SUPERVISOR 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	: Zamzuri Hamedon
Date	

Signature	: ……………………………………
Name of Panel	:
Date	:

STUDENT 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 of candidate	: Hudzaifah Mohd Azam
Date	:

DEDICATION

When a Man Lies He Murders

Some Part of the World

These Are the Pale Deaths Which

Men Miscall Their Lives

All this I Cannot Bear

to Witness Any Longer

Cannot the Kingdom of Salvation

Take Me Home

"To my beloved Father and Mother"

ACKNOWLEDGEMENTS

First and foremost**,** I feel grateful to Allah s.w.t. because this paper has successfully completed in the given. I would like to convey my full appreciation and thankful to my supervisor, Mr. Zamzuri Hamedon for his attention and guidance on teaching and delivering his knowledge related to this paper and lead me for encouragement, support, critics and advise to complete this paper. Without his motivation, I could not finish this paper as presented here. Besides that, I would like to dedicated my appreciation to all the people that that helping me in learning the Moldflow software including Mr Mohamed Reza Zalani, my plastic technology lecturer for sharing his knowledge about the software. Lastly, I would like to give my appreciation to *Khairul Izwan Zulkafli, Wan Mohd Firdausi Wan Sembak, Hazwan Aizat Hussin, Zul Amri Abu Bakar, Mohd Adzwan Mohd Nor* who are my fellow friends under same supervision by Mr. Zamzuri Hamedon. With their supports, help and advice keep me strong and confident for completing this paper. Also a special thanks to my beloved family for their endless supports for me to finally finished this project

ABSTRACT

This project presents the Simulation Study of the Plastic floor tile for Two plate and Three plate mold using Plastic simulation software. The product design the analysis is hexagonal in shape, and has interlocking features for attaching purpose. The product was designed using Solidwork software, Version 2005. Analyzing of the product was completed using Moldflow software, which is specialize in simulating the broadest range of injection molding processes to predict and avoid potential manufacturing defects. For the analysis in this paper, the Moldflow Plastic Adviser is used as the main software for analyzing the product designed. The perimeter to consider in this study is the two plate and three plate mold type of cold runner injection molding. The three plate mold is believed to be better than two plate mold in term of mass production and quality. The injection perimeters to be considered are actual filling time, actual injection pressure, cycle time, maximum clamping force during cycle, runner system volume and shot volume. The best type of mold with the best injection parameters that save cost is selected to produce the product, which is a plastic floor tile for sport usage.

ABSTRAK

Projek ini membentangkan tentang “Pengajian Simulasi tentang Tile Lantai Plastik untuk Acuan Dua Keping dan Acuan Tiga Keping menggunakan Perisian Simulasi Plastik“. Produk yang direka untuk dianalisa berbentuk heksagon, dan mempunyai bahagian yang membolehkan bahagian produk tersebut bercantum. Produk tersebut direka menggunakan perisian *Solidwork*, versi 2005. Proses menganalisa produk tersebut dijalankan menggunakan perisian *Moldflow Plastik Adviser,* yang dikhususkan dalam penganalisaan pelbagai proses injeksi acuan untuk meramal dan menghindari kesan – kesan buruk pada produk dalam proses pembuatan. Untuk projek ini, perisian *Moldflow Plastik Adviser* telah dijadikan sebagai perisian utama untuk menganalisa produk yang direka. Paramiter yang perlu dikaji ialah diantara acuan dua keping dan acuan tiga keping dari jenis *cold runner* dalam proses injeksi. Acuan tiga keping dipercayai lebih baik dari segi kualiti dan pengeluaran secara besar- besaran. Paramiter-paramiter dalam injeksi acuan yang perlu dikaji ialah masa pemenuhan, tekanan semasa proses injeksi, masa kitaran sesebuah proses, jumlah tekanan maksima oleh mesin, jumlah sistem *runner* dan jumlah bahan yang diinjek. Acuan yang mempunyai paramiter- paramiter injeksi acuan yang paling baik dan menjimatkan akan dipilih untuk menghasilkan produk, iaitu tile lantai plastik untuk kegunaan sukan.

TABLE OF CONTENTS

LIST OF FIGURES

LIST OF TABLES

LIST OF ABBREVIATIONS

MPA	Moldflow Plastic Adviser
MPI	Moldflow Plastic Insight
ABS	Acrylonitrile-Butadiene-Styrene
PVC	Polyvinyl chloride
TPE	Thermo Plastic Elastromer
FEA	Finite Element Analysis
mm	millimeter
lit/min	litter per minutes
C	Celsius
Mpa	Mega Pascal
cm^3	centimeter cubic

LIST OF APPENDIXES

APPENDIX A

GANTT CHART FOR SIMULATION STUDY OF THE PLASTIC FLOOR TILE FOR TWO PLATE AND THREE PLATE MOLD USING PLASTIC SIMULATION SOFTWARE

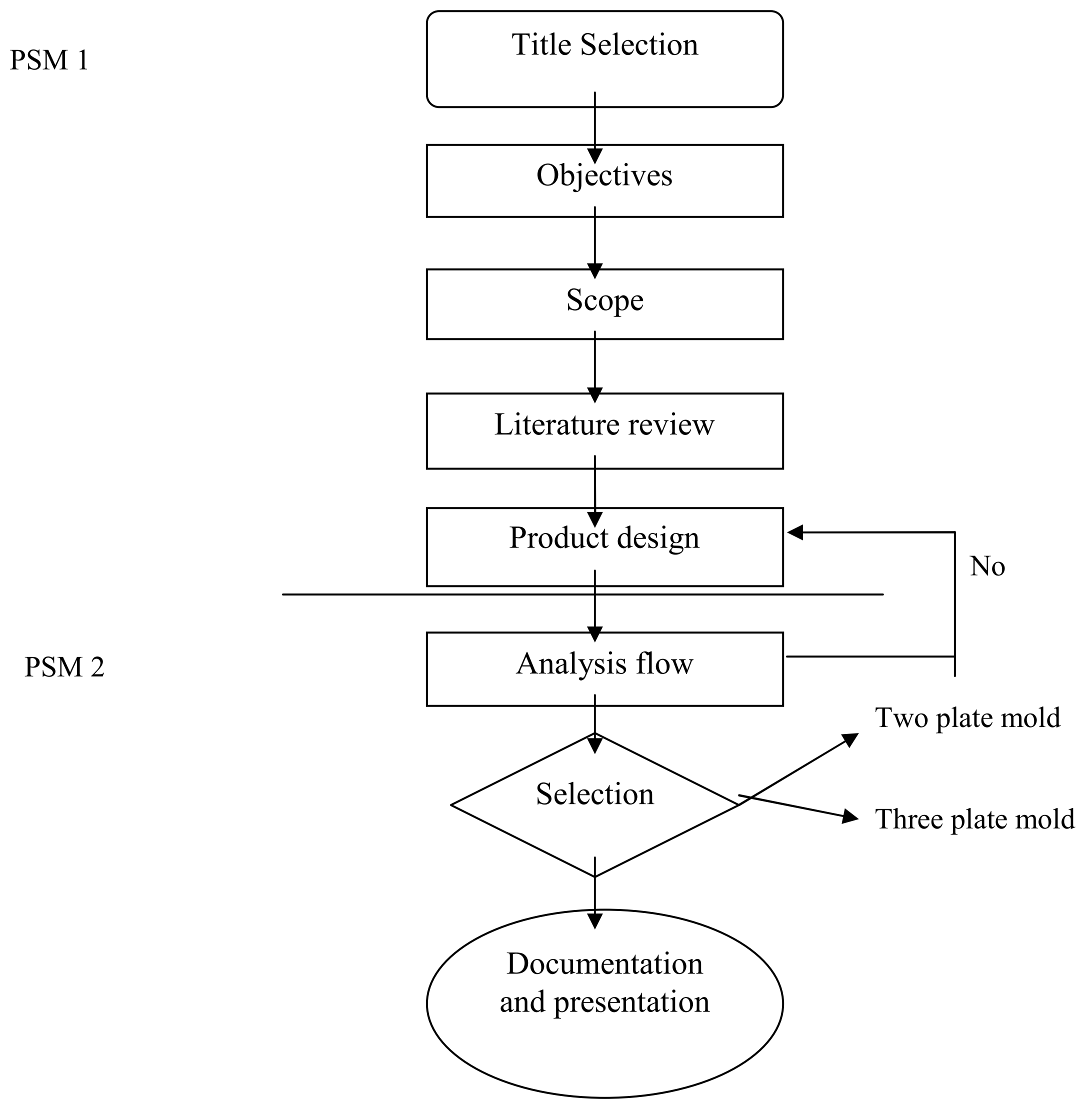
PSM 1

Project Activities	Jan				Feb				Mar				Apr			
	W 1	W 2	W 3	W 4	W1	W 2	W 3	W 4	W 1	W 2	W 3	W 4	W 1	W 2	W 3	W 4
Verify Topic	■															
Discuss about the literature review		■														
Submit the literature review			■													
Discuss about the project introduction				■	■											
Mid sem holiday						■										
Submit the introduction and literature review						■	■									
Submit the objective, scope, literature review and methodology							■									
Learn about the MOLDFLOW software								■	■	■						
Design the product to analyze										■						
Finish the full report											■					
Do the slide for presentation												■	■			
Presentation and submit the full report														■		

PSM 2

Project Activities	June				July				August				September				October			
	W1	W2	W3	W4	W1	W2	W3	W4	W1	W2	W3	W4	W1	W2	W3	W4	W1	W2	W3	W4
Holiday.	■	■	■	■	■															
continue from PSM 1					■	■														
Analyze the product							■	■	■	■	■	■	■							
make the selection of mold														■	■					
Hari Raya Puasa break.																■	■			
Finish the full report.																	■	■		
Prepares for slide presentation.																			■	
Presentation and submit full report.																			■	■

APPENDIX B


PSM 1
Title Selection
Objectives
Scope
Literature review
Product design
No
PSM 2
Analysis flow
Two plate mold
Selection
Three plate mold
Documentation
and presentation

CHAPTER 1

INTRODUCTION

1.1 INTRODUCTION

This project title is simulation study of the plastic floor tile for two plate and three plate mold using plastic simulation software and focused on getting the best result to produce the plastic floor tile between the two plate and the three plate mold using plastic simulation software such as Moldflow software. The design will be in hexagonal shape, with the interlocking features for the attaching purpose. For having the most optimize mold design, the plastic simulation software will be use to determine and analyze, whether the two plate or the three plate mold is most suitable to produce the plastic floor tile. Therefore, the Moldflow analyzing ability is useful to determine the manufacturing feasibility of the design.

1.2 OBJECTIVES

1. To get the best result to produce the plastic floor tile between two plate and three plate mold using plastic simulation software.
2. To analyze the result using Moldflow software.
3. To choose between the two plate and three plate mold as the mold to produce the product, which is the plastic floor tile

1.3 PROJECT SCOPE

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

1. Analyze between the two most common type of cold runner mold, the two plate and the three plate mold
2. Using the Moldflow Plastic Adviser (MPA) as the main software to analyze and get the best results.
3. Comparison between the two plate mold and the three plate mold, and choosing the mold with the results according to the parameters, material effects and cooling system effects.

1.4 PROBLEM STATEMENT

The trends of producing a plastic product in injection molding industries are recently changing from traditional method to using FEA analysis. The used of simulation software run concurrently with designing and fabrication become more popular and applicable. For example, to produce a plastic floor tile with the right mold type to optimize the production and also the profit from the products, there are many aspects to be considered.

