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) .

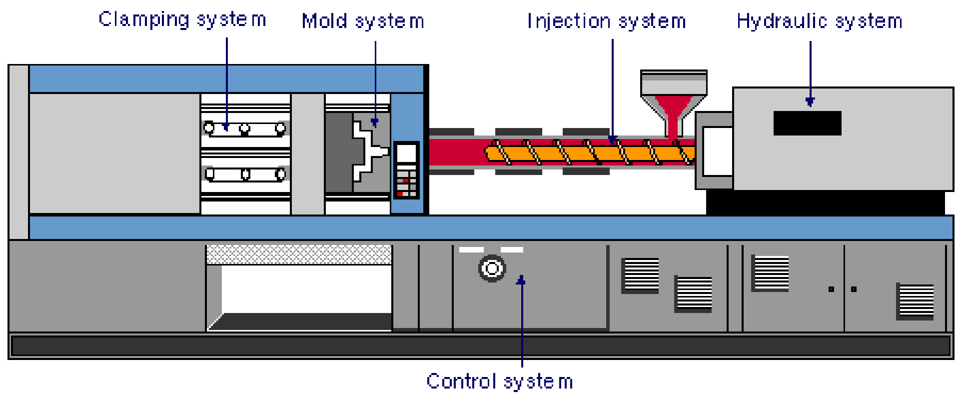


Figure 2.1: Injection moulding

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).

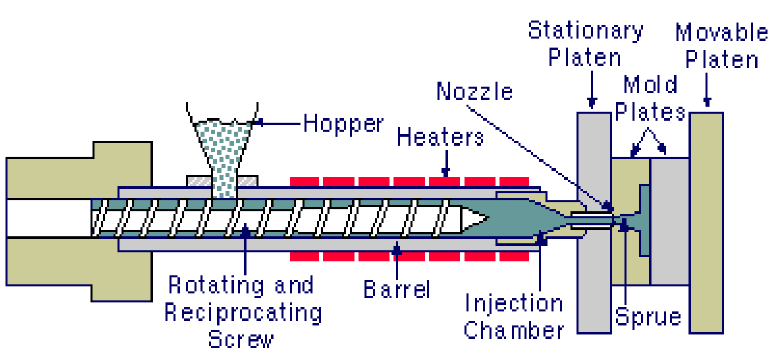


Figure 2.2: A single screw injection molding machine for thermoplastics

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).

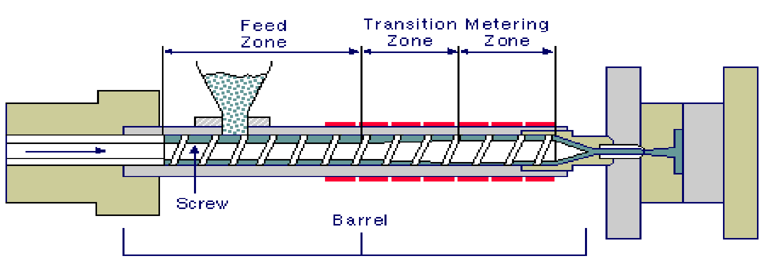


Figure 2.3: A reciprocating screw, showing the feeding zone, transition zone, and metering zone.

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).

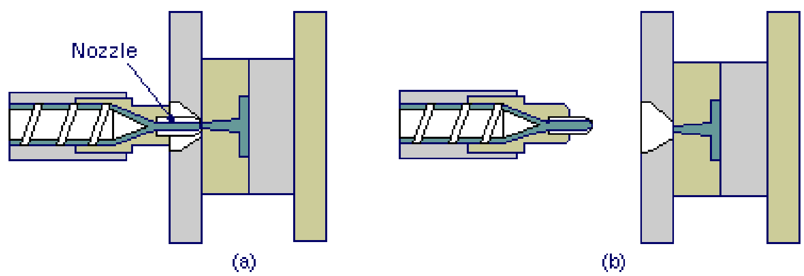


Figure 2.4: (a) Nozzle with barrel in processing position. (b) Nozzle with barrel backed out for purging.

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)

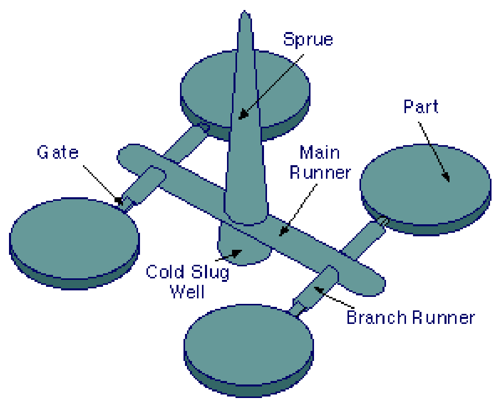


Figure 2.5: The molded system includes a delivery system and molded parts.

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.

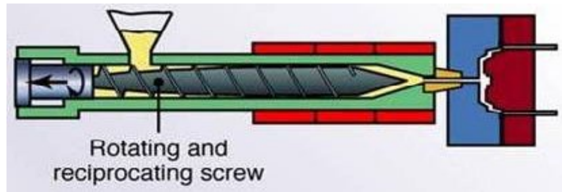


Figure 2.6: Build up polymer in front of sprue bushing. Pressure pushes the screws backwards. When sufficient polymer has built-up, rotation stops.

Source: Harper, Charles A. (1999)

ii.

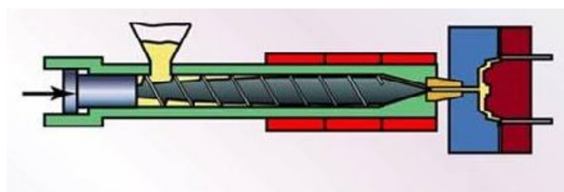


Figure 2.7: When the mold is ready, the screw is pushed forward by a hydraulic cylinder, filling the sprue bushing, sprue, and mold cavity with polymer. The screw begins rotating again to build up more polymer.

Source: Harper, Charles A. (1999)

iii.

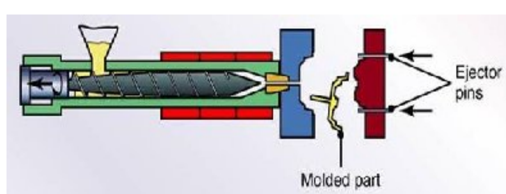


Figure 2.8: After is solidified or cured, the mold opens, and ejector pin remove the molded part.

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).

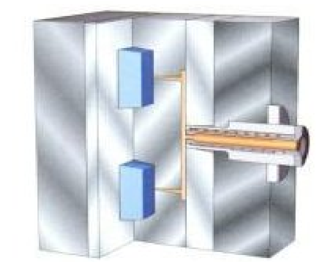
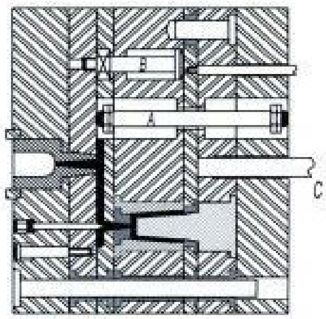


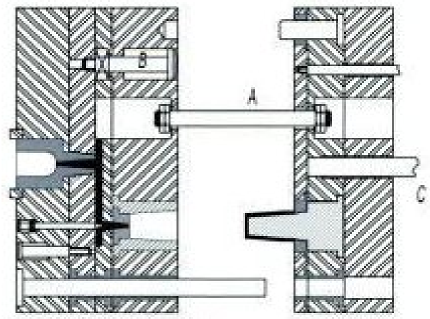
Figure 2.9 : Three plate runner mold

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

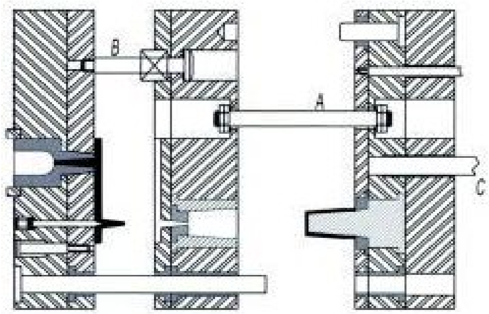
The three plate cold runner mold has a second parting plane located behind the cavity plate. The second parting plane, between cavity plate and top clamp plate, provides for a runner to travel under the mold cavity to any position relative to the part cavity. A secondary sprue transfer the melt from 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.



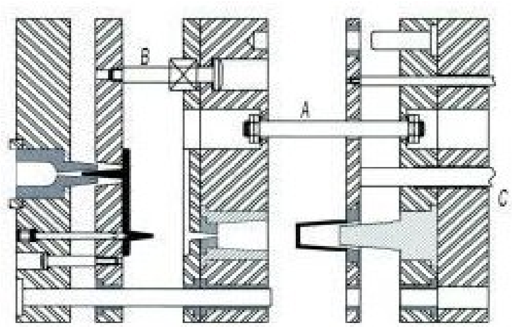
Step 1 – Mold closed



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

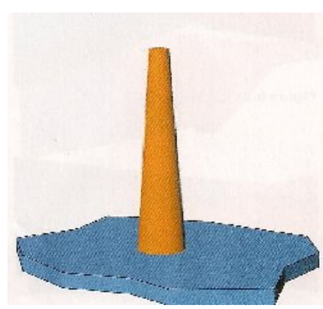


Figure 2.11: Sprue gate

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

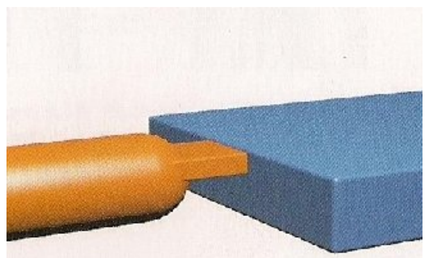


Figure 2.12: Common edge gate

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

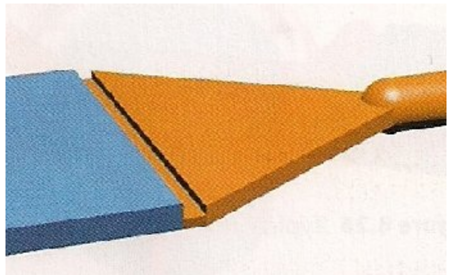


Figure 2.13: Fan gate

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

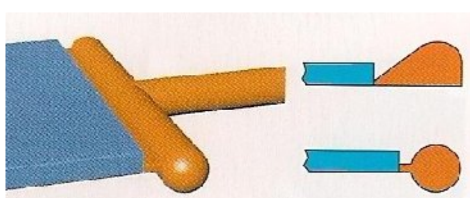


Figure 2.14: Film gate

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

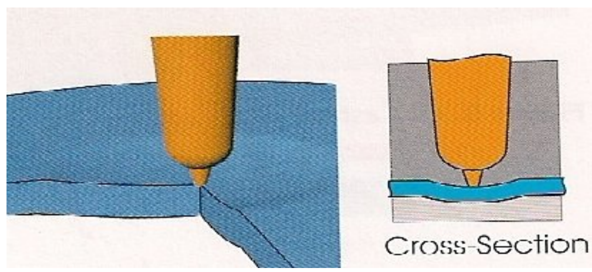


Figure 2.18: Pin point gate

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

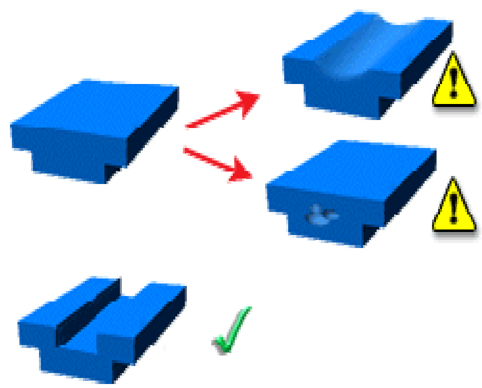


Figure 2.21: Sink and voids surface

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

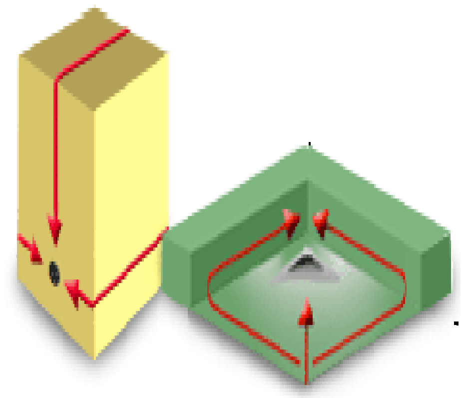


Figure 2.22: Air traps forms

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).

CHAPTER 3

METHODOLOGY

3.1 INTRODUCTION

This chapter discuss about methodology of the project. Besides that, this chapter shows the time line from the start till finished. The time line starts when receive the project title and start it with some briefing session with supervisor.

The methodology is as stages or steps that need to be follow and this will ensure the project done according to the planning. Methodology as an algorithm that finds a solution in the given environment of the multi-layered finite space consisting of the problems statement, project scopes and objective, literature review, product selection, dimensioning drawing, material selection, Moldflow analysis, modeling the design and documentation.

Analysis by using Moldflow software, MPI 5.0, is the main step in getting result. Through the analysis, comparison of the result will be done. It is important that the analysis have followed the objective and also the project scope. The results also have to achieve the project objective.

3.2 PROJECT FLOW DIAGRAM

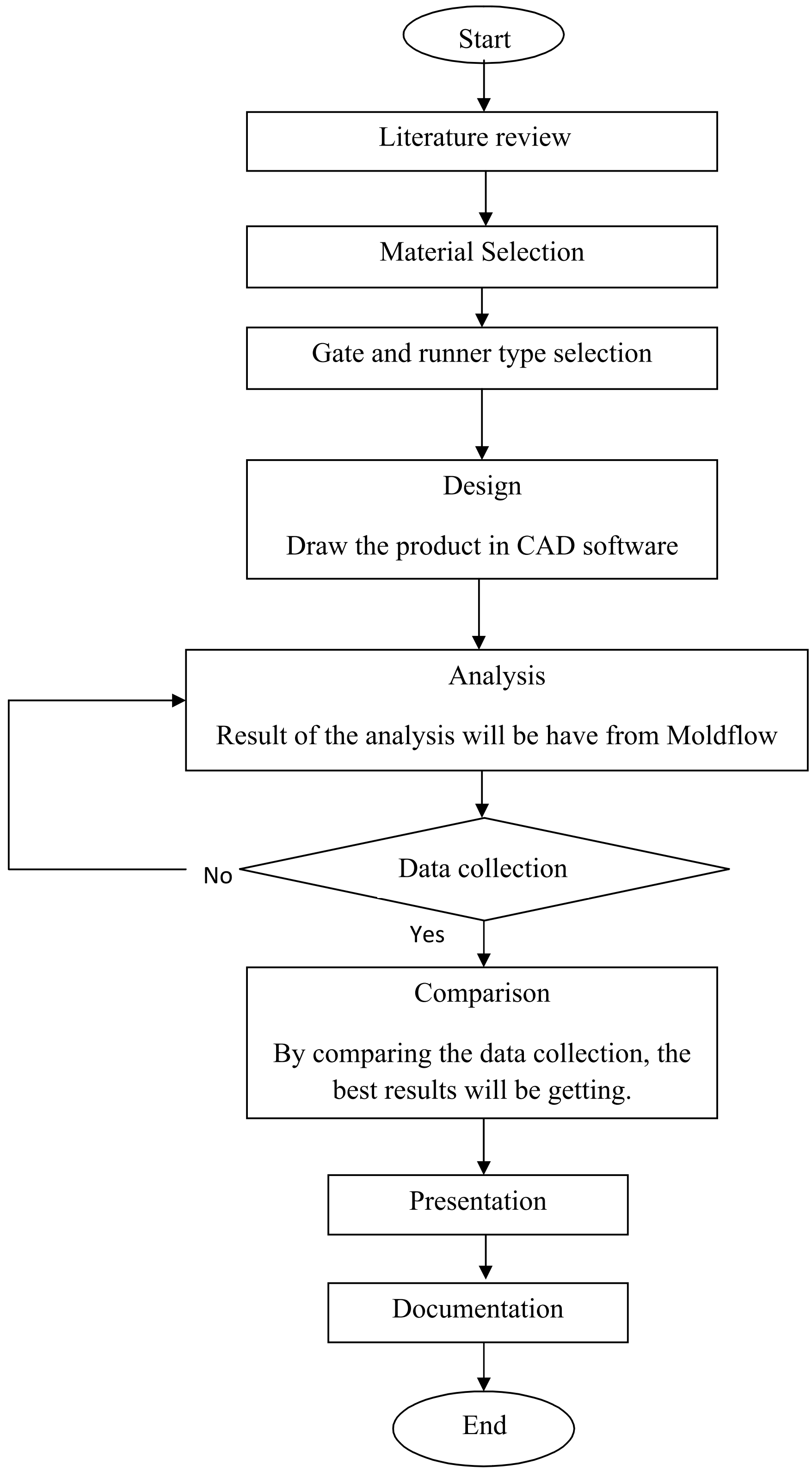


Figure 3.1: Project Flow Chart

3.3 LITERATURE REVIEW

The flow for this project will start with gather information by research and literature review via internet, journal, reference books, supervisor and other relevant academic material that related to this project. The literature is more about the injection moulding process and parameter, gate mechanism, types of defects, cycle time, and Moldflow plastic insight (MPI) software.

3.4 MATERIAL SELECTION

The product selected for this project is book tray. Based on function and characteristic of book tray, there are a few suitable materials for this product.

i) Polycarbonate (PC)
ii) Acrylonitrile Butadiene Styrene (ABS)
iii) Polyethylene (PE)
iv) Polystyrene (PS)

Table 3.1: Table Material Selection Matrix

Properties									
Resin	**Properties 1: Impact resistance**		**Property 2: Toughness**		**Property 3: Surface Finish**		**Property 4: Cost**		**Total score**
	Value/ Rank	**WF=5 Score**	**Value/ Rank**	**WF=5 Score**	**Value/ Rank**	**WF=3 Score**	**Value/ Rank**	**WF=4 Score**	
PC	4	20	4	20	3	9	1	4	53
ABS	3	15	3	15	3	9	4	16	55
PE	2	10	2	10	2	3	3	12	35
PS	1	5	1	20	1	3	2	8	36

From the table 3.1 of material selection matrix, Acrylonitrile Butadiene Styrene is chosen because the material has good characteristic such as impact resistance, toughness, surface finish and less cost.

