Cooling Channel Design for Multi-Cavity Plastic Injection Moulds

K. Poornima[1], M. N. M Ansari[2]

[1, 2]Centre for Advanced Materials,
Universiti Tenaga Nasional, Kajang, Selangor, 43000, Malaysia

Abstract: ***Plastic injection moulding is vastly used in today's manufacturing industry. It is a preferred method compared to other types of manufacturing processes because less surface finishing is needed for the parts produced by injection moulding. In manufacturing field, finishing processes such as surface treatment, metallization and heat treatments are carried out in order to improve the surface quality. The additional processes contribute to higher manufacturing costs and can be avoided by employing an automated process such as plastic injection moulding. The mould can be repeatedly used, making plastic injection moulding appropriate for mass production. The whole cycle time starts from filling, packing, cooling and part ejection (mould opening). Cooling phase takes more than 2/3 of the whole cycle time making it the most dominant factor among the other components of the cycle. The influence of cooling channels will be discussed by comparing straight drilling cooling channels (SDCC) with conformal cooling channels (CCC). A multi-cavity rectangular plate was analysed by employing 4 different cooling systems. Through-out the analyses, chilled water and thermoplastics which represent at least 90% of all plastic were considered and set as constants.***

Keywords: conformal cooling channels, straight drilled cooling channels, cooling time, cooling channels design.

1. Introduction

The complexity of the products has increased with more features. The current development of the industry is responsible for producing difficult geometry shapes besides ensuring shorter cycle time for the whole process to take place. Coping with their rapid and vast production, the products must be manufactured in the shortest time possible. However, the intention of increasing productivity should not comprise the quality of the products. Cooling stage plays a vital role; in fact, more than 2/3 of the cycle time depends on it solely [1].

A common practice in the industry is when the cooling design is left to the be considered at the last stage of the design and the cooling channels are built in whatever space left after the ejector pins or other elements have been placed [2]. The complexity of the design is given more time and effort as channels are done based cooling on the toolmaker's determination besides their experience. In this research, since chilled water is the dominant coolant, only thermoplastics which represent at least 90% of all plastics and water as the coolant are considered. Based on the background study done on the types of cooling channels available currently [3, 4, 5], it can generally be divided into two main categories which are straight drilling cooling channel (SDCC) and conformal cooling channels (CCC). Through SDCC, the part will be forced to cool unevenly as the temperature distribution is not uniform due the variation of the distances between the cavity's wall and cooling lines [6]. The conventional way can be efficient for simple parts but when the level of intricacy drops, the efficiency follows suit. Time allocation is not given specifically when it comes to designing cooling system. It is usually squeezed in whatever space left. Of course, by avoiding this and optimizing the design, efficiency can be increased.

The plastic injection moulding cycle is considered cyclic where a series of stages are repeated in each cycle [7]. The process which is continuous starts from the molten polymer, in this case, thermoplastics, is injected in through the gating system after the moulds have closed. It is then followed by the filling stage; the melt spreads with the help of pressure gradient. Cooling system aids in the solidification of the melt. As soon as it reached the ejection temperature, the mould opens for ejection to take place. A basic mould unit comprises of the sprue, the runners, the gates and the cavities.

The various phases involved in plastic injection moulding are influential when it comes to preserving the product's quality [8]. Filling phase, as explained in the previous section, is the melt flow into the cavity from injection location through sprue, runners and gate. Variation in pressure or pressure gradient enables the fill-up to happen. However, the injection location is also a contributing factor [9]. Just like how current would choose a path with lesser resistance to travel through, the melt does it the same where the path which has the least restriction or obstruction will be chosen to fill up first.

Packing phase is ensuring the volume of the part is the same as the volume of the original model after the melt having filled up the cavity. Under-packing happens when the weight of the plastic is lesser than the theoretical value and over-packing happens when it is more. Over-packing will have built-in stresses that usually will be relieved after the moulding process as warpage while decrease in volume will result in voids and shrinkages [10]. It is very essential in the case of identifying shrinkage allowance that should be given to the part in order to compensate it. During the filling time, pressure gradient helps in filling up the cavity. In a similar manner, during the packing phase, the melt flows from a denser region to a less dense one till it is symmetrically distributed. Two other parameters closely related to this phase are compressibility and thermal expansion [11].

