UNIVERSITI TEKNIKAL MALAYSIA MELAKA

DESIGN A DUSTPAN AND ANALYZE THE OPTIMUM INJECTION MOLDING PARAMETERS FOR A GATE LOCATION

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

by

NADIAH BINTI AHMAD TERMIZI
B051210224
910811085304

FACULTY OF MANUFACTURING ENGINEERING
2015

UNIVERSITI TEKNIKAL MALAYSIA MELAKA

BORANG PENGESAHAN STATUS LAPORAN PROJEK SARJANA MUDA

TAJUK: **Design a Dustpan and Analyze the Optimum Injection Molding Parameters for a Gate Location**

SESI PENGAJIAN: **2014/15 Semester 2**

Saya **NADIAH BINTI AHMAD TERMIZI**

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 yang berdarjah keselamatan atau kepentingan Malaysia sebagaimana 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.45,Jalan Kasturi 3A,Seksyen BB8

Bukit Beruntung,48300 Rawang

Selangor Darul Ehsan

Cop Rasmi:

Tarikh: _____ Tarikh: _____

** 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 “Design a Dustpan and Analyze The Optimum Injection Molding Parameters for A Gate Location” is the result of my own research except as cited in references.

Signature : ……

Author’s Name : NADIAH BINTI AHMAD TERMIZI

Date : ……

APPROVAL

This report is submitted to the Faculty of Manufacturing Engineering of UTeM as a partial fulfillment of the requirement for the degree of Bachelor of Manufacturing Engineering (Manufacturing Design) (Hons.). The member of the supervisory committee is as follow:

…….

(Project Supervisor)

(Dr. Rosidah binti Jaafar)

ABSTRAK

Projek ini adalah mereka bentuk dustpan dan menentukan parameter acuan suntikan optimum untuk lokasi pintu. Dalam mengurangkan kecacatan berlaku di bahagian-bahagian plastik acuan suntikan, keperluan untuk menganalisis dan mengesahkan parameter acuan suntikan sebelum pengeluaran kumpulan adalah penting untuk menjadi amalan. Kajian ini dijalankan kerana kekurangan prosedur teori yang sedia ada untuk diikuti, proses reka bentuk biasanya dilakukan pada kaedah percubaan dan kesilapan. Analisis proses melibatkan simulasi aliran bahan lebur ke dalam rongga sistem Pemakanan suntikan acuan termasuk seriawan, pelari dan pintu telah direka untuk menganalisis aliran plastik dalam rongga semasa proses pengacuan suntikan. Proses analisis aliran plastik acuan suntikan menggunakan perisian simulasi digunakan untuk mengoptimumkan parameter acuan suntikan yang tekanan suntikan, mencairkan suhu, mengisi masa dan masa penyejukan. Terdapat empat reka bentuk konsep yang berbeza dustpan di mana setiap konsep mempunyai manufacturability yang berbeza untuk dihasilkan menggunakan Allrounder Arburg 420C 800-250 Injection Molding Machine. Reka bentuk dustpan dijana dengan menggunakan SolidWorks 2013 dan geometri sistem pemberian makanan telah direka dengan menggunakan SolidWorks Plastik. Bahan digunakan dalam simulasi komputer adalah polipropilena. Dari hasil analisis, optimum suntikan acuan parameter iaitu tekanan suntikan, suhu leburan purata, mengisi masa dan masa penyejukan telah diperolehi bagi lokasi pintu telah dipilih.

ABSTRACT

This project is about designing a dustpan and determined the optimum injection molding parameters for a gate location. In minimizing defects occurred in injection molding plastic parts, the need to analyze and confirmed the injection molding parameters before batch production is crucial to be practice. This study was conducted due to the lack of existing theoretical procedures to follow, the design process normally carried out on a trial and error method. The process analysis involves the simulation of molten material flow into the cavity of the injection mold Feeding system included sprue, runner and gate were designed to analyze the plastic flow in cavity during injection molding process. The process of analysis plastic flow in injection molding using simulation software is used to optimizing injection molding parameters which are injection pressure, melt temperature, fill time and cooling time. There were four different concept designs of dustpan where each concept has a different manufacturability to produce by using Allrounder Arburg 420C 800-250 Injection Molding Machine. The dustpan designs generated by using Solidworks 2013 and the geometry of feeding system was designed by using SolidWorks Plastics. Material applied during the computer simulation is Polypropylene. From the analysis result, optimum injection molding parameter which is injection pressure, average melt temperature, fill time and cooling time were obtained for a gate location been selected.