The best plastic material will selected from four materials above by using the table 3.1 of material selection matrix. Above show the table of material selection matrix. Book tray needs this type of material because this product usually needs a good resistant to impact and toughness which is comes from books or papers work in order to avoid the product broken while used by the user. These materials also have good surface finish and looked attractive. The most important is the price for this material is cheaper than other material. It will become the factor to user buy this product which is more economic.

Acrylonitrile Butadiene Styrene (ABS) has been choosing as the plastic material that will be used in the analysis. The trade name and manufacture of material that are available in Moldflow software is shows in table 3.2. The type of material is required to run the analysis. Type of material determine from the where manufacture it is and the specific name for that material in the software. The analysis parameter also requests the melt temperature of material that we have defined. Then table 3.3 shows the recommended melting temperature for several materials for injection molding process. The mold temperature state as constants parameters like melting temperature and it temperature is recommended in table 3.4.

Table 3.2: Table selected material with their manufacturer

Material	**Trade name**	**Manufacturer**
Acrylonitrile Butadiene Styrene (ABS)	Toyolac 100	Toray industry incorporation
Polycarbonate (PC)	Panlite L-1225	Teijin chemicals
Polyethylene (PE)	Dowlex 2517	Dow chemical (USA)
Polystyrene (PS)	Austrex 103	Polystyrene Australia

Source: Biron,M. (2009)

Table 3.3: Table of suggested melt temperature for selected plastics material

Plastic material	**Temperature (^{o} C)**
Acrylonitrile Butadiene Styrene (ABS)	216
Polycarbonate (PC)	288
Polyethylene (PE)	204
Polypropylene (PP)	177
Polystyrene (PS)	199
Polyethylene terephthalate (PET)	232
Acetal (POM,Polyacetal)	218
Polybutylene terephthalate (PBT)	218

Source: Biron,M. (2009)

Table 3.4: Table of suggested mold temperature for selected plastics material

Plastic material	**Temperature (^{o} C)**
Acrylonitrile Butadiene Styrene (ABS)	85
Polycarbonate (PC)	104
Polyethylene (PE)	43
Polypropylene (PP)	49
Polysulfone (PS)	82
Polyethylene terephthalate (PET)	99
Acetal (POM,Polyacetal)	99
Polybutylene terephthalate (PBT)	82

Source: Biron,M. (2009)

From the table 3.3 and table 3.4 above, to obtained the best result the suggested melt and mold temperature will used in the gate mechanism analysis which is will be setup in Moldflow software.

3.5 GATE TYPE SELECTION

From table 3.5, the design engineer can choose from a large number of runner systems to offer the optimum quality and economics to the user. These are: (Hanser, 2001)

I Runner which remains with the molded part and have to be cut off afterwards

II Runner which are automatically separated from the molded part and are demold separately.

III Runner which are automatically separated from the molded part during demolding but remain in the mold.

Table 3.5: Classification of runner

Gating System	
I	1. Sprue gate 2. Edge gate 3. Disk gate 4. Ring gate
II	5. Tunnel gate (submarine gate) 6. Pinpoint gate (in three-plate mold)
III	7. Pinpoint gate (with reserved sprue) 8. Runnerless gating 9. Runner for stack mold 10. Insulated runner 11. Hot manifold

Source: Beaumont, John P. (2004)

From Design of Plastic Moulds and Dies, L.Sors and Balazs,1989, the inlet dimensions (gate) for the injection molding are depending on the mass of the injected object as shown as Table 3.6 below.

Table 3.6: Gate dimensions based on mass(g) of workpiece or product.

Mass of workpiece (g)	Sprue (direct gate) Ø mm	Needle gate (pinpoint or tunnel) Ø mm	Rectangular gate mm
to 10	2.5 – 3.5	0.8	1.0 x 2.0 – 2.0 x 2.0
11 – 20	3.5 – 4.5	0.8	1.5 x 2.5 – 2.5 x 3.5
21 – 40	4.0 – 5.0	1.0 – 1.2	2.0 x 3.0 – 2.5 x 3.5
41 – 150	4.5 – 6.0	1.5 – 2.5	2.5 x 3.5 – 3.5 x 4.5
151 – 300	4.5 – 7.5	1.5 – 2.8	2.5 x 3.5 – 3.5 x 4.5
301 – 500	6.0 – 8.0	1.8 – 3.5	-
501 – 1000	8.0 – 10.0	-	-
1001 – 5000	10.0 – 15.0	-	-

Source: L.Sors and Balazs (1989)

From the table 3.5, the classification of runner will indicate the suitable gates to use in mold. So, from there the classification of II is the best which are automatically separated from the molded part and are demold separately. The types of gate to choose goes to pin point gate because the mold to produce book tray is in the single cavity while the submarine gate used in multi cavity.

Based on table 3.6, the gate dimension will be base on the mass of the work piece or product. The mass for book tray is in 151 grams to 300 grams range and the dimension will be in range 1.5mm to 2.8mm. This range will be used in gate size analysis.

3.6 RUNNER SELECTION

Runner is important as its function to reduce the pressure when plastic material injected into the mold. To valid this analysis the runner diameter must be constant, in order to get the best result in gate mechanism analysis. Since the actual using 8mm in runner, so this diameter will use in this analysis. Shape of the runner according to the table below and the most possibly runner shape to use in the three plate mold is full round cross-section.

Table 3.7: Gate dimensions based on mass(g) of workpiece or product.

Cross-section		Descriptions
(a) Full Round	**Advantages**	Smallest surface relative to cross-section, slowest cooling rate, low heat and frictional losses, center of channel freezes last therefore effective holding pressure.
	Disadvantages	Machining into both mold half is difficult and expensive.
(b) Trapezoidal	**Advantages**	Best approximation of circular cross-section, simpler machining in one mold half only (moveable half)
	Disadvantages	More heat losses and scrap compare to circular cross-section

Source: L.Sors and Balazs (1989)

3.7 DESIGN OF THE PRODUCT

The product has been selected is book tray. The technical drawing Solidwork 2006 has been used. The part that have been draw will use in moldflow analysis. The dimensions of this part almost the same with the exact one. The drawing file needs to save as in 'Iges file'. This type of file will be export to the Moldflow plastic insight (MPI) software for simulating process. Below are front view, side view, top view and isometric drawing of product. Please refer to appendix B for technical drawing.

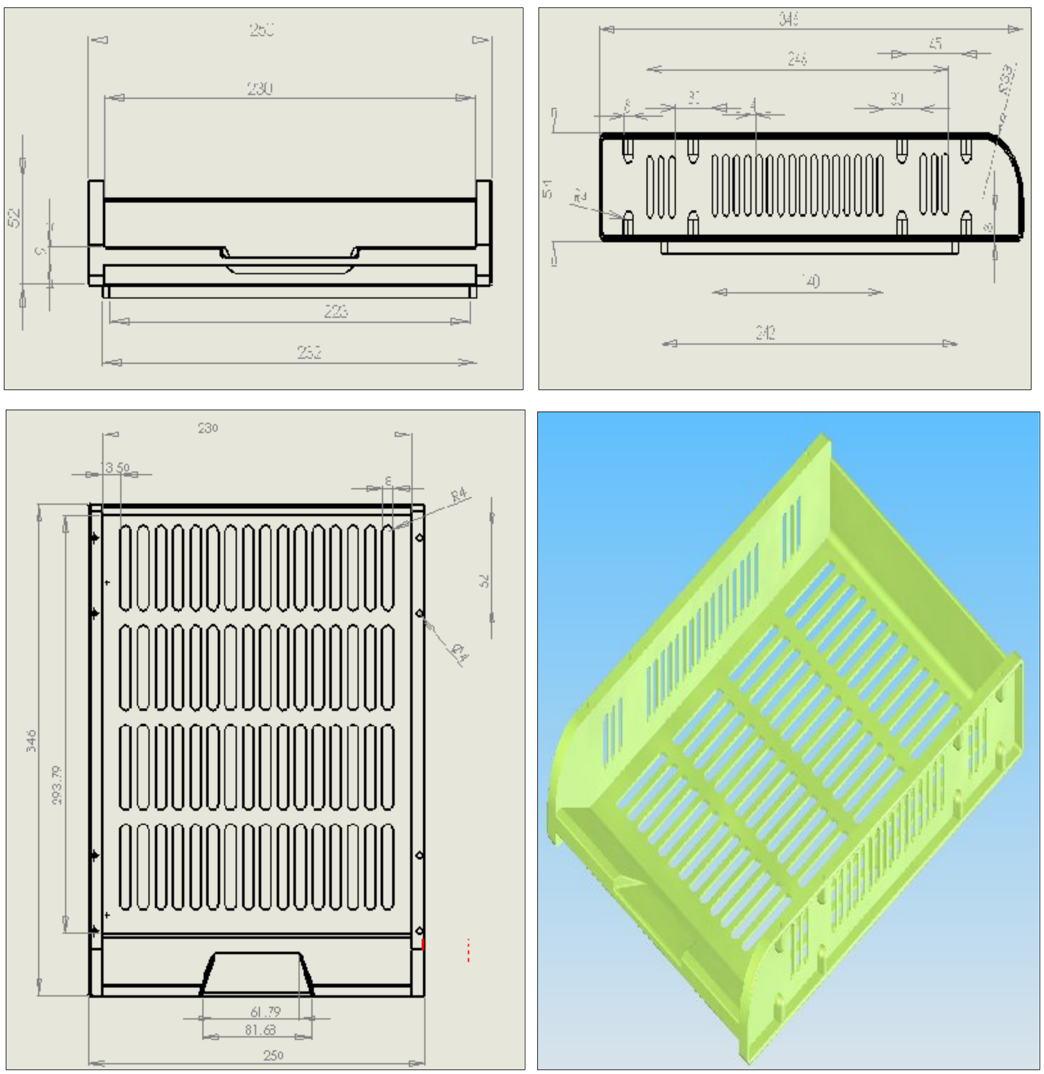


Figure 3.2: Drawing of the book tray that will be analyzed in MPI 5.0. The dimension of the mold will use in the software.

3.8 ANALYSIS

The analysis of project will be perform by simulate the plastic product using Moldflow Plastics Insight software. The purpose of the analysis is to know the effect of gates mechanism on injection molding parameters and the defects occurred to product. In analysis setup, all the mould parameter follows the actual mould such as get location, type of gate, cooling system, and runner system. The others parameters such as melt temperature, mold temperature constant and certainly based on gates characteristic. The analysis need to repeat eight time with different material. Below are a few steps in using the Moldflow software and the constant process parameters.

Step in running the Moldflow simulation

a) Create new project

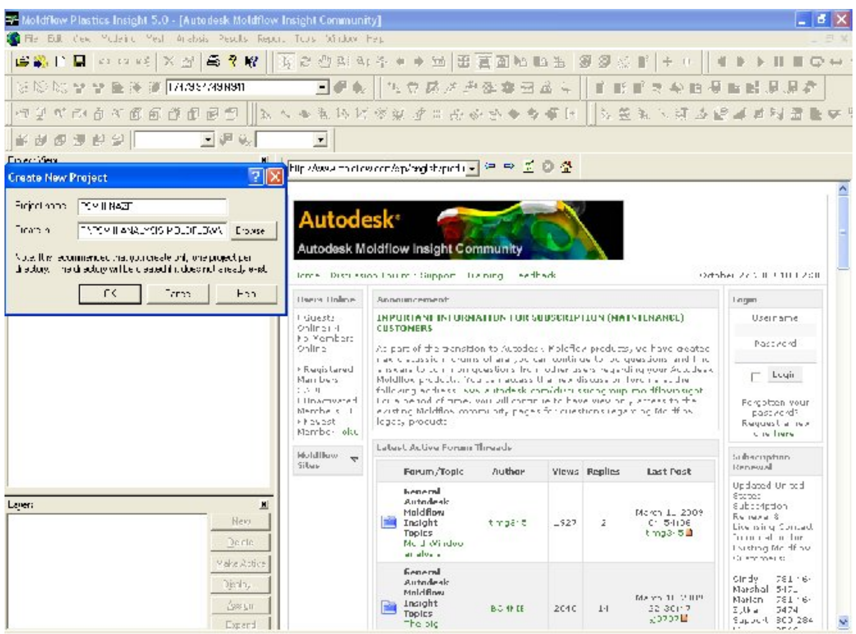


Figure 3.3: Create new project

b) Import 3-D CAD file

i.Import ‘iges’ file

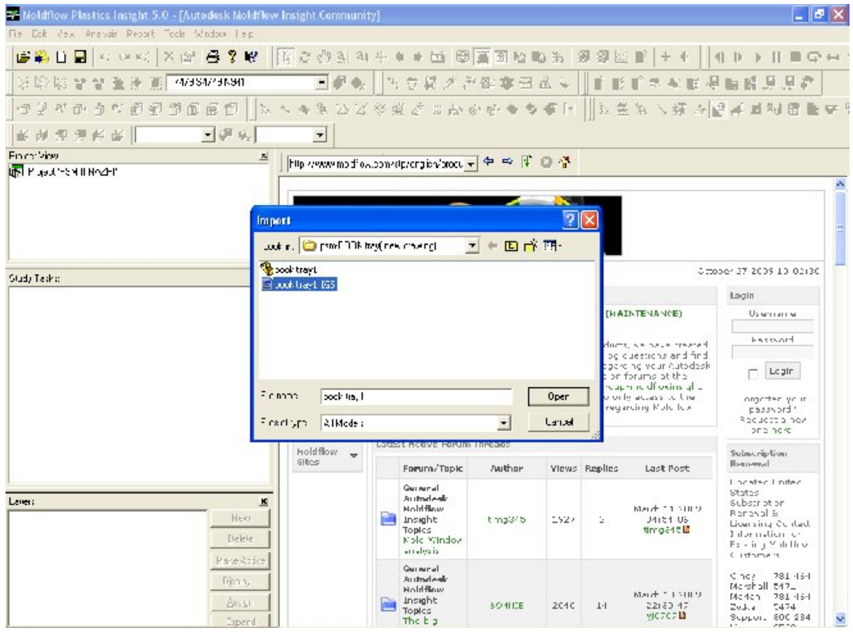


Figure 3.4: Import ‘iges’ file from CAD drawing

ii.Generate mesh

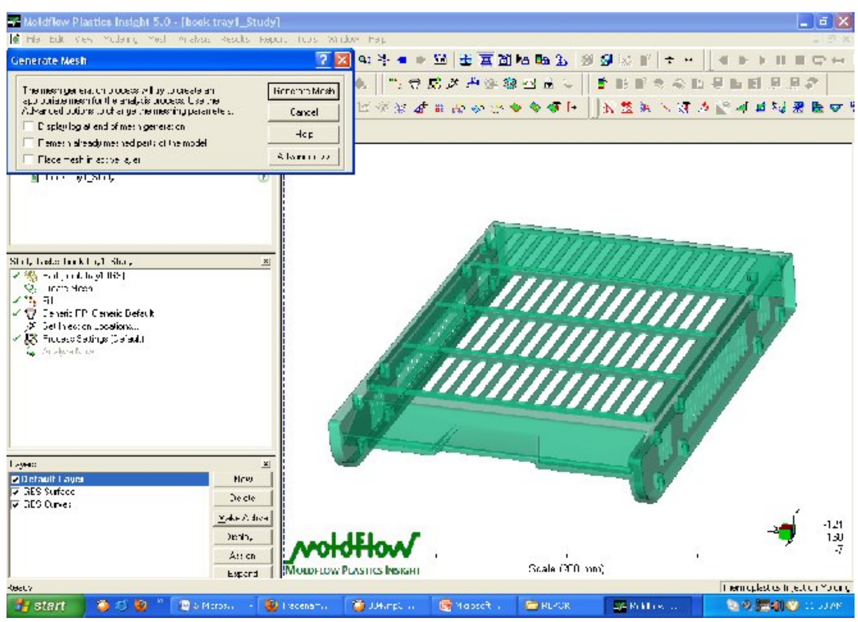


Figure 3.5: Generate the meshing entity

iii.Mesh pattern was appeared as selected mesh type

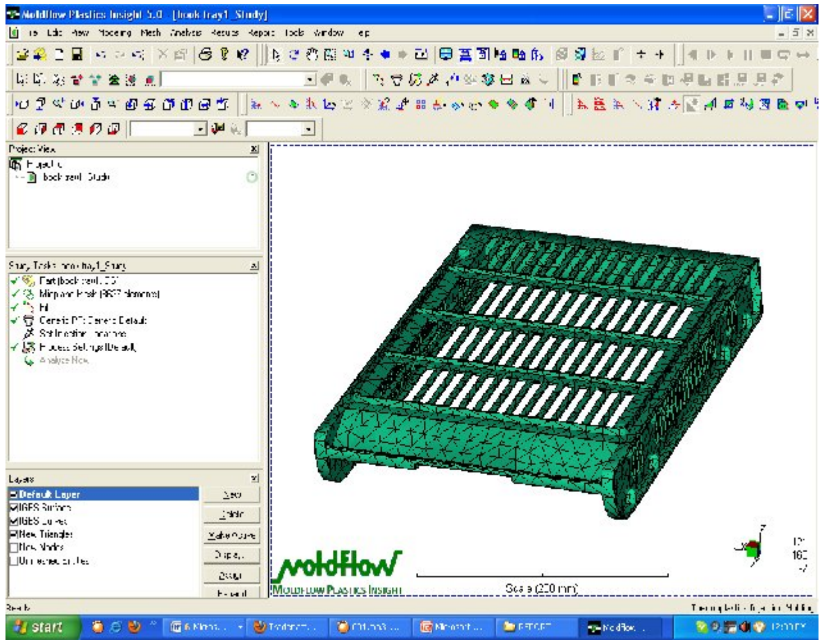


Figure 3.6: Mesh pattern appeared on Book Tray product (fusion)

