

## Design and Simulation of Plastic Injection Moulding Process

Wong, C. T., Shamsuddin Sulaiman, Napsiah Ismail & A.M.S. Hamouda

*Department of Mechanical and Manufacturing Engineering*

*Universiti Putra Malaysia, 43400 UPM, Serdang, Selangor, Malaysia*

*E-mails: choontat@tm.net.my, suddin@eng.upm.edu.my, napsiah@eng.upm.edu.my, hamouda@eng.upm.edu.my*

### ABSTRAK

Kertas ini membentangkan proses reka bentuk satu acuan suntikan plastik untuk menghasilkan satu produk. Ia dibahagikan kepada tiga bahagian iaitu: bahagian reka bentuk, bahagian simulasi dan bahagian analisa. Plastik produk ini direka bentuk kepada dua bentuk yang berlainan tetapi mempunyai fungsi yang sama iaitu satu produk menggunakan fungsi klip dan satu produk lagi menggunakan fungsi lekat. Untuk Reka Bentuk Terbantu Komputer (CAD), perisian komputer yang digunakan untuk meraka bentuk produk dalam 3 dimensi (3D) dan acuan ialah Pro/Engineer (Pro/E). Dua plastik produk yang berlainan akan direka bentuk dalam *insert* yang boleh ditukar-tukar di dalam satu acuan untuk menghasilkan dua produk yang berlainan. Untuk Pembuatan Terbantu Komputer (CAM), Pro/Manufacturing (Pro/MFG) daripada perisian komputer Pro/E digunakan untuk menjanakan beberapa kod kawalan berangka untuk proses pemesinan. Analisis terhadap pengaliran cecair plastik dijalankan dengan menggunakan perisian (MoldFlow – Part Advisor version 4). Daripada analisis ini kedudukan tempat suntikan, suhu bahan dan tekanan suntikan dapat ditentukan. Jangkaan kecacatan produk seperti *weld lines* dan udara terperangkap juga ditemui dan dianalisis.

### ABSTRACT

This paper presents the design of plastic injection mould for producing a plastic product. The plastic part was designed into two different types of product, but in the same usage function. One part is using clip function and another part is using stick function. In the computer-aided design (CAD), two plastic parts were drawn in 3 dimension (3D) view by using Pro-Engineer (Pro-E) parametric software. In the computer-aided manufacturing (CAM), Pro-Manufacturing from Pro-E parametric software was used to develop the machining program. For mould design, the product was designed into two changeable inserts to produce two different types of plastic product in one mould base. Before proceeding to injection machine and mould design, this part was analysed and simulated by using Mold Flow or Part Advisor software. From the analysis and simulation we can define the most suitable injection location, material temperature and pressure for injection. The predicted weld lines and air trap were also found and analysed.

**Keywords:** Changeable insert mould, injection pressure, air traps, injection location, mould design

### INTRODUCTION

Nowadays, the technology of the tool and die fabrication in plastic injection is one of the world's fastest growing industries. Plastic is now used in almost every application, ranging from household articles to space travel, from transportation to packing, from medicine to toys, from bridge building to sports. Generally, injection moulding is a process that forms the plastic into a desired shape by melting the plastic material and forcing the plastic material under pressure into the mould cavity. The shape of the plastic that is desired is achieved by cooling in thermoplastic or by chemical reaction for thermosetting.

Mould design and fabrication is a costly and high technology process because it uses science-based computer-aided engineering (CAE) software to analyse and simulate the

plastic parts, computer-aided design (CAD) software to design the complicated plastic product and computer-aided manufacturing (CAM) to do the programming fabrication to run the computer numeric control (CNC) for milling or latch. Advances in computer technology have led to an increasingly favorable power to cost ratio for computers. So this advantageous and costly technology will improve productivity and process consistency.

Thus, for this fast growing industry, new technologies are vital to ensure that this technology reaches perfection. So computer-aided engineering (CAE) is the assistant to process and calculate the plastic material flow inside the injection moulding.

