ShopBotTools.com

ShopBot CNC Handbook

ShopBot Control Software versions 3.8 and later

ShopBot Desktop
24” x 18”

ShopBot Desktop MAX
36” x 24”

ShopBot Desktop MAX ATC
36” x 24”

ShopBot Buddy® 32”
ShopBot Buddy® 48”

PRSstandard & PRSalpha
48” x 48”, 96” x 48” and larger

5-Axis
34” x 34” x 24”

Visit **www.shopbottools.com** for:

Products • Forum • Documentation
Training • Customer Success Stories

SALES/TECH SUPPORT

Call: 1-888-680-4466 • +1-919-680-4800
Email: sales@shopbottools.com
support@shopbottools.com

+1-919-680-4800 • 888-680-4466
ShopBotTools.com

Handbook for ShopBot CNC

Congratulations on the addition of a ShopBot Personal Robotic System to your world. Use this Handbook as a resource for you to operate the ShopBot Tool and the design (CAD), toolpathing (CAM) and ShopBot Control software safely, easily and efficiently.

MECHANICALS

ShopBots come in different models, with different levels of assembly and set up. Refer to the Quick Start Guide (or Assembly Manual for tools requiring assembly) included with the model of ShopBot that you have. The manuals for each model may also be downloaded from the ShopBot website: www.shopbottools.com > Support Resources.

SOFTWARE

The Assembly and Set Up documentation also includes information on how to load the SB3 ShopBot Control software and Vectric® Design/Toolpathing (CAD/CAM) software on your computer(s). See pages 5 and 6 for suggestions/requirements for the computer(s) assigned to your ShopBot.

ShopBot Control Software: The accompanying ShopBot flash drive contains a copy of the SB3 ShopBot Control Software that runs the tools. The latest version of the ShopBot Control Software is always available for download (free) from the website **www.shopbottools.com/support/**

Vectric® VCarve Pro Design and Toolpathing software (2D CAD/2D & 3D CAM) is included with the purchase of a ShopBot. **Aspire** (2D & 3D CAD/CAM) is an upgrade to VCarve Pro. **www.vectric.com**

All downloads of the Vectric software, as well as supplemental clip art, files and tutorials are available through the Vectric Portal. After the purchase of your ShopBot, you should have received an email with instructions on how to create an account with Vectric. Use that information to create the V&CO log in.

Important: Keep the email associated with the ShopBot order, and the User Name/password created for the V&CO log in in a safe place that can be accessed later. Suggestion: Write down the information below, or at least the location of an electronic version of the information. ShopBot Tools does not have the log in information, nor the license code. Contact **support@vectric.com** for help identifying the Vectric license.

Email associated with order:______________________Registered User Name:___________________________

License Code:____

An electronic version of the information can be found:____

Table of Contents

On the Flash Drive

ShopBot Control Software (SB3)

Do not connect the ShopBot to the Computer until you have installed the ShopBot Control Software which installs the driver for the USB connection!

The ShopBot Control Software is the software that operates the ShopBot. It must be installed on the computer that is connected to the ShopBot Control Box. While it may be installed on as many computers as you would like, the Control Software will only run in **Move/Cut** mode if there is an active ShopBot connected to the computer. Otherwise, the Control Software will open in **Preview** mode.

Detailed information on installing the Control Software on the computer can be found in the Assembly Manual that accompanies the actual ShopBot model, on the ShopBot website, or in the Quick Start Guide for Beginning ShopBotters found in the HELP menu in the Control Software. If the drivers for the ShopBot do not install correctly, documentation on installing the drivers can be found in the online FAQs *https://www.shopbottools.com/support/faq*. See the section on Computer Requirements for more information on setting up the computer that runs the ShopBot. Check the ShopBot website (*www.shopbottools.com/support/*) for any updates. The latest version is always available free of charge. With the Control Software, you can send single moves to the ShopBot, warm up the spindle, preview a part/cut file on screen, or run a ShopBot Part File (.sbp extension).

Tool Library for Vectric Software (VCarve Pro and Aspire)

The Chapter on **Bit Selection, Feeds and Speeds** has recommendations for setting the feed rate, spindle speed and pass depth for different router bits and tool operations in Tool Library. While the chapter emphasizes Vectric software, the recommendations are good starting points for all Computer-Aided-Machining software.

The Flash Drive contains a Tool Database which can be loaded into the Vectric Software as a starting point. Instructions on how to manage the Database and create a new tool (bit) for a bit not in a ShopBot Starter Bit kit are also on the Flash Drive and in the Chapters on **Trouble Shooting and Maintenance.** The Projects and Design files also have information about bit selection, feeds and speeds.

Design and Project Files

The section of the **ShopBot Safety and Basic Use** titled Intro to Using CAD/CAM to Design a Part file gives an overview of the steps to get from putting a Design in the Computer (What and Where), Toolpathing the Designs (How), and Creating the ShopBot Parts (.sbp) files. The CAD/CAM (VCarve Pro) file and the .sbp files are on the thumb drive.

The Chapter titled **Projects and Techniques** has several examples of projects which are examples of how to use the VCarve Pro design and toolpathing software. Instructions in .pdf format to follow along or to print out, and the VCarve Pro files themselves are on the Flash Drive.

Additional Resources

Explore The ShopBot Website

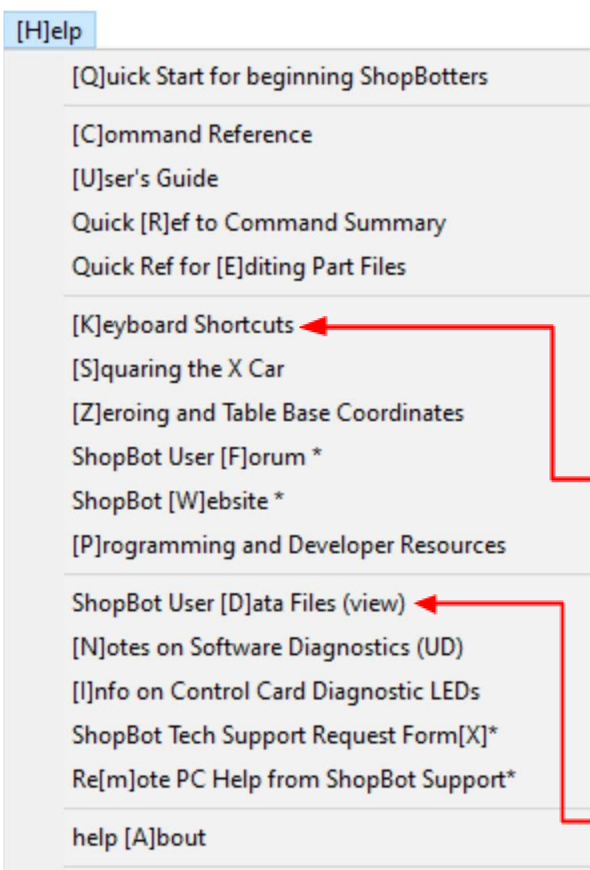
Support Resources on the ShopBot website has links to **Documentation**, written and video **Tutorials**, and a **Calendar of Events** and scheduled trainings.

Community has links to **Blogs** (check out the Original Blog Series), **Events** such as Camp ShopBots, Training, and the **ShopBot Forum**, where ShopBot owners and users themselves ask and answer questions.

Explore CNC for Case Studies and **Applications** for inspiration on how ShopBots are used in Manufacturing, Education, and the Arts. Check out the **Projects** with design files and instructions. There is also background information on CNC (Computer Numeric Design) and the history of ShopBot.

Resources Found in "Help" in the SB3 Control Software

These references are not printed out, but may provide additional resources for the ShopBot operator.



- [Q]uick Start has information about loading the software on the computer, tips for ShopBot set up, and running a part file.
- [C]ommand Reference is a summary of ShopBot commands.
- [U]ser's Guide is a historical document that was written in the early years of ShopBot. Some sections are relevant for those to wish to get deep into ShopBot code.
  - Working with G-Code begins on page 49.
  - Working with ShopBot converters begins on page 58 (.sbp to .dxf, etc.).
- Tips and Techniques in this section.
- [P]rogramming and Developer Resources gives more details about specific commands used in ShopBot code.
  - Look to this document if you wish to get into/teach Programming using ShopBot.
  - Information about using Math, User Variables (&), and System Variables (%) begins on page 7.
- The files in this section help the operator help Support in troubleshooting problems with hardware, software and computer settings.
- Click on help [A]bout to determine which version of the Control Software is loaded.

Computer Requirements and Suggestions

For the Computer that Runs the ShopBot (SB3 Control Software)

PC computer running Windows 7, 8, 8.1 or 10 (as of June 2020).

When selecting the computer to run the ShopBot, keep in mind that a significant amount of dust and debris is generated by CNC machining. The PC connects to the ShopBot Control Box running SB3 Control Software via a 10-foot USB cable supplied by ShopBot. The computer must stay in that 10 foot range. Do not replace the USB cable supplied by ShopBot with a longer cable. Peripherals such as a keyboard or mouse (corded or cordless) may be farther than 10 feet away from the ShopBot Control Box.

Dedicate one computer to running the ShopBot. While that computer is running a ShopBot Part (.sbp) file, it cannot be engaged in any other activity, including checking for Windows (or other) updates on the internet, or designing parts in the CAD/CAM software. If using an Apple Product, the Computer must be set as a dual boot, and be running in PC mode.

Minimum Requirements for the Computer:

- Pentium or Dual Core Processor, 2.0 GHz or Higher.
- 4 GB RAM (8 GB RAM preferred.
- Graphics display 1024 x 768.

Additional requirements/suggestions:

- USB Port (2.0) for the ShopBot.
- Additional ports for Spindle Speed Controller (ShopBot Gantry and Buddy Tools).
- A USB port for transferring files between the Design Computer and the ShopBot Computer via flash drive.
- Additional ports as needed for peripherals such as mouse, keyboard, monitor, etc.
- If using a laptop to run the ShopBot, always have the computer plugged in.
- Adobe Reader loaded on the computer for reading “Help” files from both Control and Design software.

The following YouTube video has information on how to install a clean version of Windows Operating System on a computer, as well as tips for setting up the computer and loading the SB3 ShopBot Control Software: https://youtu.be/pC3K6QBSrz0

If you are not installing a clean version of the Windows software, skip to the sections on loading the Control Software. While the installation file for the VCarve Pro CAD/CAM software is now downloaded from the Vectric Portal rather than sent on the ShopBot Flash Drive, the steps to install the Vectric software and load the license code are the same.

For the Computer that runs Vectric VCarve Pro or Aspire Design/Toolpathing (CAD/CAM) software

Minimum Requirements

- 2 GHz multi Core CPU.
- 4 Gb RAM with Windows 7, 8, 8.1 or 10 (as of June 2020).
- 300 Mb Disk Space (Program).
- USB drive required for USB Media Pack if purchased.
- 7.7 GB Additional disk space required if you wish to store the tutorials and clipart on your hard drive.
- Graphics display 1024 x 768.

3D Rendering Design Software such as Solidworks, Inventor and Fusion 360 have different requirements to run efficiently on the Design computer. Check with the software company for the most up-to-date requirements.

Contact ShopBot

ShopBot Tools, Inc

3333B Industrial Dr
Durham, NC 27704
www.shopbottools.com
919 888-680-4466
+1 919 680-4800

Technical Support

Email support@shopbottools.com for Technical Support
For a form to help get started with Support: https://www.shopbottools.com/support/contact-form

Office Hours

Monday–Friday: 9am–5pm EST

After Hours Tech Support

Monday–Friday: 5pm–9pm EST
Saturday–Sunday: 8am–9pm EST

Safety and Basic Use

ShopBot Control Software versions 3.8 and later

This Chapter outlines important steps in safely running a ShopBot CNC (Computer-Numerically-Controlled) machine.

The Chapter refers to other Chapters in the Handbook that have more detailed information. Other resources such as video tutorials and documents can be found on the ShopBot website https:www.shopbottools.com.

In addition to descriptions of the physical ShopBot itself, this Chapter references both SB3 ShopBot Control Software and VCarve Pro CAD/CAM software included with the purchase of the ShopBot. It assumes that both the Control and CAD/CAM software have already been loaded onto the computer that will run the ShopBot. If the CAD/CAM software of choice is something other than VCarve Pro/Aspire, general concepts such as X and Y origin and Z start position will remain valid.

Overview of ShopBot Mechanicals

ShopBots are digital fabrication machines, also known as CNC (computer-numerically-controlled) tools. The three major configurations of ShopBots are described below. All ShopBots move in at least 3 axes: X, Y and Z. Some models of ShopBot are available with an Automatic Tool Changer (ATC) option. A 4th axis Rotary Indexer can be added to all 3 axis models of ShopBots. See separate documentation for the ShopBot 5 axis tool.

ShopBot Gantry Tools (example: PRS 96-48):

- Stationary Tool.
- Gantry moves along the rails of the table (X).
- Spindle travels across the gantry (Y).
- Z axis controls height of spindle.
- Travel in the X: 96”.
- Travel in the Y: 48”.
- Travel in the Z: 6 – 7”.
- Drive System: Rack and Pinion.

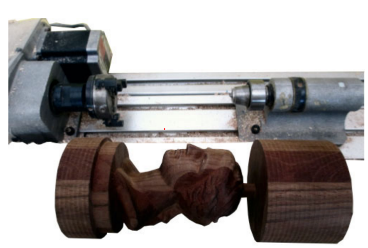
ShopBot Buddy (Factory Settings):

- Can be moved if castor kit included.
- Table supporting the material moves in the X axis.
- Spindle moves over top of material in the Y axis.
- Z axis controls height of spindle.
- Travel in the X: 24”.
- Travel in the Y: 36” or 48”.
- Travel in the Z: 6 – 7”.
- Drive System: Rack and Pinion.

ShopBot Desktop and Desktop Max:

- Can be stationed on any sturdy workspace.
- Plugs into household outlet (US: 110, 15A).
- Travel in the X: 24” (Desktop) or 36” (Max).
- Travel in the Y: 18” (Desktop) or 24” (Max).
- Travel in the Z: 5.5”.
- Drive System: Lead Screw.

Rotary Indexers:



- 4th axis Rotary to cut in the round.
- Requires a driver to control the live motor.
- Available in 3”, 6” and 12” size.

Control Box, Drivers and Motors

The ShopBot Control Box contains the drivers to control the speed and directions of the motors on the X, Y and Z axes. In all current models of ShopBots, the power to the ShopBot as well as the cutter head (spindle or router) is run through the Control Box. A relay in the Control Box allows the Control Software to turn on and off the power to the spindle or router at the appropriate time.

There are two major models of ShopBot Control Systems: Alpha Tools with closed loop stepper motors, and Standard Tools with open loop motors. The different models of ShopBot (Alpha, Standard and Desktop) can be determined by looking at the Control Box and Stop Pendants.

ShopBot PRS Alpha

Closed loop stepper motors: location of cutting head monitored by Control Software.

- E-Stop pendant includes a **Reset** button to turn on power to the VFD (see below) and reset the motor drivers when turning on the ShopBot or after engaging E-stop.
- Press **Start** button on pendant to start the spindle spinning before responding to the **OK** prompt to start the part file.

ShopBot PRS Standard

Pre- and Post- May 2020

Stepper motors are open loop: location not monitored by Control Software.

- Spindle starts spinning when **OK** given to the prompt when starting a part file.
- Single button Remote stop will pause movement of tool and spindle when engaged.
- Standard Control Box from May 2020 has a pendant like the Desktop tools.

Control Box pre-May 2020
Separate remote stop
Lockout key on front of
Control Box

Control Box post May 2020
Separate remote pendant
like Desktop Spindle
Lockout, soft stop and
E-stop on pendant

ShopBot Desktops

Stepper motors are open loop, location not monitored by Control Software.

- Control Box integral to the Desktop.
- Spindle starts spinning when **OK** given to the prompt when starting a part file.

Separate remote pendant . pindle Lockout, soft stop and E-stop on pendant

Cutting Heads: Spindle and VFD or Router

The ShopBot is equipped with a Spindle or a Router as a cutting head

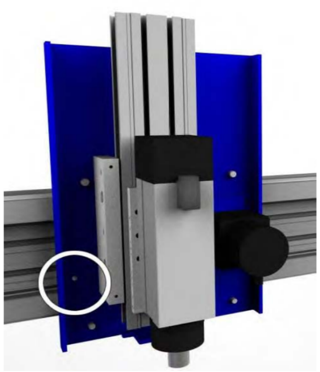
Spindles can be identified by their square shape and lower noise level while running. They have greater precision and more ability to regulate cutting speeds.

The speed of a spindle (RPMs) is controlled through the separate VFD (Variable Frequency Drive).

Routers can be identified by their round shape. The power and speed controls are integral to the router

Production Aids

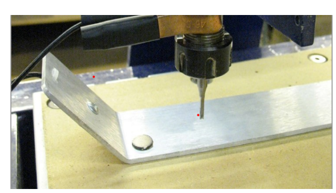
The **Dust Skirt** rides with the head of the spindle or router to help the dust collection system collect waste while cutting. Information on installing the PRS4 dust skirt where brushes are held to polycarbonate base via magnets here: https://www.shopbottools.com/ShopBotDocs/files/DustFootInstall.pdf.



Proximity or Limit Switches and their targets are found on the X and Y axis (Z Prox switch optional).

- Proximity Switches can be used to run an automatic zeroing program to zero the X and Y axis at the lower left corner of the ShopBot Table.
- Tools with a Z axis Prox switch can be set to also Zero the Z axis from the top when using the Z-Zero routine ATC models zero X, Y and Z axes with Prox Switches.
- When engaged in the ShopBot software (**VN**>Limit Switches ON) , Proximity or Limit Switches will stop the movement of the ShopBot when it has reached the extent of its useable travel.

The **Z-Zero Plate** is used to run an Automatic or Custom program to zero the Z axis at the table surface or the material surface, depending upon where it is set in the CAD/CAM files.



Requirements for Routing or Machining

Material

ShopBot is capable of machining:

- Wood, plywood or hardwood.
- Plastics.
- Aluminum and other non-ferrous metals.
- Foam (sheets) for sign making or insulation.

CAD, CAM and Control Software

The first step for machining a part on a ShopBot is to create a part or design on a computer using Computer-Aided-Design (CAD) software (“what and where”.) “How“ to machine it is programmed using Computer-Aided-Machine (CAM) software. The CAM software then generates a ShopBot Part (.sbp) file that the ShopBot Control (SB3) uses to control the cutting head to create the parts.

Software Included with the purchase of a ShopBot is VCarve Pro, a CAD/CAM software.

The ShopBot Control software is included and available for download from the ShopBot website www.shopbottools.com.

More information on using CAD, CAM and Control Software is found in other Chapters of this manual. Tutorials are available on the ShopBot website, and the websites of the CAD/CAM software providers (Example: Vectric.com, Autodesk.com).

Plans to Hold Down Material While Machining. Options Include:

- Mechanical:
  - Screws (location set in CAD file)
  - Jigs
  - Clamps
  - Doublestick tape or adhesive.
- Vacuum
  - Direct vacuum.
  - General vacuum with plenum and bleeder board.

Starting on page 35 in this **Safety and Basic Use** Chapter, there are examples of how to use the CAD/CAM software to safely locate hold down screws in a ShopBot part file, and how to add Tabs to a Profile Toolpath to keep the part attached to the substrate. The separate Chapter on **HOLD DOWN TECHNIQUES** has many examples of different hold down options.

Bits Made for CNC Machining

More information on bits is found in beginning on page 35, and in the **Bit Selection, Feeds and Speeds** Chapter of this Handbook.

Example of Bits in the ShopBot Starter Bit Kit

- **End mill** (flat on the end):
  - Used for general cutting and clearing out area such as pockets or dadoes.
  - Come with straight, up spiral and down spiral and compression options.
  - Tool geometry specific to wood, plastic or metal.
- **Ball nose** (round on the end):
  - Used for 3D carving.
- **V bit**:
  - Used for engraving and carving
  - Changing to a different angle of V bit changes the look.
- **Planing or mortising bit:**
  - Used for re-surfacing the ShopBot Table.

Collets to Hold the Bits in the Spindle or Router

More information on collets, and on **Troubleshooting** is found in other Chapters of this manual.

Collet Nut

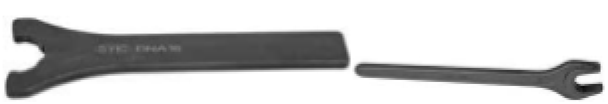
ER 25 Collet (Gantry/Buddy)

ER 20 Collet (Desktop)

ER 32 Collet (ATC)

Porter Cable Router Collet

The ER collets for spindles have two components: the collet nut and the collet itself, the collets for most routers are one piece.



Collet wrenches and collet nut (provided) are used to tighten the collet in the spindle.

Automatic Tool Changers

ShopBot Tools with Automatic Tool Changes have their own specialized instructions for collets/bits, CAD/CAM setup, and ShopBot set up. **See the Assembly Manual for ATC, and the laminated Cheat Sheet for the Automatic Tool Changer (ATC).**

Pausing or Stopping a ShopBot in a Hurry

There are two ways to stop the tool the tool while it is moving or cutting a part file. A “pause” or slow, controlled stop, will decelerate the tool, lift the Z axis to a safe height and stop the router/spindle. An emergency stop cuts power to the tool immediately.

Pause or Soft, Controlled Stop (All Models of ShopBot)

The gentlest way to stop the machine during a cut is to hit the space bar on the keyboard or the STOP button in the Red Position Screen. This will bring the machine to a “slow stop” retaining tool zero locations. Unless it is a dire emergency, this is the preferred manner to pause or stop a file.

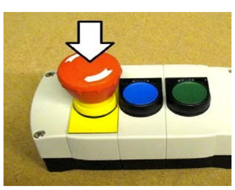
The message at left will appear. Choose “Quit” to stop the file and go back to the main console window, or “Resume” to continue the file from this point. “Insert a Command” allows to operator to change the Move Speed in a file or other select commands before resuming the file.

PRSstandard tools shipped prior to mid 2020 have a single button remote stop pendant that acts as a soft stop. For a true emergency stop, twist the main stop switch on the cover of the Control Box (below.)

Standard Tools shipped starting mid 2020 and Desktop Tools have a pendant that holds the spindle interlock and an Emergency Stop.

Emergency Stop

To immediately cut off the power to the spindle/router and all motors, use the red emergency stop button on the 3 button pendants. Older ShopBots may require turning off the switch on the Control Box. **Caution: An emergency stop is not a gentle process.** Before resuming action after an emergency stop, the operator should check the status of the bit, and re-zero all axes because the locations will no longer be accurate.



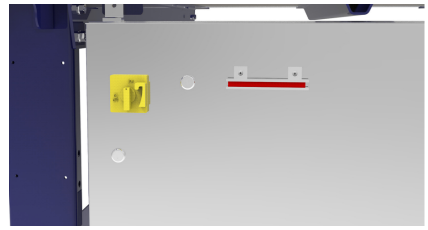
Safety: A Few Basic Rules

1. **Learn and understand safe use of the machine.** Do not allow untrained individuals to operate the machine without supervision. Be aware of the location of the Pause or Stop options at all times.
2. **Eye and ear protection MUST be worn by the machine operator as well as any bystanders or observers.** Flying sawdust, material chips, and other debris can cause serious eye injury. Sustained noise causes hearing loss.
3. **Wear closed-toe shoes at all times.**
4. **Make sure that the bit is properly placed in the collet, and the collet is properly tightened in place.**
5. **Make sure that the material is properly secured before cutting.** Be aware of any small parts that may come loose after being cut out. If a small part catches the edge of a spinning bit, it can be thrown forcefully in any direction, causing injury or damage. A loose part on the surface of the material can be captured by a spinning bit, resulting in fire. Plan hold down to avoid the bit hitting a screw or clamp.
6. **Never place your hands on the rails of the ShopBot.** Be aware that the machine may move unexpectedly in any direction, which can cause serious injury if your hands are in the path of movement.
7. **Never wear gloves while operating the machine.** As with any power tool, a glove can get caught in moving or spinning parts and pull your hand into the machinery.
8. **Tie back long hair and remove loose clothing and jewelry.** If you have an identity card on a lanyard, place the lanyard someplace where it will not be caught in the machine.
9. **Never leave a machine running and unattended.** Understand that a spinning tool generates friction and heat, creating a risk of fire. This risk is minimized by using the correct feeds and speeds (see Bit Selection, Feeds and Speeds), using sharp bits, and by always double-checking files before cutting. If leaving the work area briefly, be sure to Pause the machine or find someone to monitor the area in your absence.
10. **Be prepared to pause or stop the cut if something seems incorrect or unsafe.** If it sounds wrong, it probably is.
11. Keep a working fire extinguisher within reach of the machine.
12. **As operator of the tool, assume responsibility for ensuring that the machine you are going to work on and the surrounding area is safe and configured in a manner you expect and are familiar with.** Monitor visitors to the tool area and ensure that they are also acting in a safe manner.
13. **Shop Etiquette:** Do not talk to anyone while they are working on a piece of equipment. Never leave a machine running while unattended. Leave the work area cleaner than when you arrived. Begin clean up and store your project 10 -15 minutes before the end of your session/work day.

Turning on the ShopBot Control Box

ShopBot Gantry and Buddy Tools

Turn on the large yellow plastic switch on the front of the ShopBot Control Box. The point of the arrow should point up towards “1”.

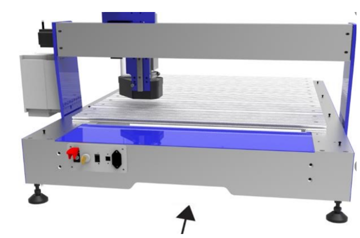


For Alpha ShopBots with a pendant like that shown here, press the blue RESET button on the pendant to engage the motor drivers and provide power to the VFD.

For Standard ShopBots, turning on the Control Box engages the motor drivers and VFD.

If the VFD does not light up when the Control Box is turned on and (if applicable) the RESET button has been pressed, check that the interlock for the spindle is engaged (picture does not show key, only where to inset the key and how to switch to engage.) For the Gantry and Buddy tools with the interlock on the Control Box, a key is attached to the collet wrench.

ShopBot Desktops



The Red On/Off switch is at the back of the ShopBot Desktops.

Older Desktops may have the On/Off switch in front and no pendant.

The Desktop Pendant has spindle interlock key, soft Stop/Start Button, and Emergency Stop button. If the ShopBot does not power up or the VFD does not light up, check the pendant to be sure that the Spindle Interlock key is engaged, and the Emergency Stop is not engaged. Finally, check on/off switch on bottom of VFD.

Starting the SB3 ShopBot Control Software

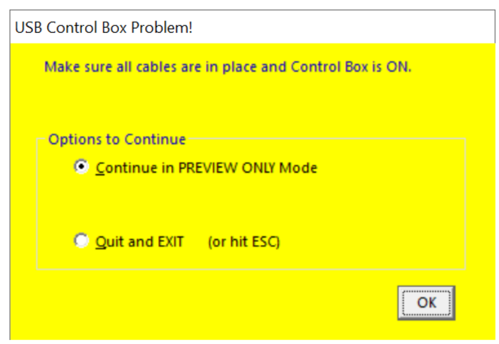
Instructions on how to load the ShopBot Control software are found in Section 10 of the ShopBot Assembly manual or in the Desktop Quick Start Guide.

- Turn on the ShopBot Control Box (above).
- Turn on the ShopBot computer (if it is not already on).
- Click on **ShopBot3.exe** icon to launch the Control software.

The red ShopBot Position Screen will appear.

Troubleshooting ShopBot Not Connecting

If a yellow screen like that shown below appears, the computer has not connected to the ShopBot. If the computer running the ShopBot is not hooked up to an actual ShopBot, Continue in Preview Mode. In Preview Mode, the user can explore the software and run previews of files on the screen. If this is the first time attempting to connect to the ShopBot, after checking the following suggestions, see the Advanced Troubleshooting links below.



- Confirm that the ShopBot Control Box is turned on.
  - For Alpha tools, try pressing the blue Reset Button on the pendant again.
  - Retry Connection.
- Check that both ends of the USB cable from the ShopBot Control Box to the computer are firmly plugged in.
  - At the computer end, make sure the cable is plugged into a USB 2.0 port (no blue line above port).
  - If plugged into a hub, make sure nothing else is plugged into the hub, especially the Spindle Speed Controller.
  - Retry Connection.
- Is an E-Stop/Remote Stop button on (#4 input flashing red on Alpha tools)?
  - Release the red button(s).
  - RESET (Alpha tools).
  - Retry Connection.

Advanced Troubleshooting: https://youtu.be/f5TAtIZHJTE

www.shopbottools.com > Support resources > FAQ

The SB3 ShopBot Control Software has Two Modes:

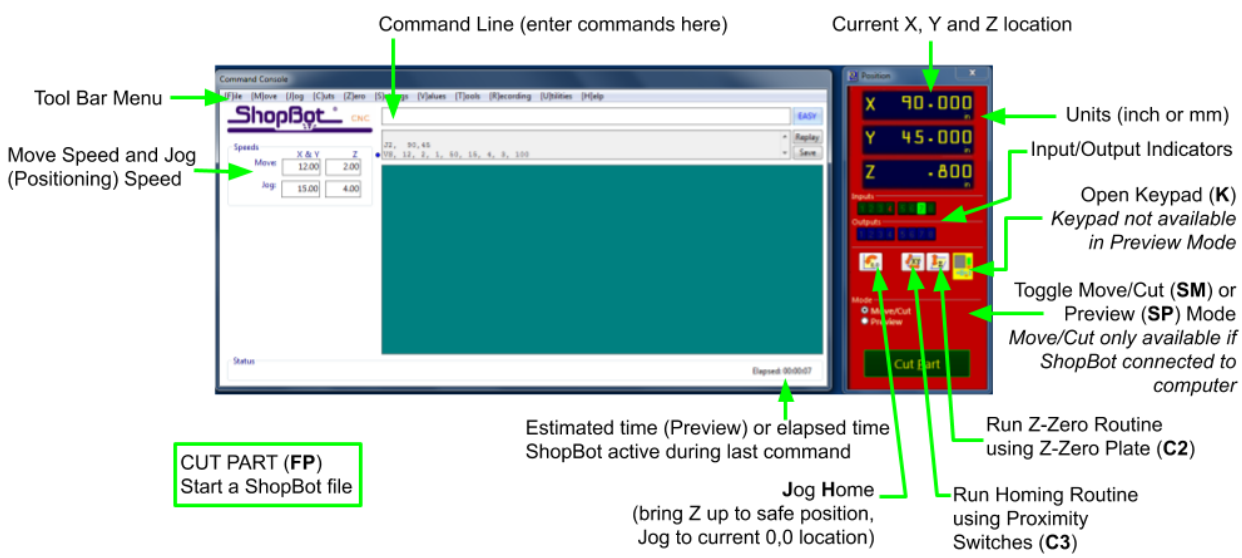
“ShopBotEASY” shows only the necessary functions to set up and run a part file.

“Full mode” shows all available functions and settings.

If necessary, switch to FULL mode by clicking on the Help (?) options button in the red position window and then **Switch to FULL**.

Full Display Overview

Icons and 2 Letter Commands



Working with the Spindle or Router

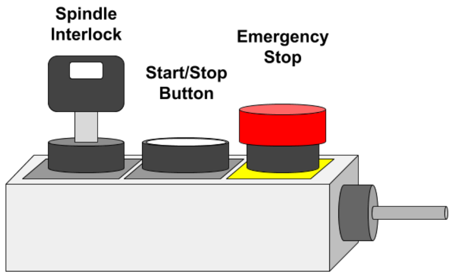
To maximize the life of the spindle bearings, the spindle should be warmed up before cutting at the beginning of each day (or after letting it sit idle for longer than 4 hours).

If the power to the Spindle or Router is run through the ShopBot Control Box (standard set up for current ShopBots) then the ShopBot Control software will give a prompt to start the spindle or router at the appropriate time when running a ShopBot Part (.sbp) file. It is also possible to run the spindle/router manually for warming it up, or for making simple cuts with the Keypad Control.

Preparing for Running The Spindle

- Turn on the ShopBot Control Box.
- Press the **Reset Button** on the Pendant (ShopBot alpha tools only).
- Launch the ShopBot Control Software and ensure that it is in **Move/Cut** mode.
- Ensure that the Safety Interlock is engaged (Older ShopBots may not have a Spindle Safety Interlock).

For Gantry and Buddy tools, the Interlock may be on the front of the Control Box. The key is attached to the Collet Wrench.



For Desktop tools and some Standard tools, the Spindle Interlock is on the Pendant. The keys are separate from the collet wrenches. Keep them in a safe place if removed from the pendant.

If working with a Spindle: Look at the face of the VFD to be sure that there is power to the VFD

If there is not a red display on the face of the VFD

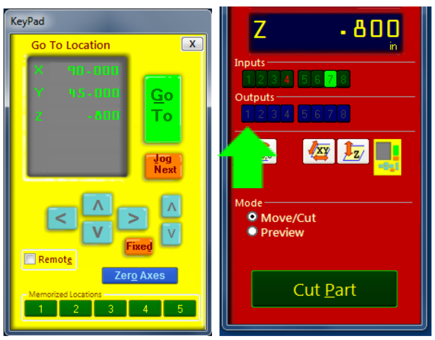
- Is the Control Box active? E-stops disengaged and Reset button hit?
- Is the Interlock engaged?
- On Desktops, is the power switch to the VFD (at the bottom) turned on?

If working with a Router: make sure the power switch on the router is on.

Starting The Spindle/Router Manually

For the Spindle or Router to turn on, the #4 and the #1 Outputs must be turned on. To run the spindle/router outside of a part file:

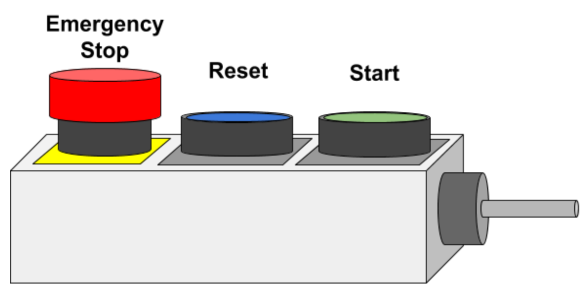
- In the main console window, type **K** or use the yellow icon to bring up the Keypad control.
- Notice that output #4 lights up in the Red Position window.
- Click on the #1 output to turn it on. This will create a prompt to start the router/spindle.
- **For PRS Alpha models only**: The green START button on the remote stop pendant must be pressed to activate the Spindle.
- For all models, press **OK** on the prompt to start the router/spindle or the part file.



Opening up KeyPad Control turns on the #4 output.

Click on the #1 output to get the prompt to start the spindle or router.

- For Standard and Desktop ShopBots, clicking on **OK** at the prompt should start the spindle or router.
- Listen to be sure that the spindle is actually spinning.
- If it is not spinning, Stop or Pause the file (hit the space bar), check above steps, and retry.



Alpha models only: press the **start** button on the pendant to start the spindle when the prompt appears.

- Listen to be sure that the spindle is spinning.
- If it is spinning, respond to the prompt to start the file.
- If it is not spinning, **cancel** at the prompt, check above steps including interlock engaged, and retry.

The safest way to turn the spindle/router off is to click on the output 1 switch again.

Setting the RPMs (Speed) of the Router/Spindle

See the Chapter on **Bit Selection, Feeds and Speeds** for detailed information on what values to choose when setting the Spindle or Router speed while running a part file.

ShopBot Models with a Spindle Speed Controller

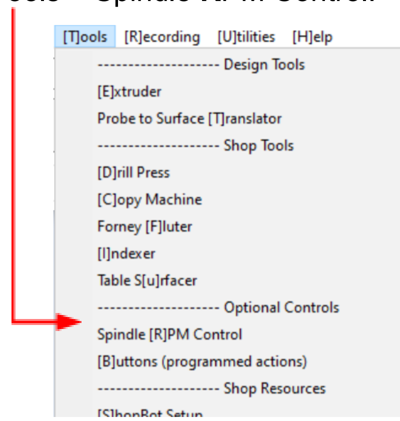
Determine whether the model of ShopBot has a Spindle Speed Controller. There will be a cable that comes from the VFD and is plugged into the Computer (ShopBot Gantry and Buddy Tools).

In models with a Spindle Speed Controller, the Control Software controls the speed of the spindle.

For best results, plug the Spindle Speed Controller into an empty USB port (2.0 or 3.0) in the computer, and not into a hub.

To change the spindle speed, open the Spindle RPM Control in the software:

TR: **T**ools > Spindle **R**PM Control.



Hint: to have the Spindle RPM screen open when the ShopBot software starts, Right Click on the RPM in the upper left corner and change Start when SB3 Starts from 0 to 1. Click OK to save the settings.

To change the spindle speed, enter the new RPM value in the box, then press **Enter.**

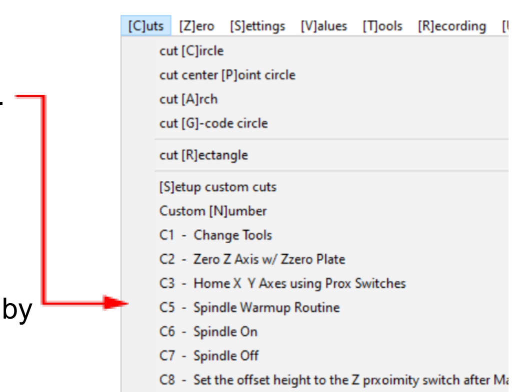
There is a line of code in ShopBot Part Files that sets the Spindle Speed for models that have a Spindle Speed Controller.

Example: TR, 12000.

Warming Up ShopBot Models with a Spindle Speed Controller

Automatic Warm Up Routine:

C5 will run a routine that takes the spindle through a 15-minute warm up at multiple RPMs, then turns it off.



Manually warm up the Spindle:

- Turn on the spindle with Keypad and #1 input.
- Set spindle RPM's to 8000 for 2 mins.
- Increase RPM's by 2000 at 2 min intervals.
- After running at 12K for 2 min, turn off spindle by turning off #1 input.

Desktop Models and Other ShopBots without a Spindle Speed Controller

If the ShopBot model does not have a Spindle Speed Controller (Desktops and some older ShopBot models), then the operator must control the RPMs of the spindle at the VFD.

The VFD on ShopBot Desktops usually displays Frequency rather than RPMs. Look for the control knob on the VFD which dials in the Frequency. To determine the RPMs, multiply the frequency displayed by the standard Hertz (Hz) for the country. North America uses 60 Hz, other countries may use 50Hz or 60Hz as the standard.

Desktop Spindle Settings

Frequency to RPMs for 60Hz
(US, Canada, and Mexico. May apply to other countries):
Example: 200 x 60 = 12000 RPMs.

Frequency to RPMs for 50Hz
(Europe and most other regions):
Example: 200 x 50 = 10000 RPMs.

Display	Spindle RPM	Display	Spindle RPM
300	18000	300	15000
266	16000	266	13300
233	14000	233	11650
200	12000	200	10000
166	10000	166	8300
133	8000	133	6650
100	6000	100	5000

Warming up ShopBot Models Without a Spindle Speed Controller

While the ShopBot Control software will run through the Automatic Warm Up Routine triggered by using **C5**, the spindle will stay at the RPMs set at the VFD for the entire program.

Manual warm up of the Spindle:

- Turn on the spindle with Keypad and #1 input. Press **OK** or **Enter** to start spindle.
- Set spindle RPM's to 8000 for 2 mins.
- Increase RPM's by 2000 at 2 min intervals.
- After running at 12K for 2 min, turn off spindle by turning off #1 input.

Display of Frequency of Desktop Spindle VFD. Use knob to change Frequency.

RPM Display of Gantry or Buddy Spindle.

When set to Local (Lo/Re) mode, use arrows to change RPM.

ShopBot Models with a Router

For Routers:

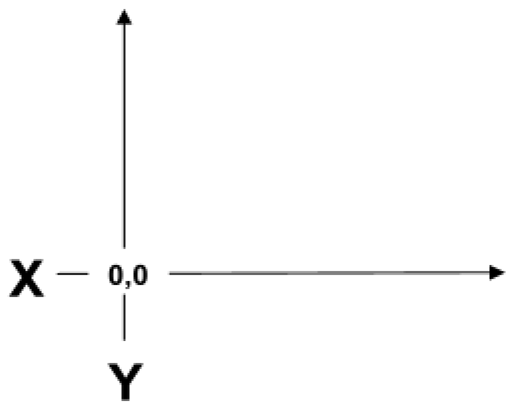
- Check that the power on the router is turned on.
- Use the dial on the router to set the speed (A router tends to have less torque with lower speed settings).
- There is no need to warm up a router before using.

Getting Oriented to the ShopBot

The movement of all digital fabrication tools (laser cutters, 3D printers, CNC machines) is based on Cartesian geometry (think: plotting.) While the physical location of the X and Y origin (0,0) differs from tool to tool (even within ShopBots), all machine code (G-code or ShopBot code) is a series of commands telling the digital fabrication tool where to go from 0,0, and what to do when it gets there.

A Review of Cartesian Geometry

- The X axis runs right to left.
- The Y axis runs away and towards you.
- Where X and Y cross is 0,0.
- Every point on a ShopBot table has an X and Y value.
- Convention for CNC tools puts X and Y 0,0 in lower left corner of table.
- The Z axis (not shown) plunges into or rises above material.
- ShopBot commands are a basically a series of “move to: X, Y, Z” commands (as measured from 0,0,0).



Understanding the Orientation of the Different Models of ShopBot

Remember that the movement of ShopBots is based on knowing where the conventional origin (0,0) is, and which are the X and Y axes. The examples below indicate which is the “FRONT” of each model of ShopBot, as defined by which axis is the X axis.

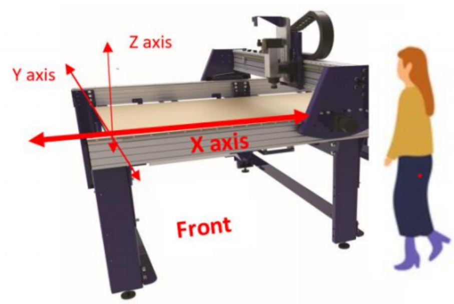
All CAD/CAM files will assume the X axis runs right to left, and the Y away and towards the operator, so it is important to stand at the true front of the ShopBot in order to have the files run correctly. When setting up material or stock, pay attention to which is the X axis in the design or CAM file.

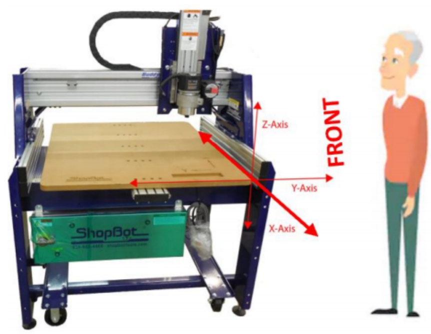
The Automatic XY zeroing routine using the Proximity Switches (page 28) will set the zero on the X axis first, then the Y axis. Setting the X and Y origin manually starts on page 27.

The FRONT is Defined as the X Axis for That Model of Shopbot

ShopBot Gantry Tools (96” x 48”)

- Common size of a gantry tool will fit a 4’ x 8’ sheet of plywood.
- The X axis is along the rails (96”).
- The Y axis is across the gantry (48”).





ShopBot Buddy Tools

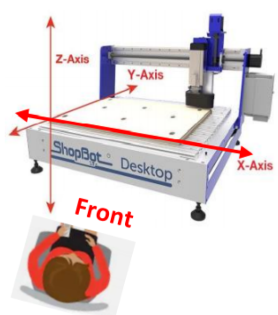
(Factory Configuration, no PowerStick accessory)

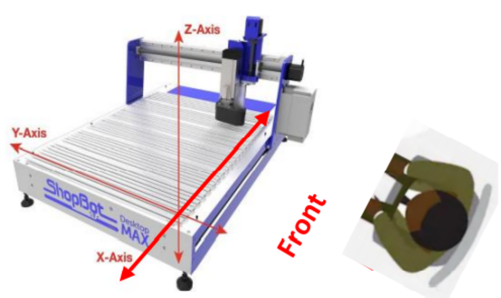
- The X axis is along the rails. Table has 24” of travel.
- The Y axis is across the gantry (32” or 48’).
- The X and Y axes can be switched (will change the location of 0,0 and the Front of the tool).

https://www.shopbottools.com/ShopBotDocs/files/SwitchingXYAxesonBuddy.pdf

ShopBot Desktop (24” x 18”)

- The X axis is along the gantry (24”).
- The Y axis runs front to back (18”).





ShopBot Desktop Max (36” x 24”)

- The X axis is along the rails (36”).
- The Y axis is across the gantry (24”).

Setting the ShopBot to Move in Inches or mm

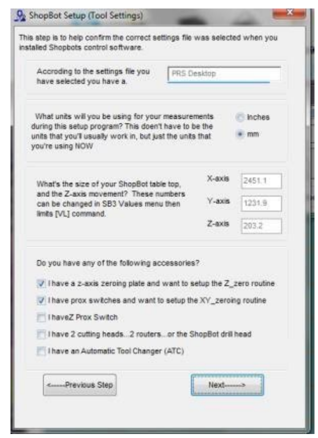
The Units (inches or mm) that the ShopBot is set to is shown under the position on the red Position screen.

For International Customers

- Windows software must be set to English US.
- ShopBot Control Software expects the decimal symbol to be a period (.) not a comma (,).
- Set design and toolpathing (CAD/CAM) software to mm (more information in later sections).
- Save ShopBot code in mm (example: ShopBot TC (mm) (*.sbp).
- Set ShopBot to mm.

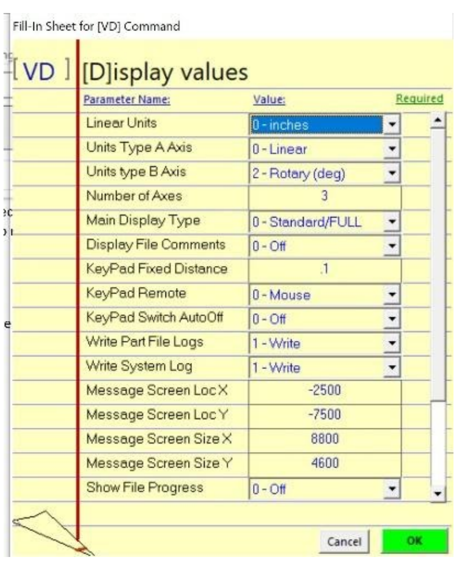
If the ShopBot part (.sbp) created in the CAM software does not match the units the machine is set to, a warning message will pop up when the file is loaded.

To change the Units (inches or mm), use **TS** (**T**ools ShopBot **S**etup) from the ShopBot Control software menu:



Note: By paging through the screens, one can:

- Confirm that the ShopBot .ini file has the correct size ShopBot.
- Set the units that the ShopBot is running in (inches or mm).
- Alter the value of the thickness of the Z Zero plate and set the Z Zero routine in the correct units.
- Set up the automatic X and Y Zero routine in the correct units.



One can also use the **VD** (**V**alue **D**isplay) command from the ShopBot Control software to:

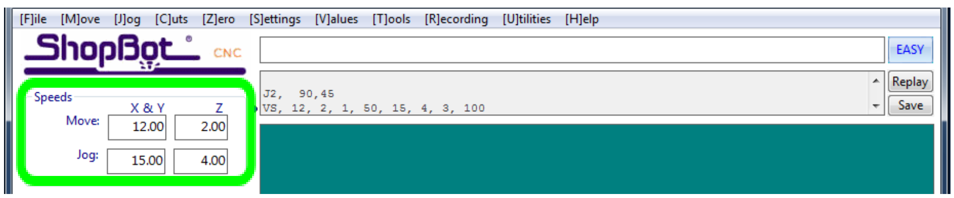
- Set the ShopBot in inches or mm.
- Change the number of axes shown on the Position Screen (Example: 3 axes for regular ShopBot, 5 axes for Rotary Indexer or 5 axis tool).

Setting Move and Jog Speeds

The Move Speed is the speed at which the ShopBot will cut through (X&Y) and plunge into (Z) the material. **M**ove **S**peed (**MS**) is set by the operator for manually moving the tool around using Keypad Control, or as a line of code in a part file generated by CAM software.

The **J**og **S**peed is the positioning speed between cutting moves. There is no load on the ShopBot when positioning itself for the next cut, so it can move faster. The operator sets the Jog Speed.

Move and **Jog** speeds are displayed on the left side of the ShopBot Control Console. Values are expressed as **inches/second** or **mm/sec**.



Knowing at what speed to cut through the material and at what RPM to set the spindle is an important skill. Generally, to keep the bit cool and get better cuts, the waste generated by machining should be larger “chips” rather than finer sawdust. See the Chapter on **Bit Selection, Feeds and Speeds** for more details on suggestions for setting optimal move speeds. A **Chip Load Calculator** found under the **Tools** menu in the ShopBot Control software.

Change the Move and Jog speeds used for Keypad Control or manual moves by:

- Typing a new value directly in the box circled above or
- Using the ShopBot toolbar menu to select the Fill-In sheets to change the:
  - **M**ove **S**peed [**MS**] or **J**og **S**peed [**JS**].
  - Both Move and Jog speed at the same time [**VS**].

Note: The gantry ShopBot and Buddy can jog at 6 – 12 ”/sec in the X&Y and 4”/sec in the Z.

Set the Jog Speed for the Desktop no faster than 4”/sec in the X&Y and 2”/sec in the Z.

In each .sbp file, regardless of the CAM software used to generate the ShopBot part file, there is a line of code that sets the Move Speed and overrides values entered manually. Use **FE** or any text editor to open a .sbp file and read the code. The MS values in the .sbp file can be edited to change the Move Speed for that file (example MS, 4, 1) without using the CAD/CAM software.

In VCarve Pro/Aspire, the Move Speed is set in the Tool Library. See sections about using VCarve Pro/Aspire CAD/CAM for more information.

For information about using G-Code with ShopBot, including setting feed rate, in the ShopBot Control Software, see **H**elp> **U**ser’s Guide.
Information on G-Code and ShopBot starts on page 49

Moving the Cutting Head Around the Table

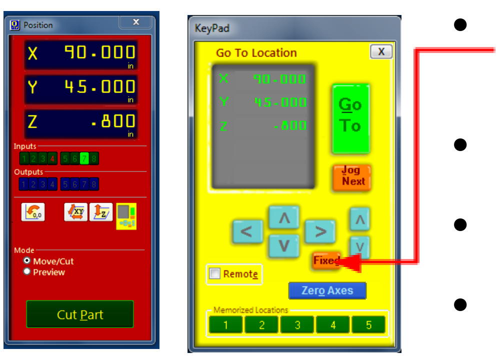
When the Control Box is on, and the drivers engaged, one should not be able to physically push the gantries to move the ShopBot around the table. (Note: If there is movement in an axis when the Control Box is on and the drivers engaged, see **Troubleshooting and Maintenance** for information on tuning up the ShopBot.)

Use Keypad Control (below) or ShopBot Commands to move the cutting head around the table.

(FYI: If the Control Box is off or drivers not engaged, it is possible to push the ShopBot around. Be careful and push slowly if the pinion gears are engaged in the rack to avoid damaging the motor drivers in the Control Box. Remember to re-zero all axes after moving the ShopBot so the computer knows where the ShopBot is.

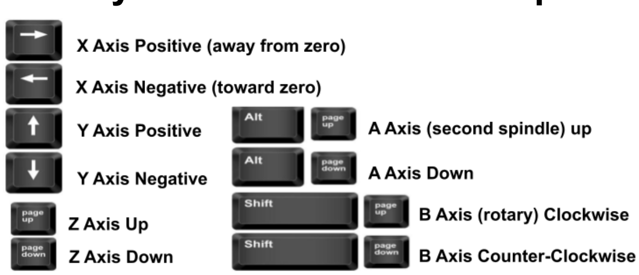
Keypad Control of the ShopBot

- Set Software to **Move/Cut** mode (Red Position Screen.) Keypad will not open in Preview.
- Press **K** on the keyboard or use Keypad icon on the Red Position Screen to open Keypad.
- **Stand at the Front of the ShopBot model and orient the computer so that the left/right arrows align with the X axis on the ShopBot.**
- Where the computer thinks the location (distance from 0) is indicated on Position screen.
- For now, don't worry about where the computer thinks the head is, just move the tool around using the Keypad Control. Do be mindful of moving too close to edge of travel to avoid faulting a driver (alpha tool) or hearing the grinding noise which indicates that the ShopBot has tried to travel beyond its limits.
- Pages 25 – 27 describe how to "Zero" the ShopBot.
- To move to a specific location, type value in Keypad's Go To Location box, press GO TO.
- Close the Keypad by pressing **ESC** or the X in the Upper Right Corner.



- To move the ShopBot in small increments, click on the orange **Fixed** button and enter the increments in the box that pops up when the Fixed button is active.
- In Fixed Mode, every time an arrow is pushed, the ShopBot will move that increment. ESC to exit Fixed.
- To move diagonally, place the cursor in yellow space between two arrows and hold the left mouse button.
- To move at Jog speed, hold the Ctrl button down while pressing the arrow keys.

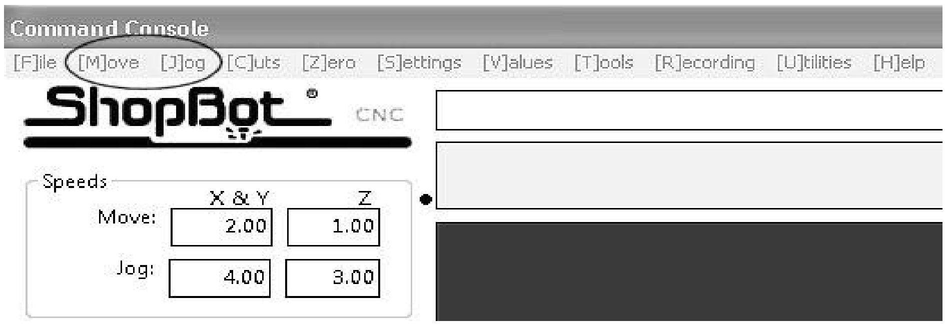
Using the Computer Keyboard to Move the ShopBot



Using ShopBot Commands to Move or Jog the ShopBot

The ShopBot Control software uses the **Move** (cutting speed) and **Jog** (positioning speed in between cuts) commands to send the cutting head to an exact location that is the absolute distance from 0,0,0. See Pages 27 – 29 for how to “Zero” the ShopBot.

Put the Cursor in the Command Line and Type **M** to pull down options for which axis/axes to move. Then, type in the values of the new location. When entering the locations for multiple axes, the order is always X, then Y, then Z. Use a comma to delineate the values for each axis. Press **Enter** to execute the command.



EXAMPLES:

- Type **MY 3** to Move the Y axis to 3” from Y=0.
- Type **M2 3,3** to Move the X and Y axes simultaneously. The result will be a diagonal move to put the head at 3” from the X=0 and 3” from Y=0.
- Type **M3 4,4,.5** to Move the X and Y axes to 4” from X,Y =0,0 and the Z axis to .5” above Z Zero. Before doing this, check where the computer thinks Zero on the Z is, and ensure that the Z will not plunge into the table or travel too high and top out!!
- NOTE: in the above screen shot, the X&Y Move speed is set to 2 (inches per second) and the Z plunge set to 1 (inch per second) This is a pretty slow speed for a ShopBot gantry tool.
- Take a few minutes to check out the other options under the Move and Jog menus Example: **M**ove the ShopBot to Home Position (**MH**).