c) Set analysis sequence

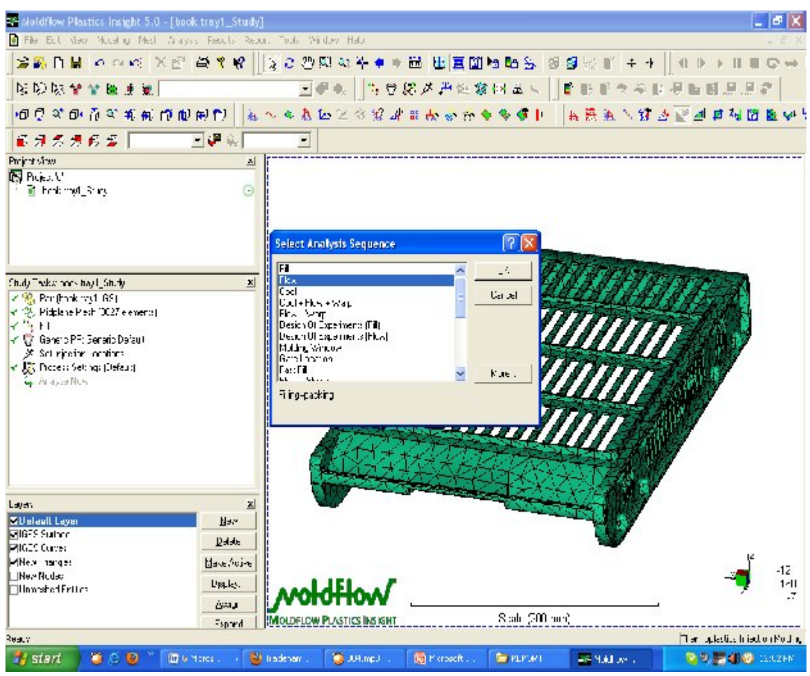


Figure 3.7: Analysis sequence wizard (flow analysis)

d) Select the material

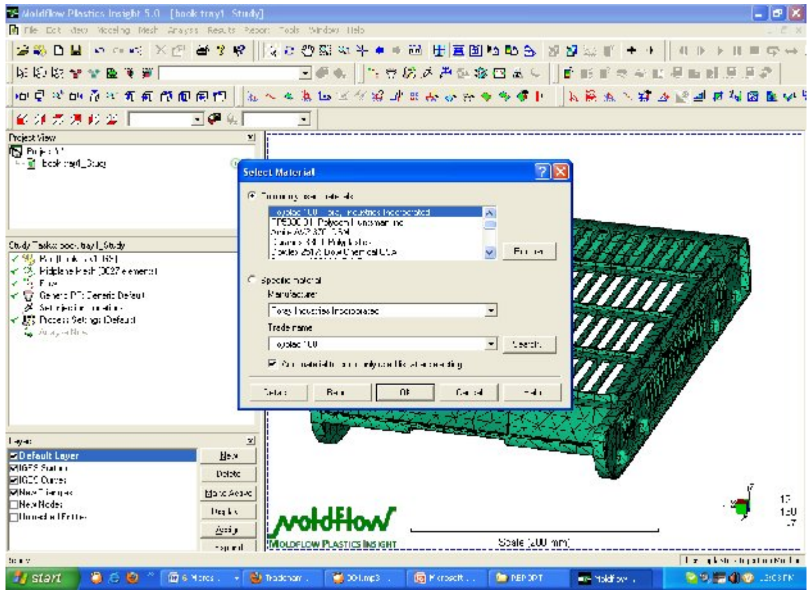


Figure 3.8: Toyolac 100 fom ABS family have been chosen

e) Set gate location and type of gate

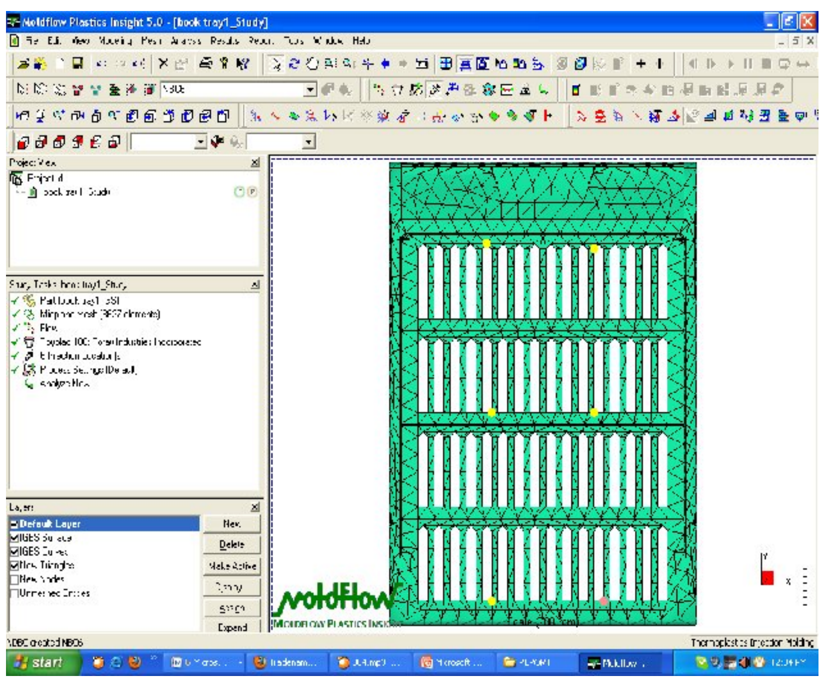


Figure 3.9: Gate location

f) Set runner and gate system

i. Set up in the small window below

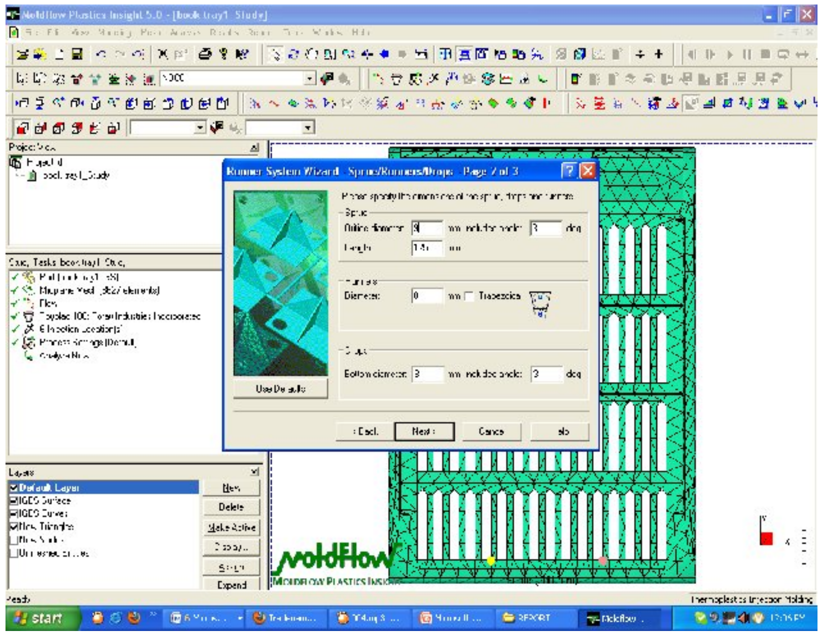


Figure 3.10: Runner and gate system to set up

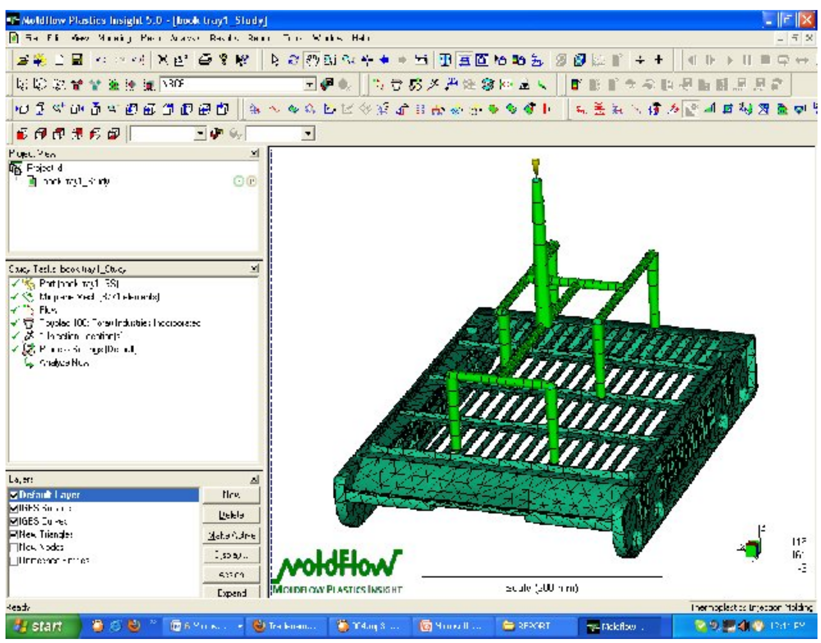


Figure 3.11: Runner and gate system be done

g) Set cooling system

i.Set cooling system wizard

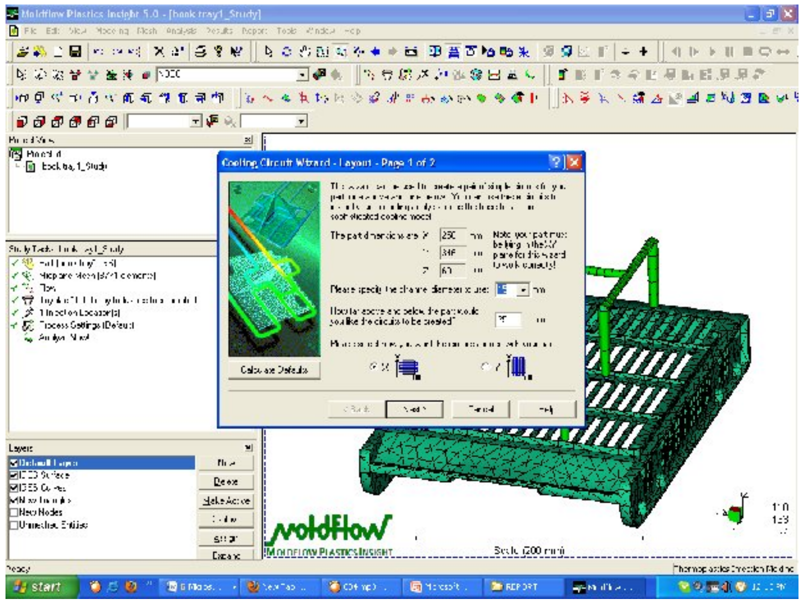


Figure 3.12: Cooling system type setup

ii.Cooling system was appeared

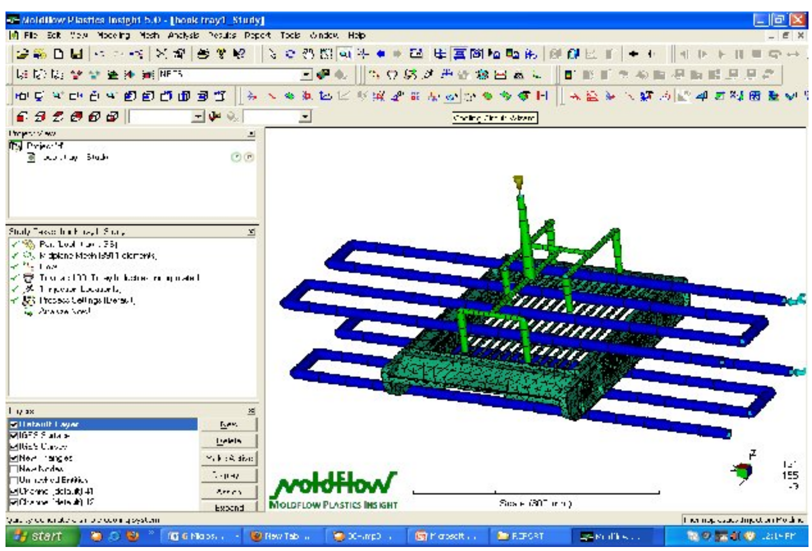


Figure 3.13: Complete modelling with cooling and runner system

h) Set process parameters

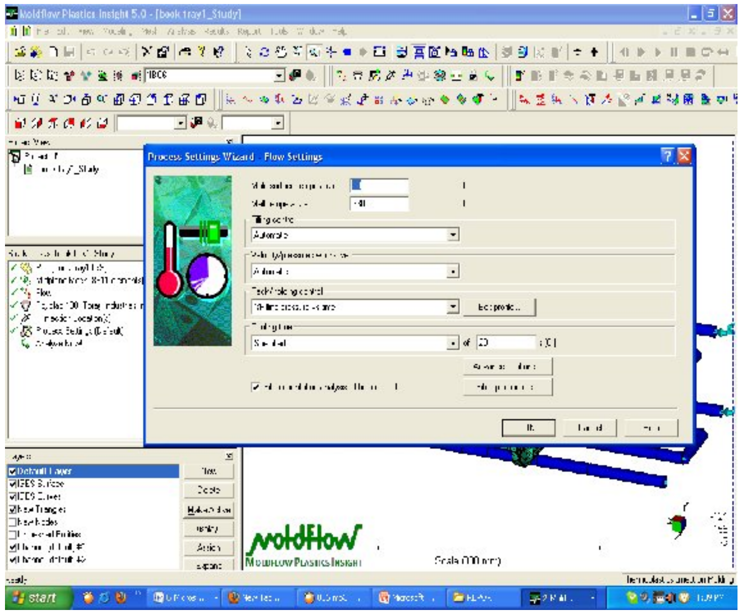


Figure 3.14: Process parameter setup

i) Perform the analysis

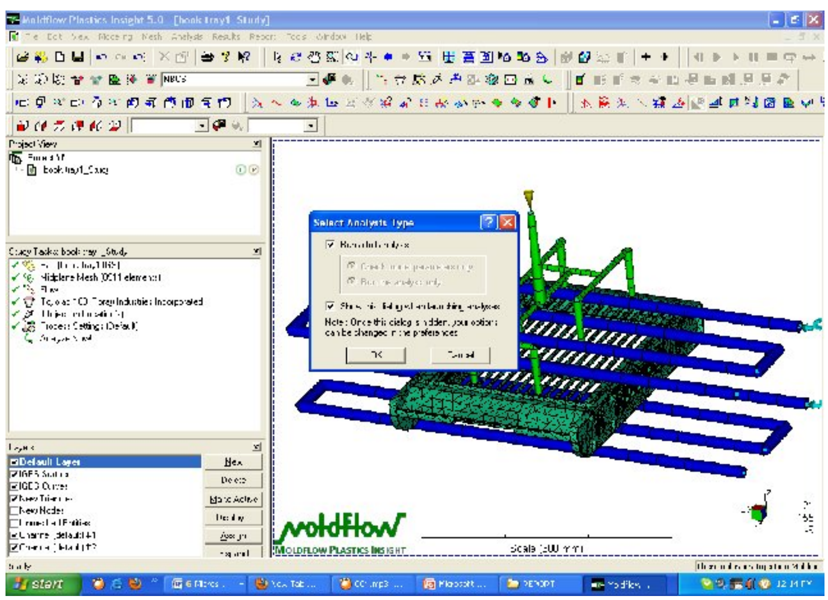


Figure 3.15: Perform the analysis

3.9 DATA COLLECTION

Data collection is the process after the analysis process. The result from the analysis needs to record as shown on the table below. The injection moulding parameter needs to consider or record are results analysis and defect analysis. Below is the table of data collection.

Table 3.8: Result analysis for different number of gates, location and size of gates

Number of Gates	Four gates	Six gates	Eight gates
Fill time (s)			
Max. injection pressure during filling (MPa)			
Max. clamp force during filling (tonne)			
Total part weight at the end of filling (g)			
Runner system volume (cm^3)			

Table 3.9: Defect analysis for different number of gates

Number of Gates	Four gates	Six gates	Eight gates
Volumetric shrinkage (%)			
Sink Index (%)			
Air traps (level)			

3.10 RESULT COMPARISON

Result comparison will be compared to the data above which is consist the filling time, maximum injection pressure, maximum clamp force, total part weight in the end of filling, and the runner system volume. For the defects analysis includes volumetric shrinkage, sink index and air traps level.

CHAPTER 4

RESULT AND DISCUSSION

4.1 INTRODUCTION

This chapter discusses the results that have been taken from the Moldflow analysis had been compared. The analyses that were done based on the gate mechanism such as number of gate, location of gates and the size of gates. After the analysis is completed, the results were compared based on parameters and defects. The trend of the result also had been investigated. Most of the discussion based on the figure and data that have been collected.

