DESIGN AND ANALYSIS OF PLASTIC INJECTION MOULD USING MULTIPLE CAVITY MOULD

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

(Manufacturing Design) (Hons)

by

MOHD IZZAT BIN ZAINAL ABIDIN

B051310233

940618-14-6467

FACULTY OF MANUFACTURING ENGINEERING

2017

DECLARATION

I hereby, declared this report entitled “Design and Analysis of Plastic Injection Mould using Multiple Cavity Mould” is the results of my own research except as cited in reference.

Signature :

Author’s Name: MOHD IZZAT BIN ZAINAL ABIDIN

Date :

APPROVAL

This report is submitted to the Faculty of Manufacturing Engineering of Universiti Teknikal Malaysia Melaka as a partial fulfillment of the requirements for the degree of Bachelor of Manufacturing Engineering (Manufacturing Design) (Hons.).

The member of the supervisory committee are as follows:

Dr Rosidah Binti Jaafar- Signature & Stamp

ABSTRAK

Kajian ini dijalankan untuk mereka bentuk dan analisis produk plastik dalam acuan suntikan plastik menggunakan pelbagai acuan rongga sebagai projek utamanya. Pakaian dilipat Hanger direka sebagai sebahagian plastik dalam pelbagai acuan suntikan rongga. perisian SolidWork digunakan untuk mereka bentuk bahagian plastik dan Moldflow Software digunakan untuk melakukan analisis pengisian dan pembungkusan. Kajian ini membincangkan aliran plastik lebur dalam proses suntikan acuan. Kajian dilakukan dengan mempertimbangkan pintu yang sesuai untuk menempatkan sebahagian yang dipilih dalam bentuk tiga plat acuan suntikan. Kemudian, Moldflow Mold Penasihat digunakan untuk menganalisis kesan aliran bahan untuk seriawan, pelari dan pintu untuk acuan tiga plat. Di peringkat akhir, Moldflow Mold Penasihat membantu dalam simulasi cadangkan kedudukan yang terbaik lokasi gate, masa suntikan setara dan suhu lebur semasa proses pengacuan suntikan plastik, untuk menunjukkan tanda sink, talian weld, memerangkap udara dianggarkan dan orientasi kulit berdasarkan gate lokasi, reka bentuk bahagian dan digunakan. Analisis Moldflow Mold Penasihat juga membantu untuk menentukan sebahagian kecacatan yang mungkin berlaku semasa proses pengacuan suntikan plastik seperti pukulan pendek, tidak sama rata mengisi, lebih pengisian dan lain-lain. Selain itu, ia dianalisis warping dan penyejukan kualiti dianggarkan berdasarkan bahagian dan saluran penyejukan direka. Untuk jumlah semuanya, reka bentuk dan analisis pelbagai acuan rongga dapat dilakukan dengan menggunakan perisian ini dan hujan yang kajian telah dicapai di atas kertas ini..

ABSTRACT

This study was conducted to design and analysis of a plastic product in plastic injection mould using multiple cavity mould as its main project. Foldable Clothes Hanger are designed as the plastic part in the multiple cavity injection mould. SolidWork software is used to design the plastic part and MoldFlow Software is used to perform the analysis of filling and packing. This research discussed the flow of molten plastic inside the injection mould process. The study done by considering the suitable gate to locate for the selected part in designing three-plate injection moulds. Then, the Moldflow Mold Adviser is used to analyze the effect of the material flow to sprue, runner and gate for three-plate moulds. In the final stage, Moldflow Mold Adviser helps in simulation to suggest the best position of gate location, equivalent injection time and melting temperature during plastic injection moulding process, to show the sink mark, weld line, air trap estimated and skin orientation based on gate location, part design and used. The analysis of MoldFlow Mold Adviser also helps to determine the part defect that might occurred during plastic injection moulding process such as short shot, unequal filling, over filling and others. Besides that, it analyzed the warping and cooling quality estimated based on part and cooling channel designed. To sum it all up, design and analysis of multiple cavity mould were able to be done by using this software and the significant of studies were achieved on this paper.

