BENCHMARKING OF THE ACTUAL INJECTION OF THE PRODUCT VERSUS PLASTIC SIMULATION SOFTWARE

ZUL AMRI BIN A.BAKAR

UNIVERSITI MALAYSIA PAHANG

BENCHMARKING OF THE ACTUAL INJECTION OF THE PRODUCT VERSUS PLASTIC SIMULATION SOFTWARE

ZUL AMRI BIN A.BAKAR

A report submitted in partial fulfillment of the requirements
for the award of the degree of
Bachelor of Mechanical Engineering with Manufacturing

Faculty of Mechanical Engineering
UNIVERSITI MALAYSIA PAHANG

NOVEMBER 2008

SUPERVISOR'S DECLARATION

We hereby declare that we have checked this project and in our opinion this project is satisfactory in terms of scope and quality for the award of the degree of Bachelor of Mechanical Engineering with Manufacturing

Signature : ------------------
Name of Supervisor :
Position :
Date :

Signature : ------------------
Name of Panel :
Position :

STUDENT'S DECLARATION

I hereby declare that the work in this thesis is my own except for quotations and summaries which have been duly acknowledged. The thesis has not been accepted for any degree and is not concurrently submitted for award of other degree.

Signature	:	------------------
Name	:	ZUL AMRI BIN A.BAKAR
ID Number	:	ME05015
Date	:	4 NOVEMBER 2008

DEDICATION

"Dedicated to my family the mission of my life"

ACKNOWLEDGEMENTS

Gratefully expressed from deep of my heart and highly gratitude dedicated toward my supervisor Mr. Zamzuri Hamedon for his precious guidance, invaluable ideas and knowledge, constantly encourage and support during easy and hard time to make this project available. Vast knowledge of him in multiple fields as well as technical and science conduction always impressing me and the feeling will persist to stay with me forever to be my model in achieving the future. I appreciate his consistent support from the first day of meeting until this concluding moment. I am truly grateful for his progressive vision about my training in science, his tolerances of my naive mistake, his patience in guiding my understanding, and his commitment to my future career. I also thankfully for the time spent for guiding me, transferring the knowledge and skills, and correcting my many mistakes and problems.

My sincere thanks to staffs of Mechanical Lab, UMP especially Mr. Khairul Azha in assisting, guiding and teaching me with his all dedications and sincere during all my days working at the lab. I am grateful for his patience in explaining my mistakes and misunderstanding which make this project available.

I am also indebted to my parents and family for their limitless love and supports during the progress of this project and as well as my whole life. I also dedicated this gratitude towards my colleagues for their ideas and support. Their encouragement and sacrifices make many unforgettable memories, knowledge and experiences for me during the years spent in UMP.

ABSTRACT

The idea of this project, benchmarking of the actual injection of the product versus plastics simulation software is to do analysis of the parameters that involved in the plastic injection molding in order to determine the best solution to overcome the problems and defects that occurred in the plastic injection molding. By actual injection processes, the results that can be observed are limited such as temperature, stress and the point of gate that are almost impossible to be analyzed by naked eye. Thus, for such details information the engineer can depend on Computer Aided Engineer (CAE) or Computer Aided Manufacturing (CAM) tools such as in this case Moldflow, in generating the accurate data of the parameter that has been analyzed. The tools are capable in assisting the designation of the mold and the parts that need to be produced by generates the data that cannot be achieved by doing the actual experiments. Hence, actual injection analysis needs quite a lot of effort to determine the optimal parameters for the injection process by experienced expertise. Try and error method was traditional way in injecting the part into fine product which consumed a lot of time and energy as well as increase the production cost. The result between software simulation and the actual injection might have slight differences because of several factors. The factors can be determined by doing both analyses and comparing the result will generated the data of error percentage of the simulation software to actual injection as same as factors of the errors.

ABSTRAK

Idea mengenai projek ini iaitu perbandingan antara injeksi sebenar produk dan perisian simulasi plastik adalah untuk menganalisa faktor-faktor yang boleh diukur yang terlibat secara langsung dalam arena acuan injeksi plastik. Ini adalah bertujuan untuk mencari jalan penyelesaian terbaik untuk mengatasi masalah dan kecacatan yang terdapat pada model injeksi plastik. Berdasarkan injeksi sebenar hasil analisis yang boleh diperhatikan adalah terhad kepada beberapa permerhatian sahaja dan faktor seperti suhu, tekanan dan titik kedudukan get adalah menghampiri mustahil untuk diperhatikan dengan menggunakan deria penglihatan manusia. Oleh itu untuk maklumat terperinci seperti perkara tersebut jurutera-jurutera boleh menggunakan perisian "*Computer Aided Engineer"* (CAE) atau "*Computer Aided Manufacturing"* (CAM) seperti dalam kes ini iaitu Moldflow untuk menghasilkan maklumat dan data yang tepat setelah menganalisa permerhatian tersebut. Perisian tersebut berupaya untuk menbantu dalam mereka cipta acuan dan model produk yang perlu dihasilkan dengan menghasilkan data yang tidak dapat diperolehi dengan melakukan eksperimen injeksi sebenar. Eksperimen injeksi sebenar memerlukan kepakaran dan tenaga yang tinggi untuk menghasilkan keadaan terbaik bagi proses injeksi tersebut. Kaedah cuba jaya adalah kaedah tradisional yang diguna pakai untuk menentukan keadaan terbaik tersebut namun ianya memerlukan masa dan tenaga yang banyak di samping meningkatkan kos pembuatan. Nilai dan keputusan yang dihasilkan oleh simulasi perisian dan injeksi sebenar berkemungkinan mempunyai sedikit perbezaan yang disebabkan oleh beberapa faktor dan faktor-faktor tersebut boleh dikenal pasti dengan melakukan kedua-dua analisa tersebut dan hasil analisa tersebut dibandingkan. Perbandingan tersebut akan menghasilkan peratusan kesilapan antara perisian simulasi dan injeksi sebenar.

TABLE OF CONTENTS

APPENDICES

LIST OF TABLES

LIST OF FIGURES

LIST OF ABBREVIATIONS

CAE	Computer Aided Engineer
CAM	Computer Aided Manufacturing
FEA	Finite Element Analysis
MPA	Moldflow Plastic Advisor
ABS	Acrylonitrile-Butadiene-Styrene
MIMO	Multiple-Input Multiple-Output
MPI	Moldflow Plastic Insight

CHAPTER 1

INTRODUCTION

1.1 INTRODUCTION

Nowadays, plastic injection molding is become one of the important industry in the world. This industry placed in the manufacturing field and most of the parts, objects and goods surrounding are based on plastic material. Injection molding is one of the manufacturing techniques from manufacturing engineering field in producing parts that based on plastic material. The molten plastic that has been melted from the hopper through the barrel will be injected at high pressure into a mold with the cavity of desired parts shape. Major problem in plastic injection molding industry is the results are somehow different from the simulation software. Thus, it will contribute serious problems for the Quality Assistant and the engineer in order to predict the suitable setting or design of the part and mold. This project will compare those results. The parameters need to be selected as not all of the data or results can be observed by naked eye by actual injection. Published software, Moldflow will be used during analysis by simulation. In the end the results gained from those two approaches will be compared and analyzed to observe if the results are same, acceptable or errors.

This analysis can be done in many approaches but based on this project's title the study has to be done by manual experimental and at the same time by plastic simulation software. Benchmarking can be defined as comparison or to differentiate two or more parameters that have been studied. From the title of this project, in other words, it means to make comparison of the results, observations and consequences between the actual manually handled plastic injection molding machine and the results that have been analyzed by the plastic simulation software. There are some analysis can be compared between the actual injection and software simulation. For an example is the common defect occurred in plastic manufacturing industry which is shrinkage. Volumetric shrinkage is the contraction of polymer due to the change in temperature from melt temperature to ambient temperature [1]. High volumetric shrinkage can cause part warpage, sink marks, critical dimensions that are too small and internal voids. Excessive wall thickness and inadequate packing can both contribute to high volumetric shrinkage in a part. The solutions to avoid this defect are altering the part design such as the wall thickness and the other critical area. Second solution is altering the gate locations and lastly altering the processing conditions by increase the packing pressure. There other analysis that can be benchmark is the deflection of the finished products. The deflection resulting from the Moldflow shows the deflection at each node of the part (warpage or stress analysis), or each node of the wire or paddle (microchip encapsulation analysis), based on a "best fit" technique, where the original geometry and the deformed geometry are overlaid in such a way that they best fit together, or based on a defined anchor plane defined. There are a number of possible variants of the deflection result according to:

- Analysis type - The result name may indicate the type of analysis that was run, that is, either small deflection or large deflection. If this is not indicated in the result name, then the results will apply to a small deflection analysis.
- Net vs component deflections - The net view of net deflections at each node, or the component of the deflection either along the X, Y or Z axis. The axis directions are determined by the defined anchor plane and are indicated in the anchor plane symbols.

- All effects versus warpage contributors - There are four sets of deflection results. To create these results, run a small deflection warpage analysis and select the Isolate cause of warpage option on the Warp Settings page of the Process Settings Wizard.

There are also analyses that can not be compared between those two approaches yet it is important to be analyzed such as for an example the fill time analysis. As in Moldflow software the results of this analysis is called fill time result. The Fill time result shows the position of the flow front at regular intervals as the cavity fills. Each color contour represents the parts of the mold which were being filled at the same time. At the start of injection, the result is dark blue, and the last places to fill are red. If the part is a short shot, the section which did not fill has no color. Secondly, the analysis of time to freeze also an important parameter yet can be observed by naked human eyes. Thus, from Moldflow judgments the Time to freeze result is generated from a Midplane and Fusion flow analysis, and shows the amount of time taken from the end of fill at 100% to the ejection temperature. This result takes into account the dynamics of both filling and packing phases, where new hot material enters the cavity. This new hot material affects the cooling time.

Shrinkage is the amount by which a molded product is smaller than the size of cavity space wherein it was produced by injecting plastic under high pressure injection and at high temperature [2]. There are three rules regarding the shrinkage behavior which the first rule is, there is a definite relationship between pressure (P), volume (V) and temperature (T). This relationship is different for various plastic. Any and all conditions that affect those parameters will affect the shrinkage. Second rule is when a volume of plastic is heated it will expands. Then when it cools to the original temperature, it will contract to the original volume. Third rule is when a plastic is compressed the volume will be reduced. When the pressure is reduced to the original pressure it will return to its original volume. The greater the temperature difference between the room temperature and injected plastic then the shrinkage also will be greater. Timing also can affect the shrinkage behavior where the longer the injection pressure is kept on the plastic in the cavities the less will be shrinkage. In term of pressure, where the pressure on the plastic (in cavity) is high, less shrinkage will take place but in the other hand when the area is low in pressure the plastic will shrink more. It also can be affected by plastic material characteristics. Each plastic has a typical coefficient of temperature expansion. In most cases it is impossible to predict with certainty the correct shrinkage of a material since it depends on so many factors.

