

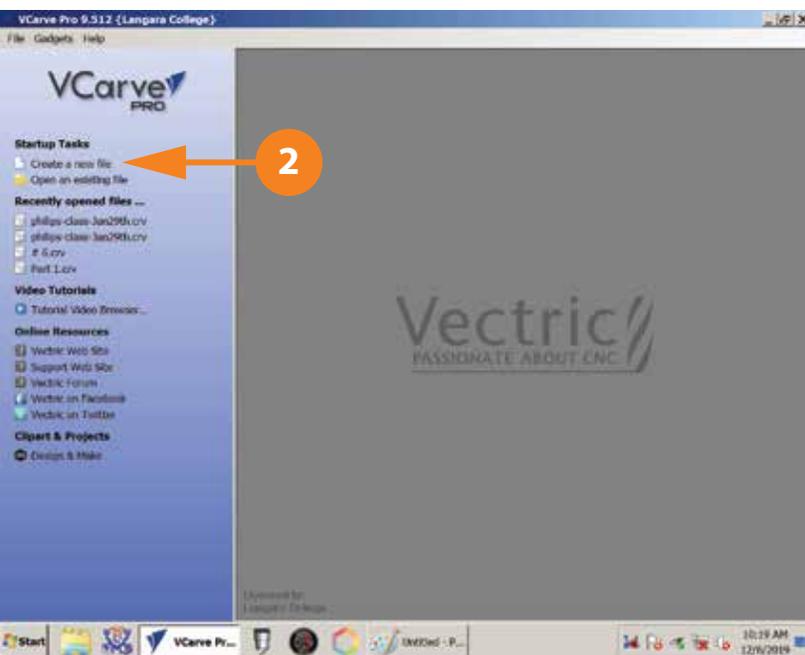


1

Open VCarve software

Make sure you have your rhino file saved to the correct format (DXF) and available on a USB stick

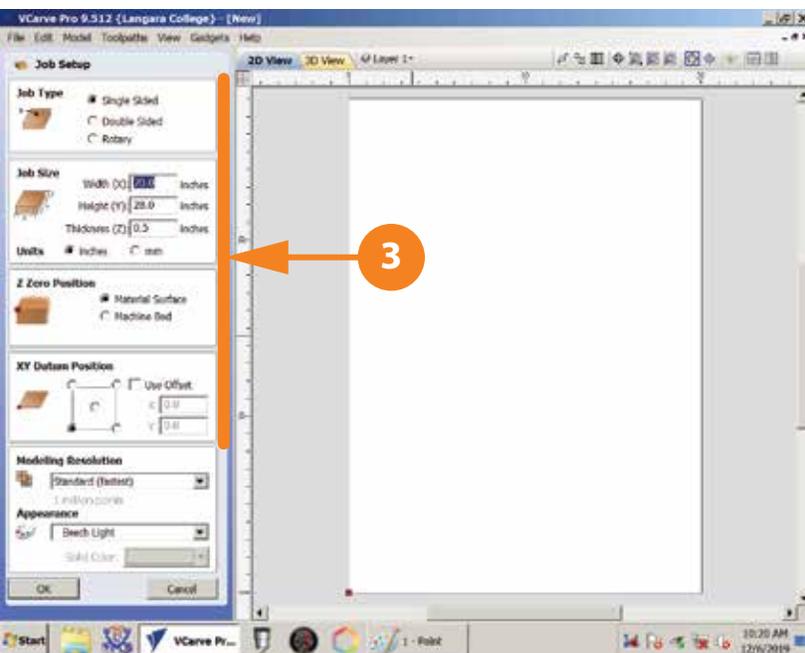
Note: your original Rhino file should be in inches



2

Create a new file

(for each individual project)

