Mould Design and Manufacturing Using Computer Technology

A thesis presented in partial fulfilment of the
requirements for the degree of

Master in Technology

at Massey University, Palmerston North
New Zealand

Yuk-sun Lam
2000

ACKNOWLEDGEMENTS

To my supervisor, Dr. L. Q. Tang, for her advice and assistance throughout the project. Your guidance was invaluable when I needed to be reminded of the purpose of the project. I am sincerely grateful for your support.

To staff within the Institute of Technology & Engineering for helping me during the past one year. I particularly wish to thank Leith Baker, whose cooperation has been gratefully appreciated.

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

To my wife, Tracy, whom I love dearly and thank you 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 have led me on an exciting and educational journey. Thank you both.

ABSTRACT

This thesis presents a research work carried out at the Institute of Technology and Engineering of Massey University. The overall goal of the project is to develop an integrated mould design and manufacturing system for teaching and research. The development involves establishing a computer-aided design, computer-aided manufacturing, CNC machining, and injection moulding program for Technology and Engineering undergraduate and postgraduate students. An integrated CAD/CAM/CAE and CNC system has been established through this project. Four laboratory courses together with the tutorials have been developed and established. This design and manufacturing system and the corresponding laboratory courses provide a great support for Engineering Design and Manufacturing programmes. The laboratory courses give the opportunities to the students to gain the experience from concept design, through hands-on project, to the final physical product.

The research work presented went through four stages. Before this project started, the Institute of Technology and Engineering (ITE) had a stand alone CNC milling machine, a CNC lathe, an injection moulding machine, a limited number of PCs and a few CAD licenses. The first stage was to develop an integrated CAD/CAM/CAE and CNC system based on the existing facilities in ITE and the limited funding provided by the fund of innovation and excellence in teaching. The second stage was focused on product design. It was realized that a product is needed for the development of this project. This product must have the features to demonstrate the applications of computer-aided technologies in product design, analysis and manufacturing. The product must also be a plastic component in order to smoothly integrate plastic injection moulding technologies within the programme. Stage three was to build and produce the die set of the product, which involves mould design and analysis, simulation, testing, tooling, and actual CNC machining. The final stage was to develop the laboratory courses, write the tutorials and produce a demonstration program. Four lab courses and the tutorials were developed,

which cover CAD, CAM, CNC, and injection moulding. These labs provide students with hands-on and practical experience.

The project has been successfully completed. The demonstration program is now used for ITE's open day and other occasions such as visitors from industries, high schools and other organisations. The four lab courses are now taken by 3rd and 4th year B.Eng and B.Tech students in the Institute of Technology and Engineering.

CONTENT

LIST OF FIGURES

LIST OF TABLES

1. INTRODUCTION

The Institute of Technology and Engineering (ITE) is an industrial-oriented Institute at Massey University. It offers a number of undergraduate and postgraduate Engineering programmes. In order to meet students' requirements and challenge in the competitive market, the Institute is moving towards the direction of delivering high quality, motivated, computer-based, up-to-date educational programmes. The project presented in this thesis is supported by the Institute with the focus to develop An Integrated Mould Design and Manufacturing System for undergraduate teaching.

The development of such an integrated mould design and manufacturing system involves developing a product to integrate CAD/CAM, CNC machining and injection-moulding techniques. It is important to come up with a product that would motivate the learning spirit of the students. This led to the idea of producing a souvenir for Engineering and Technology students, which can be memorabilia to be proud of and to portray study within the Institute. The souvenir is also expected to motivate students into learning mould design and manufacturing using CAD/CAM systems, moulding process analysis and simulation, CNC machining and injection moulding techniques. Researching, brainstorming and discussions led to a few possible choices. 3D computer models of the souvenirs were first built in the CAD system. Prototypes were then successfully made. Following the prototype development, two models, one is the business card holder, the other is the picture frame, were chosen as the souvenirs to be the outcome of the mould design and manufacturing lab course. The development of the lab content to produce the die set of the souvenir was then carried out, which includes mould design, building the assembly computer models of the die set and plastic moulding analysis. Manufacturing process planning and CNC simulation were carried out using CAM systems, which then successfully generated the CNC code for the machining of the moulds. The core and

cavity plates were finally machined out using the CNC machines in the workshop. Fig. 1 presents an illustration of the design and manufacturing process from the concept design to the final product. The lab course provides the students with an opportunity to face the real problems in mould design and manufacturing and gives them a hands-on opportunity in mould design, analysis, CNC machining and injection moulding. At the end of the course, students are required to hand in the project report which includes the computer models of the product and the die set, mould analysis and simulation results, and the results of CNC simulations of the core and cavity plates for the CNC machines in the workshop.
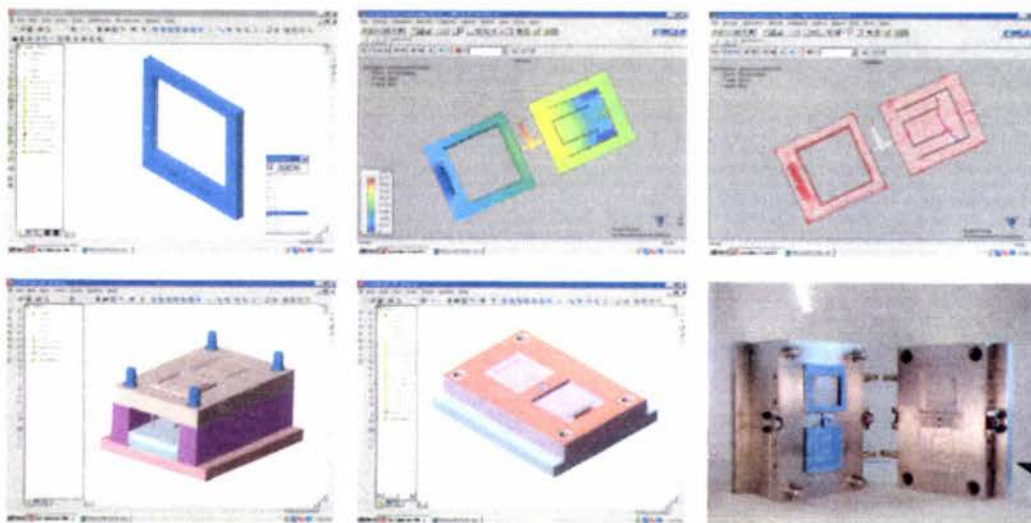
Figure 1-1 From concept design to final product.

1.1. FACILITIES IN ITE

The Institute of Technology and Engineering does have a CAD Lab and an Engineering Workshop. When the project started, the facilities available in the CAD Lab were a few Pentium 133 PCs and a few Solidworks 98 licenses. In the Engineering Workshop, there is an old CNC milling machine, a CNC lathe and an injection moulding machine. In

order to establish an integrated CAD/CAM/CAE and CNC environment, the Institute equipped the CAD Lab with new computers and up-to-date software such as the latest version of Solidworks, new arrival of Camworks, C-mould etc. With the new facilities arrived, an integrated system was able to be quickly established. However, as the CNC machine tool is an old model, it eliminated some of the powerful functions the new software provided. At the same time, some unwanted problems came out, such as the compatibility between the software and the hardware. A solid understanding of the facilities in the workshop and the new software was required in order to build a smooth communication between the hardware and the software.

1.2. MOULD DESIGN AND MANUFACTURING

Mould design and manufacturing are important parts in manufacturing industry. So many approaches are presented to shorten the product development cycles for developing a new product such as domestic appliances and car parts. As the products become more complicated, especially in free-form shapes, thin wall thickness and the complex moulding parts, to satisfy these requirements the quality of the product is now depending on the use of new technologies and the know-how human resources. In recent years, PCs have become more and more powerful and the price has significantly dropped. It is no longer a big issue for small and medium-size companies to afford PCs. At the same time, CAD, CAM and CAE systems have become more and more popular. There are so many CAD, CAM and CAE packages on the market to select for using in mould design and manufacturing field. The application of computer technologies requires today's engineers to be familiar with computer techniques, especially in an integrated CAD/CAM/CAE environment. On the other hand, universities are facing the demanding of producing graduates with the matching knowledge and skills.

3

This project developed an injection moulding lab course which aimed at the application CAD/CAM/CAE technologies in mould design and manufacturing. This lab provides the information about mould design rules, prediction of the problems which may occur during the injection moulding process, such as poor unbalanced runner system, vent location, weld-lines etc. It also provides students with an opportunity to use CAD/CAM/CAE and CNC techniques such as the simulation of injection moulding process, the selection of machining parameters and the simulation of CNC machining.

1.3. THE OBJECTIVE OF THE PRESENT WORK

To fulfill the main target of this project, which is to develop an integrated mould design and manufacturing system for undergraduate teaching, several objectives were set which involved developing an integrated CAD/CAM/CAE and CNC environment and the corresponding the lab courses.

1. Develop a product for the lab courses

Design a product which has features to demonstrate the techniques of CAD, CAM, CAE, CNC and injection moulding. The product is also suitable for using as an example for undergraduate teaching.

2. Establish an integrated CAD/CAM/CAE and CNC system

Based on the facilities available to build a smooth link between the hardware and the software. This requires a detail study both in the CAD lab and the workshop to totally understand the functions and the requirements of the software and the capabilities of the machine tools and the controllers.

3. Build the die set

According to the size and the capability of the injection moulding machine and the CNC machine tools in ITE's workshop, design and manufacture a die set to produce the product.

4. Make the physical product

To make the physical product requires choosing the proper material, assembling the die set, testing the injection moulding machine and fine tuning.

5. Develop the CAD, CAM, CAE lab courses

Develop the hands-on mould design and manufacturing lab courses including the tutorial guides.

Chapter 1 of this thesis is an introduction of the present work and Chapter 2 the product design and the requirements. Mould design and manufacturing are discussed in detail in Chapter 3. The structure of dies and the manufacturing process and the integrated mould design and manufacturing system are dealt with in Chapter 4 and 5. Chapter 6 presents the outcome of this project and the discussion and conclusions.

2. PRODUCT DESIGN

2.1. Introduction

A product was needed for the development of this project. A souvenir was created to demonstrate the design concepts, functions and the applications of 3D CAD systems. This chapter presents the souvenir design process and the final product.

2.2. Product design

A product was needed for this project. This product had to have the features to demonstrate the applications of computer-aided technologies in mould design, analysis, and manufacturing. The product also had to be a plastic component in order to integrate plastic injection moulding techniques. It was also important to come up with a product that would motivate the learning spirit of the students. This led to the idea of producing a souvenir. The souvenir was expected to motivate Engineering and Technology students into learning mould design and manufacturing using CAD/CAM systems, moulding process analysis and simulation, CNC machining and injection moulding techniques.

As the souvenir was for students the cost became an important issue. The first aim of the project was to look for an adequate material on the market. After several discussions and comparisons, a decision was made to use recycled plastics as the Institute has this type of material available. Apart from the above considerations, the design of the souvenir also had to consider the capabilities of the facilities available in ITE.

6

Following the design requirements, the early stage of the project came out with a number of ideas based on some market research. There were three designs to choose from. Following paragraphs will show these designs and the advantages of each design as well as the final selection.

2.2.1. Cardholder

Figure 2-1 illustrates the first design. This design has the ITE and Massey University letters on the top surface. In general the ideal of the business card holder is acceptable. It satisfies the design requirements. However, on mould design, the structure of the mould will require to use a three-plate mould if not using a sprue gate. Fig. 2-2 and Fig. 2-3 show the possible structure of the die set using a pin gate.

Figure 2-1 The design of the card holder.
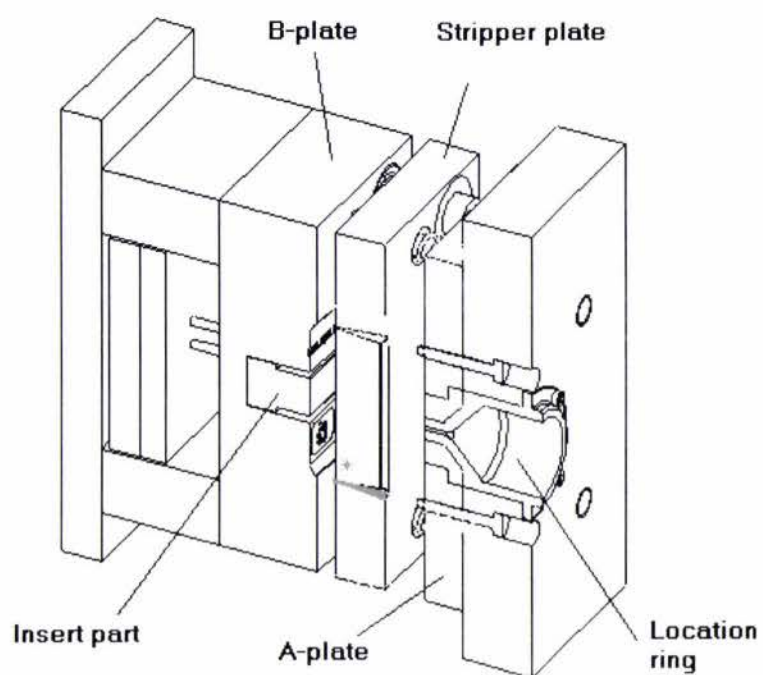
Figure 2-2 The mould structure of the card holder.
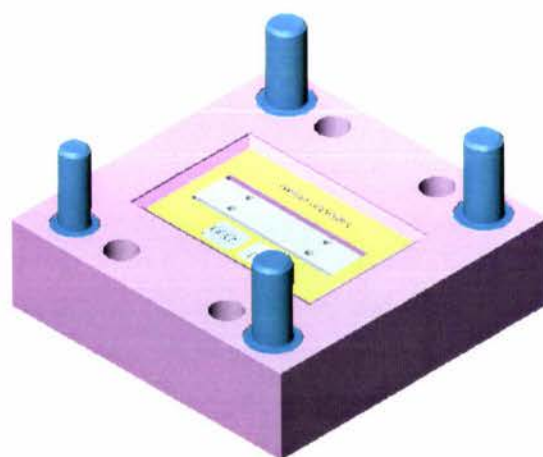
Figure 2-3 The B plate with inserted part

2.2.2. Design two – A picture frame with a 3D surface

Design a picture frame was one of the ideas. Every graduate will need a picture frame for the graduation portrait. This design was aimed to produce a picture frame which would be convenient to hold a small size graduation picture at home or office. It was hoped that the design would be simple, easy to make and assemble with a little formal, fashionable and attractive feeling. This design, as it has a 3D surface, requires at least a 3-axis CNC facility. This is the drawback of this design.

The final design of the picture frame is shown in Fig. 2-4. For this design, the standard two plate mould can be used. The picture frame and the supporting back plate can be designed in the same die set. The A-plate, B-plate and the die set are illustrated in Fig. 2-5, 2-6 and 2-7.
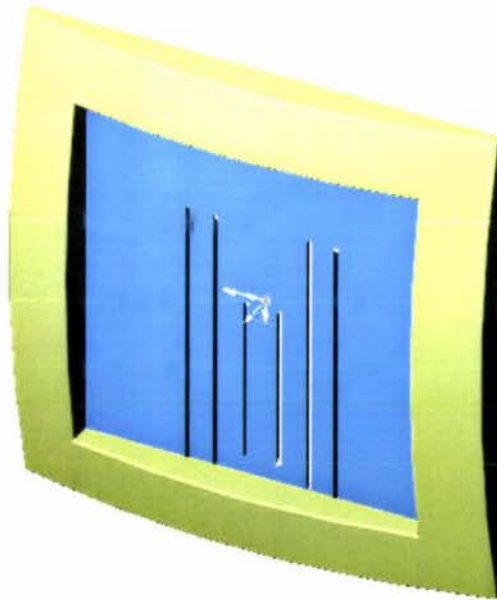
Figure 2-4 A picture frame with a 3D surface.
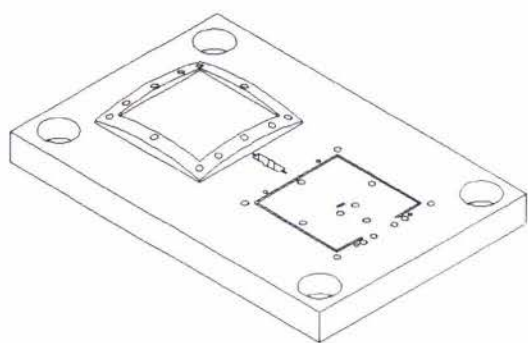
Figure 2-5 B plate of the die set.
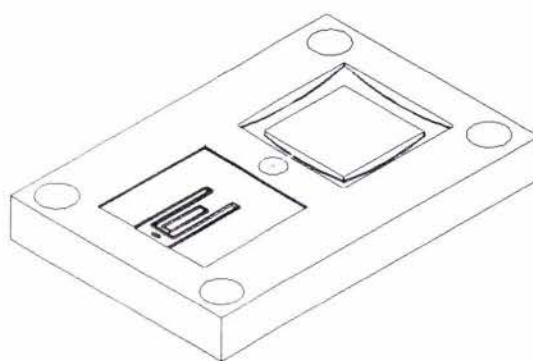
Figure 2-6 A plate of the die set.
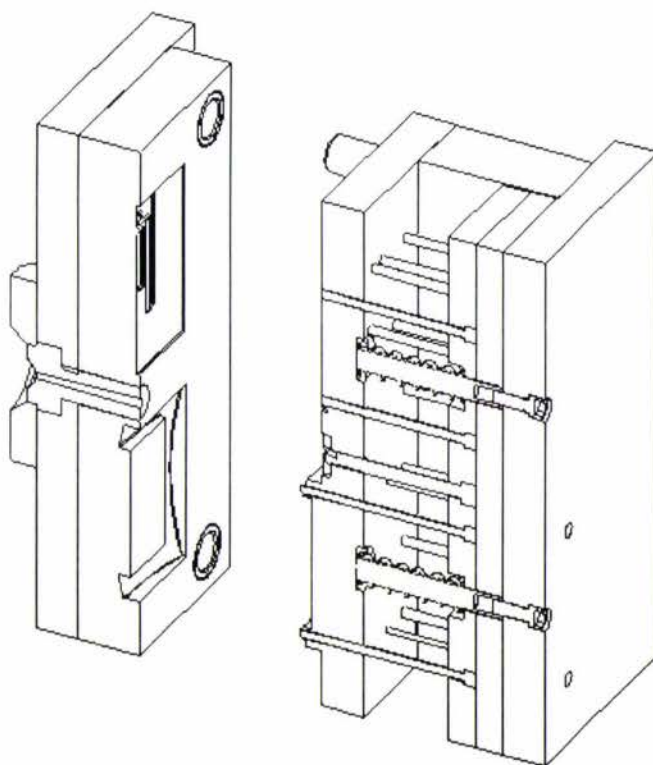
Figure 2-7 The mould structure of the picture frame.

10

2.2.3. Design three – A picture frame

Base on the existing 2 ½ axis CNC facilities in ITE and the concept of the 3D picture frame design, a modified design was made and is shown in Fig. 2-8. The modification was made on the 3D surface. The new design has a flat surface on the front. Patterns such as logos or letters can be added to the front surface using inserts. The supporting plate and the structure as well as the assembly were kept the same as design two. Fig. 2-9 shows the final design of the picture frame.
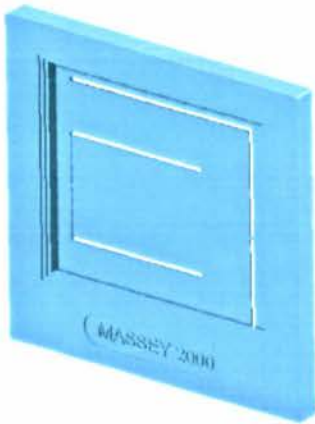
Figure 2-8 The modified picture frame.   **Figure 2-9 The final design of the picture frame.**
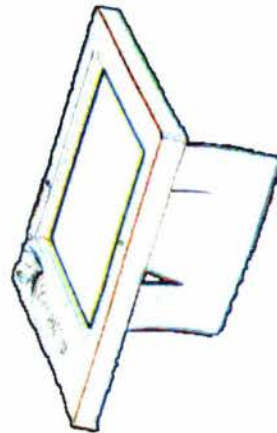
2.3.   Design selection

All the designs were modeled in a 3D CAD system (Solidworks) to present a 3D view of the product. Based on the design, the facility availabilitiy in ITE, and the selection criteria, which are simple, low cost, easy to make, and can be used to integrate Computer-

Aided Technologies in mould design and manufacturing, the final selection was Design Three, the modified picture frame. The is because Design One requires 3-plate die set, which is more complicated than 2-plate die set. Design Two is a good design. However, ITE dose not have a 3-axis CNC machine, which means the work needs to be contracted to outside companies. This will increase the cost significantly. Design Three was a suitable product. The size and the simple shape meet the requirement of this project and can be produced within the Institute.

3. FOUNDATIONS OF MOULD DESIGN AND MANUFACTURING

3.1. Introduction

The quality of a plastic product largely depends upon the mould design and manufacturing. This chapter presents the foundations of mould design and manufacturing, which covers the basic considerations and the commonly used techniques.

