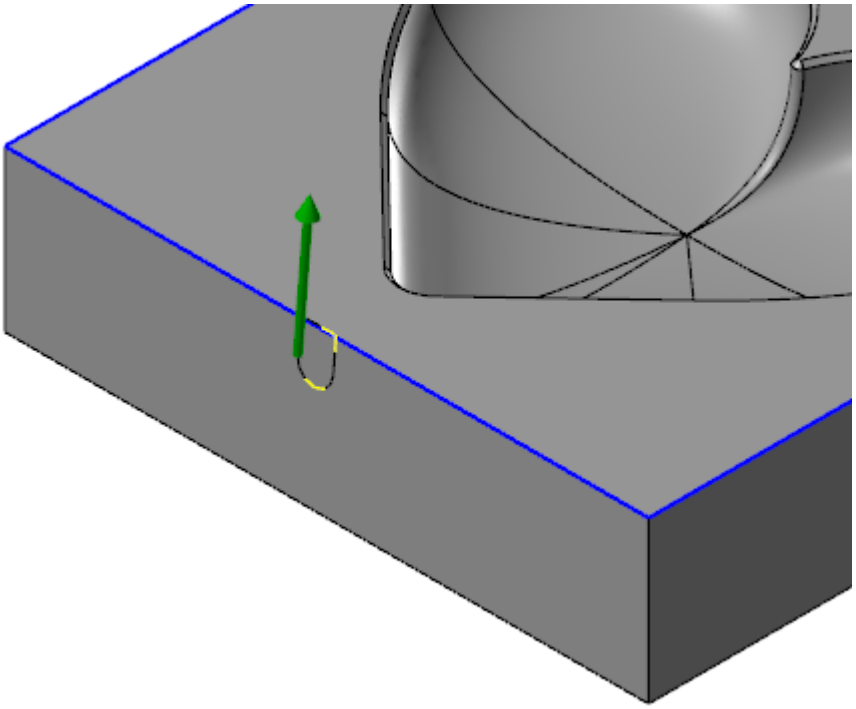
52. Right-click and choose **Right (WCS)** to verify the results.

53. Return the view to **Isometric (WCS)**.
54. On the **Solids** tab, click **Extrude**.

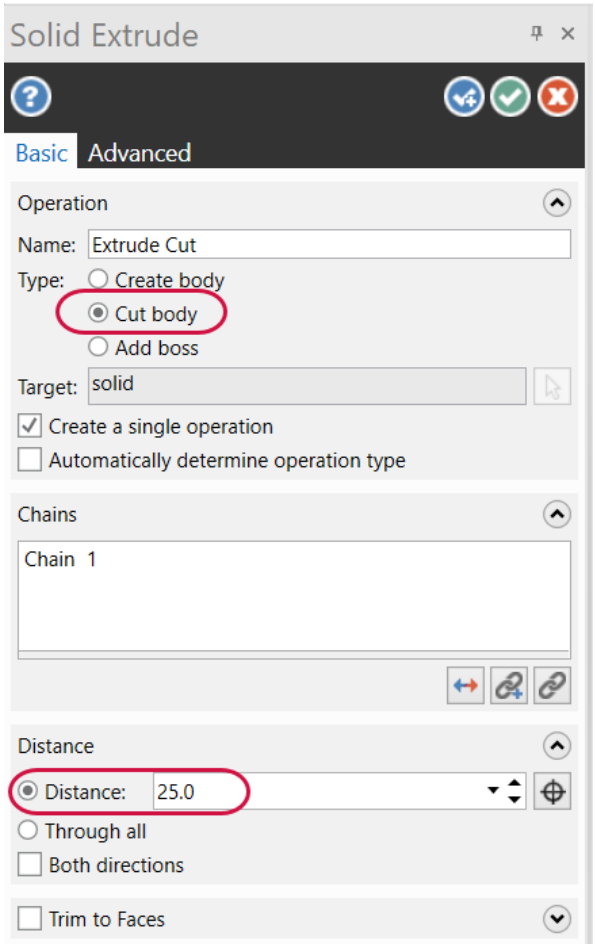


The **Wireframe Chaining** dialog box displays.

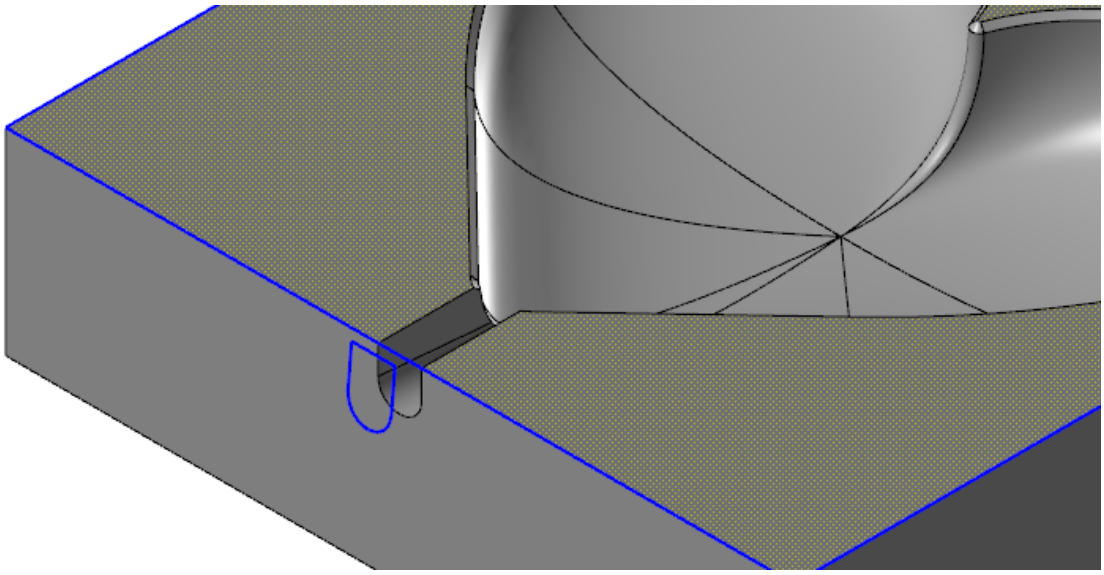
55. Select the geometry and then click **OK**.



56. In the **Solid Extrude** function panel, choose **Cut Body** as the **Type**, and enter a **Distance** of **25.0**.



57. Click **OK**.



58. Return to the Levels Manager and notice that the four entities you just created on level **3**, **Slot geometry**. Experiment with the visibility to see what shows in the graphics window.
59. Save your file.

SHOW US WHAT YOU LEARNED!

Can you answer these questions?

1. You can use an STL file in a Solids Boolean remove operation.
    a. True
    b. False
2. The 2D construction mode places geometry parallel to the current construction plane.
    a. True
    b. False
3. Which Solid Boolean option combines two solids into one solid?
    a. Add
    b. Remove
    c. Common
4. Which Solid Boolean option leaves an impression of one solid in another solid?
    a. Add
    b. Remove
    c. Common
5. Which of the following does **not** set the size of the fillet in the Face to Face Fillet function?
    a. Diameter
    b. Radius
    c. Width
6. Which function on the Transform tab changes the angle of an entity or a group of entities.
    a. Dynamic
    b. Translate
    c. Rotate
    d. Project
7. List some characteristics in a model that can hurt the chances of a casting being successfully released from the mold.

CHAPTER 2

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 to be machined, and the ability to machine around the exterior and the interior of the piece successfully, while avoiding the fixture itself.

For this tutorial, the fixture selected is a standard machinist's vise. The vise is elevated on parallels to allow machining of the exterior and interior of the part.

Goals

- Explore the machinist vice provided with this tutorial
- Understand fixturing options
- Experiment with moving your part to the machine origin

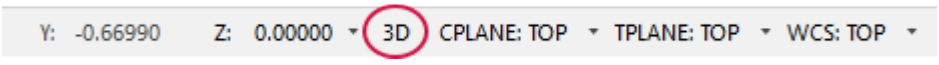
Exercise 1: Moving the Mold to the Origin

In this exercise, you will move the candy mold to the origin for the provided machinist vice on the default the mill machine.

> **WARNING**
> This project is completed using a specific machine and machinist vise model provided with this tutorial. The setup and fixturing of your machine may vary.

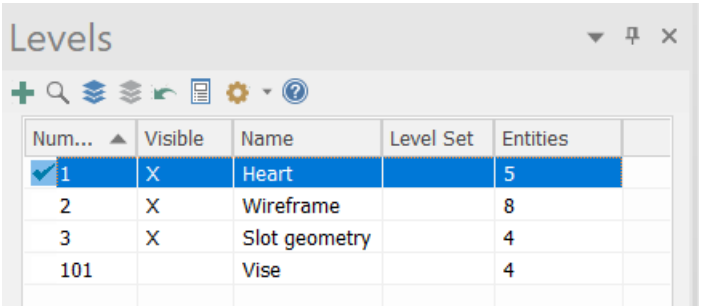
1. Your file, `Heart mm-XXX.mcam`, which you saved from the previous chapter should be open in Mastercam.
2. On the **View** tab, click **Show Axes** and **Show Gnomons** to activate them. Use the function's drop downs to choose which axes and gnomons display in the graphics window.

3. Set the **2D/3D** toggle on the Status bar to **3D**.

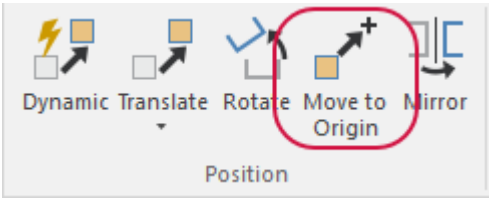


4. Right-click in the graphics window and set your view to **Isometric (WCS)**.
5. Choose **Fit** to center your part in the graphics window.

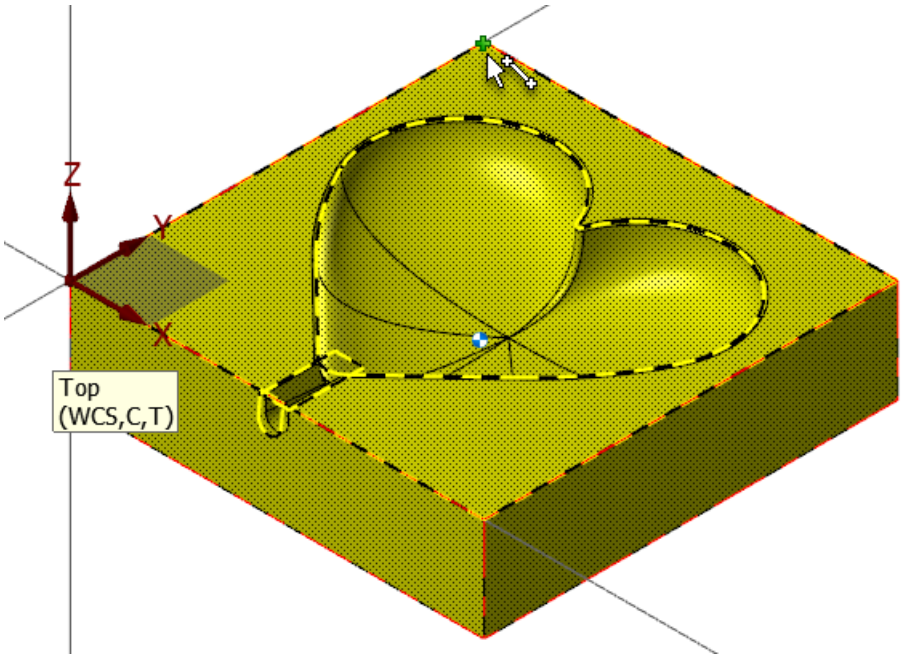
6. In Levels Manager, make the **Heart**, **Wireframe**, and **Slot** geometry levels visible and make level **1** the active level.



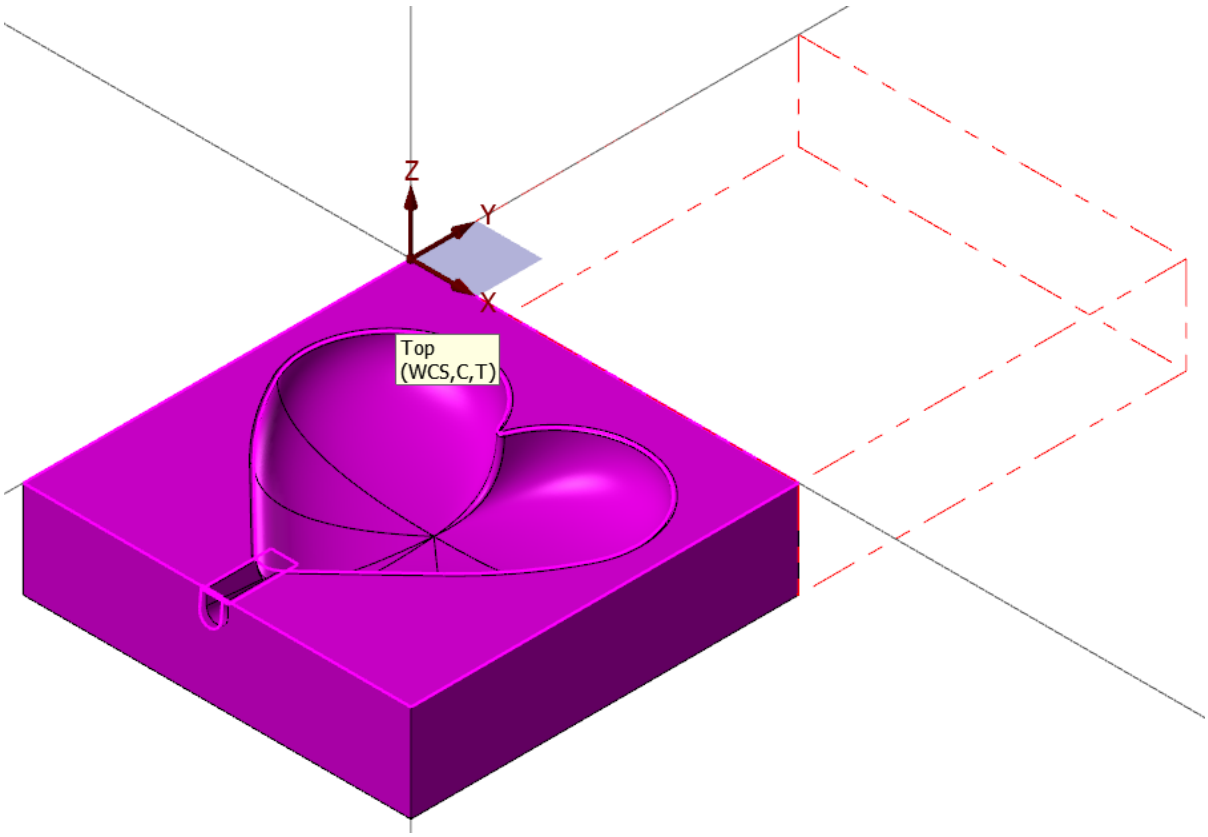
7. On the **Transform** tab, click **Move to Origin**.



8. Click the upper left corner of the mold as shown in the following image.



9. The mold moves to the origin.



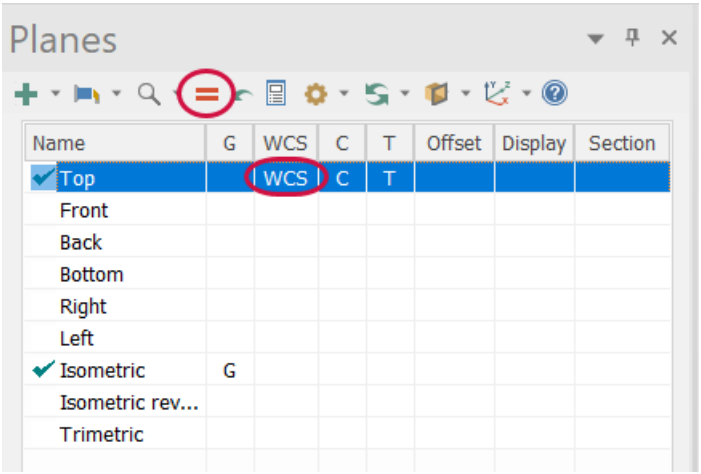
10. The mold color is now purple which is the results color for a transformation. Click **Clear Colors** on the **View** tab to return all entities their default colors.
11. You can click **Show Axes** and **Show Gnomons** to hide them in the graphics window.

Exercise 2: Choosing a Machine and Setting Up Stock

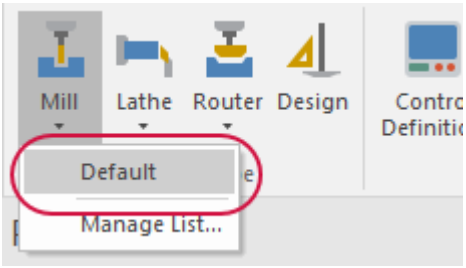
In this exercise, you will choose a machine and setup your stock.

1. Your file, `Heart mm-XXX.mcam`, which you saved from the previous exercise should be open in Mastercam.
2. Ensure that the **2D/3D** toggle on the Status bar is set to **3D**.
3. Click the **Planes** tab to bring the Planes Manager to the forefront.

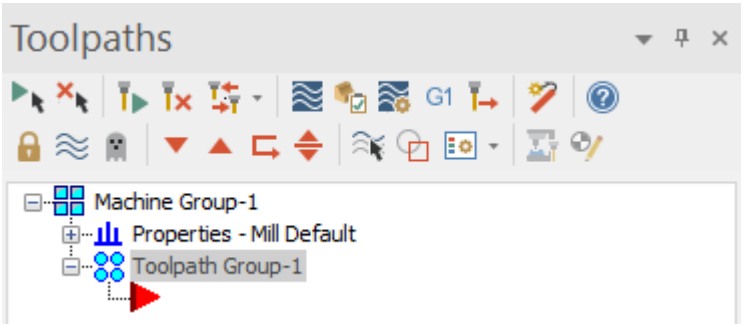
4. Click in the **WCS** column for the Top plane and then click the **Set your current WCS, construction plane, and tool plane with their origins to the selected plane**.



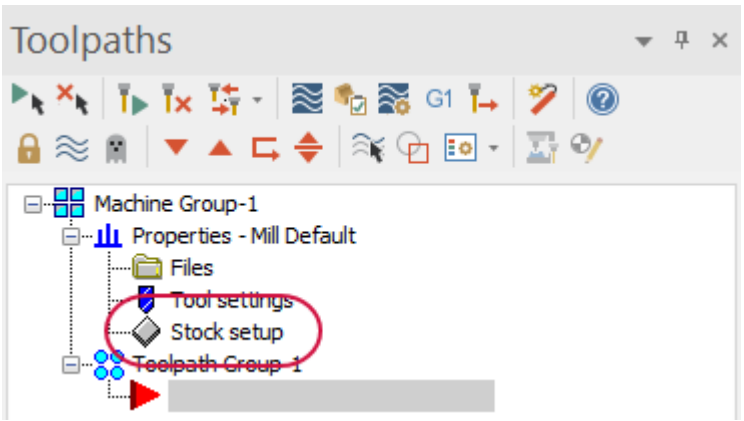
5. On the **Machine** tab, choose the **Default** mill machine from the drop-down.



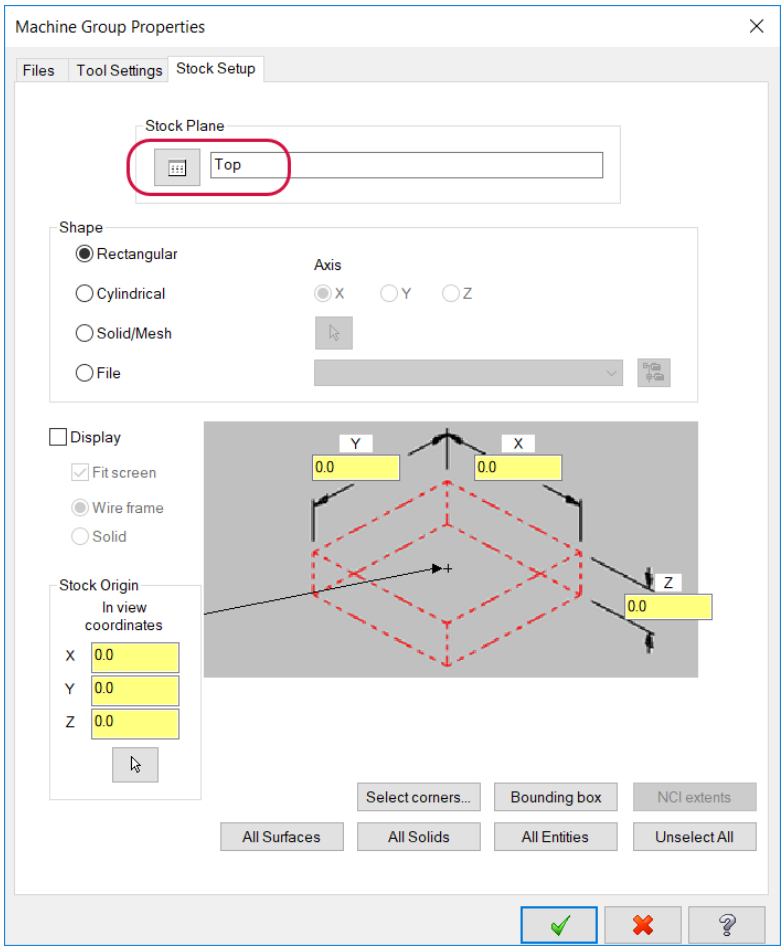
6. Click the **Toolpaths** tab to bring the **Toolpaths Manager** to the forefront. Note that **Machine Group 1** is the **Mill Default MM**.



7. Expand the **Properties** group.

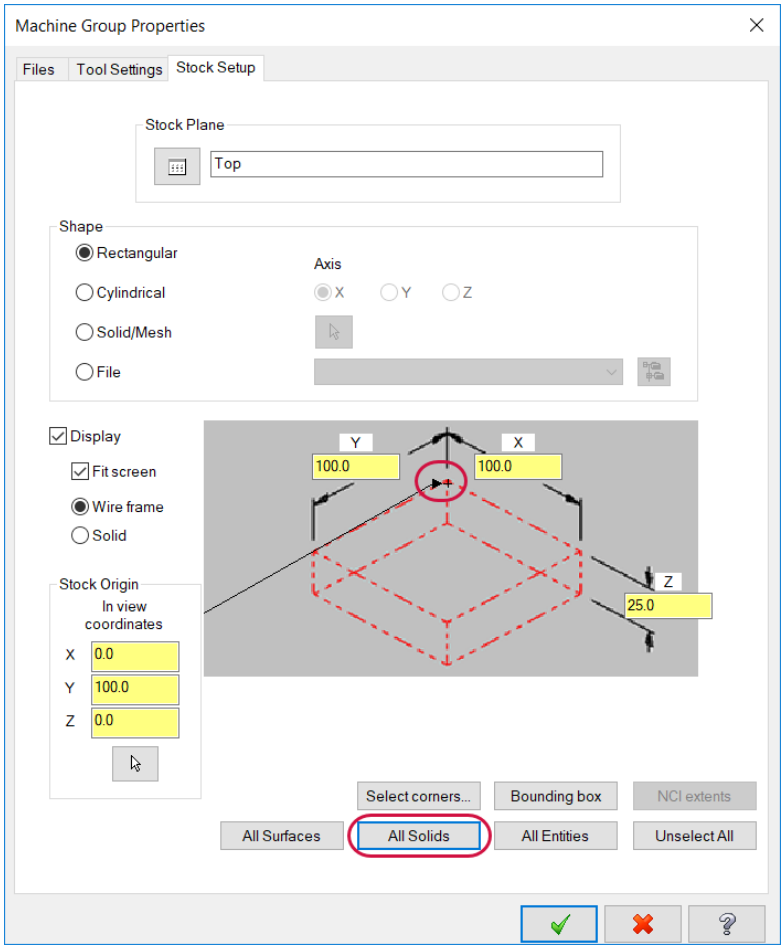


8. Click **Stock setup** to open the **Stock Setup** page of the **Machine Group Properties** dialog box. Confirm that the **Stock Plane** is set to **Top**.



NOTE
If the **Stock Plane** is incorrect, you can click the icon and choose the correct plane from the **Plane Selection** dialog box.

9. In the illustration, choose the top, back point to set the **Stock Origin** and then click **All Solids.**



The values in the **Y**, **X**, and **Z** fields update to the measurements corresponding to the stock you created (**100.0**, **100.0**, **25.0**).

