

Quick Check list 7 Mar 2019:

Rapid Z Gaps and **Home Z** are set **above** the top of your **material** in CAD/CAM software.

Mach2/3 Arcs (inch) (*.txt) is the postprocessor used.

Mark the **X,Y Datum** corner (lower left corner) & **Z Zero** (normally the machine bed (spoil board)).

Mark **safe hold-down** (nail/screw/clamp) locations.

Use your eye and ear protection!

Set Up the CNC Router

Power up: PC; Boot Mach3; select most recent Profile; Click OK

white switch; press **RUN** when "0000.0" on display.; Power up air compressor in metal shop.

Zero X & Y: Dbl-click **Reset** to green. Click **REF ALL HOME**. Router will go to table X,Y = 0,0.

If the number panel to the right of **Zero X** and **Zero Y** don't go to 0, click on each panel, enter 0 and press enter for both **X** and **Y**. (If the machine won't go to the zero X & Y corner with **REF ALL HOME** (the outlines stay red), **shut down** and restart the PC and the Router.)

Zero Z: Page Up Z axis 2". Move the spindle +6" on X and on Y axes with the arrow keys.

Click **Reset** to blinking red/green.

Install collet & loosely install the collet nut **by hand**. (CCW as seen from above.)

Install 1st mill: Press 1st mill into collet (flutes clear of collet); **finger** tighten.

Hold wrenches as **close to the nut** as possible; **tighten** the collet nut **with 1 hand only**.

Put the wrenches back on the hooks, don't leave them on the spoil board!

Do Not Overtighten - this will destroy the threads and render the \$600 Spindle useless!

Connect Z Zero probe and **place it on the spoil board** (or your project) under the mill. **Click Reset** to green.

Click **Auto Tool Zero**. After the spindle touches the probe & **rises to 1.5"**, **remove** the probe.

Set Up Your Material

Place your **material on the spoil board; secure it** (screws or nails) square to the table rails.

Center your mill **on the X,Y Datum** corner; Click **Zero X**. Click **Zero Y**.

Run Your Project

Open the Dust Collector gate. Check that the other gates in the shop are closed.

(—>) **Click Reset** to blinking red/gn. **Change mill** if necessary. **Click Reset** to green.

Page Up to put mill ~2" **above** the spoil board. **Move mill for zeroing the in the Z axis**.

Connect the Z Zero probe; place it under the mill.

If Z zero set to base of project, place probe on spoil board. If set to top of project, place probe on material.

Click **Auto Tool Zero**. When gantry stops, **remove** probe. **Place DC shroud. DC on!**

Load GCode; Check GCode for tool path name, installed mill, Speed & Feed. **Rewind**.

To test a toolpath with an "air cut", set a block of scrap **greater** in thickness than your material **on top** of your material. **Zero** mill to **top** of **block**. Set block aside; run toolpath. **Re-zero** mill to **toolpath Z Zero**.

(->>) **Click Cycle Start**

Be sure the mill is spinning @ Speed! Check that driver display is **Speed ± 10%**.

Be ready to click **Stop** or **Reset!**

If you have to **Stop** the machine before the tool path is complete: **raise the mill above your material;**

correct the problem; click **Rewind**. (The tool path will reset to the beginning.) Go to(->>).

When tool path is completed, **move the mill away** from the material.

Click **Close GCode**. Remove the dust collector shroud.

Go to (—>).

Clean Up Any Mess!

When the project is complete, chop tabs (mallet & chisel); **remove** your project.

Remove your **material**. **Remove** all **hold-downs**. **Clean up the chips** on the table and **floor!**

Return the collet to the tray. **Quit** Mach3. (alt f, shift x)

Power down the router (white switch), **dust collector & air compressor**. **Empty** the DC box!

GWizard - Feeds & Speeds in the Woodshop

Have the following info available:

material you're cutting; tool: HSS or carbide, cutter diameter, # of flutes, stickout; cut: depth & width(stepover)

Open GWizard; click on the **Feeds/Speeds** tab.

