International Journal of Technical Innovation in Modern Engineering & Science (IJTIMES)
Impact Factor: 5.22 (SJIF-2017), e-ISSN: 2455-2585
Volume 4, Issue 6, June-2018

MOLD FLOW ANALYSIS OF AN INJECTION MOLDED INTRAVENOUS COMPONENT

Nagendra J[1]

[1]*Department of Mechanical Engineering, NHCE, VTU ,Email id:nagendrajayaram@gmail.com*

Abstract— Multi Cavity Die Design of Plastic injection mould for Intravenous (IV) Regulator component was designed for Acrylonitrile butadiene styrene (ABS) plastic material. The study was focused on multi-cavity design, feeding System, ejector system, selection of suitable machine based on clamping force calculation and manufacture of mold to produce better-quality components. This project begins with the study of component for shape, size, critical parameters to aid in proper selection of Parting surface, Ejection system and Feed system. The 3D model of the component was analysed using the Mold flow plastic insight software which helps in design and manufacture of the mold. The Plastic flow analysis was been done using Plastic part Adviser software in order to check the confidence of fill, fill time, injection pressure, flow front temperature, the pressure drop and detection of defect like air traps for the molding component. These parameters were considered while designing of the mould to validate the design and to manufacture better-quality component. Computer aided design of core and cavity as well as the assembly of the tool was done using Unigraphics NX4 for better visualization. And, detailed drawings by Auto Cad 2007 software to serve as an input to manufacturing. The designed mold produced error free components during the tests and was validated accordingly.

Keywords — Mold Flow Analysis, Feeding system, Ejector system. Multi cavity.

I. INTRODUCTION

The work deals with the Design and manufacturing of sixteen cavity injection mould tool for "IV Regulator" of a medical part**.** This is a component used as a speed adjusting device in blood bag set. ABS material is used for the production of this component. The tool has to be designed to produce a good quality component considering the ease of manufacturability, assembly and Positive ejection of the component. The following points were considered as the primary objectives, a) To design a multi cavity injection mold for IV Regulator. The mold has to be designed to produce a good quality component considering the ease of manufacturability, assembly and positive ejection of the component within the minimum possible time and cost. (Design of moldings tool is carried out using software *Unigaphics NX* 4). b). To verify Plastic flow analysis by Plastic Adviser software to validate design and to check manufacturability of the plastic part. c). To Prepare detail drawing of the mold (by Auto CAD 2007) for manufacturing. The Methodology followed for this project work is,

- Study of the component: The study of the component is the most important and the first step for the design. The component drawings are carefully scrutinized to extract the maximum possible amount of information. The important information available is the critical dimensions, line of draw, parting line, suitable ejection system, and core.
- Solid model of the component: Solid modelling of component is done using Unigraphics NX considering all the critical dimensions.
- Design Calculations: It is carried out to determine the various design parameters that determine the final mould like clamp force required during injection, number of cavities, wall thickness of inserts, guide pillar design, design of feeding system, cooling calculations.
- Mould flow analysis: Mold flow analysis is carried before tool design to determine the confidence of fill, injection pressure, melt temperature distribution, possible weld lines and air traps etc. which helps in design and setting input parameters to the injection molding machine during production etc.

- Core and Cavity extraction: Extraction is done by providing proper shrinkage, tolerance is provided to the dimensions to which a cavity and core should be manufactured in order to produce a part of desired shape and size. The usual way to decide on the amount of shrinkage is to consult data supplied by the material manufacturer. While designing shrinkage is provided depending on the type of plastic material to be chosen for injection molding. A thicker piece will have a higher shrinkage value compared to thinner section.
- Solid modeling of the tool: 3-D modeling of the entire mould is done using Unigraphics NX. The required dimensions are determined by calculation, which is used during modeling of the tool.
- Selection of tooling materials: The steel used in the manufacture of mould varies depending on the applications. Proper material selection and proper combination of alloys in varying percentages are required for finished moulds. Characteristics of polishing ability, high dimensional stability and hardenability are directly introduced to the alloy steel.
- Tool manufacturing process: Raw material is transferred into the finished tool in this activity generally, during manufacturing. Each part of the tool is manufactured by referring to their respective design drawings. Before taking up manufacturing, the tool drawings are studied and a process plan for each part is prepared. Attention is bestowed on manufacturability during design stage itself. Tolerance and hardness required and related aspects are selected for implementation.
- Tool tryout and troubleshooting: After the tool is manufactured and assembled, the tool is 'tried' to see that component produced is true to the geometry and dimensions as specified in the drawing. Tryout is a procedure where the tool is subjected to actual working condition and the performance of the tool is noted. After the tool has been tried out, the component is thoroughly inspected for various defects. If any defects are found, it is suitably reworked.
- Cost Estimation: Mould cost estimation is done by considering weight of parts and the their respective cost per kg , machining cost, heat treatment cost, design cost and tool try out cost is all put together, the probable cost is computed.

