CNC Routing Overview (9 March 2019)
J.Sikorski (john2pt0)
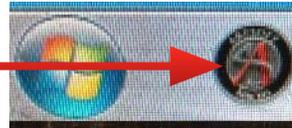
Set Up and Run Your Project

Be sure you marked the **X,Y Datum** corner & **Z Zero** (top or bottom of your material).

Be sure you know the **safe hold-down** (nail/screw/clamp) locations.

Use your eye and ear protection!

Set Up the CNC Router
Power up: **PC** on

Boot **Mach3**
Select the most recent *Profile* in
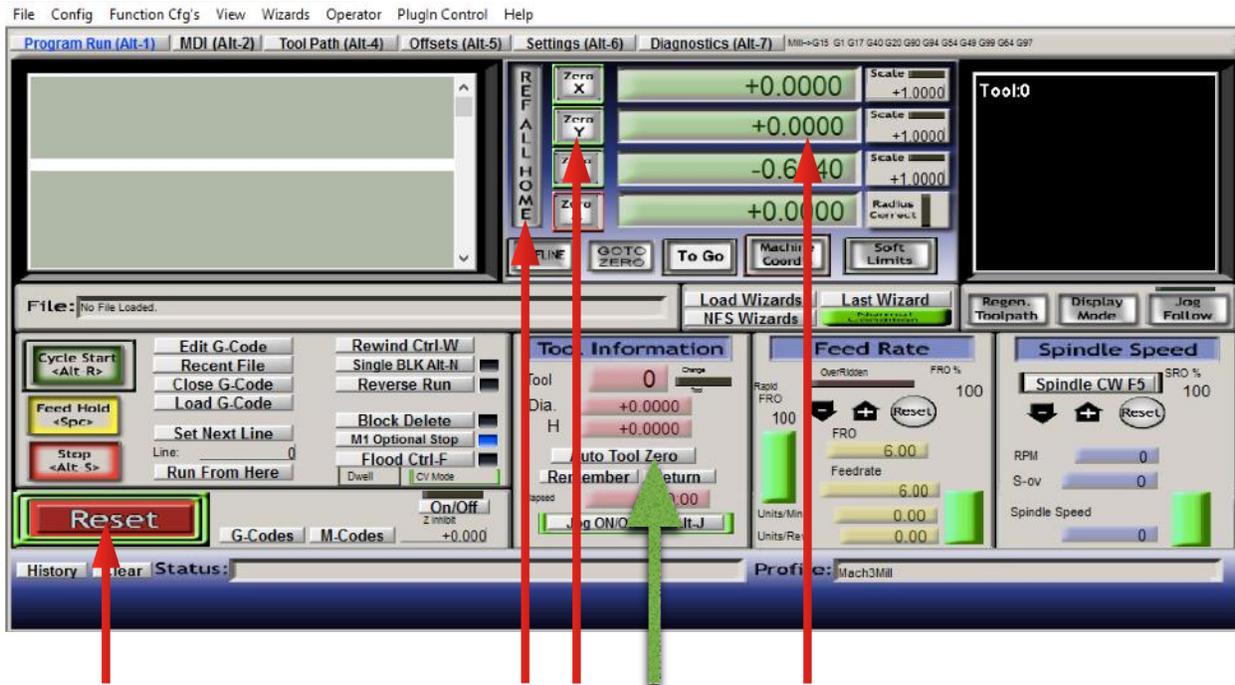the popup window.

Click *OK*

(It will complain that it can't see the router. Once you
turn the router on (below), click on "Retry", Mach3 will
now launch.)

Flip the **white switch** on the wall to power up the CNC router.

When **"0000.0"** appears on
the spindle driver display,
press RUN.

Zero the Machine X & Y:

In **Mach3:**   Double-Click *Reset* to show a steady green border.
Click *REF ALL HOME.* The router will go to table X,Y = 0,0.
The borders of *Zero X, Zero Y* and *Zero Z* will change to green.
Click *Reset* to show a blinking **red/green** border**.**
Click *Reset* to show a steady green border
Click *Zero X.* Click *Zero Y.* The numeric displays will reset to 0.
The machine is zeroed to the table **X** zero and **Y** zero ("machine coordinates").

