Key Engineering Materials
ISSN: 1662-9795, Vol. 926, pp 1821-1831
doi:10.4028/p-q2k0v8

Submitted: 2021-12-07
Revised: 2022-01-28
Accepted: 2022-02-01
Online: 2022-07-22

Comparison of Conventional and Conformal Cooling Channels in the Production of a Commercial Injection-Moulded Component

Mandana Kariminejad[1,a*], Mohammadreza Kadivar[1,b], Marion McAfee[1,c], Gerard McGranaghan[1,d], David Tormey[1,e*]

[1]Centre for Precision Engineering Material and Manufacturing Research (PEM Centre), Institute of Technology Sligo, Ash Lane, Sligo, F91 YW50, Ireland

[a]mandana.kariminejad@mail.itsligo.ie, [b]mohammadreza.kadivar@mail.itsligo.ie, [c]mcafee.marion@itsligo.ie, [d]mcgranaghan.gerard@itsligo.ie, [e]tormey.david @itsligo.ie

*Correspondence: Mandana Kariminejad, David Tormey

Keywords: Injection Moulding, Conformal cooling channels, Conventional Cooling channels, Simulation

Abstract. Cooling channels are critical in injection mould tooling as cooling performance influences component quality, cycle time, and overall process efficiency. Additively Manufactured moulds allow the incorporation of cooling channels conforming to the shape of the cavity and core to improve heat removal. These conformal channels can reduce the cycle time, reduce mould temperature, and enhance the temperature uniformity on the mould's surface, leading to improved quality of the moulded components and reduced wastage in the production. The design of such channels is more challenging than conventional channels; thus, Computer-Aided Engineering (CAE) has a significant role within the design process. In this paper, a novel design for conformal cooling channels for the production of a commercial component from an industrial partner is investigated. This component had issues of high cycle time and a high defect rate due to residual stresses, resulting in component shrinkage. First, the existing conventional drilled cooling channels in the mould were simulated in Autodesk Moldflow Insight to evaluate temperature distribution and cycle time. Based on the temperature distribution, conformal cooling channels were designed in Solidworks, addressing the problem areas. Next, a simulation of fluid flow in the conformal channels was conducted in ANSYS-Fluent to ensure equal flow distribution in the entire circuit, iteratively arriving at an optimal configuration. Finally, the results of the new conformal channels, including mould temperature and cycle time, were compared with conventional cooling channels in simulation. The results showed a significant reduction in cycle time and improvement in the temperature distribution, thereby minimising residual stresses and shrinkage.

Introduction

Injection moulding (IM) is a low-cost and reliable manufacturing process for high volume production of components with numerous shapes and configurations [1]. The cooling performance significantly influences the production rate by contributing approximately 50 to 70 % of the cycle time of the IM process [2]. High-performance mould cooling requires cooling channels close to, and following the pattern of the mould surface to improve the cooling efficiency [3] by reducing the cooling time, improving the temperature uniformity on the mould's surface [4,5], and enhancing the quality of the moulded part [6,7]. These types of channels are known as Conformal Cooling Channels (CCC), and their fabrication using conventional manufacturing methods is challenging, and mostly impossible. The advances in modern manufacturing technologies such as Additive Manufacturing (AM) have facilitated the fabrication of CCC within IM tools [8]. The design and fabrication of additive manufactured mould with internal CCC has received considerable industrial and research attention [9,10].

In injection mould industries, mould designers adopt a series of general rules to design the appropriate cooling channels within moulds, although applying these rules to complex moulds is not straightforward. The modern mould design follows a standard procedure to develop an optimised

CCC within moulds [9], utilising a computational approach and Computer-Aided Engineering (CAE). Implementing computer simulations instead of costly trial-and-error procedure can significantly reduce development time and cost [9,10]. The computational analysis provides useful data on cooling-related issues such as residual stress, shrinkage, warpage [6,7]. A large body of research has been devoted to the design and fabrication of cooling channels, found in the recent review studies of Feng et al. [8] and Kanbur et al. [9].

Different computational software is commercially available for the analysis of IM tools, including ABAQUS and ANSYS for thermal analysis [11]. Most of the computational studies have been devoted to the thermal or mechanical analysis of IM tools, while fluid flow analysis in cooling channels has received relatively less attention. The flow dynamic of coolant can influence the heat transfer and pressure drop, especially in CCCs fabricated by AM that have complex structures with rough internal surfaces [12,13]. Computational Fluid Dynamics (CFD) is a valuable tool that allows studying fluid dynamics and heat transfer in CCCs. Arrivabeni [14] reported that CFD can highlight all critical issues of the cooling flow and permit envisaging possible improvements of CCCs. Kadivar et al. [13] performed CFD studies and reported that the channel's curvature and roughness could enhance the heat transfer, improve the mould temperature uniformity, and increase pressure drop. They also stated that CFD might be incorporated with optimisation methods to find an optimal design of CCCs. Zhihong et al. [15] used CFD and reported that the CCC's structure could substantially influence the flow and heat transfer.

CFD tools such as ANSYS can accurately model fluid flow in cooling channels and analyse the thermal effects on the mould. However, simulating the effect of the resulting mould temperature distribution on the component cooling time and resulting properties such as residual stress - and hence shrinkage and warpage - is complex and time-consuming in such an environment. Therefore, some specific CAE software like Moldflow and Moldex3D were developed to simulate all the aspects of the injection moulding process efficiently and with reasonable accuracy. Li et al. [16] used Moldflow software to validate the topology optimisation approach developed for the design of CCCs. The results showed the method could improve the uniformity and efficiency of the cooling process. Marin et al. [17] designed parallel and series patterns for CCCs of the injection-moulded component. The comparison of both patterns with the conventional baffle cooling channels in CAE illustrated the parallel pattern is the most- efficient approach. Lu et al. [18] also compared the effect of conventional baffle channels and CCCs on cooling time and residual stress. First, the performance of two types of channels was analysed in Moldex3D and then the results were verified in the experiment, showing the lower residual stresses and cooling time with CCCs. Kariminejad et al. [19] investigated the most influential process parameters on the cycle time and shrinkage through simulation in Moldflow. They used Taguchi Design of Experiment (DOE) method and recommended an optimal setting for optimization of both shrinkage and cycle time together. Hanid et al. [20] applied Moldflow and optimisation methods to report that CCCs can improve the component quality by reduction of the warpage.

