4-18-2022 1:30 PM

Cooling Systems Analysis for Plastic Mold Injection Tools

Veronica Flores Quijada, *The University of Western Ontario*

Supervisor: Zhang, Chao, *The University of Western Ontario*

A thesis submitted in partial fulfillment of the requirements for the Master of Engineering Science degree in Mechanical and Materials Engineering

Abstract

The Plastic Injection Mold (PIM) industry has been searching for new technologies that improve the manufacturing of parts by reducing the production time and cost as well as increasing the quality of the product. The cooling systems in the PIM are designed initially to be straight-drilled into the mold, but this manufacturing process has traditionally not been very effective, since for molded parts with complex geometries, the cooling channels are not able to reach certain areas. This limitation has led the industry to develop conformal cooling channels that use the additive manufacturing technology, which allows the cooling channels to conform the part's working surface. So, the production cycle time can be reduced and the temperature distribution in the molded part is more uniform to provide better quality in the product. That said, the cost of using conformal cooling in the manufacturing process has been demonstrated to be higher than the cost of conventional cooling channels Therefore, this research aims to investigate the advantages of conformal cooling and the degree to which it improves product quality and manufacturing cycle time.

To be able to carried out these studies, CFD simulations using NX-FLOEFD by Siemens are performed to investigate the possibilities of improve conventional cooling channels, and then to compare them with a designed conformal cooling channel. The parametric studies are performed for different configurations and different operating points to improve these systems. A conformal cooling system is designed in this study for the same product part following design recommendations from previous studies. The study indicates that the conformal cooling channels proven to have a better cooling performance providing a higher quality product.

Keywords: Conformal cooling, Conventional cooling simulations, CFD, Plastic Injection Mold, manufacturing, heat transfer.

Lay Summary

Plastic Injection Molding tools have been developed in the industry for the manufacture of plastic parts and pieces used in the diverse fields. The process followed to create the product starts with the injection of the melted plastic into the tool followed by the solidification of the part, due to the cooling system inside of the cavity and core of the mold, until the ejection temperature. 80% of the cycle time of this process is consumed by the cooling system, so its improvement will reduce the cycle time and this will reduce the production time and cost. For this reason the design of the cooling systems in these tools is of importance.

These cooling systems developed in the mold industry have been manufactured by the straight-drilled process, which for complex geometries, it has not been the most effective design. This limitation has led the industry to develop conformal cooling channels which use additive manufacturing technology to conform the channels to the part's working surface, providing a reduction of the cooling time and a better quality of the product. The additive manufacturing technology have been demonstrated to be higher than the cost of the straight-drilled process and for this reason it has been totally implemented in the industry. This research aims to investigate the advantages of conformal cooling system over conventional cooling and the improvements of the product quality as well as the manufacturing cycle time. Computational Fluid Dynamic (CFD) using NX-FLOEFD by Simens to obtain the most optimal configuration using conventional cooling channels in the mold, to then compare them with a designed conformal cooling system in the same tool. Parametric studies are also generated for the different configurations and operating points in the search of the most optimal configuration for the purposes of reducing the cycle time while maintaining the expected quality of the product.

The study indicates that the conformal cooling channels proven to have a better cooling performance providing a higher quality product.

Nomenclature

General Fluid Mechanics

κ - Kinetic energy of turbulence

ρ - Density

u - Velocity

h - Thermal enthalpy

D - Cooling channel diameter

S_i - Mass-distributed external force per unit mass

$S_i^{gravity} = \rho * g_i$ - Mass-distributed external force per unit mass due to gravity

g_i - Gravitational acceleration component along the i-th coordinate direction

$t_{injection}$ - Injection time in PIM

$t_{packing}$ - Packing time in PIM

$t_{cooling}$ - Cooling time

$t_{ejection}$ - Ejection time

$T_{cooling}$ - Coolant temperature

T_{melt} - Plastic melting temperature

T_{mold} - Mold temperature

q_{conv} - Heat transfer rate by convection

q_{cond} - Heat transfer rate by conduction

$q_{plastic}$ - heat transfer from the melted polymer

q_i - Diffusive heat flux

Q_H - Heat source per unit volume

τ_{ij} - Viscous shear stress tensor

τ_{ij}^R - Reynolds-stress tensor

Contents

List of Figures

List of Tables

Chapter 1

Introduction

1.1 Fundamentals of Plastic Injection Molding (PIM)

In the manufacturing industry, thermoforming processes are commonly used for mass productions. This process consists of a thermally softened material that is injected into molds and then cooled to solidify it in the form of the expected shape of the product [1]. The materials that can be used in these processes are all the plastic products and some metal products. These thermoforming processes can be classified the injection molding, die casting and hot extrusion, where all of them share a common requirement, the cooling ability. The focus of this study is on the plastic injection molding (PIM) process.

The PIM process follows a sequence of stages, injection, packing, cooling, and ejection of the product. Figure 1.1 [2] shows the diagram of a typical plastic injection molding tool. In the first stage of the process, the plastic granules are melted by the reciprocating screw and then injected into the mold cavity to fill in the leaving space until the packing stage occurs, where the gate freezes. While the mold is packed, the cooling stage occurs, where the molten polymer gets solidify ideally with a uniform temperature through the entire surface until the ejection temperature. The product is ejected once it achieves this temperature and at this point the cycle is completed.

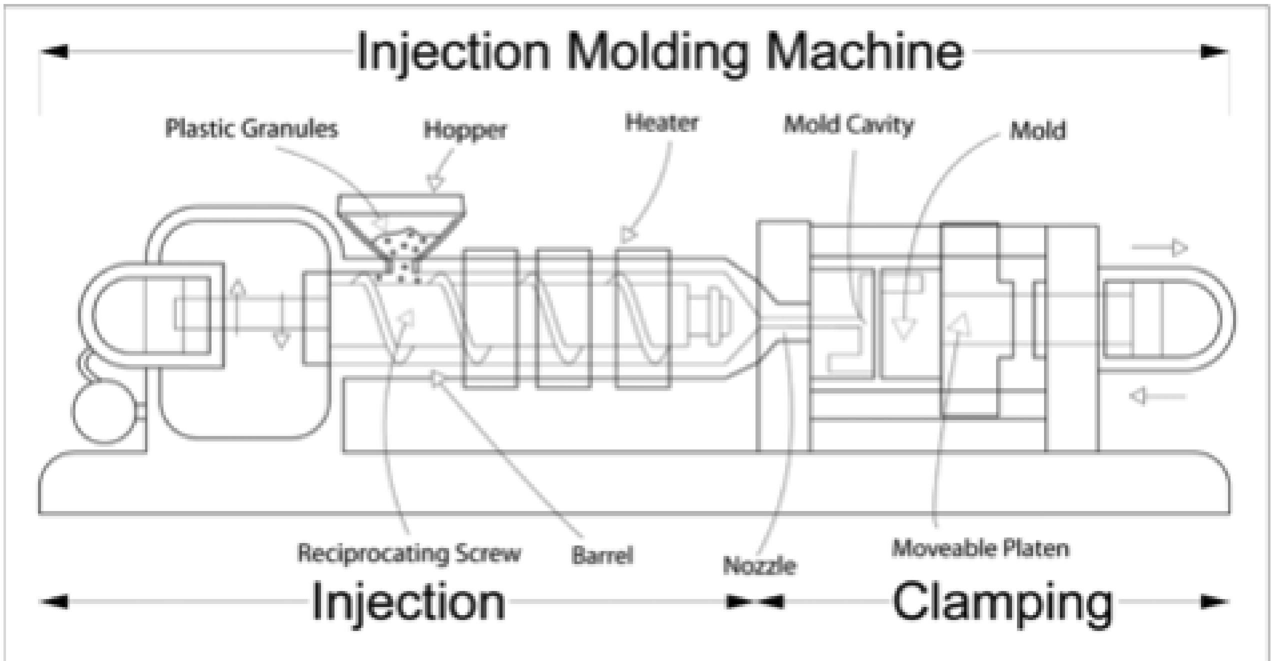


Figure 1.1: Injection mold tool description [2]

In the manufacturing industry, time and cost are strongly linked, the cost-efficiency of the process depends on the time spent in the mold cycle. For this reason, the longer the cycle time of the process is, the higher the cost of creating the product and the less efficient it is. One of the objectives to optimize the process is to reduce the cycle time without compromising the quality of the product. Figure 1.2 illustrates the estimated time that each stage takes for the entire cycle of the mold tool [3]. It is noticed that in the injection molding process, the cooling time takes around 70-80% of the entire time of the cycle [4], therefore, any reduction in the cooling time will have a positive impact on the total production time.

1.2 Overview of the cooling process in PIM process

Due to the impact of the cooling time on the cycle time of the PIM, it is important to improve the design of an optimal cooling system. The main objective is to reduce the cooling cycle time while maintaining a uniform surface temperature of the part, and the core and cavity areas. There are many parameters that affect the performance of cooling system of the mold,

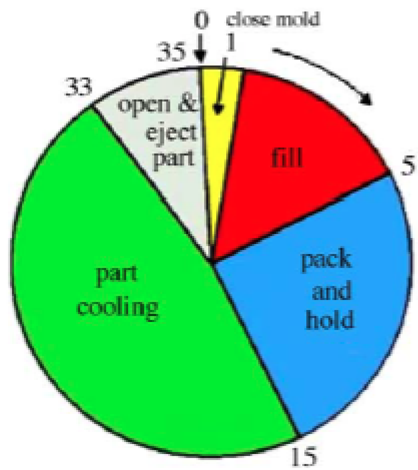


Figure 1.2: Mold injection cycle time [3]

but the most important one, which is responsible for obtaining a uniform temperature in the part and in the mold surfaces, will be the design of cooling channels. Therefore, the location of the cooling channels from the product plays a crucial role in the effectiveness of the cooling process.

The most commonly used channels inside of the mold injection tools have been straight-drilled channels, which can be machined easily. This type of cooling channel is referred as "Conventional Cooling Channel (CCC)"[5], and the location of the channels is inside of the mold and the cavity with a proper distance from the mold surface of the product shape and within the other channels. Figure 1.3 shows the commonly used layout of the cooling channel inside of the mold [5]. For this type of cooling channel, different configurations can be used based on its cooling efficiency. These configurations can be serial cooling channels, in which all the channels in the system are connected end to end to form a single loop with the respective inlet and outlet; and parallel cooling channels, where a set of cooling channels are connected to a supply manifold and a collection manifold, but not between each other directly. Compared with serial channels, parallel channels can be more flexible to adjust the cooling rate and temperature, however, serial channels are simpler and most used in cooling systems [1]. Other methods that are also used in the CCC to improve the uniformity of the cooling process in the product are bubblers and baffles, which can be added to the existed system to avoid areas with non-uniform temperature (hot spots).

