Open Works
CNC Safety Manual
CNC machine overview

The CNC and Laser Shop is equipped with two Shopbot CNC machines for wood, plastic, and other soft materials, a Laguna CNC machine for hard plastics and soft metals, and a Tormach 5-axis CNC for multi-dimensional milling in metal (which is covered in a separate guide.) Note: There are additional steps for milling metal on the Laguna, talk to a Tech for help.

The bed size of the Shopbot CNC machines and the Laguna is 48” x 96”.

Beyond shared CNC basics, the setup and controls for the Shopbot and Laguna are different. Refer to the diagrams and instructions on the following pages to operate each.

Safety and setup for CNC routing

Follow this checklist every time you use the CNC machines. This workflow will help you get the best results and operate the machines safely.

• Wear proper safety gear covered on page 10.
• Never place hands near the spindle or on the bed while the machine is running.
• Know the size of your material - length, width, and depth (thickness). Measuring tape and calipers are available in the tool chest.
• Check your tool parameters. In the software, and always check your math for Feed Rate and Spindle RPM (covered on the next page). Any user can save tools, so be sure you are using your own correct settings.
• Preview your toolpaths and verify your X, Y, and Z zeros before you begin. Check your file with a Tech before sending a job.
• Check the bed for damage and debris. Chips and sawdust can affect the level of your workpiece and quality of cut.
• Clear the table of any extra materials or tools before you begin.
• Secure your workpiece to the wasteboard using a sufficient quantity of nylon nails or screws. The wasteboard is a piece of MDF provided as a backing on the CNC machines.
• Verify that your toolpath is clear and will not hit any hardware or fasteners.
Shopbot CNC machine

1. Spindle
2. YZ Car
3. Gantry
4. Bed / Wasteboard
5. Rails
6. Frame
7. Dust hose
8. Dust boot
9. E-Chain
10. Control box
11. E-Stops

XY: 0,0

X-AXIS
Y-AXIS
Z-AXIS

Shopbot CNC overview.
Selecting a bit

The bit you select is critical to getting the best cut. It attaches to the spindle with a chuck and collet, which are sized to match the bit.

- You may bring your own bits, or find the correct bit sets in the drawer for each CNC machine. Make sure to check with a Tech to approve outside bits for use with your file and material.

- Bit size is often labeled on the shank, or can be measured with calipers.

- There are a wide variety of bits available for every type of project and material. Selecting the best bit is a matter of practice and experience.

**CNC routing bit types**

**Endmills** (spiral cutters) used for straight cutting. Range of sizes and flute types (below).

**Ballnose** (contouring) rounded end bits used for 3D carving and smooth contours.

**V bit** carves sharp grooves used for lettering and sharp details.

**Endmill flute types**

**Straight Flute** – great all around bit, decent chip removal.

**Up Spiral** – great chip removal, can tear-out the top of thin veneer such as finish grade plywood.

**Down Spiral** – poor chip removal, no tear out, slower feed rate.

**Compression** – combination of up and down spiral, great all around bit, great for plywood or laminated sheet goods.

- Check the bits for damage, including char marks, and chips in the cutting flutes (bring any damaged or charred bits to the Shop Tech).

- When working with multiple bits for a single job, use the labeled tray on the Shopbot computer stand to remember which bits are assigned to each tool number.
Toolpaths etc.
It is important to select the correct type of toolpath and bit function for the job you are trying to execute.

Profile and pocket clearing describe the path the tool makes around a shape. Refer to the examples to see profile and pocket cuts at various stages of the process.

**Profile** cuts out the contour of a shape instead of removing material inside the lines. *(24A - A Examples)*

**Pocket clearing** etches the surface of the material, milling out the space inside a shape. *(24A - B Examples)*

Climb and conventional cuts refer to the way the bit removes material across the milling surface.

**Climb cut** The bit rotates in the same direction the tool is moving. Preferred choice for CNC machines at Open Works. *(24C)*

**Conventional cut** The bit rotates opposite the direction the tool is moving. *(24D)*

- Multiple toolpaths for a single project can be saved together.
- Several passes over the same path with different settings, or a toolchange between passes may help get the best results.
- .DXF files works best. Objects from 3D software can also be used but they need to be flattened to create toolpaths. You can also design directly in Vcarve.
Selecting Toolpaths and Tool Info

When selecting a toolpath type (profile, pocket, engraving etc) you must also select what type of bit will be used. Open Works has many preset cutting parameters, but it is always good to check that they are suitable for your job.

Cutting Parameters

Pass Depth
The maximum depth of cut the tool can cut. The Pass Depth controls the number of z level passes that are calculated for a toolpath. For example, creating a pocket 1 inch (25.4 mm) deep using a tool that has a Pass Depth of 0.25 inches (6.35 mm) will result in the toolpath making 4 passes.

