



## CNC Shark Post Processors v1.5

The latest CNC Shark Post Processors provide 3 new functional features:

- Ready to use tolerance modes in the form of two different post processors.
- Contains the information needed by VCarve Pro and Aspire to enable sending the tap file from the Vectric application directly to the CNC Shark Control Panel.
- Places key information directly into the tap file about the toolpath(s) selected.

This document provides installation instructions and details of the new functional features found in the CNC Shark Post Processors v1.5.

## Installing the CNC Shark Post Processors

You can find the latest CNC Shark Post Processors at the Next Wave Automation web site's Download page. You should initially save the post processor file(s) to a location on your disk that you will remember. This is typically in a folder in your /My Documents. Once you have saved it there, copy the downloaded file to the appropriate Vectric application /PostP folder. You should not attempt to copy the post processor directly from the Downloads page to the specific /PostP folder on disk. On many Vista and most Win 7 operating systems, this will result in a failure in the ability for the Vectric application to use the CNC Shark post processor file. In addition, most current anti-virus/internet security applications will block you from attempting to copy directly from the Downloads page to the specific /PostP folder on disk.

**64bit Operating System (OS) users  
should be sure to review the note at  
the end of this section.**

## VCarve Pro/Aspire/Cut3D Post Processors and Installation

The following post processors are for use with VCarve Pro (version 3.1 and higher), Aspire (version 2.0 and higher) and Cut3D (only):

CNCShark-USB\_NewArcs\_inch (Updated 01/08/2012)  
CNCShark-USB\_NewArcs\_mm (Updated 01/08/2012)  
CNCShark-USB\_3dContour\_inch (Updated 01/08/2012)  
CNCShark-USB\_3dContour\_mm (Updated 01/08/2012)

### CUT3D

The /PostP folder for Cut3D is located in C:\Program Files\Cut3D\PostP in both WinXP, Vista and Win 7 systems

### VCarve Pro v5.5 and Earlier

The /PostP folder for VCarve Pro v5.5 and earlier is located in C:\Program Files\VCarve Pro\V5.5\PostP.



- For versions previous to v5.5, the folder name 'V5.5' will reflect that version. For example, VCarve Pro v5.0 - \V5.0\PostP

### **Aspire 2.5 and Earlier**

The /PostP folder for Aspire v2.5 and earlier is located in C:\Program Files\Aspire\V2.5\PostP

### **VCarve Pro v6.0 and Later – WinXP Operating System**

The /PostP folder for VCarve Pro v6.0 and later is located in C:\Documents and Settings\All Users\Application Data\Vetric\VCarve Pro\V6.0\PostP

- For versions later than v6.0, the folder name 'V6.0' will reflect that version. For example, VCarve Pro v6.5 - \V6.5\PostP

### **Aspire 3.0 and Later – WinXP Operating System**

The /PostP folder for Aspire 3.0 and later is located in C:\Documents and Settings\All Users\Application Data\Vetric\Aspire\V3.0\PostP

- For versions later than v3.0, the folder name 'V3.0' will reflect that version. For example, Aspire v3.5 - \V3.5\PostP

### **VCarve Pro v6.0 and Later – Vista and Win 7 Operating Systems**

The /PostP folder for VCarve Pro v6.0 and later is located in C:\Program Data\Vetric\VCarve Pro\V6.0\PostP

- For versions later than v6.0, the folder name 'V6.0' will reflect that version. For example, VCarve Pro v6.5 - \V6.5\PostP

### **Aspire 3.0 and Later – Vista and Win 7 Operating Systems**

The /PostP folder for Aspire 3.0 and later is located in C:\Program Data\Vetric\Aspire\V3.0\PostP

- For versions later than 3.0, the folder name 'V3.0' will reflect that version. For example, Aspire v3.5 - \V3.5\PostP

### **Photo VCarve Post Processors and Installation**

The following post processors are for use with Photo VCarve (only)

CNC Shark-USB\_inch (Updated 11/29/2010)

CNC Shark-USB\_mm (Updated 11/29/2010)

The /PostP folder for Photo VCarve is located in C:\Program Files\PhotoVCarve\PostP in both WinXP, Vista and Win 7 systems

### **64 bit OS (Win XP/Vista/Win7)**

For all locations listed above that begin with 'C:\Program Files\...' - on a 64 bit system this will be 'C:\Program Files (x86)\...'

### **VCarve Pro 6.0 (and newer) and Aspire 3.0 (and newer) Note**

The locations listed above refer to default installation settings. Another way you can determine the location of your VCarve Pro 6.0 (and newer) and Aspire 3.0 (and newer) Application Data folder is to



select File, Open Application Data Folder. This will open the Application Data folder in a Windows Explorer window.