The design process of moulds with conformal CCCs is time-consuming and requires the implementation of long trial-and-error processes or optimisation techniques [9]. Moreover, the structure of CCCs is more complex than straight drilled channels, and it involves curve channels and flow branches that can significantly influence the fluid flow and hence heat transfer in the CCCs. Therefore, implementing the CFD in the design process to study and enhance the flow in the CCC can improve the efficiency of the design process. Utilising CFD in ANSYS-Fluent can reduce the trial-and-error in the design process; it also provides the capacity to consider the effect of the complex flow physics such as secondary flow (due to channel curvature) and additive manufactured roughness, which can significantly influence the heat transfer in channels [13,21]. This paper investigates the design of CCC for a commercial injection mould that is being used in an industrial production line while having underlying issues, such as a high cycle time and reduced quality due to component shrinkage, which result in a high rate of out-of-spec components. A design process was utilised to address the issue with the efficient design of CCCs within the mould. In this process, CFD simulations in ANSYS-Fluent were used to study and optimise the coolant flow in CCCs. Subsequently, the efficiency of the final design of the mould with CCCs was evaluated in Moldflow. The performance of ANSYS-Fluent and Moldflow software were also studied and compared with respect to the accuracy.

Methodology

In this section the design procedure of the CCCs will be described as following steps:

1. Simulation of the current IM tool with conventional cooling channels in Moldflow
2. Design of CCC in Solidworks based on the results of the previous step
3. Flow simulation in ANSYS-Fluent
4. Process simulation and thermal analysis in Moldflow

Steps 2 to 4 may be repeated to arrive at the optimal design

Component Description. This study investigates a component referred to as a 'Clip', with the isometric view illustrated in Fig. 1. The initial process setting for optimisation was provided by an industrial partner, summarised in Table 1. The material of the Clip component is Delrin 500P NC010, and the dimension is 32.36×26.33×11.9 mm. The Clip is currently being manufactured by an IM tool with conventional cooling channels (drilled cooling channels), triggering a high cycle time and issues with component quality frequently not meeting specifications due to shrinkage problems.

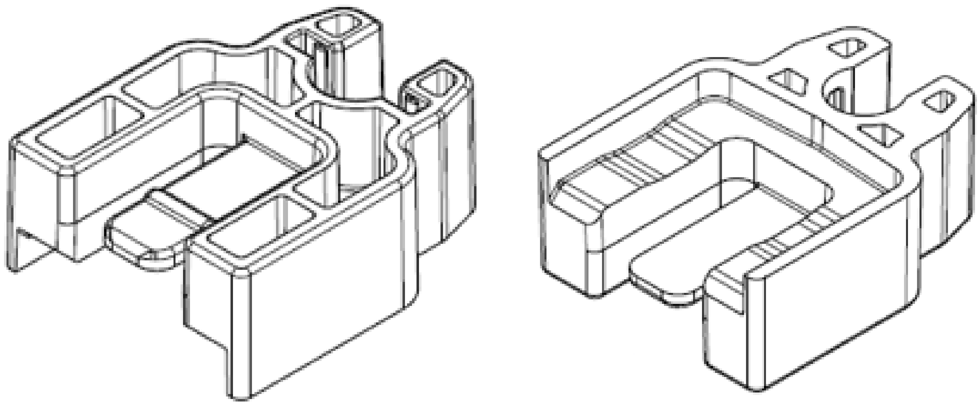

Fig. 1 Isometric view of the Clip injection moulded component

Table 1 The Initial process setting

Melt temperature [°C]	Coolant Flow Rate [lit/min]	Coolant Inlet temperature [°C]	Mould temperature [° C]	Ejection Temperature [° C]
215	7	75	90	135

Simulation of conventional cooling channels. The initial step in the design of CCC is to investigate the critical locations and hotspots within the component based on the conventional cooling channels. For this purpose, a CAD model of the mould with drilled cooling channels was created in Solidworks. Subsequently, the channels, moving mould, fixed mould, and the Clip component were imported to Moldflow for the simulation.

In Moldflow, a 3D mesh (shown in Fig. 2) was generated to capture every detail of the channels, moulds and component (Clip). Two inlets and outlets were considered for the cooling channels, and the coolant used was water. The injection location and hot runner system were also designed in Moldflow based on the specifications of the production tool.

Design of CCCs in Solidworks. The design of CCCs within the moulds was carried out in Solidworks based on the simulation results and addressing the requirements related to manufacturability, heat transfer, coolant fluid flow consideration, and mechanical properties.

