



## **UNIVERSITI TEKNIKAL MALAYSIA MELAKA**

### **SYSTEMATIC DEVELOPMENT OF INJECTION MOLD: A CONVOCATION SOUVENIR**

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

by

**ASMA NOR HIDAYAH BINTI ABDULLAH**

**B071310249**

**910521-06-5858**

FACULTY OF ENGINEERING TECHNOLOGY

2016

## **DECLARATION**

I hereby, declared this report entitled “Systematic Development of Injection Mold: A Convocation Souvenir” is the results of my own research except as cited in references.

Signature : .....

Author's Name : ASMA NOR HIDAYAH BINTI ABDULLAH

Date : 9<sup>th</sup> DECEMBER 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 (Process and Technology) with Honours. The member of the supervisory is as follow:

.....  
Mr. Mohd Faizal Bin Halim  
(Project Supervisor)

## **ABSTRACT**

Injection molding process is the one of the most important processes for the production of plastic parts which offers many advantages to alternatives manufacturing methods, including minimal losses from scrap and minimal finishing requirements. To develop a new plastic product, an injection molding machine requires a new injection mould. The current process to develop a mold for the injection molding needed well knowledge, high cost, time consuming and required experience tool and mould makers. The try an error method were still used as method to overcome the problem arise in injection molding. Usually, the mold will be test on injection molding machine and at that time, all the problem and defect at the product will be defined. However, in order to overcome the problem, there were high cost and time consuming to do the modification to the mold or find the suitable injection molding parameter. In this project, a systematic way was introduced which used several application of CAD/CAE and CAM to develop a mold. CAE software such as Moldflow has been used to find the suitable injection parameter and mold design. Several defects were identified and solved without any modification to the mold.

## ABSTRAK

*Proses acuan suntikan adalah salah satu dari proses penting dalam penghasilan produk plastik dimana proses ini mempunyai kelebihan sebagai satu kaedah pembuatan alternatif, termasuklah kerugian minimum dari lebihan dan minimum keperluan kemasan. Dalam penghasilan sesuatu produk plastik yang baru, mesin acuan suntikan memerlukan acuan yang baru. Pada masa sekarang, proses dalam menghasilkan acuan suntikan memerlukan ilmu pengetahuan yang tinggi, kos yang tinggi, jangka masa yang panjang dan memerlukan pembuat acuan yang berpengalaman. Kaedah cuba gagal masih dianggap sebagai kaedah mencari penyelesaian dalam pembuatan acuan. Lazimnya acuan akan di uji pada mesin suntikan dan pada masa ini segala permasalahan atau kecacatan pada produk dapat dikenalpasti. Namun penyelesaian kepada masalah tersebut memerlukan kos dan masa yang panjang untuk melakukan modifikasi pada acuan atau terus mencari parameter suntikan yang terbaik. Dalam projek ini suatu kaedah yang lebih sistematik diperkenalkan dengan menggunakan beberapa aplikasi CAD/CAE dan CAM untuk menghasilkan acuan. Perisian CAE seperti Autodesk Moldflow telah digunakan untuk mencari parameter suntikan dan semasa rekabentuk acuan. Beberapa kecacatan produk telah dikenalpasti dan diatasi tanpa melibatkan sebarang modifikasi pada acuan.*

## DEDICATIONS

Firstly, dissertation is dedicated to my beloved parent, Mr. Abdullah bin Abd Majid and Mrs. Faezah binti Habas, that encouraged and put their trust on me to complete this project. Not forgetting to my sibling, their supports are the reason why and who I am right now.

My deepest gratitude also were given to Mr. Mohd Faizal bin Halim as my supervisor whose passion for teaching set a new knowledge that can be apply, not only in this project, also can be implement in daily routine.

Other than that, thanks also give to technicians Faculty of Engineering Technology (FTK), Mr. Azimin, Mr. Hisham, Mr. Basri, Mr. Azizul Ikhwan, Mr. Khairul Affendi and Mr. Fauzi those who always gave good co-operation and time in order to complete this research.

