#### Getting Started with Alibre CAM

**Tutorial 3: Machining a Shaft Base** 



Using 2<sup>1</sup>/<sub>2</sub> Axis Facing, Pocketing, Hole Pocketing & Profiling Toolpath methods

## Introduction

This tutorial will illustrate machining of a prismatic part such as this Shaft Base using 2-1/2 milling operations. Even though we have created a 3-D representation of the part, it will be seen later on that we can machine this using just 2D Sketches. The reason we are able to do this is because of the prismatic nature of this model. This tutorial will introduce the usage of 2  $\frac{1}{2}$  axis machining for a simple one-sided part. We will use profiling, pocketing and hole pocketing operations.

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 Shaft Base

- We will machine the shaft base completely using 2  $\frac{1}{2}$  axis-machining operations.
- The starting material for the Shaft Base is soft wood and the size is 5.5 x 3.25 x 0.625 inches.
- The wooden sheet will be held to the machine table or the spoil sheet on the table using double-sided tape.
- The part will be machined using a single 1/4 inch flat end mill.
- Determining the sequence of machining operations
  - $_{\odot}$  The first operation would involve machining around the boss using a 2  $^{1\!\!/_2}$  Axis Facing Operation.
  - The next step would involve machining the area inside the boss. We will use 2 <sup>1</sup>/<sub>2</sub> Axis Pocketing Operation, which is ideal removing material inside a specified region.
  - $\circ~$  We are now down to the level where the step holes need to be machined. As the holes are circular we will use 2  $\frac{1}{2}$  Axis Hole Pocketing operation to machine the holes to its depth in 2 separate operations.
  - Finally we will cut out the shape of the part from the rectangular using a contour toolpath. This is accomplished using a 2 ½ Axis Profiling Operation that separates the finished part from the stock material.

### **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

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 **ShaftBase.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)
- 3. 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	- 3		3 🍅	<b>a</b> X	<b>h</b> 🛍 X	10 Ci		Machining Operations Browser	
Desig	in Exp	lorer						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.

### 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



2. Set the Machine type to 3 axis

achine			Þ
Machine Setup			
Machine Type			
🧿 3 Axis	🔿 4 Axis	🔿 5 Axi	s
- Tool Change Position	n		
×O	Y O	≥ Z 0	÷ ŀ
4th Axis (Primary Axis			
Rotary Center: X		🗘 Z 0	÷ R
Rotary Axis: 💿	X Axis O Y Axis	O Specify	
x 1	Y 0	Z 0	
- 5th Axis (Secondary)	Axis )		
Rotary Center: X 0		🔶 Z 🛛	÷ 🖓
Rotary Axis: X 0	Y 1	Z 0	
Gage Length	0		
Output all Co-o	rdinates in Rotated Co-c	ordinate System	
	UK	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
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:  Show Output Dialog when Post-Processing
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/Load Stock**

5. Select Create / Load Stock from the Setup tab and choose Box Stock



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

Stock Model Information					
Important notes about stock models:					
<ol> <li>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.</li> </ol>					
<ol><li>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.</li></ol>					
You can choose the simulation model type in the Simulation Settings dialog.					
<ol><li>Please note that Cut Material Simulation of rotated Machining Operations is available only in the Pro and the Expert configurations of Alibre CAM.</li></ol>					
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.

6. This brings up the Box Stock parameters. Set the Length (L) = 5.50, Width W = 3.25, and Height (H) = 0.75. Leave the other parameters as default, and click OK.

Box Stock
Stock Geometry
Comer Position
Height (H) Z Length (L) X
Comer Coordinates Xc 0 · Yc 0 · Zc · 0.625 · K
Dimensions L 5.5 • W 3.25 • H 0.75 •
Copy Model Bounding Box
OK Cancel Help

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



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 **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		
Align Part and Stock			
Z Alignment —			
🔿 Тор	🔿 Center	<ul> <li>Bottom</li> </ul>	
-XY Alignment-			
O North Wes	t 🔿 North	🔿 North East	
O Mid-West	💿 Center	◯ Mid-East	
O South Wes	st 🔿 South	O South East	
	OK	Cancel Hel	P

# 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

18	🕺 Setu	P 🔁	Create	🧐 Si	mulate	
œ	-	203 -	<b>917</b> •	0	· 🌮 📢	1
Ξ [	赺 Ma	chining	Operal	ions	~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~	

 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).

