Running your File on the ShopBot CNC Router
A Basic Step-By-Step Guide

This guide assumes your VCarve file is ready and that you are moving on to the ShopBot to set up your job. If you have not completed your VCarve File, begin with the ‘Creating your File in VCarve for the ShopBot CNC Router’ guide.

ShopBot CNC Router Reservation Procedure

*You must have taken the CNC Routing Shop Class with Maker Playground to be certified to reserve time on the ShopBot.

Please have your VCarve file ready prior to your reservation on the CNC Router in order to keep the ShopBot desktop dedicated to running files. This can be prepared on MPG's laptops or on your personal computer. Enter the Makerspace Client Code below in the trial version of VCarve to allow your files to be opened in VCarve on the ShopBot Desktop. Many helpful tutorials for VCarve can be found online. If you need staff assistance, complete the file to the best of your ability, and staff will be able to assist with finalizing the file for the ShopBot.

Makerspace Client Code: 251BE-8E15B-4705B-C38B2-67D8F-6E7C8-3AA39

Check your feed and speeds for the given bit and material with Maker Playground staff before exporting your toolpaths for ShopBot. If you would like to bring your own CNC bits, make sure they have either a 1/4" or 1/2" shank for our collets.

VCarve will provide a rough time estimate for your file. Be sure that the time estimate fits within your reserved time slot. Most projects take less than an hour to complete.

Maximum Job Size: 96x48x8"

Software: VCarve Pro

Compatible File Types:

vector: .dxf, .dwg, .eps, .ai, .pdf, .pvc, v3d, .v3m, .crv, .skp, .svg

image: .bmp, .jpg, .gif, .tif, .tiff, .png, .jpeg

3D model: .stl, .v3m, .3dm, .skp, .rlf, .3ds, .asc, .prj, .x, .dxf, .lwo, .wrl, .obj

Approved Materials:

- Solid wood
- Plywood
- MDF
- Dense foam
- Plastic
- Aluminum (prior clearance by MPG staff required)

Available bits:

- 1/4" Endmill
- 1/2" Endmill
- 1/8" Endmill
- 1/4" O-Flute (Acrylic)
- 60° Engraving V-Groove
- 1/4" Diameter Up Spiral Ball Nose (3D Milling)

Staff will typically have the above bits available for your use, and we may have additional bits for specific projects. Ask staff for a complete, up-to-date list of bits. If you are looking for a more specific router bit, you may need to bring in your own. If you are surfacing slabs or river tables, you will need to provide your own surfacing bit.

Some Basic ShopBot Rules:

1. **Practice Woodshop Safety**: Always wear eye and ear protection while operating the ShopBot. Keep clothes tucked in and long hair tied back. Wear close-toed shoes at all times.
2. **Be aware of the location of the emergency stop buttons and fire extinguisher.** There are two EStop buttons on the ShopBot - one is on the pendant remote on the desk, and the other is located on the front of the ShopBot on the gantry. We also have a fire extinguisher kept near the ShopBot. Take this moment to locate the stop buttons and fire extinguisher.
3. **Do not place your hands on the rails or lean on the ShopBot.** This could cause injury to you or cause the ShopBot to lose its location while operating.
4. **Learn and understand safe use of the ShopBot.** Receive training before usage. Read this guide all the way through as a refresher. More safety rules will be found throughout this guide.
5. **Ask staff if you are unsure!** This tool requires many steps that can be difficult to remember and is sometimes a bit finicky. Follow along with this guide and ask staff for any clarifications. You'll likely need help to use the ShopBot successfully a few times before being able to run a file completely on your own.

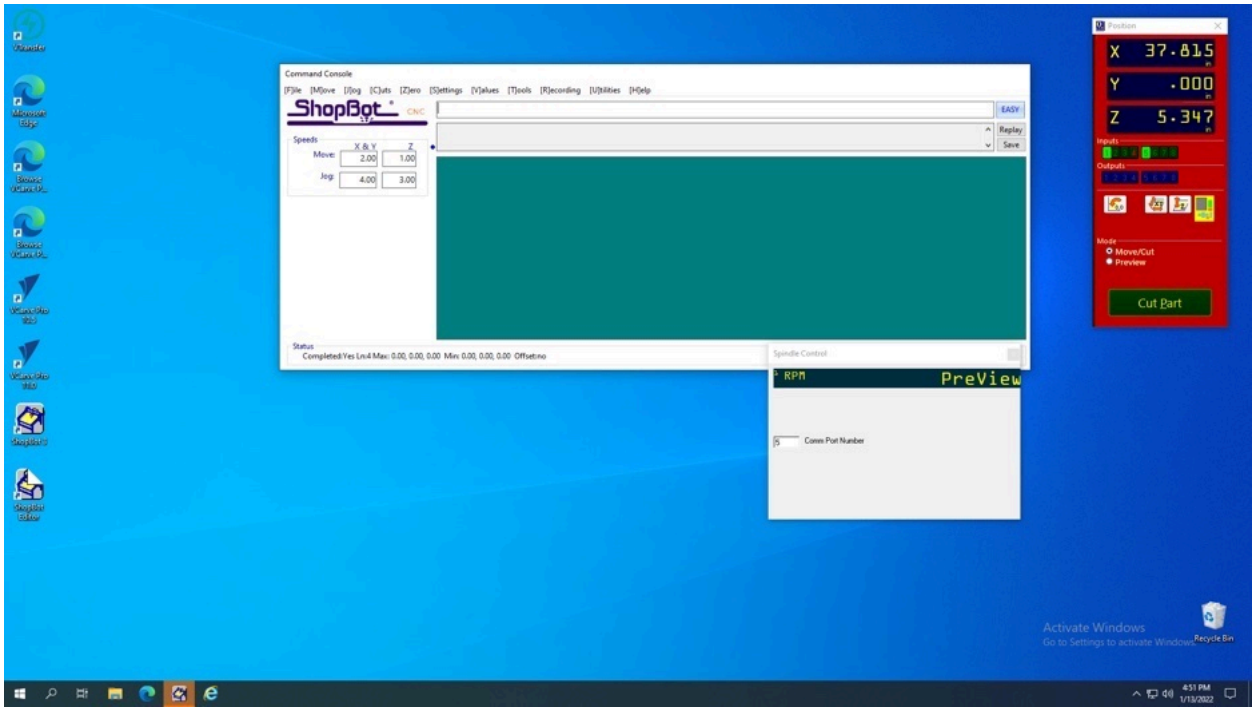
Note: This guide will show all you need to know for basic operation of the CNC. Some of these things can be completed in a different order, but until you understand the use of the ShopBot, we recommend following along with this guide. ShopBot software also has many more advanced capabilities, many of which can be found in the ShopBot CNC Handbook with additional CNC information. Staff can also assist with this information.