3.2. Mould design and manufacturing foundations

Conventional mould design is a process of developing a mould following the guidelines obtained from mould design and manufacturing experience. Mould design is based on a trial and error approach. A trial of injection cycle is to diagnose any problems that occur in the mould set. If the causes of the problems are found, they will be eliminated by changing the mould design or moulding parameter. A mould design consists of several components. Fig3.1 illustrates a simple standard mould. To design a mould , the designer must consider the filling process, ejecting process, the cooling system and the structure of the mould. The Following paragraphs present the considerations which should be taken in mould design.
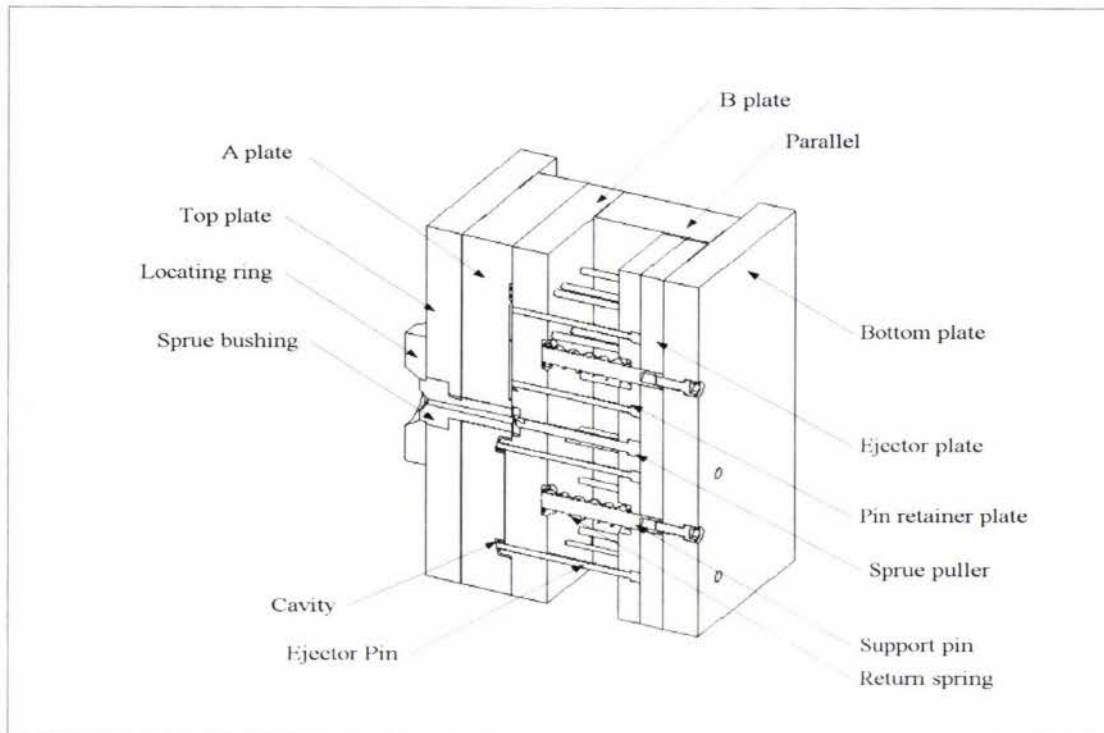
Figure 3-1 The base mould structure of a standard two-plate mould.

3.2.1. Wall thickness

Wall thickness is one of the important factors to be considered in the design of plastic components. The determination of wall thickness should be based on the requirements of structure, strength, precision etc. Basically, the wall thickness should be made as uniform as possible to eliminate part distortion, internal stresses and cracking. The wall thickness should not be too great or too small. Table 3-1 gives the preferred minimum, average, maximum wall thickness for the various thermoplastics and Table 3-2 is for thermosetting materials[11].

Thermoplastic materials	Minimum inches	Average inches	Maximum inches
Acetal	0.015	0.062	0.125
ABS	0.030	0.090	0.125
Acrylic	0.025	0.093	0.250
Cellulosics	0.025	0.075	0.187
FEP fluoroplastic	0.010	0.035	0.5
Ionomer	0.025	0.062	0.75
Nylon	0.015	0.062	0.125
Polycarbonate	0.040	0.093	0.375
Polythylene (L.D.)	0.020	0.062	0.25
Polythylene (H.D.)	0.035	0.062	0.25
Ethylene vinyl acetate	0.020	0.062	0.125
Polypropylene	0.025	0.080	0.300
Polysulfone	0.040	0.100	0.375
Modified PPO	0.030	0.080	0.375
Polystyrene	0.030	0.062	0.250
SAN	0.030	0.062	0.250
PVC-rigid	0.040	0.093	0.375
Polyurethane	0.025	0.500	1.500

Table 3-1 Suggested wall thickness for thermoplastic moulding material[11].

Thermosetting materials	Minimum inches	Average inches	Maximum inches
Alkyd-glass filled	0.040	0.125	0.500
Alkyd-mineral filled	0.040	0.187	0.375
Diallyl phthalate	0.040	0.187	0.375
Epoxy/glass	0.030	0.125	1.000
Melamine-cellulose filled	0.035	0.100	0.187
Urea-cellulose filled	0.035	0.100	0.187

Phenolic-general purpose	0.050	0.125	1.000
Phenolic-flock filled	0.050	0.125	1.000
Phenolic-glass filled	0.030	0.093	0.750
Phenolic- fabric filled	0.062	0.187	0.375
Phenolic-mineral filled	0.0125	0.187	1.000
Silicone glass	0.050	0.125	0.250
Polyster premix	0.040	0.070	1.000

Table 3-2 Suggested wall thickness for thermosetting materials[11].

Wall thickness can also be calculated using the formulas in Table 3-3.
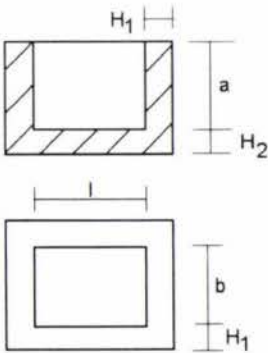
Simplified drawing	Equation		
	Wall thickness	$H_1 =$ $(C*P*a^4/(E*\delta))^{1/2}$	$H_1 =$ $(p*l/(2*[\sigma]))^{1/2}$
	Bottom thickness	$H_2 =$ $(C'*P*b^4/(E*\delta))^{1/2}$	$H_2 =$ $(p*l/(2*[\sigma]))^{1/2}$

Table 3-3 The calculation of minimum wall thickness[3].

Where:

H_1 is the wall thickness (cm)

H_2 is the bottom thickness (cm)

C=l/a, C=l/b' is constant

P is injection pressure: $(250\sim1000) \times 10^5$ Pa

$[\sigma]$ is stress (Pa)

E is the module of elasticity (steel: 2.1×10^{11} pa)

δ is the allowance of plastic

l is the length of cavity (cm)

3.2.2. Ribs

The function of ribs is to increase the rigidity and strength of a moulded piece without increasing the wall thickness. The proper use of ribs usually will prevent warpage during cooling, so the shape will be maintained.

Basically, a rib should incorporate drafts and fillets. The base of a rib may be tapered on the cross section for easy ejection from the mould. A fillet should be used to prevent the stress concentration and provide additional strength. Figure 3-2 illustrates the structure of a rib. The suggested dimensions of ribs in part design are:

$H < 3*W$

$R = W / 8$

$W_1 = 0.5\sim0.7$ W

$L = >3*W$
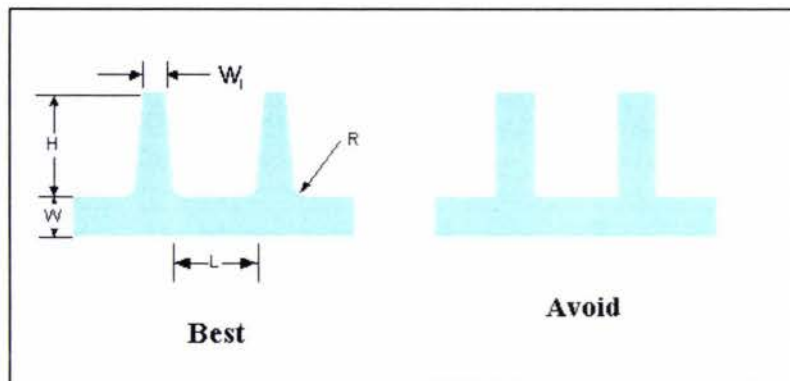
Figure 3-2 A typical rid and radii design.

3.2.3. Radii and fillet

Fillets can reduce the stress concentration, produce a moulding with greater structure strength, provide streamlined flow paths for molten polymer in filling the mould and permit easier ejection of parts from the mould. Figure 3-3 shows the recommended dimensions of a fillet.
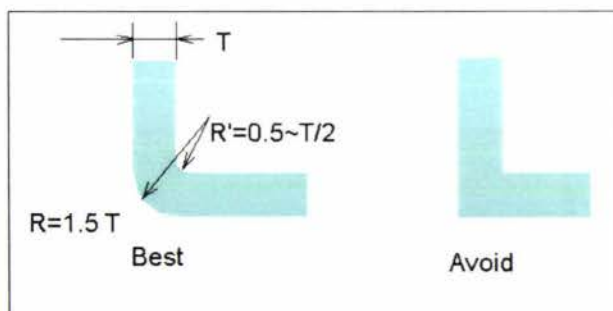
Figure 3-3 A typical fillet and radii design.

3.2.4. Draft

The main function of a draft is that the piece can be easily removed from the mould. Draft (Fig. 3-4) can be provided inside and outside. If the moulded piece is straight-sided or vertical, it is difficult to release the piece from the mould. If the taper or draft with a high polished surface is used to replace the vertical wall, the problem can be avoided.
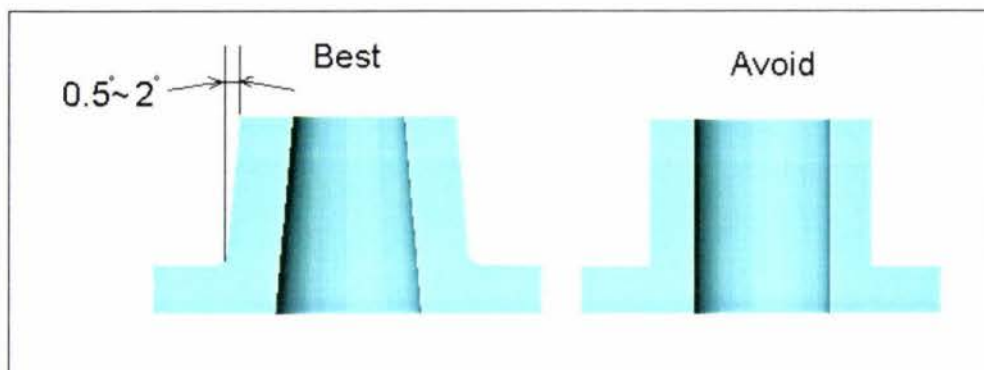
Figure 3-4 A draft design.

3.2.5. Clamping force

The minimum clamping force is derived from the reactive force of the cavity, which results from the projected area of all cavities and runners and the maximum cavity pressure [22]:

$$F = A*P$$

F: is the reactive force

A: the projected areas of cavities and runners system

P: the cavity pressure

Depending on the part designed and the material, faulty operation can increase the injection pressure. Therefore, the maximum clamping force is:

$$F_{max} = A_{max} * P_{inj} < F_{clamp}$$

3.2.6. Runner system

The runner system is to accommodate and guide the molten plastic material flows from the barrel into the mould cavity. A standard runner system usually consists of several components such as a sprue, runners and gates.

To determine the runner size and gate size of a balanced runner system relates to the surface area and the weight of the part. When impressions are not identical, to determine runner size is more difficult. The rule design of a balancing runner system and the suggested dimensions are detailed in Appendix D. Computer-aided engineering software for mould design and analysis are available, which provide a great tool for mould design. Details of using CAE package in this project are presented in later chapters.

3.2.7. Sprue puller

A sprue puller must be provided for pulling the sprue stem out of the sprue bushing and to break it away from the nozzle when the mould opens. Typical types of sprue pullers are shown in Fig. 3-2.
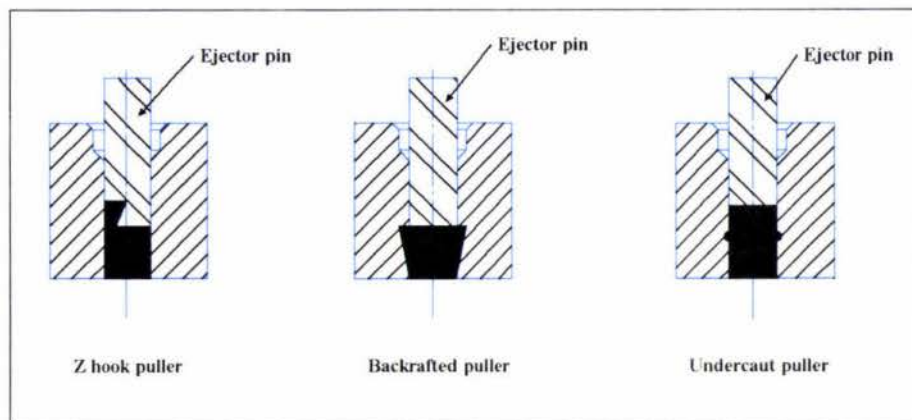
Figure 3-5 The typical sprue puller.

3.2.8. Ejector pin

To demould a moulding, ejector pins are the most frequently employed elements. They are on the market as standards in many variations and dimensions. There are two basic types of ejector pins according to their intended use (Fig. 3-3).
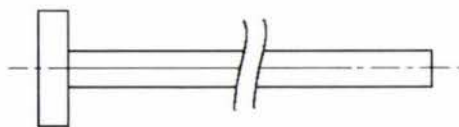
a) Ejector pin with cylinderical head
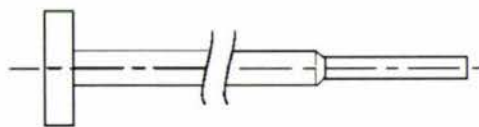
b) Shoulder type ejector pin

1.   Straight cylindrical pins are the most common for all ejection forces. The cylindrical head reduces the hazard of being pressed into the ejector plate.

Figure 3-6 The two common ejection pins.

21

2.   Shoulder-types (stepped ejector) pins are employed if only a small area of the moulding is available for ejection and little force is needed.

3.2.9. Vent location and size

Proper air or gas escapement in a closed mould is absolutely essential. A good escapement prevents short–shot, burned material, damaged mould and poor ejection. The vent should be located along the parting plane. It is not desirable to locate a vent in the middle of the part. If there is an air trap in the middle of the part, air can be removed only through a vent to an ejection pin in that location. And larger channels evacuate the air or gas into the atmosphere. Free escape of the air or gas from the mould is essential.

Normally, the vent size is from 0.0125 to 0.075 mm deep and from 1.5 to 12.7 wide.

3.2.10. Cooling system

Maintenance of the correct mould temperature greatly influences the injection speed and the distortion of the moulded part. The aim is to achieve a cooling time shorter than the recovery time. The whole cooling system should be divided into several separate cooling circuits if possible because the temperature of individual spots is easier to control.

The temperature of the cooling medium can be thermostatically regulated. This depends on the type of material to be injection moulded. Water is the commonly used cooling medium. However, if temperatures exceeding 90 C are required, oil cooling is preferred [3].

4. THE DEVELOPMENT OF AN INTEGRATED CAD/CAM/CAE SYSTEM FOR MOULD DESIGN AND MANUFACTURING

4.1. Introduction

An integrated CAD/CAM/CAE system was established through this project. It was hoped that this system will provide an opportunity for students to get familiar with the application of computer-aided technologies in mould design and manufacturing.

4.2. Mould design using 3D CAD

The 3D models of the three souvenirs discussed in Chapter 2 were built in a CAD system called Solidworks, based on requirements such as:

1. The structure of the die set.

2. The estimated cost.

3. The available CNC facilities.

4. The capabilities of the injection moulding machine in ITE.

5. The available cutting tools.

6. The available plastic material in ITE.

Further study and analysis were carried out on the three designs. Advantages and problems of each of the designs were identified, which are listed below.

- Design A (Card holder): The problem is that the three-plate mould cost is very high and the facilities of the workshop in ITE can not meet the requirements.

- Design B (3D picture frame): Two problems were identified in this design. The first one was the difficulty to machine the words "Massey 2000" on the curved surface using the existing CNC machine. The other problem was that it was difficult to use the existing CNC to machine the 3D cavity as the tool is a 2-½ CNC facility.

- Design C (2D picture frame): Based on Design B, the 3D curved surface was changed to a flat surface, this design is simple and the corresponding mould is able to be produced using ITE's facility at a low cost.

According to the analysis, Design C was chosen as the souvenir. Based on the 3D model of Design C, Mould design was carried out within the same CAD system. The mould is a two-plate mould with runner, cooling, ejecting systems. As the purpose of the mould design is to develop a hand-on course for students to learn how to use computer techniques to achieve the optimized results in mould design and manufacturing, using CAE system is a compulsory phase.

C-mold was chosen to be used in this project. In order to use C-mold to simulate the injection process, the 3D model of the mould has to be built first. This project did an investigation on how to effectively create the 3-D model of a mould for C-mold. The recommended procedure is outlined below.

1. Create a new 3D part file for runner system independently.

2. Keep the original 3D part files as they are.

3. Create an assembly model containing all the components and then save the assembly file.

4. Click the Insert, Feature, New to insert a new part.

5. Click a face or plane on one of the components.

6. Click Insert, Feature, Join; the join dialog box appears.

7. Select those components in the "Feature Manager"

8. Click OK, and save the joined file.

The advantage of designing the 3D model of the mould in this way is that all files will share the data of the injection model. If one data is changed, other files will automatically be updated when the file is opened. This is a suitable method to create a model for simulation of the injection environment and prevent data loss. Fig.4-1 shows the joined model created using the above mentioned method. It is ready for using in C-mold to simulate the injection moulding process.
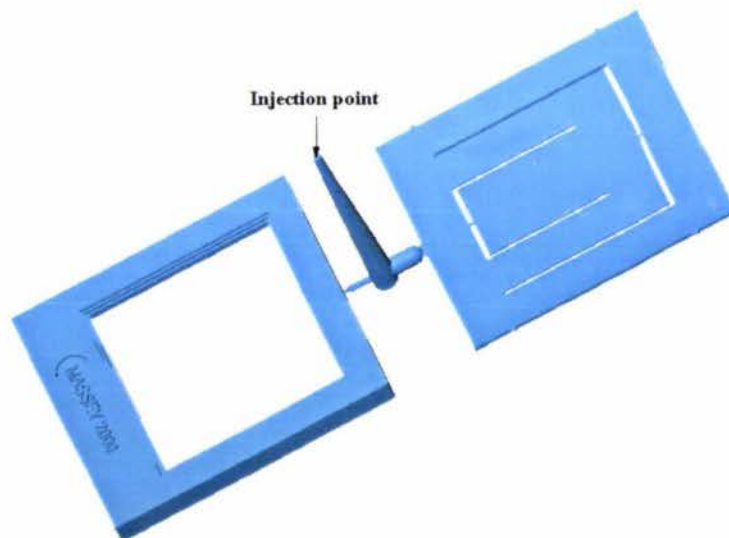

Figure 4-1 The joined model for simulating injection moulding process.

4.3. Injection moulding process simulation using CAE

CAE provides more powerful functions in the last decade. In mould design, several CAE packages are available in the market such as C-mold, Moldflow, etc. In this project, C-mold was used to simulate the injection process, which aids the designer to identify the reasonable unbalanced runner system, using the results of the analysis to identify where the problems of a mould design are likely to occur and then the mould may need redesigning. The analysis incorporates the prediction of the molten plastic flow into impressions, the distribution of temperature, the location of wildlines, air trap and shrinkage, shear stress, etc. All the results can be presented on the graphical plot; the designer can clearly understand the injection process.

The picture frame has two components, the frame and the supporting plate. The frame has a uniform thickness, but the supporter has variously thickness more than 50 %. A better way to produce the frame and the supporter is to use two die sets. However, the cost will be increased significant.

To avoid the unbalanced flow, the runner system using different runner size, gate size and runner length is necessary to reach the desirable injection condition. In this project the gate size and the length of runner system were assumed unchanged factors. Only the cross-sectional dimensions of the runner are unknown parameters.

The selection of polymer for this project had no any special requirements such as chemical resistant, oil resistant or ultraviolet. The polymer must be cheap so that the cost of the picture frame can be kept low. Recycled polyethylene was used in making the picture frame.

Base on the 3D model showed in Figure 4 – 1, simulations using C-mold under different injection conditions were made. A few cases are presented below to illustrate the simulation process and the possible results. The results of the analysis can assist us to find out the suitable vent location, heat distribution for arranging the cooling system, the location of sinkage, etc. Based on the results the mould designers can make modifications and changes accordingly to improve the quality of the mould.

