International Journal of Research in Engineering and Applied Sciences(IJREAS)
Available online at http://euroasiapub.org/journals.php
Vol. 7 Issue 4, April-2017, pp. 11~20
ISSN (O): 2249-3905, ISSN(P): 2349-6525 | Impact Factor: 7.196 | Thomson Reuters Researcher ID: L-5236-2015

OPTIMIZATION OF MATERIAL FLOW IN FAMILY MOULDS WITH UNIQUE RUNNER DESIGN

G Rajendra Prasd[1],
Research Scholar, JJTU Rajastan

Dr S Chakradhar Goud[2],
Professor & Principal,
SANA Engineering College, Kodada.

Abstract; Injection moulding is the most widely used polymeric fabrication process. It evolved from metal die casting, however, unlike molten metals, polymer melts have a high viscosity and cannot simply be poured into a mould. Instead a large force must be used to inject the polymer into the hollow mould cavity. More melt must also be packed into the mould during solidification to avoid shrinkage in the mould. The injection moulding process is primarily a sequential operation that results in the transformation of plastic pellets into a moulded part. Identical parts are produced through a cyclic process involving the melting of a pellet or powder resin followed by the injection of the polymer melt into the hollow mould cavity under high pressure. The present paper reveals the mold flow analysis and its filling and the design of gates as per the requirement study. The product developed in solid works and the fill time analysis with gate design has been optimised.
Key words: Mould fill analysis, critical filling, structural analysis case study, runner design.

1.0 Introduction

Plastics have advantages such as cost, lightweight structure, resilient, resistance to corrosion, color fastness, transparency, ease of processing, etc. They are being used in a wide variety of fields such as the medical industry, where they are used for detailed modeling of organs. In the architectural design industry, plastic forms are used to create scale replicas of proposed buildings. The wide range and various types of plastic materials that are available today can be designed and processed successfully while still meeting high quality, performance, and profitability requirements. Today and in the future companies must continue to produce high quality parts in order to remain competitive in the global marketplace. With the help of computer technology plastic design and processing limitations have been gradually reduced while the quality has improved significantly.

2.0 Literature review

Y.K. Shen, Y. J. Shie, and W. Y. Wu [1]: Microinjection is a branch of micro system technology. This paper employs analysis software to simulate three plastic materials (PP, PA, and POM) and four injection process parameters (injection time, mold temperature, injection temperature, and injection pressure) and applies them in simulation microinjection with the assistance of the Taguchi method, which is adopted in this paper. Further, the influences of these three plastic materials and four injection process parameters on microinjection moulding are analyzed. Through the simulation results, it is known that among the microinjection process parameters mold temperature is the most important moulding parameter. Also, during the microinjection process, mold temperature must be raised to be higher than the traditional mold temperature to prevent short shot that occurs when the melt 11 cools down too rapidly or when the melt's temperature is insufficient.

Y. K. Shen and W. Y. Wu [2]: Micro system technology enables product miniaturization, diversifies product functions, and improves quality, reliability and added value. This paper employs mold flow software to analyze three plastic materials (PP, PA, POM) and four injection moulding parameters (injection time, mold temperature, injection temperature, and injection pressure) and applies them in simulation microinjection moulding. During the process, the Taguchi method is used alternatively. All these are to obtain a better understanding of the relation between the three plastic materials as well as the four injection moulding parameters and microinjection moulding. Through the simulation results, it is known that mold temperature is the most important factor among the injection moulding parameters. Moreover, the mold temperature of microinjection moulding should be raised to be higher than the glass transformation temperature of the plastic material to avail the injection moulding of products. Y.

