# VCarve Pro

This software is used for 2D design and calculation of 2D and 2.5D toolpaths for cutting parts on a CNC Router. The software can import 2D designs from other programs such as FormZ, Rhino and Illustrator, but also includes a full set of drawing and editing tools with advanced layout options.

In this tutorial, we will cover basic **toolpathing** of 2D designs for cutting parts on the CNC.

1. Export your file from the program you created it in as a .dxf or .dwg - Make sure your units are in inches and that drawing is correctly scaled. Save it to a thumb drive, Google Drive or e-mail it to yourself.

2. Open VCarve Pro. Click the "Open an existing file" button and select your file.

3. In the **Job Setup** column on the left, you will type in the correct settings for your part(s).
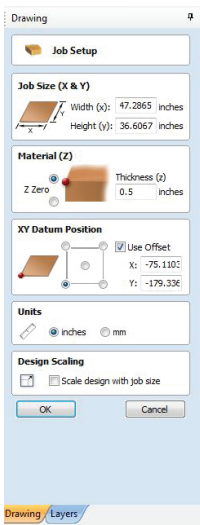
Job Size: Type in the X & Y dimensions of your material. Always give your design at least a 1" border of buffer material on all sides.
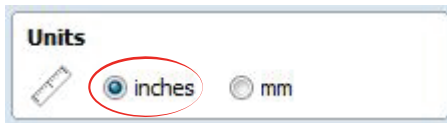The max material size for our CNC is X = 48" and Y = 96"

Material (Z): Make sure the top dot is clicked for Z Zero and type in the correct thickness of your material.

XY Datum Position: Make sure the dot at the bottom left corner is clicked. Uncheck "Use Offset" and set the X and Y positions to zero.
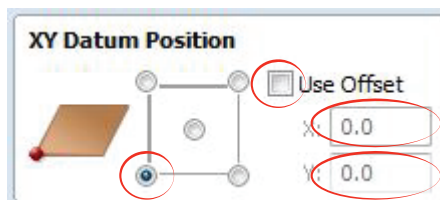
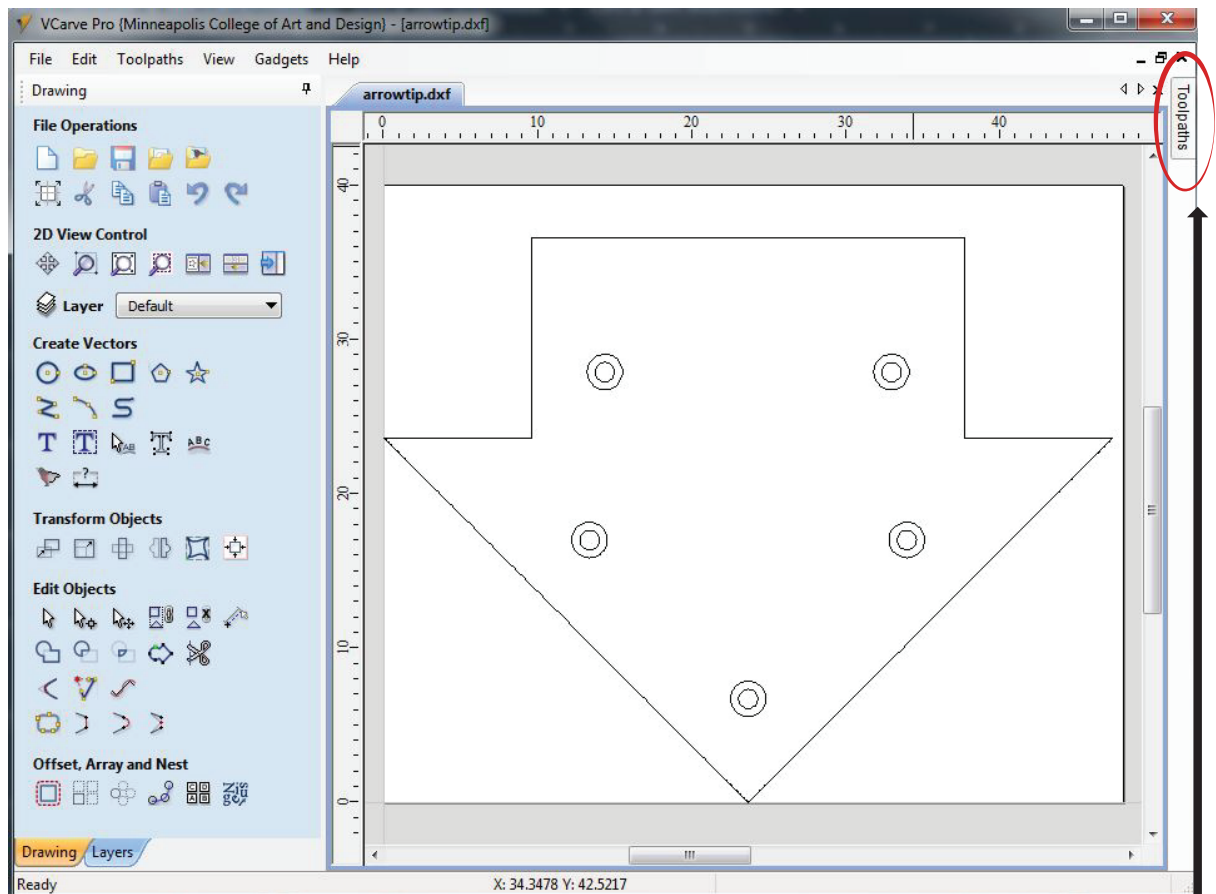**Units**: Make sure inches is selected.   **Design Scaling**: Uncheck "Scale design with job size