10. Click **OK**.
11. Save your file.

SHOW US WHAT YOU LEARNED!

Can you answer these questions?

1. If you do not need to cut the outside edge of a part, a vise can be used to hold the stock.
    a. True
    b. False
2. You can machine (cut) on the outside of the stock only if the cut depth does not go deeper than the amount of stock sticking out above the vise jaws.
    a. True
    b. False
3. The Move to Origin function...
    a. Moves the origin to a specific location on the part
    b. Moves the part to the origin
    c. None of the above
4. In Stock Setup, the only way to enter the size of the stock is to manually type in the value.
    a. True
    b. False
5. In Stock Setup, you can display the stock in the graphics window to confirm the orientation and location of the part.
    a. True
    b. False
6. What are some benefits of a vise versus hold-down clamps when machining a part?
7. What should you consider when you use a vise to hold a part?
8. List some things you can do to ensure that your part will more easily release from your mold?

CHAPTER 3

ROUGH MOLD PROGRAMMING

A roughing toolpath is typically used to remove large areas of unwanted material to produce a shape that is similar to the final shape. Using a roughing toolpath followed by a finish toolpath can reduce the time it takes to machine a part.

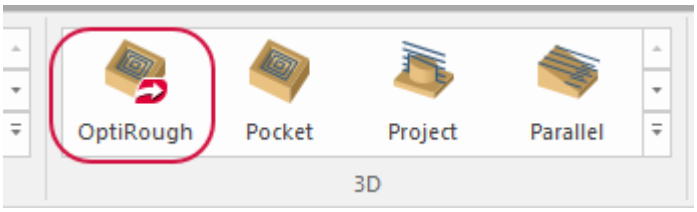
Goals

- Program a rough toolpath
- Understand the concepts of machining and avoidance geometry
- Understand and apply a containment boundary to a toolpath
- Explore tool libraries including filtering by tool type
- Experiment with toolpath verification using Mastercam Simulator

Exercise 1: Creating the OptiRough Toolpath

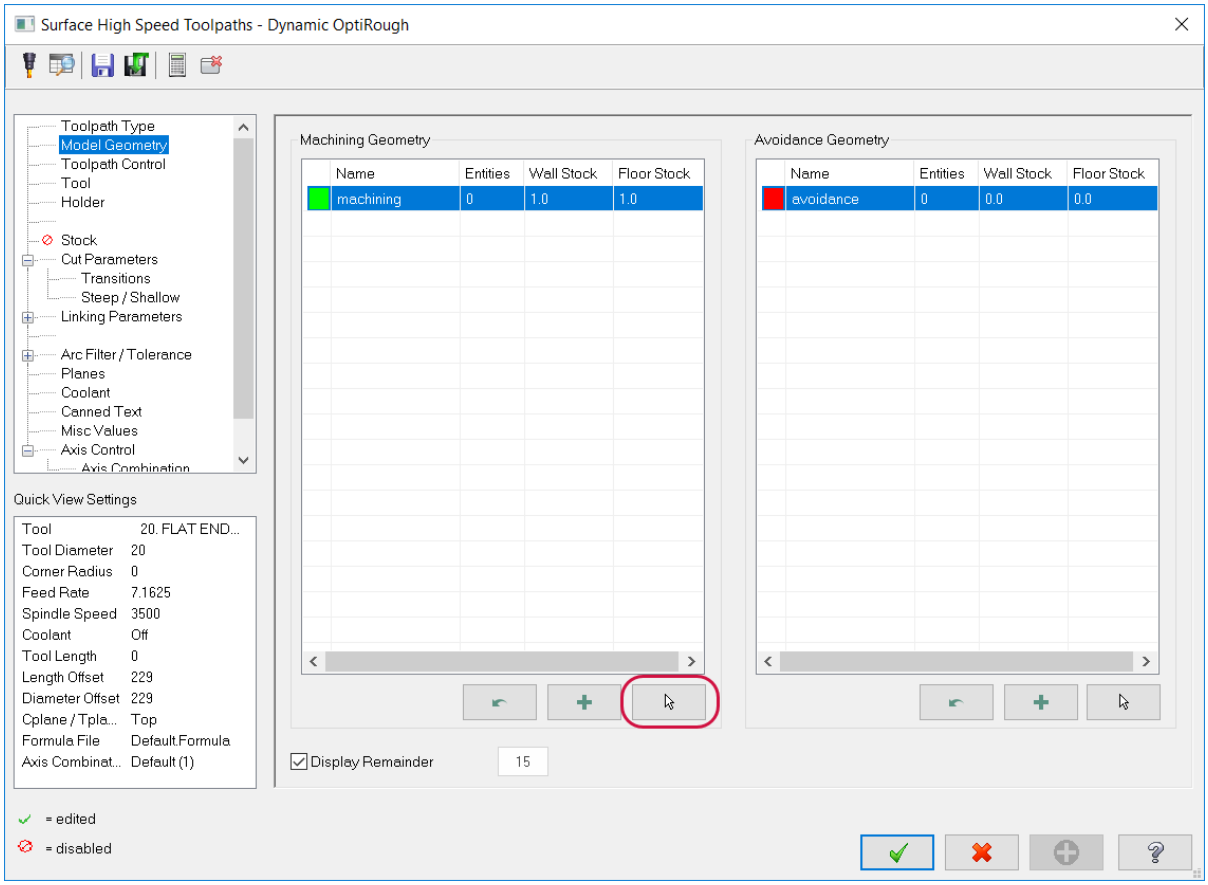
In this exercise, you will program a OptiRough toolpath to rough out the heart mold.

1. Your file, `Heart mm-XXX.mcam`, which you saved from the previous chapter should be open in Mastercam.
2. Right-click and choose **Fit** and set your view to **Isometric (WCS)**.
3. In the Levels Manager, make sure the Heart and Wireframe levels are visible.
4. On the **Mill Toolpaths** contextual tab, click **OptiRough** in the 3D group.

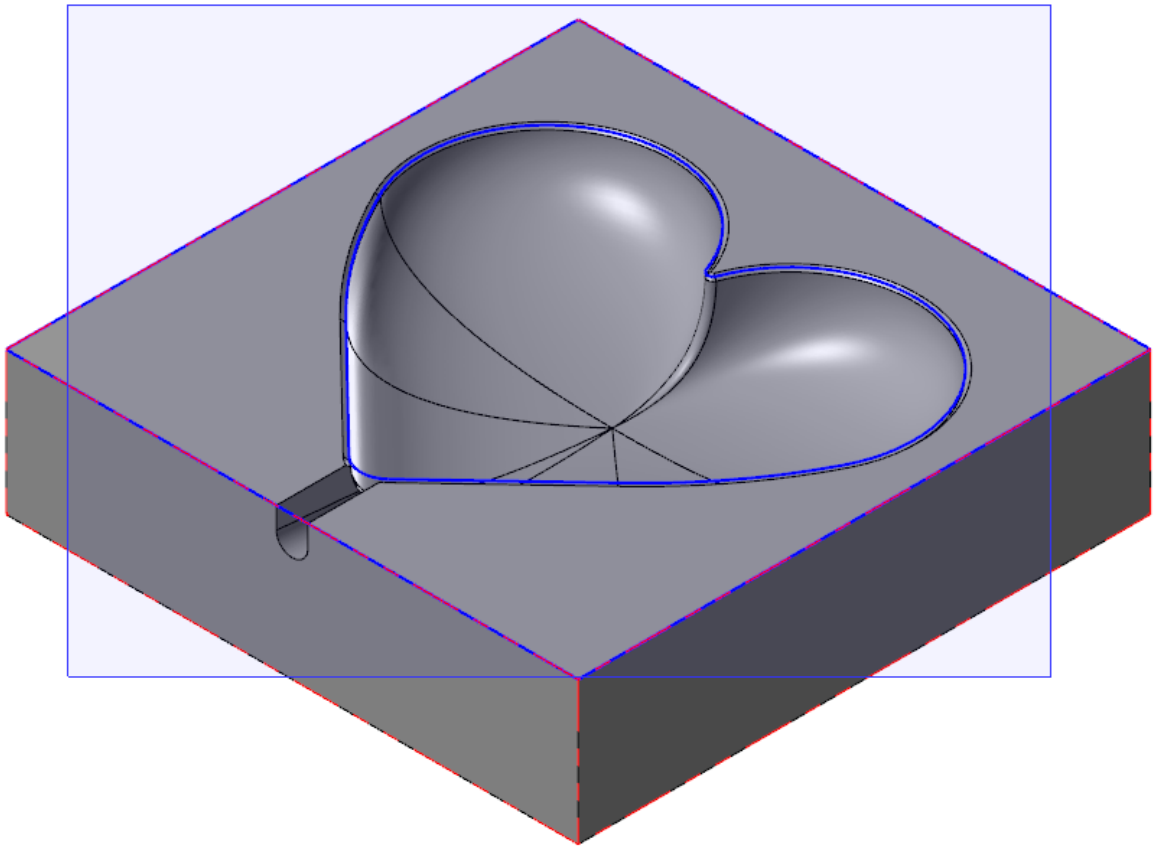


The **Model Geometry** page of the **Dynamic OptiRough** dialog box displays.

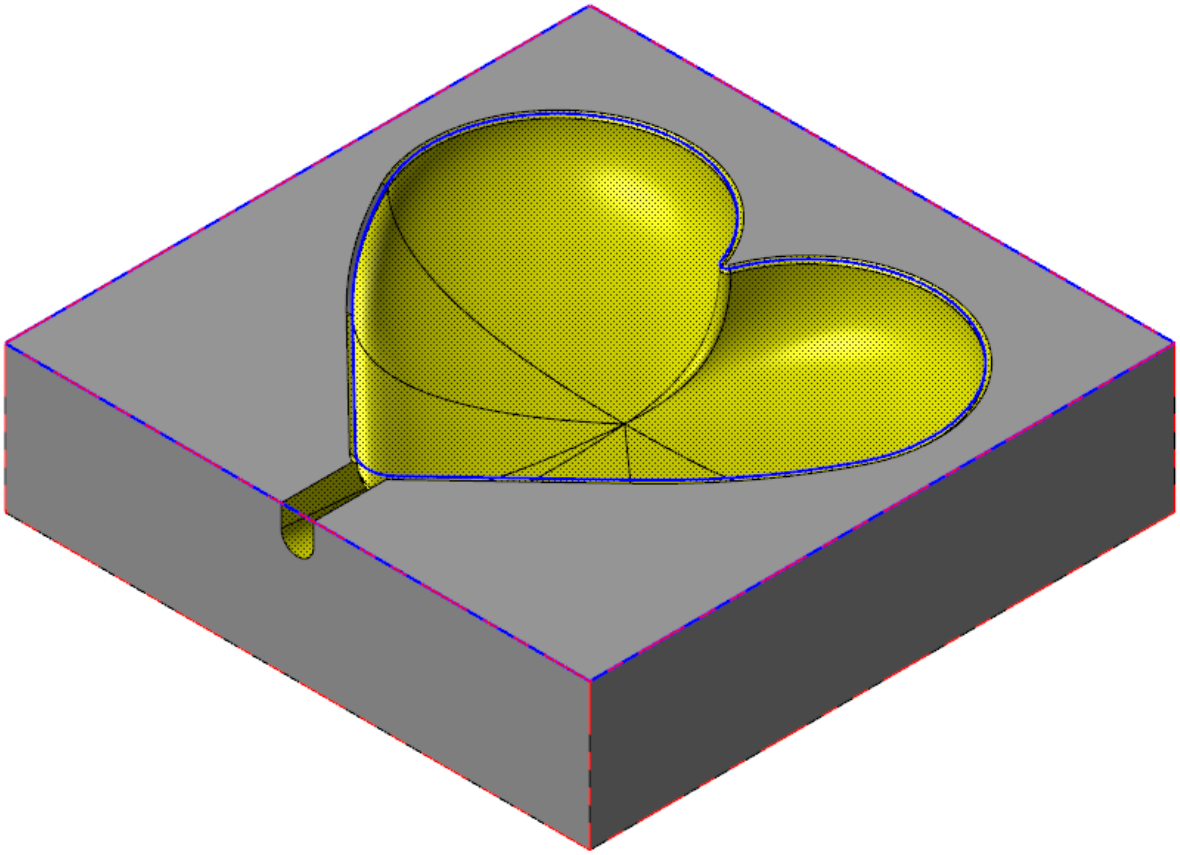
5. Click **Select entities** in the **Machining Geometry** section.



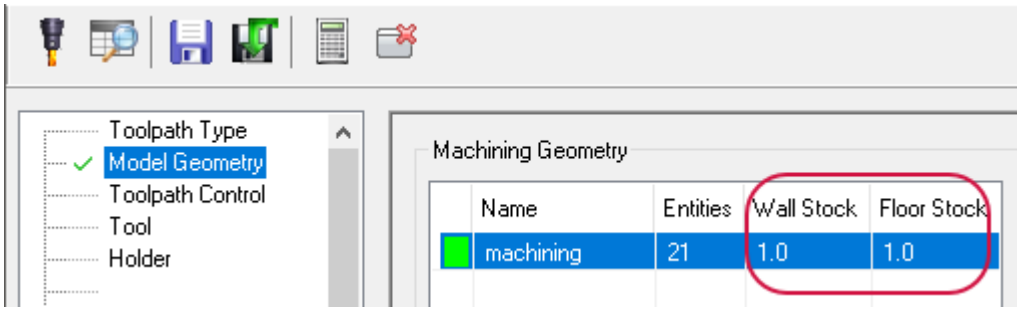
6. Select the heart and slot geometry. You can select each section or use window select as shown in the following image.



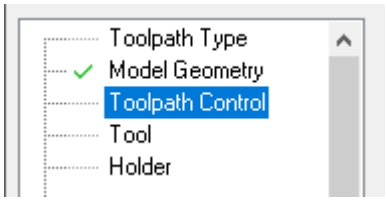
7. Rotate the part and make sure that all parts of the mold to be machine are selected.



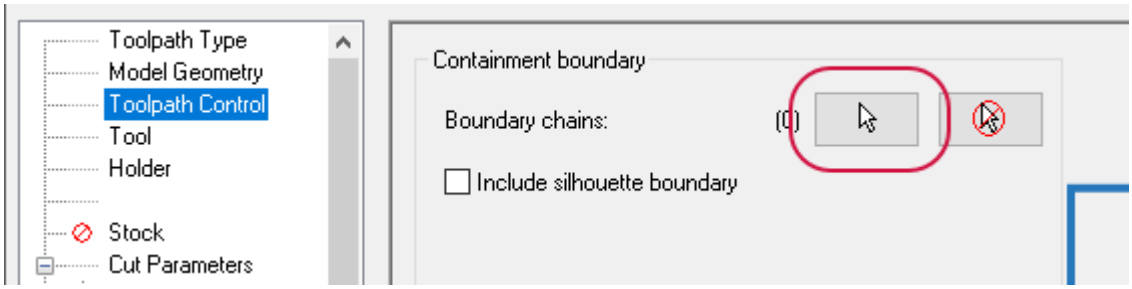
8. Click **End Selection** to return to the **Model Geometry** page.
9. For a rough toolpath you will designate a certain amount of stock to leave on the part. This stock will be removed with the finish toolpaths. Double-click the**Wall Stock** and **Floor Stock** fields of **Machining Geometry** to activate them and enter **1.0**.



10. Click the **Toolpath Control** page in the tree control.

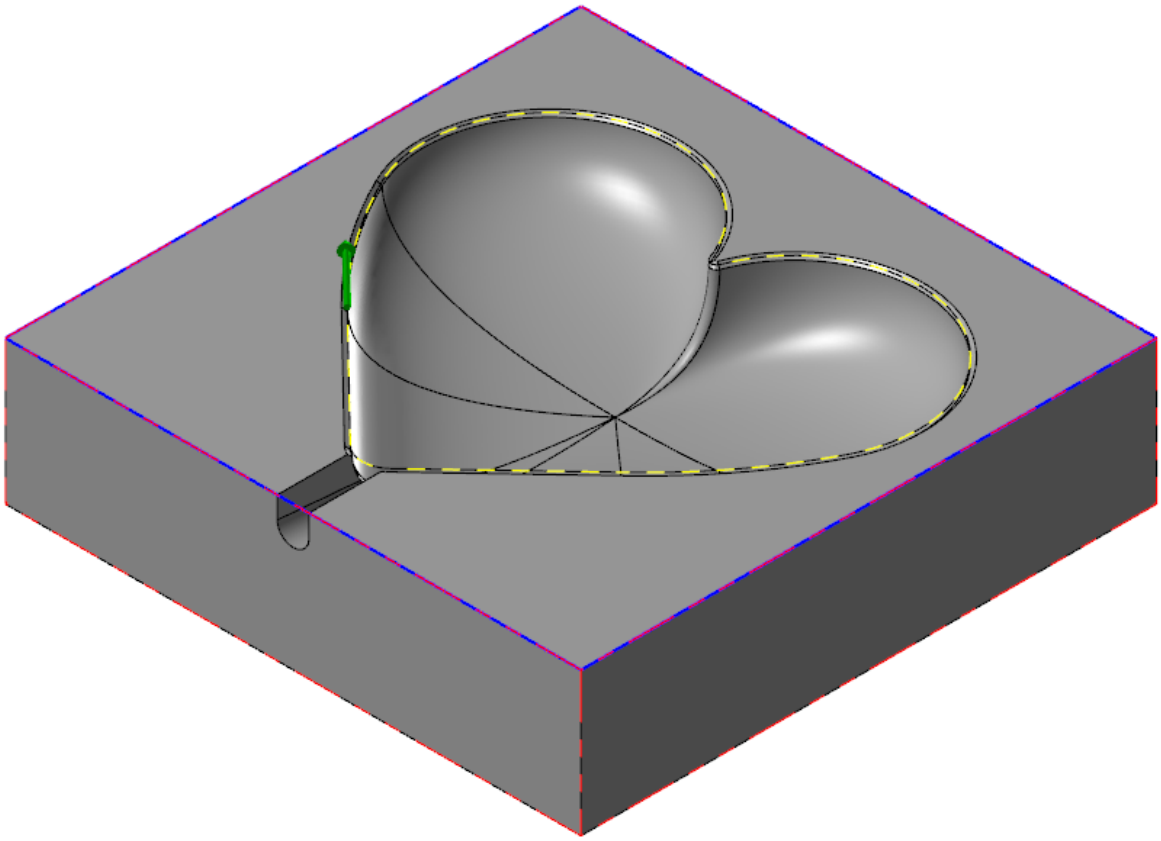


11. In this page you will choose the containment boundary to limit the area that will be machined. Click the **Boundary chains** select arrow.



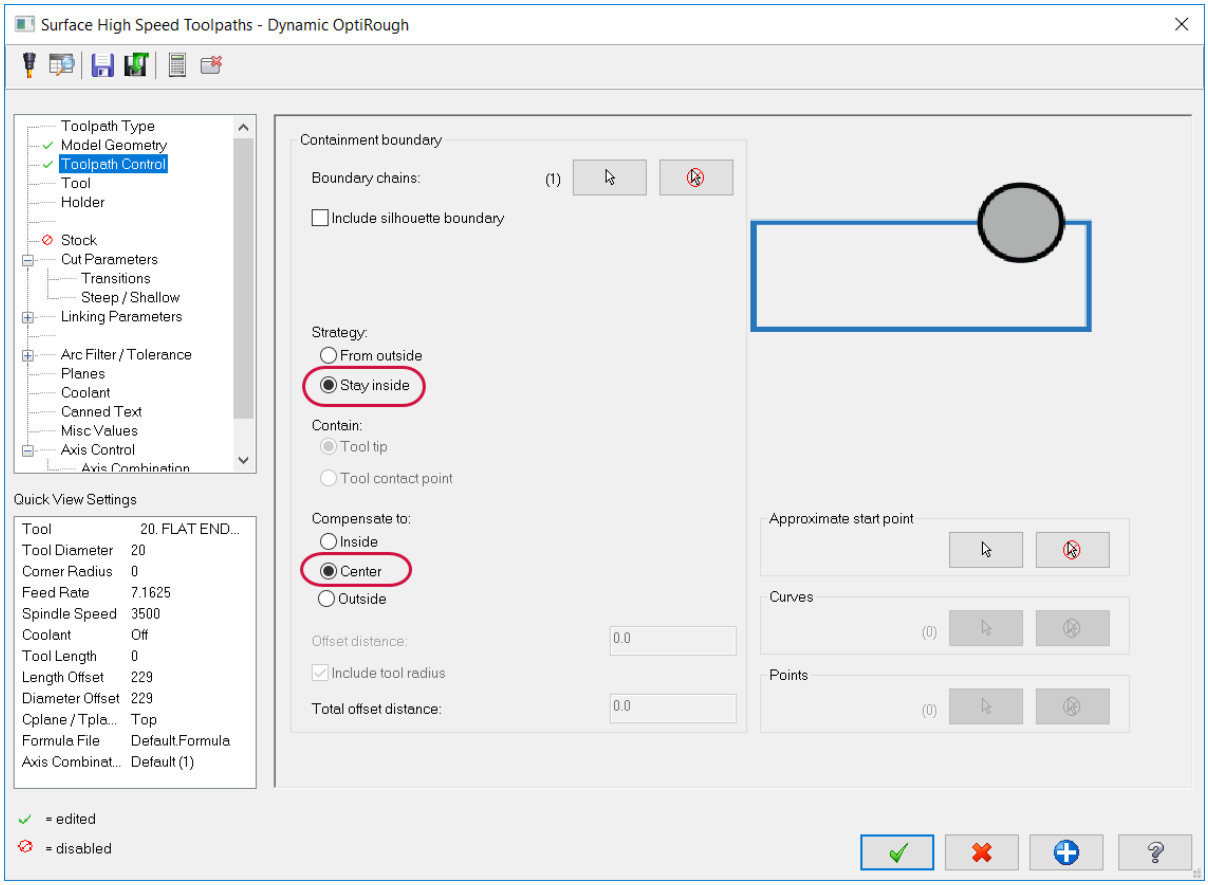
The **Wireframe Chaining** dialog box displays.

12. Hold down the [**Shift**] key and select a segment of the containment boundary wireframe. [**Shift+click**] selects all tangent segments.

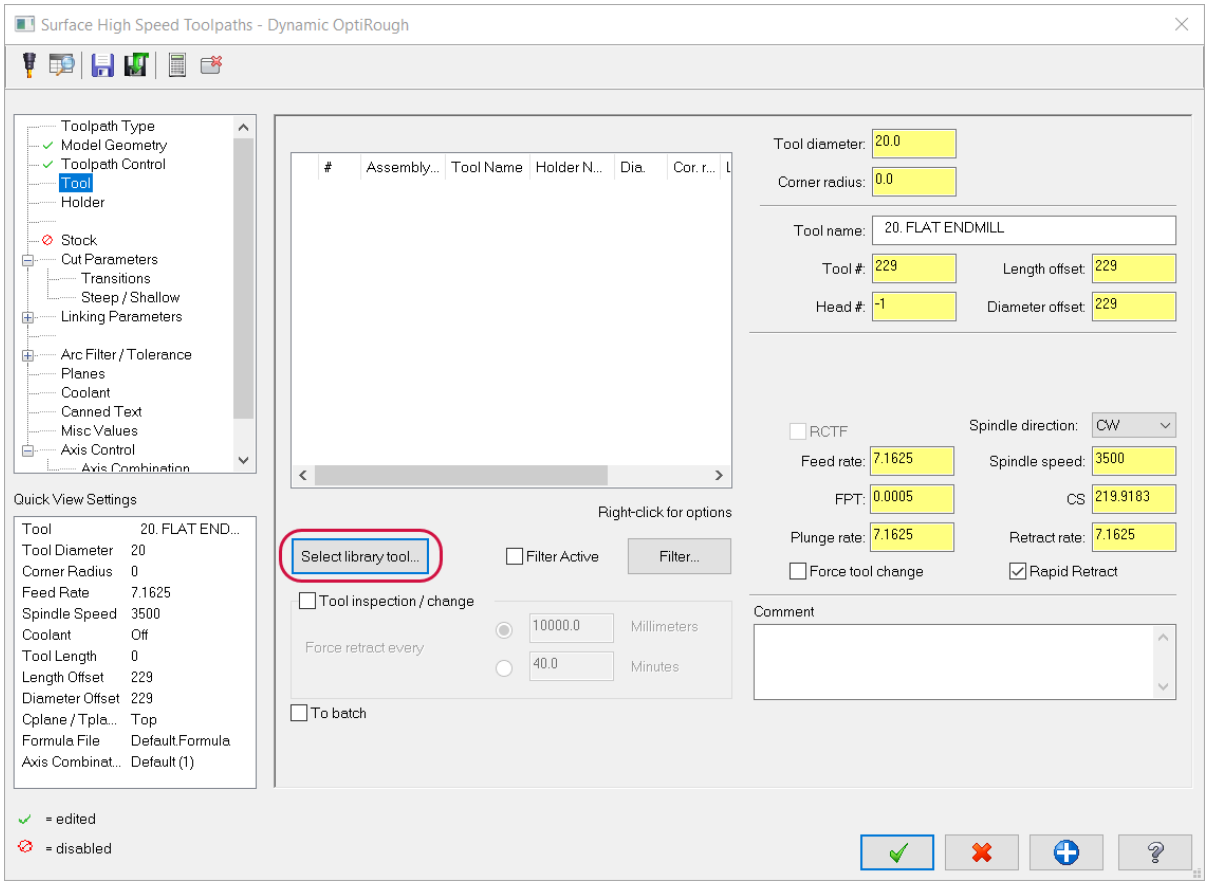


13. Click **OK** to close the **Wireframe Chaining** dialog box and return to the **Toolpath Control** page.
14. Select **Stay inside** for the **Strategy** to keep the toolpath inside the selected machining geometry.

