

# ShopBot Desktop Beginner Training

By: David Cardarelli  
version 1.1

- 1) Lift the red shield on the main power switch, and switch the machine on
  - i) Using the key, make sure that the Variable Frequency Drive unit (gray box on the side of the machine) powers on (it has a delay)
- 2) Load the "ShopBot 3" software on the PC and wait for it to connect with the machine
- 3) This will bring up the red position box
  - a) Within this box, select the yellow "keypad" icon "K Command"
- 4) With the keypad active, you may use the on screen buttons to move the gantry, car, and spindle. You can also use the arrow keys on the keyboard, as well as the page up/down keys.
  - a) Raise the Z axis so you have access to the collet, and collet nut
- 5) Turn the machine OFF at the red flip kill switch again
- 6) You will see two wrenches on the spoilboard. One is a blue park tool wrench (20mm), which is used to hold the spindle stationary. The other is the splined collet wrench. Using these two together, please loosen the collet nut and remove the nut and collet. You may be able to do this by hand without the wrenches.
- 7) Our machine has  $\frac{1}{4}$ " and  $\frac{1}{8}$ " collets, which means we can only use those sized bits
- 8) Ensure that the collet is installed properly into the collet nut. It should seat with a very pronounced "click"
- 9) Slide the appropriate bit fully into the collet until the top is flush with the collet.
  - a) Ensure that none of the cutting points, flutes, etc. are pushed into the collet... that's bad
  - b) Ensure that no more than  $\frac{3}{8}$ " sticks up out of the top of the collet
  - c) We want to be CERTAIN that at *least*  $\frac{3}{4}$  of the bit shaft is inserted into the collet. This will prevent the bit from spinning within the collet, or the bit breaking (GOGGLES!)
- 10) Back at the machine, ensure the threads are free of debris and clean, and hand thread the collet nut onto the spindle
- 11) Using the two wrenches, tighten the collet nut so it is hand tight, plus an additional  $\frac{1}{8}$  turn. The official spec is 59ft-lbs.
- 12) Now is a good time to position our materials on the spoilboard
  - a) Please check your screw depth, and T-Clamp clearance!
  - b) Please take a moment to decide where you'll be producing your piece. Now is a great time to measure!
- 13) Once the bit is secured, power the main power switch back on so we can begin the zero axis procedure
- 14) Zeroing the Z (every time, with every bit, with every material adjustment)
  - a) With the gantry and the car positioned roughly near your cut, and your bit installed, place the Z zero plate (located in a sleeve on the left front of the machine) flat on the table under the bit.
  - b) Take the grounding clamp, and attach it to the bit itself. This will allow an electrical current to flow through once the bit touches the plate, and the zero will be set.
  - c) Back at the PC, close the yellow control box, and on the RED control box, you'll see a "Z zero height" icon. Select this. The machine will load a script, and ask if you're ready to zero. With the plate flat under the bit, and the grounding clamp connected to the spindle shaft, hit enter, and the machine will go down and touch the plate. You should see the number 1 input icon light up when it makes contact. If this does not happen, contact the garage manager.
  - d) The machine should now retract the Z to .500 inches above the zero surface.
  - e) Remove the plate, and clamp, and store them ensuring they do not touch, which would cause the #1 icon to light up.
  - f) If for any reason this was unsuccessful, consider the machine INOP, and contact the garage manager.
- 15) Zeroing the X and Y axis
  - a) Select the "X & Y Zero" Icon on the red control panel
  - b) The machine will use the limit switches to calibrate.
  - c) Select OK afterwards
- 16) Raise your Z and attach the dust foot.
- 17) Next we need to warm the bearings in the spindle up to operating temperature.
  - a) Adjust the VFD to approximately "100" which equals roughly 6000 RPM. This should run for approximately ten minutes to properly warm the bearings up to operating temperature. So please note the time.
  - b) Install the front plexi shield
  - c) Open the yellow control box, and select the "#1 Output" blue icon. This WILL activate the spindle.
- 18) While the bearings are spinning up to operating temperature, let's work on our design!
  - a) Switch to the Partworks Design Guide on page 3

