Manufacturing Optimization of Plastic Injection Mould with Unique Gate Design of Family Mold

G.Rajendra Prasad[#1], Dr.S.ChakradharGoud[*2],

*Research Scholar,S.J.J.T.University[*1], Rajasthan, India & Principal, Sana Engineering College, Kodad, T.S, India[*2]*

Abstract:*Injection moulding has been a challenging process for many manufacturers and researchers to produce products meeting requirements at the lowest cost. Faced with global competition in injection moulding industry, using the trial and error approach to determine the process parameters for injection moulding is no longer good enough. Factors that affect the quality of a moulded part can be classified into four categories part design, mould design, machine performance and processing conditions. The part and mould design are assumed as established and fixed. During production, quality characteristics may deviate due to drifting or shifting of processing conditions caused by machine wear, environmental change or operator fatigue. Determining optimal process parameter settings critically influences productivity, quality, and cost of production in the plastic injection moulding (PIM) industry.*

Key words: *Multi cavity mould, gate machining, uniform flow, Practical observation.*

1.0 INTRODUCTION

Moving from a single cavity mould to one that produces two, four, or eight parts at once seems like an easy way to increase production volume and reduce part costs. This can be true in many cases, but only if the right steps are taken and the requisite homework done first. Designing a part for multi-cavity moulding is not as simple as copying the CAD file for a single-cavity mould multiple times. The physics encountered when forcing molten plastic through a mould’s sprues, runners, and gates change as moulds become larger and more complex, something that can impact moulding performance and part quality. Also, thermal variations within a multi-cavity mould body become more of a concern, and plastic must travel longer distances to reach the finish line, both of which increase the risk of partially-filled cavities and sink as well as part deformation after ejection. When moving from single- to multi-cavity tooling, it’s important to recognize that parts that behave perfectly in a single-cavity mould might not play well with others, at least not without first making some tweaks to the part, the process, or even the material.

1.1 GATE

The gates used in plastic injection moulding are no different. They allow molten plastic to flow into the mould at the beginning of the injection cycle, and then hold it under pressure until the mold cools, the plastic has solidified, and the part is subsequently ejected. In mouldmaking, there are more types of gates than there are players on a baseball team. Pin-style and hot tip gates are often employed on single-cavity moulds to solve challenges with complex part geometries and to reduce gate vestige. The small remnant of runner material that must be trimmed from the finished work piece but these are rarely if ever used on multi-cavity moulds. Here, tab gates (also known as edge gates) are the rule. Not only is gate placement far more flexible something very important when trying to squeeze multiple parts into a mould but the larger vestige that comes with tab gating works well to absorb residual flow stress around that section of the mould. Another example where costly part redesign can be avoided when making the jump to multi-cavity is gate placement. Consider a mould for a plastic water bottle lid. Initial limited production expectations might dictate placement of the gate in a single-cavity mould gate at a certain mould location. But when production ramps up and the head of supply chain decides it’s time to invest in a multi-cavity tool, the original gate location may be impossible to achieve due to the changes in part orientation required for multi-cavity moulding. The injection molding process stages starts with the feeding of a polymer through hopper to barrel which is then heated with the sufficient temperature to make it flow, then the molten plastic which was melted will be injected under high pressure into the mold the process is commonly known as Injection, After injection pressure will be applied to both platens of the injection moldingmachine(moving and fixed platens) in order to hold.

1.2 OBJECTIVES

1. To verify the injection moulding process for multi-cavity systems.

2. To check the gate feasibility practically to consider filling problems in mould trail.

3. To get practical enhancement of mould cavities for further optimise the mould with latest techniques.

2.0 LITERATURE REVIEW

Bikas, A et.al.(2002) present a NESPLAN code towards the intelligent design of the injection moulding process. Furthermore NASPLAN code has been coupled with three different numerical optimization methods in order to design the guidance of melt by tuning of gates. They suggested filling condition of mould on the basis of vacuum pressure.

Galantucci, L. M., &Spina, R. (2003) study mold filling condition with the help of Integrating FE with DOE. They attempt to study on gate location melt temperature, mould temperature, injection pressure, packing time, packing pressure process parameter.

Erzurumlu and Ozcelik (2006) used Taguchi technique to minimize warpage and the sink index. In their study they considered mold temperature, melt temperature, packing pressure, rib cross section and rib layout angle and material PC/ABS, POM,PA66. They find in their research that packing pressure is influence the factor for PC/ABS plastic products, rib cross section influence POM material plastic product and rib layout angle influence PA66 material plastic product significantly.

Shen et.al. (2007) did investigation on effect of molding variables on sink mark index using Taguchi method. They considered melt temperature, injection time, packing time, mold temperature, distance between gate and rib layout.

Huang, M. S et. al. (2007) study the cavity pressure profile in the mould of injection moulding machine. In their experimental research they find innovative switchover method yields a more uniform product weight that any traditional methods after study the filling condition in the mold.

