

## **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:

#### **Contract Set 5** Importing the machine:

Mastercam can simulate exact replicas of machine tools to ensure crash free programs. You will import these files.

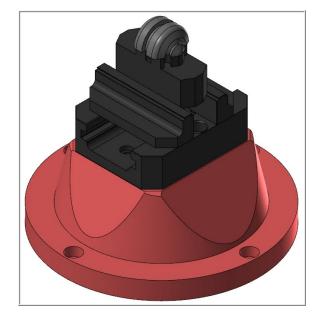
#### **Open an existing file containing:**

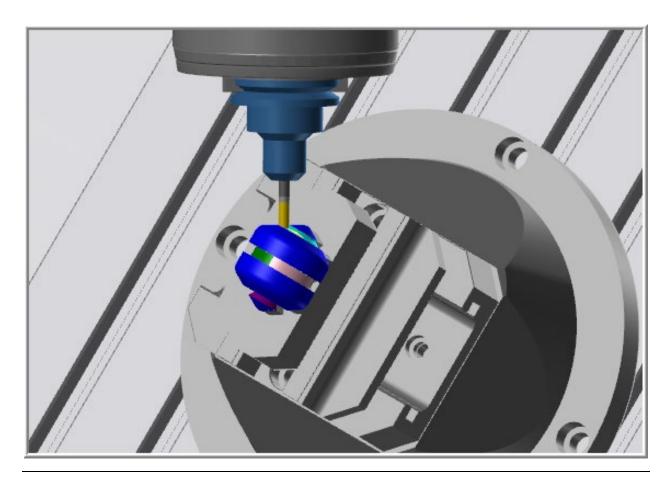
The solid geometry for the machine vise Solid geometry for the part Tools for machining

#### **Contract Compatible Consisting of:**

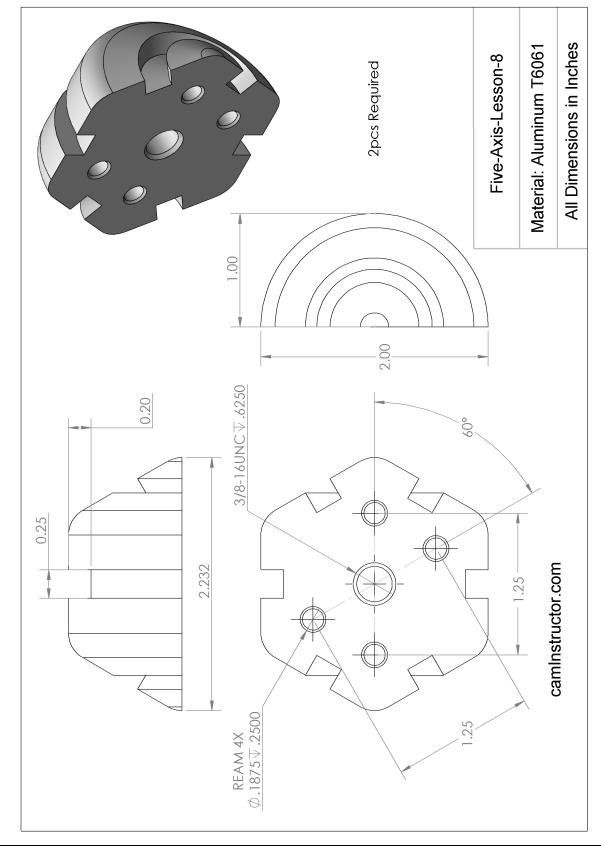
3+2 3D roughing, Flow 5 Axis, Curve 5 Axis, and Multaxis Deburr

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





Five-Axis-Lesson-8-1



### Five-Axis-Lesson-8-DRAWING

Five-Axis-Lesson-8-2

## TOOL LIST

#	Tool Name	Holder Name	Dia.	Cor. rad.	Length	# Flutes	Туре	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.
- 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.

File Home Share	View 🗲 🗰	Hand Folder Tools					,
Navigation Details pane		d icons End Small icons	<pre></pre>	☐ Group by ▼ ▲ Add columns ▼ Size all columns to fit	<ul> <li>☐ Item check boxe</li> <li>✓ File name exten</li> <li>✓ Hidden items</li> </ul>	isions Hide selected items	Options
Panes		Layout		Current view	Shov	v/hide	
-							
← → × ↑ 🕹 > Tr	is PC > Download	5		•	√ Č	Search Downloads	
🖈 Quick access		Name		Date modified	Туре	Size	
	*	📙 CamInstructor_HAAS_U	JMC-750.zip		Compressed	(zipp 3,813 KB	3
Desktop							
Documents	*						
🕂 Downloads	A						
E Pictures	*						

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**.

~	Extract Compressed (Zipped) Folders	
	Select a Destination and Extract Files	
	Files will be extracted to this folder:	
	C:\Users\mikew\Downloads\CamInstructor_HAAS_UMC-750	Browse
	Show extracted files when complete	
		Extract Cancel

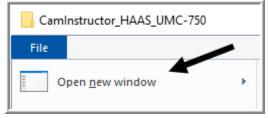
#### 4. Extraction complete...

$\leftarrow$ $\rightarrow$ $\checkmark$ $\uparrow$ $\square$ $\rightarrow$ This F	C > Downloads > CamInstructor_H	HAAS_UMC-750 >	✓ Ö Search C
	Name	^ Date modified	Туре
Quick access Desktop	🖌 🔒 Caminstructor	HAAS_UMC-750	File folder
Documents	A		

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.

	> CamInstructor_HAA	AS_UMC-750
Name	Туре	Size
GamInstructor_HAAS_UMC-750	File folder	
CamInstructor_HAAS_UMC-750.mcam-control	MCAM-CONTROL	412 KB
CamInstructor_HAAS_UMC-750.mcam-mmd	Mastercam MMD	186 KB
CamInstructor_HAAS_UMC-750.pst	Outlook Data File	12 KB

- 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.

	ents > Shared Mastercam 2020 > CNC_M	ACHINES >	
Name	Type Siz	te	Dat
CamInstructor_HAAS_UMC-750.mcam-control	MCAM-CONTROL File	412 KB	201
CamInstructor_HAAS_UMC-750.mcam-mmd	Mastercam MMD File	186 KB	201

## 9. Select the **CamInstructor\_HAAS\_UMC-750** folder. **Copy and Paste** it into the MachSim folder inside of the MachineSimulation folder.

	> Public > Public Docu	ments > Shared Mast	ercam 2020 🔉	${\sf MachineSimulation} \ \Rightarrow \ {\sf MachSim} \ \Rightarrow$
Name .	Date modified	Туре	Size	
CamInstructor_HAAS_UMC-750	2019-09-20 12:19	File folder		

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

→ This PC → Local Disk (C:) → Users →	Public > Public Docum	nents > Shared Maste	ercam 2020 → mill 🔅	Posts
Name	Date modified	Туре	Size	
CamInstructor_HAAS_UMC-750.pst	2019-09-20 12:30	Outlook Data File	12 KB	

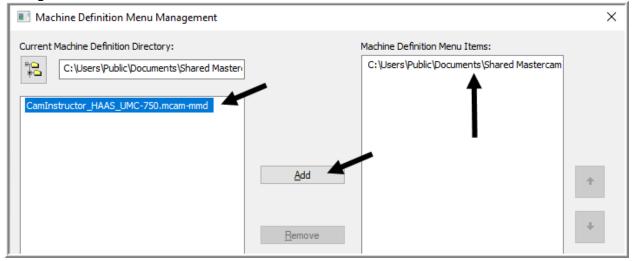
**C** This completes the file importing.

