International Journal of Engineering Research-Online
A Peer Reviewed International Journal
Articles available online http://www.ijoer.in

Vol.3., Issue.5., 2015
(Sept.-Oct.)

RESEARCH ARTICLE

ISSN: 2321-7758

DESIGN OPTIMIZATION OF GATE LOCATION, MOULD DESIGN AND COST ESTIMATION OF BASE PLATE EMF LOAD CELL

ANUMANDLA MADHUKAR[1], V.NARASIMHA REDDY[2]
[1]M.Tech Scholar, [2]Asst.Professor
Department of Mechanical Engineering, Malla Reddy Engineering College (Autonomous)
Hyderabad, India

ANUMANDLA MADHUKAR

ABSTRACT

For plastic components manufacturing process used is Injection moulding. While doing this manufacturing process we have to face some problems in filling process, clamping, cooling, and amount of material to inject into the cavity area. Due to the above problems there is wastage of material, time, poor component quality.

In this paper, mould flow analysis is conducted by using plastic advisor. In this paper, a comparative analysis has been performed by taking one, two and three gate locations for a EMF load cell component used in Weighing machines. In this study problems faced in the injection moulding processes can also be avoided. Results compared are Actual Injection Time, Actual Injection pressure, Weld lines, Air Traps, shot Volume, Estimated cycle time, Filling clamping force, Packing Clamp Force. By using above result optimizing Injection moulding manufacturing process with least defects is possible.

Mould Flow Plastic Advisor simulation software from Pro/E was used for the analysis and the optimum gate location is found with least defects. The better parameters are optimized from the mould flow analysis. Total mould base is designed which is ready for production and total die cost is estimated.

Key words; Mould flow plastic advisor, EMF load cell

1.1 INTRODUCTION TO INJECTION MOULDING

Injection molding is the most commonly used manufacturing process for the fabrication of plastic parts. A wide variety of products are manufactured using injection molding, which vary greatly in their size, complexity, and application. The injection molding process requires the use of an injection molding machine, raw plastic material, and a mold. The plastic is melted in the injection molding machine and then injected into the mold, where it cools and solidifies into the final part.

Injection molding is used to produce thin-walled plastic parts for a wide variety of applications, one of the most common being plastic housings. Plastic housing is a thin-walled enclosure, often requiring many ribs and bosses on the interior. These housings are used in a variety of products including household appliances, consumer electronics, power tools, and as automotive dashboards. Other common thin-walled products include different types of open containers, such as buckets. Injection molding is also used to produce several everyday items such as toothbrushes or small plastic toys. Many medical devices, including valves and syringes, are manufactured using injection molding as well.

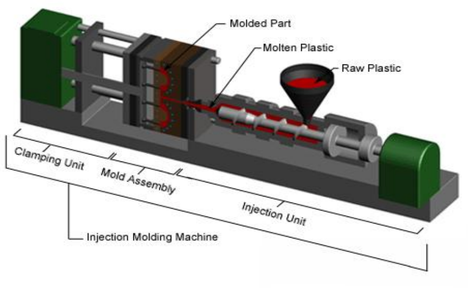


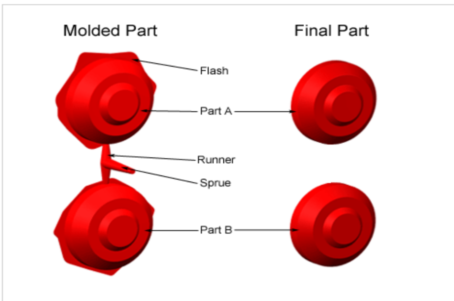
Fig:1 Injection molding machine

1.2 PROCESS CYCLE

The process cycle for injection molding is very short, typically between 2 seconds and 2 minutes, and consists of the following four stages:

