

USER GUIDE

The purpose of this User Guide is to guide the user on the steps to install my v5a (or greater) customized Post-Processor (PostP) supporting Carbide Motion (CM) gcode sender for use with Vectric design software executing on a Shapeoko CNC machine.

The v5a (or higher) PostP for Vectric software adds toolpath and tool# information to the STATUS line display of Carbide Motion (CM) similar to some display info added when using Carbide Create. It also adds 'SafeZ Rapid movement avoiding workholding' like Carbide Create adds to gcode. It also adds several additional status prompts to aid the user in the process flow, as well as an archived data section appended to the end of the gcode file for future reference (see end of this user guide for example). A typical v5a CM Status line display line might appear as follows:

Toolpath: Profile 1, cutout, Tool# 102

Note: In the example above, the user had entered "Profile 1, cutout" as the toolpath name in the Vectric software.

Normally the Vectric default Carbide Motion ATC v4 or v8 PostP always only displays the PostP Revision ID stamp on the CM Status line continuously such as: AC06FA06858F6820297A912D3D187A70, which does not mean much to the basic end user.

Additionally, this User Guide in Section 2 also describes how to use the "Static Time" or added text display option added in version 6 and above of this PostP to display whatever additional text on the CM Status line that the end user desires to be displayed.

For clarification, versions 5 and 6 are only for use with a Trim Router; version 7 is only for use with a Spindle/VFD. For most basic Trim Router users, version 5 is all you will need.

This PostP for Vectric was developed and tested with Vectric Vcarve Desktop and Pro versions 11 and 12 and 12.5 with a Shapeoko Pro CNC XXL, and Carbide Motion v635 thru v651.

Therefore this User Guide is divided into two sections: **Section 1 Post-Processor Install** and **Section 2 PostP Static Time and Text Display**.

SECTION 1 POST-PROCESSOR INSTALL

You should have received the ZIP file containing the post-processors and this User Guide or you would not be here.

Step 1. Place the extracted files into a dedicated folder on your harddrive of your choice.

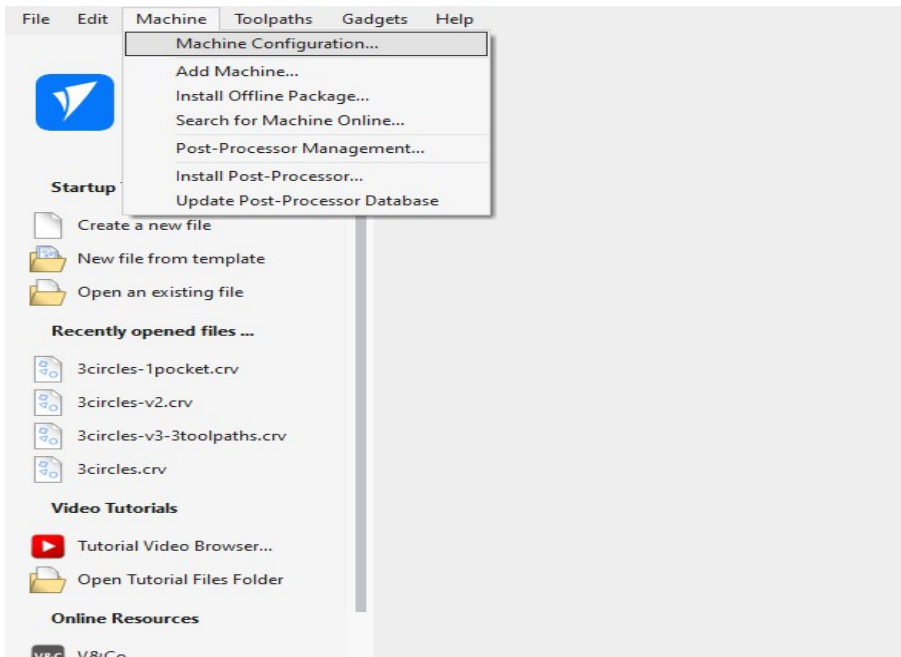
Step 2. Load your appropriate Vectric design software (Vcarve Desktop, Vcarve Pro, Aspire, etc).

Step 3. There are **TWO** options to install the post-processors: please first read the two options below and decide which you prefer to perform easiest.

