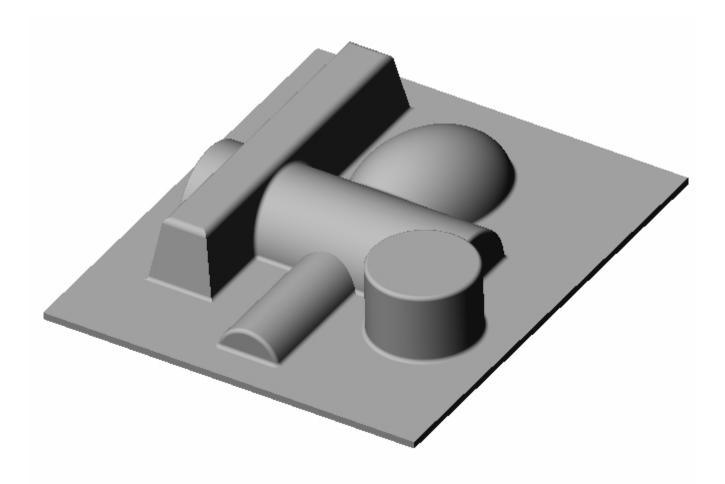
Tutorial 7: Machining a 3D Mold



Using 3 axis Roughing and Finishing Toolpath methods

Introduction

This tutorial will illustrate machining this Mold using 3 axis-milling operations. This tutorial will introduce the usage of several 3-axis operations such as horizontal roughing, parallel finishing, and horizontal 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 mold

- We will machine the mold completely using 3 axis-machining operations.
- The part itself will be machined out of a 5.5 x 6.5 inch x 1.25-inch wood block.
- The stock may be held to the machine table or the spoil sheet on the table using doublesided tape or by clamps.
- The part will be machined using 0.5" flat end mill, 0.25" & 0.125" ball end mills.

Main Programming Steps

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

- Create the Stock geometry
- Set the Machine zero point or Locate geometry with respect to the machine coordinates
- 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 regions for containing the cutter to specific areas to cut
- 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.

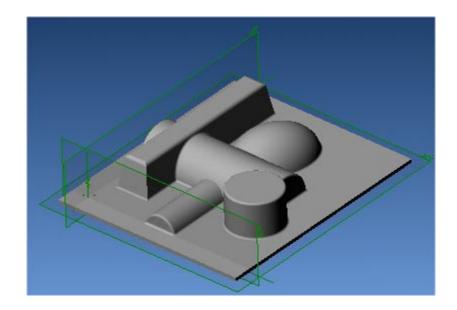
Loading the Part Model

"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 **3dMold.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
1	- 🔁 🖪 🔷 🍘 🎒 X 🖿 🏔 X い つ			Machining Operations Browser				
	n Exp							Cutting Tools Browser

3dMold - Alibre Design Professional File Edit View Insert Sketch 3D Sketch Feature Tools Alibre CAM Window Help	
] 🦉 + 🔂 🗑 🎒 🎒 👗 🏷 🗠 🗠	
Albre 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. Click on the Setup tab located under the MOPs Browser and select Setup Machine

Setup 🚰 Create 🔮 Simulate
Machine Chining Operations
E Setup
Machine - 3 Axis
Post - AbilitySystems
🔂 Stock - None
📾 🛷 💀 🎄

2. Set the Machine type to 3 axis

chine			
Aachine Setup			
Machine Type			
🧿 3 Axis	🔿 4 Axis	054	Axis
- Tool Change P	osition X 0 💽 Y 0	Z 0	
- 4th Axis (Primar			
Rotary Center:		÷ Z 0	÷ 4
Rotary Axis:	●XAxis ○Y	Axis 🔿 Speci	fy
	X 1 Y 0	Z O	
5th Axis (Secor	idary Axis)		
Rotary Center:	X 0 📫 Y 0	<u>↑</u> Z 0	÷ 🔖
Rotary Axis:	X 0 Y 1	Z 0	
Gage Length	0		
Output all	Co-ordinates in Rotate	d Co-ordinate System	
	ОК	Cancel	Help

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



4. 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

Set Post-Processor Options	3
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: Inc	
OK Cancel Help	

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.

