Fabrication of Printed Circuit Boards Using a Table Top CNC Mill
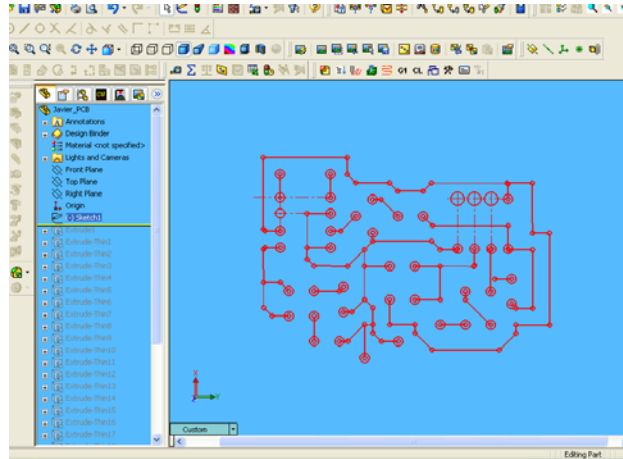
by

Christopher F. Wise
Departmental Engineer, New Mexico State University,
Department of Engineering Technology and Surveying Engineering
PO Box 30001, MSC 3566, Las Cruces, NM 88003-8001, chwise@nmsu.edu

Abstract – The purpose of this paper is to examine the fabrication of Printed Circuit Boards (PCBs) using a milling process. Historically, PCBs have been manufactured by a process in which warm ferric chloride is sprayed/rinsed (or another etchant) over the board, removing any undesired copper. In an effort to rid the task of dealing with and disposing of chemicals, an alternate solution was requested. Similar to the more common and well known chemical PCB etch process, the PCB milling process removes copper to create the traces, pads and ground planes, as necessary. However, unlike the chemical etch process, PCB milling is a chemical-free process and can be completed in a typical lab environment without exposure to hazardous chemicals. In the case of PCB milling, the quality of a circuit board is determined by the system's milling accuracy and control as well as the condition (sharpness and temper) of the milling bits and their respective feed/rotational speeds.
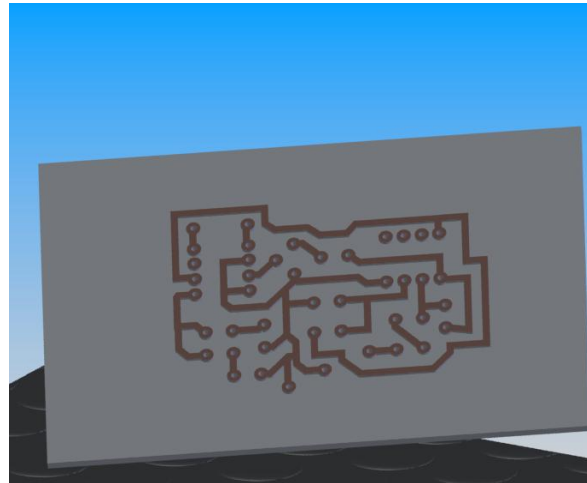
Transferring Layout into 3-D Model

The initial step in the process of PCB fabrication is the layout of the board. Board layout is currently being performed using MicroSim PCB layout software. Once the layout is complete, it is saved as a ".dxf" format. The ".dxf" file is then opened as a "model", not a drawing, using Solidworks ®. Note that any 3-D modeling software can be used, but Solidworks® was chosen due to the author's familiarity with it. When initially opened, the model appears as a sketch of the model which is then traced over to form the electrical traces and pads (See Figure 1). Finishing this process yields a 3-D model. It takes an average of 35 minutes to create the 3-D model using this process (See Figure 2).