Result of study case 1
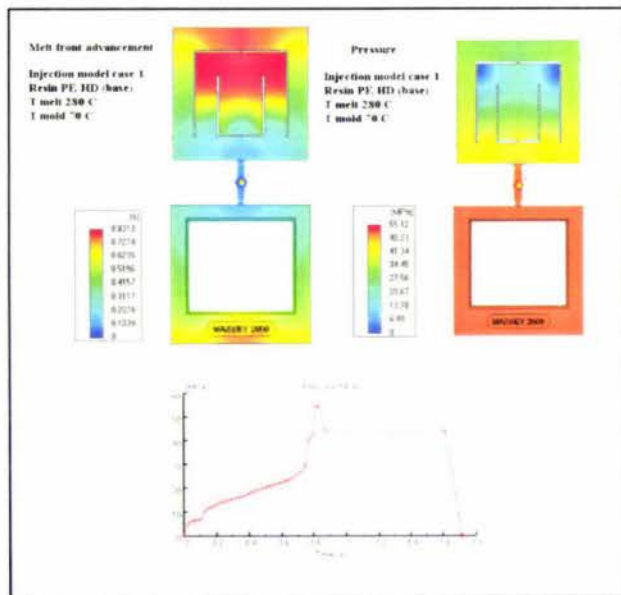
According to the Figure 4 –2, the frame was completely filled with polymer before the supporting plate. The injection pressure increased suddenly to overcome the unbalanced filling process. The unbalance flow needed a high injection pressure and a back pressure to finish the injection cycle.

Figure 4-2 The results of case 1.

Result of study case 2

The mould temperature was adjusted from 70 C to 85 C, the pressure trace changed slightly, but the results were similar to Case 1.
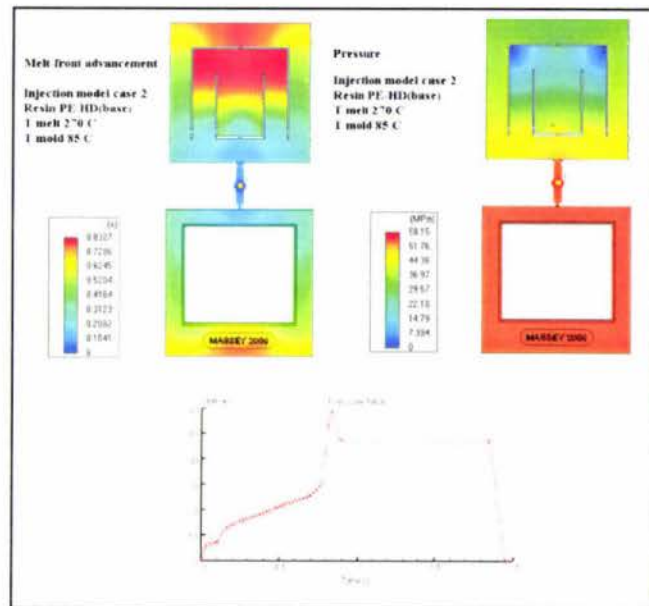

Figure 4-3 The results of case 2.

Result of study case 3

Based on the results of case 2, the designer tried to decrease the flow rate of the frame. The change of the runner diameter can change the flow rate. The runner diameter of the frame was reduced from 6 mm to 2 mm. The result is shown in Fig 4-4. The filling process was different for case 1 and 2. The support plate was completely filled before the frame. The injection pressure was reduced from 65 Mpa to 45 Mpa. The results of case 3 use improved although the filling process still was unbalanced.

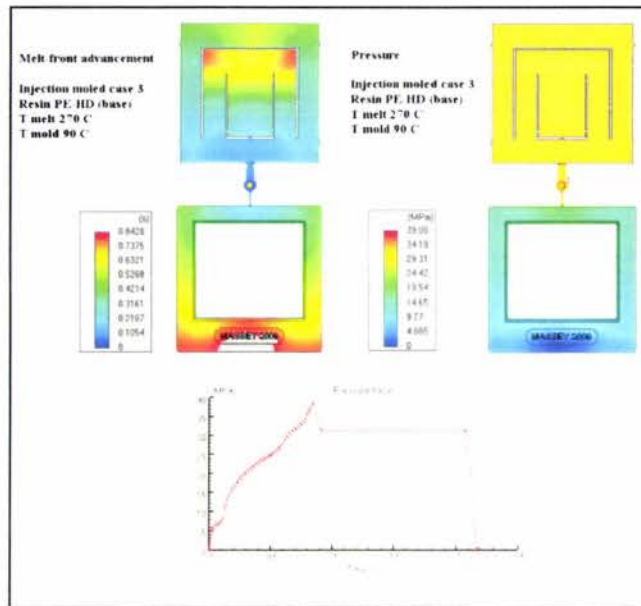**Figure 4-4 The results of case 3.**

Result of study case 4

The result of the case 3 showed that a small area at the location of "Massey 2000" on the frame side was not completely filled in the filling process. The filling process can be improved through the adjustment of the injection temperature and the mould temperature.
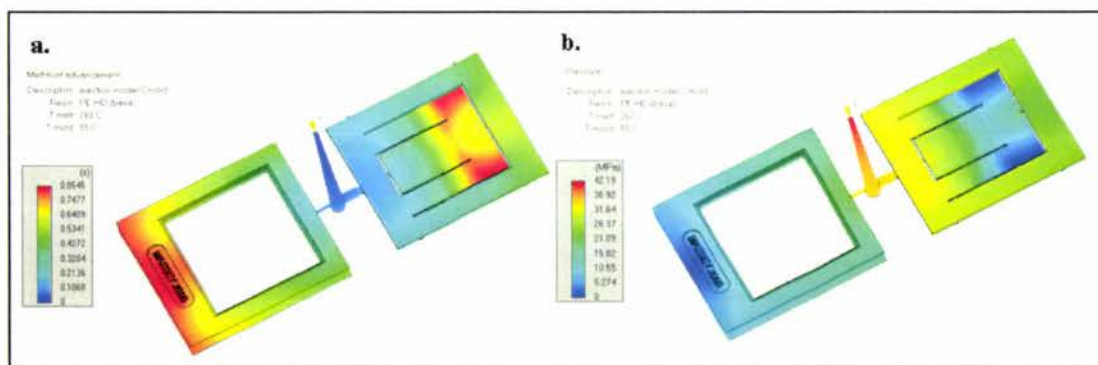

Figure 4-5 The Melt-front plot and pressure plot of the case 4.

29

The results of changing injection temperature and mould temperature are shown in figure 4-6. The various colors show the injection temperature and the pressure distribution in the filling process. The balance filling process was obtained at the reasonable injection temperature and mould temperature.
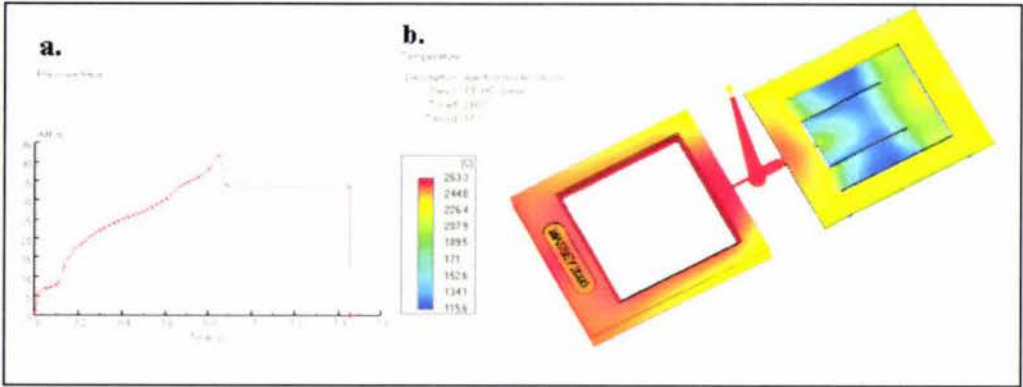
Figure 4-6 The pressure trace and temperature plot.

The pressure trace of case 4 is the best in the analysis. It has the lowest injection pressure and a fairly good filling process as shown in Figure 4-7. Injection temperature is also low compared with the former cases. However, there might be a warp in the support plate at the blue area. This is caused by the shrinkage of the thinnest wall thickness.

Figure 4-7 The time-to-cooling plot and orientation plot.

30

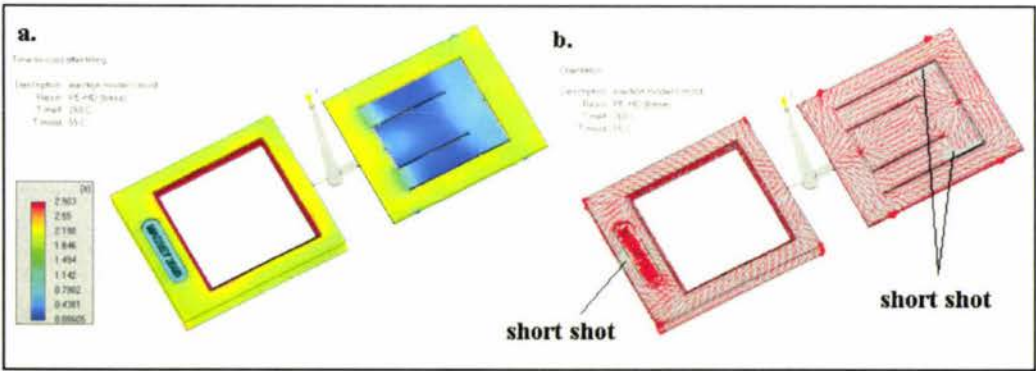A cooling system was designed in the mould. The result of the "Time-to-cooling after filling" plot is for improving the design of cooling system and the injection cycle. The orientation plot indicates (Figure 4-8 b.) that the location of short shot might occur at the end two corners of the support plate.
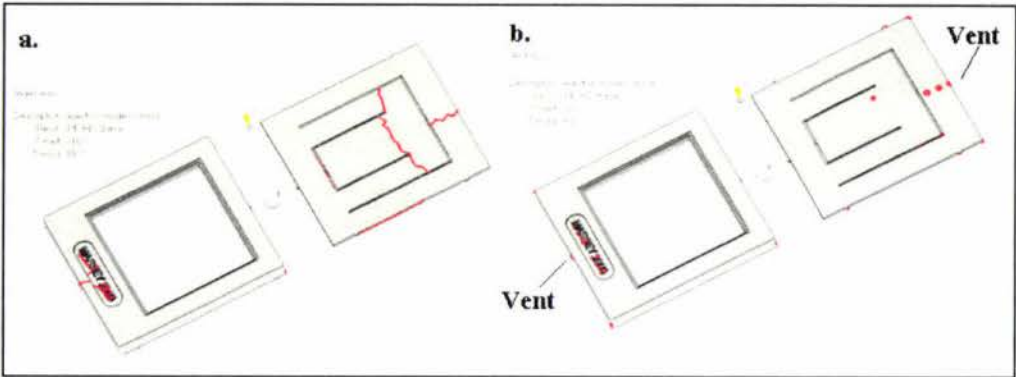
Figure 4-8 The weld line plot and air traps plot.

Figure 6-9 shows the location of the possible weld lines and air traps. The location of

weld lines indicates the potential weak areas in the part. Air traps can cause short shots and affect product quality. According to the result of the air traps, vent locations can be designed to improve the filling process and the product quality as shown in Figure 4-8 b.
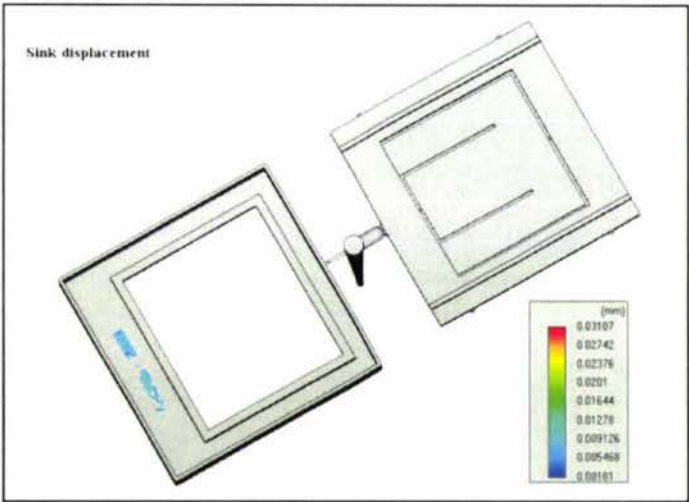

Figure 4-9 The sink plot of case 4.

Shrinkage also needs to be considered. CAE software can predicate the possible result. Figure 4-9 shows the sink occurred at the back of the frame under the injection conditions used in case 4.

Table 4-1 shows the simulation results using C-mold under different sizes of runners, the injection temperatures and mould temperatures.

Study case	Runner diameter $(D_f:D_s)$ mm	Injection temp. C	Mould temp. C	Result
1	6:6	280	70	High injection pressure, back pressure Unbalance filling process
2	6:6	280	85	High injection pressure, back pressure Unbalance filling process
3	2:6	270	90	Low injection pressure, back pressure Unbalanced filling process
4	2:6	265	55	Low injection pressure, back pressure balanced filling process

Table 4-1 The results of the study cases.

4.4. Design the mould for the picture frame

Mould design very much depends on the experience of the designer. A reasonable mould design incorporates the cavity layout and runner system design. In the runner system, the designer should consider the characteristics of polymers, weight of the product, runner layout, runner size and the location of the gate. Apart from these, to select a suitable material is another important aspect. Selection should be based on the product requirements such as the stress, product structures, shrinkage requirement, cost, etc. An important aspect of characters of the material is the shrinkage from the liquid state to solid state. The shrinkage of thermoplastic may be up to 5% for some plastics, which

causes a significant problem in precision requirements. Therefore, consideration should be given to the characters of the material when designing a mould.

Balanced runner systems are commonly used in mould design. It has the same pressure drop, equal length of runners and same type of gate. And determination of the runner size and gate size can be determined through theoretical calculation or reference to the handbook of plastic materials. However, the problem in this mould design in the project was to make an unbalanced runner system into a balanced system.

The method of solving the moulding problem is to use CAE package because the Conventional mould design relies heavily on trial and error, which can be time consuming and expensive because each mould change requires more time on machining equipment and injection mould machine. By using CAD and CAE technology, a great deal of the guess work and trial are taken out of the mould design process; thus CAD and CAE can reduce the development time from a concept.

In this project, a balanced runner is not possible, The runner system should be artificially balanced. Balanced flow into the cavities is a prerequisite for a quality part. Therefore, the balance flow can be achieved by changing the runner size, length and gate dimension.

- Changing runner diameter

If the runner diameter is too small, it may cause the runner to freeze prematurely, causing a short shot. It is important that non-standard runner diameters will increase mould manufacturing and maintenance costs.

- Changing runner length

Increasing the runner length will result in higher flow resistance; therefore, the pressure drop will be higher.

- Changing the gate dimension

Changing the gate dimension may give a seemingly balanced filling. However, it affects

the gate freeze-off time greatly, which is detrimental to part uniformity.

Based on the 3D model of the picture frame, the CAD model of the mould for the frame and the supporting plate was created in Solidworks which is shown in Fig 4-10.
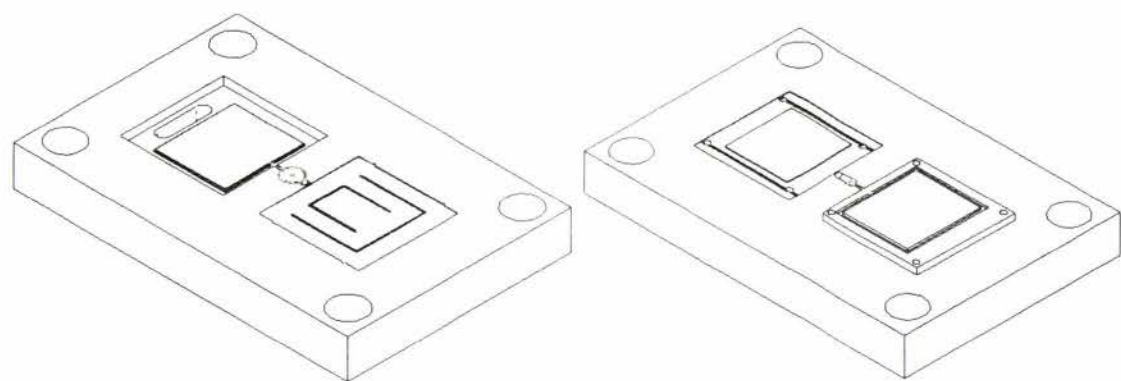

Figure 4-10 The 3D model of the picture frame.

Before actually machining the die set, analysis and cutting process simulation were carried out. Following are the detailed drawings of the mould, specifications of the injection moulding machine, cutting tools, injection parameters, and the operation planning sheets.

1. Detailed drawings:

The table 4-2 shows all the names of the drawing for this project. Each drawing has detailed dimensions of a part. All the drawings are shown in the end of the thesis.

Part Name	Drawing no.	Quantity
Frame	PF1	1
Supporter	PF02	1
A plate	PF03	1
B plate	PF04	1

Inserted part	PF11	1
Location ring	PF09	1
Sprue bush	PF10	1
Ejector 5 mm dia x 97.5	PF14	4
Ejector pin 5 mm dia x 105	PF15	4
Sprue puller 10 mm dia x 95	PF13	1
Clamping plate A	PF05	1
Clamping plate B	PF06	1
Ejector plate	PF07	1
Ejector bar	PF08	1
Supporting pillar	PF12	2

Table 4-2 A list of drawings.

2. The specifications of the injection mould machine:

Model no:	TTI-330/100
Injection capacity:	6.3~180 g
Injection pressure (theoretical):	1950 kgf/cm^2
Injection speed:	70mm/sec
Clamping force:	100 ton
Clamping stroke:	300 mm
Distance between tie bar:	355 x 355 mm
Platen size:	335 x 335 mm
Mould thickness range:	10 ~ 310 mm
Ejector stroke:	75 mm

Table 4-3 The specifications of the injection mould machine in ITE[7].

3. Injection parameters of high density polyethylene (HDPE)

Injection pressure:	10000 ~ 15000 psi
Melt temperature:	180 ~ 280 C
Mould temperature:	20 ~ 90 C
Typical runner size:	2~10 mm
Standard gate size: W x H x L	2 x 0.4
Fan gate size: Wide x Thickness	6.5 x 0.7

Table 4-4 A list of character of the high-density polyethylene[5].

4. Available cutting tools in ITE workshop

To confirm the available cutting tools is very important. The manufacturing engineer can then use these tools to prepare the machining process and plan the operation sequence. Table 4-5 illustrates the available cutting tools in the ITE workshop.

Miller		Driller		Reamer	
Dia. in mm	Description	Dia. in mm	Description	Dia. In mm	Description
2	Carbide	4	HSS	5	HSS
2	Carbide/ BN	4.7	HSS	10	HSS
3	Carbide	5	HSS	13	HSS
6	Carbide/ BN	7	HSS	16	HSS
6	Carbide/ insert	8	HSS		
10	Carbide/ insert	9.7	HSS		
12	Carbide/ insert	12.5	HSS		
1	HSS	15.8	HSS		
2~20	HSS				

Table 4-5 A list of available cutting tools in ITE

5. Operation planning sheet

A typical operation sheet lists operations that are done in sequence on a single machine, Some sheets may cover all the operations for a given part for a group of machines in manned cell. Some Operation sheets only list the required operations and the machine to be used. Speeds and feeds may be left to the discretion of the operator, when a skilled worker and small quantities are involved. However, an operation sheet commonly details the tools, speeds, feed rates and the allowed time for completing each operation, especially in a CNC machine. The operation created for machining the mould of the picture frame is illustrated in Fig 4-11.

					Design or Test		Date	
					Worker		Date	
					Part Name		File Name	
Drawing					Material		CNC Program	Remark / Test
					Size			
					Clamp no			
					Cooling Liquid			
					No of Part			
					Sheet			

Work Seq.	Operation Description	Machine	Cutter Tool	Spindle Speed	Feed Rate	Cut Depth	Operation time	
							Feed Times	Prep. time

Figure 4-11 A typical operation sheet for this project.

4.5. Mould manufacturing using CAM

CAMworks is a new CAM software to ITE, Massey University. It arrived in July 1999. This CAM software was new to everyone in ITE. The first main target is to integrate CAMworks with the CNC milling machine in ITE. The existing CNC machine in the workshop is a Bridgeport CNC milling machine with a Heidenhain Controller. The memory of the controller is very limited. It can not accept a NC program with more than 1000 blocks. As the existing Heidendain controller in the workshop is pretty old, the NC program format is not completely compatible with the output of the CAMworks. To realize the integration, two main problems must be solved. One is the size of the NC program. The other is the compatibility of the NC program format.

