ISSN 2250-3153

Analysis of Plastic Flow in Two Plate Multi Cavity Injection Mould for Plastic Component for Pump Seal

Jagannatha Rao M B, Dr. Ramni

Student of M Tech in Mechanical Machine Design, Department of Mechanical Engineering, Shridevi Institute of Engineering & technology, Tumkur -570106, Ph.: 9738580575

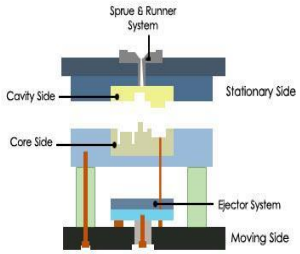
Abstract- This is focused on the analysis of plastic flow in two plate injection mould. Motor rare housing component is designed with multi cavity as the plastic part in the two plate injection mould which is used in domestic motor. Mould flow analysis software is used to perform the analysis of filling, wrap and best gate location. This will discuss the flow of molten plastic inside the injection mould. The analysis begins with the origin of the flow channels such as Barrel, nozzle, sprue, runners, and gates until the cavity is completely filled. In this analysis, plastic flow behavior is studied together with parameters settings in plastic injection mould, Based on the analysis improvement has been made to the part design and feed systems in the mould. This includes the location of the gate at the part design, size of sprue, runners and gates. From the analysis also, CAE helps to determine the part defects that might occurred during plastic injection molding process such as Short shot, unequal filling, over filling, welding lines and others. From that, the optimum parameters setting are selected in order to get a quality plastic. So this will explain the plastic flow analysis clearly.

I. INTRODUCTION

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.

The plastic product can make from the several processes like injection moulding, blow moulding, compression molding, film insert mounding, gas assist moulding, rotational moulding, structural foam moulding, extrusion and Thermoforming. This thesis will explain and study more about injection moulding. Injection moulding is a process in which the plastic material is injected into a mould forming a plastic product. Injection moulding is a manufacturing technique for making parts from thermoplastic material. The solid plastic material is fed into an injection moulding machine, heated and then pressed into the mould. In injection moulding, plastic pellets or granules are fed from a hopper into a heating Chamber. A plunger or screw pushes the plastic through the heating chamber, where the material is softened into a fluid state. At the end of this chamber, the resin is forced into a cooled, closed mould. Once the plastic cools to a solid state, the mould opens and the finished part is ejected.



Standard Two Plate Mold

Injection moulding is very widely used for manufacturing a variety of parts, from the smallest component to entire body panel. It is the most common method of production, with some commonly made items including bottle caps and outdoor furniture. Plastic moulding products can be seen everywhere such as plastic tubes, grips, toys, bottles, cases, accessories, kitchen utensils and a lot more. The mould is made by a mould maker from tool steel, usually either steel or aluminum, and precision machined to form the features of the desired part. Mould is used to produce desire product that we needed. Many elements are involved in mould such as feeding, cooling and injector system. In modern technology, CAD software can be used to design mould and after that perfiune machining raw material to produce complete mould. Feeding system is important element for plastic flow in injection mould. All of this will be explain detail inside this project. In this thesis, the analysis is concentrate for plastic flow in two plate injection mould.

II. PROBLEM STATEMENT

A parameter setting and feeding system such as gate, runner and sprue inside the plastic injection mould are located by mould makers using trial and error method. At this situation, people that have a lot of experience in injection moulding process who capable to decide the size and location of feeding system especially in two plate mould are needed. The problems occur when this person cannot perform the job with good method and needed to take much time to think and make an experiment.

Waste time and higher cost maybe happened during this period. This situation happened at past time before process analysis can perform with software.

Simulation software is the new technology that can examine the behaviors of plastic flow inside the cavity mould. It can decide the better method to select the best design for feeding system like runner, sprue, and gate and process parameter. The size of gate, runner and sprue is importance thing to consider for producing good quality plastic product. It can give effect to product if the unsuitable type or sizes are used. There are many type of simulation software now that can make work more easy and accurate.

The Plastic adviser Software is one of software used to ensure the best choice location of feeding system and size of gate, runner and sprue. It's one of the most advanced tools ever devised for the plastic injection moulding engineer. More precisely, it's a computer generated 3D simulation that models the flow of resin material into a single or multi-cavity mould. With the aid of mould flow analysis, engineers can obtain statistical data of the moulding process before the mould is actually constructed. The object is to optimize the fill process of a mould and the integrity of the moulded part. The data provided during the analysis helps the engineer select the optimum location for gate, sprue and runner. Temperature variations and all the suitable parameter are clearly defined in the simulation. The end result is accurate, economical and reliable plastic parts.

