EFFECTS OF INJECTION MOLDING GATE MECHANISM ON PARAMETERS MACHINING AND DEFECTS OF BOOK TRAY

ABDUL AZWAN BIN ABDOL ZAHAR

BACHELOR OF ENGINEERING
UNIVERSITI MALAYSIA PAHANG

EFFECTS OF INJECTION MOLDING GATE MECHANISM ON PARAMETERS MACHINING AND DEFECTS OF BOOK TRAY

ABDUL AZWAN BIN ABDOL ZAHAR

Thesis submitted in fullfillment of the requirements
for the award of the degree of
Bachelor of Mechanical Engineering with Manufacturing Engineering

Faculty of Mechanical Engineering
UNIVERSITI MALAYSIA PAHANG

NOVEMBER 2009

ACKNOWLEDGEMENTS

First and foremost, I want to thank Allah S.W.T, for giving me the source of power, knowledge and strength to finish the project and dissertation for completing my Bachelor of Mechanical Engineering final year project. By gradually following the plan of effect of injection molding gate mechanism on parameters machining and defects of book tray, I've manage to fulfill my responsibility to this project.

I would like to express my gratitude to my supervisor Mr. Mohamed Reza Zalani Bin Mohamed Suffian for his wisdom, endurance and encouragement during his supervision period. He motivates me about the importance of constant and gives me his outstanding professional conduct. I appreciate his consistent support from the first day I applied to graduate program to these concluding moments. I am truly grateful for his progressive vision about my training in science, his tolerance of my naive mistakes, and his commitment to my future career. I also sincerely thanks for the time spent proofreading and correcting my many mistakes. I acknowledge my sincere indebtedness and gratitude to my parents for their love, dream and sacrifice throughout my life. My sincere thanks go to all my lab mates and members of the staff of the Mechanical Engineering Department, UMP, who helped me in many ways and made my stay at UMP pleasant and unforgettable. And also thanks to Dean Faculty of Mechanical Engineering, Associate Prof. Dr. Rosli Bin Abu Bakar for his advice and suggestion. I also sincerely thanks for the time spent proofreading and correcting my many mistakes.

I would also like to acknowledge with much appreciation the crucial role of the staff in Mechanical Laboratory, who gave a permission to use the computer laboratory to design and run the analysis and also the plastic laboratory to make mould measurement and to use all the necessary tools in the laboratory. Special thanks too goes to Mr. Azizuddin Bin Abd Aziz against as the Final Year Coordinator, who has give some advice and share his knowledge on this final year project.

Many thanks go to the assistant instructor engineer (PJP) especially to Mr. Khairul Azha B A Jalal, and to all the supervisors who have given their full effort in guiding the through on the project progress. I acknowledge my sincere indebtedness and gratitude to my parents for their love, dream and sacrifice throughout my life. Special thanks should be given to my committee members. I would like to acknowledge their comments and suggestions, which was crucial for the successful completion of this project.

ABSTRACT

Demand on plastics product in this country is very tremendous because plastic product has better quality, design and appearance than any material product. In order to produce better quality of plastics 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 defects and follow the characteristics from actual specification. The three-dimensional solid modelling of plastic product (book tray) was developed using the computer-aided drawing software. The dimension of product based on the actual product mould. The three-dimensional solid modelling of plastic product will import to the computer aided engineering software. The computer aided engineering software was then performed is using Moldflow simulation software. The computer aided engineering model of product was analyzed using injection flow analysis. The analysis need to do in three times for each gates mechanism. Finally, the best result of number of gates, gates locations, and sizes of head of gates obtained from the analysis. From filling times, the cycle time of product producing can be calculated for finding the best gates mechanism. The cycle time can relate to the production capacity, product cost and profit. From that comparison, the best gate mechanism for plastic product (book tray) can be selected and also can know the effect of parameters machining and defects of book tray. This project is using Moldflow Plastics Insight 5.0 software to analyze the effect of gate mechanism. Book tray has been chosen as subjects of experiment. Before get the best mechanism of the gate the analysis that should be analyze are number of gates, gate location, and gates sizes. By using this analysis method also can reduce the cost in define the gate mechanism for plastic product using manual method.