4. Click "okay".

5. The screen should look like the image below. On the left are various options for **File Operations**, **2D View Control**, **Creating Vectors**, and **Transforming** and **Editing Objects**.

Hovering your cursor over each icon will give you a description of what the tool does.
If you need to move, rotate, scale or edit the vectors in your file, now is the time to do it.

6. When your file is ready to be toolpathed, click the **Toolpaths** tab on the top right corner of the screen.

By clicking the pushpin icon, you can keep the **Toolpaths** box pinned open on the screen.

In this tutorial we will review the following **Toolpath Operations**:

Profile- Allows you to machine around the profile geometry of a part.

Pocket- Cleans out an enclosed area of a part at a specifically set depth.

You are welcome to explore other Toolpath Operations options depending on your cutting needs.
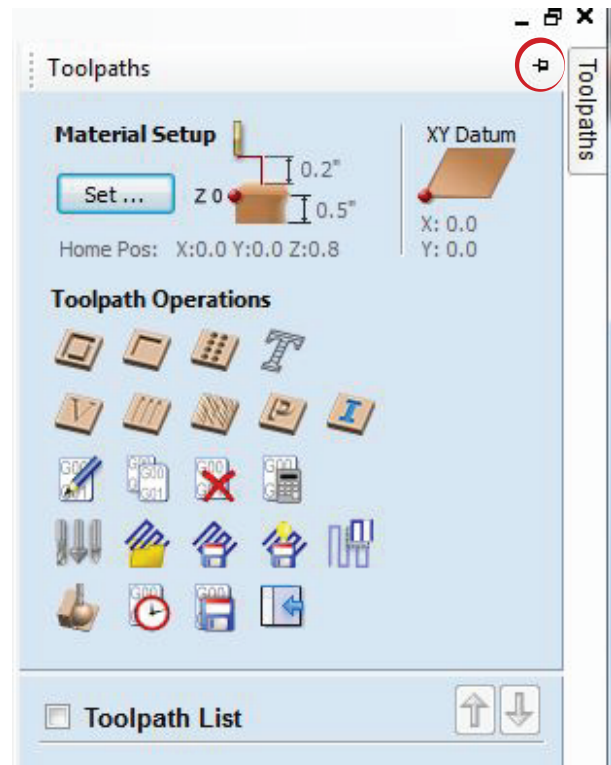

Pocket Toolpaths

Pocket cuts should always be done before Profile cuts. This is because your material has the most surface area and stability on the vacuum bed so you minimize the risk of your material moving when clearing out big areas.

