OPTIMIZATION OF MOLDING PARAMETER EFFECT TO WARPAGE OF CAR DASHBOARD BASED ON PLASTIC FLOW SIMULATION SOFTWARE

MOHD ZAHIRUDDIN BIN AZRI

BACHELOR OF ENGINEERING

UNIVERSITI MALAYSIA PAHANG

OPTIMIZATION OF MOLDING PARAMETER EFFECT TO WARPAGE OF CAR DASHBOARD BASED ON PLASTIC FLOW SIMULATION SOFTWARE

MOHD ZAHIRUDDIN BIN AZRI

Thesis submitted in fulfilment of the requirements
for the award of the degree of
Bachelor of Mechanical Engineering with Automotive Engineering

Faculty of Mechanical Engineering
UNIVERSITI MALAYSIA PAHANG

NOVEMBER 2009

UNIVERSITI MALAYSIA PAHANG

BORANG PENGESAHAN STATUS TESIS♦

JUDUL: **OPTIMIZATION OF MOLDING PARAMETER EFFECT TO WARPAGE OF CAR DASHBOARD BASED ON PLASTIC FLOW SIMULATION** SOFTWARE

SESI PENGAJIAN: 2009/2010

Saya **MOHD ZAHIRUDDINBIN AZRI (870324-11-5103)**
(HURUF BESAR)

mengaku membenarkan tesis (Sarjana Muda/~~Sarjana~~ /~~Doktor Falsafah~~)* ini disimpan di Perpustakaan dengan syarat-syarat kegunaan seperti berikut:

1. Tesis adalah hakmilik Universiti Malaysia Pahang (UMP).
2. Perpustakaan dibenarkan membuat salinan untuk tujuan pengajian sahaja.
3. Perpustakaan dibenarkan membuat salinan tesis ini sebagai bahan pertukaran antara institusi pengajian tinggi.
4. **Sila tandakan (√)

[ ]	**SULIT**	(Mengandungi maklumat yang berdarjah keselamatan atau kepentingan Malaysia seperti yang termaktub di dalam AKTA RAHSIA RASMI 1972)
[ ]	**TERHAD**	(Mengandungi maklumat TERHAD yang telah ditentukan oleh organisasi/badan di mana penyelidikan dijalankan)
[√]	**TIDAK TERHAD**	

Disahkan oleh:

______________________	______________________
(TANDATANGAN PENULIS)	(TANDATANGAN PENYELIA)

Alamat Tetap:

BLOK 41-22,	**SITI HARYANI BT TOMADI**
PERUMAHAN DESA TASIK,	(Nama Penyelia)
57000 SUNGAI BESI,	
KUALA LUMPUR.	

Tarikh: **20 NOVEMBER 2009**   Tarikh: : **20 NOVEMBER 2009**

CATATAN:
- * Potong yang tidak berkenaan.
- ** Jika tesis ini SULIT atau TERHAD, sila lampirkan surat daripada pihak berkuasa/organisasi berkenaan dengan menyatakan sekali tempoh tesis ini perlu dikelaskan sebagai atau TERHAD.
- ♦ Tesis dimaksudkan sebagai tesis bagi Ijazah doktor Falsafah dan Sarjana secara Penyelidikan, atau disertasi bagi pengajian secara kerja kursus dan penyelidikan, atau Laporan Projek Sarjana Muda (PSM).

SUPERVISOR DECLARATION

SUPERVISOR DECLARATION

I hereby declare that I have read this report and in my opinion this report is sufficient in term of scope and quality for the award of the degree of Bachelor of Mechanical Engineering with Manufacturing Engineering.

Signature	: …………………………
Name of Supervisor	: **MADAM SITI HARYANI BT TMADI**
Position	: Lecturer of Faculty Mechanical Engineering
Date	: ………………………....

STUDENT’S DECLARATION

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

Signature: ....
Name: MOHD ZAHIRUDDIN BIN AZRI
ID Number: MH06069
Date: 20 NOVEMBER 2009

DEDICATION

Another turning point,
A fork stuck in the road.
Time grabs you by the wrist,
Directs you where to go.
So make the best of this test,
And don't ask why.
It's not a question
But a lesson learned in time.
It's something unpredictable,
But in the end is right.
I hope you had the time of your life

Especially dedicated to
My beloved mother and father

ACKNOWLEDGEMENTS

First and foremost, I want to thank Allah S.W.T, for giving me the source of power, knowledge and strength to finish the project and dissertation for completing my Bachelor of Mechanical Engineering final year project. By gradually following the plan of optimization of molding parameter effect to warpage of car dashboard based on plastic flow simulation software, I've manage to fulfill my responsibility to this project.

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

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

Many thanks go to all the supervisors who have given their full effort in guiding the through on the project progress. I acknowledge my sincere indebtedness and gratitude to my parents for their love, dream and sacrifice throughout my life. Special thanks should be given to my committee members. I would like to acknowledge their comments and suggestions, which was crucial for the successful completion of this project.

ABSTRACT

This thesis present about the optimization of molding parameter effect to warpage of car dashboard based on plastic flow simulation software. The objective of this project is to study, analyze, and optimize the parameters that effect to warpage with using plastic flow simulation software. This project using the manual measurement after the original method using 3D scanner cannot be done because of the technical report. The measurement uses the tools from the Faculty of Mechanical Engineering Laboratory such as ruler, vernier calliper, threads, and protractor. Then, draw the dashboard in SolidWork 2008. Using MoldFlow Plastics Insight (MPI) 5.0 software, the injection molding parameters are optimized to get the best parameters to minimize warpage. The small changes of molding parameters will give higher impact to the plastic characteristics. The comparison between the types of analysis will be made. The comparison is between the deflections, fill times, and volumetric shrinkage including air traps and weld lines. As the conclusion, the changes of molding parameters will affect the warpage of the car dashboard.

ABSTRAK

Tesis ini membentangkan tentang mengoptimumkan pembolehubah yang memberi kesan kepada kelengkungan dasbor kereta dengan menggunakan perisian simulasi pengaliran plastik. Objektif projek ini adalah mengkaji, analisis, dan mengoptimumkan pembolehubah-pembolehubah yang memberi kesan kelengkungan plastik dengan menggunakan perisian simulasi pengaliran plastik (MoldFlow Plastics Insight (MPI) 5.0. Projek ini menggunakan pengukuran manual selepas perancangan asal menggunakan alat pengimbas 3-Dimensi tidak dapat dijalankan kerana terdapat masalah teknikal. Pengukuran ini menggunakan alat-alat yang terdapat di Makmal Fakulti Mekanikal seperti pembaris, angkup vernier, benang, dan jangka sudut. Dengan menggunakan perisian simulasi plastik, pembolehubah-pembolehubah suntikan acuan (*injection molding*) adalah dioptimumkan untuk mendapatkan pembolehubah-pembolehubah yang terbaik untuk mengurangkan kelengkungan. Perubahan yang sedikit terhadap pembolehubah-pembolehubah ini akan memberi kesan terdapat sifat-sifat plastik. Perbandingan antara jenis-jenis kajian dibuat. Perbandingan tersebut adalah antara defleksi, masa pengisian, dan isipadu pengecutan termasuklah perangkapan udara dan sempadan penyatuan. Kesimpulannya, perubahan pembolehubah-pembolehubah memberi kesan pelengkungan dasbor kereta.

TABLE OF CONTENTS

LIST OF TABLES

LIST OF FIGURES

LIST OF APPENDICES

LIST OF SYMBOLS

^0C	Degree Celsius
%	Percent
MPa	Mega Pascal
cm3/s	Centimeter cubic per second

LIST OF ABBREVIATIONS

DOE	Design of experiment
ABS	Acrylonitrile-butadiene-styrene
CAD	Computer aided drawing
CAE	Computer aided engineering
FYP	Final year project
MPI	*Moldflow plastics insight*
MoT	Mold temperature
MeT	Melt temperature
PP	Packing pressure
PPT	Packing pressure time
CT	Cooling time

CHAPTER 1

INTRODUCTION

The sole purpose of this project is to optimize molding parameter effect to warpage of car dashboard based on plastic flow simulation software. The basic perspective of this project is injection molding. The definition of this word is shape-forming process in which molten metal or plastic is injected into aluminum, ceramic, or steel molds (shaped like the end product) and squeezed under high pressure. Injection molding is employed mainly in the production of solid objects.

Therefore, as a student of mechanical engineering of University Malaysia Pahang, this project exposes student the field of manufacturing engineering as part of mechanical engineering. The optimization of molding parameter is the first step to make product better.

1.1 BACKGROUND OF PROJECT

There are several steps to finish this project. Firstly, focus on injection molding process. Injection molding is the molten plastic is injected into the mold cavity to form shape with high pressure and high temperature and hold it there until the parts are cool and stiff enough to be removed from the die.

The second step is molding parameters. In this case study, focus on four (4) molding parameters that needed in the injection molding. There are flow rate, injection pressure, mold temperature, and melt temperature. These molding parameters will give effect to the plastic depend on how it will be control. So, the small changes of molding parameters will give significant impact to the plastic characteristics.

Warpage is a distortion where the surfaces of the molded part do not follow the intended shape of the design. Warpage relates to the distortion induced by the inhomogeneous shrinkage and relaxation of residual stress in the part once outside the mold, shrinkage expresses the overall dimensional change as the unconstrained part cool down to ambient temperature. [1-4]

There are many molding parameters effect to warpage such as mold cooling, part and mold designs, and process conditions, injection pressure, back pressure, melt temperature and mold temperature. In this case study, the molding parameters that considered are stated before; flow rate, injection pressure, mold temperature, and melt temperature. All these parameters must be calculated to get the best plastic with lowest warpage compare than before.

In this case study, the needed is optimizing molding parameter effect to warpage of car dashboard based on plastic flow simulation software and reverse engineering method. Reverse engineering means that the process of duplicating an existing part, subassembly, or product, without drawings, documentation, or a computer model. It also defined as the process of discovering the technological principles of a device, object or system through analysis of its structure, function and operation. It often involves taking something apart and analyzing its workings in detail, used in maintenance or to try to make a new device or program that does the same thing without copying anything from the original. It’s used to make car dashboard more stable in contain, strength, and brittle with minimum failure. [13, 14]

1.2 OBJECTIVES

The objectives of this case study are to:

1. Study the parameters effect in injection molding to car dashboard
2. Analyze the effect of molding parameters such as melt flow rate, injection pressure, mould temperature and melt temperature to warpage of car dashboard.
3. Optimize the molding parameters by using plastic flow simulation software in order to get a good result.

1.3 PROJECT SCOPES

To start of this project, a thoroughly research of review is done with the means of the internet, books, available published articles and materials that related to the title and supervisor's guidance. This is a continuing progress until sufficient knowledge is attained to complete the project.

The original idea of this project is using the reverse engineering in 3D scanning existing product in laboratory which is car dashboard. Unfortunately, the 3D scanner in the Faculty of Mechanical Laboratory cannot be used because of the technical problem. Due to that problem, this project changed to the manual measurement method.

Next is modeling the product in design software. For this project, the software called SolidWorks is chosen. SolidWorks is a 3D mechanical CAD (computer-aided design) program that runs on Microsoft Windows and was developed by SolidWorks Corporation. The reason using this software are solid-based solid modeler, utilizes a parametric feature-based approach to create models and assemblies. [8, 9]

Here, parameters refer to constraints whose values determine the shape or geometry of the car dashboard. Numeric parameters can be associated with each other through the use of relations, which allow them to capture design intent. Building a model in SolidWorks usually starts with a 2D sketch. The sketch consists of geometry such as points, lines, arcs, conics, and splines. Dimensions are added to the sketch to define the size and location of the geometry. Relations are used to define attributes such as tangency, parallelism, perpendicularity, and concentricity. The parametric nature of SolidWorks means that the dimensions and relations drive the geometry, not the other way around. The dimensions in the sketch can be controlled independently, or by relationships to other parameters inside or outside of the sketch. [8, 9]

After do the car dashboard shape and it will analyze by using plastic flow software. The flow analysis software can predict the flow of plastic throughout the injection molding cycle to ensure that acceptable parts are designed for the manufacture. Using flow analysis, the optimizing gate locations and processing conditions assess possible part defects, and automatically determine the dimensions for a balanced feed system by considering the low effects resulting from differential mold-block temperatures and asymmetries in part geometry. One can also determine the optimum set-up for polymer valve gate timing sequences and determine whether hot or cold runners are appropriate.

Then optimize using mold flow software. Once the plastics injection and packing phases have been optimized, cooling circuits and a mold block- including mold inserts- can be modeled around a part for a mold cooling analysis. The mold design can be optimized by adjusting the size and locations of cooling circuits, as well as modifying circuit and coolant processing parameters. All the benefits of mold cooling analysis can be applied to gas injection molded parts as well.

Finally the molding process optimization can be done by using proper method and processing condition. Thus, the molding process will produce better product in a process.

1.4 PROBLEM STATEMENT

From research that have been done, determine optimal process parameter settings critically influence productivity, quality, and cost of production in the plastic injection molding industry. Previously, production engineers used trial-and-error method to determine optimal process parameter setting for plastic injection molding. However, this method is unsuitable in present plastic injection molding because the increasing complexity of product design and the requirement of multi-response quality characteristics. This research presents an approach in plastic flow software for the optimization of the parameters effect to the car dashboard.

In this project, what is needed is to use the plastic flow simulation software. This software includes all the information that needed in optimizing the warpage of car dashboard. After get the best result, the values of the parameters are applied to production line using injection molding. All these thing are very important to make sure that product can have long life time depends on its strength, capability working with higher temperature and vibration.

CHAPTER 2

LITERATURE REVIEW

2.1 INTRODUCTION

For the injection molding cycle to begin, four criteria must be met: mold open, ejector pins retracted, shot built, and carriage forward. When these criteria are met, the cycle begins with the mold closing. This is typically done as fast as possible with a slow down near the end of travel. Mold safety is low speed and low pressure mold closing. It usually begins just before the leader pins of the mold and must be set properly to prevent accidental mold damage. When the mold halves touch clamp tonnage is built. Next, molten plastic material is injected into the mold. The material travels into the mold via the sprue bushing, and then the runner system delivers the material to the gate. The gate directs the material into the mold cavity to form the desired part. This injection usually occurs under velocity control.

When the part is nearly full, injection control is switched from velocity control to pressure control. This is referred to as the pack/hold phase of the cycle. Pressure must be maintained on the material until the gate solidifies to prevent material from flowing back out of the cavity. Cooling time is dependent primarily on the wall thickness of the part but also depends on the material being molded. Production molding usually requires faster cooling. Water is often channeled throughout the dies to produce faster cooling times. During the cooling portion of the cycle after the gate has solidified, plastication takes place.

Plastication is the process of melting material and preparing the next shot. The material begins in the hopper and enters the barrel through the feed throat. The feed throat must be cooled to prevent plastic pellets from fusing together from the barrel heat. The barrel contains a screw that primarily uses shear to melt the pellets and consists of three sections. The first section is the feed section which conveys the pellets forward and allows barrel heat to soften the pellets. The flight depth is uniform and deepest in this section. The next section is the transition section and is responsible for melting the material through shear. The flight depth continuously decreases in this section, compressing the material. The final section is the metering section which features a shallow flight depth, improves the melt quality and color dispersion. At the front of the screw is the non-return valve which allows the screw to act as both an extruder and a plunger. When the screw is moving backwards to build a shot, the non-return assembly allows material to flow in front of the screw creating a melt pool or shot. During injection, the non-return assembly prevents the shot from flowing back into the screw sections.

Once the shot has been built and the cooling time has timed out, the mold opens. Mold opening must occur slow-fast-slow. The mold must be opened slowly to release the vacuum that is caused by the injection molding process and prevent the part from staying on the stationary mold half. This is undesirable because the ejection system is on the moving mold half. Then the mold is opened as far as needed, if robots are not being used, the mold only has to open far enough for the part to be removed. A slowdown near the end of travel must be utilized to compensate for the momentum of the mold. Without slowing down the machine cannot maintain accurate positions and may slam to a stop damaging the machine. Once the mold is open, the ejector pins are moved forward, ejecting the part. When the ejector pins retract, all criteria for a molding cycle have been met and the next cycle can begin.[11]

The basic injection cycle is as follows: Mold close – injection carriage forward – inject plastic – metering – carriage retract – mold open – eject part(s) Some machines are run by electric motors instead of hydraulics or a combination of both. The water-cooling channels that assist in cooling the mold and the heated plastic solidifies into the part.

Improper cooling can result in distorted molding. The cycle is completed when the mold opens and the part is ejected with the assistance of ejector pins within the mold.

The raw material for injection molding is most commonly supplied in pellet or granule form. Pellets are poured into the feed hopper, a large open bottomed container, which is attached to the back end of a cylindrical, horizontal barrel. A screw within this barrel is rotated by a motor, feeding pellets up the screw's grooves. The depth of the screw flights decreases toward the end of the screw nearest the mold, compressing the heated plastic. As the screw rotates, the pellets are moved forward in the screw and they undergo extreme pressure and friction which generates most of the heat needed to melt the pellets. Electric heater bands attached to the outside of the barrel assist in the heating and temperature control during the melting process.

The channels through which the plastic flows toward the chamber will also solidify, forming an attached frame. This frame is composed of the *sprue*, which is the main channel from the reservoir of molten resin, parallel with the direction of draw, and *runners*, which are perpendicular to the direction of draw, and are used to convey molten resin to the *gate(s)*, or point(s) of injection. The sprue and runner system can be cut or twisted off and recycled, sometimes being granulated next to the mold machine. Some molds are designed so that the part is automatically stripped through action of the mold.

Optimizing the parameters for injection molding will ensure quality products, with minimum of molded-in-stress, at lower manufacturing cost. All parameter adjustment will have an effect (positive or negative) on the physical and aesthetic properties of the molded product. Understanding this relationship allows the molder to manipulate the properties to meet specific requirement establish for the product.

2.2 PARAMETERS OF THE MOLDING PROCESS

There are numerous variable affect the injection molding process. For instance, injection pressure consisted of more than one item. There is initial injection pressure, second and up to fifth stage injection pressure, holding pressure, back pressure, and line pressure. All of these give direct effect on each other. Humidity and ambient temperature also have effect on the molding process. Shift changes, relief operators, fan blowing, housekeeping, age of equipment, size of machine, location of press, pressure of cooling water, all seemed to have direct or indirect effect on the injection molding process. Although there are so many different variables, it is not impossible to get control of the injection molding process. What is needed is a more practical approach to understanding all these parameters, and targeting those that have the greatest effect on the overall quality and cost effectiveness of the finished molded product. For all these, there are four main categories of parameters: temperature, pressure, and time. All these relative importance of the categories is shown by size of the circles. Thus, temperature is the most important, followed by pressure, and time. However, each is dependent on the other, and changing one will affect one or all of the others. [10-13]

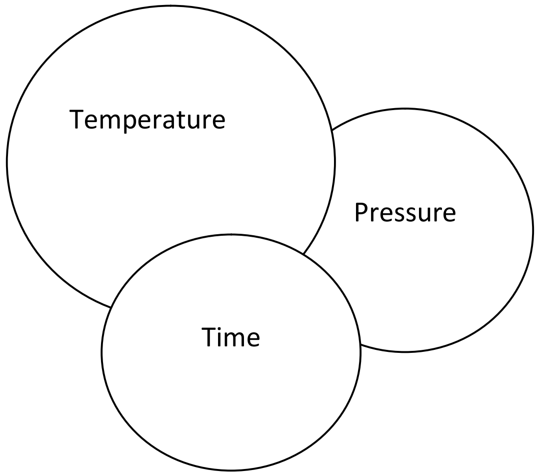


Figure 2.1: Main processing parameters [10]

2.2.1 Temperature

2.2.1.1 Melt Temperature Control

Melt temperature is the temperature at which the plastic material is maintained throughout the flow path. This path begins where the plastic material is transferred from the machine hopper into the cylinder of the injection unit. Then the material is augered through the heating cylinder and into the machine nozzle. From there it is injected into the mold, where it must travel along a runner system (if one exists), through the gates, and into the cavities that are machined into the mold. The temperature of the melt must be controlled along the path, starting with the heating cylinder. [10][12]

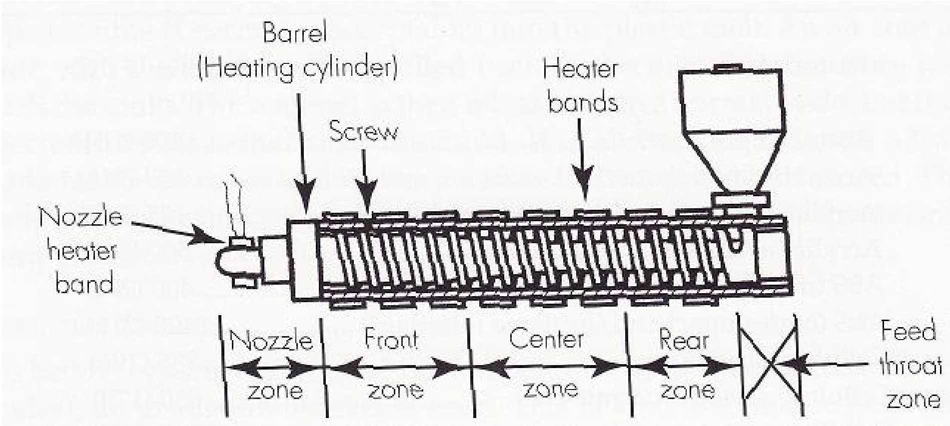


Figure 2.2: Heating cylinder (barrel) [10]

Figure 2.2 shows the heating cylinder is wrapped with heater bands. These are electrical heaters shaped like hinged bracelets that mount around the outside of the heating cylinder. There are three main heating zones to the heating cylinder: the rear zone, the center zone, and the front zone. In addition, there is usually at least one heater band fastened around the machine nozzle, an area referred to as the nozzle zone.

The plastic for the injection process should be brought up to proper temperature gradually as the material drops from the hopper into the rear zone, where the initial heat begins to soften the material. Then the material is augered forward by the screw into the center zone where the temperature is generally 10 to 20^{o}F (5.6 to 11^{o}C) higher than the in the rear zone. As the material travels to the front zone, the temperature is gain increased by 10 to 20^{o}F, and finally the material is ready to be injected into the mold. It is held at this point until the previous molding cycle is complete, at which time the mold opens, parts are ejected, the mold closes, and the next cycle begins. The charge of preheated plastic is then injected into the mold.[10-13]

Besides absorbing heat from the externally mounted heater bands, the plastic material absorbs a large amount of heat from the friction caused by the augering action of the injection screw. The screw rotates to bring fresh material into the heating cylinder and prepare it for the coming cycle. While being pulled along, the new material is squeezed between the flights of the screw and the inside wall of the injection barrel. The friction generates heat, which is absorbed by the plastic. [10-13]

The main point here is that plastic must be heated to the proper temperature for injection. Melt is measured at the nozzle as the plastic exist the machine, before it enters the mold. It is measured by taking an “air shot” and plunging a probe from a measuring instrument with a fast response time (1 second is acceptable) into the plastic melt. An air shot is made with the injection sled pulled back so the injection unit does not touch the mold. The material is then released as in a normal cycle, but it is injected into air rather than mold. It is allowed to fall onto a tray made for the purpose and its temperature is then quickly measured. The temperature at that point should be within 10^{o}F (5.6^{o}C) of the desired temperature.

2.2.1.2 Mold Temperature Control

The plastic material is now ready to flow into the mold. First, it must travel through the machine nozzle, which is the last heating zone provided by the machine. After the material exits the nozzle and enters the mold, it immediately begins to cool down as the mold absorbs heat from it. The rate at which this heat is absorbed determines how far the plastic will flow before it begins to solidify and stop moving. Each product depending on its design and plastic material, demand specific cooling rates, and this rate of cooling is critical to product quality. Therefore, the mold temperature must be regulated in order to regulate the cooling rate of the plastic. This is done by connecting the mold to a temperature control unit that normally utilizes water as medium. The water is circulated through the mold and held at a preset temperature by heating or cooling in cycles. [10-13]

Every combination of plastic and product has a specific temperature at which the mold should be maintained to ensure quality molding. The mold temperature is measured directly from the molding surface of the tool with a solid probe on a pyrometer device. Usually, readings from several areas are averaged. The object of the cooling process is to lower the temperature of the molded plastic to the point at which it solidifies again. When the plastic reaches that point, it can be ejected from the mold with relative structural safety. [10-13] That simply means that the plastic part will not move excessively, causing warpage, twisting, or other shrinkage related problem as the plastic continues to cool.