The **Jog** command is similar to the Move command

- Type **JX** 5 on the keyboard to Jog the X axis at positioning speed (fast) to 5” from the X=0 position.
- Type **J2** 5,5 to Jog diagonally so that both the X and Y axes end up at 5” from the X,Y = 0,0 position. Use a comma to separate the X and Y locations.
- NOTE: in the above screen shot, the X&Y Jog speed is set to 4 (inches per second) and the Z plunge set to 3 (inches per second).
- **JH** will bring the Z up to 1”, then jog the head back to the 0,0 location defined in the Control Software. If Z is above 1” already, it will not move the Z down (See **Hidden Gems** for how to define Safe Z height).

NOTE: Think carefully before typing in a J3 command – the router bit could crash into the table and break the bit or top the Z out beyond the limits of its travel.

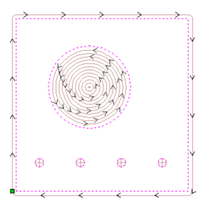
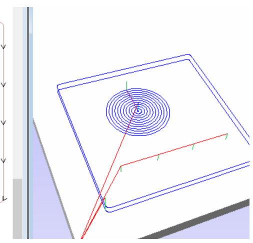
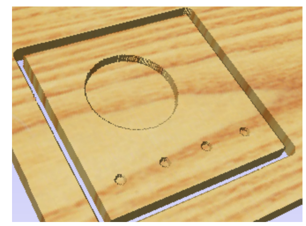
Overview of Bit Options

A later Chapter (**Bit Selection, Feeds and Speeds**) and tutorials on using VCarve Pro/Aspire CAD/CAM software have more information on types of bit to use, and the appropriate feeds and speeds for different materials and toolpathing options. The following defines general categories of router bits, and generic information on how to change a router bit.

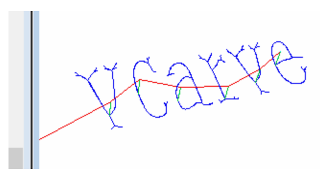
Router Bits Fall into 3 Main Categories

End mills for cutting tool paths that plunge, then stay at one depth

- Profile to cut out parts
- Pocketing
- Drilling

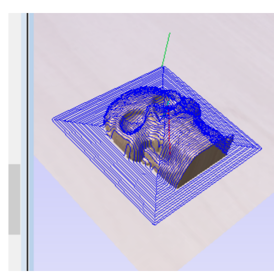
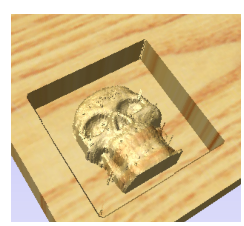





V bits for engraving or V carving


Ball nose bits for 3D carving



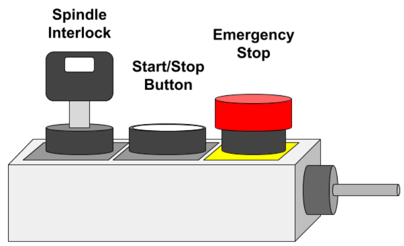



Within each of those categories, the actual geometry of the bit will vary depending upon the type of material and the desired results/quality of the cut.

Inserting a Router Bit Into the Spindle or Router

Prepare the ShopBot to Change a Bit

- Move the head of the ShopBot so that the spindle/router is over the table. The idea is that a bit coming out of the collet will fall onto the table, not the floor.
- Bring up the Z so that there is room to get access to the bit/collet.
- Pull down on the dust brush to remove it from the assembly (Older models of dust skirt may require dropping or removing the entire dust skirt).
- For Gantry and Buddy tools, disengage the collet wrench/key from the Control Box interlock.
- For Desktop Tools, disengage the interlock on the pendant.



Prepare the Collet and Collet Nut

Make sure that collet and nut are completely clean. A dirty collet will not grip the bit properly and can cause it to slip during the cut. Use a piece of paper, a wire brush, compressed air, and/or mineral spirits to remove dust or excess grease; https://youtu.be/6gMezORPi8M

For spindles only: Before inserting the bit, press the collet into the nut and listen for the “click.” Confirm that the collet is snapped in by holding the nut upside-down and letting the collet hang freely.

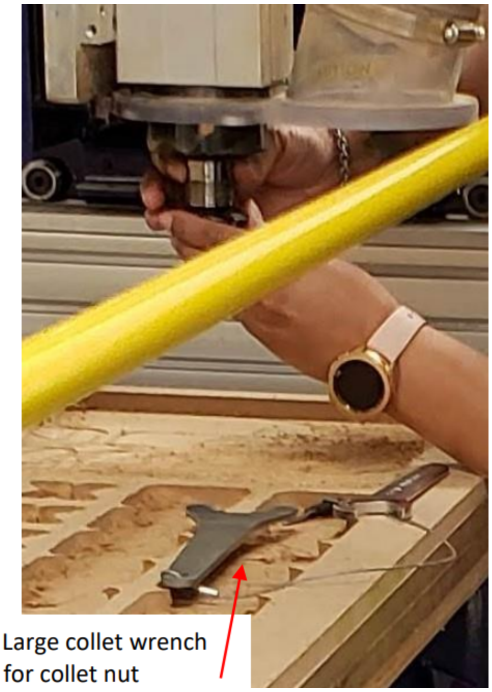
For routers only: The router collet and nut are held together by a retaining clip. It is not necessary to disassemble the components for cleaning or bit changes.

Slide the bit into the collet so that the shaft (the smooth part) fills up at least 75% of the collet. Check that the collet grips only the shank of the bit, keeping any parts of the machining or flute outside of the collet.

For spindles: snap the collet into the nut, then flip it over to test that collet is snapped in securely. Flip assembly over again and insert bit into the bottom of the assembly (shown).

With a one-part router collet (left) or a two-part spindle collet (right), all machining on bit must remain outside of the collet to avoid pulling chips up into the collet. For V-bits, expose .25 -.5” (3 – 6mm) of shaft.

Threading the Collet/Bit onto the Spindle


Large collet wrench
for collet nut

- Using two hands, thread the collet/collet assembly onto the router or spindle. It should go on very easily. If there is any resistance, do NOT apply more force. Stop, back the collet off and try again until it threads on smoothly.
- Once the collet is finger-tight, check the bit to make sure it is in the correction position in the collet.
- Use the included wrenches to fully tighten the nut.
  - For spindles, the larger collet wrench (key attached for Gantry/Buddy tools) slides onto the collet nut from under the collet.
  - The smaller collet wrench slides into the slots on the spindle.
- Righty-tighty/lefty loosey of the collet nut (bottom).
- Think “monkey tight” not “gorilla tight”.
- Gently tighten the bit into the collet, then give it a little extra tug with the wrenches.

https://www.shopbottools.com/training/tutorials has several videos on changing bits. Scroll down to the section labelled ShopBot Desktop: QuickStart Videos.

After Changing the Bit

- Engage the Lockout key after changing the bit so that the spindle will turn on when the file is started.
- If the ShopBot Control software was placed in Preview Mode during the bit change, return it to Move/Cut before actually starting the file.
- Remember to Zero the Z after a Bit Change (page 29).

Maintenance of Router Bits and Collets

Want to make bits last longer?

- Avoid dropping bits or collets on the concrete floor. When changing a bit, move the head of the ShopBot over the table so that the bit falls on the table rather than onto the floor.
- Inspect the bit and clean it before use. If it is damaged, think about getting a new one.
- Use a good, clean collet.
- Tighten the bit into the collet correctly: Not too tight, not too loose.
- Read the Chapter on **Bit Selection, Feeds and Speeds** for suggestions on setting feeds and speeds for keeping the bits cool while machining.
- Avoid hitting a hold down screw or clamp.

Collets have a working life of 400 – 700 hours if nothing untoward happens to them. See Chapter on **Troubleshooting and Maintenance** for more information on collets.

Registering or Zeroing the ShopBot to the Computer

Before starting to cut parts, the ShopBot Control software must be registered to the actual origin (0,0,0) of the tool or CAD/CAM design. This is often called “Zeroing” the ShopBot.

To start, verify that you understand which is the FRONT of the tool (page 18.) If the computer is aligned correctly to the ShopBot itself, pressing on the right arrow key will move the X axis to the right.

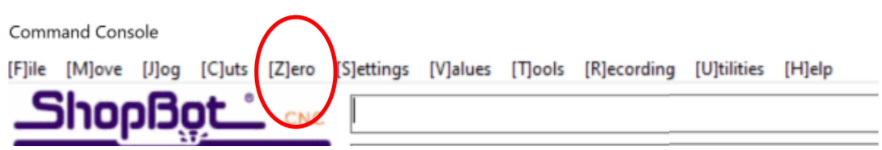
There are two techniques for Zeroing the tool:

- “Manual”: Move the head to the desired zero location, then use a ShopBot Command to set that location as zero in the ShopBot Control Software. Using a **Zero** command does not change the physical location of the bit/cutter head, but tells the ShopBot Control Software to set the current location as 0.
- Run the Automatic Zero Routines that use the Proximity Switches for the X & Y axes and the Z-Zero Plate for the Z axis.

Recommended: Treat the X and Y axes separately from the Z axis. The X and Y Origin (0,0) should stay consistent for all the part files in a given run. The Z axis will need to be zeroed every time the bit is changed or if a new sheet of material is put down (Z set to top of material in CAD/CAM file).

The “Manual” Technique: Keypad Control and ShopBot Commands

- In the first, “manual” technique, use the Keypad to move the head to the desired Zero location. Hold down the arrows consistently and avoid “bopping” the arrows in short bursts
- Use the arrow keys on the yellow Keypad screen, or the arrow keys on the computer keyboard, to move the head to the desired location
- To move the Z, use the arrows at the far right of the Keypad screen or Page Up and Page Down on the computer keyboard.
- To move the head in small increments, enter Fixed mode (orange Fixed button) Enter the increments for the tool to move each time an arrow key is pushed in the box below the Fixed button
- Once the head is at the desired location, click on the blue Zero Axes button. In the screen that pops up, select which axes to Zero. Note how the number in the Red Position screen reads 0 for the axes that have been zeroed



The “Automatic” Zeroing Routines:

“Automatic” or “Custom” zeroing routines:

- Use Proximity Switches and Targets to zero the X and Y axes. In some set ups, a Prox switch will Zero the Z axis also.
- Use the Z-Zero Plate to Zero the Z axis.
- Icons for Zeroing are found on the Position screen.
- Alternatively, two letter commands to run the Custom zeroing. routines are found under the Cut menu on the toolbar.

NOTE: The icons on the Red Position screen are not visible if the yellow Keypad Control screen is active.

Zeroing the X and Y Axes Using the Proximity or Limit Switches

A video on how to use the Proximity Switches to set the X and Y zero can be be found on the ShopBot website. https://www.shopbottools.com/training/tutorials > Desktop: Quick Start.

- Use the **XY** icon or **C3** from the Toolbar menu to run the Custom program that uses the Proximity Switches on the ShopBot to Zero the X and Y axes at the lower left corner of the Table. The ShopBot will move each axes to its respective Proximity or Limit switch target, then back off until the router bit is at 0,0 *
- Even if the desired 0,0 location is not in the lower left corner of the ShopBot Table, it is a good idea to use the automatic zeroing routine to zero at that corner, then move the router head to the correct location using Keypad Control or an M2 or J2 command. Once in the correct location, use the Zero button on the Keypad Control screen or **Z2** to Zero the X and Y axes at that location.

** If the **C3** program does not Zero the X&Y at the correct location, take a few minutes to set up the ShopBot correctly. For instructions on how to set up the ShopBot Zeroing routines, open the ShopBot Control software and use the **T**ools **S**hopBot Setup option. Page through the screens until the **XY zero** routine comes up. Click on “Make It Easy on Me” to run the set up routine for the homing routine to zero the X and Y axes*

Jog Home (JH) vs Home X & Y Axis using Prox Switches (C3)

- The icon on the left = **J**og **H**ome. This command raises the Z to a safe height (usually 1”), then sends the bit to the 0,0 location, wherever it is currently defined in the software.
- The icon on the right = **C3**: Home X & Y axis using Prox Switches. This command runs the zeroing routine to set 0,0 at lower left corner of table.

Zeroing the Z Axis Using the Z-Zero Plate:

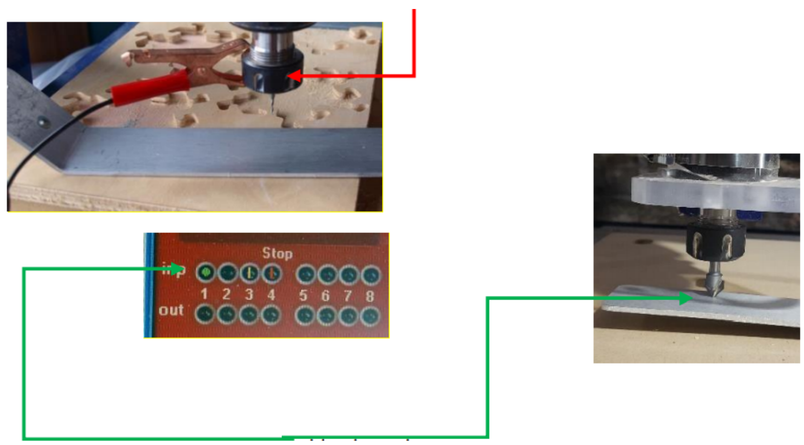
REMEMBER: The Z axis must be Zeroed every time:

- A different bit is inserted in the spindle.
- New material is laid down if the CAD/CAM files set the Z-Zero at the surface of the material.
- The Z has traveled too high and the driver faulted or stalled at the top ("nih-nih-nih" noise).
- The E-stop button on an Alpha tool triggered or the main power button turned off (Desktop).

A video demonstrating the use of the Z-Zero plate can be found at https://www.shopbottools.com/training/tutorials >Desktop: Quick Start.

Prepare to run the Z-Zero Routine with the Z-Zero Plate

Clamp the alligator clip to the tab on the spindle, the router bit, or the collet nut.



- Touch the Z-Zero plate to the router bit and look at the computer screen to make sure that the Number 1 input light is green. **If the #1 input is not lit up, there is not a complete circuit between the Z-Zero plate and the Control Box, and the Z-Zeroing routine will not work properly.**
- Place the Z- Zero plate on whichever surface set in the CAD/CAM file (top of material or on the machine bed.) Is the plate is under the bit?
- On the computer, click on the **Z-Zero icon** or type **C2.**
- Press **OK** if the Z-Zero plate is under the bit. The bit will touch Z-Zero plate twice, take the thickness of the plate into account, set the Z axis to zero , then move up to safe Z position.
- When the Z-Zero routine is complete, remove alligator clip and put away the Z-Zero plate/clip.

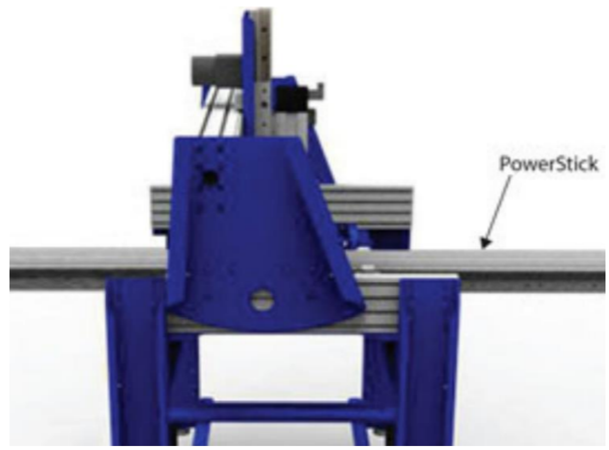
NOTE: The default value of the thickness of the Z-Zero plate is set to .121". To test that value, after running the Z-Zero program and storing the plate, send the Z to 0 (MZ,0) If the bit is not sitting on the material/table, measure the true thickness of the plate, and change the value in **T**ools **S**etup.

Staying Within the Physical Limits of ShopBot:

Each ShopBot has a Nominal Range of Travel, and an Actual Range of Travel:

ShopBot Gantry Tool

- PRS 9648 will take a 4' x 8' sheet of plywood.
- The rails are 120" long (10').
- The gantry can actually travel ~105" before it runs out of rail (X axis).
- The extra range of travel in the X can accommodate the tool rack for an ATC.
- The YZ car can travel 49 – 50" before it runs into the sides of the gantry (Y axis).
- The Z might have 8" of travel, but actual travel is dependent upon how many layers of support board, thickness of the material, length of the bit, etc.



ShopBot Buddy Tool

- The Buddy has either a 32" or 48" gantry.
- The ZY car can travel ~33" or ~49" before it runs into the sides of the gantry.
- Travel along the rails (X axis) is 24". The power stick is longer than 24" to support movement of the power stick on rollers under the table.
- The factory mounted base is larger than the available cutting area of the Buddy.
- Suggestion: Mount a spoil board that reflects the actual travel of the Buddy.
- The Z travel is same as the Gantry tool.

What happens when the ShopBot is sent to a location that is beyond its actual travel area (**or the E-stop is activated**?) It depends upon whether the model of ShopBot is a Closed-loop or Open-loop system.

Alpha Tools: The location of the ShopBot is constantly being reported back to the computer (closed loop motor/drivers).

If the ShopBot **Alpha** can't physically go to a location in one axis, it will cause ALL the motor drivers to fault. A prompt will state that the ShopBot has lost location, and the user needs to reset the drivers and re-zero the ShopBot. Example: Z travels too high, faults at the top. #4 input on position screen turns red, prompt comes up saying drivers have faulted. Physically, all drivers are off, so ShopBot can be pushed by hand. The weight of the Z itself might cause it drop down (drivers no longer holding it in position).

Recovering from a Driver Fault

- Press the Reset button on the pendant.
- Suggested: Close down ShopBot Control software and reopen.
- Re-zero all axis.
- Determine why the ShopBot tried to travel beyond its limits If necessary, rewrite CAD/CAM and .sbp files to avoid future faults.
- See below for options to turn on Proximity Switches in the SB3 ShopBot Control software to avoid future driver faults.

Gantry/Buddy Tools with Standard Control Box, and Desktops

The computer is constantly sending information to the Control Box, but the Control Box is not sending information back to the computer (Open loop motors/drivers.)

When the ShopBot reaches its physical limits, the computer continues moving towards the location, but the ShopBot itself has stalled. Listen for the dreaded nih-nih-nih sound. No damage is done to the ShopBot, but the location of the ShopBot as shown on the Red Position Screen does not have any relation to the actual location of the cutter head. This will cause a problem for the next cut in that axis.

Example: The ShopBot code has sent the Z to 5.5”, but the Z physically topped out at 5.25” above the material. Now, there is a .25” discrepancy between the actual Z location and the computer location.

In the next cut, the Z will plunge too deep into the material by that .25” discrepancy.

Recovering from Motors Stalling

- Rezero the axis where the motor has stalled. See note below for whether to rezero all axes
- Determine why the ShopBot tried to travel beyond its limits. If necessary, rewrite CAD/CAM and .sbp files to avoid future faults
- See below for options to turn on Proximity Switches in the Control software

NOTE: If the motor stall happens in the X or Y axis, but the design falls within the travel limit of the ShopBot, the ShopBot will “self-correct” and continue the part file in the new location. If that is acceptable, DO NOT rezero the X and Y axes using the Automatic Zero Routine (C3) at this time. See previous section for how to change the XY zero location/routine.

Correcting Issues That Contribute to “Over Travel”

Physical set up of ShopBot. Zero the ShopBot, then **Move** the ShopBot to the extent of that axis. If the ShopBot will not move to that location, adjust Zero routines and/or targets to ensure full travel for that ShopBot. Use **TS** to modify Automatic Zero Routines, and the Assembly Manual for physical tool set up.

ShopBot Control Software determines some routines, like how high the Z travels after running the Z-Zero routine using the Z-Zero plate. The default value for the “Safe Z Pullup” is 1” for Gantry/Buddy tools, and .5” for Desktop tools. Use **VC** to change the Safe Z Pullup in the Control software.

Aids to Avoid Faulting Drivers or Stalling Motors

Proximity Switches

- The Proximity or Limit Switches were introduced in the section on using the Automatic or Custom XY Zeroing routine **(C3).**
- When running the Zeroing routine, code turns the Proximity Switches ON while running the routine, then returns them to whatever status they were in before the routine.
- The default setting of the ShopBot Control software is to have the limit or proximity switches OFF. Turn them on to have the prox switches stop the tool before it reaches a mechanical stop. **VN** > Limits ON.
- Why keep the Limit Switches off? In some cases, things like fluorescent lights can cause the Limit Switches to trigger falsely. In this case, it's best to leave them off in the software.
- If the Limit Switches don't seem to be physically working, contact ShopBot support to troubleshoot and correct the issue.

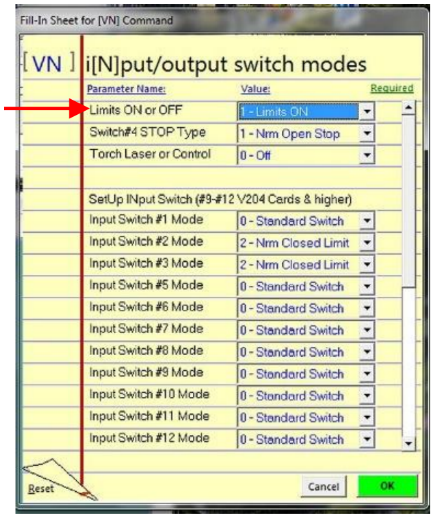
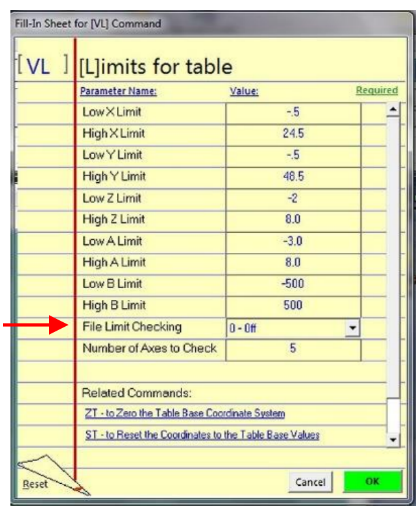
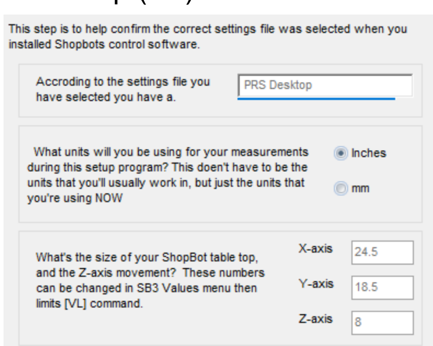


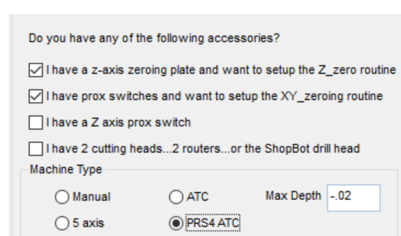
Table Limits

- The SB3 Control Software also has the ability to check that the file falls within the stated limits of the table. Turn on File Limit Checking using the **VL** command.
- The Table Limits are set when the .ini file is loaded for the model of ShopBot. They can be checked and adjusted using the Tool Setup (**TS**) command.



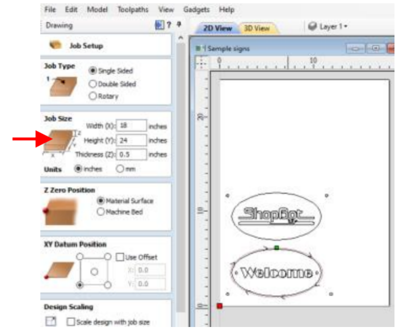


- In some cases, like setting up for an ATC, the Table Limits are turned on automatically. If the ATC will not travel low enough in the Z (example, surfacing the ShopBot table), use **TS** to adjust the value here.

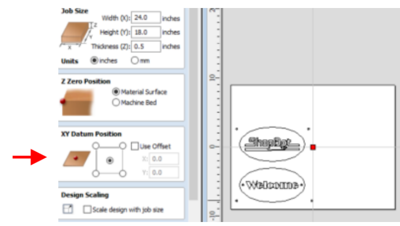


Design and Toolpathing Software (CAD/CAM) Settings

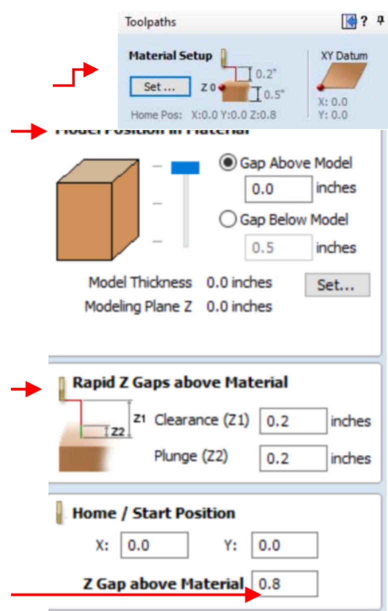
- **Total material area and material layout.**
  - In the example to the right, the material is set up for a ShopBot Desktop with a cutting area of 24” by 18”.
  - The material will fit inside the Desktop cutting area HOWEVER, the orientation of the material is incorrect.
  - The problem is in the Y axis. If the material is set up as 24” in the Y, the tool is incapable of traveling that distance. Pay attention to the FRONT of the tool.



- **Origin of X and Y.** Does the X and Y origin match where the machine is set up?
  - In the example to the right, the origin is set to center of the material.
  - If the ShopBot itself is set up with the XY 0,0 in the lower left corner of the tool, then the tool won’t be able to move into the lower and left quadrants of the design.
  - Either move the XY 0,0 location on the tool, or change the CAD/CAM file and .sbp file.



- **Z Zero Position.** Does the Z origin match where the machine is set up?
  - In the example to the right, the origin is set to the Material Surface.
  - If the actual Z is zeroed to a location other than where the CAD/CAM file is written, the bit will either float above the material, or cut too deep.



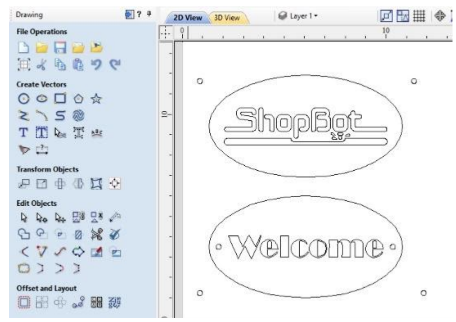
- **Safe Z and Home Z:** Is a setting in CAM causing the Z to travel beyond its physical ability?
  - Access **Material Setup** on the Toolpaths (CAM) side
  - Not relevant for 2D CAM. If doing a 3D carving, where does the model fit inside the material?
  - Rapid Z defines how far the Z is moving above the material when the ShopBot is moving to a new position.
  - Travel too high can make the Z top out; even if the Z doesn’t top out, an .sbp file takes longer if Z traveling higher than it needs to. Travel not high enough can run into clamps or other obstacles.
  - Home/Start Position is how far the bit travels above the Material at the beginning and end of the file VCarve Software will correct the values if Home Z is not high enough to clear material (example: Z Zero set to table and material is 1” thick).
  - Setting from previous VCarve file will be retained in new VCarve file, so Z home might have to be adjusted to avoid topping out Z in a new design file.

Intro to Using CAD/CAM to Design a Part File

This page gives an overview of the steps to get from idea through to creating a part file for a ShopBot. While there are step by step instructions on the flash drive, the Design File (.crv) and the ShopBot Part (.sbp) files shown are also included on the flash drive so you can practice cutting a part file without having to design and toolpath it.

This example uses VCarve Pro CAD/CAM software included with the purchase of a ShopBot. The example is heavy on the CAM side to address concerns that are important when using a CNC machine. Other helpful Chapters in this Handbook include **Bit Selection, Feeds and Speeds, Hold Down Techniques** and **Projects and Techniques.**

Lay Out the Design Using the Computer-Aided Design (CAD) Side of VCarve Pro/Aspire:

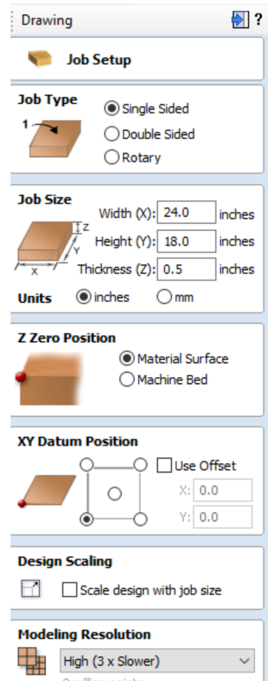


- Define the size and orientation of the material.
- Lay out the location of two ovals in the material.
- Put text in the ovals.
- Create markers to indicate where it is safe to put **HOLD DOWN** screws to hold the material and/or the ovals in place while machining.

Use the Computer-Aided Machining (CAM) software to create the toolpaths. The CAM:

- Defines the toolpath operations.
  - "Drill" for Hold Down markers.
  - "VCarve or Engrave" for text.
  - "Profile" to cut out the ovals.
- Defines the bits used for each of the operations (End mill and V-Bit). Information in the **Tool Library** for each bit defines:
  - Diameter of the bit.
  - Pass depth (determines number of passes).
  - Stepover (mostly relevant for pocketing or 3D carving).
  - Spindle speed for ShopBots with Spindle Speed Controller (a reminder for others).
  - X and Y Move speed in inches/sec (or mm/sec).
  - Z Plunge rate in inches/sec (or mm/sec).
  - Defines the tool number of the bit (used for ATC or to signal a bit change).
- In a Profile Toolpath, additional **Hold Down** options can include adding Tabs to keep the part attached to the substrate until the operator is ready to remove the parts.
- The CAM software also generates the code for the ShopBot to use to machine the parts.

Getting Started with VCarve Pro/Aspire



In a 2D CAD software such as VCarve Pro, the first step is to define the size and orientation of the **material** that will be placed on the ShopBot. Keep this in mind and review the orientation of the various models of ShopBots. Once the material is defined, where parts are laid out on the virtual material is where they will cut out on the ShopBot.

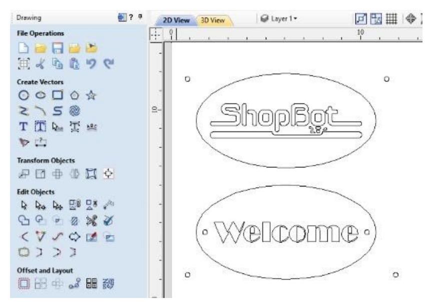
NOTE: When using a 3D rendering CAD software such as Fusion 360, the material (stock) size and X, Y and Z origins may not be defined until CAM set up.

Use Job Set up to define the Material

- Job Type: “Single Sided” for this example.
- Define the size of the material in the X, Y and Z axes.
- Define the origin of the Z Axis (top of material, or top of table surface).
- Define the origin of the X and Y axes. Lower left corner is conventional for CNC machines.

Overview: Part Design and Layout

Using the Vector Creation and Alignment tools on the CAD side of VCarvePro:



- Two Ovals are placed on the material.
- Text is created and aligned inside each of the ovals.
- After the ovals and text are in place, small circles are added as markers for where it is safe to put hold down screws: Four screws outside the ovals hold the material in place Two screws inside the bottom oval hold it in place.

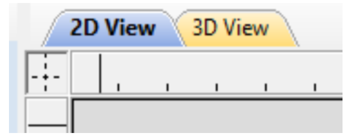
Overview: How to Machine the Parts

Using the CAM side of the V Carve Pro software,

3 different types of toolpaths are created:

- The locations of the hold-down screws are marked with the Drill option, using a ¼” end mill bit.
- The lettering is set to be **V-carved (engraved)** with a 60 or 90 degree V bit.
- The ovals will be cut out using a **Profile** toolpath set to the outside (¼” end mill bit).
- The oval with no screws inside will have tabs added to the profile toolpath to hold it in place.

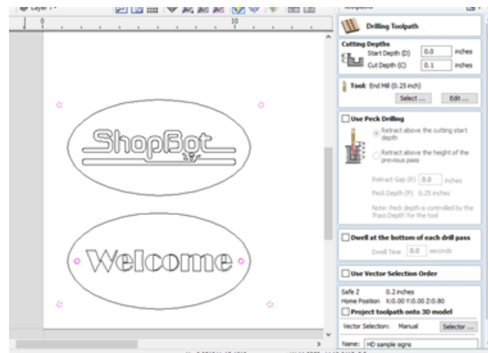
The Details of CAM



NOTE: In VCarve Pro, the default setting is that, after each toolpath is created, the software will switch to 3D view.

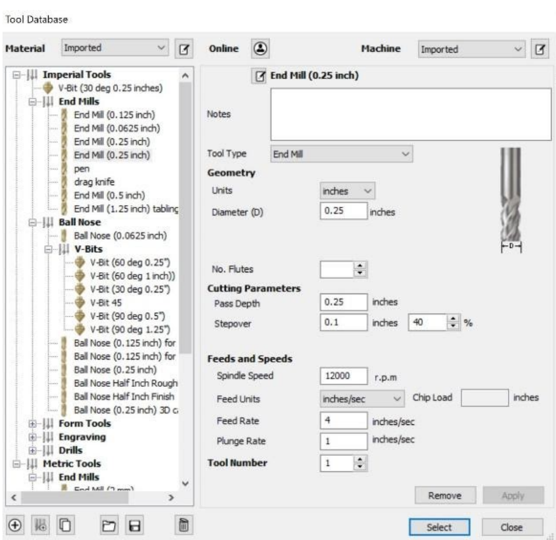
Since all toolpathing is done while in 2D view, switch back to 2D view to continue creating toolpaths.

"Drill" Toolpath to Mark the Hold Down Locations.



The Drill routine is a good option for making a shallow mark to indicate where to put a hold down screw. The bit plunges straight down and then pulls back up, so there is no lateral pressure on the material to move it out of place. If the material is large enough, there is no need for additional hold down while running the Drill toolpath.

- Select the circles that mark where it is safe to put a hold down screw (hold shift key and click on each circle).
- Open the **Drill** Toolpath.
- Leave start depth at the top of the material (0.0).
- Cut depth is how deep to plunge the bit (.1").
- Click on **Tool Selection** to open the Tool Library.



Select a ¼" End Mill bit from the list.

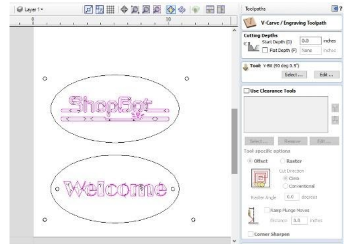
Enter/confirm data in the Tool Library.

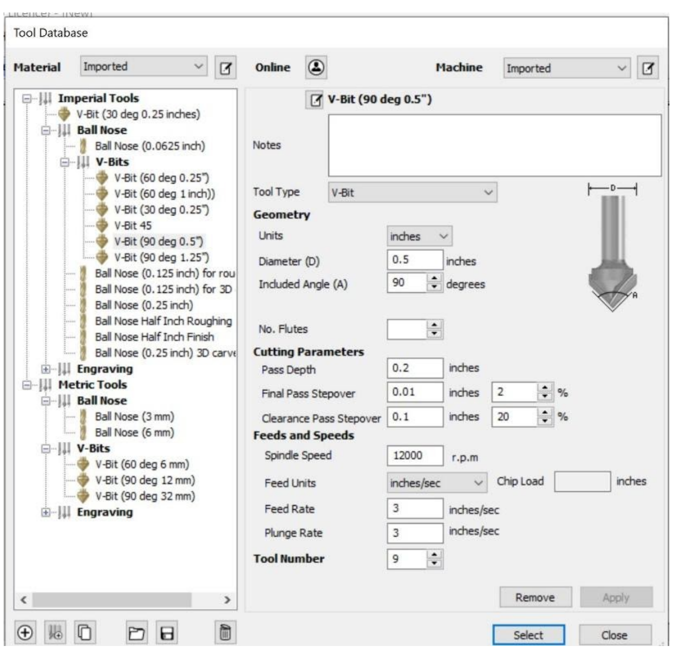
- Diameter of the bit is .25".
- Pass depth = diameter of bit (pass depth is the deepest the bit will go in any one pass).
- Spindle speed set to 12000.
- Feed rate set to 4 inches per second (3 ips for Desktop).
- Plunge rate set to 1 ips.
- Tool number set to 1 (User defines tool number For tools with a single spindle, use numbers 1 – 18).
- **Select** the tool.

No other options need to be selected for the Hold Down toolpath. Give the toolpath a name that is easy to find, **Calculate** and **Close.**

“VCarve/Engraving” Toolpath to Engrave the Letters:

- Select the Text inside the letters.
- Open the **VCarve/Engrave** Toolpath.
- Start depth default to material surface (0,0).
- Do not click on Flat Depth.
- **Select** the Tool.





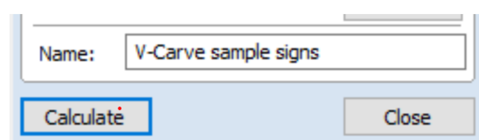
Select the V bit included in the ShopBot starter bit kit available.

(usually 60 V for big ShopBots, 90 V for Desktop Tools).

- Diameter of the bit is .5” for 90v or 1” for 60 V bits included in bit kits.
- Pass depth ~ ½ diameter of bit.
- Spindle speed set to 12000.
- Feed rate set to 2.5 or 3 ips.
- Plunge rate set to 2.5 or 3 ips.
- Tool number set to 6 or 9. User defines tool number Suggestion: use something easy to remember (example: 9 = 90 degree V) For tools with a single spindle, use numbers 1 – 18).
- **Select** the bit.

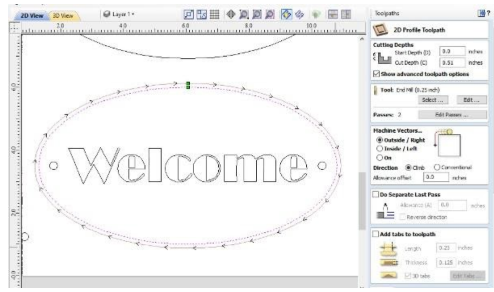
No other options need to be selected for the VCarving text toolpath. Give the toolpath a name that is easy to find (example: VCarve sample signs).

Calculate and **Close.**

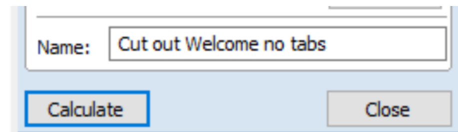


“Profile” Toolpath to Cut Out the Ovals

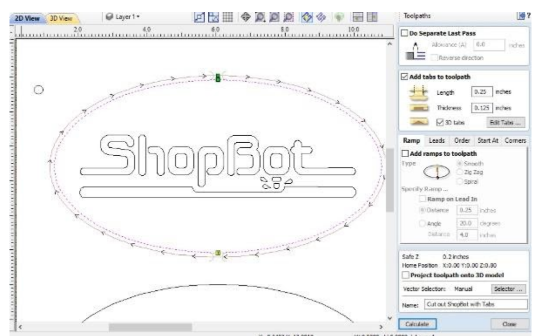
Since one oval has hold down screws inside, no additional hold down is required.



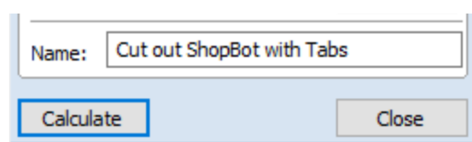
- Select the oval with the hold down screws inside.
- Open the **Profile** toolpath.
- Start depth default to material surface (0,0).
- Set the Cut Depth = to material thickness +.02”.
- Select the same End Mill bit used for the Drill.
- In order to get the ovals the correct size, the toolpath must travel **outside** the vectors to compensate for the waste from the bit.
- No other options are necessary.
- Give the toolpath a name that is easy to remember (example: Cut out Welcome no tabs).
- **Calculate** and **Close.**



The other oval does not have any hold down screws inside, so it may shift or fly out as the ShopBot completes the cut. This oval uses all the same settings as above, except that TABs will be added to the toolpath to hold the part in place.



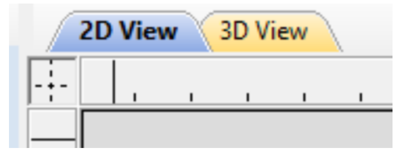
- Select the oval with no hold down inside.
- Open the **Profile** toolpath.
- Keep all the settings for cut depth, tool, etc.
- Click on **Add Tabs** and Define.
  - Length of tabs.
  - Thickness of tab.
  - Click on 3D tabs.
  - Edit Tabs to determine # of tabs.
- Give the toolpath a name that is easy to remember (example: Cut out ShopBot with Tabs).
- **Calculate** and **Close.**



Previewing the Toolpaths in VCarve Pro/Aspire

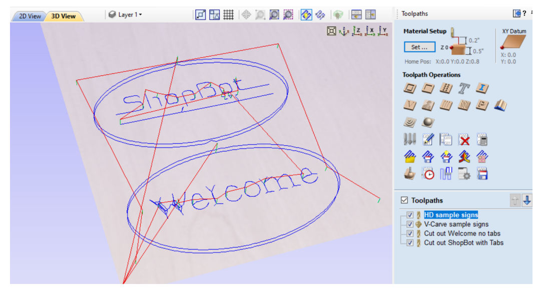
Toolpaths can be previewed and simulated on screen before generating ShopBot code.

If there are edits to be made, it is best to make them before generating the code and machining the parts.

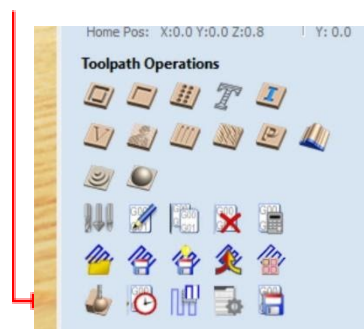


Move to the 3D view.
In 3D view, the movement of the tool can be observed.

- This example view is set to isometric (upper right corner).
- All the toolpaths are checked and visible.
- Red lines mark the positioning movement of the tool (Jog).
- Blue lines mark the cutting of the tool. Notice 2 passes for the oval cut out, and tabs in the upper oval.
- Use the mouse to rotate the view or the icons in upper right corner of screen (next to isometric) to select a given view.
- If the pass depth is too deep, or not deep enough, it should be visible here.



In 3D view, the toolpaths can be simulated on screen.



- In this example, all the toolpaths have been simulated.
- This example view is set to isometric.
- Use the mouse to rotate the view or the icons in upper right corner of screen to select a given view.
- Control + mouse allows for pan.
- Where the blue background shows through is where the toolpath cuts through the material.
- Notice the tab at top and bottom of ShopBot oval that keeps oval attached to substrate.

Generating the ShopBot Code

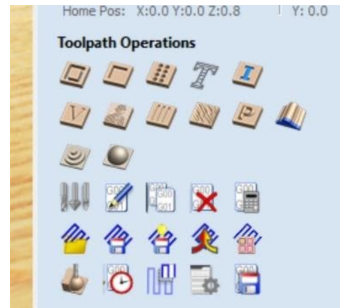
The ShopBot Control software runs ShopBot Part (.sbp) files to:

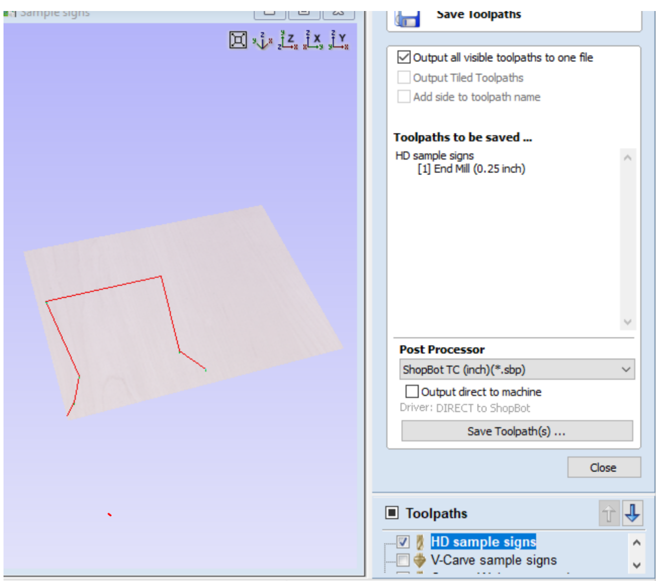
- Move the spindle/router head around the ShopBot CNC in the X, Y and Z axes.
- Set the Move Speed in the X and Y axes, and the Plunge Speed in the Z axis.
- Turn the spindle/router on and off at the appropriate time.
- For models of ShopBot with a Spindle Speed Controller, the software controls the RPMs of the spindle.

Use the **Save Toolpath** Icon to generate the ShopBot Code.

The first .sbp file will be the markers for Hold Down.

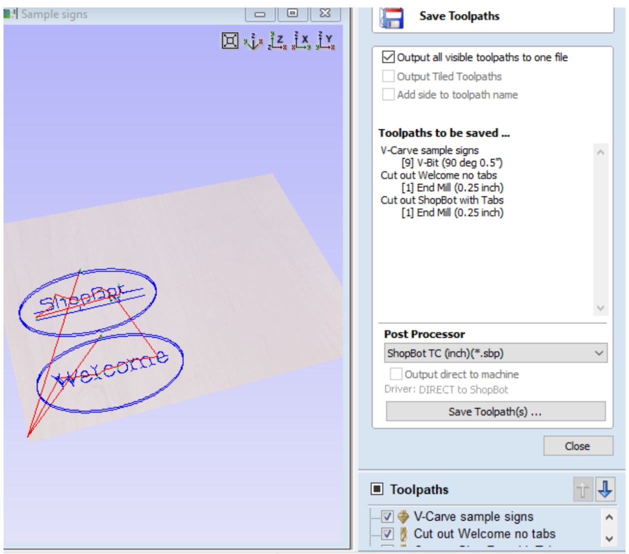
- The Hold Down file is always run first, and on its own.
- The Hold Down file uses the ShopBot to mark where it is safe to put the Hold Down Screws.
- After running the Hold Down file, move the head out of the way, then screw the material down to the table.





- If **Output of visible toolpaths to one file** is checked, the toolpath(s) checked below will be put in one file.
- Put a check in the box next to the HD sample signs toolpath.
- Verify that the correct toolpath is listed here.
- Choose the **Post Processor** Use Down arrow to choose between ShopBot TC (inch) or ShopBot TC (mm).
- **Save Toolpath** to a location that can be found later.

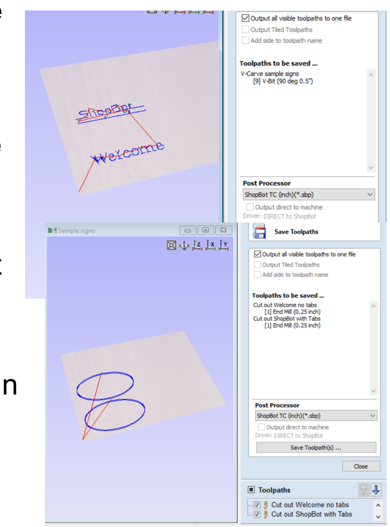
Creating one big .sbp file that includes changing a bit within the file



The **ShopBot TC Post Processor** allows for a bit change during a project.

- In this project, the text uses a V bit, while the cutouts use a ¼" end mill.
- If all 3 toolpaths are put into the same .sbp file (and the tool numbers in the tool library are different), the SB3 Control Software stops the tool & spindle, signals that there is a bit change, and asks about Zeroing the Z after the bit change.
- The order that the toolpaths are saved in is determined by the order at the bottom. Change order with blue arrow.
- ShopBotters with an Automatic Tool Changer can put all toolpaths into a single .sbp file, and the ATC will pick up the new bit at the appropriate time.

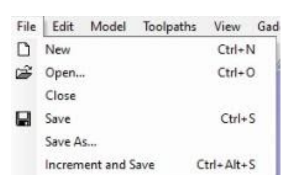
Question? Is it better to create several .sbp files that each use a single bit rather than create one big .sbp file that includes a bit change?



- While there are many safety measures that prevent the spindle from starting while changing a bit, in public maker spaces and schools, it might be safer to put toolpaths that use different bits into their own .sbp files. It is easy to start a new file after the bit has been changed.
- If any one of the files have to be rerun (for example, the ovals didn't get cut all the way through), it is easier to adjust and rerun the one file rather than having to run them all again.
- In the examples to the right, the text using a V bit is put into its own .sbp file. Both toolpaths for cutting out the ovals use the same ¼" end mill bit, so they are put into the same .sbp file.

NOTE: If the toolpath is changed in the CAD/CAM file, remember to rewrite the .sbp code also.

Saving the CAM/CAM (.crv) File

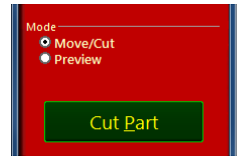


- Saving the toolpath only saves the ShopBot Code.
- Remember to save the actual Design file itself (.crv) every time an .sbp file is saved.
- Mantra: Save on the right (toolpath), save on the left (design file).

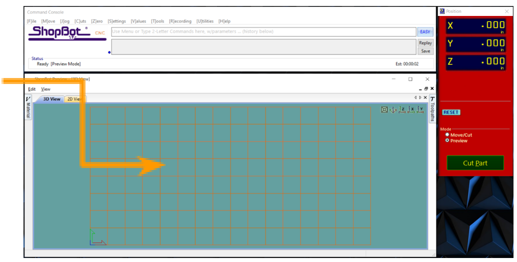
Preview Files in SB3 ShopBot Control Software:

Set the ShopBot Control Software to Preview Mode to visualize ShopBot Part (.sbp) files on the screen before actually cutting them out, and to get an estimate of the time required to cut a part.

- **NOTE:** If the operator is not sure which file to run, previewing a file onscreen can help to avoid running the wrong .sbp file.
- If no ShopBot is hooked up to the computer running the software, the software will default to the Preview mode.
- When a ShopBot is connected to the computer running the software, one can toggle between **Preview** and **Move/Cut** settings.

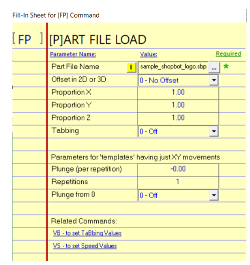


When in Preview Mode, the Preview Screen is visible
The grid size displayed is determined by the values set up by the .ini file chosen when the Control Software was first loaded. Check **TS** (**T**ools > **S**hopBot SetUp) to see the ShopBot model set up in the Control Software.

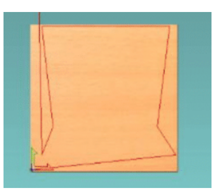


To Preview a file on screen (toolpaths only):

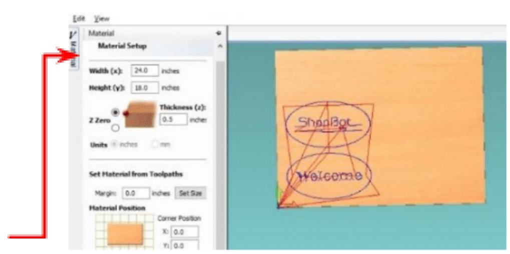
- Click on **Cut Part** or enter **FP** in the Command Box on the ShopBot Command Console.
- Select the correct .sbp file.
- When the FP Fill-in Sheet appears, accept the defaults by pressing the **Enter** key on the keyboard or using the mouse to click on **Start.**



If the size of the material has not been entered before the file is previewed, the toolpath sets the size of the material.

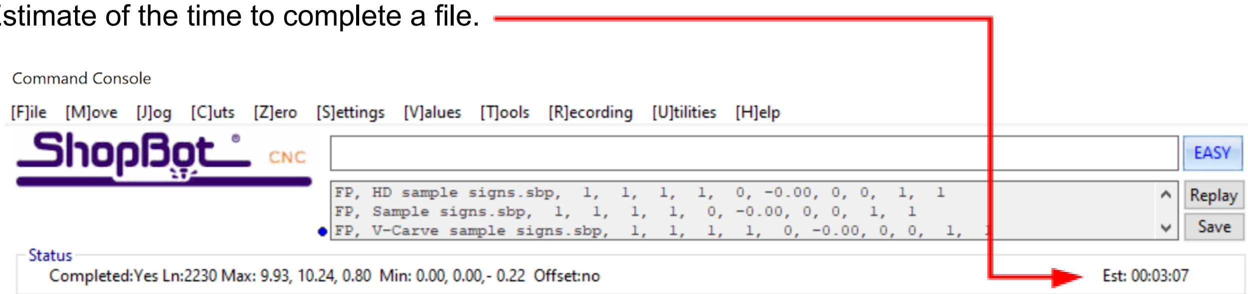


To get a more accurate preview of the part files on the material, enter the size of the material using the Material tab.



Multiple part files can be run in the same preview To clear the preview, open the **Toolpaths** tab on the right side of the Preview screen.

Estimate of the time to complete a file.



“Air Cut” a Job

During an Air Cut, the bit moves above the material in the X, Y and Z axes, but the bit does not plunge into the material. This gives the operator an opportunity to correct a mistake in the design file or the set up before any actual cuts are made. **CAUTION**: If attempting an Air Cut with an ATC, pick up the correct bit before starting the Air Cut. Do not try to pick up a new bit while performing an Air Cut.

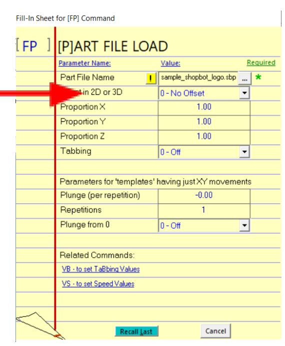
To do an Air Cut, select the 3D offset option when running a file. When 3D offset is selected, the ShopBot Control software will treat the **current location** of the router bit as if all 3 axes were at 0,0,0 . If the X and Y axes are at 0,0 and the Z axis is 1 inch above the material, the router bit should float above the **material** instead of cutting into it. As soon as the file is confirmed, **Pause** the Air Cut by hitting the space bar, then Quit the file.

Preparing for running an Air Cut

- Place the material and Zero the X & Y axes in the location defined in the CAD file.
- Set the ShopBot Control Software into **Move/Cut** mode.
- **Important**: Jog Home (**JH**) so that the bit is at the X and Y 0,0 location.
- Set the Z so it is at least 1” above the material.
  - In this example, the Z 0 is set at the material surface in the CAD file, so the bit is 1” above the material.
  - If the Z 0 is set at the table surface in the CAD file, move the Z up so that it is 1” above the material (but won’t top out).

Running the Air Cut

- Click the **CUT PART** button or type **FP.**
- Select the correct .sbp file.
- At the Fill-in sheet for the FP command, click on the down arrow by Offset in 2D or 3D and choose 3D offset.
- On the Red Position Screen, click the **Start** button, or just hit **Enter.**
- Starting the spindle is optional during an Air Cut.
- Click OK to start the gantries moving. **Pause** if there is a problem or whenever the file has been confirmed to the operator’s satisfaction.
- **WATCH THE JOB!** To **Stop** or **Pause** the file without having to re-zero everything, use the **Spacebar** on the computer keyboard, or the **STOP** button on the ShopBot Position screen.
- The gantries will stop, the spindle shut off, the bit will pull up and out of the material.
- Follow directions on the pop-up screen to Quit or Resume.
  - If all is well, Quit the file, then reload it to start the real cut. No need to JH for real cut.
  - The **FP** fill-in screen defaults to No Offset, and the file is selected and ready to go.
  - If all is not well, make adjustments, then Air Cut again.



