# Review for various design parameters considered during mould design


**[1]H. H. Makwana, [2]N. D.Gosvami**

[1]M.E. (Cad/Cam), [2]Professor(Mechanical)

[1, 2]Atmiya Institute of Technology, Rajkot, India

[1]hiren_sos@yahoo.co.in[2]ndgosvami@aits.edu.in



**Abstract**—*With increasingly short life span on consumer electronic products such as mobile phones becoming more fashionable, injection moulding remains the most popular method for producing the associated plastic parts. The process requires a molten polymer being injected into a cavity inside a mould, which is cooled and the part ejected. The main phases in an injection moulding process therefore involve filling, cooling and ejection. The cost-efficiency of the process is dependent on the time spent in the moulding cycle. Another thing was found that the properly determined gate location leads to better resin flow and shorter hesitation time. For that design guideline was induced by investigating resin flow patterns depending on several gate positions obtained by numerical analyses of a simple strip with a hinge*



**Keywords:** Injection molding; Gate location design; cooling channel.


## I. INTRODUCTION

Injection molding is one of the most common plastics processing methods. It consists of heating and melting the plastic pellets (resin); injecting the melt under high pressure by a screw type auger into a closed mold; cooling the melt inside the mold to form the part; and ejecting the finished part. Due to the elevated pressures and temperatures, great effort is needed in the design of the mold. The mold must accommodate not only the part cavity but also a runner system to distribute the melt throughout the entire part (or parts) as well as water cooling channels fabricated within the body of the mold. Commonly, the mold is tool steel and represents a significant initial cost to the plastics manufacturer. Thus any mold rework, if the cavity does not fill properly or produces defective parts, is undesirable The quality of plastic injection molding parts depends on the material, part and mold designs, and the process parameters required to manufacture them. Mould and die making is an important supporting industry since their related products represent more than 70% of the non-standard components in consumer products. The production runs, however, are typically of small lot-size and with great varieties. The high demand for shorter design and manufacturing lead times, good dimensional and overall quality, and rapid design changes has become the bottlenecks in die and mould industries.

## II. BRIEF OVERVIEW OF THE INJECTION MOULDING PROCESS

The injection moulding industry needs to reduce costs to remain competitive. This need has been addressed using various technologies ranging from design software to computer numerical control machinery. After these technologies are in place and moulding begins the cost is usually based on cycle time. Adjustments can be made to the moulding machine to help reduce the time to mould but in the final analysis the time is dictated by the ability of the mould to carry the heat away from the molten polymer. Liquid is passed through cooling channels in the mould at the required temperature. This must allow the molten polymer to flow into all sections of the cavity while at the same time remove the heat as quickly as possible. Up to now these channels have been produced by drilling which can only produce straight lines. If the channels carrying the water could be conformed to the shape of the part and their cross-section changed to increase the heat conducting area then a more efficient means of heat removal could be realized. This may also help to reduce warpage when the part is ejected, as the plastic would be cooled more uniformly.

### A. Temperature control

Temperatures such as those for the molten polymer, the mould, the surround temperature and the clamping system temperature need to be controlled. When molten plastic is injected in the mould it must be solidified to form the object. The mould temperature is regulated by circulation of a liquid cooler, usually water or oil that flows inside channels inside the mould parts. When the part is sufficiently cooled it can be ejected. Most (95%) of the shrinkage happens in the mould and it is compensated by the incoming material; the remainder of the shrinkage takes place sometime following the production of the part.

### B. Pressure control

Both the injection unit and the clamping system require pressure with the latter developed to resist the former three different pressures can be distinguished in the injection unit: initial, hold and back. All these are obtained by the action of a screw. In the clamping unit the oil pump of the hydraulic system controls the pressure needed to move the mould. Holding pressure is required to finish the filling operation and maintained during solidification to supply the shrinkage.

### C. Time control

Time is the most significant parameter in the entire operation. Cost and machine efficiency can be estimated from the cycle time. The principle temporal aspects to be controlled include: gate-to-gate time, injection time and cooling time.

### D. Thermal proprieties

Despite their large diffusion, for all plastic materials temperature range is a limit to their purpose. Both high and low temperature can create damage to plastic components. It is important to study thermal proprieties to understand and predict this behavior. Therefore cooling times in moulding ma-

The process of filling a mold cavity with a polymer melt is complex. The melt entering the cavity, via the sprue, must be evenly distributed by the runner system to the gates of the part cavity or cavities. Fig. 1 illustrates design terminology for a simple mold with two cavities.

