UNIVERSITI TEKNIKAL MALAYSIA, MELAKA

DESIGN AND ANALYSIS OF MODULAR FIXTURE OF PCB ROUTING MACHINE

This report submitted in accordance of with the requirement of the Universiti Teknikal Malaysia Melaka (UTeM) for the Bachelor Degree of Manufacturing Engineering (Manufacturing Design) with Honours.

by

MOHD FARID BIN GHAZALI

FACULTY OF MANUFACTURING ENGINEERING

2009
JUDUL: Design and Analysis of Modular Fixture of PCB Routing Machine

SES PENGAJIAN: 2009-2010

MOHD FARID BIN GHAZALI

mengaku membenarkan tesis (PSM/Sarjana/Doktor Falsafah) ini disimpan di Perpustakaan Universiti Teknikal Malaysia Melaka (UTeM) dengan syarat-syarat kegunaan seperti berikut:

1. Tesis adalah hak milik Universiti Teknikal Malaysia Melaka.
2. Perpustakaan Universiti Teknikal Malaysia Melaka dibenarkan membuat salinan untuk tujuan pengajian sahaja.
3. Perpustakaan dibenarkan membuat salinan tesis ini sebagai bahan pertukaran antara institusi pengajian tinggi.

☐ SULIT (Mengandungi maklumat yang beredar keselamatan atau kepentingan Malaysia yang termaktub di dalam AKTA RAHSIA RASMI 1972)
☐ TERHAD (Mengandungi maklumat TERHAD yang telah ditentukan oleh organisasi/badan di mana penyelidikan dijalankan)
☐ TIDAK TERHAD

Disahkan oleh:

(TANDATANGAN PENULIS) (TANDATANGAN PENYELIA)

Alamat Tetap:
No 6, Jalan Indah 2/8A, 
Puchong Indah, 
47100 Puchong, 
Selangor.

Cop Rasmi:

Tarikh: _______________________  Tarikh: _______________________
DECLARATION

I hereby, declared this report entitled “Design and Analysis of Modular Fixture of PCB Routing Machine” is the results of my own research except as cited in references.

Signature : ..................................................
Author’s Name : ..................................................
Date : .....................................................
This report is submitted to the Faculty of Manufacturing Engineering of UTeM as a partial fulfillment of the requirements for the degree of Bachelor of Manufacturing Engineering (Manufacturing Design). The member of the supervisory committee is as follow:

..............................
(Mr. Ismail Bin Abu Shah)
(Supervisor)
This report contains the research details on the design and analysis of modular fixture of PCB routing machine. Basically, this report described the research information needed according to the project title. As this report was finished, it was consisted of six chapters which were included the introduction, literature review, methodology, result and analysis, discussion, and the last was conclusion and recommendation. The first three chapters described more on initial investigation and action taken to undergo the research, which are needed for the understanding before the design and analysis stage done. The three other chapters showed and explained the result and output from the analysis. Generally, this research was focused on designing a modular fixture used for PCB cutting using the routing machine. The main purpose of this research was to design and to analyze the PCB fixture that positioned and located for holding the PCB during the cutting operation. Furthermore, the analysis was required in order to find out the most optimum position besides, and to carried out the analysis result for comparison of several condition methods used to conduct the analysis. The analysis was focused on strain measurement for PCB vibration during the cutting operation. At last, based on the analysis result, the PCB fixture that scored the lowest strain value was determined as the most optimum position for holding the PCB, in which, the objectives of this research was conducted.
ABSTRAK