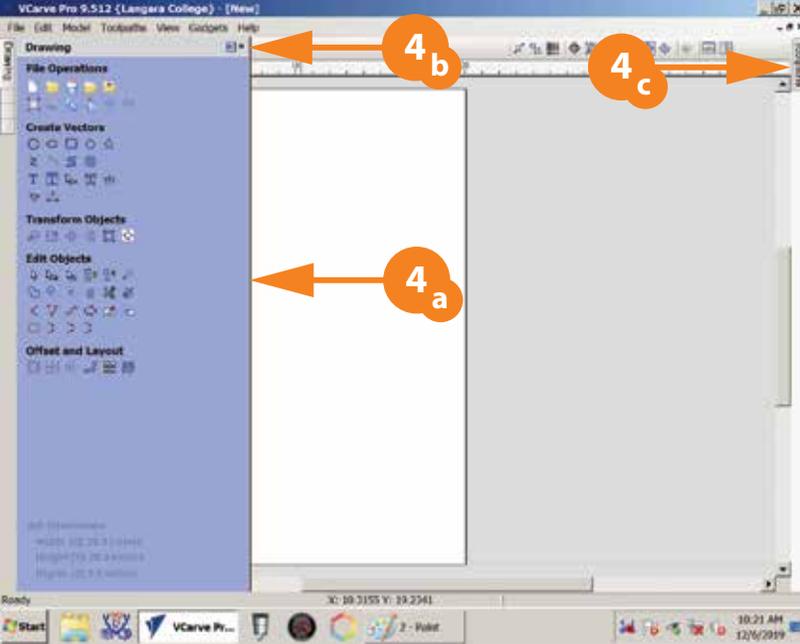


3

VERY IMPORTANT !

Specify the *Job type, Job size, Zero position and XY datum* as shown. This tells the CNC router if you are only cutting from one side (job type), how big your material is (job size), if you will start cutting from the top of the material (z zero position), and where the routers home position is (xy datum position). When done, Press OK

Note: 20" x 28" is the maximum size



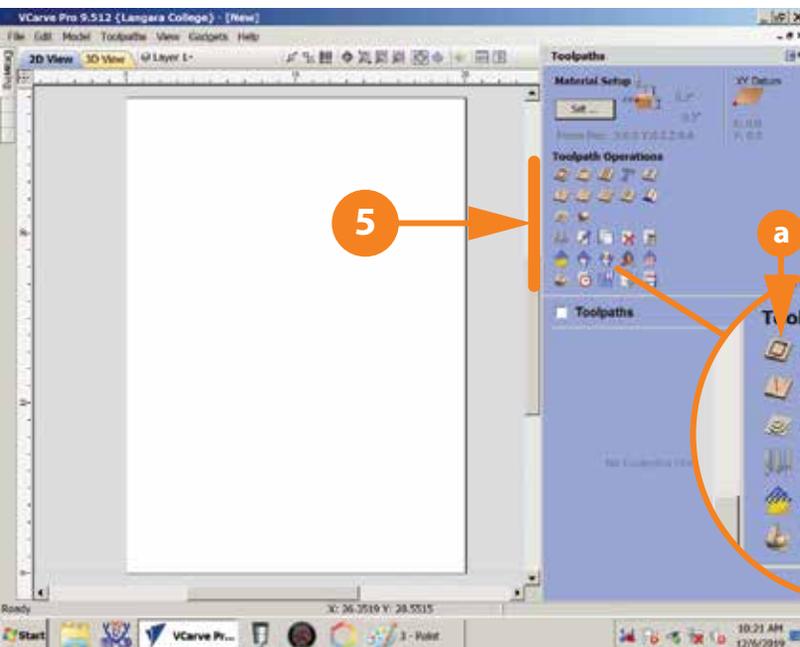
4

A file will open at the Job Size you specified.

4a - The drawing tool palette will open

4b - Close this tool palette here

4c - Open the toolpaths tool palette

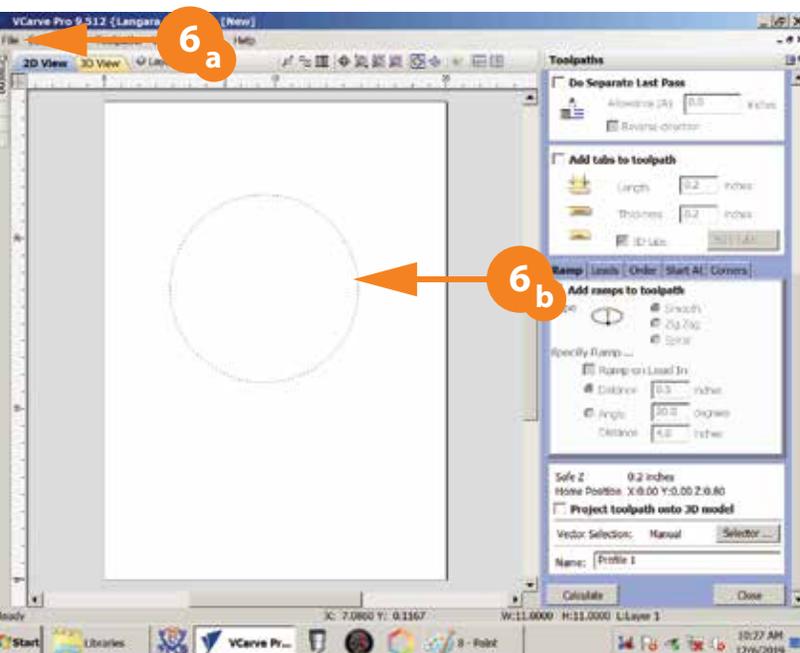


5

Toolpath Operations - note: we will only be creating profiles and pockets

5a - This button opens the *profiles* menu. Profiles cut completely through material.