Last but not least, dedications were also given to my friends that have been on by my side all the time during completing this project.

## **ACKNOWLEDGMENTS**

Primarily I would like thank to Allah S.W.T for being able to complete this project with success. Then this appreciation to Mr. Mohd Faizal Bin Halim, whose valuable guidance that helped me patches this project. Because of his guidance on suggestion and improvement bring major contributor toward completing of the project.

Thank also goes to all technicians that share their experienced and knowledge during completing this project. I would like to thank to the lecturer in Universiti Teknikal Malaysia Malacca who always gave their support and guidance to me in order to make improvement from time to time.

Lastly thanks to my family and my friend that given their supporting in mental and physically. Their help were the biggest appreciation from me. Thank you for being by my side during completing this project.

# TABLE OF CONTENT

Abstrak	i
Abstract	ii
Dedication	iii
Acknowledgement	iv
Table of Content	v
List of Tables	ix
List of Figures	x
List Abbreviations, Symbol and Nomenclature	xiii
<b>CHAPTER 1: INTRODUCTION</b>	<b>1</b>
1.0 Introduction	1
1.1 Problem statement	4
1.2 Objectives	5
1.3 Scopes	5
<b>CHAPTER 2: LITERATURE REVIEW</b>	<b>6</b>
2.0 Introduction	6
2.1 Traditional Injection Molding Process	6
2.2 Systematic Molding	7
2.3 Injection Molding Machine	9
2.3.1 Process of injection molding	10
2.3.2 Injection Molding Parameter	11
2.4 Molds	12
2.4.1 Parting Line	14
2.4.2 Feed system	14



2.4.2.1	Sprue	14
2.4.2.2	Runner	15
2.4.2.3	Gates	16
2.4.3	Insert Mold	17
2.5	Mold Making Technique	19
2.5.1	Rapid Tooling	20
2.5.2	High Speed Machining	20
2.5.3	Electrical Discharge Machining (EDM)	21
2.6	Mold Flow Simulation	22
2.7	Raw Material	23
2.8	Product Design	24
2.8.1	Solidwork	25
2.8.2	Shrinkage	26
2.8.3	Wall Thickness	28
2.8.4	Draft Angle	29
2.8.5	Corner Design (Fillet, Radius and Chamfer)	30
2.9	Defects in Injection Molding	33
2.10	Summary	36
 <b>CHAPTER 3: METHODOLOGY</b>		<b>37</b>
3.0	Overview	37
3.1	Flowchart	38
3.2	3D designs	39
3.2.1	Design 1	39
3.2.2	Design 2	40
3.2.3	Design 3	40
3.2.4	Design 4	41
3.2.5	Design 5	41
3.3	Designs for Plastic Part	42
3.3.1	Fillet	42
3.3.2	Wall Thickness	43

3.3.3	Draft	43
3.4	Measuring the Insert	45
3.5	Considering the Ejector Pin Location	46
3.6	Mold Flow Simulation	47
3.6.1	Material Selection	47
3.6.2	Part Filling Optimization	48
3.6.2.1	Determine the Number of Gate	48
3.6.2.2	Position the Gate for Balanced Filling	49
3.6.3	Molding Condition	50
3.6.4	Runner System	50
3.7	Core and Cavity Design	51
3.7.1	Parting Line	51
3.7.2	Shut-Off Surface	52
3.7.3	Parting Surface	53
3.7.4	Tooling Split	53
3.7.5	Cavity Design	54
3.7.6	Core Design	56
3.8	Mold Fabrication	57
3.8.1	Squaring Process	57
3.8.2	Machining Program	58
3.8.3	Machining Process	60
3.9	Mold Assembly	61
3.10	Injection Molding Process	63
<b>CHAPTER 4: RESULT AND ANALYSIS</b>		<b>65</b>
4.0	Overview	65
4.1	Analysis on Different Gate Location by Mold Flow Plastic Advisor	65
4.1.1	Result on Melt Temperature 220 (C) for Gate Location 1	70
4.1.2	Result on Melt Temperature 200 (C) for Gate Location 1	71
4.1.3	Result on Melt Temperature 220 (C) for Gate Location 2	72
4.1.4	Result On Melt Temperature 200 (C) for Gate Location 2	73