4.5.1. Control the size of a NC program

An NC program is a program which consists of cutter locations when machining a surface. Changing the size of the cutter, using different cutting patterns such as "Zigzag", "Pocket in" and "Pocket out" will change the NC program size. Therefore, defining the cutter size as large as possible and choosing the suitable cutting patterns can assist to control the size of the NC program and help to meet the memory requirement of the Heidendain 145 controller in the workshop. However, this is only useful to small NC programs. For large NC programs, i.e. the program size is much larger than 1000 blocks, the only suggestion is to break the program into several sub-programs. To break a program, the suggested breaking point should be based on the cutting layers. In this way, it gives clearer cutting information and reduce unnecessary errors. An example is shown in Fig.4-12.
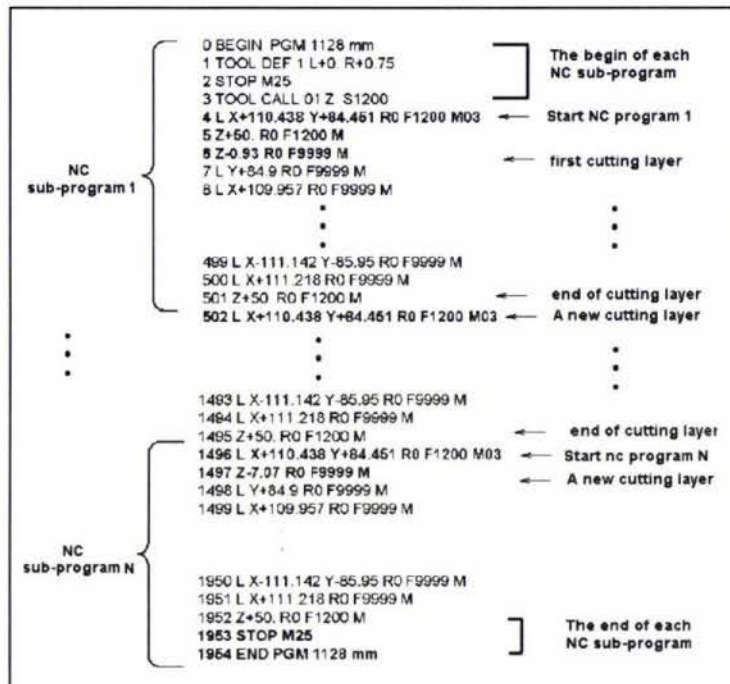
38

Figure 4-12 An example of subprograms.

4.5.2. Editing an NC program

The methods of editing an NC program are manual or using the NC editor program. Manual editing is a way which requires an experienced NC programmer to check the NC program line by line. The NC programmer needs to spend a lot of time checking and modifying the lines. In some cases, the edited NC programs need to dry run to find out problems. Manual editing is a time consuming method. Another way to edit the NC program is to use on NC editor program. Some of NC editors can "Dry Run" an NC program on the PC and rapidly correct the errors within the NC program. Based on the experience of this project, using NC editor to correct NC programs is the recommended method.

4.5.3. Toolpath simulation using CAMworks

In this project, machining parts included mould plates, ejector pin, pin plate, two clamping plates, insert part etc. All the components were designed using 3D CAD Solidworks and then passed into CAMworks system. The machining processes were defined in CAMworks and the toolpath for each component was simulated. The corresponding NC programs were then generated by CAMworks and finally sent to CNC milling machine in the workshop. The whole die set was produced in ITE's workshop. In order to demonstrate the simulation process, the insert part as shown in Fig. 4-13 is chosen as an example in this section.
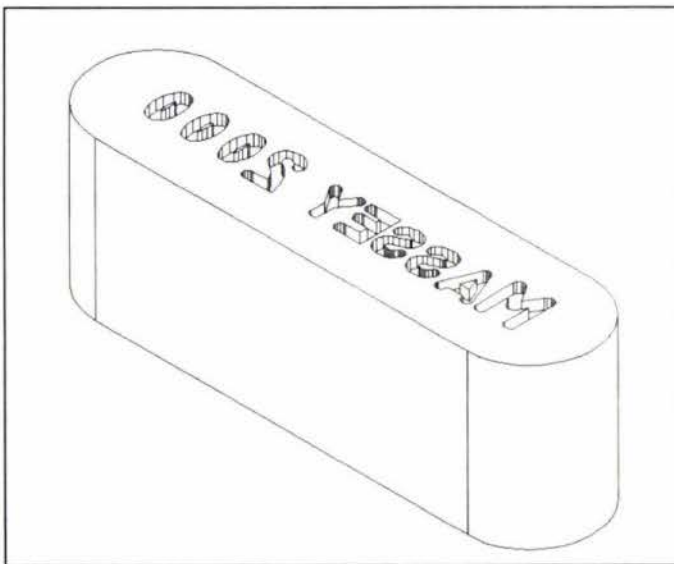
Figure 4-13 The 3D model of the insert part.

The simulation procedure of the insert part is detailed in Appendix B, which includes application of Automatic Feature Recognition (AFR), Manual Feature Recognition (MFR) and Technical database (TB). The simulation procedure of the insert part consists of the following main steps.

1. Create the model in Solidworks.

2. Define the parameters of CNC machine to be used.

3. Run the AFR to retrieve machining features.

4. Insert the machining operations if necessary.

5. Generate the toolpath.

6. Adjust machining parameters such as cutter diameter, feed rate, etc. (if necessary)

7. Regenerate the toolpath after changing machining parameters.

8. Simulate the machining process.

9. Generate the NC program.

The toolpath of the insert part is illustrated in Figure 4-14. During the simulation process, errors and mistakes can be found out. The simulation process allows the designer to change cutter size, machining sequence, etc. until the optimized toolpath obtained. The simulation of the insert part is shown in Figure 4-15. Each of the machining operations is shown in the manager tree. The operations can be easily re-arrange in the manager tree area.
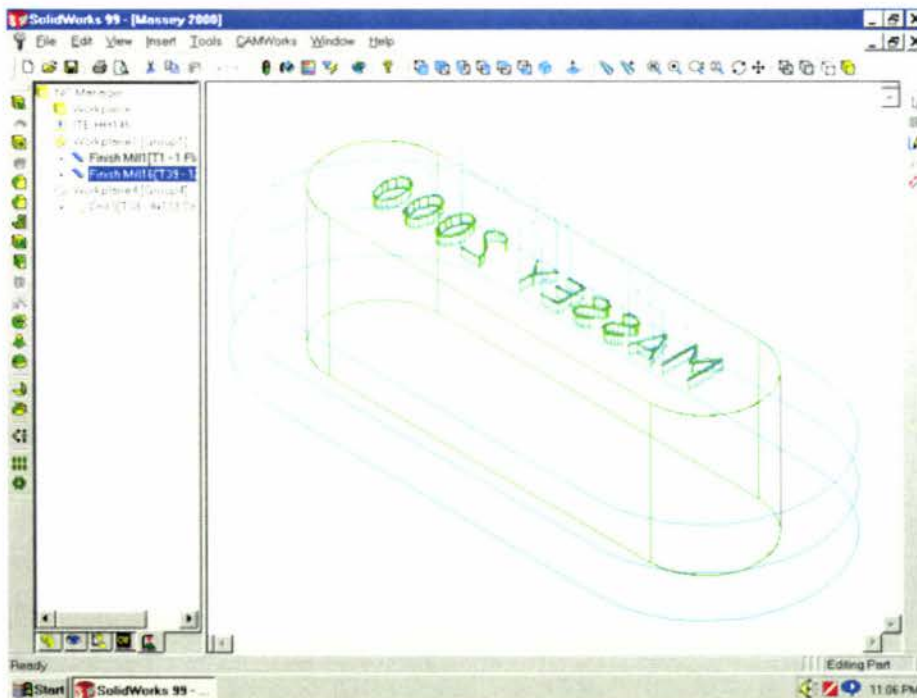
Figure 4-14 The toolpaths of the insert part.
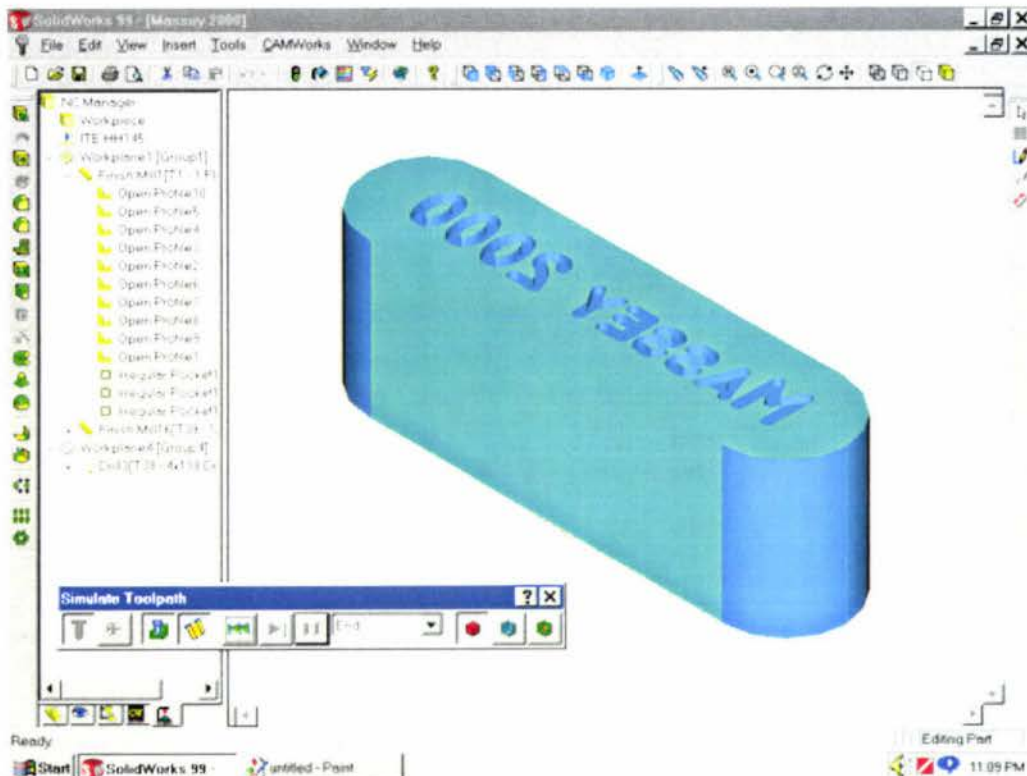
Figure 4-15 The simulation of the insert part.

4.5.4. Generate NC programs and machining the mould for the picture frame

Once the simulation stage is gone through and the designer is satisfied with the machining parameters, cutter size etc, the generating of the NC program is straight forward in CAMworks. By clicking on the post-process a NC program will be outputted from CAMworks. Details are given in appendix B. The following NC programs are the output from CAMworks for machining the insert part.

Program 1: Machining the "Massey 2000"

```
1 TOOL DEF 01 L0 R.500
```

2 (1 MM 2 FLUTE CARB E.M.)

3 STOP M25

4 TOOL CALL 1 Z S2500

5 Z+10.000 R0 F8466 M

6 L X+38.074 Y+4.222 R0 F8466 M03

7 Z+5.000 R0 F8466 M

8 **Z-.250** R0 F100 M

9 L Y+8.001 R0 F100 M

    .

    .

433 Z+10.000 R0 F8466 M

434 STOP M25

435 STOP M02

Program 2: machining edges

1 TOOL DEF 38 L0 R2.000

    .

    .

24 Z-1.500 R0 F150 M

25 L X+40.930 R0 F350 M

26 CC X+40.930 Y+5.930

27 C X+54.860 Y+5.930 DR+ R0 F350 M

28 CC

29 C X+40.930 Y+19.860 DR+ R0 F350 M

30 L X+5.930 R0 F350 M

    .

    .

163 Z+10.000 R0 F8466 M

164 STOP M25

165 STOP M02

According to the features of the components and the simulation results from CAE and CAM, the preparation to machine the component on the CNC machine tool is an important stage, which includes the preparation of the NC program, the cutting tools, clamping devices, the operation sheet etc. For an inexperienced person, to carry out this work without mistakes will be a difficult task. In order to help an inexperienced person such as students to go through this stage smoothly, the main steps are outlined below and a machining process using the NC program generated by CAMworks is shown in Fig. 4-16.

The brief of a machining process:

- Download the NC program form a PC to the CNC milling machine.
- Setup the material and the cutting tools for machining.
- Set the machining datum.
- Off set the machining datum.
- Dry run the NC program
- Offset the machining datum back.
- Machine the material to produce the component.

Figure 4-16 One of the machining processes using NC program.

5. MANUFACTURING OF THE PICTURE FRAME THROUGH INJECTION MOULDING

5.1. Introduction

This chapter presents the manufacture of the picture frame using the injection moulding machine in ITE. The injection moulding condition was first set according to the CAE analysis result, and then modified to obtain the optimized condition to produce the quality picture frame.

5.2. Injection mould process of the picture frame

To produce the plastic picture frame on the injection moulding machine in ITE's workshop, the first task was to set the correct injection parameters such as pressure, temperature etc. Base on the work done in chapter 4, the initial set up of the injection condition was the recommended injection parameters by the CAE process. However, as there were differences between the simulation specifications in CAE process and the actual situation such as the materials used etc, certain modifications were needed to get rid of short shot, improve cooling, etc. After a few adjustments, a desired injection moulding condition was achieved. This was done within a fairly short time. It was obvious that without CAE, the adjustment would take much longer time. A comparison of the values of parameters recommended by CAE process (C-mold) and the actual condition was presented in Table 5-1.

	C-mold	Actual injection condition
Melt temperature	260 C	280 C
Injection pressure	45 Mpa	55 Mpa
Mould temperature	55 C	–
Clamping force	–	60 Mpa
Injection time	0.9 s	5 s
Cooling time	2.9 s	2.5 s
Material	Polyethylene	Recycle
Supplier	Unknown	–

Table 5-1 The comparison of injection conditions.

CAE analysis can also predicte problems which might happen during the injection moulding process. The CAE analysis of the picture frame using C-mold did indicate the possible problems such as air traps, weldlines, sink marks, etc. The actual injection moulding process of the picture frame did prove the prediction of the CAE analysis. Table 5-2 presents both the predictions and the actual situation.

	Predicted problems	Observed problems in the product.
Air traps	The location as Fig 6-7 b	–
Weld line	The location as Fig 6-7 a	Same location, weakness part
Sink	At the back of "Massey 2000"	Same location, other is ribs in the supporting plate
Short shot	At the end of the melt-front	Sometimes have short shot at same injection parameters.
Warpage	At the thinness area	Same location.

Table 5-2 The Comparison of C-mold results and observed problems.

The final product was moulded using the injection moulding machine in the workshop, which is shown in Figure 5-1. The product is shown in Figure 5-13. The picture frame is an extremely simple product, but it shows that if the design and manufacturing are not carried out using CAE package to analyze the injection process and the correct manufacturing procedures, even the simple mould may result in poor products. CAE is a great tool which can help the designer to improve mould design and predict problems in the injection moulding process.

Figure 5-1 The picture frame mounts on the B plate at the moving platen.

6. CONCLUSION

This project has successfully developed an integrated CAD/CAM/CAE and CNC system for mould design and manufacturing at the Institute of Technology and Engineering. Through the picture frame, two labs and the tutorial materials were developed. This integrated system and the lab courses will provide the undergraduates with a hands-on opportunity in mould design and manufacturing. Students will gain experience from concept design to final product.

6.1. Highlights of Computer-Aided Technologies

Computer-Aided Software (CAS) has rapidly changed in various professional fields. It develops powerful functions and friendly user interface and assists designers to produce a design faster and better than before. It also provides professional design, analysis, and simulation tools both for education and industrial companies. CAS covers Computer-Aided Design (CAD), Computer-Aided Manufacturing (CAM) and Computer-Aided Engineering (CAE). This project employed CAD/CAM/CAE to present the educational and industrial trend.

1. Computer-Aided Design

Using Computer Aided Design is very popular in different fields of industry. Many CAD softwares are available on the market and they provide similar functions in detailing drawing, assembly, part design, converting file format, etc. They also can effectively manage the design information of a component and the associated assemblies.

2. Computer-Aided Engineering

The CAE software used in this project is C-mold. It can analyze the injection process and predict problems which occur in the injection process. The results of the analysis can be used to improve product design, mould design and injection process. For example, C-mold was used to successfully identify the runner size in this project and predict the problems in the filling process. The analyzed results accurately showed where the weld lines, sink, air traps were.

3. Computer-Aided Manufacturing

CAMworks was the CAM software used in this project. It can simulate the manufacturing process, organize operations, build tool libraries and generate NC programs. The machining process was divided into two parts. The first was to prepare the CNC programs for testing in the CNC milling machine. The second part was to create operation sheets which include the cutting tools, fixtures, chucks, spindle speeds, cutting dept, etc. The operation sheet is a guideline of the manufacturing process, which provides operation sequence and details of machining parameters for each of the operations.

6.2. Further work

This project has established the undergraduate CAD/CAM/CNC laboratory courses which have the ability to produce simple injection moulds for plastic components. However, to match with today's competitive global educational programs, further improvement is required. The first is to keep updating the CAD/CAM/CAE software. If possible, replace the existing C-mold educational license with the commercial license. This will overcome the constraints on the complex of 3D models as educational license only allows for simple 3D models which will have a STL file with less than 4000

triangles. Secondly, some of the facilities in ITE such as the HEIDENHAIN TNC 145 CNC machine need to be replaced. This machine is a 2 ½ CNC milling machine with very limited functions and a small memory size. It is too old to be integrated with any CAD/CAM software smoothly. In this decade, most of the plastic products using injection moulding techniques are very complex 3D designs. Without a good facility, the creativity of students will be restricted. With the new facilities, time spending on editing NC programs will be reduced. Unnecessary mistakes and errors will be decreased. Students will be able to spend more time on 3D design, product innovation, and producing challenging and competitive outcomes.

A TUTORIAL OF C-MOLD 3D QUICKFILL

A.1   Introduction

This tutorial used C-mold to evaluate different plastic material for analyzing the injection process. The system reveals potential design and manufacturing problems and provides immediate solution advice. Designers can obtain real engineering feedback in minutes. This tutorial is to teach students how to use C-mold through a study case, students can easily grasp the concept of the analysis process using C-mold.

A.2   Analysis process

The following procedure is to introduce the analysis process.

1. Import CAD models

C-mold accepts solid geometrical models from other CAD packages. The importing format is the STL (stereolithography) file, which can be in either ASII or binary format. The process is:

- Click the File from the toolbar
- Select the Import
- Open the STL file
- Set the unit of measure for the STL file

2. Mould configuration

If the model is without runner systems, this step can be skipped.

- Click the Design from the toolbar
- Select the Mould Configuration
- Set up the runner and gate location
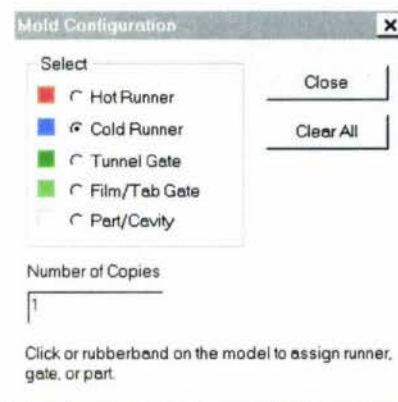
Figure A- 1 Mould configuration.

3. Set the ejection point

- Click the injection ![injection icon] to setup the injection points or select the Design in the toolbar, then select the Injection point (s).
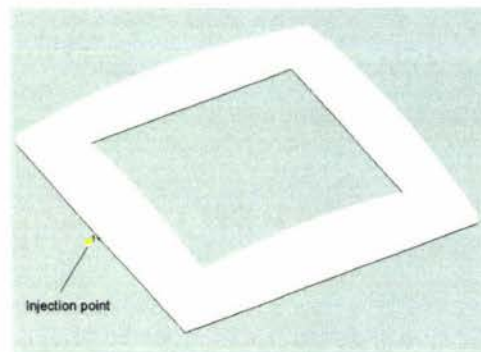
Figure A-2 The example of injection point

4. Select the resins

- Click the Select resin ![resin icon] property to choose the resin or click the Design in the toolbar, then click the Select resin.
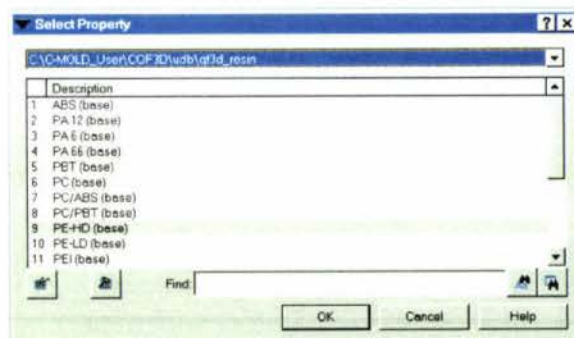
Figure A-3 The resin property.

52

5. Set the ejection condition

Click the Select process Conditions to define the process parameters or click the Design in the toolbar, then click the Select Process Conditions.
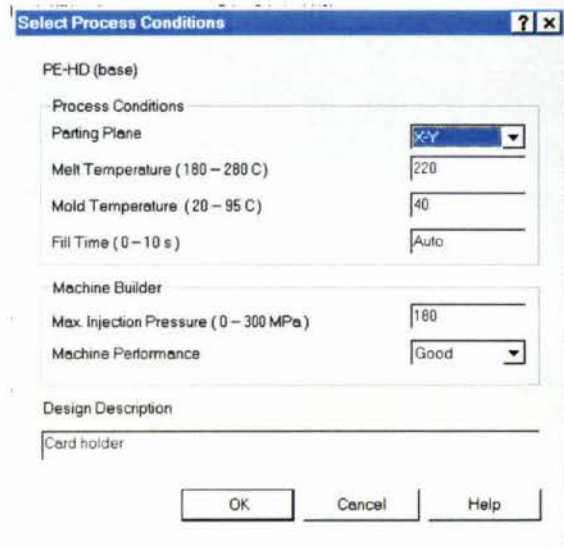
Figure A-4 The process conditions windows.