1.2 PROBLEM STATEMENT

1. The differences of the result between the software and the machine are not 100% the same
2. The detail about the defects that can not be analyzed by using simple observation methods have to be determined by using software simulations
3. The condition of mold and software capabilities might influence the results of both analyses.

1.3 PROJECT OBJECTIVES

a. To get the parameter value using the CAE or FEA software – by Moldflow software
b. To use the data from Moldflow to setup plastic injection molding machines
c. Determine the differences of results of the parameters between actual experiments and software simulation analysis

1.4 PROJECT SCOPES

a. Literature review will be done regarding to the title of this project
b. For this project Moldflow Plastic Advisor (MPA) software will be used for the software analysis method.
c. Reversed engineering will be applied according to the already available mold to obtain the parameters of the mold
d. The product designation is depends on the finished product and for this project the product is paper rack.
e. The material type will be used is Acrylonitrile-butadiene-styrene (ABS).
f. The machine that will be used is Arburg 520C Allrounder 2000-800 for the actual injection analysis.
g. The processing properties of the material will be used as reference for the software analysis
h. The model of the product is design by using CAE software Solidwork
i. The machine will be setup by using results from the software for the actual injection analysis.
j. The result from software analysis and the result from actual injection analysis will be compared.

CHAPTER 2

LITERATURE REVIEW

2.1 INTRODUCTION

Parts that based on the plastic material can be produced in many ways and the most popular approaches are by injection. The basic concept of this method is injecting the molten plastic material into the mold with the cavity of the product's shape and the material will be cooled down then forming into solid form of the desired product's shape before it ejected by pin ejector and ready to be used. In injection molding process, there are four main steps or cycles have to be taken namely filling cycle, cooling cycle, mold open cycle and part ejects. The crucial step is during the filling cycle since the quality of the goods and lifespan of the molds are depending seriously on this.

2.2 INJECTION MOLDING MACHINE

Basically, the injection molding machine functions as the holder of the mold and injecting the molten material into the cavity inside of the mold. There are several types of injection machine but most widely used are hydraulic type, all-electric and combination of both types. Generally, an all electric type machine not very different from hydraulic type in term of body mechanism [3]. However, there are also significant differences between those two types of machine and the differences as stated below:

i. the uses of AC Servo Motor
ii. the uses of ball screw

iii. the uses of gear and timing belt

The existences of these components are to substitute original hydraulic element such as hydraulic motor, directional valve, hydraulic board and cylinder. Since the electric elements are used to drive the injection machine so it is therefore called "All-Electric". The advantages of this type of injection machine are no problem with oil leakage as it does not use oil for hydraulic system thus will generate less pollution. It also has less operation noise, less energy consumption and has high accuracy of mold movement. In the other hand the operation cost of the machine is high with high cost of servo motor. The durability of ball screw also needs to be put under consideration since it has certain lifespan. This machine has slight difficulty on developing large tonnage force model which can resulted instable power supply and also unable to use accumulator to create transient high pressure. An injection molding machine is called hydraulic type when it use hydraulic system to open or closed halves mold by a reversible fluid motor actuated by a die control valve. The advantages of hydraulic type machine are the mold is easier to be setup onto the machine, the clamp pressure can be easily determined, low maintenance cost with low platen deflection since the force concentrated at the center of the platen. Vice versa the disadvantages of this machine are the oil for the hydraulic system tends to leakage and it requires large volume of hydraulic oil. The energy consumption is inefficient and overcompensate is a must due to compressibility of the oil. This machine also required large space.

2.3 IMPORTANT COMPONENT IN PLASTIC INJECTION MOLDING MACHINE

Plastic injection molding machine consists of several components that assembled into a whole machine.

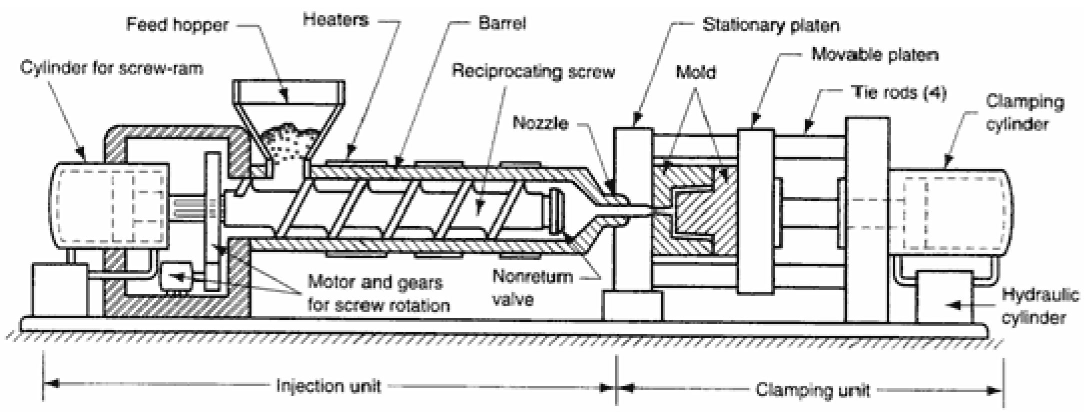


Fig 2.1: Important part of injection molding machine
Source: Plastic Technology, BMF 4713 Teaching Handout (2008)

As referring to the diagram, there are two main unit in the injection molding machines where they are stated as injection unit and clamping unit. In the clamping unit is consisted by stationary platen, mold, moveable platen, tie rods, clamping cylinder and in this case hydraulic cylinder since the machine is hydraulic type. The function of clamping unit is to holds the mold together, open and closed the mold automatically, and finish the injection process by ejects the finished product.

2.4 MOLD

Mold can be divided into two main types which are two-plate mold and three-plate mold type. The main difference of these two types of the mold is about the function of handling the runner. Three-plate mold has self-degating function which means the runner is disassembled from the finished injected products by mean of mechanical movements of the assembled mold. The three-plate mold has an extra plate compared to the two-plate mold with present of stripper plate assembled between the fixed half mold plate and top plate of sprue bushing. This function will produce two parting line instead of a single parting line for the two-plate mold type where the extra parting line located at the fixed half mold plate and stripper plate.

Three-plate mold is better when the quality of the surface finish on the products is crucial matter since the runner and sprue part do not have to be cut manually by manpower which the quality of the cutting will not be consistent with extra cost consumed for the salary man. The detached sprue and runner will be treated as wastes and depending on the material it can be recycle by crushing them back into particle or pallet form. If the material used categorized in thermoset family it can not be recycled since the chemical degradation of the material will be not resulting into a desired finished product and can be harmful for the screw where it can burning inside the barrel.

2.5 RUNNER

Runner is channel into the mould plate to connect the sprue and gate to impression. The type of runner can be defined as one of the most important factors that should be considered before fabricating process and mainly there are two types of runner namely cold runner and hot runner. They can be known by present of filament at the runner where hot runner type is chosen for one mold. There are some significant criteria differences of the two types of runner. The cold runner system has some disadvantages such as high cost of energy and workmanship, high scrap ratio, low product quality of surface appearances and requirement of high injection pressure. In the other way, hot runner system is able to provide precisely adjustable process temperature, uniform filling in multi-cavity molds, even heat distribution in the molds, improvement on mechanical properties of the injected products, cuts in production cost and shorter mold opening distance because absence of sprue while shorten the cycle time. The layout of the runner system also needs to be considered as critical factor which it depend on the shape of desired product and size. There are four main layouts such as conventional

[Fig 2.2], improved [Fig 2.3], balanced H [Fig 2.4] and circular [Fig 2.5]. When designing a mold the criteria of the mold is need to be categorized into consideration. The runner should be providing maximum cross sectional area from the standpoint of pressure transfer and a minimum contact area from the standpoint of heat transfer.

The following factors are should be considered while deciding the runner size. The first factor is about the wall section and volume of the molding. The cross sectional area of the runner must be sufficient to permit the molten material to pass through and fill the impression before the runner freezes. The second factor is the distance between impression and main runner or sprue where the resistance of flow is greatly depends on the length of the runner. When the gap between the impression and sprue or main runner is large it will make larger resistance for the flowing molten material. Thirdly is about the runner cooling system where the large size of runner will increase the cooling time.

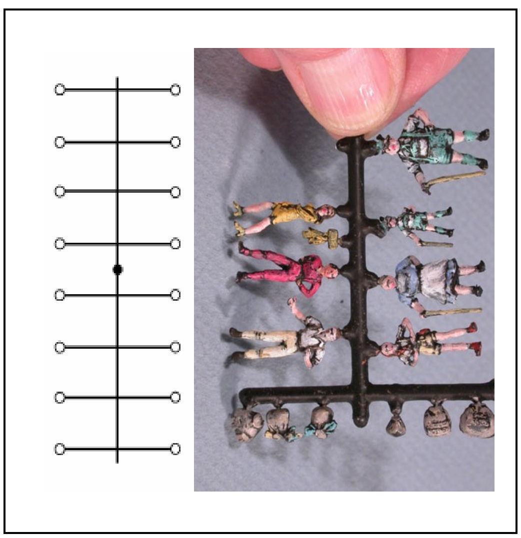


Fig 2.2: Conventional layout

Source: [3]

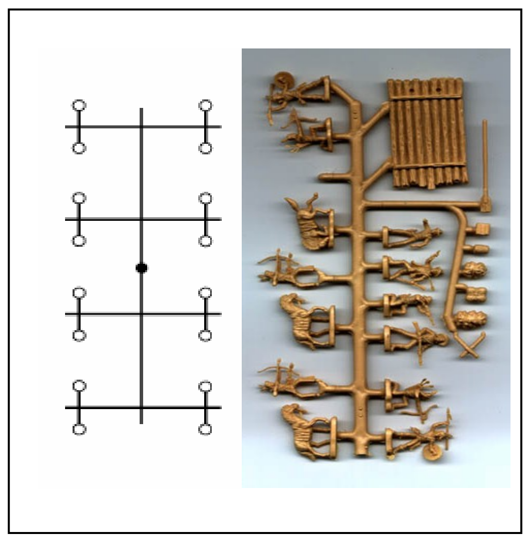


Fig 2.3: Improved layout

Source: [3]