Cooling phase starts after the filling, packing and holding process and ends just when the part reaches the ejection temperature. End of cooling phase indicates de-moulding. Heat must be transferred from the melt to ease solidification of the part. This means if the cooling is improper, it will affect the quality of the produced part. Cooling takes more than 2/3 of a cycle time where by manipulating the related parameters, cooling time can be improved tremendously. Cooling happens by removing heat through conduction, convection and radiation. The heat from the melt is transferred from the cavity to the coolant which is a moving fluid is based on convection. During heat transfer, the mould acts as a heat exchanger where it allows heat to be transferred in and out.

Logically, the more time it takes to cool; there will be a decrease in productivity. There are a lot of factors contributing to the cooling phase in the cycle time but the objective is just one that is to transfer the heat effectively out of the mould. Uniformed part thickness is a must [12]. If the part has same thickness throughout, the pattern flow will be symmetrical and when that happens, cooling takes place effectively. Besides that, having features such as undercuts and complicated part designs, the cooling will be affected as well as the fill pattern. Carrying out cooling quality analysis, results on temperature variance, cooling time variance and cooling quality can be observed before allocating the appropriate cooling systems to the mould using computer aided manufacturing software [13].

2. Methodology

2.1 Computer Aided Moulding Simulation Workflow

Figure-1 shows the methodology designed in carrying out the simulations in Autodesk Moldflow. It started by evaluating the design for nominal wall thickness as well for draft angles. Inappropriate draft angles would lead to manufacturing defects. The part was scanned through the gate location analysis for the possible locations before the other analyses were carried out.

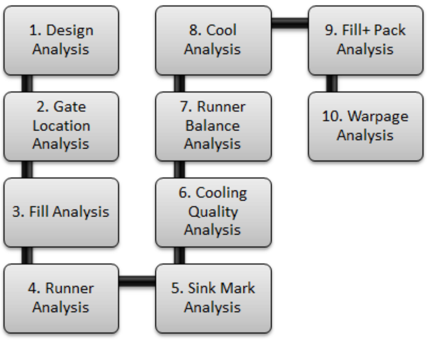

Figure 1: Sequence of Autodesk Moldflow Analyses

2.2 Part Design

The rectangular plate was designed using Pro-E Wildfire (version 5) before it was imported to Autodesk Moldflow. The overall dimension of the imported part was 120 (length) x 50 (width) x 3 (thickness) in mm. In Autodesk Moldflow, a multi-cavity was designed by employing appropriate sprue and runner system to accommodate four similar sized rectangular plates. Tables 1 till 3 show the general specifications of the imported part.

Table 1: Material properties of the melt

Injection Moulding Parameters	*Material*
Material manufacturer	Generic Default
Material trade name	Generic PP
Melt temperature (°C)	240.0
Mould temperature (°C)	40
Injection locations	4
Max machine injection pressure	180.00 (MPa)

Table 2: Injection moulding parameters of the mould

Injection Moulding Parameters	*Mould*
Mould Dimensions (mm) X Y Z	 393.80 218.80 50.00
Mold Material Material manufacturer Material trade name	 Generic Tool Steel P-20
Mold Plates Dimensions (mm) A plate B plate	 25.00 25.00

Table 3: Coolant properties

Injection Moulding Parameters	*Coolant*
Medium	Water (pure)
Temperature	25 °C
Flow rate	10.00 (lit/min)
Reynolds Number	> 30000

2.3 Cooling Channels Designs

Four designs of cooling channels were designed by taking into account the cooling design considerations on both CCC and SDCC and experimented on the 3mm rectangular plate (refer to Figures 2 till 5). Pure water of 25°C was used as the medium of the coolant. 10.00 L/min was set as the flow rate of the coolant with a Reynolds Number of above 20000 to ensure a turbulent flow. Cooling channels diameter was 6mm with the hose diameter as 12 mm. Hose diameter had to be bigger than the cooling channel diameter. By decreasing the area, the velocity of the coolant was increased and thus provided a more turbulent flow with a higher Reynolds number. It was utmost important to have a turbulent flow as it would aid in removing the heat from the mould efficiently