- For injection molding industries, time and cost is very important aspects to consider because these two aspects will directly related to the profits of a company.
- The normal selection process for the type of mold needed to use usually take time and increase cost, thus will be a lost if the wrong type of mold is selected to inject a product

- Using Moldflow software with the ability to analyze, test and even perform detailed part cost estimation can help the users to choose the most suitable mold type according to their capabilities, thus can save time and cost of selecting a mold to used to fabricate a product.

CHAPTER 2

LITERATURE REVIEW

2.1 INTRODUCTION

In this chapter, the topics discussed are about the plastic floor tile, plastic injection simulation software and the injection molding. This chapter will be organized as the following 2.1 for example of plastic floor tile, 2.2 for the plastic injection simulation software and 2.3 for introduction to the injection molding

2.2 EXAMPLE OF PLASTIC FLOOR TILE

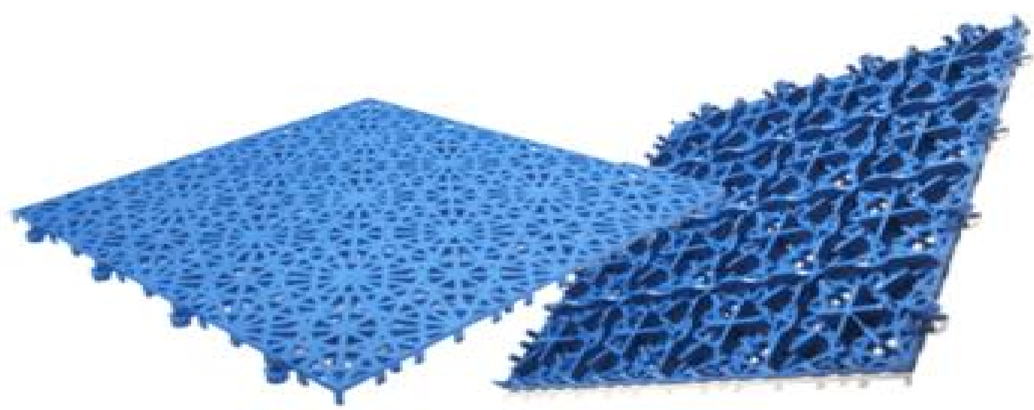


Figure 2.1: Example of plastic floor tile (TactTiles Sports Floors, 2008)

2.2.1 Plastic floor tile surface description

The plastic floor tile can be used indoors and out and is a practically maintenance free surface. Ideal for all ball games, this tile provides good traction and ball bounce, combined with free drainage and an anti-slip surface. It can be laid outside over any hard, draining surface and is ideal as a refurbishment surface in sports halls.

2.2.2 General Requirement

The product must have a very accurate dimension especially at the junction parts to ensure that the connection between one piece and another is flat. This is to make sure that the surface of the combined tiles does not have bumps. The joining edge must be close to make sure that there are no holes or large spacing between one and other tiles. The material properties of the plastic floor tile must be suitable for sports usage such as the ductility, strength, durability and plasticity of the material for correct bouncing over the whole parts. The tile must be easy to install or remove for maintenance purpose

2.2.3 Examples of Uses

- Five-a-Side Football
- Basketball
- Netball
- Handball
- Volleyball
- Tennis
- Badminton

2.2.4 Edges

Edges are very critical in having a smooth, flat surface. The junction section at the edge must be accurate to provide a neat finish and prevent a trip hazard. Edges can be attached onto any side of the plastic sport tile.

2.2.5 Expansions

In order to allow for thermal movement, Expansions should be inserted into the plastic floor. Crosspieces are available for intersections. This will allow the plastic floor tile to slightly expand without damaging the tile.

2.3 PLASTIC INJECTION SIMULATION SOFTWARE

Moldflow's is a design and analysis software that is widely used in plastics injection molding industry nowadays. This includes Moldflow Plastics Advisers (MPA) and Moldflow Plastics Insight (MPI). Overall, Moldflow products can be used to simulate the broadest range of injection molding processes to predict and avoid potential manufacturing defects .This software has proven its efficiency because the simulations available are very close to real-life situation.

2.3.1 Moldflow Plastic Adviser (MPA)

Moldflow Plastics Adviser (MPA) is easy to use plastics simulation software that allows the users to optimize their designs during the earlier stages of new product development. This software is the ideal tool to quickly check the manufacturability of your plastic part design when the cost of change is least in the early design process. Users can get quick feedback on how modifications to wall thickness, gate locations, material or geometry can affect the filling pattern and pressure and temperature distributions in the part cavity. The analysis results and detailed design advice can be used to determine the optimum part thickness and gate locations as well as to identify and eliminate cosmetic issues such as weld lines, air traps and sink marks. [1]

2.3.2 Moldflow Plastic Adviser analysis capabilities

- Plastic Filling Analysis

  This analysis 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

- Sink Marks Analysis

  This analysis indicates the presence and location of Sink Marks (and Voids) likely to be caused by features on the opposite face of the surface. Sink marks typically occur in moldings with thicker sections, or at locations opposite ribs, bosses or internal fillets. The analysis does not indicate sink marks caused by locally thick regions

- Cooling Quality Analysis

  The Cooling quality analysis shows the users where heat tends to stay in a part due to its shape and thickness. The part is considered to be located in the center of a block of metal, or a theoretical mold, without any cooling circuits and held there for a fixed period of time. The result shows the way heat will leave the hot part naturally and flow towards the extremities of the block.

- Gate Location Analysis

  This analysis shows the users the best place to set the gate location at the model being analyzed and can be a guidance for the user to get the best location to produce a product.

2.3.3 Advantages

The MPA software can be used to evaluate the manufacturing feasibility of every design. This software will allow the user to test, analyze and optimize the mold design in the early stage of the product development even before the mold is produce.

This software has the most complete material database for plastic simulation and can be used to identify the most suitable plastics material candidate. The users can easily compare the properties of two or more material and select the best material to use.

With the ability to optimize the part wall thickness to achieve uniform filling and the minimum cycle time, the software is very useful because uniform filling is critical to produce product with good quality, especially in family mold. With this software, the user can determine the best wall thickness of the part designed to achieve uniform filling and minimum cycle time

Identify and eliminate cosmetic defects is important because defects such as sink mark will affect the visual quality of the product, which is undesirable especially in aesthetic surfaces. The MPA software can identify cosmetic defects such as sink marks, weld lines, and air traps as early as at the designing process.

This software can help users to estimate the total job cost. It considers resin cost, mold manufacturing cost, molding machine operating cost. A cost summary indicates the percentage of the total cost that is associated with each of these variables.

The best gate location can be determined using this software by analyzing the part design and the software will automatically locate the potential or best gate location for the part analyzed. The user can relocate the gate location to minimize the number of weld lines.

This software is able to design and analyze all type of hot and cold runner system, including the two and three plate mold, which is in the cold runner type.

By analyzing the product design, this software can predict the minimum cycle time, the suitable clamp tonnage and the optimum shot volume for the mold design to produce the product with best quality and reduce the manufacturing waste.

The Moldflow Plastics Adviser (MPA) module allows the users to create and simulate plastic flow through single cavity, multi-cavity and family molds. Users can optimize gate type, size and location as well as runner layout, size and cross-sectional shape. Analysis results such as cycle time, clamp tonnage, and shot size is available in this software. All of this will help the users to size the injection molding machine, minimize cycle times and reduce manufacturing waste. [2]

The Moldflow Mold Adviser modules allow users to simulate more phases of the injection molding process and evaluate molded part performance and cooling circuit design. This will help the users to;

1) Increase the confidence the user's part design can be manufactured
2) Reduce part cost by lowering the amount of material used without compromising quality
3) Improve part strength and quality with a design that is optimized to the users production capabilities and selected material characteristic.

2.4 INJECTION MOLDING

Injection molding is a manufacturing technique for making parts from both thermoplastic and thermosetting plastic materials in production. Molten plastic is injected at high pressure into a mold which is the inverse of the product's shape. After a product is designed by an Industrial Designer or an Engineer, molds are made by a moldmaker (or toolmaker) from metal, usually either steel or aluminum, and precisionmachined to form the features of the desired part. Injection molding is widely used for manufacturing a variety of parts, from the smallest component to entire body panels of cars. Injection molding is the most common method of production, with some commonly made items including bottle caps and outdoor furniture. [3]

The most commonly used thermoplastic materials are polystyrene (low cost, lacking the strength and longevity of other materials), ABS or acrylonitrile butadiene styrene (a co-polymer or mixture of compounds used for everything from Lego parts to electronics housings), nylon (chemically resistant, heat resistant, tough and flexible - used for combs), polypropylene (tough and flexible - used for containers), polyethylene, and polyvinyl chloride or PVC (more common in extrusions as used for pipes, window frames, or as the insulation on wiring where it is rendered flexible by the inclusion of a high proportion of plasticiser). [3]

Injection molding can also be used to manufacture parts from aluminum or brass. The melting points of these metals are much higher than those of plastics; this makes for substantially shorter mold lifetimes despite the use of specialized steels. Nonetheless, the costs compare quite favorably to sand casting, particularly for smaller parts [3]

Mould making is an important supporting industry because their related products represent more than 70% among the components in consumer products. The high demand for shorter design and manufacturing lead times, good dimensionality and overall quality, and rapid design changes has become the bottlenecks in mould industries. It is a complicated process, and required skilled and experienced mould maker.

Injection molding machines, also known as presses, hold the molds in which the components are shaped. Presses are rated by tonnage, which expresses the amount of clamping force that the machine can generate. This pressure keeps the mold closed during the injection process. Tonnage can vary from less than 5 tons to 6000 tons, with the higher figures used in comparatively few manufacturing operations. [4]

The basic process in injection mold is heat a thermoplastic material until it melts and forces it into a hollow (cooled) cavity under pressure to fill the mold. After cooling down, the finished part is removed. [5]

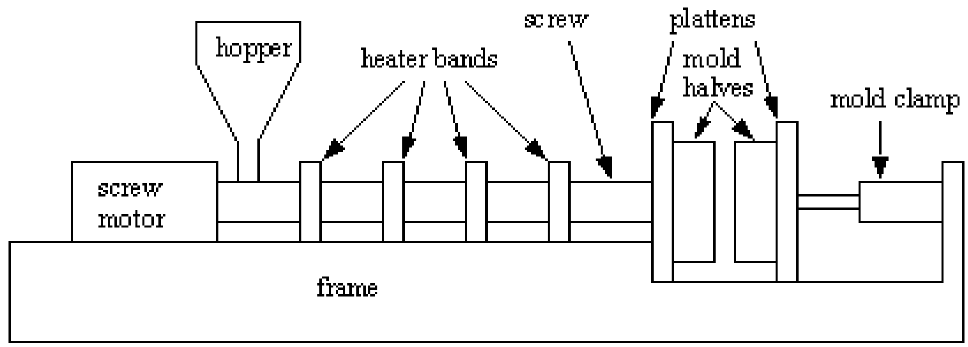


Figure 2.2: Basic injection molding machine (Injection molding, Manufacturing engineering on a disk, 2008)

A typical injection molding machine shown in Figure 2.1 with the covers removed. Plastic pellets are poured in the hopper, and finished parts emerge from the molds.