Troubleshooting an Air Cut

In 3D Offset, the ShopBot Control Software treats the current location of the bit as if it were 0,0,0. If the X and Y axes were not at 0,0 when the Air Cut was initiated, the location of the cut on the material will be wrong. **Pause** (Spacebar) and **Quit** the file, then Jog Home (**JH**). Move the Z down to 1” above the material so it doesn’t top out during the Air Cut. Start the Air Cut again.

Check List for Running an .SBP File (Ready to Cut)

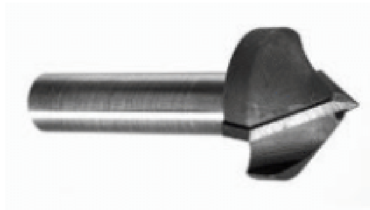
Ready to try a ShopBot file?

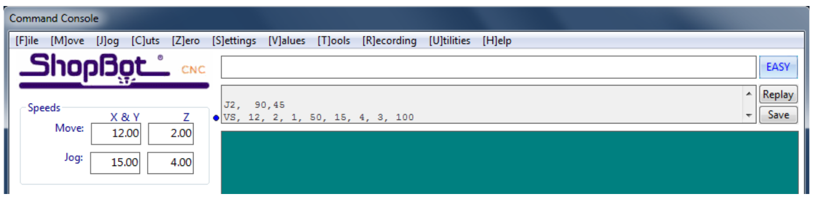
- Is the appropriate bit in the spindle/router? When changing a bit:
- Is the LockOut Key disengaged so the spindle will not turn on?
- Is the bit tight in the collet?
- Is the bit sticking out of the collet far enough to cut to full depth without dragging the collet?
- Is the material in the location and orientation defined by the CAD/CAM file?
- Is it held down temporarily or permanently so it is safe to run a file?
- Are all clamps/screws out of the way so it is safe to run a file?
- Is the ShopBot zeroed at the correct location in the X and Y?
- Is the Z Zeroed at the location defined in the CAD/CAM file?
    - Surface of the material or the table?
- Is the power to the spindle or router turned on?
    - All Tools: Is the LockOut key re-engaged after the bit change?
- Alpha Tools: has the **reset** button on E-stop pendant has been pushed and the display on the VFD lit up?
- Standard Tools/Desktop Tools with Spindle: Is the VFD showing it is powered up? If no Spindle Speed Controller, is the VFD set to the correct RPM's?
- Models with a Router: Is the power to router turned on at the router?
- Is the correct file loaded?
    - **(FP)** or **CUT PART**, then select correct file.
    - If not sure it is the correct file, **Cancel** out, and **Preview** the file on screen.
    - Maybe try an Air Cut of the File.
- Run the file. Do not change anything on the Fill-in sheet, just **Enter** or **Start** the file.
- If the software determines the current bit has a different tool number than the last file run, respond to the questions about bits.
- At the Prompt to Start the Spindle/Router.
    - Alpha tools: Press the "**Start**" button on the E-stop pendant? Listen for spindle turning before pressing **OK** to start the ShopBot moving. **If the spindle is not turning**, Cancel the file and troubleshoot why spindle did not turn on.
    - Standard and Desktop Tools: Clicking **OK** will turn on the spindle/router. There is a built-in 2 second pause before the ShopBot starts moving. **If the spindle/router does not turn on** in that interval, Pause (hit the space bar) then Quit the file. Troubleshoot why the spindle did not turn on.
- Is the Dust Collection Vacuum turned on?

WATCH YOUR JOB!

- If you wish to **Stop** or **Pause** the file without having to re-zero everything, use the **SpaceBar** on the keyboard OR click on the **STOP** button on the ShopBot Position screen. The XY movement of the tool will stop, the bit will pull out of the material and the spindle will stop turning.
- If everything is OK, **Resume** the file. Remember to restart the spindle when prompted. Otherwise, **Quit** the file.

Bit Selection, Feeds and Speeds

This **Chapter** outlines important steps in selecting the appropriate bit, and setting the correct Feed Rate (Move Speed) and Spindle Speed (RPMs) for a project.

The Chapter is sorted into categories, so read through the entire document, or go to the section that describes specific toolpaths and bits. Visit the website of the bit manufacturer for recommendations for specific bits and materials. Remember, these are starting points, so keep a record of the feeds, speeds and pass depths that work for you.

This Chapter references both ShopBot Control Software (SB3) and VCarve Pro CAD/CAM software included with the purchase of the ShopBot. If the CAD/CAM software of choice is something other than VCarve Pro/Aspire, general concepts of bit selection and feed and speed rates still apply.

Bit Selection and Feeds/Speeds for CNC Machining

It takes some practice and knowledge to figure out what bit to use, how fast to have a CNC machine move through the material (feed rate or feeds) and at what speed to set the spindle or router (RPMs or speeds).

The Goals:

- A clean cut.
- A bit that is cool at the end of the job.
- An efficient work flow for the facility.

Factors which influence Feeds and Speeds (and other settings such as Pass Depth):

- **The Type of Toolpath:**
  - 2D (once the bit plunges into the material, the bit stays at that depth for the entire operation).
  - Engrave or VCarve (the bit moves simultaneously in 3 axes (X,Y and Z) throughout the operation).
  - 3D Finish (the bit moves simultaneously in 3 axes (X, Y and Z)).
- **The Type of the Bit:**
  - End mill (flat bottom or straight end).
  - V Bit.
  - Ball Nose bit.
- **Geometry of the Bit:**
  - Up or down spiral, straight bit.
  - Meant for wood, or plastic, or aluminum.
  - Diameter of the bit.
  - Number of flutes.
- **The Material**:
  - Wood
  - Plastic
  - Non-ferrous metal (i.e. aluminum).
  - Other
- **The Type of ShopBot:** (Gantry, Desktop, Handibot).
  - An alpha tool can power through material more quickly than a Standard.
  - The Desktops and Handibot should machine at lower feed rates than the PRSalpha/Buddy.
- **The Type of Cutter (spindle or router).**
- **The Finish Quality desired.**

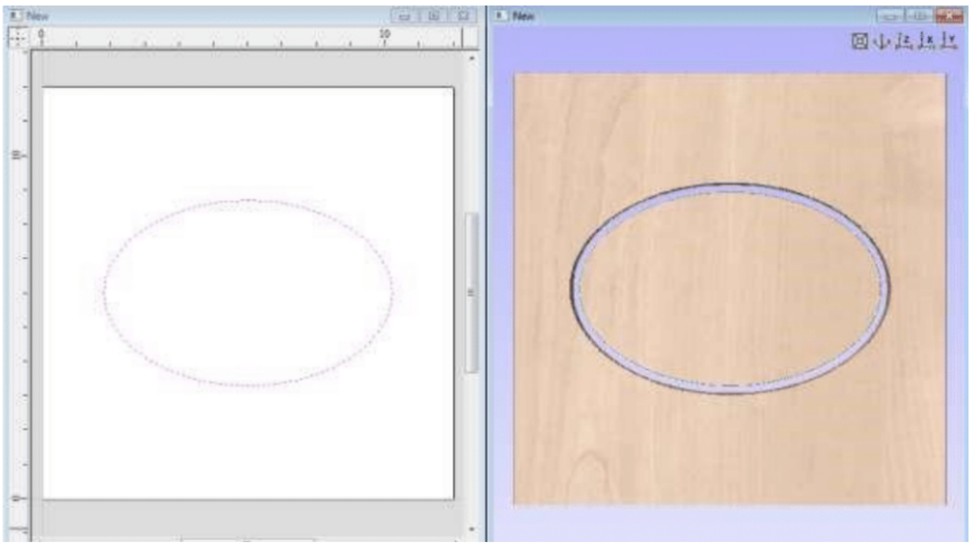
Production Needs: A school or Maker Space may have different priorities than a Manufacturing Facility.

Getting Started

Toolpaths That Stay at One Depth for the Entire Operation (2D)

Example: “Profile” is the toolpath that is used to cut out a part. The router bit plunges into the material to a depth set by the operator, then stays at that depth for the entire operation. In cutting through a thicker piece (say 1/2” or 12 mm plywood) with a ¼” (6mm) bit, it may take several passes or plunges to cut all the way through the material. However, each pass stays at the same depth once the plunge has been made.

In the example below, the CAD (design) is a simple oval that will be cut out of a piece of .5” thick plywood. The simulation of the CAM (machining or toolpath) shows that the toolpath will cut all the way through the material.



There are two main steps when creating a toolpath to cut out the part

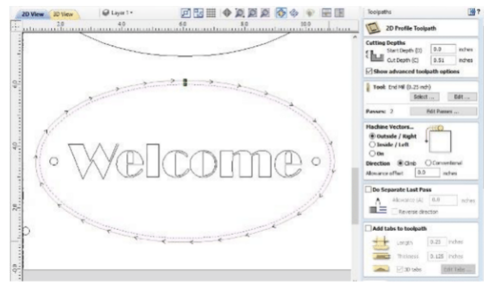
- Define the variables for the toolpath strategy.
  - Start depth
  - Cut depth
  - Where to cut (outside, inside, on the design line (vector).
  - Direction to cut (climb, conventional).
  - Add tabs, ramps, etc.

- Define the strategy for the bit (tool) in the Tool Library.
  - Tool geometry
  - Pass depth
  - Stepover
  - Spindle speed
  - Feed rate
  - Plunge rate
  - Tool number

Settings to Cut Out a Part Using a Typical “Profile” Toolpath with a Common Bit in a Common Material

Givens:

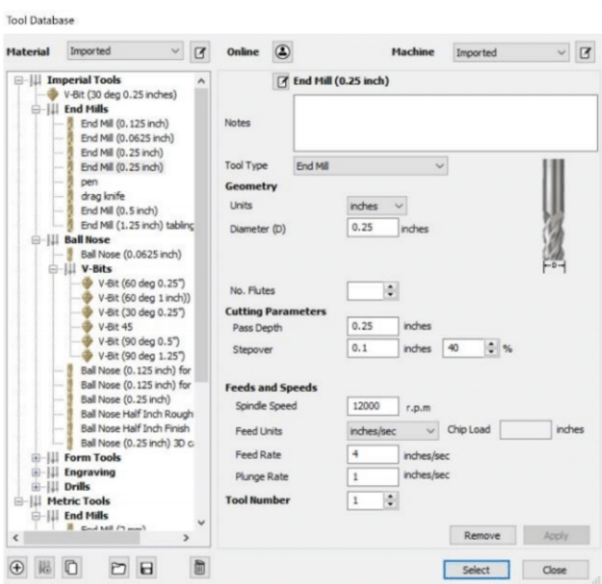
ShopBot Gantry Tool or Buddy (PRSalpha or Standard).



- Material: Wood, such as Plywood.
- Toolpath: A **Profile** that will cut around the **outside** of the vector (design line).
- Strategy: after the plunge into the material, the bit stays at that same depth for the entire pass (definition of a “2D” toolpath).
  - Start depth: 0.0 (top of material).
  - Cut depth: 0.52” (specified by user = thickness of material +.02”).
  - Bit: A Straight bit, .25” diameter.

Click on Select Button in the Tool section to Open the Tool Library.

Entering the suggested settings in the Tool Library in VCarve Pro/Aspire.



Choose a ¼” End Mill bit from the list.

Enter/confirm data in the Tool Library.

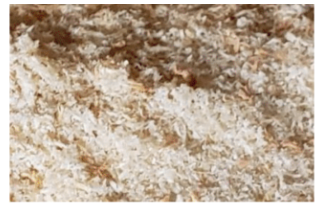
- Diameter of the bit is .25”.
- Pass depth = diameter of bit (pass depth is the deepest the bit will go in any one pass, determines number of passes).
- Stepover = 40% (stepover or overlap is only relevant for pocketing).
- Spindle speed set to 12000.
- Feed rate set to 4 **inches per second** (3 ips for Desktop; 2 ips Handibot).
- Plunge rate set to 1 ips.
- Tool number set to 1 (User defines tool number. For tools with a single spindle, use numbers 1 – 18).
- **Select** the tool.

SUMMARY: For this toolpath on this ShopBot, using this material and bit, the variables in the Tool Library are:

- Pass Depth = Diameter of the Bit .
- Spindle Speed of 12000 RPMs.
- Feed Rate (X and Y axes) of 4 ips.
- Plunge Rate (Z) of 1 ips.

Chip Load

How were these numbers determined?
One of the criteria for success is that the bit is cool (and not broken) at the end of the cut. The actual size of the waste (the “Chip Load”) that is generated when a router/spindle is machining influences how hot the bit is at the end of the cut.



- A spinning bit generates friction and heat as it moves through the material. Heat is damaging to the carbon bits generally used in CNC machining, so they will become dull and/or break if allowed to get too hot.
- When the waste generated by the machining process resembles dust (the smaller the chip load), the more heat stays on the bit.
- With a larger chip load, more of the heat is transferred to the flying chips, so the bit stays cooler.
- But there can be too much of a good thing. A chip load that is too large puts more stress on the cutter, causing a rough cut, bit deflection, and/or bit breakage.

The formula for calculating the Chip Load is:

Chip Load = Feed Rate in inches per minute / (RPM x # flutes of the bit)

In the example above, the settings were.

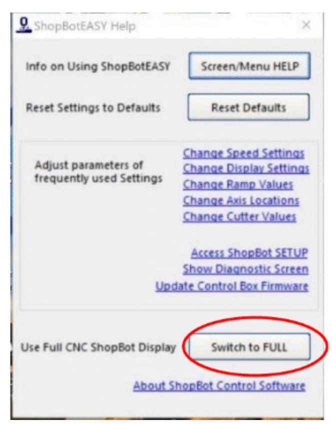
- 4 inches per second (or 240 inches per minute).
- 12000 RPMs.
- 2 flute bit (given, not specifically stated).

Chip Load (size of Chip in inches) = 240/12000 x 2 = .01

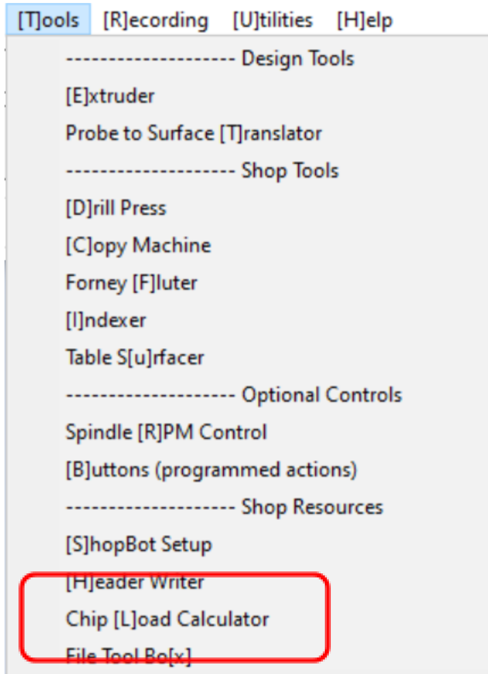
The Chip Load Calculator in the ShopBot Control software solves for the unknown in the mathematical formula for determining the chip load.

Aside: If ShopBot Control software is in EASY mode, switch to FULL mode to find the menu that includes the Chip Load Calculator.

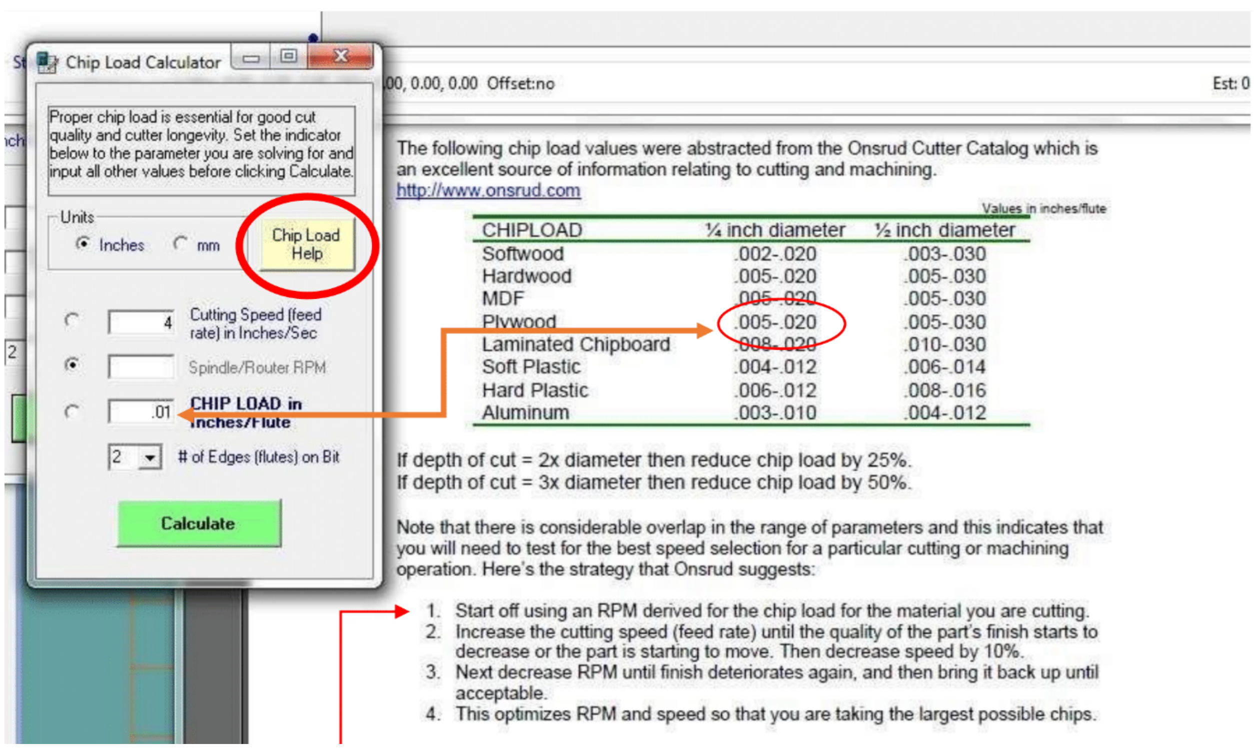
If necessary, click on the Question mark in the position screen, then select Switch to Full.



Open Tools > Chip Load Calculator



Click on **Chip Load Help** to open a chart of suggested chip loads for different materials. To solve for the correct RPMs, enter a chip load value near the middle of the suggested range and a move speed of 4 ips.



Strategy for Dialing in Optimal Speeds and Feeds

The information from the Chip Load Calculator is just a starting point to determine the correct **balance** between the move speed through the feed rate and the RPM's of the spindle. There is always some trial and error in the actual performance of the tool/bit for that material.

The needs of the facility should also be taken into account. For a school or a custom shop that is creating fewer parts, preserving the life of the bit and having successful outcomes on "one off" projects may be the most important factors. The values suggested in this document focus on that outcome.

A production facility that is processing 50+ sheets of material a day should probably push the CNC machine/spindle faster and harder to cut each sheet more quickly. It may be more cost-effective to replace bits more frequently if moving faster saves 5+ minutes per sheet of material.

To INCREASE the ChipLoad

- Increase the Feed Rate.
- Decrease the RPMs.
- Use a bit with fewer flutes.

To DECREASE the ChipLoad

- Decrease the Feed Rate.
- Increase the RPMs.
- Use a bit with more flutes.

Notes on Materials

Plastic:

- Plastic machines better with the correct bit.
- If plastic is melting back when it is being machined, start by slowing the spindle down 100 -200 RPMs.

Aluminum has a tendency to melt back also.

- Use the right bit (see below).
- **Decrease the pass depth.**
- If pocketing or clearing, decrease the stepover.
- Consider using Fusion 360's Adaptive clearing techniques for 3D machining of aluminum.
- Decrease the RPMs.
- Play with the Feed Rate.

Feeds and Speeds by Types of Bits, Types of Toolpaths and Materials

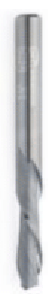
A ShopBot Starter Bit Kit includes End Mill bits for wood and plastic/aluminum, a V bit and a Ball Nose bit.

The following pages will break down each of these categories and give more detailed information about the Bits, as well as determining Feeds and Speed for the Toolpath, Type of Material (Wood, Plastic, Aluminum), and Type of ShopBot (Gantry Tool and Buddy/Desktop/Handibot).

Many Feeds and Speeds Charts based assume that the user is using an end mill bit and a toolpath in which the machining is done at one depth once the bit has made the initial plunge. The Feed rates are for the X and Y axes only. But there are more considerations.

End Mill Bits for 2D Toolpaths (Profile, Pocket, Drill)

End mills for machining with a CNC machine are different than bits used for “hand” routers, in part because they have a cutting edge on the bottom that allows them into plunge into the material rather than just cut from the side. The most common forms of end mills have a flat bottom or “square-end” and are used for most operations where the bit plunges to one Z depth (operator defined) for the entire operation. The Z does pull up to a safe distance above the material while positioning for each new start location, but it does not change height while the bit is machining the material in the X and Y axes. For purposes of this document, this type of toolpath will be called a 2D toolpath.



While all end mills have a blade at the bottom, the geometry of the fluting will vary depending upon the material being machined (e.g. wood, plastic, aluminum) and the desired results (e.g. rapid clearing of chips, smooth top and/or bottom edge.) Differences in the geometry of end mill bits are explored below.

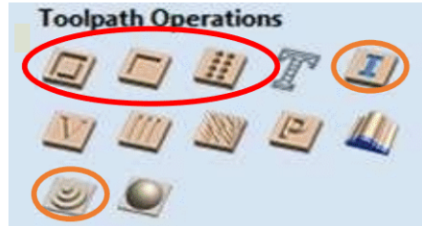
Shown: Down spiral end mill for wood.

Most feed and speed charts that depend upon the “chip load” are designed for determining the **feed rate** of the X and Y axes in these types of toolpaths. The values in this document are generic starting points for the bits in the ShopBot Starter bit kits.

Check with the manufacturer for specific recommendations, and keep records to tweak the feeds and speeds based on results.

For 2D toolpaths, the **plunge speed** (Z axis) into the material is usually slower than **move speed** (X and Y axes) through the material to reduce stress on the bit and the spindle/router as it plunges into the material.

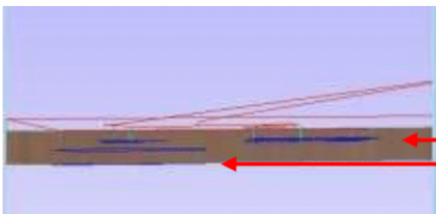
Most common Toolpath operations for End Mills

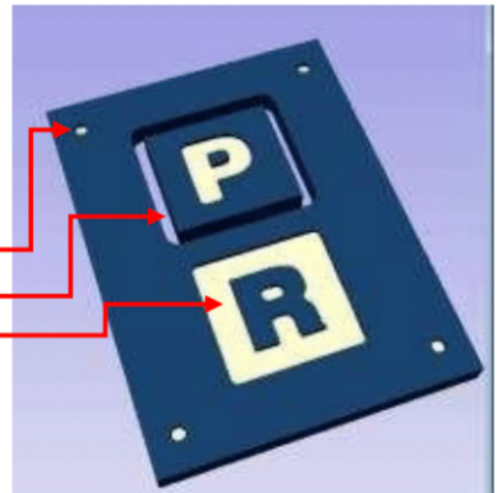


- Profile
- Pocket
- Drill

Secondary Operations:

- Inlay
- 3D Roughing





Simulation of toolpaths

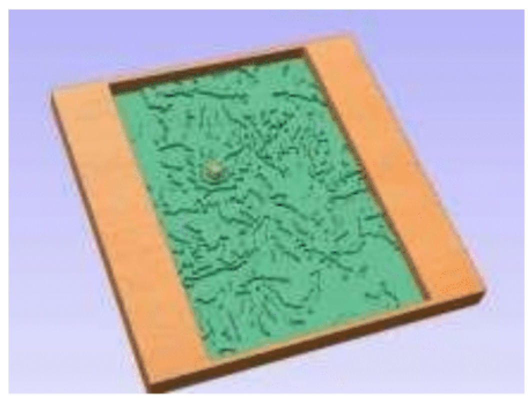
Drill

Profile

Pocket

Side view of Profile and Pocket toolpaths.
In Vectric preview:

- Red lines are positioning moves.
- Blue lines are machining into material.



Simulation of 3D Roughing toolpath with stepdown equal to .25” bit diameter.

A More Detailed Look at the End Mills and Surfacing Bit in ShopBot Starter Bit Kits

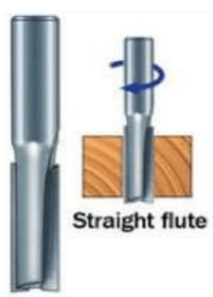
What is the difference between an end mill and a square-end bit?

An end mill has cutting flutes that extend across the bottom (end) of the bit. It is designed for plunge- cutting as well as lateral cutting. “Square-end” is simply a description of end shape. Square-end bits are not always end mills, and end mills do not always have square ends (example: ball nose bit for 3D carving).

The first image at right shows a true end mill. Notice that there is a “blade” across the bottom of the bit. The second shows a straight-fluted, square-end bit that is not an end mill. The table surfacing bit included in the ShopBot router bit is an example of a square-end bit that is not an end mill. Special care must be taken when “plunging” with this type of bit, such as starting the Z plunge off the edge of the table and moving into the surfacing routine, or ramping into the plunge. Take passes of .08” or less when using this surfacing bit.

General Geometries Available for End Mill Bits

Flute Type: There are four basic flute types: Straight, spiral up-cut, spiral down-cut, and compression. Each has its own advantages and disadvantages, which are outlined in the chart below.



Good edge quality on most materials.

Moderate chip clearing abilities.

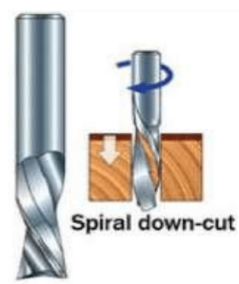
Ideal for: general-purpose cutting.

May chip or fray top face, good quality on bottom face when through-cutting.

Excels at clearing chips and dissipating heat, especially with “O-flute” bits.

Upwards force may pull up material

Ideal for: plastics, aluminum, or any material where heat buildup is a concern.

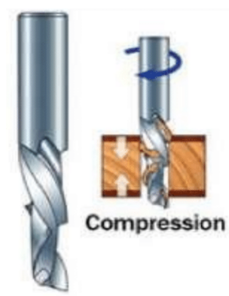


Best edge quality on top face, may chip or fray bottom face when through-cutting.

May compact chips in a groove.

Downwards force may help with cutting thin sheets.

Ideal for: plywood and laminates (pocketing).



Clean edge on both top and bottom face.

Designed to cut veneered or laminated materials at full depth in one pass.

Ideal for: plywood and laminates (profile cutting).

Bits for Wood vs Bits for Plastic or Aluminum

For best results, use the correct bit (and the correct feeds and speeds) for the material.

A ShopBot router bit kit comes with a selection of end mill bits for wood or plastic/aluminum. More complete information on bits for specific materials can be obtained from manufacturers such as www.Onsrud.com.

A note on searching the Onsrud site. If looking for a specific type of bit, it helps to know the Series Number. Start with the Series Number from the ShopBot starter bit kit   Example: 57-000 for a down-spiral bit for wood.

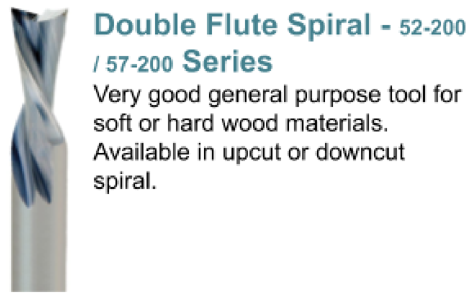
Bits for Wood

52-910 Up Spiral bit .25” diameter.

57-910 Down Spiral .25” diameter.

Note the base of the bit looks like a “fish tail” or lazy W. The flutes are separated.

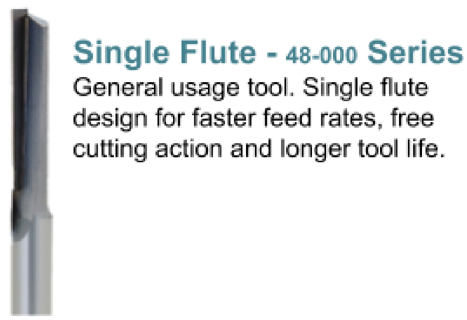
48-000 straight bits, .25” and .5” diameter.



Feeds, speeds and pass depths:

For general purpose cutting:

- Pass depth = diameter of bit or less.
- Feed Rate of 4 ips (Desktop: 3 ips; Handibot: 2 ips).
- Plunge rate (Z) of 1 ips.
- RPMs of 12000 is a good starting point Smaller bits (1/8”) may require faster RPMs.



For Production: If the focus is on reducing the time it takes to mill or machine a sheet of material rather than on saving a bit from wearing out/breaking, push the limits of the feed rate and the RPMs to increase efficiency.

Pass Depth can be increased to 1.5 times the diameter of the bit to reduce number of passes. In Profile Toolpaths, there is an option to **Ramp** the bit into the plunge to reduce stress on the spindle. See Chapter on **Hold Down Techniques** or HELP in VCarve Pro/Aspire for more information on Ramping into a plunge.

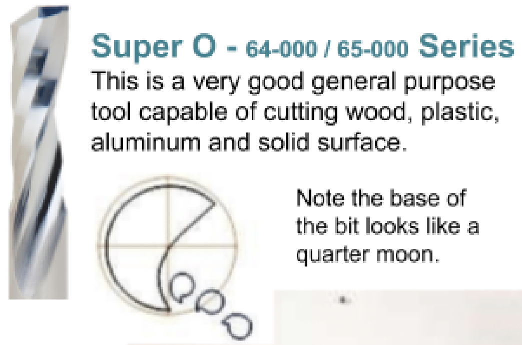
Bits for Plastic or Aluminum

Onsrud 65-000 Up Spiral bit.

Onsrud 64-000 Down Spiral bit (not included in bit kits).

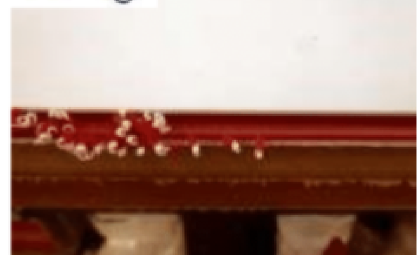
www.onsrud.com/xdoc/SuperO

The flutes are more closely spaced.



Suggestions for working with plastic:

Slow down the RPMs and/or increase the Feed Rate when working with materials such as **plastic** that melt back. Turning the spindle or router down 100 - 200 RPMS can make a significant difference in the cut quality of acrylic as well as HDPE.


Tightly curled chips flow off King Color Core HDPE with no melt back: 4 ips/11000 RPMs

For more information about bits for plastic, visit the Onsrud site: http://www.onsrud.com/xdoc/CNCPlastic-Cutting-Tools

Other bit manufacturers recommended by ShopBotters:

Harvey Tools: www.harveyperformance.com/

Amana: www.amanatool.com

Belin Bits

Suggestions for working with aluminum:


Explore the adaptive clearing techniques available in the manufacturing section of Fusion 360 for advanced projects in aluminum.

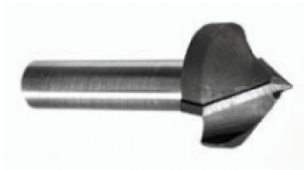
"O" flute as above as well asnOnsrud 63-000, 62-000 series. Aluminum is often an alloy with Copper, Zinc, Magnesium, Manganese, Silicon. 6061 is a good alloy for machining on a ShopBot. Aluminum can be soft and sticky "Chips" tend to get imbedded in the tooling.


With the right bit, feeds, speeds and pass depth, some aluminum alloys can be cut dry.

- **Decrease the pass depth** significantly when working with aluminum Decreasing the pass depth will increase the number of passes.
- If pocketing or clearing large areas, consider balancing pass depth and stepover.
- Slow down the RPM's.
- Play with the Feed Rate.

If needed, try **coolants** such as denatured alcohol, WD 40 or diluted dish detergent (1 part Dawn, 2 parts water.) Cool air pointed at bit can also help with cutting aluminum.

Engraving or V-Carving with V Bits



Most common Toolpath operation:

- Engraving or V-Carving

Secondary operations:

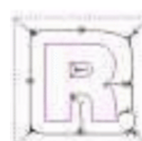
- Quick Engrave
- Prism

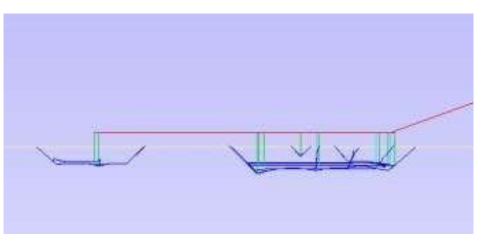
The Engraving or V-Carving toolpath is often used for making signs or other design work. Any “closed” vector such as True Type Font Text or designs can be V-Carved or Engraved.

In the Tool Library, the operator specifies the angle of the bit (example, 90 degree V, 60 degree V, etc.).

An engraving toolpath follows the center line of the closed vector.

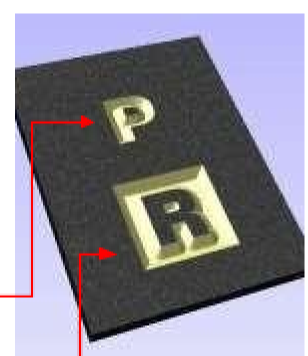
In the top example, only the letter was selected. In the bottom example, the border and the letter were selected.





An Engraving or V-Carving toolpath moves simultaneously in 3 axes. The CAM software determines how deep to plunge the V Bit to clear the area in the design. At the corners, the V bit pulls up and out, so that corners are sharp.

Simulation of V-Carving the letter, and the area between the border and letter.



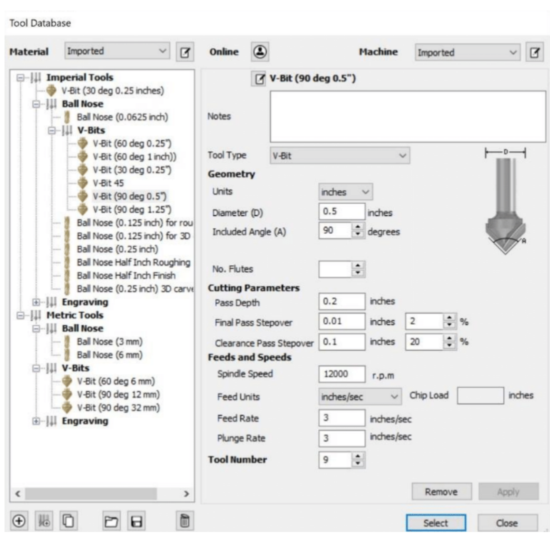
Settings for Engraving or V-Carving in Common Materials

For this type of toolpath, the bit is commonly moving simultaneously in 3 axes. In this case, it is more efficient to match the Move Speed and the Plunge Speed.

Givens:

- ShopBot Gantry Tool or Buddy (PRS alpha or standard).
- Material: Wood, such as Plywood or Plastic such as HDPE.
- Toolpath: **V-Carve/Engraving** Toolpath
  Strategy: The bit follows the centerline of the CLOSED vector, plunging deeper where the width of the design is wider, and pulling up and out of the material in the corners of the design.
  - Start depth: 0.0 (top of material).
  - Notice that “cut depth” is not an option. The software will calculate how deep to plunge the V bit during the operation.
  - Flat Depth: Leave unchecked unless otherwise specified (see below).
  - Bit: A 90 degree V bit with a diameter of .5” at the top of the bit.

Clicking on the **Select** Button in the Tool section opens the **Tool Library.**



Select the V bit included in the ShopBot starter bit kit (usually 60 V or 90 V) Example: 90V

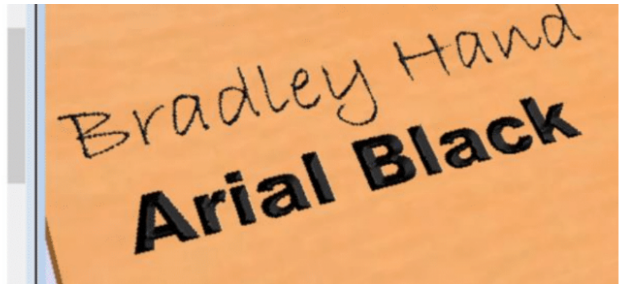
- Diameter of the bit is .5” for 90v or 1” for 60 V bits included in bit kits.
- Pass depth ~ ½ diameter of bit.
- Spindle speed set to 12000.
- Feed rate set to 2.5 or 3 ips.
- Plunge rate set to 2.5 or 3 ips (Feed Rate and Plunge matched).
- Tool number set to 6 or 9.
- Users defines tool number Suggestion: use something easy to remember (example: 9 = 90 degree V) For tools with a single spindle, use numbers 1 – 18).
- **Select** the bit.

SUMMARY: For this toolpath on this ShopBot, using this material and bit, the variables in the Tool Library are:

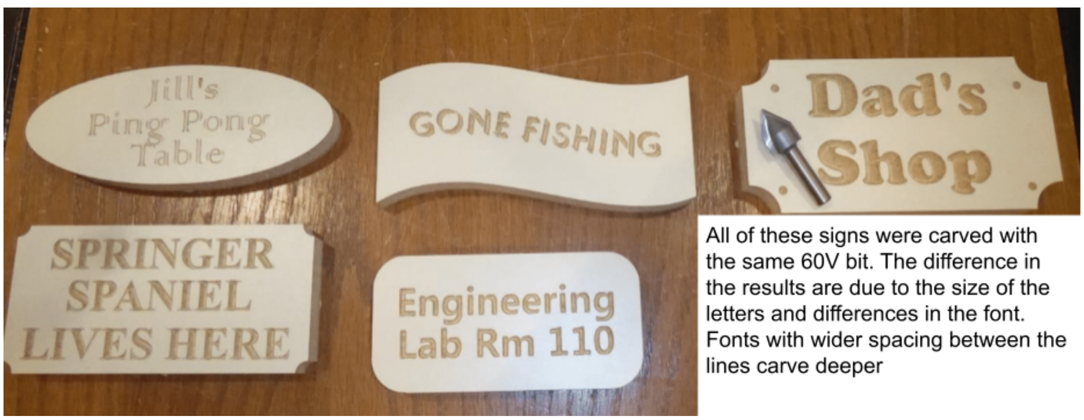
- Pass Depth = Half of the diameter of the Bit.
- Spindle Speed of 12000 RPMs (wood).
- Feed Rate (X and Y axes) of 2.5 or 3 ips.
- Plunge Rate (Z) of 2.5 or 3 ips.

Factors Affecting Engraving or V-Carving

The characteristics of the chosen font or design



Two different fonts were used in the design. Both are 1” high, but the spacing between edges in the Bradley Hand font is narrower than Arial Black. The simulation shows that when the fonts are carved with the same 90V bit, the toolpath for the wider font plunges deeper into the material with a more satisfying result.



All of these signs were carved with the same 60V bit. The difference in the results are due to the size of the letters and differences in the font. Fonts with wider spacing between the lines carve deeper

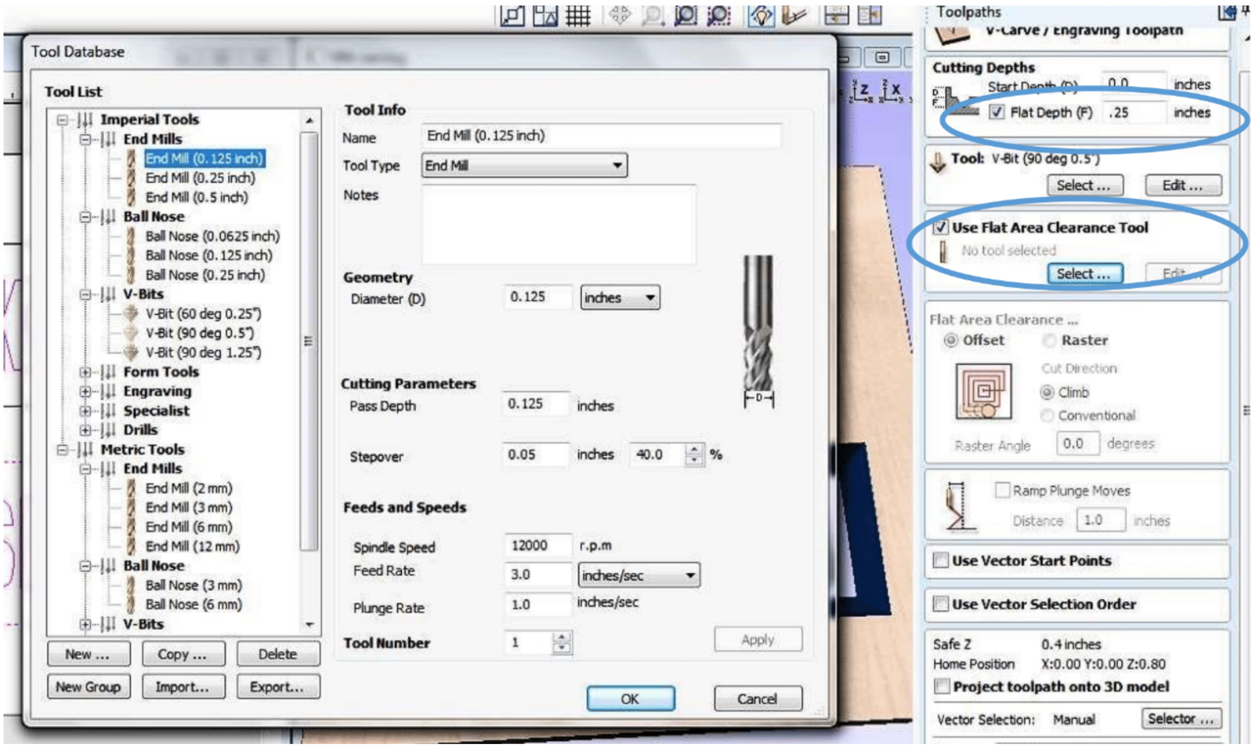
Importance of the Angle of the V Bit

V Bits come in different angles. In the example below, the font is the same, so the different looks are due to the different bits. When the user defines the angle of the V Bit in the Tool Library, the CAM portion of VCarve Pro/Aspire automatically adjusts the depth of the toolpath to reflect the geometry of the bit.

Lettering carved with a 120 V Bit will be shallower than lettering carved with a 60 V Bit. Light will reflect out from the letters differently, depending upon the angle of the engraving. Hint: for larger letters, use a wider V.

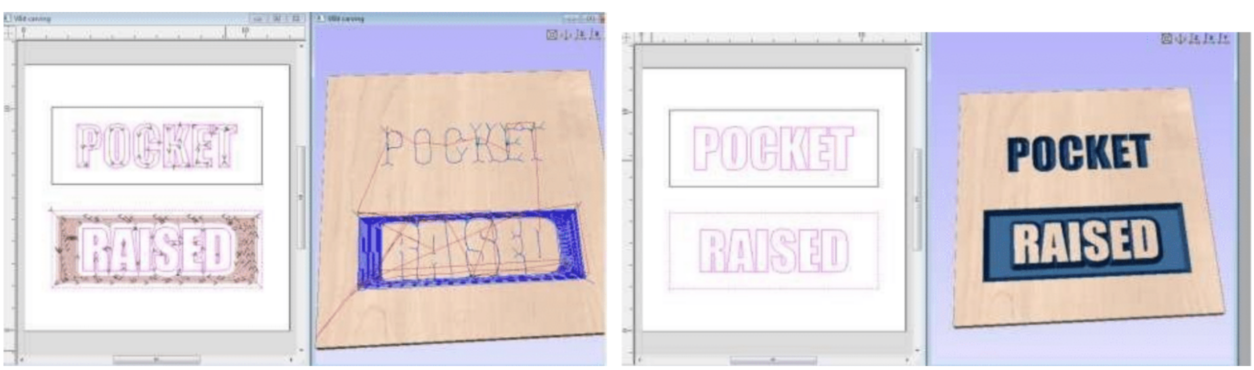
Adding a Flat Depth to a V-Carve/Engraving Toolpath

If the V-Carve/Engraving toolpath would cut all the way through the material, or to bevel the edge of the design and then pocket out the rest, use the “**Flat Depth**” option available in the V-Carve/Engraving Toolpath.



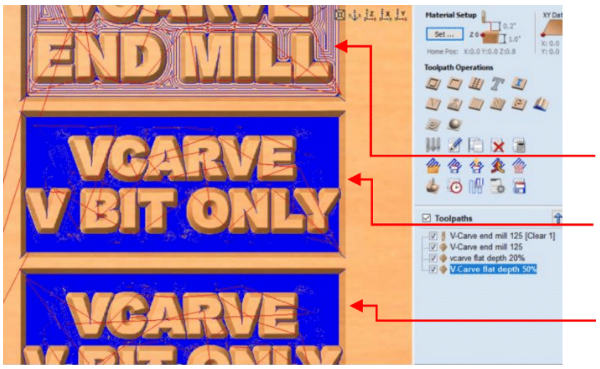
If there is a significant amount of material to remove in the Flat Depth, a second, **Flat Area Clearance Tool** may be added to the toolpath. For this second bit, use the same settings as for a 2D Pocket as described above.

If no Flat Area Clearance Tool is added to the toolpath, the software will use the tip of the V Bit to pocket out the area remaining area. This is where the stepover value is important.



Examples of Different Results When Using a Second End Mill Bit, or the V Bit Only, for the Flat Depth option of V-Carving

Simulation with toolpaths superimposed of 3 different option for V-Carving with the Flat Depth option.



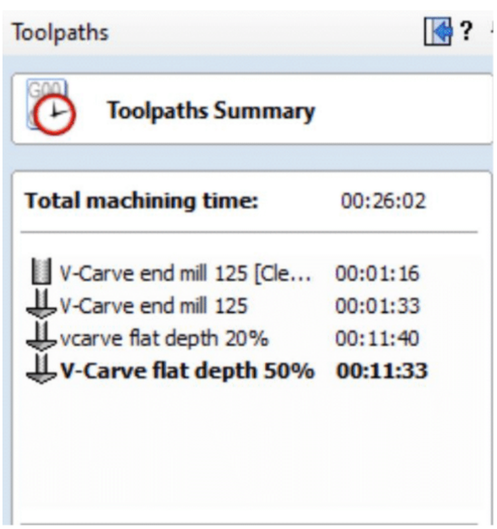
The 90 V Bit bevels the edges of the letters and the border.

- A second .125” bit pockets the area. The software creates 2 different toolpaths.
- The 90 V Bit bevels the edges, then the tip of the V bit pockets the flat area. Stepover set to 20%.
- The 90 V Bit bevels the edges, then the tip of the V bit pockets the flat area. Stepover set to 50%.

Detail of Simulation of the three options.

Create a simulation that shows the results. (A simulation on screen is easier to see than a screen shot of same.) The result is not quite as smooth when the V Bit is the only bit used.

Actual results will depend upon the material.



Estimating the Time to Machine a Toolpath.

Select all of the tools paths (put a check mark in front of them) and then choose the **Summary of All Toolpaths** icon to get an estimate of how long each option would take to machine.

See the VCarve Pro/Aspire **Help** for more information on this. This estimate is best used to compare one tool path option to another, and not for actual time to machine.

Suggestion: Preview the .sbp file in the SB3 ShopBot Control software for a better estimate of machining time (**Safety and Basic Use**).

Example of a Large Sign (~55” x 33” x 2”) V-Carved with a Flat Depth Added

- Ash was glued up and planed by a professional wood supplier.
- The blank was coated with 3 layers of black paint and 3 layers of polyurethane before carving.
- Details and border selected and toolpathed as V-Carve with Flat Depth
  - 90 degree V Bit beveled the edges of the design and border (cut down on chipping of edges).
  - Flat depth set to .125”.
  - .25” End Mill bit pocketed rest of sign in an offset strategy.
- After some sanding of the cleared area, 3 coats of polyurethane sprayed on by professional sign maker.

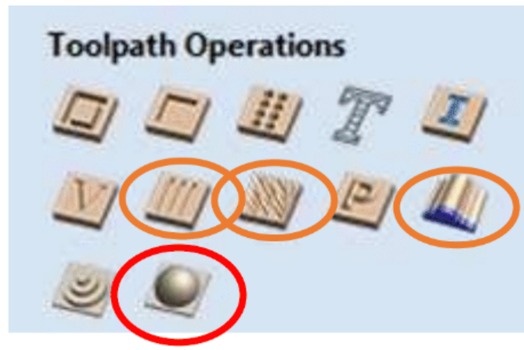
Finished Sign.

Pocketing of area between border and design in progress.

3D Carving with Ball Nose Bits

Caution: Before starting the 3D Finish pass with the ball nose bit, if working with hard material, clear away all the excess material with a 3D Roughing Pass. Treat the 3D Roughing Pass as if it were a 2D Pocket toolpath, and use the settings described above.

For true 3D Carving which uses a **Ball Nose bit**, the Z height is constantly changing as the X and Y axes are moving through the material. The bit is moving simultaneously in 3 axes, so it is more efficient to match the Move Speed and the Plunge Speed. Since most of the material has been cleared out in the roughing pass, and the stepover is small (<10%), very little material is being machined in each pass, so the Feed Rates can be faster.

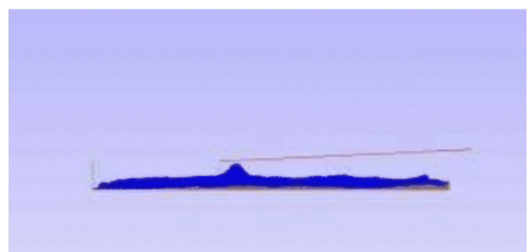


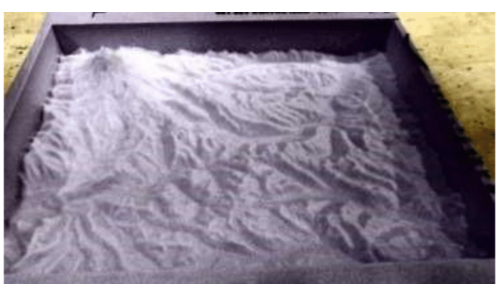
Most common Toolpath operation:

- 3D Finish pass

Secondary Operations:

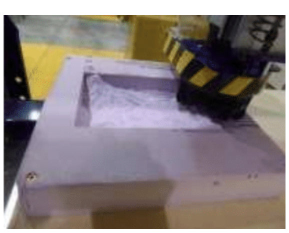
- Fluting
- Texturing
- Molding



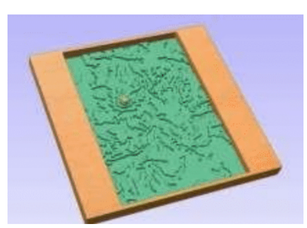


This topographic map of the Mt. Rainier region in Washington State was generated as an .stl file from the website Terrain2STL. The model was then imported into VCarve Pro for generating the 3D toolpaths.

The 3D Finish Pass was carved in pink insultation foam with a .125” ball nose bit.



Since this material is soft foam, this 3D Finish Pass is being machined with no 3D Roughing Pass.

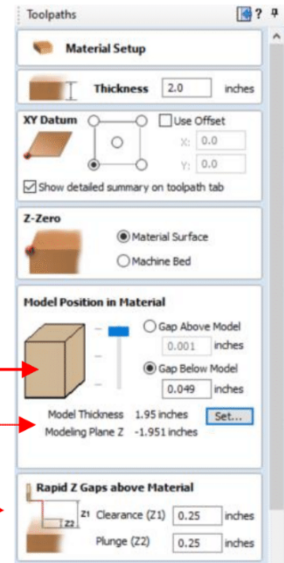


If carving in a harder material (like wood), it would be necessary to do a 3D Roughing Pass with an end mill bit before doing the 3D Finish Pass.

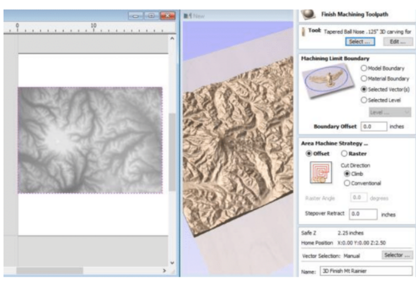
Settings for 3D Finish Pass in Common Materials

For this type of toolpath, the bit is commonly moving simultaneously in 3 axes. It is more efficient to match the Move Speed and the Plunge Speed for 3D Finish Passes.

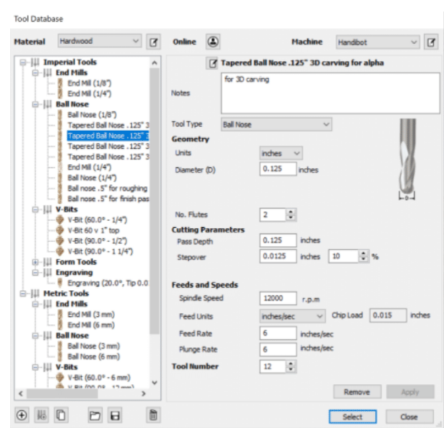
Givens:



- ShopBot Gantry Tool or Buddy (PRS alpha or standard).
- Material: Solid Wood, MDF or Foam.
- Toolpath: 3D Finish Pass.
- Strategy: The Z plunges to full model depth while moving in X/Y.
  - When creating a 3D toolpath (Roughing or Finish), the 3D model itself determines the Z depth. Where the model fits in the material determines the Start Depth. Open Material Setup to explore.
  - The first step is to make sure the model fits in the material. If the model is thicker than the material, the software will display an error. Change the material thickness, or the model size, or "slice" it. See Help in Vectric software for more info.
  - If the model is thinner than the material, position the model in the material.
  - Use the options here to determine how high the Z travels in between cuts and when going towards the Home position (not shown).



  - Clicking on the **Select** Button in the Tool section opens the Tool Library (below).
  - In this example, a vector around the 3D model determines what will be machined.
  - When Area Machine Strategy is set to **Offset** and Climb, the path starts in the middle of the model and machines toward the outside.



- Select the Tapered Ball Nose Bit, .125" diameter.
  - Pass Depth: Pass Depth is an option in VCarve Pro/Aspire starting in v10, but it is not relevant because the Z plunges to the full depth of the 3D model.
  - Stepover: Set the Stepover to 8 – 10% so that the "shoulders" of the previous pass are shaved off in the next pass. (A 10% stepover means that 90% of the area already machined will be re-machined. Let the ShopBot do the sanding).
  - Spindle Speed: 12000 RPMs.
  - Feed Rate AND Plunge rate: 6 ips for Gantry/Buddy 4 ips for Desktop.
  - Tool Number: 12 (arbitrary, 12 for .125" bit?).

SUMMARY: For this toolpath on this ShopBot, using this material and bit, the variables in the Tool Library are:

- Pass Depth: not relevant in the 3D Finish toolpath because bit plunges to full depth of model.
- Stepover: 10% or less so that all traces of machining removed and the finish is smooth.
- Spindle Speed of 12000 RPMs (wood).
- Feed Rate (X and Y axes) of 6 ips (4 ips for Desktop; 3 ips for Handibot).
- Plunge Rate (Z) of 6 ips (4 ips for Desktop; 3 ips for Handibot).