K. Shen, S. L. Yeh, and S. H. Chen [3]: Microinjection moulding is a branch of micro electro-mechanical system technology. This paper employs mold flow analysis software and draws a plan of simulation experiment items: three plastic materials (PS, PC, PMMA) and four moulding parameters (injection time, mold temperature, injection temperature, and injection pressure). The finite element method is used with the Taguchi method in microinjection moulding mainly to analyze the critical factors in the relation between the three plastic materials as well as the four injection moulding parameters and microinjection moulding. Two points about microinjection process are known through the simulation results: First, among the injection moulding parameters, mold temperature is the most important factor, and the next in importance is injection temperature. The third in line is injection pressure, and the least important among the four is injection time. Furthermore, the parameter values of mold temperature and injection temperature have mutual influences on each other. On the other hand, those of injection pressure and injection time affect each other's. Second, 12 mold temperature of microinjection moulding has to be higher than that of traditional injection, or otherwise short shot may occur. All of the above illustrate that temperature is the most critical factor of moulding parameters affecting precision products. Since products are small, high melting temperature is needed to increase the fluidity of the melt so that filling can be completed within a short period of time. However, it must be attended with meticulous care that temperature must not exceed the allowed temperature interval of the plastic material, or the plastic material may crack and cause potential problems after injecting the product such as bubbles formed in the plastic material, gas entrapping, and unsatisfactory mechanical properties.

3.0 Problem statement

This is base to determine the plastic process flow. Plastic is a material that can produce many shapes that can be used by human in routine life. All of plastic products are produce from various type of operation or process. All of product Produces with different type of plastic material depend to needed. Plastics are divided into two distinct groups' thermoplastics and thermo sets. Plastics can be moulded into various forms and hardened for commercial use. Plastic is perfect for this modern age. It is light, strong, easily moulded and durable. Although plastics are thought of as a modem invention, there have always been "natural polymers" such as amber, tortoise shells and animal horns. These materials behaved very much like today's manufactured plastics and were often used similar to the way manufactured plastics are currently applied. More important, the process and subsequently the moulded components are affected. When changes to a particular process variable or machine setting do occur (which significantly affect the stability of the moulding process so that defective components are produced) it is important that the correct process variable should be controlled so as to rectify the disturbance. For this purpose mould should be optimized after designing and the problems have to minimize before manufacturing the mould.

4.0 Objectives

1. To develop the cad model for case study from industrial application.
2. To develop mould assembly design as per the requirements.
3. To run the flow analysis to optimise the mould.

5.0 Methodology

1. Selection of component, 2.design of component, 3.Design of mold cavities 4.Design of core inserts, 5.Assembly of mold, 6.Generation of manufacturing tool paths, 7.Mold flow analysis

	Family abbreviation-LDPE Familyname-polyethylenes (PE)	Family abbreviation-ABS Familyname - Acrylonitrile copolymers (abs, Trade name -(10% Rubber)
Mold Temperature Range	20-70 °C	25-80°C
Melt Temperature	220°C	230°C
Ejection Temperature	80°C	88 °C
Maximum Shear Stress	0.11 MPa	0.28 MPa
Maximum Shear Rate	400001/s	12000 1/s
Modulus Of Elasticity	124 MPa	3500 MPa
Poison Ratio	0.41	0.36
Shear Modulus	43.97 MPa	1287 MPa
Melt Density	0.73537 g/cm^3	0.94933 g/cm^3
Specific Heat	4230 J/Kg-C	2400 J/Kg-C
Heating/Cooling Rate	-0.3333c/s	-0.3333c/s

6.0 Design and analysis

The part was designed using the solid works software. This is a CAD application, using feature-based parametrical construction tool modules that support all activities related to modeling of the plastic part as well as the injection mold. It maintains all parameters, which is also useful during the injection molding analysis, in order to identify the parting plane and define cavity layouts. The mold design is carried out by using both feature-based and parametric approaches. When the type and structure of the mold are initially defined, the values associated to the main parameters are given in order to specify the mold geometry. As the CAD and FEM process are interactive, designers are required to prioritize important elements within different parts. The data exchange between CAD and FEM is realized in several steps.

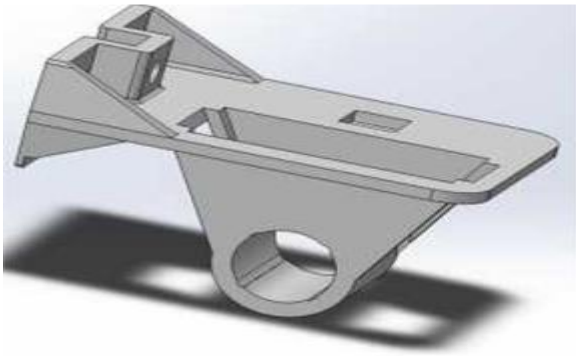

Figure No.1 shows the design in solid works.

