Semler's
WOOD SHOP
EST. 2021

Running the ShopSabre CNC

VCarve ▸ G-Code ▸ ShopSabre Controller ▸ 30” × 40” Bed

Quick-Reference Cheat Sheet

Follow every phase in order. The most common mistake is opening the G-code file in the ShopSabre controller BEFORE setting Z-zero. Always set Z-zero first — you cannot do it after the file is open.

WORKFLOW OVERVIEW

1	2	3	4	5	6	7
Get File on USB	Check Feeds & Speeds	Save G-Code	Set Z-Zero	Open File in Controller	Check Boundary	Run the Job

PHASE 1 — PREPARE YOUR FILE

1 Get Your VCarve File on a USB Drive

Bring your .vcarve file on a USB drive to the CNC computer

- Save your VCarve project file (.vcarve) to a USB drive before coming to the CNC
- Plug the USB drive into the CNC computer
- Open VCarve on the CNC computer
- Go to File ▸ Open and navigate to your USB drive
- Open your .vcarve file in VCarve — do NOT open it in the ShopSabre controller

⚠ **WARNING** — Open your file in VCarve only. Do not open it in the ShopSabre controller yet — Z-zero must be set first.

PHASE 2 — VERIFY FEEDS, SPEEDS & CUT DEPTH

2 Check Your Feeds, Speeds & Cut Depth

Verify all toolpath settings before saving G-code

- In VCarve, open your toolpaths and review each one
- Check the feed rate, plunge rate, and cut depth for every toolpath
- Cross-reference the Feed & Speed / Cut Depth chart to confirm your settings are correct
- Do not proceed until all settings match the chart

◎ **CAUTION** — Wrong feeds, speeds, or cut depth can break bits, burn material, or damage the machine. Always check the chart.

PHASE 3 — SAVE G-CODE

3 Save Your Toolpath as G-Code

Calculate and save your toolpath file to the computer

- In VCarve, go to the Toolpaths panel
- Click Save Toolpaths (or Calculate and Save)
- Make sure the post-processor is set correctly for the ShopSabre
- Save the G-code file to the Desktop or Documents folder on the CNC computer
- Note the filename so you can find it easily in the next step

ⓘ INFO Save to Desktop or Documents only. Do not run G-code directly from the USB drive.

PHASE 4 — SET Z-ZERO ⚠ DO THIS BEFORE OPENING YOUR FILE

4 Drive the CNC & Set Z-Zero

This must happen before you open your G-code file in the ShopSabre controller. Once the file is open, you can no longer set Z-zero.

Use the keyboard to jog the CNC and set Z-zero

- Open the ShopSabre controller (do NOT load your file yet)
- Use the arrow keys (left/right/up/down) to jog the CNC head over your board
- Position the head over the top surface of your material where you want X/Y zero
- Use Page Up and Page Down to slowly lower the Z axis toward the material surface
- Slide a piece of paper under the bit as you lower it
- Stop when the bit just grips the paper — you should feel slight resistance when pulling the paper
- Set Z-zero at this position in the controller
- Drive Z back UP before doing anything else — do not leave the bit at the surface

⚠ WARNING Always drive Z back UP after setting Z-zero. Leaving the bit at the surface risks breaking the bit when you jog the machine.

⚠ WARNING MOST COMMON MISTAKE: Do not open your G-code file before setting Z-zero. If you open the file first, Z-zero cannot be set and your job will cut at the wrong depth.

PHASE 5 — OPEN YOUR FILE IN THE CONTROLLER

5 Load Your G-Code File

Open your saved G-code file in the ShopSabre controller

- In the ShopSabre controller, open your G-code file from the Desktop or Documents folder
- The toolpath will appear on screen
- Zoom in to inspect the toolpath and confirm it looks correct

ⓘ INFO Z-zero must already be set before this step. If you skipped Phase 4, go back and set Z-zero now before continuing.

PHASE 6 — CHECK THE BOUNDARY

6 Jog the CNC to Verify It Stays on the Board

Manually jog the CNC around the perimeter of your toolpath

- Use the arrow keys to drive the CNC head to each corner of your toolpath
- Confirm the head stays over your board at every corner — it should not go past the edge
- Confirm the head does not go past the hold-down clamps or board holders
- If anything looks off, stop and see Mr. Semler before proceeding

◎ CAUTION	If the toolpath runs off the board or into a clamp, the bit will break and the job will be ruined. Take your time on this step.

PHASE 7 — RUN THE JOB

7 Set Speed & Start the Cut

Reduce speed, start the job, and stay ready to stop it

- Find the speed slider in the ShopSabre controller software
- Set the speed slider down to about 15% before starting
- Press the green button (top left corner of the screen) to start the job
- The first press will zoom the view out to show the full job — this is normal
- Press the green button a second time to actually begin cutting
- Watch the cut carefully — keep your hand near the spacebar
- Press the spacebar immediately if anything looks wrong — it will pause the job
- Once you are confident the cut looks correct, you can raise the speed to between 30–60%

⚠ WARNING	Never walk away while the CNC is running. Keep your hand near the spacebar at all times so you can pause instantly if something goes wrong.
◎ CAUTION	Start at 15% speed. Only increase to 30–60% once you have confirmed the first pass looks correct.
ⓘ INFO	The spacebar pauses the job. Pressing the green button again will resume from where it stopped.

STUDENT CHECKPOINT

Full Run Checklist — Check Every Step

☐	**VCarve file saved to USB drive**
☐	**File opened in VCarve on the CNC computer** *NOT in the ShopSabre controller*
☐	**Feeds, speeds, and cut depth verified against the chart**
☐	**G-code saved to Desktop or Documents folder**
☐	**ShopSabre controller opened — file NOT loaded yet**

☐	**CNC jogged over the board using arrow keys**
☐	**Z-zero set using paper method — bit just grips paper**
☐	**Z axis driven back UP after setting Z-zero**
☐	**G-code file loaded into ShopSabre controller**
☐	**Toolpath visually inspected on screen**
☐	**CNC jogged to all corners — stays on board, clears clamps**
☐	**Speed slider set to 15% before starting**
☐	**Green button pressed once — view zoomed out**
☐	**Green button pressed second time — cutting started**
☐	**Hand kept near spacebar throughout the job**
☐	**Speed raised to 30–60% only after confirming first pass is correct**