15. Select **Center** to **Compensate to**. This results in a toolpath in which the center of the tool travels to the containment boundary.

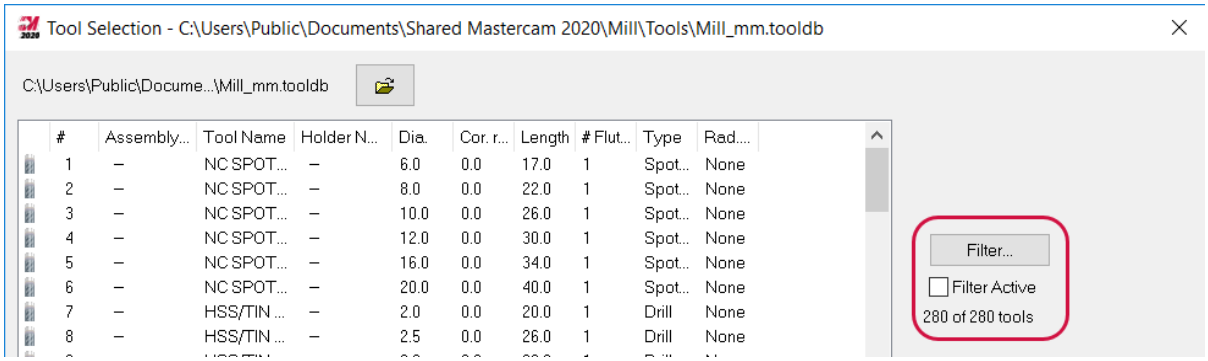


16. Click the **Tool** page in the tree control.



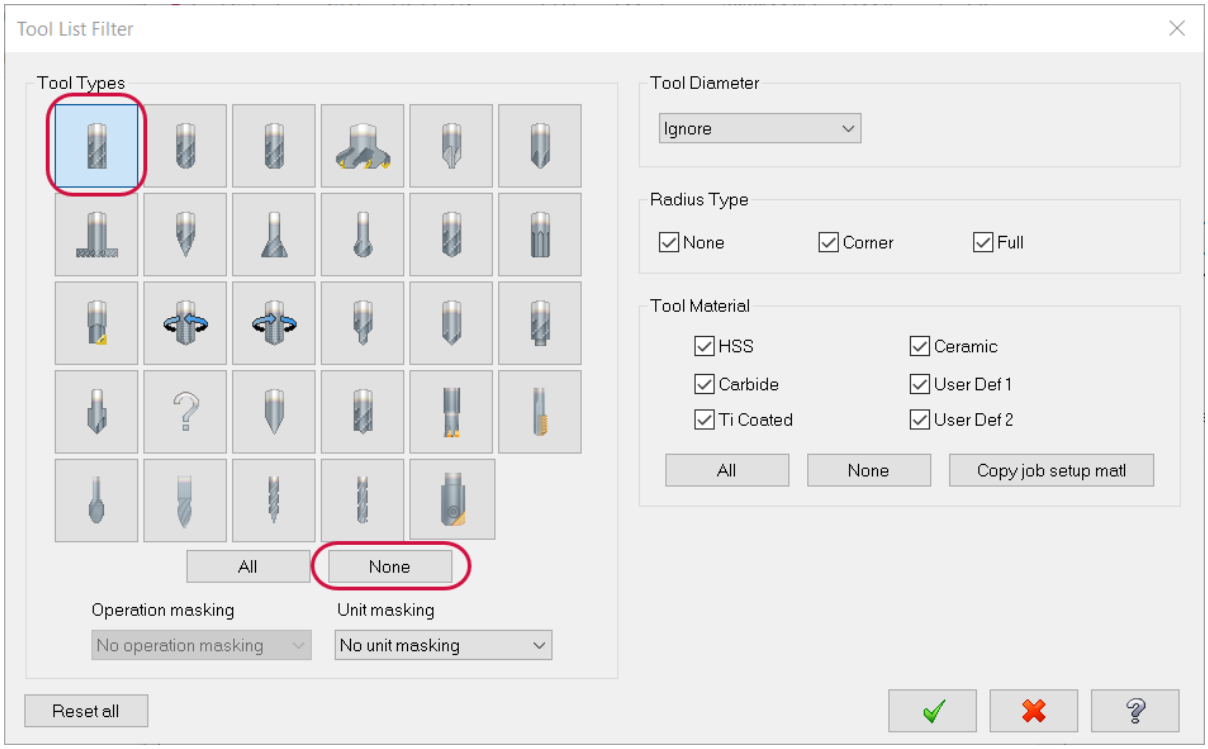
17. Click **Select library** tool to open the **Tool Selection** dialog box.

18. Deselect the **Filter Active** checkbox and then click **Filter**.



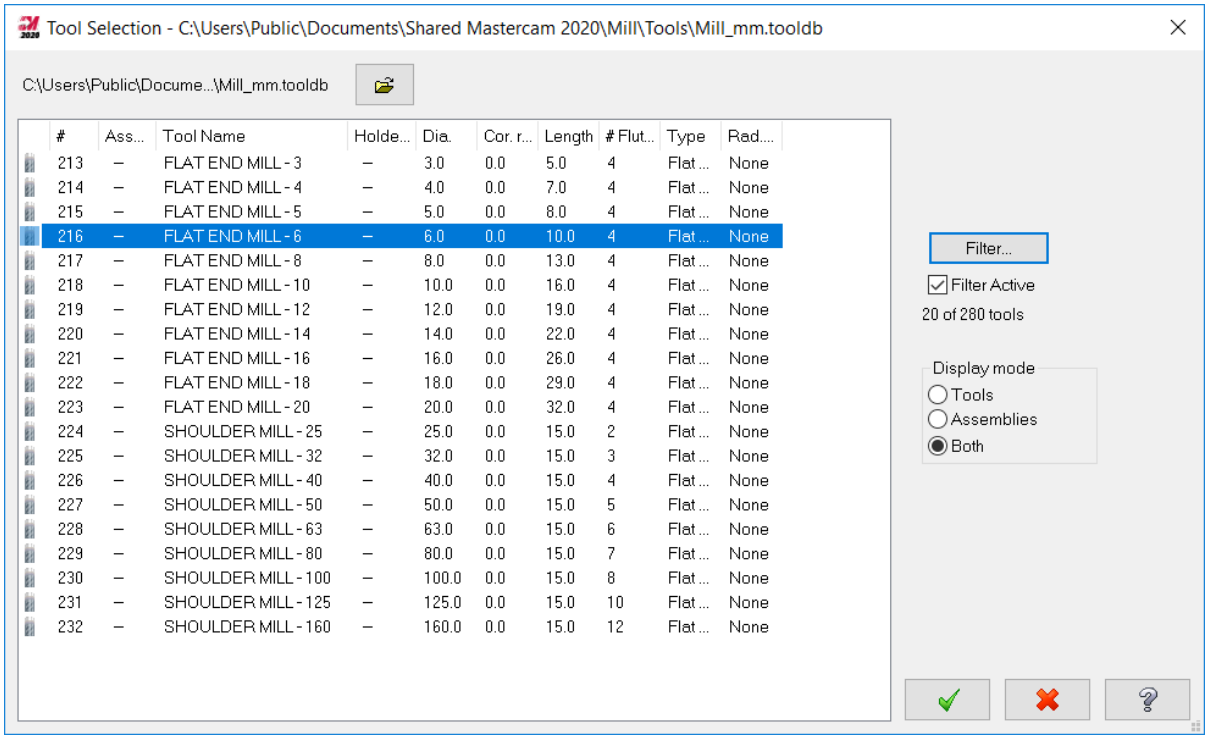
The **Tool List Filter** dialog box displays.

19. Clear the filters by clicking **None**.

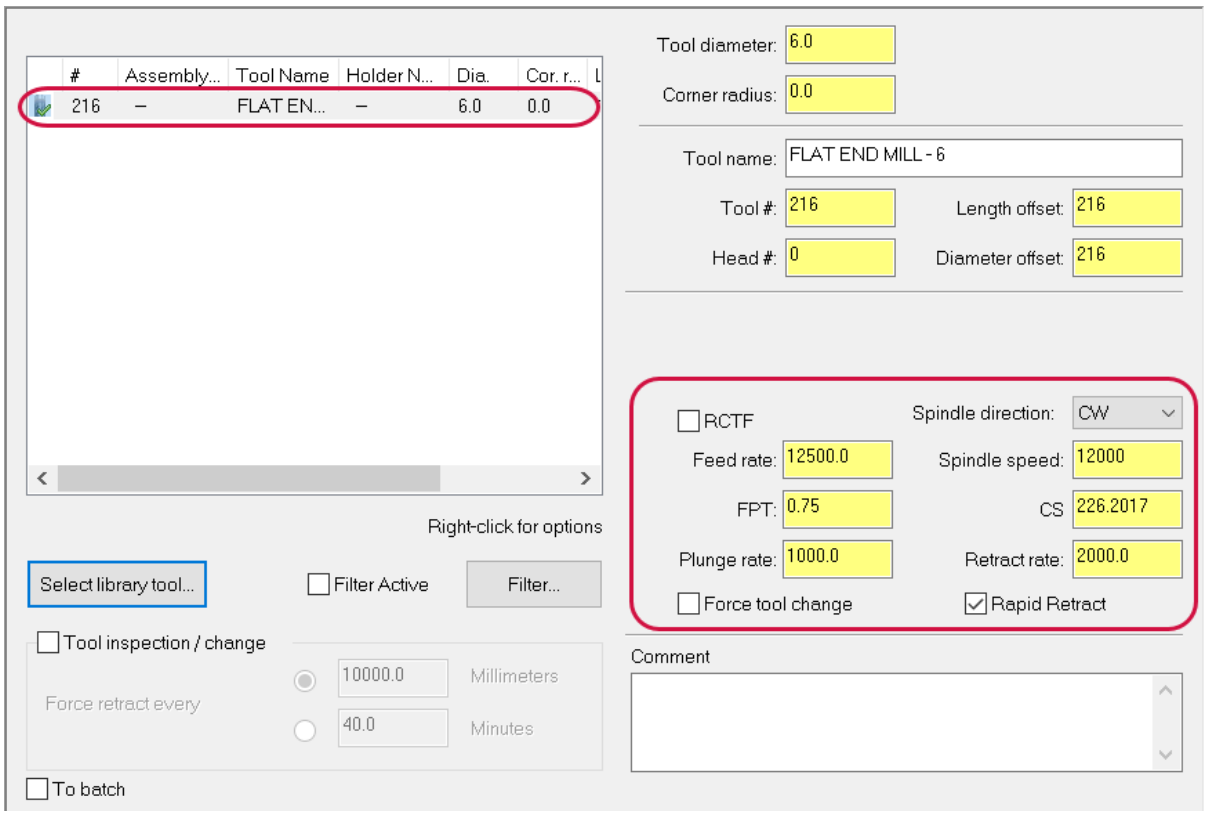


20. Choose the **Endmill1 Flat** filter and then click **OK**.

21. Select the **FLAT END MILL - 6** from the library of tools and click **OK**.



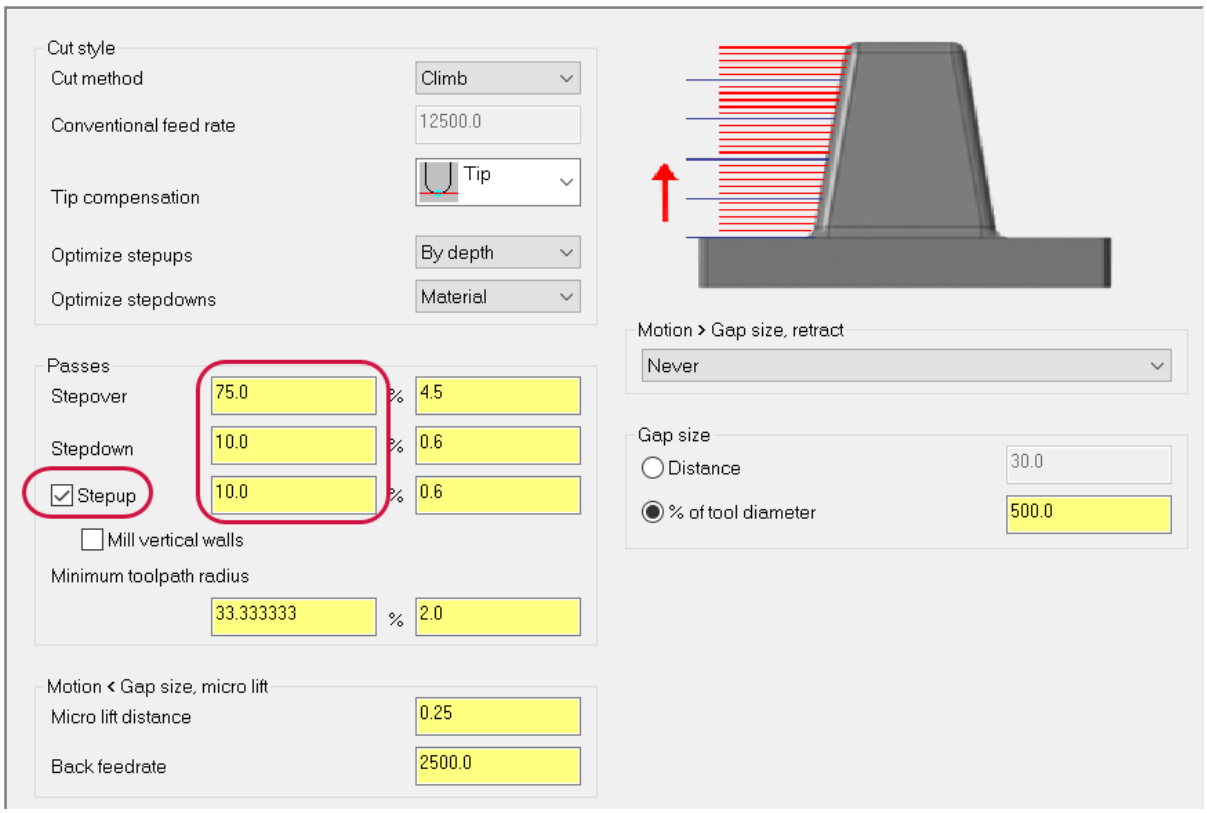
22. Enter the following values for the tool.



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

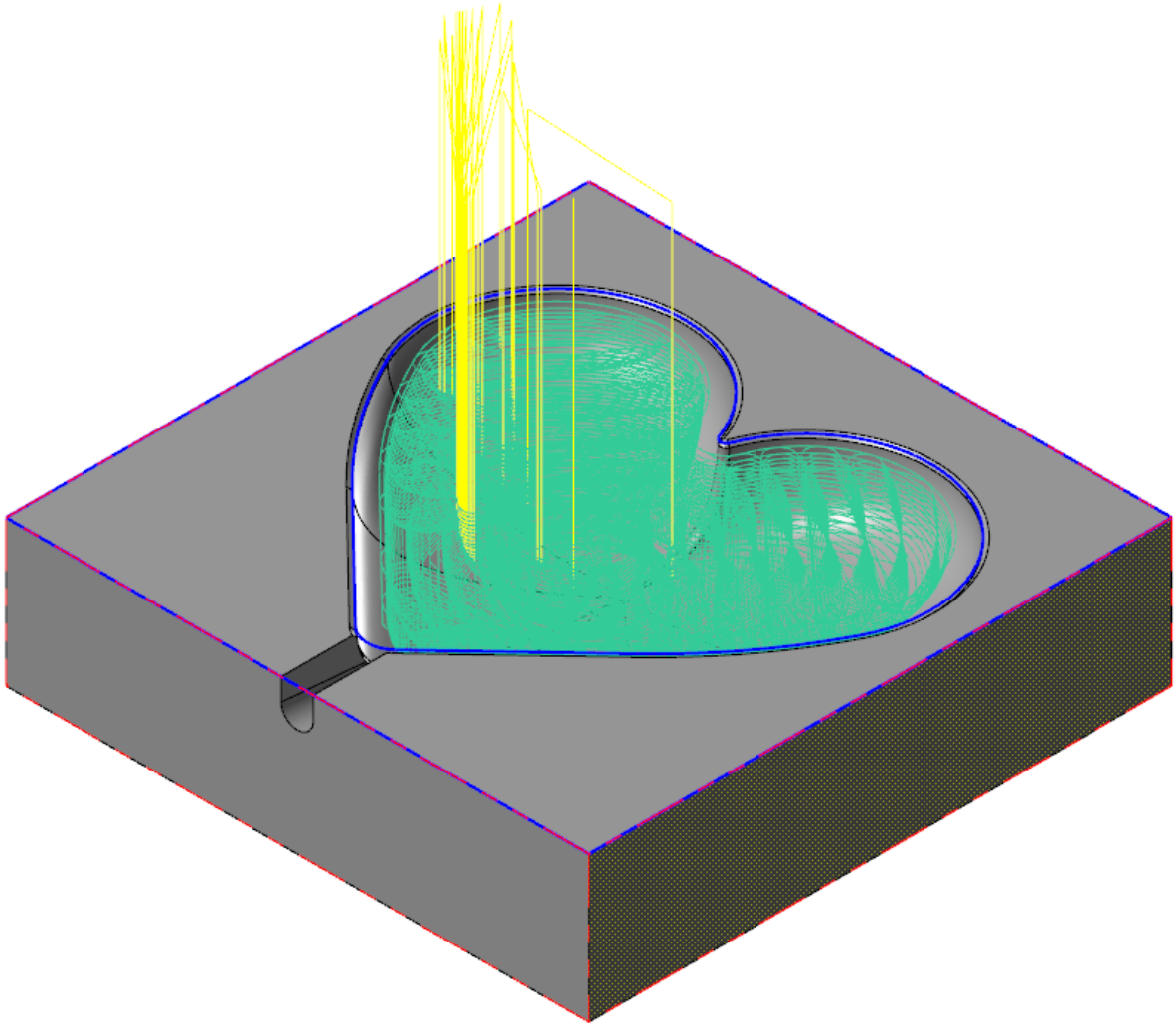
23. Click the **Cut Parameters** page in the tree control.

24. Select the **Stepup** checkbox and enter the following values for the **Passes.**



NOTE
The slope of the surface you are machining determines the Stepup value. The more vertical the surface the lower the Stepup value.

25. Click **OK**. (All other pages retain the default settings.) The toolpath appears on your part.



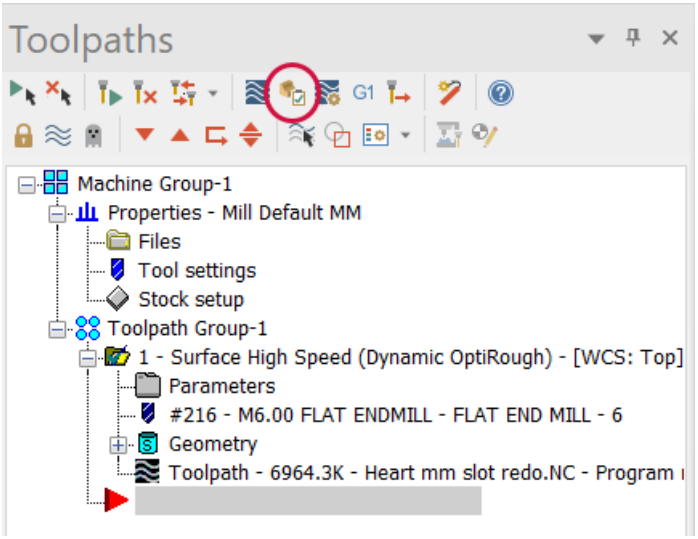
26. Save your file. In the next exercise you will verify the toolpath.

Exercise 2: Verifying the Toolpath

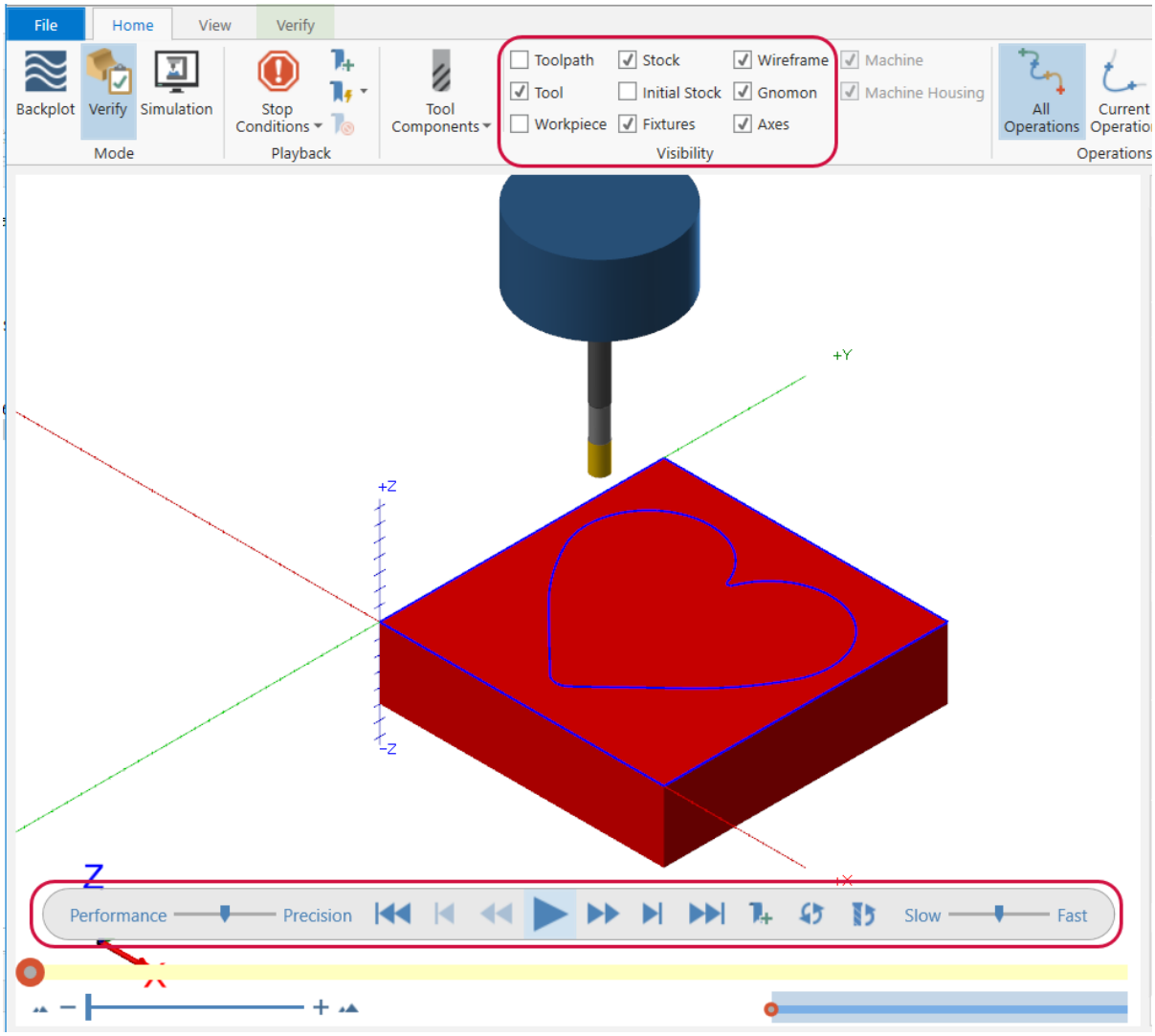
In this exercise, you will use Mastercam Simulator to view and verify the Dynamic OptiRough toolpath.