Rezavand, S. A. M et.al. (2007) presented a simplified wax model of gas turbine blade. Their study on the mold filling condition in the investment casting process, they explain that major steps in investment casting processes are injection molding of Wax pattern, ceramic coating, removing wax, drying and material casting. In the mold manufacturing they consider injection temperature and holding time as processing variables. They found that holding time to be more dominant than that of injection temperature.

Sahputra, I. H.et.al. (2007).study on the mold flow condition of injection moulding process with the help of SIMPOL and MPI software.They compare the software result.

Mathivanan D and Parthasarathy (2009a,2009b) reported modeling of sink marks using DOE based regression.

Saman, A et.al. (2009 November) study the mold condition of injection molding by the CAD/CAE tools to design optimal gating system. They suggest the optimal gating system with the help of CATIA and MOLDFLOW software.

Stanek, M.etal. (2010) study mold design with the help of Cadmould software. They explain that Cadmould software can calculates filling time, speed and vulcanization time in the mold and consequent after curing depending on the material and technological parameters.

Zhao Longzhi et.al.(2010) study the sink marks defect with simulation with the help of software mold flow and did experiment with the help of taguchimethod.In their research they study on Polypropylene material and process parameter melt temperature, mold temperature, injection time, holding pressure, cooling time.

Babur ozcelik et al (2010) attempted to study on mechanical properties of material with using taguchi method. They consider melt temperature, packing time, cooling time, injection pressure.

Gruber, D. P. et.al. (2011) study on the measurement of the visual perceptibility of sink marks on injection molding parts They study the sink marks defected plastic parts by incrementing the holding pressure and other parameters kept constant. Wang, X.et.al.(2013) study on the warpage and sink marks defect with the help of rapid heat cycle molding technology.

Vashisht, R. et.al.(2014) study the different gating system with the help of MOLD FLOW software. They check the fill cycle time with the help of MOLDFLOW for minimization of sink marks.

Ramesh, G(2015) study on sink marks minimization of a head light for alto car plastic component. They did modelling of product with the help of Pro E and Mold flow analysis with the help of MPI software. In their observation they explain single gate location can minimize sink marks in comparison to multiple gate location. It could save wastage of material.

3.0 CONSIDERATIONS OF MANUFACTURING

1. Mould base material : P-20
2. Core-cavity inserts : STAVAX
3. Sliders and Stripper rings: STAVAX
4. All inserts Machining on CNC process only.
5. Sparking also on CNC EDM.
6. Wear plates providing for sliders.
7. Taper interlocks providing for Parting line.
8. Material used for moulding-ABS

4.0 METHODOLOGY

Fill stage: During this stage, the mould cavities are filled with molten resin. As the material is forced forward, it passes over a spreader, or torpedo, within the barrel, which causes mixing. This stage is determined by an injection velocity (rate), a pressure, and a time. Injection velocity is the rate at which the plunger moves forward.Pack stage: As the melt enters the mould, it cools and introduces shrinkage. The pack stage is necessary to force more melt into the mould to compensate for shrinkage.

Gate manufacturing in CNC milling

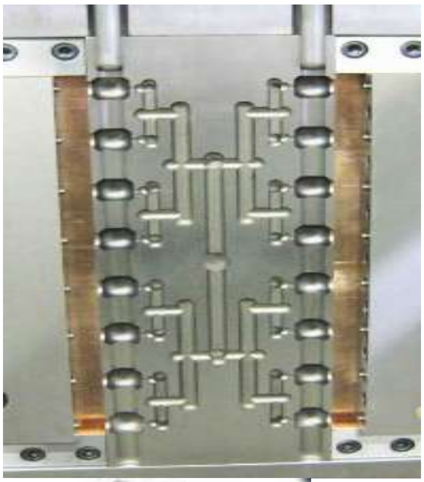

Figure 3.1 shows the multi cavity runner gate 3.2 Shows the mould assembly

Hold stage:

When no more material can be forced into the mould, melt can still leak back through the gate. The hold stage applies forces against the material in the cavity until the gate freezes to prevent leaking of the melt. In some machines, pack and hold are combined into a single second or holding stage. The aim of this paper is to identify and select the optimal conditions for the technological process of injection moulding of selected part to determine the optimal conditions. Injection process is affected by number of factors that influences on productivity and accuracy of injection moulding as well as on the overall cost associated with its production.