The only hindrance to producing a PCB this way is the spacing between any opposing geometry is limited to spacing of 0.032". Some surface mount devices have even smaller spacing limitations which can limit the use of the milling process. There are smaller endmills available but smaller endmills require slower cutting speeds and are difficult to find and work with.
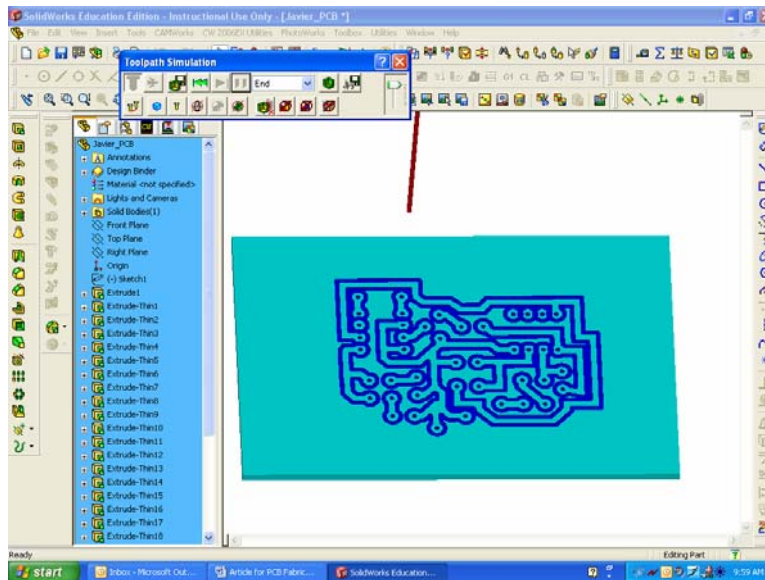
Figure 1 Initial sketch of layout

Figure 2 3-D model of finished board

Generation of G-Code

Once the model is complete, the G-code is generated using CAMWorks (SolidWorks® specific) which generates the code necessary to perform the milling operation which will create the traces and also the drilling operation for the through holes. CAMWorks is a very easy program to use, however, knowledge of machining is necessary in order to properly set up the PCB and ensure proper cutting and feed speeds. Also, CAMWorks automatically selects the cutting tools but must be checked to ensure proper selection has been made.

Once the G-code is complete and all adjustments to speeds and tools have been made, a simulation can be run to ensure that all traces and pads are properly cut and that all holes have been drilled (See Figure 3).

Figure 3 Simulation using CAMWorks

Upon satisfactory verification of the simulation, the code can then be loaded into the micro mill program, MPS2003. The MPS2003 program is a standard universal G-Code interpreter package that was designed with many full size CNC machining features in mind.

Note: the code is saved as a ".txt" file.

Simulation
Before beginning the cutting operation, two simulated dry runs are performed to verify that the interpretation was completed properly. These are as follows:

- Simulation (Screen only)
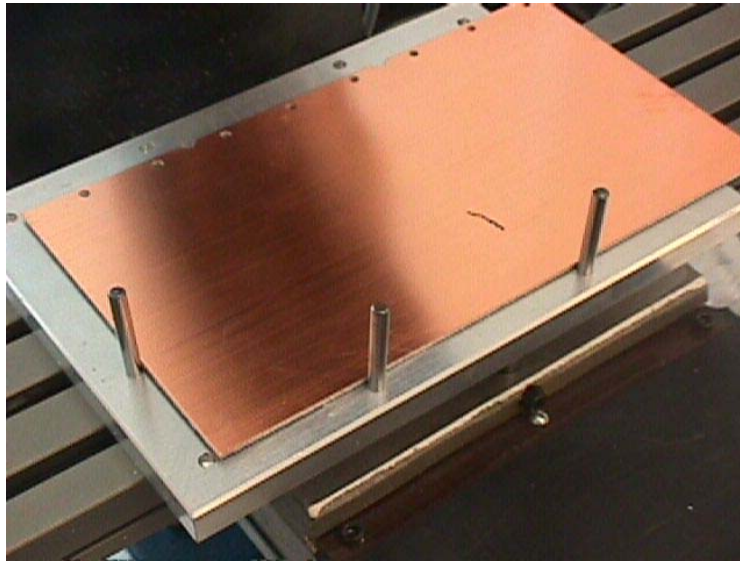- Simulation (CNC, w/out cutting)