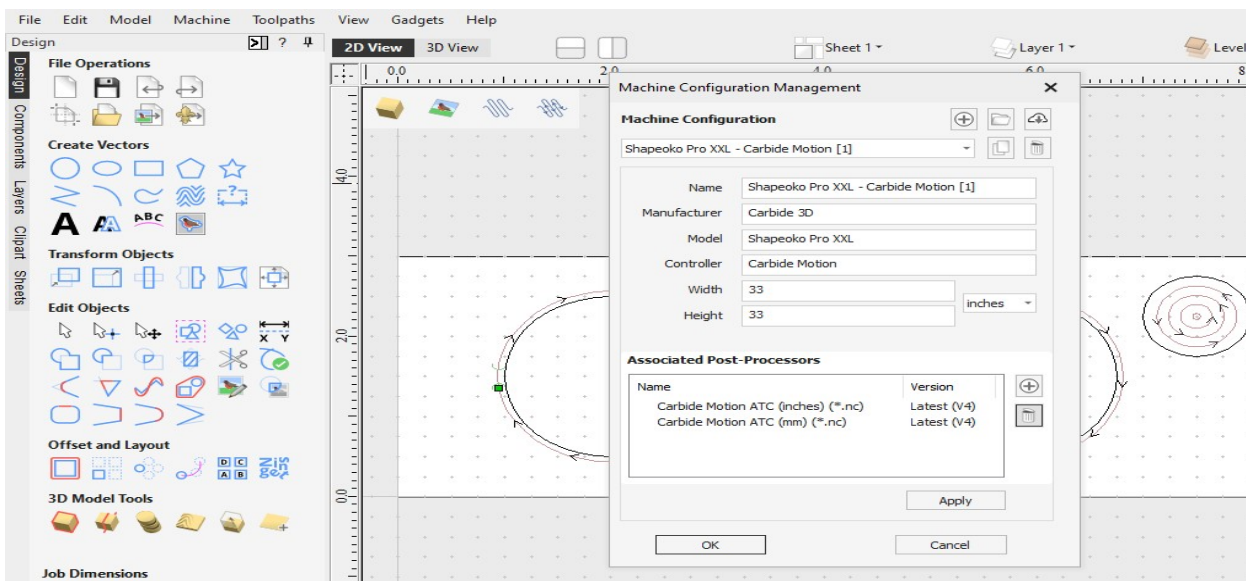
Option 1:

Step 4: Using Windows Explorer or any file management program, move or copy the two post-processor files (the two files ending in .pp) to the Vectric Users Custom Post-Processor folder named My_PostP folder. The folder path is usually as follows:
c:\ProgramData\Vectric\Vcarve Pro\V11.5\My_PostP

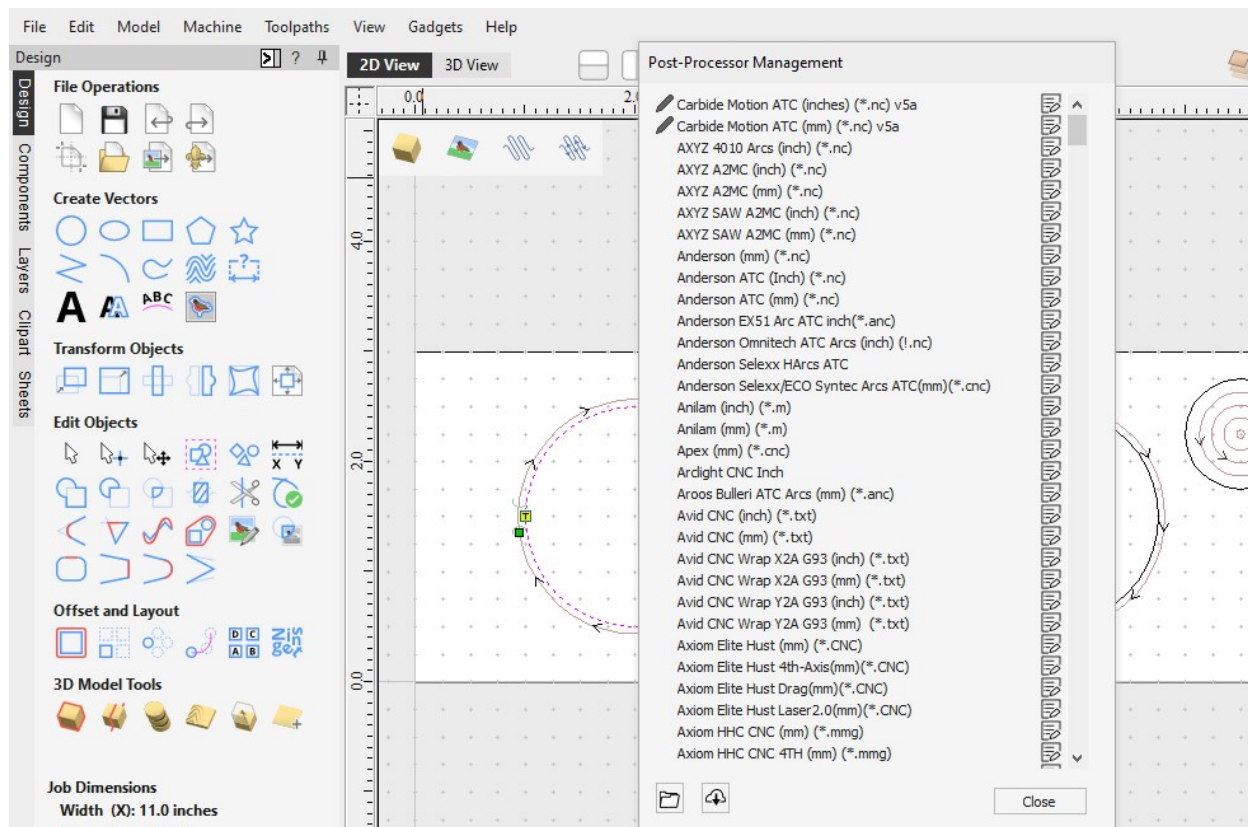
Step 5. In the Vectric software, select Machine, Machine Configuration, as shown below.