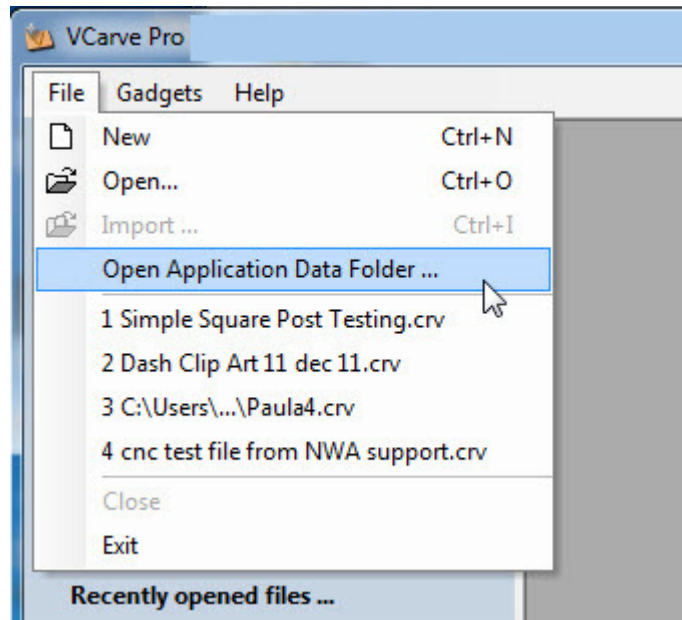


Figure 1 - VCarve Pro Open Application Data Folder

## Troubleshooting CNC Shark Post Processor Installation

### *Can't 'Find' the Folder in Windows Explorer*

The typical default setting in Windows Explorer is to hide hidden files, folders and drives.

- In Windows XP, the folder 'C:\Documents and Settings\All Users\Application Data' may be 'hidden'.
- In Windows Vista and Windows 7, the folder 'C:\Program Data' may be hidden.



To change this setting, select Tools, Folder Options from the Windows Explorer toolbar.

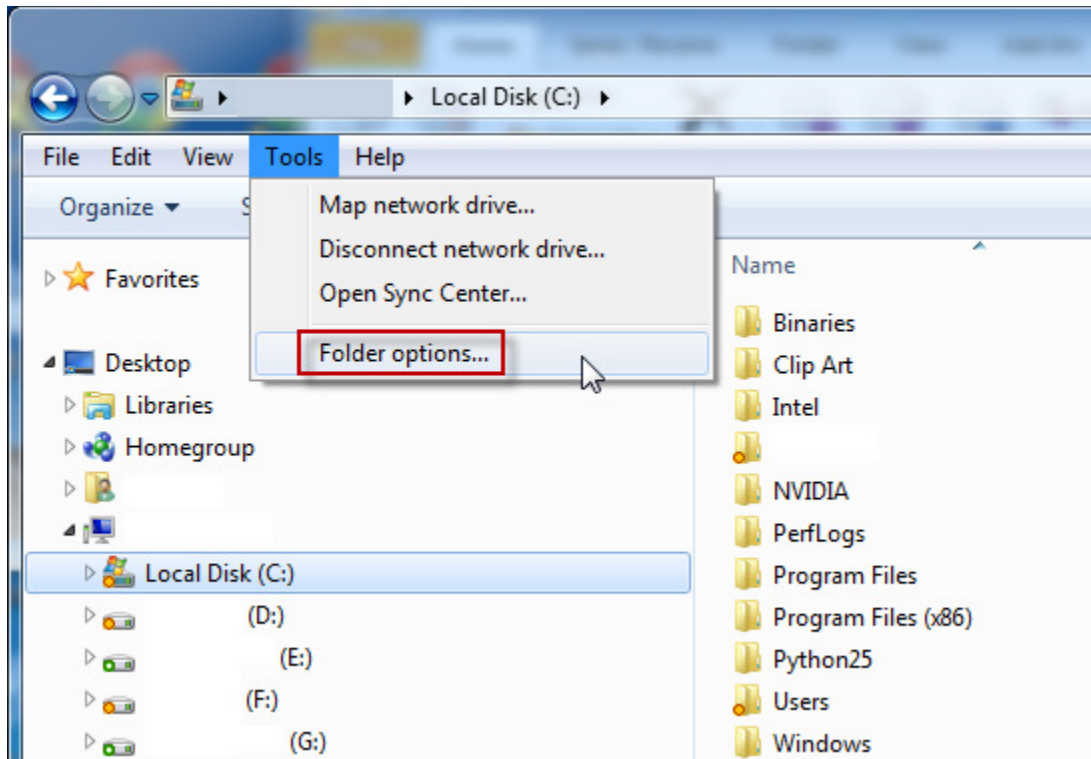


Figure 2 - Windows Explorer Folder Options



In the Folder Options, select the View tab. You may have to click on 'Hidden files and folders' to expand the tree. Click on the 'Show hidden files, folders, and drives' to activate the selection. Then hit 'Apply' and then 'OK' to close the panel.