Concerning manufacturability, the CCC design must meet the requirements of AM, including build direction, channel size and constraints on overhanging features. As a side effect of the AM process,

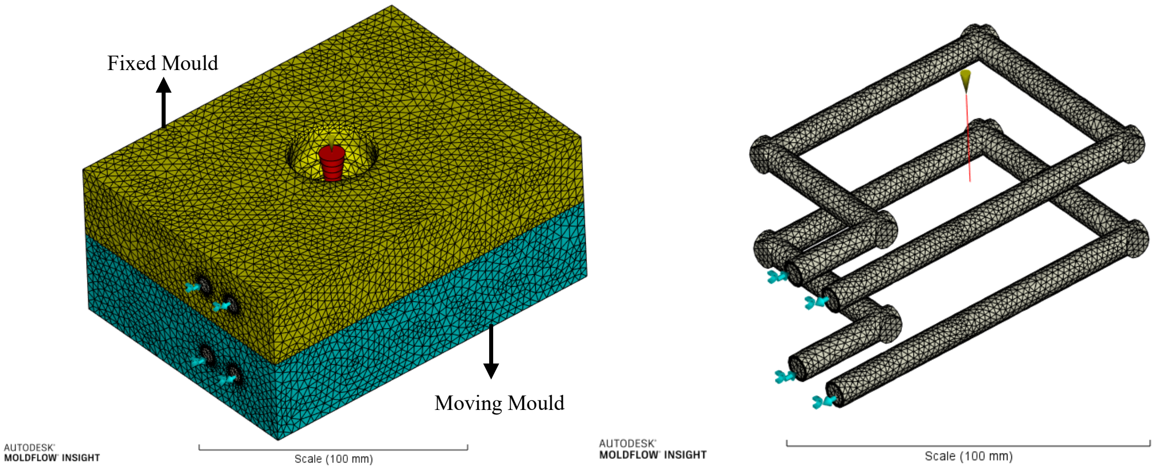

Fig. 2 The 3D meshed of moulds and conventional cooling channels

the roughness generated on the channel's internal surfaces reduces the channel's size to a certain extent [12], which depends on the AM method and process parameters. The manufacturing of circular channels with overhanging faces fails for channels with a diameter larger than certain values (depending on the AM method and process) due to the lack of support for overhanging surfaces. Another limitation is that the maximum manufacturable diameter size varies depending on the AM process and technology. A diameter of 5 mm was selected for this study based on the build constraints of the Direct Metal Laser Sintering (DMLS) system that was used. All object's downward faces (overhangs) must be supported by any supporting structures, except for faces with a build angle of 45° or larger that are self-supported and ingratiated [22]. Build direction defines the overhanging faces; therefore, it must be defined before the design of the CCCs.

The design of flow passageways was performed by following the contour of the cavity's surfaces. A distance of 2D (where D is channel diameter) was considered between the channel's centre and the surface of the cavity. This distance was reduced to D where the space limit did not allow using larger distances. Further reduction of this distance could lead to decreased mechanical strength of the mound. A distance of 3D was applied between adjacent channels. The channels were designed to give good spatial coverage over the cavity surface, with particular concentration in the regions identified as hotspots from the conventional cooling simulation. The number of turns and channels curvature was minimised to avoid large pressure drops or dead flow in the system. Fig. 3 shows the designed CCC within the mould.

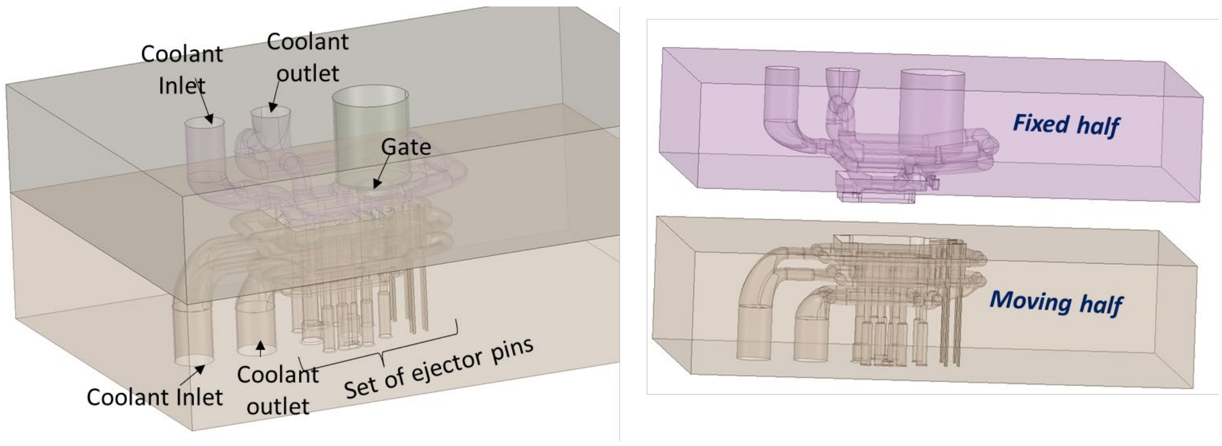

Fig. 3 The final design of CCC within moulds in Solidworks

Flow Simulation in ANSYS-Fluent. Before analysing the mould's cooling performance, the fluid dynamics in the designed CCC was evaluated using CFD in ANSYS-Fluent. In this step, the geometry of the CCC without the mould solid body is included in the simulation, which saves computational effort compared to simulating the entire mould with the component. CFD provides valuable information about the flow division between different branches, possible dead flow (flow with significant velocity reduction), and pressure drop. As an intermediate step in the design, the CFD results can be used to test for possible problems and to trial modifications before proceeding to complete simulations of the entire mould.

The CFD simulations were performed using Finite Volume method (FVM) to solve the Navier-Stokes equations in ANSYS-Fluent. The turbulent flow was modelled by using the k-ε model [23], which is a recognised numerical model that is typically used to describe turbulent flow characteristics.

Simulation of conformal cooling channels. After designing the conformal cooling channels, the efficiency of the CCCs should be evaluated and compared with conventional cooling channels. For this comparison, the entire mould with CCCs was imported to the Moldflow. A 3D mesh type was selected in the software to capture all the geometry details, as illustrated in Fig. 4. The initial process setting and conditions of the inlets and outlets are the same as conventional channels for comparison purposes, as shown in Table 1.