5b - This button opens the *pocket* menu. Pockets only cut part way through a material.

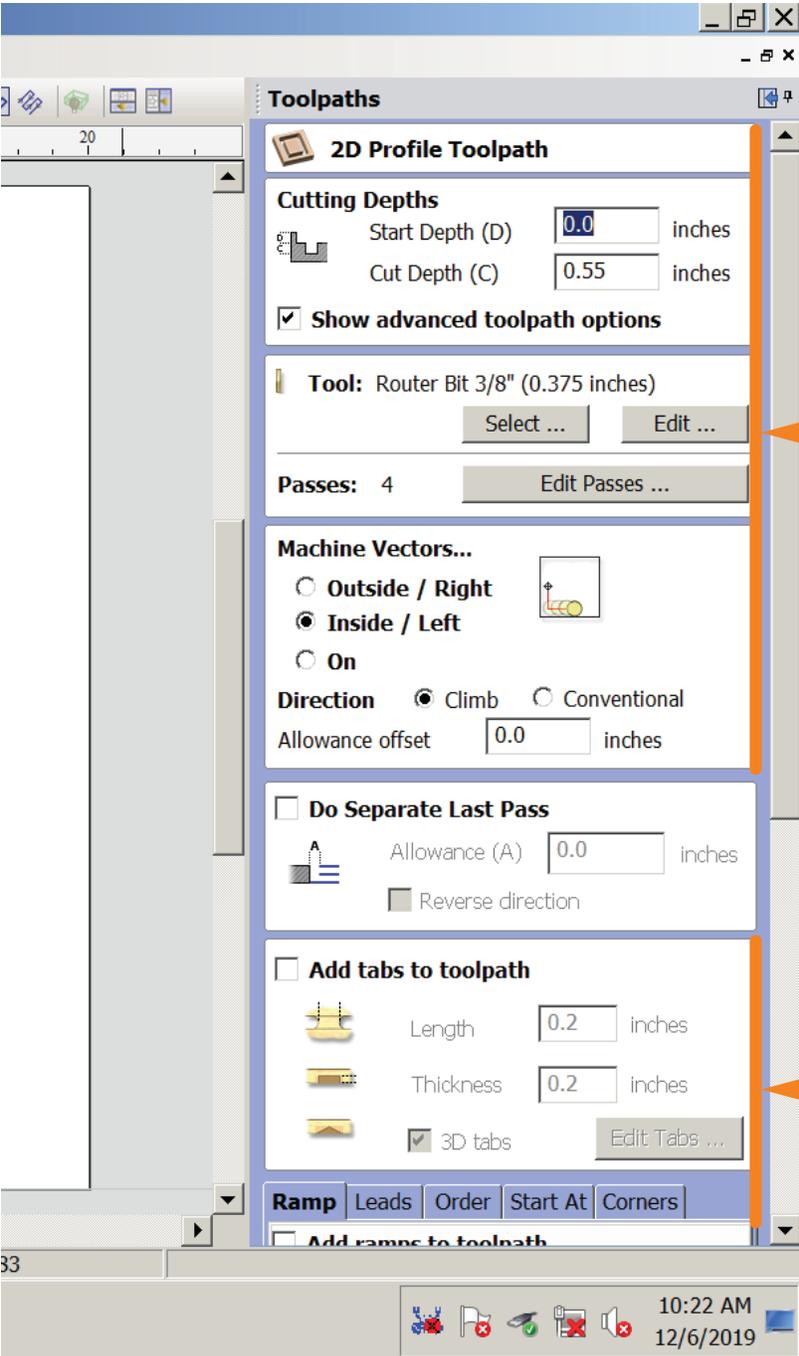


6

Toolpath Operations - profile

6a - Click on *File* and then *import vectors* navigate to your saved rhino file.

6b - Place your file within the work area and select the lines that you want to create **the same** profile cuts for (you can choose more than one set of lines at a time. Selected lines will have the same settings)



7

Toolpath operations (cont.)

7a - Cutting Depth

Start depth - 0 (top of material)
 Cut depth - measure material and add .05 to cut all the way through.

Tool - SEE 8A BELOW
 Machine Vectors - decide if you need to cut inside outside or on the line.