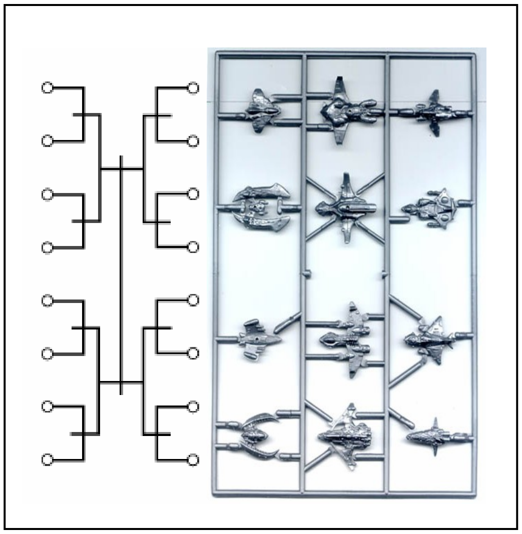


Fig 2.4: Balanced H layout
Source: [3]

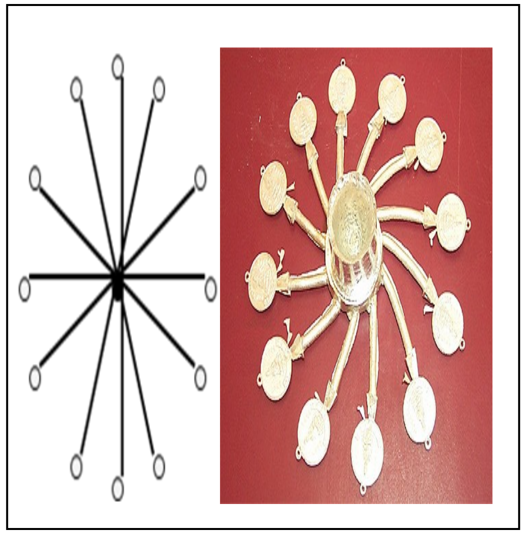


Fig 2.5: Circular layout
Source: [3]

2.6 DEFECTS ON PLASTIC INJECTION MOLDING

The defects such as burn marks or air burn which is brown or black burnt areas on the part located at furthest points from the gate is because of the tool lacks venting or the injection speed is too high [4]. The other type of defect is flash or burrs can be detected on the part when excess material in thin layer exceeding normal part geometry resulting from too much injection speed or too much material injected, clamping force too low or tool damaged. Sink marks can be detected as localized depression which happened at thicker zone of product. This defect occurred when the holding time or the pressure too low, cooling time too low with sprueless hot runners and this defect also can be caused by the gate temperature being set too high. The other type of defect is short shot where the finished product is only partial of the original shape. This is because lack of material, injection speed or pressure too low. Warping or also defined as twisting is when the part is distorted due to cool time is too short, material is too hot, lack of cooling around the tool or incorrect water temperatures (the parts bow inwards towards the cool side of the tool). Weld line or meld line is detected as discolored line where two flow fronts meet. The defect because of the mold or material temperatures set too low which mean the material is cold when they meet so they don't bond uniformly.

2.7 PREVIOUS RESEARCH ON PLASTIC INJECTION MOLDING ANALYSIS

The article written by Britton et al (2003) [5], the use of CAE software such as Moldflow, in the plastic industry is well established. The software is useful in simulating and visualizing the performance of the injection molding process. The high level of expertise needed to use it has been one of the major obstacles in its further application. One obstacle is in interpreting analysis results. The research is initiated by the tedious and difficult to interpret the results, and there is currently no commercial software that allows specifying their design intent to verify and evaluate their design.

Determining the optimal parameter setting for injection process is crucial process. Based on article by Chen,W.C. et al (2007) [6] setup the parameter to the optimal is critically influences the productivity, quality and cost in the production. Commonly engineers will use trial and error method or Taguchi's parameter design method to determine the optimal solution but recently the methods are not suitable anymore because the products nowadays are way more too complex and the requirement of multi-response quality characteristics. The optimal settings can be achieved by using a soft computing paradigm for the process parameter optimization of multiple-input multiple-output (MIMO) plastic injection process.

Generally, from article written by Kim et al (2007) [7] there are several parameters that can be compared between the actual injection and simulation software's data such as defects that occurred on the parts, filling time and residual stress. Residual stress means the internal stress that occurred in the mold but no external forces act on it and this kind of stress is one of the main problems has to be dealt in injection molding industry. This stress can produces defects on the parts when the parts used for a long period of time or exposed to high temperature. This problem can be solved by either doing the actual experiment or assisted by software. Some of the methods that can be used after experimenting by actual injection are hole-moving method or layer-removal method.

In the other hand, software such as Moldflow is used to determine the residual stresses variation with respect to the thickness and predict the residual stress in the plate. By gathering the data we can decrease or eliminate the defects. The other parameter is related with filling cycle. The article written by C.K. Au (2004) [8], filling patterns is important in designation of the parts especially in the initially conceptual design. Unfortunately the flow analysis by actual experimental is high costly since it consuming time when several design configurations need to be evaluated. The advantage of the plastic simulation software is to give an approximate solution to the basic filing patterns and geometric approaches is generated to model the space time function of these patterns without consuming too much times and materials. The filling pattern obtained by slicing the resultant space time function.

Besides, this parameter are also important to be discussed when we are injecting parts with thin wall. Based on the article from Song et al (2006) [9] thin wall has some advantages such as saving material, saving the production costs and reducing weights or shapes. In the other hand, the injection molding process will become complicated with the reducing of the part thickness which the molding characteristics are lacking systematic investigations. This problem can be analyzed by orthogonal method or Taguchi method during the actual injection and simulation software is assisting during filling analysis. The goal is to determine the most influential factor in injecting thin wall parts in order to reduce time for trial molding, improves the part quality and can be reference for further experiment of molding defects.

There are several complicated factors are considered in this experiment such as filling volume, melting temperature, injection pressures, injection rate, metering size and part thickness. The filling capability of the molten plastic material will decrease rapidly with the reducing of the part thickness as we can see that the molten plastic is difficult to be distributed in the mold cavity of a part with thin wall [9]. Metering size and injection rate can be analyzed by the software since an appropriate metering size is the necessary condition to the molding and accelerating injection rate can increase the filling ratio. The melt temperature and injection pressure are also important during injecting a part with thin wall.

Furthermore, the article by J.K.L.Ho et al (2004) [10], the thin wall part can added the cost for the production line. Minimizing the cost of producing the plastic product is very important. Currently, the approach of R&D work focuses on optimizing the dimensions of the plastics component which is reducing the wall thickness of the product.

There are some common analysis can be done by the Moldflow software such as fill time, injection pressure, melt temperature, pressure drop and quality prediction. Fill time illustrate the specific illustrates the specific flow path of a plastic material grade and the color spectrum identifies fill time intervals (sec.) across the part geometry. Viewing benefits include accurately balanced multi-cavity and family mold layouts. Injection pressure analysis is color coded to identify regions of high and low pressure across the part geometry. Melt temperature illustrates the melt flow temperature across a given part geometry [10]. Wide temperature variance across a part can potentially result in poor quality issues such as warpage, improper gate location, and sink marks. Pressure drop analysis represents the drop in pressure from the material injection location to a given location on the part. A uniform pressure distribution per unit length is desired across the part. This uniformity allows for the most efficient filling pattern. Quality prediction analysis is used to address quality issues. The quality issues are derived using pressures, temperatures, cooling time, shear rate, and shear stress data.

2.8 MATERIALS OF THE MODEL

Polymeric materials are characterized by long chains of repeated molecule units known as "mers"[11]. These long chains intertwine to form the bulk of the plastic. The natures by which the chains intertwine determine the plastic's macroscopic properties. Typically, the polymer chain orientations are random and give the plastic an *amorphous* structure. Amorphous plastics have good impact strength and toughness. Examples include acrylonitrile-butadiene-styrene (ABS), styrene-acrylonitrile copolymer (SAN), polyvinyl chloride (PVC), polycarbonate (PC), and polystyrene (PS). If instead the polymer chains take an orderly, densely packed arrangement, the plastic is said to be crystalline. Such plastics share many properties with crystals, and typically will have lower elongation and flexibility than amorphous plastics. Examples of crystalline plastics include acetal, polyamide, polyethylene, polypropylene, polyester, and polyphenylene sulfide. Most plastics can be classified as either thermoplastic or thermoset, a label which describes the strength of the bonds between adjacent polymer chains within the structure. In thermoplastics, the polymer chains are only weakly bonded which is Van der Waals forces. The chains are free to slide past one another when sufficient thermal energy is supplied, making the plastic formable and recyclable. In thermoset, adjacent polymer chains form strong cross links. When heated, these cross links prevent the polymer chains from slipping past one another. Because of the cross-linked chain structure of polymers, the tensile strength of polymers tends to degrade with increasing temperature. As the temperature raises, the parameters such as modulus either for tensile or flexural values will drop, the tensile strength also will drop. But in the other hand the elongation of the material, creep effects, stress relaxation and impact strength or toughness will be increased.

The material that will be used in this project is acrylonitrile-butadiene-styrene (ABS). ABS is an ideal material wherever superlative surface quality, colorfastness and luster are required. ABS is a two phase polymer blend. A continuous phase of styrene-acrylonitrile copolymer gives the materials rigidity, hardness and heat resistance. The toughness of ABS is the result of sub microscopically fine polybutadiene rubber particles uniformly distributed in the SAN matrix. ABS standard grades have been developed specifically to meet the requirements of major customers. ABS is readily modified both by the addition of additives and by variation of the ratio of the three monomers Acrylonitrile, Butadiene and Styrene hence grades available include high and medium impact, high heat resistance, and electroplatable. Fibre reinforcement can be incorporated to increase stiffness and dimensional stability. ABS is readily blended or alloyed with other polymers further increasing the range of properties available. Fire retardancy may be obtained either by the inclusion of fire retardant additives or by blending with PVC. The natural material is an opaque ivory color and is readily colored with pigments or dyes.

There are several important properties of the ABS material take into account for the processing of the injection molding process. As shown in Table 2.1 the value for melt flow is 1.1 gm/10 min. The melting temperature range is from 100ºC to 110ºC when the material is in amorphous state. Meanwhile for the injection process the range should be inbound from 177ºC until 260ºC and during the setup for the software simulation analysis and actual injection the value for the melting parameter for ABS material should not be lowered than 177ºC and not higher than 260ºC. The melting temperature will impact the result and defect maybe occurred for mismanipulation of the value. The molding pressure also ranged from 56 MPa until 173 MPa.

