Copyright is owned by the Author of the thesis. Permission is given for a copy to be downloaded by an individual for the purpose of research and private study only. The thesis may not be reproduced elsewhere without the permission of the Author.

PREDICTION OF PROBLEMS IN INJECTION MOULDED PLASTIC PRODUCTS WITH COMPUTER AIDED MOULD DESIGN SOFTWARE

A thesis presented in partial fulfilment of the requirements

for the degree of

Masters In Technology in

Manufacturing and Industrial Technology

at

Massey University, New Zealand

Peter Ian Cayzer

1996

ABSTRACT

Several new technologies to assist plastic injection moulding companies have been developed in the last twenty years. A number of computer software programs are now available which could revolutionise mould design. The most exciting aspect of the Computer Aided Mould Design (CAMD) software is the effect it has on reducing the lead time required to produce a working mould from a product concept.

The application of the new technology for designing moulds, however, has been slow in New Zealand. One of the main reasons for the slow progress is the perceived value of the software or consulting services. Many injection moulding companies who design and manufacture moulds do not realise the great potential of CAMD software to save many hours of mould changes and volume of polymer material, even when the program is used after the mould has been made. However, the true benefits are only seen when the mould is designed using CAMD before the mould has been manufactured. Moulds manufactured correctly the first time save a great deal of time, energy and money. The value of the software is not completely understood by injection moulding manufacturers. They perceive the immediate benefits, however, the ongoing benefits are not recognised.

A project was carried out to demonstrate the potential of CAMD software in determining moulding problems in existing injection moulded products. Four products, two of which were supplied by an injection moulding company, that had moulding problems, were simulated using Moldflow, a CAMD software package. The results of the simulation were compared with the actual moulding problems.

It was found that the Moldflow simulation results described the problems occurring in the moulds accurately. Moulding problems included warpage, air traps and weld lines in poor positions and flow marks. Warpage is a major problem in injection moulded products. Even simple products can warp if not designed correctly.

The only problems Moldflow did not identify, and does not claim to, were the flow marks caused by jetting and splashing of plastic as it entered the cavity. The designer must be aware of the problems caused by jetting and design gates to avoid it.

Moldflow, and other CAMD software, are beneficial tools for the mould designer. The advantages of CAMD include short mould development time, shorter lead times from concept to production, reduction in the amount of material used, fewer changes to machine settings and predictable, repeatable quality. These benefits are not only savings in the mould design and manufacture, they also continue on into the processing of the product since less material is used in the product and machine down time caused by moulding problems is greatly reduced.

ACKNOWLEDGMENTS

To my supervisors, Ralph Ball, Alan Wright and David Morgan for their advice and assistance in developing the study this thesis addresses. Your guidance was invaluable at times when I needed to be reminded of the purpose of the project.

To Moldflow Australia Pty. Ltd. for their help and training during the past one and a half years. Their aid in the provision of training has made this study possible. I particularly wish to thank Derek Hain and Kate Hillman for their friendliness and assistance during the three week training course.

To Dr. Simon Hurley, whose numerous words of encouragement and advice guided me to the completion of this work. I am sincerely grateful for your support.

To Sunbeam Corporation Ltd. and Napier Tool and Die Ltd., for providing projects for the study. I particularly wish to thank Craig Dais and Peter Whitburn of Sunbeam whose cooperation has been greatly appreciated. Thank you for your support.

To my family and friends, too many to name, I appreciate your love and support over the past two years which has kept me going during the struggles.

To my wife, Fiona, whom I love dearly and thank for being patient and understanding of my needs during this extended life episode.

This thesis is dedicated to my parents whose persistent encouragement and vision has led me on an exciting and educational journey. Thank you both.

My life journey has given me eternal hope in Jesus Christ and I must finish by giving Him thanks for the endless grace, peace and love He gives.

TABLE OF CONTENTS

LIST OF FIGURES

LIST OF TABLES

CHAPTER ONE

INTRODUCTION TO MOULD DESIGN

Mould design for the injection moulding process has changed from a technique requiring years of experience to one that incorporates experience and design simulation. This chapter gives a brief overview of the evolution of Computer Aided Mould Design software, presents a summary of the body of this thesis and introduces the hypothesis this thesis addresses.

1.1 Summary

Injection moulding is the most widely used process in the plastics industry for manufacturing plastic products. The injection moulding process requires three main components; an injection unit, a clamping device and a mould. The mould is the subject of this thesis, in particular, the design of the mould, how well Computer Aided Mould Design (CAMD) software can predict moulding problems and how useful the software is as a cost effective adjunct to the mould designer's judgment.

In the past, mould design has been an art, rather than a science. This is mainly due to the difficulties in moulding polymers into the required shapes using a high pressure process like injection moulding. The characteristics of polymer flow under high pressures and the intricacies of the injection moulding process were understood by very few designers, hence many designs required a great deal of rework before reaching production status. This continues to be a problem with mould design since the rheology of polymers under high stresses and changing temperatures is complex and difficult to solve.

The problem could not really be overcome until computers with the power to solve the complex simultaneous flow and heat equations were available. Now that these computers are available, it is possible to not only solve the equations, but also provide graphical results that give a clear picture of what occurs in the mould as the polymer flows through it, cools and freezes. These displays are available in software packages specially designed for assisting in the design of injection moulding moulds.

The software packages are aids to the designer. A good mould designer is required to analyse the output from the mould model simulation and make the necessary changes to the model to improve the design. The designer requires not only training in using the software package, he/she also requires knowledge of the properties of polymers and mould design procedures and concepts. The latter requirements may take several years to build up to a point where the mould designer is proficient at what he/she does.

A survey carried out in 1994 by Xaioping Pan discovered that 83% of the New Zealand plastics industry realised that "tool and die making had become an important part of the New Zealand Plastics Industry." Several of the comments from the industrial survey included: "In an effort to become cost competitive, die making and design is a key component." "High quality tool and die making is a key to success." "Sometimes it is the limitation of the die or tool that prevents the product being made."

The Production Technology department is eager to develop expertise in injection moulding technology. This thesis is part of the development. Moldflow, the sophisticated mould design software has been made available to the department for teaching and research purposes by Moldflow of Australia.

The introduction to this thesis will describe the project objectives in section 1.2 and the outline of the thesis in section 1.3.

1.2 Project Objectives

There are three distinct objectives this thesis addresses:

1. To determine the benefits of computer software for the analysis of injection moulds through verification techniques. This will be carried out by studying several products, analysing them using Moldflow and drawing conclusions regarding the effectiveness of the software to predict moulding problems in the mould design.

2. To increase the Production Technology Department's awareness of available technology for injection moulding.

3. To develop an understanding of the concepts behind Computer Aided Mould Design software.

Hypothesis : Computer Aided Mould Design software can accurately predict moulding problems, namely: quality, processability and in-service requirements in a product that are not met.

1.3 Outline of the report

Mould design is an important part of the moulding process. A well designed, manufactured mould will not only last, it will also be able to produce a consistent product. Mould design can be complicated, especially when small tolerances, thin sections and complex structures are required. A design that performs the task well may not necessarily mould well in the injection moulding machine. It is therefore important to use a procedure to design the best possible mould.

Computer Aided Mould Design is a major development in the mould design field. However, it has not, as yet, been widely adopted in New Zealand. More companies are beginning to see the rewards of such a product, but this has been slow. Gallagher Plastics and Fisher and Paykel are two New Zealand companies who have begun to use CAMD in the last five years. Other companies, such as Whitfield Design, Sunbeam and Coxen and Standish have utilised the services of one or more of the CAMD consulting agents in New Zealand or Australia. Mould designers may require further educating in the benefits of Computer Aided Mould Design so that they may understand and see the real benefits of such an aid. Most injection moulding companies who have used CAMD consulting in the past have considered the cost of the analysis too high and have discarded the need for it.

To gain an understanding of the great potential of Computer Aided Mould Design software, a familiarity with the injection moulding process and the rheology of polymers is required, as well as a solid grasp of mould design. This thesis includes a discussion of the mould design tools currently available. The background sections will be followed by the reasoning behind using CAMD. Chapter 4 develops the research methodology and chapter 5 presents the projects that have been carried out using one of the software packages. The aim of these projects was to determine the ability of Computer Aided Mould Design software to predict possible problems that occur in the mould. These problems may relate to warpage, poor material flow, poor design, air traps, etc which result in poor product quality, poor processability and in-service requirements that are not met.

A discussion of any further work that could be carried out in this area and an overview of the results of the projects concludes the thesis.

CHAPTER TWO

MOULD DESIGN TOOLS

Plastic products are only as good as the design: the product design and the mould design. A good procedure should be followed to produce a mould design that will perform the task required. Two types of design methods exist which can be used within the mould design procedure: Conventional Mould Design and Computer Aided Mould Design. This chapter discusses both of these methods.

2.1 Introduction

Design methods and procedures, previous designs and software programs are aids to assist the designer in producing an effective injection moulding mould design that will produce a consistent plastic product.

A complete design process begins at the concept stage and carries all the way through to production and even disposal of a product or system. Injection moulding mould design is one portion of the whole design process. An example of a mould design methodology is given in appendix A1. In order to benefit from such a process, key people should be chosen as required. A design team could consist of a design engineer, production personnel, sales personnel, management and even the suppliers and customers.

This chapter reviews the design tools used for the physical design of an injection mould. There are two distinct methods of mould design: Conventional Mould Design and Computer Aided Mould Design. Conventional Mould Design is still commonly used today and is a valid technique for designing and testing moulds. The main problem with Conventional Mould Design, however, is the length of development time since the mould designer can never be sure the mould design will work the first time it is used.

Computer Aided Mould Design (CAMD) has been developed to remove some of the guess work of mould design and is being successfully exploited by several injection moulding companies in New Zealand and around the world. CAMD utilises computer software to assist the designer with the mould design by providing graphical results that describe the flow of the plastic in the mould. The designer must interpret the results and make changes to the design to improve the quality, appearance and processability of the product.

Both Conventional and Computer Aided Mould Design are based on several mould design principles. A mould designer would normally use the design principles and build on them with gained experience. A set of basic design principles is presented in section 2.2. They are generic and may be employed in the Conventional and Computer Assisted Mould Design methods.

CAMD includes several additional principles which relate to the graphical output provided by the mould design software, for example, pressure and temperature ranges, maximum shear stress and shear rate. These principles can not be used without the graphical output and are therefore meaningless in the Conventional Mould Design approach.

This chapter discusses the two design concepts. Sections 2.3 and 2.4 discuss the design concepts and methods specific to conventional mould design and computer aided mould design, respectively.

A leader in CAMD development is Moldflow Pty. Ltd, based in Melbourne. Much of the literature reviewed in this chapter is based on material written by Moldflow staff. Appendix A4 describes several of the CAMD software packages currently available.

2.2 Basic Design Principles

Most designers develop their own set of design principles as they build their knowledge about mould design. Several reference manuals and text books contain rules or methods for producing moulds. Two books, published by Hanser, provide the designer with an excellent starting point (Gastrow, 1993; Menges, 1993). Gastrow identifies 108 designs which have been proven to work successfully. Menges provides detail on all aspects of tool making for the injection moulding industry from the types of materials used in moulds to the designs of runners and gates.

The basic design principles cover four main mould design areas: gates, runners, cavity wall thicknesses and filling images. Gates are used to control the flow of plastic into the cavity and therefore must be designed to give an optimal flow. Runners must deliver the plastic to the gate at the correct temperature and pressure so that it flows through the cavity without freezing before completely filling the cavity. The cavity wall thicknesses must be designed to give good flow through the cavity while avoiding large, sudden cross sectional changes. A filling image provides useful information on the flow of the plastic through the cavity, indicating any areas that may give rise to problems with the design, for example, weld line positions and air traps.

2.2.1 Gates

Gates are important, even though they are normally the smallest part of the mould. The gate is the entrance to the cavity and connects the cavity with the runner system. Improper gate dimensions or location can cause many problems in a moulded product. For example, a large gate may result in sink marks if the cavity pressure is not maintained for the entire cooling time of the gate and runner. A gate in the wrong position may compound or be the direct cause of weld lines in critical areas where stress may be applied, resulting in failure of the product.

With multi-gated products, the dimensions of the gates relative to one another are also important. The relative size of gates can determine the volume of plastic that will flow through a gate and therefore the positions of the weld lines. There are several different gate types, as described in appendix A3. The type of gate chosen for a specific mould will depend on the requirements of the product and the polymer specifications. Generally, the geometry and wall thicknesses will determine where the product should be injected and the polymer parameters will determine the size of the gate (table 2.1).

Factor	Considerations for Gate Location & Size
Product	Geometry Wall thickness Direction of mechanical loading Quality demands with respect to dimensions, cosmetics, mechanics Flow length/wall thickness
Material	Viscosity Melt temperature Flow characteristic Fillers Shrinkage
Other	Distortion Weld lines Ease of demoulding Separation from moulding Costs

Source : Menges, 1993.

Table 2.1 Factors Determining the Location, Size and Type of Gate

When a gating system is designed, three important factors should be considered: (1) uniform filling, (2) eliminate jetting and (3) gate position.

2.2.1.1 *Uniform Filling*

An important factor in multi-gated parts is the uniform filling of the cavity. It is difficult to design the gates and runners for a multi-cavity mould so that all the cavities fill at the same time. Without the use of computer aided tools, the procedure for designing multi-gated products is extremely time consuming because it requires several trials and changes to the gate dimensions before a good filling pattern is established. The basic procedure involves machining the gates smaller than necessary and, following a mould trial with successive short shots, then redimensioning the gates to provide uniform filling. This requires great care and skill, since the smallest change in a gate cross section will result in major changes in the flow rate of the plastic. A mould trial followed by redimensioning the gates may need to be repeated several times before a satisfactory result is obtained. Gate sizing for a multi cavity mould is one domain where computerised mould design tools are extremely useful.

2.2.1.2 *Jetting*

A lot of visual marks and flow marks in mouldings are caused by jetting. Jetting is the result of molten plastic swelling as it enters the cavity (White, 1992). If this swelling molten plastic continues at a high flow rate without obstruction, jetting will occur. Surface defects result from weld lines between the folds of the jet (Birley, et al, 1992) and spraying of the mould surfaces with jetting plastic. The spraying plastic freezes when it strikes the cold mould surface. As the plastic front reaches these areas it freezes around the sprayed plastic, creating the surface defect. Jetting can only be corrected through correct gate and runner design and positioning with respect to the moulding. Good gate design, which avoids flow marks, is more difficult to machine, but may be worthwhile if flow marks are thought to be a problem.

2.2.1.3 *Gate Position*

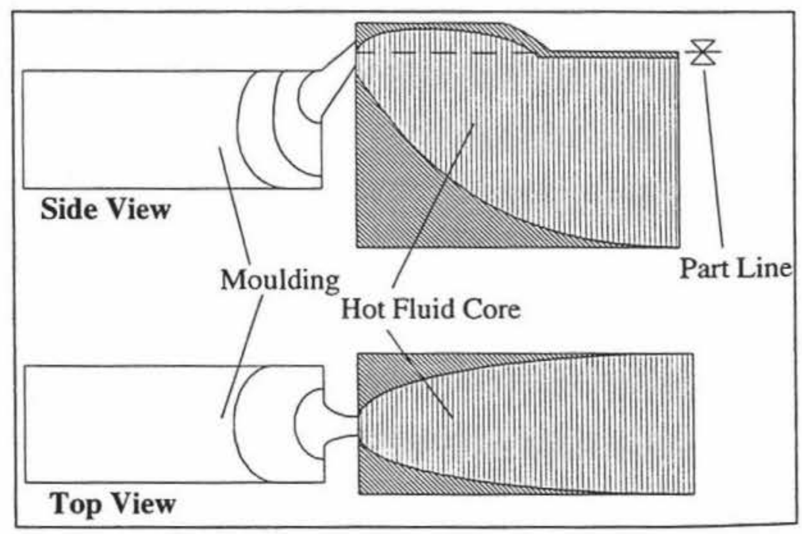
The position of a gate relative to the product is very important. Several considerations must be accounted for when deciding on the gate position, including:

1. *Flow path*: A uni-directional flow is the most desirable. When the plastic flows in two opposing directions, weld lines form at the end of flow and a weakness at that point can result.
2. *Distance from sprue*: The greater the distance between sprue and gate, the longer the plastic has to cool before entering the mould. The plastic temperature may drop resulting in flow marks. A hot runner may be necessary if the runner is too long.
3. *Thick and thin sections*: The gate should be positioned as far away from thin cavity sections as possible. When plastic enters a thin section it may hesitate while plastic finishes filling thicker sections, during which the plastic in the thin section cools and may prematurely freeze.
4. *Orientation of molecules*: Plastic products are always stronger in the direction of flow than across it. This is due to the alignment of the molecules as the polymer is injected. The alignment also means the product will shrink more in the flow direction. This is even more pronounced in products made from fibre reinforced materials since the fibres align themselves with the flow and the polymer shrinks around the particles.
5. *Principle stresses in the product*: A product undergoes stress during its normal life. This stress will be concentrated in a few regions of the product. An example is a plastic hockey stick. The stick is designed to be strongest where the greatest flexing will occur. When this product is moulded, the gate should be positioned in such a way that the principle regions of stress are the strongest. Ensuring weld lines do not meet in this section of the mould, and alignment of molecules, or crystallinity, to strengthen the product in the desired direction are two design methods that increase the strength of the product.
6. *Weld lines*: Weld lines will be present in many products and cannot be avoided. Weld lines can, however, be aligned in positions of least sensitivity by placing gates in the correct positions. For example, weld lines should not be located in regions that require the greatest strength. Weld lines should also be

avoided at the end of flow, when the plastic is coldest and may not form a good bond.

(Birley, et al, 1992; Menges, 1993; Rosato, 1991)

A gate may be positioned centrically or eccentrically with respect to the parting line of the mould (Menges, 1993). A centric gate is one that is cut into both mould halves. It is more difficult to machine and remove from the mould and may cause jetting. The advantage of such a gate is the low surface to volume ratio since it is circular in cross section. This reduces heat loss and friction. An eccentric gate is one which is cut into only one half of the mould, usually the moving half. This design makes the machining and demoulding easier. It also impedes jetting when the gate is aligned with a cavity wall. One problem with this design is the formation of a cold skin of plastic that may end up in the cavity. To avoid the cold skin entering the mould, an interesting gate and runner design was proposed by Menges (1993) (figure 2.1). The odd shaped runner and gate is more difficult to remove from the mould. The overall advantage, though, is the reduction in flow marks caused during filling. Runner volumes by the corners remove the cold slug before the plastic enters the mould and the angle and radii of the gate gives a laminar flow which will form a bubble front flow, rather than a jet.



Source : Menges, 1993.

Figure 2.1 Gate Design To Avoid Jetting

A well chosen gate location, shape and size will solve many of the problems associated with injection moulding. The design of the product is important, however, without a good gating system, the well designed product will experience many problems during filling and the functional life of the product.

2.2.2 Runners

A second major part of an injection mould is the runner system. A properly designed runner system can save the injection moulding company a lot of time later with design changes. The runner should deliver the molten polymer to the cavity at the right temperature and pressure so that it may flow through the gate and fill the cavity (Menges, 1993). Just as there are several factors which affect gates, there are several factors affecting runner position, shape and size :

- Part Volume
- Product Wall Thickness
- Plastic Material
- Length of Flow Path
- Resistance To Flow
- Surface To Volume Ratio
- Heat Losses
- Losses From Friction
- Cooling Time
- Amount Of Scrap
- Cost Of Manufacturing
- Mould Type

Source : Menges 1993

The shape of the runner is important. The runner should have the smallest area to circumference ratio to reduce heat and friction losses. The best shape is therefore a circular cross section, but is difficult to machine since it requires cutting into both halves of the mould. A good approximation is the trapezoidal cross section. Half round and rectangular cross sections should be avoided since they have high heat and friction losses. Examples of effective runner designs may be found in several mould design books (Menges, 1993; Schröer, 1983).

2.2.3 Product Wall Thickness

When designing a new product, wall thicknesses should be kept as constant as possible. This decreases the possibility of moulding problems and product quality later on by reducing differential cooling which leads to warping of the product. Wall thicknesses can be used to control many aspects of material flow through the product.

Most designers are aware of the problems that thick sections can cause when the product shrinks. Strengthening in a product should be done by ribs, rather than thickening a section. One difficulty that many plastic product designers encounter is designing bosses in corners for bolt holes. It is not good practice to fill in the whole volume with plastic, even though this would be easier when machining. Sections thicker than the body of the product will normally cause sink marks, a visual defect that is not acceptable in most cases (figure 2.2). By removing most of the material and using ribs to the boss, very little strength is lost. The product is improved by reducing its weight and eliminating the possibility of sink marks.

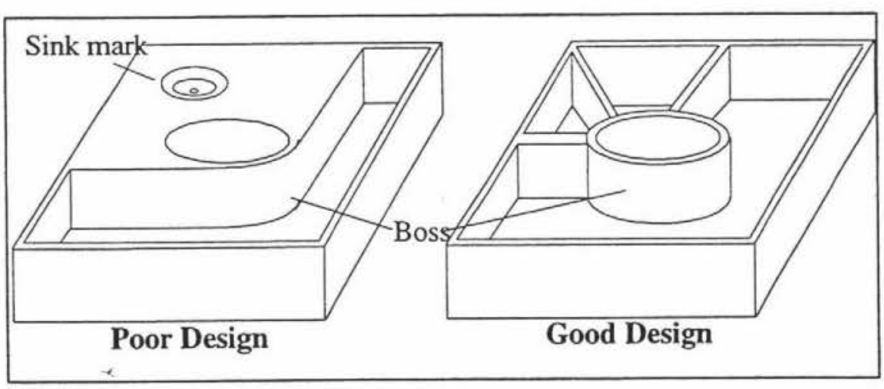


Figure 2.2 Poor Boss Design Versus Good Boss Design

Products that require thick sections can utilise other injection moulding technologies, for example, gas assisted injection moulding (Pearson, 1995).

2.2.4 Filling Analysis

Studying the filling process provides information to help determine the best location for gates and the best thicknesses of different surfaces of the product. A filling analysis is a graphical representation of plastic flow through the mould. The filling analysis provides the mould designer with information on the positions of weld lines and air traps. More sophisticated filling analysis software also provides information on the temperature and pressure ranges, and shear stress in the product. A filling analysis can be done by hand without the need for computer software, but is much faster and easier with a computer.

The Conventional Mould Design filling analysis is based on the wave theory as described by Huygens. The filling analysis is based solely on the geometry of the product. Computer Aided Mould Design has excelled in producing filling images that display what is actually happening in the mould. The programs now available describe the temperature, pressure, shear stress and other parameters based on the shape of the product, processing parameters and material used to mould the product.

2.3 Conventional Mould Design

Conventional Mould Design is the process of developing a mould from experience and a set of design principles with working mould designs as guidelines. Such mould design is an art rather than a science and relies heavily on experience and knowledge of polymer flow within a mould. The designer is responsible for the entire design process. Expertise is brought in as needed, for example a production manager may be consulted about the feasibility of manufacturing a product on the available moulding machines, or a material specialist may advise on polymer requirements. These people normally have little input in the design process, but are merely consulted for their opinions. Plastic company representatives normally have a good understanding of the materials available and their capabilities. Consulting a polymer representative near the beginning of design will aid in selecting several prospective polymers. With the material properties in mind, a design can be produced which will match the material properties.

The Conventional Mould Design concept is based on a trial and error approach. A trial is the process of running a mould in the injection moulder and diagnosing any problems. The causes of these problems are found and eliminated by design changes or moulding parameter changes. Identifying the cause of the problem in a design can take a significant amount of time.

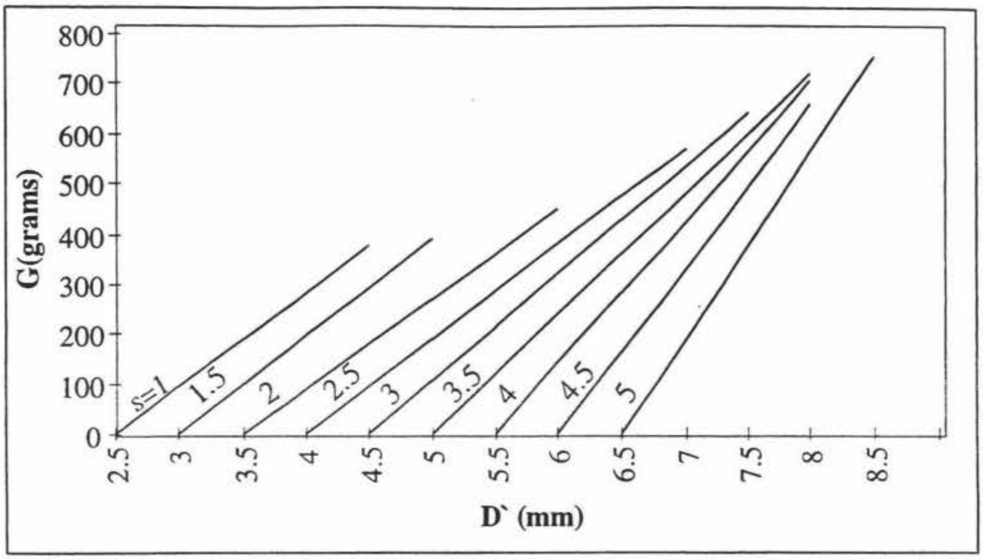
For example, a product may not be filling out and always producing a short shot. The fault could have several causes. It could be that the runner is not large enough and therefore cannot accommodate the amount of plastic required through it, or the walls of the product are too thin and the plastic is freezing off before it reaches the end of the product. When a mould designer identifies that the plastic is freezing before reaching the end of the cavity, his immediate reaction is to increase the size of the gate. This technique, however, can compound the problem since gate dimensions have a large effect on the flow rate of the polymer and even small changes in cross section can result in large volumetric flow rate changes.

Trial and error methods have been used by designers for many years. There are some principles for the designer to follow and some proven mould designs that may be copied or modified for the job at hand (Gastrow, 1993). Even with these tools, mould design is still an art. When the designer has to produce a mould for a product that is different from any of the previously successful models, trial and error must be used.

2.3.1 Conventional Runner Design

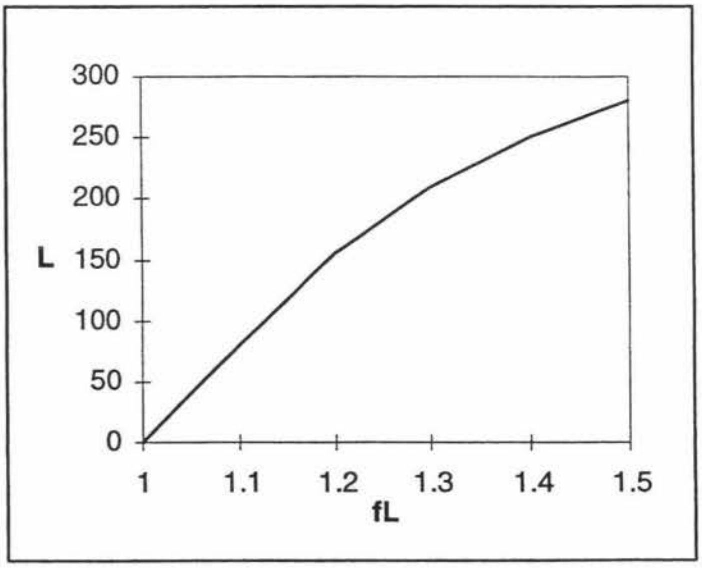
Most runner systems are over designed and end up requiring more plastic, and therefore more regrind or waste, than is really necessary. A simple heuristic for calculating the runner diameter is to design it 1.5 mm larger than the largest wall thickness of the product. This is a robust method and is a good point for a designer to start at for a simple design. As will be seen later, these methods usually over specify the runner. A more complex method of dimensioning runners is described below (Menges, 1993).

From figures 2.3 and 2.4, it is possible to determine the dimensions of the runners. The parameters of average wall thickness, s, part weight, G and runner length, L, must be known to find the diameter of the runner. From figure 2.3, the diameter, D', of the runner before the length, and therefore friction, has been considered, is found. Using the length of the runner, L, the correction factor, f_L, is found from figure 2.4. The corrected diameter can then be calculated : $D = D' \cdot f_L$.



Source : Menges, 1993.

Figure 2.3 Dimensioning Runners Applicable for PS, ABS, SAN, CAB Plastics.



Source : Menges, 1993.

Figure 2.4 Correction Term Determination

2.3.2 Filling Image

A filling analysis may be determined without the use of a computer for almost any product. The main problem is the time it takes to produce the filling analysis. A tool for examining the filling process is the filling image (Menges, 1993) (figure 2.5). It is based on the theory of wave propagation according to Huygens. The theory states that every point on a wave front is a source of a secondary wavelet. The envelope of all the wavelets describes the next primary wave front. The radius of each wavelet is identical. The filling image takes into account the product thickness. It does not, however, consider the cooling of the plastic or the viscosity of the material used, both of which have major effects on the flow properties of the polymer.

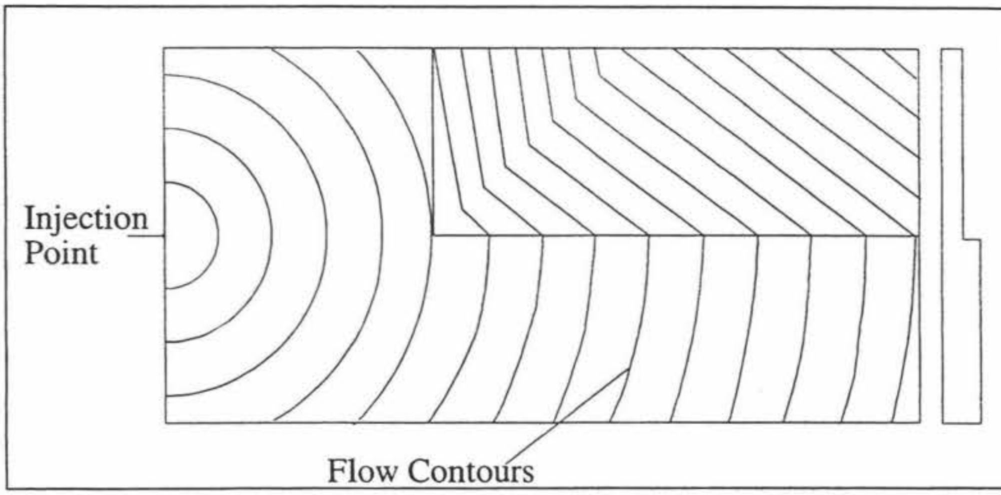


Figure 2.5 Filling Image

The filling image provides information on three key areas:

1. Gate type and location.
2. Location of weld lines and any air traps.
3. Direction of the principle orientation of the product.

Problems found by using a filling image can be rectified by :

1. Varying the position and number of gates.
2. Varying the location of bosses and varying the thickness sections.

This method has shown good results when compared to the actual performance of a mould during filling and is particularly useful if CAMD software is not available. The methodology for using filling images is well documented in Menges (1993).

2.4 Computer Aided Mould Design

Several software programs are currently available that produce a graphical analysis of a mould which may be used to improve the mould design. These software programs are beginning to set a new standard in mould design. Benefits of using CAMD software include:

- reduced lead time to production
- improved quality of the final product
- reduced costs of materials
- reduced time in trials, leaving injection moulding machines more available for production.

Computer Aided Mould Design approaches mould design in quite a different way compared to Conventional Mould Design. Rather than basing the mould design on previously well manufactured moulds, the computer aided method solves the heat and flow equations that describe the flow of the hot polymer in the mould. The equations are solved using finite element analysis. Finite element analysis is a method for solving complex simultaneous equations by using a meshed model which is created by building triangular elements on the model surfaces. The equations are solved for each of the elements, beginning with the injection points and radiating from them until each elemental equation has been solved. The results of a finite element analysis may be viewed in graphical form.

Graphical analysis provides greater detail for the mould designer to visualise what happens in the mould. By actually visualising the plastic flowing into the mould, regions that will cause problems during moulding may be found and changed to improve the flow before a mould is manufactured.

One company producing CAMD products is Moldflow Pty. Ltd., based in Melbourne, Australia. Moldflow Pty. Ltd. is one of the leading CAMD organisations who have continued to develop products to aid injection moulding companies. This section is based on the Moldflow philosophy and concepts.

2.4.1 Computer Aided Mould Design Concepts

> *"The basic Moldflow philosophy is to use a structured design procedure, based on Moldflow flow concepts, and applied using the capacity of the Moldflow software to predict pressure, temperature, shear rate, shear stress and cooling time, etc."*
>
> *(Moldflow Design Principles)*

There are twelve Moldflow flow concepts (Austin, 1985; Moldflow, 1990). Each concept states an ideal situation which should be aimed for with every mould design project. It is normally not possible to meet all the criterion, but each is important in the reduction and elimination of moulding problems.