ABSTRAK

Pada dasarnya permintaan produk plastic di negara ini sangat menggalakkan kerana produk plastic adalah setanding dengan produk yang dihasilkan dari bahan yang lain malah produk plastic juga lebih cantik dari segi rupa bentuk serta mutu. Maka dengan itu untuk menghasilkan produk plastik yang bermutu, produk plastik yang ingin dihasilkan perlu melalui beberapa proses yang sepatutnya terutama 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. Permodelan struktur pejal tiga-dimensi bagi barangan plastik (rak buku) telah dibangunkan menggunakan perisian lukisan bantuan komputer. Ukuran saiz barangan plaktik itu berdasarkan ukuran yang terdapat pada acuan. Permodelan struktur pejal tiga-dimensi untuk barangan plastik itu dimasukan kedalam perisian kejuruteraan bantuan komputer. Perisian kejuruteraan bantuan komputer yang digunakan ialah perisian simulasi Moldflow (MPI). Permodelan kejuruteraan bantuan komputer menjalankan analisis suntikan aliran. Analisis Moldflow dilakukan sebanyak tiga kali bagi setiap sifat gate itu. Pada akhirnya, keputusan terbaik dariapada bilangan gate, lokasi kedudkan gate dan size kepala gate diperolehi daripada analisis. Daripada masa memenuhi,kitaran masa pembuatan produk dapat dikira untuk mencari sifat gate yang terbaik. Kitaran masa juga dapat di kaitkan dengan kepadatan pembuatan, kos barangan dan keuntungan. Bahan plastik yang terbaik untuk barangan plastik(rak buku) akan dipilih berdasarkan perbandingan antara sifat gate juga dapat mengetahui kesan keatas parameter mesin dan akibat buruk yang terjadi kepada rak buku. Projek ini menggunakan perisian ‘Moldflow Plastic Insight 5.0’ bagi menganalisis dan mengkaji kesan keatas sifat gate. Rak buku digunakan sebagai bahan analisis. Sebelum mendapat nilai dan sifat get yang terbaik, analisis yang perlu dijalankan adalah, bilangan gate, lokasi get, saiz get. Dengan menggunakan analisis ini dapat mengurangkan kos untuk memilih sifat gate untuk barangan plastik menggunakan mesin pembentukan acuan suntikan.

.

TABLE OF CONTENTS

LIST OF TABLES

LIST OF FIGURES

LIST OF SYMBOLS

^{0}C	Degree Celsius
%	Percent
cm^3	centimetres cube
MPa	Mega Pascal

LIST OF ABBREVIATIONS

3D	Three-dimensional
ABS	Acrylonitrile-butadiene-styrene
CAD	Computer aided drawing
CAE	Computer aided engineering
FYP	Final year project
MPI	Moldflow plastics insight
PC	Polycarbonate
PE	Polyethylene
PS	Polystyrene

CHAPTER 1

INTRODUCTION

1.1 INTRODUCTION

Nowadays injection molding is probably the most important method of processing of consumer and industrial goods, and is performed everywhere in the world. 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 and also to make sure that the mold broke easily. Before this, the mold designer used manual analysis to the mould. But now, there is software that can simulate the analysis of the mold that wants to develop.

Clearly, more manufactures 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 production delays and cost overruns. Some of the materials that have been used are very expensive. Therefore, less time on the production floor working through a problem saves labor and material costs. These days, simulation software can accurately predict the fill patterns of any part. This allows for quick simulations of gate placements and helps finding the optimal location. Problem that can be avoided by performing flow analysis early in the design stages are, sink mark, air traps, shrinkages, and blush and flow marks.

1.2 OBJECTIVES

For this project, there are 3 main objectives to achieve the target. The objectives are:

(i) Investigate the gate mechanism effect on injection molding parameters and defects of book tray.
(ii) Design and proposed gate mechanism according to the results analysis.

1.3 PROJECT SCOPES

One of the most important parts in a project is the project scope. In order to get the best result, the scopes are:

(i) Analyze gate mechanism consists of number of gates, location and size of the gates.
(ii) Using the Moldflow Plastic Insight (MPI 5.0) as the main software to analyze.
(iii) Comparison selected parameters and defects which are volumetric shrinkages, air traps, and sink index on each gate mechanism.

1.4 PROBLEM STATEMENT

The trends of producing a plastics product in injection molding industries are recently changing from traditional method to using the FEA analysis. For injection molding industries, time and cost is very important aspects to consider because these two aspects will directly related to the profits at a company. The next issue to consider is the number and location of the gate. In some cases, the product designers will indicate how much and where they believe the gate should be. Number and location of gates must be selected because the function and strength of the product depend on that factor.

The filling of cavity slow or impossible to fill the cavity full before it freezes. This is because of the gate that have been is too small. However, too large gate can make the gate and the product that been joining hard to break and will make mark in the product. In order to get the best parameter for the injection molding process, plastics have been waste. Through the experiment, operator will use large amount of plastics material to get the possibly parameters to setup the machine.

CHAPTER 2

LITERATURE REVIEW

2.1 INTRODUCTION

The mechanism of the gates consist of there are various types of gate design. The gates are classified by the characteristics of the reactions by which they are formed.

