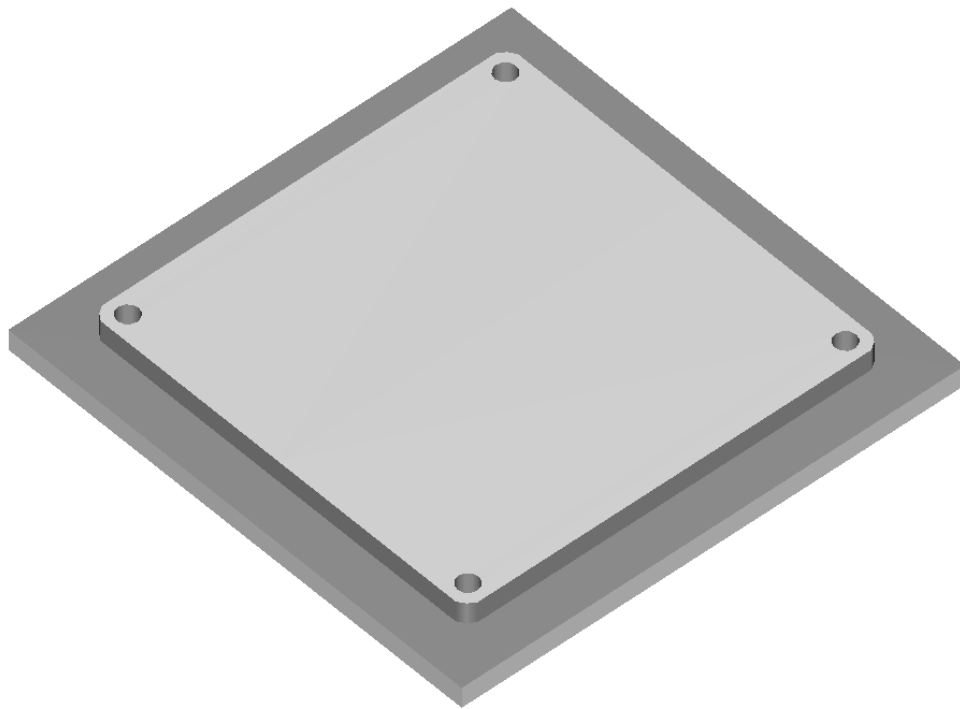


Mastercam 2020

TRAINING

GUIDE



MILL-LESSON-4

DRILL AND CONTOUR TOOLPATHS

camInstructor

Objectives

You will create the geometry for Mill-Lesson-4, and then generate a toolpath to machine the part on a CNC vertical milling machine. This lesson covers the following topics:

Create a 2-dimensional drawing by:

Creating rectangles.
Creating arcs.

Establish Stock Setup settings:

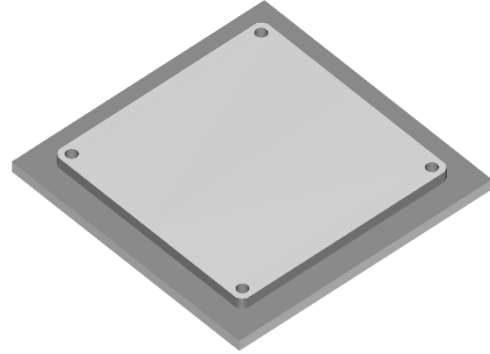
Stock size.
Material for the part.
Feed calculation.

Generate a 2-dimensional milling toolpath consisting of:

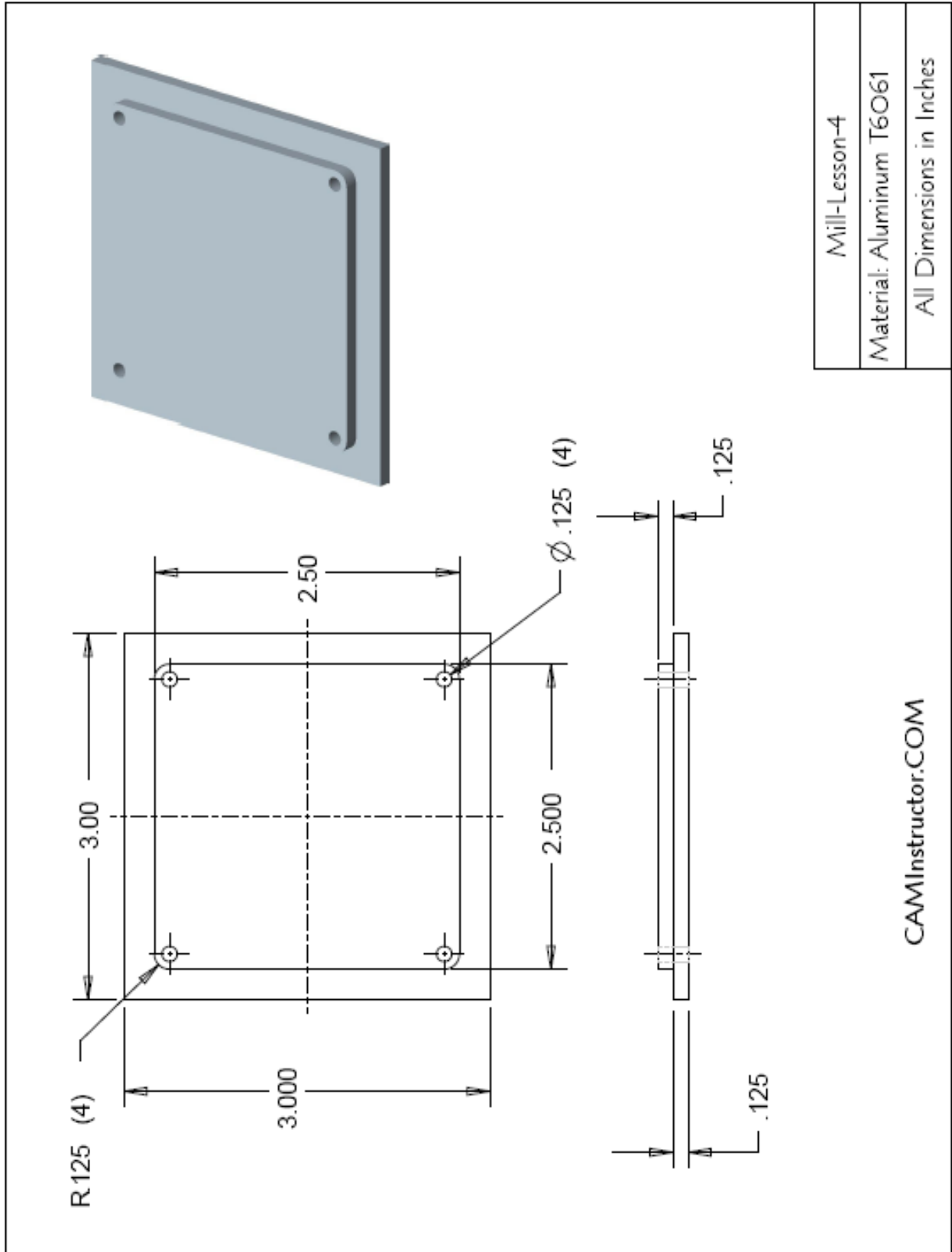
Contour toolpath.
Drill toolpath.

Inspect the toolpath using Mastercam's Verify and Backplot by:

Launching the Verify function to machine the part on the screen.
Using Backplot to identify the correctness of the toolpaths.
Generating the NC- code.



MILL-LESSON-4 DRAWING



TOOL LIST

Two cutters will be used to create this part.

The .125" diameter two flute center cutting flat end mill will be used to drill the four .125" diameter holes.

The .5" diameter two flute flat end mill will be used to machine the contour.

Center Cutting End Mills

- ☞ These two flute center cutting end mills can plunge straight down into the material.
- ☞ In this lesson the holes could be spot drilled and then drilled; using the center cutting end mill enables just one tool to complete the drilling of the holes.



MILL-LESSON-4 - THE PROCESS

Geometry Creation

TASK 1: Set the environment

TASK 2: Create a rectangle 3" x 3" – the center is at X0 Y0

TASK 3: Create the inside entities

TASK 4: Create the four .125" fillet radii

TASK 5: Save the drawing

Toolpath Creation

TASK 6: Define the rough stock using stock setup

TASK 7: Drill the 4 x .125" diameter holes

TASK 8: Machine the contour

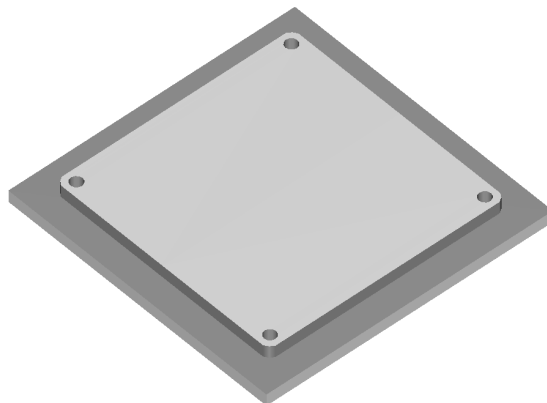
TASK 9: Backplot the toolpaths

TASK 10: Modify the contour toolpath to add roughing cuts and a finish pass

TASK 11: Verify the toolpath

TASK 12: Save the updated Mastercam file

TASK 13: Post and create the CNC code file



Geometry Creation

TASK 1:

SETTING THE ENVIRONMENT

Before starting the geometry creation you should set up the grid and machine type as outlined in the **Setting the environment** section at the beginning of this text:

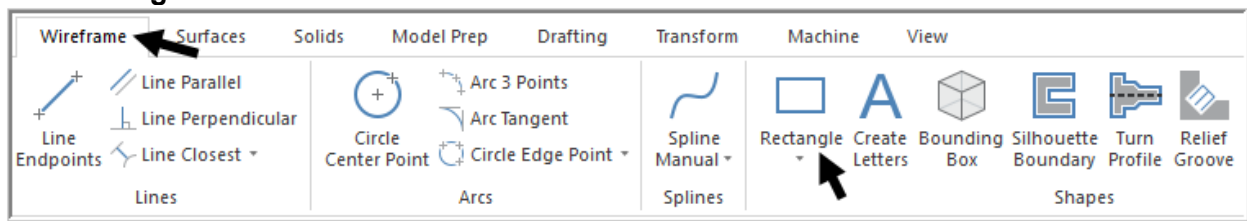
1. Set up the Grid. This will help identify the location of the origin.
2. Set the machine to the Default Mill.

TASK 2:

CREATE A RECTANGLE 3" X 3" – THE CENTRE IS AT X0 Y0.

This task explains how to create the 3" square. These four lines could be created in many different ways, this is just one option.

1. Select the **Wireframe** tab at the top of the screen and in the **Shapes** section click on **Rectangle**.



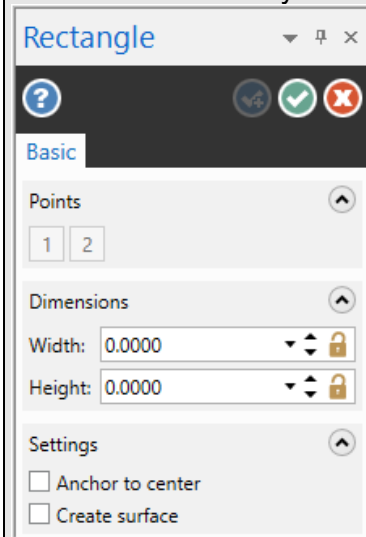
Rectangle

Use this function to create a rectangle using two points. To draw a rectangle, click to set the base point that anchors the rectangle, drag the rectangle outward, then click to set the second corner.

You can also enter X, Y, and Z positions to position the two points on the rectangle. The rectangle can be drawn outward from a center point, and can be drawn as a surface. Width and height are lockable so that you can create multiple rectangles using set dimensions.

The rectangle remains a live entity until you click the OK button or start another rectangle.

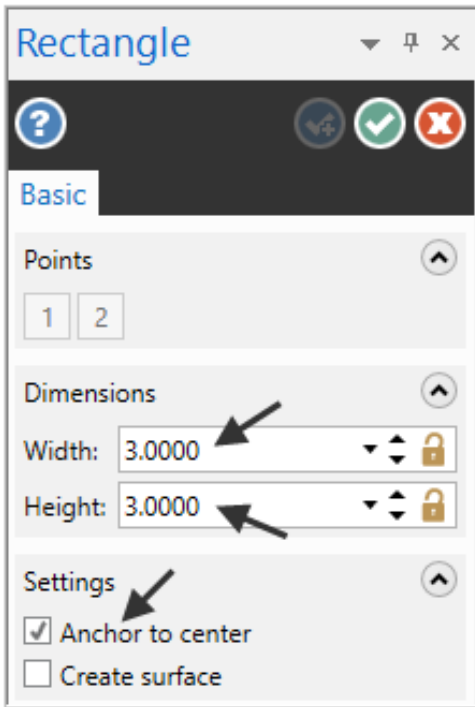
Note: While the entity is live, you can select 1 or 2 in the Point section to reposition the corners.



- On the graphics screen you are prompted: **Select a new position for the first corner** and the Rectangle panel appears.

Mastercam Training Guide

- Click in the space for **Width** and enter a value of **3.0**.
- In the **Height** section enter a value of **3.0** and then hit Enter.
- Now activate the **Anchor to Center** option.



Anchor to center:

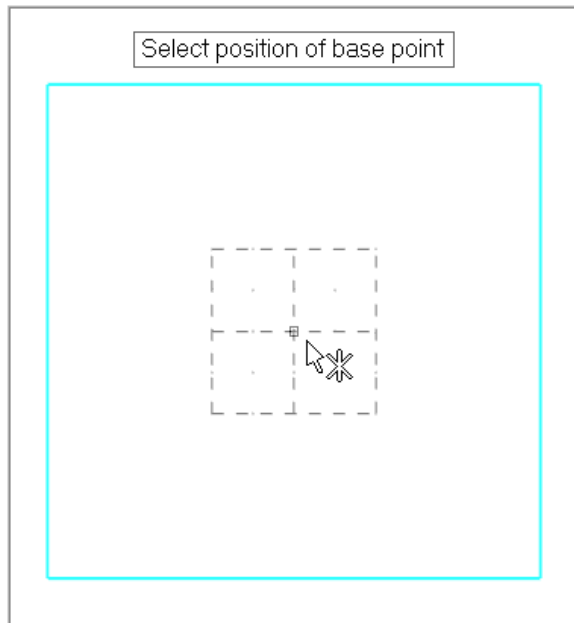
Sets the base point of the rectangle to the center point, and draws the rectangle outward from the center.


When using the **Anchor to center** and the rectangle is still a live entity, you can reposition the center point by using the **Points 1** button.

Points

1 2

- Move the cursor to the **center of the Grid** and **snap to the Origin** for the base point.

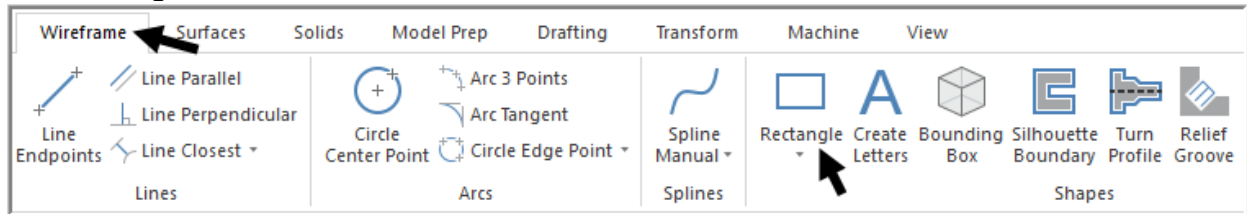


- Click on the **OK** icon  to complete this feature.
- Select the **View** tab at the top of the screen.
- Select the **Fit** icon in the **Zoom** section. Next select **Unzoom 80%** to shrink the display.

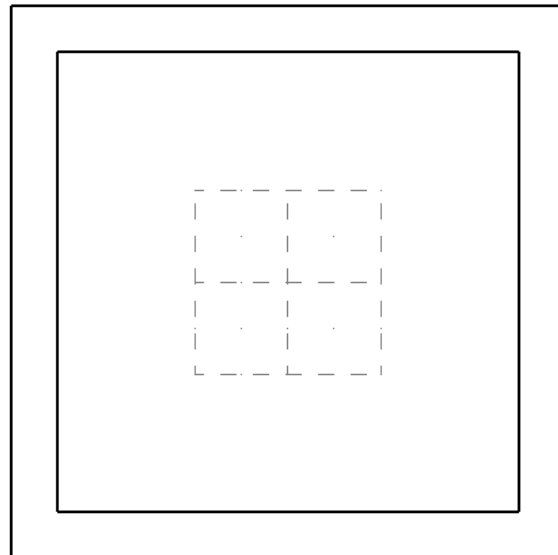
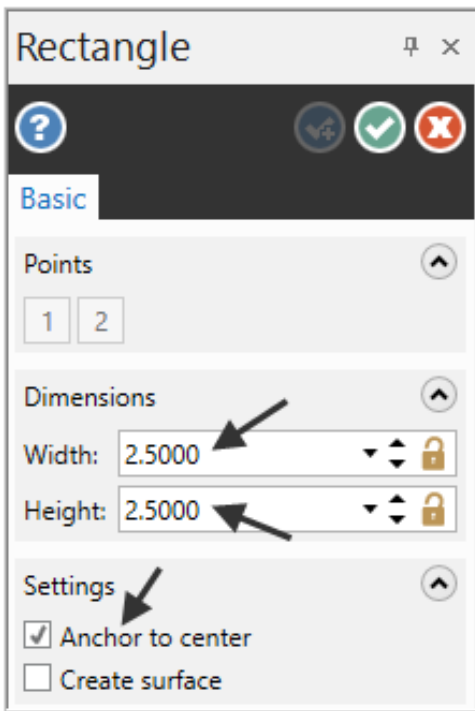
TASK 3: CREATE THE INSIDE ENTITIES


- In this task you will create the four inside lines. These lines form a 2.5" square.
- You will use Rectangle again to construct these four inside lines.

1. Select the **Wireframe** tab at the top of the screen and in the **Shapes** section click on **Rectangle**.



- On the graphics screen you are prompted: **Select a new position for the first corner** and the Rectangle panel appears.
2. Click in the space for **Width** and enter a value of **2.5**.
 3. In the **Height** section enter a value of **2.5** and then hit Enter.
 4. Ensure **Anchor to Center** is activated.

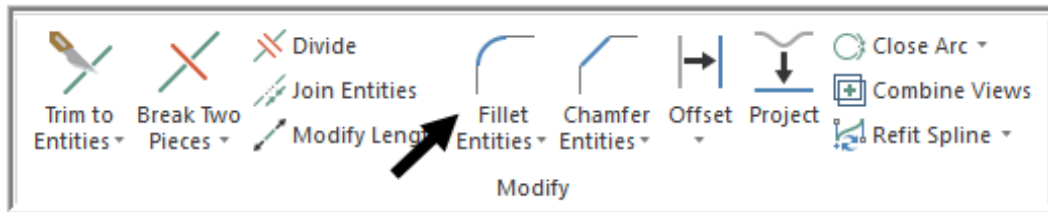


5. Move the cursor to the **center of the Grid** and **snap to the Origin** for the base point.
 6. Click on the **OK** icon  to complete this feature.
- Your part geometry should look as above right: 3.0" and 2.5" square.