Injection system,

1. a material hopper acts as an input buffer
2. a heated chamber (heater bands) melt the material
3. an injector forces the now viscous fluid into the mold

2.4.1 Mold

Molds are typically constructed from hardened steel, pre-hardened steel, aluminum, or beryllium-copper alloy. The choice of material to build a mold is primarily one of most economics. Steel molds generally cost more to construct, but their longer lifespan will offset the higher initial cost over a higher number of parts made before wearing out. Pre-hardened steel molds are less wear resistant and are used for lower volume requirements or larger components. [3] A simple mold is shown in Figure 2.2

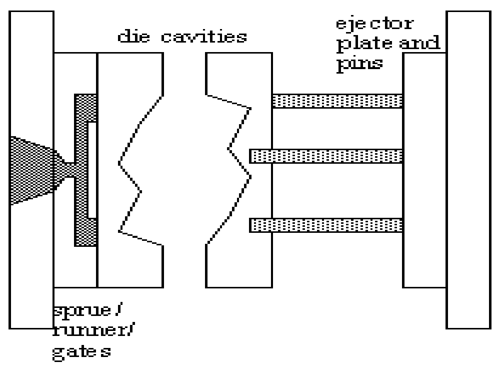


Figure 2.3: A simple injection mold (Injection molding, Manufacturing engineering on a disk, 2008)

Table 2.1: Lists of the parts of a mold and functions.

Parts	Functions
Locating ring	Guides the injection nozzle into the mold.
Sprue Bushing	Where the injected material enters the mold cavities
Clamp front plate	Secures the front cavity, locating ring, and other components to the stationary platen.
Front cavity	Holds half of the negative of the shape to be molded.

Rear cavity	The mating half for the front cavity that completes the negative of the final part. Guide pins are mounted on this to ensure correctly aligned cavities.
Spacer Blocks/Rails	Used to separate the rear cavity from the rear clamp plate
Ejector housing	Contains the ejector pins to knock the parts out of the mold and forces the cavity back when the mold is closed.
Rear Clamp Plate	Supports the rear half of the mold on the moving platen, and provides rigidity under molding pressures.

Components to consider in mold design,

1) Part design

   This is very important because all the parameters at the part such as thickness, length and shape will effect the cycle time of the mold

2) Material

   The material used is very critical because every material has its own properties such as strength and melts temperature. Different material has different cooling time.

3) Machine used

   Every machine has its own maximum injection pressure, maximum clamping pressure and maximum opening. Type of machine is different according to the users need.

Factors that are often altered in the design are,

1) Runners

   Runners can be redesign or change to get the best quality of product according to the number of cavity.

2) Mold cooling

   Cooling system is important because good cooling system can shorten the cycle time of the mold and can reduce the warpage for the product produce.

Runners carry the plastic to the injection gates and are often considered disposable or reusable. Typical runner systems are,

1) Cold runner

   For the cold runner molds, the material is heated outside the mold, at the injection system which is at the heated chamber.

2) Hot runner

   The hot runner molds has filaments at the runner in the mold to maintain the molten plastic temperature as the molten plastic filled the cavity.

There are two main types of injection molds: **cold runner (two plate and three plate designs)** and hot runner – the more common of the runnerless molds. The significant difference is the presence of a sprue and runner with every molded part in the cold runner type. This extra molded component must be separated from the desired molded part [4]

2.4.1.1 Two Plate Mold

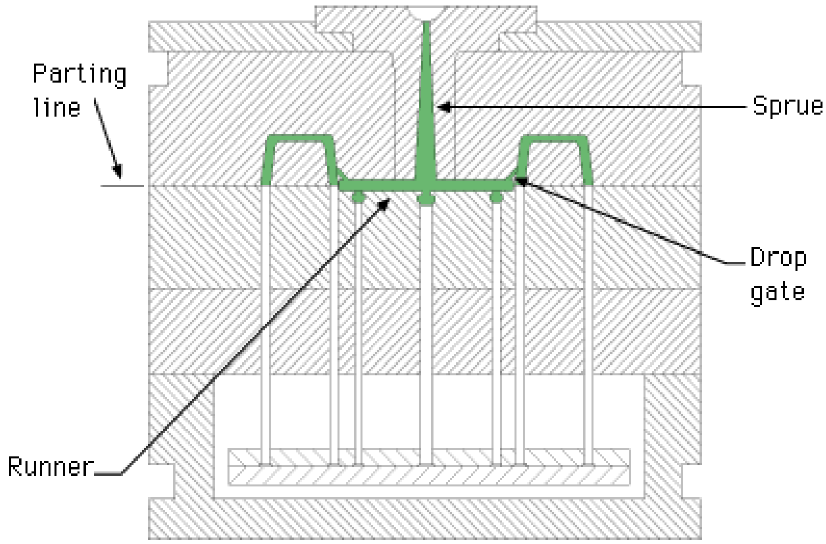


Figure 2.4: Two plate mold (RTP company, tech info 2008)

This is the simplest type of mold available because it only has one parting line and the mold splits in two halves. The mold's runner must be located at the parting line, thus the part produce can only be gated on its perimeter only. [3]

The conventional two-plate mold consists of two halves fastened to the two platens of the molding machine's clamping unit. When the clamping unit is opened, the two mold halves open, the most obvious feature of the mold is the cavity, which is usually formed by removing metal from the mating surfaces of the two halves. Molds can contain a single cavity or multiple cavities to produce more than one part in a single shot. The figure shows a mold with two cavities. The parting surfaces (or parting line in a cross-sectional view of the mold) are where the mold opens to remove the parts. [6]

A mold must have a distribution channel through which the polymer melts flows from the nozzle of the injection barrel into the mold cavity. The distribution channel consist of a sprue, which leads from the nozzle into the mold; runner, which lead from the sprue to the cavity; gates that constrict the flow of plastic into the cavity. (There are one or more gates for each cavity in the mold) [6]

2.4.1.1.1 Advantage

The two mold design is very simple because it only has one parting line and much more cheaper than the three plate mold design in term of price and cost to operate. The mold requires less maintenance and less skill to set up and operate.

2.4.1.1.2 Disadvantage

In two plate mold, the runner is still attached with the product after the injection process is finished. This will add the labor cost to cut the runner from the product and of course will increase the production cost for the product.

2.4.1.2 Three Plate Mold

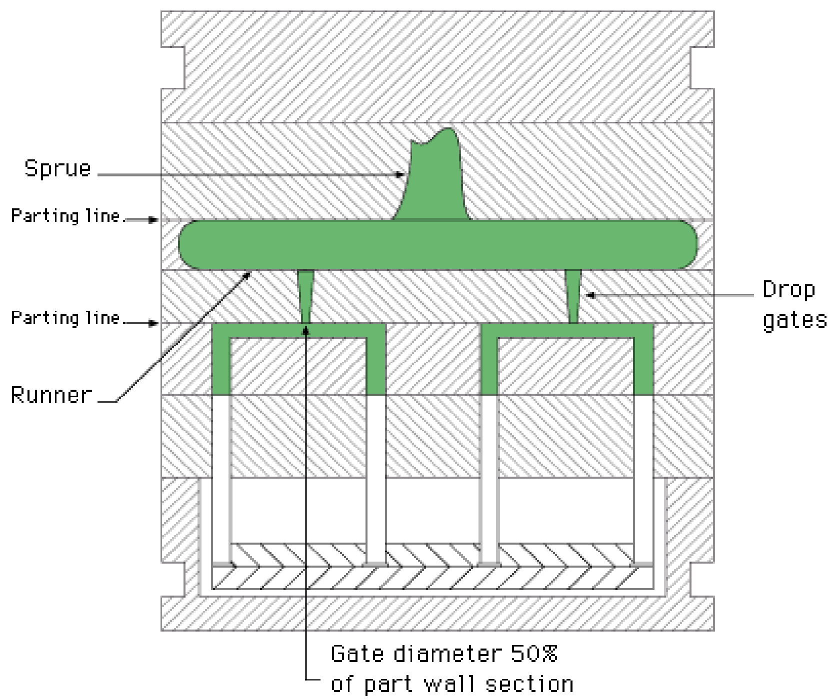


Figure 2.5: Three plate mold **(**RTP company, tech info 2008)

The three plate mold differs from the two plate because it has two parting planes, and the mold splits into three sections every time the part is ejected. Since the mold has two parting planes, the runner system can be located on one, and the part on the other. Three plate molds are used because of their flexibility in gating location. A part can be gated virtually anywhere along its surface. [3]

The flow of molten plastic in the three plate mold is through a gate located at the base of the cup-shaped part, rather than at the side. This allows more even distribution of melt into the sides of the cup. In the two-plate mold, the plastic must flow around the core and join on the opposite side; possibly creating a weakness at the weld line. The three-plate mold allows more automatic operation of the molding machine. This forces disconnection of runner and parts, which drop by gravity (with possible assistance from blown air or a robotic arm) into different containers beneath the mold [6]

2.4.1.2.1 Advantage

The three plate mold is self de-gating mold which means that the product will be disconnected automatically from the runner after the injection process is finished. The product produce is also have better surface finish because of the gate size in three plate mold is pin pointed. The gate location in three plate mold is more flexible because it has two parting line and the gate can be located virtually anywhere along the surface.

2.4.1.2.2 Disadvantage

The three plate mold is more expensive in term of cost compare to the two plate mold because it has two parting line and the mechanism to self de-gating the product. The mold also has additional waste at the runner because the runner is the part that leads sprue into the mold and it is not needed together with the part.

2.4.2 Defects and Countermeasures in Injection Molding

There are many common defects in injection molding such as blister, burn marks , color streaks, delamination, flash, embedded contaminates, flow marks, jetting, silver streaks and sink marks[3]

Blister or blistering is a raised or layered zone on the surface of the parts. It is happened when tool or material is too hot, and often caused by a lack of cooling around the tool or a faulty heater [4]

Burn marks also called as air burn or gas burn is a black or brown burnt areas on the part located at furthest points from gate, caused by tool lacks of venting and the injection speed is too high [3]

Color streak is a localized change of color because the master batch isn't mixing properly, or the material has run out and it's starting to come through as natural only. This can be cured by switching to a color agent with good thermal stability because in terms of molding conditions, it is desirable for the plastic and die temperature to be high, and for the speed of flow to be low (i.e., hot-slow molding). [7]

Delamination is thin mica like layers formed in part wall that caused by contamination of the material e.g. PP mixed with ABS and it is very dangerous if the part is being used for a safety critical application as the material has very little strength when delaminated as the materials cannot bond. Carrying out the cleaning at the injection machine such as inside the cylinder, hopper and dryer or standardize the resin temperatures such as raised the temperature of resin and die can reduce the delamination effect. [3][7]

Flash or burrs is excess material in thin layer exceeding normal part geometry, caused by tool damage, too much injection speed/material injected or the clamping force is too low. This can be eliminate by using enough clamping force or use the appropriate injection speed and amount of material when injecting process. [3]

Embedded contaminates or embedded particulates is a foreign particle (burnt material or other) that embedded in the part and caused by the particles on the tool surface, contaminated material or foreign debris in the barrel, or too much shear heat burning the material prior to injection [3]

Flow marks is a directionally "off tone" wavy lines or patterns usually caused by too slow injection speeds (the plastic has cooled down too much during injection, injection speeds must be set as fast as you can get away with at all times). This can be counter by raising the mold temperature, the runner and gates, increasing the speed of the injection flow or review the shaped of the molded component and round off corners on steps between areas of different thickness so that smoother flow may be realized. [7]

Jetting is a deformed part by turbulent flow of material. This usually occurs because of the poor tool design, gate or runner position and if the injection speed set too high. This can be overcome by increasing the temperature of the plastic to lower its viscosity. In the case of amorphous plastics, the ideal temperature of the temperature of the mold is between 20 C and 30 C lower than the plastic's thermal deformation temperature. It is also beneficial to reduce the injection speed. [3][4]

Silver streaks are a circular pattern around gate caused by hot gas. The moisture in the material, usually when hygroscopic resins are dried improperly is the main cause of this defect. If bubbles are formed during blank purging, optimization of the nozzle temperature, cylinder temperature, injection speed, or screw speed and balancing between moldings machine capacity component capacity can reduce the silver streak. [7]

Sink Marks is the most common defects in injection molding. It is a localized depression (in thicker zones of the parts) and usually caused by part thickness that is not uniform, the holding time/pressure is too low or cooling time is too low. With the sprueless hot runners, this can also be caused by the gate temperature being set too high [3] [5]

Weld lines can be minimize by shaping conditions which is using plastic with high temperature, lowering the injection speed and using high holding pressure. Although weld lines will become less obvious when the speed is increased (i.e., the depth is minimized), there is a tendency for their length to increase and marks are made less noticeable using the compression effect of the weld’s V-notch [7].

