A lifetime designing PCBs: early design adventures

Charles Pfeil - September 07, 2017

My first exposure to PCB design was in 1966 working for Northrup Corporation in the quality assurance (QA) department. My father got me a job there during the summer of my junior year in high school; I was so proud to be working for $1.75 per hour. With gas at 32 cents per gallon, my wages would more than cover my 45-mile round trip.

Working for QA sounds pretty good for a first job, but back then it really was not so glamorous. My primary responsibility was to inspect a Rubylith artwork that PCB designers created. I used an eyeglass to find X-Acto knife cuts and scratches made in the red plastic. Once the offending nicks were found, I would use a red pen to fill them in. Frankly, it got boring after just a few days, but, little did I know, this teenage experience would launch my career in the PCB industry, for which I am forever grateful!

It didn't take long to understand that PCB design was all about creating proper interconnects between components so when the power was applied it would behave and perform as the engineer imagined. Now 51 years later, I can honestly say that nothing has changed in that regard. Sure, materials, components, and signal performance have evolved to enable astounding capabilities; yet, it's still all about hooking up components. With that perspective, PCB design seems like a rather simple task. I often ponder why it hasn't been completely automated after all these decades, but that is a topic for another article.

In 1969, I was fortunate to get a job designing keyboards for Datatronics. This was a summer job while I was studying for an architecture degree. In my calculus class, we wrote little programs with punched cards for the mainframe computer. This piqued my interest in computer technology and made the idea of PCB design even more appealing.

Before working at Datatronics, I interviewed for a draftsman job at an architectural firm. Much to my surprise and embarrassment, I did not get the job because my printing was not fancy enough. Soon, I realized that in architecture, either you were Frank Lloyd Wright or you were a draftsman, nothing in between. Since I failed as a draftsman, my dream was ruined by reality. My work at Datatronics made me understand that as a PCB designer, I could fulfill my primal desire for my career, which was to be creative and get paid for it.

At Datatronics, the PCB assembly for the keyboard consisted of two layers with membrane overlays made of plated Mylar that would complete a circuit when pressed together by each key. Designing a layout was done with red and blue pencils on matte surface Mylar, and the scale of the layout was 2x. Using black tape for the routes and round dots for the pads, we could create the artwork on clear Mylar. Each sheet of Mylar had two punched holes that fit onto registration pins. The layout and the tape-up were aligned and when placed on a light table, it was easy to see if all the interconnects

were taped up properly.

My first design at Datatronics was fabricated and worked without any problems. Designers, like every creative person, want to be involved in projects that can provide a sense of accomplishment. This first design meant a lot to me and validated my decision to abandon architecture for electronics. **Figure 1** illustrates two Mylar layers of an early keyboard design.

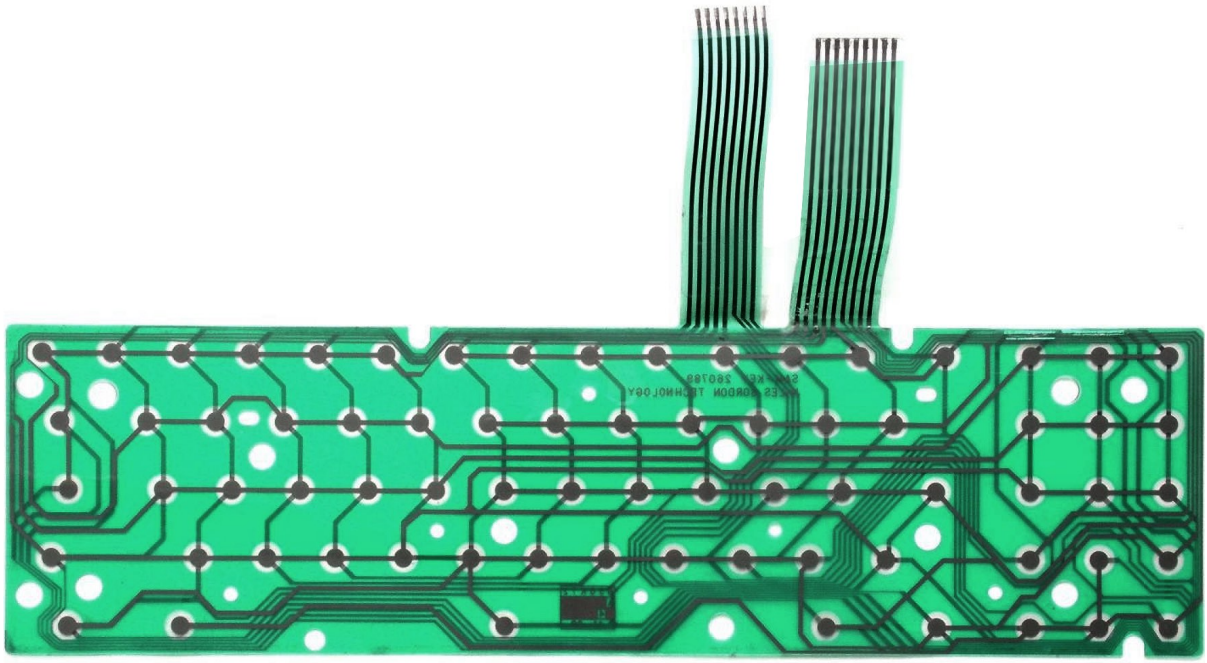

Figure 1 Keyboard membrane layers (**Source**: Velesoft)

