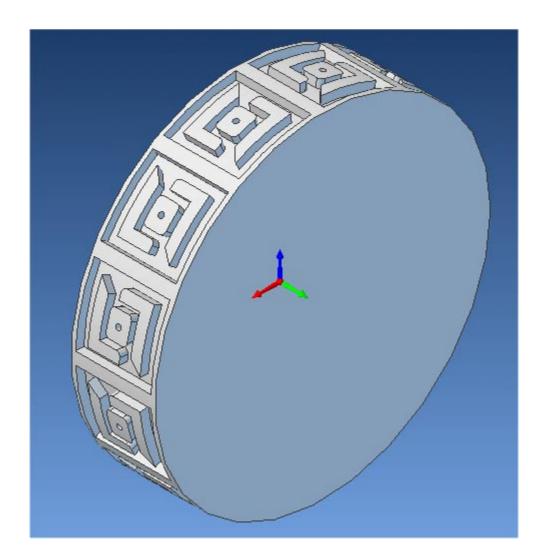
Getting Started with Alibre CAM

Tutorial 13: Pocketing and Drilling on a Ring



Introduction

This tutorial is intended to describe the 4 axis pocketing and hole making operations. Pocketing machines closed regions as if they were pockets - completely enclosed by inner and outer regions. The tool cannot go beyond the outer region, and cannot go within inner regions. This is unlike **Facing**, in which the outermost region is considered to enclose material to be removed. Hole making operations are used to create holes in a part; the hole types varying from simple drill holes, counter sunk holes, through holes to tapped and bored holes. Here you will learn to drill simple holes.

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 part

- 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 a 0.0625" Flat End Mill and 0.125" Standard Drill

Main Programming Steps

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

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

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

Loading the Part Model

Getting Started with Alibre CAM

"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.

- 1. Select File / Open Part, or click the Open Part icon from the Alibre Design standard toolbar.
- From the Open dialog box, select the 4AxisPocketing_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)

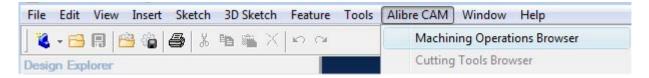
The loaded part appears as shown below.



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.



4AxisPocketing_1 - Alibre Design Professional File Edit View Insert Sketch 3D Sketch F	esture Tools	Alibre CAM	Window	Help	_ • •
		Allbre CAM		nap	
Alibre CAM					
Design Explorer Alibre CAM					

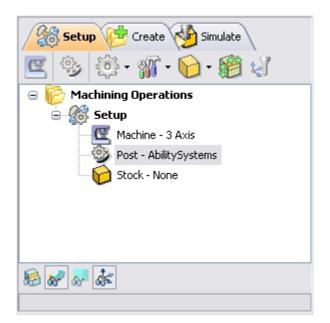
The Alibre CAM browser (MOPs and Cutting Tools) is now loaded and is docked over the Design Explorer. You can toggle between the MOPs browser and the Design Explorer from the tabs at the bottom of the window. It can be hidden by un-checking Browser on the Alibre CAM menu bar. To re-display the hidden 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			
Machine Setup			
Machine Type]
🔿 3 Axis	🧕 4 Axi	s O	5 Axis
- Tool Change P	osition × 0 ♀ Y	0 🚔 Z O	
- 4th Axis (Primar	y Axis)		
Rotary Center:	X 0 🚔 Y	0 🚔 Z 0	- k
Rotary Axis:	● X Axis ● Y X 1 Y		ecify
- 5th Axis (Secor	idary Axis)		
Rotary Center:	X 0 🔶 Y	0 🗘 Z 0	÷ 😽
Rotary Axis:	X 0 Y	1 Z O	
Gage Length	0	ed Co-ordinate Syste	m
	10	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
Set Post-Processor Options
Select Post Processor Current Post Processor: Haas Edit 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:
OK Cancel Help

Create Stock Geometry

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

Setup Create	💯 Simulate
🖪 🧐 ۞ • 🎢 •	
😑 📂 Machining Operatio	ins
	Cylinder Stock

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

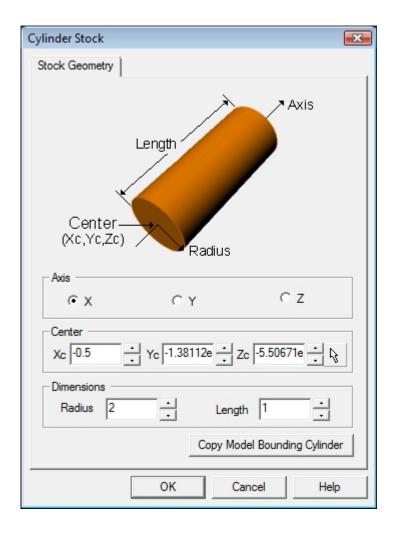
Stock Model Information
Important notes about stock models:
 Once the stock is created, the 3-D bounding box of the stock model will be rendered in the Alibre Design graphics window. Please note that this is not the actual stock model. The actual stock model will be displayed only in the simulation window.
2. 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 type 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