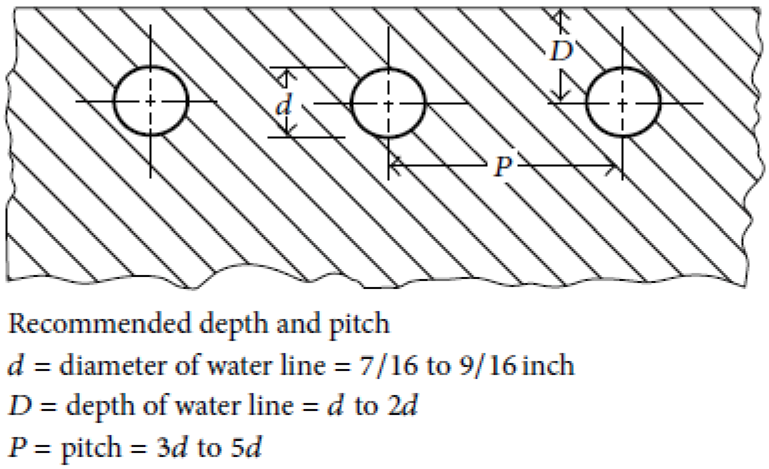


Figure 1.3: Layout of the cooling channel position inside of the mold [5]

A better cooling performance can be achieved by decreasing the distance from the cavity surface to the center of a cooling channel. In the cases where the geometry of the product is complex, CCC might not be the best approach due to the channels being straight drilled into the core and cavity, which is a limitation for this type of design. Thus, “Conformal Cooling Channels” have been developed, where the channels conform to the shape of the part in the core and cavity. This method uses a contour-like channel, which can be as close as possible to the surface of the mold. Thus, it can increase the heat absorption from the melt polymer, and the part can be cooled with a uniform surface temperature in an efficient manner [3]. The manufacturing processes used for the construction of these types of cooling channels come from additive manufacturing technologies such as rapid prototyping and laser powder bed fusion, which creates a 3D objective through laser selectively melting of a defined region of powder particles on the power bed layer by layer according to the 3D model [6].

Most of the recent studies have demonstrated that conformal cooling channels can achieve a better cooling system by reducing the cooling time and increasing the quality of the part due to the temperature uniformity on the surfaces in contact compared with the CCC. Due to the complexity of this type of cooling system, the manufacturing cost of the conformal cooling channel is higher than the one expected with CCC. Therefore, it has not been completely implemented in the industry, but studies have demonstrated the feasibility of its implementation. Figure 1.4 shows the difference between the conventional cooling channel (straight drilled) and conformal cooling channel design inside of a mold [5].

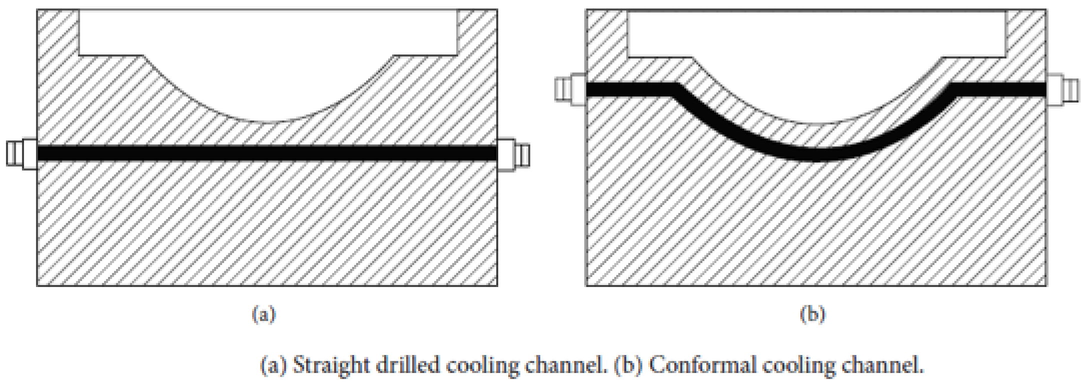


Figure 1.4: Cooling channel layout in a mold (a) Conventional cooling channel (b) Conformal cooling channel [5]

For the optimization of these cooling channels, not just the arrangement of the channels on the cavity and the core of the mold will have an influence on the process, the coolant flow characteristics will also influence the uniformity of the temperature on the surface of the part and the mold areas due to the convection heat transfer process. It is necessary to ensure that the flow is turbulent in each of the circuits since it enables the heat to be transferred more effectively. Studies showed that Reynolds numbers between 8,000 and 10,000 will provide an effective cooling process. If Reynolds number falls below the desired level, hot spots can be created, which compromises the quality of the product. The most commonly used coolant in the industry for this type of application is water and the temperatures of the flow will depend on the conditions of the molten polymer and previous studies. Flow rates and temperatures are important parameters in the design process of an optimal cooling channel configuration.

1.3 Consequences of a deficient cooling system

An effective cooling system in mold injection tools, with an optimal cooling cycle time, uniform temperature on the product and on the mold cavity and core surfaces, can avoid defect on the product and to guarantee the quality of the product. A deficient cooling system can compromise the quality of the product as well as mechanical and thermal properties. The cooling time to be achieved must be optimal since it cannot be too short or too long. Lesser cooling time leads to deformation of the part and longer cooling time will result in machine time waste that will be translated to less profits of the production [7]. Sink marks are one of the defects related to the cooling time, which are localized depression on the part, generated by the cooling time being too short.

The uniformity of the temperature in the surface of the product as well as in the mold core and cavity plays a significant role in reducing defects on the part since it influences the mechanical properties, shrinkage, warpage, and the surface quality. For the mold temperatures in core and cavity, if there is a strong gradient between the two halves, the part may warp and distort its shape [3]. These temperatures should be not too high to burn the component or not too low as the quality of the product can be compromised [7]. The uniformity of the part surface temperature is necessary since any hot spots can also affect the part shape, mechanical and thermal properties of the part at the moment of ejection as well as the quality of the product. On the design of an optimal cooling system in PIM tools, the thermal analysis on the part and the mold during the process as well as the cycle cooling time calculation will allow the designer to predict the behaviour of the cooling system and to avoid such defects. With this information it is possible to make the necessary modifications prior to the manufacturing of the mold and to the execution of the manufacturing of the part, since these defects can be translated to a high economic impact for the industry.

1.4 Use of computational approach to optimize the design of a PIM cooling system

To obtain an optimal design of a PIM cooling system, which can reduce the cycle time and maintain a uniform temperature in the product and in the mold, the effect of certain parameters on the cooling process need to be studied, such as: the cooling channel position, type, deployment connections, length of cooling channel, flow rate and temperature of the coolant used. Previous studies have developed some methodologies in order to obtain the most effective cooling system, but the product shape and geometry changes often, and sometimes it is not possible to adapt previous methodologies to new products. CAE (Computer Aided Engineering) and FEA (Finite Element Analysis) approaches have been used to predict the behaviour of the injection molding process for different designs. So, the designers can assess and optimize different variables during the design phase before the production of the mold.

Some software packages such as MoldFlow and ANSYS have been used in previous researches to conduct simulations on thermal, mechanical, and fluid flow analyses involved in the plastic mold injection tools. These tools have also been used to run parametric studies to find out the most optimal configuration and operating parameters for cooling systems inside of the mold injection tools. The simulations were carried out to be as close to reality as possible, where some of these software would provide more accurate results depending on numerical models and methods used. In most cases these simulations were validated with experimental data that to validate the accuracy of the results. The advantage of the numerical simulation is that it will provide an overview of the most effective design that will generate more profit for the industry with minimal cost, compared with the execution of the manufacturing of the mold for each attempt in the search of an optimal design.

1.4.1 CFD Software for mold cooling analysis in PIM

Considering that there are several software capable of generating simulations to study the performance of the mold injection tools prior to manufacturing of them, there is a strong interest in the search for a software with the same capabilities in the simulation of PIM cooling systems which is suitable for industry applications. NX of Siemens is a software with a wide range of applications used for thermal, mechanical, and fluid flow simulations and analysis that is looking to expand for more industrial applicability. Mold Wizard is the application of NX focused on the design of mold injection tools, using a wide range of materials and libraries based on previous usage of the mold making industry. Also, the NX platform applications are applied for the analysis of the design of mold injection tools and thermal and mechanical properties of the mold to prevent damages on the tool.

In the urge to provide mold designers with the solutions to have all the necessary tools to analyse their designs and obtain the optimal design it is necessary to add the cooling systems analysis to add on the package to make NX suitable for mold designers. FLOEFD is a CFD application in the NX package. In the current work the applicability of FLOEFD in the mold design industry will be investigated and the results will be validated against the experimental data obtained from their clients who run in try-outs for design purposes. A development of conformal cooling channels is also evaluated in the present work. Then, the studies of difference of the cooling systems for plastic mold injection tools will be performed using FLOEFD.

Other software used in the industry for the analysis of this type of applications as MoldFlow, only have the capability to generate CFD simulation of the injection, cooling and ejection stages, since it involves fluid analysis. The platform of this software only generate analysis related to flow without the capability of modify geometry or configuration of the cooling system, where this geometry would need to be modified in another platform to be able to generate analysis in this software. This is not as convenient to the industry since it will need the availability of other software to conduct a proper analysis on the cooling system and to evaluate the diverse scenarios to obtain an optimal design.

1.5 Description of the molds used

For the analysis and validation of the use of FLOEFD in thermal cooling systems analysis on mold injection tools, two molds are used. Both molds are provided the manufacturer customer for the research. These tools are originally designed with all the components illustrated in Figure 1.1. In this study, to the reduce the computational time, the components to be used for the analysis of the cooling system are only the ones involved in the cooling process, that will be described as follow for each of the molds.

1.5.1 Mold A

The plastic part to be manufactured from this mold injection tool is shown in Figure 1.5, where the CAD geometry of the product is shown. The material of the part is PC ASA, an amorphous thermoplastic alloy of polycarbonate (PC) and Acrylonitrile styrene acrylate (ASA) that provides enhanced heat resistance and mechanical properties. The melting temperature of the plastic is 240°C and the ejection temperature of the part is 104°C. This product is manufactured for the automotive industry.

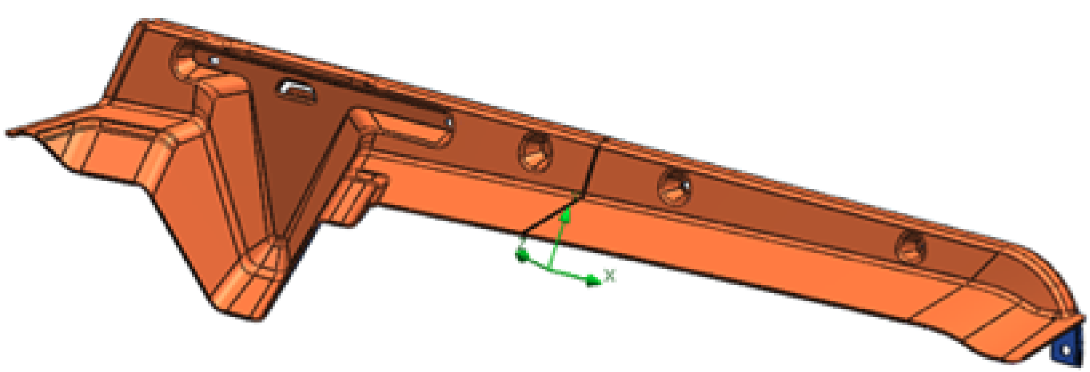


Figure 1.5: Plastic part CAD Mold A

This mold is being used for validation of the software with experimental data that was conducted by the manufacturer.

1.5.2 Mold B

Figure 1.6 shows the CAD geometry of mold B. The material used for this part is polypropylene (PP) and the usage of this part is as a lid for a gallon. The melting temperature is 190°C and the ejection temperature of the part is 102°C. This mold is used for the design of a conformal cooling channel and to be compared with the originally conventional cooling channel that the tool was designed for.