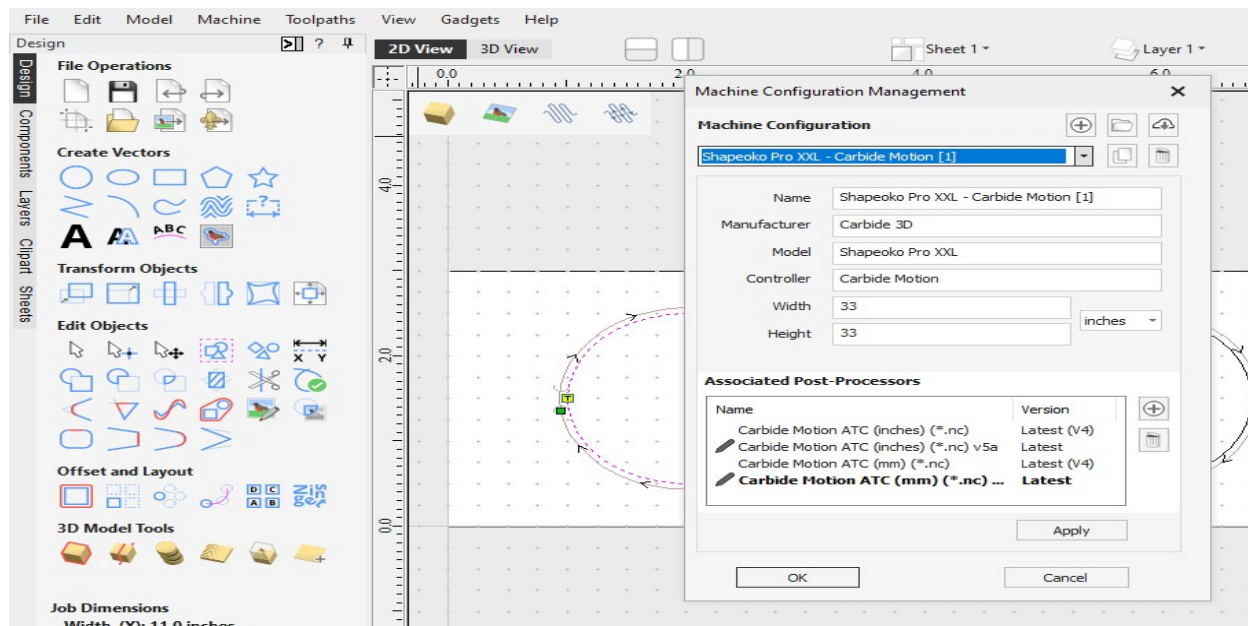
Step 6. The following below dialog box should appear, then select the + symbol to the right of the displayed associated post-processors.



Step 7. The below dialog box will appear showing the two new post-processors at the top of the listing with pencil icons beside them. Select one or both, a SELECT button will appear, press the button.