Laporan ini mengandungi butiran kajian tentang rekabentuk dan analisis lekapan modular pada mesin pemotongan PCB. Pada dasarnya, laporan ini menggambarkan maklumat kajian yang diperlukan sesuai dengan tajuk projek. Setelah laporan ini disiapkan, ia terdiri daripada enam bab yang terdiri daripada pengenalan, tinjauan karya, metodologi, hasil dan analisis, perbincangan, dan yang terakhir, kesimpulan dan cadangan. Tiga bab pertama menjelaskan lebih lanjut mengenai penyiasatan awal dan tindakan yang diambil untuk menjalankan kajian yang mana diperlukan bagi pemahaman sebelum rekabentuk dan analisis dilakukan. Tiga bab berikutnya menunjukkan dan menjelaskan hasil dan keluaran dari analisis. Umumnya, kajian ini difokuskan pada perancangan lekapan modular yang digunakan untuk memotong PCB dengan menggunakan mesin pemotong. Tujuan utama dari penelitian ini adalah untuk merancang dan menganalisis lekapan PCB yang diposisikan dan dileletakkan untuk memegang PCB semasa operasi pemotongan. Selanjutnya, analisa ini diperlukan dalam rangka untuk mengetahui kedudukan yang paling optimum selain daripada melaksanakan keputusan analisis untuk membandingkan keadaan beberapa kaedah yang digunakan untuk membuat analisis. Analisis difokuskan pada pengukuran regangan untuk getaran PCB semasa operasi pemotongan. Akhirnya, berdasarkan hasil analisis, lekapan PCB yang mencatatkan nilai regangan terendah ditentukan sebagai kedudukan yang paling optimum untuk memegang PCB, yang mana merupakan tujuan utama kajian ini dilakukan.
DEDICATION

To my beloved family and friends.
I am glad to thankful to Allah The Almighty, with His bless and mercy, I was successfully completed the PSM report as well. I would like to express my appreciation to anybody who is involved directly or indirectly during the preparation of this report. Moreover, I would like to thank to my PSM supervisor, Mr. Ismail bin Abu Shah, for his advice and guidance towards me throughout this project. He was helped me with his constructive comments and ideas to complete this report as well. Next, to my family members, thanks a lot for their support and encouragement to me to carry out this project better. Same goes to my fellow friends, a lot of thank to their cooperation to share their useful ideas and comments in order to accomplish this report. To all that I have mentioned above, I really appreciate your support and concern. I promise to improve myself to be better for the future. Thank you very much.
# TABLE OF CONTENT

Abstract i  
Abstrak ii  
Dedication iii  
Acknowledgement iv  
Table of Content v  
List of Tables ix  
List of Figures x  
List of Abbreviations xiii

## 1.0 INTRODUCTION 1
1.1 Background 1  
1.2 Problem Statement 4  
1.3 Objective of Study 6  
1.4 Scope of Study 6  
1.5 Project Outline 6

## 2.0 LITERATURE REVIEW 8
2.1 Background of Printed Circuit Board (PCB) 8  
2.1.1 PCB History 10  
2.1.2 PCB Warpage 11  
2.2 Workholding - Jigs and Fixtures 12  
2.2.1 Principles of Workholding 13  
2.2.1.1 Locating 13  
2.2.1.2 Clamping 16  
2.3 Fixtures 18  
2.3.1 Fixture Configurations 19  
2.3.2 Fixture Design Process 20  
2.4 Modular Fixture 21  
2.4.1 Modular Fixture System 23  
2.4.1.1 T-slot-based Modular Fixtures 23
2.4.1.2 Dowel-Pin-Based Modular Fixtures 24
2.4.1.3 Comparison between T-slot based and Dowel-pin based 26
2.5 PCB Routing 28
2.5.1 Pin Routing 28
2.5.2 CNC Routing Applications 29
2.5.3 CNC Operations 30
2.5.4 Mechanical Routing with CNC Equipment 31
2.5.5 Summarizing of PCB Routing 34
2.6 PCB Depanelize 35
2.6.1 PCB Strain Analysis 36
2.6.2 BGA Assembly Strain Measurement 39