### Theory

#### How Plastic Fills a Mould

The injection moulding process can be broken into three phases:

1. Filling phase
2. Pressurization phase
3. Compensating phase

#### Filling Phase

When designing plastic parts for the injection moulding process, the important element to understand is how the plastic is filling in the mould. In the mould injection filling phase, molten plastic is injected into the cavity until the cavity is just filled. As plastic flows into the cavity, the plastic in contact with the mould wall quickly freezes and this will create a frozen layer of plastic between the mould and the molten plastic.

Plate 1 shows how the flow front expands as material from behind is pushed forward. The edge of the flow front comes into contact with the mould and freezes. The molecules in the frozen layer are therefore not highly orientated, and once frozen, the orientation will not change.

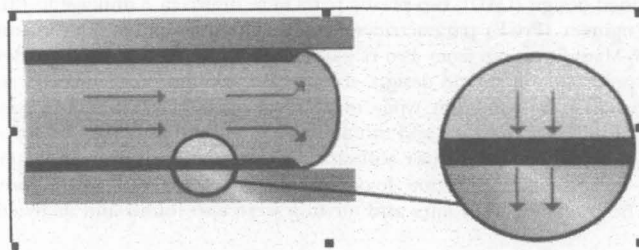


Plate 1: Molten plastic flow inside the mould

The red arrows show the flow direction of the molten plastic. The dark blue layers against the mould walls show the layers of frozen plastic. The green arrows indicate the direction of heat flow from the polymer melt into the mould walls.

The frozen layer gains heat as more molten plastic flows through the cavity, and loses heat to the mould. When the frozen layer reaches a certain thickness, equilibrium is reached. This normally happens early in the injection moulding process, after a few tenths of a second.

#### Pressurization Phase

The pressurization phase will happen after the filling phase completely fills the cavity area. At this stage, all the flow paths will be filled up by molten plastic, the edges and

corners of the filling area in the cavity may not contain plastic. To completely fill out the geometry, extra plastic is pushed into the cavity during the pressurization phase.

Plate 2 shows the difference between the end of the filling phase (left image) and the end of the pressurization phase (right image). Notice the unfilled corners (inside the blue circles) that are left at the end of the filling phase. The yellow cone indicates the polymer injection location and the plastic is red.

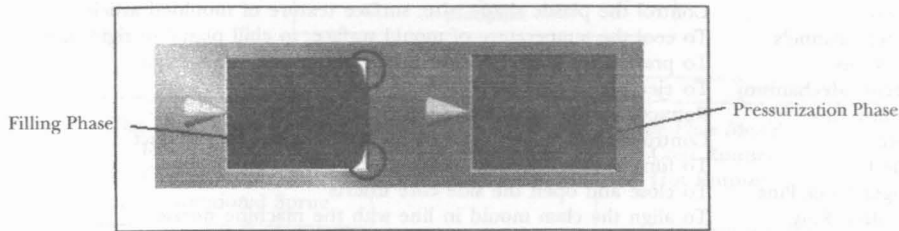


Plate 2: Difference between filling and pressurization phase

#### Compensating Phase

Plastic material has a high volumetric shrinkage, from the melting plastic to solid, it will shrinkage average of 25%. Therefore after the pressurization phase, more material must be injected into the cavity to compensate for the plastic shrinking after it cools. This is the compensating phase (MoldFlow Help Files).

#### Mould Basic Construction

Mould design and mould making can be considered the most critical processes of the injection moulding system. This is because the mould design and mould making have tremendous impact on productivity and product quality. Basically, the function of the mould is two fold. The first is imparting the desired shape to the plasticized polymer, the second is cooling the injection moulded parts. The mould, which contains one or more cavities, consists of two basic parts:

- A stationary mould half on the side where the plastic is injected
- A moving half on the closing or ejector side. The separation of the two moulds is called the parting line.

The mould will determine the final product properties such as size, shape, dimension and finishing surface. Usually, the melted plastic material is filled through the central feed channel called sprue to the cavity. The mould will coincide with the injection machine cylinder nozzle. Table 1 is the summary list for mould component and the function respectively.

