ANALYSIS OF THE POSSIBILITY OF MINIMISING THE WARPAGE IN INJECTION MOULDING

Gabbalage Anjula Ruwan Premarathna

(118315T)

Master of Engineering in Manufacturing Systems Engineering

Department of Mechanical Engineering

University of Moratuwa
Sri Lanka

August 2015

ANALYSIS OF THE POSSIBILITY OF MINIMISING THE WARPAGE IN INJECTION MOULDING

Gabbalage Anjula Ruwan Premarathna

(118315T)

This thesis was submitted in partial fulfilment of the requirements for the Degree of Master of Engineering in Manufacturing Systems Engineering

Department of Mechanical Engineering

University of Moratuwa
Sri Lanka

August 2015

DECLARATION

I declare that this is my own work and this thesis/dissertation does not incorporate without acknowledgement any material previously submitted for a Degree or Diploma in any other University or institute of higher learning and to the best of my knowledge and belief it does not contain any material previously published or written by another person except where the acknowledgement is made in the text.

Also, I hereby grant to University of Moratuwa the non-exclusive right to reproduce and distribute my thesis/dissertation, in whole or in part in print, electronic or other medium. I retain the right to use this content in whole or part in future works (such as articles or books).

Signature: Date:

The above candidate has carried out research for the Master's thesis under my supervision.

Signature of the supervisor: Date:

Abstract

This thesis focuses on the research “to investigate and analyse the warpage in a product and reduce the warpage using optimum parameters”.

Factors affecting for warpage are discussed and categorized their relative position of affecting. An article subjected to warpage is selected and factors affected for the warpage are detailed analysed one by one. The research carried out on the basis of selected major factors. Part geometry, gate location, runner system, filling and packing/ holding pressures, filling and packing/ holding times and cooling layout are analysed and changed to determine optimum parameters and minimize the warpage factor. Modified mould design was done by utilizing Computer Aided Design and analysed the mould to ensure the success of the design. The CAD Software used for design is Unigraphics NX and two software packages used for analysis of warpage are Auto Desk Moldflow Advisor and Solid Works Plastics. Finally with the justification of changed parameters the existing mould is modified to meet the required quality of product.

In this context, all above details are comprehensively discussed and summarized in the body of this report accompanied by necessary drawings, data tables and analysis results etc.

Acknowledgement

My special thanks goes to my supervisor Prof. M.A.R.V. Fernando, Department of Mechanical Engineering, University of Moratuwa for his continuous supervision, that is for the valuable guidance, advices, and real encouragement and for the effort taken to direct me towards more and more practical experiences and for the help he had given to improve my knowledge by giving his maximum attention.

Also I’m extremely grateful to my course coordinator, Dr. R.A.R.C. Gopura, Department of Mechanical Engineering, University of Moratuwa, for giving his maximum assistance.

At the same time I never forget to thanks for the support given by the staff of the Boehm Leckner Multi Moulds (Pvt) Ltd, Katunayake and OREL MFG Pvt Ltd, Navinna, to successfully complete my research.

Finally I’d like to give my heartiest thanks to those who helped me lot in many ways to make my research successful.

G.A.R. Premarathna
(anjula_1@yahoo.com)

Department of Mechanical Engineering
University of Moratuwa

CONTENT

LIST OF FIGURES

LIST OF TABLES

LIST OF ABBREVIATIONS

Abbreviation	Description
AMA	Autodesk Moldflow Adviser
SWP	Solid Works Plastic
EDM	Electrode Discharge Machining
CMM	Coordinate Measuring Machine

LIST OF APPENDICES

1 INTRODUCTION

1.1. Background

Plastic materials are commonly used in every industry. The most important reason for this is the material properties of the plastics. Some of these properties are lightness, resistance to corrosion, ease to give shape etc. The most important is that their physical and chemical properties can be changed as desired. Plastic materials can be used in packaging, aerospace, aviation, building and construction, automotive, agriculture, irrigation, sanitation, electrical conduits, and chemical processing plants etc. Plastic Injection Moulding is considered the most prominent process for mass producing plastic parts. More than one third of all plastic products are made by injection moulding, and over half of the world's polymer processing equipment is used for the injection moulding process. Plastic injection moulding is one of the manufacturing processes carried out by a five consecutive phases which are plasticization, injection, packing, cooling and ejection. This process is complex, but highly efficient means of producing a wide variety of three dimensional thermoplastic parts in a large volume of production. During production, quality problems of the plastic parts such as warpage, shrinkage, weld and meld lines, flow mark, flash, sink mark and void are affected from manufacturing process conditions which include the melt temperature, mould temperature, injection pressure, injection velocity, injection time, packing pressure, packing time, cooling time, cooling temperature etc. Among the defects associated with quality of the product, one of the frequently faced problems is warpage. Warpage, is a distortion of the shape of the final injection-moulded item, is caused by differential shrinkage; that is, if one area or direction of the article undergoes a different degree of shrinkage than another area or direction, the part will warp as shown in Figure 1.1.

Figure 1.1 Deformation caused by Warpage [7]

During plasticization, injection, packing, cooling and ejection processes, the residual stress is produced due to high pressure, temperature change, and relaxation of polymer chains, resulting in warpage of the part. In order to yield a product with high precision, optimum mould geometry and processing parameters must be found. To reduce the cost and time at the design stage, it is important to simulate warpage of the injection moulded part.

1.2. Objectives

The purpose of this project is to investigate and analyse the warpage in moulds and reduce the defect using optimum parameters. By selecting an article subjected to warpage, the possible causes are aimed to be discussed and found the optimum parameters for minimum warpage by varying those factors. Then the results obtained from the analysis are aimed to be compared with practical application for the selected product.

2 LITERATURE REVIEW

2.1. Root Causes of Warpage

Warpage of thermoplastic parts can be caused by two mechanisms, the contraction of the polymer during cooling and the tendency of high-molecular-weight molecules to "relax' if they are under stress [1]. The first is easy to understand, as it is a common property of all solids. The second may be compared to stretching a rubber band. As the stress is reduced, the band returns to its original size at a speed related to the rate of stress reduction. However, if the band is “frozen” while stretched, it retains its shape until the temperature increases sufficiently to allow it to “relax” and return to its normal state. As a polymer melt is injected into a mould or extruded through a die, a rapid cooling must take place in order to achieve economic cycles or throughput rates. All polymers have low heat transfer coefficients, so the rate of heat transfer is relatively slow. This is further complicated during injection moulding by the shrinkage that occurs allowing the part to retract from the mould surface, losing effective cooling. In the semi-crystalline polymers such as polypropylene and polyethylene, it is necessary to remove the heat of crystallization, in addition to the heat to reduce the temperature of the mass.

There is additional concern with semi crystalline polymers that internal stresses are developed during cooling due to the differential shrinkage between the crystalline and amorphous regions [2].

Thicker part sections have limited cooling available and cool more slowly than their thinner or better cooled counterparts. Ribs, bosses, corners, differential mould temperatures, etc., all contribute to variations in cooling time and rate of cooling. In the mould, a part develops a differential temperature profile. When the part is ejected, the thicker sections are still cooling while thinner sections may have reached their final temperature. As the part cools further the thicker areas, which are no longer restrained, contract and possibly cause warpage.

The second source of warpage is related to the molecular structure of the polymer. Polymers are made up of very long molecules which, when molten, resist flow because of their high viscosity. Forcing these long molecules through constricted geometries at very high velocities such as die lands, runners, gates, thin pan sections, etc., subjects the molecules to high strains (similar to a rubber band being stretched). If the stress is removed and the polymer does not cool, the molecules rearrange themselves into a lower stress condition (analogous, in respects, to the annealing of metals). However, in injection moulding, the cooling of the part does not allow this to happen and parts generally have some level of “moulded-in stress” after they have been ejected. If sections of the part are still hot, relaxation continues, incrementally contributing to warpage beyond that which may occur due to thermal contraction [2], [3].

Differential stresses may also occur due to non-uniform filling profiles. A classic example of this is a bottom, centre-gated, rectangular shallow box. Unless flow directors are used, filling the edges is not simultaneous. Relaxation begins in the edge, which fills first, i.e., the near edge. Even though the time frame is very small, there is enough differential in relaxation compared to the far edge, that non-uniform stress relief can occur after the part is ejected.

It is also possible that after complete cooling a residual degree of moulded-in stress may still exist in the part which, due to the geometry or rigidity of the part, does not cause any warpage. However, if at some point in its application, the part is exposed for a sufficient time to an elevated temperature, it is possible the part will lose some of its stiffness and allow these stresses to relax, causing warpage [2].

2.2. Root Cause Analysis

This is the most important step. Making changes to the processing parameters or to the mould without understanding the cause of the problem could make things worse. Often a modification of the moulding parameters can reduce the shrinkage and warpage enough to make satisfactory parts. This is the first and least expensive change to make, unless a significantly longer cycle-time is necessary. If the cycle time causes a significant part price increase, it may be more economical to consider one or more of the following [2].

1. Is the mould running on the same moulding machine? A different machine will probably have a different-sized heating cylinder, so the residence time will be different for the material. The actual pressure on the plastic during

injection may be different, even though the hydraulic pressure is the same. Each moulding machine has a step-up ratio between the hydraulic pressure and the actual pressure at the nozzle; the most common step-up ratio is 10 to 1, or the plastic has ten times the pressure of the hydraulic pressure in the injection cylinder. The actual temperature inside the heating cylinder may be different due to thermocouple location, heater band location, or the thermal conductivity of the heating cylinder [2].

2. Has the mould been damaged in some manner that causes an unacceptable part? For example, minor flash problems, if not stopped, usually lead to major flash problems. The flash, being thinner than the moulded part, shrinks less in the mould than does the part. As the part cools, the cavity pressure is reduced until the full tonnage of the machine is applied to the thin flash between the parting lines. This often results in progressively more deformation of the steel at the flash point and progressively more and larger flash [2].

If neither of the above applies, then the problem is probably related to the process or material:

3. Examine the processing conditions. Is the plastic being moulded at the proper temperature and pressure? Is the holding time adequate? Is the cure time adequate? Is the plastic dry enough as it enters the moulding machine? Are there variations in cycle time or ambient temperature? [2]
4. Is the mould temperature correct? Are the cooling hoses and fittings of adequate size? Are they the same size or configuration as when acceptable parts were made? Are there adequate coolant feed lines to separately feed each cooling zone? Is the temperature of the cooling water constant? Is the flow of the cooling water constant? [2]
5. Is the flow pattern, combined with molecular or fibre orientation, contributing to shrink or warp? Can a material change improve the orientation problem? Can a change in the number or location of gates improve the flow pattern? [2]
6. Are there thickness variations or ribs that are causing uneven shrinkage? Are there bosses attached to sidewalls that contribute to thickness variations? Is the part constrained in one area and not another, causing uneven shrinkage? [2]

7. Are the tolerances unrealistic? Will the part fulfil its fit and functional requirements even though it does not meet the print? One possible part-design solution is to loosen tolerances [2].
8. If good parts were never produced on the mould, then there may be a tooling problem that must be addressed [2].

2.3. Processing Considerations

The injection-moulding process is a semi continuous, sequential process with a number of phases. The packing phase of the process begins once the melt flow-fronts have reached the extremities of the cavity. Since plastics are compressible to a fair degree, the magnitude of the packing pressure determines the weight of material ultimately injected into the fixed-mould cavity volume. Holding pressure is applied to the plastic melt in the cavity via pressure on the moulding-machine screw through the Sprue, runner, and gate until the gate freezes. The frozen gate keeps any plastic from leaking out of the cavity thereafter. Until the gate freezes, the holding pressure adds material to make up for any shrinkage during cooling. Even after the gate freezes, the part continues to shrink. The extent of plastic part shrinkage and potential warpage is a direct result of the pressure transmitted to each section of the part via the gate and runner system. Areas experiencing the highest pressures will exhibit the lowest amounts of shrinkage. Those sections nearest the gate will shrink the least. The level of shrinkage will increase towards the periphery of the part. Since this situation is always present, warpage will result if the part is exposed to elevated temperatures that are high enough to allow stress relaxation to occur [4].

If the part has been designed with a uniform wall thickness, and if great care is taken in designing the gating system, wall thickness warpage still can result. For example, it may be desirable to gradually diminish the wall thickness from the gate area to the outer edges of the part to compensate or the pressure gradient throughout the part. The thicker sections will tend to shrink more and help to adjust for any imbalances created by pressure differences in the moulding process [5].

2.3.1. Melt Temperatures and Uniformity

One of the many factors that affect the repeatability of the moulding process is with the uniformity of the melt. Several factors contribute to the melt uniformity. In the old days before screw injection units, it was considerably more challenging to make a uniform melt [6]. The screw mechanism within the moulding machine is designed to encourage uniformity due to its tendency to assist in mixing the melt as it conveys the plastic forward along the screw. Additional mixing and heating is added as the backpressure on the screw is increased. Backpressure is hydraulic pressure applied to the injection side of the hydraulic cylinder that moves the screw during injection. Higher backpressure adds friction heat to the melt and increases the mixing action. The following are some of the more common sources of problems with melt temperature and uniformity [7].

- Fast cycles with the moulding machine at or near its maximum plasticizing capacity can lead to non-melted plastic pellets in the melt stream and, obviously, to non-uniform melt temperature and viscosity. Under these conditions, it is even possible for a gate to be plugged by an inadequately melted pellet of plastic before the mould cavity is filled or adequately packed. This causes short shots or erratic shrinkage.
- The moulding machine itself may be the source of a problem. For example, if the non-return valve in the injection unit is leaking, the machine may not be able to maintain injection or holding pressure (“lose the cushion”), causing greater shrinkage. No uniform heating from inadequate backpressure or burned-out heating bands can cause problems.
- Inadequate mixing can cause uneven shrinkage when colorant is added to the melt. Since colorants can act as nucleating agents, if the colour is unevenly dispersed throughout the melt, the crystalline ratio will be uneven, causing more shrinkage where the colorant concentration is highest.

2.3.2. Mould Temperatures and Uniformity

If mould temperature varies for any reason throughout a product run, there is going to be some variation in the shrinkage of the moulded part. Higher mould temperatures lead to higher post-mould shrinkage, but more stable parts in the long term. However, if the mould temperature rises without a corresponding increase in holding-pressure time, there can be backflow out of the cavity into the runner causing erratic shrinkage [8].

Changes in the environmental temperature or humidity can cause fluctuations in mould temperature during the production run. If a central cooling tower is used, the ambient temperature of the cooling tower will vary depending on the number of moulding machines running at any given time and on environmental conditions. Depending on a cooling tower without auxiliary temperature-control devices is unwise.

Many moulding shops operate in an ambient air condition. That is, they do not have temperature and humidity controls in the moulding department. Therefore, ambient air temperature can influence the temperature of the moulding machine and its clamping system [8]. Air temperature can affect the efficiency of the moulding machine cooling system as well as the temperature controls for the mould. Radiation cooling of the mould and the heating section of the moulding machine influence their temperatures. The temperature of the plastic pellets, as they are added to the moulding machine hopper, can affect the heat load required to melt and process the plastic. And if there are openings to the outside of the building, such as overhead doors or windows, breezes through these openings can influence the moulding machine and end product.

Humidity affects the efficiency of heat exchangers and the moisture content of plastic pellets. As the moisture content of the pellets rises, the effort required to remove or boil off the moisture before and during the moulding process increases. This can influence the temperature and condition of the melt as it enters the mould. The percentage of regrind and its pellet size and moisture condition contribute to the temperature and uniformity of the plastic melt. Physical properties change with each cycle through the machine and the grinder, and there may be some mechanical rupturing of the molecular chains. Regrinding may also change the lengths of any fibrous reinforcements. These variations affect the shrink rate, the strength, and the rigidity of the moulded part.

Inadequate coolant flow or too long a flow path can cause variations in mould temperature from start up until an equilibrium condition is reached. Then, any hesitation or inconsistency in cycle time will cause temperature fluctuations.

Inadequate coolant flow or too long a flow path can cause variations in mould temperature from start up until an equilibrium condition is reached. Then, any hesitation or inconsistency in cycle time will cause temperature fluctuations.

The cooling load, due to gate proximity or section thickness variations in the moulded part, may require that certain areas of the mould be cooled more aggressively in order to approximate the ideal condition of cooling all areas of the moulded part at the same rate.

One of the more common problems in moulding shops is inadequate mould cooling. The supply line to the moulding machine from the cooling tower may be too small. The pressure differential between the tower supply and return lines may be too low. There may not be a sufficient number of outlets to separately control each zone of the mould. Many moulding shops have about four supplies and return lines available for the mould, while the mould has eight or more cooling zones. The usual unsatisfactory practice is to plumb several zones in series [9].

For optimum performance, the water flow rate through the mould should be high enough that the flow is turbulent. Turbulent flow continually mixes the water in the cooling channels so that the water against the wall of the cooling channel is the same temperature as the water in the centre of the channel. If there is a noticeable difference in the inlet temperature and the outlet temperature, the flow is not adequate [8].

Are the feed lines to the mould large enough? If a mould has cooling channels that are larger than the inside diameter of the feed lines or fittings, the cooling flow is being choked and the mould cooling is inadequate [4].

In critical applications, thermostatically controlled water may be required on each cooling zone.

2.3.3. Filling, Packing, and Holding Pressures

Both higher melt temperatures and higher mould temperatures cause higher shrinkage; the influence of mould temperature is generally the greater of the two, since it usually may be varied over a greater range. But injection and holding pressures and time also have a significant influence on shrinkage. If injection or holding time and/or pressure are increased within limits imposed by machine pressure and clamping capabilities, the shrinkage decreases.

Any of the following will tend to lower shrinkage in polypropylene (and most other plastics as well) and may be used in combination with other options [10]:

- A plastic with a high melt flow index
- A plastic with controlled rheology
- An un-nucleated plastic
- Increase the injection pressure
- Raise the holding pressure
- Extend the injection (hold) time
- Decrease the mould temperature

Effective pressure in the cavity will vary with melt uniformity, melt temperature, and mould temperature. Uniform cavity pressure from cycle to cycle is required for constant shrinkage. Moulding-machine injection pressures may vary because of machine wear or moulding machine hydraulic-oil temperature variation caused by inadequate cooling [11].

Figure 2.1 shows a typical cavity-pressure trace that indicates the pressure in the cavity during a typical moulding cycle. Initially, there is no pressure in the cavity until the plastic flow-front passes the pressure-measuring transducer. Then the pressure increases as the flow front moves past the transducer, and more pressure is required to move the flow front as it moves away from the transducer [12].

When the cavity is full, there is a rapid rise in pressure as the plastic in the cavity is compressed during the packing phase. At the end of the packing phase, the pressure on the plastic is reduced for the duration of the holding phase. The rapid drop in pressure early in the holding phase is a result of the programmed machine-pressure drop. Then, as the plastic cools and becomes more viscous, the pressure at the transducer drops gradually because the holding pressure is not adequate to overcome viscous friction and maintain a constant pressure throughout the cavity. The position of the transducer relative to the gate affects the slope of the pressure gradient in this phase. The nearer to the gate the transducer is, the more constant the cavity pressure will appear to be. If the transducer is remote from the gate, the cavity pressure will drop more rapidly.

When the gate freezes, no more plastic can enter the cavity and the pressure drop is more rapid. When the shrinkage exceeds the compression on the plastic the cavity pressure drops to zero. After this point, the in-mould shrinkage causes the part to become smaller than the cavity. As long as there was positive pressure in the cavity, the part was potentially larger than the cavity. Finally, when the part has cooled enough to be structurally sound, the mould is opened and the part is removed [12].

Process variables such as the magnitude of the packing and holding pressures have a very significant effect on the shrinkage and final dimensions of a moulded part. If appropriate packing and holding pressures are not used, the volumetric shrinkage of a plastic material can reach as much as 25% [12]. Holding pressures must be high enough to compensate for shrinkage, yet low enough to avoid over packing, which can lead to high levels of residual stress and ejection difficulties.

2.3.4. Filling, Packing, and Holding Times

The filling and packing time must be sufficient to allow the plastic to reach the furthest extremities of the cavity and pressurize those areas to ensure minimum shrink there. The holding time must exceed the time required for the gate to freeze to avoid losing cavity pressure through the gate. The holding pressure is usually lower than the packing pressure to reduce the pressure gradient across the cavity, that is, to allow the region near the gate to have a cavity pressure more nearly the same as the pressure remote from the gate. Figure 2.1 shows how the cavity pressure will vary with varies phases of cycle time.

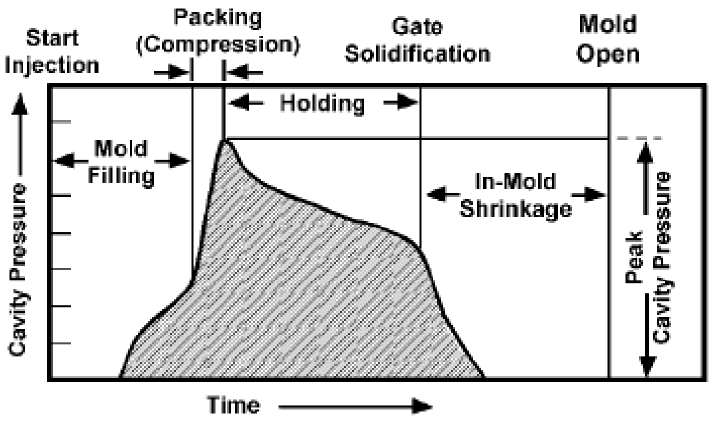


Figure 2.1 A typical Cavity-Pressure Trace

2.3.5. Part Temperature at Ejection

The part temperature at ejection must be low enough that the part will not re-melt or deform as it continues to cool out of the mould. On thick parts, it may be necessary to provide a cooling bath to keep the part from deforming [8].

2.3.6. Clamp Tonnage

The moulding machine must be able to hold the faces of the mould together with sufficient pressure to overcome the actual pressure in the projected area of the cavity perpendicular to the parting line. For example, if the projected area of the cavity and runner system was 10 square inches and the actual cavity pressure was 4,000 psi, then there would be a separating force at the parting line of 40,000 pounds or 20 tons [13]–[15]. The clamping force of the machine must exceed this separating force or the mould will open, the parting line will be damaged, and there will be flash on the part. Once flashing occurs, it will get worse and parting-line damage will increase.

A common rule-of-thumb is to select a machine that can develop at least 2½ tons (5,000 pounds) of clamping force per square inch of the projected cavity and runner area [4].

2.3.7. Post-Mould Fixturing and Annealing

The use of cooling fixtures is a last resort option. It involves extra expense to build the fixtures and extra labour to use them. It resists automation. It is more art than science. Parts must be restrained in such a manner that when cooled and released at room temperature, they are the desired size and shape.

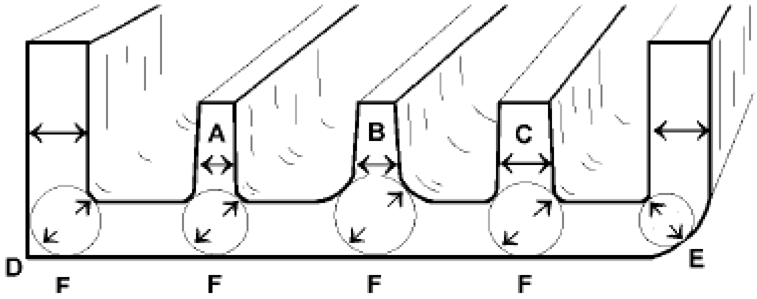
Usually, the parts have to be stressed using a weight or clamp during cooling so that they are held in a shape opposite to the undesired warpage. Thus when they are released they relax some of the frozen stress and assume the desired shape. However, if they are cooled in a fixture without annealing, they contain stresses that will eventually show themselves, after time and exposure to elevated temperature, by assuming some or the entire original undesired warp [16].

The relatively skinny core could not be cooled fast enough to maintain a temperature below that of the mould base around the outside of the part. The only way the warpage problem could be solved other than fixturing was to rebuild the mould, allowing for the inevitable warp. The in-use temperature was not excessive so post-mould stress relaxation was not a factor. A rail was built (based on trial and error) to spread the centre opening enough to make the side walls of the part parallel after the part was removed from the fixture rail. The thick walls required a long cycle so only a few parts were on the fixture at any one time [13].

2.3.8. Special Problems with Thick Walls and Sink Marks

Parts with thick wall sections are the most difficult to cool and pack. Thicker sections take longer to cool and require additional packing. When parts have both thick and thin sections, gating into the thick section is preferred because it enables packing of the thick section (provided the gates and runners are large enough), even if the thinner sections have solidified. The different cooling and packing requirements of the thick and thin sections lead to shrinkage-related internal stresses in the wall-thickness transition regions [16].

In practice, it is essentially impossible to maintain completely uniform part-wall thickness due to the complexity of part designs. As illustrated in Figure 2.2, design features such as bosses, flow leaders, or ribs result in local wall-thickness changes and, as a result, represent areas where cooling stresses can develop.



(A) Proper rib thickness and radius. (B) Excessively large radius.

(C) Excessively thick rib with proper radius. (D) Thick corner section due to square outside corner.

(E) Uniform wall thickness at corner because outside radius matches inside radius plus wall thickness.

(F) Potential areas for sink marks on the outside surface or voids in the center of the inscribed circles.

Figure 2.2 Good and Bad Wall-Thicknesses and Radius/Fillets[4]

Sink marks or voids are also common problems for parts containing reinforcing ribs on one side of the moulding. Thick ribs provide improved structural benefits and are easier to fill; however, the magnitude of sink associated with thick ribs can be excessive. The sink problem is magnified if large radii are used at the intersecting walls to reduce stress-concentration factors and improve flow. In practice, rib-wall thicknesses are typically 40% to 80% as great as the wall from which they extend, with base radius values from 25% to 40% of the wall thickness [17]. The specific rib designs are material dependent, and are influenced primarily by the shrinkage characteristics of the material.

When proper guidelines are followed, the size of the sink associated with a feature such as a rib is minimized, but some degree of sink will generally be noticeable. Localized mould cooling in the area of the sink mark can be beneficial in reducing the severity of the sink.

Various methods can be used to disguise the sink mark, as illustrated in Figure 2.3 one of the most common reasons that surface textures are used with injection-moulded plastic parts is to disguise aesthetic defects such as sink marks or weld lines. As a last resort in the fight against sink marks, moulders will sometimes add small quantities of a blowing agent to the base resin, and produce a conventional injection-moulded part with structural foam-like regions in the thicker section of the moulding (the sink is eliminated due to the internal foaming action). However, the blowing agent can create surface defects such as streaks or splay as the blowing agent creates bubbles on the surface of the moulded part. Maintaining a high air pressure in the mould during the filling phase can minimize the formation of surface bubbles.

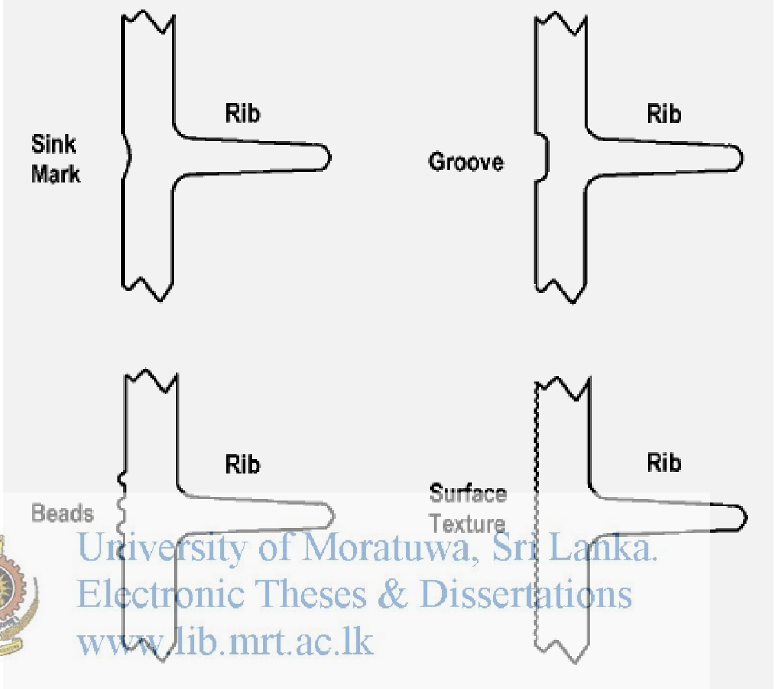


Figure 2.3 Methods of disguising sinks near heavy sections [4]

2.3.9. Nozzles

One often neglected topic in controlling shrinkage and warpage is the selection and use of nozzles at the interface between the mould and the heating cylinder. General-purpose (standard) nozzles, shown in Figure 2.4, are the most commonly used [17]. They are effectively full-bore until near the tip. A continuous-taper nozzle is shown in Figure 2.5. These encourage even flow without holdup. When materials tend toward drool, continuous-taper nozzles can help.

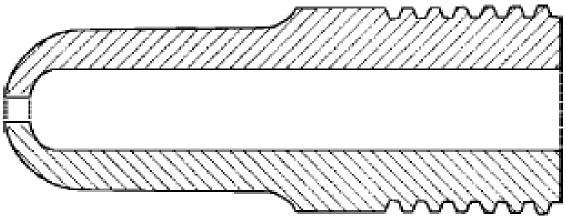


Figure 2.4 General-Purpose Nozzle [17]

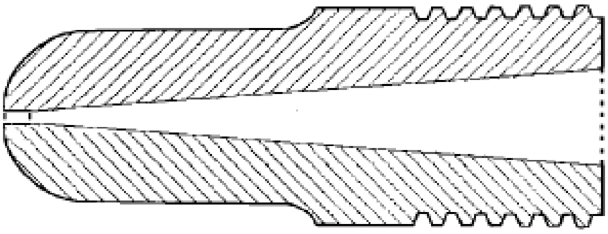


Figure 2.5 Continuous-Taper Nozzle [17]

The reverse-taper nozzle, as shown in Figure 2.6, is more commonly used with highly fluid materials like nylon, polyamides, acrylics, and similar expansive and heat sensitive materials. The sprue breaks inside the nozzle, providing expansion area and reducing drool. It has its minimum diameter near the centre of the nozzle. The minimum diameter of the nozzle must be large enough to allow adequate flow to fill the mould without undue shear-stress in the nozzle orifice. The heaters and thermocouple for the nozzle must be placed so that the temperature is as uniform as possible throughout the length of the nozzle. The controller for the nozzle should be proportional, as opposed to an off or on device, to maintain as constant a temperature as possible in the nozzle.

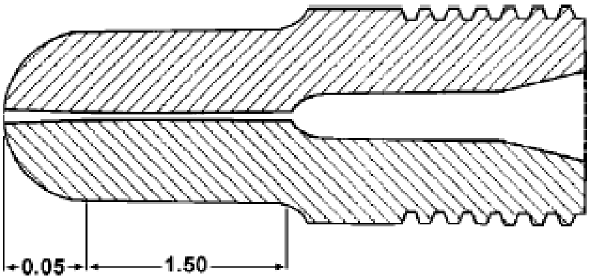


Figure 2.6 Reverse-Taper Type [17]

Important thing is the same nozzle size and type with the same size heaters in the same location and the same thermocouple location must be used each time the mould is run. All too often mould setup personnel do not change to the appropriate nozzle unless forced to. The end result is that a mould may be run with different nozzles from time to time. As a result, the moulding conditions are different. Instead of changing the nozzle, operators too often blame the material. When troubleshooting moulding problems, nozzles with very small diameters are often found feeding sprue bushings with diameters two or three times the nozzle diameter. This type of situation causes high shear heating, slow fill, and lower mould-cavity pressure relative to the machine injection-pressure setting [14].

2.3.10. Excessive or Insufficient Shrinkage

Excessive shrinkage occurs in moulded parts when the material is inadequately packed into the mould or when the melt temperature is too high. Inadequate packing, creating greater shrinkage, can result from low injection-pressures, low injection-speeds, short plunger forward times, or short clamp-time. Sometimes, however, high injection-pressures can cause excessive shrinkage by increasing the melt temperature due to the frictional heat generated [18]. High melt-temperatures cause the plastic to experience large temperature changes between the injection temperature and the temperature at which the parts can be ejected from the mould and the resulting large thermal contraction causes excessive shrinkage. However, under some combinations of conditions, an increase in melt temperature will increase the effective cavity-pressure, which will increase packing and result in a decrease in shrinkage.

Insufficient shrinkage will result if the injection pressure is too high, plunger-forward time is too long, clamp time is too long, injection speed is too fast, or melt temperature is too low. Injection pressure, injection speed, and cylinder temperature are interrelated and have a combined effect on cavity pressure and shrinkage. High injection-pressures and/or injection-speeds generate frictional heat, which increases melt temperatures and sometimes increases the shrinkage of the moulded item [18].