4.1.5	Discussion on Different Gate Location by Moldflow Advisor	74
4.2	Minimum Defect Analysis on Different Gate Location	76
4.2.1	Result on Air Trap for Gate Location 1	76
4.2.2	Result on Air Trap for Gate Location 2	77
4.2.3	Discussion on Air Trap Result	79
4.2.4	Result on Weld Line for Gate Location 1	79
4.2.5	Result on Weld Line for Gate Location 2	81
4.2.6	Discussion on weld line Result	82
4.3	Mold Fabrication Using 3-Axis CNC Machine	83
4.3.1	Machining Complexity	83
4.3.2	Machining Error	85
4.4	Injection Molding Process	85
4.4.1	Different Result between Simulation and Experimental	86
4.4.2	Part Was Unable To Eject	88
<b>CHAPTER 5: CONCLUSION AND RECOMMENDATION</b>		<b>90</b>
5.1	Conclusion	90
5.2	Recommendation for Future Work	91

## **APPENDICES**

Appendix A	-	Gantt Chart
Appendix B	-	Ejector Pin Tax Invoice
Appendix C	-	Result On CMM Measuring
Appendix D	-	Detail Drawing

## **REFERENCES**

## LIST OF TABLES

2.1	Feed System Types and Properties	17
2.2	Melt Temperature, Residence Time, and Circumferential Speed Recommend For Some Engineering Thermoplastic	24
2.3	Type of Defect and Possible Causes during Injection Molding	33
3.1	Material Quality Indicator by Moldflow Advisor for PP	48
3.2	Cutting Tool Used for Machining	60
4.1	Recommended Process for PP material	67
4.2	Result of Molding Window	67
4.3	Result on Fill Time of Condition used using Moldflow Analysis	69
4.4	Result on Melt Temperature 220 (C) for Gate Location 1	70
4.5	Result on Melt Temperature 200 (C) for Gate Location 1	71
4.6	Result on Melt Temperature 220 (C) for Gate Location 2	72
4.7	Result on Melt Temperature 200 (C) for Gate Location 2	73
4.8	Result on Air Trap for Gate Location 1	76
4.9	Result on Air Trap for Gate Location 2	78
4.10	Result on Weld Line for Gate Location 1	80
4.11	Result on Weld Line for Gate Location 2	81
4.12	Comparison of Simulation and Experimental Result	86
4.13	Comparison between PP and HDPE Recommended Process	87

## LIST OF FIGURES

1.1	Conventional Method of Injection Molding Process	2
1.2	Keris Panjang Is Symbol of the University Authority	3
2.1	Injection Molding Approach with Computer Aided Engineering Tool	9
2.2	Injection Molding Machine	10
2.3	Schematic View of Injection Molding Machine	11
2.4	Process Steps in the Injection Molding Of A Component	12
2.5	Cross Sectional View Of A Common Mold Assembly	13
2.6	Series Layout of Runner System	16
2.7	Cavity Plate for Cavity Insert	18
2.8	Core Plate for Core Insert	19
2.9	Rapid Prototype Machine	20
2.10	EDM Wire Cut Machine	21
2.11	Relationship Between Shrinkage And Characteristic Parameter	27
2.12	Possible Wall Thickness Changes	28
2.13	Exponential Increase Of Cooling Time With Respect To Wall Thickness For Three Common Plastic Materials	29
2.14	Draft On The Inside And Outside Surfaces Of A Part Should Be Equal And Parallel	30
2.15	Proper Design Of Corner In A Plastic Part	31
2.16	Comparison Of Fillet	32
2.17	Comparison Of Chamfer	32
3.1	Process Flowchart	38
3.2	Button Badge	39
3.3	Car Rearview Mirror Hanging	40
3.4	Keychain	40