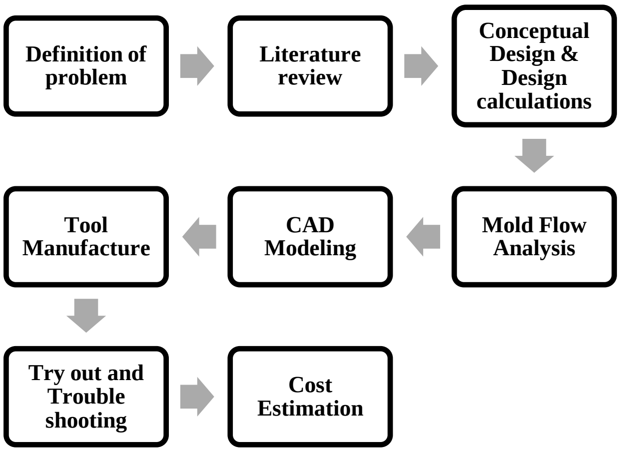

Fig. 1 Project Methodology Flow Chart

II. LITERATURE

An extensive literature survey has been conducted to collect all the information relevant to the project.

Core and Cavity extraction

J.Y.H. Fuh, et al, **[1],** recommended 6 steps for parting design module in the 3D modeling. Step1: Determination of the parting direction – Direction of the core and cavity open are the parting direction. Step 2: Recognition and patching the 'through' hole – if there is hole in the product, must indicate the parting location for the hole and determine the parting surface for these hole. Step3: Determination of parting lines and the extruding direction – Parting line is the intersection between the two groups surfaces there are core and cavity. Step 4: Generation of the parting surfaces – a surface are mating surface of the core and cavity. Usually the parting surface as splitting surfaces to separate the mould into two block, cavity and core. Step 5: Creation of containing box – Containing box is used to create core and cavity. The size of the containing box is based on the dimension of the object, strength mould, molding parameter. Step 6: Generation of mores and cavities – the box will be split into two mould halves, the core block and cavity block. The empty space inside the containing box is the place where the molten plastic will ne inject and solidify.

Runner Balancing for Multi cavity Molds

Kevin Alam, et al [2] , suggested on optimization of runner balancing in multi cavity molds. For runner systems with equivalent secondary runner diameters and lengths, cavities farthest from the injection point will experience greater shrinkage because the longer path length causes an increase of pressure drop. Accordingly, the traditional approach to runner balancing reduces the runner diameters of the closest cavities and increases the runner diameters of the furthest cavities to balance the pressure drops among them. A larger primary runner diameter also reduces pressure drops to the furthest cavities. Such designs minimize differences in shrinkage among the cavities and decrease the effects of variation on the runner balancing process. When runner length was manipulated to achieve runner balancing, it became desirable to increase the length of the runners closest to the injection point to increase the pressure drops to these cavities, while decreasing the lengths of the runners furthest from the injection point to decrease the pressure drops to the furthest cavities. Both strategies help balance the pressures among the cavities, which reduce the need to increase the diameters of the primary runner and of the furthest secondary runners to achieve that balance. This ultimately reduces the runner system volume and associated material costs. Thus, the manipulation of the runner lengths and diameters is more effective in balancing the runner system than the manipulation of runner diameters alone.