1. Select the first shape or set of shapes that you would like to apply a pocket toolpath of a single depth to. To do this, simply click on the shape in the image area of the screen. To select multiple shapes at once, hold down the shift key as you click. When a shape is selected, it will be denoted by a dashed pink line.
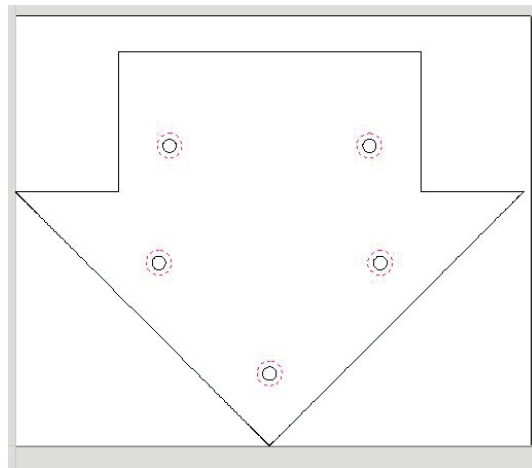
Note- If you are going to be pocket cutting different shapes at various depths, you must create separate toolpaths for each shape.

2. Once you have all the shapes you'd like selected, click on the **Pocket Toolpath** icon.
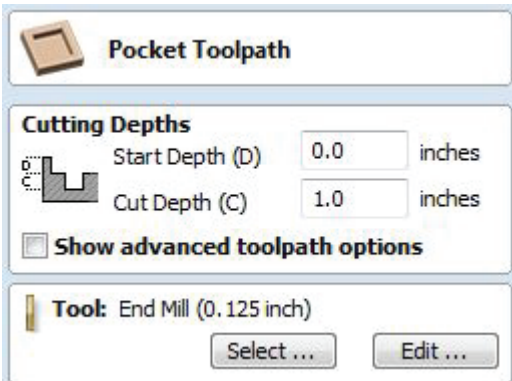
Set the **Start Depth** to zero and your **Cut Depth** to the desired depth. The **Cut Depth** MUST be less than the thickness of your material!
Uncheck **Show advanced toolpath options**.

Click the **Select...** button to change your tool bit.

In the **Tool Database**, select the bit you are using. It is recommended that an End Mill is used for pocket cuts. Here you also have the option to adjust:

Pass Depth: The general rule is that your pass depth should never be more than half diameter of your bit. So if your bit is .25", your pass depth should be .125"

Stepover:
-The stepover should be between 1/3 and 1/10 of the tool diameter.
-Use a larger stepover, in the 1/5 to 1/3 range, for soft materials that cannot hold detail well.
-Use a smaller stepover, in the 1/5 to 1/10 range, for hard materials or materials that can hold significant detail.
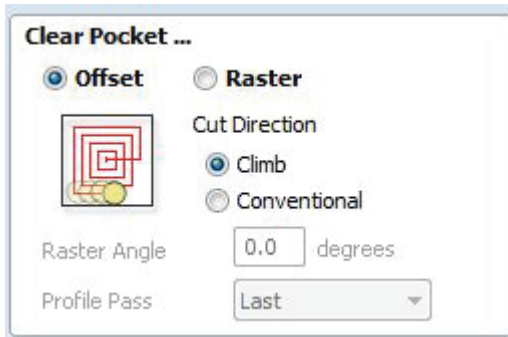-Use the largest tool that will allow you to machine your geometry.