The twelve concepts are :

1. UNI-DIRECTIONAL AND CONTROLLED FLOW PATTERN:

The plastic should flow in one direction with a straight flow front throughout filling. This gives a uni-directional orientation pattern.

2. FLOW BALANCING:

All flow paths within a mould should be balanced, i.e. fill in equal time with equal pressure.

3. CONSTANT PRESSURE GRADIENT:

The most efficient filling pattern is achieved when the pressure gradient, i.e. pressure drop per unit length, is constant along the flow path.

4. MAXIMUM SHEAR STRESS:

Shear stress during filling should be less than a critical level. The value of this critical level depends on the material and application.

5. UNIFORM COOLING TIME:

Cooling times should be uniform throughout the part to avoid warping.

6. POSITIONING WELD AND MELD LINES:

Position weld and meld lines in the least sensitive areas.

7. AVOIDING HESITATION EFFECTS:

Position gates as far away as possible from where the flow divides into thick and thin flow paths to avoid hesitation effects.

8. AVOIDING UNDER-FLOW:

Positioning gates so that the flow fronts meet at the end of filling.

9. BALANCING WITH FLOW LEADERS AND FLOW DEFLECTORS:

Use flow leaders (local increase in thickness to encourage flow in a particular direction) or flow deflectors (local reduction in thickness to divert flow) to obtain flow balance.

10. CONTROLLED FRICTIONAL HEATING:

Design runners to give a higher melt temperature in the cavity. This achieves lower stress levels in the part, without causing material degradation due to long exposure at elevated temperatures.

11. THERMAL SHUT OFF OF RUNNERS:

Design runners so that when the cavity is just filled and adequately packed the runners freeze. This avoids over pack or reverse flow in or out of the cavity, after the mould is filled.

12 ACCEPTABLE RUNNER/CAVITY RATIO:

Design runner systems for high pressure drops, thus minimising material in the runner, in order to give a low ratio of runner to cavity volume.

(Austin, 1985)

2.4.2 2D Modeling

Occasionally, the product to be analysed only requires a balancing of the runner system so that the flow into the mould is at the correct temperature and pressure. The gate locations and part thicknesses have already been set and possibly machined. Changes in the runner thicknesses can be performed from a 2D analysis.

The 2D analysis also provides information on the moulding window, which is required before further analysis on the mould can be carried out. The moulding window relates the melt temperature to the injection time based on a mould temperature. It considers the filling pressure and the stresses to establish a window in which the polymer can be injected into the mould successfully. The moulding window is found by applying the three criteria which have evolved from the twelve flow principles. The criteria are:

1. The minimum end of fill temperature is 20°C below the melt temperature
2. The maximum stress is less than that for the particular material used.
3. The maximum pressure is less than 70 MPa for the cavity and less than 100 MPa for the runner and cavity.

Figure 2.6 gives an example of a moulding window.

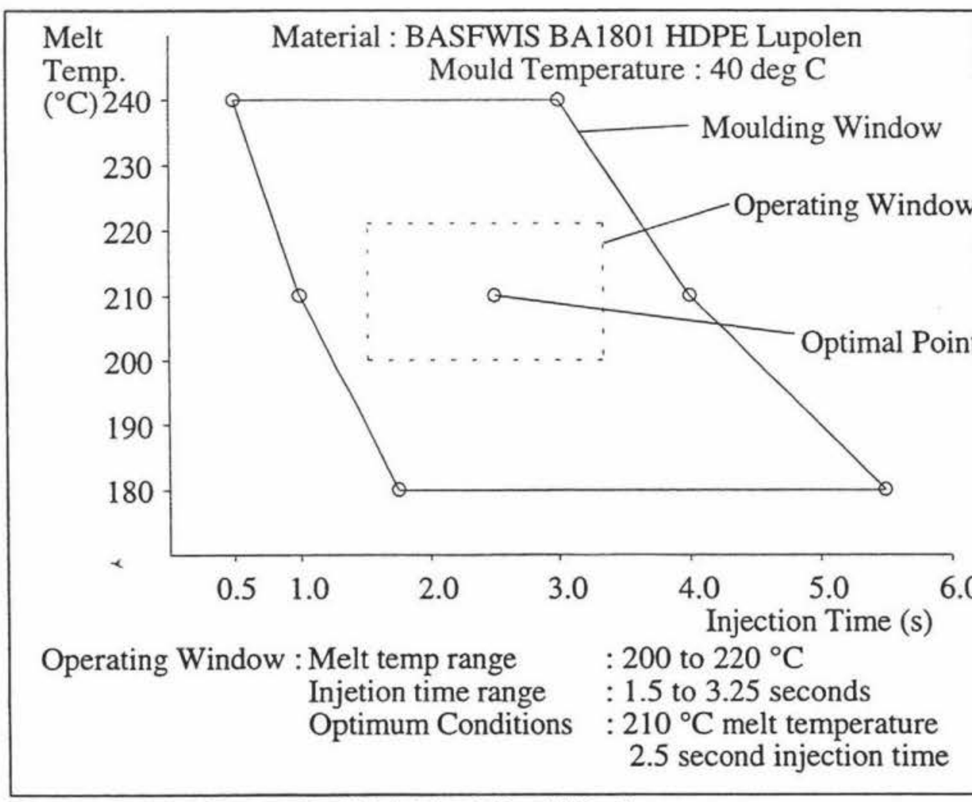


Figure 2.6 An Example Of A Moulding Window.

Three design requirements are discussed in this section; the flow path description, flow balancing of the cavity and gate and runner sizing. A flow path is required before any analysis can be performed on the product. A flow path is a description of a path that the polymer will take from the gate to the end of the product cavity. Flows are balanced so that the different flow paths complete filling of the cavity at the same time. Gate and runner sizing are important to ensure the polymer enters the cavity at the right temperature and pressure while maintaining the balanced filling.

2.4.2.1 Flow Path

A flow path is any path the plastic will take from the gate to the end of the product cavity. This may be to the point where one flow meets another or a corner of the cavity. All of the major flow paths should be described to determine the cavity pressures and ensure there are no regions which may cause problems later. An example of a product with its flow paths highlighted is given in figure 2.7.

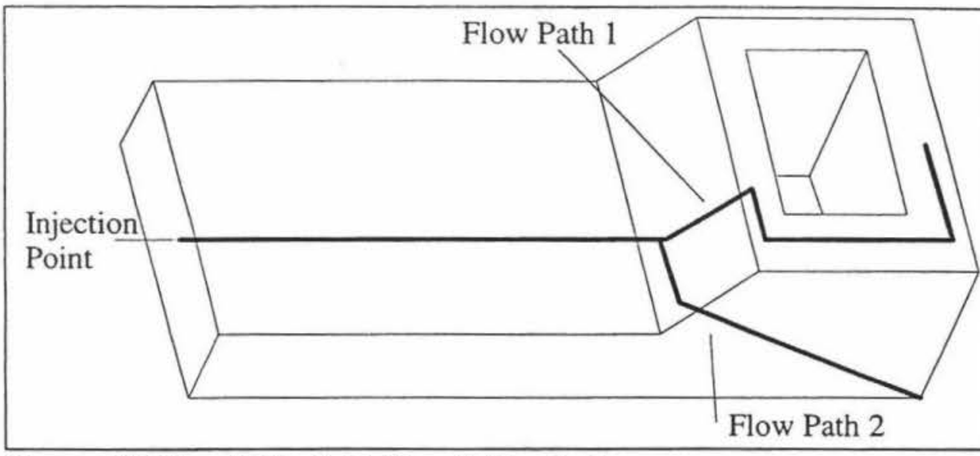


Figure 2.7 Flow Paths Of A Simple Product

A flow path consists of a series of sections, each of which has a length and a thickness. The path description may include other variables, but they are not used in the analysis of the path. The dominant flow path in a mould is used to determine the moulding window. It is the flow path which requires the greatest time to complete. In most cavities, this is also the longest flow path. From the moulding window, an optimum operating point may be chosen. This operating point sets the optimum injection time and melt temperature for the given mould temperature. Small fluctuations in these variables will not cause problems, as long as the values remain within the moulding window.

2.4.2.2 Flow Balance

A flow balance may be carried out once the moulding window has been established. A flow balance looks at all the flow paths and determines the best thicknesses of the part or the runner system based on the maximum cavity pressure. A flow balance is essential for a multi-cavity, unbalanced mould to ensure the cavities fill at an even rate, pressure and time. Flow balancing is a simple procedure using computer software.

Flow balancing is used to remove any moulding or quality problem that arises from unbalanced flow. These include over-packing and subsequent warpage, air-traps and weak weld lines.

A 3D analysis of a cavity may reveal some problems that may be overcome with changes to the wall thicknesses of the product. After changes to the geometry of the cavity, the optimum processing conditions may change. The processing conditions, or moulding window, must be recalculated and a new flow balance analysis performed. This will ensure the processing conditions are not outside the bounds of the window.

2.4.2.3 Gate and Runner Sizing

Under conventional, non-computerised mould design, runner systems are often oversized. By specifying the runner sizes according to the Moldflow analysis, savings in runner volume of 50% are possible (Moldflow 1990). A reduction in the amount of material in the runner reduces the amount of recyclate since the runner is normally reground and fed back into the feed-stock. Increased production can result since less material is required in the runner, therefore reducing the cooling time needed.

Once a flow balance of the cavity has been completed in the 2D environment, the gate and runner system may be added. If the location of the gate is known and fixed, the runner system may be balanced with the balanced cavity in the 2D environment. A runner balance is done to "reduce the runner volume, and ensure that polymer is delivered at the flow rate and temperature required to maintain optimum processing conditions at the cavities" (Moldflow, 1994).

The advantages of balancing a runner system in the 2D environment are:

1. If the dominant flow path is known, the 2D analysis is quicker than the 3D approach.

2. Multi gated cavities with different flow lengths may be analysed.

3. A result is always obtainable.

The runners are balanced when the cavities complete filling at the same time. The pressure drop in the runner system can also be maximised, resulting in frictional heating, by minimising the runner volume. Frictional heating increases the temperature from the barrel nozzle to the gate, therefore the barrel temperature may be reduced. By reducing the barrel temperature the polymer will be unaffected by high temperature degradation since the polymer is held at a high temperature for a few seconds, rather than several minutes in the barrel. A high pressure drop in the runner system allows greater control of the flow into the cavity.

The balancing of a multi cavity mould is quite simple using Moldflow software, whereas the conventional method is time consuming and usually results in only an acceptable mould design, rather than a good design (Moldflow, 1990).

2.4.3 3D Modeling

The 3D modeling technique requires the user to create a full 3D model of the product. This model is used to determine where problems with a mould are likely to occur. The 3D model is created in the MF/View environment. Within this environment a designer may create and analyse a wire frame model. The analysis results indicate sections of a mould which may need redesigning.

Analysis available within the MFView environment includes prediction of the plastic flow into a cavity, the temperature and pressure of the cavity, shear stresses, shear rates, weld lines and air traps. Graphical representations are compared to the Moldflow concepts and design changes are made to improve the flow, temperature, pressure, shear stress and shear rate.

The main project types 3D modeling is utilised for are described in the sections that follow.

- Gate positions are established in the 3D modeling environment.

- A flow analysis is performed in the 3D environment. The simulation provides the graphical results describing the polymer flow through the cavity,.
- Warpage is one of the main problems with injection moulded products. The graphical results from the 3D analysis can provide valuable information on where warping could be expected.
- The cooling design of the mould may also be analysed using the 3D environment.

2.4.3.1 Gate Positioning

As stated earlier, gates and their positions are important. By using CAMD software, it is relatively simple to change the number and location of gates for analysis. By examining the results, conclusions can be drawn based on the different designs without a mould ever being produced. This type of simulation stimulates creativity and allows the exploration of design changes without the need to manufacture a mould.

Positioning a gate is the first step in the CAMD design procedure. A gate position is chosen by analysing different gate arrangements and examining the results of each. The Moldflow approach to gate positioning is to divide the cavity up into sub-mouldings that are approximately similar in pressure requirements and volume. The gates are positioned to give both smooth and balanced filling.

The number of gates may also play a part in deciding where the gates will be placed. One gate in the center of a product may replace two gates at either end. This decision will be based on the temperature, pressure, shear rate, shear stress, weld line positions and air traps. Each of these factors must be considered when determining the location and number of gates.

2.4.3.2 Flow Analysis

"In all design projects the object is to get a smooth filling pattern i.e. one in which the plastic fills the mould with a straight flow front, without changing direction throughout filling." (Moldflow, 1990).

A flow analysis provides information on the cavity that can be used to design the mould better under the moulding conditions and material specified by a user. A flow analysis follows one of three approaches: finite difference approach, branching flow approach or finite element approach (Austin, 1985) for solving the complex simultaneous flow and heat equations. The finite element approach of solving the equations is used most often in CAMD software. Equations of flow and heat transfer are developed for each element in the cavity and then solved simultaneously. A distinct advantage of the finite element approach is the independence of the model from the flow. This allows the gating positions to be selected anywhere in the cavity without changing the model description or elements. With the other methods, the elements are dependent on the positions of the gates and therefore the elements must be reassigned each time the gate positions are changed.

Results of a flow analysis may be viewed graphically in the MFView environment. Information contained in the graphical displays include time to fill, pressure, temperature at the end of flow and the flow front, shear stress and shear rate. Weld lines and air traps may also be identified. The maximum and minimum values of each of these parameters is compared to the acceptable maximum and minimum values. If there are discrepancies between the acceptable values and the actual values, changes may be made to the cavity, gates or runners to improve the performance of the flow into the cavity, and hence the final performance of the mould.

2.4.3.3 Warpage Control

Warpage is due to differential shrinkage, that is two different shrinkage rates at two different parts of the moulding. There are three main reasons for differential shrinkage:

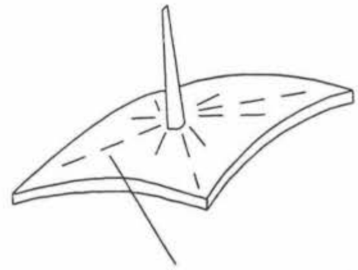
- Differential orientation.
- Differential crystallinity.
- Differential cooling

Each of these is illustrated in figures 2.8, 2.9 and 2.10, respectively (Moldflow 1990).

Warpage can be predicted by the user from the graphical results obtained in the flow analysis. The fill time plot will reveal possible product warpage if it is not an even filling image. The pressure plot will also reveal regions that could warp by differential warpage contours.

Warpage problems may be solved by changing the design of the cavity and the cooling system so that changes in the levels of shrinkage are reduced. This is of great advantage when moulding very flat products which are highly susceptible to warpage. No injection moulded product is free from the problems of warpage and therefore its prediction is very useful.

Normally, orientated material shrinks more than non-orientated material. Warping is caused by differential shrinkage and therefore highly orientated products will shrink more than non-orientated products.



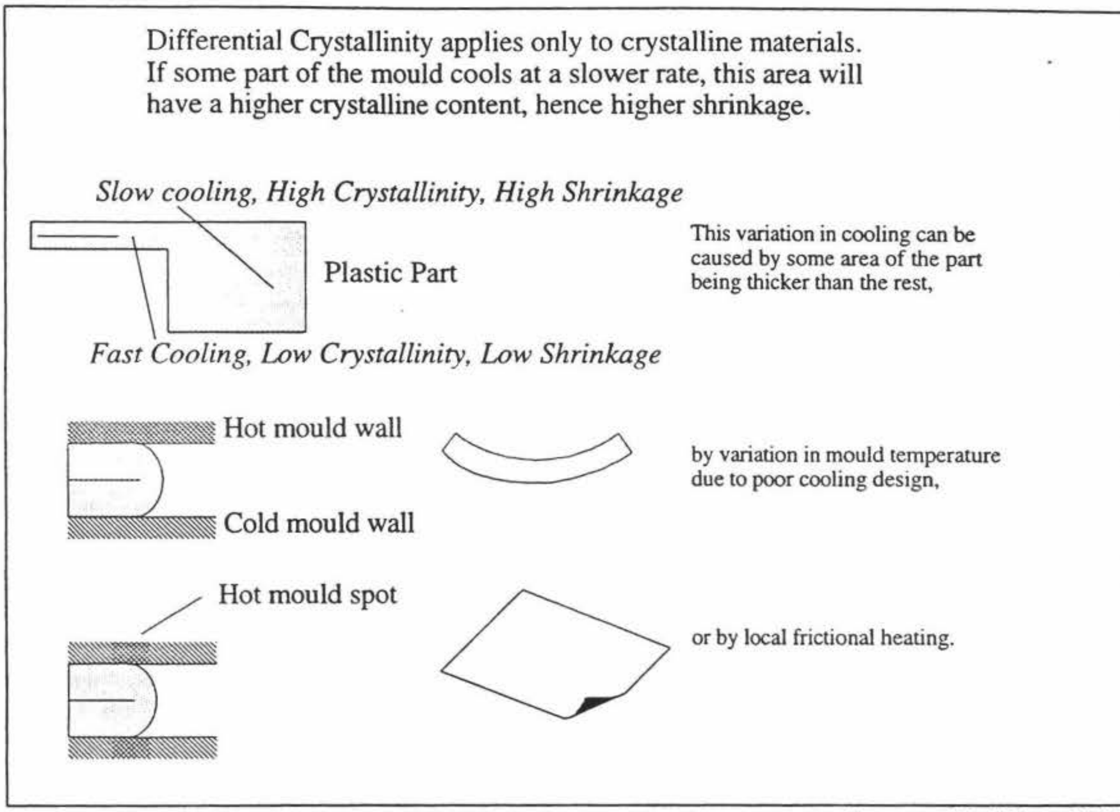
A centrally gated flat product will warp due to orientation since the orientation is greater along the diagonals than it is along the edges of the product. This causes the corners to curl.

Orientation of polymer

Source : Moldflow, 1995

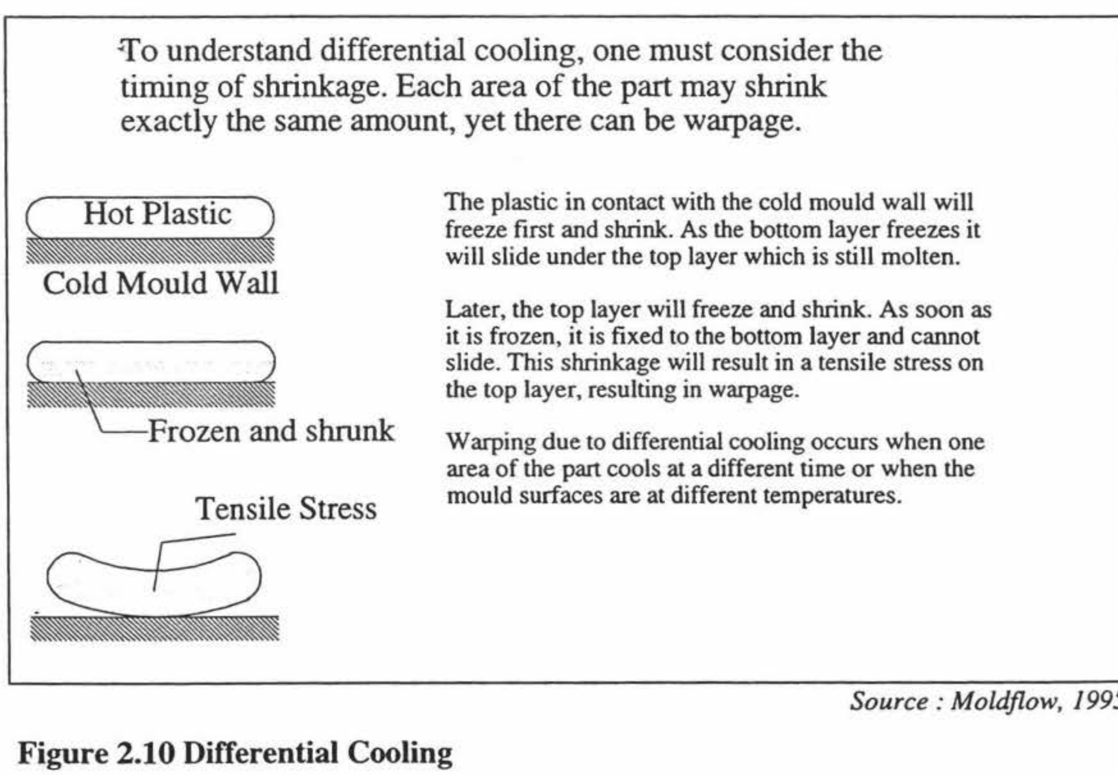
Figure 2.8 Differential Orientation

The type of warping encountered will indicate the required changes to the mould. For example, a centrally gated flat part, will warp due to the differential orientation of the polymer molecules. Some strengthening around the edge of the part could help reduce the warping of the part, but will not reduce the differential orientation of the molecules. The gate position could also be changed to eliminate warpage problems. An edge or fan gate may reduce the warpage since differential orientation is reduced with this type of gate.



Source : Moldflow, 1995

Figure 2.9 Differential Crystallinity



Source : Moldflow, 1995

Figure 2.10 Differential Cooling

2.4.3.4 Cooling System Design

Differential cooling is a cause of warpage. The cooling design is therefore important since it can reduce the possibility of warpage due to differential cooling. A cooling system may be analysed using Moldflow software and the design changed to improve the cooling of the mould. While designing cooling channels, the designer must avoid other parts of the mould, like ejection pins, sliding plates and side pins.

It is not difficult to design the cooling system in the fixed half of the mould since there are usually few other mould parts in the fixed half to interfere with the placement of cooling channels. The moving half, however, presents more difficulties to the designer since the ejection system is located in the core half. A cooling system before considering uniform heat transfer, might look like that in figure 2.11a. The isotherms show differential cooling. After a cooling analysis has been performed and changes made to the cooling layout, a uniform heat transfer cooling system might look like figure 2.11b. Uniform cooling, designed using Moldflow/Cool, reduces differential cooling due to poor cooling design, one of the main causes of warpage.

The zones in figure 2.11b are areas where more or less cooling has been used to improve the uniformity of cooling. Zone 1, requiring more cooling, has new in-designed cooling channels. The cooling channels have been moved closer to the cavity in zone 2 areas to remove the heat from the plastic faster and in zone 3 the cooling channels have been moved away from the surface of the cavity since less cooling is required in this area.

The software, however, does not design the cooling system, but rather guides the user to make good decisions based on the analysis results.

The cooling system is designed in a similar nature as the mould itself. It is built around the cavity, and cooling lines are placed in the positions the user considers as a good starting point. Analysis of the cooling system provides results regarding the temperature on either side of the cavity, the temperature difference between the two surfaces and the cooling time. The differential temperatures should be as close to zero as possible throughout the moulding. The designer continues to change the flow rate,

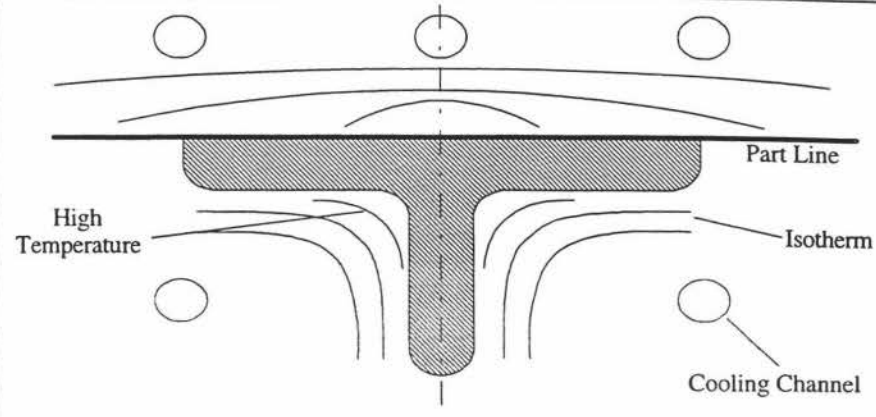


Figure 2.11a Typical Cooling Circuits

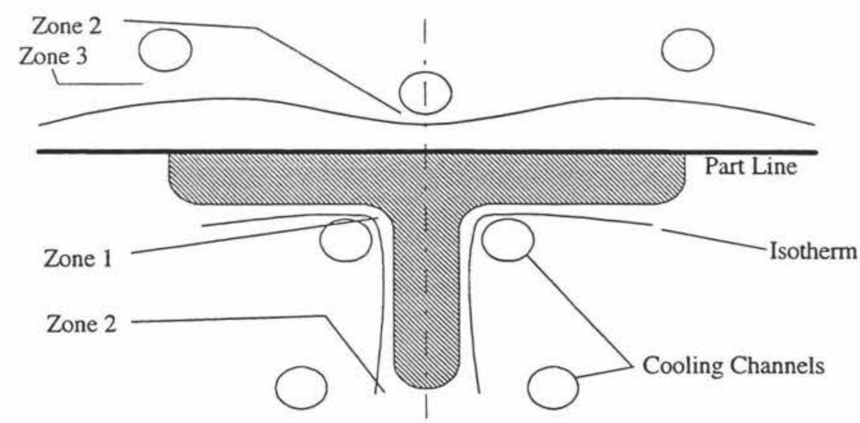


Figure 2.11b Uniform Heat Transfer Cooling Circuits

Source : MF/Cool 3.1 Cooling Analysis Manual, 1992

Figure 2.11 Typical Cooling Circuits.

coolant temperature and cooling line positions or types to increase or decrease localised cooling until the mould is cooling evenly. By designing the mould cooling system in this manner, warpage from differential cooling may be significantly reduced or eliminated.

2.4.4 Capabilities

Computer Aided Mould Design packages are beginning to make an impact on injection moulding mould design as they become accepted as being capable of producing better moulds, faster and cheaper.

CAMD software is able to assist the mould designer in making good decisions in all areas of mould design, from the cavity and runner system to cooling channel placement and design. The software does not design the product, however, rather it points the designer in the right direction, giving guidelines on when and where to make changes to the design.

Many of the problems associated with injection moulding are based on poor design or incorrect processing parameters. CAMD gives the designer the parameters which will optimise the filling of the product. These parameters are used in the processing to provide results on the design.

2.4.5 Limitations

With constant changes and upgrades to the software, limitations are being eliminated. CAMD does not give information on surface defects, for example flow marks due to jetting. To eliminate jetting, the designer should have a good understanding of correct gate design.

The product to be analysed must be drawn in the 3D environment, which can be very time consuming. This can be overcome by importing the design file from another CAD package and formatting it to the required wire-frame. If these files are not available, the designer must return to the MFView method of building the design.

2.5 Conclusion

Many injection moulding companies who design their own moulds continue to use the conventional mould design method. The conventional design method is still valid and many very good moulds have been manufactured in this fashion. Yet computer aided mould design adds to the conventional method to produce designs faster without the need for time consuming trial and error methods to improve the mould once it has been manufactured.

Proof needs to be supplied to the injection moulding companies that computer aided mould design will improve their mould design and reduce the lead time from concept to full manufacture of the part. Chapter three will present the problems associated with moulding injection moulded products and present several reasons why computerised mould design aids the mould designer.

CHAPTER THREE

MOULD DESIGN - THE PROBLEM

Mould design has changed over the past two decades. However, mould designers have been slow to recognise the benefits of Computer Aided Mould Design software. The software twenty years ago required high powered machines and were not within the grasp of the common mould designer. This has changed recently with the introduction of software for the PC. This chapter describes the reasoning behind using CAMD.

3.1 Introduction

Since the first plastic product was manufactured, the plastics industry has grown at a phenomenal rate. Many parts which were once made from metals or woods are now manufactured in plastics, and in most cases the plastic products perform better than the original product. One example is the electric kettle. It was originally manufactured in metal, however, since the introduction of plastic kettles, very few are made in the original material. Some of the advantages of the plastic kettle include:

- lower heat transfer coefficient and therefore less chance of scalding

- lighter weight

- cheaper to manufacture

There are hundreds of thousands, perhaps millions of other examples of products that were once manufactured in metal, glass, wood or composite materials which are now manufactured in plastic and performing better than their predecessors.

With such an increase in the number of plastic products, a good design basis is required to produce products in such a way that they will continue to do the job for which they were designed. This requires very good designers and design tools. The design tools should assist the designer to create a feasible design quickly, effectively and to the best of his or her ability. One of the design tools that a great many designers are now using is Computer Aided Design, or CAD, tools. CAD software allows the designer to create drawings or designs for a product or mould that are easy to modify when changes are required. Copies of the design may be reproduced quickly without the need for a draughts person or blueprinting. Detail is drawn on the same "page" and large drawings are easier to manage. CAD is a design tool which assists the designer to create the best possible product.

The greatest challenge in designing an injection moulding mould is ensuring the plastic will flow through the mould without causing any moulding problems, for example visual defects, processing problems or incomplete filling. The root cause of a great deal of the problems encountered in injection moulding products is based on a lack of understanding of how the plastic flows in the mould and consequently poor mould designs result. The flow properties, or rheology, of polymers are an important domain for a designer to grasp. Rheology has generally been considered within the chemist's sphere of knowledge. However, mould designers must learn and understand how the polymer properties effect the flow of the material in the mould so that they can produce the best possible product.

This chapter will set out the difficulties with mould design using conventional design methods. Section 3.2 discusses the reasons polymers are difficult to mould at high pressures, the rheological factors involved in injection moulding and several other reasons Computer Aided Mould Design software is needed. Section 3.3 discusses how the difficulties may be overcome using CAMD and presents the hypothesis under study. Section 3.4 discusses the benefits of CAMD software.

3.2 Moulding Difficulties

The main reason Computer Aided Mould Design software was first introduced was because injection moulded products were produced without any thought about how the plastic would perform in the mould and how the product might perform under applied stresses. The flow and heat characteristics of polymers were not understood which led to poorly manufactured products which failed under applied stresses. The rheology of polymers is an important area for the mould designer to understand.

The result of the inability to determine how polymers flow in moulds led to several distinct problems in mould design:

1. Long, usually unknown, development time, from concept to production.
2. High tooling cost.
3. Excessive material in the runner system.
4. Low quality, attributable to: high internal stresses.
   warpage.
   surface defects.
5. Small moulding window.

6. Low creativity in designs.

These problems are not necessarily in any specific order.

3.2.1 Polymer Rheology

The rheology of polymers is important in understanding how plastics will respond in the liquid state under high pressures and shear stresses and varying temperatures. Rheology of polymers is a large topic. Several points regarding rheology of polymers will be made here and further information may be found in the references.

Polymers are mostly pseudoplastic materials, which means the viscosity decreases with increased shear rate, i.e. the higher the shear rate, the easier the material will flow. Unfortunately there is an upper boundary to the shear rate at which point the polymer will begin to deteriorate. If a polymer is pushed into a mould at a very high shear rate, the deterioration in the polymer will cause cracks in the product and visible scarring.

Polymers are compressible, up to 15%, which introduces even greater problems if the cavity is not designed to fill with polymer correctly. The polymer may be compressed more in one region of the product than another. The difference in pressure can create a shrinkage differential between the two parts of the product.

A third, and perhaps most important property of polymers when they are injection moulded is the shrinkage from the liquid state to the solid state. This shrinkage may be up to 25% for some plastics. The problem here is that the polymer can be compressed by only 15%, therefore there is a deficit of 10% of plastic which must be injected into the mould while the product cools. This compensation phase is occasionally referred to as the hold time.

A number of papers (Beyer, C. E. and Spencer, R. S. 1960; Bird, R. B., Armstrong, R. C. and Hassager, O., 1987; Cheremisinoff, N. P., 1993; Shah, Pravin L., 1990) have evaluated the rheology of polymers as it relates to injection moulded products. Several equations have been recognised as valid for certain limited regions of the flow of polymers, however there is no simple method of calculating how a polymer will flow in a particular mould. Without the aid of a computer and mould design software, the mould designer cannot solve the complex flow and heat equations associated with injection moulding. Mould designers have designed good moulds in the past without CAMD, however, the procedure has been time consuming and occasionally ends in the failure of the mould. A failed mould means more time and energy must be spent to produce a new mould design, which may still have problems that need to be ironed out using a trial and error approach.

3.2.2 Long, usually unknown, development time

The development time of the mould should be as short as possible, and known, if a mould manufacturer is to gain a competitive advantage. One of the greatest unknowns in conventional mould design is the time required to progress a project from concept to production. A large portion, perhaps 50% of the total time will be required to try out the new mould and make necessary changes to it to ensure the cavity fills correctly and the quality of the product is acceptable. This time would not be required if the mould were designed correctly before manufacture.