2.5 CONCLUSION

This chapter generally discussed about the plastic floor tile, the Moldflow Plastic Adviser and the injection molding. A plastic floor tile will be design later on; will be analyzed using Moldflow software. The Moldflow software, which is the main software used for analysis in this project is explained briefly in this chapter. The two plate and the three plate mold, which is the most common type of mold in the cold runner type, is discussed here and these two are the type of molds that will be compared in this project

CHAPTER 3

METHODOLOGY

3.1 INTRODUCTION

This chapter will describe how the project had been done and the method used to achieve the project objectives. This chapter also will show the project progress from the beginning until the end. The project starts with collecting the information about the two plate and three plate mold. Then, the best type of mold will be selected.

3.2 OVERVIEW OF METHODOLOGY

Basically, there are three main parts in this project, which are;

- Product design
    - Floor tile
    - Shape - hexagon
    - Material - Thermoplastic
    - Functions - floor tile for indoor games such as. Futsal, volleyball, takraw and badminton
- FEA for plastic flow using Moldflow software

- Get the optimize parameter between the two plate and the three plate mold

- Mold selection

  - Determine the best type of mold and select the mold base on the result achieved from analysis

The first stage of this project is the product design. For this project, the plastic sport tile is chosen. This is plastic tile is for sport usage such as the futsal courts.

The second stage is analysis of flow using software like Moldflow, the project flows continue with analyzing the mold flow for the molds designed. Software such as Moldflow is used to analyze the design. All the data from the design is taken after the design is analyzed.

The next stage is the selection stage. All the data from the two plate and three plate mold will be compared using appropriate software. The two plate and three plate mold the two must common type in cold runner in the injection molding. Both of these mold have their own advantages and disadvantages. This is the stage whether to choose two plate or three plate mold.

All the information will be documented again. The presentation for the project also can be continue after finish the documentation

3.2.1 Product Design

Generally for this project, a product must be designed first before performing the analysis using Moldflow software and then based on the result obtained; the best type of mold will be selected. The product design is a plastic floor tile for sport usage. It will be hexagonal in shape and has the interlocking features for attaching purpose. The interlocking features are to make sure that the applying and removing the plastic floor tile is easier and does not damage the product.

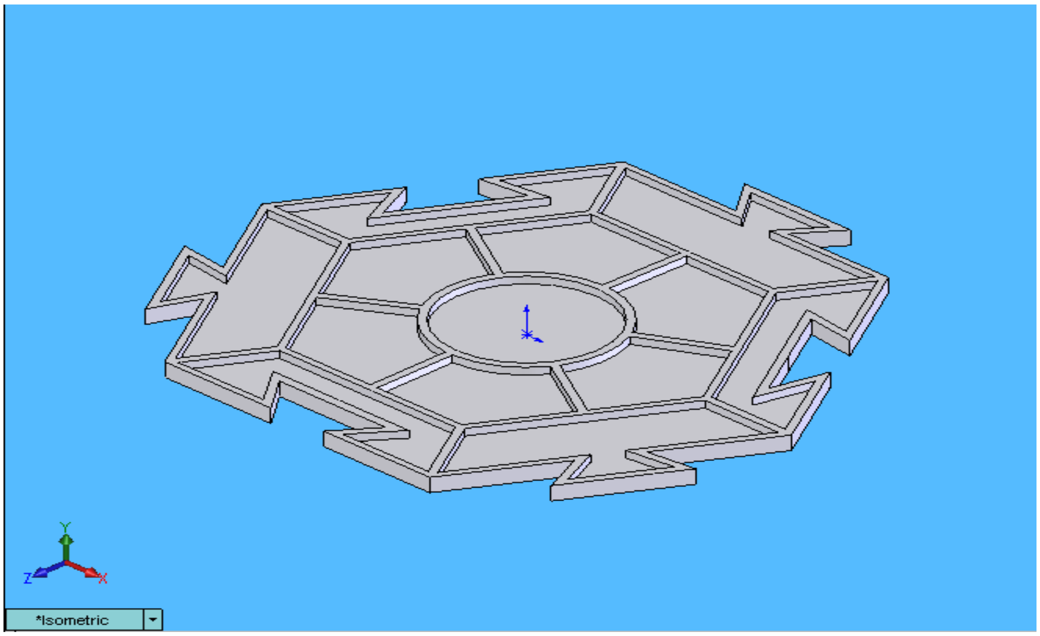


Figure 3.1: Isometric view of product design

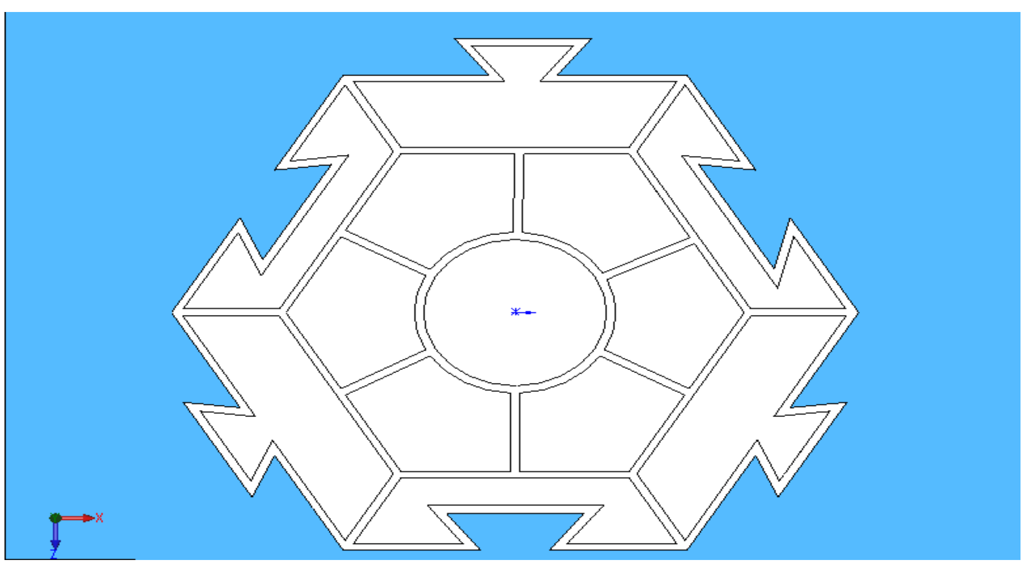


Figure 3.2: Top view of the product design

One of the most critical features of the plastic floor tile is the surface of the floor tile must be flat. In injection molding, when producing a product, there will be a tendency of the product ejected to have warpage, which is an injection defect that will make the product tend to bend. Thus the design of the product must have the features to at least reduce the warpage effect.

The product is designed using Solidwork software, V 2005. The length from end to end is 300 mm with 10 mm thickness. The design is a hexagonal plastic floor tile for sport usage. The floor tile usage is for indoor games such as futsall, volleyball, takraw and badminton. The rib, which is 2 mm in thickness, is design to strengthen the product and to reduce the warpage effect that might occurs after the injection process.

Figure 3.1 show the isometric view of the product design and figure 3.2 show the top view of the product. The interlocking features are designed for easier applying and removing the product for maintenance purpose without damaging it.

3.2.2 Analysis using Moldflow Plastic Adviser software

After the designing of the product is complete, the product is analyzed using Moldflow Plastic Adviser software to choose the best type of mold, between the two plate and the three plate mold in terms of the mold parameters, the type of materials and cooling system.

In doing so, the product which is design in Solidwork software is converted to IGS type of file before the model can be generated in the Moldflow Plastic Adviser software. After modeling, the next step is to set the gate location. Because the product is thin walled, the appropriate analysis is using Dual domain analysis. Using the best gate location analysis, the Moldflow software will show the area with the best gate location for the model generated. Several analyses are run to set the gate location.

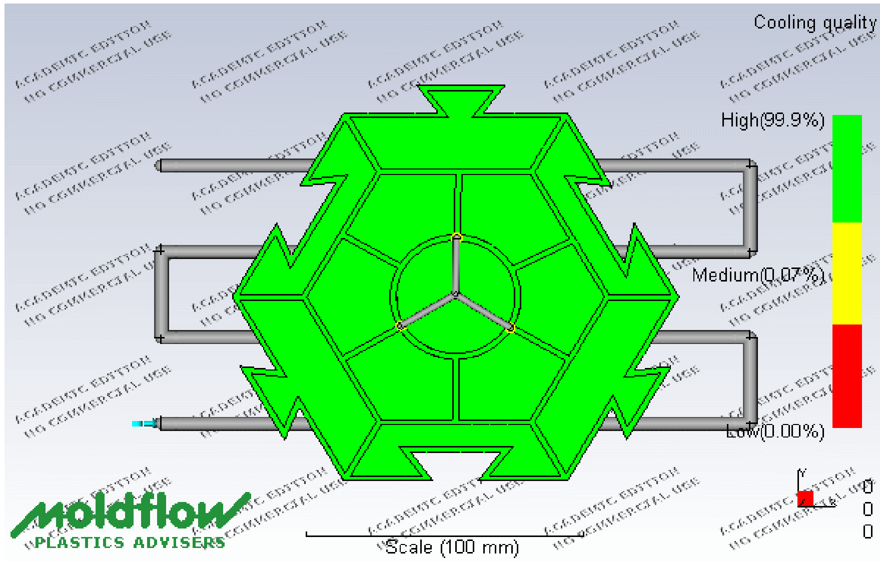


Figure 3.3: Three plate mold with three gate location

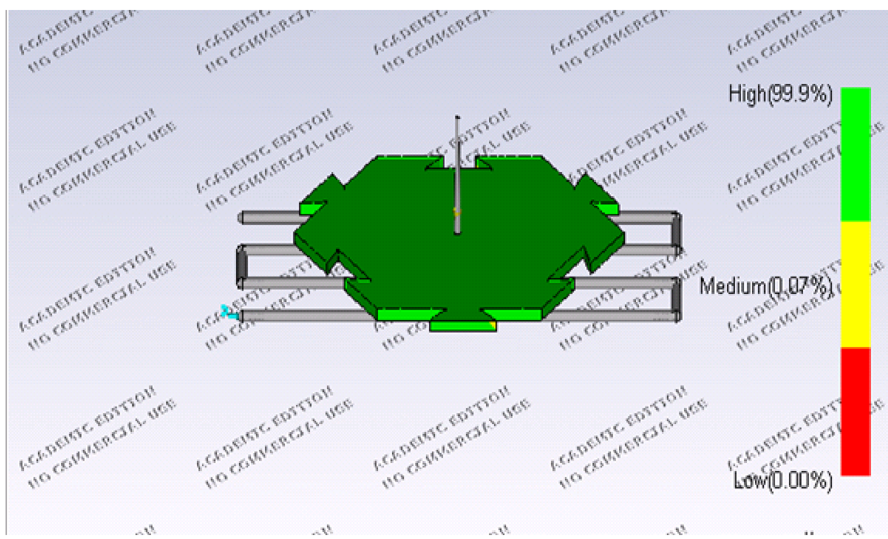


Figure 3.4: Two plate mold with one gate location

Figure 3.3 and figure 3.4 show the two plate and the three plate mold with one gate location and three gate location. The gate location for two plate mold must be located at the parting line of the mold, thus the mold's gate location is not as flexible as the three plate mold

The next step is material selection. For the first analysis with bottom cooling system for both type of molds, the ABS material with the trade name of Lustran ABS 1146. The melt temperature, mold temperature and maximum machine injection pressure is set to default setting as shown in Table 3.1. The Fill analysis, Fill + Pack analysis, cooling quality, sink mark and warp analysis is selected as the desired analysis to analyze the product.