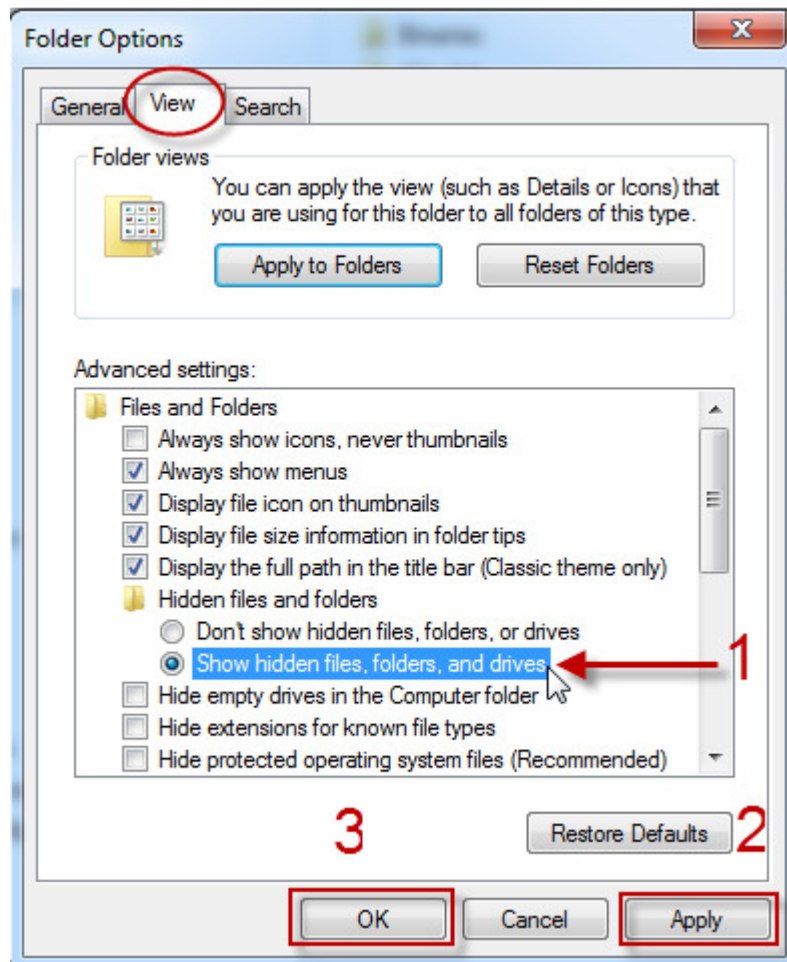


Figure 3 - Folder Options Panel





After doing this, all folders are now visible in Windows Explorer.

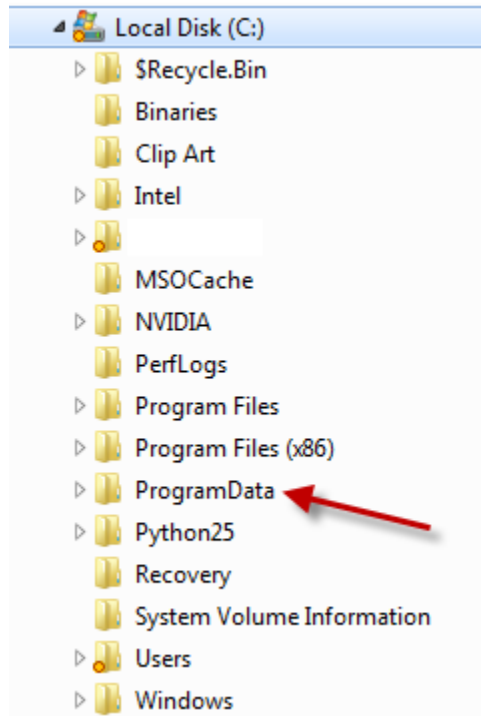


Figure 4 - C:\Program Data Folder Visible

#### *Copied the Post Processors, but Can't 'See Them' in VCarve Pro/Aspire*

Your OS can be pretty sensitive to files that are downloaded from the web. This is particularly true with Window 7.

While you may be able to download the post processor(s) from the Next Wave Automation web site directly to the appropriate /PostP folder, it will definitely result in not being able to 'see' the downloaded post processor file in the post processor file list in the Vectric application as this file is blocked from use by the OS.

Even if you download the files to a known location on your computer, and then copy them from that location to the appropriate /PostP folder, the operating system may block its use by the Vectric application.



If you are in this state, open Windows Explorer, find the post processor in the appropriate folder, right click on the post processor file, and select 'Properties'. From the 'General' tab, click on the 'Unblock' button, then hit 'Apply', and lastly 'OK' to close this panel. Open the Vectric application and you should now 'see' the post processor in the post processor list.