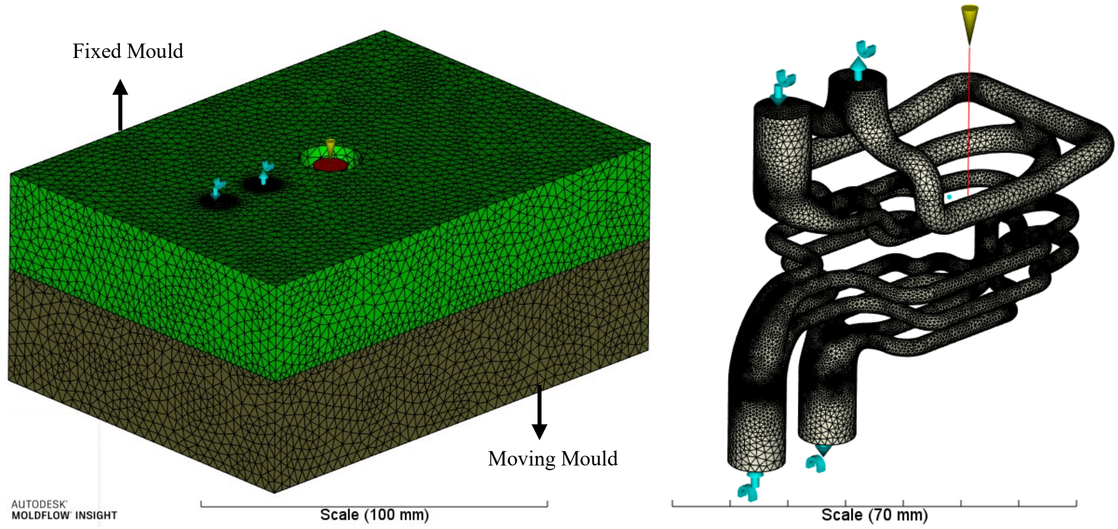

Fig. 4 The 3D meshed of moulds and CCCs

To analyse the cooling stage and temperature distribution, the Cool (FEM) method was selected in Moldflow, and the solver method was flow analysis. This approach takes the initial flow data to calculate the mould temperature distribution and then uses the temperature information to recalculate the flow solution. This process reiterates until it converges to a solution. The flow solver also considers the excessive shear heating of flow and the difference in wall thickness where the thin regions cool quickly.

Results & Discussion

Design of conformal cooling channels. After the simulation of the conventional (drilled) cooling channels in Moldflow, hotspots and critical locations of the Clip were identified, as shown in Fig. 5. The red regions are hotspots that create non-uniform temperature distribution in the Clip component, leading to residual stresses and consequently shrinkage. Hence based on these problem locations, the design of the CCCs was carried out in Solidworks, as mentioned in the previous section.

CFD simulations in ANSYS-Fluent were performed to evaluate flow dynamics and distribution in the CCCs. Based on the CFD data, the design of the CCCs was iterated three times until an acceptable flow distribution and velocity were achieved in all branches with a satisfactory pressure drop. The target was to obtain uniform flow distribution between CCC branches and reduce the pressure drop while considering the design limitations mentioned in the previous section. The CFD results of the final design of CCC are reported in Fig. 6 and 7.

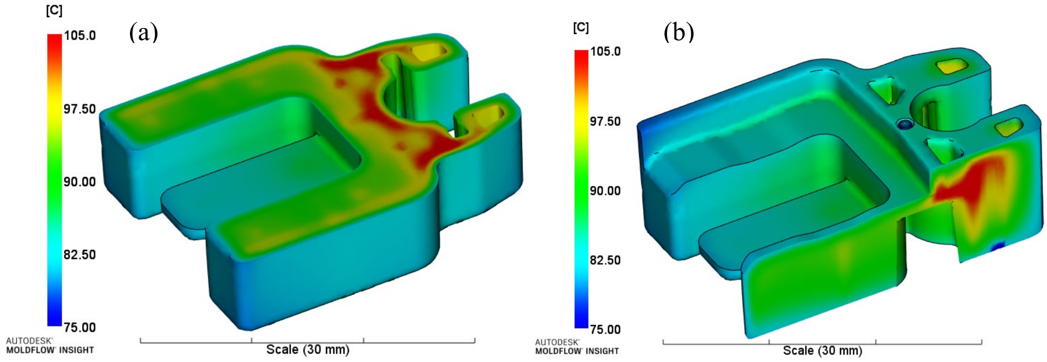

Fig. 5 Temperature profile in Clip with conventional cooling channels (a) xy cross-section (b) zx cross-section

Fig. 6 illustrates the flow path-lines coloured by velocity in the CCC. For both mould halves, relatively uniform flow distributions (with an average of 3 m/s) are observed between the lines branched from the inlet and those from the outlet. Moreover, in the regions with hot spots (close to the injection gate), the flow velocity is higher than the other regions, resulting in greater heat removal which contributes to the enhancement of temperature uniformity.

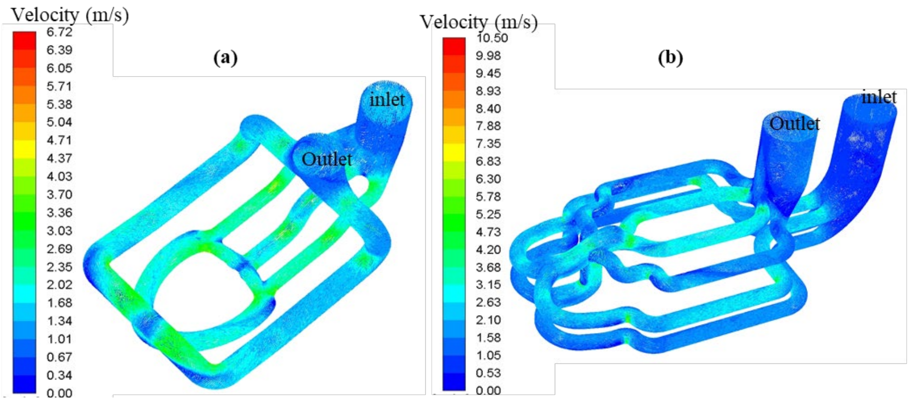

Fig. 6 CFD results of flow path-lines coloured by velocity in the CCC of (a) the fixed half and (b) the moving half of the mould