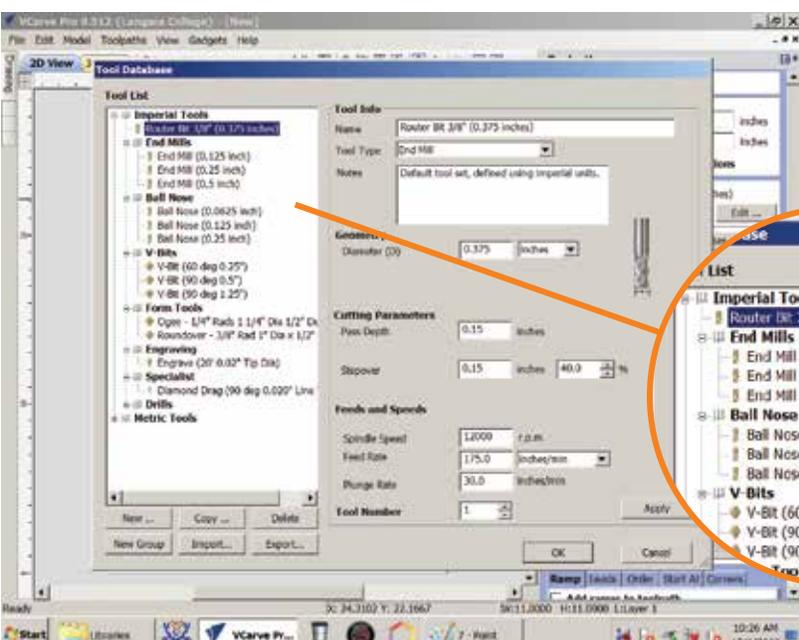
7b- Add Tabs to Toolpath

Tabs help hold pieces in place. Activate this option and place a click on *Edit Tabs* to place them on your selected lines (minimum of 3 tabs per piece).