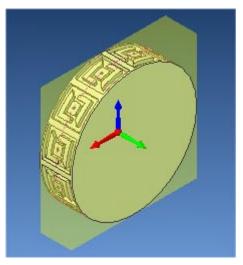
User can turn off this dialog by selecting Do not show this dialog again located on the bottom of the message window.

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 = 2'' and Length (L) = 1". Leave the other parameters as default and click OK.

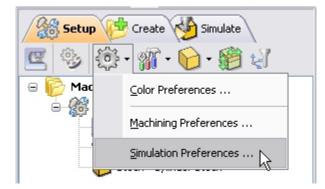


The stock geometry is now created, and a semi-transparent stock is displayed on top of the part geometry.



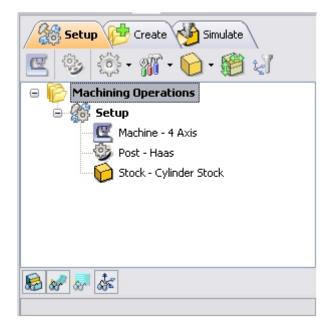
Note: The stock model created in Alibre Design Graphics window is a 3d bounding box of the stock dimension. The actual stock model will be displayed only in the stock simulation window.

3. 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 O Voxel Model
Simulation Speed Min Max 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 Solid C Transparent C Wireframe C None
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	
Align Part and Stock		
Z Alignment	 Center 	O Bottom
XY Alignment		
O North West	O North	O North East
◯ Mid-West	📀 Center	O Mid-East
O South West	🔿 South	🔿 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.

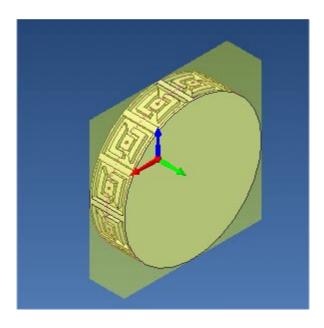
1. Select Set MCS from the Setup tab

198	👌 Setu	P 🔁	Create	🤡 Si	mulate		
E	-	203 -	% ·	0	1		
Θ [🁌 Ma	chining	Operat	ions		N	

2. Switch to SetMCS Origin tab and choose **Set to Stock Box**, the Zero Face to **Mid Z**, and Zero Position to **West** corner. This sets the machine home to the center of the stock material and the left most edge of the part geometry.

Set MCS	×
Set Machining Coordinate System	
Align MCS With Set MCS Origin Set Fixture Offsets	
C Pick Set to Stock Box Set to Part Box	
Zero Face C Highest Z Mid Z C Lowest Z	
Zero Position C North West C North C North East	
C South West C South C South East	
x -0.5 → Y 0.00050 → Z 2.22045€ → b	
Generate Cancel Save Hel	p

Click Save As to save the work and specify a file name as 4AxisPocketing_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.

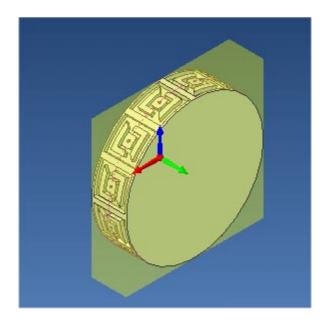
Specify Rotary Center

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, Y and Z = 0 which is the center of the stock geometry.

In this tutorial both the Machine Zero and the Rotary center are at the same location.

