

MAKER WEB

CNC SOFTWARE CERTIFICATION VCARVE PRO



CNC TUTORIAL DESCRIPTION

A CNC Milling machine is a computer controlled machine that can be used to make very precise 3D parts. The machine works by cutting or removing material from the stock material with a rotating cutting tool. The machine does this by guiding the tool in all three directions of the cartesian coordinate system, that meaning along the X, Y and Z axis. This tutorial will certify you to create and submit files for the CNC Machine. Before proceeding with this tutorial you must visit the CNC Lab website and review the CNC Basics PDF.

TABLE OF CONTENTS

Part 1: Startup and Interface
Part 2: Skills and Learning Objectives
Part 3: Parameters and Cutting Area
Part 4: VCarve Startup Material Settings
Part 5: Importing Geometry
Part 6: Viewing, Modifying and Moving Geometry
Part 7:Creating Roughing Toolpath
Part 8: Creating Finishing Toolpath
Part 9: Creating Pocketing Toolpath
Part 10: Creating Profiling Toolpath
Part 11: Creating Job Sheet
Part 12: Exporting Toolpaths
Part 13: Visiting the CNC Lab

SUPPLEMENTAL RESOURCES

https://www.vectric.com/support/tutorials/vcarve-pro

Part 1: Startup and Interface



In order for the CNC machine to work it needs coded instructions to tell it what to do. While a design in made in CAD (computer Aided Drafting) software, "toolpaths" and the resulting code is generated in CAM (Computer Aided Machining) software. There are many CAM software packages available but at Union we use a software called VCarve Pro. This software uses a 2D or 3D design and allows the user to create various toolpaths which can be exported as code files for the CNC machine.



VCarve Pro can create 2D toolpaths to cut out material and can also create 3D tool paths to sculpt complex 3D objects. This CNC certification tutorial is intended to introduce you to the basic steps to create toolpath operations which will be checked and run by CNC operators in the CNC Lab. The software is installed on a computer in the Digital Arts Lab in the Visual Arts Building. Please visit the Laser Cutting Lab during our open hours to use VCarve Pro.

Part 2: Skills and Learning Objectives



This tutorial will enable you to create 4 toolpath operations that can be exported and subsequently cut on the CNC machine by one of the operators. The CNC Lab supplies all the bits and materials for the certification tutorial. Once your file is cut correctly you will be placed on the certified list and can subsequently create files, schedule cut times and make use of the CNC machine. Please note that only Maker Web employees and specifically trained faculty and students will OPERATE the CNC Machine. This tutorial DOES NOT cover operation of the machine.



As part of the tutorial you will create the following operations: 1) Horizontal Roughing 2)Parallel Finishing 3)Pocketing 4)Profiling. VCarve offers several other toolpath types and within each toolpath type there are many options. However, these 4 are the most common and will enable you to create a variety of work on the CNC in the future.

Part 3: Parameters and Cutting Area



The template file includes a 3D surface and 2D lines for various inputs. The CNC Lab provides the template file, bits and an 8" x 8" square pieces of MDF, plywood and foam for the tutorial. All you need to do is create all the appropriate files and bring them to the Lab during our open hours.



Before getting started there are some important things to note about the CNC machine and the geometry it can and can not cut. Take note of undercuts and overall cut depth compared to bit length.

Part 4: VCarve Startup Material Settings



Start VCarve and Create a new file.

Image: Second	VCarve Pro Trial Edition 10.010 - [New] File Edit Model Toolpaths View	Gadgets Help	- <u> </u>
NOTE: This stock material size is specific to this tutorial and will have to be modified for each future cut based on the actual material being used.	🤏 Job Setup	2D View 3D View Q Layer 1 -	[2] [2] [2] [2] [2] [2] [2] [2] [2] [2]
Image: Control of the control of th	Job Type Single Sided Double Sided Rotary		
Image: State of S	Job Size ridth (r); 203.2 mm Height (r); 203.2 mm m,		
Working Resolution will have to be modified for each future cut based on the actual material being used. Indeling Resolution each future cut based on the actual material being used.	Z Zero Position Material Surface Machine Bed		NOTE: This stock material size is specific to this tutorial and
Hodeling Resetution The High (2 Solver) The High	XY Datum Position	 8	will have to be modified for each future cut based on the
	Hodeling Resolution Very High (7x Slower) ~ 4 million points Appearance 2 Canadan Naple ~		actual material being used.
	Solid Calor:		
	Cont of the second seco	• •	

The home screen has some material options. Most of the defaults are correct but you will have to fill in the material size in millimeters. For this tutorial we will be using a 8" x 8" x $\frac{3}{4}$ " piece of sheet material. In millimeters this comes to 203.2mm x 203.2mm x 19.05mm. Input these values and make sure the Z=0 position is set to Machine Bed and the resolution is set to Very High. Click OK.