#### Mould Design

To start up a new mould design, the designer should know some important points to avoid some mistakes before going further i.e. product outlook design, material usage, correct shrinkage of the material, number of cavities and selection of the mould base. Making a mistake can cause an ill-fitting mould at the final assembly and also make the plastic product out of tolerance and be rejected by the customer. Fig. 1 shows the mould designer's chart.

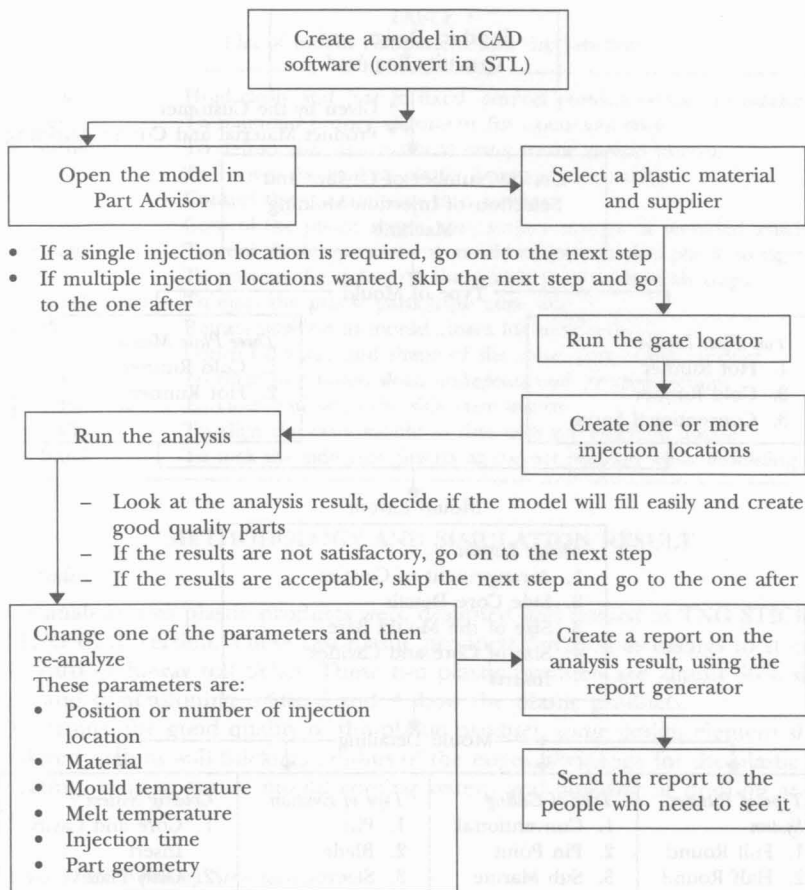


Fig. 2: Procedure diagram for Part Advisor simulation

Moldflow software. *Plate 5* shows the solid model with gate location placing on top of the model. The yellow colour cone shape shows the injection location in this analysis.

In injection and transfer moulding, the gate is considered the portion of the moulded piece that allows the molten resin to flow the runner or from the sprue into the cavity. The gate may be the same size and shape as the runner, but generally it is much smaller. There is an optimum gate size. It should be large enough for suitable fill rate and small enough to seal off and prevent backflow or overpacking.

#### Product Simulation Result

Before starting the simulation process, some information has to be set up, such as material supplier and material grade. This is normally already set in the Moldflow database and the user just selects the specific material specification. Table 2 shows the element used for this analysis:

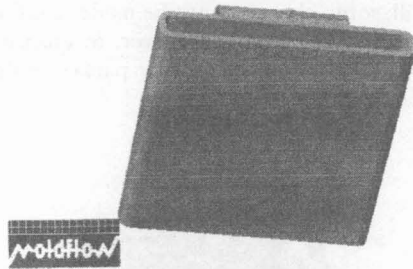


Plate 5: Solid model with gate location

TABLE 2  
Moulding process specification

Part Name	TNG CLIP VERSION
Material Grade	ABS 650
Mould Temperature	50 deg.C
Melt Temperature	235 deg.C
Max Injection Pressure	60 MPa

*Fill Time*

Fill time is the time taken to fill up the part inside the cavity, it is also to show how the plastic material flows to fill the mould. From that we know that the short shot (part of the model which did not fill) part will be displayed. From that result one can also understand how the weld line and air trap will form. Plate 6 shows the material filling into the mould.

