

Using Indexer as a Lathe



Table of Contents

Introduction	5
Overview.....	5
Getting started	5
Job setup	6
Drawing the shape	6
Creating the Toolpath	7
Cutting.....	9

Introduction

This document will show how to use the indexer like a lathe to turn radially symmetric objects such as baseball bats, stair spindles, etc.

V-Carve and Aspire now have a gadget that can machine such objects very easily, and for most situations this gadget is the easiest way to carve such objects. To install the gadget, in V-Carve or Aspire go to Gadgets – About Gadgets. Click on the link under “Downloading and Using Gadgets,” and then on Gadget Library. Click on the Turned Profile Gadget to download and read instructions on using this tool. If new to indexing, STOP READING AND GO DOWNLOAD THIS GADGET.

This gadget does not, however, turn objects quite like a traditional lathe, with the indexer spinning at a constant RPM. And for some situations – particularly in production settings – that technique is desirable.

Overview

In V-Carve Pro (or Aspire), you will draw the profile of the shape you wish to turn, and use a special post processor that tells your ShopBot to spin the indexer and cut out the profile from above. This post processor exports the Y-values in your V-carve file as Z-values, allowing you to precisely control the passes of your cut.

Getting Started

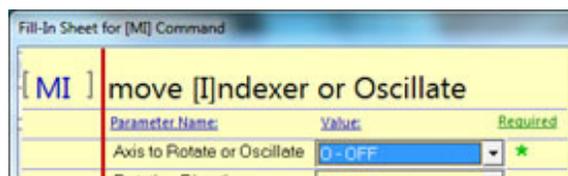
The required post processors are called ShopBot_Lathe_X_Parallel_TC_inch.pp (for X-parallel indexers), ShopBot_Lathe_Y_Parallel_TC_inch.pp (for Y-parallel indexers), or the corresponding metric posts. They are located in: C:/SbParts/Post Processors/Indexer Posts/V-Carve & Aspire.

If these files are missing, they can be downloaded from <http://www.shopbottools.com/msupport/drivers.htm> (they are included in the file Indexer Post Processors for VCarve and Aspire.zip)”.

To install the post processors, open V-carve Pro, and go to File – Open Application Data Folder. Open the folder PostP, then copy and paste the post processors into this folder. Close V-Carve, and open it again. The names of these posts will appear in the drop down menu next time a toolpath is saved.

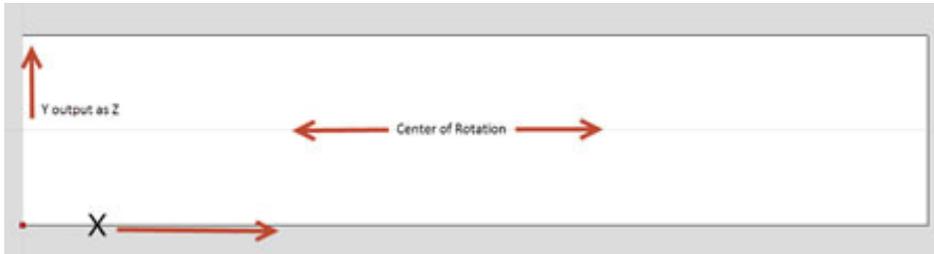
The ShopBot Lathe post processors call the MI command ([M]ove->Move [I]ndexer or Oscillate), which causes the indexer to spin at a set RPM. The indexer will continue to spin during subsequent commands, allowing the toolpath to cut the outline of the desired shape.