3.5	Car Dashboard Accessories	41
3.6	Pen Holder	41
3.7	Fillet Design	42
3.8	Wall Thickness	43
3.9	Draft	44
3.10	Detail Drawing for Pen Holder	44
3.11	Coordinate Measuring Machine (CMM)	45
3.12	Measuring Process	45
3.13	Ejector Pin	46
3.14	Location of the Part by Considering the Location of the Ejector Pin	46
3.15	Location Gate 1	49
3.16	Location Gate 2	49
3.17	Runner System	50
3.18	Parting Line	52
3.19	Shut-Off Surface	52
3.20	Parting Surface	53
3.21	Tooling Split	54
3.22	Cavity Plate	55
3.23	Details Drawing For Cavity Insert	55
3.24	Core Plate	56
3.25	Details Drawing For Cavity Insert	56
3.26	Squaring Process Using Milling Machine	58
3.27	Machining Program for Cavity	58
3.28	Result of Machining Program for Cavity	59
3.29	Machining Program For Core	59
3.30	Result of Machining Program for Core	59
3.31	3-Axis CNC Machine	61
3.32	Machining Process of Core Insert Using CNC Machine	61
3.33	Core Insert Assemble To the Core Plate	62
3.34	Mold Assembly Configuration	62
3.35	Fixed Half Assembly	63

3.36	Moving Half Assembly	63
4.1	Gate Location 1	66
4.2	Gate Location 2	66
4.3	Result of Melt Temperature Using Molding Window Analysis on Gate Location 1	68
4.4	Result of Melt Temperature Using Molding Window Analysis on Gate Location 2	68
4.5	Results on Gate Location 1 with 40 (C) Mold Temperatures, 220 (C) Melt Temperature And 180 Maximum Machine Injection Pressure	75
4.6	Results on Core and Cavity Machining	83
4.7	Depth and Narrow Space	84
4.8	Holes Machining	84
4.9	Cutting Tool Mark	85
4.10	Convocation Souvenir by Injection Molding Process	86
4.11	Part Stick to the Mold	88
4.12	Draft Angle Improvement Location	89

## **LIST OF ABBREVIATIONS, SYMBOLS AND NOMENCLATURE**

PP	-	Polypropylene
PS	-	Polystyrene
CAD	-	Computer Aided Design
CAM	-	Computer Aided Manufacturing
CAE	-	Computer Aided Engineering
CNC	-	Computer Numerical Control
3D	-	3 Dimension
EDM	-	Electrical Discharge Machining
MRR	-	Material Rate Removal
s	-	Second
HDPE	-	High Density Polyethylene
ABS	-	Acrylonitrile-Butadiene-Styrene
PC	-	Polycarbonate
C	-	Celsius
m/s	-	meter per second
Vs	-	Versus
CMM	-	Coordinate Measuring Machine
STL	-	Stereolithography
MPa	-	Megapascal
NC	-	Numerical control
KBS	-	Knowledge Based Systems



# **CHAPTER 1**

## **INTRODUCTION**

### **1.0 INTRODUCTION**

In manufacturing, one of the most important processes for the production of the plastic part is injection molding. This process is the most versatile process compare to other plastic processes such as blow molding, extrusion molding, vacuum molding and others. Injection molding is a highly cost-effective, precise and competent manufacturing with high rate and little or no finishing is required on the plastic product. The injection molding process has capabilities to produce part either a simple part or even apart with a complex shape. The application of injection molding is widely used nowadays in many industries such as automotive, aerospace, construction and household product. An example of injection molding product is the bottle cap, key holder, clothes peg, comb and a car bumper.

However, to develop a new plastic part is quite difficult. In current practice there are many problem arise during develop a new molded part. Usually, the mold makers used try an error method to overcome the problem. For example, to get the quality of the component the process parameters like injection time, injection pressure, temperature, clamping force and shear stress, mold maker have making correction repeatedly. These trials are again required if design is changed.