Feeds and Speeds:
The spindle speed should be set to 18,000 rpm or less.
In general, the **plunge rate** should be roughly half of the **feed rate**.
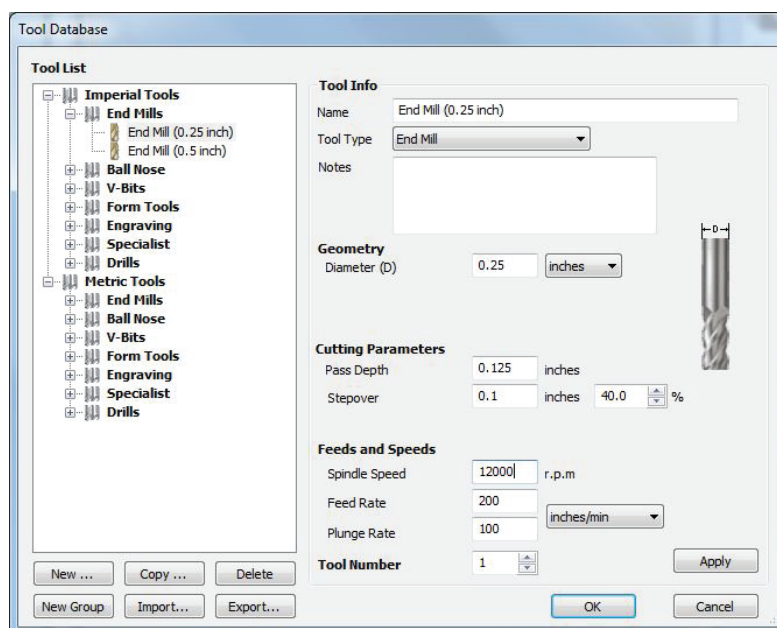
When your bit settings are correct, click "Okay" and continue to adjust your settings in the Pocket Toolpath dialog box.
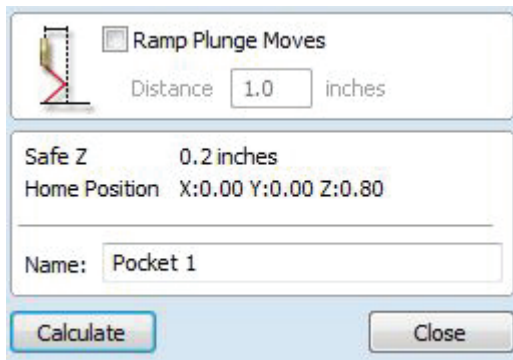

You have the option to clear your pocket by an offset or a raster.

Climb milling is when the direction of cut and rotation of the cutter combine to try to "suck" the mill up over (hence it's called "climb" milling) or away from the work. It produces the best surface finish.
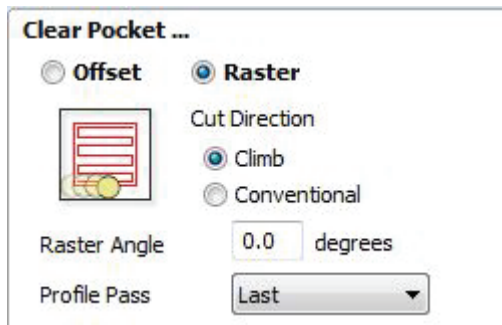
**Conventional milling** is preferred for rough surfaces.

Check **Ramp Plunge Moves** if you want your bit to enter your material on a ramp rather than a direct plunge.

Name your toolpath appropriately and click **Calculate**.

You will get a preview screen that shows your toolpaths in blue and gives you the option to watch an animated version of the toolpath.

