Tutorial 11: Machining a Ring



Introduction

This tutorial will illustrate machining this Ring using 4 axis-milling operations.

This tutorial will introduce the usage of several 4-axis operations such as 4 axis roughing and finishing.

The stepped instructions are accompanied by explanatory and introductory text. Reading this text will help you understand the tutorial methodology and provide information about additional options available.

Don't forget to save your work periodically! You may want to save the file under a different name so that the original file will be preserved.

Strategy to Machine the ring

- We will machine the ring completely using 4 axis-machining operations.
- The part itself will be machined out of a cylindrical blank.
- The stock will be held to the machine table using a rotary chuck.
- The part will be machined using 0.125", 0.0625", and 0.03125" ball end mills.

Main Programming Steps

In creating programs for each setup, the following steps will be followed:

- 1. Create the Stock geometry
- 2. Set the Machine zero point respect to the machine coordinates.
- 3. Set the rotary axis and rotary center.
- 4. Create / Select the tool used for machining
- 5. Set the feeds and speeds
- 6. Set the clearance plane for the non-cutting transfer moves of the cutter
- 7. Select the machining operations and set the parameters
- 8. Generate the toolpath
- 9. Simulate the toolpath.

You may have to repeat either all or part of these steps for subsequent operations.

Loading the Part Model

"Part" refers to the geometry that represents the final manufactured product. Typical you would create this in Alibre Design. Use the Alibre Design menu bar or the Standard toolbar buttons to create, load and save part geometry.

Getting Started with Alibre CAM

- 1. Select File / Open Part, or click the Open Part icon from the Alibre Design standard toolbar.
- From the **Open** dialog box, select the **RingExample_1.AD_PRT** file from the **Tutorials** folder in the Alibre CAM installation folder. (Default location C:\Program Files\MecSoft Corporation\Alibre CAM 2.0\Tutorials)

The loaded part appears as shown below.



Note: You must work in shaded mode in order to be able to visualize toolpaths created in Alibre CAM. It is suggested for best visual performance with Alibre CAM to work with only one view port open and the view port operating in shaded mode.

Loading the Alibre CAM Browser

1. Select Alibre CAM from the menu bar and click Machining Operations Browser.

File	Edit	View	Insert	Sketch	3D Sketch	Feature	Tools	Alibre CAM Window Help
🍓 - 🔁 🗒 🚰 🍓 👗 睧 🎕 X いつ			Machining Operations Browser					
Desig	n Exp	lorer						Cutting Tools Browser

RingExample_1 - Alibre Design Professional File Edit View Insert Sketch 3D Sketch Edit	eature Tools	Alibre CAM	Window	Help	_ • •
1 1 2 2 1 2 1 2 1 2 1 2 1 2 1 2 1 2 1 2	າ ດ [ູ]	Albre CAM	window	Theip	
Alibre CAM Alibre CAM Image: Setup Imag					
Design Explorer Alibre CAM					

The Alibre CAM browser (MOPs and Cutting Tools) is now loaded and is docked over the Design Explorer. You can toggle between the MOPs browser and the Design Explorer from the tabs at the bottom of the window. It can be hidden by un-checking Browser on the Alibre CAM menu bar. To re-display the hidden Browser window, you can re-check **Browser** from the Alibre CAM menu entry. You can also resize it by dragging one of its sides.

Preparing the part for Machining

The Setup tab allows the user to specify Machine Setup, Select Post Processor, Stock Geometry, Machine Coordinate System (Machine Zero) & Preferences.