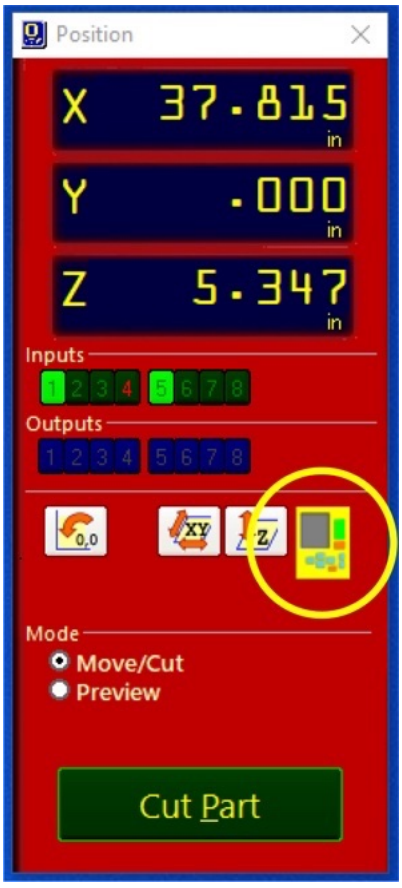
1. **Turn on the ShopBot** using the red button on the control box at the back of the CNC.

2. Next, **press the blue reset button** on the pendant remote on the desk. This needs to be done every time the CNC is started so it can find its exact location after being turned off.

3. Now we can **open 'ShopBot 3' software**. This software will allow us to move the machine,set our origins, and load our file.

This will launch the software in move/cut mode. A few windows should pop up:



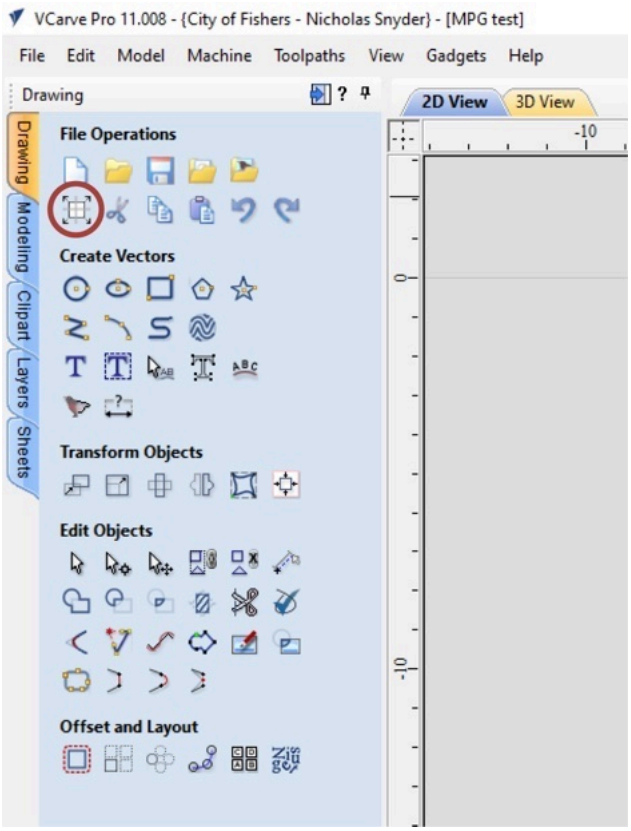


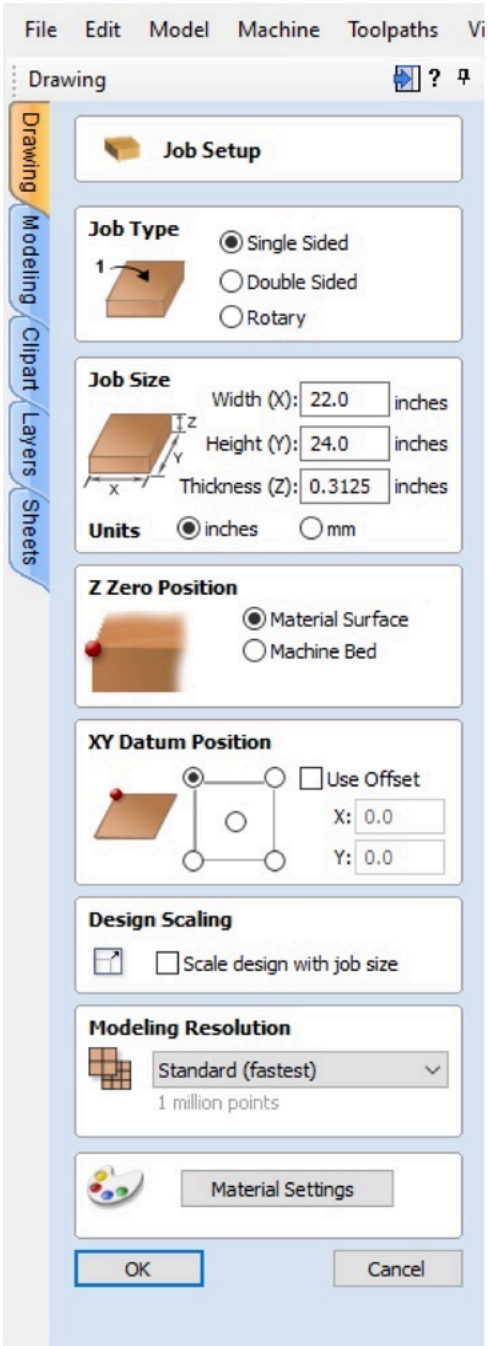
4. In order to move the gantry, **click the yellow keypad button** and a keypad window will open.