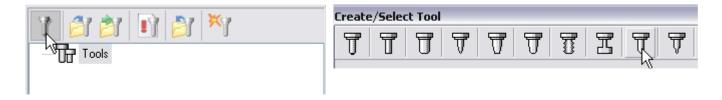
nchine				
Machine Setup				
Machine Type				
🔿 3 Axis	0	4 Axis	🔿 5 Axis	
Tool Change F	osition			
	× 0	Y O	🚔 Z 0	÷ 👌
4th Axis (Prima	ry Axis)			
Rotary Center:	X 0	Y 0	🚔 Z 0	÷ 🔊
Rotary Axis:	X Axis	OY Axis	O Specify	
	X 1	Y 0	Z O	
5th Axis (Seco	ndary Axis)			
Rotary Center:	X 0	Y O	÷ Z 0	\$
Rotary Axis:	X 0	Y 1	Z 0	
Gage Lengt	h O	* *		
Output a	Il Co-ordinates ir		rdinate System	
<u> </u>				
	ſ	ок	Cancel	Help



Create Tools

To machine the above part, we will now create a 0.125'' Drill (Standard Drill) and a 0.0625'' Flat End Mill.

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 Drill.



2. Set the tool name as **Drill-0.125**, Tool Diameter = **0.125**, Tip Angle = **120**, Flute Length = **1**,Tool Length = **1.5** Under the Properties tab set Tool Number = **1**

Create/Select Tool		
7 7 7 7 7 7 3	I I I V V T V	t t t
Tools In Library Drill-0.125	Name Drill-0.125	Properties Feeds & Speeds
	1 ÷	Material HSS 💌
	Holder Diameter Holder Length Flute Length Tool Length 1.5 Tool Length 1.5 Tip Angle 120 Tool Diameter	Number of Flutes 2 Tool Number 1 Adjust Register 0 Cutcom Register 0 Zoffset 0 Coolant None Comments
		e Edits to Tool Delete 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.

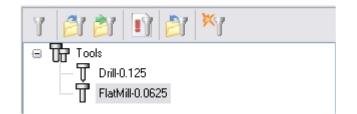
Spindle Spe	ed 5000	RPM
Feed Rates		
Plunge:	20	🚔 in/min
Approach:	20	🚔 in/min
Engage:	20	🚔 in/min
Cut:	30	🚔 in/min
Retract:	20	🚔 in/min
Departure:	20	🚔 in/min
Transfer Fe	edrate (Tf)	
💿 Use Ra	pid	
🔘 Set	50	🗧 in/min

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.

- 5. Create a 2^{nd} tool, a Flat End Mill with the following parameters.
 - a. Tool Name: **FlatMill-0.0625**, Tool Diameter = 0.0625, Flute Length = **1**,Tool Length = **1.5**, 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.

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



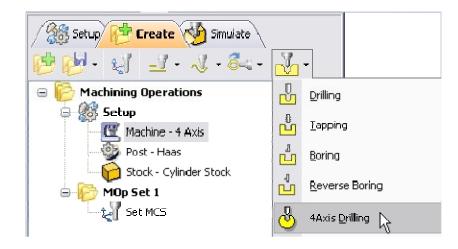
Create Machining Operations

First we will drill the holes on the cylinder using 4 Axis Drilling.

As in any other 4 axis operations, the tool is positioned normal (perpendicular) to the rotary axis. Once the holes (regions) are selected, the dialog boxes are similar to the 3 axis hole making operations. Sorting of holes is also possible to optimize the tool motion.

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

4 Axis Drilling



1. Select Holes and choose 4 Axis Drilling.

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.

The Rotation Center does not lie	on either the X or the Y axis!
Make sure your machine can han	idle this condition.
If not not the Potation Center out	rrectly using the Machine Tool definition dialog.

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.

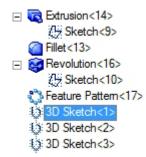
This brings up the 4 Axis Drilling Operation Dialog. We will now go over the steps for creating the toolpath.

Select Hole Features

2. Go to the Hole Features tab and click **Select Drill Points/Circles**. The drilling operation dialog is now minimized and allows selection of the sketch geometry.

ole F	eatures	Tool	Feed	s & Speeds	Clearance	Cut Parame	ters Sor	ting
#	Curren	t Hole F	eature(s) Selected	Minimum Maximun	Range Filter diameter filter Diameter: Diameter:	0	
						Select Dril	Points/Ci	
()				>				
×	Remove	All	XRem	ove Active				

3. Switch to Design Explorer and select **3D Sketch<1>**.

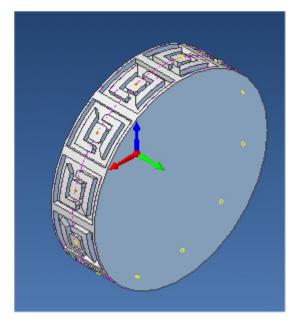


Note: You can also select the sketch from Alibre Design View. When select containment region is selected, Alibre CAM turns on Sketch selection filter to allow the user to select sketches. Hold the Shift key down to select multiple sketches.