1. Your file, `Heart mm-XXX.mcam`, which you saved from the previous exercise should be open in Mastercam.

2. In the Toolpaths Manager, click **Verify**.

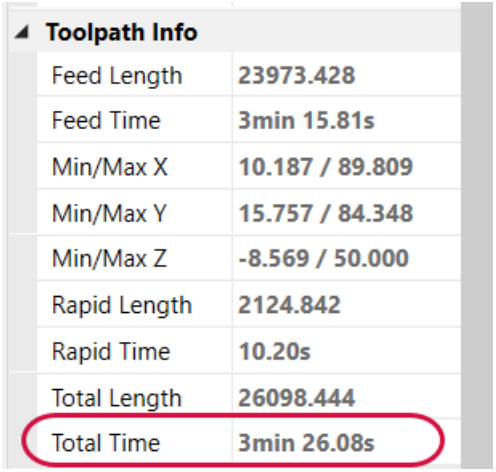


The part opens in Mastercam Simulator.



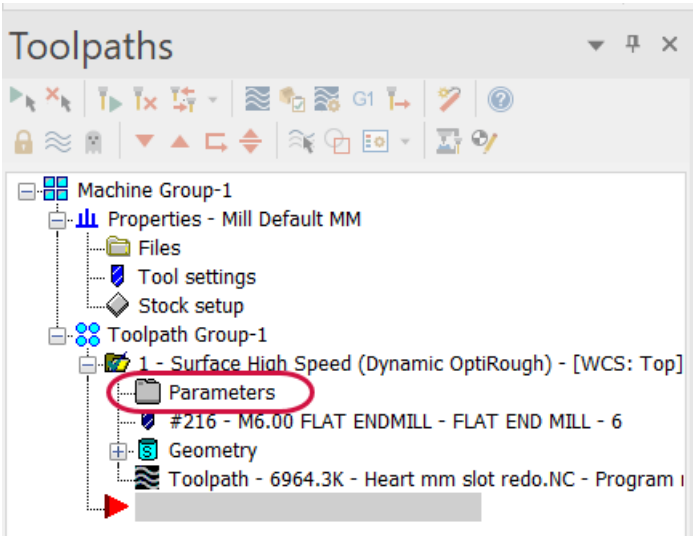
3. Select the **Visibility** checkboxes on the ribbon to experiment with what shows in the graphics window.

4. Use the playback bar at the bottom of Mastercam Simulator to run the simulation. The simulation shows that the toolpath has many very shallow passes and the total time for the OptiRough toolpath is lengthy.

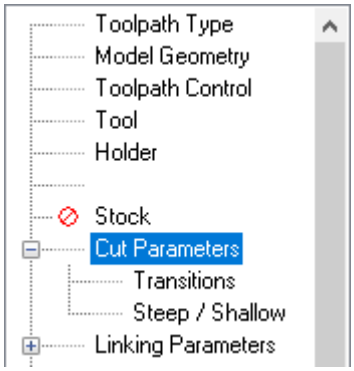


> **NOTE**
> Your **Toolpath Info** time may vary.

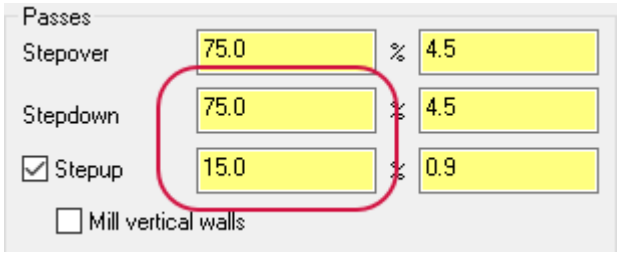
5. Close the **Mastercam Simulator** dialog box and return to Mastercam to make adjustments to the OptiRough toolpath.
6. In Toolpaths Manager, click **Parameters** for the Dynamic OptiRough toolpath.



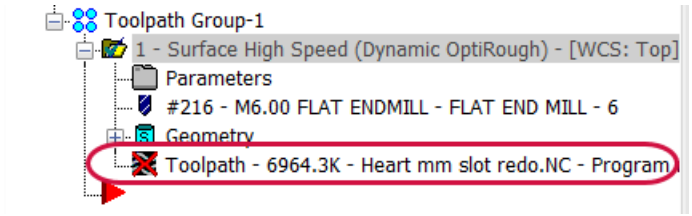
7. Click the **Cut Parameters** page in the tree control.



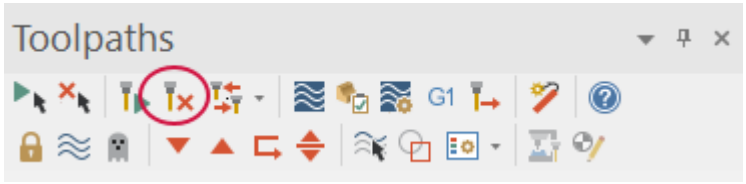
8. Enter a larger **Stepdown** value which determines the spacing between adjacent passes.
9. Enter a larger **Stepup** value as shown in the following image.



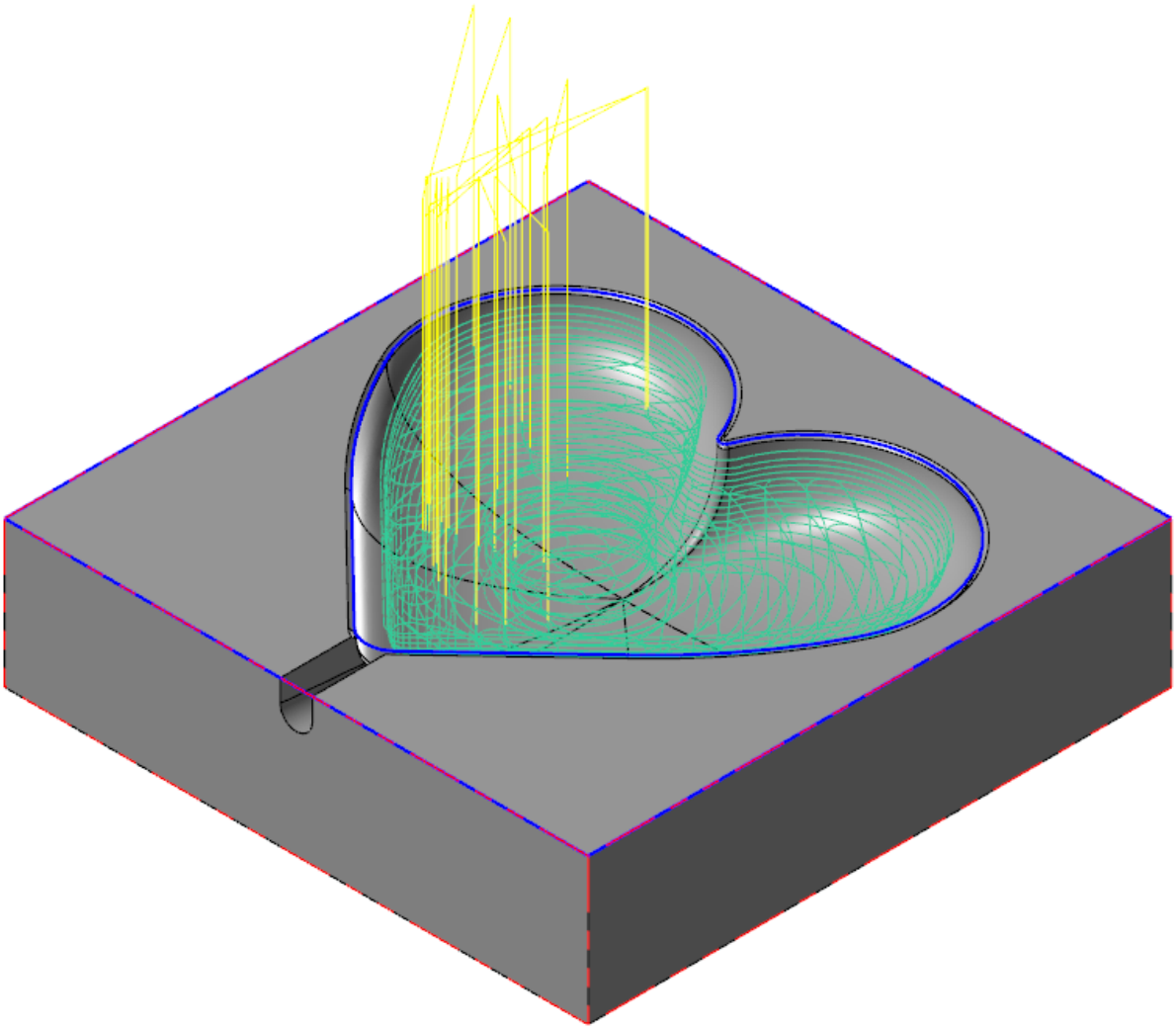
10. Click **OK**.
11. In the Toolpaths Manager you will notice that the Dynamic OptiRough toolpath is marked dirty, indicating that it must be regenerated.



12. Click **Regenerate all dirty toolpaths** on the Toolpaths Manager toolbar.

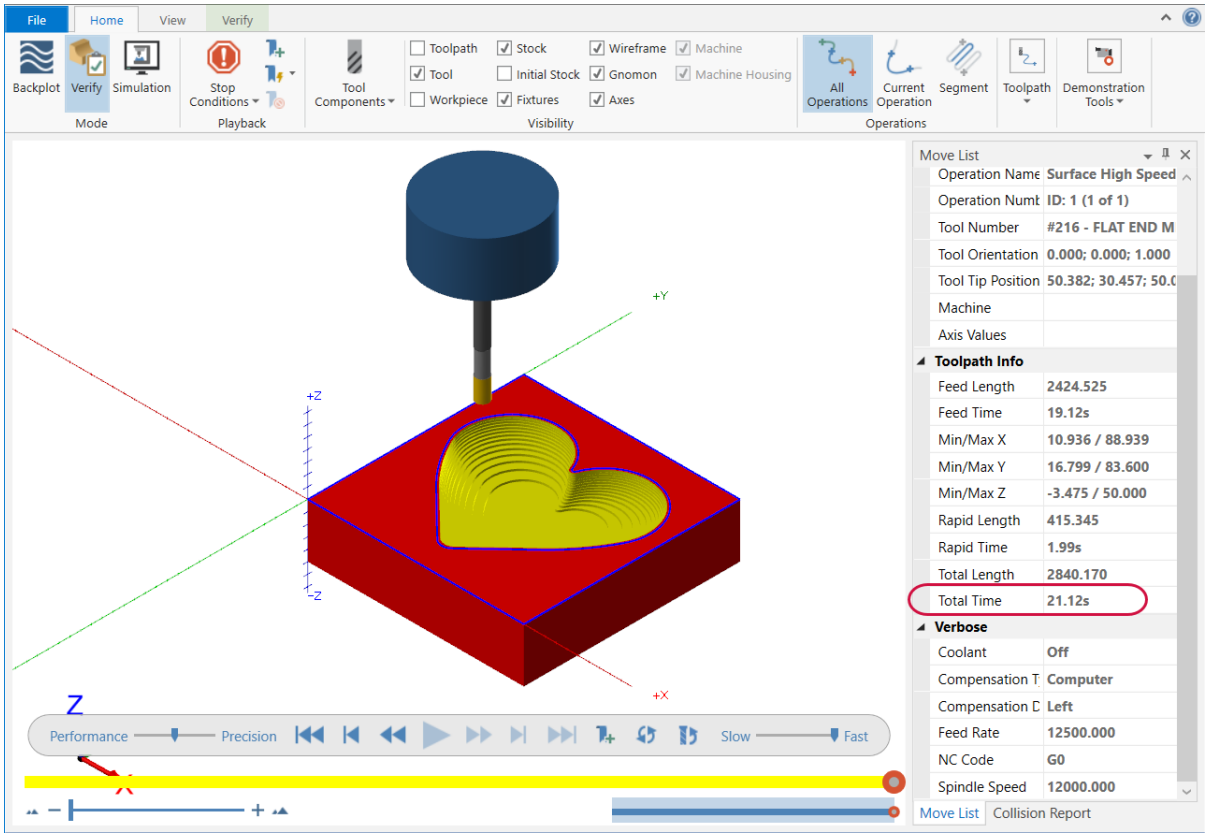


The regenerated toolpath shows in the graphics window. Note that the number of toolpath passes has been significantly reduced.



13. Click **Verify** to return to Mastercam Simulator.

14. Run the simulation again and see that the total time has been reduced but the results are still acceptable.



15. Save your file.

SHOW US WHAT YOU LEARNED!

Can you answer these questions?

1. A finishing toolpath always follows a roughing toolpath.
    a. True
    b. False
2. Verify only simulates toolpath motion.
    a. True
    b. False
3. Once they are created, toolpaths can not be modified.
    - True
    - False
4. When you create a roughing toolpath, you must enter a stock to leave amount.
    a. True
    b. False
5. The cutting tool for a roughing toolpath must be same size or less than the smallest, inside radius of the surface you are cutting.
    a. True
    b. False
6. The primary purpose of a roughing toolpath is to remove large areas of unwanted material.
    a. True
    b. False
7. Roughing toolpaths can reduce the amount of time it takes to completely machine a part.
    a. True
    b. False

CHAPTER 4

FINISH MOLD PROGRAMMING

Finish toolpaths remove a small amount of material and produce the final surface quality of the part. Typically a finish toolpath follows a roughing toolpath, but that is not required.

When a finish toolpath is used in conjunction with a roughing toolpath, the finish toolpath removes the material designated as stock to leave on walls or floors of the part. Programming a part using rough and finish toolpaths usually produces a part in the least amount of time.

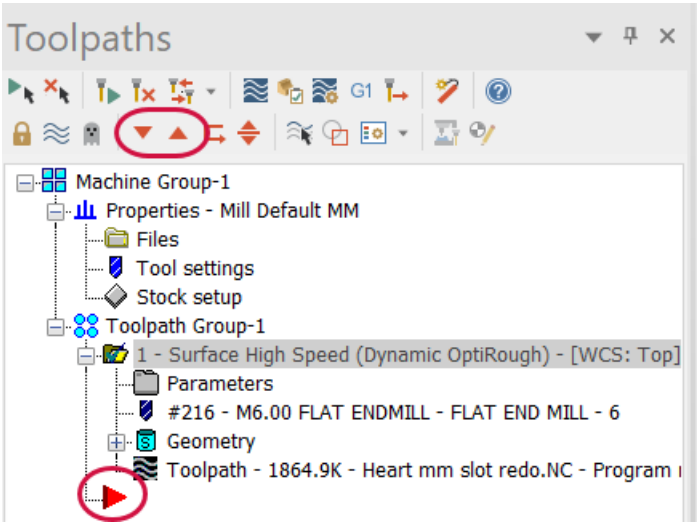
Goals

- Program a finish toolpath
- Understand the concepts of machining and avoidance geometry
- Understand and apply a containment boundary to a toolpath
- Explore tool libraries including filtering by tool type
- Experiment with toolpath verification using Mastercam Simulator

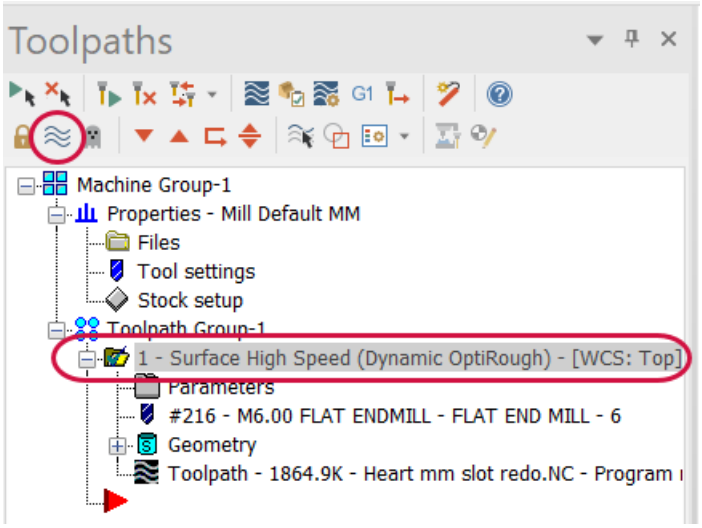
Exercise 1: Creating the Scallop Toolpath

In this exercise, you will program a finish toolpath on top of the OptiRough toolpath to create a smooth finish on the mold surface.

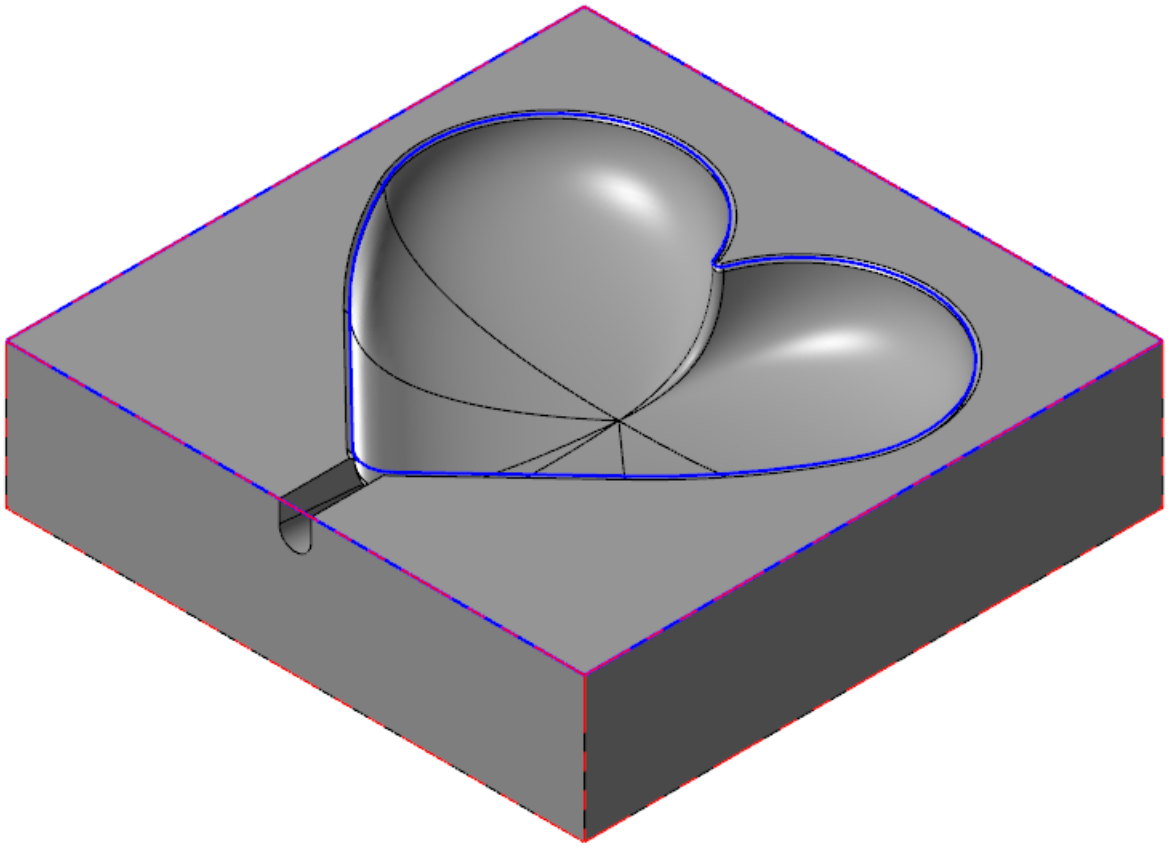
1. Your file, `Heart mm-XXX.mcam`, which you saved from the previous chapter should be open in Mastercam.
2. Right-click and choose **Fit** and set your view to **Isometric (WCS)**.
3. In the Levels Manager, make the heart and the wireframe containment boundary levels visible.
4. In Toolpaths Manager, verify that the red insert arrow is below the OptiRough toolpath. Use the **Move insert arrow** controls to move the arrow up and down in the Toolpaths Manager.



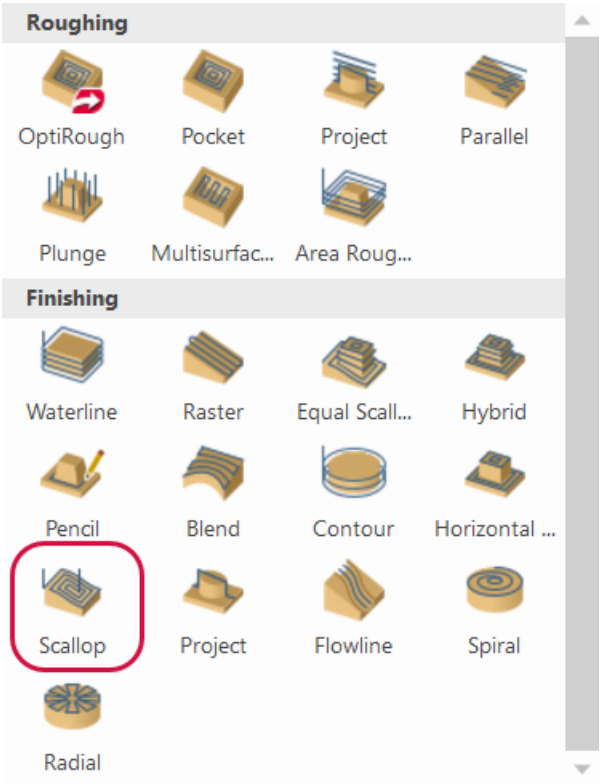
5. Select the Dynamic OptiRough toolpath. A green check on the folder indicates that it is selected.



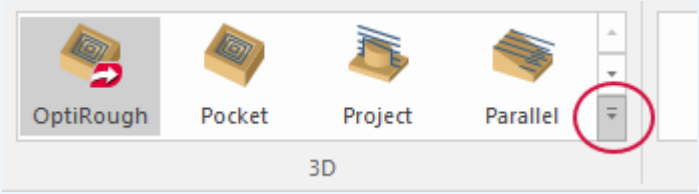
6. Click **Toggle display on selected operations** as shown in the previous image. The toolpath is hidden in the graphics window.



7. On the **Mill Toolpaths** tab, click **Scallop** in the 3D group.

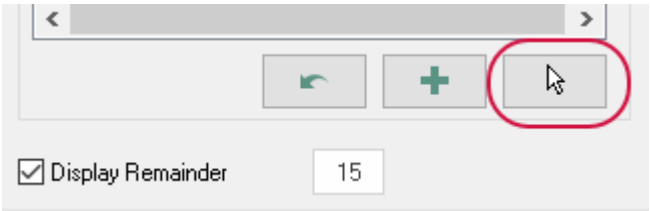


NOTE
Click the expander button in the 3D group to view the entire toolpath gallery.