WARNING: If for some reason a file is interrupted, the indexer may continue to spin. To turn it off, type MI in the ShopBot command console, and select O-OFF:



Job Setup

The turning blank we are using for this example measures 2.5" x 2.5" x 12". Begin by creating a new file in Vcarve Pro using a Width (x) of 12" and the Height (y) of 2.5 (do this regardless of whether your indexer is set up parallel to the x-axis or the y-axis). The Thickness (z) should be half of the Y-Height (1.25") - this will only be used to know how far to pull up for a rapid move. Check the box Use Offset and set the Y to be -1.25" (note the negative sign). This moves the y=zero line to the centerline of your blank. Installing VCarve Pro application



Job Setup

Job Size (X & Y)

Width (x): 12.0 inches
Height (y): 2.5 inches

Material (Z)

Z Zero  Thickness (z): 1.25 inches

XY Datum Position

Use Offset
X: 0.0
Y: -1.25

Units

inches mm

Modeling Resolution

Standard (fastest)

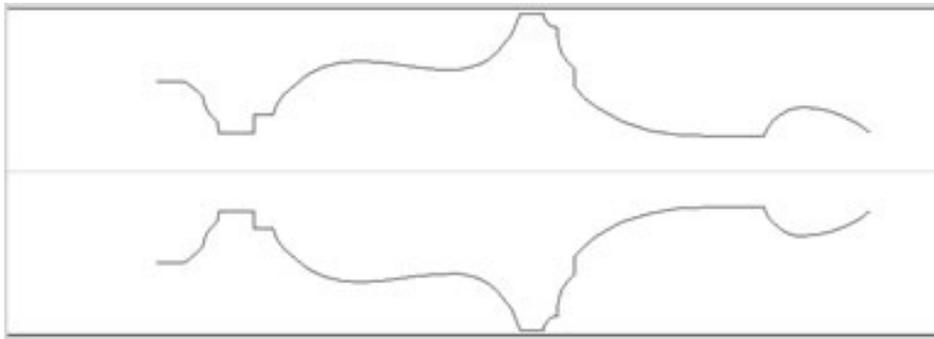
Appearance

Canadian Maple
Solid Color:

OK Cancel

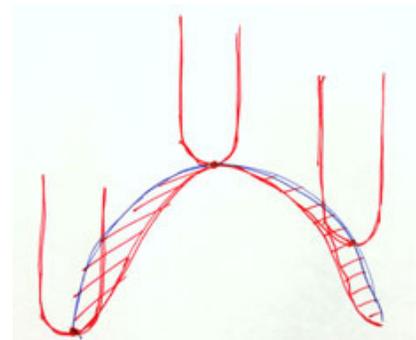
Drawing the Shape

Design the profile to turn in the top half of the blank. Make sure not to draw past the center of rotation. Once the profile is drawn in the top half, select the toolpath, and use the keyboard short-cut Ctrl+Shift+V to mirror it about the job center. This will help to visualize the full turning.



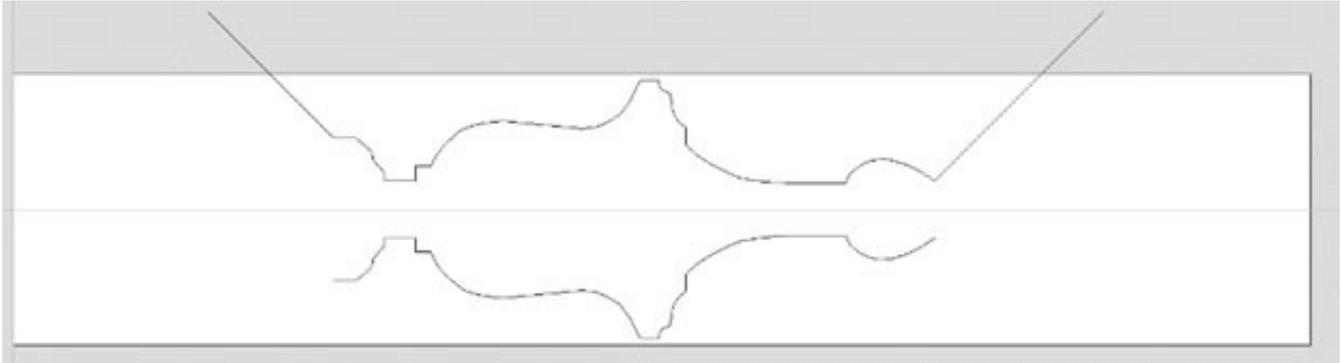
Keep in mind that the profile drawn will not exactly match the shape that is cut, due to the cutter's geometry. To illustrate this, consider machining the dome shape sketched below. The blue line illustrates the desired, drawn shape. The red line illustrates the actual shape that will be cut by a ball-nose cutter. The hashed area represents excess machined material.