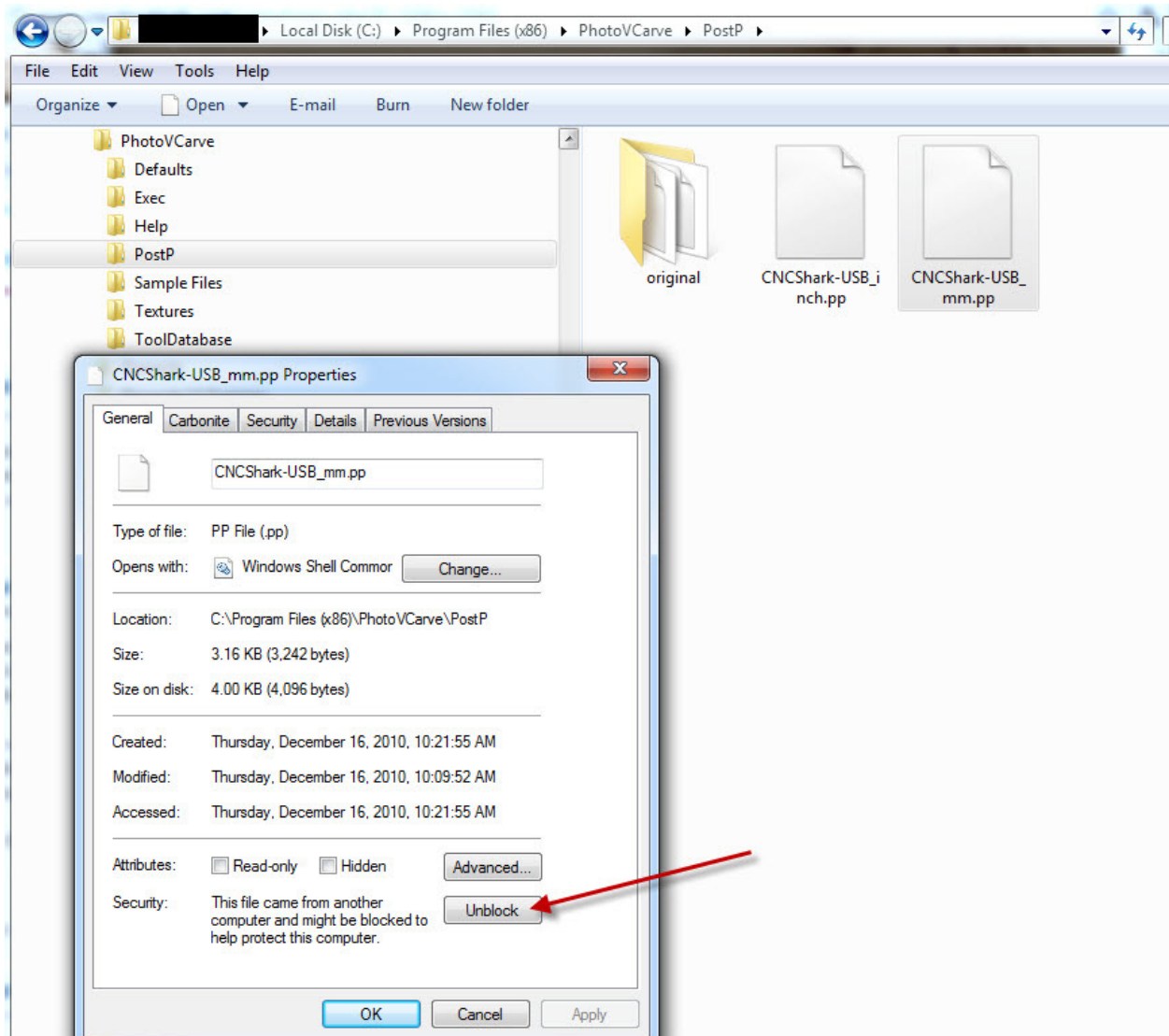


Figure 5 - File Properties, Unblock



## Tailored Post Processors

There are now 2 post processors available for use with no modification, each having an inch and millimeter mode:

- CNCShark-USB\_3dContour\_
- CNCShark-USB\_NewArcs\_

The difference between the 2 is the Set Path Control Mode G64 Px.x (Continuous Mode Best Possible Speed with Motion Blending Tolerance) setting. The 'Contour' inches post processor uses G64 P0.1 (2.54 mm), and the 'NewArcs' inches uses G64 P0.01 (0.254 mm). Previous versions of the CNC Shark Post Processor inches used G64 P0.1.

The G64 Px.x is a way to fine tune the Shark system “for best compromise between speed and accuracy. The Px.x tolerance means that the actual path will be no more than Px.x away from the programmed endpoint. The velocity will be reduced if needed to maintain the path within the tolerance. In continuous mode, sharp corners of the path may be rounded slightly so that the feed rate may be kept up (but by no more than the tolerance).”<sup>1</sup>

You can edit this setting in either post processor file using Notepad or any text editor. A smaller tolerance will result in a more accurate cut to the design. Using a smaller tolerance can also dramatically increase the machining time. A smaller tolerance may be required or desired if you need sharp inside corners for example. For projects involving 3D elements, a larger tolerance will often produce excellent results, as well as resulting in a reasonable machining time.

You also influence the outcome with the feed rate of the tool you are using. You will get a slightly different outcome for a profile toolpath for example with the same end mill when using a feed rate of 100 ipm and then a second time using a feed rate of 60 ipm.

For toolpaths involving pockets, corners and lettering, using the CNCShark-USB\_NewArcs\_ post processor is recommended as a starting point. For toolpaths involving 3D components, or where a faster machining still provides results that meet your requirements, the CNCShark-USB\_3dContour\_ post processor is recommended as a starting point.

---

<sup>1</sup> Common Machining Center, LinuxCNC.org, URL: [linuxcnc.org/docs/html/gcode.html](http://linuxcnc.org/docs/html/gcode.html)





You can always manually edit the G64 Px.x tolerance value to fine tune the Shark system to meet your requirements. Users should keep in mind that this is just one of several factors that influence the overall 'accuracy' of the outcome. Router TIR, tool settings, table setup, etc – all contribute to this.

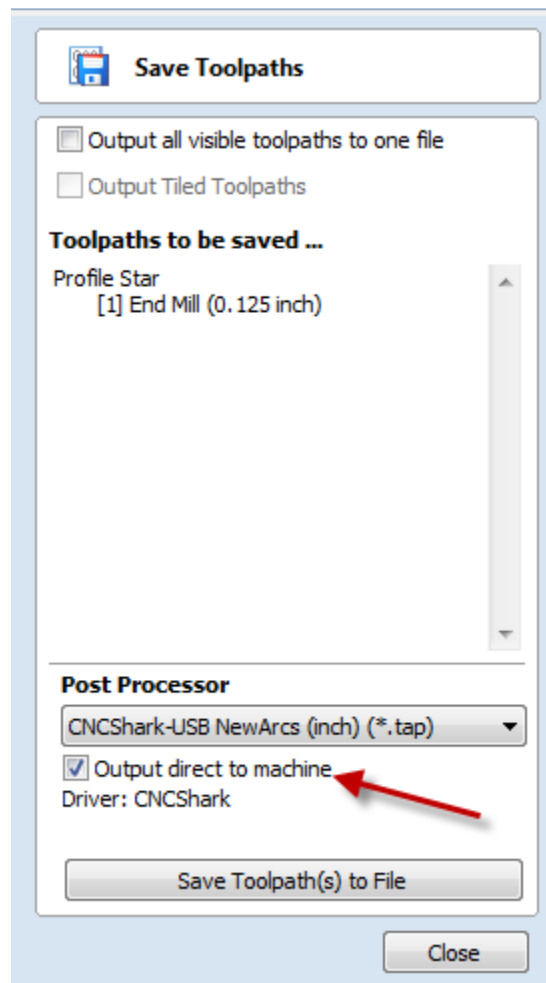
```
"([TOOLS_USED])"  
"([TOOLNAME])"  
"(|-----)"  
"(| Toolpath:- '[TOOLPATH_NAME]' )"  
"(|-----)"  
"G90"  
"G20"  
"[FC]"  
"G64 P.1"  
"S 2000"  
"M3"  
"G0 [ZH]"  
  
+-----+  
+  Commands output for rapid moves  +  
+-----+  
  
begin RAPID_MOVE  
"[FC]"  
"G00[X][Y][Z]"
```

