

LEAD CNC Preflight Checklist

- ☐ Plug in the Lead CNC
- ☐ Open "OpenBuilds Control" on the Lead CNC computer
- ☐ Connect using "/dev/ttyUSB0"
- ☐ Clear the alarm
- ☐ Verify nothing is in the way of the spindle's travel to the Machine Home Point.
- ☐ Press "*Home All*" to home the machine.
- ☐ Align your work material to the machine axis Ways to align work material:
 - Option A - Use a Right Angle Framing Square from the Wood Shop
 - Option B - Carefully run the spindle back and forth along one edge of your work piece until it is parallel to axis of machine.
- ☐ With the reusable screws in the drawer, secure work material (with spoilerboard) to CNC's spoilerboard
- ☐ Take the collet nut and collet. **Blow them out** using the air compressor. Then screw on to spindle.
- ☐ Install the correct tool bit in the spindle
 - ☐ **If cutting completely through** your work piece, make sure the tool bit extends far enough to cut all the way though.
- ☐ Use the **Torque Wrench** to tighten the collet nut onto the spindle with **15 lb.ft.** of force. STOP tightening spindle when torque wrench clicks - **OR TURNS 45 DEGREES**, whichever comes first!!!
- ☐ Release tension on Torque Wrench - **make sure Green Line is visible**
- ☐ Place 30mm head, and torque wrench back in box in drawer
- ☐ Set the Work Home Point. See "Setting Work Home Point"
- ☐ Load your gcode file by clicking *Open G-CODE > Open GCODE from File*
- ☐ Manually move the spindle over top of each screw that you used to secure your work piece. While there, look at the spindle's position in the 3D display of your gcode in OpenBuilds Control. Make sure the spindle's path does NOT intersect ANY screw.
 - ☐ **If the spindle's path overlaps a screw, the bit will hit the screw when it runs!!!** Either adjust the screw's position or change the gcode path!!!
- ☐ Say a prayer to the deity of your choice
- ☐ In OpenBuilds Control, click on *Tool On > Tool On 100%*
- ☐ **Wait** for spindle to be completely up to speed
- ☐ Press the "*Run Job*" button in the OpenBuilds Control.

IF S%#^ HAPPENS...

- ☐ Press the red button on the left side of the machine ASAP. Resolve the issue. Twist the red button to disengage it. Restart from top of Preflight checklist. (Note: After re-homing, you may find that your Work Home Point was saved from before the accident.)

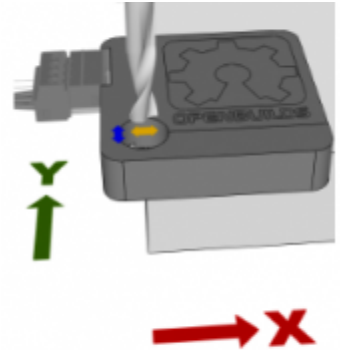
Setting Work Home Point

When you set the **Work Home Point**, you are telling the CNC the location where you want **each axis to be set to zero**. The Work Home Point is the starting point of your design tool paths. It is the location in 3D space that all of your gcode's movements are based on.

There are 2 ways to do this.

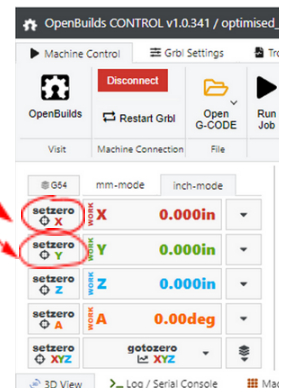
Option A: Use the Touch-off Probe

- ☐ Open the Probing Wizard
- ☐ Select the "XYZ Zero" tab.
- ☐ Place touch-off probe on the bottom left corner of the work piece.
- ☐ Place probe magnet on the collet nut
- ☐ Position the tool over the hole in the probe **as depicted in the image at right**.
- ☐ In Open Builds Control, click the *Troubleshooting* tab along the top of the window. Find the Pin labeled "Probe." Its status should be OFF.
- ☐ Touch the Touchoff Probe to the bit - **Continue ONLY if the Probe's status changes to ON**, otherwise rearrange magnet and try again.
- ☐ Enter the correct Endmill Diameter in mm in the computer window.
- ☐ Press "*Confirm Probe Position*" and then, if ready, press "*Probe*".
- ☐ **Remove** the Touchoff probe AND magnet back to their cradle.
- ☐ Switch back to *Machine Control* tab in OpenBuilds Control.



Option B: Manually

- ☐ Using the manual controls very carefully position the tip of the bit to be exactly over the lower left corner of your work piece.
- ☐ In OpenBuilds Control, click **setzero X** and **setzero Y** buttons.
- ☐ Move the bit up, and position it somewhere over the surface of your work material



Three options to set Work Home Point on Z axis:

Option B1: Manually Set Z zero

- ☐ Manually position the tip of the bit so it is exactly even with the surface of your work material. (Get close using Continuous Jog, then switch to Incremental Jog and use smaller increments.)
- ☐ Click **setzero Z** button.