During the ‘70s, I worked as a contract designer. The electronics age was emerging and PCB designers were in demand. Employment contract companies, called “job shops,” which normally supplied engineers to this industry were now being asked to provide PCB designers. But where were these designers found? There weren’t any school courses for PCB design, and most electronic engineers wanted to focus on circuit design. This is when the opportunity arose for all those who are now considered the old artistic designers. The typical designer was creative and artistic, while also very detail oriented. Count me in!

As a job shopper, I didn’t have much security. Often, my role lasted as long as a single project and then I would jump to the next company. The good news was it enabled me to gain a wide variety of experience and to meet a lot of wonderful people. **An essential attraction for PCB designers: new technology and products**
My initial encounter with a Faraday cage was while working at Marantz, which developed high-end audio receivers. The building was more like a renovated warehouse than an office. As I was escorted to my drafting table on my first day of work, we walked by this huge cage made of copper. As big as an office, inside the cage there were two men doing something with electronic gear. As a joke, my escort said something along the lines of, “Don’t touch that if you want to live.” But I didn’t get the joke and feared it for some time. What I loved about Marantz was that although it felt like I was working in a garage, the employees were extremely smart and always wanted to innovate.

My desk was a drafting table in a room with three others and the door to the sound room was right next to my desk. Marantz was working on a new project called quadraphonics. At that time, mixing stereo with precise and distinct separation of the instruments and voices was essential. With quadraphonics, this separation was applied across four speakers, two in front and two in back. As

rock bands began recording in this format, the sound room at Marantz became the place to enjoy it.

This is an essential attraction for PCB designers: being part of a team that creates new technology and products. Everyone on the team knows that the PCB designer must apply creative ideas and techniques to lay out each board, and most recognize this is essential for the completion of the project.

In the ‘70s, I also worked at a company that had many projects going on at once, mostly requiring small boards. I was given a schematic for a circuit with less than a dozen components. My thought was to impress everyone by completing it in a very short time. Indeed, my boss was surprised how quickly I passed it on to QA, but it came back even faster. The pads for the resistors were not the right spacing. Oops. I fixed that and re-submitted it. It came back again and I found out the polarity for a capacitor was wrong. This time, my boss intercepted the design and said to me, “I’m not going to let you work on simple designs anymore because you don’t pay attention to what you are doing unless it is complex.” The good news is that I wasn’t fired. For my next project, I was happy to receive a difficult design to focus on.

From that experience, I began to develop a critical skill that experienced designers have: the sense that something is wrong with a design. The sense can be strong enough that it becomes an overriding obsession. Sometimes, it is accompanied by a hint about the location and the type of problem. Other times, it’s just a dull but strong concern.

Unfortunately, the timing of this sense isn’t always very good. I started my own PCB design service bureau in 1978, and my first customer was a company for whom I had previously done contract work. The design took me one week to complete, including documentation. It wasn’t that difficult, maybe 40 dual in-line package (DIP) integrated circuits (ICs) with two edge connectors. I brought it to the customer and started the design review. As I explained what I had done, the sense that something was wrong started growing in my mind. I didn’t say anything about it, but I had two threads of thought running at the same time: “Here is how I did this design” and “Oh no, what is wrong with it?”.

Just before finishing up the review, I realized that I had placed the edge connectors in reverse order; I had placed J1 on the left and J2 on the right, but it was supposed to be the opposite. A wave of disappointment, failure, and fear washed over me. I sheepishly told my customer what had happened and that I would have to re-design it. He was gracious enough to accept the situation and allow me to do so.

The fix required me to completely start over. Nothing could be reused because swapping the connectors meant the entire placement was invalid. It took exactly one week to re-design it, same as the first time. The lesson learned? Double check the placement before routing. I don’t recall if I ever got another job from this customer, but I did learn to be more critical of my work and to do a more thorough job of checking.