## TASK 2: SETUP THE MACHINE

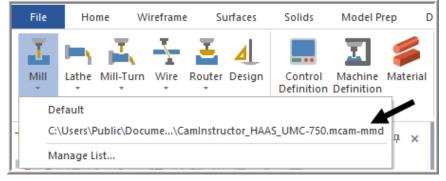
- 1. Launch Mastercam.
- 2. On the Machine tab, from the Mill pull down menu, select Mange List...

File	Hom	e	Wirefran	ne S	urfaces	Solids	Model Pr	ер	Dra	afting	Transfer	Ma	chine
L Mill	Lathe	Wire	Router	A Design	Control Definition	Machine Definition		Ba	<b>e</b> ckplot	Verify	Simulate	G1 Generate	Create
D	efault	2 1	ýpe			Job Setup			Sir	nulator	L2	Post	:
- M	anage Lis	t 🛉	<b>—</b>				<b>⊸</b> д :	×					

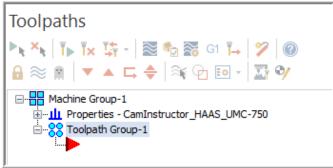
3. Select the inch version of the UMC-750 and click **Add**. The machine will then appear in the right menu. Click OK to exit.



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.

Toolpaths	Files Tool Settings Stock Setup
▶ × To Tx Tr - S	Group name Machine Group-1
	Toolpath directory C:\Users\mikew\Documents\My Mastercam 2020\Mastercam\MILL
	Group comment
Machine Group-1     Oroperties - CamInstructor HAAS UMC-750	v
Files	Machine - Toolpath Con
Tool settings	Edit 💡 🔭 Replace 😭
□ -88 Toolpath Group-1	Description CamInstructor_HAAS_UMC-750
	From file CamInstructor_HAAS_UMC-750.mcam-mmd

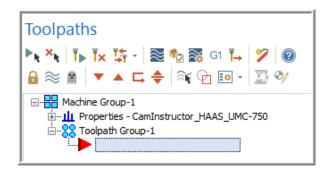
6. The folder locations for the Post and Control should match those used in the previous task.

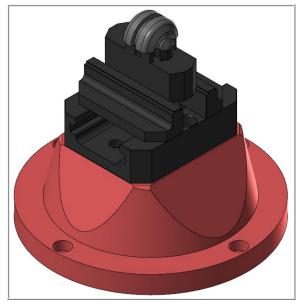
Machine Definition Manager - Machine Group-1	;
🖵 🕶 🛊 🕴  🔛 🚟 🛠	
Unused Component Groups	Description: CamInstructor_HAAS_UMC-750
	C:\Users\Public\Documents\Shared Mastercam 2020\CNC_MACHINES\CamInstructor_HAAS_UMC-750.mcam-control     Post-processor
-	Folder: C:\Users\Public\Documents\Shared Mastercam 2020\mill\Posts CamInstructor_HAAS_UMC-750.pst ~

- 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.





То	ol Ma	inager							
achi	ine Gr	oup-1 N	r 📗 = Tool	used in an op	eration				
:	#	Tool Name	Holder Name	Dia.	Cor. rad.	Length	# Flutes	Туре	Rad. Type
	5	1/4 FLAT ENDMILL	C4E4-0375	0.25	0.0	0.625	4	Flat endmill	None
2	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.

les Tool Settings Stock Set	up						
Default program number	3						
Feed Calculation		Toolpath Configuration					
From tool		Assign tool numbers sequentially					
O From material		Warn of duplicate tool numbers					
◯ From defaults		└ Use tool's step, peck, coolant					
O User defined		Search tool library when entering a					
Spindle speed	5000.0	Lool number					
Feed rate	50.0	Advanced options					
		Override defaults with modal values					
Retract rate	125.0	Clearance height					
Plunge rate	25.0	Retract height					
Adjust feed on arc mo	ve	Feed plane					
Minimum arc feed	5.0						
		Sequence number					
		Start 100.0					
		Increment 10.0					
Material							
ALUMINUM inch - 6061		Edit Select					
Ites Tool Settings Stock Setup Stock Plane Iffi Top Shape Pectangular Cylindrical Solid/Mesh							
○ File							
Display ✓ Fit screen ④ Wire frame Solid Stock Origin In view coordinates X 0.0 X 0.0							
Y 0.0 Z 1.2							

#### 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.

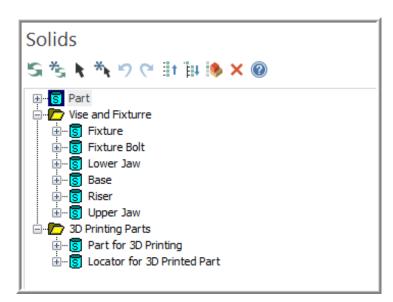
Level 100 contains all the solids used to represent the fixtures.

#### 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.

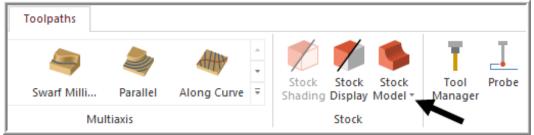
Groups can be used when many solids are used to represent an assembly. The Group can be collapsed to de-clutter the Solids Manager.

Nu     Name     Level Set     Entities       ✓ 1     X     Part     1	Levels								
✓1 X Part 1	+ 🔍 📚 🛣 🗐 🔅 * 🔞								
	Nu 🔺								
5 Surface 15	🖌 1								
	5								
6 Flow geometry 2	6								
20 Parts For 3D printing 9	20								
100 X Vise+Riser+Fixture 6	100								

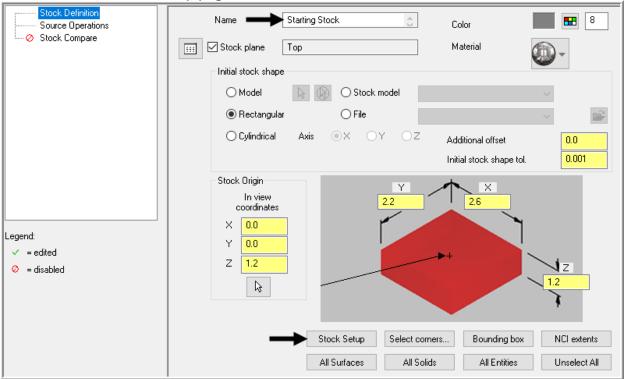


## TASK 4: ROUGH THE PART

- 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**.

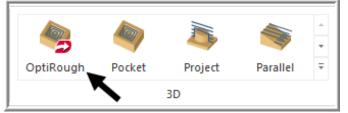


3. Turn the stock model on/off by using **Alt - T** or by clicking on the **Toolpath** Toggle display icon.



Five-Axis-Lesson-8-12

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

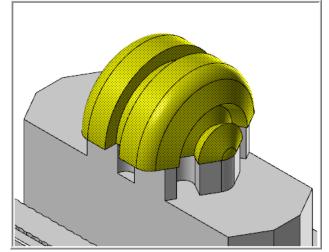


5. On the Model Geometry page set the Machining Geometry Wall and Floor stock to 0.01

Toolpath Type	^	Machining Geometry				Ave	idance Geometry			
Toolpath Control		Name	Entities	Wall Stock	Floor Stock		Name	Entities	Wall Stock	Floor Stock
Holder		machining	0	0.01	0.01		avoidance	0	0.0	0.0

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**.

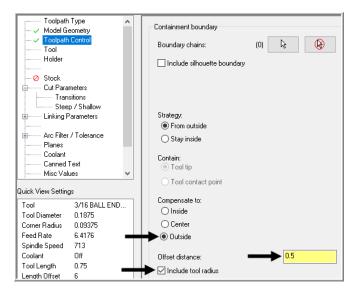
	Name	Entities	Wall Stock	Floor Stock
	machining	0	0.01	0.01
			<b>\</b>	
۲				



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.