*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.

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

## **Create Tools**

To machine the above part we will now create a  $\frac{1}{4}$  inch (0.25") Flat End Mill.

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





 Set the tool name as FlatMill-0.25, Tool Diameter = 0.25. Under the Properties tab, set Tool Number = 1.

Create/Select Tool		×
77773	1 1 1 7 7 7 7	t t t
Tools In Library	Name FlatMill-0.25	Properties Feeds & Speeds
		Material HSS 💌
	← Holder Diameter → T Holder	Number of Flutes 2
	Length	Tool Number
	1.5	Adjust Register 0
		Cutcom Register 0
	Length Length	Zoffset 0
	2.5	Coolant None 💌
		Comments
	¥ → → × Diameter	
	0.25	
		Edite to 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 & Speed	st
- Spindle Spe	ed	
	5000	🚔 RPM
Feed Rates		
Plunge:	35	🚔 in/min
Approach:	35	🚔 in/min
Engage:	40	🚔 in/min
Cut:	45	膏 in/min
Retract:	50	🚔 in/min
Departure:	50	🚔 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. 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.



The created tool is now listed under the Alibre CAM Tools browser.

### **Create Machining Operations**

We will machine the Shaft Base using 4 different machining operations – Facing, Pocketing, Hole Pocketing and Profiling.

The stock geometry has a thickness of 0.75" and the finished part is 0.625". We will create a 2.5 axis facing operation to mill the 0.125" thickness of material from the stock geometry.

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

18	setu	📭 🥵 Create 🤡 Simulate	
œ	-	尊 • 🍿 • 🏠 • 🍘 🕹	,
Θ.	👌 Ma	chining Operations	

## 2 <sup>1</sup>/<sub>2</sub> Axis Facing

1. Select 2.5 Axis Milling and choose Facing.



2. This brings up the 2  $\frac{1}{2}$  Axis Facing Operation Dialog. We will now go over the steps for creating the toolpath.

### Select Machining Features/Regions

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

Regions are sketches that already exist in your model, or separate new sketches you create within Alibre Design that coexist with your part, but do not contribute to its geometry (the sketch is not used for a feature). Regions serve different purposes in  $2\frac{1}{2}$  and 3 axis milling. In  $2\frac{1}{2}$  axis milling, however, regions determine the entire area to be milled. Part and stock geometry are not considered, and the tool always moves at a fixed Z level. Therefore, regions act as the drive curves, and their geometry is projected onto the stock.

Regions must be selected before they can be used in an operation. Creating a region does not make it active; you must use one of the **Select Regions** tools before creating the toolpath.

4. Switch to Design Explorer and select **Sketch7**. This selects the outside rectangle.

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 2  $\frac{1}{2}$  Axis Facing operation dialog comes back up displaying the selected region. The selected region is also highlighted on the part.



- 6. Switch to the Tools tab inside the 2 ½ Axis Facing operation and select FlatMill-0.25.
- 7. 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.
- 8. Switch to the Clearance Tab and set the Clearance Plane Definition to Automatic and Cut Transfer Method to Clearance Plane.

## **Specify Cut Parameters**

- 9. Click on the Roughing tab.
- 10. Set the Tolerance to **0.01**, Stock to leave to **0**, Cut Pattern to **Island Offset Cuts**, and Step Distance to **50** (% Tool Diameter).