7a

7b

33



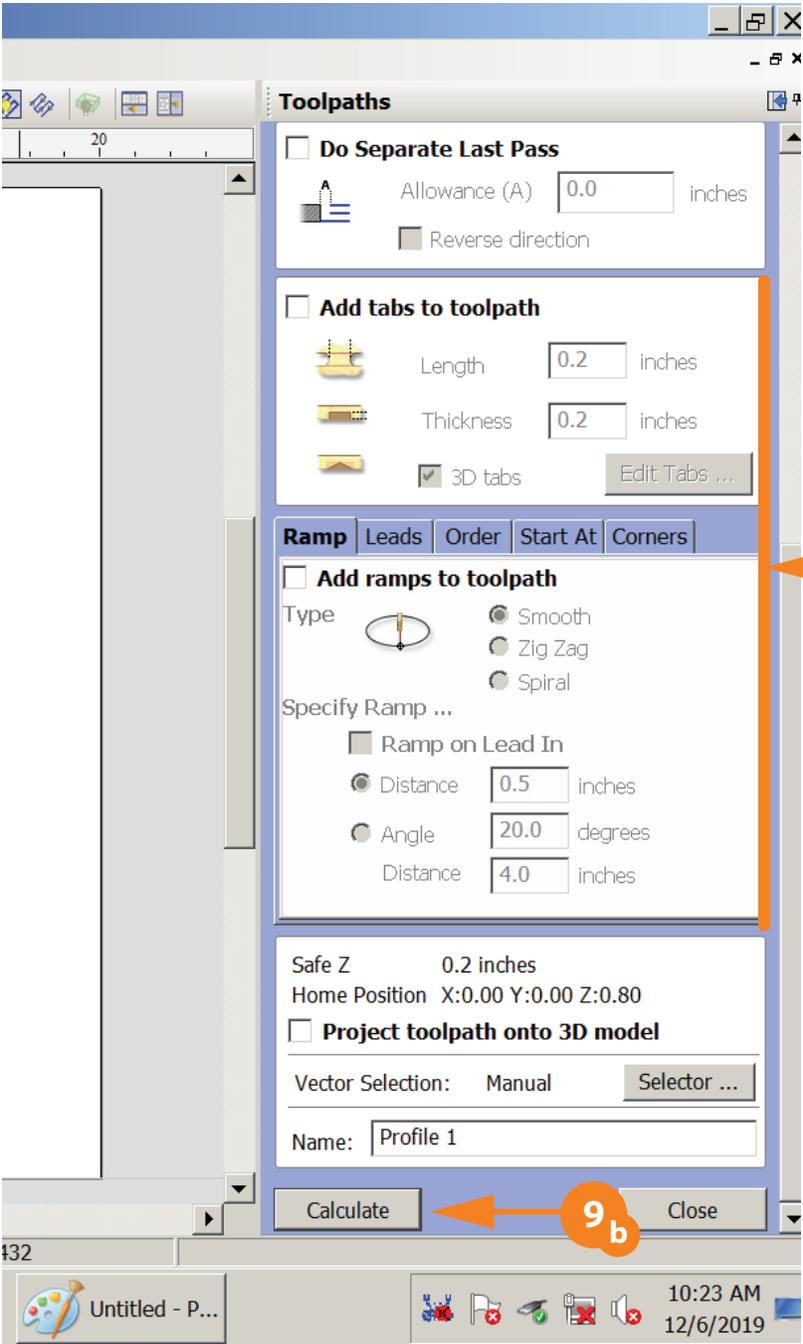
8

Toolpath operations (cont.)

8 - Tool (Within 7a, above)

Press the *select* button above and select the correct router bit we will be using. This file has the correct feed rate and speed build into the profile.

8



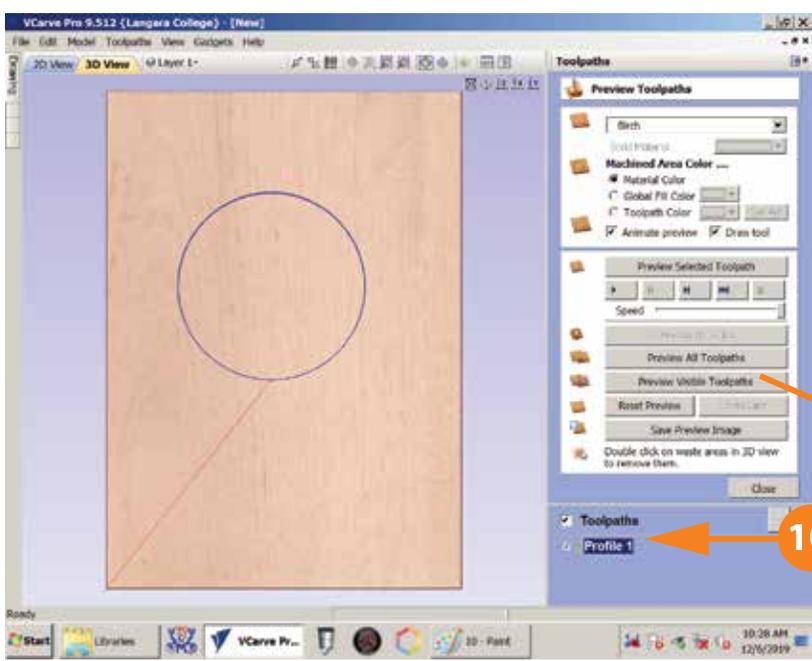
9

Toolpath Operations - (cont.)

9a - Select *Add Ramps to Toolpath* and select the *smooth* option. This makes the router start cutting on a slow downward ramp rather than a straight plunge.

9b - Click on the *Calculate* button and this will switch to a preview under the *3D tab* (top left of screen).

Note: you should get a warning indicating that your depth of cut is greater than the thickness of your material, this is fine as we want to cut slightly through the material.

