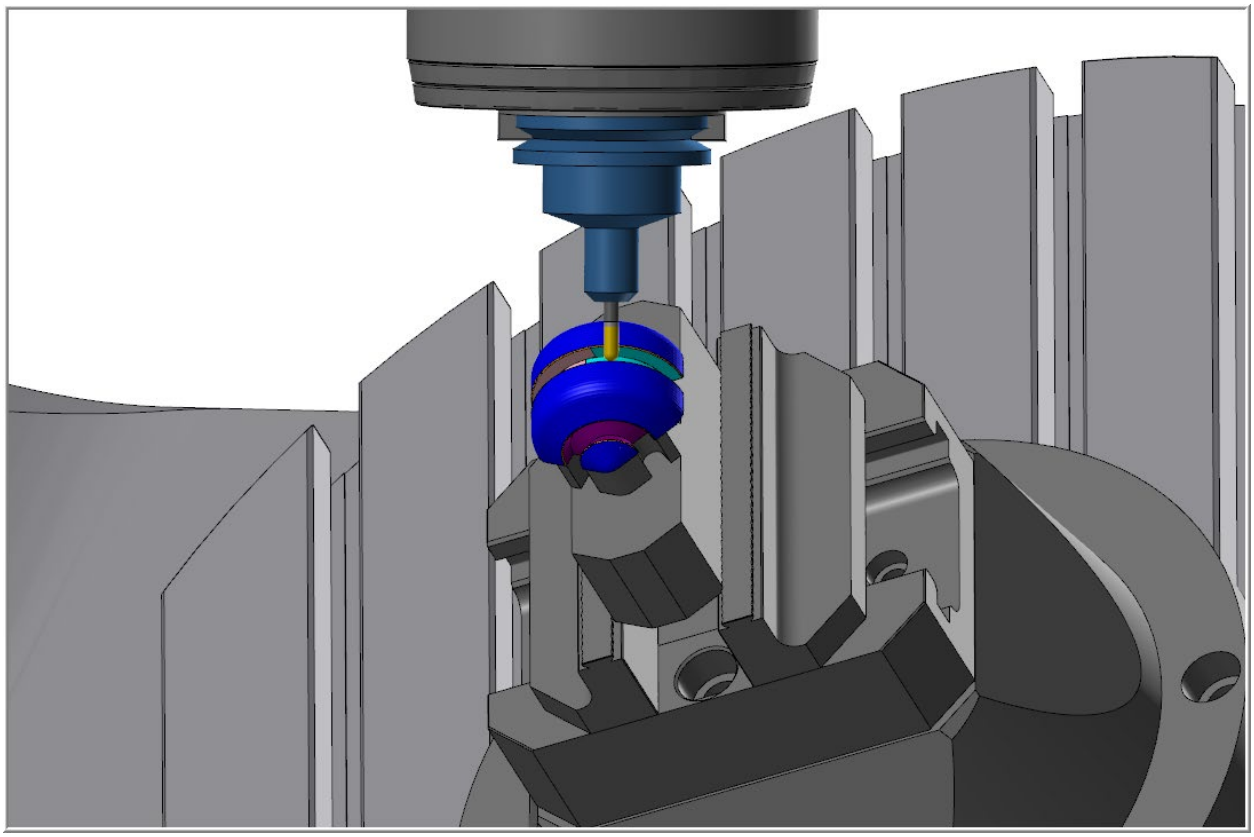
Mastercam 2020

Training Guide



5-Axis-Lesson-8

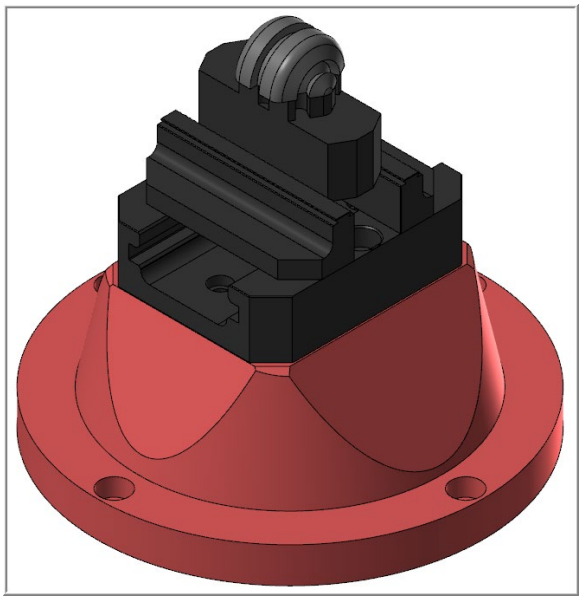
UMC-750-OptiRest-Flow & Curve 5 Axis

camInstructor

OBJECTIVES

For **5-Axis-Lesson-6** you will generate the toolpaths to machine the part on a Haas UMC-750 universal milling machine. The part will be held in a 5-axis vise.

This Lesson covers the following topics:

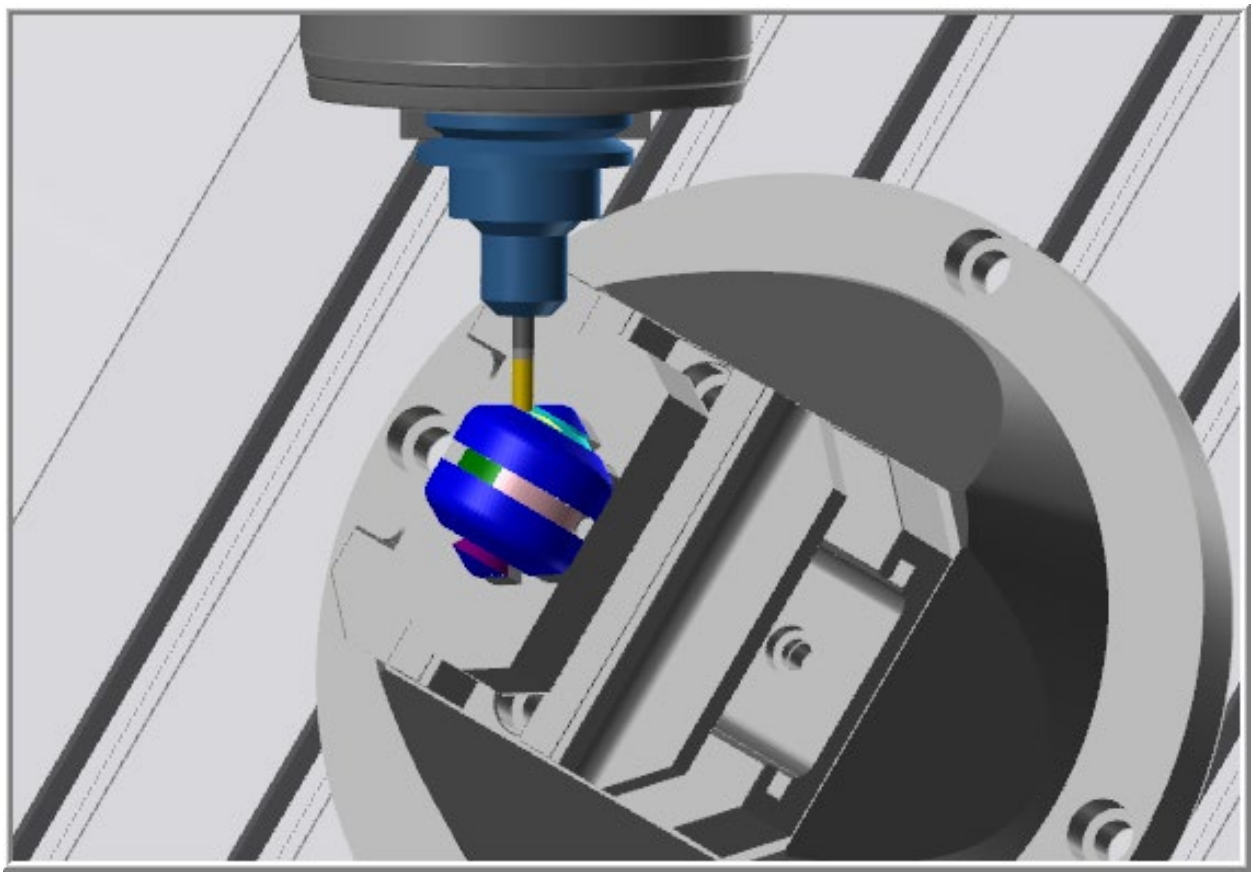


➲ **Importing the machine:**
Mastercam can simulate exact replicas of machine tools to ensure crash free programs. You will import these files.

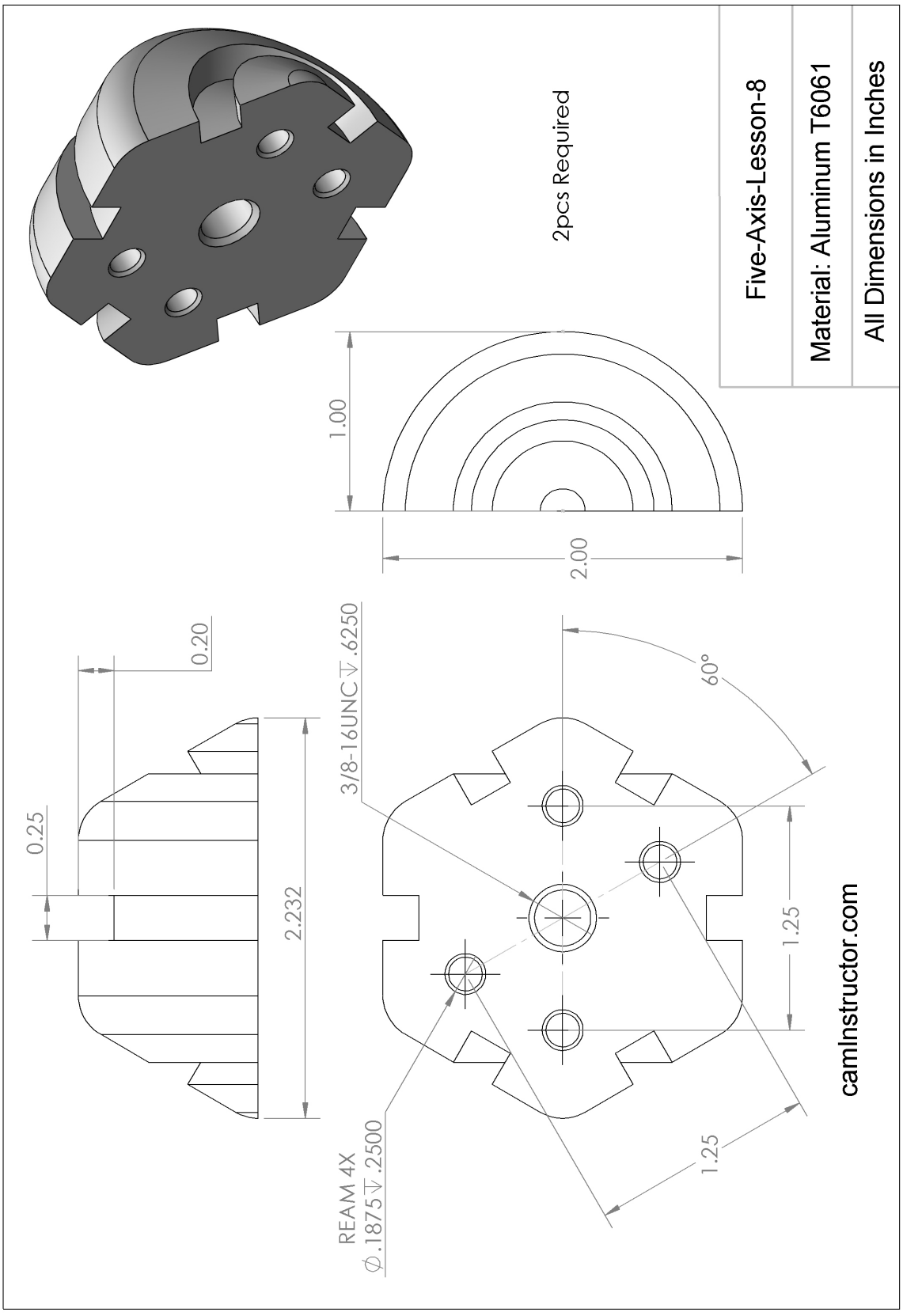
➲ **Open an existing file containing:**
The solid geometry for the machine vise
Solid geometry for the part
Tools for machining

➲ **Generate toolpaths consisting of:**
3+2 3D roughing, Flow 5 Axis, Curve 5 Axis, and Multaxis Deburr

➲ **Inspect the toolpath using Mastercam's Verify and Backplot and Machine Sim by**



Five-Axis-Lesson-8-DRAWING



TOOL LIST

#	Tool Name	Holder Name	Dia.	Cor. rad.	Length	# Flutes	Type	Rad. Type
5	1/4 FLAT ENDMILL	C4E4-0375	0.25	0.0	0.625	4	Flat endmill	None
6	3/16 BALL ENDMILL	C4E4-0187	0.1875	0.09375	0.4375	4	Ball endmill	Full
7	3/8 BALL ENDMILL	C4E4-0375	0.375	0.1875	0.75	4	Ball endmill	Full

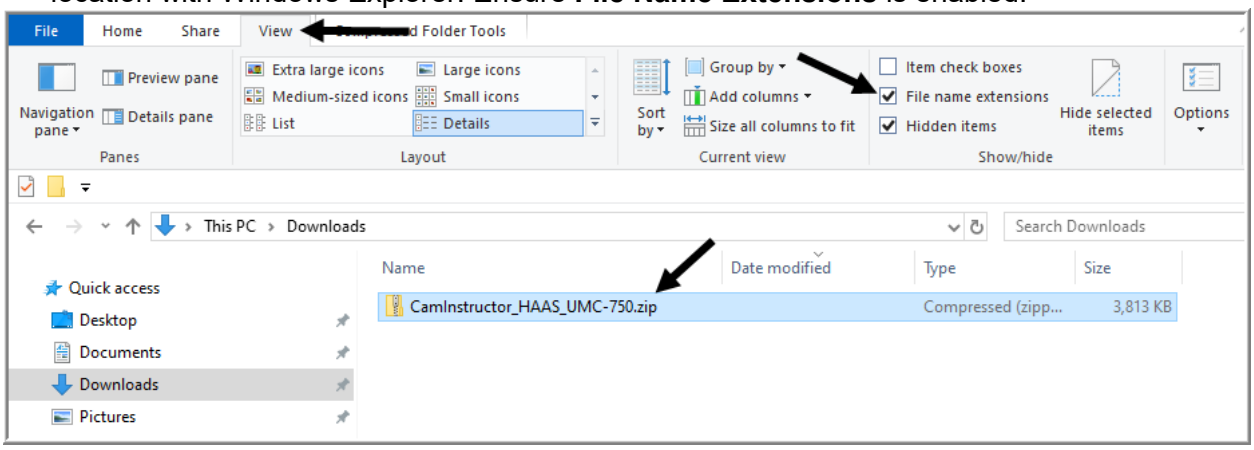
5-AXIS-LESSON-8 - THE PROCESS

TASK 1: Import Machine files
TASK 2: Setup the Machine
TASK 3: Open the Mastercam file
TASK 4: Rough the part with OptiRest
TASK 5: Finish the surface with Flow 5 Axis
TASK 6: Finish the end slots with Curve 5 Axis
TASK 7: Finish the middle slot with Curve 5 Axis
TASK 8: Deburr the part
TASK 9: Setup the Verify and Simulate options
TASK 10: Verify and Simulate the part

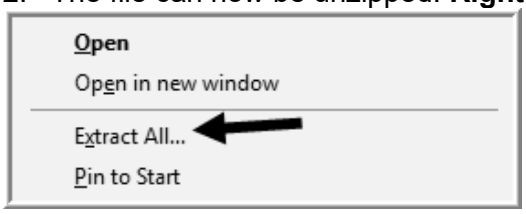
TASK 1:
IMPORTING THE MACHINE FILES

- If Machine Importing was done in the previous lessons, Task 1 and 2 of this lesson can be skipped.
- When importing a machine, the following steps may vary slightly depending on the files your provided and the format they are provided in. If your reseller has provided instructions for file importing, you should follow those instructions instead of these.
- Watching the videos for this step may be a beneficial first step.

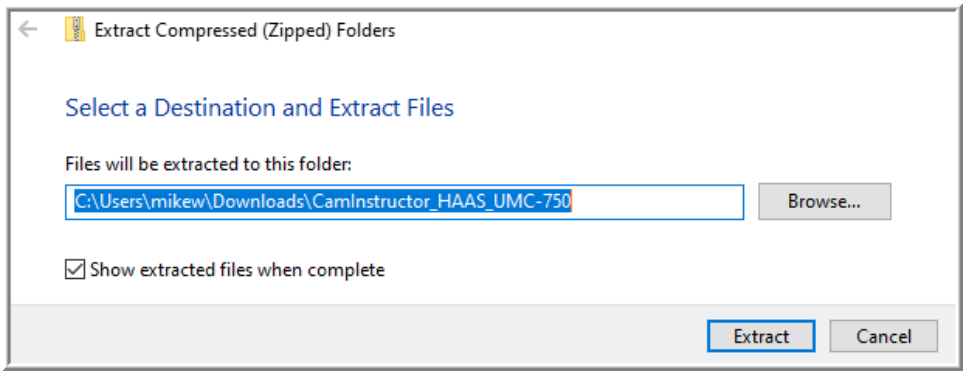
1. We provide a **single file** download as one of the lesson steps. This file contains all the necessary files for the UMC-750 in this lesson. It **does not** include a post processor that will produce gcode. This file is in a **zip** format. Save the file in a known location. Open that location with Windows Explorer. Ensure **File Name Extensions** is enabled.



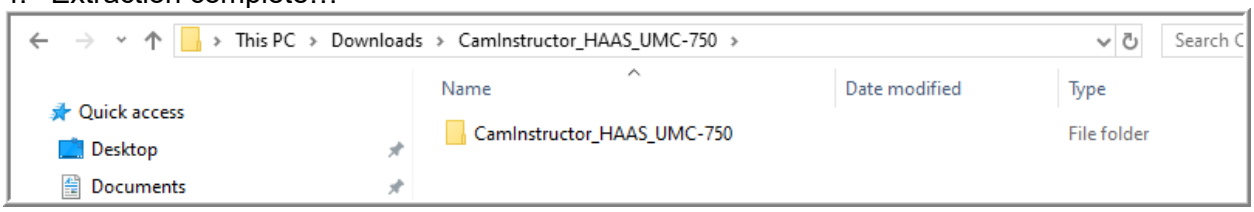
2. The file can now be unzipped. **Right click** on the file and choose **Extract All…**



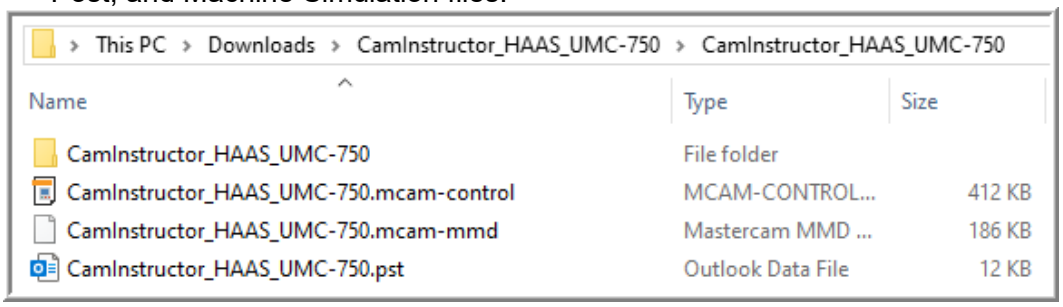
3. By default, the extraction should be set to output into the same location as the file being extracted but into its' own folder. Click **Extract**.



4. Extraction complete…

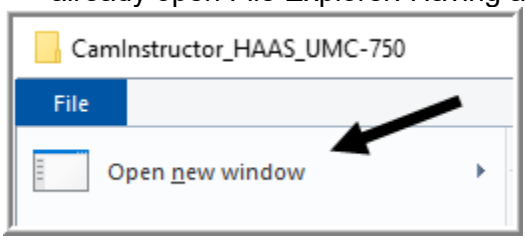


5. Open the extracted folder. The contents included may vary depending on the supplied machine and its options. Our download contains a Control Definition, Machine Definition, Post, and Machine Simulation files.

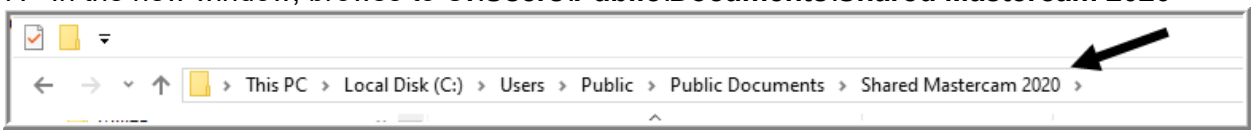


➲ The included post will only populate text blocks within toolpaths. It will not create any gcode.
➲ The files now need to be transferred to the default locations for their file types.

6. Open a second instance of Window Explorer by clicking **File>Open new window** in the already open File Explorer. Having a second window open will make coping files easier.

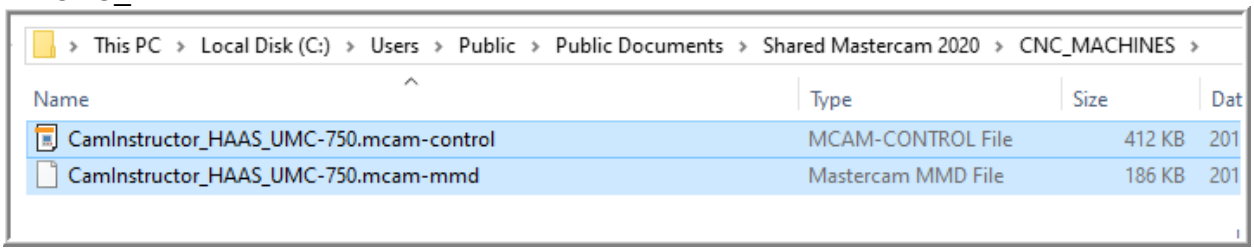


7. In the new window, browse to **C:\Users\Public\Documents\Shared Mastercam 2020**

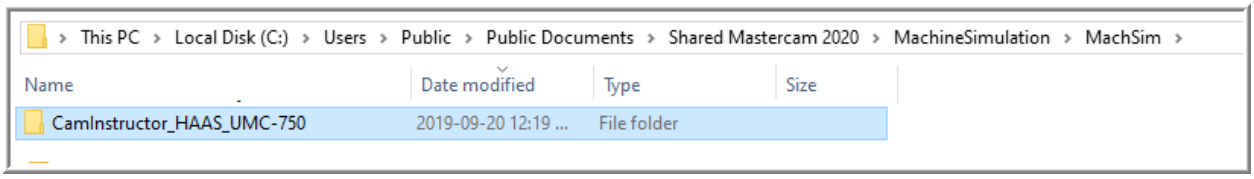


➲ We will be copying files into the **CNC_MACHINES**, **MachineSimulation/MachSim,** and **mill/Posts** folders in this directory.

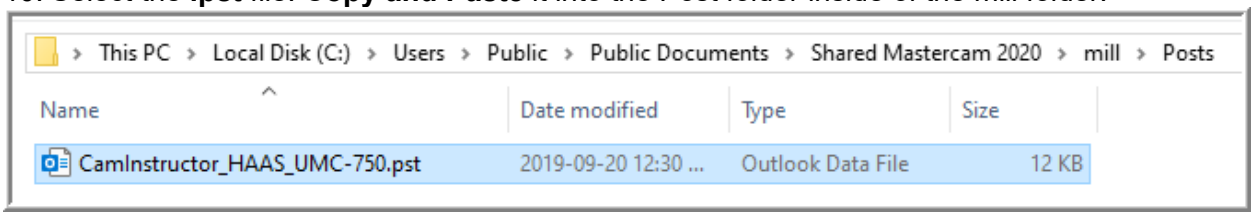
8. Select the **mcam-control** and **mcam-mmd** files. **Copy and Paste** them into the CNC_Machines folder.



9. Select the **CamInstructor_HAAS_UMC-750** folder. **Copy and Paste** it into the MachSim folder inside of the MachineSimulation folder.



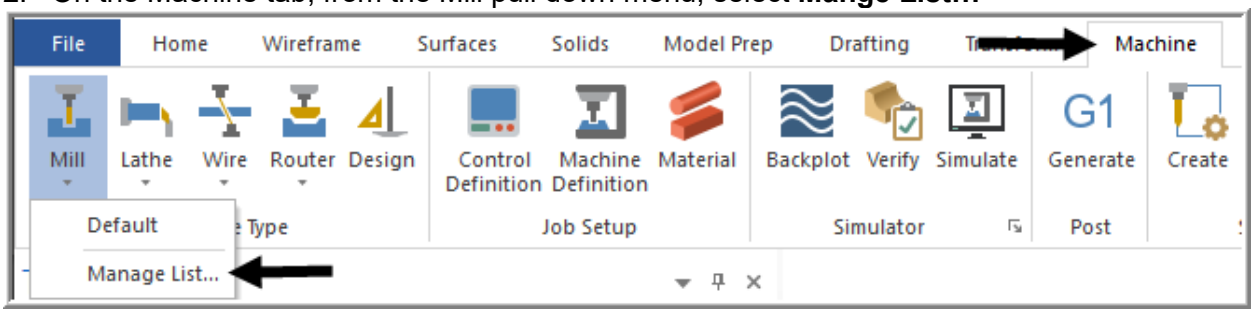
10. Select the **.pst** file. **Copy and Paste** it into the Post folder inside of the mill folder.



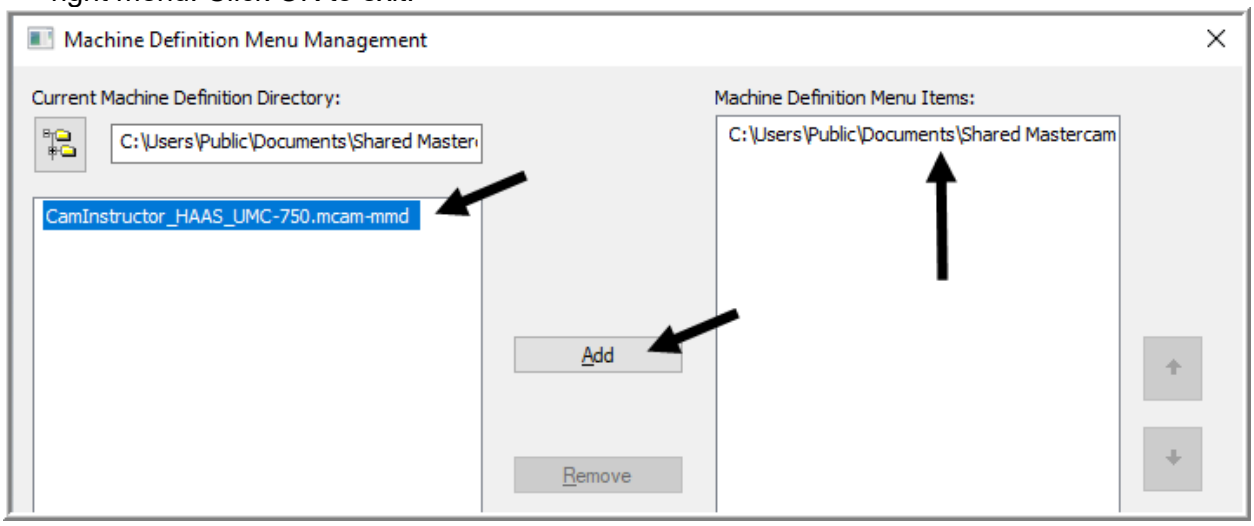
➲ This completes the file importing.