Zero the Machine to the first mill stickout (Z0):

Use *Page Up* on the keyboard to move the Z axis up about 2".
Use the arrow keys on the keyboard to move the spindle about +6" on
     X and on Y axes.
Click *Reset* to show a blinking **red/green** border**.** (This locks the spindle.)


Collet

Install the collet**:** place the nut on the spoil board and snap the collet in.
Finger tighten the assembly into the spindle (CCW, seen from above).
Install the 1st mill: press the mill into collet (keep the flutes outside the collet);
     finger tighten.

Hold the wrenches as **close to the spindle** as possible & **tighten** the collet nut using only **one hand**. It should only rotate **about 1/4 turn (CCW** from above) to firmly secure the mill.

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

Spindle wrench

Spindle

Collet nut wrench

Do Not Overtighten!
this will **destroy the threads** and render the **$600 $pindle usele$$!**
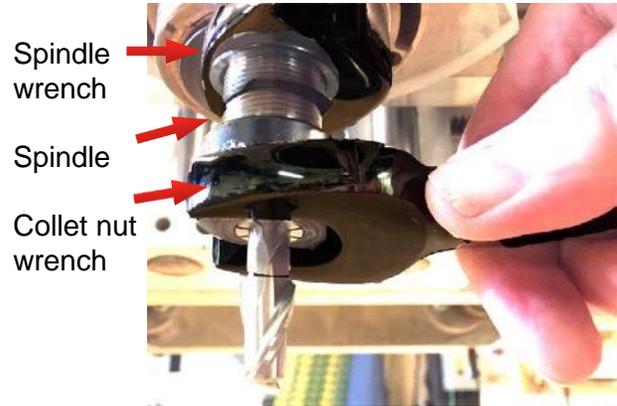
Connect the Z Zero **probe.**

Connect the yellow RCA jacks.
Clip the red-shrouded alligator clip to the mill.

(Watch that it doesn't disconnect while the router

is moving to touch the contact button.) Place the

contact button on the spoil board.

Click *Reset* to show a steady green border.
Click *Auto Tool Zero.* (green arrow, fig. above) The router will descend until the mill touches the contact button.
It will then rise to 1.5" above the spoil board.

After the spindle touches probe & **rises to 1.5",** **remove** the probe.

The machine is now zeroed to this mill stickout (Z0).

<u>Set Up Your Material</u> (zero to your material X0 & Y0)
Place your material on the spoil board; secure it square to the machine rails.
Center your mill on your project **X,Y Datum** corner.
Click *Zero X.* Click *Zero Y.*
This tells the machine where your **project/material XY Datum Corner,**
("Home X,Y") is. All toolpaths start here. ("Project coordinates")
Open the Dust Collector (DC) gate on the router.
Check that the other gates in the shop are closed for maximum suction.

Run Your Project (zero to the mill Z0 *each time you change mills*)

(—>) Click *Reset* to show a blinking red/green border.

