A Simulation Study on Warpage Analysis of Injection Moulded Plastic Part

Sanam Shikalgar¹, Mahesh Zope², Pratik Sonawane³ and Deepit Marathe⁴
¹Student, School of Polymer Engineering, MIT-WPU, Pune 411038, INDIA
²Student, School of Polymer Engineering, MIT-WPU, Pune 411038, INDIA
³Student, School of Polymer Engineering, MIT-WPU, Pune 411038, INDIA
⁴Associate Professor, School of Polymer Engineering, MIT-WPU, Pune 411038, INDIA
³Corresponding Author: shikalgarsanam@gmail.com

ABSTRACT

A part to be injection molded is evaluated by simulation for warpage analysis. The plastic part is a supporting plate to be used in the oil filter and it’s made out of nylon material. The effect of various parameters from design to processing of plastic parts is considered and validated by simulation results. The research involved in this was designing mould, computer-aided engineering, simulation analysis, and determination of plastic part processing conditions.

In this work PA66 (Grade name – Zytel 70G13HS1LNC010) material is used and the material contains 13% of fiber. Fiber orientation is nothing but the distribution of plastic melt inside the cavity and it also plays important role in deciding the warpage of part.

The effect of process parameters on part warpage is investigated from various aspects in comparison with the conventional runner system. Hot runner mould system with innovative cooling channel designs is considered. Results of simulations reveal that elevated mould temperature reduces the unwanted freezing time during the injection phase and thus improves mouldability and enhances part quality. Under similar mould temperature conditions, the effect of process parameters on warpage decreases according to the following order: packing time, packing pressure, melt temperature, injection pressure, and cooling time respectively.

Keywords-- Nylon, Runner System, Simulation, Warpage

I. INTRODUCTION

Injection molding is a key polymer processing technology capable of high accuracy net shaping of high added value products in mass production. The plastic injection molding process consists of filling, packing, cooling, and ejection of a part process[1]. Cooling and solidification of plastic parts need to be accomplished in the shortest possible time with part warpage held within specified limits. Warpage is a critical quality factor for injection moulded parts and warpage on the part occurs as it experiences uneven shrinkage during cooling from high to low temperature[2]. The part needs to shrink uniformly and in a specific amount to achieve the perfectly shaped plastic part. Higher packing pressure helps in reducing the shrinkage and thinner gates give more uniform shrinkage. The difference in cavity pressure, different orientation of fibers, different crystallinity, inhomogeneous thermal stresses, the difference in cooling water temperature on each side of the cavity, and differences in the thickness of mould from the surface of the cavity to the cooling channel are causes of warpage[3].

A commercial software Moldex 3D is used for simulation studies. Simulation software provides designer/engineer with visual and numerical feedback of part behavior & eliminates the traditional trial and error approach for optimization. Warpage is controlled by optimization of process parameter setting which relies on experience and knowledge of experts and involves a great deal of trial and error[4]. Optimization of a part involves optimization for filling, balance, or size runners, optimize cooling, optimize packing profile, and optimize warpage. In optimizing the filling of part, one has to select material, gate location, molding machine, and determine the moulding conditions. Balance/size runners involve adding runners to the part and determine the balance pressure, automatic balancing of runners changes the size of runners so the pressure to fill the part is within tolerance limits. Optimizing cooling channels involves modelling cooling components and run the analysis for various cooling system designs, different coolant temperatures, and cycle time. Optimization of packing profile involves running the analysis for initial packing pressure and packing time and after reviewing the results, determine the packing profile so pressure decay makes more uniform shrinkage. Optimization of warpage involves determining warpage magnitude based on current process conditions, if the warpage is not within acceptable range one has to determine the cause of the warpage. Warpage caused by differential cooling can be controlled by adding cooling channels, adding inserts, and changing coolant temperatures. Warpage caused by area shrinkage can be improved by reducing thickness variation of part, reduce mould temperature variation, reduce packing profile. To overcome the warpage caused due to orientation effects one has to
change processing conditions, change the gate location and reduce mould temperature variation[5].