2.2.1.3 Hydraulic System Temperature Control

Besides melt temperature and mold temperature, there is the temperature of the hydraulic system of the press to be considered. The temperature of the hydraulic oil in these system must be maintained between 80 and 140^{o}F (27 and 60^{o}C), in most case. If the oil is too cool, it will be thick (viscous) and cause sluggish action of hydraulic component. If it is too hot, it will break down, causing components to stick or valves to malfunction. The temperature of the oil is regulated by a heat exchanger mounted on the injection machine. This exchanger acts like a radiator on car and cools the oil by circulating it around tubes filled with circulating water. These tube must be kept clean and require periodic flushing with an acid cleaner. If the oil is allowed to overheat, that will eventually transfer throughout the entire machine, including the platens to which the mold is mounted. This will cause the mold to overheat and result in poor quality parts. [10-13]

2.2.2 Pressure

There are two areas in the injection machine that required pressure and pressure control: the injection unit and the clamp unit. They are closely related in that they are opposing pressure – the clamp unit must develop enough clamp pressure to overcome the pressure developed by the injection unit during the molding process.

2.2.2.1 Injection unit

Three basic types of pressure are developed by the injection unit: initial pressure, hold pressure, and back pressure.

Initial Injection Pressure

This is the first pressure that applied to the molten plastic. It develops as result of main system hydraulic pressure pushing against the back end of the injection screw (or plunger)(figure 2.3).

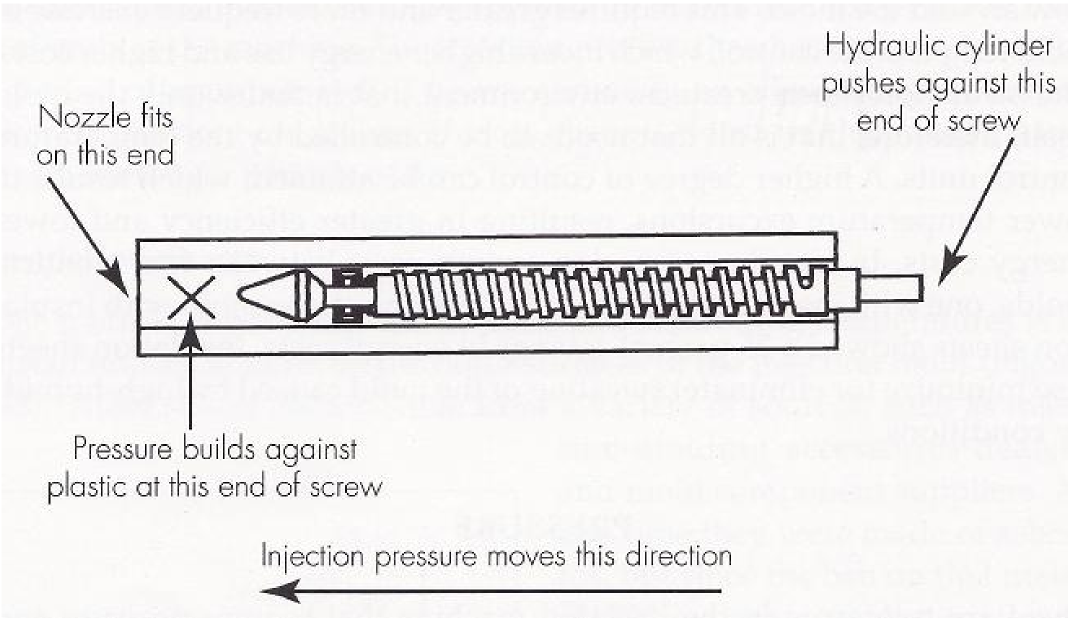


Figure 2.3: Developing injection pressure [10]

The amount of pressure developed by the main system is on the order of 2000 psi (13,789 kPa). Some systems are capable of producing more than that, but 2000 psi is the common line pressure. This pressure is converted to a maximum of 20,000 psi (137,890 kPa) at the nozzle of the injection unit (where the plastic first enters the mold) by the design and shape of the injection screw. In most cases, the full 20,000 psi is not required for filling a mold, and most products can be molded in a range of from 5000 to 15,000 psi (34,472 to 103,418 kPa). The pressure actually required depends on the plastic being molded, the viscosity and flow rate of the plastic, and the temperatures of the plastic and the mold. [10-13]

The ideal situation is to be able to fill the mold initially with the highest practical pressure in the shortest practical time. Normally, the initial fill can be accomplished in less than 3 second. Note that even though “the highest pressure practical pressure” should be used, a constant effort should be made to keep that practical pressure requirement low so molded in stresses are minimized.

To summarize, initial injection pressure is to used to create the initial filling of the mold. It should be set at the highest practical value to fill the mold with the fastest practical speed.

Holding Pressure (Secondary Pressure)

This pressure is applied at the end of the initial injection stroke (figure 2.4) and is intended to complete the final filling of the mold and hold pressure against the plastic that was injected so it can solidify while staying dense and “packed”. As a rule, the amount of pressure used here can be half the initial injection pressure or less.. so, if initial pressure was 12,000 psi (82,734 kPa), the holding pressure can be approximately 6000 psi (41.367 kPa). The holding pressure is actually applied against a cushion or pad of material. [10-13]

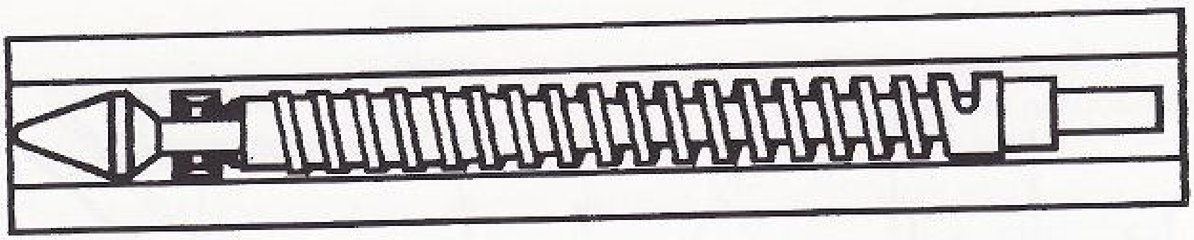


Figure 2.4: Applying holding pressure [10]

To summarize, holding pressure is used to finish the filling of the mold and pack the plastic material into the cavity image.

Back Pressure

Back pressure is applied after the injection phases mentioned above. When the hold pressure phase is complicated, a signal is sent to the machine to start turning the screw to bring new material to the front of the barrel in preparation for the next cycle, or shot (so called because plastic shoots into the mold). The screw is not pulled back. Instead, the churning, or augering, action of the screw brings new material forward, and as that material fills up in front of the screw, the material itself begins to push the screw backward (figure 2.5). [10-13]

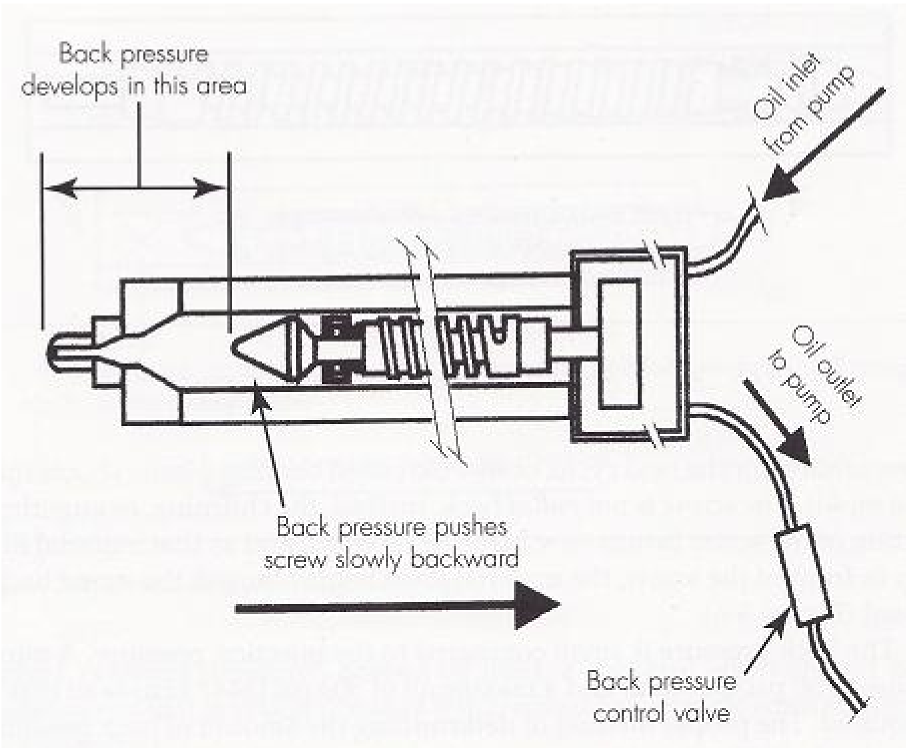


Figure 2.5: Applying back pressure [10]

The back pressure is small compared to the injection pressure. A minimum of 50 psi (345 kPa) and maximum of 500 psi (3447 kPa) is all that is required. The proper method of determining the amount of back pressure is to begin at 50 psi increase, only if necessary, in increment of 10 psi (69 kPa) until the proper mix and density are achieved. Use of back pressure helps ensure consistency in part weight, density, and material appearance. It also helps to squeeze out any trapped air or moisture not eliminated by predrying the material. This minimizes (or even eliminates) voids in the molded product. If less than 50 psi back pressure is attempted, the controls and gages are not consistent or accurate enough to maintain or indicate the actual pressure being developed. Thus, faulty readings and settings can occur. If more than 500 psi back pressure is attempted, and the screw may not return at all, or it will stay forward mush too long, and the plastic material will degrade under the extreme shear imparted to it from the continued churning action of the screw. In this case of reinforced plastic (such as glass filled), the reinforcement material will break down, and this results in much less strength than is required in the molded product. [10-13]

2.2.2.2 Clamp Unit

The purpose of developing clamp pressure is to keep the mold clamped shut against the force developed when injection pressure pushed plastic into the closed mold. Therefore, the amount of clamp force must be at least equal to the amount of injection force. Clamp pressure is applied to the mold either hydraulically or mechanically. There are advantages and disadvantages associated with each method.

Hydraulic Clamp System

In this method, the clamping force is developed by hydraulic cylinder. A piston from the cylinder is attached to a moving platen on which the mold is mounted.

The greatest advantage of this type of clamp system is that the clamp pressure can be regulated over a wide range. For instance, if the machine is rated at a 250-ton clamp force, the clamp force can be set anywhere from approximately 50 tons to the full 250 tons (445 to 2225 kN). This allows the proper clamp tonnage to be use for the specific job and minimizes the amount of energy expended. Using more tonnage than necessary not only wastes money, but may cause extensive damage to the mold or the machine or both because of the crushing forces applied. The greatest disadvantages of the hydraulic clamp is that when tonnage requirements approach the maximum rating, extreme injection pressures may overcome the clamp force and blow the mold open. For instance, if a mold requiring 225 tons is placed in a 250-ton machine and the injection pressure is on the high side, the potential exists for the injection pressure to overpower the clamp pressure, in which case the mold will open while plastic is being injected. This results in flash, short shots, and possible cycle interruption. [10-13]

Mechanical Clamp (Toggle) system

The mechanical system utilizes a knuckle and scissors (toggle) mechanism to close the mold.. the toggle is attached to the moving platen on which the mold is mounted. When the clamp is open, a small hydraulic cylinder actuates the arms by pushing along their centerline. As the piston moves forward, it pulls the arms together, closing the mold. [10-13]

For the mold to close under full tonnage, the knuckle must actually pass center to lock. if they do not lock, they will not hold in the forward position and the injection pressure will blow the mold open. This can be demonstrated by watching a person push an arm straight out from the body. When the elbow is past center and straight, the arm is locked in the forward position and is difficult to push back until the elbow is relaxed. [10-13]

The principle advantage of the mechanical system is that once it is locked in place, it is virtually impossible to blow the mold open even if injection pressures are beyond those required. Of course, there are limits to the pressure it can sustain, and eventually machine damage will occur if the injection pressures are held beyond requirements for extended periods. But once the system locks, there is no doubt that full tonnage force is available.[10-13]

There are two distinct disadvantages to this system. First, there is considerable wear on the knuckle linkages and bushings must be replaced regularly. Second, there is little accommodation for adjustment on this system. If the machine is rated at 250 tons, the only tonnage available is 250 tons. It cannot be reduced, except minimally. Thus a smaller, borderline mold could not be run in this press without the distinct possibility of damage from crushing.

Some machines combine both hydraulic and mechanical systems for mold clamping, and some even incorporate electric motors for performing the mechanical action instead of hydraulic cylinders. [10-13]

2.2.3 TIME

2.2.3.1 Gate-to-gate Cycle Time

During the injection molding process, many internal activities are taking place. Some occur simultaneously with others (parallel activities), and some must wait until others are completed (serial activities). The overall cycle time provides a measure of the time required for all these activities. This is usually referred to as the gate-to-gate cycle time because it is common to start timing the overall cycle as soon as the machine operator closes the safety gate of the machine. The timing continues until the operator closes the same gate of the machine. The timing continues until the operator closes the same gate to start the next cycle. The entire amount of time elapsed between these two action is the gate-to-gate or overall cycle time. The cycle time provides the only way to get an accurate picture of how long it takes to mold a product. This number is then used to determine the actual cost the involved to manufacture the product. This, in turn, is used to determine the selling price of the product. [10-13]

2.2.3.2 Initial Injection Time

When the mold closes completely, either a limit switch or pressure buildup signal the injection screw to push forward and inject the molten plastic into the closed mold. The screw does not turn at this point, but only acts as a plunger to force the material into the mold. This initial injection is performed at the highest practical pressure for the specific application in the fastest practical amount of time. In most cases, the time is less than 2 second and rarely more than 3 seconds. Sometimes, depending on machine design, this action is divided into two or three smaller actions. Then, the total injection time normally does not exceed 4 to 5 seconds. The initial injection time is controlled by a timer. If booster injection is available, it will be included in the first stage of the initial injection time. When boaster phase is included, the injection machine's entire hydraulic system is combined to push a large volume of oil through the system. This can increase the speed at which the material is injected into the mold. [10-13]

2.2.3.3 Injection Hold Time

On most machines, the timer for initial injection time controls the total amount of time that the injection screw is pushing forward. The initial injection time is the first part of that time, and injection hold is the latter part. On some machines, the hold time and initial time are on separate timers. [10-13]

The hold time is the amount of time the injection screw maintains pressure against the plastic after it has been injected into the mold. This pressure is applied against the cushion or pad long enough for the gate to freeze off. The molten plastic enters the mold cavity image through a gate. The gate is the first point at which the plastic actually "sees" the cavity image. Once all the required material goes through the gate and packs the cavity image, the plastic is allowed to cool under hold pressure, down to the point at which it all solidifies. But, because it is normally the thinnest part of the cavity image, the gate is the first thing to solidify. When it does, there is no reason to maintain pressure because the plastic that is in the cavity lies beyond the solidified gate and the pressure from the injection unit no longer has any effect on it. So, pressure is held against the gate only long enough for the gate to freeze. In most cases, this is only a matter of a few seconds. A gate with a thickness of 0.16 cm would take approximately 6 seconds to solidify. [10-13]

2.2.3.4 Cooling time

Cooling time is probably the most important time in the entire injection process. It is the amount of time required for the plastic material to cool down to the points at which it has solidified and an extra amount of time to allow the plastic part to become rigid enough to withstand the ejection process. Even though the plastic may cool enough to solidify, it may not be rigid enough to be ejected. This is because the curing process actually takes as long 30 days to finalize. The initial curing is rapid, and 95 percent of the total curing takes place in the mold. But other 5 percent takes place outside the mold. If the outer skin of the plastic product is solidified to a sufficient depth, the remaining cooling will not have an appreciable effect on the molded part. But if the skin is too thin, the remaining cooling will cause shrinkage stress to build up and the molded part may warp, twist, blister, or crack. [10-13]

The key to minimizing this problem is to keep the part in the mold for a sufficiently long time, but no longer than necessary because time is money, and long cycle is expensive. Most material suppliers are more than happy to share cooling time requirements for their materials at varying thickness, but, on average, a 1/16-in. thick wall should take approximately 9 to 12 second to solidify to the point at which it can be ejected from the mold without undue distortion.

2.2.3.5 Ejection Time

When the cycle is completed and the mold has opened fully, the ejection system is allowed to come forward and knock the parts out of the mold. This action is normally starts by limit switch that actuates upon the full opening of the mold. However, sometimes it is performed through mechanical stops and actions. The ejection stroke itself is normally controlled by another limit switch that actuates when the right amount of ejection has taken place, but the speed at which the system comes forward must be controlled and this is what determines the amount of time required. There is still a partial vacuum in the cavity image. Therefore, the ejection system is pushed forward at slow enough rate to overcome the vacuum, but fast enough to be practical. Ejection normally lasts for 1 to 2 seconds, depending on the length of the ejection necessary. [10-13]

The ejection system then must return before the next cycle can start. In some cases, it is not necessary to return the system because the closing of the mold will perform that action. Such practice is not recommended, however, because mold damage may occur. [10-13]

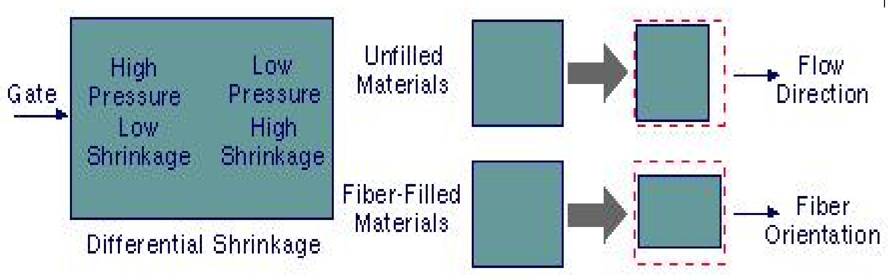
Also, there are cases when double ejection is required to “kick” a reluctant part off the ejector pins. This means that the ejector system comes forward, returns, and comes forward again before finally returning in preparation for the next cycle. This practice is called pulsing and, in effect, it doubles the amount of ejection time and the amount of wear on the ejection system. It is not recommended. [10-13]

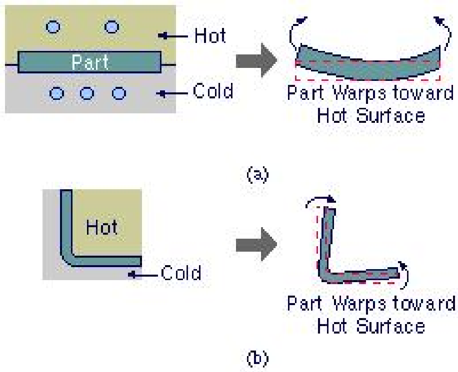
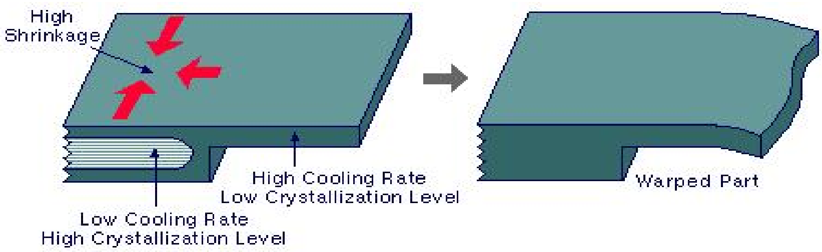
Parameter	**Average value**
Gate closing time	1 second
Mold closing time	4 seconds
Initial injection time	3 seconds
Injection hold time	5 seconds
Cooling time	12 seconds
Screw return time	8 seconds
Mold open time	4 seconds
Ejection time	1 second
Part removal time	2 seconds
Mold inspection, clean, spray, etc	2 seconds
Total time/cycle	**42 seconds**

Table 2.1: Average Times for Cycle Activities [10]

2.3 WARPAGE

Warpage is always due to differential shrinkage. Warpage in molded parts results from differential shrinkage. Variation in shrinkage can be caused by molecular and fiber orientation, temperature variations within the molded part, and by variable packing, such as over-packing at gates and under-packing at remote locations, or different pressure levels as material solidifies across the part thickness. [1-4]

Differences in filled and unfilled materials	• Differential shrinkage for filled and unfilled materials is shown below. When shrinkage is differential and anisotropic across the part and part thickness, the internal stresses created can lead to part warpage.  **Differential shrinkage for both unfilled and filled materials** • For fiber-filled thermoplastics, reinforcing fibers inhibit shrinkage due to their smaller thermal contraction and higher modulus. Therefore, fiber-filled materials shrink less along the direction in which fibers align (typically the flow direction) compared to the shrinkage in the transverse direction. Similarly, particle-filled thermoplastics shrink much less than unfilled grades. • On the other hand, if an unfilled molded part contains high levels of molecular orientation, shrinkage is anisotropic because aligned chains shrink to a greater extent in the direction of orientation.

Non-uniform mold cooling across the part thickness	• Non-uniform cooling in the part and asymmetric cooling across the part thickness from the mold cavity and core can also induce differential shrinkage. The material cools and shrinks inconsistently from the mold wall to the center, causing warpage after ejection.  **Part warpage due to (a) non-uniform cooling in the part, and (b) asymmetric cooling across the part thickness.**
Part thickness variation	• Shrinkage increases as the wall thickness increases. Differential shrinkage due to non-uniform wall thickness is a major cause of part warpage in unreinforced thermoplastics. More specifically, different cooling rates and crystallization levels generally arise within parts with wall sections of varying thickness. This causes differential shrinkage, resulting in part warpage, as shown below.  **Larger volumetric shrinkage due to the high crystallization level in the slow cooling areas (e.g., the thick sections) leads to differential shrinkage and thus part warpage.**

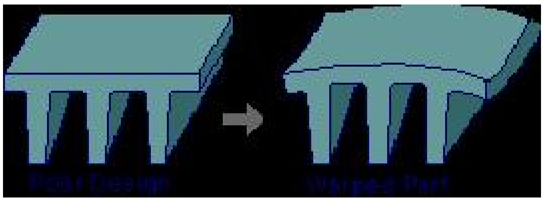
Part geometry asymmetry or curvature	• Geometric asymmetry (e.g., a flat plate with a large number of ribs that are aligned in one direction or on one side of the part) will introduce non-uniform cooling and differential shrinkage that can lead to part warpage, as shown below.  **The poor cooling of the mold wall on the ribbed side causes a slower cooling of the material on that one side, which can lead to part warpage**

Figure 2.6: Types of shrinkage

2.4 MOLDFLOW

Moldflow offers a range of a products and services in the plastics injection molding industry. It is easy to learn 3D solids based plastics flow simulation products allow you to determine the manufacturability of your part in the early design stages and avoid potential downstream problems which can lead to production delays and cost overruns. [19]

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

Cooling analysis (MF/COOL); it analyses the effect cooling on flow, optimizes cooling line geometry and processing conditions. Process Optimization Analysis (MF/OPTIM); it gives optimized processing parameters for a component considering injection molding conditions. Warpage Analysis (MF/WARP); this analysis simulates the effect of moldings on product geometry, isolates the dominant cause of warpages so that the correct remedy can be applied. [19]

2.4.1 MOLDFLOW PLASTICS INSIGHT (MPI)

Moldflow Plastic Insight products are a complete suite of advanced plastics process simulation tools for predicting and eliminating potential manufacturing problems simulations tools for predicting and eliminating potential manufacturing problems nd optimizing part design, mold design and the injection molding process. MPI products simulate the broadest range of manufacturing processes. With MPI, one can simulate the filling, packing and cooling stages of the thermoplastics injection molding process and also predict the resultant fiber orientations and take that into account when predicting part warpage. MPI users can also simulate other complex molding process such as gas assisted injection molding, co-injection molding, injection-compression molding, microcellular molding, reactive molding, and microchip encapsulation. [19]

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

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

The Moldflow Plastics Insight suite of software is the world leading product for the in-depth simulations to validate part and mold design. Companies around the world have chosen Moldflow's solution because they offer; Unique, Patented Fusion Technolgy. MPI/Fusion, which is based on Moldflow's patented Dual Domain^{TM} Technology, allows you to analyze CAD solid models of thin-walled parts directly, resulting in a significant decrease in model preparation time. The time savings allow you to analyze more design iterations as well as perform more in depth analyzed. [19]

CHAPTER 3

METHODOLOGY

3.1 INTRODUCTION

This chapter discuss about methodology of the project. Besides that, this chapter will show the time line of this project from the start until the project is finished successfully. For this project, the time line starts when receive the project title and start it with some briefing session with supervisor..