6. Run the analysis

- Finally, click the Launch Simulation to simulate the filling process.

A.3 Results of analysis

1. Melt-front advantage

The Figure A-5 shows the melt front advance as part filling. Students can easily understand the filling path, the picture and the filling time. Filling path shows the color change from blue color (start position) to red color (end position). If the part is a complex part, the suitable gate location is not easy to determine where the location can achieve the balanced flow. Using C-mold, the designer can adjust the location. Balanced flow is one of the most important mould design principles.
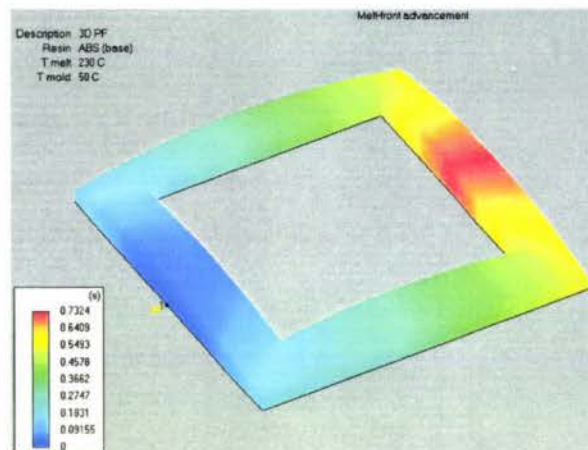
Figure A-5 The melt-front plot.

2. Pressure and pressure trace

Figure A-6 shows the pressure distribution. Pressure is the driving force that overcomes the resistance in the mould. The pressure is to push the molten polymer to fill the mould cavity. Therefore, factors to influence the injection pressure are the cavity geometry and the viscosity of the material. The cavity geometry incorporates the wall thickness, surface area, gate size, flow length, filling time, melting temperature, moulding temperature, ram speed and flow rate. All of these directly influence the viscosity of molten polymer.
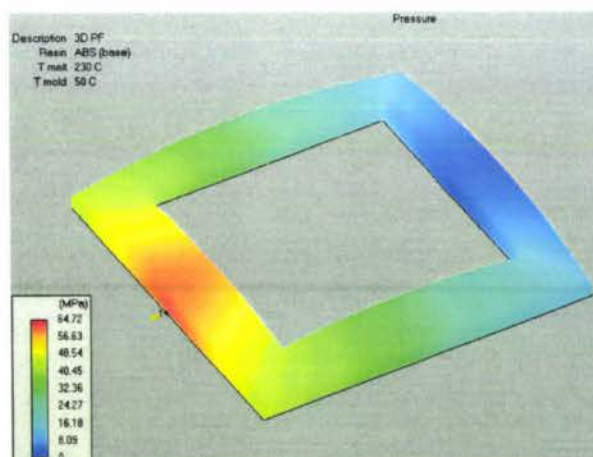
Figure A-6 The pressure plot.

The pressure trace shows the pressure change from the beginning of filling until the end. Figure A-7 shows the different pressure changes during a filling process. The following is an explanation of the pressure trace plot.

- Red line is the pressure trace which has a smooth and constant slope, but it is not a desirable trace. The hesitation occurs at the end of the filling. So the required clamping force will be increased.
- Pink line is the unbalanced filling. The slope of the pressure trace (1) will increase when a flow path is filled. So the injection flow rate is redistributed to other unfilled flow paths.
- Black line shows a desired trace, which has a smooth (4) and constant slope (4). The maximum pressure is indicated.
- Blue line shows that the melt front encounters a thin section (2), so the slope of pressure trace is suddenly increased.
- Green line shows that the hesitation occurs (3). It can either be caused by the short shot or the pressure spike at the end of the filling. This is not desirable, because the required clamping force can significantly be increased due to the pressure spike.
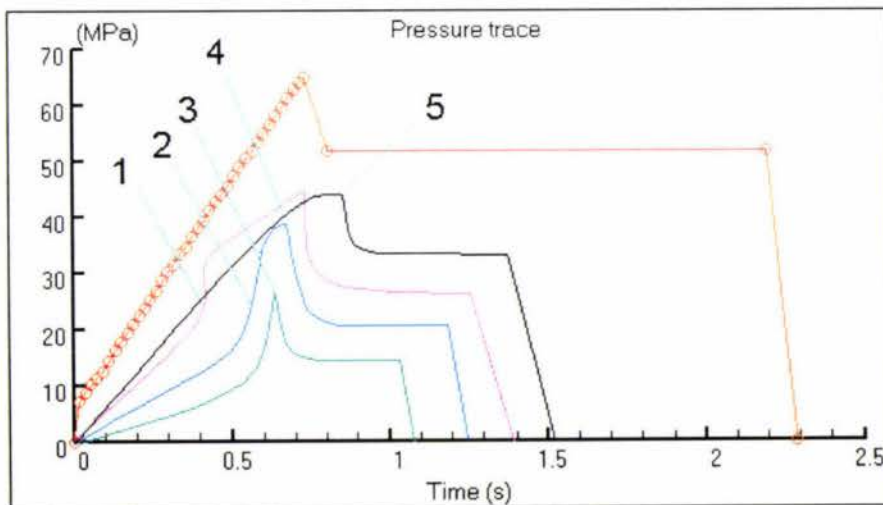
Figure A- 7 The pressure trace plot.

55

3. Temperature

The temperature plot shows the variation of temperature distribution as shown in Fig. A-8. Based on the distribution of the temperature, if the variation is too large, there is a potential that the part might be warped. The process condition can be adjusted to the location of the gate or the part thickness can be changed to prevent warpage occurrence.
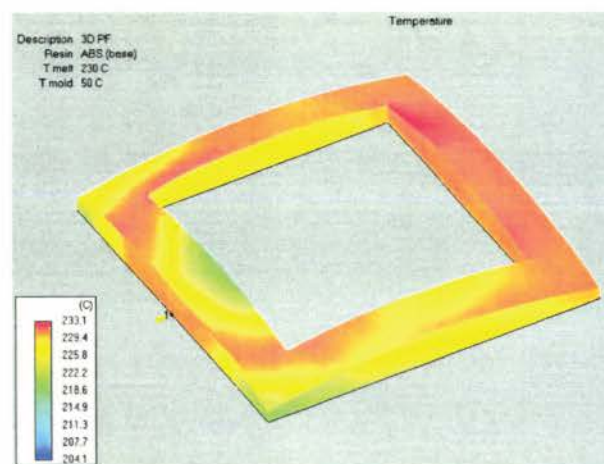
Figure A- 8 The temperature plot.

4. Cooling time

The cooling time plot as shown in Figure A-9, shows the different cooling time at different positions of a part.
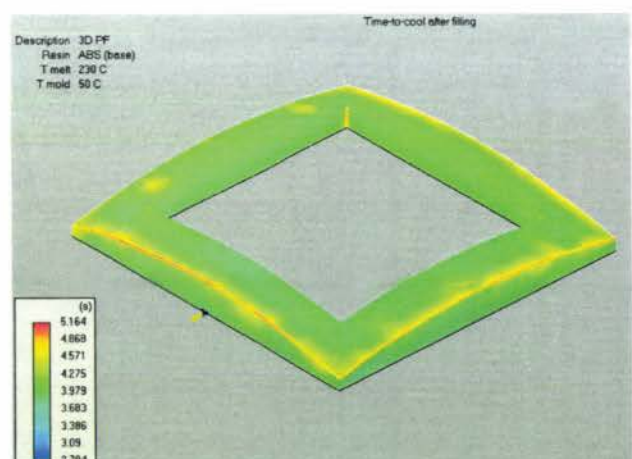
Figure A- 9 The cooling time plot.

5. Orientation

The orientation plot shows that different skin orientation can cause uneven shrinkage and part warpage as shown in Fig.A-10. Different skin orientation can also lead to poor surface quality.
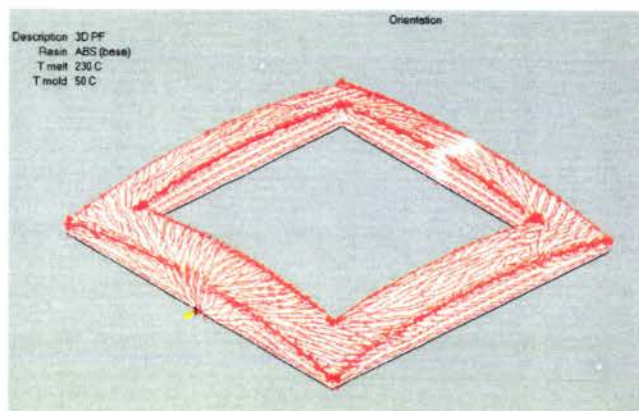
Figure A- 10 The orientation plot.

6. Weldlines

The weldlines plot shows the possible weld lines that are the weakest area of the part and the potential failure locations as shown in Fig. A-11.
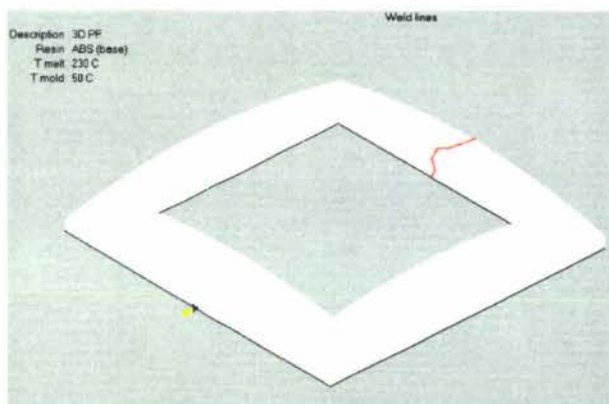
Figure A- 11 The weldlines plot.

7. Vent location

Figure A-12 shows where the air traps occur in the mould. Base on the location of air traps, a proper venting system can be designed to release the air to the atmosphere and to avoid short shots and burn marks.
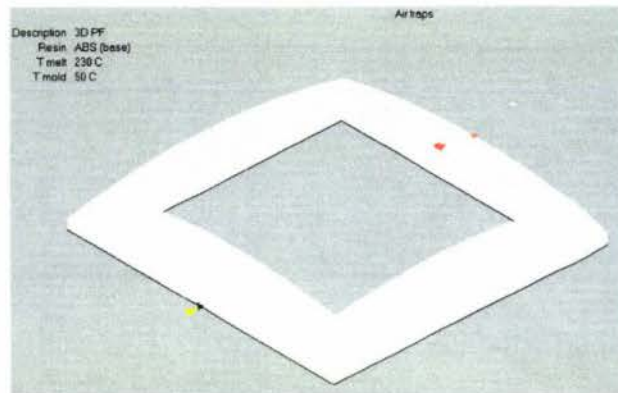
Figure A- 12 The air traps.

8. Design advice

This report provides moulding advice and suggestions for improving mould design such as part thickness, clamping force and the injection pressure.

9. Design specialization

A.4   The report of design specializations includes mould design information, machine setup, machine specifications and resin selections.

B A TUTORIAL OF CAMWORKS

B.1   Introduction

Computer aided Manufacturing is an important part of integrated mould design and Manufacturing. This tutorial includes use of technical database and CAMworks.

B.2   Create a customer tool library using technology database

As CAMworks is a third-party software built on Solidworks, it can be accessed very easily from Solidworks. The procedure to create a customer tool library is listed below:

1. In Solidworks click CAMworks, Technology

- The CAMworks 99 Technology Database should appear as shown on the right.

- Click Mill in the Mode Menu: the main Menu for milling should appear, and then set units to Metric.

2. Define CNC milling machine

- click Mill in the Mode Menu
- Define the CNC milling machine. The definition includes the Machine Name, Controller Name, Maximum spindle speed; Maximum feed rate, etc.
- Click the Close button to save the information.

3. Click Tools library and then select the Tools Crib in Tool menu.
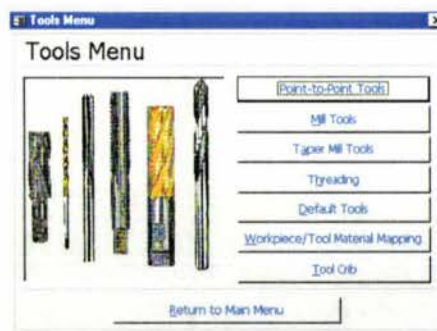
4. Create a customer tool crib

- Enter the Name of the Tool Crib.
- Pull down the arrow on the Dtn. No.
- Pull down the arrow on the Type of Tool.
- Add the cutter tool information from Tool Library.
- Click the Close button.

60
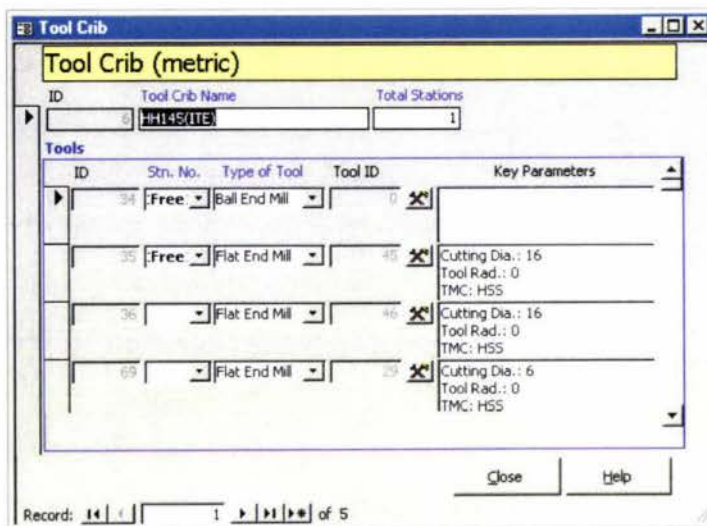
B.3 Applying CAMworks to Solidworks model

This section is to describe the simulation of the insert part using CAMworks. Through the insert part, students can grasp the basic use of CAMworks. The insert part with a logo "Massey 2000" is a particular part of this project. The thickness of each letter is 1 mm. Flat end cutter with 1 mm diameter was used to mill these letters. The insert part is shown on the right.
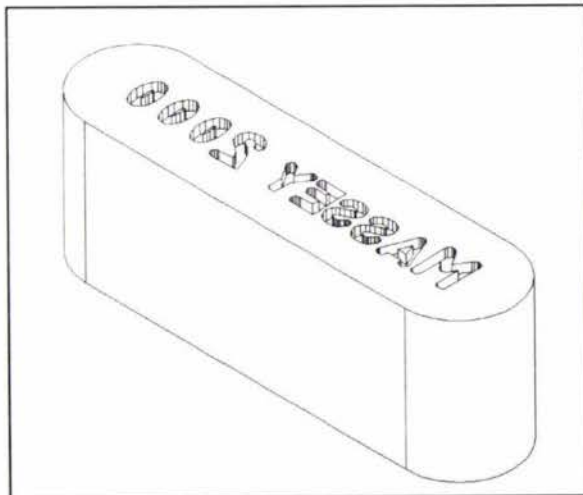
1. Define the machine parameters

• Open the part file "Insert plate".SLDPRT.

- Click the CAMworks Machinable Features tab at the bottom of the feature tree.

- Right click the Machine (ITE HH145) in the Machinable Features tree and select the Parameters command on the shortcut menu.

- Define the available machine: ITE HH145 and click select.

- Click the Tool set tab.

- Highlight the Crib 5 and click select.

- Click the Controller tab

- Highlight the HH145 and click select

- Click the Parameters tab

- Enter a program number. Leave Material blank.

- Click the "OK" bottom.
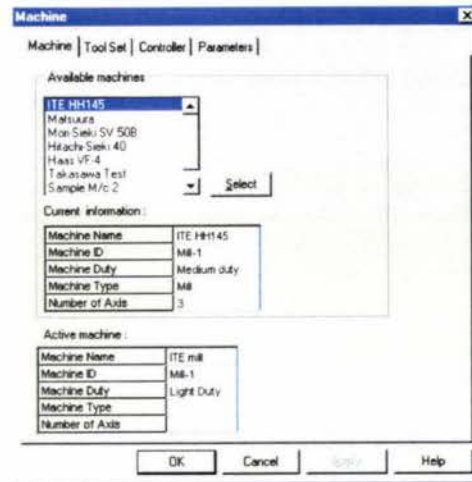
2. Define the size of workpiece

- Right click Workpiece in the Machinable Features tree and select the Edit Definition.

- Click the Material tab to define the material if a special material will be used.

- Click OK.
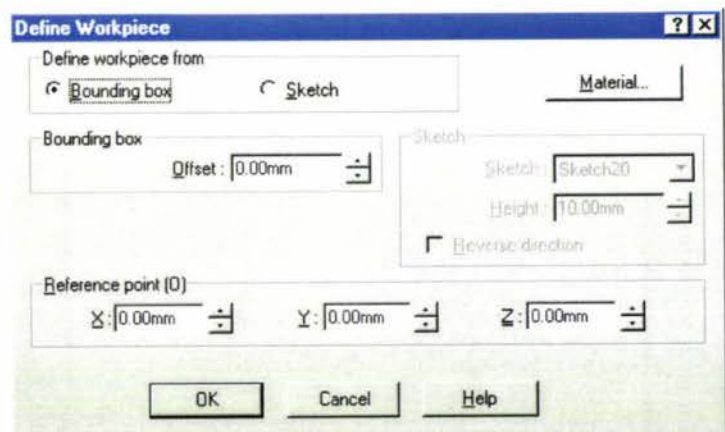
3. Generate operation plan

- Right click NC Manager in the Machinable Features tree and select Generate Operation plan.

When the Automatic Feature Recognition (AFR) process is finished, the Machinable Features tree displays the machinable features.

4. Insert a workplane

- Right click Workpiece and select Insert Workplane.
- Select a reference plane from the Insert Plane dialog box, if necessary.

5. Inserting a 2 axis feature

- Right click Workplane 1 in the Machinable Feature tree and select Insert 2 Axis Feature.
- The 2 Axis Feature Wizard dialog box appears.
- Select "M" from Available Sketches and click Next button.
- The 2-Axis Feature Wizard: Attribute dialog box appears
- Define Feature Type: Open profile; End Condition: bland and Depth: 1 mm.
- Click Finish bottom.
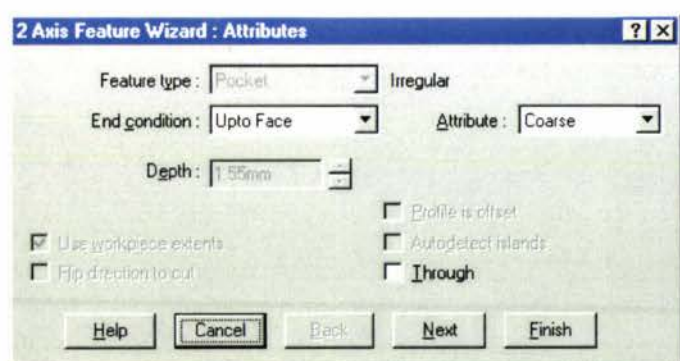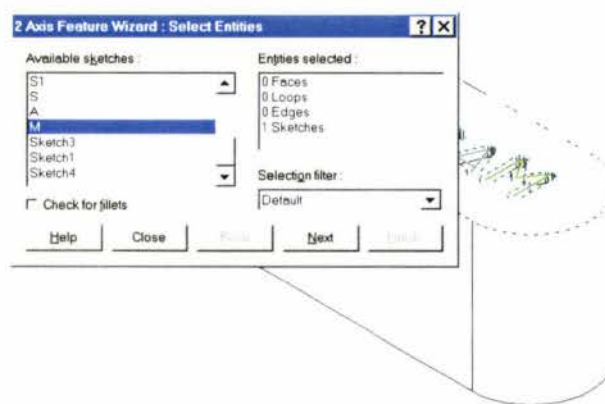
6. Re-generate the Extracting Machinable features

- Right click NC Manager in the Machinable Features tree and select Generate Operation plan.
- The Generate Operation Plan appears and click Update or Regenerate

7. Generate operation toolpath

- Click Operations tab at the bottom of the Machinable feature tree.
- Click the Workplane 1, and then right click the Generate Toolpath on the shortcut menu.
- The operations are listed in the operation tree.