How to Choose the Correct Size Ball Nose Bit

The amount of detail required and the amount of time available to create the 3D Carving determines the size of the ball nose bit. The smaller the bit, the more detail, but also the longer machining time.

Suggestion: Generate several toolpaths in VCarve Pro/Aspire using different bits, then **Preview** the toolpaths to determine the best results. Use the **Summary of all Toolpaths** icon to estimate different machining times.

3 different sizes of Ball Nose Bits are displayed to the right:

- .125” ball nose (not tapered) requires a .125” collet.
- .25” ball nose requires a .25” collet.
- .5” ball nose requires a .5” collet.
- ShopBot Router Bit kits include tapered ball nose bits that use a .25” collet, then taper down to the tip (.125” or .0625”) This tapering gives strength to a bit with a longer cut length while still creating details in the 3D carving.

The map of Egypt behind the 3 bits was carved with the .125” ball nose bit displayed.

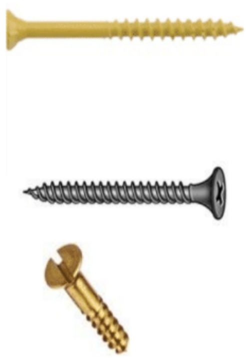
(Protective wax covering on the largest bit).

3D models that have sweeping curves and do not need to retain small details can use a bigger ball nose bit. The photo above has 3D toolpaths that utilized a .25” ball nose bit.

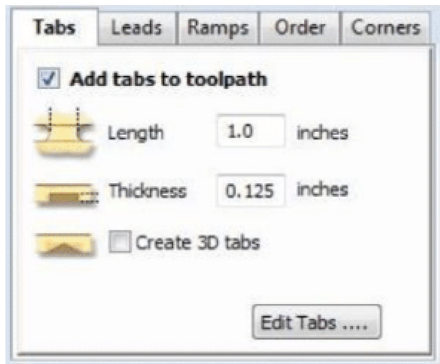
The 2D toolpaths (Pocket and Drill) utilized a .25” end mill bit.

Hold Down Techniques

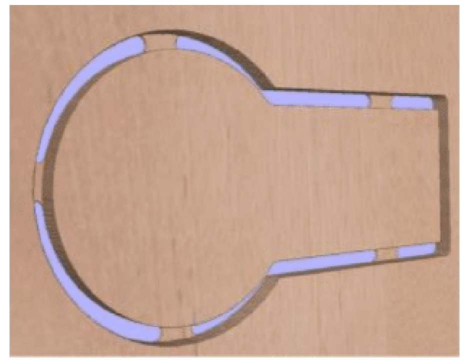
Adequate hold down is critical for safety, efficiency and good part production. Plan for the hold down technique early in the design process.



Fasteners



Adding tabs in V-Carve Pro



Example of tabs on a part

This chapter offers suggestions for holding down the material while machining/milling it on the ShopBot CNC machine.

The Chapter is sorted into two major categories of hold down: Mechanical Hold Down and Vacuum Hold Down. One can manipulate the Design software to locate where it is safe to put a hold down screw, or the CAM software to allow for safe Z travel to avoid hitting a clamp if that is the method of hold down. Even with Vacuum Hold Down Systems, there are times when mechanical hold down or adjustments to the CAD/CAM process must be considered.

This Chapter references both SB3 ShopBot Control Software and VCarve Pro CAD/CAM software included with the purchase of the ShopBot. If the CAD/CAM software of choice is something other than VCarve Pro/Aspire, general concepts of hold down still apply.

Thank you to ShopBotters who have provided examples of their hold down techniques.

Overview of Hold Down Techniques

There are two general categories of hold down techniques.

Mechanical: the material is held in place with a physical object, such as:

- Screws and nylon nails
- Adhesives
- Jigs
- Clamps

Vacuum: the material is held in place with a vacuum system.

- Direct vacuum: jigs and templates.
- Whole Table or Universal vacuum.

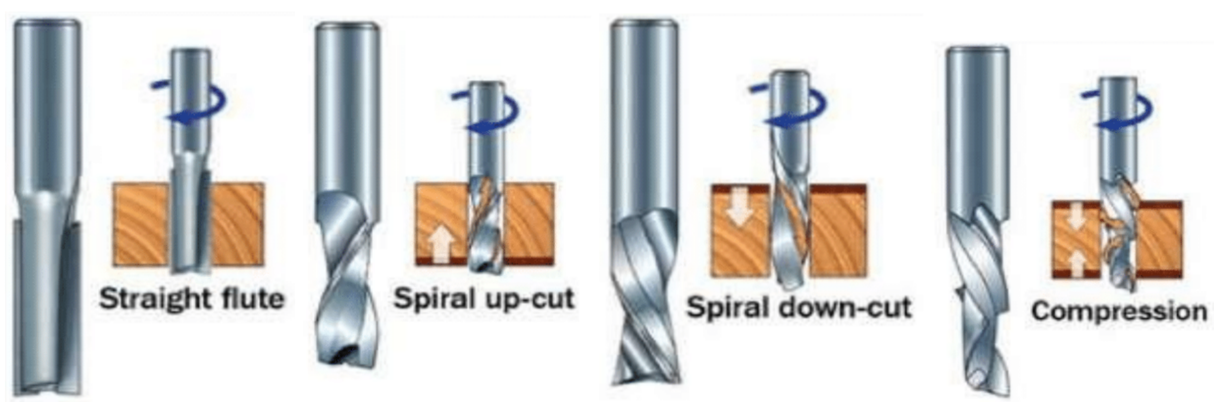
Hold down may require more than one technique.

Regardless of the technique, planning for the hold down should be part of the actual CAD design.

ASIDE: BIT SELECTION AND HOLD DOWN

Flute type: There are four basic flute types: Straight, spiral up-cut, spiral down-cut, and compression. The flute type can interact with the hold down technique:

- An up-cut spiral will clear chips easily, but also pulls the material/parts up as it is machining (put in more hold down screws).
- A down-cut spiral pushes the material down as it is machining and pushes the waste chips down into the kerf, often packing the parts in place (may not need as many hold down screws).
- A compression bit gives the most satisfying results (both top and bottom edge of material clean) when it cuts through in one pass. That single pass puts more lateral stress on the material/parts, which may result in parts moving during machining unless there is adequate hold down.



Mechanical Hold Down Options

Screws and Nylon/Plastic Nails

- This is the most straightforward technique for hold down.
- Have a cordless drill and the appropriate driver available for inserting and removing the screws.
- Note: The screws screw right into the spoil board, which can be resurfaced when it gets torn up too badly (see **Projects and Techniques**).
    - Screws leave dimples in MDF spoil boards, so have sandpaper nearby to level the dimples.
    - Plywood can be used as a spoil board instead of MDF when not using a Universal Vacuum System as hold down.

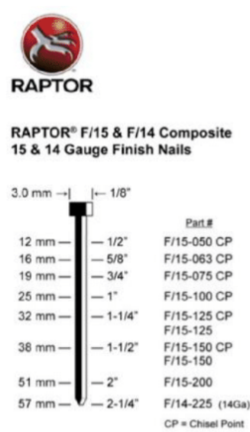
Many different types of screws are appropriate for hold down. The advantages of several types are listed below, with the type of driver needed mentioned.

- Deck screws have no threads at top so the material is pulled down against the sacrificial board: Star drive head (usually T20 or T25).
- Drywall screws are a reasonable option: Philips head.
- Brass screws are softer and help protect bits from breaking if they are hit: Flat head.

Nylon nails are an option with advantages and disadvantages.

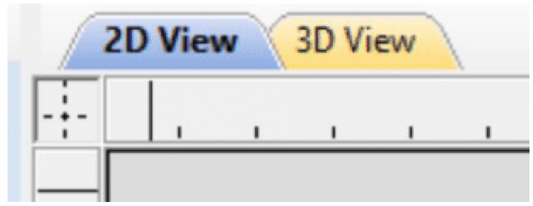
- Hitting a nylon nail won't break a bit.
- When ready to remove the material, the nail itself can be trimmed off at the table level. The remainder of the nail stays in the spoil board.
- The nails are expensive.
- They require a specialized nail gun and compressor.

Metal nails are not recommended since hitting one will break a bit, and they would be difficult to remove from the material.

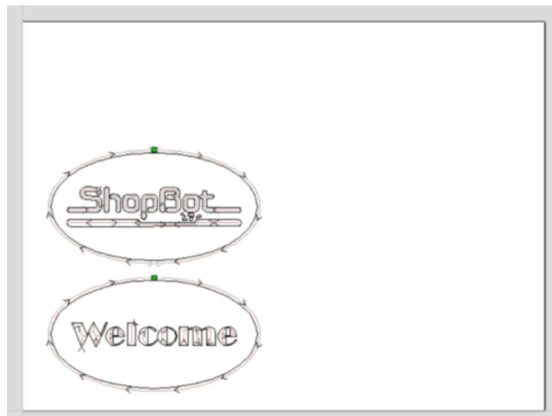


Use CAD Software to Mark Where it is Safe to Put a Hold Down Screw

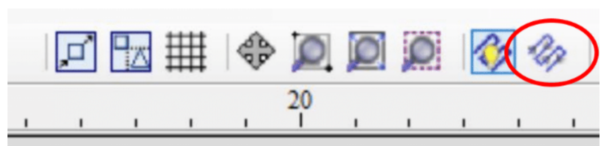
Design the parts to be machined and lay them out on the material in the desired location before placing the hold down locations.



It is a good idea to toolpath the parts to have an idea of where the bit will travel when creating the parts. If needed, switch to 2D view to see the toolpaths (and check the toolpaths to make them visible).



- In the example to the right, two ovals with text inside were laid out on a Desktop-sized piece of material.
- The text on the inside is toolpathed as a V Carve or Engraving file.
- The ovals are cut out using a Profile toolpath.
- In 2D view, the toolpaths default as “wireframe” drawing style which indicates the center point and direction of travel for the bit.
- Switch to “solid” drawing style to get an indication of how much material will be taken out by the bit. It is a toggle, so click on the icon to move between wire frame and solid.
- In earlier versions of VCarve Pro/Aspire, the “solid” option is found on the toolpath screen under the list of toolpaths.
- The blue shows that the bit will travel on both sides of the wireframe. Be sure to put hold down locations outside of the travel of the bit.

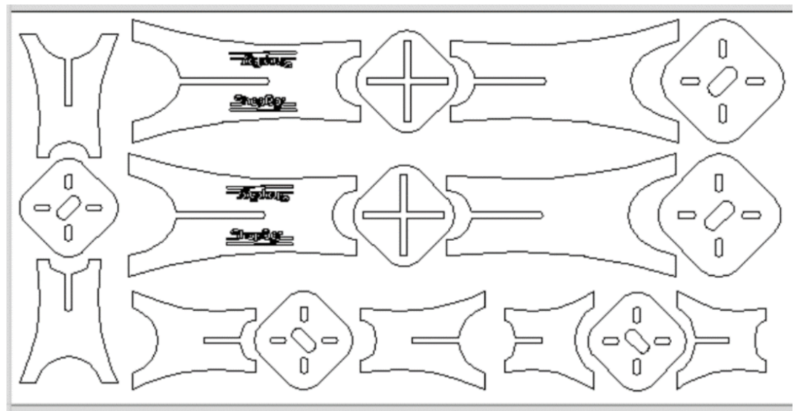


Use the CAD software to draw a circle wherever it is safe to put a hold down screw or nylon nail.

- In the example to the right, there are four .25” circles outside the ovals to hold the material in place.
- The Welcome sign on the bottom has two additional circles on the inside of the oval so that screws will hold the oval in place even after the sign is cut out of the material.
- Assuming that the head of the screw is equal to or less than .25” in diameter, no screws will be in the way of the travel of the bit.

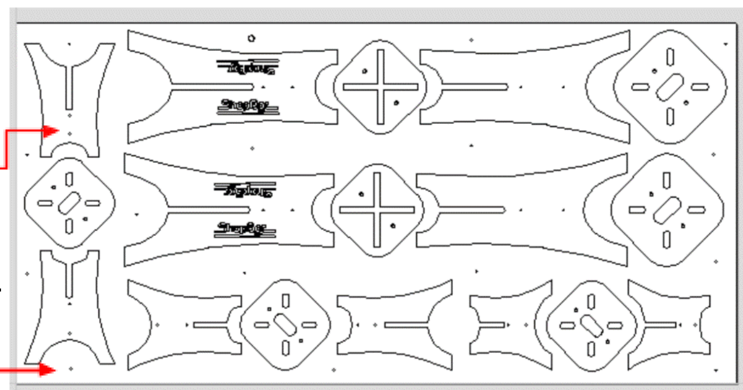
Hold Down Locations in a Full Sheet of Material

On the ShopBot website, under **Explore CNC > CNC Projects** is a “Family of Step Stools”. The design file is a sheet of plywood laid out to create multiple slide-together step stools. It looks something like this:



In this example, there are two sets of hold down markers.

- Markers inside the bases will be hidden by the other side of the base.
- Markers outside the parts keep the material flat against the table. Their size and location are somewhat random.



Putting hold down screws throughout the plywood, including the middle, helps keep sheet goods like plywood that might bow in the middle flat against the table, resulting in better parts.

Here’s a link to the project:
https://www.shopbottools.com/files/Projects/Instructions_A%20Family%20of%20Shop%20Stools.pdf

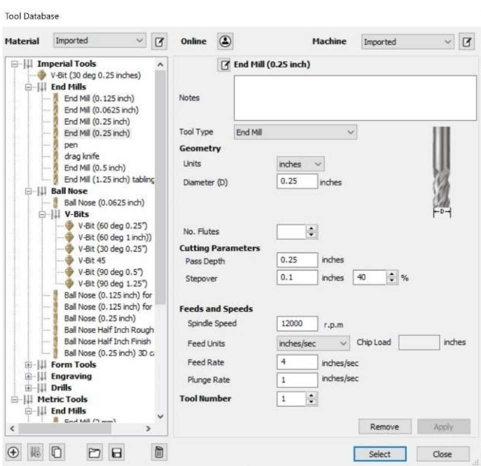
Creating Toolpaths and ShopBot Code for the Hold Down Screw Locations

If this looks familiar, it is similar to **Safety and Basic Use: Design and Toolpathing a Part File.**

The Drill routine is a good option for making a shallow mark to indicate where to put a hold down screw. The bit plunges straight down and then pulls back up, so there is no lateral pressure on the material to move it out of place. If the material is large enough, there is no need for additional hold down while running the Drill toolpath.

Note: In the Drill toolpath, it is the size of the bit rather than the size of the circle that determines that size of the marker.

- Select the circles that mark where it is safe to put a hold down screw (hold shift key and click on each circle).
- Open the Drill Toolpath.
- Leave start depth at the top of the material (0.0).
- Cut depth is how deep to plunge the bit (.1").
- Click on **Tool Selection** to open the Tool Library.

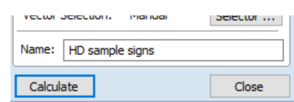


Select a ¼" End Mill bit from the list.

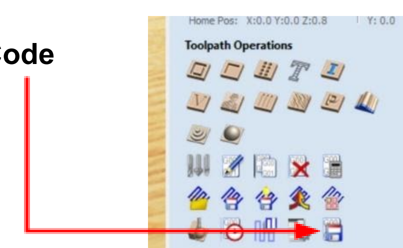
Enter/confirm data in the Tool Library.

- Diameter of the bit is .25".
- Pass depth = diameter of bit (pass depth is the deepest the bit will go in any one pass).
- Spindle speed set to 12000.
- Feed rate set to 4 **inches per second** (3 ips for Desktop).
- Plunge rate set to 1 ips.
- Tool number set to 1 (User defines tool number. For tools with a single spindle, use numbers 1 – 18)
- **Select** the tool.

No other options need to be selected for the Hold Down toolpath. Give the toolpath a name that is easy to find, **Calculate** and **Close**.



Use the Save Toolpath Icon to generate the ShopBot Code



The Hold Down File is Always Run First, and On Its Own

- Use the ShopBot to mark where it is safe to put the Hold Down Screws.
- Move the head out of the way, then screw the material down to the table before continuing.

Adding Additional Hold Down in the CAM File

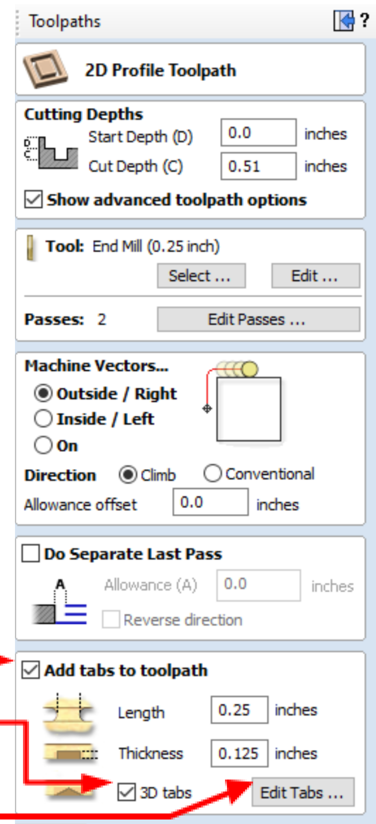
The Welcome oval and the ShopBot oval should be toolpathed separately for cut out. The two toolpaths can be saved in the same .sbp file and run at the same time.

Use the Profile toolpath with no additional features to cut out the Welcome sign because the screws inside will hold down the sign. There will be holes from the screws in the finished sign, but they can be used to mount the sign.

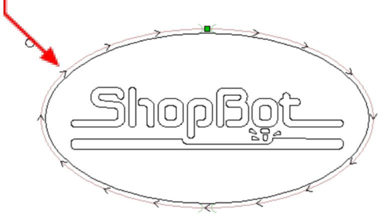
Since the top ShopBot oval does not have any hold down inside the oval, the oval may shift when the cut through is complete. Several options in the Profile Toolpath may help keep the oval from shifting at the end of the final pass:

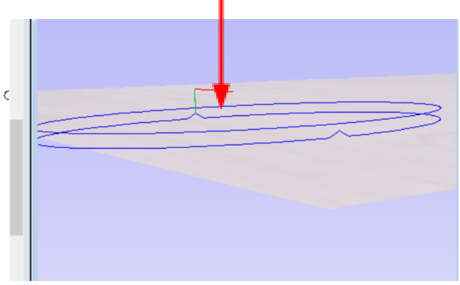
Option 1: Add **Tabs** to the Profile toolpath. The tab(s) will keep the oval attached to the substrate until the operator is ready to remove the oval.

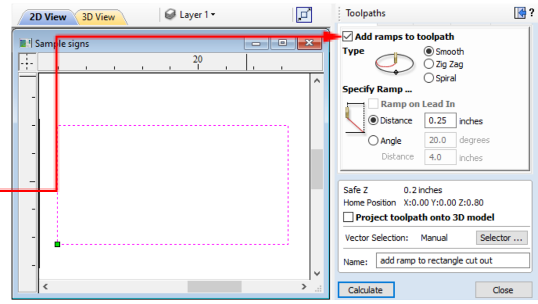
- Checking the 3D tabs box will make the tabs easier to remove later.
- By clicking on the **Edit Tabs** button, the number and location of tabs can be adjusted.
- See the **HELP** in VCarve Pro/Aspire for more details.



Indication of tabs in 3D simulation, 2D wireframe and 3D toolpath.

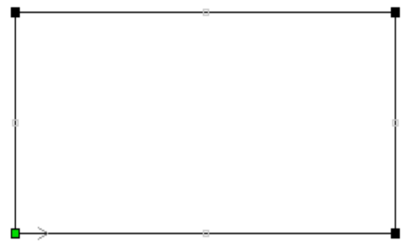




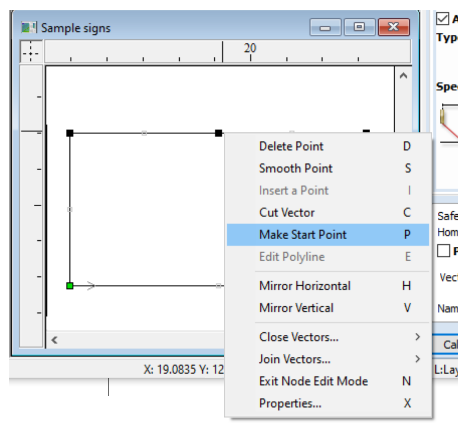


Option 2: Add a **Ramp** to the Profile toolpath. A ramp would create a temporary tab where it starts the last plunge, then cut it off at the end.

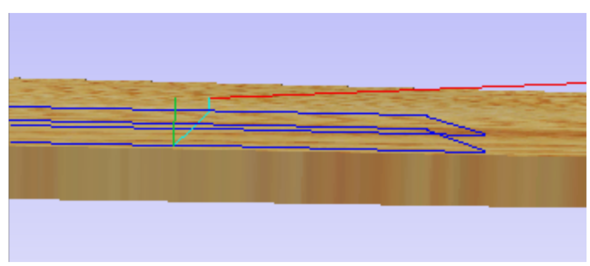
- Moving the start point to a location where the ramp occurs where bit is traveling in a straight line works best for this technique.
  - Click on the rectangle, and move into "node editing" mode instead of "select". The "N" key on the keyboard toggles between **node editing** and **select** mode. Move the cursor around and pay attention to how it changes when the cursor is on a point (node) or on a span.



  - In the top rectangle, the Start Point (green) is in the lower left corner.
  - In the bottom rectangle, a point was inserted in the top span. Put the cursor on the span (line) where the new point wants to be, then Right Click and Insert a Point. "I" is the shortcut.
  - Put the cursor on the new point, and Right Click. Select Make Start Point ("P" is shortcut) to move the start point. The new start point will turn green.



- The aqua lines in the rotated 3D view of the Profile toolpath show how the Z is ramping into the material at an angle when it plunges to the next level. For the final pass, there will be a "tab" at the beginning of the plunge, which will then be cut off when the profile is complete.



- While this does not always keep the part from jumping at the end, it does reduce the effort to get rid of the tabs. Play with the length of the ramps to see what works best.

Suggestion: try the diameter of the bit as a starting point for the ramp length.

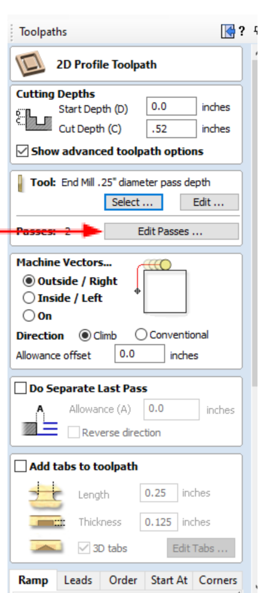
Option 3: Leave a thin **skin** of material on the bottom of the oval:

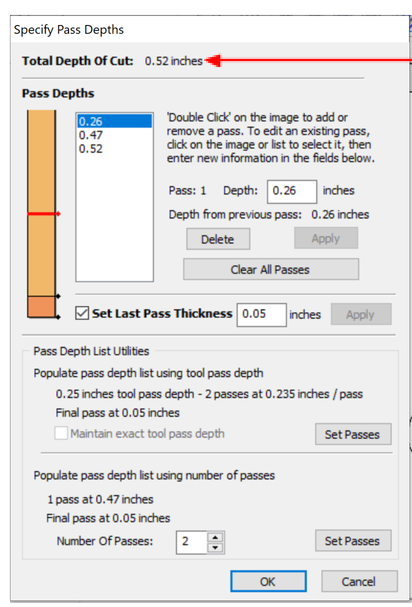
If the desired result is to remove the skin later with a trim router or by planing the project, just reduce the cut depth to be slightly less than the actual material thickness:

- If the Z origin is set to the table surface in the CAD/CAM set up, any differences between the actual material thickness and the cut depth will be at the top of the material, so irrelevant. Remember to Zero the Z at the ShopBot table surface before machining the project **OR**
- If setting the origin at the material surface in the CAD/CAM set up, measure the thickness of the material very accurately. Set the cut depth to leave a thin skin at the bottom of the part. Remember to Zero the Z at the material surface before machining the project.

If the desired result is to cut the part all the way through, but have the last pass be very shallow to reduce the amount of lateral pressure on the material during the cut:

- After setting the Cut Depth (. 52”) and selecting the Tool in the Tool Library, the CAM software will set the number of passes based on Pass Depth, set in the Tool Library.
- In this example, the number of passes = 2.
- Click on **Edit Passes** to open the Pass Depth Editor.
  - Note Total Depth of Cut = .52 inches.
  - Click **Set Last Pass Thickness** and enter 0.05 inches in the box. This allows for extra depth to cut through .5” material + .02” extra, and to leave a .03” skin for the last pass.
  - **Apply**, so now the number of passes will be 3, instead of two. See schematic of passes.
  - The first two passes will remove .47” of material, almost cutting all the way through.
  - The final pass will cut through the material, but the last pass is so shallow that there will not be much lateral pressure on the material, and the part will be less likely to shift during the last pass.
  - After clicking **OK** to accept the changes, the number of passes on the main screen will change from 2 to 3.





Double Sided Tapes and Adhesive

Double-sided duct tape
(image ©Duck Brand)

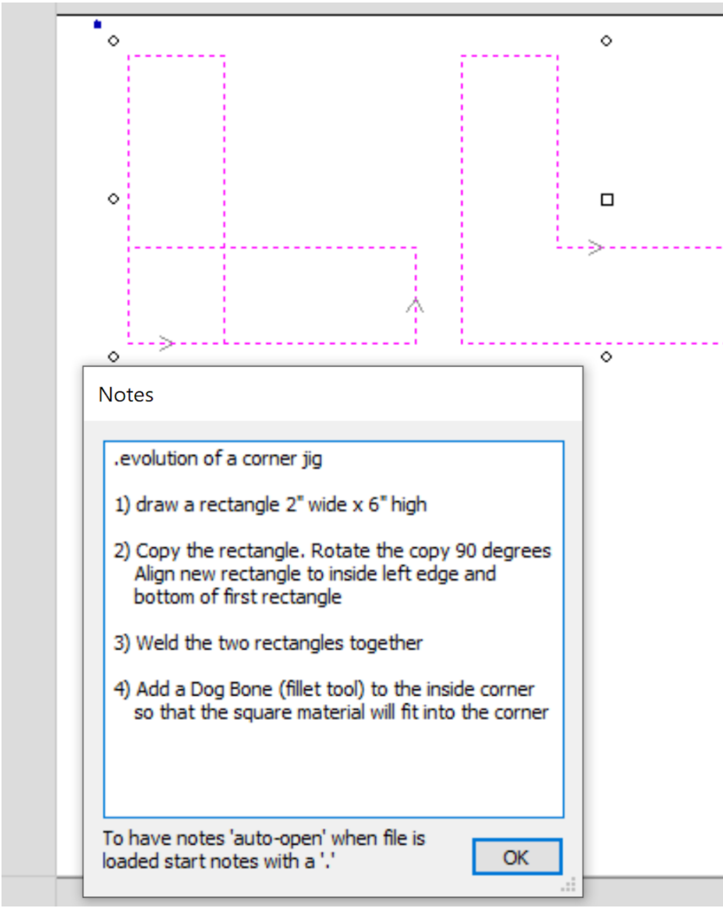
VHB foam tapes
(image ©3M)

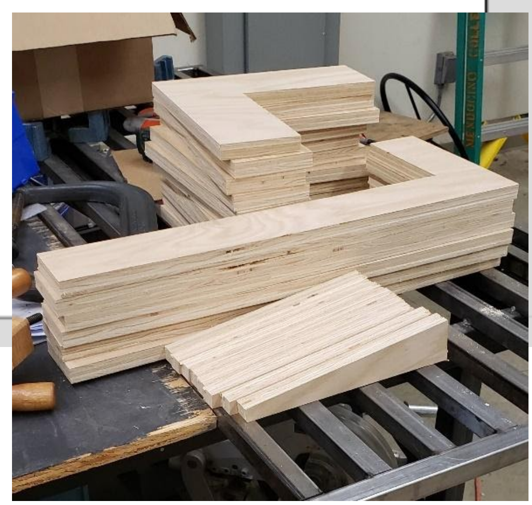
Super 77 spray adhesive
(image ©3M)

- Adhesives and Double Stick Tape are options if there is not a big load on the material.
  - Some ShopBotters like double-sided carpet tape.
  - Use experience to know if the thickness of the tape must be accounted for in the cut depth.
- Avoid wrinkles in tape to ensure material remains flat.
- The longer the adhesive is adhered to the material/table, the more difficult it will be to remove.
  - Some customers have applied masking table to the table surface and to the back of the material.
  - Then, they spray the adhesive on the tape and press the material to the table.
  - Remove the tape from the table and material as soon as the machining is complete to avoid permanently adhering the two together.
- The ShopBot forum is a great place to explore what ShopBot users find effective brands of double-sided tapes and adhesives. www.shopbottools.com/community or http://www.talkshopbot.com/forum/forum.php.

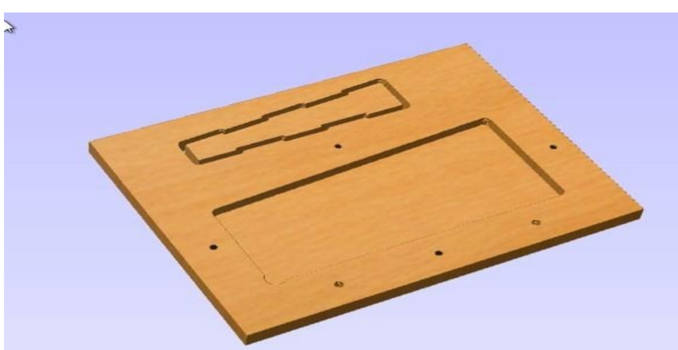
Jigs

Use CAD/CAM Software and ShopBot to create jigs and shims. Example:

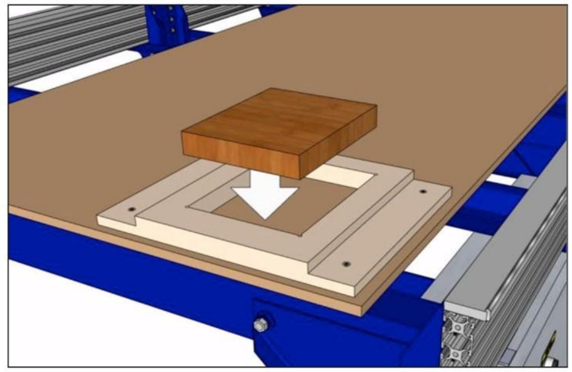




Look in the **Projects and Techniques section** for an example of jig creation.



“Press Fit” Jigs

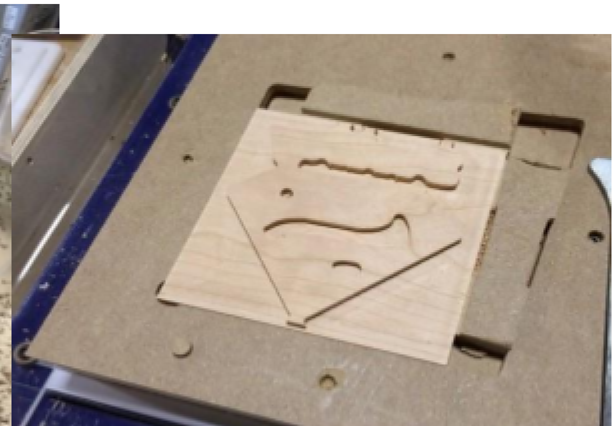


- “Press Fit” jigs are a good option when doing multiples of blanks that are all the same size. However, the fit is subject to changes in humidity or other variables.
- Note how the jig is attached to the table with screws.
- The inside corners of the jig have dogbone fillets to allow the square corners of the blank to fit in the jig.
- The blank is jammed into the jig for machining.
- Consider where the lower left-hand corner of the blank is on the table and adjust the CAD/CAM file accordingly (see below).

Adding a “shim” allows for slight variations in the size of the blank.

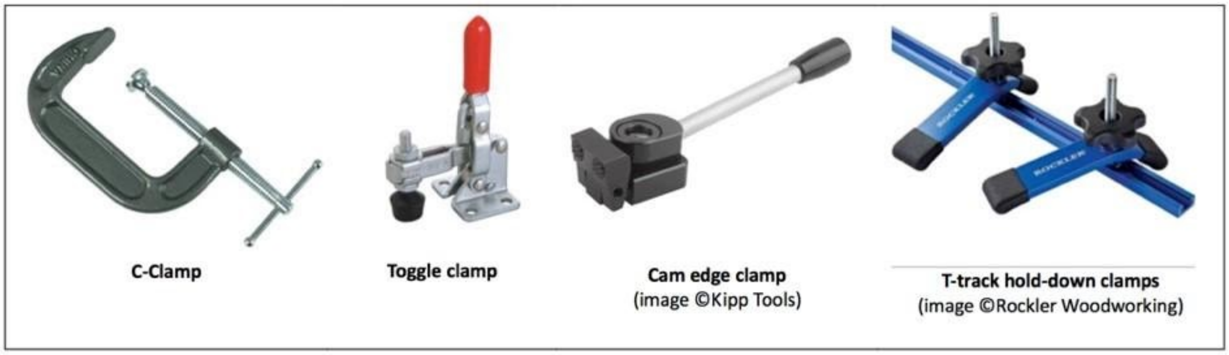
In these examples:

- The jig is attached to the spoil board of the Desktop.
- The jig is big enough to hold the blank and the shims.
- The blank is pressed into the lower left corner of the jig (note dogbone fillets in cut out).
- The tapered shims are tapped into the space between the blank and the jig to firmly hold the blank in place while machining. Note the shims point in different directions.
- In this example, the lower left corner of the design area for the blank is not the same as the lower left corner of the ShopBot table. Compensate for this by:
  - Using the size of the blank as the Job Size for the CAD/CAM file, then zeroing the X and Y at the corner of the blank **OR**
  - Use the size of the Jig as the Job Size for the CAD/CAM file, then draw the location of the blank and shims into the file. Put the lower left corner of the blank at the inside corner of the jig.



Clamps

Commercial Options



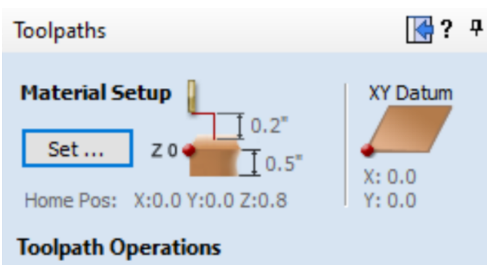
C-Clamp

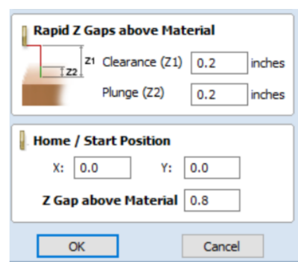
Toggle clamp

Cam edge clamp
(image ©Kipp Tools)

T-track hold-down clamps
(image ©Rockler Woodworking)

Note: When using clamps, remember to use **Material Setup** on the CAM side to set the Z height high enough so the bit misses the clamp when moving around table (Rapid Z Gaps) or going “home” (Home/Start Position).



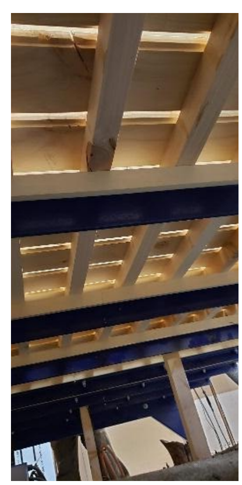


Combining Clamps with a Customized ShopBot Bed

The example below is a custom support/sacrificial level for a 5 x 10 PRT alpha table. Instead of a solid piece of plywood or MDF, the sacrificial level is a series of 1” x 5” boards with an underlay of 1” x 5” boards attached to the ShopBot table cross supports and 1” x 3” boards running parallel to the X axis.

Holes near the edges and slots between boards allow clamps to be slid into position around the material.

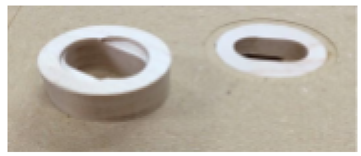
(Occasionally, the bit has been known to touch the end of the clamp…see clamp on end.) Be careful using clamps.



In this example, T-track was inserted into the bed of the ShopBot so that the clamp could be slid into position for different sized material.

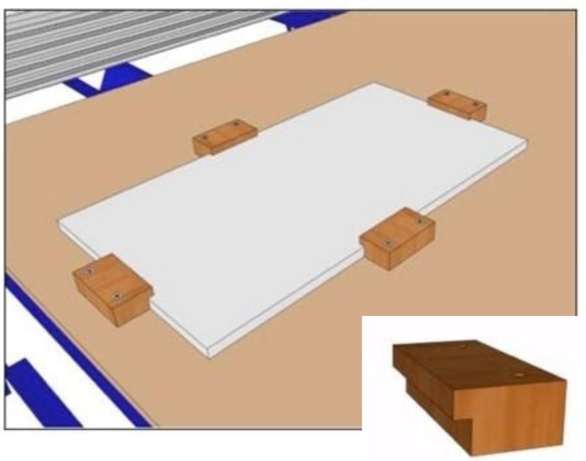
Inlay the track below surface to avoid hitting when cutting through the material.

In this example, homemade blocks were inlaid into the bed.

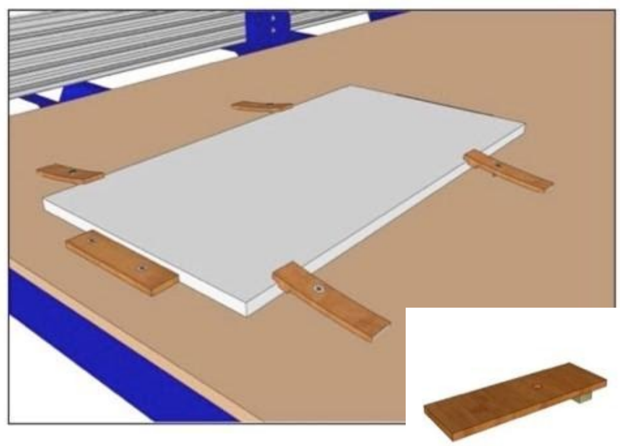


Clamp-like options: Make Your Own

The important point is to capture the material on all sides so that it doesn't move while being machined.



A block of wood with a rabbet can be used on each side of the material. The rabbet height should be slightly less than the material height, allowing it to pull down tightly when screwed in place.

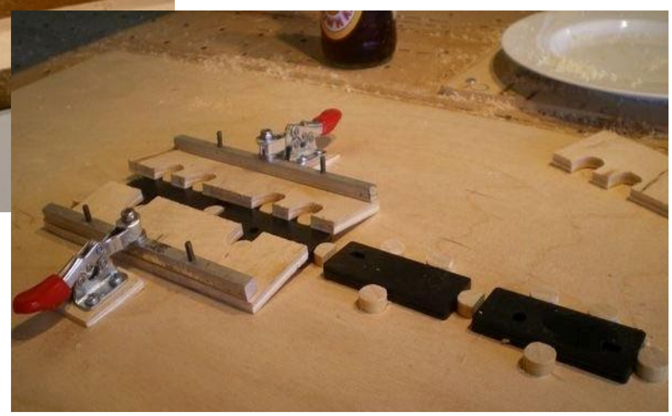


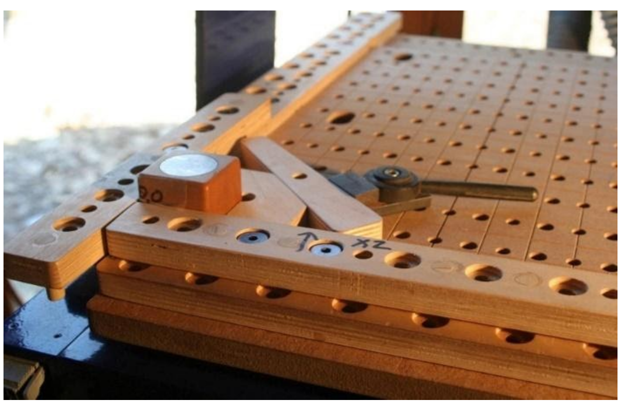
A thin slightly flexible strip of wood can be used on material with varying thickness. Glue a smaller strip across it to butt against the material to prevent slipping.

Improvise: Capture the material on all sides with scrap pieces of wood to prevent movement while machining.

Clamps and Jigs Combined

Examples of ShopBot-created jigs for use with clamps.





Desktop surface modified with holes to receive jigs/clamps.

Vacuum Hold Down

Direct vacuum: jigs and templates

Direct vacuum and jigs or templates are inexpensive solutions that are most practical when making the same parts repeatedly or for machining parts that do not cut all the way through the material.

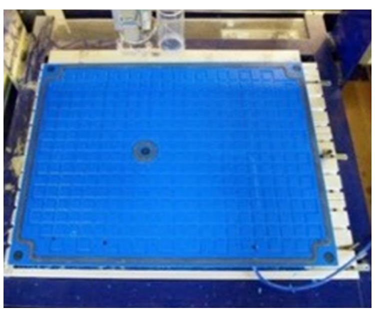
The system makes use of a high vacuum, low flow volume motor such as a shop vacuum, compressed air venturi vacuum pump, mall squirrel cage blower or rotary vane blower.

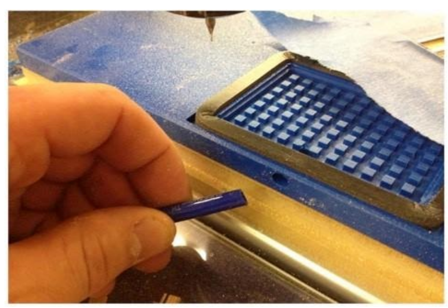
The compromise for using an inexpensive vacuum is that a small leak will cause a large drop in vacuum power. One solution is to create a template for each set of parts to be machined or held in place so that the vacuum is directly under the parts that need to be held in place. The through-cuts take place outside of where the vacuum is holding the part in place, thus not releasing the vacuum.

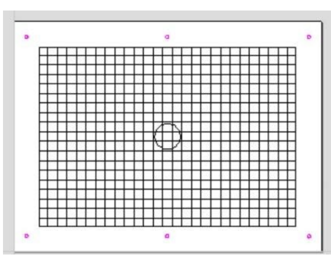
The ShopBot can be used to create the layers necessary for this type of vacuum system.

The Plenum

The plenum is basically a grid with channels through which the air can flow. The plenum includes holes to attach the vacuum supplied by the motor. Gasketing or taping around the edges of the plenum is important to reduce leakage at that level. The examples below are rectangular plenums which would then take a custom template as the next layer.

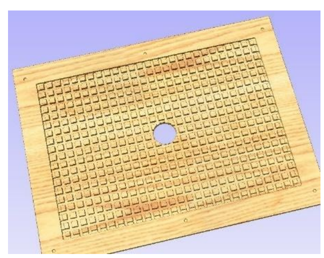






A grid is drawn on the .5” thick material Included:

- A hole for attaching the vacuum hose.
- Markers for attaching the plenum to the table.



A simulation of the completed plenum:

- Markers for hold down: Drill toolpath .1” deep.
- Edge for gasket: Profile on vector .125” deep.
- Grid: Profile on vector .25” deep.
- Hole for vacuum hose: Profile to inside .52” deep.

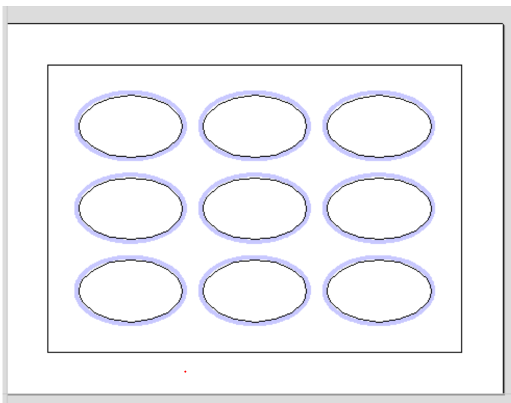
The Template or Jig

A template would sit on top of the plenum to provide a layer between the plenum and the material.

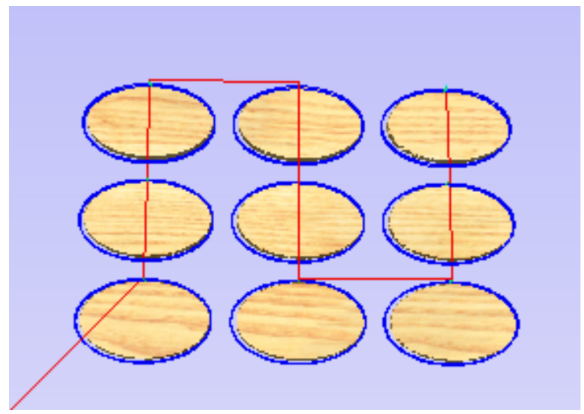
The parts would be held in place with the vacuum, while the through cuts would be made in such a way to avoid breaking the vacuum.

Goal: Cut Out Blanks for Future Use

Step 1: Plan for how the ovals will fit on the material which will fit on the area of the plenum.

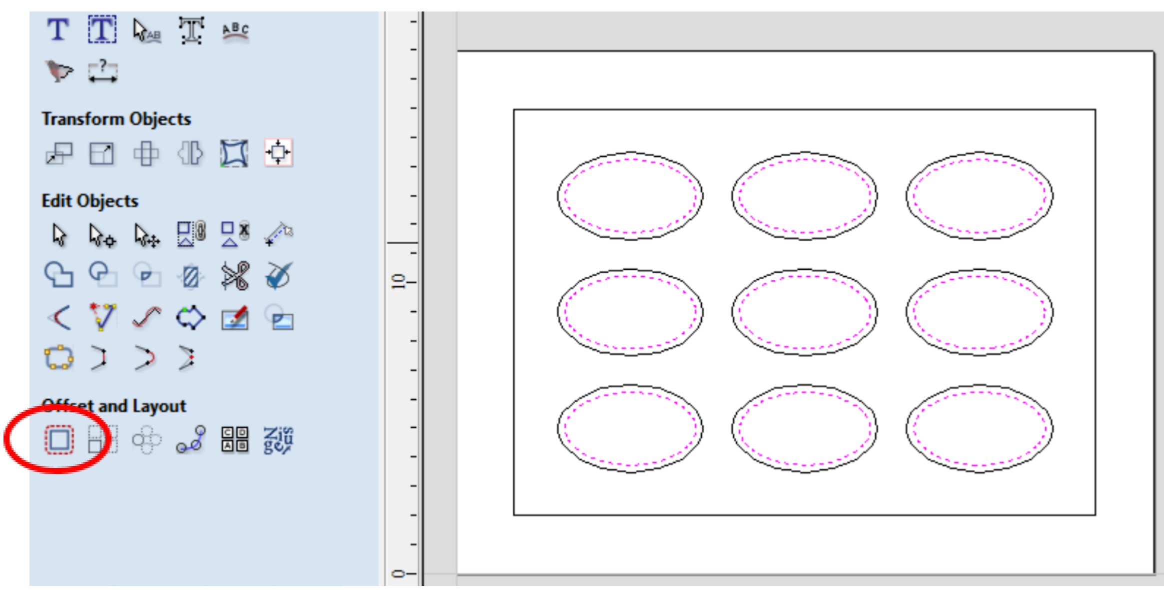


Drawing of ovals on the material
All ovals fit inside the grid of the plenum.

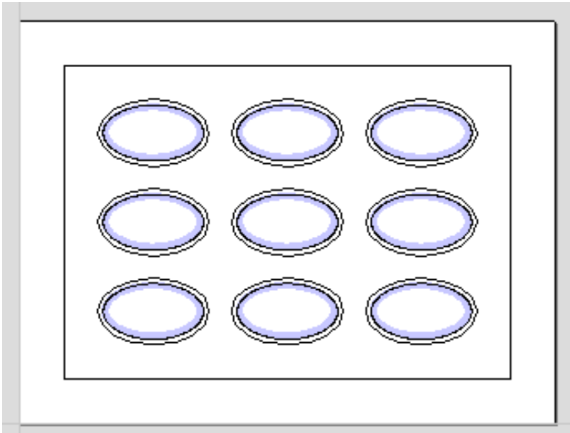


Simulation of the ovals after being
cut out of the material.

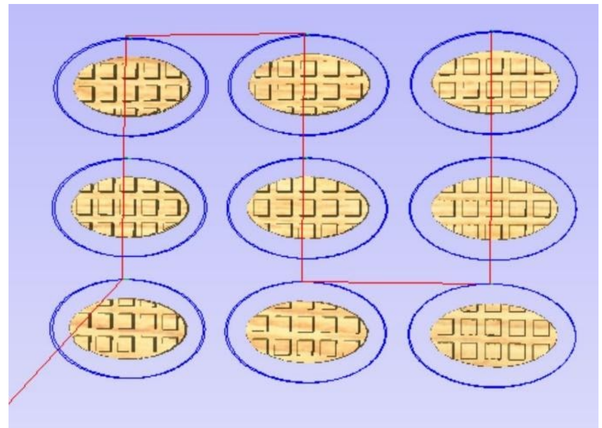
Step 2: Create the template to hold the ovals with vacuum.



Use the Offset tool to offset the cut line for the template inside the cut line for the ovals.



Toolpath the offsets: Profile to the inside .52" (or the thickness of the template material.) Add a hold down for the template to the plenum.



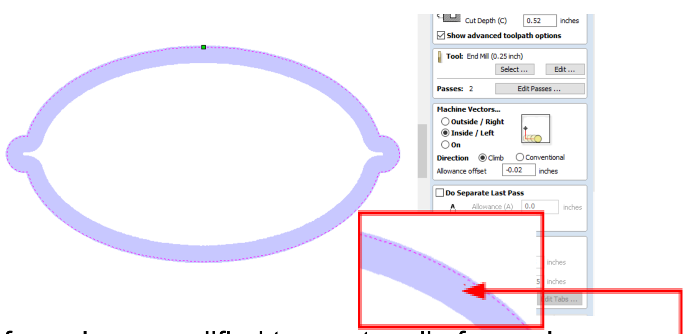
Simulation of the plenum grid showing through the template. The blue cut lines indicate where the through-cuts for cutting out the ovals will go.

Step 3. At this point, it is possible to stop

- Lay the material for the ovals on top of the template.
- Turn on the vacuum.
- Customize the ovals and toolpath/machine them (example: VCarve text in each oval).
- Cut out the ovals using an endmill: Profile to the outside .52" deep.

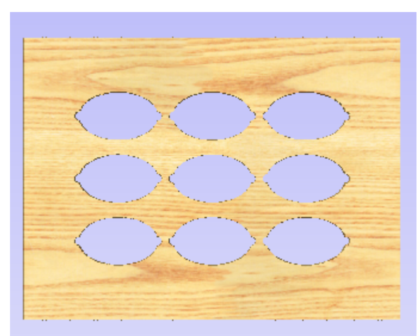
Optional Step 4. Create a jig to hold the ovals for machining later

- Instead of customizing the ovals, then cutting them out, just cut out the ovals.
- Create a jig to place the ovals in for later use. This is a good strategy for classrooms or demos. If VCarving, there is no need to change bit to an end mill to cut out the ovals.
- When ready, place the oval in jig, then customize in the CAD and machine.
- Be sure that all the holes for ovals are filled or covered to keep the vacuum intact!



Detail of new layer modified to create a jig for ovals.

- Circles welded to edges of ovals to create finger holds to easily remove ovals from the jig.
- Toolpath: **Profile** to the inside with a -.02" allowance added to allow ovals to fit inside jig.
- Detail (toolpath 2D View Solid) shows holes will be slightly larger than the drawn vectors.



Simulation of the jig for ovals Remember to add hold down for jig.

A Note About Gasketing

Templates and jigs hold down parts better when they have a gasket to help seal the vacuum.

Gasketing material works best when it is compressed into a groove when the material is lain on top and the vacuum is turned on.

Use the ShopBot to create the groove for the gasketing material.

One source of gasketing materials:

http://www.allstaradhesives.com/

Example from ShopBot Production:

- On the left side, parts have been cut, and waste material removed.
- The finished parts are still sealing the vacuum.
- The right three columns had no material in place, so holes for vacuum hold down have been taped off to keep the vacuum intact.

Universal Vacuum Systems: Plenum and Bleeder Board

Whole Table or Universal Vacuums are a great solution when project designs change frequently or production requirements demand a quick turnaround of material. The material is laid down on the bleeder board. When the pump is turned on, a vacuum is pulled through the bleeder board, holding the material in place during machining.

ShopBot Table Set Up Includes:

- Support or base board on the bottom layer.
- The middle "plenum" layer has channels for air flow to pull the vacuum. The plenum is divided into zones which can be opened or closed with the manifold.
- The top layer is a bleeder board through which the vacuum is pulled. The bleeder board is also the sacrificial board.

The vacuum pump must be very powerful to pull a vacuum through the bleeder board to hold the material in place. Two options available through ShopBot include the Becker Rotary Vane, and the Republic Regenerative.

Pumps are heavy and noisy, so plan for placement of the pumps when designing the space.

https://www.shopbottools.com/products/accessories/vacuum

Plumbing under the table connects the zones to the manifold. The operator uses valves on the manifold to open/close zones on the vacuum table to reflect the size of the material being machined.

Suggestion: Label the zones on the manifold for easy reference.

Visit the ShopBot website for more information on installation and maintenance of universal vacuum systems: https://www.shopbottools.com/support/documentation/accessories

Zones and Vacuum Hold Down

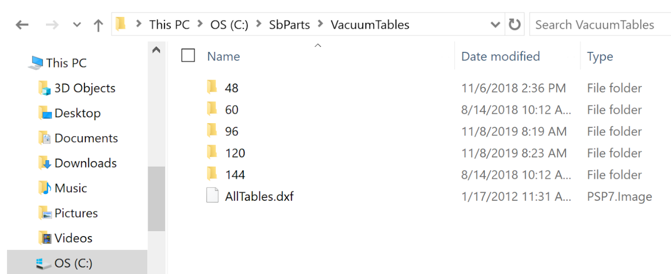
The ShopBot creates the plenum (in zones) and through-holes for vacuum plumbing. The set up shown is awaiting its bleeder board, which will be glued to the plenum.

Notice in the photo to the right that there are no channels for vacuum around the edges and in between zones. If the parts to be held down are large so that there is plenty of surface area to be held in place with the vacuum, the “dead” areas of the vacuum table shouldn’t be relevant.

If the parts are smaller, it is important to take the location of the dead areas into account when laying out the parts on the material.

Creating the Plenum

There are .sbp files for creating the zones in the SBPart folder.



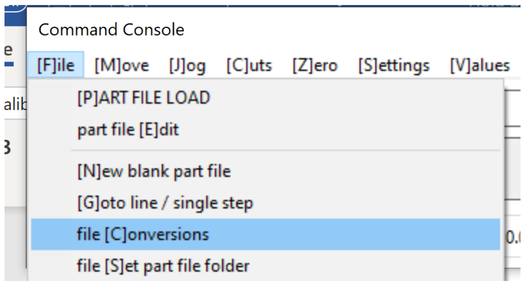
To find the SBParts folder, go to This PC > OS(C:) > SBParts > VacuumTables > then choose the size of the ShopBot table. Tables bigger than 4 x 8 have more zones.