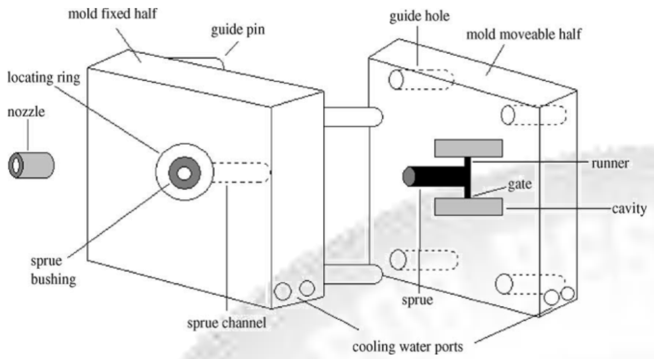


Figure 1: Typical plastic injection mold

### E. Cooling channels

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. if the cooling channels can be made to conform to the shape of the part then the cooling system the cycle time can be significantly reduced with cooling taking place uniformly in all zones. Furthermore, if the part is ejected with the same temperature in everypoint the subsequent shrinkage outside the mould is also uniformand this avoids post-injection warpage of parts. Anotheradvantage is that a mould equipped with conformal channels reaches the operation temperature quicker than a normal one equipped with standard (or drilled) cooling channels.

## III. Methodology adopted for reducing defects and improving die design:

H.S. Kim, J.S. Son, Y.T. Imdeveloped Gate location design in injection molding of an automobile junction box with integral hinges in which a design guideline was induced by investigating resin flow patterns obtained by numerical analyses for several different gate positions of a simple strip with a hinge. The analyses of the simple strip part showed that the resin at the hinge did not flow until the other side of the part was filled. Once the resin at the hinge did not flow for a long time enough to be solidified, short shots or hesitation marks formed due to the secondary resin flow over the partly solidified flow front occurred. Therefore, a design guideline to properly locate gate positions to minimize flow hesitation at hinge areas will be useful to avoid formation of defects. As a practical application of the guideline determined, four gate systems for an automobile junction box cover were designed. Finally, injection molding tryouts using a mold that was designed by one of the proposed gate systems were conducted using polypropylene that contained 20% talc (LUPOL HI-5205).

The predicted flow front for each case was shown in figure 2. As shown in this figure, short shot occurred in Case 1 and 2 while filling was completed successfully in Case 3 and 4.figure 2(a) shows the flow front of Case 1 reached H4 at about 1 s and then, H1, H2 and H3 were filled within 1.6–1.8 s. However, severe flow hesitations occurred at H1 and H4 since the flow fronts at the hinge regions did not advance until the filling of the left and upper sides of the part was completed., flow front temperatures at H1 and H4 reduced to 130 ◦C, which was the transition temperature of the resin and short shot occurred at the corner of the door connected to H4. In addition, the premature hinge failure of H1 was predicted since severe hesitation marks were formed at the hinge due to secondary resin flow over the solidified flow front. As a result, the mold designer's selection without considering the possible hesitation effect may lead to fatal defects.

As shown in figure (b) and Table 1, although H1, H2 and H3 were filled successfully with the reduced hesitations, it was predicted that short shot would occur at H4, where the flow front became colder than the transition temperature. Because H4 was laid vertically, flow front reached the top of the hinge earlier than bottom one so that severe hesitation occurred at the top of H4.

| Case | Flow front temperature at hinge(C) | | | |
|---|---|---|---|---|
| | H1 | H2 | H3 | H4 |
| 1 | 130.0 | 189.3 | 206.1 | 130.0 |
| 2 | 217.7 | 215.8 | 219.8 | 130.0 |
| 3 | 216.6 | 214.2 | 219.8 | 192.5 |
| 4 | 209.8 | 212.2 | 219.2 | 199.8 |