Panel Name (panels may be in a different order in different versions of GWizard)

Machine	Generic CNC Router (Material:) Wood (hard, soft, MDF, plywood), Plastic or Aluminum (Tool:) Carbide Endmill or HSS Endmill
Tool	<u>Tool Dia.</u> (dia. of <i>cutting</i> end, not shank) <u>Flutes:</u> 1 for plastic or aluminum; 2 for wood, plywood, MDF <u>Stickout:</u> (how far the mill sticks out beyond the collet: at least the length of the flutes, at most 1/2 the length of the mill; shorter stickout is better)
Mfg	Skip for basic setup
Cut	<u>Cut Depth:</u> wood: start with the radius of the cutting end ("Axial Engage" 50%) <u>Cut Width:</u> less than the radius of the cutting end ("Radial Engage" 10% - 48%) ("Cut Width" is often called "stepover". It is the width of the cuts which occur after the first cut.)
Results	Write these down for use in VCarve: RPM ("Spindle Speed") Feedrate ipm ("Feed Rate") (Plunge ("Plunge Rate", if shown)
HSM	Skip for basic setup
Tips	May have some interesting suggestions

(**Tortoise - Hare panel**) Click on the dot & adjust with mouse or cursor keys between 1% & 100%.

Vary the % to change Feedrate & thus smoothness of finish.

The higher the Feed, the faster, but rougher the cut. 1-29% is a good starting range.

Adjust the % until **RPM** and/or **Feedrate** turn red, then back off until both are black.

Moving between 1% & **red** numbers will show the calculated min & max Speed & Feed.

A **black Feedrate** is necessary; if you cannot get the Feedrate black, consider a different dia. mill)

Use **RPM, Feedrate & Plunge** in the "Edit" panel after you "Select" a tool in a *VCarve* tool path.

Limits (Verify the following)

HP Limit: 2

RPM Limit: **20000**

Min RPM: **0**

Feed Limit: **200**

Skip the rest for basic setup

Mini Calc Skip for basic setup

Cut KB skip for basic setup

End mill info (no endorsement by our facility or guarantee of quality / results)

Intro to end mills: <http://makezine.com/2014/09/10/endmills/>

Intro to CNC: NYC CNC.com

Basic assortment ("carbide" lasts longer):

1/4" 2 flute upcut end mills (for profiles & pockets in hardwood and MDF)

1/4" 2 flute *downcut* or *compression* for plywood

1/4" 1 flute upcut end mills for Aluminium or plastic

1/4" 60° v-bit (for engraving lettering, etc.)

1/4" 2 flute ball nose mill (for 3D contours)

Sales: (local but pricy) Rockler, Woodcraft, McMaster-Carr, Fastenal

Online: kodiakcuttingtools, toolstoday, Eagle America, Southwest Tools, Whiteside, Amana (via Amazon), MSCDirect

CNC intro summary

Safety

**Protect yourself: safety glasses, ear protection, no dangling hair, long sleeves or hoodie ties
no gloves! closed shoes**

Protect your colleagues: secure the mill, the material & avoid loose knots in wood

Protect the machine: secure the mill, the material (a nail every 6-12", including along cuts of the center of the board).

Don't cut more than 0.04" (1mm) into the spoil board

Do Not Leave the Machine While It's Running!

CNC work include 3 stages:

CAD (computer aided design) - Illustrator, Sketchup, Fusion 360, **VCarve**, etc.

CAM (computer aided manufacturing) - Fusion 360, **VCarve**, etc.

GCode (the text file that drives the CNC machine) Fusion 360 (no **G28**), **VCarve**, etc.

The Hive13-built CNC router has standard parts: the table which supports the spoil board, the gantry which moves the spindle in 3 axes in response to the GCode, the spindle, which holds and spins the cutting mill & the dust collector hose, gate and shroud (to...collect dust...).

The CNC works by moving the mill at the **feed** rate (**inches** per minute) while spinning it at a **speed**, both of which are determined by the **GCode** text file of the **toolpath** along which it is moving.

When not cutting, the mill is moved quickly *between* tool paths at a set speed called the "**rapid**".