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

In this project, analysis by using Moldflow software, MPI 5.0, is the main step in getting result. Through the analysis, comparison of the result will be done. It is important that the analysis have going through follow the objective and also the project scope. The results also have to achieve the project objective. Some sequences have been decided to run through the Moldflow software.

3.2 PROTON SAGA DASHBOARD

Figure 3.1 Proton Saga Dashboard [17]

A car dashboard is actually made from plastic injection molding. To find the suitable material for lower cost but has many advantages is much difficult. Now a day, many plastic components especially Perusahaan Otomobil Nasional Sdn. Bhd use a Acrylonitrile Butadiene Styrene (ABS) Plastic Resin in many part of it cars including a car dashboard.

ABS is an amorphous thermoplastic blend. The recipe is 15-35% acrylnitrile, 5-30% butadiene and 40-60% styrene. Depending on the blend different properties can be achieved. Acrylnitrile contributes with thermal and chemical resistence, and the rubberlike butadiene gives ductility and impact strength. Styrene gives the glossy surface and makes the material easily machinable and less expensive.

Generally, ABS has good impact strength also at low temperatures. It has satisfactory stiffness and dimensional stability, glossy surface and is easy to machine. If UV-stabilizators are added, ABS is suitable for outdoor applications.

Recommended mold temperature (◦C)	60
Recommended melt temperature (◦C)	250
Melt density (g/cm3)	0.98843
Solid density (g/cm3)	1.0541
Eject temperature (◦C)	85
Maximum shear stress (MPa)	0.3
Maximum shear rate (s−1)	50000
Thermal conductivity (W/m ◦C)	0.23
Elastic module (MPa)	2600
Poisson ratio	0.38

Table 3.1: Material properties of ABS

3.3 MANUAL MEASUREMENT METHOD

As mentioned before, the original idea cannot proceed because of the technical problem in Faculty of Mechanical Laboratory. The 3D Scanner has a technical problem to do the scanning method for this project. When all this happen, the scope of this project need to change but still use the reverse engineering method. The change is made from using the 3D Scanner to the manual measurement method. This means that all the dimensions of the saga dashboard is measured by using the mechanical tools such as rulers, vernier caliper, threads, and protractor.

3.4 SOLIDWORKS 2008

After all the dimensions of the saga dashboard has been measured, then all dimensions are transferred to SolidWork. In this project, the SolidWorks 2008 is used. All the dimensions are sketched to get the solid part which is a saga dashboard that has been measured earlier by using the manual measurement methods. All the dimensions of the car dashboard must be draw correctly in the correct axis. The part that produce in the SolidWorks 2008 need to save as IGES(*igs) file. The main reason is to export the part to the MoldFlow Plastics Insight (MPI) 5.0 software for simulating process. Below are front view, side view, top view and isometric drawing of product. Please refer to appendix B for technical drawing.

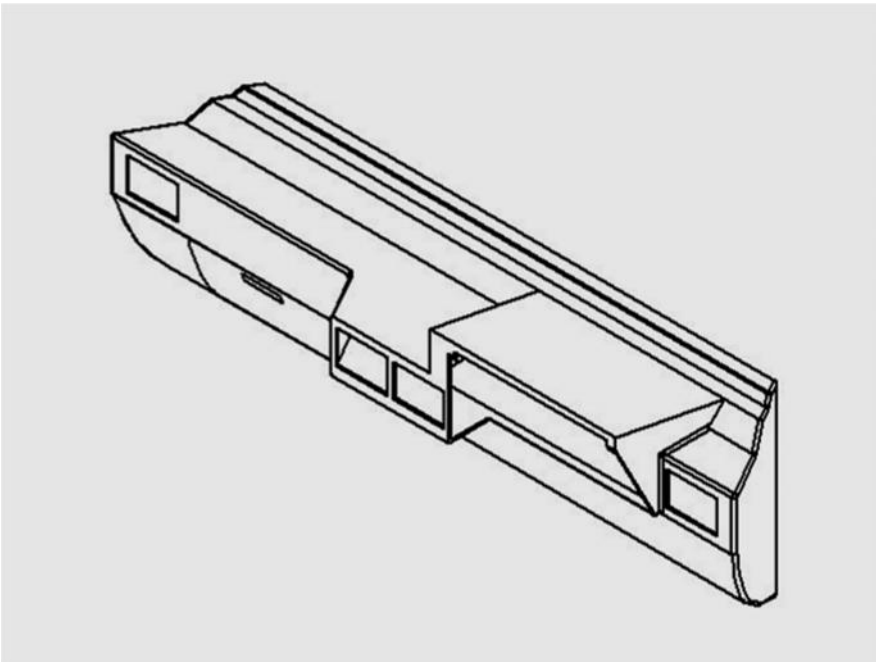


Figure 3.2: Isometric view of Proton Saga Dashboard

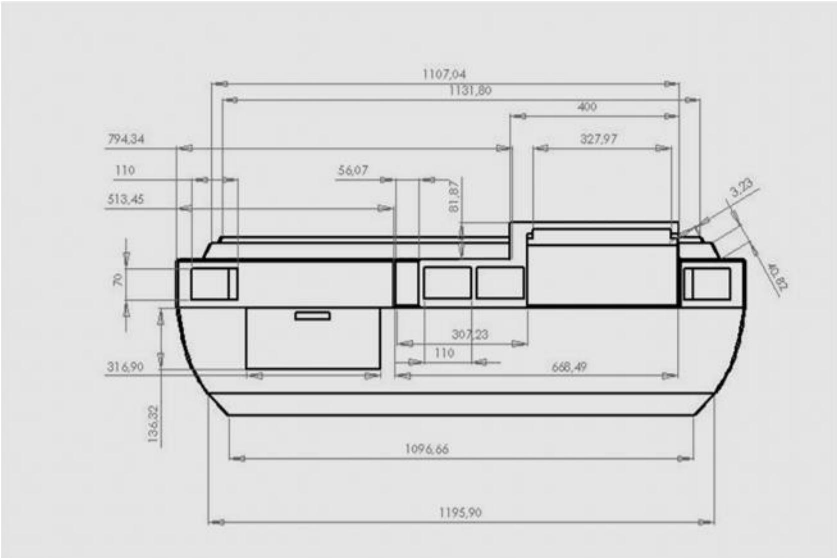


Figure 3.3: Front view of Proton Saga Dashboard

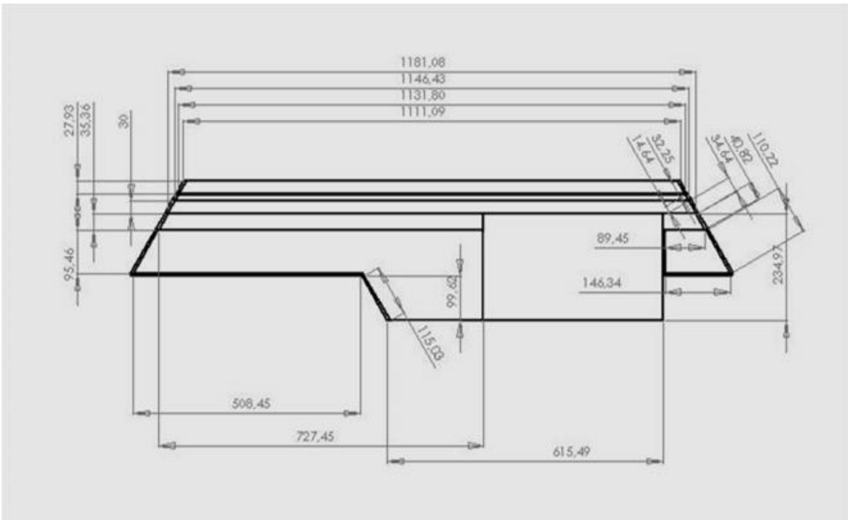


Figure 3.4: Top view of Proton Saga Dashboard

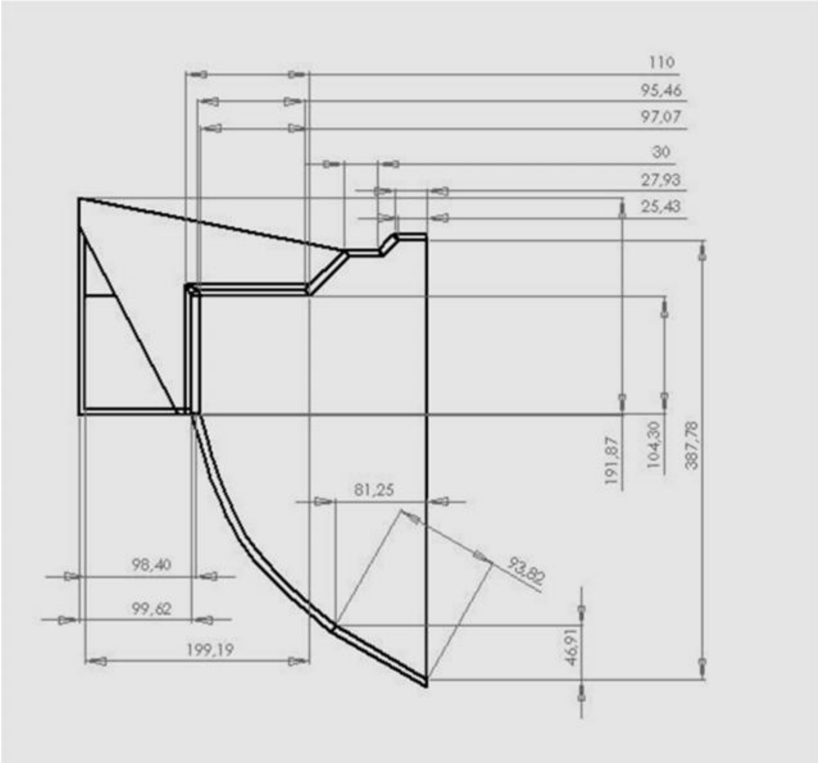


Figure 3.5: Side view of Proton Saga Dashboard

3.5 DESIGN OF EXPERIMENT

Warpage values are calculated by a process parameters generated by full factorial experiment design. In factorial design, a variable range is divided into level between the lowest and the highest values. A three-level full factorial design creates 3^n training data, where *n* is the number of variables. In this study, 3^4=81 training data are generated and utilized for four variables or parameters. Ranges for process parameters are shown in Table 3.2.

Table 3.2

Low-middle-high values of process parameters in three-level full factorial design

Mold temperature (◦C)	40-60-80
Melt temperature (◦C)	230-250-270
Flow rate (cm^3/s)	80-100-120
Injection pressure (MPa)	100-130-160

From the data, the lowest value of warpage displacement has been choosing to optimize again to study the effect parameters to the car dashboard. This lowest value is declared as the best parameters in this project.

3.6 MOLDFLOW PLASTICS INSIGHT (MPI) 5.0 SOFTWARE

The problem is already mentioned in chapter one that to optimize the molding parameter effect to warpage of car dashboard based on plastic flow simulation software. It is important because the warpage during the injection molding process can be controlled by optimized it parameters. The proper way to control this warpage is by using moldflow plastic insight 5.0.

Moldflow Plastics Insight (MPI) software represents the most comprehensive suite of definitive tools for simulating, analyzing, optimizing, and validating plastics part and mold designs. MPI 5.0 have 19 distinct modules that can be used to simulate nine unique molding processes, including plastics flow and packing, mold cooling, and part shrinkage and warpage for thermoplastic, and gas-assisted injection molding environments, as well as co-injection molding and injection-compression molding processes. The software also offers the world's largest database of its kind with more than 7,800 materials characterized for use in plastic CAE analysis

3.7 ANALYSIS

The analysis of project will be perform by simulate the plastic product using Moldflow Plastics Insight software. The purpose of the analysis is to know the effect of injection molding parameters to warpage of car dashboard and the defects occurred to product. The parameters are melt temperature, mold temperature, injection pressure, and flow rate. The cooling time is constant in this project. The analysis need to repeat five (5) times with different parameters. Below are a few steps in using the Moldflow software and the constant process parameters.

Step in running the Moldflow simulation

a) Create new project

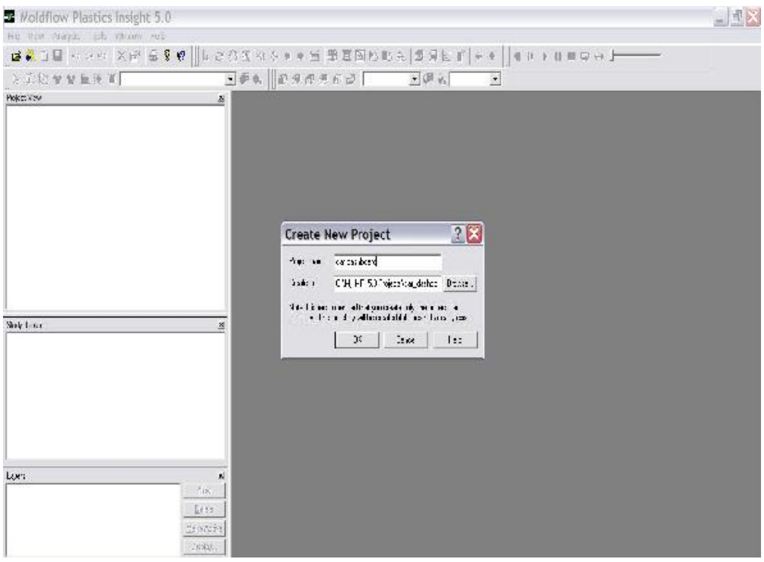


Figure 3.6: Create new project

b) Import 3-D CAD file

i.Import 'iges' file

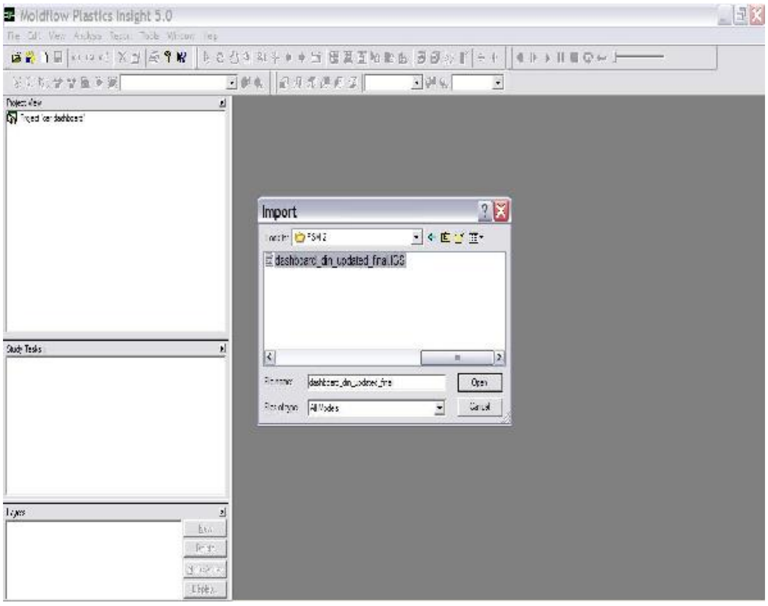


Figure 3.7: Import 'iges' file from CAD drawing

ii.Generate mesh

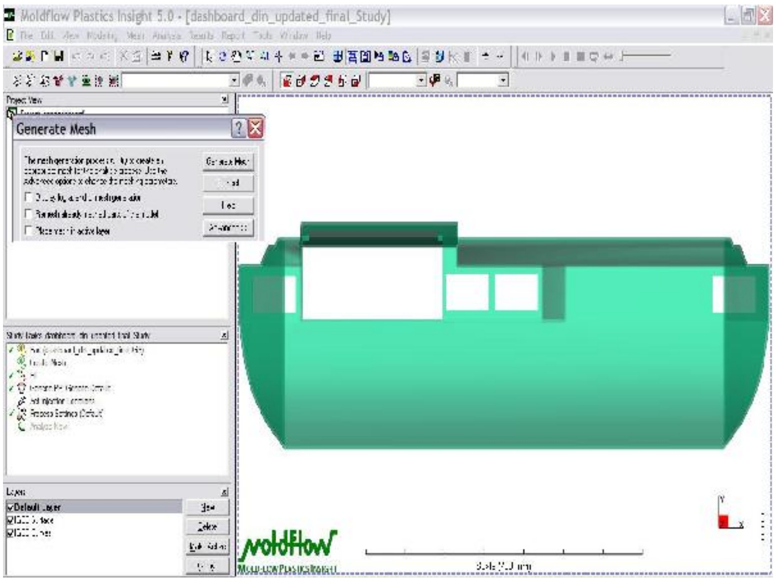


Figure 3.8: Generate the meshing entity

iii.Mesh pattern was appeared as selected mesh type

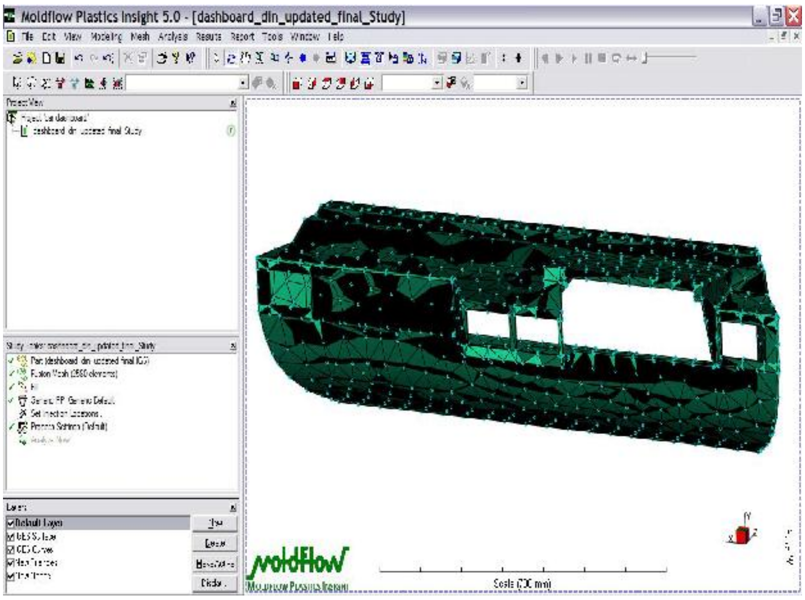


Figure 3.9: Mesh pattern appeared on Saga Dashboard (fusion)

c) Set analysis sequence

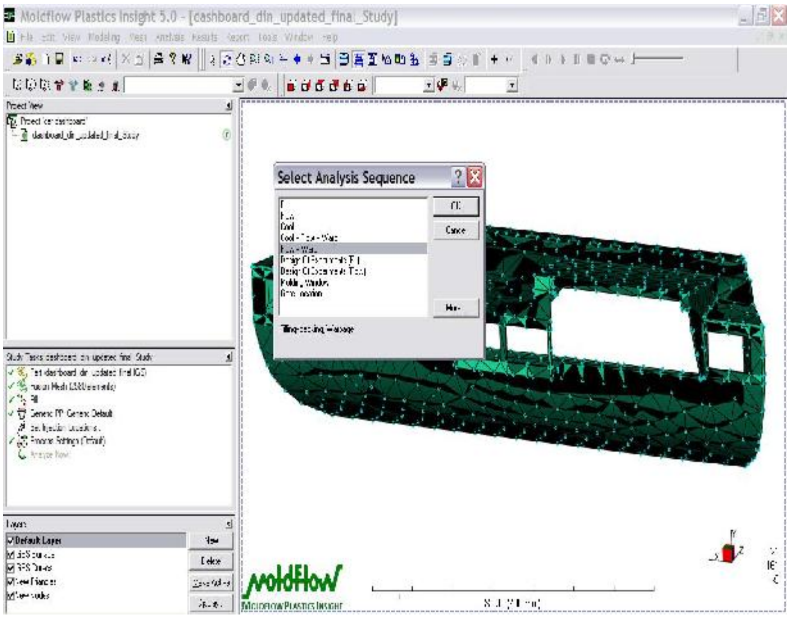


Figure 3.10: Analysis sequence wizard (flow + warp analysis)