In plastics in general, and polyethylene in particular, shrinkage can be reduced by many means. All too often, customers strive for a less expensive part by using a lower quality or lower strength plastic or too low a mould temperature, which, in the long run, causes end user dissatisfaction and a bad name (again) for plastic. The cheapest price is not always the best bargain [4].

2.3.11. Secondary Machining

If a part that is essentially flat is machined over a significant portion of its flat surface, the machining operation removes some of the surface material that is in compression. The surface compression is a natural result of the surface of a moulded part cooling sooner than the core of the part. When the material in compression is removed, the centre of the part, which is in tension, is moved closer to the finished surface. This causes a tendency for the part to bow concave toward the machined surface. Figure 2.7 shows how the compressive stress in the surface of a part is machined away, and the distribution of stresses is changed [19].

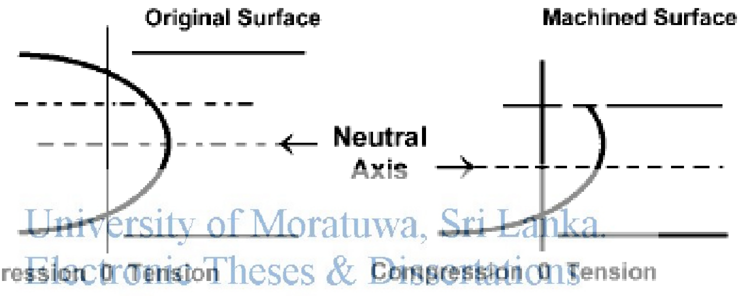


Figure 2.7 The moulded-in stresses are affected by secondary machining [19]

2.3.12. Quality Control

There are many factors that are under the control of the moulder. Some of these are the injection pressures at various times during the cycle, the time that the pressures are applied, the injection rates, the plastic material, and the mould temperature [20]. Figure 2.8 shows a schematic of a system that monitors some of these variables. This type of system can be a closed loop system to change machine settings if the system detects unauthorized changes.

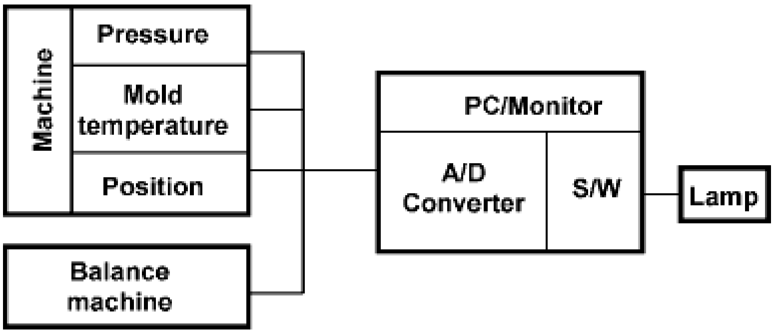


Figure 2.8 Schematic of a Quality Monitoring System [20]

This type of closed-loop system improves the quality and consistency of moulded parts, but does not guarantee the quality of the finished product. Since moulded parts continue to shrink over time, and the majority of that shrinkage occurs over the first forty-eight hours after moulding, one cannot reliably determine that a part is satisfactory until the part has been examined at least two days after it is moulded. Since it is possible to mould thousands of parts in some cases over a 48-hour period, some immediate indication of quality must be used [4]. Some of the indirectly controlled measurements are the weight of the finished part, the maximum cavity pressure measured at a particular point in the cavity, the cavity pressure at the end of the holding cycle, the time required for the pressure in the cavity to reach the maximum, and the time at which the cavity pressure reaches zero. Several directly controlled parameters affect each of these indirectly controlled variables. Some of these indirectly controlled measurements are more closely correlated to the quality of the finished part. A study done by B. H. Min among others has determined that the highest correlation between shrinkage and the quality of the finished part is the weight of the finished part. In other words, if two parts weigh the same and one part is known to be good, the likelihood that the other part is good is greater than 91% [4].

The next highest correlation between two acceptable parts is in the maximum cavity pressure measured during the moulding cycle for the two parts. If two parts are moulded with the same peak cavity pressure and one of the two parts is known to be good, then the likelihood that both are good is better than 84% [4]. Since both of these variables can be measured at the time a part is moulded, they provide the quality-assurance personnel a method to immediately determine if a moulded part is satisfactory.

If both weight and maximum cavity pressure are within limits for a given part, it is virtually certain that the parts are acceptable. For maximum quality assurance, mould sample parts at a variety of weights and maximum cavity pressures and after forty-eight hours determine which of these parts meet quality requirements. Then any parts that are moulded that fall within the established limits are good. Figure 2.9 shows the relationship between allowable tolerance limits and the range of indirectly controlled parameters.

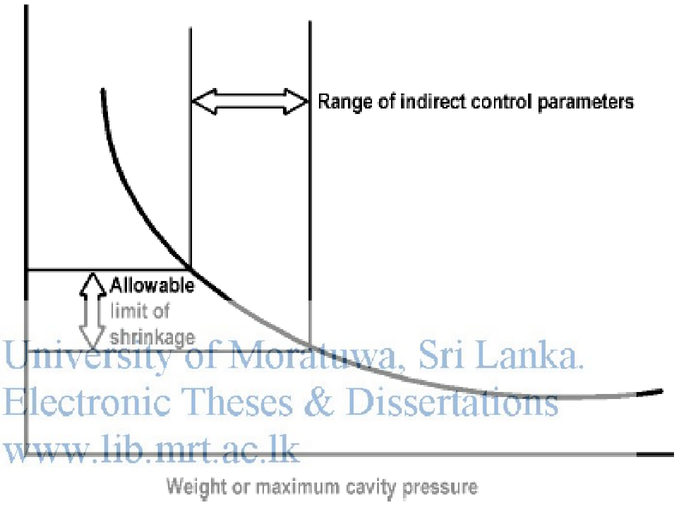


Figure 2.9 Quality-control relationship [4]

2.4. Material Considerations

There are a hundred or so commercial generic plastics and more than 41,000 grades [20]. It is very difficult to control shrinkage and warpage, and consequently the dimensions, of a part made of a semi crystalline plastic than one made of an amorphous plastic. Amorphous plastics have lower and more uniform shrink rates than do semi crystalline plastics. If tight tolerances and minimum warpage are of primary concern, and if an amorphous plastic with the necessary physical properties can be found, then it should be the preferred choice.

The injection-moulding process is generally used to produce parts that require fairly tight dimensional tolerances. In some cases very tight tolerances are required. For example, moulded plastic parts that must mate with other parts to produce an assembly must be moulded to accurate dimensional specifications. Many plastic materials exhibit relatively large mould-shrinkage values, and unfortunately, mould shrinkage is not always isotropic in nature. If a plastic material exhibits anisotropic mould-shrinkage behaviour, establishing cavity dimensions is no longer a simple “scale up” procedure. In addition, anisotropic shrinkage will lead to a degree of warpage (out-of-plane distortion) or internal stress [19].

Where close tolerance and stability are a concern, the coefficient of thermal expansion must be considered. Some applications depend on different coefficients of thermal expansion in order to perform their function, even with metal materials. A common example is the bimetallic spring in home thermostats. As temperatures change, the thermostat spring coils tighter or uncoils to open or close a mercury switch to start the heating or cooling cycle as appropriate. When parts with tight tolerances must operate over a wide range of temperatures, the materials used must have compatible coefficients of thermal expansion. If not, parts can come apart or break as a result of temperature-induced size change and stress. The plastic chosen for an application must be compatible with the end-use temperature range for the expected stress loads [19].

In some respects, mould shrinkage can be compared to linear thermal contraction or expansion. A mass of molten polymer cooling in a mould contracts as the temperature drops. Holding pressure is used to minimize shrinkage, but is only effective as long as the gate(s) remains open. If the polymer is homogeneous, all parts should shrink essentially the same amount even after the pressure is removed or the gates freeze. This generally is the case with amorphous polymers such as polystyrene, polycarbonate, ABS, etc. Published values for mould shrinkage of these materials is very low and do not exhibit a broad range. Generally they are in the order of less than 0.010 units/unit [21]. Why are polypropylene, polyethylene, nylon, acetyl, etc., different? Unlike amorphous polymers, these semi crystalline resins are not homogeneous; they have a structure containing both amorphous and crystalline components. As these resins cool, a multitude of crystals form that are surrounded by amorphous regions. The crystalline regions shrink much more than the amorphous regions. This imbalance in shrinkage causes a net increase in shrinkage and introduces sensitivity to other moulding parameters, which have additional effects on the shrinkage.

Another factor influencing shrinkage is the viscos-elastic characteristic of high molecular-weight polymer melts. The long molecular-weight chains are literally stretched, and placed under tensile stress, as they fill the mould. As the stresses are relieved during cooling, the chains try to relax, analogous to stretching a rubber band and slowly letting it return to its original size. This relaxation also influences the shrinkage, especially in different flow directions. Both the average molecular weight and the molecular weight distribution are key material factors that influence this facet of mould shrinkage.

The relative proportion of crystalline to amorphous components changes shrinkage. This is a very critical variable with polyethylene, but is not as significant with polypropylene, as evidenced by the much narrower range of specific gravity, another property affected by the degree of crystallinity [22].

Strength may be an important factor. If so, consideration must be given to creep-characteristics. Will the plastic support the proposed load over long periods of time or will it gradually give way? Will the proposed part distort under load in such a manner that the product will become unsatisfactory over time? Closely related to strength is the heat-deflection temperature. This property gives an indication of the effect of heat on the plastic’s strength.

Chemical resistance is frequently important. Will the chemicals in the environment cause swelling or cracking? Remember that water is a chemical and many plastics, especially nylon, absorb significant amounts of water. If the size of the plastic part changes significantly due to chemical absorption, the part may fail or become unusable. Aromatic hydrocarbons, for example, attack many plastics such as polycarbonate [23].

Coefficient of friction can be important in gears or bearings where there is sliding contact. Acetyl and nylon have low coefficients of friction while others in a similar environment will wear quickly.

Toughness is indicated by various types of impact tests. When impact loads are expected, the impact ratings give an indication of toughness for comparison purposes between various plastics. Environmental variables can affect toughness. For example, nylon is typically much tougher after it has absorbed some water than it is dry. Typically, increasing toughness is accompanied by a reduction in rigidity [24].

Low shrinkage is usually desired for parts requiring low warpage and tight tolerances, although low shrinkage is often associated with plastics with high long-term creep. Electrical conductivity is important where the plastic must isolate electrical charges. In other cases, some conductivity is necessary to avoid the build-up of a static charge. Tensile modulus is a measure of the stiffness of a plastic part. Thermal conductivity may be important to help dissipate heat [5].

2.4.1. Filler or Reinforcement Content

Fibrous fillers cause amorphous plastics that are essentially isotropic in their shrinkage behaviour to become anisotropic [22]. The cross-flow shrink rate becomes greater than the flow-direction shrink. On the other hand, the addition of small amounts of fibrous reinforcement to a semi crystalline plastic can make it become more isotropic in its shrink behaviour. The addition of flake or particulate filler to semi crystalline plastics reduces the overall shrink-rate and improves the shrinkage predictability.

Flake or particulate fillers that have lubricating characteristics can be added to amorphous materials to make them more satisfactory for a wear or bearing application without creating anisotropic shrinkage behaviour.

2.4.2. Degree of Liquid Absorption

Different plastics absorb different liquids. The amount of liquid that a plastic will absorb and the effects of the liquid on the dimensions and the physical characteristics of a plastic part must be considered. If a part changes size considerably while absorbing a liquid, it can become unusable due to interference with an adjoining part. If the molecular structure of a plastic is attacked by a fluid or gas, the plastic may become brittle, crack, or even dissolve. If a plastic loses a fluid (such as a plasticizer that can leach out as a fluid or vapour) during use, it may be come unsatisfactory because it changes colour, shrinks, or becomes brittle and cracks [24].

2.4.3. Regrind

Shrinkage is affected by the amount of regrind used. Each time the material passes through the moulding machine, the material is degraded somewhat. If the percentage of regrind varies from time to time, the shrinkage and warpage will also vary. This is especially true of glass-fibre–reinforced plastics. Some glass fibres are broken each time the material is processed, and they are broken more when the material is reground in preparation for reuse [18].

2.5. Tooling Considerations

Simply making a void in the mould that is the size and shape of the part to be moulded plus the average predicted shrink is not adequate for making even a simple part. A competent mould builder and designer must consider many different things to adequately design a quality mould [25].

2.5.1. Gate Locations

Gate location is one of the more critical aspects of mould design. First of all, if the part has thickness variations, the gate must be placed to fill the thicker section first [4]. Then the mould designer must visualize the flow patterns from the gate throughout the mould, and use that visualization to predict any likely flow or shrinkage variations. If thickness variations are such that a thick area surrounds a thinner area, a void can form in the molten plastic in the thin area, trapping air and preventing the moulding of a complete part. Often this trapped air is compressed and heated by the compression to the point that the plastic around the void is burned, leaving a charred surface [26].

Multiple gates may be required to fill the part adequately with a minimum pressure drop across the moulded part. Where multiple gates are present, the flow pattern within the mould is more difficult to predict, but the mould designer must consider the total flow pattern, especially for anisotropic materials [27].

The use of many gates often gets around the problems of differential shrinkage that leads to warpage. With multiple gates, the flow length is cut down, and cavity pressures tend to be more uniform (therefore mould shrinkage is more uniform) since all areas of the part are then "near" the gate. Alternatively, if the appropriate shrinkage data is available, the cavity dimensions can be cut to compensate for the different shrinkage values, but that is not a common practice. That data is more often used to design the multiple gates layout [28].

Shrinkage data generated on larger, plaque-type test moulds with well-defined linear flow is preferred to that generated using the oversimplified, standard ASTM testing technique. Using these larger parts, materials suppliers can generate both inflow and cross flow shrinkage values close to and far away from the gate region.

2.5.2. Types and Sizes of Gates

Gate location may be influenced by the appearance of the moulded part [4]. Certain surfaces may be cosmetically important and a gate mark on these surfaces may be restricted or forbidden. Small gates are cosmetically desirable but usually increase the shrink of the moulded part. Where control of shrink is of paramount importance, larger gates must be used.

Where small gates direct the flow of plastic across a flat surface, there is likely to be a tendency to jet a thin stream of plastic across the surface. Later, plastic flow will fill in around the initial jet of material. This leaves an undesirable surface blemish showing the profile of the initial jet of material. To avoid jetting, the gate should direct the flow of plastic against a core pin or wall to cause the plastic to "puddle" immediately. Tab or fan gates discourage jetting and encourage "pudding." See an example of jetting in Figure 2.10.

Figure 2.11 shows the method of causing immediate pudding as plastic enters the mould cavity. As the cavity pressure builds, the core is pushed away from the plastic and into its retracted position, providing a wall in the retracted position for the completed part.

Tunnel gates are preferred by many moulders to automatically separate the part from the runner. This avoids secondary hand trimming and sorting of the runner system from the moulded parts. On the other hand, if the moulder is using robotic systems and is keeping each cavity separated from all the others, it may be desirable to select a gate that keeps the parts on the runner until the robot places the parts and they are separated from the runner with some sort of die. Good communication between the mould designer and the moulder is of utmost importance [29].

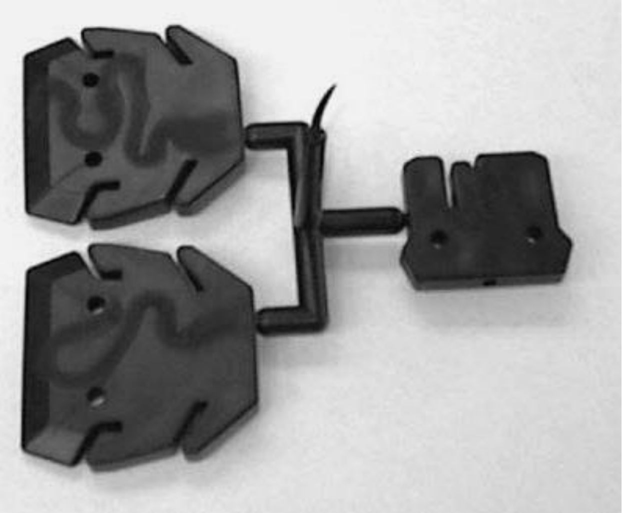


Figure 2.10 An example of jetting in an Injection Mould [4]

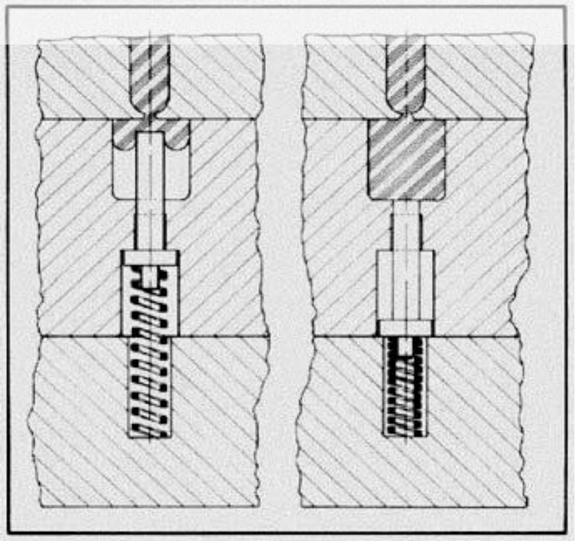


Figure 2.11 A movable core that inhibits jetting [4]

Gate size must be adequate to control shrinkage. For semi crystalline materials, gate size should be between 50% and 100% of the maximum part-thickness [4]. The larger the gate, the better control has on the part shrinkage.

2.5.3. Runner Systems

For minimum shrinkage in moulded parts, any runner between the moulded part and the moulding machine nozzle must be greater in its minimum dimension than the maximum thickness of the part being moulded [4]. Furthermore, the runner should increase in cross section toward the sprue at any intersection or abrupt change in direction. The size of the runner must be large enough that the runner remains fluid until after the part has solidified. If the runners are too small, then the runner solidifies before the part, causing higher shrink rates. On the other hand, if the runners are too large, then the cycle time must be extended far beyond what is necessary for the part to solidify so that the runners will not be molten when the mould opens [30].

In any multiple-cavity design where all cavities are identical, the runner system must be balanced so that the pressure drop and temperature distribution through the runner system is equal to each cavity gate. Runner design must strive to mix or distribute the shear heat in the runner so that all cavities receive material at the same temperature.

If the mould contains several cavities of different sizes, then a flow analysis should probably be made to ensure that each cavity fills at the same time. Runner size and gate size can be adjusted to achieve this goal.

2.5.4. Mould-Cooling Layout

One aspect often overlooked in mould design is the need for uniform filling and cooling. In a part having a complex geometry, even with relatively uniform wall thickness, it is not unusual to observe different shrinkage rates in different sections of the part. This may be due to non-uniform cooling and/or non-uniform filling patterns [31], [32], [33]. The use of computer analysis to study the filling and cooling pattern is a useful tool to identify these problems and provide guidance for their minimization or elimination [34].

Cooling channels must be arranged to remove heat in a manner so that the entire moulded part and runner system cool at the same rate. Where there are both thick and thin moulded-part sections, the cooling capacity of the system in the thick areas must be greater so that the thick sections cool at the same rate as the thin sections [35]. Core pins and outside corners of cores need special attention to maximize heat transfer into the cooling system. Heat pipes or high-conductivity material can be used to encourage better cooling [36], [37].

The runner system and gates, being of larger cross section, typically require extra cooling to bring their temperature down at the same rate as the thinner sections of moulded parts [38].

Processes are available through companies that permit the placement of cooling lines at a uniform distance from a profiled surface. Such systems are sometimes called conformable or conforming cooling, where the cooling channels conform to the profile of the part. This invention has brought injection moulding process into a new era. This method is capable of reducing the cooling time more than 40% of existing conventional cooling layouts. Also uniform cooling reduces warpage significantly [39], [40].

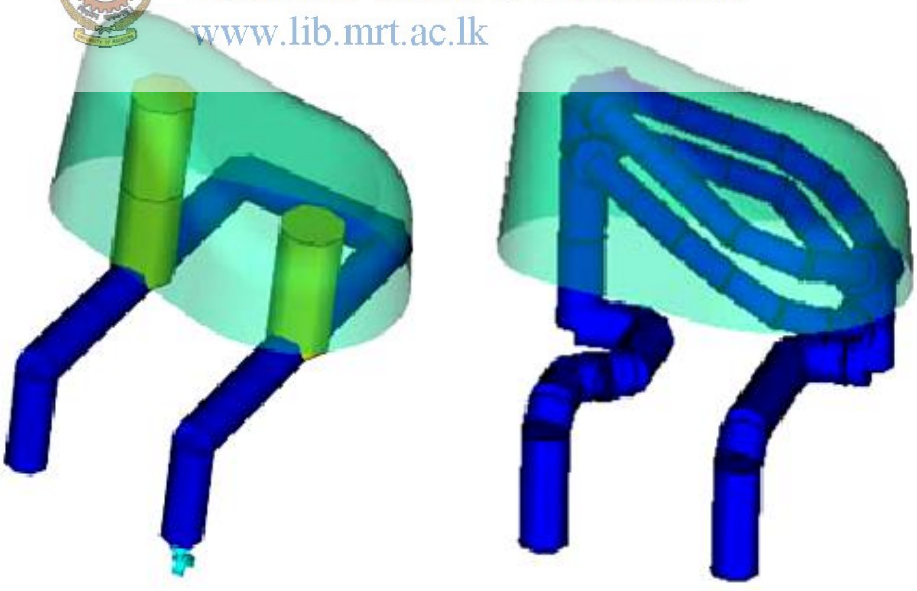


Figure 2.12 Conventional Core Cooling versus Conformal Core Cooling [41]

These processes create a part out of liquid, molten, or powdered polymer or metal. Parts are created by using lasers or ink-jet technology. In the CAD software, solids models are represented by their outside surfaces. In order for the CAD model to be used in an additive process, it has to be converted to a format that the rapid prototyping machine can understand. The files that represent the outside surfaces are called STL files. In this format, triangles represent the surfaces. In most machines this is done from the bottom of the part to the top, because the equipment to manufacture the prototypes is mounted on the top of the machine [42].

2.5.5. Tool Tolerances

The part designer and the end user must consider the inevitable variations in shrinkage and warpage of any moulded part of any type of plastic. The question is not, “Will the part shrink or warp?” The question is, “How much will it shrink and warp?” Furthermore, the manufacture of a moulded part includes two distinct and separate sets of tolerances: one for the moulding process and one for the manufacture of the mould (the mould builder). By far the larger tolerance is required for the moulder because of the lack of predictability and consistency in the moulding process as compared to the accuracy possible on modern machine tools.

Thus, some of the tolerance available for the moulded part is of necessity used by the mould builder. There is no such thing as a perfect mould or mould component. Some tolerance is always required when machining anything, even precision reference-blocks and gages (although in the latter case, the tolerance may be only a few millionths of an inch).

Typically, a mould builder will use as little of the total tolerance available for the moulded part as possible in building the mould. Normally the mould will be within 10% to 20% of the optimum size of the part, including the best estimate of the shrinkage for the plastic selected [43]. For example, if a part to be moulded of polycarbonate is one-eighth inch thick and six inches long, the expected shrink is from 0.005 to 0.007 units per unit of length. If the part is restrained from shrinking by cored holes or other restraining agents at the edges of the part, the shrink is likely being nearer 0.005 units per unit of length [43]. On the other hand, if the part is unrestrained and essentially flat, the shrink rate is more likely to be nearer 0.007

units per unit of length. Assuming the latter, a 6-inch-long part would require a mould that is 6 in. × 1.007 = 6.042 in. long. A reasonable tolerance for this length of a plastic part might be ± 0.008 in [43]. The mould builder would likely use no more than ± 0.001 inches. This does use up some of the tolerance, but moulder is left with most of the tolerance available for his use.

The tool designer can hold very tight tolerances in the manufacturing of the mould. However, neither the tool designer, moulder, a mould-filling analyst nor the material supplier can absolutely sure of the exact shrink-rate at any given location within a mould. While tool tolerances are tight, they are aimed at an assumed shrink rate. Sometimes the only way to hold extremely tight moulded-part tolerances is to build the mould twice. The first mould is a “best guess” for shrinkage prediction. This mould is then thoroughly analysed for shrinkage in every part of the mould. The second, rebuilt mould is based on the shrinkages actually observed in the first mould [44].

2.5.6. Draft Angles

Draft on surfaces that are perpendicular to the parting line of a mould is necessary. Walls that are parallel to the opening motion of a mould will cause scuffmarks on the part surface as the part slides past the mould-cavity surface during mould opening or ejection. When the part is moulded, the shrinkage through the thickness of the part is frequently so low that when the mould opens, the outside of the moulded part rubs against the cavity walls (shown in the figure by the arrows pointing out). When texture is present, the draft requirements are increased dramatically to allow the texture to slide free of the mould cavity as the mould opens and the part is ejected [45].

Draft on the mould core is important. In the first place, draft on the core allows easier ejection of the part from the core and reduces the number and size of ejectors necessary. If the draft is not sufficient to allow the part to unload the shrink stresses as it moves off the core, the last part of the core to exit the moulded part will scratch, scuff, or raise a burr on the open edge of the moulded part [46].

The plastic shrinks as the part is pushed off the core, relaxing these forces (stresses). This causes the sharp edge at the top of the core to scrape some plastic from the inside of the plastic part, producing some plastic dust or shavings. Some of these shavings may remain in the cored hole and others may remain in the mould to contaminate the next shot or cause damage to the mould face. Usually in this type of situation, the open edge of the cored hole is stretched or distorted, and a raised lip or burr is left around the hole [46].

2.5.7. Ejection-System Design

A typical mould is shown in Figure 2.13. The operating ejection section is shown toward the bottom of the figure (the ejector plate), with the return pins and sprue puller. This mechanism moves forward, carrying the ejection system, to press or strip the plastic parts from the mould. Figure 2.14 shows the cross section of a typical mould and one of several ejector pins in each cavity.

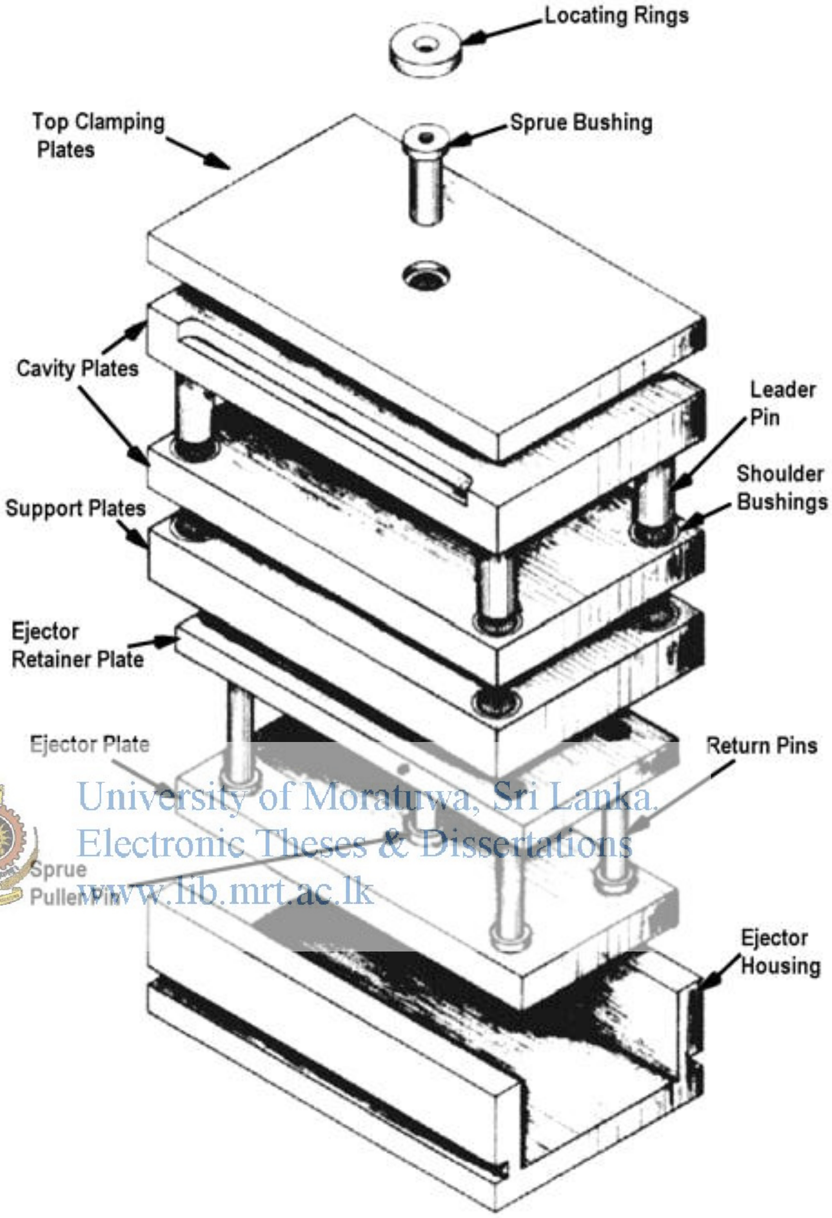


Figure 2.13 A Typical Mould Construction [4]

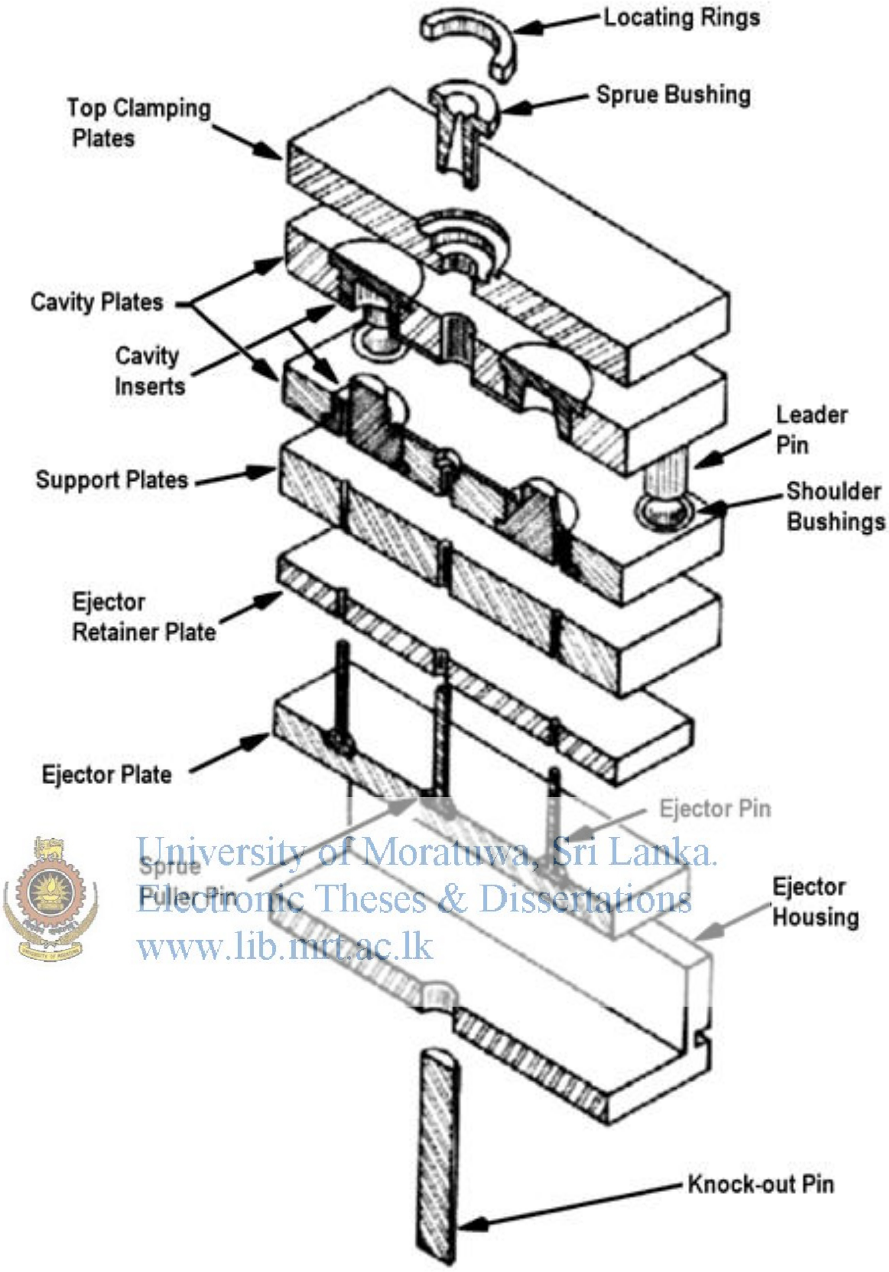


Figure 2.14 Cross Section of a Typical Two-Plated Injection Mould [**4**]

A number of ejection schemes are available, including, but not limited to, ejector pins or blades and stripper sleeves or plates, as shown in Figure 2.14, and special lifts that move away from the part while forming an undercut. The goal of the mould designer, from a shrink/warp standpoint, is to provide a sufficient number of ejection devices to remove the part from the mould without distorting the part in any way. If any portion of the moulded part sticks or lags behind the rest of the part as it is ejected, there is a potential for the moulded part to be stressed beyond its yield point, that is, bent or warped. The stripper plate design shown in Figure 2.15 is the type of ejection system that applies equal pressure around the periphery of a part to remove it from the mould. Often an air inlet is designed into the centre of the core to permit air to enter and reduce the force required to eject the part.

