Processing Gerber files for manufacturing printed circuit boards

Arturo Moreno-Báez\textsuperscript{a}, Gerardo Miramontes-de León\textsuperscript{a}, Ernesto García-Domínguez\textsuperscript{a}, Claudia Sifuentes-Gallardo\textsuperscript{a}

\textsuperscript{a}Universidad Autónoma de Zacatecas
C.A. de procesamiento e instrumentación óptica
López Velarde 801, Zacatecas, México

Abstract

The development of electronic prototypes requires the manufacture of printed circuit boards (PCBs), which is the support and the interconnection of electronic devices. PCBs became of vital importance when the complexity of prototypes requires the connection of a large number of components. This paper presents a technique for manufacturing of PCBs by milling machines, which is intended to reduce the time of the manufacturing process. The approach used in this work is based on image processing algorithms (dilation, erosion, and coding) and a microcontroller circuit. The results are promising, because the time required to perform the PCB is less than with other techniques used by the hobbyist.

© 2012 Published by Elsevier Ltd. Selection and/or peer-review under responsibility of the Organizing Committee of the ENIINVIE-2012. Open access under [CC BY-NC-ND license](https://creativecommons.org/licenses/by-nc-nd/).

Keywords: Gerber file, CAD/CAM, PCB

Nomenclature

- CAD/CAM: Computer Aided Design/Computer Aided Manufacturing
- PCB: Printed Circuit Board
- rs274x: Gerber format standard
- CNC: Computer numerical control

1. Introduction

In the development of experimental research, there is the need of a specific system, either to solve an interconnection problem between a signal from one computer to another system or computer, or to control the movement or speed of a motor, among many others. For some of these needs, there is no commercial solution, or if it exists, does not completely solve the problem. Then, the investment of time and money in electronic design is justifiable. The development of electronic systems involves a series of steps, which leads us to obtain a circuit suitable for our problem. The steps of the process are:

\textit{Email address:} morenob20@hotmail.com (Arturo Moreno-Báez)
Within this list, some common steps that must be repeated several times are steps b) through g). Step
d) is of great importance because of the resources invested to develop the system, but promptly making a
PCB, to cover a very specific need, demands too many human and material resources, so this is the area
of opportunity where the present work is focused.

The implementation of electronic circuits has gone through an evolution from the use of bridge-terminals,
wire wrap, and the PCBs. When testing small circuits, it is feasible to use a breadboard, which is constructed
as semi-fixed (since can be changed) circuit, but its maintenance and versatility are limited. Some other
drawbacks of the breadboard are parasitic capacitances, false contacts, loose components, etc. The imple-
mentation of a prototype in a PCB ensures that those problems listed above will not be present, minimizing
the risk of failure.

Although there are companies that manufacture printed circuit boards, for a prototype, the cost is not
justified. So, it is important to seek cost-effective solutions. Acceptable electronic development tools include
CAD extensions, which help in drawing paths of the PCB. These tools reduce the time spent on that task,
but going from CAD drawing to the implementation of the circuit, generates particular problems. One of
these problems is the transfer of the PCB paths to a cupper clad circuit. The most widely used technique
for the construction of PCBs is the silk-screen, which is slow and for a small volume that is built, its cost is
high. Besides the time spent, if one takes into account that a student or a researcher is the one who should
perform this task, the cost is higher.

For the manufacture of PCBs, CAD tools have a format suitable for special machines CAD/CAM (Com-
puter Aided Design/Computer Aided Manufacturing). The format is called Gerber [1] (RS-274D and
RS-274X), which uses a photo-plotter to create a silk-screen mesh. The Gerber file (see Table [1].a)) contains
openings (Fig. 1.a)) and trajectories (Fig. (fig. 1.c). With the combination of openings and trajectories, each
of the tracks is formed (Fig. 1.c). The pads are defined in the same way (Fig 1.d). The tracks and paths
information contained in the file is the PCB (Fig. 1.e).
The usual way to manufacture PCBs, for prototyping, is through chemical attack with iron chloride \( (\text{FeCl}_3) \). The printed pads and tracks are protected from the \( \text{FeCl}_3 \). The chemical attack would be responsible for removing excess material. Another way is the use of a CNC milling machine, which takes care of removing the material around the tracks. The Gerber format cannot be used directly to control the milling machine, because it does not define the geometry of the conductive tracks, but it defines instead the travel of pattern openings of the photo-plotter’s lamp. For the use of a milling machine it is necessary to obtain the path to be followed by the tool to generate the track (see Fig. 1.f and Table 1.b).

There are software tools that are capable of transforming the source code from Gerber to CNC milling machines (Fig. 2). The use of management programs for small CNC milling machines, in conjunction with code transformation programs, entails some problems, because of the way these programs implement the standards and introduce mismatch errors in the instructions.

The intended application of this work is to eliminate these errors and make the use of the milling machine as simple as a control system, so the use of the milling machine resembles the use of a conventional printer. Debugging, either Gerber or CNC code, is a time-consuming task, and the complexity of the generated code does not allow the geometry to correspond easily to the instructions (see Table 1). So, this is a problem to be avoided in building prototypes [2].

![Fig. 2. Software tool converted Gerber Format to CNC Code](image)
2. Materials and Methods

The use of a simple automaton for controlling a milling machine, shown in Figure 3, requires the following settings: a) the space work is limited into a plane, b) the motion is also coded in a plane, c) the Z axis is displaced in a non-concurrent way, d) the axes are actuated by stepper motors.

These limitations allow to have a low complexity automaton to control the milling machine. Figure 4 describes the way data is delivered to the controller, these data is obtained after the Gerber file computational tool performs the following steps:

1. Rendering of a bitmap image.
2. Expanding the image to provide clear paths necessary for the roughing tool.
3. Edge detection
4. Encoding using the chain-code.
5. Complete the information with the motion in Z axis.

These five steps allow to build the PCB, from the data provided by the Gerber file.

![Fig. 3. Mini-mill machine](image)

![Fig. 4. Proposed technique](image)

![Fig. 5. Application of the technique](image)
Figure 5 shows the steps of the proposed technique. The first block corresponds to the bitmap obtained from the Gerber file, over which, in the following step, it is applied a dilation and thickening of the image lines; the dilation is applied so the grinding tool generates the PCB to the nominal dimensions required for the next step. Then, edge detection is performed, followed by a coding procedure, which results in a string of numbers (the chain-code) indicating the direction the tool must follow to manufacture the PCB.

3. Results

A PCB was designed and manufactured using the proposed approach. The designed PCB is a SD memory card socket, which is necessary for some memory based systems. The SD card socket is a surface-mount type component, so it is not possible to test this component using a breadboard, thus a PCB is needed. Figure 6 shows the manufactured PCB; on the left side the electronic design is shown, the image of the center corresponds to the finished PCB, and to the right the mounted SD card socket can be seen.

The time required to manufacture the PCB, shown in Figure 6, taking into account the activities of setup of the milling machine, drilling and tools changes, were less than 8 minutes. This time is lower than the time needed by serigraphic methods, or using thermally transferable sheets.

4. Conclusions

The results of the proposed technique are encouraging, since the complexity of manufacturing the PCB was reduced, achieving the goal of spending more time in the development of a research. Its use does not require training or highly trained personnel. Thus the proposed approach becomes a laboratory tool useful for the development of electronic systems, that may be necessary in experimental research. An unexpected result, due to the way the information is encoded, is the control system, which was implemented with a simple automaton of very low complexity. A work in progress is to make the system independent of a personal computer.

References

[1] The gerber format specification: Rs-274x or extended gerber, revision h (Jan 2012).