

Instructions for the BNC CNC Mill and the dxf2fgc Conversion Program

Eric Chu
Mathies Group, UC Berkeley
May 21, 2009

1 Introduction

If you're reading this to learn how to use the BNC's CNC mill and you don't concern about what is happening behind the scenes, skip ahead to the section entitled "Operation." This document was rewritten from Will Grover's CNC operation manual, for the new version of FlashCut v3 program.

1.1 Theory of operation

Our application requires precise drilling of holes relative to features etched into glass wafers. Arbitrarily place the etched wafer on the mill and use a microscope to determine the location of the wafer relative to the mill's fixed coordinates, then use this information to offset (rotate and translate) the hole coordinates in the CAD file so that the holes are drilled in the correct places on the otherwise arbitrarily placed wafer.

The process of converting a DXF file (containing circles at the locations of the holes to drill) to a FGC file (encoding all the mill motions necessary for drilling the wafers) may be summarized as follows:

1. By using the drill bit to nick the surface of the wafer in an empty region, the height of the wafer surface in the Z-dimension of the mill's internal coordinate system is determined.
2. By sighting the nick in the microscope, the offset between the drill bit and the microscope crosshair in the XY plane of the mill's internal coordinate system is determined.
3. By using the microscope to sight the locations of two alignment marks on the wafer, the locations of these two marks in the XY plane of the mill's internal coordinate system are determined.
4. The operator provides the locations of the alignment marks in the XY plane of the DXF file's coordinate system.
5. Using the two sets of coordinates for the alignment marks (one set relative to the mill/wafer system and the other relative to the DXF system), a two-part offset (rotation and translation) between the DXF alignments and the mill/wafer alignments is calculated.
6. This offset is applied to the coordinates of each hole to be drilled in the DXF file, thereby converting their locations to the mill/wafer coordinate system.
7. The XY locations of the holes in the mill/wafer coordinate system are converted to "G code" instructions and combined with operations in the Z dimension that encode the "pecking" motions necessary for drilling glass. The resulting FGC file is saved for use by the mill's FlashCut software.

2 Operation

2.1 Wafer preparation

- Attach your glass substrate to a blank 4-inch backing wafer with pine resin.
- Place a blank backing wafer on an aluminum foil-covered hotplate and sprinkle ground pine resin over the wafer's surface. When the resin has melted place the etched wafer on the backing wafer and gently press the bubbles out of the trapped layer of pine resin. Any bubbles remaining at the locations of drilled holes may cause wafer breakage when those holes are drilled
- Scrape excess pine resin off the back of the backing wafer to improve seal on the mill's vacuum chuck.

2.2 Preparation of DXF file(s)

The locations of *circles* in your DXF file(s) are used to determine where to drill holes on your substrate. The radius, layer, and other parameters of the circles are irrelevant to the conversion program - all circles present in a given DXF file will be drilled; other shapes like lines or polylines will be ignored. Remember to remove any big circles representing the outline of your wafer (unless you want a hole drilled in the center of your wafer!). If you use AutoCAD for designing masks, run a purge command to eliminate any empty blocks, layers, etc. (they confuse the dxf2fgc program). Export your mask as DXF files (any of the DXF formats before 2000/R13 will work) and save them to a USB drive. You should have one DXF file *for each drill bit size* that you plan to use, containing only the circles/holes that you plan to drill with that bit. For example, say you have a wafer with 100 large holes and 40 small holes. If you want to drill 50 large holes per bit (using two large bits in the process) and 40 small holes per bit (using only one small bit in the process), you'll need a total of 3 DXF files with names and contents something like this:

1. bigbit1.dxf includes circles at locations of first 50 big holes
2. bigbit2.dxf includes circles at locations of remaining 50 big holes
3. smlbit1.dxf includes circles at locations of all 40 small holes

This may sound like a lot of work but you only have to generate these DXF files once for a given mask and drilling regime. The dxf2fgc program will accept DXF files with a scale of either 1 mm/unit or 1 um/unit. Try to limit the filenames to the traditional DOS "8.3" limits (eight character filename plus .DXF) because the FlashCut program is unable to read longer filenames.

You'll also need to know the DXF file coordinates of two alignment marks from your substrate. For greatest alignment accuracy, the two alignment marks should be as far apart as possible, ideally straight across from each other (left to right across the wafer), and should be clearly visible. Use a Sharpie pen to draw a circle around tiny alignment marks for easy observation under the microscope. Circles for drilling can be used as one or both of the alignment marks, but it can be difficult to align to large circles.