Figure 6 - G64 Px.x Line to Edit in Post Processor



## Enabling Use of Output Direct to Machine in VCarve Pro and Aspire

The CNC Shark Post Processors v1.5 contain the command line necessary to enable use of the 'Output direct to machine' option in VCarve Pro and Aspire.



**Figure 7 - Save Toolpaths, Output direct to machine**


When used with the CNC Shark Control Panel v1.5.0.22 installed on the same computer as VCarve Pro or Aspire, and connected to a Shark Control Box, you can choose to send the tap file directly to the Shark Control Panel from VCarve Pro or Aspire.

One additional file must be placed in the appropriate /PostP folder – CNCShark\_run.ini. This file should be placed in the same folder as the post processors (described above).



The post processor contains the command 'DIRECT\_OUTPUT'. This enables the process to open the CNC Shark Control Panel and load the tap file when that option is checked in the VCarve Pro or Aspire 'Save Toolpaths' panel.

```
POST_NAME = "CNCShark-USB 3D Contour (mm) (*.tap)"  
FILE_EXTENSION = ".tap"  
UNITS = "MM"  
DIRECT_OUTPUT = "CNCShark|CNCShark_run.ini"  
SUBSTITUTE = "{ }"
```

A red arrow points from the right side of the image towards the 'DIRECT\_OUTPUT' line in the code block.

**Figure 8 - DIRECT\_OUTPUT Command Line in the Shark Post Processor**

The 'Output direct to machine' option in the VCarve Pro or Aspire 'Save Toolpaths' panel will only be active when a post processor that contains this command line is selected for use.

**Remember, the CNCShark\_run.ini file must be in the same folder as the CNC Shark post processor you are using for 'Output direct to machine'. You are able to select this option even if this file is not properly installed. In that case, the process will fail and you will receive a message indicating the failure.**

Using this work flow is straight forward and easy. First, ensure that the CNC Shark Control Panel is closed (not running on the computer). From VCarve Pro or Aspire, in the 'Save Toolpaths' panel, select the desired CNC Shark Post Processor v1.5, and enable the 'Output direct to machine' by clicking on the box. When you click on 'Save Toolpath(s) to File' with 'Output direct to machine' selected, you will still be prompted to name and save the tap file to disk. After writing the file to disk, the CNC Shark Control Panel will be started and the tap file will be loaded.

**The CNC Shark Control Panel cannot be open (running) when you start this process. Be sure to check and ensure that the CNC Shark Control Panel is closed.**

You will very quickly see the CNC Shark Control Panel 'splash' screen appear. The CNC Shark Control Panel will display after the tap file has been loaded. This is very different than the behavior when you start the CNC Shark Control Panel directly, and then load the tap file. In that case, the CNC Shark Control Panel appears immediately after the splash screen. The time it takes for the CNC Shark Control Panel to appear when using 'Output direct to machine' will be as long or short as the time it takes to 'Load G Code' directly from the CNC Shark Control Panel. Be patient when using this method with large



tap files. If you see the splash screen, you know that the CNC Shark Control Panel has been successfully opened and the tap file is being loaded. When that is complete, the CNC Shark Control Panel will be visible and ready for use.



**Figure 9 - CNC Shark Control Panel Splash Screen Displayed During Output Direct to Machine Startup**

You must close the CNC Shark Control Panel after a job is finished before using the direct output method to run a different tap file. Once the tap file is loaded, you can run that job as many times as desired – i.e. you do not have to close the CNC Shark Control Panel and resend the same tap file to CNC Shark Control Panel from VCarve Pro or Aspire between runs of the same job. You could also choose to use the 'Load G Code' function from the CNC Shark Control Panel to load a different tap file (without closing and re-opening the CNC Shark Control Panel first).



## Troubleshooting Output Direct to Machine

### *The 'Output Direct to Machine' is 'Grayed Out'*

If the post processor selected in the 'Save Toolpaths' panel in VCarve Pro or Aspire is grayed out, this means that the CNC Shark post processor selected does not support this. Select a CNC Shark post processor that does support the use of this option.