4.2 DATA

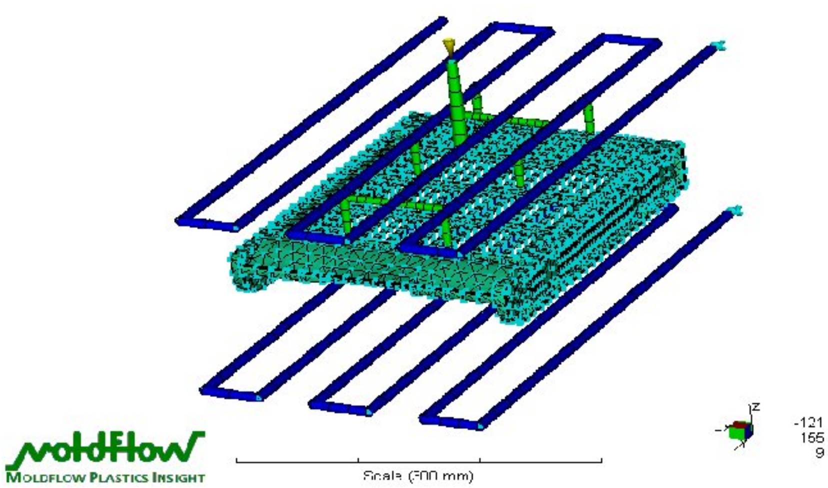


Figure 4.1: Meshing with runner and cooling system

Figure 4.1 shows the book tray with the meshing entity, runner system and cooling system. The mesh is a web that consists of elements, with each element containing a node at every corner. The mesh provides the basis for a Moldflow analysis, where molding properties are calculated at every node. The cooling system use in this analysis of gate mechanism is circular type with 10.0mm in diameter. Pure water acts as the cooling with 10.0lit/min flow rate with constant time of cooling which is 20 second in each analysis. The sprue opening should be as small as possible but must fill the cavity effectively. The sprue used is circular type with 9mm orifice diameter included angle, 3 degree and 100mm in length. The runner diameter is 8mm and the drops will be 8mm too. The type of gates is pin point gate which is suitable gates on a three-plate mold design with 1.6mm in diameter. This model will analyzed in terms of gate mechanism which are consist number of gates, location of gates, exclude size of gates which is the size of the head will be different.

Table 4.1: General properties of gate mechanism analysis using ABS

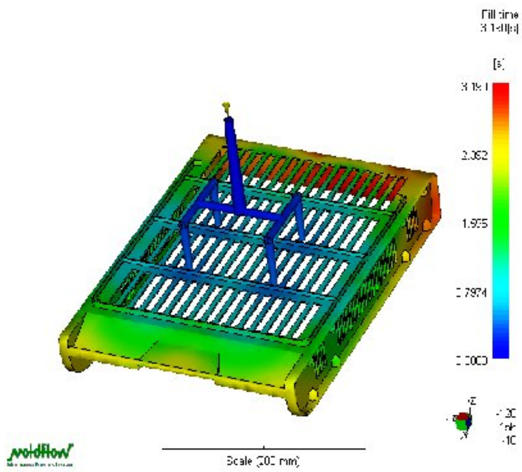
GENERAL	
Material trade name	Toyolac 100
Max machine injection Pressure (MPa)	180.0000
Melt temperature (C)	216.000
Mold temperature (C)	85.0000
Part volume to be filled (cm^3)	251.0330
Cooling time (second)	20

Source: Toray plastic (2009)

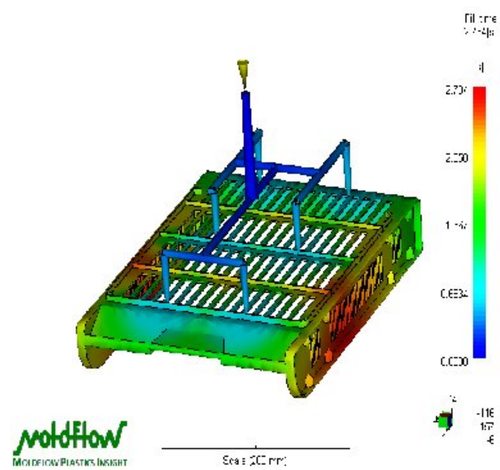
Table 4.1 shows maximum injection pressure is 180 MPa which is injection pressure will not exceed this pressure. The melt and the mold temperature for this material are 216 C and 85 C. The part volume to be filled is the actual part volume exclude the runner system is 251.033 cm^3.

4.2.1 Analysis using different number of gates

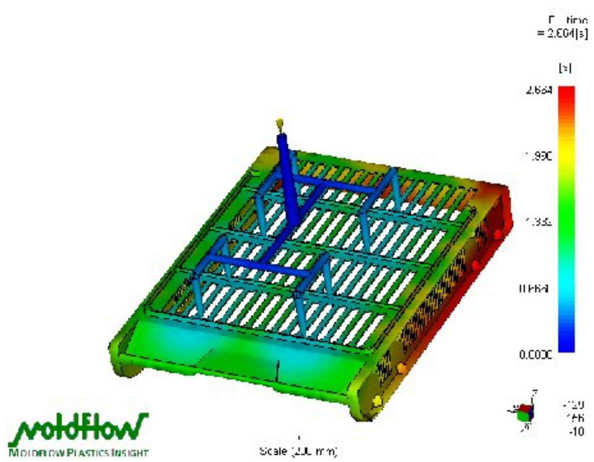
i. Fill time



a) Four gates



b) Six gates

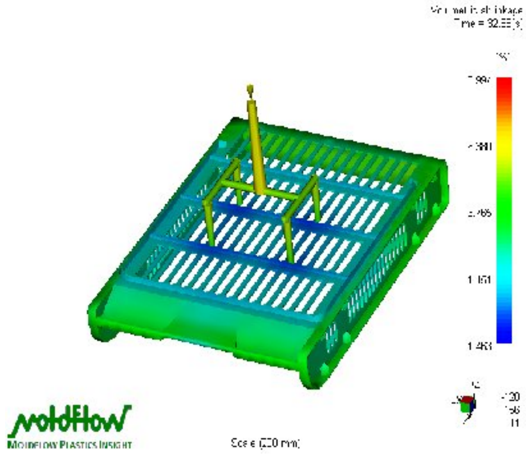


c) Eight of gates

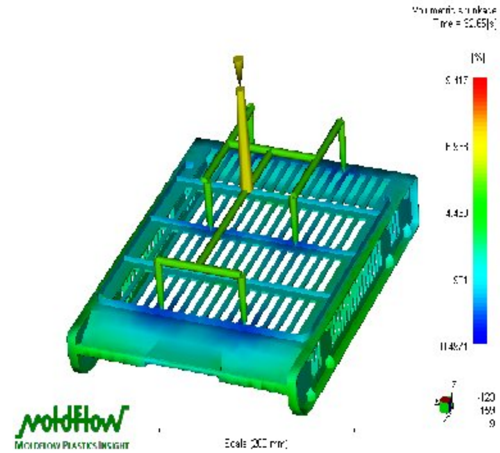
Figure 4.2: Fill time for different number of gates

Figure shows the fill time gate location of (a) four gates (b) six gates (c) eight gates. At number gate of two, the fill time is 3.190s. At six gates, the fill time is 2.734s. For eight gates, the product fills with the melt plastic in 2.664s.

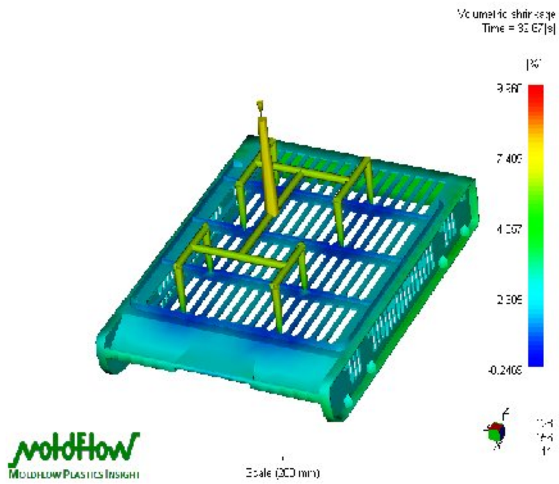
ii. Volumetric shrinkage



a) Four gates



b) Six gates

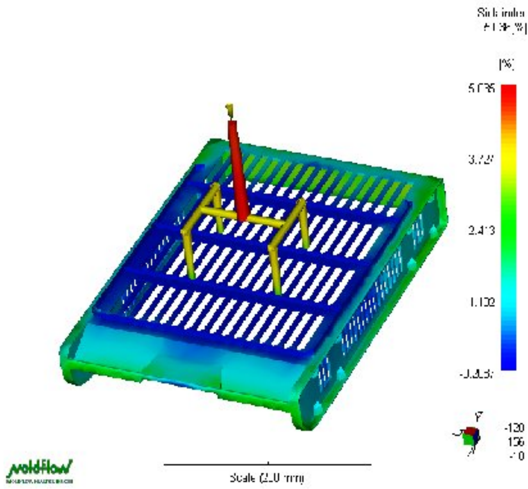


c) Eight gates

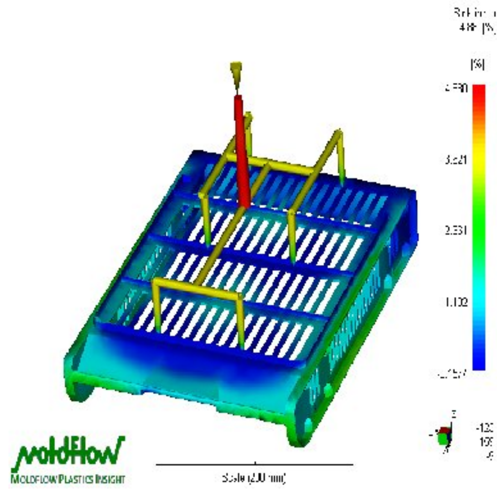
Figure 4.3: Volumetric shrinkage for number of gates

Figure shows volumetric shrinkage in the product that has been analyzed. The volumetric shrinkage for four gates is 8.994%. While for six gates, the volumetric shrinkage is 9.411%. For the product with eight gates, it takes 9.960% to shrink. The area that marks with blue color is the lowest area of shrinkage. While the biggest area shrinkage represented by red color.

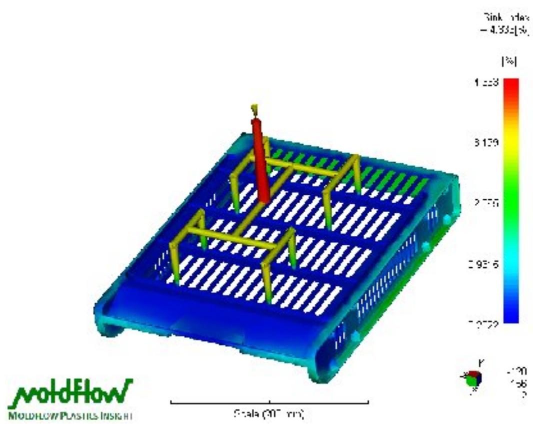
iii. Sink index



a) Four gates



b) Six gates



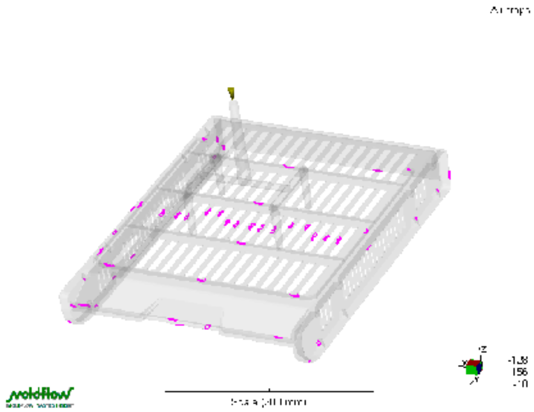
c) Eight gates

Figure 4.4: Sink index for number of gates

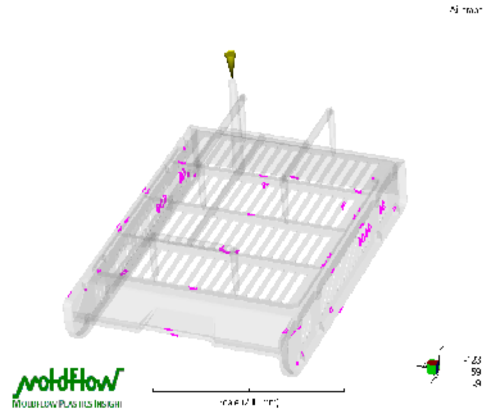
Figure shows sink index in the product that has been analyzed. The sink index for four gates is 5.035%. While for six gates, the sink index is 4.578%. For the product with eight gates, it takes 4.333%. The area that marks with blue color is the lowest area of sink index. While the biggest area sink represented by red color.

iv. Air traps

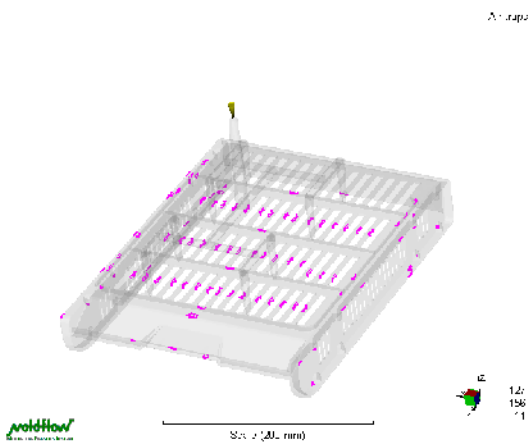
a)



Four gates



b) Six gates



c) Eight gates

Figure 4.5: Air traps number of gates

Figure shows the air traps defect in the product that has been analyzed. The air traps for eight gates has the highest defects. While for six and four gates, it takes lowest air traps to occur defect on the product. The air traps represent by the pink color circle.

Table 4.2: Result analysis for different number of gates

Number of Gates	**Four gates**	**Six gates**	**Eight gates**
Fill time (s)	3.190	2.734	2.644
Max. injection pressure during filling (MPa)	125.589	94.324	93.630
Max. clamp force during filling (tonne)	441.099	268.341	244.189
Total part weight at the end of filling (g)	301.020	321.027	326.443
Runner system volume (cm^3)	33.916	57.408	60.325

The filling time is the time needed for the molten plastics to finish the cavity. As shown in Table 4.2, the filling time for four gates and six gates are 3.190 sec and 2.734 sec while the eight gates is 2.644 sec. The maximum injection pressure during filling time for four gates is 125.589 MPa while six gates and eight gates are 94.324 MPa and 93.630 MPa. This show how much pressure is needed to push the molten cavity plastic into cavity. Maximum clamping force during filling for four gates, six gates and eight gates are 441.099 tonne, 268.341 tonne, and 244.189 tonne each. Total part weight at the end of filling shows the part weight after the molten plastics finish filling the cavity. From the result above the four gates and six gates are 301.020 grams and 321.027 grams while for eight gates is 326.443 grams. Runner system volume shows the volume of sprue, gates and runners used. For four gates is 33.916 cm^3 and for six gates and eight gates are 57.408 cm^3 and 60.325 cm^3each.

Table 4.3: Defect for different number of gates

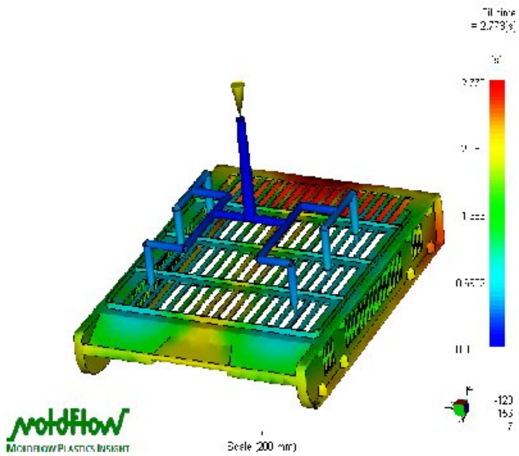
Number of Gates	**Four gates**	**Six gates**	**Eight gates**
Volumetric shrinkage (%)	8.994	9.411	9.960
Sink Index (%)	5.035	4.578	4.333
Air traps (level)	low	low	medium

Table 4.3 shows volumetric shrinkage in the product that has been analyzed. The volumetric shrinkage for four gates is 9.480%. While for six gates, the volumetric shrinkage is 9.411%. For the product with eight gates, it takes 9.260% to shrink. The sink index for four gates is 5.035%. While for six gates, the sink index is 4.578%. For the product with eight gates, it takes 4.333%. For eight gates has the highest air traps while for six and four gates, it takes lowest air traps to occur defect on the product. Assess the probability of air traps actually appearing at these locations.

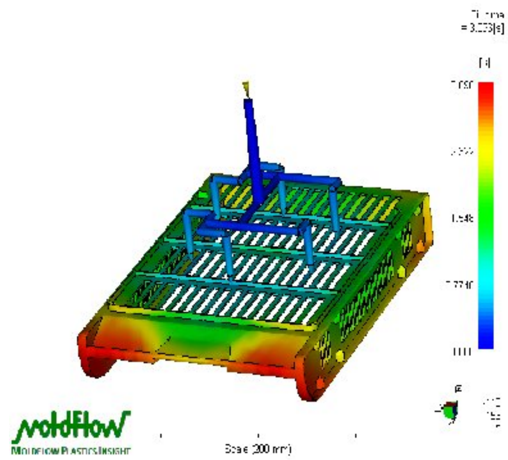
The Volumetric shrinkage result can be used to get shrinkage percent on the product. High shrinkage values could indicate voids inside the part. Volumetric shrinkage should be uniform across the whole part to reduce the defects, and it should be less than the recommended maximum value for the material. Volumetric shrinkage can be controlled by the use of packing profiles. The sink index indicates where sink mark will happen. Sink mark usually occurs at the part which is thicker compare to other parts such as at the ribs of the product. Air traps is a bubble inside the part where the melt stops at a convergence of at least 2 flow fronts. An air trap will cause a burn mark if the air is under enough pressure, causing the air to ignite and burn the plastic.