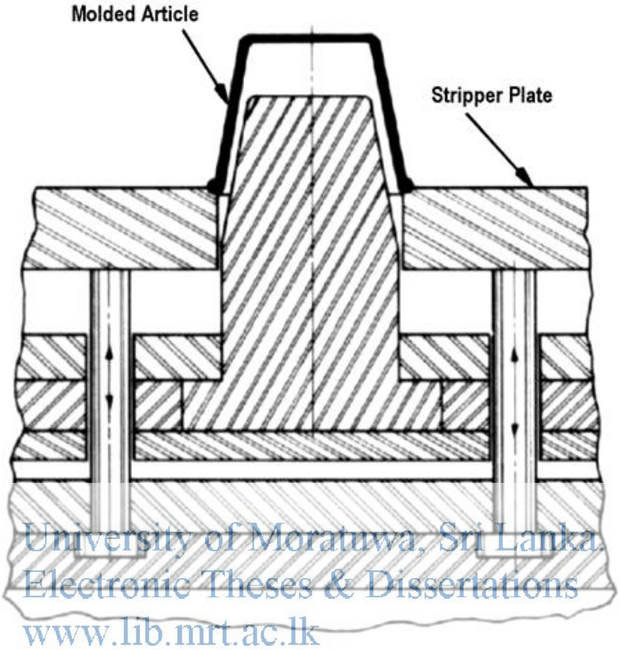


Figure 2.15 A stripper plate ejection assembly [4]

2.5.8. Elastic Deformation of a Mould

A mould must be manufactured with sufficient rigidity to resist the immense forces that attempt to open the mould or bend the mould plates. If a mould deflects a measurable amount, that deflection will show up in the moulded part. Usually the deflection causes an increase in part thickness and may be accompanied by flash around the part or over core pins that are intended to form through holes in the part. If the moulded part has side walls that form a deep bucket or boxlike shape, then inadequate mould rigidity may allow the mould plates to flex under injection pressure and allow the side walls of the moulded part to thicken or bow.

The mould may be designed with adequate strength to resist the internal pressure of the plastic without bending, but that is not adequate. It must resist the internal forces without measurable deflection [18]. Deflection calculations are often overlooked and are often beyond the knowledge and ability of a mould designer. The moulding machine itself may be a source of shrinkage problems. The platens on a moulding machine must be flat in order to support the mould over its entire surface. If the moulding-machine platens are damaged so that they are concave in the centre, no amount of mould rigidity can be depended upon to resist the opening forces generated by the pressure of the injected plastic. Distortions in moulding-machine platens have caused part thickness variations, mould flash, and even mould damage [47].

2.5.9. Mould Wear

When moulding plastics with abrasive fillers or glass fibre fillers the mould areas at or near the gate are subjected to high wear. This is especially true if the plastic entering the gate immediately impinges against a wall or a core pin. Sometimes areas at the end of the flow path are also subject to significant abrasive wear [44]. Mould builders often provide replaceable inserts in these areas. Variations due to wear in these areas do affect the part's dimensions. The softer the material used in mould construction, the more rapidly wear of this type can occur. Wear and impressions made when material is trapped between the mould faces as the mould closes under many tons of pressure can damage the parting line at the edge of the cavity. It is important that an appropriately hard material be used in the mould construction to avoid early failure of this type. Any variations in the parting line or any flash as a result of parting line impressions increase the apparent size of the part and soon lead to out-of-tolerance parts [48].

2.5.10. Mould Contamination

Deposits on mould surfaces can come from a number of different sources. If the part design and mould design are such that excessively high melt temperatures are necessary to fill the part, moulder may find that some degradation of the plastic material takes place which can deposit plastic decomposition products on the surface of the mould [49].

If the mould is not adequately vented, air pressure in the mould builds up as the cavity fills. It is a principle of physics that as pressure builds rapidly on a fixed weight of a gas (air); the temperature of that gas rises dramatically. This is essentially what happens in a diesel engine to ignite the fuel. In an injection mould, the pressures can increase to the point that the leading edge of the plastic material ignites. This usually leaves a dark deposit in the mould at the last point to fill, and leaves a burned spot on the moulded part. If the venting is marginal, the part may not show a burned area, yet products of decomposition will accumulate in the mould in the region of the last area to fill [49].

The high amounts of fillers such as flame retardants, lubricants, pigments, impact modifiers, etc., that are required in some applications often bleed out of the moulded part in tiny amounts that accumulate in the mould. After a while they build up a film of measurable thickness. Such deposits reduce the apparent size of the mould and the moulded product [4].

High shear-rates caused by too small a gate or too high an injection pressure contribute to degradation of the plastic and the separation of fillers. The deposits tend to bond to the mould surfaces that are hottest, such as core pins, inside corners, and any area where air is trapped. If the vents are barely adequate, sometimes the deposits will build up in the vents themselves, aggravating the problem.

Excessive heat-time history such as might be experienced in hot-runner moulds or when small parts are being moulded on machines with large shot capacity, sometimes causes degradation products. When moulding shear-sensitive plastics, use generously sized runners and gates. Sometimes multiple gates will help with shear-sensitive materials. Use an adequate number and size of vents.

Whatever the cause of the mould deposits, they eventually affect the dimension of the moulded part. The first line of defence is to adjust the moulding conditions or modify the mould to eliminate the cause of the deposits. If that is not possible, then the deposits should be removed before they build up any significant thickness. The thicker they are, the harder they are to remove without potential mould damage. On highly polished moulds, the best approach is to find a solvent that will not attack the mould surface. Such diverse products as oven sprays and lemonade with caffeine have worked. Cryogenic blasting may be a good way to remove deposits.

Commercial mould-cleaning sprays often work. If a solvent cannot be found, then the mildest possible abrasive may be necessary. In a polished mould, only a trained mould polisher can safely use abrasives [49].

2.5.11. Position Deviations of Movable Mould Components

Movable components are part of every mould, and they may be subject to positioning variations. Even the simplest mould has moving parts. The two halves of the mould are aligned by leader pins or by parting-line locks. There must be some clearance for these components to slide with respect to one another. Therefore, they may shift from side to side within the clearance provided from one shot to the next. Core pins within sleeve ejectors have clearances between the core pin and the sleeve, and between the sleeve and the mould. Each of these clearances allows some shift in the position of the core pin from shot to shot. Slide components that form side holes or undercuts have clearances to allow them to move freely. Each time the mould cycles, the slide can move within the clearance envelope so that it is positioned differently each time the mould is closed. Injection-pressure variations can cause mould deflection that affects the positioning of slides and cores and the thickness of the moulded part. Each of these potential variations is quite small nevertheless; they are measurable and can be significant in moulded parts with tight tolerances [30].

2.6. Part Geometry

Section thickness variations are quite common in designs from inexperienced designers. Another common problem is a design with excessively close or unrealistic tolerances. Inexperienced designers apply unnecessary and unrealistic tolerances to the dimensions of a plastic part. Creep failure of plastic parts is another common problem often overlooked by designers [4]. Moulder and mould builder can save their customer untold dollars and the customer’s reputation if they can council their customer to avoid creep failure [50].

The earlier moulder and mould builder get involved in the design process, the more likely endues customer is to accept changes to the part design. Most of the time, end users are open to design suggestions provided they do not compromise the general appearance and function of the part. Potential problems should be cited no later than when the part or mould is quoted, and solutions should be offered at that time. Possible solutions may include design changes or material changes to resolve the problem. If the problems cannot be resolved, it is better to decline the project. It is never a good idea to approach the customer with sample parts from the mould and say, “Oh, by the way, we can’t mould the parts to print” [45].

2.6.1. Overall Part Dimensions

Overall tolerances and dimensions of a moulded part are frequently designed too tightly. Consider this common situation. The designer selects a material with published shrink rates of 1.5% to 3%. He then designs a plastic part that is 100 mm long and specifies a length tolerance of ±0.1 mm. The published shrink data indicates that under normal moulding conditions, a 3-mm thick tensile test bar may vary as much as 1.5%. Therefore, the 100 mm long dimension may vary as much as 1.5 mm under normal moulding conditions. That is 15 times the tolerance specified above [51].

In this situation, the designer needs to review the tolerance requirements to see if they really need to be so tight. If they do, then he should specify a different material with a lower and more predictable shrink rate and/or redesign the part to allow greater latitude in the tolerances. Unrealistic tolerance specifications lead to excessive rejects, high part-costs, and general conflict between moulder and the customer.

2.6.2. Wall Thickness

The wall thickness of a plastic part should be no greater than necessary to provide structural integrity and to provide adequate thickness for the plastic to flow easily into the most remote corners and details. Too thin a part will narrow the process window available to moulder, which in turn will increase the likelihood of rejects and will lead to price increases. Too thick a part will also lead to price increases because the cycle time will be greater than necessary and the quantity of plastic in the part will be more than is needed. The thickness of a plastic part should be as uniform as possible to avoid moulded-in stresses, warpage, anisotropic shrinkage, and excessive cycle time [4]. Where parts do require different wall thicknesses, some design options are available for minimizing shrinkage problems. Figure 2.16 illustrates wall thickness transitions, from poor to best, for a part designed with different wall thicknesses. Note that the best design has a tapered section between thick and thin sections at least three times as long as the material is thick. Figure 2.17 shows another example of a part designed with non-uniform wall thickness, one given to asymmetrical shrinkage. The thicker section shrinks more than the thinner. For a part of this design type, the asymmetrical shrinkage can be corrected by ribbing the thick section or by making the thickness uniform [52].

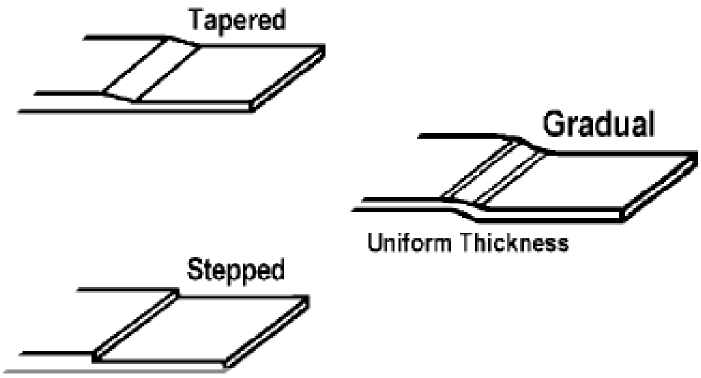


Figure 2.16 Changes in section thickness [4].

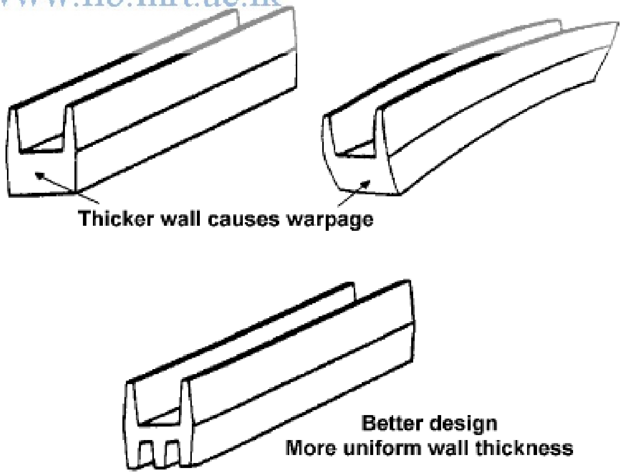


Figure 2.17 Non uniform wall thickness [4]

Wall thickness problems can become excessive when features such as bosses are incorporated into the Side wall of the moulding [30]. The excessive thickness is likely to cause the formation of sink marks or shrinkage voids. Sinks form when the walls are not sufficiently strong to resist the negative pressure caused by shrinkage of the thick section. Voids form when the solid skin is strong enough to withstand the negative pressure that builds as the polymer melt cools and shrinks without compensation. Sink marks are undesirable from an aesthetic point of view, while shrinkage voids are discontinuities that act as stress concentration areas during end-use loading. Voids are also aesthetic defects for transparent or translucent parts. Figure 2.18 and Figure 2.19 illustrate correct and incorrect boss designs for the control of sink marks.

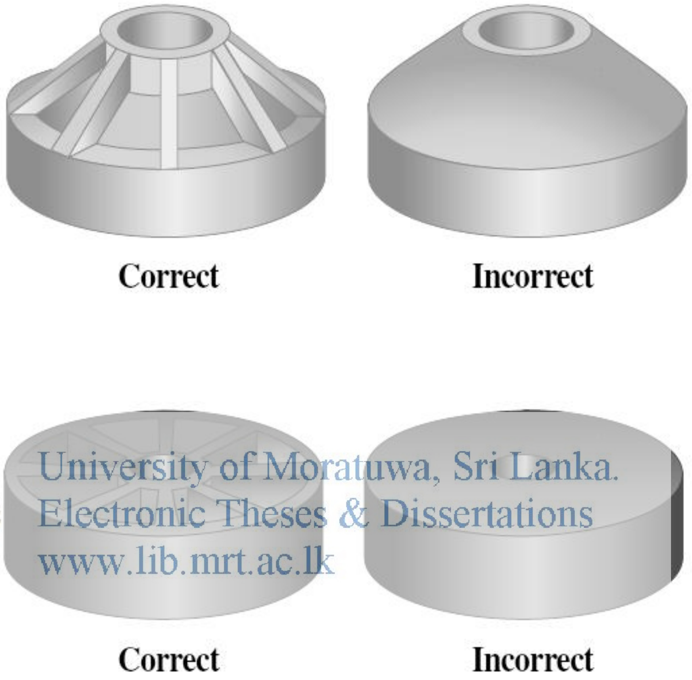


Figure 2.18 Avoid Thickness Variations

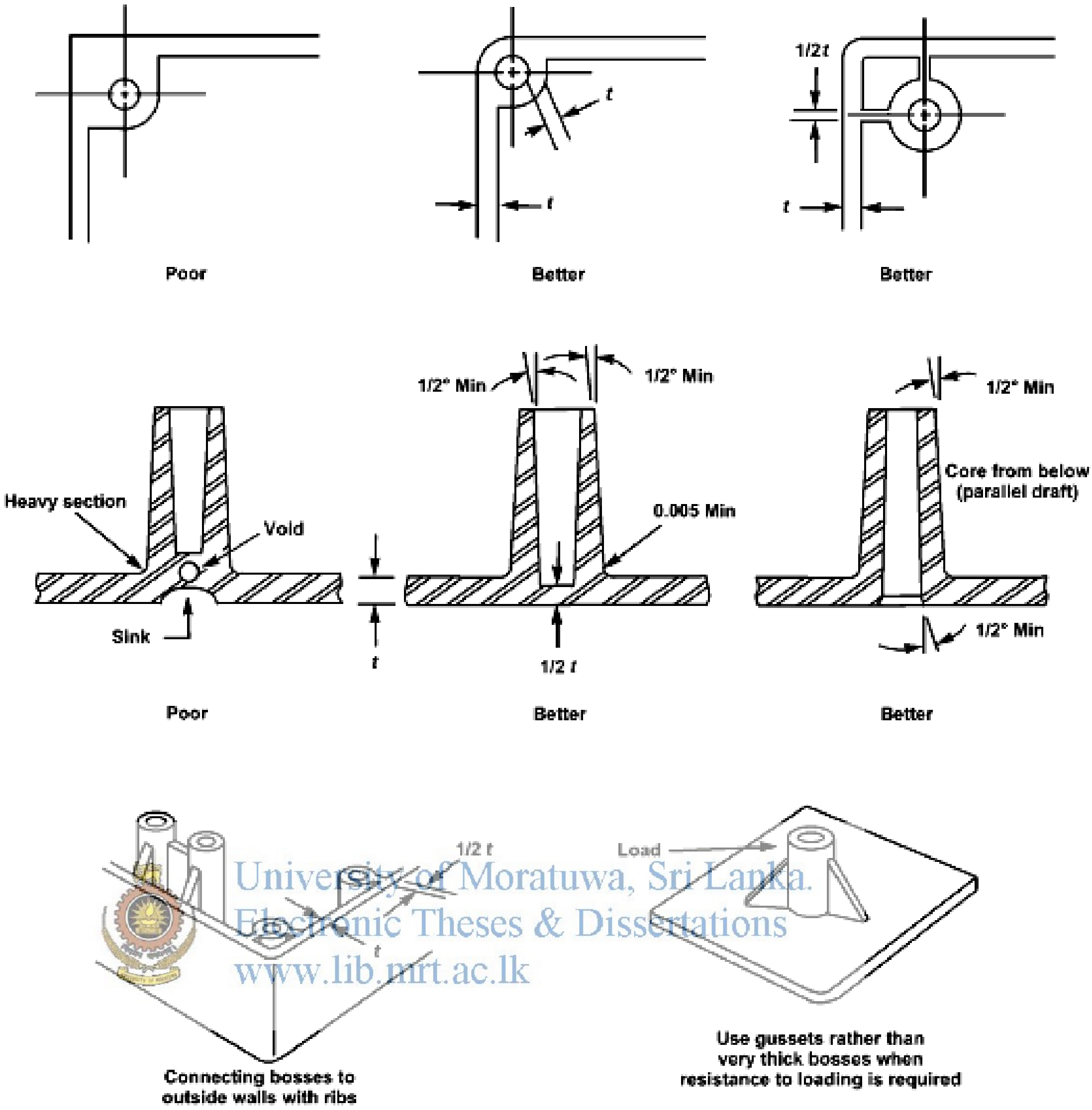


Figure 2.19 Incorrect Boss Designs and Correct Boss Designs [4]

2.6.3. Sharp Corners

Other important factor which effect for sink marks and warpage is sharp corners. Also sharp corners will increases more ejector pins. Figure 2.20 shows an example for how prevent sharp edges at product design the ejection force required and ejector pin marks on the product. Then the ejector system requires.

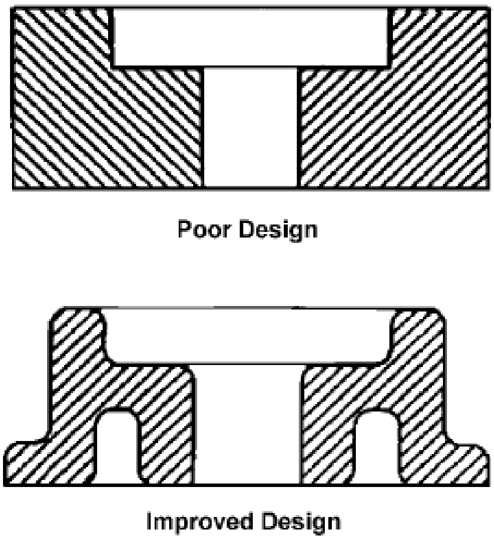


Figure 2.20 Design for Uniform Thickness and corner with Radius [9].

2.7. Comparison of Factors

After evaluating the results obtained from researches at the literature review, factors that effects for warpage can be classify as shown in Table 2.1 [2], [4]–[6], [51]. There may be deviations of the level of effectiveness of these factors for special products [4].

Table 2.1 Classification of factors affecting for warpage

Level of Effectiveness	Factor
High	Cooling layout
	Gate Location
	Part Geometry
	Runner systems
	Filling, Packing, and Holding Pressures
	Filling, Packing, and Holding times
Low	Types and Sizes of Gates
	Melt Temperatures and Uniformity
	Mould Temperatures and Uniformity
	Part Temperature at Ejection
	Clamp Tonnage
	Post-Mould Fixturing and Annealing
	Special Problems with Thick walls and Sink Marks
	Nozzles
	Excessive or Insufficient shrinkage
	Secondary Machining
	Quality Control
	Filler or Reinforcement Content
	Degree of Liquid Absorption
	Regrind
	Tool Tolerances
	Draft angles
	Ejection-System Design
	Elastic Deformation of a Mould
	Mould wear
	Mould Contamination
	Position Deviations of Movable Mould Components

3 METHODOLOGY

3.1. Identification of Main Features

There are no easy solutions to eliminate warpage, but with careful consideration of the factors contributing to warpage, many potential pitfalls may be avoided. Since there is large number of factors effecting for warpage, following are the selected major factors which are to be discussed under this research.

- Part Geometry
- Gate Location
- Runner systems
- Filling, packing/ holding Pressures
- Filling, packing/ holding times
- Cooling layout

3.2. Process Modifications

This study will lead to more improved injection moulding process with quality product by minimum modifications and minimum cost. The research plan is as follows,

1. Select the product.
2. Investigate the warpage and how it affects for the functions of product assembly.
3. Identify possible reasons for warpage from selected factors.
4. Analyse the situation by varying the parameters with the Autodesk Moldflow Adviser and Solidworks Plastics software.
5. Discuss the various options and decide the modifications.
6. Determine optimum parameters to reduce warpage and do the design modifications with Siemens NX 9.0.
7. Sample production and comparison of the results with software analysis.
8. Discuss the future improvements.

The main aim of the plastic manufacturers is to deliver parts at low costs, with a short delivery time, and with required quality. The quality can be defined differently depending on the usage of the product, but one important issue for the manufacturers today is the warpage of the final plastic products since they are often parts of a system assembled together.

The selected part to analyse the effect of the warpage is a part related to a solar panel which is called as a hanger (Figure 3.1). Warpage of this part will cause the failure of the assembly. As this part is moving within the assembly, effect of warpage is relatively high.

Figure 3.1 Selected Product (Hanger)

Product assembly is shown in Figure 3.2 and assembly drawing of the mould is shown in appendix A.

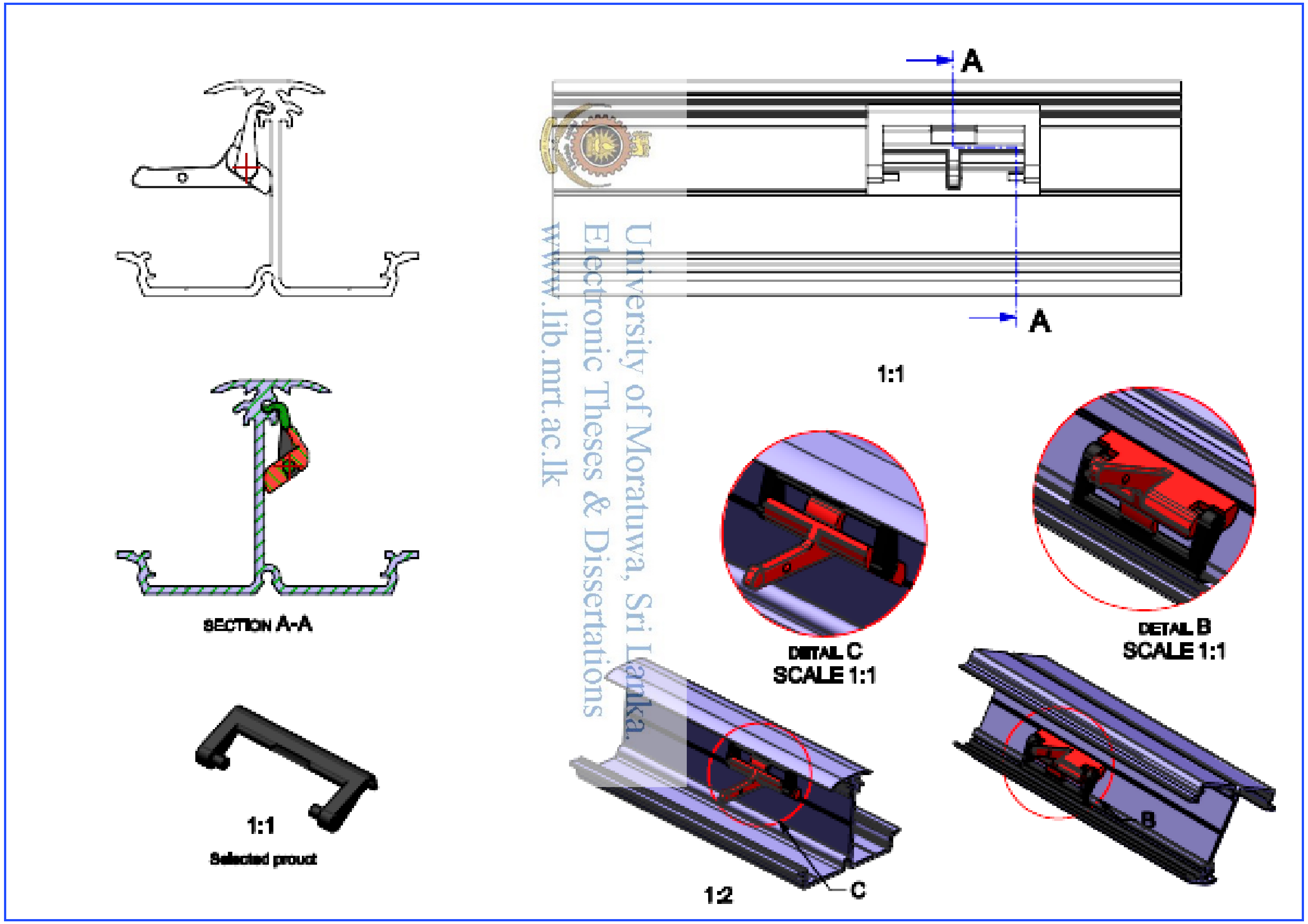


Figure 3.2 Product Assembly

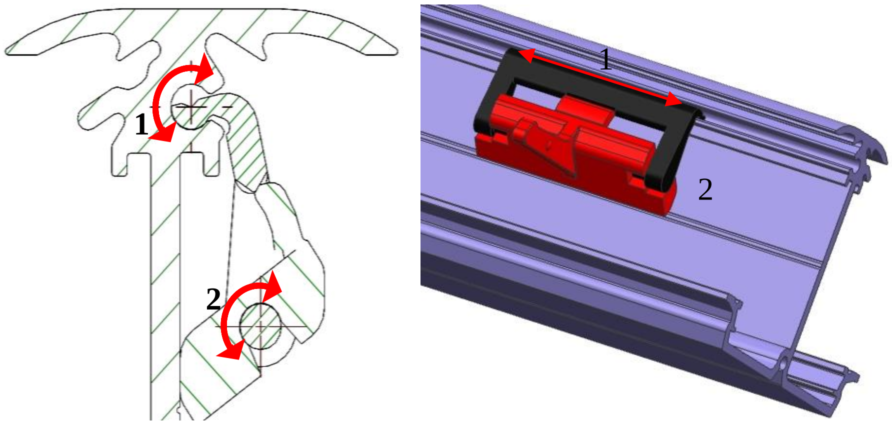


Figure 3.3 Type of Joints

Joint no 1 which is shown in Figure 3.3 is a roller joint and the joint no 2 is a hinge.

3.2.1. Investigation of Warpage

When analysing this selected part, warpage minimization is a very significant factor as product assembly is not functioning as expected in the product design. The deformation occurred in this product in each direction has measured and shown Figure 3.5 and Figure 3.6.

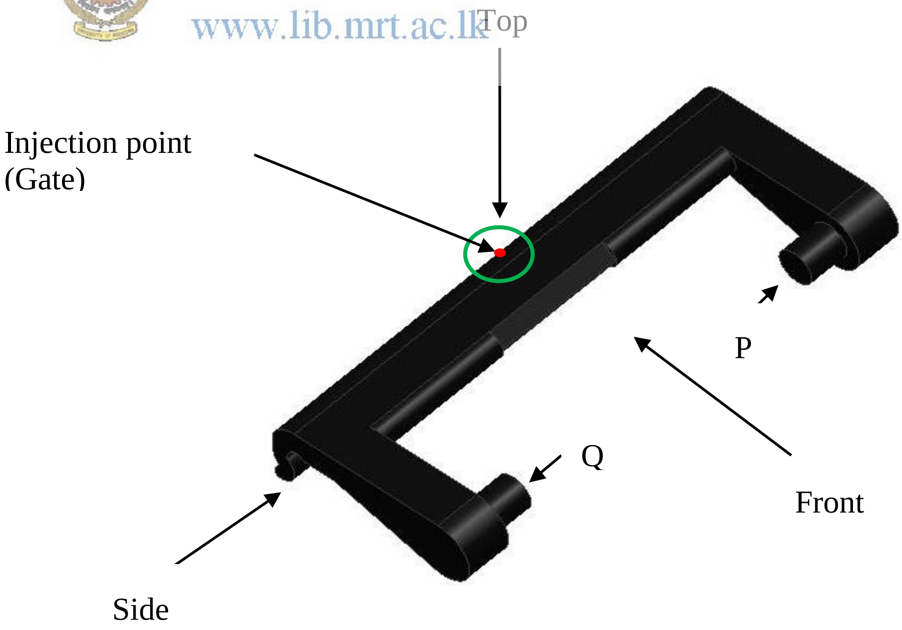


Figure 3.4 Product (Isometric View)

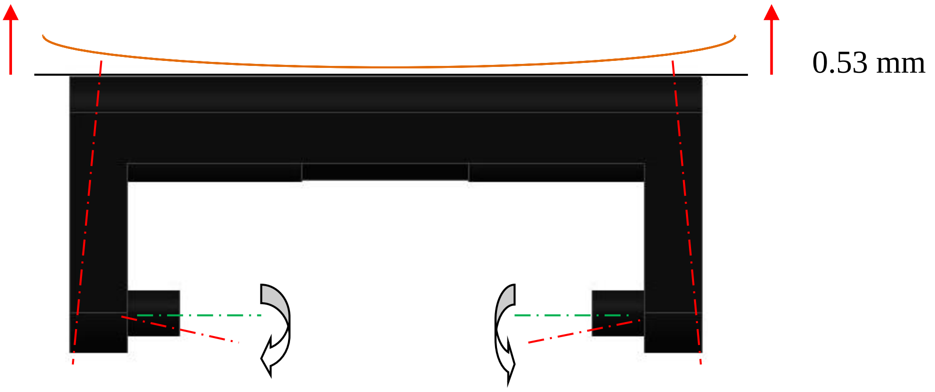


Figure 3.5 Top view

To obtain a justify value, ten samples were measured and the average value was calculated. Measurements were obtained by using CMM.

Figure 3.6 Front View

Figure 3.7 Side View

According to the design, the hinges should be collinear. But it shows two way deviations in top view and front view. Therefore the warpage causes difficulty in assembling and obstructs the smooth operation of the hinge. These Conditions are applicable for the roller joint also. The maximum permissible tolerance for both hinge and roller joint is 0.2 mm in either way. But calculated averages value highly exceed the permissible value.

3.3. Data Analysis

Analysis was done using following two softwares.

1. Autodesk Moldflow Adviser
2. Solid Works Plastics

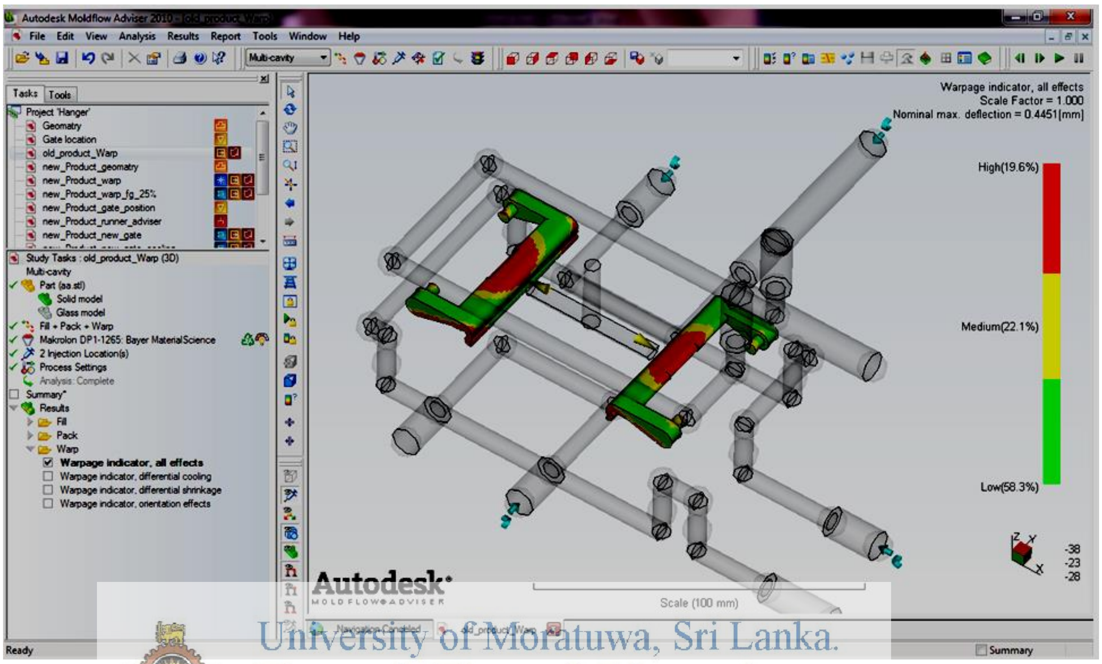
Table 3.1 shows the current processing parameters.