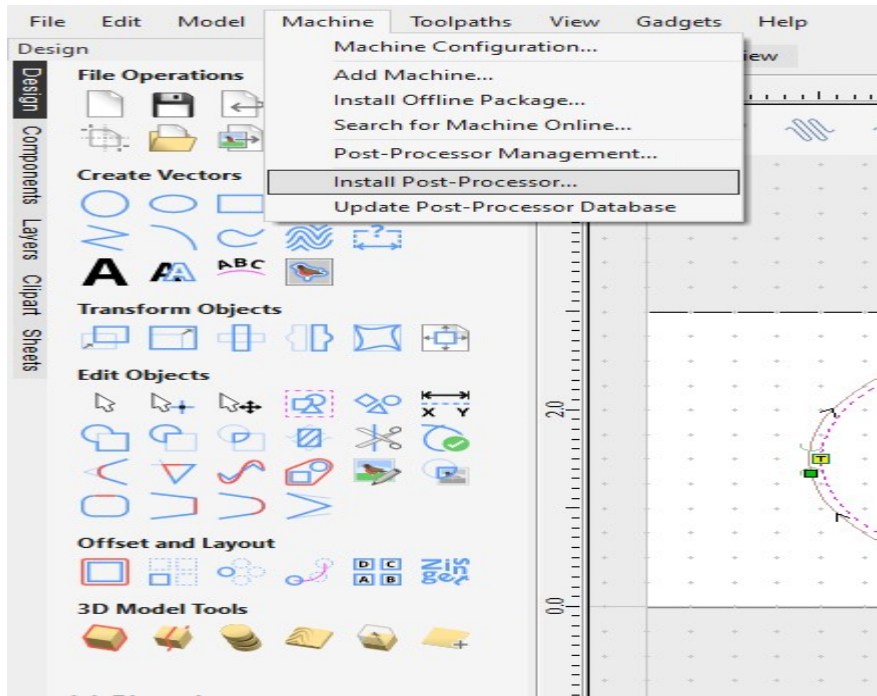


Step 8. The below dialog box will appear, double click the new (v5a) post-processor to be the default and click the APPLY button. Then click the OK button. The dialog will close, your new PostP is installed. Proceed to Step 9 (following Option 2 description).

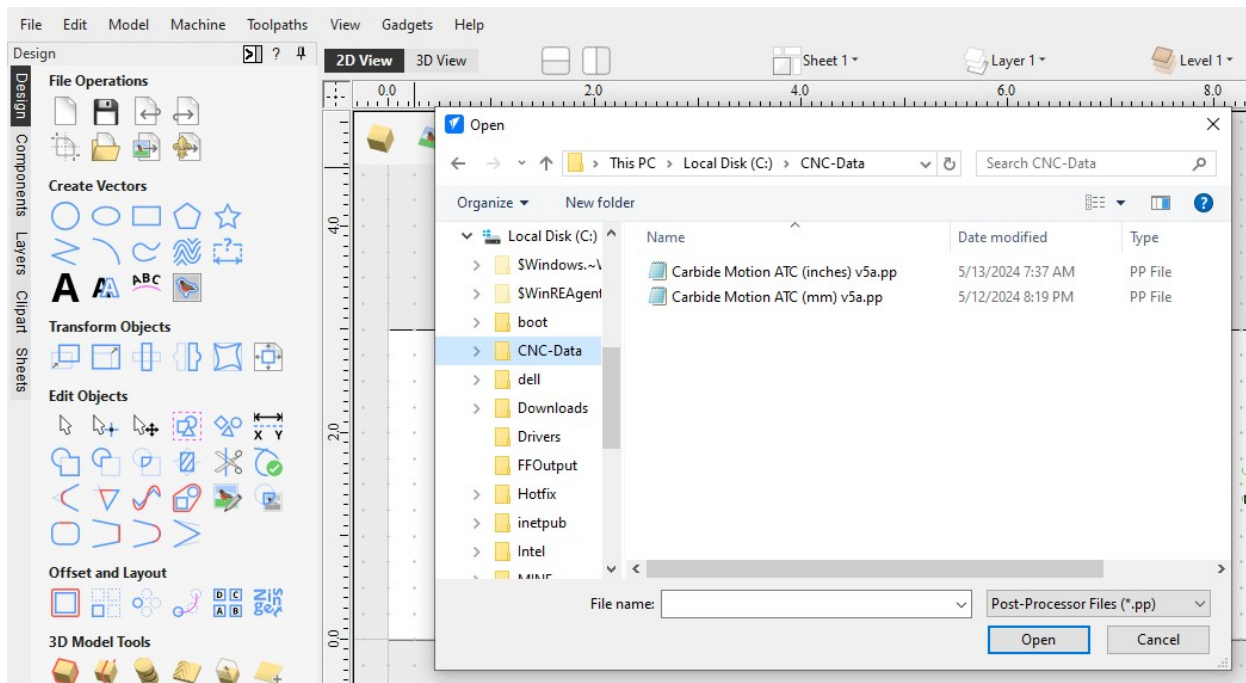


Option 2:

Step 4. In the Vectric software, select Machine, Install Post-Processor, as shown in below figure.