The Drilling operation dialog comes back up displaying the selected regions. The selected regions are also highlighted on the part.

#	Selected Machining Region(s)
1	Region 1



5. Switch to the Tools tab inside the 4 Axis Drilling operation and select Drill-0.125.

Features Tool Feeds & Spe	eds Clearance Cut Parameters	s Sorting
Tools	Tool Geometry	y
T [Drill-0.125]	Diameter	0.125
E 101110.123	Corner Radius	0
4	Taper	0
	Tip Angle	120
	🖃 Tool Propertie	S
	Tool Name	Drill-0.125
	Tool #	1
	# of Flutes	2
	Cutcom Register	0
	Adjust Register	0
	Z-Offset	0
	Material	HSS
	Coolant	None
	Comments	
	E Feeds & Speed	ls
	Spindle Speed	5000
	Feed Rate	30
	Edit/Create/S	Select Tool
	Provio	w Tool
	Fievie	W 1001

6. 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.

lole Features To	ol Feeds & S	Speeds	Clearance	Cut Parame	ters Sorting
- Spindle Speed -	5000	RPM			
Feed Rates			3		
Plunge (Pf)	20	in/mi	n 📋 .		
Approach (Af)	20	in/mi	n Pf		
Engage (Ef)	20	in/mi		1	Cf Tf
Cut (Cf)	30 🚖	in/mi	n Af	Y	
Retract (Rf)	20	in/mi	n Ef	4	Rf
Departure (Df)	20	in/mi	n		
Transfer (Tf)					
💿 Use Rapid					
🔘 Set	50	in/mir	n		
	ad From Tool		=		
	From Table	—ĥ	<u>ک</u>		
112111221					

Getting Started with Alibre CAM

7. Switch to Clearance tab and set the Clearance Plane Definition to Stock Max R + Dist = **0.25**. Set the cut transfer method to Clearance Plane.

Hole Features Tool Feeds & Speeds Clearance Cut Parameters Sorting
Clearance Plane Definition
 Automatic Part Max R + Dist Stock Max R + Dist Absolute Z Value 25 10.25 10.25
Cut Transfer Method Skim Skim Clearance (C) Clearance Plane Clearance Plane
Generate Cancel Save Help

Specify Cut Parameters

- 8. Click on the Cut Parameters tab.
- 9. Set the Drill Type = **Standard Drill**, Drill Depth = **0.1**, **Check** Add Tool tip to Drill Depth and Approach Distance = **0.1**

Axis Drilling Hole Features Tool Feeds & Speeds Clearance Cut Parameters Sorting
Drill Types Standard Drill Depth Control Drill Depth 0.1 Image: Add Tool tip to Drill Depth Image: Add Tool tip to Depth Image: Add Too
Generate Cancel Save Help

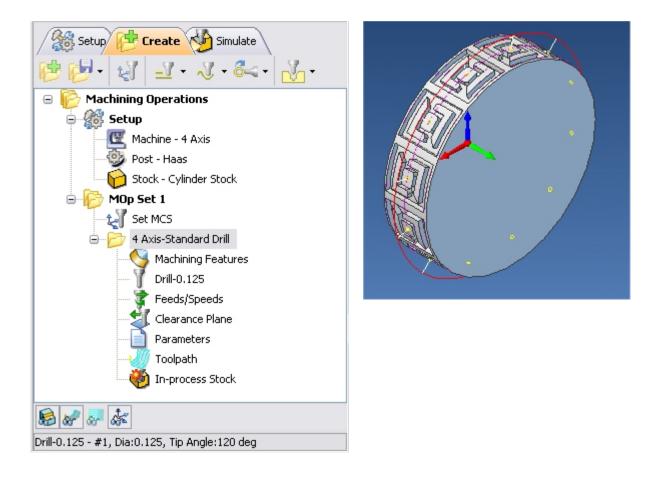
10. Switch to Sorting tab in the 4 Axis Drilling dialog.