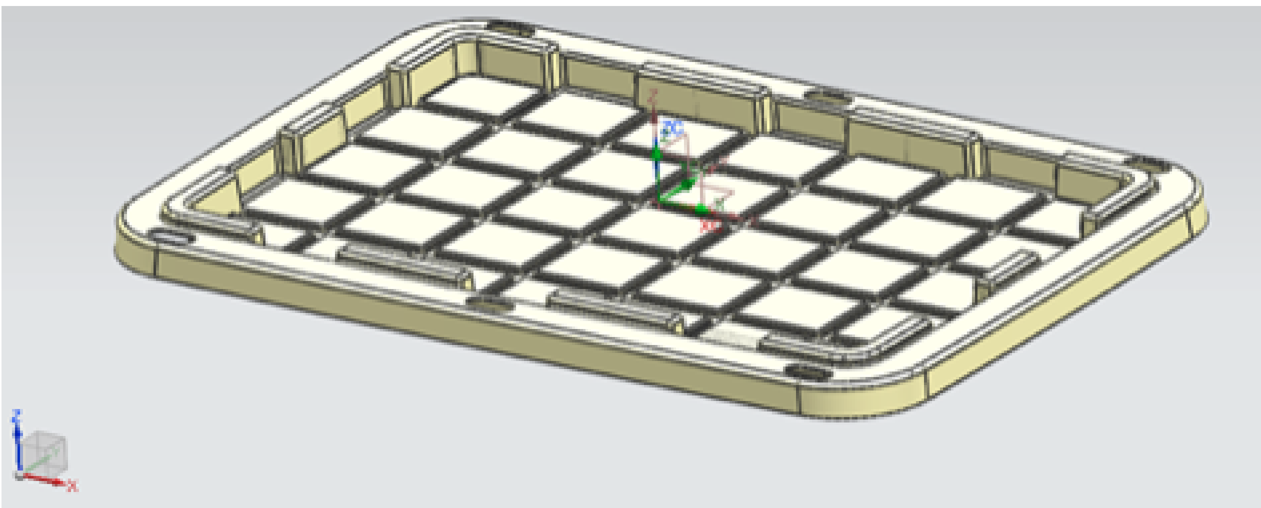


Figure 1.6: Plastic part CAD Mold B

1.6 Objectives of the research

The analysis on the design of plastic injection mold tool has a positive impact on the industry for the manufacturing of parts, specifically on the study of optimal cooling systems, which will improve the manufacturing process. For this reason, it is important to have software that can provide the analysis of these systems accurately to allow the study of the feasibility on the design. In the current research, the first objective is to evaluate the accuracy and the feasibility of the use of the software NX-FLOEFD of Siemens as a tool to analyse the behaviour of cooling systems inside of mold injection tools since NX of Siemens already has the Mold Wizard application for the CAD modelling of injection molds. This is performed using molds A and B of the manufacturer and comparing the results with the experimental data obtained from the operating tools.

Another objective of the current research is to analyze the behaviour of the conventional cooling systems by running parametric studies. These studies are carried out for different operating parameters, the volume flow rate, temperatures, and different channel diameters. The parametric study will allow to recognize the behaviour of the conventional cooling system as well as its limit of improvement.

The novel conformal cooling channels can improve the cooling efficiency of PIM by reducing the cooling time and achieve a more uniform temperature distribution. In the current research, a conformal cooling system is designed for the mold B with a serpentine configuration. The thermal analysis is conducted for the designed conformal cooling channel to evaluate the effectiveness of this type cooling system.

The results obtained for both types of cooling systems, the conventional and conformal cooling systems, are compared for the cooling time and the temperature distribution to evaluate the advantage of the use of conformal cooling channels over the conventional cooling channels.

After evaluating both cooling systems, the last objective of the current work is to analyse the applicability of the software in the industry applications and it convenience. This will be done considering the current software available in the industry and the previous work performed as well as the industry requirements for this purpose.

Chapter 2

Literature Review

2.1 Overview

Considering the high demand on the use of PIM technology and the need to achieve minimum cycle time, in order to reduce the production cost without compromising the quality of the part, many researchers and mold designers have been developing different methods towards these goals. Since 60-80% of the cycle time is on the cooling stage, most previous studies were focused on the development of an optimal cooling system that can be functional, applicable to the industry, and to be able to meet the requirements of the manufacturer.

In this chapter, an analysis of previous investigations is being conducted. The reviews are first conducted for, a one-dimensional method for the heat transfer between the cavity, the part, and the core in order to estimate the cooling time and analyse the temperature distribution through the part and the mold. Then, the reviews are conducted for two types of cooling systems, the straight-drilled cooling channels and the conformal cooling channel, and the different tools used to reduce the cycle time and to keep a uniform temperature through the part and the mold. Another critical point needed to be evaluated is the defects developed in the product the cooling systems design.

Previous design optimization practices and research are also reviewed, such as as different configurations of the cooling channels, their arrangement and location, the most optimal diameter of the channel based on the part thickness as well as its influence on the quality of the final product. Also, the influence of the materials of the mold in the cooling process is considered. Since most of these studies on the optimization of plastic mold injection tools were done using a computational approach, different computational methods and models are reviewed in order to understand their applicability for different cooling systems. Added to this, some other studies used different computational methods for thermal and mechanical analysis to guarantee that the selected cooling design was able to handle the thermal load and the mechanical load.

The selection of the software to be used for computational simulations has also been part of previous studies of cooling systems. In the current study FLOEFD of Siemens NX is used for the CFD analysis of the cooling system in this study, and in this section some theoretical background is being reviewed.

Manufacturing processes of these tools have been a constraint in the development of effective cooling systems. For the straight-drilled channels, the limitations are due to the channels not being able to get closer to the part because of mechanical failures on the mold; at the same time, since the channels are drilled, there cannot be sharp edges since they affect the characteristics of the flow, which affects the effectiveness of the heat transfer rate. For the conformal cooling channels, many studies have been carried out to improve the manufacturability of these types of channels considering that they affect the cost of production of the mold.

2.2 Types of cooling systems

After reviewing the physics of the cooling phenomenon on plastic injection mold, it is noticed that the effectiveness of the cooling process and the possible reduction on the cooling time will depend on the disposition of the cooling channels on the mold. Since both sides of the mold in contact with the product, core and cavity must have cooling circuits in order to provide a uniform temperature distribution during the cooling process at both sides of the product, which will avoid and decease the warpage after the ejection of the part [8]. Based on the manufacturing process for the channels inside of the mold tool, there are two different types of cooling channels: conventional cooling channels, which are straight-drilled holes inside of the mold following a certain circuit that will go along the geometry of the product as much as possible; and conformal cooling channels which conform the shape of the geometry of the part and it is possible be used for geometry cooling channels.

2.2.1 Straight-drilled cooling channels

The conventional cooling channels in plastic mold injection tool is referred to the cooling channels that are straight-drilled machined into the mold tool. This was the original method of manufacturing by drilling machining following as possible the shape of the product. However, using this drilling method drilling could not be possible to have uniform temperature distribution for the parts with complicated shapes. In the search for the uniform temperatures during the cooling processes different configurations of the conventional cooling channels have been studied, as well as accessories added to these channels in order to reduce the cooling time and maintain the uniformity of the temperature.

Parallel cooling channels

These type of cooling channels is straight drilled to the channels with a configuration of a supply main manifold that goes into a collection manifold. Figure 2.1 shows an example of a parallel configuration. In these circuits, the flow rate along various cooling channels may be different, this will depend on the flow resistance of each of the cooling channels. Therefore, if the flow rate changes along the different channels, the heat transfer efficiency from the coolant to the tool and to the product might vary and this could translate in a non-uniform temperature in the mold and the part [9]. One of the advantages of this type of cooling circuit is that it is more flexible in configuring the cooling rate and coolant temperature, and it can be more useful on large molds and dies where more than one set of conventional straight-drilled cooling channels are implemented to obtain a more efficient and uniform coolant effect [1].

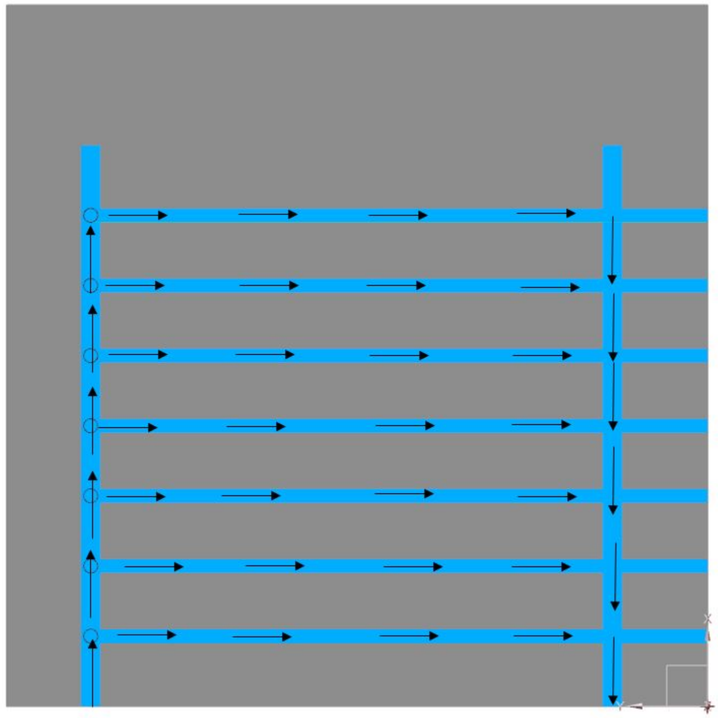


Figure 2.1: Parallel cooling channels

Serial cooling channels

These cooling channels are connected in a single loop from the flow inlet to its outlet. It is the most commonly used set up of circuits in the mold injection industry. In the design of this type of cooling channels, the fluid flow rate through the channel need to be high enough. So, the flow is turbulent in the cooling lines to enhance the heat transfer. When the molds are large, to ensure uniform cooling in the entire mold, more than one serial channel might be used to have uniform temperature as well as keep the turbulent flow inside of the channels [9].

Cooling devices

Straight-drilled cooling channels are limited by the manufacturing process in the mold to reach all the areas of the part to obtain a uniform change in temperature on the cooling process.

For these areas where conventional cooling channels cannot reach, some cooling strategies have been developed using some cooling devices in the channels, such as baffles and bubblers, which are the most common used in the PIM industry.

Baffles are a straight drilled cooling channel that are perpendicular to a main cooling channel which has a blade inside that separates the channel into two parts, allowing the flow coming in to the channel from one of the sides and leaving from the other side. Figure 2.2 is an side view of the baffles used in the mold A. These devices are in areas where the straight cooling channels cannot be conveniently drilled as the figure shows. One end is connected to the channel while the other end of the channel is not connected to any channel. It will just force the flow pass through a smaller surface area and return to the other side of the blade to enhance the cooling efficiency and uniformity in these areas where a normal straight cooling channel cannot reach [10].

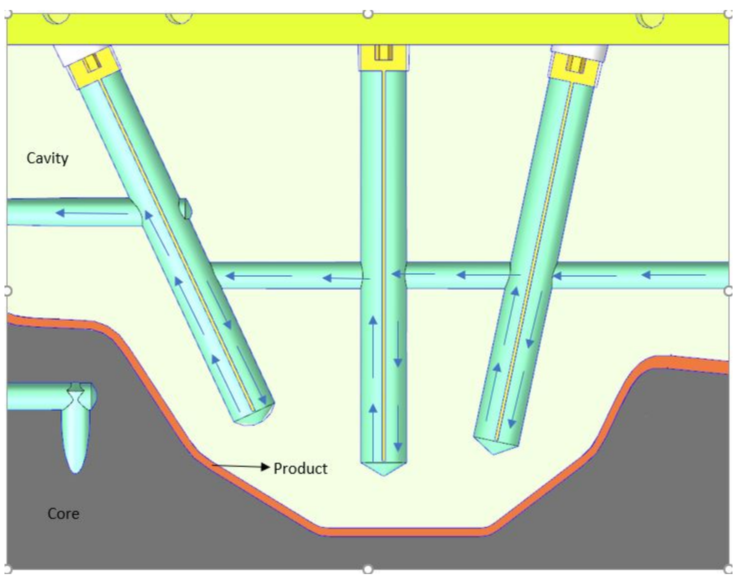


Figure 2.2: Use of baffles in cooling systems

Another type of cooling devices used, which are similar to baffles with the difference that the blade is replaced by a small tube. This is a tubular cavity with a larger diameter than a straight-drilled cooling channel with a concentric tube where the outermost end of the bubbler is places as the outlet. This device is also used for concave areas that cannot be reached by conventional cooling channels.