Create Stock Geometry

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



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

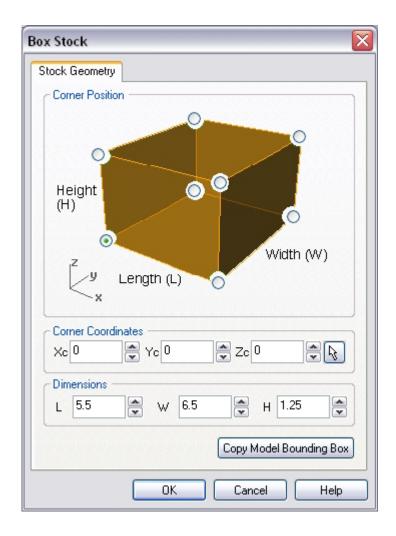
Stock Model Information						
Important notes about stock models:						
 Once the stock is created, the 3-D bounding box of the stock model will be rendered in the Alibre Design graphics window. Please note that this is not the actual stock model. The actual stock model will be displayed only in the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

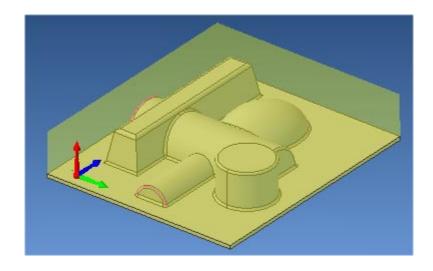
User can turn off this dialog by selecting Do not show this dialog again located on the bottom of the message window.

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

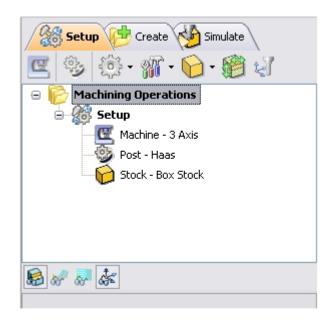
2. This brings up the Box Stock parameters. Set the Length (L) = 5.50, Width W = 6.5 and Height (H) = 1.25. Leave the other parameters as default and Click OK. Set Corner coordinates for Xc, Yc and Zc = 0.



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



3. 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

12	Setu	P 🔁	Create	🧐 si	mulate	
œ	-	203 -	M •	0	R	
Θ 🔓	🁌 Ma	chining	Operal	ions	20	

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

Align Part and Stoc	k Models		X
Align Part and Stock			
	1		
Z Alignment —			
	🔿 Center	 Bottom 	
-XY Alignment-			
O North Wes	t 🔿 North	🔿 North East	
O Mid-West	💿 Center	◯ Mid-East	
O South Wes	t 🔿 South	🔘 South East	
	ок 🗌	Cancel Hel;	

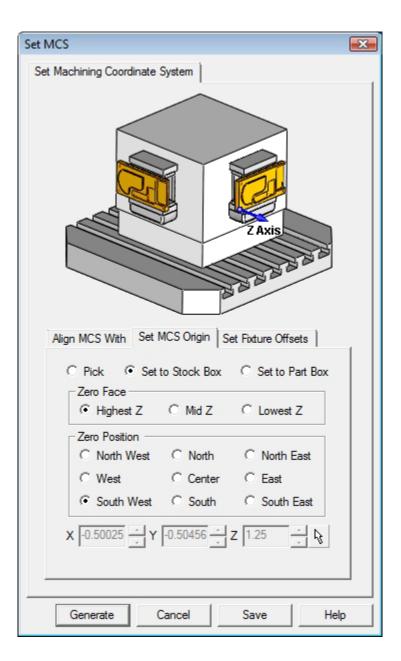
Set Machine Coordinate System (MCS)

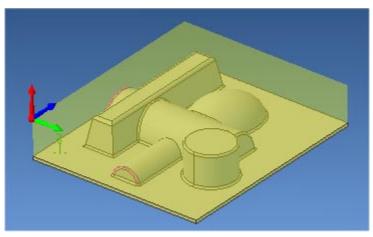
The steps below help you determine the machine home (also known as machine zero or tool touch off point) for the part/stock geometry.

1. Select Set MCS from the Setup tab



2. Switch to SetMCS Origin tab and choose **Set to Stock Box**, the Zero Face to **Highest Z**, and Zero Position to **South West** corner. This sets the machine home to the top of the stock material and the southwest corner of the part geometry.





(This sets the machine home to Southwest corner and top of the stock material).