III. OBJECTIVE

The main objectives of this research are:
i. To design plastic part.
ii. To design feeding system like sprue, runner and gate in two plate injection mould.
iii. To set optimum process parameter like injection pressure, speed, temperature and other.
iv. Analysis plastic flow in two plate injection Mould

IV. COMPONENT DETAILS

The component is made of nylon66 (PA66) polyamide materials have high mechanical strength and superior resistance to wear and organic chemicals. NYLON 66 GF30 has more than doubles the strength and stiffness of unreinforced nylons and a heat deflection temperature which approaches its melting point. NYLON 66 is 30% glass-fiber-reinforced nylon 66 materials whose important properties include high tensile and flexural strength, stiffness, excellent heat deflection temperature, and superior abrasion and wear resistance.

The below figure gives a 3d model of Pump Seal which is designed using CATIA software

Component name: Pump Seal

Component material: Polyamide (Nylon 66) PA66
Shrinkage: 1.5 %
Component weight: 10.5 grams
Moulding type: Multi Cavity injection mould tool

V. MOLDFLOW ANALYSIS SIMULATION

Numerical simulation of polymer processing is getting more and more popular in the Industry because it helps to forecast the problems that may occur due to the wrong tool design or use not optimized processing conditions. As result, a lot of money and time can be saved. The simulation is done before the final tool design phase, when the changes in tool design are still possible.

The simulation is usually done with special commercial programs, not only for injection moulding, but also for other processing technologies, like extrusion, blow extrusion, calendaring, thermoforming, etc. The target of numerous research conducted with the use of numerical simulation of the injection moulding process is usually to obtain the pressure, temperature and shear rate distribution in a closed mould and investigation of melt flow front movement in cavity as well as determining the flow length in different injection moulding conditions. A mathematical model which allows evaluating the orientation of short fibers used as filler in injection moulded parts was also worked out. It is possible to calculate the pressure required for total cavity filling, clamping force after cavity filling, evaluating the optimal processing conditions.

The optimal processing conditions can be found during the filling, holding and cooling stage simulation. The criteria of optimization are the proper values of temperature, pressure, and shear rate and even their distribution in the moulded part, what can guarantee the good properties of finally formed parts. The relationship between processing conditions and parts properties is the topic of many research works.

VI. MOLD FLOW SOFTWARE

Mold flow software has been developed by Mold flow International Pvt. Ltd., Australia. It helps in finite elemental analysis used in the design of plastic product, mould design and production of plastic components. Following are the modules of MOLDFLOW software:

Flow Analysis: The Flow analysis is used to determine the gate position and filling pattern. It analyses polymer flow within the mould, optimizes mould cavity layout, balances runners and obtains mould processing conditions for filling & packing phases of the Moulding cycle.

Cooling Analysis: It analyses the effect of cooling on flow, optimizes cooling line geometry & processing conditions.

Process Optimization Analysis: It gives optimized-processing parameters for a component considering injection-moulding conditions.

Warpage Analysis: This analysis simulates the effect of Moulding on product geometry, isolates the dominant cause of warpage so that the correct remedy can be applied.

Shrinkage Analysis: This analysis gives dimensions of mould cavities, using shrinkage determined from specific grade material shrinkage data & flow analysis results.

The analysis is carried out using the software Mold flow Plastic Adviser.

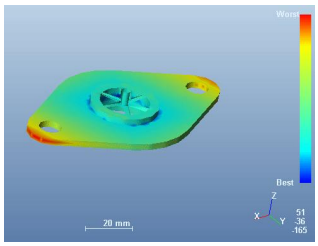
The Benefits of Predictive Analysis are as below:

To avoid the high costs and time delays associated with problems discovered at the start of manufacturing, it is necessary to consider the combined effects of part geometry, material selection, mold design and processing conditions on the manufacturability of a part. Using predictive analysis tools to simulate the injection molding process, organizations and industries can evaluate and optimize interactions among these variables during the design phases of a project before production begins, where the cost of change is minimal and the impact of the change is greatest.

The productive analysis can simulate the filling, packing and cooling phases of thermoplastics molding processes using materials with or without fillers and fiber reinforcements, as well as predict post-molding phenomena such as part warpage, also simulate material flow and cure of reactive molding processes.

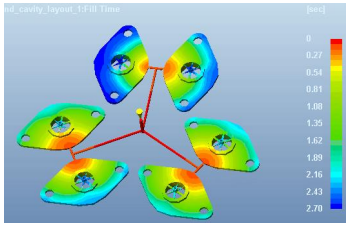
MPI also offers the world's largest material database of its kind with more than 7,800 thermoplastic materials characterized for use in plastics CAE analysis, as well as thermoset materials, coolants and mold materials, and injection molding machine-specific analysis capabilities.

