Faculty of Manufacturing Engineering

EFFECT OF INJECTION MOULDING PARAMETERS ON THE SHRINKAGE USING TAGUCHI METHOD

Wan Noor Azrina Binti Wan Azhari

Master of Manufacturing Engineering (Manufacturing System Engineering)

2015

EFFECT OF INJECTION MOULDING PARAMETERS ON THE SHRINKAGE USING TAGUCHI METHOD

WAN NOOR AZRINA BINTI WAN AZHARI

A thesis is submitted
in fulfillment of the requirement for the degree of Master of Manufacturing Engineering (Manufacturing System Engineering)

Faculty of Manufacturing Engineering

UNIVERSITI TEKNIKAL MALAYSIA MELAKA

2015

DECLARATION

I declare that this thesis entitled “Effect Of Injection Moulding Parameters On The Shrinkage Using Taguchi Method” is the result of my own research except as cited in the references. The thesis has not been accepted for any degree and is not concurrently submitted in candidature of any other degree.

Signature : .....

Name : .....

Date : .....

APPROVAL

I hereby declare that I have read this dissertation/report and in my opinion this dissertation/report is sufficient in terms of scope and quality as a partial fulfillment of Master of Manufacturing Engineering (Manufacturing System).

Signature :…….....

Supervisor Name :…….....

Date :…….....

DEDICATION

Thanks to Allah s.w.t., my husband my parents, my family, siblings and all my friends.

ABSTRACT

This project focuses on the optimization of injection molding machine using High Density Polyethylene (HDPE) material. The objective of this experiment is to provide statistical evidence for optimizing parameters of the injection molding process. Design of Experiment (DOE) is a powerful technique in order to achieve the high quality of product especially in mass production industries. This experiment was conducted to find out the optimal parameter setting in Injection Molding process in order to minimize the time, waste and cost. In this case study, the Taguchi method is used. It is an alternative way compared with the traditional method which is inefficient and unreliable with One Factor At A Time (OFAT) approach. The Taguchi method had been chosen in these studies because instead of having to test all possibilities, it only performed as per standard orthogonal arrays to investigate the effects of the entire machining parameters through small number of experiments. The parameter and factor that involved in this case study are shot size, injection pressure, injection speed and clamping force. After gaining all the parameter and results from the experiment, it will be interpreted in Minitab 17 software using ANOVA to verify the optimal parameter. The samples yet to be evaluated for weight changes to find the best set of parameter with less value of shrinkage percentage. Finally, the result proved that the optimum parameter reduced the percentage of shrinkage and ANOVA identified shot size parameter as the most significance input towards volumetric shrinkage.

ABSTRAK

Kajian ini adalah bertujuan untuk mencari aturan parameter yang sesuai dalam proses suntikan supaya dapat mengurangkan masa, bahan sisa dan kos menggunakan bahan plastik High Density Polyethylene (HDPE). Rekabentuk experimen merupakan kaedah yang berkesan bagi mencapai kualiti yang tinggi terutama dalam industri pengeluaran secara besar-besaran. Di dalam kes kajian ini, kaedah "Taguchi" akan digunakan. Ia adalah merupakan alternatif dari kaedah tradisional yang kurang cekap dan tidak dapat dipercayai kesahihannya kerana pendekatan ujikaji yang menggunakan perubahan hanya pada satu faktor. Kaedah "Taguchi" dipilih dalam kajian ini kerana ia hanya menguji sebagaimana di dalam standard jujukan "orthogonal" ke atas kesan parameter mesin berbanding terpaksa menguji semua kebarangkalian. Parameter dan faktor yang terlibat dalam kajian ini adalah "shot size", "injection pressure", "injection speed dan "clamping force". Selepas mengumpul kesemua data dan hasil keputusan, data tersebut akan dianalisa menggunakan perisian Minitab 17 melalui ANOVA untuk mengesahkan parameter yang terbaik. Kemudian sampel tersebut akan diujii bedasarkan pengurangan berat untuk mendapatkan set parameter terbaik yang mempunyai kurang peratusan pengecutan. Akhirnya, parameter yang optimum telah mengurangkan peratus pengecutan dan ANOVA membuktikan "shot size" adalah parameter yang paling mempengaruhi pengecutan produk.