Toolpath Type					/		Tool diameter: 0	.25		
Toolpath Control		#	Tool Name		Dia.	Cor.	Corner radius: 0			
Holder		a 5	1/4 FLAT EN	DMILL	0.25	0.0				
Holder		6			0.1875	0.05	Tool name: 1	/4 FLAT EN	NDMILL	٦.
🤣 Stock	1	7	3/8 BALL EN	DMILL	0.375	0.18				
Cut Parameters							Tool #: <mark>5</mark>		Length offset: 5	
Transitions Steep / Shallow							Head #: 0		Diameter offset: 5	1
Linking Parameters										_
							•			
Arc Filter / Tolerance										
Planes Coolant							×			
Canned Text							RCTF		Spindle direction: CW $\sim$	1
Misc Values 🗸						_	Feed rate: 91	.6800	Spindle speed: 7640	
Quick View Settings	Ľ	<				>	FPT: 0	003	SFM 500.0	
					Right-click for c	options			3FM 000.0	
Tool 1/4 FLAT ENDM Tool Diameter 0.25	D						Plunge rate: 4	0.0	Retract rate: 150.0	
Corner Radius 0	1	select lib	rary tool	Filter Act	tive Filter.		Force tool	change	Rapid Retract	

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.

Toolpath Type     Model Geometry     Toolpath Control     Tool     Holder		operties polpath Group-1 1 - Stock model - [WCS: Top] - [Tplane: Top] - Starting
Cut Parameters	One other operation     Roughing Tool	- Parameters - Triangles = 12 - 0.4K

- 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.

Toolpath	Туре	~						
🚽 🗸 Model Ge	eometry		Cut style					
🚽 🗸 🗸 Toolpath	Control		Cut method			Climb 🗸 🗸		
- V Tool						91.68		
Holder			Conventional fee	d rate		31.00		
Stock		rs i i i i i i i i i i i i i i i i i i i		Tip compensation		Tip ~		
	sitions		Optimize stepups			By depth 🛛 🗸		
	o / Shallow							
Emme Linking P	arameters		Optimize stepdow	vns		Material $\sim$		
Arc Filter	/ Tolerance						Motion > Gap size, retract	11
Planes	7 Tolerance		Passes		,		Never ~	
Coolant			Stepover	30.0 -	%	0.075		
Canned T	ſext				1		Gap size	
Misc Valu	les	~	Stepdown	200.0	%	0.5	O Distance 1.25	
			Stepup	10.0	2	0.025		
Quick View Setting	ļs						└	
Tool	1/4 FLAT END	м	Mill vertica	al walls				1
Tool Diameter	0.25		Minimum toolpath	n radius 🛛 🚄				
Corner Radius	0			30.0	%	0.075		
Feed Rate	100.031			00.0	%	0.010		
Spindle Speed	7640							
Coolant	Off		Motion < Gap size	e, micro lift				
Tool Length	1.125		Micro lift distance	9		0.01		
Length Offset	5					100.0		
Diameter Offset	5		Back feedrate			100.0		

Five-Axis-Lesson-8-14

- 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.

Transitions		
Linking Parameters	Z depth	
	Adjust for stock to leave	
Arc Filter / Tolerance		
Coolant	Detect limits	
Canned Text	Minimum denth 0.0	
Misc Values	Minimum depth 0.0	
Quick View Settings	-0.05	
Tool 1/4 FLAT ENDM		

- **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.

Toolpath Type A Model Geometry Toolpath Control Tool Holder	Retracts Clearance plane Absolute	8.0 k	Minimum Vertical Retract
Holder → Stock → Cut Parameters → Transitions → Steep / Shallow ← Linking Parameters → Arc Filter / Tolerance → Planes	Curl up Curl down Part clearance	0.15 0.15 0.25	
Coolant Canned Text	Leads		
< >	Linear entry/exit (incremental)	0.1	Fitting
Quick View Settings	Vertical arc entry	0.075	Minimize Trimming ~
Tool 3/8 FLAT EN Tool Diameter 0.375	Vertical arc exit	0.075	Max trimming 0.05 distance
Corner Radius 0 Feed Rate 98.0196	Horizontal arc entry	0.075	
Spindle Speed 7640 Coolant On	Horizontal arc exit	0.075	
Tool Length 1.625	Max ramp angle	10.0	
Length Offset 22 Diameter Off 22	Ramp height	0.1	

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.

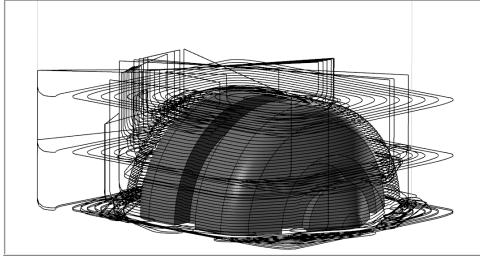
Toolpath Type Model Geometry Toolpath Control	el Geometry						
Holder     Holder     Cut Parameters     Transitions     Steep / Shallow     Linking Parameters	tolerance	Line/Arc Smoothing tolerance tolerance					
Arc Filter / Tolerance     Planes     Coolant     Canned Text     Misc Values	Create arcs in:	Smoothing Settings Use Fixed Segment Length Segment Length 0.02					
Quick View Settings Tool 1/4 FLAT ENDM Tool Diameter 0.25 Corner Badius 0	Minimum arc radius 0.005 Maximum arc radius 100.0	<ul> <li>Shift points randomly along toolpath</li> <li>Minimize number of points</li> </ul>					
Field Rate 100.031 Spindle Speed 7640 Coolant Off Tool Length 1.125	Use maximal tolerance value for both     Tighten Line filtering tolerance     Tighten Arc filtering tolerance	Present arcs as line segments  Output 3D arc entry motion					

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.

Toolpath/Surface	<				
A containment boundary is required for rest roughing, the min/max limits of the machining data will be used					
$\checkmark$					
Do not show this warning again this session					

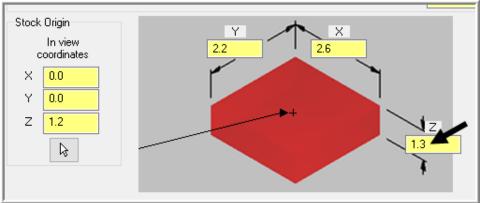
The resulting toolpath, fixture hidden for clarity. Backplot the toolpath, notice the estimated cycle time. 6m44s. When verified, you may notice a depth cut at 0.0108 and the final depth cut at 0.0. The toolpath specified a depth of -0.05, which has not happened.



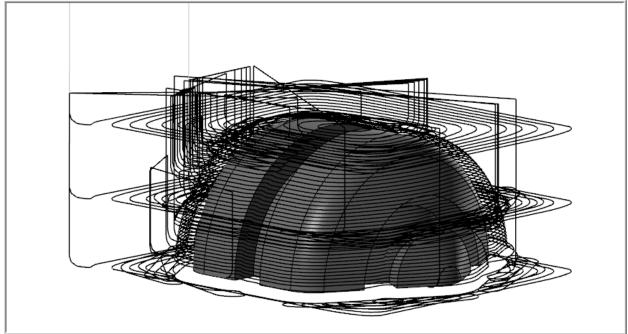
- This is a stock aware toolpath so motion will only be generated where there is stock to cut. In order to cut below Z0, the toolpath needs to see stock there to cut. Next, we will adjust the stock model to have stock in this area.
- The depth cuts at 0.0108 and 0.0 are not efficient. Cutting a slightly 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**.