Table 3.1 Processing Parameters

Parameter	Value
Material	PC
Part volume	2.8 cm^3
Shot weight	35.03g
Article weight	25.13g
Melt temperature	295 [^{0}C]
Injection Speed	85 cm^3/s
Injection Pressure	138 bar
Injection / fill time	4 S
Holding Time	5 S
Mould Temperature	80 [^{0}C]
Cooling Time	20 S
Mould open Time	1 S
Mould close Time	2 S
Ejection time	2 S
Holding pressure	60 bar
Coolant Temperature	20 [^{0}C]
Ambient Temperature	30 [^{0}C]

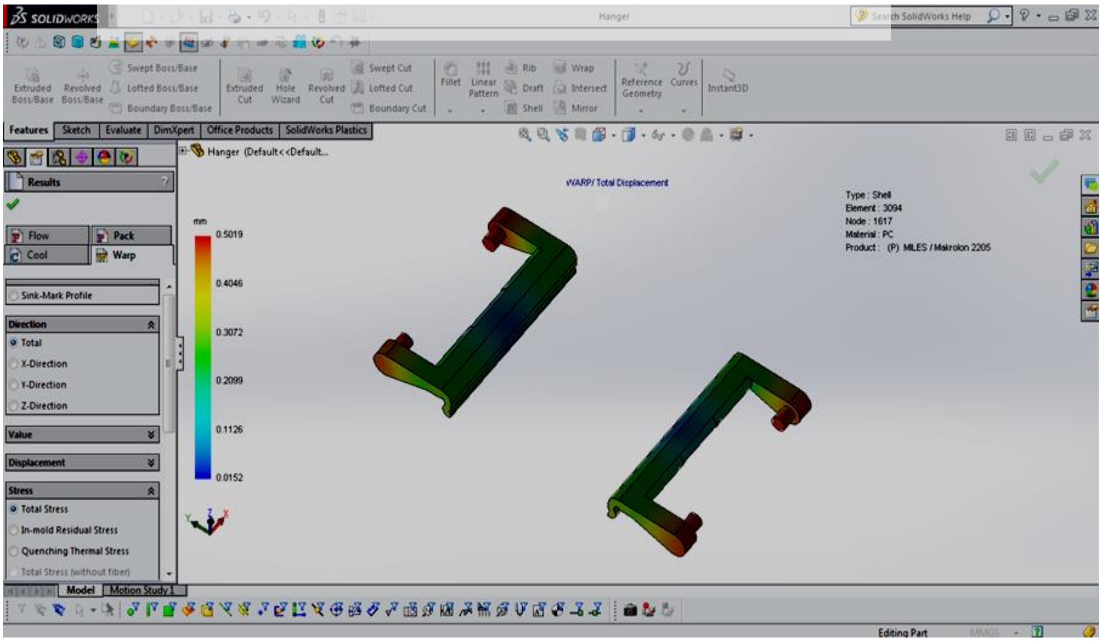
3.3.1. Part Geometry

The part was analysed against the deflection occurred due to warpage. Figure 3.8 and Figure 3.9 show the analytical result with Autodesk Moldflow adviser and Solid works plastics respectively.



Warpage indicator, all effects
Scale Factor = 1.000
Nominal max. deflection = 0.4451[mm]

Figure 3.8 Warpage (AMA)



WARP/ Total Displacement

Type : Shell
Element : 3094
Node : 1617
Material : PC
Product : (P) MILES / Makrolon 2205

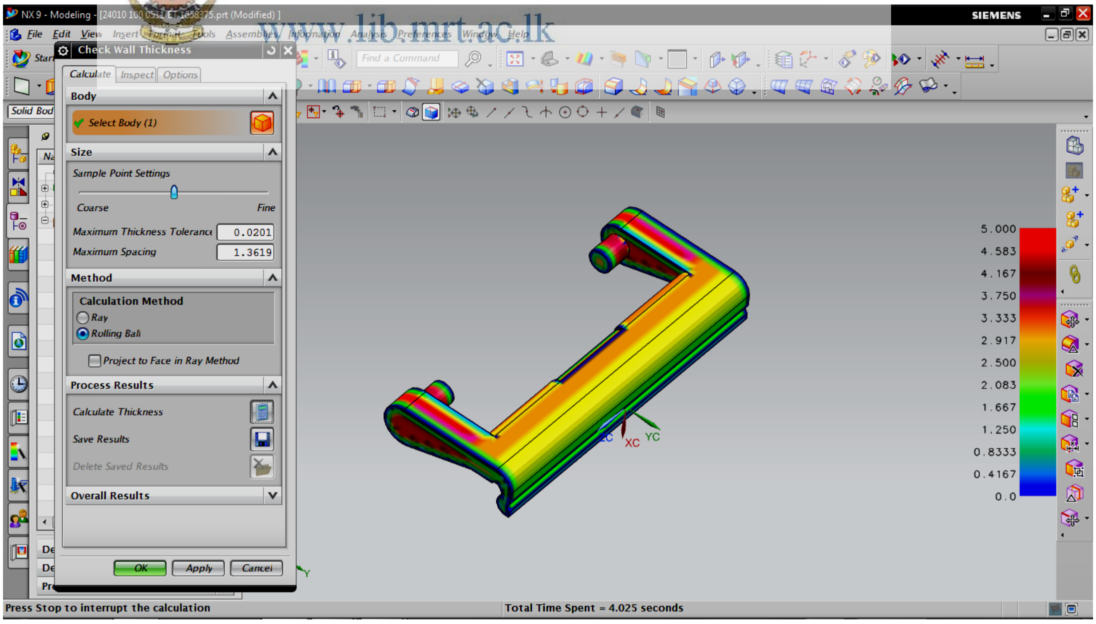
Figure 3.9 Warpage (SWP)

Figure 3.8 shows the analysis result of Autodesk Moldflow Adviser. It gives the normal maximum deflection as 0.4451mm. Also it indicates that the deflection can occur in the middle portion of the article as well as in both ends.

Figure 3.9 shows the nominal maximum deflection as 0.5019 mm, which is given by the analysis result of Solidworks Plastic. It indicates that the deflection can occur in the hinged points.

Both softwares give lesser values than the real deflection of 0.7 mm. The value given from the solid works is much nearer to the actual value than the value given by Autodesk Moldflow adviser. But the pattern of the deflection took place in the article, is similar to the pattern shown in the result given by Autodesk Moldflow adviser than in the Solid Works.

When examining the geometry of the part, there aren't any sharp thickness variations. Also there are no boshes with sharp edges. But there are smooth thickness variations through the part which is shown in Figure 3.10. The thickness analysis was done with Siemens NX9.0.



NX 9 - Modeling - [24010 100 0511 ET 1658375.prt (Modified)]

Check Wall Thickness

Select Body (1)

Sample Point Settings

Maximum Thickness Tolerance 0.0201

Maximum Spacing 1.3619

Calculation Method
○ Ray
● Rolling Ball

☐ Project to Face in Ray Method

Press Stop to interrupt the calculation

Total Time Spent = 4.025 seconds

Figure 3.10 Wall Thickness

Part geometry was modified as shown in Figure 3.11 without affecting the function of it. This modification improved the product by reducing the product weight. It caused 14.56% reduction in original weight 12.57g.

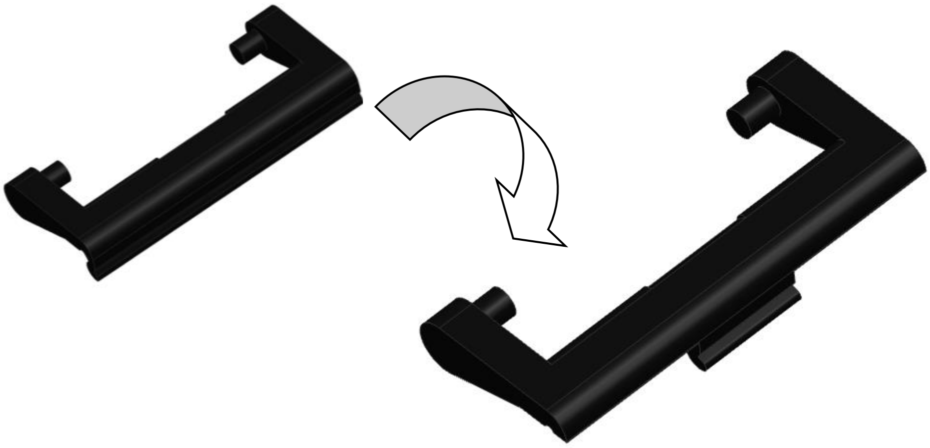
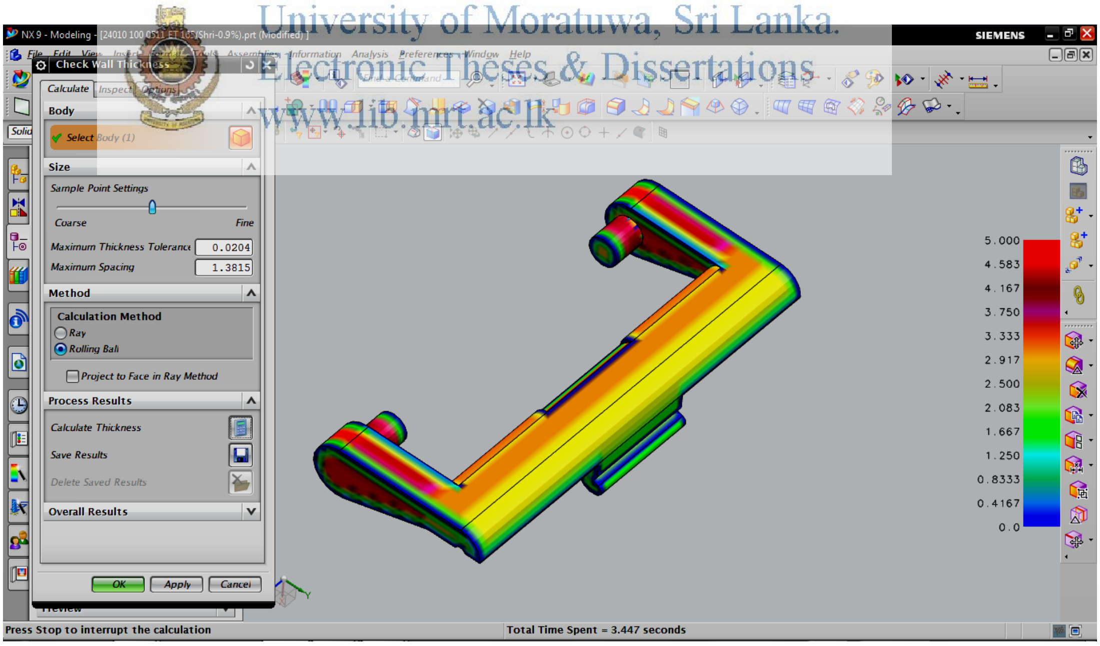


Figure 3.11 Modified Product

Then the modified part was analysed for thickness variations and the result is shown in Figure 3.12.



NX 9 - Modeling - [24010 100 0511 ET 165(Shri-0.9%).prt (Modified)]

Check Wall Thickness

Select Body (1)

Sample Point Settings

Maximum Thickness Tolerance 0.0204

Maximum Spacing 1.3815

Calculation Method
○ Ray
● Rolling Ball

☐ Project to Face in Ray Method

Press Stop to interrupt the calculation

Total Time Spent = 3.447 seconds

Figure 3.12 Wall Thickness of Modified Product

There are some areas that thicknesses have reached about 5mm. Most of the areas, the thickness is about 3mm. Therefore the product can be further modified to reduce the thickness of the thicker areas which are shown in Figure 3.13.

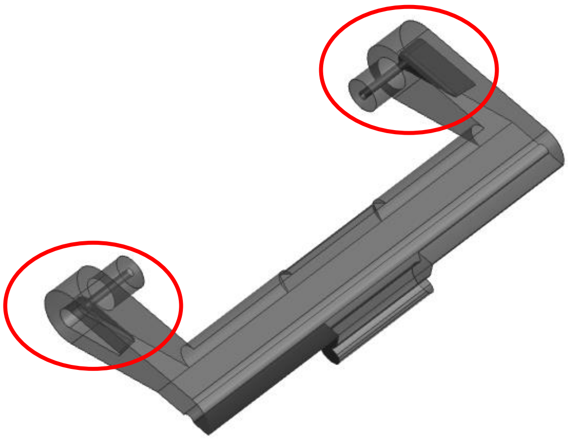
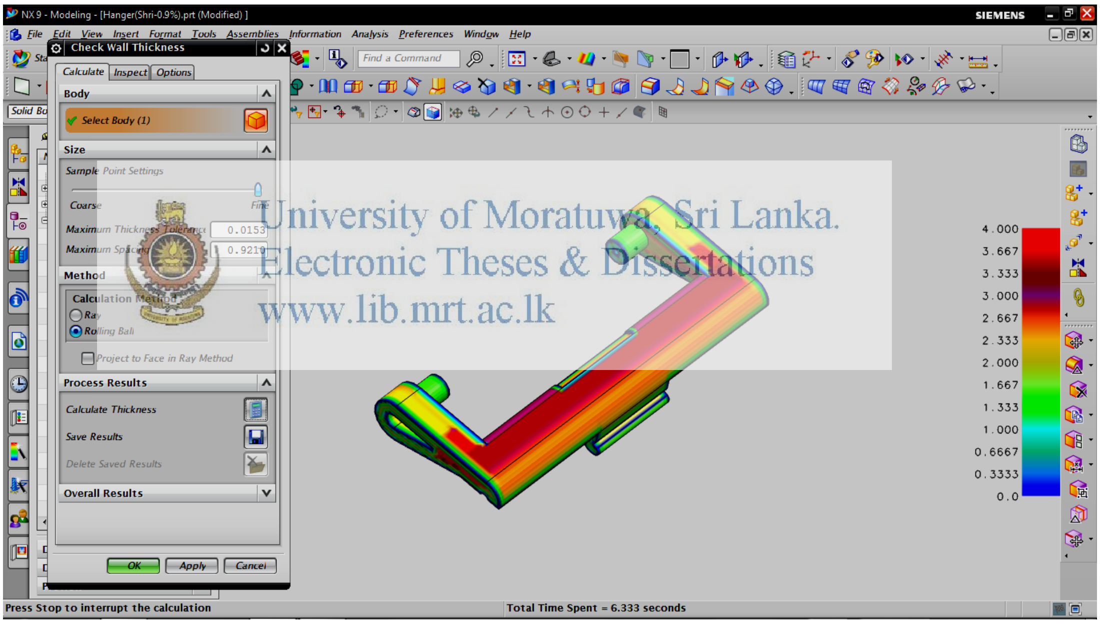


Figure 3.13 Modified Product

Two pockets and two holes were created to maintain the uniform thickness. The Figure 3.14 shows the new thickness distribution.



✓ Select Body (1)

Sample Point Settings: Coarse — Fine

Maximum Thickness Tolerance 0.0153

Maximum Spacing 0.9210

Calculation Method
○ Ray
● Rolling Ball

☐ Project to Face in Ray Method

Press Stop to interrupt the calculation

Total Time Spent = 6.333 seconds

Figure 3.14 Wall Thickness of Modified Product

Due to the modification, the above result shows excellent uniform thickness distribution. But because of this modification, undercuts have to be created. So mould needs sliders for ejecting this part. These undercuts have to be placed in the same axis of current sliders. But practically it is not possible with the existing arrangement of the mould. Therefore existing sliders need to be replaced with larger slider. Then the pocket of ejector side cavity insert has to be widening to match the new sliders. But that arrangement will cause the pocket to merge with a cooling line and main screw hole as shown in Figure 3.15 and appendix A. Therefore this modification cannot be done for the existing mould and need to design a new mould to tally with the requirement.

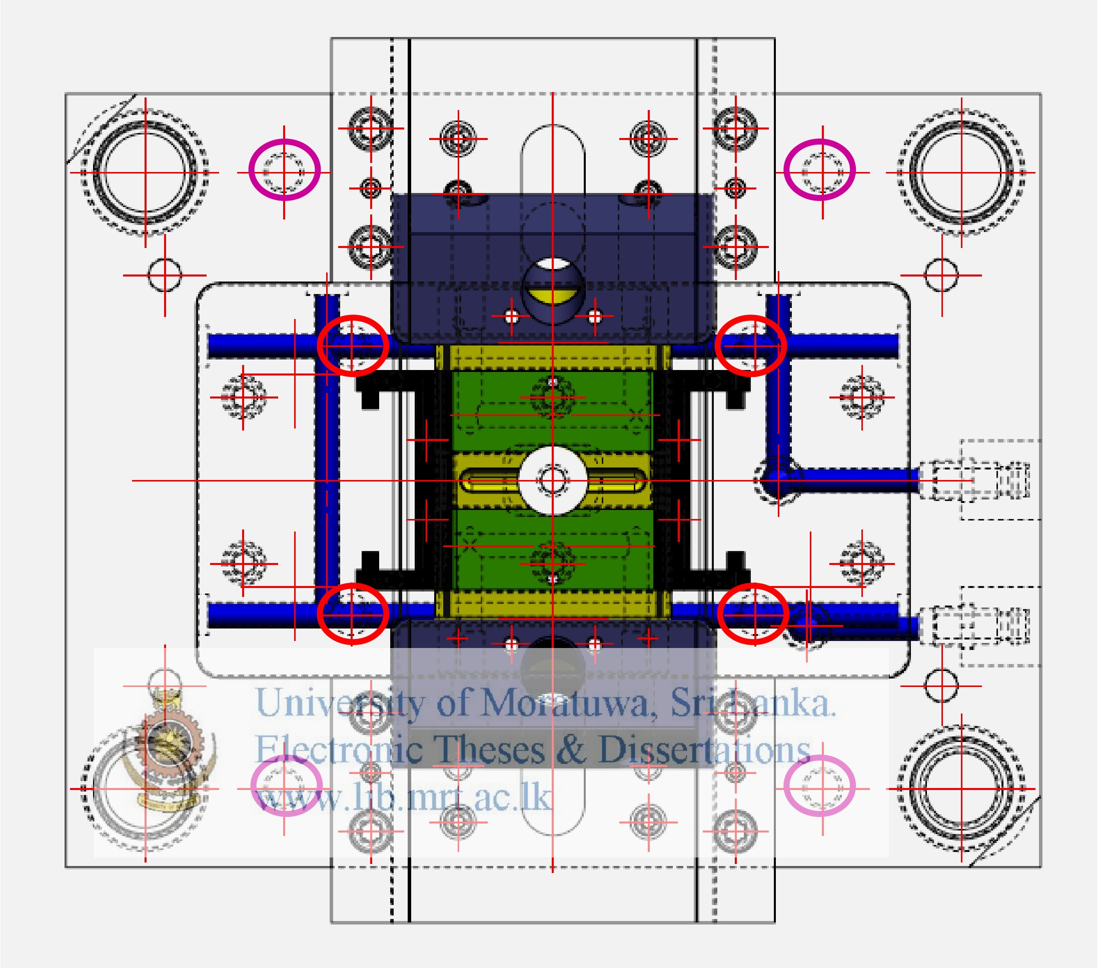


Figure 3.15 Cavity Layout

In Figure 3.15, Circles marked in red shows vertical cooling lines which are needed to be merged and circles marked in violet shows the main screw holes which are needed to be merged. So this modification will need larger size new mould. As manufacturing a new mould is associated with relatively higher cost, the first trial was done to check the warpage of the sample, ignoring this modification.

Therefore the previous design was analysed and the results are shown in Figure 3.16 and Figure 3.17.

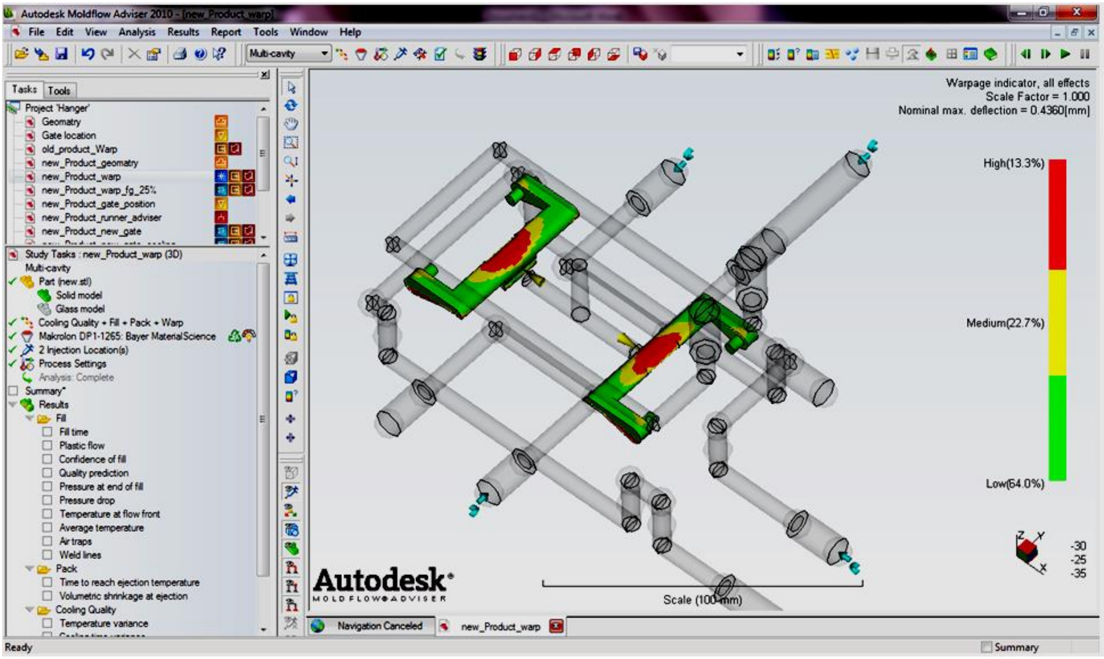


Figure 3.16 Warpage of Modified Product

Results shows that the normal maximum deflection as 0.4360mm. Also this shows that the deflection can occur in the middle portion of the article as well as in both ends.

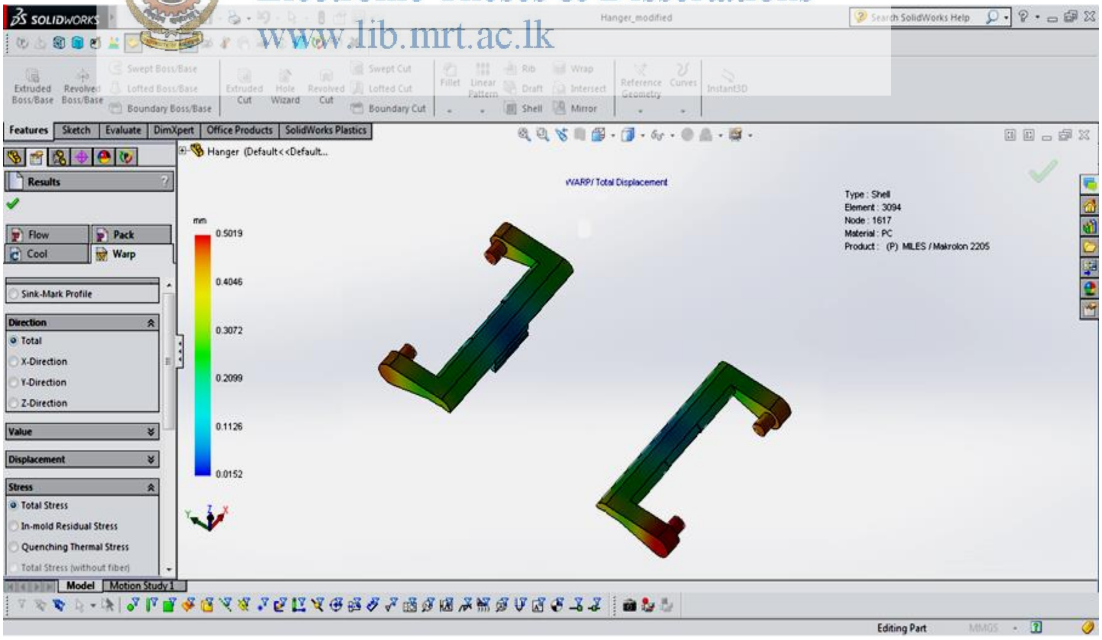


Figure 3.17 Warpage of Modified Product

Results of Figure 3.16 show the nominal maximum deflection as 0.5019 mm. The results of the warpage indication gain from Solid works plastics for both geometries are same. But Autodesk Moldflow Adviser shows slight reduction of deflection as shown in Table 3.2.

Table 3.2 Classification of factors affecting for warpage

Modified Part Geometry	Existing Part Geometry
Nominal Maximum Deflection - 0.4360mm	Nominal Maximum Deflection - 0.4451mm
Percentage of area of deflection Low – 64.0% Medium – 22.7% High – 13.3%	Percentage of area of deflection Low – 58.3% Medium – 22.1% High – 19.6%

The percentage of higher deflection positions is lower in modified part geometry than in existing part geometry though the nominal maximum deflection is same. Therefore modified part geometry has caused some change in the deflection as it minimized the higher deflection range. Also it reduced the weight of the product and minimized the material requirement, without affecting the function in the assembly. Therefore the modified part geometry is selected for the analysis.

Another important factor is that, due to this modification, there is a reduction of the length of the roller joint which reduces the effect of deflection. The results which are shown in Figure 3.16 and Figure 3.17 verified this factor.

3.3.2. Gate Location

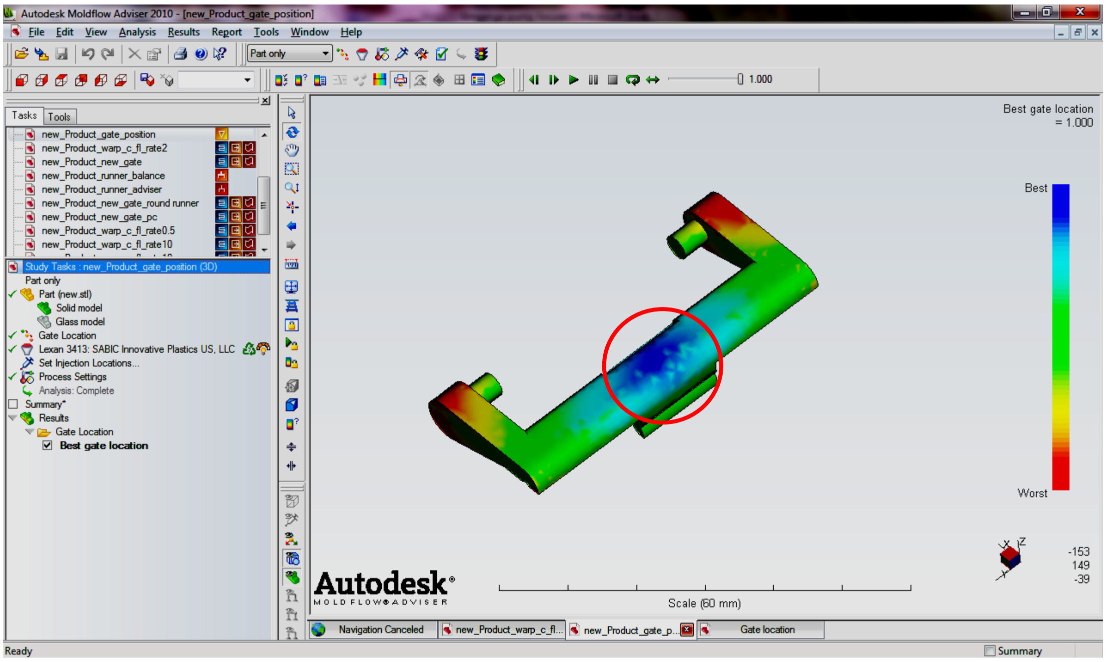


Figure 3.18 Gate Location Analysis

Analysis was done using the modified part geometry to find the best gate location for filling. Filling flow pattern is playing a major role in warpage reduction in injection molding. Figure 3.19 and Figure 3.21 show the variation of the flow pattern with changing gate point position.

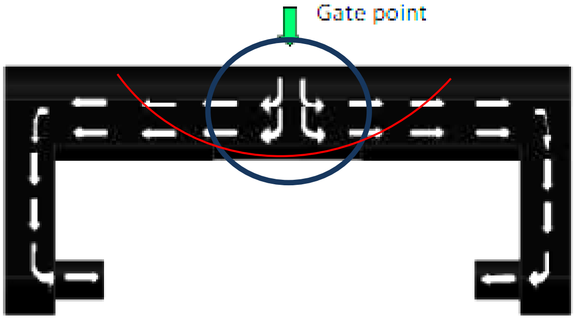


Figure 3.19 Flow Pattern of Mid-Point Gate Position

Significant warpage can be observed in the area where gate is located. Material Flow pattern from the gate point to both ways is symmetrical and warpage is due to the different shrinkage characteristics along the flow paths as well as perpendicular to flow paths.

The effect of the difference in wall thickness on shrinkage is relatively slight. The main cause of warping is the difference between the fibre orientations. That is difference between longitudinal orientation and perpendicular orientation of the fibres to the direction of flow. So the warpage occurs due to the wall thickness distribution, gate location and flow pattern of the moulded part.

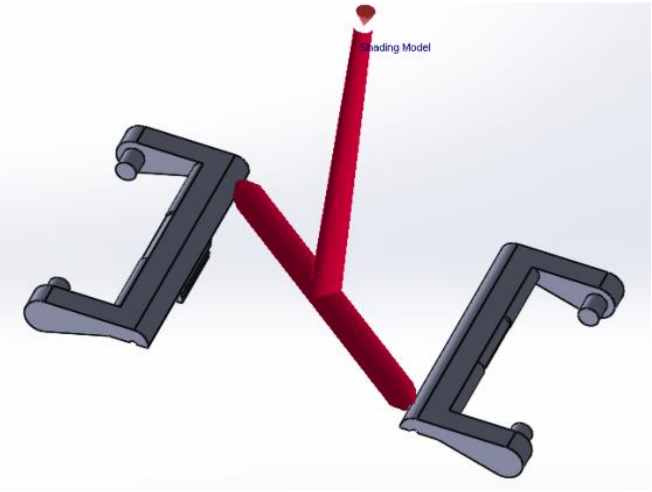


Figure 3.20 Gate Position at a Side

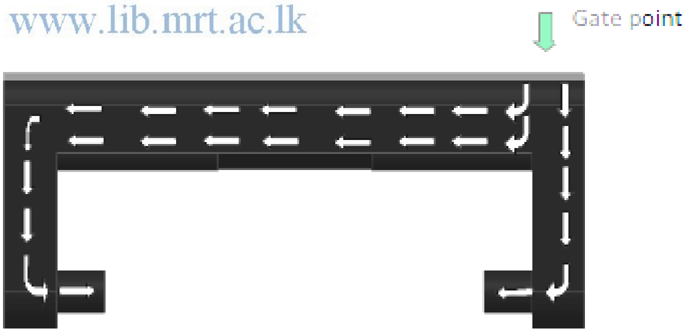


Figure 3.21 Flow pattern of new gate position

When the gate point is moved to the end point, the flow pattern is in a single way through the middle base. Also same fill pattern can be observed through the side arms. The Figure 3.22 shows the software analysis of how this change effect for the warpage.