- 19) After ten minutes, adjust the VFD to read approximately "200" which will be 12000 RPM. You should hear the speed audibly change as you increase the RPM
  - a) Select the "#1 Output" icon again to turn off the spindle
- 20) With the bearings fully up to operating temperature, it's time to run a Part File!
  - a) First we should run our Part File in "Preview" mode
    - i) Enter an "SP" command: [S]et, [P]review Mode
    - ii) Next select an "FP" command "[F]ile" > "[P]art file load" to load a sample Part File "El Garage Manager SB Training" or your part file that you have previously designed
    - iii) This will bring up the "Fill-in sheet"
    - iv) For the sample, we can leave the values as default and simply hit "enter" which will load the design onto the screen for you to preview
      - (1) This is the path that you have programmed into the tool.
      - (2) Red lines indicate travel moves
      - (3) Blue lines indicate cutting moves
    - v) Observe this file in 3D, and expand the "Material" tab on the left to find any errors
      - (1) Check material thickness, and orientation for Z
- 21) Select the "Toolpaths" tab on the left
  - a) Ensure that the "Table", "Material", and "Origin" boxes are checked, and CONFIRM that these make sense. Click them on and off to be sure.
  - b) Select "Preview Cut" to ensure your finished (digital) model matches what you'd like to have in real life
- 22) Now that the sample looks good, we can switch to "move/cut mode" with an "SM" command: [S]ettings > set to [M]ove/Cut Mode
- 23) From here, we should run an "air cut" to be *certain* the machine does what we want before actually cutting the material.
  - a) Air-cutting allows us to run our actual toolpaths with the machine zeroed a distance ABOVE our material. This is an excellent way to ensure that the machine has interpreted your toolpaths correctly.
  - b) To do this, we should run a "K Command" and raise our Z to a specific (appropriate) height
  - c) We can now set this as our zero position by typing a letter "O" command and a "Z" command
  - d) ESC out of the yellow control box
- 24) Select an "FP" command to load your file once again and press "Enter" at the Fill-In Sheet
  - a) Ensure our safety shielding is in place
  - b) Safety glasses are mandatory
  - c) The machine will ask if the correct tool is in the spindle
  - d) The machine will ask if the Z axis is zeroed
  - e) This will ask you to select "OK" to run your PartFile. THIS WILL START MOVING THE MACHINE
  - f) When the PartFile is done running, the machine will re-home, and ask you to hit "OK" to acknowledge completion of the cut
- 25) Now to make our actual cut, we need to send the Z axis back to it's original zeroed position, and use another "OZ" command to zero it
  - a) We can then send the Z to the safe height of .500 inches and exit the Yellow control box
- 26) Run your last "FP" command, and hit enter at our "Fill-in Sheet"
- 27) The machine will ask you if it's ok to start the spindle and run the PartFile
- 28) THIS WILL BEGIN YOUR ACTUAL CUT!!!!
  - a) Ensure all safety measures are in place
  - b) Ensure that our two primary environmental controls are active (Fein Dust Extractor, and JET Air Filtration)
  - c) Select "OK"
- 29) Once the machine has finished the cut, it will re-home
- 30) We should inspect our work before we remove the stock to ensure we don't need to make any adjustments
- 31) Once we're sure our work is satisfactory, we can position the gantry and car in a position that will allow us to easily remove the bit, and our work
- 32) Power the machine off at the VFD and the main red kill-switch
  - a) Remove the bit, and reinstall the collet, and collet nut
  - b) Once the bit is safely stored, remove your materials, tidy up and log your time

