CNSI Microfluidics Lab - A simple approach for drilling holes in glass slides
Dave Bothman – DRAFT 14 May 2014

Note: Operating the Flashcut/Sherline CNC Mill requires some familiarity with the Flashcut software user interface. If you are new to this tool there is a good tutorial available in section 5 of the Flashcut manual, and is also available on the Microfluidics Lab website at:
http://microfluidics.cnsi.ucsb.edu/tools/CNC_drill/FlashCut%20CNC%204.5%20Tutorial.pdf

Edit the sample program for the holes that you want to drill

1. The program below can be edited to drill your hole pattern. Simply change the X,Y coordinates to the coordinates that you want drilled. Add or remove G73 lines as needed. (The sample program is available for download on microfluidics.cnsi.ucsb.edu)
2. The program origin is the lower-left corner of the slide, so all of the x,y coordinates should be positive.
3. Notes on the syntax
   3.1. (Comments are in parentheses)
   3.2. The setup blocks configure the drill correctly for drilling programs
   3.3. The G73 command drills a hole
      3.3.1.1. X is the x coordinate of the hole
      3.3.1.2. Y is the y coordinate of the hole
      3.3.1.3. Z is the depth of the hole measured from the top of the slide. Z+ moves the drill away from the glass, and Z- moves the drill towards the glass
      3.3.1.4. Q is the "peck" distance – the depth drilled before the drill retracts to let coolant in
      3.3.1.5. R is the height above the surface that the drill moves between holes
      3.3.1.6. F is the feed rate in mm/min
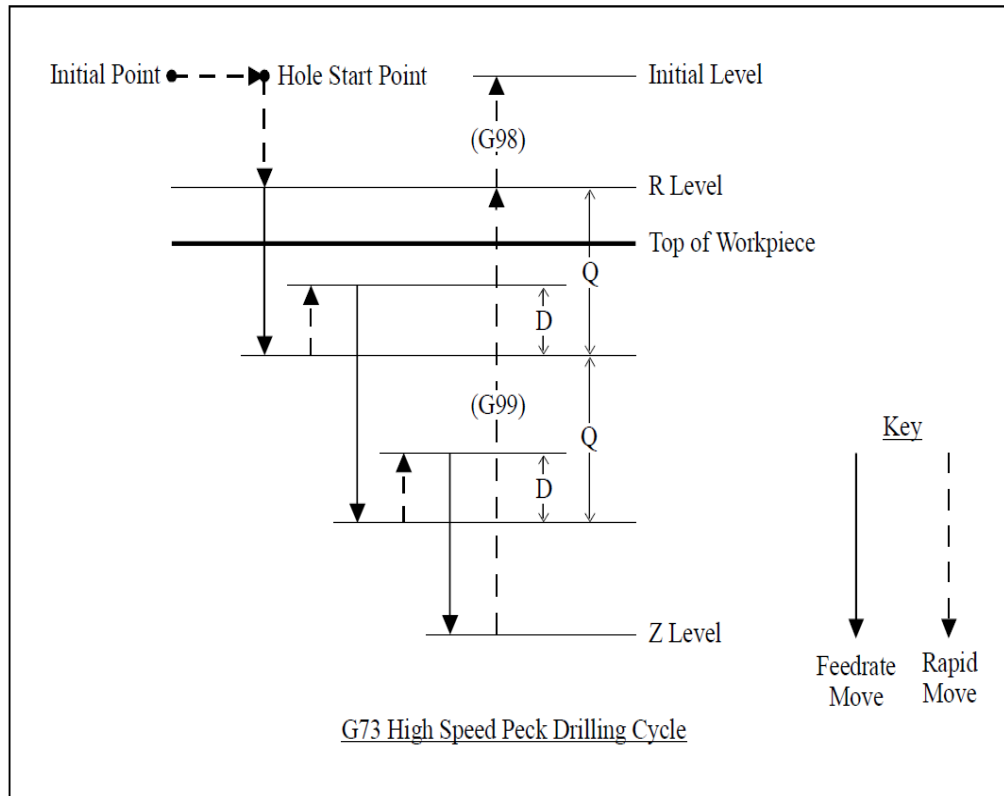4. Save the file as a TEXT file - .txt in word, or edit it in notepad.

Figure 1- Illustration of G73 Command

```
(Sample program to drill six holes on the Microfluidics Lab Drill)
(Dave Bothman, 2 July 2012)
(Change the x,y coordinates of the G73 commands for your hole locations)
(Setup blocks)
G21 (select mm units)
G49 G80 G90 (cancel offsets, cancel cycles, absolute coordinates)
G98 (Move the drill away from the part when drilling is finished)

G00 X0 Y0 Z5 (move to the lower left corner of slide, 5mm above)

(Hole drilling blocks)
G73 X22.5 Y12.5 Z-1.2 Q.1 R0.5 F5
G73 X29.5 Y4.5 Z-1.2 Q.1 R0.5 F5
G73 X45.5 Y4.5 Z-1.2 Q.1 R0.5 F5
G73 X52.5 Y12.5 Z-1.2 Q.1 R0.5 F5
G73 X45.5 Y20.5 Z-1.2 Q.1 R0.5 F5
G73 X29.5 Y20.5 Z-1.2 Q.1 R0.5 F5

(Closing block)
M30
```