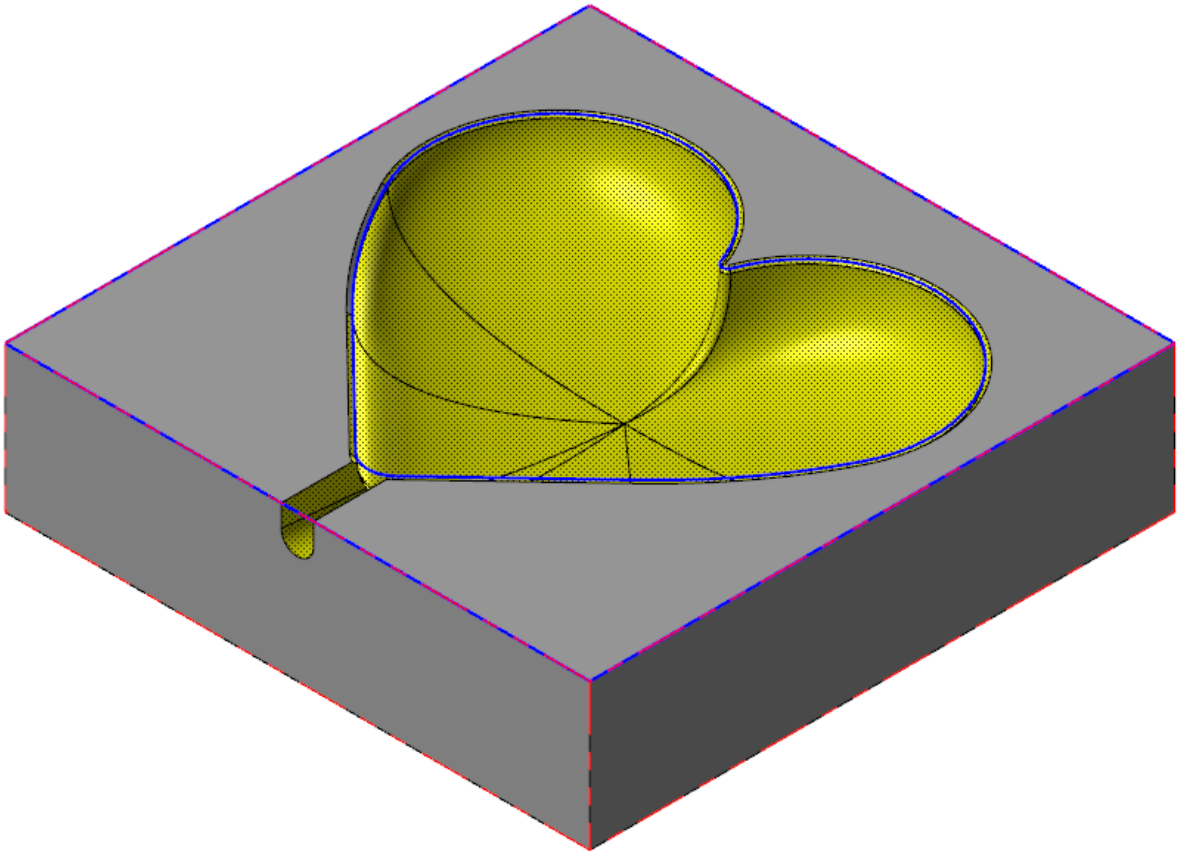
The **Model Geometry** page of the **Scallop** dialog box displays.

8. Click **Select entities** in the **Machining Geometry** section.

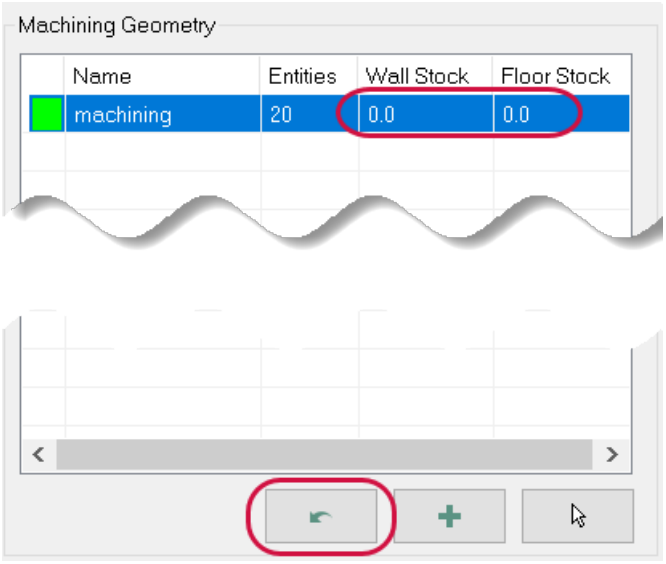


9. Use window select to select the heart and slot geometry.

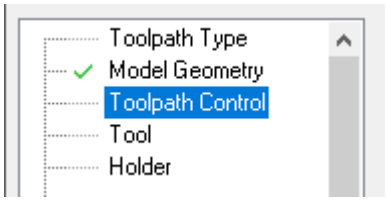
10. Rotate the part and make sure that all parts of the mold to be machine are selected.



11. Click **End Selection** to return to the **Model Geometry** page.
12. The finish toolpath will remove the stock to leave amounts that were entered for the OptiRough toolpath. Click **Reset stock values** to return **Wall Stock** and **Floor Stock** to **0.0**.

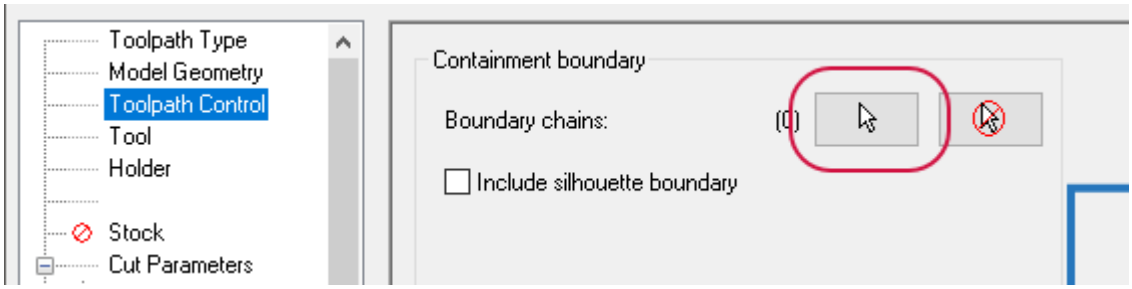


13. Click the **Toolpath Control** page in the tree control.



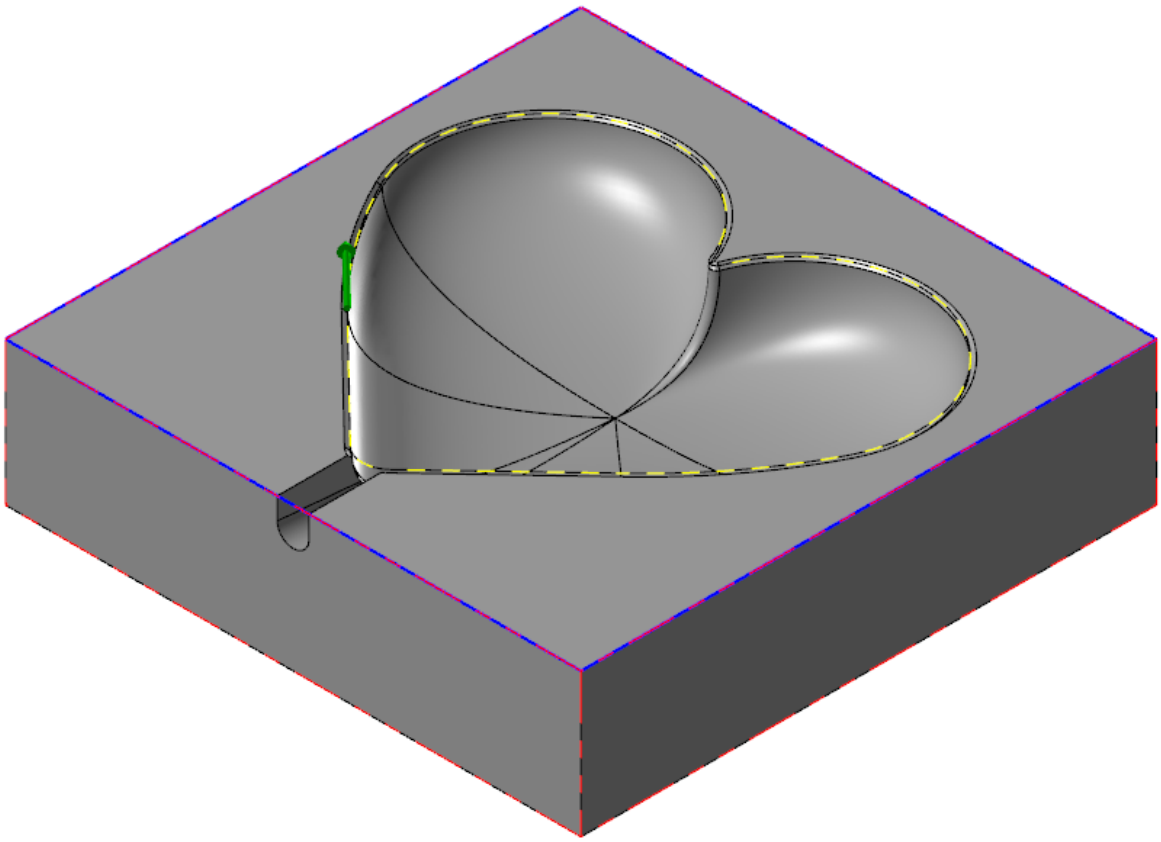
In this page you will choose the containment boundary to limit the area that will be machined.

14. Click the **Boundary chains** select arrow.



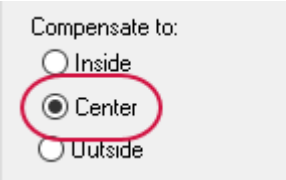
The **Wireframe Chaining** dialog box displays.

15. Select the containment boundary wireframe.

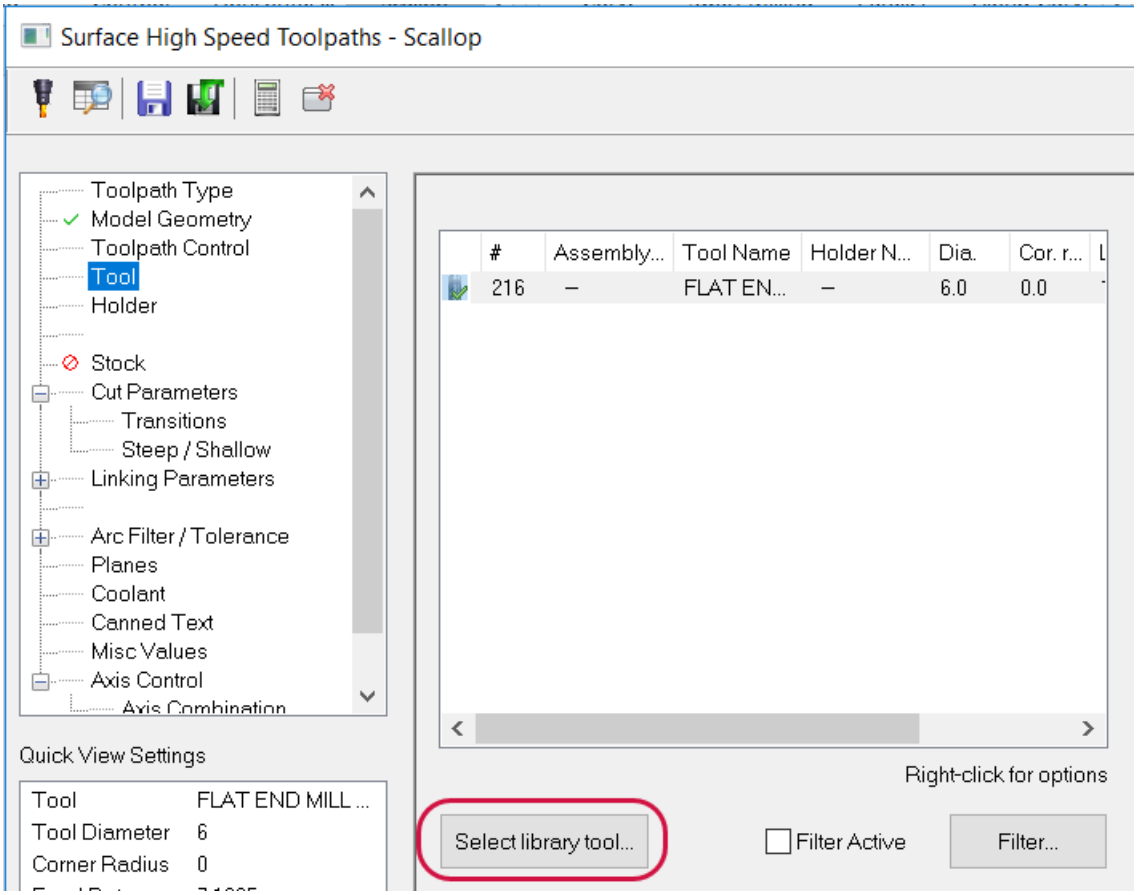


16. Click **OK** to close the **Wireframe Chaining** dialog box and return to the **Toolpath Control** page.

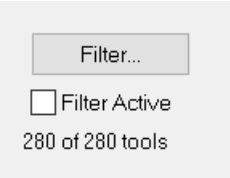
17. Select **Center** to **Compensate to**.



18. Click the **Tool** page in the tree control.
19. Click **Select library tool.**

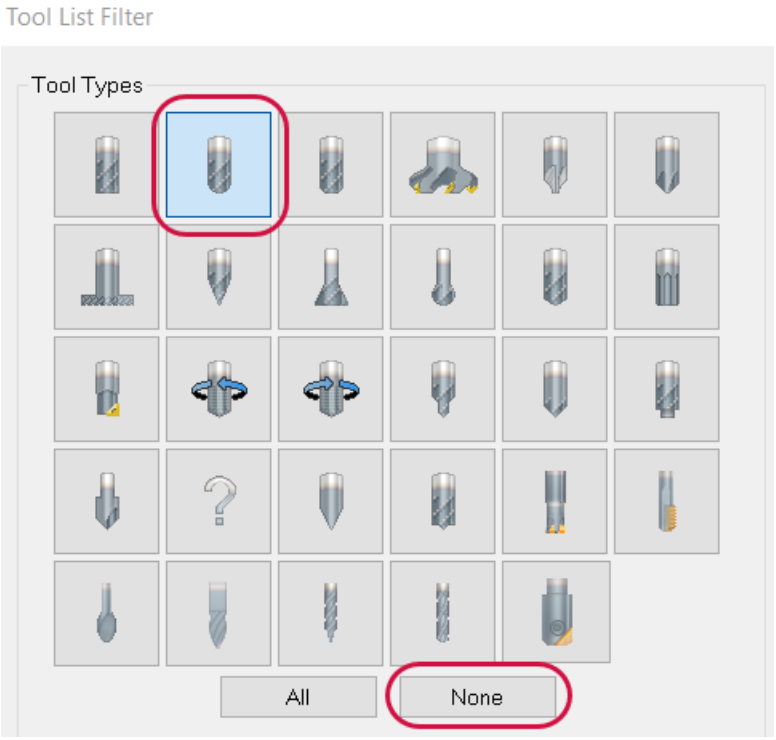


20. Deselect the **Filter Active** checkbox and then click **Filter**.

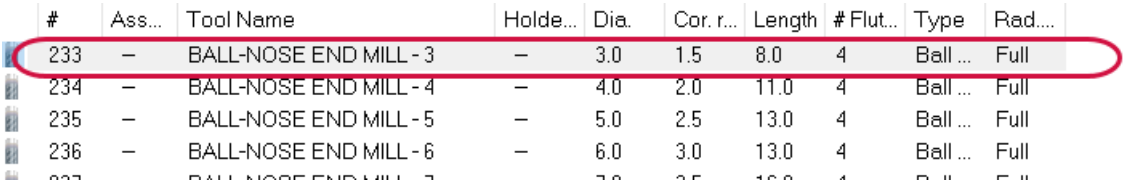


The **Tool List Filter** dialog box displays.

21. Clear the filters by clicking **None**.



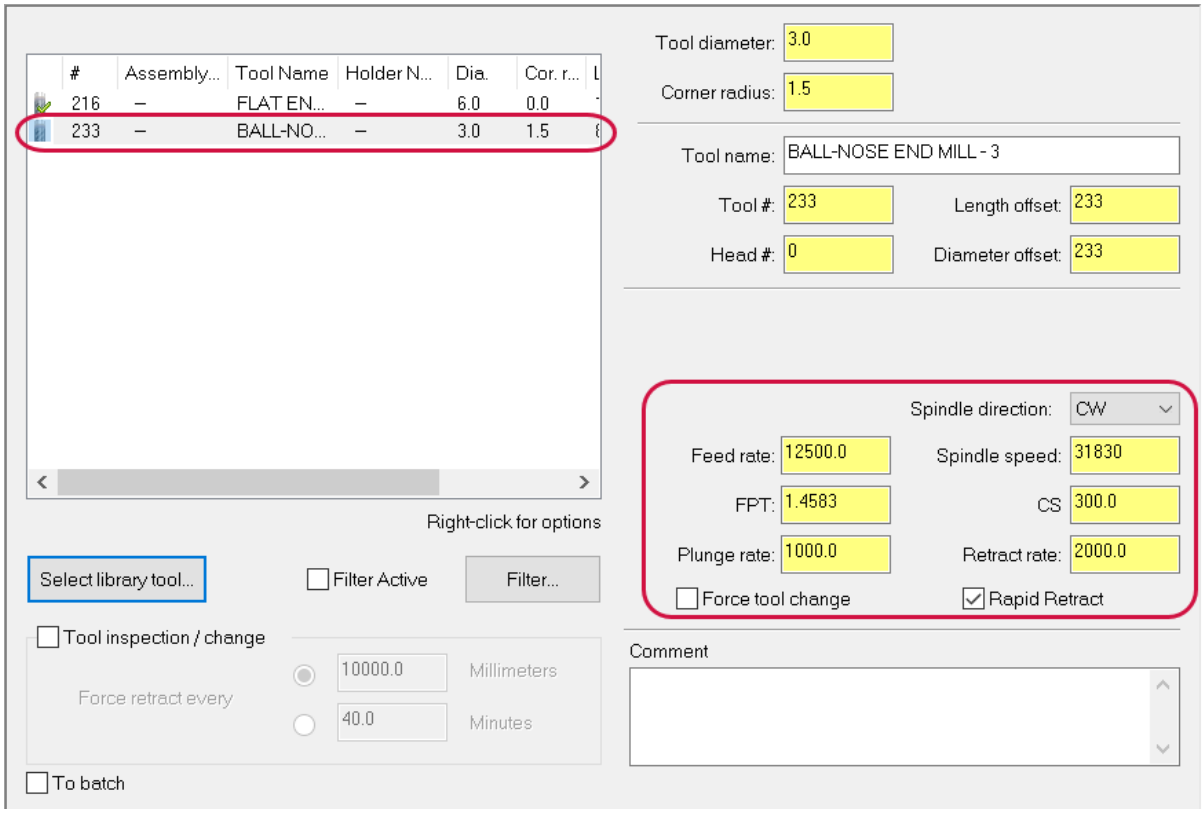
22. Choose the **Endmill2 Sphere** filter and then click **OK**.
23. Select the **BALL-NOSE END MILL - 3** from the library of tools and click **OK**.



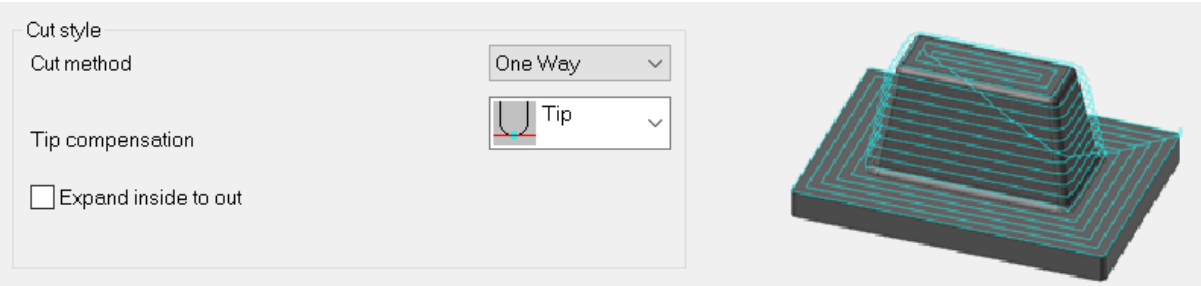
#	Ass...	Tool Name	Holde...	Dia.	Cor. r...	Length	# Flut...	Type	Rad....
233	–	BALL-NOSE END MILL - 3	–	3.0	1.5	8.0	4	Ball ...	Full
234	–	BALL-NOSE END MILL - 4	–	4.0	2.0	11.0	4	Ball ...	Full
235	–	BALL-NOSE END MILL - 5	–	5.0	2.5	13.0	4	Ball ...	Full
236	–	BALL-NOSE END MILL - 6	–	6.0	3.0	13.0	4	Ball ...	Full

24. Enter the following values for the tool.

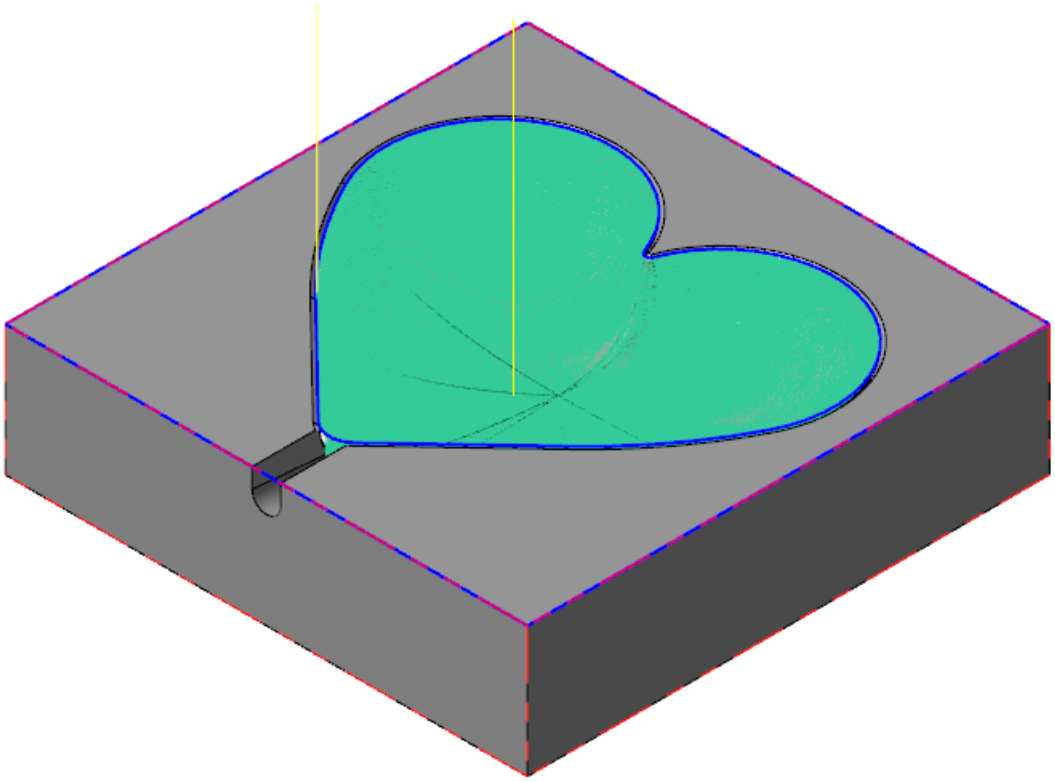
> **CAUTION**
> These values may vary depending on your machine and material.



25. Click the **Cut Parameters** page in the tree control.
26. Deselect **Expand inside to out** to have the toolpath cut from the perimeter of the heart toward the center.

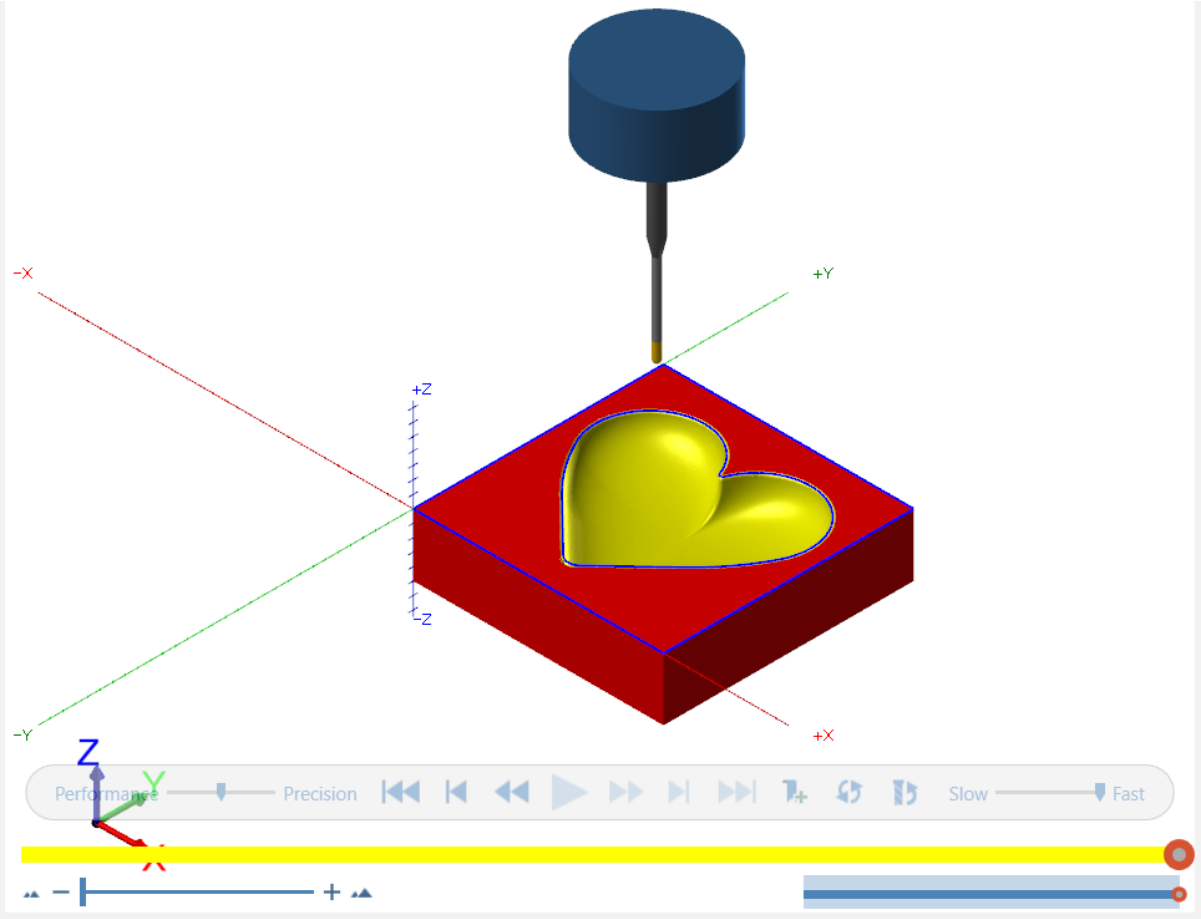


27. Click **OK**. (All other pages retain the default settings.) The toolpath appears on your part.



28. In the Toolpaths Manager, click **Verify** and open Mastercam Simulator.

29. Use the playback bar to run the simulation.



30. Save your file.

SHOW US WHAT YOU LEARNED!