d) Select the material

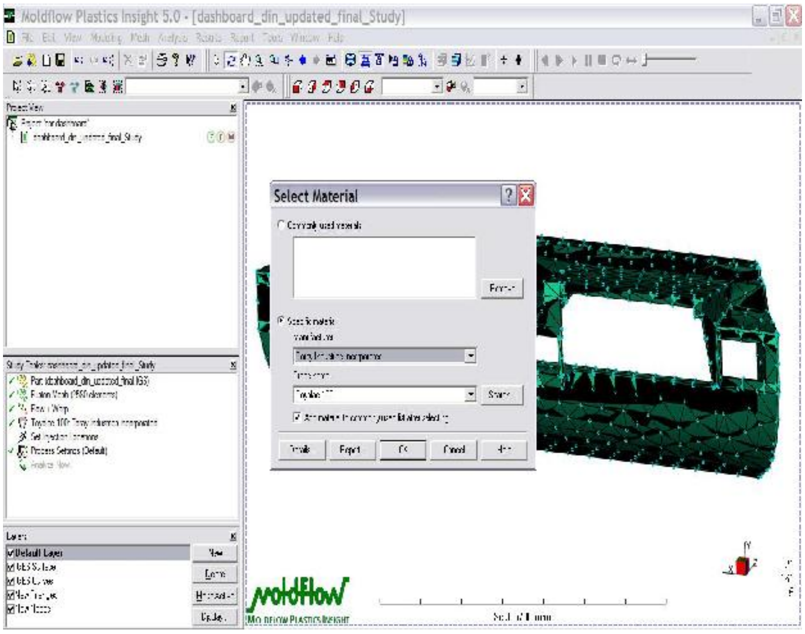


Figure 3.11: Toyolac 100 fom ABS family have been chosen

e) Set gate location

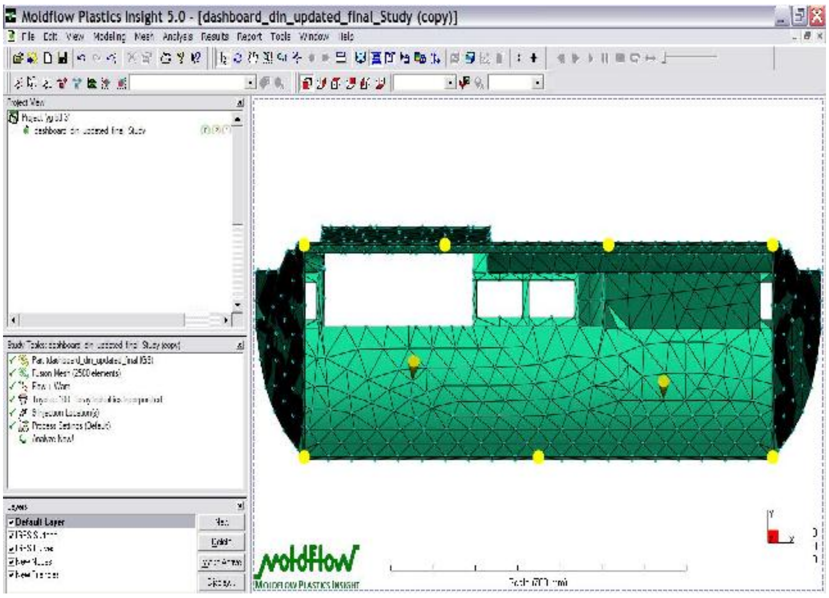


Figure 3.12: Gate location

f) Set runner and gate system

i. Set up in the small window below

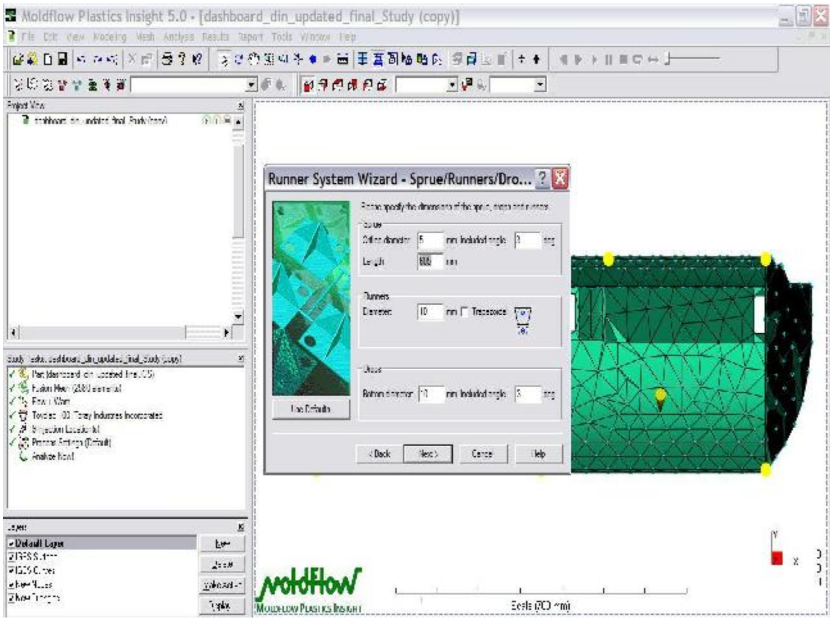


Figure 3.13: Runner and gate system to set up

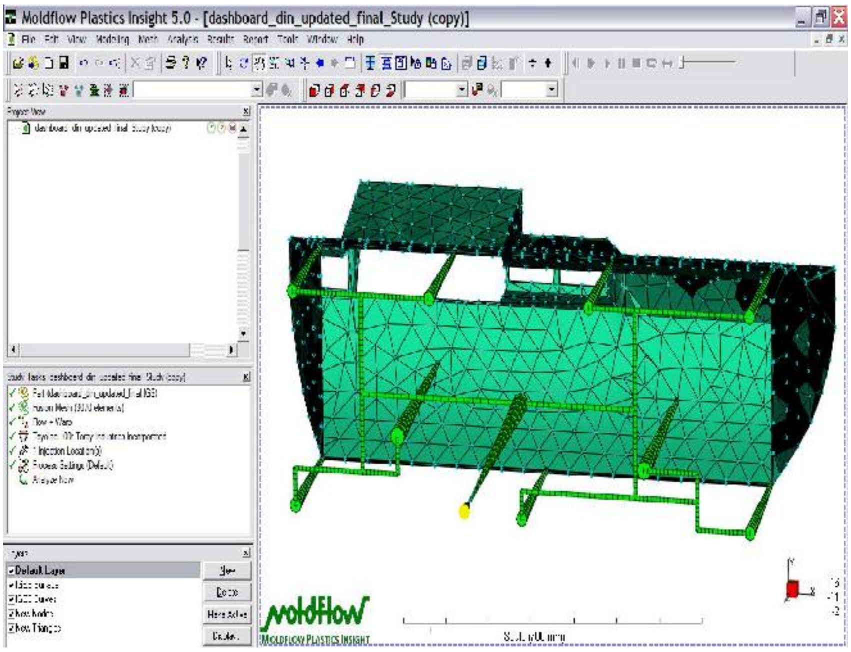


Figure 3.14: Runner and gate system be done

g) Set cooling system

i.Set cooling system wizard

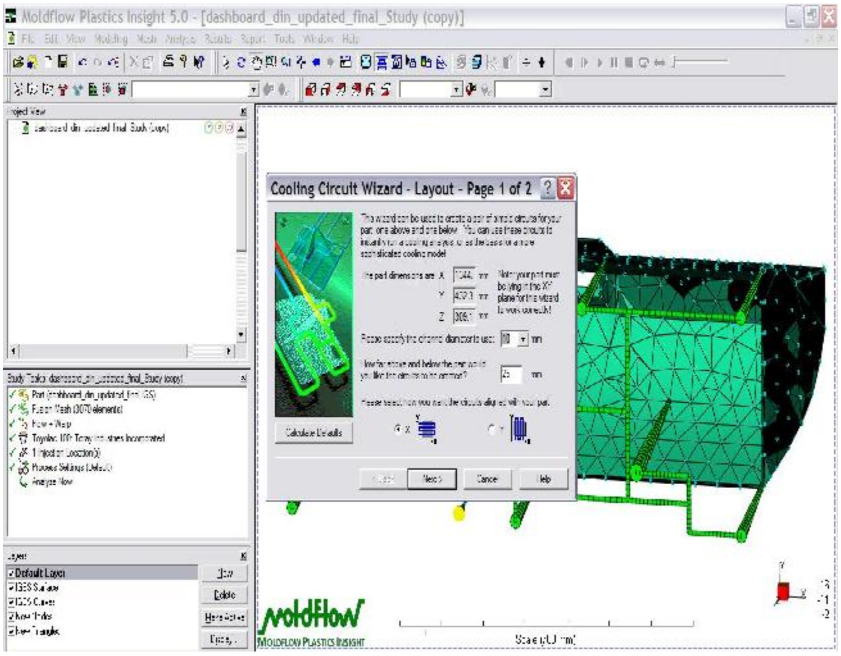


Figure 3.15: Cooling system type setup

ii.Cooling system was appeared

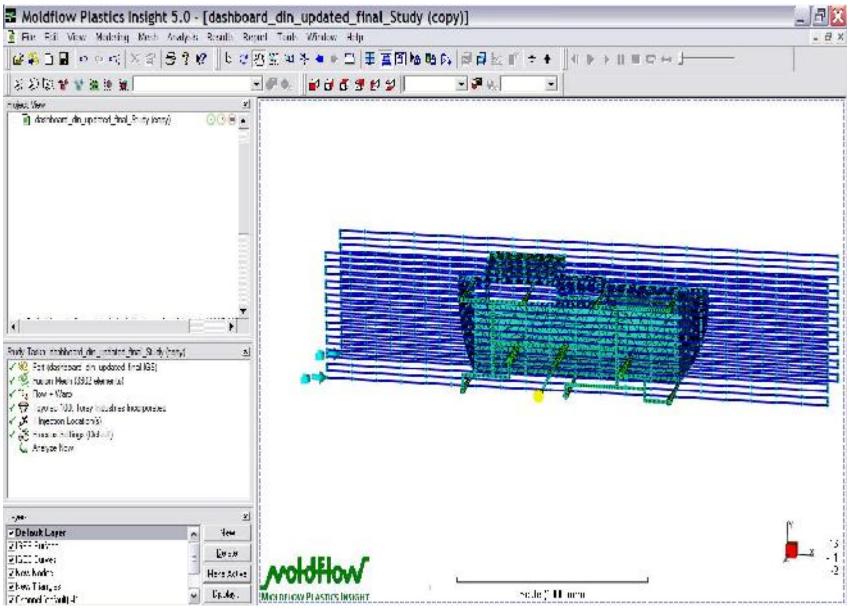


Figure 3.16: Complete modelling with cooling and runner system

h) Set process parameters

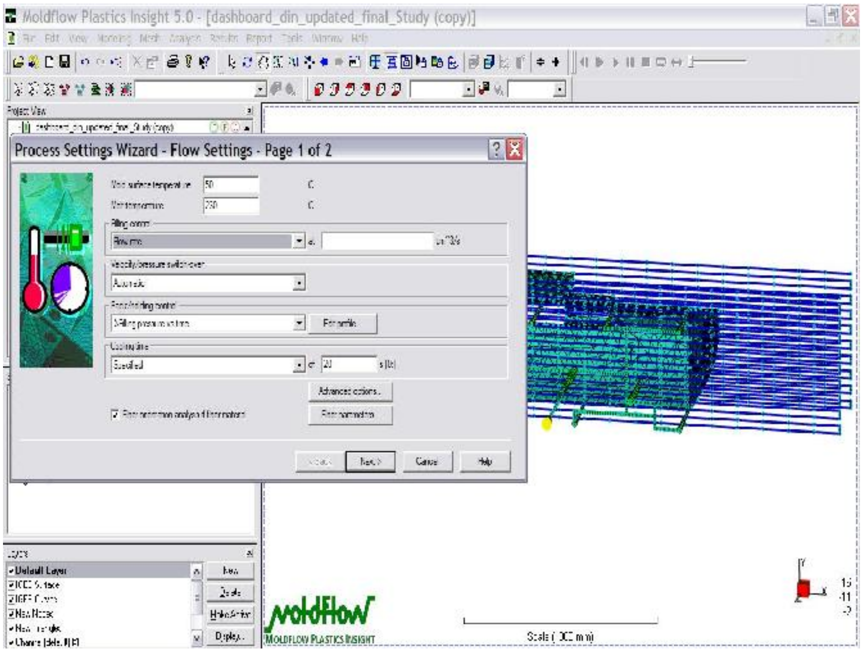


Figure 3.17: Process parameter setup

i) Perform the analysis

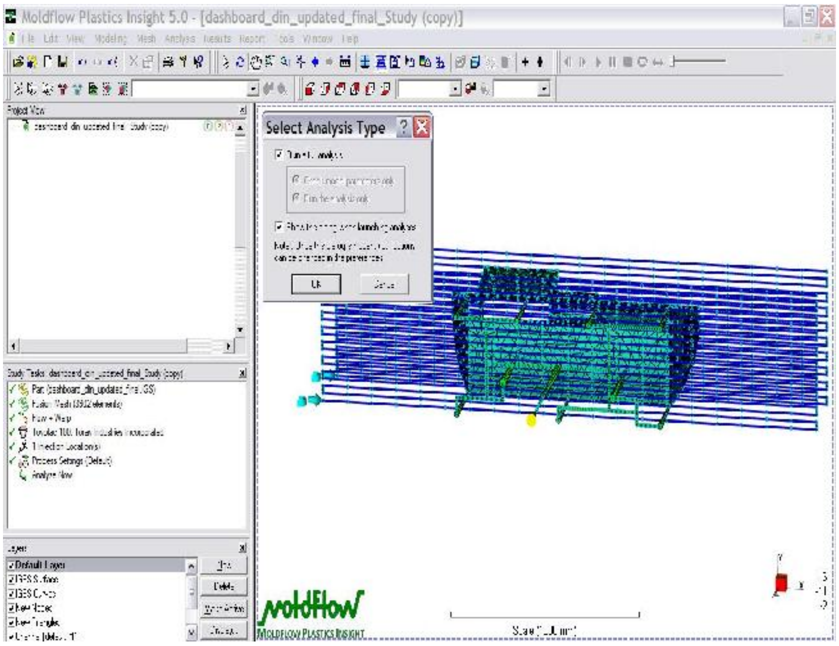


Figure 3.18: Perform the analysis

3.8 FLOW CHART

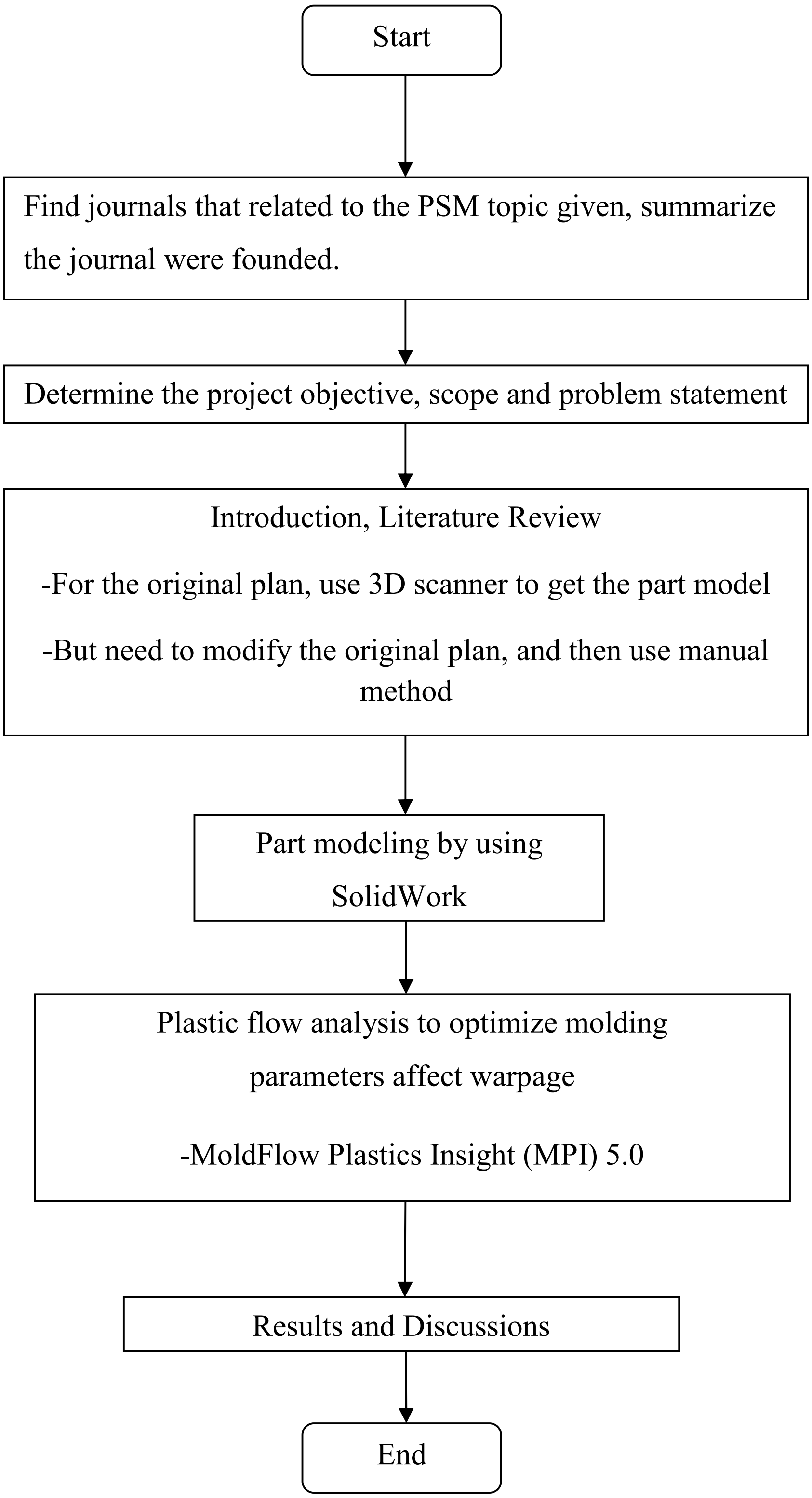


Figure 3.19: Flow chart

CHAPTER 4

RESULT AND DISCUSSION

4.1 INTRODUCTION

In this chapter, the results that have been taken from the Moldflow analysis had been compared. The comparison based on the air traps, weld lines, fill times, volumetric shrinkage and deflection. The results based on the parameter that are been chosen. These parameters are mold temperature, melt temperature, flow rate, and injection pressure. Most of the discussion based on the figure and data that have been collected.

4.2 DATA

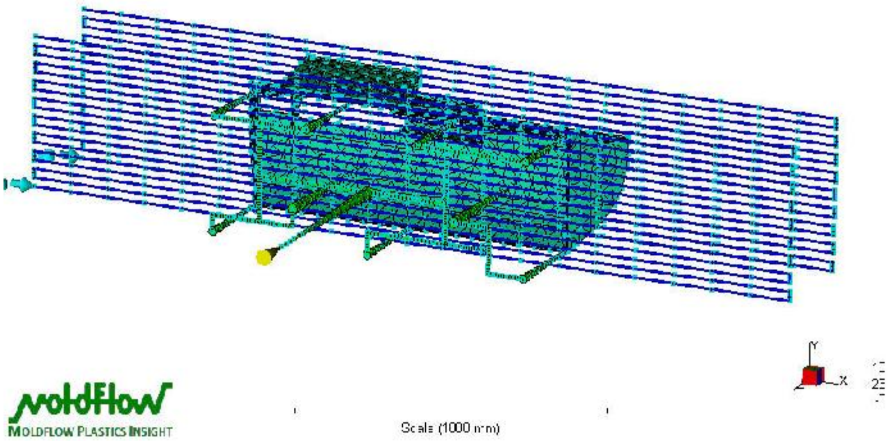
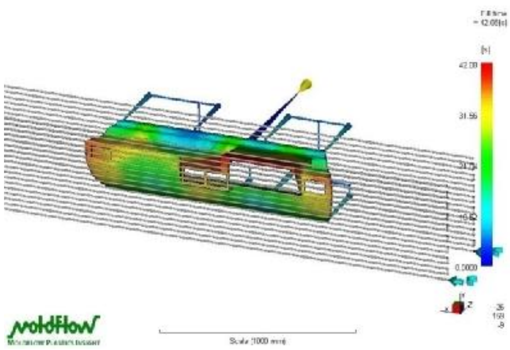


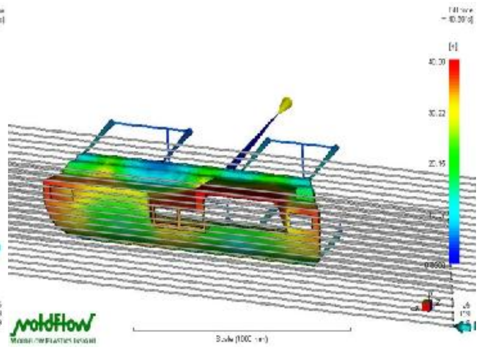
Figure 4.1: Car dashboard in MoldFlow Plastics Insight 5.0

Figure 4.1 shows the dashboard with the meshing entity, runner system and cooling system. The mesh is a web that consists of elements, with each element containing a node at every corner. The mesh provides the basis for a Moldflow analysis, where molding properties are calculated at every node. The cooling system use in this analysis of gate mechanism is circular type with 10.0mm in diameter. Pure water acts as the cooling with 10.0lit/min flow rate with constant time of cooling which is 20 second in each analysis. The sprue opening should be as small as possible but must fill the cavity effectively. The sprue used is circular type with 5mm orifice diameter included angle, 3 degree and 655 mm in length. The runner diameter is 10 mm and the drops will be 10 mm too. This model will analyzed in terms of different types of analysis which are injection pressure analysis, flow rate analysis, melt temperature analysis, and mold temperature analysis.

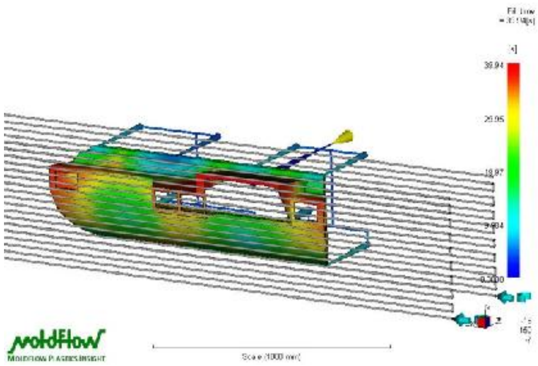
4.3 INJECTION PRESSURE ANALYSIS



a) 160 MPa



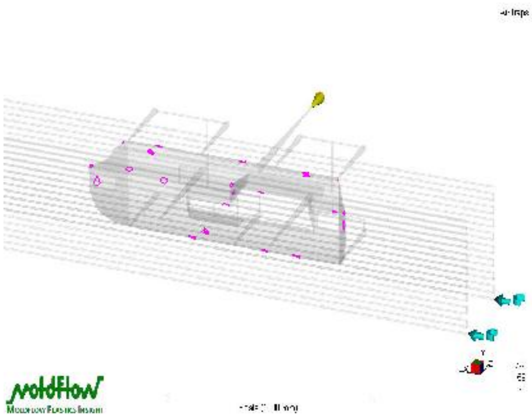
b) 170 MPa



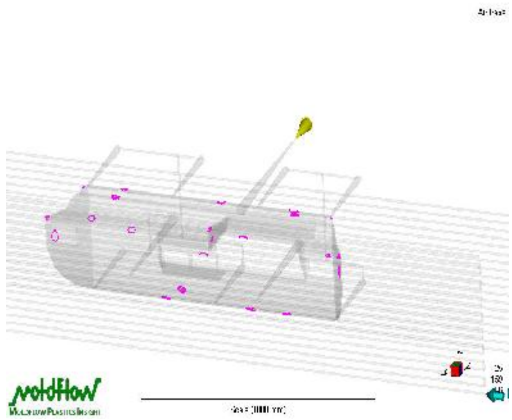
c) 180 MPa

Figure 4.2: Fill time for different injection pressure (a) 160 MPa (b) 170 MPa (c) 180 MPa

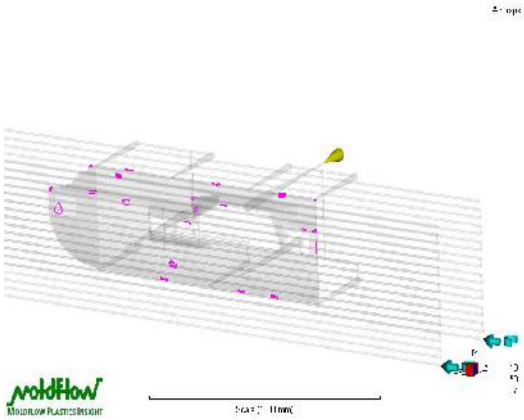
Figure 4.2 compare the fill time for the different injection pressure. The recommended pressure, 160 MPa gives 42.08s of filling time. By increasing the injection pressure to 170 MPa, the fill time that calculated by the software is 40.30s. When the injection pressure changes to 180 MPa, it resulted 39.94s of fill time.



a) 160 MPa



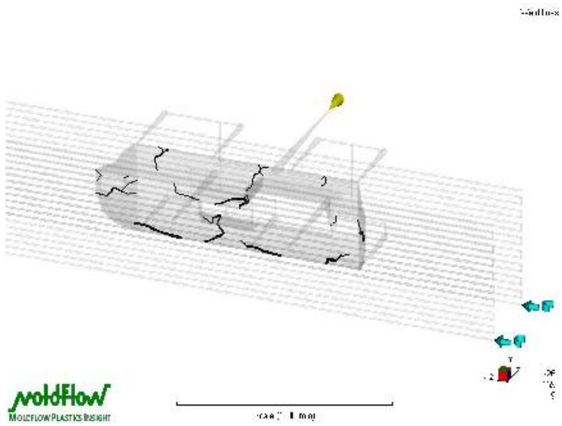
b) 170 MPa



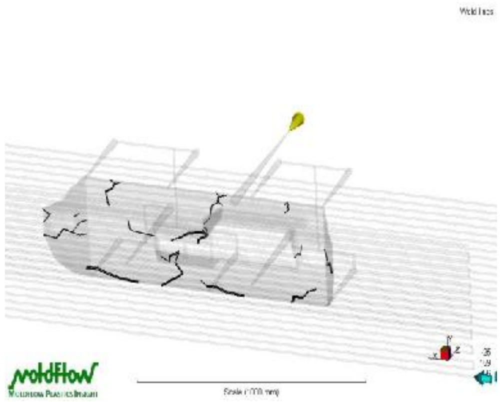
c) 180 MPa

Figure 4.3: Air traps for different injection pressure (a) 160 MPa (b) 170 MPa (c) 180 MPa