Prepare the mill to run your program

1. Start Flashcut CNC and Centre CAM programs (desktop icons)
2. In Flashcut home the axes: **Home Tab, Seek Home**
3. In Flashcut jog the axes so that the spindle is about 3cm above the approximate center of the vacuum chuck: **Jog tab, X,Y,Z, fast, med, slow** as appropriate
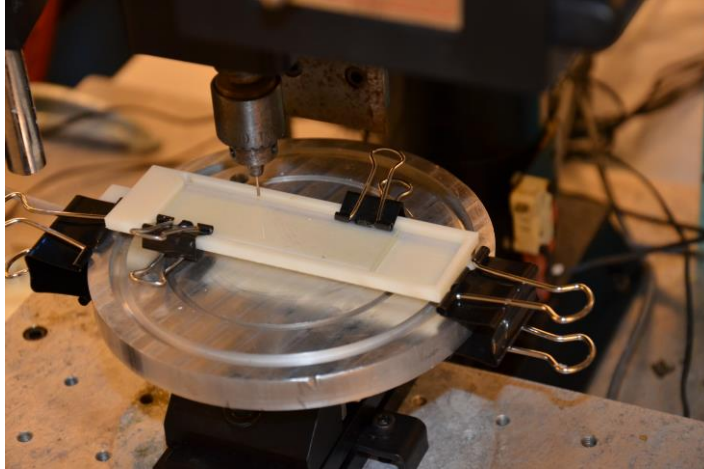4. Attach the glass slide drill chuck to the vacuum chuck using binder clips (figure 2)

Figure 2 - Microscope slide chuck clamped onto the Microfluidics Lab CNC Mill

5. Align the chuck with the bed of the mill using the camera
    5.1. Jog the chuck under the camera and align the cross-hairs with one edge of the chuck (figure 2)
    5.2. Jog back and forth along the edge, adjusting the angle of the chuck until the cross-hairs stay on the edge.
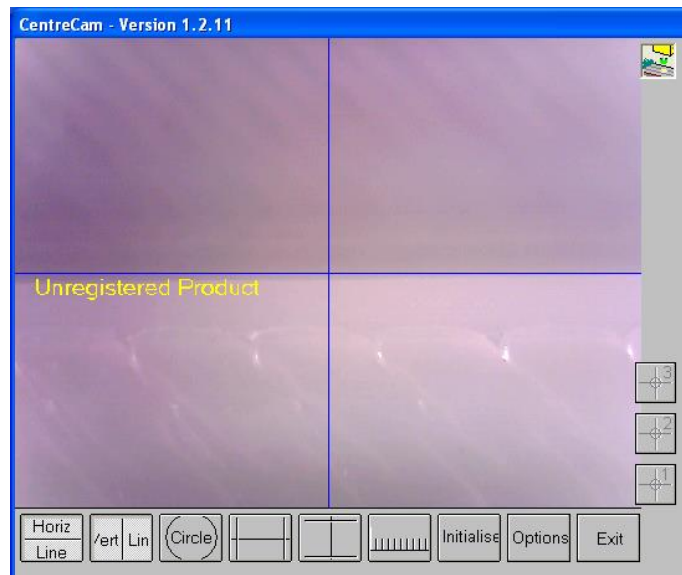

Figure 3 - CentreCAM screen shot showing alignment of chuck with cross-hairs

6. Clamp the slide that you plan to drill, and a backing slide in the chuck using binder clips. Push the slides to the lower left-hand corner. It's not necessary to bond the backing slide, but if you want to use Crystalbond, set the hot plate to 121C.
7. Put the pointy-tipped scribe in the drill chuck and align it visually with the lower-left hand corner of the slide. (figure 4)
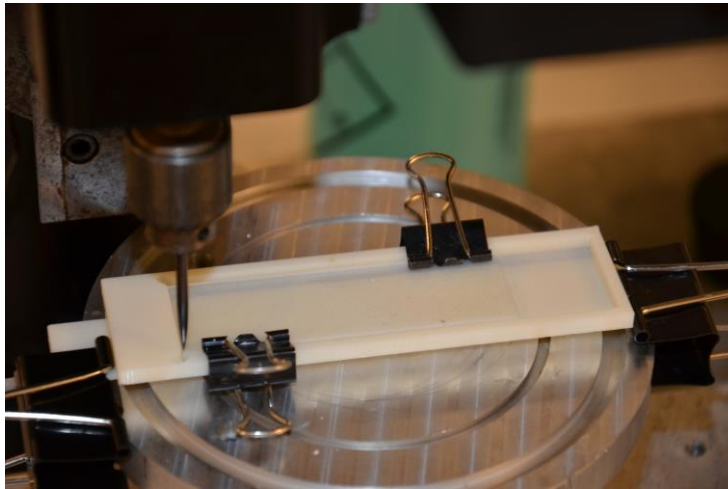
Figure 4 - Using the pointy-tipped scribe to approximately locate the lower left corner of the slide

