CONCEPTUAL DESIGN OF TW0-PLATE INJECTION MOLD FOR UMP'S KEYCHAIN

AZHAR BIN YUSOP

A report submitted in partial fulfillment of the requirement for the award of degree of Mechanical Engineering

Faculty of Mechanical Engineering
Universiti Malaysia Pahang

MAY 2008

ABSTRACT

UMP's keychain is a small product made from plastic same as UMP (University Malaysia Pahang) logo. Demand on plastic product in this country is very tremendous because plastic product has better in quality, design and appearance than any material product. To produce better quality of plastic product, it needs to have some processes and most important is initially in design stages. High quality plastic product can be produced using injection molding process that required good design of mold. In design the plastic mold the most important thing is get the suitable size runner and gate system. This project is using Moldflow Mold Adviser software in order to get achieve optimum size runner and gate system for two-plate injection mold for UMP's keychain

ABSTRAK

Rantai kunci UMP ialah satu produk kecil yang di perbuat dari plastik berbentuk logo UMP (Universiti Malaysia Pahang). Pada dasarnya permintaan produk plastik di negara ini sangat menggalakkan kerana produk plastik adalah setanding dengan produk yang dihasilkan dari bahan yang lain malah produk plastik juga lebih cantik dari segi rupa bentuk serta bermutu. Maka dengan itu untuk menghasilkan produk plastik yang bermutu, produk plastik yang ingin dihasilkan perlu melalui beberapa proses yang sepatutnya terutamanya yang penting sekali ialah pada proses permulaan yang melibatkan proses reka bentuk. Produk plastik yang baik dapat dihasilkan dengan menggunakan proses *injection molding* yang memerlukan reka bentuk acuan yang baik. Untuk mereka bentuk acuan plastic, yang paling penting adalah mendapatkan ukuran *runner* dan *gate* sistem yang sesuai. Kajian ini menggunakan perisian Moldflow Mold Adviser untuk mendapatkan ukuran *runner* dan *gate* sistem yang optima untuk 2-plate suntikan acuan untuk rantai kunci UMP.

TABLE OF CONTENTS

LIST OF TABLES

LIST OF FIGURES

LIST OF SYMBOLS

ABS	-	Acrylonitrile-Butadiene-Styrene
PA	-	Polyamide
PC	-	Polycarbonate
PP	-	Polypropylene
PS	-	Polystyrene
P/L	-	Parting line
HREG	-	Hot runner edge gating
TG	-	Tunnel gating
EG	-	Edge gating
h_r	-	Depth of recess
t	-	Thickness
D	-	Diameter
H	-	Height

LIST OF APPENDICES

CHAPTER 1

INTRODUCTION

1.1 Project Background

Injection molding nowadays have been one of the most important industry in the world. By using this method, the production became faster and more productive. The developing of injection molding becomes a competition from day to day. This process now integrated with computer control make the production better in quality and better quantity.

In designing the mold for injection molding, the accuracy in making mold very important in order to reduce cost and also to make sure that the mold broke easily. Before this, the mold designer used manual analysis to the mold. But now, there is software that can simulate the analysis of the mold that wants to develop.

Clearly, more and more manufacturers are using computational and analytical techniques to reduced design time and cost while significantly improving yield and quality. By using plastics flow simulation products, the determination of manufacturability of part in the early design stages and avoids potential downstream problems which can lead to production delays and cost overruns.

Simulation software allows to do some trouble shooting very easily. Some of the materials that were use are very expensive. Therefore, less time on the production floor working through a problem saves labor and material costs. By using software, designers been able to run simulations and locate and eliminate unsightly nit lines.

Problems that can be avoided by performing flow analysis early in the design stages are air traps, warpage, sink marks and voids, shrinkage, weld lines and meld lines.

1.2 Problem Statement

The problems statement for this project is:

i. Normally, more than three times mold testing required to balancing the material flow into the part especially for multi-cavities and family molds.
ii. Improper gate size, gate location and runner size will caused the long time for mold testing to get the desired final appearance of the product.

1.3 Project Objective

The objective of the project is to analysis material flow for injection molding process in order to get the suitable runner and gate system and design the two-plate injection mold for UMP keychain.

1.4 Project Scope

In order to achieve the objective, the project scopes must include:

i. Literature review on injection molding mold.
ii. Design the UMP keychain using Solid works.
iii. Analysis the material flow using Mould flow software in order to select the suitable runner and gate system.
iv. Design the two-plate injection mold for UMP keychain.
v. Injection mold die is from standard die

CHAPTER 2

LITERATURE REVIEW

2.1 Introduction

The best way is the molds and parts designer must have a good knowledge on the basic of injection molding process. It will help them to design for manufacturer and not just a design which is very nice but can't be manufactured. In this chapter, we will look at the basic theory of plastic injection molding including type of material, product design, mold design and process.