4.2.2 Analysis using different of gates location

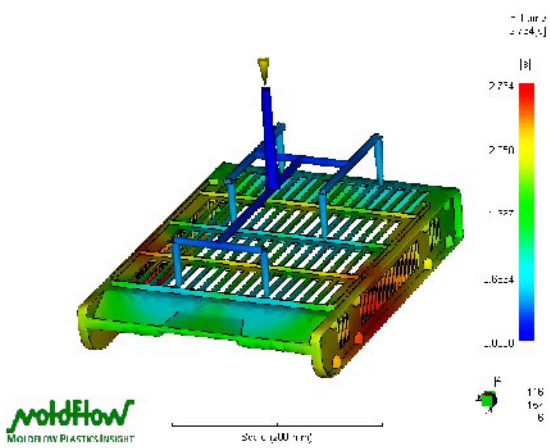
i. Fill time



a) Position A



b) Position B

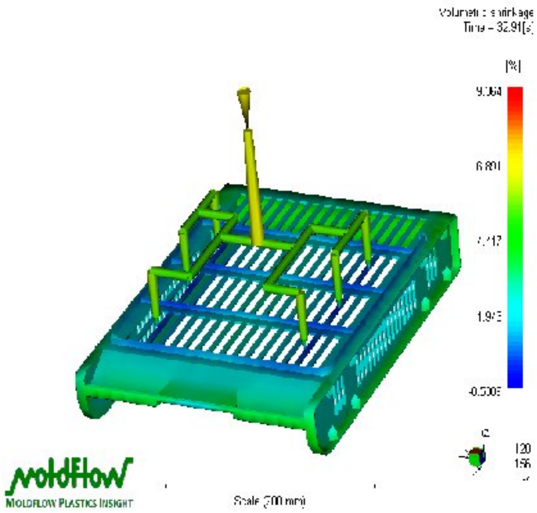


c) Position C

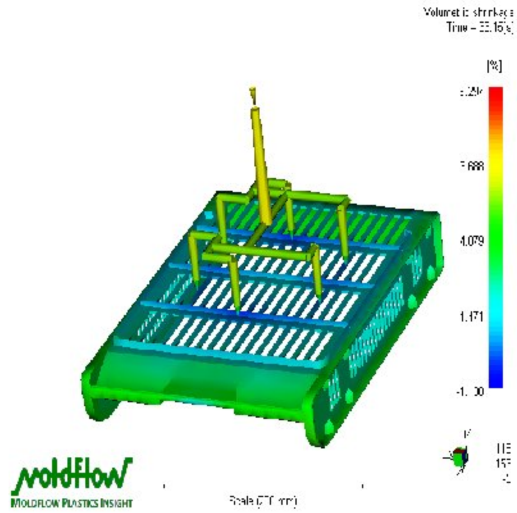
Figure 4.6: Fill time for different gate location

Figure shows the fill time gate location of (a) Position A, (b) Position B, (c) Position C. At gate position A, the fill time is 2.773s. At gate position B, the fill time is 3.096. For gate position C, the product fills with the melt plastic in 2.734s.

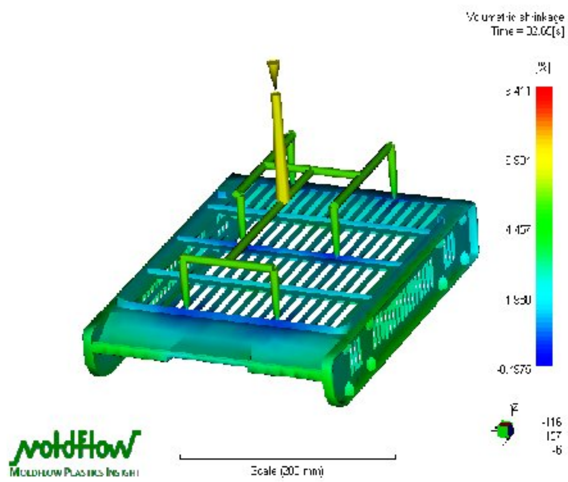
ii. Volumetric shrinkage



a) Position A



b) Position B

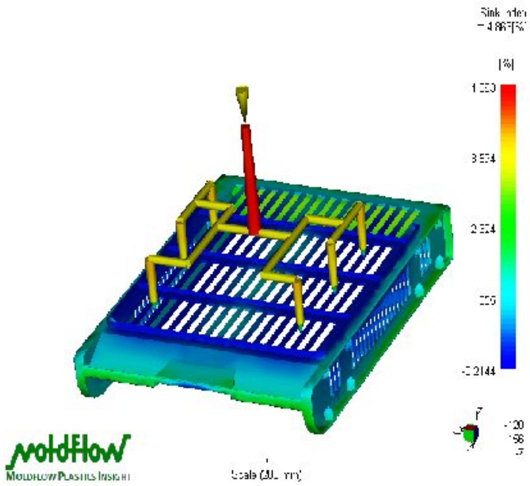


c) Position C

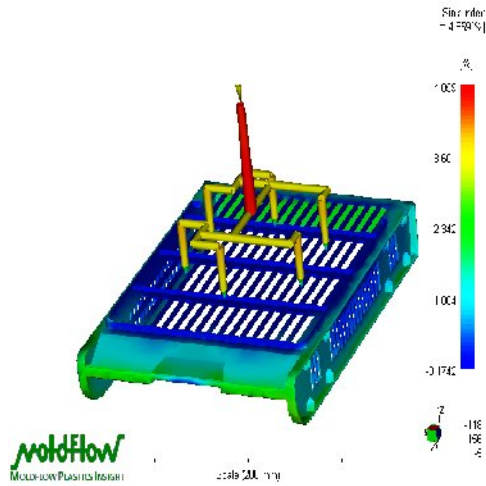
Figure 4.7: Volumetric shrinkage for gate location

Figure shows volumetric shrinkage in the product that has been analyzed. The volumetric shrinkage for gate position A is 9.364%. While for position B, the volumetric shrinkage is 9.297%. For the product with gate position C, it takes 9.411% to shrink. The area that marks with blue color is the lowest area of shrinkage. While the biggest area shrinkage represented by red color.

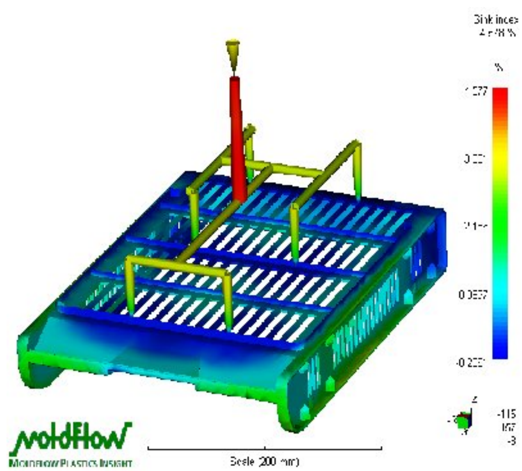
iii. Sink index



a) Position A



b) Position B

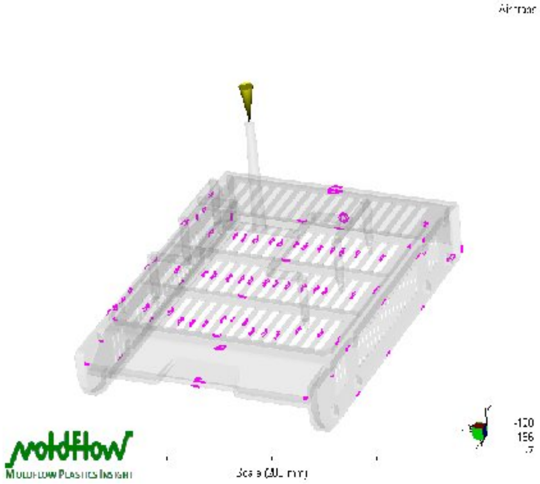


c) Position C

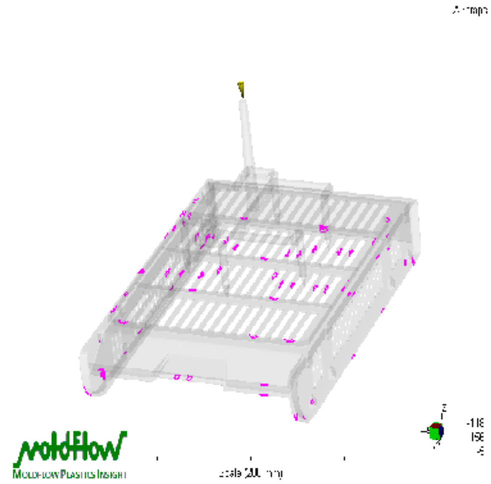
Figure 4.8: Sink index for gate location

Figure shows sink index in the product that has been analyzed. The sink index for gate position A is 4.863%. While for position B, the sink index is 4.859%. For the product with gate position C, it takes 4.578%. The area that marks with blue color is the lowest area of sink index. While the biggest area sink represented by red color.

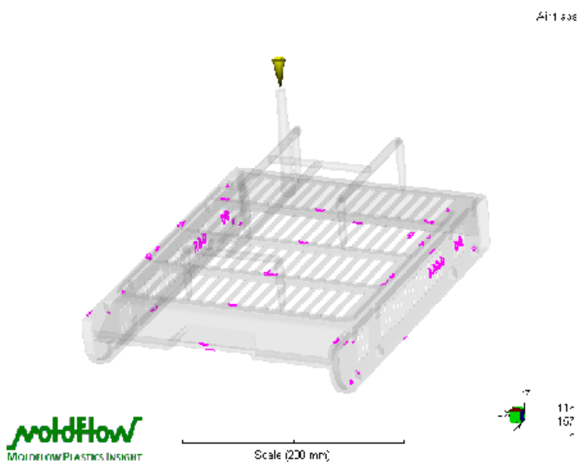
iv. Air traps



a) Position A



b) Position B



c) Position C

Figure 4.9: Air traps for gate location

Figure shows the air traps defect in the product that has been analyzed. The air traps represent by the pink color circle. The air traps location and also the number of the three gate location above are different. From figure above, gate location for position C has a lower number and location of bubble compare to the other gate location.

Table 4.4: Result analysis for different gates location

Gate Location	Position A	Position B	Position C
Fill time (s)	2.773	3.096	2.734
Max. injection pressure during filling (MPa)	95.664	98.742	94.324
Max. clamp force during filling (tonne)	275.715	290.921	268.341
Total part weight at the end of filling (g)	320.148	319.210	321.027
Runner system volume (cm^3)	56.789	54.489	57.408

The filling time is the time needed for the molten plastics to finish the cavity. As shown in Table 4.4, the filling time for gate position A and gate position B are 2.773 sec and 3.096 sec while the gate position C is 2.734 sec. The maximum injection pressure during filling time for gate position A is 95.664 MPa while gate position B and gate position C are 98.742 MPa and 94.324 MPa. This show how much pressure is needed to push the molten cavity plastic into cavity. Maximum clamping force during filling for gate position A, gate position B and gate position C are 275.715 tonne, 290.921 tonne, and 268.341 tonne each. Total part weight at the end of filling shows the part weight after the molten plastics finish filling the cavity. From the result above the total weight for gate position A and gate position B are 320.148 grams and 319.210 grams while for gate position C is 321.027 grams. Runner system volume shows the volume of sprue, gates and runners used. For gate position A is 56.789 cm^3 and for gate position B and gate position C are 54.489 cm^3 and 57.408 cm^3each.

Table 4.5: Defect for different gates location

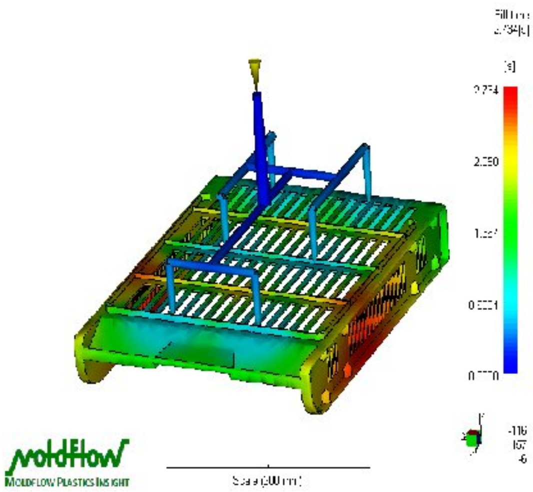
Gate Location	Position A	Position B	Position C
Volumetric shrinkage (%)	9.364	9.297	9.411
Sink Index (%)	4.863	4.859	4.578
Air traps (level)	high	medium	low

Table 4.5 shows volumetric shrinkage in the product that has been analyzed. The volumetric shrinkage for position A is 9.364%. While for gate position B, the volumetric shrinkage is 9.297%. For the product with gate position C, it takes 9.411% to shrink. The sink index for four gates is 4.863%. While for gate position B, the sink index is 4.859%. For the product with gate position C, it takes 4.578% to sink index. The air traps for position C, it takes lowest air traps from the others which are possibility to occur on the product. Assess the probability of air traps actually appearing at these locations.

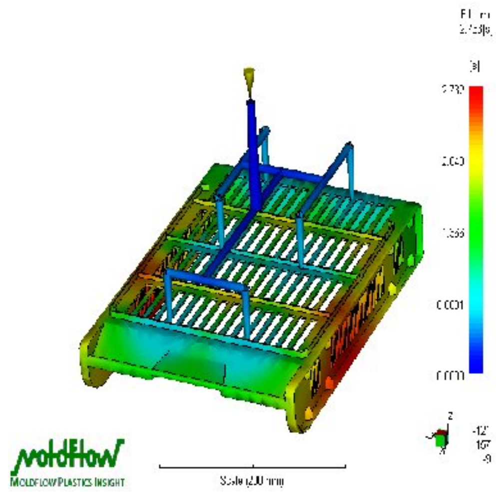
The Volumetric shrinkage result can be used to get shrinkage percent on the product. High shrinkage values could indicate voids inside the part. Volumetric shrinkage should be uniform across the whole part to reduce the defects, and it should be less than the recommended maximum value for the material. Volumetric shrinkage can be controlled by the use of packing profiles. The sink index indicates where sink mark will happen. Sink mark usually occurs at the part which is thicker compare to other parts such as at the ribs of the product. Air traps is a bubble inside the part where the melt stops at a convergence of at least 2 flow fronts. An air trap will cause a burn mark if the air is under enough pressure, causing the air to ignite and burn the plastic.

4.2.3 Analysis using different size of gates

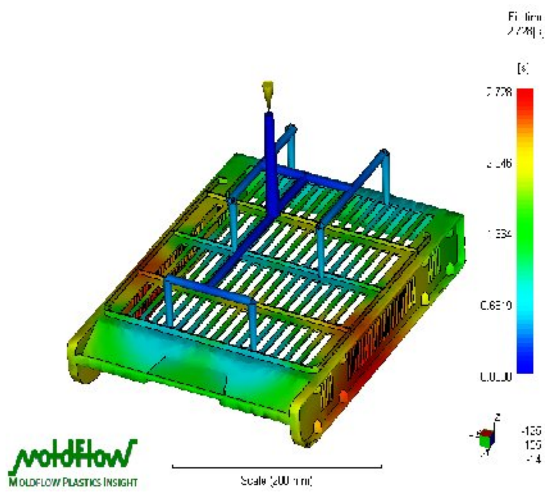
i. Fill time



a) 1.5mm



b) 2mm

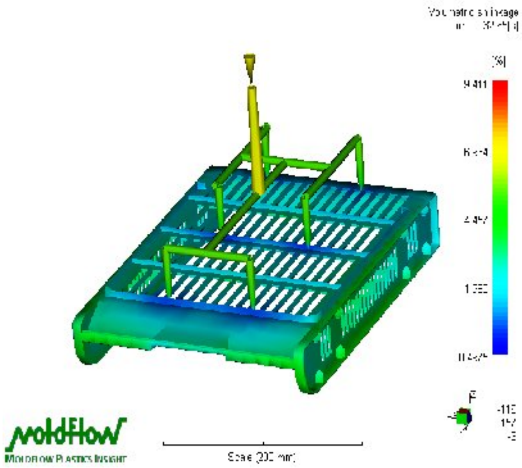


c) 2.5mm

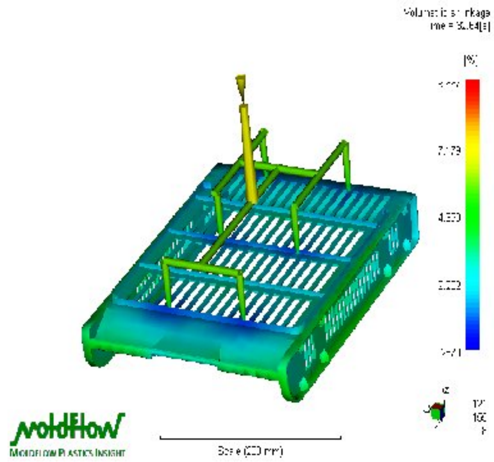
Figure 4.10: Fill time for different gates size

Figure shows the fill time for different gates size (a) 1.5mm (b) 2mm (c) 2.5mm. At gates size 1.5mm, the fill time is 2.734s. At gates size 2mm, the fill time is 2.733s. For gates size 2.5mm, the product fills with the melt plastic in 2.728s.

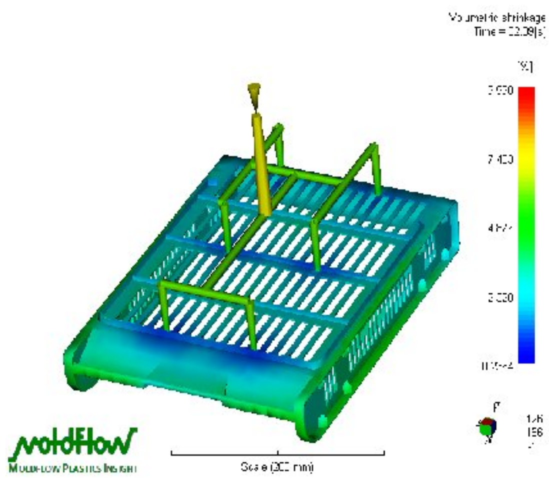
ii. Volumetric shrinkage



a) 1.5mm



b) 2mm

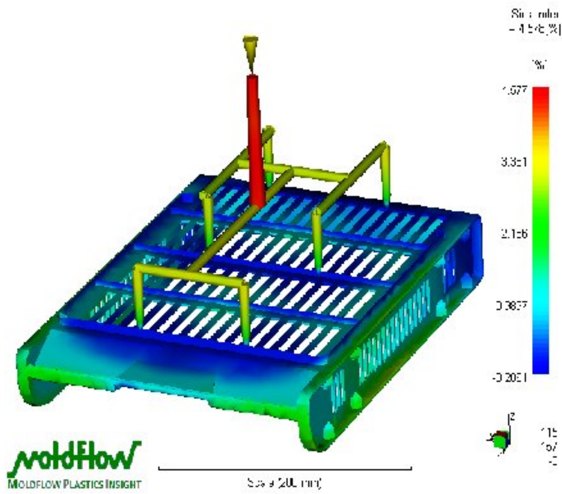


c) 2.5mm

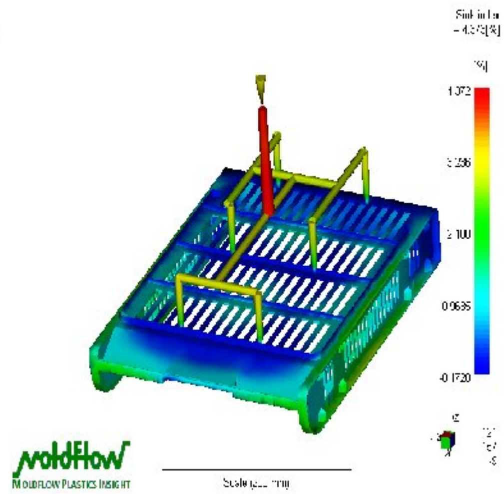
Figure 4.11: Volumetric shrinkage for gate size

Figure shows volumetric shrinkage in the product that has been analyzed. The volumetric shrinkage for gates size 1.5mm is 9.411%. While for gates size 2mm, the volumetric shrinkage is 9.667%. For the product with gates size 2.5mm, it takes 9.990% to shrink. The area that marks with blue color is the lowest area of shrinkage. While the biggest area shrinkage represented by red color.