Setup Tab

1. Go to the Alibre CAM MOps browser and click on the Setup tab



2. Select Machine Setup from the setup tab.



3. Set the Machine type to 4 Axis and Rotary Axis to X Axis. For most controllers rotation along X represents A axis and rotation along Y represents B axis. We will set the Rotary Center once we determine the Machine Zero.

achine			2
Machine Setup			
Machine Type			
🔿 3 Axis	🧕 4 Axis	🔿 5 Axi	is
Tool Change Position	,		
× O	► Y 0	≥ Z 0	
- 4th Axis (Primary Axis)		
Rotary Center: X 0	Y 0	🚔 Z 0	
Rotary Axis: X 1	X Axis O Y Axis Y 0	Specify	
- 5th Axis (Secondary /	Axis)		
Rotary Center: X 0		🔶 Z 0	÷ 🖓
Rotary Axis: X	Y 1	z 0	
Gage Length	0	vordinate Sustem	
	ramates in notated CC	-ordinate bystelli	
	OK	Cancel	Help

4. Select **Post** from the setup tab to specify the post processor options



5. Set the current post processor that is on your controller. We will select Haas as the post processor for this exercise. Set the posted file extension type to .nc

Note: By default post processor files are located under C:\Program Files\MecSoft Corporation\Alibre CAM 2.0\Posts The program to send the posted output is set to notepad. This would output the G code to a notepad.

Set Post-Processor Options	3
Set Post-Processor Options	
Select Post Processor Current Post Processor: Haas Folder where post-processor files are located: C:\Program Files\MecSoft Corporation\Alibre CAM 2.0\Posts	
Program to send posted file to	
Options Posted File Extension: Inc	
OK Cancel Help	

Create Stock Geometry

1. Select Create/Load stock from the setup tab and create a Cylinder Stock.



The stock model information dialog may be displayed when a stock geometry is created.

Stock Model Information						
Important notes about stock models:						
 Once the stock is created, the 3-D bounding box of the stock model will be rendered in the Alibre Design graphics window. Please note that this is not the actual stock model. The actual stock model will be displayed only in the simulation window. 						
 Also make sure that you use the Polygonal Stock model when working with rotated MCS operations. The Voxel Stock model can only be used when the MCS is parallel to the global XYZ system. You can choose the simulation model tupe in the Simulation Settings dialog. 						
 Please note that Cut Material Simulation of rotated Machining Operations is available only in the Pro and the Expert configurations of Alibre CAM. 						
Do not show this dialog again.						
OK Cancel Help						

Click OK

User can turn off this dialog by selecting Do not show this dialog again located on the bottom of the message window.

To display this dialog during stock creation, select Alibre CAM Preferences->Simulation Preferences and select Invoke 'Stock Model Information' dialog.

2. This brings up the Cylinder Stock parameters. Set the Axis (rotary) = X, Radius = 0.52'' and Length (L) = 0.6. Leave the other parameters as default and click OK.

Cylinder Stock	×
Stock Geometry	
Center (Xc,Yc,Zc) Radius	Axis
Axis • X C Y	z
Center Xc -0.3 Yc 0 Zc 0.124	105 <u>·</u> k
Dimensions Radius 0.54 • Length 0.6	•
Copy Model Bound	ding Cylinder
OK Cancel	Help

3. The stock geometry is now created, and a semi-transparent stock is displayed on top of the part geometry.



Note: The stock model created in Alibre Design Graphics window is a 3d bounding box of the stock dimension. The actual stock model will be displayed only in the stock simulation window.

4. You must switch the simulation model to Polygonal model to run 4 axis simulations. Select Preferences->Simulation Preferences from the Setup Tab and switch the simulation model to Polygonal if set to Voxel.



Set Simulation Preferences						
Simulation Model						
C Voxel Model Polygonal Model						
Simulation Speed Min Max Maximum Display Interval: 100						
Simulation Accuracy Standard Medium Fine						
Stock Model Transparency						
Opaque Transparent						
Simulation Mode						
Tool Holder Display						
Display Tool Holder During Simulation						
Tool Display						
Stock Model Information						
Invoke 'Stock Model Information' dialog.						
Run Simulation After Regeneration						
OK Cancel Help						

The setup tab now displays the following information: Machine Type, Post Processor, and Stock type as show below.



Align Part and Stock

Once the stock model is created, user can move the stock geometry relative to the part geometry and use the stock box to specify the machine zero (home position).