2.2 INJECTION MOULDING

Injection molding is a manufacturing technique for making parts from both thermoplastic and thermosetting plastic materials in production. Molten plastic is injected at high pressure into a mould, which is the inverse of the product's shape. Molding is widely used for manufacturing a variety of parts, from the smallest component. Injection molding is the most common method of production, with some commonly made items including bottle caps and outdoor furniture. The most commonly materials used is thermoplastic materials are polystyrene because they are low cost, lacking the strength and longevity of other materials, acrylonitrile butadiene styrene (ABS) is a co-polymer or mixture of compounds used for everything from Lego parts to electronics housings, nylon are chemically resistant, heat resistant, tough and flexible - used for combs, polypropylene also tough and flexible and used for containers, polyethylene, and polyvinyl chloride or PVC is more common in extrusions as used for pipes, window frames, or as the insulation on wiring where it is rendered flexible by the inclusion of a high proportion of plasticize as reported by Osswald, Tim A., Turng, Lih-Sheng, Graman, Paul J. (2002) .

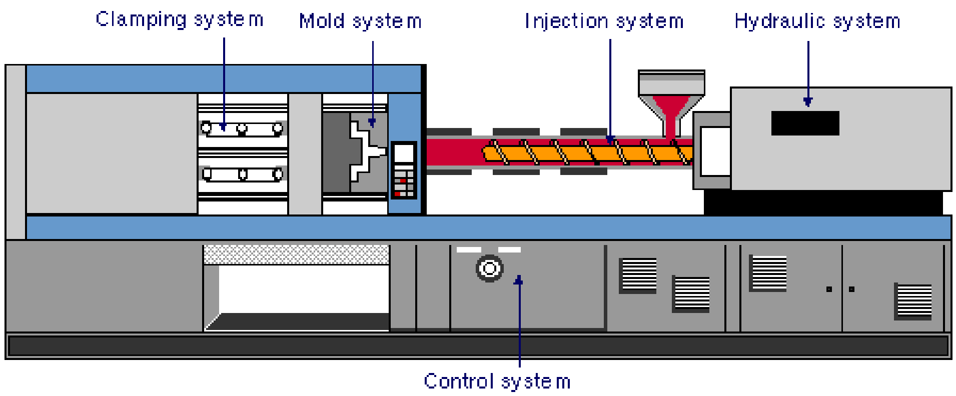


Figure 2.1: Injection moulding

Source: Osswald, Tim A., Turng, Lih-Sheng, Graman, Paul J. (2002)

2.2.1 Machine components

The injection system consists of a hopper, a reciprocating screw and barrel assembly, and an injection nozzle, as shown in Figure 2. This system confines and transports the plastic as it progresses through the feeding, compressing, degassing, melting, injection, and packing stages as reported by Beaumont, J. P., Nagel, R., and Sherman, R. (2002).

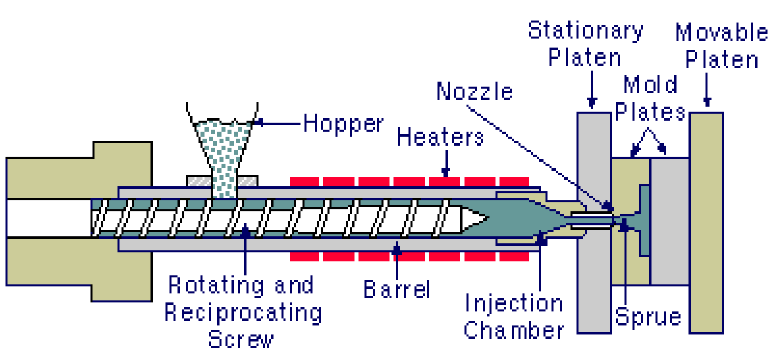


Figure 2.2: A single screw injection molding machine for thermoplastics

Source: Beaumont, J. P., Nagel, R., and Sherman, R. (2002)

The hopper **-** Thermoplastic material is supplied to molders in the form of small pellets. The hopper on the injection molding machine holds these pellets. The pellets are gravity-fed from the hopper through the hopper throat into the barrel and screw assembly.

The barrel **-** The barrel of the injection molding machine supports the reciprocating plasticizing screw. It is heated by the electric heater bands.

The reciprocating screw **-** The reciprocating screw is used to compress, melt, and convey the material. The reciprocating screw consists of three zones (illustrated below):

i. The feeding zone.
ii. The compressing (or transition) zone.
iii. The metering zone.

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. Typically, a molding machine can have three or more heater bands or zones with different temperature settings as reported by Beaumont, J. P., Nagel, R., and Sherman, R. (2002).