2.3 Starting the CNC mill

- Turn on the two power supply/controller boxes near to the CNC mill. Log on the computer if it isn't already on.
- Run the program FlashCut CNC (shortcut on the desktop). When asked whether the program should establish a connection to the signal generator, click *Yes*, then click *Agree*, then click *Yes* when asked whether to run backlash compensation.
- Check the breadboard around the CNC mill and make sure nothing is obstructing the path of the mill. The screen should look like Figure 1.

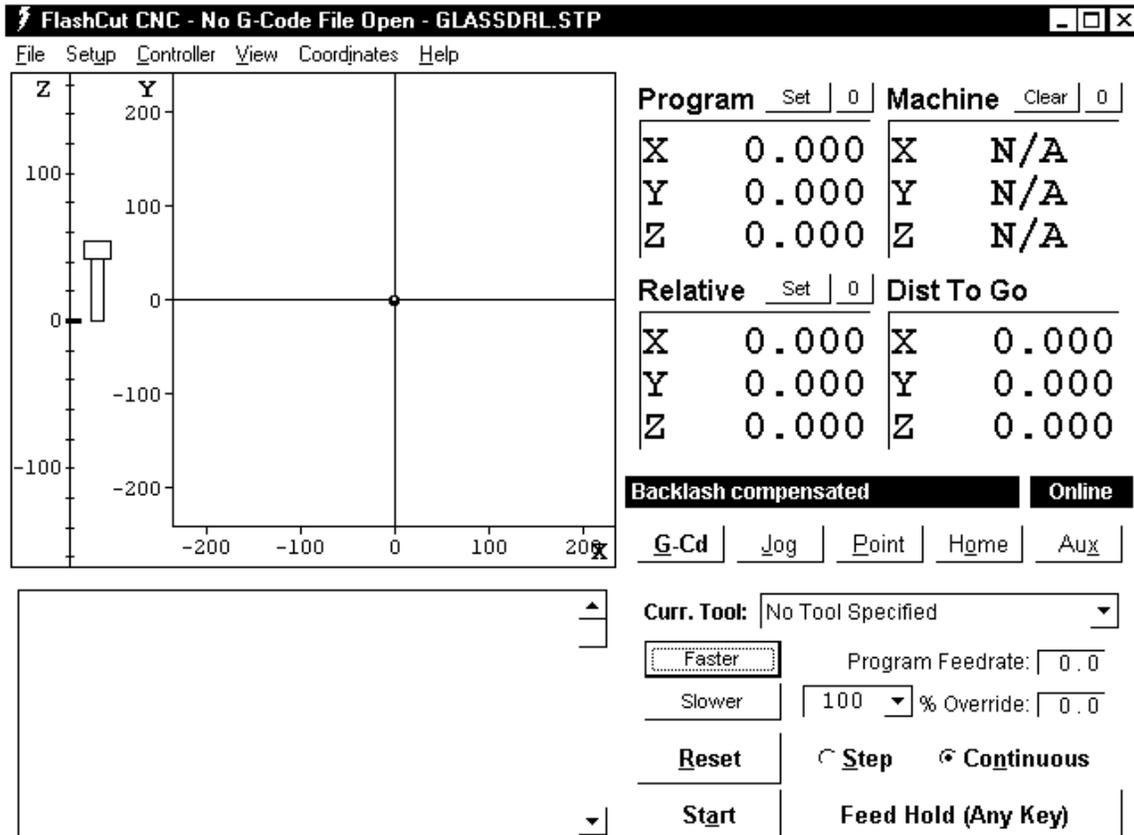


Figure 1: FlashCut program start up screen

- Click *Jog* the three mill axes near to their limits. Do not *Seek home* because the chuck may extend beyond the limit switch and hit the pillar of Z-axis.
- *Important:* when you find the ideal origin location, click the 0 next to the *Program* coordinates to zero the program coordinates (the mill can only move towards the positive from now on). These program coordinates should always agree with the machine coordinates, as shown in Figure 2. You can also zero the relative coordinates so that all three coordinate systems are in agreement.