Click Save As to save the work and specify a file name as 3dMold-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.

Create Tools

To machine the above part we will now create a $\frac{1}{2}$ inch (0.5") 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 Flat End Mill.



 Set the tool name as FlatMill-0.5, Tool Diameter = 0.5, Under the Properties tab set Tool Number = 1.

Create/Select Tool		×
777777	1 3 3 7 7 7 7	T I I I
Tools In Library	Name FlatMill-0.5	Properties Feeds & Speeds
	1 ► Holder Diameter →	Material HSS
	↑ Holder Length	Number of Flutes 2 v Tool Number 1 v
	1.5	Adjust Register 0
	Flute Tool Length	Cutcom Register 0
	2.5	Coolant None 💌 Comments
	± I → H Diameter	
	0.5	
	Save as New Tool Save	e Edits to Tool Delete Tool 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.

Properties F	eeds & Spee	ds				
- Spindle Spe	ed 5000	* *	RPM			
- Feed Rates						
Plunge:	35	* *	in/min			
Approach:	35	•	in/min			
Engage:	40	* *	in/min			
Cut:	45	 • •	in/min			
Retract:	50	*	in/min			
Departure:	50	-	in/min			
Transfer Fe	edrate (Tf)					
💿 Use Ra	💿 Use Rapid					
🔿 Set	50	*	in/min			

5. Click **Save as New Tool**. The tool is now created and listed under Tools in Library. Click OK to close the dialog.

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 Ball End Mill with the following parameters.
 - a. Tool Name: BallMill-0.25, Tool Number = 2.
 - b. Switch to Feeds & Speeds tab set Spindle Speed = 5000 rpm, plunge & approach feed = 35 ipm, approach feed = 40 ipm, cut feed = 45 ipm, retract and departure feeds = 50 ipm. Set the Transfer Feedrate to Use Rapid.
 - c. Click Save as New Tool.

Create/Select Tool		×
777773	33307	T I I
Tools In Library	Name BallMill-0.25	Properties Feeds & Speeds
		Material HSS 💌
	← Holder Diameter → T Holder	Number of Flutes 2
	Length	Tool Number 2
		Cutcom Register 0
	Flute Tool Length Length	Zoffset 0
	2.5	Coolant None 💌
		Comments
	↓ → ↓ Diameter	
	0.25	
		Edits to Tool Delete Tool
	ОК	Cancel Help

- 7. Create another Ball End Mill with the following parameters.
 - a. Tool Name: BallMill-0.125, Tool Number = 3.
 - b. Switch to Feeds & Speeds tab set Spindle Speed = 5000 rpm, plunge & approach feed = 35 ipm, approach feed = 40 ipm, cut feed = 45 ipm, retract and departure feeds = 50 ipm. Set the Transfer Feedrate to Use Rapid.
 - c. Click Save as New Tool.

The created tools are now listed under the Alibre CAM Tools browser.

1 ³⁴ 16 11 16 16 1	
 Tools FlatMill-0.5 BallMill-0.25 BallMill-0.125 	

Create Machining Operations

We will machine the mold using 3 different machining operations – Horizontal Roughing, Parallel Finishing, and Horizontal Finishing.

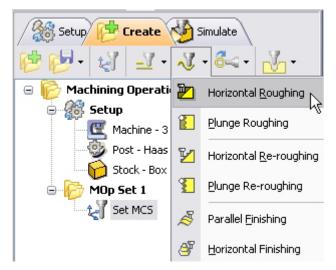
The first step in machining the mold will be a roughing operation. This type of machining is very efficient for removing large volumes of material and is typically performed with a large tool. Roughing is typically followed by semi-finishing or finishing toolpaths.

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



3 axis Horizontal Roughing

1. Select 3 Axis Milling and choose Horizontal Roughing.



- 2. This brings up the 3 Axis Horizontal Roughing Operation Dialog. We will now go over the steps for creating the toolpath.
- 3. Switch to the Tools tab inside the 3 axis Horizontal Roughing operation and select FlatMill-0.5.

Cut Parameters Cut Levels Enga	age/Re	tract Advan	ced Cut Parameters
Machining Features/Regions Tool		Feeds & Speeds	Clearance
□		Tool Geometry	
		Diameter	0.5
FlatMill-0.5		Corner Radius	0
BallMill-0.25		Taper	0
🖵 📅 BallMill-0.125		Tip Angle	0
		Tool Propertie	5
		Tool Name	FlatMill-0.5
		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	45
		Edit/Create/S Previe	Select Tool w Tool
		200 200	

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.