Option B2: Manually Set Z Zero Using Paper or Foil

- ☐ Move the tip of the bit so that it's extremely close to the surface of your work piece.
- ☐ Switch to *Incremental Jog*
- ☐ Click the box labeled **0.1mm**.
- ☐ Place your small piece of paper or foil on your work piece, underneath the bit.

- ☐ Incrementally lower the bit until you can no longer move the paper or foil around under it.
- ☐ Click *setzero Z* button.

Option B3: Set Only Z Zero With Touch-off Probe

- ☐ Open the Probing Wizard and select the *Z Zero* tab.
- ☐ Place the probe **up-side down** on the work piece under the bit.
- ☐ Move the bit to within half an inch above the surface of the upside down probe.
- ☐ Place the Probe Magnet on the spindle collet nut.
- ☐ In Open Builds Control, click the *Troubleshooting* tab along the top of the window. Find the Pin labeled "Probe." Its status should be OFF.
- ☐ Touch the Touchoff Probe to the bit - **Continue ONLY if the Probe's status changes to ON**, otherwise rearrange magnet and try again.
- ☐ Press "*Confirm Probe Position*" and then, if ready, press "*Probe*".
- ☐ **Remove** the Touchoff probe AND magnet back to their cradle.
- ☐ Switch back to *Machine Control* tab in OpenBuilds Control.

LEAD CNC Shutdown Checklist

- ☐ Unscrew your creation.
- ☐ Return screws to the Reusable Screw bin, in drawer (If any screws are stripped, throw them away.)
- ☐ Place screwdriver bit from drill back in drawer.
- ☐ **Remove collet and collet nut** from the spindle using provided wrenches in drawer
- ☐ **Blow out collet, and collet nut** using air compressor in auto shop.
- ☐ **Apply little bit of oil** to the collet; brush oil on with bottle brush.
- ☐ Place collet and collet nut back in containers in drawer.
- ☐ Hang drill back up in Wood Shop. Put battery on charger.
- ☐ Vacuum up all the chips that were created
- ☐ C'mon... don't half-ass this. Do a good job.
- ☐ Sweep up chips on the floor.
- ☐ Unplug the Lead CNC
- ☐ Lock Lead CNC computer by holding down "Windows key" + "L" on keyboard

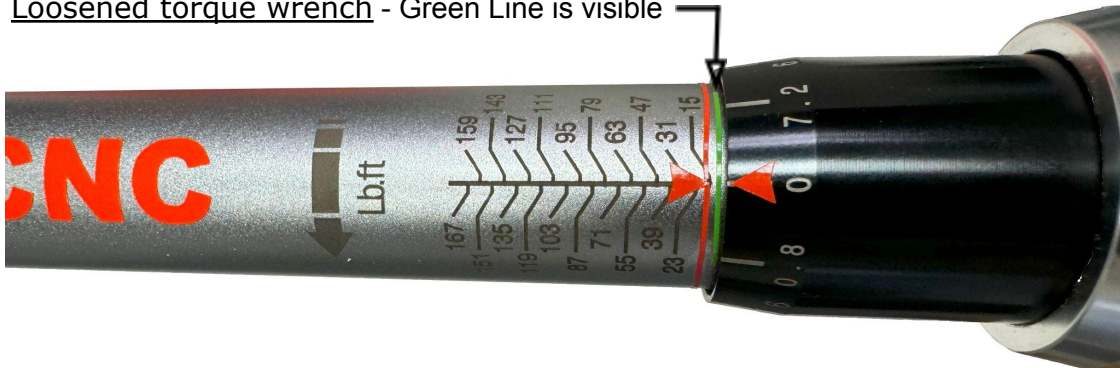


- ☐ Bask in the glory of another job well done.

Using the Torque Wrench

Torque wrenches click once a certain amount of force is applied. THEY DO NOT PREVENT OVERTIGHTENING. You have to stop when you hear the *click*

Loosened torque wrench - Green Line is visible



Torque wrenches must be stored with the torque set to zero

When the Green Line is visible, the wrench is in the zero position.

Therefore,

the Torque Wrench must be stored with the
Green Line VISIBLE

Setting wrench to 15 lb. ft. of force:

- ☐ Pull down the silver ring
- ☐ Twist the black handle to the right 1 complete revolution
- ☐ Let the silver ring spring back up
- ☐ The orange arrows should be lined up, and the green line no longer visible.
- ☐ Attach 30mm head to end of torque wrench

Torque wrench - set to 15 lb. ft.

No visible Green Line. Only Orange Line Visible

