MOULD FLOW ANALYSIS BASED PRODUCT DEVELOPMENT FOR PLASTIC INJECTION MOULDING PROCESS

Submitted in accordance with the requirement of the Universiti Teknikal Malaysia Melaka (UTeM) for the Bachelor Degree of Manufacturing Engineering (Hons.)

By

MUHAMMAD ADIB BIN ISMAIL

B051520018

950407-10-5699

FACULTY OF MANUFACTURING ENGINEERING

2019

UNIVERSITI TEKNIKAL MALAYSIA MELAKA

BORANG PENGESAHAN STATUS LAPORAN PROJEK SARJANA MUDA

Tajuk: **MOULD FLOW ANALYSIS BASED PRODUCT DEVELOPMENT FOR PLASTIC INJECTION MOULDING PROCESS**

Sesi Pengajian: **2018/2019 Semester 1**

Saya **MUHAMMAD ADIB BIN ISMAIL (950407-10-5699)**

mengaku membenarkan Laporan Projek Sarjana Muda (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 yang berdarjah keselamatan atau kepentingan Malaysiasebagaimana yang termaktub dalam AKTA RAHSIA RASMI 1972)

☐ TERHAD (Mengandungi maklumat TERHAD yang telah ditentukan oleh organisasi/ badan di mana penyelidikan dijalankan)

☐ TIDAK TERHAD

Disahkan oleh:

Alamat Tetap:
No 459, Taman Cermai Fasa 1,
Jalan Bukit Wang, 06000,
Jitra, Kedah Darul Aman.

Tarikh: 9th JANUARY 2019

Cop Rasmi:

Tarikh: 9th JANUARY 2019

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

DECLARATION

I hereby, declared this report entitled "Mould Flow Analysis Based Product Development for Plastic Injection Moulding Process" is the results of my own research except as cited in reference.

Signature :……..

Author's Name : MUHAMMAD ADIB BIN ISMAIL

Date : 9th JANUARY 2019

APPROVAL

This report is submitted to the Faculty of Manufacturing Engineering of Universiti Teknikal Malaysia Melaka as a partial fulfilment of the requirement for the degree of Bachelor of Manufacturing Engineering (Hons.). The members of the supervisory committee are as follow:

……..

(Dr Rosidah Bt Jaafar)

ABSTRAK

Tujuan kajian ini adalah untuk menilai analisis aliran acuan berdasarkan perkembangan produk untuk pengacuan suntikan plastik. Pertama, plastik adalah reka bentuk berdasarkan keperluan produk yang diperlukan dari borang tinjauan. Dengan menggunakan reka bentuk dibantu komputer (CAD) dari perisian Autodesk Inventor 2017, bahagian plastik telah ditarik dalam 3 dimensi (3D) kemudiannya meneruskan analisis aliran acuan untuk menganalisis bahagian plastik. Seterusnya, dengan menggunakan perisian Autodesk Mold Flow Adviser 2018, ia membolehkan untuk menghasilkan simulasi maya bagaimana aliran plastik dalam acuan. Daripada simulasi, ia boleh dianalisis parameter yang paling sesuai untuk pengacuan suntikan seperti masa mengisi, tekanan suntikan, suhu di hadapan aliran, variance suhu, lapisan beku, kualiti penyejukan dan kecacatan yang berlaku di bahagian plastik seperti perangkap udara, weld garis dan tanda tenggelam. Dalam simulasi bahan yang digunakan ialah Acrylonitrile butadiene styrene (ABS). Akhir sekali, selepas simulasi dilakukan, teruskan membuat reka bentuk acuan menggunakan dua acuan asas acuan. Komponen asas acuan dibuat dengan menggunakan Autodesk Inventor 2017 dengan menggunakan asas acuan bahagian standard FUTABA.