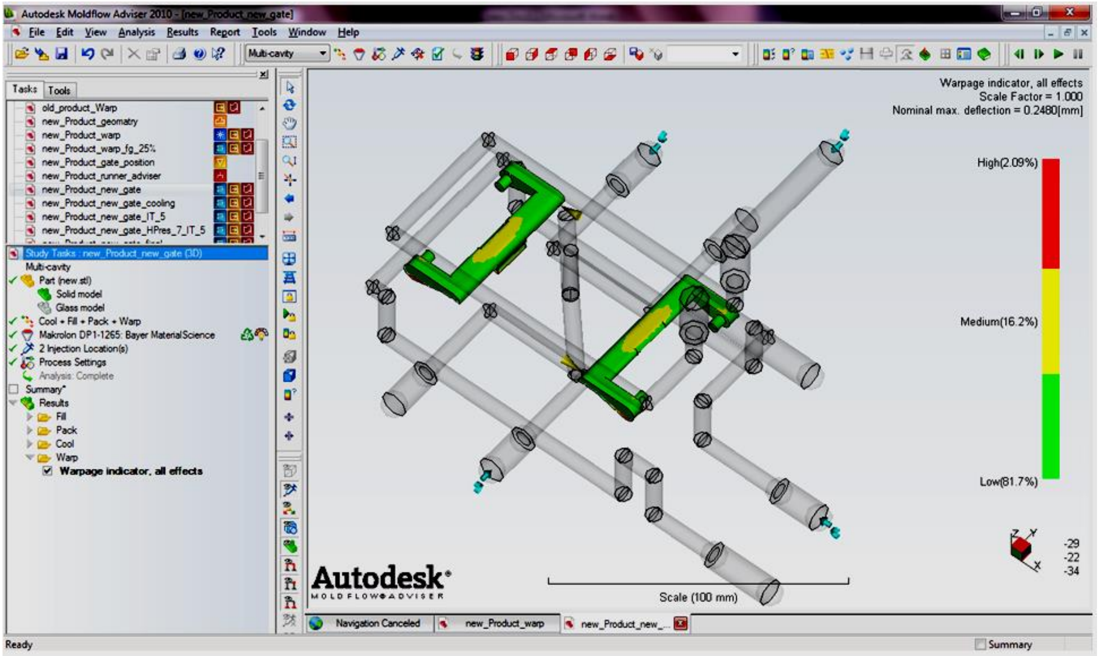


Figure 3.22 Warpage for New Gate Position (AMA)

Autodesk Mould flow adviser shows that, placing the gate at the end point reduces the Warpage to 0.2480 mm.

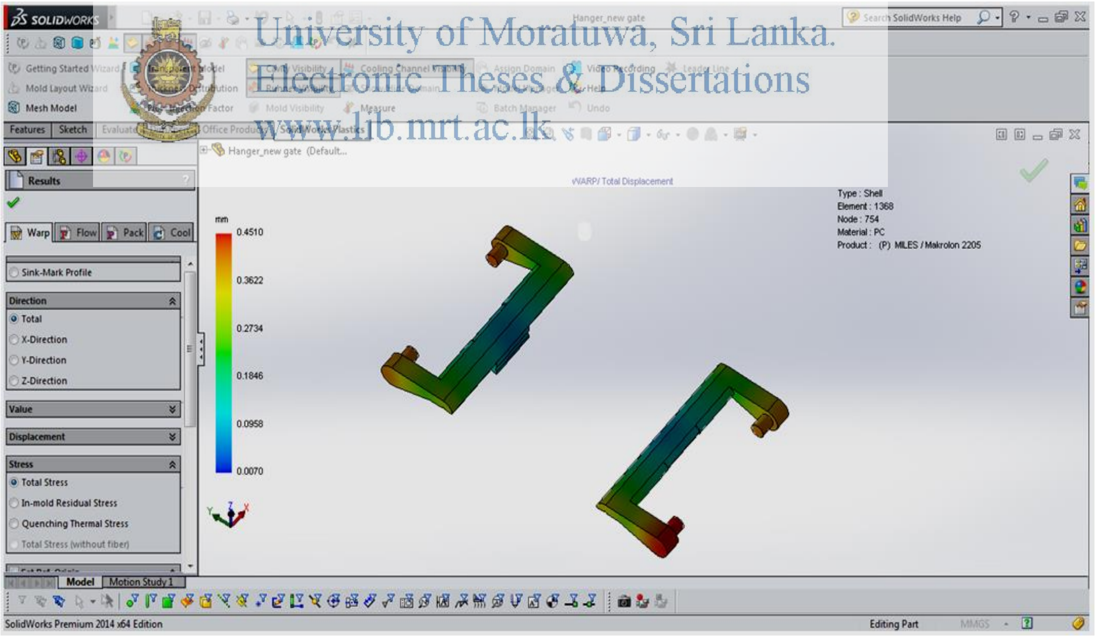


Figure 3.23 Warpage for New Gate Position (SWP)

The Solid works plastic shows the deflection as 0.4510mm. This value is slightly deviated from the value obtained when the gate was at the mid-point.

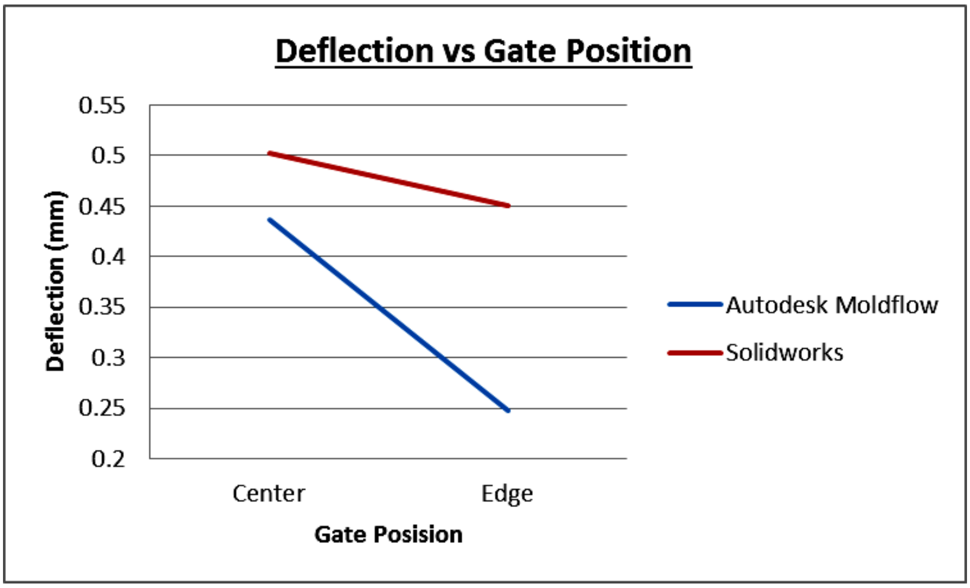


Figure 3.24 Change in Deflection by Gate Position

Filling can be improved by using two gate points at both ends. Figure 3.25 shows the runner system and the location of gate points related to it.

Figure 3.25 Twin gate points

Autodesk Moldflow Adviser shows that the use of twin gate positions reduces the injection pressure requirement up to 5.64Mpa, while Solid works plastic shows that value as 10.94Mpa. These results are shown in Figure 3.26 and Figure 3.27 respectively.

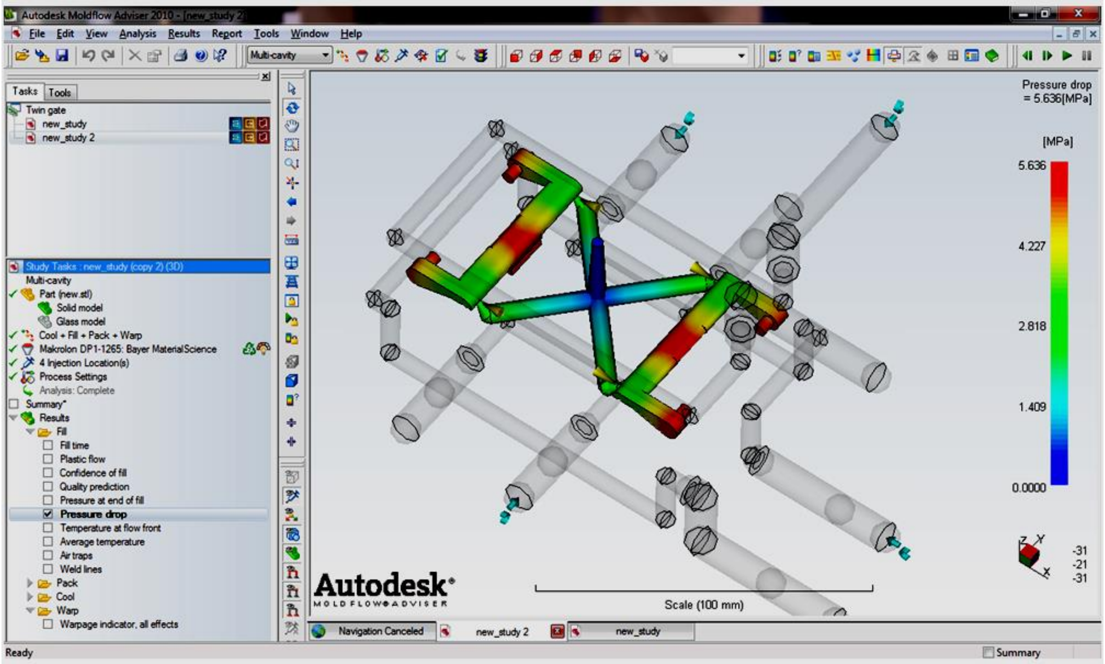


Figure 3.26 Pressure drop through the Runner and Gates (AMA)

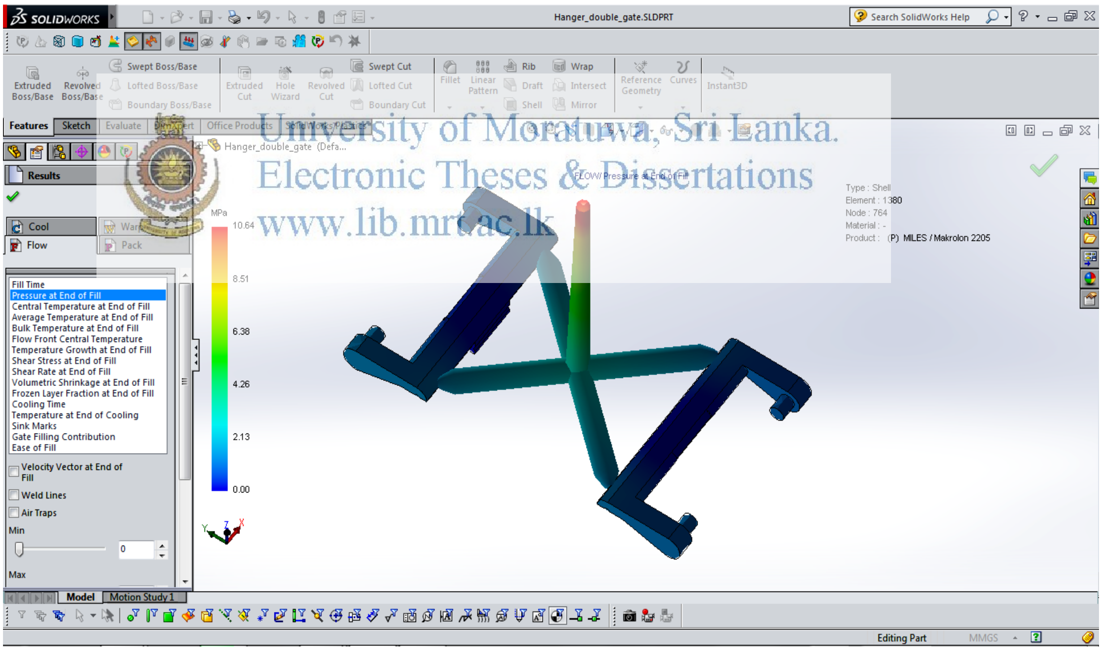


Figure 3.27 Pressure at End of Fill (SWP)

In Figure 3.28 and Figure 3.29, it shows the reduction of the injection time which is due to the acceleration of filling caused by this modification.

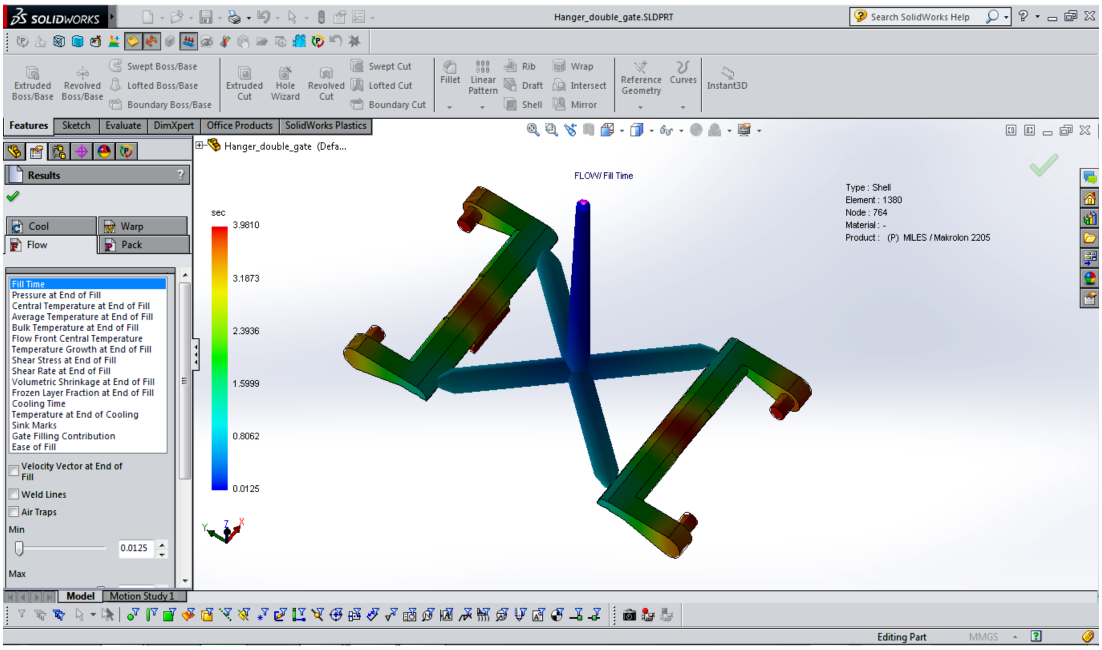


Figure 3.28 Fill Time (SWP)

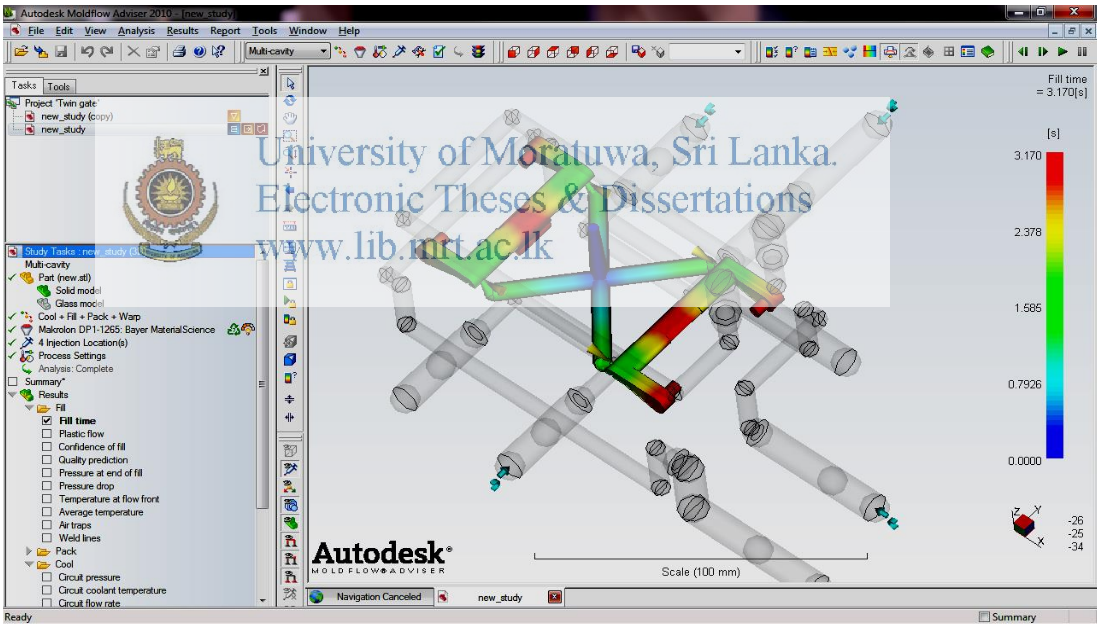


Figure 3.29 Fill Time (AMA)

Two injection points can be placed in both ends of the part to accelerate the filling. But this will lead to following effects.

i. Increase Runner Weight

Table 3.3 shows the percentage weight of the material of the product and runner with respect to the total material weight of initial product, modified product with single gate and modified product with twin gates respectivly. The weight values are taken using Autodesk Moldflow Adviser software.

Table 3.3 Change of runner weight percentage

	Initial product		Modified product with Single Gate		Modified product with Twin Gates	
	Weight(g)	Percentage (%)	Weight(g)	Percentage (%)	Weight(g)	Percentage (%)
Product	25.13	71.73	21.47	61.05	21.47	52.61
Runner	9.90	28.27	13.70	38.95	19.34	47.39
Total shot (product + runner)	35.03		35.17		40.81	

In the initial product the percentage of the runner material weight to total material weight is 28.27%. In modified product with single gate, the runner weight to total weight is 38.95%. when twin injection points are used the percentage of runner weight to total weight become 47.39%. This is nearly half of the shot weight and it is not economically advantage which leads to material wastage. Though the recycling can be done, only maximum 5% of recycled material can be used with virgin material to keep primary mateiral properties. Therefore this will increase material recycling cost too.

ii. Weld line

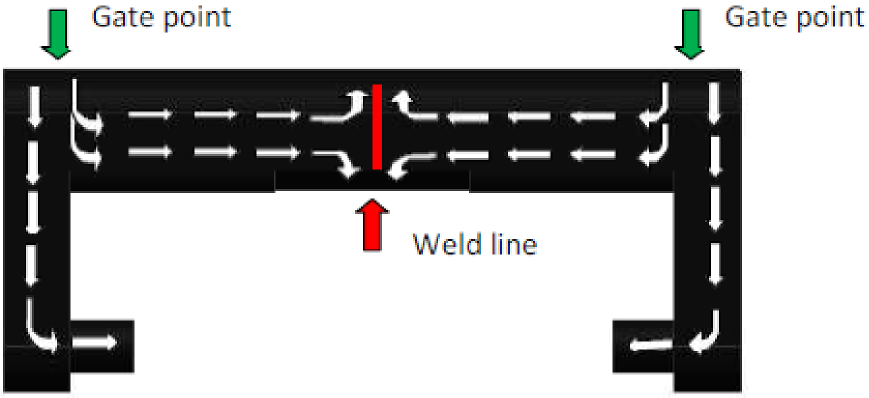


Figure 3.30 Weld Line

Weld lines (Figure 3.30) can occur when two or more melt streams unite in the mould. This happens, when the parts are gated at several points. Quality will reduce as a result of weld lines. Also air entrapment (air bubbles) occurs when air that should be expelled from the mould is enclosed by melt streams and cannot escape. Weld lines and air entrapment reduce the mechanical properties, particularly impact strength. Figure 3.31 and Figure 3.32 shows that how these weld lines can occur.

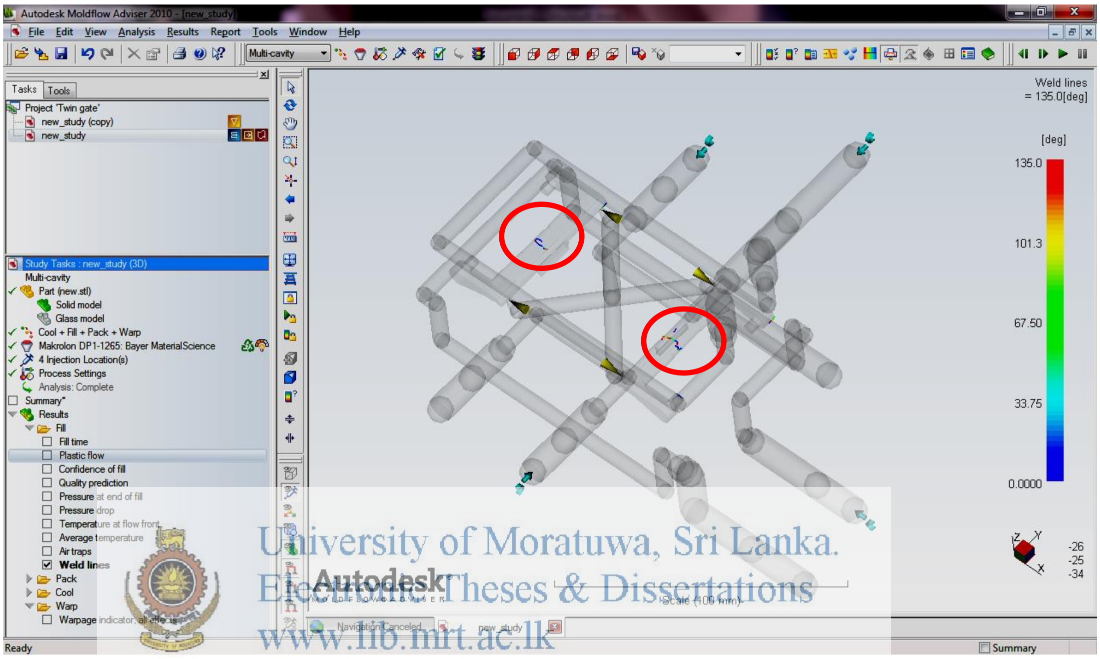


Figure 3.31 Weld Line (AMA)

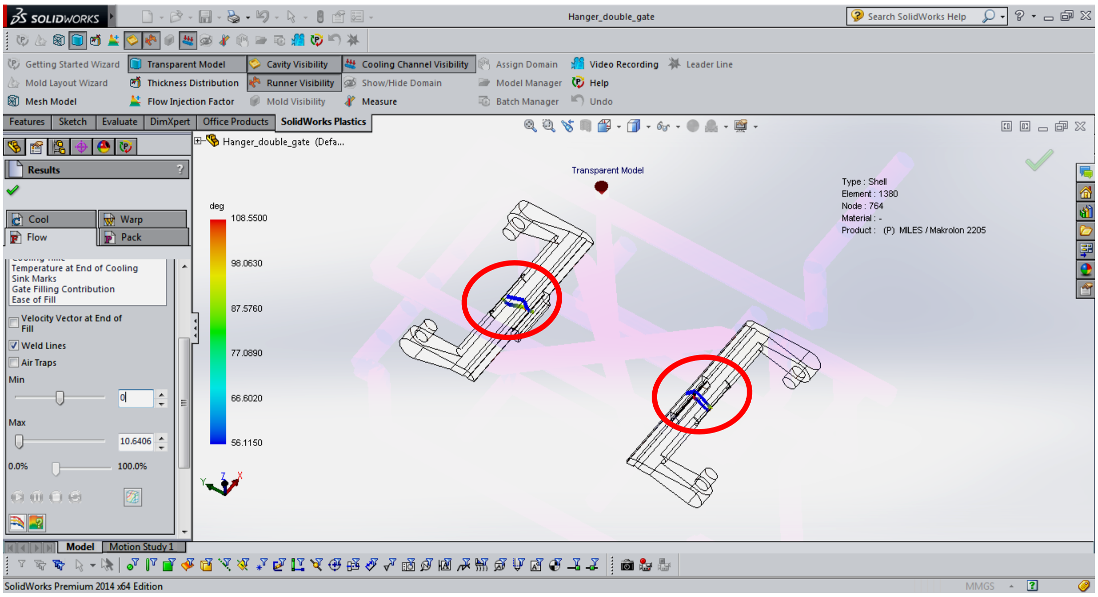
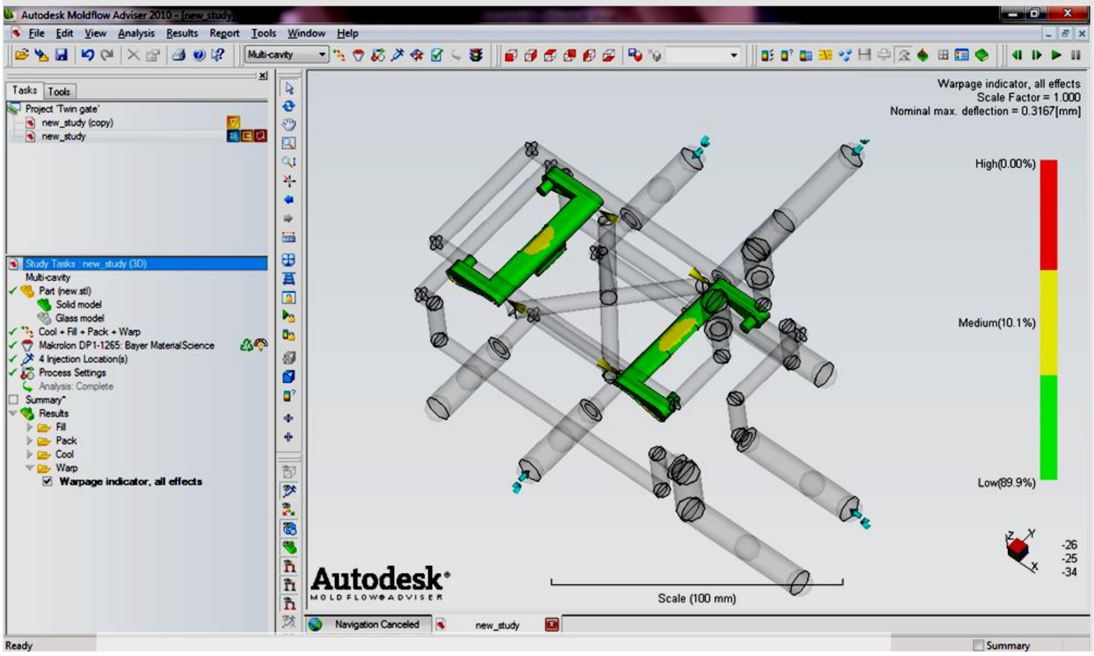


Figure 3.32 Weld Line (SWP)

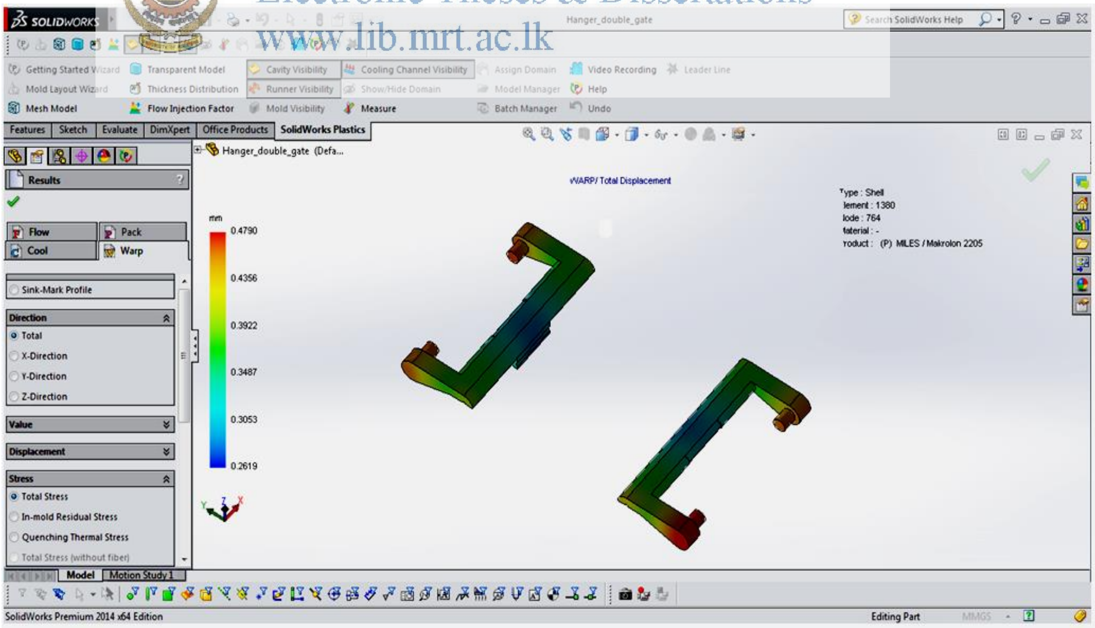
iii. Warpage

When using twin gates, it shows lesser deflection than old single gate position but higher than new single gate position as shown in Figure 3.33 and Figure 3.34.



Warpage indicator, all effects
Scale Factor = 1.000
Nominal max. deflection = 0.3167[mm]

Figure 3.33 Warpage for Twin Gates (AMA)



Type : Shell
Element : 1380
Node : 764
Material : -
Product : (P) MILES / Makrolon 2205

Figure 3.34 Warpage for Twin Gates (SWP)

Therefore considering above facts, new single gate position is selected to proceed with the research.

3.3.3. Runner Systems

Runner system conveys the molten material from sprue to gate. The cross section of the runner should have maximum cross-sectional area and minimum perimeter. Runners should have a high volume-to-surface area ratio. Such a section will minimize heat loss, premature solidification of the molten resin in the runner system, and pressure drop. Balancing the runner system ensures that all mould cavities fill at the same rate and pressure.

i. Cross sections of runner

   a) Unfavourable cross sections shown in Figure 3.35 have to be avoided.

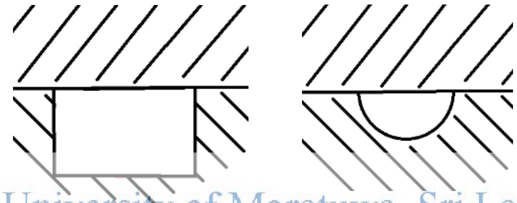


Figure 3.35 Unfavourable Cross Sections

   b) Trapezoidal cross section

This is an alternative to parabolic cross section. Disadvantage of this cross section is that more frictional losses and scrap compared with circular cross section.

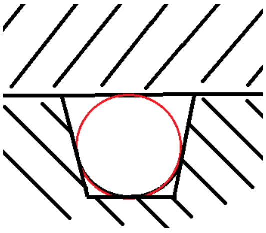


Figure 3.36 Trapezoidal Cross Section

   c) Circular cross section

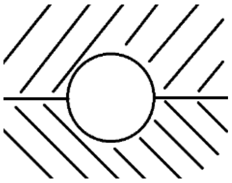


Figure 3.37 Circular Cross Section

Slowest cooling rate, low heat and friction losses, smallest surface relative to cross section and centre of channel freezes last are the main advantage of choosing this cross section. The main disadvantages are difficulty in machining both mould halves and higher cost associated with it.

d) Parabolic cross section

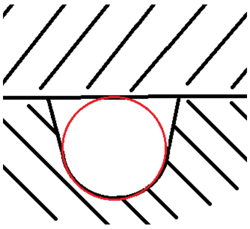


Figure 3.38 Parabolic Cross Section

This is the best approximation for circular cross-section. Machining is needed in only one half of the mould. Usually machining is done in movable side due to easy ejection of the runner. But more scrap is there, compared with circular cross-section.

Consequently, a runner with a circular cross section is the most ideal one. However, since it is necessary to carve the runner in both the fixed side and the movable side, the cost of manufacturing the mould increases. In order to solve this problem, a parabolic cross-sectional shape runner is used.

Since Autodesk Moldflow Adviser and Solid Works do not facilitate to select parabolic shape as runner shape, for the purpose of analysis, the circular section which behave much similar to parabolic runner, was selected. But for the weight calculation, actual parabolic cross section parameters were considered.

ii. Runner Diameter

Ideally, when deciding the size of the runner diameter, it will take many factors into account. Those are part volume, part flow length, runner length, machine capacity, gate size, and cycle time. Generally, runners should have diameter equal to the maximum part thickness, but within 4 mm to 10 mm diameter range to avoid early freeze-off or excessive cycle time [4]. The runner should be large enough to minimize pressure loss and small enough to maintain satisfactory cycle time. Smaller runner diameter has been successfully used as a result of computer flow analysis where the smaller runner diameter increases material shear heat, thereby assisting in maintaining melt temperature and enhancing the polymer flow. Large runners are not economical because of the amount of energy that goes into forming, and then regrinding the material that solidifies within them.