The pressure drop is another important factor in the design of the CCC. Significant local reduction in pressure can interrupt the uniformity of the flow distribution or even prevent flow movement in some flow branches. Increasing the overall pressure drop between inlet and outlet increases the required pumping power to feed the coolant through the system, contributing to higher cost and energy consumption. Fig. 7 shows the pressure distribution in the CCC. A relatively uniform pressure distribution between all lines branching from the inlet and outlet channels is observed for both mould halves, with no significant local pressure drop noted. The overall pressure drop in the CCC of the fixed and moving halves are 21.7 kPa and 35.2 kPa, respectively. The negative pressure in Figure 7 corresponds to the inner wall of the bends where the flow separates from the wall due to the channel curvature.

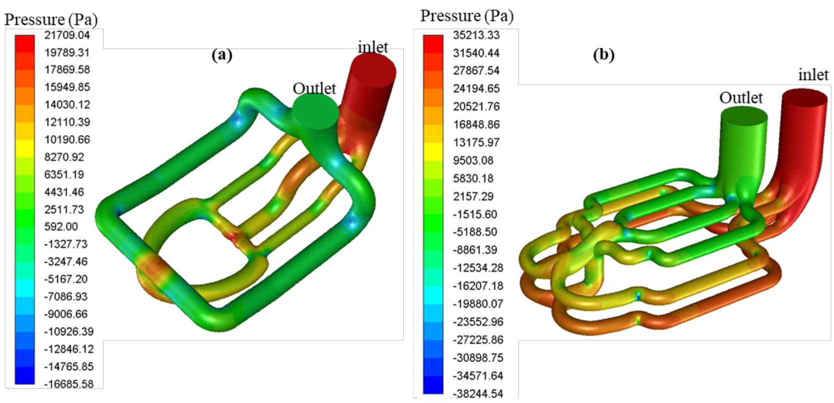

Fig. 7 CFD results of pressure distribution in the CCC of (a) the fixed half and (b) the moving half of the mould

After assessing the flow behaviour in the CCCs by CFD, the design was evaluated and compared to the conventional cooling channels in Moldflow. The effect of the CCCs on the temperature distribution in the fixed and moving halves demonstrates that the conformal cooling channels cause the temperature to be more uniform in both halves. Fig. 8 shows the temperature of the hotspots at ejection time in the moving half of the mould declined from 99 °C to almost 85 °C. A similar result was achieved for the fixed half with a reduction in the hotspots' temperature from 91 °C to 80 °C.

The thermal analysis results of the component (Clip) with conventional cooling (Fig. 5) were then compared to the results obtained for the CCCs. In Fig. 5, the temperature of the part with drilled channels varied from 75°C to 105°C. So, the temperature difference in part (ΔT) is around 30°C and there are a lot of hotspot regions with 105°C. While in Fig. 9 (the part with designed CCCs), the temperature difference in the part decreased and varied from 75 °C to almost 90 °C (ΔT= 15 °C) for most of the regions. Also, the average part temperature decreased from 86.2°C to 80.5°C with CCCs. Therefore, most of the hotspots, shown in red coloured in Fig. 5, disappeared and replaced with the green coloured with the temperature of about 90°C. Based on this thermal analysis, Fig. 8 and 9 show a significant improvement in the uniformity of the mould surface temperature and the component, using CCCs compared to straight drilled channels.

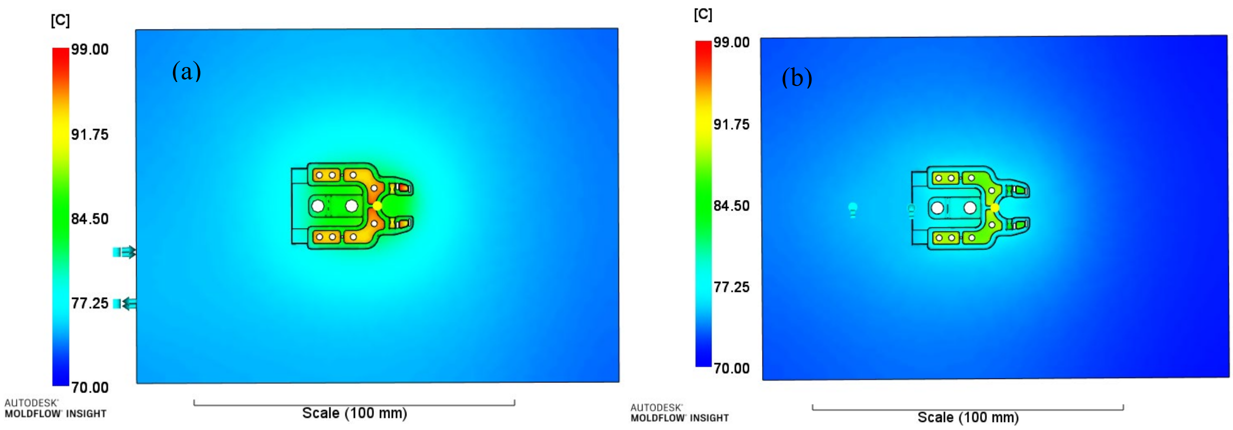

Fig. 8 Temperature distribution at ejection time in (a) moving half with conventional cooling and (b) with CCCs