Can you answer these questions?

1. When finishing, you must use a cutting tool the same radius or smaller than the smallest, inside radius of the surface you are cutting to clean-up any unmachined areas.
    a. True
    b. False
2. You can use the same tool to both rough and finish cut.
    a. True
    b. False
3. Your stepover should be less than 10% of your cutter's diameter to get the best surface finish, .
    a. True
    b. False
4. You can always achieve a perfectly smooth finish with one finish toolpath.
    a. True
    b. False
5. Typically, the cutter you would use to get the smoothest surface on a finish toolpath would be:
    a. Flat endmill
    b. Drill Bit
    c. Ball endmill
    d. Spherical endmill
    e. Both C and D
6. The Finish Scallop toolpath does not have a stock to leave option .
    a. True
    b. False
7. When you set the cut parameters of a Finish Scallop toolpath, you can control whether the cutting motion starts from the center and moves in or the perimeter and moves out.
    a. True
    b. False
8. List some benefits of using Verify to review your toolpaths?

CHAPTER 5

SLOT PROGRAMMING

The slot is a long narrow shape that will be programmed using a rough and finish Scallop toolpath. Using the Toolpaths Manager **Copy** function, you will quickly create the finish toolpath based on the rough toolpath with minimal edits required.

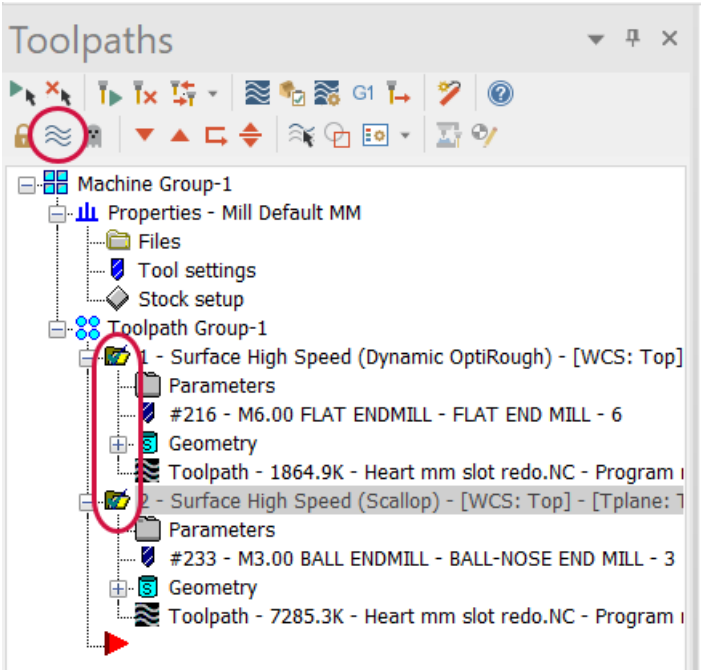
Goals

- Program a rough Scallop toolpath
- Use Copy to quickly create a finish Scallop toolpath
- Create a containment boundary using Wireframe tools
- Explore tool libraries including filtering by tool type
- Experiment with toolpath verification using Mastercam Simulator

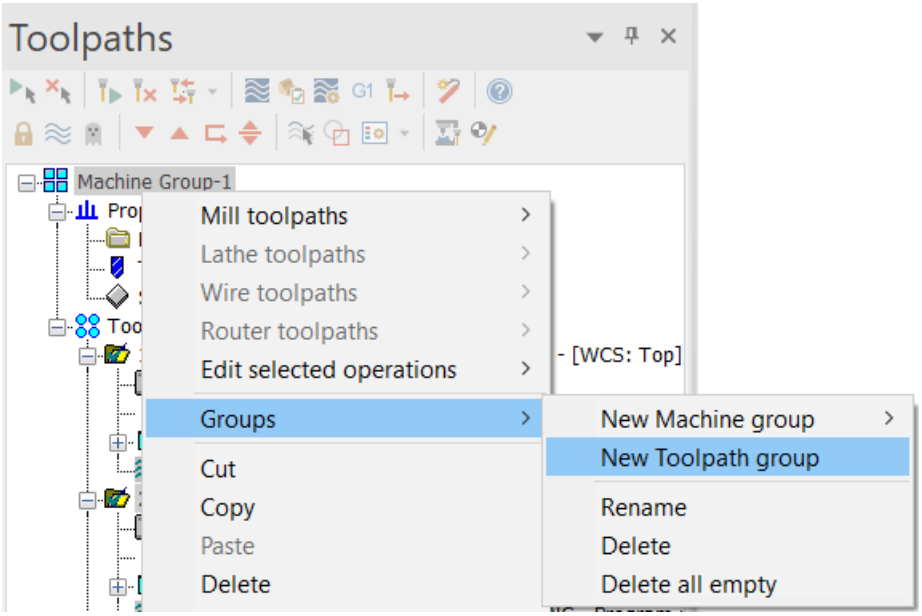
Exercise 1: Creating the Rough Scallop Toolpath

In this exercise, you will create a containment boundary for the slot area of the mold and then program a rough Scallop toolpath.

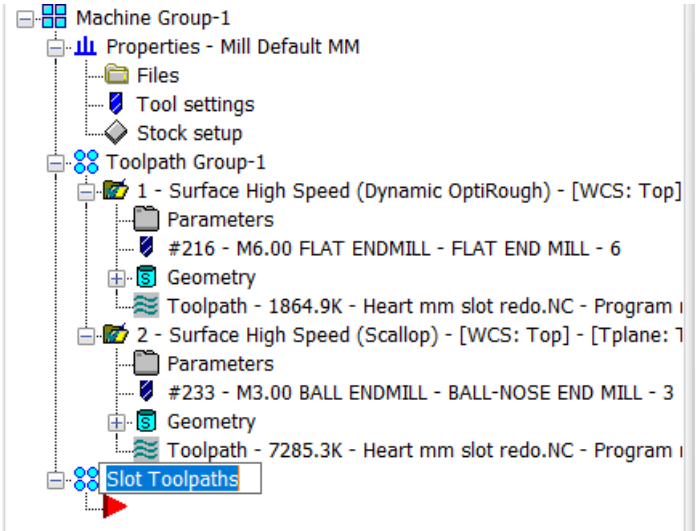
1. Your file, `Heart mm-XXX.mcam`, which you saved from the previous chapter should be open in Mastercam.
2. Right-click and choose **Fit** and set your view to **Isometric (WCS)**.
3. In the Levels Manager, make the heart and the containment boundary levels visible.
4. Select both toolpaths and then click **Toggle the display of selected operations** or press [**Alt+T**] to hide the toolpaths in the graphics window.



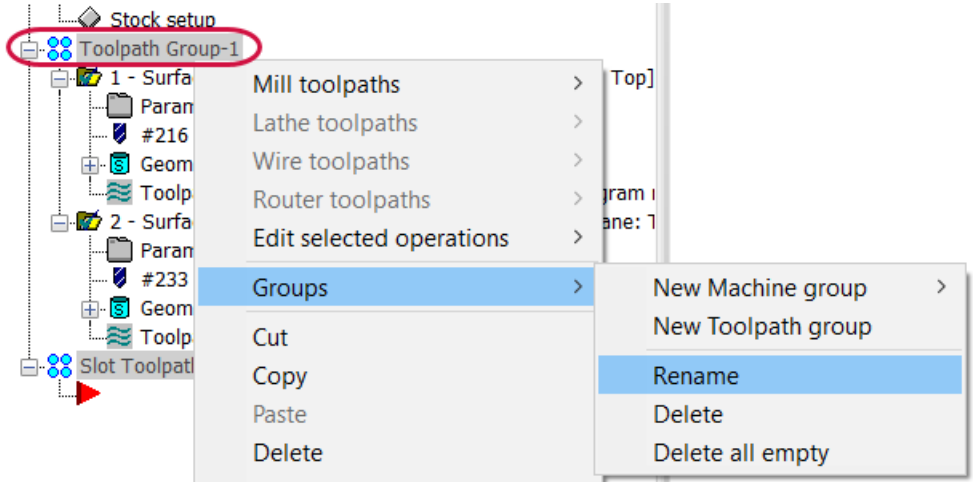
5. Adding a new toolpath group will help you organize the file. Right-click **Machine Group-1** and select **Group**, **New Toolpath group**.



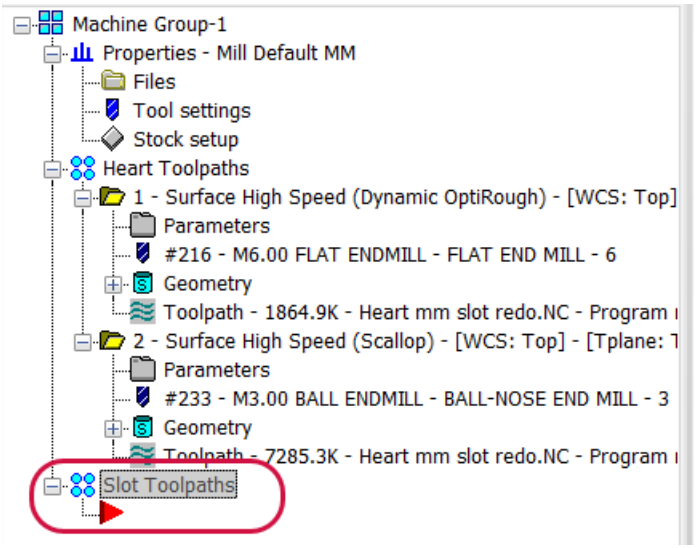
6. In Toolpaths Manager enter **Slot Toolpaths** for the name of the new group. Press [**Enter**] to accept.



7. Right-click **Toolpath Group-1** and select **Groups**, **Rename**.

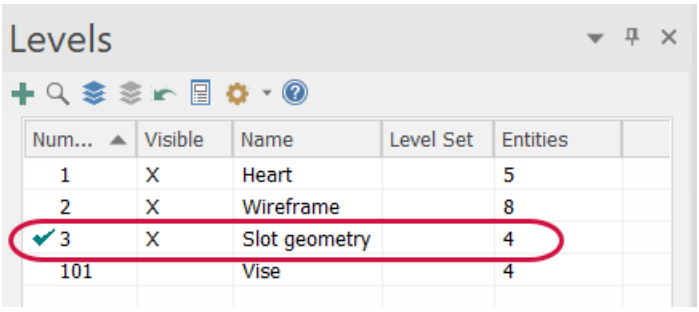


8. Enter **Heart Toolpaths** for the new name.
9. Ensure that the red insert arrow is below the **Slot Toolpaths** group.

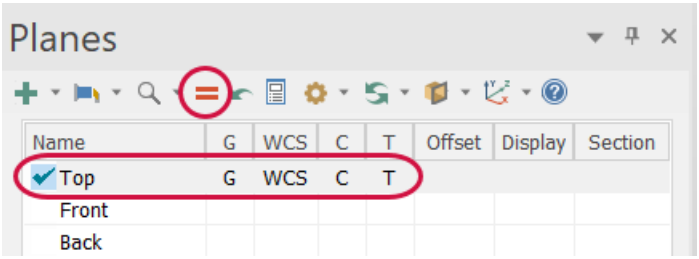


In the next steps you are going to create a containment boundary to keep the machining inside the slot.

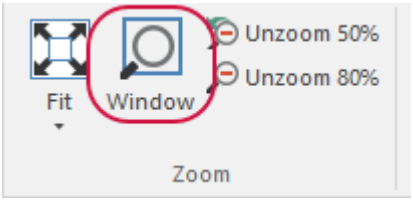
10. On the Levels Manager, make level **3 Slot geometry** the active level and turn on the visibility.



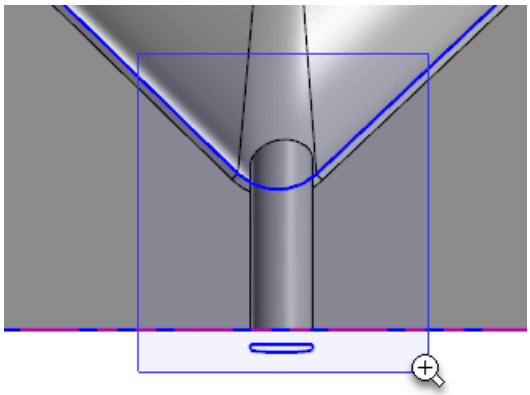
11. In Planes Manager, make the Top plane active and then click **Set your current WCS, construction plane, and tool plane with their origins to the selected plane**.



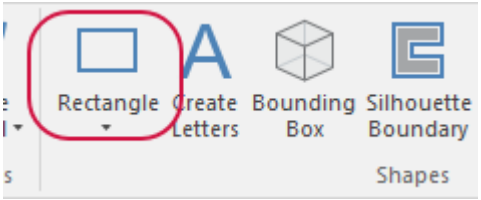
12. Right-click in the graphics window and set your view to **Top (WCS)**.
13. On the **View** tab, click **Window**.



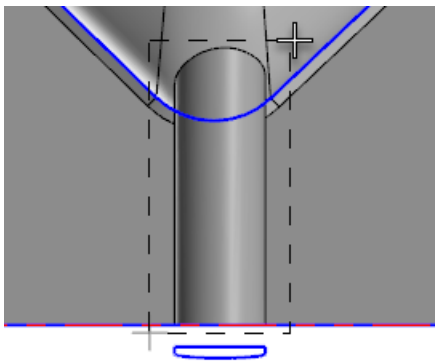
14. Window select the slot area to magnify that region.



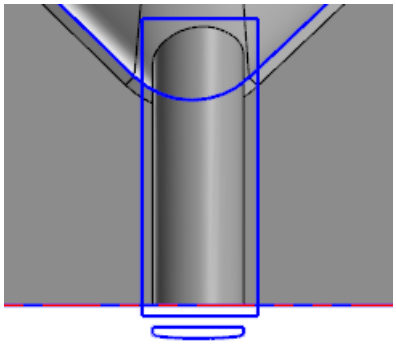
15. On **Wireframe** tab, click **Rectangle**.



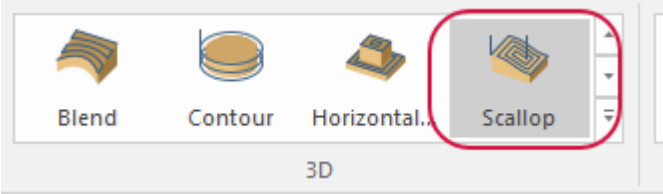
16. Draw a rectangle around the slot geometry as shown in the following image.



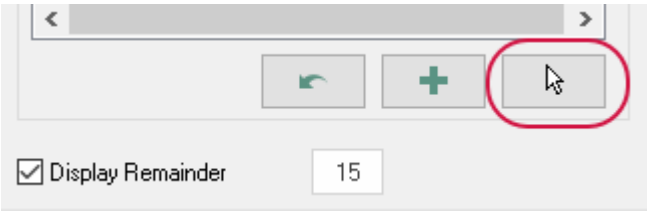
The rectangle can be placed any where on the Z plane. It will be used as a containment boundary when machining from the top.



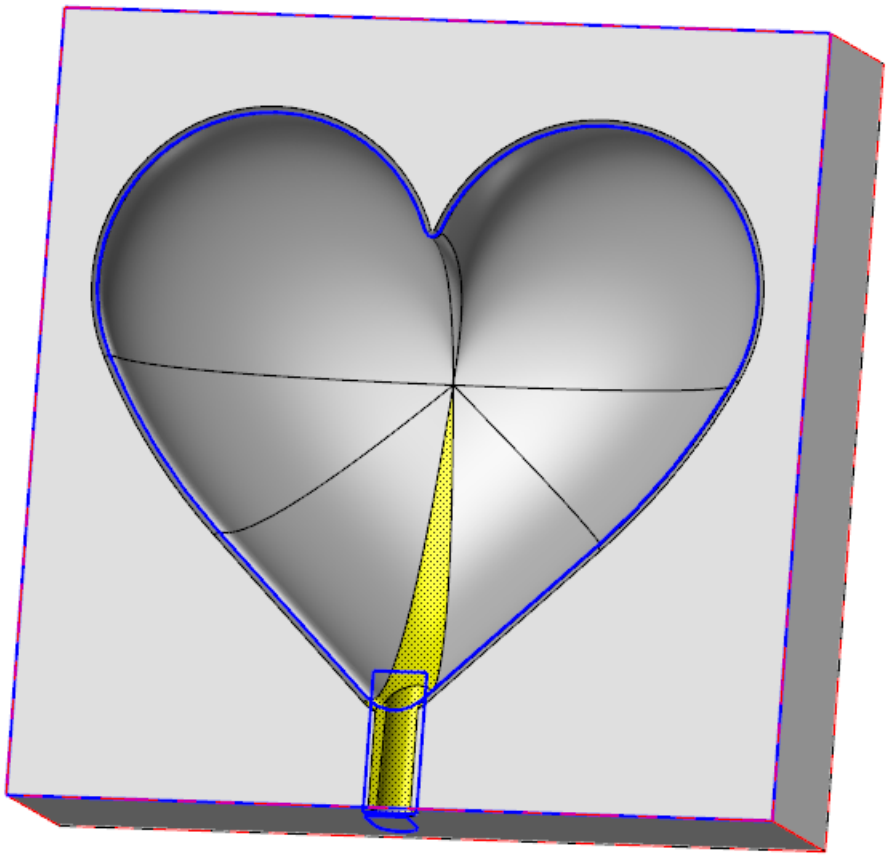
17. On the **Mill Toolpaths** tab, click **Scallop** in the 3D group.



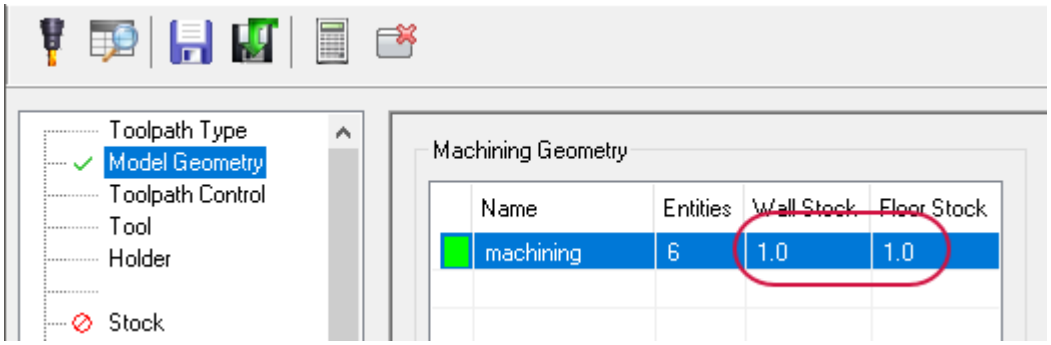
18. On the **Model Geometry** page, click **Select entities** in the **Machining Geometry** section.



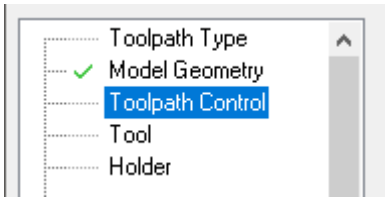
19. Select the slot surfaces and any surfaces tangent to the slot.



20. Click **End Selection** to return to the **Model Geometry** page.
21. For a rough toolpath you will designate a certain amount of stock to leave on the part. This stock will be removed with the finish toolpath. Double click the **Wall Stock** and **Floor Stock** fields of Machining Geometry to activate them and enter **1.0**.

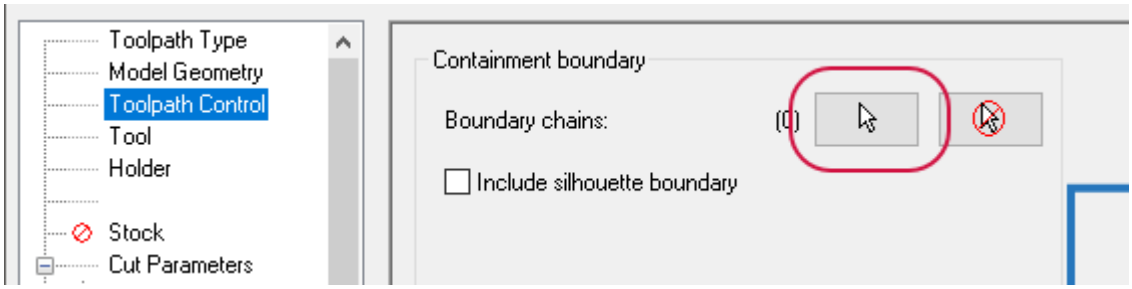


22. Click the **Toolpath Control** page in the tree control.



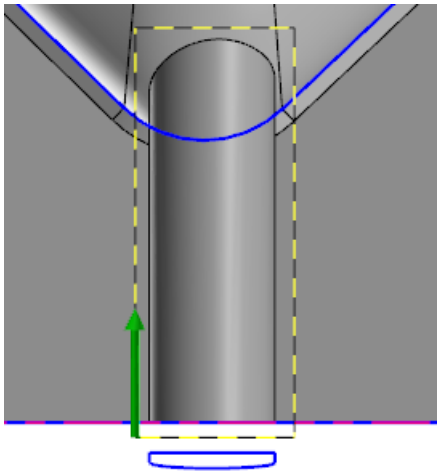
In this page you will choose the containment boundary to limit the area that will be machined.

23. Click the **Boundary chains** select arrow.



The **Wireframe Chaining** dialog box displays.