1. *Clamping* - Prior to the injection of the material into the mold, the two halves of the mold must first be securely closed by the clamping unit. Each half of the mold is attached to the injection molding machine and one half is allowed to slide. The hydraulically powered clamping unit pushes the mold halves together and exerts sufficient force to keep the mold securely closed while the material is injected. The time required to close and clamp the mold is dependent upon the machine.This time can be estimated from the dry cycle time of the machine.
2. *Injection* - The raw plastic material, usually in the form of pellets, is fed into the injection molding machine, and advanced towards the mold by the injection unit. During this process, the material is melted by heat and pressure. The molten plastic is then injected into the mold very quickly and the buildup of pressure packs and holds the material. The amount of material that is injected is referred to as the shot. The injection time is difficult to calculate accurately due to the complex and changing flow of the molten plastic into the mold. However, the injection time can be estimated by the shot volume, injection pressure, and injection power.
3. *Cooling* - The molten plastic that is inside the mold begins to cool as soon as it makes contact with the interior mold surfaces. As the plastic cools, it will solidify into the shape of the desired part. However, during cooling some shrinkage of the part may occur. The packing of material in the injection stage allows additional material to flow into the mold and reduce the amount of visible shrinkage. The mold can not be opened until the required cooling time has elapsed. The cooling time can be estimated from several thermodynamic properties of the plastic and the maximum wall thickness of the part.
4. *Ejection* - After sufficient time has passed, the cooled part may be ejected from the mold by the ejection system, which is attached to the rear half of the mold. When the mold is opened, a mechanism is used to push the part out of the mold. Force must be applied to eject the part because during cooling the part shrinks and adheres to the mold. In order to facilitate the ejection of the part, a mold release agent can be sprayed onto the surfaces of the mold cavity prior to injection of the material. The time that is required to open the mold and eject the part can be estimated from the dry cycle time of the machine and should include time for the part to fall free of the mold. Once the part is ejected, the mold can be clamped shut for the next shot to be injected.

After the injection molding cycle, some post processing is typically required. During cooling, the material in the channels of the mold will solidify attached to the part. This excess material, along with any flash that has occurred, must be trimmed from the part, typically by using cutters. For some types of material, such as thermoplastics, the scrap material that results from this trimming can be recycled by being placed into a plastic grinder, also called regrind machines or granulators, which regrinds the scrap material into pellets. Due to some degradation of the material properties, the regrind must be mixed with raw material in the proper regrind ratio to be reused in the injection molding process.



Injection molded part

1.3 EQUIPMENT

Injection molding machines have many components and are available in different configurations, including a horizontal configuration and a vertical configuration. However, regardless of their design, all injection molding machines utilize a power source, injection unit, mold assembly, and clamping unit to perform the four stages of the process cycle.

1.4 INJECTION UNIT

The injection unit is responsible for both heating and injecting the material into the mold. The first part of this unit is the hopper, a large container into which the raw plastic is poured. The hopper has an open bottom, which allows the material to feed into the barrel. The barrel contains the mechanism for heating and injecting the material into the mold. This mechanism is usually a ram injector or a reciprocating screw. A ram injector forces the material forward through a heated section with a ram or plunger that is usually hydraulically powered. Today, the more common technique is the use of a reciprocating screw. A reciprocating screw moves the material forward by both rotating and sliding axially, being powered by either a hydraulic or electric motor. The material enters the grooves of the screw from the hopper and is advanced towards the mold as the screw rotates. While it is advanced, the material is melted by pressure, friction, and additional heaters that surround the reciprocating screw. The molten plastic is then injected very quickly into the mold through the nozzle at the end of the barrel by the buildup of pressure and the forward action of the screw. This increasing pressure allows the material to be packed and forcibly held in the mold. Once the material has solidified inside the mold, the screw can retract and fill with more material for the next shot.

1.5 CLAMPING UNIT

Prior to the injection of the molten plastic into the mold, the two halves of the mold must first be securely closed by the clamping unit. When the mold is attached to the injection molding machine, each half is fixed to a large plate, called a platen. The front half of the mold, called the mold cavity, is mounted to a stationary platen and aligns with the nozzle of the injection unit. The rear half of the mold, called the mold core, is mounted to a movable platen, which slides along the tie bars. The hydraulically powered clamping motor actuates clamping bars that push the moveable platen towards the stationary platen and exert sufficient force to keep the mold securely closed while the material is injected and subsequently cools. After the required cooling time, the mold is then opened by the clamping motor. An ejection system, which is attached to the rear half of the mold, is actuated by the ejector bar and pushes the solidified part out of the open cavity.

2. MATERIALS AND METHODS

There are many types of materials that may be used in the injection molding process. Most polymers may be used, including all thermoplastics, some thermosets, and some elastomers. When these materials are used in the injection molding process, their raw form is usually small pellets or a fine powder. Also, colorants may be added in the process to control the color of the final part. The selection of a material for creating injection molded parts is not solely based upon the desired characteristics of the final part. While each material has different properties that will affect the strength and function of the final part, these properties also dictate the parameters used in processing these materials. Each material requires a different set of processing parameters in the injection molding process, including the injection temperature, injection pressure, mold temperature, ejection temperature, and cycle time. A comparison of some commonly used materials is shown below.