4.1 Injection temperatures and process timings for the mould trail from BHANODAYAM industry, cherlapalli., Hyderabad

Selected temperature profiles				
Label of profile	Temperature of jet(^{0}c)	Temperature at screw(^{0}c)		
		Phase 1	Phase 2	Phase 3
1	340	330	320	310
2	325	315	305	295
3	320	310	300	290
4	315	305	295	290
5	300	290	280	270

Temperature profile	Weight of mould(g)	Dosing time (sec.)	Material pillow(g)	Screw path (mm)	Max. pressure of melt (bar)	Mold filling time (sec.)
1	7.654	2.453	2.820	40	585.33	1.500
2	7.642	2.463	2.842	40	707.33	1.603
3	7.639	2.486	2.853	40	847.33	1.636
4	7.628	2.503	2.875	40	1004.30	1.816
5	7.619	2.530	2.890	40	1214.30	2.003

5.0 DISCUSSIONS

Discussion of measured values for different temperature profiles from Table 1 and Table 2 shows that when the temperatures at the nozzle and the screw are higher, the time of the dosage lowers. Dosing Time is at all temperature profiles less than 3 seconds, and cooling of moulds takes up to 10 seconds. At the temperature profile No.1 is temperature of nozzle and screw very high. At these temperatures, the risk of degradation possibility of the material often occurs. This profile would be able to shorten the production cycle, but does not guarantee the quality pressings. For the production of these moulds, even temperature profile No.5 was inconvenient with a temperature of 300°C at nozzle. The reason was the relatively high pressure of molding and low quality. More time is needed for the injection and probably would require a longer was excluded and temperature profile No.4 with nozzle temperature 315 °C. In the remaining two profiles it was necessary to consider the advantages and convenience of use. Although the difference between the weight and filling time were minimal, there was a significant difference in the maximum filling pressure. For this reason, as the optimum temperature profile was chosen profile No. 2, with a temperature of nozzle 325 °C. After evaluating of moulds at different speeds of injection was injection rate of 30 ccm/s chosen as the most suitable. The reason for this choice was high quality of moulds, without failures and burns. Determination of the holding pressure was primary based on the weight of the moldings. By the increasing time of holding pressure, the cooling time was gradually reduced. Reducing the cooling time with raising the holding pressure phase did not affect the quality of the moldings. High-quality surface of samples was observed at 4 sec holding pressure time.

6.0 CONCLUSION

The task of the optimization was to create moldings with required quality in the shorter injection cycle, without compromising their quality. As the optimal temperature profile was chosen profile No.2. Injection cycle after optimization decreased from 22.9 seconds to 20.2 seconds, i.e. 2.7 seconds. Optimization of the injection cycle and error analysis yielded shortens production cycle and reduced the proportion of waste during production because of visual defects. Thus optimization of this part and process saves the energy and material costs.

6.1 FUTURE SCOPE

The simulation included a test of process parameters obtained during the optimization of injection molding process. Fill analysis of the cavity and injection pressure analysis revealed the cause of burns in two places between inlet moldings. Analysis of quality prediction revealed the possible problems during production and confirms problems from praxis.

References

(1) Mathivanan, D., M. Nouby, and R. Vidhya. "Minimization of sink mark defects in injection molding process–Taguchi approach." International Journal of Engineering, Science and Technology 2, no. 2 (2010): 13-22.

(2) G.Singh,M.K.Pradhan,A.Verma." A review of the effect of process parameters on the performance of plastic injection molding process to control the warpage in Plastics

(3) A Thesis on optimization of injection molding cooling by Laris-erikRanner

(4) Erzurumlu, Tuncay, and Babur Ozcelik. "Minimization of warpage and sink index in injection-molded thermoplastic parts using Taguchi optimization method." Materials & design 27, no. 10 (2006): 853-861.

(5) Galantucci, L. M., and R. Spina. "Evaluation of filling conditions of injection moulding by integrating numerical simulations and experimental tests." Journal of materials processing technology 141, no. 2 (2003): 266-275.

(6) Vashisht, Rahul, and ArjunKapila. "Analyzing Effects of Different Gates on Component and Molding Parameters." (2014): 2352-2359.

(7) Dimla, D. E., M. Camilotto, and F. Miani. "Design and optimisation of conformal cooling channels in injection moulding tools." Journal of Materials Processing Technology 164 (2005): 1294-1300.

(8) Ozcelik, Babur, AlperOzbay, and ErhanDemirbas. "Influence of injection parameters and mold materials on mechanical properties of ABS in plastic injection molding." International Communications in Heat and Mass Transfer 37, no. 9 (2010): 1359-1365.

(9) Saman, Alias Mohd, Abdul Halim Abdullah, and MohdAsriMohd Nor. "Computer simulation opportunity in plastic injection mold development for automotive part." In Computer Technology and Development, 2009.ICCTD'09. International Conference on, vol. 1, pp. 495-498. IEEE, 2009.

(10) Longzhi, Zhao, Chen Binghui, Li Jianyun, and Zhang Shangbing. "Optimization of plastics injection molding processing parameters based on the minimization of sink marks." In Mechanic Automation and Control Engineering (MACE), 2010 International Conference on, pp. 593-595. IEEE, 2010