Machining Fe	atures/Regions	Tool	Feeds & Speeds Clearan
Roughing	Cut Levels	Entry/Exit	Advanced Cut Parameter
Global Par	ameters		Desition
To	lerance: 0.01		Region Toolpath
	Stock: 0		/ the chord
Compe	ensation: AUTO/N	-	* tolerance
			🛶 🖊 Stock
۲	Island Offset Cuts		O Linear Cuts
~ Cut Di	rection		
💿 Clin	nb (Down Cut)		
O Cor	nventional (Up Cut)		
O Mix	ked		
- Step D	)istance		+
0 % آ	Fool Dia. 50		
⊖ Dis	tance 0.125	A	
Cor	mer Cleanup		₩ Stepover
15161515			

11. Switch to the Cut Levels Tab.

- 12. Use the Following Settings.
  - a. Pick Top = **0** (As the selected region at Z = -0.125", we would need to start the first cut from Z =0).
  - b. Total Cut Depth = 0.125, Rough Depth = 0.125, and Rough Depth/Cut = 0.0625.

Machining realu	res/Regions	I ool	Feed	ls & Speeds	Clearance
Roughing	Cut Levels	Entry	v/Exit	Advanced	Cut Parameters
Location of Cu At Top Pick Top Cut Depth Co Total Cut Dep Rough Deptt 0.125	At Bottom At Bottom At Bottom O At Bottom O At Bottom O O O O O O O O O		Rough Depth otal ut epth Finis Dept	Set Top Z	– Rough - Depth/Cut - Finish Depth/Cut
Rough Depth	/Cut: 0.0625		Finish De	pth/Cut: 0	
Clear Islan	id lops		Use 3D Mo	del to Detect	Depth
Cut Levels On O Depth First O Level First	dering t			5 6 7	

- 13. Switch to the Entry/Exit tab and set the Entry and Exit parameters to none.
- 14. Click **Generate**. The 2½ Axis Facing 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.



- 1. Switch to Simulate tab, Select the 2 ½ Axis Facing Operation and click to launch the Alibre CAM Stock Simulation window.
- 2. Click Simulate from the Stock Simulation window to run simulation.



#### **Getting Started with Alibre CAM**

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.

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

## 2 <sup>1</sup>/<sub>2</sub> Axis Pocketing

We will now use 2<sup>1</sup>/<sub>2</sub> axis Pocketing operation to machine the area inside the boss.



### **Creating the Pocketing Operation #1**

1. From the Create Operations tab, select 2<sup>1</sup>/<sub>2</sub> axis Milling and Pocketing.

Setup	1	Create 🔗 Simulate
🖻 🛃 •	ŧJ	🚽 🗸 🗸 • 🖧 • 🔥 •
😑 📄 Mac	hining	Operations

This brings up the 2  $\frac{1}{2}$  Axis Pocketing Operations dialog. We will go over the steps for creating the pocketing operation.

- Go to the Machining Features/ Regions tab and click Select Containment Regions. The Pocketing operation dialog is now minimized and allows selection of the sketch geometry.
- 3. Switch to Design Explorer and select **Sketch5**.



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.

4. Click Select Sketch(s) to complete the selection.

The 2  $\frac{1}{2}$  Axis Pocketing operation dialog comes back up displaying the selected. The selected regions are also highlighted on the part.

Ma	chining Features/Regions Tool	
#	Selected Machining Region(s)	
1	Region 1	

- 5. Switch to the Tools tab inside the 2<sup>1</sup>/<sub>2</sub> Axis Pocketing operation and select FlatMill-0.25.
- 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 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 to 0, Cut Pattern to Offset Cuts, and Step over distance to 25 (% Tool Diameter).

Machining Featu	res/Regions	Tool	Feeds & Sp	eeds Clearan	се
Cut Parameters	Cut Levels	Entry/Exit	Advanced Cu	it Parameters 📗 Sor	tin
Global Param	eters		Dogion		
Toleranc	e: 0.001	×	Region	Toolpath	
Stock	c 0		- M	chord	
Compensation	1 AUTO/NON	IE 🗸		tolerance	
			→	⊢ Stock	
💿 Offset Cu	ts OLine	ar Cuts 🤇	) Spiral Cuts	O Radial Cuts	
<ul> <li>○ Conve</li> <li>⊙ Mixed</li> <li>Start Poin</li> <li>⊙ Inside</li> <li>Step Distant</li> <li>⊙ % Too</li> <li>○ Distant</li> </ul>	entional (Up Cut) It ance I Dia. 25 Ice 0.1	side	↓ ↓ ↓ ↓ ↓ ↓ ↓ ↓ ↓ ↓ ↓ ↓ ↓ ↓	r	
Cornel	r Cleanup				

3. Switch to the Cut Levels Tab.

- 4. Use the Following Settings.
  - a. Location of Cut Geometry at **Top**.