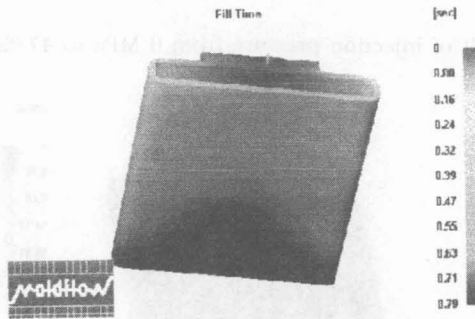
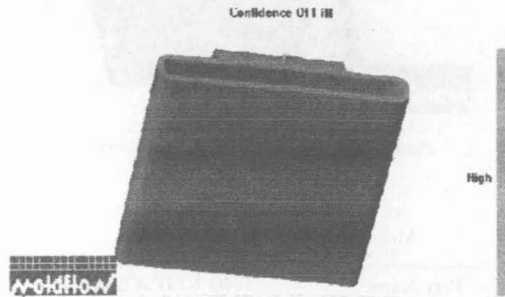


Plate 6: 100% materials fill of plastic part

*Confidence of Fill*

The confidence of fill result displays the probability of the region within the cavity with plastic. This result is derived from the pressure and temperature results. The confidence of fill will display in three colours: green, yellow, red and translucent. Showing in green will definitely fill, in yellow may be difficult to fill or may have quality problem, in red will be difficult to fill or will have quality problem and in translucent will not fill (short shot).

If the cavity does not fill, some changes must be made to either design, gate location, choice of plastic or processing conditions. However, to ensure the finished part is of good quality, the cavity must also be adequately packed with plastic. *Plate 7* shows confidence of fill in this analysis.



*Plate 7: Confidence of fill*

#### *Injection Pressure*

The colour at each place on the model represents the pressure at the place on the model. Two colours show the highest pressure (red) and lowest pressure (blue). The injection pressure can be used in conjunction with pressure drop result. For example, even if a section of a part has an acceptable pressure drop, the actual injection pressure in the same area may be too high. High injection can cause overpacking.

To reduce the chance of this happening, follow these steps:

1. Increase the maximum injection pressure
2. Alter the polymer injection location
3. Alter part geometry
4. Select a different material

*Plate 8* shows the result of injection pressure from 0 MPa to 47.02 MPa.



*Plate 8: Injection pressure*

#### *Pressure Drop*

*Plate 9* shows the pressure drop result. When the place is filled, the pressure will drop from the injection location to the end of the filled part. The pressure drops result uses

a range of colours to indicate the region of highest-pressure drop (coloured red) through to the region of lowest pressure drop (coloured blue). The pressure drop is one factor used to determine the confidence of fill result.



Plate 9: Pressure drop

#### Air Traps

Plate 10 shows the air traps result after analysis. The small blue bubbles are showing the air traps in the parts. Air traps result shows the regions where the melt stops at a convergence of at least 2 flow fronts or at the last point of fill, where a bubble of air becomes trapped. To prevent air traps occurring when converging flow fronts surround and trap a bubble of air, balance flow paths by either:

1. Using flow leaders / deflectors
2. Changing part wall thickness
3. Changing polymer injection locations

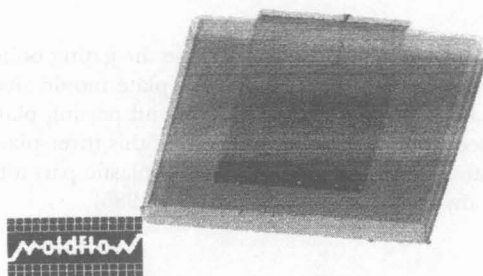


Plate 10: Air trap

#### Mould Development

After doing all the Moldflow simulation for the product, mould design will be preceded based on the simulation result. Only one mould base is used to produce two different plastic parts; this is because almost 70% of the plastic product shapes are similar, so changeable inserts are used in this case. Core insert will be as permanent insert and change the slider and cavity insert for TNG Clip version and TNG Stick version. Plate 11 shows the cavity and core insert for TNG Clip and Stick version.