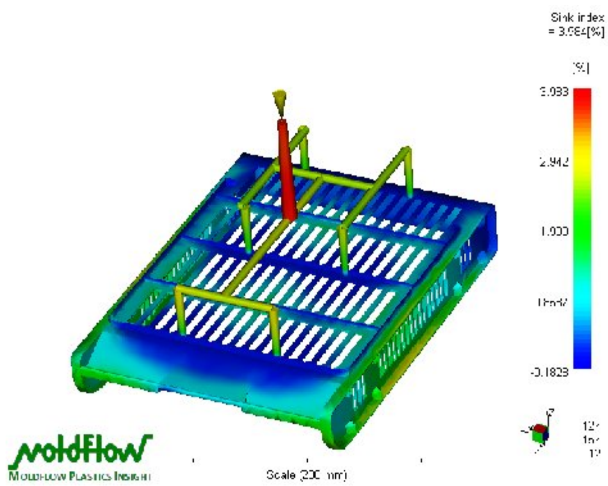
iii. Sink index



a) 1.5mm



b) 2mm

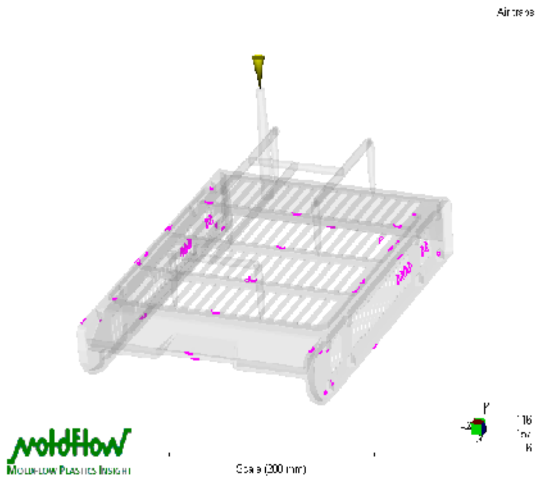


c) 2.5mm

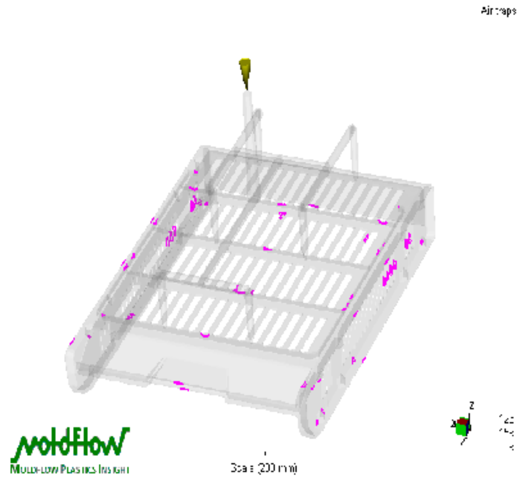
Figure 4.12: Sink index for gate size

Figure shows sink index in the product that has been analyzed. The sink index for gates size 1.5mm is 4.578%. While for gates size 2mm, the sink index is 4.373%. For the product with gates size 2.5mm, it takes 3.984%. The area that marks with blue color is the lowest area of sink index. While the biggest area sink represented by red color.

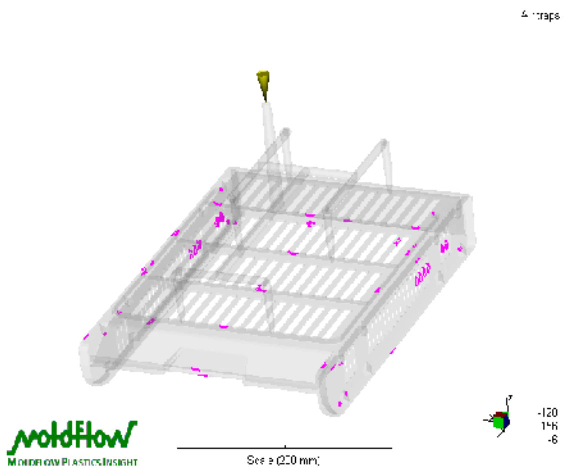
iv. Air traps



a) 1.5mm



b) 2mm



c) 2.5mm

Figure 4.13: Air traps for different gates size

Figure shows the air traps defect in the product that has been analyzed. The air traps represent by the pink color circle. The air traps location and also the number of the three gate location above are same. There is no change in the air traps defect for different gates size.

Table 4.6: Result analysis for different size of gates

Gate size	1.5mm	2mm	2.5mm
Fill time (s)	2.734	2.733	2.728
Max. injection pressure during filling (MPa)	94.324	92.905	92.462
Max. clamp force during filling (tonne)	268.341	267.973	266.325
Total part weight at the end of filling (g)	321.027	321.413	321.900
Runner system volume (cm^3)	57.408	57.823	58.360

The filling time is the time needed for the molten plastics to finish the cavity. As shown in Table 4.1, the filling time for gates size 1.5mm and gates size 2mm are 2.734 sec and 2.733 sec while the gates size 2.5mm is 2.728 sec. The maximum injection pressure during filling time gates size 1.5mm is 94.324 MPa while gate gates size 2mm and gates size 2.5mm are 92.905 MPa and 92.462 MPa. This show how much pressure is needed to push the molten cavity plastic into cavity. Maximum clamping force during filling for gates size 1.5mm, gates size 2mm and gates size 2.5mm are 268.341 tonne, 267.973 tonne, and 266.325 tonne each. Total part weight at the end of filling shows the part weight after the molten plastics finish filling the cavity. From the result above the total weight for gates size 1.5mm and gates size 2mm are 321.027 grams and 321.413 grams while for gates size 2.5mm is 321.900 grams. Runner system volume shows the volume of sprue, gates and runners used. For gates size 1.5mm is 57.408 cm^3 and for gates size 2mm and gates size 2.5mm are 57.823 cm^3 and 58.360 cm^3each.

Table 4.7: Defects for different size of gates

Gate size	**1.5mm**	**2mm**	**2.5mm**
Volumetric shrinkage (%)	9.411	9.667	9.990
Sink Index (%)	4.578	4.373	3.984
Air traps (level)	low	low	low

Table 4.7 shows volumetric shrinkage in the product that has been analyzed. The volumetric shrinkage for gates size 1.5mm is 9.411%. While for gates size 2mm, the volumetric shrinkage is 9.667%. For the product with gates size 2.5mm, it takes 9.990% to shrink. The sink index for gates size 1.5mm is 4.578%. While for gates size 2mm, the sink index is 4.373%. For the product with gates size 2.5mm, it takes 3.984% to sink index. The air traps location and also the number of the three size gates above are low. There is no change in the air traps defect for different gates size. Assess the probability of air traps actually appearing at these locations.

The Volumetric shrinkage result can be used to get shrinkage percent on the product. High shrinkage values could indicate voids inside the part. Volumetric shrinkage should be uniform across the whole part to reduce the defects, and it should be less than the recommended maximum value for the material. Volumetric shrinkage can be controlled by the use of packing profiles. The sink index indicates where sink mark will happen. Sink mark usually occurs at the part which is thicker compare to other parts such as at the ribs of the product. Air traps is a bubble inside the part where the melt stops at a convergence of at least 2 flow fronts. An air trap will cause a burn mark if the air is under enough pressure, causing the air to ignite and burn the plastic.

4.3 DISCUSSION

Discussion will be made based on the finding result for the flow analysis and defects analysis for the number of gates, location gates and the sizes of gates.

i. Number of gates analysis

Graph filling time (second) vs Number of gates

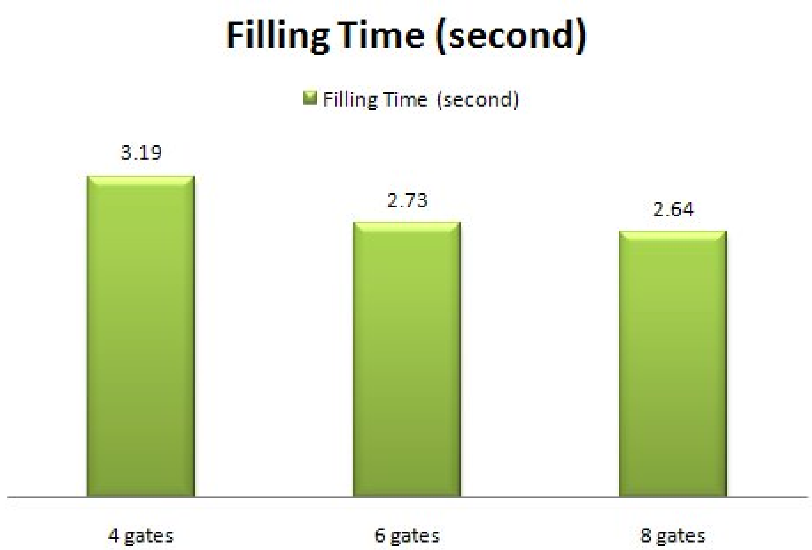


Figure 4.14

For the number of gates analysis, as shown as figure 4.14, the filling time at number gate of eight which is 2.664 second, is faster compare to the product fills with the melt plastic using six and four gates where are 3.190 and 2.734 each. This is because of the capacity of the runner system volume which is 60.325 cm^3 is bigger than six and four gates with 57.408 cm^3 and 33.916 cm^3.

Graph parameters vs Number of gates

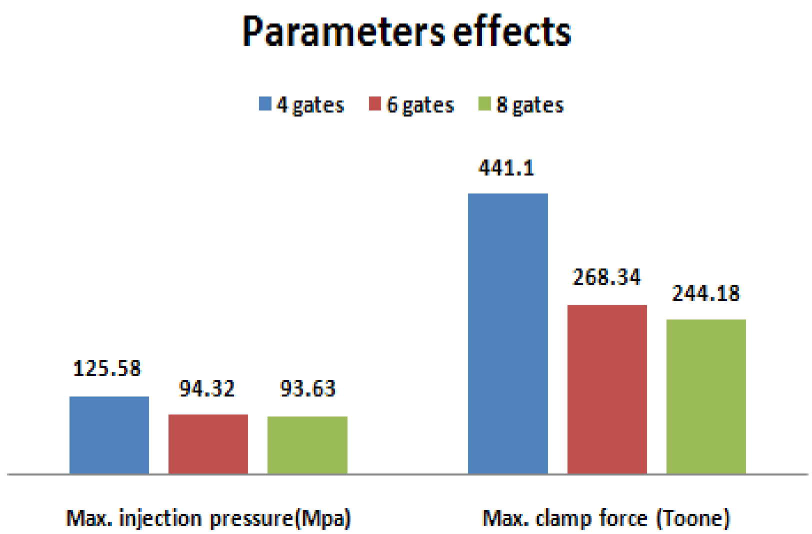


Figure 4.15

Figure 4.15, show the maximum injection pressure using in four gates during filling phase which is 125.589 MPa, is higher compare to six and eight gates with 94.324 MPa and 93.630 MPa each. These situations happen because more force is needed to push the molten plastics into the cavity by using four gates, compare using six and eight gates where allows the molten plastic fill cavity easier and faster. The maximum clamping force for four gates is higher than six and eight gates with 441.099 tonne, 268.341 tonne, and 244.189 tonne each. This occurred because four gates using more injection pressure during filling phase. The clamping force must be higher than injection pressure during filling to ensure the mold will not opened during the injection process running. The total part weight at the end of filling by using eight gates is higher than using six and four gates with 326.44 gram, 3 321.027 gram, and 301.020 gram each.

Graph defects vs Number of gates

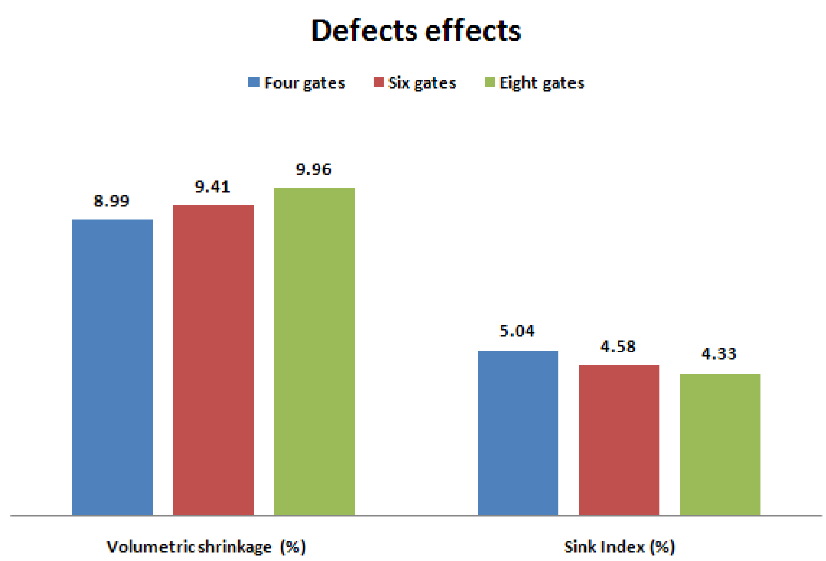


Figure 4.16

The volumetric shrinkage occurred in mold with many gates is higher compare to the mold with fewer gates. It is because from figure 4.16, mold with eight gates used lower injection pressure compare to mold using four gates which is used higher injection pressure. Sink index is higher in mold which is using fewer gates compare to mold using many gates. It is because using more gates in mold will delay the gate freeze-off time and this also will allow more material to be packed into the cavity compare to fewer gates used in the mold. Air traps occur when converging flow fronts surround and trap a bubble of air. From table, the mold using eight gates has higher possibility to air traps on the product. This is because many flow fronts from gates come to converge and makes air traps.

ii. Gates locations analysis

Graph filling time (second) vs Gates locations

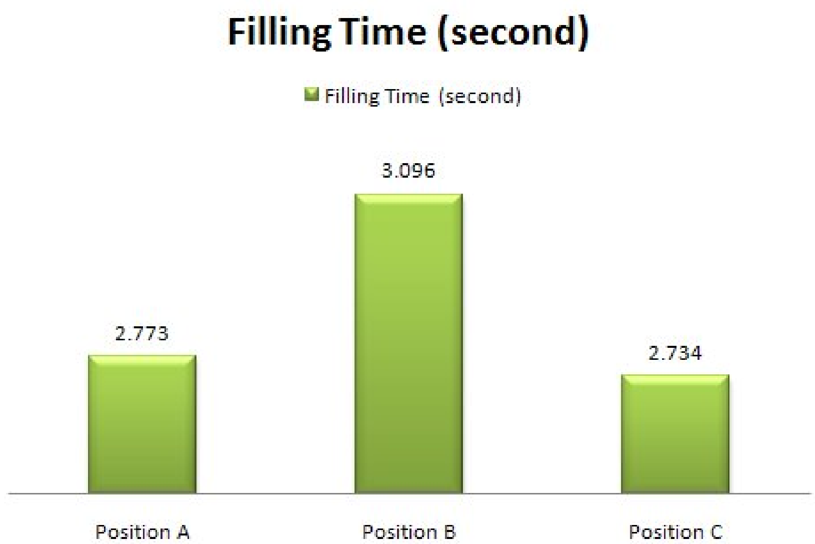


Figure 4.17

From figure 4.17, gate locations also contribute to the time that fill the cavity depend on the runner volume system. Runner system volume shows the volume of sprue, gates and runners used. For gate position A is 56.789 cm^3 and for gate position B and gate position C are 54.489 cm^3 and 57.408 cm^3each. More runner volume used faster the molten plastics fill the cavity. The filling time is the time needed for the molten plastics to finish the cavity. As shown in Table 4.1, the filling time for gate position A and gate position B are 2.773 sec and 3.096 sec while the gate position C is 2.734 sec.