Despite the usage of these devices, in order to obtain a better cooling system capable of reducing the cooling time while maintaining uniform temperature in the product and in the mold, for complex geometries of the product to be molded, conventional cooling systems might not provide less cooling time or uniform cooling due to the limitation of the manufacturing process of straight drilled channels. This incapability of the conventional cooling channels for a complicated contour-like geometry enhanced the search for methods that provide a cooling system that can "conforms" to the shape of the part in the core and cavity.

2.2.2 Conformal cooling channels

Conformal cooling systems are the latest approaches on plastic injection molding in which channels are designed to follow the geometry of the part or conform to its shape. This type of channels allows the coolant to access most of the areas of the part in core and cavity uniformly, so that the cooling process gets more efficient and consistent without the need of the use of any additional devices for areas where the channels cannot reach using conventional cooling channels. Initially, Sasch et. Al.[11] conducted a comparison between the use of conformal cooling channel against the machined straight conventional cooling channels in mold injection tools. In this study, the numerical and experimental results were compared between the conventional and the conformal cooling systems indicating that the conformal cooling channels provided a more uniform surface temperature than the conventional cooling channels.

On another studies, Dimla et. al. [3] conducted a set of computational simulations with 3D CAD models for a design and optimization of conformal cooling channels to predict the most optimal location of the channels to reduce the cooling time. This design was compared with straight-drilled channels and showed to have better results as well. Park et. al [12] generated a study on the improvement of the cooling time and cycle time by designing conformal cooling channels for a complex geometry where straight-drilled cooling channels were difficult to reach some areas of the plastic part. In this study, the manufacturing process was also analysed since the model was manufactured and tested to validate the design of the cooling system.

The effect of different types of cooling channels for plastic injection mold was also studied by Khichadi et.al. [13], for a plastic part with a thickness of 5 mm. This study conducted computational simulations to optimize the cycle time and reduce the cooling time as well as any defects due to cooling process such as air traps and sink marks. Sink marks are local surface depressions in the molding interior due to an uneven heat removal during the cooling process. The study showed that conformal cooling channels provide less sink marks than the other types of channels as well as a higher percentage on the reduction of the cooling time. The conformal cooling channel in this study also showed to have uniform cooling process which makes it a more favourable cooling system.

Marques et. al. [14] analysed the use of different configurations of conformal cooling channels (parallel and serial) and compared them with conventional cooling channels by CAE simulations. The author indicated the importance of a higher pressure on the conformal cooling circuits due to the pressure drop that might experience due to the complexity of the geometry. Also, it was demonstrated that in the serial conformal cooling system, the flow rate achieved turbulent flow with Reynolds number close to 10,000, which is an optimal value for this type of application. Serial conformal cooling channels had better cooling efficiency. This serial conformal cooling, which was the best option for the case investigated, only achieve up to 6% reduction in the cycle time. On another hand, parallel conformal cooling channels did not have enough flow rate to achieve turbulent flow in most areas of the circuit, the Reynolds number was too high compared to the optimal Reynolds number for this application.

Additive manufacturing is the method used for the fabrication of conformal cooling channels inside of the mold. Different techniques of additive manufacturing are available in the industry, due to the development of solid free-form fabrication (SFF) technology [15]. The process allows the manufacture of 3D with the use of an additive process to print it out, where layers of material powder are laid down one after the other where the laser melts off the desired cross section following a CAD model. This additive process allows the creation of channels that are possible to conform the part following certain standards to also achieve the structural stability [8].

The constraint in this type of cooling system is the manufacturing cost being too expensive compared to the machining of a conventional cooling channel. Considering the low demand for the use of conformal cooling channels due to its current state of development, it might increase on its use if the cost of manufacturing gets lower. For this reason, the analysis of the optimization on cooling channels is of importance since this provides information on the performance of the cooling system and its impact on the reduction of the cycle time, therefore, an increase on the production of the part and reduction of its cost of manufacturing which can offset the cost of additive manufacturing.

2.3 Design parameters for cooling systems for PIM

In order to generate an analysis of the behaviour of the cooling systems on PIM, some factors are necessary to consider. To improve the design of the cooling systems in plastic mold injection tools, there are several factors that affect the performance of the cooling system. Some of these factors are described as follows [4] :

- Thickness of the model product: For thicker molded product, a longer cooling time is needed.
- Shape of the molded product: For those parts that have a complex geometry, the cooling in certain areas may be more difficult, which affects the cooling time needed for the part to achieve the ejection temperature.
- Quality of plastic material: The thermal diffusivity and thermal conductivity for different plastic materials used for this type of application will affect the cooling time, since the higher the conductivity and diffusivity the lesser the cooling time.
- Injection and ejection temperatures: When the difference between the injection and ejection temperatures is higher, the cooling time will be longer.

- Mold material: The metal materials used for mold injection tools with a higher thermal conductivity requires a shorter cooling time.

- Number, position, and size of cooling channel: The more cooling channels, the closer the cooling channel to the parting area or the larger the diameter of the channel, the better the cooling effect and shorter of the cooling time.

- Quality of coolant: different coolants have different specific heat, density, and viscosity, and they will affect on the heat transfer results.

- Coolant flow rate and temperature: Considering the heat transfer rate increases when the flow inside the channels is turbulent, the coolant flow rate must reach the condition of turbulent flow in order to have a better cooling. Also, in most cases, lower temperatures also will shorten the cooling time.

Location of cooling channels and selection of the cross section

Several studies have been conducted to find the most effective way to locate cooling channels in order to reduce the cooling time and maintain the uniformity of the cooling temperature through the part geometry and the mold.

Table 2.1 shows the parameters used in the study by Meyer [16], where a range of configurations were examined based on the thickness of the part, since for a thicker molded part a higher cooling time is expected. The diameter of the cooling channel (D), is determined by the thickness of the molded product. A range of diameters were recommended based on the thickness of the part in this research considering that it is necessary to have turbulent flow in the channels. This positioning will depend on the pitch between the cooling channels (P), and it is recommended to be between 3D and 5D. The distance between the channel centerline and the mold wall, L, was recommended to be between 1.5D and 3D. The length of the channel, and the number of the channels (N) have to be set up uniformly along the core and cavity area to maintain a uniform temperature. The parametric was conducted using different configurations to find the most optimal design.

Table 2.1: Correlations for design cooling systems in PIM (Meyer, 2009)

Thickness [mm]	Channel Diameter (D)[mm]	Pitch (P)	Distance from Centerline (L)
0 - 2	4 - 8	2D - 3D	1.5D - 2D
2 - 4	8 - 12	2D - 3D	1.5D - 2D
4 - 6	12 - 16	2D - 3D	1.5D - 2D

Different studies have been performed to develop a methodology for an optimal design on conformal cooling channels since its manufacturing process allows more flexibility in the development of the cooling channels. Dimla et.al. [3] investigated the position of the conformal cooling channels. To keep the temperature uniform some physical aspects must be considered, such as convex areas in the product need more cooling due to the heat concentration in this area, where concave areas need less cooling because the presence of more material helps the diffusion of heat in the mold. Khan et. al.[5] showed on their study that for the same pitch and cooling channel diameter, conformal cooling channels connected in series provides 6.39% less cooling time compared with the use of conventional cooling system. From this study, it is noticed the use of the configuration proposed by Mayer et. al.[16].

Marques et.al. [14] studied the design of the conformal cooling system using the diameter, pitch and the location of the channels based on those listed in Table 2.1 with a topology designed following the shape of the product. From this study, it was concluded that conformal cooling design in series was the most optimal configuration since it achieves Reynolds numbers for turbulent flow, with homogeneous mold temperature and the ejected product with smaller deformations. Parallel conformal cooling channels did not have enough flow rate to achieve turbulent flow in most areas of the circuits. Therefore for conformal cooling channels, parallel configuration is not the best approach.

Wang [17]proposed an spiral layout for the design of conformal cooling channel. All studies were done using MoldFlow as the software for the analysis but the study did not provide an experiment to validate the use of this type of cooling channels.

Bin et. al. [18] did a numerical study on the different configurations of the conformal cooling channels including longitudinal channels and serpentine channels. It was demonstrated that the serpentine cooling channel was best under low Reynolds number since it can provide a greatest coolant turbulence and highest velocity, which intensifies the heat transfer. Longitudinal conformal channels showed to have a better performance due the larger cooling areas, and more uniform temperature distribution, this configuration was better for higher Reynolds numbers.

Tomasoni et.al. [19], studied the optimization of a thermoforming mold cooling system by designing and analysing three different geometries of the cooling channel (serpentine, rectangular and tank), which all have circular cross sections. From this study it was concluded that the serpentine cooling system was the best approach to cool the mold, since this configuration allows the coolant fluid to follow a precise path and to reach almost every point of the part in study. Kuo [20] studied the influence of the cooling channel layout on the cooling efficiency using silicon rubber molds. In this study different conformal cooling layouts were used for configurations with various inlets and outlets, and it was demonstrated by computational simulations that the cooling systems in serpentine with four inlets and outlets provided the best result.

Marques et. al [14] showed by the comparison conformal cooling channels in series and parallel with conventional cooling channels for the cooling time and temperature distribution. It is showed that if the conformal cooling channel is not properly designed it cannot provide reasonable results. Therefore it is important to analyse the different scenarios and run parametric studies in order to find the cooling system that will achieve the expected results and reduce the cycle time of the PIM.

Another parameter that has been studied on the improvement of conformal cooling channels is the selection of the cross section. It was demonstrated by Shayfull et al.[21] that the use of square cross section provides a better cooling system since it reduces the warpage appearance as well as the reduction of the cooling time. The constraints on the use of the square cross section was its manufacturing since if the geometry of the product is too complex, the durability of the mold would be compromised.

2.4 Computational approach for cooling analysis on PIM

Before the invention of computational methods for analysis of models in the design of plastic mold injection tools, the best approach to evaluate the performance of a cooling system was by observing the product ejected from the mold or by doing a pre-production run. These methods increased the production cost since any changes that were going to be made on the system to improve the performance of the cooling process implied the manufacturing of a new mold. Despite of the experience on the development of these tools that the industry used, it was common to find defects on the products as warpage, surface cracking, damage on the mold, which was a consequence of the poor design of the cooling systems.

Different methods have been developed for the design and manufacture of plastic injection mold tools in order to improve the process of the design and obtain better performance. These methods have been part of studies using computed-aided design and computed-aided engineering has been mostly used in this process as the ultimate tools for a better design.

Kapila et. al.[7] with the use of MoldFlow simulations and using experimental methods, conducted a study for different flow rates and the influence of it on the cooling time, mold temperature and part temperature distributions. The study was conducted with the same flow temperature for all the different volume flow rates at $20°C$. Both computational simulation and experimental results showed how the increase of the flow rate can reduce the cooling time, but after a certain flow rate, in this case is $28[l/min]$, the cooling time does not reduce any more and in same cases the cooling time increases. Therefore in order to analyse the effect of the study of the flow rate on the cooling systems, it is necessary to generate a parametric study when a cooling system is being designed for a plastic mold injection tool. Other studies have been done for conventional cooling analysis to predict the effect of the addition of accessories into the channels on the cooling time and the temperature distribution in the mold.