3.0 METHODOLOGY 41
3.1 Introduction 41
3.2 Project Planning 41
3.2.1 Statement of Work (Gantt chart) 42
3.3 Project Flow 44
3.3.1 Identify Problem and Defining of Project Objectives and Scope 45
3.3.2 Literature Review and Existing Method 45
3.3.3 PCB Modular Fixture Design 46
3.3.4 Experimental Analysis, Simulation and Verification 46
3.3.5 Discussion and Conclusion 47
3.4 Gathering Information 48
3.4.1 Online Articles and Journals 48
3.4.2 Reference Books 48
3.4.3 Video Presentation 49
3.5 Design and Analysis Tools 49

4.0 RESULT AND ANALYSIS 50
4.1 PCB Used 50
4.2 Cutting Area 51
4.3 Routing Machine 52
4.4 Fixture Design 54
4.4.1 Conceptual Design 54
<table>
<thead>
<tr>
<th>Section</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>6.0 CONCLUSION AND RECOMMENDATION</td>
<td>97</td>
</tr>
<tr>
<td>6.1 Conclusion</td>
<td>97</td>
</tr>
<tr>
<td>6.2 Research Barriers</td>
<td>99</td>
</tr>
<tr>
<td>6.3 Recommendations</td>
<td>99</td>
</tr>
</tbody>
</table>

REFERENCES 102
APPENDIX 104
# LIST OF TABLES

<table>
<thead>
<tr>
<th>Table</th>
<th>Title</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>2.1</td>
<td>Comparison of T-slot and Dowel-pin modular fixtures</td>
<td>27</td>
</tr>
<tr>
<td>2.2</td>
<td>Various depaneling methods</td>
<td>35</td>
</tr>
<tr>
<td>3.1</td>
<td>Project Gantt chart for PSM 1</td>
<td>40</td>
</tr>
<tr>
<td>3.2</td>
<td>Project Gantt chart for PSM II</td>
<td>41</td>
</tr>
<tr>
<td>4.1</td>
<td>Locating pin and supporter used for 3 difference conditions</td>
<td>70</td>
</tr>
<tr>
<td>4.2</td>
<td>Cutting parameters</td>
<td>71</td>
</tr>
<tr>
<td>4.3</td>
<td>Strain gauge results</td>
<td>79</td>
</tr>
<tr>
<td>5.1</td>
<td>Comparison of PCB jamming and without PCB jamming</td>
<td>84</td>
</tr>
</tbody>
</table>
CHAPTER 1
INTRODUCTION

This report described a project on the design and analysis of modular fixture for Printed Circuit Board (PCB) routing machine. In this research, the modular fixture becomes a work holding device for PCB cutting operation by using the routing machine.

1.1 Background

The work holding device can be defined as a device used to locate and hold a workpiece. The work holding device references the tool performing the operation on the part being held. Basically, the work holding device is widely used in manufacturing industries, where, most of manufacturing processes include machining, assembly, joining, and also inspection of part required the use of work holding device.

Technically, the work holding device can be divided into four categories; standard devices, jigs, fixtures, and modular fixturing system. Each type has it own capability to hold the part in certain conditions. Furthermore, it can be designed either in automated or manually operated.

In addition, the work holding device is essential for the manufacturing process due to the greater accuracy and quality needed in manufacturing industries. Other than that, the reproducibility of the similar part using the work holding device can lead to the consistent productivity and accuracy of the part.
A printed circuit board, or PCB, is a self-contained module of interconnected electronic components found in devices ranging from common beepers, or pagers, and radios to sophisticated radar and computer systems. The circuits are formed by a thin layer of conducting material deposited, or "printed," on the surface of an insulating board known as the substrate. Individual electronic components are placed on the surface of the substrate and soldered to the interconnecting circuits. Contact fingers along one or more edges of the substrate act as connectors to other PCBs or to external electrical devices such as on-off switches. A printed circuit board may have circuits that perform a single function, such as a signal amplifier, or multiple functions.

Depaneling is a process step in high-volume electronics assembly operation. In order to increase the throughput of PCB manufacturing and surface mount technology lines, PCBs are often designed so that they consist of many smaller individual PCBs that will be used in the final product. This PCB cluster is called a panel or multiblock. The large panel is broken up or “depaneled” as a certain step in the process depending on the product, it may happen right after SMT process, after in-circuit test (ICT), after soldering of through-hole elements, or even right before the final case up of the assembly.