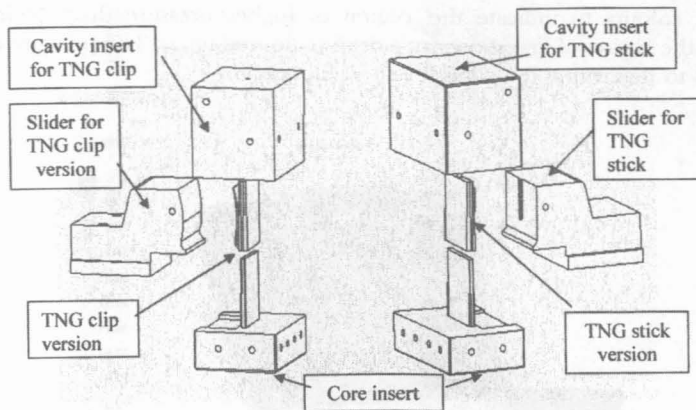


Plate 11: Cavity, core and slider insert

### Mould Base Size

For this project, three-plate type system mould base is used with pinpoint gating. The sizes for the mould base are 250 mm x 250 mm with FCI type, A-plate size 100 mm, B-plate size 70 mm, Spacer block size 100 mm and with stripper plate (referred from LKM (Lung Kee Metal) mould base handbook). The function of the stripper plate is to remove the runner from the plastic part and the plastic part will be considered a good product with no need to touch up.

For this mould base just one cavity will be injecting each time, the molten material of plastic will flow through sprue to the cavity. After cooling, the plastic parts will harden and stick to the core and then be ejected by using the shoulder-type ejector pin.

### Three Plate Mould

Three plate moulds are used in this project because the gating point to inject the molten plastic is small and the runner system for a three-plate mould sits on a second parting plane parallel to the main parting plane. This second parting plane allows the runners and sprue to be ejected when the mould is open. In this three-plate mould, the runners will be ejected separately from the cavities and the plastic part after injection does not need to go through any touch up process (Rosato 1986).

### Slider Core

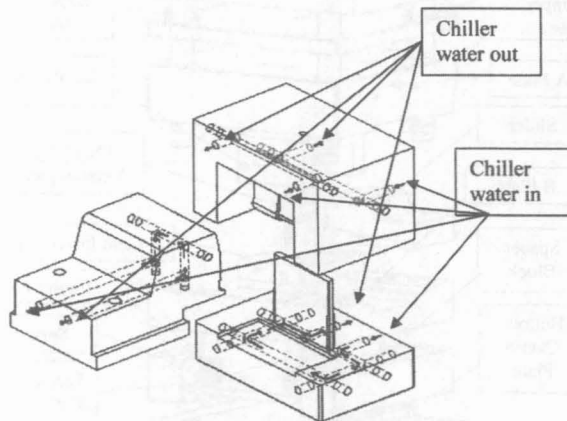
Slider core is used when the plastic parts are design undercut for example, side hole, side boss, etc. Slider core is stationary attached to plate B (refer to Plate 14) and two angular pins are attached to plate A. When the mould opens, the angular pin will move the slide core horizontally out from the mould position. The amount of the slide core movement will depend on the angle and how far the slider moves on the angular pin. By the time the mould is opened, the slide core has moved enough so that the ejector pin has been fully withdrawn from the plastic part.

### Cooling System (Water Channel)

A water channel is to control the temperature of the mould surface and to chill up the molten plastic material to become rigid states that eject from the core side. For this



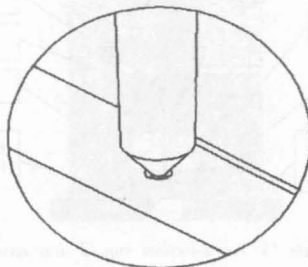
project the water channels are designed to be drilled along the length of cavities, core and slider. The cooling system is vertically to the mould. *Plate 12* shows the flow of the water channel in the mould insert.