🗄 🧹 Cut Para			l ip compensatio	n		
Tran: Stee	sitions p / Shallow		Optimize stepups			By depth $\sim$
Linking Parameters		Optimize stepdov	Material $\sim$			
Arc Filter	/ Tolerance		Passes			
Coolant			Stepover	30.0	%	0.075
Canned <sup>-</sup> Misc Valu			Stepdown	220.0	%	0.55
Quick View Setting	js		🗹 Stepup	10.0	%	0.025
Tool	1/4 FLAT ENDM		Mill vertica	al walls		
Tool Diameter	0.25		Minimum toolpath	n radius		
Corner Radius	0			30.0	2%	0.075
Feed Rate	100.031				~	
Spindle Speed	7640					
Coolant	On		Motion < Gap siz			
Tool Length	1.125		Micro lift distance	9		0.01
Length Offset Diameter Offset	5 5		Back feedrate			300.0

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**.

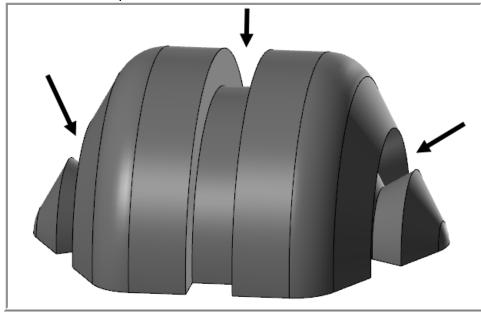


- 18. Both the Stock Model and Toolpath are dirty. Regenerate both.
  The adjusted toolpath. Backplot and make note of the new cycle time. 5m20s. The toolpath now cuts down to -0.05 as well.

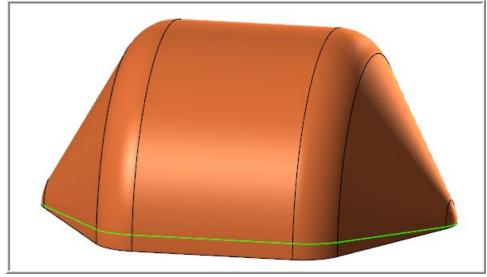


## TASK 5: FINISH THE SURFACE WITH FLOW 5 AXIS

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.



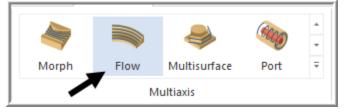
These surfaces also extend slightly past the bottom of the part. Check the Tips and Techniques section for steps to create this surface.



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

Levels			
+ 9, 📚 🛣 🗏 🔅 - 🞯			
Visible	Name	Level Set	Entities
	Part		1
х	Surface		15
	Flow geometry		2
	Parts For 3D printing		9
	Vise+Riser+Fixture		6
	/isible	Visible Name Part X Surface Flow geometry Parts For 3D printing	Visible Name Level Set Part X Surface Flow geometry Parts For 3D printing

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



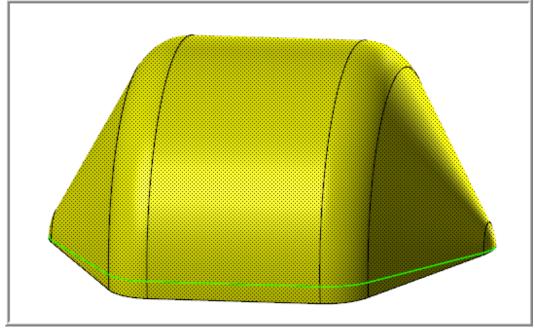
3. Select the **3/8 Ball Endmill**. Speeds and Feeds should be set automatically.

Toolpath Type			Tool diameter: 0.375
Holder	# Tool Name	Dia. Cor.	Comer radius: 0.1875
	5 1/4 FLAT ENDMILL	0.25 0.0	Comer radius: 0.1073
Stock Cut Pattem	6 3/16 BALL ENDMILL	0.1875 0.09	Tool name: 3/8 BALL ENDMILL
Tool Axis Control	7 3/8 BALL ENDMILL	0.375 0.18	
Collision Control			Tool #: 7 Length offset: 7
Linking Roughing		•	Head #: 0 Diameter offset: 7
Filter		-	
Additional Settings			
			Spindle direction: $\square$ CW $\checkmark$
	<	>	Feed rate: 150.0 Spindle speed: 8000
Quick View Settings		Right-click for options	FPT: 0.0047 SFM 785.3403
Tool 3/8 BALL EN 🔺		-	Plunge rate: 100.0 Retract rate: 150.0
Tool Diameter 0.375	Select library tool Filter Activ	ve Filter	Force tool change Rapid Retract
Comer Radius 0.1875			

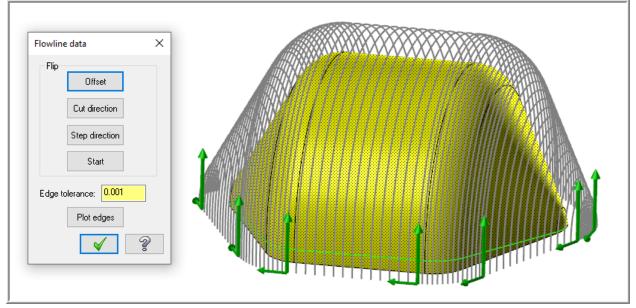
4. On the Cut Pattern page, click the **Surface selection** button.

Toolpath Type ✓ Tool Holder	Surfaces (0)	Flow parameters
Stock	Cutting method	Zigzag ~
<u>Cut Pattem</u> Tool Axis Control     Collision Control	Compensation type	Computer ~

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.

Toolpath Type	Surfaces (7)	Flow parameters
Stock	Cutting method	Zigzag ~
Cut Pattem	Compensation type	Computer 🗸
Collision Control	Compensation direction	Left 🗸 🖒
Roughing Filter	Tip compensation	Tip 🗸 🔰
	Stock to leave on drive surfaces	0.0
	Diameter (for simulation)	0.25
	Distance increment	0.1
Quick View Settings	Step along	
Tool 3/8 BALL EN A Tool Diameter 0.375	Distance	0.05
Comer Radius 0.1875 Feed Rate 150	Cut tolerance	0.001
Spindle Spe 8000	Step across	
Coolant On	Distance:	0.05
Tool Length 1.125		
Length Offset 7	O Scallop height:	0.001

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

Toolpath Type ✓ Tool Holder	Tool axis control 🗪 Patt	ern surface 🗸 (7) 🗟 🛞
Stock	Output format	5 axis V
Cut Pattem ⊕✓ Tool Axis Control	Backplot rotary axis	imes axis $ imes$
Collision Control	Lead/lag angle	0.0
Roughing Filter	Side tilt angle	0.0
	Angle increment	3.0
	Tool vector length	1.0
	Minimum tilt	0.0

#### **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.

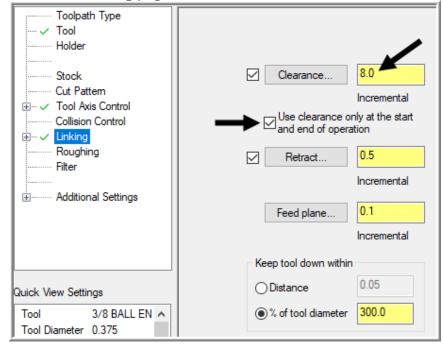
From point: Limits the tool axis to originate from a selected point.

**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.

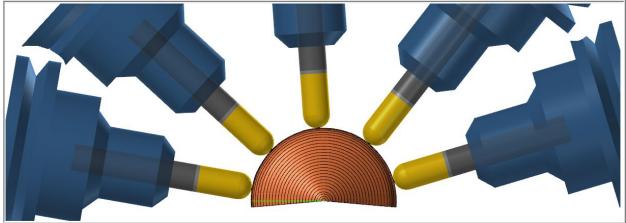
#### 10. On the Linking page, set the Clearance to 8.0 and enable Only at the Start and End.