You can move the gantry by clicking the buttons on the screen, or by using the arrow keys on the keyboard and 'PgUp' & 'PgDn' for the Z-axis.

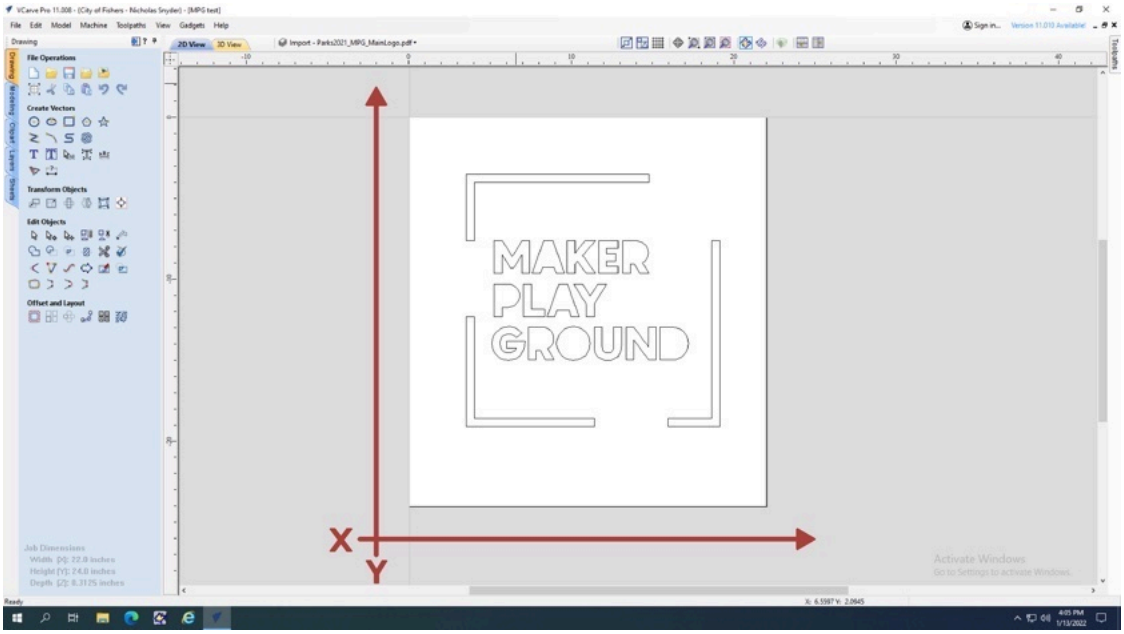
You can also type in coordinates on the yellow keypad and click 'Go To,' but be careful to not crash the spindle into anything on the bed or the bed itself.

5. **Open your VCarve file.** Having the VCarve file open will allow you to reference it while setting up the ShopBot, and save your final toolpath immediately before running so you know what to expect. This will also allow you to easily make any changes. Press the 'Set Job Dimensions and Origin' button under 'File Operations' to display some important set-up settings.





6. **Place your material on the bed and check its orientation.** The below images show the orientation of the file in relation to the orientation of the bed. Think landscape format facing the computer: X will be your width - longways across the bed, and Y will be your height, shortways across the bed. If your material is not oriented correctly, the tool may move to a different area of the bed than you expect and ruin your material or hit a screw or clamp, breaking the bit.