ACKNOWLEDGEMENTS

First and foremost, I would like to take this opportunity to express my sincere acknowledgement to my supervisor Dr. Mohd Amran Bin Md Ali from the Faculty of Manufacturing Engineering Universiti Teknikal Malaysia Melaka (UTeM) for her essential supervision, support and encouragement towards the completion of this thesis.

Special thanks to my parents, my husband, siblings, friends and colleagues at Institut Latihan Perindustrian Ledang for their moral support in completing this master. Lastly, thank you to everyone who had been to the crucial parts of realization of this project.

TABLE OF CONTENTS

CHAPTER 4

CHAPTER 5

LIST OF TABLES

LIST OF FIGURES

LIST OF ABREVIATIONS

ANOVA	- Analysis of Variance
ASTM	- American Society for Testing Materials
DOE	- Design of Experiment
HDPE	- High Density Polyethylene
ISO	- International Standard Organization
RSM	- Response Surface Methodology
SS	- Short size
IP	- Injection pressure
IS	- Injection speed
CF	- Clamping force

CHAPTER 1

INTRODUCTION

1.1 Introduction

Basically, in order to get a high-speed moulding of plastic such as thermoplastics, injection moulding technique was developed. This machine works by injecting molten thermoplastic into a mould cavity. This process is subjected to high pressure so that the melted thermoplastic is injected by plunger action with the help of plunger system. (Ghosh, 2011)

In addition, injection moulding process is a rapid process when comparing with other type of moulding processes. Furthermore, after a cycle of the process is completed, feed system such as sprues and runners can be recycled. (Harper, 2002). So this process can be considered as a process that generates low waste. It is also called as a mass production process and the production of products must worth enough to compensate the construction cost of the mould. In addition, complex shapes and intricate shapes can be manufacture by using this process.

Injection moulding process is indeed can be optimized to reduce the unnecessary wastes in production by conducting Design of experiment (DOE). DOE is an operation carried out under control conditions in order to discover an unknown effect or establish a hypothesis. It is an experiment which conducted a series of tests or trials that produce quantifiable outcomes and generally performed to explore, estimate or confirm. The exploration are refers from the data gathering to learn about the process or product characteristic (Anthony et. al., 2000), besides that it's involved with the manipulating of

controllable factors in order to not only determine their effect, but to find the optimum combination of factors that will yield the desired outcome. Therefore the overall of DOE is all about determining the optimum setting or levels of the resources to produce the product that meet quality characteristic (Fryman, 2002).

DOE benefit many manufacturing processes without exception of injection moulding process because can reduce the cost of materials instead of trial and error, therefore it's effected the cycle time of experiment and working percentage, it is indirectly can help to shorten the time and effort that required to discover the optimal conditions to produce Six Sigma quality in delivery of product and services (Arthur, 2007).

Instead of practicing trial and error problem solving, this project implements the DOE process of Taguchi approach by using Minitab software to obtain the L9 orthogonal array and then performing the Analysis of variance (ANOVA). The inputs parameters involved are shot size, injection pressure, injection speed and clamping towards the volumetric shrinkage response. The material employed for the project is High Density Polyethylene (HDPE). The purpose is to identify the optimum parametric combination to reduce the volumetric shrinkage of the two plate family mould.

1.2 Problem Statement

According to Hasan et. al (2007), one of the plastic processing techniques is using injection moulding machine. The injection moulding process actually is the most practical and cost effective to produce plastic products. Furthermore, the versatility of the process allows the production of high quality products regardless the product complexity. The injection moulding process has four phases, which are plasticization, injection, packing and cooling (Seaman, 1994). The quality of the product is depending on material selection, design of the mould and process parameter setting. The process starts with the plastic granules are melted and then forced into the cavity of a close mould and the product is produced.