Stepover
The distance the cutter moves over when doing area clearance cutting. The greater the stepover the faster the job will be machined, but this must be balanced with the material being cut and the tooling being used, to ensure that the tool does not break.

Feeds and Speed
Spindle Speed
Speed of tool rotation, specified in revolutions per minute.

Feed Rate
The surface cutting rate at which the cutter is moved in the material. The units can be specified in distance per second or minute.

Plunge Rate
The cutting rate at which the cutter is moved vertically into the material or during ramping moves. The units can be specified in distance per second or per minute.
CNC formulas

In order to cut safely and cleanly, it is critical to calculate the correct feeds and speeds. The wrong settings can damage your workpiece, the machine, and endanger you and those around you. The settings for each job or toolpath depend on each of these four variables:

- **Chipload** is the measurement of the material removed by each cutting edge during a cut. Can be used to make adjustments to feed and speed parameters for cleaner, safer, and more efficient cutting.

- **Drill (# of cutting edges)** a rotary end cutting tool has one or more cutting lips and one or more flutes to pass chips and emit cutting fluid. Generally, the number of flutes = the number of cutting edges.

- **Feed rate (IPM/IPS)** is the speed of the tool (bit) traveling through the material. It is measured in either inches per second (IPS) or inches per minute (IPM). Most calculations are in IPM, to get IPS - divide by 60. Start with a slow feed rate (around 2 inches per second) and increase as needed.

- **Revolutions per minute (RPM)** is the speed at which the spindle is rotating. Note: The Laguna calculates speed in rotations per second (RPS) so convert formulas accordingly.

Utilize the recommended settings and the variables you know to solve for the missing variable using any of the formulas below:

\[
\text{chip load} = \frac{\text{feed rate (IPM)}}{\text{RPM} \times \# \text{ of cutting edges}}
\]

\[
\text{feed rate (IPM)} = \text{RPM} \times \# \text{ of cutting edges} \times \text{chip load}
\]

\[
\text{RPM} = \frac{\text{feed rate (IPM)}}{\# \text{ of cutting edges} \times \text{chip load}}
\]
File setup and running the job

- Open your file in the VCarve software, and enter the “Job Size”, “Z Zero Position” and “XY Datum” in the Job Setup window. (34A) Note: Measure exact material thickness using calipers (Example: 0.734 inches) and input into VCarve. Never cut more than 0.02” into the wasteboard.

- Review your toolpaths with a Tech and save your VCarve file.

- Select the toolpaths you want to cut by checking the box next to them, and make sure they are in order. Typically, pockets should be cut before throughcuts. (34B)

- Select the correct post processor. “Shopbot TC Inch” (.SBP) This is crucial for communicating correctly with the Shopbot. (34C)

- Save toolpaths to file. This outputs your toolpaths into Shopbot’s version of G-code. (34D)

- Select “Start” which will appear at the bottom of the window if the file has loaded correctly. Code will run, but the job will not begin yet.

- Run through the final safety checklist:
  √ Spindle warmed up?
  √ Material secure?
  √ Z zeroed?
  √ File correct?
  √ Spindle interlock key engaged?
  √ Safety gear on?
27A Shopbot CNC onboard controls.

27B Controller.

27C E-stop button (located on each side of gantry).

27D Spanner wrench.

27E Crescent wrench.

27F (Left to right) Chuck, collet, bit.
Shopbot controls
Follow these steps for controlling the Shopbot.

Turning the machine on
• Turn on the power by turning the knob on the control box from Off to On. (28A)

• Disengage both e-stops by twisting and pulling out. One is located on the gantry, and the other is by the controller next to the computer. (28B, 28C)

• Press the reset button on the controller. Make sure you hear a click sound from the control panel. (If not, the machine may be off, or one of the e-stops not disengaged). (28D)

• Engage the spindle by turning the key on the control box from “Disengaged” to “Engaged”. (28D)

Moving the gantry and spindle
The Shopbot is controlled using the SB3 software which is located on the desktop of each machine’s corresponding computer. Any time you are moving the machine, observe the following for safety:

• Keep hands away from all moving parts.

• Be sure the spindle is not running, and the Z axis is raised enough to clear the surface of the material.

• Make sure the bed is free from stray tools or materials.

• Familiarize yourself with the axis orientation. The X axis is the long dimension of the table. The Y axis is the gantry where the spindle operates. See diagram 26A.