Part 5: Importing Geometry

VCarve Pro Trial Edition 10.010 - [New]			o ×
File Edit Model Toolpaths View O	Sadgets Help		- 8 ×
Drawing 💽 ? 👎	2D View 3D View Q Layer 1 -	P 🔣 🏥 🗇 🔎 👰 🧖 🚫 🧇 🐨 📰	Too
File Operations	FFF	200	
L 📁 🗔 🗁 💆 H 4 h 6 v 9 9			^ _
Create Vectors			
2 7 5 0 T T & T *	-		
Fransform Objects	-		
Edit Objects			
< 7 2 0 M 2 0 1 > 1	8-		
Offset and Layout			
Job Dimensions			
Width (X): 203.2 mm Height (Y): 203.2 mm Depth (Z): 19.05 mm			
Drawing Modeling Clipart Layers	6		~

This is the VCarve user interface. From here you can import or create geometry, create and edit and preview toolpaths and finally export them. For this tutorial you will want to import the accompanying template file. It is an STL file.

VCarve Pro Trial Edition 10.010 - (New)	Inets Help	
Rew Ctri-N Rew Ctri-N Rew Ctri-N Save Ctri-O Close Save Ctri-S Save As Increment and Save Ctri-Alt-S Print	200 Miew 30 View @ Layer 1 - 2	
Print Preview Print Setup Import Export Open Application Data Folder	Import Vectors Ctrl+1 Import Bitmsp Import Component / 3D Model	NOTE: There are many types
1 0:\UNION DOCS\Photo file_1 2 0:\UNION DOCS\lotmot2 layer 3 0:\UNION DOCS\CAM file_1 4 C:\Users\\191025_CAM file Exit	Import PhotoVCarve, Machinist or Cut3d Toolpaths	of 2D and 3D files that can be imported and there are many
 < ♥ ∽ ⇔ ☎ ☎ ∞ ○ > > > Offset and Layout ○ ⊕ ∞ ∞ ∞ ∞ ∞ ∞ 	2- - - -	ways to extract geometry from 3D files. For the balance of information see the VCanve
		Manual.
Job Dimensions Width QQ: 203.2 mm Height (Y): 203.2 mm Depth (Z): 19.05 mm		
Impact a component or 20 model	<	× 28 2771 V 82 0002

Click New > Import > Component/3D Model. Navigate to where you have the template files located and select the "surface" file and import it.

Part 6: Viewing, Modifying and Moving Geometry



Once you have selected and imported the file there will be some options about position and scale. All of the defaults can stay the same except the Zero Plane. This has to match the zeroing strategy defined by the material description. Make sure the zero plane is **ALWAYS** set to the lowest it can be or the bottom of the material. In this case it's set to 19.05mm.



Once you click okay, you will see a 3D Preview of you model. It should be in the appropriate XYZ position but if necessary it can be moved by clicking on the 2D View Tab. To navigate around the model use the mouse. The **Left** mouse button rotates the model. The **Right** mouse button zooms in and out and the **Middle** mouse button pans around the model.

Part 6: Viewing, Modifying and Moving Geometry



If you click on the surface it selects it. Click the surface again to show the grips. Moving these grips can move, rotate and scale the surface. We don't want to modify the surface at all in this tutorial but it is important to know.



Next we want to import the vector lines that will define various boundaries for milling. Click File > Import > Vectors. Navigate to where you have the template files located and select the "boundaries" file and import it.

Part 6: Viewing, Modifying and Moving Geometry



You should see the outlines in purple. Click off them to deselect them. They should now be black.



Now that we have all the geometry files imported we want to go back to the 3D View and start making tool paths. Click on the 3D View button then the Toolpaths Button on the upper right. This will open the toolpath dialog window.

Part 7: Creating Roughing Toolpath



It is important to sequence your toolpaths correctly. For this tutorial we want to carve the surface then cut the pocket and finally cut the piece out. In order to carve the smooth surface we need to cut deeply into the material and also use a small bit. Because that will exceed the recommended cut depth we must first rough out the surface in steps and then finish it with a different operation. To do this we will first choose "3D Roughing Toolpath".