DEDICATION

My special dedication to my beloved mother, Jamiah binti Ramli, and my beloved father, Ahmad Termizi bin Bidin for their loves and supports which never end and with the loves and supports given to me, I managed to go through 4 years of my study which full with challenges and hunches. To beloved friends of 4BMFR who have been with me through my journey in education. Also thank you for all the motivation and their beliefs towards me.

ACKNOWLEDGEMENT

I would like to express my appreciation to all those who provided me the possibility to complete this report. A special gratitude give to my final year project supervisor, Dr Rosidah binti Jaafar, whose contribution in suggestion and encouragement, helped to coordinate my project especially in writing this report.

TABLE OF CONTENT

LIST OF TABLES

LIST OF FIGURES

LIST OF ABBREVIATIONS, SYMBOLS AND NOMENCLATURE

PIM	Plastic Injection Molding
PP	Polypropylene
PC	Polycarbonate
ABS	Acrylonitrile Butadiene Styrene
ANOVA	Analysis of Variance
CAE	Computer Aided Engineering
CES	Cambridge Engineering Selector
3D	3 Dimensional

CHAPTER 1
INTRODUCTION

This chapter will explained about the project background all about the design and analysis of the plastic flow in injection molding. This also includes the problem statement and objective of the research. The scope on what being study is also important point need to elaborate.

1.1 Project Background

This project is about designing and analysis the mold flow in plastic injection molding. Plastic is a material that can be produced by injection molding machine to form any shape for consumer product. Plastic for injection molding may be classified into two types: thermoplastics and thermoset. Many different commercial methods are accustomed to produce thermoplastic products. Each of the method used to produce thermoplastics products has its specific design requirements, along with limitations.

Plastic is suitable for injection molding because it is relatively easy to mold into complex shapes, versatility, coloring throughout, transparency, lightweight, high strength, relatively low energy requirement for processing and able to perform mechanical properties. In injection molding, plastic is a material been chosen as a material due to these reason, but plastic have their limitation to implement this type of injection molding process. The limitations of plastic part by injection molding can bring to the occurrence of defect on the part itself.

This project will explain about the most frequent processing method for thermoplastics, injection molding which can be forcing a molten plastic into molds at high pressure. The plastic then forms to the structure of the mold as it cools and solidifies. Usually a quick-cycle process, injection molding can produce large quantities of parts, accommodate a wide selection of part sizes, offer excellent part-to-part repeatability, and make parts with relatively tight tolerances (Yeager, 2000).

Design phase is the most crucial in product development to achieve high quality of product produce by injection molding. The designing process plays important role in injection molding parts, there is several limitation need to be consider when producing with injection molding such as the design must have uniform wall thickness, ribs thickness should be low that wall thickness, rounded corner design, draft angle, bosses thickness and undercuts.

This study also is to analyze the plastic flow in injection molding mold as already known, mold simulation is a method to observe the material flow in mold during injection process. The purpose of this method is to analyze the optimum parameter need to be confirmed to produce high quality of plastic injection part without any defect. Material flow simulation may determine optimal gate location and optimized runner system, forecast weld line, determine the location of air trap, shear stress, filling time, filling possibility, melt temperature, gate freezing time, pressure holding time and pressure. By applying this mold flow simulation before manufacture in real injection molding, manufactures can predict and avoid manufacturing defect on part in earlier stages of part and mold design.

1.2 Problem Statement

Injection molding plastic parts continuously having a defects problem, although the improvements have been made to the part design, mold and material but the occurrence of defect is present not consider the design geometry and type of material. Until now, method used by engineers to determine the parameter including melt temperature, injection pressure, packing pressure , cooling time, injection time

and injection velocity is trial-and-error method, where this process depends on engineers experienced and intuition in determining the initial process parameter setting. It can be prove that this method is not the best and can be practice for future product productivity because it leads to costly process and time consuming (Lahoti, et al., 2013).

