

Printed Circuit Board Fabrication with EAGLE Light

Printed Circuit Board Fabrication with EAGLE Light

SOON CHIN FHONG
IGNATIUS AGUNG WIBOWO



PENERBIT UTHM

CONTENTS

<i>Preface</i>	ix
<i>Acknowledgment</i>	xiii
Chapter 1 INTRODUCTION	1
What is Printed Circuit Board?	2
Printed Circuit Board Option	5
PCB CAD Software	5
PCB Layout	6
The EAGLE Software	9
System Requirements for EAGLE Software	10
Chapter 2 PCB MAKING TECHNIQUES	11
Basic PCB Manufacturing Process	11
Mass PCB Fabrication Technique	19
Summary	28
Chapter 3 FAMILIARIZATION WITH EAGLE	29
Getting the Software	29
Software Installation	32
Running EAGLE for the First Time	36
Control Panel	37
Schematic Editor	41
Board Editor	42
Summary	51
Chapter 4 PCB LAYOUT FOR A MICROCONTROLLER CIRCUIT	53
Create a New Project	55
Adding and Parts Arrangement	55
Changing the Value and Name of the Part	61
Changing the Position of the Value and Name of a Part	63
Viewing the Footprint of the Parts in Board Window	64

Wiring the Power Supply and Ground	64
Wiring the Defaults Wires	67
Electrical Rule Check (ERC) and Troubleshooting	68
Display Control of the PCB Layout	73
Auto Route Board	78
PCB Layout Touch Up	86
Silk Screen Production	96
Parts List Generation	98
Summary	100
Chapter 5 PCB LAYOUT FOR OPTOISOLATOR AND LPF	103
Creating a New Project	104
Creating New Schematic	105
Adding Parts	105
Deleting Parts	108
Changing the Value and Name of the Parts	108
Arranging Components	109
Wiring the Circuit	110
Troubleshooting	111
Changing the Position of Value and Name of the Components	116
Board Layout	117
Arranging Components in the Board	118
Autorouting	120
Adding Component after the Board is Ready	123
Adding a Frame	125
Adjusting Track Width	128
Manual Touching	133
Printing the Tracks and Pads	133
Printing the Silk Screen	136
Bill of Material	138
Generating Part List	140
Backup Files	140
Summary	141
Chapter 6 MICROCONTROLLER AND MEMORY DEVICES	143
Creating Microcontroller and Memory Devices Circuit and Layout	143

Summary	148
Chapter 7 MAKING A LIBRARY	149
Create New Library Sub-Directory	150
Create New Library in mylibrary Sub-Directory	154
Create a Symbol for the New Library	155
Create Package for the New Library	163
Create Device for New Library	171
Testing out the New Library	176
Summary	178
Chapter 8 MANUFACUTRING DATA PREPARATION	179
Generating the Excellon Drill File	181
Generating Gerber File	183
Summary	184
Chapter 9 EASY PCB FABRICATION PROCESS	185
Creating the Transfer	186
Transfer of Printed Circuit	188
Etching Process	191
Removing Printer Toner Traces	194
Drilling Holes	196
Summary	197
Bibliography	199
Index	201

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, BMW, 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!

ACKNOWLEDGEMENT

This is our combined effort in writing the second book. A high degree of commitment has been put into it.

The drive of writing this book comes from our dear graduates Nicholas Tan Jin Sheng and Low Si Khim and a lot of other students who have used Eagle to fabricate the PIC microcontroller programmer and adopted Easy PCB fabrication technique in their undergraduate project development. The expression of many other students' need to pick up CAD skill in PCB design has made this book to be possible. To our colleague, Madam Cheng Leong Yeok who has shown interest in finding out Eagle's capability from us, your enquiries has turned into motivation for us to carry on.

We are grateful to Mr. Khairul Bin Hj. Nani, our technician in the PCB Fabrication laboratory of Universiti Tun Hussein Onn Malaysia who has been very helpful in preparing tools and facility for the graphic production in this book. We thank Mr. Mohd Jais Bin Che Soh and Mr. Mahmod bin Munajat who has generously shared their knowledge with us during our work in Advance PCB Laboratory.

We are also indebted to our Dean from Faculty of Electrical and Electronic Engineering, Prof. Ir. Dr. Zainal Abidin Bin Ahmad, our department heads Associate Prof. Siti Hawa Bt Ruslan and Associate Prof. Dr. Zainal Bin Alam bin Haron. Their continuous support and encouragement are very valuable and meaningful to us.

Last but not least, to our beloved family who has supported us unreservedly.

Thank you very much!

Soon Chin Fhong
soon@uthm.edu.my

Ignatius Agung Wibowo
agung@uthm.edu.my

Faculty of Electrical and Electronic Engineering
Universiti Tun Hussein Onn Malaysia
Batu Pahat, Johor