8. Adjust operation parameters

- Right click on the Finish Mill in the operations tree.
- Click the parameters command on the shortcut menu.
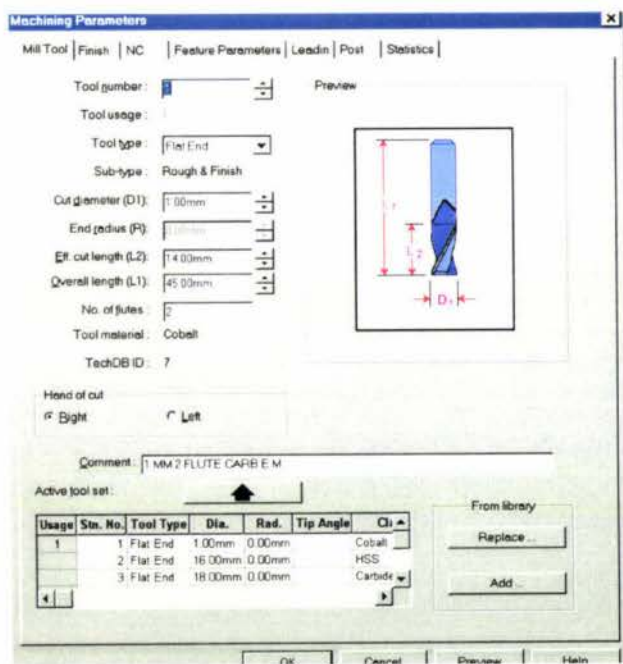
Mill Tool tab

Active tool set: 1 mm (Flat end)

If a cutter size can not find in the table, the cutter size can be added into the table from the Tool Library.

Finish tab

Depth parameters :

First cut amt = 0 .25 mm.

Max cut amt = 0.25 mm.

64

Final cut amt = 0.25 mm.

Sides parameters

Cut amt = 0.75 mm.

Other = 0 mm.

Cut methods: conventional.

Depth process: to depth by region.
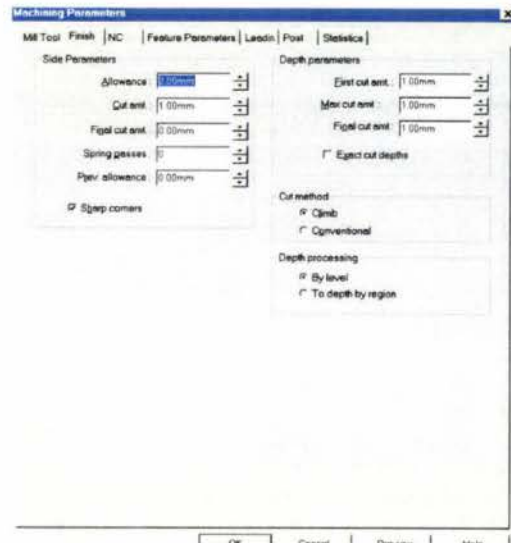
NC tab

    Rapid plane   : 10 mm

    Clearance plane   : 5 mm

    Z feed rate   : 40 mm/min

    XY feed rate   : 40 mm/min

    Spindle speed   : 2900 rpm (Max speed of CNC milling machine)

    CNC compensation   :select the "Off"

    Toolpath center   : select the "without compensation"
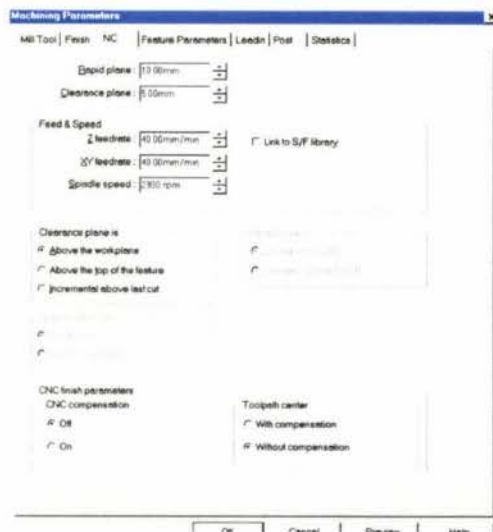
Feature parameters tab

    Entry type: No

    Feature list: check the machining depth.

Leadin tab

    Lead in/out: click the start point.

Post tab

    Absolute incremental : Absolute.

9. Set the origin

- Right click the Workplane1 in the feature operation tree and click the parameters on the shortcut menu.
- The Workplane dialog displays.
- Set the origin of the machining material.
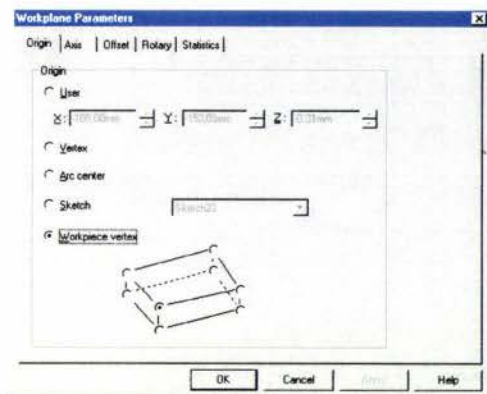
10. Generating toolpath and simulating the machining process

- Right click Workplane 1 in the operations tree and click Generate Toolpath on the shortcut menu.
- Click an operation in the operations tree-the toolpath.
- Right click operation in the operations tree and click Simulate Toolpath on the shortcut manu.

The simulation is shown as below.
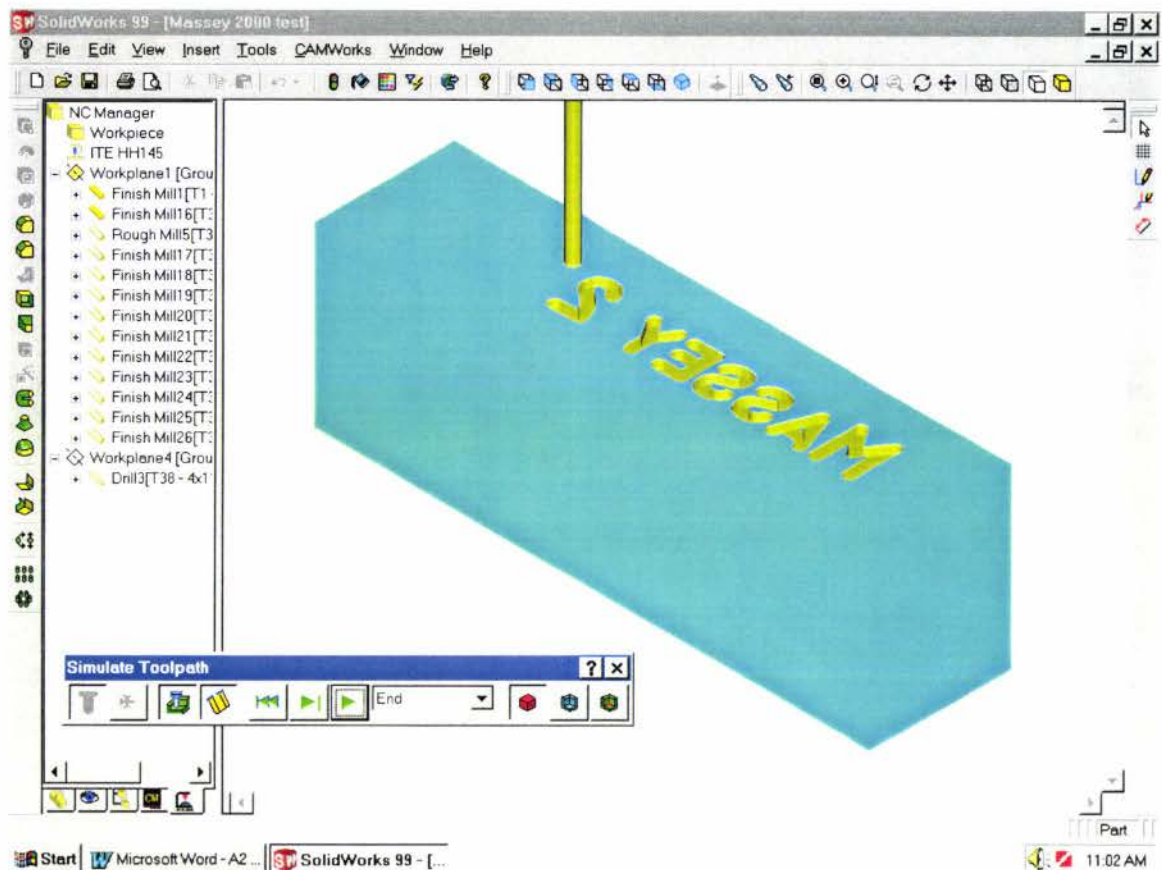
11. post process toolpaths

- Right click NC machine and click the Post Process on the shortcut menu.

- Enter the name M2K and click save

- Click Run and then the NC program appears.

- Save the NC program.
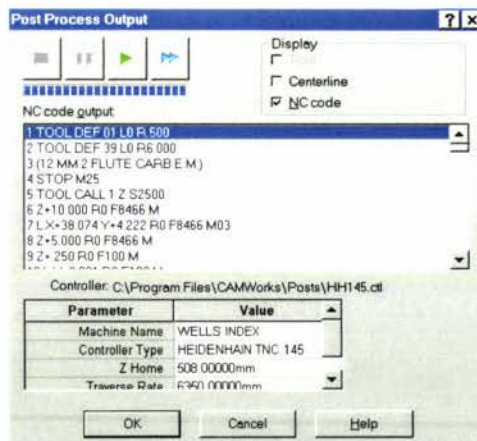
C. INFORMATION OF MACHINING PROCESS

C.1   Introduction

Metal cutting is the removal of unwanted metal from a workpiece in the form of chips to obtain a finished part of desired size, shape and finish. There are seven basic chip formation processes: turning, milling, drilling, shaping, sawing, broaching and grinding.

C.2   Tools selection

The tool selection of the project was based on the existing facilities of ITE and mold requirements. The selection includes the measuring instruments, cutter tools, polish, machining parameters.

C.2.1   Measuring instruments

The measuring instruments have machine-mounted measuring devices and manual measuring instruments. The range in accuracy of machine-mounted measuring devices permits measurements from 0.5 to 0.0003mm. For manual measuring instruments, precision and accuracy of the instruments can be affected by the skill of the person, the elastic deformation of the instrument and the temperature of the machined material. But the advantage of a manual measuring instrument is that it is easy to use and costs less.

Most small or medium sizes usually use. The CAD/CAM laboratory in ITE is a typical small size workshop. The existing measuring instruments are illustrated in Fig C-1.

1. Vernier caliper

The vernier caliper end-measuring instrument is available in various sizes. The vernier caliper can be used to measure the diameter of a hole, a cylinder, depth of a bland hole and length. But the problem of the vernier caliper is that the difficulty remains in obtaining uniform contact pressure between the legs of the instrument. The accuracy of vernier caliper is 0.01.

Figure C- 1 The existing measuring instruments of the workshop: Venier caliper (top), indicator (middle) and machinist's rules

2. Dial indicator

Dial indicators are a widely used form of deviation-type gauges. Most dial indicators have a spindle travel equal to 2 ½ revolutions of the indicating pointer and are read in either 0.02 or 0.002. The indicator is often mounted on an adjustable arm to permit its

being brought into proper relationship with the work. It is important that the axis of spindle can be exactly aligned between the workpiece to achieve accuracy.

C.2.2   Milling

The milling process can be classified into broad categories. The first is peripheral milling, the other is surface milling. In peripheral milling, the generated surface is parallel with the axis of rotation of the cutter, and this process is often called slab milling. In surface milling, the generated surface is at right angles to the cutter axis. In the CAD/CAM laboratory, CNC milling machine is a surface milling machine and the controller is HH145. The determining parameters of milling process are:

1. Cutting speed:

$$V = \pi * D * N / 1000$$

Where
V: cutting speed.
D: cutter diameter.
N: the spindle speed.

3. Feed rate:

$$F = f_t * N * Z$$

Where
F: feed rate
f_t : feed of the table

N; spindle speed

Z: No of teeth in the cutter

The materials of milling cutters in this project are high-speed steel (Fig. C-2) and carbide milling cutter (Fig.C-3). The minimum size of HSS milling cutter is 1 mm slot mill, and maximum is 16 mm. The Figure C-3 shows some of HSS milling cutters. Those consist of Insert-tooth face milling cutter, carbide slot milling cutters and ball-end milling cutters.

Figure C- 2 Typical milling cutter.

Figure C- 3 Typical insert carbide milling cutters and ball end milling cutters.

According to equations, the feed rate and spindle speed are shown in Table. The milling process is based on determining parameters to machine A plate, B plate and their components.

raw material	V m/min	D mm	Tool material	Type	N rpm	n	f mm/t	feed rate mmpm
P20	12	1	HSS	slot mill	2800	2	0.01	56
P20	45	2	carbide	end mill	2800	4	0.07	784
P20	45	2	carbide	ball end	2800	4	0.07	784
P20	45	6	carbide	ball end	2389	4	0.07	669
P20	60	6	carbide	end mill	3185	1	0.08	255
P20	60	10	carbide	end mill	1911	2	0.1	382
P20	60	16	carbide	end mill	1194	2	0.1	239
Steel	70	6	carbide	end mill	2800	1	0.15	557
Steel	70	10	carbide	end mill	2229	2	0.15	669
Steel	70	16	carbide	end mill	1393	2	0.15	418

Table C- 1 The spindle speed and feed rate in this project.

C.2.3 Drilling

In manufacturing, drilling is a very important process because a large proportion of holes are made by drilling. Most drilling is done with a tool having two cutting edges. The most common types of drills are twist drills such as Fig. C-4.

For this project, the parameters of drilling process are:

1.   Cutting speed

$N = 1000*V / (\pi*D)$

Where
Drill diameter: D mm
Rotation of the spindle: N rpm
Cutting speed: V

2. Feed rate

Feed rate (mmpm) = Feed (mmpm) x (rpm)

The feed rate and spindle speed are shown in table C-2.

Figure C- 4 Twist drills were used to drill holes in this project.

Machining material	V (m/min)	D (mm)	N (rpm)	f (mm/r)	feed rate (mmpm)
steel	32	4	2548	0.08	204

P20	26	4.8	1725	0.03	52
steel	30	5	1911	0.1	191
P20	26	6	1380	0.04	55
steel	28	7	1274	0.14	178
P20	12	8	478	0.17	81
P20	12	9.8	390	0.22	86
steel	26	10	828	0.18	149
steel	24	13	588	0.24	141
P20	12.5	15.7	254	0.32	81
steel	22	16	438	0.4	175

Table C- 2 The feed rate of drilling.

C.2.4 Reaming

Reaming removes a small amount of material from the surface of holes. It is done for two

purposes: to bring holes to a more exact size and improve the finish of an existing hole. In order to obtain proper results, only a minimum number of materials should be left for removal by reaming. A proper reamed hole will be within 0.001 in (0.025mm) of correct size and have a fine finish.

Figure C-5 The example of straight-fluted hand reamer.

Reaming parameters

Machining material	V (m/min)	D (mm)	N (rpm)	f (mm/r)	feed rate (mmpm)
P20	18	5	1146	0.1	115
P20	18	10	573	0.19	109
P20	21	16	418	0.26	109

Table C- 3 The feed rate of straight-fluted hand reamer.

C.2.5   Thread tapping

The cutting of an internal thread using a multiple-point tool is called thread tapping, and the tool is called a tap. A hole must already exist, which is made by drilling. The diameter of a hole is slightly larger than the minor diameter of the thread. The solid hand taps are usually used for small holes. Figure C-6 illustrates a file of top with drill.
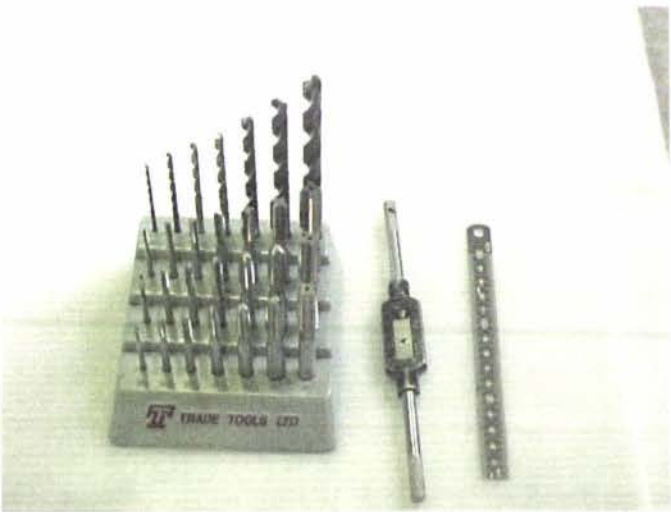
Figure C- 6 A file of taps sets.

The following table shows the required diameter of the holes for tapping.

Bolt size	Diameter of a hole
M5	4.2 mm
M8	7 mm
M10	9 mm

Table C- 4 The size of hole for tapping in this project.

C.2.6   Polishing

Most of the mold polishing is done by hand with the aid of pine sticks and polishing compound, especially in A plate and B plate. Most mold makers use the "Hand grinder" to finish grinding to the close fit requirements. A "Hand grinder" can operate at speeds up to 22,000 rpm, which is a very useful tool for polishing.

The polishing operation in this workshop only can use basic tools to polish cavities. At this stage, a number of grit papers and abrasive stones were used to polish the mold without the hand grinder. First, files were used to remove machining marks, and then the polishing operation started the small grade grit paper (180), and then up to high grade grit paper (1200) to polish surfaces. Some abrasive stones were used to polish the vertical walls. Figure C-7 shows some tools for polishing.

Figure C- 7 Tools for polishing in this project.

D DESIGN OF RUNNER SYSTEMS

D.1   Runner system

The runner system accommodates the molten plastic from the barrel and guides it into the impression. The runner is a channel machined into the mould plate, which connects to the sprue with the gate to the impression. The runner is positioned on the parting line (surface) in the basic two-plate mould. But some runners may be positioned below the parting line. Wherever the position of the runner, it has to be demoulded with the moulding after each shot.

D.2   Types of runner system

There are two types of runner -gate systems, naturally-balanced and naturally-unbalanced. Naturally-balanced means that all flow paths are identical in shape, size, and with the same pressure drop at the same instance during cavity filling. Naturally-unbalanced runner-gate system leads to a non-uniform filling condition for each cavity.

The preferred type of runner layout is known as the balance runner, as shown in Fig D-1. The balance runner permits best uniformity of flow of the material from sprue into the various impressions since all impressions are at equal distance from sprue. This design is beneficial for all types of material.

D.2.1   Balance layouts

The balanced runner provides equal distance and runner size from sprue to all the cavities, so that the filling conditions of all cavities are the same.

The H runner system
Advantages: Equal flow lengths to all cavities, easy demoulding and without gate correction.
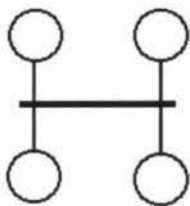
The "H" runner system

Disadvantages: only limited number of cavities can be accommodated

The radical runner system
Advantages: Equal flow lengths to all cavities, easy demoulding especially of parts requiring unscrewing device and without gate correction.
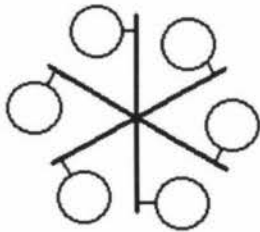
The Radial runner system

Disadvantages: same as H runner system
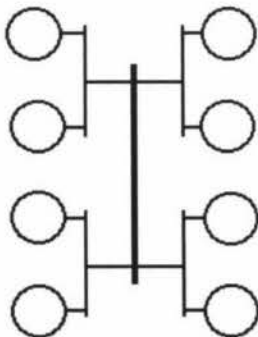
The symmetrical runner system
Advantages: same as H runner system

The Symmertrical runner system

Disadvantages: large runner volume.

Figure D- 1 Typical balance runner layout.

D.2.2 Unbalanced layout

Although the herringbone as shown in Figure D-2 is unbalanced, it has more space to accommodate more cavities than with radial or H layout, with minimum runner volume and tooling cost. The disadvantage is the unequal flow lengths to individual cavities. The unbalanced runner system can be artificially balanced by changing the runner diameter using CAE software such as C-mould.

Typical unbalance runner system
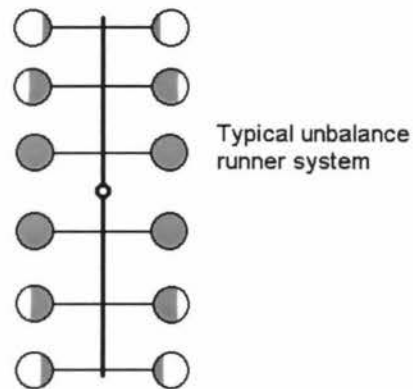
Figure D- 2 An unbalance runner layout.

D.3 Considerations of runner design.

D.3.1 Runner size

Before deciding the size of runner, the following factors must be considered:

- The cross-section of the runner.

It must be sufficient to permit the molten plastic to pass through and fill the impression before the runner freezes. On the other hand, the molten plastic can be applied for shrinkage compensation under the packing pressure if required.

- The distance between the sprue and the impression

79

if the distance from the sprue to the impression is increased, the diameter of the runner need to increase for reducing the flow resistance

- The range of mould-maker's cutters available

The size chosen for the runner should be considered according to the range of cutters in stock or on the market. The more common sizes are from 2 to 13 mm in 1mm steps and 1/8 – ½ in 1/16 steps in the imperial.

- The plastic material to be used.

The various plastic materials have different chemical characters such as velocity and molecular structure. Those material require minimum runner diameters, which are shown in table D-1.