In most of the injection molding production unit's, during production problem is faced when filling the material inside the cavity area. The manufacturers change the processing parameters by trial and error method. By trial and error method, there is lot of wastage of material, time, and power. At the same time price of the component will also be increased.

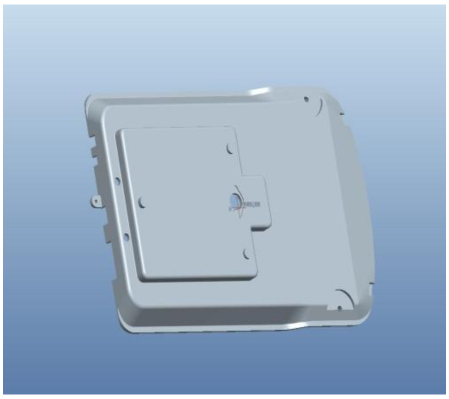
In this project above problem is rectified by taking software support of Plastic Advisor. Using this software, the quality of filling of material by changing processing parameters before going to manufacturing can be analyzed.

Mould flow analysis is also by done by taking one gate location, two gate locations and three gate locations.

Material name	Abbreviation	Trade names	Description	Applications
Acetal	POM	Celcon, Delrin, Hostaform, Lucel	Strong, rigid, excellent fatigue resistance, excellent creep resistance, chemical resistance, moisture resistance, naturally opaque white, low/medium cost	Bearings, cams, gears, handles, plumbing components, rollers, rotors, slide guides, valves
Acrylic	PMMA	Diakon, Oroglas, Lucite, Plexiglas	Rigid, brittle, scratch resistant, transparent, optical clarity, low/medium cost	Display stands, knobs, lenses, light housings, panels, reflectors, signs, shelves, trays
Acrylonitrile Butadiene Styrene	ABS	Cycolac, Magnum, Novodur, Terluran	Strong, flexible, low mold shrinkage (tight tolerances), chemical resistance, electroplating capability, naturally opaque, low/medium cost	Automotive (consoles, panels, trim, vents), boxes, gauges, housings, inhalors, toys
Cellulose Acetate	CA	Dexel, Cellidor, Setilithe	Tough, transparent, high cost	Handles, eyeglass frames
Polyamide 6 (Nylon)	PA6	Akulon, Ultramid, Grilon	High strength, fatigue resistance, chemical resistance, low creep, low friction, almost opaque/white, medium/high cost	Bearings, bushings, gears, rollers, wheels
Polyamide 6/6 (Nylon)	PA6/6	Kopa, Zytel, Radilon	High strength, fatigue resistance, chemical resistance, low creep, low friction, almost opaque/white, medium/high cost	Handles, levers, small housings, zip ties
Polyamide 11+12 (Nylon)	PA11+12	Rilsan, Grilamid	High strength, fatigue resistance, chemical resistance, low creep, low friction, almost opaque to clear, very high cost	Air filters, eyeglass frames, safety masks
Polycarbonate	PC	Calibre, Lexan, Makrolon	Very tough, temperature resistance, dimensional stability, transparent, high cost	Automotive (panels, lenses, consoles), bottles, containers, housings, light covers, reflectors, safety helmets and shields

Material name	Abbreviation	Trade names	Description	Applications
Acetal	POM	Celcon, Delrin, Hostaform, Lucel	Strong, rigid, excellent fatigue resistance, excellent creep resistance, chemical resistance, moisture resistance, naturally opaque white, low/medium cost	Bearings, cams, gears, handles, plumbing components, rollers, rotors, slide guides, valves
Acrylic	PMMA	Diakon, Oroglas, Lucite, Plexiglas	Rigid, brittle, scratch resistant, transparent, optical clarity, low/medium cost	Display stands, knobs, lenses, light housings, panels, reflectors, signs, shelves, trays
Acrylonitrile Butadiene Styrene	ABS	Cycolac, Magnum, Novodur, Terluran	Strong, flexible, low mold shrinkage (tight tolerances), chemical resistance, electroplating capability, naturally opaque, low/medium cost	Automotive (consoles, panels, trim, vents), boxes, gauges, housings, inhalors, toys
Cellulose Acetate	CA	Dexel, Cellidor, Setilithe	Tough, transparent, high cost	Handles, eyeglass frames
Polyamide 6 (Nylon)	PA6	Akulon, Ultramid, Grilon	High strength, fatigue resistance, chemical resistance, low creep, low friction, almost opaque/white, medium/high cost	Bearings, bushings, gears, rollers, wheels
Polyamide 6/6 (Nylon)	PA6/6	Kopa, Zytel, Radilon	High strength, fatigue resistance, chemical resistance, low creep, low friction, almost opaque/white, medium/high cost	Handles, levers, small housings, zip ties
Polyamide 11+12 (Nylon)	PA11+12	Rilsan, Grilamid	High strength, fatigue resistance, chemical resistance, low creep, low friction, almost opaque to clear, very high cost	Air filters, eyeglass frames, safety masks
Polycarbonate	PC	Calibre, Lexan, Makrolon	Very tough, temperature resistance, dimensional stability, transparent, high cost	Automotive (panels, lenses, consoles), bottles, containers, housings, light covers, reflectors, safety helmets and shields