Part chosen for simulation is plastic holder for cables and coolant tube found in Honda Civic EJ9. Part in real due to vibrations and stress always has problems with cranks and deformations after time. At first, part was designed in SolidWorks 2012 with real dimensions. Next using Solid Works Simulations deformation of holder was analysed by using real values. Maximum experimentally measured force at end of the holder was 30N.

Next using Solid Works Simulations deformation of holder was analysed by using real values. Maximum experimentally measured force at end of the holder was 30N. Material was chosen to DuPont Engineering Polymers, Zytel 73G30, PA6 30% Glass Fiber Filled. By simulation maximum von Misses stress was calculated to 45 189 676 Nm2 while Yield strength of selected material is 103 648 886 Nm2, showed on Figure 4. Displacement of deformation is showed on Figure 5, when maximum URES was calculated to 3,357mm.

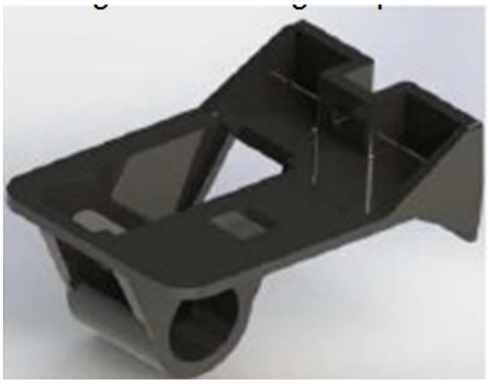

Figure No. 2: Renderation with high gloss surface

Next using Solid Works Simulations deformation of holder was analysed by using real values. Maximum experimentally measured force at end of the holder was 30N. Material was chosen to DuPont Engineering Polymers, Zytel 73G30, PA6 30% Glass Fiber Filled. By simulation maximum von Misses stress was calculated to 45 189 676 Nm2 while Yield strength of selected material is 103 648 886 Nm2, showed on Figure 3. Displacement of deformation is showed on Figure 5, when maximum URES was calculated to 3,357mm.

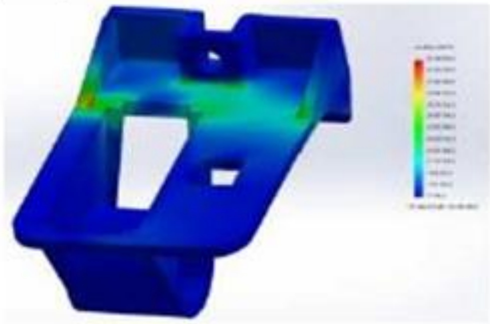

Figure No.3: Displacement of von Misses stress

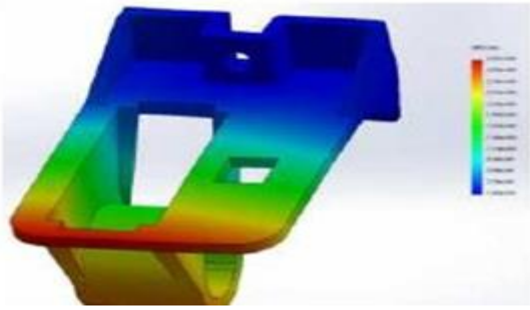

Figure No.4: Displacement of stress

Next factor of safety (FOS) was set to value of 3. Simulation showed small area of possible deformations and cranks as shown on Fig.6.5. Next step was the analysis of the shape of part due to technological conditions of manufacturing. One rib at bottom side of holder was added, chamfer and fillets were changed due to stress conditions. All changes are in state of negative cut, so they can be done to finished form. Addition of rib and final shape is shown on Figure 6.6.