Cut Parameters	Cut Level	s Engag	e/Retract		Cut Parameter
Machining Featu	res/Regions	Tool	Feeds &	Speeds	Clearance
- Spindle Speed -	5000	BPM			
Feed Rates					
Plunge (Pf)	35 🚖] in/min		-	
Approach (Af)	35] in/min	Pf		
Engage (Ef)	40	j in/min		Cf	Tf
Cut (Cf)	45	in/min	Af		Df
Retract (Rf)	50	in/min	Ef		Rf
Departure (Df)	50	in/min			_
Transfer (Tf)					
💿 Use Rapid	50	T			
O Set	50	in/min			
Lo	ad From Tool				
	From Table	he			

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

rizontal Roughing					
Cut Parameters Cut I	evels	Engage/	Retract	Advanced	d Cut Parameters
Machining Features/Regio	ons T	ool	Feeds	& Speeds	Clearance
Clearance Plane Definition	n ———			ot	_Stock Max Z
 Automatic 		<		•	Part
◯ Part Max Z + Dist	0.25		1		Max Z
🔘 Stock Max Z + Dist	0.25				1
O Absolute Z Value	0.25	4			
Cut Transfer Method Skim Skim Clearance (C) Clearance Plane	0		Clea Part M	arance Plan	
ſ	Generate		ancel	Save	Help

Specify Cut Parameters

- 6. Click on the Cut Parameters tab.
- Set the Intol and Outol = 0.01, Stock to leave =0.025, Cut Pattern to Stock Offset, Cut Direction = Mixed, Step over distance = 40 (% Tool Diameter).

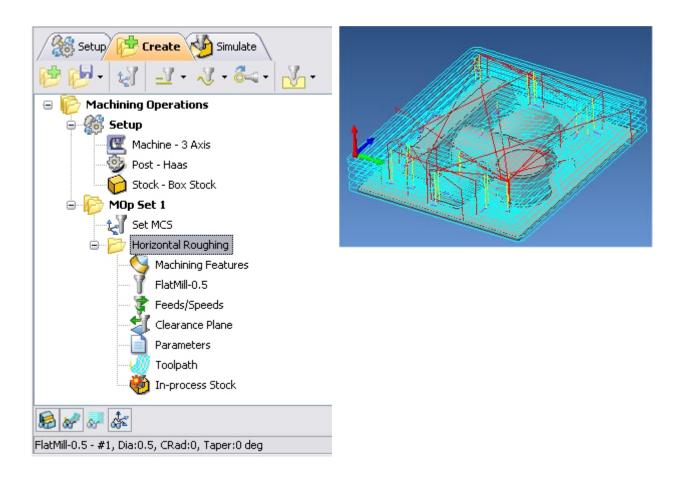
Machining Features/Regi	ions	Tool	Feeds	& Speeds	Clearance
Cut Parameters Cut	Levels	Engage/	'Retract	Advanced	Cut Parameter:
- Global Parameters-			Teelneth		
Intol	0.01		Toolpatł 	Stock (Duttol
Outol	0.01		Intol	-	
Stock	0.025		1	10	Z
	0.020		И Ра	rt Geometry	
Cut Pattern					
O Part Offset (Facir	ng) 💿 S	tock Offse	t (Pocketing) 🔘 Linea	ır 📄
- Cut Direction					
Climb (Down Cut)) OC	onvention	al (Up Cut)	💿 Mixed	ł
C Offset					
Pocket Start Poi	nt Inside	~			
Linear		3			
O Start at Bottom	O Start at	Тор		1 1	
Angle of Cuts	: 0				
Cleanup Pass				1	
- Stepover Control					T
	25		+		1
O Distance	0.125				
0	0.01		Ų		
C) Scallop			→ + St	epover	
O Scallop					

8. Switch to the Cut Levels Tab.

- 9. Use the Following Settings.
 - a. Step Down Control (dZ) = 50 (% Tool Diameter).
 b. Check Clear Flats under Cut Levels.