3.2.3 Small Moulding Window

The moulding window indicates the operating temperatures and injection time that a particular polymer should be used at with a design. The moulding window depends on the flow paths, the path wall thicknesses and the polymer viscosity. When a mould design is carried out using conventional methods, the moulding window is not normally known before manufacture. It is found by trial and error once the mould has been manufactured, and the result is normally a small moulding window. The moulding window can be enlarged by using a less viscous material or by changing the design. However, once a good operating point has been found, the mould normally will not under-go further costly changes.

3.2.4 Tooling Cost

Mould tooling for injection moulding is expensive. A simple mould can easily cost $15,000 and $100,000 moulds are not unheard of. There are two main reasons for the high cost. The first is the cost of the materials used for mould making. The second is the time required to develop the mould concept into a functioning mould. The development time includes an unknown amount when the tool is tested using the trial and error method, as explained in chapter 2. The mould normally requires several changes after a number of mould trials have been carried out. This additional cost is passed on to the injection moulding customer who is attempting to keep his/her costs low since the injection moulding industry is very competitive. Any aid or tool which will reduce the cost of the mould is truly an asset to an injection moulding mould design company by introducing a competitive edge.

The basis of conventional mould design is the trial and error approach. This method consumes time, energy and tool material. Occasionally a tool will be manufactured which requires a complete change because the material won't fill the cavity, or the design was flawed in some way and develops quality problems. In this case, the mould must be scrapped and a new mould designed and manufactured. This is an extreme case scenario, however, many mould designers and injection moulding companies do have problems when running trials on moulds and end up spending many hours trialing, error finding and repairing moulds that do not perform correctly. The end result is a tool which has cost possibly twice as much to manufacture and may only produce a satisfactory product.

3.2.5 Excessive Material in the Runner System

Once a plastic product has been manufactured, the runner system is normally recycled or discarded. A runner system should contain a small proportion of the polymer in the entire cavity for two reasons.

1. If the polymer is to be recycled back into the product, the recycled polymer should make up no more than 10% of the total shot weight. More than 10% can cause a reduction in the properties of the polymer, affecting the properties of the end product. 10% is only a general guide. Some polymers can be recycled with little effect to their properties and therefore larger amounts of recyclate may be used in the product. An example is polyethylene. Glass reinforced polymers lose a great deal of their strength when they are recycled because grinding shortens the length of the glass particles to such a degree that the tensile strength they provide is reduced.

2. The polymer in the runner is not normally part of the product. A large runner requires a greater amount of time to cool. It therefore unnecessarily increases the cycle time of the process, adding to the cost of the product, especially if there is a high demand or constraint on the injection moulding machine.

Mould designers who use the conventional mould design methods normally over-specify the runner sizes, increasing the amount of material required, because :

- the runner is considered merely as a transporting mechanism, moving polymer from the nozzle to the cavity.

- large runners are thought to introduce low stress levels in the polymer. The low stress levels are transferred to the product.

These two reasons seem justified, however, the runner may be used as more than merely a transporting device and experimentation has shown that "parts moulded with smaller runners have lower stress levels and less tendency to warp." (Moldflow, 1990).

Multi-cavity moulds are difficult to balance using Conventional Mould Design. One reason for this is designers attempt to use the gate as a control for the rate at which the polymer flows into the mould. The gate is a poor flow control device since small changes to its size yield large changes in the flow rate of the polymer. By sizing the runners so that the cavities fill at the same time, a true flow balance may be achieved. Unfortunately, this is very difficult using Conventional Mould Design because the runners must be sized according to the volume of the cavity and the wall thicknesses through the cavity. Once again, the volume of the runners are over specified in multi-cavity moulds leading to wasted material and increased energy use.

3.2.6 Low Product Quality

The quality of the product will depend on its purpose. A product that is not seen, for example a switch casing, may require high strength; the quality of the product will depend on the strength of the final product. Another plastic product may be quite visible, for example, a child's building block. The block must be free from surface defects as well as have good tensile strength. Quality is expected in both products, however, the quality is measured differently.

Plastic products have, in the past, had a bad name. "Cheap and nasty" comes to some people's minds when they are asked about plastic products. However, this attitude is changing as manufacturers pay attention to the quality of the products they are producing. Poor quality is visible in three main aspects of the product; (1) warpage, (2) surface defects and (3) premature fracture. Warpage is any unwanted twisting or bending of a product. Surface defects cover a large range of moulding problems including sink marks, visible weld lines, flow marks and air bubbles. The defects are caused by poor mould design or poor manufacturing techniques. Premature fracture occurs when the internal stresses are too high or the material used does not meet the requirements. The internal stresses are moulded in at the time of manufacture. High internal stress may be due to poor mould design, poor material choice or incorrect machine settings.

Most of the quality problems can be designed out by following a design procedure and avoiding such things as large, rapid changes in product thickness and poor boss design, as described in chapter 2. However, some of the problems cannot be predicted. These problems are ironed out once the mould has been manufactured. Trials on the mould will disclose the design faults and these may be rectified. High stresses and premature failure cannot be known until a product has been manufactured and tested. This can be a costly experience if the mould requires large or difficult changes. Many moulds require repairs or pins in sections that were not manufactured correctly the first time.

3.2.7 Low Creativity

Many moulds are designed based on previous mould designs and experience. Much of the creativity is taken out of product and mould design because the outcome of a "creative mould" is not known. A mould that is based on previous successful products will itself be more successful. Creativity must take a back seat since a working mould that will produce the expected product is required.

The problems explained in sections 3.2.1 to 3.2.7 can best be solved by implementing Computer Aided Mould Design software to assist the mould designer in making good decisions based on facts, rather than educated guesses. Creativity enters the picture since moulds can be designed using the software where the analysis results explain whether or not the product design is feasible.

3.3 Resolving Moulding Problems

There are two basic methods for resolving moulding problems: Conventional Mould Design and Computer Aided Mould Design, as outlined in chapter two. Both of these mould design methods are valid, however, the Conventional Mould Design relies heavily on the trial and error approach which can be time consuming and expensive. Conventional Mould Design is time consuming because several mould trials are necessary before a mould can be sent for manufacturing. Each mould trial is followed by design and mould changes. It is also expensive because each mould design change requires more time on machining equipment and injection moulding machines as well as the mould maker's time. By developing moulds using Computer Aided Mould Design software, a great deal of the guess work and trial work is taken out of the process, thus reducing the time taken to produce a mould from a concept.

Computer Aided Mould Design saves time, resources and money. It saves time by reducing the lead time from concept to manufactured product, saves resources by decreasing the amount of time required in trials using injection moulding machines and mould manufacturing services when mould changes are required. CAMD saves money in all areas of mould design and manufacture.

Thousands of products have been designed using CAMD software with very good results. Some of these successful mould designs have been published in plastic journals (Vallens, 1993; Mapleston, 1994).

This thesis addresses the ability of Computer Aided Mould Design software to accurately predict the way the plastic flows in a mould and the problems that could be expected in a specific, existing mould. The methodology for this study is presented in chapter 4.

3.4 Benefits Of CAMD

There are nine main benefits that result when CAMD software is used. These have been outlined in the chapter 'Computer-Aided Part And Mould Design' by C. Austin in the book Developments in Injection Moulding - 3, edited by A. Whelan and J. P. Goff (1985).

> *1. Very little tool development time. This accrues benefits not only from the development engineer's time saved, but also an increase in available capacity due to the machine being out of circulation less frequently.*
>
> *2. A reduction in the amount of material used.*
>
> *3. Reductions in regrind due to smaller runner systems used.*
>
> *4. Reductions in scrap, and therefore savings in regrind, and materials, and a reduction in the production of replacement for the scrap.*
>
> *5. Increases in productivity from faster cycle times.*
>
> *6. Less corrective machine setting.*
>
> *7. Cheaper tooling.*
>
> *8. Predictable and repeatable quality.*
>
> *9. Faster turnaround between concept and production.*

Other benefits which are not directly apparent include:

1. With a short development time, contracting for design projects can be accurately assessed. The short development time also allows shorter lead times from the project concept to manufacturing of the product.

2. Since the simulation gives the designer information on the flow and heat of the product, a consistent high quality product can be provided which will lead to a good reputation for high quality products and moulds.

3. Mould flow simulation gives the mould designer the largest possible moulding window therefore allowing flexibility in materials and processing parameters that can be used with the mould.

4. Creativity is enhanced with Computer Aided Mould Design since simulations can be run to see how a creative design might respond. Existing working mould designs do not have to be followed since the simulation of the design assists the designer by describing what will happen in the mould under specific circumstances.

CHAPTER FOUR

RESEARCH METHODOLOGY

To investigate the hypothesis, a research method was created. Each of the projects used in a simulation followed the guidelines. This chapter presents the methodology for the research.

4.1 *Introduction*

A study was carried out to compare the results of mould filling analysis using Computer Aided Mould Design software with the actual results experienced in a mould to produce an injection moulded product. The research hypothesis was:

> **Computer Aided Mould Design software can accurately predict moulding problems, namely: quality, processability and in-service requirements in a product that are not met.**

Several injection moulded products were selected to investigate the hypothesis.

The analysis involved creating a model of the injection moulded product using the Moldflow software and analysing the model with the Moldflow analysis suite to produce results describing how the plastic would flow through the mould. Analysis results indicated regions of the product design that would lead to moulding problems. The moulding problems identified were compared to the observed problems the users of the mould were experiencing. This comparison led to conclusions regarding the hypothesis. Since the results of the Moldflow analysis require interpretation by the mould designer, several projects were done to build the author's experience in using the software and in interpreting the results. Once the experience was established, projects could be carried out to prove or disprove the hypothesis.

Two companies provided products that could be used in the experience building stage and one company provided products that could be used in the study. Four products were chosen as study projects, two mould designs developed at the Production Technology Department, Massey University, and two products provided by Sunbeam, a local company in the injection moulding industry. The benefits to the companies participating in the study included:

- finding out what was occurring in the mould as the plastic was injected, at the end of filling the cavity and during the cooling phase.

- gaining feedback on the mould design in the form of recommendations to improve the mould design which will lead to an improvement in the quality, processability and/or the in-use requirements of the product.

This chapter describes the methodology followed in the study and analysis of the projects presented in chapter five. Section 4.2 explains how products were selected. Section 4.3 discusses the information required for analysis and how it was collected and collated. Section 4.4 presents the methodology for the analysis of each product.

4.2 Project Selection

Several companies who produce injection moulded products were contacted to determine if they had products which did not meet their quality, processability or in use requirements. Most of the companies contacted were experiencing these problems with several of the products they produced.

From the companies contacted, projects were chosen that represented common problems encountered by many moulding companies. Most injection moulding companies have problems with warpage of flat products, excessively long injection times and poor surface quality of the product. The last two problems are normally connected since the product with surface defects will normally be run at a slower injection time to eliminate the flow marks.

Two of the products analysed for the study were produced at the Production Technology Department, Massey University. The two industry projects were provided by Sunbeam Limited.

4.3 Data Collection

The data required for the mould analysis included the processing parameters (eg. mould temperature, temperature of the molten polymer and the injection time), a mould design drawing detailing injection points and the design of the cooling system. The drawing was used to create the computer model and the processing parameters were used in the flow simulation. The cooling system design was only required if warpage was a major concern in the product.

A data recording sheet was created to collect the required data in a format that could be quickly interpreted and entered into Moldflow. The information was required in three distinct areas; mould design, processing parameters and cooling design. The data recording sheet followed the format below.

Mould Design

A fully dimensioned drawing of the product must be attached.

Where are the injection locations? (Indicate on the drawing).

Moulding Parameters

1. *What is the injection time for this product?*

2. *What is the temperature of the plastic in the nozzle?*

3. *What is the mould temperature?*

4. *Is a pressure or flow profile used? If yes, what is the pressure profile?*

5. *What material is used in this mould? (Include supplier and grade).*

Cooling System

1. *What is the mould made of? (tick the box of the material used in each part of the mould.*

Mould Material	*Cavity*	*Core*	*Inserts*
Stainless Steel (420)			
Medium Alloy Steel (P20)			
Carbon Steel (1020)			
Be-Cu Alloy (C17000)			
Be-Cu Alloy (C17500)			
Other (Specify material and thermal conductivity)			

2. *Include a drawing of the cooling lines with respect to the cavity. Include labels on bubblers and baffles and the path of the coolant.*

3. *What cooling fluid is used?*

4. *What is the inlet temperature of the cooling fluid?*

5. *What is the outlet temperature of the cooling fluid?*

6. *What is the flow rate of the cooling fluid?*

The information collected was used in the Moldflow analysis software for the simulation of the mould design. The Moldflow flow analysis results had to be interpreted to establish the regions of the mould which could cause moulding difficulties. The analysis was relatively simple to interpret to find weld lines and air traps that may have caused weaknesses or may be visible, or find regions that do not fill completely. Other moulding problems like weak areas due to high stresses and low temperatures and warping had to be interpreted from several graphical outputs by the designer using experience in reading the results.

To avoid bias in the interpretation of the output, it was preferable that the problems experienced when using the mould were not discussed until the model analysis had been completed and the results interpreted by the author.

4.4 Design Analysis

The design analysis followed several steps, as outlined in this section. First, a moulding window was developed for the product using the 2D flow analysis environment, as described in chapter 2. The company's processing parameters were then used in the 2D flow analysis to find the shear stress, pressure and temperature ranges for the design. This answered the question: Are the processing parameters inside the moulding window for this mould and polymer material? If the answer to this question was no, there was a high probability that the processing parameters were a cause of the moulding problems. Figure 4.1 shows the quadrants outside the moulding window. If the mould were operated in one of the quadrants some or all of the moulding problems may be attributable to the processing parameters.

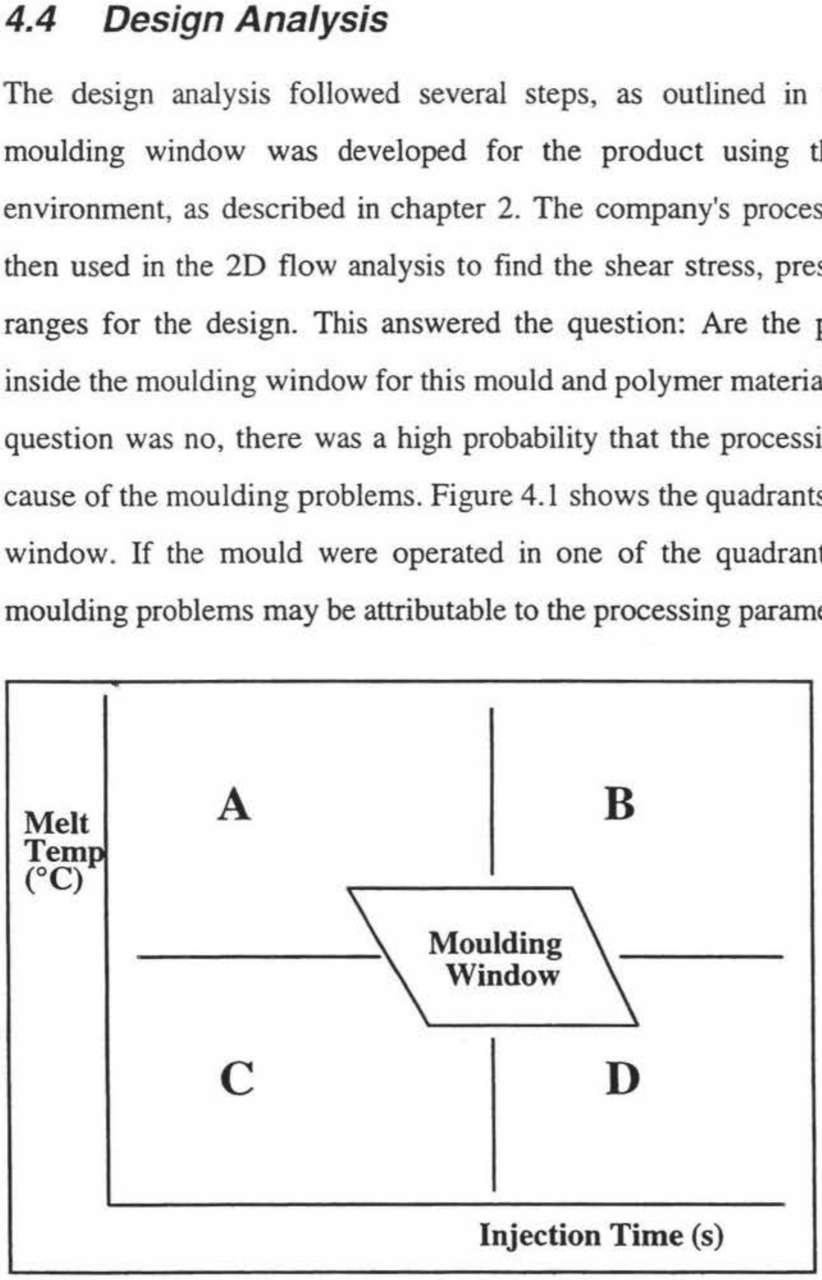


Figure 4.1 Moulding Window Quadrants

Quadrant A: High melt temperature and fast injection time.

The combination of high melt temperature and fast injection time gives rise to problems like high shear stress and high shear rate. These two factors produce high residual stress in the product which is noticeable when the product experiences stress cracking or easy breakage.

Quadrant B: High melt temperature and slow injection time.

A product running in quadrant B would experience a large end of fill temperature range and a high shear stress. The large "End of Fill" temperature range gives rise to temperatures lower than that which is ideal for the particular product. Low temperatures lead to differential cooling since several sections of the product will be colder than other sections at the end of cavity filling. Differential cooling results in warpage of the product.

The high shear stress will result in high internal stresses which could cause warpage, especially if the product is flat and has thin walls.

Quadrant C: Low melt temperature and fast injection time.

A product moulded under conditions dictated by quadrant C would experience high shear stress and high shear rate. High shear stress will result in high residual stress in the product which will cause stress cracking and easy breakage of the moulded part.

The shear rate concerns the gate design and size. A high shear rate at the gate could result in visual defects on the surface of the mould. Surface defects may result even if the shear rate at the gate is below the recommended shear rate if the plastic enters the gate and does not strike an opposing mould wall relatively quickly.

Quadrant D: Low melt temperature and slow injection time.

The combination of low melt temperature and slow injection time results in incomplete filling of the product. A high moulding pressure is required to overcome the low viscosity of the plastic at the cool temperature. A large end of fill temperature range would also be expected since the injection time is long. This situation gives rise to incomplete packing of the material in the product, since the plastic cools to a point where it is difficult for the plastic to move through the mould.

Table 4.1 summarises the expected results when a mould is operated outside the moulding window.

Quadrant	Problems	Expected Results in Product
A	High shear stress	Stress cracking Easily broken
	High shear rate	Surface marks around gate Bubbles in product
B	Large end of fill temperature range	Warpage Burning of plastic
C	High shear stress	Stress cracking Easily broken
	High shear rate	Surface marks around gate Polymer jetting Bubbles in plastic
D	Large end of fill temperature range	Warpage Incomplete filling Polymer rippling

Table 4.1 2D Analysis Problems

The next step in the analysis was to create a 3D model of the product and run a simulation using the processing parameters provided by the company. The 3D model was created either from the drawings provided with the product or, in the case of no drawings, from the sample product provided. The design was then meshed to produce a model on which the finite element analysis software could perform the necessary flow and heat calculations. Sensible responses from the software could only be produced if the mesh was correctly designed. Following the design of the meshed model, a flow simulation could be carried out. The results of the simulation were then examined in the 3D modeling environment and interpreted based on the Moldflow flow concepts presented in chapter 2 and the experience of the author.

The 2D analysis provided information on the temperature of flow fronts and the shear stress which was used to establish regions where problems such as high stress leading to cracking or warping, or incomplete filling of products might arise.

The 3D model analysis provided greater detail of the flow of the polymer through the mould cavity in the form of contour plots. The contour plots revealed the possible moulding problems which may be encountered in the moulding of the product. Table 4.2 lists the graphical outputs from the 3D model simulation and the problems which could be expected from the results. Some problems are indicated by the results of a combination of the graphical results. Normally, if there is evidence for a particular problem, for example, warpage, in the fill time plot, other graphical plots should reinforce the fill time plot result.

4.5 Comparison of Moldflow Results and Observed Problems

The Moldflow results were compared with the observed moulding problems to ascertain how effective Moldflow was at identifying the moulding problems. This comparison and the discussions with the moulding companies that followed provided the information required to prove or disprove the hypothesis.

The comparison was done by first listing the problems Moldflow analysis described. These were then compared to those problems experienced during the moulding of the product. Any spurious problems were then discussed with the injection moulding company and conclusions drawn as to whether Moldflow was accurately predicting the observed problems. The input from the injection moulding company was vital to the project since all of the information about the projects was based on the data provided by them.

The following chapter discusses the results of the projects as described in section 4.4. Each of the projects was analysed with the Moldflow software. The moulding problems were determined from the analysis results and the experience of the author to interpret the results.

MF Contour Plot	Possible Graphical Result	Problem With Product
End of Fill Pressure	Above maximum allowable pressure Large variation in pressure gradient from injection point to ends of fill	Warpage
Flow Front Temperature	Below no flow temperature Range > 20°C Above injection temperature	Incomplete filling Warpage Product burns Poor surface quality
Fill Time	Uneven filling pattern	Warpage
Flow angle	Reverse flow	Poor weld line - breaks easily
% Frozen at End of Fill	100% frozen at end of fill	Incomplete filling
Cooling Time	Long cooling time for runners and sprue	Excessive cooling time required
End of Fill Temperature	Range > 20°C Below no flow temperature Above injection temperature	Warpage Incomplete filling Excessive shear heating
Air Traps	Traps in unventable position	Bubbles in product Incomplete filling
Maximum Shear Rate	Gate shear rate > maximum allowable Product shear rate high (near maximum allowable)	Poor product quality around gate Possible surface defects Excessive shear heating
Maximum Shear Stress	Shear stress in product > maximum allowable	Stress cracking Breaks easily
Weld Lines	Weld lines in cold positions in product	Breaks easily at weld

Table 4.2 3D Model Simulation Results

CHAPTER FIVE

PROJECT DESCRIPTIONS AND RESULTS

This chapter analyses the four projects that were carried out to test the hypothesis. Two of the projects, the storage box and the planter pots, were products manufactured in the Production Technology Department. The knob and the strip were products supplied by Sunbeam Corporation Limited. The figures and further discussion of results are located in appendix 4.

5.1 *Storage Box - Massey University, Production Technology*

5.1.1 Summary

The storage box was a simple product for storing small objects like nuts and bolts. It was to be manufactured at low cost from a commodity plastic material, possibly recycled plastic.

The box, when manufactured, functioned well. However, the mould had several defects that required repairing before any consistency in quality of the product could be achieved. The mould itself had been poorly constructed. The cavity was made out of several different steel plates, and the core was not sunken into the bolster, rather, it was merely bolted to the surface of the bolster. The mould parts moved when the plastic was injected and could not hold the cavity shape. A better construction would have resulted in a mould which would last longer and have greater stability when used.

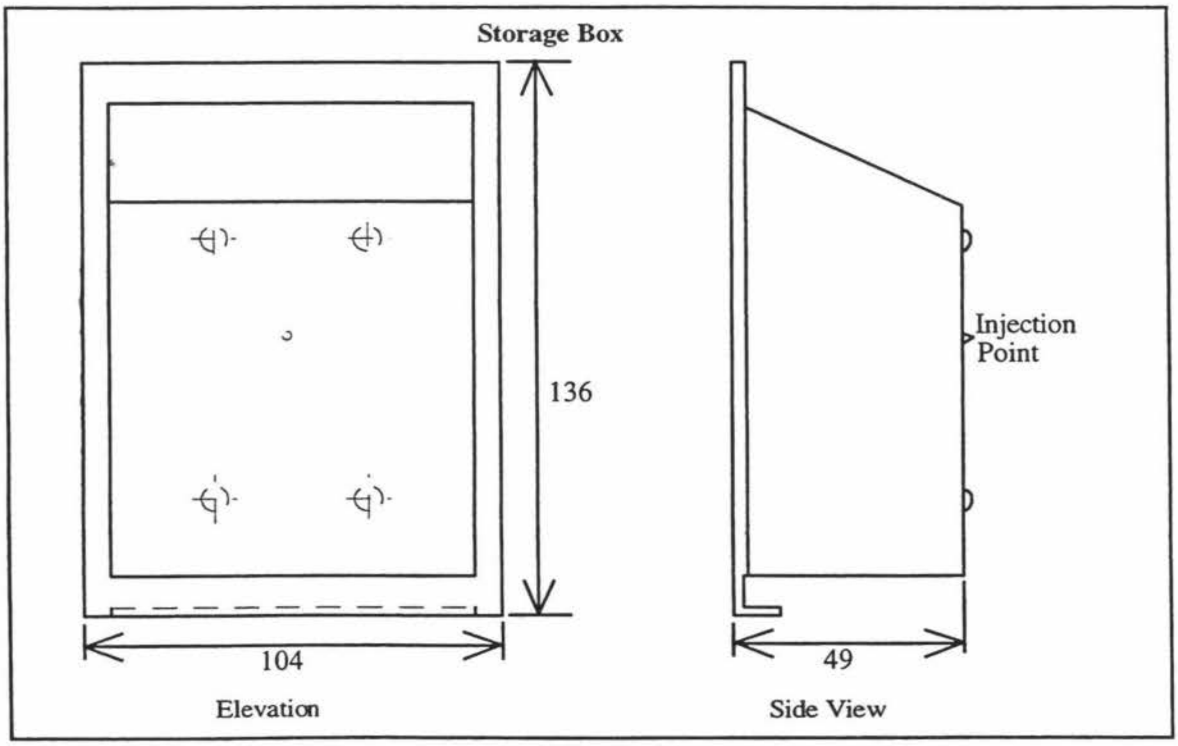


Figure 5.1 Storage Box

The material for the storage box had very simple requirements. It should be resistant to most chemicals, especially oils and greases. It should be tough to withstand knocks associated with small storage boxes. The material also must be cheap so that the cost of the box is kept low. All of these requirements are met in polyethylene. High density polyethylene is also widely available as a recycled material.

The following information was gathered from the technicians in the Technology Department regarding the box and the processing parameters that were used for manufacturing the storage box:

- Product supplied, with injection point indicated
- Injection time for the box: 4.5 seconds
- Melt temperature of the plastic: 220°C
- Mould temperature: 30°C
- Maximum injection pressure of the machine used: 200 MPa
- Maximum clamp force of the machine used: 100 tonnes
- No flow profile was used for this product.
- Material: Polyethylene
- Supplier: unknown, a recycled material will be used
- Grade: unknown

5.1.2 Moldflow Simulation of the Box Model

Moldflow was used to analyse the existing mould design and predict the problems encountered in manufacturing the product. Since drawings of the storage box were not available, a 3D wire frame was created from the actual product. The drawings would not give an accurate picture of the actual product anyway, since the movement of the core changed the shape and wall thicknesses of the product.

A polyethylene material was chosen from the database of materials that had middle of the range flow properties. Lupolen 5031L, a BASF polyethylene, was used for the analysis.

A 2D model was constructed to determine the moulding window (figure 5.2).

The processing parameters used for the box were just outside this boundary, however, they were not outside the moulding window. The window for the Lupolen polyethylene may be different from the window for the recycled material. Therefore a robust product design would be needed to ensure the changes in the recycled material properties will be accommodated.

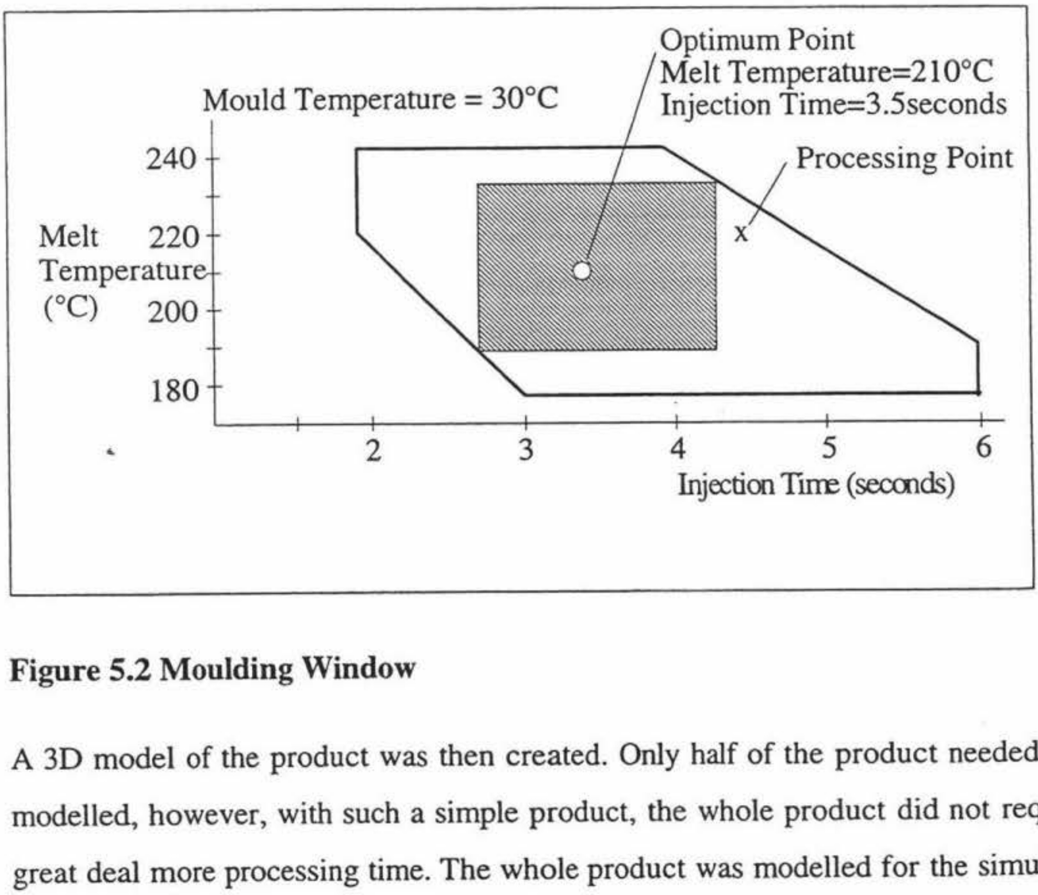


Figure 5.2 Moulding Window

A 3D model of the product was then created. Only half of the product needed to be modelled, however, with such a simple product, the whole product did not require a great deal more processing time. The whole product was modelled for the simulation. The model consisted of 37 surfaces, which were meshed into 670 elements and 357 nodes. A finer mesh was used to ensure the results of this mesh were sufficiently accurate. Similar results were obtained.

The analysis revealed an air trap in the corner of the box. The evidence for the air trap included the freezing of the polyethylene before it reached the corner. This was observed in the time plot (figure A4.5, appendix 4), and the air trap and weld line plots (figure A4.6, appendix 4) confirmed the results. The flow front temperature plot (figure A4.7, appendix 4) also validated the existence of an air trap since the temperature in the corner of the box was below the no flow temperature.

A second problem was the unbalanced flow of plastic between the front and rear of the box. This was due primarily to the gate position and the difference in wall thickness between the two regions. The root cause of the unbalanced flow, improper core construction, could not be resolved by re-design of the product, however, balancing the flows between the front and rear of the box could be solved with a Moldflow simulation. The unbalanced flow was apparent in the time plot since the front of the box filled first. A plot of the pressure at the end of fill (figure A4.6, appendix 4) and during filling verified this result. High pressure in the front of the box indicated that it had filled and was over packed.

The differential packing was caused by the difference in length of the flow paths and the differential thicknesses of the front and rear of the box. The front of the box filled first by more than one second which allowed further plastic to compress into this volume. Once the rear of the box was filled, the front had been significantly compressed, resulting in differential shrinkage. The difference in thicknesses also resulted in differential cooling which compounded the warpage problem.

The high pressure would also cause high internal stresses which would be seen as warpage. Warpage in the model is verified by the low end of fill temperature at the rear of the box.

5.1.3 Storage Box Moulding Problems

On careful examination of the product, the main problems were quite discernible. The cavity was not filling correctly and an air trap was evident in the right rear corner of the box (figure A4.12, appendix 4). The air trap was caused by a race track effect, in turn caused by a thin side wall around the air trap region and a thick edge around the top of the box. This effect is quite noticeable in the product via the pattern of the flow lines.

A second problem in the finished product was the warping of the box (figures A4.8 and A4.9, appendix 4). The warping was pronounced in the finished product. The side walls gave the product stability, however, the box still suffered from warpage.