1. Open "PartWorks X.x"

2. Select “Create a New File” which will bring up our “Job Setup” window
  - a. Here we’ll need to define our material parameters
    - i. The ShopBot can support material of 18” in Y, and 24” in X
  - b. Select the position you would like to set the “Z Zero” to zero from, and the ACTUAL thickness of your material
  - c. Select the “XY Origin Position”
    - i. For the demo, we can use the center since it makes it easy to manipulate our design
  - d. Ensure your “Units” are correct for your drawing/design
3. When you select “OK” this will bring up the “Drawing” left hand tab, and open your basic workspace
4. Select the “Draw Rectangle” icon
  - a. Set the “Anchor Point”, and the XY coordinates (units based) for where you’d like it positioned.
    - i. For our demo, let’s try a center anchor at 0, 0.
  - b. Choose your “Corner Type”
  - c. Determine your “Size”
    - i. For the demo let’s use 3(x), 1.12(y)
5. Next, select the “Import Vectors From A File” icon under “File Operations”
  - a. Navigate to the “El Garage Manager ShopBot Training” folder, and select the “NCState El SB Training” Illustrator file
  - b. Then, since the PartWorks Vector software will load that file in whatever position it was originally saved in, it’s best to select the “Zoom to fit” icon under the “2D View Control” section
6. Next, with our logo selected, under the “Edit Objects” section we can select the “Move, Scale, Rotate Selection” icon
  - a. If we hold “shift” and drag a corner to scale the object, it will scale about its center point
  - b. If we don’t hold shift, it will scale about its opposite corner
  - c. Position the center at our 0, 0 point using the “Move Selected Objects” Icon on the “Edit Objects” section
7. Select the “Set Selected Objects Size” icon under the “Edit Objects” section
  - a. Set it to “2.5” and be sure that “Link XY” is checked
8. Now, select our entire design with click and drag, and click the “Move Selected Objects” icon under the “Edit Objects” section
  - a. Choose our anchor to be the lower left, and set our XY coordinates to be where we’d like it on our board
9. With our design set, we can select the “Switch to Toolpaths Tab” icon in the “2D View Control” section which will bring up and lock our right-hand “Toolpaths” tab
10. First, let’s check our “Material Setup” by selecting the first icon
  - a. What should we know about our rapid Z gaps?
11. Next, highlight the two outer rectangles of our design, and select the “V-Carve/Engraving Toolpath” icon
  - a. We want to have “Flat Depth” selected, and at 0.2 inches
  - b. Our “Start Depth” should of course be 0.0
  - c. Under “V Tool” we need to ensure we have the proper bit selected for this toolpath. In this case, we’ll be using the 60° ¼ inch bit Whiteside #1541
    - i. These values should be set, so there should be no need to adjust. But it’s well worth taking a moment to be *sure* they’re correct.
  - d. Select “Calculate”
12. Now, click back over to the 2D “PartWorks” view tab, open our “Drawing” tab, and select the “Zoom Selected” icon.
  - a. This shows us our toolpath in 2D, and is a very useful check to ensure its doing what we want it to do!
13. Now, select all of our lettering, and choose another “V-Carve/Engraving Toolpath”
  - a. Using identical settings, setup our second toolpath, and cross check it.
14. Lastly, we need to save each individual toolpath by highlighting it, and selecting “Save Toolpath” It should be saved in .sbp (ShopBot) format
15. Then head back over to step 19 on page 2 of our main guide

## Spindle RPM and Feed Rate Calculations | Chip Load Formula

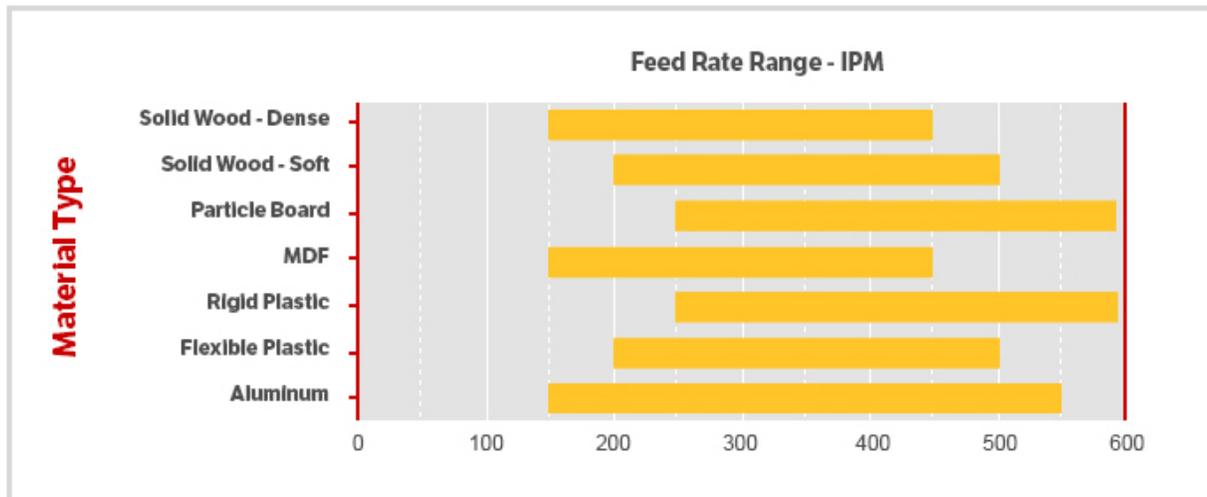
To optimize chip removal the rotating speed to the cutting tool and the linear axis speed that the tool moves must be balanced to maximize metal removal, machine performance, and cutting tool performance. The following information provides background information of their relationship to help maximize machining performance.

