

Introduction

This tutorial will illustrate machining a Sign using 2-1/2 axis-engraving operations. We can engrave the sign using 2-D Sketches. This tutorial will introduce the usage of $2-\frac{1}{2}$ axis simple V Carving using V bit.

V carving refers to a cutting strategy employed by sign makers to create sharp corners. V carving is performed using a tapered bit or conical tool (as shown below) usually known in the industry as a V Bit.



The V-bit is made to rise from the cutting depth to the top of the surface at the corners in such a way that the tapered sides of the cutter are always in contact with the corners. When the cutter finally reaches the top surface, only the bottom tip of the tool will be in contact with the corners, thereby creating clean and crisp cuts at the corners.

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

- V carving is performed using the 2 ¹/₂ axis Machining Operation.
- The part itself will be machined out of a 10.75 inch x 4 inch x $\frac{1}{2}$ inch poplar wood sheet
- The part would be machined using a single V-Groove bit.
- The wooden sheet will be held to the machine table or the spoil sheet on the table using double-sided tape.

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 **V-Carve1.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 imported part appears as shown below



Loading the Alibre CAM Browser



1. Select Alibre CAM from the menu bar and click Machining Operations 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. 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 3 axis

lachine			Þ
Machine Setup			
Machine Type			
🧿 3 Axis	🔿 4 Axis	🔿 5 Axis	
- Tool Change Position			
×O	Y 0	≥ Z 0	÷ ->
- 4th Axis (Primary Axis)			
Rotary Center: X 0	÷ Y 0	🔶 Z 0	÷ R
Rotary Axis: ()>	KAxis OYAxis	O Specify	
X 1	Y O	Z 0	
- 5th Axis (Secondary A	wis)		
Rotary Center: X 0		🔶 Z 0	÷ N
Rotary Axis: X 0	Y 1	Z 0	
Gage Length	0		
Output all Co-ore	dinates in Rotated Co-c	rdinate System	
	ОК	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

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: _nc 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 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 hotes about stock models:
 Once the stock is created, the 3-D bounding box of the stock model will be rendered in the Alibre Design graphics window. Please note that this is not the actual stock model. The actual stock model will be displayed only in the simulation window.
 Also make sure that you use the Polygonal Stock model when working with rotated MCS operations. The Voxel Stock model can only be used when the MCS is parallel to the global XYZ system. You can observe the simulation model type is the Simulation Settings dialog.
Tou can choose the sinulation model type in the sinulation 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) = 10.75, Width W = 4.00 and Height (H) = 0.5. Make sure to set the corner position to Southwest corner Top of Stock as shown below.

Box Stock 🛛 🔀
Stock Geometry
Corner Position
Height
(H)
Z Width (W)
- Corner Coordinates
Xc 0 Yc 0 Zc 0 🕀
C Dimensions
L 10.75 🖤 W 4 🗭 H 0.5 💌
Copy Model Bounding Box
OK Cancel Help

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



4. 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

1 Sta	setu	p 🔁	Create	🧐 s	imulate	
œ		203 -	% •	0	• 🙀 🖓	1
Θ [赺 Ma	chining	Operat	ions	~	

2. Set Z alignment to **Top** 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	$\overline{\mathbf{X}}$
Align Part and Stock		
	-	
Z Alignment —		
⊙ Top	🔿 Center	◯ Bottom
XY Alignment		
O North West	O North	O North East
O Mid-West	💿 Center	O Mid-East
O South West	🔿 South	O South East
L		
		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.

1. Select Set MCS from the Setup tab



 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.

Set MCS	×
Set Machining Coordinate System	
Z Axis	
Align MCS With Set MCS Origin Set Fixture Offsets	
C Pick Set to Stock Box Set to Part Box Zero Face Highest Z Mid Z Lowest Z	
Zero Position C North West C North C North East C West C Center C East	
South West South South	
Generate Cancel Save Hel	p



(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 V-Carve1-Rev1.

Create Tools

To machine the above part, we will now create a 60-degree Taper Tool.

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 VeeMill.

r* 15 (1 15 15)	
Tools	

Create	e/Selec	t Tool:		
Ţ	T	T	Ŗ	7

2. Set the tool name as **VeeMill1**, Taper Angle = **30**, Flute Length = **0.4**, Tool Length = **2**. Under the Properties tab set Tool Number = **1**.

Create/Select Tool	×
Create/Select Tool T	Properties Feeds & Speeds Material HSS
H Holder Diameter → Ť Holder Length ± Taper Angle 30 ♥ Flute Length 0.4 ♥	Number of Flutes 2 Tool Number 1 Adjust Register 0 Cutcom Register 0 Zoffset 0 Coolant None Comments
Save as New Tool Save B	Edits to Tool Delete Tool Cancel Help

Note: Taper Angle represents the included angle for a taper tool. For example a 60-degree taper tool would have a included angle of 30 degrees. If you have a taper tool with a diameter select Chamfer Mill or Taper Mill under Create/Select Tool.

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 Fe	eeds & Speed	ls		
- Spindle Spe	ed 8000	*	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 Feedrate (Tf)				
Ose Rapid				
🔿 Set	50	*	in/min	
L				

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 tools are now listed under the Cutting Tools browser.

Create Machining Operations

In this process we will create a 2.5 axis V-Carve operation.