Jahan and El-Mounayri ([22][23][24]) conducted studies on the optimal design for conformal cooling channels in plastic injection molding using numerical methods and validated the results with experimental data used in conjunction with Design of Experiments (DOE). They developed a design and used the numerical analysis for a conformal cooling channel for plastic injection molding. They used ANSYS work bench to predict the cooling time and the ejection temperature [22]. Also an DOE was conducted to analyse the influence of the channel cross section, where circular, square, rectangular, elliptical and semi-circular were evaluated. From these studies it was possible to obtain an optimal design of the conformal cooling channels following the DOE approach, where a redesign of the conventional cooling system was obtained for the cooling channel. From the second DOE in this study, it was also concluded that rectangular cross section channel proved to be most effective but it is limited in applications in industry due to manufacturing constraints.[22]

In another study, Jahan et.al. [23] generated thermo-mechanical design optimization study of conformal cooling channels using a similar approach of Design of Experiment (DOE) as they used on the previous study [22] where the computational results were validated by the experimental data. Two analysis were conducted in this study, a thermal analysis to obtain the cooling time and the temperature distribution on the product and the mold, and a structural analysis to predict the deformation and distribution of Von-mises stress on the mold body for the structural stability of the designed system. The combinations of these two analyses, showed that the cooling system with the lowest cooling time provided higher Von-mises stress that exceed the allowable stress limit. However, when the Von-mises stress was the lowest, the cooling time was not the lowest but would be less than the cooling time of the traditional conventional cooling system. From this study, it is noticed the increase in the channel pitch distance increases the cooling time. It was proven that the conformal cooling improves the cooling efficiency compared with conventional cooling channels, providing an increased uniformity in the temperature distribution. [23].

With the advances in the computational methods studies have been conducted to develop methodologies to predict the behaviour of the cooling system in plastic injection mold tools that can help the mold designers to improve their design. Jahan et.al. [24] based on the DOE technique proposed a design methodology with basis and guidelines for mold makers to support their conformal cooling channel designs. The methodology consisted of the analysis of three different thickness of the plastic part, as well as circular, cylindrical and conical shapes of the plastic part. The channel cross-section design was also part of the study including circular, rectangular and square ones. A total of 27 designs were proposed based on a thermo-mechanical analysis in order to find the best one. From this study, it was found that the best design solution may not be feasible in practical industrial applications.

Different software have been used to predict the performance of plastic mold injection tools. For the analysis on the cooling system, MoldFlow has been the software most used in the industry to predict the behaviour of cooling system since it integrates, the analysis on the appearance of defects on the plastic part as warpage and sink marks. A computational analysis using ANSYS was conducted by Jahan et. al. ([24][22] [23]) where the cooling time was obtained. Kuo et.al.[25] used Moldex3D to investigate the cooling time of molded parts, the mold temperature difference, the part temperature difference and the warpage of molded parts of two different conformal cooling channels, one is a profiled conformal cooling channel and the other one is a regular conformal cooling channels. In this study two margins steel injection molding tools with the two designs of conformal cooling channels were fabricated as well by direct metal printing technology to validate the numerical results.

Chapter 3

Methodology

3.1 Introduction

The PIM process comprises four stages: filling, packing, cooling and ejecting the molded part. In this process, the cooling stage is to reduce the temperature of the molded plastic part, uniformly to the ejection temperature. The objective of the analysis of the cooling stage is to obtain the temperature distribution in the molded product and mold cavity.

Based on the procedure of the plastic mold injection tool of injection, packing, cooling and ejection, as showed on Figure 1.2, the total cycle time used in the PIM is the sum of the time for each stage.

$$t_{cycle} = t_{injection} + t_{packing} + t_{cooling} + t_{ejection} \tag{3.1}$$

Equation 3.1 shows the calculation of the cycle time, where $t_{injection}$ is the time from the moment where the melted plastic is injected until it fills the cavity area of the product, $t_{packing}$ is the time where pressure is sustained after the cavity is filled, $t_{cooling}$ is the time that takes to solidify the part until the ejection temperature, $t_{ejection}$ is the time that the product is ejected from the mold. The time that the mold takes on each stage will vary depending on the design of each of these processes. Since the cycle time needs to be reduced to reduce the production cost, the major impact on this cycle time reduction is the cooling time. It is important to design an optimal cooling system.

The cooling process consists of the removal of unwanted heat from a certain area. For the case of the plastic injection mold, heat needs to be removed from the molded plastic part in order to solidify the melted polymer and achieve the ejection temperature. The cooling process starts from the moment when the plastic is being injected into the mold cavity and followed by the packing procedure. Based on the objective of the cooling system of removing heat from the part, a key consideration for the design of the cooling channel is the effectiveness of the heat transfer between the melted polymer close to the parting surface to the nearest cooling channel. Most of the heat transfer occurs due through conduction.

The heat transfer process in a tool is shown in Figure 3.1. The coolant flow passes through the channel, the melted plastic is on the cavity section at a distance D from the center of the channel. The heat transfer occurs from the melted plastic material by conduction to the interface between the melted polymer and the mold. This heat transferred passes through this interface followed by the mold body and finally to the interface between the mold and the coolant interface through conduction as well. Then the heat is transferred to the coolant from the interface through convection until finally the temperature of the body reaches the ejection temperature.

Applying the energy balance principle to the mold injection cooling process, the heat flows into the mold by the melted polymer $q_{plastic}$ is transferred via conduction and convection in the cooling system. Based on the previous model proposed by Lin et. al. [4] the mold exterior faces can be treated as adiabatic, and consequently the heat transfer by radiation is not considered in the study.

$$q_{plastic} = q_{cond} + q_{conv} \tag{3.2}$$

The rate of conduction heat transfer is described by Fourier's Law. The convection heat transfer happens through the cooling channel. The Newton's law of cooling can be applied for the convection heat transfer.

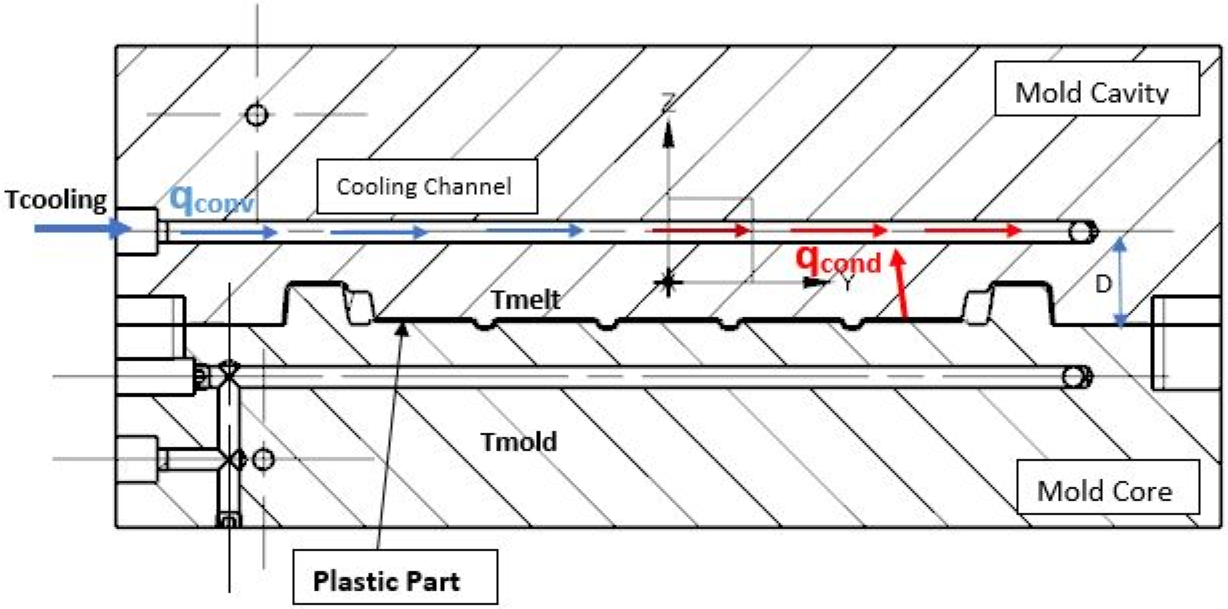


Figure 3.1: Heat transfer process in a mold

From the methods of heat transfer involved in the cooling process of the polymer for plastic mold injection tools, it is possible to notice that heat conduction is the process that governs the effectiveness of the cooling system. Previous studies have been mostly for the cooling systems design methods based on heat conduction theory. [4] [15] [3]

In this chapter, the methodology used for the analysis and its application on the cooling system is explained. On the first stage of this research, the applicability of FLOEFD of Siemens NX for the simulations of the cooling system in plastic mold injection tools is also analysed.

To evaluate the capabilities of the software, the mold A is used for simulations to analyse the cooling time of the part, the temperature distribution in the part ad well as in the mold core and cavity to identify hot spots in the part. In order to validate the results obtained from the simulation, a comparison is done with the experimental data. Once the computational results are validated with the mold A, the numerical study for the mold B will be done.Then, a parametric study will be carried out to improve the performance of the conventional cooling systems.

Once the analysis of the conventional cooling systems for both molds are performed, the design of a conformal cooling system is conducted for the product part of the mold B.The computational simulations for the conformal cooling system are performed on this mold to evaluate the improvements on the cooling time and the temperature distribution for the new design compared it with the conventional cooling system.

3.2 Use of FLOEFD as a CFD software to analyse the design of the cooling system in Plastic Injection Molding tools

FLOEFD-NX from Siemens is a fully integrated software with the capability of simulating fluid flows inside or outside models, including the heat transfer due to convection, radiation, and conduction with the use of computational fluid dynamics technology (CFD). FLOEFD possess an interface that includes pre-processor for specifying data for calculation, co-processor for monitoring and controlling the calculation, post processor for viewing the results. This software has been used in many industries for various applications, which allows the analysis of laminar, turbulent and transitional flows, heat transfer within fluids, between fluids and solids, steady-state and transient flows, two and three dimensional analysis. Considering that most of the fluid flows encountered in engineering practice are turbulent, FLOEFD was mainly developed to simulate turbulent flows.

FLOEFD has not been used in the PIM analysis, specifically for the cooling system analysis. Considering the scope of the software, it is important to analyse the feasibility of its use on the analysis and optimization of cooling systems for the PIM. From the Technical Reference handbook of Simcenter FLOEFD for NX, version 2021.3, it is possible to use the software to conduct CFD simulations for cooling system for PIM.

3.2.1 Governing Equations

Computational Fluid Dynamics describes the numerical solution of the partial differential equations that governs fluid flow and associated processes. These equations that governs the fluid flows are, based on the laws of the conservation of mass, momentum and energy for fluid flows.

The Favre-averaged Navier-Stokes equations are used in this study. The Favre-averaging approach takes the time-averaged equations and simplify them using the density-weighted averaging. Using this procedure, terms as the Reynolds stresses appear in the equations for which turbulence models must be used. The turbulence model used in FLOEFD for the turbulent kinetic energy and its dissipation rate is the standard $\kappa - \varepsilon$ model, where two additional transport equations are used to describe the turbulent kinetic energy and the dissipation rate.