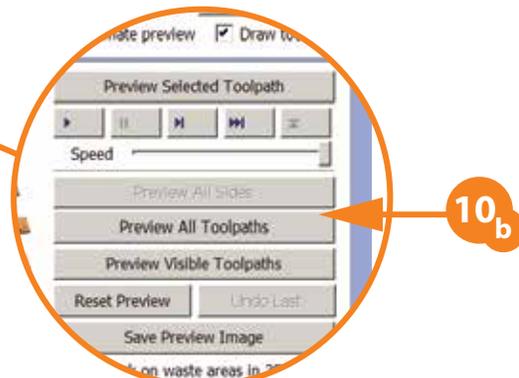


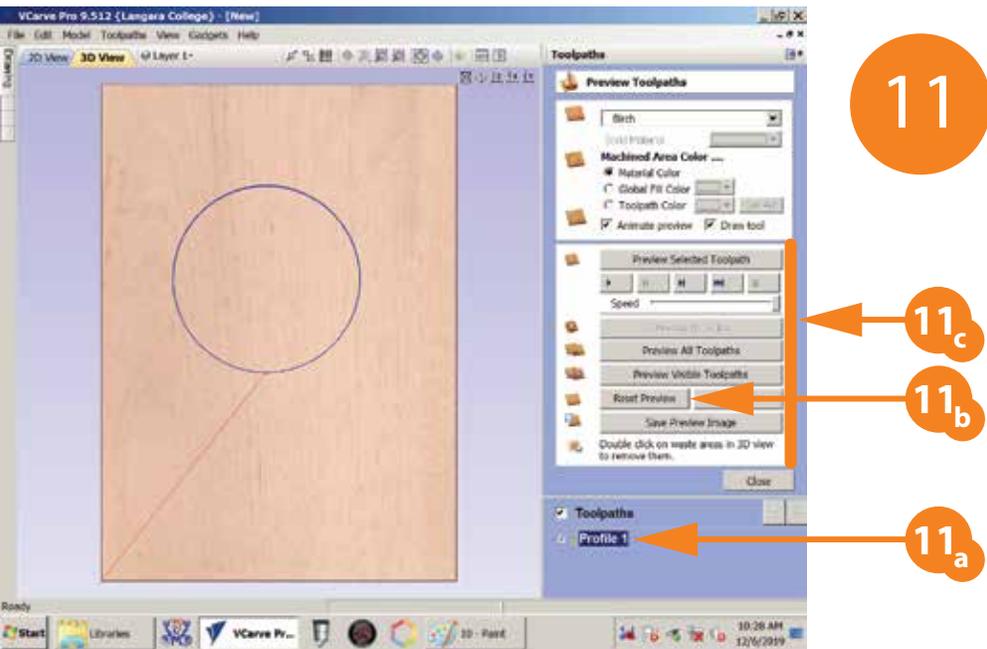
10

Preview Toolpaths

10a - select the single *toolpath* (*Profile 1*) you have just created to make it active (you can rename the toolpath here).

10b - Click on *Preview All Toolpaths* to get an animated preview of the toolpath



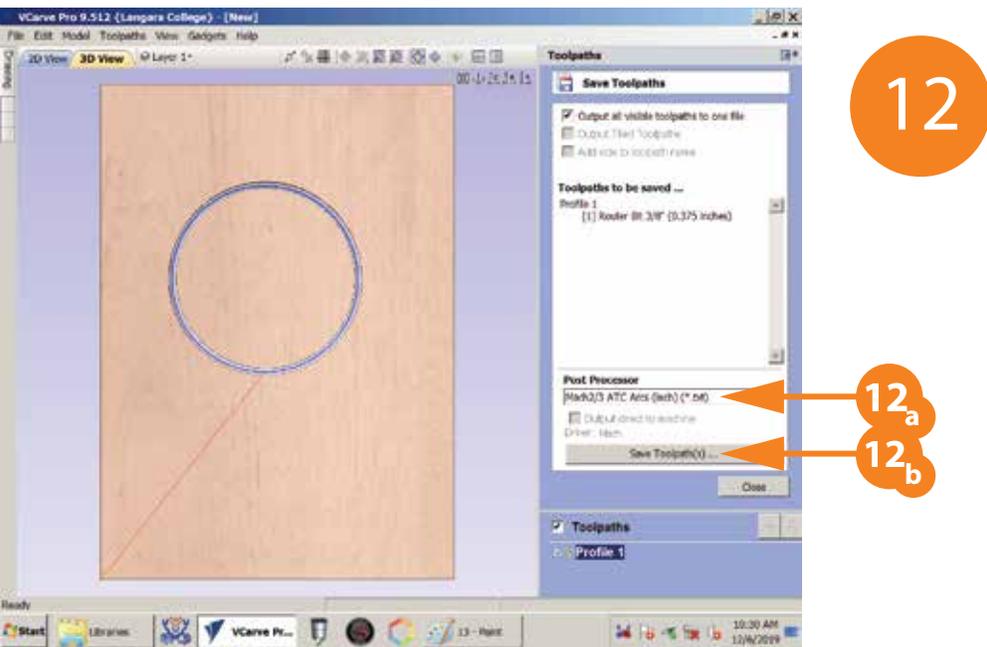


At this point you can click on the *2D View* tab and create *pocket* toolpaths. the main difference to profiles being the depth of the pocket and the type of path the router takes to clear out the pocket.