DEDICATION

Only

my beloved father, Zainal Abidin

my appreciated mother, Rohati

my supervisor, Dr Rosidah

for giving me moral support, cooperation, encouragement and also understandings

Thank You So Much For The Encouragement

ACKNOWLEDGEMENT

My appreciation goes towards my supervisor, Dr Rosidah Binti Jaafar for her guidance and encouragement throughout the progress of this project. Without her critiques that is very constructive, this project would be able to be accomplished

Special thanks goes to my parents, Zainal Abidin and Rohati for their endless support into finishing this project. Not to forget to En Hairuddin for helping me with the design and analysis of this project who also helped to create a 3D printing prototype in his lab.

Finally, a special thanks to friends who gave a final push and tremendous courage to finish the project together. Their support was invaluable.

TABLE OF CONTENTS

CHAPTER 3: METHODOLOGY

CHAPTER 4: RESULT AND DISCUSSION

LIST OF TABLES

LIST OF FIGURES

LIST OF ABBREVIATIONS

Three Dimensional	–	3D
Design of Experiment	–	DOE
Mold Plastic Advisor	–	MPA
Acrylonitrile Butadiene Styrene	–	ABS

LIST OF SYMBOLS

°C	–	Degree Celcius
%	–	Percentage

CHAPTER 1

INTRODUCTION

1.1 Background

The ever growing industry in this world in terms of many field is the plastic industry. Over the years, plastic material has evolved into many things such plastic polymers into creating and innovating products that could only be an imagination to the old age. Plastic also starts to replace the old conventional materials such as metals, ceramic and paper. Plastic is considered to be dynamic in terms of its chemical and physical properties as it is easily manipulated into different things according to requirement. Thus, this makes the possibility of making cheap every day product higher than before.

Nowadays, researcher in the plastic industry is investing heavily into plastic that is cheap, strong and biodegradable as environmental issues arises with the old plastic type. The conventional synthetic polymer which obtained from petrochemical-derived monomers is non-biodegradable resulted to the environmental issues (Sikorska et al., 2008; Musiol et al., 2011). From the objective, researchers came up with Acrylonitrile butadiene styrene (ABS), which is a thermoplastic polymer that is biodegradable or recyclable. Not only that, this plastic also has a high impact resistance, toughness and heat resistance

Injection moulding is one of the cost effective process in this challenging economical industries. Its technology has been improved tremendously compared to the past which is not

viable for the industry. With the innovation of plastic products, this process is the most suitable into producing various types of polymeric materials with complex geometry and shapes with low manufacture and cost (Chen et al., 2009). Plastic injection moulding process involves three main stages in each cycle. Firstly, hot plastic material are injected into the mould cavity at injection temperature. In cooling channels, the heat of the plastic is dispersed out and will be solidified and ejected out of the mould (Hassan et al., 2009). Many researchers found that the injection moulding parameters have the crucial effects on designing the economical and good quality of mould for thermoplastic product (Kwong et al., 1997; Lotti et al., 2002; Chen et al., 2005; Patcharaphun and Mennig, 2007).

In single cavity mould, only one product can be created at a single time using that mould. While in multiple cavity, multiple product can be produced using the same mould. Other than that, single cavity uses only two plate mould that parameters of plastic injection moulding can be set easily as compared as multiple cavity that uses three or more plate mould and is prone to defects due to the varied parameters.

As in modern industries, the time consuming non-simulation approaches are substituted with computer aided engineering (CAE) analysis software such as Moldflow® , C-MOLD™, and Moldex3D® . These CAE software assisted in injection moulding simulation by providing output results such as flow pattern, fill time, air traps, frozen layer fraction, orientation at skin, weld lines, etc. which virtually explained the flow pattern of the melted polymer in the mould during filling, packing and cooling stages (Moldflow Corporation, 2004). Moreover, injection moulding simulation not only helps in modelling the process and flow pattern analysis, it also developed the visual and numerical feedback interpretation results as the guidance in achieving optimum moulding parameters, compatibility of materials used, and reduced the process cycle time and cost expenses in mould modification.

1.2 Problem Statement

Multi-Cavity Mould process has a lot of parameters that can affect the final products of the moulded parts. Multi cavity mould also could be used to produce the same product with the same material and parameters to improve production rate or to produce different product with different parameters and material. The parameters that need to be studied is the injection temperature, pressure, time and its cooling time during solidifying process.

Injection Moulding has come a long way since its invention and throughout the time, humans have been innovating injection moulding to be more efficient and cost effective. Multi-cavity injection moulding was used in manufacturing industry to shorten manufacturing time greatly. Though this innovation was great to the industry, it has its flaws.