*Plate 12: Water channel flow inside mould insert*

#### *Gating System*

The gate is the connection between the runner system and the moulded part. It must permit enough material to flow into the mould to fill out the part plus additional material as is required to overcome the normal thermal shrinkage of the part. In this project, pin point gates are used, when the mould is open the part will automatically be removed from the runner and there is no need to touch up for the plastic product. *Plate 13* shows the pinpoint gate system.



*Plate 13: Pin point gate system*

#### *Complete Mould Assembly*

*Plate 14* shows the completed mould assembly for TNG Clip Version. *Plates 15* and *16* show the cross-section for mould close assembly and open assembly position.

#### *CAD/CAM for Mould Design*

In the plastic industry, CAD/CAM has emerged to the point where it now shows the promise of being one of the most significant technological advances of the century.

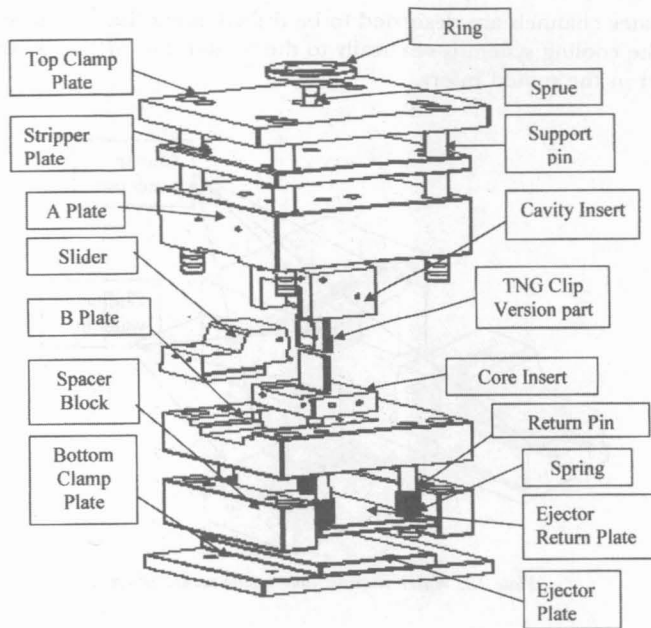


Plate 14: Completed mould assembly

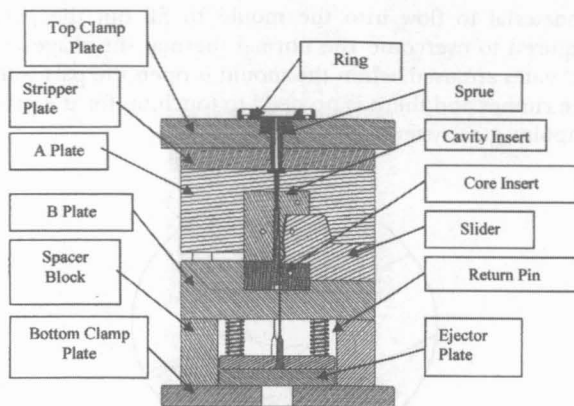


Plate 15: Cross-section mould close assembly

CAD/CAM is enabling the creative energies of plastic part and mould designers to be spent in producing better designs in a shorter time period rather than in doing repetitive mould design tasks.

#### The Advantages of CAD/CAM

The advantages of the CAD/CAM in mould design are:

- a) CAD/CAM technologies are normally used for the numerically controlled (NC) machining technology to fabrication for the moulds and also its ability to create three-dimensional product models in the data base and to generate automatically.