The governing equations are given as follows:

$$\frac{\partial \rho}{\partial t} + \frac{\partial \rho u_i}{\partial x_i} = 0 \tag{3.3}$$

$$\frac{\partial \rho u_i}{\partial t} + \frac{\partial (\rho u_i u_j)}{\partial x_j} + \frac{\partial P}{\partial x_i} = \frac{\partial \tau_{ij} + \tau_{ij}^R}{\partial x_j} + S_i \tag{3.4}$$

$$\frac{\partial \rho H}{\partial t} + \frac{\partial (\rho u_i H)}{\partial x_i} = \frac{\partial (u_i(\tau_{ij} + \tau_{ij}^R) + q_i)}{\partial x_i} + \frac{\partial P}{\partial t} - \tau_{ij}^R \frac{\partial u_i}{\partial x_j} + Q_H \tag{3.5}$$

i = 1,2,3 , j = 1,2,3

$$H = h + \frac{u^2}{2} + \frac{5}{3}k \tag{3.6}$$

From these equations, u is the fluid velocity, ρ is the fluid density, S_i is a mass-distributed external force per unit mass, for the current case is only due to gravity $S_i = S_i^{gravity}$, h is the thermal enthalpy; Q_H is the heat source or sink per unit volume, τ_{ij} is the viscous shear stress tensor,τ_{ij}^R is the Reynolds-stress tensor that follows the Boussinessq assumption, q_i is the diffusive heat flux. The subscripts are used to denote summation over the three coordinates directions. (FLOEFD Technical Reference 2021)

3.2.2 Numerical Solution Technique

To solve the governing equations, the discrete numerical technique based on the finite volume method (FVM) is used. To use the FVM, the computational domain is discretized into a non-overlapping set of finite regions which fill the domain of interest completely. In this case, Cartesian rectangular coordinate system is the one employed. To obtain the space discretization, the axis-oriented rectangular grid is used far from a geometry boundary. The approach of near-boundary mesh is used. The spatial derivatives are approximated with implicit difference operators of second-order accuracy. Time derivatives are approximated with an implicit first-order Euler scheme.

Two-Scales Wall Functions Model

Considering the behaviour of the flow in the boundary layer region, a Two-Scales Wall Functions model is used. "*Thick-boundary-layer*", is used on a fine mesh (the number of cells across a boundary later is 6 or greater). "*Thin-boundary-layer*", is used on a coarse mesh (number of cells across a boundary layer is 4 or less). An appropriate boundary-layer approach will be selected automatically by the software based on the computational mesh.

3.3 Configurations of the molds

Two molds are used in this study and both are provided by the mold manufacturer. Figure 3.2 shows the complete tool for the mold A, and figure 3.3 shows the complete tool of the mold B. These figures show all the components in the tool.

Since the entire geometries of both mold have a high level of complexity, it is necessary to simplify the geometry in order to run a CFD simulation. To reduce the computational time and cost, for each of the mold, just the components of the tool that have contact with the cooling system are used in the CFD simulations.

This reduction of the mold will not affect the accuracy of the results since the heat transfer

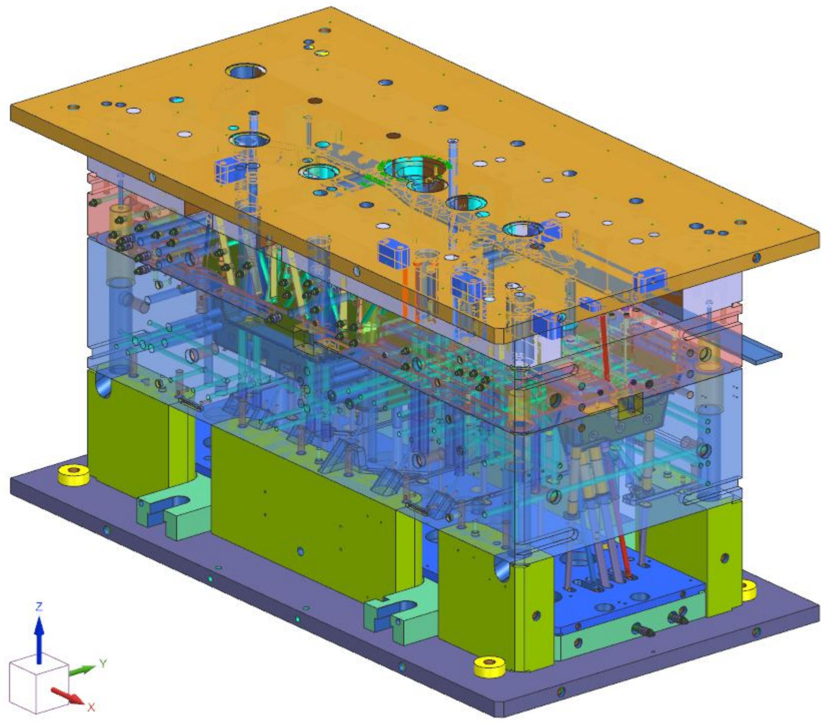


Figure 3.2: Configuration of mold A

will occur only in the areas where the cooling channels are displaced. The rest of the geometry involved in the PIM process does not affect the cooling process, only the injection and the ejection process that are not being simulated but are defined by a heat source.Figures 3.4 and 3.5 show the simplified configurations, which will be used in simulations for molds A and B, respectively.

The main objective of the simulation is to obtain the cooling time, the temperature distribution during the cooling process in the part and in the mold, and the temperature distribution at the end of the cooling process when the part is about to be ejected. Since the only process analysed in this study is the cooling process, to simulate the injection stage, the initial time of the process is going to be taken when the entire cavity is filled with the melted plastic. To simulate the injection and ejection of the plastic into the mold, a cyclical heat source will be set up in the product surface area. This source is a heat source that starts turned on with the melting temperature of the plastic. Once the coolant starts to flow through the channels, and

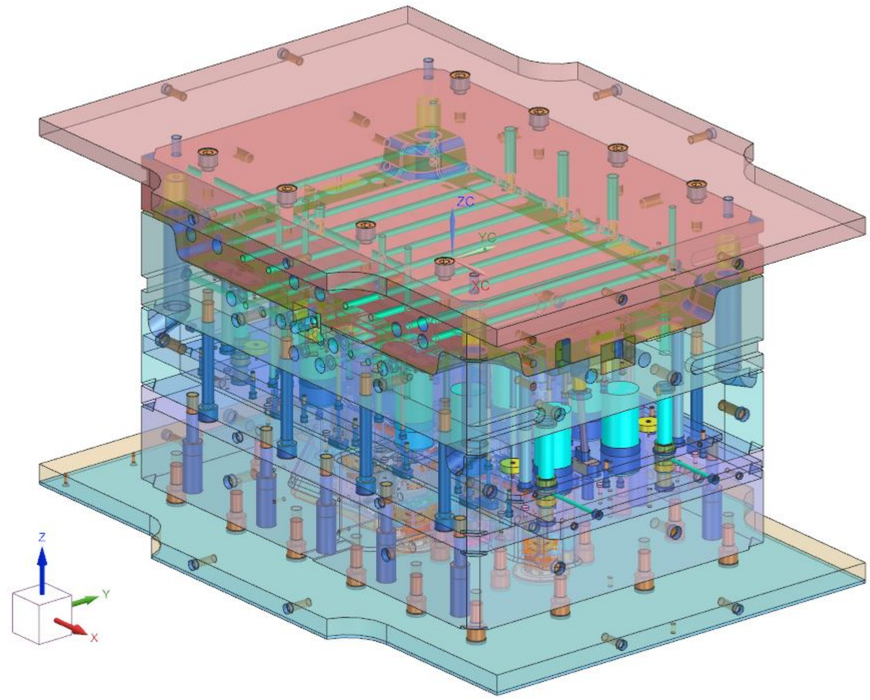


Figure 3.3: Configuration of mold B

the heat transfer occurs, the source is turned off until the part reaches the ejection temperature. The source is turned on again when the temperature of the part is bellow the ejection temperature, to simulate the injection of the next part. The cooling time is the time between the injection temperature of the part and the ejection temperature and is calculated from the averaged temperature on the entire part surface.

Since it is necessary to know the time that the cooling system takes to cool down the product, this is a transient simulation. The type of flow to be considered in the study is an internal turbulent flow. Heat transfer bin the mold and the part is also taken into account. The initial temperatures of the core and cavity and any other mold component are set up based on the temperature that the mold would have once the operations are running.

The coolant used in the study is water. The material used for the mold core and cavity is steel P20, and Table 3.1 shows its properties. This material is a commonly used material mold for the mold in the industry. This type of steel is characterized for its moderate strength levels

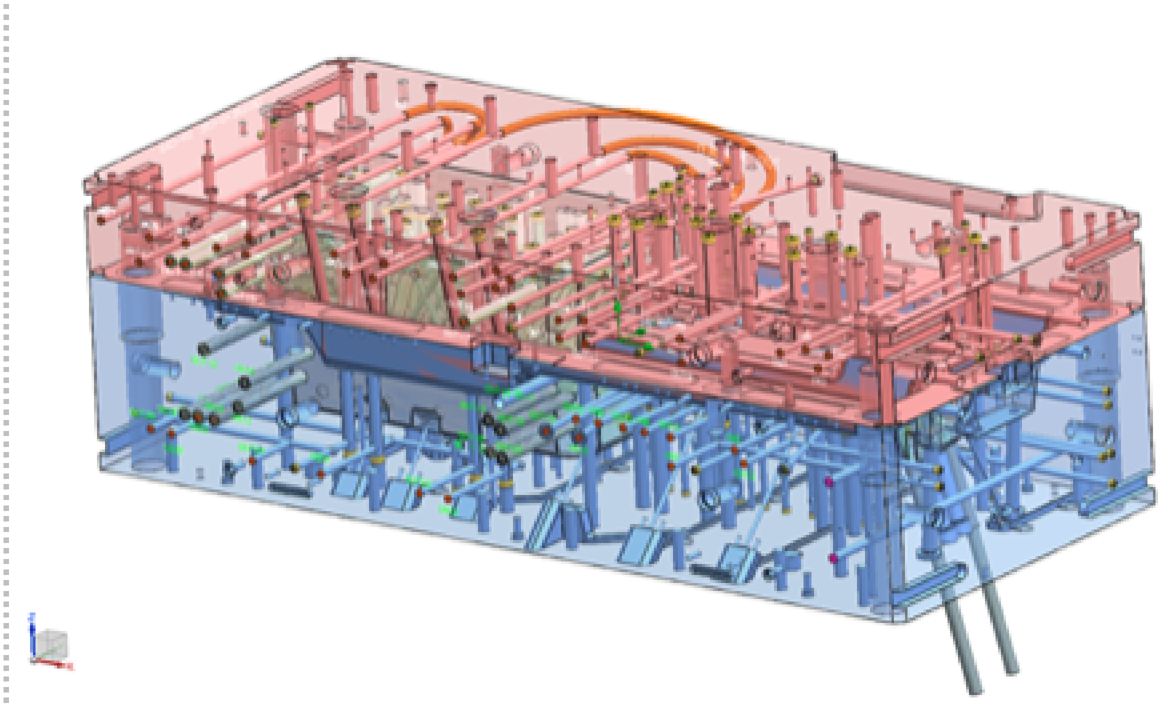


Figure 3.4: Simplified geometry for mold A

and its acceptable toughness, and it is ideal for plastic injection mold tools. The materials of the plastic are different for the two mold and both are used in the industry.

Table 3.1: Material properties of P20 Mold Steel

Property	*Value*
Density $[kg/m^3]$	7810
Specific Heat $[J/Kg * K]$	460
Thermal Conductivity $[W/m * K]$	30

3.3.1 Mesh

FLOEFD is a CFD Software that generates their own mesh. The mesh is generated for the entire geometry first and for areas where the gradient of flow parameters is high, then it is refined. 3D Tetrahedral mesh is used. Global mesh is used for the entire geometry, while local mesh is refined on the area of the product to be molded.