Cooling Channel Design and Gate Design

D.E. Dimla, M. et al, [3], states that conformal cooling channels in injection molding design are advantageous then straight cooling passages. His Results from an investigation of the effectiveness of conformal channels through the construction of three different moulds with and without conformal cooling showed that the latter technique led to significant improvements and a general reduction of the cycle time while ameliorating heat transfer. According to this study, as with most manufacturing fields, production time and costs are strongly correlated. The longer it takes to produce parts the more are the costs, and with injection molding production industries cooling time is often taken as the indicator of cycle time. Improving cooling systems will reduce production costs. A simple way to control temperature and heat interchange is to create several channels inside the mould where a cooler liquid is forced to circulate. Conventional machining like CNC drilling can be used to make straight channels. Herein, the main problem is the impossibility of producing complicated channels in three-dimension, especially close to the wall of the mould. This produces an inefficient cooling system because the heat cannot be taken away uniformly from the mould and the different shrinkage causes warpage and cooling time increase. On the other hand, if the cooling channels can be made to conform to the shape of the part as much as possible, then the cooling system the cycle time can be significantly reduced with cooling taking place uniformly in all zones. **Charles Shaw, [4],** states that the melts front moves through the mould, molten plastic contacts with wall of the mould and immediately begins to cool and solidify. Thus the first resin to contact the wall is further along in the cooling process at the end of filling than the last resin to be injected. In additional, thicker portions of the mould cool slower than thinner portions. This difference in cooling time aerates uneven volumetric contraction in the part, which lead to warpage of the plastic product. Thus gate point should always be provided at the higher thickness of the part. If the part does not design properly, the warpage to the product will exceed the required tolerances, the produced part is fail.

Mold Flow Analysis

M.K.A.Ariffin, et al [5], states that, integrating the computer software such as CAD/CAM, Moldflow makes mould making industries becomes easier compare to the traditional method which has been prove time consuming and very costly. A mould is actually the part of the injection moulding which produce the product. The product quality depends on the mould itself. Thus any defect or design errors will affect the quality of end product. So it is important to verify the mould using mould flow analysis for defect free product.

Manufacturing and Process Planning

Shaw C. Feng et, al (2003) [6] states that the process planning is a key activity for designers to evaluate manufacturability and manufacturing cost and time in the early product development stage plastic injection mould industry. So it is important to integrate the design and process planning message exchange to save time and cost of development. **Z.lou, et al [7],** states the new procedure for the development of the mold. The procedure should implement through an integrated, knowledge-based system to improve mould design efficiency and shorten design time. Normally, the conventional designer will separate the mould design are divide into three stages: detailed design of the mould, design of the mould-base and ordering the mould base from the manufacturer. In the new mould development process was purpose by X.Ruan, firstly the design mould-base according to dimension of the product. Secondly, begin to design the detail of the mould while ordering the mould–base from the manufacturer. The step of the design the detail of mould and ordering the mould-base are carry out at the same time , enable to shortening the product development cycle.

Wong Choon Tat et, al (2002) [8], Injection moulding is where the molten plastic fill into the mould(desired shape) so the Injection of plastic into the mould has been categories into 3 phase: Filling Phase, Pressurization Phase and Compensating Phase. Filling Phase – this phase where the molten material are filling the cavity of the product inside the mould. The molten material contact to the wall of the mould will solidify very fast than other area. Pressurization Phase – Even all the cavity is filled with the molten material but at the corner or edges, or the angle may not fulfill with the molten plastic because of the path is too narrow, so extra pressure and material need to complete the fullness of the cavity. Compensating Phase – Plastic is a high shrinkage material. From the molten phase to the solid phase it will shrinkage around 25% after the pressurization phase. **S.H. Tang, et, al (2007) [9]** suggested that, the sequence of manufacturing of mold by following steps can reduce time and helps in easy of manufacturing. According to him, the sequences of machining processes like drilling, milling and tapping are important to get a good quality of work and also save the machining time. In machining process, there are some steps that should be highlight and pay attention. For drilling process, especially for cooling channel, the drilling machine that has been used is radial drilling machine instead of vertical drill press. This is because the spindle speed for radial drilling machine is controlled by gear while the vertical drill press is used belt. Thus, the radial drilling machine can produce higher drilling torque as compared to the vertical drill press. **Suqin Yao (2003) [10]** states that process sequence divide into 2 principles Principle 1: The setup sequence must be arranged according to the sequence of design features. Principle 2: The setup sequence must be arranged according to the feature's pre-defined process sequence. The process sequence should always follow the basic manufacturing principle, there is doing rough cuts first, and semi and finish cuts in a prescribed order.