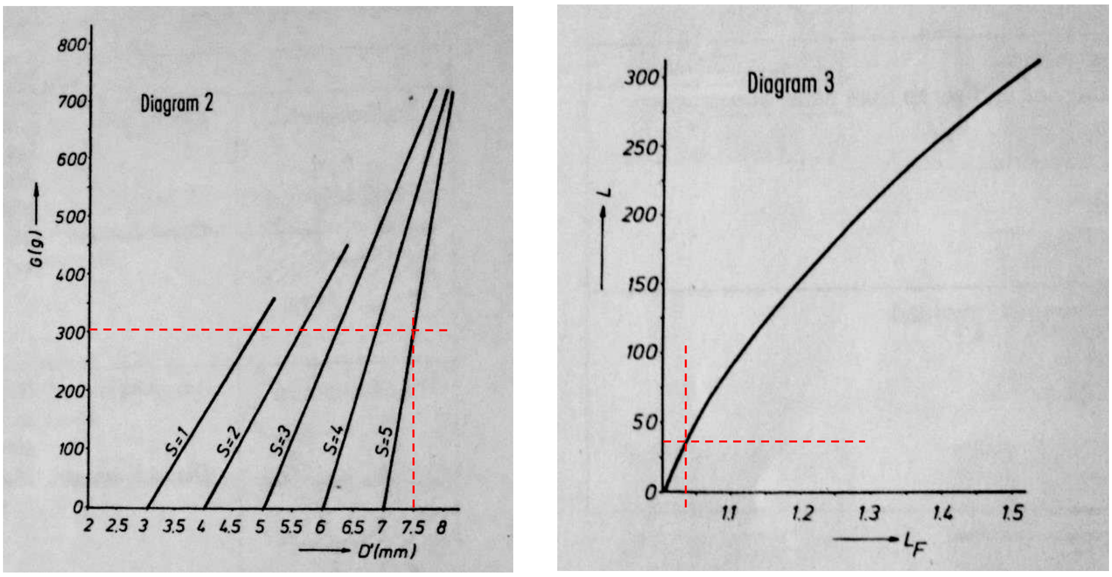


Figure 3.39 Diagrams for Runner Diameter Calculation [4]

Maximum thickness of part (S) = 5mm

Part weight (G) = 10.735g

Gradient of line S=5 = 300/ (7.5-7)

= 600

ΔD' for part weight = 10.73/600

= 0.018

So, D' = 7.018

Runner length per cavity (L) = 34.1

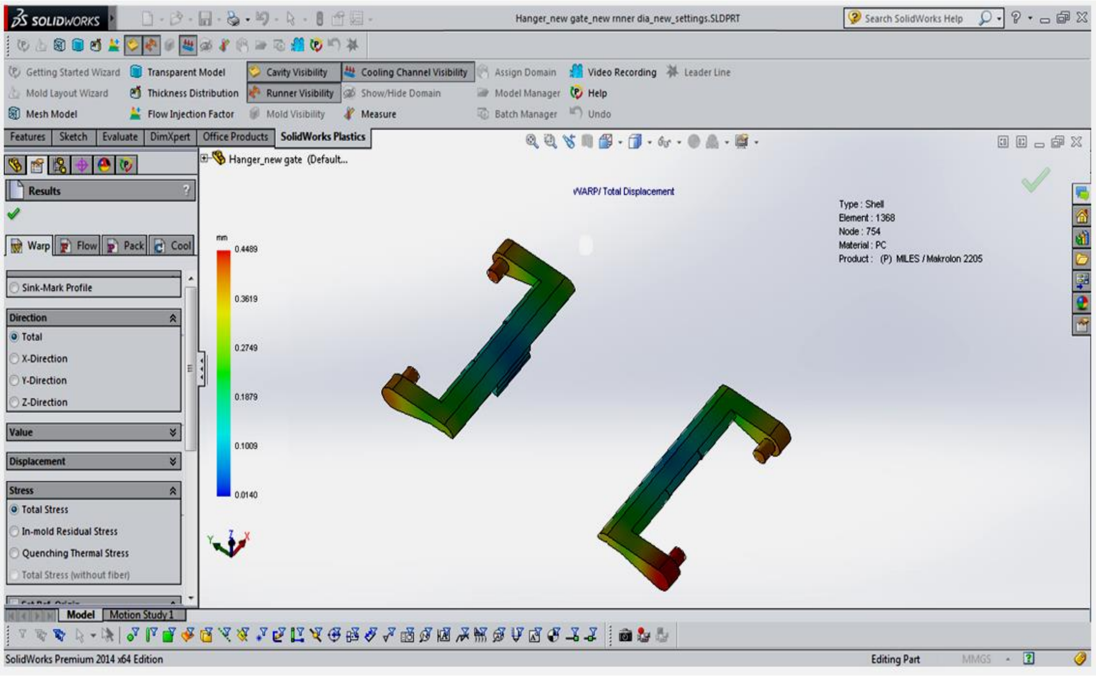
L_F = 1.03

Correct diameter (D) = D' x L_F

= 7.018 x 1.03

= <u>7.23mm</u>

Results of the warpage analysis after introducing the new runner system, was shown in Figure 3.40 and Figure 3.41.



WARP/ Total Displacement

mm

0.4489

0.3619

0.2749

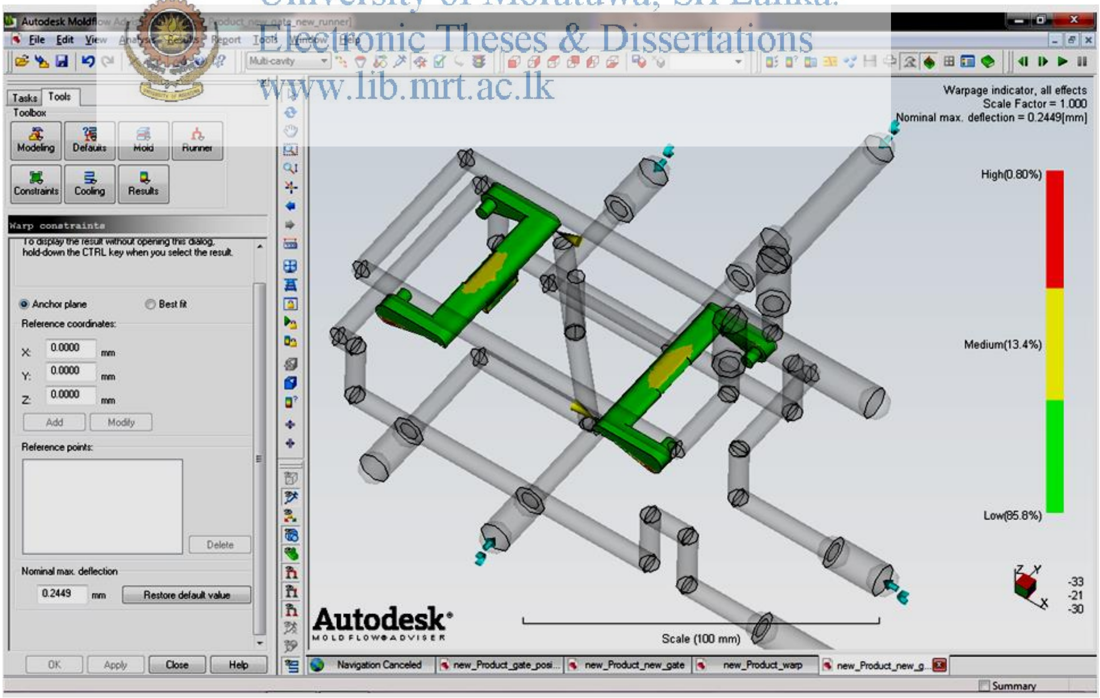
0.1879

0.1009

0.0140

Type : Shell
Element : 1368
Node : 754
Material : PC
Product : (P) MILES / Makrolon 2205

Figure 3.40 Warpage for New Runner System (AMA)



Warp constraints

To display the result without opening this dialog, hold-down the CTRL key when you select the result.

● Anchor plane ○ Best fit

Reference coordinates:

X: 0.0000 mm

Y: 0.0000 mm

Z: 0.0000 mm

Reference points:

Nominal max. deflection

0.2449 mm

Warpage indicator, all effects
Scale Factor = 1.000
Nominal max. deflection = 0.2449[mm]

Figure 3.41 Warpage for New Runner System (SWP)

Modified mould design is shown in appendix B.

3.3.4. Filling and Holding Pressures

Injection pressure and holding pressure are very important factors which determine the properties of the injection moulded parts. Injection pressure need to adequate to fill the mould totally with molten polymer. The switching from injection pressure to holding pressure should happen very smoothly. The molten polymers are compressible and at higher pressure it will compensate the shrinkage of materials during the cooling. The holding pressure can compensate the shrinkage during the cooling period as long as the injection channel from the injection cylinder to the mould is in molten state. After the solidification of the injection channel the injection moulded part will shrink freely.

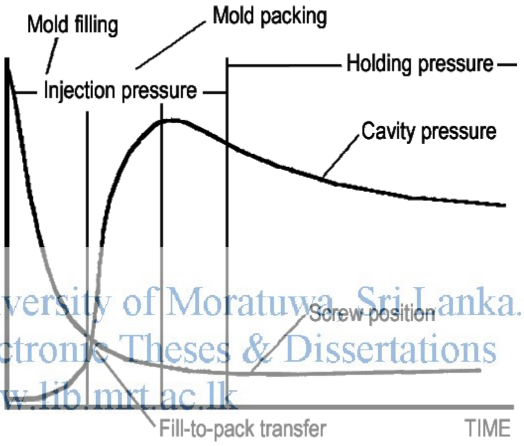


Figure 3.42 Injection Moulding Pressure Cycle

1. Fill pressure

Initially there is no resistance to flow of melt. Resistance increases as the cavity is being filled up. Fill pressure is the measure of resistance to flow of melt. No variation of warpage is observed when increasing the injection pressure from both softwares.

Change in the gate position and runner diameter may change the current injection pressure of 13.8MPa. The injection pressure required should be equal to maximum pressure drop. Maximum pressure drop can be seen in the end points which are far away from the gate position. The result shows that Maximum pressure drop is 14Mpa (Figure 3.43).

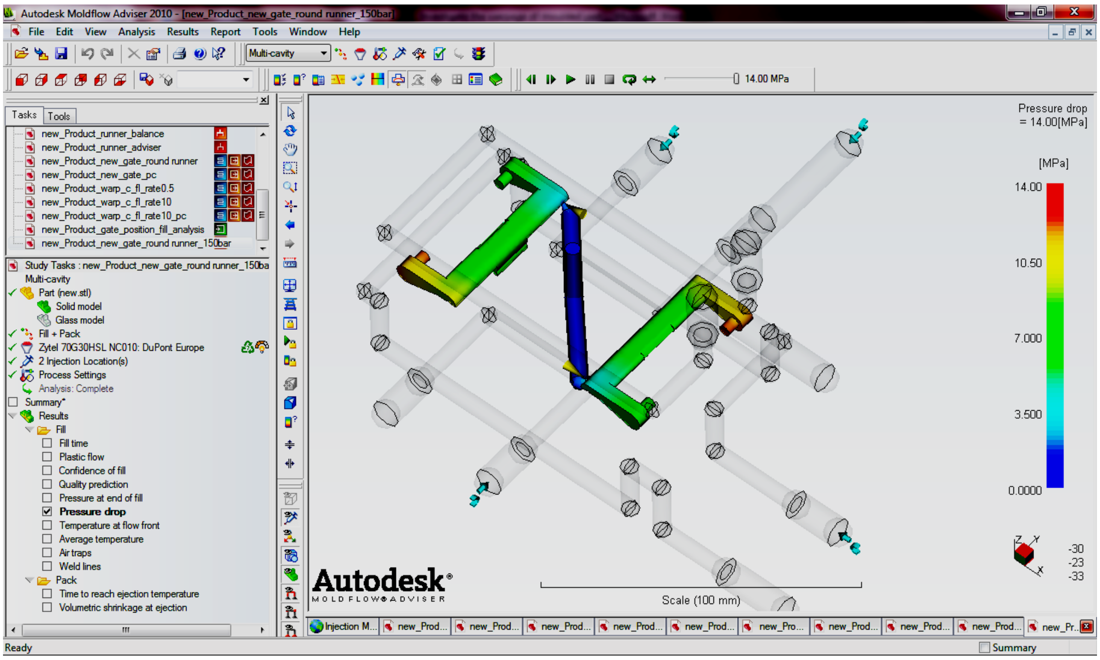


Figure 3.43 Pressure Drop (AMA)

As Solid Works Plastics does not give the result of the pressure drop, in analysis it is consider pressure at end of fill which gives the opposite result of pressure drop. Therefore according to this result, injection pressure requirement is 13.15 Mpa.

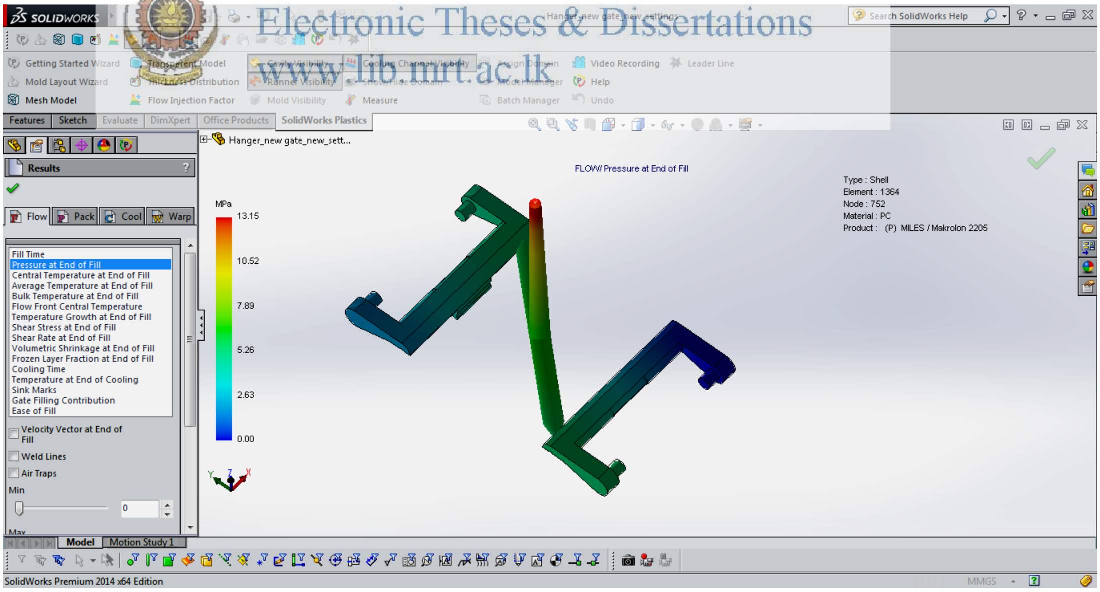


Figure 3.44 Pressure at End of Fill (SWP)

Injection pressure should be maintained at minimum possible value as higher injection pressure can cause higher power consumption and increase the wear of machine components.

2. Packing Pressure

The holding pressure was varied from 6Mpa to 11Mpa and the results are shown in the graph of Figure 3.45.

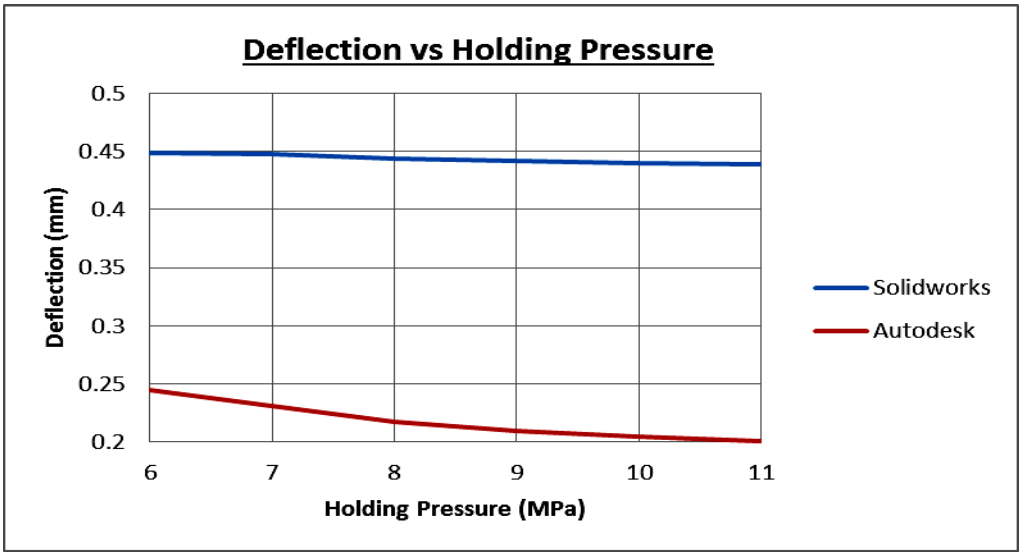


Figure 3.45 Variation of warpage with respective to Holding Pressure

Figure 3.46 Warpage for Holding Pressure of 11Mpa (AMA)

When the holding pressure is 11Mpa, the result given by the Autodesk Mould Flow adviser, is shown in Figure 3.46. But Solid Works Plastic gives a maximum deflection as 0.4392 mm under same condition.

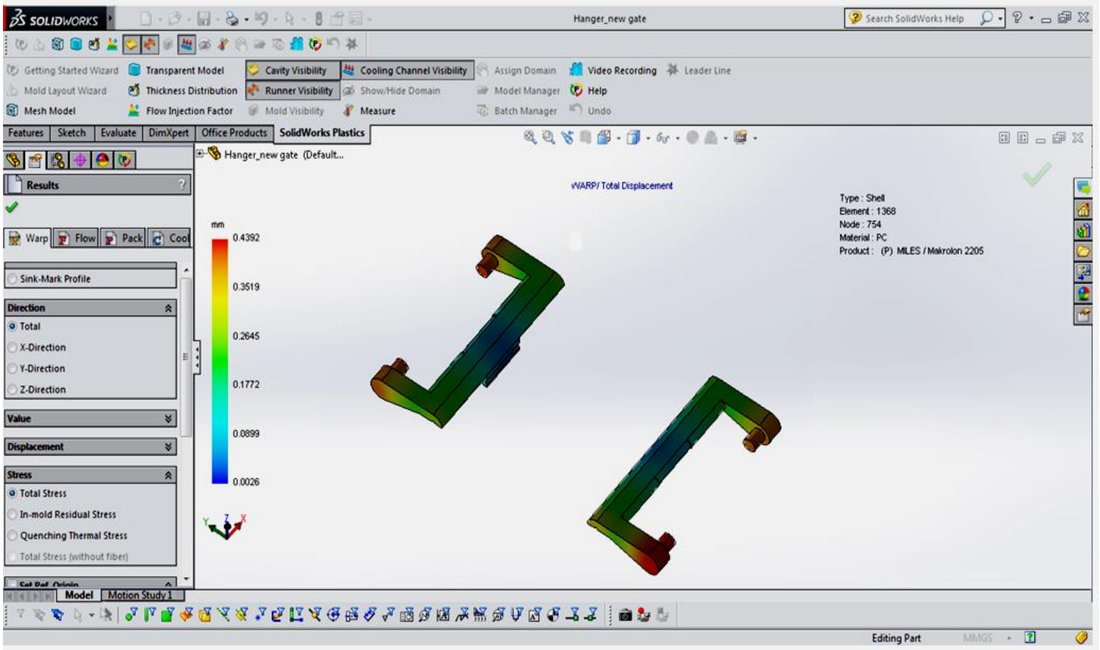


Figure 3.47 Warpage for Holding Pressure of 11Mpa (SWP)

3.3.5. Filling and Packing Times

1. Fill Time

Fill time of 4.852s is the result of the fill time analysis done using Autodesk Moldflow adviser.

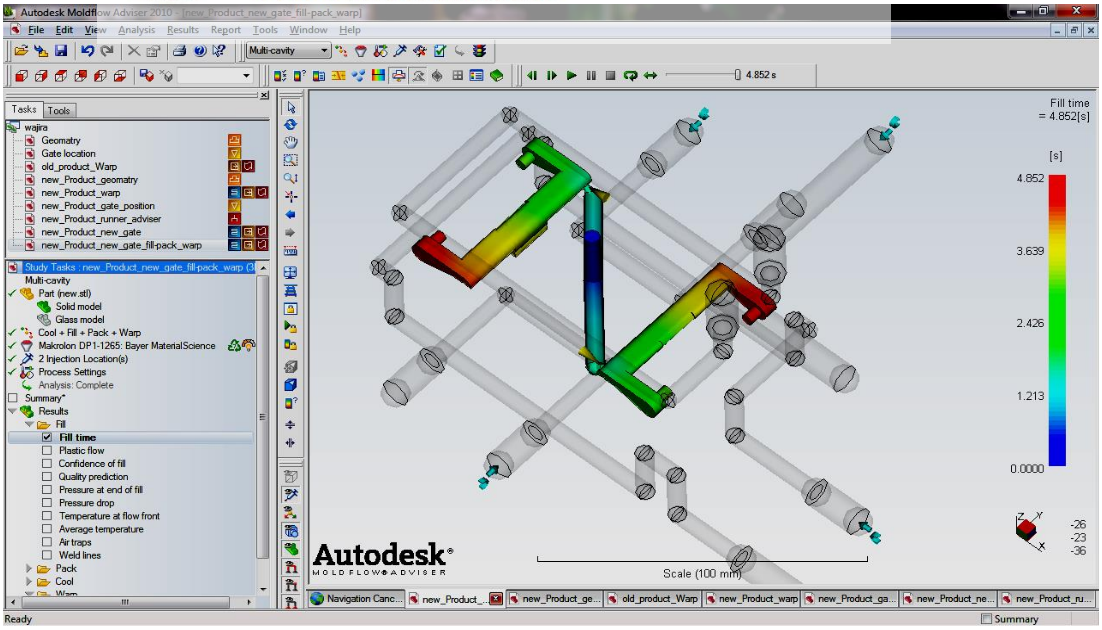


Figure 3.48 Fill Time (AMA)

Fill time of 4.9568s is the result obtained by the fill time analysis which is done using Solid Works plastic.

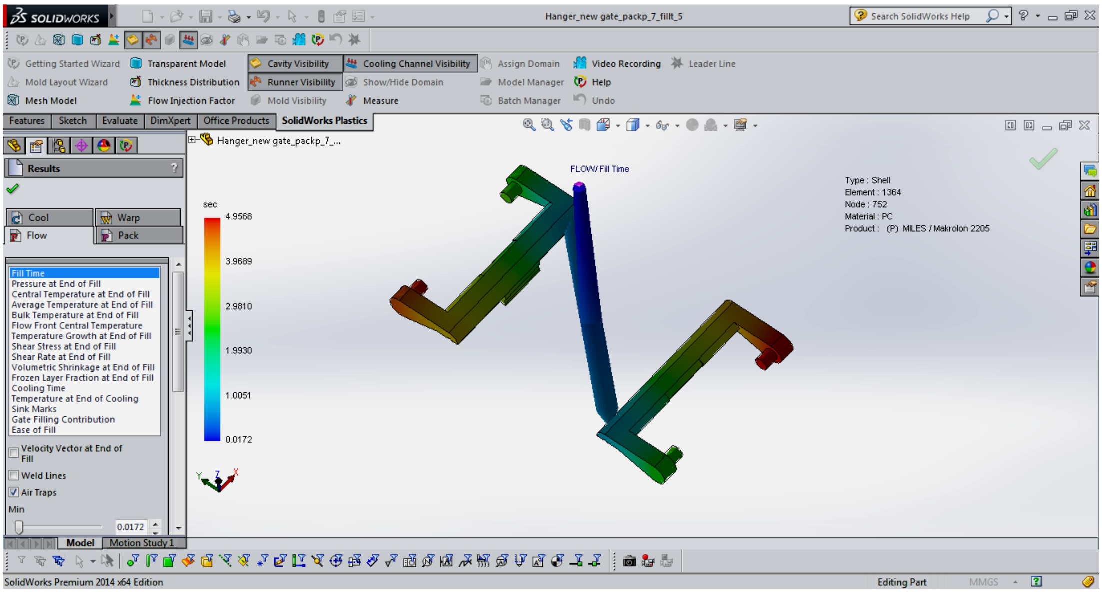


Figure 3.49 Fill Time (SWP)

The results shown in Figure 3.48 and Figure 3.49 prove that the gate point modification results the injection time to rise up to 5s. This can be verified using the analysis results of confidence of fill.

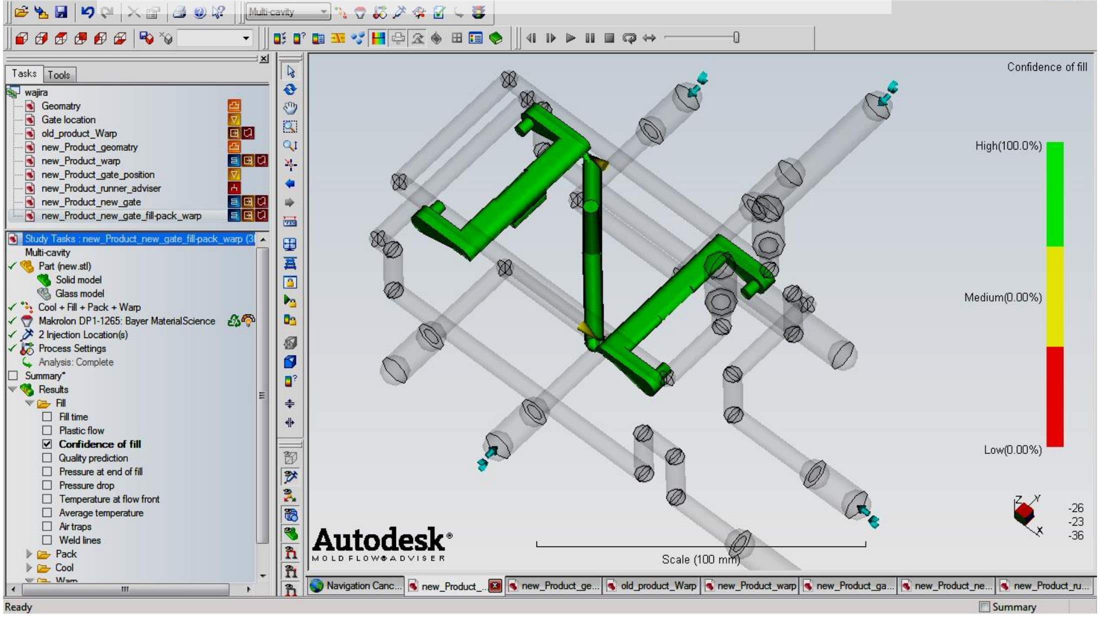
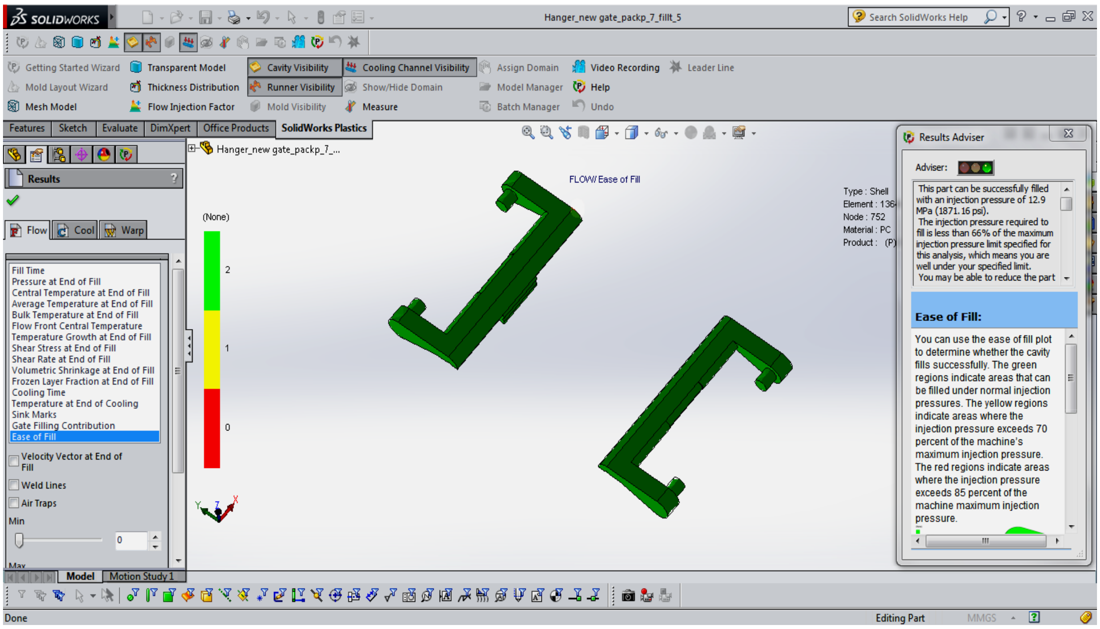


Figure 3.50 Confidence of Fill (AMA)



This part can be successfully filled with an injection pressure of 12.9 MPa (1871.16 psi).
The injection pressure required to fill is less than 66% of the maximum injection pressure limit specified for this analysis, which means you are well under your specified limit.
You may be able to reduce the part

Ease of Fill:

You can use the ease of fill plot to determine whether the cavity fills successfully. The green regions indicate areas that can be filled under normal injection pressures. The yellow regions indicate areas where the injection pressure exceeds 70 percent of the machine's maximum injection pressure. The red regions indicate areas where the injection pressure exceeds 85 percent of the machine maximum injection pressure.

Figure 3.51 Confidence of Fill (SWP)

There is no deviation of warpage which is given by both softwares with respect to the increase of injection time to 5S.

2. Holding time

Analysis was done with both softwares by varying holding time from 5s to 10s and the results are shown in the graph in Figure 3.52.

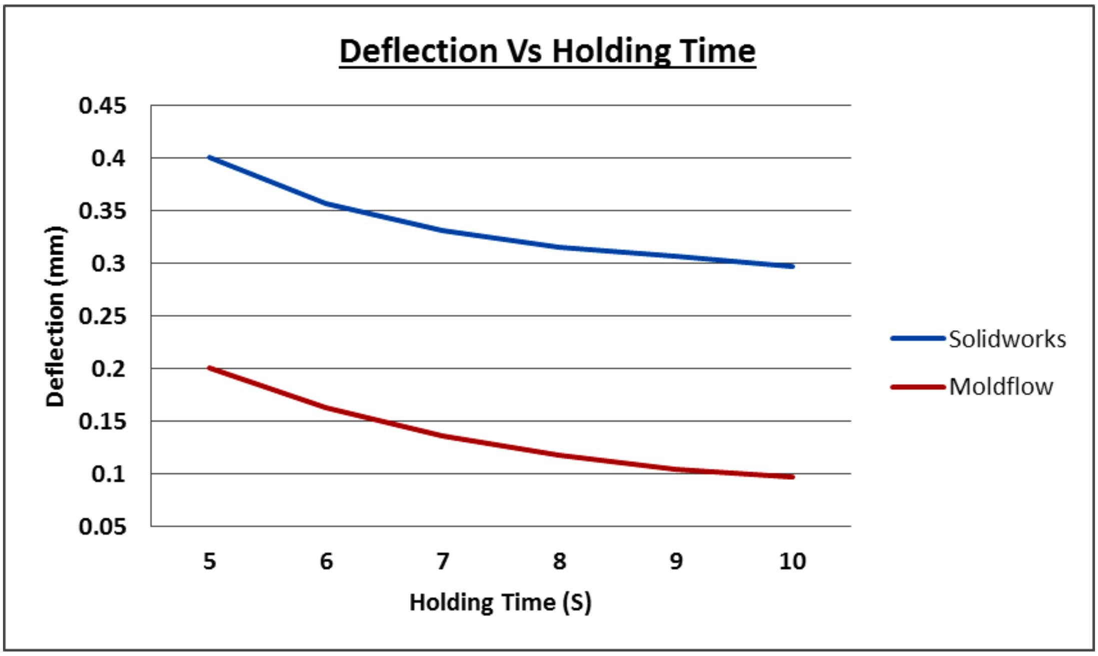
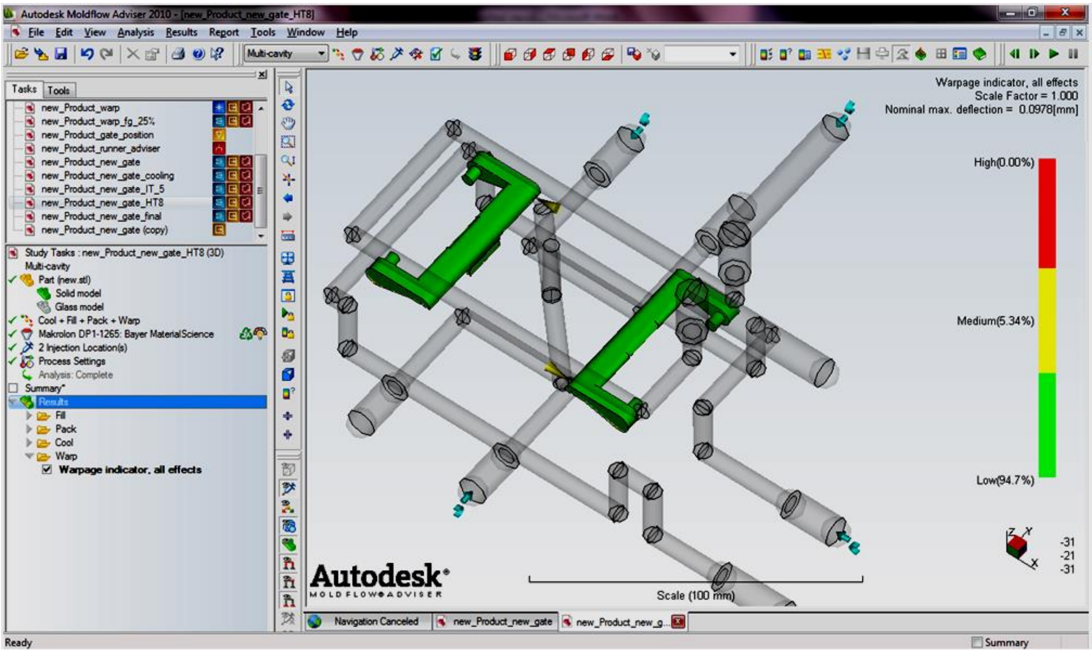


Figure 3.52 Variation of Warpage by Holding Time

Results at the holding time of 10s are shown in Figure 3.53 and Figure 3.54.