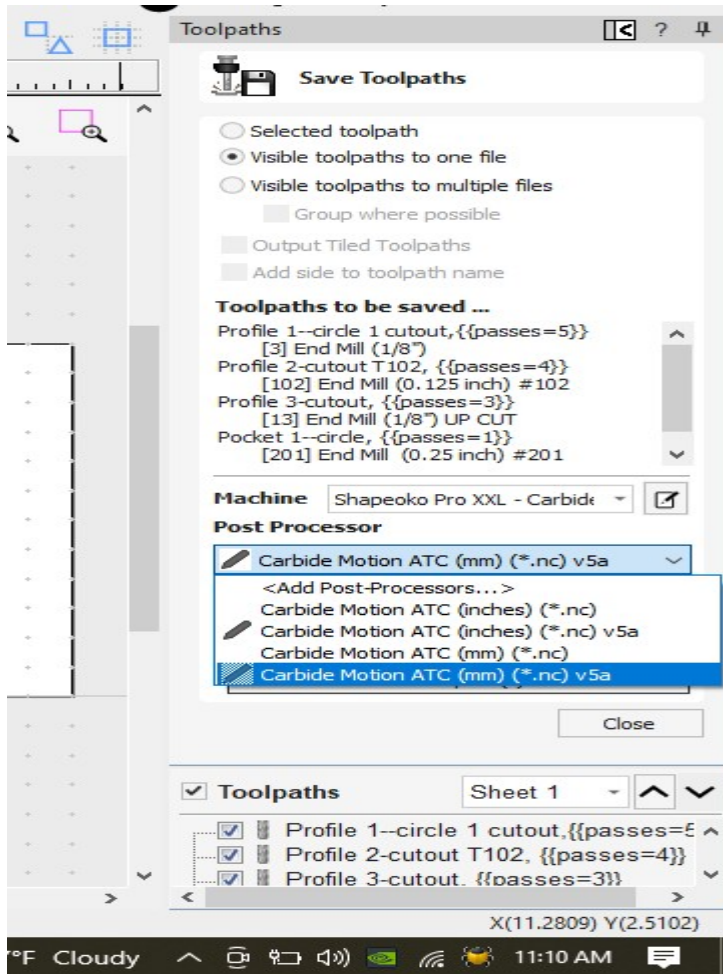


Step 5. The below dialog box will appear, navigate to the folder where you saved the new .pp files and select them, one at a time, selecting OPEN button. Vectric will move a copy to the My_PostP folder automatically for you.



Step 6. Repeat Steps 5 to 8 of Option 1 above. Then proceed to Step 9.

Step 9. Your new post-processors are installed and ready to be selected when you generate a toolpath. See below example. (If not shown, select 'add processors...' from pulldown arrow and select it).



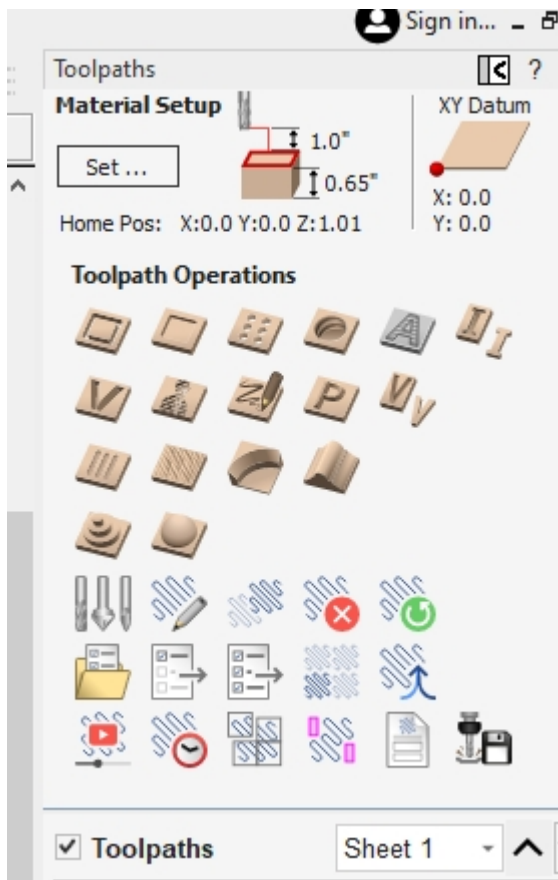
Step 10. This completes the post-processor install SECTION 1.

