INVESTIGATION AND ANALYSIS OF FINDING THE INJECTION MOLDING GATE BY USING MOLDFLOW

MUHAMAD FRIDY NIZAM BIN HARUN

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

Faculty of Mechanical Engineering
Universiti Malaysia Pahang

NOVEMBER 2007

ABSTRACT

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. The design has to be correct and can produce better product, so it needs to be analyzing before fabricate the mold. Material flow analysis should be running to the plastic product to ensure no defect and follow the characteristics from actual specification. This project is using Moldflow Plastic Insight software to analyze and investigate the effects of gate and the parameter on the product. UMP cardholder has been chosen as a subject of experiment. The result that use in this study are the fill time, time to freeze, the injection pressure, the clamping force, and also defects.

ABSTRAK

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. Reka bentuk ini mestilah tepat dan menghasilkan produk yang baik, maka reka bentuk ini hendaklah dianalisis terlebih dahulu sebelum acuan dihasilkan. Analisis aliran bahan hendaklah dilakukan ke atas produk bagi memastikan tiada kerosakan dan mengikut spesifikasi yang ditetapkan. Projek ini menggunakan perisisan 'Moldflow Plastic Insight' bagi menganalisis dan mengkaji kesan get ke atas produk dan nilai yang sesuai bagi menghasilkan produk. Pemegang kad UMP digunakan sebagai bahan analisis. Keputusan yang dibincangkan ialah masa yang diambil bagi mengisi acuan, masa plastik membentuk, tekanan suntikan, daya tahan, dan kerosakan didalam produk.

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 are software that can simulate the analysis of the mold that want 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

In designing mold for injection molding, gate one of the elements that very important. There are problems if the gate of the mold didn't design correctly. The filling of the cavity slow or impossible to fill the cavity full before it freeze. This is happen because the gate that have been use too small. The wall of the cavity breaks because the runner ends too close to the cavity in order to get short land. Other than that, the plastic hard to break where it joins the products. This is happen because the gate is too large.

Other than that, in order to get the parameters for the injection molding process, plastic have been wasted. Through the experiment, operator will use large amount of plastic in order to get the best parameters for the setup of the machine. The best parameters very important so that the cycle time become faster, produce high quality and quantity product, saving energy, and also didn't risk the mold.

However, every problems need to be solve. Some of the mold designer use past experience from earlier designs or from information given by experienced designers or molders. Commonly method, often practical, but may be misleading when applied to mold similar to another mold. Other than that, some designers also use an empirical approximation based on experiments. Adjust gate sizes to achieve uniform filling of all cavities and to compensate for variations in pressure drops in runners. But, nowadays, using computer software analysis, these problems can be overcome easily.

By the software analysis, it can easily determine the way to get the better gates for mold. More over, the analysis also recommends the best and optimum parameters. The costs on develop mold decrease and also reduce risk for the mold to get wear. The production rate increase and produce better product.

1.3 Project Objective

The objective of this project is to investigate the injection molding gating mechanisms and parameters for two plate mold.

1.4 Project Scope

1. Injection mold die is from standard die.
2. Analysis by using Mold Flow software.
3. The analysis should be injection molding material, injection molding parameter, injection molding system, molding cycle time, and defects.

CHAPTER 2

LITERATURE REVIEW

2.1 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.

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.

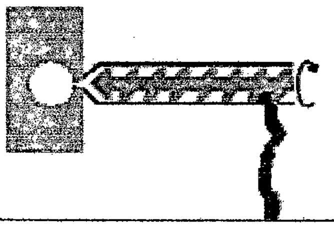
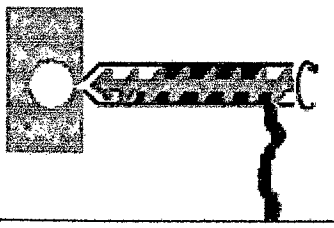
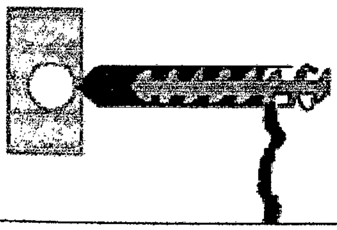
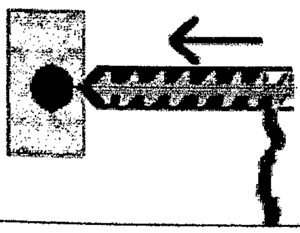
2.1.1 Raw Materials

