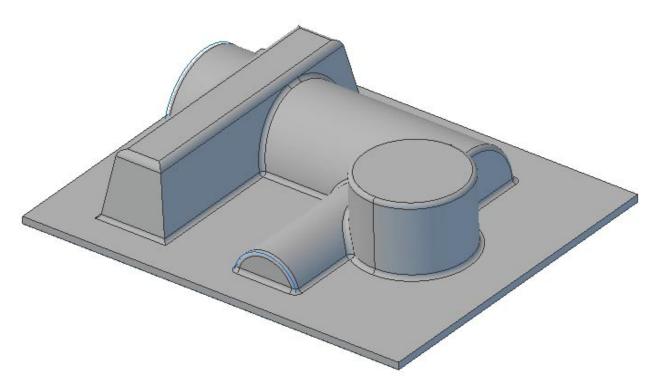
Tutorial 10: Re-Machining a 3D Mold



Using 3 axis Pencil Tracing and Valley Re-Machining Toolpath methods

## Introduction

This tutorial will illustrate machining this Mold using advanced 3 axis-milling operations.

This tutorial will introduce the usage of advanced 3-axis operations such as Pencil Tracing & Valley Remachining.

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 4.5 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 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.

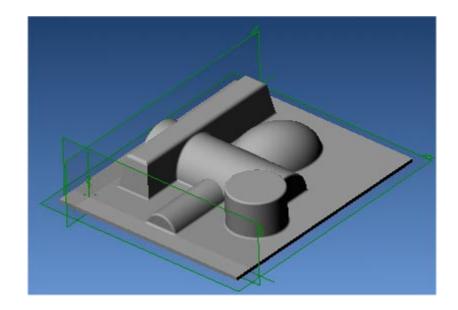
You may have to repeat either all or part of these steps for subsequent operations.

### 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 **3dMold2.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.

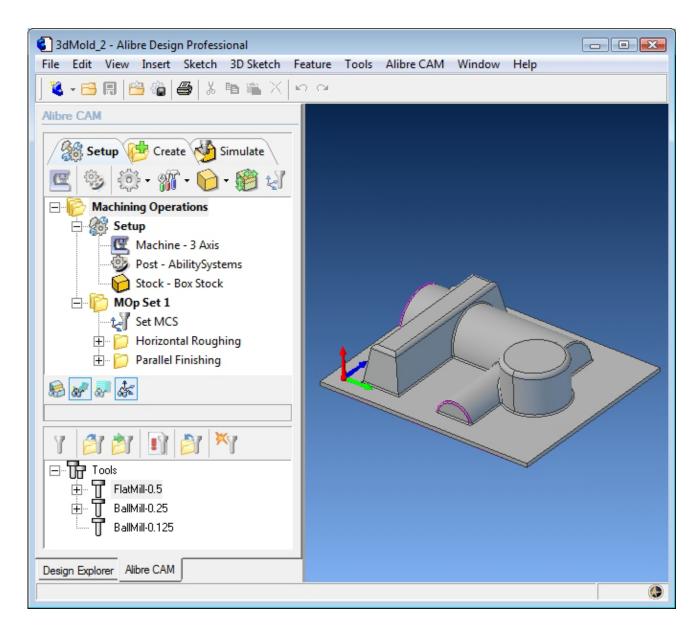


**Note:** You must work in shaded mode in order to be able to visualize toolpaths created in Alibre CAM. It is suggested for best visual performance with Alibre CAM to work with only one view port open and the view port operating in shaded mode.

#### 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
🌂 - 🖴 🖪  🆓 🍯	3   X 🖻 🛍 X	50 04		Machining Operations Browser
esign Explorer				Cutting Tools Browser



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.

### **Create Machining Operations**

Note: The part is pre-programmed with 2 machining operations. Horizontal Roughing and Parallel Finishing.

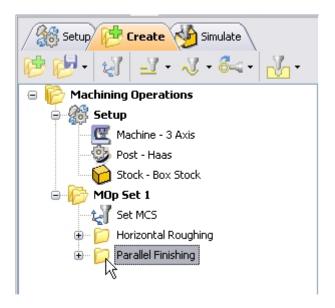
We will now use Remachining to finish the mold using 2 different machining operations – Pencil Tracing and Valley Re-Machining.