This opens the toolpath dialog box where you can modify all the settings for the cut. Most of the default setting are fine but we do need to select the bit and set the appropriate parameters. Click on the Select Bit button which will open the dialog box.

Part 7: NOTE: The values given are **Creating Roughing Toolpath** for this tutorial only and will be different for different bits and materials. Always refer to the File Edit Model Toolpaths View Gadgets Help Feeds and Speeds Guide for Modeling 🛃 ? 👎 2D View 3D View @ Default 3D Model T speeds and note that the cut 📂 🔎 🗇 🕎 👌 🛋 🛸 Tool Database depth should equal the Material MDF ~ 🗷 ٢ ~ 3 Components Hingerial Tools End Mil (1/2') Bal Nose (1/2') Hingerial End Mills End Mills End Mill (1/8') End Mill (1/8') Hall Nose diameter of the bit and that the Ball Nose (1/2") □ - V <u>∩</u> Level 1 - V ∩ surface Notes stepover should be in the Tool Type range of 20-50% depending on Ball Nose (1/8") Ball Nose (1/4") V-Bits V-Bit (30.0° - 1/4") V-Bit (90.0° - 1/2") V-Bit (90.0° - 1 1/4") Form Tools Units 0 desired finish. Ogee - 1/4" Rads 1 1/4" [Roundover - 3/8" Rad 1" No. Flute Z Level **O 3D Raste** (20.0°, Tip 0 ev End Mills End Mill (3 mm) End Mill (6 mm) Profile Last \$ 9 Stepove Order Level by level ~ Raster Direction Along X ~ Ball Nose (3) Ball Nose (3) Ball Nose (6) Spindle Speed Reverse step direction Feed Units 0.009 Ball Nose (6 mm) U-Bits V-Bit (20.0° - 6 m V-Bit (90.0° - 12 V-Bit (90.0° - 32 Ramp Plunge Moves Feed Rate Distance 0.0 mm Plunge Rate - Form Tools III Engraving Tool Nu Safe Z 44.05 mm Home Position X:0.00 Y:0.00 Z:44.05 ove Apply Rem Vector Selection: Manual Selector ... ⊕ ₩ 0 ₽ 8 同 Select Close Name: 3D Roughing 1 Close Calculate Modeling Clipart Layers

First, select the ½" Ball Nose. The default values may not be correct so be sure to check EACH of the parameters. First make sure the **Diameter** is set to .5". Next make sure the **Number of Flutes** is set to 2. Make sure the **Pass Depth** and **Stepover** are set to .25". Finally set the Spindle Speed to 10,000 rpm, the Feed Rate is set to 180 inches/minute and the Plunge Rate is set to 30. Click Apply and then Select. For more information on these values please refer to the CNC Basics PDF.



Once you have set all of the parameters an clicked Select. The dialog box will disappear and you are ready to process the toolpath. Click "Calculate". The software will do some processing and show you a preview of the toolpath and give you a chance to simulate the movement of the bit.

Part 7: Creating Roughing Toolpath



Once you see the toolpaths you can click on the Play button to the right to begin the simulation. To view it again you can click the Reset Preview and Play again.



Inspect the preview of the cut to be sure everything looks okay. If everything looks appropriate click on the Close button to finalize the toolpath.

Part 8: Creating Finishing Toolpath



This will bring you back to the root toolpath dialog box. You will now see your first toolpath operation has been created at the bottom of the dialog box. Note that you can edit this toolpath by double clicking on it which opens up the previous parameters dialog box. Next we want to create our next toolpath, 3D Finishing. Click the 3D Finishing button.



The cut parameters dialog window should look familiar as it is similar to the dialog box for each operation because they each have some similar parameters. Like before we want to leave most of the default settings as they are but we need to change the Boundary Offset to 3mm. Also we need to select and modify the Bit and it's cutting parameters. While we used a ½" ball mill for roughing, we will use a ¼" flat end mill for the finishing operation. Click the Bit Selection button.



Select the ¼" End Mill to open the its parameters. These should look familiar to you. There are some differences though and like before it is important to review each setting. make sure the **Diameter** is set to .25". Next make sure the **Number of Flutes** is set to 2. Make sure the **Pass Depth** is set to .25" and the **Stepover** are set to **25%**. Finally set the Spindle Speed to 10,000 rpm, the Feed Rate is set to 180 inches/minute and the Plunge Rate is set to 30. Click Apply and then Select. For more information on these values please refer to the CNC Basics PDF.