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

Toolpath Type ── ✓ Tool ── Holder	☑ Filter	
Stock Cut Pattem 	Tolerance Look ahead Dne way filtering	0.001

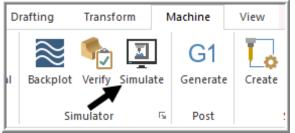
- 12. Click **Ok** to complete the operation.
- 13. **Backplot** the toolpath. Notice, particularly about half-way through the toolpath when machining the mid-section, the tool does not disengage from the cut.



- Next, confirm this same motion in **Simulation** which uses the machines limits to determine toolpath motion.
- 14. Check that the correct machine is selected.

Toolpaths       ▶ × ×   T ▶ T × IF ·   ≥        > ≈ ≈   ▼ ▲ □ ≑   ≥
Components Data Simulation
Automatic

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



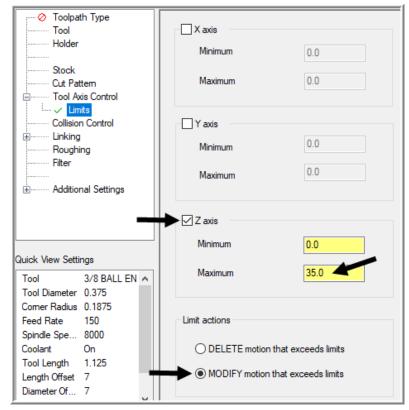
16. The first operation can be skipped over by clicking the Next Operation button.



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

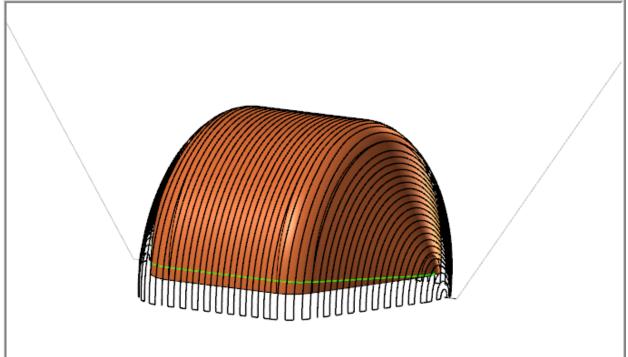
Enable Z Axis limits and set the Maximum to 35 degrees.

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.

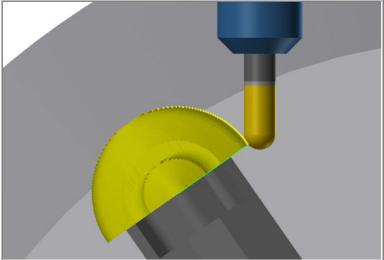


20. Click **Ok**, and the **Regenerate** the dirty operation.

#### ➔ The modified toolpath.

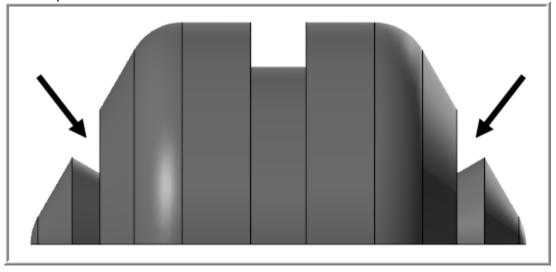


Running both toolpaths through Simulate will show no retracts happening in the mid-section of the part this time.



## 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.

Γ	Levels				
ŀ	+ 🔍 📚 🕿 🗐 🔅 - 🞯				
L	Nu 🔺	Visible	Name	Level Set	Entities
L	🖌 1	х	Part		1
L	5		Surface		15
L	6		Flow geometry		2
L	20		Parts For 3D printing		9
L	100		Vise+Riser+Fixture		6
<u> </u>					

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



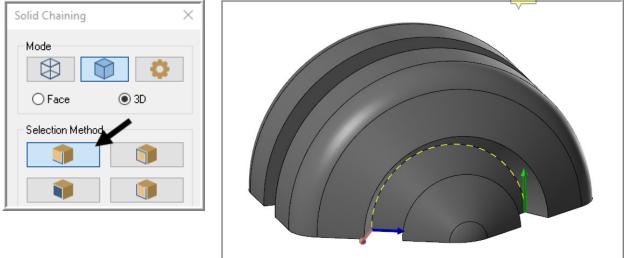
- 3. Select the **1/4 Flat Endmill** from the tool list.
- 4. Reduce the feedrate to 40 ipm. This path will be cutting a full slot so reducing the feedrate is necessary.



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**.

	tool, our of type will be bet to ob our too and then ollow the belowion batten.			
Toolpath Type Tool Holder	Curve type	3D Curves 🗸 (0) 🗟		
Stock	Compensation type	Computer V		
Cut Pattem     Tool Axis Control     Collision Control	Compensation direction	Left 🗸		
⊡ Linking Roughing	Tip compensation	Tip 🗸 <u> </u>		
Filter	Radial offset	<b>0.125</b>		

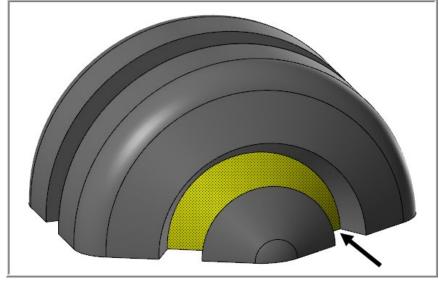
6. The Chaining Manager should be in the **Solid Mode**. Enable only **Edges** selection. Select the edge shown. Direction is important. Reverse the chain if needed. Click Ok.



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

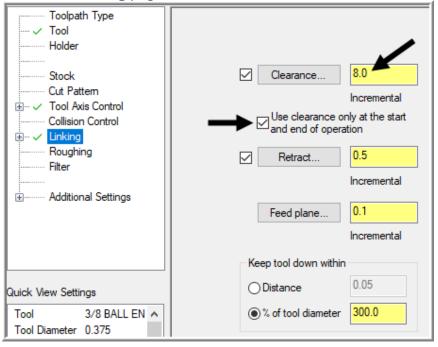
Toolpath Type		
Holder	Tool axis control 🗕	Surface V (0) 🔖 😡
Stock	Output format	5 axis V
Cut Pattem     Tool Axis Control	Backplot rotary axis	X axis V
Collision Control	Lead/lag angle	0.0
Roughing Filter	Side tilt angle	0.0
	Angle increment	3.0
	Tool vector length	1.0

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).

Toolpath Type Tool Holder	<ul> <li>✓ Entry/Exit</li> <li>✓ Entry curve</li> </ul>
Stock     Cut Pattem     Tool Axis Control     Collision Control     Linking	Always use       Length       55.0 % 0.1375       Image: Constraint of the second se
Entry/Exit     Home/Ref. Points     Safety Zone     Roughing	► Exit curve
Filter 	Always use
Quick View Settings	Thickness     0.0     %     0.0       Height     0.0     Direction     Image: Arrow angle of the second s
Tool     1/4 FLAT EN       Tool Diameter     0.25       Comer Radius     0	Curve tolerance 0.001

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.