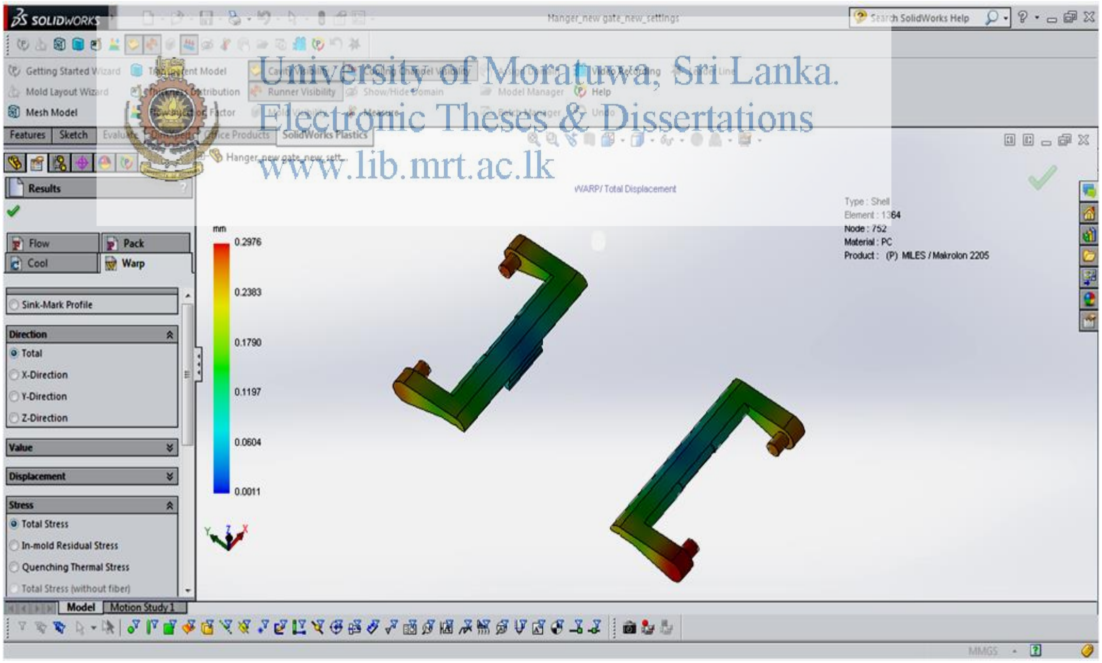

Autodesk Moldflow Adviser 2010 - [new_Product_new_gate_HT8]

Study Tasks : new_Product_new_gate_HT8 (3D)
Multi-cavity
Part (new.stl)
Cool + Fill + Pack + Warp
Makrolon DP1-1265: Bayer MaterialScience
2 Injection Location(s)
Process Settings
Analysis: Complete

☑ Warpage indicator, all effects

Warpage indicator, all effects
Scale Factor = 1.000
Nominal max. deflection = 0.0978[mm]

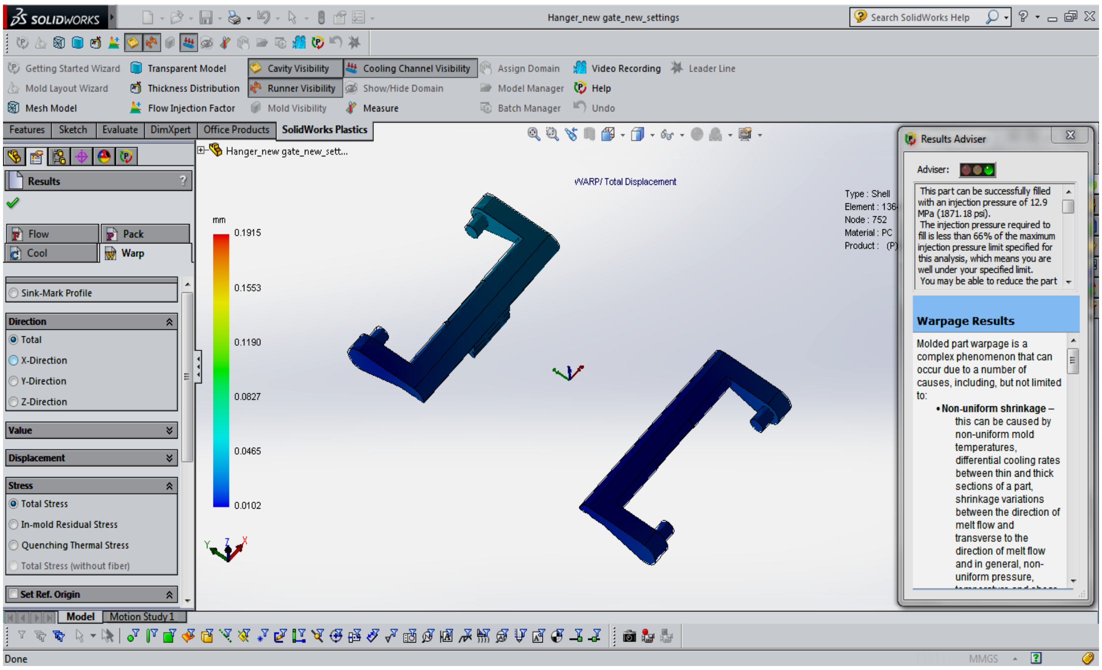
Figure 3.53 Warpage for Holding Time of 10s (AMA)


Type : Shell
Element : 1364
Node : 752
Material : PC
Product : (P) MILES / Makrolon 2205

Direction: ● Total
Stress: ● Total Stress

Figure 3.54 Warpage for Holding Time of 10s (SWP)

The total displacement is 0.2976 mm at the holding time of 10S. Solidworks plastic showed that it needs a holding time of 19S to get a lesser warpage value than 0.2mm. The total displacement is 0.1915 mm at holding time of 19s.



This part can be successfully filled with an injection pressure of 12.9 MPa (1871.18 psi).
The injection pressure required to fill is less than 66% of the maximum injection pressure limit specified for this analysis, which means you are well under your specified limit.
You may be able to reduce the part

Warpage Results

Molded part warpage is a complex phenomenon that can occur due to a number of causes, including, but not limited to:

- **Non-uniform shrinkage** – this can be caused by non-uniform mold temperatures, differential cooling rates between thin and thick sections of a part, shrinkage variations between the direction of melt flow and transverse to the direction of melt flow and in general, non-uniform pressure,

Figure 3.55 Warpage for Holding Time of 19s (SWP)

3.3.6. Cooling Layout

Cooling layout of a mould should be able to provide uniform cooling from each side to cavity or the product. Cooling layout depend on variables of cooling channel diameter, number of cooling channels and the placement of them. The sizing of cooling channels is dependent on the rate of cooling and temperature control, needed for the controlling part quality.

Cooling channels must be arranged to remove heat in a manner, so that the entire moulded part and runner system cool at the same rate. Where there are both thick and thin moulded-part sections, the cooling capacity of the system in the thick areas must be greater so that the thick sections cool at the same rate as the thin sections [21]. Core pins and outside corners of cores need special attention to maximize heat transfer into the cooling system. Heat pipes or high-conductivity material can be used to encourage better cooling [19].

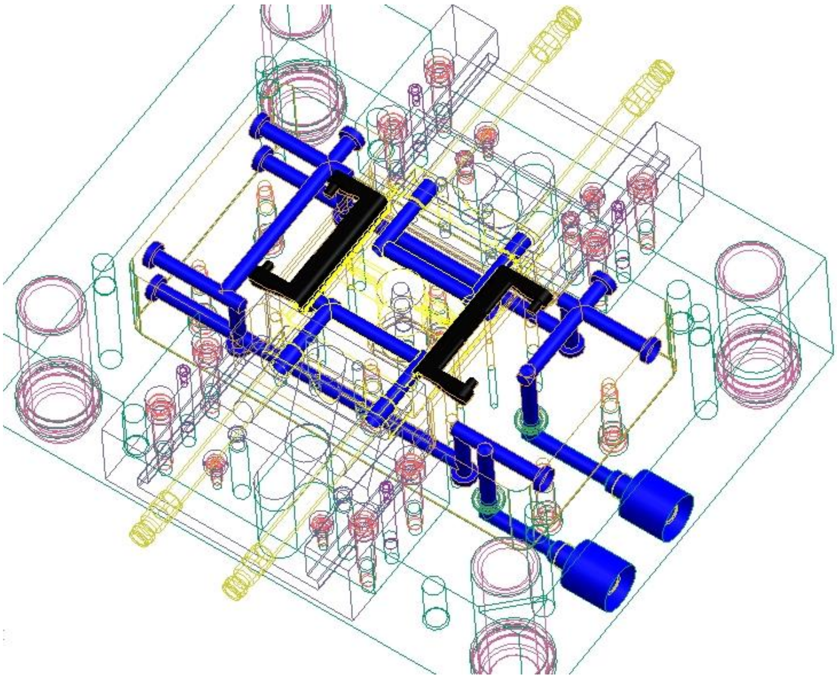


Figure 3.56 Cooling Layout of Ejector Half

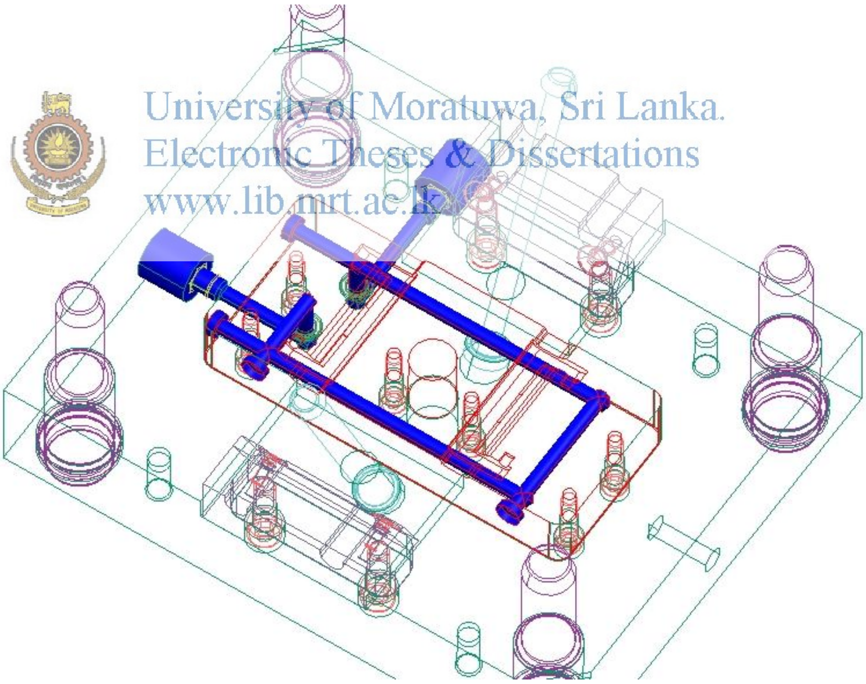


Figure 3.57 Cooling Layout of Injection Half

Existing cooling layouts of ejector side and injection side are shown in Figure 3.56 and Figure 3.57 respectively. Cooling layout analysis was done for existing layout using Autodesk Moldflow Adviser and results are shown in Figure 3.58 and Figure 3.59.

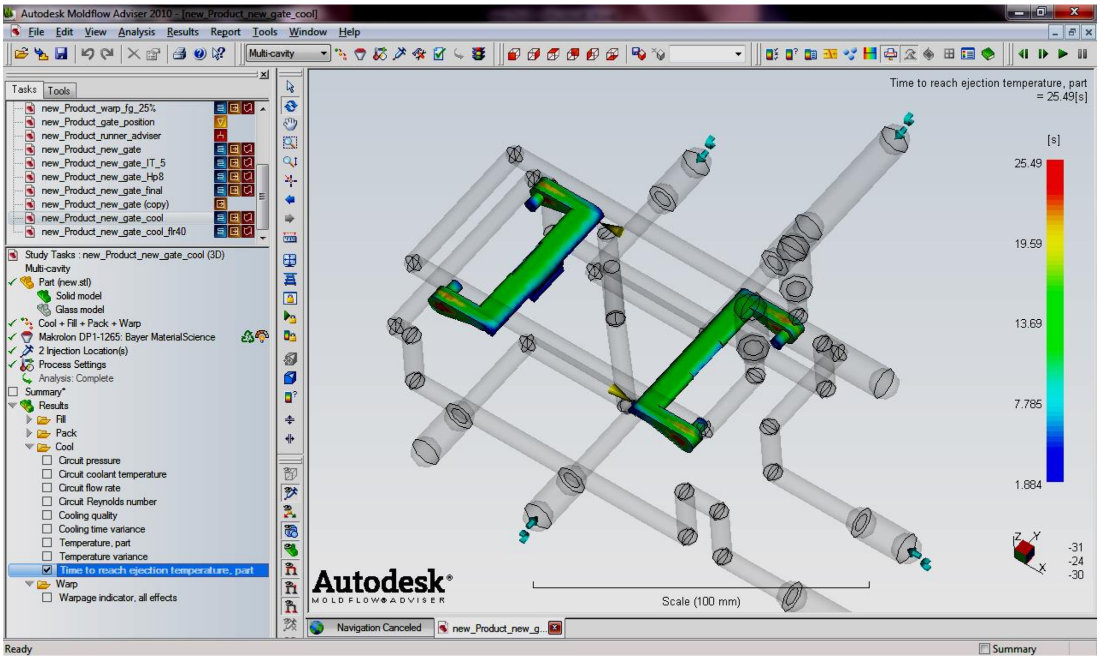


Figure 3.58 Time to reach Ejection Temperature (AMA)

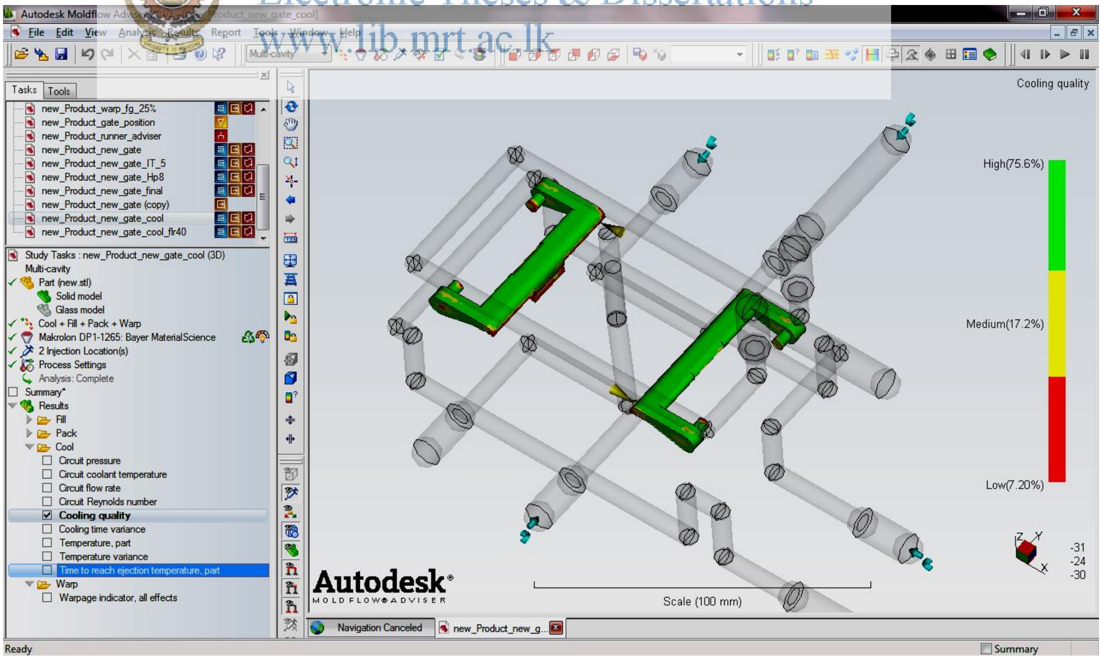
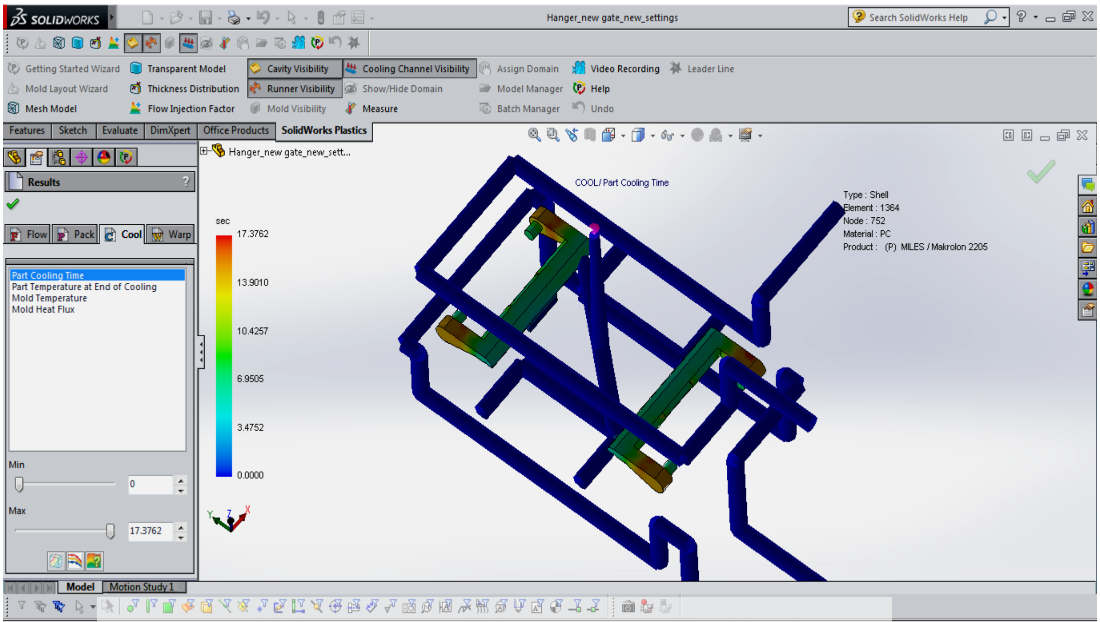


Figure 3.59 Cooling Quality (AMA)

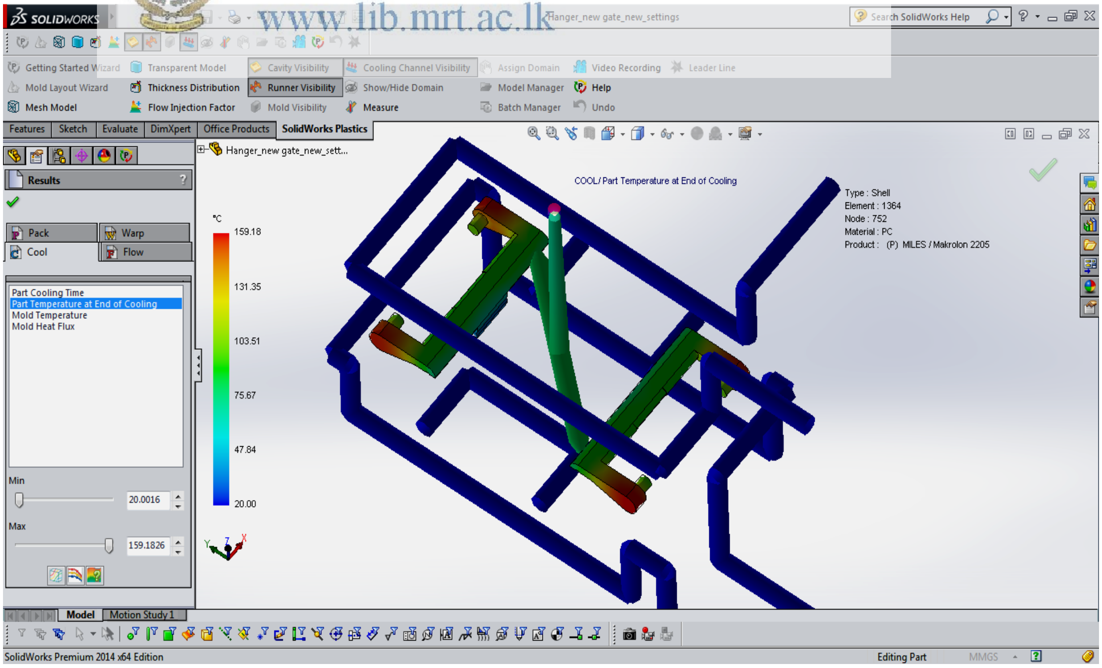
Result shows almost uniform cooling of part except the thicker sections. Also the cooling quality is much acceptable. So it proves the existing cooling layout is well balanced and fulfilling the requirements. According to above results, with new settings the maximum time needs to reach the ejection temperature is 26s.



COOL/ Part Cooling Time

Type : Shell
Element : 1364
Node : 752
Material : PC
Product : (P) MILES / Makrolon 2205

Figure 3.60 Part Cooling Time (SWP)



COOL/ Part Temperature at End of Cooling

Type : Shell
Element : 1364
Node : 752
Material : PC
Product : (P) MILES / Makrolon 2205

Figure 3.61 Part Temperature at End of Cooling (SWP)

It shows that the cooling time required to reach the part to ejection temperature is only 18s. So the actual cooling time requirement needs to be verified at the practical application. Quality of existing cooling layout was analysed by both above analysis to check any compulsory modification. Results prove the quality of cooling system is good enough for a quality product as modification of cooling layout is too difficult after hardening the cavity insert. Both analysis shows that the uniform cooling of the part limited the effect of warpage of this part.

3.3.7. Modifications

Summarised list of the proposed modifications are

- Modify the part geometry
- Change gate position
- Increase runner diameter up to 7.23mm
- Change process parameters as shown in Table 3.4

Table 3.4 Recommended Processing Parameters

Parameter	Previous Value	Proposed Value
Material	PC	PC
Part volume	2.8 cm^3	2.4 cm^3
Shot weight	35.03g	35.17 g
Article weight	25.13g	21.47 g
Melt temperature	295 [^{0}C]	295 [^{0}C]
Injection Speed	85 cm^3/s	85 cm^3/s
Injection Pressure	138 bar	132 bar -140 bar
Injection / fill time	4 S	5 S
Holding Time	5 S	6S-19S
Mould Temperature	80 [^{0}C]	80 [^{0}C]
Cooling Time	20 S	18S-26 S
Mould open Time	1 S	1 S
Mould close Time	2 S	2 S
Ejection time	2 S	2 S
Holding pressure	60 bar	110 bar
Coolant Temperature	20 [^{0}C]	20 [^{0}C]
Ambient Temperature	30 [^{0}C]	30 [^{0}C]

4 CASE STUDY

In early chapters, all modification proposals have been clearly defined and in this chapter it will be discussed about the practical application of those and the results obtained.

4.1. Modified Mould Design

The modified mould design is shown in appendix B.

4.1.1. Slider Modification

In the existing mould, as a first step, sliders had to be modified to comply with the new product design. Therefore a sub insert was fixed to fulfil that requirement which is shown in green colour in Figure 4.1. The sub insert and the pocket of the sub insert were created using a hardened work piece with the process of EDM wire cut.

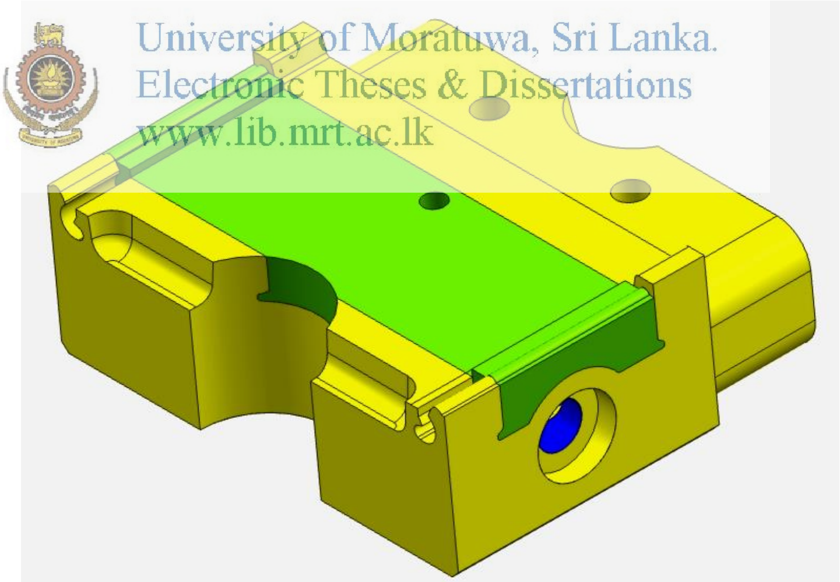


Figure 4.1 Slider Modification

After assembling, the outer surfaces were grinded to match surfaces properly. Actual part is shown in Figure 4.2 and the sub inserts are circled in red colour.

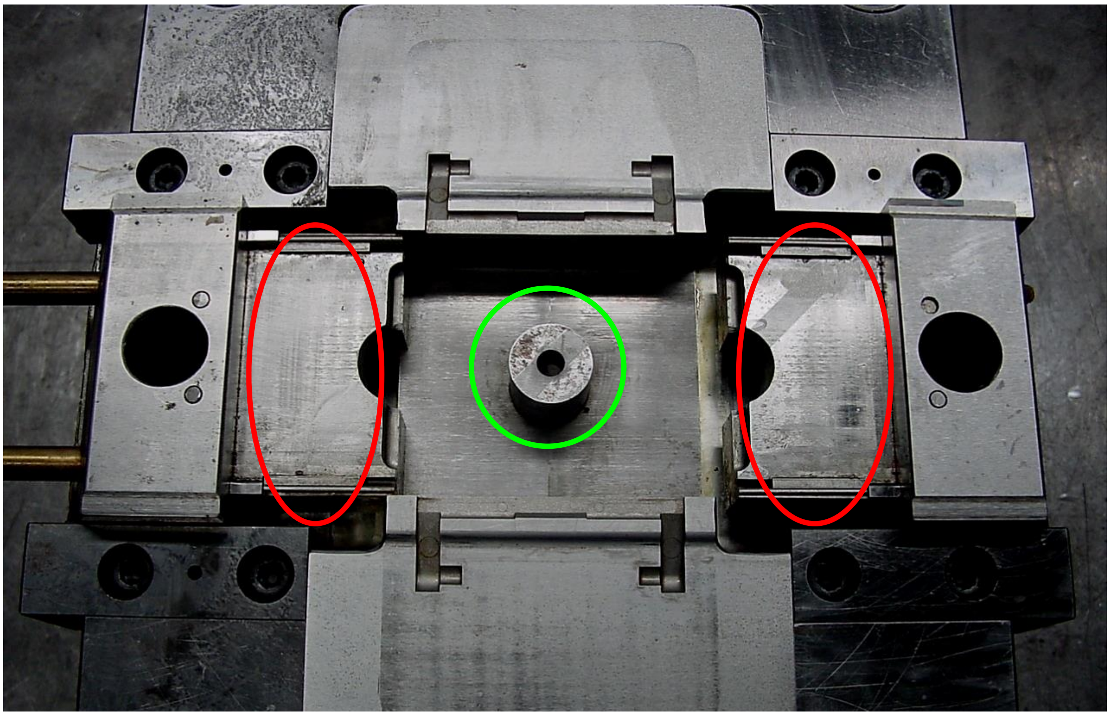


Figure 4.2 After the Slider Modification

4.1.2. New Gate Position with New Runner Section

The second step was making of new runner and gates. Before the modification is done, the runner was located in the centre of the part in the ejector side on sliders. After the modification if it remains there, the flow of molten material can easily go through it. So the puller insert which is circled in green colour in figure 4.2 is replaced with new one. The existing old gates at the injection side cavity insert were closed using laser welding. Excess welding material in the cavity was removed using EDM and Excess welding material in top surface of the insert was removed using surface grinding. These areas are shown in figure 4.3 using green circles. A new runner was machined by CNC machining and gates were produced by EDM. The modified insert is shown in Figure 4.3.

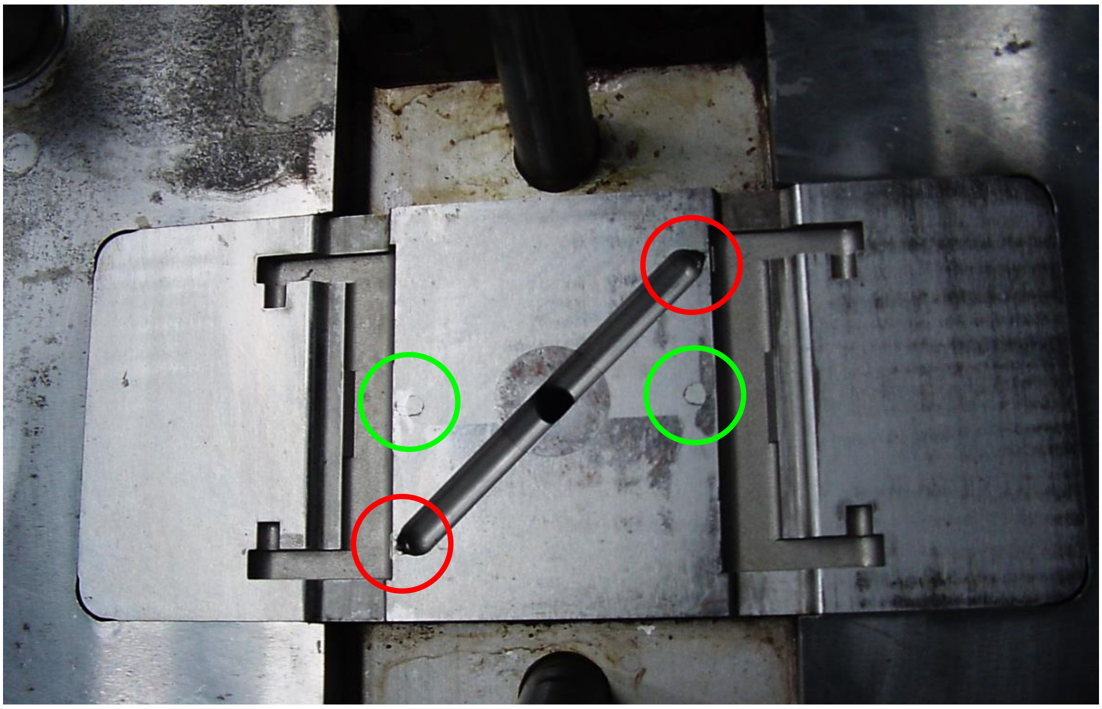


Figure 4.3 Modified Runner and New Gates

Completely modified mould assembly is shown in Figure 4.4.

Figure 4.4 Modified Mould Assembly

4.2. Sample Production and Results

Samples were produced with previous settings and following problems were detected.

i. Short filling

ii. Ejector pin marks

Then injection pressure and injection time were changed up to 140 MPa and 5S respectively. Then the short filling problem was solved, but the cooling time had to be increased up to 23S to avoid ejector pin marks.

After solving those defects warpage was measured and it has reduced to 0.314 mm. The holding pressure was increased while other parameters remain same and the observed results were taken into the graph shown in Figure 4.5.

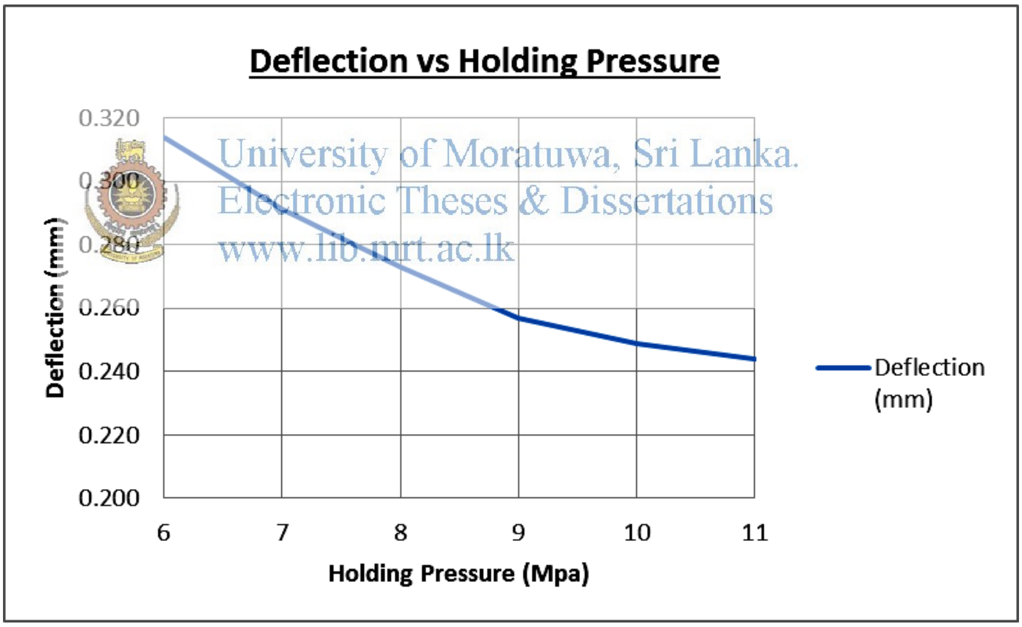


Figure 4.5 Variation of deflection by Holding Pressure

Finally holding time was increased while other parameters were unchanged and warpage was measured. Maximum deflections were taken into graph shown in Figure 4.6.