Once you have set all of the parameters an clicked Select, the dialog box will disappear and you are ready to process the toolpath. Click "Calculate". The software will do some processing and show you a preview of the toolpath and give you a chance to simulate the movement of the bit.

Part 8: Creating Finishing Toolpath



Once you see the toolpaths you can click on the Play button to the right to begin the simulation. To view it again you can click the Reset Preview and Play again. You will also see your new toolpath added to the list at the bottom. Click Play to watch the simulation.



Inspect the preview of the cut to be sure everything looks okay. If everything looks appropriate click on the Close button to finalize the toolpath.

Part 9: Creating Pocketing Toolpath



This will bring you back to the root toolpath dialog box. You will now see your list of toolpath operations at the bottom of the dialog box. Note that you can edit these toolpaths by double clicking on them. Next we want to create our next toolpath, Pocketing. Click the Pocketing Button.



The cut parameters dialog window should look familiar. Because a Pocketing operation requires a boundary we will switch to a 2D view and select the inner vector line as the boundary. It should turn purple. Like before we want to leave most of the default settings as they are but we need to change the Cut Depth parameters. Set the Start Depth to 0 and the Cut Depth to 15mm. Also we need to select and modify the Bit and it's cutting parameters. Click the Bit Selection button.



For the Pocketing Operation we will be using the same bit as the Finishing Operation, the ¹/₄" End Mill. It should retain its settings from before but review to be sure. Make sure the **Diameter** is set to .25". Next make sure the **Number of Flutes** is set to 2. Make sure the **Pass Depth** is set to .25" and the **Stepover** are set to **25%**. Finally set the Spindle Speed to 10,000 rpm, the Feed Rate is set to 180 inches/minute and the Plunge Rate is set to 30. Click Apply and then Select.



Once you have set all of the parameters an clicked Select, the dialog box will disappear and you are ready to process the toolpath. Click "Calculate". The software will do some processing and show you a preview of the toolpath and give you a chance to simulate the movement of the bit. For better viewing you can switch back to the 3D View.

Part 9: Creating Pocketing Toolpath



Once you see the toolpaths you can click on the Preview all Toolpaths button to the right to begin the simulation of each sequential toolpath. To view it again you can click the Reset Preview and Play again. You will also see your new toolpath added to the list at the bottom.



Inspect the preview of the cut to be sure everything looks okay. If everything looks appropriate click on the Close button to finalize the toolpath.

Part 10: Creating Profiling Toolpath



This will bring you back to the root toolpath dialog box. You will now see your list of toolpath operations at the bottom of the dialog box. Next we want to create our final toolpath, Profiling. Click the Profiling Button.



A Profiling operation also requires a boundary so we will switch to a 2D view and select the outer vector line as the boundary. It should turn purple. First, we need to change the Cut Depth parameters. Set the Start Depth to 0 and the Cut Depth to 20mm. We also want to set the cut to be Outside/Right of the vector line. Finally we need to select and modify the Bit and it's cutting parameters. Click the Bit Selection button.

Part 10: NOTE: The values given are Creating Profiling Toolpath for this tutorial only and will be different for different bits and materials. Always refer to the File Edit Model Toolpaths View Gadgets Feeds and Speeds Guide for Modeling 2 7 @ Default 2D View 3D View speeds and note that the cut Tool Database " A 🕸 🖾 A 👟 MDF Online (1) ~ 2 Machine Desktop × 1 depth should equal the Components End Mill (1/4") rial Tool End Mill (1/2" diameter of the bit and that the ■ I Level 1 I C Surface stepover should be in the 1 range of 20-50% depending on Geometr Ball N Diameter (D) desired finish. No. Flute: Distance Cutting Pa 100.0 mm Pass Dept 0.0625 inches 25 Add tabs to toolpath Length 12.0 mm Ŧ Feeds and Sp Spindle Sp 3.0 mm Feed Units 0.009 inche Edit Tabs ... Feed Rate V-Bit (20.0° V-Bit (90.0° V-Bit (90.0° Form Tools Plunge Rab ition X:0.00 X:0.00 7:44 05 Name: Profile 1 Close Calculate ⊕ № Ĉ 🖻 🖯 Close Select X:412.3101 Y:275.7342 H-136.071 L-Defau

For the Profiling Operation we will be using the same bit as the Finishing Operation, the ¹/₄" End Mill. It should retain its settings from before but review to be sure. Make sure the **Diameter** is set to .25". Next make sure the **Number of Flutes** is set to 2. Make sure the **Pass Depth** is set to .25" and the **Stepover** are set to **25%**. Finally set the Spindle Speed to 10,000 rpm, the Feed Rate is set to 180 inches/minute and the Plunge Rate is set to 30. Click Apply and then Select.

