MOLD DESIGN FOR PLASTICS INJECTION MOLDING PROCEES (TWO COLORS) AND ANALYSIS OF GATES LOCATION

AHMAD ZULHELMI BIN MUHAMAD

BACHELOR OF ENGINEERING

UNIVERSITI MALAYSIA PAHANG

UNIVERSITI MALAYSIA PAHANG

FACULTY OF MECHANICAL ENGINEERING

We certify that the project entitled "Mold design for plastics injection molding process (two colors) and and analysis of gate location" is written by *Ahmad Zulhelmi bin Muhamad*. We have examined the final copy of this project and in our opinion; it is fully adequate in terms of scope and quality for the award of the degree of Bachelor of Engineering. We herewith recommend that it be accepted in partial fulfillment of the requirements for the degree of Bachelor of Mechanical Engineering with Manufacturing Engineering.

Pn Nurul Shahida Mohd Shalahim	....
Examiner	Signature

MOLD DESIGN FOR PLASTICS INJECTION MOLDING PROCEES (TWO COLORS) AND ANALYSIS OF GATES LOCATION

AHMAD ZULHELMI BIN MUHAMAD

A report submitted in partial fulfillment 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

SUPERVISOR'S DECLARATION

I hereby declare that I have checked this project and in my opinion, this project is adequate in terms of scope and quality for the award of the degree of Bachelor of Mechanical Engineering with Manufacturing Engineering.

Signature : ....

Name of Supervisor : EN. ASNUL HADI AHMAD

Position : LECTURER OF FACULTY MECHANICAL ENGINEERING

Date : ....

STUDENT’S DECLARATION

I hereby declare that this thesis entitled *“Mold design for plastic injection molding process and gate location analysis”* is the result of my own research except as cited in the references. The thesis has not been accepted for my degree and is not concurrently candidature of any other degree.

Signature : ....
Name : AHMAD ZULHELMI BIN MUHAMAD
ID Number : ME 06011
Date : ....

Dedicated to my parents

PUAN SITI AZIMAH BINTI HUSAIN

ENCIK MUHAMAD BIN JUSOH

ACKNOWLEDGEMENTS

I am Ahmad Zulhelmi Bin Muhamad, would like to acknowledge and extend our heartfelt gratitude to my supervisor En. Asnul Hadi Ahmad; Lecturer of Faculty Mechanical Engineering for his continues support, helpful advice and valuable guidance throughout my thesis. This thesis could not have been done without En. Asnul who not only served as my supervisor but also encouraged and guide me through the writing up thesis process by giving his best effort. I also wish to express my sincere appreciate to the lecturers, technical staffs of Faculty Mechanical Engineering, University Malaysia Pahang for their teaching and help during the period of the project.

I also wish to express sincere appreciation to all my friends for their advice to do the study. I benefited greatly from the comments and wisdom these reviewers generously shared with me.

Most importantly, I would like to thank to my family especially my parents, Puan Siti Azimah binti Husain and Encik Muhamad Bin Jusoh, which have guided me throughout my life. They have always sacrifices their time and continuous support me to achieve my dreams and goals. I would like to thank them for all support and encouragement they done for me. I truly could not have done my thesis without them.

ABSTRACT

This thesis describe about the mold design for plastic injection molding process for two colors and the analysis of gate location in order to find the best gate location to apply it in to the design. The molds for injection molding is not the cheap one it is expensive and needed an accurate step when apply the design on the mold because a slice mistake can follow by big loss. First step is about finding the best gate location from several gate location have been choose but before that the design of the product and it materials should be choose. After the analysis have done, the collected data from several gate location will be compare in order to find best gate location and apply it in the mold design. To designing mold, the several system like runner system, injector system and the ways material enter the mold will be considered.

ABSTRAK

Dalam tesis ini mengandungi huraian tentang rekaan acuan bagi proses injection molding untuk produk dua warna dan analisis tentang tempat letaknya gate bagi membolehkan untuk mencari kedudukan gate yang paling sesuai pada acuan. Dalam membuat acuan bagi injection molding process yang digunakan haruslah tepat dan sesuai dalam membuat reka bentuk acuan kerana sedikit kecacatan kepada acuan boleh membawa kepada kerugian yang besar. Bahagian pertama dalam process membuat acuan ini adalah membuat analysis tentang kedudukan gate ke atas produk tetapi sebelum itu bahan bagi produk itu hendaklah sesuai. Apabila analisis produk telah dijalankan, data data yang diperolehi haruslah dibuat perbandingan agar dapat mencari kedudukan gate yang paling sesuai bagi acuan. Dalam mengahsilkan acuan , beberapa system seperti runner system, injector system dan cara bahan bagi produk memasuki acuan diambil kira.

TABLE OF CONTENTS

CHAPTER 4 RESULTS AND DISCUSSIONS

LIST OF TABLES

LIST OF FIGURES

LIST OF SYMBOLS

Mpa	Mega Pascal
Mm	Milimeters
%	Percentage
^{o} C	Degree of Celcius

LIST OF ABBREVIATIONS

AISI	American Iron Steel Institute
ABS	Acrylonitrile Butadiene Styrene
PVC	Polyvinyl Chloride
CAD	Computer Aided Design
2D	Two Dimensional
3D	Three Dimensional
TPE	Thermoplastic Elastomers
PP	PolyPropylene
FEA	Finite Element Analysis
FE	Finite Element

CHAPTER 1

INTRODUCTION

1.1 Project Background

Injection molding process is a manufacturing technique for making parts from plastic material. The process started when the molten plastic was injected at high pressure into a mold, which was followed of the desired shape. The mold was made by a mold maker from metal, usually either steel or aluminum and precision-machined to form the features of the desired part. The product can be produce in better quality and better quantity by using process which integrated with computer control.

Injection molding nowadays have been one of the important industry in the world because of from the injection molding we can produce many type of product. The product from injection molding process expand from just one color product to multi color product, that show and make the developing in this industry becomes competitive from a day to day.

The most important requirement when design the mold for injection molding process is to get the high accuracy mold in order to reduce the cost and ensure that the mold have good specification because of this process was expensive in early set up. Year before, mold designer use manual analysis process to analyze the mold but nowadays the designer can use software to simulate the analysis of the mold. The results from analysis give very high accuracy to compare with manual analysis.

From software simulation we can avoid the problem easily. If we make the process several times without make a proper analysis the problem like warpage, shrinkage, voids and many more can be occurs and it will add the processing cost. By using the software, the designers can easily make the simulation, locate the problem and solve it with proper ways.

1.2 Problem Statement

In plastic injection there a few problem may occurs that can affect the product. The problem like warpage of the part, weak welds, and unmelted particles in molding, jetting, voids, shrinkage and many more can occurs to the products if we not design the mold correctly. The mold components are an expensive therefore the mold design should be precise and accurate in order to reduce the cost.

Different gate location give different results of defects when making the analysis and therefore the analysis about the gates location by using the Moldflow software should be analyze. The problem occurs in order to choose several gate locations to be analyze and then to understand the application of Moldflow software to make the analysis.

Then the software which is will be use to design the mold for two colors plastic injection molding process is AutoCAD and SolidWork2007. To design the mold some application for two colors mold like the product material, mold material mold plate, materials injection system and ejection system which can affect in order to design the mold should be considered.

1.3 Objectives of the Project

This project can teach the student to practice the knowledge and used the skill to apply it in problem solving. This project also important to train and increase the student capability when to answering, researching, data gathering, decision making and to solve the problem occurs to apply it in working life. The objectives of the project are;

1.3.1 To design a mold for two colors plastic injection molding process.
1.3.2 To determine the best gate location to produces the best mold design.

1.5 Scope of Study

This project will conduct about the designing the mold for two color plastic injection molding by using the software AutoCAD and Solidwork2007. To designing the mold, the best gate location need to be determine in order to apply it to the design. Moldflow software will be used for analysis the gate location. The parameter of the analysis is about the effect of gate location from several gate locations in order to find the best gate location and apply it to the design mold.

CHAPTER 2

LITERITURE REVIEW

2.1 Injection Molding

The injection molding age started when John wesley hyatt became the first whom inject hot celluloid into a mold, producing billiard balls in 1868. He and his brother Isaiah patented an injection molding machine that used a plunger in 1872, and the process remained more or less the same until 1946, when James Hendry built the first screw injection molding machine, revolutionizing the plastics industry. Roughly 95% of all molding machines now use screws to efficiently heat, mix, and inject plastic into molds. The injection molding process is a manufacturing process for making product both from thermoplastics and thermosetting plastic materials. The process start when the molten plastic was injected at high pressure in to a mold which is the mold followed the product shape. The material to make the mold usually use steel or aluminum, and precision-machined to form the features of the desired part. The element that influent injection molding process are:

a) the molder
b) the material
c) the injection machine
d) the mold

The moulder it was about the person who basically was the engineer who design and analyze the mold to make the perfect mold and get the best product from that mold process. Then the material which the most commonly used in thermoplastic materials were the polystyrene (low cost, lacking the strength and longevity), Acrylonitrile Butadiene Styrene (ABS), polyamide, polypropylene, polyethylene and polyvinyl chloride or PVC.

2.1.1 Injection Molding Machine

There are several differences between the injection molding machine for one color and for two colors. From figure 2.1 it shows about the machine for injection molding for one color. One of the differences we can see is the quantity of barrel which is to contain a material use in injection.

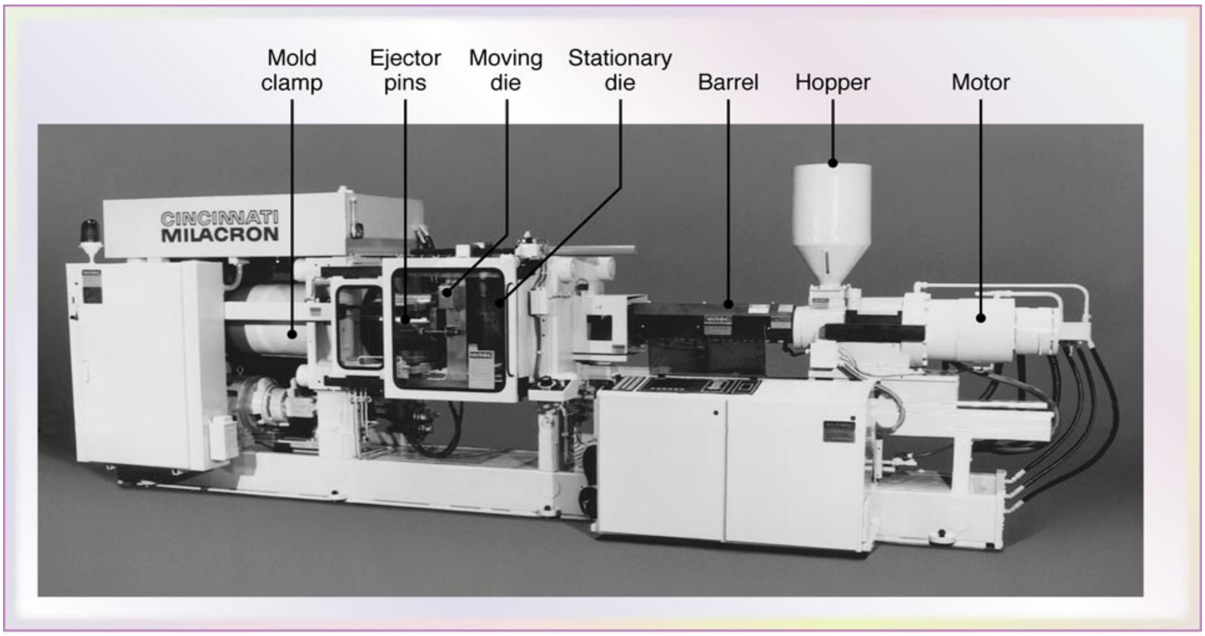


Figure 2.1: Injection Molding Machine for one color

Figure 2.2 and Figure 2.3 show the machine that for multi injection molding (two colors). It has same principle like machine in figure 2.1 which is to inject the molten material into the mold but for figure 2.2 it have a two hopper, also for figure 2.3 as use for two different type or colors of the materials. The way the machine for figure 2.2 and machine in figure 2.3 work have differences because it from difference manufacture and different axis of injection which is for figure 2.2 it use horizontal and for figure 2.3 it use vertical type of injection.

Figure 2.2

Figure 2.3

The mold design will based on the machine that be like the picture in figure 2.2 which for horizontal axis. The part on the machine still same but it have add some same part as to add the different type or different color of the materials. The basic parts which have on the machine are shown in figure 2.4;

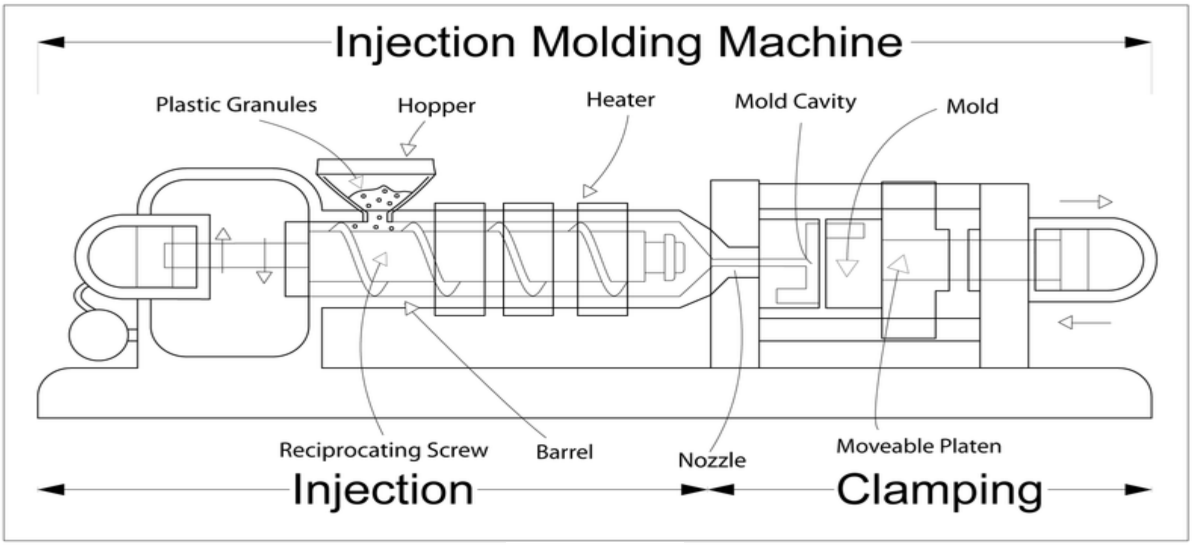


Figure 2.4

2.1.1.1 Injection Molding Parts

In the injection parts consist of the hopper, barrel, heater, and nozzle. Each of it has different function. The hopper can be fed manually or pneumatically (either semi automatically or fully automated). Several hoppers can feed one machines so that filters, colorants or other additives can be introduced simultaneously. In this case the injection molding machine also acts as a mixer. The resin enters the machine through the hopper. Then the barrel is a heavy steel cylinder built to withstand the pressure and temperature involve in melting the resin. The reciprocating Screw was to melt and inject the resin, the entire screw moves forward (usually driven by hydraulic mechanism at driver at the end of the machine) and pushed the molten resin out through the end of the barrel.

To ensure that the resin does not flow backward, a check valve or non return valve is usually used be as attachment to the end of the screw. When the resin moves forward through the nozzle, the open end of the barrel that is shape to fit into a bushing on the back of the mold. The nozzle is moved into place against this bushing during start-up and remains in contact with the bushing during all the normal operational cycles. Then the reciprocating screw combines the melting of the resin and injects the resin into the single chamber or mold.

2.2 Mold

There are three-basic types of moulds such as cold runner with two plate mold which it need trimming process to removed the channel connecting mould cavity to the end of barrel. Also cold runner with three plate mold which the runner automatically separated from the part when the mould open and last the hot runner mold which it have no gates, runners or sprues attached to the molded part.

Types of moulds used in injection molding:

(a) two-plate mold

(b) three-plate mold

(c) hot-runner mold

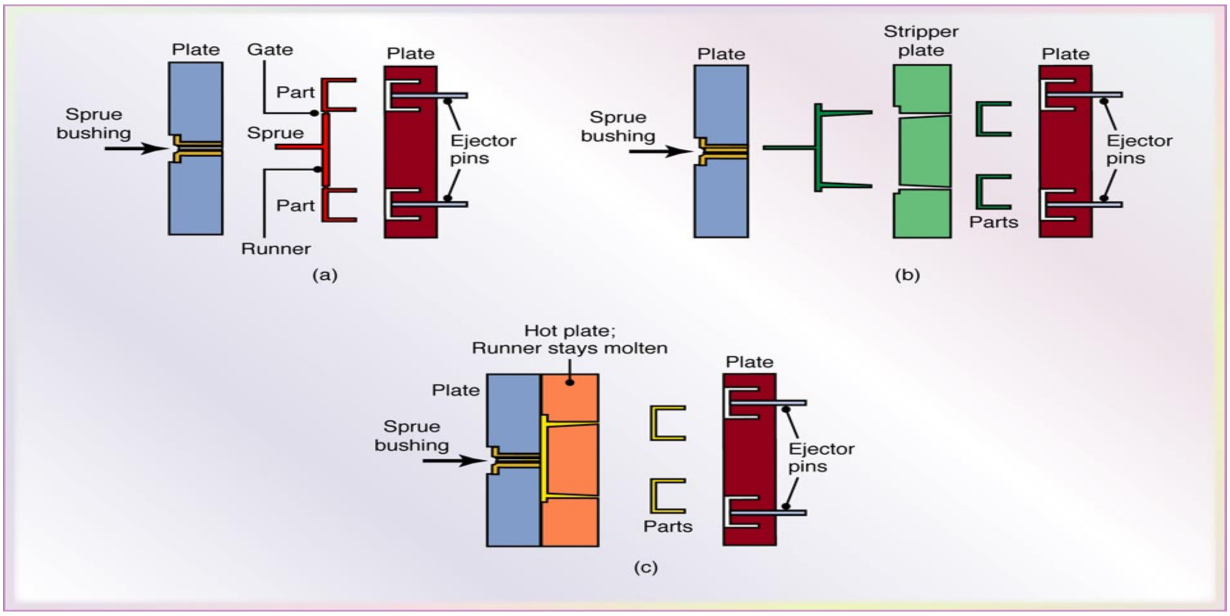


Figure 2.5: Type of Mold

There are several advantages and disadvantages for two plate and three plate mold. The type of mold to choose in making a design must depend on the dimension of the product, the quantity of the cavity or product design on the mold and the quantity of product to be produce. The opening system for this two mold type also different and the system for remove the runner also different.

As for two plates mold it use a direct cold sprue and no stripper plate are needed. This system use simple opening mechanism which is one opening between fix and moveable half plate. This two plate mold used runner on parting lines and self drop. This is shown in figure 2.6. The moveable plate of the moves with the moveable platen of the injection molding to allows the parts (products) to be removed (shown in figure 2.5).

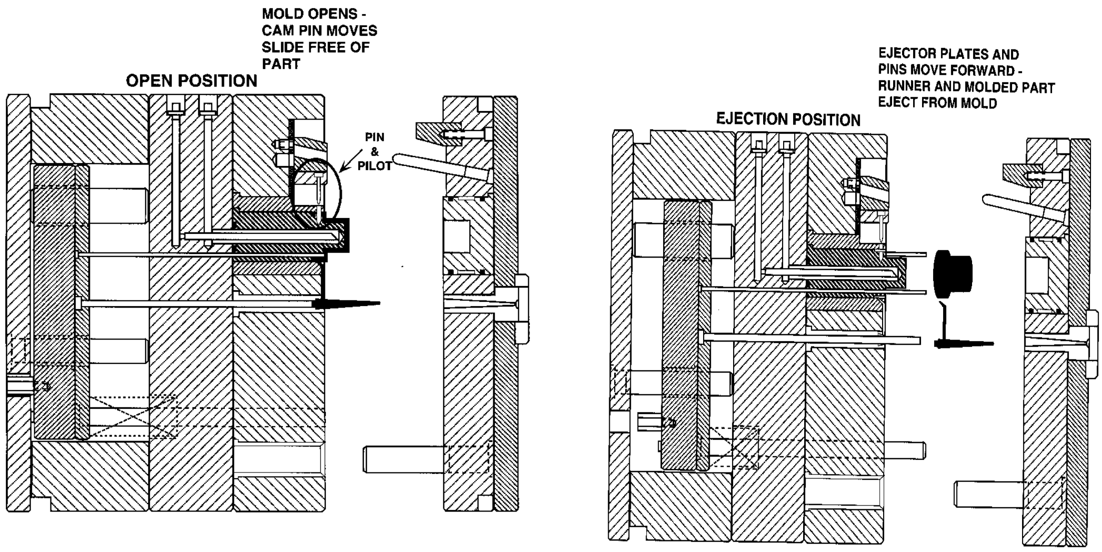


Figure 2.6: Two Plate Mold

Then for three plates mold it use indirect cold sprue and it also have a stripper plate which is to pull the runner out. The opening mechanism for this mold type is two opening between fix and moveable half. From figure 2.7 it shows about the three plate mold and it also shows there are two opening between fix plate and stripper plate and the other opening at stripper plate and moveable plate.

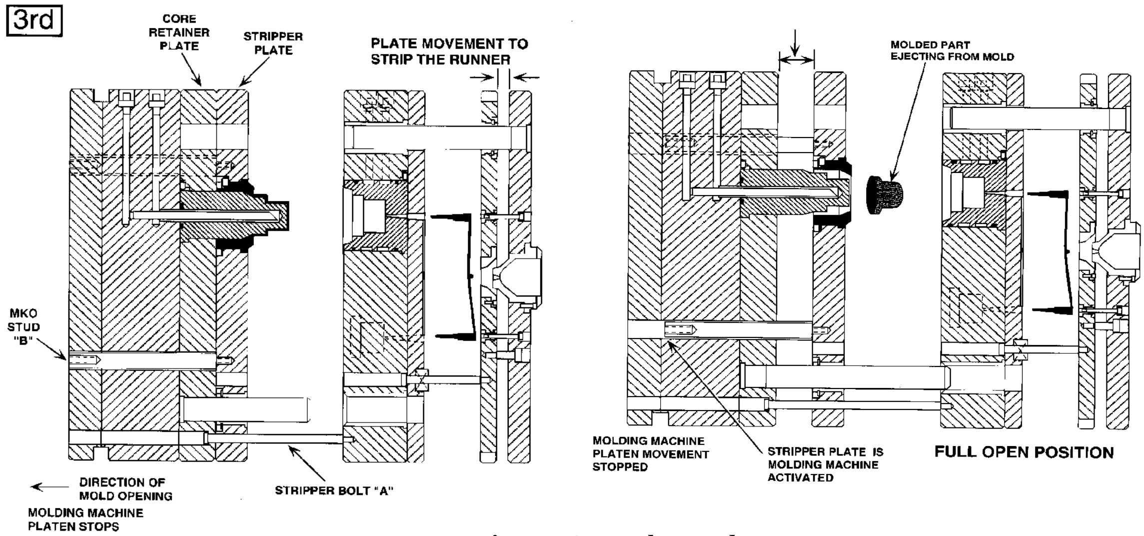
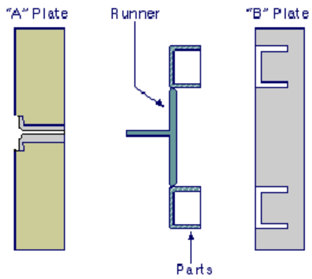
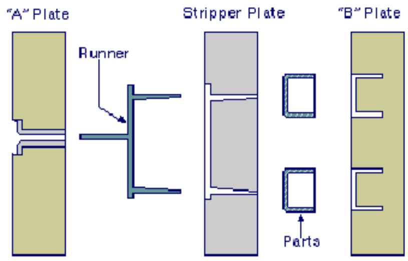


Figure 2.7: Three Plate

Table 2.1: Comparison between two and three plate mold.

Type of mold	Two plate mold	Three plate mold
		
Advantages	• Low fabrication cost. • Simple mold design. • Easy to maintenance the mold. • Short lead time for mold fabrication. • Easy to fitting and assembly the mold. • Simple ejection system of mold.	• The runner automatic separated from the part during mold opening. • Reduced production cost. • Reduced cycle time.
Disadvantages	• Need secondary process to trim of the runner. • The gate area can be seen clearly after cut.	• The mold structure is complex.

2.2 Runner

The runner is the feed system which is channeling from sprue to the cavities. Flow characteristic (viscosity), temperature and some others factors are important in order to determine runner diameter and length. If the diameter of the runner is too small or the length is too long, the resin can freeze in the runner before the mold is completely full and if the runner system is too large, excess material would be ejected and too much regrind created

2.3.1 Standard Runner Systems

Standard runners are directly machined into the mold plates, which form the main parting line and the temperature is same as the mold temperature. The frozen materials in the runner has to be remolded along with the molded part after each shot. For thermoplastic, the frozen material can generally be recycled as regrind. For the case of thermosets, it has limited scope for re use and unrecoverable material.

2.3.2 Hot-Runner Systems

Hot runners can be view as the extended injection nozzles in the form of block. Heat barriers isolated it from the cold mold. It contains the runner system consisting of central sprue bush, runners and gates or nozzles. The temperature of the block is about the melting range of the thermoplastic melts. Hot-runners have several advantages and disadvantages:

Table 2.2: Advantages and disadvantages of Hot-runners

Advantages	Disadvantages
-No loss of melt, less energy and work input.	-High cost
-Easier fully automatic operation	-The risk decomposition and production stoppages in the case of materials with low thermal resistance
-Higher quality because melts can be transfer into the cavity at the optimum sites.	-problem from thermal isolation from the hot-runner increase.

2.3.3 Cold-Runner Systems

Like the hot runner is use in thermoplastics, cold runner are used in molds for reactive materials such as thermosets and rubber. The hot mold which is kept at 160-180˚C, the cold runner must kept at 80-120˚C in order that the material may not react prematurely in the runner. The advantages are same as for thermoplastics but there a few addition like pressure consumption in cold runners is very high, a fact which make design more expensive.

The runner system which use in this project is the cold-runner systems. Multi-cavity tool layouts should be balanced. In a balanced runner system, molten material flows into each cavity at equal times and pressure. Examples of balanced spider and cross-runner systems are shown in the figures 2.8 and figure 2.9 below . The figure 2.10 shown typical runner cross-section which is best cross-section and which is poor cross-section for the runner.