TASK 2:
SETUP THE MACHINE

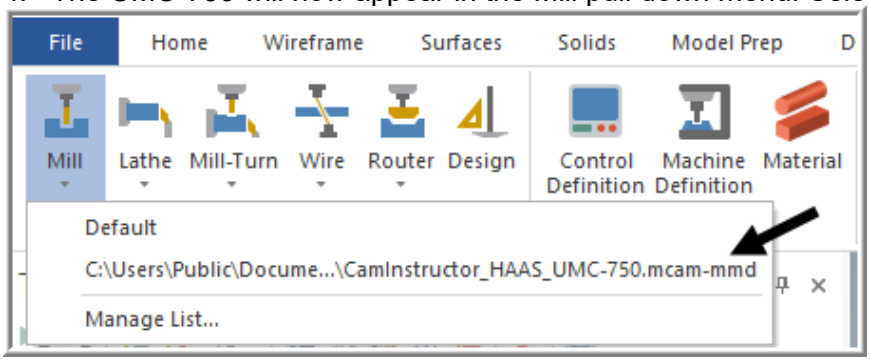
1. Launch Mastercam.
2. On the Machine tab, from the Mill pull down menu, select **Mange List…**



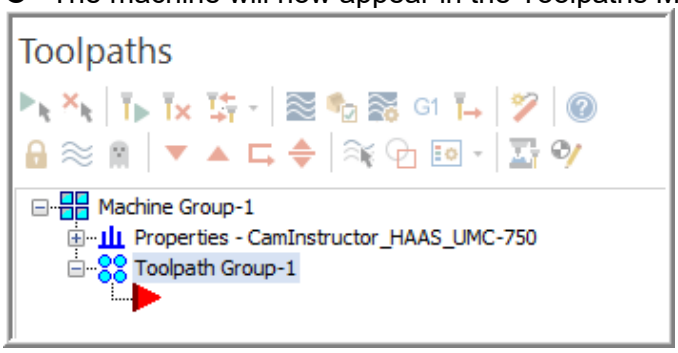
3. Select the inch version of the UMC-750 and click **Add**. The machine will then appear in the right menu. Click OK to exit.



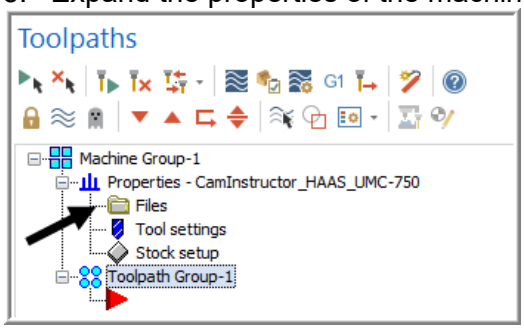
4. The UMC-750 will now appear in the Mill pull down menu. Select it now.

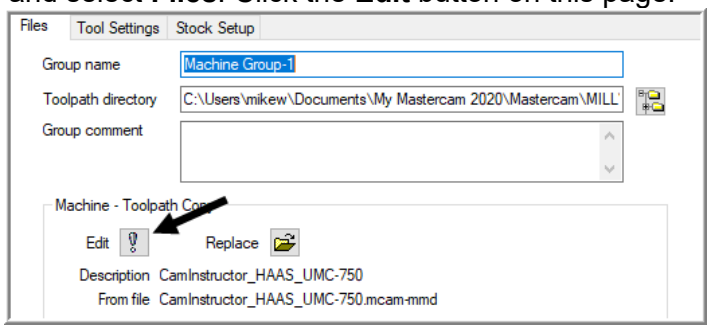


➲ The machine will now appear in the Toolpaths Manager.

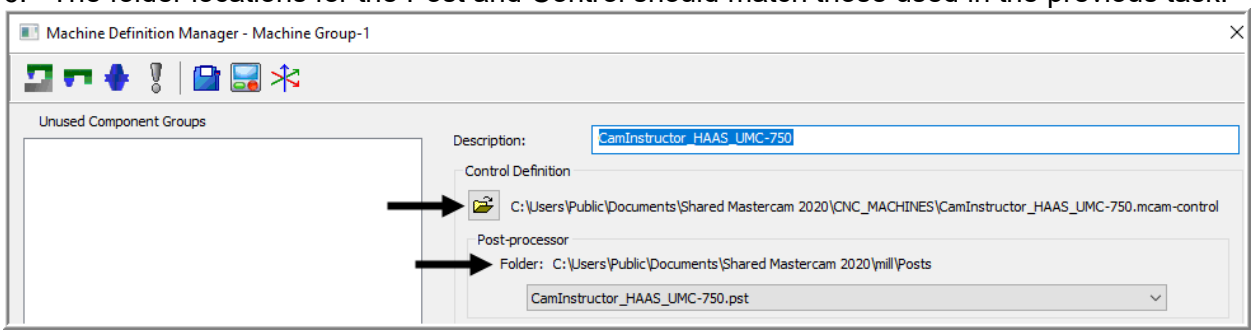


5. Expand the properties of the machine and select **Files**. Click the **Edit** button on this page.





6. The folder locations for the Post and Control should match those used in the previous task.



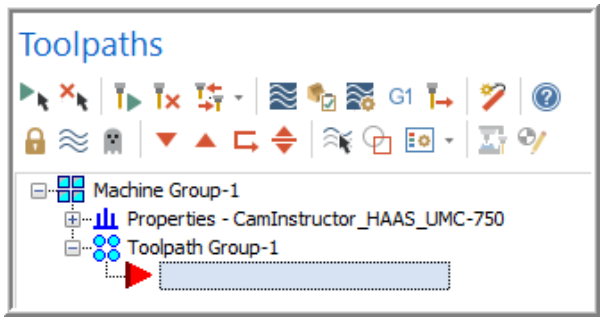
➲ You are now ready to program with the UMC-750.

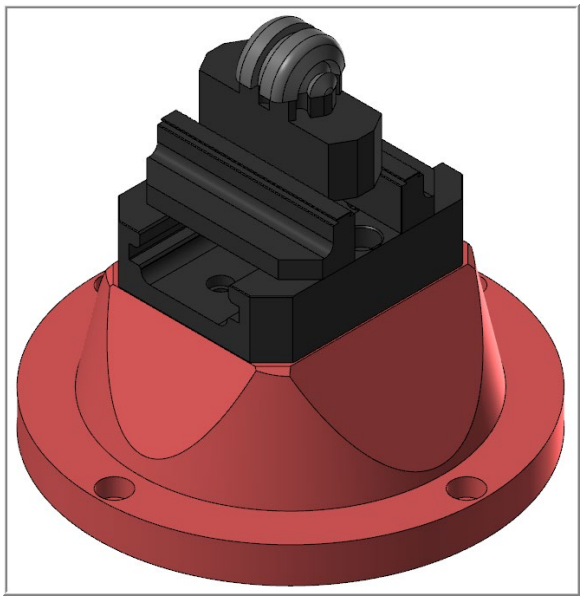
➲ Machine Simulation setup will be done later when there are available cutting paths.

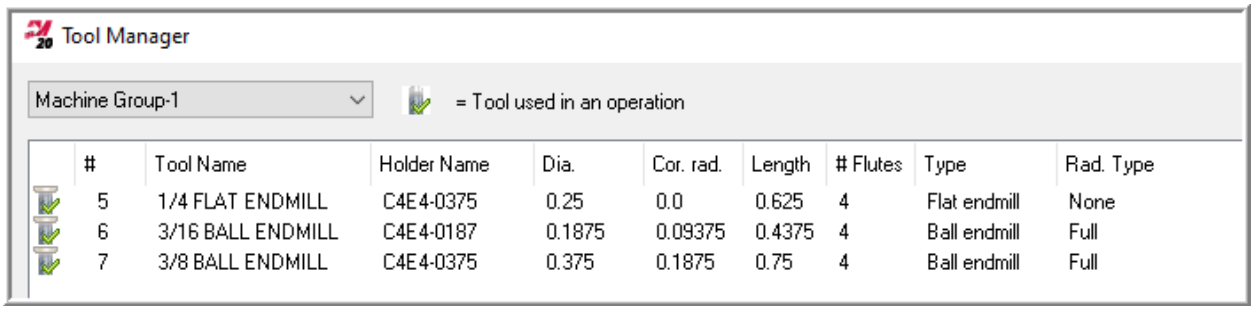
TASK 3:
OPEN THE MASTERCAM FILE

- This lesson will start by using an existing Mastercam file which has the part and fixtures loaded and in location already, tools with holders defined and speeds and feeds stored. Future lessons will walk through the placement of the part and the fixtures as well as tool holder selection/modification and speed and feed calculations.
- The machine loaded in this file is the camInstructor definition of the Haas UMC-750. Machine Simulation will also use camInstructor definitions. If you do not have this machine, the Default Mill can be used and a Trunnion VMC can be used for Machine Simulation.

1. The file can be **downloaded** as a step in the Online Lesson. Make note of the save location and open the file using the **File>Open** command in Mastercam.







#	Tool Name	Holder Name	Dia.	Cor. rad.	Length	# Flutes	Type	Rad. Type
5	1/4 FLAT ENDMILL	C4E4-0375	0.25	0.0	0.625	4	Flat endmill	None
6	3/16 BALL ENDMILL	C4E4-0187	0.1875	0.09375	0.4375	4	Ball endmill	Full
7	3/8 BALL ENDMILL	C4E4-0375	0.375	0.1875	0.75	4	Ball endmill	Full

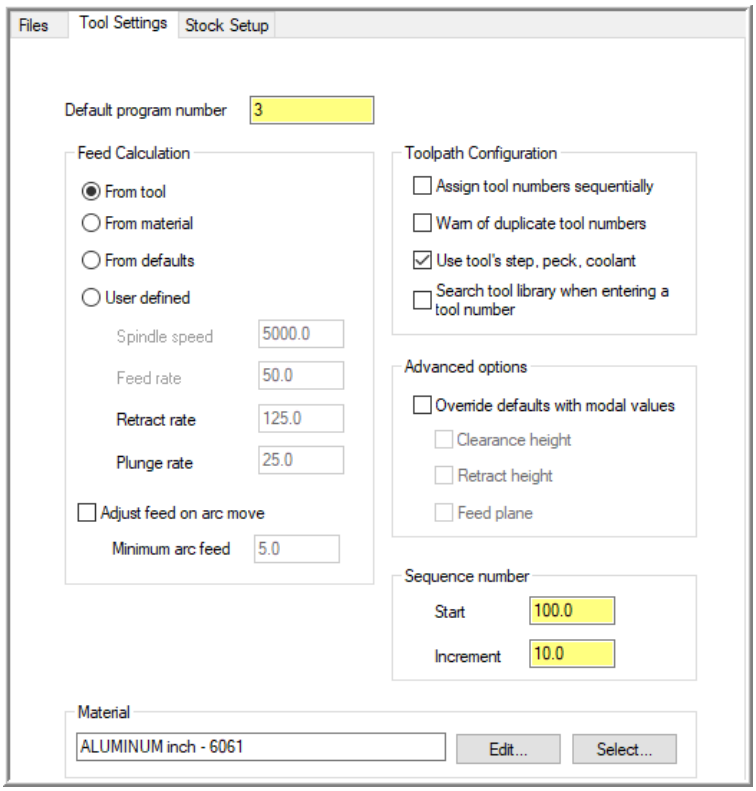
➲ Tool Settings

Feed is set to **From Tool** so the speeds and feeds set in the tools parameters will be used.

Use Tools Step, Peck, Coolant is enabled and these values will be populated into the created operations automatically from the tools parameters.

Override Defaults is enabled so any adjustments made to the retract values will carry over to subsequent toolpaths.

The **Material** type is set correctly but will have no effect on the toolpaths, it is set for reference only here.



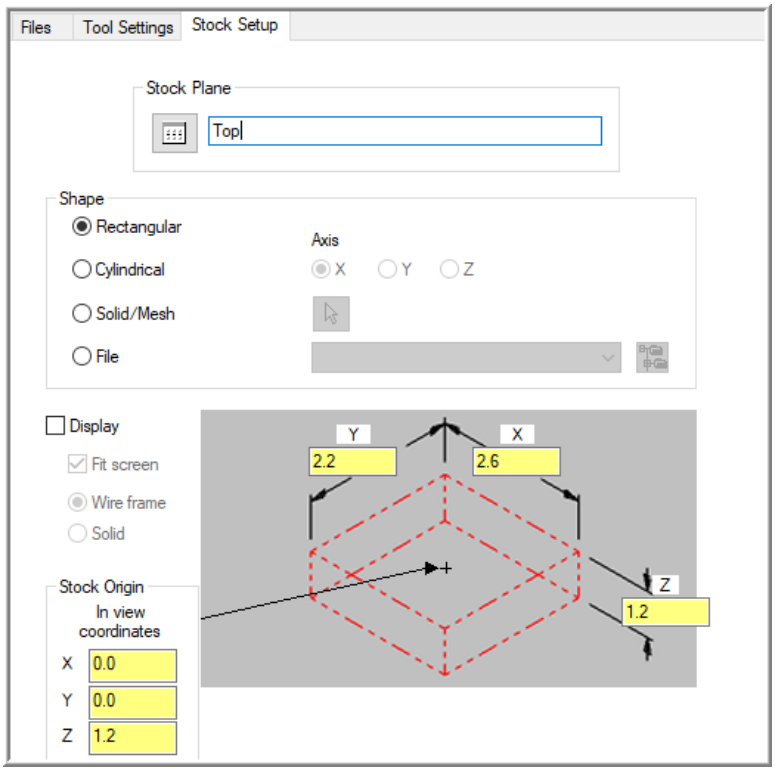
➲ Stock Setup

The stock has been defined as a **2.2x2.6x1.2 block**. This leaves stock on all faces except the bottom, which is bolted to the fixture.

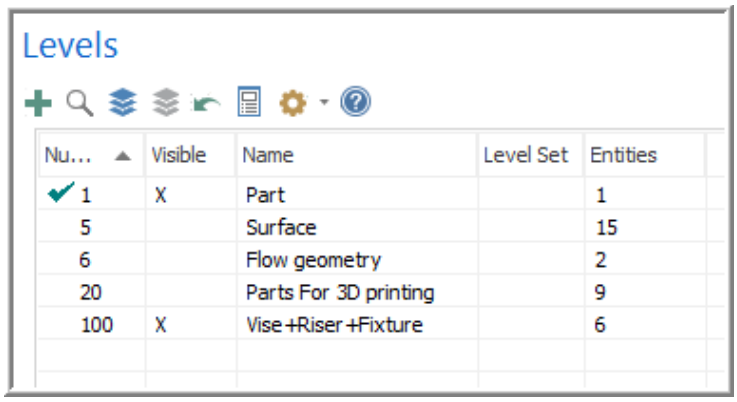
Depending on the stock you cut, adjust the block to accurately reflect its dimensions.

The stock block used on the machine should have the bottom face machined as well as the holes already drilled into it.

Z0 for the stock setup is at the bottom of the block, which is at the top of the fixture.



➲ Levels



Level 1 contains the solid model for this project.

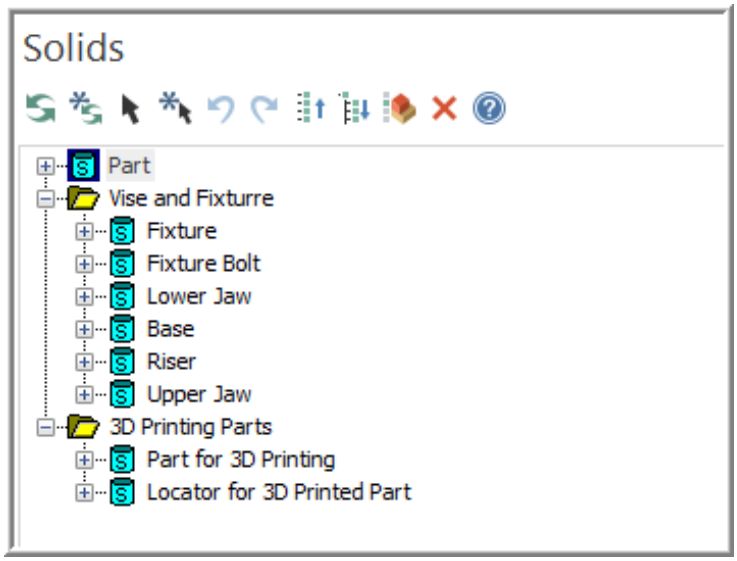
Level 5 contains a surface which will be used in a toolpath. This surface is used to bridge the gaps left by the slots.

Level 6 contains geometry which will be used for a toolpath.

Level 20 contains solid models that can be used for 3D printing.

Level 100 contains all the solids used to represent the fixtures.

➲ Solids



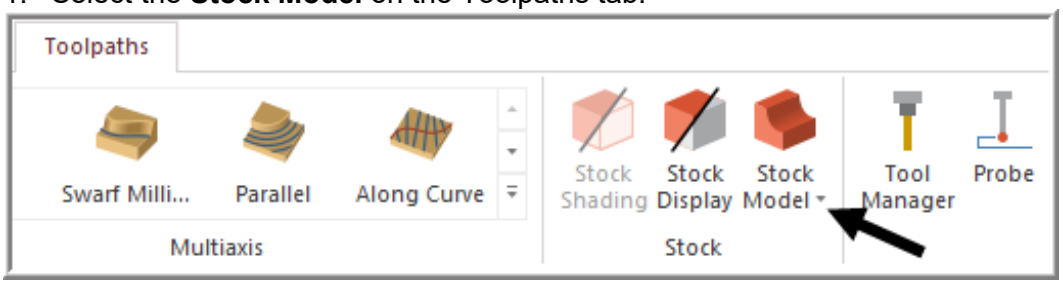
There are several solids in the file. The solids used to represent the vise and fixture have been grouped together as have the 3D printing models. The part model remains ungrouped.

Groups can be used when many solids are used to represent an assembly. The Group can be collapsed to de-clutter the Solids Manager.

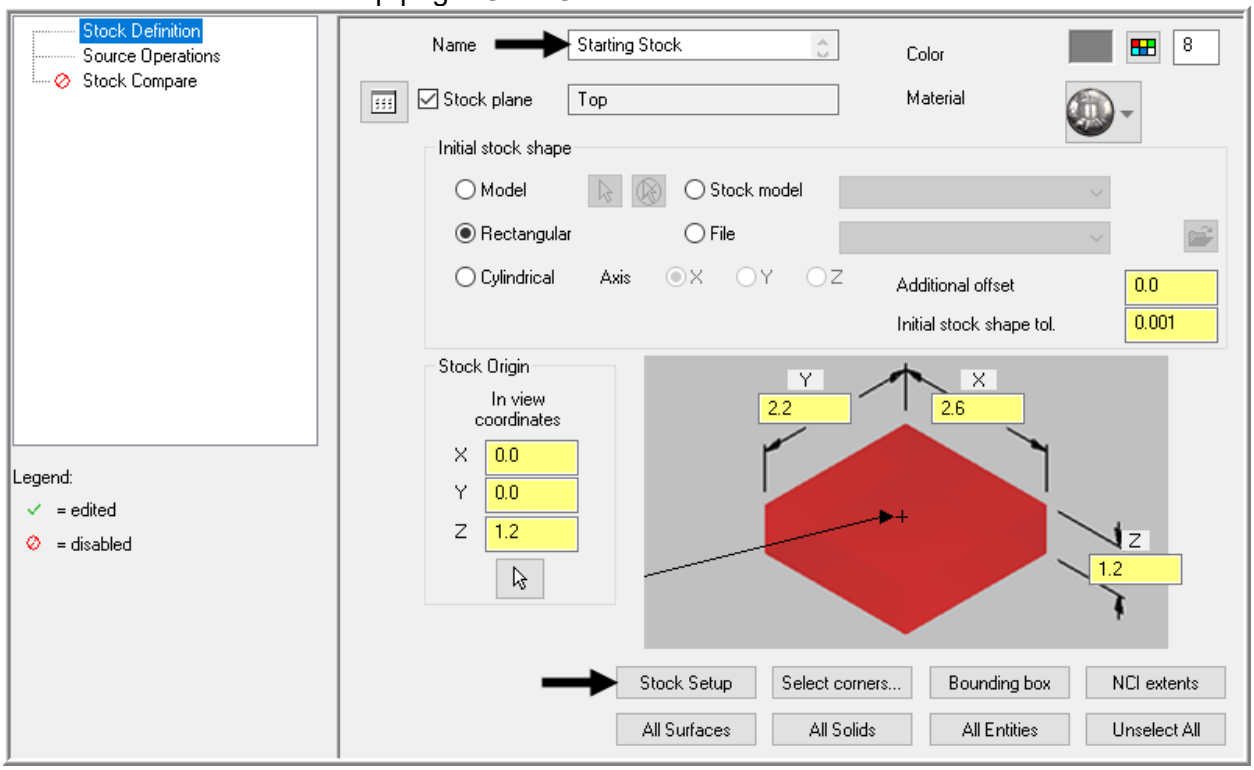
TASK 4:
ROUGH THE PART

➲ The part will be roughed out with a 3D OptiRough toolpath. To make the toolpath stock aware, a Stock Model must first be made then the OptiRough toolpath can be used as a Rest milling operation.

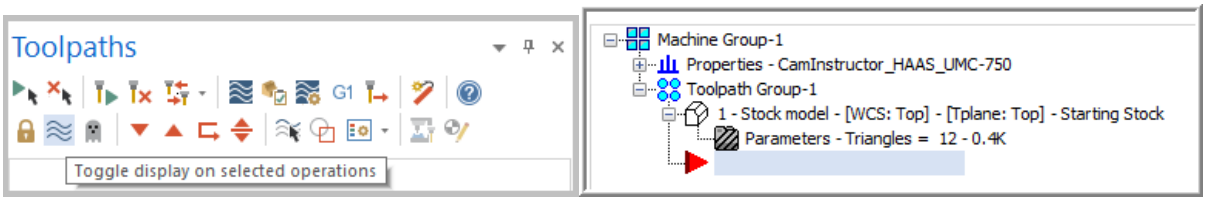
1. Select the **Stock Model** on the Toolpaths tab.



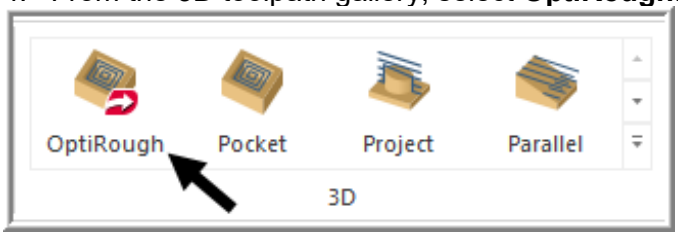
2. Give the Stock Model a **name**, set the Initial Stock Shape to **Rectangle** and then click the **Stock Setup button** to populate the origin and size of the rectangle from the values used in the Machines Stock Setup page. Click **Ok**.



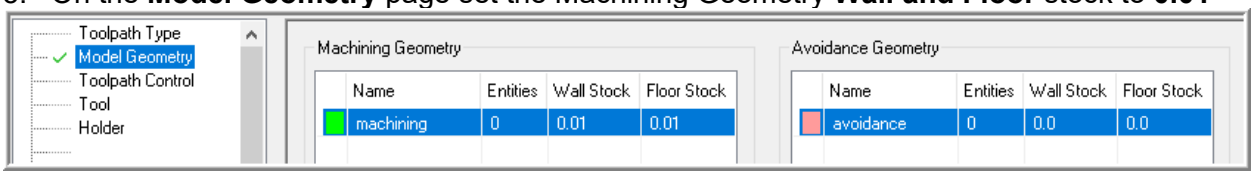
3. Turn the stock model on/off by using **Alt - T** or by clicking on the **Toolpath** Toggle display icon.



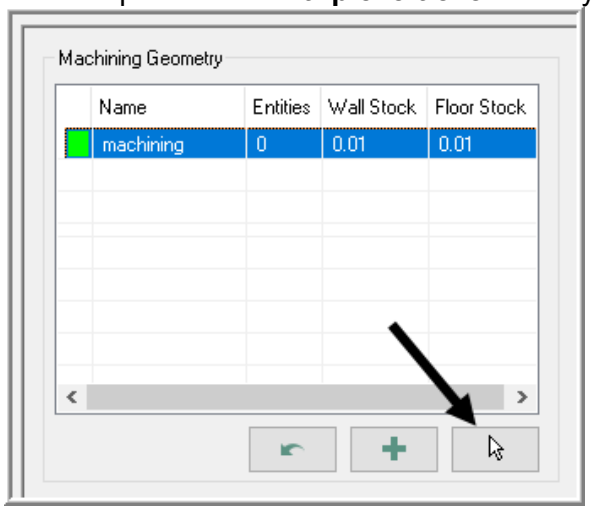
4. From the 3D toolpath gallery, select **OptiRough**.

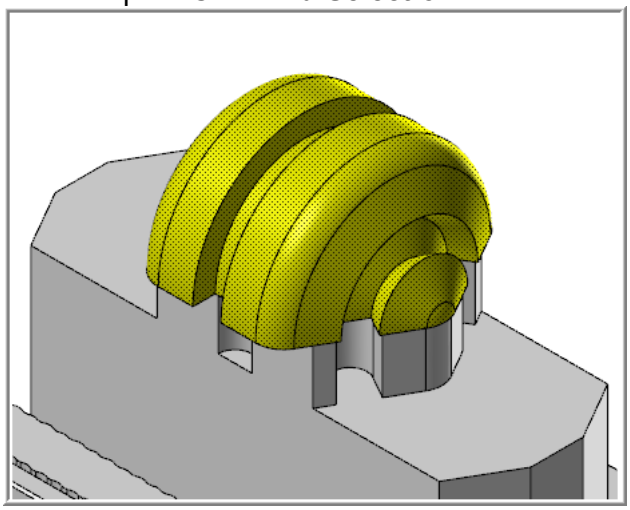


5. On the **Model Geometry** page set the Machining Geometry **Wall and Floor** stock to **0.01**



6. Use the selection button at the bottom of the Machining geometry section to select the entire solid part. You can **triple left click** on any face of the part. Click **End Selection**.