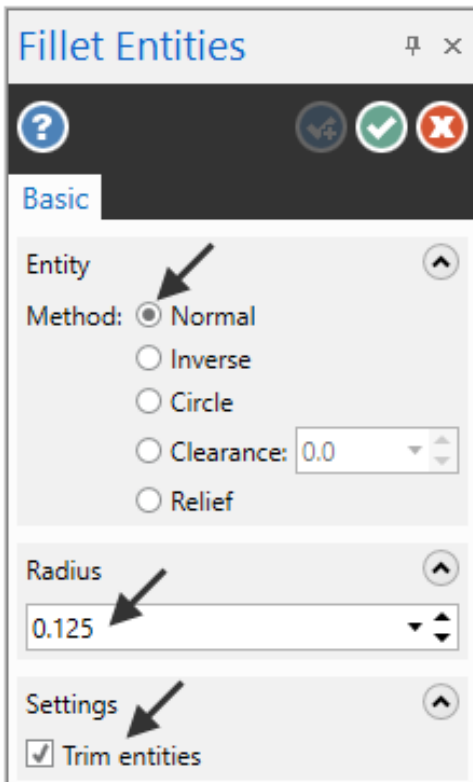
TASK 4: CREATE THE FOUR .125" FILLET RADII

➤ In this task you will create the four .125 fillet radii on the corners of the 2.5" square you have just created.

1. Select the **WIREFRAME** tab at the top of the screen and in the **Modify** section click on **Fillet Entities**.



2. The **Fillet Entities** panel appears. If required activate the **Method** to **Normal** and enter a value of **0.125** for **Radius**.
3. Ensure the **Trim entities** box at the bottom of the panel is check marked to turn the trim on.



Use **Fillet Entities** to apply fillets to existing entities.

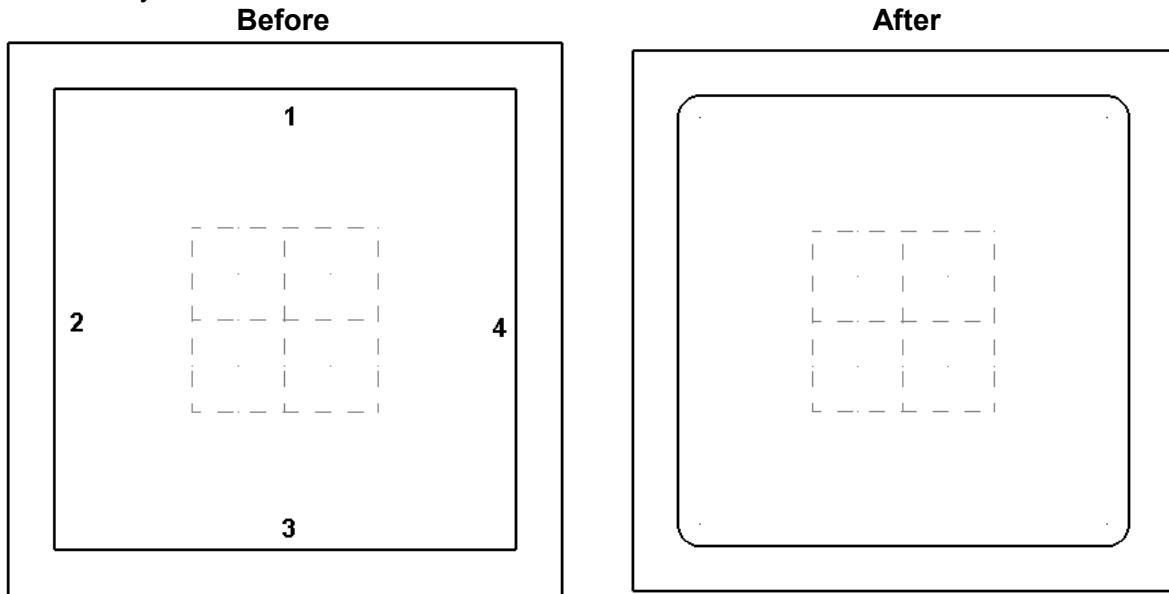
Before selecting the entities to fillet, choose the fillet Method (Normal, Inverse, Circle, Clearance, Relief), and enter the desired radius value. Then choose the first and second entities.


Mastercam's auto-preview feature displays a temporary fillet at the selected location.

You can also choose to trim to the fillet or to leave the selected lines untrimmed.

Trimming is the default, so for no trimming, uncheck **Trim entities**.

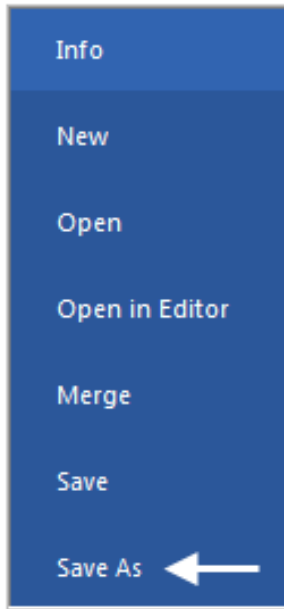
4. When prompted to “**Fillet: Select an entity**”, select Line 1 and 2 as shown below. The fillet radius appears at the corner of line 1 and 2.
5. To complete the remaining three fillet radii, select:
6. Line 2 and 3.
7. Line 3 and 4.
8. And finally line 4 and 1.



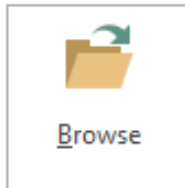
9. Click on the **OK** icon  to complete this feature.
- ➡ Your completed part geometry should look like the figure on the right above.

TASK 5: SAVE THE DRAWING

1. Select **File** at the top left corner of the screen.
2. Select **Save As**.



3. Click on the Browse icon.



4. In the **File name** box, type **Mill-Lesson-4**.
5. Browse to an appropriate location.
6. Select the **Save** button to save the file and complete this function.

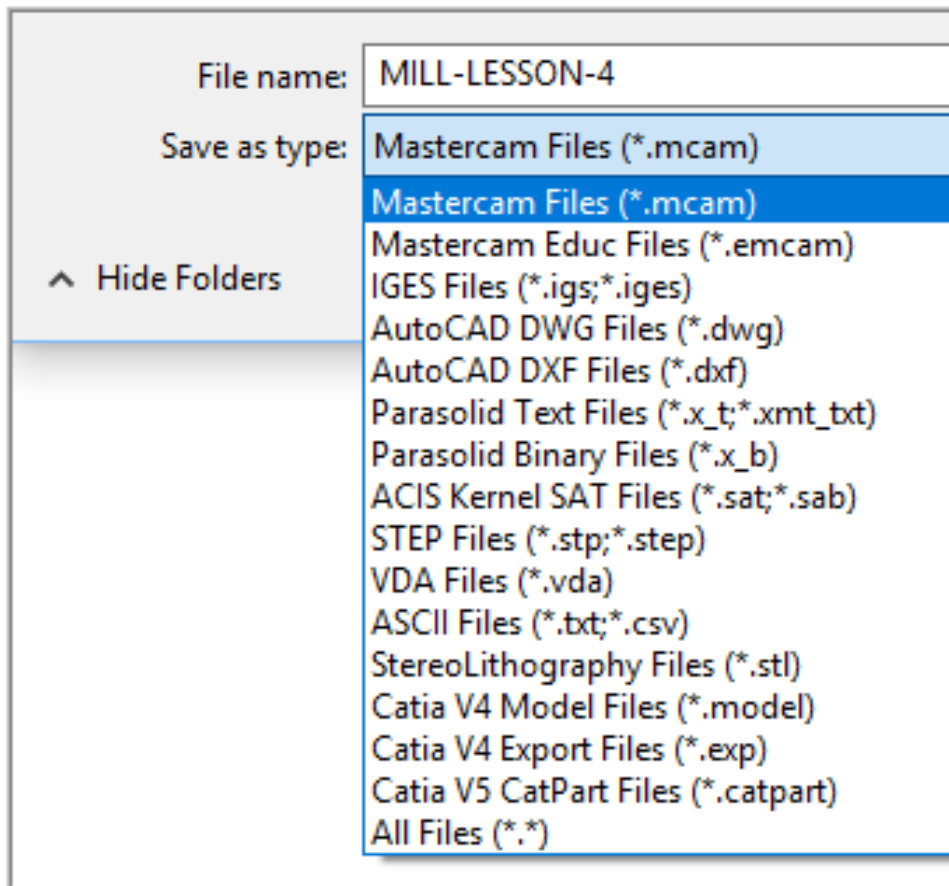


File Save As

Mastercam lets you read (import) a variety of CAD file types into the Mastercam database.

You can also write (export) Mastercam files to a variety of different file formats. See your Mastercam Reseller for more information on the options available for your Mastercam configuration.

Below is an example of the various **Save as type** file options.

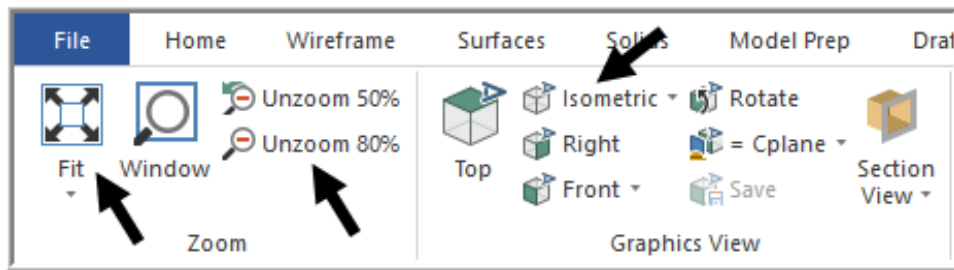


Toolpath Creation

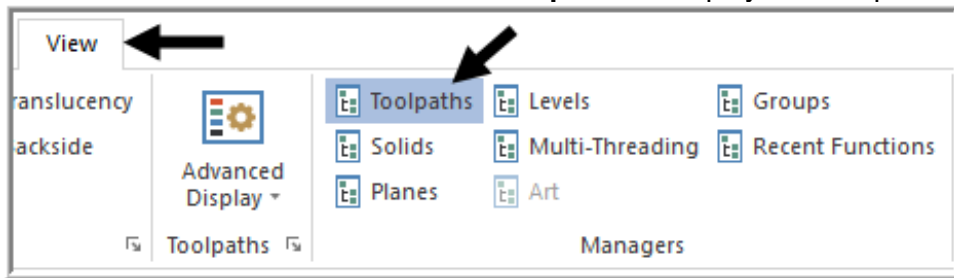
TASK 6:

DEFINING THE ROUGH STOCK USING STOCK SETUP

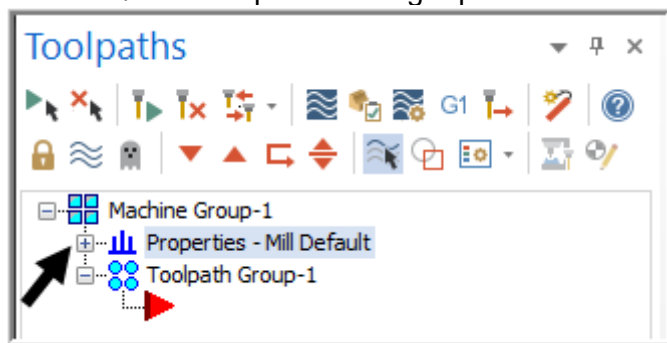
1. Select the **View** tab at the top of the screen.
2. For a better view of the part select **Isometric** in the **Graphics View** section.
3. Select the **Fit** icon in the **Zoom** section. Next select **Unzoom 80%** to shrink the display.



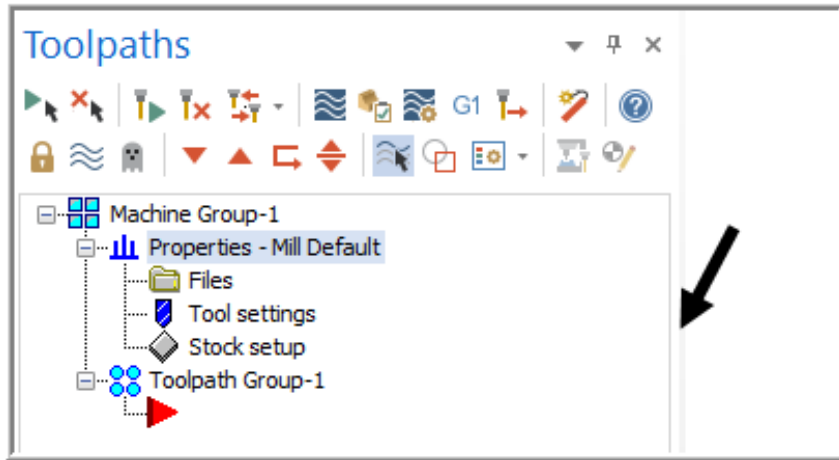
4. Still on the **View tab** and click on **Toolpaths** to display the Toolpaths Manager.



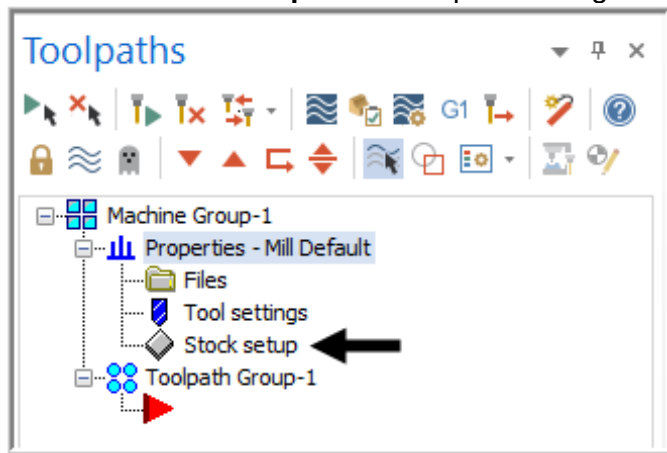
5. Select the plus in front of **Properties** to expand the **Machine Group Properties**. **Alt-O** will Show/hide Toolpaths Manager pane.



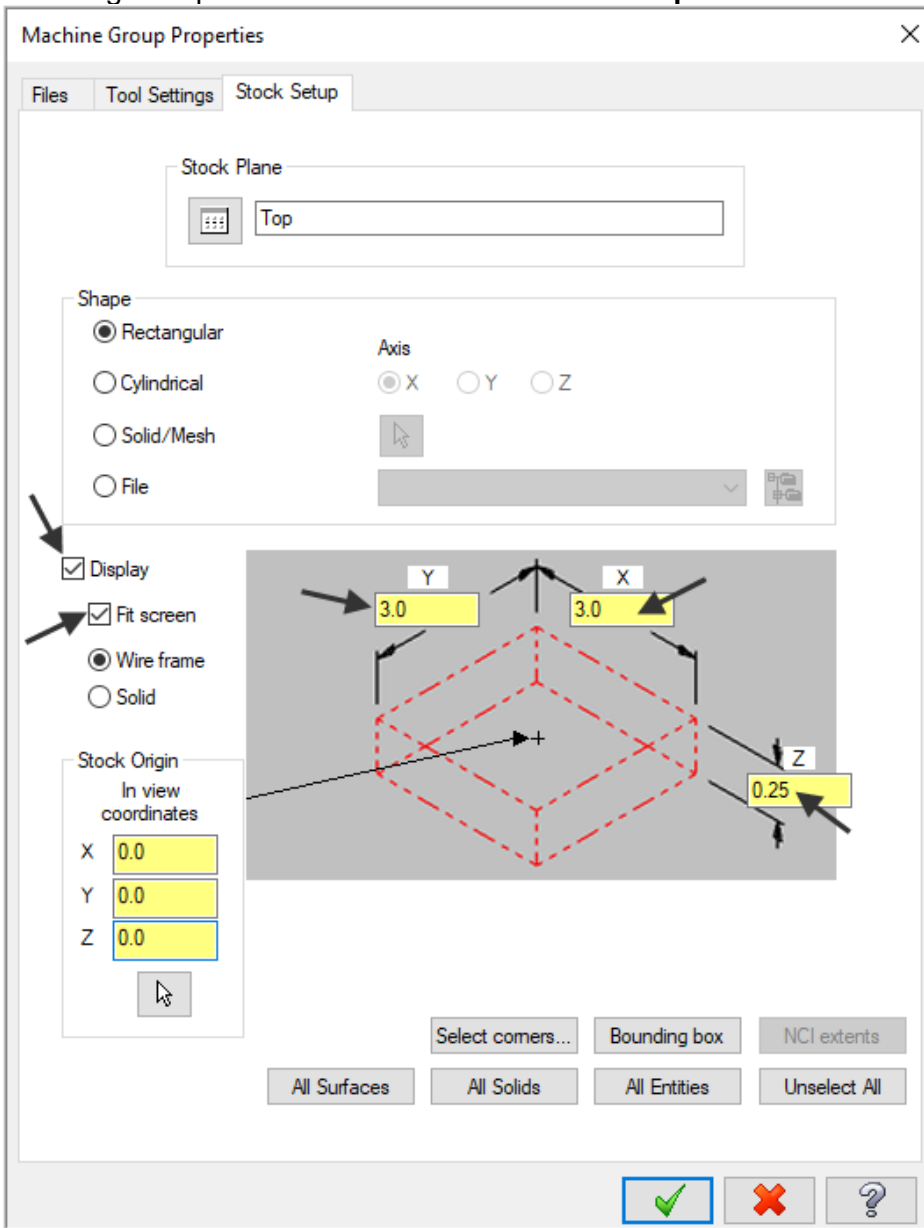
6. You may need to extend the toolpaths manager window, if so click on the right hand pane, hold the mouse button down and drag to the right.



7. Select **Stock setup** in the Toolpath manager window.



8. Change the parameters to match the **Stock Setup** screenshot below:



Stock Setup Stock Origin

The stock origin is the X-Y-Z coordinate position of the point indicated by the cross in the picture of the stock model.

Use it so Mastercam knows where your stock model is located relative to your part and coordinate system.

You can either type in the X-Y-Z coordinates directly in the fields, or choose the Select button to return to the graphics window and select a position.