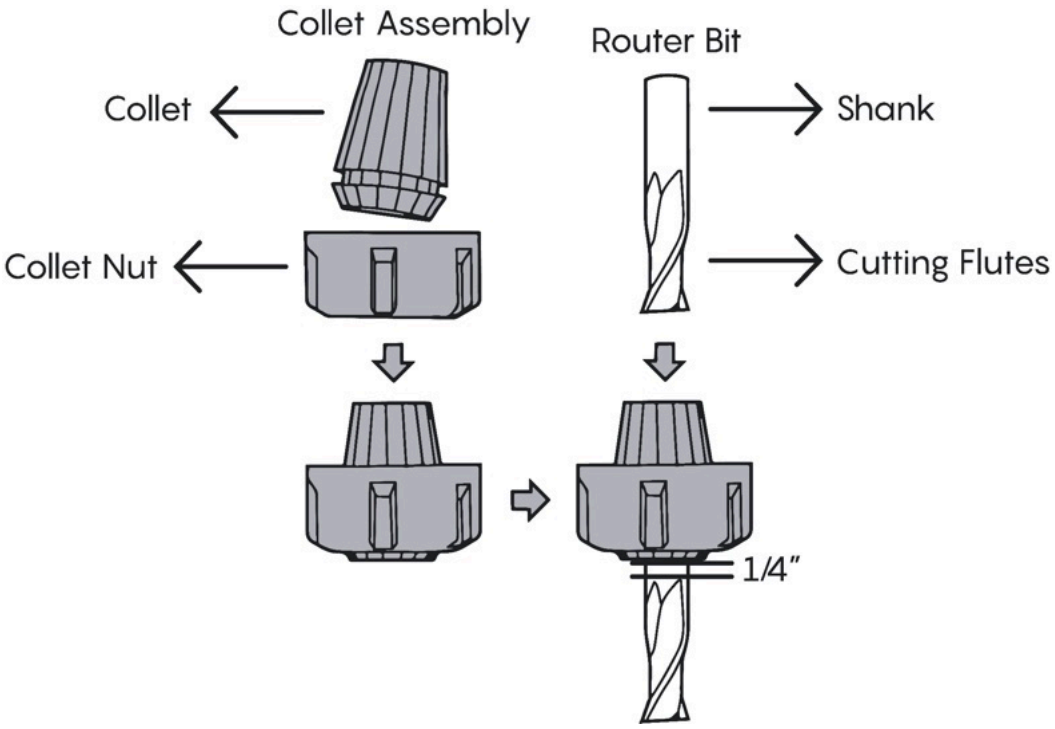


7. **Secure your material to the spoilerboard.** This can be done in a variety of ways and will differ with your project. Screws are common if you have excess material that will be cut off of the piece later. (Just make sure no screws are in the way of the toolpath!) Using carpet tape can also be appropriate for some projects, but we recommend you use it in combination with another technique, such as using scrap wood blocks screwed into the spoilerboard around your material as "clamps." As stated in A Basic Step-By-Step Guide for VCarve, when cutting out any shapes completely, it's recommended to use tabs on your VCarve file to make sure any shapes cut out don't separate from the bed. Not securing your material properly can send pieces flying or trap the bit inside of material and break it. For more hold down techniques, look in the ShopBot CNC Handbook or **ask staff if you need assistance!** NOTE: Our ShopBot does not have a vacuum table, so vacuum hold-down techniques are not a viable option.

8. Change the bit to the one chosen in your file. You may need to remove the dust boot to access the bit, collet, and cover nut. Remove the previous bit by loosening the cover nut with the wrenches on the control box.

   You'll notice one of the wrenches is connected to a key. This key engages and disengages the spindle, so the router will never turn on while you are changing the bit. Make sure that the key is reengaged before loading your file.

   If your bit requires a different sized collet, you'll need to find the appropriate collet size and install it in the cover nut. You'll hear a "click" when the collet is secure.

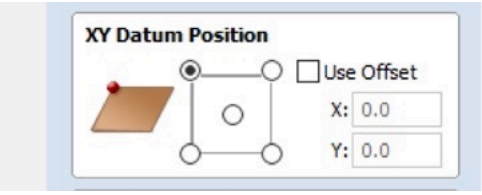


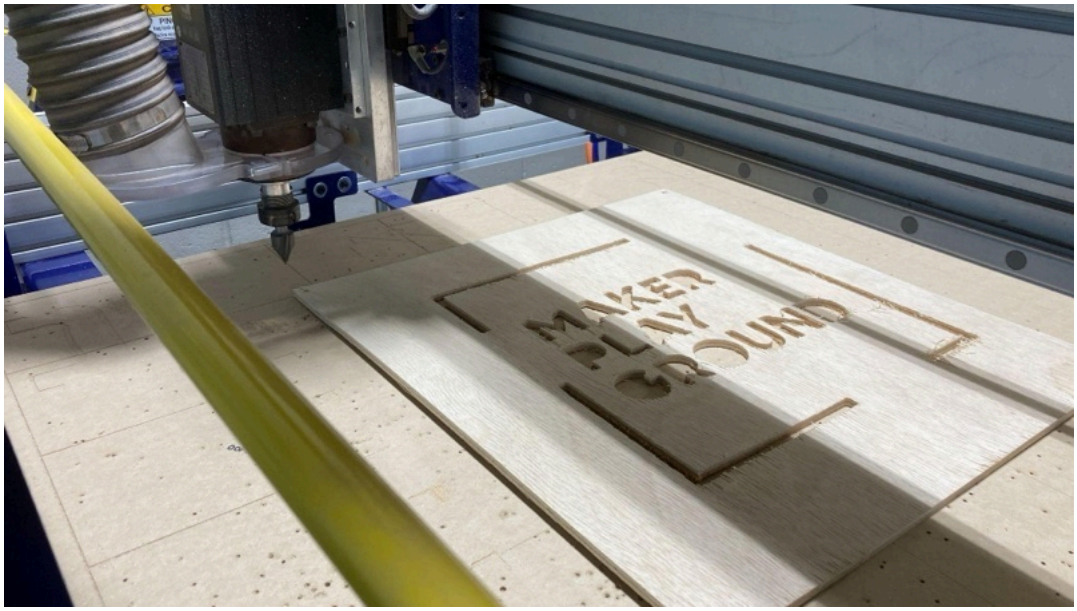
Loosely screw the cover nut onto the spindle, and then insert the bit for your file. **Insert the bit so that just ¼" of shank is exposed above cutting flutes.** The collet should not be touching any of the cutting flutes. Firmly secure the cover nut, but **do not overtighten it**. Extreme overtightening can cause tool failure. Think "monkey tight, not gorilla tight."

Take this time to check that your bit is long enough to cut your project at its deepest point by measuring with a tape measure and comparing to the cut depth in your file.

Return the wrenches to the magnet on the control box and reengage the spindle key.