Machining Featu		Tool	Feed	s & Speeds	Clearance
Cut Parameters	Cut Levels	Enga	ge/Retract	Advanced (Cut Parameter:
Stepdown C % Tool D Distance Number	iameter 50 0.125		Ţ.		
Cut Levels 0					
O Level Fir				1 2 3 4	
Cut Levels Top (T) Bottom (I)		<	Т F		
hł.					

- 10. Switch to the Engage/Retract tab and leave the entry/exit parameters as default.
- 11. Click **Generate**. The 3 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.

83	6	<mark>89</mark> 7	\$s	63 ^{II}	63	—J-	÷	
	St	tock	Visib	ility				

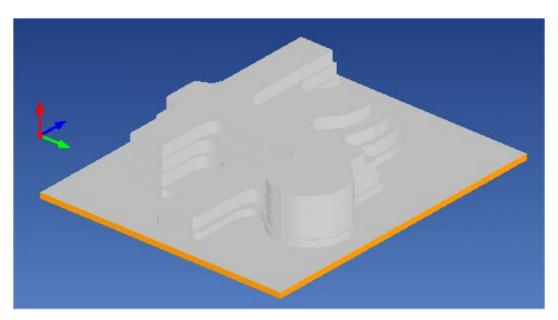
1. Switch to the Simulate tab in the Alibre CAM -MOps browser.



- 2. Select the 3 Axis Horizontal Roughing 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.

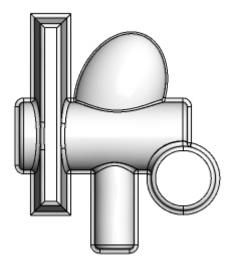
Note: You can adjust the simulation speed by selecting Simulation Preferences that is located to the bottom right corner of the Simulate tab Alibre CAM-MOps browser or from the Stock simulation window.



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

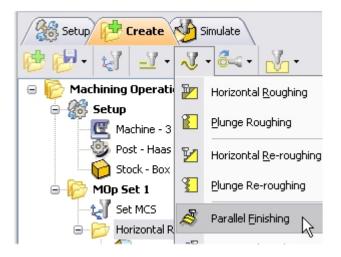
3 axis Parallel Finishing

We will now use 3 axis Parallel Finishing operation to pre-finish the part using a 0.25'' Ball End Mill.



This is an efficient method of finishing or pre-finishing, typically used when part surfaces are relatively flat. A 2D linear zigzag pattern is generated on the XY plane above the part geometry. The tool moves along this cut pattern, following the contours of the part geometry below.

1. From the Create Operations tab, select 3 axis Milling and Parallel Finishing.



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

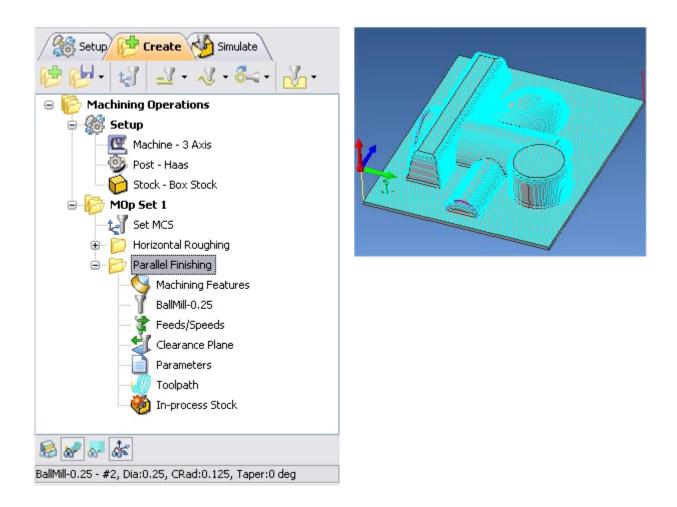
- 2. Switch to the Tools tab inside the Parallel Finishing operation and select BallMill-0.25.
- 3. Click on the Feeds and Speeds tab and select Load From Tool. Alibre CAM will now get the feeds and speeds information that was set when the tool was defined.
- 4. Switch to the Clearance Tab and set the Clearance Plane Definition to Automatic and Cut Transfer Method to Clearance Plane.

Specify Cut Parameters

- 5. Click on the Cut Parameters tab.
- Set the Tolerance to 0.001, Stock to leave =0, Cut Direction = Mixed, Step over distance = 15 (% Tool Diameter).