• Jog slowly to avoid hitting proximity switches at the threshold of each axis, or damaging the bit or your material.
There are several ways to move the gantry and the YZ Car along it:

**Jog X, Y, and Z manually** using the Keypad panel. Click the yellow button in the red Position panel, or press K on the keyboard to bring up the controls. Use the mouse *only* to click the arrows *not* the computer keyboard arrows - to move the gantry. *(29A)*

**Use the homing buttons.** In the red Position panel, click the “0,0”, “X,Y” and “Z” home buttons, or type C3 in the command line to move each axis to its home location. (While running a command, your cursor will be locked in the red window). *(29B)*

**Type coordinate values** in the red Position panel, type a value and click the Go-To button to move the spindle to that location on the X, Y, and Z axis. *(29C)*
Machine setup

Run the spindle warmup routine

- Make sure the spindle is empty or has a tool securely in place.
- Type C5 into the command line, or go to Cuts > C5 Spindle Warmup Routine in the menu.
- Press the green Start button on the controller next to the computer, wait to hear the spindle turn on, then click “OK” in the dialog box. (The warmup procedure takes about 10 minutes). (30A)

Load material

- Check the wasteboard for debris and level. You can clear off any burs with a chisel.
- Move the gantry out of the way. If you type an X value of 90 in the red Position box, the gantry will move to the far end of the bed.
- Place material on the bed. Align material with the bottom right corner of the Shopbot, or X-o, Y-o position. (30B) (To place material somewhere else on the bed, ask a Tech for help resetting X, Y zero.)
- Secure the material to the wasteboard using the pneumatic nailer or screws. Be sure to use a sufficient quantity of nylon nails at appropriate distances to hold your piece in place (approximately one every 6 to 10 inches, depending on material size). See page 48 for instructions on using the pneumatic nailer.
Install the bit / Toolchange

- Jog the gantry to a reachable location and height where you can comfortably install the tool.

- **Lower the dust boot** (if attached) enough to access the spindle head. See page 33 for instructions.

- **Turn the key and remove the** spanner wrench from the power box. This safety feature will ensure the spindle does not turn on while installing the bit. The crescent wrench is also located on top of the power box. *(31A)*

- **Place the collet inside the chuck** then insert the bit, until about 1/2 an inch of the shank above the cutting flutes is visible outside the collet. *Note: If a bit is already installed, remove the chuck and collet and clean out any dust with the overhead compressed air gun.* *(31B)*

- **Attach the chuck to the spindle and hand tighten** taking care not to drop the bit which may damage it or your workpiece. *(31C)*

- **Hold the crescent wrench in your left hand and the spanner wrench in your right** and push away from each other to tighten the collet. *Do not overtighten* - it only needs to be as tight as a food jar lid. *(31D)*
Set Z zero height

• Make sure you are wearing safety glasses.

• Remove the copper alligator clip from the top of the YZ car, and clip to the bit just below the collet. (32A)

• Place the aluminum plate below the spindle. (32B)

• Select the Z-0 button in the red Position panel. (32C)

• Hold the plate down with your fingers to account for flex, but keep a safe distance from the bit. (32D)

• Press “OK” in the dialog box to touch off the plate. Keep hands away from descending bit.

• Check for the second dialog box to confirm the Z has been zeroed.
Attach dust boot

1. **Locate the dust collector adapter** on the cart for the corresponding Shopbot.

2. **Loosen the dust collector knob** by turning counterclockwise and carefully lower the dust collector frame.

3. **Slide the adapter onto the frame.**

4. **Pull the spring-loaded pin** to secure it in place.

5. **Raise the frame** and secure in place by tightening the knob.

6. **Attach the black hose** to the dust collector adapter. This will connect to the vacuum system.
• **Start the spindle** by pressing the “Start” button on the controller box. *(35A)*

• **Click “Ok”** in the dialog box that pops up on-screen to run the job. *(35B)*

• **Supervise cut** while hovering over the “Stop” (Pause) button in the red *Position* window in case your file starts cutting in an unexpected way. Be prepared to engage the E-stop in the event anything goes wrong - this will cut power and cancel the job.

![Image of the control box](image)

**Clean-up**

• **Remove the collet and bit.** Dust off with the compressed air gun, and return to the tool chest. Let a Tech know of any tool damage or cleaning necessary.

• **Turn machine off** following **ALL** the steps to turn the machine on, in reverse, unless you know someone will be using it next. See page 28. *(35C)*

• **Save files to a flash drive** and delete off of computer.

• **Remove piece carefully** trimming tabs and removing any remaining nylon nails only with the designated chisels and mallets. *(35D)*

• **Clean debris from the wasteboard** and let a tech know if it needs to be resurfaced.

• **Place all small material and dust in the trash;** break down large pieces and place them in the dumpster outside.

![Image of the control box](image)

35A Push start on the control box.

35B Click OK in dialog box.

35C Turn machine off.

35D Remove piece from wasteboard.