Table-1: The minimum values of the flow front temperature at the hinges

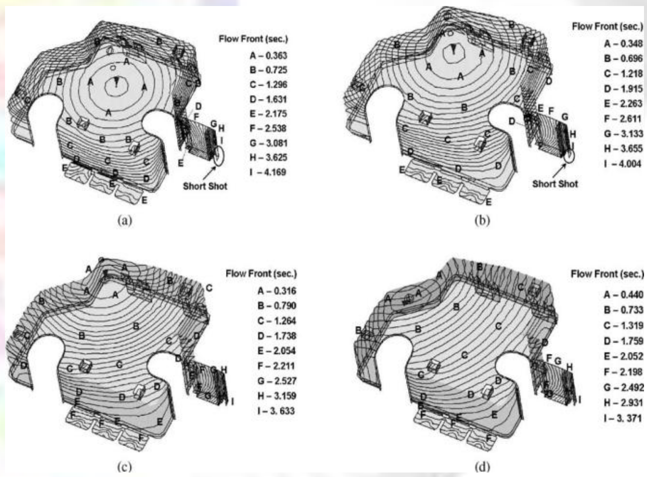


Fig. 2. Predicted flow front for each gate location: (a) Case 1, (b) Case 2, (c) Case 3 and (d) Case 4.

Comparing the flow pattern of Case 3 with those of the previous ones, it can be seen that flow front reached the hinges after the left and upper sides of the part was filled. If there were unfilled regions at the left and upper sides, flow front would not advance through the hinge area since the flow resistance at hinges was larger than that of regions with the basic thickness. Cooling of the flow front of Case 3 was reduced remarkably in comparison to Case 1 and Case 2, as shown in Table 1. Therefore, it was predicted that all hinges were filled successfully in this case. As shown in Fig. 2(d), no possible defects were predicted in Case 4 as well. The flow front, which reached H1 and H4 at about 1.9 s, did not show severe hesitations since the left and upper side of the part had already been filled. Case 4 had the largest value for the flow front temperature at H4 among all four cases, as shown in Table 1.

In this study, flow patterns and occurrence of defects that varied with the different four gate locations were investigated by numerical analyses.

As a result, it could be seen that the proper selection of gate locations can avoid defects such as short shot or hesitation marks by minimizing hesitations at the hinges. Of course, resin flow can be affected by several process conditions such as the melting temperature, mold temperature, injection speed and so on. In general, hesitations at hinges can be reduced by increasing the injection speed and melting temperature due to better fluidity of melted polymer. However, it would be safer to select proper gate locations with moderate process conditions at the mold design stage because there are limitation to the machine capacities and variation of the operator's skill

D.E. Dimla, M. Camilott and Miani developed Design axnd optimization of conformal cooling channels in injection moulding tools in which A design and optimization of conformal channels in cooling an injection-molded component has been conducted using virtual prototypes. The method pursued involved constructing a 3D CAD model of the object, from which the core and cavity of the tool was created. The ensuing simulations showed that it was possible to optimize and predict the best location for such channels to reduce the cooling times when compared to straight-drilled channels.

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. Further-more, if the part is ejected with the same temperature in every point the subsequent shrinkage outside the mould is also uniform and this avoids post-injection warpage of parts. Another advantage is that a mould equipped with conformal channels reaches the operation temperature quicker than a normal one equipped with standard (or drilled) cooling channels.

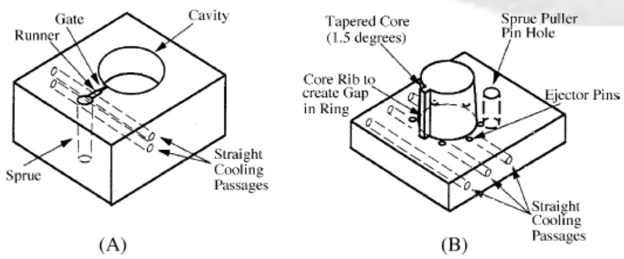


Figure.3 :Cavity (A) and core (B) of a drilled channels mould

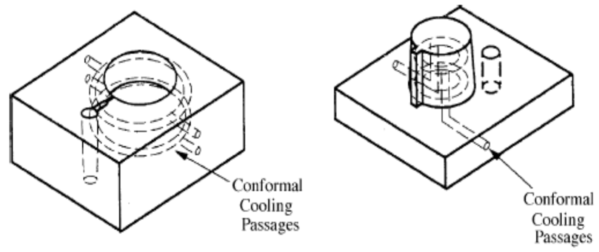


Figure. 4: Same mould as Fig. 4 with conformal channels [3].