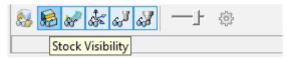
Machining Featu		Tool Feeds & Speed
Clearance	Cut Parameters	Z Containment Entry/
Global Parameters		
Intol	0.001 💌	Toolpath I Stock Outtol
	×	Intol
Outtol	0.001 🚔	The second secon
Stock	0	Part Geometry
Cut Control		
Cut Directio		
Mixed	🔘 Climb (Down	n Cut) 🛛 🔿 Conventional (Up Cut)
- Start Side -		1
 Bottom 	🔿 Тор	
-Angle of Cu		
	0	
Stepover Control -		
💿 % Tool Diam	eter 15 🚔	+ +
O Distance	0.125	
	0.01	
O Scallop	0.01	→ + Stepover

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



Simulate Toolpath

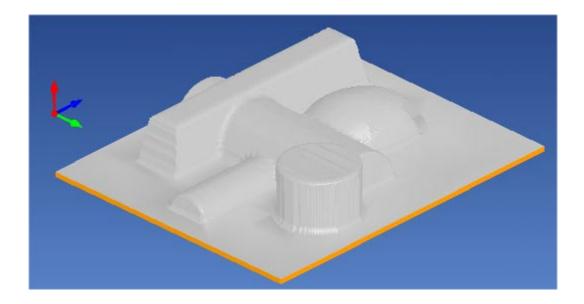
The generated toolpath can now be simulated. Make sure to turn on Stock Visibility under the Simulate tab.



- 1. Switch to Simulate Tab, Select the 3 Axis Parallel Finishing and click lock to launch the Alibre CAM Stock Simulation window.
- 2. 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.



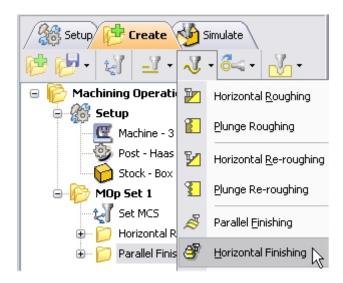
3. Once the simulation is complete, you can close the Stock Simulation window and return to the Alibre CAM browser.

3 axis Horizontal Finishing

We will now create a Horizontal Finishing operation for machining steep area regions.

This method is used for pre-finishing or finishing in constant Z levels, typically used when the part has large vertical surfaces and when Parallel Finishing will not yield satisfactory results.

- 1. Switch to the Create Operations tab.
- 2. Select 3 axis Milling and Horizontal Finishing.

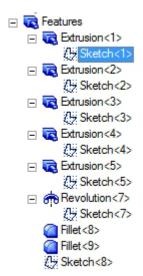


Select Machining Features/Regions

1. Go to the **Machining Features/ Regions** tab and click **Select Containment Regions**. The Horizontal Finishing operation dialog is now minimized and allows selection of the sketch geometry.

Horizontal Finishing	×
Cut Parameters Cut Levels Machining Features/Regions Tool	Optimized Machining Entry/Exit Feeds & Speeds Clearance
# Selected Machining Region(s)	Select Containment Regions Select Flat Area Containment Regions
Currently Selected Avoid Regions	Containment Region
	Select Avoid Regions
4	Select Flat Area Avoid Regions
Remove All Remove Active	
Generate	Cancel Save Help

2. Switch to Design Explorer and select **Sketch1** located under Extrusion1.

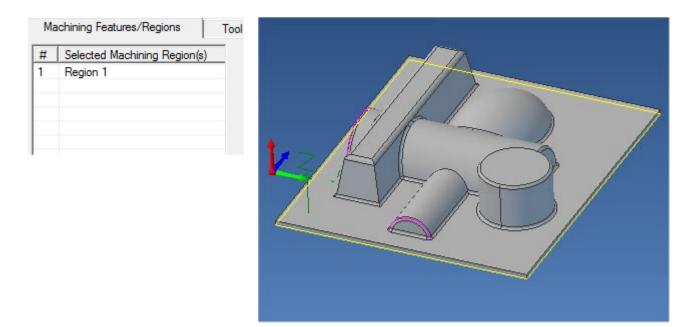


Note: You can also select the sketch from Alibre Design View. When select containment region is selected, Alibre CAM turns on Sketch selection filter to allow the user to select sketches. Hold the Shift key down to select multiple sketches.



to complete the selection.