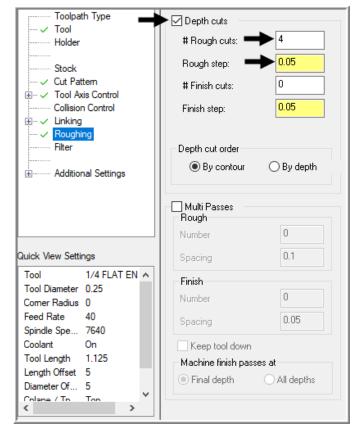
💮 🔗 Toolpath Type		Safety Zone
Tool	Safety Zone	Salety Zone
Holder	- Tool Motion	
	Axis of Rotation 5 Axis	3
Stock		Basic Advanced
Cut Pattern	Angle Step 3.0	Dasic Advanced
Tool Axis Control     Collision Control		Entities
	Use feed rate 0.0	
Entry/Exit		Select:  Manual:
Home/Ref. Points	Shape	<ul> <li>All shown</li> </ul>
Safety Zone		
Roughing	7	Shape
Filter	I I I	○ Rectangular
		O Cylindrical
Additional Settings	Y	Spherical
		Rectangular Settings
Quick View Settings	×	
Tool 1/4 FLAT EN A	n in the second s	Cylindrical Settings
Tool Diameter 0.25		
Comer Radius 0	Rectangular	Spherical Settings
Feed Rate 40	Define Shape	Radius: 1.5
Spindle Spe 7640		
1		

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

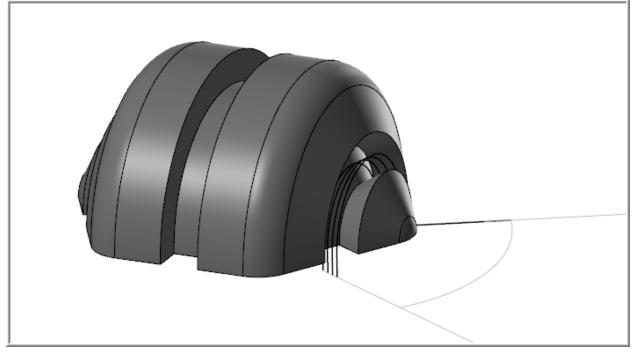
> Knowing the slot is 0.20 deep, setting 4 cuts of 0.05 each result in 4 even depth cuts starting from the top of stock.

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

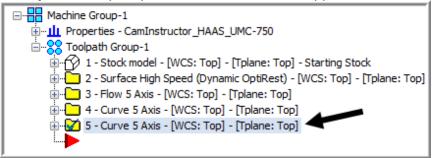
This scenario, the endmill is the same width as the slot so no Multi Passes are needed.



- 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.



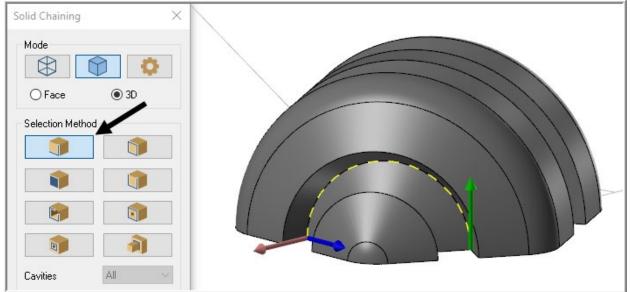
15. Copy and Paste the Curve 5 Axis toolpath. This will duplicate the toolpath just made. We will adjust this copied path to do the slot on the opposite side of the part.



- 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.

👓 🤣 Toolpath Type				
Holder	Curve type	3D Curves 🗸 (0) 🔓 🛞		
Stock	Compensation type	Computer V		
Collision Control	Compensation direction			
Linking	Tip compensation	Tip 🗸 📶		
Roughing				
Filter	Radial offset	0.125		

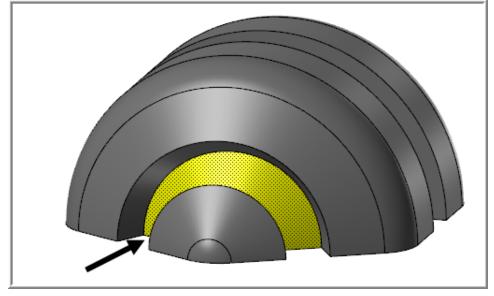
18. Select the curve at the other end of the part. Click Ok.



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

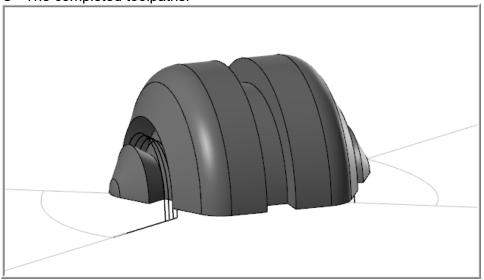
···· ⊘ Toolpath Type ······ Tool ····· Holder	Tool axis control	Surface	v (0) k 🛞
Stock	Output format	5 axis $\sim$	7 7
	Backplot rotary axis	imes axis $ imes$	a a
Collision Control	Lead/lag angle	0.0	

#### 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.

Γ	Levels				
Н	+ 🔍 📚 🕿 🗐 🔅 - 🞯				
	Nu 🔺	Visible	Name	Level Set	Entities
Ш	🖌 1	x	Part		1
Ш	5		Surface		15
H	6	Х	Curve geometry		2
	20		Parts For 3D printing		9
	100		Vise+Riser+Fixture		6

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



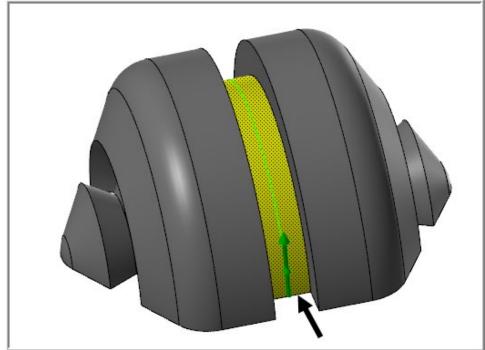
- Selection of a toolpath that was already made in the file will populate the values with those used in the previous instance, except for any geometry selections.
- 3. The same tool will be used, 1/4 Flat Endmill, with the Feedrate reduced to 40 ipm.
- 4. The Cut Pattern page will use some different settings. Set the **Radial Offset value to 0**. Then click the Curve **Selection button**.

Toolpath Type Tool	_	
Holder	Curve type	3D Curves V (0) 🗟 🛞
Stock	Compensation type	Computer V
Coll Patern     Tool Axis Control     Collision Control	Compensation direction	Left 🗸 🔿
Linking     Roughing	Tip compensation	Tip 🗸 🔰
Filter	Radial offset	▶ <mark>0.0</mark>

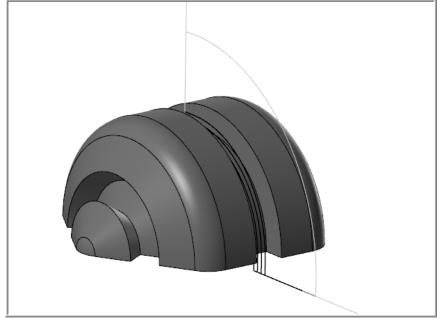
5. Switch to the **Wireframe** Selection Mode with the **Single** selection method. Select **one** of the arc segments. Click **Ok**.



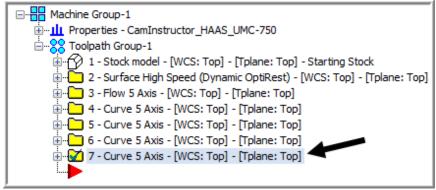
6. The **Tool Axis Control** will be left to **Surface**. Click the selection button and select the face under the previously selected wireframe arc. Click **End**.



- 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.

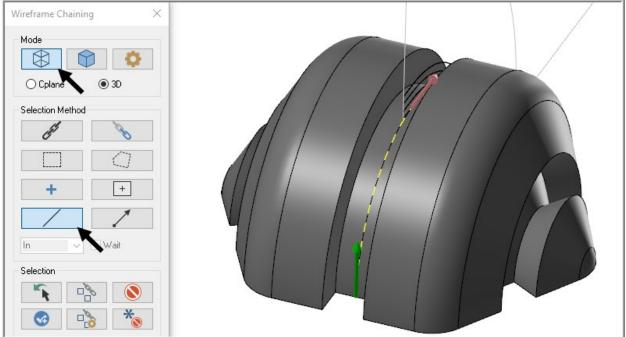


9. On the new toolpaths **Cut Pattern** page, clear the selected curve and then click the selection button to choose a new curve.