Understanding this discrepancy will allow modification of the drawn shape to better match the actual toolpath that bit will take. To modify the drawn outline precisely, use node editing by selecting the toolpath and typing N on the keyboard.

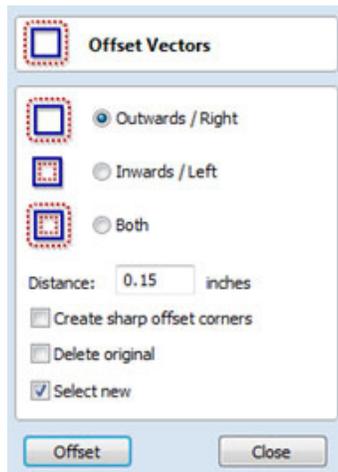
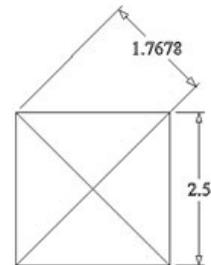


Creating Toolpaths

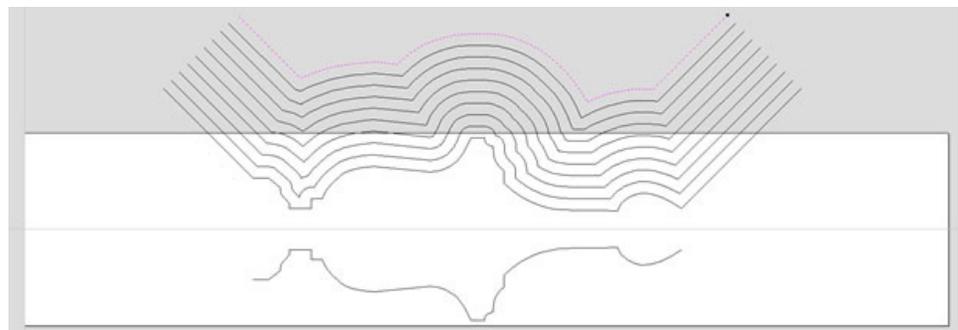
The next step is to draw all the passes that the tool will use to create this outline. Because of the way the post processor exports Y values as Z moves, it will be necessary to manually add lines for roughing. First, draw lines extending the end points of the top profile, to direct how the tool will enter and exit the blank.