We will determine the Total Cut Depth from the 3D model by snapping at 2 points.

b. Select the Depth measuring tool located to the right of Total Cut Depth. This will minimize the Pocketing Operation parameters dialog.

Total Cut Depth:	0
Rough Depth:	Finish Depth:
0	0
A 1 1 1 1 1 1	- i - i - i - i - i

c. Pick the top edge of the boss as the start point of axis vector and the bottom edge of the boss as the end point as shown below.



- d. The pocketing operation dialog shows up and determines Total Cut Depth = 0.25.
- e. Set the Rough Depth = **0.25** and Rough Depth/Cut = **0.05**.

Machining Features/Regions Too Cut Parameters Cut Levels Entry/	ol Feeds & Speeds Clearance /Exit Advanced Cut Parameters Sortin
Location of Cut Geometry At Top At Bottom Pick Top Cut Depth Control Total Cut Depth: 0.25 Rough Depth: Finish Depth: 0.25 D	Total Cut Geometry at Top Pepth Total Cut Depth/Cut Finish Depth/Cut Finish Depth
Rough Depth/Cut: 0.05	Finish Depth/Cut: 0
Clear Island Tops	Use 3D Model to Detect Depth
Out Levels Ordering Depth First Level First	

This would machine the pocket in steps of 0.05 resulting in 5 cut levels.

*Note:* You can also specify the Total Cut Depth by entering the depth values under Total Cut Depth.

5. Switch to the **Entry/Exit** tab.

6.	Use the following settings for Entry/Exit. Make sure to check Apply Entry/Exit at all cut
	levels.

	res/Regions	Tool	Feeds & Speeds	Clearance
Cut Parameters	Cut Levels	Entry/Exit	Advanced Cut Parame	ters Sortin
				Entry Motions
Approach	Motion		2010/02/02/02/02	
	Length (L)	0.025	1	1±
Engage M	lotion		///	Тн
📀 Path	Angle (A)	10 🚔	A	>
O Linear	Height (H)	0.05	1	/
	Distance (D)	0.05	/ /	
◯ Helix	Radius(R)	0.0625	$\sim$	/
				-
Retract M	lotion Length (L)	0.1	Rapid <b>→</b>	
Retract M	lotion Length (L) Angle (A)	0.1	Rapid <b>→</b> Departu	re
Retract M Linear Radial	lotion Length (L) Angle (A) Radius (R)	0.1 × 45 × 0 ×	Rapid∔ Departu D≠ T	re
C Retract M C Linear C Radial	Length (L) Angle (A) Radius (R)		Rapid → Departu D= T T	re
Retract M C Linear Radial Departure V	Length (L) Angle (A) Radius (R) Motion ertical Dist (D):	0.1 × 45 × 0 ×	Rapid+ Departu P+ FL Retract	re
Petract M C Linear Radial Departure V	lotion Length (L) Angle (A) Radius (R) Motion ertical Dist (D):	0.1 × 45 × 0 ×	Rapid+ Departu D+ Retract	re
Retract M C Linear Radial Departure V Apply entry/ex	Iotion Length (L) Angle (A) Radius (R) Motion ertical Dist (D): it at all cut leve	0.1 45 0 0.6 0.6 0.6	Rapid+ Departu D+ Retract	re

7. Click **Generate**. The 2½ Axis Pocketing toolpath is now generated and the Operation is listed under the 2½ Axis Facing Operation in the Alibre CAM MOps browser.

Note: You can rearrange the operations in the MOps browser by selecting the operation and dragging and dropping.

