A STUDY ON PROCESS PARAMETER OF REDUCING THINWALL THICKNESS BY USING MOLD FLOW SIMULATION ANALYSIS

MUHAMMAD ZULKIFLI BIN UMOR

PROJECT SUPERVISOR
MR. JASRI BIN MOHAMAD

TOOL & DIE
PRODUCTION TECHNOLOGY DEPARTMENT
KOLEJ UNIVERSITI TEKNOLOGI TUN HUSSEIN ONN
KUITTHO

MARCH 2003

VERIFICATION OF PROJECT SUPERVISOR

"I hereby declare that the Thesis has been read and we have the opinion that the project paper is appropriate in terms of scope coverage and quality for awarding a Bachelor of Technology in Tooling"

Signature : ……………………………………

Name of Supervisor : JAFRI MOHAMMAD

Date : 28/3/2003

A STUDY ON PROCESS PARAMETER OF REDUCING THINWALL THICKNESS BY USING MOLD FLOW SIMULATION ANALYSIS

OATH

"I hereby declare that this project paper is originated from my own initiative and free from plagiarism"

Signature	:	....
Name of Student	:	MUHAMMAD ZULKIFLI BIN UMOR
I.C Number	:	760104-11-5403
Date	:	....

DEDICATION

I would like to dedicate this project to my beloved parents and to all my family who gave support and confidence to me to overcome all the obstacles on my journey of success.

ACKNOWLEDGEMENT

I would like to send my special thanks, greatest appreciation and respect to my project supervisor Mr. Jasri bin Mohamad for his contribution, continuous supervision and guidance given to me throughout the implementation of this project.

Deepest thanks to my ex Degree Program Manager, Mr. George Koshy and to all my lecturers for all of your supports and cooperation.

Last but not least, my extended appreciation to all my family and friends for their understanding and support given to me to overcome all the problems.

ABSTRACT

This paper presents about the study of process parameter towards the plastic part thickness reducing produced by injection moulding method. The model has been designed in such a way in order to minimize the running time analysis. This is because the objective through this project is basically to study the process parameter of the two different thicknesses. This project mainly focused on the effect of different process parameter setting for the thickness of (2mm and 1.5mm) x 400mm x 400mm which are based on the big size of the computer part such as monitor and casing. Beside that, in this study a simulation of flow analysis (Moldflow) has been utilized to investigate filling image, temperature and pressure reactions, weld lines and air traps phenomena in cooperating of moulding process.

TABLE OF CONTENT

LIST OF TABLES

LIST OF FIGURES

LIST OF ABBREVIATIONS

I. Abbreviations of Technical Standards

ASTM	American Society for Testing and Materials
DIN	Deutsches Institute für Normung
ISO	International Organization for Standardization
SI	System International d'Unites *(metric units)*
JIS	Japan Industrial Standards

2. Abbreviations of Polymer Materials

ABS	Acrylonitrile-butadiene-styrene
PC	Polycarbonate, Lexan
PP	Polypropylene
PS	Polystyrene

3. Abbreviations of (Others)

CAD	Computer Aided Design
CAE	Computer Aided Engineering
CAM	Computer Aided Manufacturing
IGES	International Geometry Exchange Standard
DSC	Differential Scanning Calorimetry
MFI	Melt Flow Index
MFR	Melt Flow Rate
Tg	Glass transition temperature
RP	Rapid Prototyping
STL	A Stereolithography file format

LIST OF APPENDIX

CHAPTER I

INTRODUCTION

1.0 BACKGROUND OF THE PROJECT

The study of reducing the thin wall thickness will be focused on the effect of different process parameter setting with the selected size actually targeted for the big part like monitor and casing which are actually will reducing the cost because of the minimum usage of material.

The use of FEA for the prove of mechanical properties also can't be carried out because this software currently is not available in GMI and after brief discussion with person in charge for this software in SIRIM, it's impossible to perform this task. Due to above problems, the project proceed to use Mold Flow to perform the analysis study with the selected model which consists of (2mm & 1.5mm) x 400 x 400mm thickness.