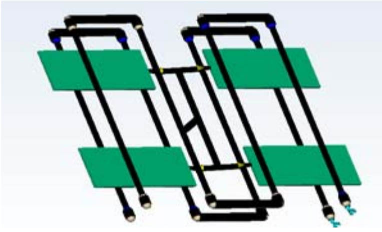

Figure 2: In-line-transverse (In-line-T) Cooling Channel Type1- (D1)

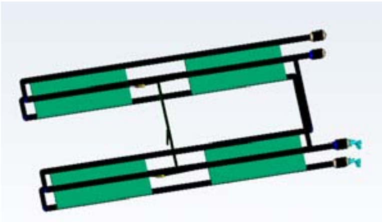

Figure 3: In-line-longitudinal (In-line-L) Cooling Channel Type2- (D2)

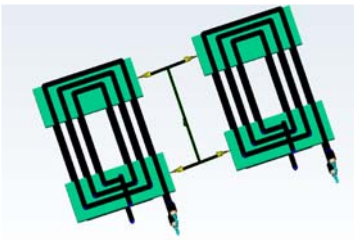

Figure 4: Spiral Cooling Channel Type3- (D3)

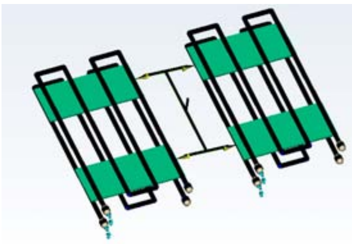

Figure 5: Zig-Zag Cooling Channel Type4- (D4)

3. Results and Discussion

3.1 Before the Integration of Cooling Channels

The melt chooses the path which had the least restrictions or obstructions to fill up first. Gate point was where the melt started entering into the cavity and it was very essential to ensure a well-balanced flow. This was achieved throughout the analyses carried out on a multi-cavity part (refer to Figures 6 till 8) and it was proved by observing the color distribution (refer to Figure 6) where the melt started to fill up the cavities uniformly and reached the edge of the each cavities at a same time. The flow of the melt was indicated by different colours at each region. The region in red was the end of the cavity and took the longest time to get filled if compared with the entire part. A well balanced flow was ensured with a proper runner system and injection locations. As Figure 6 depicts, the runner system designed was functioning well because the melt flow was regulated within the mould cavities. The fill time obtained as in Figure 6 would be lower than the fill time obtained after the integration of cooling channels. It will be further elaborated at later sections

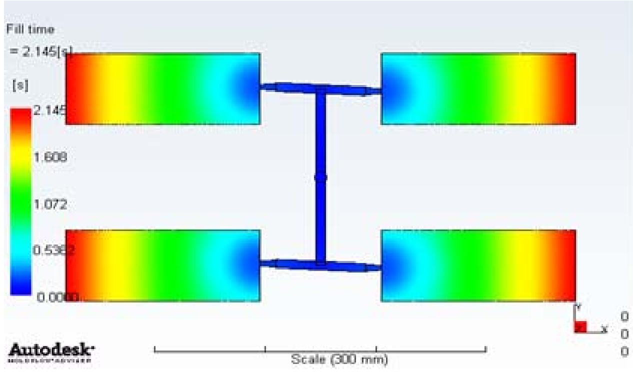

Figure 6: Fill time of the multi-cavity part in Fill Analysis

Difference in pressure was the driving force which allowed the melt to fill up the cavity. The pressure started increasing at a specific location when the melt reached the location. The increase in pressure at a particular location would push the melt forward to another location of lesser pressure during the filling process. The result of injection pressure required for the part before the integration of cooling channels (refer to Figure 7) is 18.51 MPa which was much lesser compared to the injection pressure required for the part experimented with different cooling channel designs. This was because when the fill time increased, the melt cooled down parallel to that. Increase in fill time means there was an increase in viscosity and thus it was valid to have a higher injection pressure to fill up the cavity. As it was observed in Figure 7, the pressure drop was uniformed throughout each of the cavity and this was mainly due to a properly designed runner system, without neglecting the injection locations as well. The uniformity in pressure drop was indicated by the different colours indicating each region of the part. Red region indicated the highest injection pressure while darker blue, the lowest.

