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 :  9th DECEMBER 2016
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 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 and 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

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

<table>
<thead>
<tr>
<th>Section</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>Abstrak</td>
<td>i</td>
</tr>
<tr>
<td>Abstract</td>
<td>ii</td>
</tr>
<tr>
<td>Dedication</td>
<td>iii</td>
</tr>
<tr>
<td>Acknowledgement</td>
<td>iv</td>
</tr>
<tr>
<td>Table of Content</td>
<td>v</td>
</tr>
<tr>
<td>List of Tables</td>
<td>ix</td>
</tr>
<tr>
<td>List of Figures</td>
<td>x</td>
</tr>
<tr>
<td>List Abbreviations, Symbol and Nomenclature</td>
<td>xiii</td>
</tr>
</tbody>
</table>

**CHAPTER 1: INTRODUCTION**

1.0 Introduction 1
1.1 Problem statement 4
1.2 Objectives 5
1.3 Scopes 5

**CHAPTER 2: LITERATURE REVIEW**

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
3.3.3 Draft
3.4 Measuring the Insert
3.5 Considering the Ejector Pin Location
3.6 Mold Flow Simulation
   3.6.1 Material Selection
   3.6.2 Part Filling Optimization
      3.6.2.1 Determine the Number of Gate
      3.6.2.2 Position the Gate for Balanced Filling
   3.6.3 Molding Condition
   3.6.4 Runner System
3.7 Core and Cavity Design
   3.7.1 Parting Line
   3.7.2 Shut-Off Surface
   3.7.3 Parting Surface
   3.7.4 Tooling Split
   3.7.5 Cavity Design
   3.7.6 Core Design
3.8 Mold Fabrication
   3.8.1 Squaring Process
   3.8.2 Machining Program
   3.8.3 Machining Process
3.9 Mold Assembly
3.10 Injection Molding Process

CHAPTER 4: RESULT AND ANALYSIS

4.0 Overview
4.1 Analysis on Different Gate Location by Mold Flow Plastic Advisor
   4.1.1 Result on Melt Temperature 220 (C) for Gate Location 1
   4.1.2 Result on Melt Temperature 200 (C) for Gate Location 1
   4.1.3 Result on Melt Temperature 220 (C) for Gate Location 2
   4.1.4 Result On Melt Temperature 200 (C) for Gate Location 2
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

CHAPTER 5: CONCLUSION AND RECOMMENDATION 90

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

4.1 Gate Location 1
4.2 Gate Location 2
4.3 Result of Melt Temperature Using Molding Window Analysis on Gate Location 1
4.4 Result of Melt Temperature Using Molding Window Analysis on Gate Location 2
4.5 Results on Gate Location 1 with 40 (C) Mold Temperatures, 220 (C) Melt Temperature And 180 Maximum Machine Injection Pressure
4.6 Results on Core and Cavity Machining
4.7 Depth and Narrow Space
4.8 Holes Machining
4.9 Cutting Tool Mark
4.10 Convocation Souvenir by Injection Molding Process
4.11 Part Stick to the Mold
4.12 Draft Angle Improvement Location
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 compared 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 process with a high rate and little or no finishing required on the plastic product. The injection molding process has the capability to produce parts, whether simple or complex in shape. The application of injection molding is widely used nowadays in many industries such as automotive, aerospace, construction, and household products. 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 problems that arise during the development of a new molded part. Usually, the mold makers use a trial-and-error method to overcome these problems. For example, to get the quality of the component, the process parameters like injection time, injection pressure, temperature, clamping force, and shear stress, mold makers have to make corrections repeatedly. These trials are again required if the design is changed.

There will be a severe increase in design turnover cost which causes 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.](image)

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 Processs

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 and 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