Sorting

11.	Use	Minimum	Distance	Sort and	specify	the	Start	Point as	S Lower	Left

 Upper Left Oupper Right Lower Left Lower Right Directional Sort Primary Sort Direction (P) Start Angle (A) Secondary Sort Direction (S) Low to High High to Low Traversal Pattern Zig ZigZag 	 Lower Left Lower Right Directional Sort Primary Sort Direction (P) Start Angle (A) Secondary Sort Direction (S) Low to High High to Low Traversal Pattern 	Lower Left Lower Right Directional Sort Primary Sort Direction (P) Start Angle (A) Secondary Sort Direction (S) Start End
Primary Sort Direction (P) Start Angle (A) Secondary Sort Direction (S) Low to High O High to Low Traversal Pattern	Primary Sort Direction (P) Start Angle (A) Secondary Sort Direction (S) Low to High High to Low Start Point Traversal Pattern	Primary Sort Direction (P) Start Angle (A) Secondary Sort Direction (S) End
C Low to High C High to Low Start Point Point Point	C Low to High C High to Low Start Point Point Point	End
		Point Point

12. Click **Generate**. The 4 Axis Drilling 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 Drilling Operation and click Simulation window.

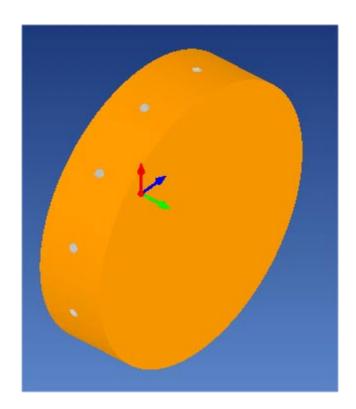
to launch the Alibre CAM Stock

Getting Started with Alibre CAM

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

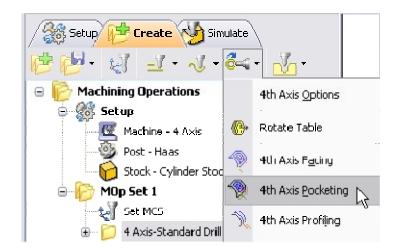
The simulated part is as shown below.

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 Pocketing



1. Select 4 Axis from the Create operations tab and choose 4 Axis Pocketing

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	dle this condition.
If not, set the Rotation Center corr	rectly using the Machine Tool definition dialog.
Do not show this dialog again	OK Help 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.

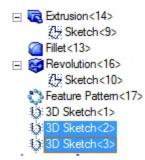
This brings up the 4 Axis Pocketing Operation Dialog. We will now go over the steps for creating the toolpath.

Select Machining Features/Regions

 Go to the Machining Features/ Regions tab and click Select Containment Regions. The 4th Axis Pocketing operation dialog is now minimized and allows selection of the sketch geometry.

4 Axis Pocketing	X
Cut Parameters Cut Levels Entry/Ex Machining Features/Regions Tool	it Advanced Cut Parameters Sorting Feeds & Speeds Clearance
# Selected Machining Region(s) # Selected Machining Region(s) * III * III # Currently Selected Avoid Regions	Select Containment Regions
 ✓ III ► ★ Remove All ★ Remove Active 	Avoid Region
Generate	Cancel Save Help

3. Switch to Design Explorer and select **3D Sketch<2> and 3D Sketch<3>**.

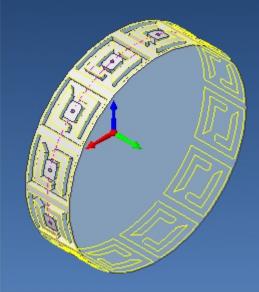


Note: You can also select the sketch from Alibre Design View. When select containment region is selected, Alibre CAM turns on Sketch selection filter to allow the user to select sketches. Hold the Shift key down to select multiple sketches.



The 4 Axis Pocketing operation dialog comes back up displaying the selected regions. The selected regions are also highlighted on the part.

#	Selected Machining Region(s)	
1	Region 1	
2	Region 2	1303
		_



5. Switch to the Tools tab inside the 4 axis pocketing operation and select FlatMill-0.0625.

Axis Pocketing				
Cut Parameters Cut Levels	Entry/Exit	Advanced Cut Par	ameters Sorting	
Machining Features/Regions	Tool	Feeds & Speeds	N. Contraction of the second s	
🖃 📅 Tools		Tool Geometry		
FlatMill-0.0625		Diameter	0.0625	
		Corner Radius	0	
~~		Taper	0	
		Tip Angle	0	
		Tool Propertie		
		Tool Name	FlatMill-0.0625	
		Tool #	2	
		# of Flutes	2	
		Cutcom Register		
		Adjust Register	0	
		Z-Offset	0	
		Material	HSS	
		Coolant	None	
		Comments		
		E Feeds & Speed	ls	
		Spindle Speed	5000	
		Feed Rate	30	
			Gelect Tool w Tool	
Ge	enerate C	ancel Sav	e Help	

- 6. 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.
- 7. Switch to the Clearance Tab and set the Clearance Plane Definition to Stock Max Z + Dist = **0.25** and Cut Transfer Method to Clearance Plane.