In this paper, a comparative analysis of feeding and cooling system configurations has been done in terms of ejection time temperature, shrinkage, temperature profile, warpage, sink marks to determine which system configuration is appropriate for this part providing uniform cooling, minimum cycle time, uniform flatness, less warpage and shrinkage[6].

II. EXPERIMENTAL WORK

Moldex 3D is a set of CAE programs used with CAE simulation and practical data to determine optimum results. Unigraphics NX software is used to draw the geometry of the plastic cap and part is imported in Moldex 3D R20 studio mould flow analysis which is carried out by setting parameters like injection pressure, packing pressure, holding time, and mould temperature. The main objective of the experiment is to determine the optimum combination of the feed system and cooling system for the part. According to the functional requirement of part, a part needs to be flat and without sink marks. The part is allowed to warp up to 0.9mm, thus to fit the part for its functional use one needs to lower the warpage of the part.

The conventionally cold sprue feed system is used for injection moulding of plastic parts but as there are issues like temperature drop, high injection pressure affects the duplicity of mould. To overcome this drawback hot sprue moulds are now being used in injection moulding of plastic parts. The cooling system for moulds needs to be built inside the mould plates. Generally, cooling channels are drilled straightway in the XY plane of mould plates and this is termed as simple cooling channels. When the part is of critical cross-section and requires effective cooling, a simple cooling system fails to cool down the part. In this case cooling system need to be designed in such a way that effective cooling of part is achieved. Thus, cooling systems like baffle cooling, bubbler cooling, & thermal pins are used in such cases. Conformal type of cooling is one of the best effective cooling systems, as cooling channels conform to the shape of the part and thus provides uniform cooling. At the same time for critical parts, it becomes difficult to create a cooling system design and thus its manufacturing becomes costly. Thus, moulds with cold sprue and simple cooling channels are cost-effective [7].

Initial simulation runs are carried out only with the feed system to get the idea of heat distribution in mould. The results of this run give the heat distribution profile for the part and this helps in designing the cooling system for the mould. As a simple cooling system is easy to build, maintain and cost-effective too, simulation runs are carried out with the same by varying the temperature of the coolant, the type of coolant, number of cooling channels, and size of cooling channels. Results of runs with cold sprue and simple cooling system fails to reach the target and thus further runs are carried out with hot sprue and various cooling system designs. Thus experimental work of this case is divided into three sections as – 1. Part design 2. Tool and process design 3. Simulation

2.1 Part Design

In this case, a plastic cap has been modelled using Unigraphics NX 11 (Figure – 1). Part is of 113 mm diameter, 13mm height, and 3mm thickness. The part has ribs on the upper side of 2.5mm height and ridges on the inner side of 0.5mm.

![Figure 1: Part design Model](image)

The stp (standard for exchange of product model data) file of the plastic part has been imported to Moldex 3D R20 studio to perform mould flow analysis.

The part in this case study is a functional component of the oil filter of a four wheels vehicle. The oil filter has three main sections base, filter media, and upper cover plate. This part is located below the filter media section and thus needs to withstand the load. Thus the part needs to be stiff and for proper fitment, there is a requirement of flatness. The part has ridges on the inner side for proper fitment with the filter base and ribs on the upper side to provide additional strength and support.

The material to be used for this part is PA66 polymer supplied by DuPont of Zytel 70G13HS1L NC010 grade and it consists of 13 % of glass fibres. The material has good physical properties such as good stiffness and toughness, good electrical and flammability properties, good abrasion and chemical resistance. The material has good high-temperature performance and good melt stability, thus it is mouldable in hot runner moulds. The mechanical properties of the material are shown in Table 1.

2.2 Tool and Process Design

The injection moulding process requires the use of an injection moulding machine, raw plastic material, and a mould. The plastic is melted in the injection moulding process and then injected into the mould where it gets cool
and solidifies into the final part. In this case study, the material is already assigned for the part, so a study of warpage for this part has not concentrated more on the material section. The core and cavity design of the injection mould is what gives the final product its shape. The tool also places a large role in achieving effective cooling. If the plastic material sets at the wrong speed at injection may cause distortion and stresses in part, so here process design plays its role. Under process design, one has to evaluate the optimum combination of sets of parameters like melt temperature, mould temperature, injection speed, injection pressure, packing time, cooling time, etc.