Figures No. 5, 6 and 7: Shows the problem and the rectification respectively

Next simulation of stress was done again and deformation was improved. Von Misses stress was calculated to 32 960 602 Nm2 in comparison to 45 189 676 Nm^{2}it is improvement by 37%. Maximum deflection was calculated to 2,227mm, what is almost 50% improvement. By analysis of deformations with Factor of safety set to 3 there were no calculated areas that can be possible areas of cranks. Stress, deflection and FOS analysis is shown on Fig 8 and 9. Next performed simulations were injection simulations using Autodesk Mold flow.

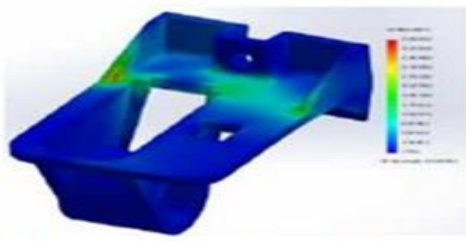

Figure No. 8: Displacement von-misses stresses

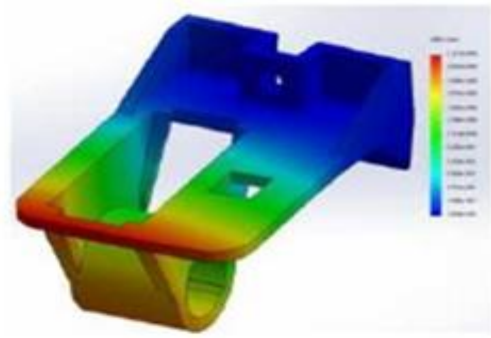

Figure No. 9: Displacement of stresses

The first step in analysing is importing the design into the moldflow software using IGES Format. After importing the design the next step is to mesh the product design using moldflow software. In next part of paper, the main analyses of AMA Material for simulations was set to DuPont Engineering Polymers, Zytel 73G30, PA6 30% Glass Fibre Filled, same as material at stress simulations with properties obtained from material list. Nominal wall thickness The Nominal wall thickness result displays thickness variations relative to the wall thickness of the part. Nominal wall thickness was analyzed to 2,292mm, end of nominal range 20% with end of low deviation range set to 50 %.

Molding analysis

The Molding Window display is a plot over the ranges of the melt temperature and the injection time for a given mold temperature. The three colors (red not feasible, yellow and green preferred) represent how good a particular combination of processing conditions is.Melt temperature for analysed material was set to 275°C and analysed injection time was calculated to 0,6598s and mould temperature set to 100°C.

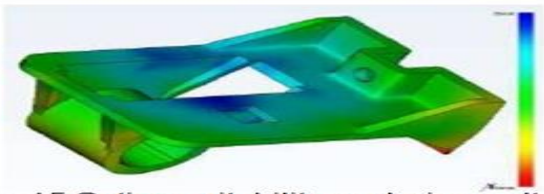

Fig. 15 Gating suitability analysis results

Figure No.10: Displacement of stresses

7. Fill time analysis

The Fill time result shows the position of the flow front at regular intervals as the cavity fills. In the following figure, the contour colors represent the flow of plastic into the part. All regions with the same color filled at the same time. In a part with a good fill time result, the flow pattern is balanced, meaning:

All flow paths finish at the same time. All flow fronts should reach the edges of the model simultaneously. This means that in the illustration, each flow path should end with red contours. The contours are evenly spaced. The contour spacing indicates the speed at which the polymer is flowing. Widely-spaced contours indicate rapid flow, while narrow contours indicate that the part is filling slowly. After the analysis, the fill time of part was calculated to 0,3976s and the variance of fill is displayed in Fig.8.

Confidence of fill

The Confidence of fill result displays the probability of plastic filling a region within the cavity under conventional injection moulding conditions. This result is derived from the pressure and temperature results. Simulation had shown that part has highest possible confidence of fill (100%) with entered material properties and moulding parameters.

