MARYLAND MICROROBOTICS LABORATORY

Milling Standard Operating Procedure

Alexi Charalambides
alexigc1@gmail.com

July 18, 2016

Motivation

When producing molds for elastomer skins, milling is a highly favorable approach over other conventional techniques, such as 3D printing and clean room processing. 3D printing is quick and can easily interpret .stl files from traditional CAD software, but lacks single micron resolution in X, Y, and Z, and produces rough surface finishes. 3D printed materials also release a gas when heated that inhibits curing of polymers. Clean room processing can achieve single micron resolution, but is limited to a 4” silicon wafer, and is extremely resource intensive with respect to both time and money. Milling achieves the best of both worlds, by enabling microscale features over a large area while having a low barrier-to-entry.

Overview

NC (numerical control) code is a low-level machine language that tells the endmill where to move. When milling complicated geometries, writing NC code from scratch is impractical. Meanwhile, milling from .stl files often produces long cut times, or fails to automate a path that captures microscale geometries. Using a freeware software, like CAMotics, tool paths can be programmed at a higher level using Tool Path Language (TPL) and then exported as NC code (also known as g-code). However, when NC code is exported from CAMotics, the format of the gcode isn't quite compatible with the Roland machine in the Robotics Realization Lab. Therefore, additional formatting is necessary before it can be read from the machine. Formatting includes placing decimal points on every value except spindle speed (but no more than 4 decimals), changing G0 and G1 to G00 and G01, and correctly formatting code for specialized operations like tool changes.

Once the NC code is prepared, you're ready to mill. Molds are frequently milled out of acrylic since it's clear (easy to tell if the mold is refilled with conductive elastomer), cuts well, is inexpensive, is sufficiently planar over a large area, and doesn't warp when subject to 80 C to cure PDMS. Acrylic also laser cuts well, and is typically laser cut into smaller pieces (ex: 12" by 4") to save material and fit inside the milling machine (which can accommodate a stock up to about 14" by 14"). The endmills are typically between 50 microns and 400 microns in diameter and are purchased from Microcut USA. Different sizes are cut at different speeds, for example, a 100 micron endmill is cut at 6 mm/min and 12000 rpm, and a 400 micron endmill is cut at 40-100 mm/min and 8000-10000 rpm. The 50 and 100 um endmills tend to fatigue and break after about 1-2 hours. Endmills are mounted on chucks which are placed inside the milling machine's magazine (can hold 4). This makes changing tools during cutting easy. Before executing a cut, the origin needs to be defined by manually moving the endmill, and the stock needs to be leveled. A leveling procedure was developed called, "gradient alignment marks", which cuts four trenches of varying depths, 0 um, 100 um, 200 um, and 300 um deep, in each corner of the stock. Then, based on the result of the cut the stock can be adjusted, typically in angle increments of 0.05 degrees or Z increments of 10-100 um, and the code can be run again until three trenches are visible in each corner.

From start to finish, the process flow looks like: design in CAMotics, format g-code, place endmills in magazine and measure z-offset, mount the stock, level it and set the origin, and execute the cut. After the cut, the mold is ready to be refilled with a conductive elastomer, and 7 wt% CNT/PDMS is commonly used. Once perfected, this manufacturing process from start to finish (from designing in CAMotics to peeling the elastomer skin from the mold) can be accomplished in less than a day.

Creating NC Code

1. Write TPL in CAMotics (highly recommended tutorial: http://tplang.org/)
2. CAMotics > Export > Gcode, metric
3. Change file ending from .gcode to .txt
4. Open with Notepad++
5. Replace…
    a. “X” with “X “
    b. “Y” with “Y “
    c. “Z” with “Z “
6. Copy all (Ctrl+A, then Ctrl+C)
7. Open Excel, paste special
8. Format columns C, E, and G to “Number”, Decimal = 4
9. Copy all (Ctrl+A, then Ctrl+C), and paste back into Notepad++
10. Replace…
    a. “   “ with “”, you can double click on a “tab” copy it since you can’t type a tab
    b. “G0” with “G00”
    c. “G1” with “G01”
11. Format headers, tool changes, and feed rate or spindle speed changes
    a. Header:

%	(start data feed)
G90	(set absolute coordinates)
G21	(set metric [all units in mm])
G54	(set coordinate system)
G00Z1.0001	(rapid move to h_safe)
F6.01S8000M03	(set feed rate and spindle speed, turn on spindle)

    b. Footer:

M05	(stop spindle)
M02	(end of program)
%	(end data feed)

    c. Tool change (if applicable):

M05	(stop spindle)
M06T4	(change to tool 4, corresponds to magazine number)
S12000	(set spindle speed)
M03	(turn on spindle)
G04X5.0	(pause for 5.0 s)
F6.01	(set feed rate [mm/min])
G00X0.0Y0.0	(rapid move to origin, where Z is still high)
G00Z1.0001	(rapid move to h_safe)

Milling with Roland Machine

1. Turn on machine
    a. Open air valve (blue hose on wall behind machine)
    b. Turn on power (located to the right of the emergency stop button)
    c. Open “VPanel” software > OK > set “NC code” > press green Enter button on machine
2. Prepare tools
    a. Move tool > Move to view
    b. Use wrenches to change endmills
    c. “Register tool” in magazine
    d. Calibrate Z offset
3. Grab your starting tool (typically 82016). Attach/detach > select tool: # in magazine > Replace
4. Mount stock, set origin, and level
    a. Move tool > Get Current Coordinates > Rotate to 357.75
    b. Affix stock to the clamp on the left, then cone on the right. Adjust position of cone using hex key if needed.
    c. Manually move endmill to approximate origin (corresponds to origin used in TPL)
    d. Base point > Set “XYZ Origin” > Apply
    e. Cut “gradient alignment marks” code
    f. Adjust origin, and move X over by at least 2.5 mm
    g. Iterate until 3 trenches appear in each corner
5. Cut > open file > Output. Press big red button if something goes wrong.
6. Clean up
    a. Attach/detach > Return tool
    b. Remove stock from machine and air gun it. Inspect under stereoscope.
    c. Close VPanel and log off
    d. Turn off Roland power
    e. Close air valve

Debugging

"The endmill moves very high during milling and doesn't cut."

The endmill reached an internal bound in either the X or Y directions. Reposition your origin.

"The tool path that it's cutting isn't what I coded."

Your code either has more than 4 decimal points somewhere in it, or it's missing a Z move to h_safe.

"VPanel isn't responding to anything I'm doing, it's all grayed out."

Shut the milling machine hatch. If it's already closed, physically push on the red X to make sure the latch is in good contact with the rest of the machine. Don't place objects inside the latch because it could damage it.