There is an option in the MPS2003 software that allows the user to verify the milling operation as a simulation only. This is used to again ensure that the code was properly interpreted before the cutting process begins. Once this is verified, the "Z" axis is set about an inch above the board and the machine is then run through its paces to ensure that the "X" and "Y" axis "zero" are at the proper locations. If the "X" and "Y" locations are not at the "center" of the material then cuts outside the board parameters are possible, resulting in a scrapped board.
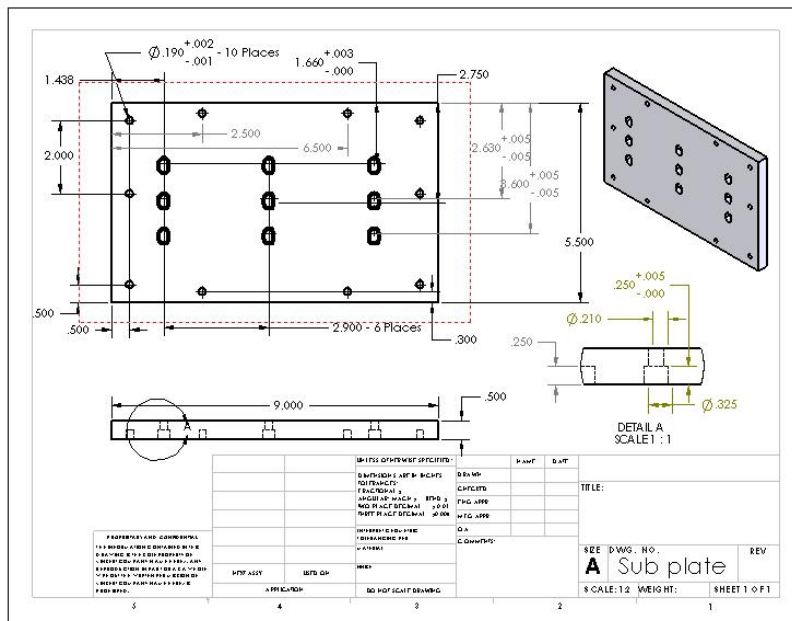
Mounting PCB to Sub-Plate
Once the verifications above have been completed, the PCB material is now ready to mount to the sub-plate. The sub-plate protects the steel table from damage during the drilling and routing process. A double-sided tape manufactured by 3-M for attaching the material to the sub-plate has worked very well in this application (See Table I). Several types of tape from carpet tape to masking were experimented with, but the 3-M was

found to be the most suitable. If the board adheres too tightly to the sub-plate it can be very difficult to remove and thus may be damaged. Alignment pins are used to align the printed circuit board material so that it is square with the "X" and "Y" axis (See Figure 4). The sub-plate was designed and manufactured in-house and is made from 6061-T6 aluminum plate (See Figure 5 for drawing).

Note: Use of a vacuum plate works equally well and eliminates the need to remove the tape upon completion of the board routing and drilling operations.

Figure 4 Alignment pins

Figure 5 Sub-plate drawing

Milling Operation
The Milling operation requires a spindle speed of a minimum of 10,000 RPM in order to get an accurate and "clean" cut. This is one reason that hinders the use of a typical floor mill which would have a maximum speed of about 3500 rpm. Industry routing is usually done at 20-40kRPM. At a slower RPM the material tends to tear/rip as opposed to cut. Also, the feeds and speeds play a big role in the accuracy as well as the life of the end mill or drill bit. A complete equipment list can be found in Table I below.
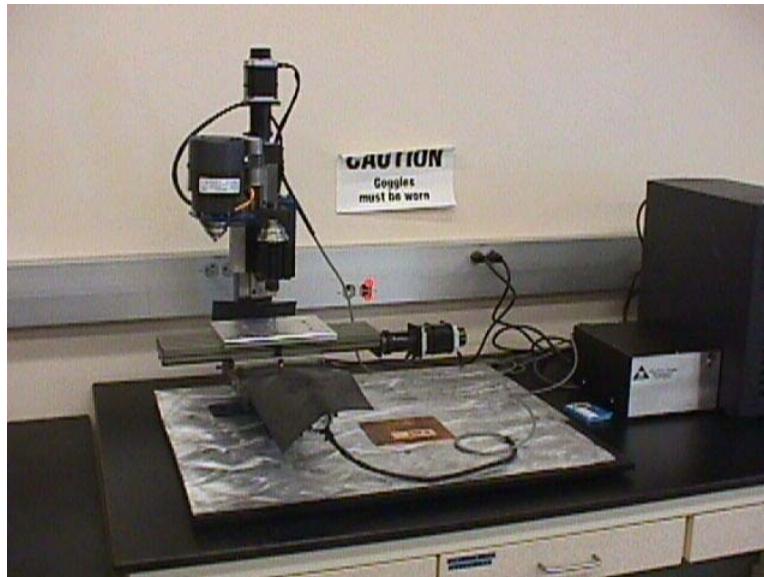
Table I Equipment List