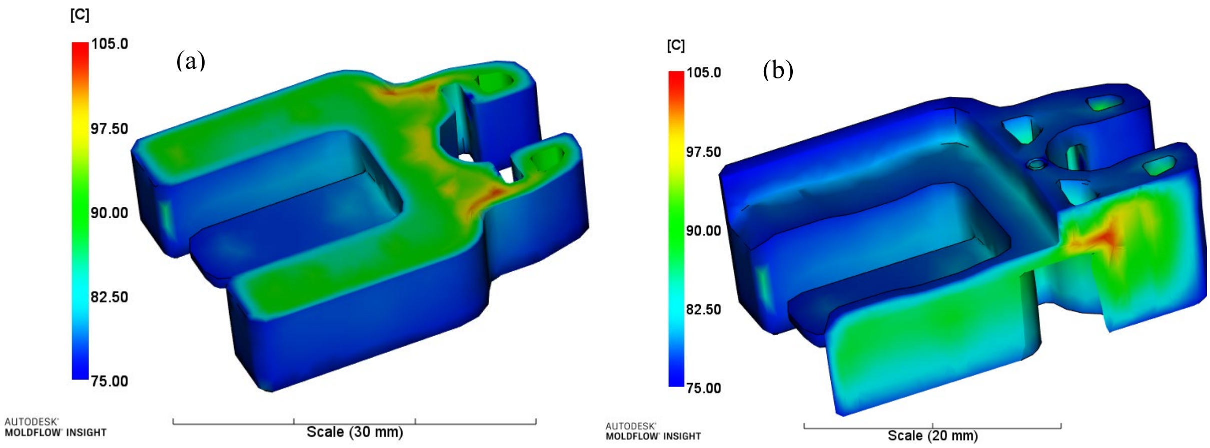

Fig. 9 Temperature profile in Clip with CCCs at ejection time (a) xy cross-section (b) zx cross-section

Another critical factor in the production of this component is cycle time. Fig. 10 compared time to reach ejection temperature (see Table 1) for both channel types, proving a reduction of ejection time from 19.66s to 14.42s. The cycle time of the Clip with conventional cooling channels is about 35s; however, it decreased significantly with CCCs to about 20s. The comparison of drilled channels and CCCs at ejection time is also summarized in Table 2.

Comparing Fluent and Moldflow software. Fig. 11 illustrates the velocity field in the cross-section of the CCC lines in both mould halves resulting from Moldflow and ANSYS-Fluent (CFD). The fluid velocity in the feed lines (designed in the core) is higher than the return lines. Since higher

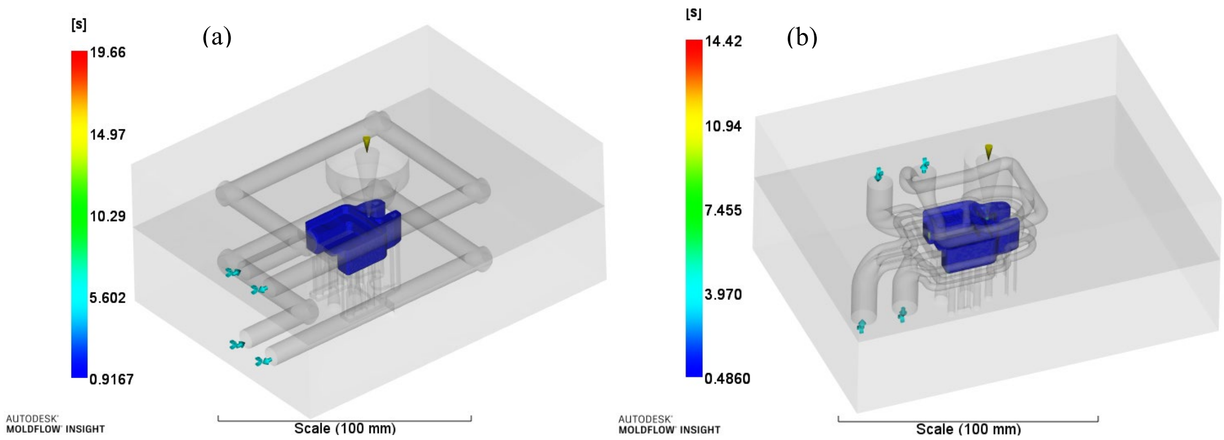

Fig. 10 Time to reach ejection temperature for (a) conventional channels, (b) CCCs

Table 2 Comparison of the efficiency of the conventional channels and CCCS at ejection time

Cooling channel type	Part temperature difference ΔT [°C]	Fixed mould hotspot temperature [°C]	Moving mould hotspot temperature [°C]	Cycle time [s]
Drilled Channels	30	91	99	35
CCCs	15	80	85	20

coolant velocity generally contributes to greater heat removal efficiency, more heat is removed by feed lines than return lines. Therefore, more heat will be removed from the core and the region close to the gate (which has greater heat generation), leading to temperature reduction and enhanced temperature uniformity, as observed in Fig. 8. However, the heat distribution over the entire cavity surface of the moving half is more uniform; therefore ensuring a relatively uniform flow distribution between all lines, as observed in Fig. 7 (b).

Fig. 11 also compares the results of the velocity fields from Moldflow and ANSYS-Fluent. Both profiles have similar velocity characteristics for the specific cross-section position shown. However, a few differences are observed between the profiles produced from each modelling tool, which this likely originates from different computational and numerical analysis methods. ANSYS-Fluent encompasses more fundamental fluid physics such as secondary flow [24], flow mixing due to roughness [13], and flow separation [25] in the solution calculation than that of Moldflow; therefore, it has more capacity to simulate more complex flow in CCCs. However, the flow simulation in Moldflow provides the capacity to simulate the complete IM process by including more physics phenomena specific to polymer flow, stress analysis, etc. In general, the comparative results between the two modelling tools showed a good agreement between the velocity profile of both analyses, which is very important for simulating and designing industrial mould tools.