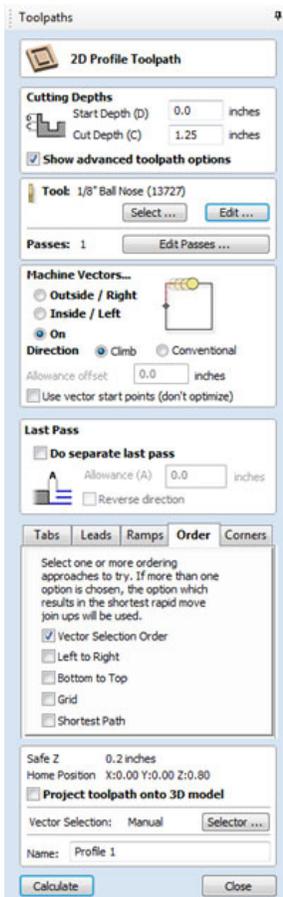
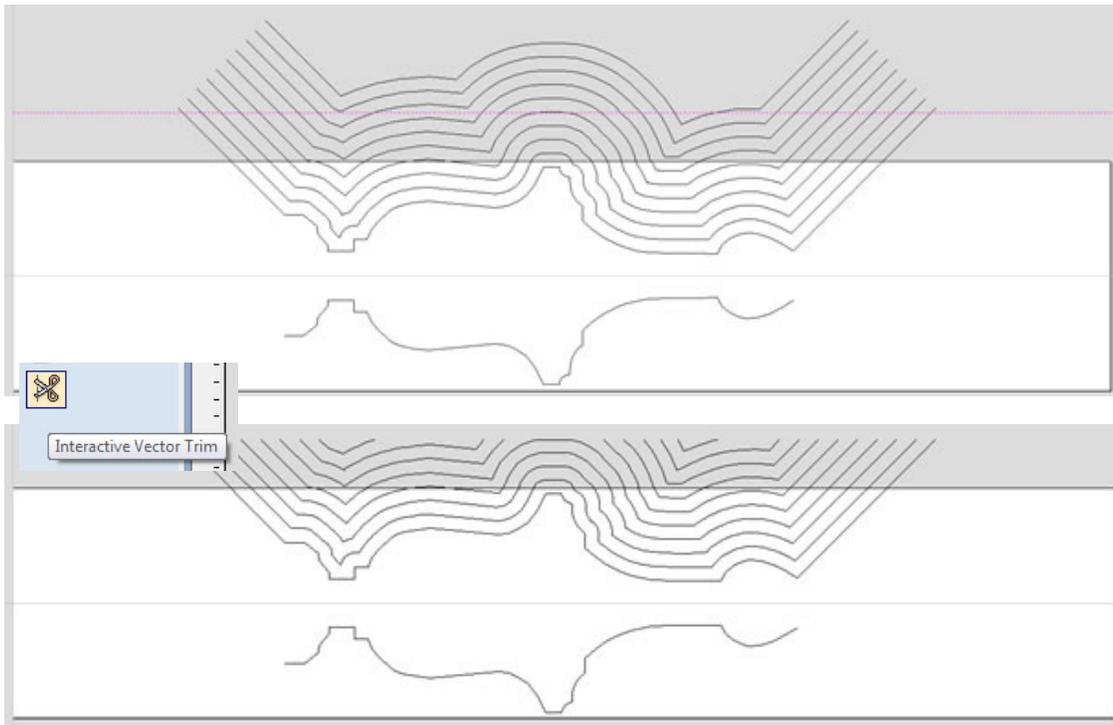
If starting with a square blank, it will be necessary to extend these lines well beyond the top surface of the blank, in order to account for the square corners. In this example, even though the blank is only 2.5", the toolpath needs to start at a height of 1.77" above the centerline.



Next, add the roughing lines. The easiest way to draw the roughing lines is to use the offset selected vectors tool, located under Offset and Layout in the drawing tab. Check Outwards / Right and under distance enter the pass depth. In this example, use a pass depth of .15 inches. Check the box Select New, and then click the Offset button multiple times until the offset curves extend well beyond the edge of the blank.



Next, draw a line at Y=1.77, and use the trim tool to cut off the vectors at that point.



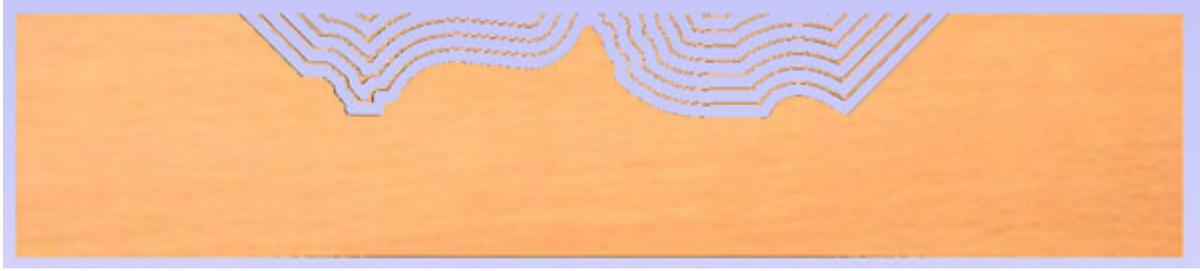
To create the toolpath, select the vectors one by one, starting with the first roughing pass, and then select the profile toolpath icon. The cut depth will not be used, but if it is set to the thickness of the material it will aid in visualizing the cut.