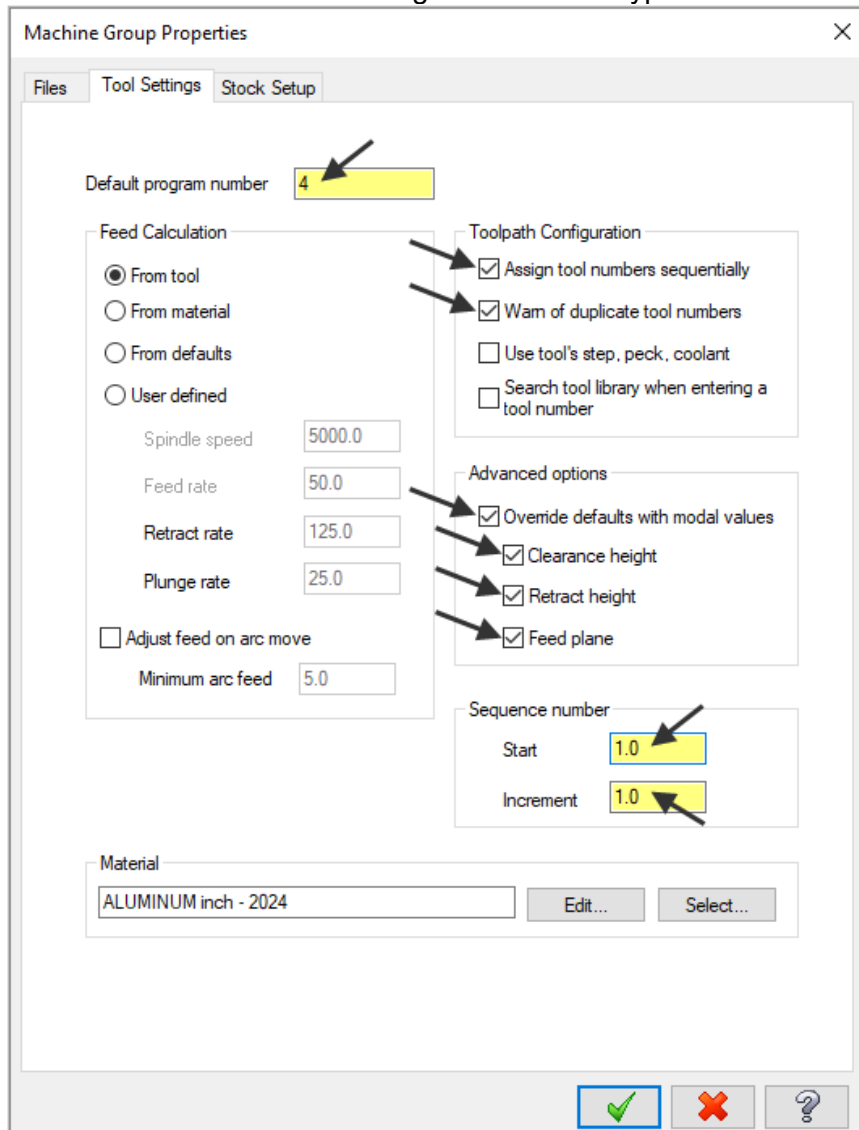
Display

Choose this option to display the stock model in the graphics window.

Fit screen

Choose this option so that the stock boundary lines are included when you use the **Fit to Screen** function.

9. Select the Tool Settings tab and change the parameters to match the **Tool Settings** screenshot below. To change the **Material** type follow the next set of instructions.



Program #

The post processor will use this field if your machine tool requires a number for a program name.

Sequence #

Enter the starting sequence number and the increment in the proper fields.

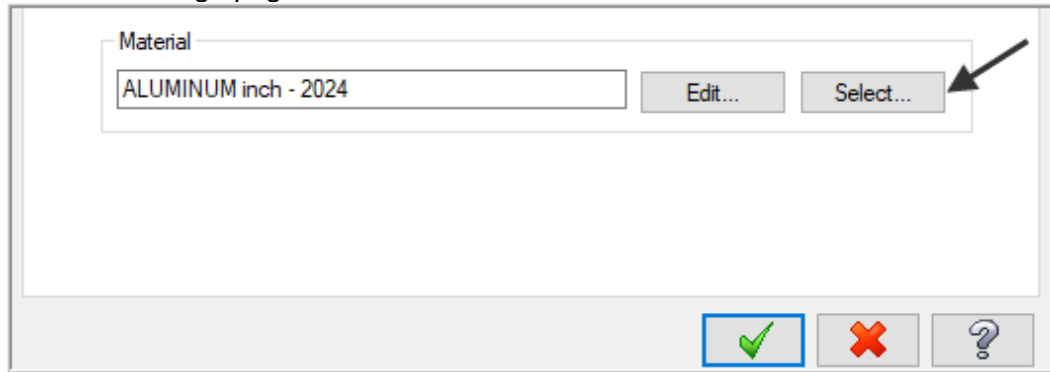
The default values and specific format for these fields is determined by the NC Output section of the control definition.

In the code file below note the block numbers are sequenced in **increments of 1**, N9 N10 etc.

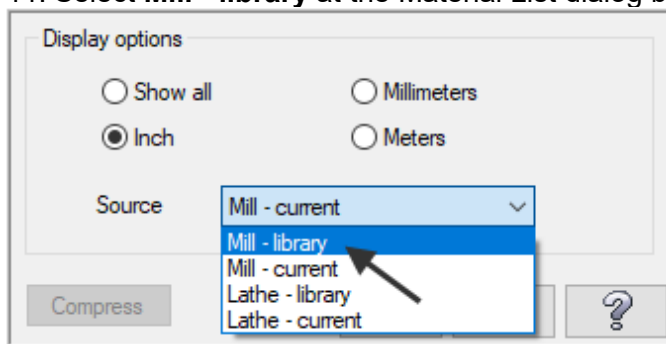
```
N9 Y-1.125
N10 X1.125
N11 Y1.125
N12 G80
N13 Z2.
N14 M5
```

Mastercam Training Guide

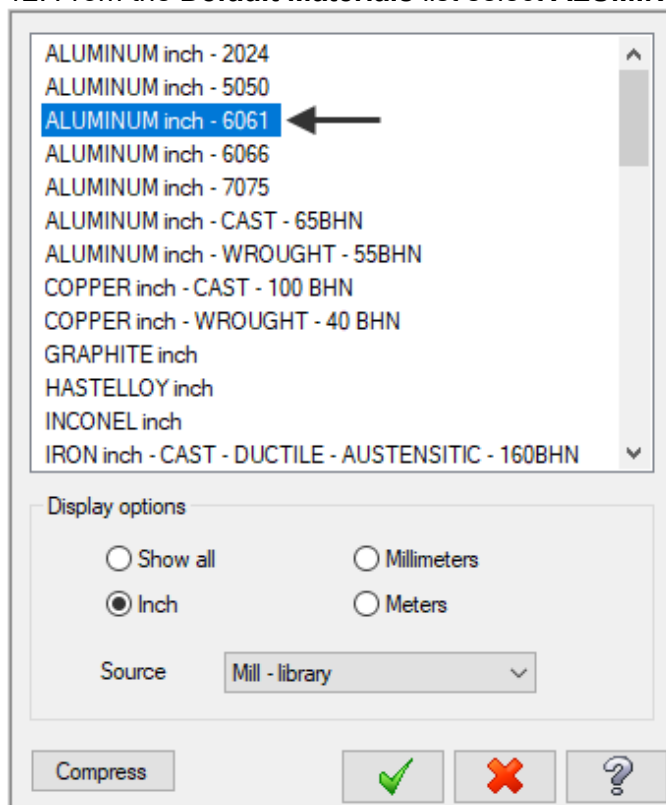
10. To change the **Material** type to **Aluminum 6061** pick the **Select** button at the bottom of the Tool Settings page.




11. Select **Mill - library** at the Material List dialog box open the Source drop down list.

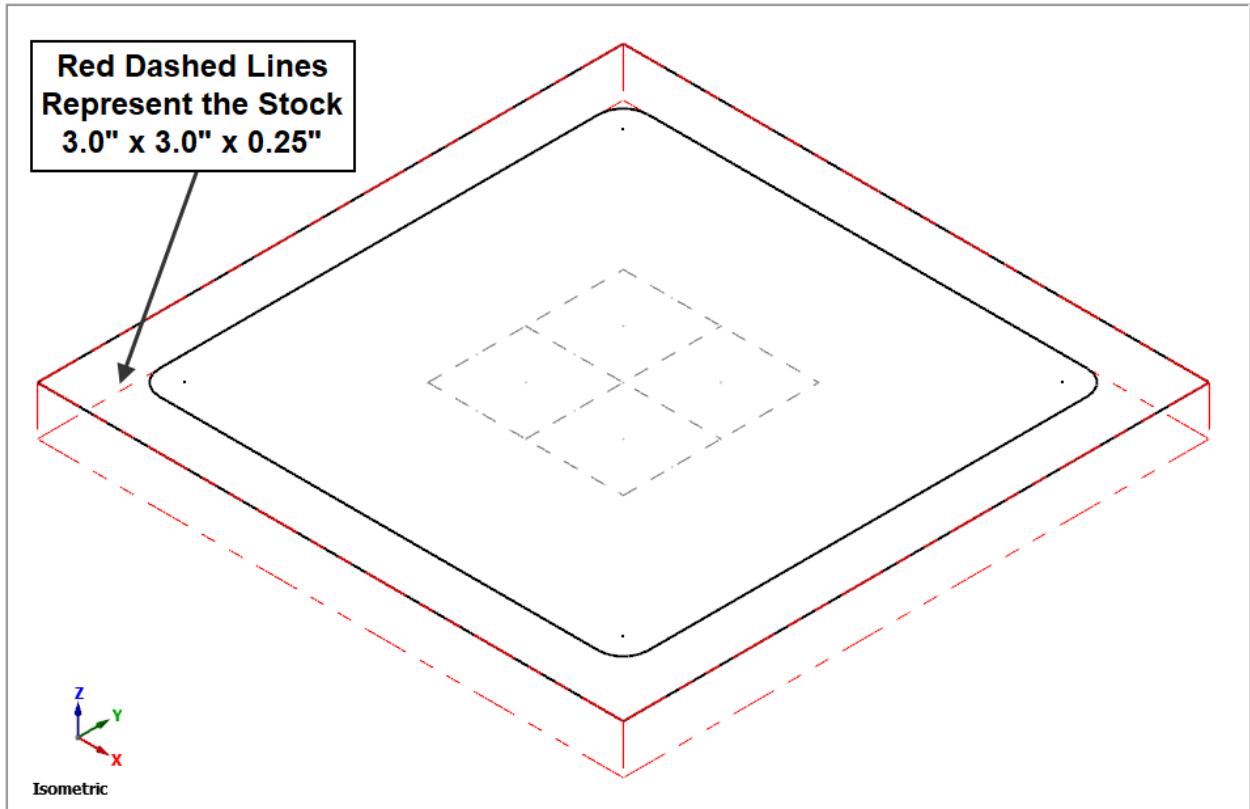


12. From the **Default Materials** list select **ALUMINUM inch -6061** and then select .



13. Select the OK button  again to complete this Stock Setup function.

☞ Your screen will look similar to the screenshot below:

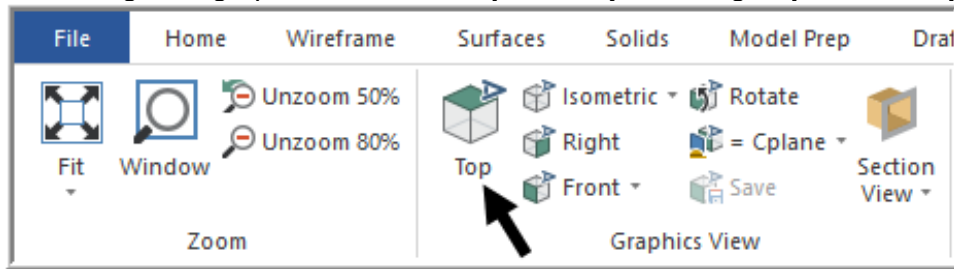


TASK 7:

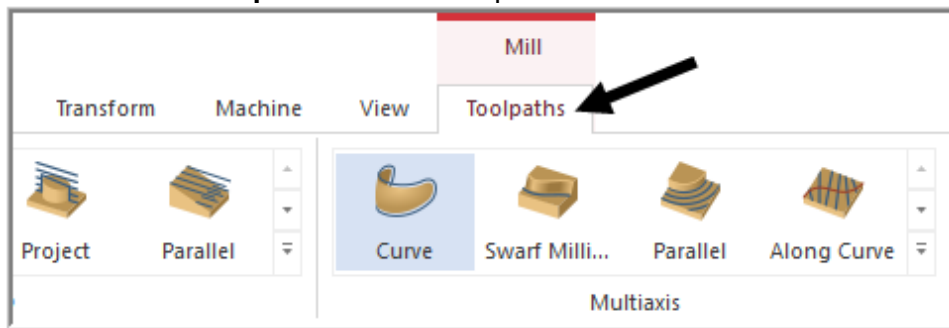
DRILL THE 4 X .125" DIAMETER HOLES

- ☞ In this task you will drill the four .125" diameter holes through the part with a center cutting two flute end mill that is .125 " diameter.
- ☞ As the four holes will be drilled at the center of the .125" fillet radii you need to create points at the center of these arcs, when creating the drilling toolpath you can snap to the center of the arcs.

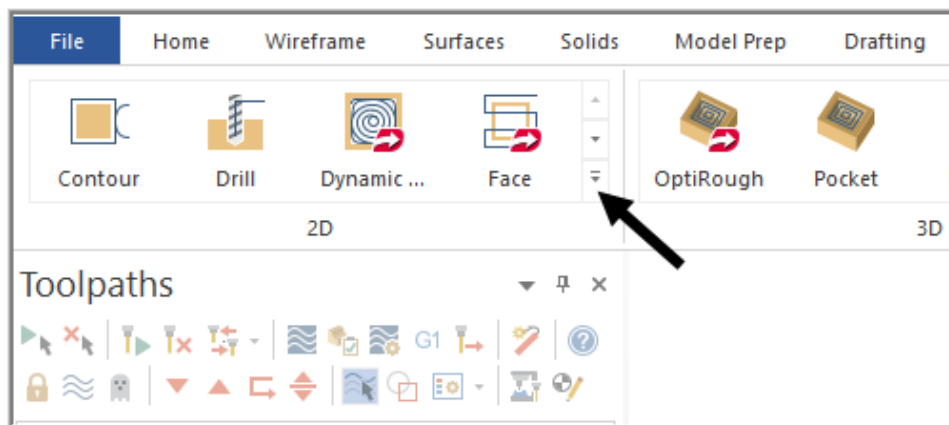
1. Select the **View** tab at the top of the screen, if required.
2. Change the graphics view to a **Top View** by selecting **Top** in the **Graphics View** section.



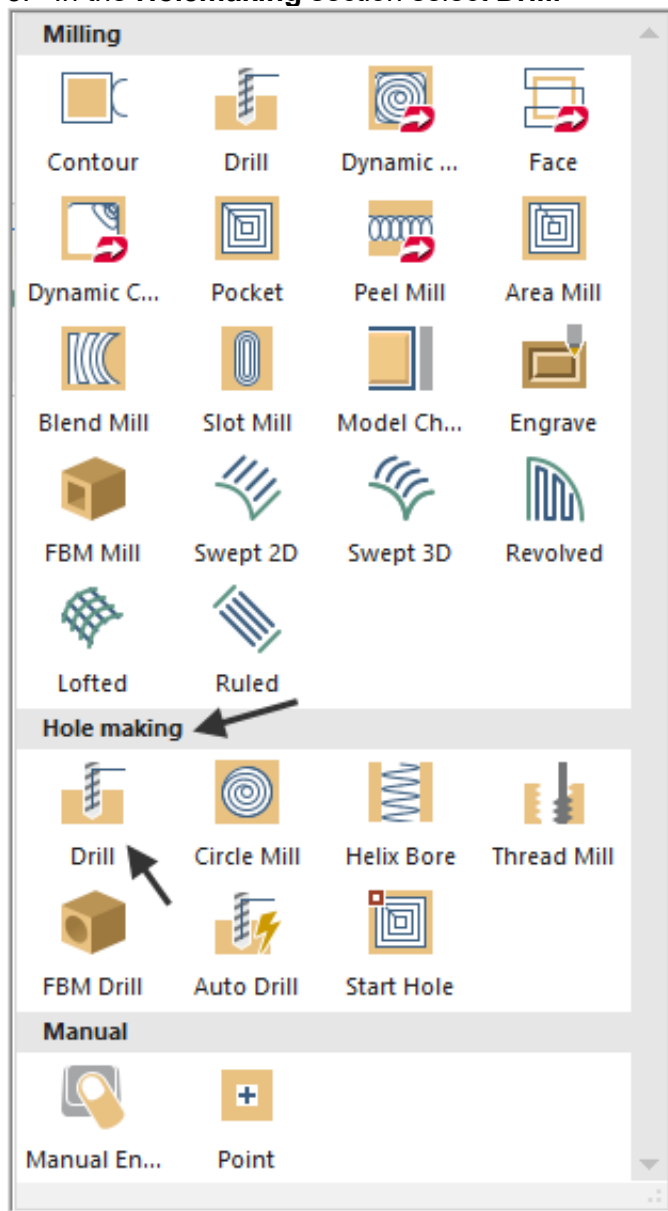
3. Select the **Toolpaths** tab at the top of the screen.



4. Click on the **Expand gallery** down arrow to view additional options in the **2D** toolpath section.



5. In the **Holemaking** section select **Drill**.

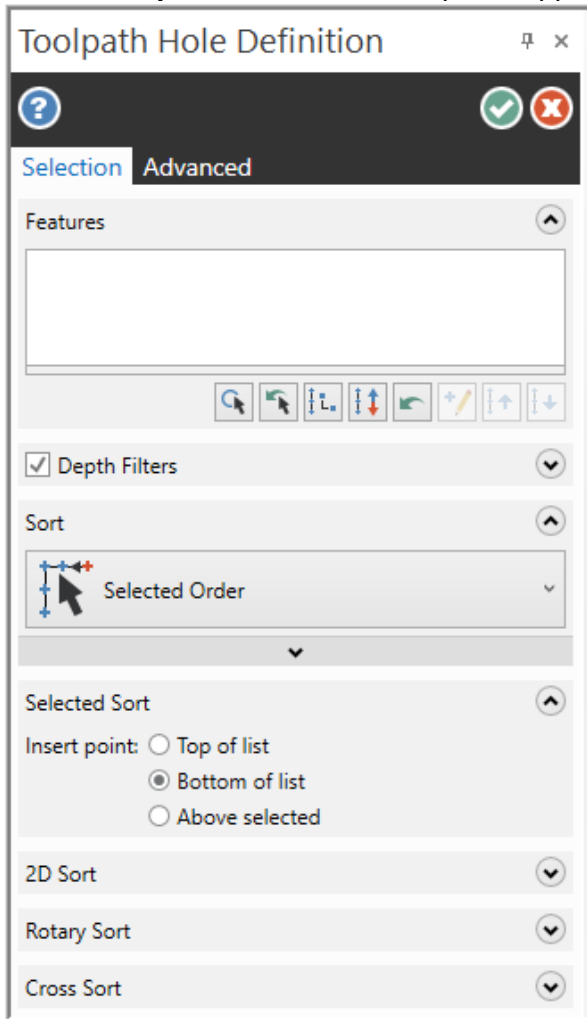


Mastercam Training Guide

Now you are prompted to:

Select one or more entities to add or remove from the features list.

6. The **Toolpath Hole Definition** panel appears:

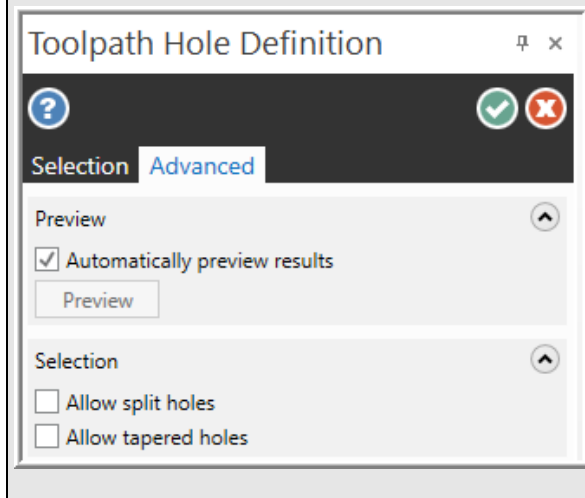


Toolpath Hole Definition

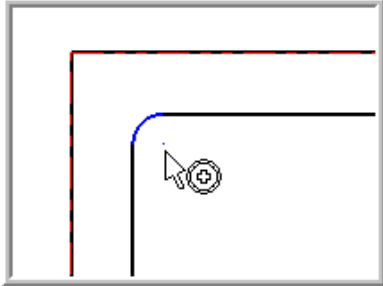
Use the Toolpath Hole Definition function panel to select holes for a hole making toolpath.