By literature review it can be understood that, runner balancing in multi cavity mold design depends on length and diameter rather than diameter alone and how these two should be manipulated for effective balancing, Steps to be followed for Core and Cavity extraction, Conformal cooling has more effective then straight cooling system, gate point provision, Mold flow analysis which helps to validate mold design, suggests the possible defects in the mold and Manufacturing and process planning aspects suggest the sequence should be followed for manufacture of mold in order to reduce machining time and cost.

III. METHODOLOGY AND EXPERIMENTAL INVESTIGATION

3.1 Component Description: The component IV Regulator is used in with blood bag to adjust the speed of fluid flowing through the tube. It should be an aesthetic component. The cavity surface has an aesthetic value and hence it should have glossy finish and no visible marks are allowed on the surface like witness marks. Isometric view of the component is shown in Fig.3.1. No Weld lines sink marks or other visible defects are permitted in the component. Surfaces must be smooth and clean.

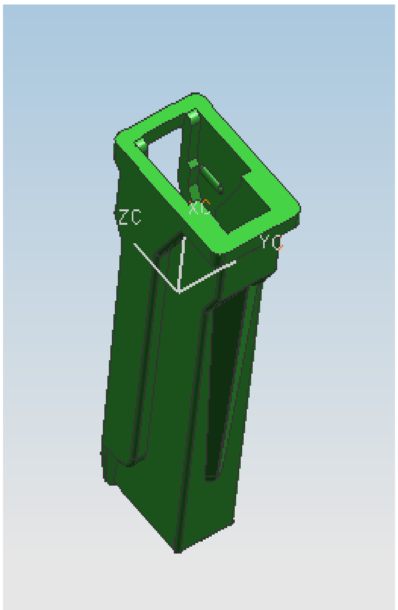

Fig.3.1 3D Model of IV Regulator

Plastic Flow Analysis: Plastic flow analysis is carried out using the Mold Flow Plastic Adviser software. The adviser is designed to determine the manufacturability of the plastic part design. It allows us to create a model of the component in a CAD package and then read the model into the advisor. We need to input some information including the polymer and injection location. Other settings such as material temperature, mould temperature, pressure, are selected automatically according to the material properties. The following parameter are evaluated during the analysis, Confidence of fill, Fill time, Injection pressure, Pressure drop, Flow front temperature, The weld lines, Air traps formation. The results are shown as colour shaded pictures, colour plots. These are set norms for acceptability of results of these analyses. If the results of the analysis are within set norms, then the part and mould design becomes acceptable. Also we can review the inputs and try to resolve the molding problems.

3.2 Inputs to Plastic Advisor Software:

Plastic Material : ABS
Injection Pressure : 110 Mpa
Melt Temperature : 220°C
Mold Temperature : 85°C

3.3 Confidence of Fill

The Confidence of fill result displays the probability of plastic filling a region within the cavity under set injection moulding conditions. This result is derived from the pressure and temperature results. Fig. 3.1 shows the confidence of fill.

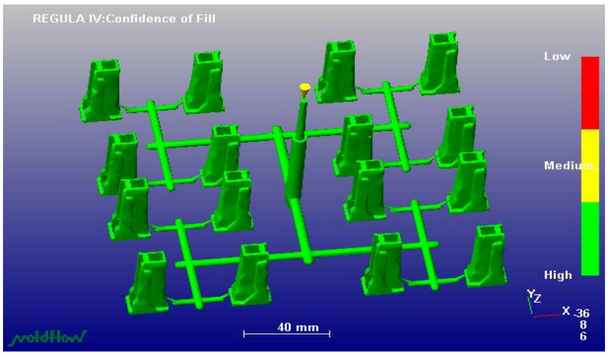

Fig.3.1 Confidence of fill

The colour displayed in the confidence of fill results are interpreted as, Green colour section will definitely fill, Yellow colour section may be difficult to fill or having quality problems and Red colour section will be difficult to fill or have quality problems. Confidence of fill is determined by both pressure and melts temperature we may need to adjust both processing conditions before an acceptable confidence of fill result is obtained.