The 3 Axis Horizontal Finishing operation dialog comes back up displaying the selected regions. The selected regions are also highlighted on the part.

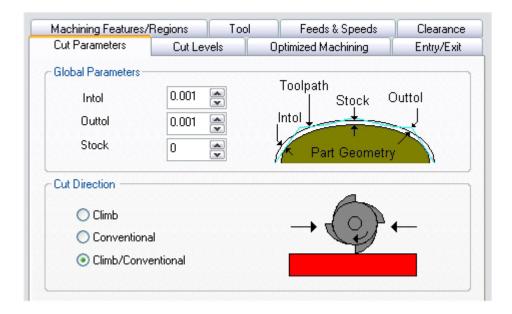


- 4. Switch to the Tools tab inside Horizontal Finishing operation and select BallMill-0.125.
- 5. Click on the Feeds and Speeds tab and select Load From Tool. Alibre CAM will now get the feeds and speeds information that was set when the tool was defined.

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

Specify Cut Parameters

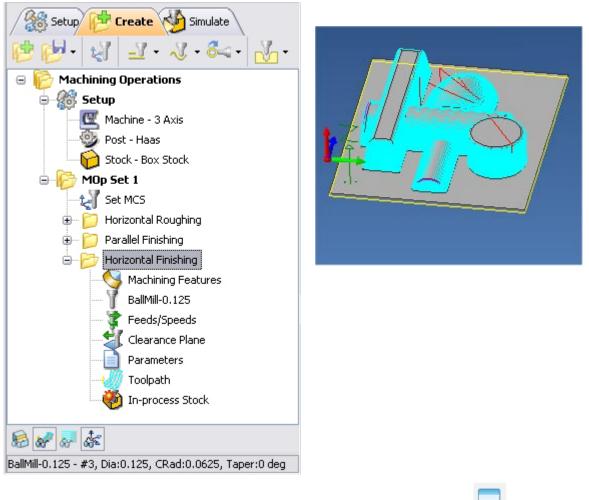
- 7. Click on the Cut Parameters tab.
- 8. Set Intol, Outol = **0.001**, Cut Direction = Climb/Conventional.



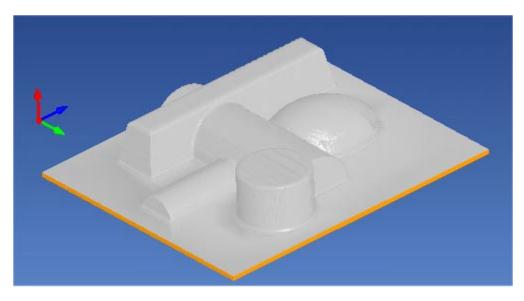
- 9. Switch to the Cut Levels tab.
- 10. Set the Step Down Control (dZ) = 15 (% Tool Diameter).

Cut Parameters Cut Levels Optimized Machining Entry/Exit Stepdown Control (dZ) ③ % Tool Diameter ① Distance ① 125 ② Number of Levels ⑤ Number of Levels ⑤ Depth First 〇 Depth First 〇 Depth First 〇 O O O O O O O O O O O O O O O O O O O	rizontal Finishing	
 % Tool Diameter Distance Number of Levels Number of Levels Cut Levels Ordering Level First Depth First 		
Cut Levels		Ţ dZ
□ Top (T) □ ● F	C Level First	
Bottom (B) -1.25	□ Top (T) 0 💌 □ Bottom (B) -1.25	F
	Generate	Cancel Save Help

- 11. Leave the entry/exit parameters at default.
- 12. Click **Generate**. The Horizontal Finishing operation is now created and is listed under the MOps browser.



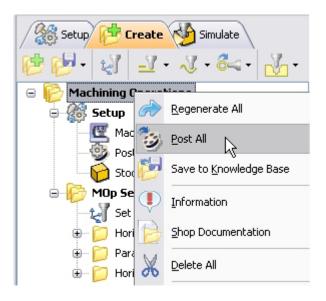
- 13. Switch to the Simulate tab, select Horizontal Finishing, and click by to launch the Alibre CAM Stock Simulation window.
- 14. Click Simulate from the Stock Simulation window to run simulation.



15. 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 **3dMold.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