Mastercam gives you many techniques for selecting points; you can use as many methods as are appropriate for your application

Choose OK when you have selected all the points.

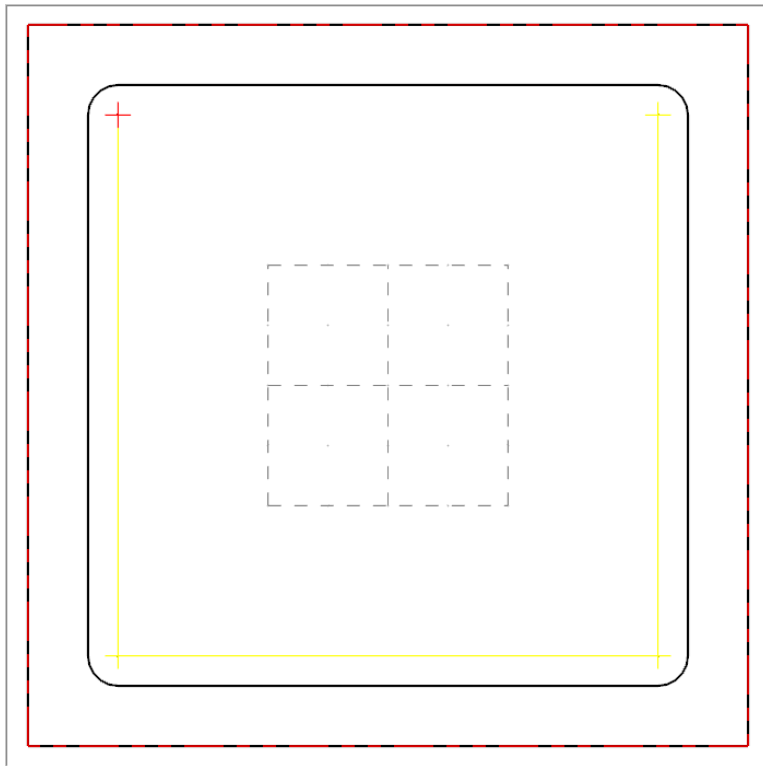


- As you need to drill the four holes at the center of the four fillet radii all you need to do now is snap to the center of the fillet radius.
- 7. Move the mouse and position the cursor over the **center of the top left hand fillet radius**, you will notice a circle appears as you move closer to the center of the radius, this is a visual cue.
- 8. This circle demotes you are snapping the center of this arc. Use the left button of your mouse to pick this center point.



- 9. Using the method outlined above continue on and pick the remaining three center points, in this order:
- 10. Lower left.
- 11. Lower right.
- 12. Upper right.

- Your screen should look like the screenshot below after selecting the four center points:

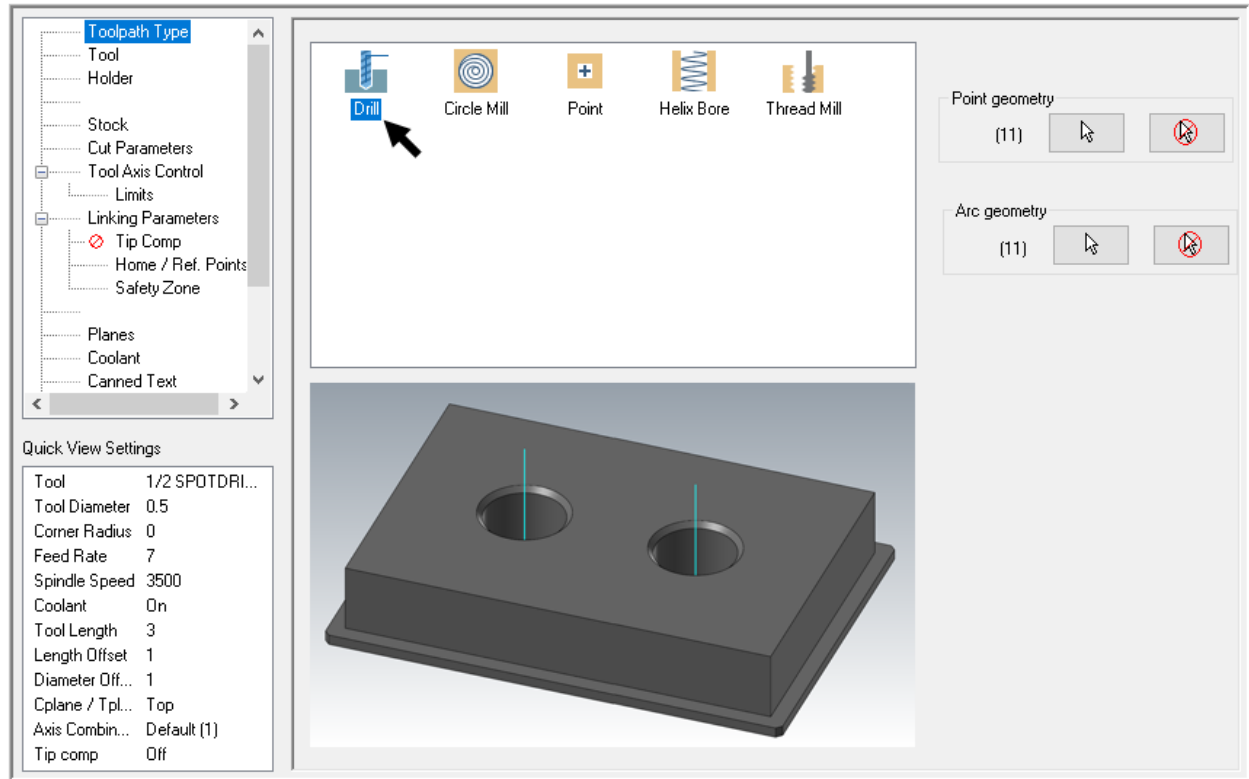


- 13. Select the OK button  in the **Toolpath Hole Definition** panel.

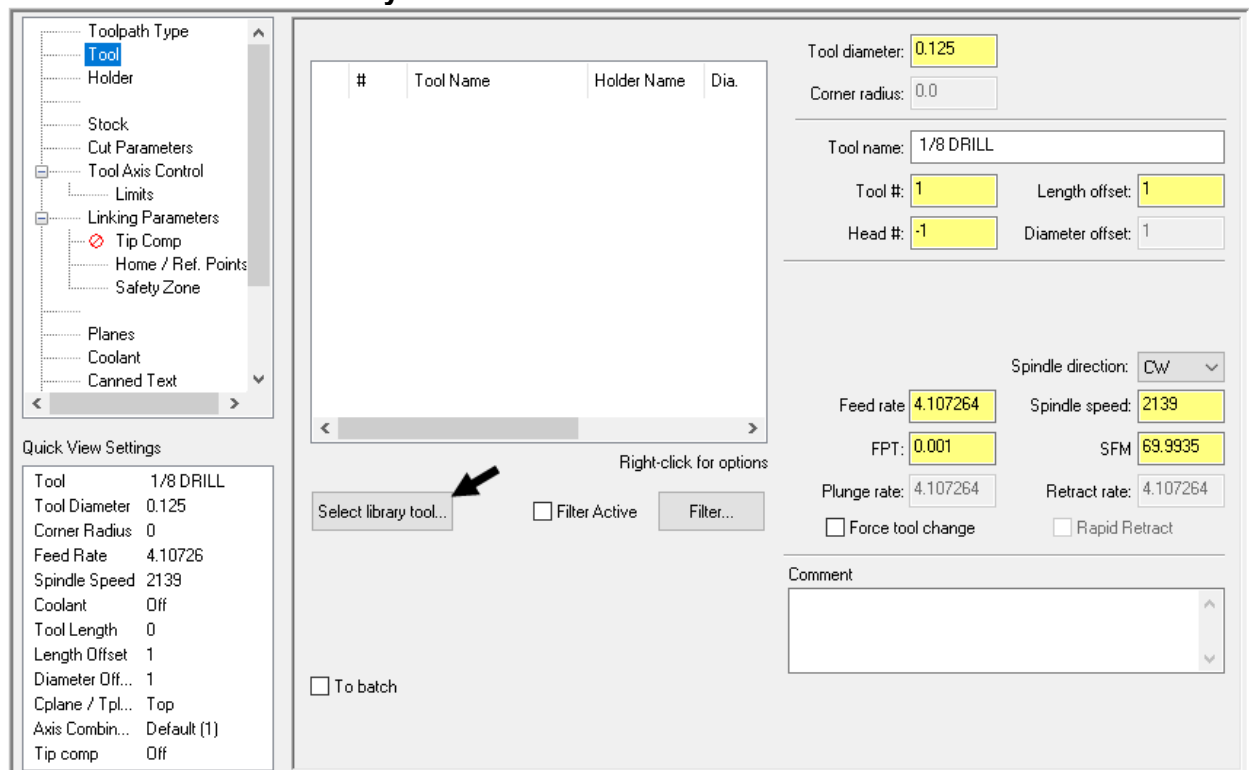
Mastercam Training Guide

➤ After selecting the OK button, you are confronted with the Drill Toolpath Type page. The first task here will be to select a .125" diameter end mill.

14. Ensure the **Toolpath Type** is set to **Drill** as shown below and then select **Tool** from the list on the left.



15. Click on the **Select library tool** button in the lower left corner.



16. Use the slider bar on the right of this dialog box to scroll down and locate a .125" diameter flat end mill.
17. Select the **.125" diameter flat end mill** by picking anywhere along the .125 end mill row, as shown below:

#	Assembly Name	Tool Name	Holder Name	Dia.	Cor. rad.	Length	Type	Rad. Typ
281	--	3/32 FLAT ENDMILL	--	0.09375	0.0	0.375	Endmill1 Flat	None
282	--	1/8 FLAT ENDMILL	--	0.125	0.0	0.375	Endmill1 Flat	None
283	--	5/32 FLAT ENDMILL	--	0.15625	0.0	0.375	Endmill1 Flat	None
284	--	3/16 FLAT ENDMILL	--	0.1875	0.0	0.4375	Endmill1 Flat	None

18. To resize a column in the **Tool Selection dialog box**, click on the divider between the columns with your left mouse button, as shown below, hold the left mouse button down and move to the right or left.

#	Assembly Name	Tool Name	Holder Name	Dia.
281	--	3/32 FLAT ENDMILL	--	0.09375
282	--	1/8 FLAT ENDMILL	--	0.125
283	--	5/32 FLAT ENDMILL	--	0.15625

19. Select the **OK** button to complete the selection of this tool.

Tool Selection dialog box

Use this dialog box to select tools from a tool library and add them to the current machine group. This means that they can be selected and used in a toolpath. Double-click on a tool to select it, use it in the current operation, and close this dialog box.

To see tools from a different library, select the new library from the drop-list. Click on the folder button to see libraries from different folders.

You can also filter the tool list by a number of different criteria to make it easier to find the right tool. Select Filter active to apply a tool filter, or click the Filter button to edit the selection criteria.

Mastercam Training Guide

20. Make changes to the **Tool** parameter page as shown below:

Toolpath Type

- ✓ Tool
- Holder
- Stock
- Cut Parameters
- Tool Axis Control
- Limits
- Linking Parameters
- Tip Comp
- Home / Ref. Points
- Safety Zone
- Planes
- Coolant
- Canned Text

Quick View Settings

Tool	1/8 FLAT EN...
Tool Diameter	0.125
Corner Radius	0
Feed Rate	7
Spindle Speed	5000
Coolant	Off
Tool Length	2.25
Length Offset	1
Diameter Off...	1
Cplane / Tpl...	Top
Axis Combin...	Default (1)
Tip comp	Off

#	Tool Name	Holder Name	Dia.
1	1/8 FLAT ENDMILL	-	0.125

Tool diameter: 0.125
 Corner radius: 0.0
 Tool name: 1/8 FLAT ENDMILL
 Tool #: 1 Length offset: 1
 Head #: 0 Diameter offset: 1

Spindle direction: CW
 Feed rate: 7.0 Spindle speed: 5000
 FPT: 0.0003 SFM: 163.6126
 Plunge rate: 6.160896 Retract rate: 6.160896
 Force tool change Rapid Retract

Comment

To batch

21. Select **Cut Parameters** from the list on the left and make changes to this page as shown below if required. The Cycle should be set to Drill/Counterbore.

Toolpath Type

- ✓ Tool
- Holder
- Stock
- Cut Parameters**
- Tool Axis Control
- Limits
- Linking Parameters
- Tip Comp
- Home / Ref. Points
- Safety Zone
- Planes
- Coolant
- Canned Text

Quick View Settings

Tool	1/2 SPOTDRI...
Tool Diameter	0.5
Corner Radius	0
Feed Rate	7
Spindle Speed	3500
Coolant	Off
Tool Length	3
Length Offset	1
Diameter Off...	1
Cplane / Tpl...	Top
Axis Combin...	Default (1)
Tip comp	Off

Cycle: Drill/Counterbore

First peck: 0.1
 Subsequent peck: 0.1
 Peck clearance: 0.1
 Retract amount: 0.1
 Dwell: 0.0
 Shift: 0.1

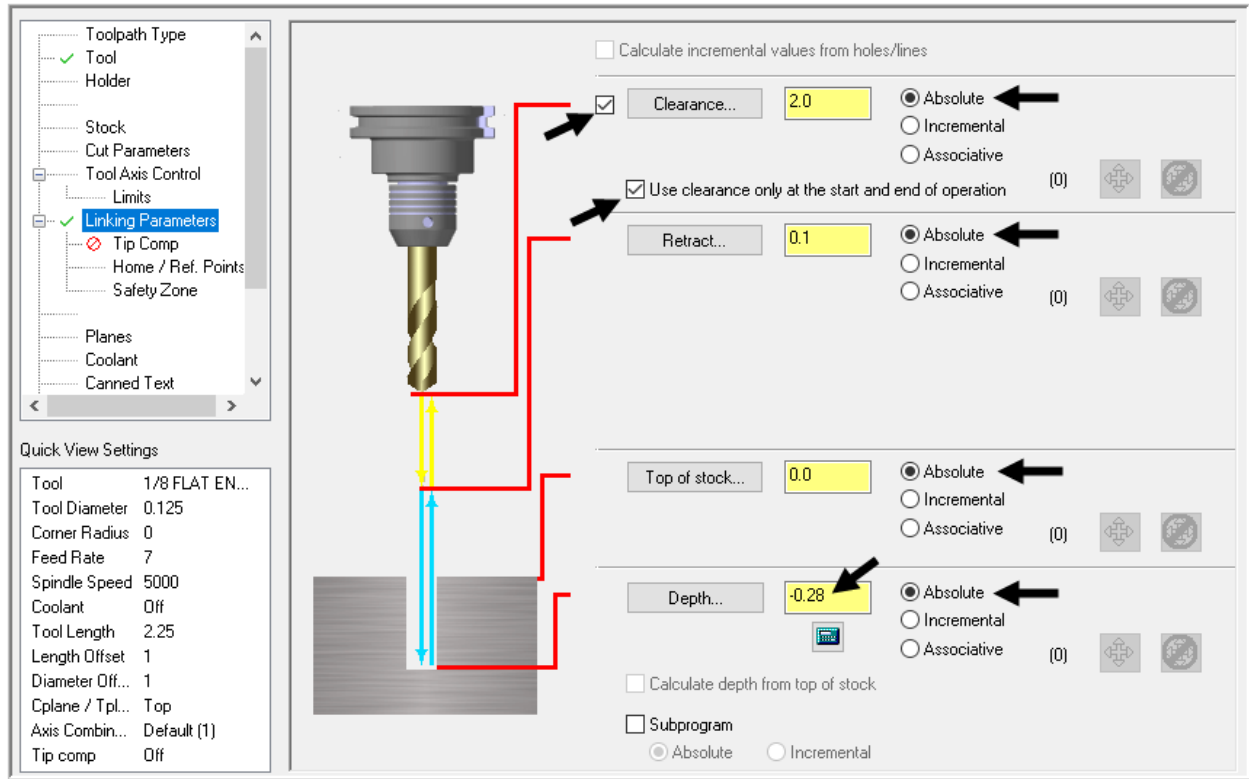
Apply custom drill parameters

1-Drill parameter #1	0.0	1-Drill parameter #6	0.0
1-Drill parameter #2	0.0	1-Drill parameter #7	0.0
1-Drill parameter #3	0.0	1-Drill parameter #8	0.0
1-Drill parameter #4	0.0	1-Drill parameter #9	0.0
1-Drill parameter #5	0.0	1-Drill parameter #10	0.0

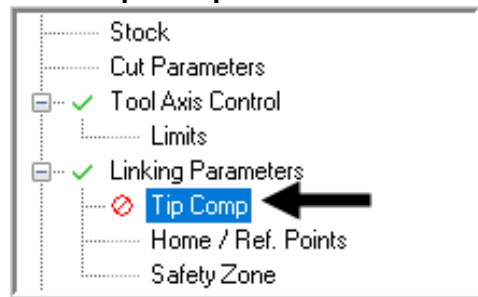
Drill/Counterbore:

Recommended for drilling holes with depths of less than three times the tool diameter

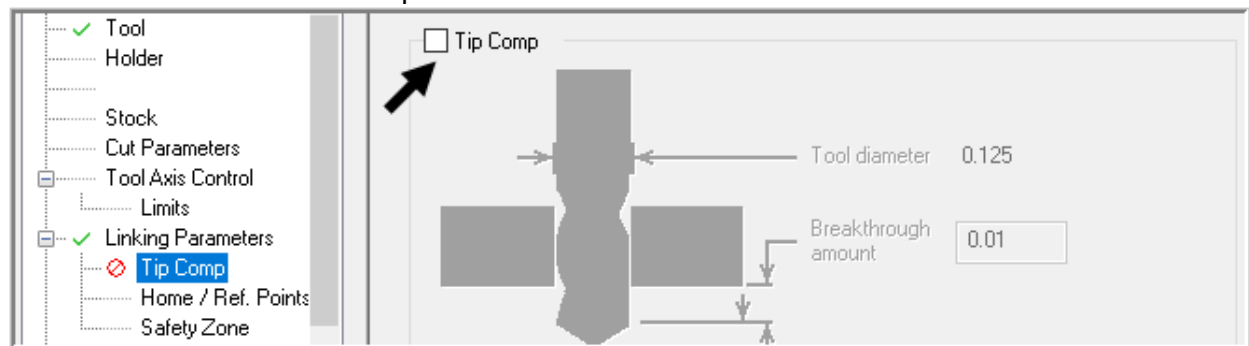
22. Select **Linking Parameters** from the list on the left and make changes to this page as shown below. Input the **depth of -0.28** and the other values as shown below. **Note** all the values are set to **Absolute**.