Program		Set	0	Machine		Clear	0
X	0.000			X	0.000		
Y	0.000			Y	0.000		
Z	0.000			Z	0.000		

Relative		Set		Dist To Go	
X	0.000			X	0.000
Y	0.000			Y	0.000
Z	0.000			Z	0.000

Figure 2: FlastCut program after zeroing *Program* and *Relative* coordinates

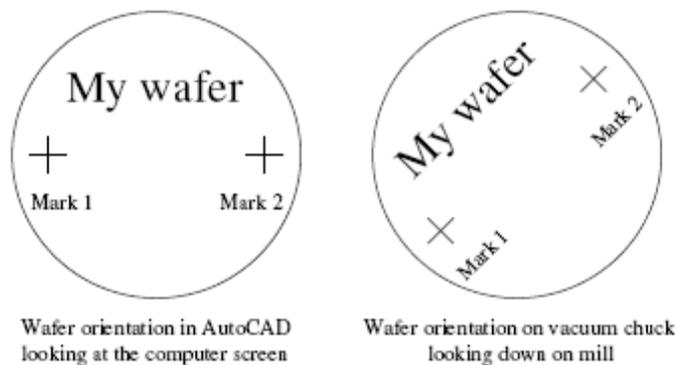


Figure 3: Wafer orientation

2.4 Loading your wafer

Place the wafer on the chuck so that the first alignment mark is to the *left* of the second alignment mark and the line between the two alignment marks makes approximately a 45° angle with the horizontal (x) axis (see Figure 3). The exact angle is *not* critical, but it is important for this angle to be somewhere between 0° and 90°. This is necessary because the conversion program requires that the both the sine and the cosine of this angle be greater than zero. An angle close to 45° will minimize any errors introduced in the conversion process. Now connect the vacuum line to the chuck. You will need to turn on the valve on the wall behind the CNC mill. The wafer should now be stuck firmly to the top of the vacuum chuck. Install your drill bit in the chuck and make sure the bit is centered in the chuck by watching the bit while rotating the chuck by hand.