5.1.4 Comparison of Results

The results from Moldflow compared favourably with the manufactured product. Moldflow predicted the problems encountered with the box. The air trap in the corner of the box was foreseen, and the warpage of the product could be predicted from the results of the filling, pressure and temperature plots.

The warping could easily be explained in the simulation since the two ends of the box filled at different times and the temperature plots showed a large variation in the end of fill temperature, which led to differential cooling and warpage. Deciding where and when the product is likely to warp requires interpretation of the contour plots. If one section of the cavity fills before another and the end of fill temperature range is greater than 20°C, warpage will normally result, especially if the product is flat or thin walled. In the case of the storage box, the two plots revealed the presence of warpage. The warping was extraordinary because it was able to overcome the molecular resistance even though the thickness of the side walls was between two and three millimetres.

The box is an extremely simple product, but the example shows that even simple moulds may result in poor products if the design and manufacturing of the mould is not carried out using tested guide-lines and correct manufacturing procedures.

Moldflow predicted the problems encountered in the storage box. Table 5.1 is a summary of the moulding problems and whether Moldflow was able to predict the user defined problems.

Mould Problems	Moldflow Predicted	User Defined
Air trap in the corner	✔	✔
Warpage at the ends	✔	✔
Warpage of the base	*	✔
Hesitation effect	✔	
Race track effect	✔	

** Warpage in specific location is difficult to predict without the warpage software. However, warpage in the box was expected.*

Table 5.1 Moldflow Results and User-Defined Problems for the Storage Box

5.1.5 Design Changes and Results

The second part of this project was to use Moldflow to recommend design changes that would allow the product to fill completely and eliminate the shifting of the core. This could be done by ensuring the front and back of the box filled at the same time. One method would be to grade the thickness of the base of the box, creating flow leaders and flow deflectors. This method is normally used with products that have flat bases.

A simpler approach was taken. The runners on either side of the channel, used for holding the box divider, could be used as flow leaders to the rear of the box. This design is not ideal since it would set up internal stresses in the box. However, the box is thick and the dimensions of the rim are such that the box would be able to withstand the stresses. The box is not a high performance part and does not require precise dimensions. Increasing the size of the runners would therefore be an acceptable method of improving the performance of the mould. Moldflow was used to simulate different runner thicknesses until the box filled evenly. The runners were then machined to the size recommended by Moldflow.

The second problem was the hole in the right side of the box. To eliminate this, the thickness of the right side had to be increased. This could be simulated on Moldflow, however, the solution was simple. The core required grinding on the thin side to remove the difference in thickness between the two sides. This was carried out and the result was a box that filled well without a hole appearing. This was a case where rational thinking was required, rather than computer results. Moldflow was used to simulate the changed model for comparison to the manufactured product.

The results of the simulation showed a filling pattern similar to that which occurred in the injection moulding machine (figures A4.14 - A4.17, appendix 4). The severe warping of the product, caused by the high levels of over- and under-packing, is not seen in the finished product because the product fills more evenly. This was predicted from the Moldflow results.

5.2 Planter Pots - Massey University, Production Technology

5.2.1 Summary

The planter pot was designed for raising seedlings. There are many seedling pots manufactured in New Zealand, however, very few are manufactured by the injection moulding process. Injection moulding could be used to manufacture the pot from a recycled polymer.

In the design, four pots are produced in one shot and are connected by a top plate which is strengthened with a three millimetre rim around the outside. Each pot has 20 square runners down the length with a solid reinforcing ring around one third of the length of the pot (figure 5.3).

A simulation was carried out on a model of the product to establish where problems might lie. The analysis gave results that showed the product was easy to fill with polyethylene, however, the shape of the pots produced a complex flow pattern. Welds at the ends of the fill were concerns since the temperature in these regions was low by the time the polymer arrived, resulting in weak welds.

The shape of the product added to the difficulty of filling it without warping. The four pots were filled from one gate at the centre of the top plate. A desirable flow pattern would be one where the polymer flowed around the top quickly and then evenly down the sides of the pots. In the moulding of the product, the top did not fill quickly and the polymer flowed down the inside of the pots first. A warpage problem was evident from the filling results since the top was not filled evenly and would suffer from over-packing.

The material requirements were fairly simple. The plastic had to be cheap, have good UV stability and be chemically stable. A recycled polyethylene would be ideal for this product since the product does not come into contact with any food. For the analysis, BASF Lupolen 5031L was used since it is a high density polyethylene with medium range flow characteristics.

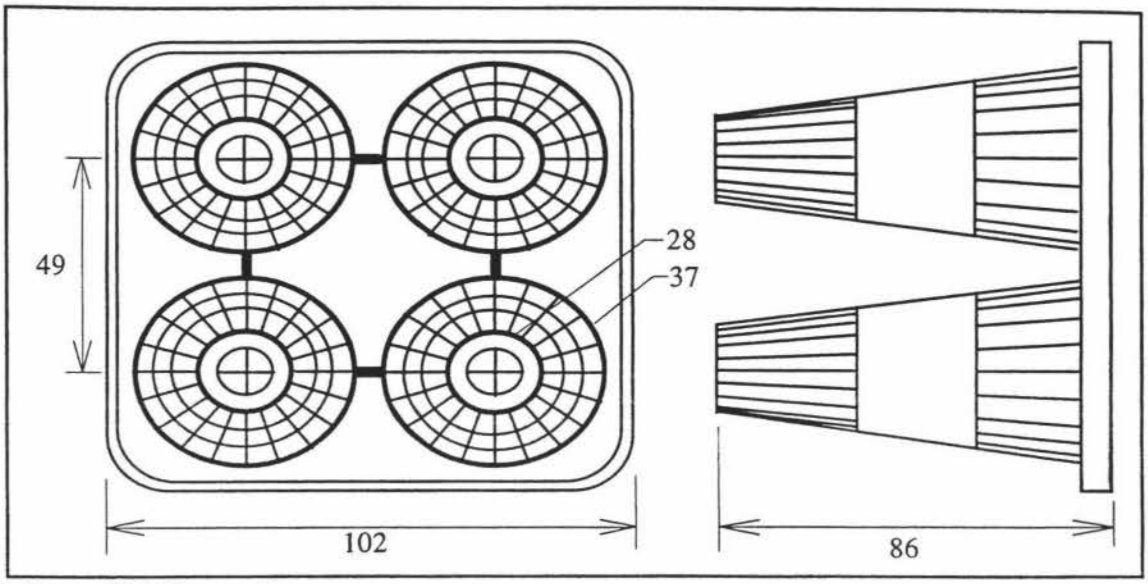


Figure 5.3 Planter Pots

The following information was gathered from the Production Technology technicians regarding the planter pot mould design and the processing parameters used for manufacturing the product:

- Sample of the product, noting the injection point.
- Injection time for the pots: 5 seconds
- Melt temperature of the plastic: 260°C
- Mould temperature: 40°C
- Maximum injection pressure of the machine used: 200 MPa
- Maximum clamp force of the machine used: 100 tonnes
- No flow profile was used for this product.
- Material: Polyethylene
- Supplier: unknown, a recycled material will be used.
- Grade: unknown

5.2.2 Moldflow Simulation Results for the Planter Pot Model

The analysis of the planter pots started with the creation of a 2D flow path file to find the moulding window and compare the moulding parameters with the window. One dominant flow path exists in the product (figure A4.19, appendix 4). It begins at the gate and continues down to the base of the pot. This flow path produces the moulding window shown in figure 5.4.

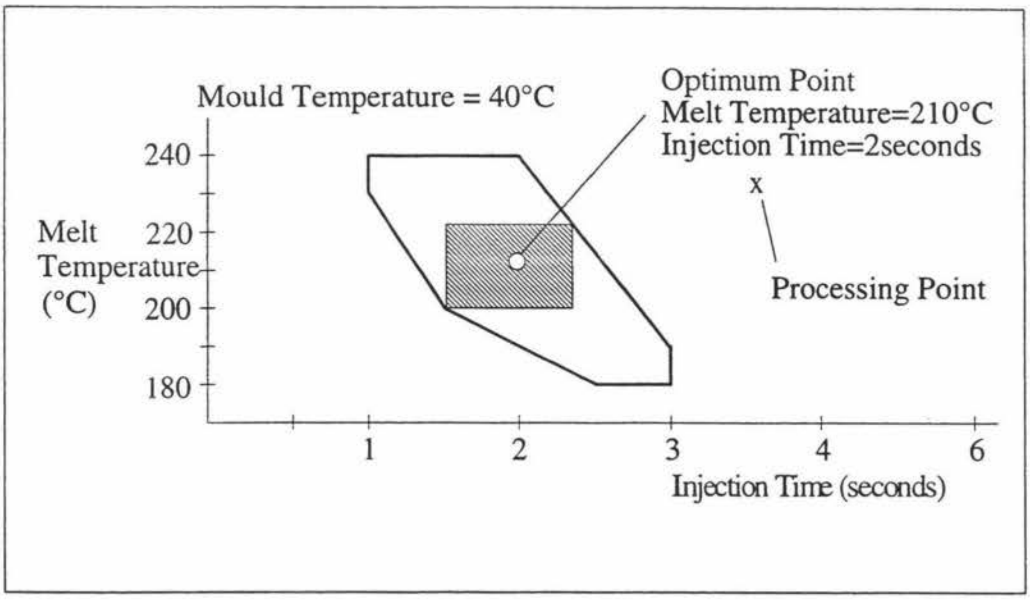


Figure 5.4 Moulding Window

The moulding window is small for the planter pot model and Lupolen material. The moulding window could be enlarged if a material with a higher viscosity were used. The processing parameters used are obviously not within the moulding window, rather they are in quadrant B. The expected results would include warpage of the product and weak weld lines at the end of the flow path caused by a large end of fill temperature range. The temperature at the end of the flow path would be near the no flow temperature.

A 3D model was created for a multi laminate filling simulation. Only one quarter of the product needed to be modelled since the product is symmetrical about two perpendicular planes. The model consisted of 97 surfaces which were meshed into 1239 elements and 898 nodes. The large number of elements were necessary since the 20 runners down the length of the pots were narrow.

A simulation was carried out using the processing parameters provided by the technicians. The simulation revealed two main problems with the design. The first was the low end of fill temperature. The temperature at the base of the pots was too low for a good weld to form and consequently a weak region would be expected where the polymer flow fronts met. The meeting point is in the bottom of the pots, which is also the thinnest section. The thickness of the cross at the base is less than one millimetre. Plots of the weld line positions and the flow front temperature (figures A4.21 and A4.22, appendix 4) revealed the weakness of the base.

The second problem the simulation results implied was the warpage of the top. The sides of the top fill before the corners and over-packing occurs in the side regions. The corners fill at the end of filling and therefore are under-packed. This differential packing causes warpage in the product since the two regions will shrink at different rates. The warpage is suggested by the time and pressure plots of the planter pots (figures A4.23 and A4.24, appendix 4).

5.2.3 Planter Pot Moulding Problems

The planter pots showed visible warpage along the tops (figure A4.25, appendix 4). This warpage caused the top to bow downwards. The warpage did not impede the function of the product and therefore was not a concern. However, it was still a visible problem and could be solved by redesigning the product using Moldflow principles.

The base of the pots was very weak. The weakness of the base may have been due to the thin cross section of the material in the area, however, the part tended to snap rather than bend and stretch as would be expected with polyethylene. The snapping of the polyethylene would indicate a poor welding of the polymer and probably low crystallinity. This would be a problem since the pots were designed for repeated use. If the base broke through, the soil in the pot would begin to fall out the bottom. The weak base needed to be redesigned.

5.2.4 Comparison of Results

The results obtained from the Moldflow simulation compared favourably with the manufactured product. The simulation predicted the weak region at the base of each of the pots, due to the low temperature welds. The point at which the weld met in each of the pots was the weakest point in the product, hence one could conclude that the weakness was due to the poor welds. The author predicted the warpage at the top of the product by the details given in the time and pressure plots.

Low temperature welds were predicted from the flow front temperature plot and the weld line plot. A weld line in a region where the flow front temperature is near the injection temperature will bind well and remain strong. However, when the temperature is low (near the no flow temperature) the weld will be poor, creating a weak join. In the case of the planter pot, the flow front temperature was low in the weld region at the base of the pots. This weld would be weak and susceptible to cracking under low stress.

Warpage of the top of the product was apparent from the fill and pressure plots. An uneven filling always results in internal stress in the product. The internal stress will oppose the molecular bonding of the polymer. If the internal stress is greater than the bonding, the product will warp. Harder plastics can absorb greater internal stresses before they will warp. Polyethylene is a soft, pliable plastic which will warp under low internal stresses.

Moldflow predicted the problems encountered in the planter pots. Table 5.2 is a summary of the moulding problems and when Moldflow was able to predict the user defined problems.

Mould Problems	Moldflow Predicted	User Defined
Warpage of the top	✔	✔
Weak welds	✔	✔

Table 5.2 Moldflow Results and User-Defined Problems for the Planter Pots

5.3 *Crockpot Lid Knob - Sunbeam*

5.3.1 Summary

A knob was designed in polycarbonate for the Crockpot lid. Previous Crockpot lids were manufactured from glass. Sunbeam changed to manufacturing the lids out of plastic because the plastic product was much cheaper to manufacture and the company could manufacture the part at the factory rather than contracting out the manufacture of a glass lid. Polycarbonate is less likely to break when dropped or knocked and therefore is safer than the glass equivalent.

The knob is a fairly simple design (figure 5.5). It consists of a round top for gripping and a stepped, hollow shaft for fitting into the top of the Crockpot lid. The knob press fits into the Crockpot lid and is held by the friction between the two components. The outside diameter of the shaft must be accurately manufactured since it must press fit onto the lid.

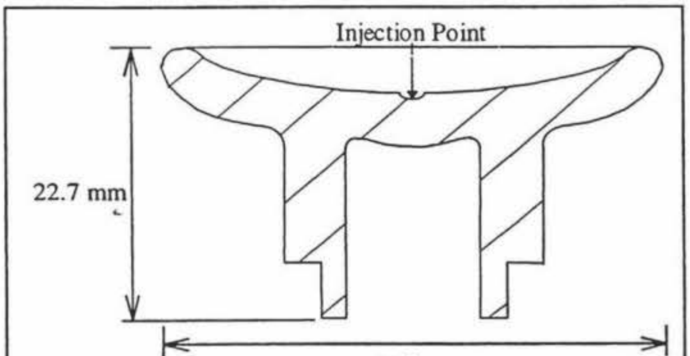


Figure 5.5 Crockpot Lid Knob

The material chosen for the product required the following specifications:

- food and drug approved
- ultra violet light stabilised
- good chemical resistance
- transparent
- stable at temperatures of 120°C

The material selected had all of these properties. Polycarbonate is a very strong, high temperature engineering polymer. It is transparent and may be tinted to many different colours. Polycarbonate is resistant to most chemicals and is food and drug approved. The heat distortion temperature at 1.82 MPa is 127°C and therefore is well suited to the temperatures and pressures it would experience in contact with the Crockpot.

The following information was gathered from Sunbeam regarding the design of the knob and the processing parameters used for manufacturing the product:

- Drawing of the product supplied, with injection point indicated.
- Injection time for the knob: 12 seconds
- Melt temperature of the plastic: 270 °C
- Mould temperature: 80 °C
- Maximum clamp force of the machine used: 50 tonnes
- No flow profile was used for this product.
- Material: Polycarbonate
- Supplier: Dow Chemicals
- Grade: Calibre 201-15, Calibre 301-15 was used for the analysis

Calibre 201-15 was not in the Moldflow database of materials. A suitable substitute was required. Calibre 301-15 is very similar in flow characteristics and the differences between these two grades did not affect the results of the analysis.

5.3.2 Moldflow Simulation of the Knob Model

A moulding window for the product was first found using the 2D Moldflow software. With such a simple product the flow paths were simple to find. Two flow paths exist in the knob, one to the edges of the radiused handle and one to the base (figure A4.27, appendix 4). The longer of the two was the flow path from the injection point to the base and produced the moulding window figure 5.6.

The processing parameters, namely, the polymer melt temperature, mould temperature and injection time, used by Sunbeam for the knob, when compared with the moulding window, revealed that the injection time was too long. The optimum time for the injection, according to the moulding window, was one second. Sunbeam had been using twelve seconds. The processing parameters are in quadrant D where a high moulding pressure and a large end of fill temperature range would be expected. The effects of these problems are warpage, incomplete filling and polymer rippling. Since the knob is such a thick, chunky product, warpage would not be a problem, however the high moulded-in stress would remain in the product and any excessive heating may cause the knob to change shape. Incomplete filling should be apparent, as should rippling since the knob is transparent.

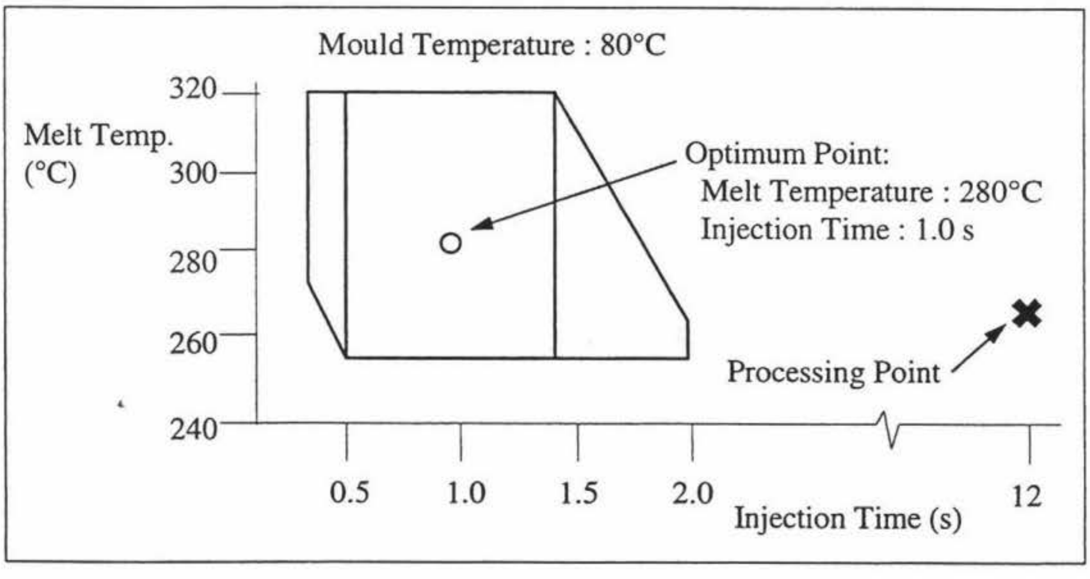


Figure 5.6 Moulding Window for the Knob

The second exercise was to construct a model of the knob using the 3D software. The model required only one quarter of the total product since the knob is symmetrical about the central axis. The knob model consisted of 42 surfaces, which were meshed into 228 elements and 136 nodes. To ensure the results of this analysis were reliable, a finer mesh was also created. The two analyses gave very similar results and therefore the coarser mesh was quite sufficient for the analysis.

The simulation was run using the processing parameters provided by Sunbeam. The results of the simulation showed the main concerns with the mould. The first observable problem was the flow front temperature range (figure A4.29, appendix 4). Normally, the designer would attempt to maintain the temperature at the flow front between 20°C below the injection temperature and the injection temperature. The range for this analysis was clearly too large (214°C to 270°C). The visible defects in the product would include marks, for example rippling of the plastic below the surface, that indicate the temperature had dropped below the acceptable lower limit (20°C below the injection temperature). Since the product is transparent, it was expected that these marks would be quite visible.

5.3.3 Knob Moulding Problems

The knob mould was built as a single cavity mould so that it could be manufactured quickly and production started with a short lead time. The mould is therefore not ideal and should be a four cavity die. Even so, the mould is not running as well as it could.

Sunbeam realise the cycle time is too long. An injection time of twelve seconds is used to eliminate the surface defects due to splashing of the plastic as it enters the mould. The main reason for this splashing is the length of the flow before it strikes a wall. This causes a significant amount of plastic to enter the cavity as a jet, rather than a flow front. The jetting plastic freezes when it strikes a wall and remains there as a surface defect. By increasing the injection time, the flow enters as a front, rather than a jet, eliminating the splash marks.

The injection time could not be decreased without the visible flow marks forming on the surface of the product. Unfortunately, increasing the time to fill introduced rippling on the outside of the product opposite the gate and out towards the end of the knob (figure A4.30, appendix 4). The rippling of the product was quite noticeable, but acceptable as far as the customer was concerned.

5.3.4 Comparison of Results

The rippling marks in the knob were predicted by the author from the Moldflow software results. The reason behind this conclusion was the temperature range at the end of fill resulting in a folding of the layers of the material as it continued to enter the product. The trial run of the product and the resulting knobs that were produced showed the rippling on the surface of the product. This rippling is due to the knob not filling completely and pulling away from the surface of the mould as it cooled and shrunk. The ripples were quite visible in the product around the underneath of the knob.

Moldflow predicted a low end of fill temperature and incomplete filling. The knob was not being filled completely because the time was too long for the injection moulding machine to compensate for the shrinkage of the polycarbonate. This allowed the knob to shrink and pull away from the surface of the mould. Moldflow predicted a premature freezing of the material before it reached the end of filling. This was not the case experienced by Sunbeam. The product did not fill completely, however, the ends of the knob were complete. The incomplete filling was over the surface of the product and was caused by the low temperature at the end of filling. As the polymer froze it pulled away from the mould walls. The ripples are due to the difference in shrinkage over the surface of the product.

Moldflow predicted the problems encountered in the Crockpot lid knob. Table 5.3 is a summary of the moulding problems and when Moldflow was able to predict the user defined problems. Moldflow did not explicitly identify the incomplete filling in the same manner as was encountered in the mould. However, the product did not fill completely in other areas and therefore the Moldflow results were still quite relevant. The thick wall dimensions would have aided the filling to the ends of the product.

Mould Problems	Moldflow Predicted	User Defined
Incomplete filling	✔	✔
Rippling of the surface	✔	✔

Table 5.3 Moldflow Results and User-Defined Problems for the Planter Pots

5.4 *Strips For Bench top Oven - Sunbeam*

5.4.1 Summary

Sunbeam produce a range of electrical appliances, one of which is a bench top oven. The controls for the oven are housed in a white unit, which is where the strips are located.

The strips are 11 mm wide and 213 mm long (figure 5.7). The length to width ratio raises concern regarding warping. Filling of the product must be done correctly to reduce the probability of warpage of the part. The strips should not be over-packed or under-packed at the end of a flow path and cooling must also be planned carefully to give even cooling throughout the whole strip.

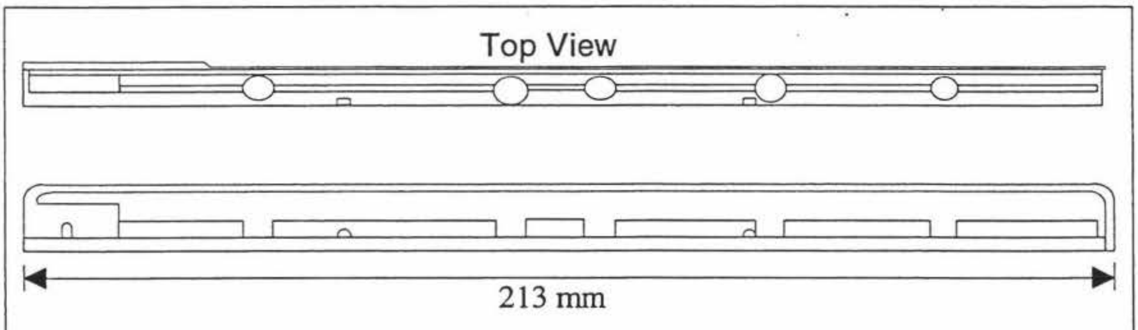


Figure 5.7 Strips for the Bench Top Oven

The polymer material chosen for the part required the following specifications:

- Strength at high temperatures

- FDA approved

The polymer selected for the strips was polybutylene terephthalate (PBT) with a 30% glass fibre reinforcing. The material was coloured black. The particular PBT chosen was Valox 420 produced by General Electric Plastics. This material was available in the Moldflow database.

The following information was gathered from Sunbeam regarding the design of the strips and the processing parameters used for manufacturing the product:

Drawing of the product supplied, with injection point indicated.

- Injection time for the strips: 6 seconds
- Melt temperature of the plastic: 270 °C
- Mould temperature: 70 °C
- Maximum clamp force of the machine used: 50 tonnes
- No flow profile was used for this product.
- Material: Polybutylene terephthalate (PBT)
- Supplier: General Electric Plastics
- Grade: Valox 420, 30% glass fibre

5.4.2 Moldflow Simulation of the Strips

The mould has two cavities, one for the left hand strip and one for the right hand strip. To reduce some of the warping of the product the strip mould was changed from two gates into each cavity to one gate per cavity.

The two strips are very similar and therefore the simulation was carried out on the right hand strip first. The left hand strip was added later. The flow paths for the product are simply from the gate to either end of the strips. This yields two distinct flow paths from the gate. The gate is not at the centre of the mould and therefore the two ends of the strip do not fill at the same time. This causes over-packing at the end that fills first and under-packing at the end that fills last.

A moulding window for one of the strips was first found using the 2D Moldflow software. The longest flow path was obvious and was used to find the moulding window (figure 5.8).

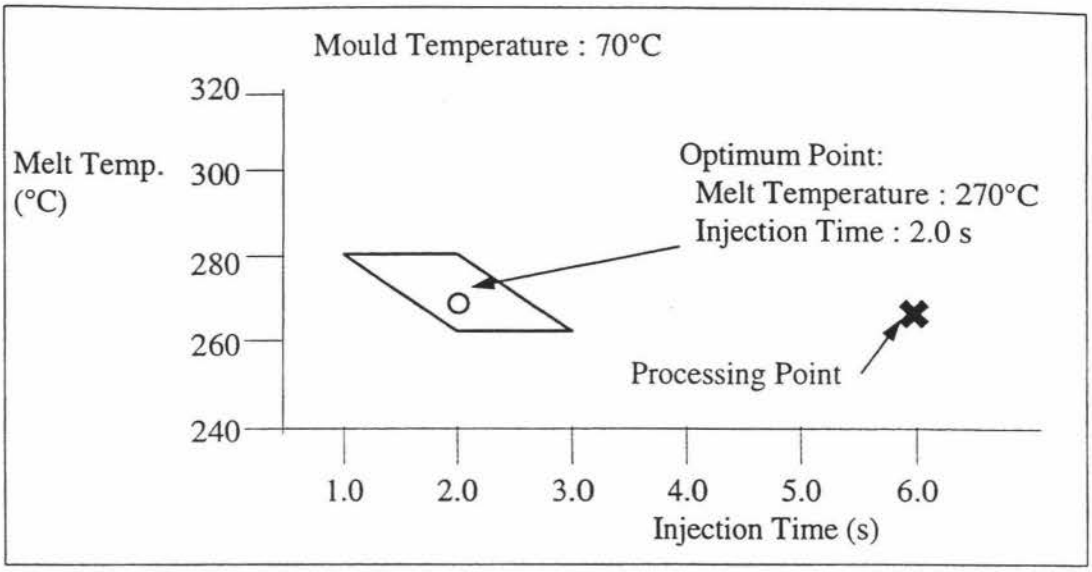


Figure 5.8 Moulding Window for the Strips

The processing parameters used by Sunbeam were compared to the moulding window. The injection time was too long. An injection time of 6 seconds (Sunbeam) would result in the product freezing off just before it had filled completely. The main problem with such a long injection time is the high shear stresses and possible warpage of the product due to the low flow front temperature.

A 3D model was created from the drawings supplied. The strip was designed with 126 surfaces which were meshed into 499 nodes and 701 elements.

The multi laminate simulation on the 3D model of the strips produced an expected result for the gating system of the strip mould. The nearer end filled first and began to over pack while the far end continued to fill (figure A4.33, appendix 4). The pressure in the near end was around 40 MPa when the far end finished filling (figure A4.34, appendix 4). This pressure differential causes internal stresses at one end of the product which results in differential shrinkage. Differential shrinkage causes warpage of the product.

5.4.3 Strip Moulding Problems

The warpage of the strips was quite visible (figure A4.35, appendix 4). The deflection over the length of the strip was around seven millimetres. The warpage was such that the product was still operational. The strip was held in place, in the control box, at either end which pulled the strip straight. However, the warpage was a problem and changes to the design to produce a straighter product were desirable.

5.4.4 Comparison of Results

The warpage in the product could be predicted by the Moldflow analysis since one end of the product filled before the other. This differential filling caused differential packing which lead to differential shrinkage of the product. The differential shrinkage resulted in the warpage of the product. The amount of warpage could not be predicted without the MF/Warp software. However, the fact that the product would warp was quite apparent from the flow simulation carried out, and even from the design and position of the gate.

Moldflow predicted the problems encountered in the strips. Table 5.4 is a summary of the moulding problems and when Moldflow was able to predict the user defined problems.

Mould Problems	Moldflow Predicted	User Defined
Warpage	✔	✔

Table 5.4 Moldflow Results and User-Defined Problems for the Strips

5.5 Summary

The four projects were all fairly simple products, however, they all demonstrated the ability of Moldflow to predict moulding problems. Even without the warpage software, warpage can be detected to a certain extent, by recognising the contour patterns that predict possible warpage.

Warpage was apparent in several of the projects. The different plots that revealed the presence of warpage were the fill time plot and the pressure plot. When the polymer fills one region of the mould before another, at different pressures, the product may warp. The contour lines in the plots also provide information on whether or not the product will warp. The warpage in each of the products that had a warpage problem was detected from the Moldflow simulation results. The warpage was mainly due to unbalanced filling in three of the products. It was especially noticeable in the strip and the storage box. The unbalanced filling in both of these products was considerable. A comparison between the storage box before and after the design changes showed great improvements in the warpage since the polymer flow in the product had been balanced.

The planter pots had a problem with low temperature welds. The Moldflow simulation results included the flow front temperature plot and the weld line plot which, when compared, gave information on the temperature at the locations of the weld lines. In the case of the planter pots, the temperature in these regions was low, hence weak weld lines would be expected. When the welds were tested on moulded products, they were weaker than expected.

The storage box suffered from an air trap in a position where it was very difficult to vent. The cause of the air trap was the differences in thickness between the side wall of the box and the rim around the top. Three Moldflow plots were examined to determine the extent of the air trap. The fill time plot showed the region where the hole was present to fill last. The air trap plot showed air traps in the corner and along the side of the box. The percent frozen plot displayed the amount of plastic that was frozen at the end of the filling stage. A large region around the suspect corner was completely frozen. These three plots led to the conclusion that the product would have a hole, or at the very least a very poor weld. The moulded product had a large hole the size of a five cent piece in the corner of the box.

Two problems in the knob project were not detected by the Moldflow results. Moldflow does not show the designer how to design a product without surface defects. Jetting caused the plastic to spray onto the surface of the mould at short injection times. The results from Moldflow produced a moulding window which gave a 0.5 to 1.5 second injection time range. When the plastic was injected in one second, the surface of the product was sprayed with jetting plastic. The injection time had to be increased to reduce this problem.

The gate design can cause jetting and spraying of plastic. A mould designer must be aware of this and design the gate so that the plastic does not jet as it enters the mould.

Table 5.5 summarises the types of problems that were predicted by Moldflow and whether the predicted problems were confirmed in the results.

Moldflow Detects:	Confirmed	Not Tested	Questionable
Warpage from Over-packing	✔		
Warpage from Differential Shrinkage	✔		
Air traps from Racetrack Effect	✔		
Incomplete Filling from Hesitation		✔	
Weak Weld Lines	✔		
Sink Marks		✔	
Warpage from Unbalanced Flow	✔		
Stress Cracking from High Shear Stress		✔	
Poor Strength from Internal Stresses		✔	
Poor Quality from Low Temp. Filling	✔		

Table 5.5 Types of Problems Solved by Moldflow, Confirmed in Results

CAMD software packages overcome the problems outlined in chapter 3 and restated below:

1. Long, usually unknown, development time, from concept to production.
2. High tooling cost.
3. Excessive material in the runner system.
4. Low quality, attributable to: high internal stresses.
   warpage.
   surface defects.

5. Small moulding window.

6. Low creativity in designs.

From the results of the projects reported in chapter five, several conclusions can be established relating to the five problems regions.

1. The development time for a mould can be "tied down". The box model was redesigned with minimum effort and the new model showed good results. The trial and error and redesigning stages had unknown time periods. These two stages are practically eliminated when CAMD is used. With CAMD, the development time includes:

   Concept to CAD drawing.

   CAD drawing to Moldflow model (now faster with MF/Midplane).