VII. MOLD FLOW RESULT



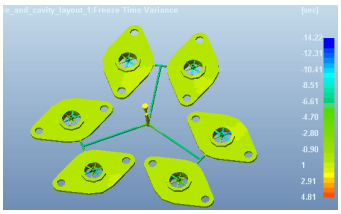
BEST GATE LOCATION

The Gate Location result rates each place on the model for its suitability for an injection location. The most suitable areas, colored blue, are rated as best, and the least suitable areas of the model, colored red, are rated as worst.



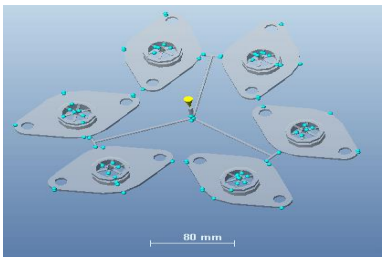
ANALYSIS FOR FILL TIME

This 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 from red, to indicate the first region to fill, through to blue to indicate the last region to fill.



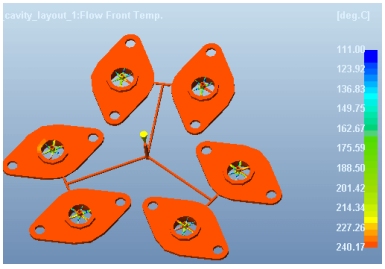
ANALYSIS FOR FREEZE TIME

The Freeze Time Variance result plots the deviation of the time it takes the polymer to freeze in any region of the part from the average time to freeze for the entire part. Areas that are plotted as positive values (red) take longer to freeze than the average time to freeze, and areas that are plotted as negative values (blue) freeze more quickly.



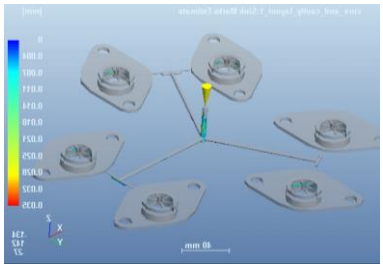
ANALYSIS FOR AIR TRAPS

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.



TEMPERATURE AT FLOW FRONT

The flow front temperature result uses a range of colors to indicate the region of lowest temperature (colored blue) through to the region of highest temperature (colored red). The colors 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.



VOLUMETRIC SHRINKAGE AT EJECTION

As the hot plastic material cools in the mould, it contracts towards the center, and will stick to core because of shrinkage factor. Thus, while designing a tool, the shrinkage allowance must be added to the core and cavity. Different materials will have different shrinkage.

VIII. CONCLUSION

Mold flow analysis was carried out on the component and feed system of injection molding tool. This gave satisfactory results and the same was confirmed from analysis such as injection pressure, fill time, flow front temperature, quality of fill, weld line, air traps etc. The results indicated that the injection molded components could be manufactured with minimum molding defects. The tool was manufactured using the CNC and NC machining process according to DME standard and manufactured elements were assembled.

The trial out of the injection mold tool revealed the components produced without defective. Further work can be carried out by performing the stress analysis to core and cavity inserts using ANSYS software for more effective design. Fatigue analysis can also carried out for the tool which results in improving the life of the tool. The mold flow analysis can further be used to carry out for design of experiments for fill analysis, Wrap analysis, the best gate location and the result can be utilized for further optimization of the tool design.

- From the Mold flow analysis report large numbers of air traps were observed. This is eliminated during manufacturing of the tool by providing the air vents at proper locations.
- Mold flow analysis helped in deciding the process parameters like mould and melts temperature, fill time, injection pressure. This reduced considerable amount of time and cost during trail run.

CAD/CAE technology facilitates the use of numerically controlled machining technology in fabrication of mould. In, turn this reduces number and complexity of manual setup operations

REFERENCES

[1] Effects of injection molding conditions on the mechanical properties of polyamide / glass fiber composites by prof. Dr. lk YILMAZER Supervisor.

[2] Adekunle A. Fagade and David O. Kazmer, Early cost estimation for injection molded parts, a Thesis by University of Massachusetts

[3] J.A. Brydson, Plastic materials

[4] British Cataloging sixth edition, 1995, Butterworth Heinemann Ltd.

[5] Haddout,A & Villoutreix, G.,An Experimental study of fiber orientation in injection moulded short glass fiber reinforced polypropylene/polyarylamidecomposites (Conservatoire National des Artset M6tiers, France), pp.147-153, 1992.

AUTHORS

First Author –Jagannatha Rao M B, IV Sem, M Tech (Machine Design), Department of Mechanical Engineering, Shridevi Institute Of Engineering & Technology,Tumkur-572106, Karnataka, India., Email: get_jagannath@rediffmail.com Mob:+91-9738580575.

Second Author – Dr. Ramni Professor and Head , NTTF, Peenya, Bengaluru-560058,Karnataka, India