Table 1: Mechanical properties of material

| Mechanical Property | Value          |
|---------------------|----------------|
| Polymer Density     | 1.14 g/cc      |
| Polymer Modulus     | 2e^10 dyne/cm² |
| Fiber weight        | 13 %           |
| Fiber density       | 2.55 g/cc      |
| Fiber shear modulus | 3e^11 dyne/cm² |
| Fiber L/D ratio     | 20             |

Injection mould tool consists of the following components: Fixed and moving backplate, fixed and moving supporting plate, ejector plate, ejector plate retainer, bolster sleeves, guide bushes, core retaining plate, mould core, and cavity plate, cavity retaining plate, sprue bush, locating ring, etc. Essential elements of the core and cavity are gate and runner. Designing of gate needs to be done carefully to ensure correct filling of core and cavity. As placement of gates always remains visible on a moulded part so the aesthetic of the part needs to be considered.[8] In this case study best gate location analysis is carried out and as the part is uniform in shape it is found to be located in the centre of the base side. Figure 2 and Figure 3 show the injection mould tool and its cross-sectional view respectively.

Runners are passages that distribute molten material from sprue to gates around the cavity inside a mould. Runners slow down and smooth the outflow of liquid and are designed to provide approximate flow rates to various parts of mould cavity[8]. As there exist two types of runner system namely cold runner and hot runner, both these types of runner system are analyzed in this case.

2.2.1 Cold Sprue Mould

Cold sprue moulds are generally used in an injection molding process. All types of material can be moulded in cold sprue moulds. They are easy to manufacture and their maintenance is also easy. The main drawbacks of a cold runner system relate to cost, waste, and ability to hold tolerances. As cold runner consumes more material in each cycle it adds to the cost and waste. During demoulding of part, both core and cavity plates are moved apart to eject the part. After the opening of plates, the surface for moulding comes in contact with the environment and there is the chance to happen temperature drop in the mould. The difference in temperature in both mould plates may lead to warpage of part, also because of temperature changes.
2.2.2 Hot Sprue Mould

As cold sprue moulds generate scrap, reprocessing of this material is generally difficult and that’s why hot sprue moulds come into the picture. To maintain the temperature in both halves of hot sprue mould is a difficult task. Hot sprue moulds have material limitations for moulding as they require materials possessing good thermal stability and a generous processing window. There is more need to balance the flow compared to cold sprue moulds. These moulds are more advantageous over cold sprue moulds as 100% of the material is used during the moulding process. At the same time, these moulds are critical to manufacturing and thus are expensive ones. These moulds also require higher maintenance over time. Hot runner systems are better suited for parts that require tight tolerances. In a hot runner system as cavity and core plates are at high temperature, melt reaches easily to fill the part, and thus it requires lower injection pressure. As melt front reaches the corners of part at high temperature, lower packing is required and as melt fronts meet at high temperatures, there are fewer chances of weld line [8].

2.3 Design of Cooling System

The cooling system is a key component of the well-designed injection moulding process. The appropriate location for cooling channels is inside mould plates with the proper distance apart from mould surface and between successive cooling channels. Fig. 4 shows that distance(D), from cavity surface to centre of cooling channels should be one to two times of cooling channel diameter. The pitch of the cooling channel should lie between 3d to 5d, whereas the typical channel diameter ranges from 10 to 14 mm. The coolant inlet and outlet of a cooling system can be placed on the same side or opposite side [9].

![Figure 4: Cooling channel dimensional parameters](image)

In this case, water is selected as the coolant liquid for all analysis as it has a high cooling property, economically viable, and environmentally friendly. The coolant Reynolds number was calculated to be 9750 and the temperature at the coolant inlet was 25°C, which indicates that the flow of water was fully turbulent. Here, the following layouts of cooling channels are analysed.