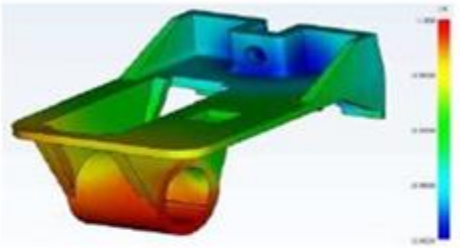

Figure No. 11: Confidence of fill analysis

Figure No.12: Confidence of fill

Pressure at end of fill

The Pressure at end of fill result shows the maximum injection pressure value obtained during the whole duration of the filling phase. This result is output by a Fill analysis and is displayed on the surface only. Pressure at end of fill was calculated to 123,9 MPa. **Temperature at flow front**

Temperature at flow front shows the temperature of the polymer when the flow front reached a specified point, in the center of the plastic cross-section. This result is output by a Fill analysis. Maximum temperature at flow front was calculated to 277oC and progress is shown in Fig.24. Temperature variance 7 and maximum variance measured between centre of part and edge was only 2,798oC. After these simulations, time to reach ejection temperature was calculated to 23,56s, only few air traps were discovered and minimum weld lines showed up in the part. Confidence of fill was rated with maximum possible value of 99,96%. Runner adviser analysis showed, that engaged system met all requirements and fulfil the criterions. Cooling system adviser calculated maximum flow rate to 10lit/min with distilled water as cooling medium.

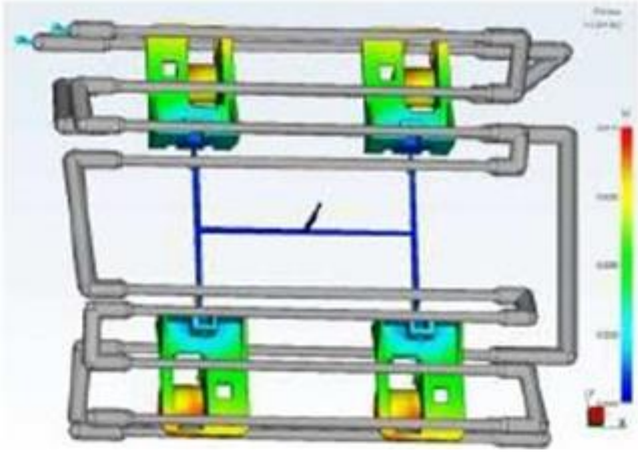

Figure No.13: Results of fill time analysis

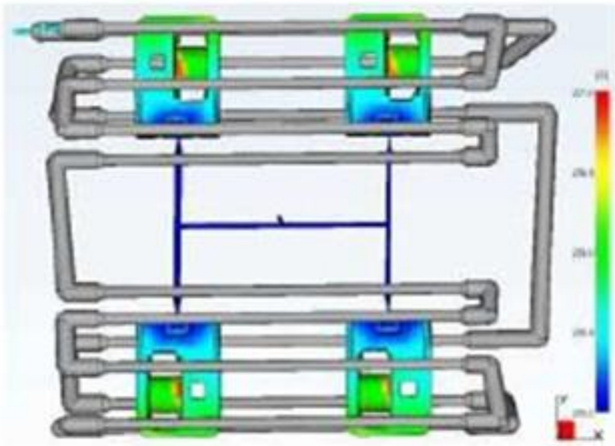

Figure No.14: Temperature flow front results

Circuit pressure result

The Circuit pressure result is generated from a Cool analysis and shows the distribution of pressure along a cooling circuit, averaged over the cycle. The pressure inside the cooling circuits should remain evenly distributed from the inlet circuit pressure to the outlet circuit pressure. Calculated value at simulation was 281,7KPa and progress is shown in Fig.25. Variance between inlet and outlet was calculated to 2,74oC. Minimum Raynolds number of turbulent flow was calculated to 11809 and maximum to 39363.