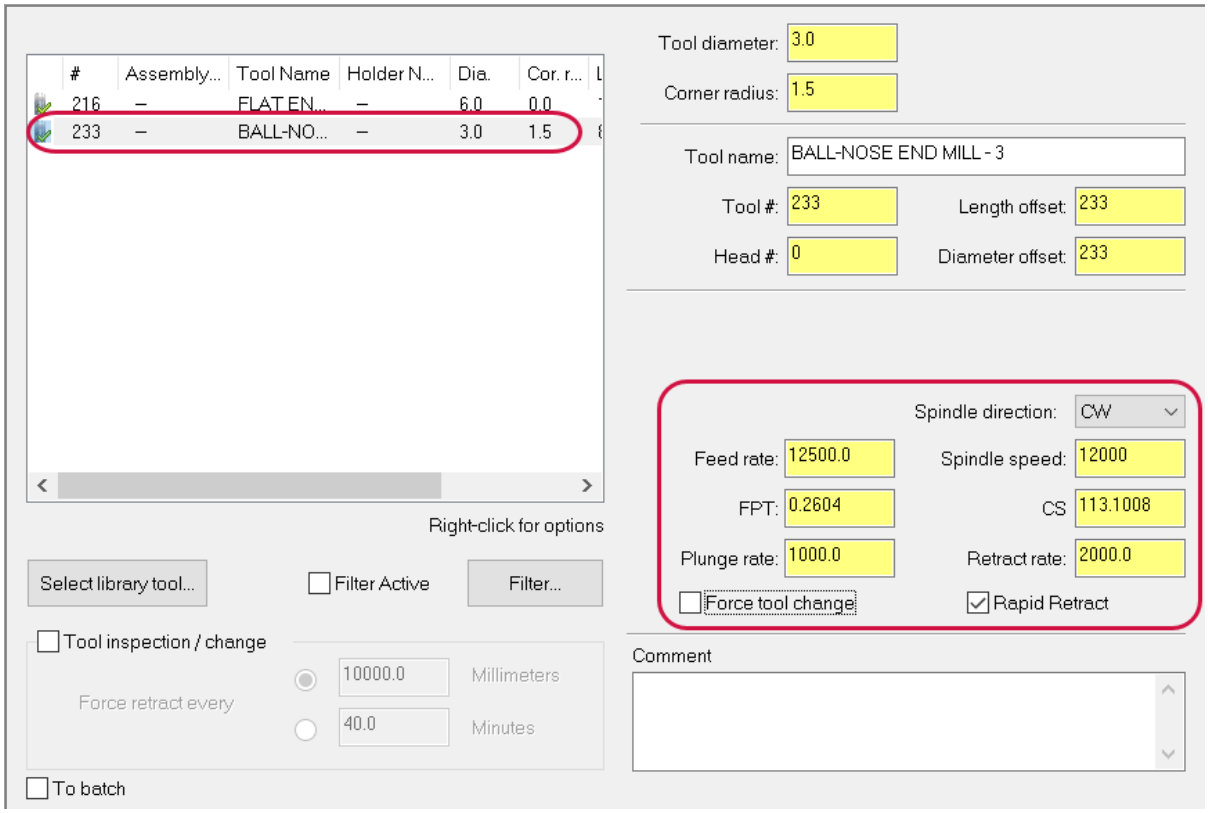
Hold down the [**Shift**] key and select a segment of the containment boundary wireframe. [**Shift+click**] selects all tangent segments.



24. Click **OK** to close the **Wireframe Chaining** dialog box and return to the **Toolpath Control** page.

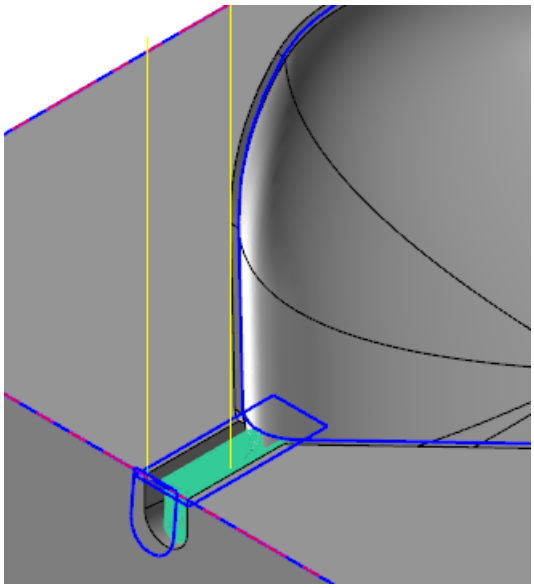
25. Select **Inside** to **Compensate to**. This forces the tool to machine inside the containment boundary.

26. Click the **Tool** page in the tree control.
27. Select the **BALL-NOSE ENDMILL - 3** as the tool and enter the values shown in the following image.

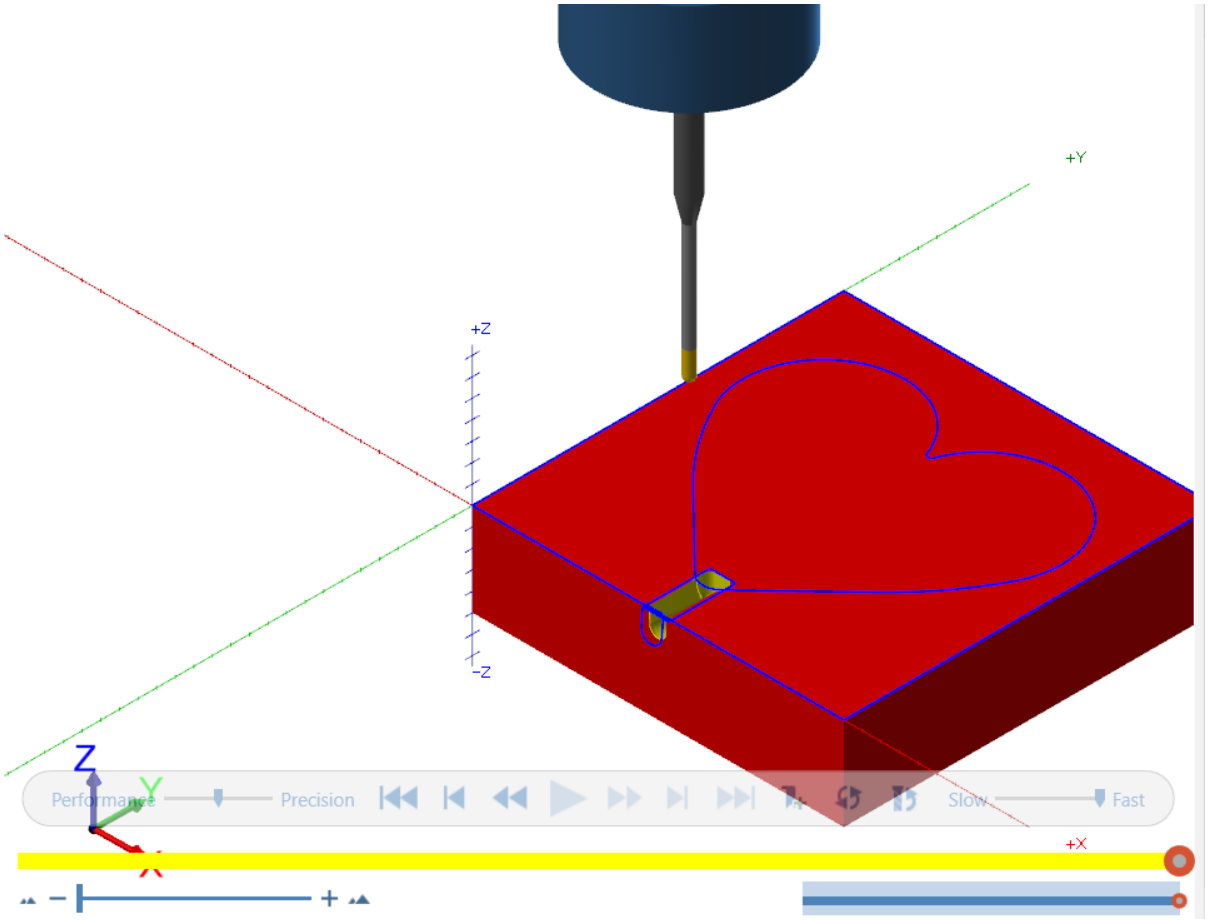


CAUTION
These values may vary depending on your machine and material.

28. Click **OK**. (All other pages retain the default settings.) The toolpath appears on your part.



29. In the Toolpaths Manager, select **Toolpath 3** and click **Verify**. Only the selected toolpaths will be verified in Mastercam Simulator.



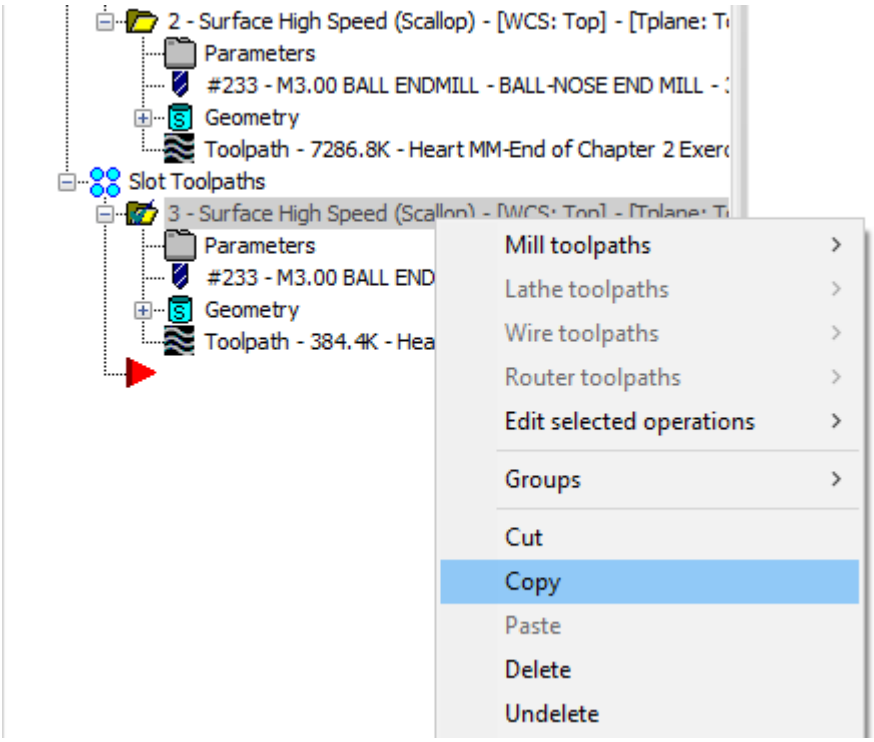
30. Use the playback tools to view the simulation and then close Mastercam Simulator.

31. Save the file.

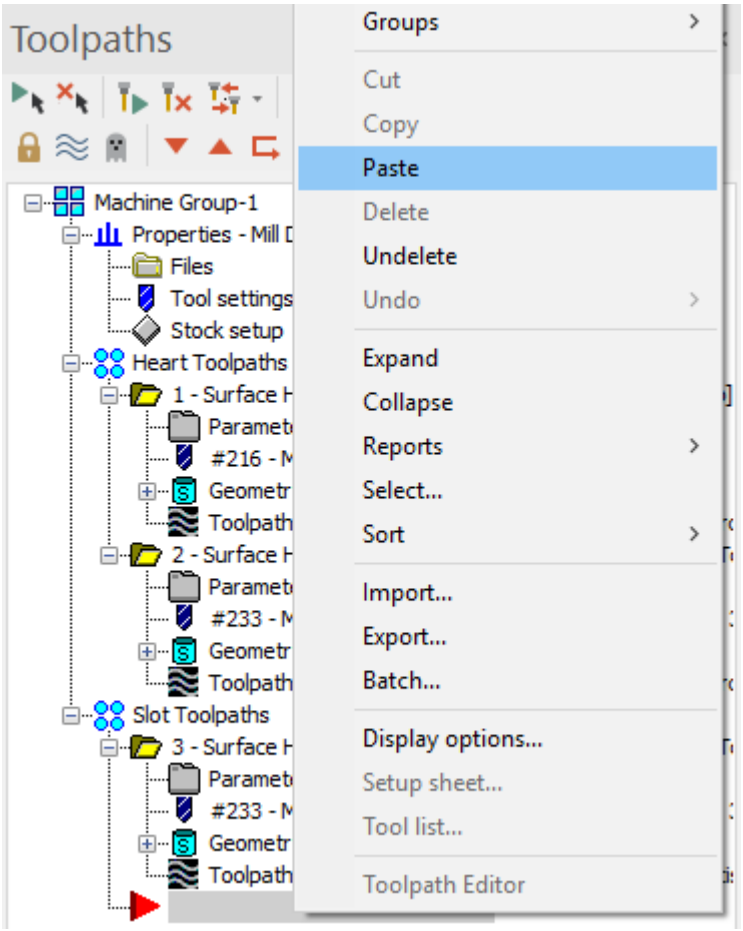
Exercise 2: Creating the Finish Scallop Toolpath for the Slot

In this exercise, you will copy the rough Scallop toolpath and make modifications so it can be used as a finish toolpath.

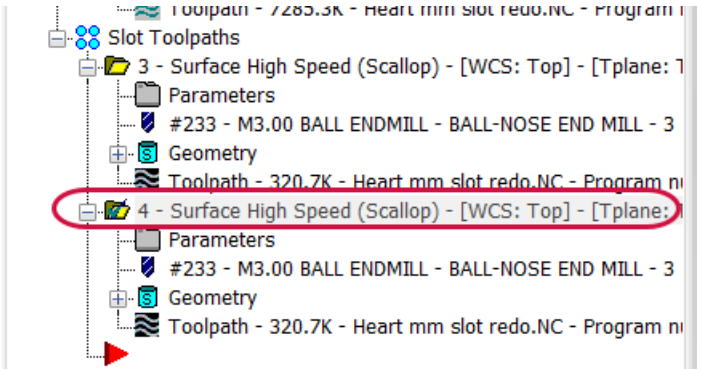
1. Your file, `Heart mm-XXX.mcam`, which you saved from the previous exercise should be open in Mastercam.
2. In Toolpaths Manager, right-click **Toolpath 3** and choose **Copy**.



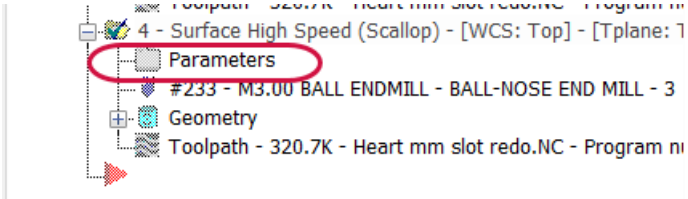
3. Ensure that the red insert arrow is below **Toolpath 3** and then right-click and choose **Paste**.



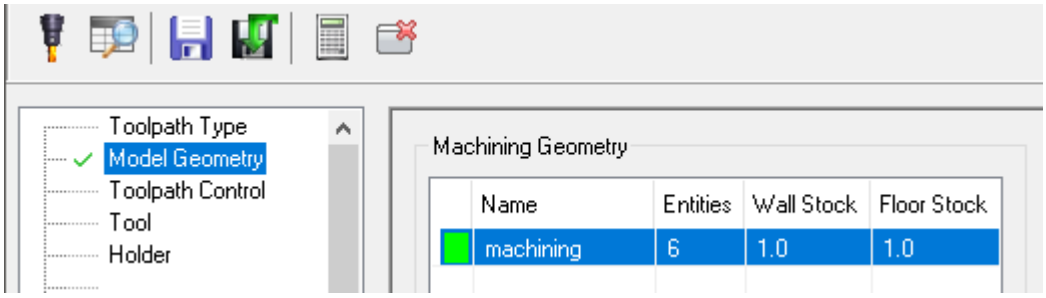
Toolpath 4, which is a duplicate of **Toolpath 3** appears in the Toolpaths Manager.



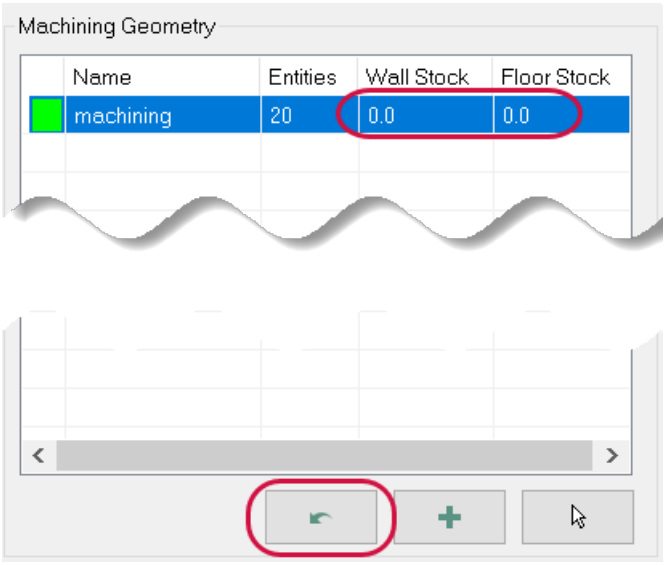
4. Click **Parameters** for **Toolpath 4**.



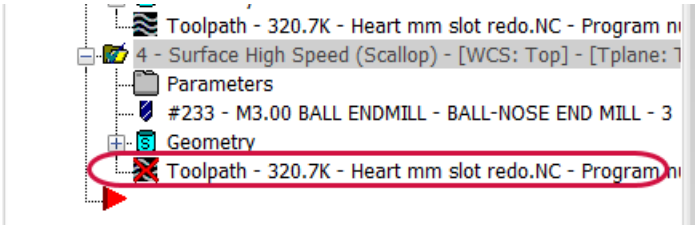
5. Open the **Model Geometry** page.



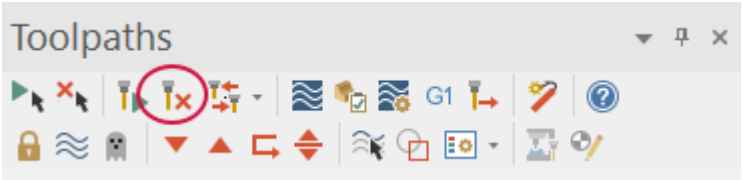
6. The finish toolpath will remove the stock to leave amounts that were entered for the rough Scallop toolpath. Click **Reset stock values** to return **Wall Stock** and **Floor Stock** to **0.0**.



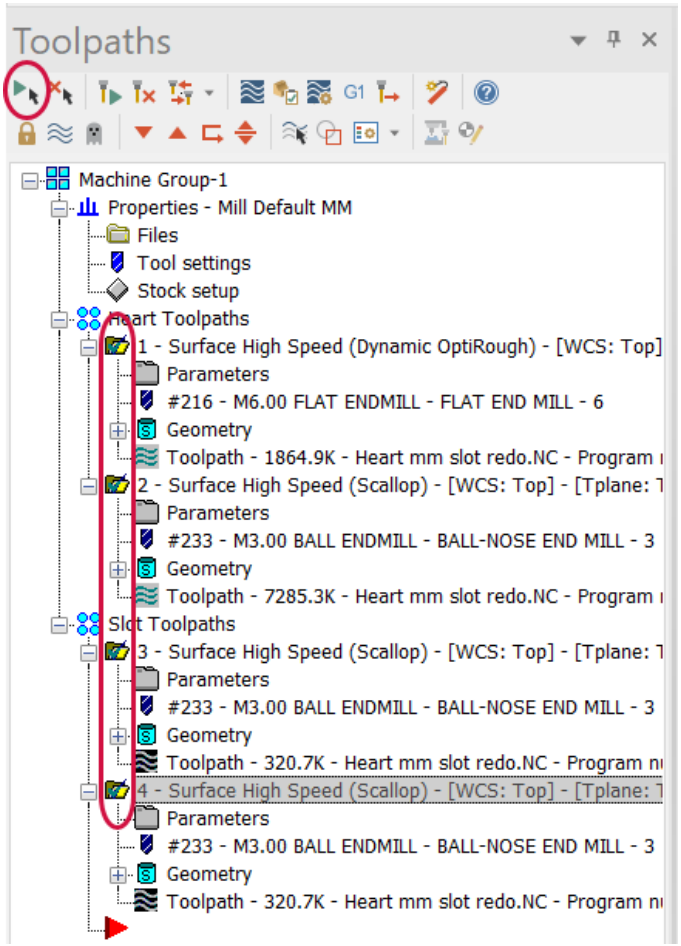
7. Click **OK**. (All other pages retain the default settings.)
8. In the Toolpaths Manager, **Toolpath 4** is marked dirty, indicating that it must be regenerated.



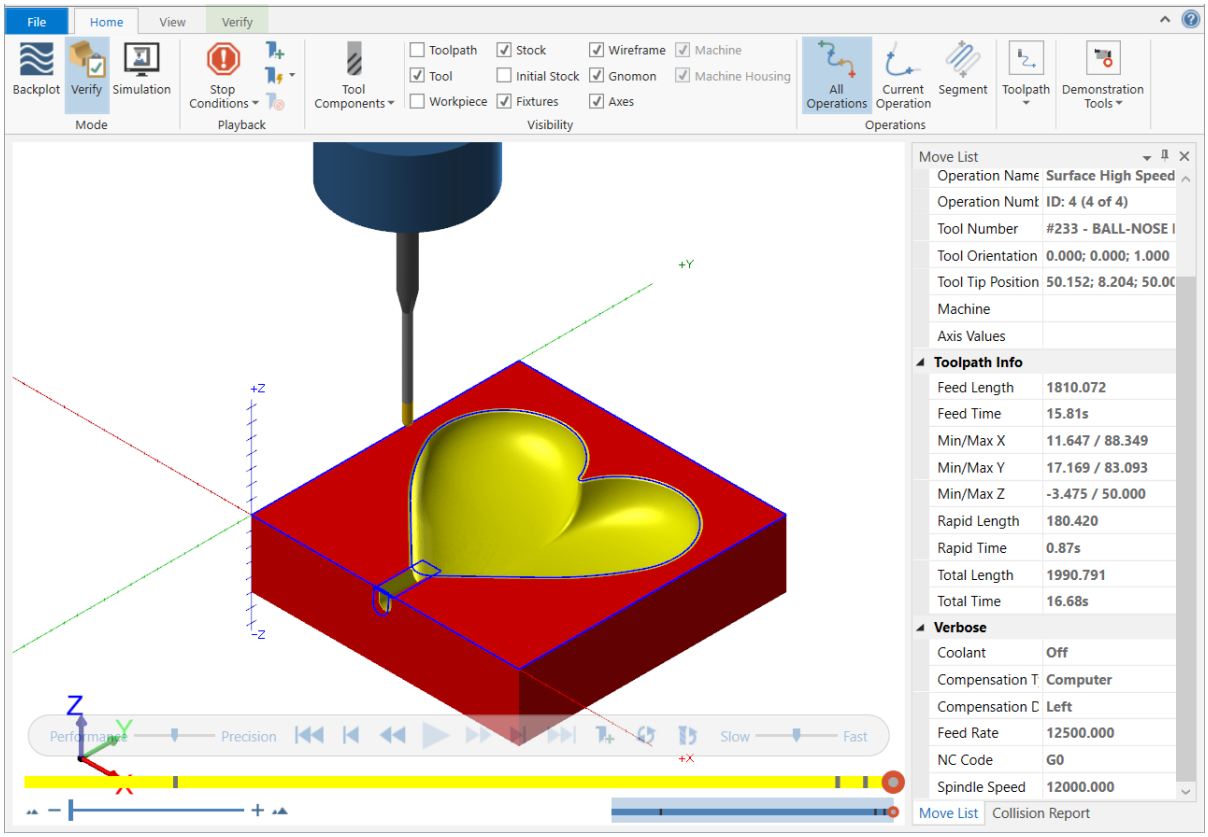
9. Click **Regenerate all dirty toolpaths** on the Toolpaths Manager toolbar.



10. In the Toolpaths Manager, click **Select all operations**. Each toolpath will have a green check indicating it is selected.



11. Click **Verify** to open Mastercam Simulator.



12. Play the simulation through then use the **Visibility** controls on the ribbon to change what you see. Experiment with the playback controls at the bottom of the screen to move forward and backward through the simulation.

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

13. Close Mastercam Simulator when you are done.
14. Save the file. You are now ready to send your file to the post processor and create the NC program.

SHOW US WHAT YOU LEARNED!

Can you answer these questions?

1. You can use filters to narrow the type and number of tools you see in the tool library.
    a. True
    b. False
2. What is the best way to reuse a toolpath?
    a. Copying and pasting it in the Toolpaths Manager
    b. Exporting the parameters into an Excel file and reimporting the operation with a different name into your part file.
3. You must select toolpath geometry one entity at a time.
    a. True
    b. False
4. You can select machining surfaces and set stock to leave values on the Model Geometry page of a toolpath dialog box.
    a. True
    b. False
5. Which of the following is not a compensation option for Containment Boundary?
    a. Inside
    b. Above
    c. Center
    d. Outside
6. The only way to tell how long an operation will take to complete is to run it on the machine.
    a. True
    b. False
7. Describe how you could get a better view of an issue with a deep cut that you notice when you are using Verify. Make sure you mention the controls you would use.

CHAPTER 6

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 mill machine chosen for this project. If you use a different machine, it will require a different post.