1. Simple Cooling System

![Figure 5: Simple cooling system](image)

A simple cooling system is a conventionally used cooling system in injection moulds and this has the straight drilled cooling channels in both the core and cavity side of mould plates shown in Fig. 5. The distance between the cooling channel and the moulded product is controlled at about 20 mm, coolant may be water or oil is passed through the cooling channel and purpose heat dissipation is achieved.

In this case, a simple cooling system is designed with a circular cross-section as per the following dimensions.

- Diameter of the cooling channel – 8 mm
- Length of the cooling channel – 100 mm
- Distance between centre of the cooling channel and mould plate base – 20 mm

2. Baffle Cooling System

![Figure 6: Baffle cooling system](image)

In this system baffle is a cooling system component that is constructed by inserting a metal plate in the cooling channels. Areas of the mould which cannot be cooled effectively by simple cooling may require the use of baffles. Baffles divert the coolant flow into areas that would normally lack cooling.

In this design system, both cavity and core sides of mould plates have a baffle type of cooling system embedded in it. For this case dimensions of the cooling system are as follows:

- Diameter of the channel – 8 mm
- Diameter of baffle – 3 mm
- Length of baffle – 5 mm
Distance from center of cooling channels to the base of mould plate – 20 mm

3. Conformal and Plate Type of Cooling System

Conformal cooling channels actually outlines the geometry of moulding cavity and therefore it promotes faster and uniform temperature control and significantly increases the performance of the injection cycle on both production rate and quality of plastic part [4]. In this case, conformal cooling is accomplished with the help of a plate, the coolant flows through that plate and provides effective cooling on the core side. Ribs are provided on the core side of mould and it takes more time for cooling as compared to the other side and thus it requires more cooling on that side.

Dimensions of cooling channel for the conformal cooling system is as follows
- Diameter of the cooling channel – 8mm
- Length of the cooling channel in X & Y direction – 100mm
- Diameter of the plate – 50mm

Table 2: Experimental condition and test observation.

| Sr. No. | Melt Temperature | Injection pressure | Packing pressure | Packing time | Cooling time | Warpage | Cycle time |
|---------|------------------|--------------------|------------------|--------------|-------------|---------|------------|
| 1       | 285              | 190                | 190              | 10           | 2.9         | 1.846   | 19.1       |
| 2       | 285              | 190                | 190              | 10           | 3           | 1.832   | 19.2       |
| 3       | 285              | 190                | 190              | 10           | 3.1         | 1.815   | 19.3       |
| 4       | 285              | 200                | 200              | 15           | 2.9         | 1.811   | 19.1       |
| 5       | 285              | 200                | 200              | 15           | 3           | 1.806   | 19.2       |
| 6       | 285              | 200                | 200              | 15           | 3.1         | 1.796   | 19.3       |
| 7       | 285              | 210                | 210              | 20           | 2.9         | 1.774   | 19.1       |
| 8       | 285              | 210                | 210              | 20           | 3           | 1.761   | 19.2       |
| 9       | 285              | 210                | 210              | 20           | 3.1         | 1.753   | 19.3       |
| 10      | 285              | 210                | 210              | 20           | 3.1         | 1.741   | 19.2       |
| 11      | 295              | 210                | 190              | 20           | 2.9         | 1.732   | 19.3       |
| 12      | 295              | 190                | 190              | 10           | 2.9         | 1.736   | 19.3       |
| 13      | 295              | 200                | 200              | 15           | 3           | 1.736   | 19.3       |
| 14      | 295              | 200                | 200              | 15           | 3           | 1.721   | 19.2       |
| 15      | 295              | 200                | 200              | 15           | 3.1         | 1.713   | 19.3       |
| 16      | 295              | 210                | 210              | 20           | 2.9         | 1.709   | 19.1       |
| 17      | 295              | 210                | 210              | 20           | 3           | 1.700   | 19.2       |
| 18      | 295              | 210                | 210              | 20           | 3.1         | 1.692   | 19.3       |
| 19      | 295              | 210                | 190              | 10           | 2.9         | 1.685   | 19.1       |
| 20      | 305              | 190                | 190              | 10           | 3           | 1.677   | 19.2       |
| 21      | 305              | 190                | 190              | 10           | 3.1         | 1.665   | 19.3       |
| 22      | 305              | 200                | 200              | 15           | 2.9         | 1.661   | 19.1       |
| 23      | 305              | 200                | 200              | 15           | 3           | 1.654   | 19.2       |
| 24      | 305              | 200                | 200              | 15           | 3.1         | 1.648   | 19.3       |
| 25      | 305              | 210                | 210              | 20           | 2.9         | 1.639   | 19.1       |
| 26      | 305              | 210                | 210              | 20           | 3           | 1.628   | 19.2       |
| 27      | 305              | 210                | 210              | 20           | 3.1         | 1.612   | 19.3       |
Graphical representation of simulation run results for optimization of process parameters