#### **Getting Started with Alibre CAM**



### **Simulate Toolpath**

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



- 1. Switch to Simulate tab, Select the 2 ½ Axis Pocketing Operation and click low to launch the Alibre CAM Stock Simulation window.
- 2. Click Simulate from the Stock Simulation window to run simulation.



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

## **Creating the Pocketing Operation #2**

We will now create a 2<sup>nd</sup> pocketing operation for machining the region around the boss.

1. Switch to the Create Operations tab.

## **Copying a MOp**

- 2. Select the 2 <sup>1</sup>/<sub>2</sub> axis Pocketing Operation created from the previous step, right mouse click, and select **Copy**.
- 3. Right click and select **Paste**.

#### **Getting Started with Alibre CAM**



4. This would create a copy of the 2 <sup>1</sup>/<sub>2</sub> axis Pocketing Operation listed below the first pocketing operation as show below.



5. Expand the 2 <sup>1</sup>/<sub>2</sub> Axis Pocketing-1 folder and double click on Machining Features.



- 6. Click Remove All under Machining Features and click **Select Containment Regions**.
- 7. Select **Sketch7** and **Sketch6** from the Design Explorer by holding the Ctrl key down.



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.



- 9. Regions are now listed under Machining Features/Regions.
- 10. Switch to the **Cut Levels Tab**.
- 11. Use the Following Settings.
  - a. Location of Cut Geometry Select Pick at Top = 0.125
  - b. Total Cut Depth Set this to **0.375**
  - c. Set the Rough Depth = 0.375 and Rough Depth /Cut = 0.05
  - d. Switch to the Entry/Exit tab and set the Retract Motion to Linear, Length = 0.1 and Angle = 0
  - e. Click Generate.



12. The pocketing toolpath is now created and displayed in the MOps browser.



## Simulate Toolpath

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



to

- 1. Switch to Simulate tab, Select the 2 ½ Axis Pocketing-1 Operation and click launch the Alibre CAM Stock Simulation window.
- 2. Click Simulate from the Stock Simulation window to run simulation.



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

## **Hole Pocketing Operation**

In order to machine the 6 holes, we will now use 2 ½ axis hole pocketing operation.



## Creating the Hole Pocketing Operation #1

- 1. Select 2 1/2 Axis Milling and Hole Pocketing.
- 2. Go to the Machining Features/ Regions tab and click Select Containment Regions.
- Switch to Design Explorer and select Sketch2 located under Extrude4. This selects the 6 holes.



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.

- 5. Switch to the Tools tab inside the Hole Pocketing operation and select FlatMill-0.25.
- 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 Automatic and Cut Transfer Method to Clearance Plane.

#### **Specify Cut Parameters**

- 1. Click on the **Cut Parameters** tab.
- 2. Use the following Settings
  - a. Tolerance to **0.001**,
  - b. Hole Depth (H) =0.0625, Uncheck Use 3D model to Detect Depth,
  - c. Hole Diameter (D) = 0.5, Check Cleanup pass at each Z level
  - d. Step over distance = 25 (% Tool Diameter),
  - e. Step Down Control (dZ) = 50 (% Tool Diameter),
  - f. Cut Direction = Climb (Down Cutting).

Machining Fea	atures/Regions		Tool		Feeds & S	peeds
Clearance	Cut Pa	arameters	En	try/Exit		Sorting
- Global Param	eters					
Tolerance (t	) 0.001					
- Location of C	ut Geometry -		+		1	
💿 At Top	O At Bottom			O A	Ś	2
O Pick Top	0	\$		0 0	<pre></pre>	
Hole Paramet	ters		н		S⇔s	
Hole Depth (	(H) 0.0625	₩Ţ.			K	
Use 3D N	Iodel to Detect	Depth		$\Rightarrow$	\$	2
Hole Diamete	er (D) 0.5				1	
	Dia las Hole Dia	a for	±	(CE	D)	
any Arcs/	Circles selecte	d	18.		$\sim$	
🔽 Cleanup I	Pass at Each Z	Level		r D		
- Cut Direction						
0 C	limb (Down Cut	ting)	O Convent	ional (Up C	Cutting)	
- Stepover Cor	ntrol		Stepdown	Control (d	Z)	
💿 % Tool Dia	ameter 25	×	💿 % Too	l Diameter	50	Ŧ
O Distance	0.125	5 <u>*</u>	🔘 Distan	се	0.25	< >
			O Numb	er of Level	s 5	~ >

- 3. Switch to the Entry Exit Tab and set the Helix Diameter = **0.25**.
- 4. Check Create full (360 degree) helixes only and output each helix individually.