Mills ("router bits" in the woodshop) are specified by **diameter** (of shaft & cutter, if different), numbers of **flutes** (cutting edges, 1-2 for a CNC router), **geometry** of the end of the cutter (end mill, ball nose, etc), **cutting length** (length of the flutes), **stickout** (the length sticking out beyond the collet - should not exceed 1/2 the total mill length) and **material** - HSS (high speed steel) or carbide.

When purchasing mills, you will want to specify **diameter** of cutter & shaft (if different), **numbers of flutes**, **geometry**, **cutting length** and **material** of which the mill is made.

A starter set of mills could contain a couple of 1/4-3/8" end mills & a same size shaft 60° v-bit.

Do not use hand-held (or table) router bits in the CNC router. They are not built to tolerate the lateral stress of CNC machining and can break and send metal and cutters flying!

The only exception is the use of hand-router V-bits. These can be used for engraving lettering, provided only a small part of the bit (usu. ~0.125") cuts into the wood to engrave the letters.

When calculating **Feeds & Speeds**, you will need the **cutter diameter**, **number of flutes**, **stickout** and **material**. (See the handout for details of using GWizard for feeds and speeds.)

Mills are made to cut horizontally. Don't try to drill holes with them. (Cutting out a circle that has a larger diameter than the mill is OK, it's still a horizontal cut.) If you want to drill a hole, put a drill bit into the collet & use a "peck drilling" toolpath.

VCarve Pro 9 (short overview! much more in the Vectric website tutorials)

F “finds” all vectors & “backs out” to show them all in the 2D window; function key **f9** centers a selected vector on your material in the 2D window; Type “8(space)1/16=” into a numeric panel to calculate the fraction & show “8.0625”

Create a new file:

Single Sided

enter **Width** (x), **Height** (y) and **Thickness** (z) of material; verify your **Units**

Set **Z Zero** to **Machine Bed**

Set **XY Datum Position** to lower left corner of material.

Click **OK**

CAD

Mark the placement of the hold downs (nails, etc.) with 1.5” circles.

Import or make up your design.

Remember to include a path for cutting it out of the material, if needed.

Offset under **Offset and Layout**

Click the right-pointing blue arrow icon (**Switch to Toolpath commands**) at the top of the **Drawing** panel to get to the CAM section.

CAM

Click **Set...** and enter values for **Clearance(Z1)**, **Plunge(Z2)**, and **Home/Start Z Gap above Material**

Z1 to clear clamps & screws, usu 0.5-1” (or above any clamps!)

Z2 to prevent mill crashing into material, 0.2”

Z Gap above Material - easiest is **Thickness + Z1 + Z2**

Click **OK**.

Choose each vector in turn & define the tool path parameters.

Select your tool from the database via **Tool: Select**

Click **Tool: Edit...** to enter your **feed**, **speed** and (if needed) **plunge** values

Name tool path so you know what project, what path, what mill (type & size).

Calculate.

If you are **cutting** your project **out** of the blank, be sure the **Profile** tool path you are using to cut it out is only **0.04”** deeper than the material. Be sure you get a **WARNING** from VCarve that you are going to cut through the material.

Click **OK** for the warning.

After all tool paths are calculated & checked, **Save** the file via the **File** menu.

Postprocessing

At the Hive, import your file into VCarve, affirm that you want to register it.

Go to the **CAM** window, as above; save the toolpaths via the **Save Toolpath** floppy disc **icon**.

These are the GCode text files you will **Load** into **Mach3**.

References

CAD/CAM/postprocessing

VCarve vectric.com - downloads, FAQs, tutorials

Fusion360 autodesk.com/products/fusion-360/overview (if generating GCode in Fusion, *de-select G28*)

NYCCNC nyccnc.com - CNC, Fusion 360, start a business, classes in Zanesville (east of Columbus)

GWizard: CNCCookbook cnccookbook.com/GWizardGWizardInstall1SC.html

GCode: Tormach http://www.tormach.com/machine_codes_gcodes.html

descriptions of GCode “words” (commands) like the “S” word for speed and the “F” word for...feed rate.