When you are finished, click the **Close** button to return to the **Toolpath dialog box**.
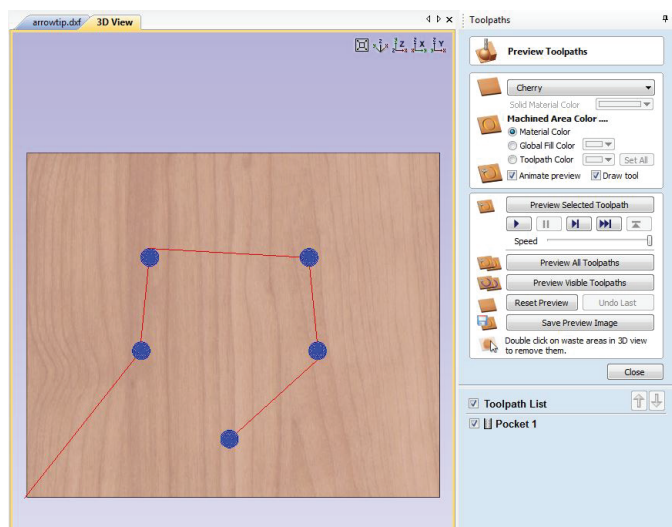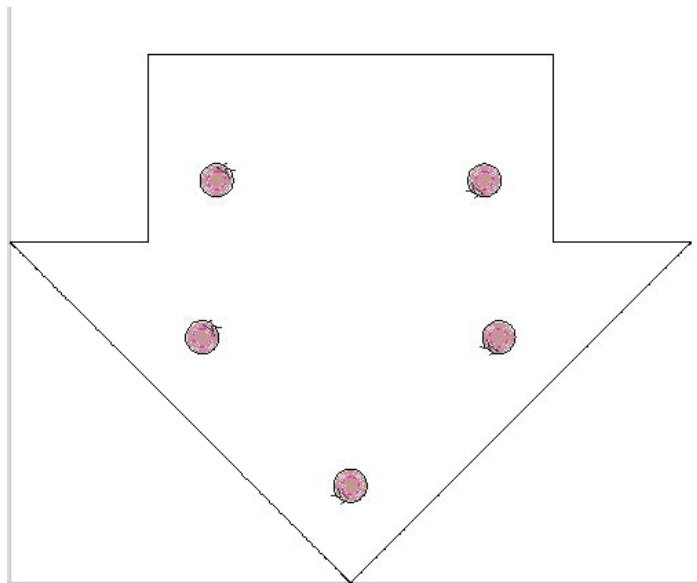
Profile Toolpaths

To get out of 3D view and return to your vector image, click the .dxf tab in the top left corner of the image area.
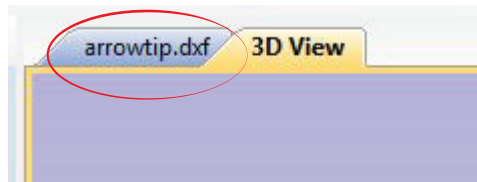
1. Select the first shape or set of shapes that you would like to apply a profile toolpath to. To do this, simply click on the shape in the image area of the screen. To select multiple shapes at once, hold down the shift key as you click. When a shape is selected, it will be denoted by a dashed pink line.

2. Once you have all the shapes you'd like selected, click on the **Profile Toolpath** icon.
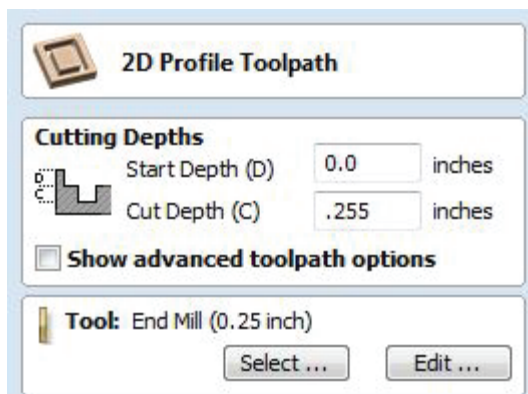
Set your **Start Depth** at zero and you **Cut Depth** to the thickness of your material plus an extra .05" to cut a bit deeper and into your spoil board. So if your material is .25" you'll want a Cut Depth of .255"

Click **Select...** to change your tool bit.