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



#### **3 axis Pencil Tracing**

1. Highlight the Parallel Finishing operation in the Create operations tab. This would ensure the next machining operation created would below the Parallel Finishing toolpath.



2. Select 3 Axis Milling and choose Pencil Tracing.

Setup	Cre	ate 😏	Simulate		
19 19 ·	21 :	- T - 🕄	• 🚑 •	•	
		2	<u>P</u> encil Tra	cing 🗼	

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

earance Cut Parameters	Advanced Parameters	Entry/Exi
Machining Features/Regions	Tool F	eeds & Speeds
Tools	Tool Geometry	
FlatMill-0.5	Diameter	0.125
BallMill-0.25	Corner Radius	0.0625
	Taper	0
La Ballin 10.125	Tip Angle	0
.0	Tool Propertie	
	Tool Name	BallMill-0.125
	Tool #	3
	# of Flutes	2
	Cutcom Register	0
	Adjust Register	0
	Z-Offset	0
	Material	HSS
	Coolant	None
	Comments	
	E Feeds & Speed	ls
	Spindle Speed	8000
	Feed Rate	30
	Edit/Create/S	Gelect Tool

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.

Clearance	Cut Param	eters	Advanced	Parameters	Entry/Exi
Machining Fe	atures/Region	s	Tool	Fee	eds & Speeds
Spindle Speed -			-		
	5000 🚖	] RPM			
Feed Rates			7		
Plunge (Pf)	35	in/min		_	
Approach (Af)	35 🌲	in/min	Pf		
Engage (Ef)	40	in/min		Cf	Tf
Cut (Cf)	45	in/min	Af		
Retract (Rf)	50	in/min	Ef		Rf
Departure (Df)	50	in/min			
Transfer (Tf)					
💿 Use Rapid		-			
O Set	20	in/min			
			_		
	ad From Tool				
Load	From Table				

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

Machining Features/Regions		Tool	Feed	ds & Speeds	
Clearance	Cut P	arameters	Advanced Pa	rameters	Entry/Exi
Clearance Pla	ne Definitio	n ———		at	Stock Max Z
Automati	с			.+	Part
O Part Max	Z + Dist	0.25			Max 2
O Stock M	ax Z + Dist	0.25			
O Absolute	Z Value	0.25	5		
OR Clearance	earance (C) e Plane	0	Part M	ax Z	

# **Specify Cut Parameters**

- 7. Click on the Cut Parameters tab.
- Set the Intol and Outol = 0.001, Stock to leave =0.0, Cut Direction to Climb, Select Do multiple cuts and set Number of Cuts = 2 and Step over control = 20 (% Tool Diameter).

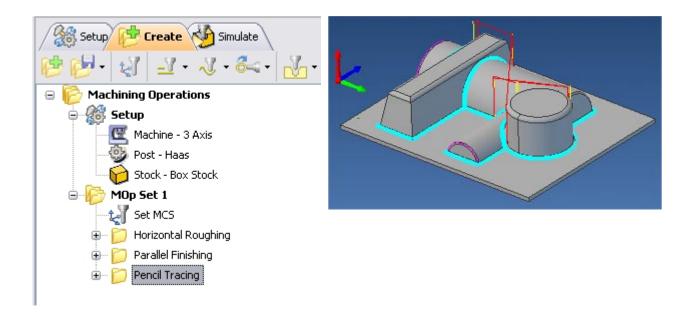
Machining F	eatures/Regions	Tool	Feed	s & Speeds
Clearance	Cut Parameters	Advanced Para	meters	Entry/Exil
Global F Intol Outol Stock	Parameters 0.001 0.001 0 0	Intol	Stock O	uttol
Cuts Dir Cuts Dir Clir Cor			Ì	
Number	ool Dia. 20			

## **Advanced Cut parameters**

- 9. Switch to the advanced cut parameters Tab.
- 10. Use the Following Settings.
  - a. Limiting Steep Angle = 90 and Maximum included angle =180.