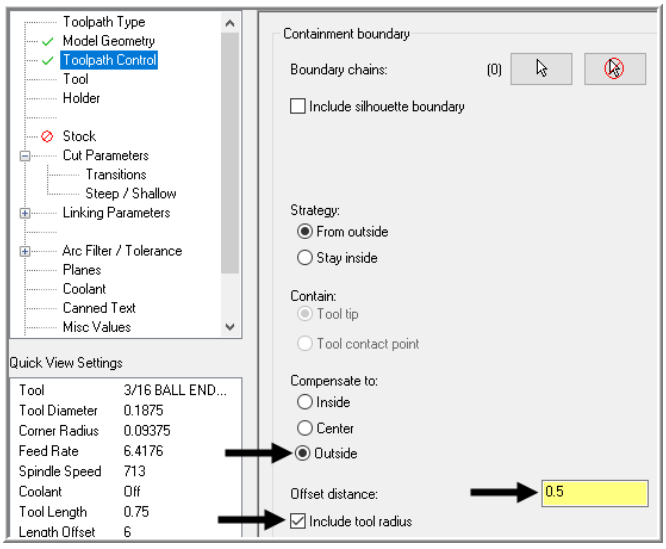




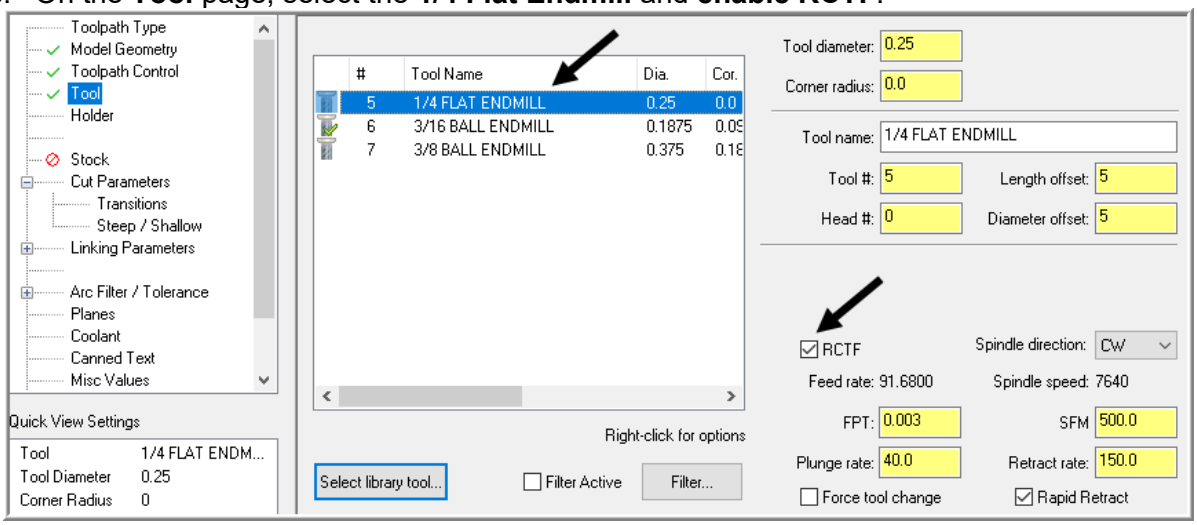
7. **Toolpath Control** page.

   Strategy set to **From Outside**.

   Compensate set to Outside with the **Offset Distance set to 0.5** and **Include Tool Radius checked**. This will allow the toolpath to look outside the Boundary chain for material to cut. If Stock is not used with this operation, care must be taken in setting of the Offset Distance.



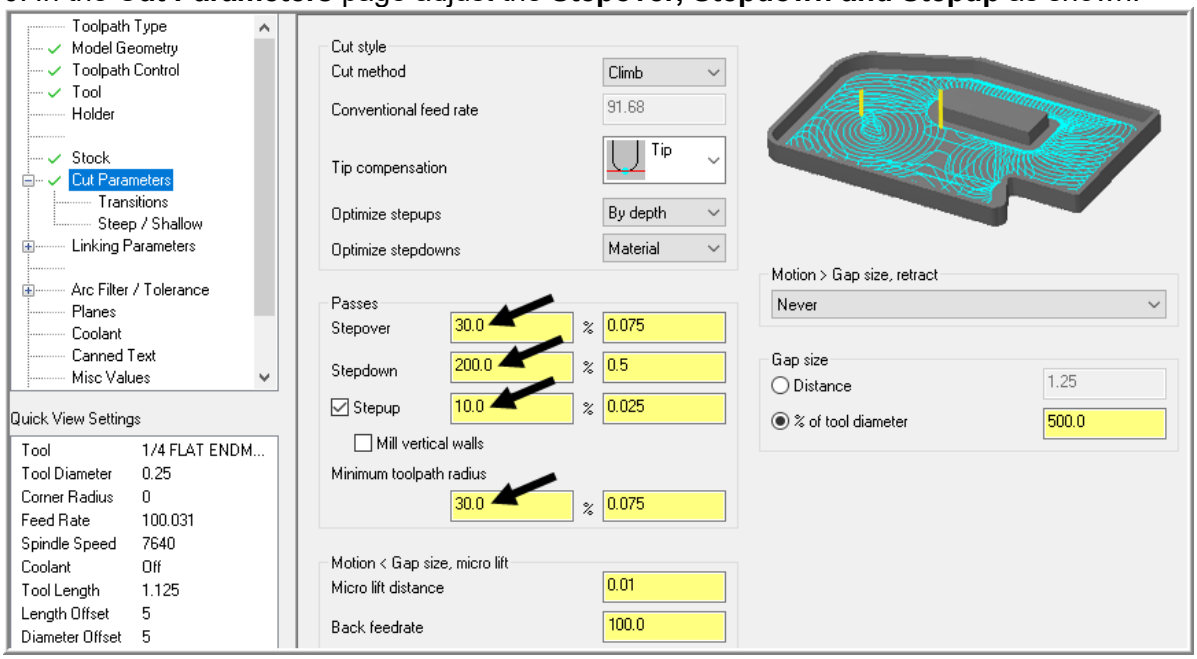
8. On the **Tool** page, select the **1/4 Flat Endmill** and **enable RCTF**.



9. **Stock** page. Check the Rest Material box. Set the stock to One Other Operation and ensure the created stock model is selected.

➲ Enabling the Stock will limit where the toolpath will cut. The Toolpath Control settings are allowing the operation to start cutting outside of the part boundary but the Stock settings are reducing the created toolpaths to locations where there is stock to cut.

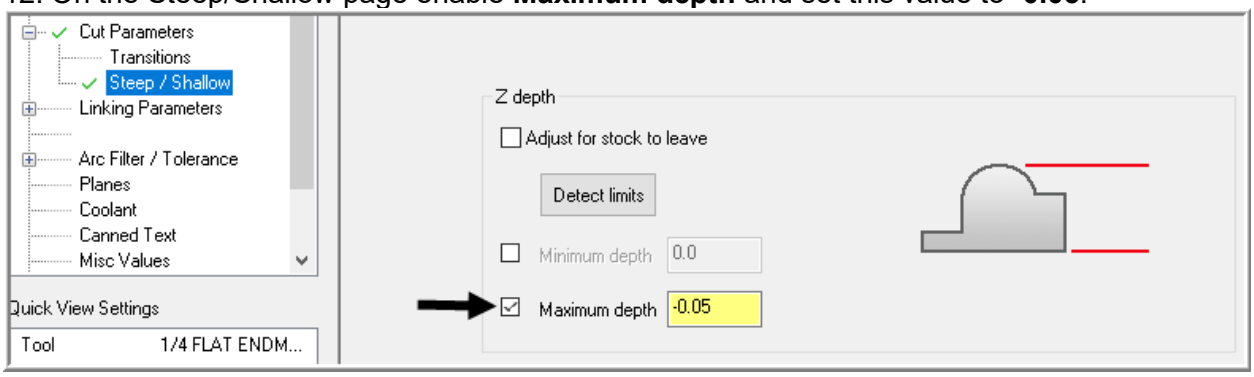
10. In the **Cut Parameters** page adjust the **Stepover, Stepdown and Stepup** as shown.



11. No settings are needed on the **Transitions** page. This toolpath will not be entering any pockets so there is no need to define this motion.

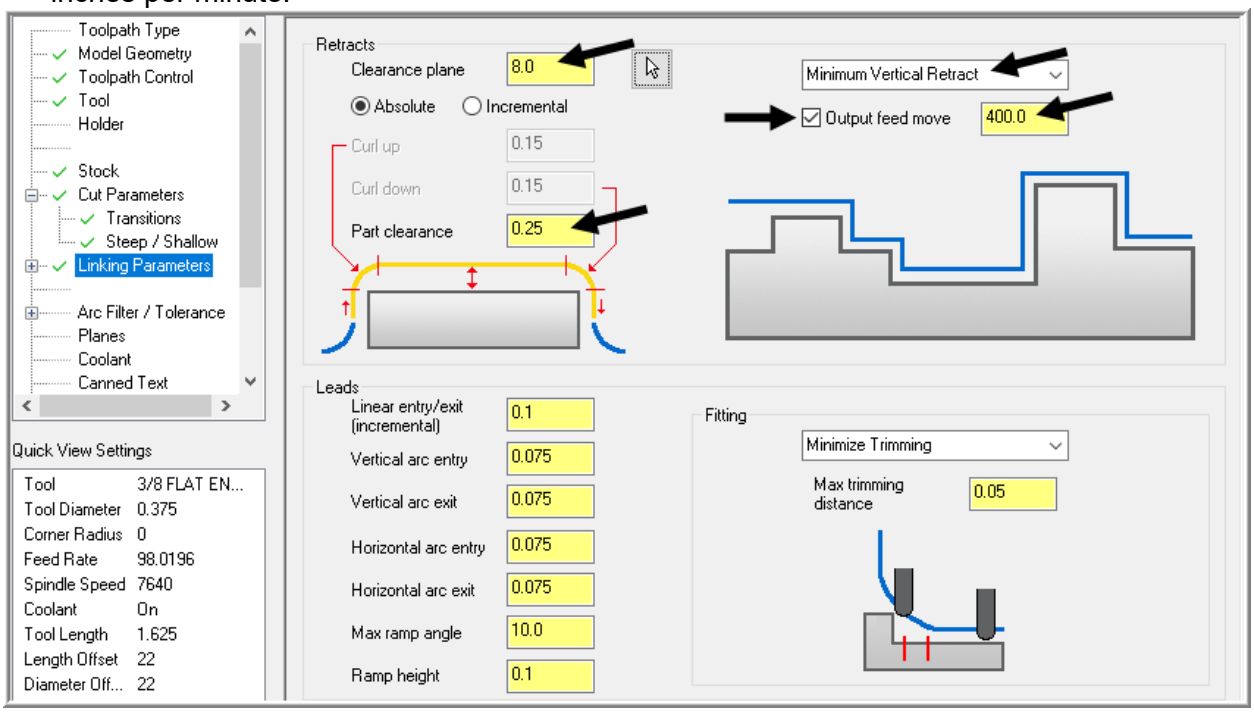
➲ The Steep/Shallow page will be used to limit the depth of the toolpath. The path needs to cut slightly below the bottom of the part.

12. On the Steep/Shallow page enable **Maximum depth** and set this value to **-0.05**.



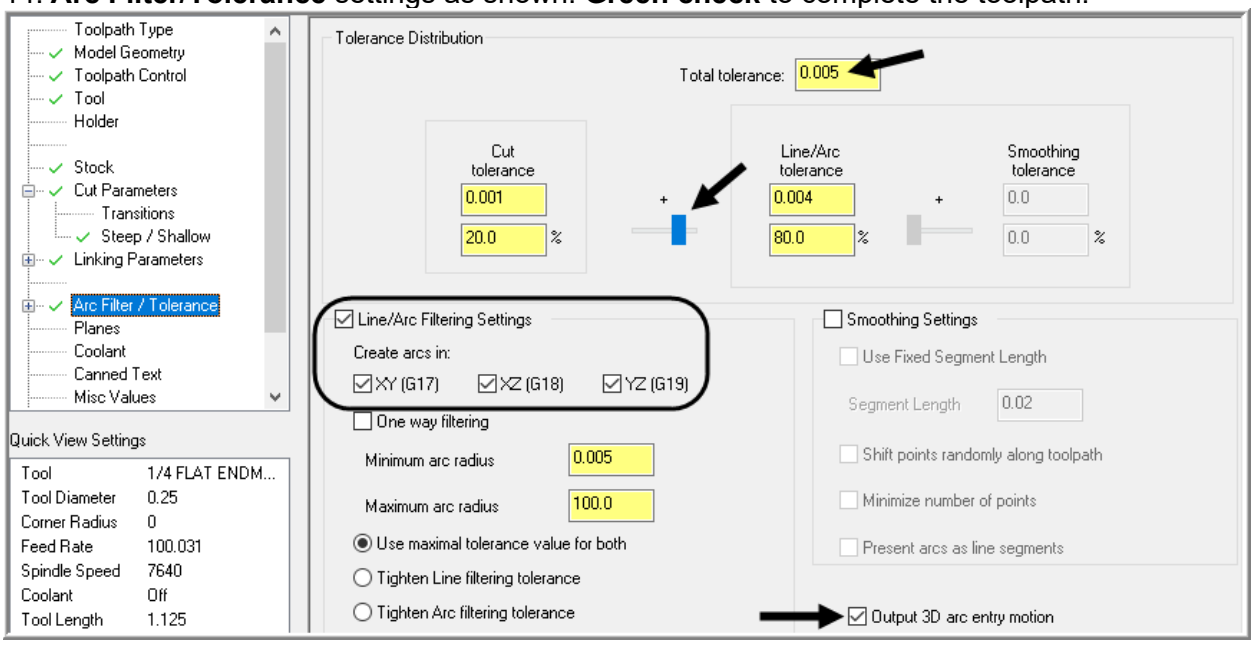
➲ The Steep/Shallow setting will be re-visited later.

13. Linking Parameters. Clearance Plane set to **8.0**, Part Clearance set to **0.25**. From the retract style pulldown select Minimum Vertical Retract, enable Output Feed Move and set to **400** inches per minute.

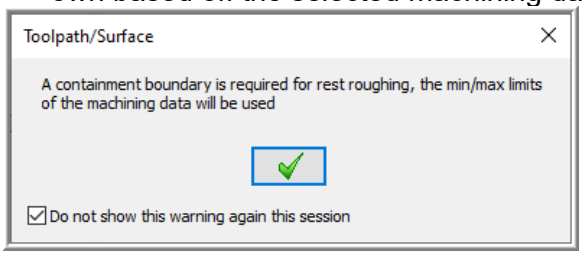


➲ The above settings will allow the toolpath to rapid around the part with 0.25 inches of clearance. The Output Feed move allows for straight line ‘rapids’ to be performed. Typically, machines will dogleg rapid which may lead to unsimulated crashes on the machine.

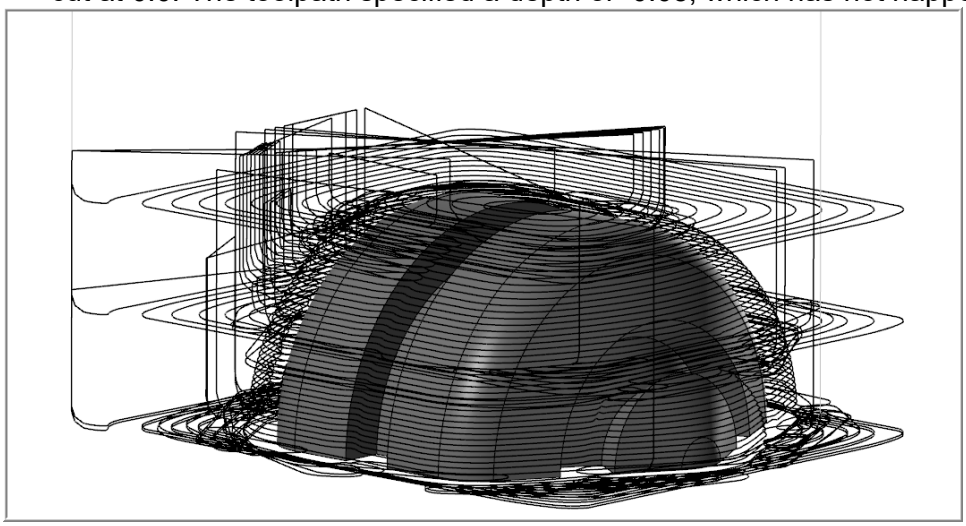
14. **Arc Filter/Tolerance** settings as shown. **Green check** to complete the toolpath.



15. If you are prompted about a containment boundary, **click Do Not Show and then click Ok**. The message is stating that since a boundary was not selected, Mastercam will create it's own based off the selected machining data.

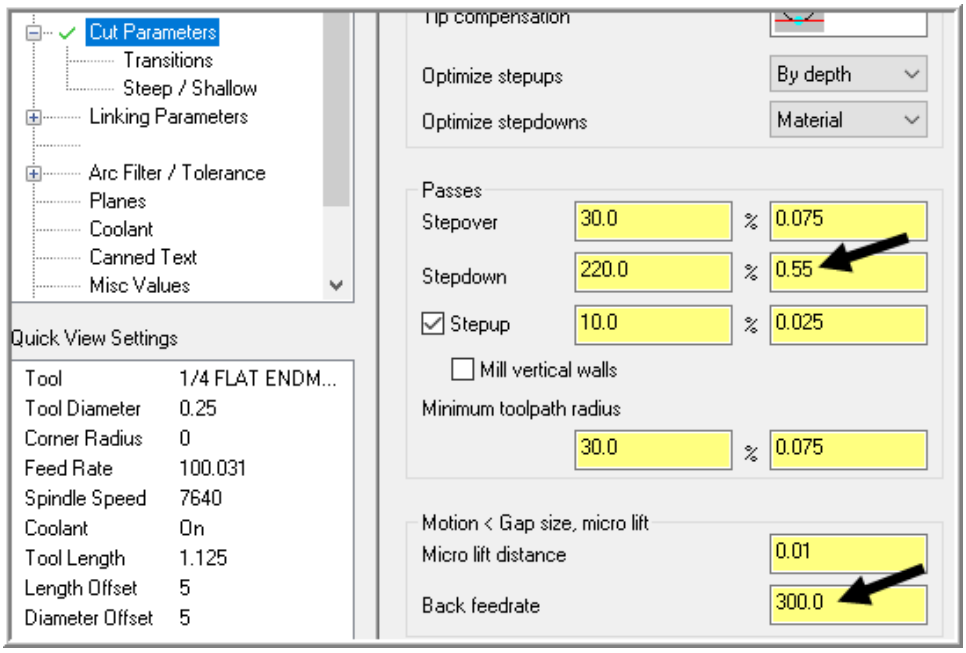


➲ The resulting toolpath, fixture hidden for clarity. Backplot the toolpath, notice the estimated cycle time. 6m44s. When verified, you may notice a depth cut at 0.0108 and the final depth cut at 0.0. The toolpath specified a depth of -0.05, which has not happened.

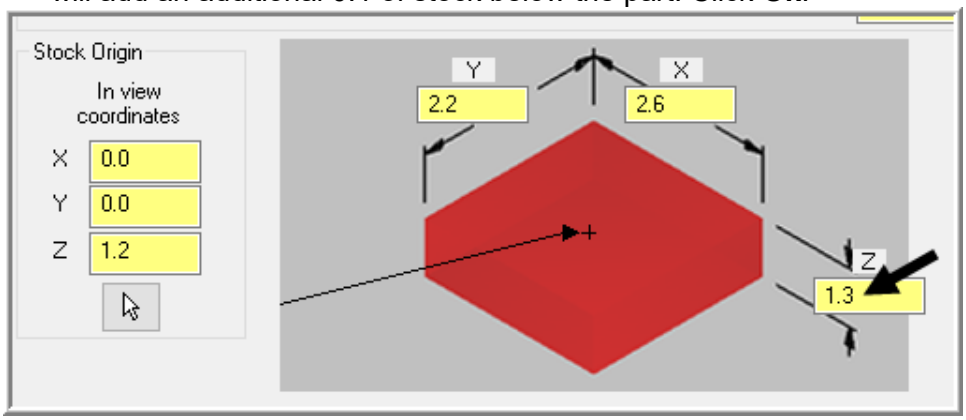


- This is a stock aware toolpath so motion will only be generated where there is stock to cut. In order to cut below Z0, the toolpath needs to see stock there to cut. Next, we will adjust the stock model to have stock in this area.
- The depth cuts at 0.0108 and 0.0 are not efficient. Cutting a slightly larger stepdown will eliminate this extra depth cut.
- Time can also be saved by moving faster during non-cutting portions of the toopath. This is controlled with the Back Feedrate setting.

16. Click on the Toolpaths Parameters folder to access the toolpaths settings. On the Cut Parameters page **increase the Stepdown to 0.55** and increase the **Back feedrate to 300**. Click **Ok**.

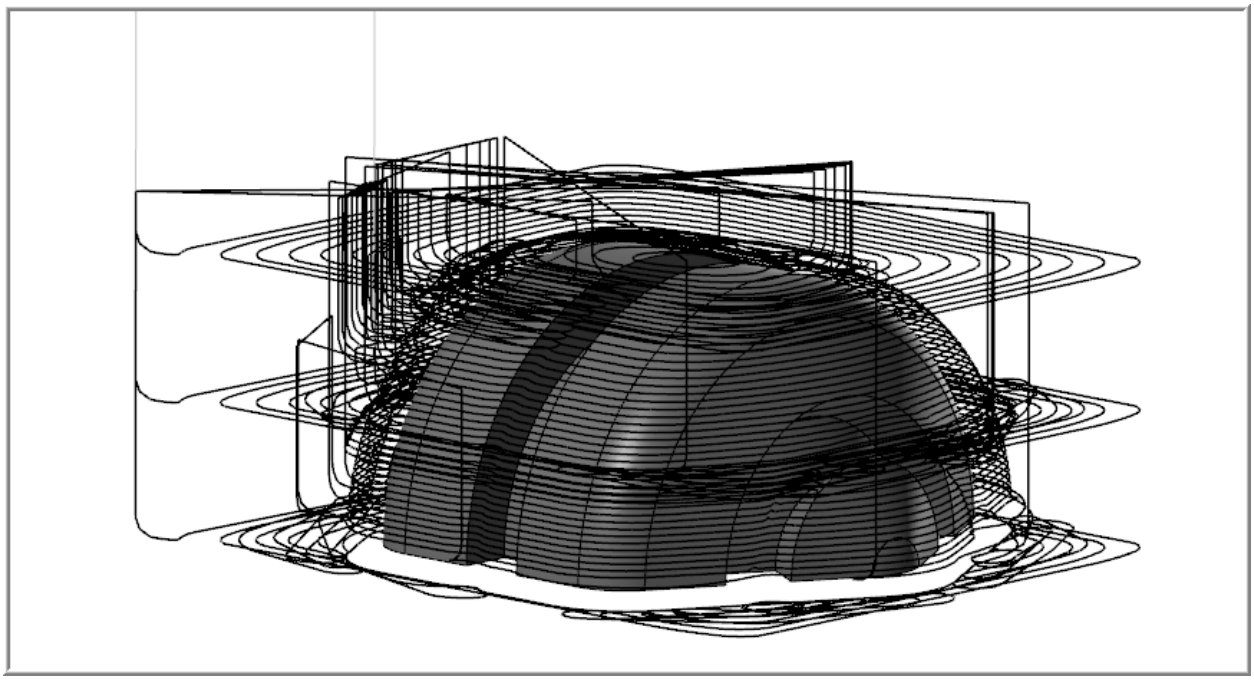


17. Click the **Parameters folder for the Stock model**. Adjust the **Stock Thickness to 1.3**. This will add an additional 0.1 of stock below the part. Click **Ok**.



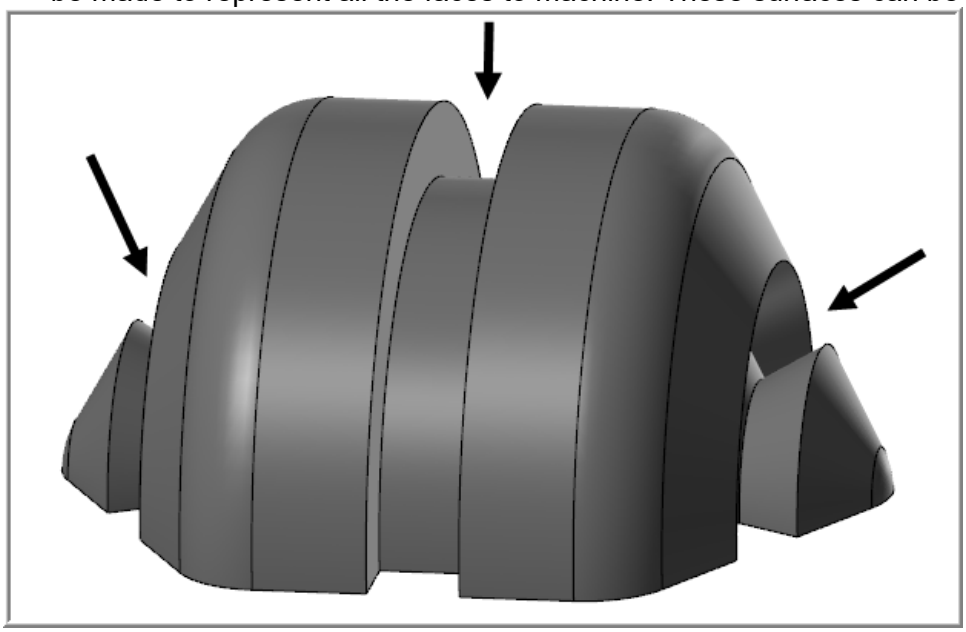
18. Both the Stock Model and Toolpath are dirty. **Regenerate both**.

➲ The adjusted toolpath. Backplot and make note of the new cycle time. 5m20s. The toolpath now cuts down to **-0.05** as well.

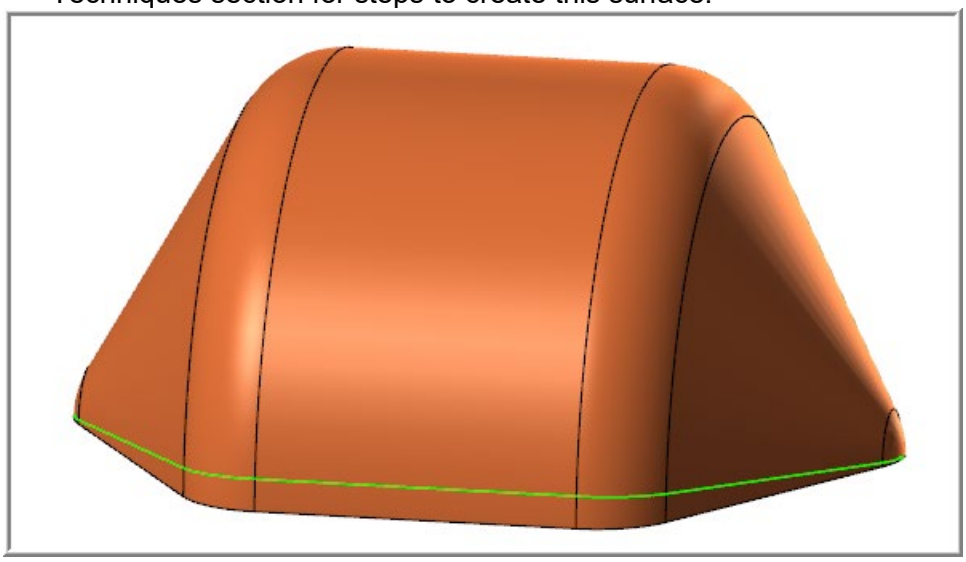


TASK 5:
FINISH THE SURFACE WITH FLOW 5 AXIS

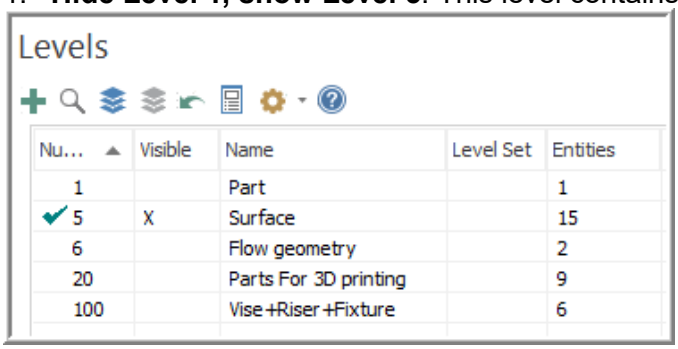
➲ The outer surface will be finished with Flow 5 Axis. The solid model could be used for this toolpath, but the toolpaths would end at the exact edge of the slots shown below. Extending the toolpath over these edges is ideal. One solution is to create surfaces to fill these voids and add them to the solid faces for toolpath geometry selection. Alternatively, surfaces can be made to represent all the faces to machine. These surfaces can be found on level 5.