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



V-Carving

2. Select 2.5 Axis Mill and choose V-Carving.



This brings up the V-Carving Operations dialog. We will 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 V-carve operation dialog is now minimized and allows selection of the sketch geometry.

V-Carving	×
Clearance Cut Parameters Sorting Machining Features/Regions Tool Feeds & Speeds	
# Selected Machining Region(s) Image: Select Containment Regions Image: Select Containment Regio	
Select Avoid Regions Select Flat Area Avoid Regions Remove All Remove Active	
Generate Cancel Save Help	

4. Switch to Design Explorer and select **Sketch2**. This will select the text.



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.

6. V-Carve operation dialog comes back up displaying the selected regions. The selected regions are also highlighted on the part

t	Selected Machining Region(s)
	Region 1



Selecting the Tool

Clearance	Cut Parameters	Sorting
Machining Features/Regions	Tool	Feeds & Speed
-		
	E Tool Geomet	ry
VeeMill1	Diameter	0
<u>↓</u>	Corner Radius	U
12.18	Taper	30
	Tip Angle	U
	Tool Propert	es
	Tool Name	VeeMill1
	Tool #	1
	# of Flutes	2
	Cutcom Regist	er O
	Adjust Registe	0
	Z-Offset	0
	Material	HSS
	Coolant	None
	Comments	
	E Feeds & Spe	eds
	Spindle Speed	8000
	Feed Rate	30
	Edit/Create	/Select Tool ew Tool

7. Switch to the Tools tab inside the V-Carving operation.

8. Select VeeMill1. VeeMill1 is now selected as the active tool and the Tool parameters are displayed to the right of the Tools window.

Set Feeds and Speeds

9. Click on the Feeds and Speeds tab.

/-Carving			X
Clearance	Cut Parameters		Sorting
Machining Features/Regions		Tool	Feeds & Speeds
Spindle Speed	RPM		
- Feed Rates			
Plunge (Pf) 20 🚔	in/min		
Approach (Af)	in/min	Pf	
Engage (Ef) 20	in/min		Cf Tf
Cut (Cf) 30 🚔	in/min	Af	
Retract (Rf) 20	in/min	Ef	Df/
Departure (Df) 20	in/min		RI RI
Transfer (Tf)			
💿 Use Rapid			
Set 50	in/min		
Load From Tool			
Load From Table			
		bee bee	
Gene	erate	Cancel	Save Help

10. Select Load From Tool. Alibre CAM will now get the feeds and speeds information that was set when the tool was defined.

Clearance Control

11. Switch to Clearance Tab.

Machining Features/R	egions	Tool	Feeds & Speeds
Clearance	С	ut Parameters	Sorting
Clearance Plane Definition	n		Stock Max Z
O Automatic			Part
🔘 Part Max Z + Dist	0.25		Max.
O Stock Max Z + Dist	0.25		
Absolute Z Value	0.25		
● Clearance Plane		Part Ma	
		ang kang kan	-

12. Set the Clearance Plane Definition to Absolute Z Value = **0.25** and Cut Transfer Method to Clearance Plane.

Setting Cut Transfer to Clearance Plane would apply the Absolute Z value clearance between transfers when the tool moves from a machining region to another.

Specifying Cut Parameters

13. Switch to Cut Parameters tab.

Machining Features/Regions	Tool	Feeds & Speeds
Clearance	Cut Parameters	Sorting
Global Parameters		
Tolerance: 0.001		
Cut Side	_	
⊙ Inside 🔿 Outside		
Location of Cut Geometry	_	
💿 At Top 🛛 🔿 At Bottom		
O Pick Top		ut Geometry at Top
Cut Depth Control		→ ± Depth/Cut →
Total Cut Depth: 0.2		Finish Depth/Cut
Rough Depth: Finish Depth:	Depth	
0.2		
`		
Rough Depth/Cut: 0.05	Finish Depth/	/Cut: 0
	<u> </u>	

14. Set the Tolerance = **0.001**, Cut Side = **Inside**, Location of Cut Geometry = **At Top**, Total Cut Depth = **0.2**, Rough Depth = **0.2** and Rough Depth/Cut = **0.05**.

Sorting

- 15. Select the Sorting Tab.
- 16. Use Minimum Distance Sort & set Start Point as Lower Left.

Machining Features/Reg	gions	Tool	Feeds & Speeds
Clearance	С	ut Parameters	Sorting
◯ No Sort (Minimum 	Distance Sort 🛛 🤇) Directional Sort
Minimum Distance Sort			
Olloper Left	Inner Bight		
	ower Bight		
Lonor Lon VI	onerngrit		
Directional Sort			
Primary Sort Direction	(P)		
Start Angle (A)	*		
- Secondary Sort Direct	tion (S)	<u> </u>	
O Low to High O	High to Lov	Start Point	End Point
Traversal Pattern			
O Zig O	ZigZag		1. 1. 1. 1. 1. 1. 1. 1. 1. 1. 1. 1. 1. 1

This will sort based on the minimum distance between 2 text based on their start points.

17. Click **Generate**. The V-Carving toolpath is now generated and the Operation is listed under the Alibre CAM MOps browser.

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





Simulate Toolpath

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



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



- 2. Select the V-Carving 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 **V-Carve.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 www.lista.it info@lista.it