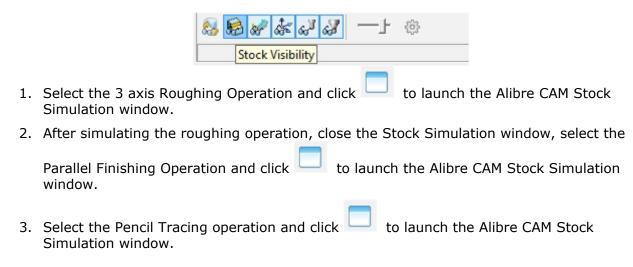
Machining F	eatures/Regions	Tool	Feeds & Speeds
Clearance	Cut Parameters	Advanced Param	eters Entry/Ex
Slope C	`ontrol		
	lit Cuts at Angle (A)		
			-
Cut Sp	blit Limiting Length:		
	0.25		
00	itput Flat & Steep Cuts		
	itput Flat Cuts Only		
00	iptut Steep Cuts Only		
Change	Cuts Direction		
Steep t	cuis Direction		A CONTRACTOR
OAs	cending		
ODe	scending		
	seenang		74.3.74.3.76
			. 100
Limiting St	eep Angle: 90	Maximum Included A	ingle: 180
	10.00		100
	Generate	Cancel	Save Help

11. Click **Generate**. The 3-axis Pencil Tracing toolpath is now generated, and the Operation is listed under the Alibre CAM MOps browser.

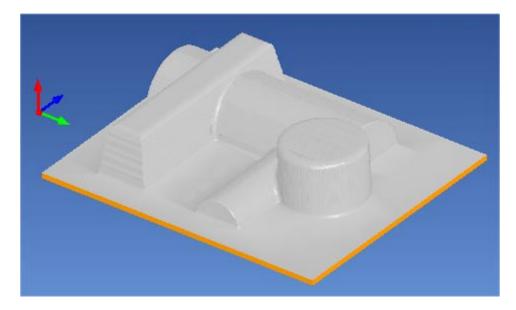


### Simulate Toolpath

Make sure to turn on Stock Visibility under the Simulate tab.



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



Note: You can pause/stop, step, and skip the simulation using the controls available on the Simulation window. To control the simulation speed, select Simulation Settings 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 Valley Re-Machining

We will now use 3 axis Valley Re-Machining operation to finish the part using a 0.125" Ball End Mill.

- 1. From the Create Operations tab, select 3 axis Milling and Valley Re-Machining.
- 2. This brings up the Valley Re-Machining Operations dialog. We will go over the steps for creating the pocketing operation.
- 3. Switch to the Tools tab inside the Valley Re-Machining operation and select BallMill-0.125.
- 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.
- 5. Switch to the Clearance Tab and set the Clearance Plane Definition to Automatic and Cut Transfer Method to Clearance Plane.
- 6. Click on the Cut control tab.

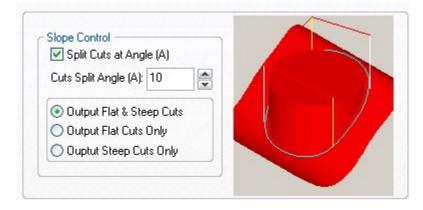
# **Specify Cut Control**

- 7. Click on the cut control tab.
- Set the Tolerance to 0.001, Stock to leave =0, Under Reference Tool Parameters set Tool Diameter (D) = 0.25 (This is the diameter of the tool that was used for parallel finishing).

Clearance     Cut Control     Cutting Parameters     Entry/Ex       Global Parameters     0.001     Image: Control     Toolpath     Outloi       Outol     0.001     Image: Control     Image: Control     Toolpath       Stock     0     Image: Control     Image: Control     Toolpath       Note     0.001     Image: Control     Image: Control     Toolpath       Note     0.001     Image: Control     Image: Control     Image: Control       Reference Tool Parameters     Image: Control     Image: Control     Image: Control       Reference Tool Parameters     Image: Control     Image: Control     Image: Control       Reference Tool Parameters     Image: Control     Image: Control     Image: Control       Naximum Included     180     Image: Control     Image: Control	Machining Features/Regions			Tool		Feed	s & Speeds
Intol Dutol Dutol Stock D Part Geometry	Clearance	Cut Cont	rol	Cuttin	g Parameter:	; )	Entry/Exi
Intol Dutol Dutol Stock D Part Geometry							
Intol Dutol Stock 0 Stock 0 Stock 0 Part Geometry Part Geometry Part Geometry Part Geometry Current Tool Cut Overlap: Limiting Steep Angle: 90 Maximum Included 180 Stock 0 Stock 0 Stock Outtol Intol Part Geometry Stock Outtol Part Geometry Stock Outtol Current Tool Current Tool Current Tool Current Tool Current Tool	Global Pa	presidentia		То	olnath		
Stock          Stock       Image: Construction of the second s	Intol	0.0	101	Cardo -	Sto	ick O	uttol
Stock       O       Part Geometry         Reference Tool Parameters       Tool Diameter(D):       0.25         Minimum Cut Length:       0.05       Current Tool         Cut Overlap:       0.025       S         Limiting Steep Angle:       90       Reference Tool         Maximum Included       180       Reference Tool	Outol	0.0	101				1
Reference Tool Parameters         Tool Diameter(D):       0.25         Minimum Cut Length:       0.05         Cut Overlap:       0.025         Limiting Steep Angle:       90         Maximum Included       180	Stock	0			Part Gr	ometry	
Tool Diameter(D): 0.25					Fait Of	omeny	N
Tool Diameter(D): 0.25	Deferre	- TI D					1
Minimum Cut Length: 0.05 Cut Overlap: 0.025 Limiting Steep Angle: 90 Maximum Included 180		E					
Cut Overlap: 0.025							ent Tool
Limiting Steep Angle: 90	Minimum	Cut Length:	0.05			U	
Limiting Steep Angle: 90	0	Cut Overlap:	0.025		T	S	Lun -
	Limiting 9	teep Angle:	90				
	Maxim		180			eference	Tool
		Angle:	100				

