Calculating Toolpath(s)

Once you have imported and positioned all the vectors you wish to cut, click the “Toolpaths” tab on the right side of the screen and pin it open.

Select the vector(s) which you would like to generate toolpaths for. If you have multiple types of cuts you will make (for instance, a large pocket cut of a certain depth and a single cut around the part at a separate depth to cut it out of the surrounding material) select only the vectors associated with one type of cut. Next, under Toolpath Operations, choose the type of cut you would like to make. With 2D vectors, you will likely only use the first two listed: “Profile Toolpath” and “Pocket Toolpath”. The Profile Toolpath is used to cut along a single or set of vector(s) to a certain depth, creating a path the width of the diameter of the bit. The Pocket Toolpath generates a cutting pattern which mills out the entire area of material inside or between a closed set of vectors to a defined depth.

Examples of a Profile and Pocket cut are shown below. These were created using the same vector.

Profile Toolpath

To create a Profile Toolpath, select the icon under Toolpath Operations. Make sure “Start Depth” is set to zero, and set “Cut Depth” to the distance into your material you would like to cut. Under Tool, click the “Select …” button and choose the tool which you plan to use for your cut. Once you have selected your tool, make note of the spindle speed listed (in rpms) as you will need this later. Click “Ok”. Under Machine Vectors, Select one of 3 options: “Outside/Right”, “Inside/Left”, or “On”.

The “Climb” and “Conventional” cuts found in the above options depend on the material (wood vs. plastic (also grain direction of wood)) you are cutting and the type of bit (upcut, downcut, compression) you are using. In general, climb cut should work fine, but it is suggested that you test your cut first on a scrap piece to determine what works best for your conditions.
Leave “Ramp Plunge Moves” unchecked. Adding tabs to your toolpath will leave some material to hold the object so it doesn’t come loose during the last pass of the cut and potentially damage the piece being cut or breaking a bit. Check “Add tabs to toolpath” to select the length and thickness of the tabs, and click “Edit Tabs …” to change the positioning of the tabs. You can do this by following the onscreen instructions under Interactive tab entry. You can also automate the process of adding tabs by inputting a “Constant number” (the number of tabs per selected vector, and modifying the tabs placed by the software by deleting or moving them however you please. Once you have finished this step, click “Close”. Name your toolpath so that you can recognize what it is later, such as “Outside cut” or “Half inch cut”. This is especially useful when you have many toolpaths in your file. Click “Calculate”.

Pocket Toolpath

Note: A Pocket Toolpath requires a closed vector (a shape with a clear inside and outside, not a single line. Depending on the software used to create your vectors, they may not be closed even if they appear that way onscreen. Use the VCarve tool titled “Join open vectors” under Edit Objects in the Drawing tab to select and close vectors.

To create a Pocket Toolpath, select the icon under Toolpath Operations. Make sure “Start Depth” is set to zero, and set “Cut Depth” to the distance into your material you would like to cut. Under Tool, click the “Select …” button and choose the tool which you plan to use for your cut. Once you have selected your tool, make note of the spindle speed listed (in rpms) as you will need this for setting up the job. Click “OK”. Under Clear Pocket, you are presented with two options: “Offset” and “Raster”.

- The “Offset” option creates a spiraling path from the inside out or outside in, depending on the cut direction. This is generally best for circular or organic shapes.

- The “Raster” option creates a path which runs back and forth in straight lines to fill the selected area. The angle of the line direction can be changed (this angle is measured from the x axis, so 0 degrees is horizontal and 90 degrees is vertical cuts). The order in which the profile pass (path around the outside of the shape) is cut, it can be set to be before or after the raster of the inside area.

Name your toolpath so that you can recognize what it is later, such as “Circular cut” or “Half inch cut”. This is especially useful when you have many toolpaths in your file. Click “Calculate”.

Preview your cut and “Save Toolpath” (.sbp)