The most common problems with multi-cavity mould is uneven filling to the mould. Multiple Cavity Moulding may cause the material used to be melted partially that disrupts the flow into the runner which cause the runner to be stuck with unmelted. There were studies proven that the differences in mould temperature results in problematic plastic parts like warpage (Beaumont, 2004). These kind of problems will cause the production to stop functioning as repair and/or maintenance is needed on the machine. This will cost the industry time, expert labor, money and etc.

The purpose of this paper is to avoid multi-cavity mould to fill unevenly during injection process by considering the critical parameters during design process of the mould which is number of cavities, methods of injection, types of runners, gating method, methods of injection, capacity and features of the injection moulding machines.

1.3 Objectives

There are several objectives that is established in order to undergo this study:

1. To investigate the cause of non-uniform melted plastic on runner
2. To investigate the effect of molten polymer temperature on the finishing product
3. To investigate the optimum velocity control (pressure) during filling

1.4 Scopes

The scopes of this project is rather limited to follow the title of this report and achieving the objectives that have been stated above. First of all, creating and designing a plastic product is a must. Next would be able to do literature studies on multiple cavity mould and plastic injection moulding itself. Furthermore, utilizing the CAE software is needed in order to analyze the mould based on parameter and its defect to complete the objectives. Lastly would be using CAD software in designing the product and mould itself.

1.5 Significance of Study

The purpose of this paper is to design and analyze multi-cavity plastic injection moulding for the selected product by designing a new optimized and analyzed mould. The significance of this paper would be :

- Able to understand mould structure for injection moulding process
- Able to design multi-cavity with three plate mould for the part and/or product selected
- Able to use MoldFlow software to do analysis on the material flow and other optimization for the mould designed

CHAPTER 2

LITERATURE REVIEW

2.1 Plastic Injection Moulding

Plastic injection moulding is a process of injecting molten polymer or plastic through a small gap called gate through a closed mould. Injecting the molten plastic uses high pressure that is directly inverse from the product's shape through the mould. Then, molten plastic will start to solidifies into crystallize polymer inside the mould and final product is obtained by removing the plate mould.

H.T. Paro et al (1986) found that injection moulding is one of the most important polymer processing methods for producing plastic parts. Process parameters in addition to moulding material and part design are major factors affecting the quality of plastic parts produced by injection moulding. Quality of these parts is often associated with warpage. Effects of process parameters on non-uniform shrinkage leading to warpage are investigated from several aspects in literature (A.I. Isayav, Marcel Dekker, 1987).

Calculations in injection moulding is a complex one that requires every type of resin unique shrinkage value to be factored in and the mould used must compensate for it. Unprecise calculation will cause the final product to be incorrectly sized with many other flaws. Though, this problem could be solved by filling the mould with resin, holding it under pressure and adding more resin to compensate for contraction if there is any. Other complications may include

burned parts resulting from the melt temperature being set too high, warpage resulting from an uneven surface temperature, or incomplete filling due to a too slow of an injection stroke.

2.1.1 Injection moulding stages

Injection moulding is a cyclic process where the molten polymer is inserted into the mould cavity then solidifies into the desired part or product based on the mould. Through every cycle, there are three significant stages. To begin with, melted polymer is injected into the mould cavity based on its parameter temperature which is then called by filling and post-filling stage. After that, the heat for the polymer is cut off from the mould to let the channels to be cooled down which is the cooling stage. Lastly, when the plastic part is solidifies, it is ejected out from the mould (Hassan et al., 2009).

2.1.1.1 Filling and packing stage

A viscosity function or a model is required to model the injection moulding process for different types of polymer. In this stage, viscosity model is one of the most important factors that could influence the moulding parameters. In order for the filling process to be in control, the high viscosity polymers need to be in laminar flow. The turbulence generated will cause the process to be out of control and causes different multiple flow as the chain reaction on the surface or within the completed plastic part.

Local viscosity will increased significantly when the polymer flowing inside the mould comes in contact with the mould surface with lower temperature compared to the molten plastic temperature and will produce no flow of polymer against the mould wall. The continuous flow of polymer is insulated by the non-flowing polymer from the cold mould wall. Thickness of the