23. Select the plus sign to the left of **Linking Parameters** to expand the list if required and click on **Tip Comp**.

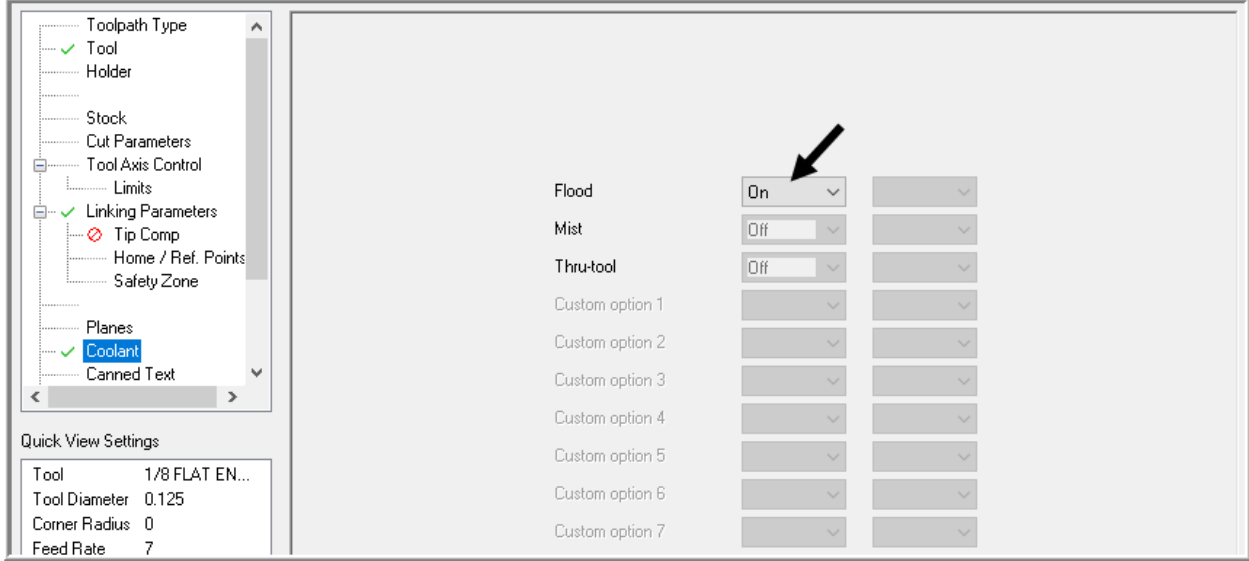



24. Ensure **Tip Comp** is **not activated** as shown below. The **Tip Comp** box is empty. The red circular icon denotes this option is disabled.



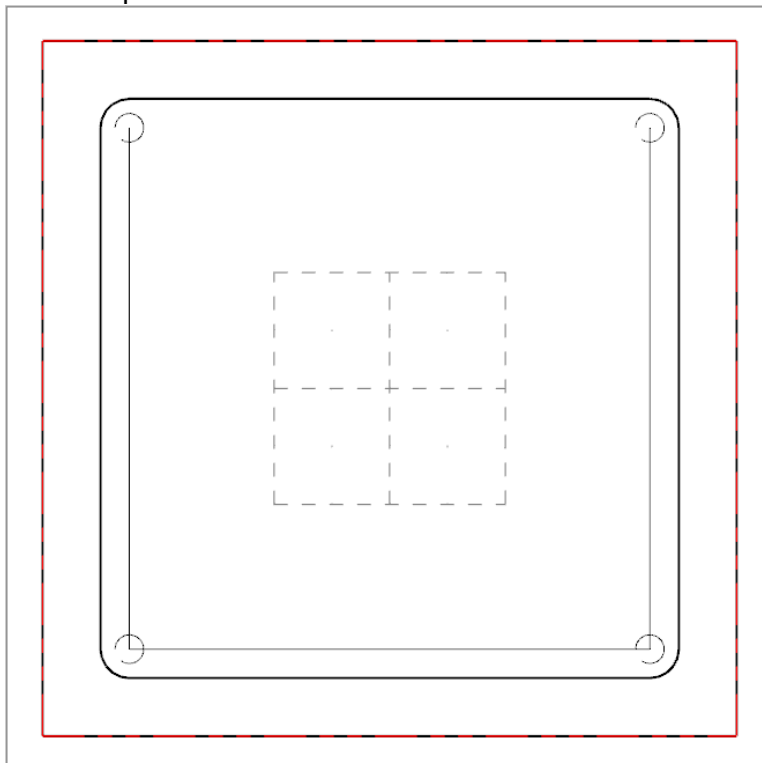
Mastercam Training Guide

25. Select **Coolant** from the list on the left. Open up the drop down menu for **Flood** and set it to **On**.



26. Select the **OK** button  to complete this function.

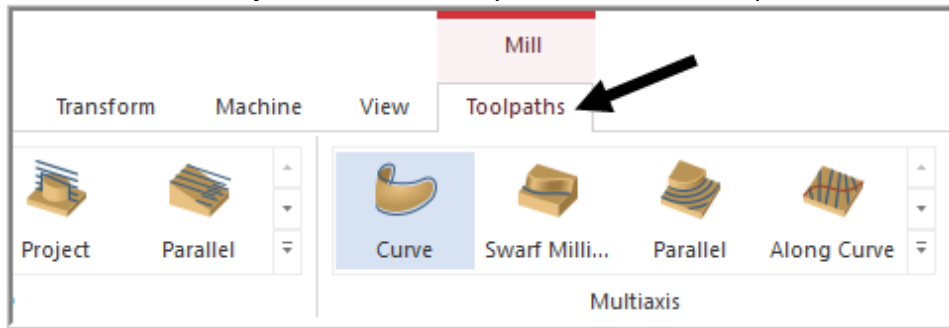
☞ Your part should look like the screenshot below:



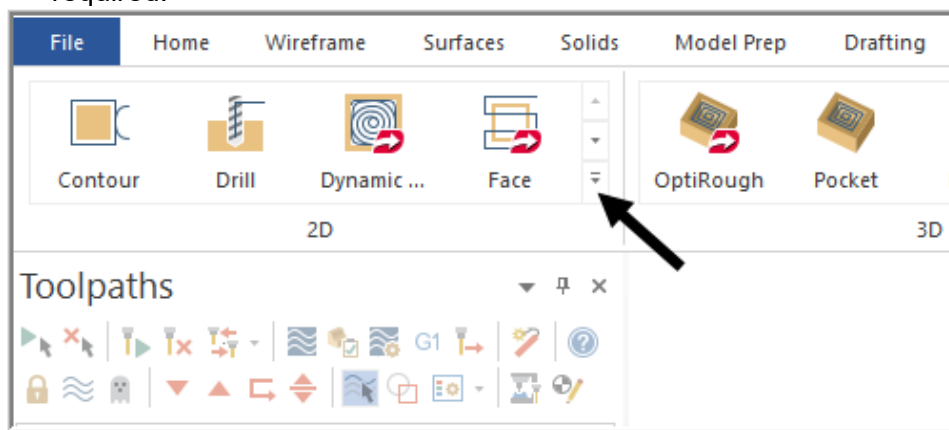
**TASK 8:
MACHINE THE CONTOUR.**

- In this task you will machine the contour with a .5" diameter 2 flute end mill.
- Initially you will machine the contour in one cut at a depth of -.125" and then later in this Lesson **add roughing and finishing cuts** using **Depth of Cuts** and **Multi Passes**.

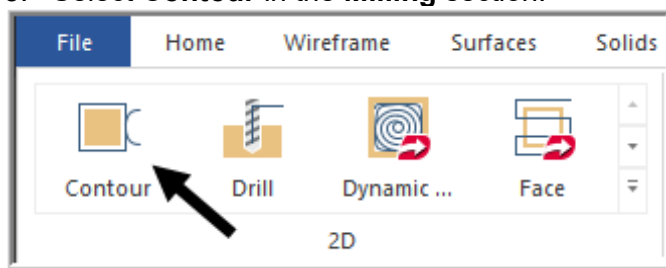
1. Select the **Toolpaths** tab at the top of the screen if required.



2. Click on the **Expand gallery** to view additional options in the **2D** toolpath section, if required.



3. Select **Contour** in the **Milling** section.



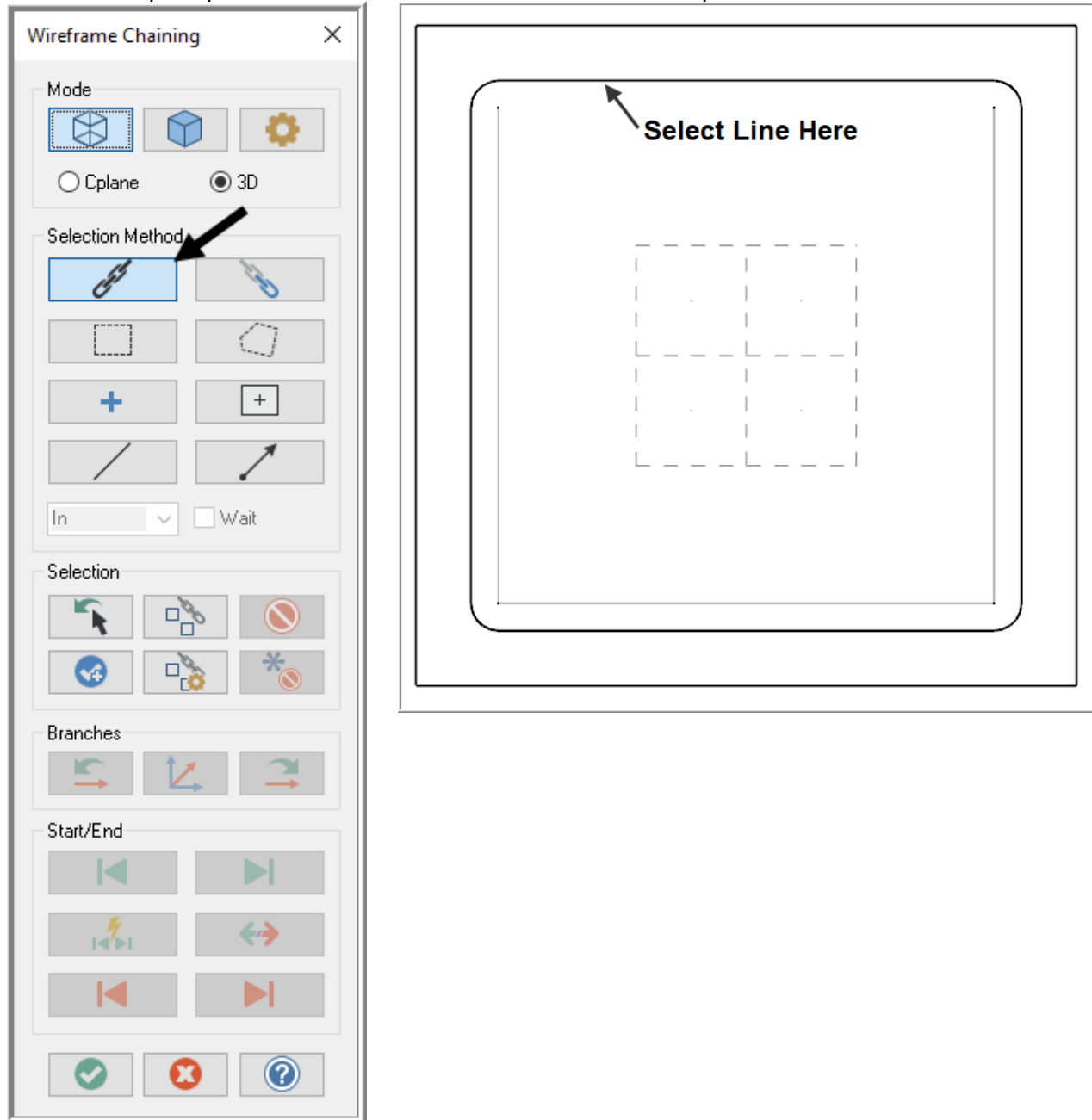
Contour toolpaths remove material along a path defined by a chain of curves.

Contour toolpaths only follow a chain; they do not clean out an enclosed area.

You can select an unlimited number of chains for each toolpath. You can create either 2D or 3D contour toolpaths.

Mastercam Training Guide

4. On the screen you will now see the **Chaining** dialog box with **Chain** set and in the graphics screen a prompt to “**Select Contour chain**”. Select the top horizontal line as shown below:



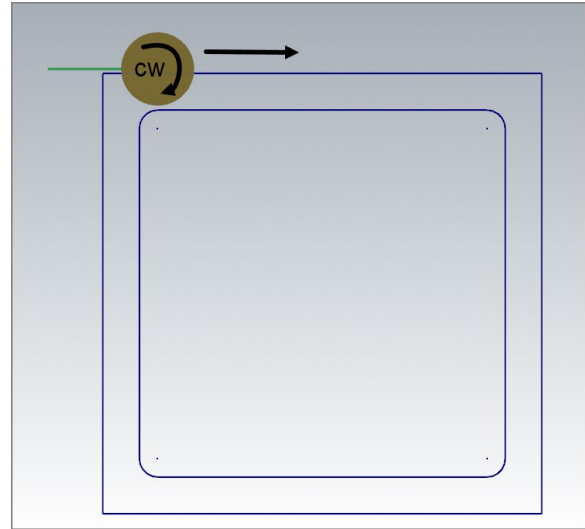
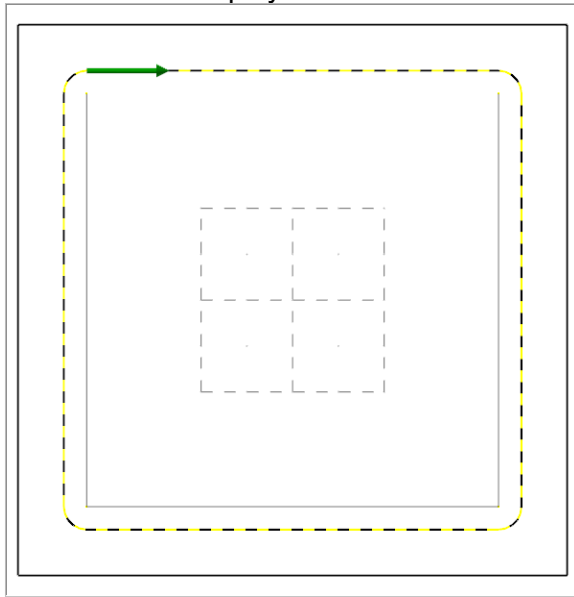
Chaining dialog box

Use this dialog box to select chains of entities for generating surfaces, solids, or toolpaths, or for performing certain Analyze, Xform, or other operations.

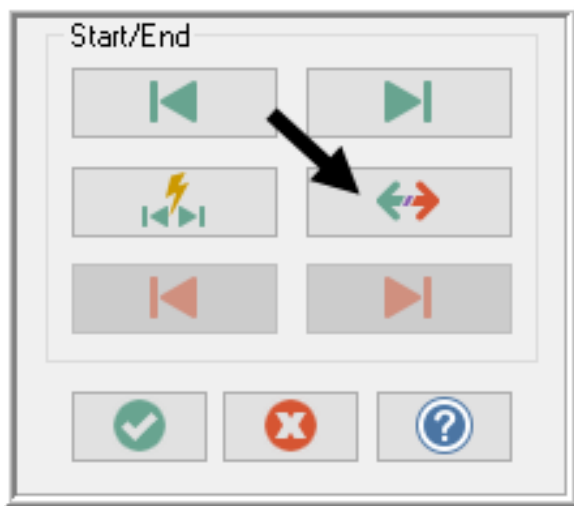
A chain of entities consists of one or more entities linked together in order and direction.

The chaining dialog box operates in two modes: Wireframe (default) and Solids. Two buttons at the top let you switch between modes; however, if either solid or wireframe geometry is not present, its button is unavailable.

- After selecting the top horizontal line your graphics screen should look like the screenshot below, with the green arrow pointing to the **right, clockwise direction**.
- The material for this part is aluminum so to attain a good finish when contouring climb milling should be employed.



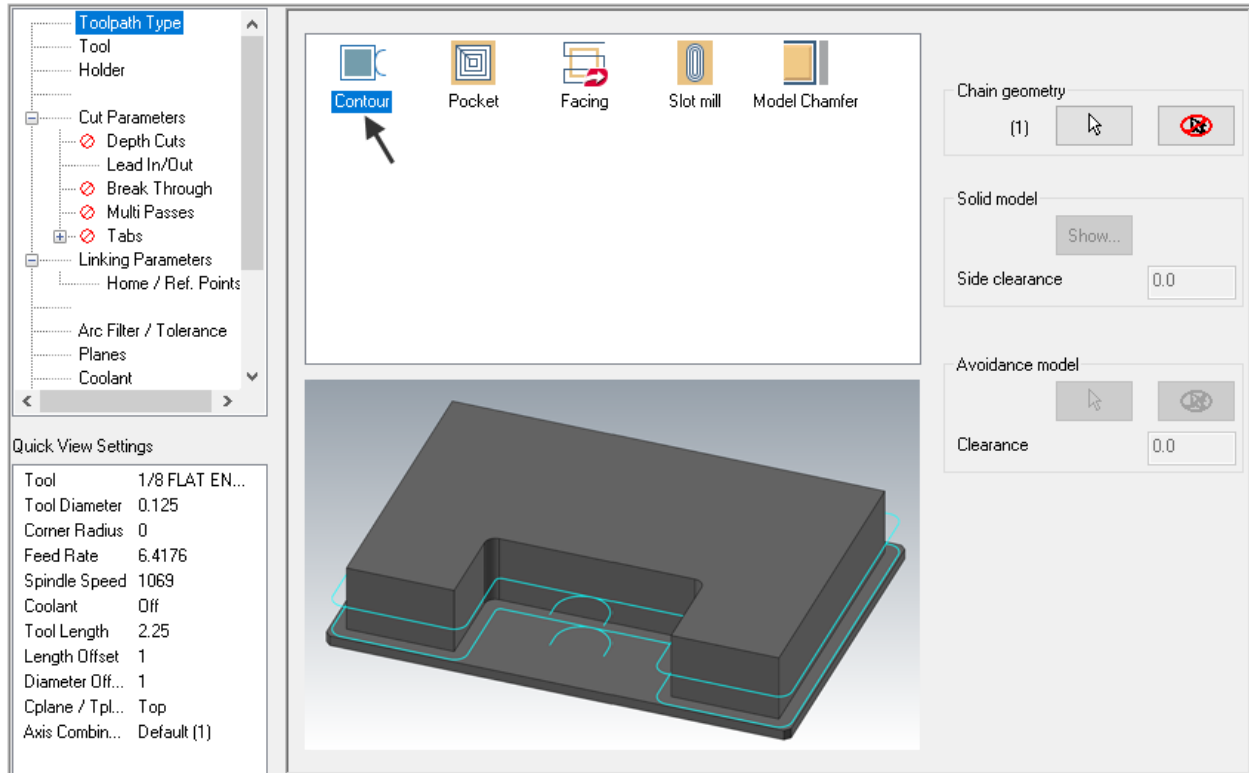
5. If the arrow is not pointing to the right select the arrow from the **Chaining** dialog box shown below to reverse the direction:



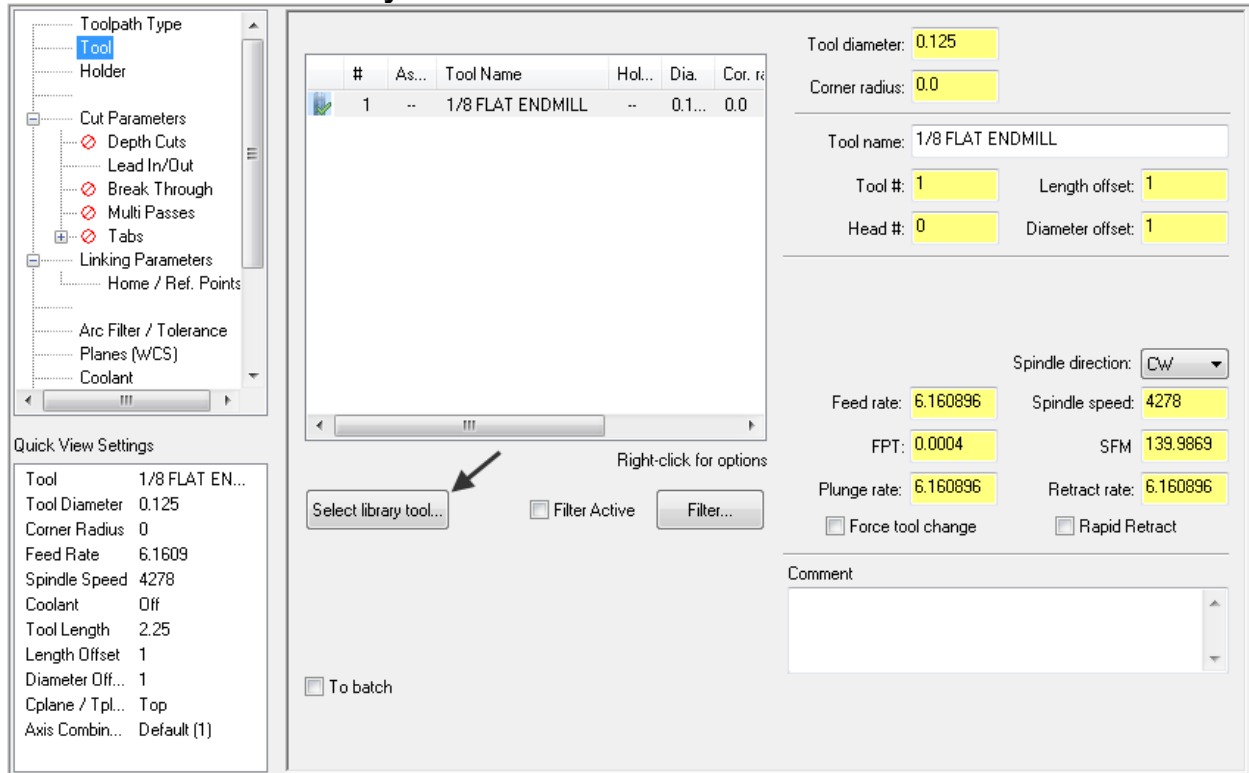
6. After the contour has been successfully chained select the **OK** button at the bottom of the Chaining dialog box.