Goals

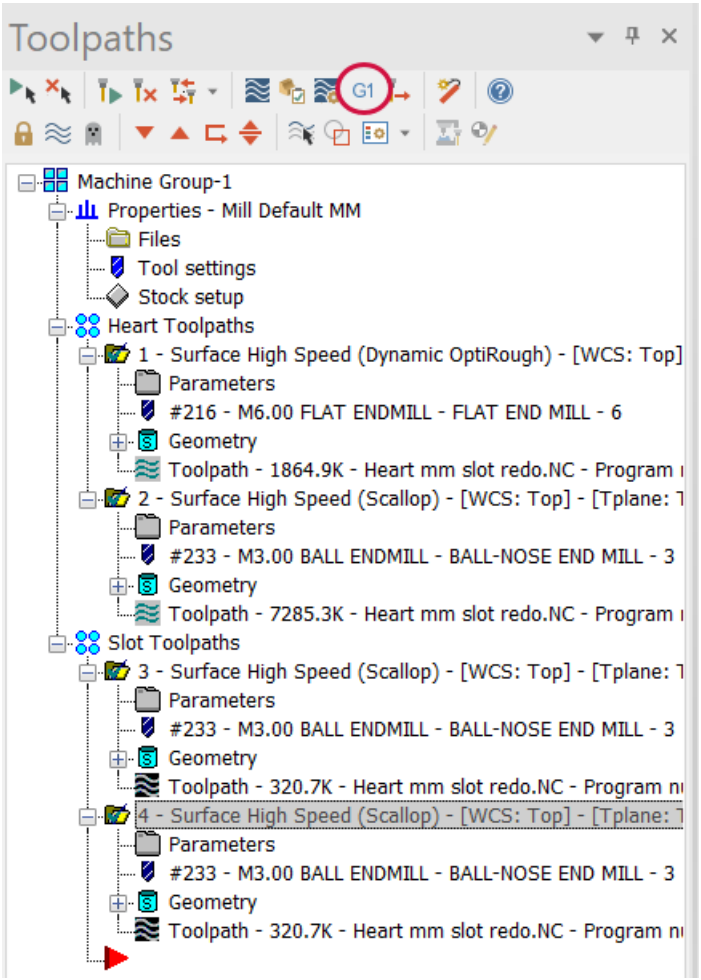
- Understand post processing concepts
- Explore Code Expert and the NC code

Exercise 1: Post Processing the Operations

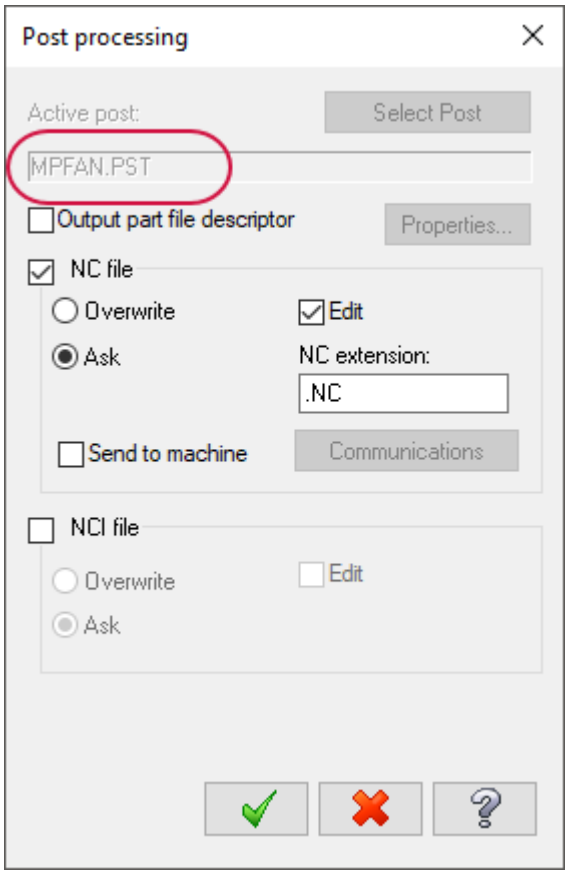
In this exercise, you will send the completed file to post processor. The post processor creates a machine-readable NC program from the Mastercam file.

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

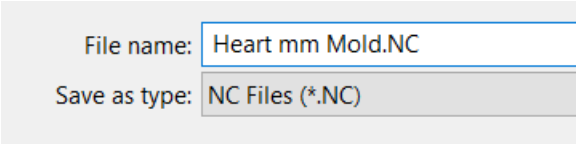
3. Click **G1** to post the toolpaths for the selected operations. The post processor creates a machine-readable NC program from the Mastercam file.



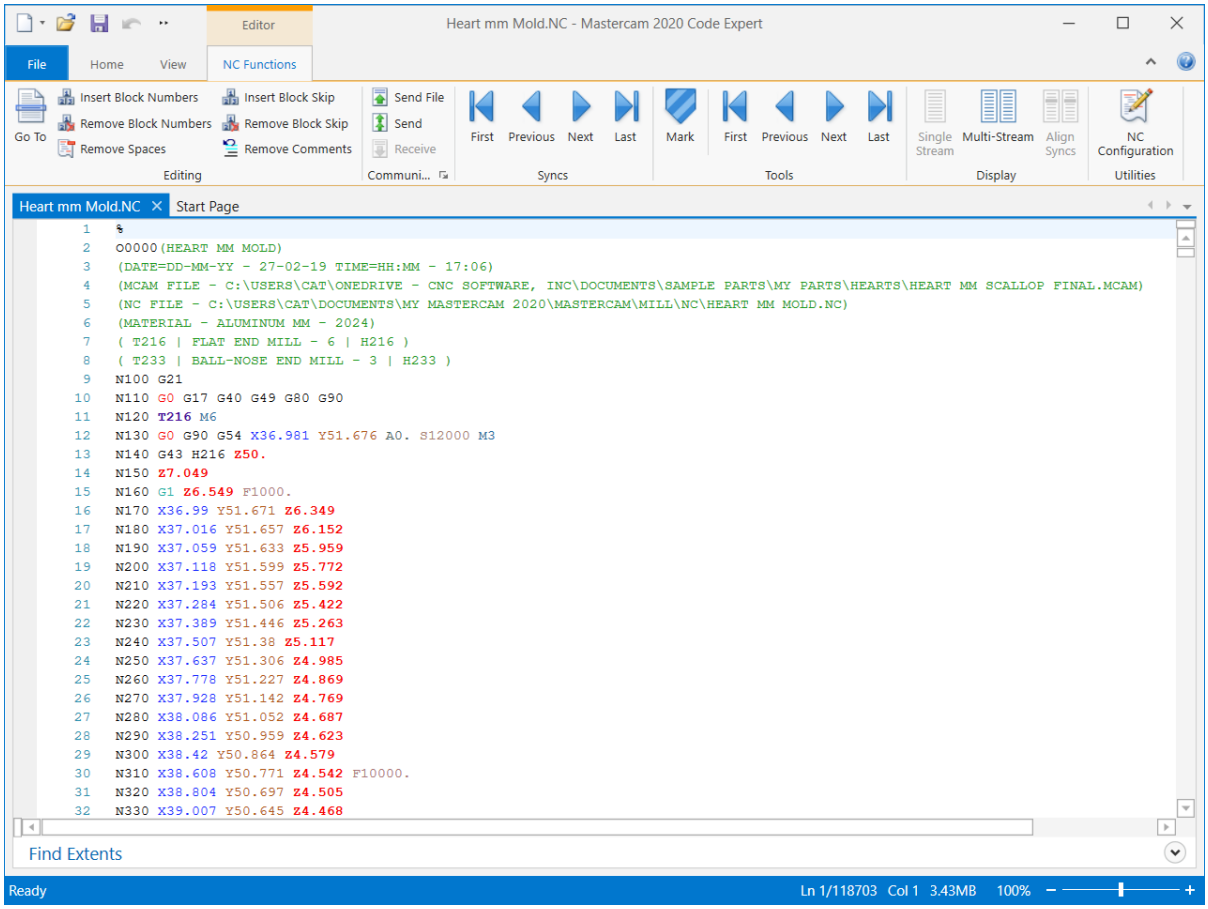
The **Post processing** dialog box displays. The name of the post processor for the machine you have chosen displays as the **Active post** field in read-only mode.



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.

> **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 on the CNC machine.
    a. True
    b. False
4. Describe how you transfer the information in your Mastercam file to your machine?

CHAPTER 7

CHALLENGE: SELF-GUIDED PROJECT

Now that you have a basic understanding of designing and programming a mold in Mastercam, it's time to make one of your own. For this self-guided project, you will be creating a mold in Mastercam with a maximum size of 100 mm x 100 mm x 25.4 mm. The model you use for your mold may be obtained from a number of online resources including GrabCAD (www.grabcad.com) and the SOLIDWORKS CAD model library (https://my.solidworks.com/cadmodels/explore) or created on your own using the powerful CAD features found in Mastercam.

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

- How to use Mastercam's wireframe and solid model design tools to create a machinable, solid model.
- Programming a part with roughing and finishing toolpaths.
- Securing stock to the machine.
- Confirming toolpath operations with Mastercam Simulator (Verify).

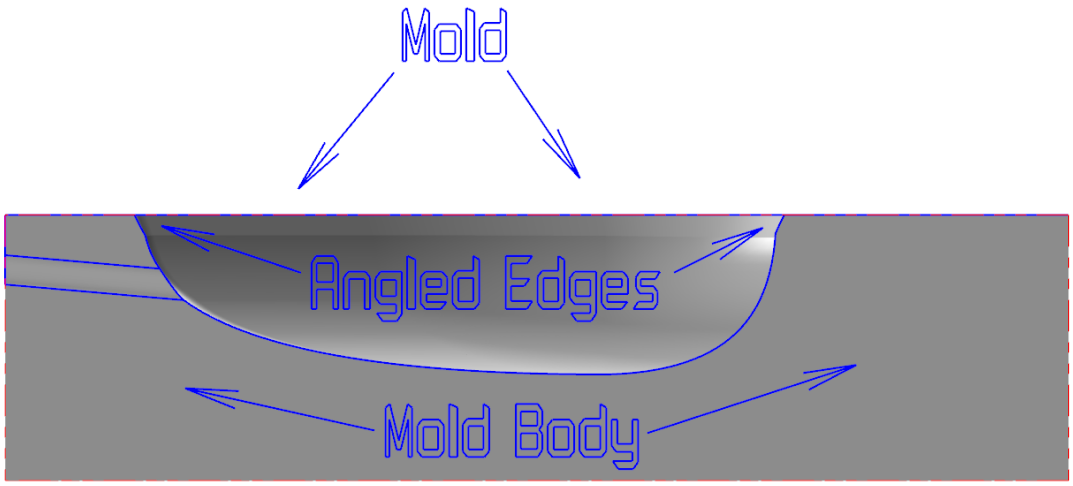
As part of this project, you will be expected to prepare a brief presentation. The presentation should be no longer than 5 minutes and clearly outline the steps that you have taken to complete the project, including which ideas you explored, what new technology you learned to facilitate the process, projects 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:

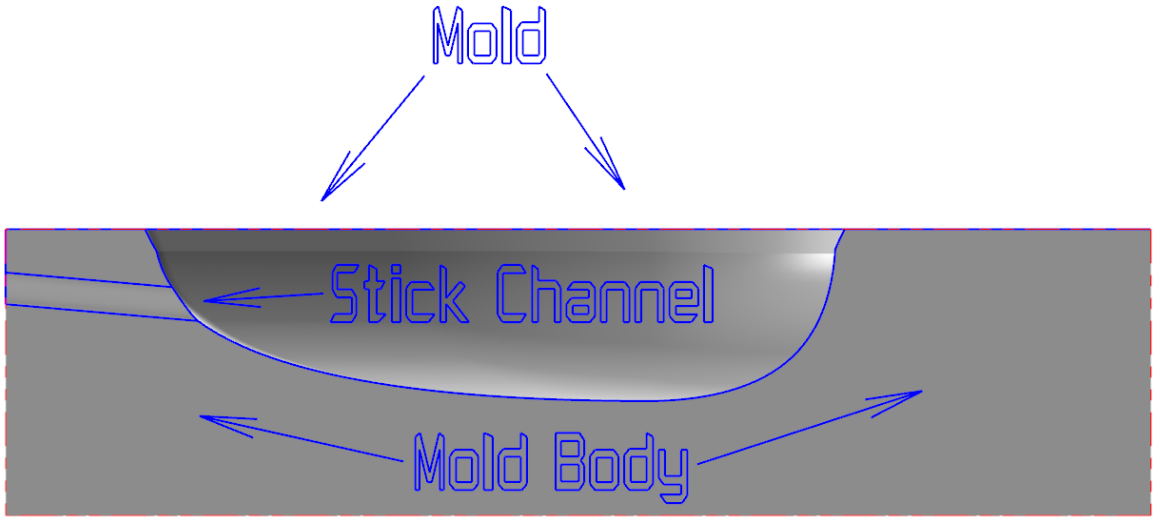
- Creativity – the process and ability to explore and express multiple ideas in a unique way.
- Initiative – the ability to overcome projects independently and with a positive attitude.
- Iteration – the ability to develop the product with progressive improvement over time.
- Continued Learning – the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

MOLD MODEL TIPS

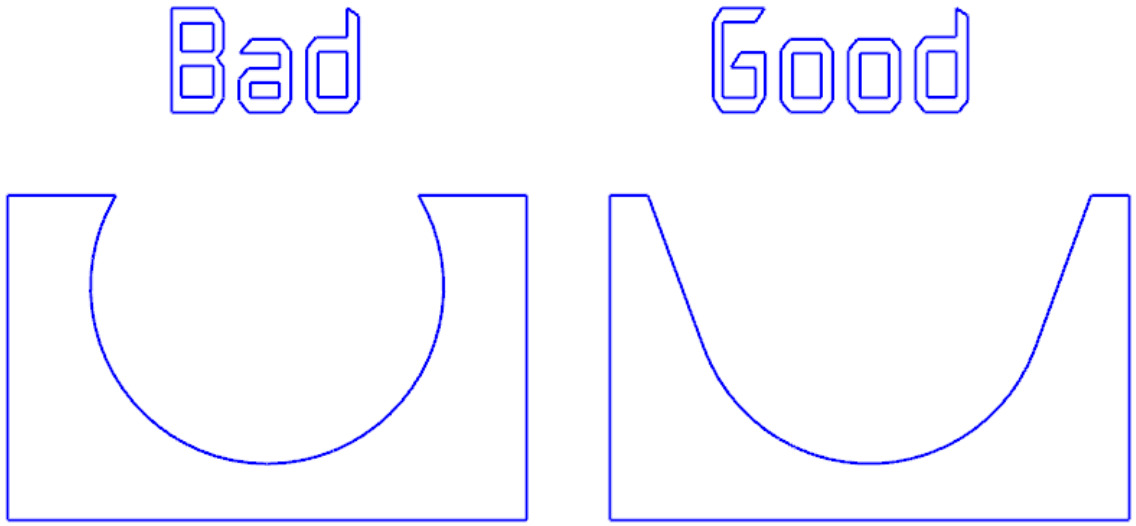
- **Use draft** when possible to make removing your part easier. A good mold would be shaped like an upside-down pyramid, where the widest part is at the top and the narrowest at the bottom. A good example of a mold is an ice cube try, where there is a taper from the top down and inward to the bottom.
- **Add an angled or tapered edge** to the top to help release the mold part from the mold body if the walls of the mold are very steep.



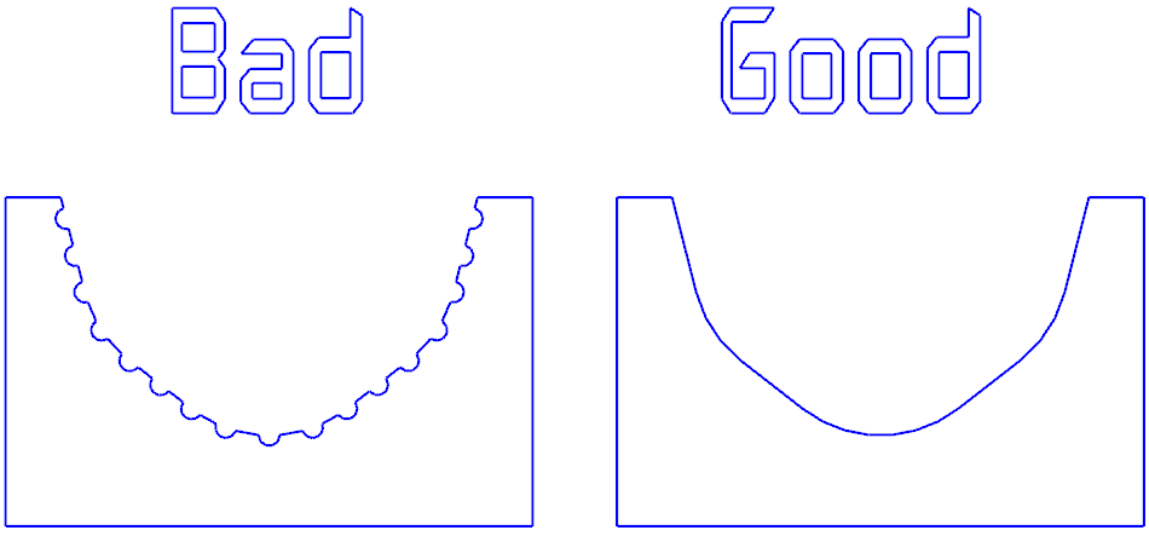
- **Put a stick in it!** A stick in the mold can be useful in pulling the part cleanly out of the mold. Remember, you will have to create and cut a channel add this stick to your mold, and you'll need to consider how to keep the part material leaking out.



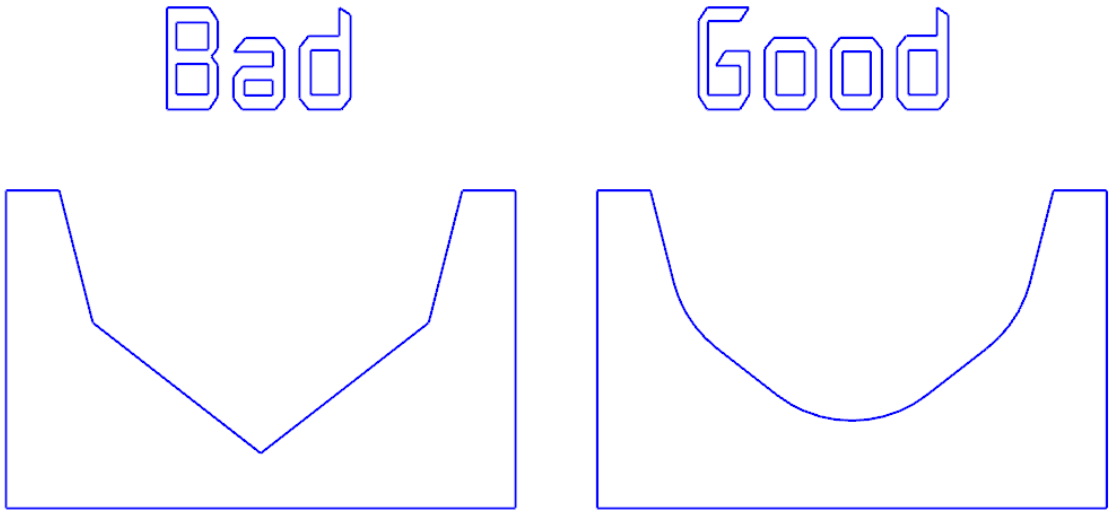
- **Avoid models with undercuts.** An undercut is when the top of the model is narrower than the bottom.



- **Avoid textured surfaces.** The smoother the surface, the easier it will be to remove the part from the mold. Textured surfaces will grip your material. It is likely that your part will not release cleanly from the mold and could be damaged in the process.



- **Avoid sharp corners and knife-edges.** Use fillets and chamfers to soften these when they can't be avoided.



- **Be flexible!** Rigid material can make it difficult to remove a part from your mold, even in the best of circumstances. Consider using a flexible mold body like silicone or even a multiple piece mold so that the part can be removed without damage.
- **Use a release agent.** A mold may be the perfect shape to release and they part could still not come out easily or at all. Consider coating your mold with a release agent, such as petroleum jelly, baby powder, nonstick spray or anything else the will help keep the mold and part from holding on to each other. Keep in mind, any possible physical or chemical affects your release agent will have on the finished part!
- **Consider shrinkage.** Shrinkage is a measurement based on change and is represented as either a ratio or as a percentage. Most materials will expand when heated and will shrink when cooled. This can cause the part to either pull away from the mold and make it easier to release, or cause the mold to tighten around the part and make it harder to release. It is not uncommon when making a candy with a mold, to chill the mold after the part material is added to help the candy shrink and release from the mold.

  If the finished part must be a specific size, you may need to be scale the mold for shrinkage so that the part is the correct size. Use the following equation to determine the difference between the mold and part dimension is:

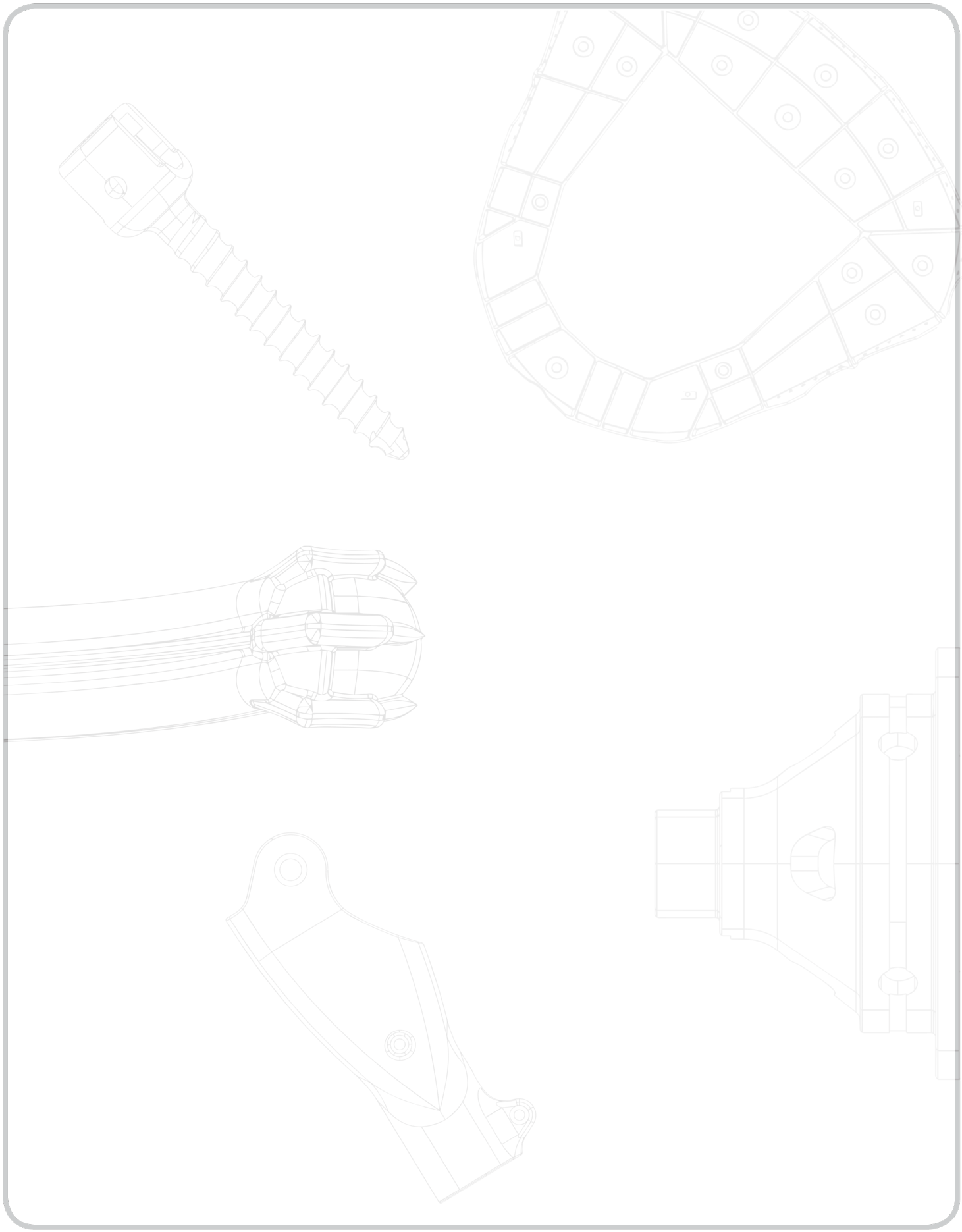
  Ratio = (mold – part) /mold

  % = (mold – part) (100%) / mold

CONCLUSION

Congratulations! You have completed the *Quick Part Series - Chocolate Mold*! Now that you have mastered the skills in this tutorial, explore Mastercam's other features and functions.

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



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

www.mastercam.com

Mastercam