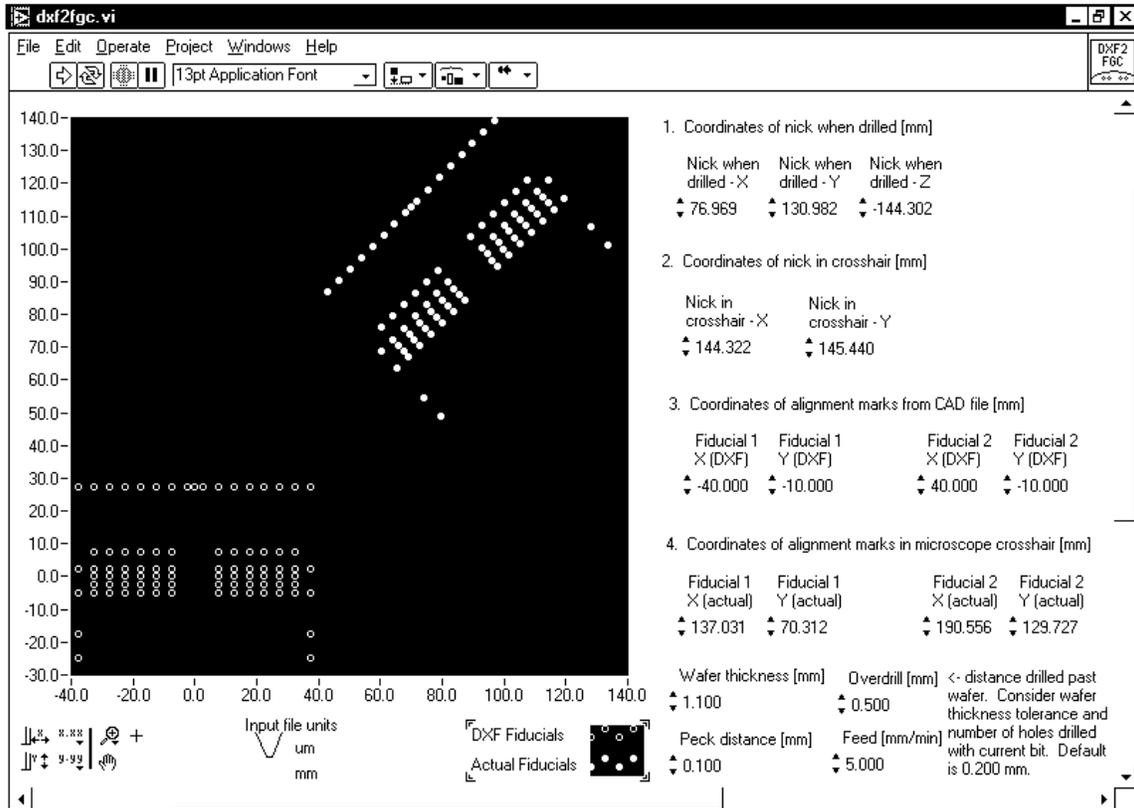


Figure 4: dxf2fgc program after start up

2.5 Converting your *first* DXF file

Start the dxf2fgc program as shown in Figure 4. Hit Ctrl+M to switch to run mode. Converting your first DXF file takes longer than the remaining files because a number of alignment steps must be performed. On subsequent DXF conversions (for subsequent drill bits) only a single parameter must be adjusted; see *Converting subsequent DXF files*.

i. Important: notice that when the mill is in the home position, the vacuum chuck is very close to the pillar that supports the Z axis. When the mill is in this position, *first move the chuck out (+y) clear of the pillar before moving the chuck left (+x)*. Clicking +x when the chuck is at the back position will drive the chuck into the pillar and damage the mill!

ii. Using the *jog* controls in the FlashCut program as shown in Figure 5, *use (+y) first, then use the remaining jog controls to position the drill bit near the surface of the substrate in a region that can be nicked without hurting the final device*. Place a drop or two of milling fluid on the substrate beneath the bit.

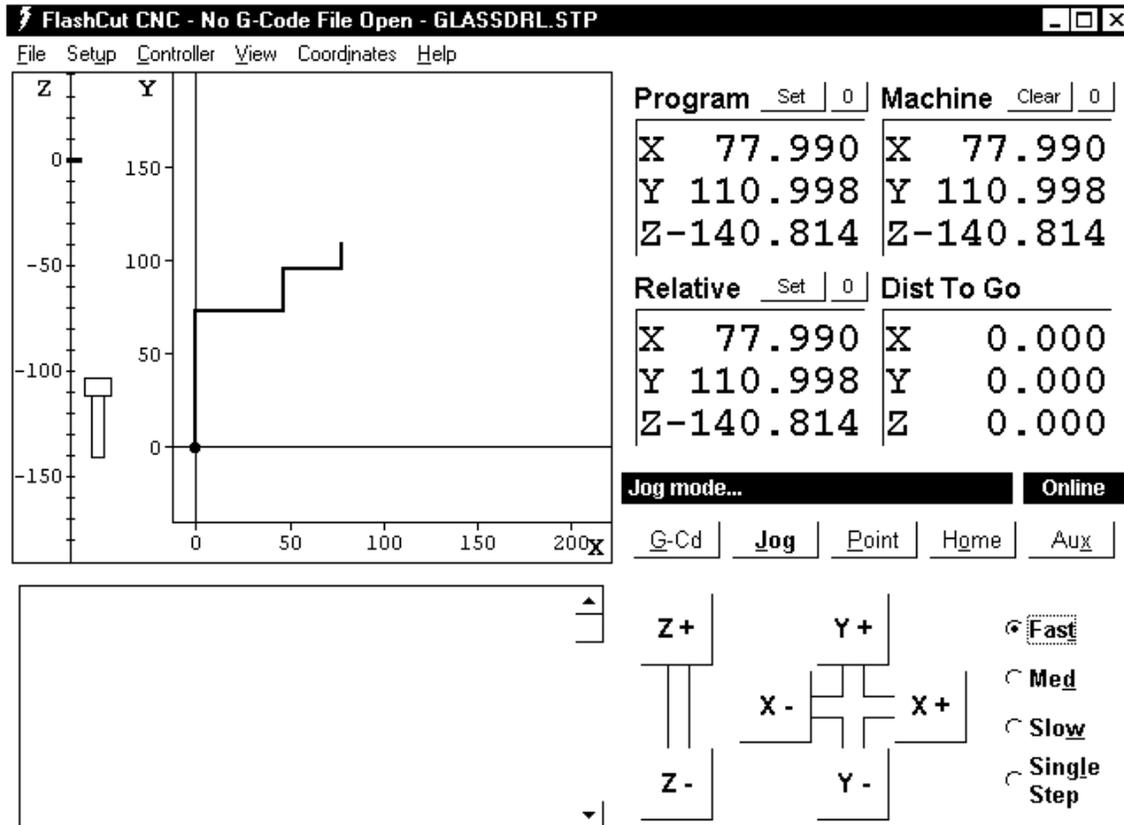


Figure 5: FlastCut program after jogging to nick position. Note: jog in y-direction first.

iii. Turn on the drill motor switch and gently ramp the speed up to the highest setting. Using single mouse-clicks on the *medium* speed, slowly bring the bit down until it enters the fluid drop and *just* touches the surface of the substrate (a tiny plume of glass milk will be seen when the rotating bit contacts the wafer surface).