Furthermore, the depaneling process is relatively similar as the PCB cutting in industry. Basically, the cutting operation is done manually or automatically based on the accuracy needed for the final product of the PCB. In term of accuracy and the ability of the cutting operation, the automatic approach is usually used together with the other elements that include fixture and holding elements. Besides that, the movement of cutting is controlled by robot movement that will cut the PCBs according to the robot program. Moreover, this cutting operation is also known as the PCB routing operation.

In routing process, it involves two most important parameters which are feed rate and rotational speed. They are chosen according to the bit type and diameter and should remain proportional. The routers generate vibrations of the same frequency as their rotational speed, which might be important if there are vibration sensitive components on the surface of the board. By mean of that, in order to make the
routing process is succeed without damaging the components and the board itself, a proper fixture should be used together to hold the PCB during the cutting of PCB. The fixture is not only to hold the PCB, but also to maintain the accuracy of cutting and to prevent any abnormalities to the board including crack propagation and warping.

Other than that, the fixture should be a modular type fixture which is able to locate and hold the PCB at several points in order to find out the greatest location for the locating pin to hold the PCB for the cutting operation. The most less strain level resulted from the PCB cutting will become the most appropriate location for the locating pin to hold the PCB.
1.2 Problem Statement

PCB cutting or depaneling is not an easy process. In modern manufacturing environment where surface-mount components are the norm, depaneling methods such as V-scores and various hand-break methods may be risky. Reliably and repeatably controlling the forces required to depanel the PCB is difficult. If a ceramic capacitor is damaged, for example, the defect may then show up as a field failure. These kinds of defects can slip right through a functional test.

One significant problem related to the PCB cutting is crack or warping cause by high force given during the operation. Normally, in manual cutting operation which is usually using the hand to break the PCB, the force given is not constant due to the energy of human being. Since the force is not consistent, the PCB is highly risk to be damaged in term of warping or crack. It will lead to failure for the PCB to be functioned. Furthermore, there is no specific equipment used to analyze and measure the PCB if any abnormality happened. By using the naked eye, it is not impossible to find out which part of the PCB got crack or warping, but if the amount of crack is too little, it might be difficult to the user to determine the infected area.

The selection of automatic or manual method to break or to cut the PCB also will affect the PCB. Basically, the manual hand break method is not a good way to cut the PCB. The force given by human energy to break the PCB will lead to the PCB damage due to the improper way of cutting. Besides that, the hand break method also involves inconsistent force given to cut the PCB. Moreover, the automatic machining that used to cut the PCB also has several drawbacks. The cutting strategy is the main cause that will lead to the PCB warping, damage and so on. In term of that, the cutting movement to cut the PCB and the cutting point which is to start the PCB cutting need to be considered first before the cutting operation is done. The wrong direction of cutting movement and the improper selection of cutting start point will affect the PCB. Furthermore, the machine capability also will make the cutting operation fail due to several things that related to the machine capability. As an example, the speed of cutting and the routing bit used to cut the PCB. In order to prevent failure for the PCB cutting operation, the right selection of cutting speed
and the tool bit used are the other things to consider as the cutting strategy by using the automatic cutting methods.

Another problem associated with the PCB cutting is that the holding method used during the PCB cutting operation. By mean of that, the jigs or fixtures used for holding the PCB are not good enough to prevent the PCB from warping or damage. Basically, the problem is happened due to the unsuitable jigs or fixtures used. The jigs or fixtures should not be used only for holding the PCB. It is required for the jigs or fixtures to absorb the force, stress and strain during the cutting operation. Moreover, most of the jigs or fixtures used are not modular, that is much better to hold and locate the PCB well. In addition, the locating of PCB onto the fixture is not proper enough. Technically, it will lead to the unstable condition to the PCB for the cutting operation. The vibration from the cutting operation will damage the PCB and makes the PCB fail to function.