ABSTRACT

The aim of this study is to evaluate the mould flow analysis based on the product development for plastic injection molding. Firstly, the plastic was design based on the product requirement that required from the survey form. By using computer aided design (CAD) from the Autodesk Inventor 2017 software, the plastic part were drawn in 3 dimension (3D) then its proceed to mould flow analysis for analysing the plastic part. Next, by using Autodesk Mould Flow Adviser 2018 software, it allows to generate virtual simulation how the plastic flow in the mould. From the simulation it can be analysed the most suitable parameter for injection moulding such as fill time, injection pressure, temperature at flow front, temperature variance, frozen layer, cooling quality and the defect that occur in the plastic part such as air traps, weld line and sink marks. In the simulation the material that used is Acrylonitrile butadiene styrene (ABS). Lastly, after the simulation was done,proceed to make mould design by using two plate mould base. The component of the mould base is making by using Autodesk Inventor 2017 by using FUTABA standard part mould base.

DEDICATION

This project work is dedicated to my parent and my supervisor, who thought us different roads of life and direct us towards our destination. To all those friends and companions who helped us while doing this work and made the journey of university career easier. I also dedicate this work to my lovely father; Ismail Bin Ahmad and appreciated mother; Nadzifah Binti Md Noor who has encouraged me all the way and whose encouragement has made sure that I give it all it takes to finish that which I have started. Thank you. My love for you all can never be quantified. God bless you.

ACKNOWLEDGEMENT

First of all, I am deeply grateful to Allah. Comingfrom the intense gratification of my supervisors whom their contribution has raised the quality of this thesis. They have always supported me and have given me enthusiasm for for this project. They have patiently guided me. I am very grateful to their supervision and I owe them the greatest degree of appreciation I am deeply grateful and Iwish to express my deepest gratitude to Dr Rosidah Binti Jaafaar

I would like to take this opportunity to say warm thanks to all my beloved friends, who have been so supportive along the way of doing my final year project. They had given their critical suggestion and comments throughout my research Thanks for the great friendship.

I also would like to express my wholehearted thanks to my family for their generous support they provided me throughout my entire life and particularly through the process of pursuing the degree. Because of their unconditional love and prayers, I have the chance to complete this final year project

TABLE OF CONTENTS

LIST OF TABLE

LIST OF FIGURE

LIST OF ABBREVIATIONS

3D	-	3 dimensional
BBA	-	Boolean Base Approach
BRDO	-	Boolean regularized difference operation
CAD	-	Computer Aided Design
CAE	-	Computer Aided Engineeering
DCC	-	Direct Computer Control
FDM	-	Fused Deposition Modelling
MFA	-	Moldflow Analysis
RE	-	Reverse Engineering

LIST OF SYMBOLS

±	-	tolerance
Mm	-	millimetre
ºC	-	degree calcius
MPa	-	Megapascal

CHAPTER 1

INTRODUCTION

This research focus on the analytical studies with regard to mould flow analysis for polypropylene/ recycle polypropylene blend based product development of injection moulding process. This chapter will introduce background of study for this project, problem statement, objectives and scope of the project.

1.1 Research background

Injection moulding process is the most common method to create parts of plastic material. This method typically used for thermoplastic materials where it may be successively melted, reshaped and cooled. After a designer had been designed the product, moulds are made by a mould maker or toolmaker from metal. Commonly the mould is made from steel or aluminium and use an accurate machine to produce the features of the desired part. This moulding method is usually used for manufacturing a various parts, from the smallest parts of the biggest parts such as in automotive parts. The process of injection moulding includes injecting molten plastic at high pressure into a mould shaped in the form of a part. Then, the plastic will cool and solidifies, the mould open and the product is ejected out from the mould. Injection moulding parts can be simple and complex based on the customer requirement. It also can be solid, foamed, reinforced or filled. This moulding can produce small, large, thick, flexible and rigid product.

The appliance of injection moulding is used to create many things such as wire spool, packaging, bottle automotive part and others part. Plastic moulding machine is newly used in manufacturing plastic parts or rubber parts and components. This process is the best way

of producing high volume of the same design product. Besides that, it also can give high production rates, repeatable high tolerances, and the ability to use a wide range of materials. The important one is the labour cost can be lower, minimal scrap losses and also reduce defect of the product. Injection moulding also have disadvantages such as need costly equipment investment, potentially high running cost and also his costing in machining mould parts. Lastly, the need to design mouldable parts also the disadvantage of this method.