3.3 Fill Time

This result shows the flow path of the plastic material through the part. Each colour shown represents the part of the mould which was being filled at the same time. A short shot will be displayed as translucent. At Fill time is shown in Fig.3.2. The fill time uses a range of colours to indicate the first region to fill (coloured red) through the last region to fill (coloured blue). The primary use of the fill time result is to determine if all the flow paths fill at the same time (balanced flow paths).

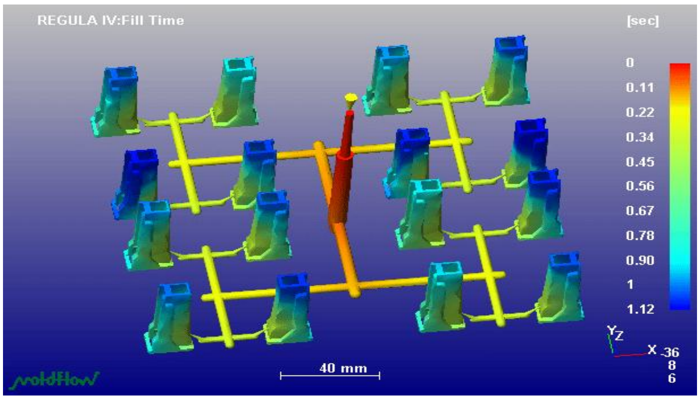

Fig: 3.2 Fill Time

3.4 Injection Pressure

The colour at each place on the model represents the pressure at each place on the model, at the moment the part is filled completely. The Pressure at end of fill result is used with the Pressure drop result to locate areas where the actual injection pressure is too high. Injection pressure result uses a range of colours to indicate the region of lowest pressure (blue) through the region of highest pressure (red). Injection Pressure is as shown in Fig.3.3. High injection can cause over packing and also to improve the confidence of the part filling, the injection pressure can be decreased.

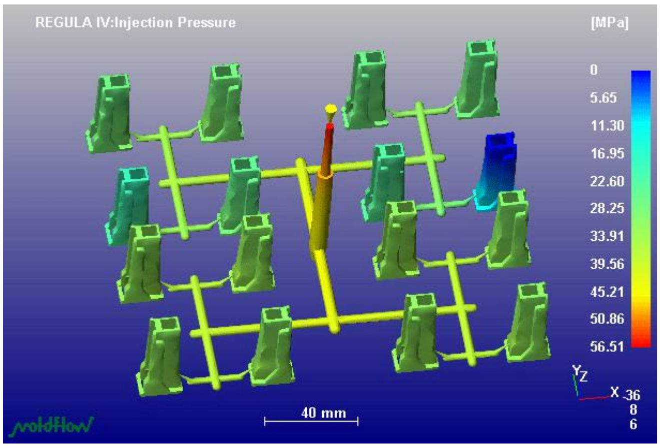

Fig.3.3 Injection Pressure

3.5 Flow Front temperature

The colours represent the material temperature at each point as that point was filled. The result shows the changes in the temperature of flow front during filling. Fig.3.4 Shows flow front temperature variation. Flow front temperature uses a range of colours to indicate the region of lowest temperature (coloured blue) through to the region of highest temperature (coloured red). If the flow front temperature is too low in a thin area of the part, hesitation or short shot may occur. In areas where the flow front temperature is too high, material degradation and surface defects may occur. Make sure that the flow front temperature is always within the recommended temperature range for the polymer we are using.