Mastercam Training Guide

7. Ensure the **Toolpath Type** is set to **Contour** as shown below and then select **Tool** from the list on the left.




8. Click on the **Select library tool** button in the lower left corner.

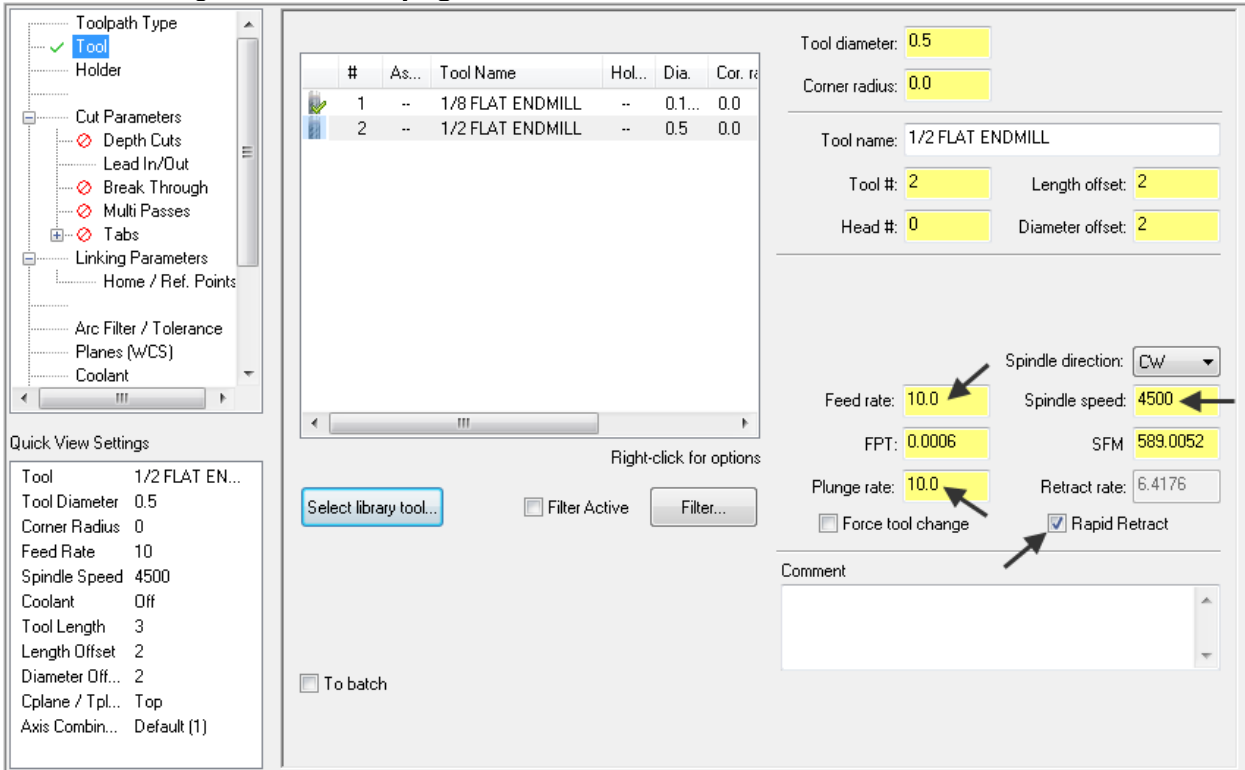


9. Use the slider bar on the right of this dialog box to scroll down and locate a **.5" diameter flat end mill**. Select the .5" diameter flat end mill by picking anywhere along the .5 end mill row, as shown below:

#	Assembly Name	Tool Name	Holder Name	Dia.	Cor. rad.	Length	# Flutes	Type
288	--	13/32 FLAT ENDMILL		0.40625	0.0	0.8	4	Endmill1 Flat
289	--	7/16 FLAT ENDMILL		0.4375	0.0	0.8	4	Endmill1 Flat
290	--	1/2 FLAT ENDMILL	--	0.5	0.0	1.0	4	Endmill1 Flat
291	--	17/32 FLAT ENDMILL	--	0.5312	0.0	1.0	4	Endmill1 Flat
292	--	5/8 FLAT ENDMILL	--	0.625	0.0	1.5	4	Endmill1 Flat

10. Select the **OK** button  to complete the selection of this tool.

11. Make changes to the **Tool page** as shown below:



The screenshot shows the Mastercam Tool dialog box with the following settings:

- Toolpath Type:** Tool
- Tool List:**

#	As...	Tool Name	Hol...	Dia.	Cor. r...
1	--	1/8 FLAT ENDMILL	--	0.1...	0.0
2	--	1/2 FLAT ENDMILL	--	0.5	0.0
- Tool diameter:** 0.5
- Corner radius:** 0.0
- Tool name:** 1/2 FLAT ENDMILL
- Tool #:** 2
- Length offset:** 2
- Head #:** 0
- Diameter offset:** 2
- Spindle direction:** CW
- Feed rate:** 10.0
- Spindle speed:** 4500
- FPT:** 0.0006
- SFM:** 589.0052
- Plunge rate:** 10.0
- Retract rate:** 6.4176
- Force tool change:**
- Rapid Retract:**
- Comment:** (Empty)

Rapid retract

Select this option to retract from the part at the rapid rate set at the machine tool. If you do not select this option, Mastercam will use the programmed Retract rate set in this dialog box.

SFM (inch), CS (metric)

Displays the cutting speed in surface feet per minute or surface meters per minute.

FPT

Displays the amount of **feed per tooth**, based on the number of flutes in the tool definition.

12. Select **Cut Parameters** from the list on the left and make changes to this page if required.

Quick View Settings

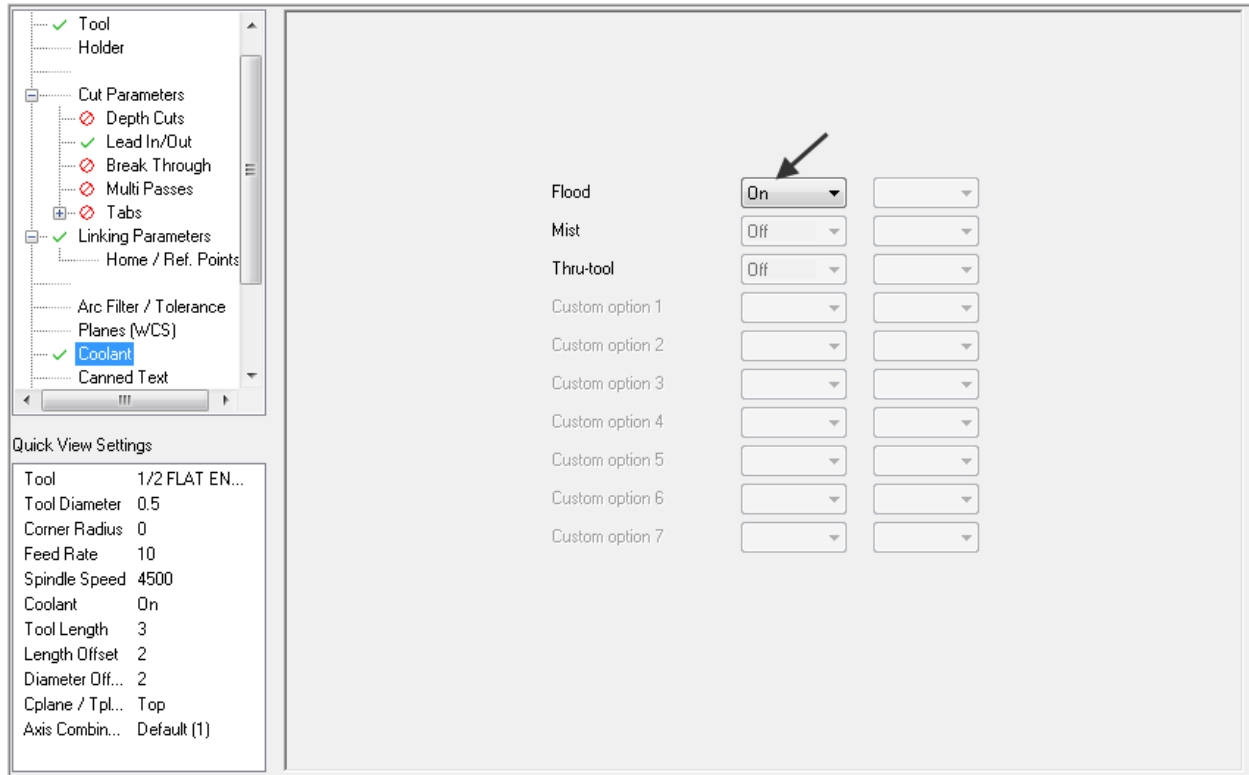
Tool	1/2 FLAT EN...
Tool Diameter	0.5
Corner Radius	0
Feed Rate	10
Spindle Speed	4500
Coolant	Off
Tool Length	3
Length Offset	2
Diameter Off...	2
Cplane / Tpl...	Top
Axis Combin...	Default (1)

13. Select **Linking Parameters**. Input the depth of **-0.125** and the other values as shown below. **Note** all the values are set to **Absolute**.

Quick View Settings

Tool	1/2 FLAT EN...
Tool Diameter	0.5
Corner Radius	0
Feed Rate	10
Spindle Speed	4500
Coolant	Off
Tool Length	3
Length Offset	2
Diameter Off...	2
Cplane / Tpl...	Top
Axis Combin...	Default (1)


14. Select **Coolant** from the list on the left. Open up the drop down menu for **Flood** and set it to **On**.

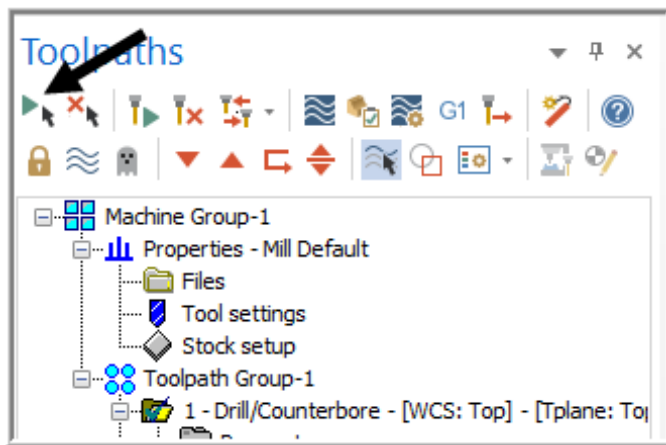


15. Select the **OK** button  to complete this function

TASK 9: BACKPLOT THE TOOLPATHS

- In this task you will use Mastercam's Backplot function to view the path the tools take to cut this part.
- Backplot will enable you to review the cutting motions and identify any problem areas when cutting the part.
- When the toolpaths are being Backplotted Mastercam displays tool path information on the right of the screen. Information such as the current tool position in X, Y, and Z coordinates.
- **For more information on Backplot see the Tips and Techniques section in the Online Course Instruction.**

1. To pick all the operations to Backplot pick the **Select all operations** icon  circled below:



- Another method to **Select all** the operations is by clicking on the **Toolpath Group-1** in the Toolpaths Manager as shown by the arrow above.

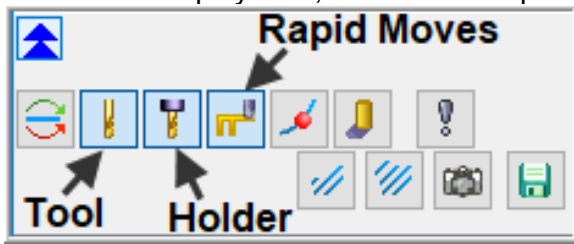
2. The next step is to select the **Backplot selected operations** icon shown below:



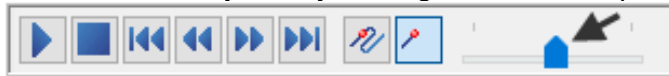
Please Note:

Selecting this icon launches "Classic Backplot" if enabled in the Advanced Configuration as shown in the Setting the Environment document and video

3. Ensure Display Tool, Holder and Rapid moves are activated.



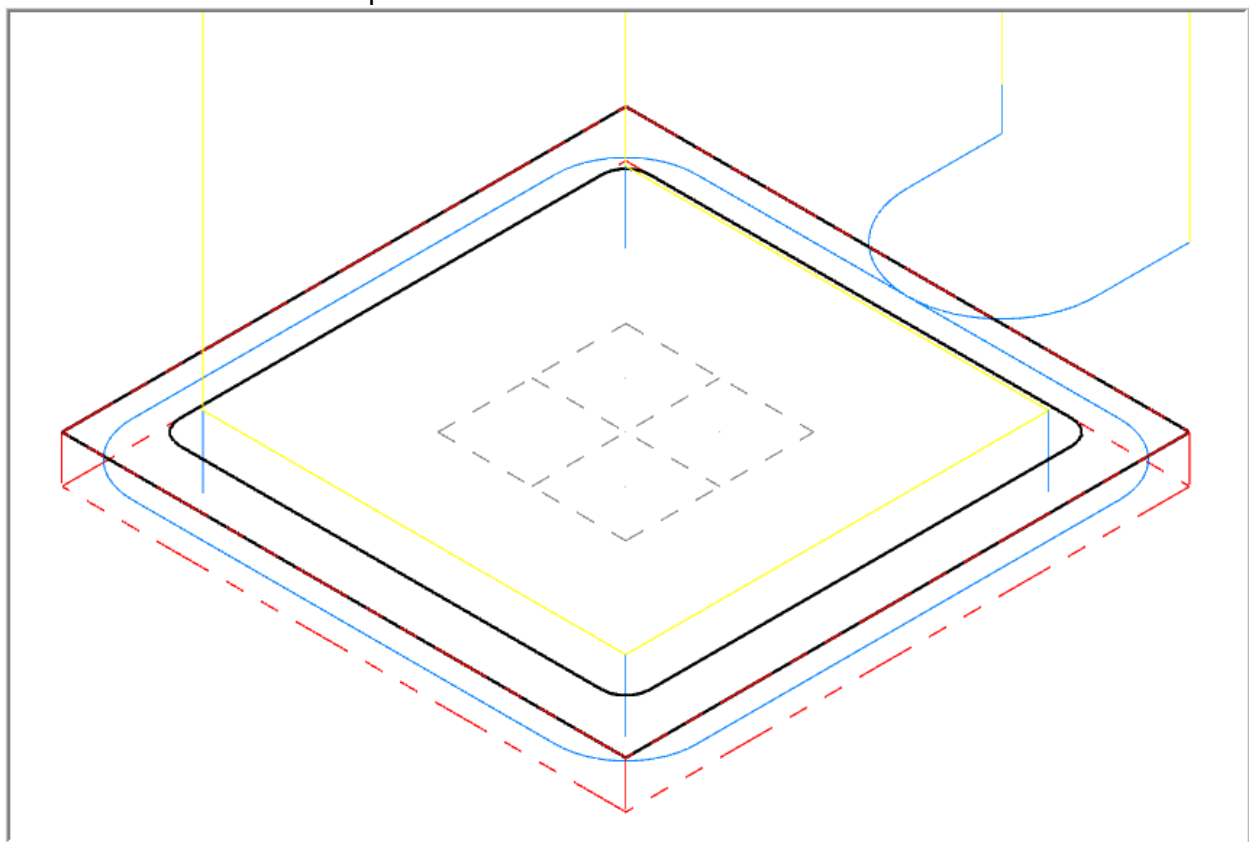
4. Set the run **Speed** by moving the slider bar pointer to the midpoint as shown below.



5. Now select the **Play** button to review the toolpaths.



6. After reviewing the Backplot of the two toolpaths click the Close button  in the top right hand corner to exit Backplot.