Mold parameters such as mold width and offsets from the cavity is set after the gate, runner and sprue for both molds is design. The cooling system is design at both molds before running all the selected analysis.

The same steps is repeated when analyzing the product for the second analysis using TPE material with the trade name of Riteflex 672 but with top and bottom cooling system for both molds. The melt temperature, mold temperature and maximum machine injection pressure is set to default as shown in Table 3.2.

Table 3.1: General properties for both molds using **ABS** (Lustran ABS 1146) with bottom cooling system

GENERAL	**2PLATE**	**3 PLATE**
Material trade name	Lustran ABS 1146	Lustran ABS 1146
Melt temperature (C)	260.0	260.0
Mold temperature (C)	80.0	80.0
Injection locations	1	3
Max. machine injection pressure (MPa)	180.000	180.000

Table 3.2: General properties of both molds using **TPE** (Riteflex 672) with top and bottom cooling system

GENERAL	**2PLATE**	**3 PLATE**
Material trade name	Riteflex 672	Riteflex 672
Melt temperature (C)	250.0	250.0
Mold temperature (C)	50.0	50.0
Injection locations	1	3
Max. machine injection pressure (MPa)	180.000	180.000

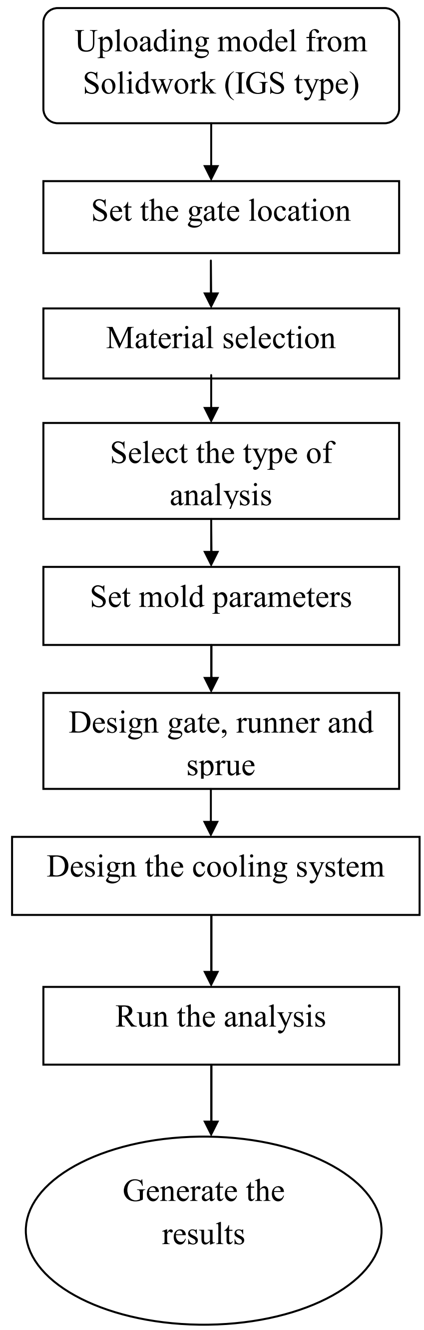


Figure 3.5: Moldflow Plastic Adviser software flow chart

3.2.3 Mold Selection

All the data from the Fill analysis, Fill+Pack analysis, cooling quality, sink mark warp analysis is gather and then compared between both two plate and three plate mold molds

The mold selection is based on the parameters effect for both molds such as actual filling time, actual injection pressure, injection cycle time, maximum clamp force during cycle, effect of material in term of cooling time and effect of cooling system with warpage for both molds

The best mold with faster operation time and save more cost will be selected as the type of mold to produce the hexagonal plastic floor tile for sport usage, which is the product design.

3.3 CONCLUSION

This chapter discussed about the methodology of this project. Starts with the product designed, which is using Solidwork software, then the FEA using Moldflow software and the mold selection stage. This will be used as a guideline to finish the whole project.

CHAPTER 4

RESULT AND DISCUSSION

4.1 RESULT

The materials used in these analyses are Riteflex 672 (TPE) and Lustran ABS 1146 (ABS) .The two plate mold is design with one gate location, which is at the parting line of the mold, while the three plate mold is design with three gate location. The type of gate use in these analyses is tip point type with 1.0 mm head. All the analysis is performed using Dual domain analysis because of the thin walled product from Moldflow Plastic Adviser software.

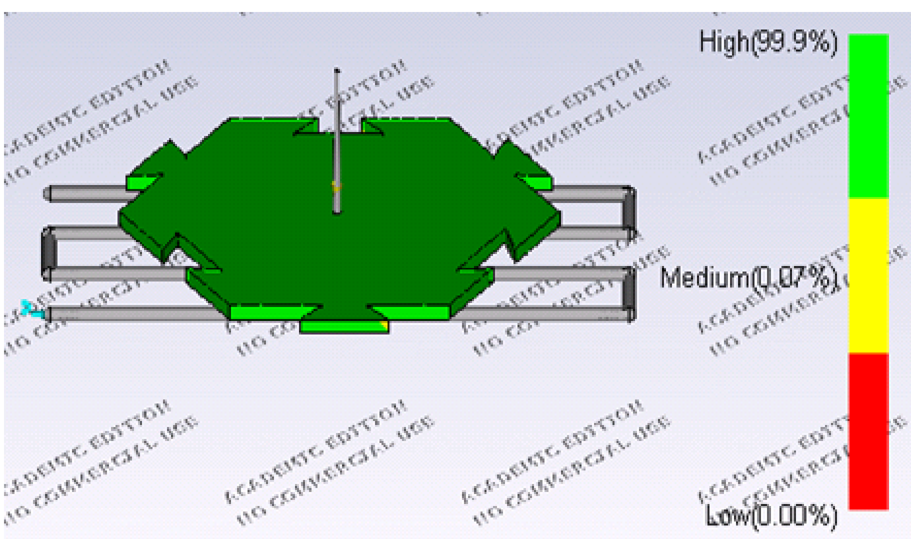


Figure 4.1: The two plate mold with bottom cooling system

Figure 4.1 show the two plate mold with bottom cooling system only. This analysis is performed while using the ABS material (Lustran ABS 1146). The figure shows that the gate location for the two plate mold is situated at the center of the mold to ensure that the molten plastic is evenly distributed to fill the cavity. The sprue used is circular tapered type with 3.0 mm start and 6.0 mm end diameter. The cooling system use in this analysis is circular type with 10.0 mm in diameter. Pure water act as the cooling liquid with 10.0 lit/min flow rate

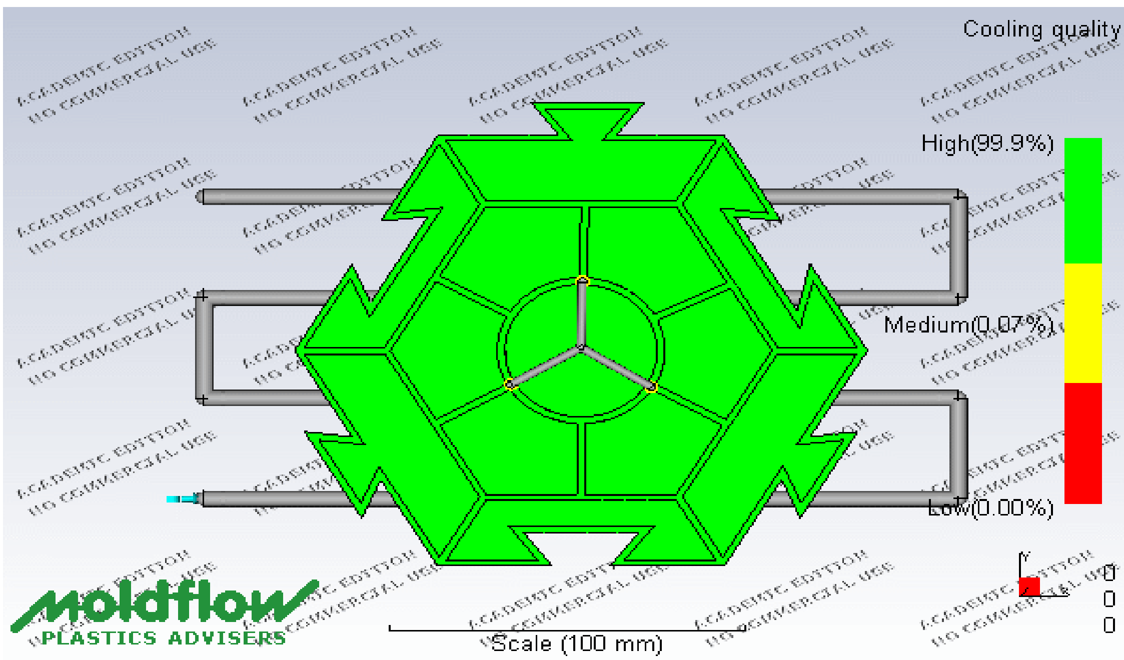


Figure 4.2: The three plate mold with bottom cooling system

Figure 4.2 show the three plate mold with bottom cooling system. Three gate locations are placed at the bottom part of the model. This model is analyzed while using the ABS material (Lustran ABS 1146). All the gates used in this analysis are pin point type with 1 mm end diameter. The branched runners are circular cold runner type with 5.0 mm diameter and the sprue used for this analysis is circular tapered with 3 mm start and 6.0 mm end in diameters. The cooling system use in this analysis is circular type with 10 mm in diameter. Pure water act as the cooling liquid with 10.0 lit/min flow rate