1.6
1.65
1.7
1.75
1.8
1.85
1.9
200
220
240
260
280
300
320
340
MELT TEMPERATURE/INJECTION PRESSURE/PACKING PRESSURE/PACKING TIME/Cooling TIME

0
20
40
60
80
100
120
140
160
180
200
220
240
260
280
300
320
340
WARPAGE VALUE

warpage Injection pressure Packing pressure Packing time Cooling time Melt temperature
2.2.4 Process Design

Several process parameters influence the quality of injected parts during injection moulding. The shrinkage and the warpage are observed factors, and Moldex 3D is used to conduct this parametric study to investigate experimentally the effect of each process parameter on shrinkage and warpage. Designing the process is nothing but optimization of process parameters and to conduct this study on case study part four parameters at three levels are considered. The orthogonal array L27 is as shown in table -2 which has 27 rows corresponding to the number of simulations runs with required columns.

| Table 3: Process parameters for study |
|--------------------------------------|
| Melt temperature                     | 285  | 295  | 305  |
| Injection pressure                   | 190  | 200  | 210  |
| Packing pressure                     | 240  | 250  | 260  |
| Packing time                         | 10   | 15   | 20   |

Table 4: Optimized Process Parameters

| Maximum injection pressure          | 200 Mpa |
|-------------------------------------|---------|
| Maximum packing pressure            | 220 Mpa |
| Melt temperature                    | 295℃    |
| Mould temperature                   | 80℃     |
| Filling time                        | 1.2 sec |
| Packing time                        | 20 sec  |
| Cooling time                        | 3.1 sec |
| Ejection temperature                | 190℃    |

Melt temperature is assigned by the first column, the injection pressure is assigned by the second column, packing pressure is assigned by the third column, and packing time is assigned by the fourth column[10]. The material process parameters and their levels are discussed in table -3.

Table 2 shows the experimental result of warpage and cycle time at different combinations of process parameters. Throughout the experiment filling time and mold open time are 1.2 sec and 5 sec respectively. All the observations from table -3 are plotted on the graph as shown in fig. 8. The figure clearly shows the process parameters that give the lowest warpage value and these are the optimized process parameters. Table 4 shows the optimized process parameters.

2.3 Simulation

Mould flow analysis is performed after assigning PA66 DuPont of Zytel 70G13HS1L NC010 grade material to the plastic part. Moldex 3D R20 the studio has been used to perform the analysis. Solid mesh is used for a part with 1,81,583 elements. First of all gate location analysis is performed and the best location is in the center and base of the part. After this moulding window analysis is performed to evaluate optimum material conditions required for part production. Moulding window of this case study is shown in Figure - 9, injection time is shown on the x-axis whereas the y-axis shows mould temperature.

![Figure 9: Moulding window for case study](image)

The moulding window is divided into several regions from which region of blue colour is preferred over others and regions in green and yellow colours are feasible ones whereas the red-coloured region is not feasible. At an optimum point in moulding window, the melt temperature is 295℃, mould temperature is 75℃, and injection time is 1 second. Using these optimal conditions Fill+Pack+Warp+Cool analysis is performed. Figure 10 shows the Filling melt front time result for the run and it clearly shows that part fills 100% at the optimum point of moulding window.