In this way one can reduce the time required when the moulding machine is started. When the polymer is injected, it solidifies immediately touching the wall of the mould. If the volume of the part is sufficiently big and its thickness is too small, polymer solidified can obstruct the flow and hinder a complete filling of the cavity. In this case the mould must be heated to a particular temperature in order to permit the polymer to flow. Despite all these advantages it may be noticed that new technologies involved in the production of moulding tools with conformal channels can increase initial costs for the additional complexity of the construction process.

M.C. Song, Z. Liu, M.J. Wang, T.M. Yu and D.Y. Zhao Researched on effects of injection process parameters on the molding process for ultra-thin wall plastic parts and they give suggestion from the researched work as

1) Part thickness is the decisive parameter to the molding of ultra-thin wall plastic parts, the filling capability of the melt declines rapidly with the reducing of part thickness.
2) Metering size and injection rate are the principal factors in ultra-thin wall injection molding. Appropriate metering size is the necessary condition to the molding, accelerating injection rate can make a great increase in the filling ratio.
3) Melt temperature and injection pressure are the secondary factors, but higher melt temperature and injection pressure are also necessary in molding process of ultra-thin wall plastic parts.

Hasan Oktem, TuncayErzurumlu and Ibrahim Uzman were determined the Application of Taguchi optimization technique in determining plastic injection molding process parameters for a thin-shell part. From analysis results, the findings can be drawn as below:

1) Based on ANOVA results, it is apparent that packing pressure (PP) is most important parameter with a percent value of 58.03 on warpage and this was followed by packing pressure time (PPT) of 23.03%, injection time (It) of 15.17%, and cooling time (CT) of 3.68%, respectively.
2) It is also seen that packing pressure time (PPT) have most significant parameters with a percent value of 84.054 on shrinkage this was followed by packing pressure (PP) of 7.83%, injection time (It) of 5.528%, and cooling time (CT) of 2.588%, consecutively.
3) After optimization process, the minimum warpage is calculated to be 0.405 mm, which is less than0.414 mm

in FE analyses of L9 orthogonal array. It can be inferred that warpage is improved by about2.17%.

4) The minimum shrinkage is calculated to be 4.878%, which is less than 4.910% in FE analyses of L9 orthogonal array. It can be concluded that shrinkage is improved by about 0.7%.
5) Regarding verification test, it can be observed that the error between the predicted value of the minimum warpage and the analyzed value of that is found as 1.26%. It can also show that the error between the predicted value of the minimum shrinkage and the analyzed value of that is found as 0.2%.

## IV. Conclusion

Different process parameters like gate positions and conformal channels in cooling are investigated which affect the mold design. Proper gate positions can decrease the shrinkage defect and conformal channels in cooling reduce the production time. Other parameters like part thickness is the decisive parameter to the molding, metering size and injection rate are the principal factors in molding process, accelerating injection rate can bring a great increase in the filling ratio. Taguchi optimization technique is also the way to find the optimum levels of process parameters used in injection part by improving the warpage problem with shrinkage variation.

## References

[1] Hasan Oktem, TuncayErzurumlu, Ibrahim Uzman,“Application of Taguchi optimization technique in determining plastic injection molding process parameters for a thin-shell part”,Materials and Design 28, 2007, pp 1271–1278.

[2] M.C. Song, Z. Liu, M.J. Wang, T.M. Yu, D.Y. Zhao , “Research on effects of injection process parameters on the molding process for ultra-thin wall plastic parts”,Journal of Materials Processing Technology 187–188, 2007, pp 668–671.

[3] D.E. Dimla, M. Camilotto , F. Miani , “Design and optimisation of conformal cooling channels in injection moulding tools”, Journal of Materials Processing Technology 164–165 ,2005, pp 1294–1300.

[4] H.S. Kim, J.S. Son, Y.T. Im , “Gate location design in injection molding of an automobile junction box with integral hinges”, Journal of Materials Processing Technology 140 ,2003 , pp 110–115

[5] R.A. Tatara, R.M. Sievers, ValentinHierzer, “Modeling the injection molding processing of a polypropylene closure having an integral hinge”, Journal of Materials Processing Technology 176 , 2006, pp 200–204