	0.25	×	Poir	ıt 🟋
Angle (A)	10	* *		<b>A</b> ≹∰
🖲 Height (H)	0.25			нΣ₹
pproach Distance (D)	0.025			Æ

5. Click **Generate**. The Hole Pocketing operation is now created and is listed under the MOps browser.





- 4. Switch to the Simulate tab, select Hole Pocketing, and click by to launch the Alibre CAM Stock Simulation window.
- 5. Click Simulate from the Stock Simulation window to run simulation.



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

## Creating the Hole Pocketing Operation #2

#### **Getting Started with Alibre CAM**

1. Switch to the Create Operations tab.

#### **Creating the Hole Pocketing Operation**

- 2. Select the Hole Pocketing Operation created from the previous step, right mouse click, and select **Copy**.
- 3. Right click and select **Paste**.
- 4. This would create a copy of the Hole Pocketing Operation listed below the first Hole Pocketing operation as show below.



- 5. Expand the Hole Pocketing-1 folder and double click on Machining Features.
- 6. Click Remove All under Machining Features and click **Select Containment Regions**.
- 7. Switch to Design Explorer and select **Sketch8**.



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.

9. Switch to Cut Parameters tab and set the Hole Depth = **0.1875**, Hole Diameter = **0.315**.

Machining Feal	tures/Regions	Tool	Feeds & Speeds
Clearance	Cut Parameters	Entry/Exit	t Sorting
- Global Parame	eters		
Tolerance (t)	0.001		4
- Location of Cu	it Geometry	+	Ð
At Top	O At Bottom		Ð
O Pick Top	0 \$ B	9	$\mathbb{R}$
-Hole Paramete	ers	L CE	5 Js
Hole Depth (H	н) 0.1875 🚔 Ҭ		¥∕°
Use 3D M	odel to Detect Depth	<	×
Hole Diamete	r (D) 0.315 🛤		₿~
	ia as Hole Dia for	+ ((C	R)
any Arcs/	Circles selected		
Cleanup F	'ass at Each Z Level		<u>D</u> ''
-Cut Direction-			
💽 Cli	mb (Down Cutting)	🔘 Conventional (L	Jp Cutting)
Stepover Con	trol	- Stepdown Contro	ol (dZ)
💿 % Tool Dia	meter 25	💿 % Tool Diam	eter 50 🚖
O Distance	0.125	🔘 Distance	0.25
		Number of Le	evels 5
		6	

10. Switch to the Entry/Exit tab and set the Helix Diameter to **0.09**.

Helix Filch				°''' 2	52
Angle (A)	10	4 >		A	R
🖲 Height (H)	0.25			нт≥	R
oproach Distance (D)	0.025		-	Æ	K

11. Click **Generate**. The Hole Pocketing Operation for the inner holes is now created.

Setup 🔁 Create 🖉 Simulate	
🖻 🛃 - 🕹 - 🗞 - 💑 -	
😑 🞼 Machining Operations	
🖨 🎊 Setup	
Machine - 3 Axis	
Stock - Box Stock	
🖨 🦻 MOp Set 1	
‡ar Set MCS	
🕀 💬 2 1/2 Axis Facing	
🗊 💬 2 1/2 Axis Pocketing	
🗊 💬 2 1/2 Axis Pocketing-1	
🕀 📁 Hole Pocketing	
🕀 💬 Hole Pocketing-1	
😥 🛷 💀	
FlatMill-0.25 - #1, Dia:0.25, CRad:0, Taper:0 deg	

12. Switch to the Simulate tab, select Hole Pocketing-1, and click CAM Stock Simulation window.

to launch the Alibre

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

🗞 😥 🛷 केंद्र 🖓 6 Stock Visibility



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

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

# **Creating the Outer Profile Toolpath**

We will not create a 2 ½ Axis Profiling toolpath to cut out the shaft base. We will use the same Tool (FlatMill-0.25) and settings for Feeds /Speeds and Clearance plane for the profiling toolpath.