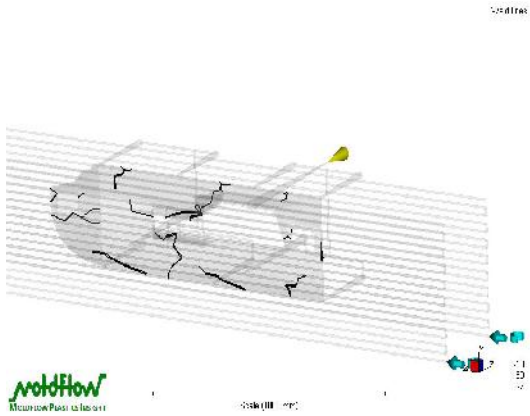
Figure 4.3 compared to each other to see the different in air traps that occur since the injection pressure different with each other. As a result, there is no differences in air traps that occur in the product with different injection pressure. The pattern of air traps same. The air traps represent by the pink color circle



a) 160 MPa



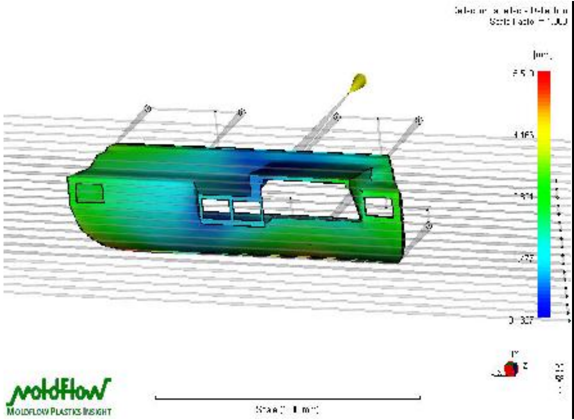
b) 170 MPa



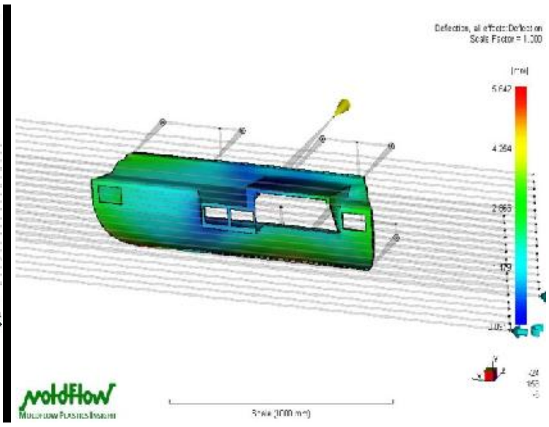
c) 180 MPa

Figure 4.4: Weld lines for different injection pressure (a) 160 MPa (b) 170 MPa (c) 180 MPa

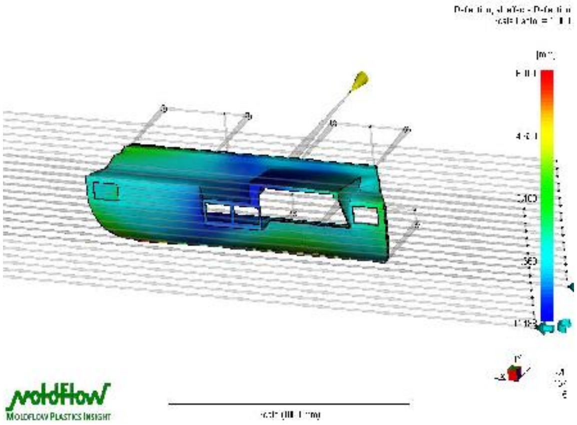
Based on the Figure 4.4, there is weld line occur after analyze the product. By comparing the pattern of weld line in Figure 4.3(a), (b), and (c), there are nothing different. The locations and also the numbers of the weld line are equal even though the injection pressure different.



a) 160 MPa



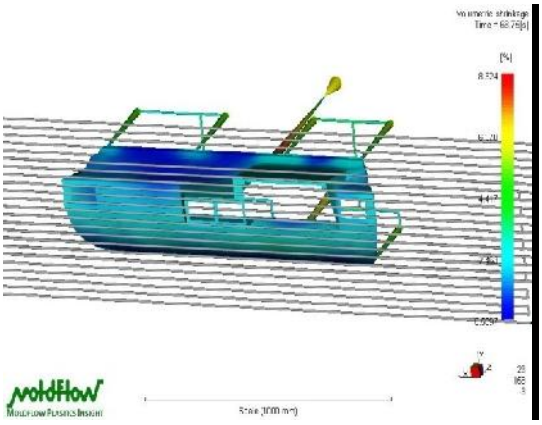
b) 170 MPa



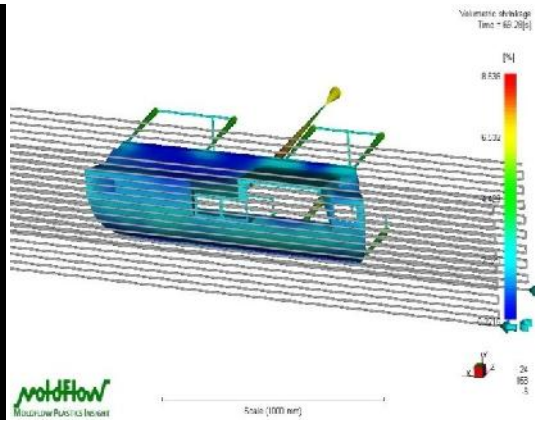
c) 180 MPa

Figure 4.5: Deflections for different injection pressure (a) 160 MPa (b) 170 MPa (c) 180 MPa

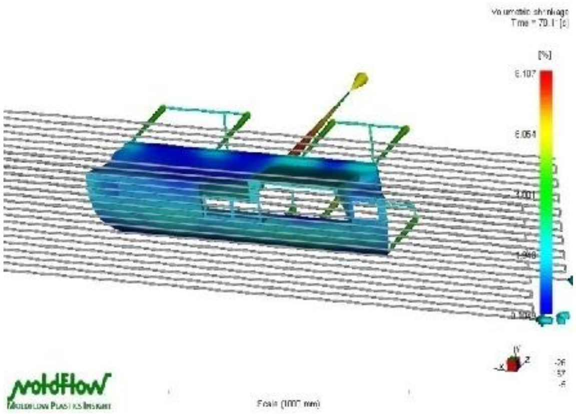
From the Figure 4.5 above, it show the deflection for the product with injection pressure 160 MPa, 170 MPa, and 180 MPa. The deflection for injection pressure 160 MPa is 5.510mm after start to deflect at 0.1327mm at x, y, and z axis. While the deflection for injection pressure 170 MPa, the deflection start at 0.0910mm and end at 5.642mm. With injection pressure of 180 MPa, the product starts to defect at 0.2188mm and end at 6.101mm. The area that marks with blue color is the area where the deflection in the lowest value. While the highest value deflection represented by the red color.



a) 160 MPa



b) 170 MPa



c) 180 MPa

Figure 4.6: Volumetric shrinkage for different injection pressure (a) 160 MPa (b) 170 MPa (c) 180 MPa

Figure 4.6 show the volumetric shrinkage for different injection pressure (a) 160 MPa (b) 170 MPa (c) 180 MPa. At injection pressure 160 MPa, the volumetric shrinkage is 8.324%. At injection pressure 170 MPa, 8.636%. For injection pressure 180 MPa, the volumetric shrinkage is 8.107%. The area that marks with blue color is the lowest area of sink index. While the biggest area sink represented by red color.

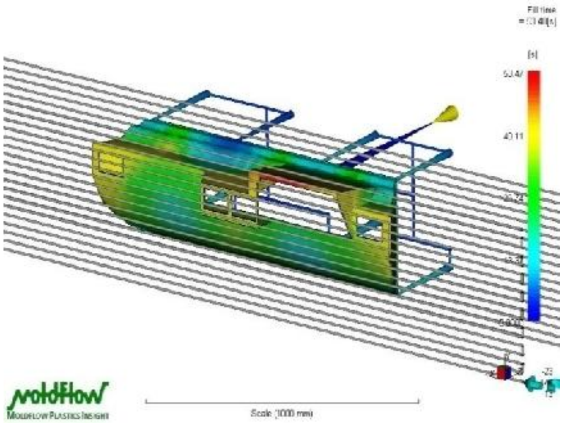
Table 4.1: Results summary for injection pressure analysis.

Mold Temperature (^oC)	Melt Temperature (^oC)	Flow Rate (cm^3/s)	Injection Pressure (MPa)	Maximum Warpage Displacement (mm)	Fill Time (s)	Volumetric Shrinkage (%)
40	230	200	140	5.469	48.71	8.704
40	230	200	150	5.480	45.29	8.678
40	230	200	160	5.510	42.08	8.636
40	230	200	170	5.642	40.30	8.324
40	230	200	180	6.101	39.94	8.107

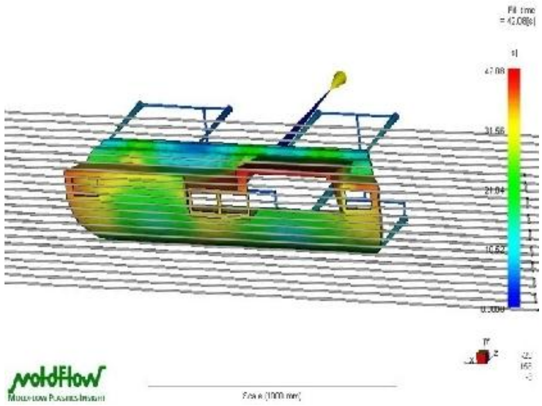
From the Table 4.1, the mold temperature, melt temperature and the flow rate are remained constant while the injection pressure is changed. At the injection pressure 180MPa gives the warpage displacement is 6.101 mm, the fill time is 39.94 s, and the volumetric shrinkage is 8.107 %. At the injection pressure 140MPa gives the warpage displacement is 5.469 mm, the fill time is 48.71 s, and the volumetric shrinkage is 8.704 %. When the injection pressure increase, the warpage displacement that calculated by the Moldflow software increase. The fill time and the volumetric shrinkage are decrease due to the increasing of the injection pressure.

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

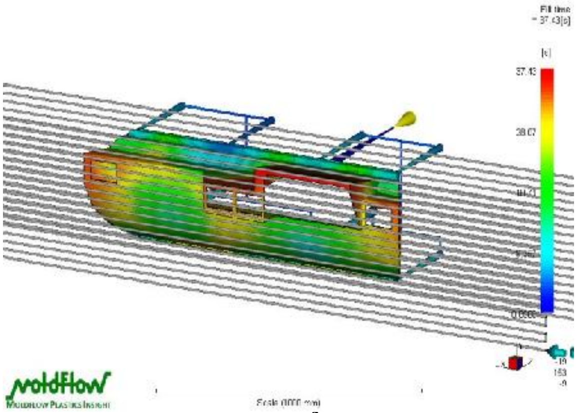
4.4 FLOW RATE ANALYSIS



a) 180 cm^3/s



b) 200 cm^3/s



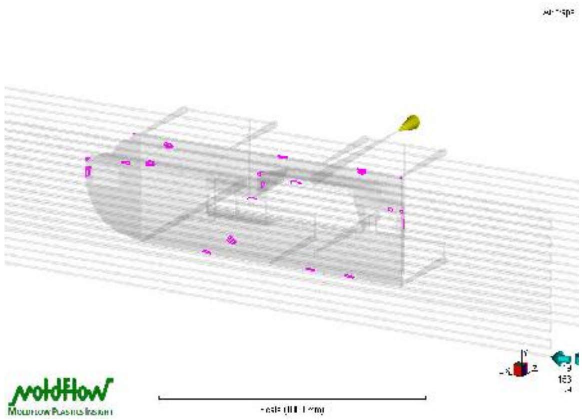
c) 220 cm^3/s

Figure 4.7: Fill time for different flow rate (a) 180 cm^3/s
(b) 200 cm^3/s (c) 220 cm^3/s

Figure 4.7 compare the fill time for the different flow rate. The recommended flow rate, 200 cm^3/s give 42.08s of filling time. By increasing the flow rate increased to the value of 220 cm^3/s, the fill time that calculated by the software is 37.43s. When the flow rate changes up to 180 cm^3/s, it resulted 53.47s of fill time.

a) 180 cm^3/s

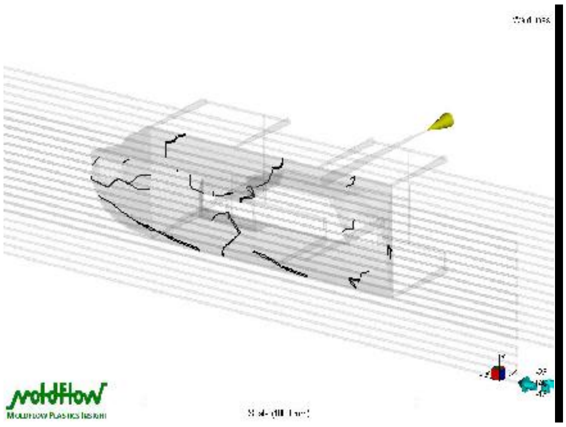
b) 200 cm^3/s



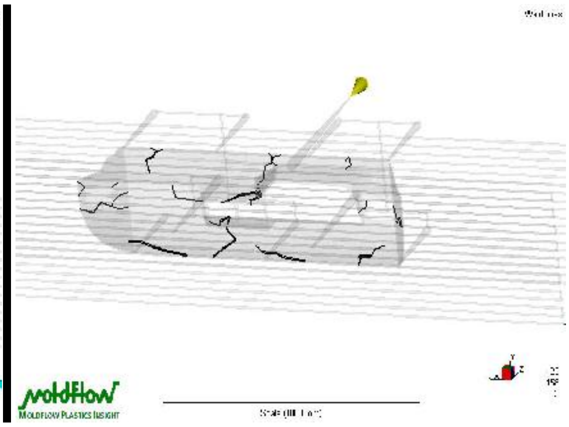
c) 220 cm^3/s

Figure 4.8: Air traps for different flow rate (a) 180 cm^3/s (b) 200 cm^3/s (c) 220 cm^3/s

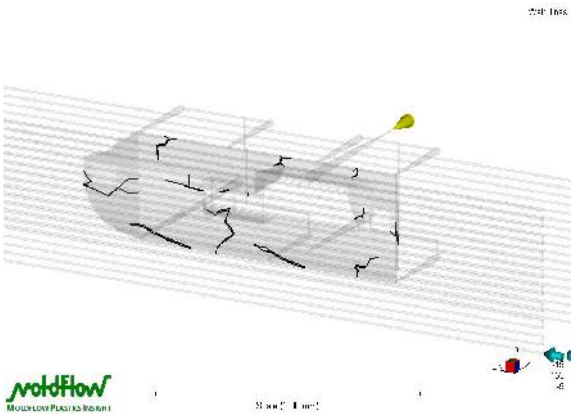
Figure 4.8 compared to each other to see the different in air traps that occur since the flow rate different with each other. As a result, there's different in air traps that occur in the product with different flow rate. But the pattern of air traps is nearly the same.



a) 180 cm^3/s



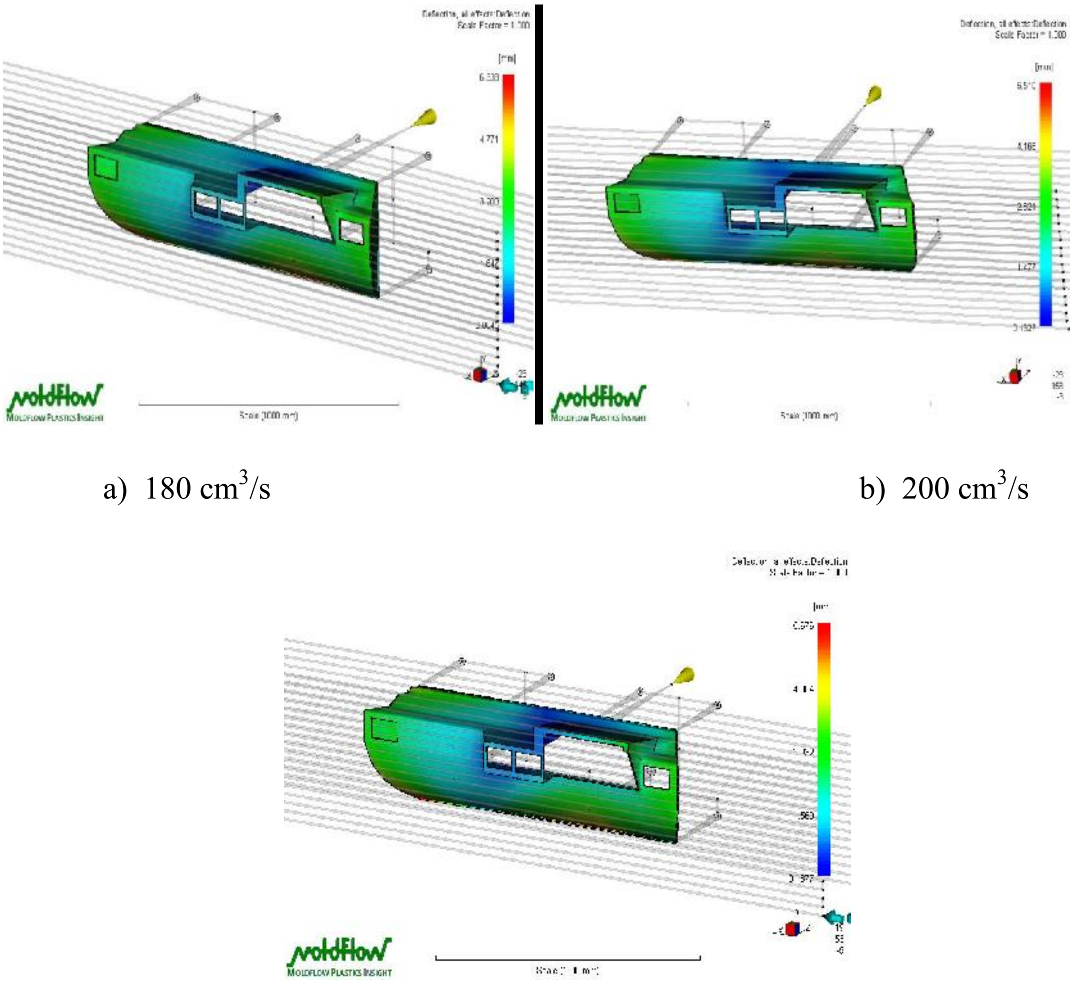
b) 200 cm^3/s



c) 220 cm^3/s

Figure 4.9: Weld line for different flow rate (a) 180 cm^3/s (b) 200 cm^3/s (c) 220 cm^3/s

Figure 4.9 compared to each other to see the different in weld line that occur since the flow rate different with each other. As a result, there's different in weld line defect that occur in the product with different flow rate.

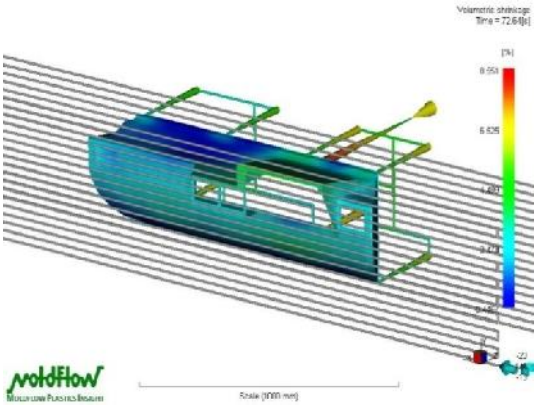


a) 180 cm^3/s b) 200 cm^3/s

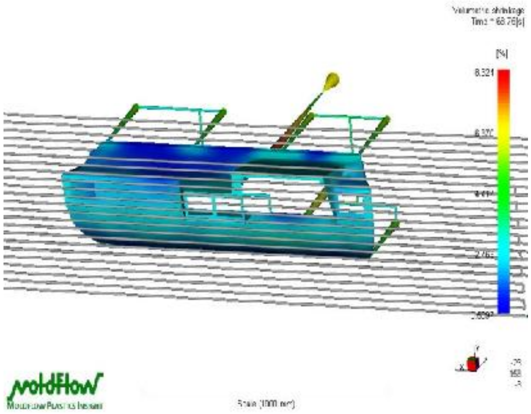
c)220 cm^3/s

Figure 4.10: Deflection for different flow rate (a) 180 cm^3/s (b) 200 cm^3/s (c) 220 cm^3/s

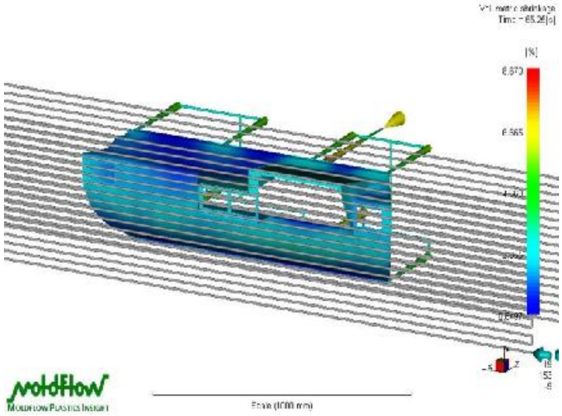
From the Figure 4.10 above, it show the deflection for the product with flow rate 180 cm^3/s, 200 cm^3/s, and 220 cm^3/s. The deflection for flow rate 180 cm^3/s is 6.333mm after start to deflect at 0.0845mm at x, y, and z axis. While the deflection for flow rate 200 cm^3/s, deflection start at 0.1327mm and end at 5.510mm. For product with flow rate 220 cm^3/s, its starts to deflect at 0.1877mm and end at 5.676mm. The area that marks with blue color is the area where the deflection in the lowest value. While the highest value deflection represented by the red color.



a) 180 cm^3/s



b) 200 cm^3/s



c)220 cm^3/s

Figure 4.11: Volumetric shrinkage for different flow rate (a) 180 cm^3/s (b) 200 cm^3/s (c) 220 cm^3/s

Figure 4.11 show the volumetric shrinkage for different flow rate as (a) 180 cm^3/s (b) 200 cm^3/s (c) 220 cm^3/s. At flow rate 180 cm^3/s, the volumetric shrinkage is 8.551%. At flow rate 200 cm^3/s, 8.324%. For flow rate 220 cm^3/s, the volumetric shrinkage is 8.670%.

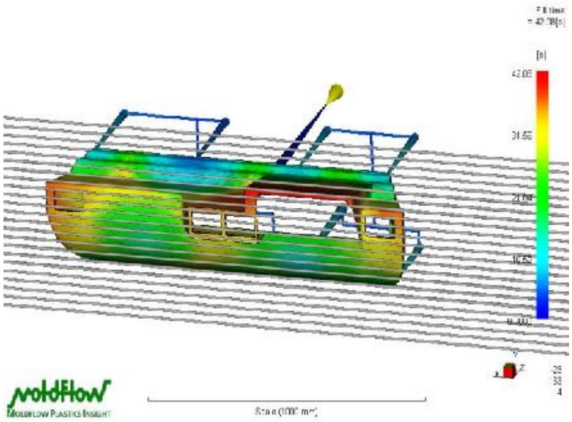
Table 4.2: Results summary for the flow rate analysis

Mold Temperature (^oC)	Melt Temperature (^oC)	Flow Rate (cm^3/s)	Injection Pressure (MPa)	Maximum Warpage Displacement (mm)	Fill Time (s)	Volumetric Shrinkage (%)
40	230	180	160	6.333	48.03	7.905
40	230	190	160	5.694	45.76	8.103
40	230	200	160	5.510	42.08	8.324
40	230	210	160	5.479	39.97	8.490
40	230	220	160	5.476	37.43	8.670

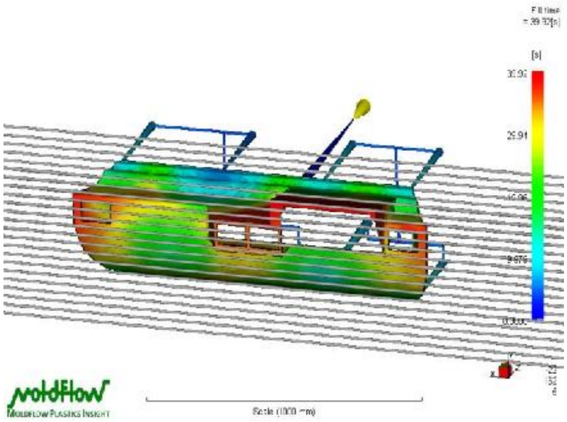
Form the Table 4.2, the mold temperature, melt temperature, and injection pressure are constant but changed in the flow rate. Different flow rate of filling control gives different warpage displacement. The flow rate 220 cm^3/s gives the smaller warpage displacement with 5.476 mm, the fill time is 37.43 s, and volumetric shrinkage is 8.670 %. The flow rate 180 cm^3/s gives the higher warpage displacement with 6.333 mm, the fill time is 48.03 s, and volumetric shrinkage is 7.905 %. The warpage displacement increases with the increasing of the flow rate.