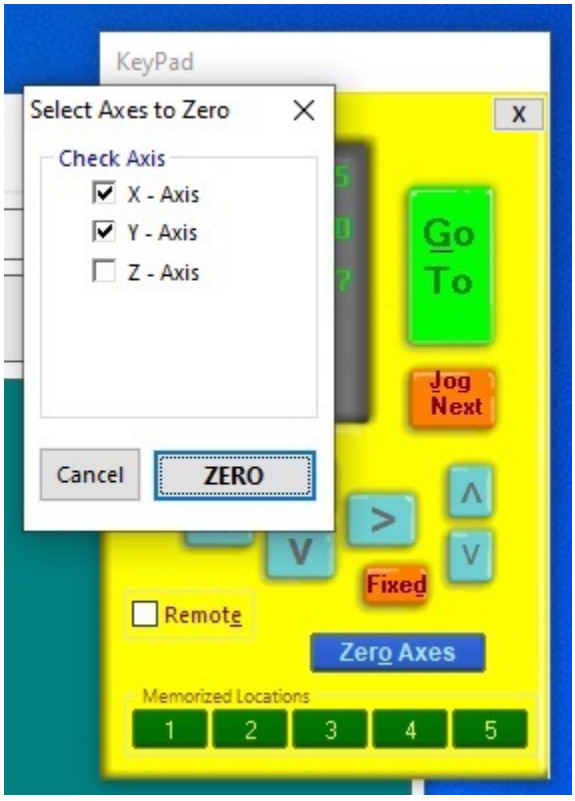
9. Next, we're ready to **set the X and Y axis home according to your file.** Refer to 'XY Datum Position' in your job setup. In this instance, our origin would be in the upper left corner. Move the spindle to the location of your origin, again keeping in mind the bed orientation. The center of the bit should point to your origin.





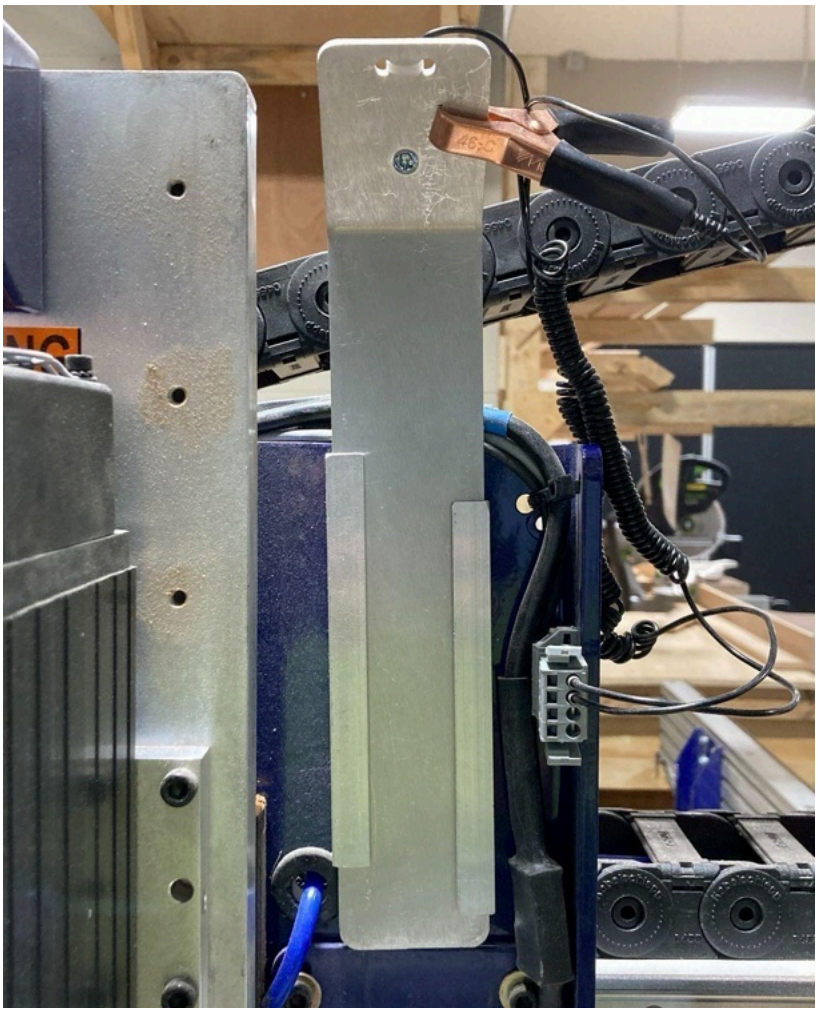
Click 'zero axes.'

Check just the X and Y axes and then click 'ZERO.' We'll be zeroing the Z axis separately in the next step.