- 1. Switch to the Create Operations tab and select 2.5 Axis Milling and choose Profiling.
- 2. Go to the **Machining Features/ Regions** tab, click Remove All under Machining Features, and click **Select Containment Regions**.
- 3. Switch to Design Explorer and select **Sketch1**.



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.

5. The selected region is now displayed under Machining Features/Regions.

Ma	chining Features/Regions	Tool
#	Selected Machining Region(s)	
1	Region 1	



- 6. Switch to the Tools tab inside 2 <sup>1</sup>/<sub>2</sub> Axis Profiling operation and Select the FlatMill-0.25.
- 7. 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.
- 8. Switch to Clearance Tab. Set the Clearance Plane Definition to Automatic and Cut Transfer Method to Clearance Plane.

# **Specifying Cut Parameters**

- 1. Switch to Cut Parameters tab and use the following Settings
  - a. Tolerance = **0.001**,
  - b. Stock = **0**,
  - c. Cut Start Side- Check Use Outside/Inside for closed curves and pick **Outside**.

Machining Featu	es/Regions To	ool Feeds & Speeds Clearance
Cut Parameters	Cut Levels Entry	VExit Advanced Cut Parameters Sortin
Global Parame	eters	
Tolerance	e 0.001	Region
Chevel	. 0	An chord
Stock		
Lompensation	AUTO/NONE	→ ← Stock
- Cut Direction		
Climb (Dov	vn Cut)	10 <b>1</b>
O Convention	nal (Up Cut)	////
Mixed		
- Cut Start Side		
Right	🔿 Left	
🔽 Use Outsi	de/Inside for Closed Cu	rves ///
💿 Out	side 🔘 Inside	
	using 3D Model	Corner Cleanup
Stepover Con	trol	Total Cut Width
Total Cut W	/idth: 0	100
Step.		
A	4 - 1 - 1 - 1 - 1 - 1 - 1	
-		
		₩ Step/Cut

Select the Cut Levels Tab and specify Location of Cut Geometry At Bottom, Total Cut Depth = 0.25, Rough Depth/Cut = 0.05.

Machining Features/Regions Too Cut Parameters Cut Levels Entry/	ol Feeds & Speeds Clearance 'Exit Advanced Cut Parameters Sorting
Location of Cut Geometry At Top At Bottom Pick Top Cut Depth Control Total Cut Depth: 0.25 Rough Depth: Finish Depth: 0.25 0	Total Cut Depth Epinish Finish Depth/Cut Finish Cut Geometry At Bottom
Rough Depth/Cut: 0.05	Finish Depth/Cut:
	Use 3D Model to Detect Depth
Out Levels Ordering Depth First Level First	

- 3. Switch to Entry/Exit Tab and Set the Entry and Exit Type to None.
- 4. Click **Generate**. The 2<sup>1</sup>/<sub>2</sub> Axis Profile toolpath is now generated and the Operation is listed under the Alibre CAM-MOps browser.

Toolpath display can be turned on/off by selecting Toolpath Visibility under the MOps browser.





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

8	😣 😥 🐼 🐼 😣	
	Stock Visibility	

5. Switch to the Simulate tab, Select the 2 ½ Axis Profiling Operation and click launch the Alibre CAM Stock Simulation window.

to

- 6. Click Simulate from the Stock Simulation window to run simulation.
- 7. Once the simulation is complete, you can close the Stock Simulation window and return to the Alibre CAM browser.



## **Post Processing**

Once the toolpath has been generated, it can be post-processed to a specific machine controller.

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



2. Specify the File Name as **Shaftbase.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