VCarve Pro Trial Edition 10.010 - [New]		— 🗆 ×
File Edit Model Toolpaths View	Gadgets Help	_ 8 ×
Modeling	30 View 🖉 Default -	Toolpaths 📑 ? 🍟 🚽
3D Model Tools 🚰 🖉 🐟 🛸	H = 31	2D Profile Toolpath
Components □-♡ <u>∩</u> Level 1		Start Depth (D) 0.0 mm Cut Depth (C) 20 mm
- I Surface		Tool: End Mil (1/4") Select Edit
	Vcave Pro Trial Edition X WARNING - Tool will cut through material The current tool cutting depth will exceed the material thickness. Material thickness = 19.050 Maximum tool depth 20.000 Pressing O.C. will continue with toolpath calculation, However, the resulting toolpath WILL CUT THROUGH the base of the material possibly damaing the machine bed. If you do not have a secrificial layer under the material being out you should cancel this operation. OK Cancel	Machine Vectors. Image: Conventional Disade / Right Image: Conventional Disade / Left Image: Conventional Discrete 0.00 mm Discrete 10.00 mm Add tabs to toolpath Image: Conventional Image: Conventional Image: Conventional
Drawing Modeling Clipart Layers		

Once you have set all of the parameters an clicked Select, the dialog box will disappear and you are ready to process the toolpath. Click "Calculate". The software will warn you that you are cutting through the material. Since your Profiling toolpath is set to cut to 20mm. It is .95mm deeper than the material to be sure it's a clean cut. Click okay. Next you should see a preview of the toolpath and have a chance to simulate the movement of the bit. For better viewing you can switch back to the 3D View.

Part 10: Creating Profiling Toolpath



Once you see the toolpaths you can click on the Preview all Toolpaths button to the right to begin the simulation of each sequential toolpath. To view it again you can click the Reset Preview and Play again. You will also see your new toolpath added to the list at the bottom.



Inspect the preview of the cut to be sure everything looks okay. If everything looks appropriate click on the Close button to finalize the toolpath.

Part 11: Creating Job Sheet



This will bring you back to the root toolpath dialog box. You will now see your list of toolpath operations at the bottom of the dialog box. Next we need to create a Job Sheet which describes all of the cutting operations. Click the button and save the HTML file in a location you can find. This file must be opened in a browser window, saved as a PDF and brought to the CNC Lab with your other files.



Simply double click the HTML file to open it in a browser window and then print as a PDF.

Part 12: Exporting Toolpaths



Your Job Sheet should be a multipage page document similar to what you see above. Note all of the valuable information including cut times, cut parameters, bits used and stock material size.



Finally we want to export each of the toolpaths individually. Highlight each of the tool paths individually and click the Save Toolpath button. When you save the files it's very important to be sure you have the correct post processor. On the pulldown menu be sure to choose **Laguna Swift CNC (mm) (*.mmg)**. Save the files folder you can locate. You should end up with 4 .mmg files. Be sure to name them as follows: Your initials_cut number_cut type (for example CB_1_RT, CB_2_FT, CB_3_PoT, CB_4_PrT).

Part 13: Visiting the CNC Lab

Please note you must be present for the duration of your cut. It should take about 30 minutes and it is an important part of the process to be sure there are no issues with the cuts



You are now ready to have your files cut in the CNC Lab. Please visit the CNC Lab Website and submit your files (1-"Cutting Checklist" 2-Your VCarve Pro file (.CRV), 3-Your Job Sheet (.PDF), and 4- Your cut Files x4 (.MMG). We will contact you to schedule a time for the cut. Please do not visit the Lab without a scheduled appointment.

NOTE: You are responsible for providing materials for all your projects after the certification tutorial. The Lab provides materials for the tutorial ONLY.

NOTE: The CNC Lab provides a limited number of bits to students but it's best to check with us to verify bit availability before making subsequent toolpaths after the tutorial.

NOTE: For every future cut you want to make on the CNC you must submit it through our website. You will need to upload a completed "Cutting Checklist", Your VCarve Pro file (.CRV), Your Job Sheet (.PDF), and your Cut Files x4 (.MMG).

Once your file is cut correctly you will be placed on the certified list and can subsequently create files, and have them cut on the CNC machine. Please note that only Maker Web employees and specifically trained faculty and students will OPERATE the CNC Machine. This tutorial DOES NOT cover operation of the machine.