The model design concept added with ribs & holes instead of using full thickness of 2mm & 1mm because these ribs & holes can reduce the usage of material and provide strengths of plastic structure like prevents from warping even can provide the heat release from the electrical components mechanism which produce heat. The effect of weld line can also be studied by adding these holes because this phenomenon normally occurred at the hollow profiles.

MoldFlow simulation software, *Moldflow Plastics Insight (MPI/Fusion) 3.1* is used in this analysis. This is because it will assist in identifying potential problems in the molding process and allow to vary gate location, process parameter conditions, and/or geometry to predict problems and determine solutions.

Process parameters are variables that typically correspond to molding machine settings. Commonly used process parameters are melt & mold temperature, cycle time and injection pressure. Mould temperature has to be choose careful because the excessive mould temperatures will extend the cooling time of the part and extend the cycle time, also the melt temperature should be chosen carefully as excessive melt temperatures can cause polymer degradation. But to make job easier the recommended mould and melt temperatures are available for most materials in the Standard database.

Keyword : Moldflow, Flow analysis

1.1 OBJECTIVE

The objective of this project is:

a.) To study the effect and their limitations of different setting process parameter towards the thickness reducing.

1.2 HYPOTHESIS

a.) What is the optimum process parameter for the selected thickness and size.

b.) What are the factors contributed towards the different setting process parameter for this reducing process.

1.3 PROBLEM STATEMENT

Basically thinner the plastic parts, higher will be the pressure to inject the material into the cavity. Uncompleted filling will be the main problem for the thinner parts with big sizes which short-shot will occurred. Deflection and warping also will the major defects for this process. And of course the cost will be higher due to the manufacturing process (tooling) and the cost of buying the machine with higher tonnage.

1.4 LIMITATION

Limitation through this study :

a.) Unable to prove the mechanical properties by experimental method such as Finite Element Analysis (FEA) due to the time constraint.

1.5 LITERATURE REVIEW

Thin wall plastic injection molding, already well established as a way to reduce the weight and real-estate requirements of small molded parts, has spilled over into large parts as engineers strive to take the cost out of computer, automotive, and appliance components. As an added benefit, thin walls also mold faster, cutting piece-part costs even further. Key influence such as component geometry, wall thickness, the number and position of the gates, material choice, shrinkage allowances and mould design are all interrelated. Part and mould design cannot be based purely on form and function but must also consider the effect of manufacturing [1].

The latest example of thin wall technology's translation from small to large parts comes in the form of computer monitor housings. Even with the thinner walls, the new housings still attain the required flame rating and fulfill all the company's mechanical performance requirements. This uniform thickness is recommended to ensure a good plastic part. Due to the ribbing on the part, there is a possibility for warpage. It is recommended to reduce the thickness of the ribs to approximately half that of the nominal wall. This will prevent some of the warpage and other cosmetic defects that could occur otherwise [1].

Thin-walled plastics are now commonly used for cellular phones and portable computers. Several new plastic materials and processes have been developed which allow the production of high quality thin-walled plastic parts. These new technologies have enabled the production of cosmetic injection moulded parts with wall thicknesses of less than 1 mm. Typical computer housings, such as those made by Sun Microsystems, are produced from two to four parts each in the order of 400 mm long by 400 mm wide. These housing panels must be structurally sound, cosmetically attractive, and resistant to weathering flammability requirements for office equipment. Consequently, most computer housings are made from engineering resins such as flame retardant ABS, PC/ABS blends or polycarbonates, which meet these specifications at reasonable cost. And it is the structural properties, cosmetics and cost that drive design. Reducing the nominal wall thickness affects all three of these areas :

- Stiffness
- Load bearing capacity
- Impact strength

Designers enhance the ruggedness of the boxes by selecting materials with high impact strengths. Decreasing the wall thickness of a housing does not directly change its impact strength. However, impact strength may be lost when stiffer, stronger materials are substituted in these applications. The cosmetics of housing may also be affected by thinning its nominal wall if it increases the occurrence of sink marks or weld lines. In a conventional 3 mm design, the wall thickness of internal ribs and bosses are specified as 60% of the nominal wall in order to minimize visual sink. Using the same approach in the design of thinner walled parts yields much weaker ribs and bosses, which may not support their loads. However, GE Plastics has reported that 1 mm panels made from glass-filled materials can have rib widths 100% of the nominal wall without sink [1].