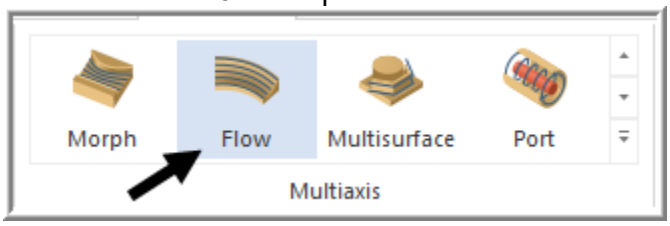
➲ These surfaces also extend slightly past the bottom of the part. Check the Tips and Techniques section for steps to create this surface.



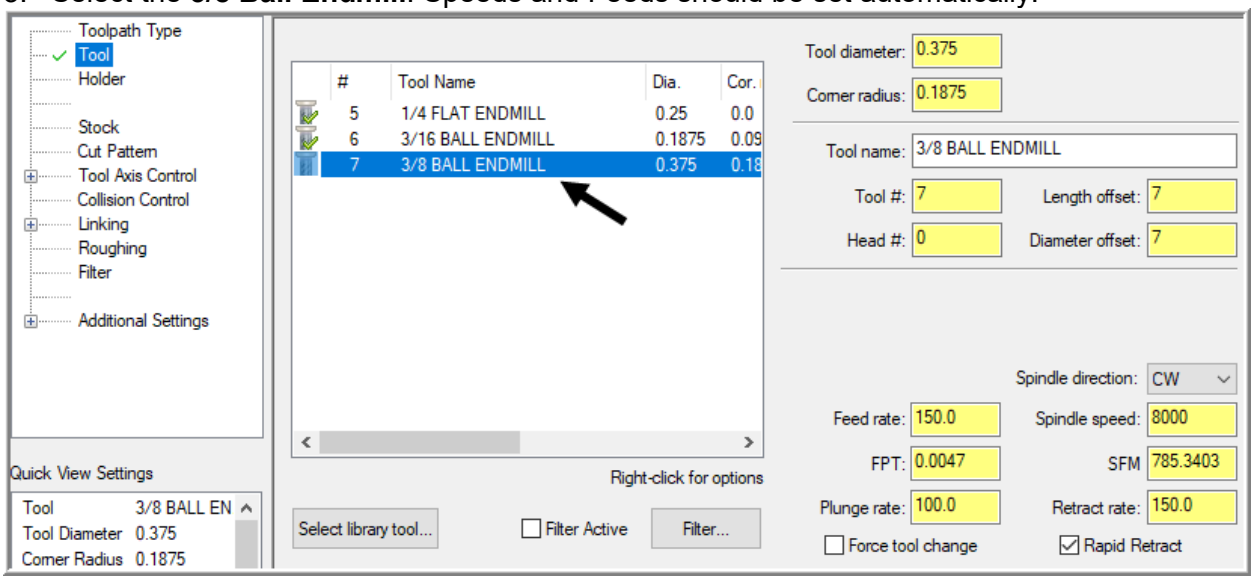
1. **Hide Level 1, show Level 5**. This level contains the surface for the Flow toolpath.



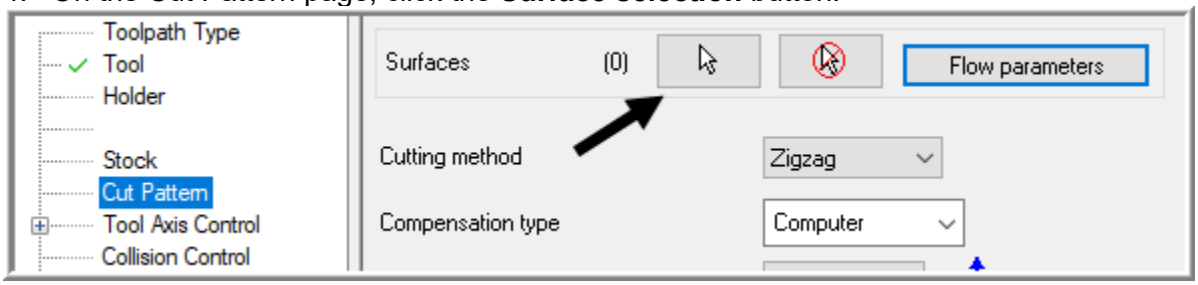
2. Select the **Flow** Toolpath from the Multiaxis toolpath gallery.



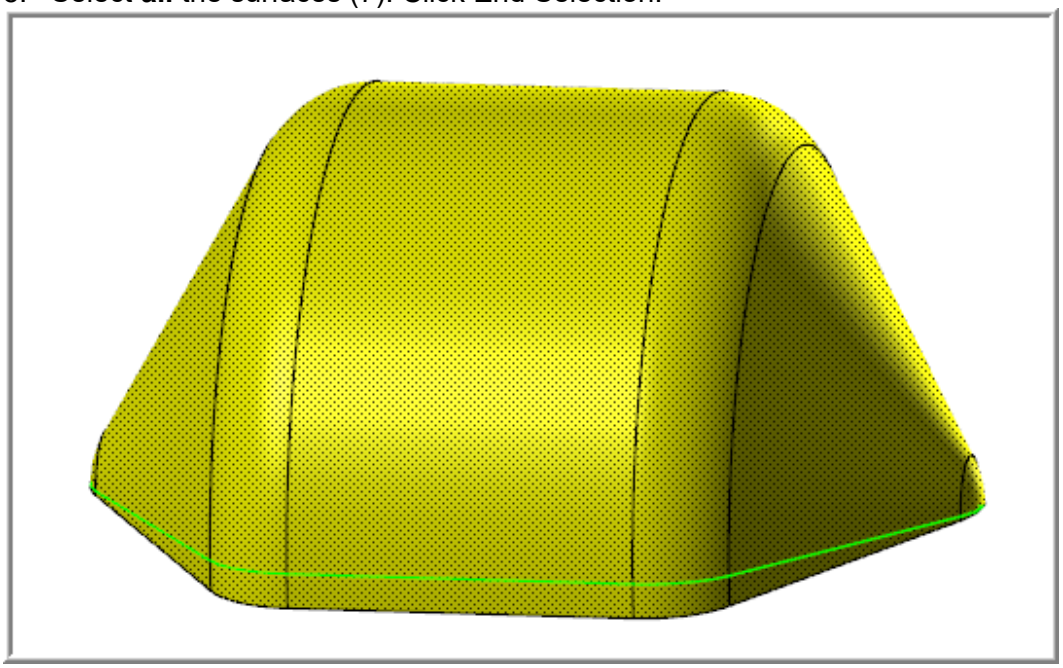
3. Select the **3/8 Ball Endmill**. Speeds and Feeds should be set automatically.



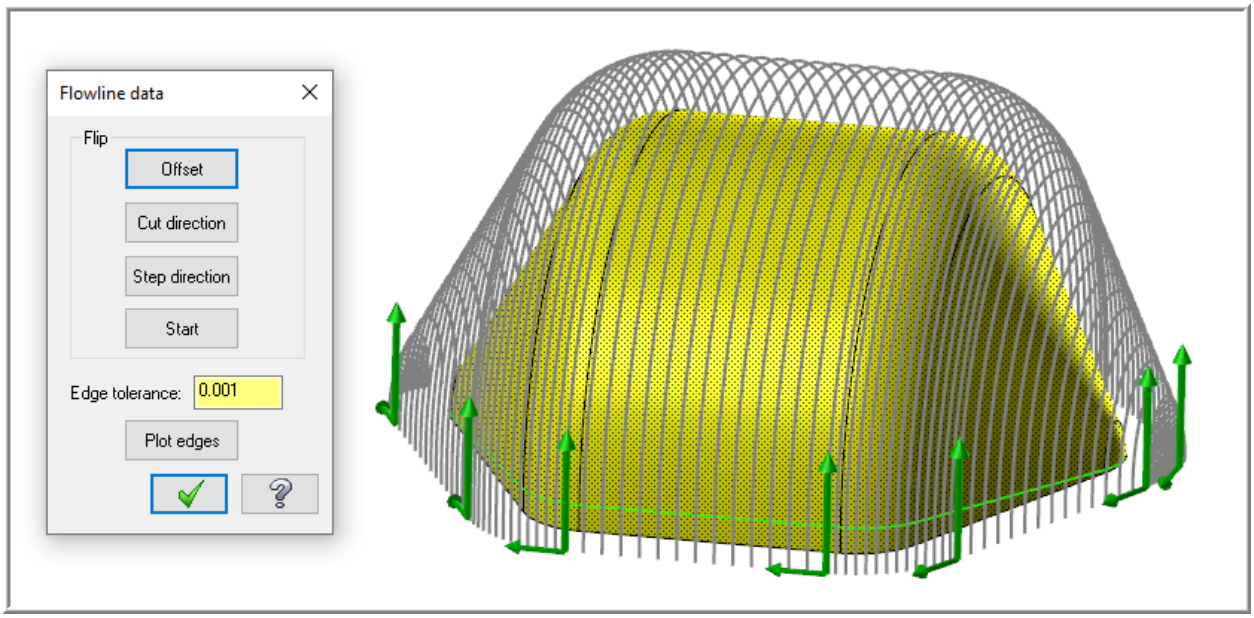
4. On the Cut Pattern page, click the **Surface selection** button.



5. Select **all** the surfaces (7). Click End Selection.

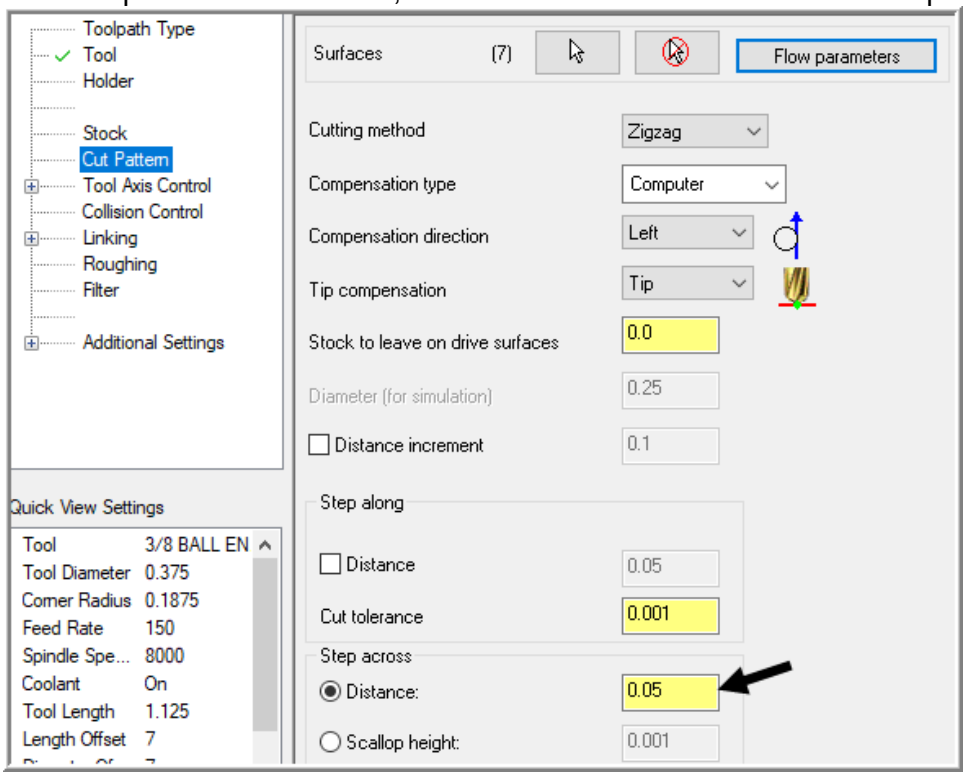


6. Next, the Flowline Data menu will appear. Click the Cut Direction button so the cut matches that shown below. Step Direction and Start can be of your choosing. If you do not see the preview on the outside of the part, click the Offset button to have the toolpath switch sides of the surface.

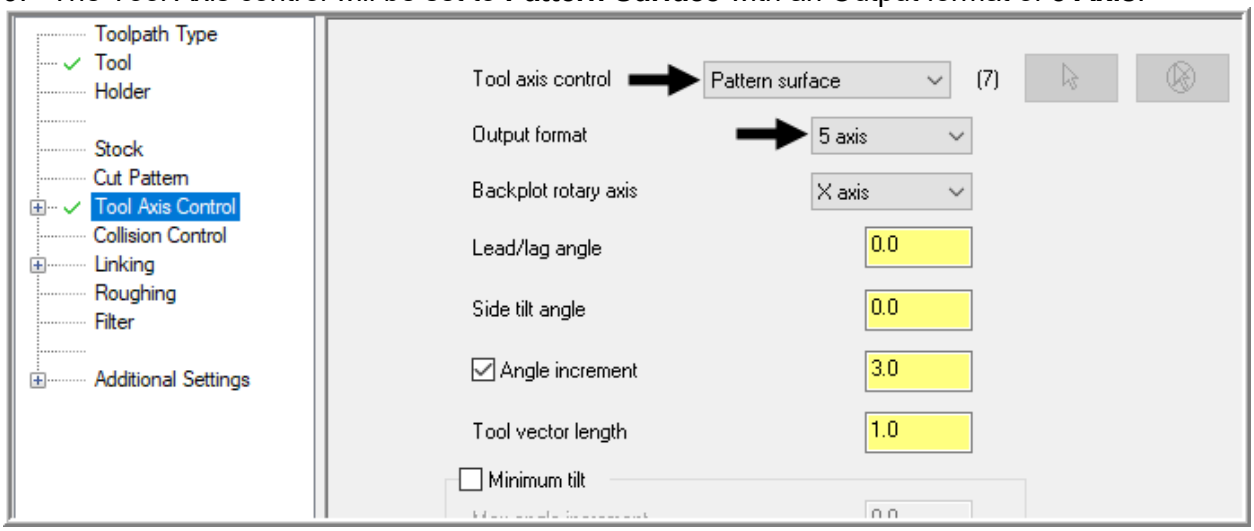


7. Click **Ok**.

8. The remaining settings on the Cut Pattern page will be left at the default setting. If you want an improved surface finish, reduce the value for Distance under Step Across..



9. The Tool Axis control will be set to **Pattern Surface** with an Output format of **5 Axis**.



Output format
Select 4-axis or 5-axis from the drop-down list.

4-axis: Allows one plane of rotation selected under Rotary axis.
5-axis: Allows tool axis rotation in any plane.

Tool axis control
Use the drop-down list to select the tool axis control method. Click Select to return to the graphics window to select appropriate entities. The number of entities is displayed to the right of the Select button.

Lines: Aligns the tool axis along the selected lines. The tool axis will be interpolated for areas between the selected lines. Select the lines in such a way that the chaining arrow is pointing towards the tool spindle.

Pattern surface: Keeps the tool axis normal to a selected surface. Pattern surface is the only option available for 3-axis output. For 3-axis output, Mastercam projects the curves onto the tool axis surfaces. The projected curves become the tool contact positions.
Plane: Keeps the tool axis normal to a selected plane.

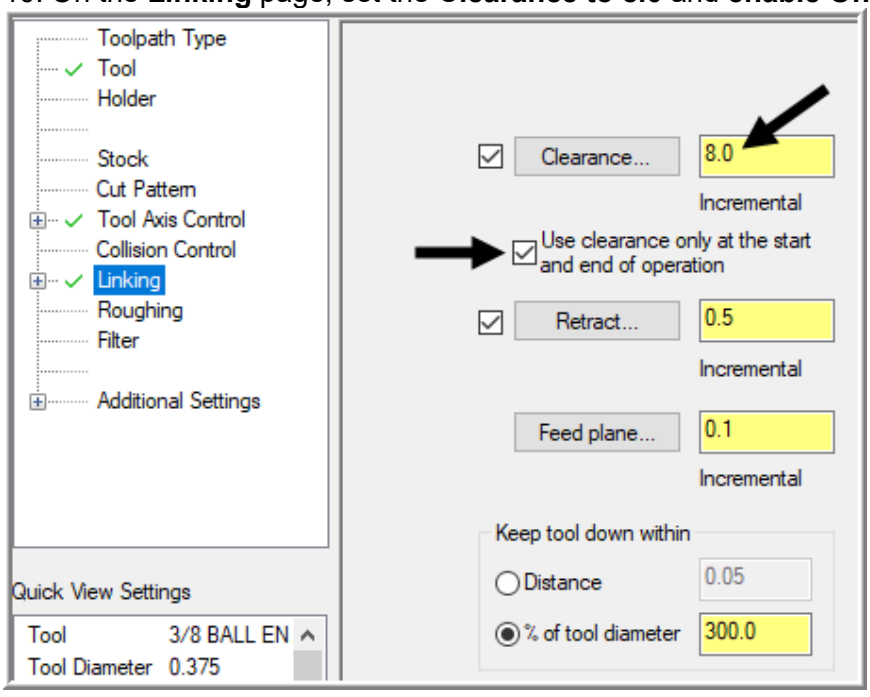
From point: Limits the tool axis to originate from a selected point.

To point: Limits the tool axis to terminate to a selected point.

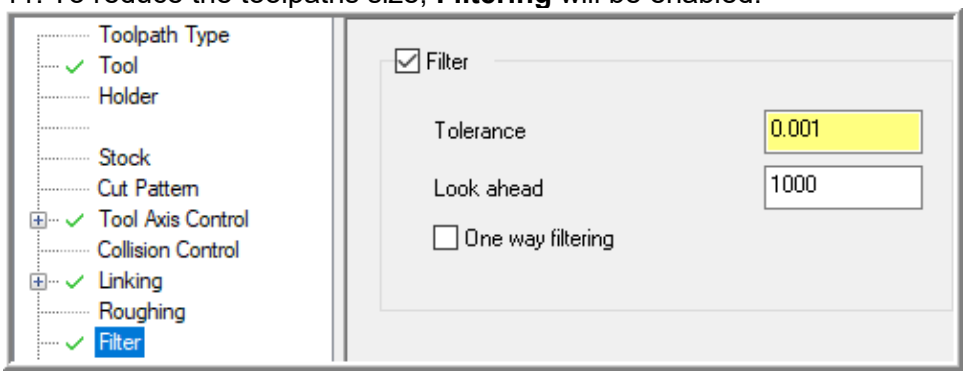
Chain: Aligns the tool axis along a line, arc, spline, or chained geometry.

Boundary: Aligns the tool axis within or on a closed boundary. If the cut pattern surface normal is within the boundary, the tool axis stays aligned with the cut pattern surface normal.

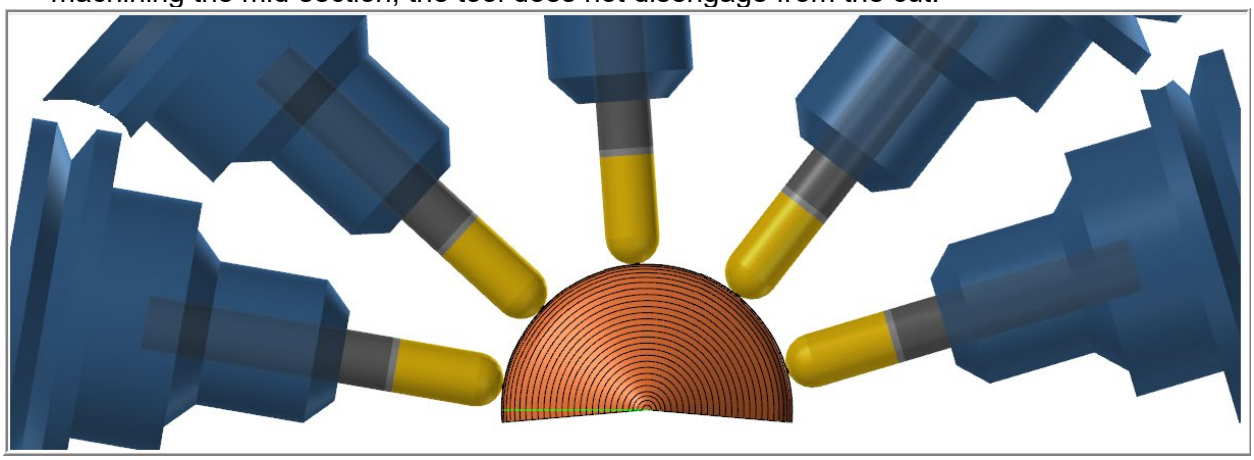
10. On the **Linking** page, set the **Clearance to 8.0** and **enable Only at the Start and End**.



11. To reduce the toolpaths size, **Filtering** will be enabled.

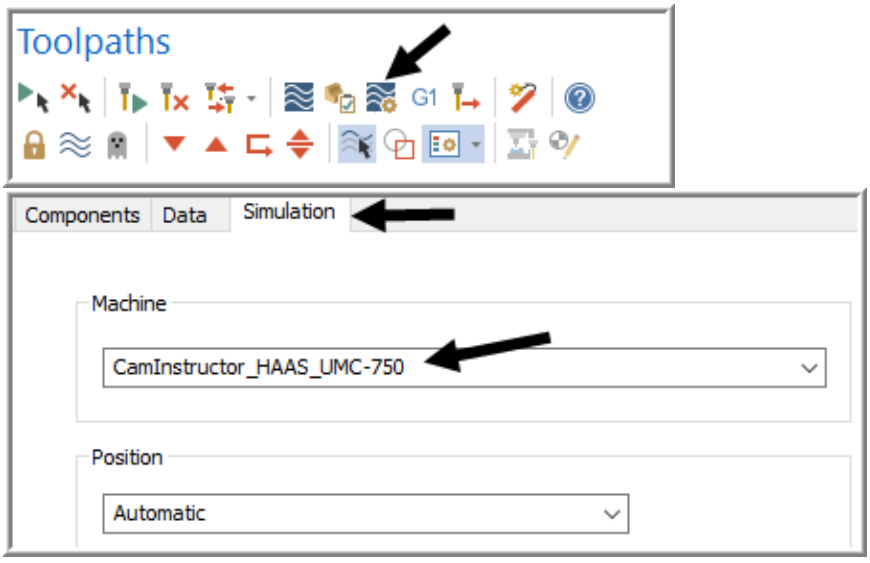


12. Click **Ok** to complete the operation.
13. **Backplot** the toolpath. Notice, particularly about half-way through the toolpath when machining the mid-section, the tool does not disengage from the cut.

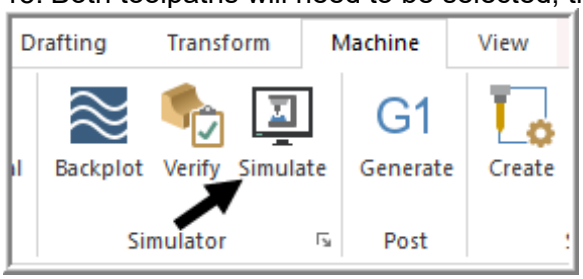


➲ Next, confirm this same motion in **Simulation** which uses the machines limits to determine toolpath motion.

14. Check that the correct machine is selected.



15. Both toolpaths will need to be selected, then launch Simulation from the Machine tab.



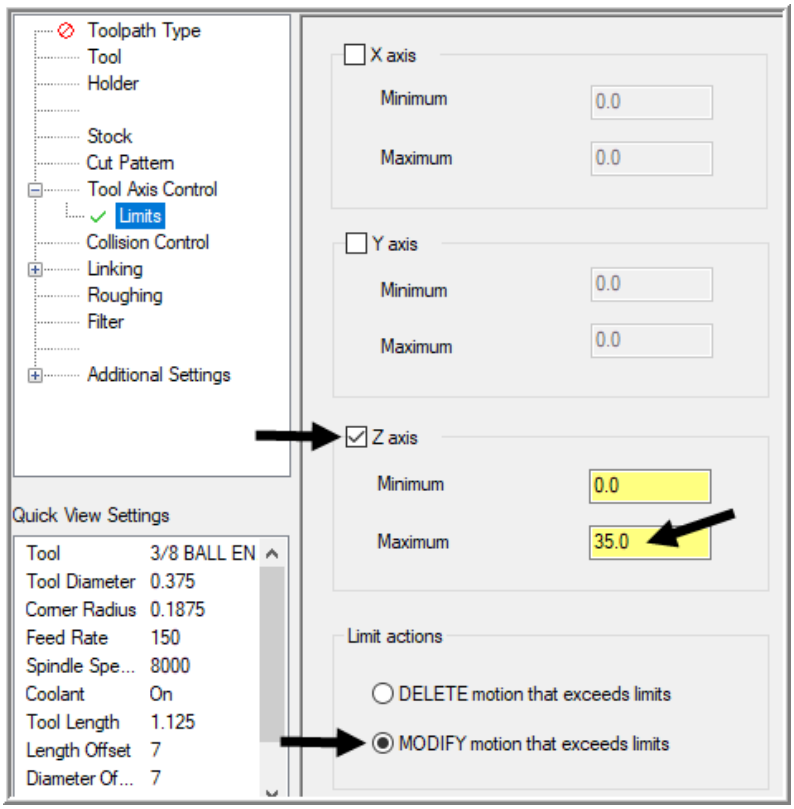
16. The first operation can be skipped over by clicking **the Next Operation** button.

- Once the toolpath gets to the middle section of this part, you will see a retract, then the table (C axis) will rotate 180 degrees and then tool will re-engage the cut. This will repeat until the mid-section is complete. You may need to single step through the toolpath to see the retracting happen.
- This is happening due to the machine limits in the B axis. The Haas UMC-750 is limited to a B axis rotation of +110 and -35. To cut this feature in one complete motion a minimum of +90 and -90 would be needed. Alternatively, we can try to limit the toolpath to stay inside of the machine's limits.

17. Close the Simulation
18. Open the **Flow 5 Axis** toolpath by clicking on the **Parameters** folder
19. Expand the **Tool Axis Control** to show the **Limits** page.