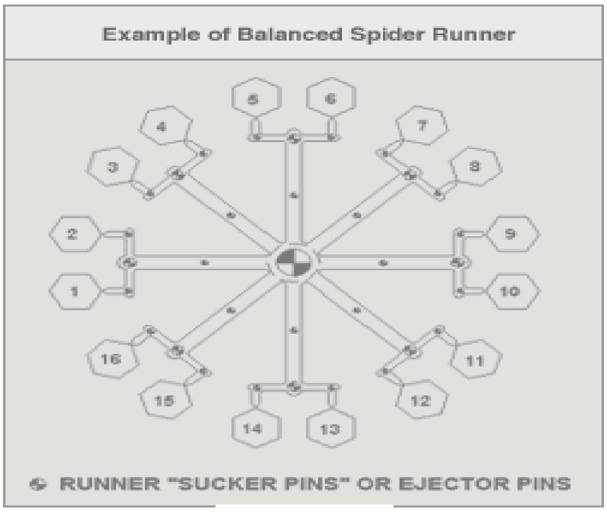


Figure 2.8

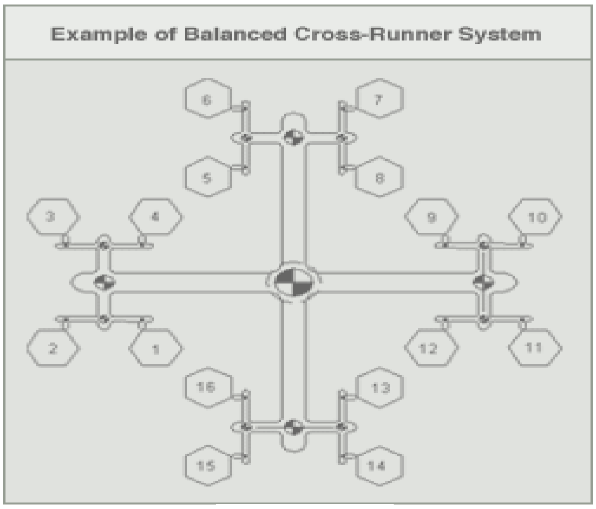


Figure 2.9

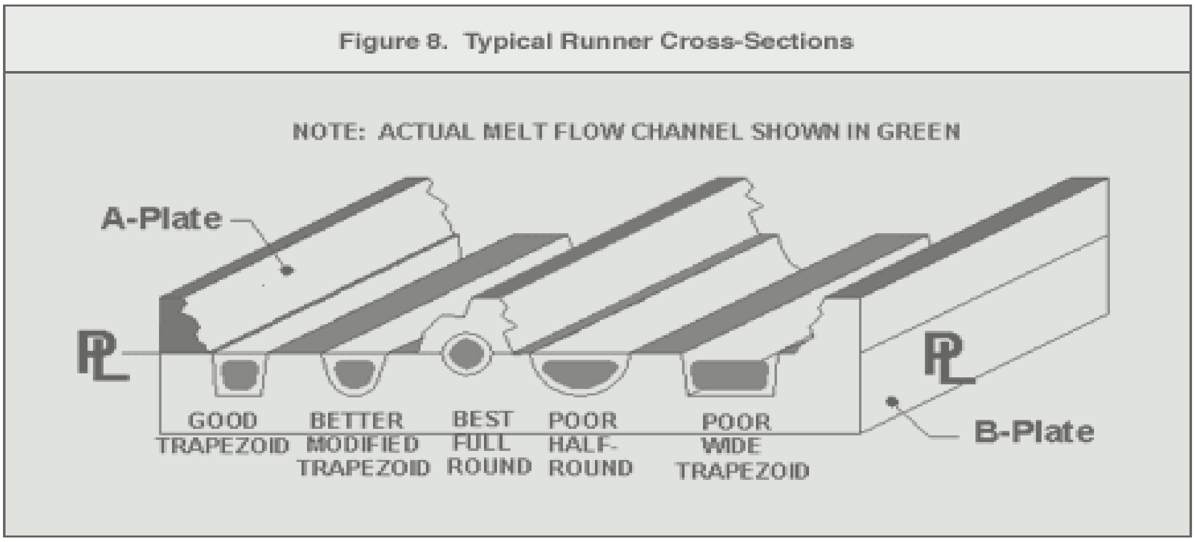


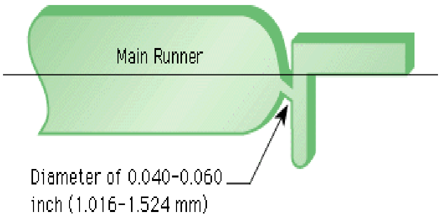
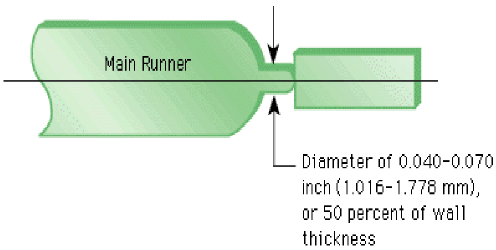
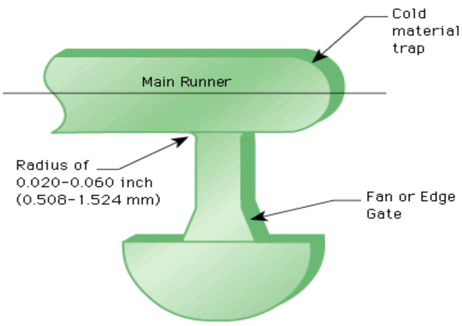
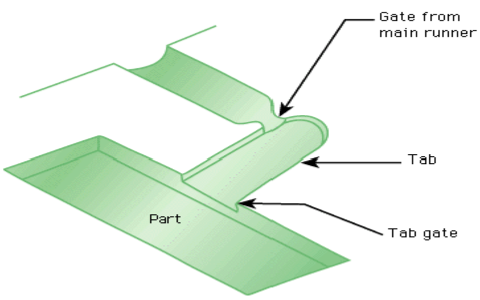
Figure 2.10

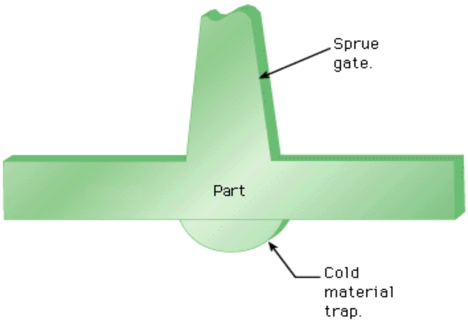
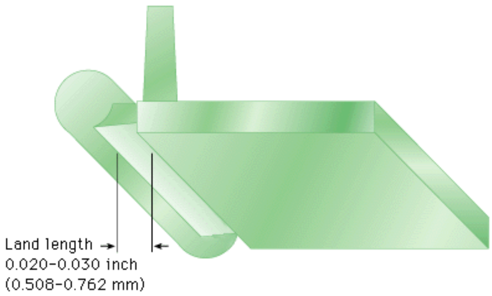
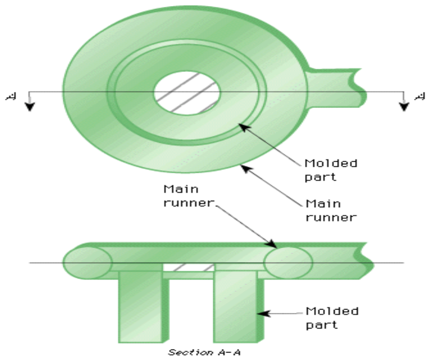
2.3 Gates

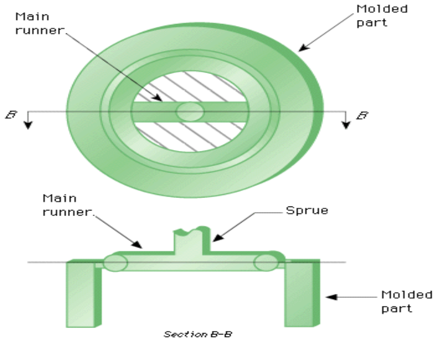
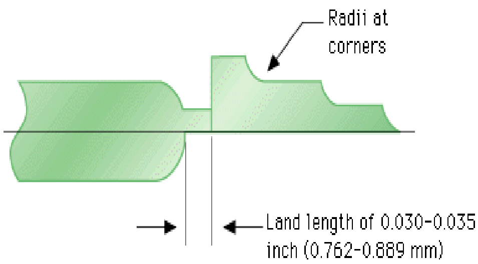
The gate is an opening through which allow the molten plastic injected into the cavity of the mold. The type and size of gate can affect the product. The mold maker must make a very accurate determination of the size and type of gate for the injection mold because of in the custom injection molding business, margins are extremely tight. The big problem like increase of cost, the damage product can occur if there have a mistake about the sizes and type of gates was wrong.

Gates can be divided in many sizes and shape depends on the types of plastic being molded and also depends on shape of the part as well. Larger parts required larger gates or need several gates. A cavity can have more than one gate and the gate should be small enough to ensure the product will have easy separation with the runner but the gate must suitable to prevent early freeze-off the melt flow, which it can affect the quality of the product. The designer should choose which gate location, type or sizes is suitable and the best for the design.

Table 2.3: Type of gates and description [6]

Type of gates	**Figure**	**Description**
a)Submarine or Tunnel Gate		-An edge gate located below the parting line or molded surface
b)Pinpoint or Restricted Gate		-A restricted opening between the runner and molded part. Normally used with thin wall parts.
c)Fan or Edge Gate		-A common gate located in the sidewall of the part to prevent restriction of resin flow. Normally used with multi-cavity, two plate molds.
d)Tab Gate		-Used for melt orientation when a large volume is needed for mold fill. The tab will helps avoid surface

		splotches due to high shear, direct gating or jetting.
e)Sprue Gate		-Recommended for single cavity molds requiring symmetrical filling. Usually used with circular parts.
f)Flash Gate		-A long, shallow, rectangular edge gate.
g)External Ring Gate		-A system used when concentricity and a smooth interior surface are important. Can be used in multi-cavity molds.

h)Internal Ring gate		-A system used when large circular parts when concentricity and smooth outer surface are required. Can only be used with single-cavity molds.
i)Thick to thin Wall Gating.		-Avoid sudden changes in wall thickness by using transition zones to eliminate stress concentrations and reduce sinks, voids, and warping in the molded part.

2.5` Part Ejections

Part ejection is more difficult in long draws areas. Ejector pins should be located at runner transitions and in areas of the part where they will not affect the aesthetics. Ejector blades, ejector sleeves and stripper rings can be used for part ejection.

Use the largest diameter pins possible to minimize pin push-through marks. Larger pins also allow the ejection of warmer parts, which can reduce cycle time. Use a 3Â° - 5Â° draft per side on all long draw areas.

Air ejection and the use of poppet can help strip large undercuts providing the material has room to deform when the air is applied. Mold surface texturing and special mold surface treatments can also help to pull the parts from the "A" half. Advancing cores are used usually when trying to strip large internal undercuts.

There is several injection components in injection molding system and this component function as to eject the cavity or product out from the mold. The components are:

Table 2.4: injector component in the mold system

Injector component	Function
Ejector retainer plate	This plate retains the ejector head pins, ejector return pins, and sprue puller pin through counter bored holes.
Ejector plate	The ejector plate acts as a back support plate for the ejector pins, return pins, and the knockout bar.
Stop pins	The stop pins are used as stops for the ejector housing when the ejector system returns as the mold closes.
Support pillars	The support pillars are used as additional support to avoid deflection of the cavity plate on the movable side of the mold
Sprue puller pin	The sprue puller is used to pull the solid sprue out of the bushing automatically when the mold opens and the molded parts and runner system are ejected.
Ejector pins	The ejector pins enter the cavity to make contact with the molded part.

2.6 CAD Software

Through the consistent of development in modern information system, many companies have increased their competitiveness in these recent years. The outcome from this brought to introduction of a CAD system or the change-over to a more powerful one is frequently measured in term of saving on time and cost during the design process.

Compared with molded part design and its sometimes complex description of freeform surfaces, mold design commands much greater proportion of CAD activities in the drawing field since the mold is largely made up of simple geometrics objects (rectangular, cylindrical, prismatic)

The CAD model of a design is a representation of the geometry in computer. The type of internal representation leads to models with different information contents. The basic design will be drawn between 2D graphics systems or 3D graphics systems.

The use of 2D system is restricted to drafting at screen level. For 2D is help user to draw all necessary views and cross-section one by one. The various view is independent to each other. The advantages of these 2D CAD drawings over a sketch are primarily that a major change does not entail having to do another complete drawing. Only 3D system describes the complete mold-part geometry. They can be divided up according to difference descriptive techniques.

First we will design the mold and the product by using CAD software to create 2D graphics system. We will include with complete dimensional for the mold and the product.

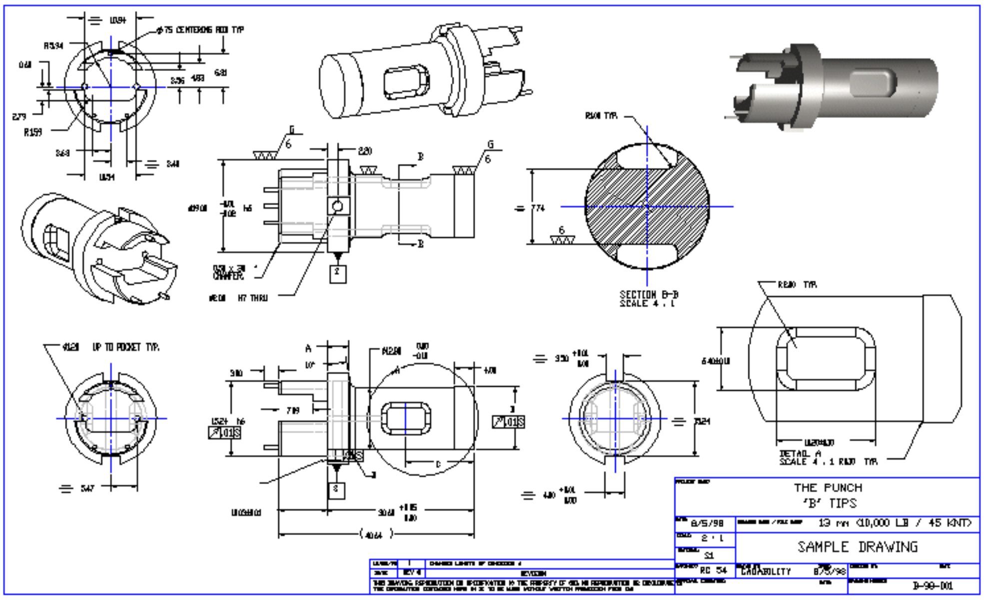


Fig 2.11: Example of picture for 2D

For 3D the CAD model of the shave type Schick exacta 2 and the mold will be created using SOLIDWORK 2007 with the dimensions followed the original physical.

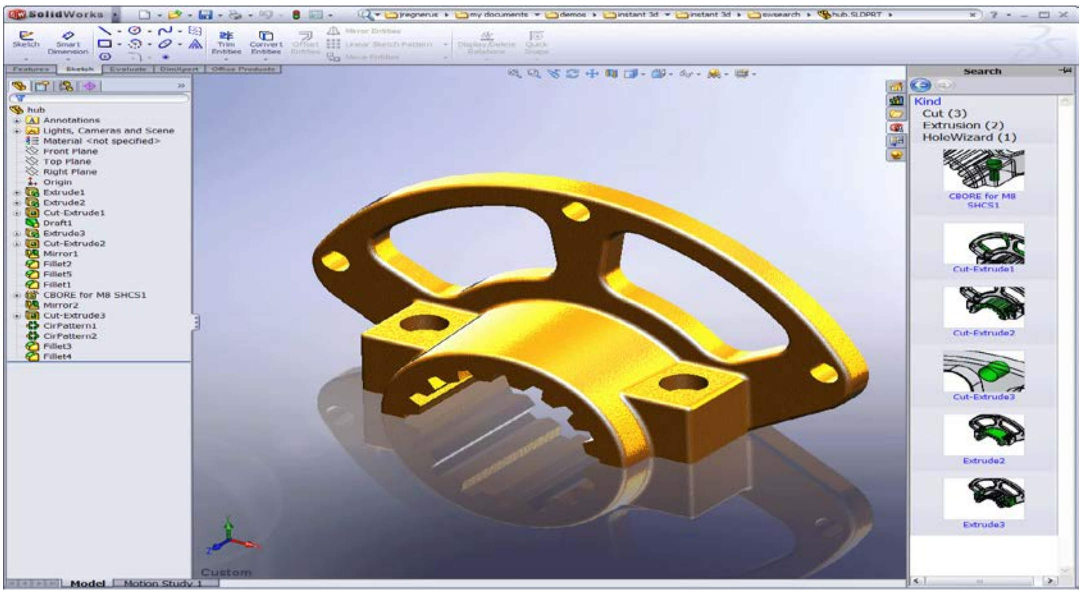


Fig 2.12: Example of picture for 3D

The design will be save in specific format because of to move it to Moldflow software after the design in the Solidwork is complete.

2.7 MoldFlow Software

Autodesk Moldflow Insight helps to simulate the filling and packing phases of the injection molding process, it help to make better predict the flow behavior of plastic melts and to achieve higher-quality manufacturing.

2.7.1 Sequence of Analysis

The procedure for mold design always start with the cavity by testing option for gate positions, optimizing molding conditions within cavity, then using that gate layout and making corrections until the cavity conditions is acceptable. Then, using these conditions the procedure addresses upstream tasks, such as defining runner dimensions. In this procedure, optimizing molding is a part of design process.

Once the gates position has been fixed and the molding condition was made, the flow rate, melt temperature, and required pressured in the runner system are determined. Which mean the cavity analysis determines a specification for the runner design.

When the filling part has been optimized, the cooling system for the part can be analyzed. For design the cooling system of the mold uniformly extracted heat from the part. This will minimize the cycle time while producing high quality parts.

Even though filling and packing are closely related, packing is the best optimized after the cooling analysis. The packing and compensation phase are dominated by heat transfer, while filling is dominated by fluid flow. The cooling analysis provides an accurate picture of how the part’s heat is extracted, so it is best to optimize the packing of the part after cooling.

The final step is to determines the warpage of the part. When the part is properly analyzed, the warpage analysis is a conformation that the part and process optimization is finish.

2.7.2 Moldflow Flow Concept

The moldflow flow concepts are a set of rules that influences both the design of a part and tools to optimize the part's filling. When these principles are followed, higher quality parts and faster cycle times are the result. Not following the principles will lead to problematic designs.

Table 2.5: Mold flow concept

Unidirectional and controlled flow pattern	Balancing with flow leaders and flow deflectors
Flow balancing	Avoid underflow
Constant pressure gradient	Avoid hesitation affects
Maximum shear stress	Controlled frictional heat
Uniform cooling	Thermal shutoff of runner
Positioning weld and meld lines	Acceptable runner/cavity ratio

For example, the software analysis results can help us determine the gate location, type and size, the approximate cycle time, the fill time of the part, the proper cooling of the mold, potential part defects (flash, shorts, warp, sink etc.) and the alternate design possibilities.

2.7.3 Injection molding for two colors concept

In this project there will be discuss about the design of mold for two colors product with two different materials will be used to it. There process for this injection system was different compare to common injection system which is for one color. The differences we can see when the injection will twice at shot to inject two materials into the cavity.

The process for the injection molding for two colors have several step and first step will the empty mold closed and the first material was injected into the mold system like you can see in figure 2.13 (d). Then the mold will opened and the mold at moveable plate will rotate by force rotatable plate like you can see in figure 2.13 (c).

As you can see at figure 2.13 (b) the mold closed and the second material been injected and for the second cavity it be injected first material. Then the mold open and the complete part are ejected by ejector system.

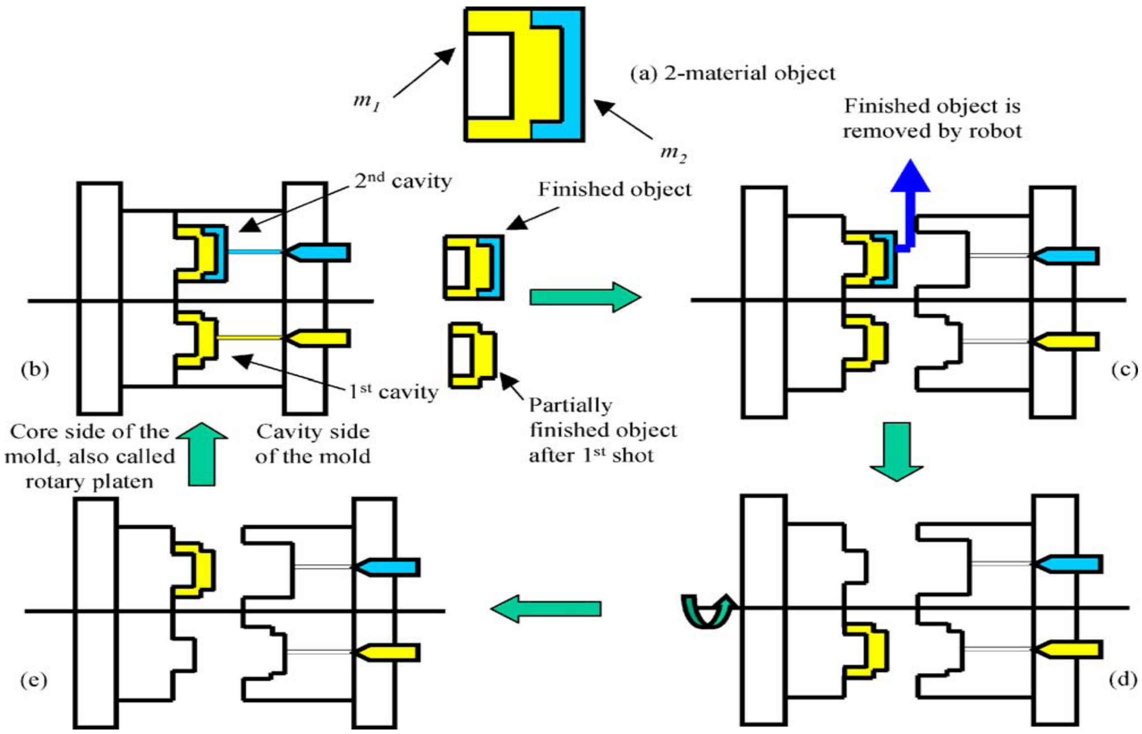


Fig 2.13: Process for injection molding two colors

Source: Xuejun Li, Satyandra K. Gupta

CHAPTER 3

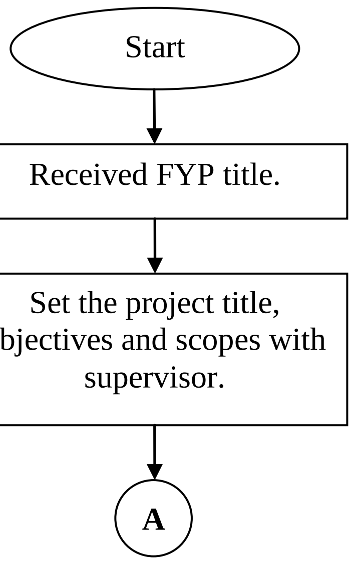
METHODOLOGY

3.1 Introduction

Methodology is about the appropriate method to conduct a project or study which is importance to ensure a smooth development of the project or study.

In this chapter there will be discussion about the method to design the mold and method we use to analysis the gate sizes to produces high quality product. To design the mold, Solidwork2007 will be used and to make analysis about the gate location Moldflow software will be used. Refer to Gantt chart in appendix D to get more information about flow chart FYP 1 and FYP 2.

3.2 Project Flow Chart for FYP 1



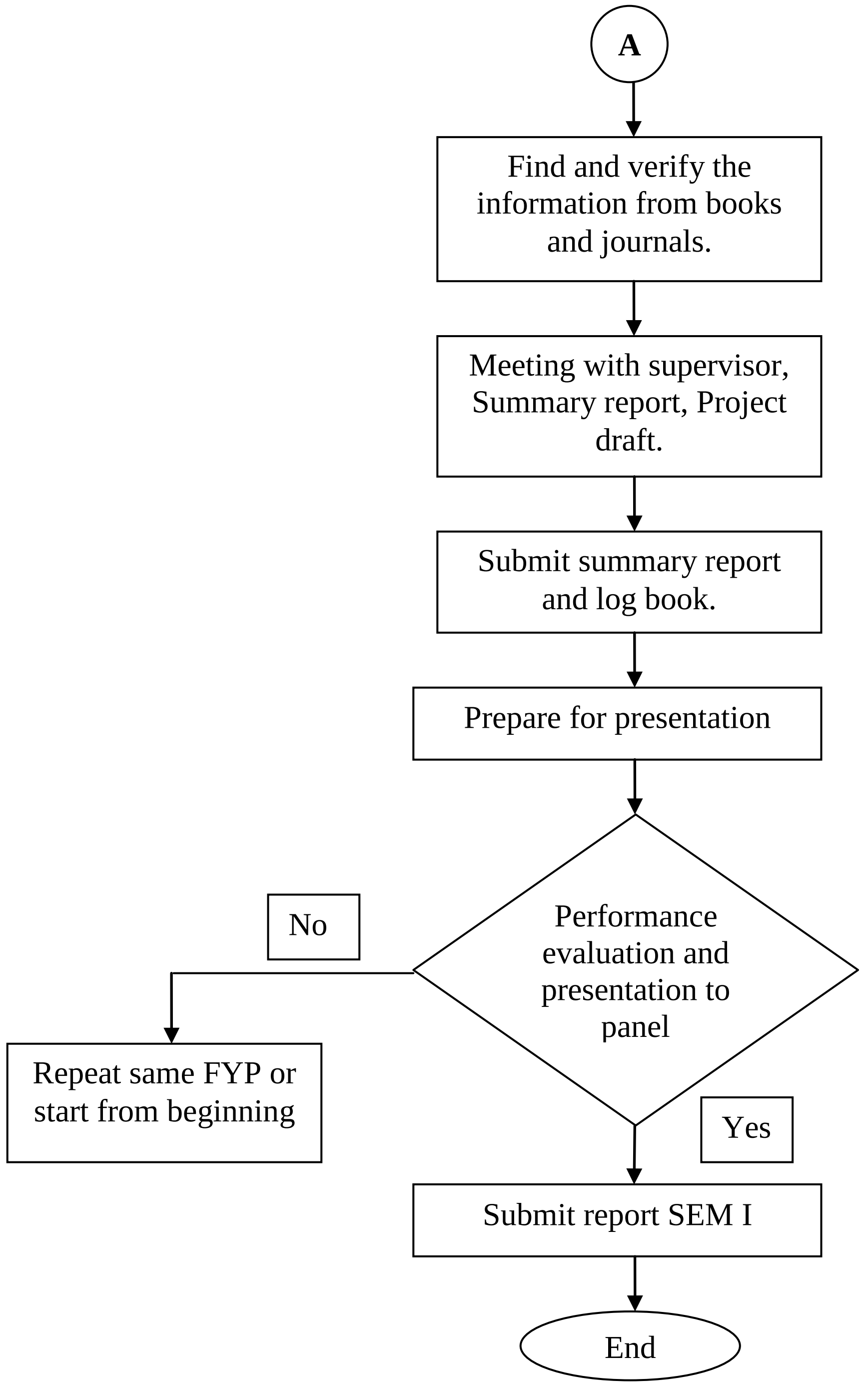


Fig 3.1: Flow chart for FYP 1

3.3 Project Flow Chart for FYP II

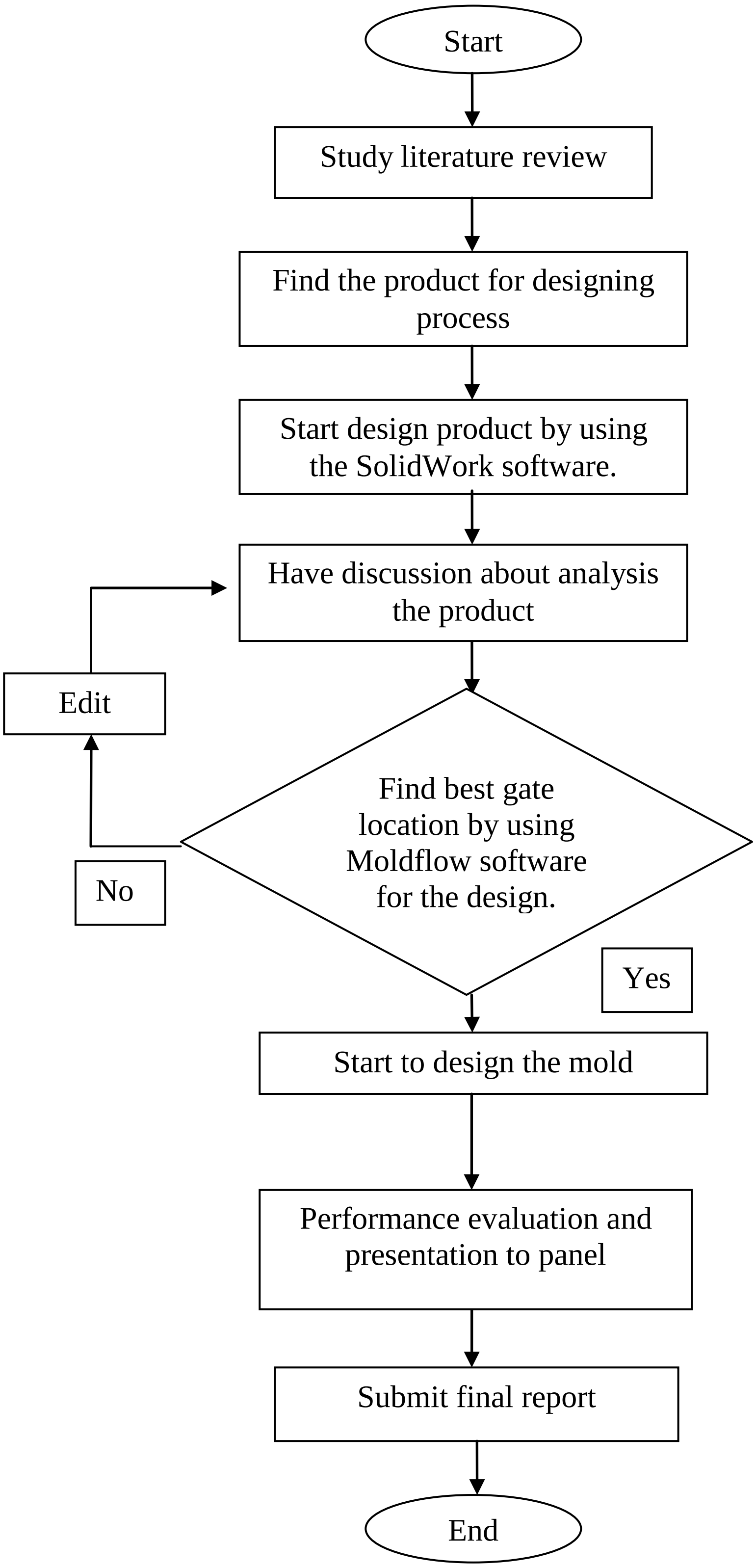


Fig 3.2: Flow chart for FYP II

3.4 Determination of design factor

Shave type Schick exacta 2 will be use as a product for designing the mold. Then we will analysis the gates sizes parameter. The figure 3.3, figure 3.4, figure 3.5 and figure 3.6 show the side view, top view, front view and bottom view.