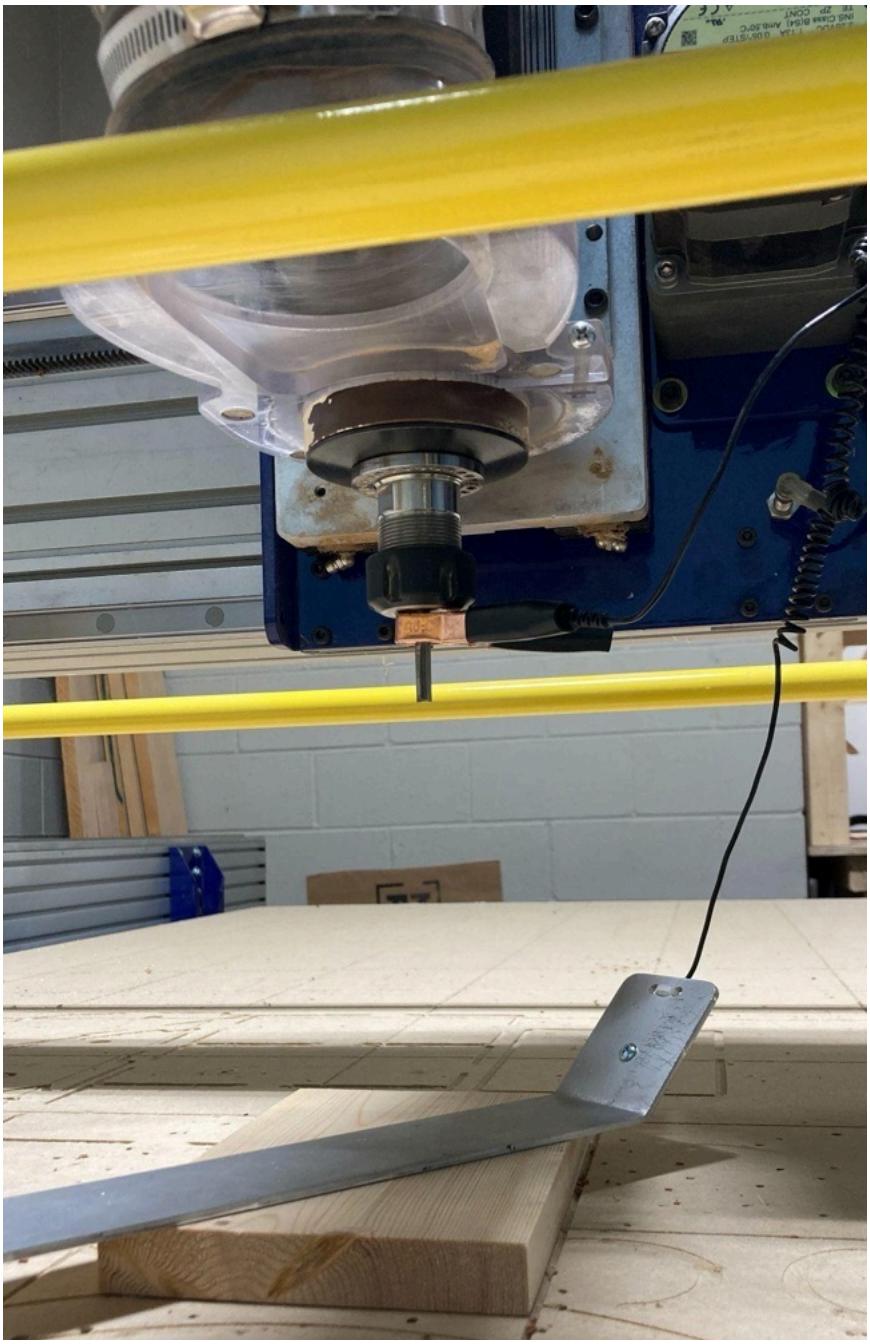


10. **Zero the Z axis using the Z zeroing plate.** In Job Setup, if you set your Z Zero Position to 'Material Surface,' you will want to zero on top of your material towards the centermost point. If you selected 'Machine Bed,' you'll zero on the machine bed next to your material. Move the spindle of the CNC over the location to zero, then close the yellow keypad.

Above the Spindle, you'll see an aluminum plate and a copper gator clip.



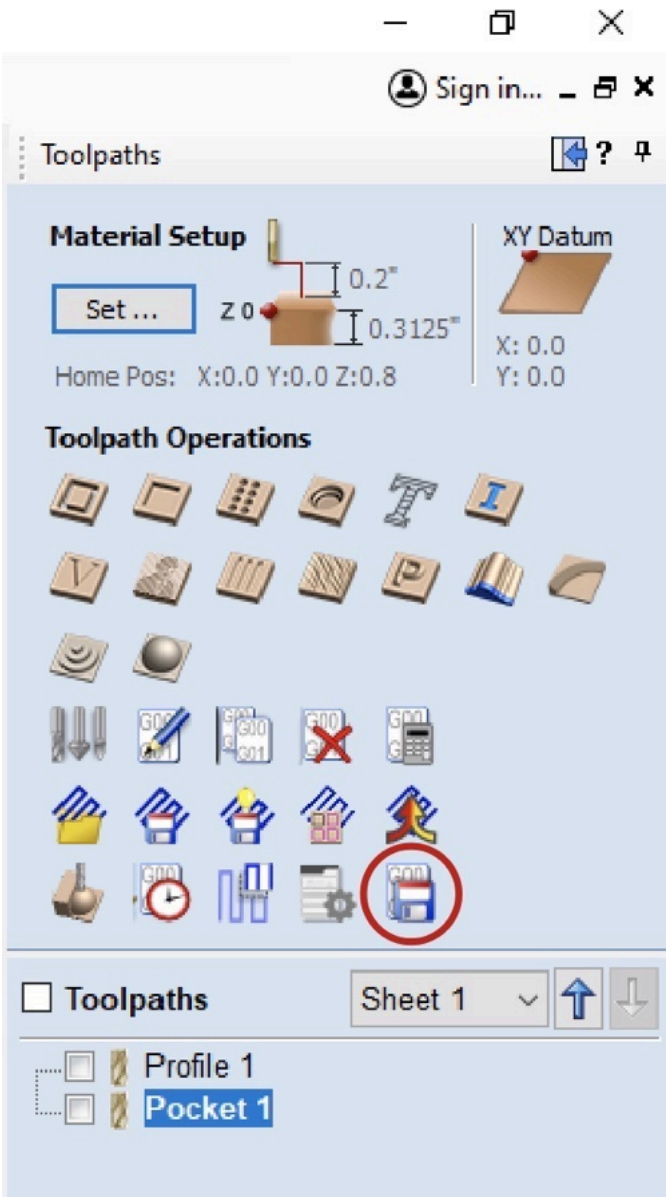
Attach the gator clip to the bit and set the aluminum plate directly underneath.



Perform a "tap test" by tapping the plate to the bit and checking if the #1 input on the red window lights up as you tap. If it does, you're good to go. If it doesn't, readjust the gator clip and repeat until it does. Place the aluminum plate back down beneath the router bit.