However, despites of the advantages, many defects occur during the process due to various factors. Unfortunately, production of defective parts contributes to massive cost and waste. Most common defects are flashing and short shot. Consideration of mould design, there are numerous parameters which affect the quality and productivity of products. The parameters contribute to defects are gate size and location, runner and cooling channel design, injection pressure, injection speed, cooling time, injection time, clamping force, shot size and etc. The effects from combination of the various parameters are quite difficult to analyse since they require numerous studies as well as much time and cost.

Beforehand, engineers practiced trial-and-error processes, which depend on the engineers' experience and intuition, to determine initial process parameter settings for the injection moulding process (Chiang and Chang 2006). However, trial-and-error processes are costly and time consuming, not optimal for complex manufacturing processes (Lam, 2004).

In order to gain an optimized process parameter, the design for injection moulding processes is sought by using Design of experiment (DOE) and numerical

analysis. DOE is designed to consider two-way interactions, and have been applied progressively for both mould and process design.

1.3 Objective

The main objective of this project is to find out the effects of injection molding process parameters on volumetric shrinkage for two-plate family injection mould employing High Density Polyethylene (HDPE) as plastic materials.

In order to fulfill the main project objective, the following sub-objectives are proposed:

- To determine the input and parameters such as short size, injection speed, injection pressure and clamping force that affects the quality of injection moulding machine for High Density Polyethylene (HDPE).
- To evaluate the part shrinkage value and weight of four differences plastic parts in two-plate family injection mould.
- To optimize the input parameters on volumetric shrinkage, using Taguchi method and to analyse the percentage contribution using analysis of variance (ANOVA).

1.4 Scope

In this research, the study approach is using one of the designs of experiment (DOE) method which is called as Taguchi method to analyze the optimal parameter in injection molding process. The type of machine used for this experiment is DEMAG Sumitomo 100T, the types of parameter involve are short size, injection pressure, injection speed and clamping force that used to analyze through Minitab software using ANOVA. Pre-screening test is conducted to evaluate the significant changes of the product weight with the selected parameters. There are 4 factor and 3 levels to be analyzed to get 9 simulation run and 3 repetition samples are taken for each run. The material used in this project is High Density Polyethylene (HDPE). The weight of finished product produced is analyzed to ensure the optimization of the parameter in order to gain the appropriate weight. Samples taken are yet to be evaluated for the shrinkage value to get set of parameters with the minimum value of shrinkage.

1.5 Organization of the Report

This report consists of five chapters. Chapter 1 is mainly about the introduction of injection moulding machine, its background, problem statement of study, scope of the study and objectives of the study. Chapter 2 consists of literature reviews that relevant to the study which previously done by other researchers that are related to the study. The next Chapter 3 is about the methodology to accomplish the whole project. It clarifies the approaches and methods used to conduct the study. Essentially, Chapter 4 is about the analysis and discussion of the results. All the outcomes of the study are thoroughly discussed in this chapter. The last chapter is Chapter 5 that concludes overall findings of the study and provides recommendations for future work.

CHAPTER 2

LITERATURE REVIEW

2.1 Injection Moulding Process

Injection molding is the most commonly used manufacturing process for the fabrication of plastic parts. A wide variety of products are manufactured using injection moulding, which vary greatly in their size, complexity, and application. The injection moulding process requires the use of an injection moulding machine, raw plastic material, and a mold. The plastic is melted in the injection moulding machine and then injected into the mold, where it cools and solidifies into the final part. The steps in this process are described in greater detail in the next section. Figure 2.1 shows the typical injection moulding machine.