   Simulation and analysis of Moldflow model.

   Mould design based on Moldflow model.

   Mould manufacture.

   Short trial before hardening the mould.

   Mould to manufacture centre.

   All of these times are known and can be accounted using, for example, a GANTT Chart. There are no tasks in the process which could lengthen the time to manufacture the mould.

2. The relatively high tooling cost will be reduced. The models created in Moldflow functioned in a similar manner to the actual moulds used in the injection moulding machine. Therefore a model created and analysed in Moldflow will give similar results to the actual product. The time needed to make changes to the mould after it has been manufactured is reduced considerably and may even be eliminated all together. This reduces the amount of machining required and therefore the tooling cost is rêduced.

3. The simulations did not include runner assessments and therefore no comment can be made regarding the runner material savings.

4. In all of the products assessed, the quality was lower than what was wanted. Most of the products suffered from warpage. The Moldflow analysis results identified warpage and high internal stress which could be eliminated, or at least reduced to an acceptable level, through design changes using Moldflow. A product could therefore be manufactured without quality defects through the use of Moldflow before the mould is created.

5. Small moulding windows can be enlarged when using Moldflow since the moulding window is identified before the mould is manufactured. By selecting a different material or enlarging the thickness of several sections, the window can be widened. this was quite evident in several of the cases. The strips, for example, had a very small moulding window. This window could have been increased in size through the use of a different grade of PBT.

6. Creativity is difficult to measure. The simple products could have been manufactured differently, however, they all suited the purpose for which they were designed. Creative designs could have been modelled before manufacturing the moulds. The results of the models would provide a good indication as to whether they would mould successfully. This would assist in the creativity of the design.

5.6 Conclusions

The three project objectives and the hypothesis were:

1. To determine the benefits of computer software for the analysis of injection moulds through verification techniques. This will be carried out by studying several products, analysing them using Moldflow and drawing conclusions regarding the effectiveness of the software to predict moulding problems in the mould design.

2. To increase the Production Technology Department's awareness of available technology for injection moulding.

3. To develop an understanding of the concepts behind Computer Aided Mould Design software.

Hypothesis : Computer Aided Mould Design software can accurately predict moulding problems, namely: quality, processability and in-service requirements in a product that are not met.

The four projects completed for the study produced results which illustrated the benefits of using Moldflow. The problems predicted by Moldflow correlated well with the problems observed in the products. Three of the four products suffered from warpage which was established from the Moldflow results. The other moulding problems, including weak weld lines and air traps, were also diagnosed using the software.

With the exception of the flow marks in the lid knob, all of the problems were identified and could be eliminated or reduced by changing the design of the product. The flow marks in the knob were caused by the gate position and the type of material used. A designer must be aware of these difficulties and design the product and choose the correct material for the product.

The Production Technology Department is interested in continuing to expose students to the technologies available in many different industries. Plastics and the design of plastic products is one of the industries which is promoted in several of the degree options. With the availability of Moldflow for training purposes, the department is exposed to one of the leading technologies in the plastic product design and manufacture. The availability of this thesis to other students and staff will highlight the benefits of the software.

The concepts of CAMD were developed in the introduction chapters. CAMD is a very useful tool for assisting mould designers with plastic product and mould design.

The four projects analysed with Moldflow showed accurate predictions of moulding problems. One can conclude, therefore, that Moldflow is capable of determining moulding problems such as quality, processability and in-service needs that are not met. These problems may be caused by improper mould design or poor material selection.

CHAPTER SIX

CONCLUSIONS

Mould design software has redefined what is possible in the injection moulding process. Moldflow has proven that it can successfully simulate the mould and identify the expected results. These results can then be used to determine the problems that could be expected in a particular mould. Further work that could be carried out in mould design and the effect of software simulation is presented here.

6.1 Summary

Without doubt, computers have changed the way we operate. Many software packages change time consuming tasks to tasks that require a lot shorter period and fewer resources. One of the best examples of this change is in the area of design. Many computer software packages have been developed to assist the designer to produce a design faster and better than before. They include Computer Aided Design (CAD), Computer Aided Manufacturing (CAM) and, more recently, Computer Aided Mould Design (CAMD), specifically for the plastics injection moulding industry. These software products have brought about a new awareness of the ability to create new and seemingly futuristic product designs which may now not only be designed, but manufactured also.

The injection moulding of plastics is by no means outside this computer age. Software programs are now available that transform the whole attitude to plastic product design. Mould designers and manufacturers are now able to change designs and observe the results of the changes without ever cutting into a block of steel. These Computer Aided Mould Design software systems are finite element analysis simulation programs for plastics. By simulating the flow of plastic into a mould, a designer can know what will happen with a particular design and a specified material.

Improvements in designs and creative new products have emerged from the injection moulding realm. Some of the credit for these products must go to the software that was used to develop the products. CAMD software is now available for running on PCs, a great step forward. The cost of purchasing the software, training and hardware has come down in the last few years and is now within the grasp of many injection moulding mould design companies. However, the benefits of these programs is still not widely accepted. Many companies believe the cost of the software, and the consulting, outweighs the benefits.

The Production Technology Department at Massey University, is eager to develop expertise in injection moulding technology and introduce it to students and industry. Moldflow Pty. Ltd. supplied the Department with their sophisticated software for the purpose of training and research. To develop expertise in using Moldflow, a project was carried out that explored the abilities of one of these software packages. Several products that did not operate in the injection moulding machine as expected were examined and simulated using the software The problems the software identified were compared with what the moulding company was experiencing for that particular product.

The collection of data, simulation and comparison were carried out following the methodology as outlined in chapter four. Four products were chosen for the study, two manufactured at the Production Technology Department and two manufactured by Sunbeam Ltd. To avoid any preconceived knowledge of the moulding problems, the simulation was performed before the problems were discussed. With the two products manufactured in the Production Technology Department this was difficult because no drawings were available and the model had to be created from a sample product.

The results of the simulations compared favourably with the actual moulding occurrences. Several of the products suffered from warpage. Warpage is a big problem in the injection moulding of products. Warpage is caused by improper gate positioning, poor mould design and poor cooling design. The warpage encountered in two of the four products was caused by poor mould design. The flow of the polymer caused regions of over and under packing. This differential packing caused the products to warp. The third product also suffered from warpage, however, it was due to the location of the gate. The result was the same, plastic over packed at one end and under packed at the other, causing the product to bend.

Moldflow successfully detected the major air trap in the storage box. A race track effect, combined with a very thin wall section, resulted in a large air trap in the corner of the box. This air trap could be seen in the plot of air traps and weld lines. The time to fill plot showed the race track effect and how the plastic filled the sections around the hole.

One problem found in the knob cannot be identified by Moldflow software. Jetting and splash marks must be designed out by the mould designer by using tabs to gate into or by using a slow injection speed to begin with and then a fast speed once the flow front has formed.

Moldflow is a very useful tool for aiding in the development of moulds. The word "aid" must be stressed since the designer still requires knowledge of basic design principles. One such principle is that outlined above regarding jetting and splash marks. Moldflow, or any other CAMD package, does not make the designer an expert, rather, it assists the designer to make sensible decisions regarding the design of the product, gate and runner system and cooling. Designs must still be created regarding the product specifications. These decisions must be based on real data and knowledge, not only a computer print out of the best flow.

Computer Aided Mould Design is slowly beginning to impact the whole domain of mould design so that plastic is no longer thought of as "cheap and nasty" and mould design is no longer merely an art.

6.2 Further Work

This thesis is a starting point for several other studies of injection moulding mould design. The database of products is a small sample. Other products could be analysed to add to the results and further the study of the software. These products should involve other problems encountered in the industry which might be detected by Moldflow software.

Other software packages are currently being produced which analyse several processes related to injection moulding. An example is gas assisted injection moulding. Currently, the process is not used in New Zealand, however, several companies are exploring the possibility. The software for this process is available from several of the larger CAMD companies, including Moldflow. A project that compared the results of a CAMD simulation with the actual manufactured product results would provide useful conclusions regarding the software and the gas injection moulding process. A project of this nature would further the industry relevant research in New Zealand.

Warpage is a major problem in injection moulded products, especially flat products. Warpage software has been developed for the PC and can be used to determine the extent of warping in a particular product. Further work could be done to explore the realms of the warp software and how the simulation results compare with the manufactured product.

Research could also be carried out to determine the causes of jetting and how it could be solved using software simulation. Detection is difficult before the mould is manufactured. A possible joint project between Moldflow and the Production Technology Department could be carried out to study the effects of jetting and how it can be avoided.

APPENDIX A1

MOULD DESIGN PROCEDURE

Procedures are required in many areas of manufacturing. Design is one of them. A good design procedure will enhance the final product and problems encountered can be recorded and referred to during the next design. A simple but effective design procedure would consist of steps similar to those presented in this appendix.

A typical design procedure would consist of the following sections.

1 Project Plan
2 Define End Use Requirements
3 Preliminary Concept Sketch
4 Material Selection
5 Part and Mould Design
6 Modify Part For Manufacturing
7 Prototyping
8 Production Tooling
9 Manufacturing

A1.1 Project Plan

The project plan includes all the areas of the project that must be considered to give an informed insight into the required detail and importance of the project. For example, if the market is very small and only a small volume of the product will be required, it will influence later decisions regarding the manufacturing method. A project may be required in several weeks, whereas another in several months. All these factors must be included in the project plan so that the schedule and procedure of the project may be known. Some areas that must be kept in mind include ensuring the real need has been defined, establishing the main function, and an analysis of previous products, their shortcomings and good features.

Care must be taken to develop the overall plan of the project, rather than the nuts and bolts at this stage.

Areas that need to be addressed include:

- Project identification
- Customers
- Projected cost and volume
- Market and competition
- Appoint project leader and team members
- Organise time-line/GANTT chart

A1.2 Define End Use Requirements

The end-use requirements include the expected physical loading on the product, environmental conditions, dimensional requirements, proposed method of manufacture, marketing constraints, regulations and standards of compliance. Under each of these headings are areas that must be explored to ensure all the requirements are met.

Physical loading requirements

- type of loading - tension, compression, flexural, point, area
- frequency of loading
- duration of the load
- average and worst case scenarios
- modes of failure (Failure Mode Effects and Criticality Analysis)

Environmental conditions

- indoor versus outdoor use
- UV resistance requirements
- resistance to chemicals
- operating temperatures
- ergonomic concerns

Dimensional requirements

- critical dimensions, surface finishes, product flatness
- tolerances, radii, tapers

Regulations and standard compliance

- testing requirements
- safety factor
- FDA regulations

Marketing constraints

- service life
- maximum production cost
- shape, size, colour
- disposal, recycling

Proposed method of manufacture

A1.3 Preliminary Concept Sketch

A three dimensional conceptual sketch of the product to be moulded is a good starting point and removes any ambiguous design expectations. This sketch should be used to highlight any areas requiring detail or regions which cause concern. Functional surfaces that cannot be changed should also be highlighted. This would include surfaces that connect to another product or surface, or regions that perform a task. As an example, consider a nut cracker. The functional surfaces are the areas that contact the nut and the hinge region.

A1.4 Material Selection

The material selection determines the materials which would suit the requirements. Usually three to six materials should be chosen at this stage. The polymer materials suppliers can provide useful information about polymers and have expertise in this field which should be utilised.

From the three to six materials chosen, a rating table may be set up to determine the best overall material to use. A variable rating system may be used so that the most important factors have a higher rating. As an example, refer to table A1.1. Based on part geometry, an arbitrary rating system (scale of 1 - 10) may be determined and used to compare the desirable properties, grading them according to their importance. For example, the properties, processability, creep resistance, chemical resistance, cost per part and flammability resistance may be important. An average rating for each is calculated.

PROPERTY		Material 1		Material 2		Material 3	
	Multiplier	Rating	Value	Rating	Value	Rating	Value
Processability	1.0	5	5.0	9	9.0	7	7.0
Creep resistance	0.8	7	5.6	8	6.4	9	7.2
Chemical resistance	0.8	4	3.2	4	32	9	7.2
Flammability resist.	1.0	7	7.0	6	6.0	6	6.0
$/part	0.5	9	4.5	7	3.5	10	5.0
Average			**6.2**		**6.9**		**7.9**

Table A1.1 Material Selection Rating

From the table, the most suitable material for the part may be chosen, in this case material 3, based on an unbiased approach. A materials company should be consulted as they have some expertise in material selection.

A1.5 Part Design

The part design should consider all the requirements and any material boundaries, for example, the material chosen will set the required thickness of the product for a given compressive strength. From these requirements a basic design may be established using design techniques such as form design, function/means tree and structure modeling.

Once the basic design has been modelled, a flow analysis may be carried out using computer aided mould design software. The flow analysis considers the material properties and flow characteristics which will affect the design of the product, especially the thicknesses of walls. Further analysis may be carried out to determine the best cooling as well as analysis to reduce possible warping.

A1.6 Modify Part for Manufacturing

At this stage the product may have hard corners and zero draft angles. The correct radii and draft angles should now be included in the design. A tooling engineer is important in this stage of the design process. Depending on the design system used, a design may not be complete when it is sent to the tooling engineer. Concurrent engineering involves designing the tool right up to the point when the tool is being cut. This method of manufacturing allows last minute changes while speeding up the design process.

A1.7 Prototyping

Prototyping may be carried out in a number of ways. There are now rapid prototyping tools that can produce the prototype rapidly from a 3 dimensional CAD image. This process can be costly, but may be worth while for an important project. Another method is to produce a mould that may be modified later. The flow analysis should give results that show where to expect weld lines and air traps that may need to be vented. A prototype allows the checking of these and other possible faults, such as sink and jetting marks. The prototype tool should include all the major features that the production tool will require, but may leave out the ejection system and finishing operations.

A1.8 Production Tooling

If a prototype tool was made, there may be some small changes necessary to bring the tool up to production requirements. This would include any changes found necessary during the prototyping, as well as hardening and any other surface finishing and putting in the ejection system.

If the prototype was made by the rapid prototyping methods, the production tool will need to be manufactured.

A1.9 Manufacturing

At this stage the tool may be handed over to manufacturing, although this is not the final stage of the design. Ongoing design changes may still be made to improve the product and the manufacturing of it.

APPENDIX A2

COMPUTER AIDED MOULD DESIGN SOFTWARE

Since its inception, computer aided mould design software has become a very useful tool. A number of companies produce software packages for the mould design market. Most of the packages are based on similar concepts and principles. The cost of the software packages and the hardware required to run the software has come down in price and is now within the reach of most mould manufacturers.

A2.1 Moldflow Pty. Ltd. - Moldflow

Moldflow Pty. Ltd. is one of the leading CAMD software companies and has been servicing the injection moulding industry since 1978. It has been a pioneer in the computer aided mould design technology and continues to provide new software to assist the injection moulding company. One of the newest achievements is the Intelligent Process Control, software that analyses the output from the mould and intelligently makes required changes to improve the performance of the injection moulding machine. IPC is closed loop control for the injection moulding machine.

The simulation software uses two different algorithms. Multi laminate is a more accurate model of the flow, however, it is also more time consuming. The fast filling analysis uses a simplified model of the flow front and hence a simulation of the model is a lot faster.

Moldflow products include:

MF/Flow	Flow analysis software.
MF/Cool	Cooling analysis software
MF/Shrink	Shrinkage prediction software
MF/Warp	Warpage prediction software
MF/View	Pre/post processor model and results viewer
MF/Stress	Predicts stress in loaded parts.
MF/Gas	Flow analysis for gas injection moulding
MF/Tsets	Flow analysis for thermoset materials

A limited version of the MF/Flow software is also available for companies who wish to use CAMD software and want to find out the benefits to the company without spending a lot of money at first. MF/Flow-E is a simplified version which does not use the Multi-Laminate algorithm and has a limit of 10000 nodes. MF/Flow-E also does not have facilities for a packing analysis simulation.

The intelligent process control has two modules:

MF/Optim	optimises the machine set up parameters
MF/Smartmold	monitors and controls the injection moulding machine

Moldflow provides six services:

- CAE Technology
- Material Testing
- Intelligent Process Control
- Research
- Consulting Services
- Training and Support

CAE technology develops the software required to analyse mould designs. Continuous improvements to both the operation and user friendly environments are ongoing. Moldflow were the first to release many of the software modules for mould design.

Moldflow's material testing laboratory tests materials for the material database which contains data on several thousands of polymer grades. Specific materials for a particular product are also tested on request.

Intelligent process control (IPC) is one of the latest developments to come from Moldflow. IPC allows closed loop control of an injection moulding machine where the software evaluates and makes the necessary changes to the injection moulding machine parameters when the process requires alterations. Using IPC the parameters set from the flow analysis can be monitored and controlled in the injection moulding machine.

Research is carried out at Moldflow in Australia, as well as in connection with universities and industrial researchers to provide up to date developments.

For small companies who cannot afford to buy and maintain the software, hardware and personnel required to run Moldflow software, Moldflow offer a consulting service. The consulting can be used at any stage from "art to part".

Moldflow provide training in all the software they develop and support the software with help at any stage. The support also includes upgrades of the software as well as technical assistance with the software.

Moldflow have many years of experience in the injection moulding industry. The software has been used in many different companies to produce successful mould designs.

A2.2 Plastics & Computer - TMconcept

Plastics & Computer are one of the top four computer aided mould design software companies. Their product, TMconcept, is based on 'a vigorous application of fundamental principles'. There are seven modules in the TMconcept product, all linked as described in figure A2.1.

MS is the material selection module. It is required for any analysis.

MCO is the Mouldability and Cost Optimisation module for finding the optimum moulding conditions.

FA is the Flow-warpage Analysis module. FA is one of the main analysis modules. It includes five types of analyses: fill, hold, cool, shrink and warp. The fill analysis sets moulding parameters using fuzzy set theory. Recommendations for moulding condition changes, based on the input data, are given a rating. The higher the rating, the greater the expected impact on the finished product.

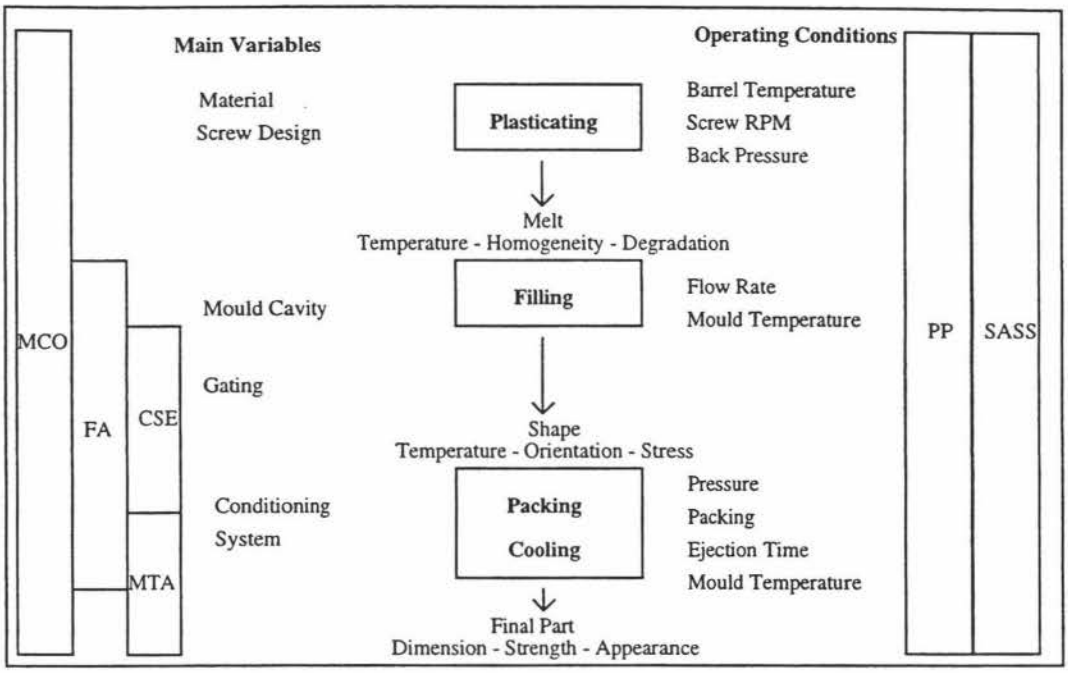
CSE is the Computerised Shrinkage Evaluation module, used to dimension the cavity to accommodate the expected shrinkage. This module would be used where small tolerances were required.

MTA provides information on the Mould Thermal Analysis. The cooling layout is designed using MTA.

PP is the post processor where all of the analysis results can be changed to machine settings.

SASS is the Situation Analysis and Solution System. It allows trouble shooting moulding problems on the shop floor so that machine setting changes can be made by the operator or machine setter.

Not all modules are required for an analysis. For example, the design of the part only requires Material Selection software, Mouldability analysis and Cost Optimisation software, and Flow-warpage Analysis software.



TMConcept System Overview Document

Figure A2.1 TMConcept Software Requirements

A2.3 SDRC - I-DEAS Master Series

SDRC have produced a range of products for CAE/CAD/CAM which operate together as integral software system. The software is available as packages and may be used in all areas of engineering. The modules specifically designed for plastics processing link up with the basic design and finite element analysis software packages.

There are four packages the mould designer can choose from:

1. Mold Analysis Package.
- I-DEAS Mold Filling
- I-DEAS Mold Cooling
- I-DEAS Material Data System
- I-DEAS Material Data Catalog - Plastics

2. Mold Filling Package
- I-DEAS Master Modeller
- I-DEAS Finite Element Modeling
- I-DEAS Mold Filling
- I-DEAS Material Data System
- I-DEAS Material Data Catalog - Plastics

3. I-DEAS Plastics Processing Package

- I-DEAS Master Modeller
- I-DEAS Finite Element Modeling
- I-DEAS Mold Filling
- I-DEAS Mold Cooling
- I-DEAS Warp and Shrink
- I-DEAS Material Data System
- I-DEAS Material Data Catalog - Plastics

4. I-DEAS Plastics Designer Package

- I-DEAS Master Modeller
- I-DEAS Finite Element Modeling
- I-DEAS Material Data System
- I-DEAS Material Data Catalog - Plastics
- I-DEAS Weld Locater

An advantage of a system where the design is carried out in a modeling package is the time saved in translating the model or even drawing it from scratch. Complex shapes can be produced and an unlimited model size allows any configuration to be modelled. The filling analysis is based on "tested and reliable numerical methods".

The system also allows the model, once it has been used for analysis, to be used as the basis of the design of the mould without the need to create another model of the cavity.

A2.4 AC Technology - C-Mold

"CAE Solutions For Plastics"

AC Technology have been producing CAMD software for a number of years and have an academic link with the Cornell University Injection Molding Program. The process simulations and software are based on research done at Cornell University. This link with the academic world has provided some of the necessary research to validate the software.

C-Mold have divided the modules into two distinct groups, one for interaction software and one for analysis software. The interactive software includes the design and viewing modules and the interfacing software, used to translate models from CAD to C-Mold models. The analysis software includes the flow, cool and warping analysis packages as well as modules for analysing gas injection moulded products and reaction injection moulded products.

The software is licensed in three different solution packages, depending on the needs, and finances, of the user.

1. **Process Solution** includes software to analyse the mould and optimise the design and the processing conditions. The software produces results which can be used to:

- position gates
- identify air traps
- prevent short shots
- identify weld line locations
- set the optimum ram speed profile
- optimise the runner sizes
- position vents
- estimate the fill time
- balance flows
- minimise the injection pressure

2. **Productivity Solution** adds the cooling analysis software to the process solution software. Using the cooling software, the following may be performed:

- predict the transient mould temperature
- identify hot spots
- define the refrigeration requirements
- select the appropriate machine
- predict the polymer melt temperature
- optimise the cooling channel layout
- minimise the cycle time

3. **Performance Solution** includes two more of the software modules used for analysing the packing and warping of the injection moulded product. The pack and warp software allow several additional benefits above the productivity solution software:

- improve the appearance
- predict shrinkage
- minimise the residual stresses
- predict part strength
- achieve tolerance requirements
- reduce the hesitation
- minimise the shrinkage
- identify the mechanisms causing warpage
- improve dimensional conformity

C-Design is the software module in which the polymer material and moulding machine may be chosen. The software contains databases of over 8,500 polymers and 2,500 moulding machines. Data may be displayed on any of the polymers, 300 of which include PVT data.

Two specialist software products are also produced by C-Mold. C-Gasflow is used for determining the design of a mould that is manufactured using gas assisted injection moulding. The second, C-Set, is used for reactive moulding processes that involve thermoset materials.

The software is based on finite element analysis. The model can be changed and analyses performed on alternative designs may be compared.

A2.5 IKV - CADMould

IKV have created the CADMould software package for injection moulding mould design assistance. The software has been produced as a set of modules, once again.

Modules.

1. **Mefisto.** Five versions of Mefisto have been produced to date. The first three are the basic modules and use simple algorithms to determine the flow of plastic in the mould. The fourth and fifth versions incorporate a more rigorous algorithm to establish rheological data. The IKV material database is included in all but the first version. The latter versions provide more information about the flow of the polymer and may also give more accurate data, especially on complicated moulds with several different surface thicknesses.

2. **Mefibo.** Two versions of Mefibo are available. Mefibo describes the direction of the fibres in the product, based on the flow information from Mefisto. Mefibo require flow data from Mefisto versions 3 onwards.

3. **Mestro-T and Mestro-E** provide information on the "physical material states of temperature, velocity and shear speed along the flow path", presented as profiles over the thickness of the component. The modules give details of the shear stress, percentage of the frozen layer, the degree of cross linking and a scorch index. Mefisto version 2 or later is required.

4. **Mehold** calculates the local volume shrinkage of the part.

5. **Meclamp** calculates the clamping force required to maintain the closure of the mould during injection. The parting plane is defined by the user.

6. **Mewarp** calculates the shrinkage and warpage of the part once it has been moulded. The graphical data can be viewed just before the product is removed from the mould or once the product has cooled to ambient temperature. Versions of Mefisto and Mehold are beneficial.

7. **Metemp** is the software module used to calculate the wall temperatures of the cavity during cooling. Cooling lines are created and the temperature profile of the product may be viewed.

Several other modules are available which assist in the design of gas injection moulded parts, thermoset parts, reaction injection moulded parts and co-injected products.

A2.6 Summary Of Software

Table A2.1 Summarises the Computer Aided Mould Design software presented here.

Supplier	Program	Hardware[1]	Mold filling	Cooling	Problem solving	Price[2]
Moldflow	Moldflow	MF,WS,PC	MF/Flow	MF/Cool	MF/Flow	$80,000
AC Technology	C-Mold	MF,WS,PC	C-Flow	C-Cool	C-Design	$18,000
Plastics & Computer	TM concept	WS,PC	faBest	MTA1	Saff	$30,000
SDRC	I-Deas	WS	I-Deas Mold Fill	I-Deas Mold Cool	I-Deas Results Advisor	$56,000
IKV	CAD-Mould	WS,PC	Mephisto	MFCool	Mephisto	$30,000

1 MF - mainframe computer, WS - work-station, PC - personal computer.
2 Price is only indicative and depends on the software and hardware requirements.

Source: Vallens, A. 1993.

Table A2.1 Summary of CAMD Software.

The list of CAMD software listed in this section is not extensive. Several other companies also produce CAMD products, however inquiries regarding their products produced no results.

APPENDIX A3

GATE DESIGNS

Several different gate designs have been created to suit the many different methods used to inject plastic into the mould. Each of these gates suits a different need and all are valuable. A designer should always consider the gate design and not adhere to only one type .

A3.1 Introduction

There are several types of gates which may be used, each having its own particular purpose. The simplest of the gates is a sprue gate. In this design there is no real gate, rather a straight flow of polymer from the sprue into the mould. This type of gate would be used for products that may be injected in the centre. A second type of gate which is becoming more popular is the submarine gate. Its main attribute is that it shears off on ejection. The product requires no further processing to remove the sprue and runner system.

Other gates include large, restricted, fan and ring. Each of these is described further in the sections that follow.

A3.2 Large

Where a product is large and requires a great deal of polymer, or where a restricted gate gives surface blemishes that are not acceptable, a larger than normal gate is used. Large gates are normally rectangular in shape and are usually greater than 6.3 mm in width. The width range from 3 to 6.3 mm is not used because it leads to adverse affects on the viscosity. The velocity decreases as gate opening increases, and flow actually decreases in this range (Rubin, 1972).

There is less of a chance that the product will suffer from flow marks, internal stresses and surface blemishes when a large gate is used, except at the entry point where a large mark is left when degating. Polymer may flow back into the injection barrel if pressure is removed before the gate freezes. Pressure must be held on the product for a longer time, compared to smaller gates, in order to reduce the possibility of back flow. The gate should be located in a position where the mark left after the sprue has been removed is not seen.

A3.3 Restricted

The faster a polymer moves, the less viscous it becomes since polymers are pseudoplastic, nonNewtonian fluids and the apparent viscosity is a function of the shear rate. The velocity of the polymer increases as it is forced through the gate aperture, at which time the temperature of the polymer increases since some of the kinetic energy is transformed to heat. Once the flow stops, the gate freezes off rapidly, since it is so small. For this reason, if the flow ceases during filling, the gate may freeze off and a short shot will result. A balanced gate system is required for multi cavity restricted gate moulds to avoid gates freezing off before the polymer fills the cavities.

A restricted gate is easily degated because the connection between the part and the runner system is only a narrow gate, less than 3 mm in diameter. The gate is prone to freezing off early, and is therefore only used for relatively small products. Small gate diameters may give rise to jetting effects. Jetting occurs when the pressure on the plastic is large enough and the aperture small enough that the plastic forms a 'spaghetti strand', rather than the normal bubble flow front. Jetting causes surface defects and flow marks. It can also be the cause of internal stresses. The 'spaghetti strand' normally freezes against the mould wall before the flow front approaches it and therefore not only causes flow marks, it also causes stress to build up around it resulting in a weakness in the product.

To avoid jetting, restricted gates should flow into a mould opposite a near mould surface. Any jetting quickly ceases when it strikes a wall, and the bubble flow front will form.

A3.4 Sprue

The sprue gate, as described earlier, has no runner system. Polymer enters the cavity directly from the sprue. It is a form of large gate and is mainly used for products that can be injected in the center. When using this type of gate, the designer must be careful to place it in a position where the mark it leaves when the sprue is removed is acceptable and does not detract from the appearance of the product.

The main advantage of such a gate is the direct, short flow from the injection unit to the cavity. Minimal pressure drop is attainable with the sprue gate, although there is also the possibility of sinking around the gate since it is normally thicker than the product wall thickness at the entry point. The direct feed also leaves no room for a cold slug well to remove plastic which has begun to solidify during travel from the injection unit to the cavity. Occasionally, the well may be incorporated into the product design as a nodule opposite the point where the plastic enters the mould. This nodule will also reduce the stress around the gate and give a better appearance around the gate.

A3.5 Fan

A fan gate is normally used in conditions where a flat product is required and one end may act as the gating area. The gate is normally thinner than the product so that it is easily detached from the moulding in a post-processing procedure.

A fan gate reduces the possibility of warping in a flat product by lessening the effects of overfill and under fill. These two defects may be seen in flat products which fill one area first (overfill) and only just fill the last section (under fill). The width of the fan gate, **a**, is normally ½ to ¾ of the width of the product, **b**, and the gate thickness, **g**, is ¼ to ½ the thickness of the product, **t**.

A flat product is usually difficult to produce because the internal shear stresses in the moulding cause the product to warp. Normally strengthening ribs are used to give the product strength under normal loads and to provide internal strength therefore reducing warp. By ensuring an even filling of the flat product, warp may be reduced to a great deal and even eliminated.

A3.6 Diaphragm and Ring

The diaphragm and ring gates are used for products where the main shapes are hollow tubes. The diaphragm is needed to reduce the possible warpage that can occur if a tube is injected at only one or two points. In order to obtain excellent dimensional accuracy, an even flow down the sides of the tube is important. This is possible with the diaphragm and ring gates (see figure A3.1).