There are several common defects occurred at plastic part such as sink marks, flashing, flow marks, short shot, warping, weld line, burn marks, color streaking, spotted whitening, and jetting. Continuously producing a part with low quality (defect) will cause a waste in term of cost and material. By using simulation, it can reduce the occurrence of defect on plastic part due to the analysis of mold flow in injection molding. From the mold flow analysis, it can determine the optimum parameter need to be apply at the plastic part for injection molding and indirectly defined if the injection molding for the part might cause defect or not.

Improve the plastic flow in injection mold. By applying simulation and analysis on mold flow in injection molding before the real production, we can improve the plastic part quality in terms of its design, strength and defect. Plastic is a material that will not degrade for a quite long terms, thus it is important to design and produce a plastic part with high quality for long terms useable. It will reduce consume of material and time producing the parts.

By applying the material simulation method before the visual injection molding process, improvement of process and material cost can be reduce in large range. The process of mold built up can be reduced by determine the requirement in software first. The parameter, design capabilities, expected defects can be determine from mold flow in software application. This proves that, the analysis can be done first and the problem can be identified early, thus the cause of problem can be avoided when the real injection of product executed. It can save the expenditure of repairing, cost of material, cost of designing and cost of production while proceed to the high productivity and good quality of plastic part using injection molding process.

1.3 Objective

The main objectives of this research are:

a) To design a new dustpan.
b) To design feeding system for plastic flow analysis.
c) To perform plastics flow analysis of a dustpan.
d) To obtain the optimum the injection molding parameters for a gate location.

1.4 Scope

The research is focusing in design a plastic part. In designing plastic part, CAD software, Solidwork 2013 software is used to design a dustpan to produce by injection molding. The software use in designing the feeding system for sprue, runner, and gate for one cavity in injection mold is Solidwork Plastic embedded in Solidwork software. The analysis of plastics flow implement by using Solidwork Plastic application. The material selected is Polypropylene. Plastics flow begins when the material or resin is injected into mold through the sprue, runner and gate to fill cavity with material. Plastic flow is analyze to optimize the injection pressure, clamping force, filling time and cooling time towards to shorter cycle time of injection molding process.

CHAPTER 2
LITERATURE REVIEW

This chapter explains about the previous research of injection molding related topics. The main topics discussed in this chapter are injection molding process, common defect in injection molding, injection molding parameter, material in injection molding, type of mold, feeding system design, part design guidelines, and analysis of plastic flow in injection molding process.

2.1 Injection Molding

Injection molding is a process of manufacture plastics part according to required specifications by melting the plastic material and forcing it under certain pressure into sprue, runner, gate and reach to mold cavity. Injection molding technologies reach up 32% to be used in plastic part production due to its ability to produce complex geometry shape with accurate dimension (Shakkarwal & Yadav, 2013).

Plastic injection molding (PIM) is highly precision tools widely used to provide a large volume of plastic part in industries such as consumer product, electronic, automotive, and medical sector. Plastic injection molding results in good dimensional intricate shape of plastics and can be produced in the finished state indirectly provide many advantages such as short product cycle, good mechanical properties, light weight, high quality part surfaces and low cost so it is becoming increasingly in today's plastic production industries (Jain, et al., 2013).

2.1.1 Injection Molding Machine

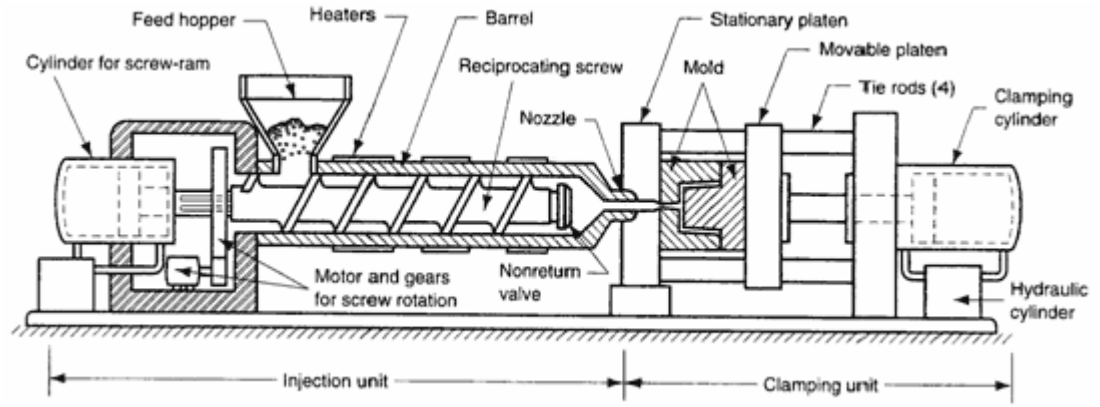

Figure 2.1: Typical injection molding machine **(Narote, et al., 2014)**.