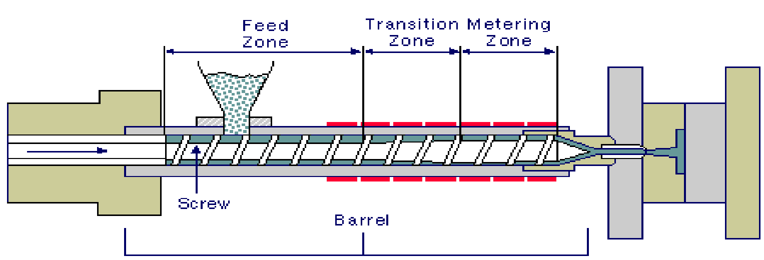


Figure 2.3: A reciprocating screw, showing the feeding zone, transition zone, and metering zone.

Source: Beaumont, J. P., Nagel, R., and Sherman, R. (2002)

The nozzle - The nozzle connects the barrel to the sprue bushing of the mold and forms a seal between the barrel and the mold. The temperature of the nozzle should be set to the material's melt temperature or just below it, depending on the recommendation of the material supplier. When the barrel is in its full forward processing position, the radius of the nozzle should nest and seal in the concave radius in the sprue bushing with a locating ring. During purging of the barrel, the barrel backs out from the sprue, so the purging compounds can free fall from the nozzle. These two barrel positions are illustrated below as reported by Beaumont, J. P., Nagel, R., and Sherman, R. (2002).

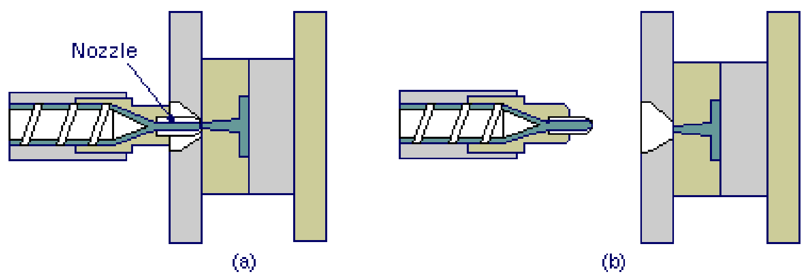


Figure 2.4: (a) Nozzle with barrel in processing position. (b) Nozzle with barrel backed out for purging.

Source: Beaumont, J. P., Nagel, R., and Sherman, R. (2002)

Clamping system - The clamping system opens and closes the mold, supports and carries the constituent parts of the mold, and generates sufficient force to prevent the mold from opening. Clamping force can be generated by a mechanical (toggle) lock, hydraulic lock, or a combination of the two basic types.

Molded system - A typical molded system consists of the delivery system and the molded part(s), as shown in figure 5. The delivery system which is provides passage for the molten plastic from the machine nozzle to the part cavity, generally includes:

i. A sprue
ii. Cold slug wells
iii. A main runner
iv. Branch runners
v. Gates

The delivery system design has a great influence on the filling pattern and thus the quality of the molded part as reported by Rees, H. and Catoen, B. (2006)

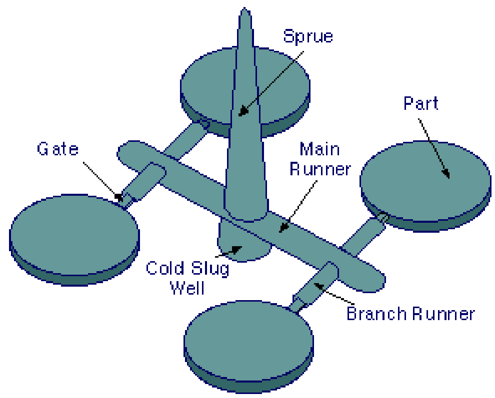


Figure 2.5: The molded system includes a delivery system and molded parts.

Source: Rees, H. and Catoen, B. (2006)

2.2.2 Machine sequence

Below is the sequence of operations in the injection molding of a part with a reciprocating screw. This process is used widely for numerous consumer and commercial products, such as toys, containers, knobs, and electrical equipment as reported by Harper, Charles A. (1999).

i.

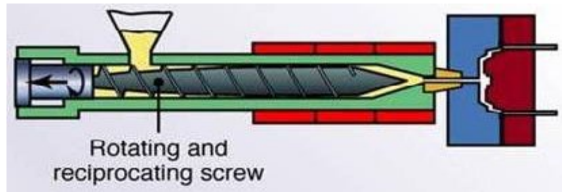


Figure 2.6: Build up polymer in front of sprue bushing. Pressure pushes the screws backwards. When sufficient polymer has built-up, rotation stops.