In the **Tool Database**, select the bit you are using. It is recommended that an End Mill is used for profile cuts. Here you also have the option to adjust:
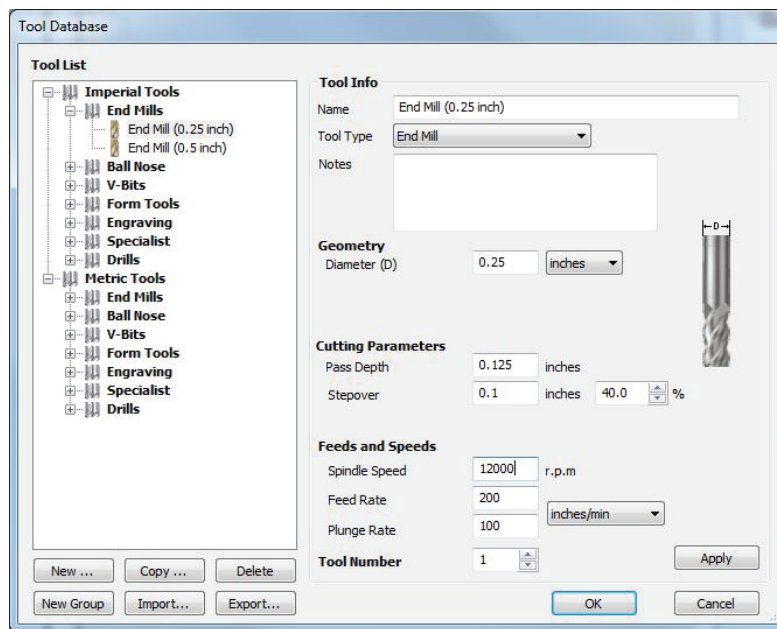
Pass Depth: The general rule is that your pass depth should never be more than half diameter of your bit. So if your bit is .25", your pass depth should be .125"
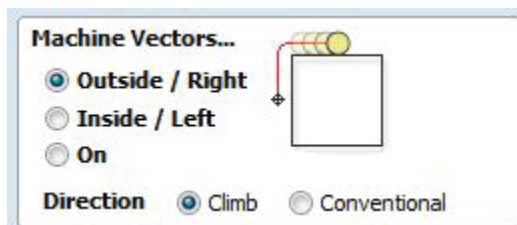
Stepover is not relevant for profile cuts.

Feeds and Speeds:
The spindle speed should be set to 18,000 rpm or less.
In general, the **plunge rate** should be roughly half of the **feed rate**.


When your bit settings are correct, click "Okay" and continue to adjust your settings in the Pocket Toolpath dialog box.


Select from **Outside/Right**, **Inside/Left** or **On** depending on what side of the path you want the cut to be made on.

Climb milling is when the direction of cut and rotation of the cutter combine to try to "suck" the mill up over (hence it's called "climb" milling) or away from the work. It produces the best surface finish.

Conventional milling is preferred for rough surfaces.
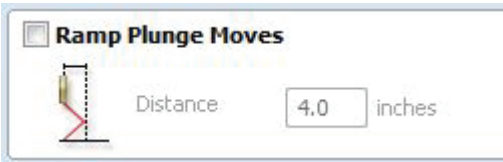
Check **Ramp Plunge Moves** if you want your bit to enter your material on a ramp rather than a direct plunge.
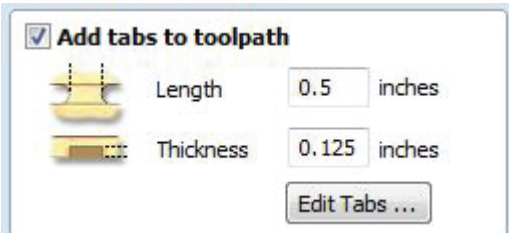

Check Add tabs to toolpath and enter the length and thickness you want for the tabs.