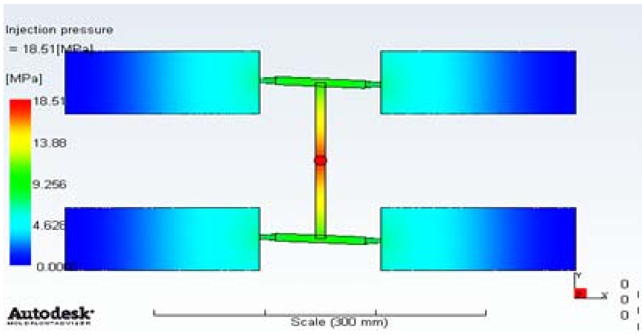

Figure 7: Injection pressure of the multi-cavity part in Fill Analysis

Cooling time variance was the time taken by the melt to actually solidify in any location of the part with respect to the average time the whole part takes to freeze. The area around the injection location seemed to have the highest difference as that was when the hot melt entered through the sprue, runners and gate. The region having the lowest cooling temperature variance which means it took lesser time to freeze in comparison with average time. The significance of this analysis was to facilitate cooling designs. Based on Figure 8, it was obvious that the middle portion of the part takes the longest time to freeze in comparison with the entire part. An appropriate cooling channels design can be designed based on the obtained result in order to aid the injection moulding process. Different cooling channels design will give different results and the effectiveness of each designed will be discussed later.

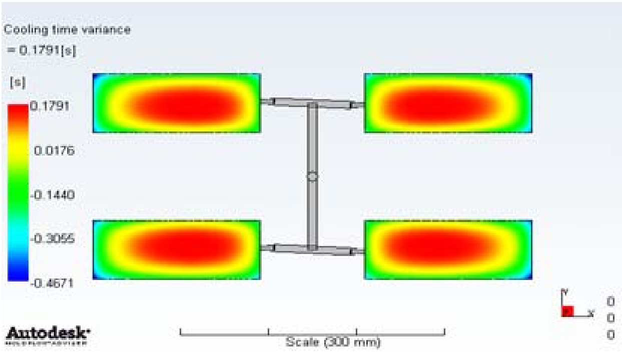

Figure 8: Cooling time variance of the multi-cavity part in Fill

The estimated cycle time was 26.00s (refer to Figure 9) inclusive of three main phases which are the filling phase, estimated pack and cool phase and the mould opening phase. The estimated cooling time was without the integration of cooling channels.

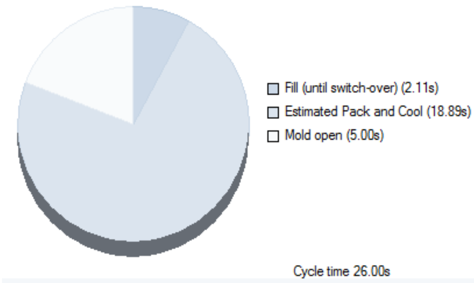

Figure 9: Cycle time

3.2 After the Integration of Cooling Channels

The Based on Figure 6, the filling analysis showed a fill time of 2.145s which was without the integration of cooling channels. There was an obvious difference before and after the integration of cooling channels as there was an increase in fill time (refer to Table 4). Basically, the temperature of the part was higher without cooling channels. A decrease in temperature would decrease the viscosity level and by all means, restricting the fluidity of the melt. That was why the fill time of the part without the cooling channels was lesser as the viscosity level was lower and melt flow was not restricted. By having employed cooling channels, the temperature of the part decreased, reducing the viscosity as well as increasing the resistance to flow freely. This could be observed in Table 4 where after the integration of cooling channels, the temperature decreased, increasing the viscosity, thus it took more time for the melt to fill up and it ranges from 2.310s to 2.313s. The more effective the cooling channel design, the longer the time it took to fill up. Among the four designs, spiral cooling channel had the highest fill time as the temperature of the part was much lower in comparison to the other 3 designs.