## Spindle Speed

Incorrect spindle speed is a common error in CNC machining. Each material type and type of cut has an ideal tool profile and cutting speed. Larger diameter tools require slower cutting speeds. The spindle speed and feed rate for a given cut must be balanced for best work quality, tool life and spindle life. The speed of the spindle is controlled by a frequency inverter. All spindles are 3-phase asynchronous motors with infinitely variable speed from 0 rpm to the maximum rated rpm. The speed can be achieved by correctly programming your frequency inverter drive to match the spindle.

## Feed Rate

The feed rate of the cutting tool must be balanced proportionally with the spindle rpm. Changing one influences the other. Feed rates set too slow decrease tool life by causing overheating and this condition may leave burn marks on the work. A tool moving too slowly causes the tool to heat up because not enough material is removed to cool the cut interface. Determining the best feed rate is determined with experience and trial-and-error to achieve best results. The feed rate chart shown here provides a range for starting point values. For first time setups, begin at the mid-range. Your cutting tool supplier can advise the cutting data for your specific application.



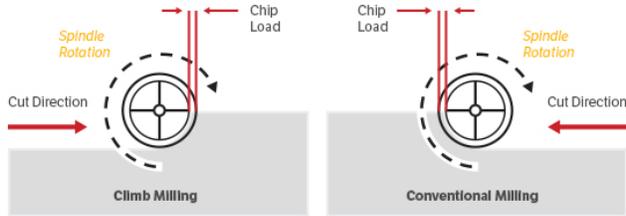
<http://www.pdsspindles.com/engineering-speeds>

## Chip Load

Chip Load is a term used to describe the thickness of a chip removed by one cutting edge of the tool. Chip load is sometimes referred to as 'feed per tooth'. The chip load is the radial depth of cut of the cutting tool in one revolution and calculated as follows:

## Chip Load = feed rate ( ipm ) ÷ ( cutting rpm x number of cutting edges )

Chip load is one factor used to determine a starting point to determine cutting speed (rpm) and feed rate values for the setup. For wood, a chip load that is too low generates too much heat, the chips produced are ‘dust-like’ and may leave burn marks on the part. A chip load that is too high is pushing the cutter through the material. This generates high radial loads on the spindle bearings, and over time, can cause spindle failure, excessive tool wear, and tool failure. There are two styles of milling: climb milling and conventional milling. For woodworking, ‘climb milling’ produces a better finish with less tear out, and some say, provides longer tool life.



Chip load is one of many factors used to size the spindle and determine machine requirements, such as:

- The speed range and power requirements for the spindle
- Tool load to determine type and grade of cutting tools
- The amount of torque generated to make the cut

We strongly encourage you to consult PDS on new spindle applications to assist in establishing your baseline cutting values. Our staff is here to discuss your technical questions by phone or email. The chart below provides chip load values for common router tools. These are ‘not to exceed’ starting point values for first time setups. Actual values will vary as a result of many machine factors, such as: rigidity, horsepower, collet condition, spindle integrity, part clamping, hold down, and others.

MATERIAL	CHIP LOAD FOR COMMON TOOL DIAMETERS (Inches)			
	0.125" Dia.	0.25" Dia.	0.375" Dia.	0.500 Dia. & Up
HARDWOOD	.003 to .005	.009 to .011	.015 to .018	.019 to .021
SOFT PLYWOOD	.004 to .006	.011 to .013	.017 to .020	.021 to .023
MDF / PARTICLE BOARD	.004 to .007	.013 to .016	.020 to .023	.025 to .027
HIGH PRESSURE LAMINATE	.003 to .005	.009 to .012	.015 to .018	.023 to .025
PHENOLIC	.004 to .005	.011 to .012	.017 to .018	.024 to .026
HARD PLASTIC	.002 to .004	.006 to .009	.008 to .010	.010 to .012
SOFT PLASTIC	.003 to .006	.007 to .010	.010 to .012	.012 to .016
SOLID SURFACE	.002 to .004	.006 to .009	.008 to .010	.010 to .012
ACRYLIC	.003 to .005	.008 to .010	.010 to .012	.012 to .015
ALUMINUM	.003 to .005	.005 to .007	.006 to .008	.008 to .010

<http://www.pdsspindles.com/engineering-speeds>

Notes: