UNIVERSITI TEKNIKAL MALAYSIA MELAKA

UNIVERSITI TEKNIKAL MALAYSIA MELAKA

DESIGN AND DEVELOPMENT OF CUSTOMIZED DRINKING BOTTLE CAP FOR SME

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

by

YONG CHIN YI

B071210114

920227-06-5776

FACULTY OF ENGINEERING TECHNOLOGY

2015

UNIVERSITI TEKNIKAL MALAYSIA MELAKA

BORANG PENGESAHAN STATUS LAPORAN PROJEK SARJANA MUDA

TAJUK: **Design and Development of Customized Drinking Bottle Cap for SME.**

SESI PENGAJIAN: **2015/16 Semester 1**

Saya **YONG CHIN YI**

mengaku membenarkan Laporan PSM ini disimpan di Perpustakaan Universiti Teknikal Malaysia Melaka (UTeM) dengan syarat-syarat kegunaan seperti berikut:

1. Laporan PSM adalah hak milik Universiti Teknikal Malaysia Melaka dan penulis.
2. Perpustakaan Universiti Teknikal Malaysia Melaka dibenarkan membuat salinan untuk tujuan pengajian sahaja dengan izin penulis.
3. Perpustakaan dibenarkan membuat salinan laporan PSM ini sebagai bahan pertukaran antara institusi pengajian tinggi.
4. **Sila tandakan (✓)

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

Disahkan oleh:

Alamat Tetap:

64, Taman Raub Jaya 5,

27600 Raub,

Pahang,

Tarikh: 25/1/2016

Cop Rasmi:

ENGR. HASSAN BIN ATTAN
Jurutera Pengajar Kanan
Jabatan Teknologi Kejuruteraan Pembuatan
Fakulti Teknologi Kejuruteraan
Universiti Teknikal Malaysia Melaka

Tarikh: 25/1/2016

** Jika Laporan PSM ini SULIT atau TERHAD, sila lampirkan surat daripada pihak berkuasa/organisa berkenaan dengan menyatakan sekali sebab dan tempoh laporan PSM ini perlu dikelaskan sebag SULIT atau TERHAD.

DECLARATION

I hereby, declared this report entitled "Design and development of customized drinking bottle cap for SME" is the results of my own research except as cited in references.

Signature : .....

Name : YONG AIN YI

Date : 25/1/2016

APPROVAL

This report is submitted to the Faculty of Engineering Technology of UTeM as a partial fulfillment of the requirements for the degree of Bachelor of Manufacturing Engineering Technology (Product Design) with Honors. The member of the supervisory is as follow:

ENGR. HASSAN BIN ATTAN
Jurutera Pengajar Kanan
Jabatan Teknologi Kejuruteraan Pembuatan
Fakulti Teknologi Kejuruteraan
Universiti Teknikal Malaysia Melaka

.....

(Project Supervisor)

.....

(Project Co-supervisor)

MOHD KAMAL BIN MUSA
Jurutera Pengajar
Jabatan Teknologi Kejuruteraan Pembuatan
Fakulti Teknologi Kejuruteraan
Universiti Teknikal Malaysia Melaka

ABSTRACT

The title of this project is 'Design and development of customized drinking bottle for Small-Medium Enterprises (SME)'. The purpose of this project is to improve the injection mould inserts bottle cap in the lab from two cavities to four cavities and to fasten the processing time for injection of bottle cap. Besides that, this project also purposely wants to fabricate prototype of injection mould inserts for new design bottle cap by using rapid prototyping. In the first part, an introduction of this project was presented. It includes the background of this project, problem statements, objectives, scope, and report organizations. Next, research and literature review regarding to the topic of this project were done. All the research and literature review conducted from journals, articles, books, websites and etc. All the important information get form the research such as plastic injection moulding, patents, and reverse engineering were as the guideline and helped to accomplish this project successfully. In the next step, the methodology was identified. The project was planned with the aid of the Gantt chart and process flow chart. The measurement of the existing injection mould inserts for drinking bottle cap was conducted. Moving on, this project was continued with mould flow simulation analysis to analyse the result of the injection moulding processing time. In the last stage, conclusion of this project was made. The conclusion was the summary of this project. It also including the achievement, significance and problem faced during implementation of project. Lastly, there was some recommendation for the future work.

ABSTRAK

Tajuk projek ini adalah 'Reka bentuk dan pembangunan penutp botol minum disesuaikan untuk Perusahaan Kecil Sederhana (PKS). Tujuan projek ini adalah untuk menambahbaikkan suntikan memasukkan acuan penutup botol di makmal daripada dua rongga hingga empat rongga dan untuk mencepatkan masa pemprosesan untuk suntikan penutup botol. Selain itu, projek ini juga bertujuan untuk menfabrikasi prototaip memasukkan suntikan acuan untuk reka bentuk baru penutup botol.Dalam bahagian pertama, pengenalan projek ini telah dibentangkan. Ia termasuk latar belakang projek ini, pernyataan masalah, objektif, skop, dan organisasi laporan. Seterusnya, penyelidikan dan kajian literatur mengenai dengan topik projek ini telah dilakukan. Semua kajian penyelidikan dan kesusasteraan yang dijalankan daripada jurnal, artikel, buku, laman web dan lain-lain. Semua maklumat yang penting yang berkaitan dengan tajuk projek seperti acuan suntikan plastik, paten, dan teknologi membalikkan kejuruteraan merupakan garis panduan supaya mencapai projek ini dengan jayanya.Metedologi telah dibentagkan dalam langkah terakhir untuk peringkat pertama Projek Sarjana Muda, Projek ini telah dirancang dengan bantuan daripada carta Gantt dan carta aliran proses. Seterusnya, analisis simulasi moldfow telah dijalankan untuk menganalisis suntikan masa pemprosesan acuan. Kesimpulannya adalah ringkasan projek ini. Ia juga termasuk pencapaian, kepentingan dan masalah yang dihadapi semasa pelaksanaan projek. Akhir sekali, terdapat beberapa cadangan untuk kerja-kerja masa depan.

DEDICATIONS

To my parents,

Yong Swee Chuan and Chang Siew Foong

for raising me become who I am today.

ACKNOWLEDGMENTS

First of all, I would like to express my deepest gratitude to my supervisor Engr. Hassan bin Attan for giving me an opportunity working under his supervision throughout this project. Furthermore, I would like to thanks my co-supervisor Mr. Mohd Kamal bin Musa for giving me a helping hand along this project so that I can accomplish my project successfully. This project would not be completed under the time frame without their supervision and guidance.

Moreover, I would like to express thank to the Assistant Engineer Mr. Basri bin Bidin and Mr. Zulkifli bin Jantan. Thanks for all the advices and guidance during my laboratory session. I am very appreciate their alimentation to make my project can run smoothly.

Special thanks to my course mates and friends to had been providing me remarkable ideas to improve this project. Last but not least, I would like to thank my family who gave me physical, emotional and financial support throughout the project.

TABLE OF CONTENTS

LIST OF FIGURES

LIST OF TABLE

LIST OF SYMBOLS AND ABBREVIATIONS

2D	=	Two-dimensional
3D	=	Three-dimensional
CAD	=	Computer-Aided Design
CAE	=	Computer-Aided Engineering
CAM	=	Computer-Aided Manufacturing
CFD	=	Computational Fluid Dynamics
CMM	=	Coordinate Measuring Machine
CNC	=	Computer Numerical Control
FEA	=	Finite Element Analysis
HDPE	=	High Density Polythene
HSM	=	Hardware Security Module
P/L	=	Parting line
PE	=	Polyethylene
PP	=	Polypropylene
RP	=	Rapid Prototyping
SME	=	Small-Medium Enterprises
STL	=	Standard Triangulation Language
UTeM	=	Universiti Teknikal Malaysia Melaka

CHAPTER 1

INTRODUCTION

1.0 Introduction

The purpose of this project is to design and develop customized drinking bottle cap for small-medium enterprises (SME) by using Computer-Aided Design (CAD) and rapid prototyping technology. In this project, design and fabrication of prototype for core and cavity of injection mould inserts for drinking bottle cap are done. There are several methods needed to complete this project such as do research relevant to drinking bottle cap, study the structure and mechanism of the mould and characteristics, and apply Reverse Engineering technology in existing drinking bottle cap and mould base. Besides, design the new bottle cap as well as the core and cavity inserts of mould by using CAD optimization and simulation. The fabrication of prototype which is the injection mould inserts for new design bottle cap is conducted by rapid prototyping.

1.1 Background

A drinking bottle cap is used to seal the top of a drinking bottle. It is normally has the name or logo of the brand of beverage on it. This project is propose to do some research on the existing drinking bottle caps in the market in order to design and develop the customized drinking bottle cap for small and medium-sized enterprises (SME). Reverse Engineering technology is conducted on existing drinking bottle cap and mould base in order to get the CAD data. From the data, the new design and optimization of drinking bottle cap is conducted by using CAD optimization and simulation. Improvement of the injection mould inserts for bottle cap in the lab of Faculty of Engineering Technology UTeM has done from two cavities to four cavities.

1.2 Problem Statement

The current injection moulds for bottle cap in the lab of Faculty of Engineering Technology UTeM only have two cavities. This cause the processing time is long with only two cavities in the mould and the productivity is low for every time injection. The capabilities of facilities in the lab need further improvement.

1.3 Objectives

The objectives of this project are:

i. To improve the injection mould inserts for bottle cap in the lab from two cavities to four cavities.

ii. To fasten the processing time for plastic injection of bottle cap and improve the productivity.

iii. To fabricate prototype of injection mould inserts for new design bottle cap.

1.4 Scope

The scope of this project includes study the functions and structures of the injection moulding and apply Reverse Engineering technology on existing drinking bottle cap and mould base. Next, design and develop the drinking bottle cap and injection mould inserts for bottle cap. Besides that, use rapid prototyping technology to produce the prototype of the caps and its injection mould inserts.

1.5 Report Organization

This project focuses on the design and development of customized drinking bottle cap for SME. In Chapter One, a short introduction to the project is given. It describes briefly about the background, problem statements, objectives and scope of project.

Chapter Two is relevant to the literature review which is the study on the existing bottle cap, plastic injection moulding and Reverse Engineering technology. It provides useful information as the references along this project. Next, Chapter Three is all about the methodology. The procedure and process flow to finish this project are described briefly in this chapter. It includes the process planning, Gantt chart, and process flow to accomplish this project.

Chapter Four describes briefly about the result and discussion regarding to the design of new bottle cap and the injection mould inserts for bottle cap as well as the mould flow simulation analysis. Furthermore, it is also describes about the fabrication of prototype by using rapid prototyping technology. Finally, conclusions about the project are made in Chapter Five which is the summary of this project.

CHAPTER 2

LITERATURE REVIEW

2.0 Introduction

Nowadays, there are many plastics products manufactured by using injection moulding method no matter in the production of consumer or industrial goods. There a few difficult choices are needed to be chosen once the product decided to be made by using injection moulding. For examples, the number of cavities, mould design, ejection method, type of machine and etc. Therefore, there is a lot of literature reviews regarding to injection moulding and reverse engineering needed to be done as the references along this project.

2.1 The Mould Development Process

In the injection moulding process, the product design and mould design are often performed concurrently in order to reduce the product development time. When designing a mould for injection moulding, some information is needed for starting the design process. A product designer may need the process variables and process parameters. Process variables are related to the part geometry, moulding material, demands on the part, lot size and delivery date. Process parameters are the number of cavities, mould dimension, injection machine model and cost estimation for mould and injected parts. David O. K, 2007 stated that a product designer have to go through the following process to fabricate a mould successfully. From the initial design, there is a review for part design and specifications, followed by developing preliminary mould design and quote. If the project is past then it will proceed to the layout design, feed system design, cooling system design, ejector system design, structural system design and machining, polishing, assembly as well as trials. If the moulding is okay, the project is come to an end or else it will return to the step that

might need to do correction and modification. Figure 2.1 shows the development process of mould.

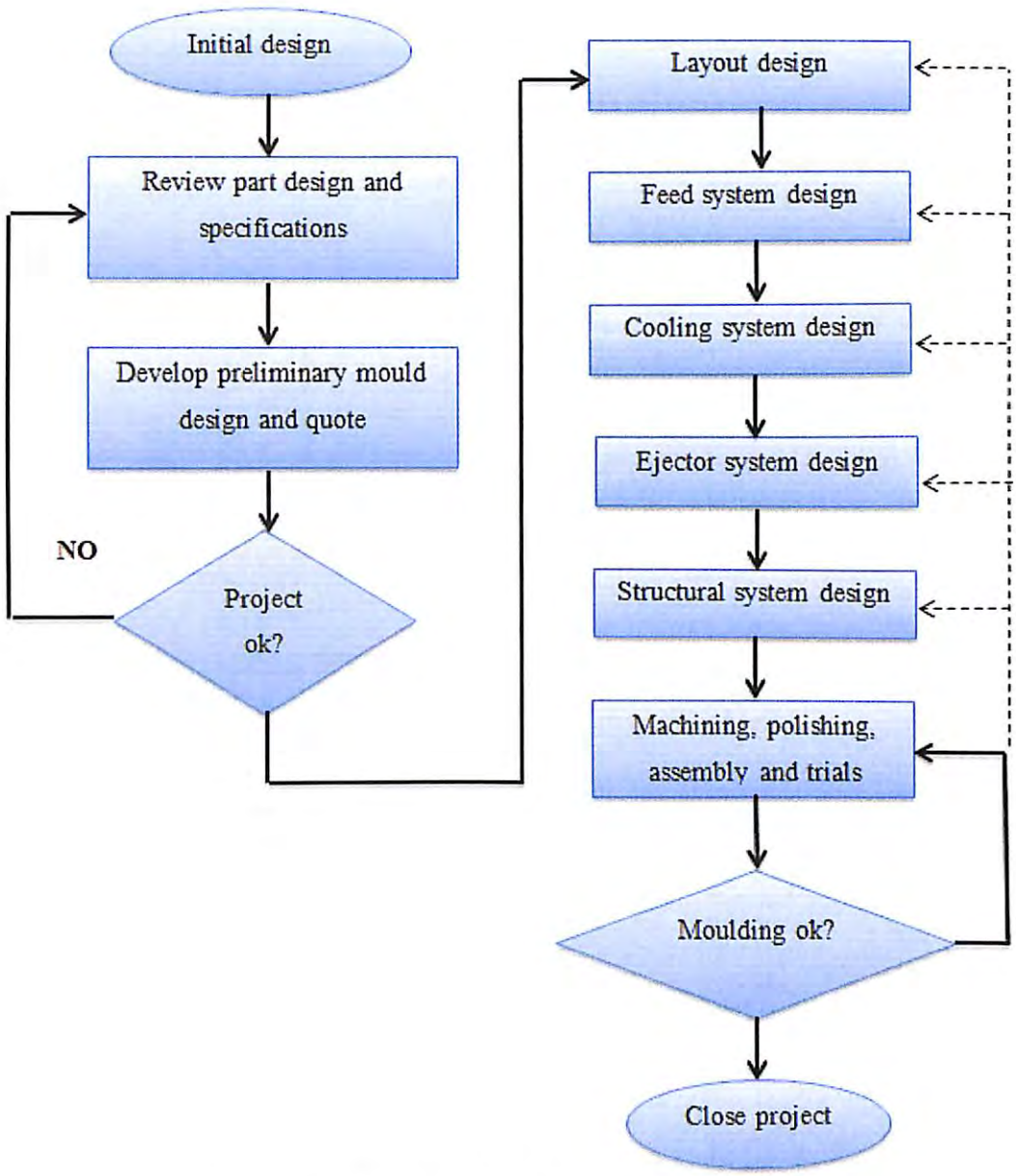

Figure 2.1: The Mould Development Process.

2.2 Introduction of Injection Mould

A manufacturing process that produces parts by injecting plastic into mould is known as injection moulding. The materials used in injection moulding normally are thermoplastic and thermosetting polymers such as Polypropylene (PP), Polyethylene

(PE) and High Density Polythene (HDPE). A mould can be used to make products in infinite variety of shapes by injecting hot plastics into the mould. The mould is made from strong and durable metals, usually either aluminium or steel by using precision machined to form the desired part's features. All moulds must be possible to remove the product after moulding without the need to destroy the mould.