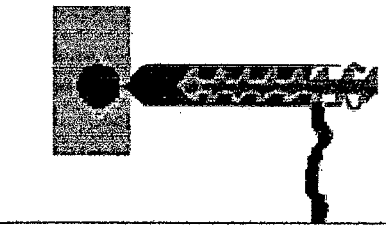
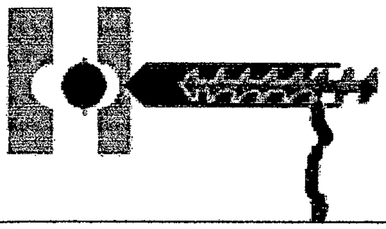
Most raw materials can be used. The resin is in pellets before processing. The examples of the material are Acrylonitrile-Butadiene-Styrene (ABS), Polyamide or Nylon (PA), Polycarbonate (PC), Polypropylene (PP), and Polystyrene (PS).

2.1.2 Injection Molding Cycle & Process

The injection molding process occurs cyclically. Typical cycle times range from 10 to 100 seconds and are controlled by the cooling time of the thermoplastic or the currying time of the thermosetting plastic. The plastic resin in the form of pellets or powder is fed from the hopper and melted. In a reciprocating screw type injection molding machine, the screw rotates forward and fills the mold with melt, holds the melt under high pressure, and adds more melt to compensate for the contraction due to cooling and solidification of the polymer. This is called the hold time. Eventually the gate freezes, isolating the mold from the injection unit, the melt cools and solidifies. Next the screw begins to rotate and more melt is generated for the next shot. In the soak time the screw is stationary and the polymer melts by heat conduction from the barrel to the polymer. The solidified part is then ejected and the mold closes for the next shot.

Table 2.1: Process shot material into mold

	**Step 1** - The uncured rubber is fed into the machine in the form of a continuous strip.
	**Step 2** - The uncured rubber is worked and warmed by an auger screw in a temperature controlled barrel.
	**Step 3** - As the rubber stock accumulates in the front of the screw, the screw is forced backwards. When the screw has moved back a specified amount, the machine is ready to make a shot.
	**Step 4** - With the mold held closed under hydraulic pressure, the screw is pushed forward. This forces the rubber into the mold, similar to the action of a hypodermic syringe.

	**Step 5** - While the rubber cures in the heated mold, the screw turns again to refill.
	**Step 6** - The mold opens and the part can be removed. The machine is ready to make the next shot, as soon as the mold closes.

2.2 Injection Molding Machine

An injection molding machine consists essentially four basic elements; clamping mechanism, plasticizing unit, injection unit, and necessary controls. There are two types of injection molding machines; plunger and reciprocating screw. Modern machines are of the reciprocating or plasticating screw type.