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

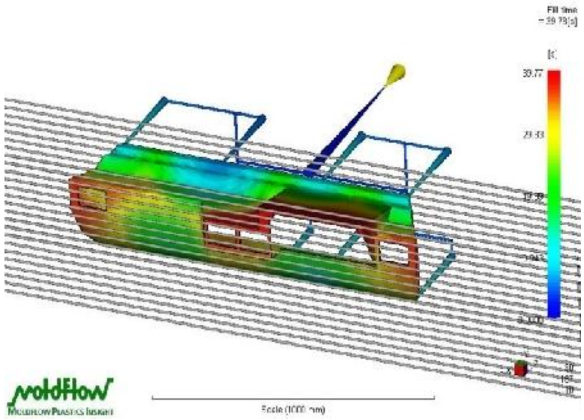
4.5 MELT TEMPERATURE ANALYSIS



a) 230^{o}C



b) 240^{o}C

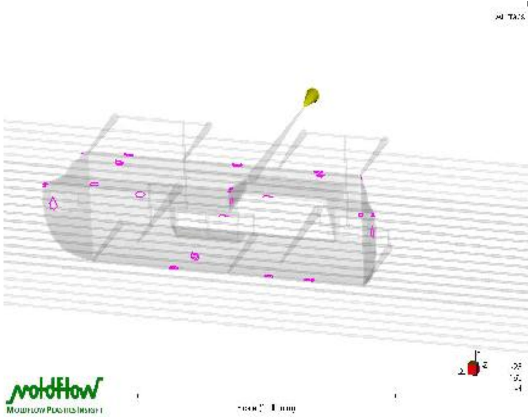


c) 250^{o}C

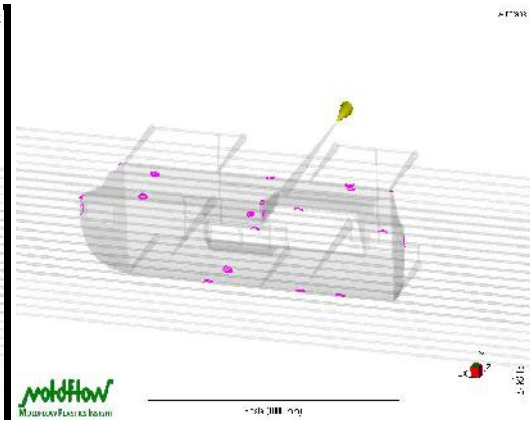
Figure 4.12: Fill time for different melt temperature (a) 230^{o}C (b) 240^{o}C (c) 250^{o}C

Figure 4.12 compare the fill time for the different melt temperature. The recommended melt temperature, 240^{o}C give 39.92s of filling time. The melt temperature increase to 250^{o}C, the fill time that calculated by the software is 39.77s. When the melt temperature decreases to 230^{o}C, the fill time is 42.08s.

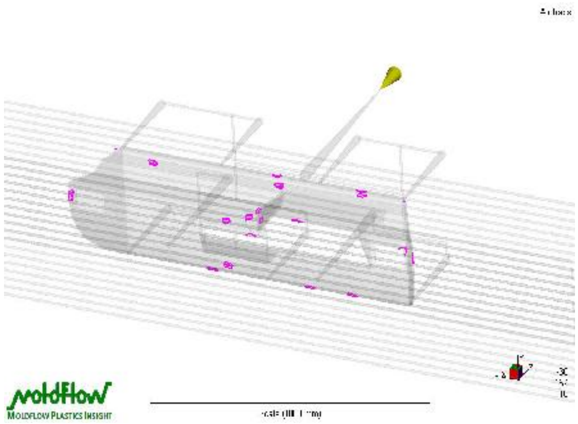
From this part, as a conclusion, the fill time decrease with the increase of the melt temperature.



a) 230°C



b) 240°C

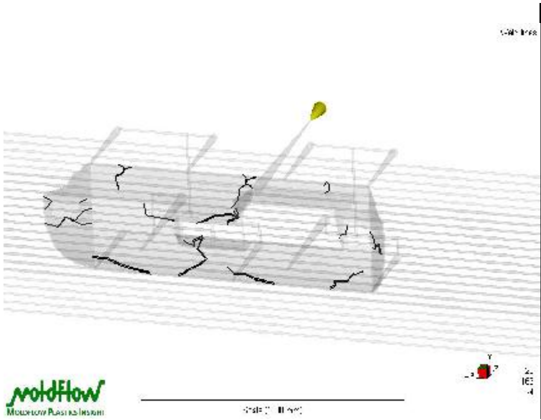


c) 250°C

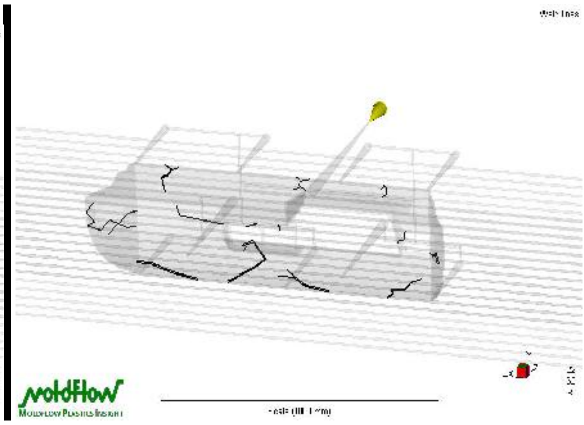
Figure 4.13: Air trap for different melt temperature (a) 230°C (b) 240°C (c) 250°C

Figure 4.13 compared to each other to see the different in air traps that occur since the melt temperature different with each other. From the Figure 4.12, the air trap in the melt temperature 230°C is bigger and air trap in the melt temperature 250°C is the smallest.

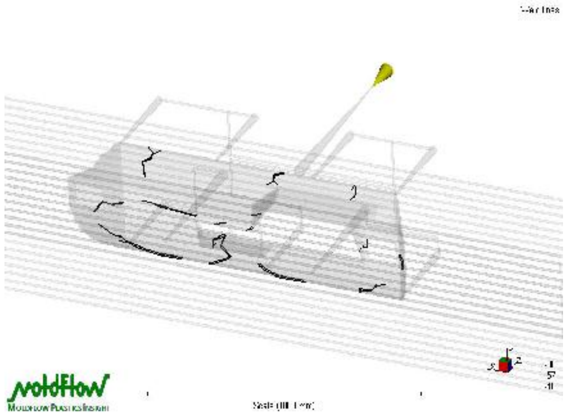
As a result, as the melt temperature increases the air trap in the dashboard become smaller. It is because the time to fulfill the dashboard is getting faster with the increasing of melt temperature.



a) 230^{o}C



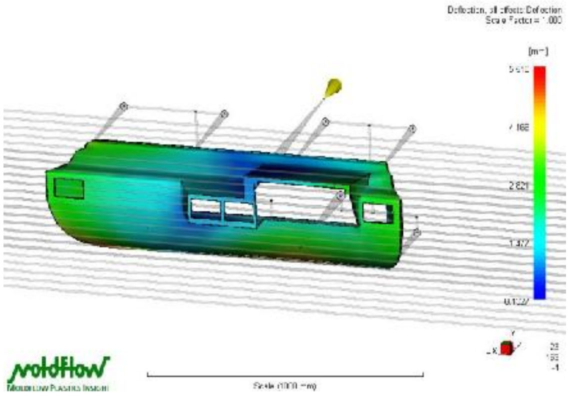
b) 240^{o}C



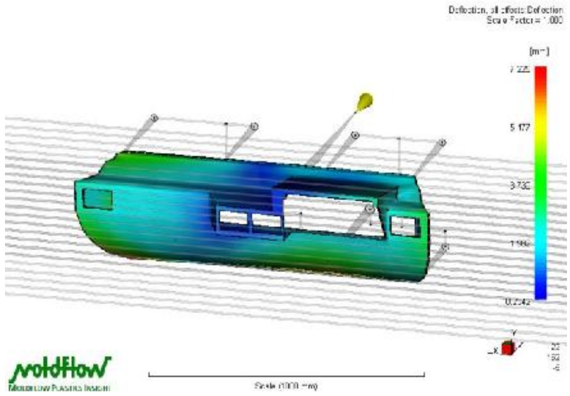
c) 250^{o}C

Figure 4.14: Weld line for different melt temperature (a) 230^{o}C (b) 240^{o}C (c) 250^{o}C

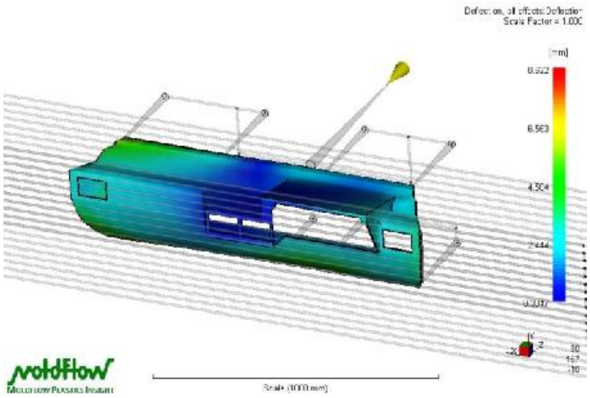
Figure 4.14 compared to each other to see the different in weld line that occur since the melt temperature different with each other. As a result, there's different in weld line that occurs in the product with different melt temperature. There are many weld line occur in the dashboard in the melt temperature 230^{o}C compare to others. The reason is when the melt temperature is lower, the plastic that want to inject is not melt enough and when it meet in mold they cannot weld very well.



a) 230^{o}C



b) 240^{o}C

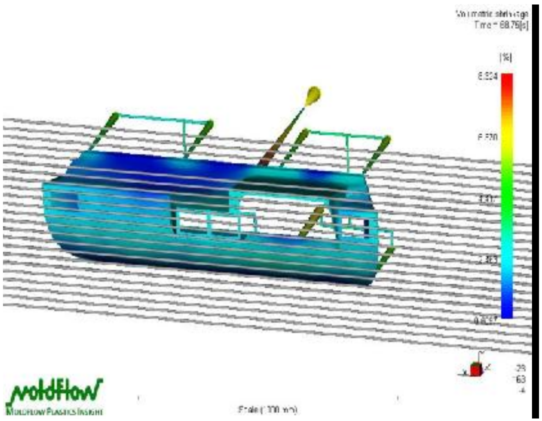


c) 250^{o}C

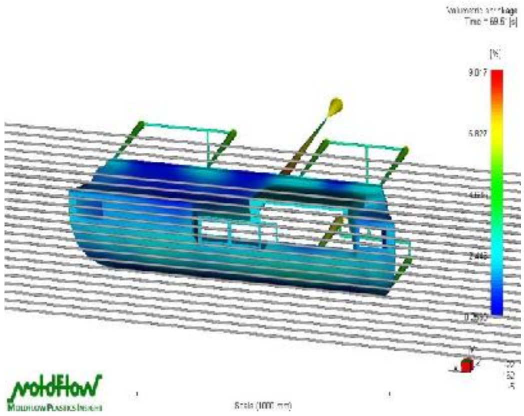
Figure 4.15: Deflection for different melt temperature (a) 230^{o}C (b) 240^{o}C (c) 250^{o}C

From the Figure 4.9 above, it show deflection for different melt temperature 230^{o}C, 240^{o}C, and 250^{o}C. The deflection for melt temperature 230^{o}C is 5.510mm after start to deflect at 0.1327mm at x, y, and z axis. While the deflection for melt temperature 240^{o}C, the deflection start at 0.2342mm and end at 7.225mm. With melt temperature of 250^{o}C, its starts to deflect at 0.3847mm and end at 8.622mm. The area that marks with blue color is the area where the deflection in the lowest value. While the highest value deflection represented by the red color.

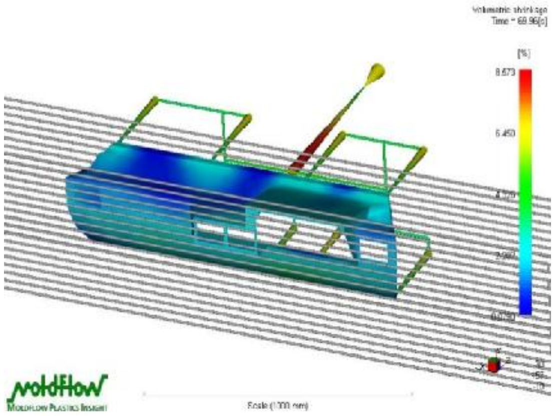
As a conclusion, the melt temperature is proportional to the value of deflection. As the melt temperature increase, the value of the deflection also increases.



a) 230°C



b) 240°C



c) 250°C

Figure 4.16: Volumetric shrinkage for different melt temperature
(a) 230°C (b) 240°C (c) 250°C

Figure 4.16 show the volumetric shrinkage for different melt temperature (a) 230°C (b) 240°C (c) 250°C. At melt temperature 230°C, the volumetric shrinkage is 8.324%. At melt temperature 240°C, the volumetric shrinkage is 9.017%. For melt temperature 250°C, the volumetric shrinkage is 8.573%.

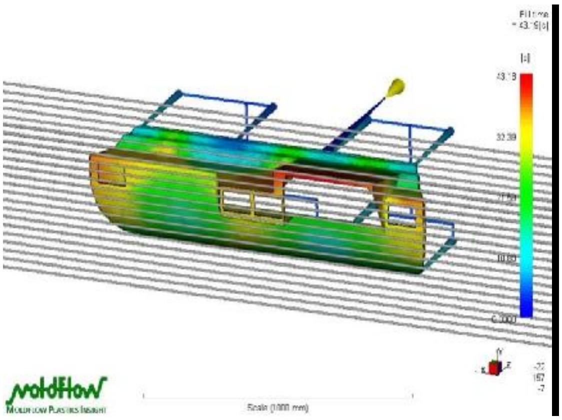
Table 4.3: Results summary for melt temperature analysis

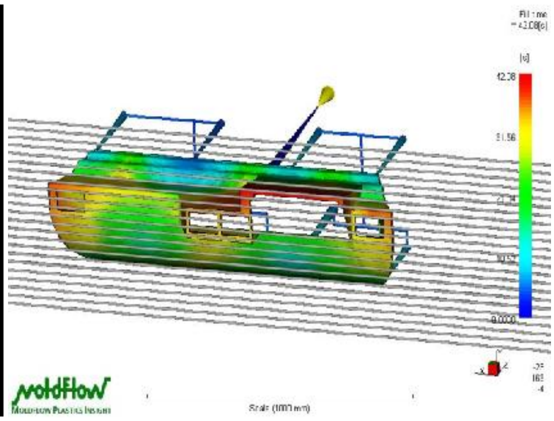
Mold Temperature (^oC)	Melt Temperature (^oC)	Flow Rate (cm^3/s)	Injection Pressure (MPa)	Max Warpage Displacement (mm)	Fill Time (s)	Volumetric Shrinkage (%)
40	210	200	160	4.304	48.96	7.843
40	220	200	160	4.928	45.98	8.097
40	230	200	160	5.510	42.08	8.324
40	240	200	160	7.225	39.92	8.573
40	250	200	160	8.622	39.77	9.017

From Table 4.3, the mold temperature, flow rate, and injection pressure are constant while the melt temperature is changed. The melt temperature also effects the warpage displacement of the dashboard. At melt temperature 210^oC, the warpage displacement is 4.304 mm, the fill time is 48.96 s, and the volumetric shrinkage is 7.843 %. At the higher melt temperature 250^oC, the warpage displacement is 8.622 mm, the fill time is 39.77 s, and the volumetric shrinkage is 9.017 %. The filling of the dashboard become faster when the melt temperature increase. The warpage displacements also increase. It is because the time that the dashboard to freeze become longer when the increasing of the melt temperature.

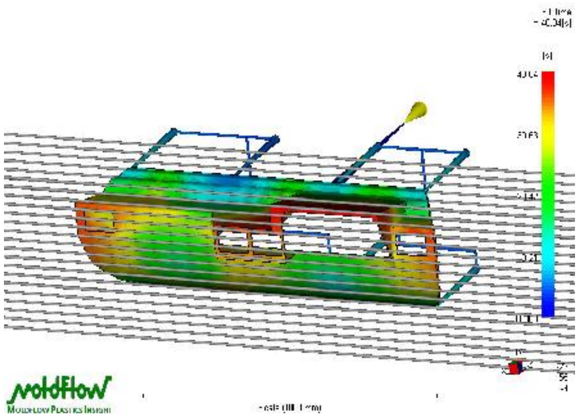
The Volumetric shrinkage result can be used to get shrinkage percent on the product. High shrinkage values could indicate voids inside the part. Volumetric shrinkage should be uniform across the whole part to reduce the defects, and it should be less than the recommended maximum value for the material.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

4.6 MOLD TEMPERATURE ANALYSIS





a) 230^{o}C b) 240^{o}C

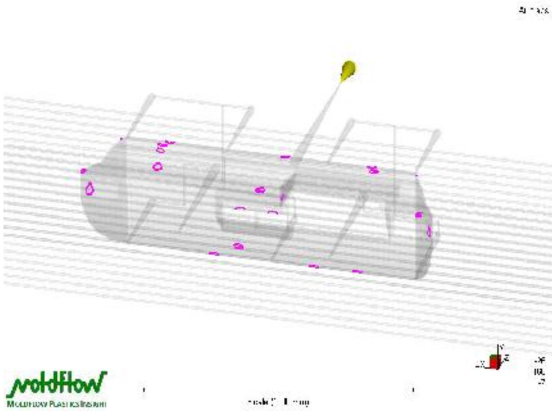


c) 60^{o}C

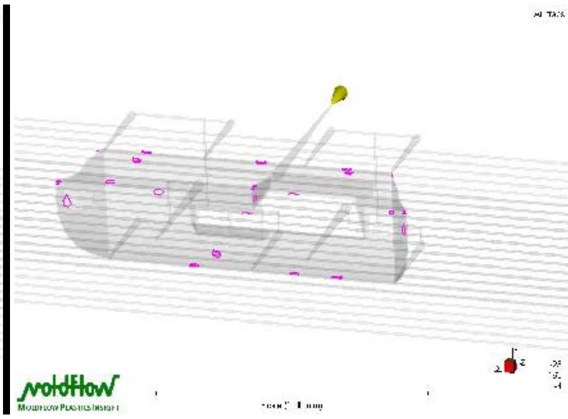
Figure 4.17: Fill time for different mold temperature (a) 20^{o}C (b) 40^{o}C (c) 60^{o}C

Figure 4.17 compare the fill time for the different mold temperature. The recommended mold temperature, 40^{o}C give 42.08s of filling time. The mold temperature increase to 60^{o}C, the fill time that calculated by the software is 40.84s. When the mold temperature decreases to 20^{o}C, the fill time is 43.18s.

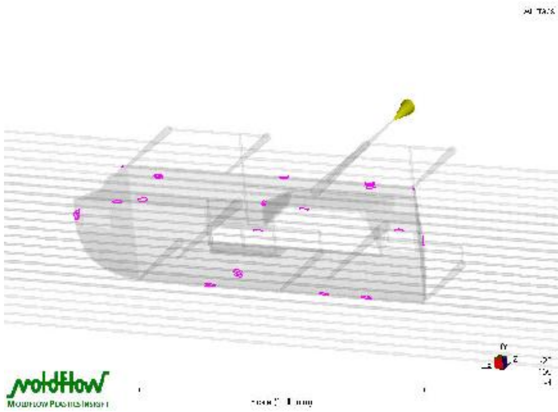
As a conclusion, the fill time decrease with the increase of the mold temperature.



a) 20°C



b) 40°C

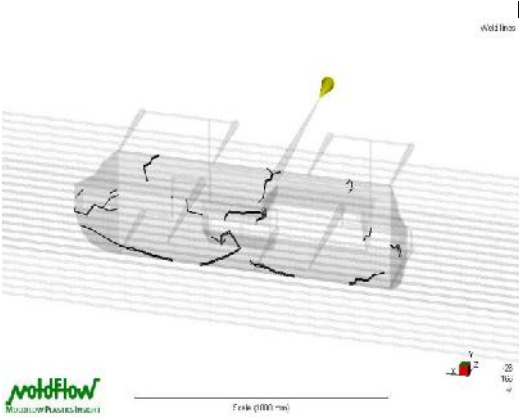


c) 60°C

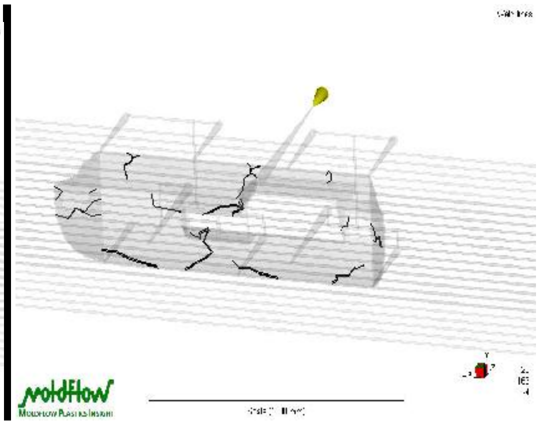
Figure 4.18: Air trap for different mold temperature
(a) 20°C (b) 40°C (c) 60°C

Figure 4.18 compared to each other to see the different in air traps that occur since the mold temperature different with each other. From the Figure 4.17, the air trap in the mold temperature 230°C is bigger and air trap in the mold temperature 250°C is the smallest.