Immediately turn down the drill speed and turn off the drill, and transfer the current (x; y; z) coordinates to Box 1 (Coordinates of nick when drilled.) on the dxf2fgc program. *Carefully jog the Z axis in the positive direction (up) clear of the wafer before proceeding.*

iv. Start the AMCAP program and place the crosshair in front of the screen. Use the *Jog* controls in the FlashCut program to move the drilled nick to the crosshair. Adjust the focus and position of the microscope as necessary, but *after this step the position and focus of the microscope should be left unchanged.* When the nick is centered on the crosshair, transfer the (x; y) coordinates to Box 2 (Coordinates of nick in crosshair.) on the dxf2fgc program, and *be careful not to bump, change the focus of, or otherwise disturb the microscope from this point on!*

v. Fill in the DXF file coordinates of the two alignment marks in Box 3 (Coordinates of alignment marks from CAD file.) on the dxf2fgc program. These are the coordinates of the two alignment marks as read directly from your original CAD file.

vi. Use the *Jog* controls to move the *first alignment mark* to the crosshair of the microscope. Transfer the (x; y) coordinates to Box 4 (Coordinates of alignment marks in microscope crosshair.) on the dxf2fgc program. Now use the *Jog* controls to move the *second alignment point* to the crosshair of the microscope and transfer these (x; y) coordinates to Box 4 as well.

vii. Set the remaining controls on the `dx2fgc` program to your desired values. Set the *Input file units* switch to either “mm” or “um” depending upon the units of your input DXF file. Specify the *thickness* of the wafer you are drilling. The *overdrill* is the distance drilled on each hole past the substrate and into the backing wafer. Naturally this distance should be more than 0 (to guarantee that the substrate has been drilled completely) but less than the thickness of the backing wafer. A typical value of 200 um should be increased to 400 or 500 um for runs requiring a large number of holes to be drilled with a single bit. Currently we are drilling 0.1 mm per peck with a feed rate of 5 mm/min during drilling and an overdrill of 200 to 500 um.

viii. When all values on the `dx2fgc` program are correct, run the program by clicking the arrow button in the upper-left corner of the screen. The program will ask for the location of the DXF file you're converting; specify the *first* DXF file you plan to drill. The program will then ask you to name the output FGC file; assign a name that associates the FGC file with the source DXF file. The program will also plot both the original DXF coordinates (open circles) and the new machine based coordinates (filled circles) for the holes to be drilled. Make sure that the angle of rotation of the machine holes on the screen resembles the actual angle of rotation of your wafer. You may even want to use the *Jog* controls on the Flash-Cut program to move the bit to the location of a hole on the substrate and check the (x; y) coordinates of this point against the approximate coordinates for the hole read from the plot on the `dx2fgc` program.

ix. **IMPORTANT:** when you're done, don't close the convert program. If you have additional DXF files to convert later, you'll reuse most of the values you entered. If you close the program those values will be lost!

2.6 Running the generated FGC code

Switch to the FlashCut program and select *File, Open G-code*. Select the FGC file that you just generated, as shown in Figure 6. Note all points should stay inside the positive quadrant of the XY window; otherwise, there will be drilling errors. Start the drill motor, gently immerse the regions of the wafer to be drilled in a little puddle of lubricating fluid, and click *G-code, Start* on the FlashCut program. Keep an eye on the mill as it runs and remember that hitting any key on the computer keyboard will instantly stop the mill's motion. If you notice anything strange, *press any key to stop the mill and then stop the drilling motor* before inserting your hands or your head into the enclosure to investigate the problem. To restart the mill after stopping mid-program, restart the motor and then click *start* on the FlashCut program.