> **Change** the **mill** if necessary.
> Click *Reset* to show a steady green border.
> *Page Up* to put the mill **2" above** your material.
> Use the arrow keys to move the mill for zeroing the in the Z axis.
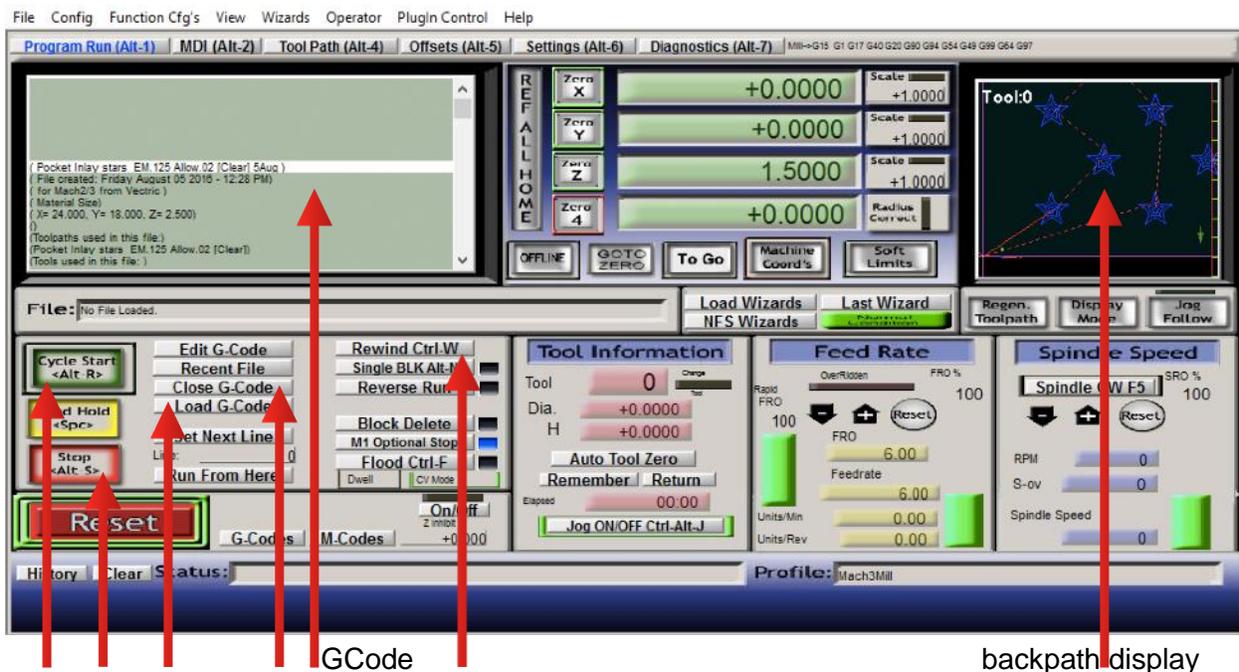> **Connect** the **Z Zero probe**.
>> If Z zero set to the base of material, place the button on the spoil board.
>> If set to the top of the material, place the Z button on the material.
> Click *Auto Tool Zero.*
>> **Remove** probe once the machine has raised the mill to 1.5" and stopped.
> **Replace** the dust collector **collar**. Turn
> the **Dust Collector on!**

GCode   backpath display

Click *Load GCode*

Check that the backpath window shows the expected toolpath pattern.

Scroll through the GCode for tool path name, installed mill,

> speed (≤ 20000) (line N150nnnnnn**Sxxxxx**) and feed
> rate (≤ 200) (line N190nnnnnnnn**Fxxx.x**).
> Click *Rewind.* Failure to click *Rewind* could result in damage to your project!

(->>) Click *Cycle Start*

> Check that the mill is spinning @ Speed!
>> (Look at the spindle driver display readout. Note: it may be up to about 10% off.)

Do Not Leave the Machine While It's Running!

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 (You may need VCarve to change the toolpath.). If you changed the toolpath, Go to (—>)

Otherwise: click *Rewind* . Go to(->>).

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

Click *Close GCode.* (fig. above)

Remove the dust collector collar.

Go to (—>).

What could possibly go wrong?
or
Do Not Leave the Machine While It's Running!

Problem: Ref All Home does not send the router the the lower left corner **or** the **machine tries to push past the stops.**(I.e., the machine won't auto-zero to the table.)

Response: Stop!

Cause: Limit sensors are off line.

Fix: Quit Mach3, shut off the PC, shut off the router, restart as above (PC, Mach3, router)

(If **Ref All Home** fails again, move the gantry to the **XY Datum** corner of your material. Click **Zero X** & **Zero Y)**

Problem: Auto Tool Zero doesn't finish by retracting to 1.5" above the Z Zero probe.

(I.e., the machine will not auto-zero to the mill stick-out.)

Response: Stop! Raise the mill off the button and remove the probe.

Cause: Limit sensor is off line.

Fix: Quit Mach3, shut off the PC, shut off the router, restart as above (PC, Mach3, router)

(If **Auto Tool Zero** fails again, move the mill to a spot where you can touch it to your Zero Z plane (the spoil board or the top of your material).

Bring the mill down to about 1/4" above the Z Zero plane using **Page Down**.