Weld lines can also be a problem. Normally, designers minimize these by limiting the number and location of gates needed to fill a part. Tool designers typically use one or two direct gates to mould housings with 3 mm walls. However, more gates are needed to fit thinner walled parts as the thinner walls constrict the flow of material and accelerate the cooling of the material.

The final and most important design element which is affected by thinning the nominal wall is the manufacturing cost of the part. This cost is made of three components: material cost; processing cost; and tooling cost. Thinning the wall obviously reduces the material cost. It also decreases process cost by reducing cooling and injection time.

Tooling costs, however, are reported to increase when moulds are designed for thinner walled parts as more steel is required to resist the higher pressures needed to push these materials into the mould. In addition, if glass filled materials are used for these designs harder, more expensive steels must be used to resist the abrasive nature of these materials. This form of moulding has been widely adopted by the cellular phone and portable computer industry because thin-walled parts provide valuable product weight and cost savings. But designers of larger parts, such as desktop or workstation computer housings, have had little interest in thinning the walls of their products and have continued to design their parts with wall thicknesses in the range of 3 mm. However, the current market trend of reducing computer cost and the environmental impact of large quantities of plastic material makes it worth exploring the potential material savings of thin walled products [2].

Mould Flow analysis is recommended be performed on this project. This is because it will assist in identifying potential problems in the molding process and allow to vary gate location, process conditions, and/or geometry to predict problems and determine solutions. A filling analysis will allow the to make these changes before the tool is cut and will reduce potential costs associated with reworking a tool. The field of flow analysis has gained increasing importance in injection moulding. Flow analysis has provided rational solution to many of the hard-to-understand effect that cause problem in he molding process. These effects have including warping, molded-in stress, excessive fill pressure, part flashing and other. The interrelationship between part design and molding process parameters that cause problems of this nature were not well understood in the industry.

There are two basis considerations in the flow of hot plastic into injection moulding which are the flow equation and the heat transfer equation. The method consists of the solution simultaneous equation of heat transfer and fluid flow. As we know plastic does not flow uniformly through thin diaphragm of a plate moulding the compensating phase, but it spread in the branching pattern. Another kind of flow behavior is where flow path are determined by part shape and gate location. Flow front will meet that head on weld together, forming a weld line. The injection period is divided into three stages; filling, compression and compensating flow. The approach is similar in each stage. In filling stage, either the pressure is set and the flow is calculated, or the flow rate calculated. In the compression and compensating stages, the holding pressure is set and the resultant flow is calculated.

Process parameters are variables that typically correspond to molding machine settings. Commonly used process parameters are melt & mold temperature, cycle time and injection pressure.

1.5.1 Advantages of reducing the wall thickness

Advantages of reducing the thickness are; provide valuable product weight and cost savings, application also can be widely used for big parts. The thin-wall technique is a method for producing plastic moldings that enables manufacturing costs to be significantly reduced. When using the thin-wall technique, it is possible to save not only material by reducing the wall thickness of a part, but also to lower the cooling time and thus shorten the cycle time. The thin walls also reduce the overall weight of the overall component.

1.5.2 Advantages of flow analysis

The CAE simulation provides engineers, designers, moulders with a visual and numerical feedback about what actually happens inside the mould cavity during the injection moulding process [3].

1.5.3 Advantages to injection moulder

Flow analysis can provided an objective view of the impact of changes of primary injection moulding process parameter such as melt temperature, mould temperature, injection speed and injection pressure. Optimization of the process parameter allows the moulder to produce parts with minimal levels of residual stress, which can result in post moulding warpage or even mechanical failure of the product. Balanced flow applied to runner and cavity design can help to reduce the amount of material used in the moulding process and eliminate problems such as warping that may be caused by local over packing in the cavity [9].