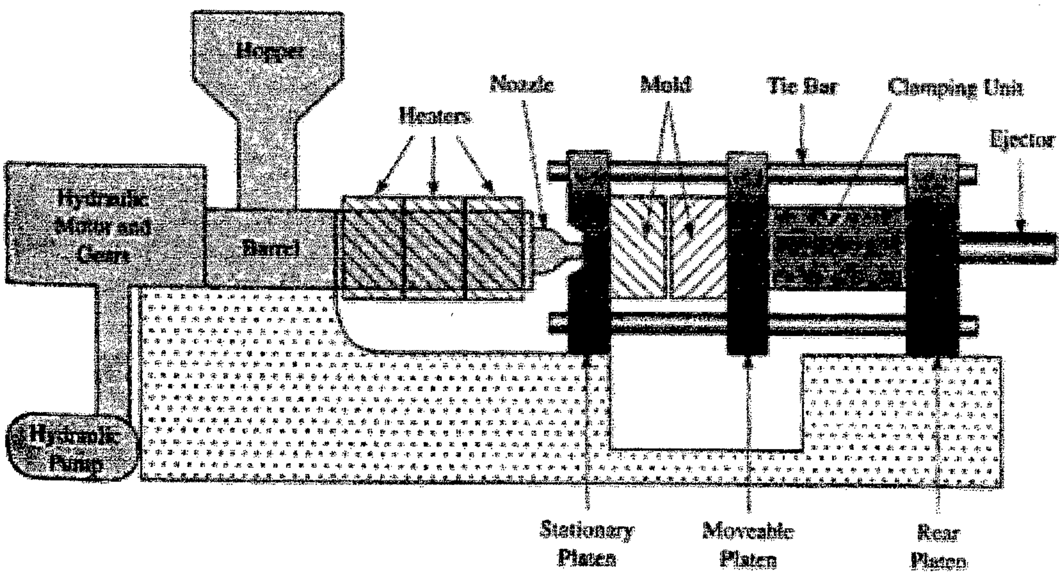


Figure 2.1: Injection molding machine

2.2.1 Reciprocating Screw Type

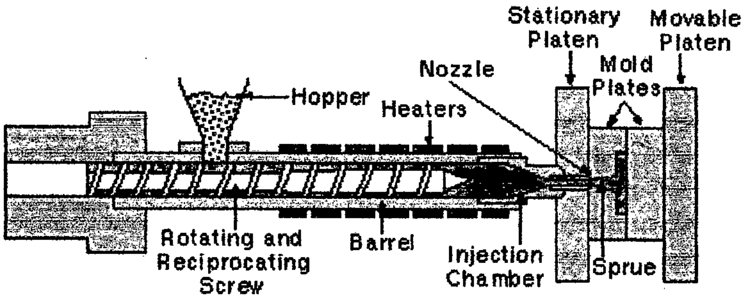


Figure 2.2: Reciprocating screw type

The injection system consists of a hopper, a reciprocating screw and barrel assembly, and an injection nozzle, as shown in Figure 2.2. This system confines and transports the plastic as it progresses through the feeding, compressing, degassing, melting, injection, and packing stages.

Reciprocating screws are capable of turning as they move backward. As the pressure builds up at the mold entrance, the rotating screw begins to move backwards under pressure to a predetermined distance. This movement controls the volume of material to be injected. The screw then stops rotating and is pushed forward hydraulically, forcing the molten plastic into the mold cavity. The pressure developed usually range from 70 to 200Mpa.

The reciprocating screw is used to compress, melt, and convey the material. The reciprocating screw consists of three zones (Figure 2.3): feed zone, transition zone, and metering zone.

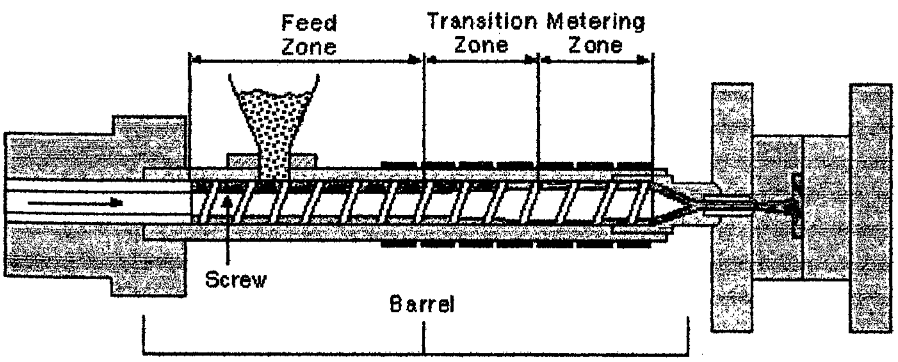


Figure 2.3: Reciprocating screw zone

A screw injection machine improves melt homogeneity, reduces variations in the molded parts, and minimizes degradations and cold spots of the polymer melt.