Table 2.1: Processing properties of ABS material

Processing Properties		**Conditions**	
		Type	**ASTM**
Melt Flow (gm/10 min)	1.1		D1238
Melting temperature (ºC)	100 - 110	Amorphous	
Processing temperature (ºC)	177 – 260	Injection molding	
	163 – 205	Compression molding	
Molding pressure (MPa)	56 - 173		
Compression ratio	1.1 – 2.0		
Linear mold shrinkage (cm/cm)	0.005 – 0.008		D955

Source: www.efunda.com (2008)

CHAPTER 3

METHODOLOGY

3.1 INTRODUCTION

The methodology indicates the works and process need to be done in order to complete this project. Briefly the methodology of this project includes from preparation of the idea by literature review and meetings with supervisor, running the simulation software, operating the injection molding machine until benchmarking the result and the progress summarized as shown below.

a. Literature review in order to gather the initial data about this project
b. Analyzing the data to get the processing parameters of the material, exposed to software simulation and plastic injection molding machines
c. Apply reverse engineering such as measures the dimension of the mold to obtain the parameters
d. Learn how to use plastic simulation software, Moldflow Plastic Advisor
    i. Introduction to the software
    ii. The systematic of the software
    iii. Parameters that need to be analyzed
    iv. Designation and setting up of part model
    v. Analysis of the model
    vi. Generating the results of the analysis
    vii. Analyzing and interpret the data that has been generated by the software

e. Learn how to operate the machines
   i. Introduction to the plastic injection molding machines
   ii. Explore how to operate the machine
   iii. Defining the parameters
   iv. Setting up the mold
   v. Controlling the parameters
   vi. Controlling the operations of the machines
   vii. Observing the parts quality

3.2 PRODUCT DESIGN

The product will be designed with Solidwork software for drawing the product. The finished product will be used as references and the dimensions are measured manually. Based on the dimension the product will be draw part by part and assembled into finished product. The rough rectangular dimension of the model is 243 mm in wide, 337 mm in long and 48 of height as shown in Fig 3.1. Then it will be the model to be used in Moldflow when doing the analysis thus inaccurate dimension of the model with the actual product will generate different results. The format of the file is STEP before the product can be imported by Moldflow software.

IGES type file format as an example, the type where the model is compressed into a whole part thus it is not encouraged to draw the model in assembly mode. It is because the simulation software not capable in reading the file correctly as the assembled model was compressed into one part and will resulting distorted and unfinished model after uploaded into the software as shown in Fig 3.5. The result of the analysis is significantly impacted by the condition of the model. As shown in the figure which is the result for confidence of fill analysis, noticeable there are some portion of the model are not filled by the molten material as estimated by the software simulation. Only 97.6 percent of the mold cavity of the model is highly confident to be filled by material thus forming short shot defect. The portions of the distorted area are the point where the parts were assembled during the process of drawing the model. The software misread the assembled parts as one whole model thus the software not capable in determine the point of assembled which lead the analysis generating the wrong result.

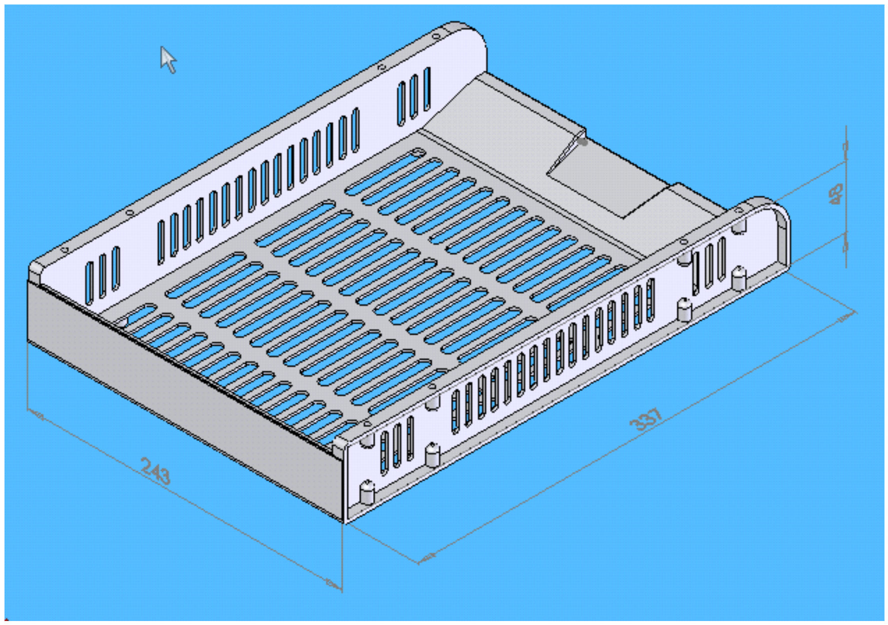


Fig 3.1: 3D view of paper rack model – Upper

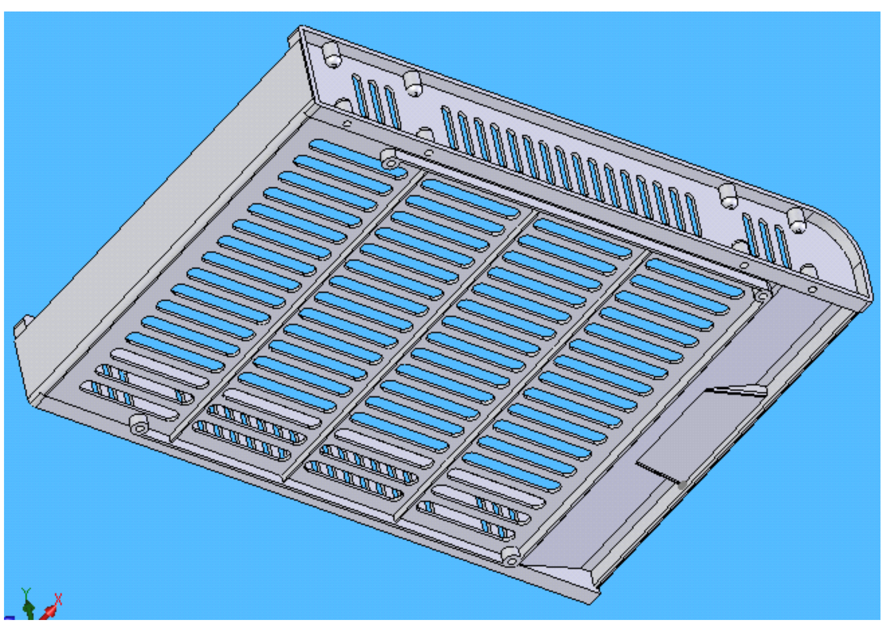


Fig 3.2: 3D view of paper rack model – Below

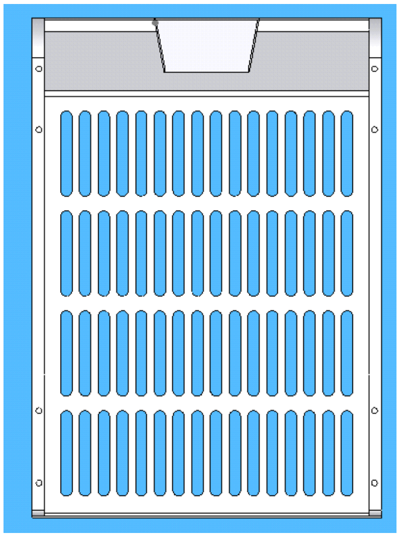


Fig 3.3: Upper view of paper rack model

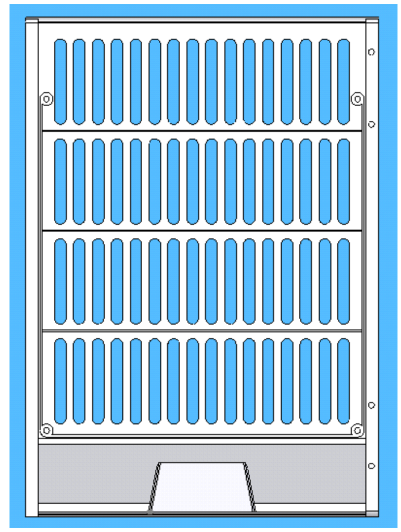


Fig 3.4: Bottom view of paper rack model

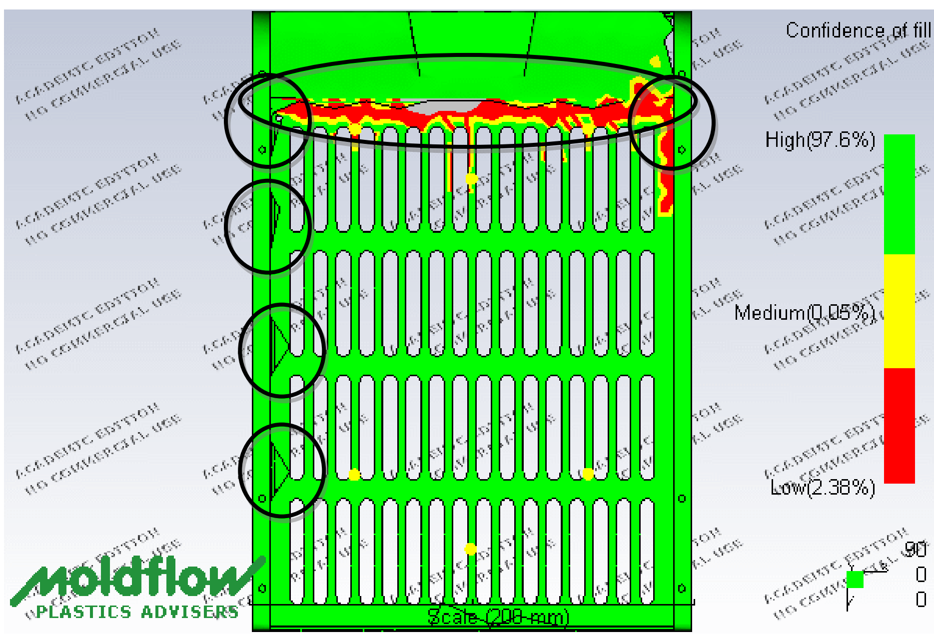


Fig 3.5: Distorted area in round circle

Other than finished product dimensions, the dimensions of the mold also measured to be used in Moldflow software during the analysis. The dimensions of the mold are including the dimensions of the each plates, overall size, dimension of the runner, branch runner, gating system and sprue [Table 3.1]. The layout of the runner system, gating system, sprue and cooling system also are observed to be designed in the software simulation.