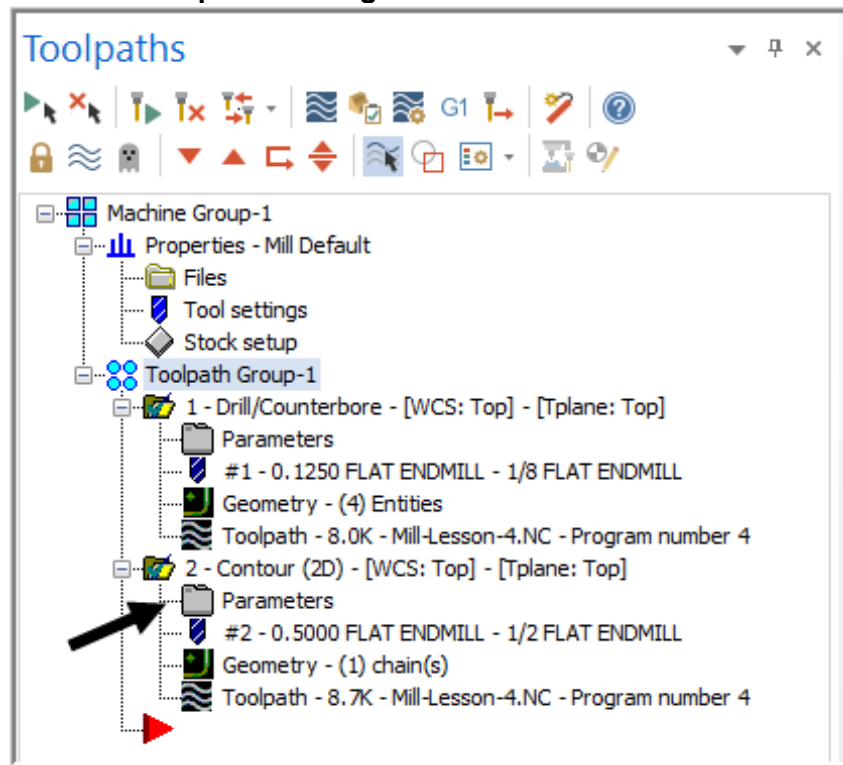


TASK 10:

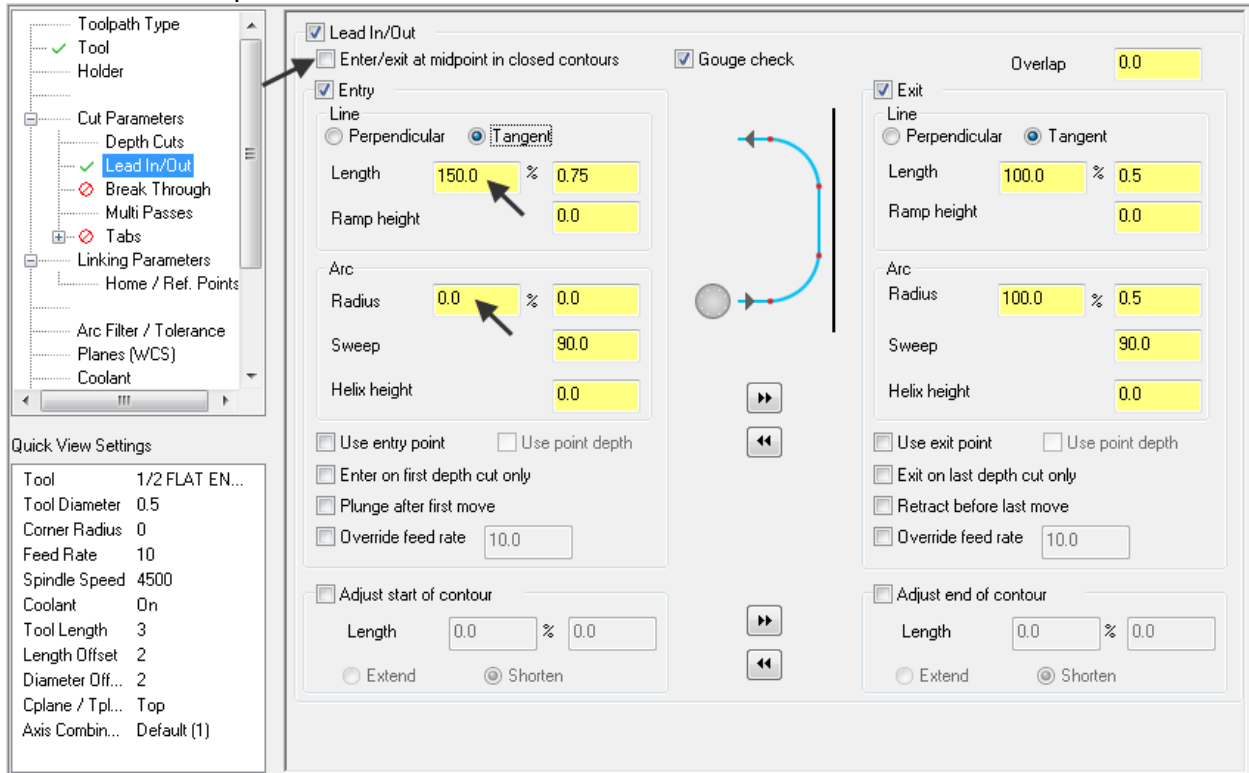
MODIFY THE CONTOUR TOOLPATH TO ADD LEAD IN/OUT, ROUGHING CUTS AND A FINISH PASS

- In this task you will use Mastercam's **Multi Passes** and **Depth of cuts** to perform a roughing and finishing operation for the contour toolpath.
- **Multi Passes** will let the tool approach the part geometry at the cutting depth in steps instead of cutting right to the part geometry.
- **Depth of cuts** can be used to set the number of depth cuts, you can enter a maximum rough step and Mastercam divides the total depth into equal steps. Or you can enter the exact number of finish steps and the size of each finish step. Mastercam never creates unequal rough depth cuts.
- For more information on Multi Passes and Depth of cuts see the **Tips and Techniques** section in the Online Course Instruction.

1. In the **Toolpaths Manager** select **Parameters** from the contour toolpath as shown below:



2. Select **Lead In/Out** from the list on the left and make changes to this page. Click the **Green check** to complete the edit.



About Lead In/Out Settings

Gouge check entry/exit motion

Ensures that the entry/exit moves do not gouge the part. If the entry/exit moves cause a gouge, they are removed from the toolpath.

Entry

Mastercam lets you create a lead in move, a lead out move, or both. Select Entry to create an entry move and turn on the Entry options.

Exit

Mastercam lets you create a lead in move, a lead out move, or both. Select Exit to create an exit move and turn on the Exit options.

Tangent


Places entry or exit lines tangent to the cut direction.

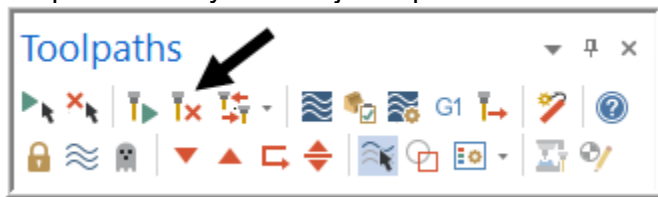
Line length

Sets the line length for entry or exit lines. Enter a percentage of the tool diameter or an exact length. Mastercam automatically updates the other field when you enter a value in one of them. Set to 0 if you just want an entry arc and do not want an entry line.

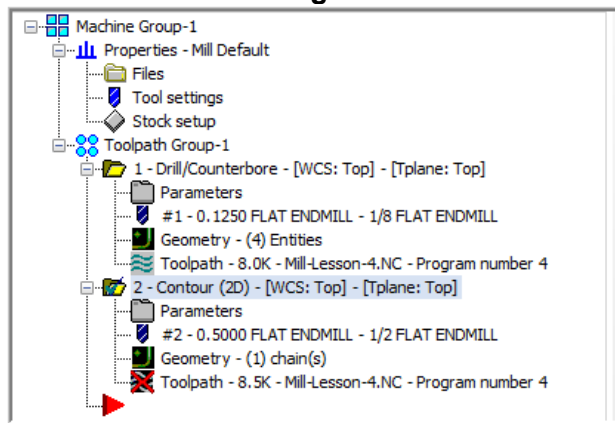
Arc radius

Defines the entry or exit arc's radius. The arc is always tangent to the toolpath. Enter a percentage of the tool diameter or an exact length. Set to 0 if you just want an entry line and do not want an entry arc.

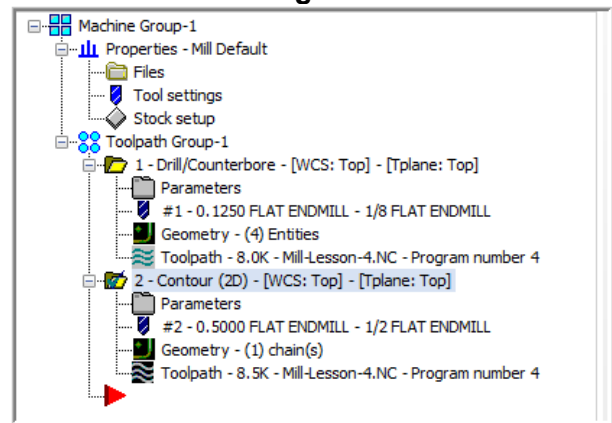
3. Select the **Regenerate all dirty operations** icon  to remove the red X from the contouring operation you have just edited. You need to update the toolpath with the new parameters you have just input.



Before Regeneration



After Regeneration



Dirty toolpath

This happens if you have changed certain parameters of the underlying geometry, or in this example you have updated a setting within the toolpath's parameters.

Toolpaths can be regenerated by clicking the Regenerate button at the top of the Toolpaths Manager shown above. There are two options for regenerating dirty operations...

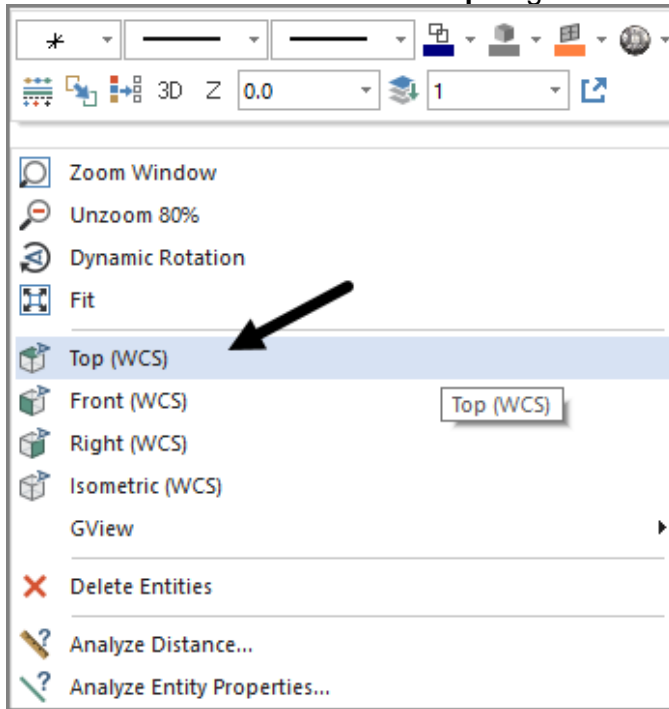


Regenerate all selected operations - Recalculates all selected operations even if they are not marked dirty.

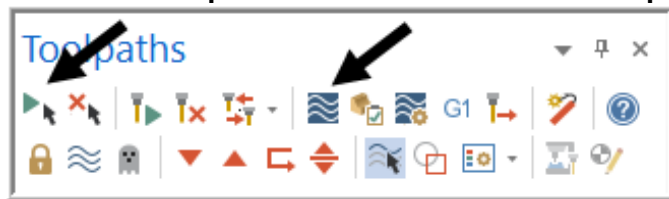


Regenerate all dirty operations - Recalculates all dirty operations even if they are not selected.

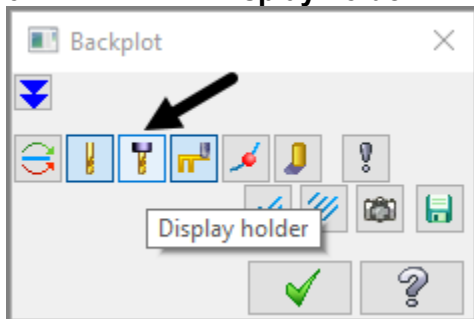
4. Make sure the View is set to **Top**. Right click on the mouse to change to Top if necessary.



5. **Select all operations** and then click on **Backplot selected operations**.

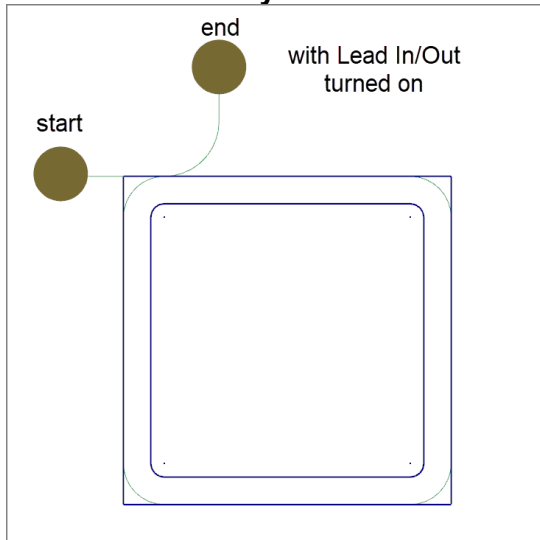


6. Turn off the **Display holder** icon. This will make the tool easier to see from the Top view.



Mastercam Training Guide

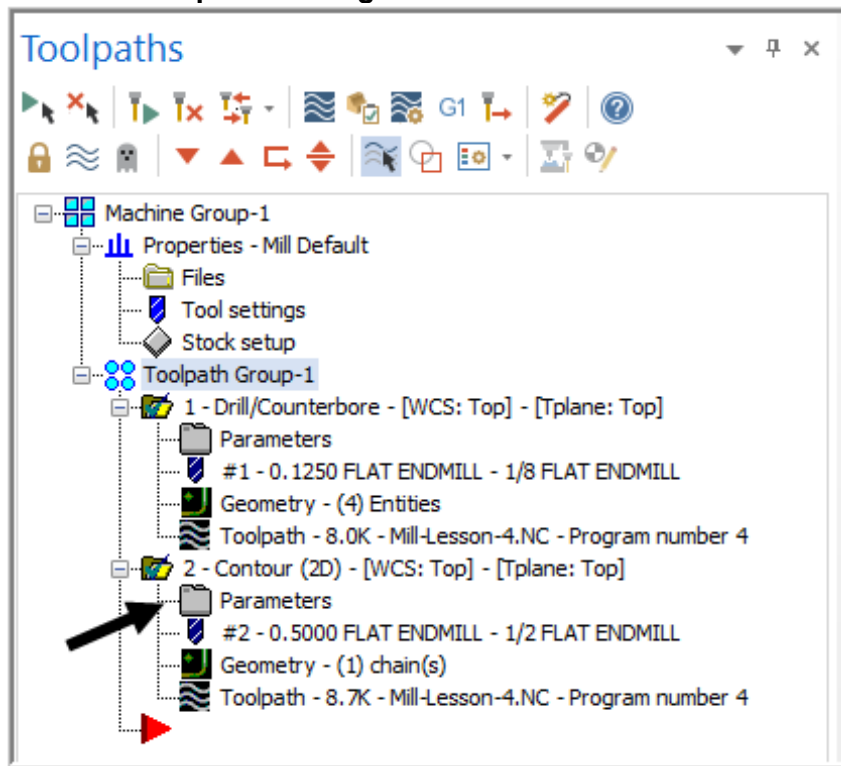
7. Click on the **Play** icon to start the backplot. The revised toolpath should match below.



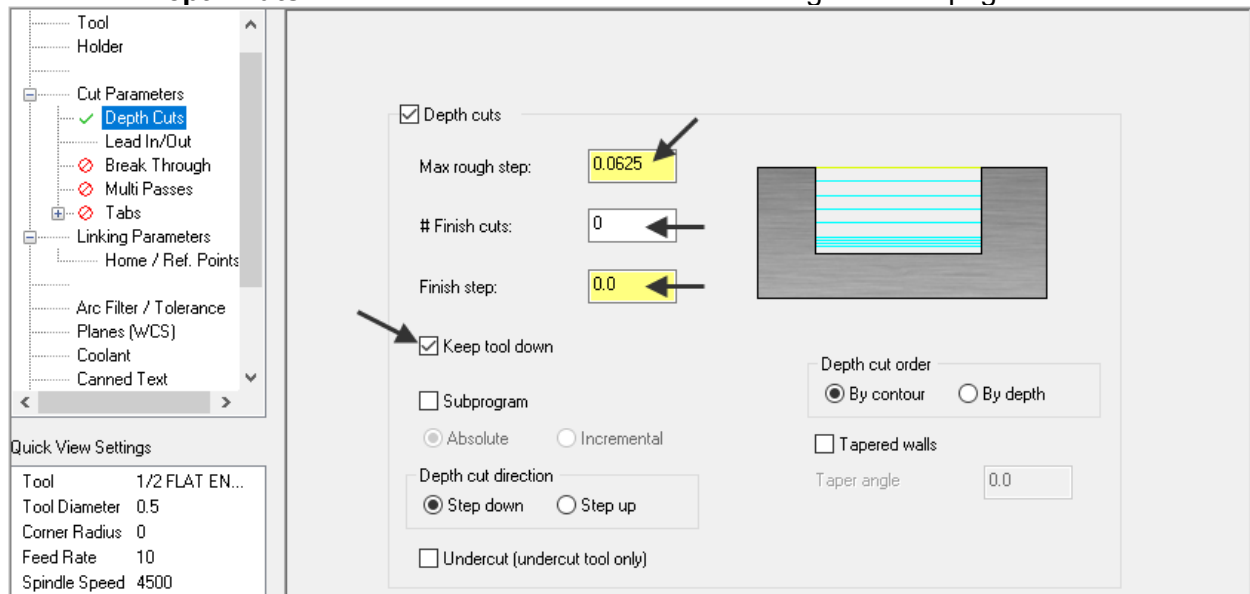
8. Click on OK to close **Backplot**.

➤ Change the **Depth cut** settings and the **Multi-Passes** settings.

1. In the **Toolpaths Manager** select **Parameters** from the contour toolpath as shown below:



2. Select **Depth Cuts** from the list on the left and make changes to this page as shown below:



Question: How many depth cuts will be created with the above settings made?

Answer: Check by backplotting or verifying the Toolpath.

About the Depth cuts dialog box

Max rough step:

Sets the maximum amount of material removed in the Z axis with each rough cut.

Finish cuts:

Sets the number of finish cuts for the contour toolpath. This number multiplied by the finish step value equals the total amount of stock cut by the finish passes. Setting the number of finish cuts to 0 creates no finish cuts.

Finish step:

Sets the amount of material removed in the Z axis with each finish cut. This number multiplied by the number of finish passes equals the total amount of stock cut by the finish passes.

Keep Tool Down:

Determines if the tool should retract between depth cuts.

Depth cut order:

By pocket/contour

Performs all depth cuts in a contour or region before moving to the next contour or region.

By depth

Creates depth cuts at the same level in every contour or region and then descends to the next depth cut level.