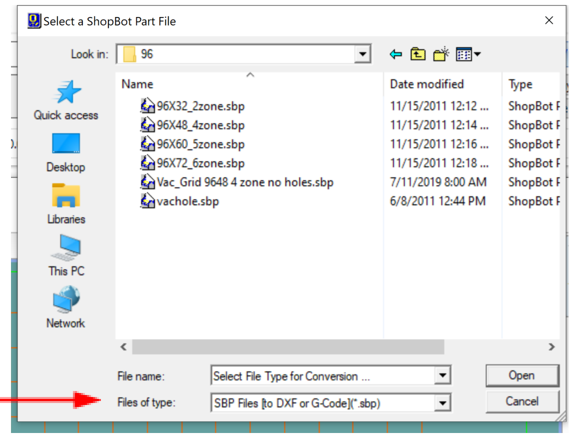
To create the plenum and accept the placement of the holes, just run the .sbp files. Instructions are included in the files.

To customize the existing design, use one of the "**Hidden Gems**" of ShopBot Control software to convert the .sbp file to a .dxf file for import into VCarve Pro/Aspire (or other CAD/CAM software).

Aside: Converting an .sbp to a .dxf for import into CAD

In the SB3 software, under **File**, there is a **C**onversion Command.

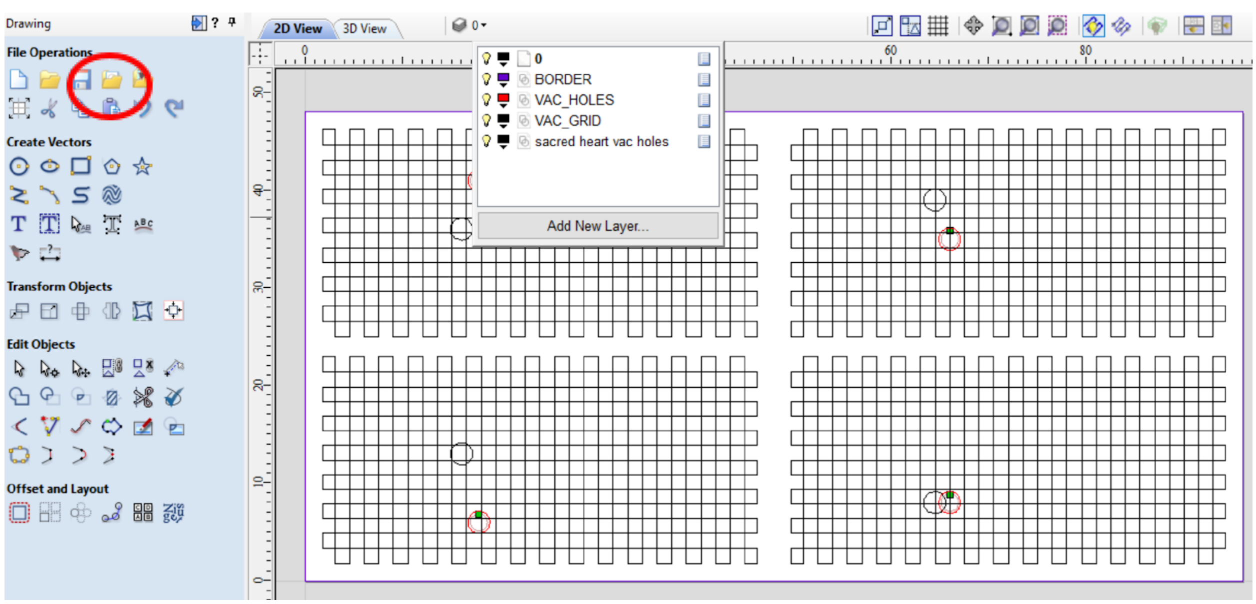




When the screen opens to select which file to convert:

- Select the VacuumTables folder in SBParts.
- Change the **Files of type** to SBP Files to DXF.
- Select the file to convert.
- Answer the questions. The converted file will be renamed and saved as a .dxf file.

When the .dxf file is imported into VCarve Pro, it is in layers, so information can be turned on or off by clicking on the light bulb next to the layer. In the example below, some of the imported layers have been deleted. The vacuum holes in the original file were not in the desired location, so another layer was created, and the holes moved. Only the new holes will be toolpathed.

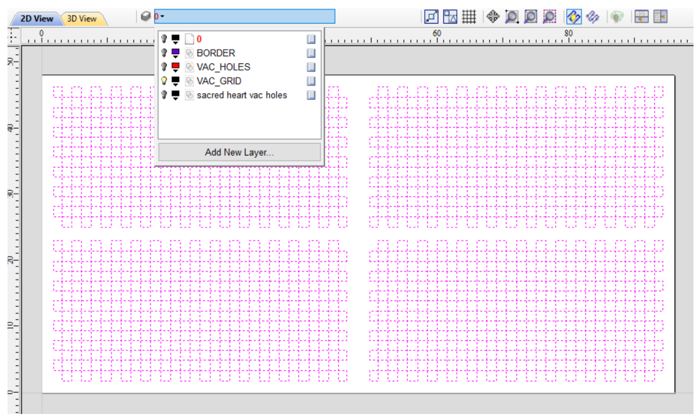


Using the Plenum Grid Layer for Laying Out Parts

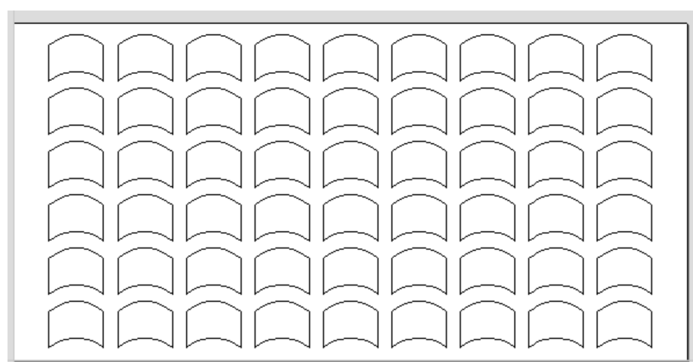
Even after the plenum has been created, having the information about the location of the zones and dead areas can be critical for vacuum hold down. It's easy to move the grid pattern into a CAD file.

One feature of VCarve Pro is that the vectors from one session of VCarve Pro can be copied to the same location in another session of VCarve. In the example below, only the vacuum grid layer is selected:

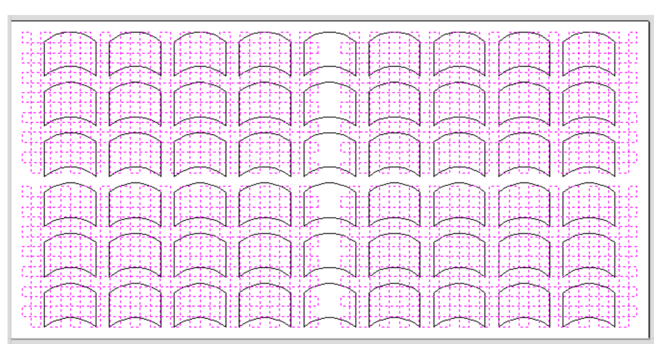
- Turn off all layers except the one with the grid.
- Select and **Copy** the grid.
- Open a fresh session of VCarve.
- **Paste.**
- Check that the grid came in on its own layer.
- Now might be a good time to create a Reference File with the grid on it. Example: **Save As** Reference Plenum Grid.
- To make designing easier in the new file, turn off the layer with the grid on it.



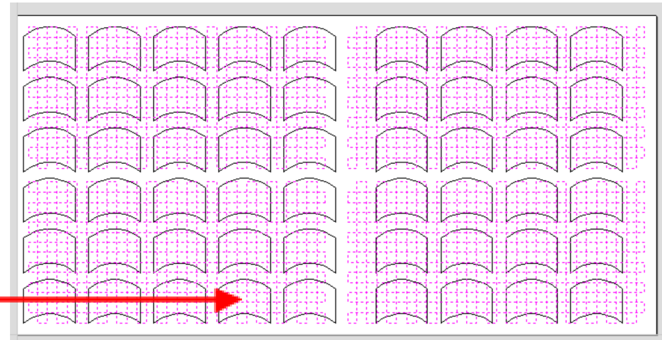
- In the example, a single part that is approximately 7.75” x 6.5” has been arrayed and the array centered on the material.
- Each part has a fairly small surface area, so a part may shift during aggressive Profiling to cut it out if there is not sufficient vacuum under the part.



- Turning on the underlying grid indicates that the center column of parts is the most vulnerable.



Options



- Delete the center column of parts.
- Add additional hold down such as Tabs to the center column.
- Shift some of the parts over to have more over the vacuum.

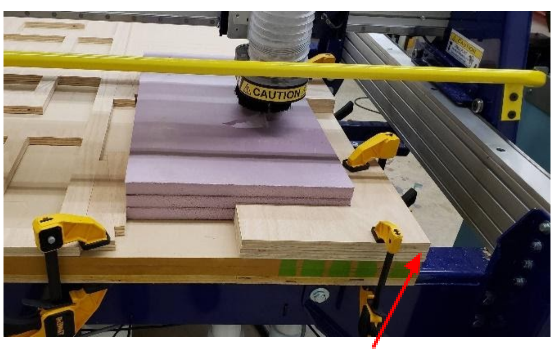
Working with Material Smaller Than the Full Table

Use the manifold to shut off zones that don't have material over them.

- In the 5-zone manifold on a 5 x 8 table shown here, all 5 zones are open (green valve parallel to the flow of air.) For a 4 x 8 sheet of plywood, all zones except zone 5 would stay open.
- If closing more than half the zones, watch the gauge on the manifold. If necessary, slightly open the relief valve on the manifold to keep the pressure (heat) from building up at the pump.

See the laminated guide for working with vacuum.

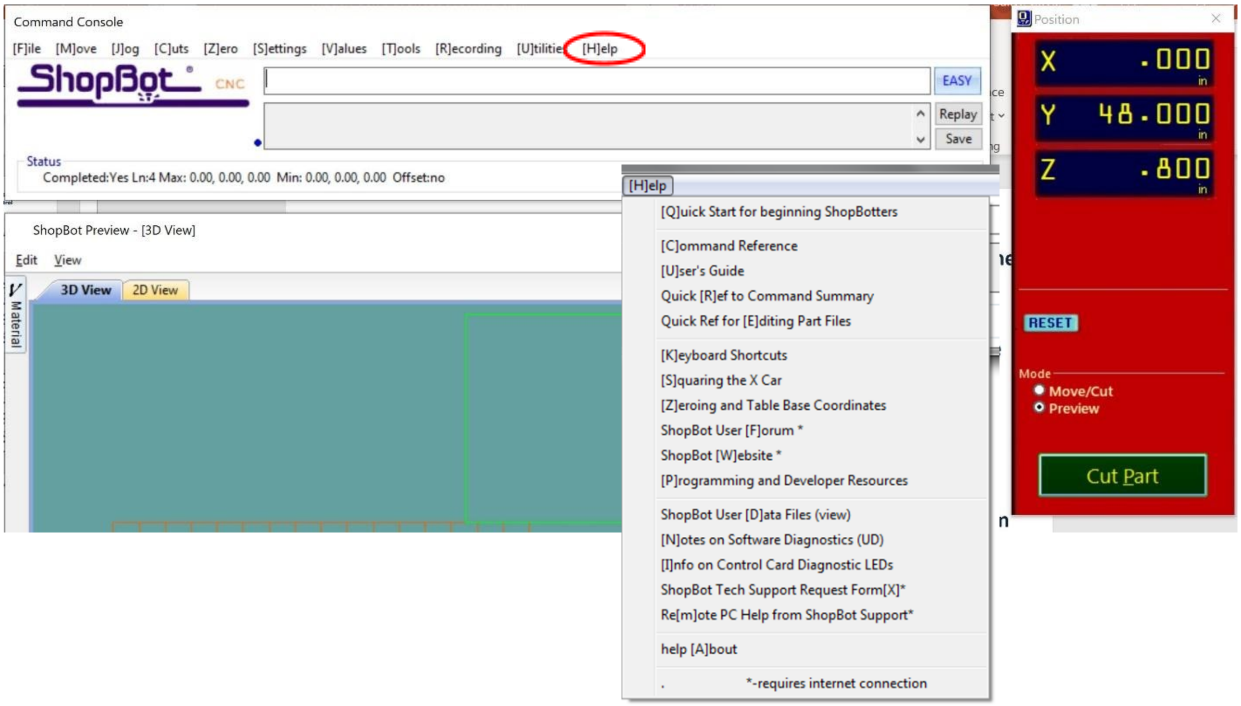
When Material Size is Smaller Than a Zone:



X, Y 0,0 set using Automatic Zeroing routine

- Additional jigs and/or clamps may be necessary (Recognize those jigs? See cut outs from a full sheet of plywood?).
- Make allowances for the corner of the material not matching the X,Y Zero set using the Auto Zero routine in the corner of the table.
- Options:
  - Set CAD file to actual size of material, then manually Zero the X and Y at the corner of the material (Safety and Basic Use Page 27).
  - Draw location of material in CAD file that has 0,0 at true lower left corner of table. Offset corner of material by dimensions of the jig.

Hidden Gems In SB3 ShopBot Control Software



This Chapter explains some of the commands in **SB3 ShopBot Control Software** that might be useful for improving the efficiency of the workflow. For example, use **[V]**alue **[A]**xis to define the value for the current location of the bit. Other commands are just fun to know about.

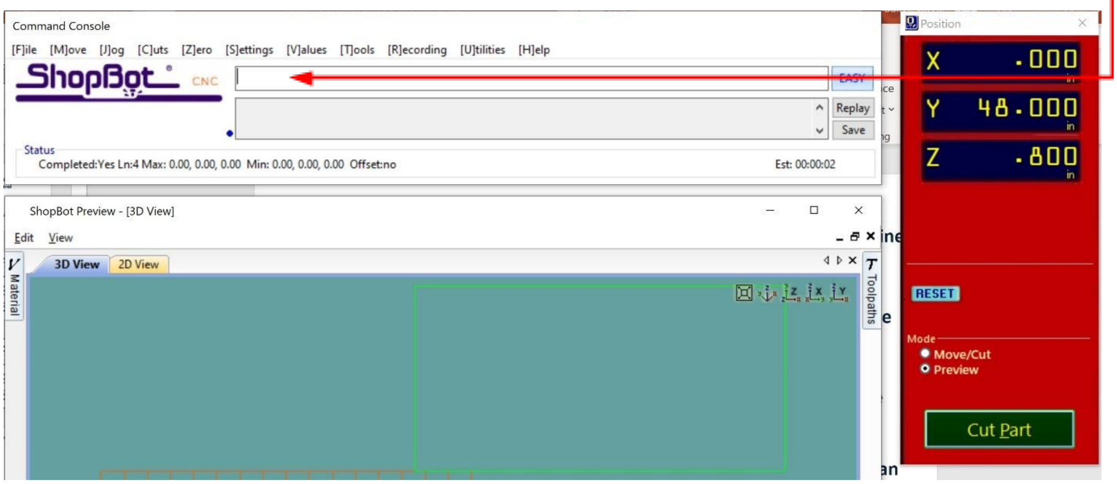
Suggestions for **Troubleshooting** problems that may be a result of Values set incorrectly begin on page 10. (Example: ShopBot slowing down too much in corners (**VR**)).

The **[H]ELP** menu of the ShopBot Control Software has documents with more detailed information about SB3 ShopBot Control Software. Check out the **[C]ommand Reference** for quick and detailed information on the commands mentioned here.

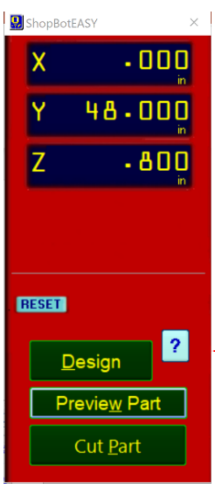
SB3 ShopBot Control Software: Full Mode

The **Tool Bar** has a list of Commands. Click on or enter the first letter of the command to access the pull down menu, then select/enter the second character.

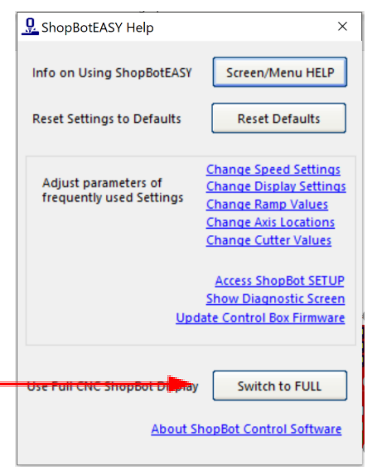
Two-character commands may also be entered in the Command Line or written as a line of code. Some commands require more information, such as value for location or file name.



If the TOOL BAR is Not Visible, the Software is Probably in ShopBot Easy Mode
To switch to **FULL.**



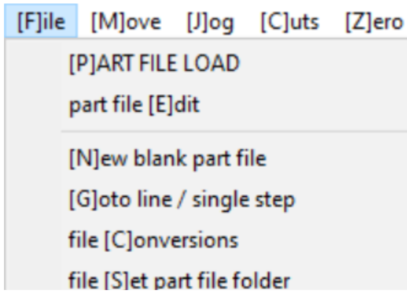
- Click on Question Mark.
- When new screen pops up **Switch to Full.**

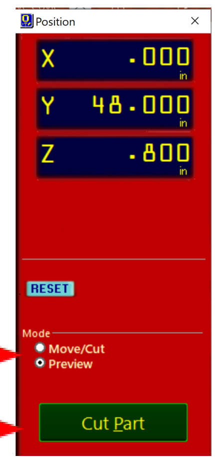


SB3 ShopBot Control Software is not the same as **G-Code**, although it does the much the same thing (tells the machine where to go, what to do when it gets there, and at what speed).

- Developed in the late 1990's, ShopBot Code contains commands that are important for the functioning of the machine.
- ShopBot Code is actually simpler and more modern than G-Code.
- If G-Code is preferred, SB3 ShopBot Control software will read some G-Code commands (see [**E**]diting and [**P**]rogramming documents in HELP).

The File Menu





FILE Commands that Run ShopBot Files.

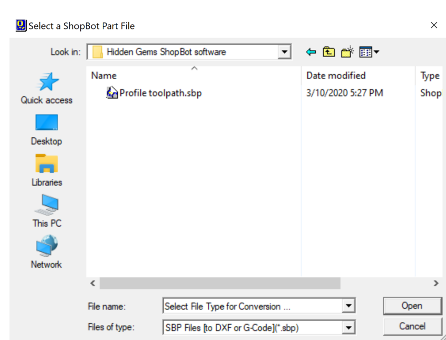
- **FP**: PART FILE LOAD loads a ShopBot Part (.sbp) File. A ShopBot file can also be loaded from the **CUT PART** icon on the Position Screen.
  - In **Move/Cut** mode, the ShopBot will move.
  - In **Preview** mode, the file will be displayed on the screen with the toolpath defining the size of the material displayed (default).
- **FG**: GoTo line/single step runs a ShopBot file from a given line in the ShopBot code. Software pulls up the last line run when a previous file was stopped before completion.

FILE Commands for Reading/Editing ShopBot Files.

- **FE**: File Edit displays the code for a ShopBot Part (.sbp) File
  - The code is a text file, so can be opened in any text program, such as NotePad, Word, etc.
  - Some aspects of the .sbp can be changed easily without having to go back to the CAM file to rewrite the code. Examples: Move Speed (MS) Spindle Speed (TR) and Tool Number.
  - Remember to SAVE the changes to the file.
- **FN**: Opens a blank page to start creating your own .sbp file.

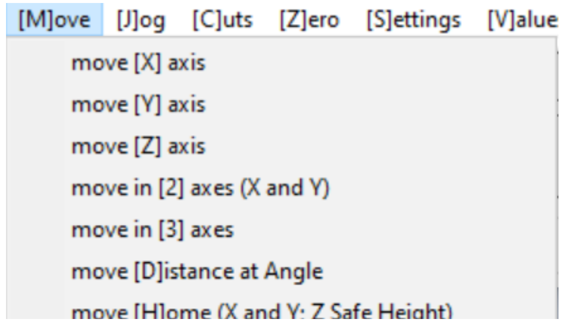
FILE Commands for Converting Code to/from .sbp

Example: You have a ShopBot Part File (.sbp) and you want to manipulate it in a CAD files such as VCarve Pro or Aspire.



- **FC**: File Conversions can convert a .sbp file to a .dxf to bring into a CAD/CAM software for manipulation.
  - Click on the Down Arrow under **File of Type** to see what can be done with a file.
  - This only works for 2D files, and is not an ideal solution…but it's there.
  - See example on page 22 of **HOLD DOWN** Chapter.

The Move Menu



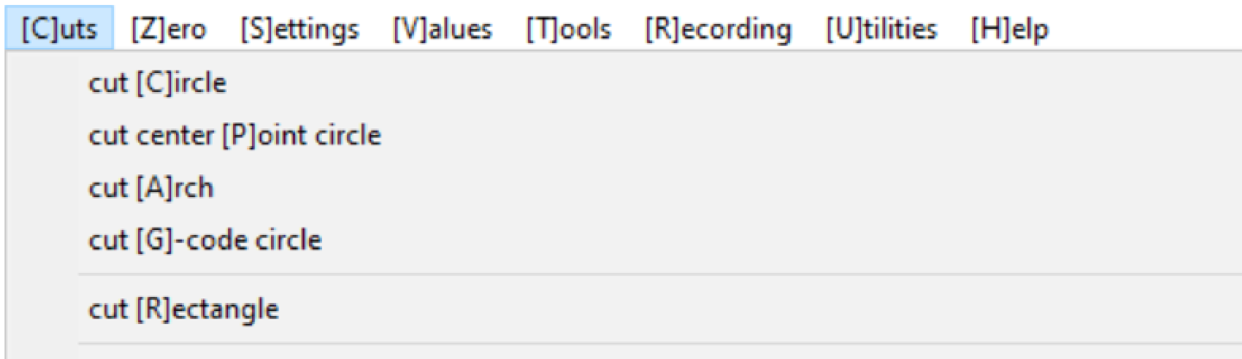
The MOVE (and JOG) Commands Tell ShopBot Where to Go
(and How Fast to Go There).

- Example: to move the X to 25, and the Y to 30, the command is **M2, 25, 30.**
  - Define the axis (axes) to move.
  - Enter the value(s) of where to go.
  - If more than one axis is defined, the order is always X, Y, Z.
  - Use a comma to delineate values.
  - If the tool is set up in Absolute mode (default), then the distance is determined from the 0,0,0 location. The tool is constantly self-correcting the distance from the origin.
  - A command to move 2 or 3 axes simultaneously (**M2** or **M3**) may result in the tool moving diagonally.
- A ShopBot Part (.sbp) file is mostly a series of **M**ove or **J**og Commands.
- To view ShopBot code generated with CAM software, or write custom code and save it, use the File Edit (**FE**) command.

Other Commands Under MOVE (and JOG).

- **MH**: Move Home.
  - Z moves up to Safe Z set under **VC**
    then moves to current 0,0 at current move speed (**MS**).
  - Note: The Z only travels up if current Z height is less than Safe Z Height
    set in VC command.
  - It does not move down to the Safe Z height.
- **JH** does the same thing at positioning speed (faster). There is an icon for JH on the red Positioning screen.
- **MS**: Sets the Move (Cutting) Speed).
  - **MS** is a command written into ShopBot Code from the CAM program (determined from values the Tool Library).
  - **JS** sets the Jog or positioning speed.
  - **VS** sets all the speeds in one place.
  - The Move and Jog speeds can also be input directly from the Command Screen.
- **MI**:
  - Sets rotary indexer to lathe mode OR
  - Sets Z to move up and down…good for knives cutting cardboard.

The Cuts and Custom Menu

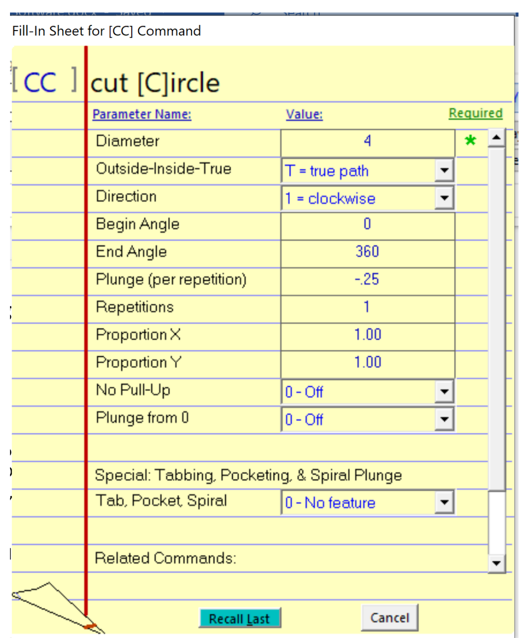


CUTS: Create Circles and Rectangles in ShopBot Software:
No Other CAD/CAM Needed.

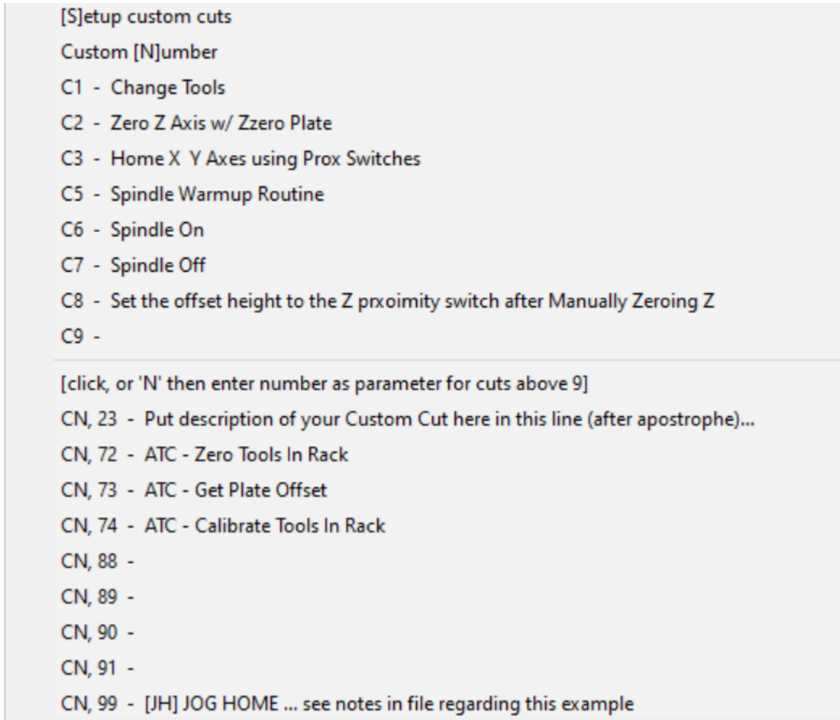
Sometimes, it's convenient to be able to cut a circle or a rectangle without having to write a CAD/CAM file. There remains in the SB3 ShopBot Control software a list of commands to create a circle, an arc, or a rectangle without any additional CAD/CAM software.

These commands run from the current location of the bit.

- Example: Cut Circle (**CC**): Enter the:
  - Diameter (Size).
  - Choose path and direction.
  - Start location (Begin angle 0 = top of circle.
  - End location (End angle 360 = top of circle).
  - Enter Plunge depth (= pass depth) as a negative value from Z-Zero. (Turn on Plunge from 0 option).
  - Repetitions = Numbers of passes. Plunge x Repetitions = total pass depth.
- Commands can be recorded (**RA**) and linked together to write a repeatable file.
- The **Quick Reference to Command Summary and Command Reference guides** found under **Help** include help writing .sbp code using the Cut Commands.



CUSTOM: Call up Macros that have been Created in the ShopBot Software.



Many commands use Variables entered by the operator, set in the CAM software, or set in routines. Variables and Custom macros are stored in the SBParts folder: C: > SBPARTS > Custom > **myvariables**. Set File Type to All Files to make the Custom files visible.

Commands for Setting up all ShopBots.

- **C2**: Run the Z-Zero routine using the Z-Zero Plate. (Use TS to change the value of the thickness of the Z-Zero Plate used in the Z-Zero routine).
- **C3**: Run the XY Zero routine using the Proximity Switches. (Use TS to set up the values used in XY Zero routine using the Proximity Switches).
- **C5**: Run the Spindle Warm Up Routine. (Tools with Spindle Speed Controller only. **TR** to open the spindle speed screen).

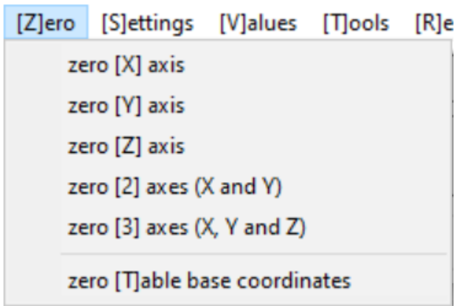
To Write a Custom file (Macro).

- **CS**: Find a template and more info here.

Commands Specific to ShopBots with an Automatic Tool Changer (ATC).

- **C1**: Change Tools in Tool Holder.
- **CN, 72**: Zero Tool in Rack. Note: Zero a Tool in the Rack whenever a new bit is inserted into a Tool Holder.
- **CN, 73**: Get Plate Offset. Note: Run the Plate Offset (Z Zero Plate) after re-surfacing the Sacrificial Board.
- **CN, 74**: Calibrate Tools (Holders) in Tool Rack. Note: Recalibrate the Tool Holders in the Tool Rack after changing the XY 0,0 routine using TS.

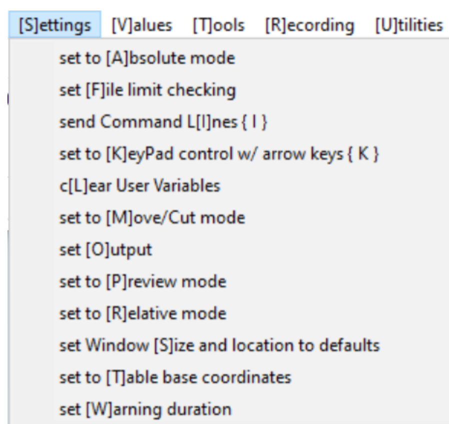
The Zero Menu



- Defines the Zero locations for the axis (axes) at the **current location of the bit.**
- The Blue Zero Button on the Keypad screen also sets the zero location.
- These commands do not move the tool, but redefine the Zero location in the SB3 ShopBot Control software (watch the location values on the red Position screen).
- See the section on Zeroing the ShopBot in the **Safety and Basic Use** Chapter for more details.
- (Use the **C2** (Z-Zero) and **C3** (X,Y-Zero) Commands or Icons to run the Automatic Zero Routines).

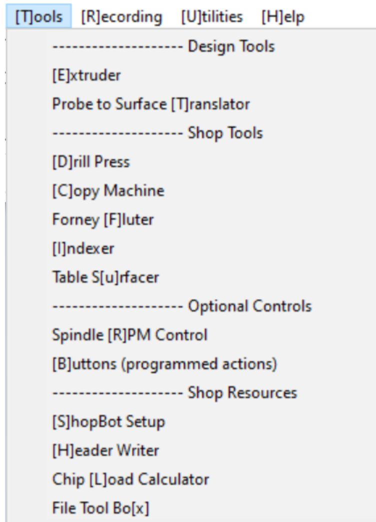
The Settings Menu

Most Commonly Used SETTINGS.



- Most Settings can be changed from icons on the Position Screen.
  - One can use 2 letter commands in the Control Console to change the settings.
  - Some Settings are written into ShopBot Code from the CAM software.
- **SA** or **SR**: Sets the software to run in Absolute mode OR Relative mode (one or the other).
  - Absolute mode (usual) defines position from Origin (0,0,0). It is the most accurate mode.
  - Relative mode defines position from last position. It may introduce error with each move.
- **SM** or **SP**: Sets the software to be able to Move the ShopBot around, OR to be able to Preview a file on the screen.
  - If there is no ShopBot connected to computer, Preview is the only option.
  - Keypad Control (**K**) will not open in Preview mode.
  - Tool will not move and spindle will not turn on in Preview mode.
- **K** (or **SK**): Opens up the Keypad to be able to move the ShopBot around using the arrow keys on the computer keyboard or Keypad on screen. Yellow icon on Position Screen also opens Keypad.

The Tools Menu



Setting up the ShopBot: Tool Setup (TS)

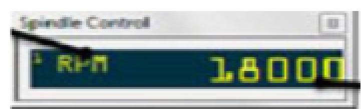
- Set Control Software to inches or mm.
- Check that settings in the SB3 Control software matches the physical ShopBot.
- Use VL to change table size when customizing ShopBot (example: Switching X & Y on the Buddy).
- Adjust the values of the thickness of Z Zero plate for Z Zero routine (C2 or Z Zero icon).
- Set up the values for the automatic X and Y Zero routine using Prox Switches (C3 or XY Zero icon).

Suggestions for Feeds and Speeds

- Chip Load Calculator (**TL**).

Turn on Spindle Speed Controller Screen

- **TR** opens the screen to adjust spindle speed. Enter new RPM value and "Enter" to change spindle speed.
- Spindle Speed Control is not functional on Desktops: Use the knob on the VFD to change the Frequency/RPMs of the spindle. See **SBU: Working with the Spindle** for more information.



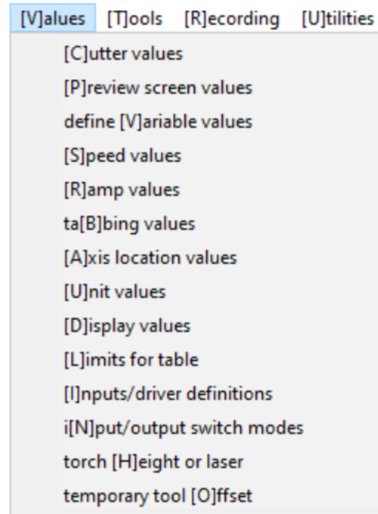
Hint: to have the Spindle RPM screen open when the ShopBot software starts, Right Click on the **RPM** in the upper left corner and change Start when SB3 Starts from 0 to 1.

Click OK to save the settings

Explore Fun Programs for Special Effects

- Fluting.
- Molding.
- Indexing (lathe).
- **Copy Machine** sets up files for a Digitizing Probe.

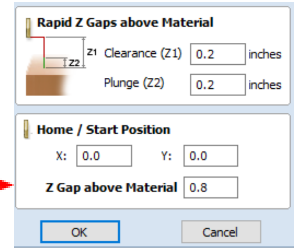
The Values Menu



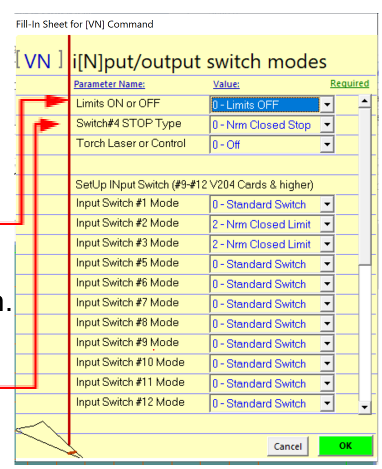
VALUES Contains Really Useful Commands.

- **VC**: Sets Safe Z Height for Z Zero plate routine. Also sets diameter of bit if using ShopBot routines like **CC.**
- **VR**: Sets "Ramp" values to control speed at which SB takes corners. See HC for more information about Ramps.
- **VA**: Set new value for current location of bit: X, Y, Z. If 5 axes are displayed, A and B location also.
  - Use this command to define a new working region of the tool, define a known location if you've lost position, or have the ShopBot cut deeper in the Z if it is not cutting through the material.
  - Example: the bit is not cutting all the way through the material.
    - Goal: Redefine the current location of the Z as slightly higher, so the bit will cut deeper.
    - Move the Z to an easy to calculate location (MZ, 1 for one inch above zero).
    - Open **VA**, and change the Z value from 1.00"to 1.02". The location of the bit does not change, but the computer thinks the Z is higher.
    - The next time the file is run, the bit will go .02" deeper than the previous run.
    - The Z axis is defined as that location until the Automatic Z-Zero routine is run again.
  - Example: to define a known location if you have lost position.
    - From the CAD file, find an object (a hold down mark?) with a known position (say X,Y=4,6).
    - Use Keypad Control to move the bit over that object (use the FIXED button to get closer).
    - Open **VA** to define the current location of the bit as X = 4, Y= 6.
    - That action also re-defines the 0,0 location until the Automatic X,Y Zero routine is run again.
- **VU**: The Unit Values are set when the .ini file is loaded at set up.
  - If the ShopBot is not moving the correct distance, Unit Values may be a place to look.
  - Use UR to reload the .ini file for the ShopBot model. See next page for more info.
- **VD**: Set up number of axes viewed (3 or 5).
  - Set A and/or B axis to linear or rotary.
  - Turn on and off log writing file (keeps a record of every .sbp file run, whether completed or not).
- **VI**: Change definition of axes in software.
  - Example: Change the X and Y axes in the Buddy ShopBot.
  - If you switch the axes in the software, use **US** to **Save** a Custom .ini file so you don't have to re-enter the changes if you have to reload the .ini file.
- **VN**: Turn on and off limit switches, set estop mode (#4) and change limit switch definition (inputs 2, 3 and 5).

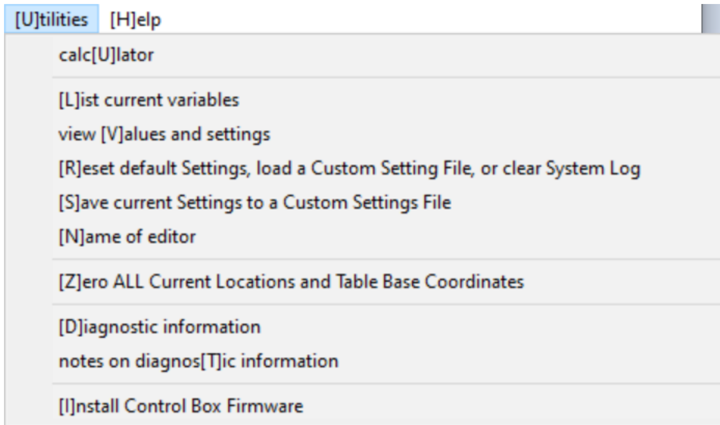
Troubleshooting: Do You Need to Change a VALUE?



- Example: Cutting thick material with a long bit. **If running the Z-Zero routine causes the Z to top out**. Use **VC** to change safe Z Height for ShopBot-controlled routines. Examples:
  - Z-Zero routine which uses Z-Zero plate.
  - Jog Home (**JH**) or Move Home (**MH**) routines.
- Caution: CAM files also set heights (Gaps) above the material when moving to a new position (Rapid Z) and going to Home at end of file.
  - VCarve/Aspire: look under **Material Setup** under Toolpathing to change Rapid Z Gap and Home Z.
  - Remember to rewrite the .sbp file after changing the toolpath.
- **ShopBot is either slowing down too much when going around corners/3D carving or banging into sides when changing directions.** Open VR. Use the "reset" button (lower left corner) to reset to factory values.
- **The ShopBot is not moving the correct distance when given a Move or Jog Command.**
  - Each type of ShopBot has a given "Unit Value" which is determined by driver, motor, pinion gear, etc.
  - When the .ini file for the model of ShopBot is loaded, standard unit values are loaded.
  - Page through **TS** to see current settings loaded in the SB3 ShopBot Control software.
  - If the settings are incorrect, use **UR** to reload the correct .ini file.
  - Use **VU** to see the unit values themselves. If the ShopBot still does not seem to be moving the correct distance, contact ShopBot Support for help with getting the correct Unit Values.
  - A negative sign in front of a Unit Value changes axis direction.
- **Turn on Limit or proximity switches to stop the tool before it hits a hard mechanical stop.**
  - The .ini file loads settings for input/output switch modes.
  - By default, the Limit switches are turned off.
  - When running the XY-Zero routine, the switches are turned on, then returned to their previous value at the end of the routine.
  - Use **VN**, TURN THE LIMIT SWITCHES ON.
  - If switches are being falsely triggered by external events (fluorescent lights, etc.), they can always be turned off again.
- **Stop Switches**: can be normally closed, or normally open.
  - Check here that the type of switch matches the tool.
- **Input switches 2, 3 and 5** are the limit or proximity switches.
  - If there is an indicator light showing up on the red Position Screen when a switch is not triggered, try changing the values to normally closed (usually input switch 5).



The Utilities Menu



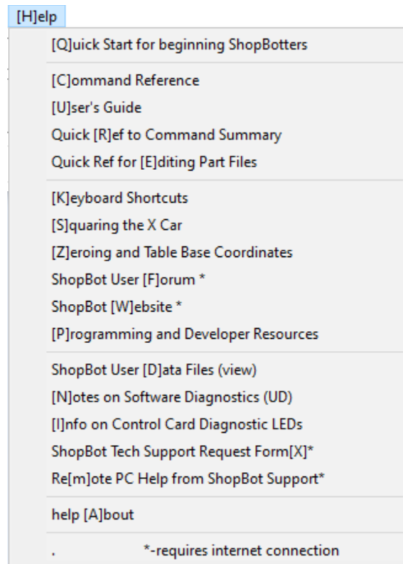
Most Commonly Used UTILITIES

- **UR**: Reset .ini file for ShopBot when:
  - First setting up or if the SB3 Control Software seems to have gotten corrupted (ShopBot misbehaving).
  - Switching between different ShopBots using the same computer (remember **TS** to change zeroing routine values).
- **US**: Save the .ini you have created for YOUR custom ShopBot (Example: switched X and Y axes on the Buddy).
- **UI**: Use when a new version of the SB3 ShopBot Control Software needs to update the firmware on the Control Box itself.

Help TECH SUPPORT Help You.

- Frankly, sometimes what you tell Tech Support is just not true.
  - You may think your machine and software is set up one way, but…it just isn't.
  - So, help yourself and Tech Support by Running the Diagnostic Information (**UD**) option and send that to ShopBot.
  - If you want to know what your Computer/ShopBot is saying about you, read the notes on Diagnostic Information.

The Help Menu



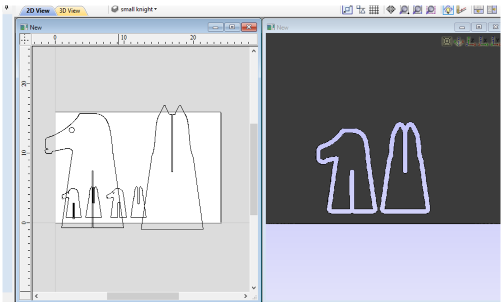
HELP is Not a 4 Letter Word.

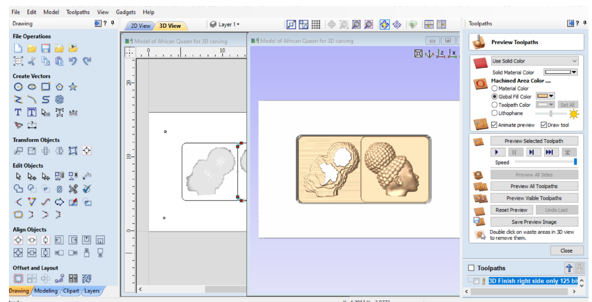
- **General reference.**
  - **Quick Start for Beginning ShopBotters** has info on setting up and using the ShopBot
  - **Command Reference**: What do SB3 commands mean?
  - **User's Guide**: Want to write your own code? G Code info (~page 51) Variables Master files and nesting.
  - **Quick Reference to Command Summary has** a list of all the variables of ShopBot Commands.
- **Shortcuts and techniques.**
  - **Keyboard Shortcuts.**
  - Techniques for **Squaring up ShopBot** and **Table Base Coordinates.**
  - Connect with other ShopBotters through the **User Forum** or **Website.**
  - **Programming and Developer Resources**: to get into/teach Programming using ShopBot. Information about using Math, User Variables (&), and System Variables (%) begins on page 7.
- **Information to help tech support help you**
  - ShopBot Tech support may ask you questions about your settings and computer.
  - Some of the documents here explain what Tech Support is asking for, and why.
  - **Help (A)bout**: to find out what version of SB3 ShopBot software is loaded.

Help is also available on the ShopBot website www.shopbottools.com and at support@shopbottools.com.

Projects and Techniques

Tutorials for ShopBot Training Using VCarve Pro





This Chapter introduces several projects or tutorials that are practical and useful for running the ShopBot. The printed document here outlines topics introduced in each project.

The VCarve Pro (v. 10) files themselves, and many of the ShopBot Part (.sbp) files are on the accompanying flash drive. For those who cannot open the v.10 VCarve files, the included vectors created from the design (.eps format) can be imported into an earlier version. A detailed .pdf for each project is included on the flash drive. The .pdf can be followed along on screen, or printed out (suggestion, 2 slides/page.) Other Chapters of the Handbook are often referenced.

The first project in the document, **Surfacing the ShopBot Sacrificial Board** (4'x 8' ShopBot, PRS9648), has the most details for creating a file from idea to execution. The tutorial on the flash drive is intended for surfacing a Desktop ShopBot with a cutting area of 24" x 18". All the tutorials on the flash drive are sized for Desktop. Expand as needed.

The **Simple Engraved Sign** project on the flash drive is used in the **Safety and Basic Use** Chapter. Make use of Information in the **Safety and Basic Use** Chapter when getting started using the ShopBot.

While each project can stand alone, or be done in any order, the later projects often rely on skills discussed in earlier projects. The Coaster Project and the Jig Project are intertwined.

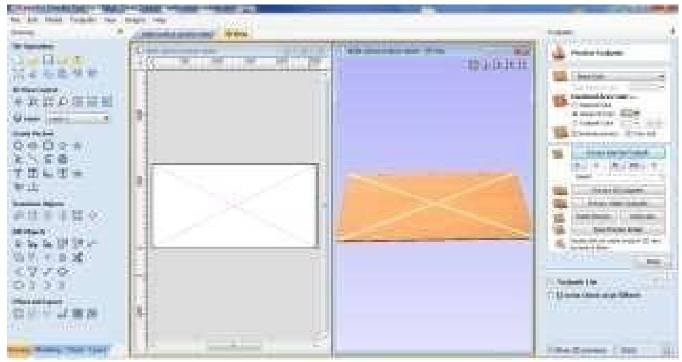
There are many other resources for becoming proficient with VCarve Pro and Aspire CAD/CAM software.

- Vectric website: http://www.vectric.com/
- ShopBot website: https://www.shopbottools.com/
- Videos and Tutorials: https://www.shopbottools.com/training/tutorials
- Projects: https://www.shopbottools.com/explore/projects

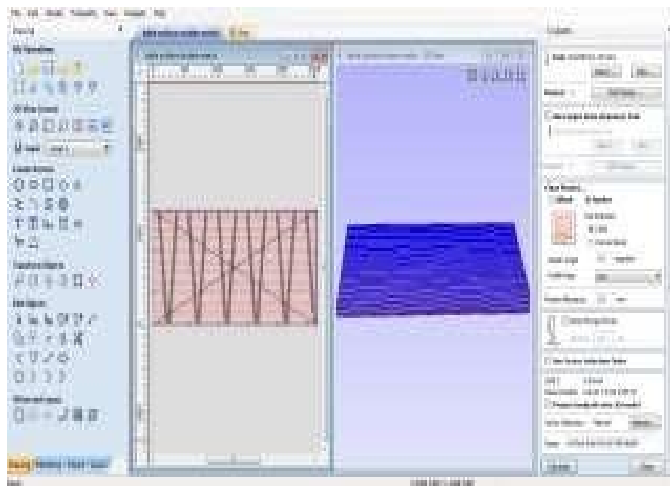
Surfacing the Sacrificial Board

Overview of Steps to Create a File to Surface the ShopBot Sacrificial Table

- Set the **job size and position** to match the size of the sacrificial board.
- Run a "cross check" across the table to make sure that the surfacing routine will reach all the low spots on the table. It may take a couple of tries before the correct depth is determined.

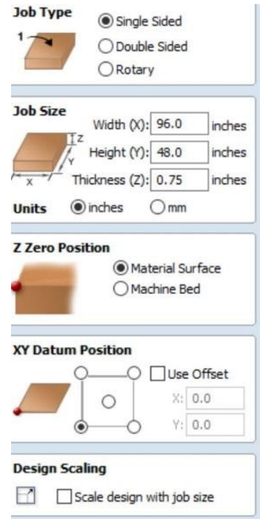


- Create a pocketing routine that **rasters** back and forth across the table to surface it, and then does a final pass around the outside edge to finish it up.
- This will create a lot of dust, so have adequate dust collection on during the surfacing routine.
- After the surfacing routine is finished, determine if the spindle is straight in the YZ car by **feeling** for ridges along the raster.



Step by Step: Surfacing the Sacrificial Board

Step 1: Measure the Size of Sacrificial Board or the Full Travel of the ShopBot

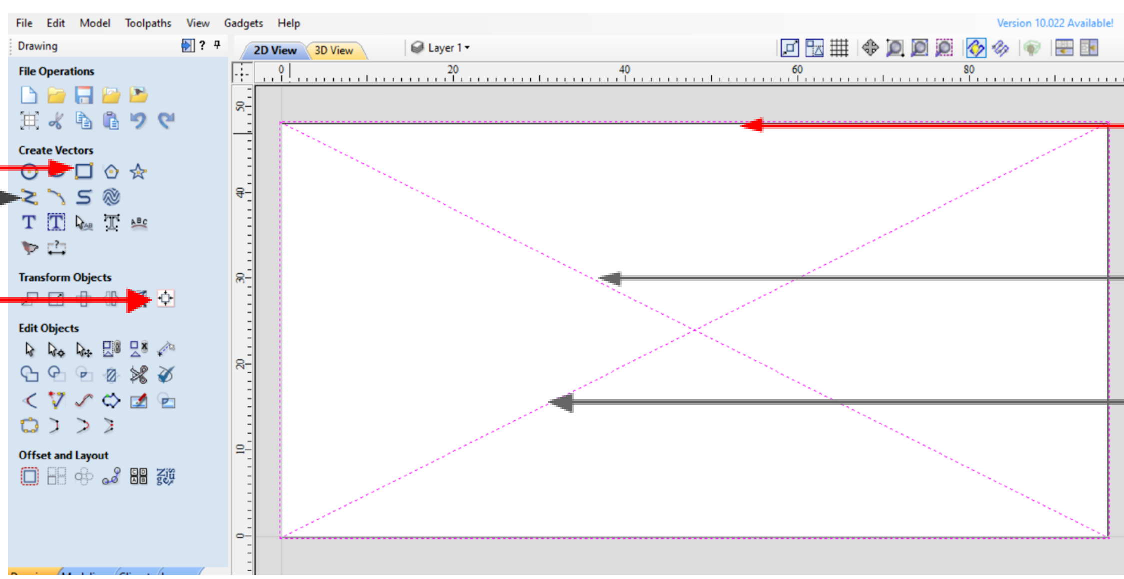


In the Job Setup screen, set up the design file for the size of the sacrificial board.

- Note: in this example, values are in inches for a PRS96-48.
  - X = 96, Y = 48, Z = .75 (for metric, multiple the inch values by 25.4 example: 96*25.4= 2438.4).
  - Set Z Zero Position to Material Surface.
  - Set XY Datum Position (0,0) to lower left corner.
  - Set Units to inches (or mm).
  - Click OK.
- The virtual representation of the material on the screen will be the size of the sacrificial board.

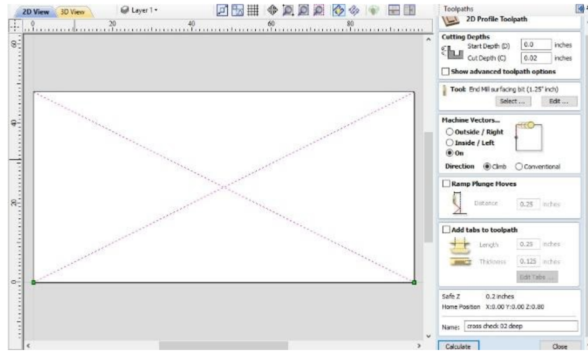
Step 2: Use the Vector Drawing Tools to:

- Draw a rectangle that is slightly larger than the support board and center it on the material.
  - The rectangle should be about .5" or 13mm larger than the material.
  - The **Centering and Aligning** options are found under Transform Objects.
- Draw two diagonal straight lines across the support board.
  - Hint: to draw a line, left click where the line should start. Hold the left mouse button and drag cursor to desired location. Left click to enter point to change direction or end.
  - Hit the **Space Bar** to finish that line. Do it again for the next line.



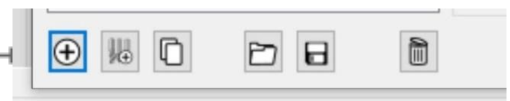
Step 3: Toolpath the Diagonal Cross Lines to Determine How Deep the Table Surfacing Routine Has To Be

- Choose **Profiling** toolpath.
  - **Start depth** is 0 (surface of material).
  - Set **Cut depth** to something shallow.
  - Value in example = .02”.
- **Select** the surfacing bit in tool library (see next image), then come back to this section.
  - Set the toolpath to profile **ON** the line.
  - Give it a name that you can remember.
  - Do not put a decimal symbol in the name of the tool path.



Step 3a: Choosing a Bit in the Tool Library

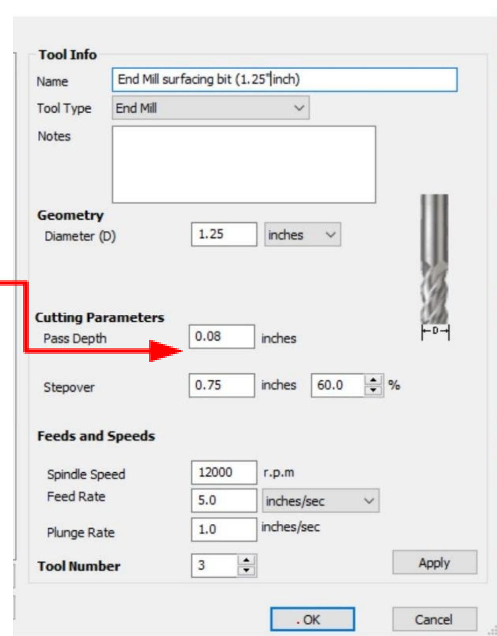
Shown: The Surfacing Bit included in the Starter Bit Kit sent with a gantry or Buddy tool. *The Desktop and Desktop Max starter bit kits include a .5” straight bit which will have different values in the Tool Library*.*



You may have to create a new bit for the surfacing bit. Add or copy and edit existing bit.

Important information for this tool is:

- The **Diameter** of the bit 1.25”.
- The **Pass Depth** should be no greater than .08” (the depth of the cutting blade on the surfacing bit).
- **Stepover** can be 40 – 80 percent (larger stepover saves time).
- The Spindle Speed is **12000.**
- The Feed Rate is something like 5 inches/sec (faster than usual).
- The Plunge rate is 1 inch/sec.



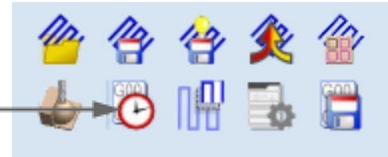
* For the .5” bit (Desktop tools).