MODEL OF EMF LOAD CELL



In most of the injection molding production units, during production problem is faced when filling the material inside the cavity area. The manufacturers change the processing parameters by trial and error method. By trial and error method, there is lot of wastage of material, time, and power. At the same time price of the component will also be increased.

In this project above problem is rectified by taking software support of Plastic Advisor. Using this software, the quality of filling of material by changing processing parameters before going to manufacturing can be analyzed.

Mould flow analysis is also by done by taking one gate location, two gate locations and three gate locations.

3. MOULD FLOW ANALYSIS

Mould flow, 3D solids-based plastics flow simulation that allows plastics part designers to determine the manufacturability of their parts during the preliminary design stages and avoid potential downstream problems, which can lead to delays and cost overruns. Following are the benefits:

- Optimize the part wall thickness to achieve uniform filling patterns, minimum cycle time and lowest part cost Identify and eliminate cosmetic issues such as sink marks, weld lines and air traps.
- Determine the best injection locations for a given part design

Mould flow analysis gives you the ability to maintain the integrity of your product designs. It provides you the tools to quickly optimize part designs and check the impact of critical design decisions on the manufacturability and quality of the product early in the design process.

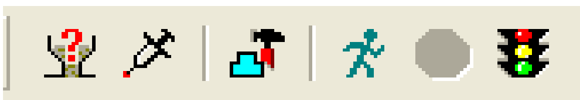
There is no need to:

- Compromise the aesthetics of your design concept for manufacturability;
- Go through a lengthy trial and error process to find the most suitable material to produce the part with the highest possible quality and the lowest possible cost
- Find out during trial runs that the produced part has visual blemishes, such as sink marks, weld lines, air traps or burn marks.

3.1 PLASTC ADVISOR

Plastic advisor is an add on analysis package for Pro/Engineer, especially for plastic injection moulding. For doing this analysis, after drawing the required object Select applications > plastic advisor > pick datum point for injection location > ok.

There will be an analysis tool bar



The icons available are:

1) moulding parameter icon
2) specify injection location
3) special analysis icon
4) Start analysis icon.

When we click on the moulding parameter icon a separate window opens and there we can select the required materials and we can specify the require injection conditions like mold temperature, injection pressure....

After specifying the materiel and processing conditions, click on the specify injection location icon and specify the required location point. Then click on the run analysis icon and select plastic flow analysis and click on start button.

Results obtained are

1) Plastic flow
2) Fill time
3) Confidence of fill
4) Injection pressure
5) Pressure drop
6) Flow front temperature
7) Quality prediction
8) Weld lines and air traps.

- Fill time result shows the flow path of the plastic through the part by plotting contours which join regions filling at the same time. These contours are displayed in a range of colors red indicates the first region to fill, blue to indicate the last region to fill. A short shot is a part of the model that did not fill, will be displayed as translucent.
- This result shows how the injection pressure distribution throughout the mold at specific time.
- Pressure drop is opposite to that of the injection pressure. This result shows how the pressure is dropped throughout the cavity at the specific time.
- The confidence of fill result displays the probability of a region within the cavity filling with plastic at conventional injection molding conditions. This result is derived from the pressure and temperature results.
- The flow front temperatures represent the material temperature at each point as that point was filled. The result shows the changes in the temperature of the flow front during filling.
- Weld lines result indicates the presence and location of weld and weld lines in the filled part model. These are places where two flow fronts have converged.
- The air trap 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. The regions highlighted in the result are positions of possible air traps.