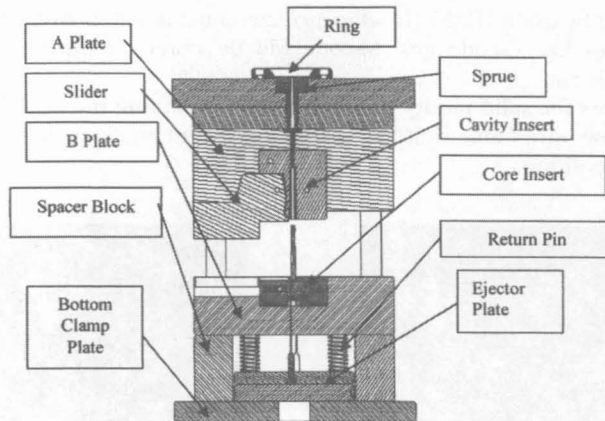


Plate 16: Cross-section mould open assembly

- b) CAD/CAM can help designers to speed up design for the plastic part and mould design process and reduce the long lead-time.
- c) CAD/CAM used in the repetitive and routine tasks of mould making will enhance the quality of work life for those skilled mould makers by providing more challenge and job satisfaction.
- d) Easy for users to redefine for part geometry; this means it can enhance quality in the mould design by reducing in the number of errors.

*Application of CAD/CAM and the Methods for Mould Design*

In this project, the type of CAD/CAM software employed is Pro/Engineer 2000i. Pro/Moldesign is an extension of Pro/Engineer to the mould manufacturing environment, covering all aspects of production mould design such as mould bases, runner system creation, waterlines, ejector-pin holes, shrinkage and hundreds of other features. The Plate 14 for mould assembly is using Pro/Moldesign to create and design the mould. Pro/NC and Pro/MFG are the manufacturing extensions of Pro/Engineer into CNC production arena, and have a wide breadth of capabilities covering from 3 to 5 axis milling, 2 and 4 axis wire EDM, turning, laser, flame, etc.

After designing all the moulds, Pro/NC and Pro/MFG for the manufacturing will play an important role again in the process of creating the detail drawing needed and generating NC toolpaths to machine the actual mould. The NC toolpaths are derived from the product model database. Product paths completed in 3-D model will be directly extracted from the mould design database for manufacturing, for example, core inserts from the mould database that want to do some manufacturing. Before starting CAD/CAM some of the important features have to be considered like tool feed rate, spindle speed, step depth, material to mill, Plunger speed, coordinate system, and what kind of end mill used.

To start CAD/CAM, dimension of work piece has to be decided first and then setting the coordinate system to the work piece. The work piece coordinate must be of the same setting with the coordinate in the CNC machine. The wrong setting will cause the machine to crash with the work piece. Decide what kind of manufacturing process to use, for example profiling, volume, pocketing, hole-making and conventional surface. After selecting the manufacturing process, Pro/NC will convert computer code to

machining code by using ICAM (Intelligence computer-aided manufacturing) software. Machining codes like G-code and M-code will be stored in the CNC and then the manufacturing is run.

Plate 17 shows the solid model simulation before running the actual manufacturing and Plate 18 shows wire frame simulation and the red line in Plate 18 is the tool running point to point position.

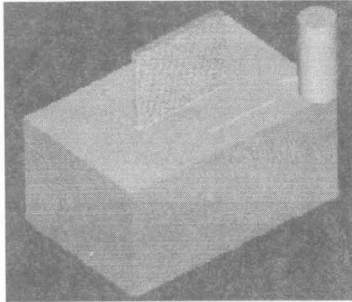


Plate 17: Solid model simulation

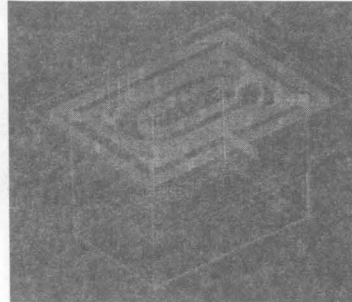


Plate 18: Wire frame simulation

Pro/MFG will create the data necessary to drive an NC machine tool to machine a Pro/Moldesign part. It does this by providing the tool to let the manufacturing engineer follow a logical sequence of steps to progress from a design model to ASCII CL data files that can be post-processed into NC machine data. The CAD/CAM illustration in Fig. 3 can basically be summarized the Pro/MFG process (Pro/MFG, R18).