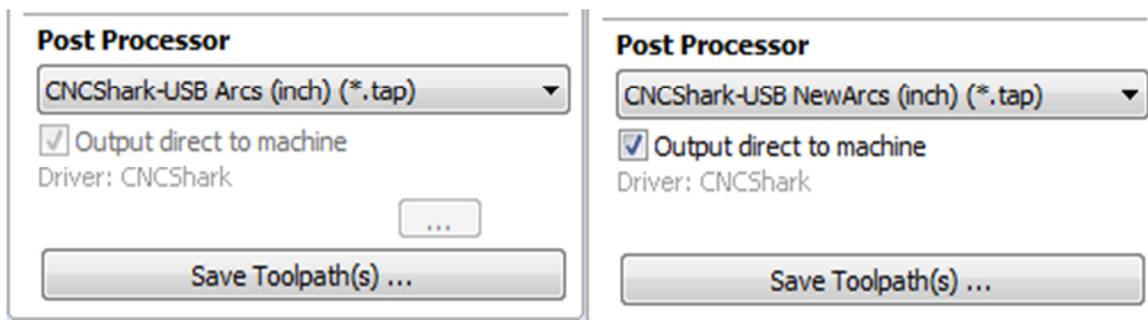


Figure 10 - Output Direct to Machine Not Available (L) and Available (R)

### *VCarve Pro or Aspire Failure Message*

If the CNCShark\_run.ini file is not present in the same folder on disk as the CNC Shark post processor you are using, the direct output process will fail, the CNC Shark Control Panel will not open, and VCarve Pro and Aspire will report this with a message box.

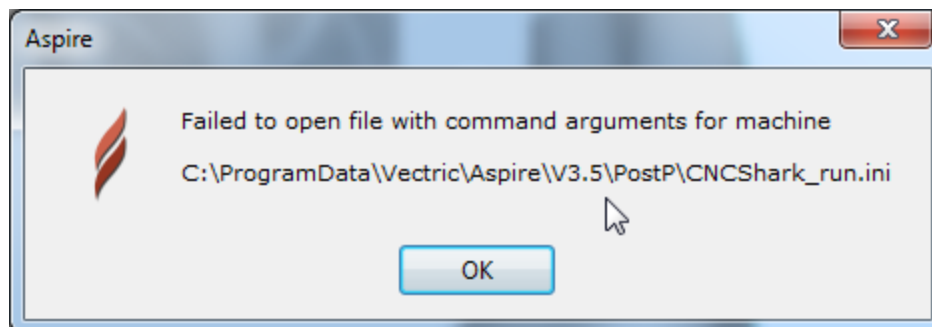


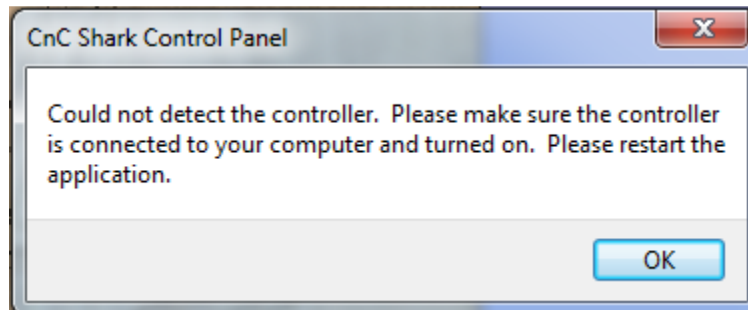
Figure 11 - VCarve Pro/Aspire Failure Message

Place a copy of the CNCShark\_run.ini file in the same folder as the CNC Shark post processors to clear this failure.

### *CNC Shark Control Panel Fail to Detect Message*

If the CNC Shark Control Panel is open (running) when you use the direct output process from VCarve Pro or Aspire, the process will fail. A CNC Shark Control Panel message indicating a failure to detect the controller will appear.





**Figure 12 - CNC Shark Control Panel Could Not Detect the Controller Message**

Close the CNC Shark Control Panel that is open (running), and try the direct output method from VCarve Pro or Aspire again.



## New Tap File Header

The CNC Shark Post Processors v1.5 create and store information about the toolpath generated by VCarve Pro and Aspire in the header of the tap file. This information will help users to know with authority specific aspects of using this tap file – like the tool (cutter/bit) the toolpath is intended to use, the size of the material, the toolpath origin, etc – when viewing the tap file after it is loaded in the CNC Shark Control Panel, or by using a text editor application like Notepad.

The header contains the following information:

1. Tap file name (name assigned to the file when it was written to disk from VCarve Pro or Aspire)
2. Date the tap file was created
3. Material size (X, Y and Z)
4. Z Origin for Material (Material Surface or Table Surface)
5. XY Origin for Material (Bottom Left Corner, Bottom Right Corner, Top Left Corner, or Top Right Corner)
6. XY Origin Position (typically 0, 0 unless the 'Use Origin Offset' in the 'Job Setup' tool in VCarve Pro or Aspire is used)
7. Home Position
8. Safe Z Height/Rapid Clearance Gap
9. Project Notes (from VCarve Pro or Aspire Project (Edit, Notes))
10. Toolpath(s) used (from your VCarve Pro or Aspire project)
11. Tool used by the tapfile

```

1 | ( Star Project Profiles )
2 | ( File created: Wednesday, January 04, 2012 - 10:53 AM)
3 | ( for CNC Shark from Vectric )
4 | { ( Material Size)
5 |   ( X= 10.000, Y= 10.000, Z= 1.000)
6 |   ( Z Origin for Material = Material Surface)
7 |   ( XY Origin for Material = Bottom Left Corner)
8 |   ( XY Origin Position = X:0.000, Y:0.000)
9 | { ( Home Position)
10 |   ( X = X0.0000 Y = Y0.0000 Z = Z0.8000)
11 | { ( Safe Z = 0.200)
12 | { (1. Quick test project to support CNC Shark Post Processor v1.5 documentation.)
13 | { (2. This project is using inches.)
14 | { (3. A third entry to enhance clarity in the screen grab and documentation.)
15 | { (Toolpaths used in this file:)
16 | { (Profile Star)
17 | { (Profile Star Inside)
18 | { (Tool used in this file: )
19 | { (End Mill {0.125 inch} PB)
  
```

**Figure 13 - Header Information in Tap File as Viewed from the CNC Shark Control Panel**

The tap file name is the name you assigned the tap file when you saved it from VCarve Pro or Aspire.