SECTION 2 PostP Static Time and Text Display

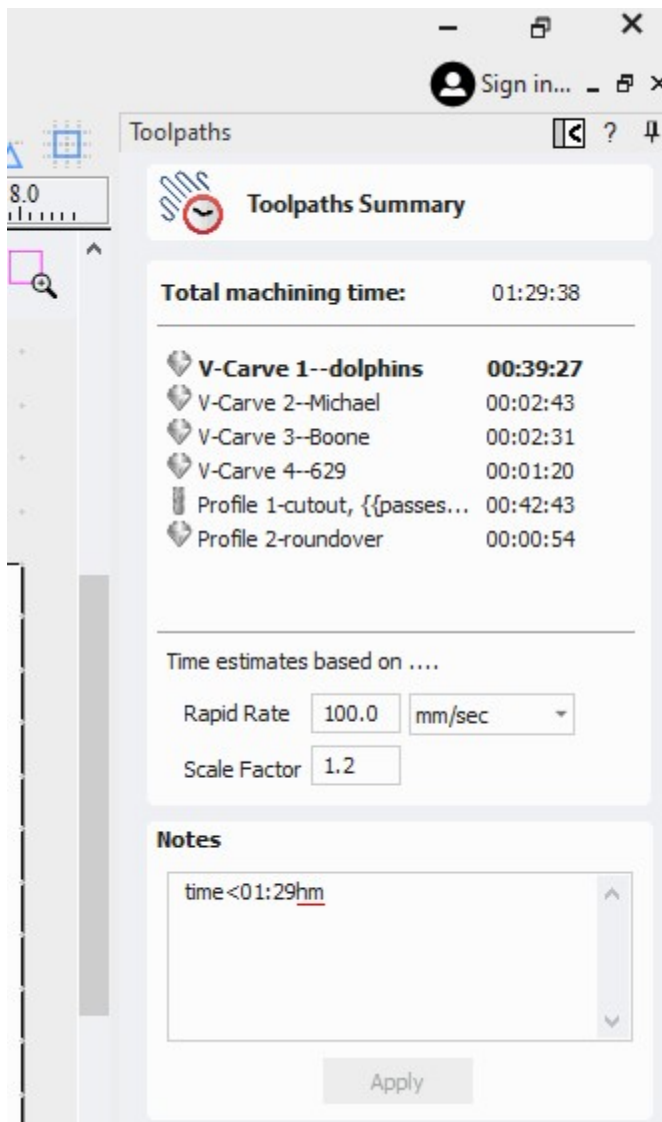
CONDITIONS OF OPERATION:

1. Must have version 6 or greater of my post-processor described above already installed.

To add the static time display, version 6 and above of the PostP makes use of the Vectric software **Toolpath Notes** function which is accessed under the **Toolpath Summary** function, or sometimes referred to as the Time Summary icon. In the Vectric v12 software screenshot below, it is the bottom row, second from left icon.



When selected, the screen on the following page is displayed.



Here a summary of all toolpaths is displayed along with their times. The user is able to enter in the **Notes** box whatever text they desire. But version 6 and above of my PostP allows a static time (or whatever text) to be entered for display on the CM STATUS line display. In the example here, I entered the total time of 1 hour 29 minutes for display. You see the format I chose of ("space" time < 01:29hm) to indicate the total cut time is less than 1 hour 29 minutes. Or I could have entered " time<39m" for just the V-Carve 1 toolpath since it is the selected toolpath. Or the user could go to each toolpath individually and enter a 'countdown' total for the time entry on each toolpath Notes box. Hope that makes sense. No entry adds nothing to display line, except an additional comma. But whatever text you type will be displayed, just be reasonable in length/content (could enter other things like spindle speed, stepover, cut depth, your name, etc, whatever text you would like to display).

A typical CM Status Line display of the example above would look like:

Toolpath: V-Carve 1—dolphins, Tool# 72, time<01:29hm

Or,

Toolpath: Profile 1-cutout, Tool# 102, time<43m, Just some test text.

Or, both times (toolpath and total),

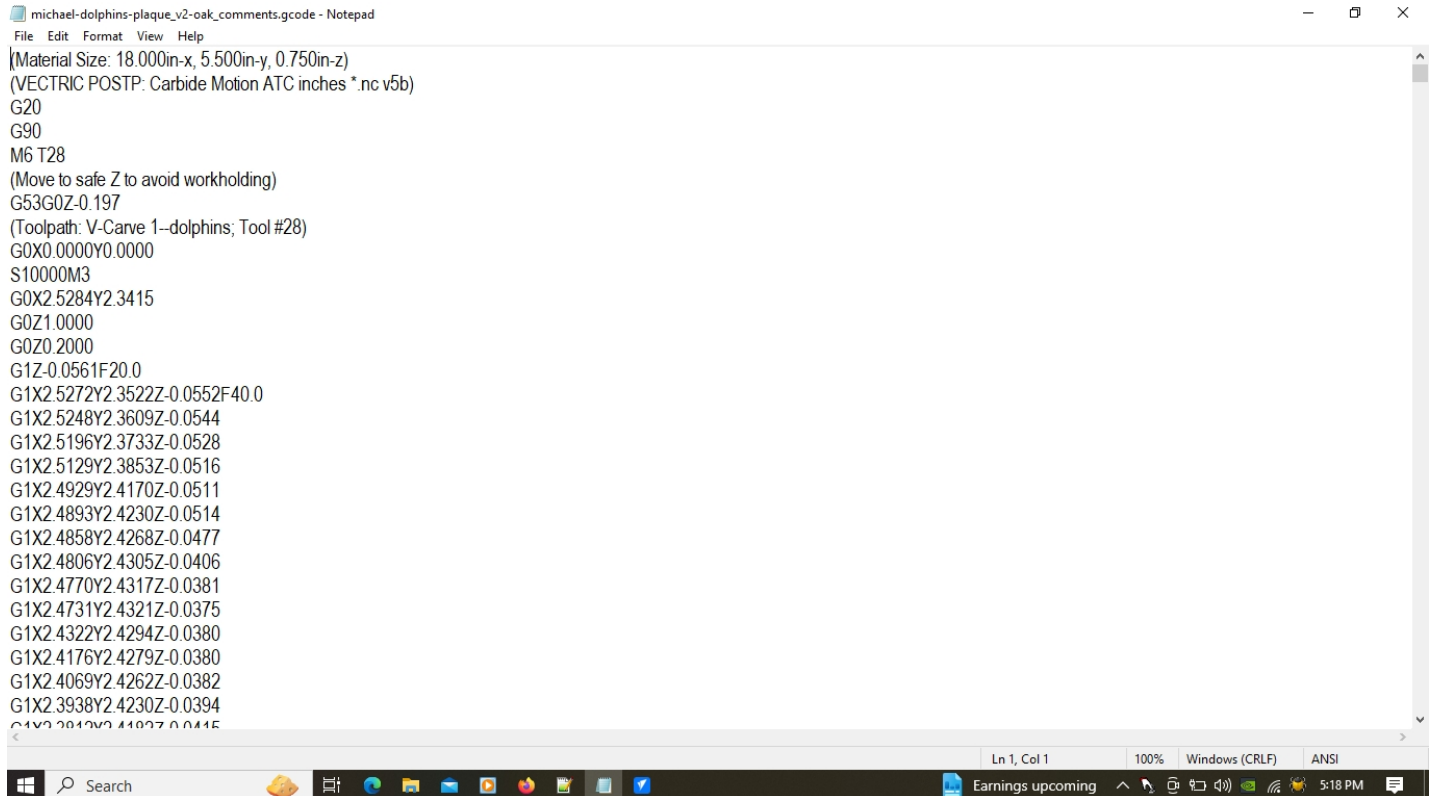
Toolpath: Profile 1-cutout, Tool# 102, Time<43m, EndT<44m