Source: Harper, Charles A. (1999)

ii.

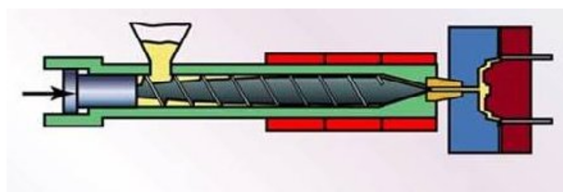


Figure 2.7: When the mold is ready, the screw is pushed forward by a hydraulic cylinder, filling the sprue bushing, sprue, and mold cavity with polymer. The screw begins rotating again to build up more polymer.

Source: Harper, Charles A. (1999)

iii.

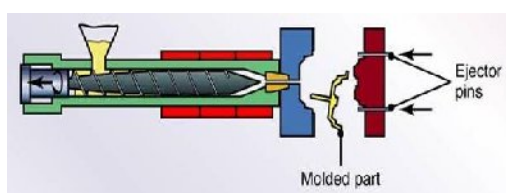


Figure 2.8: After is solidified or cured, the mold opens, and ejector pin remove the molded part.

Source: Harper, Charles A. (1999)

2.3 MOLD

Injection mold can be defined as arrangement, of one (or a number of) hollow cavity spaces built to the shapes of the desired product, with the purpose of producing (usually large number of) plastics parts, and a male mold part, called the core as reported Kalpakjian, Serope, and Schmid Steven R. (2006).

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 plastics is cooled and ready for injection, open the mold, and eject the finished products.

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

In cold-runner molds, the solidified plastic remaining in the channel connecting the mold 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 sprue attached to the mold part. Cycle times are shorter because only the molded part must be cooled and ejected.

2.3.1 Mold requirement

In designing 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, maintainence and interchangeability, ease of installation and also reasonable cost as reported by Biron,M. (2009). Injection molds must be properl designed to ensure quality plastics components. Mold design impacts productivity and profitability of molding operation.

2.3.2 Three plate runner molds

The primary advantage of the three-plate cold runner mold (Figure 2.9) over two-plate cold runner mold is the gating is no longer limited to the perimeter of the part cavity. Compared to hot runner systems, three-plate molds are low cost, relatively easy to operate, and provide for easy color changes as reported by Osswald, Tim A., Turng, Lih-Sheng, Graman, Paul J. (2002).

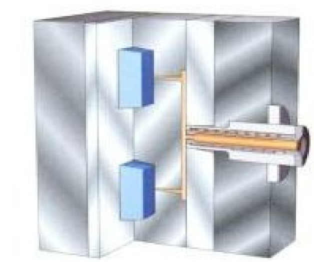
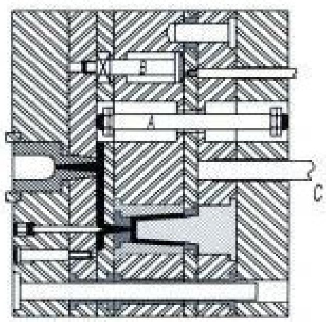


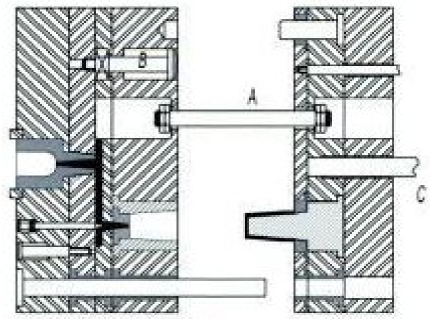
Figure 2.9 : Three plate runner mold

Source: Osswald, Tim A., Turng, Lih-Sheng, Graman, Paul J. (2002)

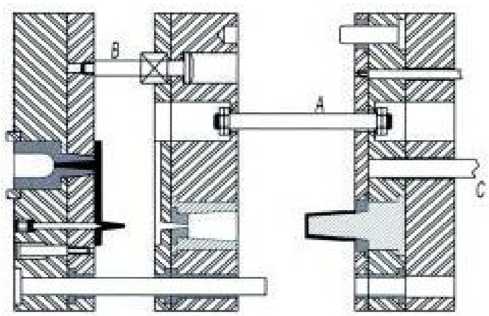
The three plate cold runner mold has a second parting plane located behind the cavity plate. The second parting plane, between cavity plate and top clamp plate, provides for a runner to travel under the mold cavity to any position relative to the part cavity. A secondary sprue transfer the melt from the runner, through the mold cavity insert, to a desired location on the part cavity. The secondary sprue is attached to the part by a small diameter pin gate. Owing to the increased flexibility in gating locations, the three plate cold runner mold might be used in multi cavity molds producing parts such as a cup, where gating in the center of the cavity is desirable.