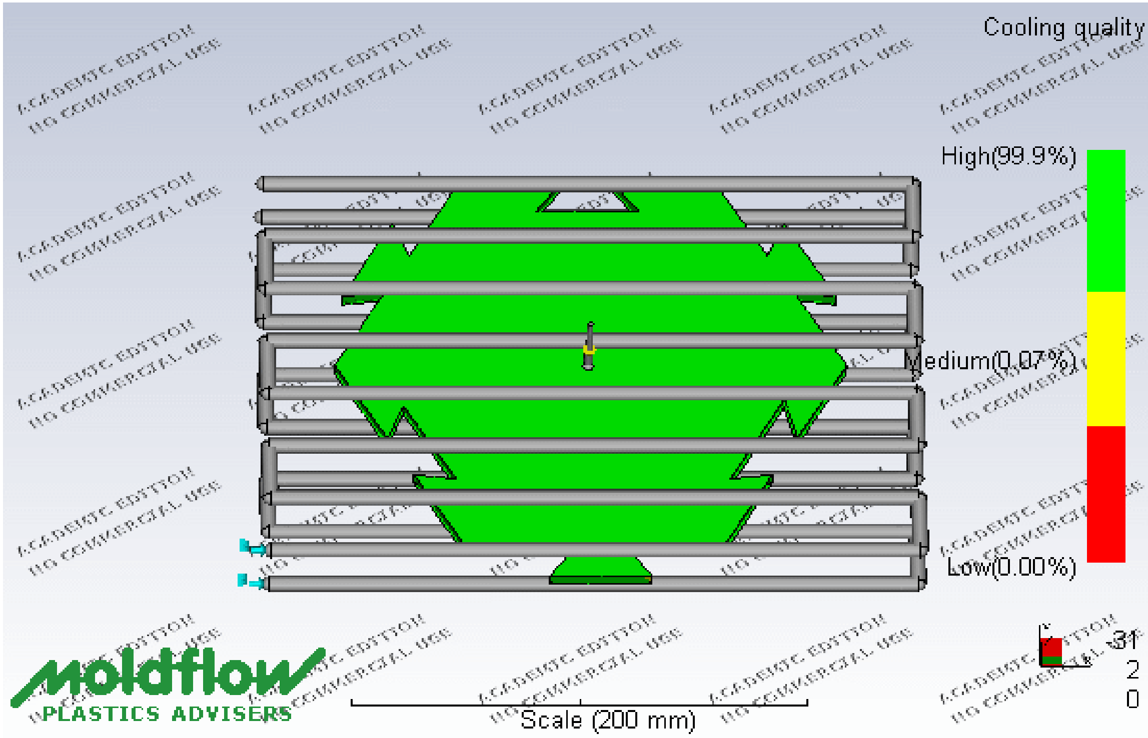


Figure 4.3: The two plate mold with top and bottom cooling system

Figure 4.3 shows the two plate mold with top and bottom cooling system. The gate location is situated at the center of the mold because for the two plate mold, the gate must be located at the center of the mold straight from the sprue. The TPE material (Riteflex 672) is used while analyzing this model. The gate used is pin point type with 1.0 mm head. The sprue used for this analysis is circular tapered type with 3.0 mm start and 6.0 mm end diameter. Same with the analysis using ABS material (Lustran ABS 1146), the cooling system is circular type with 10.0 mm in diameter. Pure water act as the cooling liquid with 10.0 lit/min flow rate

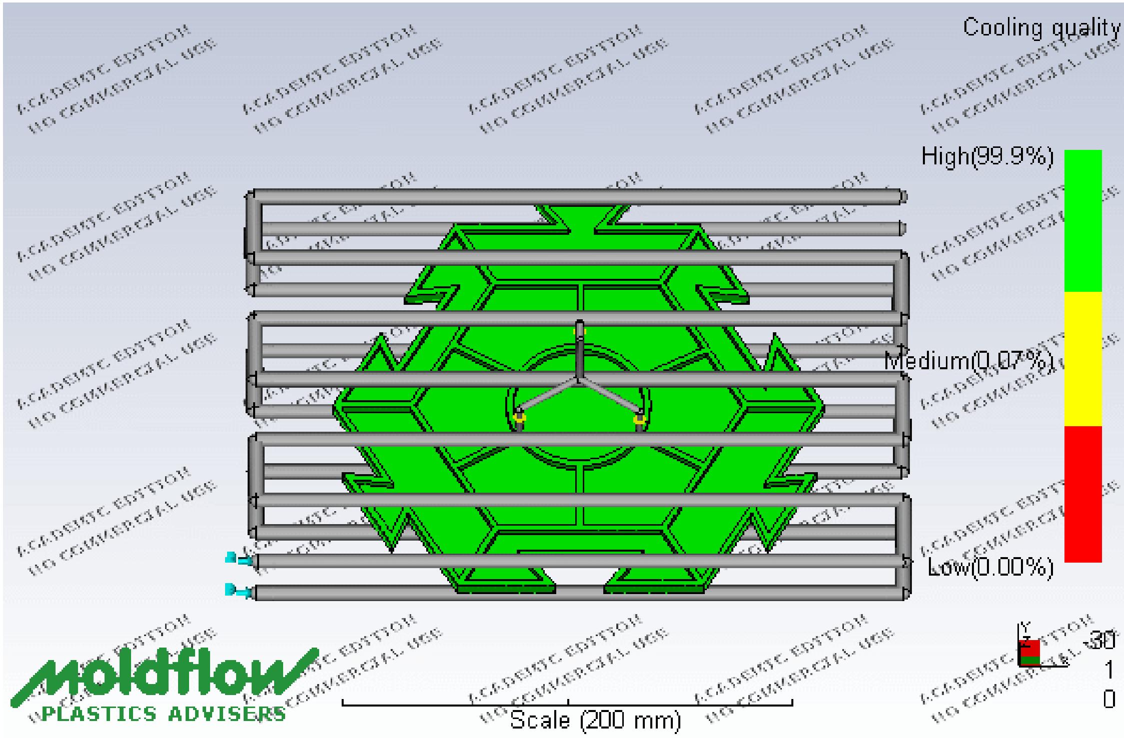


Figure 4.4: The three plate mold with top and bottom cooling system

Figure 4.4 shows the model of three plate mold with top and bottom cooling system. The three gates for this model is tip pointed type with 1.0 mm head. This model is analyzed while using the TPE material (Riteflex 672). The runners that connect the gates with sprue are circular type with 5.0 mm in diameter. The sprue for this model is circular tapered with 3.0 mm start and 6.0 mm end diameter. The cooling system used for this analysis is circular type with 10.0 mm in diameter and pure water act as the cooling liquid with 10.0 lit/min flow rate.

4.1.1 Analysis using TPE (Riteflex 672) material with top and bottom cooling system

The types of analysis used for the TPE material (Riteflex 672) are the Fill analysis, Fill+Pack analysis, cooling quality analysis, sink mark analysis and warp analysis from Moldflow Plastic Adviser software.

Table 4.1: General properties of both molds using **TPE** (Riteflex 672) with top and bottom cooling system

GENERAL	**2PLATE**	**3 PLATE**
Material trade name	Riteflex 672	Riteflex 672
Melt temperature (C)	250.0	250.0
Mold temperature (C)	50.0	50.0
Injection locations	1	3
Max. machine injection pressure (MPa)	180.000	180.000

Table 4.1 shows the general properties of the TPE (Riteflex 672) material used for both the two plate and the three plate mold. The melt temperature for both molds is 250 C and the mold temperature are 50 C. Maximum machine injection pressure is 180 MPa while the gate location for two plate mold is one and the three plate mold is three gate locations.

Table 4.2: Fill analysis for both molds **TPE** (Riteflex 672) with top and bottom cooling system

FILL ANALYSIS	**2 PLATE**	**3 PLATE**
Actual filling time (s)	2.44	2.32
Actual injection pressure (MPa)	35.914	25.920
Max. clamp force during filling (tonne)	19.906	17.238
Total part weight at the end of filling (g)	338.513	340.215
Shot volume (cm^3)	344.7180	348.9610
Cavity volume (cm^3)	343.8940	343.8930
Runner system volume (cm^3)	1.6491	5.0676

The actual filling time is the time needed for the molten plastic to finish filling the cavity. As shown in Table 4.2, the actual filling time for two plate mold is 2.44 sec while the three plate mold is 2.32 sec. The actual injection pressure for two plate mold is 35.914 Mpa and the three plate mold is 25.920 Mpa. This shows how much pressure is needed to push the molten plastic into the cavity. Maximum clamping force during filling for two plate mold is 19.906 tonne and the three plate mold is 17.238 tonne. Total part weight at the end of filling shows the part weight after the molten plastic finish filling the cavity. For two plate mold, the part weight is 338.513 grams while the three plate mold is 340.215 grams. Shot volume shows the amount of molten plastic used to fill the cavity. For the two plate mold, the amount is 344.7180 cm^3 while the three plate mold is 348.9610 cm^3. Runner system volume shows the volume of runners used. For two plate mold, the volume is 1.6491 cm^3 and three plate mold is 5.0676 cm^3.

Table 4.3: Fill+pack analysis for both molds **TPE** (Riteflex 672) with top and bottom cooling system

FILL+PACK ANALYSIS	2 PLATE	3 PLATE
Maximum clamp force during cycle (tonne)	120.54	98.831
Total part weight (g)	367.166	375.305
Cooling time (s)	21.50	21.50
Cycle time (s)	38.84	38.80

Maximum clamp force during cycle is the amount of clamping force needed for the machine to clamp the mold when injection process is running to inject the molten plastic into the mold. For two plate mold, the amount is 120.54 tonne and the three plate mold is 98.831 tonne. The total part weight is the final weight of the product after the injection process is finished. For the two plate mold, the total part weight is 367.166 grams and the three plate molds is 375.305 grams. The cooling time is the time needed for the molten plastic to cool down in the cavity. For both molds, the cooling time is 21.50 sec. Cycle time is the total amount of time to produce the product from mold opening, injection process, material cooling down and the part is ejected from the mold. For two plate mold, the cycle time is 38.84 sec and 38.80 sec.

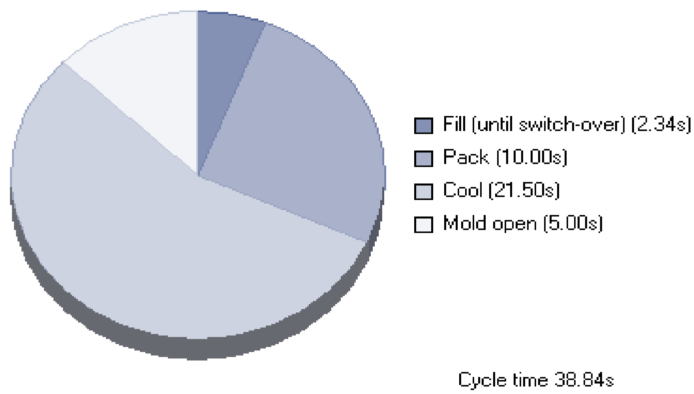


Figure 4.5: Cycle time for two plate Mold using TPE material

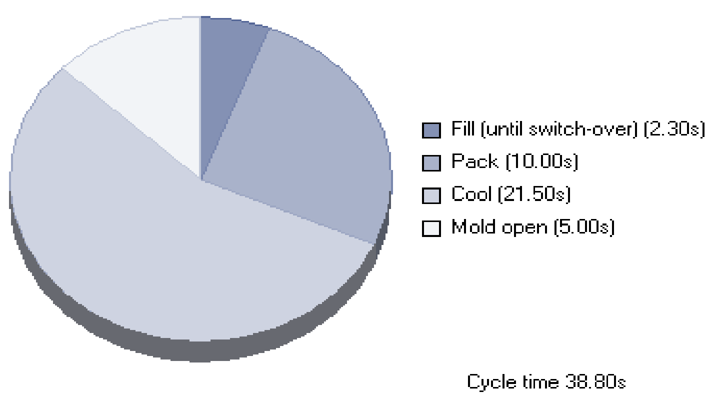


Figure 4.6: Cycle time for three plate Mold using TPE material

Figure 4.5 and 4.6 shows the cycle time for both two plate and three plate mold when using TPE material (Riteflex 672) as the analyzed material. For the two plate mold, the cycle time is 38.84 second and the three plate mold is 38.80 second.