Click **Edit Tabs...**

Click **Constant Number** and type in the number of tabs you want for each shape. Generally, the larger the shape, the more tabs you want to hold it in place.
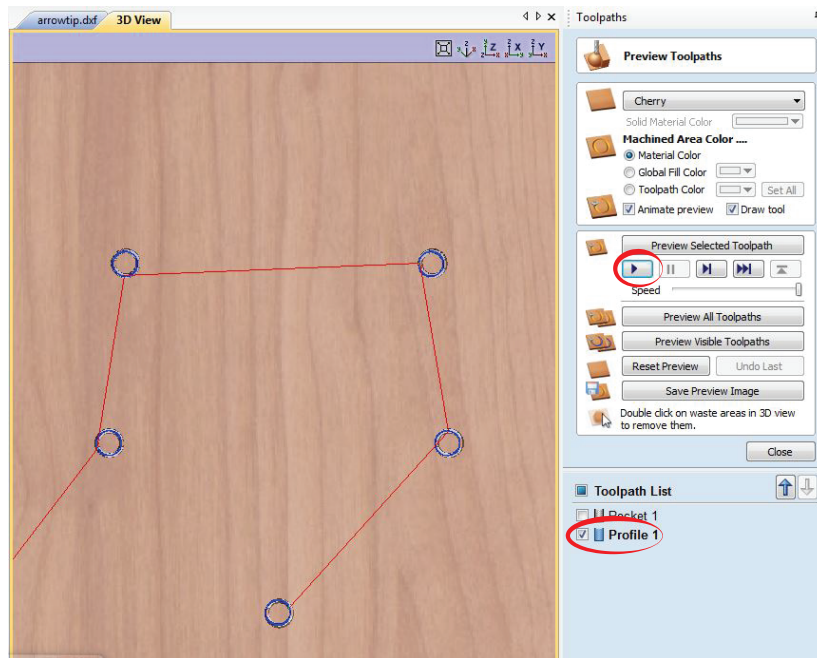
Then click **Add Tabs** and you will see them appear on your vector image represented by the letter "T". When you are finished adding tabs, click **Close** to return to the Toolpath Dialog Box.
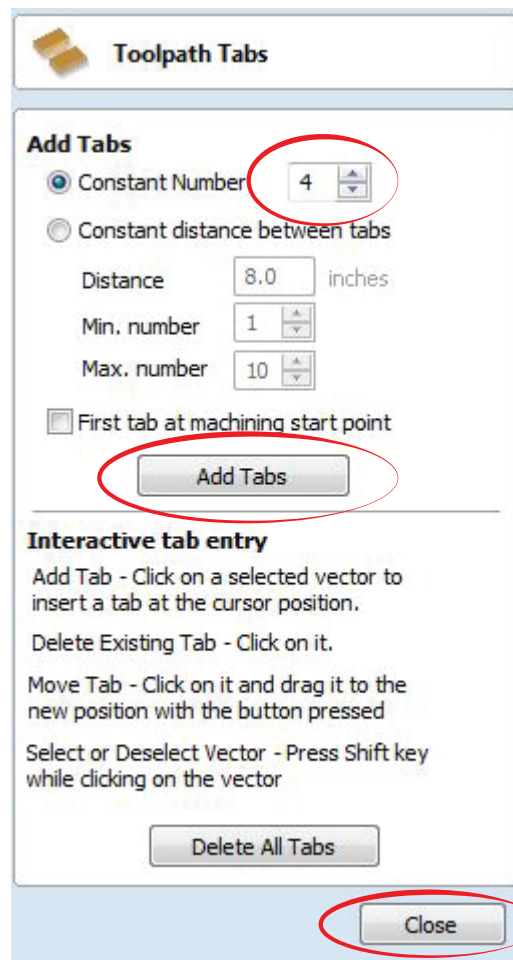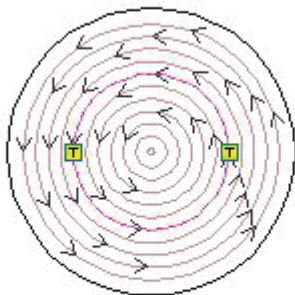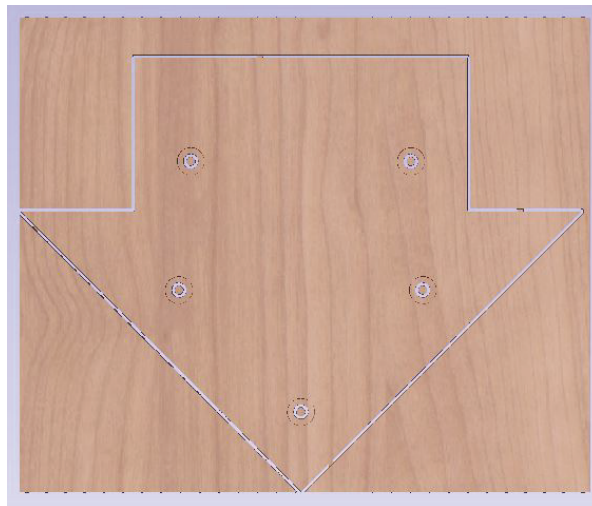
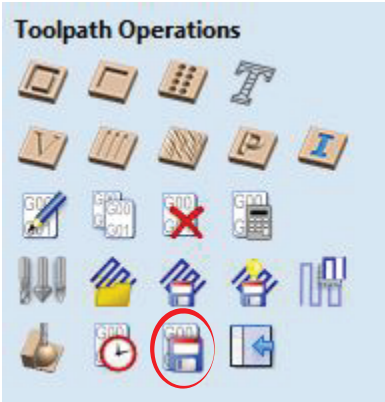Name your toolpath appropriately and then click **Calculate**.