2.2 Injection Molding

Injection Molding is the process of forcing melted plastic in to a mold cavity. Injection molding is used for processing thermoplastics, thermosets, and elastomers. This is a high-rate production process and permits good dimensional control. Injection molding is a versatile process capable of producing complex shapes with good dimensional accuracy and at high production rate. [5]

The advantages of injection molding are high production rates, high tolerances are repeatable, wide range of material can be used, low labor costs, minimal scrap losses, little need to finish parts after molding and design flexibility.

2.2.1 Raw Materials

Most raw materials can be used. The resin is in pellets before processing. The examples of the material are Acrylonitrile-Butadiene-Styrene (ABS), Polyamide or Nylon (PA), Polycarbonate (PC), Polypropylene (PP), and Polystyrene (PS).[3]

2.2.2 Injection Molding Cycle and Process

The basic injection cycle is as follows: Mold close - injection carriage forward - injects plastic - metering - carriage retracts - mold open - eject part.

The molds are closed shut, by hydraulics or electric, and the heated plastic is forced by the pressure of the injection screw to take the shape of the mold. Some machines are run by electric motors instead of hydraulics or a combination of both. The water-cooling channels then assist in cooling the mold and the heated plastic solidifies into the part. Improper cooling can result in distorted molding or one that is burnt. The cycle is completed when the mold opens and the part is ejected with the assistance of ejector pins within the mold.

The resin, or raw material for injection molding, is usually in pellet or granule form, and is melted by heat and shearing forces shortly before being injected into the mold. Resin pellets are poured into the feed hopper, a large open bottomed container, which feeds the granules down to the screw. The screw is rotated by a motor, feeding pellets up the screw's grooves. The depth of the screw flights decreases towards the end of the screw nearest the mold, compressing the heated plastic. As the screw rotates, the pellets are moved forward in the screw and they undergo extreme pressure and friction which generates most of the heat needed to melt the pellets. Heaters on either side of the screw assist in the heating and temperature control during the melting process.

The channels through which the plastic flows toward the chamber will also solidify, forming an attached frame. This frame is composed of the sprue, which is the main channel from the reservoir of molten resin, parallel with the direction of issues related to the wall but also impact to on manufacturability such as injection pressure required fill, cooling time and influence on ejection from the mold. [2]

Maintaining a constant wall thickness should be primary objective of product design. Each region in part has difference thickness will tend to shrink differently. This variation shrinkage will not only complicate achieving the desired size of part but also major contributors to warpage and residual stress. If variation in wall thickness cannot be avoid then try to keep the variation to minimum and provide a gradual transition ratter than sudden change (see figure 2.1). [2]

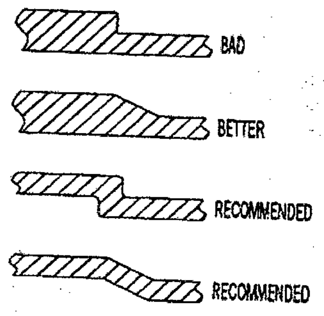

Figure 2.1: Recommended change at the intersection of think/thin region [2]

2.3.2 Corner, Fillet and Radii.

Figure 2.2 illustrated the proper design of corners in a plastic part. This design provides for a constant wall thickness throughout the corner region. The inside radius should be a minimum of 0.5 (50%) the thickness of primary wall. The outside radius should be the inside radius plus the wall thickness. This ensures that the wall thickness is constants throughout the corner. (Note: the larger the inside radius is better) [2] draw, and runners, which are perpendicular to the direction of draw, and are used to convey molten resin to the gate, or point of injection. The sprue and runner system can be cut or twisted off and recycled, sometimes being granulated next to the mold machine. Some molds are designed so that the part is automatically stripped through action of the mold. [10]

2.3 Part Design Guideline for Injection Molded Plastic Part

During part designing stage, the designers must consider whether the process has been choice has a capability to meets the design requirement such as size of part, shape, cosmetic appearance and tolerance.

The basic factors that should be considered during part designing are:

i. Use uniform wall thicknesses throughout the part. This will minimize sinking, warping, residual stresses, and improve mold fill and cycle times.
ii. Use generous radius at all corners. The inside corner radius should be a minimum of one material thickness.
iii. Use the least thickness compliant with the process, material, or product design requirements. Using the least wall thickness for the process ensures rapid cooling, short cycle times, and minimum shot weight. All these result in the least possible parts cost.
iv. Design parts to facilitate easy withdrawal from the mold by providing draft (taper) in the direction of mold opening or closing.