Typical injection machine shows in Figure 2.1 have injection unit and clamping unit where it is differs in injection molding machine. It is depends on the type of injection molding used. Common injection molding machine consist of two units:

a) Injection unit

The task of injection unit shows in Figure 2.2 is to melt the pallet material into liquid form or resin by accumulate the material in the screw chamber. The rotating screw breaks the solid material into small pieces and leads to liquid form to be injected into the cavity.

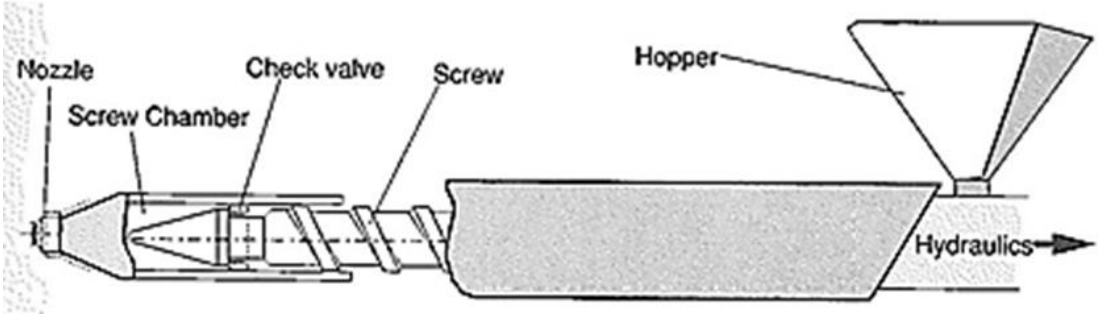

Figure 2.2 : Injection Unit **(Narote, et al., 2014)**

b) Clamping unit

The primary tasks of the clamping unit as illustrated in Figure 2.3 are expand and retract the cylinder whereby will open and close the core and cavity of the mold tightly during injection. There are three clamping types: mechanical, hydraulic and their combination.

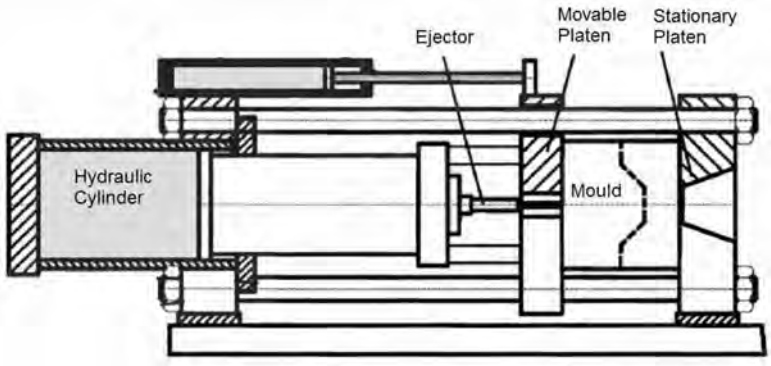

Figure 2.3: Clamping unit (Narote, et al., 2014)

2.1.2 Process and Cycle

A typical sequence of injection molding cycle as shown in Figure 2.4 is as follows:

a) Granules of plastics powder from feed hopper fed into an empty cylinder and falls into the rotating of screw which conveys the material to the front of the cylinder. The material plasticized to a fluid state during its passage along the cylinder with the help of external heaters on the barrel. The back pressure which have sufficient to push the screw back in the cylinder is to prevent some material may be escape through the nozzle. It can be used to equip a reservoir of fluid plastic in the leading of cylinder for injection.

b) Molds is closed and the cylinder moves forward on its haulage until the nozzle in contact with orifice of the mold.

c) Next, the injection of material into mold takes places due to the screw moved forward by the hydraulic cylinder. The material flow through the sprue, runner and gate to the cavity.