Table 4: Fill Analysis with the integration of cooling channels

Injection moulding parameters	*In-line-T Cooling Channel*	*In-line-L Cooling Channel*	*Spiral Cooling Channel*	*Zig-Zag Cooling Channel*
Fill time (s)	2.310	2.312	2.313	2.311
Injection pressure (MPa)	44.580	44.580	44.850	44.740
Pressure drop (MPa)	44.580	44.580	44.850	44.740
Flow front temperature (°C)	241.60	241.60	241.60	241.60
Time to reach ejection temperature (s)	22.70	20.86	20.59	20.83

Difference in pressure triggered the driving force which allowed the melt to fill up the cavity. The pressure started increasing at a specific location when the melt reached the location. The increase in pressure at a particular location would push the melt forward to another location of lesser pressure during the filling process. The result of injection pressure required for the part before the integration of cooling channels (refer to Figure 7) was 18.51 MPa which was much lesser compared to the injection pressure (44.580 MPa to 44.850 MPa) required for the part experimented on different cooling channel designs (In-line-T, In-line-L, Spiral and Zig-Zag). When the fill time increased, the melt cooled down parallel to that. Increase in fill time means there was an increase in viscosity and thus it was valid to have higher injection pressure to fill up the cavity. Spiral cooling channel had the highest injection pressure because of the increased length of cooling channels.

As for the circuit pressure, in-line-T cooling channel (refer to Table 5) triggered the least circuit pressure which was 161.300 kPa among the rest because the design did not cover the entire area and the total cooling channels length was lesser than the ones in spiral cooling channel (refer to Table 5). Spiral cooling channel was based on CCC where the cooling channels take the shape of the rectangular plate and this could be the reason which allowed more heat transfer to occur relative to uniform cooling. As the design had longer cooling channels, the circuit pressure was the highest which was 372.600 kPa. The circuit pressure was the highest at the inlet and lowest at the outlet and it gradually decreased from the inlet to the outlet. If the pressure was too high, sometimes pressure had to be supplied externally by the means of a pump or a device as such in order to aid the coolant flow. Having a pump could increase the plastic injection moulding cost.

Table 5: Cool Analysis with the integration of cooling channels

Injection moulding parameters	*In-line-T Cooling Channel*	*In-line-L Cooling Channel*	*Spiral Cooling Channel*	*Zig-Zag Cooling Channel*
Circuit pressure (kPa)	161.300	281.600	372.600	212.400
Coolant temperature (°C)	25.700	25.880	25.310	25.450
Circuit's Reynolds Number	39363	39363	39363	39363
Cooling time variance (s)	0.7969	0.2832	0.1494	0.4040
Temperature part (°C)	53.290	47.310	39.250	48.030
Temperature variance (°C)	5.351	3.167	1.884	3.76
Time to reach ejection temperature (°C)	17.82	16.75	15.71	16.89
Cycle time (s)	23.77	19.36	18.68	19.58

Temperature variance shows the difference in the temperature of a region in comparison to the average temperature with respect to the geometry of the part as well as the cooling channels at the end of the cycle. Temperature variance was the highest at the injection location. The lowest was away from the injection location. By observing the temperature variance of each part together with the cooling channels, the effectiveness of the cooling channels could be measured through the reduction of cycle time. The multi-cavity rectangular plates were experimented using the four designs with the integration of cooling channels.

In-line-T cooling channel had the highest temperature variance (refer to Figure 10) which makes the cooling channels design less desirable while spiral cooling channel appears to a better option (refer to Figure 11). The red region in Figure 10 indicated that the temperature at that particular location is high in comparison with the entire part. This was because of the improper cooling however that was not the case in Figure 11 for spiral cooling channel as the temperature distribution was even. Cooling channels in in-line-T did not cover the entire region of the part but in spiral cooling channel, the cooling channels were distributed well among each other.