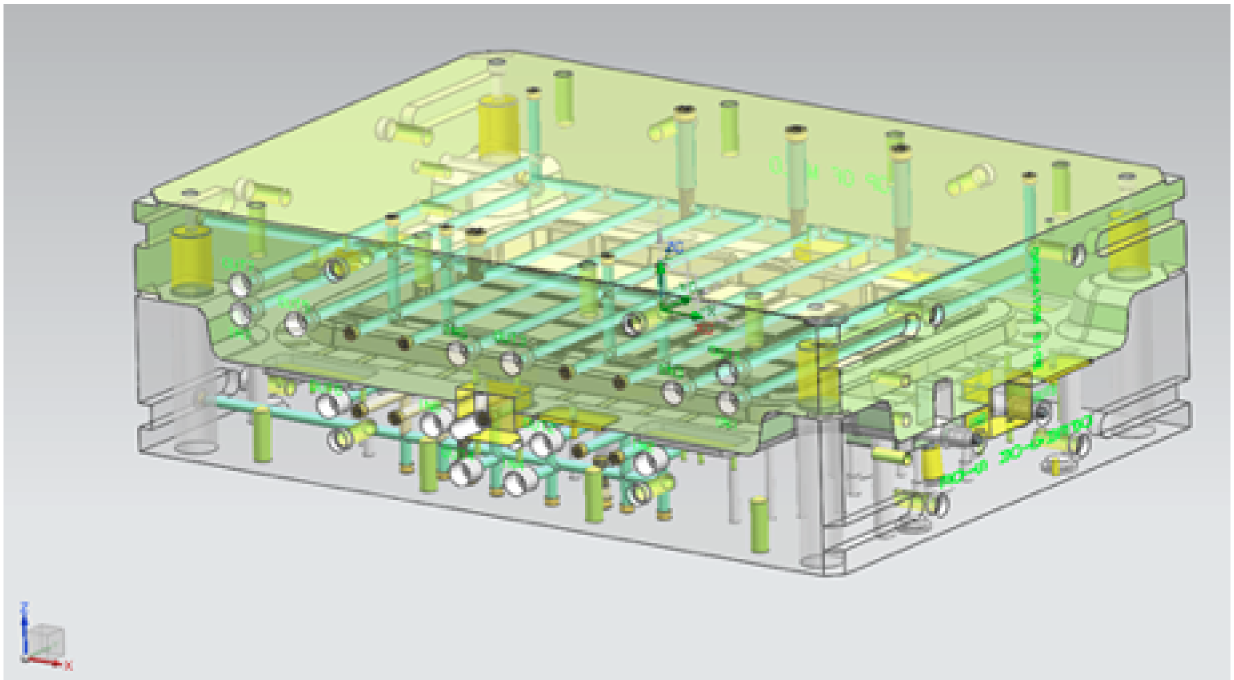


Figure 3.5: Simplified geometry for mold B

3.3.2 Heat Source

To simulate the injection and ejection of the part in the mold tool, a heat source has to be defined on the product surface area. This heat source cannot be constant since it changes based on the injection and ejection processes.

The heat source is specified based on the temperature of the part. When $t = 0s$, the average temperature of the part starts with the injection temperature (close to the melting temperature). Once the cooling process starts, this source is turned off. The cooling system removes the heat from the part, so temperature of the part decreases. When the product achieves the ejection temperature, the heat source is turned on to the injection temperature and the cycle is repeated. This allows the simulation of the injection and ejection stages in the molding process.

Chapter 4

Validation

4.1 Overview

In this chapter, FLOEFD is validated as a CFD software for the analysis of the cooling system in plastic injection mold tools. This validation is done by running simulations on the mold A, and comparing the simulation results with the experimental data performed by the manufacturer of the mold prior its construction. With the comparison between the computational results and the experimental data, it is possible to determine how accurate the simulations are. This is the mold that has detailed information on the cooling time, temperature distribution and different configurations used, it will be used to validate the FLOEFD approach for the analysis.

4.2 Configuration of Mold A

The geometry of mold A is as showed in Figure 4.1. There are 6 components in the mold. A more detailed view of the channels for the core and the cavity areas can be observed in Figure 4.2.

The cooling system for this tool is a conventional cooling system with different inlets and outlets, which form different circuits along the part in the core and the cavity. The diameter of these channels is 0.5943 in, which is selected based on the availability of the straight-drilled

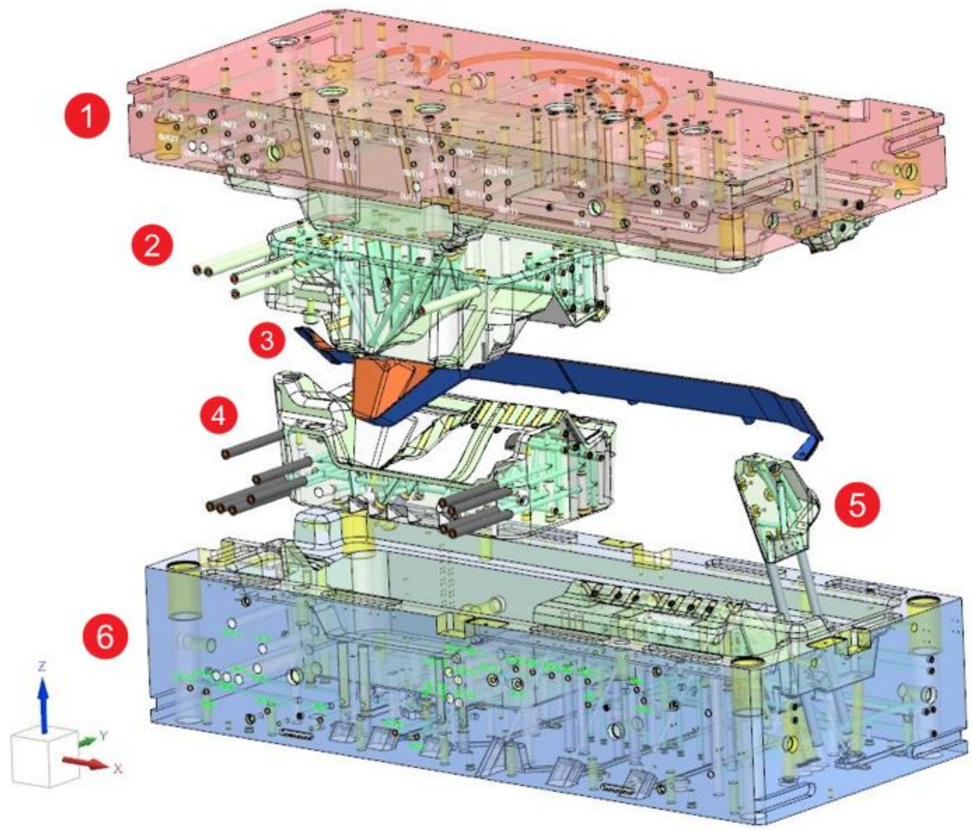


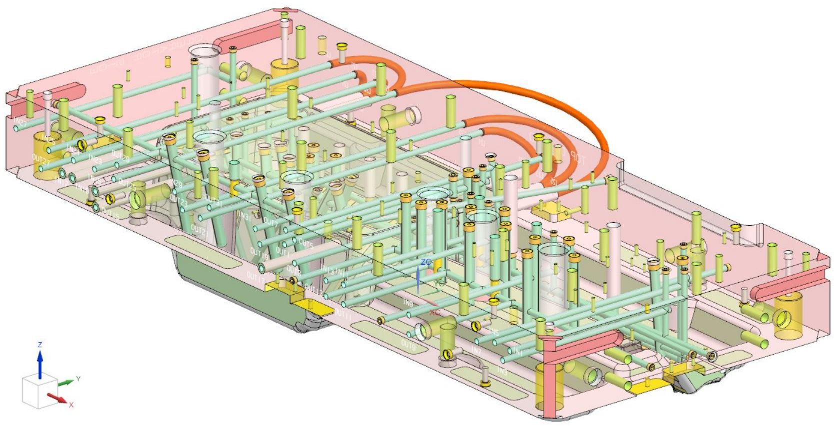
Figure 4.1: Simplified mold A: 1) Cavity, 2) Cavity insert, 3) Plastic part, 4) Core insert, 5) Core lifter, 6)Core

manufacturing process. To improve the uniformity of the cooling system part of the accessories used are diverters and bafflers for the areas where the conventional cooling channels are unable to reach.

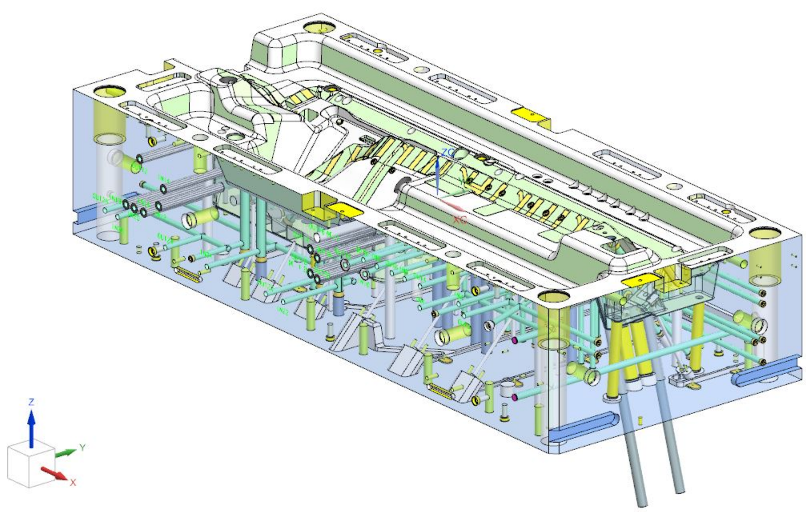
The product part material is an ABS plastic material, polypropylene used in the automotive industry due to the high impart resistance, aesthetics, flow, strength and dimensional stability. The properties of this material can be found in the following Table 4.1.

Table 4.1: Material properties of Polypropylene

Property	*Value*
Density $[kg/m^3]$	1075
Thermal Conductivity$[W/m * K]$	0.082
Melting Temperature $[K]$	533.15



(a) Cavity



(b) Core

Figure 4.2: Mold cavity and core cooling channels

4.2.1 Boundary conditions, initial conditions and source term

Boundary Conditions and Initial Conditions

- Wall : Adiabatic and non-slip
- Initial conditions: The pressure and temperature are going to be set up as the ambient conditions. The velocity will be set to $0m/s$.
- Solid Parameters: The solid temperature is set at $55°C$ as it is a initial condition.

- Inlet boundary conditions: The volume flow rate of the coolant (water) is $12l/min$. The temperature is $28°C$ for the initial set of simulations, and $60°C$ for another set of simulations. The turbulence intensity and turbulence length,are set as 2% and $0.00048m$, respectively.

- Outlet boundary conditions: The ambient pressure is the outlet boundary condition at all outlets sections for both, core and cavity.

Heat Source

Considering the plastic part used in this study, the injection temperature is the melting temperature which is $240°C$, the heat source temperature is set as $230°C$ to avoid melting issues in the simulations. For the ejection temperature, based on the studies from the manufacturing on the strength and properties of the part, it is set as $105°C$.

Table 4.2 provides a summary of the general set up for the simulations.

4.3 Grid and Time Step Independence Study

4.3.1 Grid Independence Test

Three different meshes are used for the grid independence test and their cell count are shown in Table 4.3, identified from the finer to the coarsest.The time step used for this simulation was $0.01s$.

Table 4.4 shows the results of the relative error. Figure 4.3 shows the transient heat transfer rate using the three different meshes.

The relative error is used for the selection of the mesh, with an allowance of 1-2 % of error. Since the relative error between the mesh 2 and the mesh 3 is small enough, mesh 2 is selected for the rest of the simulations in this study. Figure 4.4 shows the mesh on the solid region with the respective refinement level.