Conclusion

This work presents a novel approach, incorporating Moldflow and ANSYS-Fluent modelling tools for designing and integrating conformal cooling channels (CCCs) into a commercial injection mould tool. This component, known as Clip, had existing design issues relating to excessive cycle time and poor component quality. First, the current straight (drilled) cooling channels were simulated in the Moldflow for thermal analysis and identification of component hotspots, after which the initial design of the CCCs was completed. The CCCs were then analysed in ANSYS-Fluent to evaluate fluid flow analysis (pressure and velocity). Finally, the CCCs were simulated in Moldflow to assess their thermal analysis and efficiency when compared against that of conventional cooling channels. The last three steps were repeated two times to find the optimum design.

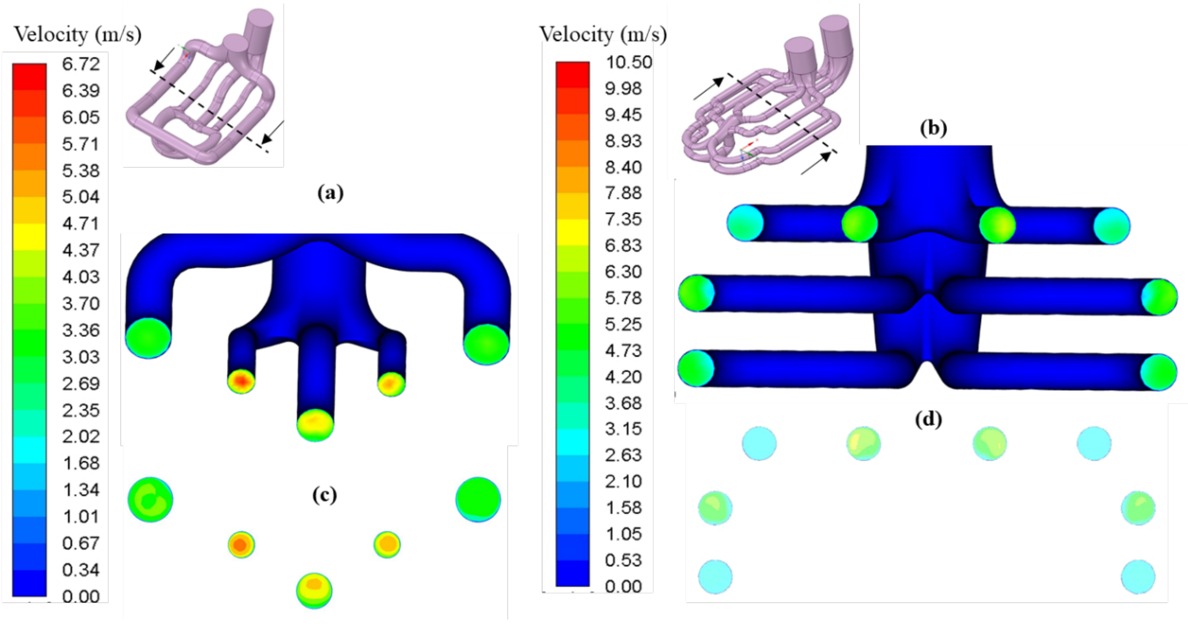

Fig. 11 Velocity field in the cross-section of CCCs from Moldflow for the (a) fixed half and (b) moving half of the mould compared with the velocity field from ANSYS-Fluent CFD for (c) the fixed half and (d) the moving half

The designed conformal cooling channels demonstrated increased fluid flow efficiency and even temperature distribution throughout the channel network. The temperature difference (ΔT) in the component halved from 30°C to 15°C (see Table 2) and the CCCs resulted in better uniformity with respect to temperature distribution in both the fixed and moving mould halves. This can reduce or eliminate the residual stresses resulting from the uneven temperature profile within the conventionally cooled industrial mould tool, thereby decreasing instances of component shrinkage and warpage.

Further, the new cooling channel design significantly decreased overall cycle time from 35s to 20s (see Table 2), which will lead to an increase in the efficiency of the process by facilitating a higher throughput rate.

Using both ANSYS and Moldflow software together is proposed as a novel way to improve the design of CCCs. The ANSYS-Fluent provides accurate and precise information about the cooling fluid flow and then Moldflow simulates the whole process to identify the efficiency of CCCs.
Finally, the comparison of coolant flow velocity predicted by each model showed a good agreement between them. Further research will involve validation of the design of CCCs in the actual injection moulding process using in-mould sensors.

Acknowledgements

This research is supported by an IT Sligo Bursary and also by a research grant from Science Foundation Ireland (SFI) under Grant Number 16/RC/3872 and is co-funded under the European Regional Development Fund and by I-Form industry partners.

References

[1] Lin, J. C., 2002, "Optimum Cooling System Design of a Free-Form Injection Mold Using an Abductive Network," J. Mater. Process. Technol., 120(1–3), pp. 226–236.

[2] Kuo, C. C., and Jiang, Z. F., 2019, "Numerical and Experimental Investigations of a Conformally Cooled Maraging Steel Injection Molding Tool Fabricated by Direct Metal Printing," Int. J. Adv. Manuf. Technol., 104(9–12), pp. 4169–4181.

[3] Papadakis, L., Avraam, S., Photiou, D., Masurtschak, S., and Pereira Falcón, J. C., 2020, "Use of a Holistic Design and Manufacturing Approach to Implement Optimized Additively Manufactured Mould Inserts for the Production of Injection-Moulded Thermoplastics," J. Manuf. Mater. Process., 4(4), p. 100.

[4] Soshi, M., Ring, J., Young, C., Oda, Y., and Mori, M., 2017, "Innovative Grid Molding and Cooling Using an Additive and Subtractive Hybrid CNC Machine Tool," CIRP Ann. - Manuf. Technol., 66(1), pp. 401–404.

[5] Cortina, M., Arrizubieta, J., Calleja, A., Ukar, E., and Alberdi, A., 2018, "Case Study to Illustrate the Potential of Conformal Cooling Channels for Hot Stamping Dies Manufactured Using Hybrid Process of Laser Metal Deposition (LMD) and Milling," Metals (Basel)., 8(2), p. 102.