Graph parameters vs Gates locations

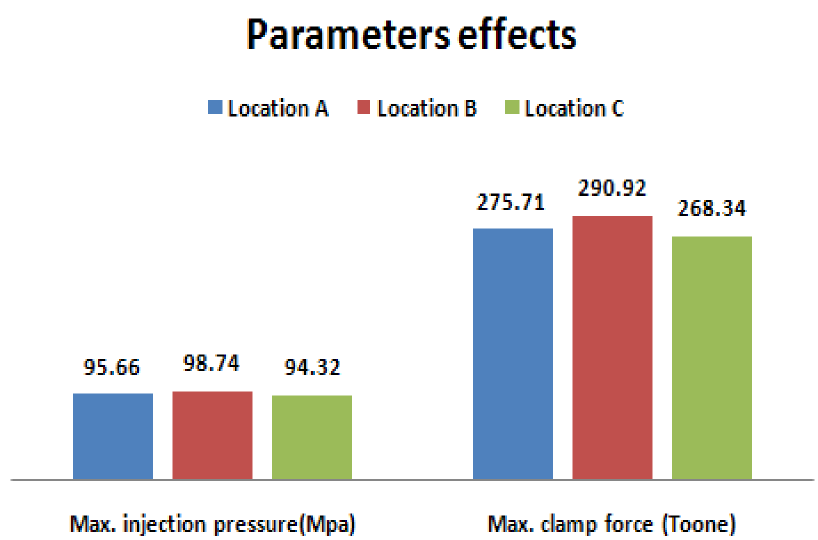


Figure 4.18

The figure 4.18 show maximum injection pressure during filling time for gate position A is 95.664 MPa while gate position B and gate position C are 98.742 MPa and 94.324 MPa. This show how much pressure is needed to push the molten cavity plastic into cavity. Maximum clamping force during filling for gate position A, gate position B and gate position C are 275.715 tonne, 290.921 tonne, and 268.341 tonne each. This occurred because gate location B using more injection pressure during filling phase. The clamping force must be higher than injection pressure during filling to ensure the mold will not opened during the injection process running.

Graph defects vs Gates locations

Figure 4.19

Based figure 4.19, gate location which is has the higher runner system volume will result lower injection pressure. As a result, the volumetric shrinkage become higher compares the mold with lower runner system volume. Location of gates contributes factor of sink index, which is occur in moldings with thicker sections. Locate the gate to or near a thicker section will allows these sections to be packed before the thinner sections freeze. As a result the sink index which is contributed to sink mark will be minimizing. Table shown the higher air traps goes to gate location A because the flow paths comes from gates is unbalance condition and also the part with uniform thickness, the physical length of flow paths may vary, and again air traps may occur.

iii. Sizes of gates analysis

Graph filling time (second) vs Sizes of gates

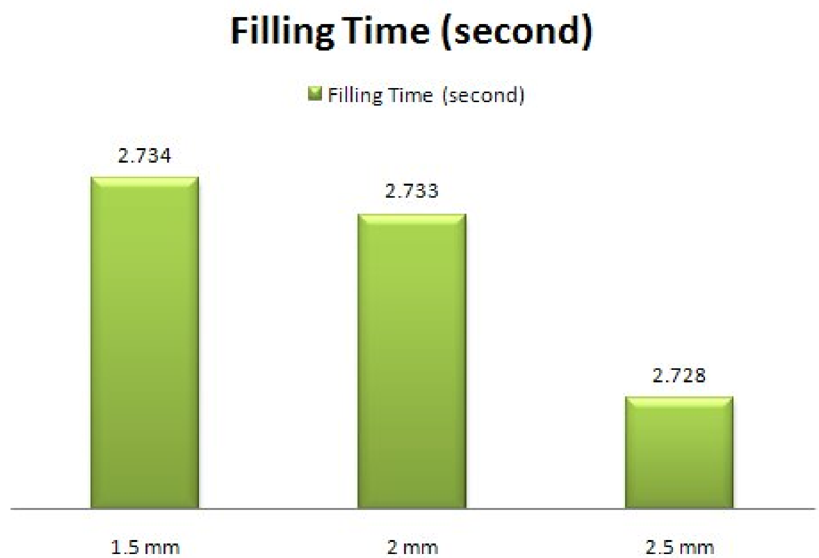


Figure 4.20

Based on figure 4.20, sizes of gates also give response to the filling time. The filling time using larger gate size is faster than the other. The filling time is the time needed for the molten plastics to finish the cavity. As shown in Table 4.1, the filling time for gates size 1mm and gates size 1.5mm are 2.734 sec and 2.733 sec while the gates size 2mm is 2.728 sec. It is because of the larger gates used more runner volume and the resulted faster in filling time. The filling time is the time needed for the molten plastics to finish the cavity.

Graph parameters vs Sizes of gates

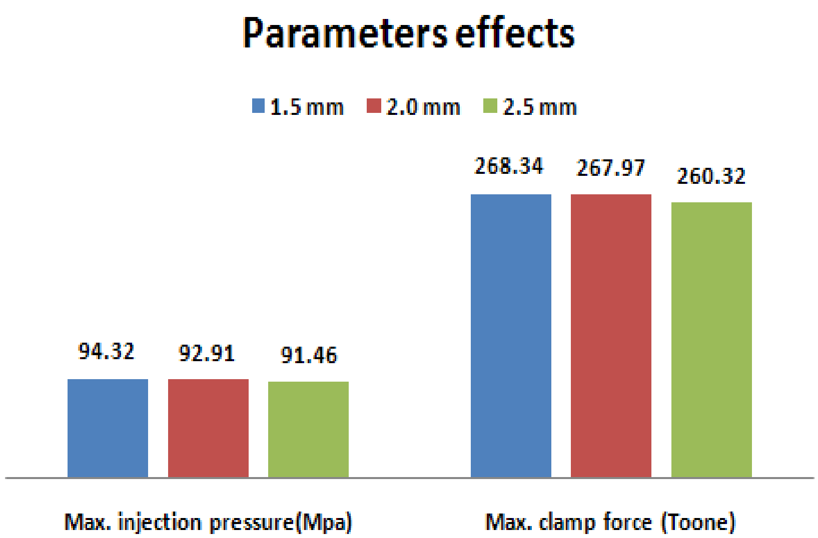


Figure 4.21

The maximum injection pressure during filling time as shown as figure 4.21 gates size 1mm is 94.324 MPa while gate gates size 1.5mm and gates size 2mm are 92.905 MPa and 92.462 MPa. This show how much pressure is needed to push the molten cavity plastic into cavity. Maximum clamping force during filling for gates size 1mm, gates size 1.5mm and gates size 2mm are 268.341 tonne, 267.973 tonne, and 266.325 tonne each. . This occurred because 1.5mm sizes of head gates using more injection pressure during filling phase. The clamping force must be higher than injection pressure during filling to ensure the mold will not opened during the injection process running.

Graph defects vs Sizes of gates

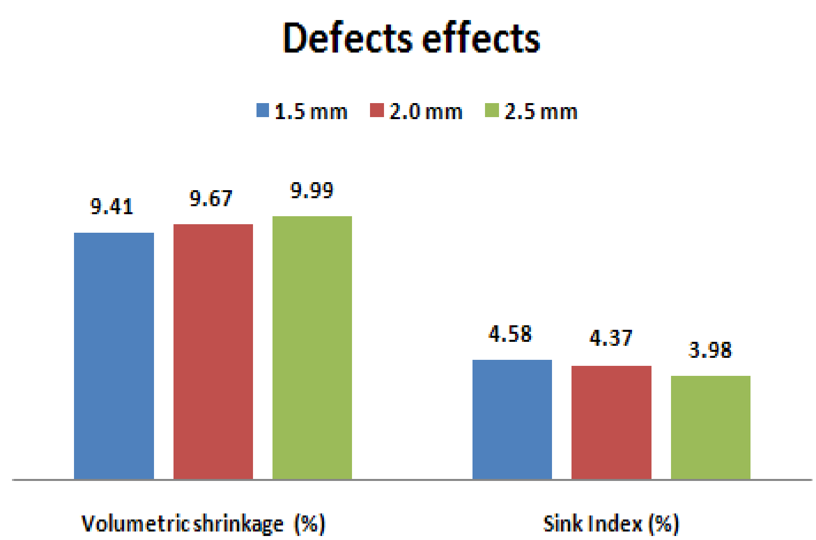


Figure 4.22

The figure 4.22 show larger size of gates will use lower injection pressure and will effected the volumetric shrinkages becomes higher. By using higher injection pressure, this will minimize the volumetric shrinkage which effect to product quality. The smaller gates in mold will result sink index higher. Increase the size of gates and runners to delay the gate freeze-off time. So, the mold using the larger gates, will allow more material to be packed into the cavity. Based on table, the air traps location and also the numbers of the three size gates above are low. There is no change in the air traps defect for different gates size. So, if air traps do exist, they should be positioned in regions that can be easily vented or ejection vent pins added so that air can be removed.

4.3.1 Parameters effect

1. Fill time

Molten plastics flow in the mold with more gates is faster compare to the mold with fewer gates, the filling time using eight gates is faster compare using six and four gates. Gate locations also contribute to the time that fill the cavity depend on the runner volume system. More runner volume used faster the molten plastics fill the cavity. Sizes of gates also give response to the filling time. The filling time using larger gate size is faster than the other. Gates should be in more amounts of gates, and larger size, in order to minimize the filling time.

2. Maximum injection pressure during filling

The maximum injection pressure in mold with many gates is lower compare to the mold with fewer gates. From table, mold with eight gates used lower injection pressure compare to mold using four gates which is used higher injection pressure. Based on table, the higher runner volume will result lower injection pressure. Also in table, larger size of gates will use lower injection pressure. This is because more force is needed to push the molten plastics into the mold with using fewer and smaller gates and also higher in runner volume. By using lower injection pressure, this will minimize the pressure needed to produce a product.

3. Maximum clamp force during filling

Maximum clamp force during filling is the amount of clamping needed for the machine to clamp the mold when injection process is running to inject the molten plastic into mold. This is because the maximum injection pressure for each gate mechanism is higher. The clamping force usually higher because to overcome the injection pressure from opening the mold during injection process.

4. Runner system volume

Based on table, results shows that the runner volume is higher in mold each characteristic of gates. It is because the runner that connected the gates to the sprue using more in length compares to the other. The mold with longer runner will result shorter in filling time. At the same time waste material on produce a product will occur but it is practical to recycle the removed of the runner system.

5. Total part weight

Total part weight of product is calculated with runner system attached. From the result obtained at Table, Table, and Table, the total part weight is constantly to the runner system. Higher runner system volume used will result higher in total part weight.

4.3.2 Defects effect

1. Volumetric shrinkage

The volumetric shrinkage occurred in mold with many gates is higher compare to the mold with fewer gates. It is because from table, mold with eight gates used lower injection pressure compare to mold using four gates which is used higher injection pressure. Based on table, gate location which is has the higher runner system volume will result lower injection pressure. As a result, the volumetric shrinkage become higher compares the mold with lower runner system volume. Also in table, larger size of gates will use lower injection pressure and will effected the volumetric shrinkages becomes higher. By using higher injection pressure, this will minimize the volumetric shrinkage which effect to product quality.

2. Sink Index

Based on table, results shows that the sink index is higher in mold which is using fewer gates compare to mold using many gates. It is because using more gates in mold will delay the gate freeze-off time and this also will allow more material to be packed into the cavity compare to fewer gates used in the mold. From table, location of gates contributes factor of sink index, which is occur in moldings with thicker sections. Locate the gate to or near a thicker section will allows these sections to be packed before the thinner sections freeze. As a result the sink index which is contributed to sink mark will be minimizing. Based on table, the smaller gates in mold will result sink index higher. Increase the size of gates and runners to delay the gate freeze-off time. So, the mold using the larger gates, will allow more material to be packed into the cavity.

3. Air Traps

Air traps occur when converging flow fronts surround and trap a bubble of air. From table, the mold using eight gates has higher possibility to air traps on the product. This is because many flow fronts from gates come to converge and makes air traps. Assess the probability of air traps actually appearing at these locations Table shown the higher air traps goes to gate location A because the flow paths comes from gates is unbalance condition and also the part with uniform thickness, the physical length of flow paths may vary, and again air traps may occur. There is no change in the air traps defect for different gates size. So, if air traps do exist, they should be positioned in regions that can be easily vented or ejection vent pins added so that air can be removed.

4.4 CONCLUSION

After all factor considered, which are the parameter effect, and the defect effect, the results shows that mold using eight gates is the faster compare to the other number of gates. But, since the volumetric shrinkage and air traps are the higher among the other number of gates, it will result low in the product quality. So, six gates will produce a good product compare to other number of gates while the filling time of six gates takes a long time compare to eight gates. The best gate location goes to gate location in position C since has the faster filling time compare to the other gate location and good result in defects effect which lower in volumetric shrinkages, sink index, and air traps. From the result in gate size analysis obtained, by using the mold with 2.5mm diameter at the head of the gates, it is faster compare to 2mm and 1.5mm head of gates. Since the product needs to be concerned on the defects occurred, the 2mm is the best choice which is faster than 1.5mm head gates because the gate with larger head will often cause a surface blemish in the final part when remove gates from the product. The volumetric shrinkages and sink mark can be minimizing by increase the packing profile and the air traps will solve by added the ejection vent pins in order to remove the air traps in the product. So, the proposed design resulted as Table 4.8 below.

Table 4.8: Proposed design of gates

Gate mechanism	**Characteristics**
Number of Gates	Six gates
Gates locations	Location C
Size of Gates	2.0mm

CHAPTER 5

CONCLUSION AND RECOMMENDATIONS

5.1 CONCLUSION

This study is focused on getting the best result to produce a book tray in terms of the gates mechanism. Using the Moldflow Plastic Insight as the main software for analysis, the results analyzed from gates mechanism analysis which are includes number of gates, location of gates and the size head of gates. Objectives of this project are investigate the gate mechanism effect on injection molding parameters and defects of book, design and proposed gate mechanism according to the results analysis.

From the observation and graph obtained, it shows that the mold using eight gates is the faster compare to the other number of gates. But, since the volumetric shrinkage and air traps are the higher among the other number of gates, it will result low in the product quality. So, six gates will produce a good product compare to other number of gates while the filling time of six gates takes a long time compare to eight gates. The best gate location goes to gate location in position C since has the faster filling time compare to the other gate location and good result in defects effect which lower in volumetric shrinkages, sink index, and air traps. From the result in gate size analysis obtained, by using the mold with 2.5mm diameter at the head of the gates, it is faster compare to 2mm and 1.5mm head of gates. Since the product needs to be concerned on the defects occurred, the 2mm is the best choice which is faster than 1.5mm head gates because the gate with larger head will often cause a surface blemish in the final part when remove gates from the product.

To get less defects, the volumetric shrinkages and sink index can be minimizing by increase the packing profile and the air traps will solve by added the ejection vent pins in order to remove the air traps in the product.

Hence, it can be conclude that base on the result obtained from the number of gates, gates locations and sizes of head gates analysis from Moldflow Plastics Insight software, the best results for producing a book tray is using he proposed design which is contains six number of gates with 2mm head sizes at location C.

5.2 RECOMMENDATIONS

In order to get a better result for the best design of gate to produce the book tray, some recommendation could be implanted in the future as below,

(i) To get the actual results, the actual fabrication of the product need to be done to know which design is better in actual situation. This is because the Moldflow software is guidance or prediction software for the users to know the overcome for the product using a variety of analysis.

(ii) To get the percentage of the Moldflow Plastics Insight (MPI) accuracy, the actual fabrication must be done.

(iii) All the mold parameters, such as mold thickness, mold size, type of gate, size of runners and the cooling system must be taken from the actual mold to get more valid data when running the analysis.

REFERENCES

Beaumont, John P. (2004). *Runner and Gating Design Handbook*. Hanser Publisher, Munich.

Beaumont, J. P., Nagel, R., and Sherman, R. (2002). *Successful Injection Molding*. Cincinnati: Hanser Publisher.

Biron,M. (2009). *Thermoplastics and Thermoplastic Composites*. Published by: Elsevier Ltd. All right reserved.

Bryce,M.D. (1996). *Plastic Injection Molding: manufacturing process fundamentals*. Published by: Society of Manufacturing Engineers Dearbone, Michigan.

Direct industry. (2002). *Mold Flow catalogue*. http://www.moldflow.com/stp/ on 30th March 2009.

Harper, Charles A. (1999). *Modern Plastic Handbook*. New York: McGraw Hill.

Kalpakjian, Serope, and Schmid Steven R. (2006). *Manufacturing Engineering and Technology*. 5th Edition. Upper Saddle River, NJ: Prentice Hall.

L.Sors and Balazs,(1989) *Mold Engineering.* Published by: Carl Hanser Verlag.Munich.

Osswald, Tim A., Turng, Lih-Sheng, Graman, Paul J. (2002). *Injection Molding Handbook*. Cincinnati: Hanser Publisher

Rees, H. and Catoen, B. (2006). *Selecting Injection Molds.* Cincinnati: Hanser Publisher.