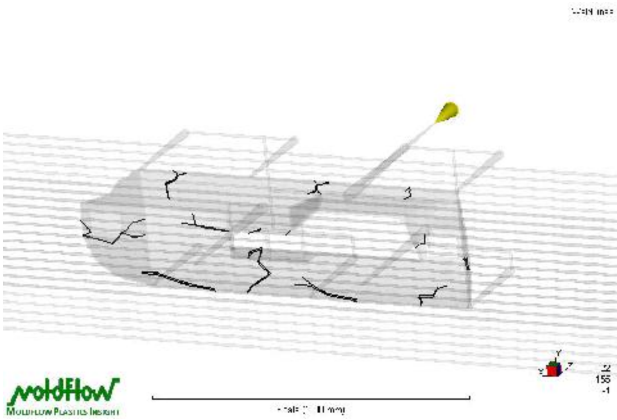
As a result, as the mold temperature increases the air trap in the dashboard become smaller.



a) 20°C



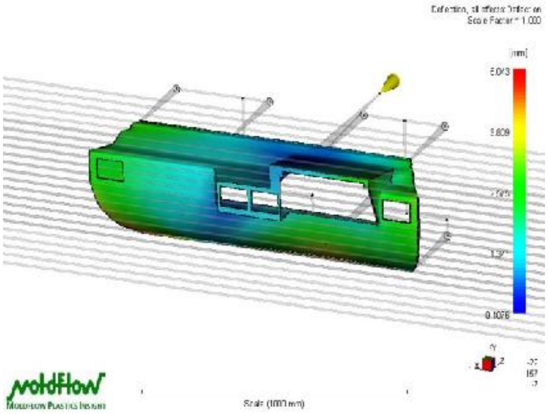
b) 40°C



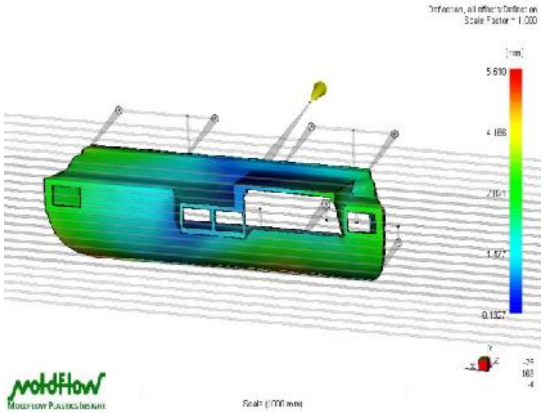
c) 60°C

Figure 4.19: Weld line for different mold temperature
(a) 20°C (b) 40°C (c) 60°C

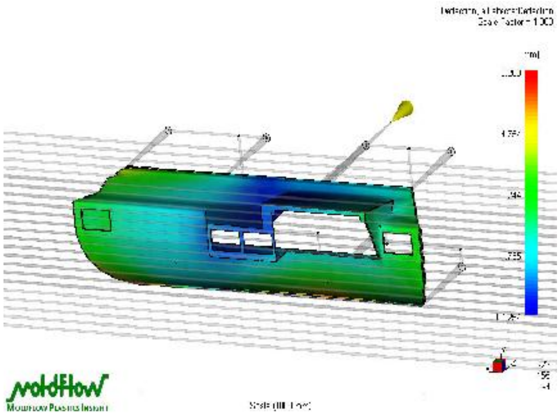
Figure 4.19 compared to each other to see the different in weld line that occur since the mold temperature different with each other. As a result, there's different in weld line that occurs in the product with different mold temperature. There are many weld line occur in the dashboard in the mold temperature 20°C compare to others. There is a few weld line occur in the mold temperature 60°C.



a) 20°C



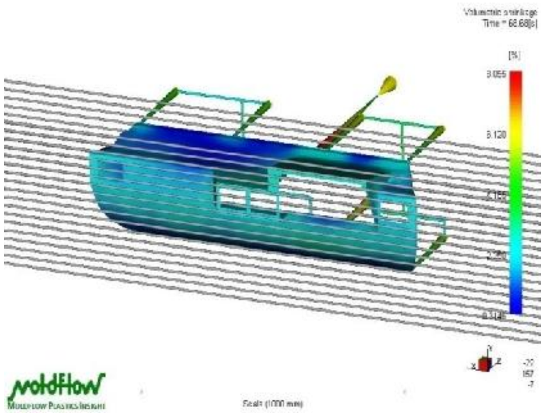
b) 40°C



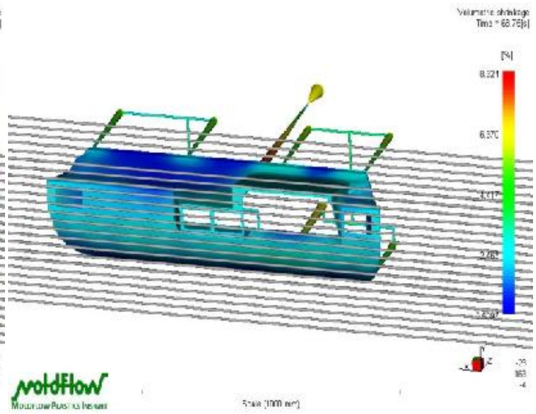
c) 60°C

Figure 4.20: Deflection for different mold temperature
(a) 20°C (b) 40°C (c) 60°C

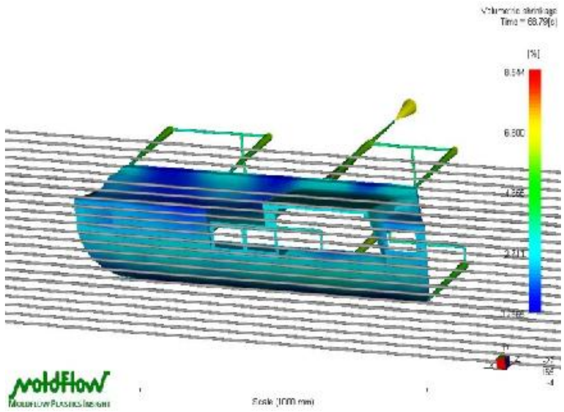
From the Figure 4.20 above, it show deflection for different mold temperature (a) 20°C (b) 40°C (c) 60°C. The deflection for melt temperature 20°C is 5.043mm after start to deflect at 0.1076mm at x, y, and z axis. While the deflection for mold temperature 40°C, the deflection start at 0.1327mm and end at 5.510mm. With mold temperature of 60°C, its starts to deflects at 0.2254mm and end at 6.263mm. The area that marks with blue color is the area where the deflection in the lowest value. While the highest value deflection represented by the red color.



a) 20^oC



b) 40^oC



c) 60^oC

Figure 4.21: Volumetric shrinkage for different mold temperature
(a) 20^oC (b) 40^oC (c) 60^oC

Figure 4.21 show the volumetric shrinkage for different mold temperature (a) 20^oC (b) 40^oC (c) 60^oC. At mold temperature 20^oC, the volumetric shrinkage is 8.055%. At mold temperature 40^oC, the volumetric shrinkage is 8.324%. For melt temperature 60^oC, the volumetric shrinkage is 8.544%.

As a conclusion, the mold temperature is proportional to the volumetric shrinkage of the dashboard. When mold temperature increase, the volumetric shrinkage also increase.

Table 4.4: Results summary for mold temperature analysis

Mold Temperature (^oC)	Melt Temperature (^oC)	Flow Rate (cm^3/s)	Injection Pressure (MPa)	Maximum Warpage Displacement (mm)	Fill Time (s)	Volumetric Shrinkage (%)
20	230	200	160	5.043	43.18	8.055
30	230	200	160	5.347	42.87	8.204
40	230	200	160	5.510	42.08	8.324
50	230	200	160	5.998	41.70	8.536
60	230	200	160	6.263	40.84	8.544

From Table 4.4, the melt temperature, the flow rate, and the injection pressure are remain constant but changed at the mold temperature. The mold temperatures also effect the warpage displacement. At mold temperature 20^oC, the warpage displacement is 5.043 mm, the fill time is 43.18 s, and the volumetric shrinkage is 8.055 %. At the higher melt temperature 60^oC, the warpage displacement is 6.263 mm, the fill time is 40.84 s, and the volumetric shrinkage is 8.544 %. When the mold temperature increases, the warpage displacement also increased. Warpage displacement proportional to the increasing of the mold temperature.

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

CHAPTER 5

CONCLUSION AND RECOMMENDATION

5.1 CONCLUSION

The main objective of this project to study the parameters effect in injection molding to car dashboard**,** analyze the effect of molding parameters such as melt temperature, injection pressure, and mold temperature to warpage of car dashboard, and optimize the molding parameters by using plastic flow simulation software in order to get a good result. Other than that, the analysis also includes the air traps, weld line, deflection, volumetric shrinkage, and fill time.This analysis meet the objectives. To sum up ;

1. The warpage displacement increase due to the increasing of the injection pressure.
2. The warpage displacement decrease when the increasing of the flow rate.
3. The warpage displacements increase when the increasing of the melt temperature.
4. The warpage displacement increase when the increasing of the mold temperature

By using the Moldflow software, the best and optimum parameters for the product have been obtained. In this study, the parameters that have been obtained are the mold temperature (40^{o}C), melt temperature (230^{o}C), flow rate (200 cm^3/s), and injection pressure (160 MPa).

This study shows that these parameters effect the warpage displacement. The changes of these parameters give the big effect to the product depending on the parameter that has changed. In the chapter 4, for every changing the parameter has shown in the results summary table.

The product design is also important in order to avoid defects such as air traps and volumetric shrinkage. The locations of the gate also have to be reconsidered. This type of analysis is used to modify the design before move to the production line. It can reduce the cost production because of the effect of the warpage. It shows the potential defect that will occur in the product. From there, the designer can repair the design in order to reduce the defects.

As a conclusion, these parameters can affect the process in making the product. The values of parameters depend on the design of the product which means that the different design will give the different values of these parameters. By using Moldflow software, the best and optimum parameter can be obtained for the product.

5.2 RECOMMENDATION

The parameters need to be obtained in order to avoid the cavity did not fill enough with material because these parameters do reach the optimum and needed. But by using the Moldflow can help to get the best parameters.

In real life, the product design need to analyze before fabricate in order to get the best design and to obtain the suitable parameter for the injection molding machine because to avoid higher cost and time production. The study by using Moldflow software can be expand in analyze design product before fabricate the mold.

REFERENCES

[1] http://www.scudc.scu.edu/cmdoc/dg_doc/develop/process/physics/b3500001

[2] Ozcelik and Erzurumlu, 2006 B. Ozcelik and T. Erzurumlu, Comparison of the warpage optimization in the plastic injection molding using ANOVA, neural network model and genetic algorithm, *Journal of Materials Processing Technology* 171 (3) (2006), pp. 437–445

[3] Huang and Tai, 2001 M.-C. Huang and C.-C. Tai, The effective factors in the warpage problem of an injection-molded part with a thin shell feature, *Journal of Materials Processing Technology* 110 (2001), pp. 1–9.

[4] Kurtaran et al., 2005 H. Kurtaran, B. Ozcelik and T. Erzurumlu, Warpage optimization of a bus ceiling lamp base using neural network model and genetic algorithm, *Journal of Materials Processing Technology* 169 (2) (2005), pp. 314–319

[5] Fausto Bernardini, Holly E. Rushmeier: The 3D Model Acquisition Pipeline. Comput. Graph. Forum 21(2): 149-172 (2002)

[6] Song Zhang, Peisen Huang, "High-resolution, real-time 3-D shape measurement," Optical Engineering, 2006, pp.123601

[7] Gabriele Guidi, Laura Micoli, Michele Russo, Bernard Frischer, Monica De Simone, Alessandro Spinetti, Luca Carosso, "3D digitization of a large model of imperial Rome," Fifth International Conference on 3-D Digital Imaging and Modeling, 2005, pp.565-572.

[8] http://www.solidworks.com/sw/why-solidworks/mechanical-engineering-software-designers.htm

[9] http://www.solidworks.com/sw/support/PDMSystemRequirements.html

[10] Kirkpatrick, S., Gelatt Jr., C.D., Vecchi, M.P. Optimization by simulated annealing, (1983) *Science*, 220 (4598), pp. 671-680.

[11] Douglas M. Bryce, "*Plastic Injection Molding , Manufacturing Process Fundamentals"*, Society of Manufacturing Engineers (SME) Michigan 48121, 1996 (29-50).

[12] Jamaludina, K. R. and Muhamad, N. and Ab. Rahman, M. N. and Amin, S. Y. M. and Murtadhahadi, I and Ismail, M. H. (2008) *Injection molding parameter optimization using the Taguchi method for highest green strength for bimodal powder mixture with SS316L in PEG and PMMA.* Advances in Powder Metallurgy & Particulate Materials 2008, 1 . pp. 186-194.

[13] http://books.google.com.my/books?id=3zzCGLjvlw0C&dq=fundamental+of+injection+molding+series/plastic+injection+molding/manufacturing+process+fundamentals&printsec=frontcover&source=bl&ots=qyHooTZRB-&sig=VWKeSRYwvTR_qGbxEyuyBI2YzY4&hl=en&ei=qXasSae2HIiO6gPO7tzjBQ&sa=X&oi=book_result&resnum=10&ct=result#PPA69,M1

[14] Mathew Schwartz, Reverse-Engineering, Computer World Development, November 12, 2001,http://www.computerworld.com/action/article.do?command=viewArticleBasic&articleId=65532

[15] Brian Hayes, Reverse Engineering, American Scientist, The Magazine of Sigma Xi, The Scientific Research Socity, March-April 2006, Volume 94, Number 2, Page: 107

[16] PolyWorks® V10.1 Beginner's Guide, March 2008 InnovMetric Software Inc. pp. 8

[17] Proton Saga Dashboard, January 19, 2007 at 2:50 pm · Filed under Cars, Local News, Malaysian Makes, Proton http://paultan.org/archives/2007/01/19/2007-proton-saga-facelift-launched/

[18] ABS - acrylonitrile butadiene styrene On Designsite.dk, lists applications. Retrieved 27 October 2006.

[19] http://www.moldflow.com/stp/ on 30th March 2009

APPENDIX A1: Gantt Chart For FYP 1

Optimization of Molding Parameter Effect to Warpage of Car Dashboard Based on Plastic Flow Simulation Software

PROJECT ACTIVITIES	Week														
	W1	W2	W3	W4	W5	W6	W7	W8	W9	W10	W11	W12	W13	W14	W15
Project confirmation															
Objectives and scopes															
Literature review															
Methodology															
Learn about MoldFlow software															
Make proposal															
Presentation															
Submit report															

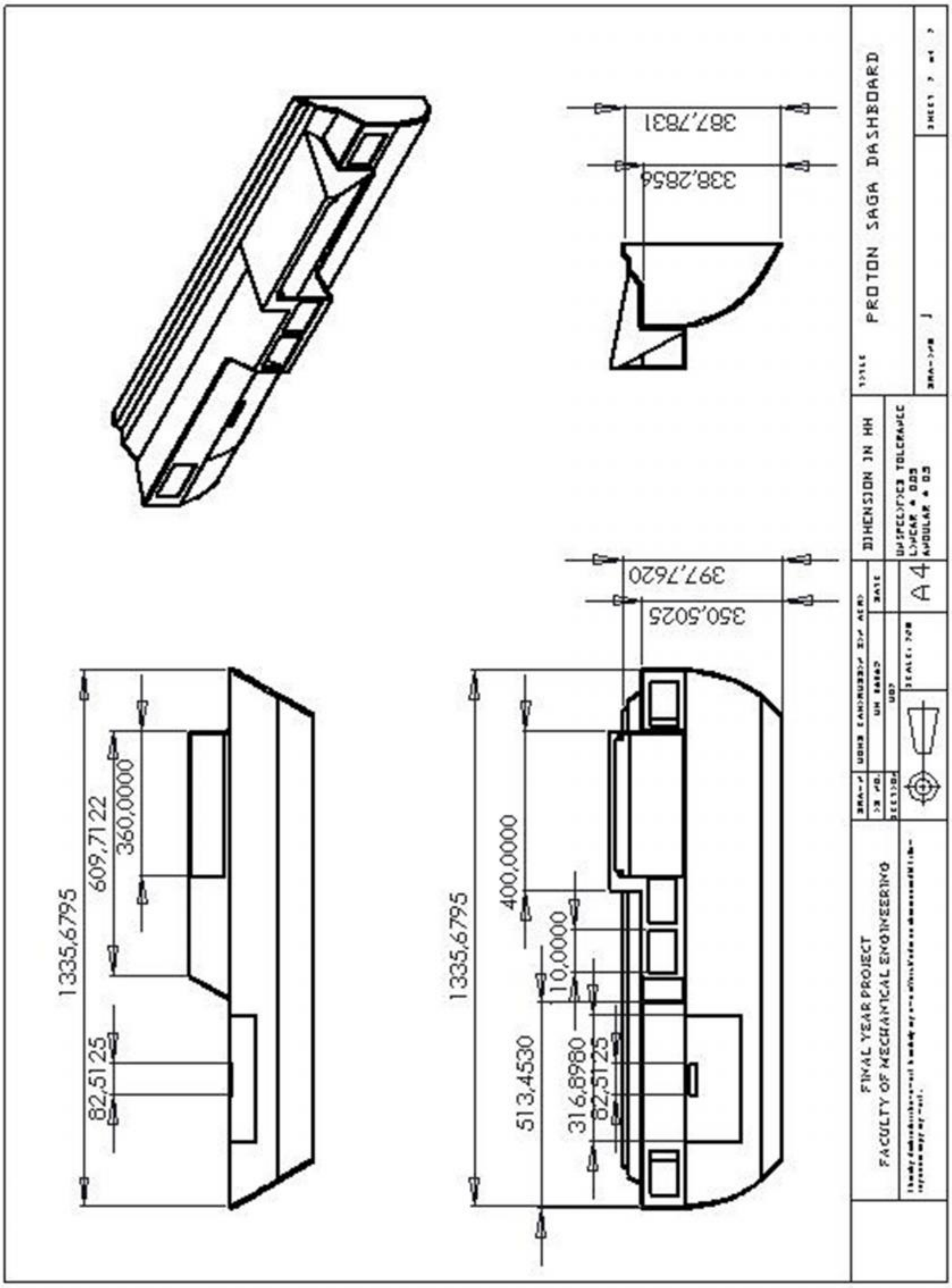
APPENDIX A2: Gantt Chart For FYP 2

Optimization of Molding Parameter Effect to Warpage of Car Dashboard Based on Plastic Flow Simulation Software

PROJECT ACTIVITIES	Week														
	W1	W2	W3	W4	W5	W6	W7	W8	W9	W10	W11	W12	W13	W14	W15
Writing report															
Reviews literature reviews															
Design of experiment															
Part modeling in SolidWorks															
MPI 5.0 analysis															
Data collection															
Result and discussion															
Slide preparation															
Final presentation															
Submit report															