![Figure 10: Run result for filling melt front time](image)

To determine the optimum combination of the feed system and cooling system following seven simulation runs are performed and the results of the warpage are studied.

1. With cold sprue and without cooling channel

This simulation is carried out to get the idea of heat distribution along the part and thus helps in designing the cooling channels.
2. With cold sprue and with a simple cooling channel
   This simulation is carried out by adding simple cooling channels to the system to study the effect of cooling on part quality.
3. With hot runner and without cooling channel and heating coil
   This simulation run is carried out to see the heat distribution along part without considering the heating coil.
4. With hot runner and with heat bush considering heating coil
   This simulation run is carried out to get an actual idea of heat distribution on part and thus area from which maximum heat needs to be removed.
5. With hot runner + heat bush + heating coil and simple cooling channel design
   Simple cooling channels are introduced in this system to configure the effect of cooling on part quality or part warpage.
6. With hot runner + heat bush + heating coil and baffle cooling channel design
   For efficient cooling and better quality of the part, a simulation run with baffle cooling channels is carried out.
7. With hot runner + heat bush + heating coil and conformal plate type cooling system
   To achieve the target of part warpage cooling system of conformal type is designed and runs are executed to study warpage on a part.

Table 5: Results of filling sprue pressure in cold and hot runner

| Sr. No. | Melt Temperature (°C) | Injection Pressure (MPa) | Cold Runner Sprue Pressure (MPa) | Hot Runner Sprue Pressure (MPa) | Sprue pressure gain with use of hot runner (MPa) |
|---------|-----------------------|--------------------------|---------------------------------|---------------------------------|-----------------------------------------------|
| 1       | 285                   | 200                      | 28.326                          | 26.899                          | 5.03                                          |
| 2       | 295                   | 200                      | 25.111                          | 22.676                          | 9.69                                          |
| 3       | 305                   | 200                      | 23.075                          | 20.477                          | 11.25                                         |

IV. RESULTS AND DISCUSSION

For determining the optimum combination of process parameters various numbers of runs are carried out as shown in table- 3. As both types of feed systems viz; cold and hot runner systems are considered in the analysis, to find the optimum feed system simulation runs are carried out and sprue pressure results for hot and cold runner systems are shown in table – 5.

4.1 Results of Cold and Hot Runner System
A. For melt temperature of 285°C and injection pressure of 200 Mpa

B. For melt temperature of 295°C and injection pressure of 200 Mpa

Figure 11: Sprue pressure (a) Cold sprue – 28.326
(b) Hot sprue – 26.899
The simulation results in terms of sprue pressure are discussed here for selecting the optimum feed system. The results show significant pressure gain while using the hot runner system. At a mean process temperature of 295°C, the cold runner system requires 25.111 MPa of sprue pressure while the hot runner system requires 22.676 MPa of sprue pressure which means a pressure gain of 9.69%. The pressure gain at other processing temperatures of 285°C and 305°C is 5.03% and 11.25% respectively. When using the cold runner system, the flow of molten plastic becomes more difficult because of heat dissipation in runners, use of the hot runner system provides significant pressure gain by eliminating the disadvantage of the cold runner system. Thus hot runner injection moulding gives pressure gain and it provides saving in required power and results in less energy consumption. (Reference) So in this case study hot
runner injection moulding is finalized as the optimum feed system.

The next parameter is the cooling system. For determining the optimum combination of the feed system and cooling system simulations are carried out as discussed earlier in section 3.