Rtp Company . (2005). *Select Mold* http://www.rtpcompany.com/info/molding/design on 12th March 2009

Toray plastic. (2009). *Plastic Raw Material*. http://www.toray.co.jp/material_ABS on 10th March 2009

Umg Abs Ltd. (2005). *ABS Selection* http://www.umgabs.co.jp/en/solutoin/t_37.htm on 2th February 2009

APPENDIX A – 1: Gantt Chart For FYP 1

Effect of Injection Molding Gate Mechanism on Parameters Machining and Defects of Book Tray

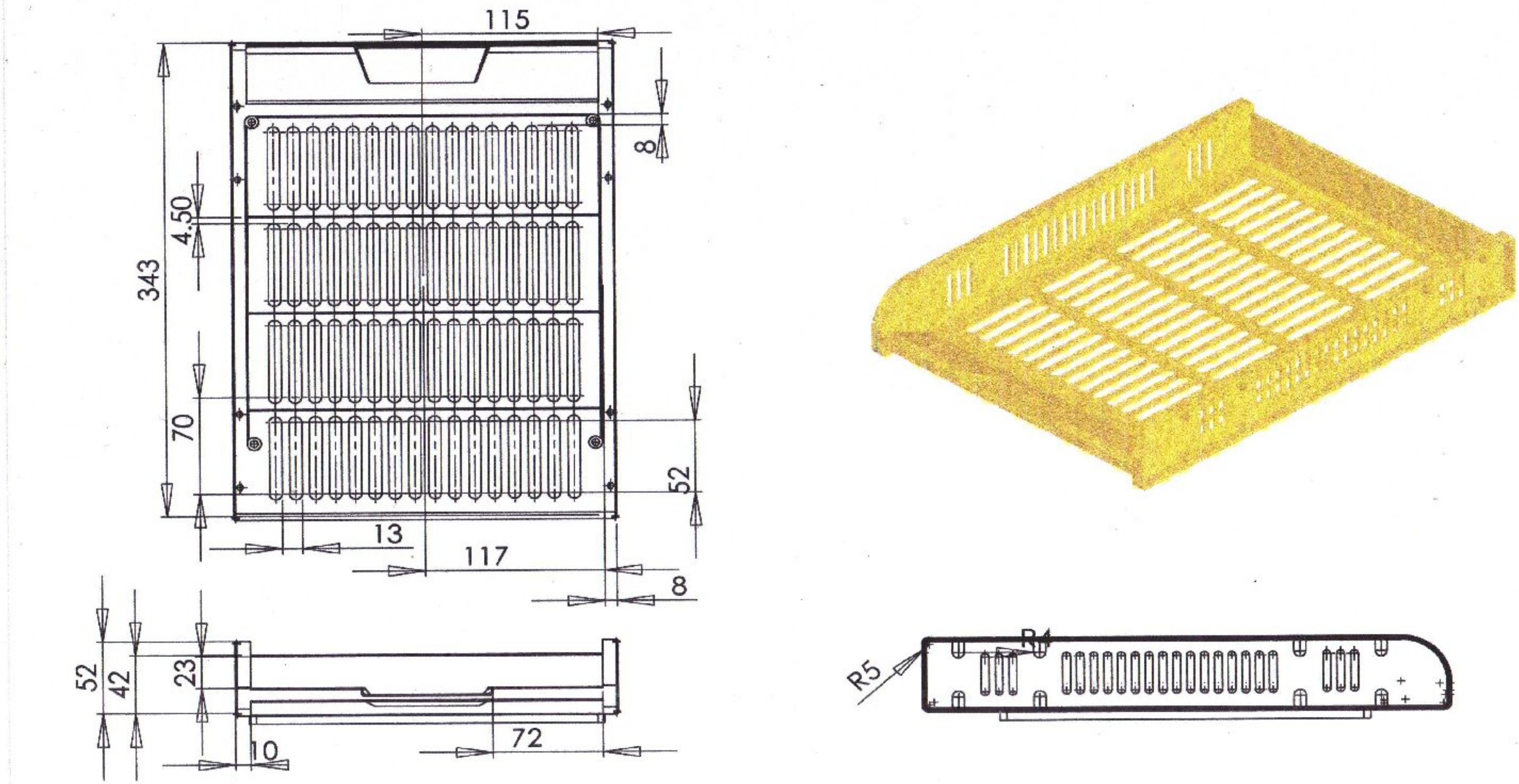
Project Activities		Week														
	W1	W2	W3	W4	W5	W6	W7	W8	W9	W10	W11	W12	W13	W14	W15	W16
1. Verify Topic																
2. Objectives and scopes																
3.Literature review																
4.Methodology																
5. Submit the full proposal																
6.Learn about Moldflow software																
6.PSM1 presentation																
7.Submit the full report																

APPENDIX A – 2: Gantt Chart For FYP 2

Effect of Injection Molding Gate Mechanism on Parameters Machining and Defects of Book Tray

Project Activities	Week															
	W1	W2	W3	W4	W5	W6	W7	W8	W9	W10	W11	W12	W13	W14	W15	W16
Writing report	■	■	■	■	■	■	■	■	■	■	■	■	■	■	■	■
Material and Mold Selection	■	■	■													
Gate type selection			■	■												
Design the product				■	■	■										
Get the result from moldflow							■	■	■	■	■					
Data collection									■	■	■					
Analysis the results											■	■	■	■		
Presentation preparation													■	■	■	
Final presentation															■	
Submit report																■

APPENDIX B – 1: Technical Drawing



UNIVERSITI MALAYSIA PAHANG	DRAWN	ABDUL AZWAN BIN ABDOL ZAHAR	TOLERANCES	+/- 0.01	
	MATRIK. NO	ME 07071	NO. DRWG	NO. 1	
	CHECK	M. REZA ZALANI BIN M.SUFFIAN	SCALE	1 : 3	A 4

APPENDIX C : Moldflow analysis result

Flow Analysis

Version: mpi500 (Build 04453)

Analysis commenced at Wed Oct 07 02:12:46 2009

Analysis running on host: sufi-4bca7194e6
Operating System: Windows XP Service Pack 2
Processor type: AuthenticAMD x86 Family 15 Model 107 Stepping 1 ~2099 MHz
Number of Processors: 2
Total Physical Memory: 1982 MBytes

Filling Analysis

Packing Analysis

Residual Stress Analysis

Process settings :

Machine parameters :

Maximum machine clamp force = 7.0002E+03 tonne
Maximum injection pressure = 1.8000E+02 MPa
Maximum machine injection rate = 5.0000E+03 cm^3/s
Machine hydraulic response time = 1.0000E-02 s

Process parameters :

Fill time = 2.3795 s
Injection time has been determined by automatic calculation.
Stroke volume determination = Automatic
Cooling time = 20.0000 s

Velocity/pressure switch-over by		= Automatic
Packing/holding time		= 10.0000 s

Ram speed profile (rel):

% shot volume	% ram speed
100.0000	100.0000
0.0000	100.0000

Pack/hold pressure profile (rel):

duration	% filling pressure
0.0000 s	80.0000
10.0000 s	80.0000
20.0000 s	0.0000

Ambient temperature	= 25.0000 C
Melt temperature	= 230.0000 C
Ideal cavity-side mold temperature	= 50.0000 C
Ideal core-side mold temperature	= 50.0000 C

NOTE: Mold wall temperature data from cooling analysis not available

---

Model details :

Mesh Type		= Fusion
Match ratio		= 83.5 %
Reciprocal match ratio		= 73.2 %
Total number of nodes		= 4967
Total number of injection location nodes		= 1
The injection location node label		= 4611
Total number of elements		= 9571
Number of part elements		= 9437
Number of sprue/runner/gate elements		= 134
Number of channel elements		= 0
Number of connector elements		= 0
Parting plane normal	(dx)	= 0.0000
	(dy)	= 0.0000
	(dz)	= 1.0000
Average aspect ratio of triangle elements		= 4.3537
Maximum aspect ratio of triangle elements		= 97.4620
Element number with maximum aspect ratio		= 8590
Minimum aspect ratio of triangle elements		= 1.1547
Element number with minimum aspect ratio		= 2302
Total volume		= 308.4410 cm^3
Volume filled initially		= 0.0000 cm^3
Volume to be filled		= 308.4410 cm^3
Part volume to be filled		= 251.0330 cm^3
Sprue/runner/gate volume to be filled		= 57.4075 cm^3
Total projected area		= 696.5510 cm^2

Filling Analysis

Packing Analysis

Residual Stress Analysis
analysis is beginning ....

Filling phase: Status: V = Velocity control
P = Pressure control
V/P= Velocity/pressure switch-over

Time (s)	Volume (%)	Pressure (MPa)	Clamp force (tonne)	Flow rate (cm^3/s)	Status
0.13	3.92	12.02	0.04	108.90	V
0.24	7.59	25.29	1.59	115.45	V
0.36	12.18	30.50	3.43	129.03	V
0.48	16.78	34.31	5.95	127.14	V
0.60	21.07	45.00	12.76	129.82	V
0.72	26.24	47.57	16.64	133.36	V
0.84	31.09	48.97	20.11	133.75	V
0.95	35.65	50.29	23.52	134.36	V
1.08	41.05	51.93	28.88	133.72	V
1.20	45.71	54.06	35.45	133.31	V
1.32	50.37	56.64	42.34	133.98	V
1.43	54.74	59.17	51.00	133.50	V
1.55	59.28	63.18	66.97	133.07	V
1.67	63.99	66.87	80.49	133.85	V
1.79	68.93	70.48	96.79	134.25	V
1.91	73.72	73.72	113.18	134.45	V
2.03	78.48	77.01	137.18	134.31	V
2.15	83.30	81.25	165.39	134.84	V
2.27	87.80	85.60	199.24	135.09	V
2.38	92.19	90.10	233.24	135.41	V
2.49	96.32	94.32	268.34	133.27	V/P
2.50	96.73	75.46	251.60	45.42	P
2.50	96.79	75.46	248.63	54.35	P
2.62	98.77	75.46	243.12	44.60	P
2.73	99.82	75.46	254.23	30.54	P
2.74	100.00	75.46	256.30	29.38	Filled

Execution time in Filling Phase = 186.75 s

Packing phase:

Time (s)	Packing (%)	Pressure (MPa)	Clamp force (tonne)	Status
17.90	50.95	0.00	5.75	P
19.40	55.99	0.00	4.44	P
20.90	61.04	0.00	3.71	P
22.40	66.08	0.00	3.29	P
23.90	71.12	0.00	3.00	P
25.40	76.16	0.00	2.78	P
26.90	81.21	0.00	2.61	P
28.40	86.25	0.00	2.50	P

| 29.90 | 91.29 | 0.00 | 2.41 | P |

Filling phase results summary :

Maximum injection pressure (at 2.486 s) = 94.3240 MPa

End of filling phase results summary :

Time at the end of filling = 2.7386 s
Total weight = 321.0270 g
Maximum Clamp force - during filling = 268.3407 tonne
Recommended ram speed profile (rel):

% stroke	% speed
0.0000	13.8253
10.0000	31.7725
17.8162	31.7725
30.0000	83.0871
40.0000	90.3197
50.0000	89.0561
60.0000	84.6610
70.0000	100.0000
80.0000	71.7146
90.0000	42.6901
100.0000	13.8614

Melt front is entirely in the cavity at % fill = 17.8162 %

Filling phase results summary for the part :

Bulk temperature - maximum (at 1.201 s) = 239.8420 C
Bulk temperature - 95th percentile (at 0.596 s) = 235.5770 C
Bulk temperature - 5th percentile (at 2.729 s) = 203.2650 C
Bulk temperature - minimum (at 2.729 s) = 80.8810 C

Wall shear stress - maximum (at 2.266 s) = 1.4034 MPa
Wall shear stress - 95th percentile (at 2.152 s) = 0.4557 MPa

Shear rate - maximum (at 2.486 s) = 5456.3701 1/s
Shear rate - 95th percentile (at 0.596 s) = 1608.6400 1/s

End of filling phase results summary for the part :

Total part weight = 263.6920 g

Bulk temperature - maximum = 234.6780 C
Bulk temperature - 95th percentile = 232.4180 C
Bulk temperature - 5th percentile = 203.2750 C
Bulk temperature - minimum = 80.8810 C
Bulk temperature - average = 222.9670 C
Bulk temperature - RMS deviation = 11.4258 C

Wall shear stress - maximum = 0.7325 MPa
Wall shear stress - 95th percentile = 0.3089 MPa
Wall shear stress - average = 0.1816 MPa
Wall shear stress - RMS deviation = 0.0823 MPa

Frozen layer fraction - maximum = 1.0000
Frozen layer fraction - 95th percentile = 0.1792

Frozen layer fraction - 5th percentile	=	0.0635
Frozen layer fraction - minimum	=	0.0116
Frozen layer fraction - average	=	0.1152
Frozen layer fraction - RMS deviation	=	0.0387
Shear rate - maximum	=	1134.7200 1/s
Shear rate - 95th percentile	=	214.3300 1/s
Shear rate - average	=	51.6178 1/s
Shear rate - RMS deviation	=	89.1149 1/s

Filling phase results summary for the runner system :

Bulk temperature - maximum	(at 2.498 s)	=	238.6130 C
Bulk temperature - 95th percentile	(at 1.084 s)	=	237.3210 C
Bulk temperature - 5th percentile	(at 2.739 s)	=	230.9140 C
Bulk temperature - minimum	(at 2.739 s)	=	230.1660 C
Wall shear stress - maximum	(at 0.952 s)	=	0.7265 MPa
Wall shear stress - 95th percentile	(at 0.240 s)	=	0.3046 MPa
Shear rate - maximum	(at 1.201 s)	=	1.6904E+04 1/s
Shear rate - 95th percentile	(at 1.084 s)	=	1873.9800 1/s

End of filling phase results summary for the runner system :

Total sprue/runner/gate weight	=	57.3347 g
Bulk temperature - maximum	=	235.7040 C
Bulk temperature - 95th percentile	=	234.5920 C
Bulk temperature - 5th percentile	=	230.9140 C
Bulk temperature - minimum	=	230.1660 C
Bulk temperature - average	=	232.7030 C
Bulk temperature - RMS deviation	=	1.0075 C
Wall shear stress - maximum	=	0.4286 MPa
Wall shear stress - 95th percentile	=	0.1862 MPa
Wall shear stress - average	=	0.1341 MPa
Wall shear stress - RMS deviation	=	0.0296 MPa
Frozen layer fraction - maximum	=	0.0938
Frozen layer fraction - 95th percentile	=	0.0849
Frozen layer fraction - 5th percentile	=	0.0326
Frozen layer fraction - minimum	=	0.0320
Frozen layer fraction - average	=	0.0511
Frozen layer fraction - RMS deviation	=	0.0196
Shear rate - maximum	=	3350.3101 1/s
Shear rate - 95th percentile	=	295.7330 1/s
Shear rate - average	=	158.5420 1/s
Shear rate - RMS deviation	=	137.6170 1/s

Packing phase results summary :

Peak pressure - minimum	(at 3.148 s)	=	28.3023 MPa
Clamp force - maximum	(at 3.148 s)	=	348.6695 tonne
Total weight - maximum	(at 12.148 s)	=	330.8730 g

End of packing phase results summary :

Time at the end of packing	=	32.6454 s
Total weight	=	329.0780 g

Packing phase results summary for the part :

Bulk temperature - maximum	(at 2.739 s)	=	234.6790 C
Bulk temperature - 95th percentile	(at 2.739 s)	=	232.4190 C
Bulk temperature - 5th percentile	(at 32.645 s)	=	50.6930 C
Bulk temperature - minimum	(at 32.645 s)	=	50.0000 C
Wall shear stress - maximum	(at 13.395 s)	=	3.7704 MPa
Wall shear stress - 95th percentile	(at 7.648 s)	=	0.7701 MPa
Volumetric shrinkage - maximum	(at 2.739 s)	=	9.4111 %
Volumetric shrinkage - 95th %ile	(at 2.739 s)	=	7.8012 %
Volumetric shrinkage - 5th %ile	(at 31.395 s)	=	0.9878 %
Volumetric shrinkage - minimum	(at 4.648 s)	=	-0.4975 %
Total part weight - maximum	(at 32.645 s)	=	271.9290 g

End of packing phase results summary for the part :

Total part weight	=	271.9290 g
Bulk temperature - maximum	=	74.7590 C
Bulk temperature - 95th percentile	=	65.9740 C
Bulk temperature - 5th percentile	=	50.6930 C
Bulk temperature - minimum	=	50.0000 C
Bulk temperature - average	=	55.0270 C
Bulk temperature - RMS deviation	=	5.2622 C
Frozen layer fraction - maximum	=	1.0000
Frozen layer fraction - 95th percentile	=	1.0000
Frozen layer fraction - 5th percentile	=	1.0000
Frozen layer fraction - minimum	=	1.0000
Frozen layer fraction - average	=	1.0000
Frozen layer fraction - RMS deviation	=	0.0000
Volumetric shrinkage - maximum	=	5.3500 %
Volumetric shrinkage - 95th percentile	=	4.2945 %
Volumetric shrinkage - 5th percentile	=	0.9878 %
Volumetric shrinkage - minimum	=	0.0768 %
Volumetric shrinkage - average	=	2.5903 %
Volumetric shrinkage - RMS deviation	=	0.9547 %
Sink index - maximum	=	3.4313 %
Sink index - 95th percentile	=	2.3811 %
Sink index - minimum	=	0.8595 %
Sink index - RMS deviation	=	0.7550 %

Packing phase results summary for the runner system :

```
   Bulk temperature - maximum         (at   2.739 s)          =   235.6920 C
   Bulk temperature - 95th percentile  (at   2.739 s)          =   234.5880 C
   Bulk temperature - 5th percentile   (at  32.645 s)          =   144.4870 C
   Bulk temperature - minimum         (at  32.645 s)          =    52.5530 C

   Wall shear stress - maximum        (at  13.395 s)          =     4.2139 MPa
   Wall shear stress - 95th percentile (at  12.499 s)          =     0.6302 MPa

   Volumetric shrinkage - maximum      (at  12.499 s)       =     7.2892 %
   Volumetric shrinkage - 95th %ile    (at  13.395 s)       =     6.8612 %
   Volumetric shrinkage - 5th %ile     (at  12.148 s)       =     2.1319 %
   Volumetric shrinkage - minimum      (at  12.499 s)       =    -0.1335 %
   Sprue/runner/gate weight - max.     (at  12.148 s)       =    59.0160 g
```

End of packing phase results summary for the runner system :

```
   Volumetric shrinkage - maximum                          =     6.7986 %
   Volumetric shrinkage - 95th percentile                  =     6.7685 %
   Volumetric shrinkage - 5th percentile                   =     4.9963 %
   Volumetric shrinkage - minimum                          =     0.3034 %
   Volumetric shrinkage - average                          =     5.5647 %
   Volumetric shrinkage - RMS deviation                    =     0.6139 %

   Sink index - maximum                                    =     4.8797 %
   Sink index - 95th percentile                            =     4.8453 %
   Sink index - minimum                                    =     3.6772 %
   Sink index - RMS deviation                              =     0.5849 %
```

Preparing interface data...
  Preparing PPC file for cooling analysis...
  Preparing LSP file for warpage analysis...
Finished preparing the interface data

Filling Analysis

Packing Analysis
Residual Stress Analysis
has completed successfully.

Weld line/air trap analysis completed

Preparing output data...
Finished preparing output data

SYNERGY Weld-line and air trap
has completed successfully.

Execution time
```
   Analysis commenced at      Wed Oct 07 02:12:46 2009
   Analysis completed at      Wed Oct 07 02:18:50 2009
   CPU time used              329.64 s
```