Set the diameter to .5”,

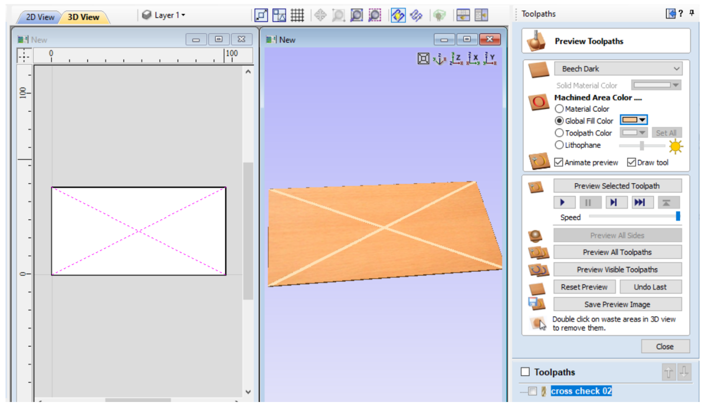
Set the pass depth to .125”,

Set the feed rate to 3 ips.

Step 4: Simulate the Toolpath on Screen

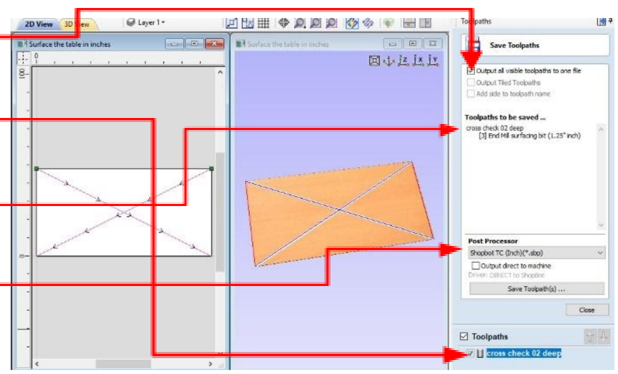


- Tile both 2D and 3D screens horizontally with the correct icon on the drawing page (top row).
- Choose the **Preview Toolpaths** icon (next to the clock).
- Highlight (click on) the desired toolpath, then click on **Preview Selected Toolpath.**
- Adjust the colors of the material and the toolpaths.
- Use the **Reset** button to clear the screen to test another toolpath.
- Remember to **Close** this screen to get back to the other screens.
- Use the VCarve **HELP** file for more details.



Step 5: Save the Toolpath as a ShopBot Part (.sbp) File

- The icon for saving a toolpath is the floppy disk.
- Put a check in this box to be able to put more than one toolpath into the same .sbp file.
- Put a check in front of the toolpath(s) to be saved into a .sbp file.
- Make sure that the correct toolpath is in “toolpaths to be saved” box.
- Use the pull-down arrow to select the correct **Post Processor**. ShopBot TC is the most common. Make sure that the ShopBot Post Processor is in the same units as the design file (inches or metric).
- Give the .sbp file a name you can remember and save it in a location you can find again.
- After the .sbp file is saved, remember to go back to the Design screen (left side of screen) and Save the CAD/CAM (.crv) file itself.



Step 6: Set Up the ShopBot and Run the Crosscheck File

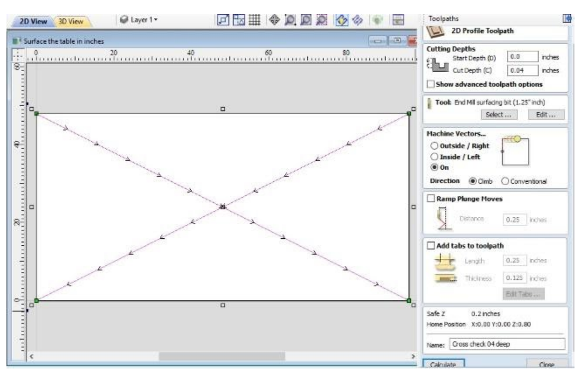
See Chapter on **ShopBot Safety and Basic Use** for more details on how to set up the ShopBot

- Insert the 1.25” bit in the half inch collet and tighten.
- Zero the X and Y with the center of the bit at the lower left corner of the table.
- Zero the Z at the surface of the material (the top of the sacrificial board).
- Make sure the lock out key is engaged and the power to the spindle VFD is on.
- Load the correct file (**FP** from menu or **Cut Part** button on Position screen).
- To do an “air cut” to test whether the file will run correctly before actually cutting into the material:
  - (Note for ShopBotters with an ATC: Pick up the correct bit before doing an AirCut.
  - If applicable, stop the AirCut before a bit change).
  - Send the bit to XY 0,0 and the Z to 1” (Jog Home usually works).
  - Load the correct file.
  - On the yellow START sheet, choose **3D offset** from the pulldown menu.
  - Press the **Space Bar** to stop the program at any time, and answer the questions on the screen to **Quit** or **Resume** the file.
  - When ready to run the file for real, the software defaults to “No Offset”, so just run it.

- Remember to “start” the spindle when ready to really run the file.
  - Is the spindle lockout key engaged so that there is power to the spindle VFD?
  - For alpha tools, has the **Start** button on the pendant been pressed at the prompt?
  - Is the spindle spinning after the **OK** question has been answered?
  - If you do not hear the spindle spinning, cancel or stop the file (hit the space bar or click on the STOP button on the Position Screen).

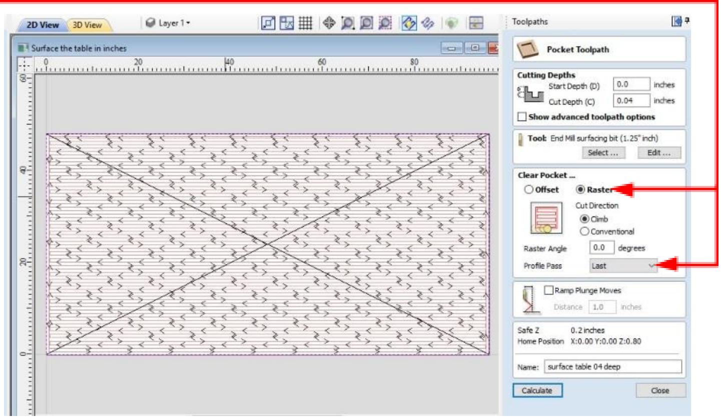
Step 7: Examine the Results of the Cross Check to See if the Bit Has Reached All the Low Spots on the Sacrificial Board

- If it is not deep enough, go back to the .crv file and redo the toolpath at a deeper depth (example, .04” instead of .02”).
- Save a new .sbp file with a name that is a reminder of the new cut depth.
- Repeat until the cross check reaches all the low spots on the table.
- Use that depth for the pocketing routine below.

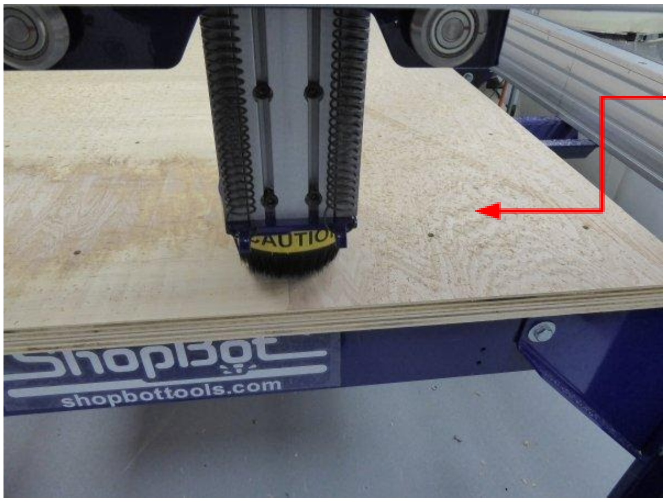


Step 8: Select the Rectangle and Create a Pocket Toolpath at the Depth Determined in the Last Step

- Set the cut depth to the depth determined in the last step.
- Select the same bit.
- **Raster** the toolpath so that it starts on the outside of the table and moves back and forth across the table, parallel to the X axis.
- Choose **Profile Pass** last to do a final run around the edge of the table.
- Give it a name that is easy to remember.
- Make sure to **save** the correct toolpath as an .sbp file.
- Remember to save the CAD/CAM (.crv) file also (on the left side of the screen).



Step 9: Run the Surfacing File as Above



Part way through table surfacing: plywood sacrificial board.

- Left side is complete, cross check still visible on right.
- Coloration of plywood is not important as long as the table feels smooth to the touch after surfacing.
- Machining lines may be visible, but usually reflect a difference in the way the light is reflecting off of the machined lines.
- If one can FEEL ridges, then adjust straightness of spindle/router on XY car. See **TroubleShooting** Chapter.

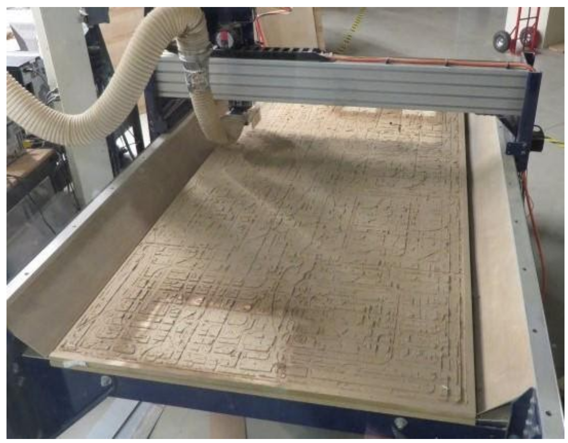
Examples of Table Surfacing

Part way through table surfacing: MDF sacrificial board

- Corner in upper right shows surfacing routine not deep enough to reach the lowest point on sacrificial board.
- Solution: stop the file, and rewrite the CAD/CAM file to remove more of the sacrificial board.
- To avoid having to rerun the entire surfacing routine, run a cross check file first.
- Note: “stripes” on table are visual.

When to resurface the table?

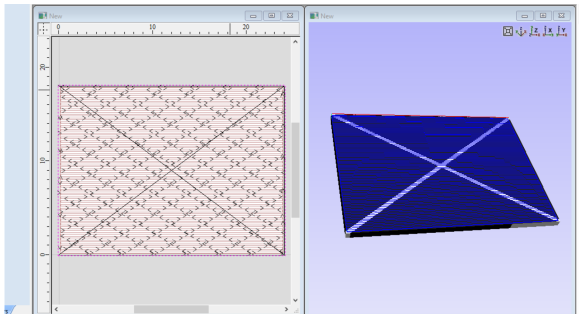
- The sacrificial board reveals a history of the use of the ShopBot.
- There comes a time when the sacrificial board becomes so scarred up that it is time to run the surfacing routine again.
- At some point, the sacrificial board must be replaced.
- If the board is glued down, the old board can be surfaced down to the support board level before gluing or screwing on another sacrificial board. Surface the new board.
- If the board is screwed down, remove the screws to remove the board. Lay down the new board and clamp in place. Use the ShopBot to create pockets to countersink the screws before attaching and surfacing the new sacrificial board.



Projects on Flash Drive

Note: If the flash drive has gone missing, the projects can be downloaded from the ShopBot website: https://www.shopbottools.com/explore/projects

Project: Surface Desktop Table
Skill Level: Beginning



Use VCarve Pro to set up table surfacing routine. (Sample.crv file on flash drive)

TOPICS:

- CAD design.
  - Set up job size and position for Desktop 24” x 18” x .75.
    - XY Origin at lower left corner.
    - Z Zero at surface of material.
  - Create lines for cross check.
  - Create rectangle for table surfacing.
- CAM design.
  - Create Profile on the line toolpath for cross check.
  - Define bit in tool library.
    - Bits selection, Feeds and Speeds.
    - RPM of spindle.
  - Create Pocket toolpath for surfacing the Desktop table top.
    - Contour vs Raster toolpath.
    - Stepover, etc.
  - Saving .sbp (ShopBot part files).
  - Saving .crv (V Carve Pro CAD/CAM files).
- Set up ShopBot for machining (see **Safety and Basic Use** Chapter).
  - Moving the tool around.
  - Warming up the spindle.
  - Changing a bit.
  - Registering machine (setting up XY and Z Zero positions).
  - Previewing a file.
    - Preview mode.
    - Aircut.
  - Running a file.
  - Stopping a file while running.

Project: Simple Engraved Sign in MDF

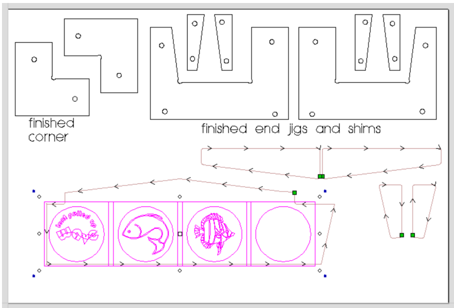
Skill Level: Beginning. This project is used in the Chapter on Safety and Basic Use

Use VCarve Pro to create a set of simple signs. (This .crv file on flash drive)

TOPICS:

- CAD design.
  - Set up job size and position for 24” x 16” x .75” MDF.
    - XY Origin at lower left corner.
    - Z Zero at surface of material.
  - Create shape for the signs.
  - Enter text and import a vector in the shapes.
  - Place circles in board to add hold down screws.
- CAM design.
  - Engrave the text using a V bit.
  - Define bit in tool library.
    - Bits selection, Feeds and Speeds.
    - RPM of spindle.
  - Create a Profile toolpath to cut out the shape.
    - Toolpath outside the design vector.
    - Add tabs and/or ramps for hold down OR
      Add hold down locations inside sign.
  - Use Drill toolpath to mark where it is safe to put a hold down screw.
    - Drill toolpath.
  - Saving .sbp (ShopBot part files).
- Set up ShopBot for machining.
  - Moving the tool around.
  - Warming up the spindle.
  - Changing a bit.
  - Registering machine (setting up XY and Z Zero positions).
  - Previewing a file.
    - Preview mode.
    - Aircut.
  - Running files.
    - Run hold down file, then screw down material.
    - Run engraving file with V bit.
    - Run Profile (cut out) files with .25” end mill.
  - Stopping or Pausing and Resuming a file while running.

Project: Jigs for Coasters and Other Projects
Skill Level: Beginning- Intermediate

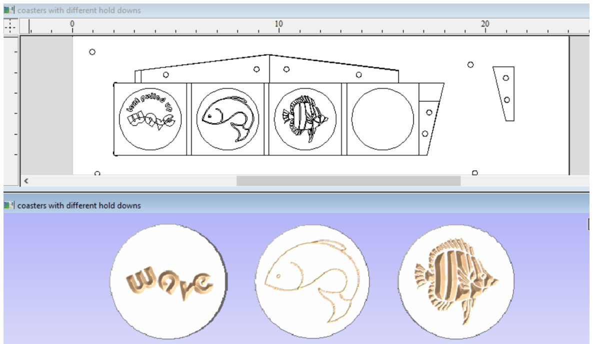


Use VCarve to Create Hold Down jigs for future projects. (Sample .crv file on flash drive.) Tutorial includes suggestions for modifying and correcting flaws in design (Design engineering, iteration).

TOPICS:

- CAD design.
  - Set up job size and position for 24” x 16” x .75” MDF.
    - XY Origin at lower left corner.
    - Z Zero at surface of material.
  - Plan ahead: Determine size of material that will be held in place with jig.
- Add space for shims to fit around material.
- Create jig for project.
  - Using Edit Vector tools to join simpler shapes.
  - Adding dog bone filets to fit square corners into internal corners.
  - Create shims to hold material.
  - Plan for hold down: locate where screws will go if using screws.
- CAM design.
  - Create a profile toolpath to cut out the jig location and shims.
    - Profile to inside to create space in board for jig.
    - Profile to outside to cut out shims and right angle jigs.
    - Add tabs and/or ramps for hold down.
    - Add allowance to toolpath where needed.
  - Create Drill toolpath if screws are hold down option.
- Save .sbp (ShopBot Part) file and .crv (CAD/CAM) file.
- Set up ShopBot for machining.
  - Changing a bit.
  - Registering machine (setting up XY and Z Zero positions).
  - Previewing a file.
    - Preview mode.
    - Aircut.
  - Running files.

Project: Creating Coasters from Dimensional Lumber
Skill Level: Beginning- Intermediate

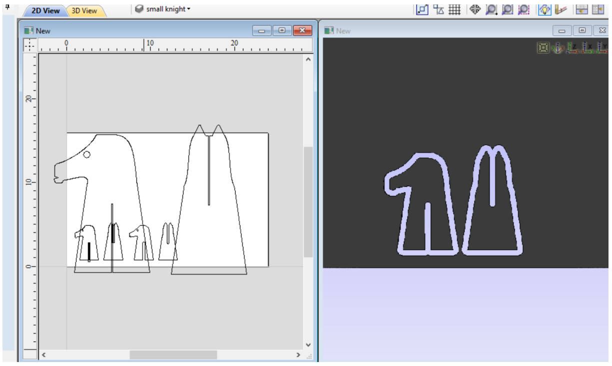



Use VCarve to create coasters. Importing vectors. Importing and tracing bitmaps. Setting up file for use of jigs for hold down. (Sample .crv file, and sample vectors and bitmaps on flash drive.) Modifying design to reflect real-world application
See also Chapter **Working with Other CAD/CAM Software**.

TOPICS:

- CAD design.
  - Set up job size/position for dimension lumber for coasters: 15.25” x 5.5” x .75” wood.
    - Take into account that using a jig for hold down will affect job size and position.
    - Z Zero at surface of material.
  - Add shape of coaster to board.
    - Use rectangles to help place coasters on material.
    - Array to lay out 4 squares with .25” gap in between them.
    - Create and center circle for coaster in each square (alignment tools).
    - Put design in each circle (include text, importing vectors and tracing bitmaps).
- CAM design.
  - Engrave the design with a V-Bit.
  - Create a Profile toolpath(to the outside or on the line) to cut out the shape.
    - Add tabs and/or ramps for hold down.
- Save .sbp (ShopBot part files) and the .crv (CAD/CAM) file.
- Set up ShopBot for machining.
  - Changing a bit.
  - Registering machine to account for offset of jig in X and Y axes.
    - Zero XY at corner of material, or corner of jig, as determined by CAD file.
    - If necessary, C3, M2 to inside corner of jig (2,2 in above jig), Z2 to zero X and Y at that location.
  - Z Zero at top of material.
  - Preview a file on screen and/or Aircut.
  - Run the files.
    - Run Engraving files with V bit.
    - Run cut out file with .25” end mill.

Project: Creating a Chess Set from a Design Found on the Internet
Skill Level: Beginning- Intermediate

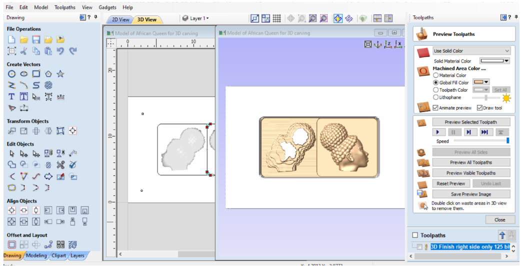



Finding a Chess Set on the internet. Importing the vectors (.dxf) into VCarve Pro. (Sample .crv and .dxf files on flash drive.) See also Chapter **Working with Other CAD/CAM Software**. Real world applications touched on by creating this chess set: intellectual property and attribution. Using Chess as an entry into other subjects: history, business, math, AI.

Finding designs on the internet, Safety on the Internet, Intellectual Property and Attribution.

- CAD design.
  - Set up job size and position for 24” x 16” x .25” (size of material).
    - XY Origin at lower left corner of **BOARD** (Adjust for jig or put at corner of Desktop surface).
    - Z Zero at surface of material.
  - Use Drawing tab to import .dxf into VCarve.
  - Use tools in VCarve to join segments and clean up vectors (See Chapter **Working with Other Software**).
  - Size the vectors to desired size of chess pieces.
  - Size the slots to match the material thickness.
  - Add adjustments for cutting inside curves.
- CAM design.
  - Create a profile toolpath to cut out the shape.
    - Toolpath outside the design vector.
    - Add tabs and/or ramps for hold down.
  - Use previews in VCarve to ensure that the chosen bit will go into the areas designated.
  - Add hold down locations for mechanical hold down (screws).
    - Drill toolpath.
  - Save .sbp (ShopBot Part) files and the .crv (CAD/CAM) file.
- Set up ShopBot for machining.
- Chess outside the Maker Space.

Project: Importing 3D files and Preparing for 3D Carving with ShopBot
Skill Level: Beginning- Intermediate



Import a 3D file (.obj, .stl) into VCarve Pro for Machining. (Sample .crv file and .stl files on flash drive.) See also Chapter **Working with Other CAD/CAM Software; Single sided vs double sided machining.**

TOPICS:

- Importing a 3D Design File into VCarve Pro.
  - Aspire, SolidWorks, Fusion360, Rhino, etc., models can be exported as a 3D model and imported into VCarve Pro for machining.
  - Websites for 3D printing are also good sources of models: Instructables, Thingiverse, Solidworks Library, etc.
- CAD design: single sided machining.
  - Set up job size and position for 24” x 16” x 2” foam (size of material).
    - XY Origin at lower left corner (Adjust for jig or put at corner of Desktop surface) .
    - Z Zero at surface of material.
  - Plan for hold down.
  - Use Modeling tab to import 3D model (.obj, .stl, etc.) into VCarve.
  - After sizing and positioning, draw vectors around the 3D model.
- CAM design.
  - Set up model in material (Z placement).
  - Create a roughing pass.
  - Create a finish pass.
    - Create another tool for finish pass.
      - Adjust stepover to 8 – 10%
      - Adjust feed rate/plunge rate for 3D carving (Chapter: **Bit Selection, Feeds and Speeds**).
  - Determine hold down technique (screws or jig).
  - Save .sbp (ShopBot part files) and .crv (CAD/CAM).
- Use Aspire to manipulate, triangulate and export 3D models.
  - Inverting an open backed model from convex to concave.
- Double-sided machining.
  - Planning ahead.
  - Setting up CAD file for flipping material.
  - Registration.
  - Setting up ShopBot for double-sided machining.

Working With Other CAD/CAM Software

ShopBotters can use other software to create their designs and models.
In general, there are two different categories of Computer-Aided Design (CAD) Software:

Engineering-based such as Solidworks, Inventor, Fusion 360, Rhino, SketchUp

- Many of these CAD programs have the option of drawing in 2D (vectors) as well as extruding the designs to render the parts in 3D on the computer screen.
- When designing in these CAD programs and preparing them for machining on the ShopBot, one must determine whether the designs are best exported as:
  - 2D files for creating out parts out of sheet good materials like plywood (or making with a laser cutter or vinyl cutter).
  - 3D files for carving in 3D with a ball nose bit (or printing with a 3D printer).

Graphics-based such as Inkscape, Corel Draw or Adobe Illustrator

- The intended goal of these software is for Desktop Printing, so they usually export 2D files.
- The Drawing Tab of VCarve Pro/Aspire is graphics-based, so should be familiar to those who use many of the graphics-based software.
- Print files can be Vector based (outlines) or Bitmaps (photos, or even fill).
- For use with the ShopBot and VCarve Pro Software, Bitmaps must be converted to Vectors.
- With Aspire CAD/CAM software, a bitmap (photo) can be traced to convert to a vector (Drawing tab), or converted to a 3D component for 3D carving (Modeling tab).
- Designers can easily import their 2D files into VCarve Pro/Aspire for toolpathing.

Many CAD software do not have a Computer-Aided-Machining or -Milling component (CAM), so the design files must be imported into another software for creating the toolpaths and generating code for a CNC machine to run the file.

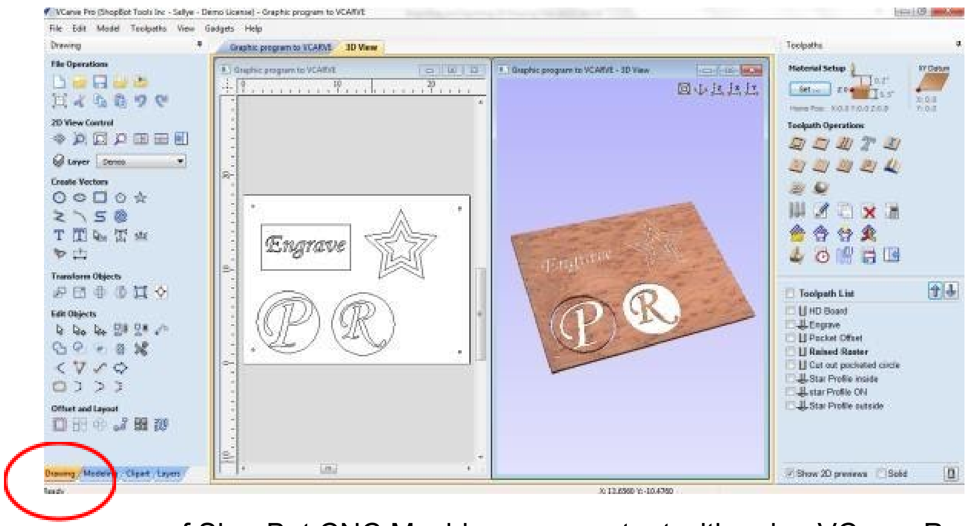
Vectric CAD/CAM software included with the purchase of a ShopBot has the capability of creating toolpaths for both 2D and 3D designs.

It is also convenient to design and lay out parts in VCarve Pro/Aspire, then export the vectors for use with another digital fabrication tool such as a laser cutter or vinyl cutter.

Importing and Exporting 2D Drawing Files

VCarve Pro/Aspire

VCarve Pro is a powerful, graphic-based 2D Computer-Aided Design (CAD) and 2D/3D Computer-Aided Machining (CAM) software that is included with the purchase of a ShopBot CNC machine. Aspire is VCarve Pro's big sister that adds the capability to design in 3D. The Drawing Tab is the same for VCarve Pro and Aspire, and the Toolpathing options are similar in the two programs.



While many users of ShopBot CNC Machines are content with using VCarve Pro as their only CAD/CAM software, others may wish to use another software to design their projects. This tutorial does not go into details on how to use the other software, but emphasizes how to bring the designs created by them into VCarve Pro or Aspire in order to lay out the parts on the material and toolpath them.

The following section outlines some of the tricks and techniques for:

- Importing 2D vector drawings into VCarve Pro/Aspire (for example: .dxf, .eps, .ai, .svg, .pdf, .skp).
- Importing bitmaps and tracing them in VCarve Pro/Aspire (for example: .jpg, .png, .bmp, .tiff).
- Exporting vectors from VCarve Pro/Aspire (.eps, .dxg, .ai, .svg).

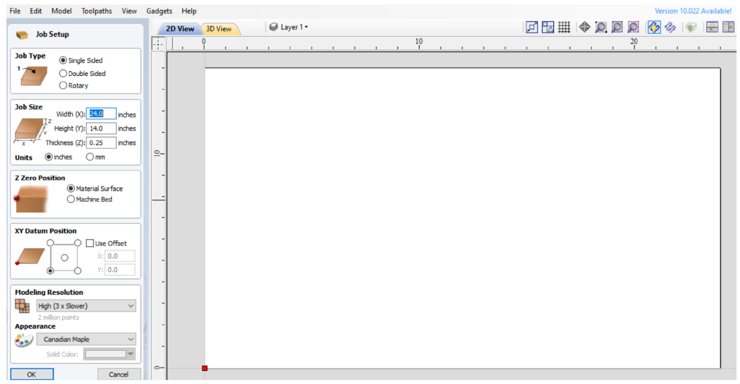
Note: the **HELP** tab in VCarve Pro and Aspire has an excellent .pdf that gives much more detailed information on importing data into the software (and anything else about the software.) The Help file is bookmarked, so scroll down to find the information, then click on the topic. The screen will jump to the information in. The **Find** option under the **Edit** drop down menu is also very helpful.

There are also tutorials on the Vectric website: www.Vectric.com, as well as projects and tutorials on the ShopBot website and in this notebook.

Getting Started

The first step is to launch a session of VCarve Pro/Aspire.

Define the **Job Size and Position** (icon: **Job Setup**) so that the virtual design screen is the same as the material that will be placed on the ShopBot:

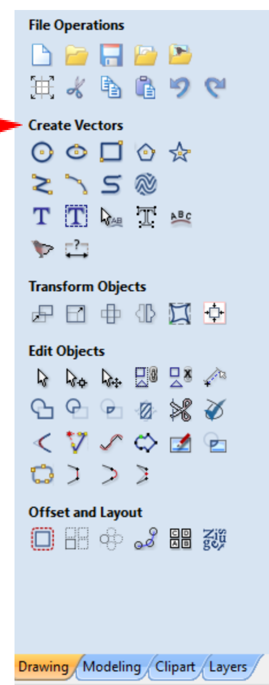


- Job Type: Single Sided.
- Job Size:
  X = 24”, Y = 14”, Z = .25”.
- Z Zero Position: Material Surface.
- XY Datum Position: Lower Left.
- Click on OK to accept.

What is a Vector?

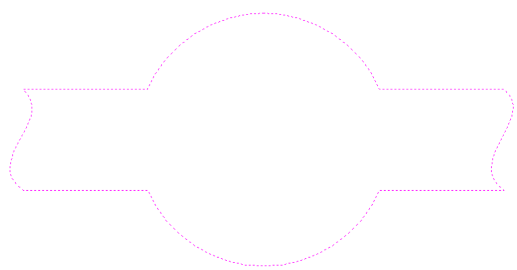
(Apologies to math teachers everywhere.) Think of vectors as line drawings. They have a start point and an end point, with a distance in between. Because they are basically mathematical formulas, they can be scaled and manipulated without a loss of resolution.

- VCarve Pro uses the term “vector” for creating circles, rectangles, lines, etc.
- Some graphic software such as Adobe or Inkscape may use the term “stroke” instead of vector.
- Word uses “line”.
- In Fusion 360, the “Sketch” mode creates vectors which can then be extruded into 3D “Bodies”.
- In Graphics software, the outline is different from the fill.



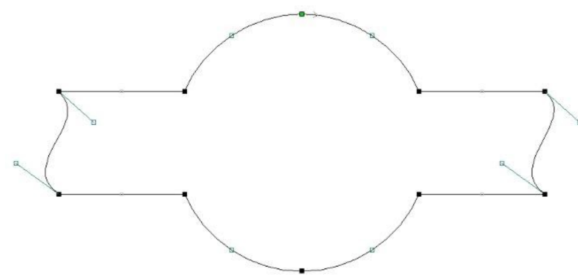
Viewing Vectors in VCarve Pro/Aspire

Vectors can be viewed in **Select Mode**, or in **Node Editing Mode** (icons under **Edit Objects**, or Right Click (RC) to get to the pull down menu.) The shortcut to toggle between modes is the "N" key on the computer keyboard.



When in Select mode, ungrouped selected vector(s) are dashed lines, usually magenta (pink).

- Vectors in Select mode can be edited, welded, aligned, centered, etc.
- Vectors that have been grouped together are solid lines.



When in Node Editing mode, the nodes (points) and spans are visible.

- Move the cursor around and note how it changes when it is on a node or a span.
- RC to see the menu of what can be done with nodes and spans.

Exporting Vector Artwork from Other Software

- Create artwork in the desired software.
- Save the vectors in the original software (specific examples follow).
- Remember where the file is saved because it will need to be called up to import into VCarve Pro/Aspire.

Note: if the file extensions are hidden on the computer, here is how to make them visible.

- In the search box, start typing “file extension”.
- When “Show or hide file extensions” appears, click on it.
- Scroll down to find the box next to “Hide extensions for known file types”.
- Uncheck the box. Now the extensions should be visible to choose the right file format.

2D Vector file extensions:

.dxf (AutoCAD, AutoDesk)
.dwg (AutoCAD)
.eps* Encapsulated post scrip)
.ai* (Adobe Illustrator)
.svg (Inkscape)
.skp (Sketch Up)
.pdf* (Adobe)
*These “metafiles” may contain both vectors (nodes and spans) and bitmap (color/fill) data

- For example, a graphics file that contains color fill has both Vector and Bitmap data.
- For importing to VCarve, the file must contain only the Vector data. .pdf files that contain both vector and bitmap data are stripped of bitmap data in later versions of VCarve Pro.
- To strip the file of the bitmap data, in the original software, try changing the “fill” to “no fill”.

Which file type is the best for importing into VCarve?

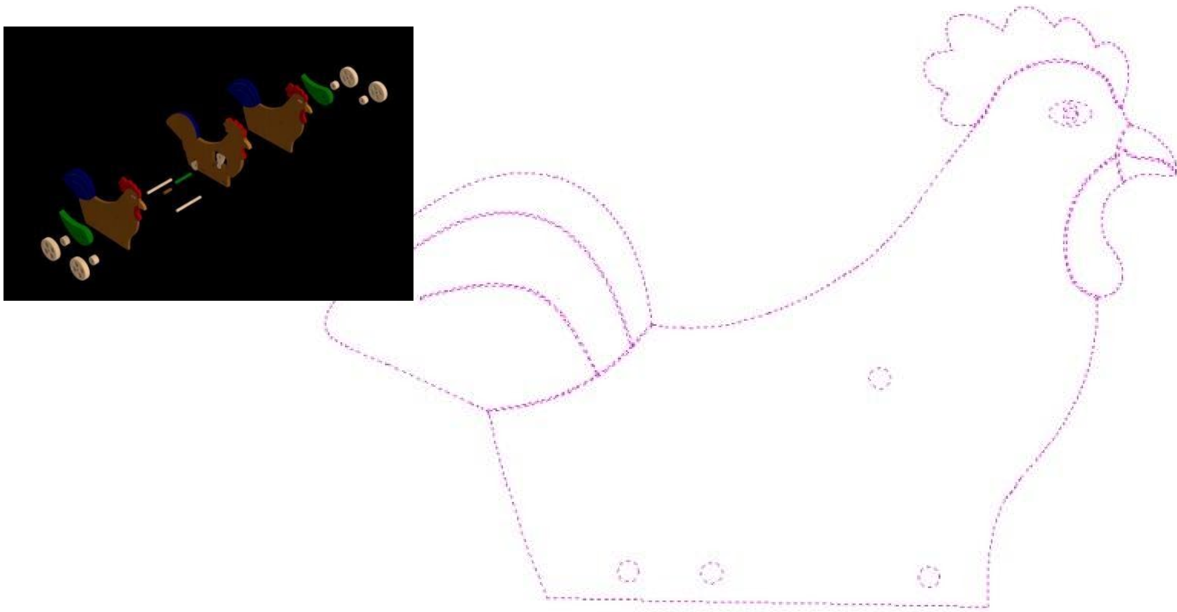
- .eps and .ai tend to come in cleaner than do .dxf and .dwg files.
- .dwg and .dxf often comes in open vectors next to each other instead of closed vectors, arcs have turned to straight line segments, duplicate vectors need to be cleaned up.
- Tools in VCarve Pro can help with cleaning up the imported data (see below).

Importing 2D files from Graphics software vs. 3D Rendering software

Since most Graphics-based software are creating 2D drawings for printing, just save the designs in one of the formats above.

3D rendering software give the option of saving designs in either a 2D format, or a 3D format (example: .stl or .obj.) For cutting out sheet goods like plywood on a CNC machine, cutting parts with a laser cutter, or using a vinyl cutter, the designs are best saved in a 2D format.

Preparing Designs Created in 3D Rendering Software for 2D Machining

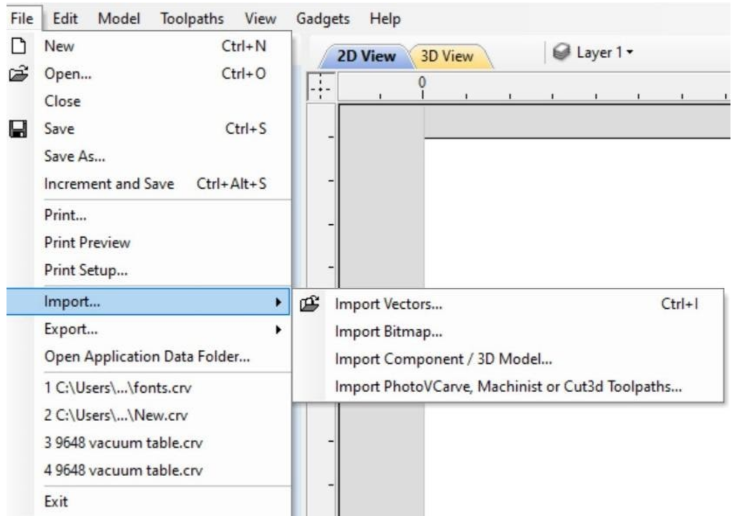


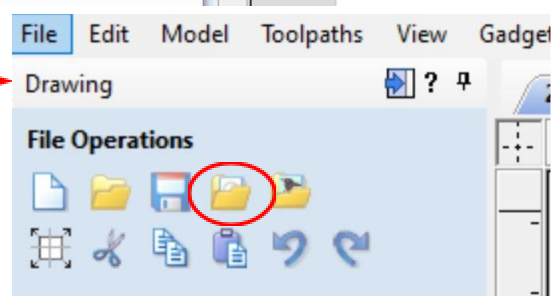
- The Chicken above was created in Solidworks.
- In the original 3D rendering software, the finished design can be.
  - Given color and shading characteristics to mimic the final product.
  - Rotated on the screen.
  - Tested in assemblies for fit and movement.
  - Exploded for viewing the details.
- In this situation, the outline of the chicken will be cut out of plywood or other sheet goods, so the vectors are the objects to be imported into VCarve Pro/Aspire.
  - In Solidworks, choose the correct plane to get the correct orientation for the cut out.
  - Use **Save As** to define the format for exporting. Solidworks gives several options.
  - Fusion 360 only allows export of .dxf files for 2D format.
    - If drawing in Sketch mode, and the sketch has not been extruded into a 3D “Body”, the sketch can be exported as a .dxf, regardless of what plane it is on.
    - One can create a Sketch from a Body and save the result as a .dxf for export for CNC machining or laser cutting or vinyl cutting.
    - Fusion 360 also has a CAM component, so toolpaths can be created directly in Fusion 360 and saved in .sbp format, but that’s another tutorial.

Importing 2D Vectors into VCarve Pro/Aspire

There are four options for importing data into VCarve Pro/Aspire.

- All four are available from the pull down menu **File > Import**.
- This section of the tutorial relates to the two file types which are options as icons on the Drawing Tab.
  - Import Vectors.
  - Import Bitmap.

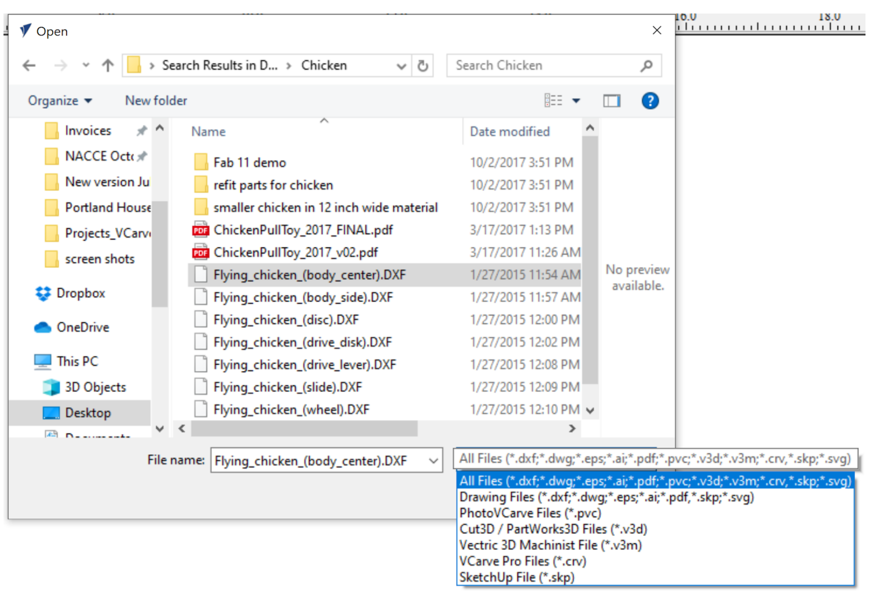




The icon for importing vectors is under **File Operations.**

Find the correct file and import it.

Vector formats that may be imported into VCarve Pro/Aspire are listed in the pulldown menu:



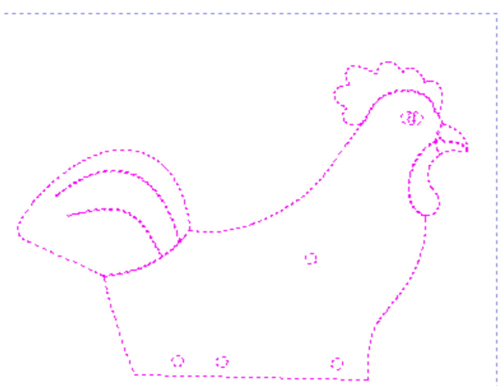
- **Sometimes, designs do not seem to have imported into the VCarve Pro/Aspire file.**
  - If you don't see the designs, the data may have imported into a location far off the material.
  - Try **Zoom to Fit** under the 2D view control.
  - With the wayward data still selected, use the **Center to Material** option under **Transform Objects > Align Selected Objects** to bring it on to the material.
  - Do check that the units of the drawing file are the same as the units of the VCarve Pro/Aspire file. Scale if need be (1" = 25.4mm).

Analyzing and Cleaning Up the 2D Files in VCarve Pro

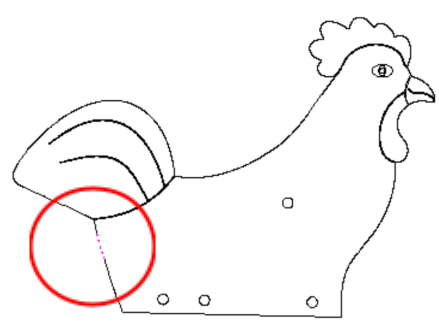
- It may be necessary to **Ungroup** the imported vectors to work with them.
  - The Icon is **Ungroup** or keyboard shortcut U.
  - Alternatively, check the layers icon to see how many layers have imported
    Select the vectors, and Right Click (RC) to open the pull down menu
    Select **Ungroup** from the menu to choose where to ungroup the data.

- Now might be a good time to check for duplicate vectors in the original file.
  - Select all of the vectors ("sweep select" so they all turn magenta (pink)).
  - RC to open the pull down menu.
  - **Select > all duplicate vectors.**
  - Delete the vectors that are still highlighted.
  - Remember that there are unlimited "undos" in VCarve.
  - Use the **Undo** icon, Control Z, or Use **Edit > Undo** to see what is being undone.
  - Use **Redo** to undo the undo.

Joining and Closing Vectors

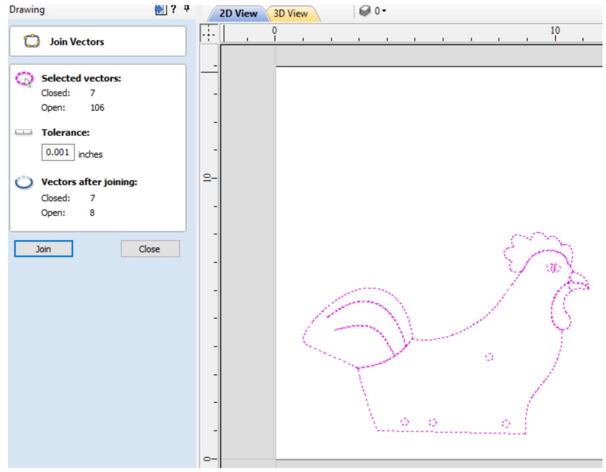


When the entire chicken is selected (sweep select), it looks as though everything is joined.

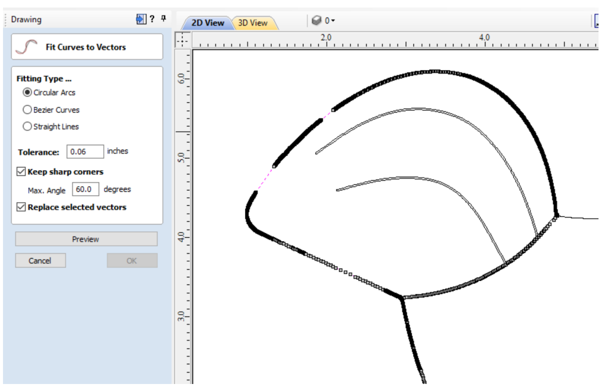


Deselect the entire chicken, then click on individual parts to see that there are many segments instead of joined and closed parts.

- The **Join Vectors** icon helps to join and close those vectors that are close together.
  - Play with the tolerance for different results.
- The **Join/Close** tools can help with segments that are too far apart for the **Join Open Vectors** tool.
- As always, HELP in VCarve Pro/Aspire is very useful.

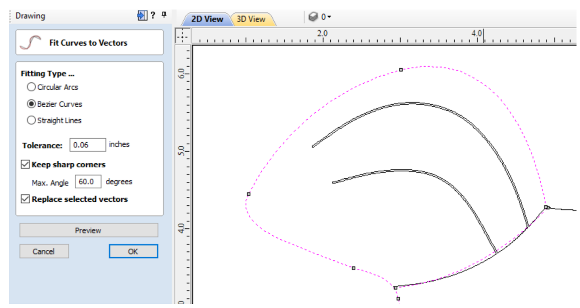


Are the Vectors Straight Line Segments or Smoothed?



Go into **Node Editing** mode to see the makeup of each vector.
("N" key is short cut to toggle between **Select** and **Node Editing** mode).

- Often, .dfx files import as straight line segments rather than arcs.
- The change in direction with each segment will show up in the cut files, and create more lines of code.



Use the **Fit Curves to Vectors** tool to smooth the segments into arcs or Bezier curves. Preview the **Fitting Type** option to see which gives the best results.

- Circular arcs may be better for circles that need to stay intact.
- Bezier curves may be better for more fluid designs.

Other parts of the drawing may need to be cleaned up to get clean vectors for toolpathing.

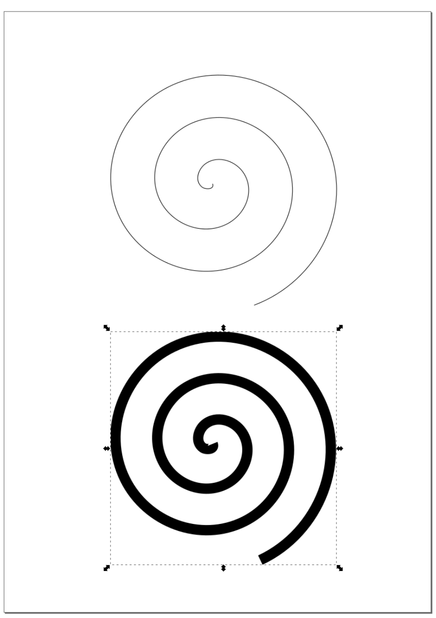
Working with Inkscape and Other Graphics-Based Software

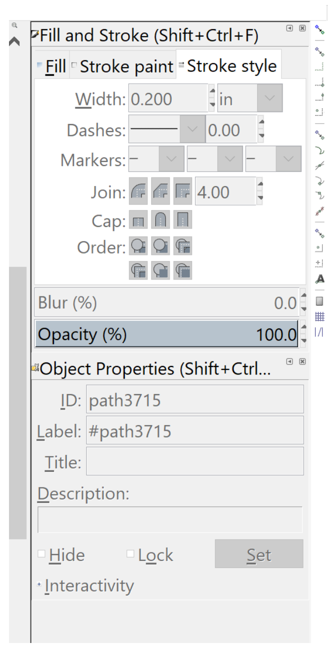
Software such as Corel Draw, Adobe Illustrator or Inkscape that were created with Desktop Printing in mind allow one to determine the width of the design line on the printed page.

- A wider line or stroke may look great on paper, but may not be the best option for the precision required for digital fabrication tools such as CNC machines or laser cutters.
- The size of the finished object can be influenced by the width of the line/stroke.
- The width of the line/stroke/outline/vector is often set in the Object Properties.
- Additionally, the manner in which the data are exported (bitmap or vector) for use in CNC machining or laser cutting makes a difference in the results.

Setting Line/Stroke Width in Inkscape

- Two identical spirals were created in Inkscape.
  - Right Clicking on the bottom spiral brings up a menu in which one can choose **Fill and Stroke.**
  - At the Stroke Style tab, one can chose the width of the stroke.
- Note the drop down menu which gives the units of the stroke width (pixels, inch, mm, etc.).
  - For illustration purposes, the two spirals were given different stroke width (in inches).
- Top spiral has a stroke width of .011”, bottom spiral has a stroke width of .2”.
  - Note: different software have a specific widths to use for vector cutting with lasers.
    - Corel Draw: Hairline.
    - AI or Illustrator: .001” or .025mm.





Options for Saving the Inkscape Design:

- **Export** the data as a bitmap (.png).
- Use **Save As** to export the data as vectors.

What is a Bitmap (or Raster) image:

A photo is a bitmap. A scanned logo is a bitmap. The color fill inside a vector outline of a character of text is a bitmap. Think of a grid where each individual square is a pixel that contains color information (including black). The number of pixels (or squares in the grid), and the amount of information in each grid square (pixel) determines the resolution of the bitmap. Bitmaps do not scale well because the closer you zoom in, the more you can see the individual squares or pixels.

Extensions for bitmaps include: jpg, .png, .bmp, .tiff

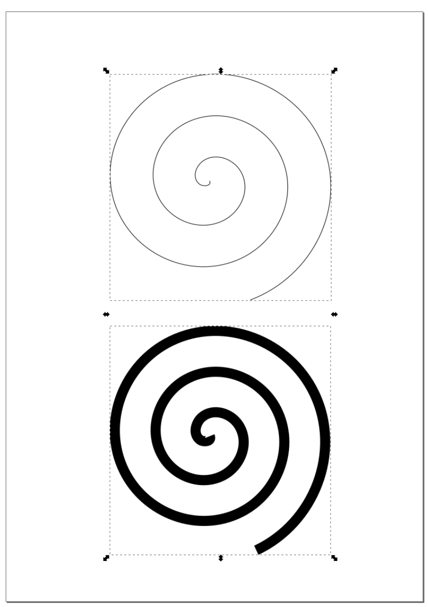
Exporting the Inkscape design as a .png (bitmap):

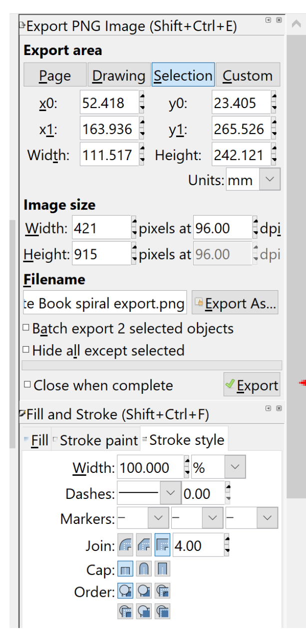
When to use this option?

- You have an original or scanned design or logo to be VCarved or Engraved with the ShopBot.
- You are creating a file to engrave on the laser (raster).
- You are using the Roland minimill and FabModules to create circuit boards.

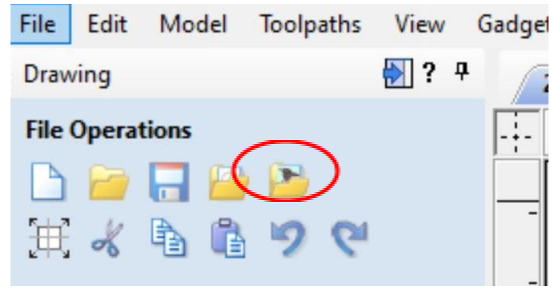
How to do it:

- Use **File > Export** to bring up the Export area screen.
- Select the objects to export.
- Fill in the blanks (including name) and click **Export.**



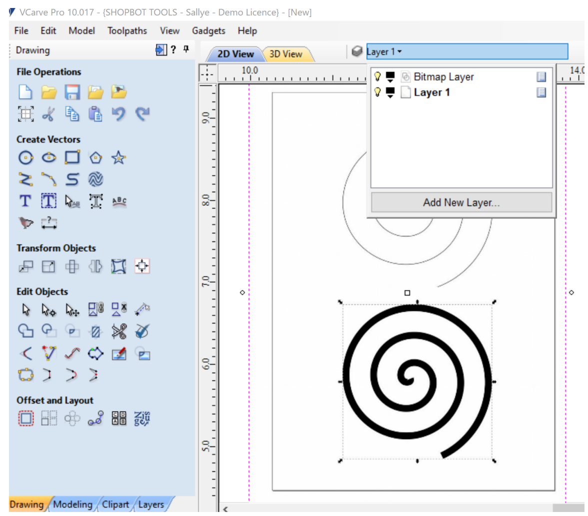


Importing a Bitmap into VCarve Pro



Use the Import Bitmap icon
or **File > Import > Import Bitmap.**

- The bitmap data will be put on its own layer.
- Click on Layers icon to bring up options.

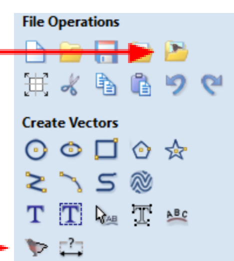


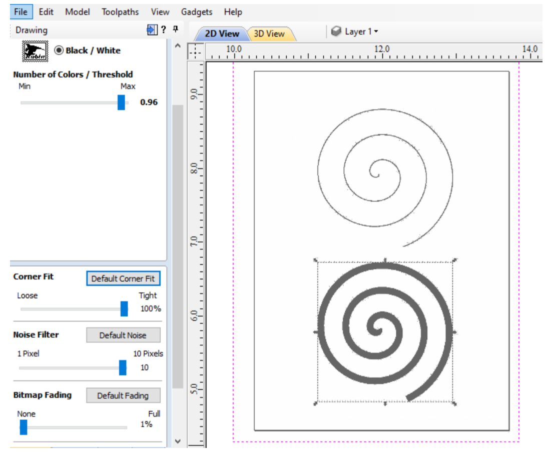
- The Bitmap will import onto its own layer into the middle of the material. It may be quite small, so zoom into it.
- Trying to enlarge it before tracing the edges may result in the vectors being rougher.
- The Traced vectors will sit on Layer 1 or whatever layer is selected.
- Use the lightbulb icon to turn layers on and off (make visible or hide).
- The traced vector is easily scaled without loss of resolution.

Tracing a Bitmap Imported into VCarve Pro

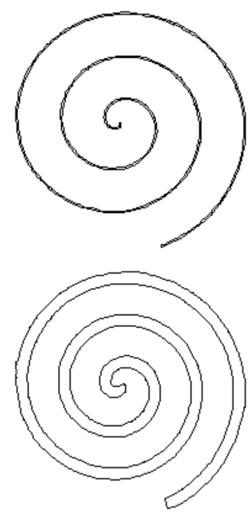
In order for the bitmap data to be used in VCarve, it must be traced and converted to vectors Changing the resolution of a bitmap changes the quality of the image/trace: higher resolution/more distinct edges = smoother trace.

- The icon to **Import a Bitmap** is a little bird in a file folder.
- The icon to **Trace a Bitmap** is a bird.

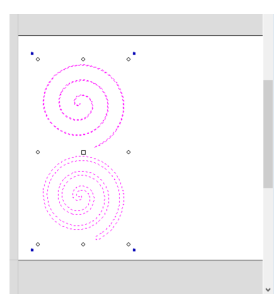




- For this example, Black and White was chosen.
    - The Threshold slider can affect how messy the bitmap to trace is.
    - Corner fit determines how closely to follow the curves.
    - Noise Filter determines how many stray pixels to filter out.
    - Bitmap fading lightens or darkens bitmap on screen but does not affect actual trace.
- Clicking on the **Preview** button shows the vectors created to follow the edge of the bitmap.
- Once the vectors are created, the bitmap layer can be turned off.
- Visit the HELP file in VCarve for more detailed information.



