

# PREFACE

Printed Circuit Board Making Using Eagle was initiated with the intention to help the undergraduate student to possess a skill in fabricating printed circuit board for their graduation project through a CAD tool known as Eagle. Indeed, the skill requirement is not just restricted to undergraduate students, all technical school and college students who are doing an electronic or electrical course must have the experience in producing a printed circuit board. This book is also written to meet the needs of the electronic hobbyist, perhaps, are interested to overcome the complexity problem in electronic design by using Eagle as a tool. A major portion of this book is practically oriented, and it has been carefully written in such a way that step by step guidance is provided to straighten the learning path. It makes your acquisition of a CAD tool skill so much easier at the end of the day!

Eagle comes in multiple languages version such as English and German. It can be installed in Windows, Mac and Linux operating system. You can export your schematic and layout in common image formats. The software has the flexibility to generate the output either in the direct PCB layout form or in Gerber files. This software has been adopted by renowned company such as AMD, AT&T, BASF, BWM, DuPont, Hewlett Packard, IBM, Intel, Motorola, National Semiconductor, Nokia, NASA, Phillips and many more.

In Chapter 1, you will gain understanding on the formation of copper clad board and printed circuit board, various types of PCB and CAD tools available, PCB layout layers and introduction to Eagle software. The difference between freeware and professional version of Eagle will be explained.

Chapter 2 offers you a idea in professional PCB fabrication techniques in lab using various types of machines.

Chapter 3 explains the familiarization with the Eagle software environment. It is for you to go through before starting to make any drawing

because you will be utilizing a lot of the tools from this chapter. When you are familiar with the graphic tools, assessing the icon is so much faster.

The explanation in Chapter 4 is done deliberately in order to guide you through steps in completing a simple digital board which is a microcontroller circuit. You will gain an appreciation on how easy it is to connect components in a schematic drawing with a very rich library of devices and how to make use of basic tools to edit and manipulate parts in a drawing. You will be shown how to apply Electrical Rule Check to detect the error and warnings in the electronic circuit. Nonetheless, print option and document preparation will be unveiled. In the board layout creation procedures, the highlight is the ground making pattern.

Chapter 5 is a compliment to Chapter 4 by increasing the complexity in the drawing though there are some repetitive steps in creating the drawing. As a result, the learners will master and enhance their skills even further. Another commonly used circuit, an optoisolator and low pass filter circuit with more samples of devices is presented in this chapter. Other example such as troubleshooting ERC error will be delineated. The additional board manipulation techniques discussed in this chapter is the single and double sided board creation, pattern correction, route and unroute technique, layer display control, silk screen production, net classes settings, BOM list generation and etc. Chapter 5 will also show you the dimensioning, putting annotation and printing of a layout.

In order to demonstrate the capability of Eagle in producing memory circuits' layout, we have decided to include Chapter 6 to enhance your learning experience. This chapter further shows you the use of bus example.

Advanced topics such as creating a new device library are included in Chapter 7. First, the concept of measurement and dimension of a device is explained through the actual device manufacturer datasheet. With that understanding in mind, the learner learns the steps in creating the symbol and package with exact dimension which is really utilizable with the correct orientation of pins. The knowledge obtained from this topic is very useful and practical if you are designing a PCB that incorporates new component released into the market.

To prepare the data for an automated PCB fabrication, Eagle provides the facility to convert the board file into Gerber data which is understandable by the photoplotter and Numerical Control drill machine. You will explore the manufacturing data preparation steps in Chapter 8.

In the final chapter of this book, Chapter 9, you will have the opportunity to explore the easy PCB fabrication technique in a very simple and low cost approach. The whole process spanning from creating a transfer, manual etching process to the preparing of PCB for component insertion will be illustrated with lots of graphics.

Follow the instructions of the book should enable you to acquire the skill in creating a useful printed circuit board with Eagle. Enjoy your learning and all the best to you!