D.3.2 Easy ejection

Runner designs must provide for easy ejection and easy removal from the moulded part, with proper cross-sectional and draft angle. The runner surface must be polished to facilitate flow and part ejection for most materials. Extended runner systems should have multiple sprue pullers and ejection locations.

D.4 Design of runner system

D.4.1 Sprue

The sprue is generally formed in the sprue bushing. After the mould is closed and the machine nozzle has been forced against the mould to seal off the point of transition between the two, the material flows directly from the barrel into the sprue. This procedure puts a very high local load upon the mould and it wears out relatively fast unless it is made of hardened steel and inserted into the mould. Thus, it can be easily replaced if damaged or worn out, as illustrated in Fig. D-3 c)

The surface of contact gains particular significance as a sealing area. Plan surfaces are seldom used in practice because they require higher sealing pressure, illustrated in Fig D-3a). In most cases, curved (spherical) contact surfaces are encountered. A shallow, spherical indentation is machined into the sprue bushing as illustrated in Fig. D-3 b [5].

a) plane area of contact between machine and sprue bushing

b) curved area of contact between machine and sprue bushing

c) another types of the curved area of contact between machine and sprue bushing
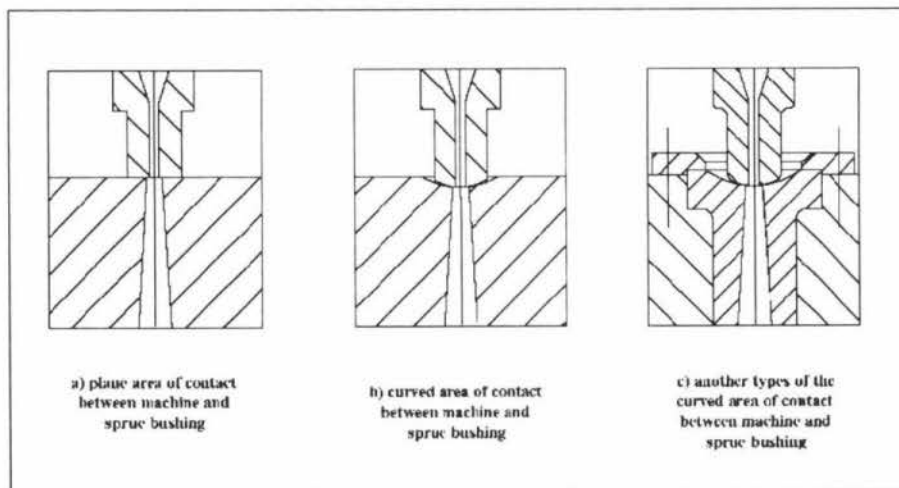
Figure D- 3 Typical sprue design.

The following general conditions determine the dimensions of the spherical contact surfaces.

$R_n + 1 \leq R_s$ (mm)

$d_s \leq d_n - 1$ (mm)

Where

R_n: spherical radius of the nozzle tip,

R_s: spherical radius of the indentation in the sprue bushing,

d_n: orifice diameter of the nozzle and

d_s: Orifice diameter of the sprue bushing.

The dimensions of the sprue depend primarily on the dimensions of the moulding. The required dimensions are:

Df >= Smax + 1.5mm

Da >= Dd + 1mm

α = 1~2 degree

tan α = (Df – Da) / (2*L)
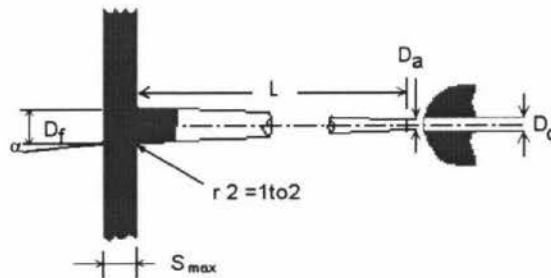
Figure D- 4 The dimensions of a sprue[5].

D.4.2 Runner cross section

Typical runner cross sections are shown in Fig D-5. The selection of the runner should consider the minimum heat loss and pressure drop. Full round runner is preferred that has minimum surface-to-volume ratio, which minimizes heat loss and pressure drop. Trapezoidal runners work well and permit the runner system to be designed on one side of the mould, which is commonly used in three-plate moulds [5].
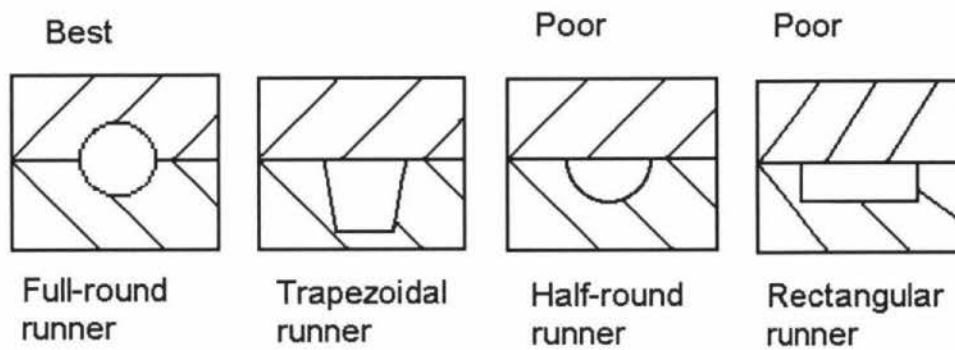
Figure D- 5 The cross sections of runners.

D.4.3 Determination of the main runner

The following formula is used for runner design:

D = (W1/2 * L1/4) /3.7

Where

D = runner diameter (mm)

W = part weight (g)

L = runner length (mm)

A typical runner diameter for various unfilled materials is given in Table D-1[5]. These approximate sizes are for conventional runner moulds and not for hot runner or insulated runner moulds.

Material	Runner Diameter (in.)
ABS, SAN	3/16 - 3/8

Acetal	1/8 – 3/8
Acetate	3/16 – 7/16
Acrylic	5/16 – 3/8
Butyrate	3/16 – 3/8
Fluorocarbon	3/16 – 3/8
Impact acrylic	5/16 – ½
Ionomers	3/32 – 3/8
Nylon	1/16 – 3/8
Phenylene	¼ - 3/8
Phenylene sulfide	¼ - ½
Polyallomer	3/16 – 3/8
Polycarbonate	3/16 – 3/8
Polyester thermoplastic:3/16 – 3/8	1/8 – 5/16
Polyetylene: low to hi-density type:1, 2, 3, 4	1/16 – 3/8
Polyamide	3/16 – 3/8
Polyphenylene oxide	¼ - 3/8
Polypropylene	3/16 – 3/8
Polystyrene: general purpose medium impact-hi impact	1/8 – 3/8
Polysulfone	¼ – 3/8
Polyvinyl (plasticized)	1/8 – 3/8
PVC rigid (modified)	¼ - 5/8

Table D- 1 Suggestion of runner diameter[5].

D.4.4   Determination of Branched runners

The diameter of the branch runners should be smaller than the main runner, because less material flows through the branches and it is economically desirable to use minimum

material in the runners. So, the relationship between the main runner diameter (d_{main}) and the branch runner diameter (d_{branch}) is

$$d_{main} = d_{branch} * N^{1/3}$$

Where
N is the number of branches

E GATE

E.1   Introduction

A Gate is a part of the runner system. It is a channel, which connects the runner with the cavity. Compared with the runner system, it has a small cross-section area.

E.2   Gate types

1)   Direct (or sprue) gate

Sprue gate is commonly used for single-cavity molds, where the sprue feeds material directly into the cavity rapidly with minimum pressure drop. The disadvantage of using a sprue gate is the gate mark left on the part surface after the sprue is trimmed off.
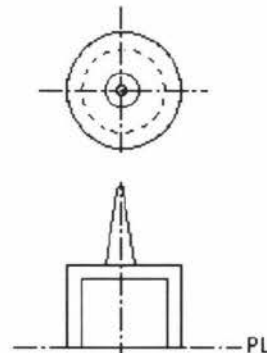
2)   Tab gate

A tab gate is typically employed for flat and thin parts, to reduce the shear stress in the cavity. The high shear stress generated around the gate is confined to the auxiliary tab, which is trimmed off after molding. A tab gate is used extensively for molding PC, acrylic, SAN, and ABS types of materials.
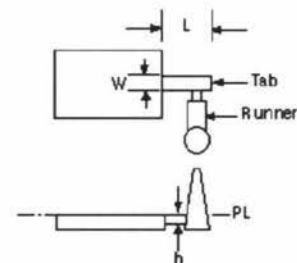
3) Edge (standard) gate:

An edge gate is located on the parting line of the mold and typically fills the part from the side, top or bottom.
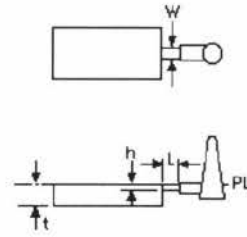
4) Overlap gate

An overlap gate is similar to an edge gate, except the gate overlaps the wall or surfaces. This type of gate is typically used to eliminate jetting.
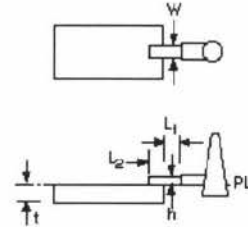
5) Fan gate

A fan gate is a wide edge gate with variable thickness. It permits rapid filling of large parts or fragile mold sections through a large entry area. It is used to create a uniform flow front into wide parts, where warpage and dimensional stability are main concerns. The gate should taper in both width and thickness, to maintain a constant cross sectional area. This will ensure that:
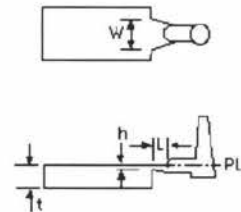
The melt velocity will be constant.
The entire width is being used for the flow.
The pressure is the same across the entire width.

6) Disk (diaphragm) gate

A diaphragm gate is often used for gating cylindrical or round parts that have an open inside diameter. It is used when concentricity is an important dimensional requirement and the presence of a weld line is objectionable. This gate is

87
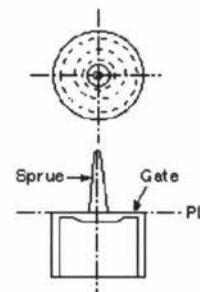
essentially a flash gate around the inside edge of the part. Since the diaphragm is fed from a concentric sprue (or stub-runner drop), uniform flow to all parts of the gate is easy to maintain.

7) Ring gate

Like a diaphragm gate, a ring gate is also used for cylindrical or round parts, but it is not always recommended. With a ring gate, the material flows freely around the core before it moves down as a uniform tube-like extrusion to fill the mold.
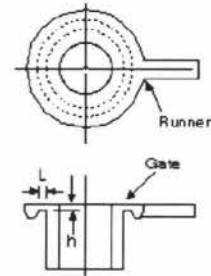
8) Spoke (spider) gate

This kind of gate is also called a four-point gate or cross gate. It is used for tube-shaped parts and offers easy de-gating and material savings. Disadvantages are the possibility of weld lines and the fact that perfect roundness is unlikely.
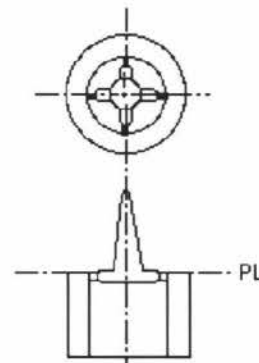
9) Film (flash) gate

A film gate is similar to a ring gate, but it is used for straight edges. It consists of a straight runner and a gate land across either the entire length or width of the cavity or a portion of the cavity. It is used for acrylic parts, and generally for flat designs of large areas where warpage must be kept to a minimum.
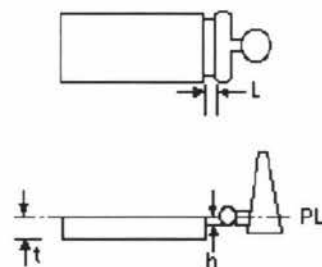
10) Pin gate

This type of gate relies on a three-plate mold design, where the runner system is on one

88

mold parting line and the part cavity is in the primary parting line. Reverse taper runners drop through the middle (third) plate, parallel to the direction of the mold opening. As the mold cavity parting line is opened, the small-diameter pin gate is torn from the part. A secondary opening of the runner parting line ejects the runners. Alternatively, the runner parting line opens first. An auxiliary, top-half ejector system extracts the runners from the reverse taper drops, tearing the runners from the parts.
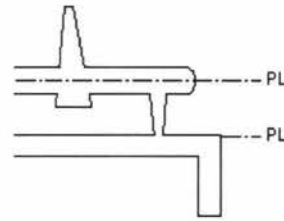
Benefits: The design is particularly useful when multiple gates per part are needed to assure symmetric filling or where long flow paths must be reduced to assure packing to all areas of the part.

11)   Submarine (tunnel, chisel) gate

A submarine gate is used in two-plate mold construction. An angled, tapered tunnel is machined from the end of the runner to the cavity, just below the parting line. As the parts and runners are ejected, the gate is sheared at the part. If a large diameter pin is added to a non-functional area of the part, the submarine gate can be built into the pin, avoiding the need for a vertical surface for the gate. If the pin is on a surface that is hidden, it does not have to be removed. Multiple submarine gates into the interior walls of cylindrical parts can replace a diaphragm gate and allow automatic de-gating. The out-of-round characteristics are not as good as those from a diaphragm gate, but are often acceptable.
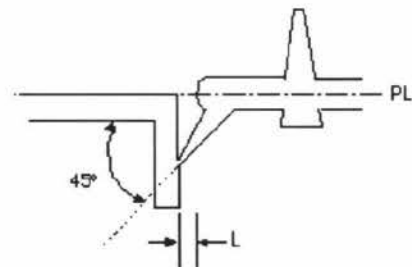
12)   Hot-runner gate

A hot-runner gate is generally used to deliver hot material through heated runners and electrically heated sprues directly into the cavity, producing runnerless moldings. The packing cycle is controlled by the freeze-off of the part near the gate. The very hot material at the gate is torn from the part as the cavity is opened.

E.3   Typical gate sizes

Gate size and location are decided by considering various requirements. The general of design, the gate should be small, simple to demold and easily separated from the part. The size of the gate is determined by the material to be processed and the section thickness of the molding. The higher the viscosity of the plastic and the thicker the section, the gate is large. The experienced gate design is shown in table E-1[11].

Item	Gate type	Dimension
1	Direct (sprue) gate	The sprue diameter here must be about 1.0 mm larger than the nozzle exit diameter.
2	Tab gate	Mini. tab width: 6.4 mm. Mini tab thickness: 75% of the impression's depth.
3	Edge (standard) gate	Gate size: 6% to 75% of the part thickness. The gate land: 0.5 to 1mm
4	Overlap gate	Gate size: Thickness: 0.4 to 6.4 mm thick Wide:1.6 to12.7 mm
5	Fan gate	Maxi. thickness: < 75% of the part

		thickness. Gate sizes: 0.25 to 1.6 mm Width: 6.4 mm to 25% of the cavity length
6	Disk (diaphragm) gate	gate thickness: 0.25 to 1.27 mm.
7	Ring gate	gate thickness: 0.25 to 1.6 mm.
8	Spoke (spider) gate	Thickness: 0.8 to 4.8 mm Wide: 1.6 to 6.4 mm
9	Film (flash) gate	Thickness: 0.25 to 0.63 mm Land area (gate length) must also be kept small, approximately 0.63 mm long.
10	Pin gate	The typical gate size: 0.25 to 1.6 mm in diameter.
11	Submarine gate	The typical size: 0.25 to 2.0 mm in diameter.

Table E- 1 Suggestion of gate size[11].

E.4   Considerations of gate design

E.4.1   Essentials of a well-designed gate

Gates should always be small at the beginning of the design process so they can be enlarged, if necessary. Reducing the gate size is not as easy as enlarging it. A properly designed gate will deliver a rapid, uniform and preferably single-directional mold filling

pattern with a proper gate freeze-off time.

E.4.2   Gate location

The gate location should be at the thickest area of the part. It is preferably at a spot where the function and appearance of the part are not impaired. This leads the material to flow from the thickest areas to thinner areas, and then helps maintain the flow and packing paths. Gate location should be central so that flow lengths are equal to each extremity of the part.

E.4.3   Avoiding common problems

Improperly positioned gates often cause the following problems:

1. Asymmetric flow paths will allow some areas to be filled, packed, and frozen before other areas are filled. This will result in differential shrinkage and probable warpage of the parts.

    The solution is to position the gate symmetrically to avoid warpage.

2. The gate location should allow the air present in the cavity to escape during injection to prevent air traps. Failure to vent the air will result in a short shot, a burn mark on the molding, or high filling and packing pressure near the gates.

    The solution is to vent properly to prevent air traps.

3. Gate location and size should prevent jetting. Jetting can be prevented by enlarging the gate or by locating the gate where the flow is directed against a cavity wall.

   The solution is to enlarge the gate to avoid jetting.

4. The gate location should might weld-lines, but there are not important if they are on the inside surfaces.

F THERMOPLASTIC

F.1   Thermoplastic

Thermoplastic resins consist of long molecules, each of which may have side chains or groups that are not attached to other molecules. Therefore, they can be repeatedly melted and solidified by heating and cooling so that the thermoplastic can be reused in processing. This chapter lists the typical application and injection molding conditions[9].

F.2   Typical applications and injection molding conditions

1. ABS (ACRYLONITRILE-BUTADIENE-STYRENE)

Typical application: automotive (instrument and interior trim parts), refrigerators, small appliance housings and power tools applications such as hair dryers, telephone housings and recreational vehicles such as golf carts.

The injection molding processing conditions are:

Drying	Suggest drying conditions are 80 - 90 C for a minimum of 2 hours. Resin moisture content should be less than 0.1%
Melt Temperature	200 – 280 C (392 - 536 F);
Mold Temperature	25 – 80 C (77 - 176 F). (Mold temperatures control the gloss properties; lower mold temperatures produce lower gloss levels)
Resin Injection	500 – 1,000 bar (7,250 - 14,500 psi)

Pressure	
Injection Speed	Moderate - high

2. PA 6 (POLYAMIDE 6, OR NYLON 6, OR POLYCAPROLACTAM)

Typical applications: PA6 is used in many structural applications because of its good mechanical strength and rigidity. It is used in bearings because of its good wear resistance.

Injection Molding processing conditions:

Drying	If the moisture content is >0.2%, drying in a hot air oven at 80 C (176 F) for 16 hours is recommended. If the material has been exposed to air for more than 8 hours, vacuum drying at 105 C (221 F) for more than 8 hours is recommended.
Melt Temperature	230 - 280 C (446 - 536 F); 250 - 300 C (482 - 572 F) for reinforced grades
Mold Temperature	80 - 90 C (176 - 194 F). If the wall thickness is greater than 3 mm, a cold mold is recommended (20 - 40 C / 68 - 104 F), which leads to a higher and more uniform degree of crystallinity. Glass reinforced resins are always processed at mold temperatures greater than 80 C (176 F).
Resin Injection Pressure	750 - 1,250 bar (~11,000 - 18,000 psi) (depends on material and product design)
Injection Speed	High (slightly lower for reinforced grades)

3. PA 12 (POLYAMIDE 12 OR NYLON 12)

Typical applications: Gear wheels for water meters and business machines, cable ties, cams, slides and bearings.

Injection Molding Processing Conditions:

Drying	If the material is exposed to air, drying in a hot air oven at 85 C (185 F) for 4 -5 hours is recommended (3-4 hours in a desiccant dryer).
Melt Temperature	230 - 300 C (446 - 580 F); Not to exceed 310 C (590 F) for standard grades.
Mold Temperature	30 - 40C (86 - 104 F) for unreinforced grades; for thin walled or large surface area components, 80 -90 C (176 - 194 F) may be used; 90 - 100 C (194 - 212 F) for reinforced grades. Increasing the mold temperature increases the crystallinity
Resin Injection Pressure	Up to 1,000 bar (14, 500 psi) Low hold pressures and high melt temperatures are recommended.
Injection Speed	High (high speeds give better finish on glass-filled grades)

4. PC (POLYCARBONATE)

Typical applications: Electronic and business equipment (computer parts, connectors, etc.), appliances (food processors, refrigerator drawers, etc.), transportation (head lights, tail lights, instrument panels, etc.).

Injection Molding Processing Conditions:

Drying	Drying conditions are 100 - 120 C (212 - 248 F) for 3 to 4 hours. Moisture content must be less than 0.02% prior to processing.
Melt Temperature	260 - 340 C (500 - 644 F) higher range for low MFR resins and vice-versa
Mold Temperature	70 - 120 C (158 - 248 F); higher range for low MFR resins and vice-versa
Resin Injection Pressure	As high as possible for rapid molding

Injection Speed	Slow injection speeds when small or edge gates are used; high speeds for other types of gates

5. PE -HD (HIGH DENSITY POLYETHYLENE)

Typical applications: Containers in refrigeration units, storage vessels, household goods (kitchenware), seal caps, bases for PET bottles, etc. Major use is in blow-molding applications (packaging applications).

Injection Molding Processing conditions:

Drying	Not normally necessary if stored properly.
Melt Temperature	180 - 280 C (356 - 536 F); for high molecular weigh resins, the suggested melt temperature range is 200 - 250 C (392 - 482 F)
Mold Temperature	20 - 95 C (68 - 194 F)
Resin Injection Pressure	700 - 1,050 bar (10,000 - 15,000 psi)
Injection Speed	Fast injection speeds are recommended; profiled speeds reduce warpage in the case of components with a large surface area.