There will be a severe increase in design turnover cost which causes in unproductive time, waste of labor, increase in overhead costs, the complain of customers

because of late in dispatch date and so on. Design and manufacture of the injection mold are time consuming, expensive process and requires highly skilled tool and mold makers.

Typically, the process flow of develop a molded part using injection molding in a current practice is product design, mold design and mold process as shown in Figure 1.1. Basically, cavity layout design depends mostly on engineer expert and knowledge. For example, for the layout patterns, the criteria to select the suitable layout pattern for design are mainly dependent on working surroundings, conditions and demands of the customer and above all based on engineer skill and experience. The design of mold is also influence by several other factors such as part mold material, geometry, parting line and number of cavities per mold. All this contribute to the quality of the molded product as well as production efficiency.

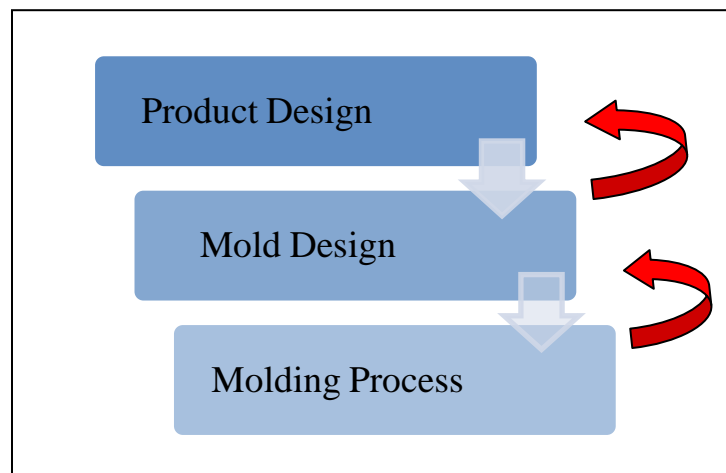


Figure 1.1: Conventional Method of Injection Molding Process.

In this project, a new design for plastic part which is souvenir part for graduation day is selected. Keris Panjang as shown Figure 1.2 will be the main design of the product. This Keris Panjang is the representation of the power of the University. To overcome the problem that may arise during development develop this part, a systematic way is introduced.



Figure 1.2: Keris Panjang is Symbol of The University Authority.

The systematic way used the application of CAD/CAE and CAM by consider the plastic part design, mould design and injection molding parameter and others factors as early as possible in the design stage by get rid of all kind of problems with optimal values without any repetition of production cycle. This systematic way can reduce time consuming, minimizing cost, and ideal use of labor time and in time dispatch with high quality. At the end of this project, a 3D molded part for injection molding (souvenir for graduation day) will be develop. Before that, the core and cavity of the mold will be fabricated. The application of CAE software such as Moldflow will be used to find the suitable injection parameter and mold design.

## **1.1 PROBLEM STATEMENT**

The current plastics industry is under great pressure to survive the competing environment, manufacturing leading times and desire to shorten design process because of the globalization of the market and high demand of the product (Raghavendra et al, 2015). The current processes of injection mold industry use the try an error method on fabricating a mold. If any problem occurs in any stage of the process, then the whole cycle has to be redone on trial an error method which will rise in overhead costs, unproductive working time, waste of labor, the dissatisfaction of customers because of delay in dispatch of product date and so on (Prakash, 2014).

Based on the problem arise, this project are introduced one of the method to overcome the problem which is the systematic way.

## **1.2 OBJECTIVES**

- (a) To develop 3D molded part for injection molding (souvenir for graduation day).
- (b) Fabricate the core and cavity of the mold.
- (c) To analyze injection molding parameter.

## **1.3 SCOPES**

- (a) Design a new plastic part by taking the design concept from existing parts.
- (b) The parts design should be considered manufacture in two plate mold.
- (c) The core cavity of the part will consider the available of existing mold base.

## **CHAPTER 2**

### **LITERATURE REVIEW**

#### **2.0 Introduction**

In this chapter, the data, analysis and information about development of plastic molded part will be stated. Development of plastic molded generally contributes three stage processes which is product design, mold design and injection molding process. Each stage has a different way to be complete.