Select the **Profile Toolpath** from the Toolpath List and click the Play button to watch an animated preview.

Finish toolpathing all the shapes in your file and do a final preview of all toolpaths.

Click the **Save Toolpaths** icon in the
Toolpath Operations box.

Toolpath Operations

Make sure everything on your **Toolpath List**
is selected.

Check the box next to
Output all visible toolpaths to one file

Save Toolpaths

☑ Output all visible toolpaths to one file

☐ Output Tiled Toolpaths

Toolpaths to be saved ...

Profile 4
    [1] End Mill (0.25 inch)
Pocket 3
    [1] End Mill (0.25 inch)
Profile 5
    [1] End Mill (0.25 inch)

Select the appropriate **post processor**
for the CNC you will be using.

The **Large CNC** uses
Mach 2/3 ATC Arc (inch) (*.txt)

The **Small CNC** uses
vELOCITY_NO_ATC (inch) (*.ncc)

Click **Save Toolpath(s)...** to save to
your drive.

Post Processor

Mach2/3 ATC Arcs (inch) (*.txt)

☐ Output direct to machine

Driver: Mach

Save Toolpath(s) to File

Close

☑ **Toolpath List**

☑ ▐ Profile 4
☑ ▐ Pocket 3
☑ ▐ **Profile 5**
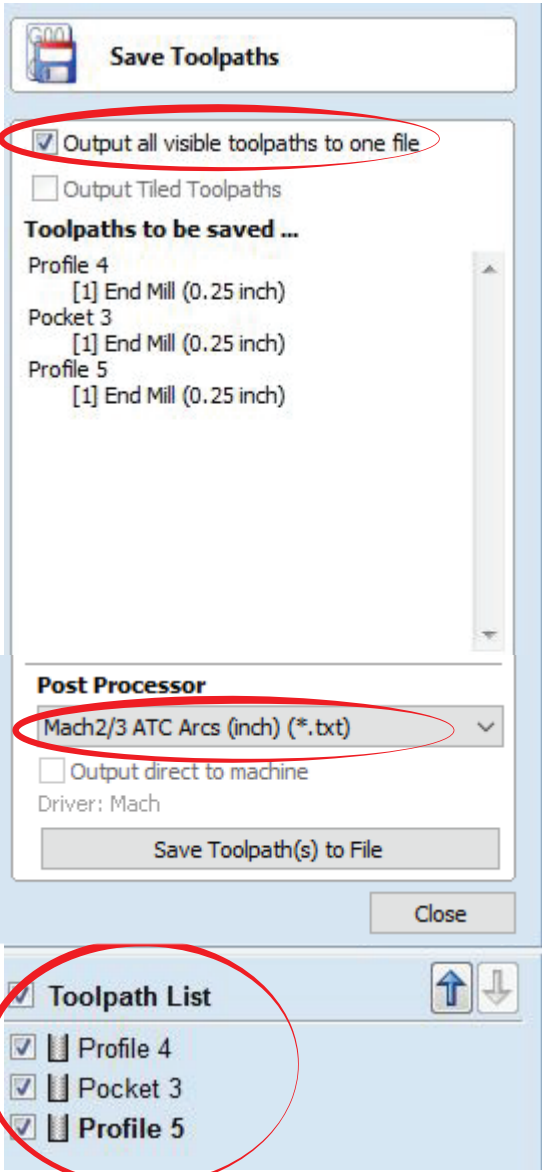
You are now ready to load your toolpaths in Techno CNC!