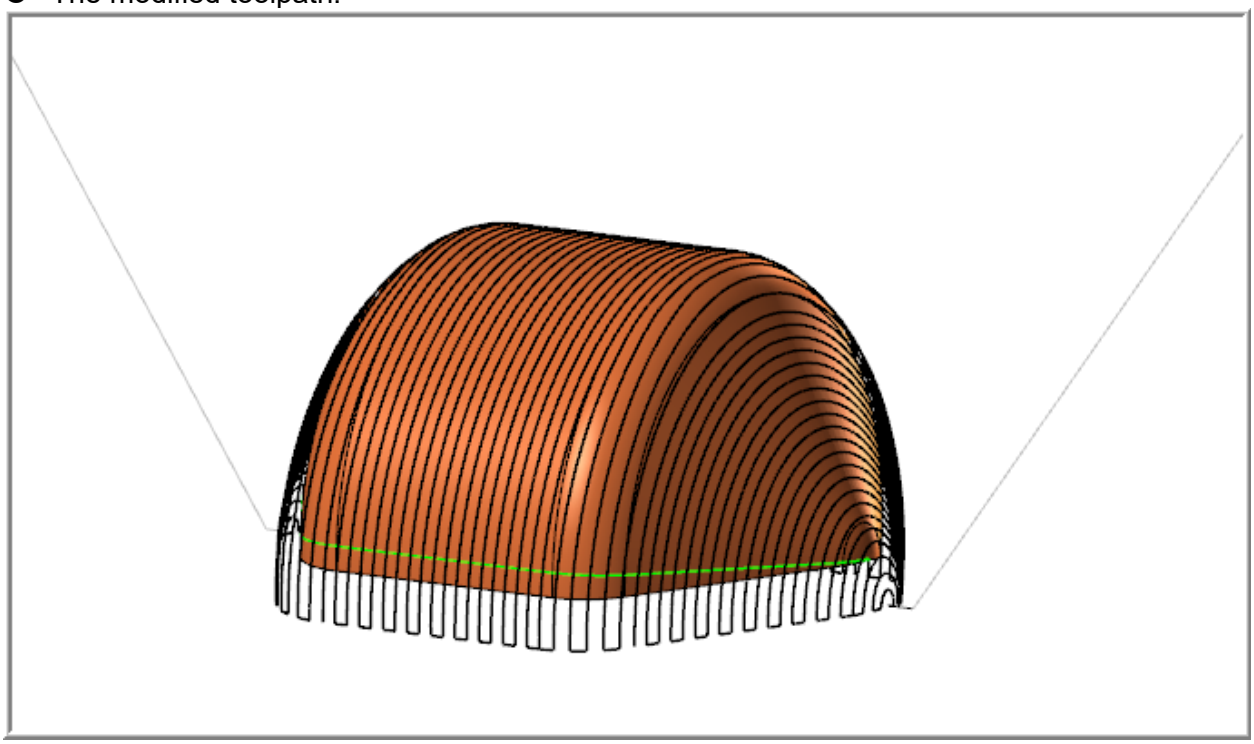
    Enable Z Axis limits and set the **Maximum to 35** degrees.

    In the Limit actions, set this to **MODIFY the motion** that exceeds. This will try and keep the portion of toolpath that exceeds 35 degrees but keep the B rotation at 35. DELETE would remove any part of the toolpath that needs to rotate beyond 35.

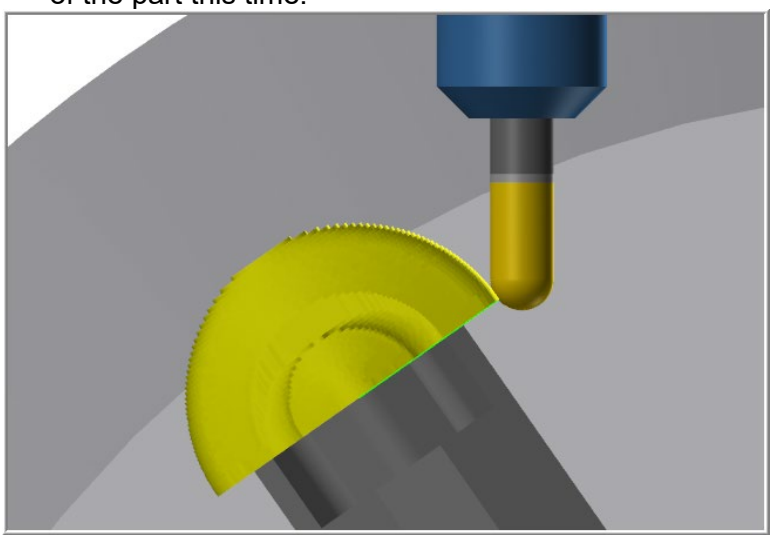


20. Click **Ok**, and the **Regenerate** the dirty operation.

➲ The modified toolpath.

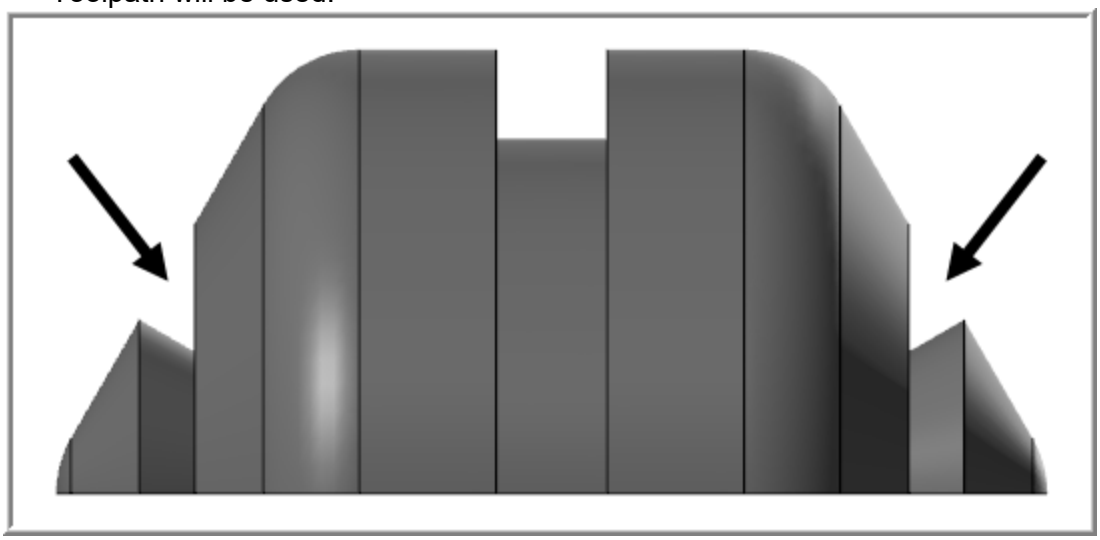


➲ Running both toolpaths through **Simulate** will show no retracts happening in the mid-section of the part this time.

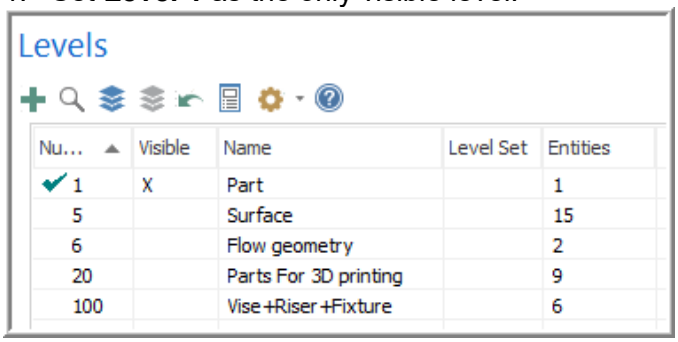


TASK 6:
MACHINE THE END SLOTS WITH CURVE 5 AXIS

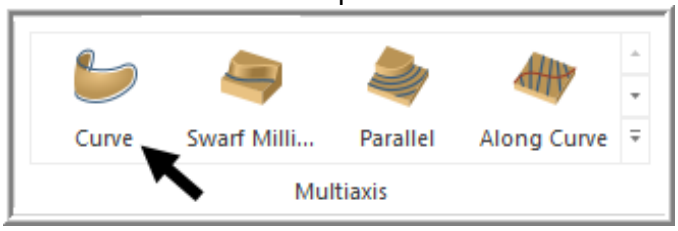
➲ The two slots at the ends of the part will be machined next. Machining will be done with a flat endmill. This tool must be kept 'normal' to the floor surface of the slot. The 5 Axis Curve Toolpath will be used.



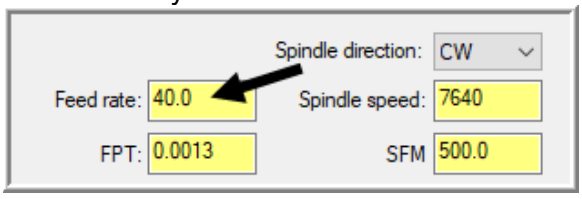
1. Set **Level 1** as the only visible level.



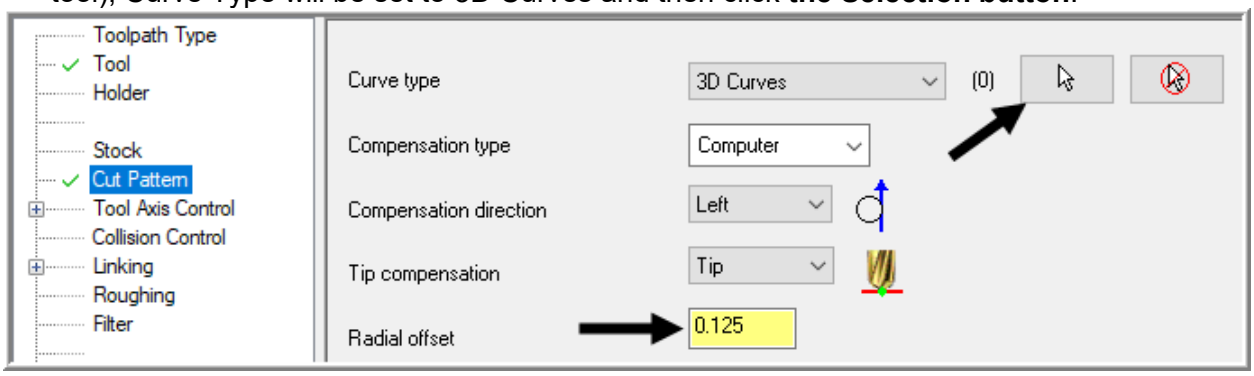
2. Select the **Curve** Toolpath from the Multiaxis toolpath gallery.



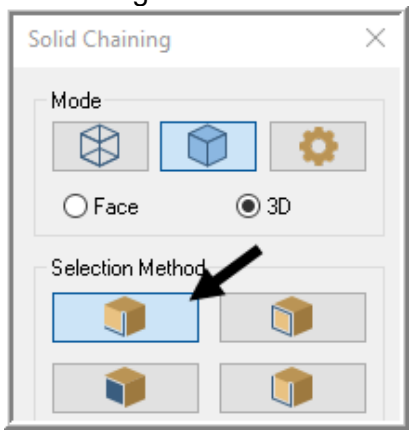
3. Select the **1/4 Flat Endmill** from the tool list.
4. Reduce the feedrate to 40 ipm. This path will be cutting a full slot so reducing the feedrate is necessary.

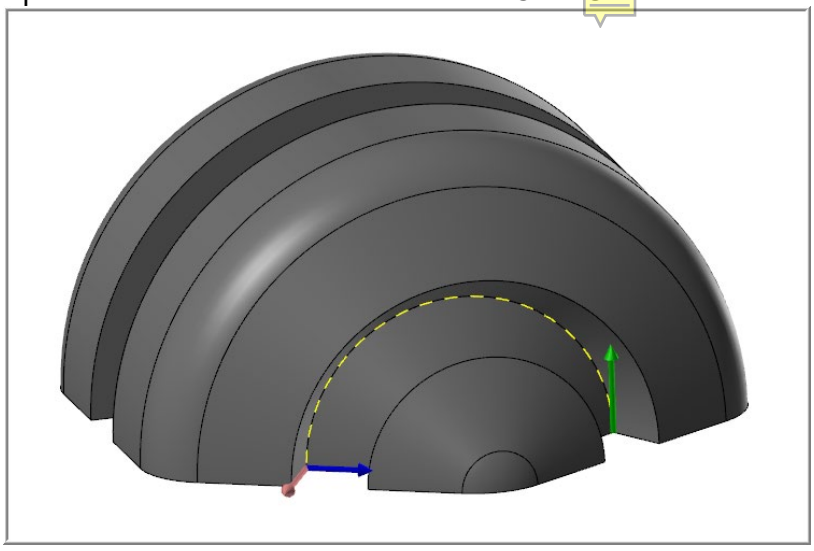


5. On the Cut Pattern page, set the Radial offset to **0.125** (half the diameter of the selected tool), Curve Type will be set to 3D Curves and then click **the Selection button**.

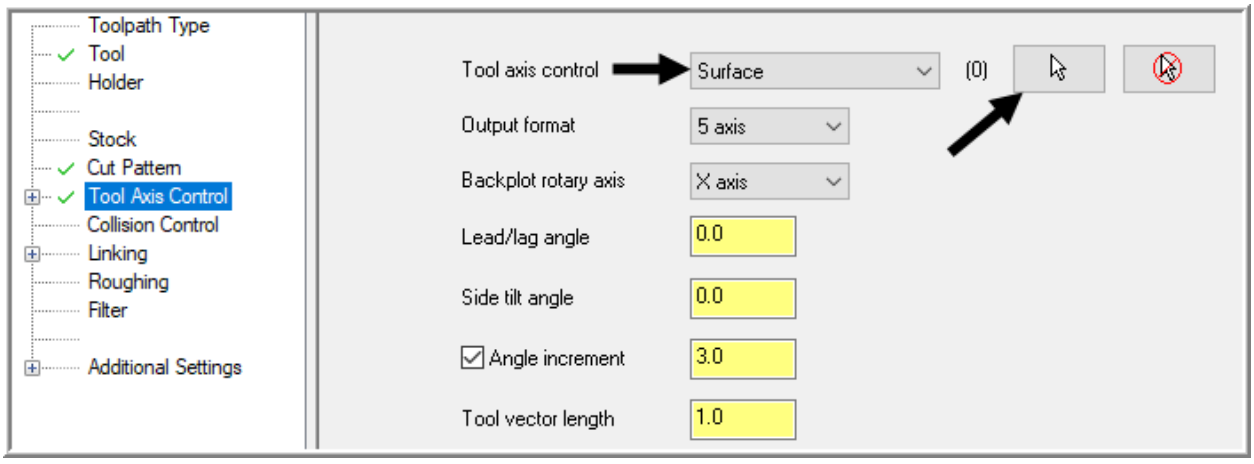


6. The Chaining Manager should be in the **Solid Mode**. Enable only **Edges** selection. Select the edge shown. Direction is important. Reverse the chain if needed. Click **Ok**.

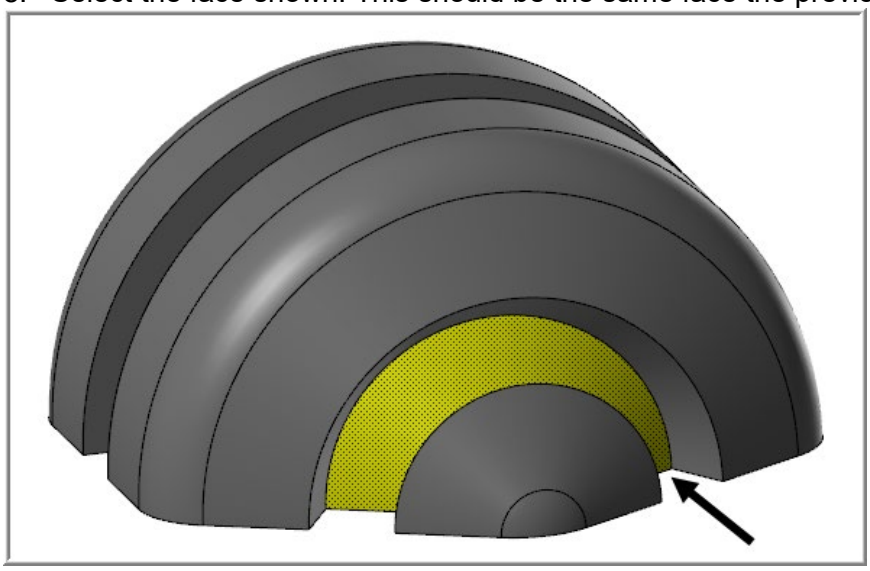




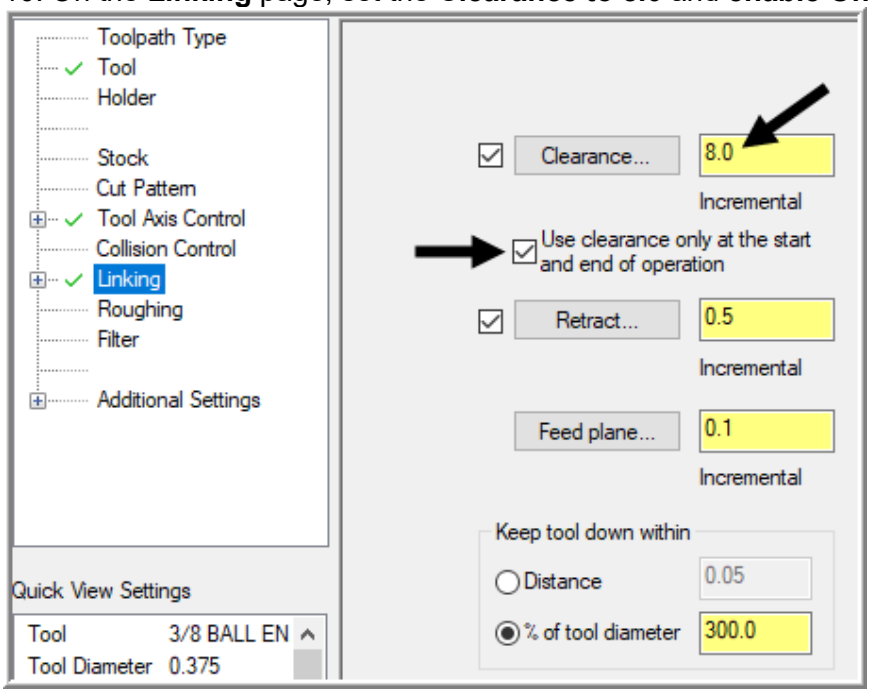
7. On the Tool Axis Control page, set the Tool Axis Control to **Surface**. Then click on the **selection button**.



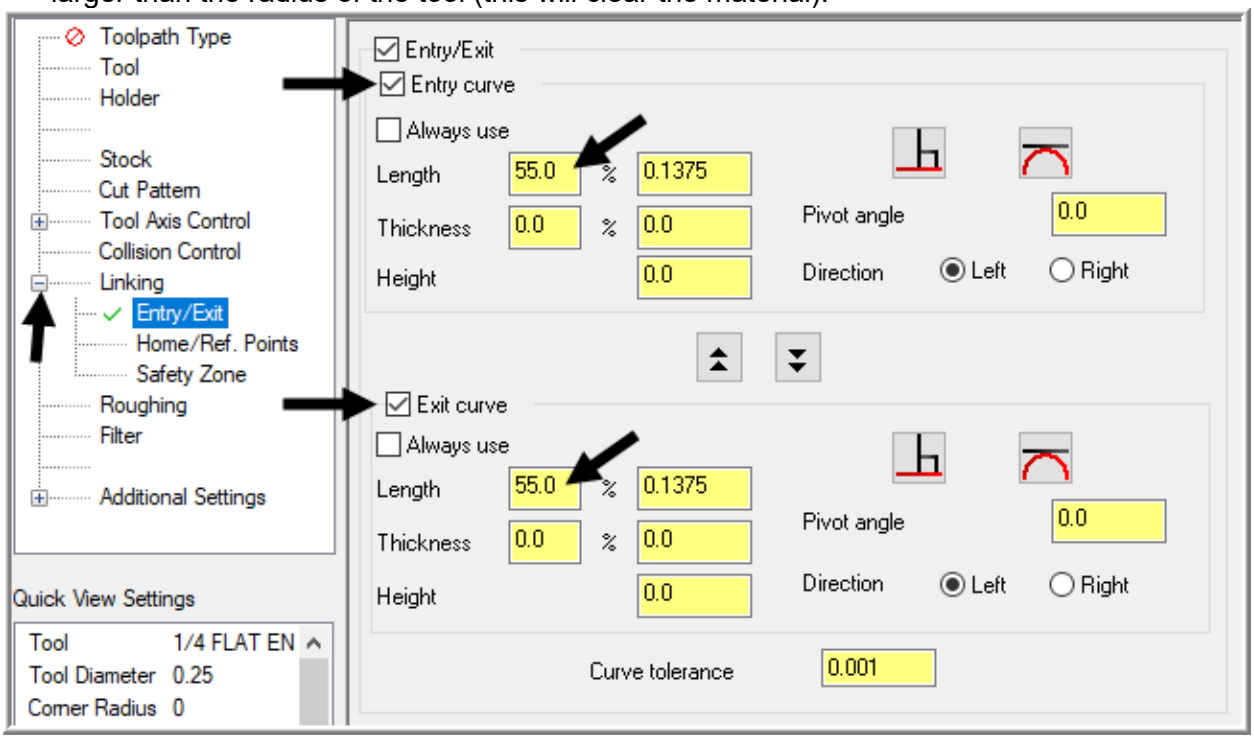
8. Select the face shown. This should be the same face the previous edge was selected from.



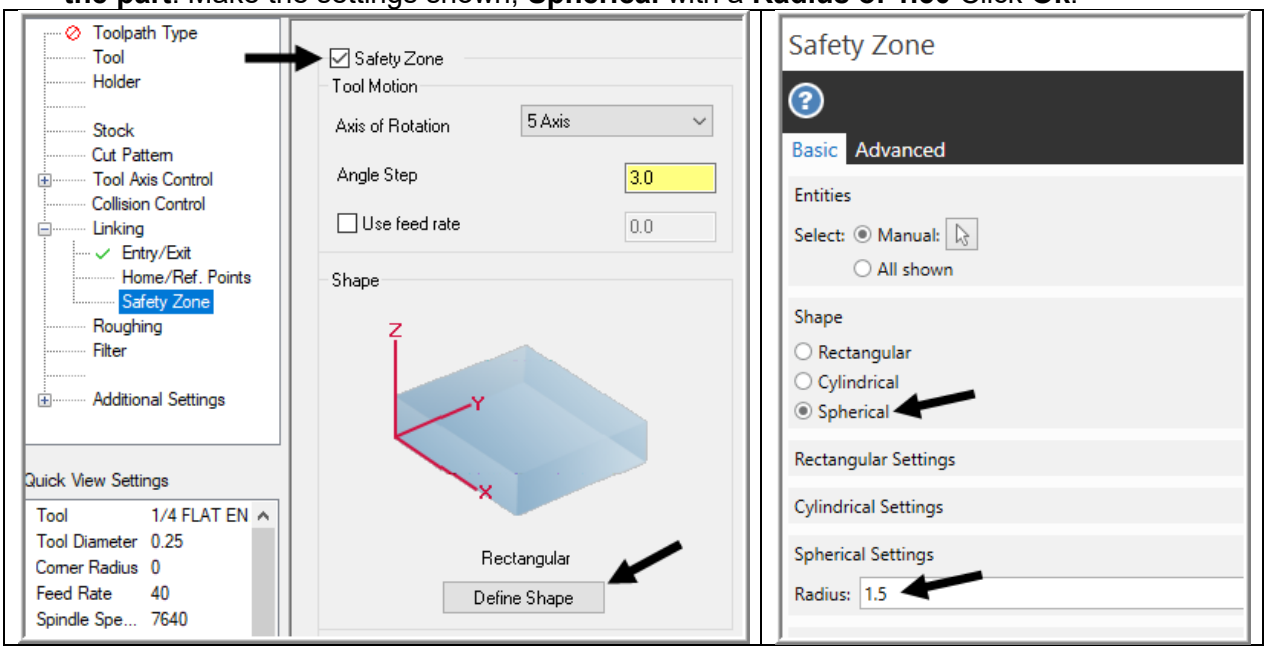
9. Click End Selection.
10. On the **Linking** page, set the **Clearance to 8.0** and **enable Only at the Start and End**.



11. Expand the **Linking** settings and choose **Entry/Exit**. Enable both **Entry** and **Exit Curve** with a Length of **55%**. This will provide a straight lead into and out of the cut that is slightly larger than the radius of the tool (this will clear the material).



12. **Enable Safety Zone**. With the depth cuts being used, controlling where the retract and repositioning moves occur is suggested. Click the **Defined Shape** button and then **select the part**. Make the settings shown, **Spherical** with a **Radius of 1.50** Click **Ok**.

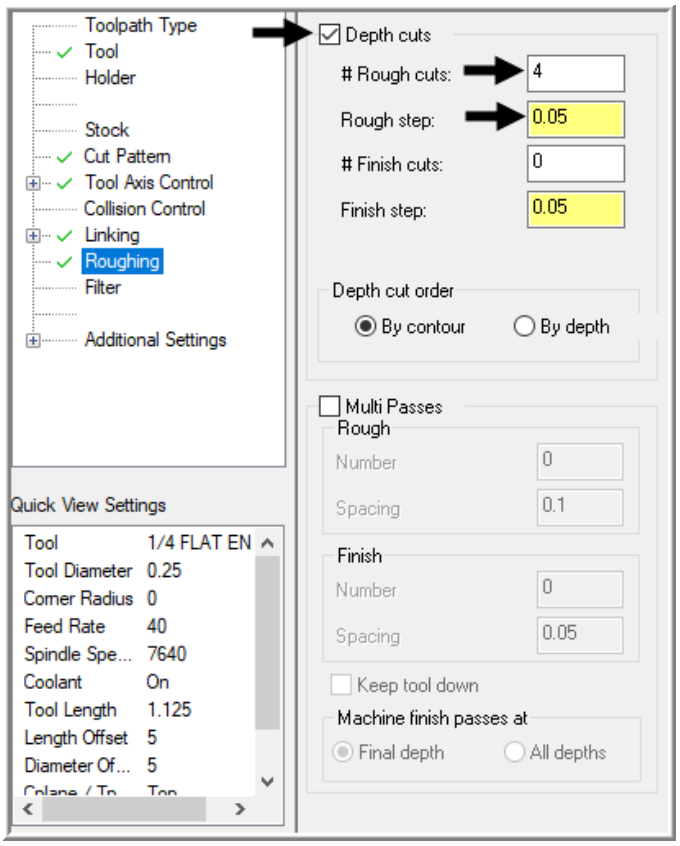


13. On the Roughing page **enable Depth Cuts** and set **the # Rough Cuts to 4** and the **Rough Step to 0.05**.

    Knowing the slot is 0.20 deep, setting 4 cuts of 0.05 each result in 4 even depth cuts starting from the top of stock.