In this example you will perform:

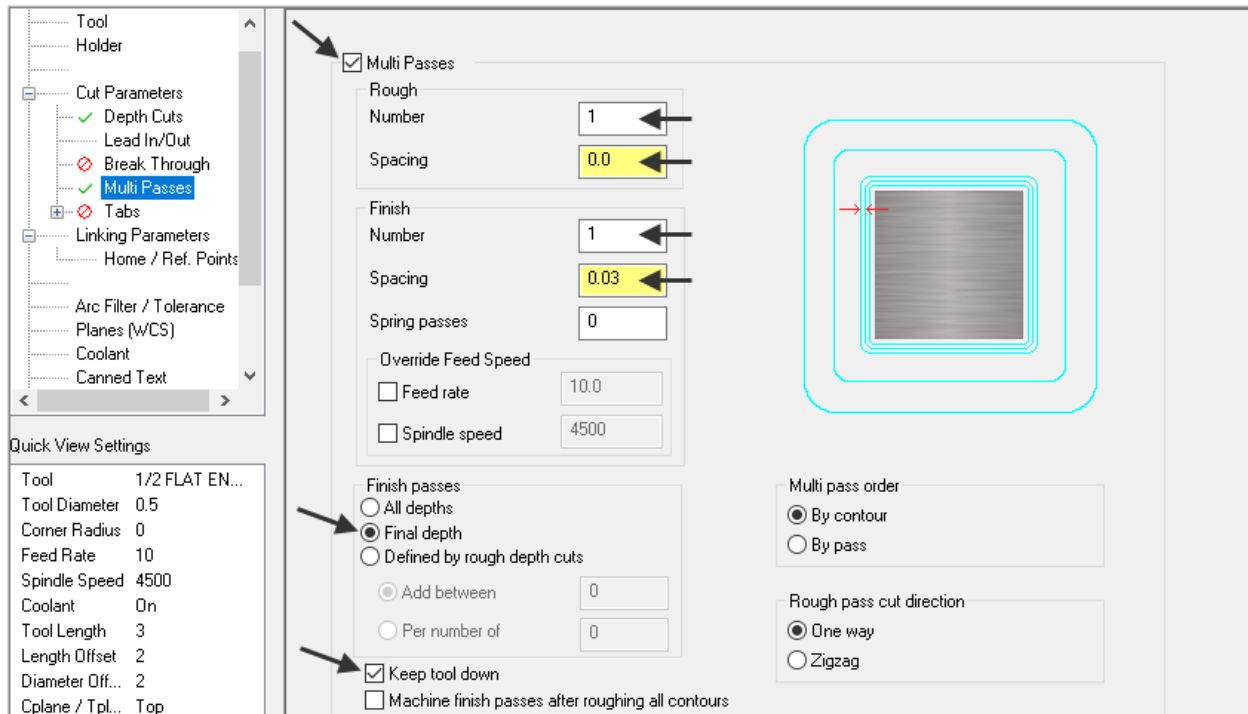
Each depth of cut will not exceed .0625", therefore as your final depth is -.125" you will perform only two rough cuts. The first at -0.0625 and the second rough cut at -.125

Only one finish pass at the final depth

The finish pass will only take place at the final depth, this final cut will machine the .030" from the contour that you set up using Multi Passes

In between passes the tool will be kept down.

3. Select **Multi Passes** from the list on the left and make changes to this page as shown below:



About the Multi Passes dialog box

Roughing passes:

Number: Enter the number of cutting passes you want Mastercam to create.

Spacing: Enter the amount of stock to remove with each cut.

Finishing passes:

Number: Enter the number of cutting passes you want Mastercam to create.

Spacing: Enter the amount of stock to remove with each cut.

Machine finish Passes at:

Final Depth: Performs a single finish pass at the final depth.

Keep Tool Down: Determines whatever the tool should retract between multi passes.

In this example you will perform:

No roughing cuts in the XY plane.


Only one finish pass at the final depth.

While cutting at the various depths you will stay .030" away from the contour.

The finish pass will take place at the final depth.


In between passes the tool will be kept down.

4. After reviewing the values input in the **Multi Passes** dialog box select the OK button  to exit.

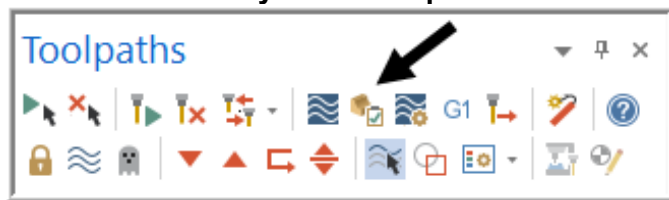
5. Select the **Regenerate all dirty operations** icon  to remove the red X from the contouring operation you have just edited. You need to update the toolpath with the new parameters you have just input.

TASK 11: VERIFY THE TOOLPATH

- Mastercam's Verify utility allows you to use solid models to simulate the machining of a part. The model created by the verification represents the surface finish, and shows collisions, if any exist.
- This allows you to identify and correct program errors before they reach the shop floor.
- Backplot and Verify are very similar. The differences between these two functions are that Backplot offers basic simulation options. Whereas Verify offers material removal, collision checking and precision control.
- **For more information on Verify see the Tips and Techniques section in the Online Course Instruction.**

1. In the **Toolpaths Manager** pick all the operations to verify by picking the **Select all operations** icon .

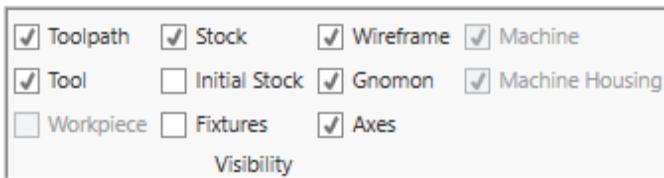
2. Select the **Verify selected operations** icon shown below:



3. **Maximize** the Backplot/Verify window if required.
4. Activate the **Both** option in the Toolpath section of the **Verify** tab.
5. Now select the **Home** Tab.

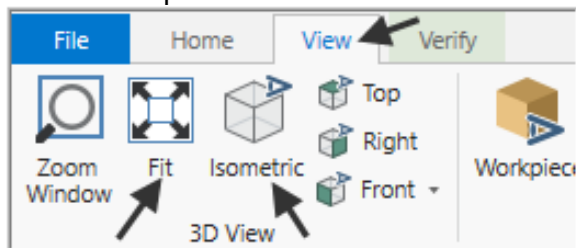


6. Activate the options shown below in the **Visibility** section of the **Home** tab. **Initial Stock** not activated.

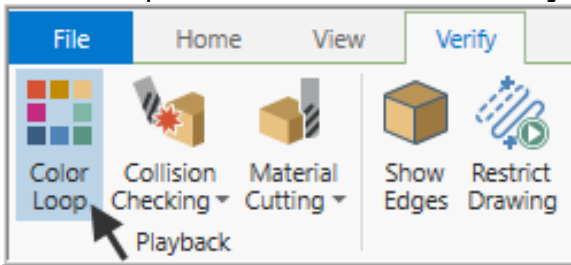


Visibility
Click on the individual functions to cycle through the **three states**:
On, Translucent, or Off.

7. At the top of the screen select the **View** tab, the **Isometric** icon and then select **Fit**.

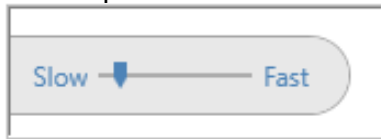


8. At the top of the screen select the **Verify** tab and activate the **Color Loop**.




Color Loop
Changes the color of the toolpath or cut stock by operation or by tool change.
Choose **File, Options** to set the colors.

9. In the lower right corner of the screen now set the run **Speed** to slow by moving the slider bar pointer over to the left as shown below.

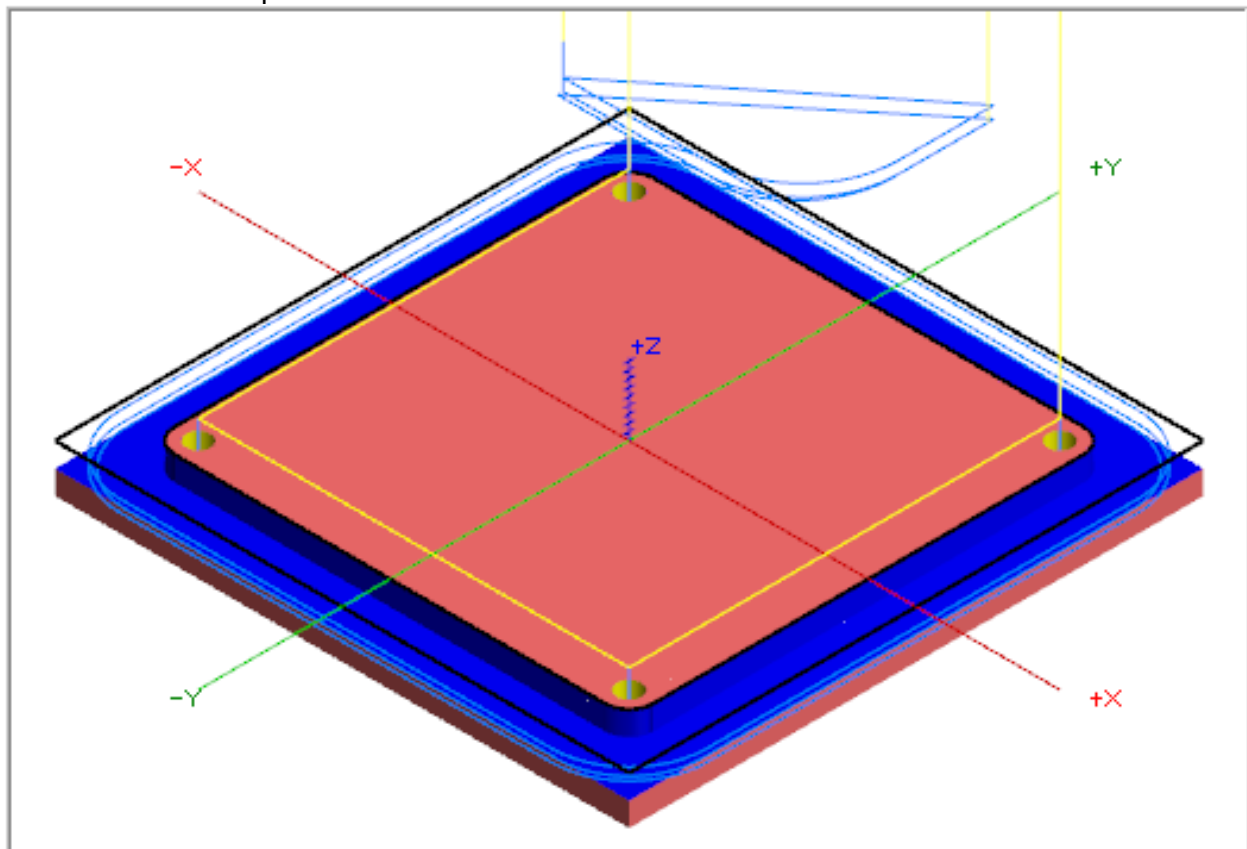


10. Now select the **Play Simulation** button to review the toolpaths.



11. Select the **Close** button  in the top right hand corner to exit Verify.

- ☞ The verified toolpaths are shown below:



TASK 12: SAVE THE UPDATED MASTERCAM FILE


1. Select the **Save** icon from the **Quick Access** toolbar at the top left of the screen.

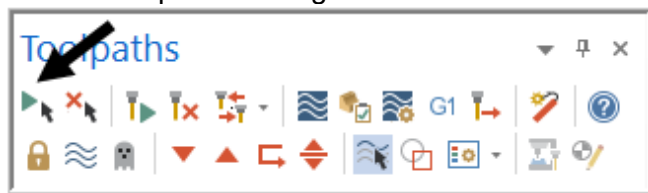


TASK 13: POST AND CREATE THE CNC CODE FILE

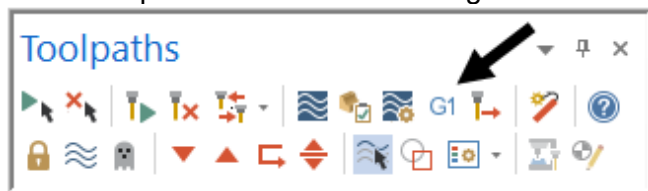
Please Note:

Users of the Mastercam **Home Learning Edition** (HLE) will not be able to Post and Create the CNC code file.

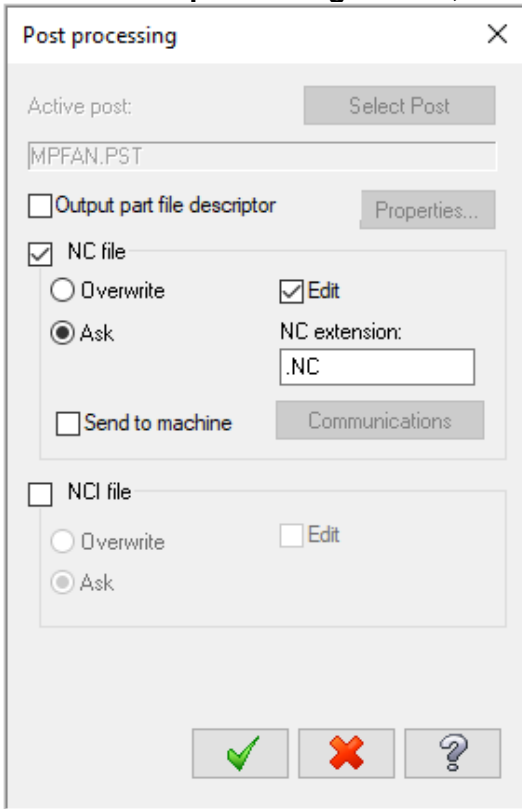
1. Ensure all the operations are selected by picking the **Select all operations** icon  from the Toolpaths manager.




2. Select the **Post selected operations** icon **G1** from the **Toolpaths manager**.
➤ **Please Note:** If you cannot see G1 click on the right pane of the Toolpaths manager window and expand the window to the right.

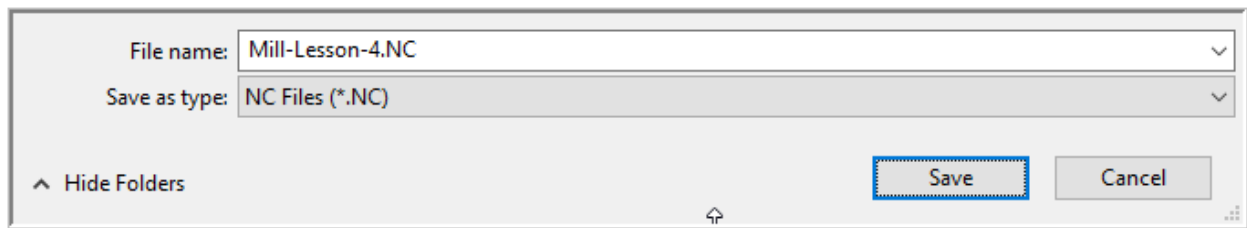


3. In the **Post processing** window, make the necessary changes as shown below:



About NC Processing
NC file: Select this option to save the NC file. The file name and extension are stored in the machine group properties for the selected operation.
 If you are posting operations from different machine groups or Mastercam files, or batch processing, Mastercam will create several files according to the settings for each machine group.
Edit: When checked, automatically launches the default text editor with the file displayed so that you can review or modify it.


4. Select the OK button  to continue.
5. Ensure the same name as your Mastercam part file name is displayed in the **NC File name** field as shown below:



6. Select the **Save** button.

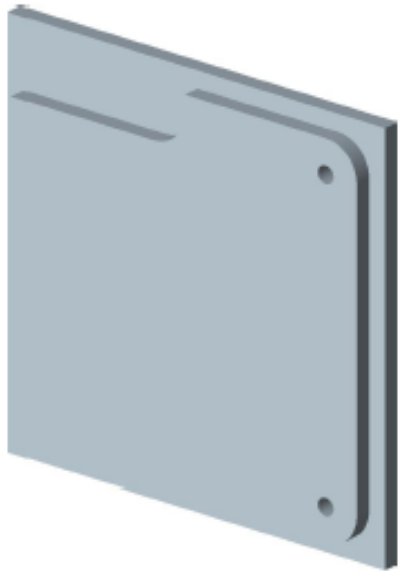
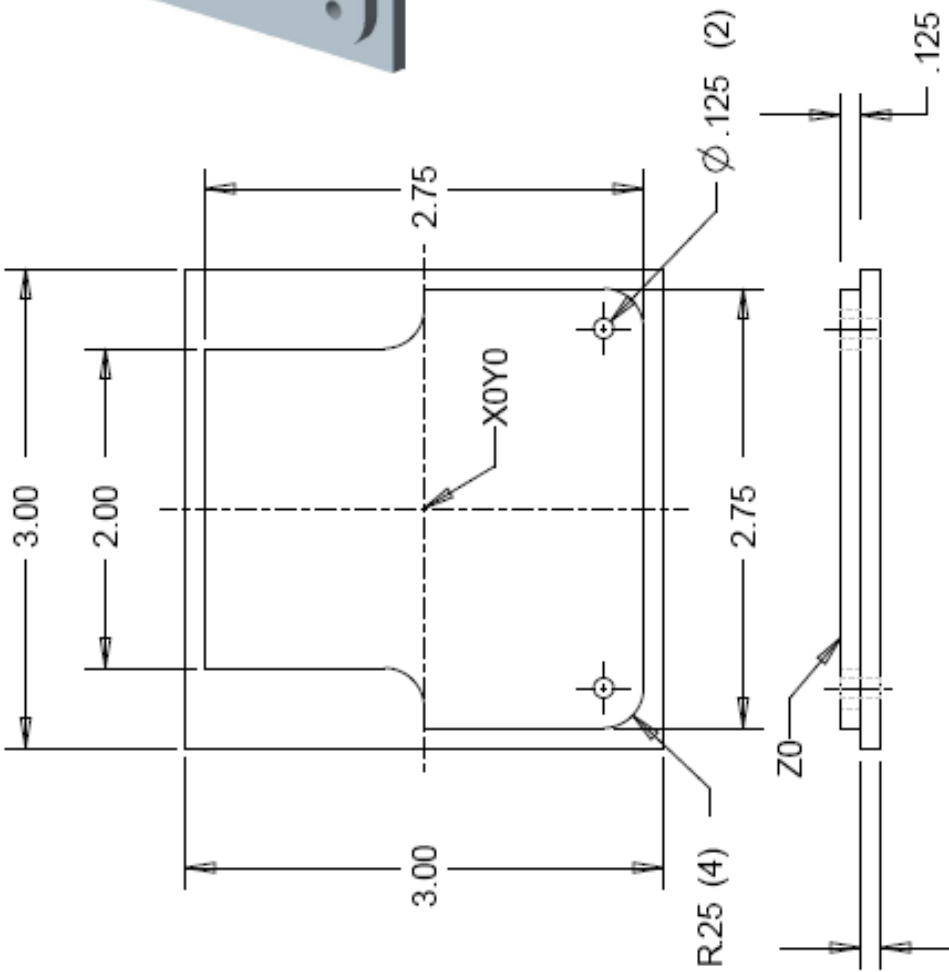
7. The CNC code file opens up in the default editor.

```
MILL-LESSON-4.NC ×
6 (MATERIAL - ALUMINUM INCH - 6061)
7 ( T1 | 1/8 FLAT ENDMILL | H1 )
8 ( T2 | 1/2 FLAT ENDMILL | H2 )
9 N1 G20
10 N2 G0 G17 G40 G49 G80 G90
11 / N3 G91 G28 Z0.
12 / N4 G28 X0. Y0.
13 / N5 G92 X10. Y10. Z10.
14 N6 T1 M6
15 N7 G0 G90 X-1.125 Y1.125 A0. S5000 M3
16 N8 G43 H1 Z2. M8
17 N9 Z.1
18 N10 G99 G81 Z-.28 R.1 F7.
19 N11 Y-1.125
20 N12 X1.125
21 N13 Y1.125
22 N14 G80
23 N15 Z2.
24 N16 M5
25 N17 G91 G28 Z0. M9
26 N18 A0.
27 N19 M01
28 N20 T2 M6
29 N21 G0 G90 X-1.875 Y1.53 A0. S4500 M3
```

8. Select the close button  in the top right corner to exit the CNC editor.

➤ This completes **Mill-Lesson-4**.

MILL-LESSON-4 EXERCISES

3.00
2.00
2.75
X0Y0
R.25 (4)
Ø.125 (2)
Z0
.125
.125

Mill-Lesson-4 Exercise
Material: Aluminum T6O61
All Dimensions in Inches

CAMInstructor.COM