4.2 The Results of Warpage Analysis
1. With cold sprue and without cooling channel - (Warpage Value – 1.739)

2. With cold sprue and with simple cooling channel – (Warpage Value – 1.695)

3. Hot runner system without cooling channel and heat bush – (Warpage value -1.832)

4. With hot runner and with heat bush considering heating coil – (Warpage value – 1.987)

5. With hot runner + heat bush + heating coil and simple cooling channel design – (Warpage Value - 1.652)

6. With hot runner + heat bush + heating coil and baffle cooling channel design - (Warpage Value – 1.454)

7. With hot runner + heat bush + heating coil and conformal plate type cooling system – (Warpage value – 1.391)
Table 6: Summary of warpage analysis results

| Run    | Warpage Value |
|--------|---------------|
| Run – 01 | 1.739         |
| Run – 02 | 1.695         |
| Run – 03 | 1.832         |
| Run – 04 | 1.987         |
| Run – 05 | 1.652         |
| Run – 06 | 1.454         |
| Run – 07 | 1.391         |

In the above warpage analysis results, the highest value of warpage is displayed as red colour. The summary of warpage results is displayed in Table 6 and thus it shows that run – 07 i.e.; hot runner system with heat bush, heating coil, and conformal and plate type of cooling system shows the minimum warpage among other runs. Run -01 & Run -02 are carried out on the cold runner system to investigate the difference in the warpage value with the hot runner system. From the results of run -02 and run -05, we can surely say that hot runners are better at reducing warpage compared to the cold runner system. Results of Run -05 are for hot runner and simple cooling channel design, so to evaluate the effect of cooling on part quality effective cooling with baffle and conformal type of cooling systems are provided in run -06 and run -07 respectively. Thus based on results we can say that conformal and plate type of cooling system with a hot runner is most efficient and suitable for the case study part among others. It has higher sprue pressure gain and lower warpage value, thus it leads to better part quality with minimum cycle time.

V. CONCLUSION

In this case study two feed systems are studied viz; Cold and hot runner with three types of cooling channel designs namely, simple cooling, baffle cooling, conformal and plate type of cooling. Simulations for filling, packing, cooling, and warpage are performed on Moldex 3D software. From simulation results, we can conclude that from the result of sprue pressure, hot runner provides more pressure gain compared to cold runner and thus it is of optimum use. Whereas results of warpage analysis taken for different cooling channel designs show that conformal and plate type of cooling system provides more uniform cooling in less cooling time.

REFERENCES

[1] Guo W, Hua L, & Mao H, et al. (2012). Prediction of warpage in plastic injection molding based on the design of experiments. J Mech Sci Technol, 26, 1133–1139.
[2] Chen DC, Huang CK, & Lin JW. (2017). Warpage analysis of a plastic spur gear injection molding. Smart Sci; 5, 206–213.
[3] Liao SJ, Chang DY, & Chen HJ, et al. (2004). Optimal process conditions of shrinkage and warpage of thin-wall parts. Polym Eng Sci, 44, 917–928.
[4] Mohamed OA, Masood SH, & Saifullah A. (2013). A simulation study of conformal cooling channels in plastic injection molding. Int J Eng Res, 2, 344–348.
[5] http://marciaswan.weebly.com/uploads/1/3/5/8/13585698/mfdg_low-res_proof_part2.pdf.
[6] Amran M, Salmah S, & Izamshah R, et al. (2015). Warpage analysis of different number cooling channels for dumbbell plastic part in injection moulding. Appl Mech Mater, 761, 8–11.
[7] Venkatesh G, Ravi Kumar Y, & Raghavendra G. (2017). Comparison of straight line to conformal cooling channel in injection molding. Mater Today Proc, 4, 1167–1173.
[8] Hidayah MHN, Shayfull Z, & Nasir SM, et al. (2017). Warpage analysis in injection moulding process. AIP Conf Proc; 1885. DOI: 10.1063/1.5002244.
[9] Kitayama S, Yokoyama M, & Takano M, et al. (2017). Multi-objective optimization of variable packing pressure profile and process parameters in plastic injection molding for minimizing warpage and cycle time. Int J Adv Manuf Technol, 92, 3991–3999.
[10] Singh G, Pradhan MK, & Verma A. (2018). Multi response optimization of injection moulding process parameters to reduce cycle time and warpage. Mater Today Proc, 5, 8398–8405.