- Confidences of fill results indicate that there is a low Confidence of Fill due to the high pressure required to fill the cavity. The advice topic on pressure drop indicates that to fill the part you will need to change one of the following things:
  1) material
  2) injection location
  3) injection pressure
  4) part geometry
  5) injection pressure
- The Quality Prediction plot uses several results to give an indication of the overall quality of the part. When the Quality Prediction result indicates a problem the following things have to be checked and changed if necessary.
  1) Shear stress is too high
  2) Cooling time is too high
  3) Pressure drop is too high
  4) Flow front temperature is too high
  5) Flow front temperature is too low
- Weld lines are not all critical and do not always lead to visual or structural defects. Weld lines are often unavoidable but we can change the flow of the material to place them in areas that are less sensitive. Weld lines should only be in areas that are not visually or structurally sensitive
- Air traps appear when flow fronts meet, trapping pockets of air. Like weld lines these are also unavoidable so we should take care that air traps should only be in locations where the air can be vented
- The Confidence of Fill and the Quality Prediction results are good starting points when checking an analysis. They will show you where there may be problems with the part.
- If flow front is too low in an area where weld lines are present, the weld lines may appear worse. In areas where the flow front temperature is too high, material degradation and surface defects may occur. We must make sure that the flow front temperature is always within the recommended temperature range for the material used.

3.2 INPUT PARAMETERS

	INJECTION PRESSURE (MPa)	MOLD TEMPERATURE (^{0}C)	MELT TEMPE RATURE (^{0}C)
CASE 1	180	40	200
CASE 2	200	60	230
CASE 3	230	80	300

MATERIAL – ABS - ACRYLONITRILE-BUTADIENE-STYRENE

GATE 1

FILL TIME

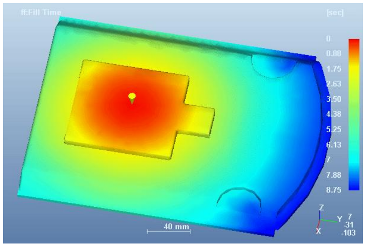


FIG 2: FILL TIME OF GATE 1

GATE2

PLASTIC FLOW

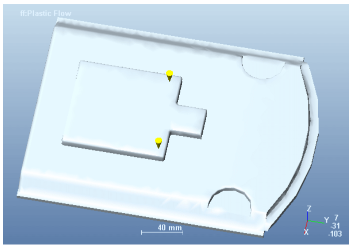


FIG 3: PLASTIC FLOW OF GATE 2

GATE 3

GLASS MODEL

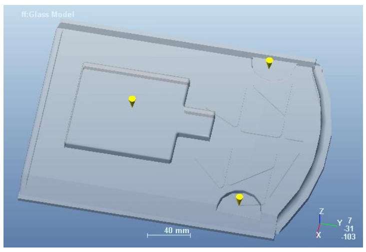


FIG 4: GLASS MODEL OF GATE 3

4. RESULTS AND DISCUSSION:

4.1 SINGLE GATE

	CASE 1	CASE 2	CASE 3
CONFIDENCE	MEDIUM	MEDIUM	MEDIUM
FILL TIME (Secs)	8.75	7.43	5.22
INJECTION PRESSURE (MPa)	7.89	4.95	2.21
WELD LINES	NO	NO	NO
AIR TRAPS	YES	YES	YES
CYCLE TIME (Secs)	118.56	158.29	247.16
QUALITY PREDICTION	MEDIUM	MEDIUM	LOW
SINK MARKS	9 % of your model was found to be prone to sink marks.	9 % of your model was found to be prone to sink marks.	9 % of your model was found to be prone to sink marks.

4.2 TWO GATES

	CASE 1	CASE 2	CASE 3
CONFIDENCE	MEDIUM	MEDIUM	MEDIUM
FILL TIME (Secs)	9.05	7.58	5.17
INJECTION PRESSURE (MPa)	6.23	4.11	1.77
WELD LINES	YES	YES	YES
AIR TRAPS	YES	YES	YES
CYCLE TIME (Secs)	117.76	157.69	247.11
QUALITY PREDICTION	MEDIUM	MEDIUM	LOW
SINK MARKS	8 % of your model was found to be prone to sink marks.	8 % of your model was found to be prone to sink marks.	9 % of your model was found to be prone to sink marks.