Click the button with the Z and up and down arrows on it. The spindle will move down to touch the Z plate twice, and then ShopBot will be zeroed. **Put the gator clip and Z-plate back above the spindle.** Reattach the dust boot if it still allows the bit to reach the deepest parts of your file. If the dust boot inhibits the length of the bit too much to cut to the set cut depth, leave it off and proceed without.

11. **Save your Toolpath(s).** If you haven't already, or have made any changes in your VCarve file, save your toolpaths by using the 'Save Toolpath' button, using the .spb post-processor. Toolpaths with the same bit can be saved in the same .spb file to run consecutively in the same file. If one or multiple toolpaths use separate bits, you'll need to save them in separate files. Name your files something easy for you to identify them.



Tip: I prefer to name my files with the title, the number in the order in which they need to be run, and then the type of bit. For example: "cool sign 1 .25 vbit" then "cool sign 2 .25 endmill" and so on.

12. Check the following:

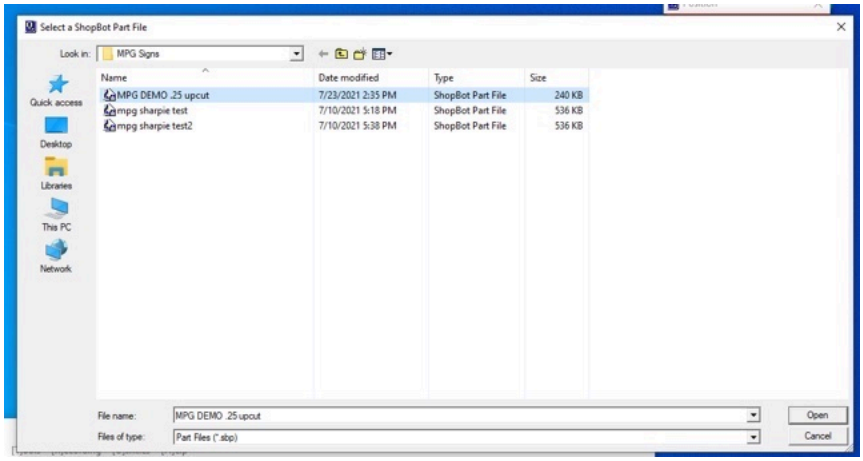
Is the material in the correct orientation?
Is the material secured to the bed?
Are any screws out of the way of the path of the bit?
Is the right bit for your file in the collet?
Is your X&Y origin set where it is located in your file?
Did you run the Z-plate zeroing process as it is set in your file?
Does the bit have enough length to cut to the depth set in the file?
Is the spindle key engaged?
Is dust collection on?
Is the dust boot on? (or should it be off to allow your bit enough cutting depth?)
Is the CNC bed clear of tools or other objects?

Once confirming everything on the list, you may continue to the next step.

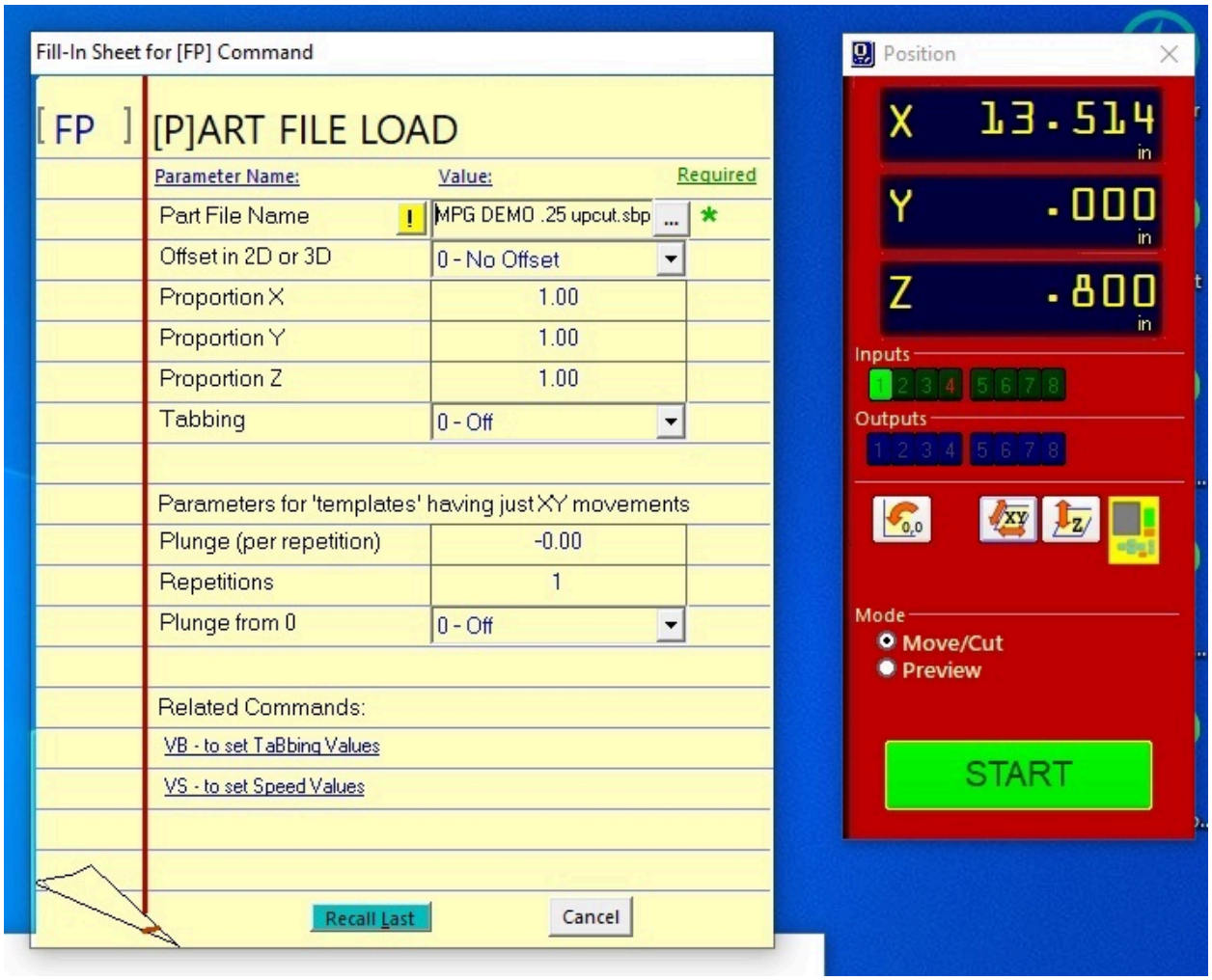
13. Load your file.