A modular fixture is essential to design as the solution to overcome the problems during the PCB cutting or depanelizing. The modular fixture should be good enough to hold and to locate the PCB for the cutting operation. Besides that, it can prevent the PCB damage, crack or warping by holding and locating the PCB at the right condition. In addition, the fixture should be capable to use together with the routing machine in order to run the PCB cutting automatically. Once the PCB is ready to cut, the modular fixture can be managed to make the PCB cutting in stable condition without affected the PCB. The strain gauge can be used together with the modular fixture during the cutting operation as the tool to measure the vibration, and to analyze the amount of crack propagation or warping for the PCB. By mean of that, the right placement of holding and locating the PCB on the modular fixture can be made according to the result shows from the using of strain gauge.
1.3 **Objective of Study**

There are two main objectives from this research. There are as follows:

a. To design a PCB modular fixture to hold and locate the PCB during the cutting operation for cutting stability.

b. To analyze the optimum holding and locating method for PCB cutting by using strain gauge.

1.4 **Scope of Study**

This study will cover the design and analysis of PCB fixture for PCB cutting operation. The fixture will be a modular fixture used to cut the PCB by using the routing machine. The experimental analysis will be conducted to find out the most optimum holding and locating condition for the PCB during the cutting operation. The PCB fixture will be designed according to the routing machine standard by using computer software, while the experimental analysis will use the strain gauge to measure the vibration, crack propagation, and warping during the PCB cutting operation. The most optimum holding and locating condition will be identified then.

1.5 **Project Outline**

Based on the thesis for Projek Sarjana Muda (PSM), an organisation has been constructed for the process flow of completion. These organisations are used for students purposely follow the format and understand the sequence of doing the project as good as possible to fulfil course of Degree in UTeM. The formats of organisations are as follows:

a. Chapter 1 – Introduction

This chapter represents the general introduction of the project which consists of project background, problem statement, objective and scope of the
project, and also the project outline which is briefly explain the subtopics related to this project.

b. Chapter 2 - Literature Review
   This chapter is all about the knowledge and information of the project. Basically, it is the theoretical knowledge regarding to the project title which is help in understanding the project as well.

c. Chapter 3 – Methodology
   This chapter is focusing on the flow of the project from the beginning until the project done. Besides that, it also shows the general explanation of each process of methodology involves in this project. Others, the tools and analysis techniques use for designing and analyzing the PCB fixture are also stated.

d. Chapter 4 – Result and Analysis
   This chapter will show the result of the PCB fixture design for the cutting operation. Other than that, the method of conducting the experimental analysis also shows in this chapter. Furthermore, the analysis result from the PCB cutting operation also included.

e. Chapter 5 – Discussion
   This chapter will describe in detail the results of this project. It will focus on the explanation of PCB fixture, the routing machine used to cut the PCB, and the process involved for the PCB cutting operation. Others, the most optimum holding and locating condition for PCB also stated in this chapter.

f. Chapter 6 – Conclusion
   This chapter will present the overall conclusion of this project. Any recommendations and suggestions for future research will be described as well as the barriers that maybe give some difficulties in order to finish this project.
2.1 Background of Printed Circuit Board (PCB)

A printed circuit board, or PCB, is a self-contained module of interconnected electronic components found in devices ranging from common beepers, or pagers, and radios to sophisticated radar and computer systems. The circuits are formed by a thin layer of conducting material deposited, or "printed," on the surface of an insulating board known as the substrate. Individual electronic components are placed on the surface of the substrate and soldered to the interconnecting circuits. Contact fingers along one or more edges of the substrate act as connectors to other PCBs or to external electrical devices such as on-off switches. A printed circuit board may have circuits that perform a single function, such as a signal amplifier, or multiple functions. (Carvette C., 2000).