1. Select Align Part and Stock from the Setup tab



2. Set Z alignment to **Center** and XY alignment to **Center.** (This would align the stock to the top of the part in Z and center in XY)

Align Part and Stock	Models	\sim
Align Part and Stock		
Z Alignment	💽 Center	OBottom
XY Alignment		
O North West	O North	O North East
O Mid-West	💿 Center	O Mid-East
O South West	O South	O South East
	ок	Cancel Help

Set Machine Coordinate System (MCS)

The steps below help you determine the machine home (also known as machine zero or tool touch off point) for the part/stock geometry.

The MCS is represented as a triad with 3 arrows, the Green arrow indicating X axis, Blue the Y axis and Red the Z axis. It is always a good practice to set the machine zero before generating a toolpath.

3. Select Set MCS from the Setup tab



4. Switch to SetMCS Origin tab and choose **Set to Stock Box**, the Zero Face to **Highest Z**, and Zero Position to **East** corner. This sets the machine home to the top of the stock material and the right most edge of the part geometry.

Set MCS	×
Set Machining Coordinate System	
Aign MCS With Set MCS Origin Set Fixture Offsets	
C Pick © Set to Stock Box C Set to Part Box Zero Face	
Zero Position	
O North West O North O North East	
O West O Center ⊙ East	
C South West C South C South East	
X 0.3 [▲] Y 0.000138 [▲] Z 0.664037 [▲] B	
Generate Cancel Save Hel	p

Click Save As to save the work and specify a file name as RingExample_1-Rev1.



Note: You can change the stock model transparency under standard mode by selecting Simulation Preferences that is located at the bottom of the MOps browser.

Specify Rotary Center

In this step we will determine the rotary center for the part geometry. The rotary center must pass thro' the entire part geometry. Alibre CAM will not compute a toolpath if the part/feature is below the rotary center as this is considered as an undercuts in the part.

- 1. Select Machine Setup from the setup tab.
- 2. Set the rotary center in X =0, Y = 0 and Z = -0.54 which is the center of the stock geometry.

chine		
fachine Setup		
Machine Type)
🔘 3 Axis	💿 4 Axis	🔿 5 Axis
Tool Change I	Position	
	X 0 Y 0	≠ z 0 ≠ 🕏
4th Axis (Prima	ry Axis)	
Rotary Center:	X 0 🚔 Y 0	🚔 Z -0.54 🚔 🔖
Rotary Axis:	• X Axis X 1 Y 0	O Specify Z 0
- 5th Axis (Seco	ndary Axis)	
Rotary Center:	X 0 Y 0	🗘 Z 0 🗘 🖓
Rotary Axis:	X 0 Y 1	z 0
Gage Leng	h 25	
Output a	III Co-ordinates in Rotated Co-o	ordinate System
	ОК	Cancel Help

Create Tools

To machine the above part, we will now create a 0.125", 0.0625" and 0.03125" Ball End Mills.

1. Go to the Cutting Tools browser that is located below the Alibre CAM MOps browser and select Create/Edit Tools. Select the Tool Type to Ball End Mill.

Y 🞒	21	1	37	×
Nor To	ols			

Create/Select Tool								
Ţ	T	T	T	T				

2. Set the tool name as BallMill-0.125 and Tool Diameter = 0.125. Under the Properties tab, set Tool Number = 1.

Getting Started with Alibre CAM

Create/Select Tool		×
l l l l l l l l i	III2 V V T V	TIII
Tools In Library	Name BallMill-0.125	Properties Feeds & Speeds
	1	Material HSS
	Holder Diameter -+	Number of Flutes 2
	Length	Tool Number 1
	1.5	Adjust Register 0
	₹	Cutcom Register 0
	Length Length	Zoffset 0
	1	Coolant None 🖌
		Comments
	Diameter 0.125 💌	
	Save as New Tool Save	Edits to Tool
	OK	Cancel Help

Setting Feeds and Speeds

You can assign Feeds & Speeds to a tool or you can load from a table. In this exercise, we will assign feeds and speeds to the tool.