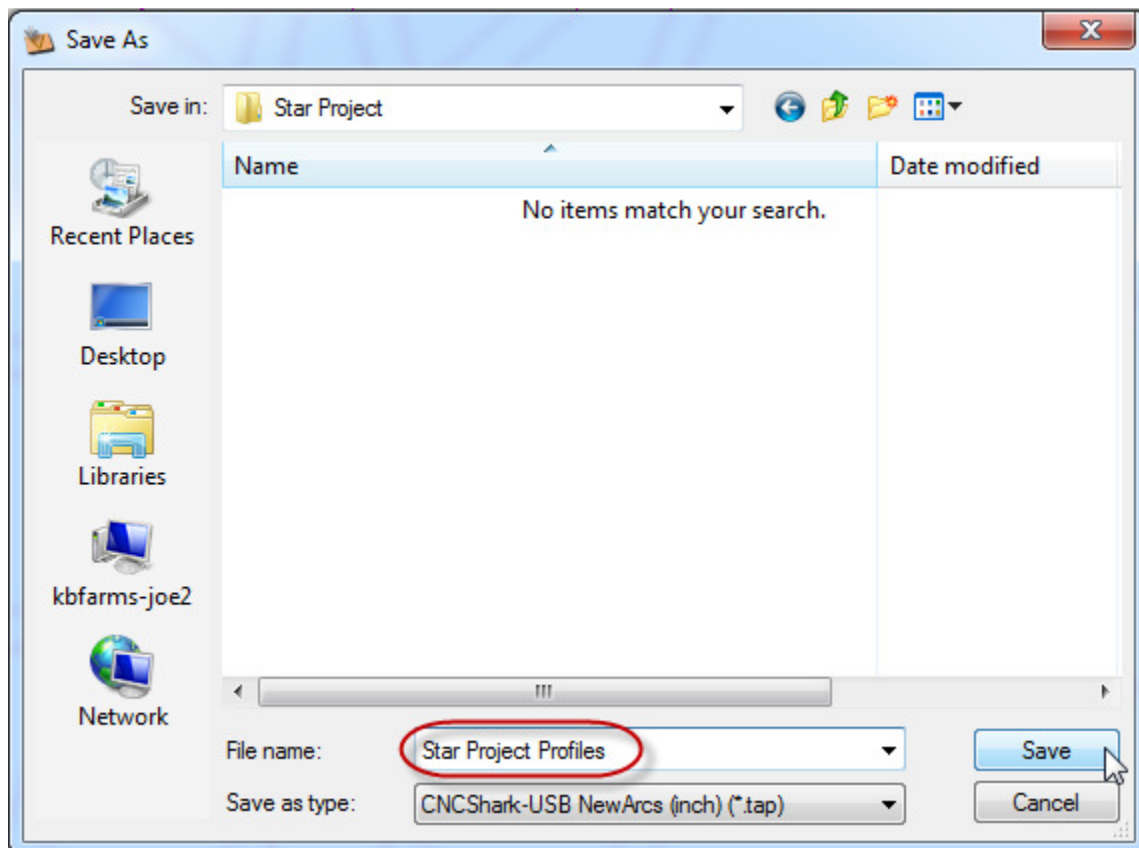


Figure 14 - Tap File Name Creation Dialogue Panel (VCarve Pro or Aspire)



The Material Size, Z Origin for Material, XY Origin for Material, and the XY Origin Position are from the settings used in the 'Job Setup' panel. From VCarve Pro or Aspire, select Edit, Job Size and Position, to open this tool.

 The "Job Setup" panel is a dialog box with a light blue border. At the top, there is a title bar with a small orange cube icon and the text "Job Setup". Below the title bar, the panel is divided into several sections:
 

- Size:** Contains two input fields. "Width (x):" is set to "10.0" inches, and "Height (y):" is set to "10.0" inches.
- Material:** Contains a radio button labeled "Z Zero" which is selected, and a "Thickness (z):" input field set to "1.0" inches. There is a small 3D model of a material block with a red dot indicating the Z-zero position.
- XY Origin Position:** Contains a diagram of a square with a central point and four corner points. The bottom-left corner point is selected with a blue dot.
- Use origin offset:** A checkbox that is currently unchecked. Below it are "X:" and "Y:" input fields, both set to "0.0".
- Data Scaling:** Contains two unchecked checkboxes: "Center data in job" and "Scale data with job".
- Units:** Contains two radio buttons: "inches" (which is selected) and "mm".

 At the bottom of the panel are two buttons: "OK" and "Cancel".

Figure 15 - Job Setup Panel in VCarve Pro and Aspire



The Home Position and Safe Z Height / Rapid Clearance Gap are from the settings used in the 'Material Setup' panel. From VCarve Pro or Aspire, open the Toolpath Tab, and select the Setup the Material and Rapid Gaps tool button to open this tool.