Table 4.4: Sink mark analysis for both molds **TPE** (Riteflex 672) with top and bottom cooling system

SINK MARK	2 PLATE	3 PLATE
Max sink depth (mm)	0.14	0.13
Average sink mark depth (mm)	0.10	0.10
Percentage of model prone to sink marks (%)	5.18	5.18

Table 4.4 shows the sink mark depth for both two plate and three plate mold. Sink mark is an injection molding defect. It is a localized depression in thicker zones of the parts**.** Maximum sink mark depth for the two plate mold is 0.14 mm and the three plate mold is 0.13 mm. The average sink mark depth for the product in two plate mold is 0.10 mm, which the same for the three plate molds. The percentage of the model to have sink mark is the same for both two plate and three plate molds, which is 5.18 %**.**

Sink mark usually occurs at the part which is thicker compare to other parts such as at the ribs of the product. For the two plate mold, the sink mark is 0.14 mm, while the three plate mold is 0.13 mm. The sink mark can be eliminated by redesign the product with uniform wall thickness.

Tables 4.5: Warp analysis for both molds **TPE** (Riteflex 672) with top and bottom cooling system

WARP	2 PLATE	3 PLATE
Nominal max. deflection (mm)	1.05	1.02
Percentage exceeding Nominal max. deflection (%)	1.38	0.59
Percent within Nominal max. deflection (%)	98.62	99.41

Warpage is also one of injection moldings defects, which usually occurs when the part injected in the mold is not cooled uniformly, for example the top part cools faster than the lower part, thus will make the product produce tends to bend upward or down ward. When using the top and bottom cooling system, the maximum nominal deflection for two plate mold is 1.05 mm, 1.38 % of the product exceeding the maximum nominal deflection and 98.62 % of the product is within the maximum nominal deflection. For the three plate mold, the maximum nominal deflection is 1.02 mm. 0.59% of the product is exceeding the nominal deflection and 99.41 % is within the maximum nominal deflection

4.1.2 Analysis using ABS (Lustran ABS 1146) material with bottom cooling system

The types of analysis used for the ABS material (Lustran ABS 1146) are the Fill analysis, Fill+Pack analysis, cooling quality analysis, sink mark analysis and warp analysis from Moldflow Plastic Adviser software.

Table 4.6: General properties for both molds using **ABS** (Lustran ABS 1146) with bottom cooling system

GENERAL	2PLATE	3 PLATE
Material trade name	Lustran ABS 1146	Lustran ABS 1146
Melt temperature (C)	260.0	260.0
Mold temperature (C)	80.0	80.0
Injection locations	1	3
Max. machine injection pressure (MPa)	180.000	180.000

Table 4.6 shows the general properties for ABS (Lustran ABS 1146) material used for the two plate and three plate mold. The melt temperature for both molds is 260 C and the mold temperature is 80 C. The maximum machine injection pressure is 180 MPa. The only difference is the two plate mold has one gate location while the three plate mold has three gate locations

Table 4.7: Fill analysis for both molds using **ABS** (Lustran ABS 1146) with bottom cooling system

FILL ANALYSIS	2 PLATE	3 PLATE
Actual filling time (s)	3.92	3.73
Actual injection pressure (Mpa)	53.306	41.106
Max. clamp force during filling (tonne)	61.755	42.192
Total part weight at the end of filling (g)	316.613	317.184
Shot volume (cm^3)	345.5430	349.6430
Cavity volume (cm^3)	343.8940	343.8930
Runner system volume (cm^3)	1.6491	5.7500

Table 4.7 shows the Fill analysis for both two plate and three plate mold. Actual filling time which is the time needed for the molten plastic to finish filling the cavity, for the two plate mold is 3.92 second and the three plate mold is 3.73 second. The actual injection pressure for two plate mold is 53.306 Mpa and the three plate mold is 41.106 Mpa. The actual injection pressure shows the amount of pressure needed to push the molten plastic into the cavity. Maximum clamping force during filling for two plate mold is 61.755 tonne and three plate mold is 42.192 tonne. The total past weight at the end of filling shows the total part weight after the molten plastic finish filling the cavity. For two plate mold, the part weight is 316.613 gram and the three plate mold is 317.184 gram. The shot volume is the amount of material used for filling the cavity. For two plate mold, the shot volume is 345.5430 $cm^{3.}$ For the three plate mold, the amount of material used is 349.6430 cm^3. The runner system volume for two plate mold is 1.6491 cm^3 and the three plate mold is 5.7500 $cm^{3.}$ This shows the amount of runner used for both type of molds.

Table 4.8: Fill + Pack analysis for both molds using **ABS** (Lustran ABS 1146) with bottom cooling system

FILL+PACK ANALYSIS	2 PLATE	3 PLATE
Maximum clamp force during cycle (tonne)	150.140	123.888
Total part weight (g)	325.462	330.414
Cooling time (s)	139.25	138.75
Cycle time (s)	158.11	157.41

For the two plate mold, the maximum clamp force during cycle, which is the amount of clamping force needed for the machine to clamp the mold when injection process is running to inject the molten plastic into the mold, is 150.140 tonne and the three plate mold is 123.888 tonne. The total part weight for the two plate mold is 325.462 gram while the three plate mold's total part weight is 330.414 gram. This is the total part weight after the injection process is finished. The cooling time is the amount of time needed for the molten plastic to cool and solidified in the cavity of the mold. For the two plate mold, the cooling time is 139.25 second and the three plate mold is 138.75 second. Cycle time is the amount of time needed to produce a product, right from the mold opening, injection process, material cooling down and the part is ejected from the mold. For the two plate mold, the cycle time is 158.11 second and the three plate mold is 157.41 second.

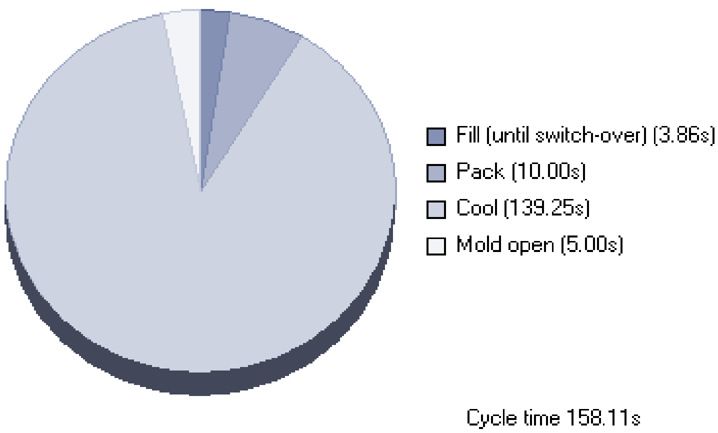


Figure 4.7: Cycle time for two plate mold using ABS material

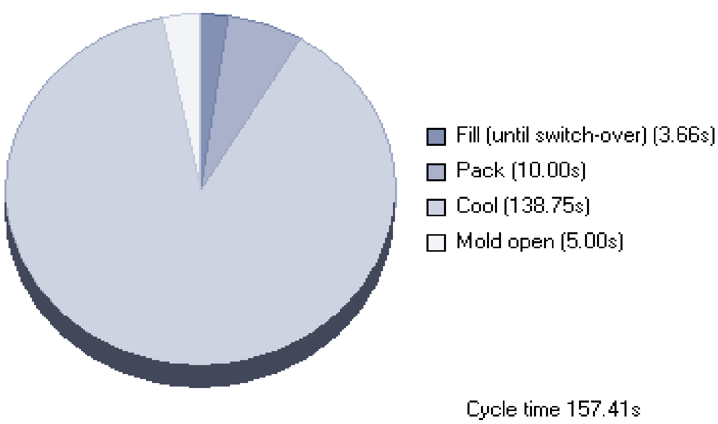


Figure 4.8: Cycle time for three plate mold using ABS material

Figure 4.7 and 4.8 shows the cycle time for both two plate and three plate mold when using the ABS material (Lustran ABS 1146) as the analyzed material. For the two plate mold, the cycle time is 158.11 second and the three plate mold is 157.41 second.

Table 4.9: Sink Mark analysis for both molds using **ABS** (Lustran ABS 1146) with bottom cooling system

SINK MARK	**2 PLATE**	**3 PLATE**
Max sink depth (mm)	0.11	0.12
Average sink mark depth (mm)	0.09	0.09
Percentage of model prone to sink marks (%)	5.18	5.19

The sink mark depth for two plate mold is 0.11 mm while the three plate mold is 0.12 mm. Sink mark is a molding defect that usually occurs at the part which is thicker than the others. The average sink mark depths for both molds are 0.09 mm. For the two plate mold, 5.18 % of the model prone to have sink mark and for the three plate mold, 5.19 % of the model likely to suffer from sink mark. To reduce or eliminate the sink marks; the design of the product must have uniform wall thickness.

Table 4.10: Warp analysis for both molds using **ABS** (Lustran ABS 1146) with bottom cooling system

WARP	**2 PLATE**	**3 PLATE**
Nominal max. deflection (mm)	2.29	2.29
Percentage exceeding Nominal max. deflection (%)	0.00	0.00
Percent within Nominal max. deflection (%)	100.00	100.00

The maximum nominal deflection for both molds is 2.29 mm when using bottom cooling system. Both products of the molds have 100.00 % within nominal deflection.

4.2 DISCUSSION

Discussion will be made base on the finding result for the Fill analysis, Fill+Pack analysis and Warp analysis for both TPE (Riteflex 672) and ABS (Lustran ABS 1146) material.

For TPE material (Riteflex 672), as shown is Table 4.2, the actual filling time for three plate mold which is 2.32 second, is faster compare to two plate mold with 2.44 second because the three plate mold has three gate locations which allows the molten plastic to fill the cavity faster, compare to the two plate mold with one gate location only. The actual injection pressure for the two plate mold is 35.914 tonne and the three plate mold is 25.920 tonne. The two plate mold's actual injection pressure is higher because more pressure is needed to push the molten plastic into the cavity because it has only one injection location. The shot volume is the amount of material injected in the injection molding to fill the cavity. With 340.215 gram of total part weight and 348.9610 cm^3 of shot volume, the three plate mold is higher total in total part weight and shot volume compare to the two plate mold with 338.513 gram of total part weight and 344.7180 cm^3. This is because it has additional runner due to the number of gate location the mold have, which is three gates.

From Table 4.3, when using TPE material, the maximum clamping force for two plate mold is 120.54 tonne while three plate mold is 98.831 tonne. The maximum clamping force for two plate mold is higher because the plate's actual injection pressure is higher compare to the three plate mold. The clamping force must be higher than the injection pressure so that the mold will not opened during the injection process is running. When using the TPE (Riteflex 672) as the analysis material, the cooling time for both molds is 21.50 second, and the cycle time for two plate mold is 38.84 second, while three plate mold is 38.80 second.