Figure 3.3: Side view

Figure 3.4: Top view

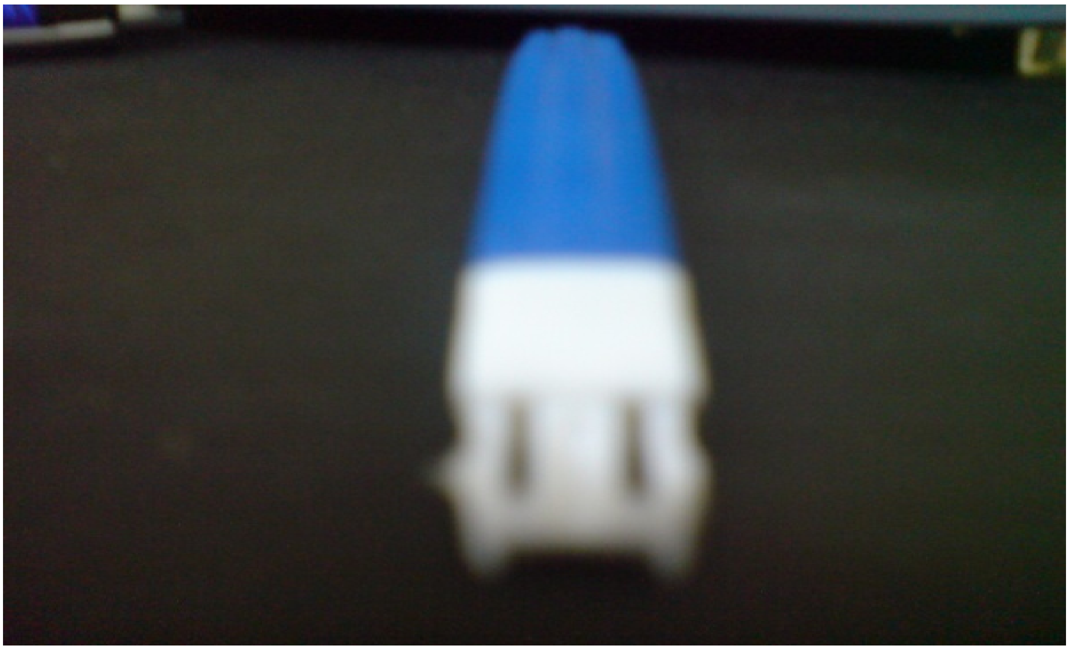


Figure 3.5: Front view

Figure 3.6: Bottom view

At this stage, first step will define about the dimension of this product like for the length, height and thickness. To get this product dimension, the real product was use as a guide. After that sketching stages started, as a guide for next step for CAD software development.

3.4.1 CAD Data Development

First step is to design the mold and the selected product to apply on mold by using CAD software to create 2D graphics system. The complete dimensional for the mold and the product will be included.

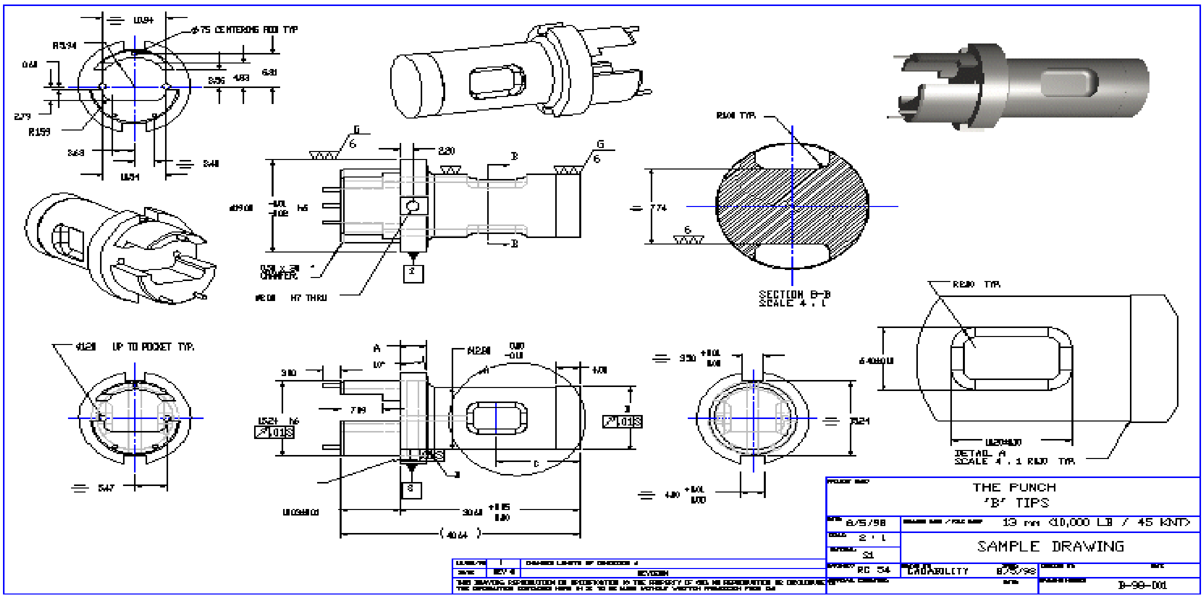


Fig 3.7: Example of picture for 2D dimension

For 3D the CAD model of the shave type Schick exacta 2 and the mold will be created using SOLIDWORK 2007 with the dimensions followed the original physical.

3.5 Moldflow Analysis

The simulation on mould flow of the designed injection mould using Moldflow software is to test whether the part produced is free from defects. If defect does exist, modification on injection mould should be done or parameters setting such as injection temperatures and pressures should be altered in order to eliminate the defects.

3.5.1 Moldflow Analysis Step Framework

The exact steps taken to conduct an analysis on parts are as varied as the parts and problem to be solved. Results from an analysis will guide in order to a solution path that will change depending on the problem and choices to make to fix problem. Generally, there are several ways to solve a problem, better than the others. Followings are the basics steps for conducting flow, cooling and warpage analysis on a part.

To run the analysis of the product by using Moldflow software several step need to be followed. First step is run the Moldflow software, then select import in order to import the file and this shown in figure 3.8.

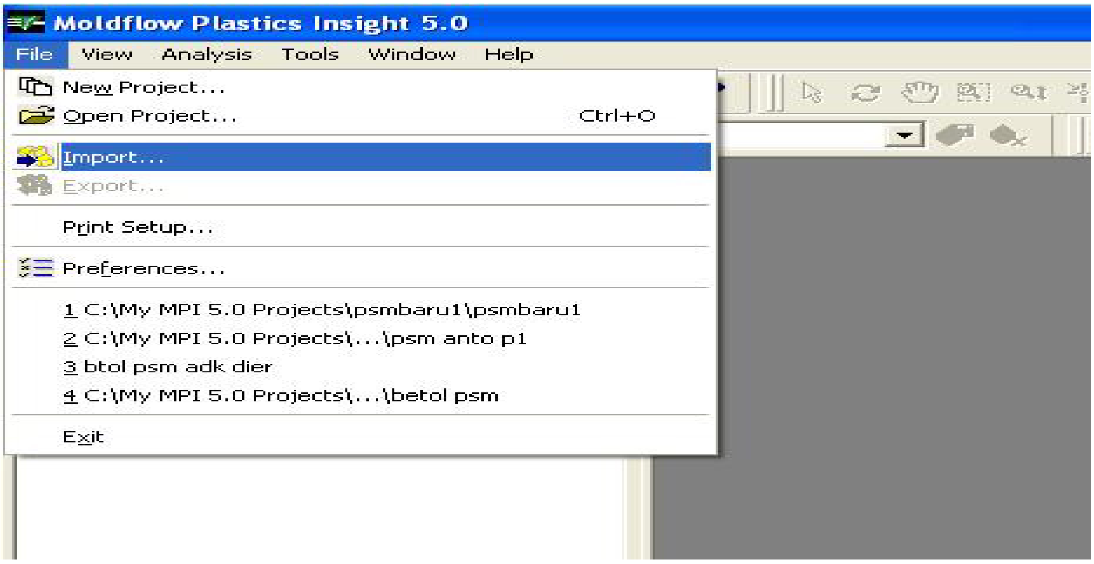


Figure 3.8: Import File

Then the second step is set the design as a fusion type and set the name for the design as a PSM Part 1. These were shown in figure 3.9 and figure 3.10.

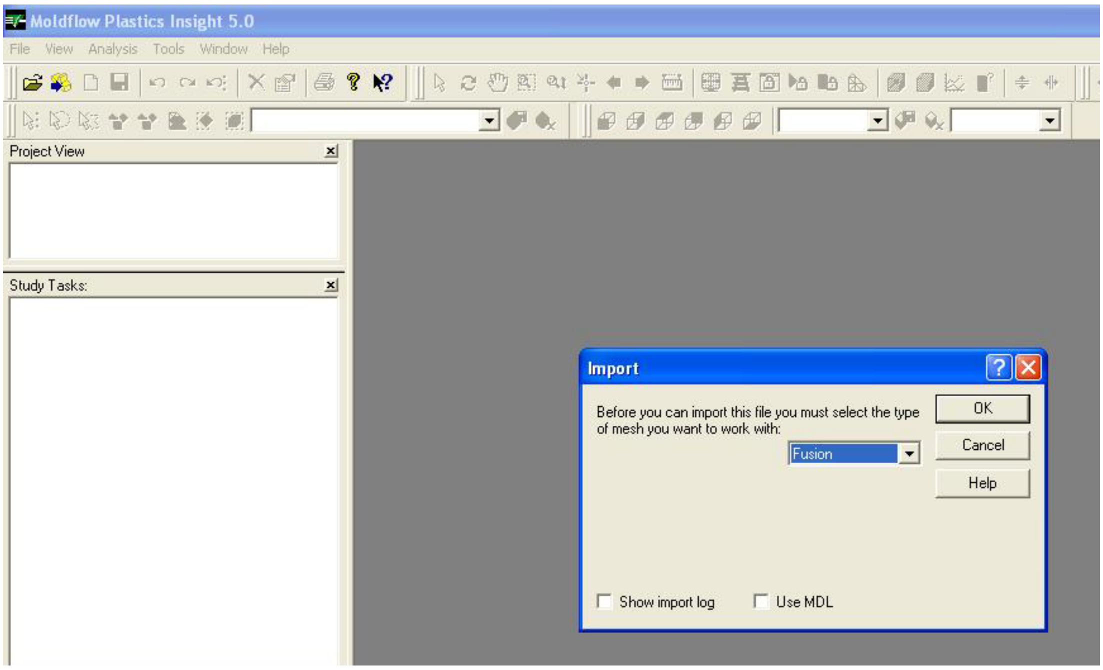


Figure 3.9: Set as fusion type

Figure 3.10: Set Part name

After part name was setup and then ‘OK’ button was clicked, the model 3D for the product will show and it shown in figure 3.11. This is shown that the process can proceed to the next step.

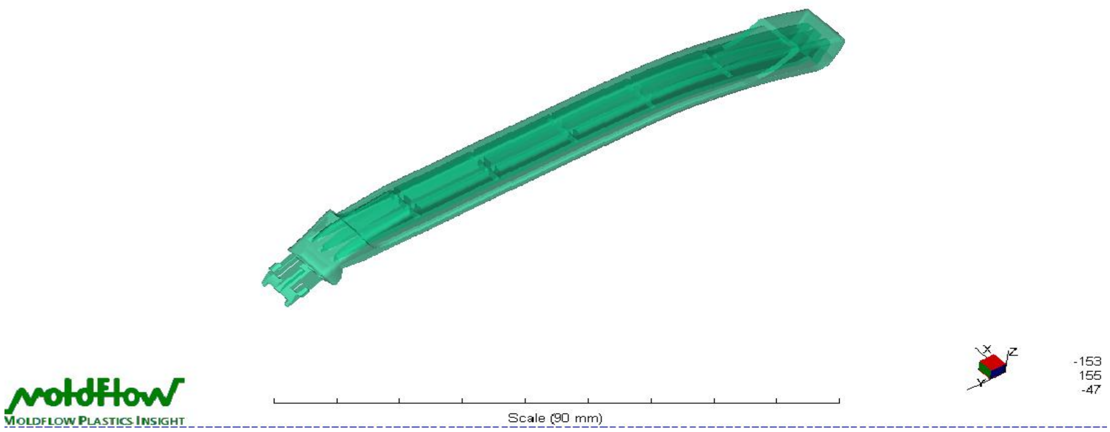


Figure 3.11: 3D Model

The next step is to duplicate the model. This is because for one injection there will inject material for two cavity. To duplicate the model, by clicking on ‘Modeling’ and then choose ‘Cavity Duplicate Wizard’, which is shown in figure 3.12. After that, the Cavity Duplicate Wizard will appear and set 2 for number of cavity, put one on row box, for column and row spacing we set as 26 and then click ‘Finish’, this is shown in figure 3.12 and figure 3.13.

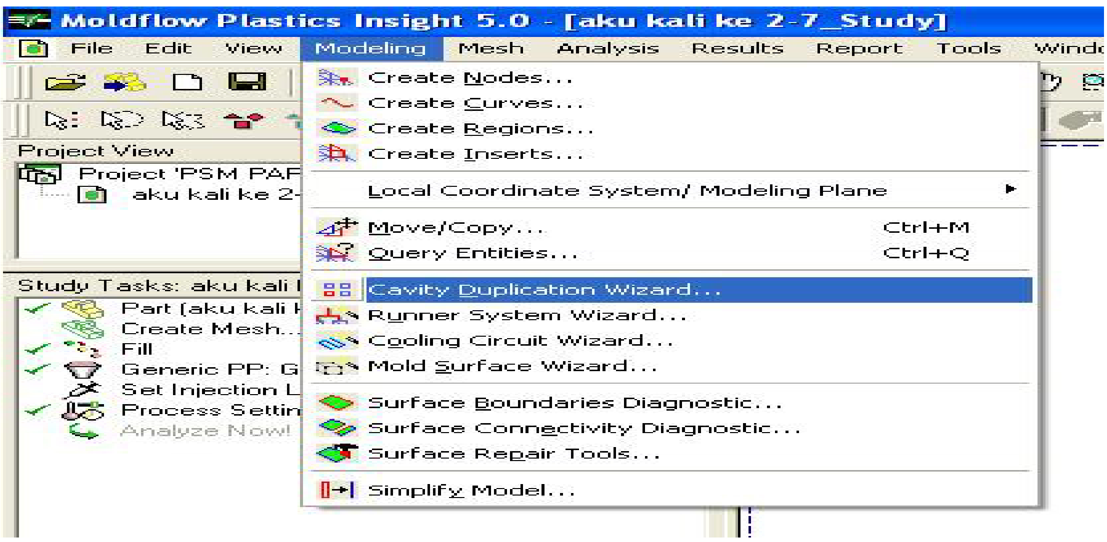


Figure 3.12: Duplicating the cavity

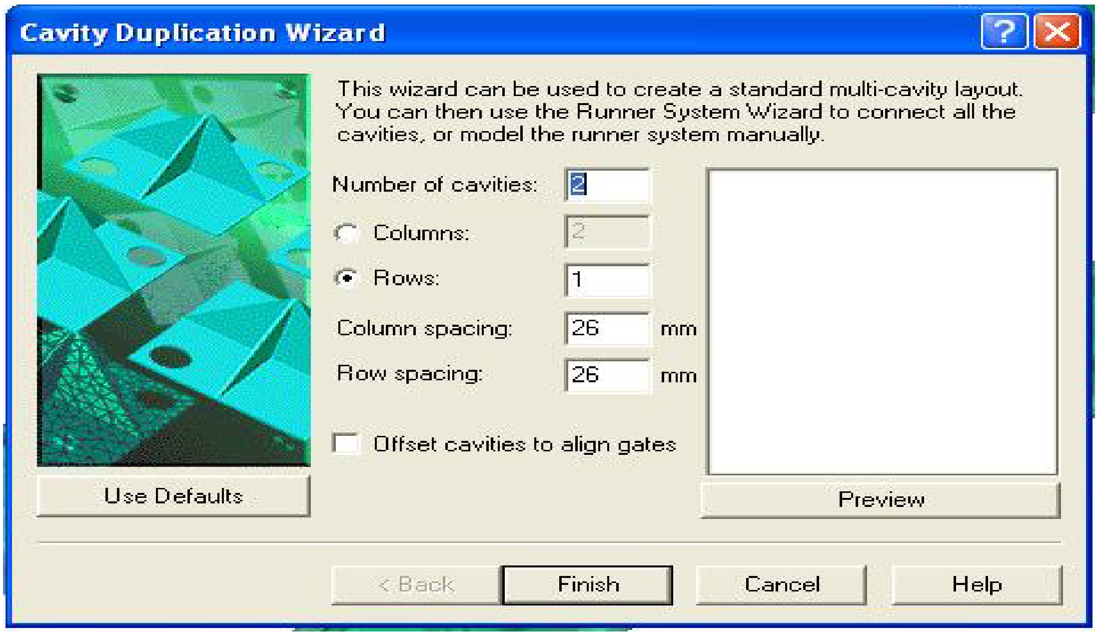


Figure 3.13: Cavity Duplication Wizard

Then, from 'Study Task', 'Create Mesh' was choose and after that, choose generate mesh, this will shown on figure 3.14. Then from figure 3.15, it will shown the meshing model which was been generated.

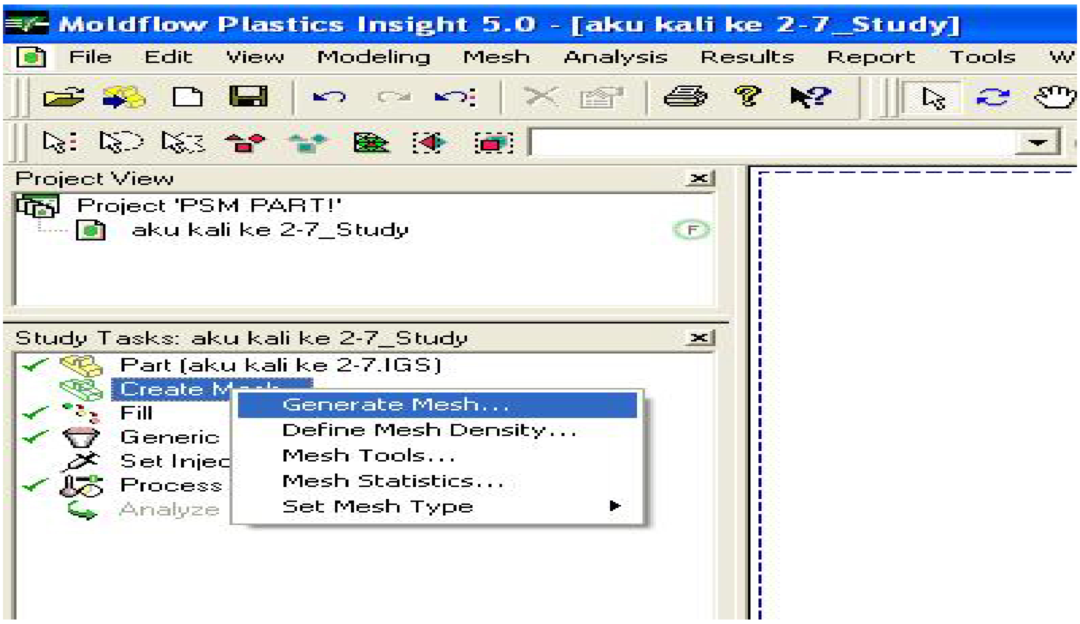


Figure 3.14: Generating mesh

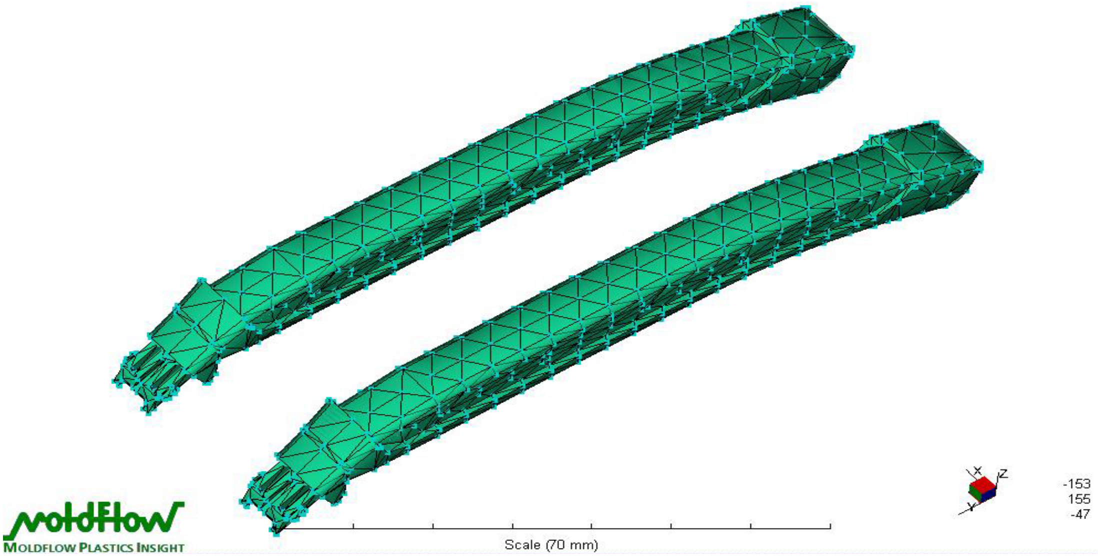


Figure 3.15: 3D Meshing Model

After that, first step which is start to setting the analysis sequence by click at 'Set Analysis Sequence' at the study task and then choose flow and warpage sequence in order to make the analysis. The figure 3.16 and figure 3.17 shows that the way to set the analysis sequence.

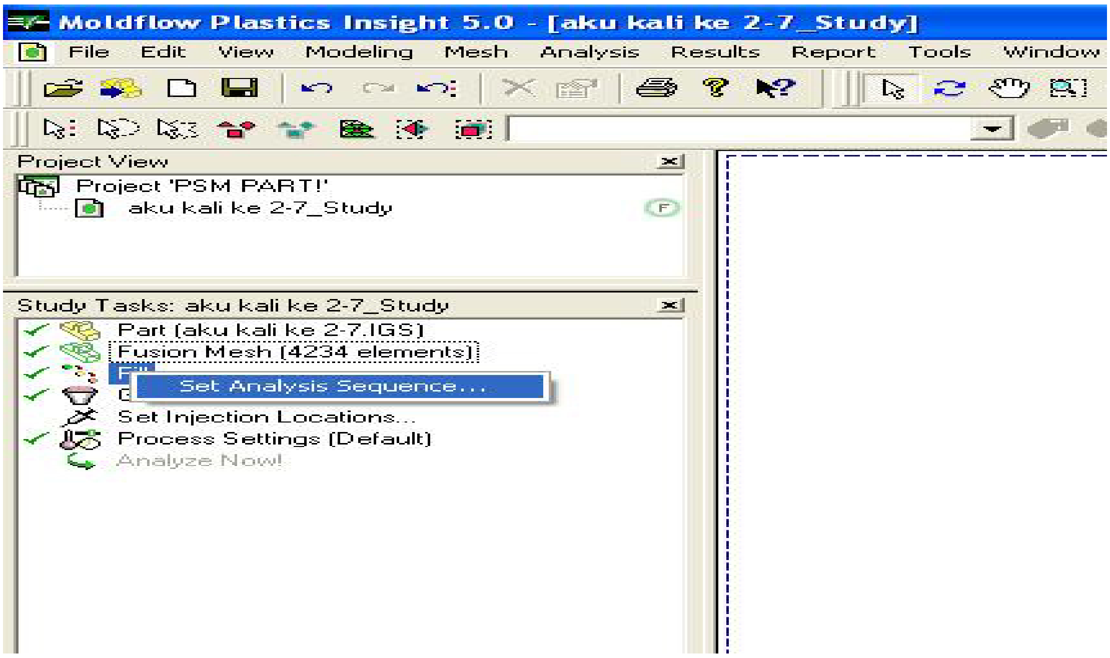


Figure 3.16: Set Analysis Sequence

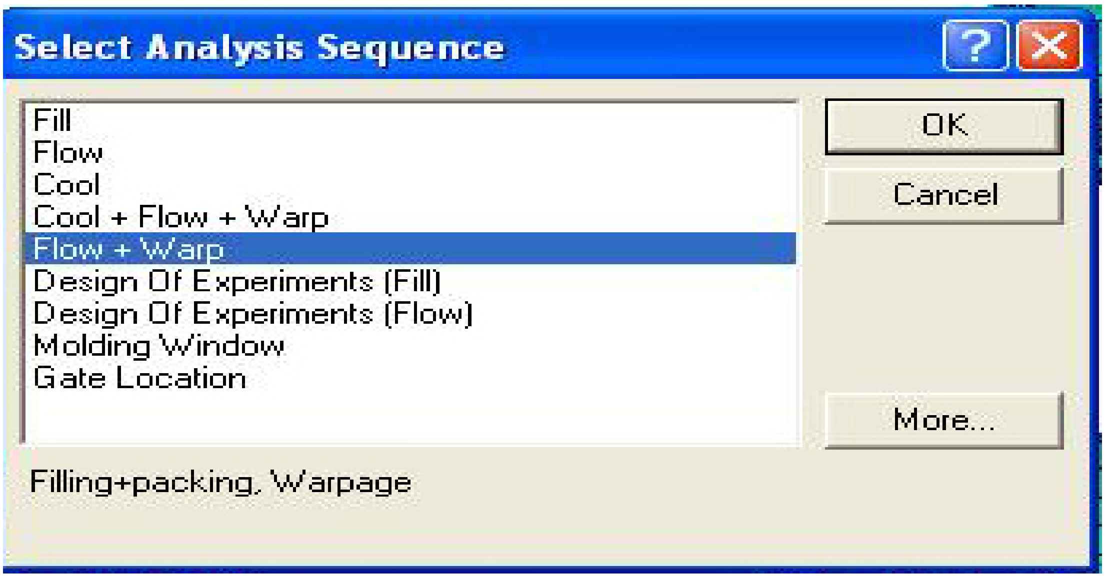


Figure 3.17: Select Analysis Sequence

After analysis sequences were selected, the next step was about choosing the material. From the ‘Study Task’, ‘Select Material’ been clicked on order to select material for the part 1 and this shown in figure 3.18 and figure 3.18 which is the material was been set up.