8. Set the **program X and Y** coordinates to zero (figure 5)
   8.1. Select *Set (in the program coordinate readout)*
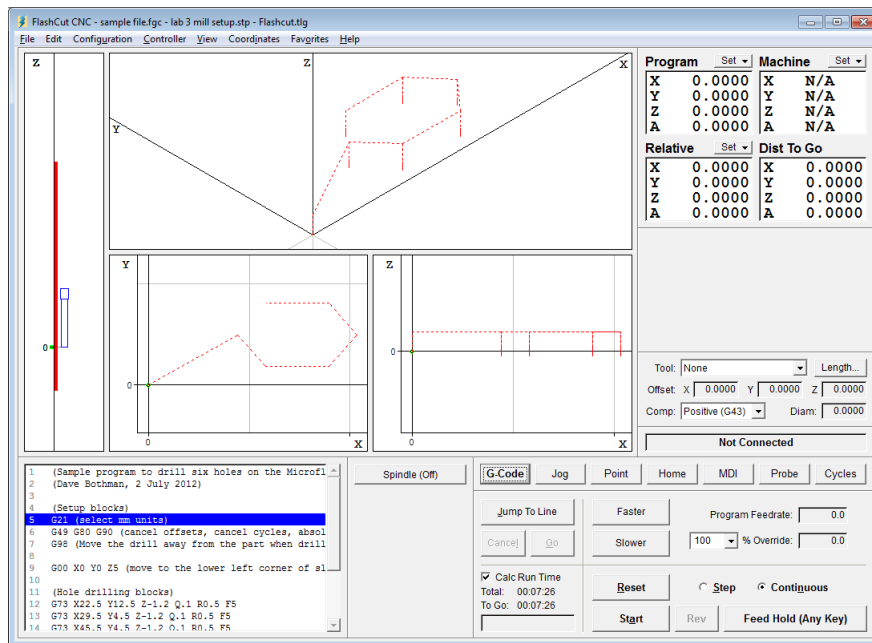   8.2. Select *Zero X and OK, repeat for Y*

Figure 5- Flashcut CNC Screen Shot – Program DRO is in the upper-right hand corner

9. Remove the pointy-tipped scribe and put in the drill that you want to use. **Make sure that the drill bit is centered, sometimes it gets caught between chuck jaws. You can see if the drill is centered by rotating it by hand.**
10. Tighten the chuck just enough so that you can still slide the drill bit in and out of the chuck.
11. Jog the drill until it just touches the glass slide. **THE DRILL MUST BE LOOSE IN THE CHUCK AT THIS POINT - BE CAREFUL NOT TO CRASH THE DRILL INTO THE GLASS – IT'S BAD FOR THE MACHINE, THE DRILL, AND YOUR PART!**
12. With the drill bit touching the glass tighten the drill chuck.
13. Set the program Z coordinate to zero (see step 5 above) **Congratulations – you've set the origin of the CNC mill to match the program!**

Run the program

Safety note: The mill, like all automated machinery, can be dangerous. When using the mill:

- **Make sure to keep your body away from areas where you could be pinched or cut.**
- **Wear safety glasses at all times**
- **Tie back your hair, tuck necklaces or anything that could get caught in the machine into your clothing.**
- **Remove finger rings.**
- **Keep the work area clean and neat.**
- **Pressing any key when a program is running will stop all of the axes**

1. Start Flashcut
2. Load your program
   2.1. File, open G-code
   2.2. Select your file
       2.2.1. The code should be visible in the lower left-hand corner
       2.2.2. **Check to make sure that the code is correct**
   2.3. The graphics window should show the drilling pattern – dashed lines are rapid moves, solid lines are drilling moves.
3. Cover the slide with coolant – the drill will be ruined quickly if it runs dry
4. Turn on the spindle and ramp up to top speed (~10,000 rpm)
5. Select **step** in the lower left corner of the Flashcut screen – this will advance the program by one line at a time.
6. Press **start** and advance the code to the G00 X0 Y0 Z5 line
   6.1. You will get an error message warning you that the program coordinates are not zeroed – start anyway? **Press Yes.**
   6.2. The axes should move so that the drill is 5mm above the glass at the lower-left hand corner of the slide. If it isn't there is a problem with your code.
7. Press **start** again until the first G73 line and drill the first hole.
8. If that works fine, **select continuous** and **start** to drill the rest of the holes automatically
9. Add coolant as needed.
10. Ramp the spindle speed down and turn the spindle off when you're done.
11. In the log book record the number of holes that you've drilled. This helps us plan periodic maintenance.

Clean up

1. Use the suction hose to suction the coolant off of the slides
2. Remove the slides from the chuck.
3. Drill any additional slides.
4. If the drill bit is still good note the number of holes that you've drilled and save it where you can use it again. If it's worn out throw it out in the red sharps bin.
5. Use the suction and paper towels to clean the mill.
6. Close open windows on the computer and turn
7. Put supplies away and tidy-up.