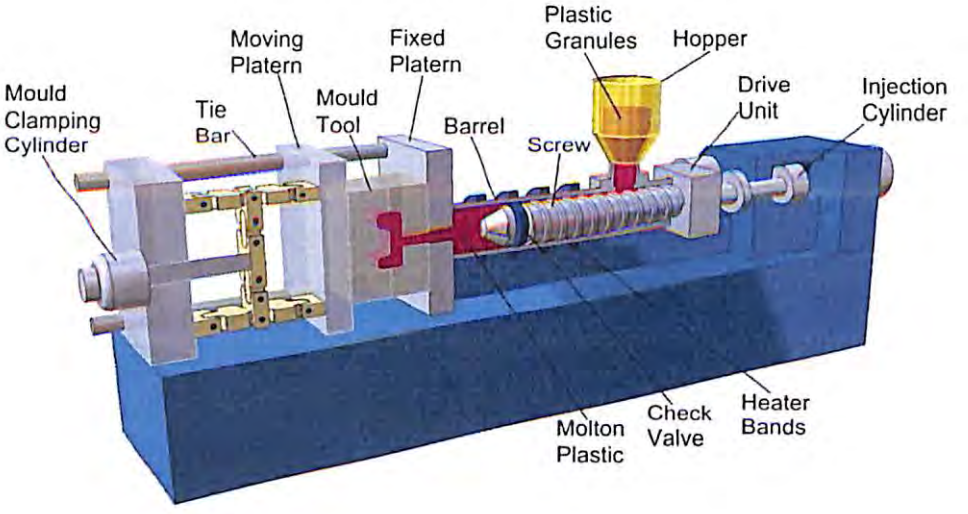

Figure 2.2: Injection Moulding Machine

There are many rules for designing a mould. Therefore, the designers must understand clearly about the basic of mould. This includes the mould cavity space, number of cavities, cavity shape and shrinkage.

2.2.1 Mould Cavity Space

A shape inside the mould is called as the mould cavity space. The plastic is forced into this space and it will take on the shape of the cavity space and form the desire product. Nowadays moulds are usually made from strong and durable materials such as aluminium and metal alloys. This is because of the hot plastic is injected into the cavity space with high pressure during injection moulding. Therefore, the mould must be strong enough to prevent deformation (Herbert R, 2006).

2.2.2 Number of Cavities

Single-cavity moulds are normally used for limited production part or when the part is very large. This can helps to prevent the excessive of size requirement of the injection machine.

Multi-cavity mould refers to there are more than one injected part are made in the same mould. The purpose of multi-cavity mould is to produce multiple identical parts within each mould injection cycle. The production increases in proportion to the number of cavities in mould. Thus, when the more cavities in mould, the more economic in production and the higher the profit.

For a successful multi-cavity mould, the melt conditions introduced to each of the cavities in the mould should be the same and balance. A fishbone or tree runner layout of multi-cavity mould will use less material compare to most runners. Yet, there is a disadvantage of it due to the imbalance filling of cavities when precision moulding is needed. A balanced flow to each of the cavity in multi-cavity mould will help to maximize the potential to produce parts with quality.

There is one type of mould known as family mould, which is the mould with cavities are different from each other, in order to inject different parts with the same mould. The advantage of family mould is only one mould needs to be used to make all the parts in the assembly. However, there are also some disadvantages by using family mould. In order to get a same and balance filling to all of the mould cavities with different shapes simultaneously, the runner system should be sized. There is more complicated of sizing runner system with non-uniform parts. If the cavities not designed properly, it will cause the decrease in part accuracy.

The number of cavities of a mould also depends on the available production time, machine shot size, product quality required, mould costs, shape and size of the moulding as well as the plasticizing capacities. (Anonymous, 2015)

Table 2.1 shows the cavity layout of mould. The arrangement of even number of cavities usually is arranged in a rectangular pattern whereas odd number of cavities arranged in a circular pattern. Figure 2.3 shows the sprue and runner layout for four cavities.