- 3. Switch to the Feeds & Speeds tab inside the create/select tool dialog.
- 4. Use the following settings for feeds and speeds.

Spindle Spe	ed 5000	🚔 ВРМ
Feed Rates		
Plunge:	20	🚔 in/min
Approach:	20	🚔 in/min
Engage:	20	🚔 in/min
Cut:	30	🚔 in/min
Retract:	20	🚔 in/min
Departure:	20	🚔 in/min
Transfer Fe	edrate (Tf)	
💿 Use Ra	pid	
🔘 Set	50	🔶 in/min

5. Click **Save as New Tool**. The tool is now created and listed under Tools in Library.

Note: You can edit the tool properties and click Save Edits to Tool to save the changes. You can create additional tools by assigning a different name and specify the tool parameters.

- 6. Create a 2nd Ball End Mill with the following parameters.
 - a. Tool Name: BallMill-0.0625, Tool Number = 2.
 - b. Switch to Feeds & Speeds tab set Spindle Speed = 5000 rpm, plunge, approach & engage feed = 20 ipm, cut feed = 30 ipm, retract and departure feeds = 20 ipm. Set the Transfer Feedrate to Use Rapid.
 - c. Click Save as New Tool.

Getting Started with Alibre CAM

- 7. Create a 3rd Ball End Mill with the following parameters
 - a. Tool Name: BallMill-0.03125, Tool Number = 3.
 - b. Switch to Feeds & Speeds tab set Spindle Speed = 5000 rpm, plunge, approach & engage feed = 20 ipm, cut feed = 30 ipm, retract and departure feeds = 20 ipm. Set the Transfer Feedrate to Use Rapid.

The created tools are now listed under the Cutting Tools browser.



Create Machining Operations

We will machine the ring using 2 different machining operations – 4 axis Roughing and Finishing.

The first step in machining the ring will be a roughing operation. In this cut method, the tool cuts the stock in successive levels. The spacing between these levels are specified by the user. This type of machining is very efficient for removing large volumes of material and is typically performed with a large tool. Roughing is typically followed by semi-finishing or finishing toolpaths.

Switch to the Create Operations tab in Alibre CAM Mops browser.



4 axis Roughing

1. Select 4 Axis Milling and choose Roughing.

Setup 🔁 Create 🐼 Sir	ulate	
10 10 - 11 - 1 - 11 - 11 - 11 - 11 - 11	64	- 🕹 -
😑 Machining Operations		4th Axis <u>O</u> ptions
Machine - 4 Axis	@ +	<u>R</u> otate Table
🧐 Post - Haas	Ŷ	4th Axis F <u>a</u> cing
🖻 🕞 MOp Set 1	-	4th Axis <u>P</u> ocketing
Set MCS	-N.	4th Axis Profiling
	۲	4U i Axis <u>R</u> oughing 💦
😒 🛷 💀	Ø	4th Axis Einishing
	Ø	4th Axis Engraving

If the rotary center is not set to the same location as the Machine Zero, a warning message dialog would be displayed at all times when a 4 axis machining operation is selected. Users can override this message by clicking OK in the dialog.

otation Center Warning!	
The Rotation Center does not lie of Make sure your machine can han	on either the X or the Y axis! dle this condition.
If not, set the Rotation Center con	Cancel

Note: You can check Do no show this dialog again to stop the warning message appearing again when you create/edit a 4 axis machining operation.

- 2. This brings up the 4 Axis Roughing Operation Dialog. We will now go over the steps for creating the toolpath.
- 3. Switch to the Tools tab inside the 4th axis Roughing operation and select BallMill-0.125.