While the outside diameter of the screw remains constant, the depth of the flights on the reciprocating screw decreases from the feed zone to the beginning of the metering zone. These flights compress the material against the inside diameter of the barrel, which creates viscous (shear) heat. This shear heat is mainly responsible for melting the material. The heater bands outside the barrel help maintain the material in the molten state. Typically, a molding machine can have three or more heater bands or zones with different temperature settings.

2.3 Mold

Injection mold can be defined as arrangement, in one assembly, of one (or a number of) hollow cavity spaces built to the shape of the desired product, with the purpose of producing (usually large number of) plastic parts, or products. The cavity space is generated by a female mold part, called the cavity, and a male mold part, called the core.

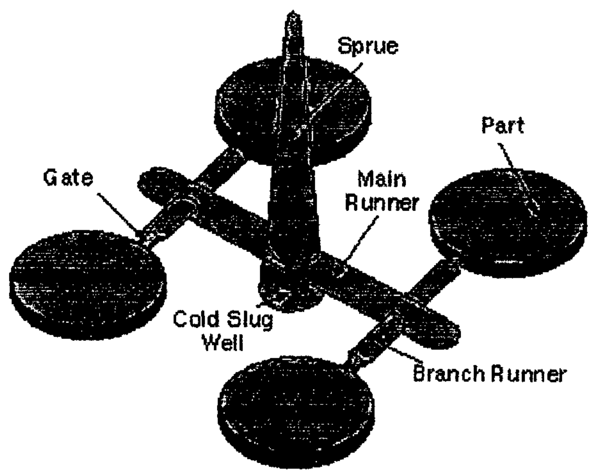


Figure 2.4: The molded system includes a delivery system and molded parts

To fill the cavity spaces, the mold is mounted in an injection molding machine that is timed to close the mold, inject the plastic into the cavity spaces, keep the mold closed until the plastic is cooled and ready for ejection, open the mold, and eject the finished products.

The molding cycle is defined as the time from the moment the mold is closed for one injection, or shot, until it is closed again for the following shot. Usually, the numbers of shots per minute (or shots per hour) are given to indicate productivity of a mold, rather than the molding cycle in seconds.

To accommodate part design, molds may have several components, including runners, cores, cavities, cooling channels, inserts, knockout pins, and ejectors. Injection mold classified into 3 basic types. Hot runner 3 plate mold, cold runner 3 plate mold (Figure 2.5 (b)), the runner system is separated from the part when the mold is opened. The other one, cold runner two plate mold (Figure 2.5 (a)), also called runnerless mold: the molten plastic is kept hot in a heated runner plate.

The effective size of the runner can be calculated from this equation [2.1]:

$$D = \frac{\left(W^{1/2} L^{1/4}\right)}{3.7} \tag{2.1}$$

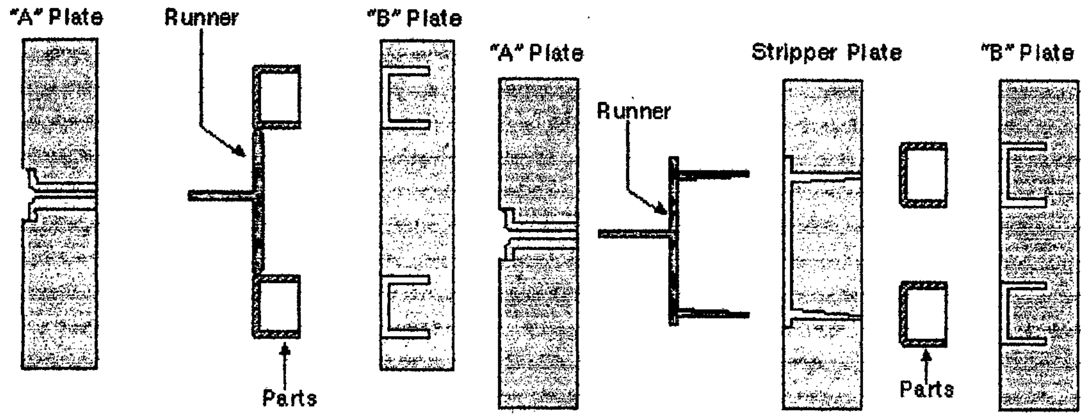
Which is: D = diameter of the runner (mm)
W= weight of the part (g)
L = Length of runner (mm)