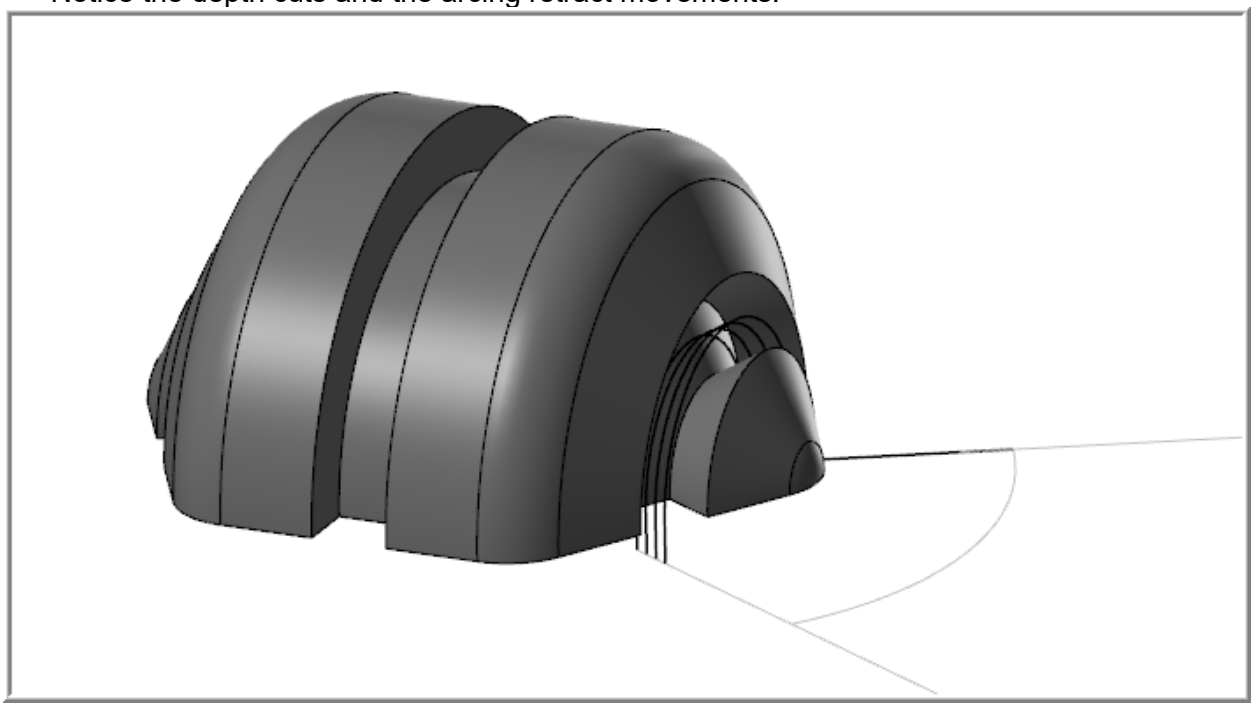
    If stepovers were needed in addition to depth cuts, Multi Passes can be enabled.

    This scenario, the endmill is the same width as the slot so no Multi Passes are needed.

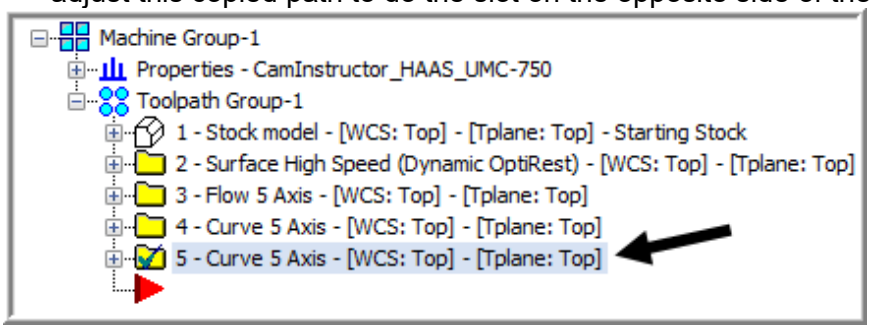


14. Click **Ok** to complete the toolpath.

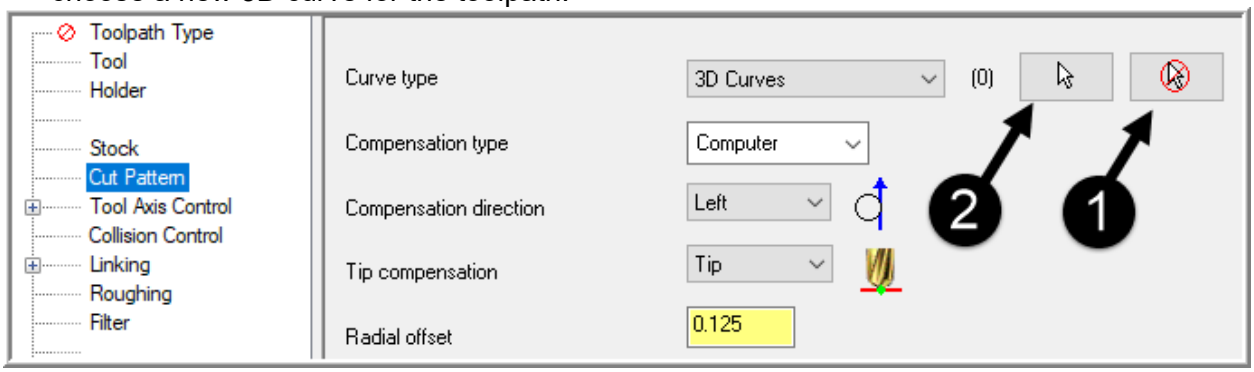
➲ The completed toolpath. (Wireframe view, fixtures hidden, toolpaths black only for clarity) Notice the depth cuts and the arcing retract movements.



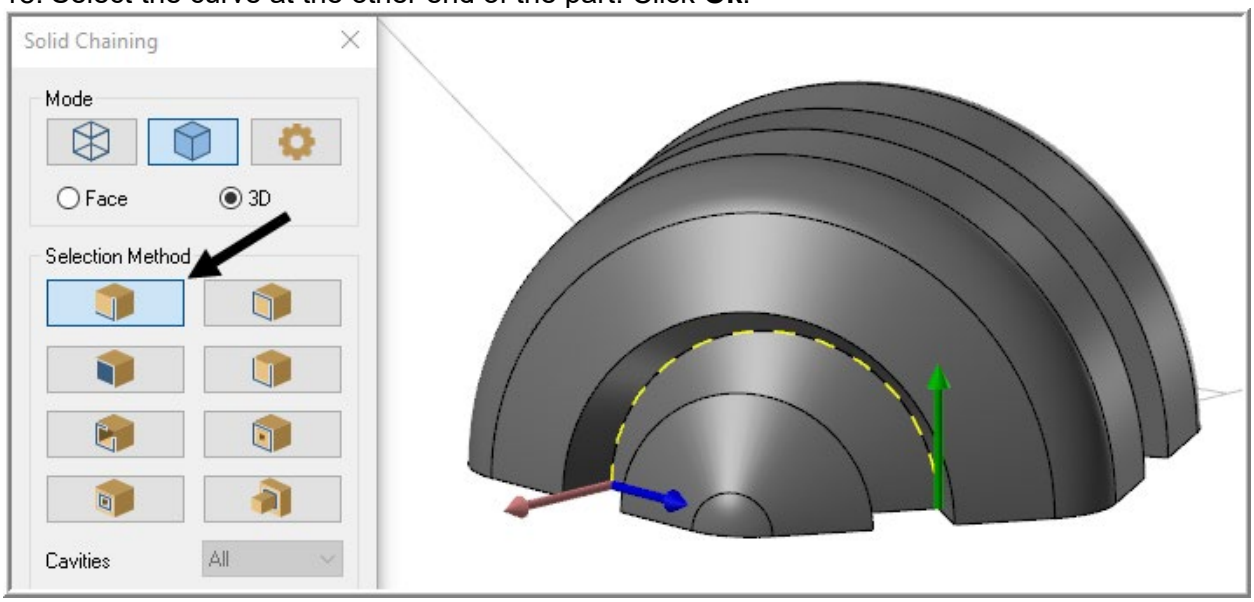
15. Copy and Paste the Curve 5 Axis toolpath. This will duplicate the toolpath just made. We will adjust this copied path to do the slot on the opposite side of the part.



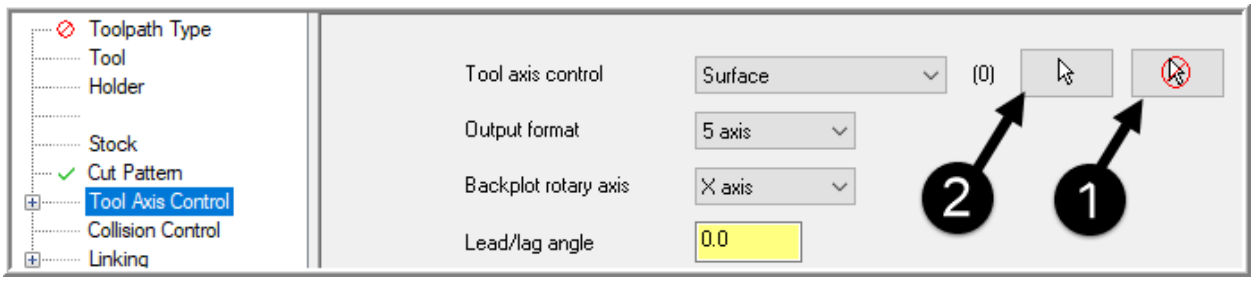
16. Open the copied operation by click on its Parameter Folder.
17. On the Cut Pattern page, **clear the current selection**, then click the **selection button** to choose a new 3D curve for the toolpath.



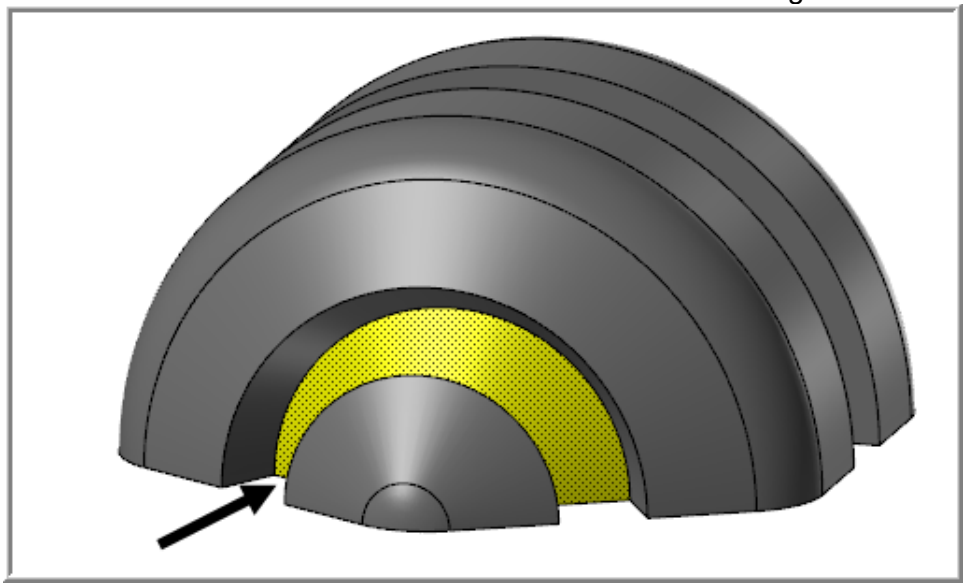
18. Select the curve at the other end of the part. Click **Ok**.



19. On the **Tool Axis Control** page, clear the current selection, then click the selection button to choose a new surface.

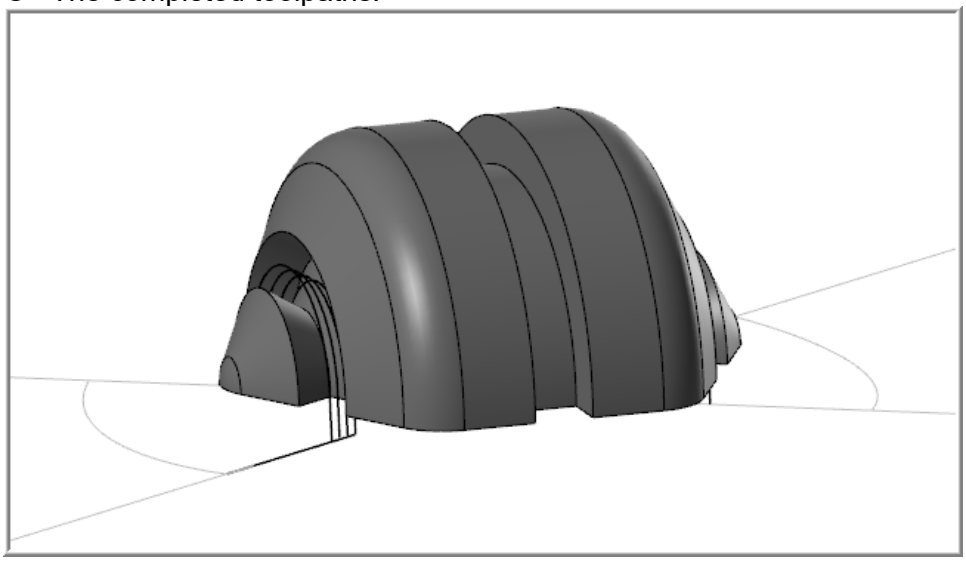


20. Select the face shown. It should be the same face the edge selection was done from.



21. Click **End Selection**.
22. Click **Ok** to complete the toolpath edits. Regenerate the operation.

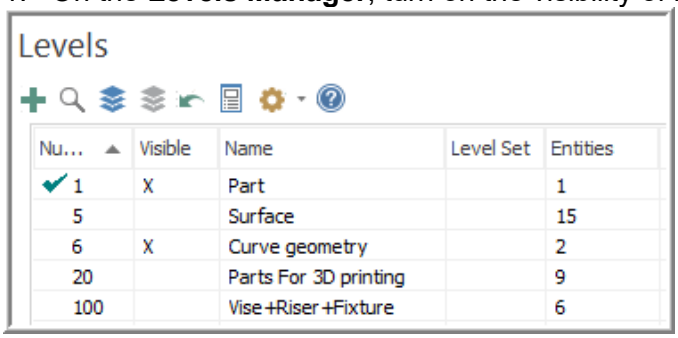
➲ The completed toolpaths.



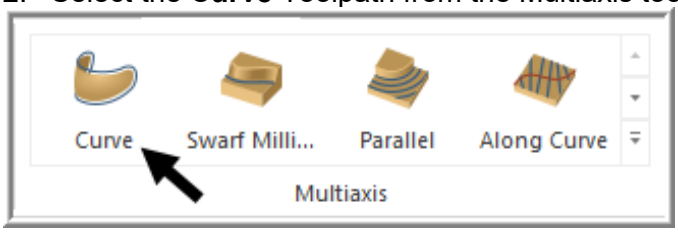
TASK 7:
MACHINE THE MIDDLE SLOT

- The middle slot of the part will use the same toolpath as the previous slots, but a slightly different strategy will have to be used due to the machine's rotation limits in the B axis.
- The slot will be cut in two 90-degree increments, basically halfway each time starting from the outside.
- Chaining this type of motion will require an additional step. Two pieces of wireframe geometry have been created on level 6 for this toolpath. For instructions on how to make this geometry, see the Tips and Techniques section.

1. On the **Levels Manager**, turn on the visibility of level 6.

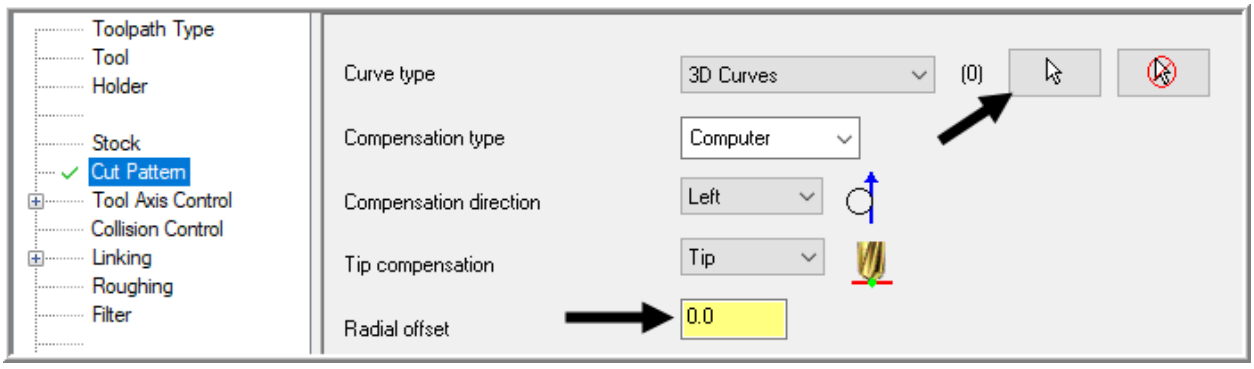


2. Select the **Curve** Toolpath from the Multiaxis toolpath gallery.

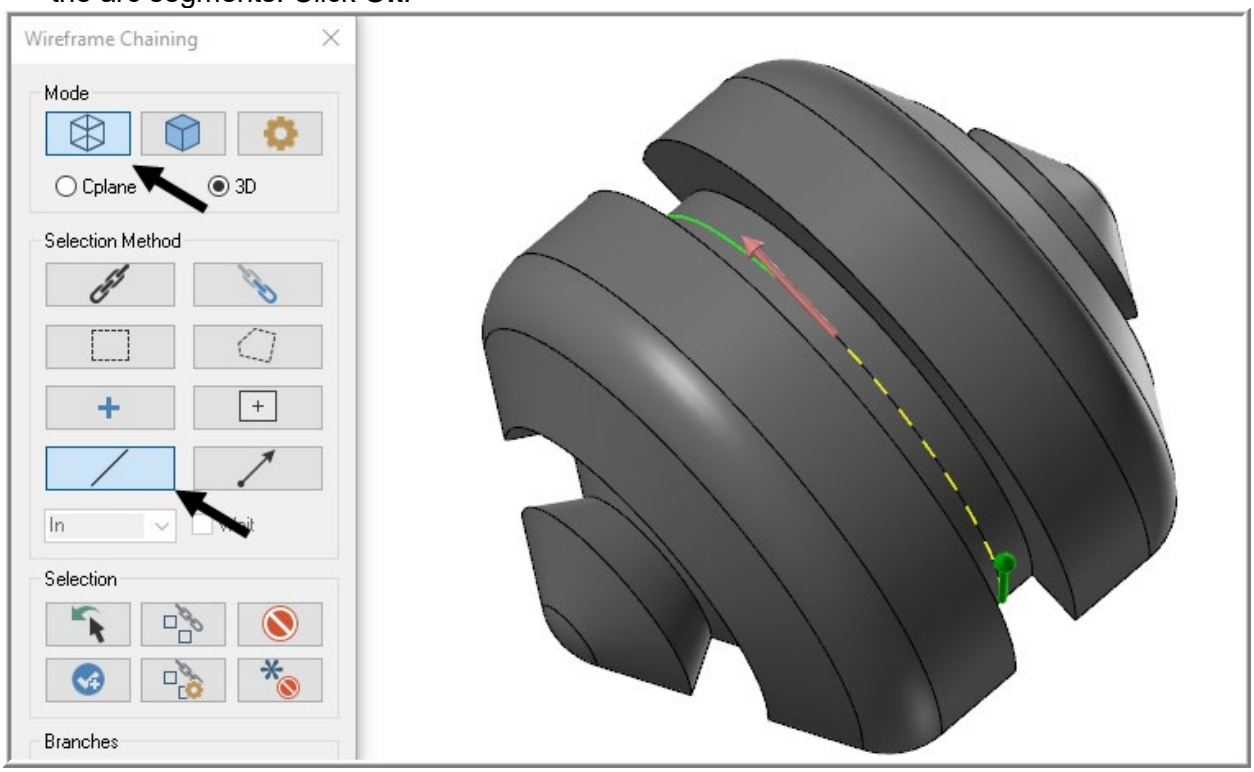


- Selection of a toolpath that was already made in the file will populate the values with those used in the previous instance, except for any geometry selections.

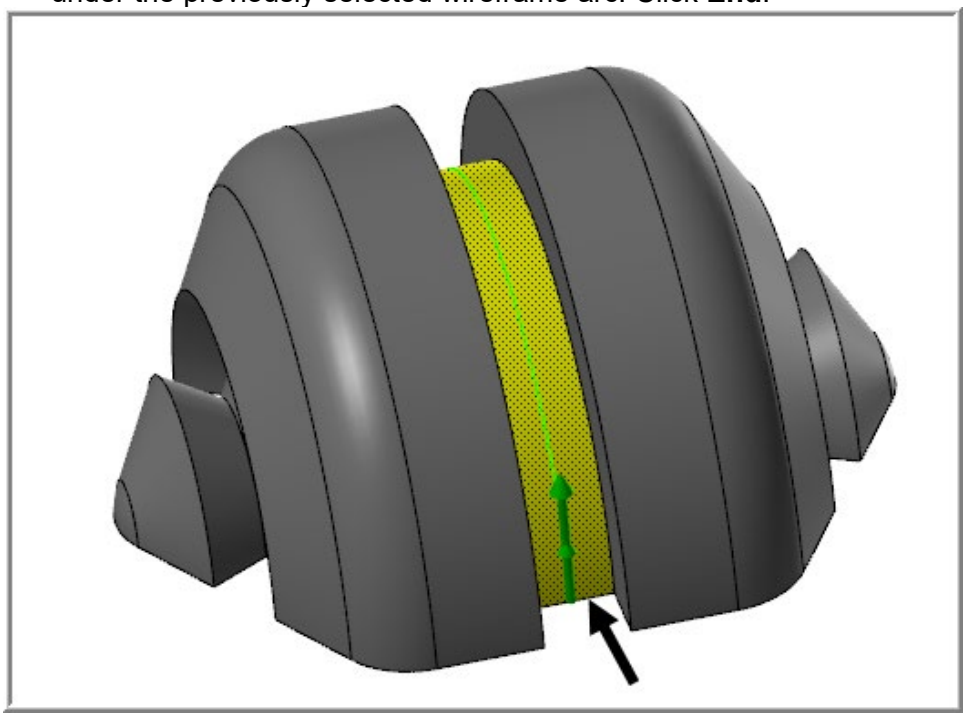
3. The same tool will be used, **1/4 Flat Endmill**, with the Feedrate reduced to 40 ipm.
4. The Cut Pattern page will use some different settings. Set the **Radial Offset value to 0**. Then click the Curve **Selection button**.



5. Switch to the **Wireframe** Selection Mode with the **Single** selection method. Select **one** of the arc segments. Click **Ok**.

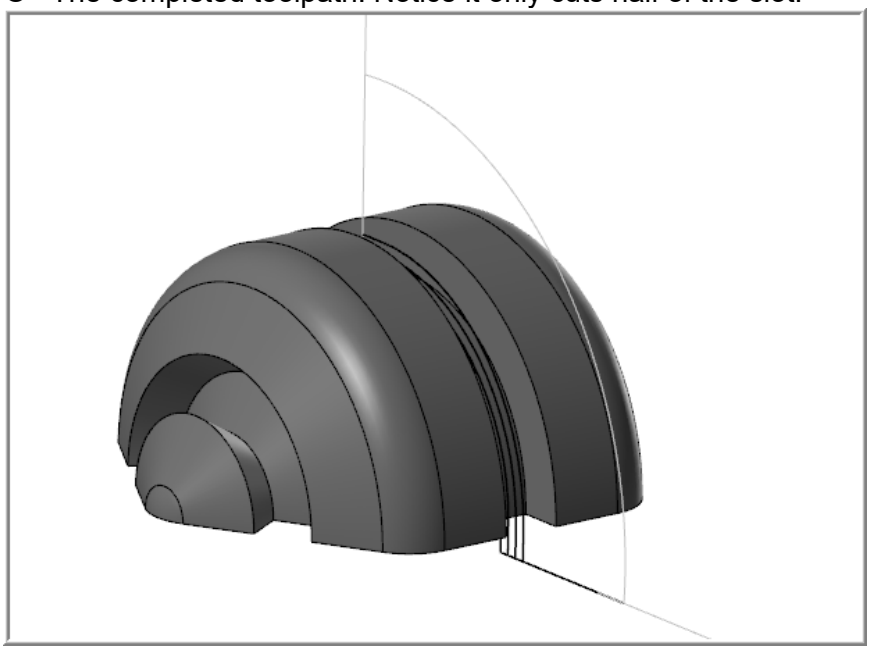


6. The **Tool Axis Control** will be left to **Surface**. Click the selection button and select the face under the previously selected wireframe arc. Click **End**.

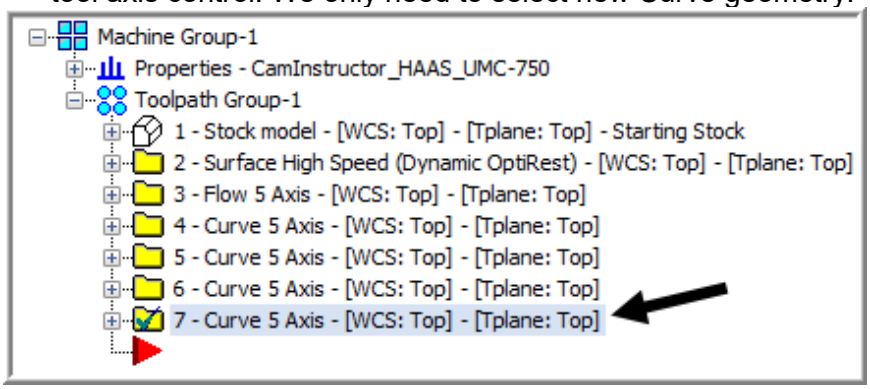


7. The remaining settings will be populated from the settings used in the previous Curve Toolpath. No further changes are needed. Click **Ok** to complete the toolpath.

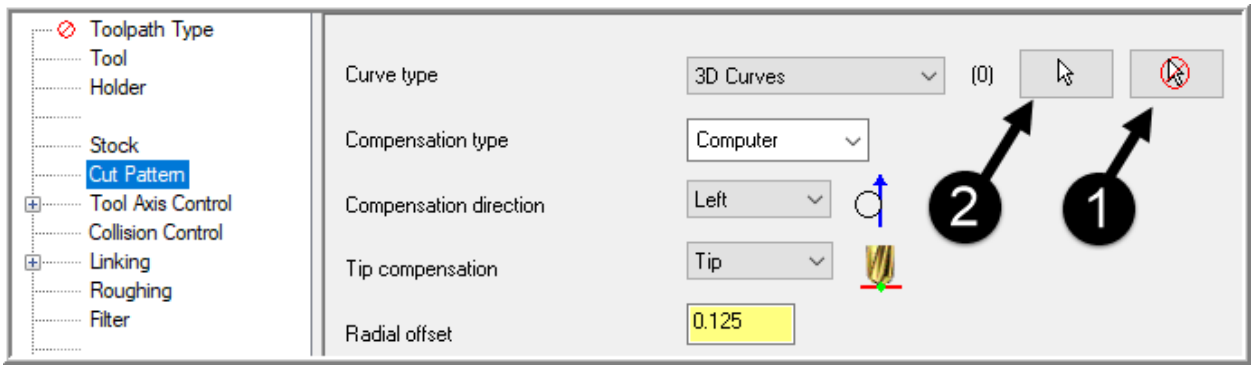
➲ The completed toolpath. Notice it only cuts half of the slot.