- With the Bitmap layer turned off, the vectors from the trace are visible.
- Use the **Scale** tool to measure the two spirals.
- Since the trace follows the edges of the bitmap, the lower spiral is larger than the upper spiral.
- The distance between the vectors is larger in the lower spiral that had the wider stroke.
- The vectors can be scaled with no loss of resolution or increased pixelation.

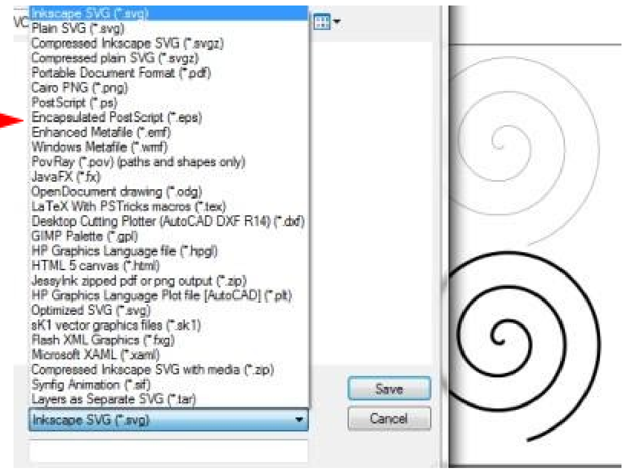



- In this example, the two spirals were scaled up after the trace.
- The VCarving or Engraving of the two spirals shows that the lower one is much deeper than the upper one because the width between the edges is greater.
- Moral of this demo: The width of the line or stroke when importing bitmaps and tracing them will determine engraving results AND the final size of the part.

Saving an Inkscape File as a Vector

Use **File > Save As** to bring up the menu that includes all vector formats.

In this example, .eps was chosen.

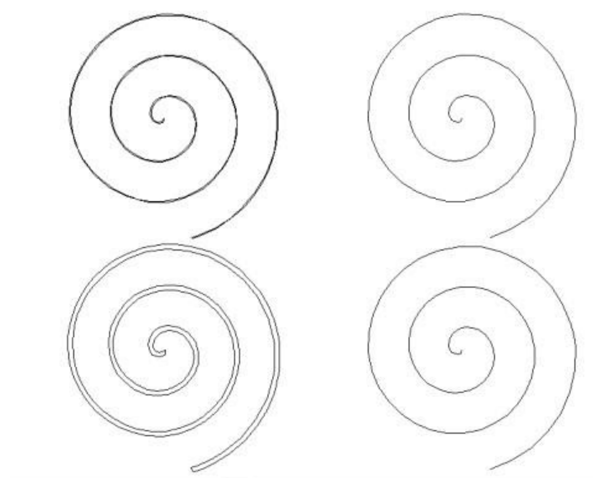


Import the vector file as above

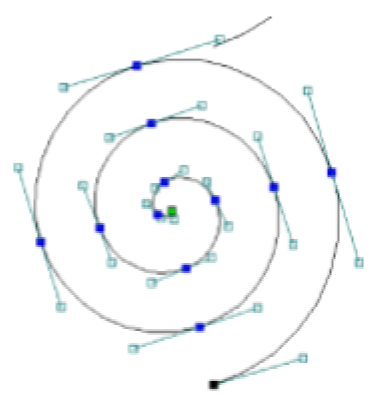
Import Vectors icon or **File > Import > Import Vectors.**

Compare the spirals imported as a bitmap and traced (left) to the spirals imported as vectors (right).

Both spirals on the right come in as the same size because only the mathematical data from the nodes and spans are saved.



The .eps format came in as Bezier curves rather than the straight line segments shown in the .dxf format in the chicken example above.



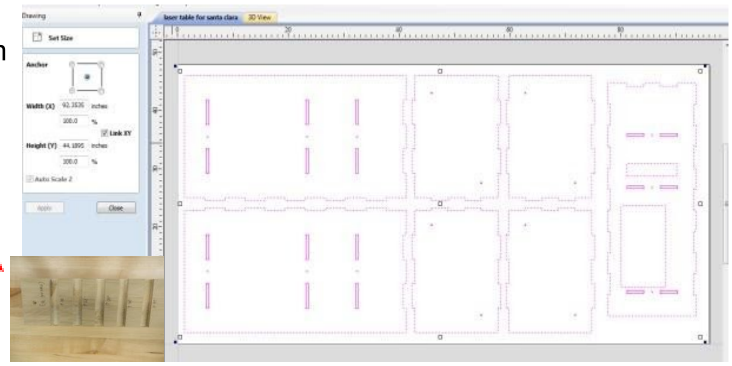
Exporting VCarve Pro Vectors for us in Another Software

(or transfer designs from later versions of VCarve Pro/Aspire to earlier versions)

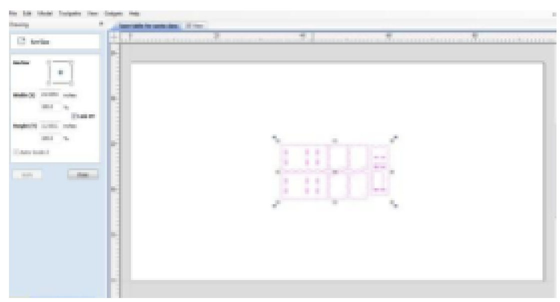
In some cases, you may want to prototype or test the concept of something that you designed for the ShopBot before you actually cut it out full size in expensive material.
Hint: Even though you cannot open .crv files created on a later version of VCarve Pro in an earlier version, you can export the vectors from the later version and import them into the earlier version.

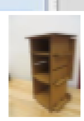
Use **File > Export** to save VCarve designs in .dxf, .eps, .ai and .svg format.

- The design at the right uses a full sheet of plywood to create a podium for a computer.
- The slot size was determined by creating a sampler with different sized slots.
  then using it to determine the correct thickness for the specific piece of plywood (.46”).



- The full sheet design was copied onto a new layer in VCarve Pro (RC for menu).
- The design on the new layer was scaled to test the concept with cardboard cut on a laser cutter.
- To scale the design:
  - The Link XY box is checked.
  - In the per cent box, divide the thickness of the cardboard (.125”) by thickness of the original slots.
  - Do the calculations right in the box in the software (125/460=).





To Export a VCarve Pro/Aspire Vector Design

Select the vectors to export.
Use **File > Export** to save the vectors in the format for the digital fab tool.

- .eps and .ai work well for many lasers.
- .dxf may be needed for Silhouette vinyl cutter.

Bring in the saved vector file to the desired software and lay out.

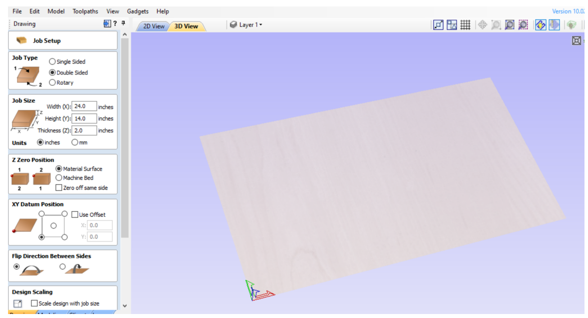
- If bringing the data in for use in laser cutting, do not forget to set the vector (stroke, line) thickness to that needed for vector work in the laser (refer to manual for a specific laser for more details).
- The color of the **vector** in the new software can be used to set speed and power of the laser.

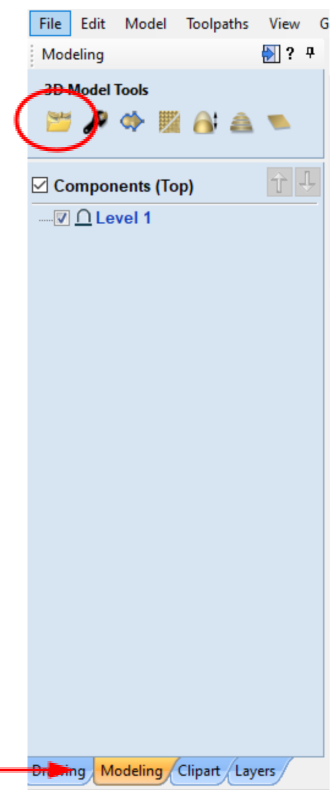
Importing and Toolpathing 3D Files in VCarve Pro/Aspire

While VCarve Pro is a 2D CAD software, it is a 2D and 3D CAM or toolpathing software. In Aspire, the designer can design in 3D, as well as import and manipulate 3D designs created in another software.

Setting up for importing a 3D file, especially one that will be two-sided, starts at the Job Set Up and Positioning screen for both VCarve Pro and Aspire.

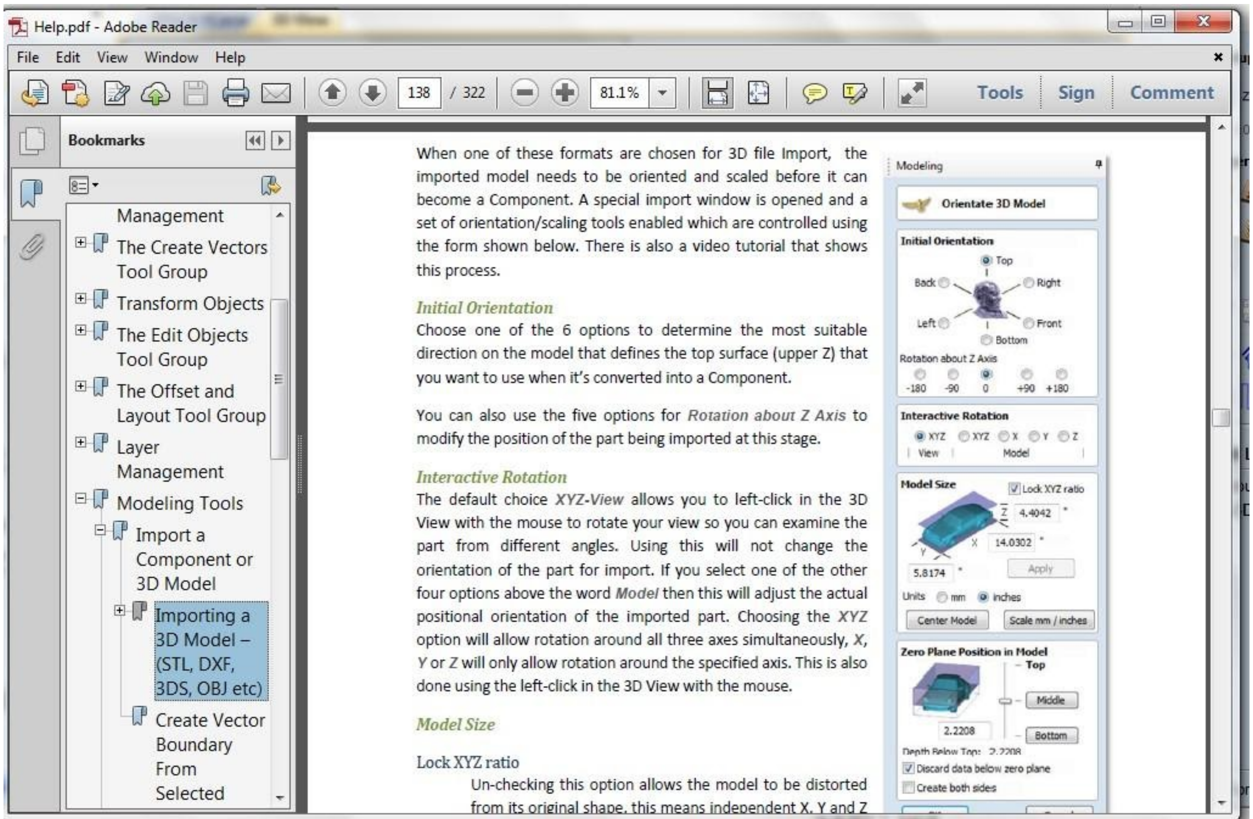
- In v 10 and later, the user can set the Job to be Single sided, Double sided, or Rotary.
- X = 24”, Y = 14”, Z = 2”
- Single-sided jobs are as above.
- With Double-sided comes more choices.
- Where the Z Zero is set defines the Z value of the ShopBot code, and where to actually Zero the Z when setting up the ShopBot.
- Define which way the material will be flipped.





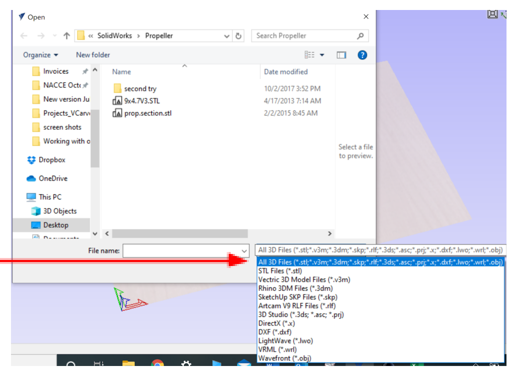
- Select the **Modeling** Tab.
- Use the Import Component or 3D model icon.
- Alternatively: **File > Import > Component/3D Model.**
- The process is the same for VCarve Pro and Aspire, with these differences:
  - Only one 3D model can be imported into a session of VCarve Pro. Multiple 3D models can be imported into Aspire.
  - The options for what you can do with a model are greater in Aspire, reflecting its advanced 3D modeling capabilities.

The **HELP** file in VCarve and Aspire is excellent, please refer to it for details. The Help is bookmarked, so when clicking on the item will open that page.



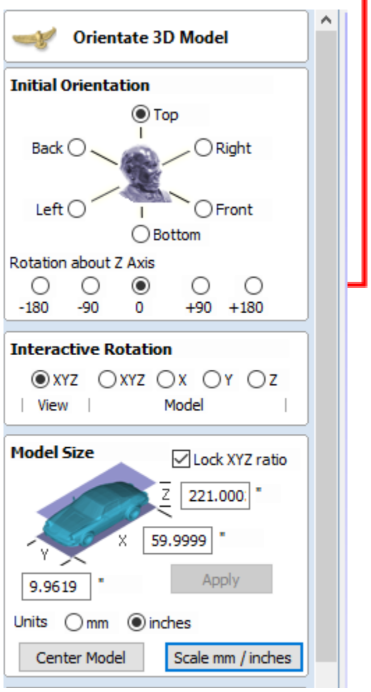
Bring the .stl into VCarve Pro or Aspire
(Note: The version of VCarve Pro must be 7.5 or later):

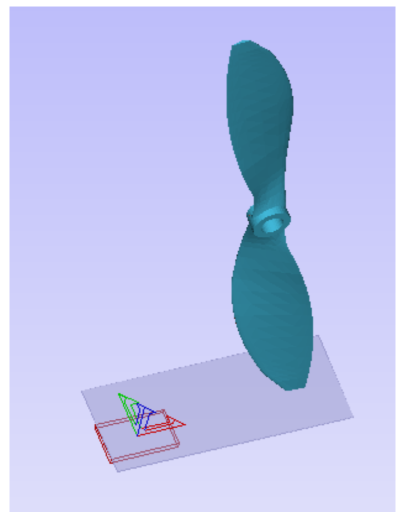
- The types of 3D models that can be brought into VCarve Pro or Aspire are listed here.



The component may not come in in the orientation best suited for machining in the material

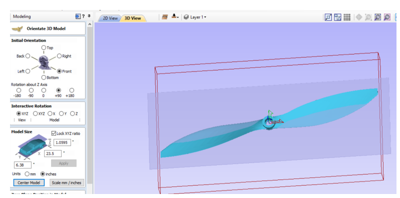
Keep clicking on the orientation to align the model with the material.



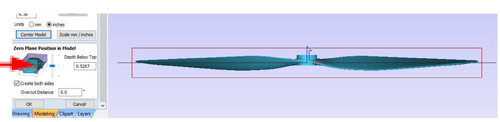


The model size can be adjusted with any axis as long as the XYZ ratio is locked
.stl files do not have a unit associated with them, so scale in mm or inches as desired.

Model fit into material.

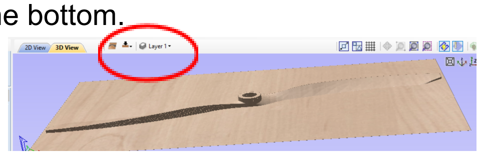


When the model is centered in the material and both sides checked, VCarve/Aspire will create two sides which can be machined by flipping the material.

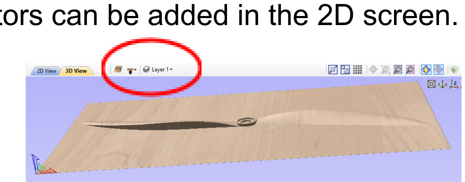


Where the two sides are sliced can be adjusted with the slider.

The imported file can be viewed from the top or the bottom.



Vectors can be added in the 2D screen.



A blog on the ShopBot website has suggestions for how to set up and machine 2-sided projects
http://www.shopbotblog.com/2019/10/machining-two-sided-3d-model-created-solidworks/

Importing and Machining More than One 3D file in VCarve Pro

3D scanners have become popular for creating models for 3D printing. Many of the same files used for 3D printing can be machined on a CNC like ShopBot in a larger format on different materials much faster than 3D printing.

How could a teacher use this in a class project? How about creating a board (2 x 4 or 2 x 6) with multiple students' images ? Is this like a totem pole? What can one teach about history and culture by studying totem poles? Are the more important images placed at the top of the totem pole, or at the bottom?

Only one 3D file or component can be imported into a session of VCarve Pro. A workaround to do multiple 3D carvings on a single board takes advantage of one of the features of Vectric software.

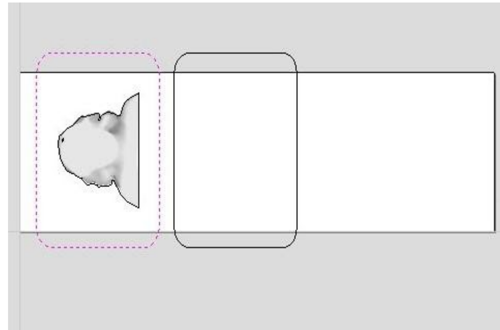
Copy and Paste in VCarve Pro/Aspire

- When something is **Copied** in VCarve Pro/Aspire, the software remembers the object(s) AND its location.
- So when the object is **Pasted**, it pastes into the original location.
  - If the original object was not moved, then multiple copies are pasted on top of each other.
  - If the original object was moved to a new location, the copy will paste back into the original location.
  - That is important to know when copying and pasting within a session of VCarve Pro/Aspire.
- The **Copy** and **Paste** feature is also true when an object is **Copied** in one session of VCarve Pro/Aspire and **Pasted** into another session of VCarve Pro/Aspire.
- This feature can be used to place multiple 3D carvings onto one piece of material, even when the 3D files are created in multiple sessions of VCarve Pro.

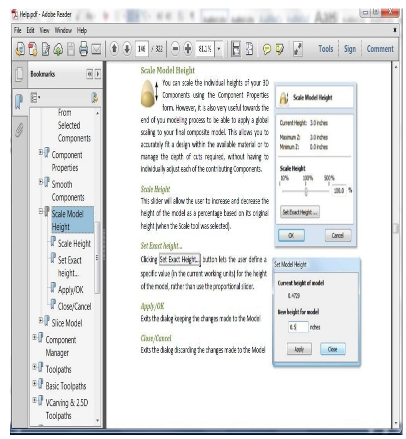
Getting Started

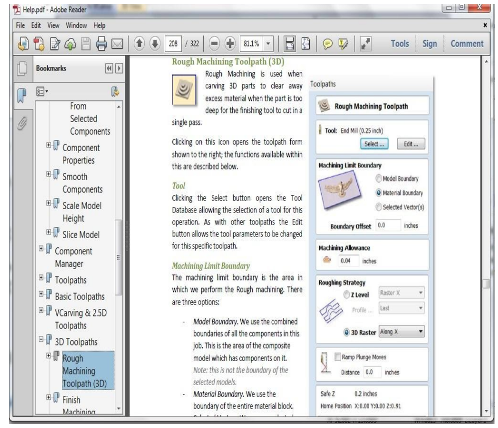
First Session of VCarve Pro: Plan for the placement of all the models in the 2 x 6 board.

- Set the Job Size and Position for the full board.
  - X = 24” (arbitrary length of design area)
    Note: to allow for a safe area to put hold down screws, the actual board is longer than the 24” design area.
  - Y = 5.5”, Z = 1.5” (true size of a 2 x 6).
  - Z Zero set at top of material.
  - **OK.**
- On the virtual representation of the full board, draw in rectangles for where each of the 3D models will go on the material.



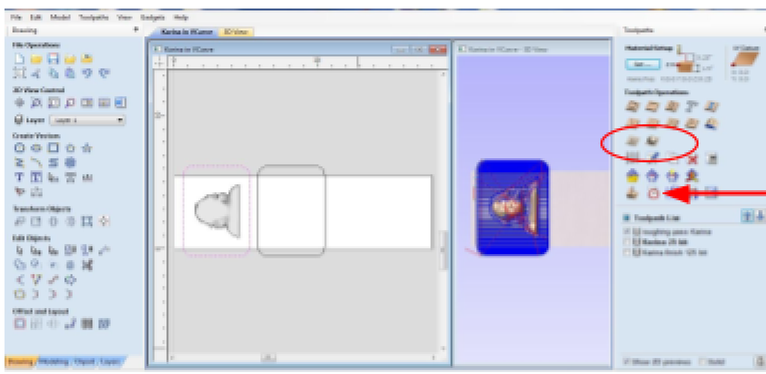
- **Import the first 3D model (.stl) as above.**
  - In this case, only the front of the model (the face) will be machined.
  - Set the slider bar until the front half of the model with no undercuts is selected.
  - Check the **Discard Date Below Zero Plane** box.
- Size and rotate the model until it fits into the first rectangle.
- If the height of the model in the desired X and Y dimensions is too tall for the thickness of the material, the model can be flattened to keep the features of the full 3D, but would not be as thick as the original when viewed from the side.
  This is thinner model is called a bas (pronounced “baa”, French for “low”) relief.
- The HELP file in VCarve has more details.





- **Create the Roughing Pass for the first 3D Carving.**
- Select the rectangle surrounding the component.
- In this case, a .25” end mill bit was used for the roughing passes.
- See the section of **Bit Selection, Feeds and Speeds** for more information about settings in the Tool Library.

- **The screen below shows both 2D and 3D views.**
  - The 2D (left view) shows the placement of the component surrounded by a rectangular vector.
  - The 3D (right view) shows the 3D rendering of the component with the roughing passes overlaid.
  - The CAD menu screen (left side) and CAM menu screen (right side) are also shown.
  - The 3D Toolpathing options are circled.



Create multiple 3D Finish Pass toolpaths to test the relationship between the detail provided by a bit vs the amount of time required to machine the 3D carving.

In this first sample, for the 3D finish pass, two bit options were tested (virtually): a .125” ball nose and a .25” ball nose.

- Create two 3D Finish toolpaths, one for each bit. See section on **Bit Selection, Feeds and Speeds** for setting in Tool Library for 3D Finish Pass.
- As above, choose the rectangle vector around the 3D model to set where to machine the model.
- Use the icon that looks like a clock to see the estimated time for each of the toolpath options.
- Use the **Preview** icon to simulate each of the toolpaths Remember to **Reset** the preview in between simulating the toolpaths.
- While the .125” ball nose bit would give more detail, it would take more time to machine than would the .25” ball nose bit This particular model does not really need the detail from a .125” bit, so the decision was to save the toolpath using the .25” ball nose.

Save the roughing and finish toolpaths (.sbp files: use the “floppy disk” icon).

- Save the roughing pass separately from the finish pass.
- Give each toolpath a name that can located later
  Perhaps include the name of the person, roughing or finish pass, bit used.
- Save all the toolpaths in the same folder so that they can be called up easily one after the other.

Save the .crv (VCarve CAD/CAM) file itself.

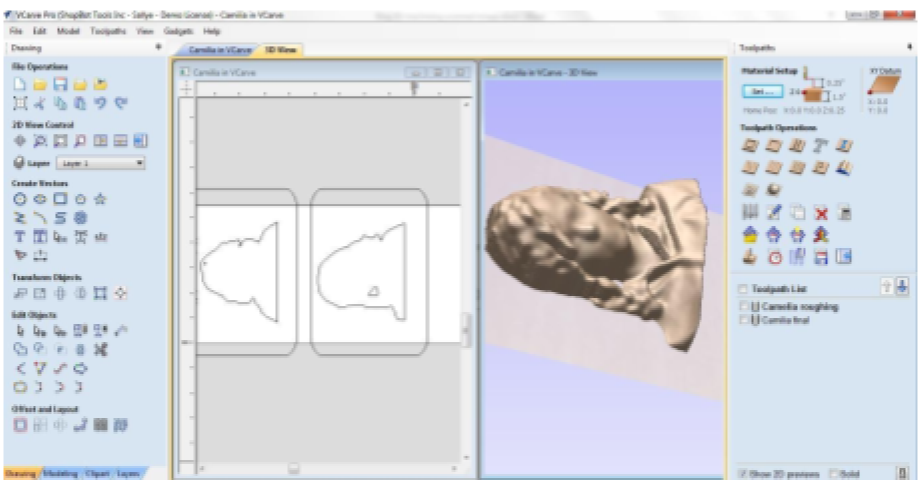
- Give it the name of the first person scanned.
- **Copy** the rectangles from the first file.

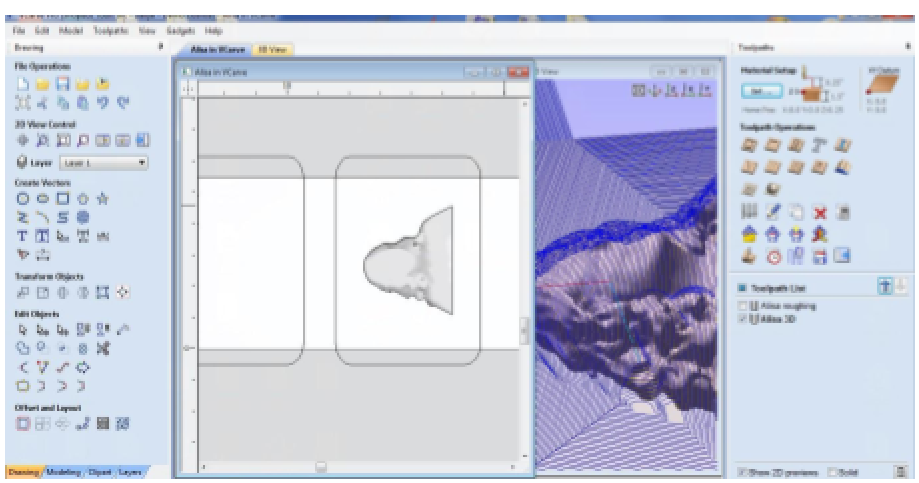
Prepare to Import the Second (and other) 3D Files.

Open a New VCarve File.

- The model size should be the same as the one just closed or saved.
- After the **OK**, so the virtual model of the 2 x 6 is on the screen, **Paste.**
- The rectangle created in the first session of VCarve should now be in the same place as the newest version of VCarve.

Repeat the process for importing each of the other 3D files, but in the next location on the board





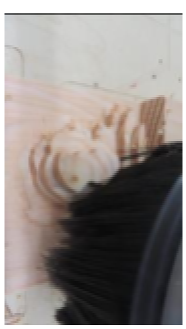
Prepare the Board for Machining on the ShopBot

Attaching the Board to the Table and Zeroing the X and Y axes: The photo below shows the board after all 3D carving completed to emphasize how the **hold down** was placed outside the area of travel for the machining.

- The board was placed on the table so that the bit would not go outside the range of the travel in the Y axis (board moved away from the long edge (X axis) of the table).
- The board itself is longer than the 24" defined in the Job Set Up. In order to have a hold down area that would be safe from the bit, the X and Y are zeroed with a blank area to the left of the start point of the machining.
    - Arrow points where to position the bit at the location where first file would start (use Keypad).
    - Use the **Z2** command (or button on Keypad screen) to set that location as X and Y Zero (0,0).
- In the blank area at left side of board, and at a location on the right (out of the picture), holes for screws to hold down the board were drilled through the 2 x 6 to keep the board from splitting.
- Sheet rock screws were screwed through the board to the sacrificial board under the 2x6 See red rectangles for placement of screws at the top of the board.

Run all Three Roughing Passes

- Insert the .25" end mill bit in the spindle, and Zero the Z at the top of the material.
- Run each of the ShopBot (.sbp) files that use the .25" inch endmill to rough out the faces, one after the other.



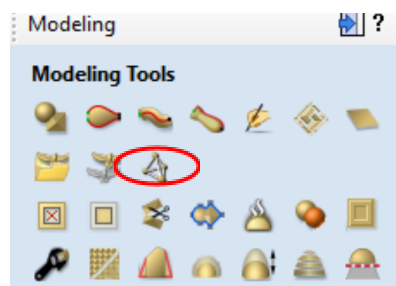
Run all Three Finish Passes

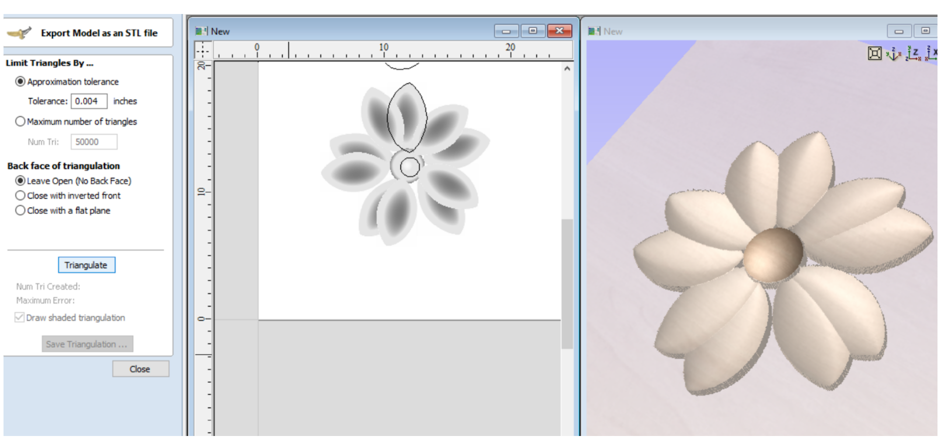
- Insert the .25” ball nose bit in the spindle, and Zero the Z at the top of the material.
  - Run each of the ShopBot (.sbp) files that use the .25” inch ball nose bit to finish the faces.

After machining, the files were cut apart using a scroll saw

Exporting 3D Models from Aspire

- 3D models cannot be exported from VCarve Pro.
- 3D models created or modified in Aspire can be triangulated and exported as .stl files for use in other 3D modeling software or 3D printing.





Comparing VCarve Pro/Aspire to Fusion 360

VCarve Pro/Aspire are very powerful CAD/CAM program with an emphasis on graphic- or artist-based design.

Fusion 360 is a very powerful 3D Rendering CAD/CAM program with an emphasis on engineering design. It is partially cloud-based, but still requires a powerful computer to run efficiently. (It currently does not run on Chromebooks…try TinkerCAD or Easel.) Cool feature: Fusion 360 is parametric: Under Modify, Users can define parameters. Change the value of the parameter, the drawing will change accordingly.

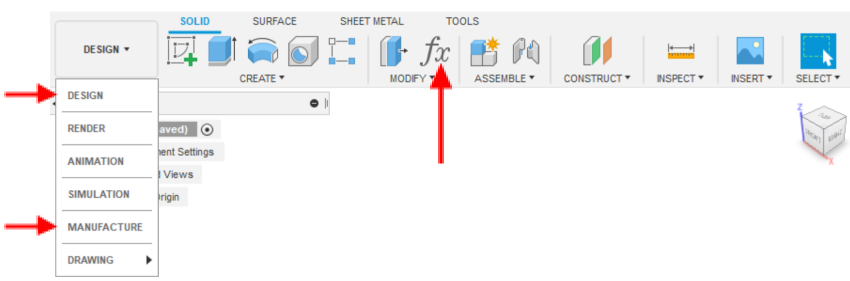
The two software use many of the same terms (example: component), but they mean different things. Likewise, they have similar abilities that are called different things (example: level vs. component.) Laying out the parts on material (stock) occurs at different times. The following two pages compare the two software.

Creating A Fusion 360 Account

A year-long license for Fusion 360 Included with the purchase of a ShopBot CNC is (information in email from ShopBot). Fusion 360 is free for educators and small businesses. Try this link (as of Aug 2020).
https://knowledge.autodesk.com/support/fusion-360/learn-explore/caas/sfdcarticles/sfdcarticles/How-to-activate-start-up-or-educational-licensing-for-Fusion-360.html

Getting Started:

Fusion 360 has many capabilities that may not be relevant for someone creating a file for machining out of sheet goods on a ShopBot. The two most relevant sections are Design (2D Sketch and 3D Bodies), and Manufacture. Since the learning curve for Fusion can be extensive, when getting started, concentrate on Tutorials that emphasize those two features. The User Interface for Fusion is frequently updated, so be flexible when watching older videos.

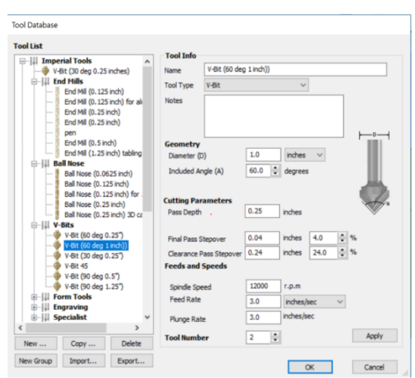


There are many strategies when designing in Fusion 360. Spend a little time thinking ahead of how you are going to “print” the design, and how you might want to change it later:

- 3D print it (additive)
- Machine it (subtractive)
  - 3D carve (save as .stl).
  - 2D out of sheet goods (save as .dxf) Planning: How many components (different sides) do you need for cubes like these?

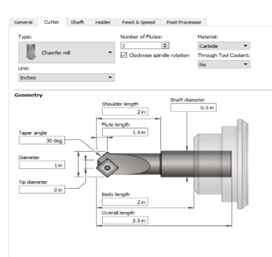
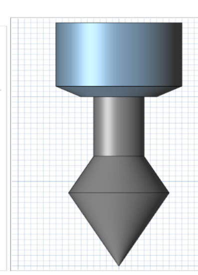
VCarve Pro/Aspire

Artist/Graphics Based Z up by default

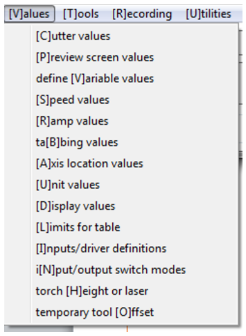
2D Drawing (Vectors)	2D View Drawing Tab	● Only draws in one plane (top or Z down view). ● Vector drawing must be "closed" to create some toolpaths: Profile outside/inside; pocket; VCarving. ● 3D models show up in grayscale in 2D view.
3D Rendering	3D View Modeling Tab Tile 2D and 3D view to see both 2D and 3D on same screen.	● Simulated toolpaths are rendered in 3D in 3D view. ● **VCarve: import .slt for toolpathing.** ○ 2D view: grey scale. ○ 3D view: 3D rendering. ○ Can only import one 3D model at a time, no export 3D ● **Aspire: create 3D model from 2D vector.** ○ 2D view: grey scale. ○ 3D view: 3D rendering. ○ Can create 3D component from bitmap in Modeling. ○ Can export mesh as 3D object for 3D printing.
Layer vs. Component	Vectric uses Layers to organize data.	● Layers can contain Vector drawings, 3D components, sides, etc. ● Layer visibility can be turned on and off. ● Definition of a Component = 3D model.
Toolpathing	Set X and Y Origin Set Z origin. Set Z Cut Depth.	● Origins are set in Job Set Up before design phase (CAD). ● If origins/locations change once toolpaths are generated, all toolpaths and .sbp files must be regenerated. ● 2D Vector Drawings. ● 2D Toolpaths: Profile, Pocket, Drill, Engraving. ○ X and Y movement set in drawings. ● Z value (Depth) is added in CAM. ● 3D Roughing pass is treated like a 2D file. ● 3D Designs: Finish pass. ○ 3D model sets the Z depth.
Bit information Feeds/ Speeds	Add/Edit Tool (Bit) information in Tool Library.	

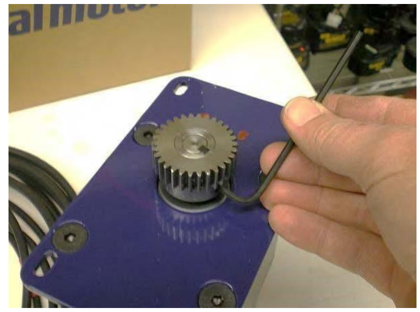
Fusion 360

Engineering/Model Making Based Define Z up in Parameters (under Profile name)

2D Drawing (Vectors)	Menu: **Sketch**	● Chose Face or Plane to start Sketch on (X, Y, Z or custom) ● Constrain and "close" the sketch to extrude into Body. ● "Stop" Sketch when it is completely constrained. ● "Edit" Sketch by clicking on Component tree or timeline.
3D Rendering	Menu: **Create**	● 3D objects are called "Bodies". ● Create 3D renderings of model by Extruding from Sketch or Creating directly as 3D object. ● Orbit, rotate, home to move around in 3D rendering. ● Export 3D model as .stl for 3D printing or CNC machining.
Layer vs. Component	“Component” in Fusion is more like a “Layer” in Vectric	● A component serves as a “container” to gather a variety of design objects, including: Bodies, Sketches, Construction Geometry, etc. ● Component visibility can be turned on and off. ● One difference between a Body and a Component. ● Copy and Paste a Component: change something in the original and it is reflected in the Copy. ○ Copy and Paste a Body, each Body remains separate. ● A Component can also be an object that has its own origin and is capable of movement on screen.
Toolpathing	Set X and Y Origin	● Origins for machining are set up in CAM (Manufacturing) Origins are set up in the “stock” or material. ● For 2D machining on sheet goods like plywood, all sketches created in other planes must be “joined” to the plane where the stock is defined. ● If origins/locations change once toolpaths are generated, all toolpaths and .sbp files must be regenerated. ● Cut Depth (Z axis), whether in 2D profile for sheet goods or for 3D carving, is set by the model. ● To export design for laser cutter or 2D CNC: convert Sketch and export .dxf. A Sketch can be created from a Body. ● To export design for 3D printer: send through Meshmixer or export as .stl.
Bit information Feeds/ Speeds	Need Collet nut info. Can select bits from inventory or create own library.	

Troubleshooting and Maintenance





While ShopBot Tech Support is just an email or phone call away, knowing where to start looking when a problem crops up saves time for ShopBotter and Support Tech alike. This section offers some suggestions for troubleshooting common problems.

Running a ShopBot is a marriage of software (CAD/CAM and Control), and hardware/mechanicals (bits, collets, the ShopBot itself.) The first part of this Chapter looks at software settings in both ShopBot Control software, and CAD/CAM software. The second part looks at the “hardware” and mechanicals.

Like any mechanical device, taking a few minutes to tune up and maintain the ShopBot on a regular basis will prevent frustration later when a part is not coming out the way it should. Tips for maintenance are offered at the end of each sub-section, with references to maintenance schedules for each model of ShopBot.

This section references both ShopBot Control Software (SB3) and VCarve Pro CAD/CAM software included with the purchase of the ShopBot. If the CAD/CAM software of choice is something other than VCarve Pro/Aspire, general concepts for many settings in the design or machining files still apply.

Overview: Troubleshooting and Maintenance

Read through the list of problems to start with a solution in the software settings (ShopBot Control software or CAD/CAM files), or the mechanicals or accessories of the ShopBot.

Problems that are Usually Related to Software Settings

- ShopBot seems to be moving at the wrong speed.
  - The Move and Jog Speeds can be set in the ShopBot Control software directly, and/or in the Tool Library in the CAM software.
- The Z axis is not cutting at the right depth: too deep, or not deep enough.
  - The material is not being cut through, even though the pass depth is correct for the material thickness. Automatic Zeroing Routines may not be sending the ShopBot to the correct zero location.
  - Z-Zero location set in the CAM file and the Z-Zero set on the ShopBot do not match.
- ShopBot is slowing down too much OR slams into sides in when it changes direction.
- ShopBot was working fine, but is now acting up.
- The ShopBot is moving the wrong distance and/or the wrong direction.

Problems that are Usually Related to Mechanical Issues

- Problems with Bit or Collet.
  - Bit is breaking at feeds and speeds that use to work.
  - Bit is not holding position in Z.
- Cuts aren't as precise as they should be.
- Circles aren't round.
- Rectangles are not cutting well on one or two sides.
- Z is not holding its position.
- Ridges occur when surfacing the Table.

Doing Maintenance on the ShopBot

The maintenance routines for all models of ShopBots are included in the Assembly Manuals for the particular model, or available on the website www.shopbottools.com.

ShopBots are designed so that the Operator can tune up and maintain most models with a few readily available tools quickly and easily. If you would prefer an on-site maintenance call from a ShopBot Technician for a thorough tune up and some hands-on training, contact Support@shopbottools.com or Sales@shopbottools.com.

Software Solutions

Problem: ShopBot Seems to be Moving at the Wrong Speed
While it may be confusing at first, after some experience with a ShopBot, you will develop a sense of when the ShopBot is not moving at the correct speed. A table below has some suggestions for the starting settings of different models of ShopBot.

The current Move Speeds (Cutting speed in X&Y and Z) and Jog Speeds (Positioning Speed in between Cuts) are displayed on the ShopBot Control Console.

The Speeds are set by:

- Entering the speeds directly in the boxes shown on the Control Console.
- Entering the **MS** (**JS**) or **VS** commands from the Command Tool bar.
  - The speed displayed is valid when using the Keypad Control to move the ShopBot around, or entering Move or Jog Commands directly into the Command Line.
- Lines of code in ShopBot Macros like the XY Homing Routine or Z-Zero Plate routine.
- Entering Move and Plunge Speeds in the Tool Library in a CAM file.
  - A line of code in a .sbp file will override a Move Speed set manually in the boxes on the Control Console.

Suggestions for Move and Jog Speeds for Different Models of ShopBots

Gantry Tools and Buddy (alpha)

Bit Type	Toolpath Type	XY Move Speed	Z Plunge Rate
End Mill	2D Profile, pocket, drill, 3D roughing	4 ips (100 mmps)	1 ips (25 mmps)
V Bit	V Carving, Engraving	2.5-3 ips	2.5-3 ips
Ball Nose Bit	3D Finish (simultaneous XYZ moves)	6 ips	6 ips
	JOG SPEEDS	6 ips	3 ips

Gantry Tools and Buddy (Standard): may need to slow down if ShopBot is struggling

ShopBot Desktops

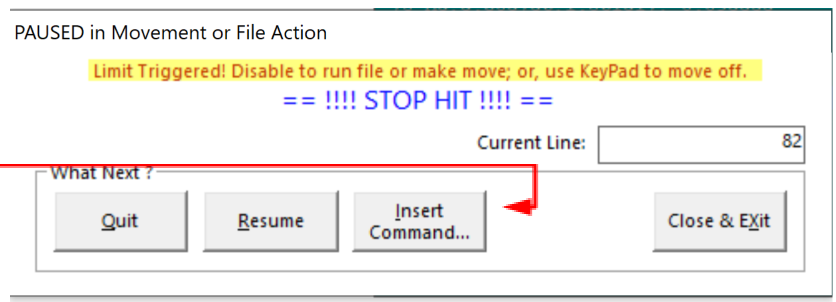
Bit Type	Toolpath Type	XY Move Speed	Z Plunge Rate
End Mill	2D Profile, pocket, drill, 3D roughing	3 ips (100 mmps)	1 ips (25 mmps)
V Bit	V Carving, Engraving	2.5 ips	2.5 ips
Ball Nose Bit	3D Finish (simultaneous XYZ moves)	4 ips	4 ips
	JOG SPEEDS	4 ips	2 ips

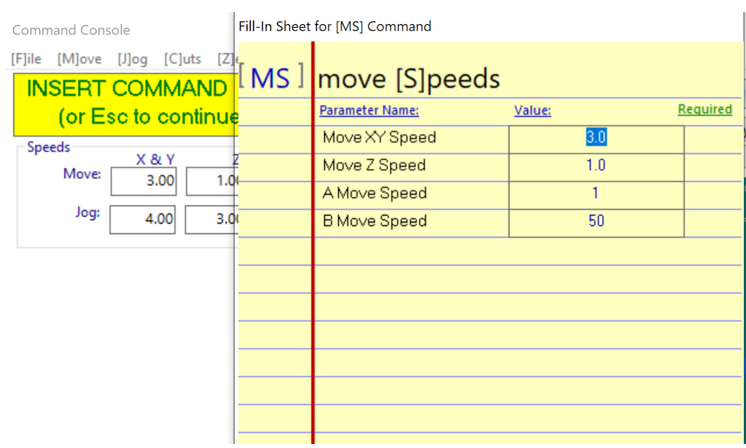
More information on setting Move and Jogs Speeds can be found in the **ShopBot Safety and Basic Use**, as well as **Bit Selection, Feeds and Speeds** Chapters of this Handbook.
If the move speed seems wrong in a file, there are ways to correct it:

- Temporarily in the middle of a run.
- Permanently in a specific file.
- In the Tool Library for a specific bit/type of toolpath for future files.

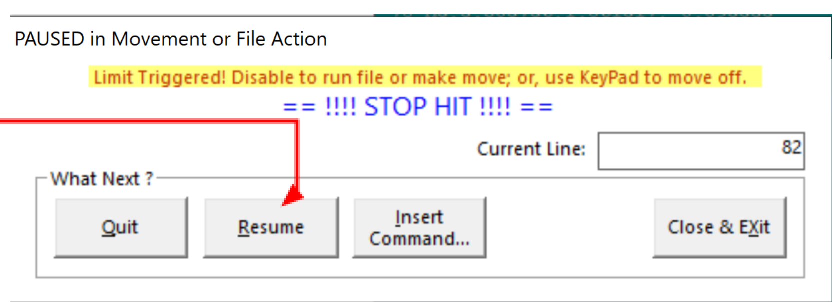
Changing the Move Speed in the Middle of Running a ShopBot File
If the ShopBot is moving at the wrong speed, but now is not the time to either edit the .sbp file directly or the settings in the Tool Library for creating future toolpaths, **Pause** the ShopBot by hitting the **Space Bar** or Clicking on the **Stop** Button on the Red Position Screen.

- Click on the **Insert Command** button given as one of the options.





- Type **MS** into the Command Line.
- Enter the new Values in the Yellow Move Speed Screen.
- Enter **OK** at bottom of yellow screen.



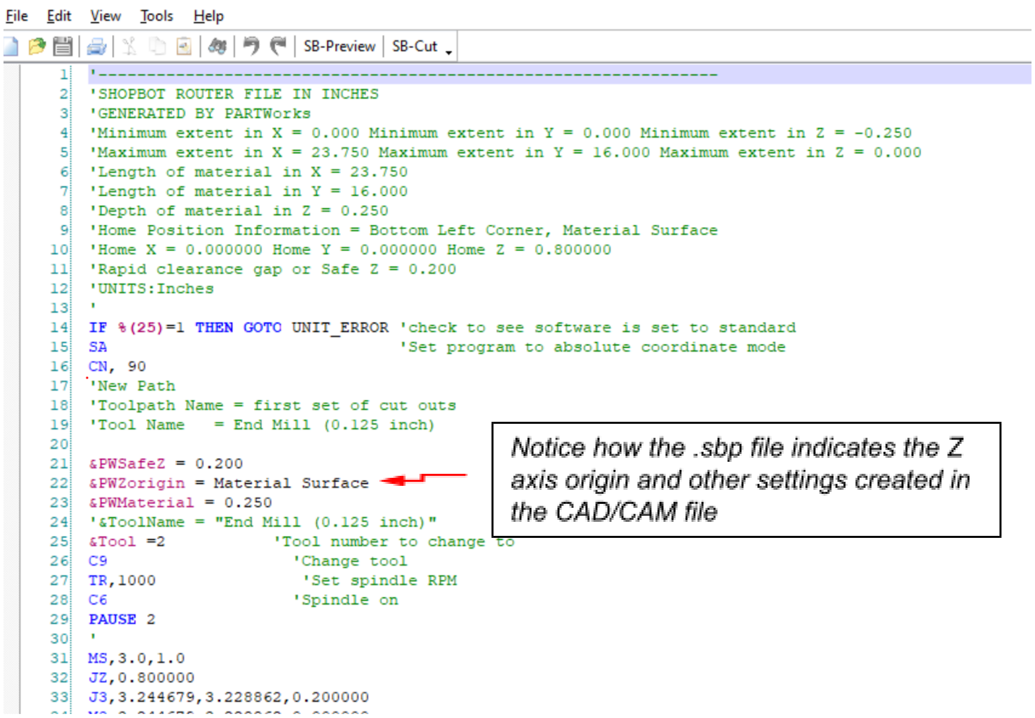
- **Resume** the file, remembering to restart the spindle or router at the prompt.
- It is also an option to **Quit** the file.

The Move Speed will stay at that value until a new Move Speed (**MS**) command is encountered by the ShopBot Control software.

Editing a .sbp File to Change the Move Speed in that Specific File

ShopBot Parts (.sbp) files are text files with commands to set Move Speed, Turn the Spindle On and Off, Trigger a Bit Change, and send the ShopBot to the correct location in the X, Y and Z (or more) axes.

ShopBot Parts (.sbp) can be viewed and edited in any text program. From the Command Console, **FE** (File Edit) will bring up the ShopBot code.




```
1  '----------------------------------------------------------------
2  'SHOPBOT ROUTER FILE IN INCHES
3  'GENERATED BY PARTWorks
4  'Minimum extent in X = 0.000 Minimum extent in Y = 0.000 Minimum extent in Z = -0.250
5  'Maximum extent in X = 23.750 Maximum extent in Y = 16.000 Maximum extent in Z = 0.000
6  'Length of material in X = 23.750
7  'Length of material in Y = 16.000
8  'Depth of material in Z = 0.250
9  'Home Position Information = Bottom Left Corner, Material Surface
10 'Home X = 0.000000 Home Y = 0.000000 Home Z = 0.800000
11 'Rapid clearance gap or Safe Z = 0.200
12 'UNITS:Inches
13 '
14 IF %(25)=1 THEN GOTO UNIT_ERROR 'check to see software is set to standard
15 SA                              'Set program to absolute coordinate mode
16 CN, 90
17 'New Path
18 'Toolpath Name = first set of cut outs
19 'Tool Name   = End Mill (0.125 inch)
20
21 &PWSafeZ = 0.200
22 &PWZorigin = Material Surface
23 &PWMaterial = 0.250
24 '&ToolName = "End Mill (0.125 inch)"
25 &Tool =2             'Tool number to change to
26 C9                   'Change tool
27 TR,1000              'Set spindle RPM
28 C6                   'Spindle on
29 PAUSE 2
30 '
31 MS,3.0,1.0
32 JZ,0.800000
33 J3,3.244679,3.228862,0.200000
```

- In the above example, Line 31 reads: **MS**, **3.0**, **1.0** (Move Speed, 3 ips in X&Y, 1 ips in Z).
  - This was written for a ShopBot Desktop, so the X&Y Move Speed was set at 3 ips. To change the MS to run on a ShopBot alpha Gantry Tool at 4 ips, change the 3 to a 4.
  - The Move Speed for any file is set in the Tool Library on the Computer that writes the Toolpath. If Tool Libraries on different computers don't match, it is very common for the Move Speed to be different than intended. If the only thing wrong with the File is the Move Speed, just edit it here and save the .sbp file.
  - If there is more than one toolpath in a .sbp file, do an **Edit** and **Find** for **MS** to correct all the Move Speeds in a .sbp file.
- As long as there is a .sbp file open, here are some other things to look at:
  - Line 27: **TS** sets the spindle speed in Tools with a Spindle Speed Controller.
  - Line 25: **&Tool** sets the Tool number (important for ATC users).
  - Note: An **&** followed by a name signals a variable. Many variables are set in the Vectric File, either in Job Setup or Tool Library. Reading the variables will give a clue where they came from.
  - 'Anything with an apostrophe in front of it is a comment for the user, not the ShopBot.

Editing the Tool Library for Specific Bits and Toolpaths

REPEAT: The Move Speeds for any file are set in the Tool Library on the Computer that writes the Toolpath. If a Toolpath was generated on one computer, then modified on another computer, and the settings in the Tool Libraries don't match, it is very common for the Move Speed to be different than intended.

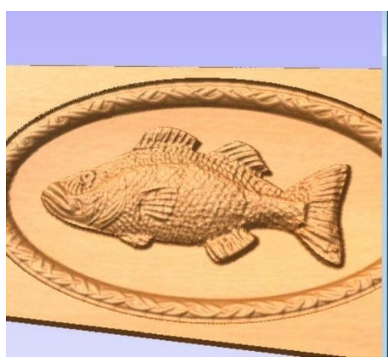
While VCarve Pro, ShopBot Edition, came with a Tool Library installed, ShopBotters using Aspire or VCarve Pro V10 and later have to install or create their own Tool Library. The Tool Library has to be created or modified on every computer that will be designing and toolpathing for the ShopBot.

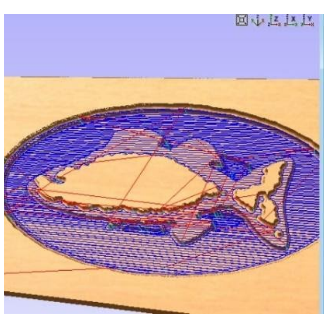
Fortunately, a tool library can be saved from one computer, then installed on another computer. A Tool Liibrary for the ShopBot Starter Bit Kit can be found on the flash drive accompanying this notebook, or obtained from ShopBot support. The HELP in VCarve Pro/Aspire is another resource for details and specifics.

The Chapter on **Bit Selection, Feeds and Speeds** has more information on what values to enter. A laminated insert in this Handbooks has suggestions for getting started with bits from the ShopBot Starter Bit kit plus a few others.

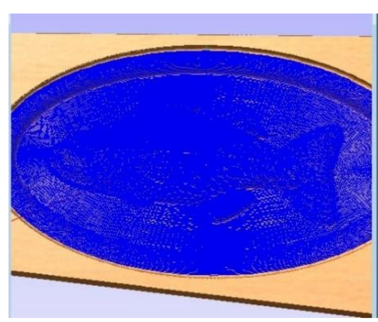
The following is an example of creating a new bit for 3D carving. In the example, the same bit is used for both 3D Roughing and 3D Finish. But separate tools are created in the Tool Library for each function. The bit created for 3D Roughing has different characteristics (feed rate, stepover, etc.) than the bit created for 3D Finish.

See page 8 for information on exporting a Tool Library, and importing it onto a different computer.