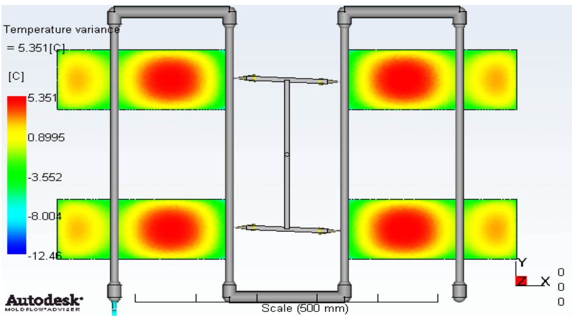

Figure 10: Temperature variance of in-line-T cooling channel

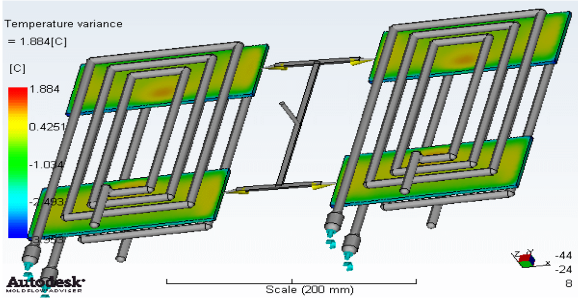

Figure 11: Temperature variance of spiral cooling channel

The time taken to reach ejection temperature was by taking into account every region on the surface whereby the part was safe to be ejected without any manufacturing defects. The lesser the time it took to be ejected out, the higher the productivity level would be. However, the quality should not be compromised. This was where the efficiency of cooling channels played an important role. If they were properly designed, more heat extraction was allowed, aiding in a balanced cooling of the part in overall. With the integration of the cooling channels, the time it took to reach the ejection temperature was decreased (refer to Table 5). Spiral cooling channel proved to be effective as it was from 20.59 s (refer to Table 4) to 15.71 s (refer to Table 5). A balanced cooling was needed to ensure even heat removal rate. This was because, if the rate of removal varied from one region to another, it affected the cooling rate as well. When the cooling rate differed, the melt solidified at different rate causing manufacturing defects such as shrinkages or warpage.

Cycle time was a dominant parameter when it came to determining the efficiency of a manufacturing process in terms of productivity and cost. Cooling time took more than 2/3 of the whole cycle time which means an effective cooling design would be able to reduce the time taken for the whole process to happen. By reducing cooling time, cycle time was reduced as cooling time served as a predominant factor in the cycle time. In order to reduce cooling time, the efficiency of cooling channels design should be improved. Among the four designs, the least cycle time was the one integrated with spiral cooling channel which was 18.68 s and in-line-T cooling channel has the highest cycle time which was 23.77 s.

Figure 12 shows the significance of cooling channels in order to reduce the cooling time as well as the cycle time. The part experimented without the integration of cooling channels and had undergone natural cooling showed an estimated cooling and packing time of 18.89 s with a cycle time of 26 s. However, that was not the case when the cooling channels were employed. Cooling channels designs helped in reducing the cooling time. The maximum cooling time was obtained from in-line-T cooling channel of 6.52 s and the minimum was from spiral cooling channel which was 1.43 s.

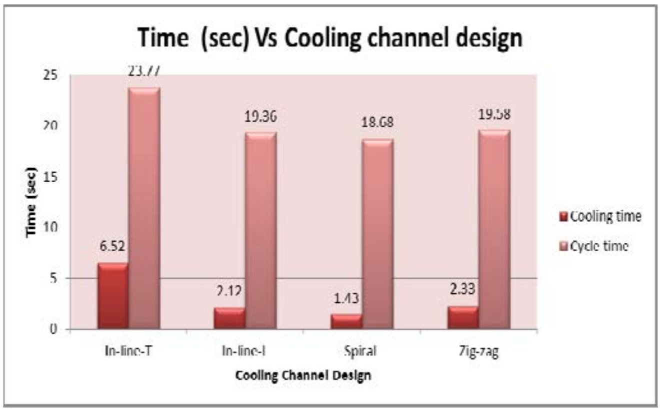

Figure 12: Cooling time (sec) vs cooling channel designs

4. Conclusions

In plastic injection moulding, cooling time is the dominant factor in reducing the cycle time. By decreasing the cooling time, cycle time can be reduced. The results of the fill and cool analyses on all the four designs indicated that spiral cooling channel needed the least cooling time in comparison with the rest. Integration of cooling channels had contributed in decreasing the cycle time by reducing the cooling time needed before the part reaches the ejection temperature. It can be observed that the coolant temperature at the outlet of spiral cooling channel showed a higher value compared to the rest and this also showed that more amount of heat was transferred out. The pressure drop was higher compared to single cavity because it had more cavities and more pressure was needed for the filling to occur. Among the four designs, spiral cooling channel was the better option compared to in-line-T cooling channel.