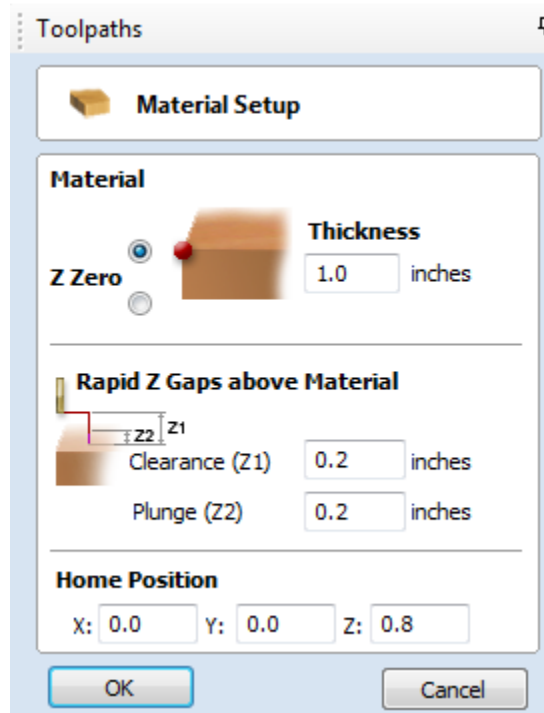


Figure 16 - Material Setup Panel in VCarve Pro and Aspire





The Project Notes are from the 'Notes' tool. To open From VCarve Pro or Aspire, select 'Edit', 'Notes'.

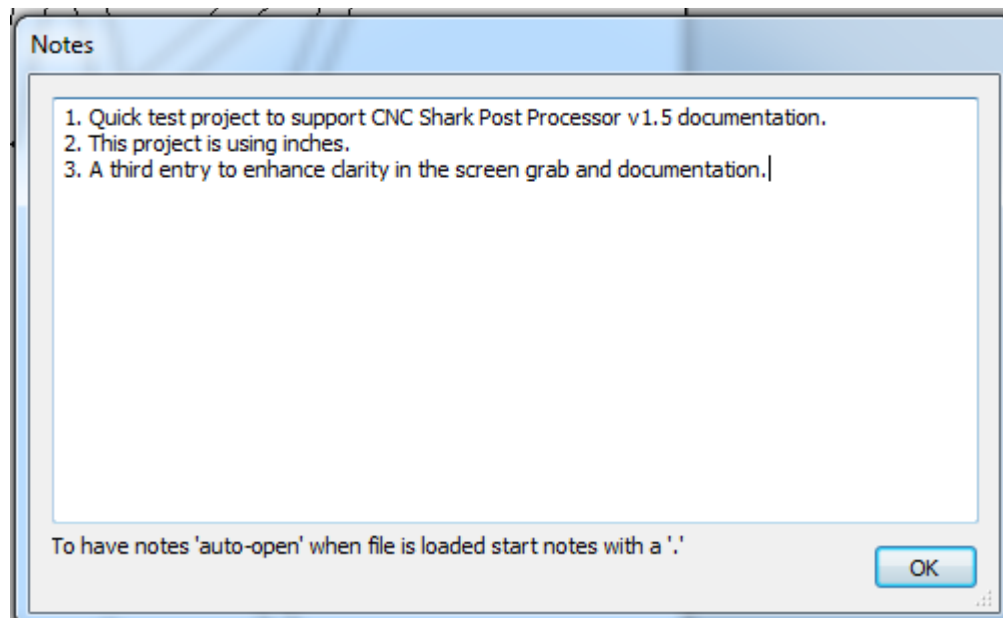


Figure 17 - Notes Tool in VCarve Pro and Aspire



The toolpaths are the VCarve Pro or Aspire project toolpaths that were selected when creating the tap file.

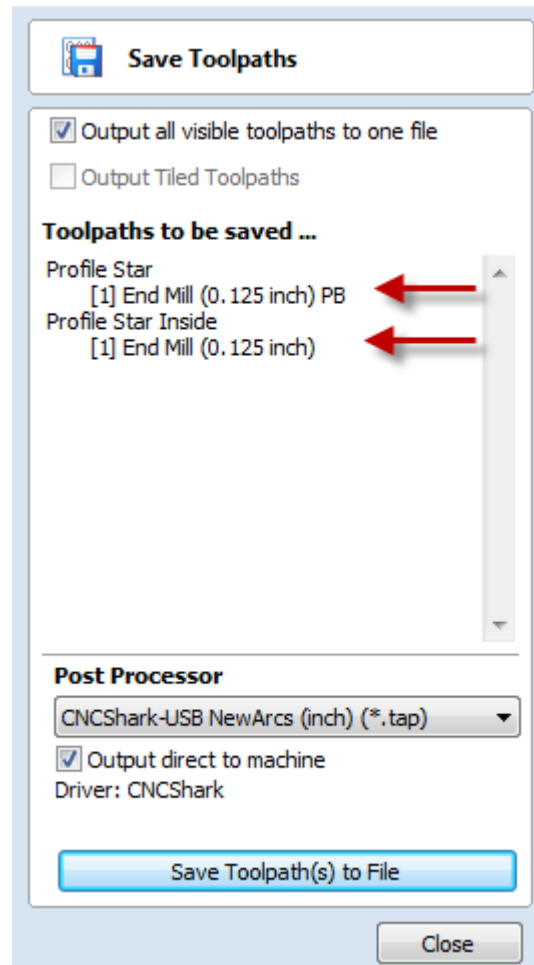


Figure 18 - Toolpaths to be Saved in VCarve Pro or Aspire

The tool of course is from the toolpath setup itself. The name of the tool in the tap file header is the same as that set in the VCarve Pro or Aspire Tool Database.