'Or' according to the experience of mold maker, they determine the runner size by:

$$D = t x \, (2.0 \text{ to } 3.0) \tag{2.2}$$

Which is: D = diameter of the runner (mm)
t = the wall thickness of the part (mm)

In cold-runner molds, the solidified plastic remaining in the channels connecting the mold cavity to the end of the barrel must be removed, which usually is done by trimming. Later, this crap can be chopped and recycled. In hot-runner molds, there are no gates, runners, or sprues attached to the molded part. Cycle times are shorter because only the molded part must be cooled and ejected.



(a) Two plate mold (b) Three plate mold

Figure 2.5: Mold types (a) Two plate mold (b) Three plate mold

2.3.1 Mold Requirement

In design and fabricate the mold the factor and requirement that need are accuracy and finish, productivity, physical strength (tensile strength, compressive strength, and plate deflection), wear resistance, safety in operation, maintenance and interchangeability, ease of installation and also reasonable cost.

Injection molds must be properly designed to ensure quality plastic components. Mold design impacts productivity and profitability of molding operation.

2.3.2 Two Plate Cold Runner

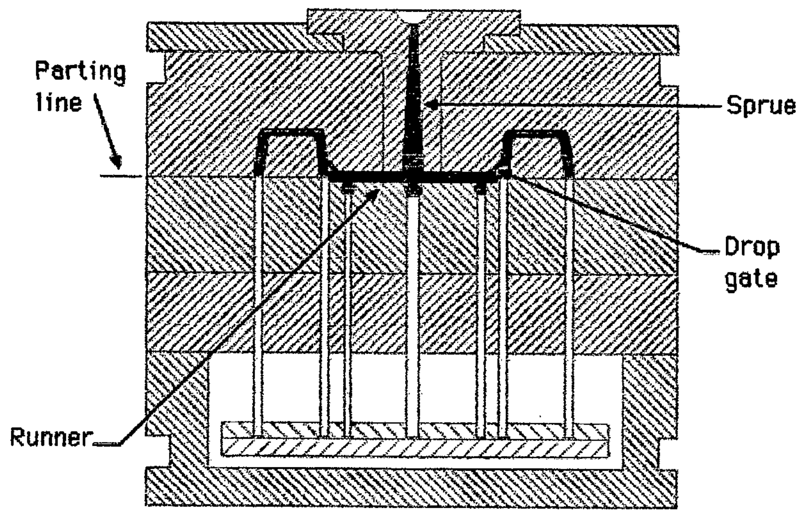


Figure 2.6: Two plate cold runner mold

The conventional two-plate mold consists of two halves fastened to the two platens of the molding machine's clamping unit. When the clamping unit is opened, the two mold halves open, as shown in Figure 2.7 (b). The most obvious feature of the mold is the cavity, which is usually formed by removing metal from the mating surfaces of the two halves. Molds can contain a single cavity or multiple cavities to produce more than one part in a single shot. The figure shows a mold with two cavities. The parting surface (or parting line (P/L) in a cross-sectional view of the mold) is where the mold opens to remove the part(s).