From Table 4.5, when using the top and bottom cooling system, the maximum nominal deflection for two plate mold is 1.05 mm, 1.38 % of the product exceeding the maximum nominal deflection and 98.62 % of the product is within the maximum nominal deflection. For the three plate mold, the maximum nominal deflection is 1.02 mm. 0.59% of the product is exceeding the nominal deflection and 99.41 % is within the maximum nominal deflection. The top and bottom cooling system gives more uniform cooling to the product, during the cooling process for both top and bottom parts of the product so that the part will not warp or bend after the cooling process.

When performing the analysis using ABS material (Lustran ABS 1146), as shown in Table 4.7, the actual filling time for two plate mold is 3.92 sec., which is higher than three plate mold with 3.73 sec. The result is the same while using TPE material (Riteflex 672) in term of filling time. This is because the molten plastic takes more time to fill the cavity in the mold because it has only one gate location. Comparing with three plate mold with three gate location, the filling of the cavity process is faster. The actual injection pressure for two plate mold which is 53.306 tonne, is higher compare to three plate mold with 41.106 tonne. This is because more force is needed to push the molten plastic into the two plate mold with one gate location only, compare to three plate mold with three gate location, which allow the molten plastic fill enter the cavity easier and faster. Because of the additional gates, the three plate mold has the higher total part weight and shot volume comparing to the two plate mold. The three plate mold has higher total part weight, shot volume and runner system volume compare to the two plate mold because of the additional gate location that connect with the runner and then the sprue. This is why the three plate mold has more waste at the runner part for every product produce.

From Table 4.8, using the (Lustran ABS 1146) material, the cooling time for both molds is exceeding 2 minutes. The two plate mold with 139.25 sec, while the three plate mold is 138.75 sec. It is very long compare than usual because the average injection molding cycle time is less than 1 minute. The clamping force for two plate is

150.140 tonne, which is higher compare to three plate with 123.888 tonne, same results when using the TPE (Riteflex 672) material. This is because the actual injection pressure for two plate is higher compare to three plate.

Based n Table 4.10, the maximum nominal deflection for both molds is 2.29 mm when using bottom cooling system. Both products of the molds have 100.00 % within nominal deflection. When using the bottom cooling system only, the lower part of product will tends to cool much faster compare to the top section of the part, thus the deflection occurs is higher compare to deflection when using both top and bottom cooling system.

4.2.1 Parameters effect

1. Actual filling time

Molten plastic flow in the mold with many gates is faster compare to the mold with fewer gates for both materials. The filling time in three plate mold is faster compare to two plate mold because it has three gates that will allow the molten plastic to fill the cavity faster. Based on Table 4.2 and Table 4.7, both the three plate mold analyzed with TPE and ABS material have lower filling time compare to two plate mold, for both different material used.

2. Actual injection pressure

Injection pressure in molds with many gates is lower compare to the mold with fewer gates. From Table 4.7, the two plate with 1 gate use 53.306 MPa of force during injection process while the three plate only use 41.106 MPa of force for the Lustran ABS 1146 material. Based on Table 4.2, for Riteflex 672 material, the two plate molds use 35.914 (MPa), while three plate mold use 25.920 (MPa). This is because more force is needed to push the molten plastic into the two plate mold with one injection location, compare to the force needed for the three plate mold which is three gate locations. This will minimize the pressure needed to produce a product, using three plate mold.

3. Cycle time

Based on Table 4.3 and table 4.8, the cycle time for three plate mold using ABS material is 157.41 second, 38.80 second when using TPE material. Comparing with two plate mold when using ABS material, the cycle time is 158.11 second and 38.84 second when using TPE material. The cycle time of three plate mold is faster compare to the two plate mold because the fill time is faster, for both materials. This is because the fill time in three plate mold is faster, with three gate locations, the molten plastic can flow into and fill the cavity faster compare to the two plate mold because of the additional gate, thus shorten the cycle time.

4. Runner system volume

From Table 4.2 and Table 4.7, result shows that the runner volume is higher in three plate mold. Three plate mold has higher amount of runner volume because it has three gates that connected with the runner to the sprue, compare to one gate only in two plate mold that connect directly with the sprue. The three plate mold have additional runner compare to the two plate mold, thus, more waste is produce at the runner part of the three plate mold's products.

5. Shot volume

The shot volume in the three plate mold is higher compare to two plate mold because there are three gates location at the mold, and the weight of the product is calculated with runner attached, thus the three plate mold will produce more waste at the runner part. From the results obtained at Table 4.2 and Table 4.7, the three plate mold has higher total part weight and shot volume because it has additional runner due to the number of gate location the mold have, which is three .

6. Maximum clamp force during cycle

Maximum clamp force during cycle is the amount of clamping force needed for the machine to clamp the mold when injection process is running to inject the molten plastic into the mold. From Table 4.4, the maximum clamp force during cycle for the two plate mold is 120.54 tonne and the three plate mold's maximum clamp force during cycle is 98.831 tonne. From Table 4.8, the maximum clamp force for the two plate mold is 150.140 tonne and the three plate mold is 123.88 tonne. The two plate mold use higher clamping force compare to three plate mold. This is because the actual injection pressure in two plate mold is higher between these two molds, for both material analyzed. The clamping force usually higher because to overcome the injection pressure from opening the mold during injection process.

4.2.2 Material effect

The materials used in both analyses are ABS (Lustran ABS 1146) and TPE material (Riteflex 672). When using the ABS material (Lustran ABS 1146), from Table 4.8, the cooling time for both molds are exceeding 2 minutes. The two plate mold with139.25 second, and the three plate mold with cooling time of 138.75 second ,which greatly effect the cycle time of the injection process to a long 158.11 second for two plate mold and 157.41 second for three plate mold. It is very long compare than usual because the average injection molding cycle time is less than 1 minute. Based on table 4.3, when using the TPE (Riteflex 672), the cooling times for both molds are greatly decreased to 21.50 second. These greatly improve the cycle time to 38.84 second for two plate mold and 38.80 second for three plate mold.

4.2.3 Cooling system effect

Based on Table 4.5 and Table 4.10, when using the bottom cooling system for both two plate mold and three plate mold the warpage occurs is 2.29 mm but when using top and bottom cooling system, the warpage value drop to 1.05 mm for two plate mold and 1.02 for three plate mold. This is because ,with bottom cooling system only, when cooling process is running, the cooling rate for the top side of the product is not the same as the bottom part of the product , so the molten plastic in the cavity tends to cool not uniformly, resulting the deflection occurs.

4.3 CONCLUSION

After all factor considered, which are the parameter effect, the material effect and the cooling system effect, the results shows that the actual filling time for three plate mold is faster compare to two plate mold. The actual injection pressure for the three plate mold also is lower than the two plate mold. The three plate mold‘s cycle time is also lower compare to the two plate mold. The two plate mold has lower runner system and shot volume compare to three plate mold.

From the results obtained, the material cooling time for TPE material (Riteflex 672) is faster compare to the material cooling time for the ABS material (Lustran ABS 1146)

The results shows that the usage of top and bottom cooling system can reduce the nominal maximum displacement caused by warpage effect compare than using only the top bottom cooling system.

CHAPTER 5

CONCLUSION AND RECOMMENDATION

5.1 INTRODUCTION

This study is focused on getting the best result to produce a plastic floor tile between the two plate or the three plate mold. Using the Moldflow Plastic Adviser as the main software for analysis, the results analyzed from both molds using TPE material (Riteflex 672) and ABS material (Lustran ABS 1146) are then compared and analyzed to get the best type of mold with the best injection parameters.

5.2 CONCLUSION

The two plate mold is the most common used mold in injection molding and the mold splits into two halves. The mold is simple and easy in design and less expansive compare to three plate mold, but runner still attached to the product after the injection process. Thus, additional labor cost is needed to remove the runner from the product. From the result obtained using Moldflow Plastic Adviser, the two plate mold is better in term of lower runner volume and lower shot volume compare to the three plate mold. Although the two plate mold has lower material usage than three plate mold, the additional labor cost to remove the runners from the product will make the two plate mold cost higher.

The three plate differs from the two plate because it has two parting planes, and the mold splits into three sections every time the part is ejected. Three plate molds has more flexibility in gating location. A part can be gated virtually anywhere along its surface. The three plate mold is also better in term of mass production because it can save more time using the self-degating mechanism in the striper plate of the mold. Based on the result obtained from Moldflow software, the three plate mold is better than two plate mold in term of faster actual filling time, faster cycle time, lower actual injection pressure and lower maximum clamp force during cycle compare to the two plate mold. Although three plate mold has more waste at the runner parts, it is more practical in mass production because it used less power of the injection machine and the cycle time is faster compare to the two plate mold.

From the analysis of material comparison and the results in table 4.3 and 4.8, it can be concluded that by using TPE (Riteflex 672) is better than ABS (Lustran ABS 1146) for faster material cooling time, thus quicken the cycle time of injecting a product. It it crucial issue in the relevant industry, time is a very important aspect that directly connected with the profits.

In order to get less warpage, the cooling system is very important. Based on the results obtained from Warp analysis in Table 4.5 and 4.10 in chapter 4, it showed that by using top and bottom design of cooling system give better result to get products with less warpage.

In general it can be conclude that base on the result obtained from the Fill, Fill+Pack ,cooling quality, Sink mark and Warp analysis from Moldflow Plastic Adviser software , the best result for producing a plastic floor tile is using the three plate mold with top and bottom cooling system and using TPE (Riteflex 672) material.

5.3 RECOMMENDATION

In order to get a better result for the best type of mold to produce the plastic floor tile, some recommendation could be implemented in the future as below,

1) To get the actual results, the actual fabrication of the product need to be done to know which type of molds is better in the actual situation. This is because the Moldflow software is as guidance or prediction software for the users to know the overcome for the product using a variety of analysis.
2) The Moldflow Plastic Insight (MPI) is another option for analyzing the product. With the meshing function, the software can give more accurate result.
3) All the mold parameters, such as mold thickness, mold size, type of gate, size of runners and the cooling system must be taken from the actual mold to get more valid data when running the analysis.

REFERENCESS

[1.] Moldflow Certified Consultant & Reseller, software package modul, (http://www.moldflowanalysis.com/) on February 2008

[2.] Moldflow Plastic Advisers® (http://www.moldflow.com/st)

[3.] Plastic Wiki, free encyclopedia, (http://plastics.inwiki.org/Cold_runner_mold) on February 2008

[4.] Dominic V Rosato, Injection Molding Handbook, 2008, published by Springer, p 236

[5.] Hugh Jack, (1993-2001), Manufacturing engineering on disk, version 1.0

[6.] C.A. (ed.) Harper, Modern Plastic Handbook, published by McGraw-Hill Professional, 2000

[7.] UMG ABS, Ltd, molding trouble shoot (http://www.umgabs.co.jp/en/solution/trouble) on February 2008

[8.] S.H. Tang , Y.M. Kong, S.M. Sapuan, R. Samin and S. Sulaiman, *Design and thermal analysis of plastic injection mould*
Department of Mechanical and Manufacturing Engineering, Universiti Putra Malaysia, 43400 Serdang, Selangor, Malaysia
Received 3 September 2004; accepted 21 June 2005.

[9.] DOUGLAS M. BRYCE
Plastic Injection Molding…manufacturing process fundamentals, published by Society of Manufacturing Engineers