4.3 THREE GATES

	CASE 1	CASE 2	CASE 3
CONFIDENCE	MEDIUM	MEDIUM	MEDIUM
FILL TIME (Secs)	8.59	7.71	5.20
INJECTION PRESSURE (MPa)	5.40	3.37	1.68
WELD LINES	YES	YES	YES
AIR TRAPS	YES	YES	YES
CYCLE TIME (Secs)	117.12	157.78	247.11
QUALITY PREDICTION	MEDIUM	MEDIUM	LOW
SINK MARKS	9 % of your model was found to be prone to sink marks.	9 % of your model was found to be prone to sink marks.	9 % of your model was found to be prone to sink marks.

MOULD DESIGN CALCULATIONS

NOMENCLATURE:

F_c = Clamping Force

P_c = Cavity Pressure

A_P = Projected area

N = No of cavities

Clamping Force

$F_c = P_c x A_P x N$

=2.34x23447.3x1

=54866.682kg/s=5.5929tonnes

Mass=vxſ=420485x0.000001

=0.420485kg

=420.485gm

Shot Capacity = 420.485 x $(\frac{1.45}{1.05})$

= 580.66

Cooling:

Cooling Capacity = 580.66 x 255.77

= 148517.905 x 5^{o}

= 742589.525

Cycle Time = 10s (min time of eject) + fill time

= 10s + 6.43

= 126.59s+6.43

= 133.02sec

No of Components Produces / hour production

= $(\frac{3600}{133.02})$

= 27.06 c/h

TOTAL DIE COST

1. Material cost
   - Rs. 125742/-
2. Machining cost
   - Rs. 120500/-
3. Transportation
   - Rs. 2000/-
4. Risk cost
   - Rs.36936.3/-
   (15% of material & machine)
5. Profit (20% of above 4)
   - Rs.57035.66/-

   Total Die cost - Rs.342214/-

Estimating component cost

1. Component material cost - 1500gms component weight
   1kg ABS – Rs.150/-
2. Each component material cost – Rs.220/-
3. Per shift production cost
   – Rs. 4000/-
   For each component production time is 252.28secs
   No.of components per each shift - (7hrs)= 7x60x60/252.28
   = 100
4. Production cost per shift
   = 4000 =Rs.40/-
   100
   Total cost = Material cost + Production cost + Packing charges + Printing
   = 220+40+30+5
   Total cost per component = Rs.295/-

5. CONCLUSION

- In this thesis, the optimal process parameters and the optimal number of gates required to fill the component EMF load cell with least defects is analyzed. The number of gates taken is one, two and three. The process parameters considered in three cases,
- **Case-1:** Max Injection Pressure: 180MPa, Mold Temperature: 40 deg C, Melt Temperature: 200 deg C,
- **Case-2:** Max Injection Pressure: 200MPa, Mold Temperature: 60 deg C, Melt Temperature: 230 deg C and
- **Case-3:** Max Injection Pressure: 230MPa, Mold Temperature: 80 deg C, Melt Temperature: 300 deg C.
- The material is ABS Plastic.
- By observing the analysis results of ABS, use of two gates is better since 8 % of model was found to be prone to sink marks when two gates is used but when single gate or three gates are used 9% of model was found to be prone to sink marks respectively. When the process parameters are considered, considering Case 1 and Case 2 parameters is better the quality prediction is medium.
- Mould is designed and modeled in 3D modeling software Pro/Engineer. The cost of mould is estimated and cost per component is estimated. The total cost of die is Rs. 3,42,214/- abd cost per each component is Rs. 295/-.

6. REFERENCES

[1]. Plastic Injection Molding: Manufacturing Process Fundamentals by Douglas M. Bryce

[2]. Injection Mould Design: A Design Manual for the Thermoplastics Industry by R G W HYPERLINK "http://www.alibris.com/search/books/author/R-G-W-Pye?aid=4073329&cm_sp=author-_-searchAuthLink"Pye

[3]. The Mould Design Guide by Peter Jones

[4]. Rosato, D. Injection Moulding Handbook. Second Edition, Chapman & Hall, London, 1995.

[5]. Lam, Y.C., Jin, S. Optimization of the gate location for plastic injection moulding. Journal of Injection Moulding Technology, 2001

[6]. Sahputra, I.H. Comparison of two flow analysis software for injection moulding tool design. Proceedings of the International Conference on Industrial Engineering and Engineering Management, 2-4 Dec. 2007, Singapore.

[7]. Saman, A.M., Abdullah, A.H., Nor, M.A.M. Computer simulation opportunity in plastic injection mould development for automotive part. International Conference on Computer Technology and Development, 13-15 Nov. 2009, Kota Kinabalu, Malaysia.