Getting Started with Alibre CAM

Clearance	Cut Parameters			Step	Down Control
achining Features/R	egions	Tool		F	eeds & Speed
.			Tool Cor	makeu	,
			Diameter	meny	0.125
BallMill-0.1	25		Corper D	seline	0.120
¶∕ BallMill-0.0	625		Taner	aalas	0.0020
T BallMill-0.0	3125		Tip Apole		0
			Tool Pro	nertie	0
			Tool Nam	Perue	BallMilL0 125
			Tool #	×	1
			# of Fluts	20	2
			Culcom P	enicher	0
			Adjust Re	aicher	0
			7-Offset	giscor	0
			Material		HSS
			Coolant		None
			Comment	2	none
			Feeds &	Sneed	5
			Spindle St	need	5000
			Feed Rat	e	30
			rooditat	<u>с</u> ,	00
		_			
			Edit/0	Create/S	elect Tool
		2			
				Preview	v Tool

4. Click on the Feeds and Speeds tab and select Load From Tool. Alibre CAM will now get the feeds and speeds information that was set when the tool was defined.

Clearance	0	ut Paramet	ers	Step Down Control
Machining Fe	atures/Regions		Tool	Feeds & Speeds
Spindle Speed -				
	5000	RPM		
Feed Rates				
Plunge (Pf)	20	in/min		
Approach (Af)	20	in/min	Pf	
Engage (Ef)	20	in/min		Cr
Cut (Cf)	30	in/min	Af	
Retract (Rf)	20	in/min	Ef	R
Departure (Df)	20	in/min		
Transfer (Tf)				
💿 Use Rapid				
◯ Set	50	in/min		
Lo	ad From Tool	N		
Load	From Table	N 2		
2000				

Getting Started with Alibre CAM

5. Switch to the Clearance Tab and set the Clearance Plane Definition to Automatic and Cut Transfer Method to Clearance Plane.

reactioning reaction	Regions Tool	Feeds & Speeds
Clearance	Cut Parameters	Step Down Control
learance Plane Definiti	on	
 Automatic 		Part Max R
◯ Part Max R + Dist	0.25	
◯ Stock Max R + Dis	t 0.25	
🔿 Absolute Z Value	0.25	
Olearance Plane		

Specify Cut Parameters

- 6. Click on the Cut Parameters tab.
- Set the Intol and Outol = 0.001, Stock to leave =0.005, Cut Pattern to Across Axis, Zig Zag and Low to High, Cut Containment Low Value = -0.6, High Value = 0 (as the machine zero is set to the right edge of the stock/ part), Step over distance = 25 (% Tool Diameter).

Machining Feature	es/Regions	Tool	Feeds & Speed
Clearance	Cut Param	eters	Step Down Control
⊂ Global Parame	eters		th
Intol	0.001	1001pa	Stock Outtol
Outol	0.001	Intol 🛓	
Stock	0.005	F	Part Geometry
- Cut Pattern-			
Across Axi	s 🔿 Alona Axis		
07:-7	07-		~
€Zig∠ag	O∠ig		102
Sow to Hig	ıh 🔘 High To Low	V.V	(/// .xe
Note: These region contain Low Value (L) -0.6 Start Angle (S) 0	settings are ignored if iment is selected High Value (H) 0 End Angle (E) 360		
C Stepover Cont	rol		+
💿 % Tool Dia	meter 25	÷	
O Distance	0.125		Stepover

8. Switch to the Step Down Control Tab.

Step Down Control

- 9. Use the Following Settings.
 - a. Cut Levels, Check Top (T) and specify Top (T) = 0.53 (radius of stock material)
 - b. Step Down Control (dR) = **75** (% Tool Diameter).

	s/Regions	Tool	Feeds & Speed
Clearance	Cut Par	ameters	Step Down Control
- Radial Cut Lev	els		
Top (T)	0.53		
🔲 Bottom (B)	0.12631		
Stepdown Con	trol (dR)		(♥)))
⊙ % Tool Diar	neter 75		\leq
O Distance	0.25		

10. Click **Generate**. The 4th axis Roughing toolpath is now generated, and the Operation is listed under the Alibre CAM MOps browser.