Table 3.1: Dimension and properties of sprue, runner and gate

Parts	Type	Shape	Dimension	
			Start (mm)	End (mm)
Sprue	Cold	Circular tapered	3.00	8.00
Runner	Cold	Semi-circular	8.00	
Gate	Cold	Circular tapered	3.00	1.00

Table 3.2: Dimension of the mold

Dimension	Length (mm)	Height (mm)	Wide (mm)
Mold	510	493	430

3.3 SIMULATION SOFTWARE ANALYSIS

There are two general methods have to be used and they are manually experimenting method and analysis by plastic simulation software. For the experiment by using software, Moldflow will be used to analyze the parameters and generates the results. The process of analysis for the simulation software as shown in Fig 3.6 started with upload the file of the model in format of STEP. Second step is defines the type of the cavity as single-cavity as the mold can only produces one product per cycle which mean there is only one cavity of the product on the mold. Then define the runner and sprue properties as shown in Table 3.1 as same as the design at the mold before design of both runner and sprue drew onto the model. Next is defining the mold properties and dimension. The next step is setup the material that will be used for the actual injection and in this case the material is defines as trade name Toyolac 100 in ABS family. Next the process move to setup the analysis processing. The parameters that can be setups are mold and melting temperature, injection time and maximum machine's injection pressure and it concluded all the preparation before run the analysis. After the analysis is completed the report is generated by the software for interpretation.

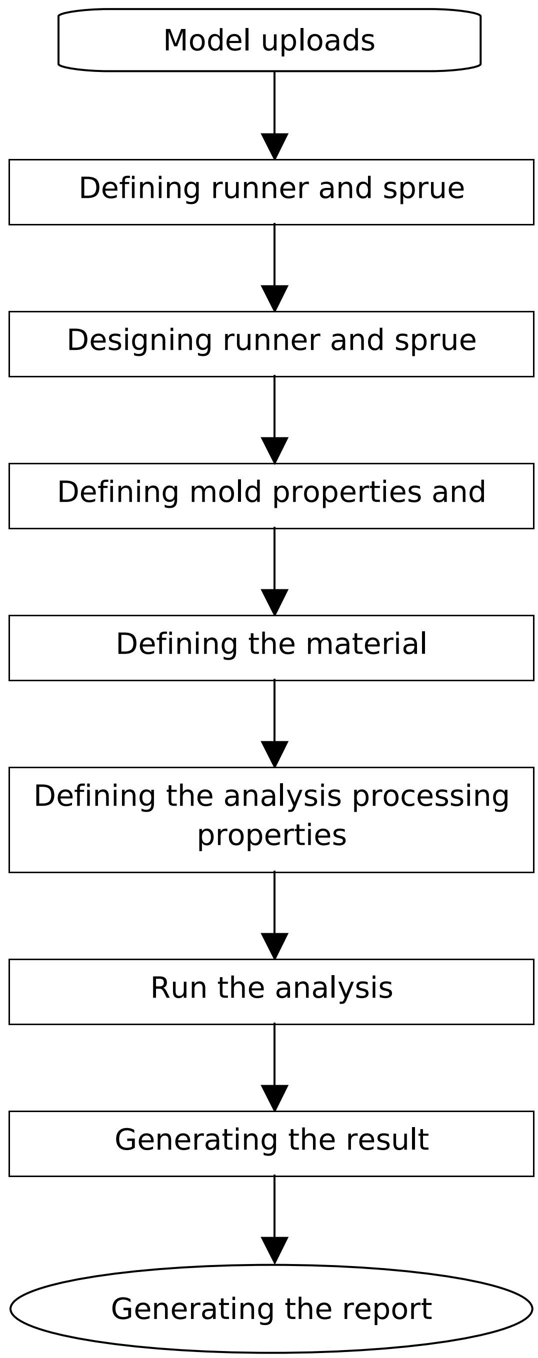


Fig 3.6: Flowchart of software simulation operation

The parameters will be set before processing with both methods and the parameters are should be same for both so those can be compared at the end of this project to benchmark the results. The parameters also should be available to be observed for both analyses and the values will be depending on the material that will be used in the injection. Thus the parameter that will be setup during the software analysis is depending on ABS processing properties. In order to do the software analysis, reverse engineering will be applied. The already available mold will be measured to obtain the information regarding the dimensions of the mold and the other properties such as the location of gate and size of the sprue. The material is selected based on the available option which is ABS material trade name Toyolac 100 is used to do the analysis. Hence, the parameters such as mold temperature, melting temperature and injection time of the software were setup as default setting which has automatically generated by the software. The mold temperature is set at 50ºC, melting temperature at 230ºC and clamping force at 2000 MPa which is at the maximum setting and depends on the machine capabilities.

3.4 ACTUAL INJECTION EXPERIMENT

Actual Injection analysis will use plastic injection molding machine horizontal type with brand name Alburg model 520C Allrounder 2000-800. The parameter will be set by referring to the data generated by the software. The already available mold as shown in Fig 3.7 for the product will be setup onto the machines and the parameters is set and the molten plastic raw material will be injected.

Fig 3.7: Mold of paper rack product

3.5 BENCHMARKING

After finishing both of the analysis the data will be recorded for observation and further analysis. The data will be differentiate and benchmark to analyzed the similarity or differences between those data from each analysis. As for the data the result is recorded in form of table and differences of the value can be determined and calculate. Meanwhile for the physically comparison list of figures will be determined to differentiate both analyses.

3.6 METHODOLOGY FLOWCHART

Generally this project started with meetings with supervisor to get the rough ideas and guidelines during the whole progress as shown in Fig 3.8. After literature study the progress move to designing the product which the already available mold was disassembled to get the dimension and layout of the runner and sprue. The dimension of the mold also took into account to setup the parameter for simulation software analysis. The product is designed by Solidwork.

The next stage is simulation software analysis where the designed model uploaded for analysis preparation. The pre-analysis run to detect whether there are error or defects on the model and if the errors present then the model need to be altered and the other hand if absent of error the preparation for the software analysis is completed and can proceed to the software analysis. The report of the software analysis result is generated by the software.

Next step is to use the results generated by the software to setup the injection molding machine for the actual injection experiments. The model is injected until the finished product is obtained and the results for the actual experiment are compiled by observing the physical state of the finished product and the data taken from the machine.

Then the results from software analysis and actual experiments are compared for the benchmarking process. The physical state of the finished product and the parameters are compared to the results generated by the simulation software before move to the calculation of the error percentage between those results. Then the last process is analyzing the data into discussion for the final report submission.

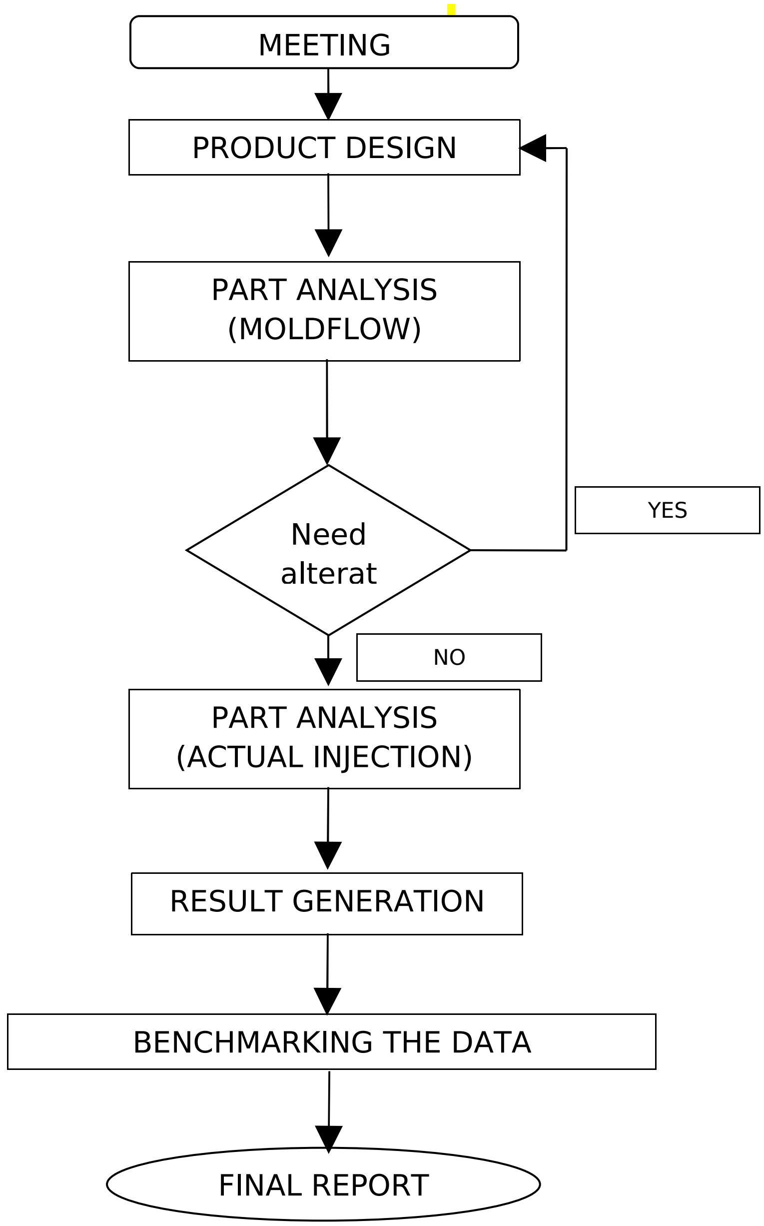


Fig 3.8: Methodology flowchart

CHAPTER 4

RESULT AND DISCUSSION

4.1 Software analysis result

The analysis of plastic injection molding by using simulation software can bring many advantages to the engineer in order to cut time consumption in setting the parameter on the injection molding machine instead of by using try and error system. The software will generates the optimum parameter for the injection process by analysis the model and the setup of desired criterion onto thus the result will be used in preparing the injection molding machines. MPA is one of the tools can used in doing the analysis of the plastic injection molding. This tool capable of analysis up to thirty important parameters involves in the injection process where some of those can not be determined by human capabilities. The parameters is setup at default setting as 230.0 C for injection temperature, 50.0 C for mold temperature and injection time is set as automatic. The other result as generated by the software is compiled as the report in form of tables.

The filling analysis result as shown in Table 4.1 indicates that the dosage volume to fill the whole cavity which mean including the runner system, gating system and product cavity inside the fixed half mold is 315.9390 cm^3. The value for the molten material to fill the cavity of the product is at 293.5810 cm^3 and the balance will be 22.3576 cm^3 that acts as waste. The fill time is at 2.53 s and estimated 120.886 MPa or 1208.86 bar of injection pressure is needed to fill the cavity. The clamp force needed to hold the mold is at 251.357 tonne.