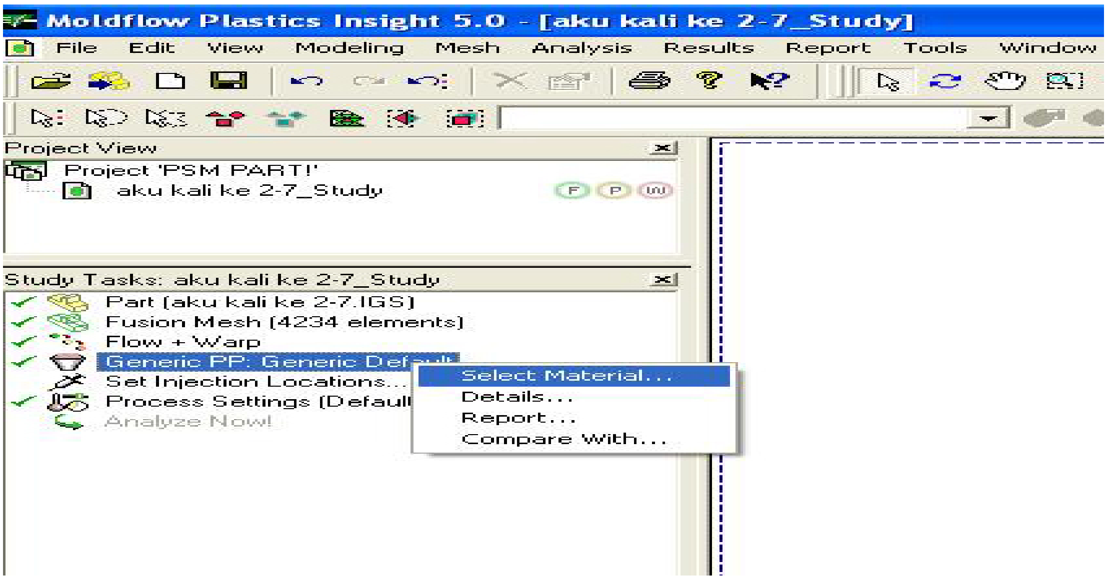


Figure 3.18: Product Material Selection

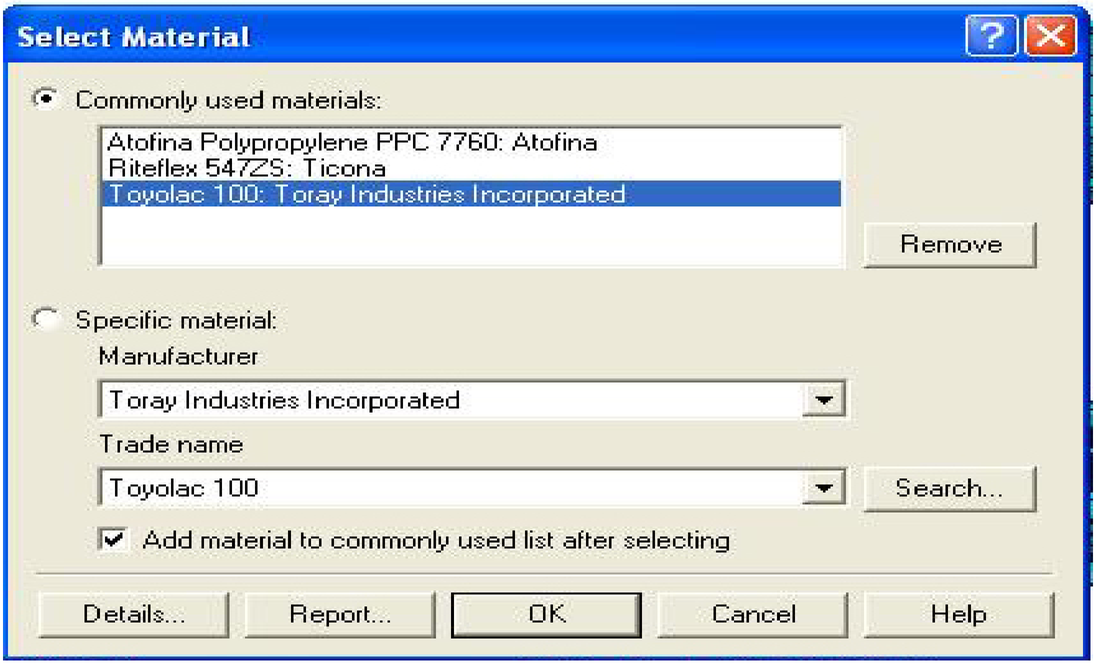


Figure 3.19: Material For Product Part 1

Next step is to set up the injection locations which are representing gate location to the product and it shown in figure 3.20 and figure 3.21.

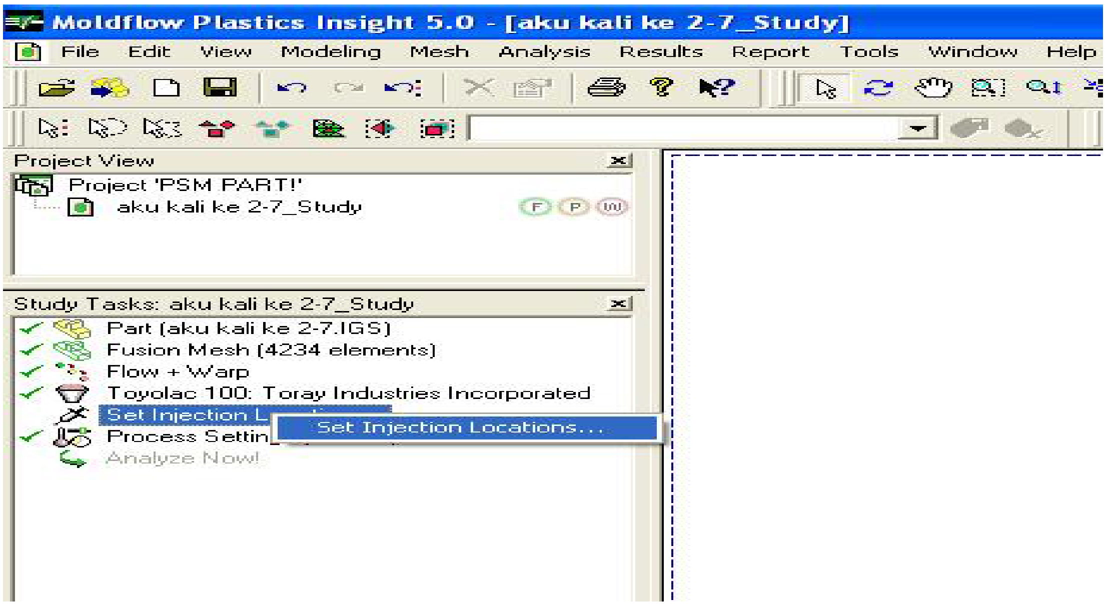


Figure 3.20: Injection Location set up

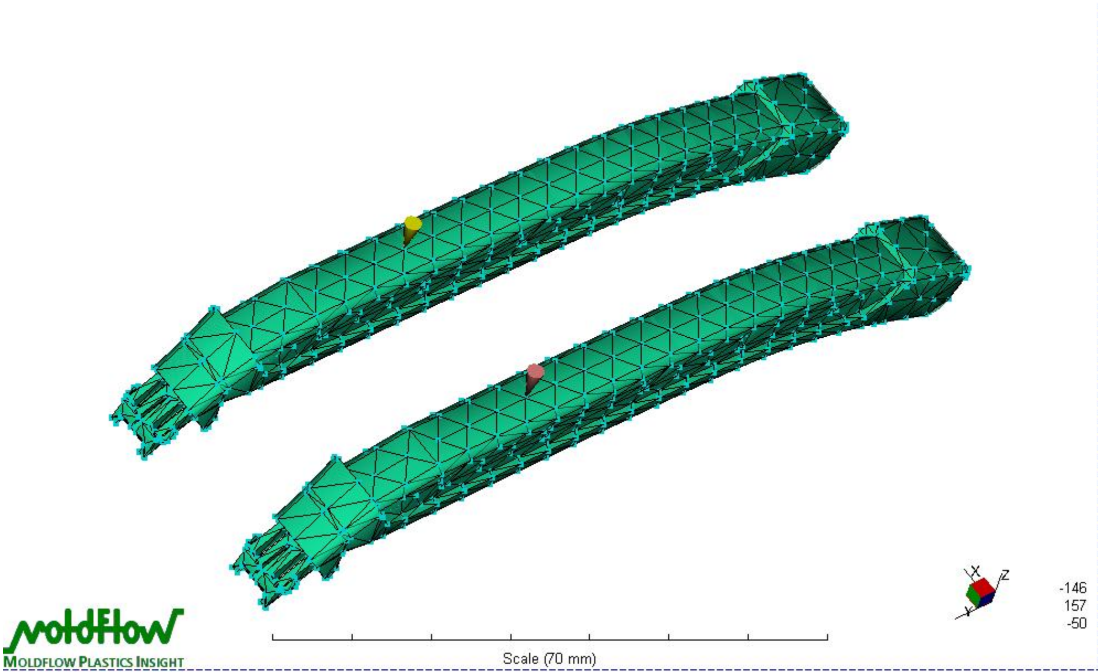


Figure 3.21: Injection Location

After injection location were set up then move to set up runner system it show on figure 3.22 and then begin to set up sprue orifice diameter 2.5mm with length 25mm then set the runner diameter for 3mm. Then put 3mm with length 1mm for the gate sizes in runner system wizard which has shown in figure 3.23, figure 3.24 and figure 3.25.

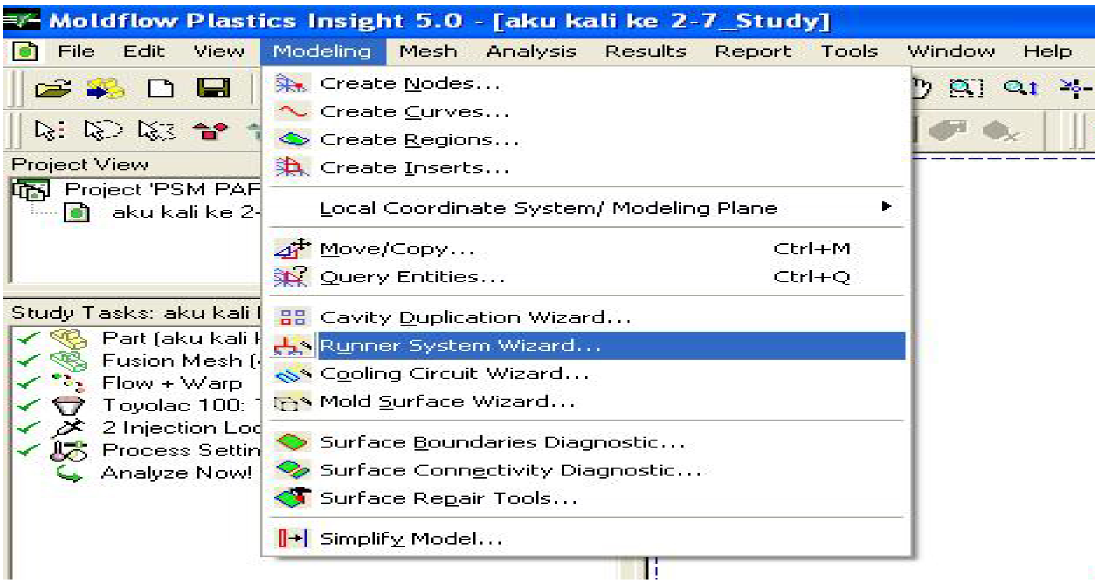


Figure 3.22: Set the Runner System

Figure 3.23: Setting the Runner System page 1

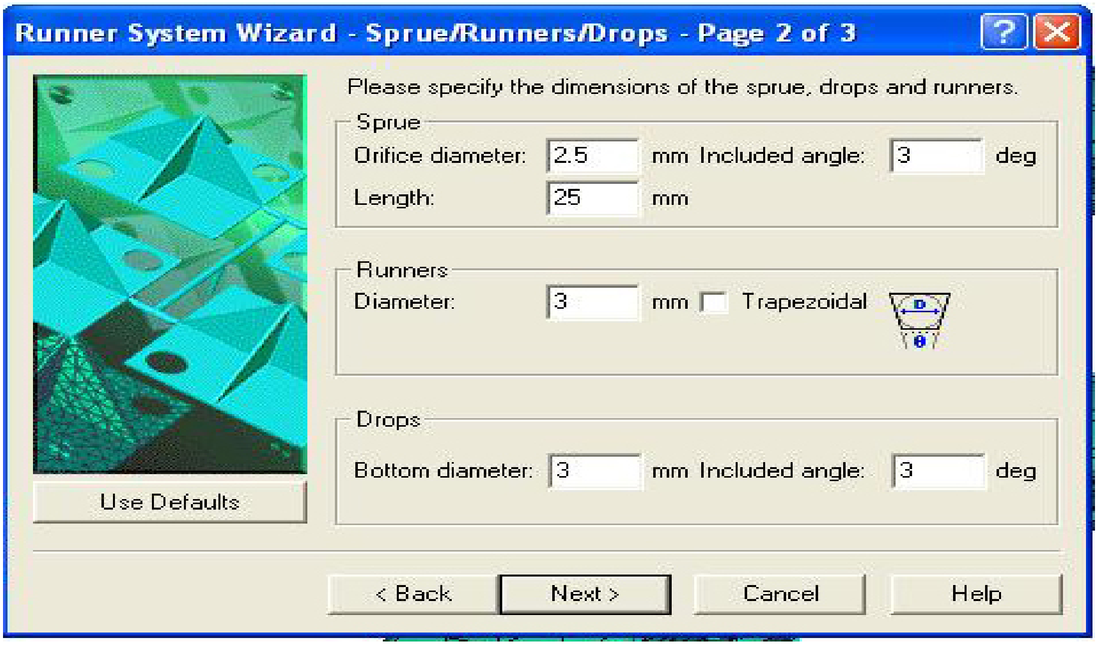


Figure 3.24: Setting the Runner System page 2

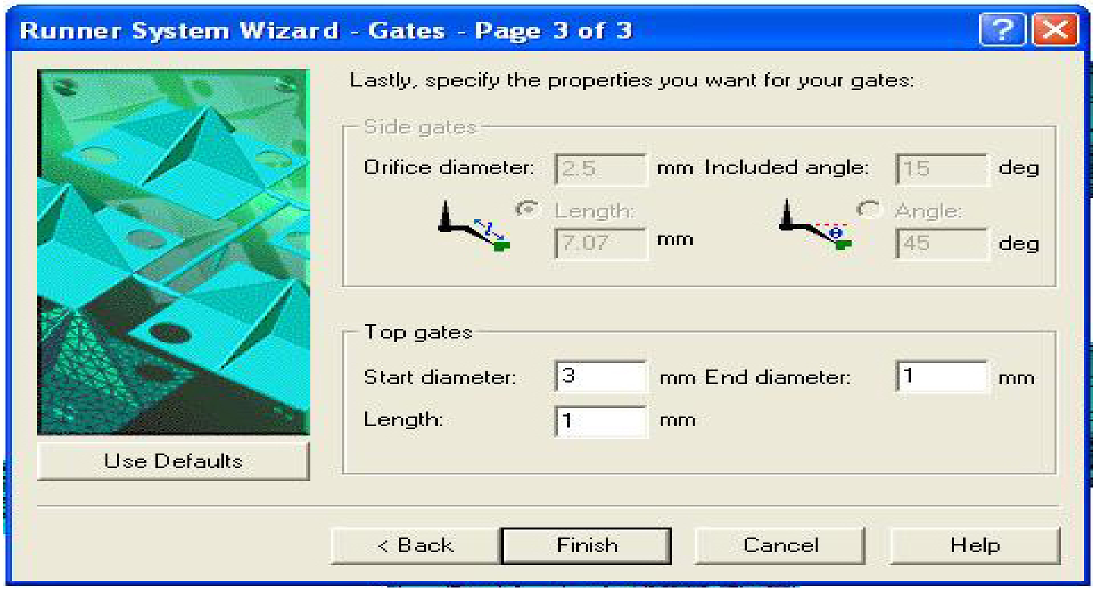


Figure 3.25: Setting the Runner System page 3

After runner system was set up to the product model, the product with the runner system shown in figure 3.26 and the 'Analysis' been click at the 'Study Task' in order to run the analysis. Steps continue from the first step when the new gate location needed to analyze and the same step repeated for the part 2.

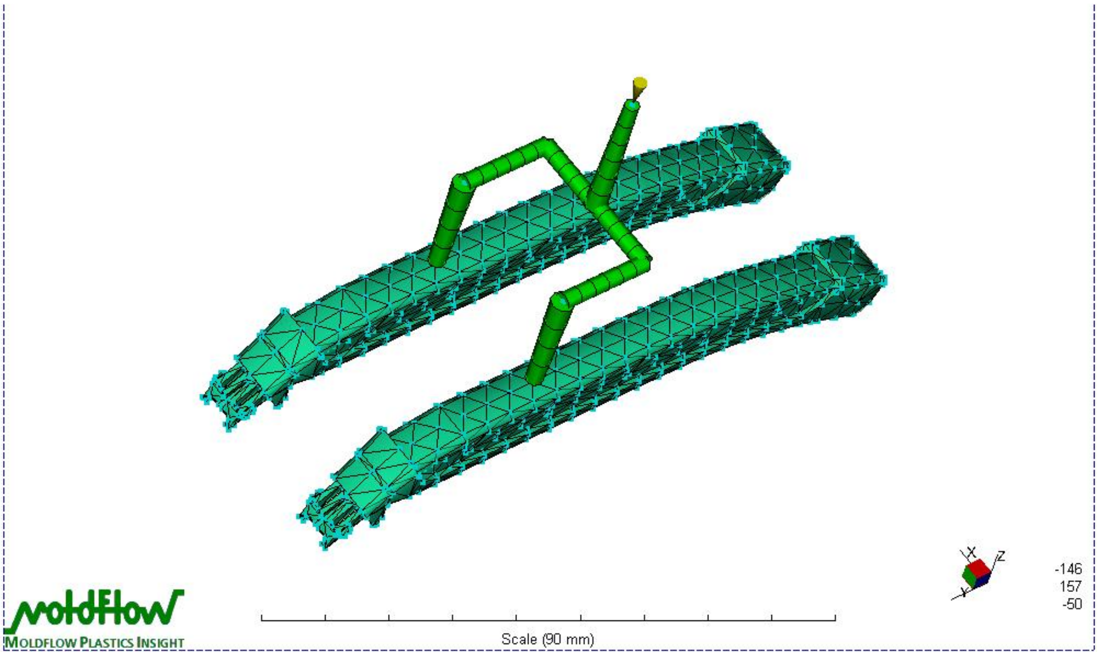


Figure 3.26: Runner System on the Product Model

3.5.1.1 The whole Process

The procedure for analyzing a part from filling to warpage. It follows the Moldflow design principle, the filling of the part, followed by the runners cooling system, packing the warpage.

3.5.1.2 Optimize Fill

Optimizing the filling of the part is the first major step in part optimization. This is the foundation for all other analysis work. The step to optimize the fill are determine analysis objectives then prepare FE mesh, then select material after that we select gate location ,molding machine, molding conditions, then set molding parameters and run the analysis.

3.5.1.3 Balance and Size the Runners

After the filling of the part has been optimized, the sizing and balancing of the runner system needs to be analyze. Depending on the layout and complexity of the runner system, the runners simply need to be sized to minimize volume, and other times the runners must be balanced to achieved the required filling pattern within and between parts

3.5.1.4 Optimize cooling

Once the runner sizing/ balancing is done, the cooling of the mold can be considered. The part of the process contain of determine analysis objectives, model cooling components, review cooling results and then revise input to the cooling analysis.

3.5.1.5 Optimize the Packing Profile

Packing is the best done after a cooling analysis because the packing and compensation phases are heat transfer dominated. When a packing analysis is conducted after cooling analysis, the tool's ability to extract heat from the part is accurately modeled. The step which is involve in optimizing a packing profile is determine initial Packing Pressure and Time and then reviews the results.

The primary output from a packing analysis is the volumetric shrinkage. The amount and distribution of the volumetric shrinkage is critical for determining the linear shrinkage and warpage of the part.

3.5.1.6 Gate Location Analysis

There will be analysis about gate location in order to determine the best gate location to apply at mold design. The best gate location should be determined when we want to produce the product with fewer defects. In order to run this gate location analysis, we will use several parameters to take as a constant parameter.

The constant parameter like mold temperature, melt temperature, mold cavity side temperature, and maximum clamp force will be determined. This will help to get the best gate location result. The parameter that we consider will be shown in table 3.1.

Table 3.1 Parameter for the gate location analysis

Process Parameter	**Value**
Maximum Pressure	180.00 MPa
Melt Temperature	230.00 ºc
Mold Cavity Side Temperature	50.00 ºc
Mold Core Side Temperature	50.00 ºc
Machine Maximum Clamp Force	7002.12 tonne
Maximum Clamp Force Required	47.57 tonne

For the product that we want to analysis was the product with two color, therefore we divided the product in two part because of it use different materials with different colors, it use different process like two nozzle will be used and one material injected per nozzle then the second material will continue to injected by using the second nozzle. The separate two parts will be shown in figure 3.27.

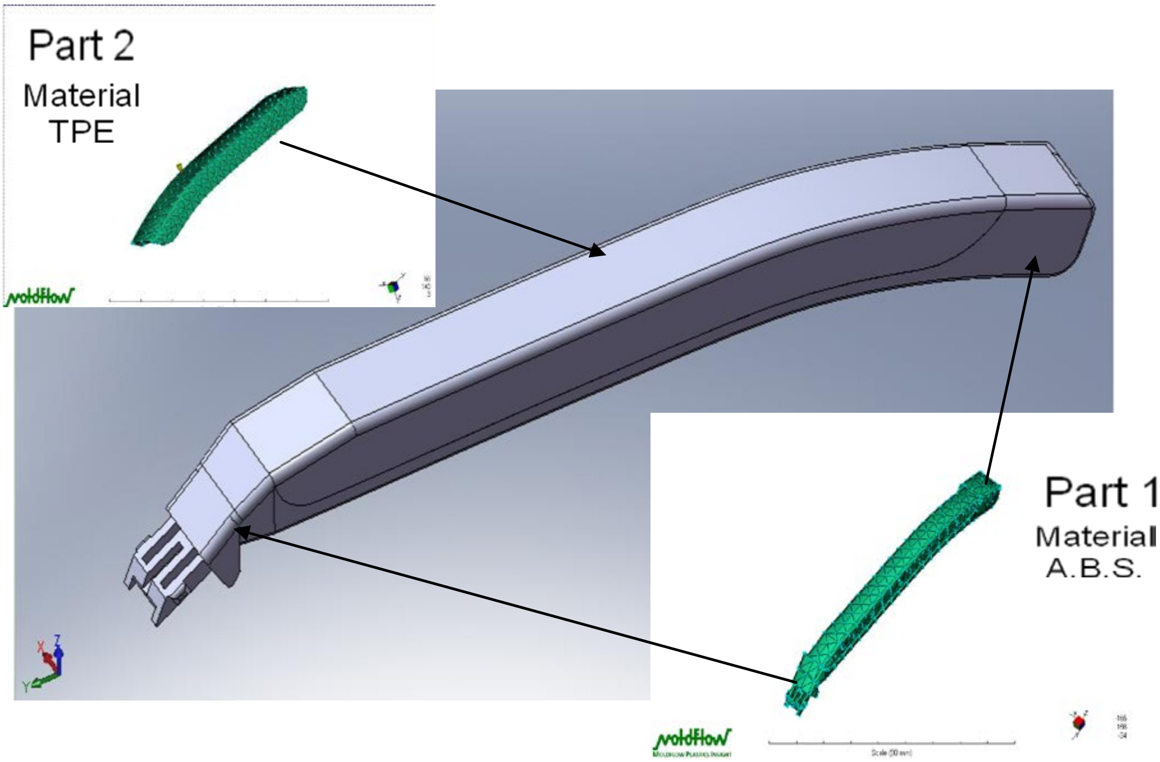


Figure 3.27: 3D product designs

In this gate location analysis several results will be taken in order to make the comparison. The result like fill time which is to determine the fill time result which is to shows the position of the flow front at the cavity fills. Time to freeze is the times for the molten materials take to solidification and when converging flow fronts surround and trap a bubble of air will create air traps.

Volumetric shrinkage is the contraction of polymer due to the change in temperature from melt temperature to ambient temperature therefore this also will considered in this analysis. Weld lines occurred when the thin frozen layers at the front of each flow path meet. This can affect the product and that why this also been considered in this analysis.

For this analysis we will take about 6 gate locations for part 1 and part 2. Figure 3.28 and figure 3.29 below show the gate locations which were considered.