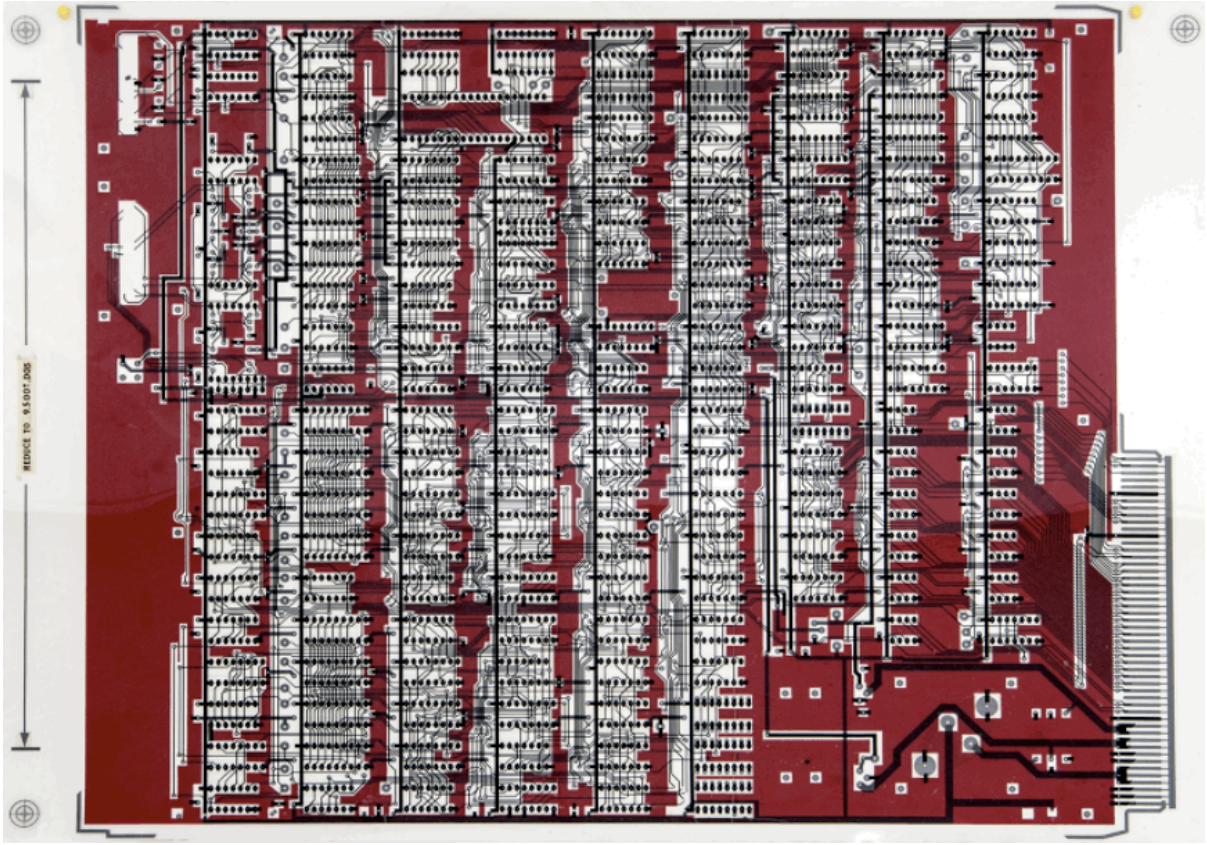

Figure 2 Black tape layout with Rubylith plane (**Source**: Charles Pfeil)

Figure 2 is one of the designs done at my service bureau 38 years ago. It has 154 ICs on two layers. For the routing, the tape-up has three Mylar layers. On the bottom is the “pad master” which has all the pads for pins and vias that are the same on both layers. Next is the solder side (or bottom side) using through-hole round pads for the ICs because the pins are soldered on the bottom. The top side is a more complicated layer. On the underside is a Rubylith sheet and on the top is the black tape. The Rubylith is cut out to make the ground plane after the tape-up is done. The IC pads on the top side are elongated so they can have traces run between them. Note the yellow plastic registration pins at the top. Some of the pads have fallen off because the glue has dried up over the years.

Designers are puzzle solvers, and at the beginning of the design, the puzzle is trying to figure out how to fulfill all of the physical and electrical requirements. Although we know how to design, we are acutely aware that most designs cannot be completed without some compromise. Coupling our inner puzzle solver with our sense of right and wrong enables us to execute on the necessary compromises without sacrificing the project goals.

This is the first of a series of articles that describe my experience as a PCB designer growing up in the age of electronics, and eventually working in the software industry to provide better tools for design. It is my hope that older designers will enjoy reading about my experience, because I am sure they have first-hand knowledge of the same ups and downs. Although newer designers may not have experienced manual design, I hope they will better appreciate the background behind the tools they are now using.

Charles Pfeil is a Senior Product Manager at Altium, working on definition of their products with a primary focus on routing tools.

Also see:

- Ten best practices of PCB design
- PCB design basics
- How did you learn PCB layout?