2.3.1 Wall Thickness

Plastic parts are generally designed around the use of relatively thin walls. Unless the injection molded plastic part is produced with gas assist injection molding, its walls are normally less than 5mm.When determining the thickness of the primary wall one must consider not only the structural, functional and aesthetic

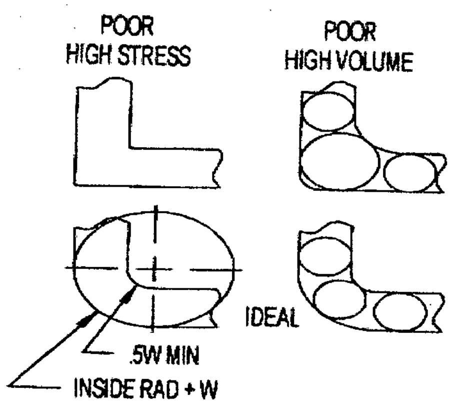

Figure 2.2: Proper designs of corners in a plastic part [2]

2.3.3 Rib

Ribs should follow the proportional thickness guidelines shown in figure 2.3. If the rib is too thick in relation to the part wall, you may experience sinks, voids, warpage, weld lines, and longer cycle times. Position ribs in the line of flow to improve filling and prevent air entrapment [8]

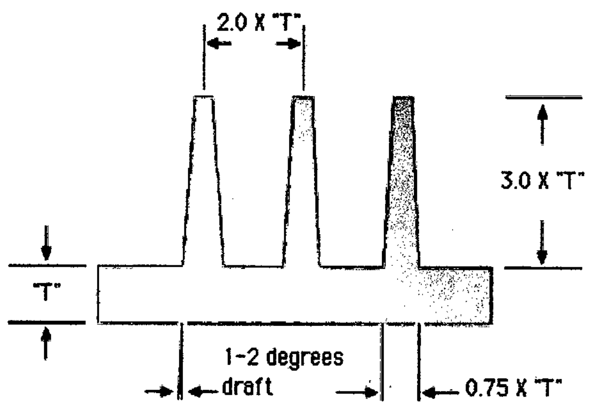

Figure 2.3: Guideline for design of rib respect to wall thickness [8]

2.6.1 Bosses and Gussets.

Bosses are used in parts that will be assembled. Connect the boss to a wall or rib with a connecting rib as shown in Figure 2.4. If the distance of the boss from the wall makes a connecting rib impractical, design the boss with gussets as shown in Figure 2.5.[8]

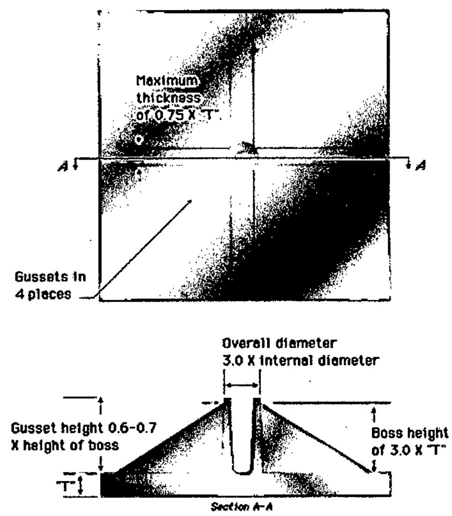

Figure 2.4: Free standing boss with four gussets. [8]

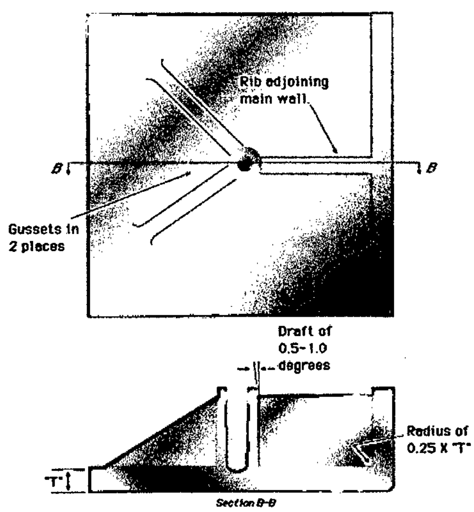

Figure 2.5: Free standing boss with two gussets [8]

2.3.5 Taper and Draft Angles

Draft or tapers are angles put on vertical walls of injection molded part to provide easier ejection from the mold. These angles generally range 1/8° to several degrees depending on material, anticipated ejection problems, and product design requirements. Draft on inside and outside surfaces of side walls should be equal and parallel (Refer figure 2.6).The greater of draft, less potential for ejection problems. [2]