1.2 Problem Statement

Car grab handle help to assist the passengers get out of the car. They can reach up and use the handle to assist raise themselves out of the seat. The driver does not have one as a result of they need the steering wheel, they will push out on that. These handles assist you to sit down well once the vehicle is in motion and also the car is taking sharp turns. These handles offer the passengers a feeling of ease. It is same like holding a public bus handle that is supposed to provide the passengers a body support.

The project is the improvement of the existing product to a new innovative product. The product that needs to innovate is car grab handle. Other utilizes incorporate going about as a place to hang clothing. The users that always go for outstation job, they need to hang their clothes because their clothes had been ironed. So, they had to find a way to transport their ironed out clothes in their car. They commonly hang the clothes on the car grab handles. We all know that this is not the best way to hang the clothes in your car. The hangers keep on banging the window and also have potential the clothes that hang on the car grab handle will fall on the car floor because of the vibration that receive.

Besides than that, the hanger will slide through car grab handle follow the momentum of the car. So the surface of car grab handle become rough due to the friction that receive from sliding hanger. When the surface of car grab handle become rough it will give uncomfortable situation to the user to grab it.

This project intends to make a new innovation in this existing product by using design development of this product by using survey from the user. Then by making mold flow analysis to see the quality of the product and make injection moulding design.

1.3 Objective

The objectives of this project are as follows:

i. To investigate the design parameter for car grab handle.
ii. To analyse the parameter in injection moulding process by using mould flow analysis.
iii. To study the design of making an injection moulded part and system.

1.4 Scope of project

The scopes of this project focus the user that always goes for outstation. They need to hang their clothes that had been ironed in properly in the car. The function of this product is to involve the hanger of the shirt without damaging the surface of the car hand bar and gives the grip of the clothes hanger so it does not slide and falls down. This project are focused on designing and then prototyping the part itself to specification. After getting the product requirement from survey form, the design product is making by using Autodesk Inventor based on product requirement. The initial prototyping, development is sometimes made up on a fused deposition modelling (FDM) based on the material that we want to use than the final part are created in. This FDM printer is utilization of 3 dimensional knowledge to accumulate a multitude of X, Y, Z coordinates on the surface of an object. After that the scope of this project is to analyse the design of the injection moulding system which is the gate locations, the sprue, the runners, cooling channel, the material flow in the mould and the whole set of the injection moulding. The mould geometry design and analysis using Autodesk Inventor Professional 2017 and Autodesk Mould Flow software.

CHAPTER 2

LITERATURE REVIEW

2.1 Introduction

This chapter will explain theory and literature review on topics that related to injection moulding process. Related data from the past studies is gather as a reference and discussed based on their research on mould flow analysis for polypropylene/ recycle polypropylene blend based product development for injection moulding process. The topics that covers in this chapter is product design, mould design, mould flow analysis, and lastly is injection moulding process.

2.1.1 Design product

Nowadays, people express their ideas through design and come out with a product. Product design start with some research before and become a problem statement and then develop the conceptual based on the problem that want to solve (Morris, 2009). Design product need to face with difficulty to understand the customer requirement and transfer the customer requirement into products (Cagan & Vogel, 2008). Need to focus on the social, economic and technology to come out with a better product design and can expose the targeted market with the product.

The main thing to generate product design is needed to know the opportunity, conceptualisation and the realization of that opportunity. A product that wants to market should be have some attraction to the eye of customer that have perceived value which is aesthetics, identity of the product, ergonomics and must have good quality. So, product design can assist the designer to express their decision based on the customers requirement

2.1.2 Reverse engineering

Reverse engineering (RE) is currently thought of one among the technologies that offer business advantages in shortening the product development cycle (Raja, 2008). Reverse engineering begins with an existing product. For something to be redesigned, it obviously must be pre-existing. Reverse engineering also uses to improve our product, as well to analyse a challenger product and to analysing a subject matter system to spot the system's elements and their interrelationships and make illustrations of the system in another form or at the next level of conception.