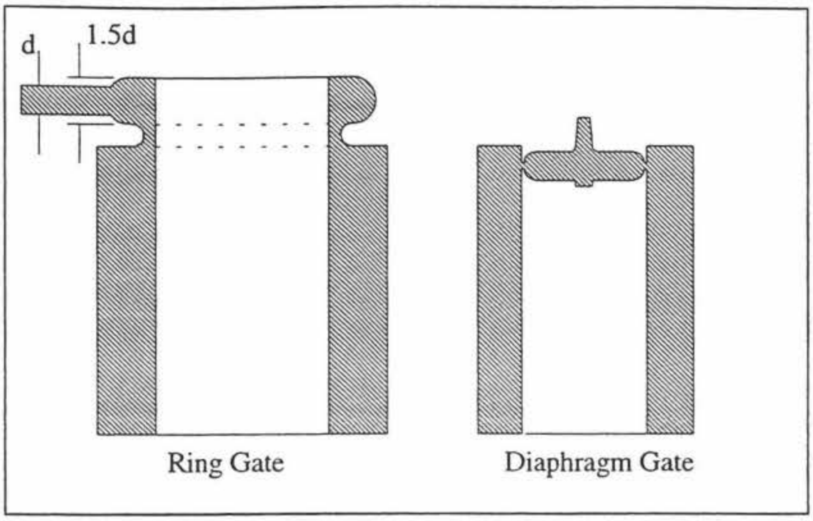


Figure A3.1 Diaphragm and Ring Gates

A3.7 Submarine

For automatic removal of the runner and gate system, a submarine gate may be used. The submarine gate is the most difficult to design and machine, but provides the only method of automatic gate removal in a two plate mould. It must be a small gate and for this reason may be used on only relatively small products. The gate must be kept small so that it may shear off without any plastic remaining in the mould. (See figure A3.2).

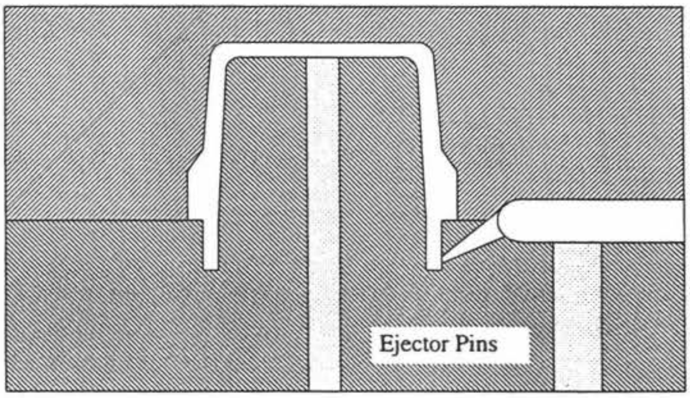


Figure A3.2 Submarine Gate

APPENDIX A4

PROJECT SIMULATIONS AND ANALYSIS

Four products were chosen for simulation and analysis using Computer Aided Mould Design software. The four projects covered a range of problems encountered in the injection moulding mould design industry. The software was used to determine the problems associated with each of the products. The results from the simulation were then compared with the actual problems encountered in the mould.

A4.1 Storage Box - Massey University, Production Technology

This section presents the storage box project results including a product description, the simulation and a comparison of the simulation results and the actual problems.

A4.1.1 Product Description

The whole product included two moulds, one producing the boxes, the other producing a bracket to hold the boxes on a wall and partitions for the centers of the boxes. The box is fairly small, 90 mm by 95 mm at the base, 46 mm in height with a 6 mm lip around the top, 3 mm thick (figure A4.1). The front of the box is graded so that at the top the length of the box is 110 mm. The rear of the box has a lip to hook it onto the wall bracket and the base has four dimples for the box to stand on when

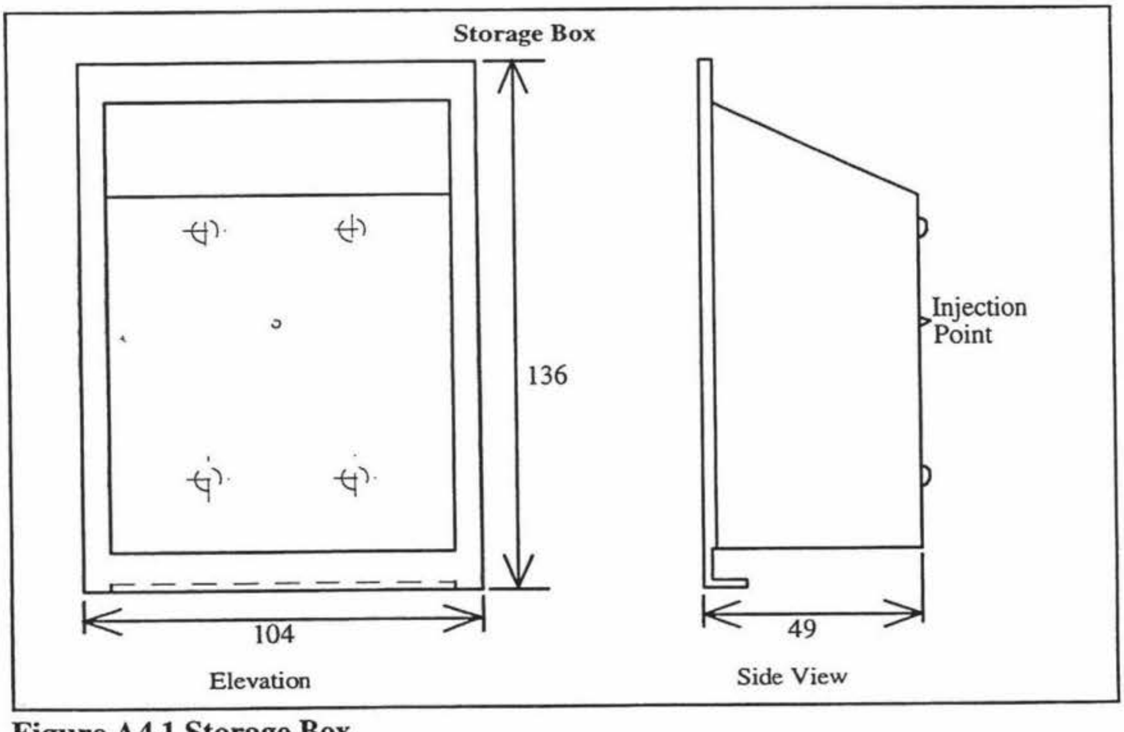


Figure A4.1 Storage Box

placed on a flat surface. Two raised tracks run down the center of the box to provide a slot for the divider. The injection point is in the center of the underside and four ejector pins have been positioned in the four corners of the mould core. A small ridge is also present at the top of the sides to assist in holding the product on the core during removal from the cavity.

A4.1.2 Material Specifications

Polyethylene is a very common polymer. Most products made in high density polyethylene are stamped with the recycle number 2. Recycled polyethylene is widely available through recycling centres or private companies. The material used for the simulation was BASF Lupolen 5031L.

Polyethylene has the following specifications when designing a mould using Moldflow software:

Mould temperature range:	20 to 60°C
Melt temperature range:	180 to 240°C (max. 280°C)
Maximum shear stress:	0.200 MPa
Maximum shear rate:	40,000 1/s

A4.1.3 Flow Paths

The longest flow path was found so that the time required to fill the mould, cooling time, mould pressure and melt temperature could be observed. The flow paths for this product are fairly simple; beginning at the center there are four main paths that can be followed to the corners. Only two of these flow paths needed to be modelled since the product is symmetrical. The flow paths are shown in figure A4.2.

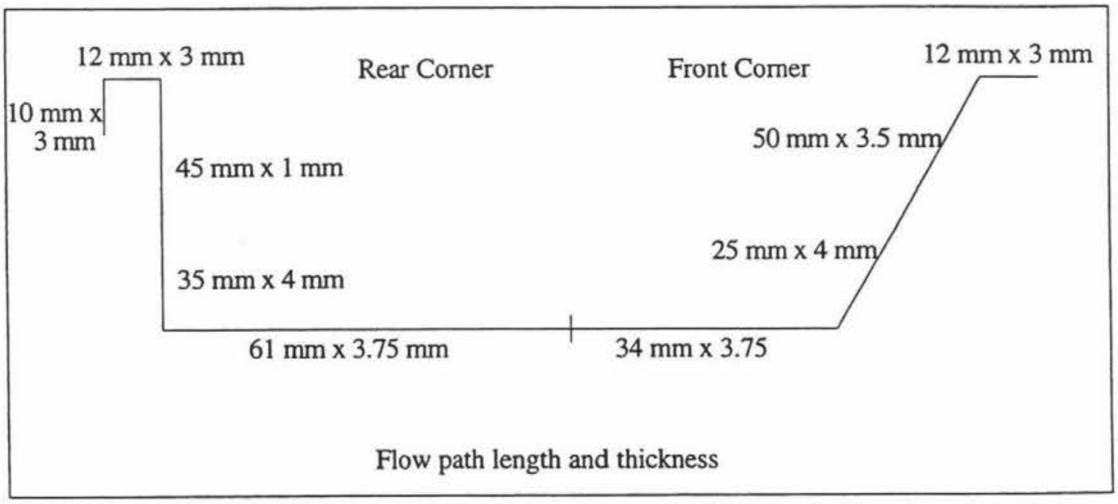


Figure A4.2 Flow Paths

A4.1.4 Results of a 2D Path Analysis

The moulding window was found for the storage box (figure A4.3). The optimum processing melt temperature and injection time are in the centre of the moulding window. For the box, the optimum melt temperature is 210°C and the optimum injection time is 3.5 seconds. The processing point used in the injection moulding machine was outside the expected range however it was still within the mouldable area. It is advisable to change the processing point to the optimum values so that any change in the parameters will not cause a change in the product quality.

A4.1.5 3D Model Simulation Analysis

The model filling simulation was performed using the moulded product after the core had shifted since the core moved during the first shot.

The back of the box has a nominal thickness of 0.8 mm, although it was less near the right corner. The front of the box had a nominal thickness of 3.5 mm and the two sides of the box differed in thickness by 1.6 mm (0.7 mm and 3.3 mm). Figure A4.4 shows the wall thicknesses of the model. The filling pattern for this design showed the front filling at about 2.6 seconds and the back not filling completely to the corners until 4.5 seconds had elapsed. The thin side of the box developed an air trap which could not be vented. This resulted in a weakness at that point and a hole in the side. Figure A4.5 is a contour plot of the fill time and the air traps for the box. The contours are close together near the bottom of the hole because the polymer is flowing very slowly in this region because it is a thin section. The polymer flows through the thicker sections first because there is less resistance. The plastic is unable to fill the side because the temperature of the material has dropped below the no flow temperature of 120°C, hence the hole forms. The square points show the air traps. The air traps of importance are those on the right rear corner.

These cannot be vented and will therefore form a poor weld or a hole. In this case, a hole forms since the plastic is too cold to fill the area and a large amount of air is trapped.

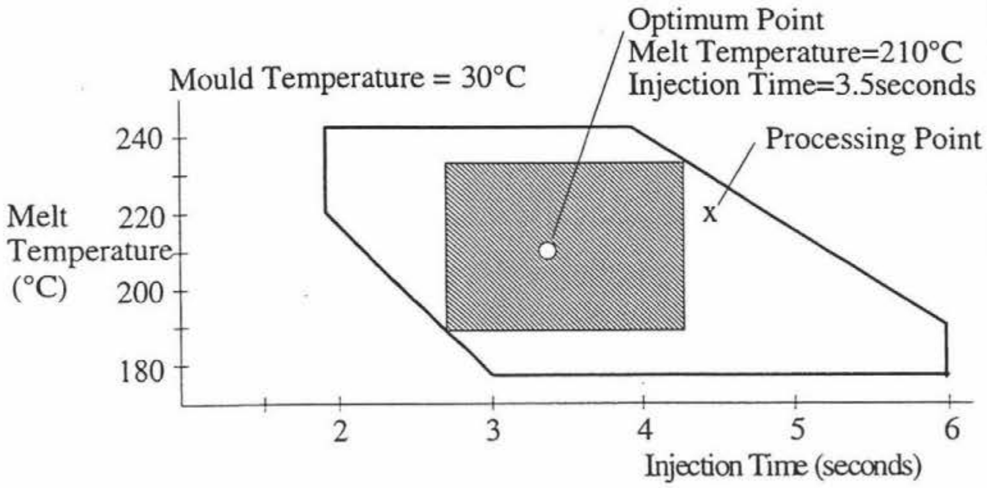


Figure A4.3 Moulding Window

The pressure and temperature contour plots revealed considerable warpage effects. Figure A4.6 shows the pressure plot. The difference in pressure between the front and rear of the box provides an indication that the front of the box is filling first and is packing before the rear of the box is filled. The end of fill temperature plot confirms this conclusion (figure A4.7). The temperature over the front and most of the base of the box is around 220°C; the injection temperature. The right rear corner is well below the no flow temperature, in fact it is below the freeze temperature at the end of the filling.

A4.1.6 Comparison

The mould flow simulation gave accurate results which could be clearly seen in the actual moulding of the product. The 70% fill time plot (figure A4.11) compares favourably with the short shot of the moulding (figure A4.10). The flow of the plastic in the simulation results is consistent with the short shot. The flow lines can be seen in the side wall of the actual box, revealing a similar pattern to the simulation contour plot (figures A4.12 and A4.13).

A4.1.7 Design Changes and Results

The results of the simulation and the actual box filling after the changes had been carried out are shown in figures A4.14 to A4.17.

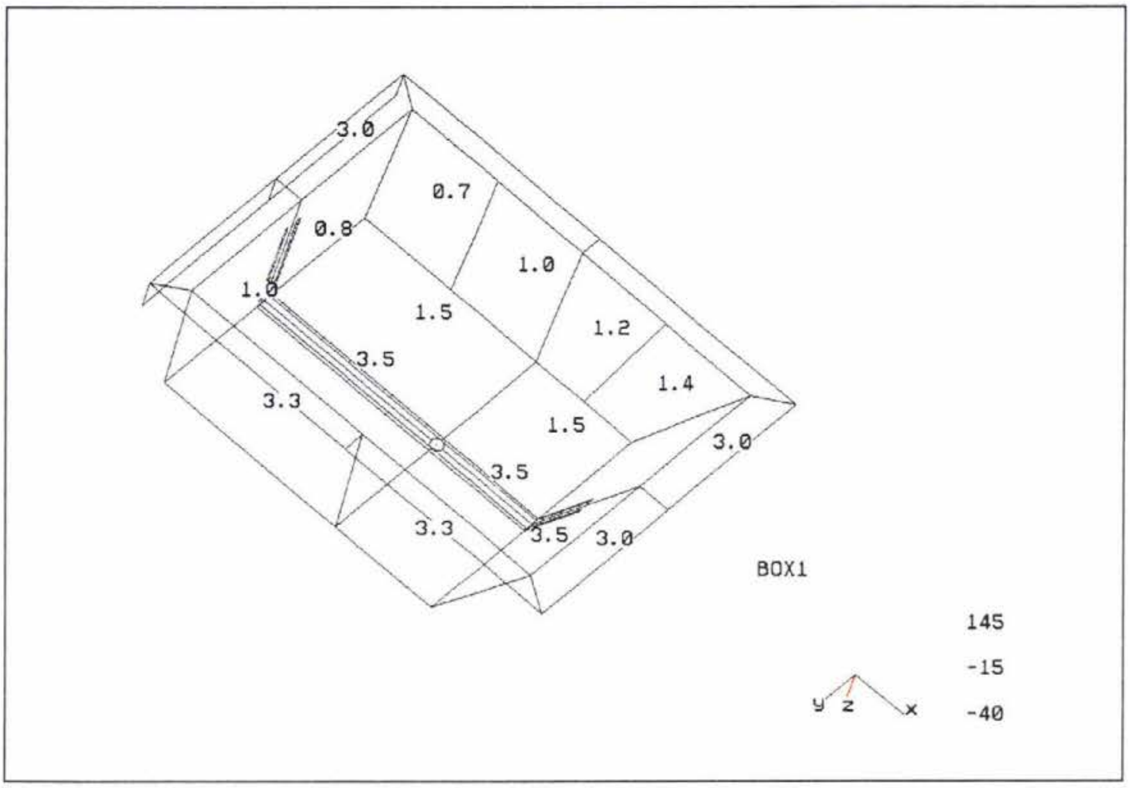


Figure A4.4 Thicknesses Plot (dimensions : millimetres)

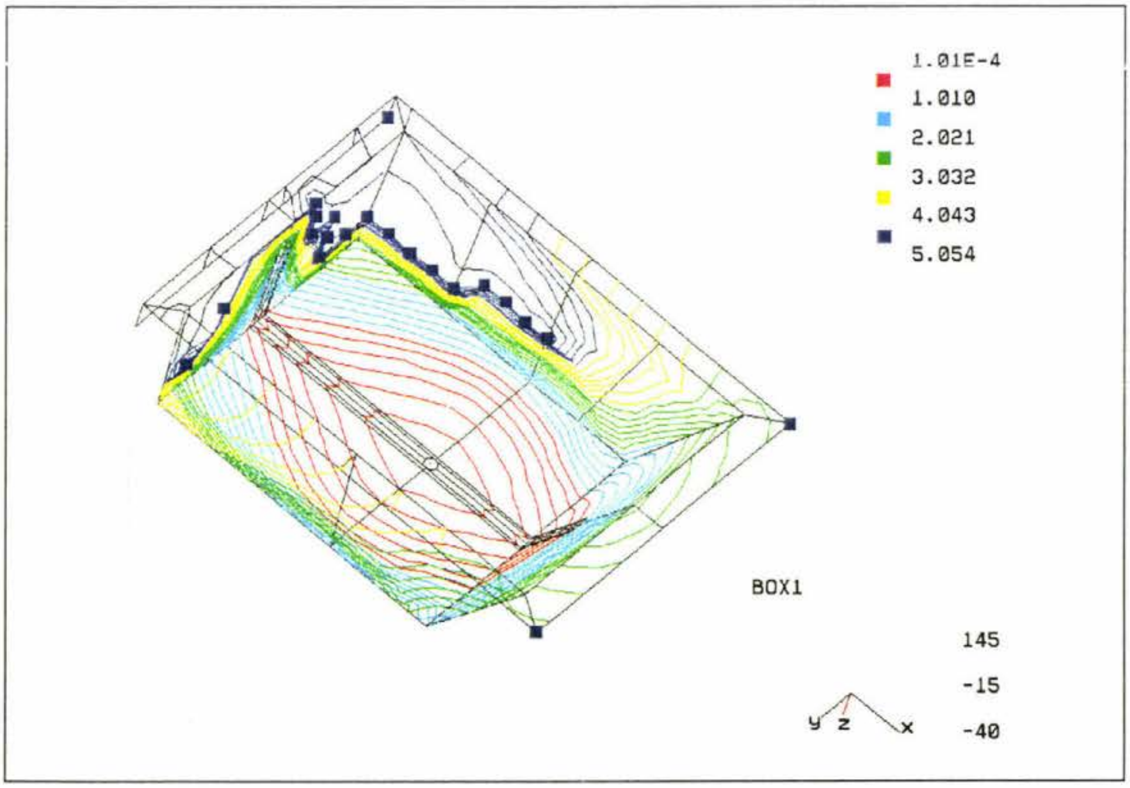


Figure A4.5 Filling and Air Traps Plot (scale : seconds)

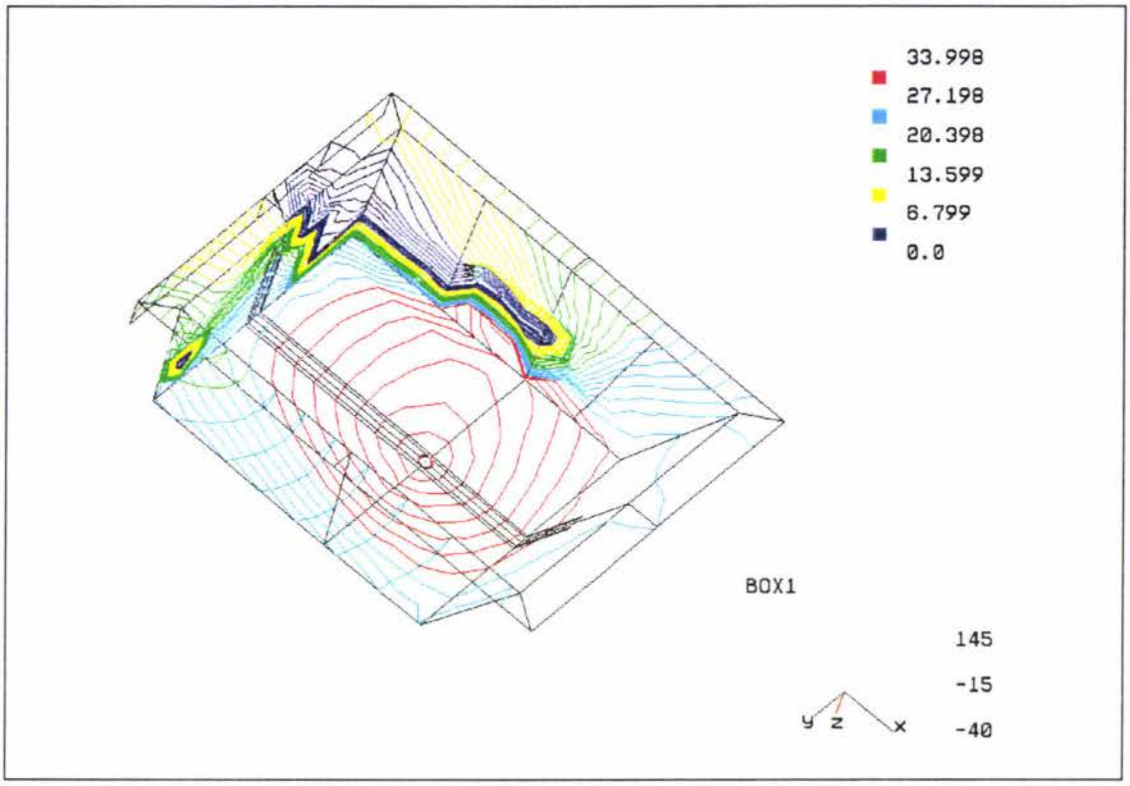


Figure A4.6 End of Fill Pressure Plot (scale : MPa)

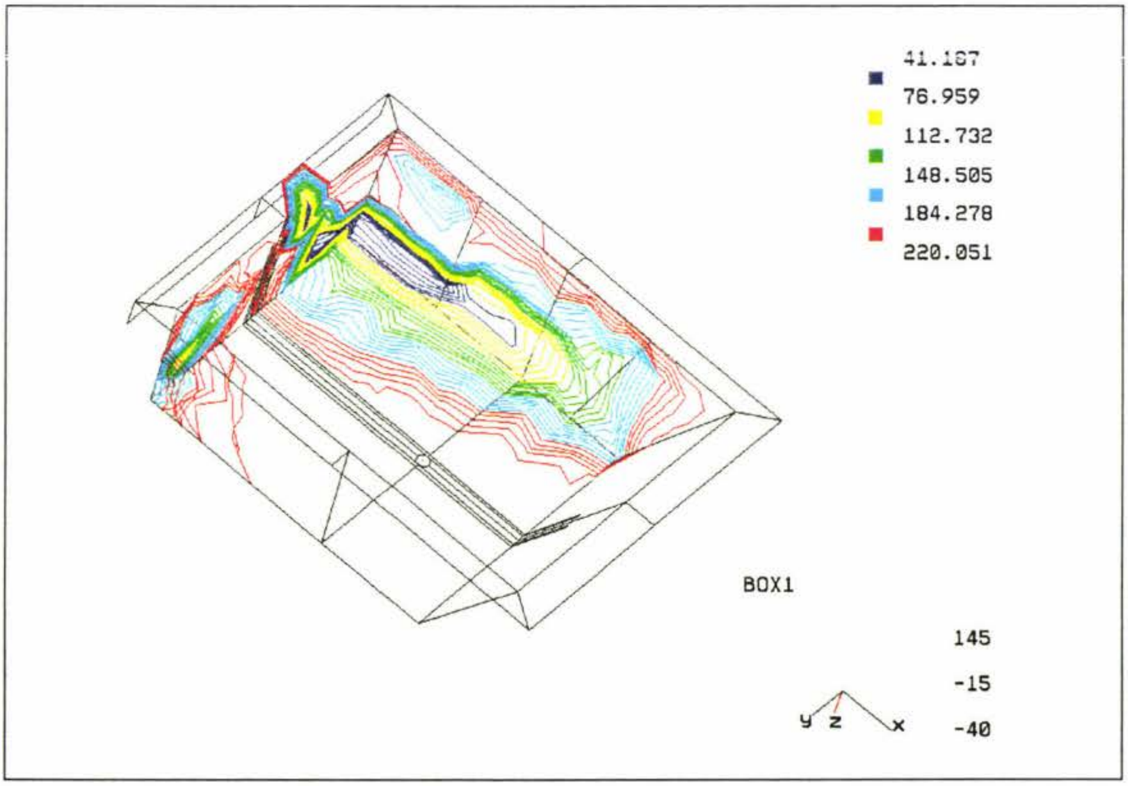


Figure A4.7 End of Fill Temperature Plot (scale : °C)

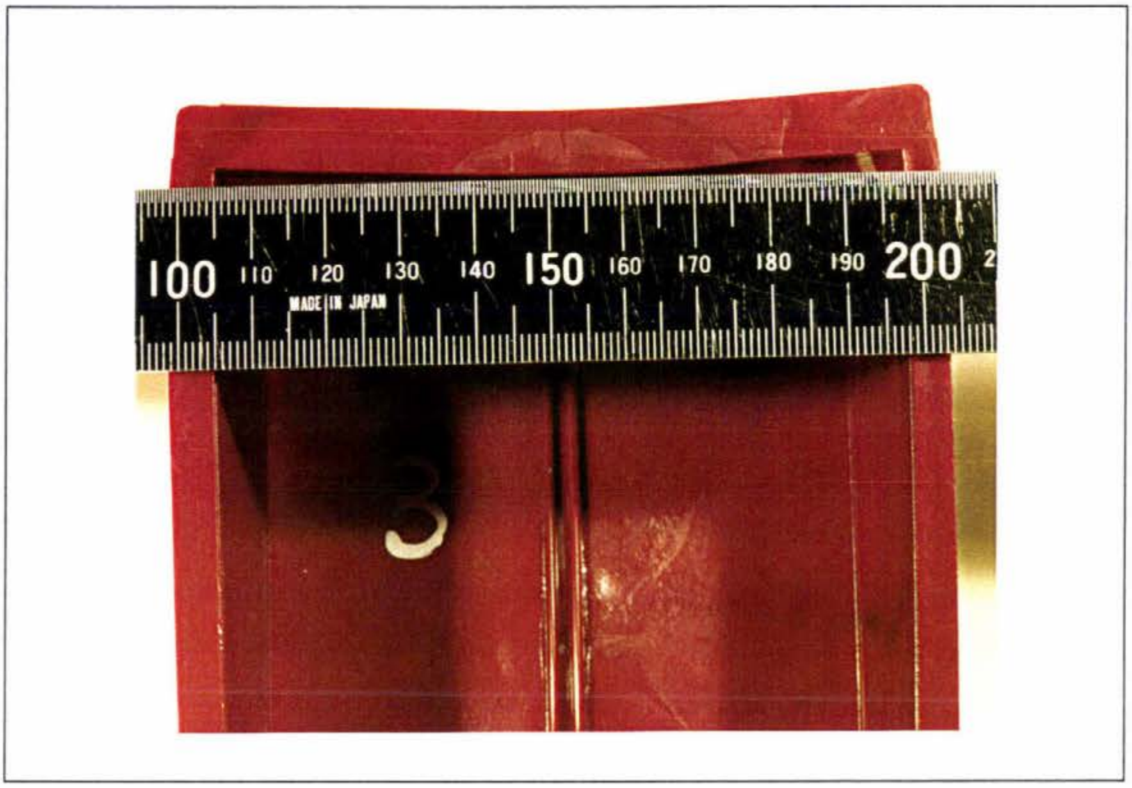


Figure A4.8 Warpage in the Rear Wall of the Product

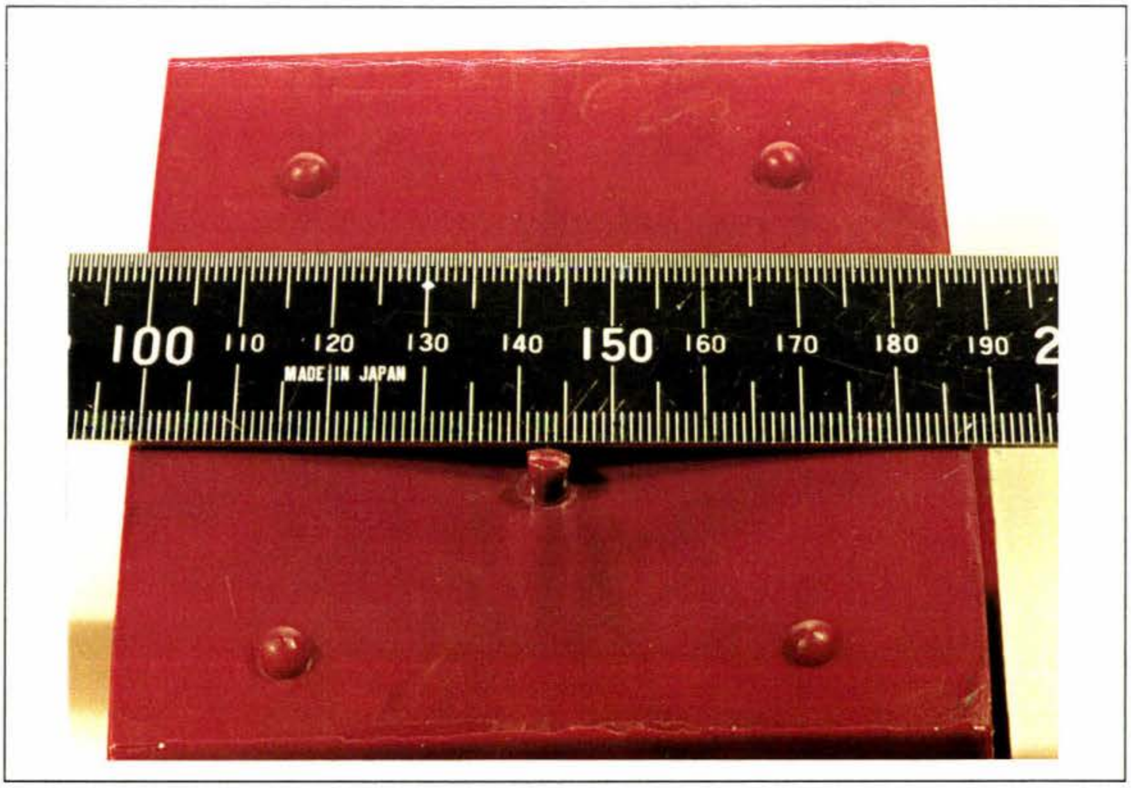


Figure A4.9 Warpage in the Base of the Product

Figure A4.10 Short Shot at 70% Filled

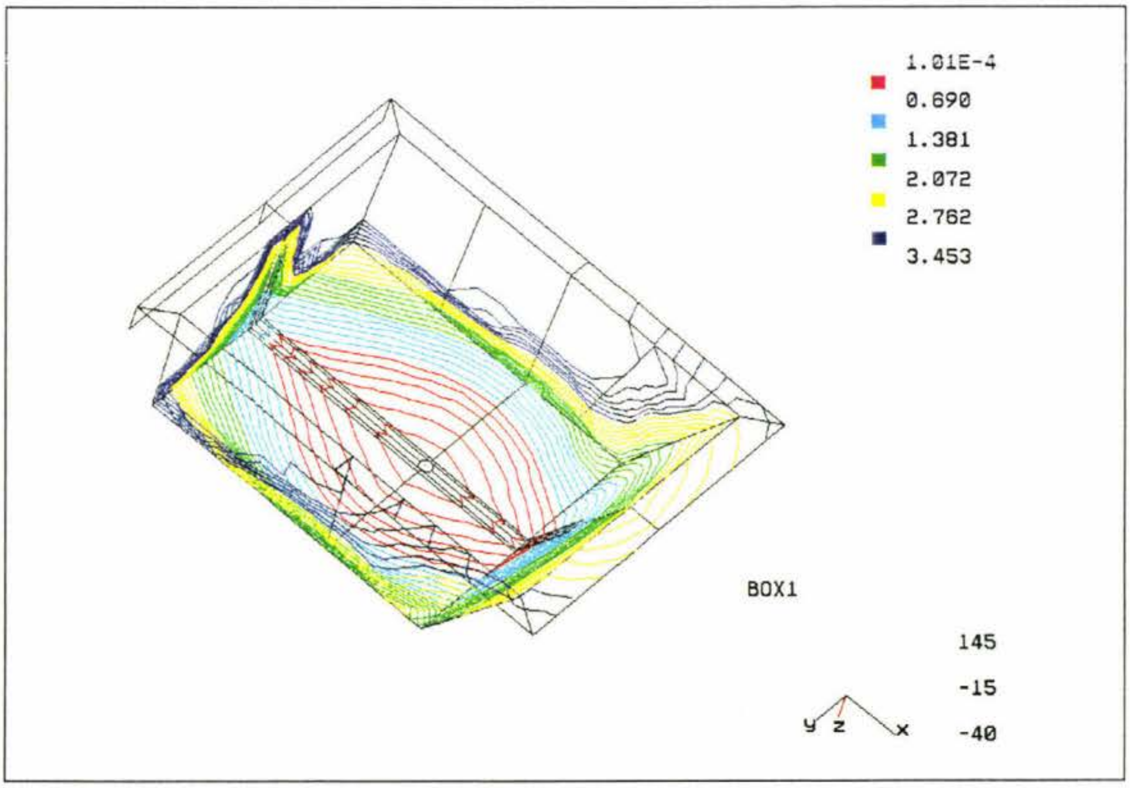


Figure A4.11 Fill Time Contour Plot at 70% Filled (scale : seconds)