References

[1] E.Sachs, S.Allen, M.Cima, E.Wylonis and H.Guo: Production of Injection Molding Tooling with Conformal Cooling Channels using the 3D Printing Process, Polymer Engineering and Science pp.448-467,(2000)

[2] X.Xu, E.Sachs, S.Allen and M.Cima: Designing Conformal Cooling Channels for tooling, Proceedings of the Solid Freeform Fabrication Symposium, University of Texas at Austin, pp. 131–146. (1998)

[3] X.Xu, E.Sach and S.Allen:The Design of Conformal Cooling Channels in Injection Moulding Tooling, Polymer Engineering and Science,pp.1269-1272,(2001)

[4] D.E.Dimla, M.Camilotto and F.Miani: Design and Optimization of Conformal Cooling Channels in injection moulding tools, Journal of Material. Processing Technology, pp1294-1300,(2005)

[5] A.B.M Saifullah, S.H. Masood and Igor Sbarski: New Cooling Channel Design for Injection Moulding, Proceedings of the World Congress on Engineering, (2009)

[6] Duleba, Branislac,Ing. (2011). Conformal Cooling for Plastics Injection Moulding. Department of Technologies and Materials, Technical University of Kosice, Letna. 1 (1), pp1-5.

[7] Wong, C.T, Shamsuddin, S, Napsiah, I, Hamouda, A.M.S. (2004). Design and Simulation of Plastic Injection Moulding Process. Pertanika J.Sci & Technol. Supplement. 12 (2), p85-99.

[8] Yadav.A.J, Dravid.S.V, Rajput.V.D (2012). Taguchi Technique in Optimization of Injection Molding Process Parameters for Manufacturing Plastic Parts. Proceedings of the NCNTE, Third Biennial National Conference on Nascent Technologies. 3 (1), p1-8.

[9] Huang.M.C, Tai.C.C. (2001). The Effective Factors in the Warpage Problem of an Injection-moulded Part with a Thin Shell Feature. Journal of Materials Processing Technology. 110 (1), pp1-9.

[10] Hassan, Hamdy. (2008). Effect of Cooling System on the polymer temperature and solidification during Injection Molding. Applied Thermal Engineering. 29 (1), 1786-1791.

[11] Galantucci.L.M, Spina.R. (2003). Evaluation of filling conditions of injection moulding by integrating numerical simulations and experimental tests. Journal of Materials Processing Technology. 141 (2), pp266-275.

[12] Malloy, R.A. (1994). Manufacturing Considerations for Injection Molded Parts. In: Plastic Part Design for Injection Molding. Munich, Vienna, New York: Hanser Publications. Pp99-104.

[13] Singh,Kamar.J. (1987). Design of Mold Cooling System. In: Avraam, I.Isayev, Injection and Compression Molding Fundamentals. New York: Marcel Dekker, Inc. pp597-605.

Author Profile

K. Poornima received Bachelor of Mechanical Engineering (Hons) from Universiti Tenaga Nasional in 2012. She is now pursuing Master of Mechanical Engineering at Universiti Tenaga Nasional. Her research interests are pre-dominantly in the manufacturing field as well as Thermodynamics and Thermo-science (heat transfer, thermo-fluids, energy conversion and HVAC systems).

M. N. M Ansari / Dr. Mohamed N.M. Ansari is currently holding a position as Senior Lecturer in Mechanical Engineering, College of Engineering, Universiti Tenaga Nasional (UNITEN), Malaysia and a Visiting Research Scientist, RMIT University, Australia since August 2010. He received his Bachelor of Engineering degree in Mechanical Engineering from University of Madras, India in the year 1994. He received his P.G.Diploma in Plastics Engineering from CIPET, Ahmedabad (1995). He earned his Masters Degree in Computer Integrated Manufacturing from Anna University (2002). He joined the School of Materials, Polymers, and Mineral Resources Engineering at Universiti Sains Malaysia in August 2004 and completed his PhD degree in Polymer Engineering (2009). He has supervised more than 30 students Projects and published several technical articles in Journal and Conferences. He is also a Reviewer for UPM Journal, Malaysia, Nanotech 2012 & 2013 International Conference, USA, and International Journal of Automation Technology, Japan. He has travelled more than 10 countries worldwide.