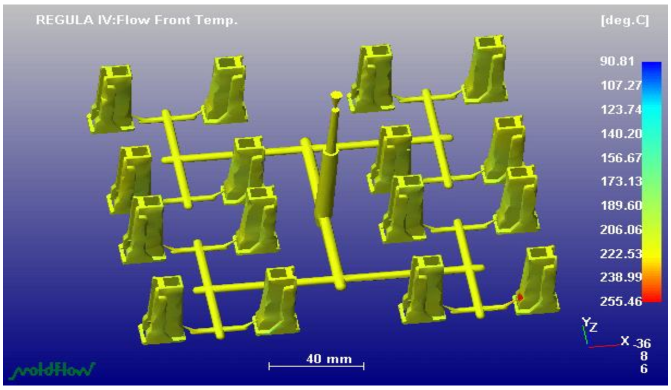

Fig.3.4 Flow front temperature

3.6 Pressure Drop

The Pressure drop result uses a range of colours to indicate the region of highest pressure drop through to the region of lowest pressure drop. This result indicates how much pressure is necessary to fill the different areas of the part. As shown in the following diagram, the colour at each place on the model represents the drop in pressure from the injection location to that place on the model at the moment that place was filled, that is, the pressure required to force material to flow to that point. The Pressure drop result is one factor that is used to determine the Confidence of fill result. Pressure drop is shown in Fig.3.5.

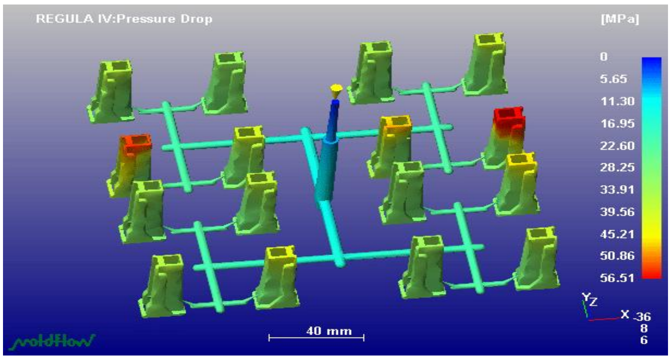

Fig.3.5 Pressure Drop

3.7 Air Traps

An air trap is air that is caught inside the mold cavity. It becomes trapped by converging polymer melt fronts or because it failed to escape from the mold vents, or mold inserts, which also act as vents. Air-trap locations are usually in areas that fill last. Lack of vents or undersized vents in these last-to-fill areas are a common cause of air traps and the resulting defects. Air trap is shown in Fig.3.6. Entrapped air will result in voids and bubbles inside the molded part,a short shot or surface defects such as blemishes or burn marks. To eliminate air traps, you can modify the filling pattern by reducing the injection speed, enlarging venting, or placing proper venting in the cavity. Vents are typically positioned at discontinuities of mold material, such as at parting surfaces, between the insert and mold wall, at ejector pins, and at mold slides.

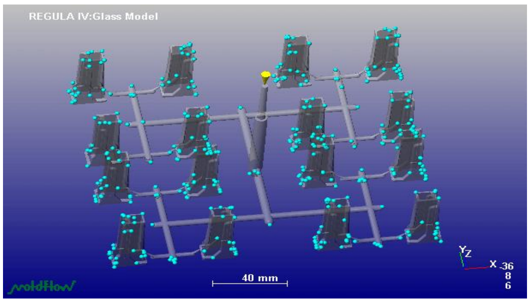

Fig.3.6 Air Trap

IV. RESULTS AND CONCLUSION

During the trials following defects were occurred and they were solved by adjusting some parameters in the machine. Following Table no.4.1 shows the injection molding defects occurred for the part during trial and their remedies.

Sl.No.	Defects	Remedies
1.	Short Fill	Provide sufficient feed Increase melt & nozzle Temperature Provide sufficient vents in the mold
2.	Sink Mark	Increase dosing Increase holding pressure
3.	Weld Line	Increase injection speed Provide air vents in the mold
4.	Flash	Increase Clamping force Decrease injection speed Decrease injection pressure
5.	Black Specks	Reduce Excessive melt temperature Reduce Excess back pressure
6.	Warp	Reduce mold temperature Decrease injection fill rate
7.	Burning	Clean vents, increase vent size Reduce melt temperature

Table 4.1 Defects and remedies

Conclusions

- The design of the injection mould tool was completed with the required standards and quality set by the organization.
- The Plastic flow analysis was carried out to the feed system of the injection tool and the component. The analysis revealed that the design of the feed system was satisfactory. This was judged by evaluating the results obtained from the various parameters of the analysis such as injection pressure, fill time, flow front temperature, air traps. The results indicated that the molding could be manufactured with minimum molding defects.
- The tool was manufactured using the in-house tool room facilities according to standards specified by the organization.
- Trial out of the injection mould tool revealed that moldings obtained were satisfactory. Hence the design and manufacturing of the injection mould tool for the components was successful. The Parts produced were free from any defects. Tool cost estimation of the injection mould tool suggested that the tool fabrication was financially feasible and acceptable by the organization.