d) After a short intervalrance of the mold which called holding time, the screw rotates backward and generate pressure in the cylinder, thus the screw force backward against low pressure in the barrel until energize the limit switch to stops the rotation of screw. The plasticized molten material is ready for the next cycle. The mold open and the article is ejected, a mold close again for next cycle (Narote, et al., 2014)

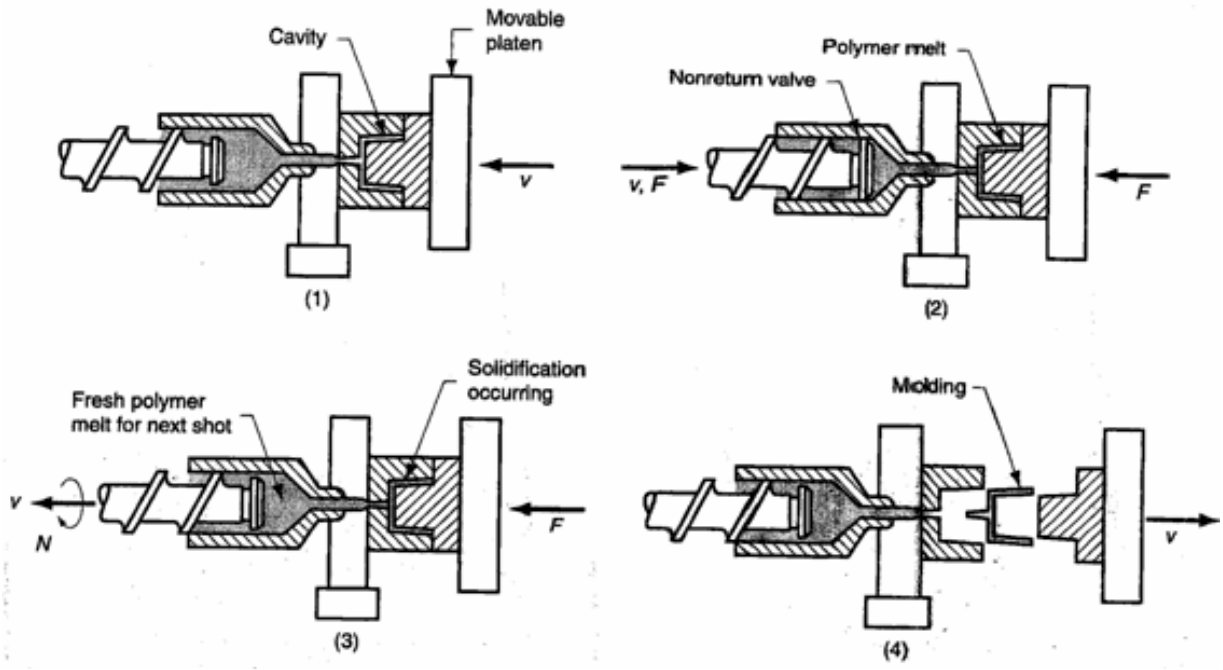

Figure 2.4: Injection Molding Cycle (Narote, et al., 2014)

2.1.3 Common Defect in Injection Molding

Defects on the plastic part are one of the disadvantages of injection molding process. Much reason can be the cause of this problem such as the material used, the mold problem, the setting parameter of machine itself and also the design required to

produce plastic part. There are several common defects occurred on plastic part for instance sink marks, flash, short shot, warping, weld line and flow marks.

2.1.3.1 Sink Marks

Refer to Figure 2.5, sink is the depression or a deep recess or notch on the edge or surface of a part that do not mimic the mold steel surface. Sink and cavity are the signal of the distribution of stress on uniform across the part and are alert signs that the part may not perform as required. The possible cause of sink mark is thick ribs walls of part itself and should be reduced by design the thickness of the ribs lower than thickness of the part. (Bozelli, 2007)

Figure 2.5: Sink Mark (Beaumont, 2008)

2.1.3.2 Short Shot

Part produced is short or some section of the part, like a rib or corner is not completely filled out is shows in Figure 2.6 This situation happened due to incorrect shot size during injection process, trapped of gas or air in the mold thus liquid material cannot filled the trapped area of gas or air. Melt temperature and mold temperature also might be the reason of this problem. (Bozelli, 2007)