Figure A4.12 Visible Flow Marks and Air Trap

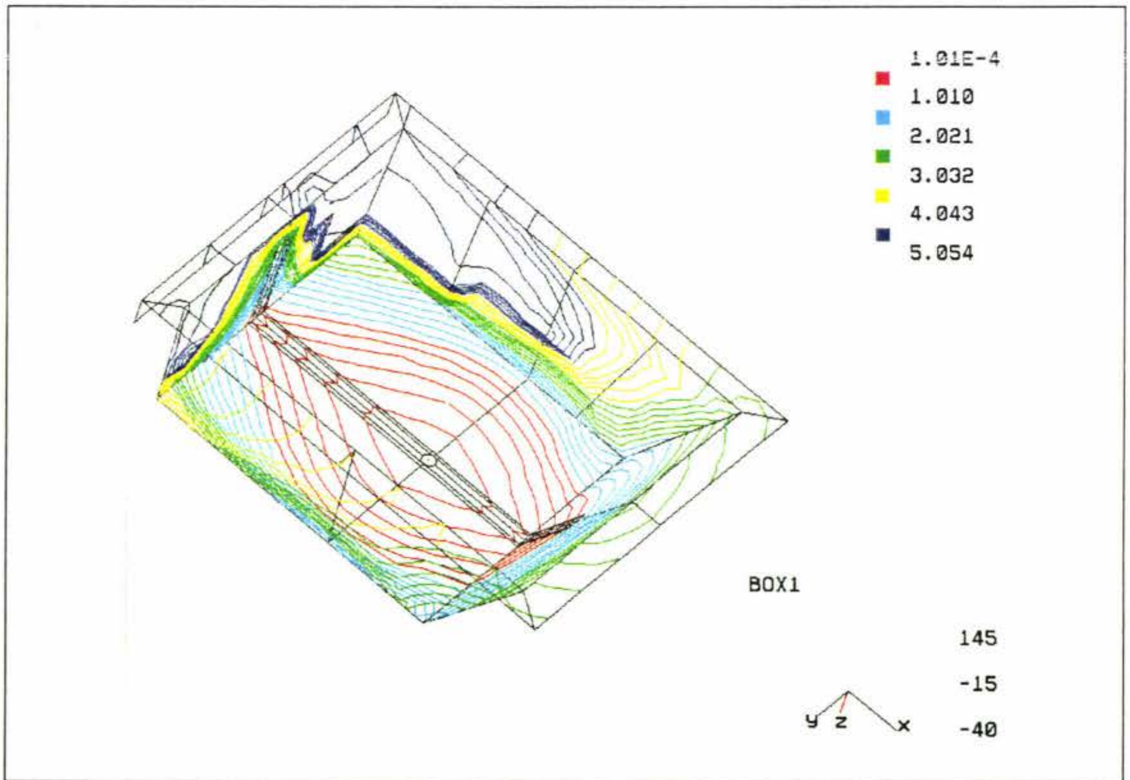


Figure A4.13 Fill Time Contour Plot at 100% Filled (scale : seconds)

Figure A4.14 Short Shot at 70% Filled, After Changes

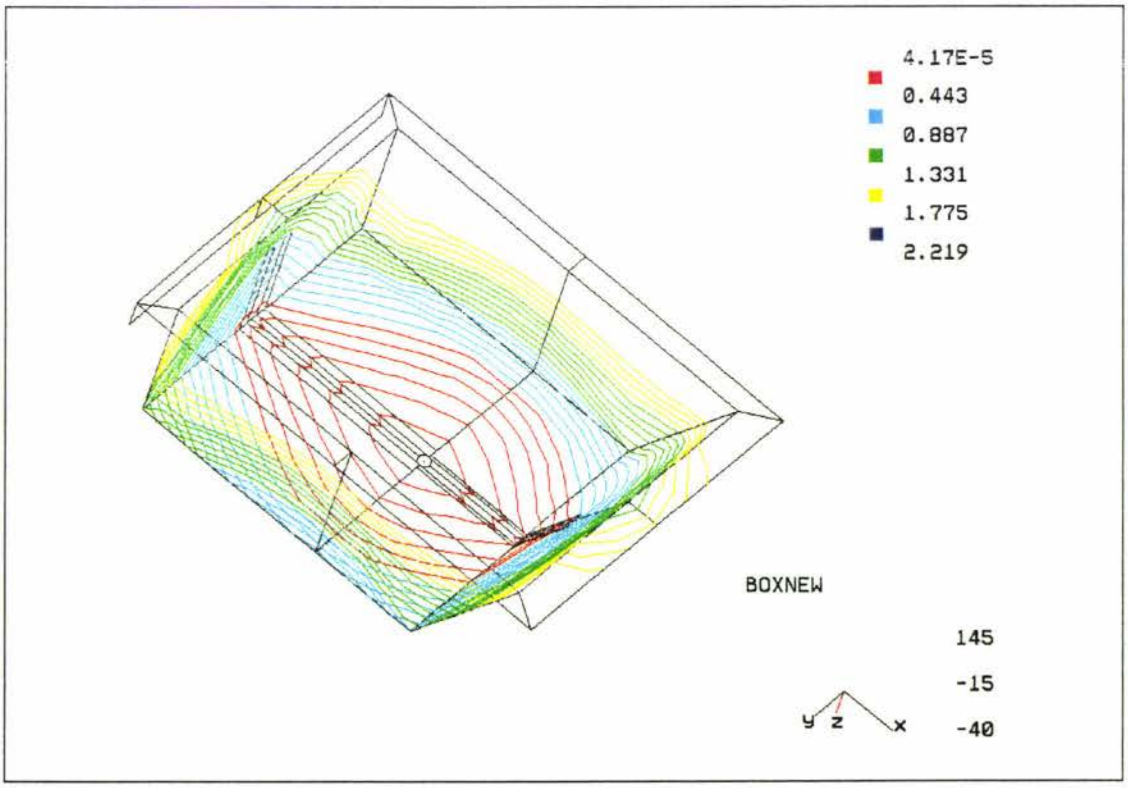


Figure A4.15 Fill Time Plot at 70% Filled, After Changes (scale : seconds)

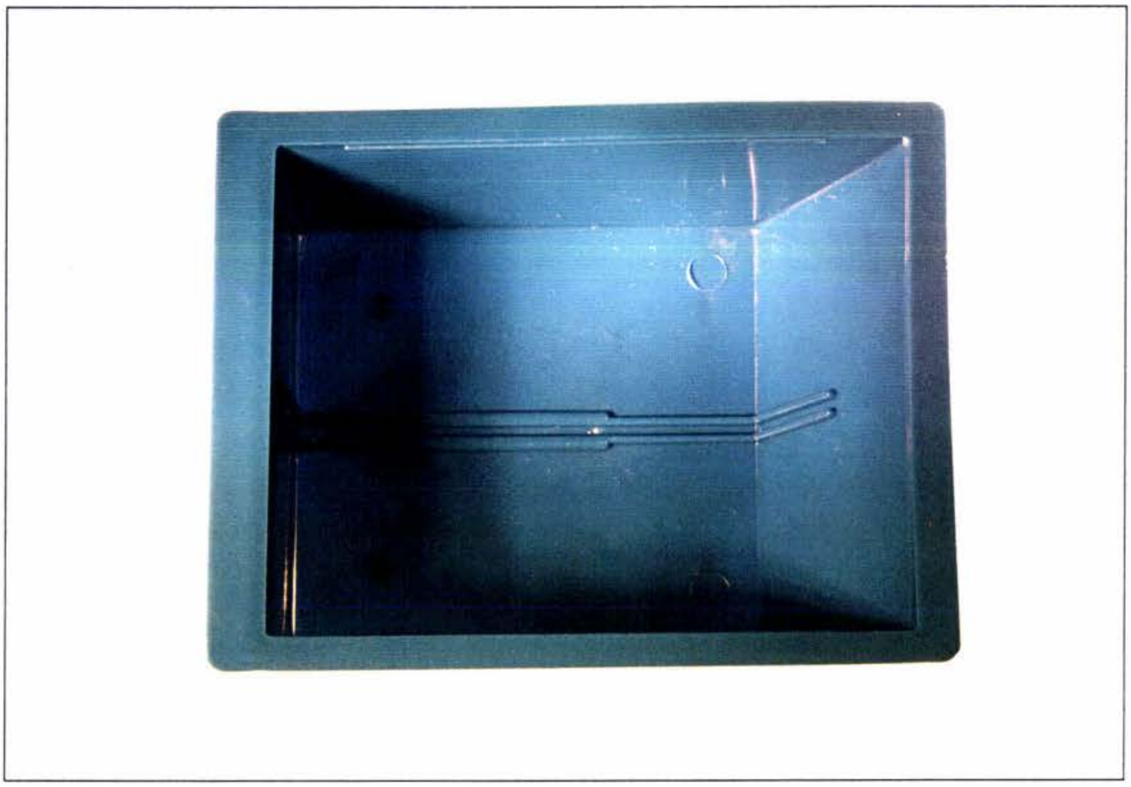


Figure A4.16 Complete Storage Box After Changes

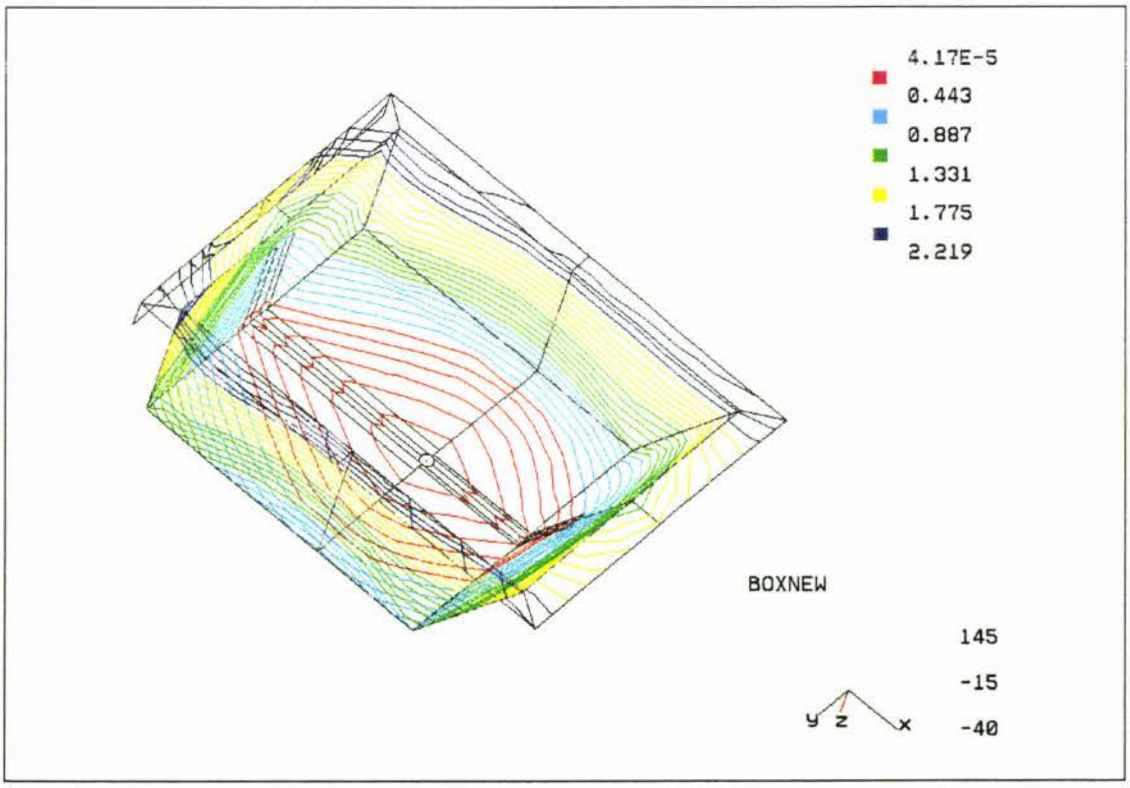


Figure A4.17 Fill Time Contour Plot, After Changes (scale : seconds)

A4.2 Planter Pots - Massey University, Production Technology

This section details the planter pots project results including a product description, the simulation and a comparison of the simulation results and the actual moulding problems.

A4.2.1 Product Description

The planter pot mould was manufactured at Massey University by the Production Technology technicians. It was to be manufactured in recycled polyethylene or polypropylene.

Four pots are connected together with a top plate. Each pot is 37 mm in diameter at the top, narrowing to 20 mm at the base. Each pot consists of twenty ribs down the length of the pot, each with a 2 mm square cross section. A 30 mm strengthening rib runs around the middle of each pot. The total height of the product is 86 mm and the top is 102 mm square (figure A4.18)

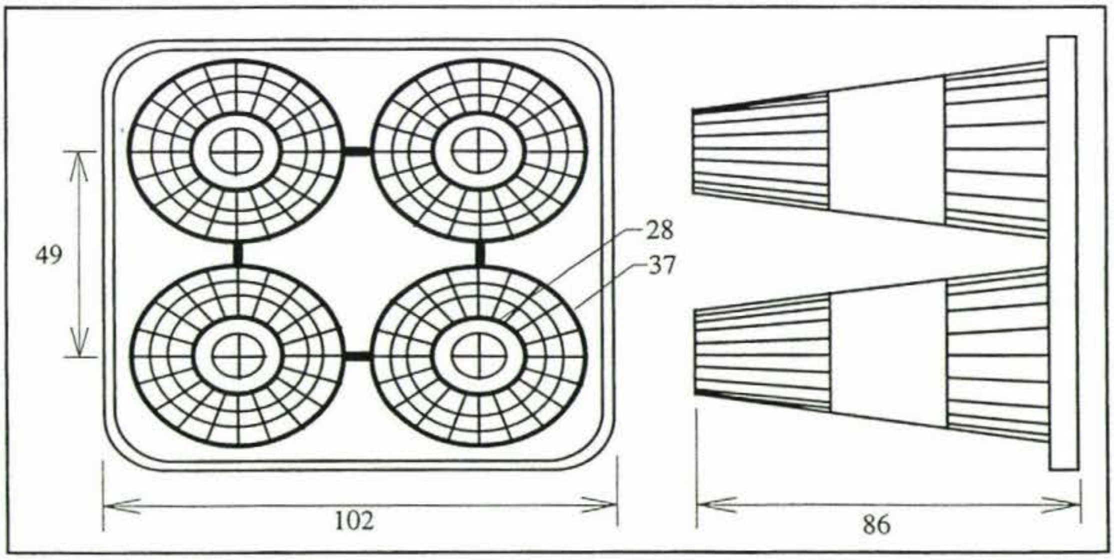


Figure A4.18 Planter Pots

A4.2.2 Material Specifications

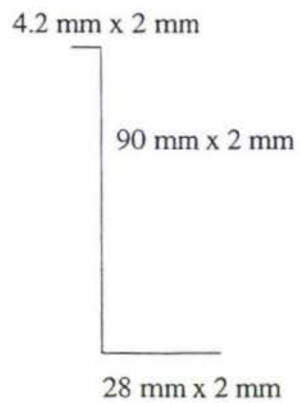
The planter pots were to be manufactured from recycled polyethylene and polypropylene. For the simulation, a polyethylene was used, specifically BASF Lupolen 5031L.

Polyethylene has the following moulding specifications:

Mould temperature range:	20 to 60°C
Melt temperature range:	180 to 240°C (max. 280°C)
Maximum shear stress:	0.200 MPa
Maximum shear rate:	40,000 1/s

A4.2.3 Flow Paths for Planter Pots

The pot has one distinctly dominant flow path, that from the gate to the base of one of the pots (figure A4.19).



Flow path length and thickness

Figure A4.19 Flow Paths

A4.2.4 Results of a 2D Path Analysis

The moulding window for the pots was found for a mould temperature of 40°C. The processing point used for the pots was outside the moulding area in quadrant 2; high melt temperature and long injection time (figure A4.20). This would result in a large end of fill temperature range in the product which, in turn, would result in warping of the product.

The product did show warping, however, the warping was due to more than just the out of control processing point. These factors will be discussed in section A4.2.5.

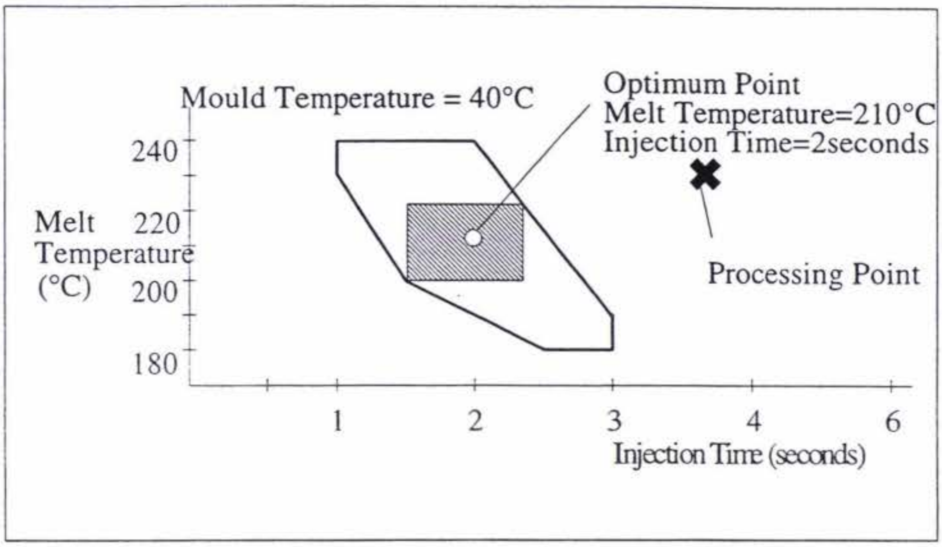


Figure A4.20 Moulding Window for Planter Pots

A4.2.5 3D Model Simulation Analysis

Two problems were diagnosed in the planter pot design model; (1) poor weld lines and (2) warpage of the top. The poor weld lines were only a problem because the temperature of the plastic as it met was too cold for the weld to form properly. The weld line plot (figure A4.21) and the flow front temperature plot (figure A4.22) together reveal the low temperature welds at the base of the pots.

The warpage of the top of the pots was detected in both the fill time plot and the pressure plot. The fill time plot (figure A4.23) shows the corners of the top filling last. This filling pattern is suspicious. A pressure plot (figure A4.24) of the product while it is filling shows the pressure in the sides higher than in the corners. The two contour plots describe a product which would have a tendency to warp.

A4.2.6 Comparison

The product showed similar problems as those found in the simulation. The product did have weak welds at the base. When the product was loaded in a tensile fashion, the welds were the points that broke. The break was also clean, signifying little overlap of molecules had occurred. The warpage of the product was also noticeable (figure A4.25). The last point to fill (figure A4.26) is the same for both the simulation and the actual problem.

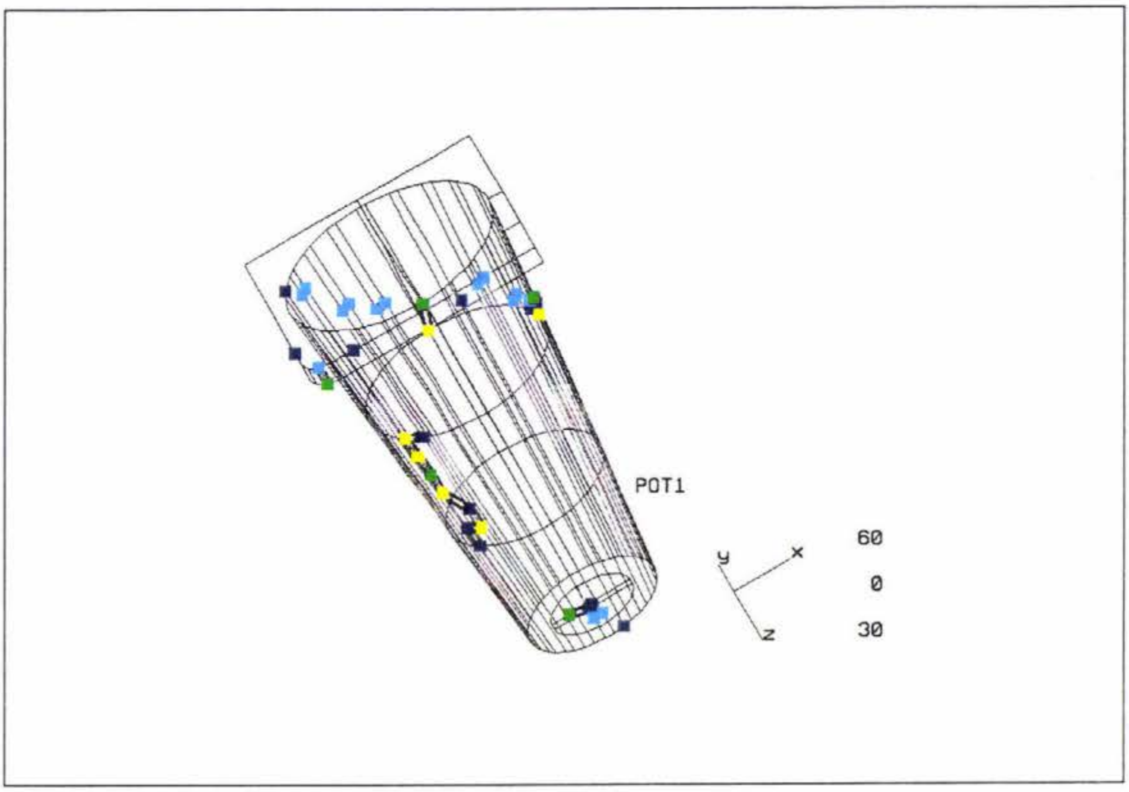


Figure A4.21 Weld Line Plot

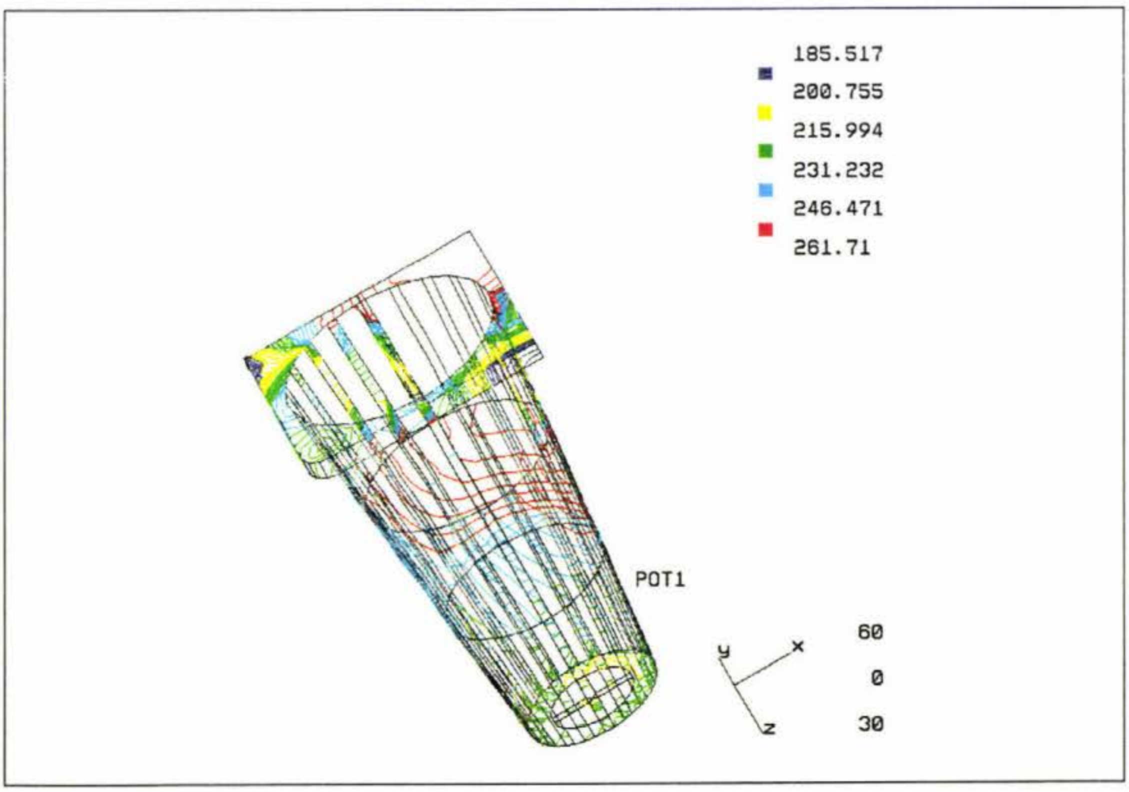


Figure A4.22 Flow Front Temperature Plot. (scale : °C)

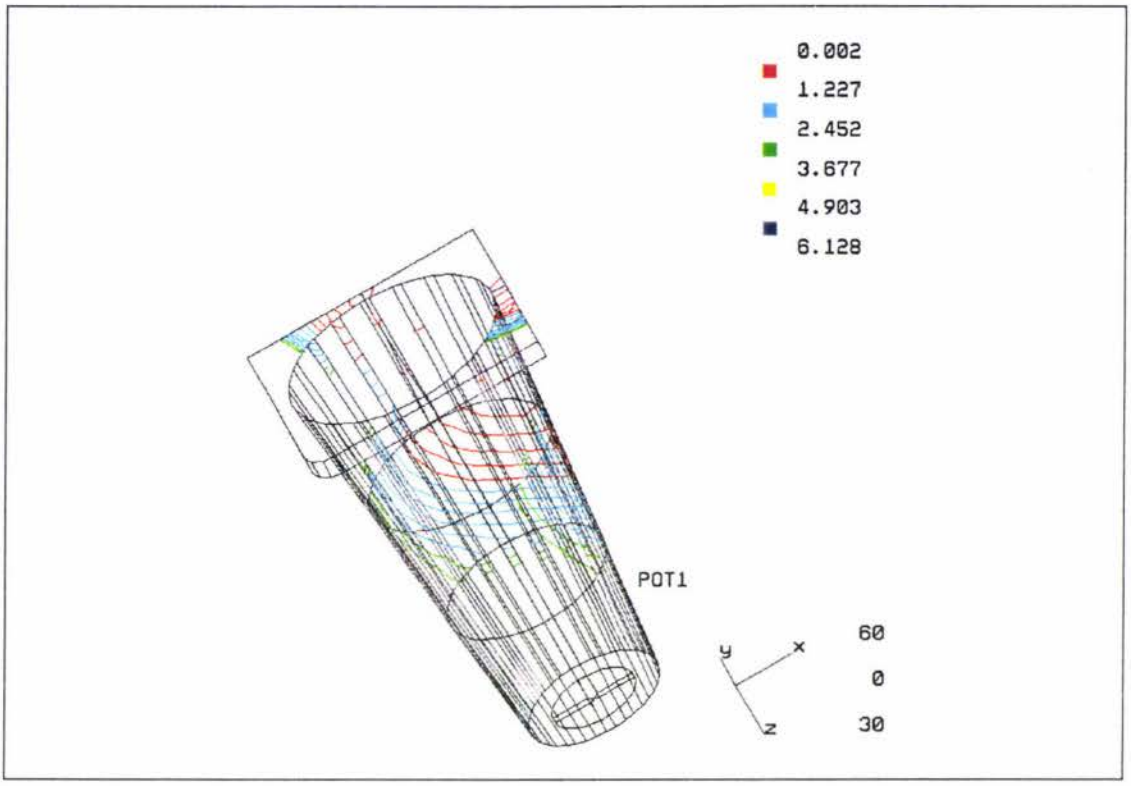


Figure A4.23 Fill Time Plot, Short Shot (scale : seconds)

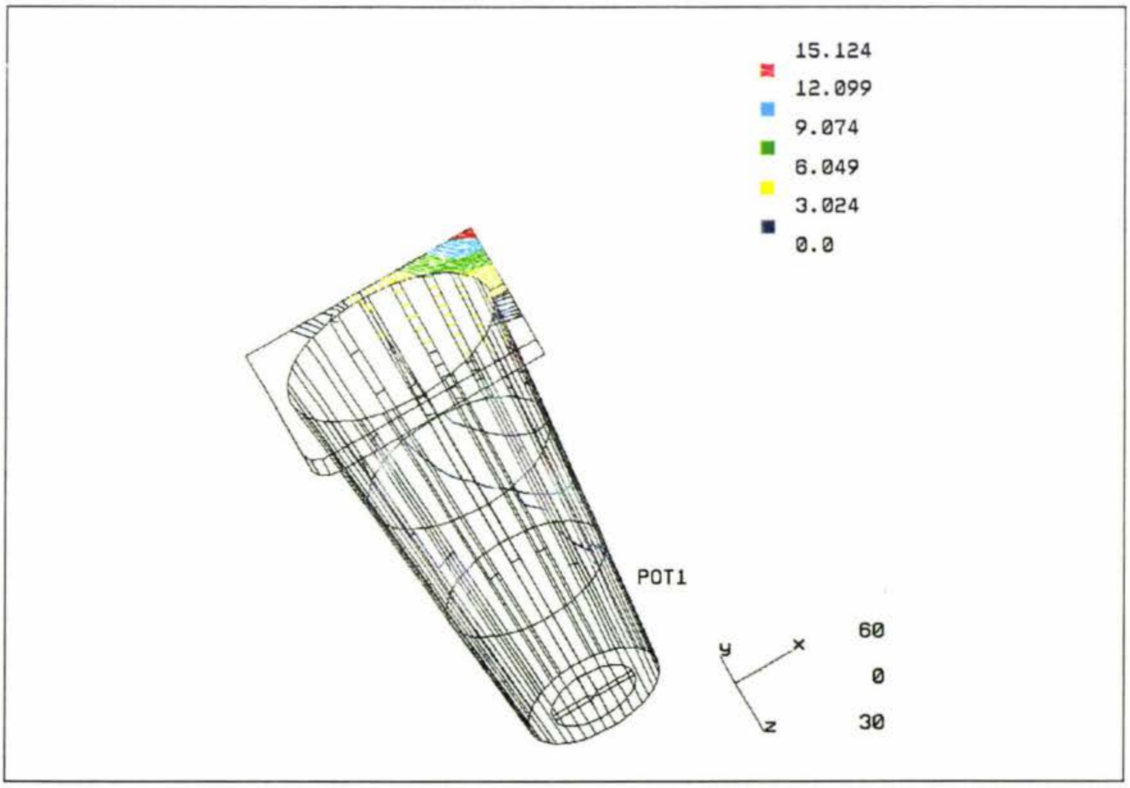


Figure A4.24 Pressure Plot, Short Shot (scale : MPa)

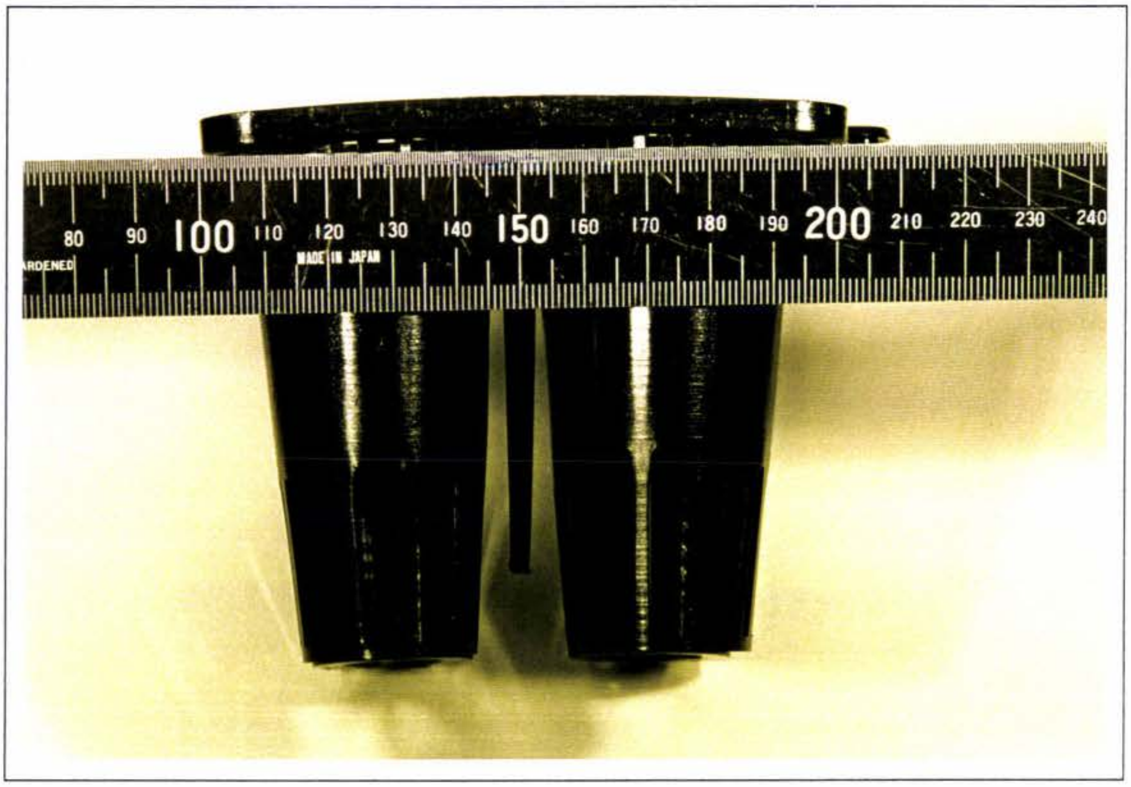


Figure A4.25 Warpage in the Top of the Planter Pot

Figure A4.26 Last Point to Fill

A4.3 Crockpot Lid Knob - Sunbeam

This section presents the results of the knob simulation and moulded knob comparison.