## **Specify Cutting Parameters**

- 9. Click on the Cutting Parameters tab.
- 10. Use Split Cuts under slope Control. Specify the Cuts Split Angle (A) = 10
- 11. Select Output Flat and Steep Cuts.



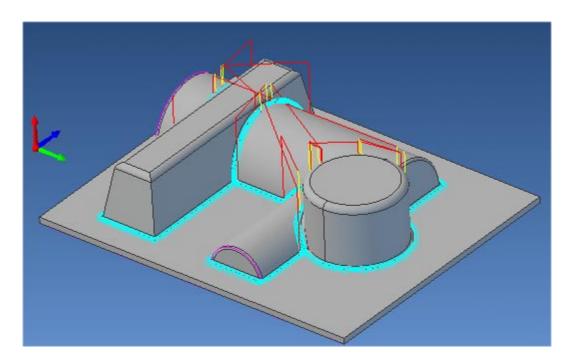
- 12. Select Flat Cut Parameters.
- 13. Select Cut Pattern as Along, Cut Direction as Along, Cut Direction as Mixed and specify Step Over control = 20 (% tool diameter).

Flat Cuts Parameters S	iteep Cuts Para	ameters					
	Along	O Parallel	O Horizontal				
Cut Direction	tional 💿 Mixe	ed	No.				
Stepover Control	20	•					
O Distance	0.125	•					
Angle of Cuts:	Angle of Cuts:						
Cut Start 💿 B	ottom 🔿 Top						

- 14. Select Steep Cut Parameters.
- 15. Select Cut Pattern as Across, Cut Direction as Along, Cut Direction as Mixed and specify Step Over control = 20 (% tool diameter).

Flat Cuts Parameters Steep Cuts	Parameters	
Across O Along	🔘 Parallel	O Horizontal
Cut Direction		ANN OTA
◯ Climb ◯ Conventional ⊙	Mixed	
Stepover Control		
O Distance 0.125	×	
Angle of Cuts:		
Cut Start    Bottom	Тор	

16. Click **Generate**. The Valley Remachining toolpath is now generated, and the Operation is listed in the Alibre CAM MOps browser.



17. Switch to Simulate tab, select Valley Remachining, and click Simulate to run the simulation.

## **Post Processing**

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

Setup [ Create 🖉 Simulate				
🖻 🔁 • 🔣 🖃	• 💐 • 🏍 • 🔥 •			
😑 📄 Machining 🗠	- <b>L</b> <sup>2</sup>			
Setup 🛷	<u>R</u> egenerate All			
🖉 Ma 🌍	Post All			
St 🦻	Save to Knowledge Base			
🗎 🦻 MOp S	Information			
🕀 💆 Но 🍺	Shop Documentation			
⊕… 🃁 Pa ⊕… 🎦 Pe 🐰	<u>D</u> elete All			
🗄 🗁 🔽 🔂	New <u>M</u> Op Set			
تي اي 🗞 😥				
BallMill-0.125 - #3, Dia:0.125	, CRad:0.0625, Taper:0 deg			

2. Specify the File Name as **3axisMold2.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