According to the Abella *et al.,* (1994) the basic conception of manufacturing a part based on a resourceful or physical model without the utilization of an engineering drawing. The process of retrieving new geometry of a manufactured part by digitizing and modifying an existing CAD model (Yau *et al*., 1993). Designing in CAD becomes more difficult once product become additionally complicated in form, and there is no guarantee that the CAD illustration can replicate the sculpted model precisely. Reverse engineering will solve this problem because the physical model is the supply of data for the CAD model. This can be additionally stated as the physical-to-digital process represented in Figure 2.1 reverse engineering can also reduce product development cycle times.

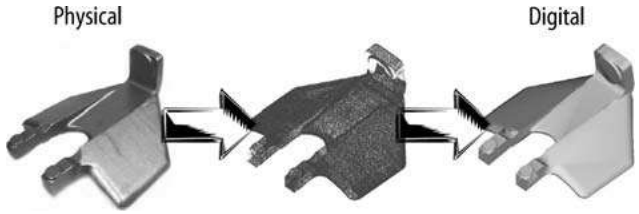

Figure 2.1: Product development life cycle where physical to digital process (Raja, 2008)

2.2 Mould design

Based on Shen, (2010), the mould design should basically specify a tool that may operate satisfactorily in production. To design the mould, the designer needs to achieve a few objectives such as:

i. It should operate at the desired production rate or higher and last a minimum of for the expected design life.
ii. Need well design and produce to required specification.
iii. It must not be liable to frequent breakdowns and may not need frequent maintenance or servicing

According to the Mikó, (2013) an injection mould is a customize product so the production process also need to customize. The workflow of design process will explain from the initial process to the end process as shown in Figure 2.2.

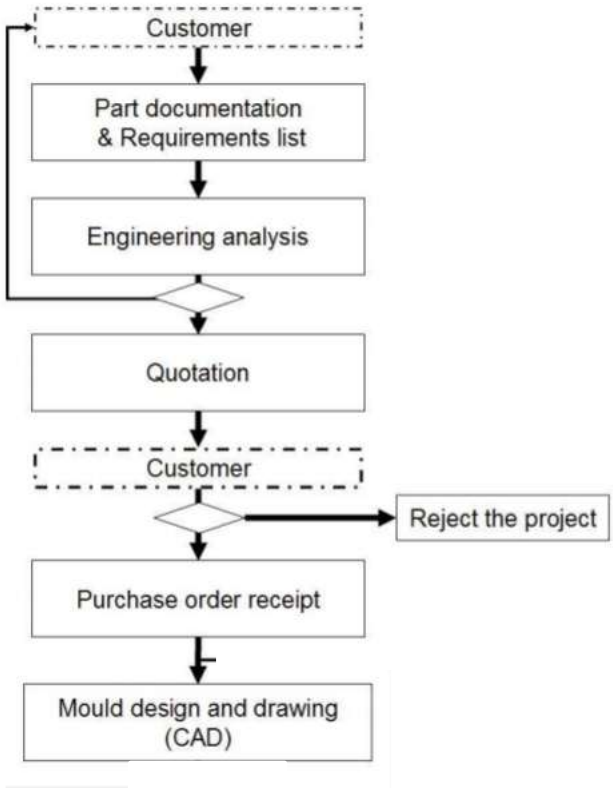

Figure 2.2: The workflow of design process (Mikó, 2013)

Firstly is the customer, who wants to make a plastic part by using an injection mould process. The customers will give their requirement and description of parts to the designer. The form of data can be in engineering drawing, digital computer model and physical model. Next workflow will be the requirement that contains information about mould design. The most important data that contain such as:

i. Moulding material, containing grade and colour.
ii. Quantity of impressions
iii. Type of mould such as two plates, three plates, split, hut runner and cool runner.
iv. Type or runner system.
v. Type of gate
vi. Method of gating

2.2.1 Cavity design

There are two simple steps to generate the core and cavity block. The first one is a Boolean base approach (BBA) where using the Boolean regularized difference operation (BRDO) to generate core and cavity blocks. In BBA to create a swept solid and construct a cavity solid, the procedures that need to take are sweeping the moulded part within the parting direction (Hui and Tan, 1992). The BRDO generated a cavity preform between the cavities solid and swept solid in the cavity side. The BRDO also generate core solid and a core preform is created between cores solid and swept solid in the core side. Then, the core and cavity block that created by the BRDO between the core and cavity preforms in the position that required.