#### DISCUSSION AND CONCLUSION

The conceptual design on the plastic parts is based on the need of the car driver. The plastic parts holder casing is to place the Touch N' Go card or PLUS hi-way ticket for the driver to have easy access to it when he/she reaches the toll. In this project the plastic parts were designed into two different types of product but design in one changeable insert mould base. Just using one mould base will reduce the cost and time in manufacturing. Pro/Moldesign, Pro/MFG and Pro/NC from Pro/ENGINEER are used to design the mould and fabricate the plastic parts. In the mould-making industries, accuracy and tolerance are the important issues; so using CAD/CAM will enhance quality of the work in the mould design and reduce the number of errors in manufacturing.

From the analysis simulation, Moldflow provides sufficient information results such as fill time, injection pressure and pressure drop. With this result, users can avoid the defect of the plastic in actual injection such as sink mark, hesitation, air traps, and overpacking. The analysis will also help the mould designer to design a perfect mould with minimum modifications and it will also reduce the mould setup time. With this analysis and simulation it will help to reduce time and cost.

## Design and Simulation of Plastic Injection Moulding Process

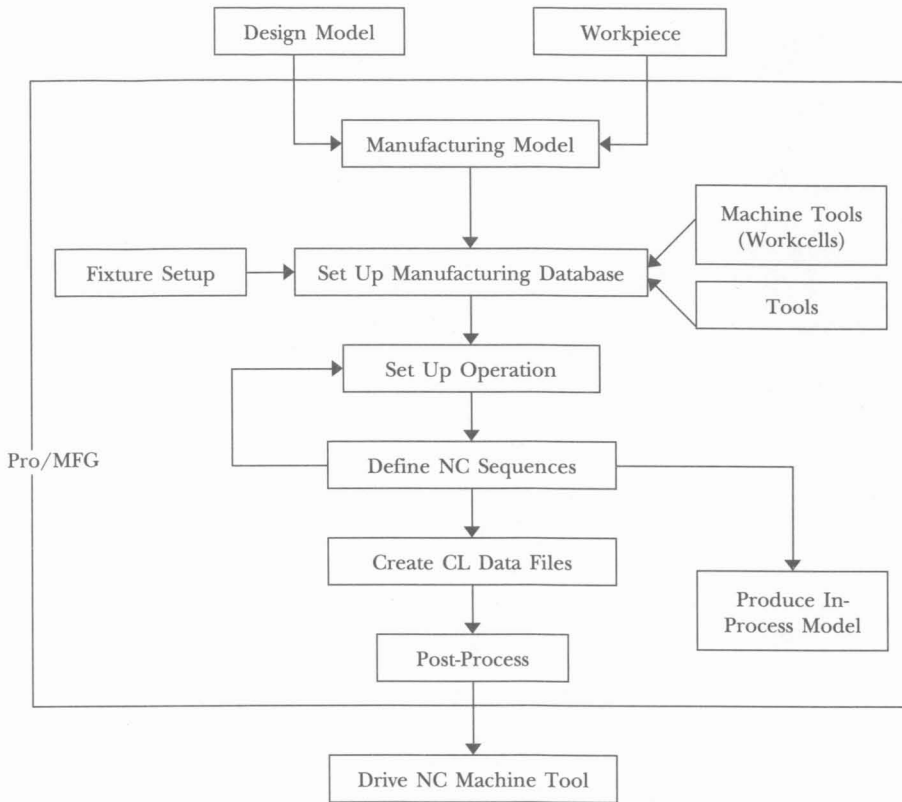


Fig. 3: Summarizes the Pro/MFG process

### REFERENCES

- FRADOS, J. 1976. *Plastic Engineering Handbook (The Society of the Plastic Industry)*. Fourth edition. Van NostRand Reinhold Company.
- Moldflow Help Files.
- CAMPBELL, P.D.Q. 1996. *Plastic Component Design*. First edition. New York, USA: Industrial Press Inc.
- Pro/MFG and Pro/NC-CHECK Users guide, Release 18.
- ROSATO, D.V. 1986. *Injection Molding Handbook*. Applied Publisher Ltd. Essex.
- BECK, R.D. and V. NOSTRAND. 1980. *Plastic Product Design*. Second edition. Reinhold Company.