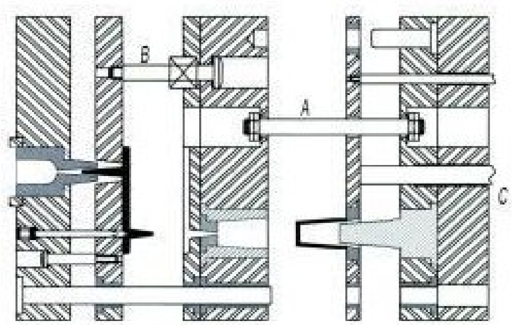
Step 1 – Mold closed



Step 2 – Main parting lines opens



Step 3 – Secondary parting lines opens



Step 4 - Part and runner are ejected

Figure 2.10: Opening and ejection action of a three plate mold with a hot sprue

Source: Kalpakjian, Serope, and Schmid Steven R. (2006)

Figure 2.10 show one variation of a three plate cold runner mold as reported by Kalpakjian, Serope, and Schmid Steven R. (2006). Here, the ejection of the part and runner begins with the mold first opening at a primary parting line, defined between the core and cavity plates (Step 2). At a position where the part has been fully retraced from the cavity, a pull rod (A) will begin to pull a floating cavity plate to open the mold at a second parting line (step 3). As the secondary parting line opens, a stationary sprue from puller with an undercut holds the base of the secondary sprue, or cold drop, such that the mold opens sufficiently for the secondary sprue to be fully relieved. The runner stripper plate, which ejects the molded part is triggered by the action of the ejector plate acting on push rod (C). the stripper plate ejecting the runner is activated by a second puller pin (D), which pulls the plate forward as the mold is opening.

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. As reported by Bryce,M.D. (1996). 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 shrinkage plastic in the cavity space connected for a longer period with the hot plastics supplied from the injection. This permits packing before the gate freezes. Small gates freeze faster and produce to facilitate separation of products from the runner, and to make the gate mark, or vestige, more inconspicuous.

2.4.1 Gate vestige

Vestige is the visual appearance (on the product) of the point of separation of the plastic between runner and product (break-off point). Its outline conforms to the shape of the gate. The gate shape is usually round, occasionally elliptical, half moon shaped, rectangular, or trapezoidal. The surface appearance of the break may be separated from hot plastic or dull when cold plastic is broken as reported by Rtp Company. (2005).

2.4.2 Number of gates

A minimum number of gates is generally desirable for the part as reported by Rtp Company. (2005).. However, manufacturing often requires that multiple gates are used. The fewer the gates, the fewer product problems resulting from gate vestige, blush, high residual gate stress, and welds from multiple flow fronts. However, a single gate can result in processing problems such as excessive fill pressure and clamp tonnage concerns. In some cases it may be desirable to reduce material shear rates and shear stresses by distributing flow during mold filling between multiple gates.

2.4.3 Gating position on a part

There are number of factor to be considered when a gating position on apart. Some of these are obvious, whereas others require a more in depth understanding of the plastic part formation process. The placement of the gate is an important consideration that can often affect shrinkage, molding efficiency, and part performance. However, as reported by Rtp Company. (2005). such suggested location may not always be the best for filling the cavity space or for the best strength properties of the product. At this point of the development, the input by a molder designers should be encouraged to find the best location for the gate.

2.4.4 Gate design

The gate serves as the entrance to the cavity and should be designed to permit the mold to fill easily. A cavity can. Gates should be small enough to ensure easy separation of the runner and the part but large enough to prevent early freeze-off of polymer flow, which can adversely affect the consistency of part dimensions as reported by Beaumont, John P. (2004). A variety of gate designs and location shown:

2.4.4.1 Sprue gate

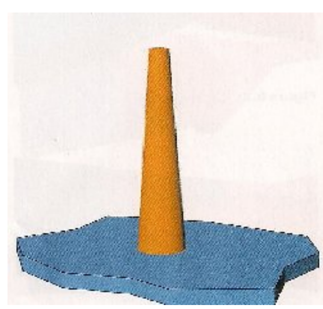


Figure 2.11: Sprue gate

Source: Beaumont, John P. (2004)

It is recommended for single cavity molds or for parts requiring symmetrical filling. This type of gate is suitable for thick sections because holding pressure is more effective. A short sprue is favored, enabling rapid mold filling and low-pressure losses. A cold slug well should be included opposite the gate. Typically, the part shrinkage near the sprue gate will be low; shrinkage in the sprue gate will be high. This results in high tensile stresses near the gate.