Preview Toolpaths

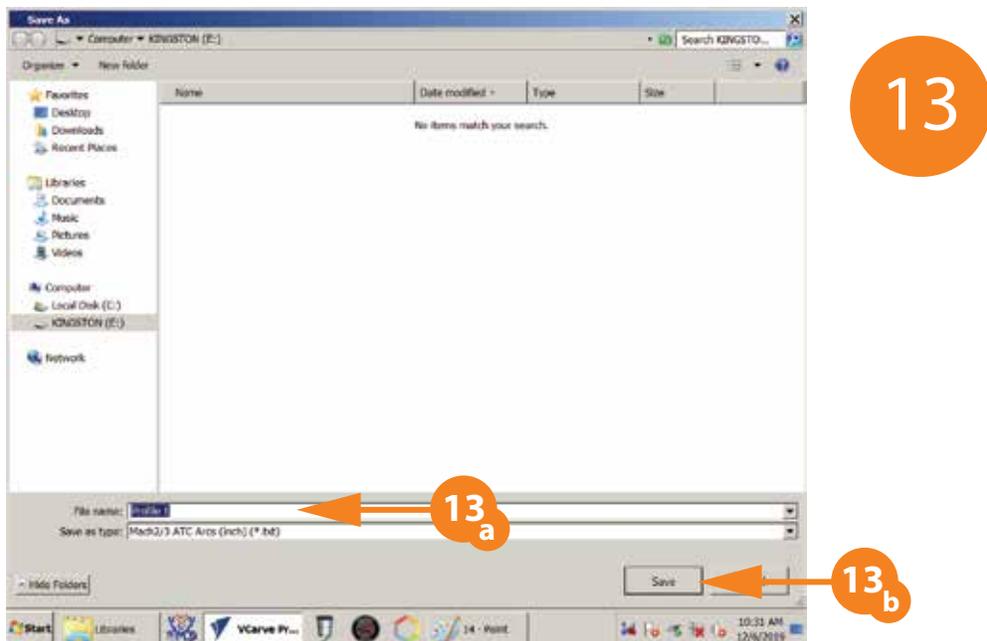
- 11a - Select all your profiles and pockets
- 11b - Click on *Reset Preview* to clear previous previews
- 11c - click on *Preview all Toolpaths* to get a graphic preview of how all your profiles will cut. .

Note: you must drag all your toolpaths into the correct order for them to cut properly and safely for your particular file.



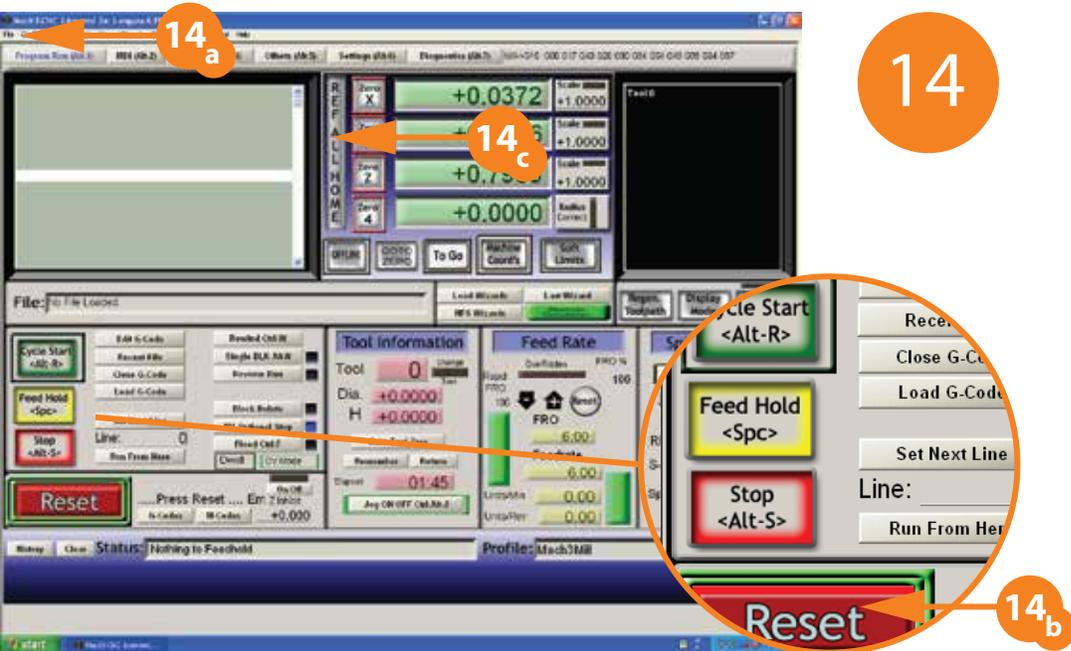
Save Toolpaths

- 12a - make sure "output all visible toolpaths to one file" is checked. IMPORTANT - make sure the correct post processor is selected (*Mach 2/3 ATC (inch) (*.atc)*) as this prepares the toolpaths for our specific output software on the CNC router.
- 12b - click on *Save Toolpath(s)* to save the file to your usb stick



Save As

- 13a - name your file , save it to your usb stick
- 13b - click save, then eject your usb stick, bring the file to the CNC router