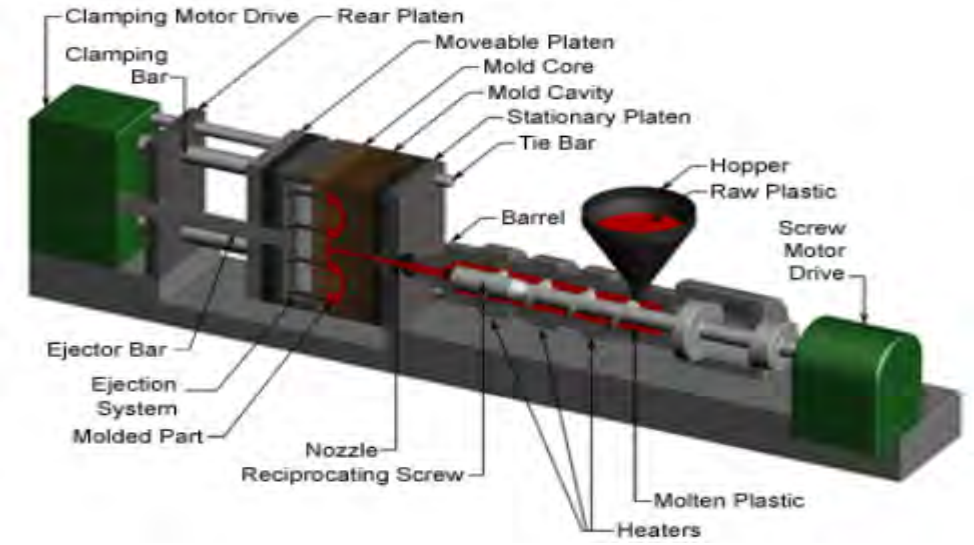

Figure 2.1: Injection Moulding Machine

Injection molding machines are typically characterized by the tonnage of the clamp force they provided. The required clamp force is determined by the projected area of the parts in the mold and the pressure with which the material is injected. Therefore, a larger part required a larger clamping force. Also, certain materials that require high injection pressures may require higher tonnage machines. The size of the part must also comply with other machine specifications, such as shot capacity, clamp stroke, minimum mold thickness, and platen size.

According to Barient (2000), the wide variety in the types of parts that can be produced by injection molding process is the key reason that more injection molding process are selected for plastics processing than any other type of molding equipment. The injection molding process is the common method of converting plastics from the raw materials and most typically used for thermoplastic materials which may be successively melted, reshaped and cooled. The versatility of the process allows the production of high quality simple or complex parts. Some of the machines operate in fully automated as to fulfil the high demand of the production rate. The process is conceptually simple, the plastic granules are melted at certain temperature and then forced into the cavity of a closed mold and then shape of the parts is produced accordingly.

The key for success in injection molding stated by Brent (2000):

i. The proper machine for good melting and injection of the resin
ii. The proper resin for appropriate part performance
iii. A good mold for part definition and removal
iv. Proper operation for efficient molding cycles

The injection molding consists of two essential components and that are injection and clamping unit. The function of injection unit is to melt the resin and inject the melted resin into the mold. Generally the molding process has 3 stages, filling, packing-

holding and cooling. The reciprocating screw which is located in the barrel helps to melt the resin in under certain temperature and pushes the melted resin into the mold cavity.

Normally, the screw stays in the forward position until the resins begin to solidify. The mold pressure is hold to ensure the mold fills completely. Then, the next process, packing-holding, in this stage, the additional resin is added under a certain pressure to compensate shrinkage.

2.2 Injection Moulding Defects

Table 2.1 exhibits the most common defects and its causes that might occur during the operation of the injection moulding process.

Table 2.1: Injection Moulding Defects (Schauer, 2007).

Defects	Causes
Flash	• Injection pressure too high • Clamp force too low
Warping	• Non-uniform cooling rate
Bubbles	• Injection temperature too high • Too much moisture in material • Non-uniform cooling rate
Unfilled section	• Insufficient shot volume • Flow rate of material too low
Sink mark	• Injection pressure too low • Non-uniform cooling rate
Ejector mark	• Cooling time too short • Ejection force too high

Many of the above defects are caused by a non-uniform cooling rate. A variation in the cooling rate can be caused by non-uniform wall thickness or non-uniform mold temperature.