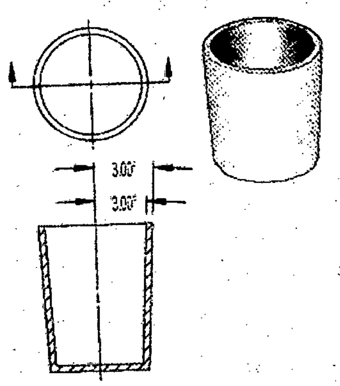

Figure 2.6: Draft on the inside and outside surface of part should be equal and parallel. [2]

2.4 Injection Mold Design

An injection mold must satisfy the following basic requirements:

i. Contain a core and cavity set that defines the features of part will form.
ii. Provide means for molten plastic to be delivered from the injection molding machine to the part forming cavities.
iii. Act as heat exchanger, which will cool the part rapidly and cool the part uniformly
iv. Provide for the molded part to be ejected from mold
v. Have a structure that will resist internal melt pressure.

vi. In multicavity molds, provide uniformly to each cavity through dimension, melt delivery, and cooling

All of the above is done with mold steel tolerances often as little as ±0.00508 mm (0.0002 in). [1]

2.4.1 Type of Injection Mold

There are three basic types of molds:

i. The cold-runner two plate mold (design is simplest)
ii. The cold-runner three-plate mold, in which the runner system is separated from the part when the mold opens.
iii. The hot-runner mold (also called runnerless mold), in which the molten plastic is kept hot in a heated runner plate.

In cold-runner molds, the solidified plastic in the channels that connect the mold cavity to the end of the barrel must be removed, usually by trimming. This scrap can be chopped and recycled. In hot-runner molds, which are more expensive, there are no gates, runners, or sprue attached to the molded part. Cycle times are shorter, because only injection molded part must be cooled and ejected. [6]

2.4.2 Standard Mold Assembly

Basic type mold called "two plate molds". When the mold opens, the mold is divided into two blocks of fixed side (cavity side) and the movable side (core side). The ejector mechanism is provided on the movable side.

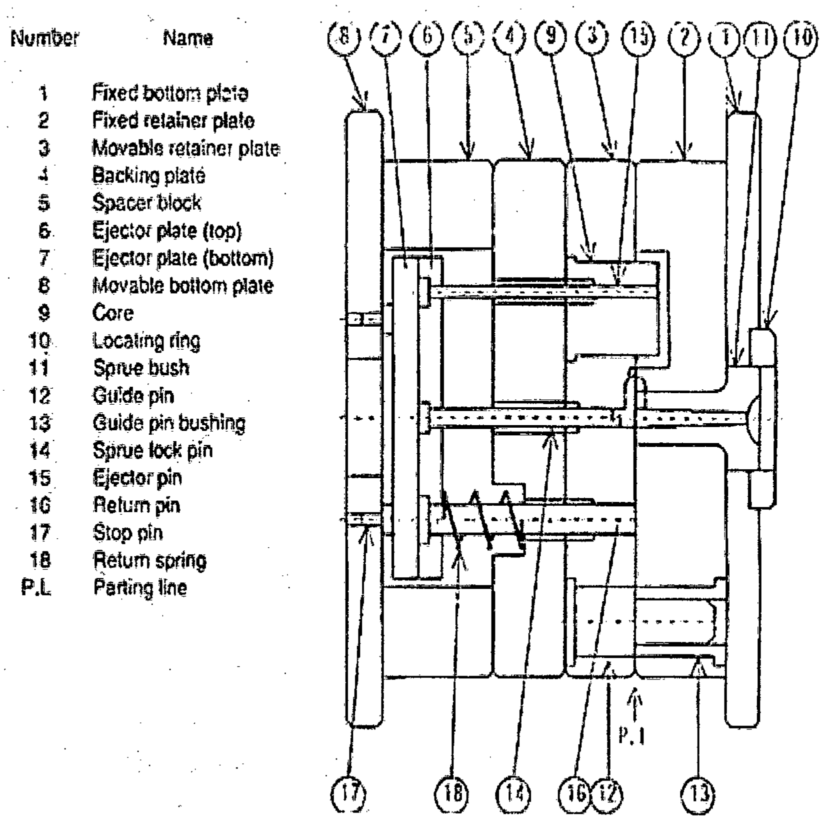

Figure 2.7: Standard mold [7]

2.4.2.1 Mold Part Name and Functions

Explanations are given here on names and function of mold parts based on standard mold show in figure 2.7.

The right side from the boundary surface of mold in opening, i.e. the parting line (P.L), is called the "fixed side", and the left side the "movable side". The fixed side means that the parts are fixed to the nozzle side of the molding machine. The movable side means that parts are mounted on the movable platen sliding for opening/closing of mold and move per injection shot.

The locating ring (10) on the fixed side is fitted on a hold of same diameter in the bottom plate of a molding machine for location of a mold when it is mounted on the molding machine.