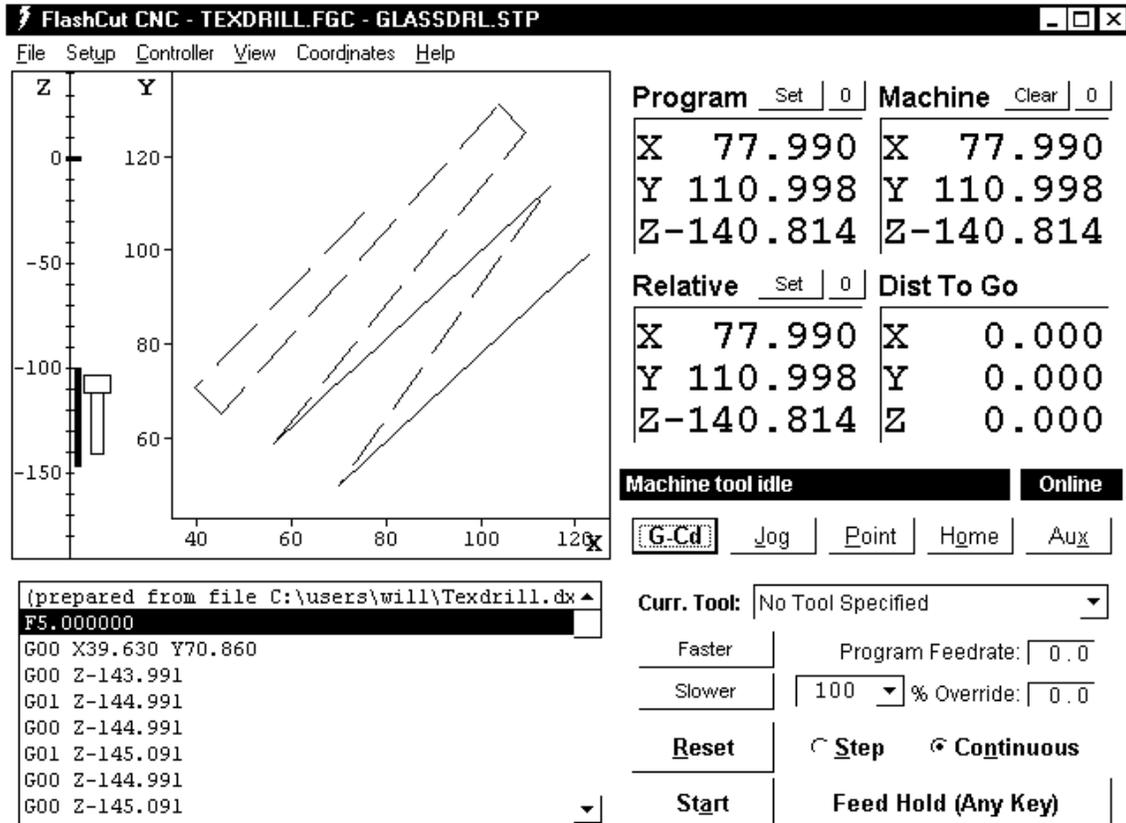


Figure 6: FlashCut program after running the generated FGC file

2.7 Converting subsequent DXF files

Once you have finished drilling the holes in the first DXF/FGC combination, if you have remaining holes/bits, change to your next bit and use the FlashCut *Jog* controls to nick the wafer again. Turn off the drill with the bit in contact with the wafer and this time *transfer only the Z coordinate* to Box 1 of the convert program. Then back the drill off from the surface (+Z) as usual. All other coordinates on the program should still be correct from the first DXF conversion. Now run the *dx2fgc* program on your second DXF file and you'll create your second FGC file, ready to run in the FlashCut program. Continue this alternating process until all your DXF files are converted and all your holes are drilled.

2.8 Shutdown

Remove the lubricant from your wafer and detach the wafer from the chuck. For good karma, move the chuck back to your origin by *Point* - $x,y,z = 0,0,0$. Close the FlashCut and *dx2fgc* programs but leave the computer on, then turn off the two power switches on the CNC mill control boxes and you're done!

3. Note

3.1 Mill maintenance

We lubricate the mill's leadscrews and axes every 500 holes drilled (which happens pretty quickly). Here is the protocol:

1. Home the mill and zero the *program* coordinates.
2. Wipe away any visible gobs of old lubricant from the leadscrews and axis surfaces. The X-axis is inverted and difficult to reach, but a quick wipe beneath the axis with a Technicloth should suffice.
3. Apply a small amount of lubricant onto the leadscrews and the axis surfaces.
4. Jog the three mill axes nearly to their limits.
5. Wipe away any excess lubricant from the leadscrews and axis surfaces.