There are three major types of printed circuit board construction: single-sided, double-sided, and multi-layered. Single-sided boards have the components on one side of the substrate. When the number of components becomes too much for a single-sided board, a double-sided board may be used. (Ford D.N., 2001). Electrical connections between the circuits on each side are made by drilling holes through the substrate in appropriate locations and plating the inside of the holes with a conducting material. The third type, a multi-layered board, has a substrate made up of layers of printed circuits separated by layers of insulation. The components on the surface connect through plated holes drilled down to the appropriate circuit layer. This greatly simplifies the circuit pattern.
Components on a printed circuit board are electrically connected to the circuits by two different methods: the older "through hole technology" and the newer "surface mount technology." (Ford D.N., 2001). With through hole technology, each component has thin wires, or leads, which are pushed through small holes in the substrate and soldered to connection pads in the circuits on the opposite side. Gravity and friction between the leads and the sides of the holes keeps the components in place until they are soldered. With surface mount technology, stubby J-shaped or L-shaped legs on each component contact the printed circuits directly. A solder paste consisting of glue, flux, and solder are applied at the point of contact to hold the components in place until the solder is melted, or "reflowed," in an oven to make the final connection. Although surface mount technology requires greater care in the placement of the components, it eliminates the time-consuming drilling process and the space-consuming connection pads inherent with through hole technology.

Two other types of circuit assemblies are related to the printed circuit board. An integrated circuit, sometimes called an IC or microchip, performs similar functions to a printed circuit board except the IC contains many more circuits and components that are electrochemically "grown" in place on the surface of a very small chip of silicon. A hybrid circuit, as the name implies, looks like a printed circuit board, but contains some components that are grown onto the surface of the substrate rather than being placed on the surface and soldered.

Figure 2.1: Populated PCB (Meskauskas, 2006)
2.1.1 PCB History

Printed circuit boards evolved from electrical connection systems that were developed in the 1850s. Metal strips or rods were originally used to connect large electric components mounted on wooden bases. In time the metal strips were replaced by wires connected to screw terminals, and wooden bases were replaced by metal chassis. But smaller and more compact designs were needed due to the increased operating needs of the products that used circuit boards. In 1925, Charles Ducas of the United States submitted a patent application for a method of creating an electrical path directly on an insulated surface by printing through a stencil with electrically conductive inks. This method gave birth to the name "printed wiring" or "printed circuit." (Carvette C., 2000).

In the 1943, Paul Eisler of the United Kingdom patented a method of etching the conductive pattern, or circuits, on a layer of copper foil bonded to a glass-reinforced, non-conductive base. Widespread use of Eisler's technique did not come until the 1950s when the transistor was introduced for commercial use. Up to that point, the size of vacuum tubes and other components were so large that the traditional mounting and wiring methods were all that was needed. With the advent of transistors, however, the components became very small, and manufacturers turned to printed circuit boards to reduce the overall size of the electronic package. (Ford D.N., 2001).

Through hole technology and its use in multi-layer PCBs was patented by the U.S. firm Hazeltyne in 1961. The resulting increase in component density and closely spaced electrical paths started a new era in PCB design. Integrated circuit chips were introduced in the 1970s, and these components were quickly incorporated into printed circuit board design and manufacturing techniques.
2.1.2 PCB Warpage

Warpage is the out of plane deformation of the artefact (which is PCB in this research), caused by differential (non-homogenous) shrinkage or expansion of elements composing the artifact. (Klein et al., 2004). In this research, it will focus on thermally induced warpage, i.e. warpage of the PCB when it is subjected to the thermal loading. Increasingly, local warpage, e.g. warpage in the region of critical component footprint, is a more critical issue than global warpage, the warpage of the PCB as a hole. Changes in the contour of the component footprint can create shorts or opens in the PCB component solder joints during cutting operation or build stresses into the assembly that appear as later reliability problem (Klein et al., 2004).

![Figure 2.2: Out of plane deformation of a linear element (Klein, 2004)](image)

![Figure 2.3: Saddle deformation (Klein, 2004)](image)