Select the desired cutter. Typically, a ball nose cutter will provide the best results; in this example we have used a 1/8th inch ball nose. The pass depth and step-over parameters will not be used, but the spindle speed, feed and plunge rates will be. Set the feed and speeds much lower than normal in order to avoid carving spirals.

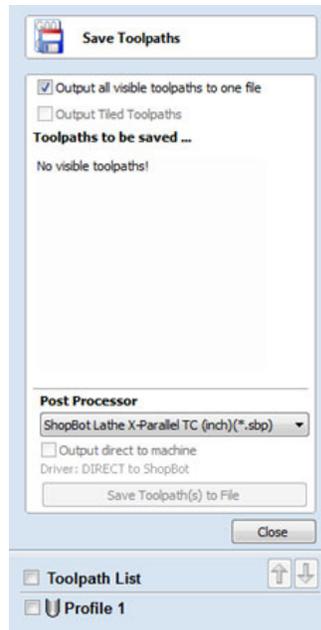
Set the Machine vectors option to On line. Under the Order tab, check the box to Vector Selection Order. This option will cut the vectors in the order that they were selected.

Once the toolpath is calculated, preview it and make sure that the vectors are cut in the desired order.





Save the toolpath using the lathe post processor that fits the indexer setup.



Cutting

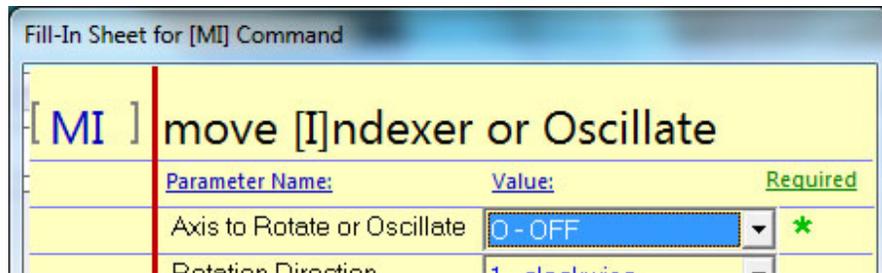
WARNING: If using the post processor for the first time, it is highly recommended to run an air-cut to make sure that the file has been properly set up. This can be easily done by zeroing the x, y, and z axis at some point in space, well away from the blank or table.

When ready to cut, zero the bit to the center axis of the indexer, at one end of the blank, such that moving in the positive x direction (for x-parallel indexers) or positive y direction (for y-parallel indexers) would move the bit toward the other end of the blank. There are a couple of ways to zero the z axis to the center axis of your indexer without removing the blank; one of the easiest is to bring the bit to the top of the blank, then type VA in the command console and enter half the diameter of the blank.

When running the file, a prompt will appear to choose the direction of rotation. The direction that gives the best finish depends on the material and cutter.

Next, enter the RPM. The default of 150 is usually sufficient; on 6" and 12" indexers can go as high as 250. Keep in mind that higher RPM means lower torque; depending on how aggressive the passes are - it may stall the motor with too high of an RPM. Play around with this setting to find the optimal speed for the job.

If there is an error in the file, or you pause it for some reason, you will need to manually turn off the indexer by typing MI in the ShopBot command console, and select O-OFF:



Parameter Name:	Value:	Required
Axis to Rotate or Oscillate	O - OFF	*
Rotation Direction	1 - clockwise	

After the file has finished running, sanding can be done by spinning the project on the indexer. Start the indexer spinning again by using the MI command, selecting B-Axis, and typing in the desired RPM.

Happy turning!