Table 4.1 : Fill analysis result

Descriptions	**Values**
Actual filling time, s	2.53
Actual injection pressure, MPa	120.886
Clamp force area, cm^2	623.5470
Max clamp force during filling, tone	251.357
Velocity/pressure switch-over at % volume, %	97.03
Velocity/pressure switch-over at time, s	2.37
Total part weight at the end of filling, g	214.352
Shot volume, cm^3	315.9390
Cavity volume, cm^3	293.5810
Runner system volume, cm^3	22.3576

The analysis also showed the estimated time taken to complete one cycle of the injection process as shown in Fig 4.1. It is divided into four main stages namely fill stage, pack stage, cool stage and mold open stage. The longest time taken is during the cooling process where it consumed at 17.63 s.

Fill stage, the shortest time consumed, indicates as the molten material injected into the cavity and wholly fill at 2.37 s. Pack and cool is one of the crucial factor in determine the quality of the finished injected products where efficient manipulation of the cooling system will reduce the warpage defect phenomenon and prevent the finished product brake or crack during the ejection due to the premature solidify molten material in the cavity.

The natural behavior resulting of shrinkage of the solidifying molten material where resulting warpage or deflection normally at wide shallow area and designing rib structure can overcome this problem.

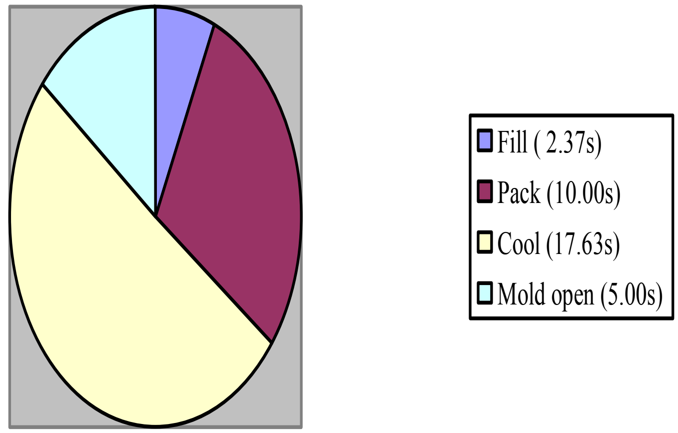


Fig 4.1: Pie chart of cycle time

One of the defects estimated by the software simulation is sink mark. It occurred at thick part where the area thickness is inconstant with the overall area of the product. It also can be happened at the area near of walls. Sink mark is localized depression where the natural behavior make the molten material shrink above their tolerances value. The defects can be overcame by increase the cooling time and injection pressure or redesign the product with avoiding inappropriate varies thickness of the products. From the result as shown in Table 4.2 the maximum value for the sink of the molten material is 0.07 mm at depth at covered 0.76% from the total products. The average of the depth for the sink mark is at 0.02 mm.

Table 4.2: Estimated sink mark

Description	**Value**
Max sink depth, mm	0.07
Average sink mark depth, mm	0.02
Percentage of model prone to sink marks, %	0.76

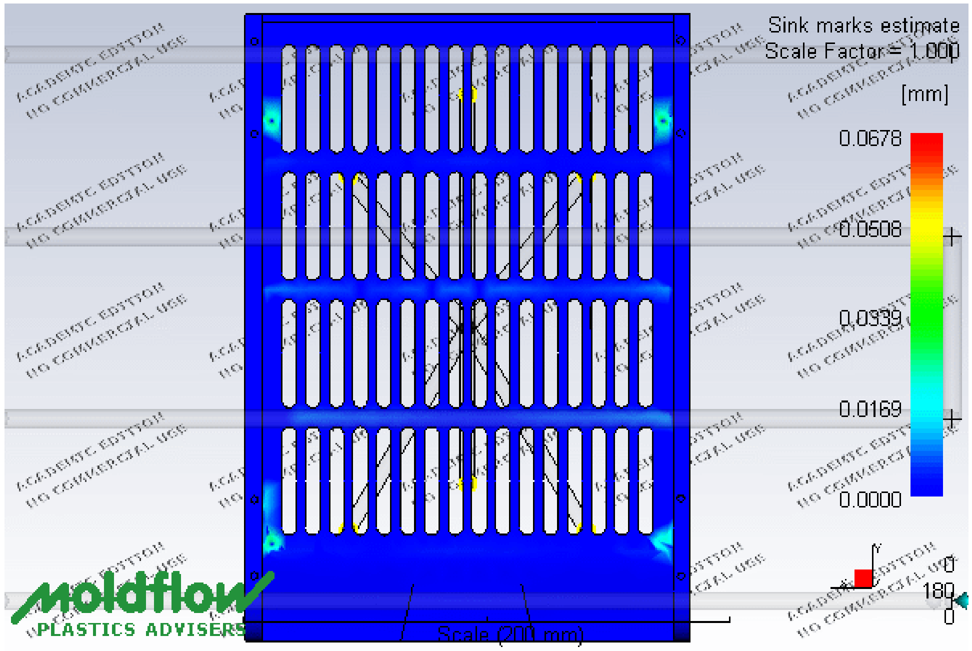


Fig 4.2: Sink mark

The other defect estimated by the simulation software is warpage or also can be defined as warping or twisting. It can be observed as distorted part of area where there is deflection of the product from the original dimension. It usually occurred when the material is too hot during the injection, cooling time is too short and inefficient design of cooling system where the product was not wholly solidifies into rigid. From the analysis as shown in Table 4.3 it shows that the nominal maximum deflection at 1.04 mm where 22.73 % of the area are exceeding the nominal maximum deflection which put 77.27% balance within the maximum deflection.

Table 4.3: Estimated warpage

Description	Value
Nominal max deflection, mm	1.04
Percentage exceeding nominal max deflection, %	22.73
Percent within nominal max deflection, %	77.27

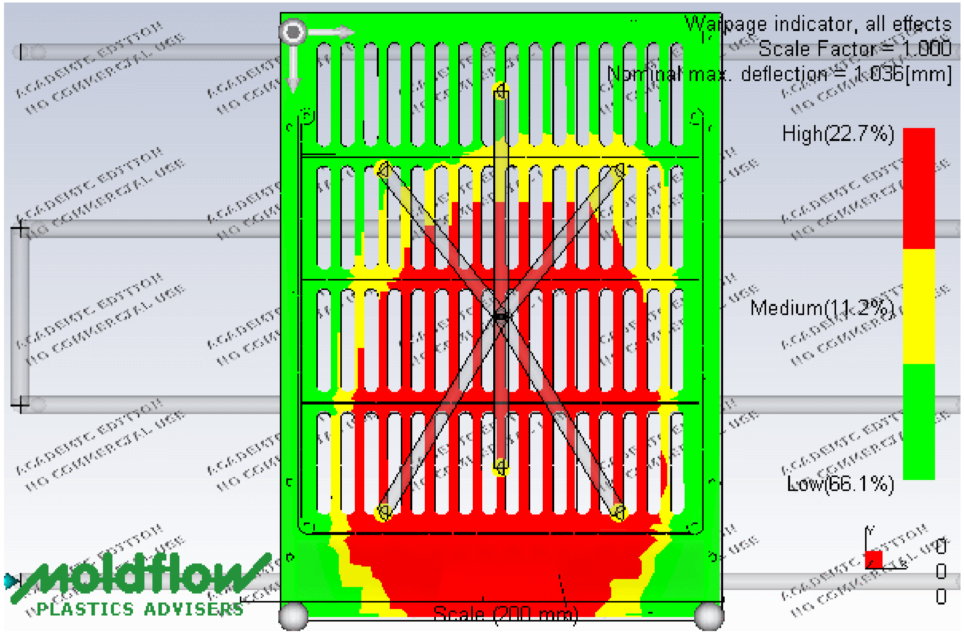


Fig 4.3: Warpage

4.2 Actual injection experiment result

The actual injection experiment use injection machine brand name Arburg 520c Allrounder 2000-800 horizontal type. By using the parameter from the software analysis the machine will inject the part and the result is observed. There are three main parameters that has been setup on the machine such as injection flow in unit of ccm/s, injection pressure in unit of bar and dosage volume at ccm. The parameter that has been set in the machine is described as in Table 4.4. For this experiment the cooling and cycle time value is remain constant as the result from the simulation software where it does not influence the filling analysis. The cooling time is taken approximately due to the simulation software result which at 17.63 s and decided as 18.00 s. The injection indicates the speed of the flow by ccm per second and in this case the material is flowing at 110 ccm per second but this parameter also does not affect significantly the fill analysis result.

Table 4.4: Actual injection analysis result

Parameters	Average value
Dosage volume (ccm)	350.00
Injection pressure (bar)	1300.00
Injection flow (ccm/s)	110.00
Cooling time (s)	18.00
Cycle time (s)	35.00
Clamp force (MPa)	2000.00

4.3 Comparison between simulation software and actual injection.

As the title of this project is benchmarking between both analyses result thus in this section the comparison is done by two category which are comparing the parameters after finished product have been obtained and the second category is by comparing the visual physically results of both analyses. For the parameter category it will include the value for dosage volume, injection pressure, injection flow, cooling time, cycle time and clamp force as those will affect the finished product. Then the second category will observe the physical result of the injected product and the prediction figure by the plastic simulation software.

4.3.1 Comparison of parameters

The parameters to inject the finished product for both analyses are compared in this section. The results are picked when the product is fully filled into finished product. The data for both analyses will be compared by the Table 4.5.

Table 4.5: Comparison of results from actual injection and software simulation

Parameter	Software simulation	Actual injection	Differences
Dosage volume (ccm)	315.9390	350.00	+34.061
Injection pressure (bar)	1208.86	1300.00	+91.140
Injection flow (ccm/s)	110.00	110.00	0.000
Cooling time (s)	17.63	18.00	+0.370
Cycle time (s)	35.00	35.00	0.000
Clamp force (33one)	218.946	200.00	-189.460

As the analysis use the exact value of the parameter from the simulation software the cavity will be not fully filled and short shot is present as shown in Fig 4.4 as lack of pressure and dosage volume during the actual analyses. It is known that there are some errors for the results of simulation software as there are present of different values of results. The mold also in average condition and not well maintained as have not been in used for two years made the mold was rusty and interfere with the result of actual injection analysis. However several precaution step and maintenance have been made especially the rusty area at one of the branch runner area. It also noticed that the value of clamping force generated by the simulation software is slightly higher than the actual injection at 18.946 tonne since the machine is only capable provide 200 tonne maximum of clamping force. It resulting in flashing defect onto the actual injection product where the clamping force is not enough in clamping the moveable and fixed plate together thus some of the molten plastic injected through the gap of less clamping force as shown in Fig 4.5.