2.4.4.2 Common edge gate

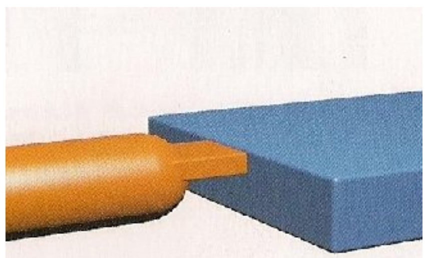


Figure 2.12: Common edge gate

Source: Beaumont, John P. (2004)

Common edge gates are the most basic type of gate. They are normally rectangular in cross-section and attach to the part, along its parameter, at the parting line of the mold. An edge gate would be preferable in a multi-cavity mold where parts are to be positioned for automated post-molding assembly. The edge gate will remain with the part maintaining the molded part’s position and orientation on the runner, which will provide for easy post mold handling, such as assembly, decoration, or inspection.

2.4.4.3 Fan gate

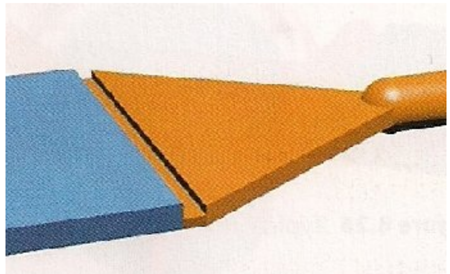


Figure 2.13: Fan gate

Source: Beaumont, John P. (2004)

Fan gates are similar to a basic edge gate in that they are attached to the part at the parting line. The difference is that the fan gates expend out from the runner in the shape of a fan with its widest end opening to the cavity. The fan region can be relatively thick and feeds a thin gate land, which is attached directly to the part. This design spreads and slows the melt as it enters the cavity. The benefits of the slower flow and the broad uniform melt flow from include improved melt orientation, reduce shear rates through the gate. Therefore the fan gate is use to create a uniform flow front into wide parts where warpage and dimensional stability are main concern.

2.4.4.4 Film gate or flash gate

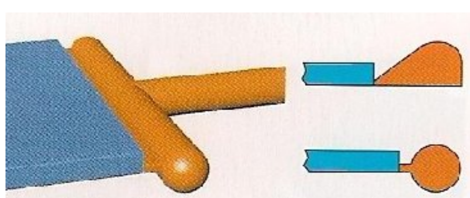


Figure 2.14: Film gate

Source: Beaumont, John P. (2004)

The film, or flash gate attempts to capture the advantages of the fan gate, while it uses less space and material. In this design runner attach to a get manifold that distribute the melt along abroad thin get land attach directly to the part. The disadvantage of this type of gate is the fact that the flow distribution across the gate and the flow through the gate is lee predictable then in the fan gate. The resulting filling pattern are sensitive process variation. Film or flash, gets work best at fast fill rate where hesitation is minimized.

2.4.4.5 Pin point gate

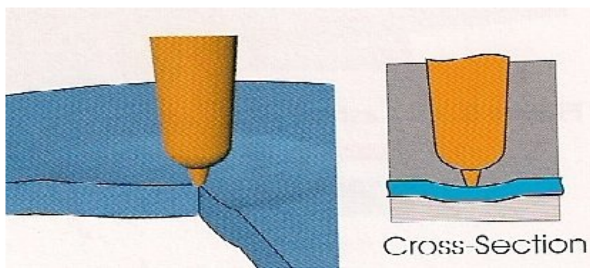


Figure 2.18: Pin point gate

Source: Beaumont, John P. (2004)

A pin point gate is restricted gate used in three-plate cold runner molds, where the runner system is located on the secondary mold parting line and the part cavity is on the primary parting line.

2.4.5 Defects on gate

Gate defects are always related to the design. These are some defects that has been detect on product produce by injection molding as reported by Direct industry. (2002). Defect of gate region shown:

2.4.5.1 Sink mark and voids

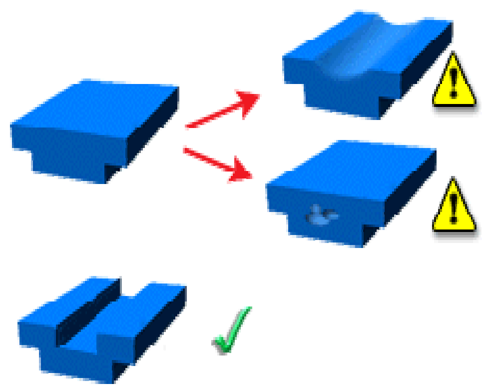


Figure 2.21: Sink and voids surface

Source: Direct industry. (2002)

Descriptions:

Sink marks and voids both result from localized shrinkage of the material at thick sections without sufficient compensation.