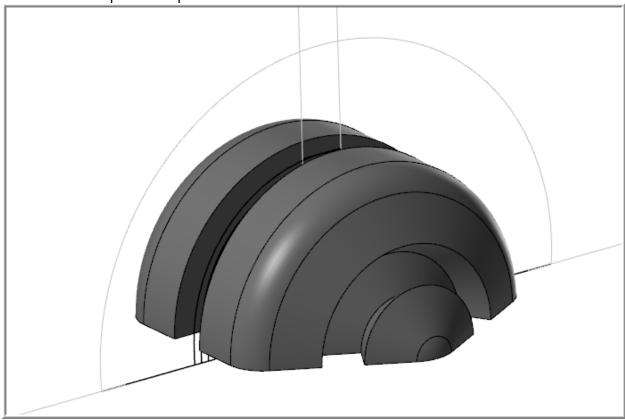
O Toolpath Type		
Holder	Curve type	3D Curves 🗸 (0) 🔓 🛞
Stock	Compensation type	Computer V
Cut Pattern		
Tool Axis Control	Compensation direction	
Linking	Tip compensation	Tip V
Roughing		
Filter	Radial offset	0.125
]		

#### Five-Axis-Lesson-8-36

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.



Five-Axis-Lesson-8-37

## 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.

Toolpath Type					Tool diameter: 0.1875	
Holder	#	Tool Name	Dia.	Cor.	Comer radius: 0.09375	
	7 5	1/4 FLAT ENDMILL	0.25	0.0	Comertadids.	
	<b>C</b>	3/16 BALL ENDMILL	0.1875	0.09	Tool name: 3/16 BALL ENDMILL	
Linking	7	3/8 BALL ENDMILL	0.375	0.18	Toormanie.	
					Tool #: 6 Leng	th offset: <mark>6</mark>
i Additional Settings					Head #: 0 Diamet	ter offset: 6
					Spindle o	direction: CW 🗸 🗸
					Feed rate: 150.0 Spind	le speed: 8000
Quick View Settings	<			>	FPT: 0.0047	SFM 392.6702
			Right-click for o	tions		150.0
Tool 3/16 BALL E. A	Select lib	rary tool Filter Act	tive Filter.		Plunge rate: 100.0 Ret	ract rate: 150.0
Tool Diameter 0.1875	Select lib		riller		Force tool change	Rapid Retract
Comer Radius 0.09375						

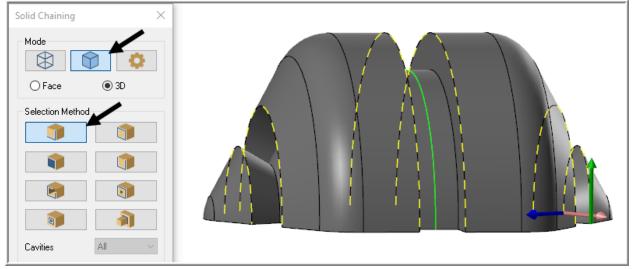
3. On the **Cut Pattern** page, click the **Part Surfaces selection** icon. Select the entire part model.

Toolpath Type	Geometry input	$\sim$
Holder	Part surfaces	🐻 🐼 🔞
	Edge definition	Auto detect 🗸 🗸
Tool Axis Control	Exclude edges	(0) 🔀 🚱
Additional Settings	Check surfaces	(0) 🔽 🛞

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**.

Path parameter	
Edge shape	Constant width V
Inner corners	Trim
Tip comp	Tip V
Extension/Overlap	
Length	0
Surface Quality	
Cut Tolerance	0.0005

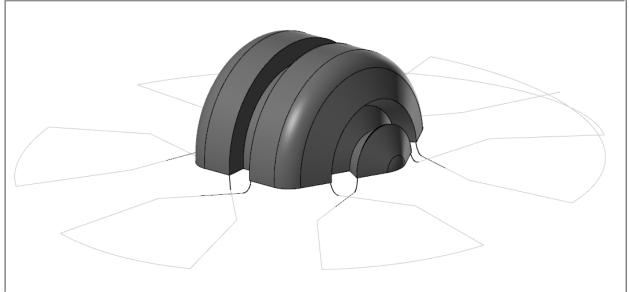
7. Tool Axis Contr	ol. Make the se	ettings as shown	l <b>.</b>	
Toolpath Type	Tilting			
Holder	Machining type	5 axis (simultaneous)		
Cut Pattem     Tool Axis Control	Strategy	Normal to contour		
Linking	Direction	Z axis	(0) 🐼 😣	
Additional Settings	🕨 🗹 Tilt range	Minimum	0.0	
		Maximum	60.0	
				A CONTRACTOR OF A CONTRACTOR OFTA CONTRACTOR O
	Lead angle to cutting dire	ection	0.0	
Quick View Settings				
Tool 1/4 BALL EN A	Clearances			
Tool Diameter 0.25	Holder		0.1	
Comer Radius 0.125	Shank		0.025	
Feed Rate 110	ondrik			
Spindle Spe 8404	Shoulder		0.01	
Coolant On				
Toollength 175				

- 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.

Toolpath Type	Clearance plane	
Holder	Type Sphere	$\sim$
	Position Automatic	$\sim$
···· 🗸 Linking	Radius Automatic	~
	Rapid distance	0.8
	Feed distance	0.4
	Air move safety distance	0.4
	Smooth corners	0
	Links	
	Type Retract to clearance area	$\sim$
Quick View Settings		
Tool 3/16 BALL E. A	Lead in/out	
Tool Diameter 0.1875	Radius	0.12
Comer Radius 0.09375		
Feed Rate 150	Min. radius	<mark>0.04</mark>
Spindle Spe 8000		
Coolant On	Feed rate control	
Tool Length 0.75		
Length Offset 6	Replace Rapid with Feed	399
Diameter Of 6		
Colone / To Too		

Five-Axis-Lesson-8-40

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.

Components	Data Simu	lation					
Stock							
Stor	ck Setup						
	Min point:	Max point:	Margins:				
×	-1.3	1.3	0.0	Scan toolpath(s)			
Y	-1.1	1.1	0.0	Use Stock Setup values			
z	0.0	1.2	0.0	Pick stock corners			
OBox							
⊖ Cylir	nder						
Cyli	inder axis:	() X	⊖y ⊖z				
Cyli	inder diameter	0.0	✓ Cent	er on axis			
	d	Ø					
OFile		~>		Brow	Se		
		Charling Charle		~			
Ostoc	ck model	Starting Stock		×			
Fixt	ures						
<ul> <li>Leve</li> </ul>	els	Number	Name				
			1 Part				
			Surface				
			6 Curve geometry				
			20 Parts For 3D printing     100 Vise+Riser+Fixture				
				-			
🔾 Solid	ds	B					
() File					se		

- 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.

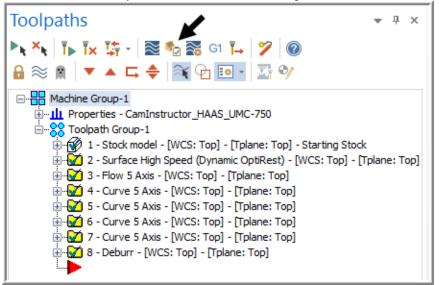
Components	Data	Simulation				
Machir	ne					
Can	Instruct	or_HAAS_UM	C-750		$\sim$	Post Settings
Positio	n					
A., b						
Aut	omatic			$\sim$		

- 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**.