[6] Brooks, H., and Brigden, K., 2016, "Design of Conformal Cooling Layers with Self-Supporting Lattices for Additively Manufactured Tooling," Addit. Manuf., 11, pp. 16–22.

[7] Venkatesh, G., Ravi Kumar, Y., and Raghavendra, G., 2017, "Comparison of Straight Line to Conformal Cooling Channel in Injection Molding," *Materials Today: Proceedings*, Elsevier Ltd, pp. 1167–1173.

[8] Feng, S., Kamat, A. M., and Pei, Y., 2021, "Design and Fabrication of Conformal Cooling Channels in Molds: Review and Progress Updates," Int. J. Heat Mass Transf., 171, p. 121082.

[9] Kanbur, B. B., Suping, S., and Duan, F., 2020, "Design and Optimization of Conformal Cooling Channels for Injection Molding: A Review," Int. J. Adv. Manuf. Technol., 106(7–8), pp. 3253–3271.

[10] Wei, Z., Wu, J., Shi, N., and Li, L., 2020, "Review of Conformal Cooling System Design and Additive Manufacturing for Injection Molds," Math. Biosci. Eng., 17(5), pp. 5414–5431.

[11] Shinde, M. S., and Ashtankar, K. M., 2017, "Additive Manufacturing–Assisted Conformal Cooling Channels in Mold Manufacturing Processes," Adv. Mech. Eng., 9(5), p. 168781401769976.

[12] Liu, C., Cai, Z., Dai, Y., Huang, N., Xu, F., and Lao, C., 2018, "Experimental Comparison of the Flow Rate and Cooling Performance of Internal Cooling Channels Fabricated via Selective Laser Melting and Conventional Drilling Process," Int. J. Adv. Manuf. Technol., 96(5–8), pp. 2757–2767.

[13] Kadivar, M., McGranaghan, G., and Tormey, D., 2021, "Effect of Surface Roughness on the Performance of Additive Manufactured Conformal Cooling Channels for Injection Moulds," *37th International Manufacturing Conference (IMC37)*.

[14] Arrivabeni, E. B., Tomasoni, D., Giorleo, L., and Barbato, M. C., 2021, "A Methodology for Mould Conformal Cooling Channels Optimization Exploiting 3D Printing," ESAFORM 2021.

[15] YAN, Z., QIAN, Y., HUANG, W., ZHOU, X., and GONG, X., 2018, "Research on Heat Transfer Enhancement of Variable Cross Sectional Conformal Cooling of Injection Mold Based on Fluent," J. Mech. Eng. Res., 10(2), pp. 7–20.

[16] Li, Z., Wang, X., Gu, J., Ruan, S., Shen, C., Lyu, Y., and Zhao, Y., 2018, "Topology Optimization for the Design of Conformal Cooling System in Thin-Wall Injection Molding Based on BEM," Int. J. Adv. Manuf. Technol., 94(1–4), pp. 1041–1059.

[17] Marin, F., de Miranda, J. R., and de Souza, A. F., 2018, "Study of the Design of Cooling Channels for Polymers Injection Molds," Polym. Eng. Sci., 58(4), pp. 552–559.

[18] Lu, C. T., Chen, G. S., and Tseng, S. C., 2018, "Effectiveness of Conformal Cooling for a V-Shaped Plate and Its Influence on Warpage," *Proceedings of the 2017 IEEE International Conference on Information, Communication and Engineering: Information and Innovation for Modern Technology, ICICE 2017*, Institute of Electrical and Electronics Engineers Inc., pp. 79–82.

[19] M Kariminejad, D Tormey, S Huq, J Morrison, M. M., 2021, "Optimization of a Commercial Injection-Moulded Component by Using DOE and Simulation," *37th International Manufacturing Conference*.

[20] Mohd Hanid, M. H., Abd Rahim, S. Z., Gondro, J., Sharif, S., Al Bakri Abdullah, M. M., Mohd Zain, A., El-Hadj Abdellah, A., Mat Saad, M. N., Wysłocki, J. J., and Nabiałek, M., 2021, "Warpage Optimisation on the Moulded Part with Straight Drilled and Conformal Cooling Channels Using Response Surface Methodology (RSM), Glowworm Swarm Optimisation (GSO) and Genetic Algorithm (GA) Optimisation Approaches," Materials (Basel)., 14(6).

[21] Kadivar, M., Tormey, D., and McGranaghan, G., 2021, "A Review on Turbulent Flow over Rough Surfaces: Fundamentals and Theories," Int. J. Thermofluids, 10, p. 100077.

[22] Gaynor, A. T., and Guest, J. K., 2016, "Topology Optimization Considering Overhang Constraints: Eliminating Sacrificial Support Material in Additive Manufacturing through Design," Struct. Multidiscip. Optim., 54(5), pp. 1157–1172.

[23] Doshmanziari, F. I., Kadivar, M. R., Yaghoubi, M., Jalali-Vahid, D., and Arvinfar, M. A., 2017, "Experimental and Numerical Study of Turbulent Fluid Flow and Heat Transfer of Al 2 O 3 /Water Nanofluid in a Spiral-Coil Tube," Heat Transf. Eng., 38(6), pp. 611–626.

[24] Doshmanziari, F. I., Kadivar, M. R., Yaghoubi, M., Jalali-Vahid, D., and Arvinfar, M. A., 2017, "Experimental and Numerical Study of Turbulent Fluid Flow and Heat Transfer of Al2O3/Water Nanofluid in a Spiral-Coil Tube," Heat Transf. Eng., 38(6), pp. 611–626.

[25] Ramamurthy, A. S., Han, S. S., and Biron, P. M., 2013, "Three-Dimensional Simulation Parameters for 90° Open Channel Bend Flows," J. Comput. Civ. Eng., 27(3), pp. 282–291.