1.5.4 Disadvantages of reducing wall thickness

Disadvantages that contribute to this thickness are; structural of the stiffness is not strong, impact strength may be lost and manufacturing especially tooling because more steel required to resist the higher pressures needed to push these materials into mould. The moldflow software cannot show exactly where the flashing but it can be detected and identified by analyze the result from clamp tonnage analysis and over packing pressure. Flashing can be the result of a poor match of the cavity and core at the parting line, high pressure in the mould, a poorly supported mould, and/or using a machine with inadequate clamp tonnage capacity.

1.6 DEFINITION OF TERMS

air vent	a small outlet, usually a groove or grounded step, to provide a path for air to flow out of a mould cavity as the material enters
air trap	is an air or gas bubble that has been trapped by converging flow fronts or trapped against the cavity wall.
cavity	depression in mould, which usually forms the outer surface of the molded part
clamping force	force that required to provide motion for closing, clamping and opening the mould
cooling phase	a phase that begin simultaneously with injection as melt starts to cool right from the beginning of injection phase
core	male element in the mould which produces a hole or recess in part
degating	task of separating gate from molded part
ejection phases	a stage where the solidify molded part is ejected out from the mould
filling phase	stage of where the melt plastic fill up the impression, once material is injected into the mould

flow	a qualitative description of the fluidity of plastic material during the molding process
flow line	a mark on a piece made by the meeting of two flow fronts during molding, also called 'weld line'
flow marks	wavy surface appearance on a molded part caused by improper flow of material into the mould
gate	restricted section of the runner at the entrance to the cavity of an injection or transfer mould
holding phase	a process after the injection phase where the axial screw speed slow, as barely sufficient melt is forced into cavity to compress to thermal contraction of material
flashing	occurs when the polymer is not totally contained within the mold cavity
flow leader	is an increase in thickness along a flow path to increase the rate of flow along that path.
flow deflector	is a decrease in thickness along a flow path to decrease the rate of flow along that path.
impression	a part or column in the mould which impart shape to the molding
injection molding	a method of shaping certain materials, as thermoplastic substances, by forcing the heated, syrupy resin into water-chilled molds for cooling and setting
mesh density	number of elements per unit area
nominal part thickness	the thickness of the part at the thickest section.
packing phase	a stage where melt material is packed into the mould
pin point gate	a particular type of gate that not require manual degating entered from the top of part
prototype	a perfect example of a particular type in full-scale, operational model, used for demonstration or testing, that incorporates a new design or features
rapid prototyping	an additive technology that produces models and prototype parts from 3D CAD model data
runner	the channel that connects the sprue with gate to the cavity

shrinkage	a different between the size of the molded part after it reach equilibrium at room temperature and the size of cavity in which it was molded
short-shot	the incomplete filling of a mold cavity which results in the production of an incomplete part.
specific gravity	the density (mass per unit volume)
sprue	feed opening provided in the injection or transfer mould
sub marine gate	a particular type of gate that not require manual degating entered from the side or below of part
warpage	the injection phenomena that occurs on the molded parts, bend or distort
weld line	is a weakness or visible flaw created when two or more flows meet and converge while filling a part.

CHAPTER II

METHODOLOGY

2.0 METHODOLOGY SEQUENCES

Following are the methodology sequences:

1. The first stage is preparing model by using CATIA. This CAD model must be design in solid. It then needs to be transfer to STL (StereoLithograpy) format.

2. Translate the STL file into a Mould Flow file. This is done automatically by the Translator Wizard. For the initial translation, it is better to take the defaults selected by the Translator Wizard. The STL file will be translated into a mould flow file with a matched mesh.

3. Check the mesh using the Mesh Diagnostic icon. The result will report any issues with the mesh, such as overlapping elements, high aspect ratio, connectivity regions, and non-manifold edges.

4. If problem were reported, such as more than one connectivity region, use the Edit Mesh icon to activate mesh editing capabilities.

5. Once the Fix Mesh routine is complete, run the Mesh Diagnostic again to verify that the problems were fixed.

6. Proceed to check and repair the mesh where required.

7. Run the analysis.

8. View the result and interpret the analysis.