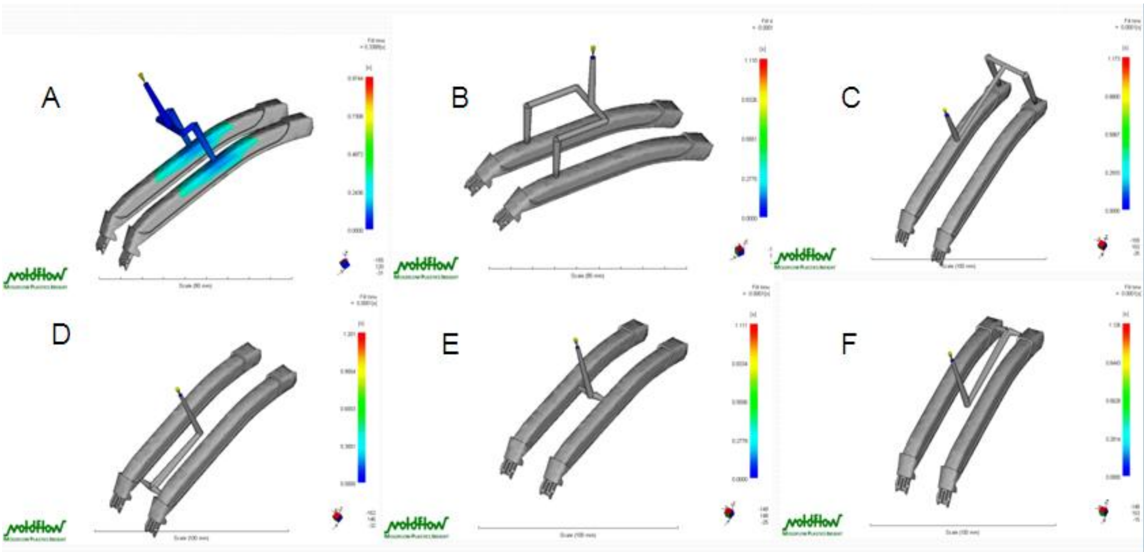


Figure 3.28: Gate Location for Part 1

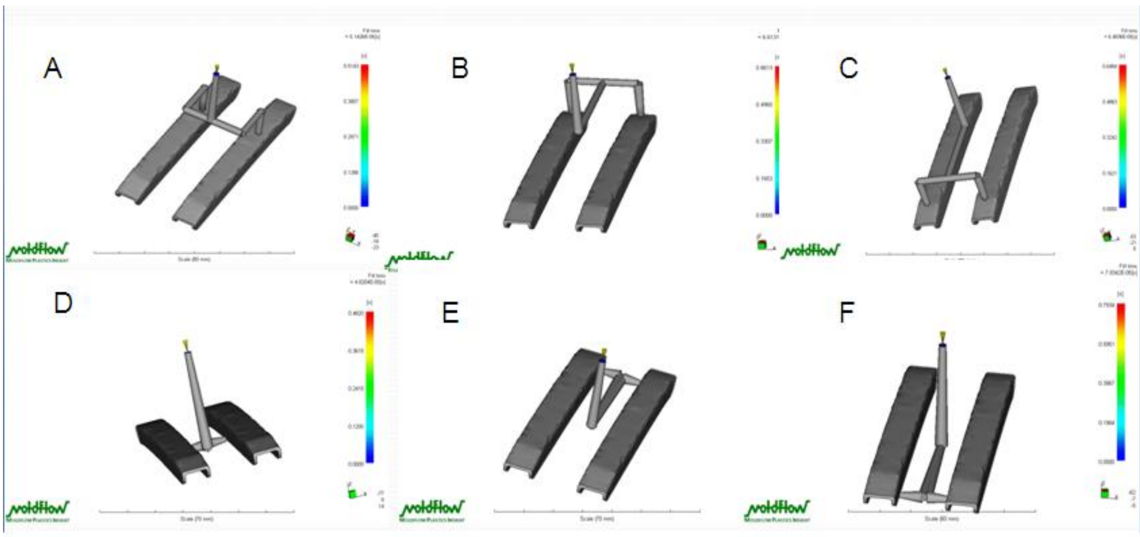


Fig 3.29: Gate Location for Part 2

In this analysis, we make the stages in order to differentiate the quantity of the air traps and weld lines that occurs in the product. These stages have 5 values which are dividing from 5 for too many and give value 1 for the gate location which is produce very small quantity of air traps or weld lines. The figure 3.30 and figure 3.31 show the air traps and weld lines stage for the part.

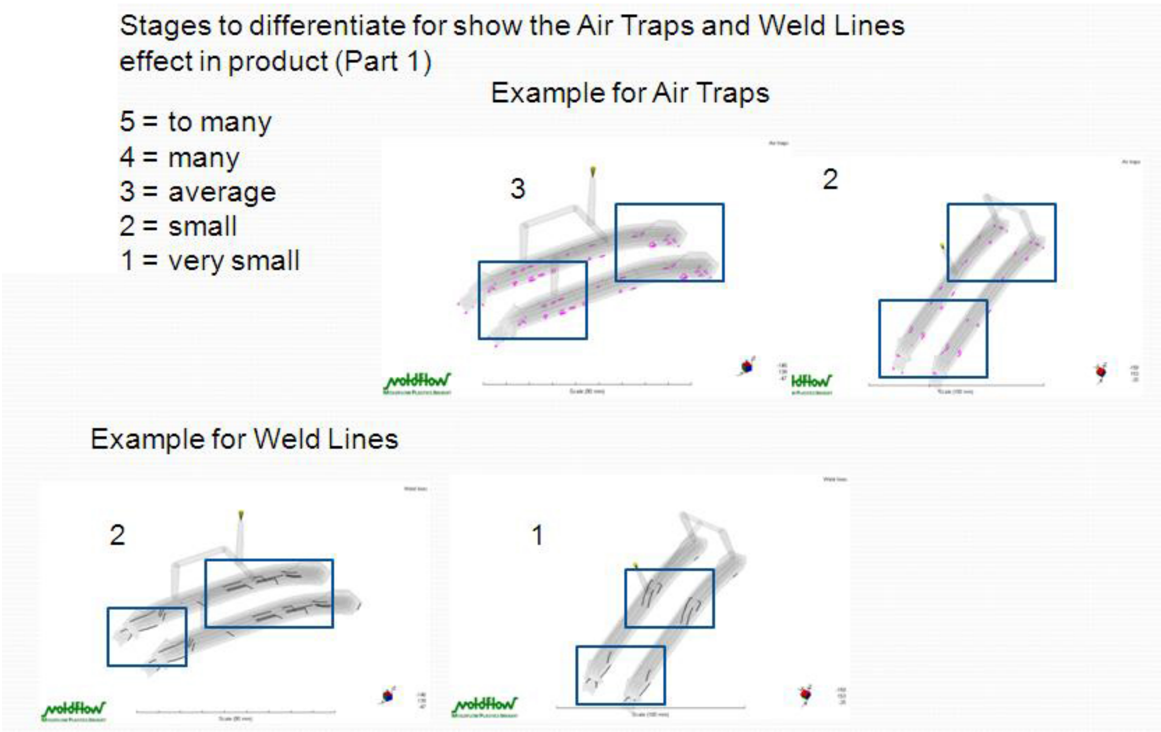


Fig 3.30: Air Traps and Weld Lines Stage for Part 1

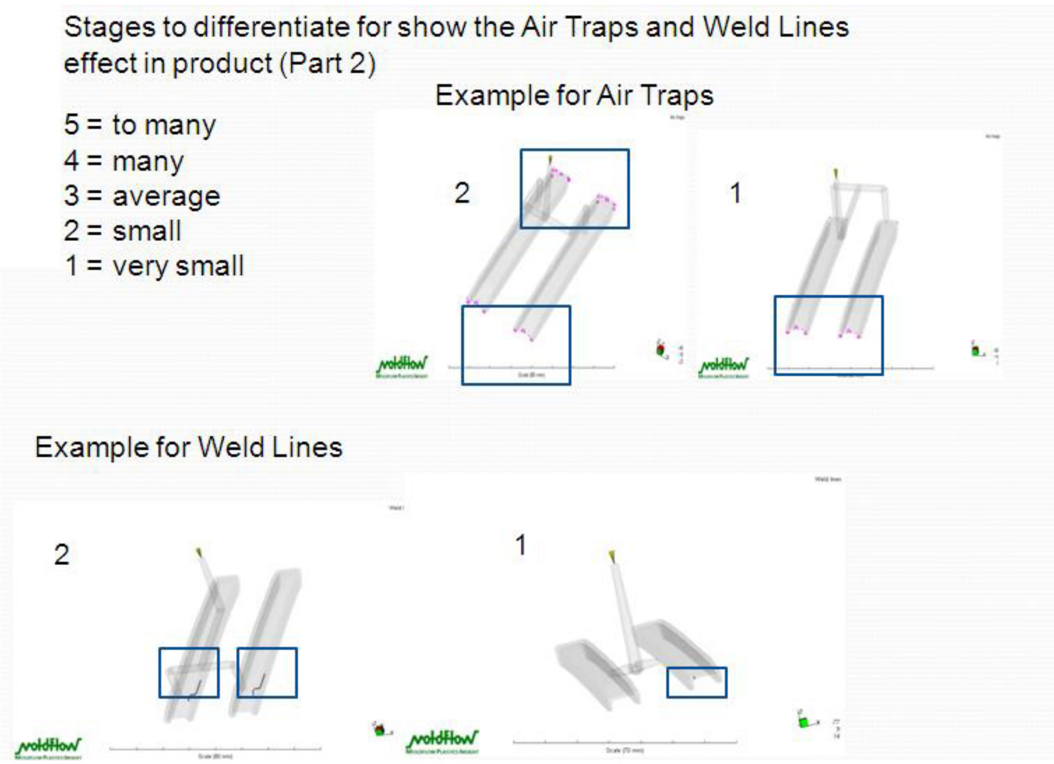


Fig 3.31: Air Traps and Weld Lines Stage for Part 2

3.6 Design the mold

When the process for analyze the gate location have done then we proceed with designing the mold with applying the best gate location on it. We also use the software Solidwork 2007 in order to design the mold. There also about to choosing a material for the mold making.

3.7 Results and Discussions

After the analysis done we can get the complete result and from that we can make the discussion about the project

.3.8 Conclusion

From this project we can know more about the Solidwork software and how to make the perfect design with it. From the Solidwork software, we have make the design for plastics injection molding process for two colors. Then, from this project we also can learn more about plastics injection molding process with two colors and its component. From Moldflow software we can learn about how to collect the data input for the experiment to make the analysis, learn how to run this software and how to collect the data output to make the discussion.

3.9 Divided work for PSM 1 and PSM 2

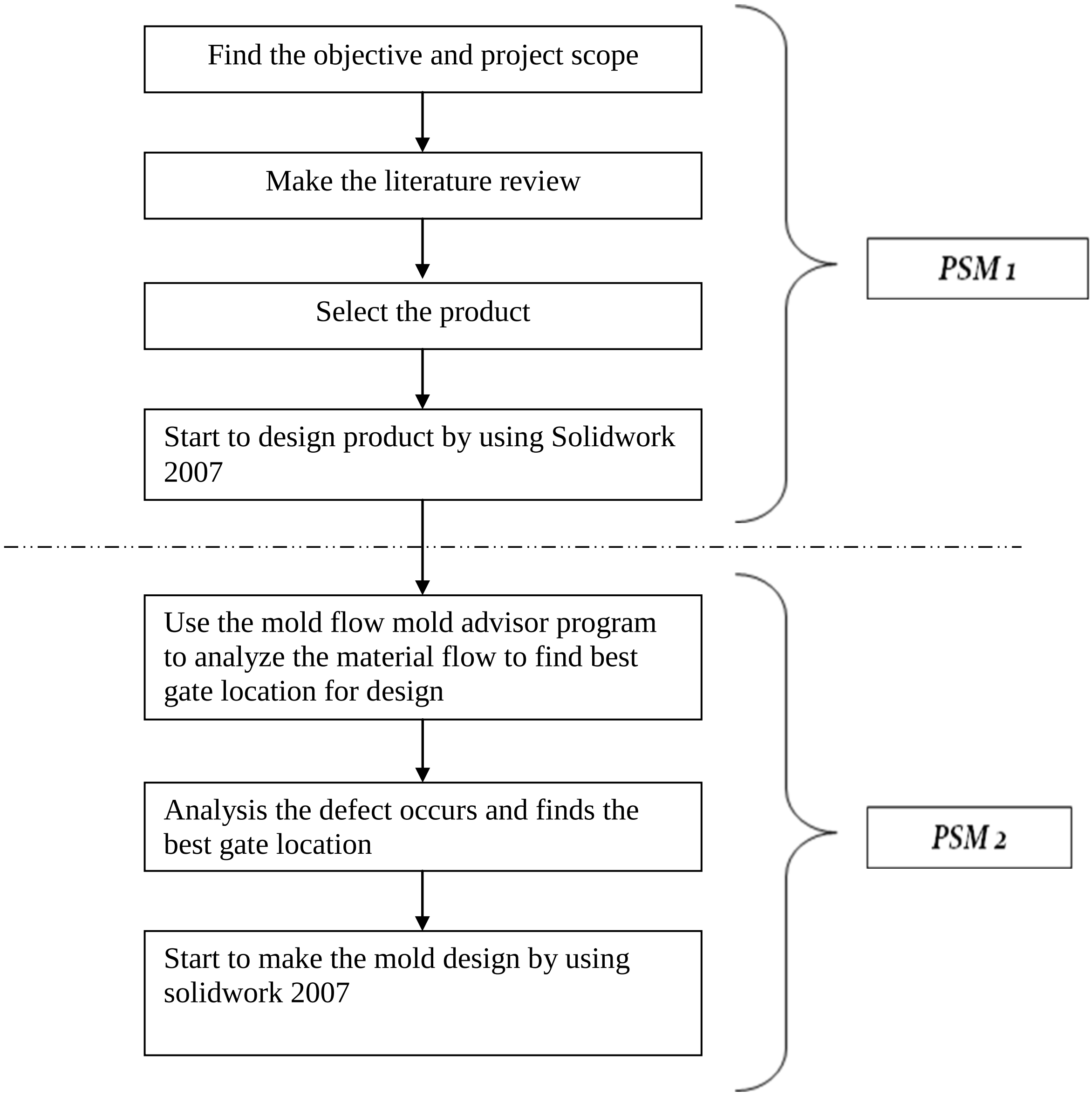


Fig 29: Work for PSM 1 and PSM 2

CHAPTER 4

RESULTS AND DISCUSSIONS

4.1 INTRODUCTION

In this chapter, the result of the analysis design for product for plastic injection molding with two colors and the mold design is discussed. The discussions of the analysis are based on the analysis and guidelines that have been discuss in the chapter two and chapter three.

4.1.1 3D Product Design

First stage in design the two color mold for plastic injection molding process is to select the product. After the product was selected then product been design with the same dimension as the real product by using Solidwork2007. The figure 4.1 and figure 4.2 show for trigonometry and top view. Then figure 4.3 and figure 4.4 shows the bottom and side view.

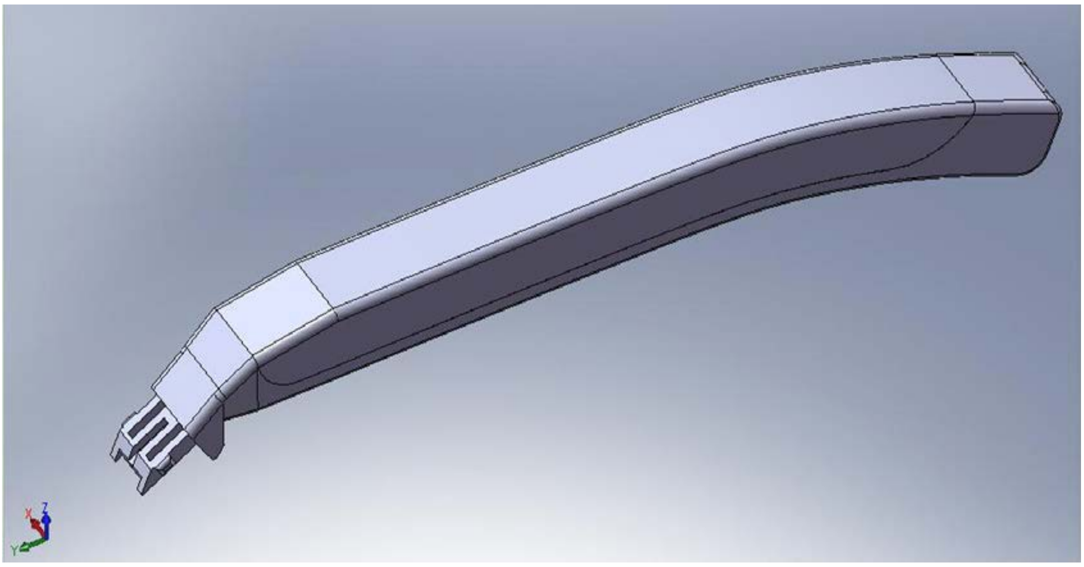
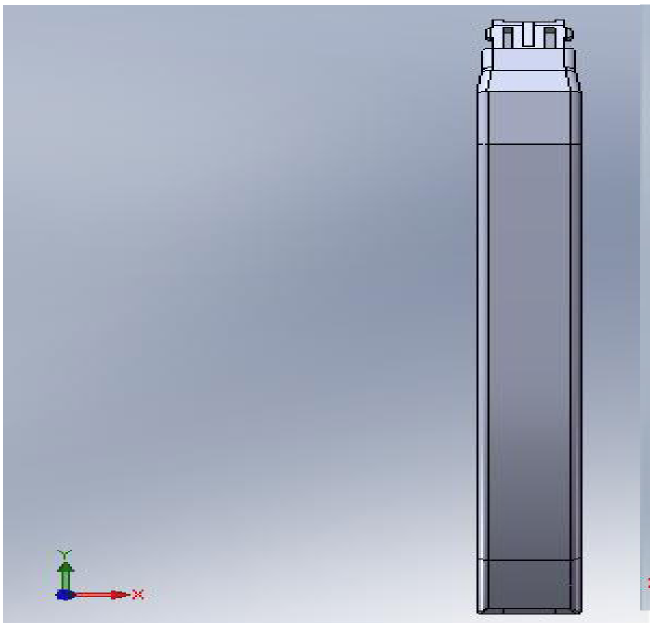


Figure 4.1: Bottom view



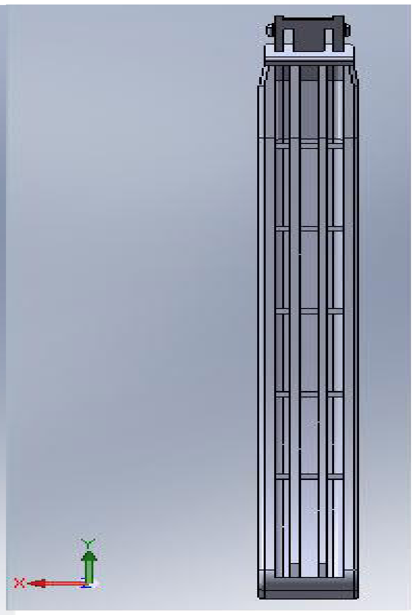


Figure 4.2: Top View **Figure 4.3:** Bottom View

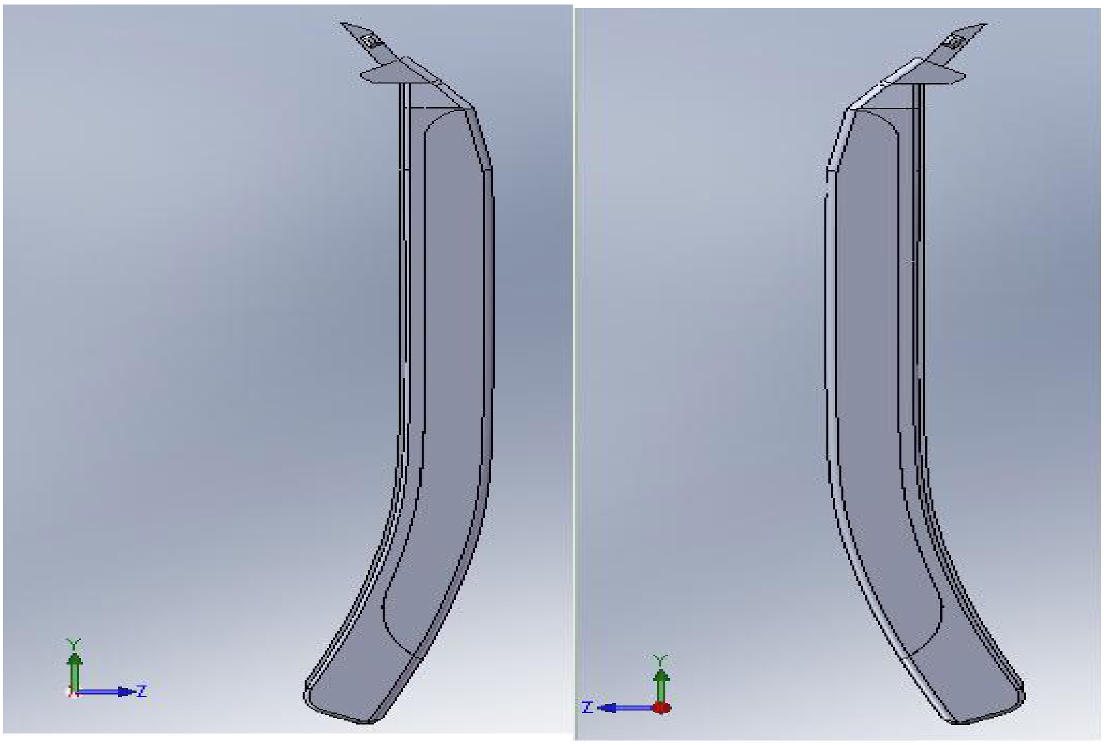


Figure 4.4: Side View

4.2 ANALYSIS RESULT

4.2.1 Gate Location for Part 1

The gate location will be analysis by using MoldFlow and then the data will be collected. The comparison data of the several gate locations shown in table 4.1.

Table 4.1: Comparison of Gate Location of part 1

Gate Location	Fill time (s)	Air traps	Weld lines	Sink index (%)	Time to Freeze (s)	Cycle time (s)	Volumetric shrinkage (%)
A	0.975	3	2	4.231	16.410	31.077	6.161
B	1.110	3	2	4.757	16.910	31.226	6.688
C	1.173	2	1	3.867	17.410	31.238	5.794
D	1.201	3	2	5.003	17.160	31.313	6.934
E	1.111	3	2	4.874	17.110	31.140	6.803
F	1.126	2	1	3.967	16.91	31.205	5.892

Table 4.2: Analysis of Gate Location of part 1

Gate Location	Fill time (s)	Air traps	Weld lines	Sink index (%)	Time to Freeze (s)	Cycle time (s)	Volumetric shrinkage (%)
A	0.975	3	2	4.231	16.410	31.077	6.161
B	1.110	3	2	4.757	16.910	31.226	6.688
C	1.173	2	1	3.867	17.410	31.238	5.794
D	1.201	3	2	5.003	17.160	31.313	6.934
E	1.111	3	2	4.874	17.110	31.140	6.803
F	1.126	2	1	3.967	16.91	31.205	5.892

From this table we can see that in yellow painted it show the best result to be considered. For gate location at C show that 4 best reading been counted and follow by gate location at F for two best reading and at gate location A have one. Therefore for best gate location choice was at C. Next are the reviews for best gate location.

4.2.2 Fill Time

The figure 4.5 shows that time taken to fill in to the mold. The colors that occur at the cavity represent the scale for time taken to fill the part. The maximum time taken to inject all the part is 1.173 second.

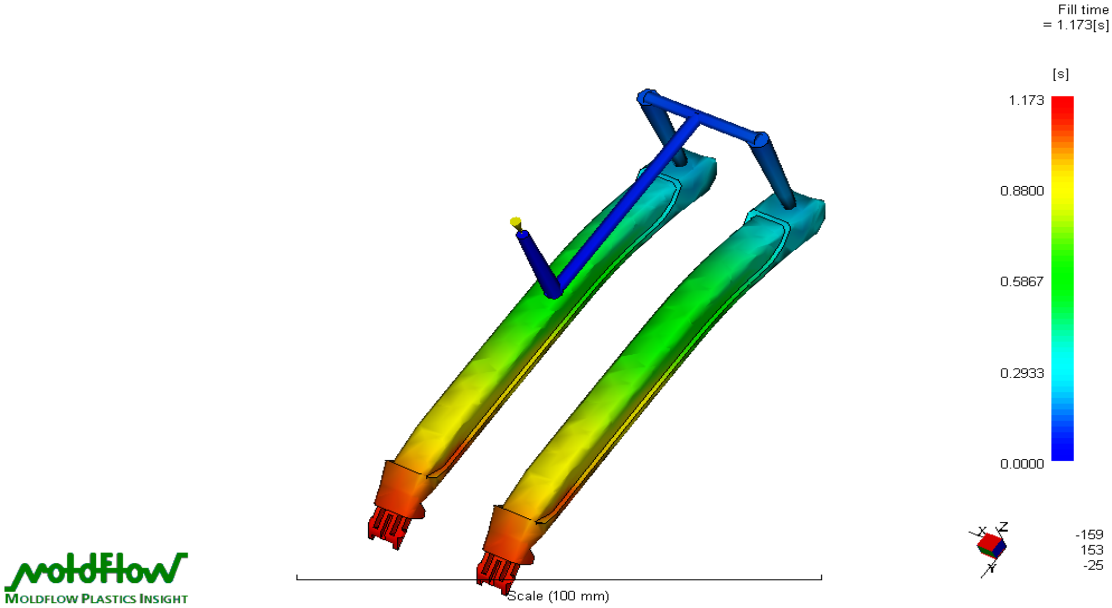


Figure 4.5: Fill time

4.2.3 Air Traps

From the Figure 4.6, we can see the air traps occurs at the product. The air traps is represent by the pink color in the figure. Normally this air traps occurs during the solidified process of plastic injection. The air traps will cause the burn mark to happen in the product. The burn mark is defined as discoloration which is usually black, brown or dark yellow/brown depending upon severity. Because of the quantity air traps for this gate location is about average and small in quantity therefore we put it in stage 2.

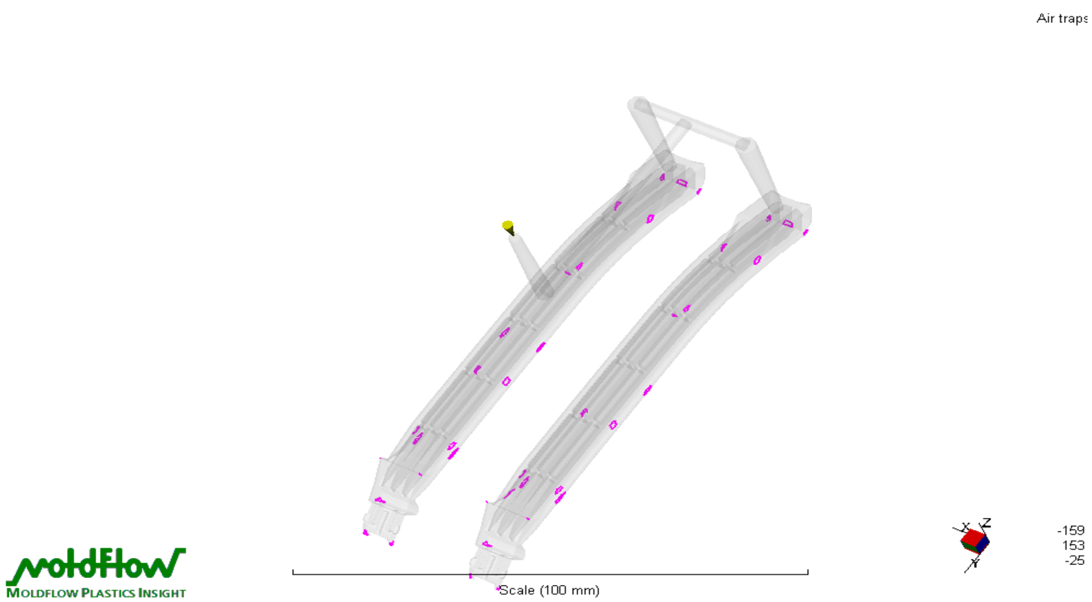


Figure 4.6: Air traps

4.2.4 Weld lines

Figure 4.7 show the welds a line which is occurs to the product. A weld line (also called a weld mark or a knit line) is formed when separate melt fronts traveling in opposite directions meet. A meld line occurs if two emerging melt fronts flow parallel to each other and create a bond between them. Weld and meld lines can be caused by holes or inserts in the part, multiple gates or variable wall thickness. From the quantity of it weld lines it be in the stages 1. It a quiet small quantity of weld lines and we can see it in figure 4.7.