Item	Source	Part /model #	Link
CNC Mill	Taig Tools	Micro Mill 2018	http://www.taigtools.com/mmill.html
End Mills	PMT	ET-2-0312-F	http://www.pmtnow.com/
Drill Bits	Drill Bit City	05WS66	http://www.drillbitcity.com/
Computer	*		
Tape	3-M	Poster Tape (holds up to 0.25 pounds)	
PCB Material	Dalpro	62P1/01212G5 **	http://www.dalpro.net/CopperClad.html

*Capable of running DOS based software
**Copper clad FR-4 laminate/circuit board material, 0.062" thick, 1 sided, 1 ounce

Milling Machine
The selection for the milling process was to use a CNC table top mill, specifically the Micro Mill model 2018 (See Figure 6). The software used to run the micro-mill is DOS based which requires special formatting for newer computers. This formatting is necessary because the DOS language is not compatible with FAT 32 and newer versions. The hard drive will have to be re-formatted for FAT 16 or FAT 12.

Figure 6 CNC Mill, Processor and Computer

Route and Drill Time

The boards that have been produced take approximately 15-20 minutes to route and drill (See Figure 7). Quicker times have been achieved but tend to increase the risk of breaking an end mill or drill bit and that can result in the loss of the board. The drill bit size is 0.033" and the end mill size is 0.032" which are quite small as compared to a typical 0.250" end mill (See Figure 8). After experimenting with different configurations, these sizes work best for maintaining maximum pad size which in turn helps to eliminate the lifting of pads during the soldering operation.

Figure 7 Completed Printed Circuit Assembly, Component Side

Figure 8 Comparison of 0.250" and 0.032" End Mill

Conclusion:

The fabrication of Printed Circuit Boards using a CNC mill versus a chemical etch has worked very well for New Mexico State University's Engineering Technology program. It eliminates the environmental hazards and mess of the chemical etch based process and allows the process to be used in practically any environment. This project also allows electrical engineering undergraduates an opportunity to witness the mechanical side of the fabrication process. The waste produced through the manufacture of PCBs on an industrial scale, through the use of etchants and chlorides, has been blamed for serious pollution in the North Sea (North coast of England and Scotland). Within the last 15 years these pollutants have been blamed for the decline in fishing stocks and in particular, the decline of the seal / dolphin populations

Below are some of the warning symbols that appear on etchants and ferric chloride solutions. They are present to warn the user of their potential danger if mishandled.

There are machines available that are specifically designed to fabricate PCBs and can cost upwards of tens of thousands of dollars, but with a cost of approximately $2500.00 plus a computer this application was much more cost effective than other sources at the time.

Appendix

Definitions:

CNC - The abbreviation **CNC** stands for **computer numerical control**, and refers specifically to a computer "controller" that reads G-code instructions and drives the machine tool, a powered mechanical device typically used to fabricate metal components by the selective removal of metal.

Endmill - is a shank-type Milling cutter, a cutting tool used in industrial milling applications. It is distinguished from the drill bit, in its application, geometry, and manufacture.

G-code - a common name for the programming language that controls NC and CNC machine tools. Developed by the Electronic Industries Alliance in the early 1960s, a final revision was approved in February 1980 as RS274D. Due to the lack of further development, the immense variety of machine tool configurations, and little demand for interoperability, few machine tool controllers (CNCs) adhere to this standard. Extensions and variations have been added independently by manufacturers, and operators of a specific controller must be aware of differences of each manufacturer's product.

References:

http://en.wikipedia.org/wiki/Main_Page

http://www.teksoft.com/

http://www.solidworks.com/

http://www.engineerlive.com/european-electronics-engineer/manufacturingassembly/page_2/1332/the-performance-of-cemented-carbide-in-machining-pcbs.thtml

http://www.technologystudent.com/pcb/pcbcnc.htm