Simulate Toolpath

The generated toolpath can now be simulated. Make sure to turn on Stock Visibility under the Simulate tab.



1. Switch to the Simulate tab in the Alibre CAM -MOps browser.



2. Select 4th Axis Roughing Operation and click Simulation window.

to launch the Alibre CAM Stock

Getting Started with Alibre CAM

3. Click Simulate from the Stock Simulation window to run simulation.

The simulated part is as shown below.

Note: You can adjust the simulation speed by selecting Simulation Preferences that is located to the bottom right corner of the Simulate tab Alibre CAM-MOps browser or from the Stock simulation window.



4. Once the simulation is complete, you can close the Stock Simulation window and return to the Alibre CAM browser.

4 axis Finishing

We will now use 4th Axis Finishing operation to finish the part using a 0.0625" Ball End Mill.

In this method, the tool is always normal to the axis of table rotation (continuous mode). The tool motions can be parallel to or normal to the rotation axis.

1. From the Create Operations tab, select 4 axis Milling and 4 Axis Finishing.



This brings up the Finishing Operations dialog. We will go over the steps for creating the pocketing operation.

2. Switch to the Tools tab inside the 4 Axis Finishing operation dialog and select BallMill-0.0625.

h Axis Parallel Finishing			
Machining Features/Regions Tool	Feeds & Spee	ds Clearance	Cut Parameters
		Teel Constant	
		Tool Geometr	y 0.0005
🕀 🔐 📅 BallMill-0.125		Diameter	0.0625
T BallMill-0.0625		Corner Radius	0.03125
۲۶		Taper	0
		Tip Angle	0
		Tool Propertie	15
		Tool Name	BallMill-0.0625
		Tool #	2
		# of Flutes	2
		Cutcom Register	0
		Adjust Register	0
		Z-Offset	0
		Material	HSS
		Coolant	None
		Comments	
		Feeds & Speed	ds
		Spindle Speed	5000
		Feed Rate	30
		Edit/Create/	Select Tool
		Previe	w Tool
Genera	ite Cano	el Sav	e Help

- 3. Click on the Feeds and Speeds tab and select Load From Tool. Alibre CAM will now get the feeds and speeds information that was set when the tool was defined.
- 4. Switch to the Clearance Tab and set the Clearance Plane Definition to Automatic and Cut Transfer Method to Clearance Plane.

Specify Cut Parameters

- 1. Click on the Cut Parameters tab.
- Set the Tolerance to 0.001, Stock to leave =0, Cut Pattern = Along Axis, Zig Zag, Low to High, Set Cut Containment Low Value = -0.5, High Value = -0.1, Step over distance = 5 (% Tool Diameter).

4th Axis Parallel Finishing 🛛 🔀
Machining Features/Regions Tool Feeds & Speeds Clearance Cut Parameters
Global Parameters D.001 Intol 0.001 Outol 0.001 Stock 0
Cut Pattern Across Axis ZigZag ZigZag Low to High High To Low Cut Axial Containment Note: These settings are ignored if region containment is selected Low Value (L) High Value (H) 0.5 O Start Angle (S) End Angle (E) 0 360
Stepover Control
Generate Cancel Save Help

3. Click **Generate**. The Finishing toolpath is now generated, and the Operation is listed under the 4th Axis Roughing Operation in the Alibre CAM MOps browser.

Getting Started with Alibre CAM



Simulate Toolpath

The generated toolpath can now be simulated. Make sure to turn on Stock Visibility under the Simulate tab.



1. Switch to the Simulate tab in the Alibre CAM -MOps browser.



2. Select 4th Axis Parallel Finishing Operation and click by to launch the Alibre CAM Stock Simulation window.

3. Click Simulate from the Stock Simulation window to run simulation.

The simulated part is as shown below.

Note: You can adjust the simulation speed by selecting Simulation Preferences that is located to the bottom right corner of the Simulate tab Alibre CAM-MOps browser or from the Stock simulation window.