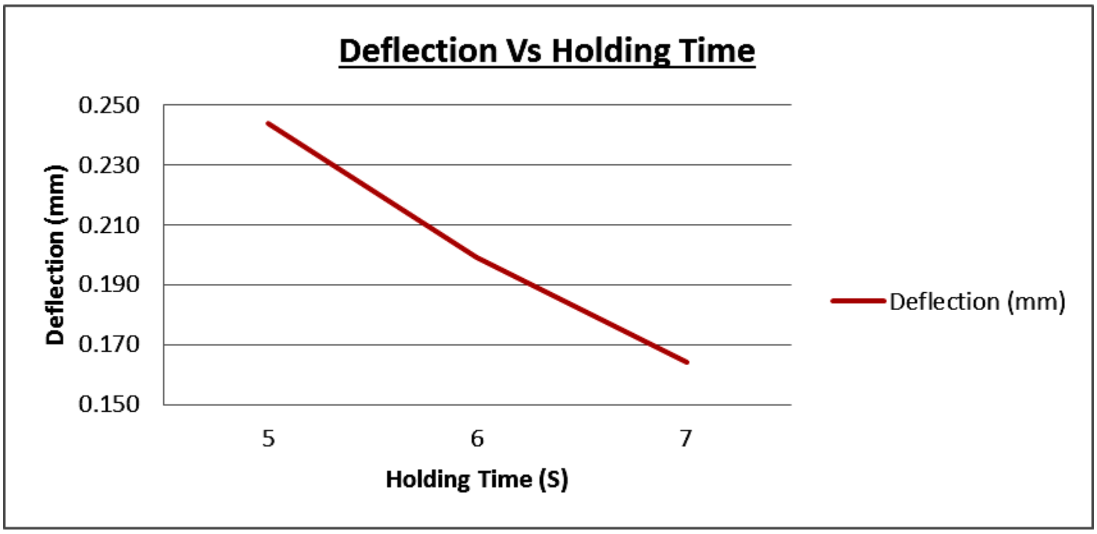


Figure 4.6 Variation of deflection by Holding Time

The new product dimensions were measured using CMM and the new values are as shown in Figure 4.7 and Figure 4.8. Here also ten articles were selected randomly and got the average values of them.

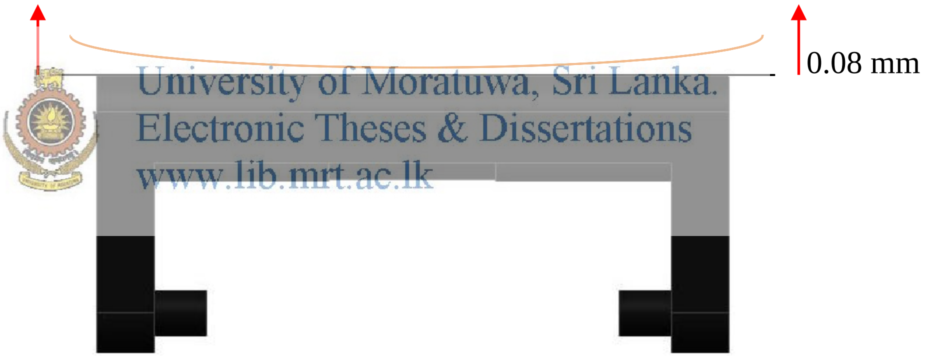


Figure 4.7 Top View

Figure 4.8 Front View

It can be clearly seen that the deflection occurs due to the warpage is well within the tolerance limit of 0.2 mm. Table 4.1 shows the comparison of practical process parameters with software analysis proposed values.

Table 4.1 Comparison of Processed Parameters

Parameter	Previous Value	Proposed process parameter	Processed value
Material	PC	PC	PC
Part volume	2.8 cm^3	2.4 cm^3	2.4 cm^3
Shot weight	35.03g	35.17 g	35.17 g
Article weight	25.13g	21.47 g	21.47 g
Melt temperature	295 [^{0}C]	295 [^{0}C]	295 [^{0}C]
Injection Speed	85 cm^3/s	85 cm^3/s	85 cm^3/s
Injection Pressure	138 bar	132 bar-140 bar	140 bar
Injection / fill time	4 S	5 S	5 S
Holding Time	5 S	6S-19S	7S
Mould Temperature	80 [^{0}C]	80 [^{0}C]	80 [^{0}C]
Cooling Time	20 S	18S-26 S	23S
Mould open Time	1 S	1 S	1S
Mould close Time	2 S	2 S	2S
Ejection time	2 S	2 S	2S
Holding pressure	60 bar	110 bar	110 bar
Coolant Temperature	20 [^{0}C]	20 [^{0}C]	20 [^{0}C]
Ambient Temperature	30 [^{0}C]	30 [^{0}C]	30 [^{0}C]

Warpage can be further reduced by changing process settings. But then the production cost will increase. The warpage reduction is required for higher quality of products, as warpage basically affect for the dimensional tolerance and the appearance of the product. Best practice should be able to identify the quality level and achieve that through lower cost. Therefore sticking to that phenomenon, application changes were done in gate point, product design and runner systems which produced relatively lower operation cost in comparison with process parameters. When the deflection caused by the warpage is within the tolerance limit, it means that the required quality level has been achieved. Further attempts to reduction of warpage will cause additional cost.

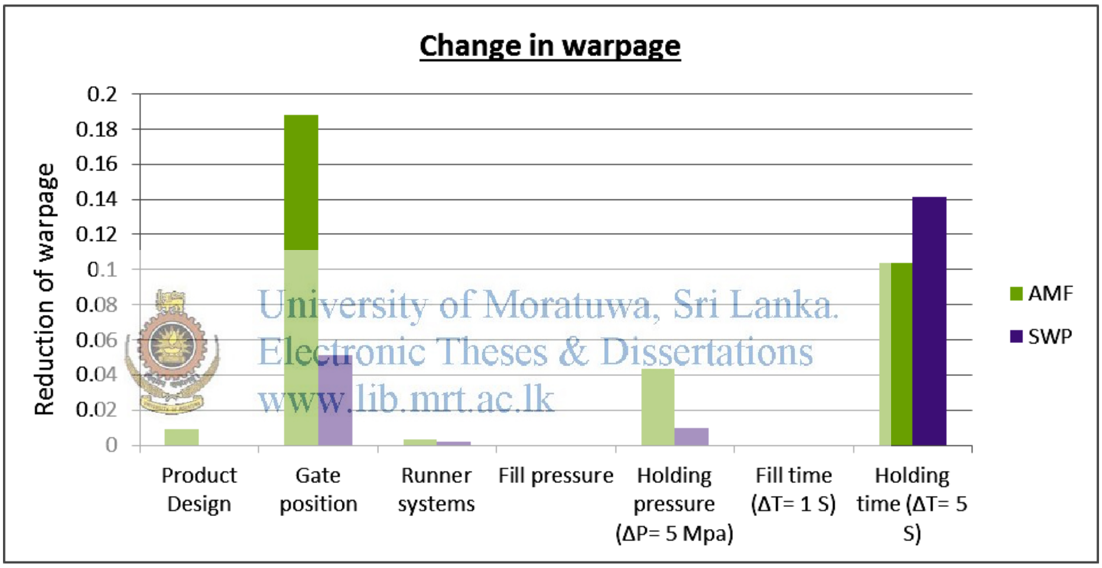


Figure 4.9 Effect of different factors for the Warpage Reduction

The results obtained from the analysis, are summarised in the graph shown in Figure 4.9. It shows the comparison of various factors for the warpage reduction. The changes done in gate position and holding time resulted more than 85% of reduction of warpage.

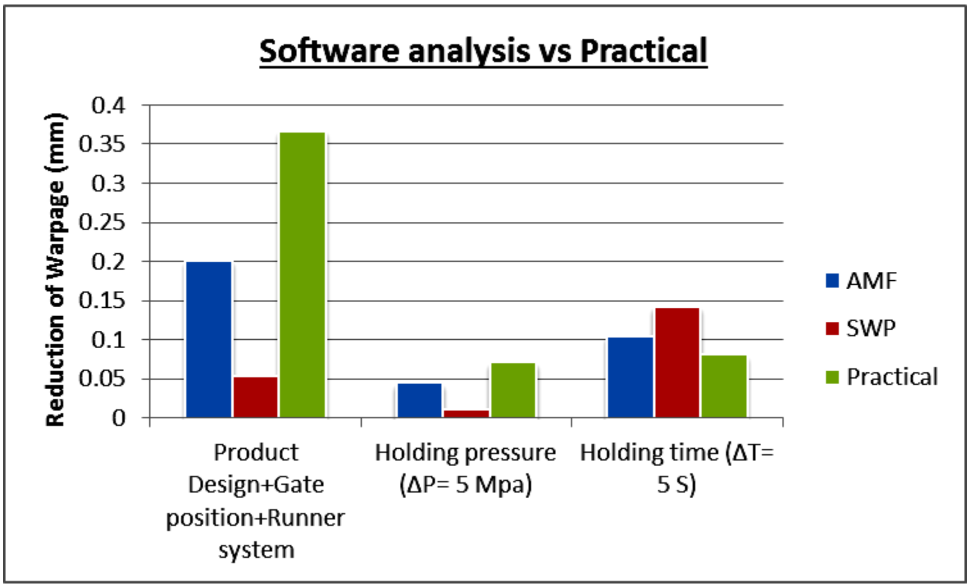


Figure 4.10 Comparison of Practical Results

Modification to product and the mould was more effective than what software analysis results in warpage reduction. But holding pressure and the holding time have given nearly equivalent result to software analysis as shown in Figure 4.10. Autodesk Moldflow was given closer value to the real value.

Table 4.2 Comparison of processed parameters with Software Analysis

Parameter	Solidworks Plastic	Autodesk Moldflow Advisor	Processed value
Injection Pressure	132 bar	140 bar	140 bar
Injection / fill time	5 S	5 S	5 S
Holding pressure	110 bar	110 bar	110 bar
Holding Time	19 S	6 S	7 S
Cooling Time	18 S	26 S	23S

Autodesk Moldflow Adviser has given closer values to real values. But the cooling time given by Autodesk Moldflow Adviser is higher than the real processed value. In Autodesk Moldflow Advisor 2010, it is not possible to set ejection temperature as this feature is not available with this version of the software. Results of Autodesk Moldflow Adviser show the part temperature at ejection as 34.34^{0}C which is shown in figure 4.11. It shows the result that it needs the cooling time of 26s to reach the final part to 35^0C.

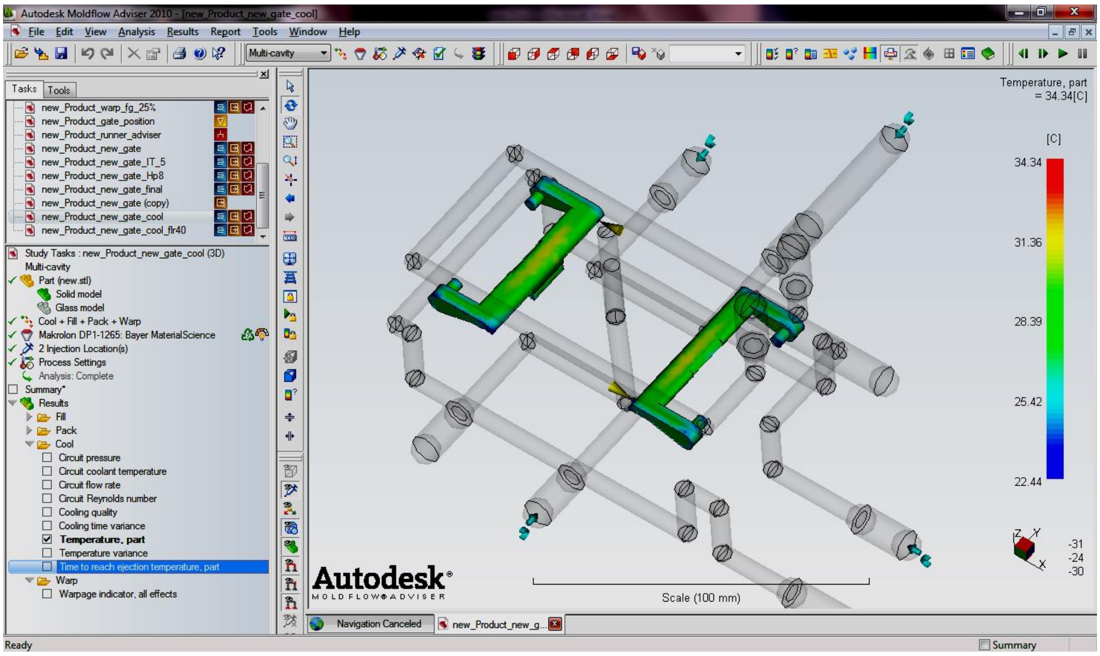


Figure 4.11 Temperature of Part

Solid Works Plastic showed higher warpage values, but lower variations with set variables. It needs holding time of 19s to reach the required warpage reduction. In this software it facilitates to set the ejection temperature of the final part which leads to obtain a cooling time which is relatively close to real value.

When reducing the warpage to the tolerance limit, it has increased cycle time by 6s. It has also increased the power consumption due to increase of pressures and the scrap materials. Therefore the maximum additional cost that can be born for the improvement of the quality of the certain product has to be decided in the production process. Every product needs an analysis for warpage reduction. The position and the magnitude of affecting the above discussed factors depend on the product. Prior analysis will lead to save money as well as time. Also it gives a chance to decide and obtain the warpage reduction level according to product requirement.

4.3. Future Improvements

4.3.1. Improved Product Design

In order to reduce the warpage and to get a uniform thickness the product improvement as shown in Figure 3.12 can be applied when manufacturing a new mould. But when such a modification is done the size of the mould will become lager and it will increase the initial cost associated with manufacturing. But as this will reduce the cooling time and product weight, the cost of production will reduce. There will be a very high reduction of cooling because of uniform thickness of the part as verified by the results shown in Figure 4.12 and Figure 4.13. Cooling time will reduce to the range of 15S - 17S from 23S.

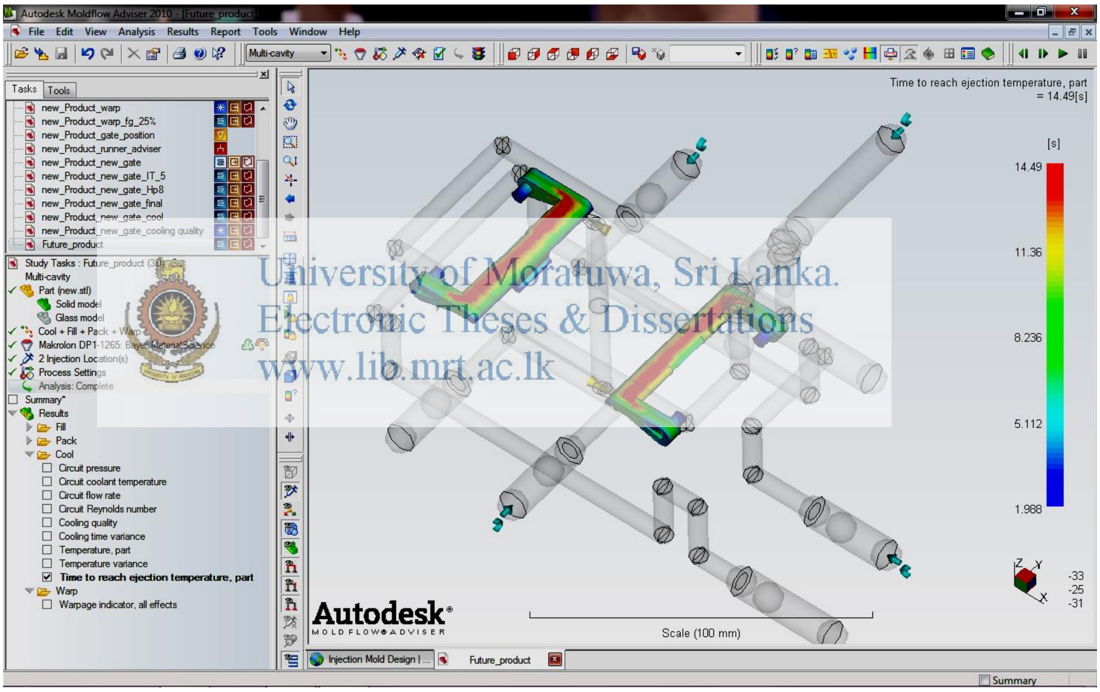
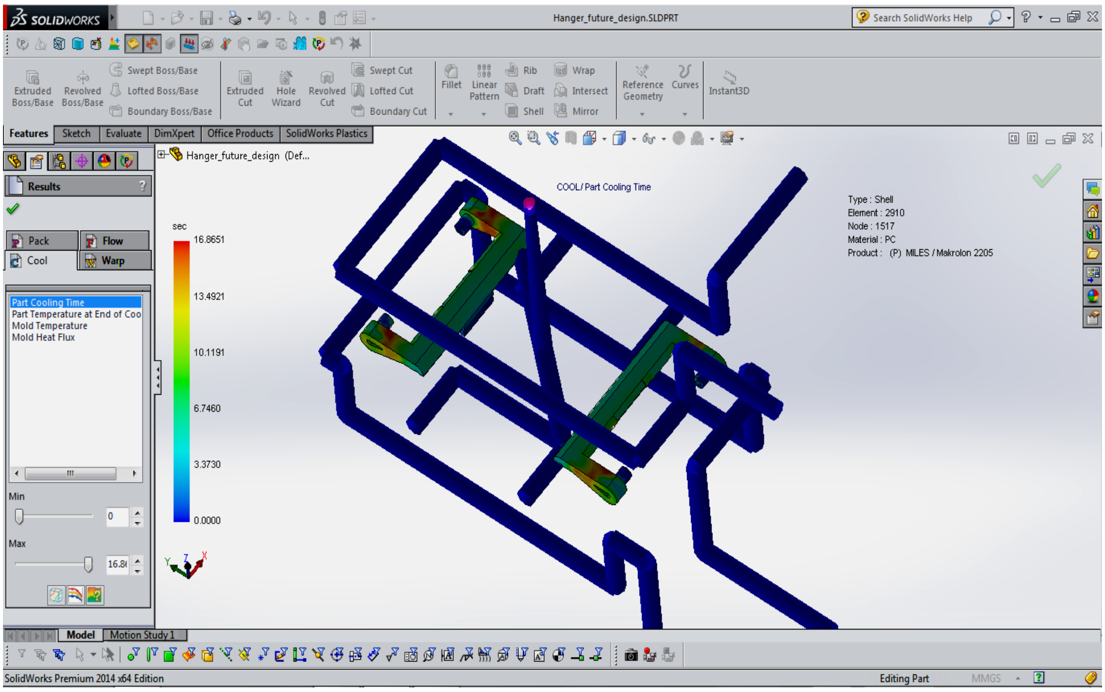


Figure 4.12 Cooling Time (AMA)



Type : Shell
Element : 2910
Node : 1517
Material : PC
Product : (P) MILES / Makrolon 2205

Figure 4.13 Part Cooling Time (SWP)

As shown in Table 4.3, though the material saving is smaller, the total reduction of operating cost is relatively high as both cooling time and shot weight effects for the operation cost.

Table 4.3 Weight Reduction

	Initial Product		Modified Product with Single Gate		Future Design	
	Weight(g)	Percentage (%)	Weight(g)	Percentage (%)	Weight(g)	Percentage (%)
Product	25.13	71.73	21.47	61.05	20.73	60.21
Runner	9.90	28.27	13.70	38.95	13.70	39.79
Total shot (product + runner)	35.03		35.17		34.43	

4.3.2. Conformal Cooling

The method of conformal cooling not only reduces the cooling time, but also the warpage due to uniform cooling. But this method has to be applied at the beginning of the design process as this method cannot be applied to the existing insert. Since the cost of this process is very high, the cooling time reduction, product complexity, quality requirement and the quantity requirement will decide the selection of this process.

5 CONCLUSION

To minimise the warpage of any product with minimum cost, it needs to consider the main affected factors in following sequence. This is a general scenario for all mould makers.

i. Part Geometry

Referring to this concept, it needs to maintain uniform thickness throughout the part at the same time minimizing product weight.

ii. Gate Location

Gate location should be selected in such manner that the part should have uniform flow pattern.

iii. Cooling Layout

Cooling line arrangement should be designed to obtain uniform cooling rate in all areas of the cavity.

iv. Runner System

Adhere to select circular cross section for runner with minimum possible runner length.

v. Holding Pressure

Stick to maintain minimum holding pressure which gives required warpage limit.

vi. Holding Time

Keep minimum holding time which gives required warpage limit.

5.1. Future Work

Researches on following areas will improve this process further.

- Verification of the methodology for other materials
- Shapes and gate locations of products
- Effectiveness of conformal cooling

References

[1] K. Prashantha, J. Soulestin, M. F. Lacrampe, E. Lafranche, P. Krawczak, G. Dupin, and M. Claes, “Taguchi analysis of shrinkage and warpage of injection-moulded polypropylene/multiwall carbon nanotubes nanocomposites,” *Express Polym. Lett.*, vol. 3, no. 10, pp. 630–638, 2009.

[2] P. Shrinkage, “Controlling Mold and Post-Mold Shrinkage and Warpage,” pp. 107–130, 2013.

[3] B. S. Reddy, J. S. Kumar, V. K. Reddy, and G. Padmanabhan, “Application of soft computing for the prediction of warpage of plastic injection molded parts,” *J. Eng. Sci. Technol. Rev.*, vol. 2, no. 1, pp. 56–62, 2009.

[4] G. Menges, W. Michaeli, and P. Mohren, Georg Menges , Walter Michaeli , Paul Mohren "How to Make Injection Molds.”

[5] A. Kramschuster, R. Cavitt, D. Ermer, Z. Chen, and L. S. Turng, “Quantitative study of shrinkage and warpage behavior for microcellular and conventional injection molding,” *Polym. Eng. Sci.*, vol. 45, no. 10, pp. 1408–1418, 2005.

[6] K. Cho and I. Jeon, “Numerical analysis of the warpage problem in TSOP,” *Microelectron. Reliab.*, vol. 44, no. 4, pp. 621–626, 2004.

[7] P. S. F. F. Alves, “Shrinkage and warpage behaviour on injection moulding parts,” 2008.

[8] E. Bociaga, T. Jaruga, K. Lubczyńska, and A. Gnatowski, “Warpage of injection moulded parts as the result of mould temperature difference,” *Arch. Mater. Sci. Eng.*, vol. 44, no. 1, pp. 28–34, 2010.

[9] Z. Shayfull, M. F. Ghazali, M. Azaman, S. M. Nasir, and N. A. Faris, “Effect of Differences Core and Cavity Temperature on Injection Molded Part and Reducing the Warpage by Taguchi Method,” *Int. J. Eng. Technol. IJET-IJENS*, vol. 10, no. 6, pp. 133–140, 2010.

[10] S. Kamaruddin, Z. A. Khan, and S. H. Foong, “Quality characteristic improvement of an injection moulding product made from blends plastic by optimizing the injection moulding parameters using Taguchi method,” *Int. J. Plast. Technol.*, vol. 14, no. 2, pp. 152–166, 2010.

[11] P. K. K. Kennedy, "*Practical and Scientific Aspects of Injection Molding Simulation*." 2008.

[12] M. Hayaty and M. H. Beheshty, “Shrinkage, Cure Characterization and Processing of Unsaturated Polyester Resin Containing PVAc Low-profile Additive,” *Iran. Polym. J.*, vol. 13, no. 5, pp. 389–396, 2004.

[13] S. Asproiu and E. Strǎjescu, “Influence of mold properties on the quality of molded parts,” *UPB Sci. Bull. Ser. D Mech. Eng.*, vol. 69, no. 3, pp. 39–46, 2007.

[14] K. S. R. Wilkinson, E.A. Poppe, Karl Leidig, “Engineering Polymers : the ‘ Top Ten ’ Injection Moulding Problems,” *DuPont Tech. Rep.*, pp. 1–4.

[15] S. Mohamed, M. Yusoff, W. a N. Harun, W. a N. Hamid, and E. Ramly, “A Plastic Injection Molding Process,” *Design*, vol. 41, pp. 1–16, 2007.

[16] C. T. Wong, S. Shamsuddin, I. N, and A. M. S. Hamouda, “Design and Simulation of Plastic Injection Moulding Process,” *Pertanika J. Sci. Techno!. Suppl.*, vol. 12, no. 2, pp. 85–99, 2004.

[17] Y. Q. Lu, P. L. Liu, X. M. Ding, and Y. S. Chung, “Mould Conceptual Design Method and System,” pp. 323–329.

[18] A. J. Pontes, “Shrinkage and Ejection forces in injection molded products,” p. 234, 2002.

[19] D. Stan, A. Tulcan, L. Tulcan, and T. Iclanzan, “Influence Factors on the Dimensional Accuracy of the Plastic Parts,” no. 1, pp. 119–124, 2008.

[20] M. H. E. Van Der Beek, G. W. M. Peters, and H. E. H. Meijer, “Characterization of polymers for improved shrinkage prediction in micromolding .,” no. C, pp. 1–4.

[21] V. Speranza, U. Vietri, and R. Pantani, “Monitoring of injection moulding of thermoplastics: Adopting pressure transducers to estimate the solidification history and the shrinkage of moulded parts,” *Stroj. Vestnik/Journal Mech. Eng.*, vol. 59, no. 11, pp. 677–682, 2013.

[22] A. Peng, Y. Chang, A. Yang, and V. Yang, “3D Fiber Orientation and Warpage Analysis of Injection-Molded Throttle Valve,” *Polymer (Guildf).*, no. 7.

[23] R. Bonart, “Thermoplastic elastomers,” *Polymer (Guildf).*, vol. 20, no. 11, pp. 1389–1403, 1979.

[24] J. Wang, “PVT Properties of Polymers for Injection Molding,” *Some Crit. Issues Inject. Molding*, 2012.

[25] C. Schauer, “Inherent problems in early IM,” 2007.

[26] “Gate Location Optimization In Injection Molding Based On Feasible Space" Ming Zhai , NERC for APPT , Zhengzhou University , Zhengzhou , China Haiwen Wang , School of Art and Information Engineering , Dalian Polytechnic University , China Abstract Opt,” pp. 1–5.

[27] Moldflow Corporation, “Moldflow Design Guide,” pp. 1–346, 2006.

[28] B. J. Hasenauer, D. Küper, J. E. Laumeyer, and I. Welsh, “Correct gate location,” pp. 15–18.

[29] Prof T. Gutowski, “Injection Molding.” 2002.

[30] J. Shen, “Design and Molding Simulation of a Plastic Part,” 2010.

[31] H. S. Park and X. P. Dang, “Design and Simulation-Based Optimization of Cooling Channels for Plastic Injection Mold,” *New Technol. Innov. Res.*, pp. 19–45, 2012.

[32] V. Maag and K.-H. Küfer, “Optimal Cooling in Injection Molding and Die Casting,” no. June, pp. 1–5, 2008.

[33] Lars-Erik Rännar, *On Optimization of Injection Molding Cooling*, no. April. 2008.

[34] O. A. Mohamed, S. H. Masood, and A. Saifullah, “A Simulation Study of Conformal Cooling Channels in Plastic Injection Molding,” *Int. J. Eng. Res.*, vol. 2, no. 5, pp. 344–348, 2013.

[35] C. Özek and Y. H. Çelık, “Calculating Molding Parameters in Plastic Injection Molds with ANN and Developing Software,” *Mater. Manuf. Process.*, vol. 27, no. January 2015, pp. 160–168, 2012.

[36] A. G. Smith, L. C. Wrobel, B. A. Mccalla, P. S. Allan, and P. R. Hornsby, “Optimisation of Continuous and Pulsed Cooling in Injection Moulding Processes.”

[37] J. Meckley and R. Edwards, “A Study on the Design and Effectiveness of Conformal Cooling Channels in Rapid Tooling Inserts,” *Technol. Interface Journal/Fall*, vol. 10, no. 1, 2009.

[38] Y. Wang, K. M. Yu, C. C. L. Wang, and Y. Zhang, “Automatic design of conformal cooling circuits for rapid tooling,” *CAD Comput. Aided Des.*, vol. 43, no. 8, pp. 1001–1010, 2011.

[39] R. Hill, “Dynamic Temperature Control in Injection Molding with New Conformal Heating / Cooling System,” pp. 5–11, 2010.

[40] E. Sachs, E. Wylonis, S. Allen, M. Cima, and H. Gu, “Production of Injection Molding Tooling With Conformal Cooling Channels Using the Three Dimensional Printing Process,” *Polym. Eng. Sci.*, vol. 40, no. 5, pp. 1232–1247, 2000.

[41] M. M. Bunch and P. Delp, “Prediction of Shrinkage in Plastic Injected Parts Due to Cooling Introduction to the Project.”

[42] T. Kimerling, “Injection Molding Cooling Time Reduction and Thermal Stress Analysis,” pp. 1–5, 2002.

[43] D. Lee, W. A. Chen, T. W. Huang, and S. J. Liu, "Factors influencing the warpage in in-mold decoration injection molded composites," *Int. Polym. Process.*, vol. 28, no. 2, pp. 221–227, 2013.

[44] S. Teklehaimanot and D. Thesis, "Simulation and Design of a plastic injection Mold," pp. 1–53, 2011.

[45] R. A. Malloy, *Plastic Part Design for Injection Molding*. 2010.

[46] N. Mehat, S. Kamaruddin, and A. Othman, "Reducing the Shrinkage in Plastic Injection Moulded Gear via Grey-Based-Taguchi Optimization Method," *Proc. World Congr. Eng. 2012 Vol III*, vol. III, pp. 1–4, 2012.

[47] S. Kuriakose, S. K. George, and P. V. Mathew, "Study of Moulding Defects in Automobile Relay Cover," vol. 2, no. 6, pp. 134–138, 2012.

[48] H. P. Kale and U. V Hambire, "Review on Optimization of Injection Molding Process Parameter for Reducing Shrinkage of High Density Polyethylene (HDPE) material," vol. 4, no. 4, pp. 2847–2850, 2015.

[49] D. G. Ahn, D. W. Kim, and Y. U. Yoon, "Optimal injection molding conditions considering the core shift for a plastic battery case with thin and deep walls," *J. Mech. Sci. Technol.*, vol. 24, no. 1, pp. 145–148, 2010.

[50] A.L. Gershon, L.J. Gyger, Jr., H.A. Bruck and S.K. Gupta, "Thermoplastic Polymer Shrinkage in Emerging Molding Processes," *J. Chem. Inf. Model.*, vol. 53, no. 9, pp. 1689–1699, 2013.

[51] Z. Kamal, "Trouble Shooting In Plastic Injection Molding Machines," *Public Health*, p. 125, 2007.

[52] V. Shah, "Scientific Approach to Injection Molding What is Scientific Approach to Injection Molding ?," 2004.

Appendix A – Mould Assembly Drawing

Appendix B – Modified Mould Assembly Drawing

Appendix C – Product Drawing

Appendix D – Properties of Polycarbonate

Rheological properties	**Value**	**Unit**
Melt volume-flow rate, MVR	19	cm³/10min
Temperature	300	°C
Moulding shrinkage, parallel	0.7	%
Moulding shrinkage, normal	0.7	%
Ejection temperature	130	°C
Mechanical properties		
Tensile Modulus	2400	MPa
Yield stress	66	MPa
Yield strain	6	%
Nominal strain at break	>50	%
Tensile creep modulus, 1h	2200	MPa
Tensile creep modulus, 1000h	1900	MPa
Test specimen production		
Injection Moulding, melt temperature	280	°C
Injection Moulding, mould temperature	80	°C
Injection Moulding, injection velocity	200	mm/s
Injection moulding		
Drying Temperature	120	°C
Drying Time	3-4	Hrs
Maximum moisture content	0.02	%
Vent Depth	0.025-0.075	mm