But again, the static time entry is a static time value that you enter, it is not a real time counting running time, but it gives a ballpark idea of time based on the Vectric software calculation, without having to print a Job Sheet for reference. **NOTE: Vectric v12.008 and earlier currently has a bug preventing Toolpath Notes printing on Job Summary Sheet properly formatted.**

NOTE: The above dialog box also contains the Vectric "Scale Factor" setting. It defaults to 2.0, however some experts recommend changing it to 1.0 for more accurate time estimates. The more complex the design, the higher the scale factor number should be to increase the time estimate. I generally keep mine set at 1.2.

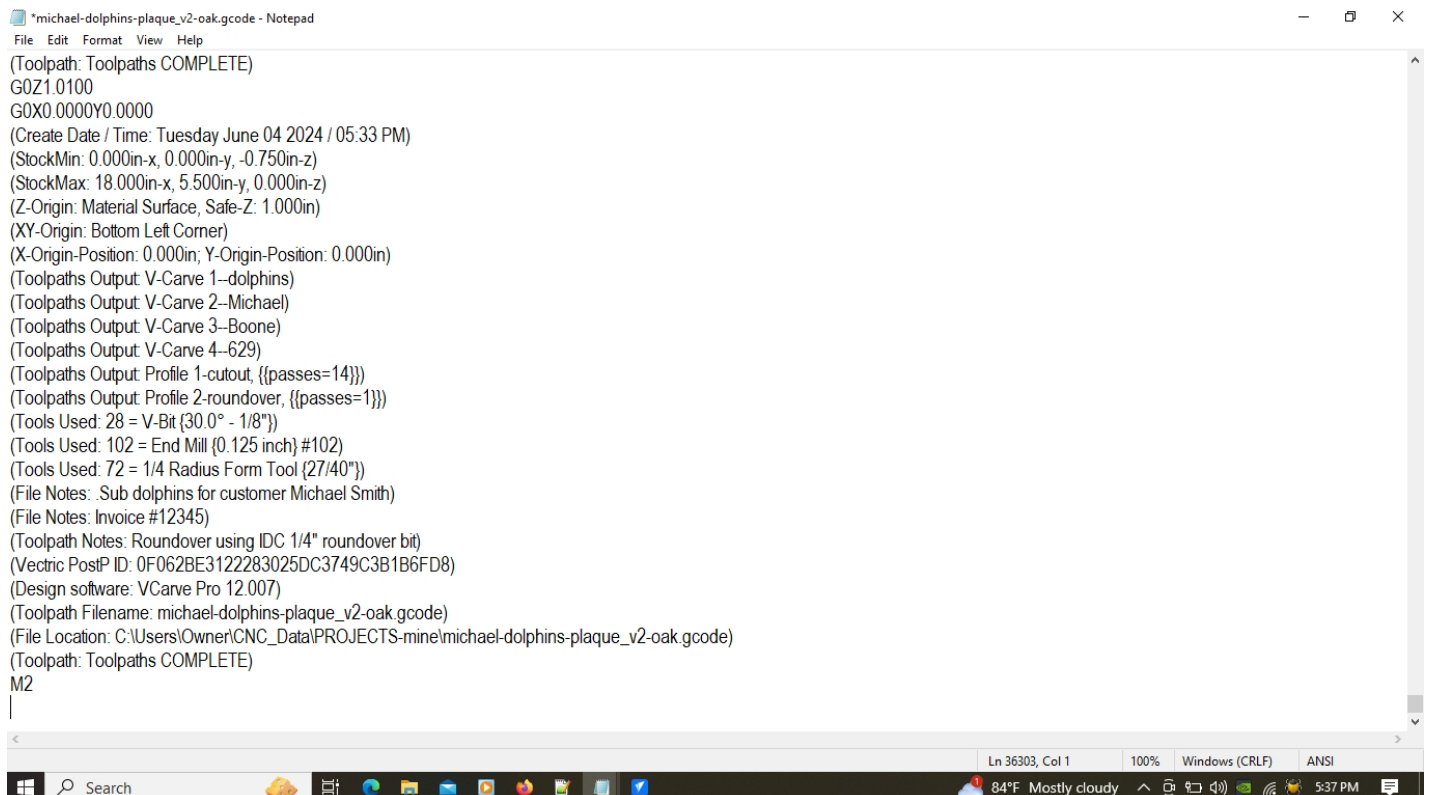
Typical v5b and above PostP Gcode Archived Info File Example