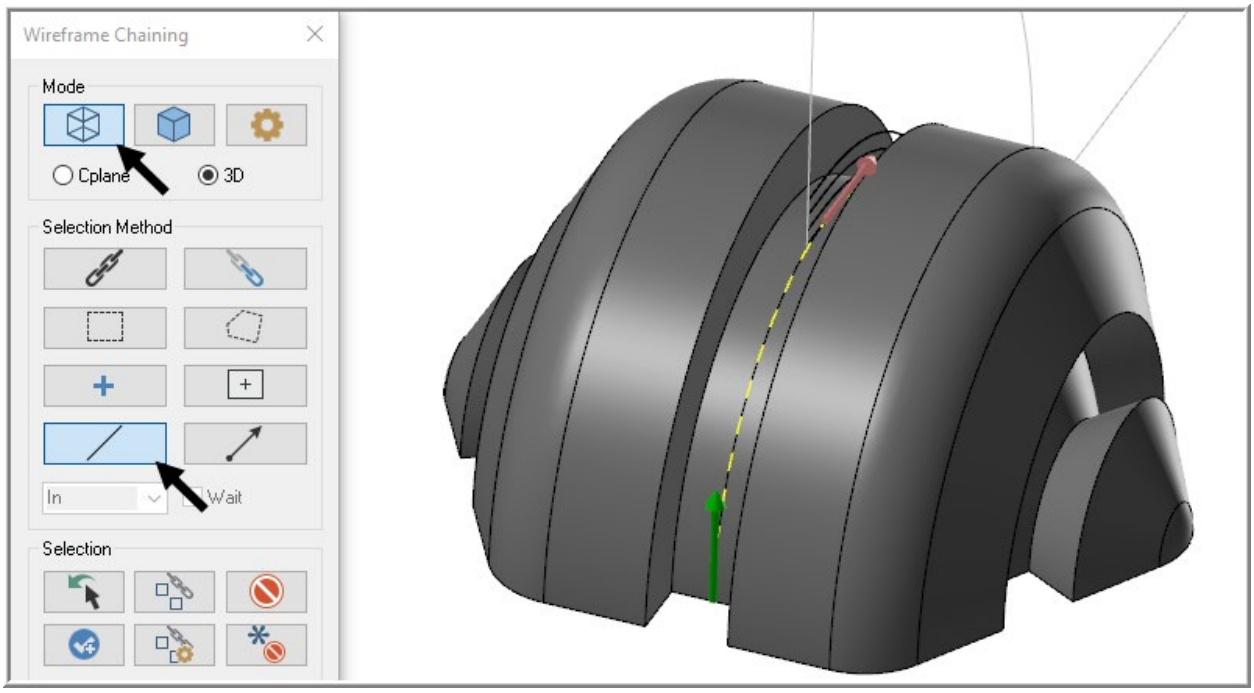
8. Copy and Paste the **Curve** toolpath just made. This will keep all settings and all selected geometry. By copying the toolpath, we save having to reselect the under lying face for the tool axis control. We only need to select new Curve geometry.



9. On the new toolpaths **Cut Pattern** page, clear the selected curve and then click the selection button to choose a new curve.

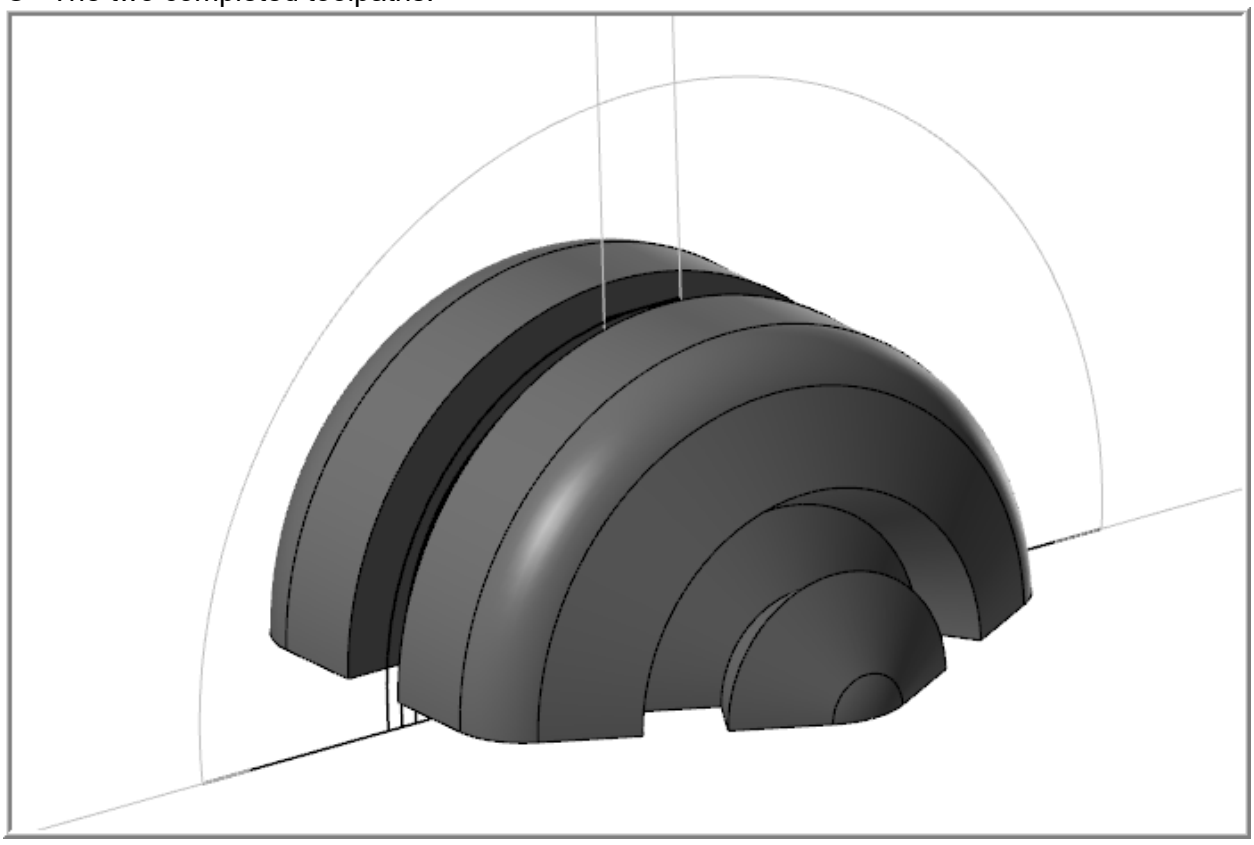


10. Ensure you are still in the Wireframe Mode and in the Single selectin method, choose the other half of the arc.



11. Click **Ok** to complete the toolpath.
12. **Regenerate** the toolpath.

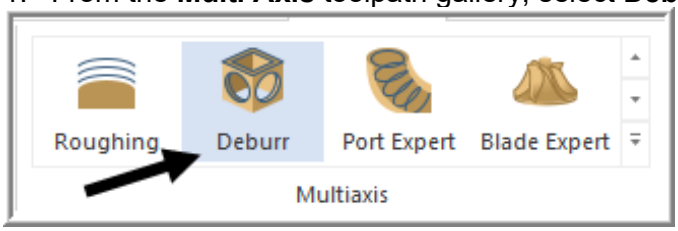
➲ The two completed toolpaths.



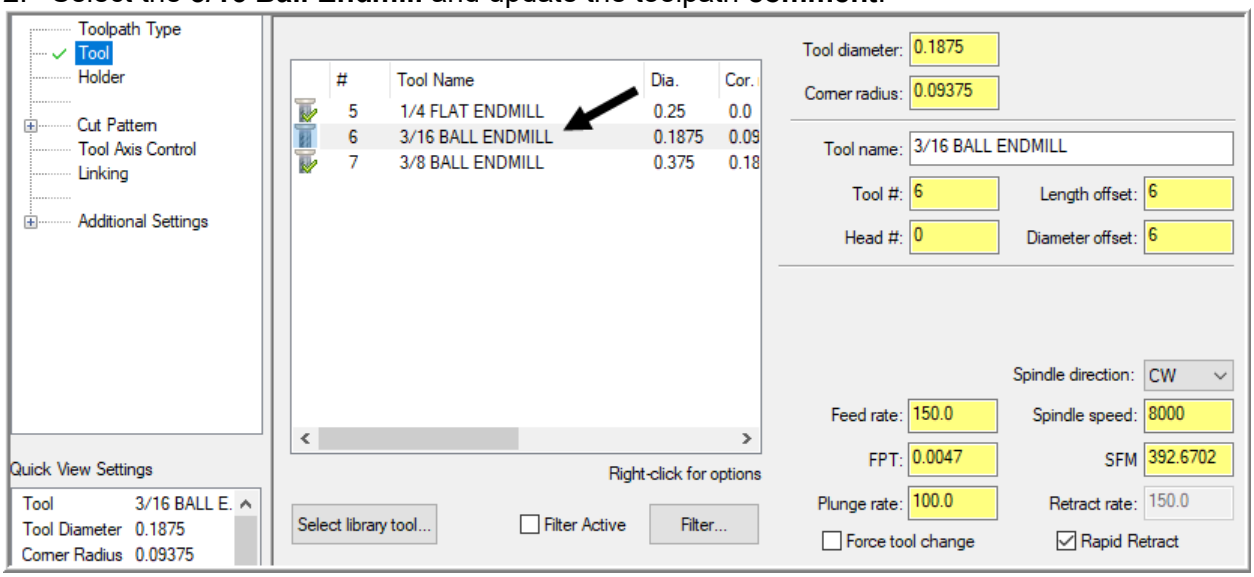
TASK 8:
DEBURR THE PART

- The final toolpath for this part will break the sharp corners and edges, it will deburr the part.
- The Multi Axis Deburr toolpath in Mastercam makes complex 5 axis deburring very simple.

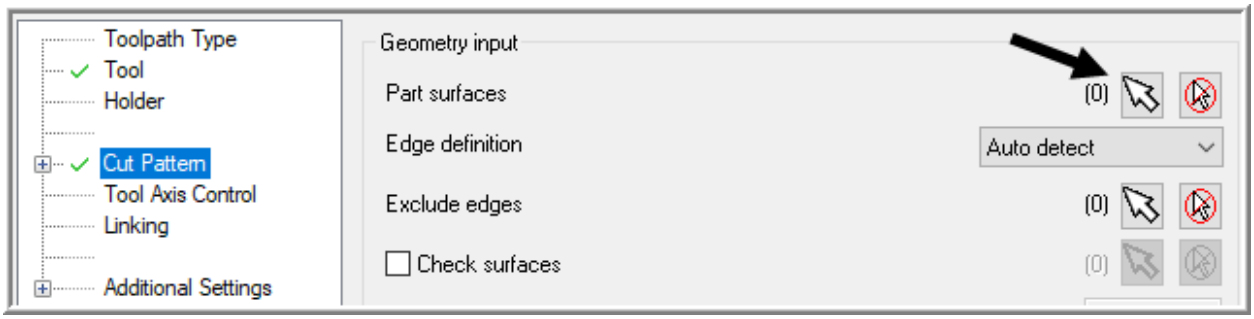
1. From the **Multi Axis** toolpath gallery, select **Deburr**.



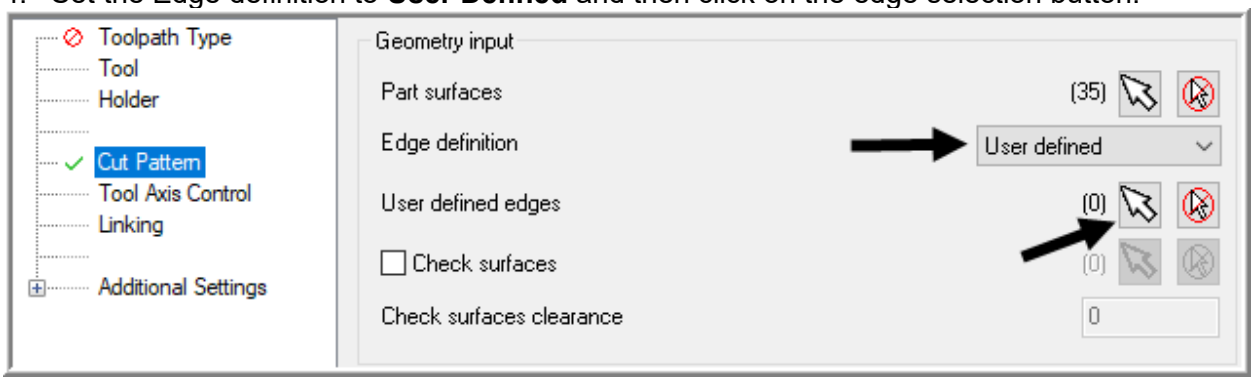
2. Select the **3/16 Ball Endmill** and update the toolpath **comment**.



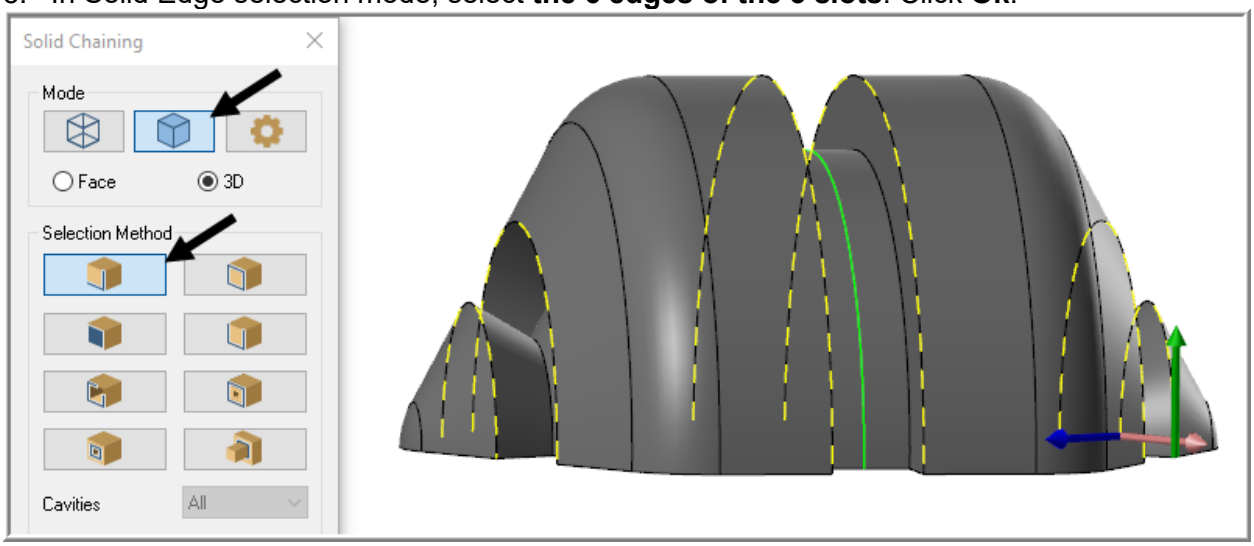
3. On the **Cut Pattern** page, click the **Part Surfaces selection** icon. Select the entire part model.



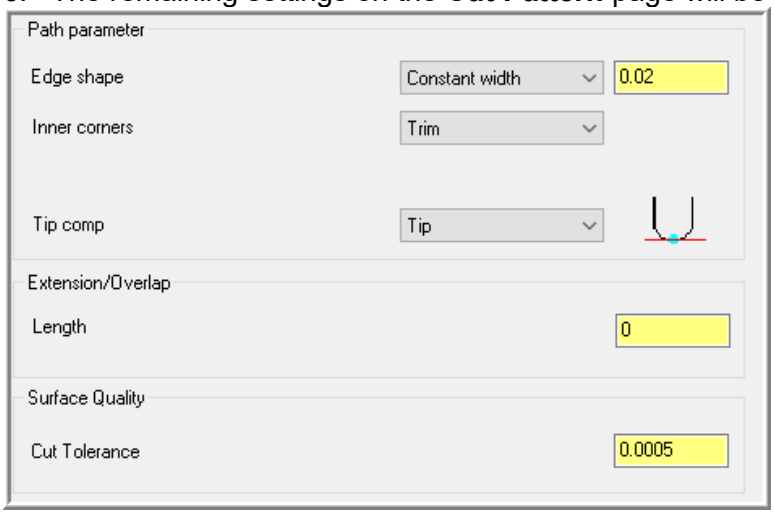
4. Set the Edge definition to **User Defined** and then click on the edge selection button.



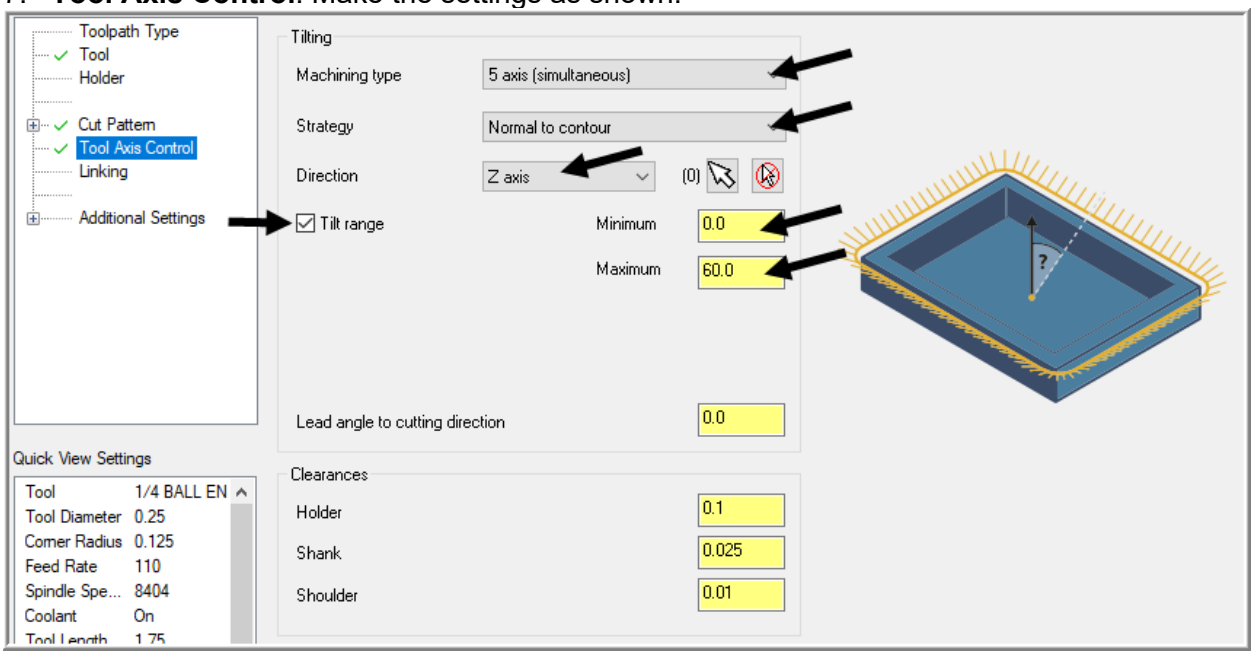
5. In Solid Edge selection mode, select **the 6 edges of the 3 slots**. Click **Ok**.



6. The remaining settings on the **Cut Pattern** page will be left at the **default values**.

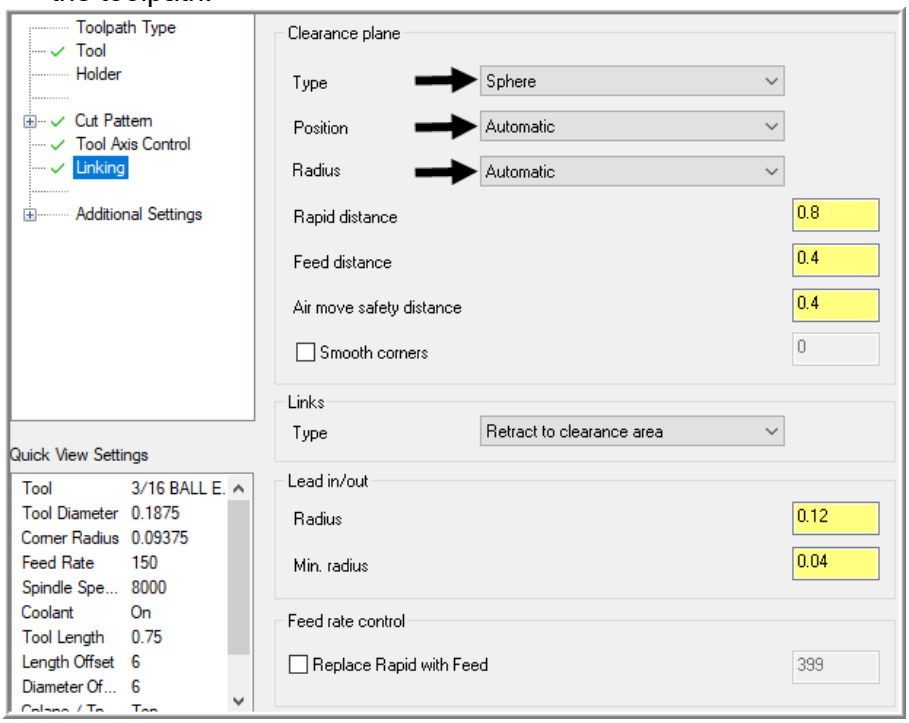


7. **Tool Axis Control**. Make the settings as shown.

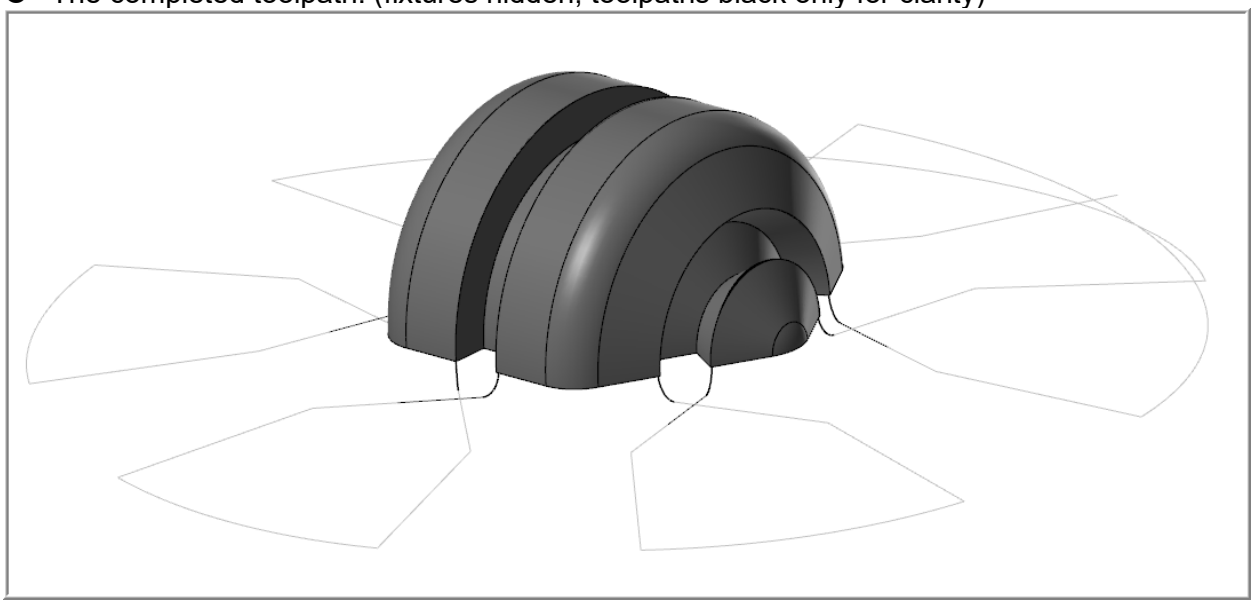


➲ The **Machining Strategy** is set to **5 axis**, this will allows the machine to move any of its axes to reach the needed cutting area. With this, the **Strategy** is set to **Normal** to Contour. The toolpath will try and keep the tool normal (perpendicular) to the cutting edge. However, since we have features on all sides of the part, this Normal cutting strategy will lead to some extreme rotations which could lead to problems. To solve this, we have enabled limits on the tilting range of the toolpath.

8. **Linking, Type** set to **Sphere**, **Position and Radius set to Automatic**. Click **Ok** to complete the toolpath.



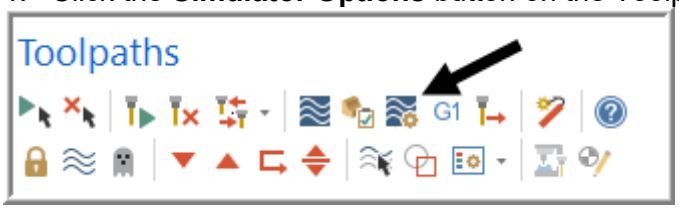
➲ The completed toolpath. (fixtures hidden, toolpaths black only for clarity)



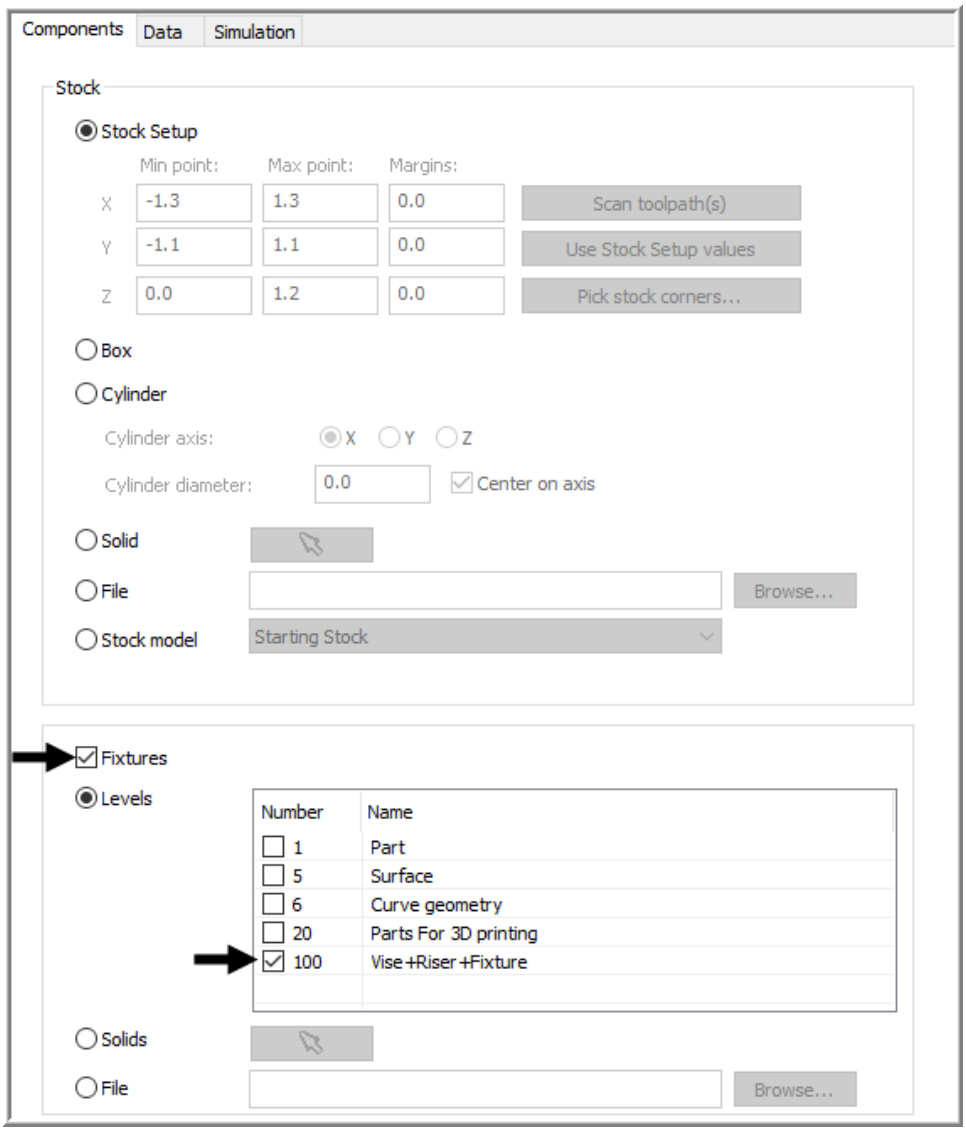
TASK 9:
SETUP THE VERIFY AND SIMULATE OPTIONS

- Before launching either **Verify** or **Simulate**, we need to setup some information in the options to get useful, accurate verification and simulations.
- **Verify** = part cutting. Simulate = part cutting and machine motion.