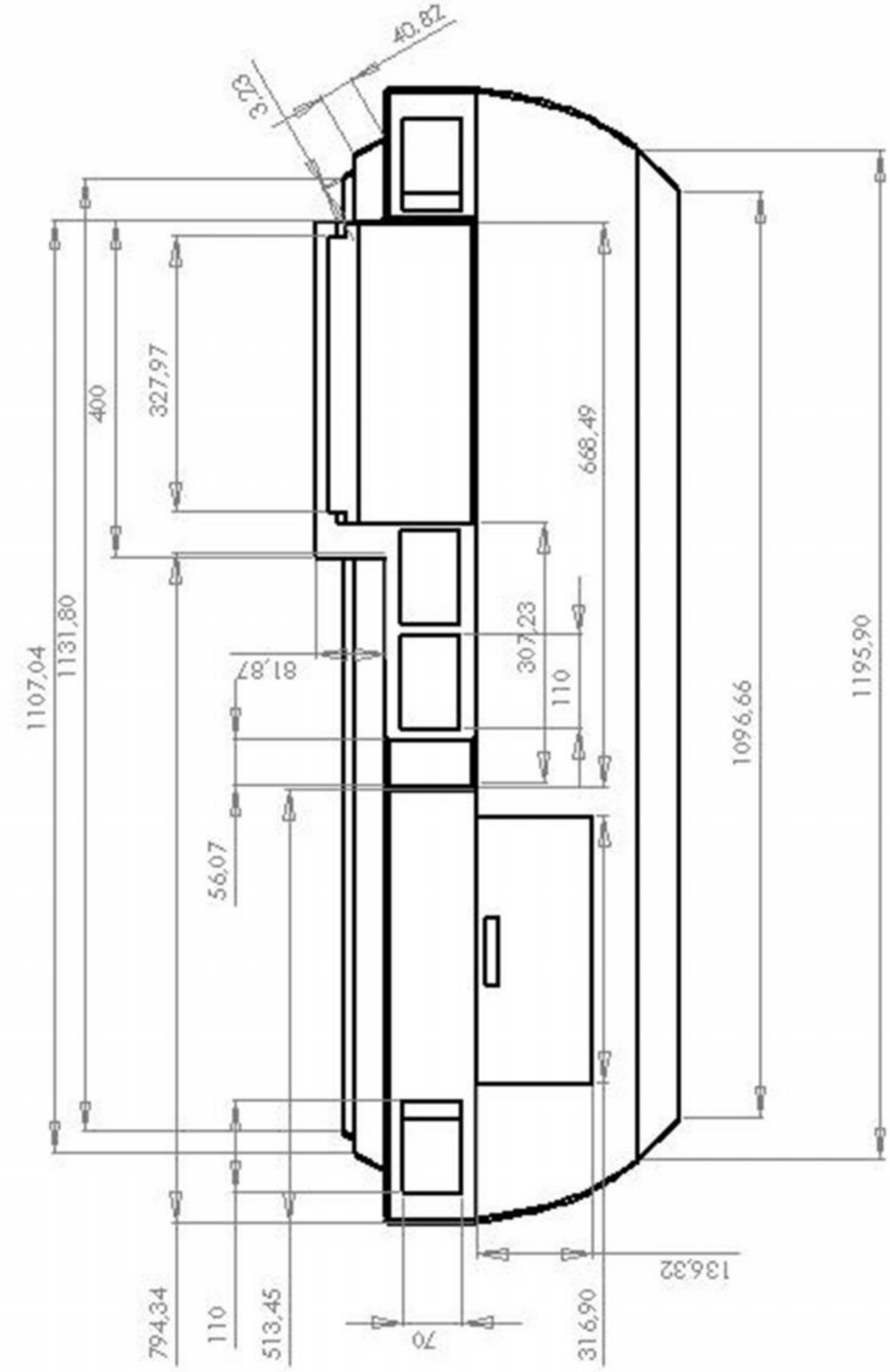
APPENDIX B1

Dashboard Drawing



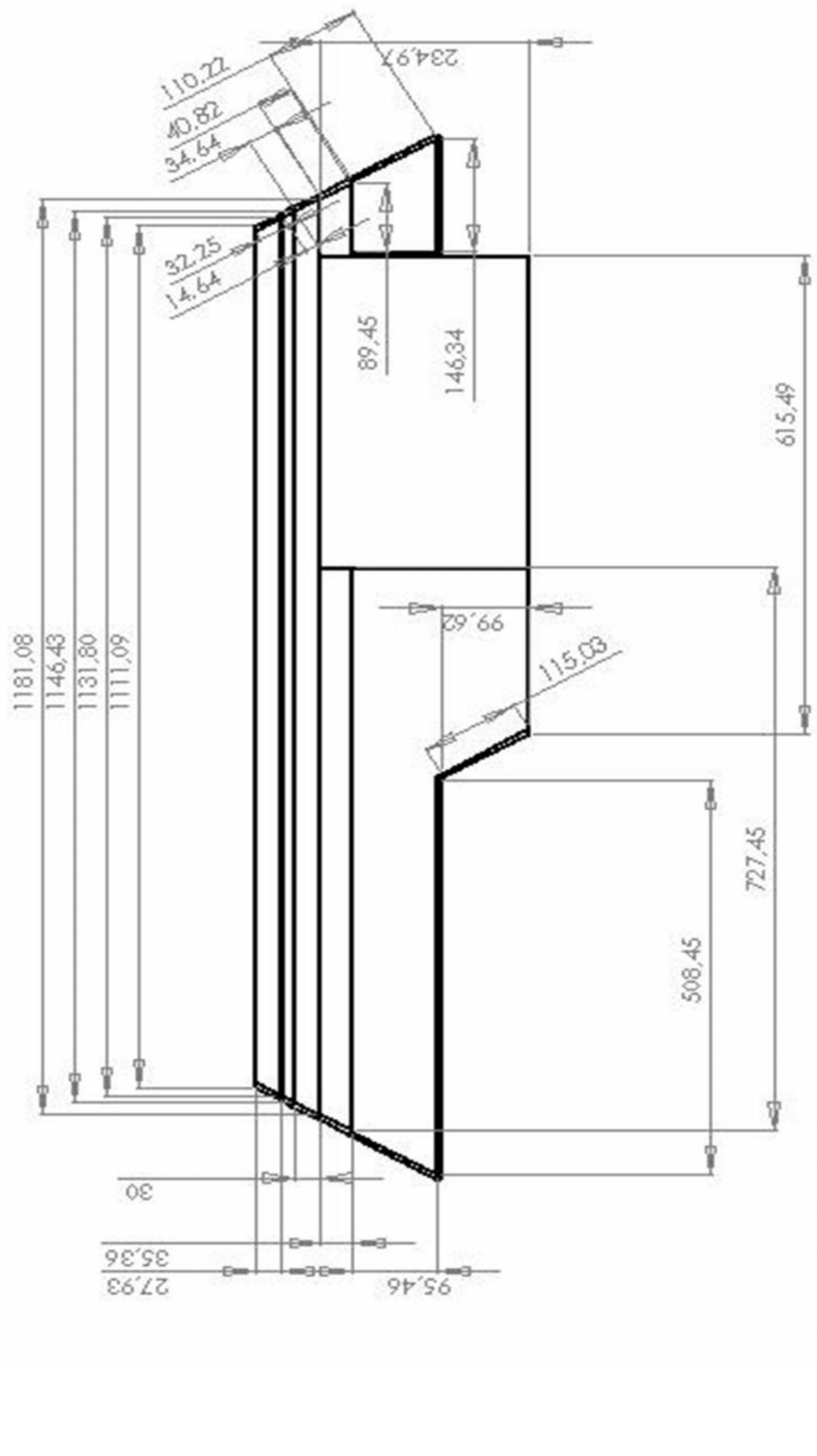
APPENDIX B2

Front View of Design



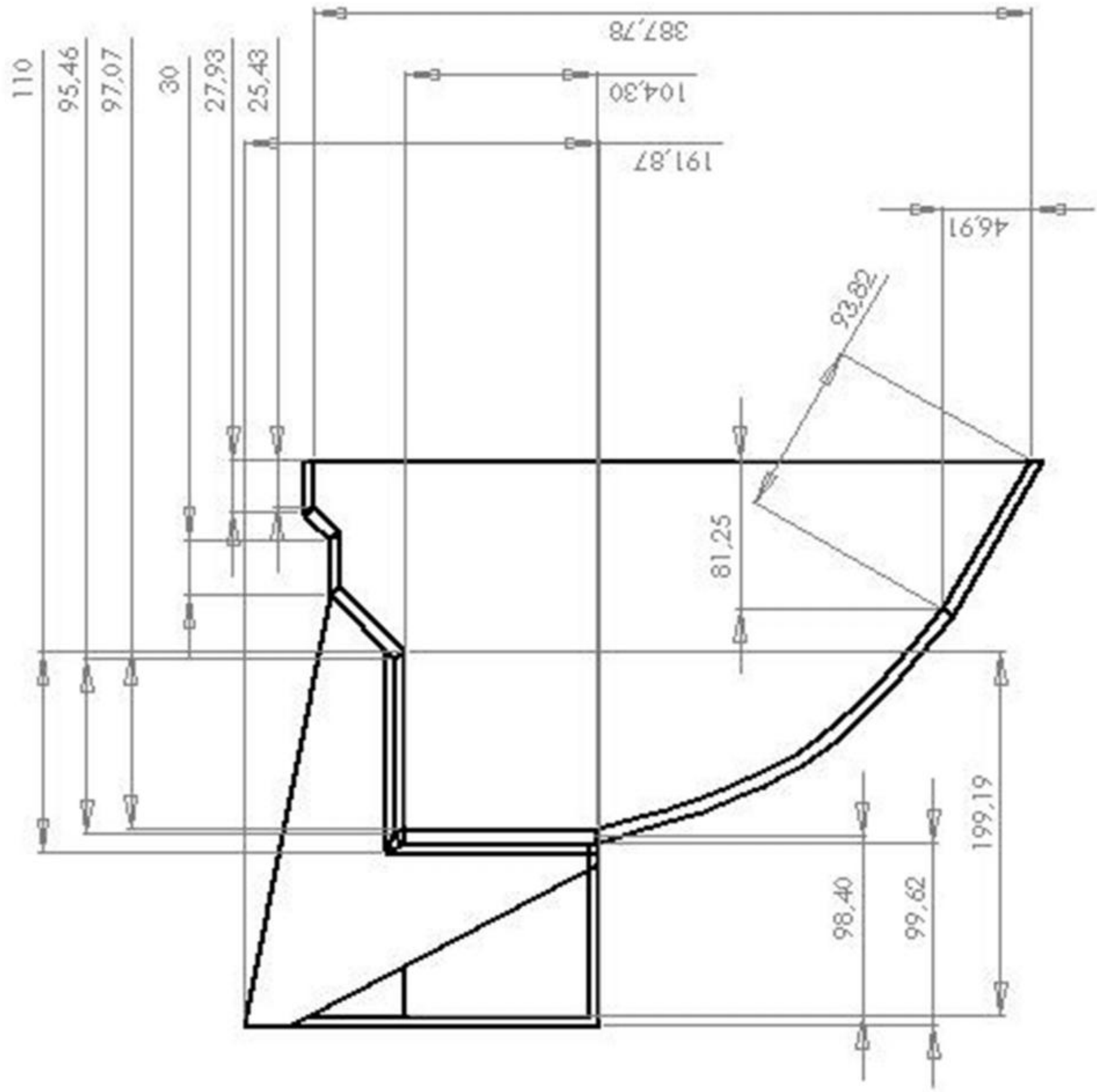
APPENDIX B3

Top View of Design



APPENDIX B4

Right View of Design



APPENDIX C

Moldflow Analysis Results

Flow Analysis

Version: mpi500 (Build 04453)

Analysis commenced at Wed Oct 28 13:35:03 2009

Analysis running on host: sparkz
Operating System: Windows XP Service Pack 3
Processor type: AuthenticAMD x86 Family 15 Model 36 Stepping 2 ~1794 MHz
Number of Processors: 1
Total Physical Memory: 1406 MBytes

Filling Analysis

Packing Analysis

Residual Stress Analysis

Date : OCT28-09
Time : 13:35:03

Allocating memory for analysis...
... finished allocating memory
Processing fusion mesh...
... finished processing fusion mesh
Reading input data...
File name : saga_dashboard_(40-230-200-160)~1
Reading solver parameters...
Reading material data...
Reading process settings...
Reading finite element mesh...
Reading cooling data...

NOTE: In the analysis sequence for this study, a cooling analysis has not been run before the flow analysis. The flow analysis will use the constant mold temperature setting in the Process Settings Wizard. Running a cooling analysis before flow provides more detailed information

```
    about mold temperatures and heat fluxes.

  No mesh for the cores was found.
  Core shift analysis switched OFF
  Reading restart data...
  Note: No restart data was found.
Finished reading input data
Checking input data...

... finished checking input data
Optimizing memory usage...
... finished optimizing memory usage
Initializing variables...
... finished initializing variables


Summary of analysis inputs :

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

Solver parameters :

  No. of laminae across thickness              =       12
  Intermediate output options for filling phase
    No. of results at constant intervals          =       20
    No. of profiled results at constant intervals   =       0
  Intermediate output options for packing phase
    No. of results at constant intervals          =       20
    No. of profiled results at constant intervals   =       0
  Melt temperature convergence tolerance          =     0.2000 C
  Mold-melt heat transfer coefficient            =  2.5000E+04 W/m^2-C
  Maximum no. of melt temperature iterations       =      100
  Pressure trace sample frequency               =      10 Hz
    Total number of pressure trace nodes          =       1
      Node    1                     =      2563

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

Material data :

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

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

  Thermal conductivity                    =      0.1800 W/m-C
```

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

Transition temperature = 100.0000 C

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

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

Residual stress model without CRIMS

Process settings :

Machine parameters :

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

Process parameters :

Flow Rate = 200.0000 cm^3/s
Stroke volume determination = Automatic
Cooling time = 20.0000 s

Velocity/pressure switch-over by injection pressure= 160.0000 MPa
Packing/holding time = 10.0000 s
Ram speed profile (rel):

% shot volume	% ram speed
100.0000	100.0000
0.0000	100.0000

Pack/hold pressure profile (rel):

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

Ambient temperature = 25.0000 C
Melt temperature = 230.0000 C
Ideal cavity-side mold temperature = 40.0000 C
Ideal core-side mold temperature = 40.0000 C

NOTE: Mold wall temperature data from cooling analysis not available

----

Model details :

Mesh Type = Fusion

Match ratio = 90.7 %

Reciprocal match ratio = 91.1 %
Total number of nodes = 2568
Total number of injection location nodes = 1
The injection location node labels are:
2563
Total number of elements = 3084
Number of part elements = 2580
Number of sprue/runner/gate elements = 504
Number of channel elements = 0
Number of connector elements = 0
Parting plane normal (dx) = 0.0000
(dy) = 0.0000
(dz) = 1.0000
Average aspect ratio of triangle elements = 3.9985
Maximum aspect ratio of triangle elements = 177.2240
Element number with maximum aspect ratio = 1470
Minimum aspect ratio of triangle elements = 1.1559
Element number with minimum aspect ratio = 2357
Total volume = 7026.8898 cm^3
Volume filled initially = 0.0000 cm^3
Volume to be filled = 7026.8898 cm^3
Part volume to be filled = 5302.9000 cm^3
Sprue/runner/gate volume to be filled = 1723.9900 cm^3
Total projected area = 5545.7401 cm^2

Filling Analysis

Packing Analysis

Residual Stress Analysis
analysis is beginning ....

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

Time (s)	Volume (%)	Pressure (MPa)	Clamp force (tonne)	Flow rate (cm^3/s)	Status
1.79	1.87	36.81	0.00	202.25	V
3.52	5.27	60.07	12.56	90.81	V
5.28	9.26	77.11	44.72	191.76	V
7.09	14.16	80.24	50.51	205.28	V
8.84	19.04	79.78	53.73	195.16	V
10.63	23.68	83.75	65.37	192.91	V
12.35	28.01	87.80	78.28	196.56	V
14.30	32.56	91.73	100.64	195.69	V

15.92	36.79	94.51	116.57	197.04	V
17.72	41.70	97.26	137.40	197.34	V
19.44	46.40	99.77	160.69	197.93	V
21.39	51.79	102.55	196.22	198.16	V
22.99	56.39	104.75	233.37	198.34	V
24.64	60.88	106.96	278.97	198.69	V
26.61	66.25	109.87	349.77	198.91	V
28.33	70.95	112.77	459.14	198.80	V
29.97	75.32	115.92	583.47	199.03	V
31.64	79.76	120.58	789.99	198.86	V
33.61	84.88	127.60	1147.51	199.47	V
35.38	89.58	134.47	1558.46	199.97	V
37.08	93.86	143.89	2137.12	200.00	V
38.59	96.97	163.20	3538.92	188.79	V/P by inj. pressure
38.68	97.13	130.56	3503.05	66.93	P
40.48	99.07	130.56	3182.43	50.87	P
42.07	99.79	130.56	3224.75	32.12	P
42.08	100.00	130.56	3226.88	32.01	Filled

Execution time in Filling Phase = 46.58 s

Packing phase:

Time (s)	Packing (%)	Pressure (MPa)	Clamp force (tonne)	Status
42.08	0.00	130.56	3226.91	P
42.49	1.54	130.56	3300.86	P
43.99	7.20	130.56	3449.90	P
45.24	11.92	130.56	3496.51	P
46.49	16.63	130.56	3499.80	P
47.74	21.35	130.56	3474.74	P
48.61	24.61	0.00	3442.65	P
48.61				Pressure released
49.25	27.05	0.00	3291.83	P
50.50	31.76	0.00	3090.31	P
51.75	36.48	0.00	2896.45	P
53.25	42.14	0.00	2678.04	P
54.50	46.85	0.00	2507.91	P
55.75	51.57	0.00	2344.62	P
57.00	56.28	0.00	2187.15	P
58.50	61.94	0.00	2004.81	P
59.75	66.65	0.00	1857.67	P
61.00	71.37	0.00	1714.69	P
62.50	77.03	0.00	1548.23	P
63.75	81.74	0.00	1413.50	P
65.00	86.46	0.00	1282.24	P
66.50	92.11	0.00	1129.18	P
67.75	96.83	0.00	1006.31	P
68.75	100.00	0.00	912.54	P

Filling phase results summary :

Maximum injection pressure (at 38.595 s) = 163.1980 MPa

End of filling phase results summary :

Time at the end of filling = 42.0825 s
Total weight = 7240.2101 g
Maximum Clamp force - during filling = 3538.9185 tonne
Recommended ram speed profile (rel):

% stroke	% speed
0.0000	10.0000
10.0000	20.9080
24.3080	20.9080
30.0000	44.4203
40.0000	80.9029
50.0000	100.0000
60.0000	92.0558
70.0000	76.8750
80.0000	56.0432
90.0000	31.4936
100.0000	10.0000

Melt front is entirely in the cavity at % fill = 24.3080 %

Filling phase results summary for the part :

Bulk temperature - maximum (at 15.925 s) = 236.9000 C
Bulk temperature - 95th percentile (at 7.089 s) = 235.7880 C
Bulk temperature - 5th percentile (at 42.072 s) = 161.4220 C
Bulk temperature - minimum (at 42.072 s) = 138.8430 C

Wall shear stress - maximum (at 38.595 s) = 1.0045 MPa
Wall shear stress - 95th percentile (at 38.595 s) = 0.5732 MPa

Shear rate - maximum (at 38.595 s) = 822.7230 1/s
Shear rate - 95th percentile (at 7.089 s) = 140.7100 1/s

End of filling phase results summary for the part :

Total part weight = 5492.7802 g

Bulk temperature - maximum = 231.8770 C
Bulk temperature - 95th percentile = 222.7080 C
Bulk temperature - 5th percentile = 161.4230 C
Bulk temperature - minimum = 138.8440 C
Bulk temperature - average = 196.2060 C
Bulk temperature - RMS deviation = 18.3244 C

Wall shear stress - maximum = 0.6285 MPa
Wall shear stress - 95th percentile = 0.3344 MPa
Wall shear stress - average = 0.1465 MPa
Wall shear stress - RMS deviation = 0.1006 MPa

Frozen layer fraction - maximum = 0.4009
Frozen layer fraction - 95th percentile = 0.3292
Frozen layer fraction - 5th percentile = 0.1484
Frozen layer fraction - minimum = 0.0000
Frozen layer fraction - average = 0.2309
Frozen layer fraction - RMS deviation = 0.0518

Shear rate - maximum = 87.6738 1/s
Shear rate - 95th percentile = 11.6807 1/s

Shear rate - average = 2.2989 1/s
Shear rate - RMS deviation = 5.2076 1/s

Filling phase results summary for the runner system :

Bulk temperature - maximum (at 5.284 s) = 250.7610 C
Bulk temperature - 95th percentile (at 7.089 s) = 244.9450 C
Bulk temperature - 5th percentile (at 42.004 s) = 230.1320 C
Bulk temperature - minimum (at 42.004 s) = 223.5830 C

Wall shear stress - maximum (at 38.595 s) = 2.2604 MPa
Wall shear stress - 95th percentile (at 37.081 s) = 0.1821 MPa

Shear rate - maximum (at 28.327 s) = 4.0803E+04 1/s
Shear rate - 95th percentile (at 26.608 s) = 796.8290 1/s

End of filling phase results summary for the runner system :

Total sprue/runner/gate weight = 1747.4300 g
Bulk temperature - maximum = 239.3470 C
Bulk temperature - 95th percentile = 238.5370 C
Bulk temperature - 5th percentile = 230.1450 C
Bulk temperature - minimum = 223.5990 C
Bulk temperature - average = 233.7120 C
Bulk temperature - RMS deviation = 2.4852 C

Wall shear stress - maximum = 0.7056 MPa
Wall shear stress - 95th percentile = 0.1301 MPa
Wall shear stress - average = 0.0379 MPa
Wall shear stress - RMS deviation = 0.0425 MPa

Frozen layer fraction - maximum = 0.2598
Frozen layer fraction - 95th percentile = 0.1942
Frozen layer fraction - 5th percentile = 0.0391
Frozen layer fraction - minimum = 0.0389
Frozen layer fraction - average = 0.1120
Frozen layer fraction - RMS deviation = 0.0502

Shear rate - maximum = 8517.6396 1/s
Shear rate - 95th percentile = 66.5166 1/s
Shear rate - average = 19.5509 1/s
Shear rate - RMS deviation = 63.0155 1/s

Packing phase results summary :

Peak pressure - minimum (at 45.242 s) = 40.2455 MPa
Clamp force - maximum (at 38.595 s) = 3538.9185 tonne
Total weight - maximum (at 48.607 s) = 7306.3998 g

End of packing phase results summary :

Time at the end of packing = 68.7539 s
Total weight = 7272.9301 g

Packing phase results summary for the part :

Bulk temperature - maximum (at 42.083 s) = 231.8770 C
Bulk temperature - 95th percentile (at 42.083 s) = 222.7080 C
Bulk temperature - 5th percentile (at 68.754 s) = 77.8220 C

Bulk temperature - minimum (at 68.754 s) = 69.1180 C

Wall shear stress - maximum (at 65.004 s) = 0.9796 MPa
Wall shear stress - 95th percentile (at 48.607 s) = 0.3531 MPa

Volumetric shrinkage - maximum (at 42.083 s) = 8.3240 %
Volumetric shrinkage - 95th %ile (at 42.083 s) = 5.0639 %
Volumetric shrinkage - 5th %ile (at 49.254 s) = 0.9683 %
Volumetric shrinkage - minimum (at 53.254 s) = 0.5097 %

Total part weight - maximum (at 50.504 s) = 5551.2900 g

End of packing phase results summary for the part :

Total part weight = 5550.6902 g

Bulk temperature - maximum = 173.5200 C
Bulk temperature - 95th percentile = 151.2730 C
Bulk temperature - 5th percentile = 77.8220 C
Bulk temperature - minimum = 69.1180 C
Bulk temperature - average = 119.7900 C
Bulk temperature - RMS deviation = 17.4382 C

Frozen layer fraction - maximum = 1.0000
Frozen layer fraction - 95th percentile = 1.0000
Frozen layer fraction - 5th percentile = 0.3459
Frozen layer fraction - minimum = 0.2776
Frozen layer fraction - average = 0.5614
Frozen layer fraction - RMS deviation = 0.1647

Volumetric shrinkage - maximum = 3.3524 %
Volumetric shrinkage - 95th percentile = 2.7574 %
Volumetric shrinkage - 5th percentile = 1.0355 %
Volumetric shrinkage - minimum = 0.5601 %
Volumetric shrinkage - average = 1.9150 %
Volumetric shrinkage - RMS deviation = 0.5308 %

Sink index - maximum = 1.6553 %
Sink index - 95th percentile = 1.0136 %
Sink index - minimum = 0.3961 %
Sink index - RMS deviation = 0.3676 %

Packing phase results summary for the runner system :

Bulk temperature - maximum (at 42.083 s) = 239.3470 C
Bulk temperature - 95th percentile (at 42.083 s) = 238.5370 C
Bulk temperature - 5th percentile (at 68.754 s) = 190.6660 C
Bulk temperature - minimum (at 68.754 s) = 132.3150 C

Wall shear stress - maximum (at 68.754 s) = 2.5174 MPa
Wall shear stress - 95th percentile (at 48.607 s) = 0.1429 MPa

Volumetric shrinkage - maximum (at 68.754 s) = 7.7864 %
Volumetric shrinkage - 95th %ile (at 57.004 s) = 7.6829 %
Volumetric shrinkage - 5th %ile (at 48.607 s) = 2.0576 %
Volumetric shrinkage - minimum (at 68.754 s) = 1.3054 %
Sprue/runner/gate weight - max. (at 38.595 s) = 1759.0700 g

End of packing phase results summary for the runner system :

Total sprue/runner/gate weight = 1722.2400 g
Bulk temperature - maximum = 233.5910 C
Bulk temperature - 95th percentile = 232.6840 C
Bulk temperature - 5th percentile = 190.6660 C
Bulk temperature - minimum = 132.3150 C
Bulk temperature - average = 222.2350 C
Bulk temperature - RMS deviation = 13.7376 C

Frozen layer fraction - maximum = 0.6583
Frozen layer fraction - 95th percentile = 0.3893
Frozen layer fraction - 5th percentile = 0.0910
Frozen layer fraction - minimum = 0.0824
Frozen layer fraction - average = 0.2054
Frozen layer fraction - RMS deviation = 0.0958

Volumetric shrinkage - maximum = 7.7864 %
Volumetric shrinkage - 95th percentile = 7.6576 %
Volumetric shrinkage - 5th percentile = 2.9692 %
Volumetric shrinkage - minimum = 1.3054 %
Volumetric shrinkage - average = 5.2336 %
Volumetric shrinkage - RMS deviation = 1.3543 %

Sink index - maximum = 5.8836 %
Sink index - 95th percentile = 5.7597 %
Sink index - minimum = 3.4900 %
Sink index - RMS deviation = 1.2419 %

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

Filling Analysis

Packing Analysis

Residual Stress Analysis
has completed successfully.

Weld line/air trap analysis completed

Preparing output data...
Finished preparing output data

SYNERGY Weld-line and air trap
has completed successfully.

Execution time
Analysis commenced at Wed Oct 28 13:35:03 2009
Analysis completed at Wed Oct 28 13:37:21 2009
CPU time used 91.70 s

US patent 6,096,088 ,
Australian Patent No. 721978 ,
and foreign patents and pending applications

Warpage Analysis

Version: mpi500 (Build 04453)

Analysis running on host: sparkz
Operating System: Windows XP Service Pack 3
Processor type: AuthenticAMD x86 Family 15 Model 36 Stepping 2 ~1794 MHz
Number of Processors: 1
Total Physical Memory: 1406 MBytes

Analysis commenced at Wed Oct 28 13:37:26 2009

Model file name: dashboard_din_updated_final_study_perfect!!!_terbaik_(40-230-200-160).udm

Fusion mesh statistics:
Percentage of edge element (number) 11.28 %
Percentage of edge element (area) 1.94 %
Percentage of matched elements 90.7 %
Percentage of reciprocal_matched elements 91.1 %

Reading solver parameters...
Corner effect is OFF
Reading mechanical property and residual stress data...

Analysis model: residual stress without crims.

Establishing MPC constraint relationship...

Defining anchor plane...
Number of separate cavities = 1

Writing input file for structural analysis program...

Launching structural analysis program...

Reading structural analysis input file...
...finished reading structural analysis input file.

Beginning load incrementation loop...

Load Case 1: Total Shrinkage Effect ......

Setting structure information...

Assembling stiffness matrix...

Solving finite element static equilibrium equations...

Kstep Kstra Nref Nite Node Ipos Negpv Detk Rfac Displacement

```
 ----------------------------------------------------------------------
   1    1    1    0    273   3    0  1.0e+00  1.000e+00  -1.005e+01
```

Minimum/maximum displacements at last step (unit: mm):

	Node	Min.	Node	Max.
Trans-X	154	-2.3250e-01	1267	5.1965e+00
Trans-Y	251	-6.2919e+00	600	2.3019e+00
Trans-Z	273	-1.0047e+01	50	2.7505e+00

Load Case 2: Differential Shrinkage Effect

```
-----------------------------------------
   2    0    0    0    273   3    0  1.0e+00  1.000e+00  -1.005e+01
```

Minimum/maximum displacements at last step (unit: mm):

	Node	Min.	Node	Max.
Trans-X	154	-2.3250e-01	1267	5.1965e+00
Trans-Y	251	-6.2919e+00	600	2.3019e+00
Trans-Z	273	-1.0047e+01	50	2.7505e+00

Load Case 3: Differential Orientation Effect

```
-------------------------------------------
   3    0    0    0    273   3    0  1.0e+00  1.000e+00   1.625e-08
```

Minimum/maximum displacements at last step (unit: mm):

	Node	Min.	Node	Max.
Trans-X	243	-6.0838e-09	490	1.9058e-08
Trans-Y	456	-5.5149e-08	1157	5.5840e-08
Trans-Z	1106	-4.1700e-08	996	5.3687e-08

Load Case 4: Differential Cooling Effect

```
---------------------------------------
   4    0    0    0    273   3    0  1.0e+00  1.000e+00  -2.170e-05
```

Minimum/maximum displacements at last step (unit: mm):

	Node	Min.	Node	Max.
Trans-X	1166	-2.3014e-05	256	4.1266e-05
Trans-Y	251	-1.8215e-05	880	8.7760e-05
Trans-Z	849	-3.5056e-05	1151	3.8795e-05

Structural analysis completed in 4.56 secs.

Writing result file...

Best-fit transformation will be used to display warpage deflections
if no anchor plane is defined.

Execution time
  Analysis commenced at Wed Oct 28 13:37:26 2009

Analysis completed at Wed Oct 28 13:37:33 2009
CPU time used 5.27 s
Warpage analysis has completed successfully.