#### **2.1 Traditional Injection Molding Process**

Since World War II (1940), injection molding process got very high demand. Injection molding has become the most important process for manufacturing plastic parts due to its capability to produce intricate shapes with good dimensional accuracy. In the traditional injection molding, the process to produce the plastic part completely depends on the experience of mold designer. Mold designers are needed to possess thoroughly and widely experience, because the specific decisions require the knowledge of the interaction of various parameters. The traditional injection molding used try an error method to find the suitable method, process, parameter and others before the actual part is produce. For example, in producing the plastic parts, size of the mold fabricate will be bigger than actual size to determine the amount of shrinkage. After determine the shrinkage and suitable parameter of injection molding has been defined, the mold is

once again rebuilt by considering the shrinkage and injection parameter. Consequence of this, the result indicated that conventional method will consume more time and also increases the cost of machining and delay in dispatching the product. If change in design happens before pre-production and after production by trial an error method then there will be a severe increase in design change cost which effects on cycle time, waste of raw material, increase in process time and wastage of labor cost. This lack of fundamental engineering analysis during mold design frequently result in mold that may fail and require extensive rework, produce molding of inferior quality and less cost effective than may have been possible. Conventional method of plastic injection molding is based on the type of artifact, hence the numbers of trials are required to understand like injection time, injection pressure, temperature, clamping force and shear stress (Prakash, 2014). These trials are once again repeated if design is changed.

## **2.2 Systematic Molding**

With the advances in computer technology and artificial intelligence, efforts have been directed to reduce the cost and lead time in the design and manufacture of an injection mold to become more systematic process. Prakash, 2014, expressed that systematic molding is where the product development has changed from the traditional serial process of design, followed by manufacture, to a more organized concurrent process where design and manufacture are considered at a very early stage of design. The pictorial view of systematic concept with computer aided engineering method is shown in Figure 2.1. Injection mold design has been the main area of research since it is a complex process involving several sub-designs related to various components of the mold, each requiring good knowledge and experience. Mold design also affects the productivity, manufacturability of mold, mold maintenance cost and the quality of the molded part. Most of the work in mold design has been directed to the application of Knowledge Based Systems (KBS) and artificial intelligence to eliminate or supplement the vast amount of human expertise required in traditional design process. Several

studies have also been made on improving the design of specific components of an injection mould.

- i. Computer Simulation for Finding Optimum Gate Location in Plastic Injection Molding Process by Babu and Vardan (2013). In this project an analysis has been performed to investigate different gate locations for a head light cover of an Alto car a plastic component. Plastic advisor simulation tool from Pro/E was used for the analysis to optimize the gate location with slightest defects and it requires less time to reach a quality result with no material waste as compared with conventional trial error method.
- ii. Sustainable injection molding: The impact of materials selection and gate location on part warpage and injection pressure by Huszar et al. (2015). This research comes up to on how the warpage (i.e. part deflection) and injection pressure of an intricate. The computational work used Moldflow Insight's "Fill + Pack +Warpage" analysis sequence to investigate the molding characteristics of a partially symmetrical, hollow geometry using four potential gate locations geometry could be minimized by selecting an maximum thermoplastic material and injection gate location (through which the molten plastic flows into the cavity). The numerical analyses for mold filling considered four gate locations along with a PP (polypropylene), PS (polystyrene) and a fibre-filled PP material. Each had different shrinkage characteristics, mechanical property and viscosity.
- iii. Gating System Design Optimization for Injection Molding by Dr. M P Singh et al. (2015) develop a system with the help of CAD or CAM which facilitates the user to work on injection molding operations starting from the cavity to the selection of gates and then to the efficiency of runner and finally to sprue. This results in time reduction of designing process and to a certain extent also reduces manufacturing cost.
- iv. A Design and Molding Analysis of Two Plate Mould Tool for Motor Rare Housing Thermoplastic Product by Raghavendra N et al. (2015). The purpose of this research is to design two plate mold tool and analyze the material flow