Specify Cut Parameters

- 8. Click on the Cut Parameters tab.
- 9. Set Tolerance = 0.001, Stock = 0, Cut Pattern = Offset Cuts, Cut Direction = Mixed, Start Point = Inside, Step Distance = 25% (Tool Diameter), Check Corner Cleanup.

	res/Regions	Tool	Feeds & S		Clearance
Cut Parameters	Cut Levels	Entry/Exit	Advanced C	ut Parameters	Sortin
Global Param			Region		
Toleranc	e: 0.001			/ / /	olpath
Stock	c 0		ſ	hord 🔨	
Compensation	AUTO/NON	EV		tolerance 🕴	•
			-	← Stock	
Offset Cu	ts 🔘 Line	ar Cuts () Spiral Cuts	🔘 Radial C	iuts
	ıt		Ţ		
- Step Dist	ance		+		1
💿 % Too	l Dia. 25				
O Distar	ice 0.1	25	→ + Stepov	er	
Corne	r Cleanup				
-					

- 10. Switch to Cut Levels Tab.
- 11. Use the following Settings
 - a. Location of Cut Geometry At Top.
 - b. Total Cut Depth = **0.1**, Rough Depth = **0.1**
 - c. Rough Depth/Cut = 0.025.
 - d. Cut Level Ordering = **Depth First**.

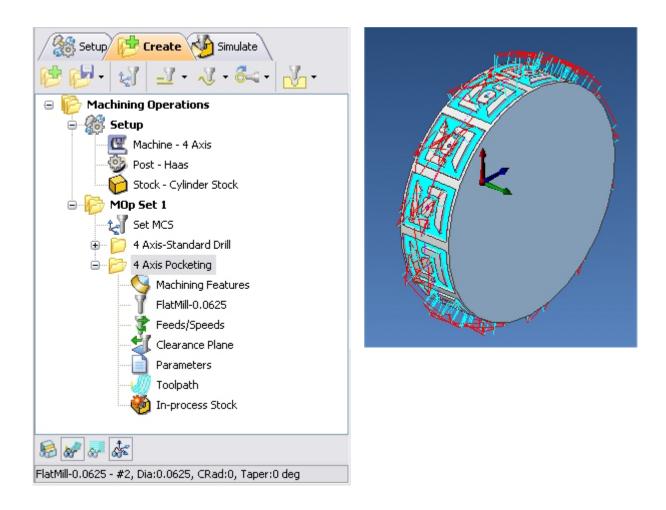
Machining Featu	res/Regions	Tool	Feeds & Speeds	Clearance
Cut Parameters	Cut Levels	Entry/Exit	Advanced Cut Para	ameters Sorting
	At Bottom	epth:	Rough Depth	etry at Top ↓ Rough Depth/Cut ★ Finish Depth/Cut
Rough Depth	1 1 1 1 1		Finish Depth/Cut: 0	
Cut Levels Or	st			

Sorting

12. Switch to Sorting tab and use Minimum Distance Sort and select Lower Left as Start Point

13. Click **Generate**. The 4 Axis Pocketing toolpath is now generated, and the Operation is listed under 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.

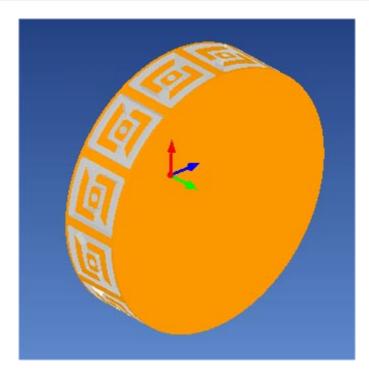


2. Select 4th Axis Pocketing Operation and click by to launch the Alibre CAM Stock Simulation window.

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

The simulated part is as shown below.

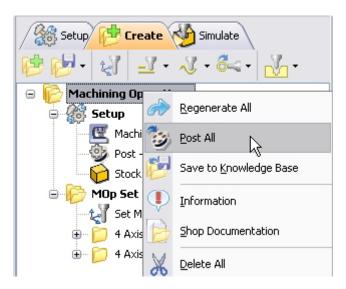
Note: You can adjust the simulation speed by selecting Simulation Preferences that is located to the bottom right corner of the Simulate tab Alibre CAM-MOps browser or from the Stock simulation window.



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

Post Processing

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



2. Specify the File Name as **4AxisPocketing.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