Cause:

1. Early gate freeze-off or low packing pressure may not pack the cavity properly.
2. High volumetric shrinkage.
3. Insufficient material compensation.

Solution:

Optimize packing profile. Alter part design to avoid thick sections and reduce the thickness of any features that intersect with the main surface. On long thin flat parts the gate is best placed between 60-70% down the part length to minimize warp.

2.4.5.2 Air traps

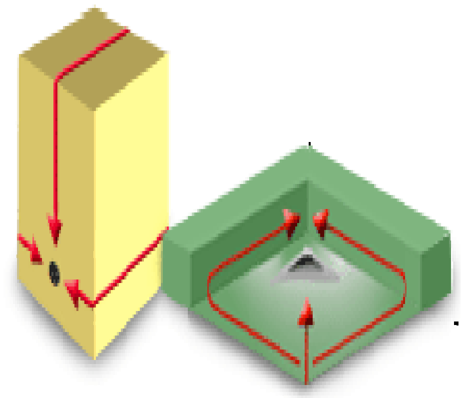


Figure 2.22: Air traps forms

Source: Direct industry. (2002)

Descriptions:

Air traps occur when converging flow fronts surround and trap a bubble of air. The trapped air can cause incomplete filling and packing, and will often cause a surface blemish in the final part. Air trapped in pockets may compress, heat up and cause burn marks.

Cause:

Flow paths do not need the racetrack effect or hesitation to have unbalanced flow. In a part with uniform thickness, the physical length of flow paths may vary, and again air traps may occur. Also lack of vents or undersized vents in these last-to-fill areas are a common cause of air traps.

Solution:

Place the gate in such a manner as to push the knit lines into obscure areas. Changing the runner system can alter the filling pattern in such a way that the last-to-fill areas are located at the proper venting locations. If air traps do exist, they should be positioned in regions that can be easily vented or ejection and/or vent pins added so that air can be removed.

2.5 MOLDFLOW

Moldflow offers a range of a products and services in the plastics injection molding industry. It is easy to learn 3D solids based plastics flow simulation products allow you to determine the manufacturability of your part in the early design stages and avoid potential downstream problems which can lead to production delays and cost overruns.

Moldflow software has been develop by moldflow international Pvt. Ltd., Australia. It helps in finite element analysis used in the design plastics product, mould design and production of plastic components. Following are the modules o moldflow software. Flow analysis (MF/FLOW); The flow analysis is used to determine the gates position and filling patter. It analyses polymer flow within the mould, optimizes mould cavity layout, balance runner and obtains mould processing conditions for filling and packing phases of the molding cycle as reported by Direct industry. (2002).

Cooling analysis (MF/COOL); it analyses the effect cooling on flow, optimizes cooling line geometry and processing conditions. Process Optimization Analysis (MF/OPTIM); it gives optimized processing parameters for a component considering injection molding conditions. Warpage Analysis (MF/WARP); this analysis simulates the effect of moldings on product geometry, isolates the dominant cause of warpages so that the correct remedy can be applied.

2.5.1 Moldflow Plastics Insight (MPI)

Moldflow Plastic Insight products are a complete suite of advanced plastics process simulation tools for predicting and eliminating potential manufacturing problems simulations tools for predicting and eliminating potential manufacturing problems nd optimizing part design, mold design and the injection molding process. MPI products simulate the broadest range of manufacturing processes. With MPI, one can simulate the filling, packing and cooling stages of the thermoplastics injection molding process and also predict the resultant fiber orientations and take that into account when predicting part warpage. MPI users can also simulate other complex molding process such as gas assisted injection molding, co-injection molding, injection-compression molding, microcellular molding, reactive molding, and microchip encapsulation.

MPI also allows to do some trouble shooting very easily. Some of the material we use are very expensive. Therefore, less time on the production floor working through a problem saves labor and material costs. Using MPI, we have been able to run simulations and locate and eliminate unsightly nit lines.

MPI is being employed in both tooling design and simulation of molding. MPI used to simulate mold designs before the tool is actually built. The simulations helps user determine different gate designs and locations, placement of cooling lines, and melt overflows.

The Moldflow Plastics Insight suite of software is the world leading product for the in-depth simulations to validate part and mold design. Companies around the world have chosen Moldflow's solution because they offer; Unique, Patented Fusion Technolgy. MPI/Fusion, which is based on Moldflow's patented Dual Domain^{TM} Technology, allows you to analyze CAD solid models of thin-walled parts directly, resulting in a significant decrease in model preparation time. The time savings allow you to analyze more design iterations as well as perform more in depth analyzed as reported by Direct industry. (2002).