Fig 4.4: Injected product by using simulation software result

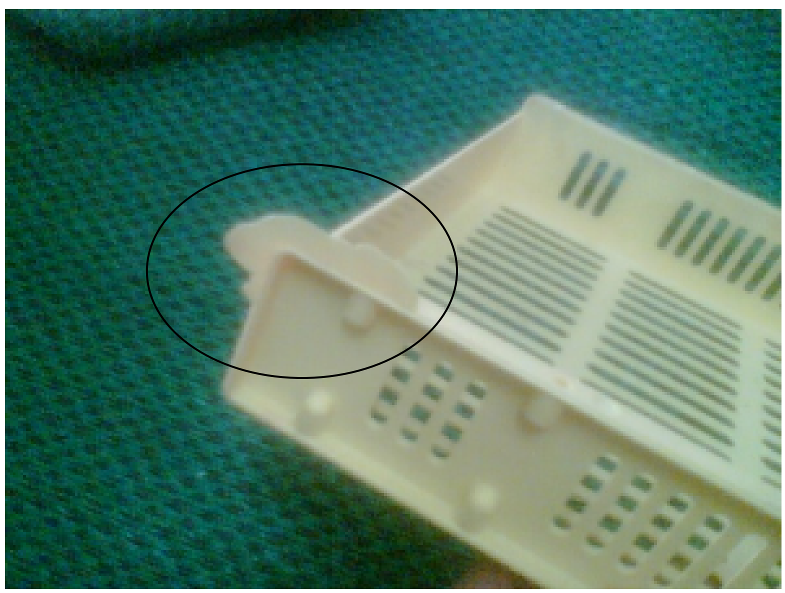


Fig 4.5: Flashing defect

4.3.2 Comparison of physical analysis result

The physical result of both analyses is compared by using the generated figure from simulation software and the actual injection. The parameters can be observed are limited such as defects as air trap [Fig 4.6], weld line [Fig 4.7], warpage and sink mark [Fig 4.8]. From the actual injection the air trap defects mostly are eliminated by the venting system of the mold. The vent hole aiding the escape of the air or gas inside of the cavity and prevent the bubbles trapped inside the molten plastic become the source of the air trap defects. Although the defects present in the software analysis but the defects does not present on the actual injection product. Warpage defect also present on both analyses at the same area and mostly accurate at the value of deflection. Sink mark defect shows accurate result for both analyses where the area covered by the defect on actual product as same as simulation software estimated. The weld line defect also occurred on actual injection product where there areas covered by the defects are almost the same as estimated by the simulation software.

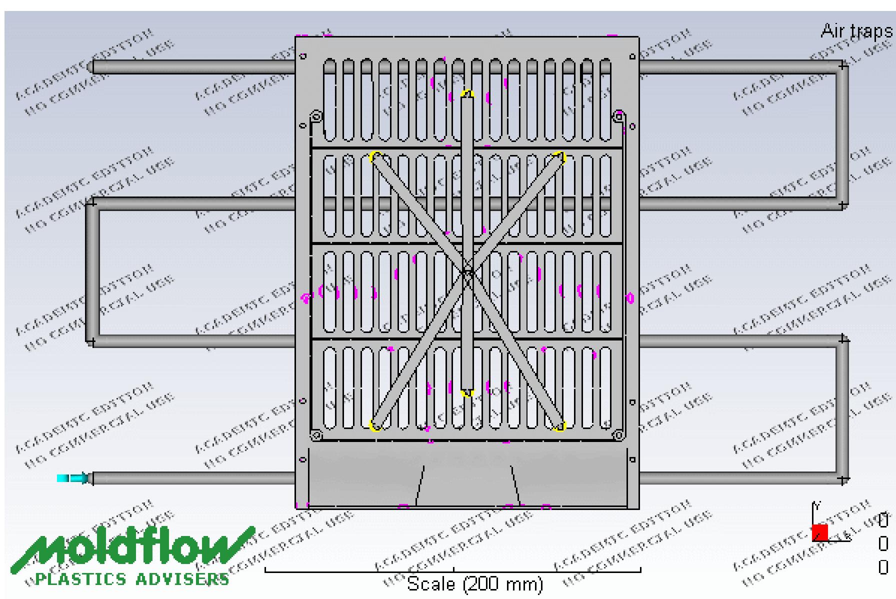


Fig 4.6: Air trap defect indicated by software in pink round shape

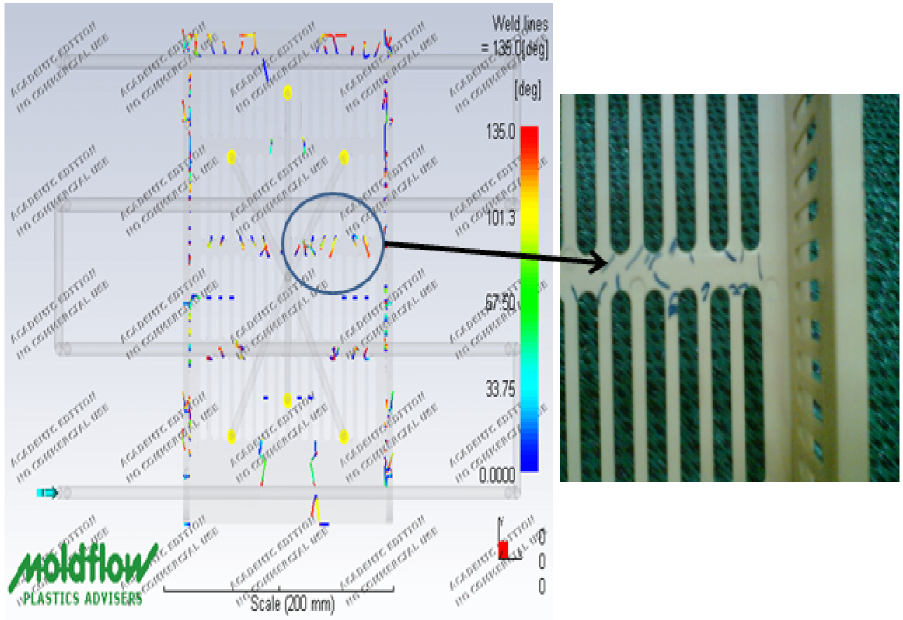


Fig 4.7: Weld line estimated by the colored line

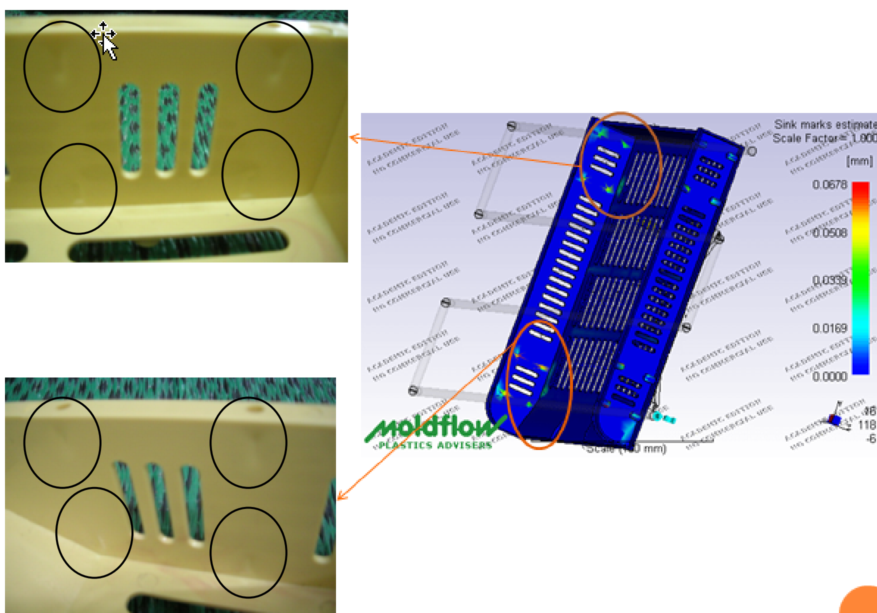


Fig 4.8: Sink mark on both analyses

4.4 Error percentage calculation

The error percentages are calculated based on the result of actual injection as following calculation step for dosage volume and injection pressure. Both of these parameters are considered important in this analysis and the only parameters taken into account in calculation because those are mostly influencing in the analyses. In the other hand, the other parameters such as cooling time although vary for both results are not significantly affecting the analyses thus the differences are negligible.

4.4.1 Dosage volume error percentage calculation

$\frac{34.061}{350}$ x 100.= 9.73% error

4.4.2 Injection pressure error percentage calculation

$\frac{91.140}{1300}$ x 100 = 7.01% error

From the calculation it can be estimated that the simulation software has error percentage at range of 7.01 to 9.73 percent or also can be put as 90.27 to 92.99 percent accurate. These errors can be resulted from several factors. The condition of the mold which is already wear because of the lifespan cycle affect the flow of the material where the cavity for the runner and branch runner are rusty and interfering the ejection process which increase the time consumption per cycle which resulting inconstant molten material temperature. Human error occurred when measuring the dimension of the mold where it is one of the crucial factor in determine the accuracy of the results such as the dimension of the gate can be considered as 1 mm of diameter but the accurate measuring can be some offset from the measured dimension. Moldflow is one build by human and it owned limited capabilities.

CHAPTER 5

CONCLUSION AND RECOMMENDATION

5.1 Conclusion

The results for simulation software are obtained after several run because of some errors. The results are acceptable with logically value and can be used for actual injection experiments. The results are compiled together according to the categories which are by physical and parameters comparison and the differences of the results determined by calculations and comparing the figures.

Designing the model has been through several stages and can be concluded that assembly type of drawing by mating many parts into one whole product with IGES type of file format are no suitable combination for this simulation software. The software capable of read the file but it misinterprets the model thus the result generated are entirely inacceptable. The model should be draw into STEP file format.

The preparation for both analyses is done by doing the literature study based on injection molding books and articles from previous research. The previous research which including the use of simulation software and plastic injection molding indicates that there are very rare of 100% same of results for software analysis and actual injections obtained. However the results are not totally wrong but it can be the guidelines or benchmark for the actual injection setup rather than start from scratch which consumed a lot of time.

The simulation software analysis results are acceptable since there are absent of errors during the analysis and the offset of the values for the parameter between simulation software and actual injection are at minimal. The parameters that need to be setup onto the software such as dimension of the model and mold, dimension and layout of the sprue and runner, melting temperature and mold temperature should be setup as accurate as possible to get the most accurate results from the software. Thus the values should be manipulated with details observation.