Put a regular (not thick) sheet of paper under the mill.

Lightly tap the **Page Down** key repeatedly, until the paper just fails to slide out from under the mill. Click to highlight the green panel next to **Zero Z** in Mach3. Type in **0.003** (the thickness of the paper), and press **Enter**.

The panel will reset to 0.003.

The router is zeroed to the current mill stickout, i.e., **Z Zero**

The above procedures will zero the machine to your material and your current mill.)

Problem: The router plunges through your material and cuts directly into the spoil board.

Response: Stop! Raise the mill!

Cause: You **zeroed the mill Z** to the **top** of your material in **VCarve**, but to the **spoil board** at the **CNC router**.

Fix: Zero the mill **Z** to the proper surface. Replace the material. **Rewind** the GCode and restart the toolpath.

Problem: The mill breaks.
Response: Stop! Raise the mill!
Cause: A break usually means either the feed is **too fast** or speed is **too slow** or you are cutting too greedily & **too deep**.
Check that you didn't crash into a screw or clamp.
Check that you didn't hit a knot if you are cutting wood.
Fix: Correct feed, speed, path, pass depth (or number of passes) or material!
Replace mill. Zero to mill stickout. Rewind the GCode & restart the toolpath.

Problem: You smell or see smoke.
Response: Stop! Raise the mill! Move it away!
If there is **fire**, **RACE** (see below)!
If there is **smoke**, clear the chips from the path, but keep them on the table.
If the smoke persists, check the dust collector box. **RACE**!
Cause: Rubbing (spinning without cutting): feed and/or speed is **too slow** or **chips are not being cleared** fast enough.
Fix: If the smoke clears when you move the chips, use the ShopVac® to manually assist the DC as it clears chips while the toolpaths are cutting.
        Rewind the GCode and restart the toolpath.

Fire Safety - RACE

Rescue others & yourself - yell "Fire! Tell others to clear out; leave.

Alarm   Call 911 - from outside if it's a large fire!
           Stay to talk to Fire Dept. & report any missing persons

Contain if it's a large fire; close the doors as you leave.

Extinguish if small (and you've trained with an extinguisher) or

Exit if the fire is large.

Catch These Problems Early: Do Not Leave the Machine While It's Running!

Clean Up Any Mess!
When the project is complete, chop the tabs (mallet & chisel) and remove your project.
Remove your material.
Remove all hold-downs.
        To leave a smooth spoil board for the next person, pull out or sand down any plastic nail remnants sticking out of the spoil board. **Clean up the chips** on the table and *floor!*

*Please do **not** use an air gun to blow chips off the spoil board!*
*Metal shop chips fall to the floor. **Wood chips & sawdust float fer-ever!***

Return the collet to the tray and the collet nut to the left table rail, near the wrenches.
Quit **Mach3**.
Power down the **router** (white switch), **dust collector** and PC.
Empty the **DC box, if needed!**

Please remember to **power down the air** Lever **compressor** (in the back left corner of the metal   shop)!

      (The lever points ~ up for off,
~ 30º below horizontal down for on.)Off

              On

To be **certified** to use the Hive 1 3 CNC router, you will need to take the Hive 1 3 CNC Intro Course and then arrange for a Hive 1 3 **CNC machinist** to observe you running a project. Post your proposed time & date on the hive13.org website mailing list page to ask for coverage.

Acknowledgements

Screen shots for Feeds and Speeds… from GWizard Software by CNCCookbook; download at https://www.cnccookbook.com/index
Screen shots for CAD / CAM from Vectric's VCarve; download at vectric.com
Screen shots for Set Up and Run Your Project from Newfangled Solutions' Mach3; download at machsupport.com

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*)

GWizard

CNCCookbook cnccookbook.com/GWizardGWizardInstall1SC.html (& info on all aspects of CNC milling)

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.

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

Intro to end mills: http://makezine.com/2014/09/10/endmills/
Basic assortment ("carbide" lasts longer):
      1/2" & 1/4" 2 flute upcut end mills (for profiles & pockets in hardwood and plywood)
          (also: 1 flute for Al or plastic, downcut or compression for plywood or laminates)
      1/2" 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