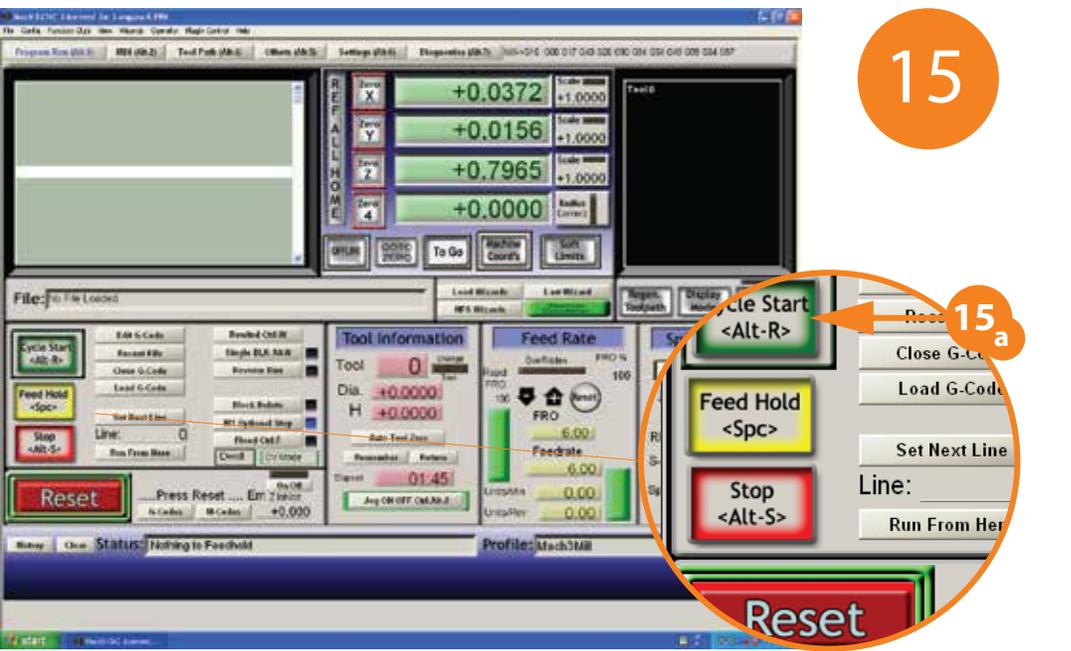
14

CNC Router Setup

14a - Insert your usb stick. Click *File* and select import text file. Navigate to your .txt file. Open.

14b - click on *RESET* to make sure the software is connected to the CNC router (the border on the button will go green when connected)

14c - **IMPORTANT!** (homeing the router). using the page up/down keys move the router AWAY from the work surface. Then, using the arrow keys, move the center of the router bit so that it is over the **bottom left corner** of the workpiece. lower the router bit with the up/down keys until it **just touches the work surface**. Press *REF ALL HOME*. :)

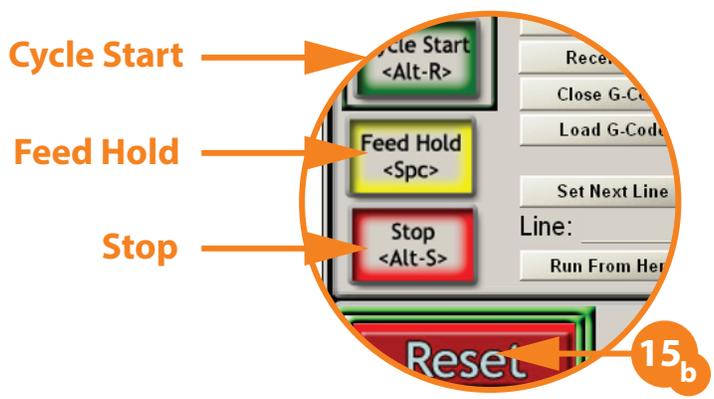


15

Cutting Toolpaths

15a - click *Cycle Start* to run your toolpaths. The router will move up from the home position, start spinning, move over to the first toolpath, then move down and start cutting.

15b - If there is a ****PROBLEM**** click on the **RESET** button or push in the **Emergency Stop** button. This immediately stops the machine. You must then start your file again from the beginning.



If you want to pause the machine without resarting the file from the beginning, click on:

Feed Hold . This uses up the remaining code in the buffer then stops the CNC from moving (the bit will still be spinning). then click...

Stop. This stops the rotation of the router bit and moves the router up (arrow keys can now move the router out of the way). When you are ready to restart click...

Cycle Start...