Altium PCB mill Workflow

* In PCB, File->FabricationOutputs->Gerber: get Gerbers of necessary files (TopLayer, BottomLayer)
* In PCB, File->FabricationOutputs->NCDrill: get the drill outlines for the holes. Select option to include board outline
* Then open these in CAMtastic
* Then for each file, File->Export->BMP
  + Set DPI=1000
  + Confirm that size is correct
  + Monochrome
* Convert to PNG online

In the PCB mill software, the tool follows the edge of the black and white. If you want to remove all of the copper, you need to set the “offset” parameter to 0 (which works as infinite). Otherwise, an offset of 1 will only separate the copper by one tool width.

For Traces:

* White = Remaining Copper
* Black = Removed Copper

For Edges and Holes:

* Black = holes
* White = left alone

In Altium:

* Design Rules:
  + no copper within 1/64” = 0.015625”
  + no holes smaller than 1/32” = 0.03125”
  + May need to cheat in the MIT software (set bit smaller than actual size (maybe 0.01”) otherwise small features will not get cut at all)

Lessons Learned from 1st attempt:

* + Need a better method of lining up the part.
  + Get the black & white correct
  + Prefer 2” x 3” circuit boards if possible (many in stock)
  + Flip the BottomLayer’s x axis
  + Set offset very high/0
  + For cutting traces, trick the machine into working smaller features by setting the tool diameter to 0.01” for the 1/64” end mill
  + In Altium, need to set the polygon pour connections to traces so that the features are still quite large, it prevented proper milling without it.
  + 20mil traces are okay but wouldn’t want to go smaller.
  + Need to set the outline/hole cutting depth deeper than 0.16” (didn’t go through all the way)
  + All of the through holes except for the TO220 are too small.
    - 6 pin header
    - Relay
    - Pogo Pins
  + The TO223 pads have 43 mil holes and 67 mil pads:
    - Use 45 mil holes and 70 mil pads where possible
  + Need to design bracket mechanism to keep the board aligned through all of the steps
  + The FR-1 material used with the mill is much more flexible than the FR-4 I was planning on. It will require reinforcement.
  + The NC drill file is 64 pixels wider than the TopLayer and BottomLayer because of the board outline width. Need to shift the TopLayer and BottomLayer x and y by +64/1000” or 0.1872 mm. For example, if the NC drill origin was (30, 20, 0), the Top and Bottom layers origin should be (30.8128, 20.8128, 0).
  + Make the BottomLayer features as large as possible to allow for misalignment when flipping the board.

Notes on the rev01.1 test 2

* Can still go larger on pads for the through hole parts (especially the Relay)
* Some of the trace edges are a bit rough on the edges, much more so than the first attempt
* The pads for the SOIC20 are not clean
* Probably need to sand down the top, which may clean it up (the bent edges might be reflecting the light to look worse)
* The board didn’t cut the full depth over the full area, need to either deepen the cut or flatten the board.
* Can move the ground plane even further away