A4.3.1 Product Description

The knob is a very simple product and consists of a handle on the top and a shaft below to connect to the rest of the Crockpot lid. The whole product is 22.7 mm high and 41 mm wide (figure A4.27). The small product lends itself to a multi cavity mould, however, Sunbeam chose to manufacture the product in a single cavity mould because the mould was faster to produce.

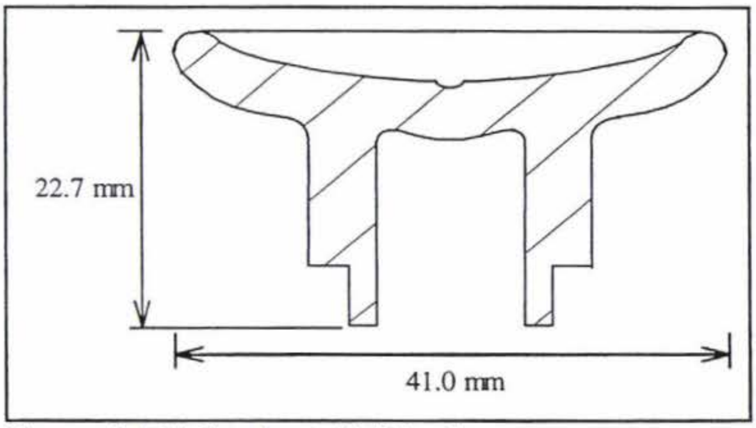


Figure A4.27 Crockpot Lid Knob

A4.3.2 Material Specifications

Polycarbonate is a very hard, rigid polymer. Many products are manufactured from polycarbonate because it is cheaper than most of the engineering plastics, yet has good strength properties. Another very useful property is the high levels of transparency that are possible with polycarbonate.

The following specifications were used to obtain the moulding window:

Mould temperature range:	80 to 120°C
Melt temperature range:	280 to 320°C
Maximum shear stress:	0.500 MPa

Maximum shear rate: 40,000 s^{-1}

A4.3.3 Flow Path

The knob has two flow paths; one to the rim of the handle, the other to the base of the shaft. The longer of the two flow paths is the latter (figure A4.28).

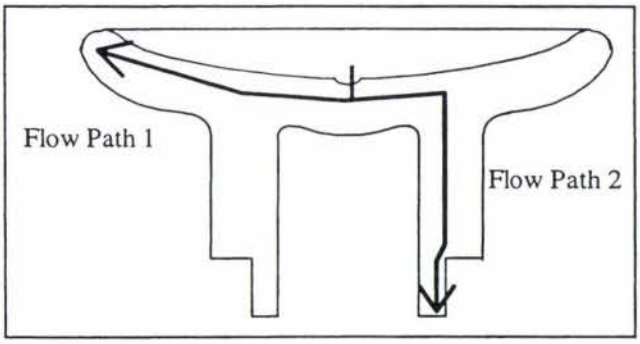


Figure A4.28 Flow Paths for the Crockpot Lid Knob

A4.3.4 Results of a 2D Path Analysis

The moulding window for the knob is shown in figure A4.29. The processing point was outside the moulding window since the injection time used was 12 seconds.

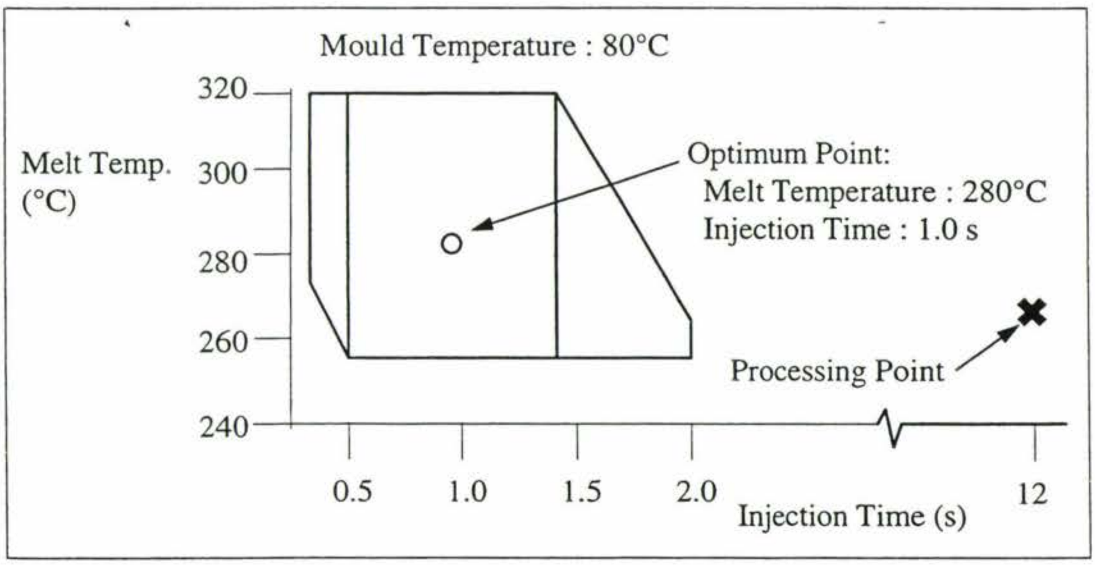


Figure A4.29 Moulding Window for the Knob

A4.3.5 Model Simulation Analysis

The long injection time was the cause of the problems in the knob. The temperature at the end of the filling was very low near the ends of the flow paths. This low temperature (figure A4.30) filling resulted in surface defects.

A4.3.6 Comparison

The low flow front temperature near the ends of the flow paths predicted in the simulation caused the ripples in the manufactured product (figure A4.31).

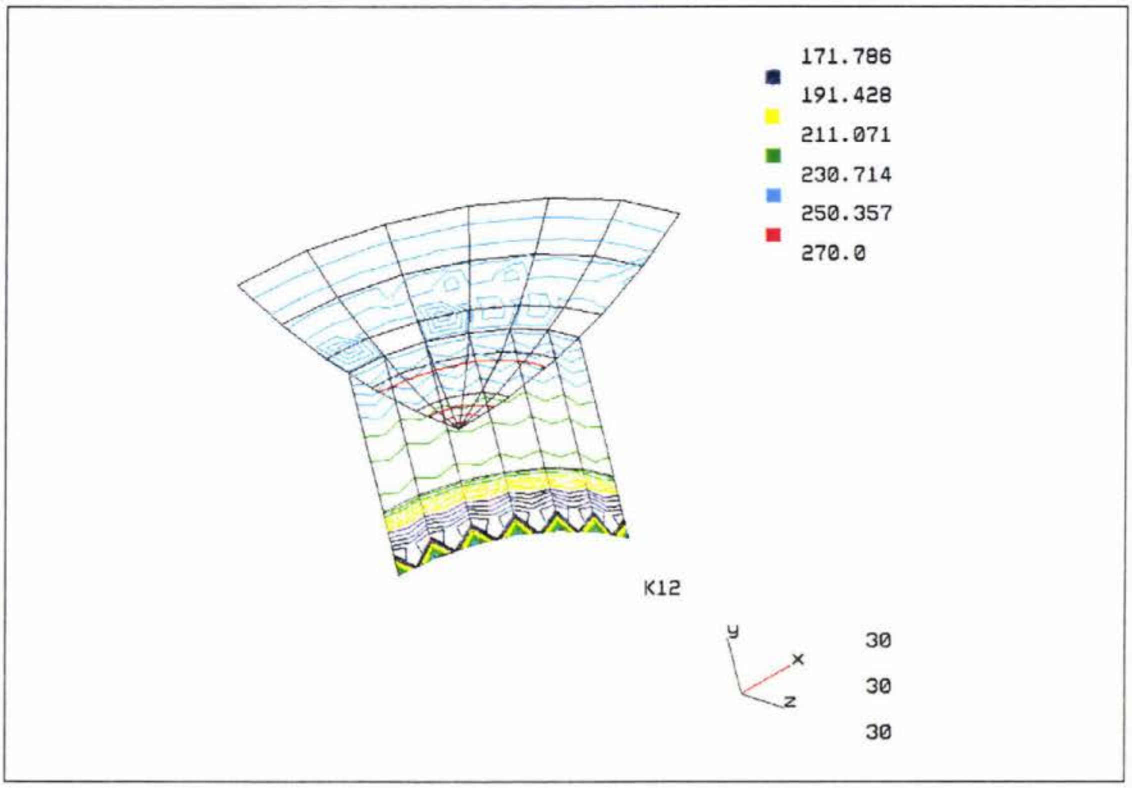


Figure A4.30 Flow Front Temperature for the Knob (scale : °C)

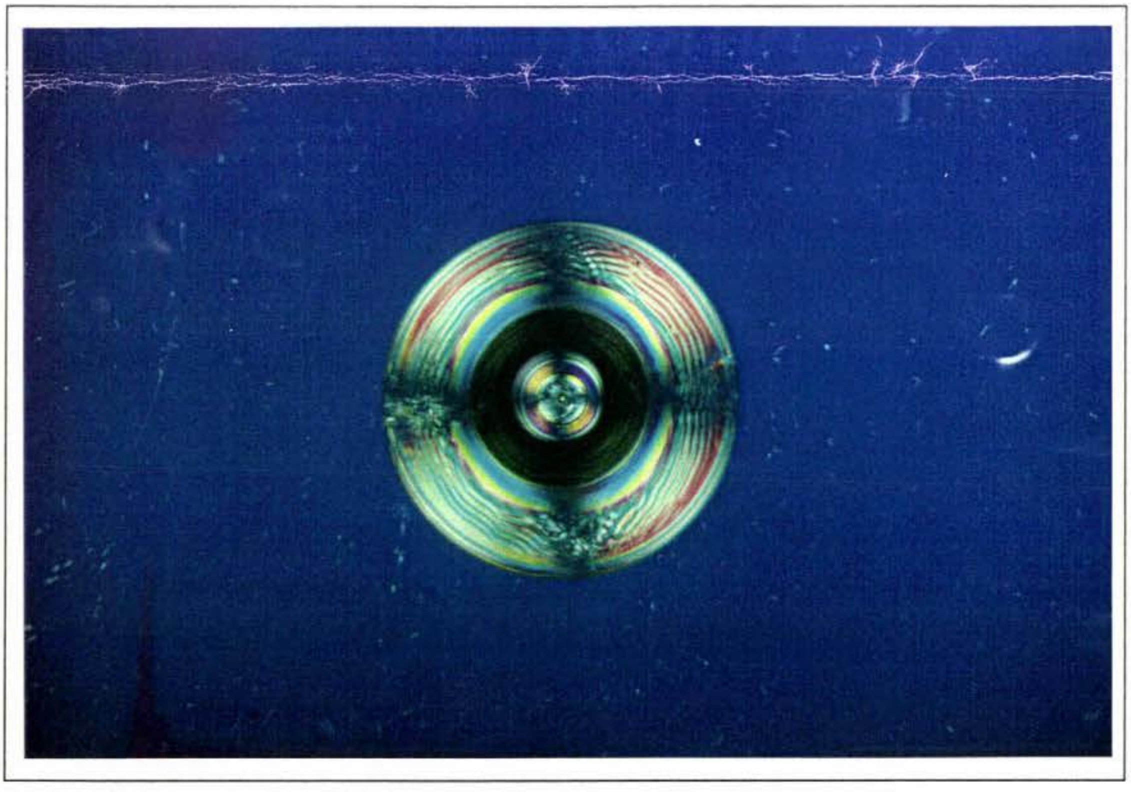


Figure A4.31 Ripples in the Knob, Through Polarised Filters

A4.4 Bench Top Oven Strip

This section presents the results of the analysis performed on the strip for the bench top oven.

A4.4.1 Product Description

The bench top oven is manufactured from several different materials, a number of which are plastic and produced using injection moulding. Two of the parts, the strips, are manufactured in the same mould. The strips are 213 millimetres long and only 11 millimetres wide. There are six holes in the base of the product and a raised edge that runs down the middle, only interrupted when it meets a hole. Two tabs have been included in the design as injection points. (figure A4.32). The second strip was similar, but had fewer holes and a cut-out at one of the ends.

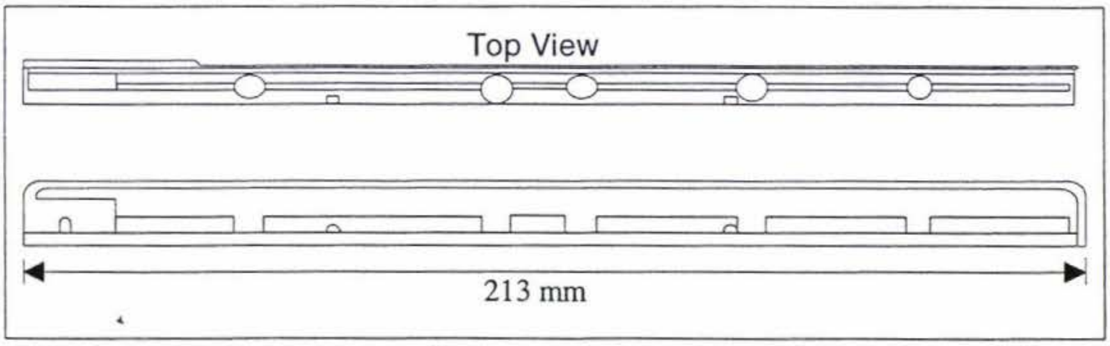


Figure A4.32 Strip

A4.4.2 Material Specifications

Polybutylene terephthalate (PBT) is an engineering plastic used for products that require high strength at high temperatures. Typical heat distortion temperatures can be as high as 205°C for PBT. The polymer is available in flame retardant grades, a useful property for the strips. The particular material used for the analysis was GE Valox 420 with 30% glass reinforcing.

PBT has the following specifications for Moldflow moulding window creation:

Mould temperature range:	40 to 80°C
Melt temperature range:	220 to 260°C (max. 300°C)
Maximum shear stress:	0.400 MPa
Maximum shear rate:	50,000 s^{-1}

A4.4.3 Flow Paths

The flow paths in the strips were very simple. The plastic enters the gate and flows to either end along the base and then up the side. The dominant flow path is obviously the longer of the two flow paths.

A4.4.4 Results of a 2D Path Analysis

The moulding window for the strips was found using the Moldflow software (figure A4.33). The moulding window was very small for the material chosen. It could have been increased by selecting a similar polymer with a higher viscosity.

A4.4.5 Model Simulation Analysis

The strip model filled well. The only problem, which was noted before the analysis, was the unbalanced flow. One end of the product always filled first, setting up a differential pressure between the two ends, resulting in warpage. This is documented in the fill time plot and the flow front pressure plot (figures A4.34 and A4.35, respectively).

A4.4.6 Comparison

The warpage is quite noticeable in the strips, as shown in figure A4.36.

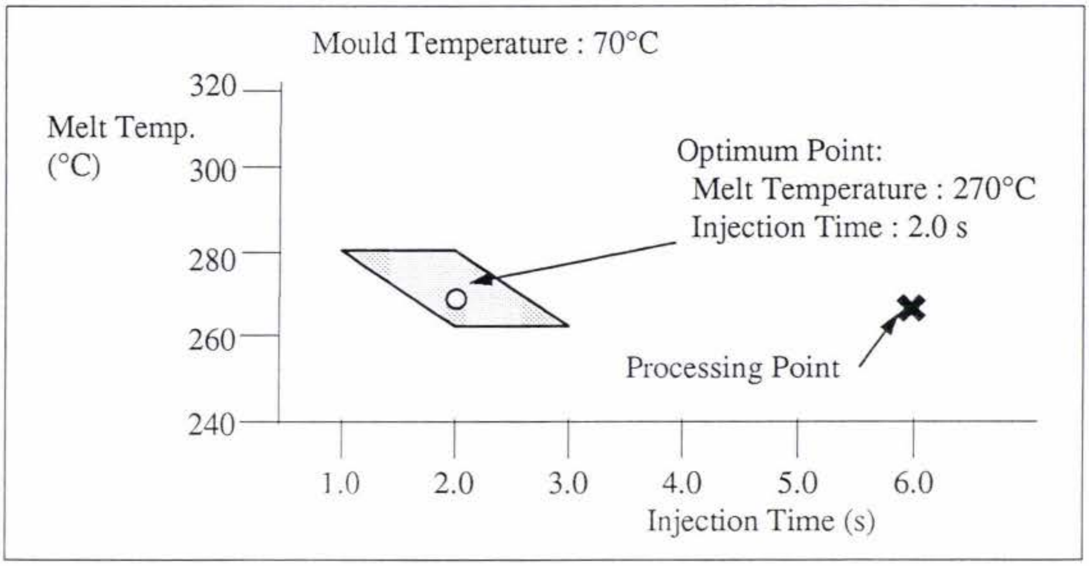


Figure A 5.33 Moulding Window for the Strips

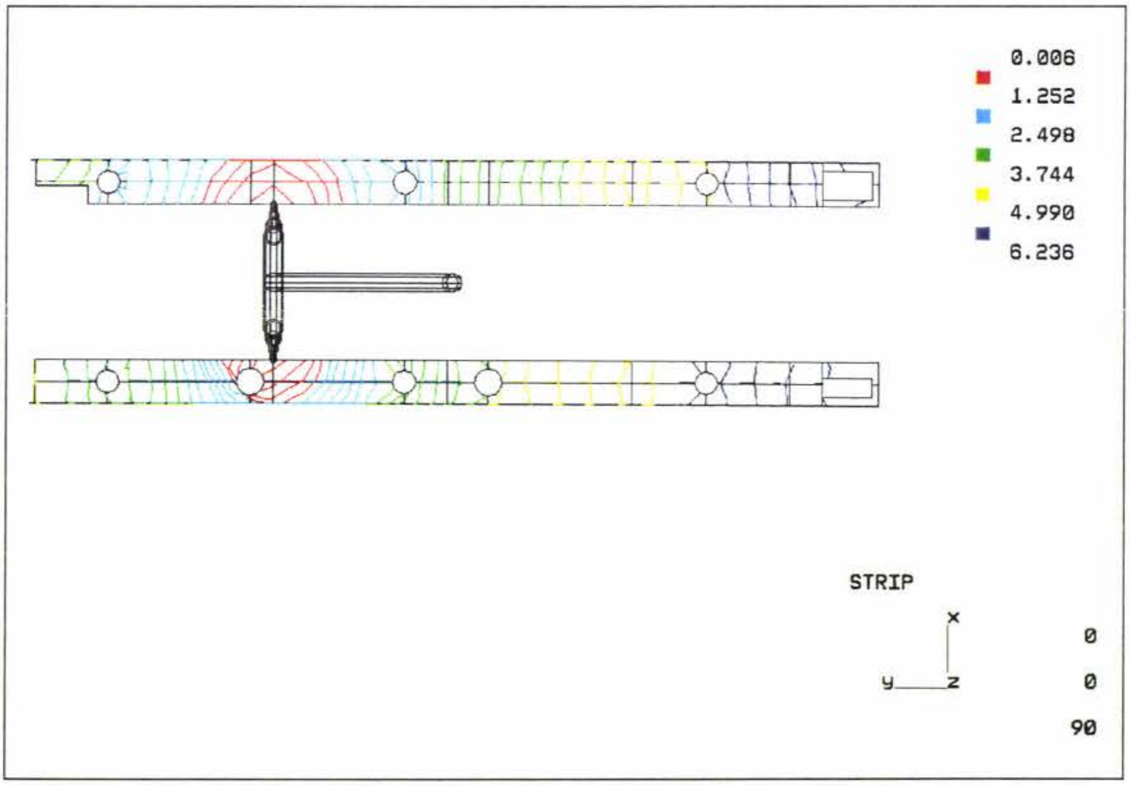


Figure A4.34 Fill Time Plot for the Strip (scale : seconds)

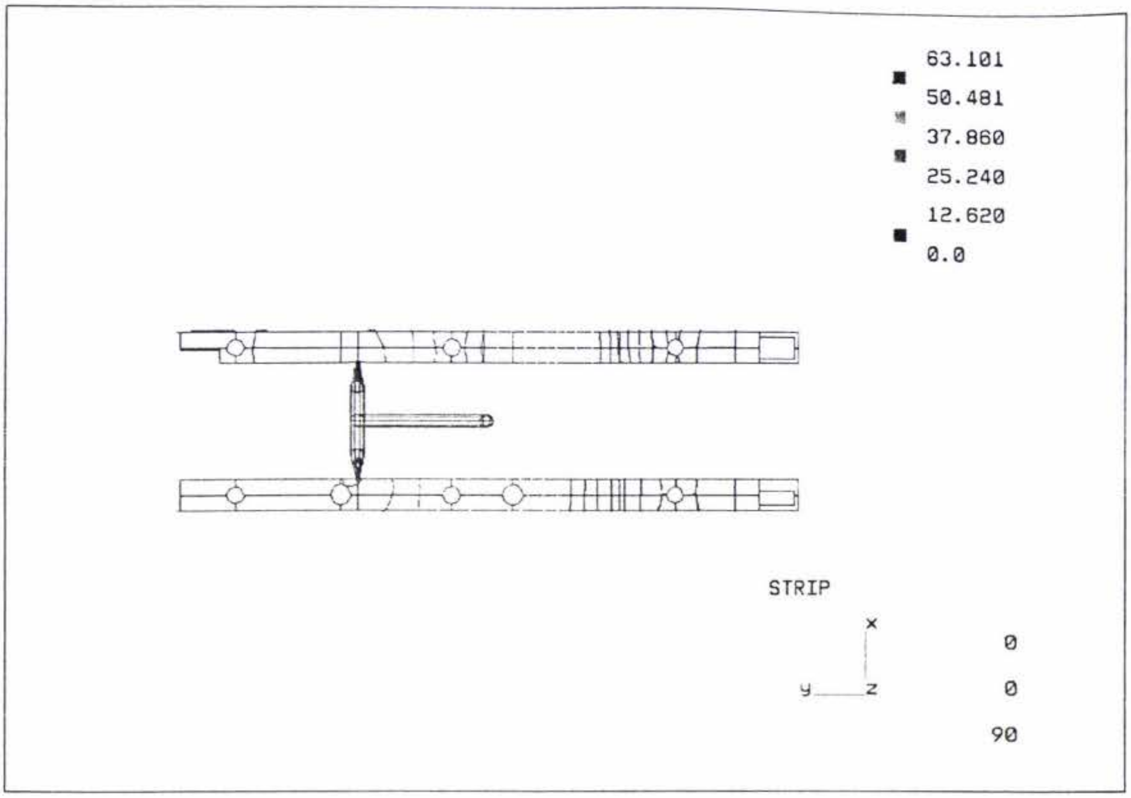


Figure A5.35 Flow Front Pressure for the Strip

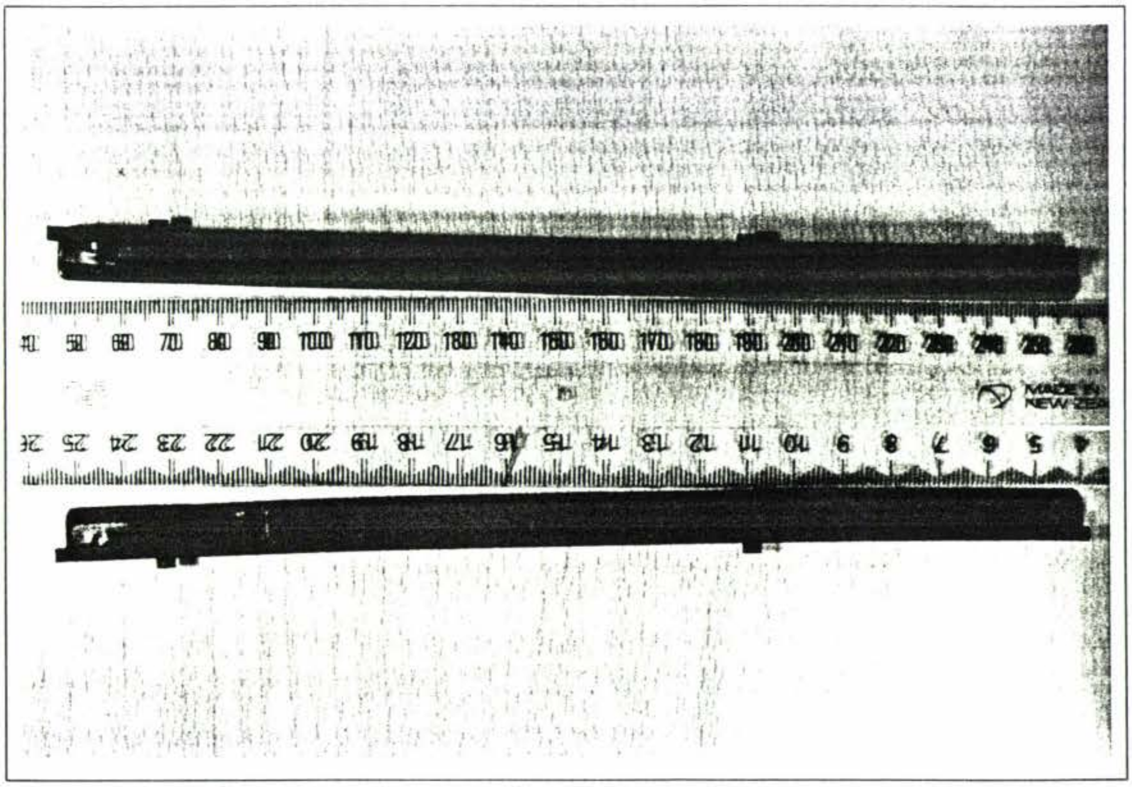


Figure A4.36 Warpage in the Strips

BIBLIOGRAPHY

Agassant, J. F. and others. (1993).

General principles of polymer processing modeling. vol 68 p 153-167. Macromolecular Symposia. Basel : Hüthig and Wepf Verlag.

Austin, C. (1985).

Computer-aided part and mould design. p 111-160. Developments In Injection Moulding - 3. Edited by Whelan A. and Goff J. P. Essex : Elsevier Applied Science Publishers Ltd.

Beyer, C. E. and Spencer, R. S. (1960).

Rheology in Molding. vol. 3 p.505-551. Rheology. Edited by Eirich, F. R. New York : Academic Press.

Bird, R. B. and others. (1987).

Dynamics of Polymeric Liquids. vol. 1 Fluid Mechanics. 2nd ed. New York : John Wiley & Sons, Inc. 649p.

Birley, A. W. and others. (1992).

Physics of plastics. Processing, Properties and Materials Engineering. New York : Oxford University Press. 528p.

Brydson, J. A. (1989).

Plastics Materials. 5th edition. London : Butterworths. 839p.

Bush, S. F. (1984).

Plastics in engineering design: an overall analysis. p.487-499. In Design Engineering Conference (7th : 1984 : Birmingham, UK). Proceedings of the 7th Annual Design Engineering Conference : IFS : Oxford : B. J. Davies, Editor, 510p.

Cheremisinoff, N. P. (1993).

An Introduction to Polymer Rheology and Processing. Boca Raton : CRC Press, Inc. 280p.

Fisher, J. M. and Halford, J. D. (1985).

Computer Assisted Design. vol. 4. p.109-117. Encyclopedia Of Polymer Science And Engineering. 2nd Edition. Editor in Chief Jacqueline I. Kroschwitz. New York : John Wiley and Sons, Inc.

Friedl, C. and Kennedy, P. (1992).

Injection Molding of Polymers : Computer Simulation. p.375-380. Concise Encyclopedia of Polymer Processing and Applications. Edited by Patrick J. Corish. Oxford : Pergamon Press. 771p.

Gastrow, H. (1993).

Injection Molds : 108 Proven Designs. 2nd Edition. Edited by E. Lindner and P. Unger. Munich : Carl Hanser Verlag. 248p.

Hall, C. (1989).

Polymer Materials - An Introduction For Technologists and Scientists. 2nd edition. London : MacMillan Education Ltd. 224p.

Innace, J. J. and Myers, J. (1994).

CAD/CAE programs are proliferating. p. 54-58. Modern Plastics International, November 1994.

Kamal, M. R. (1992).

Injection Molding of Thermoplastics : Mold Filling. p.380-385. Concise Encyclopedia of Polymer Processing and Applications. Edited by Patrick J. Corish. Oxford : Pergamon Press. 771p.

Menges, G. and Recker, H. (1985).

Computer Assisted Engineering. Supplementary volume. p.130-153. Encyclopedia Of Polymer Science And Engineering. 2nd Edition. Editor in Chief Jacqueline I. Kroschwitz. New York : John Wiley and Sons, Inc.

Menges, G. and Mohren, P. (1993).

How to make injection Molds. 2nd ed. Munich : Carl Hanser Verlag. 540p.

Moldflow Design Principles. (1990). Melbourne : Moldflow Pty Ltd. 54p.

Moldflow MF/Cool 3.1 Cooling Analysis Manual. (1992). Melbourne : Moldflow Pty. Ltd.

Moldflow MF/Flow 9.0 Training Manual. (1994). Melbourne : Moldflow Pty. Ltd.

Mapleston, P. (1994).

Versatile CAE software speeds part development. p.28-31. Modern Plastics International, February 1994.

Morton-Jones, D. H. (1989).

Polymer Processing. London : Chapman and Hall Ltd. 260p.

Pearson, T. (1995).

Gas Assisted Moulding for Improved Process Economics. p119-121. Materials World.

Pan, X. (1994).

Technology and Innovation Management - Analysis of the NZ Plastics Industry - Technology Status, Problems and Opportunities. Thesis, MTech, Massey University.

Powell, P. C. (1983).

Engineering with Polymers. London : Chapman and Hall. 318p.

Rees, H. (1985).

Molds. Supplementary volume. p.507-533. Encyclopedia Of Polymer Science And Engineering. 2nd Edition. Editor in Chief Jacqueline I. Kroschwitz. New York : John Wiley and Sons, Inc.

Rosato, D. V. and others. (1991).

Designing With Plastics and Composites. A Handbook New York : Van Nostrand Reinhold. 979p.

Rubin, I. I. (1972).

Injection Moulding Theory and Practice. New York : John Wiley and Sons. 657p.

Shah, P. L. (1990).

Rheology. p.1543-1559. Handbook Of Plastic Materials and Technology. Edited by Rubin, Irvin I. New York : John Wiley and Sons, Inc.

Schröer, P. (1983).

Injection Molds. p. 23-52. Mold-Making Handbook For The Plastics Engineer. Edited by Stoeckhert, Klaus. München : Carl Hanser Verlag.

Society of the Plastics Industry, Inc. (1991).

SPI Plastics Engineering Handbook of the Society of the Plastics Industry, Inc. 5th ed. Edited by Berins, Michael L. New York : Van Nostrand Reinhold. 869p.

Throne, J. L. (1979).

Plastics Process Engineering. New York : Marcel Dekker Inc. 924p.

Vallens, A. (1993).

Software brings speed, economy to moldmaking. p. 47-49. Modern Plastics International, April 1993.