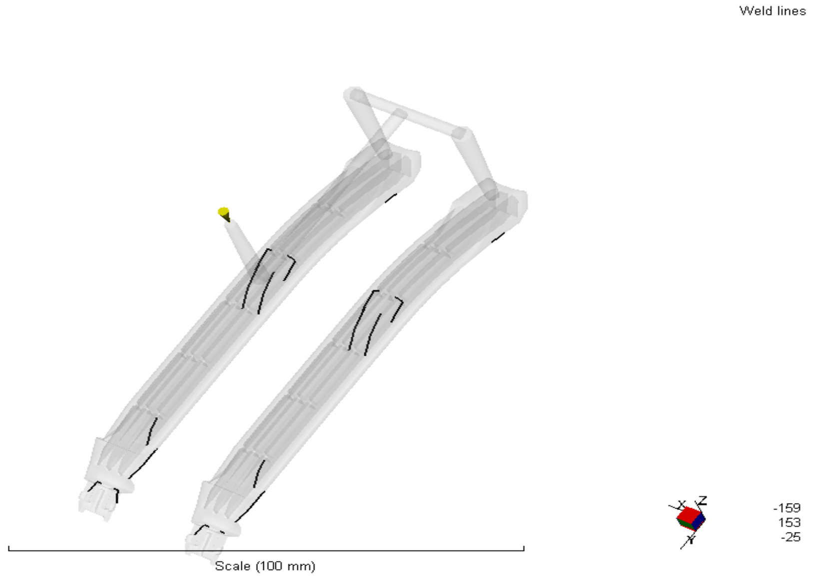


Figure 4.7: Weld lines

4.2.5 Times to Freeze

From the figure 4.8 it shows the time to freeze of the product. It shows about the time for molten plastic in the product to freeze. As we can see the time to freeze for this cavity or product is 17.41 second.

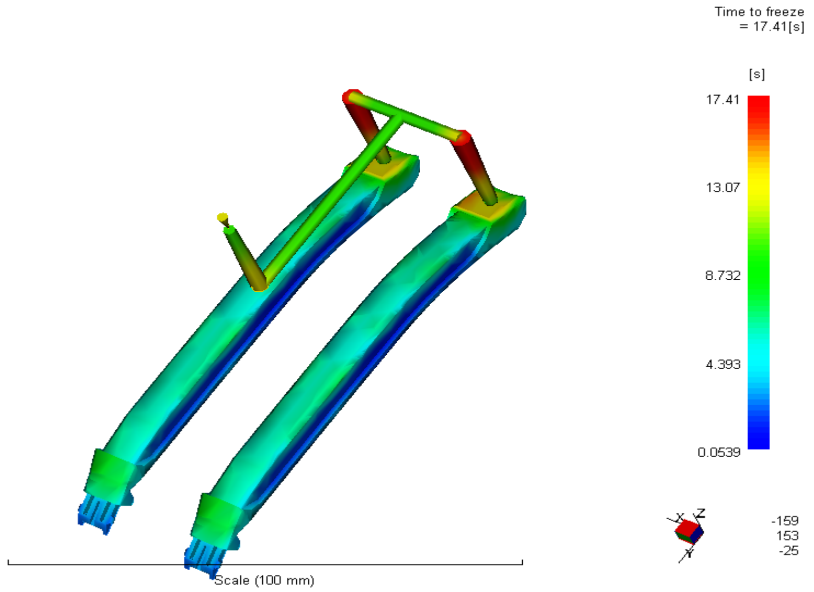


Figure 4.8: Times to freeze

4.2.6 Volumetric Shrinkage

The figure 4.9 show that the volumetric shrinkage for the product. Volumetric shrinkage is the contraction of polymer due to the change in temperature from melt temperature to ambient temperature. The volumetric shrinkage for gate location at C is 5.794% at 31.24 second.

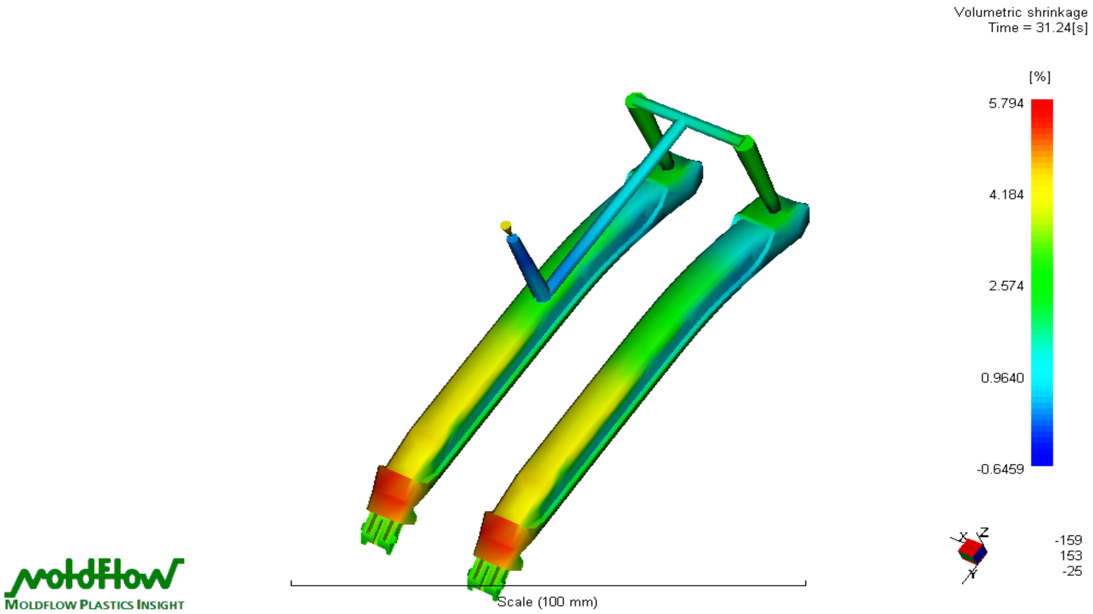


Figure 4.9: Volumetric Shrinkage

4.2.7 Mold Flow Analysis General Data

Table 4.3 showed the general property and parameter that was used in the MoldFlow analysis. This material used is Arylonitrile butadiene styrene (ABS) which is manufactured by Toray Industries Incorporated. This material and its manufacturer is very important before the analysis done by the software in order to get the exact result of the simulation. ABS has the highest impact resistance of all polymers. It takes color well. Integral metallics are possible. The acrylonitrile monomer is nasty stuff, almost as poisonous as cyanide. Once polymerized with styrene it becomes harmless. ABS can be recycled, and can be incinerated to recover the energy it contains. It been choose also because it the suggested material for this product. For more information about the analysis data can refer appendix A.

Table 4.3: Parameter that was used in the MoldFlow analysis.

Material manufacture	Toray Industries Incorporated
Material trade name	Toyolac 100
Melt Temperature	230ºC
Mold Temperature	50ºC
Injection Location	1
Max. Machine Injection Pressure	180.00MPa
Injection Time Selected	Automatic

4.2.8 Mold Flow Analysis Runner System

The runner system is very important in order to get the best quality of the product from the simulation. The function of the runner system is used to carry out the molten plastic during injection plastic process to the part. We must determine the best runner system to decrease the potential of defects. For the sprue, circular tapered is used and the diameter of the sprue is about 2.5mm inside diameter while the end diameter is 3mm. It length is 25mm

For the runners, circular shape 3mm diameter is used while for the top gates is used and its diameter is 3mm.. This is because the gate will have small diameter at the end (1mm) in order to avoid the molten plastic from flow back into runner system. The system is cold runner system and the system is very important to get quality of the product.

4.2.9 Gate Location for Part 2

The gate location for part 2 also will be analysis by using MoldFlow and then the data will be collected. The comparison data of the several gate locations will be shown in table 4.4.

Table 4.4: Comparison of Gate Location of part 2

Gate Location	Fill time (s)	Air traps	Weld lines	Sink index (%)	Time to Freeze (s)	Cycle time (s)	Volumetric shrinkage (%)
A	0.514	2	1	0	9.677	30.658	11.27
B	0.661	1	1	0	9.929	30.802	10.66
C	0.648	2	1	2.33E-6	9.928	30.794	10.90
D	0.482	2	2	3.65E-7	12.660	30.626	11.46
E	0.649	1	1	0	9.683	31.140	10.97
F	1.126	2	1	3.967	16.91	31.205	5.892

Table 4.5: Analysis of Gate Location of part 2

Gate Location	Fill time (s)	Air traps	Weld lines	Sink index (%)	Time to Freeze (s)	Cycle time (s)	Volumetric shrinkage (%)
A	0.514	2	1	0	9.677	30.658	11.27
B	0.661	1	1	0	9.929	30.802	10.66
C	0.648	2	1	2.33E-6	9.928	30.794	10.90
D	0.482	2	2	3.65E-7	12.660	30.626	11.46
E	0.649	1	1	0	9.683	31.140	10.97
F	1.126	2	1	3.967	16.91	31.205	5.892

From this table we can see that in yellow painted it show the best result to be considered. For gate location at B show that 5 best reading been counted. It stated it have fewest air traps and weld lines, no sink index and also have low percent volumetric shrinkage. Then second best gate location it at gate location at F for three best reading. Therefore for best gate location choice was at C. Next are the reviews for best gate location.

4.2.10 Fill Time

The figure 4.10 shows that time taken to fill in to the mold. The colors that occur at the cavity represent the scale for time taken to fill the part. The maximum time taken to inject all the part is 0.6614 second.

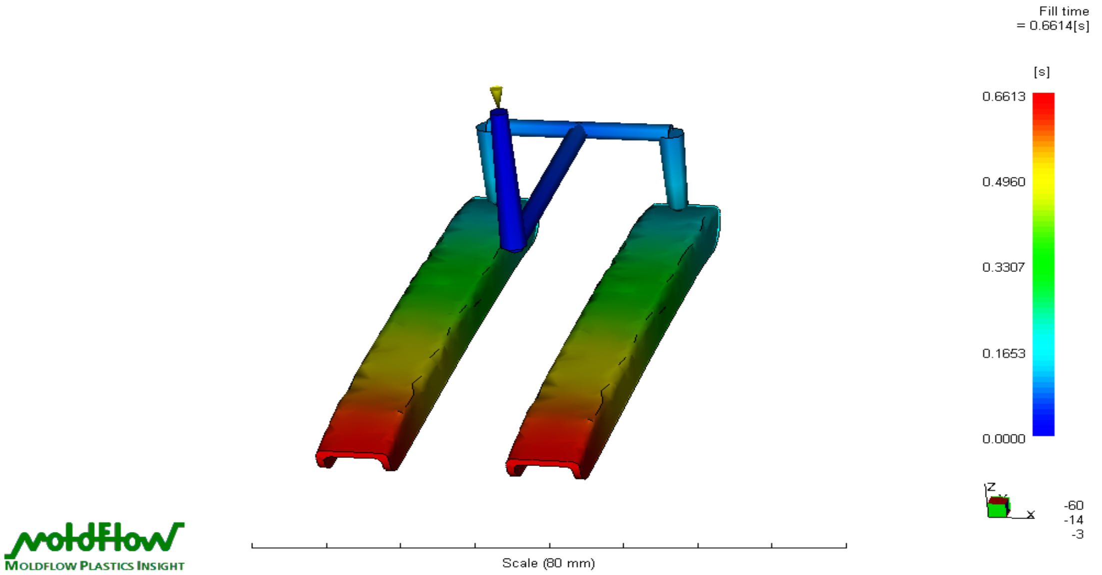


Figure 4.10: Fill time

4.2.11 Air Traps

From the Figure 4.11, we can see the air traps occurs at the product. The air traps also be represent by the pink color in the figure. Normally this air traps occurs during the solidified process of plastic injection. The air causes same as stated in air trap description in part 1. Because of the quantity air traps for this gate location is about average and small in quantity therefore we put it in stage 1.

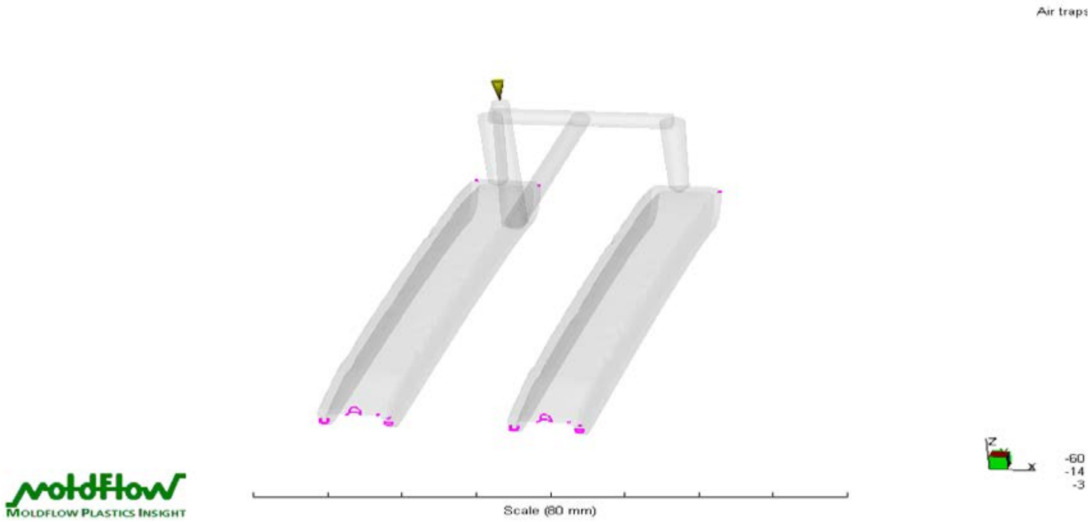


Figure 4.11: Air traps

4.2.12 Weld lines

Figure 4.12 show the welds a line which is occurs to the product. From the quantity of it weld lines it be in the stages 1. It a quiet small quantity of weld lines and we can see it in figure 4.12.

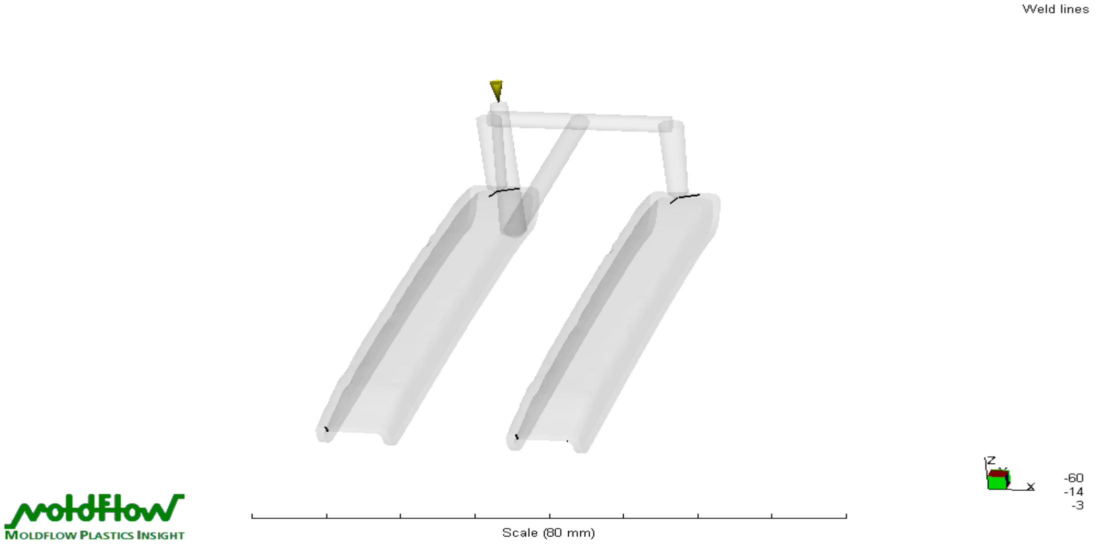


Figure 4.12: Welds lines

4.2.13 Times to Freeze

From the figure 4.13 it shows the time to freeze of the product. It shows about the time for molten plastic in the product to freeze. As we can see the time to freeze for this cavity or product is 9.929 second.

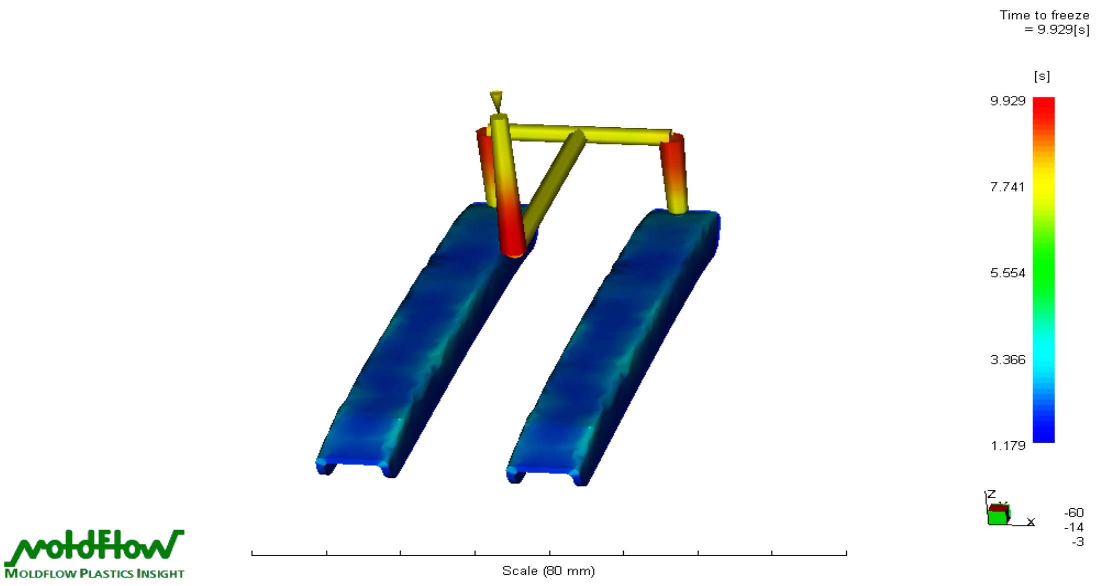


Figure 4.13: Times to freeze

4.2.14 Volumetric Shrinkage

The figure 4.14 show that the volumetric shrinkage for the product. Volumetric shrinkage is the contraction of polymer due to the change in temperature from melt temperature to ambient temperature. The volumetric shrinkage for gate location at C is 10.66% at 1.571 second.

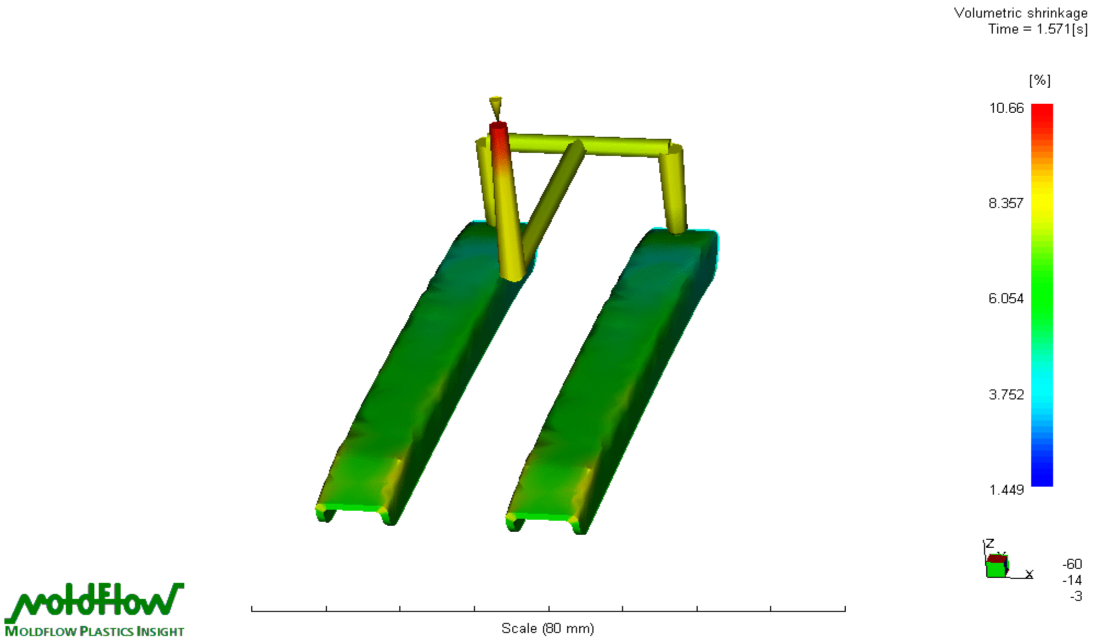


Figure 4.14 Volumetric Shrinkage

4.2.15 Mold Flow Analysis General Data

Table 4.6 showed the general property and parameter that was used in the MoldFlow analysis. This material used is Riteflex 547ZS which is manufactured by Ticona. This material and its manufacturer is very important before the analysis done by the software in order to get the exact result of the simulation. This material is a TPE (Elastomers) which is in thermoset family and it cannot be recycled. These materials also suggested material to make this shaver product. For more information about the analysis data can refer appendix B.

Table 4.6 Parameter that be used for part 2

Material manufacture	Ticona
Material trade name	Riteflex 547ZS
Melt Temperature	250ºC
Mold Temperature	50ºC
Injection Location	1
Max. Machine Injection Pressure	180.00 MPa
Injection Time Selected	Automatic

4.2.16 Mold Flow Analysis Runner System

We must determine the best runner system to decrease the potential of defects and therefore in this part 2 the runner system is still the same. For the sprue, circular tapered is used and the diameter of the sprue is about 2.5mm inside diameter while the end diameter is 3mm. It length is 25mm

For the runners, circular shape 3mm diameter is used while for the top gates is used and its diameter is 3mm. This is because the gate will have small diameter at the end (1mm) in order to avoid the molten plastic from flow back into runner system. The system is cold runner system and the system is very important to get quality of the product.

4.3 Discussions about Gate Location Analysis

In this chapter there will discuss about the best gate location based on the analysis been made by using Moldflow Software.

4.3.1 Gate Location

Based on table of Gate location for Part 1 we choose Gate location at C as a best gate location to be apply at the mold design because of it have lower stage in air trap and welds line just stage two for air traps and one for weld lines. It also have lowest sink index (3.867%) and in volumetric shrinkage (5.794%).

Then based on table of Gate location for Part 2 we have choose Gate location at B to be apply at the design. It have lower stage in air trap and welds line just stage one for air traps and one for weld lines. It also have lowest sink index (0%)and in volumetric shrinkage (10.66%), it also have Time to Freeze (9.929s).

Therefore after the finish the analyze we choose to apply the Gate location for part 1 at C and Gate location for part 2 at B to the design.

4.4 Mold Design

4.4.1 Mold Material

To design the mold the most suitable material are based on the product design like high surface finish or thermal conductivity or corrosion resistance, quantity to produce the product and the material for the product. The table 4.7 show that guide lines for selecting mold materials.

Table 4.7 Guidelines for selecting mold material.

1-3 Guidelines for selecting steel materials depends on types of plastic injection materials

Examples of selecting steel materials depends on types of plastic injection materials

	Types of Resins	Forming sample	expected specifications to resins	expected specification to steel materials	Recom-mended steel type
Thermo-plastic types of resins	PP ABS	Bumpers OA enclosures	Impact resistance	Creping control	S50C SCM440
	PS PMMA ABS	Lighting fixture Miscellaneous goods Cosmetic containers	design	Creping control Mirror finish	SKD61 prehaiden steel
	POM PA	Gears Shafts	Abrasion resistant	Abrasion resistant	SKD61 prehaiden steel
	PC PMMA	Lens Photo conductor	Transparency Optical transparency	Mirror finish easiness	SUS420J2 precipitation hardening types of steel
	PC PMMA	CD Discs DVD Discs	Optical Transparency Light retractivity	Mirror finish easiness Corrosion resistant	SUS420J2
	PVC	Gutter Pipes	Heat resistant	Corrosion resistant	SUS
	Fire resistance ABS	TV Cabinet Appliances part	Heat resistant	Corrosion resistant	SUS420J2prehaiden steel
	PBT-GF PA-GF	Camera Enclosure Electrical Equipment	resistance	Abrasion resistant	SKD11 prehaiden steel
	Magnetic powder containing PA	Printer rollers Sensor parts	Mold ability Magnetic characteristic	Non-magnetic Abrasion resistant	Nonmag-netic steel
	Mg forming	Computer enclosure Cellular phone enclosures	Heat resistance Light weight	Heat resistant Abrasion resistant	Nonmag-netic steel

From the table there are guidelines to choose the mold material based on product material. Based on the product we use materials like ABS and Elastomer. The Elastomer was combined of PP with other TPE (Thermo Plastic Elastomer) for part 2 product materials. Therefore the best mold material based on the table is S5OCSCM40. The materials was selected is the carbon steel.

4.4.2 Design mold for plastic injection molding for two colors.

The figure 4.15 show that the complete two colors mold design for this project. It also show the several mold components.

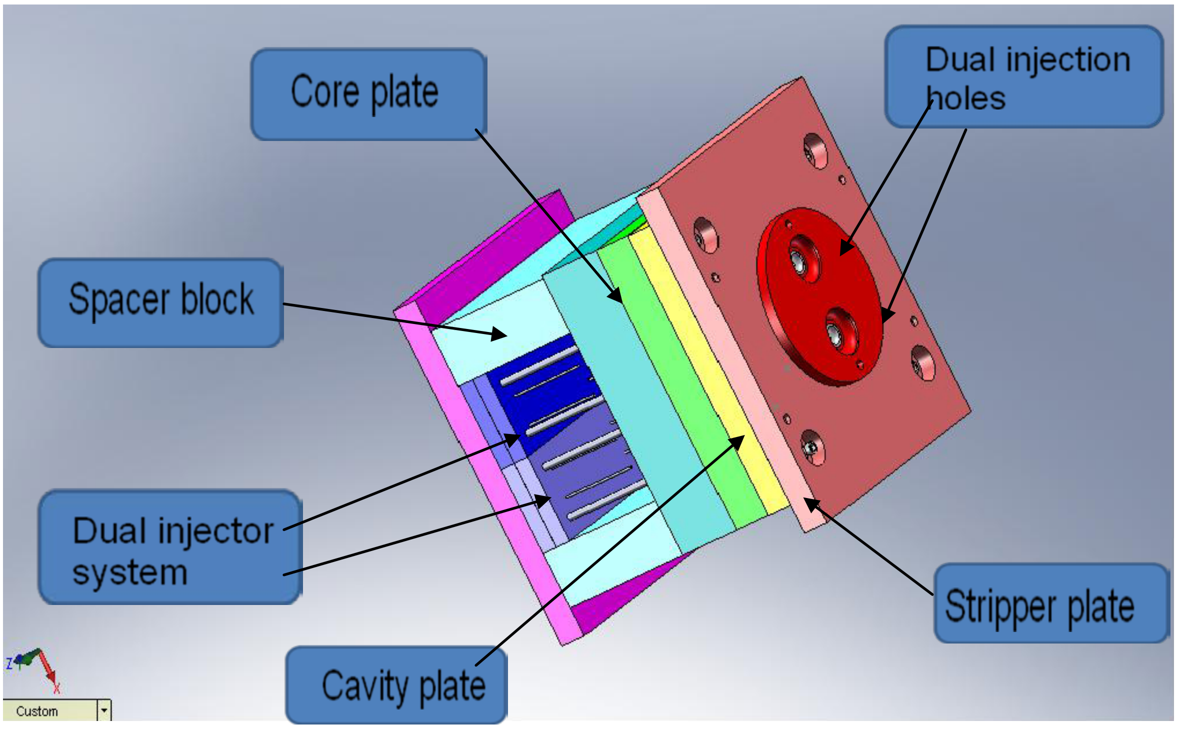


Figure 4.15: Complete Mold Design for plastic Injection molding two colors

4.4.2.1 Top Plate

This plate locate at nearest to the nozzle and at this plate where there are two hole for the nozzle from the injection molding machine for two color to inject the molten materials into the mold. At here it supports redesign locating ring which is to support two nozzles like at figure 4.16. This plate also supports sprue bushing which is it use to direct the flow of molten material into runner.