Beginning of gcode file screenshot:



```
michael-dolphins-plaque_v2-oak_comments.gcode - Notepad
File Edit Format View Help
(Material Size: 18.000in-x, 5.500in-y, 0.750in-z)
(VECTRIC POSTP: Carbide Motion ATC inches *.nc v5b)
G20
G90
M6 T28
(Move to safe Z to avoid workholding)
G53G0Z-0.197
(Toolpath: V-Carve 1--dolphins; Tool #28)
G0X0.0000Y0.0000
S10000M3
G0X2.5284Y2.3415
G0Z1.0000
G0Z0.2000
G1Z-0.0561F20.0
G1X2.5272Y2.3522Z-0.0552F40.0
G1X2.5248Y2.3609Z-0.0544
G1X2.5196Y2.3733Z-0.0528
G1X2.5129Y2.3853Z-0.0516
G1X2.4929Y2.4170Z-0.0511
G1X2.4893Y2.4230Z-0.0514
G1X2.4858Y2.4268Z-0.0477
G1X2.4806Y2.4305Z-0.0406
G1X2.4770Y2.4317Z-0.0381
G1X2.4731Y2.4321Z-0.0375
G1X2.4322Y2.4294Z-0.0380
G1X2.4176Y2.4279Z-0.0380
G1X2.4069Y2.4262Z-0.0382
G1X2.3938Y2.4230Z-0.0394
G1X2.3819Y2.4197Z-0.0415
```

End of gcode file screenshot:



```
*michael-dolphins-plaque_v2-oak.gcode - Notepad
File Edit Format View Help
(Toolpath: Toolpaths COMPLETE)
G0Z1.0100
G0X0.0000Y0.0000
(Create Date / Time: Tuesday June 04 2024 / 05:33 PM)
(StockMin: 0.000in-x, 0.000in-y, -0.750in-z)
(StockMax: 18.000in-x, 5.500in-y, 0.000in-z)
(Z-Origin: Material Surface, Safe-Z: 1.000in)
(XY-Origin: Bottom Left Corner)
(X-Origin-Position: 0.000in; Y-Origin-Position: 0.000in)
(Toolpaths Output: V-Carve 1--dolphins)
(Toolpaths Output: V-Carve 2--Michael)
(Toolpaths Output: V-Carve 3--Boone)
(Toolpaths Output: V-Carve 4--629)
(Toolpaths Output: Profile 1-cutout, {{passes=14}})
(Toolpaths Output: Profile 2-roundover, {{passes=1}})
(Tools Used: 28 = V-Bit {30.0° - 1/8"})
(Tools Used: 102 = End Mill {0.125 inch} #102)
(Tools Used: 72 = 1/4 Radius Form Tool {27/40"})
(File Notes: .Sub dolphins for customer Michael Smith)
(File Notes: Invoice #12345)
(Toolpath Notes: Roundover using IDC 1/4" roundover bit)
(Vectric PostP ID: 0F062BE3122283025DC3749C3B1B6FD8)
(Design software: VCarve Pro 12.007)
(Toolpath Filename: michael-dolphins-plaque_v2-oak.gcode)
(File Location: C:\Users\Owner\CNC_Data\PROJECTS-mine\michael-dolphins-plaque_v2-oak.gcode)
(Toolpath: Toolpaths COMPLETE)
M2
|
```