6. PE-LD (LOW DENSITY POLYETHYLENE)

Typical applications: Closures, bowls, bins, pipe couplings.

Injection Molding Processing Conditions:

Drying	Not usually necessary
Melt Temperature	180 - 280 C (355 - 535 F)
Mold Temperature	20 - 70 C (68 - 158 F)
Resin Injection Pressure	Up to 1,500 bar (21,750 psi) Pack Pressure Up to 750 bar (10,850 psi)

Injection Speed	Fast speeds are recommended; profiled speeds can limit warpage problems of large surface area parts.

7. PMMA (POLYMETHYL METHACRYLATE)

Typical applications: Automotive (signal light devices, instrument panels, etc.), medical (blood cuvettes, etc.), industrial (video discs, lighting diffusers, display shelving, etc.), consumer (drinking tumblers, stationery accessories, etc.).

Injection Molding Processing Conditions

Drying	PMMA must be dried prior to molding. Drying at 90 C (194 F) for 2-4 hours is recommended.
Melt Temperature	240 - 280 C (460 - 536 F)
Mold Temperature	35 - 80 C (90 - 176 F)
Injection Speed	Moderate

8. PP (POLYPROPYLENE)

Typical applications: Automotive (mostly mineral-filled PP is used: dashboard components, ductwork, fans, and some under-hood components), appliances (doorliners for dishwashers, ductwork for dryers, wash racks and lids for clothes washers, refrigerator liners, etc.), consumer products (lawn/garden furniture, components of lawn mowers, sprinklers, etc.).

Injection Molding Processing Conditions:

Drying	Not normally necessary if proper storage is used
Melt Temperature	220 - 280 C (428 - 536 F); not to exceed 280 C
Mold Temperature	20 - 80 C (68 - 176 F) The crystallinity level is determined by the mold temperature.
Resin Injection Pressure	Up to 1,800 bar (26,000 psi)

Injection Speed	Typically, fast injection speeds are used to minimize internal stresses; if surface defects occur, slow speed molding at a higher temperature is preferred. Machines capable of providing profiled speed are highly recommended.

9. PS (POLYSTYRENE)

Typical applications: packaging for food, housewares (tableware, trays, etc.), electrical (transparent housings, light diffusers, insulating film).

The injection molding processing condition are:

Drying	Not usually required unless stored improperly. If drying is needed, the recommended conditions are 2-3 hours at 80 C (176 F)
Melt Temperature	Not usually required unless stored improperly. If drying is needed, the recommended conditions are 2-3 hours at 80 C (176 F)
Mold Temperature	Suggested: 20 - 70 C (68 - 158 F)
Resin Injection Pressure	200 - 600 bar (3,000 - 8,700 psi)
Injection Speed	Fast speeds are recommended

10. PVC (POLYVINYL CHLORIDE)

Typical application: water distribution piping, house siding, business machine housings, packaging for foodstuffs, etc.

The injection Molding Processing Conditions are:

Drying	Not usually necessary as PVC absorbs very little water.
Melt Temperature	160 - 220 C (320 - 428 F)
Mold Temperature	20 - 70 C (68 - 158 F)

Resin Injection Pressure	Up to 1,500 bar (21,750 psi)
	Packing Pressure
	Up to 1,000 bar (14,500 psi)
Injection Speed	Relatively slow, to avoid material degradation

11. SAN (STYRENE ACRYLONITRILE)

Typical applications: Electrical (receptacles, mixer bowls, housings, etc. for kitchen appliances, refrigerator fittings, chassis for television sets, cassette boxes, etc.), automotive (head lamp bodies, reflectors, glove compartments, instrument panel covers, etc.), household appliances (tableware, cutlery, beakers, etc.), cosmetic packs, etc.

Injection Molding processing Conditions:

Drying	Dry at 80 C (176 F) for 2-4 hours prior to molding.
Melt Temperature	200 - 270 C (392 - 518 F); 230 - 260 C (446 - 500 F) for most applications; lower end of the range is used for molding thick wall components
Mold Temperature	40 - 80 C (104 - 176 F); SAN solidifies rapidly at higher temperatures; for reinforced grades, the mold temperatures should not be less than 60 C (140F). The cooling system must be well designed because the mold temperature affects the parts appearance, shrinkage and warpage.
Resin Injection Pressure	350-1,300 bar (5,000 - 20,000 psi)
Injection Speed	High speeds are recommended

G A LIST OF DRAWINGS

90

70

66

90

60

58

MASSEY 2000

R5

SECTION D-D

58

4.50

4.50

90

13

7

35

0.75

0.70

1.40

SECTION E-E

102

UNLESS OTHERWISE SPECIFIED DIMENSIONS ARE IN INCHES TOLERANCES ARE: FRACTIONS DECIMALS ANGLES + .XX+ .XX + 1 .XXX+.XXX	CAD GENERATED DRAWING, DO NOT MANUALLY UPDATE		Institute of Technology & Engineering Massey University	
	APPROVALS	DATE		
	DRAWN Yuk-sun, Lam	8-4-1999		
	CHECKED Dr. Tang		Main Frame	
MATERIAL HD PE	RESP ENG			
FINISH --	MFG ENG		SIZE A4	DWG. NO. PF 1 REV. 3
DO NOT SCALE DRAWING	QUAL ENG		SCALE	SW DRAWING FILE: SHEET 1 OF 18

103

DETAIL K

R1.03

DETAIL L

1.49
0.67
2.49

UNLESS OTHERWISE SPECIFIED DIMENSIONS ARE IN INCHES TOLERANCES ARE: FRACTIONS DECIMALS ANGLES + .XX+ .XX + 1 .XXX+.XXX	CAD GENERATED DRAWING, DO NOT MANUALLY UPDATE	**Institute of Technology & Engineering Massey University**
	APPROVALS / DATE	
	DRAWN Yuk-sun Lam 8-4-1999	
	CHECKED Dr. Tang	Supported frame
MATERIAL HD PE	RESP ENG	
FINISH --	MFG ENG	SIZE A4	DWG. NO. PF 02	REV. 3
DO NOT SCALE DRAWING	QUAL ENG	SCALE	SW DRAWING FILE:	SHEET 2 OF 18
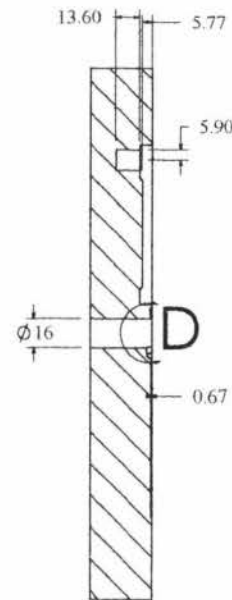
SECTION A-A

DETAIL D

DETAIL E

SECTION C-C

104

| 8 | 7 | 6 | 5 | 4 | 3 | 2 | 1 |

SECTION B-B

DETAIL D

SECTION A-A

DETAIL C

UNLESS OTHERWISE SPECIFIED DIMENSIONS ARE IN INCHES TOLERANCES ARE:		CAD GENERATED DRAWING, DO NOT MANUALLY UPDATE		*i̇e* **Institute of Technology & Engineering** **Massey University**	
FRACTIONS DECIMALS ANGLES + .XX+ .XX + 1 .XXX+.XXX		APPROVALS	DATE		
		DRAWN Yuk-sun Lam	30-5-99		
		CHECKED Dr. Tang		**B PLATE**	
MATERIAL P20		RESP ENG			
FINISH --		MFG ENG			
		QUAL ENG		SIZE A4	DWG. NO. PF 04	REV. 3
DO NOT SCALE DRAWING				SCALE	SW DRAWING FILE:	SHEET4 OF 18

106

⌀8.50

⌀60

⌀4

⌀18.50

SECTION J-J
SCALE 1 : 2.5

250

80

46

24.50

SECTION H-H
SCALE 1 : 2.5

10

300

CAD GENERATED DRAWING.
DO NOT MANUALLY UPDATE

APPROVALS	DATE
DRAWN Yuk-sun, Lam	30-5-99
CHECKED Dr. Tang	
RESP ENG	
MFG ENG	
QUAL ENG	

MATERIAL Steel

FINISH --

Institute of Technology & Engineering
Massey University

Clamping plate A

SIZE A4	DWG. NO. PF 05	REV. 3
SCALE	SW DRAWING FILE:	SHEET 5 OF 18

300

125 $^{+0.050}_{-0.050}$

Ø40

250

A ← → A

A

SECTION A-A

Ø13 — 10

Ø8

24.50

UNLESS OTHERWISE SPECIFIED DIMENSIONS ARE IN INCHES TOLERANCES ARE: FRACTIONS DECIMALS ANGLES + .XX+ .XX + 1 .XXX+.XXX		CAD GENERATED DRAWING. DO NOT MANUALLY UPDATE		Institute of Technology & Engineering Massey University	
		APPROVALS	DATE		
		DRAWN Yuk-sun, Lam	30-5-99		
		CHECKED Dr. Tang		Clamping plate B	
MATERIAL Steel		RESP ENG			
FINISH ---		MFG ENG			
				SIZE A4	DWG. NO. PF 06	REV. 3
DO NOT SCALE DRAWING		QUAL ENG		SCALE	SW DRAWING FILE:	SHEET16 OF 18

SECTION A-A

108

UNLESS OTHERWISE SPECIFIED DIMENSIONS ARE IN INCHES TOLERANCES ARE:		CAD GENERATED DRAWING. DO NOT MANUALLY UPDATE		Institute of Technology & Engineering Massey University	
FRACTIONS DECIMALS ANGLES		APPROVALS	DATE		
+ .XX+ .XX + 1 .XXX+.XXX		DRAWN Yuk-sun, Lam	8-4-1999		
		CHECKED Dr. Tang		Ejector bar	
MATERIAL MOLD STEEL		RESP ENG			
FINISH --		MFG ENG			
				SIZE A4	DWG. NO. PF 07	REV. 3
DO NOT SCALE DRAWING		QUAL ENG		SCALE	SW DRAWING FILE:	SHEET 07 OF 18
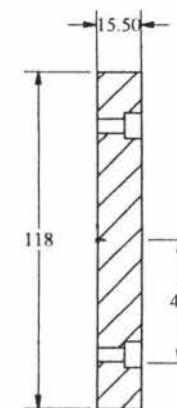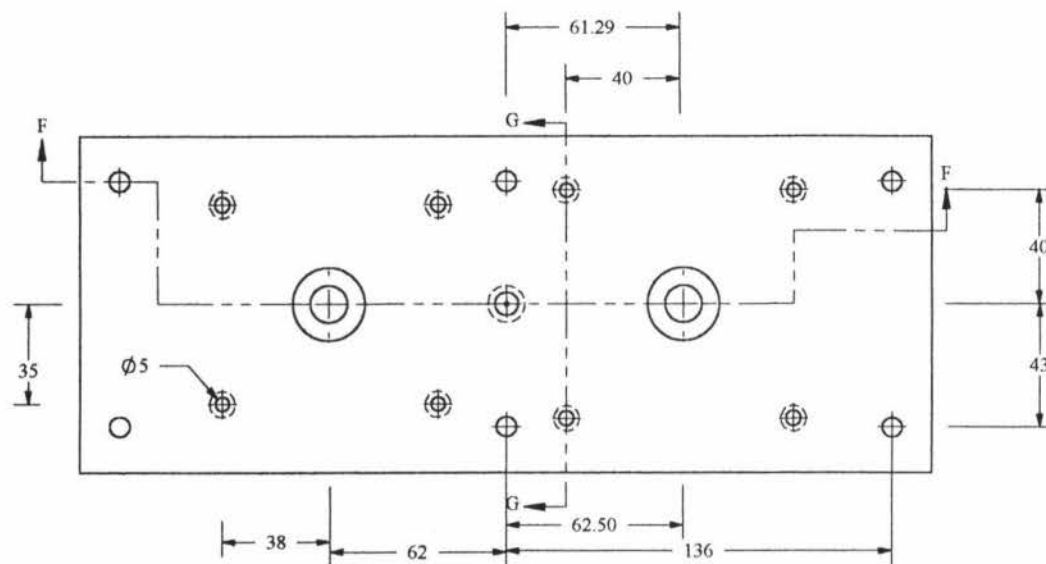
SECTION G-G
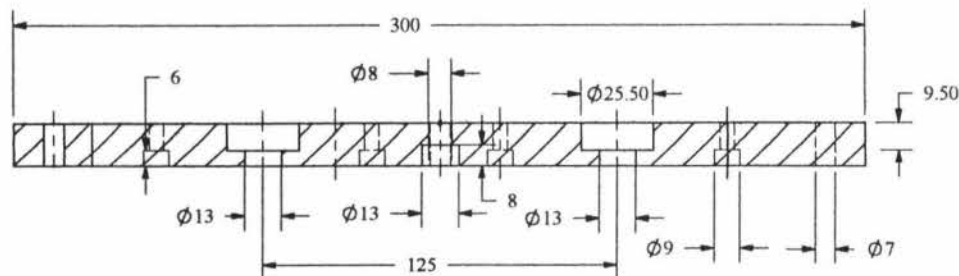
SECTION F-F

UNLESS OTHERWISE SPECIFIED DIMENSIONS ARE IN INCHES TOLERANCES ARE:		CAD GENERATED DRAWING. DO NOT MANUALLY UPDATE		*ie* **Institute of Technology & Engineering** **Massey University**		
FRACTIONS DECIMALS ANGLES + .XX+ .XX + 1 .XXX+.XXX		APPROVALS	DATE			
		DRAWN Yuk-sun, Lam	30-5-99			
		CHECKED Dr. Tang		Pin plate		
MATERIAL Steel		RESP ENG				
FINISH --		MFG ENG		SIZE A4	DWG. NO. PF 08	REV. 3
DO NOT SCALE DRAWING		QUAL ENG		SCALE	SW DRAWING FILE:	SHEET 8 OF 18

$\emptyset 16.50$ — 7.50 — 8

$\emptyset 10.50$ — 40

SECTION A-A

$15\,{}^{0}_{-0.150}$

$\emptyset 125\,{}^{0}_{-0.100}$

A   A

10

$\emptyset 35\,{}^{0}_{-0.100}$

110

1.6

$\phi 35 \, ^{0}_{-0.100}$

$\phi 17.15 \, ^{0}_{-0.050}$

1.75

49.50 $^{0}_{-0.050}$

3

B

B

$\phi 60$

3

6

$10 \, ^{0}_{-0.050}$

$6.20 \, ^{0}_{0}$

$\phi 6 \, ^{0}_{0}$

$5 \, ^{0}_{-0.050}$

$\phi 10$

$23 \, ^{0}_{0}$

$\phi 20$

$1 \, ^{0}_{0}$

SECTION B-B

111

SECTION A-A

UNLESS OTHERWISE SPECIFIED DIMENSIONS ARE IN INCHES TOLERANCES ARE:		CAD GENERATED DRAWING, DO NOT MANUALLY UPDATE		Institute of Technology & Engineering Massey University	
FRACTIONS DECIMALS ANGLES		APPROVALS	DATE		
+ .XX+ .XX + 1 .XXX+.XXX		DRAWN Yuk-sun, Lam	30-5-99		
		CHECKED Dr. Tang		Insert part	
MATERIAL Steel		RESP ENG			
FINISH --		MFG ENG		SIZE A4	DWG. NO. PF-11	REV. 1
DO NOT SCALE DRAWING		QUAL ENG		SCALE	SW DRAWING FILE:	SHEET 11 OF 18

$80_{-0.050}^{0}$

$16_{-0.050}^{0}$

A

A—A

$\emptyset 7$

$\emptyset 13_{-0.050}^{0}$

$1_{-0.050}^{0}$

SECTION A-A

Note
Quantity: 2 pcs.

8 $^{0}_{-0.050}$

89 $^{+0.050}_{-0.050}$

75.00°

6

ϕ13 $^{0}_{-0.250}$

ϕ8 $^{0}_{-0.025}$

UNLESS OTHERWISE SPECIFIED DIMENSIONS ARE IN INCHES TOLERANCES ARE:	CAD GENERATED DRAWING. DO NOT MANUALLY UPDATE		Institute of Technology & Engineering Massey University	
FRACTIONS DECIMALS ANGLES	APPROVALS	DATE		
+ .XX+ .XX + 1 .XXX+.XXX	DRAWN Yuk-sun, Lam	30-5-99		
	CHECKED Dr. Tang		Puller Dia 8	
MATERIAL Steel	RESP ENG			
FINISH --	MFG ENG		SIZE A4 DWG. NO. PF 13	REV. 3
DO NOT SCALE DRAWING	QUAL ENG		SCALE SW DRAWING FILE: SHEET 13 OF 18	

115

$6^{\ 0}_{-0.050}$

$92^{+0.050}_{-0.050}$

$\varnothing 9^{\ 0}_{-0.250}$

$\varnothing 5^{\ 0}_{-0.025}$

	APPROVALS	DATE
DRAWN	Yuk-sun, Lam	30-5-99
CHECKED	Dr. Tang	
RESP ENG		
MFG ENG		
QUAL ENG		

MATERIAL   Steel

FINISH   --

Institute of Technology & Engineering
Massey University

Ejector Dia 5

SIZE A4	DWG. NO. PF 14	REV. 3
SCALE	SW DRAWING FILE:	SHEET 14 OF 18

$6^{+0.050}_{-0.050}$　　$98.60^{+0.050}_{-0.050}$

$\emptyset 9^{+0.050}_{-0.050}$

$\emptyset 5^{+0.050}_{-0.050}$

116

Note
Quantity:4 pcs.

APPROVALS	DATE
DRAWN Yuk-sun, Lam	30-5-99
CHECKED Dr. Tang	
RESP ENG	
MFG ENG	
QUAL ENG	

MATERIAL Steel

FINISH --

Institute of Technology & Engineering
Massey University

Ejector Dia 5

SIZE A4	DWG. NO. PF 15	REV. 3
SCALE	SW DRAWING FILE:	SHEET 15 OF18

SECTION A-A

(15)
(24.50)
(35)
(40)
(49)
(15.50)
(15.50)
(24.50)

(300)
(250)
(204)
(200)

SECTION B-B

(156)
(150)
(118) (120)

117

Item	Description	Quantity
21		
20		
19		
18	Ejector bar	1
17	Pin plate	1
16	Ejector pin 5 Dia; Length: 98.38	4
15	Ejector pin 8 Dia	1
14	Hexagon socket head cap screw (metric) M8	4
13	Hexagon socket head cap screw (metric) M5	2
12	A plate	1
11	B plate	1
10	Support Block	2
9	Clamping plate A	1
8	Insert part	2
7	Support piller	2
6	Spring (MHD)	2
5	Clamping plate B	2
4	Ejector pin 5 Dia; Length: 89.5	7
3	Hexagon socket head cap screw (metric) M8	4
2	Sprue bushing	1
1	Location ring	1

UNLESS OTHERWISE SPECIFIED
DIMENSIONS ARE IN INCHES
TOLERANCES ARE:
FRACTIONS DECIMALS ANGLES

CAD GENERATED DRAWING.
DO NOT MANUALLY UPDATE

APPROVALS	DATE
DRAWN Yuk-sun, Lam	8-4-99
CHECKED L. Tang	
RESP ENG	
MFG ENG	
QUAL ENG	

MATERIAL --
FINISH --
DO NOT SCALE DRAWING

Institute of
Technology & Engineering

Massey University

The Mold set

SIZE B
DWG. NO. PF 16
REV. 3
SCALE
CAD FILE:
SHEET 16 OF 18

H REFERENCES

1. Alting, Leo *'Manufacturing Engineering Processes'* (2nd ed.). (1994). New York: Marcel Dekker, Inc.

2. Klaus Stoeckhert (Eds.) *'Mold-Making Handbook'* (1983). New York: Oxford University Press.

3. Lindner E. and Unger P. (Eds.) *'Injection Molds'* (2nd ed.). (1993). New York: Hanser.

4. Ibrahim Zeid *'CAD/CAM Theory and Practice'* (1991). USA: Mcgraw-Hill, Inc.

5. Georg Mengnes, Paul Mohren. ' *How to Maake Injection Molds'* (2nd ed.) (1993). New York: Hanser.

6. John P. Tanner. *'Manufacturing Enginnering'* (2nd ed.) (1991). UNA: Marcell Dekker, Inc.

7. *'COSMO; Operation Manual'* (1989). HK: Welltec Industrial Equipment Ltd.

8. *'Heidenhain TNC 145: Operation Manual'* Dr. Johannes Heidenhain.

9. Nicholas P. Cheremisinoff (Eds.) *'Handbook of Engineering Polymeric Materal'* (1997). USA: marcel Dekker, Inc.

10. Pual DeGarmo E., J Temple Black and Ronald A. Kohser. ' *Materials and Processes In Manufacturing'* (7th ed.) (1984). USA: Macmillan Publishing Company.

11. ' *C-Mold Design Guide'* (199). USA: Advanced CAE Technology Inc.