REFERENCES

[1] L.Kong, J.Y.H. Fuh, K.S. Lee, X.L. Liu, L.S. Ling,Y.F. Zhang, and A.Y.C. Nee, Determination of optimal parting directions in plastic injection mould design, CIRP Ann. Manuf. Technol. 2003, pp-429–432.

[2] Kevin Alam, Musa R. Kamal, A robust optimization of injection molding runner balancing, McGill University, Montreal, July 2005, pp-1934-1944.

[3] D.E.Dimla, M.Camilotto,F.Miani, Design and optimization of conformal cooling channels in injection molding tools School of Design, Engineering and Computing, Bournemouth University, UK ,2004, pp 1294-1300.

[4] Charles Shaw, Department of Manufacturing Engineering, Gate location design in injection moulding of an automobile junction Hong Kong ,1999, pp110-115.

[5] M.K.A.Ariffin, S.Sulaiman, M.M.Hamdan, Z.Leman, N.Ismail, Design and Simulation of Plastic Injection Moulding Process,University Putra Malaysia,2004, pp 85 – 99.

[6] Shaw,Sun y, The application of U shaped grooves for cooling of injection molds,Vol 216, 2002, pp 1561-1573.

[7] Z.lou ,H,Jiang, X. Ruan ,Mould manufacturing sequence in injection moulding, 2004, pp1673-1685.

[8] Wong, C. T., Shamsuddin, Sulaiman Design and Simulation of Plastic Injection Moulding Process, Malaysia,2004, pp 85-99.

[9] S.H. Tang, Chew W, Wiecek, M. M., & Zhang, J, Journal of Mechanical Design, 121, 1999,pp 179 [9]

[10] Suqin Yo, Deb, K. Multi-objective optimization using evolutionary algorithms. New York: John Wiley and Sons, 2001.

[11] Rees, H. Understanding injection mold design. Cincinnati: Hanser-Gardner Publications, 2001.

[12] Jansen, K. M. B., & Titomanlio, G. Polymer Engineering and Science,36,1996,pp-2029.

[13] J. C. Lin and C. C. Tai, The Runner Optimization Design of a Die-Casting Die and the Part Produced, International Journal of Advance Manufacturing Technology, Vol. 14, No. 2, Feb 1998,pp. 133-145.

[14] M. Zhai, Y. C. Lam and C. K. Au, Runner sizing and weld line positioning for plastics injection molding with multiple gates, Engineering with Computers, Vol. 21, No. 3, May, 2006, pp.218-224.

[15] E. Sachs, Production of injection molding with conformal cooling channels using the three dimensional printing process, Polym. Eng. Sci. 40,2000, pp 1232–1247.

[16] M. Ring, An investigation of effectiveness of conformal cooling channels and selective laser sintering material in injection moulding tools, RPD ,2002, pp 1–5.

[17] X. Xu, The design of conformal cooling channels in injection molding tooling, Polym. Eng. Sci. 2001, pp 1265–1279.

[18] S. Steadman, K.M. Pell, Expert systems in engineering design: an application for injection moulding of plastic parts, J. Intel. Manuf.,1995, pp 347–353.

[19] W.G.Ryim,Y.I. Kim, S.K. Chang, A systematic optimization of gate location and processing conditions in injection molding, J. Appl. Mech. Trans. ASME 227, 1997,pp 111–120.

[20] I. Pandelidis, Q. Zou, Optimization of injection molding design Part I: Gate location optimization, Polym. Eng. Sci. 30,1990, pp 873–882.

[21] Dominick V Rosato and Donald V Rosato,"Injection Molding Handbook", 2nd Edition,1998, Hanser Publisher.

[22] R.G.W.Pye,"Injection Mould Design", 4th Edition, 2000, Pearson Scientific and Technical Publications.

[23] Plastic Materials and Processing Second edition by A. Brent Strong, Bingham Young.