While, in the EBA, the important process to create the related core and cavity block surface based on 3D model of a moulded part in boundary representation (B-Rep) is using the Euler operation (Kwon and Lee, 1991). By applying the Euler operations, the moulded part surfaces are divided into two groups based on parting line location, and then the parting surfaces are close to each surface group. Lastly, to form the core and cavity blocks some of external surfaces are added to each surface group.

According to the Fu, *et al*., (2001) they need to put the initial design results in the earlier stage, that embrace the parting directions, parting lines and surface, and choose the single or multiple cavity type in a core and cavity design development. The single cavity procedure is usually designed the gate, runner, develop core and cavity, core insert and lastly, local tools such as side core, side cavity and split core. While for multiple cavity, the procedure usually design the cavity layout where it can perform in line, mirror symmetry and rotated symmetry. Then create a gate and runner, and design need of making core and cavity block is the cooling mechanism, ejection mechanism and standard parts. The developers of core and cavity blocks in injection mould design are shown in Figure 2.3.

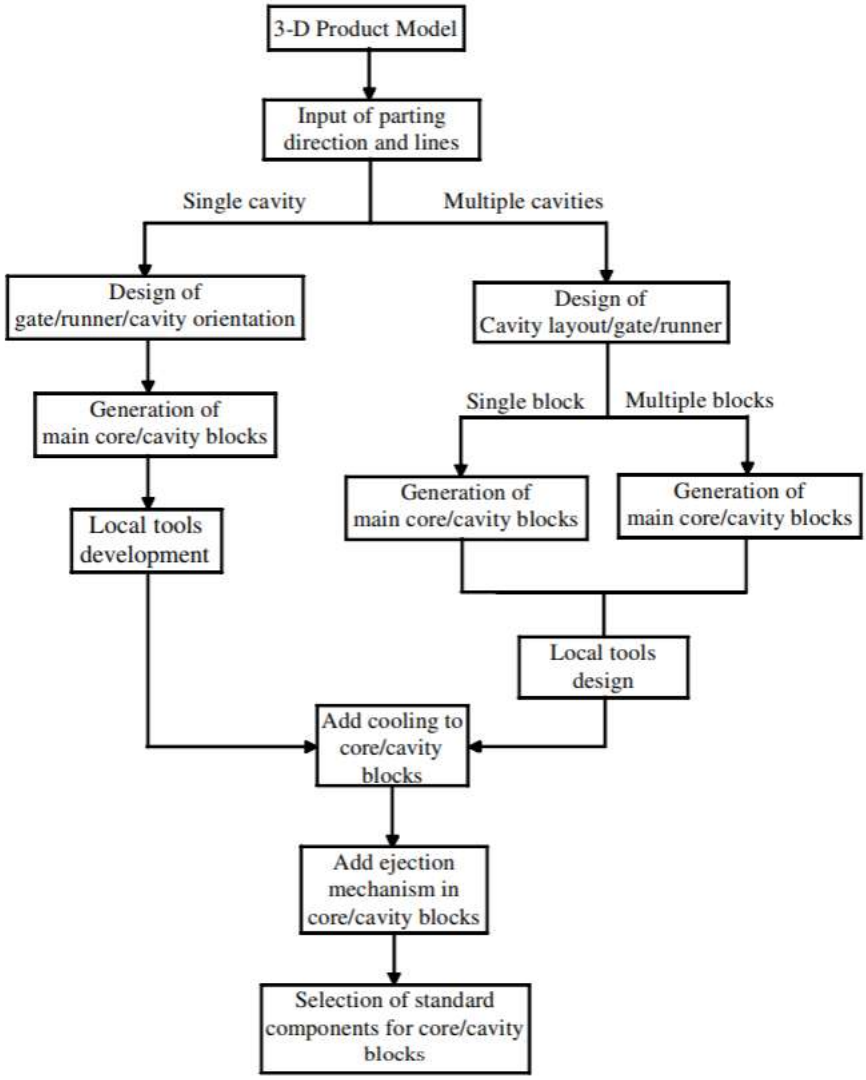

Figure 2.3: Core and cavity blocks in injection mould design development.