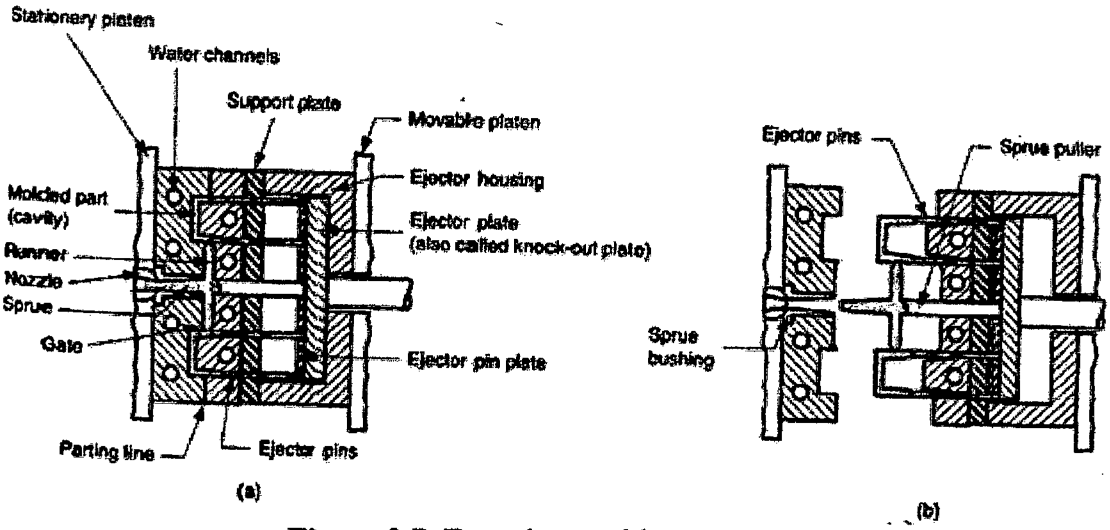


Figure 2.7: Two plate mold component

In addition to the cavity, there are other features of the mold that serve indispensable functions during the molding cycle. A mold must have a distribution channel through which the polymer melts flows from the nozzle of the injection barrel into the mold cavity. The distribution channel consists of a sprue, which leads from the nozzle into the mold; runners, which lead from the sprue to the cavity (or cavities); and gates that constrict the flow of plastic into the cavity. There are one or more gates for each cavity in the mold.

An ejection system is needed to eject the molded part from the cavity at the end of the molding cycle. Ejector pins built into the moving half of the mold usually accomplish this function. The cavity is divided between the two mold halves in such a way that the natural shrinkage of the molding causes the part to stick to the moving half. When the mold opens, the ejector pins push the part out of the mold cavity.

A cooling system is required for the mold. This consists of an external pump connected to passageways in the mold, through which water is circulated to remove heat from the hot plastic. Air must be evacuated from the mold cavity as the polymer rushes in. Much of the air passes through the small ejector pin clearances in the mold. In addition, narrow air vents are often machined into the parting surface; only about 0.03 mm (0.001 in.) deep and 12 to 25 mm (0.5 to 1.0 in.) wide, these channels permit air to escape to the outside but are too small for the viscous polymer melt to flow through.

Moulds are typically constructed from hardened steel, pre-hardened steel, aluminium, and/or beryllium copper.

2.4 Gates

Gate can be defined as a passage through which the plastic materials enter the cavity spaces. The requirements for gate are contradictory. Large gates are desirable to facilitate filling of the cavity space and to reduce stresses in the plastic and in the product. Large gates keep the slowly cooling and shrinking plastic in the cavity space connected for a longer period with the hot plastic supplied from the injection. This permits packing before the gate freezes.

Small gates freeze faster and produce higher molded-in stresses but are desirable to facilitate separation of products from the runner, and to make the gate mark, or vestige, more inconspicuous.

2.4.1 Gate Location and Number per Cavity

As a rule, one gate per cavity should be sufficient for most products and is usually satisfactory to avoid undesirable weld lines in the product; however, there are exceptions to this rule. (Herbert Rees, 2002)

2.4.2 One Gate per Cavity

2.4.2.1 Side Gating Near Top

A side gate near the top of product may be hot runner edge gating (HREG) or tunnel gating (TG). A product design may not permit gate vestige in the top for cosmetic or performance reasons, such as optical requirements.

This side location is better than gating near the rim when filling speed is important, as with thin-walled products. Because the gate is near the closed end (bottom) of the product, the plastic will usually fill the bottom first and then flow toward the rim.

It is highly recommended that the side gate be located so that the plastic stream will not flow freely into the top surface. Instead, it should be directed to hit the core. The stream could also be directed against a core pin in the top surface, near