4. Once the simulation is complete, you can close the Stock Simulation window and return to the Alibre CAM browser.

4th Axis Finishing Operation #2

We will now create a second 4 Axis finish operation to machine the prongs and pocket areas around the ring by limiting the toolpath using start, end angle cut containments.

Copying a MOp

- 1. Switch to the Create Operations tab in the Mops Browser.
- 2. Select the 4 Axis Parallel Finishing, right click **copy**.

(]

Ж

Information

Edit

Cut

⊆opy

Paste

Rename

Shop Documentation

μł

- 🖮 🕞 MOp Set 1 MOp Set 1 🛃 Set MCS 💒 Set MCS 4th Axis Roughing 4th Axis Roughing 4th Axis Decellal Finishing Ē 4H Ē <u>R</u>egenerate <u>R</u>egenerate G, Post Post 3 Simulate Simulate Simulate Until Simulate Until
- 3. Now right click on 4 Axis Parallel Finishing and select **paste**.

4. A copy of the finish operation is created below the 4th Axis Finishing operation. The operation name is labeled 4th Axis Parallel Finishing-1 and is as shown below.

0

Ж

Information

Edit

C<u>u</u>t

⊆ору

Paste

Rename

Shop Documentation

h



Specify Cut Parameters

- 5. Double click on 4th Axis Parallel Finishing-1 to edit the operation.
- 6. Switch to the tools tab and select **BallMill-0.03125**.
- 7. Switch to the Cut parameters tab and use the following parameters.

4th Axis Parallel Finishing 🛛 🛛 🔀
Machining Features/Regions Tool Feeds & Speeds Clearance Cut Parameters
Global Parameters Toolpath
Intol U.U01 Stock Outtol
Outol 0.001
Stock 0
Cut Pattern
🔿 Across Axis 💿 Along Axis
O ZigZag O Zig
⊙ Low to High ◯ High To Low
Cut Axial Containment
Note: These settings are ignored if region containment is selected
Low Value (L) High Value (H)
-0.5 -0.1 Do not cut helow:
Start Angle (S) End Angle (E)
Stepover Control
O Distance 0.125
-+ I+ Stepover
Generate Cancel Save Help

8. Click Generate. The 4th Axis Parallel Finishing-1 toolpath is now generated, and the Operation is listed under the MOps browser.



Simulate Toolpath

The generated toolpath can now be simulated. Make sure to turn on Stock Visibility under the Simulate tab.



- 1. Switch to the Simulate tab in the Alibre CAM -MOps browser.
- 2. Select 4th Axis Parallel Finishing-1 Operation and click to launch the Alibre CAM Stock Simulation window.
- 3. Click Simulate from the Stock Simulation window to run simulation.

The simulated part is as shown below.

Note: You can adjust the simulation speed by selecting Simulation Preferences that is located to the bottom right corner of the Simulate tab Alibre CAM-MOps browser or from the Stock simulation window.



4. Once the simulation is complete, you can close the Stock Simulation window and return to the Alibre CAM browser.

Post Processing

1. Select Machining Operations from the Create Operations tab and right click and select post process.

Setup 📌 Create 🖉 Simulate		
🖻 🥵 - 📢	<u>-</u> 7	• 😼 • 🗞 • 💑 •
Archining		
📮 🏀 Setu	, ,	<u>R</u> egenerate All
- <u>C</u>	18 🏐	
s s	o ita 🔛	N Save to Knowledge Base
🖨 🌈 MOp :	s 🌒	Information
⊕ · 🍎 4	tf 😥	Shop Documentation
⊕… 🃁 4 ⊕… 芦 4	tt tt 🔏	<u>D</u> elete All

2. Specify the File Name as **Ring.nc** and click save.

The post by default is set to Haas as specified under the Post processor setup. You can change the post processor by selecting a different one from the drop down menu in the list. The posted g code by default will be saved to the folder where the part file is located.

Lista Studio srl www.lista.it info@lista.it