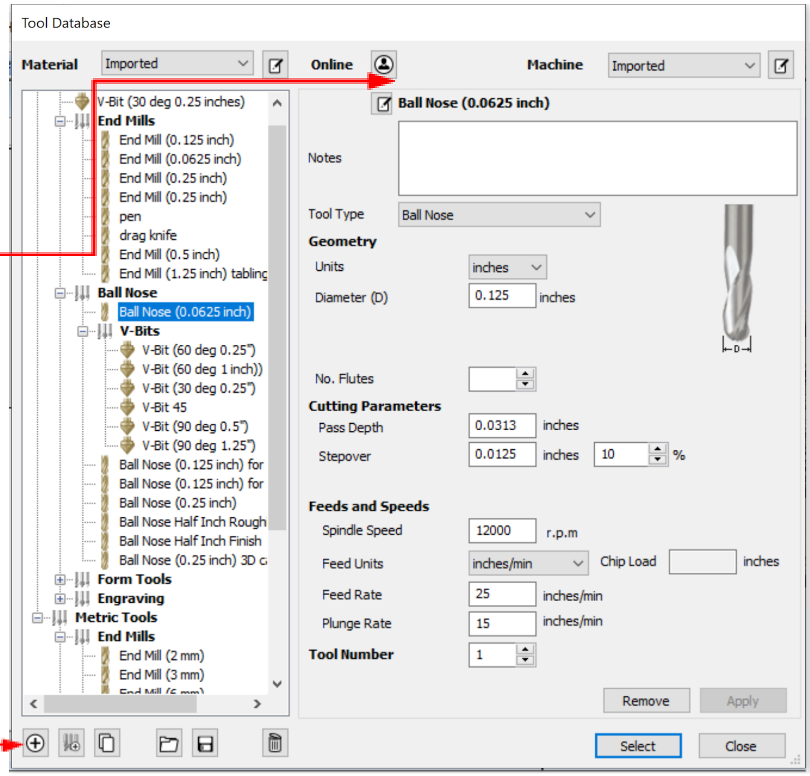


.25" Ball Nose for 3D Roughing has a Stepover of 40%, and Move Speeds of 4 and 1 ips.



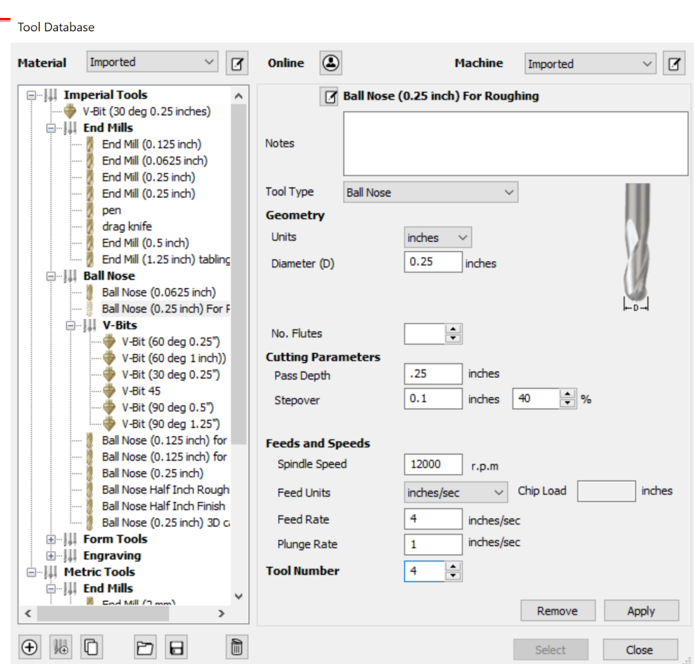
.25" Ball Nose for 3D Finish has a Stepover of 10%, and Move Speeds of 6 and 6 ips.

Getting Started



- This Tool Library has already been modified from the standard, but the highlighted bit has the default settings.
- Default Name.
- Units (inches or metric).
- Diameter (oops, in example, name and diameter don't match).
- Pass depth.
- Stepover.
- Spindle Speed.
- Feed Units (in inches/**min**).
- Feed Rate.
- Tool number (Defaults to 1).
- **Copy** the bit to start creating a new one.

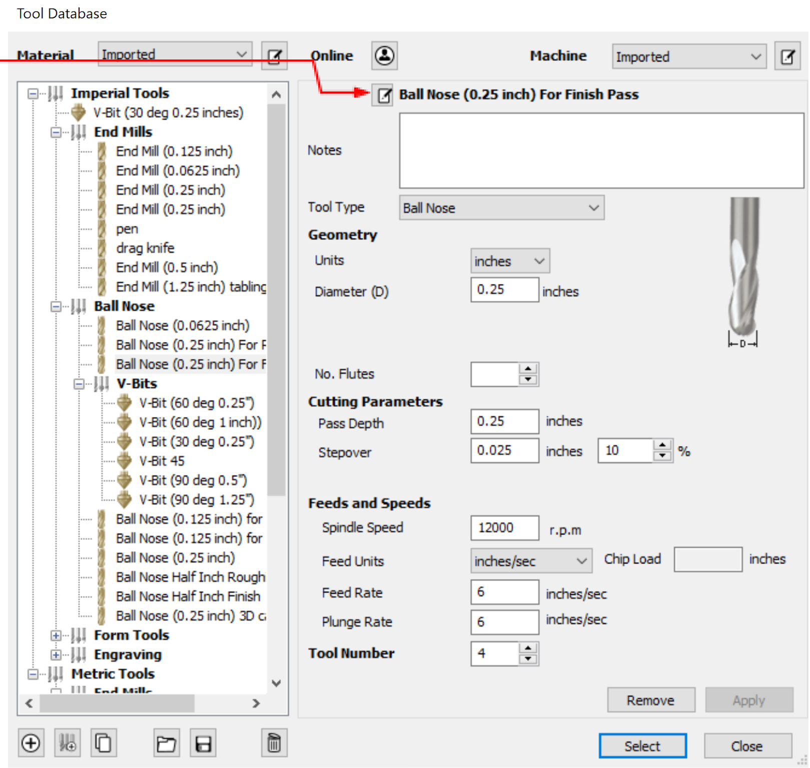
Modify the Copied Bit: 3D ROUGHING



- Rename to include size and purpose (double click to edit name).
- Units (inches or metric).
- Change Diameter.
- Pass depth (Change to diameter of bit).
- Stepover (40% for Roughing).
- Spindle Speed (12000).
- Feed Units (inches/**second**).
- Feed Rate: 4 and 1.
- Tool number (Defaults to 1. Define with different numbers if using ATC or if want ShopBot software to signal for bit change).
- After **Apply**ing, **Copy** again.

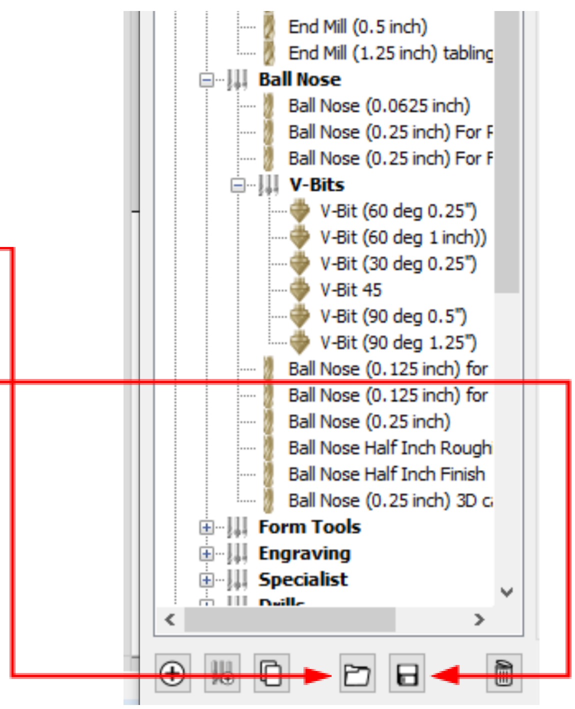
Modify the Copied Bit: 3D FINISH

- Rename to reflect bit size and purpose.
- Units (inches or metric).
- Diameter stays the same.
- Pass depth (Stays the same, but irrelevant for 3D Finish).
- Stepover: Change to 10% or less for Finish Pass.
- Spindle Speed (12000).
- Feed Units (inches/second).
- Feed Rate: 6 and 6.
- Tool number (Defaults to 1. Define with different numbers if using ATC or if you want ShopBot software to signal bit change).



Ensuring that all Computers have same Tool Library Settings

- Export a Tool or Tool Group. Save it to a flash drive and take it to each computer with VCarve or Aspire on it.
- Import the Tool or Tool Group onto another Computer from the Flash Drive.
- As always, see Vectric HELP for more information.

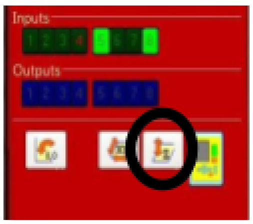


Problem: The Z axis is Not Cutting at the Right Depth: Too Deep, or Not Deep Enough

The Z is not cutting all the way through the material, even though the part file says it should be.

TEST: Is the value of the thickness of the Z-Zero plate used in the Z-Zero routine correct?

- Move the Z over an open area of the table.
- Run the Z zero routine using the Z -Zero plate
  Use **C2** or the icon shown
  (details in **ShopBot Safety and Basic Use** Chapter).
- Remove the alligator clip and store the plate.

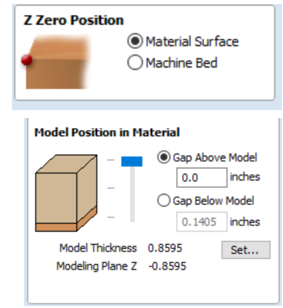


- Enter the command “**MZ, 0**” into the command line
  The Z should plunge to the surface where the Z-Zero plate was set.
- Test whether the bit is really sitting at the surface.
  - Try to slip a piece of paper under the bit.
  - If the paper won’t slide under the bit (and the bit is not plunged into the surface), the value of the thickness of the plate is correct.
  - If the bit is too high or low, the value of the Z-Zero plate needs to be modified.

- **To change the Value of the Thickness of the Z-Zero Plate in the ShopBot software.**
  - Inspect the plate to ensure that it is not warped or damaged.
  - Measure the thickness of the plate with calipers. Be as accurate as possible.
- With the ShopBot Control software open, go to (**T**)ools: (**S**)hopBot Setup.
  - Click through the screens.
  - The second screen displays what model of ShopBot the Software is set up for.
- (Run **UR** if the model is incorrect).
- It also where to set the units (in/mm).
  - The third screen (pictured) is where to enter the thickness of the Z-Zero plate.
- The default value is .121” (3.073mm).
  - The Next screen is where to set the offsets for the XY Homing Routine.

The Z is either plunging way too deep, or floating above the material

Be sure that the Z is Zeroed at the same location as in the CAD/CAM file.



- If the CAD/CAM file is set with Z Zero Position at Material Surface, but the ShopBot Z is Zeroed at the Machine Bed (aka Table), the Z will plunge too deep into the material and sacrificial board.
- If the CAD/CAM file is set to Z Zero Position at Machine Bed, but the Z itself is Zeroed at the top of the material, the Z will float above the material.
- When 3D Carving, check the position of the Model in the Material. Find this setting under **Material Setup** on the Toolpathing screen.

Problem: ShopBot is Not Taking the Corners at the Correct Speed

If the ShopBot is moving along well during straight moves, but slows down dramatically when going around corners or at the beginning of a 3D move, it is possible that the **Ramp Values** may have gotten corrupted. Another sign of Ramp Values being off is that the ShopBot is NOT slowing down when changing directions, but slamming into the sides before heading off again.

- “Ramps” in ShopBot Control Software sets how far ahead the software looks to slow down before changing direction or stopping. Imagine making a sharp turn while driving. You don’t do it at full speed, but slow down before the turn, then speed up as you come out of the turn.
- One way that Ramp values can get corrupted is if a set up routine like one of the Automatic Zeroing routines is stopped before it is complete.
- To reset the Ramps in the Software, use **VR** (Values Ramps.) Click on the “Reset” button at the lower left corner of the yellow Start page, then say **OK.**
- More information is found in the **Hidden Gems of ShopBot Software** section and the **Command Reference** under HELP.

Problem: ShopBot is Just Not Itself

Sometimes, the ShopBot just seems confused. Maybe the Software got shut down incorrectly because of a power failure. Maybe the Emergency Stop Button was pushed. Maybe a ShopBot alpha driver faulted because the ShopBot was asked to travel beyond its limits. Maybe it's a blue moon.

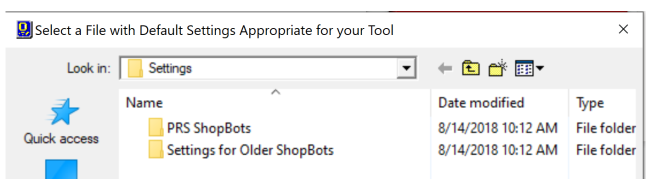
Before calling Tech Support, try doing a **UR** and reloading the .ini file. Shutting down the software, computer and ShopBot, then restarting everything may also reset ShopBot's attitude.

- Doing a **UR** for will **not.**
  - Change the values for variables set up in the Automatic Zeroing Routines.
- Doing a **UR** will reset some default values like:
  - Set the Limit Switches to Off when you'd prefer them On.
  - Change some of the Prox Switch settings.
  - Change the E-Stop Settings (Normally Open, Normally Closed).
  - Revert Axes to traditional settings if you've reversed them.
- To avoid having to change all the custom settings again, once they are as you like, SAVE a Custom .ini (**US**) with a name that can be remembered. Save it on a flash drive as well as the main computer so it can be reloaded in case a computer dies or the working .ini becomes corrupted.
- **Hidden Gems of ShopBot** Software has more details and Troubleshooting Suggestions.

Problem: ShopBot is Moving the Wrong Distance, or Wrong Direction

The ShopBot Control Software uses settings in Unit Values to determine the number of steps per unit (inches, mm or degrees) to achieve the correct distance to move. Factors that contribute to the Unit Value include type of motor, drive system (rack and pinion or lead screw), size of the pinion gears (rack and pinion tools), and step ratio of the motors. For most ShopBot models manufactured since 2004, the correct Unit Values are loaded automatically when the correct .ini is selected from the options found under **UR**.

If using the same laptop for different ShopBots, it is important to reload the correct .ini file for the ShopBot that the computer is currently connected to.



For much older ShopBots, and ShopBots that have been modified (on purpose or by accident), it may be necessary to change a Unit Value. Use the VU Command to modify the Unit Values. For Example:

- When switching the X and Y Axis on the Buddy using the Software, the X axis may run backwards. Put a negative sign in front of the X axis Unit Value, and it will run correctly. (Download the Buddy Switching Document from the ShopBot website for the rest of the modifications to change the X and Y axes).
- On occasion, the ShopBot is assembled with the gantry on backwards, or the motors plugged into the wrong drivers, or…. One solution is to put a negative in front of the Unit

Value on the axis running the wrong direction rather than changing the mechanicals. (Contact Tech Support for other options).

- On alpha Gantry Tools, the pinion gears for the X and Y axes are 30 Tooth, and the Z is 25 Tooth. On Standard Gantry Tools, the pinion gears for all 3 axes are 25 Tooth. While there are a number of other differences between the models, it is important to know if the ShopBot is an alpha or a Standard model and load the correct .ini file.
- If, by any chance, the wrong sized pinion gears are mounted on the motors, the ShopBot will not move the correct distance. It's easy enough to change the Unit Values in the ShopBot software to get it to move the correct distance until the correct sized pinion gears arrive and are swapped out. Contact Tech Support for details.
- Older PR and PRT ShopBots were shipped with a number of different configurations, so the current standard .ini files may not contain the correct Unit Values. Furthermore, an older ShopBot may have been updated with a new Control Box, Motors, Mechanicals, so it may take a bit of sleuthing to get the Unit Values Correct. The link below gives information on how to identify the ShopBot you have.
  - https://www.shopbottools.com/ShopBotDocs/files/SBG%2000171%20Settings%20Unit%20Values%202014%2012%2017.pdf
  - The document titled **Command Reference** in the ShopBot Control Software under HELP has details on how to calculate Unit Values if standard ones don't work. Click on UV for info.
- Once the Unit Values and other setting correct are correct, use the US command to SAVE a custom .ini file. The file can be loaded on a flash drive and loaded onto a different computer if needed.
- At the very least, take a photo or screen shot of the settings that work to have a record in case there is a problem with the computer running the ShopBot.

Hardware and Mechanical Solutions

This section will start with the items that are common to all ShopBots (for example, bits and collets) before moving into areas that are specific to different models of ShopBots.

Bits and Collets

A Note about Bits and Collets: For the best results, use a sharp, clean bit. Using the correct bit at the correct feeds and spindle speeds will help to maintain bit life. Working with a good, clean collet to hold the bit in place is even more critical.

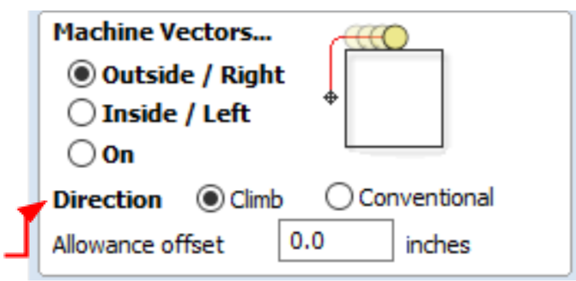
Getting the right finish on the parts:

- An upcut spiral will pull up fibers on the material, leaving the top edge fuzzy.
- A downcut spiral will push the fibers on the material down, leaving the top edge clean.

A downcut spiral also helps with hold-down by pushing down on the material.

- A compression bit is an upcut at the bottom, and a downcut at the top, so works well to leave both sides of a sheet of plywood clean. A compression bit works best when cutting in a single pass.
- A straight bit is neutral as far as leaving the top or bottom edge clean.

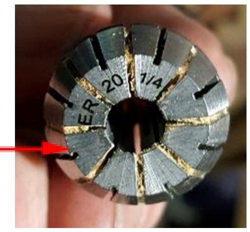
Refer to the Chapter on **Bit Selection, Feeds and Speeds** for information on choosing the correct bit (Up Spiral, Down Spiral, Straight, etc) for the material and the desired results.



Note: Settings in the CAM software such as Cut Direction (Climb vs Conventional) will also influence the results. It’s important to do sample cuts to determine the best setting.

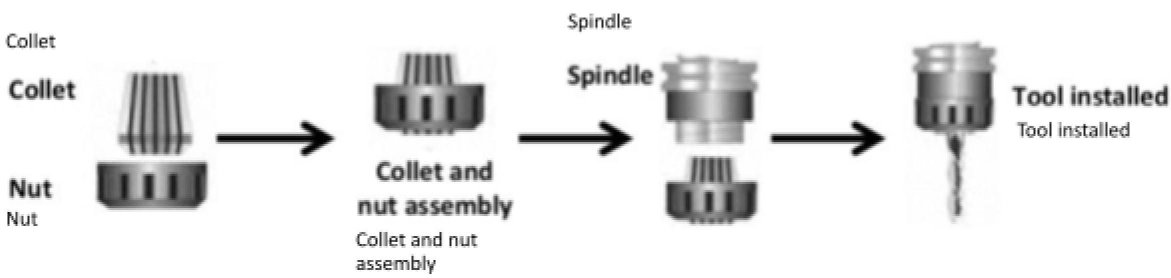
Problem: Bits are not holding position in the Z, or breaking at feeds and speeds that used to work

The Collet holds the bit in place. If the collet is dirty, or an old/damaged collet, the bit will not be held securely in place. Increased bit vibration in a bad collet may cause it to heat up and break before its time.



- Collets need to be clean in order to hold the bit in place. Before inserting a bit in the collet, check the collet and clear it of debris. Vacuum, or use a piece of paper or a brush to clear the slots of the debris generated by machining a part.
- Collets are consumables and need to be replaced after 400 to 700 hours of use, even if nothing traumatic has happened to the collet. Collets lose their spring and will no longer hold a bit properly after repeated use
- If something traumatic has happened (it fell on the cement floor, it dropped a bit while cutting, the bit ran into something major while cutting), replace the collet immediately and throw the old one away. The cost of a new collet will quickly pay for itself by preventing scrapped parts.

Proper assembly and installation of bit in collet, and collet in spindle.
See also **ShopBot Safety and Basic Use** and **Bit Selection, Feeds and Speeds.**

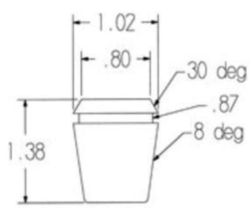


- To form the collet nut assembly, orient the collet and nut as shown at the above left and press them together until the collet clicks into the nut.
- Insert the assembly into the spindle in the orientation shown above. Use two hands to tighten the nut a few turns onto the spindle without compressing the collet.
- Insert the bit (tool) in the assembly, then use two hands to hand tighten the nut until it holds the bit securely. Refer to **ShopBot Safety and Basic Use** to see how much of the bit to insert into the collet.
- Use the provided wrenches to tighten to the bit in the assembly (monkey tight, not gorilla tight).

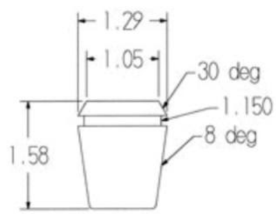
Replace Collets Regularly, and Have a Spare on Hand
(The Collet Nut must also be replaced, but not as often as the collet itself).

Different models of spindles use different **models** of collets (example: ER 25).
The shaft of the bit determines the size of the ER collets (example: ¼”, 3/8”, ½”).

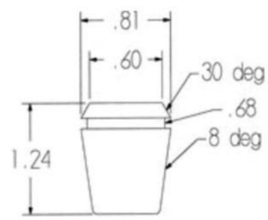
Gantry Tool/Buddy
2.2 or 4HP Spindle
ER 25



ATC
5HP Spindle
ER32



Desktop Tools
1 HP Spindle
ER 20



Tools with router
Porter Cable
Router collet

Porter Cable collets are only sold in one piece and must be replaced in their entirety

Maintenance

Bits, collets, and collet nuts should be examined for wear and cleaned before every use of the ShopBot. Check the collet when changing the bit. Prepare for the next session by checking and cleaning at the end of the day, and ordering replacements if necessary.

Trouble-shooting Rack and Pinion Tools

ShopBot Gantry Tools and ShopBot Buddies are Rack and Pinion Tools.
ShopBot Desktop and Desktop Max have a different drive system (Lead Screw).

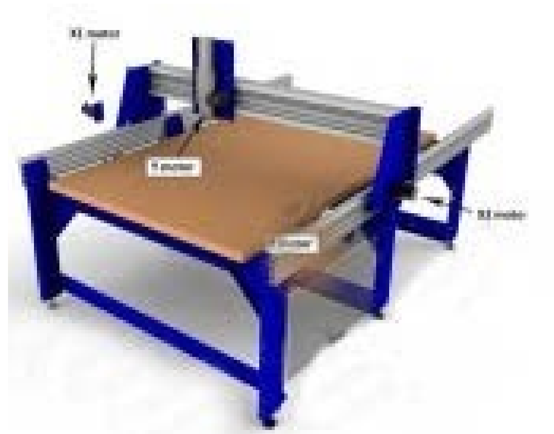
Problem: Circles Aren't Round

Possible causes

- The pinion gears are not engaging in the rack appropriately.
- The set screws that hold the pinion gears to the motor shaft have come loose.
- The pinion gears are worn and need to be replaced.
- The circle in the CAD file is not actually round.

First test: Are the pinion gears engaged in the rack

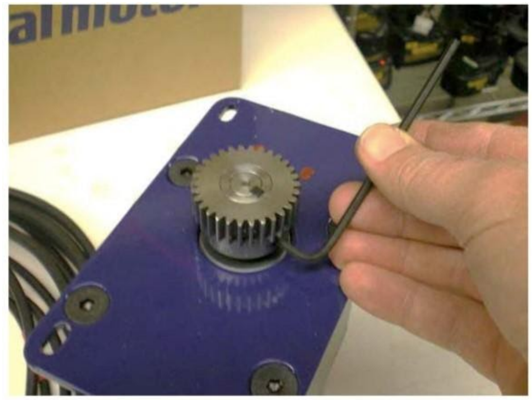
- With the control box on and the drivers reset, stand by each motor and try to move the gantry (X Motors) and YZ car (Y motor) side to side. For the X motors, grab the side of the gantry, not the yellow push bars (if applicable).
- There should be no play or side to side movement close to the motor where the pinion gear behind the motor meets the rack.
- Check the Z motor also, trying to move it up and down.
- For Buddy Tools, try to move the table itself to see if the pinion gear on the motor under the table is tight.



If there is play or side to side motion, take the following steps:

- Loosen the 4 screws holding the motor plate in place and drop the pinion gear out of the rack.
- Re-engage the pinion gear into the rack by lifting the motor in place.
  - Wiggle it a tiny bit side to side to make sure the pinion gear is fully engaged in the rack.
  - Apply significant pressure (15 – 20 lbs) to hold the pinion gears against the rack while tightening the 4 screws on the motor plate.
- Test again to see if the play in the movement of the motor has been reduced/removed.
- Be sure to check both X-axis motors.
- If there is still play in the pinion gears, move to the next step.

Second test: Are the pinion gear set screws tight?
If there is still play in the ShopBot after the pinion gears have been re-engaged, check the set screws that hold the pinion gear to the shaft of the motor.



- Remove the same screws on the motor plate and pull the entire motor/motor plate and pinion gear away from the gantry (X motors) or YZ car (Y motor).
- Be gentle and do not pull the motor assembly out too far to avoid damaging the motor wire connections.
- Wiggle the pinion gear to see if it is tight on the motor shaft. If there is play:
  - Using a 1/8” Allen wrench, tighten the set screws against the shaft of the motor, then test for play again.
  - While they are visible, inspect the pinion gears for wear. Now is a good time to order a new set of pinion gears as they are consumables. Operating while gears are loose will accelerate the wear.
  - Count the number of teeth or measure the diameter of all three pinion gears (or ask ShopBot Tech Support) to determine what size to order.

Maintenance

- Check the play in the pinion gears weekly, and before a major job.
- Pinion gears are consumables and should be replaced every 6 months to 1 year.
- The ShopBot website has a document on how to replace pinion gears: http://www.shopbottools. com/ShopBotDocs/gantry.htm > General Maintenance > Replacing Pinion Gears.
- To extend the life of the pinions, keep the rack well-greased and free from any chips or debris. A tube of grease was shipped with the ShopBot. If it is not available, use any heavy automotive or industrial grease, available from a local auto parts store.
- Do not put the heavy grease on the rails, just the rack where the pinion gears ride. Use light machine oil on the rails, like 3 in 1.
- More information on tool maintenance and checking for play in the pinion gears can be found on the ShopBot website (www.shopbottools.com) under ShopBot Docs.

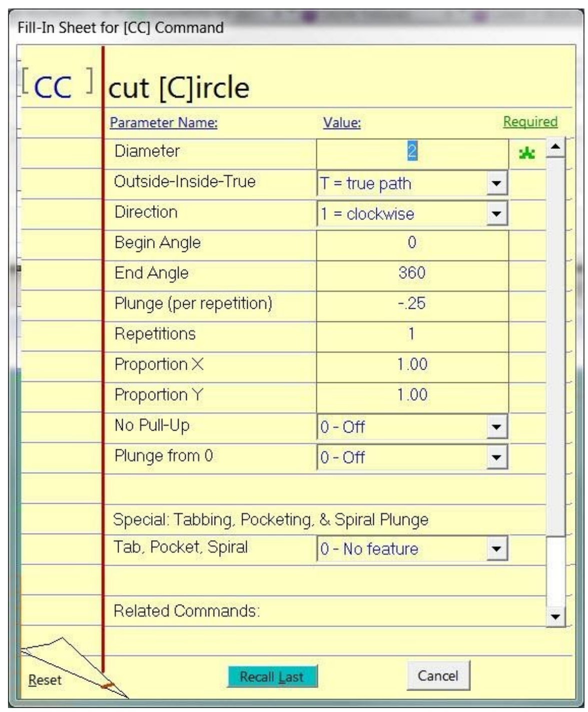
Third test: Check the CAD file

- Occasionally, the circle in a CAD file imported from a different software package is not a true circle.
- Test to see if the problem is with the CAD file or the tuning of the ShopBot by cutting a circle using the **CC** command from the ShopBot Control software.
- The ShopBot Tool Bar Menu is only available if ShopBot is set to the Full version, not ShopBot EASY (see **ShopBot Safety and Basic Use** or **Hidden Gems of ShopBot Control Software** for information on ShopBot Easy vs ShopBot Full).
- The example below will cut a 2” (50mm) circle, .25” (6mm) deep. (The values in parentheses are not an exact conversion from inches to metric, just a close approximated value).
- If the circle cut with the CC command is round, but the circle cut from the CAD file is not, look to the CAD file for the problem.
- See the **Command Reference** found under **HELP** in the ShopBot Control software for more details on the “Cut Circle” and other commands from the ShopBot Control Software.

The **CC** routine runs wherever the bit currently is.
Note: The diameter of the bit is set on the **VC** page.
The **M2** command below moves the bit to a location 6 inches (150 mm) from 0,0 in the X and Y For a full test, use several different locations on the ShopBot table.

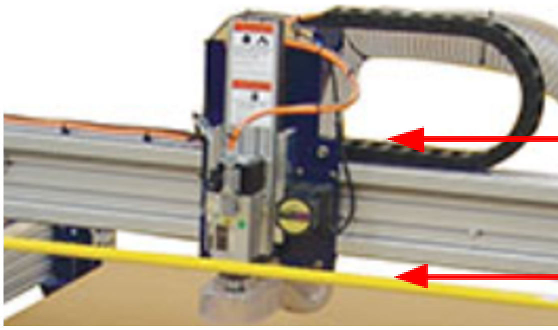
- **M2, 6,6** (Metric: M2, 150,150)
- Enter **CC** in the command line to open the Cut Circle fill-in sheet
  - Diameter (2” or 50mm)
  - T = true path (outside-inside-true)
  - 1 = clockwise (direction)
  - 0 (begin angle in degrees)
  - 360 (end angle in degrees)
  - -.25” or -6mm (plunge)
  - 1 (repetitions)
  - 1 (proportion X)
  - 1 (proportion Y)
  - 0 – Off (no pull-up)
  - 1 - On (plunge from 0)
  - 0 – no feature

ENTER to execute the command
Remember to turn on the spindle at the prompt.



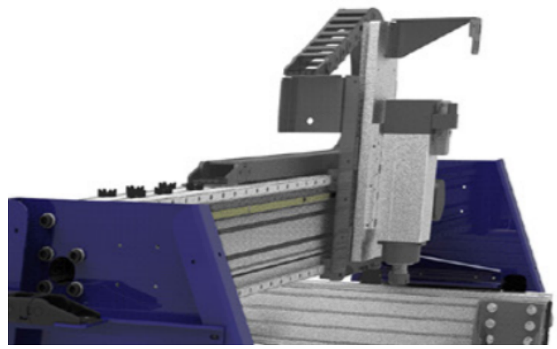
Problem: Square/Rectangular Parts Cut Correctly on 3 sides, but the 4th side is wonky

This section is relevant for PRS Gantry and Buddy ShopBots shipped before mid-2017



PRS Gantry shipped 2007 – mid 2017

- For PRS Gantry and Buddy ShopBot shipped before mid-2017, the YZ car attaches to the gantry by sets of V-Wheels on rails at the top and bottom of the gantry.
- The ShopBotter must test and adjust the V-wheels at the bottom of the gantry to ensure that the YZ car does not pull away from the gantry at the bottom.



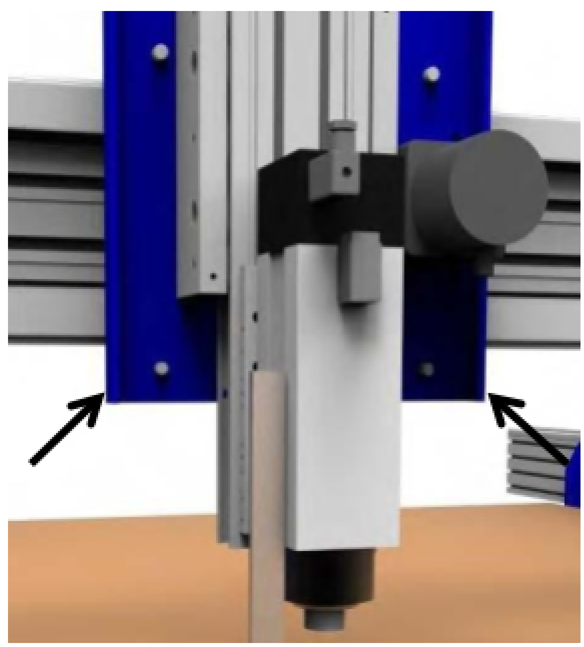
PRS4 Gantry shipped beginning mid 2017

- Beginning mid-2017, the YZ car is shipped already attached to the gantry using a linear bearing system.
- There are no adjustments to the linear bearing system for the ShopBotter to make.
- If cuts seem irregular, check that the spindle is tight on the spindle plate.

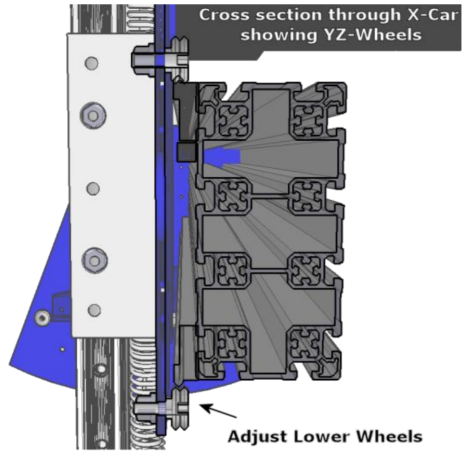
Probable Cause: The bottom V-Wheel rollers on the YZ car are not engaged properly with the rail on the cross beam. This is highly likely if the direction in which the bad cut occurs is consistent.

Test: Grab the ZY car at the bottom near the arrows shown in the illustration to the right

- There should be no motion of the YZ car away from the extruded aluminum cross beam.
- If the YZ car pulls away from the cross beam, the bottom rollers are not engaged properly on the rail.
- The result is that the parts will cut correctly if the force of the cut is driving the YZ car into the cross beam, but will cut incorrectly when the force of the cut is driving the YZ car away from the cross beam.



- Refer to document “**Adjusting the Lower Wheels on the YZ Car**” found on the ShopBot website for instructions on how to adjust the rollers on the rail.
- https://www.shopbottools.com/support/documentation/maintenance
- Hint: It’s a good idea to drop the pinion gear on the Y motor out of the rack before adjusting the lower wheels. This allows you to get a good idea of how easily the car rolls along the rails while still holding the car against the cross beam correctly.
- Don’t forget to re-tighten the pinion gear in the rack after adjusting the V-Wheels.



Maintenance: At least once a month, check that the bottom rollers on the YZ car are seated correctly.

Z Axis Trouble-shooting

Z axis problems may stem from:

- Problems with the bit/collet/collet nut .
- Pinion Gear not tight in rack (Rack and Pinion gear tools).
- Rollers in YZ car need to be adjusted (rare).
- Incorrect Spindle attachment to the mounting bracket.

The Z axis does not seem to be holding position, or the Z axis was zeroed before cutting, but seems to have lost its zero over the course of the file.

Possible cause:

- The bit is not tightened correctly in the collet/nut assembly (see above).
- The collet may not be holding the bit correctly, and needs to be replaced.
- The collet nut itself may be tired or damaged, and needs to be replaced.
- The mechanicals on the Z axis need to be adjusted/replaced.

Test: Check that the collet is holding the bit correctly.

- Run the Z zero routine with the Z -Zero plate, and test that the Z is actually zeroed: **MZ,0**; (detailed in the Software section of this document).
- Make a note of how far bit is pushed into collet.
- Run a file that requires the Z to plunge into a dense material (wood recommended).
- After the file is complete, confirm that bit is pushed into the collet at the same depth as before the file was run.
- If the bit position in the collet looks OK, run another **MZ, 0** to check for subtle loss of Z position.
  - If the bit/Z-Zero position is intact, the collet is probably holding the bit in place adequately for files cutting less aggressively.
  - If the bit has slipped up into the collet (or dropped out of the collet while the file was cutting), first make sure that the bit was tightened into the collet adequately. If it was tight, but the bit moved in the collet, time for a new collet and/or a less aggressive plunge into the material.

Maintenance: Each time, before inserting a new tool (bit) into the collet, check the following:

- Is the collet clean? Check for debris in the slots and clean as necessary.
- Is the collet the correct size for the shaft of the bit? SAE and metric collets/bits, although close in size, are not interchangeable.
- Is the collet firmly snapped into the collet nut?
- Is the bit tight in the collet assembly?
- Has the collet been overtightened and distorted by using too much force on the collet wrenches? Other damage to collet? Or damage to Collet nut?
- Are pass depths, plunge rates, and tool feeds and speeds appropriate for the material?

Other issues with Z height Movement (Up and Down)

Slop in the Z axis:

- Test that the pinion on the Z motor is tight by grasping the entire YZ car and trying to move it up and down.
- If it is loose, follow the directions for tightening the pinion gear in the rack and on the motor spindle as above.
- The rack on the Z axis run vertical, so the motor slides sideways when the hardware on the motor plate is loosened.

Adjusting the Z car bearings (Rare)

Signs bearings are too loose:

- Top and/or lower bearings spin freely and are not in. contact with inner part of the captured rail.
- There are lateral inconsistencies in Z axis, commonly called “slop”.

If bearings are too tight:

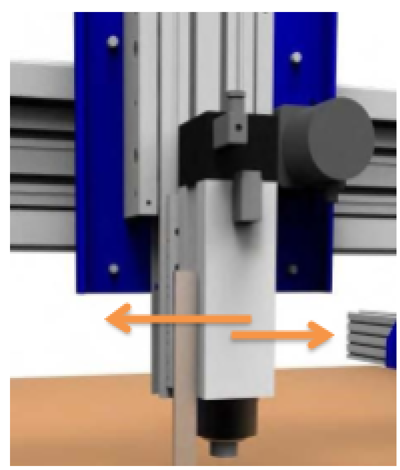
- Z axis loses steps/position, binds or even stops abruptly.
- There is premature wear to the inner part of the captured rail.

Refer to the document on the website for information on how to adjust the bearings on the Z axis.

https://www.shopbottools.com/ShopBotDocs/files/AdjustPRSZBearings.pdf

Problem: Cuts are Inconsistent on Step Downs (side to side)

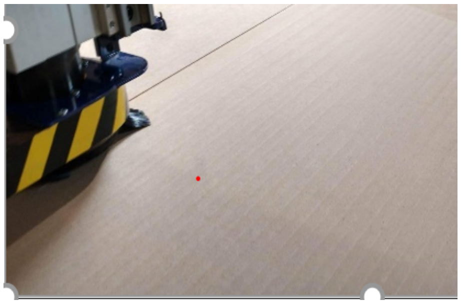
Possible Cause: The spindle is not firmly tightened onto to spindle plate
Test: grab the spindle itself and try move it on the spindle plate.
If the spindle is loose:



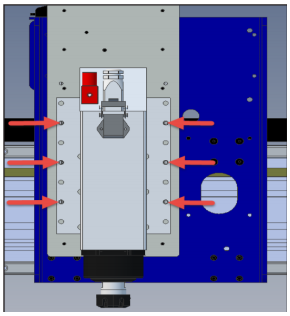
- Be sure that the spindle is square to the table (see below), then tighten the hardware holding the spindle to the spindle plate.
- Refer to the Assembly Manual for the model of ShopBot to see how the hardware fits into the spindle mounting bracket.
    - For PRS tools prior to mid-2017, the hardware is accessed from the back of the YZ car.
    - For ShopBots with the PRS4 Gantry, the hardware is accessed from the front of the YZ car.
    - The spindle for Desktop ShopBots is attached at the factory. Contact ShopBot for more info if needed.

Problems: Ridges on Table are Felt as Well as Seen

Probable Cause: Spindle is not straight on the YZ car
Test: Compare the difference between SEEING and FEELING ridges on the table when running the Table Surfacing Routine.



- The width of the 1.25” surfacing bit amplifies any problems with the Z not being perpendicular to the table
- In the photo to the right, lines are visible due to the way the light reflects on the fibers pulled up by machining. But the table feels perfectly smooth when you run your hand across it
- The photo below shows ridges on the table that can be seen and felt. One side of the bit is plunging deeper into the board, indicating that the spindle is not perpendicular to the table



- Loosen the hardware holding the spindle to the Z
  - Use a square on the spindle sides to adjust the spindle until it is perpendicular to the machined sacrificial board
  - The sides of the spindle should be parallel with the Z axis, and are the most accurate measuring points for squaring with the tool deck
  - Once square, tighten screws to hold spindle in proper alignment

ShopBot is Cutting Parallelograms Instead of Rectangles

On most ShopBot models, the axis that runs along the rails has 2 motors/drivers, while the axis that runs across the Gantry has a single motor/driver. The definition of the axis that runs along the rails depends upon the model of ShopBot.

Gantry Tools	Buddy Tools	Desktop Max	Desktop
2 X motors 1 Y motor	1 motor each on X & Y	2 X motors 1 Y motor	Beginning mid-2017: 2 Y motors, 1 X Pre mid-2017: 1 motor each on X & Y

The two motors/drivers on the same axis run on the same Channel, so the motors are perfectly matched to move simultaneously (although one runs backwards to the other so the gantry doesn't rack.) When the ShopBot Control Box is powered up and activated, the motors will stay in the same position relative to each other. If they are square, they will stay square. If they are out of alignment with each other, they have to be squared up.

There are several reasons that the motors have gotten out of alignment with each other: one motor hit something and stalled out, one of the motors got bumped when the ShopBot was powered down, the motors got out of sync when the pinion gears were re-seated in the rack.

The following section is related to squaring up the X motor mechanically so that they will cut rectangles and squares rather than parallelograms.

Options to evaluate whether the tool is "square"

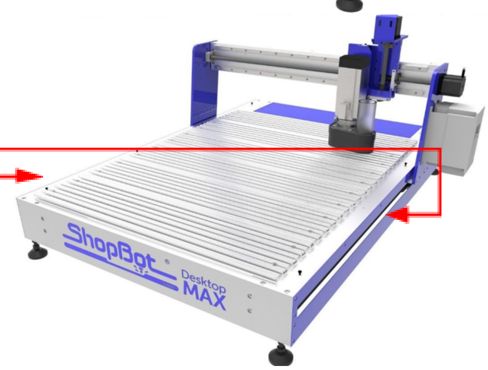
- Cut out a large rectangle, the bigger the better, and measure the diagonals. They should be equal
- Short of cutting something out, use a V Bit to inscribe a shallow, large square or rectangle as large as possible on the table surface
  - This technique can also test if the table surface is flat in relation to the mechanicals. If the V Bit plunges deeper in one section of the table compared to another, it might be time to check all the hardware around the table. Include hardware on cross supports under support board
  - For rack and pinion gear tools, run the gantry up and back on the X rails to be sure none of the the wheels on the rails are floating (not touching the rail.) Adjust feet to level the sides of the table
  - If everything seems tight, surface the table to flatten it in all areas in relation to the mechanicals of the tool (see **Projects and Techniques**)
- Turn off the power to the spindle, chuck up a pencil, and draw a rectangle
- Drill 4 holes at the corners of a square and measure the diagonals
- In all cases, if the measurement of the diagonals are equal, the tool is cutting square
- If the tool is not cutting square (the diagonals are not equal), the problem can be resolved as easily as pulling the gantry into square using the end stops, or as involved as making adjustments to the table and mechanicals. **HELP** in SB3 ShopBot software has suggestions.

Squaring Up Desktop ShopBot Models

The **Desktop Series Quick Start Guide** included with the tool, or available from the ShopBot website https://www.shopbottools.com/ShopBotDocs/files/DTQuickstartGuide_ShipAug2017.pdf has more information about squaring up the axis with two motors when getting set up, or after an event which may have gotten the motors out of alignment with each other.

- For the Desktop (24” x 18”) model manufactured after mid-2017, the Y axis has two motors (Desktops manufactured pre-mid 2017 have a single lead screw on the Y axis)

- The Desktop Max (24” x 36”) model has two motors on the X axis



Let us assume that the Desktop model was set up correctly as per the Quick Start documentation. But something happened during the regular course of activity so that the motors got out of sync with each other (Desktop moved, one motor bumped with power off, one motor stalled after hitting an object, etc.) If the motors are too far out of alignment, the Desktop/Max might stall in the axis with two motors.

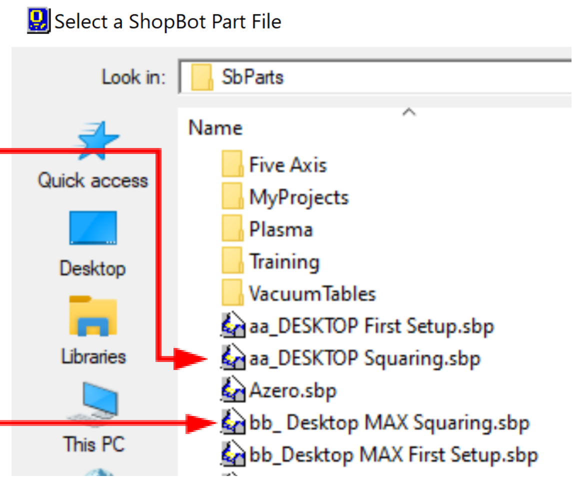
The squaring procedure runs a routine that crashes the motors against the hard stops. This will produce a loud “grinding” sound as the motors are stalling out, but it does not harm the motors in any way.

After running the squaring routine, the Automatic XY Zero Routine must be run to re-establish the proper origin.

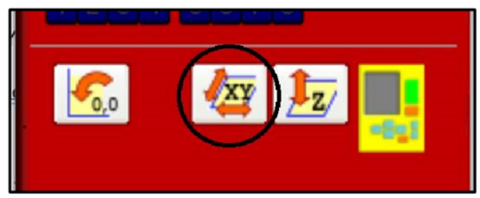
- In the SB3 Control Software, enter **FP** or click the “Cut Part” button

- Navigate to the SBParts Folder in the C: drive (Hint: Try searching for **This PC**, then scroll down to find OS (C)

- For the Desktop, run aa_DESKTOP Squaring.sbp
- For the Desktop MAX, run bb_Desktop MAX Squaring.sbp



- The grinding noise of a motor stalling signals that the ShopBot has lost its Zero location. After running the Squaring routine, run the Automatic XY Zero Routine to re-establish the 0,0 location

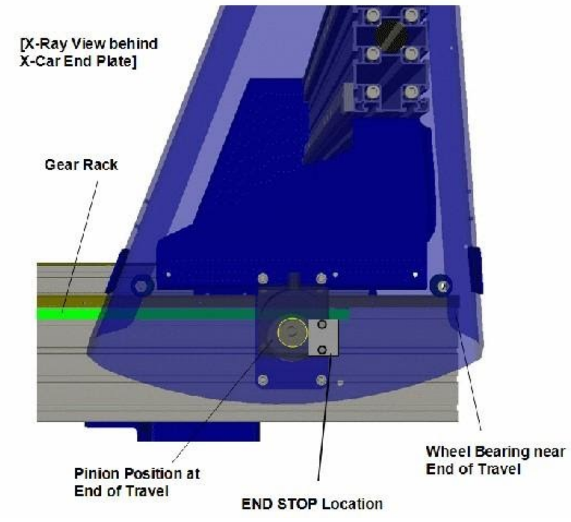


Squaring Up Rack and Pinion Models

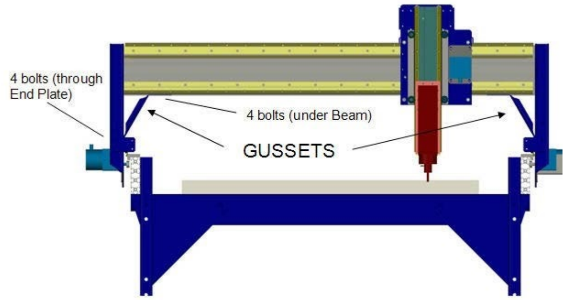
Assuming that the ShopBot was assembled correctly, the table itself should be square, and the sacrificial board squared up to the table.

If the motors became misaligned, (example: someone bumped one side of the gantry when the Control Box was not powered up, or after the pinion gears were tightened by dropping and reseating them in the rack) it is easy to realign the gantry on the rail.

- If the end stops were properly set up during tool installation, then they will be squared to each other.
  - With the Control Box powered down, pull the gantry up against the end stops.
  - Then power up the Control Box and, for alpha tools, hit the Reset Button on the pendant.
  - The motors will then hold the gantry in place as long as they are engaged and powered.



- Occasionally, there are more significant problems with the ShopBot gantry tool not being square and in alignment.
- Review the Assembly Manual for information on squaring the ShopBot during set up
- Look to **HELP** in the SB3 Control Software for the Document "Squaring the X Car" for details on how to make adjustments to the mechanicals.
- ShopBot Technical Support is always available to answer questions.

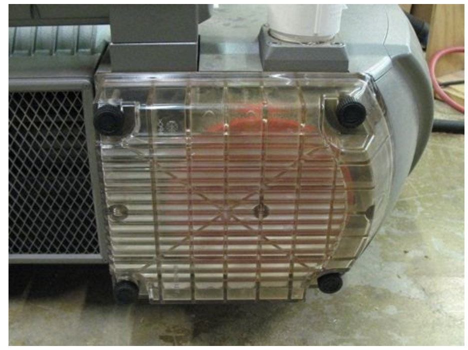


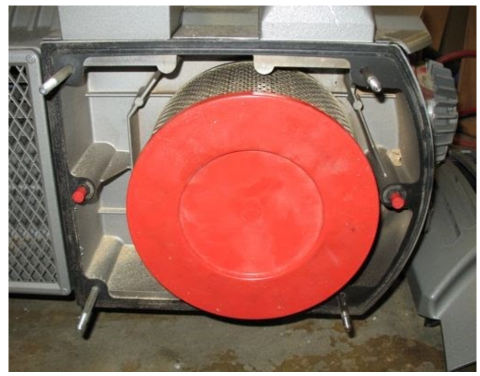
Maintenance For Becker Blower

(VTLF 2.250 Oil-less Rotary Vane Vacuum Pump)

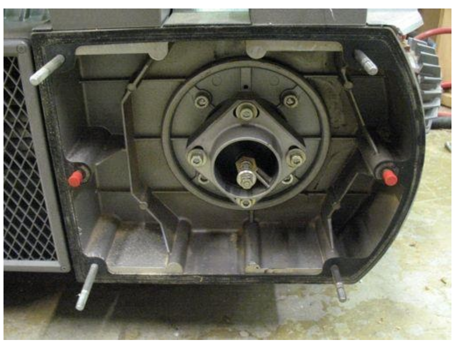
For Support, Call Becker USA Division (Ohio): 330 – 928-9966

Every 40 – 200 Operating Hours:



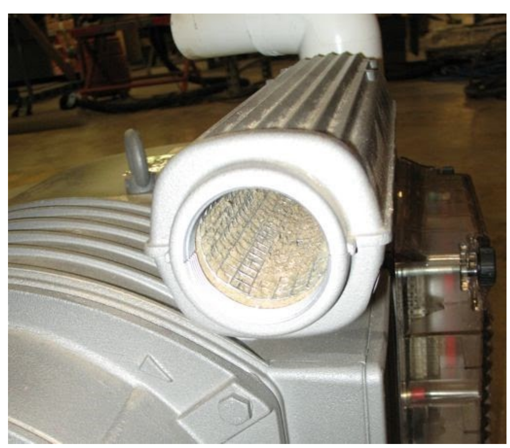


- Clean filter Cartridge and Housing
    - Unscrew black handles to remove plastic cover
    - Remove filter cartridge and clean it thoroughly with an air gun
    - Check if Cartridge needs to be replaced
        - Hold cartridge up to standard 100 watt light bulb
        - Look from the **inside** of the cartridge and see if the light comes through the filter from the **outside**.
        - If light is not visible, cartridge needs to be replaced



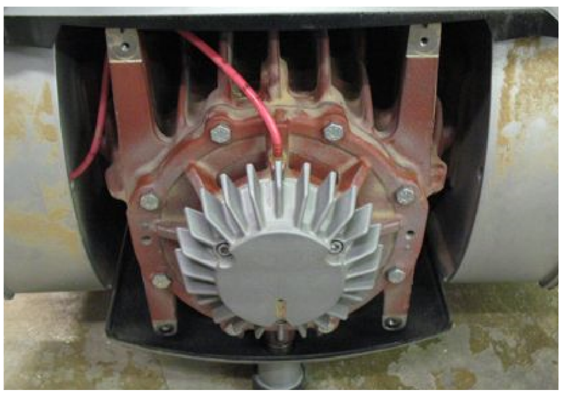
- Vacuum/clean inside the filter housing

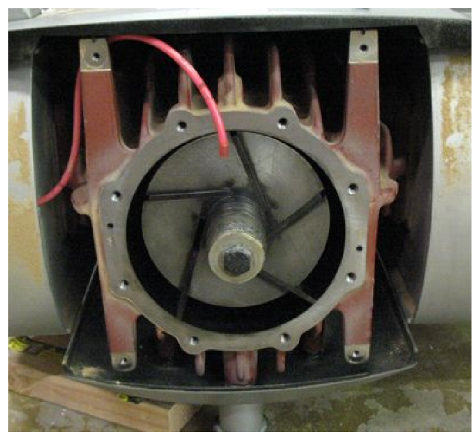
- **Check Silencer Pipes**:
    - Shine a flashlight in each to check for obstructions. Clear out excessive debris



Every 3000 Operating Hours (and Troubleshooting)

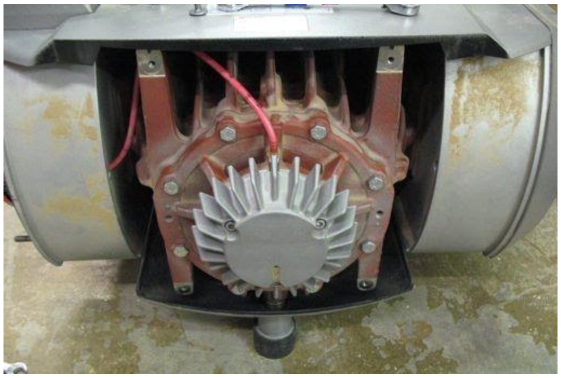
- Check Rotary Vanes
  - Remove cover hood with 5mm allen wrench.
  - Remove end-cap (13mm ~ 5” bolts).
    - Undo grease line at top of cap.
    - Do NOT remove bearing cover (attached to end cap with Allen-key bolts -gray and has fins).
    - If cap does not easily pull off, screw two of the 13 mm bolts into the slots on the left and right side of the end-cap. This will push the cap away from the pump housing.





- **Carefully** slide out a carbon vane (they are brittle).
  - Measure its width. Replace the vanes If they are less than or close to 42 mm (varies depending on model-see sticker on top of pump).
- Check outer edges of each vane. They should be smooth and free of nicks.
- Check for debris or oil in shaft. If oil is present, vanes will not function.

- **Clean Bearings**
  - Wipe grease off motor shaft.
  - Wipe grease out of the socket/bearings in the end cap. Be sure that there is no grease on the flat surface of the end cap.
  - Replace end cap, and grease lines.
  - Tighten bolts.



- **Clean End Housing**
  - With End Cap attached, but plastic cover still off, clean in and around end of pump with an air gun

- **Re-Grease**
  - Only re-grease when the **motor is running** with no resistance (vacuum not hooked up to anything).
  - Two grease nipples are located on either side of the filter cartridge.
  - Use Amblygon TA 15/2 grease, 10g per nipple.