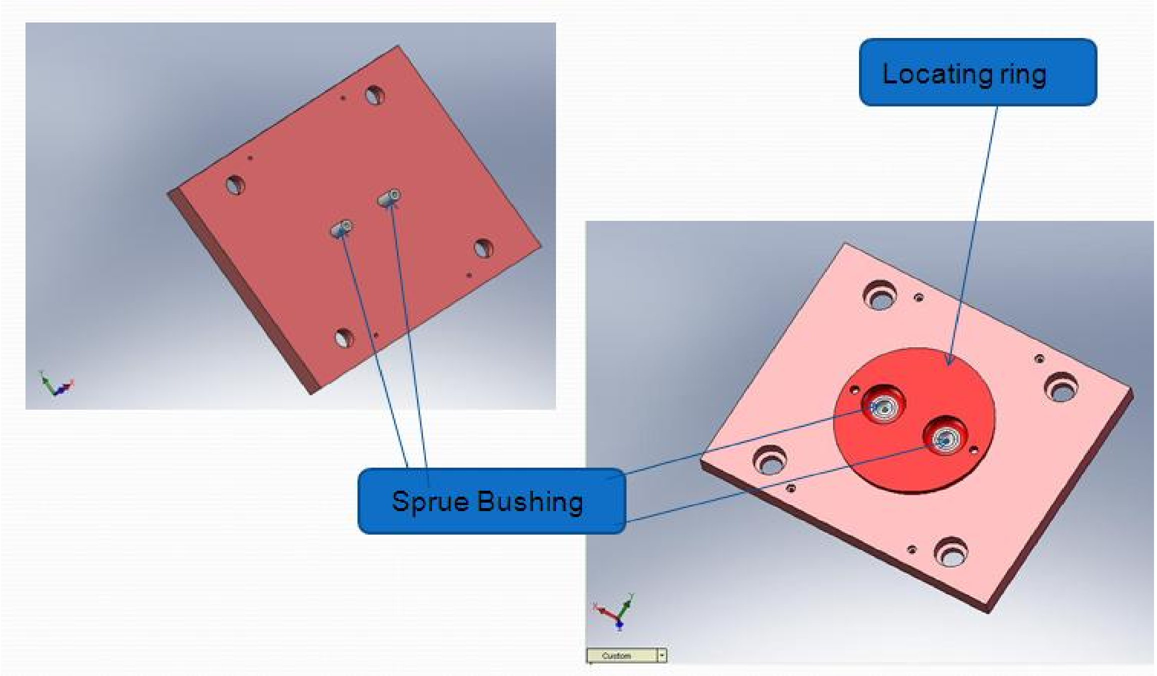


Figure 4.16: Top Plate with the Locating Ring and Sprue Bushing

4.4.2.2 Stripper Plate

For the design it have a stripper plate which is we know the stripper plate function is to remove the runner. There are half runner designs on the side of the stripper plate. The stripper plate design show at figure 4.17

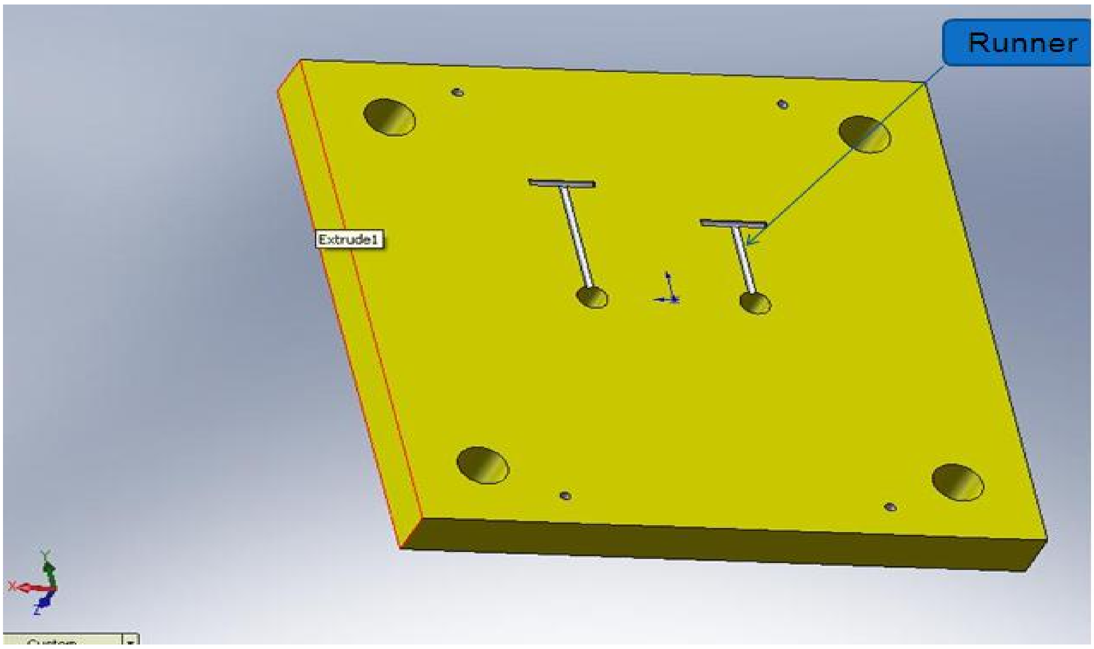


Figure 4.17: Stripper plate

4.4.2.3 Cavity Plate

Cavity plate is where the plate has section to put cavity insert in the plate show in figure 4.18. This cavity plate is between Core Plate and Stripper Plate.

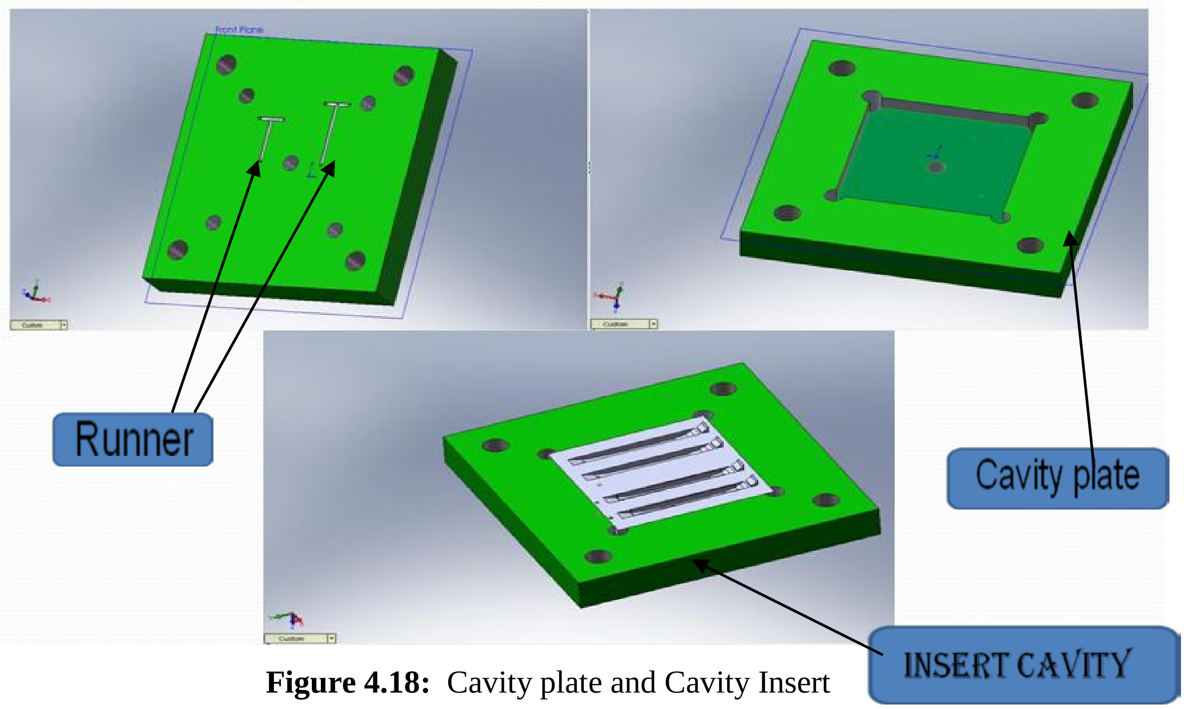


Figure 4.18: Cavity plate and Cavity Insert

4.4.2.4 Insert Cavity Plate

This is where the design been apply on a insert plate. The design is based on half of the product design. The figure 4.19 shown about inserts cavity plate.

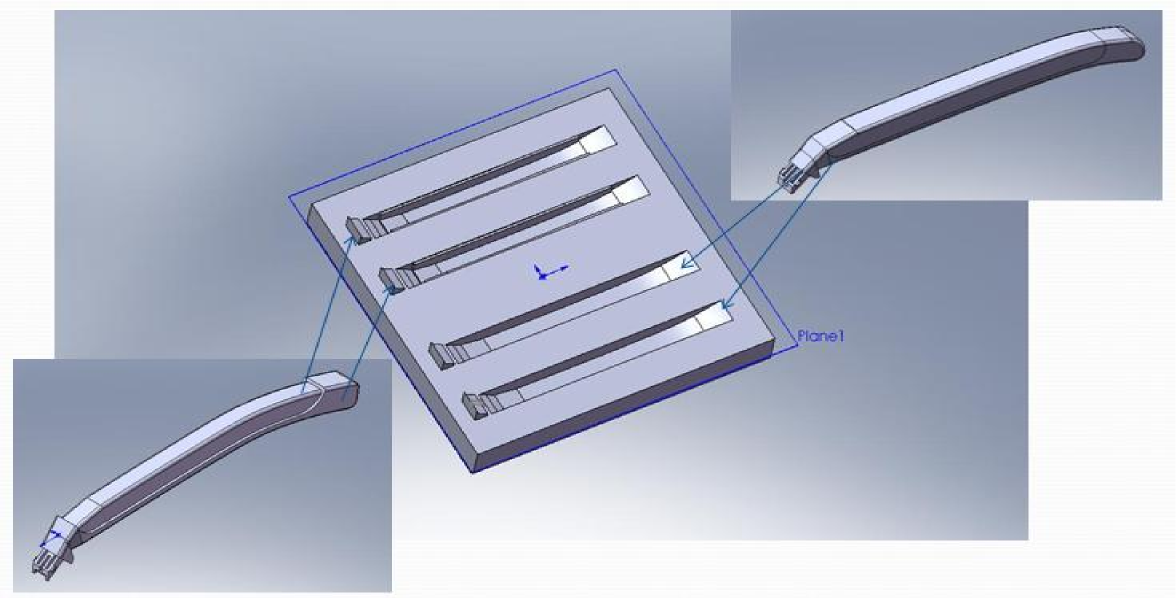


Figure 4.19: Insert Cavity plate

4.4.2.5 Core Plate

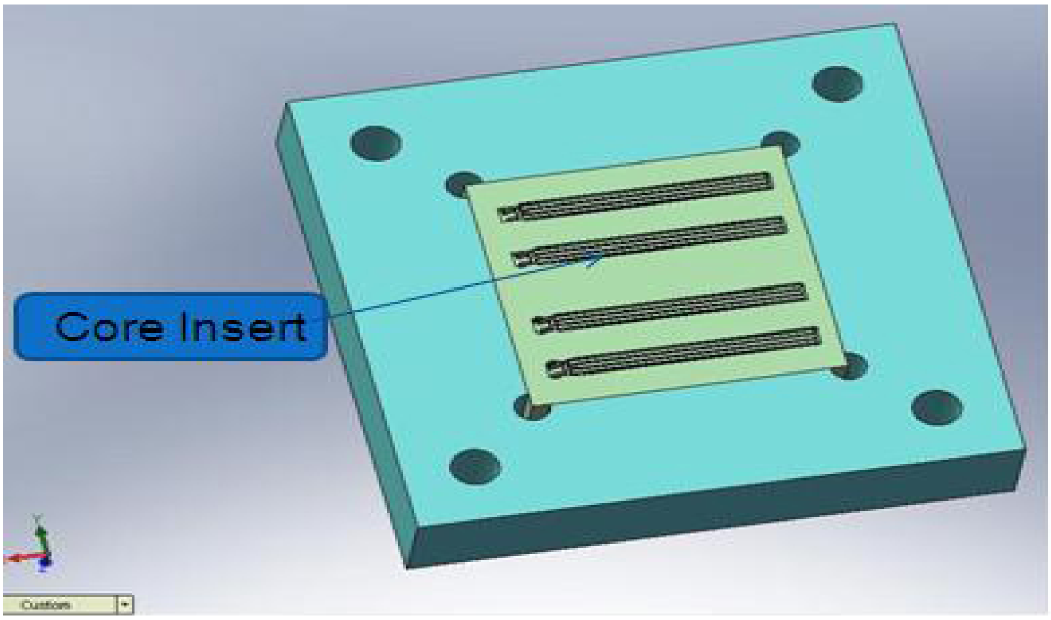


Figure 4.20: Core plate

4.4.2.6 Insert Core Plate

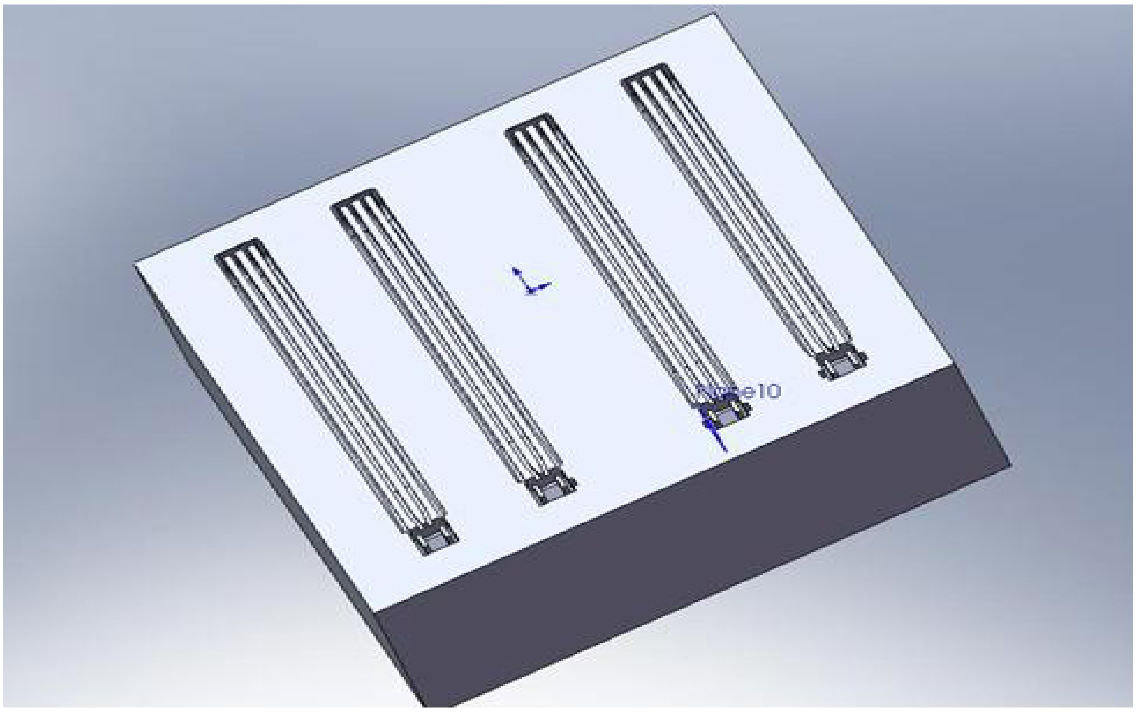


Figure 4.21: Insert Core plate

4.5 Mold Design Discussion

The design is complete and there are several part needed to be discussed. In this design, material mold usage is the Carbon Steel. The Mold is the three plate mold because of it have the stripper plate and have two opening mechanism. As you can see this injection molding design for two colors have been design with locating ring with dual injection hole. It also use two sprues bushing because of there are two injection holes needed.

From the gate location analysis we have find best gate location for this design for part 1 at C and part 2 at B and we apply it into the mold design. The half runner design on the stripper plate an the other half on the cavity plate. The upper part for the design been design at the Cavity Insert and the lower part been design on the Core Insert. The injector system for the mold design is using the dual injector system with the basic injector pin and the injector pin with the Sprue Puller. Refer to appendix C to see the 2D drawing for mold component.

CHAPTER 5

CONCLUSIONS AND RECOMMENDATIONS

5.1 CONCLUSIONS

As a conclusion of the project, for the design for plastic injection molding process for two colors was the mold design was successfully be design by using SolidWork2007. For the mold materials, the carbon steel was been choose which is take from recommended table which is make comparison between the product materials in choosing best mold material.

The FEA (Finite Element Analysis) software like the Moldflow MPI been use in this analysis in order to get the data needed to make the comparison. From this experiment it teaches how to master the FEA analysis by using moldflow software. This is very important when to find the best gate location which is want to apply it at the Mold Design.

The MoldFlow software use to analysis the best gate location which is occurs for part 1 at location C and for part 2 it occur at location B. These are the best gate location which will be used in designing two colors mold for injection molding process. When the best gate location been applied to the design, the secondary process which can increase cost or loss profit if there were problem occurs to the mold can be avoided. The parameter like the fill time, freeze time, air traps, welds lines and many more been considered in order to create differences between the gate locations.

The design for two color mold for plastic injection molding was designing by using the SolidWork2007. The best gate locations were applied to the design. The mold design for one color and two colors are different because of it have two hopper which is two ways for molten materials through to inside the cavity. Therefore the design for locating ring for one color and two colors are different which is for one color have one hole and for two colors have two holes to support the two ways molten materials into the mold that make this design needed two sprue bushing different for injection molding process for one color which is one hole for nozzle and sprue because only one material use for the process.

Then for the design mold it also have an ejector system which are eject the two cavity at the same the same time but there are four cavity on the mold. The two colors mold systems which are the core plate needed to rotate in order to make the second material flow into the cavity. When the second material injected and then the mold will open again and then cavity for one side will be ejected.

5.2 RECOMMENDATIONS

To improve this product the recommendations are needed to make the product complete. The recommendations are stated below:-

- Make other gate location analysis for two or more gates for the product, maybe the result for the defects are different.
- Use the data collected from moldFlow software for the product in order to find BEP (Break Even Point) which is use to the point where total revenue received equals total costs associated with the sale of the product

After doing the recommendation, the product will be better and more efficient.

REFERENCES:

6.0 REFERENCES FOR BOOKS

6.1.1 Single Author

Jay Shoemaker (2006). Moldflow Design Guide- A Resource for Plastics Engineer, Munich: Hanser Gardner, Inc.

Malloy R.A (1994). Plastic Part Design for Injection Molding, New York: Henser Publishers.

Rees, Herbert (2001). Injection Mold Design, Munich: Hanser Gardner, Inc.

Rees, Herbert (2002). Mold Engineering, Munich: Hanser Gardner, Inc.

6.1.2 Co-authors

Beaumont J.P / Nagel R / Sherman R. (2002). Successful Injection Molding-Process, Design and Simulation, Munich: Hanser Gardner, Inc.

Menges G., Michaeli W., and Mohren P. (2000). How to make injection molds, Munich: Hanser Gardner, Inc.

6.1.3 REFERENCES FOR JOURNALS

Babur Ozcelik and Ibrahim Sonat (2007). Warpage and structural analysis of thin shell plastic in the plastic injection molding, *Science Direct* .Retrieved on April 15, 2009, from *www.sciencedirect.com/*

Katashi Aoki (1974). *U.S. Patent No. US3807920 A1.*Retrieved on March 16, 2009, from http://www.freepatentsonline.com/complete.html

H. Yokoi, N. Masuda, and H. Mitsuda (2002). Visualation analysis of flow front behavior during filling process of injection mold cavity by two axis tracking system, *Science Direct* .Retrieved on April 15, 2009, from *www.sciencedirect.com/*

R Spina (2004). *Injection molding of automotive component, comparison between hot runner systems for case study*, Retrieved on April 15, 2009, from www.sciencedirect.com/

Yuehua Gao and Xicheng Wang (2007). Surrogate-based process optimization for reducing warpage in injection molding, *Science Direct* .Retrieved on April 15, 2009, from www.sciencedirect.com/

Xuejun Li, Satyandra K. Gupta (2003). Geometric algorithms for automated design of rotary-platen multi-shot molds,.Retrieved on April 15, 2004, from *www.*elsevier*.com/*

APPENDIX A

Result Summary for MoldFlow for Part 1

```
Material data :

   Polymer   : Toyolac 100 : Toray Industries Incorporated
   ---------
   PVT Model:    2-domain modified Tait
             coefficients: b5 =    365.4700 K
                          b6 =  2.3270E-07 K/Pa
                          Liquid phase        Solid phase
                          -------------------------------
                          b1m =      0.0010  b1s =      0.0010 m^3/kg
                          b2m =  6.1380E-07  b2s =  3.0270E-07 m^3/kg-K
                          b3m =  2.0368E+08  b3s =  2.5453E+08 Pa
                          b4m =      0.0053  b4s =      0.0044 1/K
                                        b7 =      0.0000 m^3/kg
                                        b8 =      0.0000 1/K
                                        b9 =      0.0000 1/Pa

   Specific heat (Cp)                        =   2400.0000 J/kg-C

   Thermal conductivity                      =      0.1800 W/m-C

   Viscosity model:               Cross-WLF
                         coefficients: n    =      0.3206
                                   TAUS =  5.1946E+04 Pa
                                   D1   =  5.5400E+12 Pa-s
                                   D2   =    373.1500 K
                                   D3   =      0.0000 K/Pa
                                   A1   =     28.5240
                                   A2T  =     51.6000 K

   Transition temperature                    =    100.0000 C

   Mechanical properties data:               E1  =   2240.0000 MPa
                                   E2  =   2240.0000 MPa
                                   v12 =      0.3920
                                   v23 =      0.3920
                                   G12 =    805.0000 MPa

   Transversely isotropic coefficent of
     thermal expansion (CTE) data:           Alpha1  =  8.0000E-05 1/C
                                 Alpha2  =  8.0000E-05 1/C

   Residual stress model without CRIMS

--------------------------------------------------------------------------

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 = 0.8665 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 = 82.4 %
Total number of nodes = 2149
Total number of injection location nodes = 1
The injection location node labels are:
2120
Total number of elements = 4272
Number of part elements = 4236
Number of sprue/runner/gate elements = 36
Number of channel elements = 0
Number of connector elements = 0
Average aspect ratio of triangle elements = 3.6165
Maximum aspect ratio of triangle elements = 73.0744
Minimum aspect ratio of triangle elements = 1.1859
Total volume = 9.5973 cm^3
Volume filled initially = 0.0000 cm^3
Volume to be filled = 9.5973 cm^3
Sprue/runner/gate volume to be filled = 0.8782 cm^3
Total projected area = 23.4846 cm^2

Filling phase results summary :

Maximum injection pressure (at 0.913 s) = 68.6049 MPa

End of filling phase results summary :

Time at the end of filling = 0.9794 s
Total weight = 9.8612 g
Maximum Clamp force - during filling = 6.1740 tonne
Recommended ram speed profile (rel):

% stroke	% speed
0.0000	17.5588
8.8990	17.5588
20.0000	52.5730
30.0000	61.2818
40.0000	78.2889

50.0000	91.0520
60.0000	89.7009
70.0000	100.0000
80.0000	91.2168
90.0000	54.4517
100.0000	14.8815

Melt front is entirely in the cavity at % fill = 8.8990 %

APPENDIX B

Result Summary for MoldFlow for Part 2

Material data :

```
Polymer   : Riteflex 547ZS : Ticona
---------
PVT Model:    2-domain modified Tait
           coefficients: b5 =    506.6500 K
                       b6 =  1.7250E-07 K/Pa
                       Liquid phase        Solid phase
                       -------------------------------
                       b1m =      0.0009  b1s =      0.0009 m^3/kg
                       b2m =  8.4350E-07  b2s =  4.0900E-07 m^3/kg-K
                       b3m =  1.0491E+08  b3s =  1.2317E+08 Pa
                       b4m =      0.0052  b4s =      0.0043 1/K
                                        b7 =  6.6450E-05 m^3/kg
                                        b8 =      0.0355 1/K
                                        b9 =  5.9320E-09 1/Pa

Specific heat (Cp)                          =   2030.0000 J/kg-C

Thermal conductivity                        =      0.1600 W/m-C

Viscosity model:               Cross-WLF
                         coefficients: n    =      0.4211
                                 TAUS =  3.6548E+04 Pa
                                 D1   =  2.6900E+11 Pa-s
                                 D2   =    323.1500 K
                                 D3   =      0.0000 K/Pa
                                 A1   =     25.1030
                                 A2T  =     51.6000 K

Transition temperature                      =    200.0000 C

Mechanical properties data:                E1  =      7.7000 MPa
                                 E2  =      7.7000 MPa
                                 v12 =      0.3800
                                 v23 =      0.3800
                                 G12 =      2.7900 MPa

Transversely isotropic coefficent of
  thermal expansion (CTE) data:          Alpha1  =      0.0002 1/C
                               Alpha2  =      0.0002 1/C