*Please read the next steps carefully all the way through before proceeding.

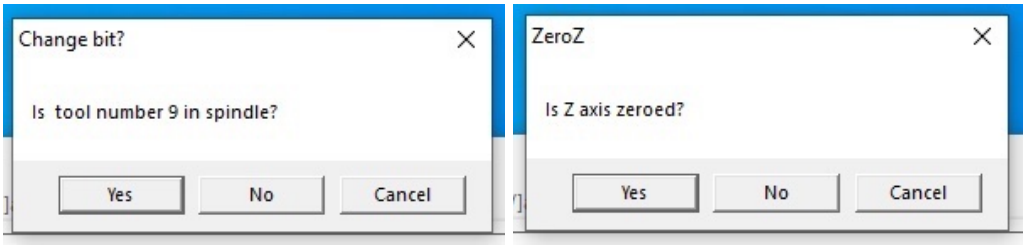
Click the 'Cut Part' button on the red window. If you do not see this button, you need to close the yellow keypad. Then, find your .spb file in the window that appears.



Once you load your file, a window will pop up with settings. You do not need to adjust any of these settings. Simply click the green 'Start' button on the red window. Your file will not start yet.

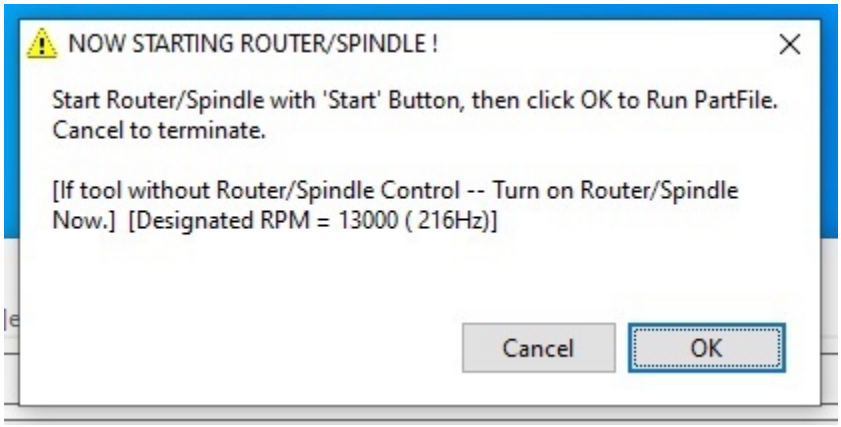


Two windows may pop up with the questions "Is tool #_ in the spindle?" and "Is Z axis zeroed?" Verify the correct bit is in the spindle and has been zeroed and click 'Yes' on both windows. If the window that pops up looks different than these, proceed to the next step.



Read directions carefully in the next step before proceeding!

14. Start the spindle.



Once you reach this window, you will need to start the spindle on the router. Do this by pressing the green button on the remote.

Do NOT click ‘OK’ without starting the spindle. The CNC will attempt to cut your file without the bit spinning. This is likely to break your bit and potentially ruin your material. If you accidentally click ‘OK’ without starting the spindle, quickly press the red E-stop button on the remote to turn off power to the CNC.

Once you have pressed the green button and the spindle has reached full speed, press the 'OK' button to start your file.

15. Watch your file run!

Stay by the CNC while it is running to watch your file. If your project is near the edge of the ShopBot facing the woodshop, watch for the dust collection hose getting pinched by the spindle and move it out of the way as needed.

If the CNC seems to follow a path you do not expect and it is in path to run into something, press the red button on the pendant. This button stops the machine immediately. However, you will lose your origin point and location information. It is best to use this in emergencies to stop the machine quickly or if you are unable to salvage the project and do not need to consider restarting the file.

To pause or "slowly stop" the CNC, press the spacebar on the keyboard. This will allow you to resume your file afterwards. Use this if you need to leave the ShopBot unattended or to investigate an issue that is not an emergency, such as adjusting feeds and speeds.

Sometimes CNC bits may break. If this happens but your project is still salvageable, you can pause your file to "slow stop" the CNC. Staff can help find a replacement bit and show you how to resume your file.

Once your file finishes:
If your project uses multiple bits saved to separate .spb files, proceed to step 16. If not, skip to step 17.

16. **Prepare for the next file with a different bit.** Repeat step 8 by changing the bit to the one set in the next file. You will want to keep your X and Y origin positions the same since this is the same project. However, you will need to repeat step 10 and re-zero the Z position with the Z-zeroing plate with the new bit in the spindle. Proceed by once again following steps 12-15. Repeat as needed with every new bit to finish your project.
17. **Your project is finished!** Remove your project from the CNC bed, and clean up any wood chips or dust created from your project. Be sure to save or move any files you want to keep to a folder with your name in the 'Member Files' folder on the desktop. Be sure to turn the ShopBot and spindle key to their off positions, as shown below.

As always, if you need assistance, don't hesitate to ask one of our staff!

Happy CNC Routing!