Actual injection of the paper rack process using the results generated by the simulation software produces short shot defects as it lack of pressure and dosage volume. Thus the values from the software act as the guideline to get the finished product. Several fine products injected to get the average results.

As for the conclusion, it is proved that during the analysis both results are not same and if the result from the actual injection taken as benchmark then it can be concluded that the result from the Moldflow Plastic Advisor are not accurate 100%. Even though, at range of 90.27 to 92.99 percent accurate for the result of simulation software can be used as a benchmark or guideline during setup the parameters onto the plastic injection molding machine rather than using try and error method which consume a lot of times and energies into waste.

The source of the errors are significantly influenced by condition of the mold where flow of the material interrupted and interfering the ejection process, technical error such as miscalculated mold dimension also interrupts the results as the simulation bound to limited capabilities.

5.2 Recommendation

Future work is one of the elements in order to enhance or further one project to a better state by the next generations. As an example a project can be enhanced in methodology to get new data and information which is lack in the previous project.

Normally the future works for this type of project are limited or almost does not have any future works. Though, one can use other plastic simulation software the software simulation analysis or use better condition mold with bigger size of product to get more accurate results since plastic simulation software are more prefer to be used for big products.

Several precautions are appropriate for this project in order to further it to the next stage. The mold should be in good condition and well maintained before one choosing the mold as a subject for the project to ensure the smooth flow of the project progress. The software and machine availability also needs to be checked and scheduled as same as preparation in termed of technical knowledge to countermeasure further difficulties smoothly.

REFERENCES

[1] Moldflow user help

[2] Mold Engineering 2nd Edition, Herbert Rees, HANSER, 2002

[3] M.S.Mohamed Reza Zalani, Plastic Technology, BMF 4713 Teaching Handout (2008)

[4] http://en.wikipedia.org/wiki/Injection_molding (24/1/08)

[5] G.A.Britton, Y.C.Lam and Y.-M.Deng, Specification Model: A First Step Towards Automatic Interpretation of Injection Molding CAE Software, 2003

[6] Chen,W.C. et al., Process parameter optimization for MIMO plastic injection molding, Expert Systems with Applications

[7] Chae Hwan Kim, Jae Ryoun Youn, Determination of residual stresses in injection-molded flat plate: Simulation and experiments, 2007

[8] C.K. Au, A geometric approach for injection mould filling simulation, 2004

[9] M.C. Song, Z. Liu, M.J. Wang, T.M. Yu and D.Y. Zhao, Research on effects of injection process parameter on the molding process for ultra-thin wall plastics parts, 2006

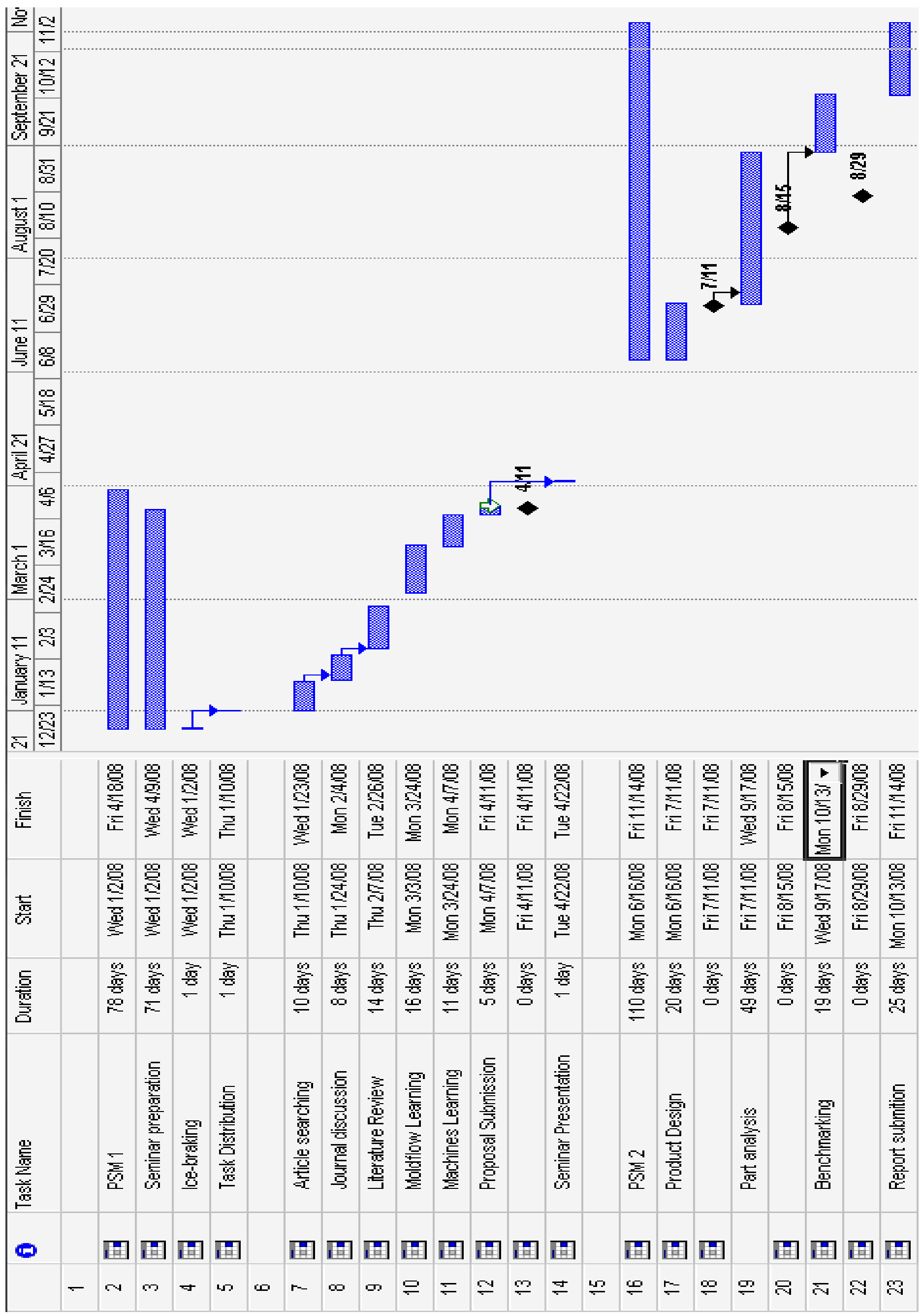
[10] J.K.L.Ho·K.F.Chu·C.K.Mok, Minimizing Manufacturing Costs For Thin Injection Molded Plastics Components, 2004

[11] http://www.efunda.com (15/3/08)

APPENDICES

Appendix A

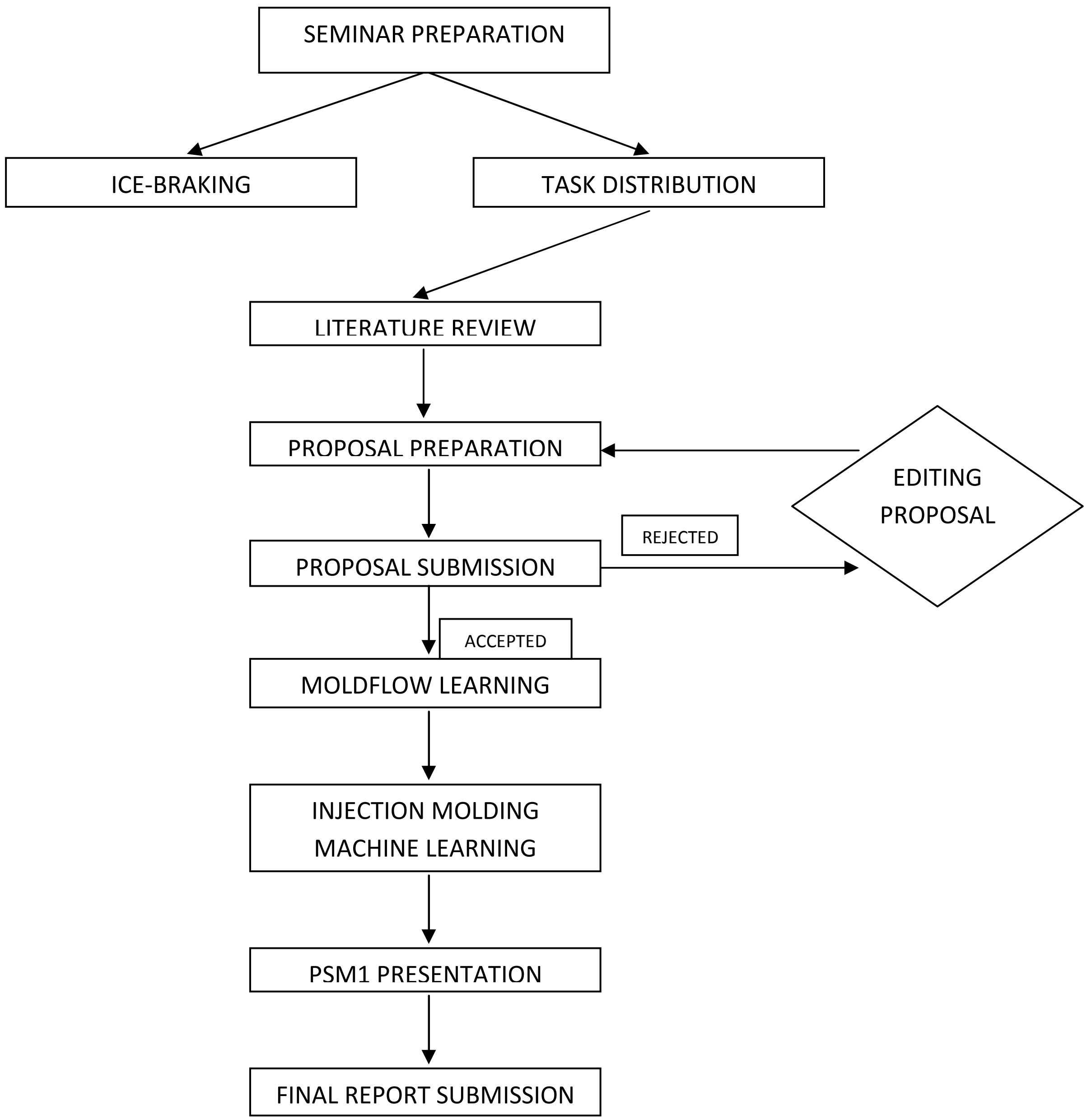
Final Year Project Gantt Chart



Appendix B

Final Year Project Flow Chart

PSM 1 Flow Chart



PSM 2 Flow Chart