Residual stress model without CRIMS
```

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                              =      0.6053 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 = 250.0000 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 = 86.2 %
Total number of nodes = 2185
Total number of injection location nodes = 1
The injection location node labels are:
2146
Total number of elements = 4322
Number of part elements = 4280
Number of sprue/runner/gate elements = 42
Number of channel elements = 0
Number of connector elements = 0
Average aspect ratio of triangle elements = 3.3136
Maximum aspect ratio of triangle elements = 53.1678
Minimum aspect ratio of triangle elements = 1.1648
Total volume = 4.6678 cm^3
Volume filled initially = 0.0000 cm^3
Volume to be filled = 4.6678 cm^3
Sprue/runner/gate volume to be filled = 0.9184 cm^3
Total projected area = 19.4279 cm^2

Filling phase results summary :

Maximum injection pressure (at 0.643 s) = 70.9364 MPa

End of filling phase results summary :

Time at the end of filling = 0.6618 s
Total weight = 5.3874 g
Maximum Clamp force - during filling = 4.6399 tonne
Recommended ram speed profile (rel):

% stroke	% speed
0.0000	51.8890
10.0000	70.8487
18.9433	70.8487
30.0000	100.0000
40.0000	87.7317
50.0000	92.6231
60.0000	90.6793
70.0000	93.0322
80.0000	88.6561
90.0000	94.0113
100.0000	81.7568

Melt front is entirely in the cavity at % fill = 18.9433 %

APPENDIX C

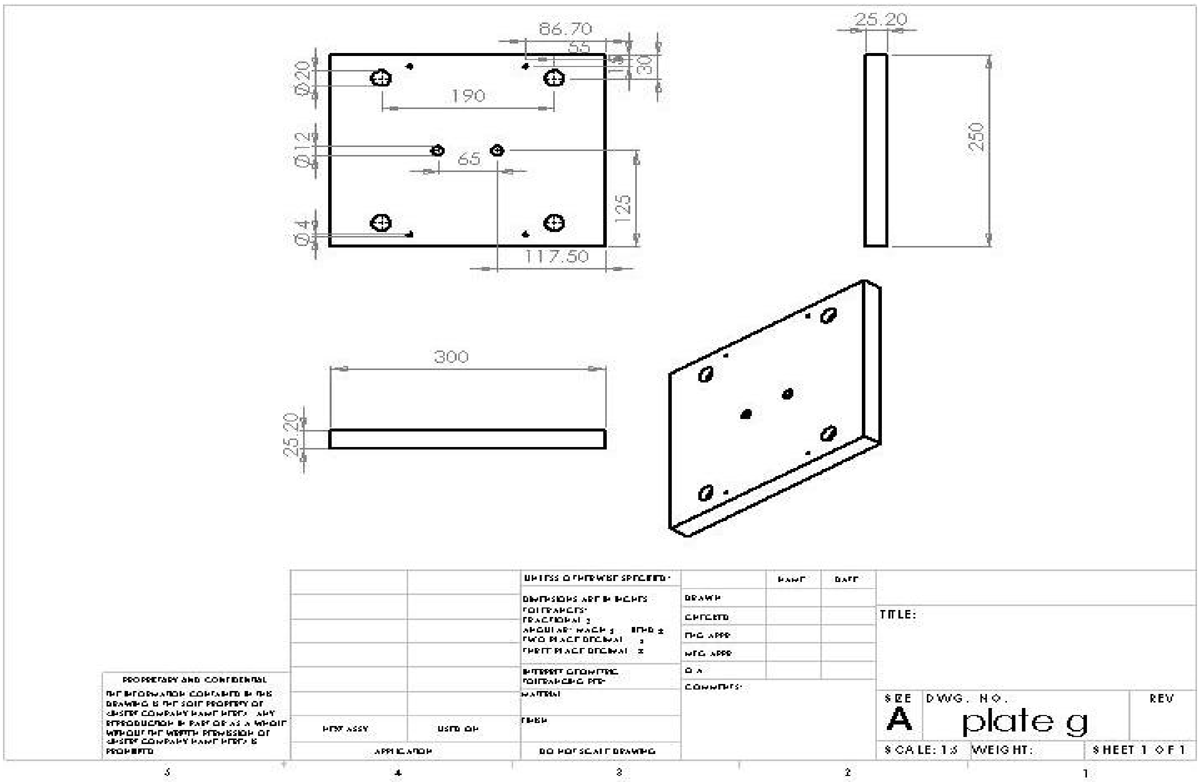


Figure C1: 2D Design for Top Plate

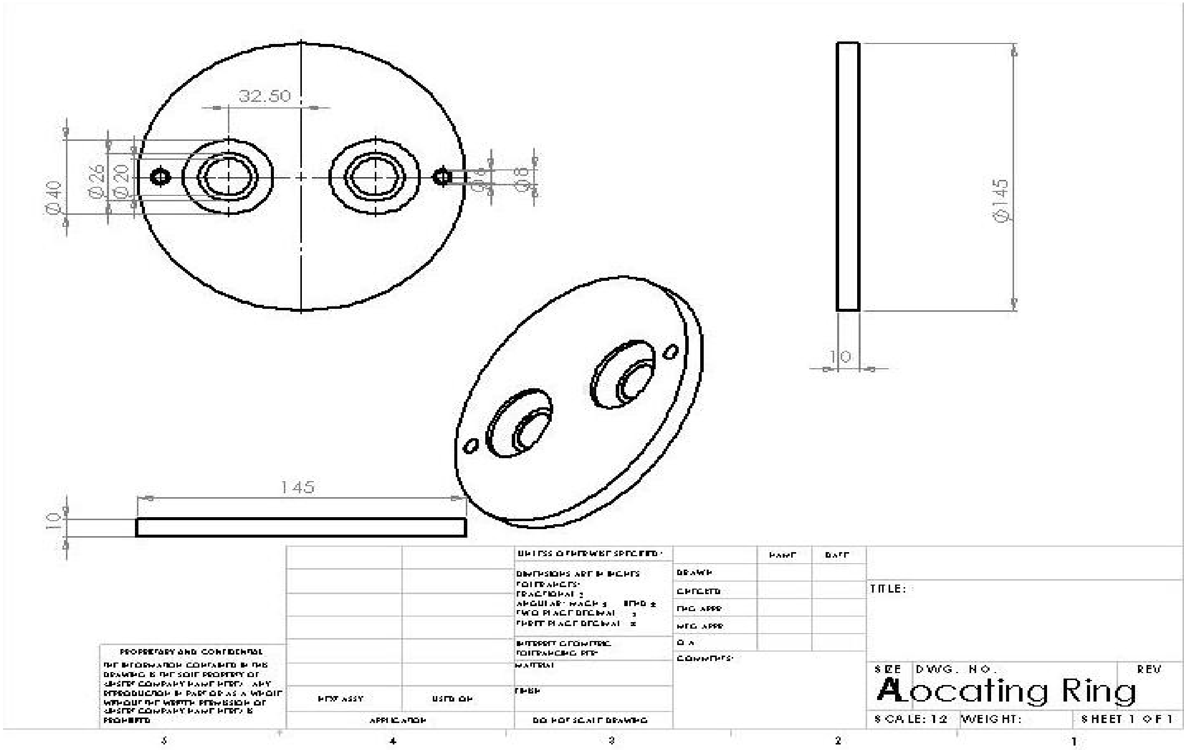


Figure C2: 2D Design for Locating Ring

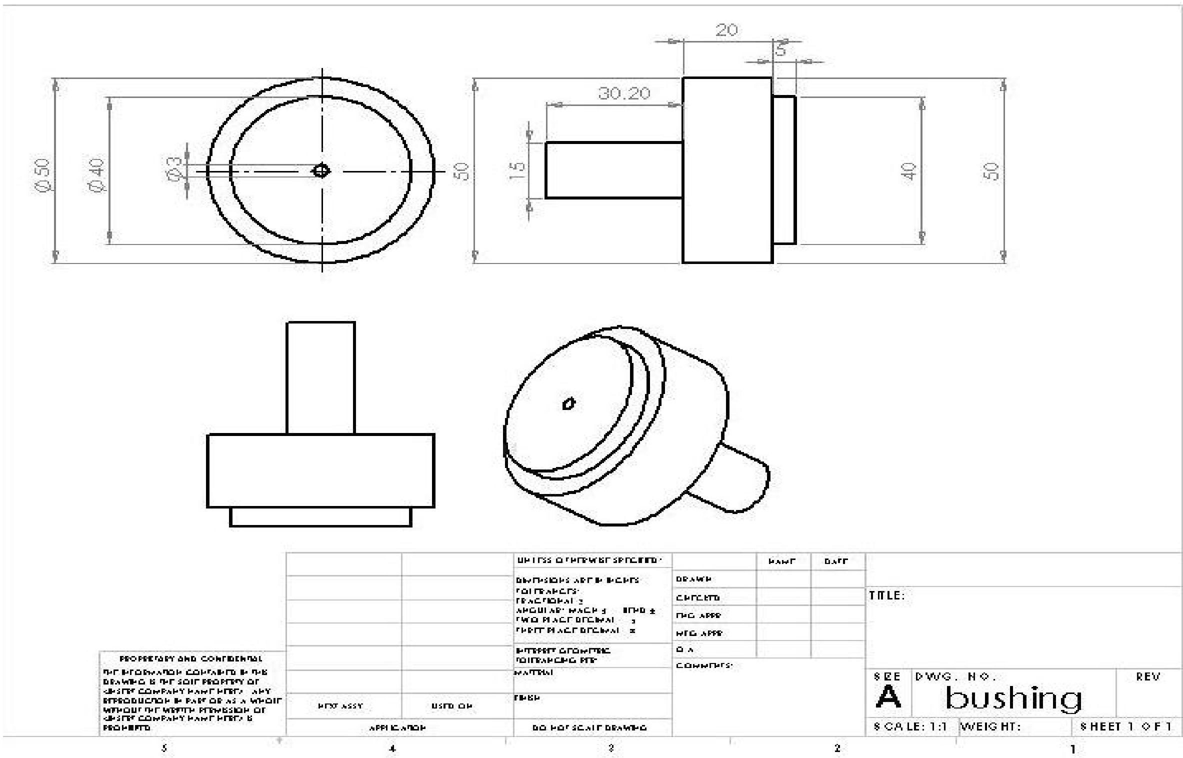


Figure C3: 2D Design of Screw Bushing

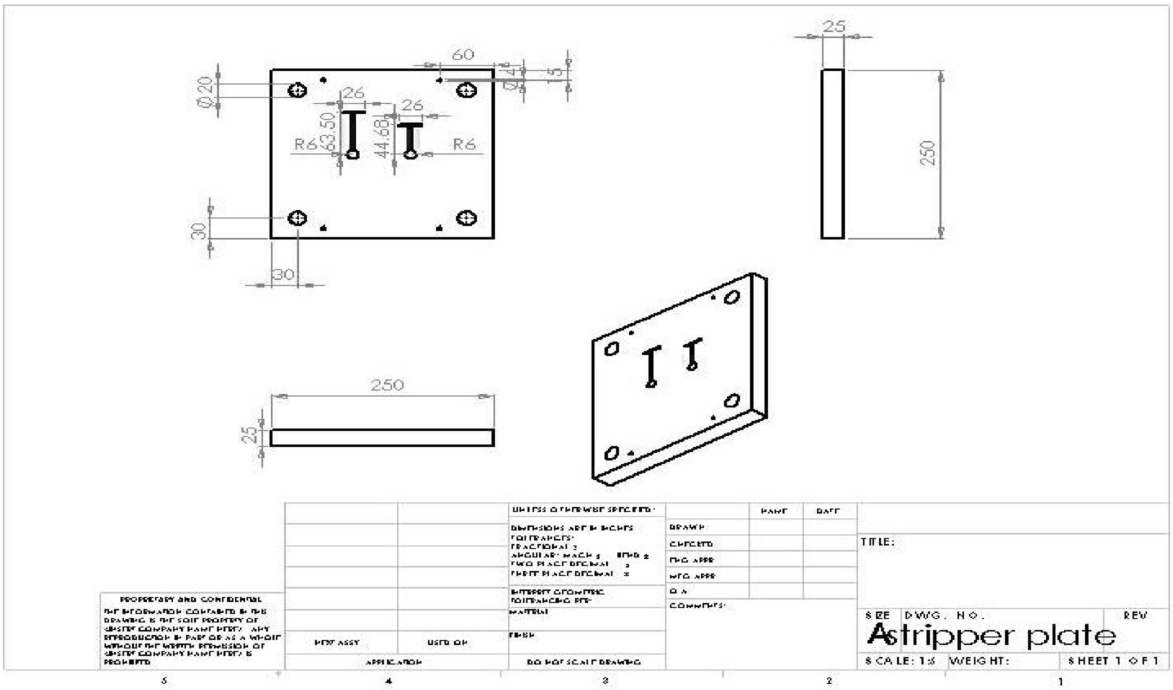


Figure C4: 2D Drawing for Stripper plate

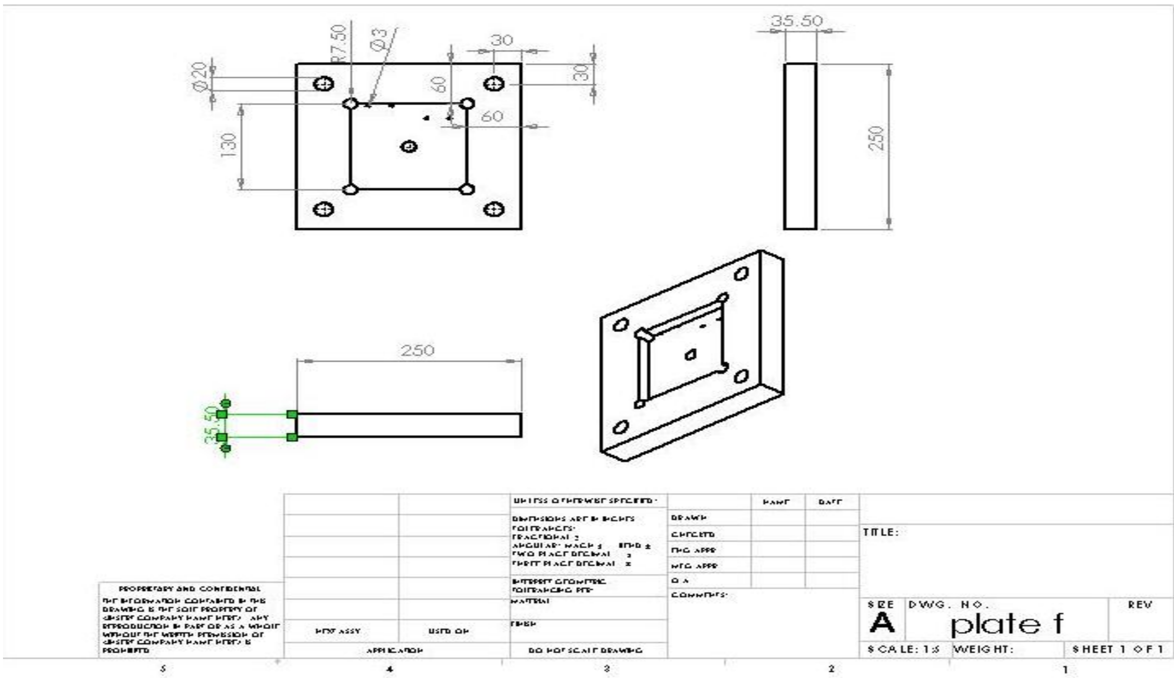


Figure C5: 2D design for Cavity plate (front).

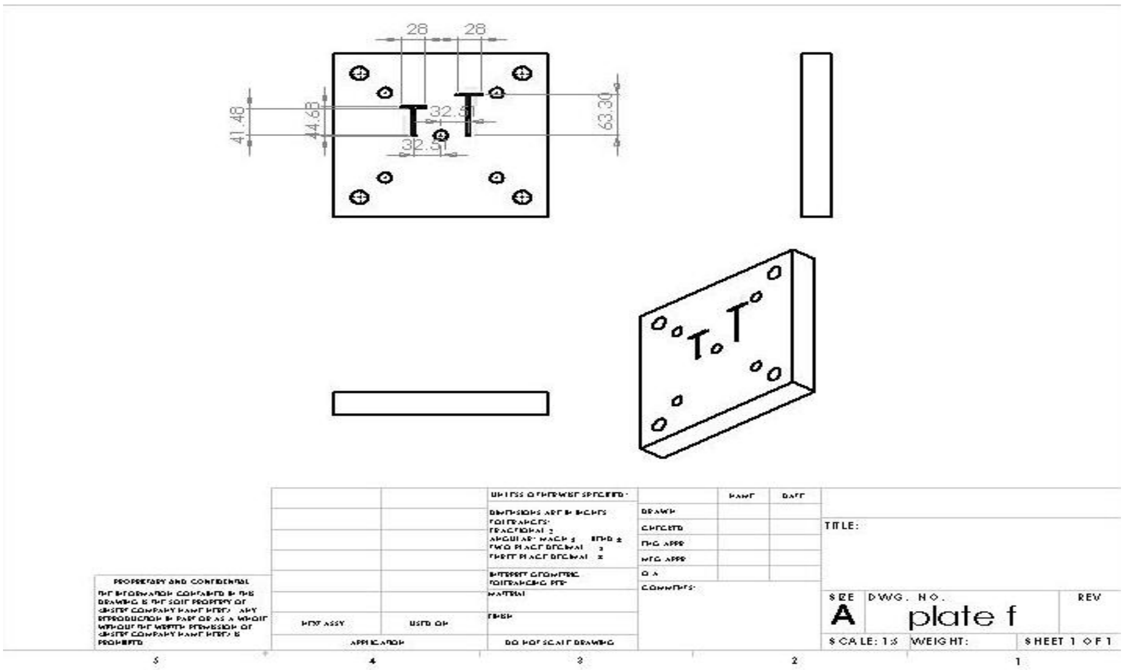


Figure C6: 2D design for Cavity plate (back).

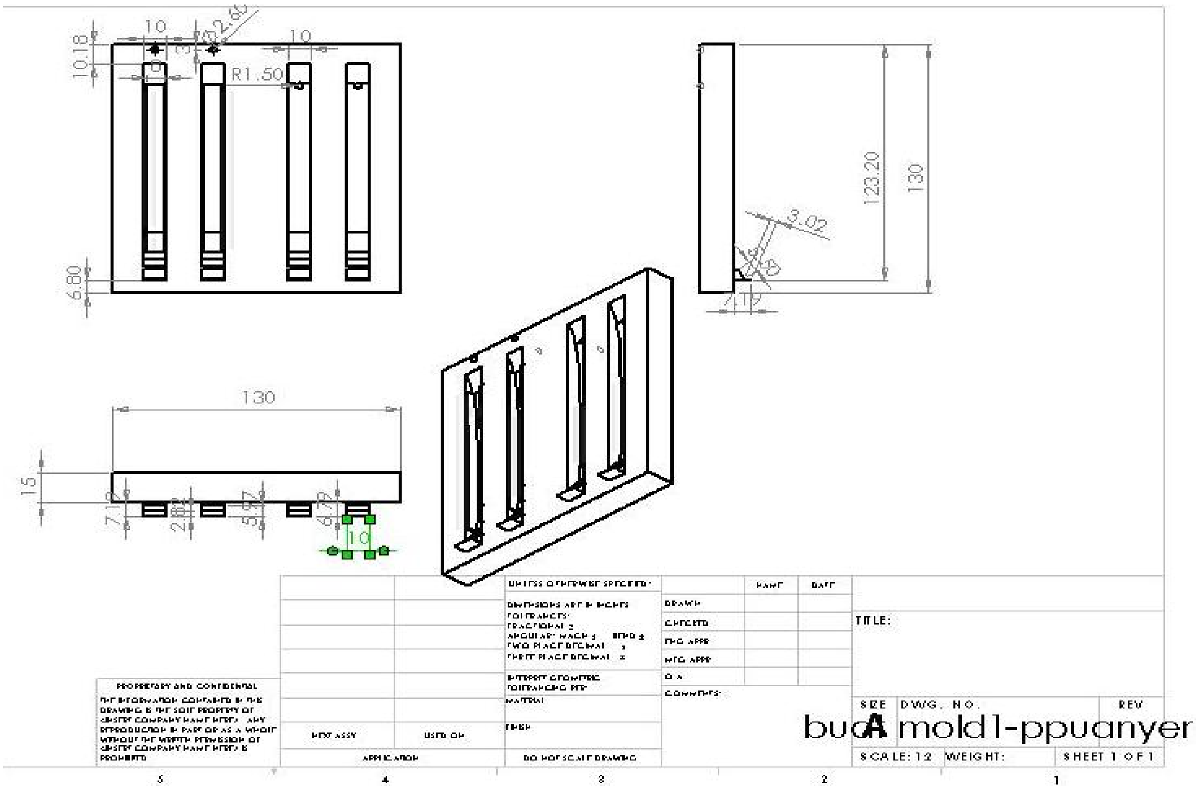


Figure C7: 2D Design for Insert Cavity plate

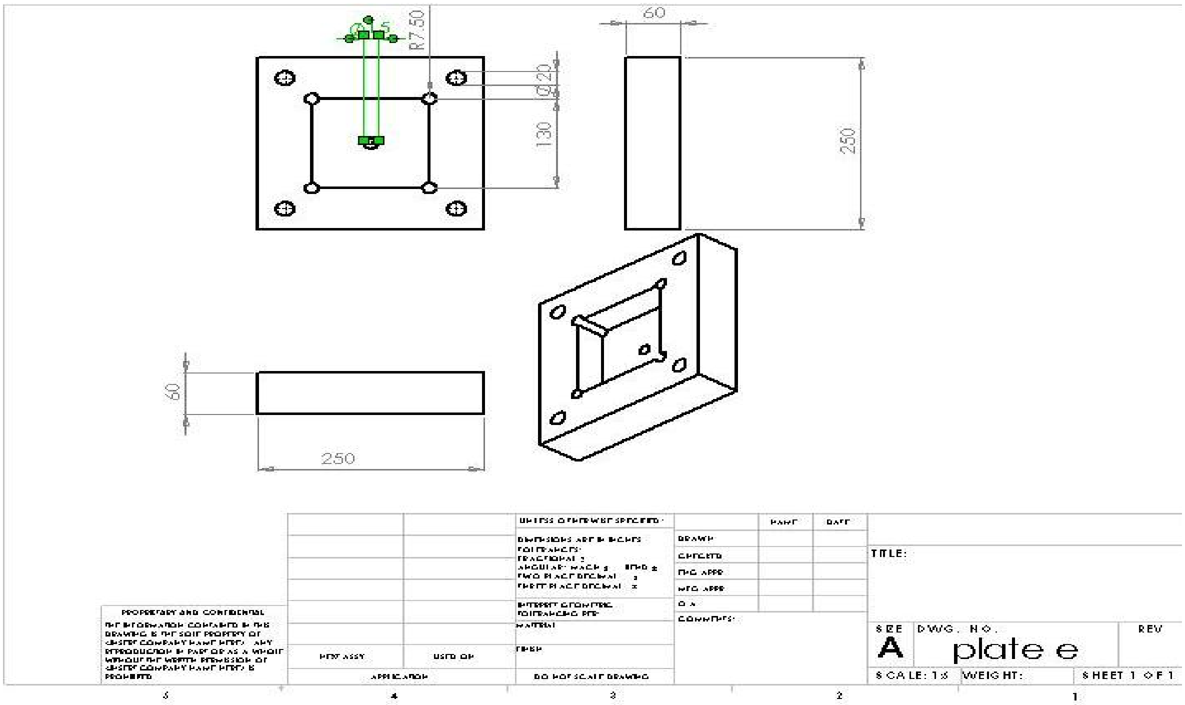


Figure C8: 2D Design for Core Plate

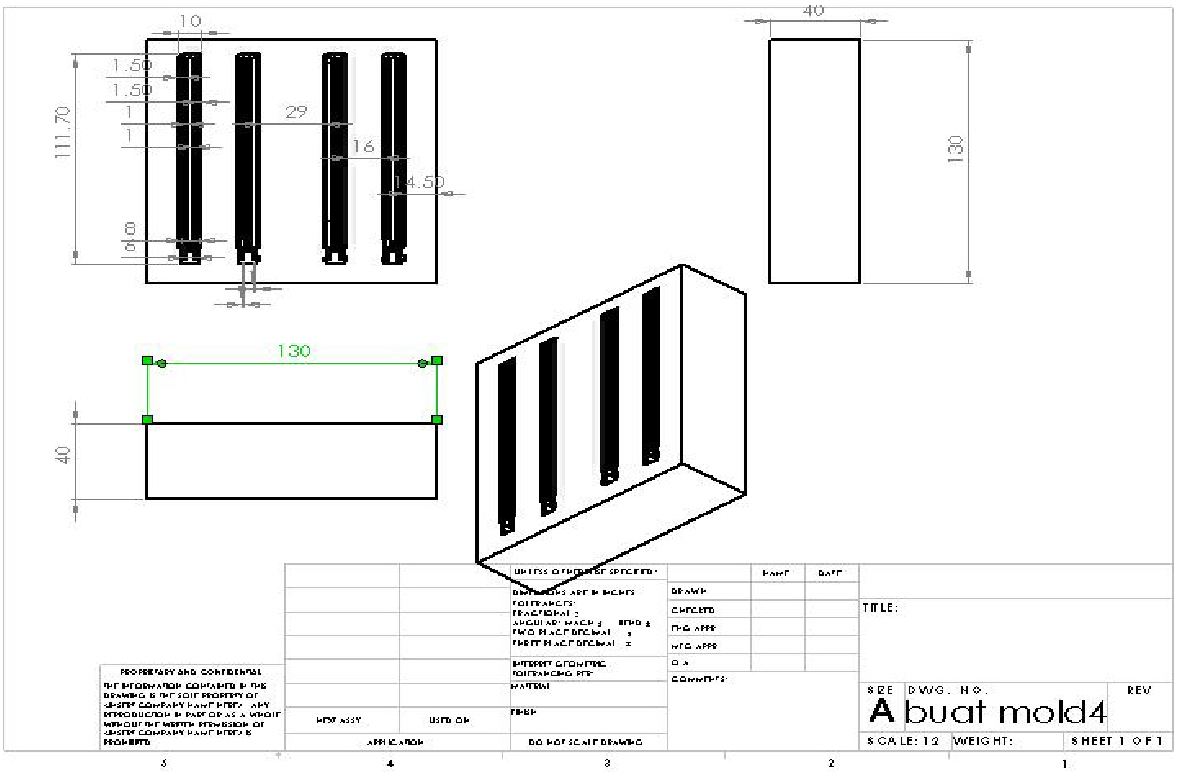


Figure C9: 2D Design for Insert Core Plate

APPENDIX D

Table D1: Gantt chart for FYP 1

Project Activity	Jobs\week	1	2	3	4	5	6	7	8	9	10	11	12	13	14	15	16
1-Understand about PSM title.	Planning	X															
	Actual	X															
2-Meeting the supervisor for the first time.	Planning		X														
	Actual		X														
3-Find the overview, project scope and make the objective and have a meeting with supervisor.	Planning			X													
	Actual			X	X												
4-Find the source like journal and go to library find previous PSM project and have a meeting with supervisor.	Planning				X												
	Actual				X	X											
5-Find more information to improve knowledge about the PSM title and have a meeting with supervisor.	Planning					X											
	Actual					X	X										
6-Make the literature review and find more knowledge about the project title.	Planning						X										
	Actual					X	X	X									
7-Collect the journal and compile it as hard copy and as a reference and have a meeting with supervisor.	Planning							X									
	Actual							X	X								
8-Compile the all source and start to find the information to make the methodology.	Planning								X								
	Actual								X	X							
9-Find more about the methodology and have a meeting with supervisor to show progress about the project.	Planning									X							
	Actual									X							
10-Make the complete methodology and have a meeting with supervisor.	Planning										X						
	Actual										X	X	X				
11-Start to make a complete report.	Planning											X					
	Actual											X	X	X			
12-Have a meeting with supervisor and discuss about the project.	Planning												X				
	Actual												X	X			
13-Have a meeting with supervisor and discuss about the project.	Planning													X			
	Actual													X	X		
14-Presentation	Planning						X	X	X	X	X	X	X	X	X		
	Actual						X	X	X	X	X	X	X	X	X		

Gant Chart for FYP 1

Table D2: Gantt chart for FYP 2

Project Activity	Jobs\week	1	2	3	4	5	6	7	8	9	10	11	12	13	14
1- Understand about Literature review	Planning	X													
	Actual	X	X	X	X	X	X								
2- Find the product for designing process and have a meeting with the supervisor.	Planning		X												
	Actual	X	X												
3- Start to make the product design by using Solidwork software and have a meeting the supervisor.	Planning			X	X										
	Actual				X	X	X								
4- Start to learn how to run Moldflow software at the FKM lab and find more source about moldflow at library.	Planning				X										
	Actual								X	X					
5- Collect the data which is to insert in analysis when to run Moldflow.	Planning					X									
	Actual									X					
6- Run the test for Moldflow, collect the result and have meeting with supervisor.	Planning						X	X							
	Actual										X				
7- Prepare the analysis result and have meeting with supervisor.	Planning							X	X	X					
	Actual										X				
8- Compile the all source and make complete result and discussion. Design the mold	Planning										X				
	Actual										X	X			
9- Start to make the conclusion and have meeting with supervisor.	Planning											X			
	Actual											X			
10- Make the complete conclusion and have a meeting with supervisor.	Planning											X			
	Actual												X		
11- Start to make a complete report.	Planning											X			
	Actual												X		
12- Have a meeting with supervisor and discuss about the project.	Planning												X		
	Actual														X
13- Have a meeting with supervisor and discuss about the project.	Planning													X	
	Actual														X
14-Presentation	Planning		X	X	X	X	X	X	X	X	X	X	X	X	X
	Actual		X	X	X	X	X	X	X	X	X	X	X	X	X

Gantt chart for FYP 2