1. Click the **Simulator Options** button on the Toolpaths Manager.



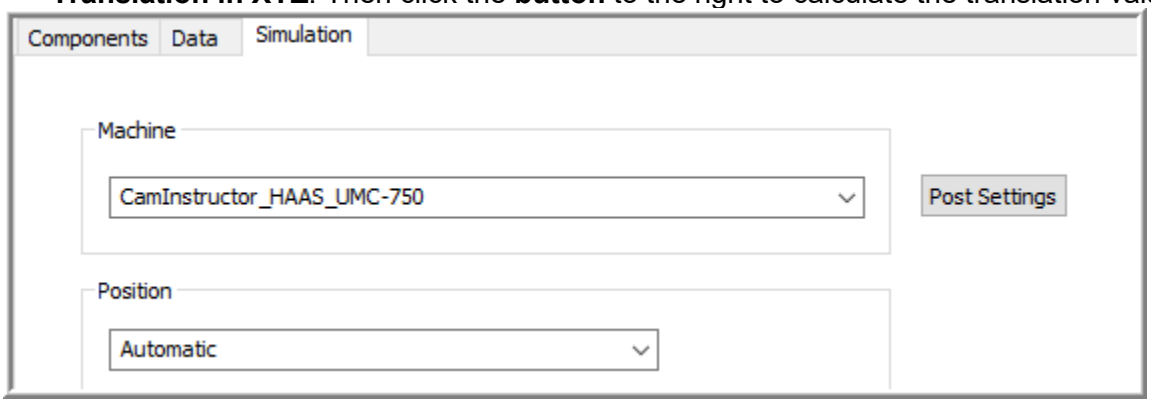
2. On the **Components** tab, turn **Fixtures on and check level 100**. Anything on level 100 will be simulated as a fixture.



3. No changes will be made on the **Data** tab.

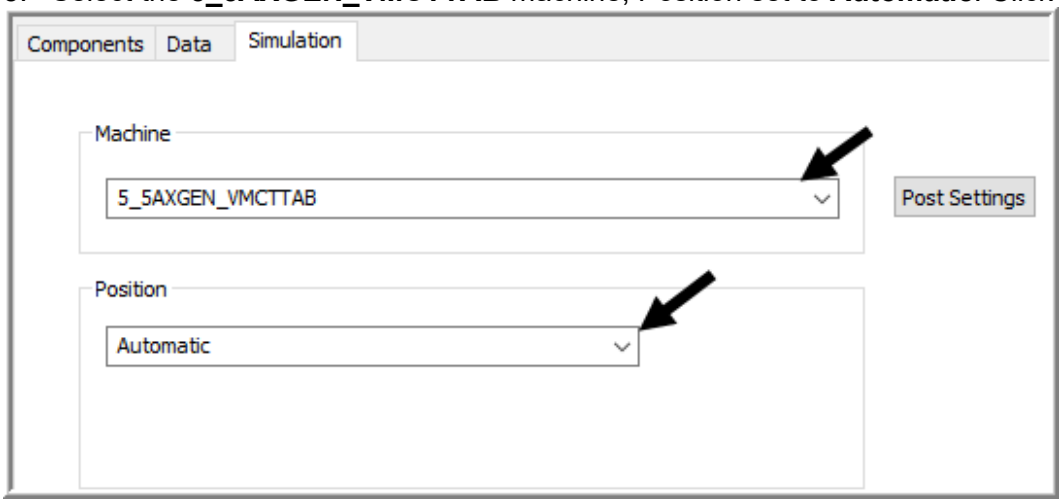
➲ If you have the **UMC750** mach sim files you can use a **VMC** with a **Trunnion**.

4. On the **Simulation** tab, select the **camInstructor_HAAS UMC750**. Set the Position to **Translation in XYZ**. Then click the **button** to the right to calculate the translation values.



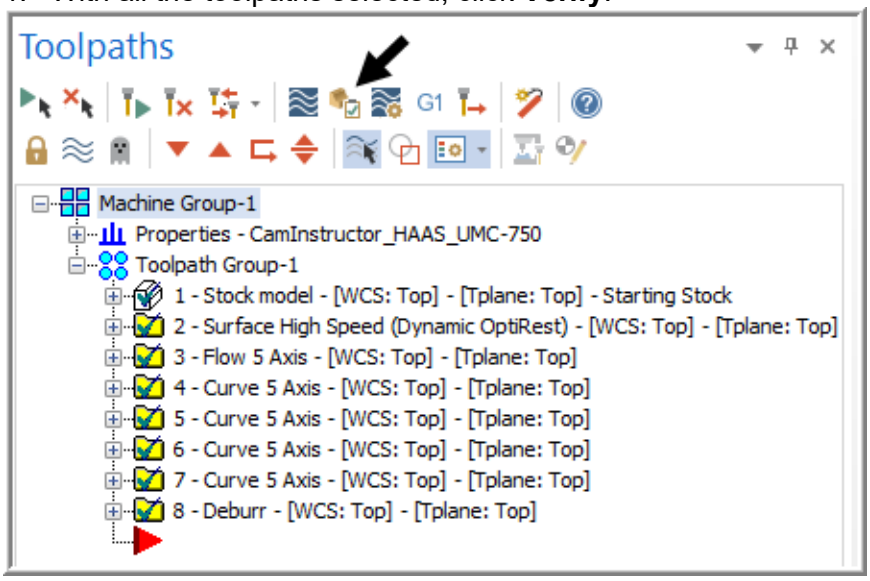
➲ If you do not have the **camInstructor_HAAS UMC750** Mach Sim files, make the following settings instead.

5. Select the **5_5AXGEN_VMCTTAB** machine, Position set to **Automatic**. Click **Ok**.

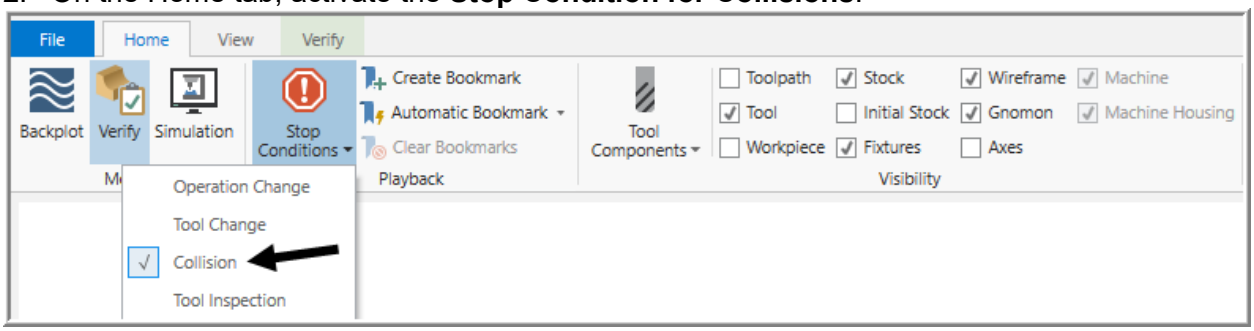


TASK 10:
VERIFY AND SIMULATE THE PART

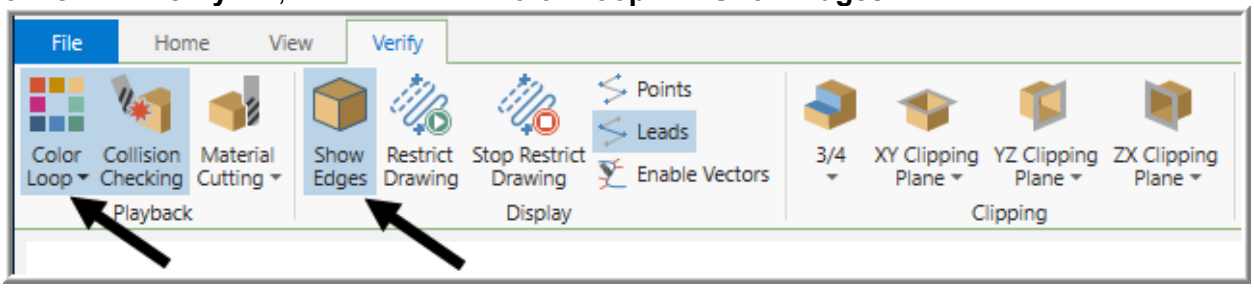
1. With all the toolpaths selected, click **Verify**.



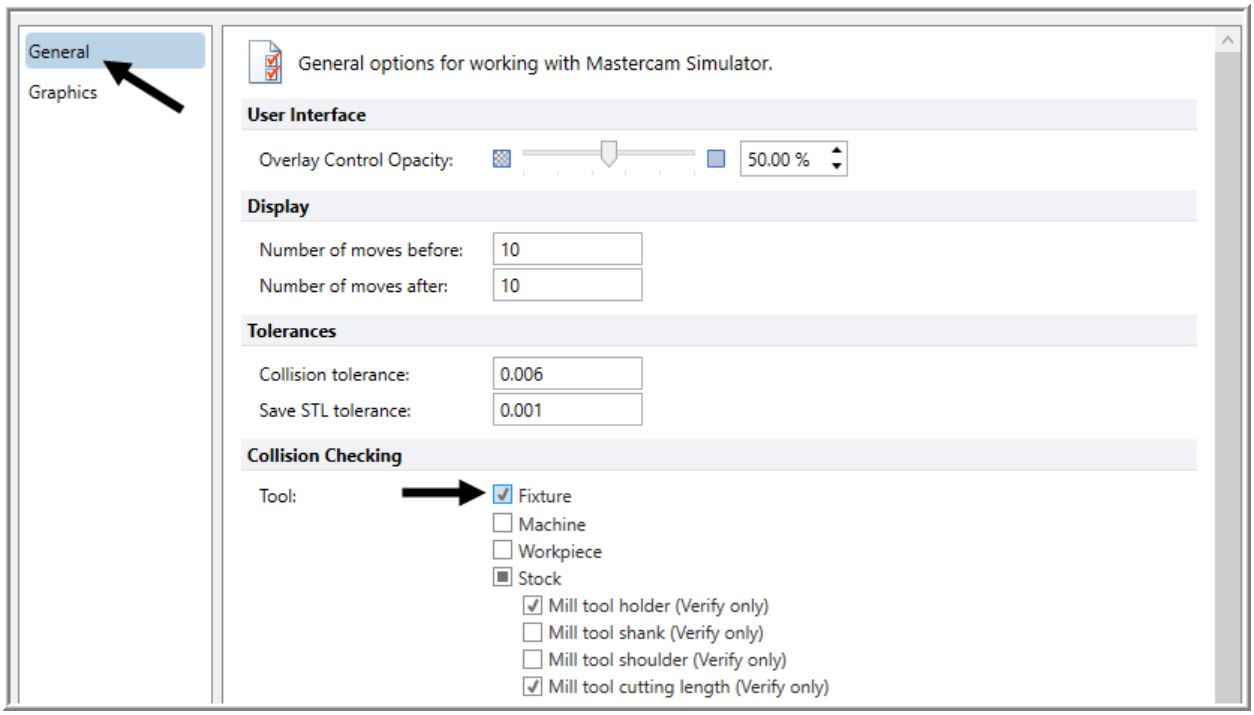
2. On the Home tab, activate the **Stop Condition for Collisions**.



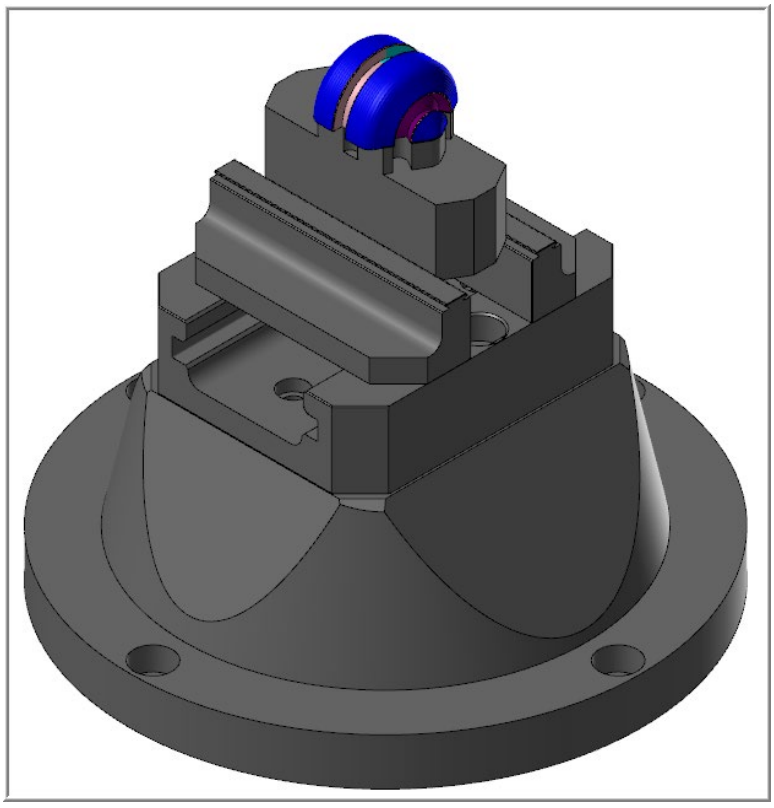
3. On the **Verify** tab, we've enabled **Color Loop** and **Show Edges**.



4. Click on **File>Options** and then on the **General** menu, enable **Collision Checking** with the **Tool to the Fixture**. Click **Ok**.

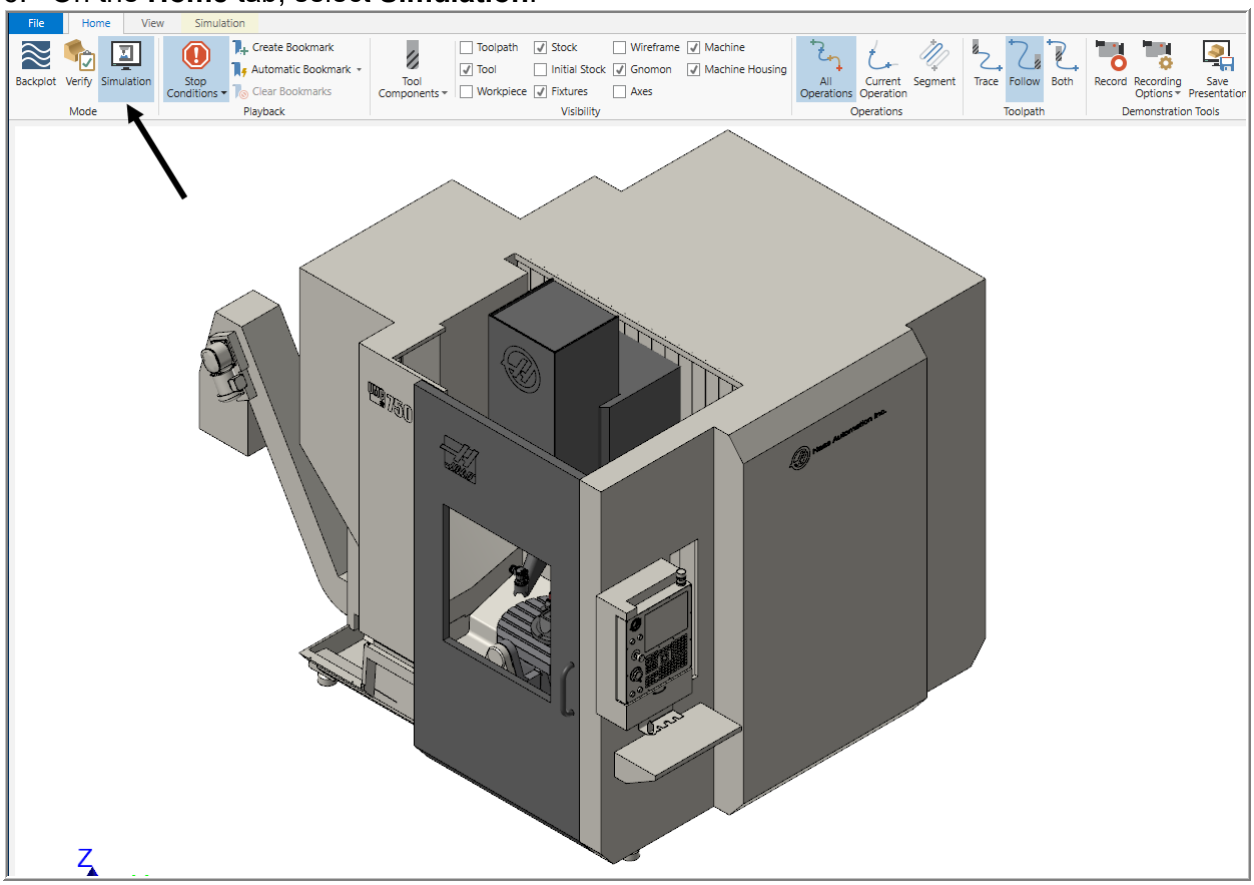


5. **Play** through the verification. Your part should remain in place while the tool rotated to the different cutting planes. The Verification should go quick. You should also see your defined fixtures on screen.

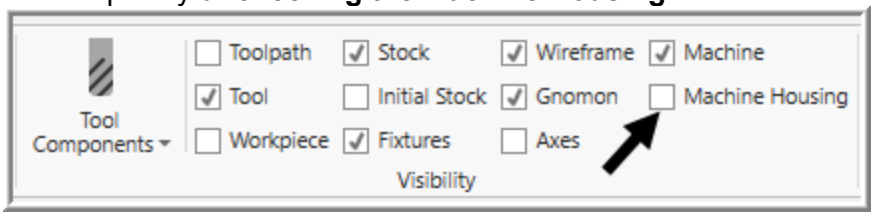


➲ The verification did not detect any collisions. Switch to **Simulate**

6. On the **Home** tab, select **Simulation**.

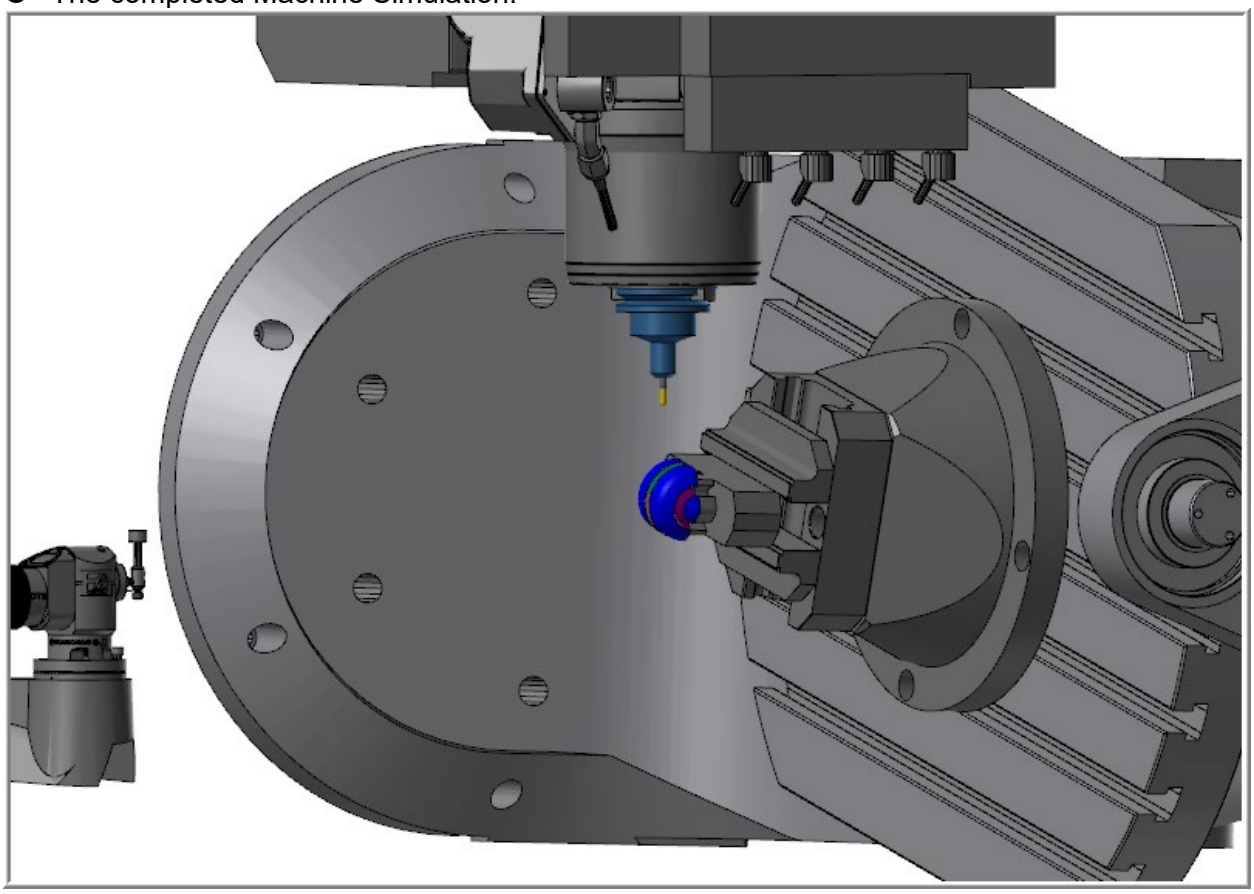


7. The **Machine Housing** looks nice, but it will be in the way during Simulation. Hide it by completely **unchecking the Machine Housing** check box.

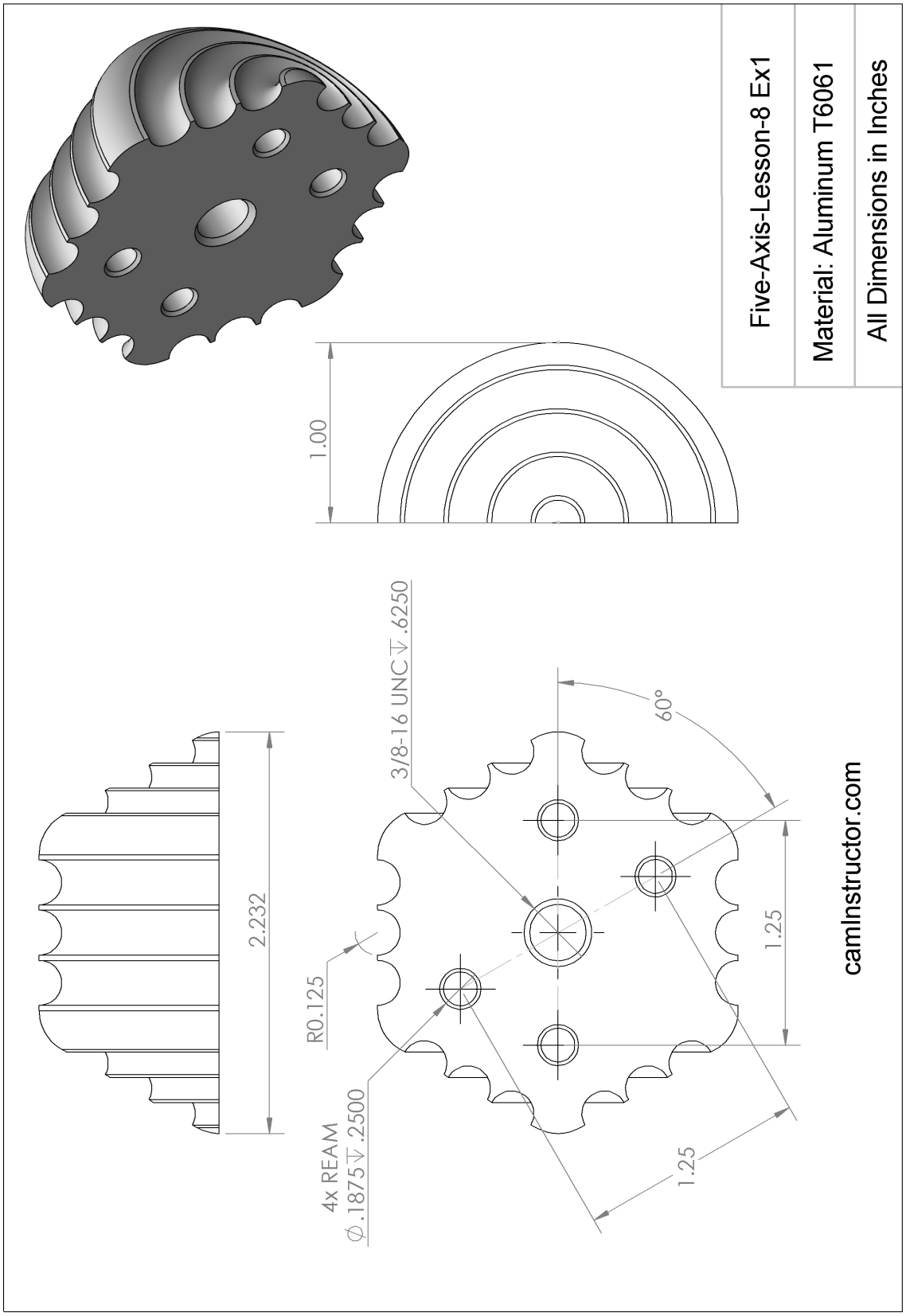


8. The Simulation will accurately show all machine movement as the actual machine will make during the machining process. This Simulation will take longer than the Verification did.
9. You can speed up or slow down the process by adjusting the Performance/Precision slider and/or the Slow/Fast slider.

➲ The completed Machine Simulation.



Five-Axis-Lesson-8-Ex1



Five-Axis-Lesson-8-Ex2