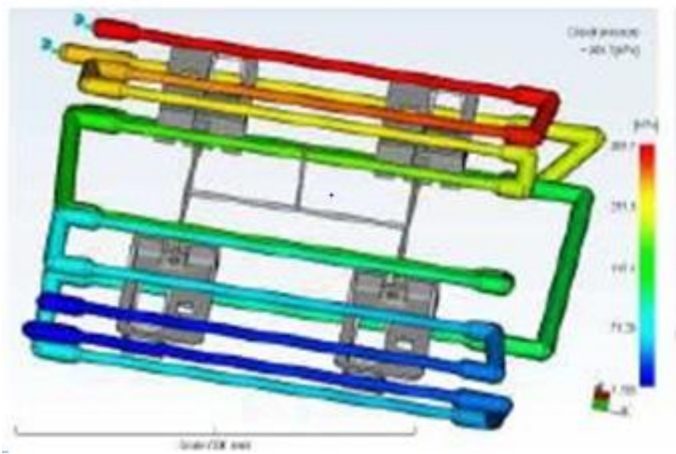

Figure No.15: Circuit Pressure Result

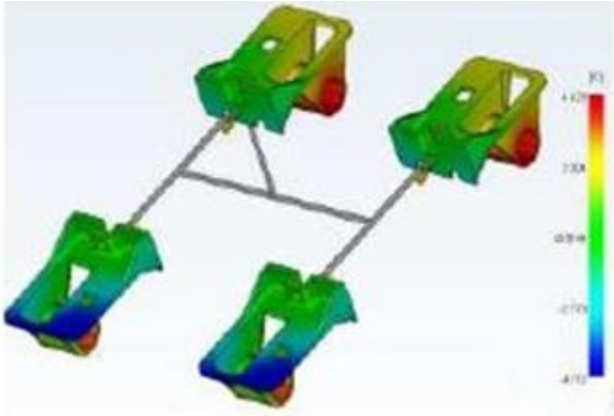

Figure No.16: Temperature Variance

Temperature, part result

The Temperature, part result shows the average temperature at the part boundary (part side of the part/mold interface) over the duration of the cycle. This result is used to find localized hot or cold spots, and determine whether they will affect cycle time and part warpage. As show in Fig. 26, temperature variance at part was minimal due to the design of part and construction of cooling system. Time to reach ejection temperature of part was calculated to 13,01s. At warpage simulations maximum nominal deflection was calculated to 1,125mm, what is acceptable value. This value can be reduced by leaving the part for longer time in mold cavity for better cooling or choosing another type of material.

8. CONCLUSIONS

Mold filling simulation is a very helpful tool for the injection mold designer and plastic part designer. The process uses software to virtually simulate the filling, packing and cooling of a molded plastic part. It allows the mold/part designer to make critical decisions about the design before the injection mold is manufactured, when design changes are significantly less expensive. When considering part and mold design, gate location and part filling are initial concerns. The simulation will provide a visual representation of how the mold will fill. This information is valuable in showing potential problems, such as areas of the part that will not completely fill with plastic. With this type of initial information, the designer can then simulate different gate locations that can improve these and other situations. The software can calculate the pressure inside the filling plastic, which could point towards potential problems. Other evaluations such as material sheering, temperature and pressure can also be determined. This and other data is not only helpful for designing the part, but can also be used to decide if a specific material will work with a given part design.

9. References

[1]. Peter Kennedy, Rong Zheng, Flow Analysis of Injection Molds, 2nd Ed., Hanser Publications, Cincinnati, U.S.A, 2013.
[2]. Ashutosh Modi, Prof. K.A. Rade, S.S. Kulkarni, Design enhancement for an injection mold using inputs from flow analysis, IJAERS, Vol. II, Issue III, April-June, 2013.
[3]. Injection Mould Design, R.G.W. Pye.
[4]. Injection Mould Design, Rapra.
[5]. Technical Directory on Design and Tooling for Plastics, CIPET.

[6]. “Material for injection moulding’’ branisler Dublea Frantisek-2012.
[7]. Auto desk Mold-Flow Insight material data warehouse.
[8]. wikimedia Foundation, Inc, 2010. InjectionMoulding. [online] Available at: [Accessed 28 August 2010]
[9]. Bryce, D. M., ‘Plastic injection molding: Manufacturing process fundamental’. Society of Manufacturing Engineers, (1996)
[10]. Sri. P V S M Varma and Sri. P N E Naveen, “Optimizing Injection Moulding Tool Cost by Using Virtual Software Techniques” International Journal of Mechanical Engineering & Technology (IJMET), Volume 4, Issue 6, 2013, pp. 227 - 240, ISSN Print: 0976 – 6340, ISSN Online: 0976 – 6359