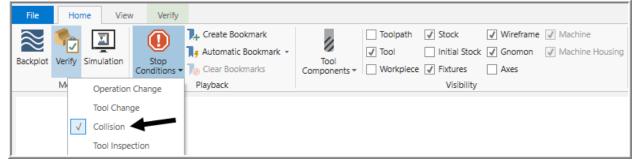
Components Data Simulation	
Machine 5_5AXGEN_VMCTTAB	Post Settings
Automatic	

### 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.



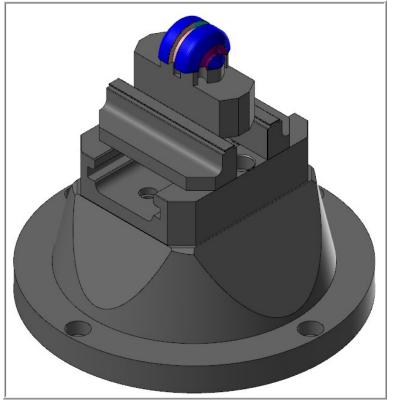
3. On the Verify tab, we've enabled Color Loop and Show Edges.

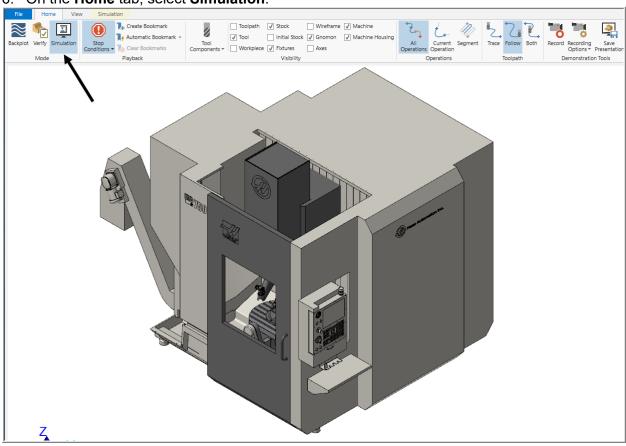


4. Click on **File>Options** and then on the **General** menu, enable **Collision Checking** with the **Tool to the Fixture**. Click **Ok**.

		_
General Graphics	General options for working with Mastercam Simulator.	^
	User Interface	
	Overlay Control Opacity: 📓 🛼 🗍 🗾 50.00 % 🗘	
	Display	
	Number of moves before:     10       Number of moves after:     10	
	Tolerances	
	Collision tolerance:     0.006       Save STL tolerance:     0.001	
	Collision Checking	
	Tool:   Fixture  Machine  Workpiece  Stock  Mill tool holder (Verify only)  Mill tool shank (Verify only)  Mill tool shoulder (Verify only)  Mill tool cutting length (Verify only)  Mill tool cutting length (Verify only)	

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.





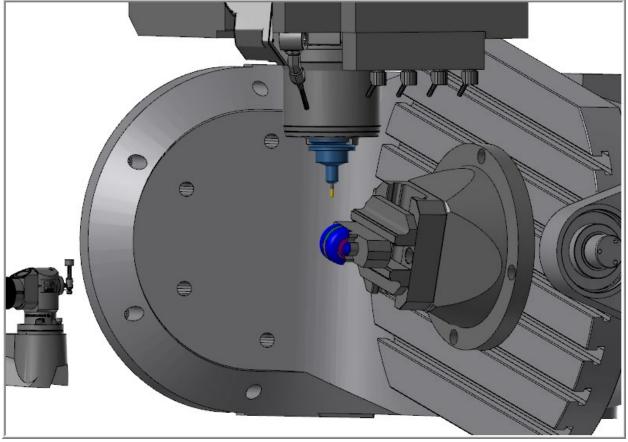
The verification did not detect any collisions. Switch to Simulate
On the Home tab, select Simulation.

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.

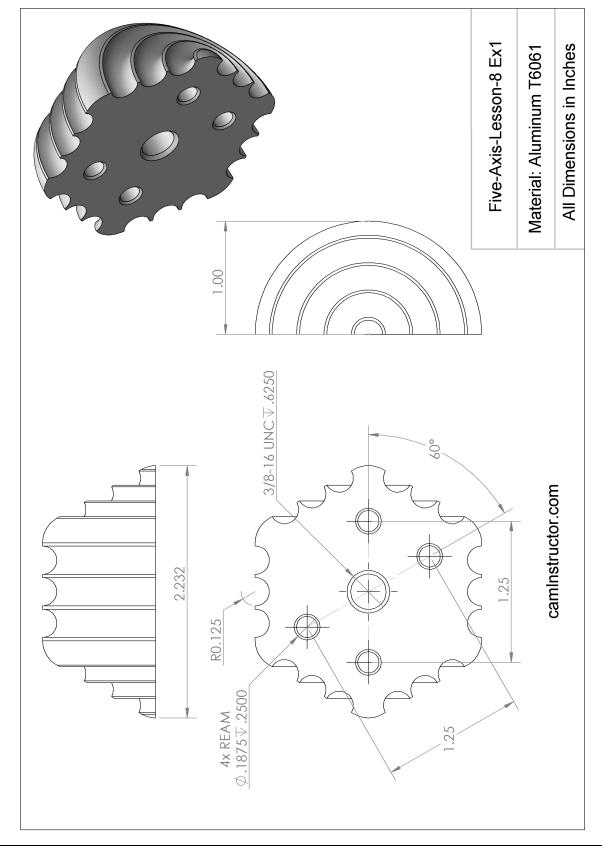


- 8. The Simulation will accurately show all machine movement as the actual machine will make during the machining process. This Simulation will take longer than the Verification did.
- 9. You can speed up or slow down the process by adjusting the Performance/Precision slider and/or the Slow/Fast slider.

➔ The completed Machine Simulation.

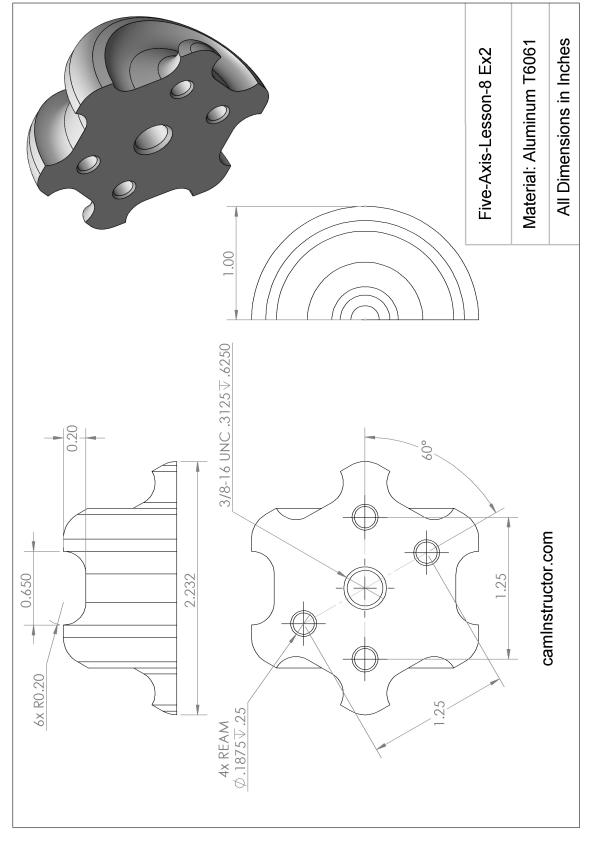


### Five-Axis-Lesson-8-Ex1



Five-Axis-Lesson-8-48

# Five-Axis-Lesson-8-Ex2



Five-Axis-Lesson-8-49