Table 4.2: Summary of Simulation Set Up

Characteristic	*SimulationSetup*
General Settings	• Internal Flow • Considers conduction between solids and convection • Fluid definition :Water • Solid Definition: for the mold Steel P20 initial temperature of $55°C$ • Wall Conditions: Adiabatic Walls, non-slip
Initial Conditions	• Pressure and temperature with ambient conditions • Fluid flow velocity $0m/s$
Materials	• Plastic Part: Polycarbonate (PC) ASA • Mold: Steel P20
Boundary Conditions	• Inlet: Volume Flow Rate: $12l/min$ / Temperature = $16°C$ • Outlet: Ambient Pressure
Heat Source	• Injection Temperature: $230°C$ • Ejection Temperature: $105°C$

Table 4.3: Meshes used in the grid independence test for mold A

$MeshID$	$NumberofCells$
1	3,756,276
2	2,507,664
3	1,805,891

Table 4.4: Grid independence test results

Mesh ID	Relative Error [%]
2-1	1.05
3-2	6.6

4.3.2 Time Step Independence Test

With the mesh from the grid independence test, another set of simulations is performed for the time step independence test. The time step independence test is conducted using three different time steps, $1s$, $0.01s$ and $0.001s$. The simulations are performed following the configuration listed in Table 4.2. The periodic set up for the freezing period is 10 iterations on freezing and 10 iterations at no freezing, starting after 100 iterations. Freezing and no freezing periods are defined for storage purposes to reduce computational cost in the simulation.

- The first simulation a time step of 0.01 will be set up with freezing period of 20 iterations and non-freezing period of 10 iterations.
- The second simulation will use a time step of 0.01 with freezing and no freezing period of 10 and 5 respectively.
- The third simulation will use a time step of 0.005 with a freezing and no freezing period of 10 and 5 respectively. Is important to mention that for a very small time step, due to the complexity of the geometry and the size of the model the software will take more than expected to provide results.

The cooling time is the parameter used to compare the results between the different time

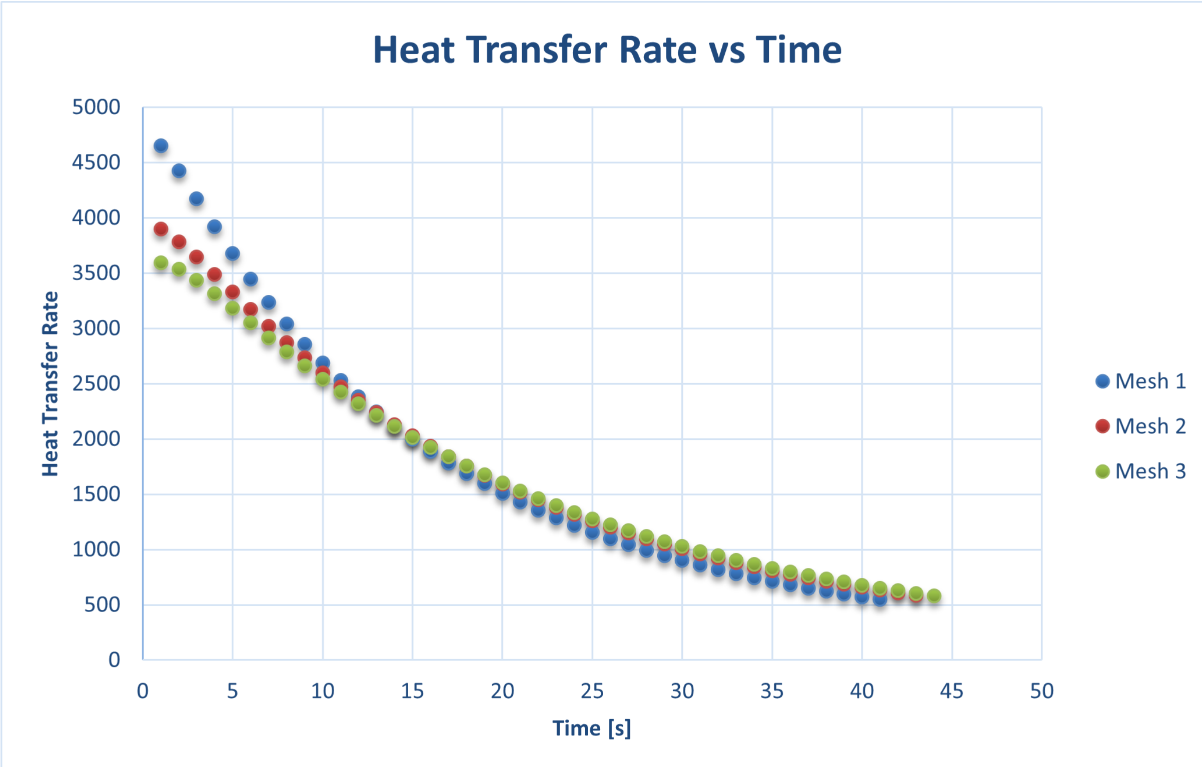


Figure 4.3: Transient heat transfer rate for three different meshes

steps. Table 4.5 shows relative errors from the different time steps, which are about 3%. For this study, between 2 – 5% of error is accepted.

Table 4.5: Time step independence test results

Time Step	Relative error %
1 - 0.01	3.77
0.01 - 0.001	3.33

Is possible to notice that from the 3 time step they all provide a relative error approximately 3% without a significant fluctuations. An smaller time step can lead to a longer computational time, which in case of looking for a more accurate result it will be a necessary practice. Since the relative error of the time step between $0.01s$ and $0.001s$ is small enough for the purpose of this study and the applicability for the industry applications. Therefore, the time step used for the rest simulations will be $0.01s$ since it provides result with a permissible error with less

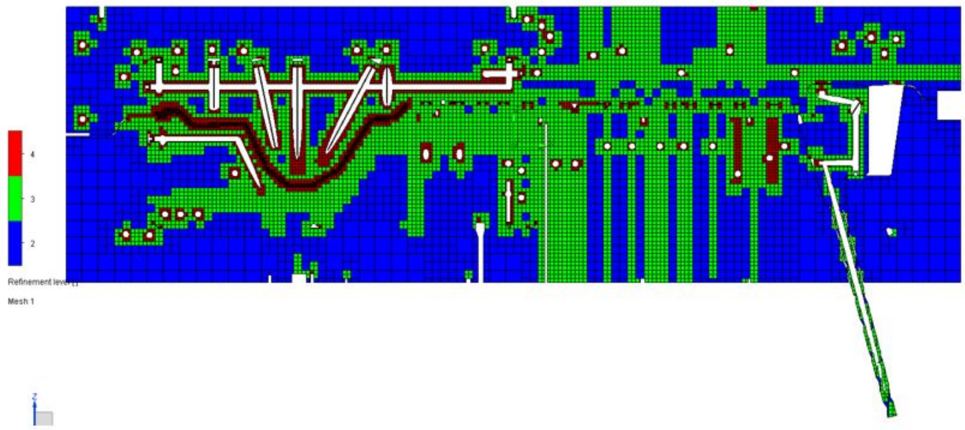


Figure 4.4: ZX Plane view of the mesh of the solid in Mold 5468

computational time.

4.4 Results

Using the mesh from the grid independence study, and the time step from the time step independence test, the simulation is performed for the mold A and the predicted cooling time is compared with the experimental data to validate the numerical model.

In order to obtain the temperature distribution on the part, as well as the cooling time in the part, an average temperature on the solid is to be set up as a goal results to take in the time that it takes to reduce the temperature from the melting point to the ejection point. Another result goal set up on the solid part is the heat transfer rate generated by this temperature changes, since it allows to evaluate the consistency on the heat transfer rate in the part.

From the flow perspective, to evaluate the consistency on the flow inside of the channels, an absolute total enthalpy on the inlets is to be set up as a goal result as well as the ambient pressure of the core and cavity velocity averaged from the outlets.

For the calculation of the cooling time, with the variation of the averaged temperature calculated on the part within time, it will be possible to obtain the time that takes the part to reduce its temperature from the injection temperature to the ejection temperature by subtracting the time from the lower temperature of the time with the higher temperature. The period of time where the part goes from the injection temperature to ejection temperature is one cooling cycle. To obtain the most accurate cooling time it is necessary to use the data after certain numbers of cycles. From previous simulations it was noticed that the predicted cooling time is stabilized after 20 cycles.

For this case the cooling time calculated from the average temperature of the part is $44.95s$ and Figure 4.5 shows the trend of the predicted cooling time of each of the cycles after 10 cycles of the simulations. Figure 4.6 shows the plot of the average temperature of the part changing with time. Also the heat transfer rate associated is shown in Figure 4.7.

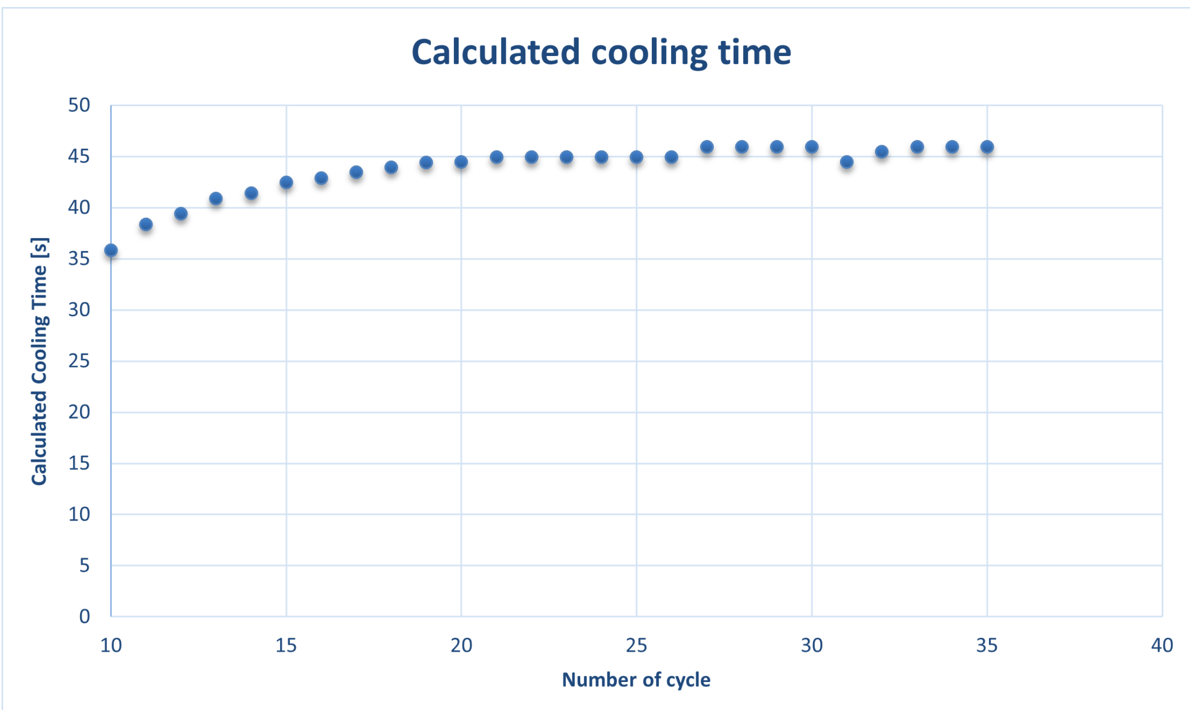


Figure 4.5: Cooling time cycles in mold A

Figure 4.8 show the temperature distribution in the part at the ejection time. So, the hot spots (higher temperatures areas) can be identified.

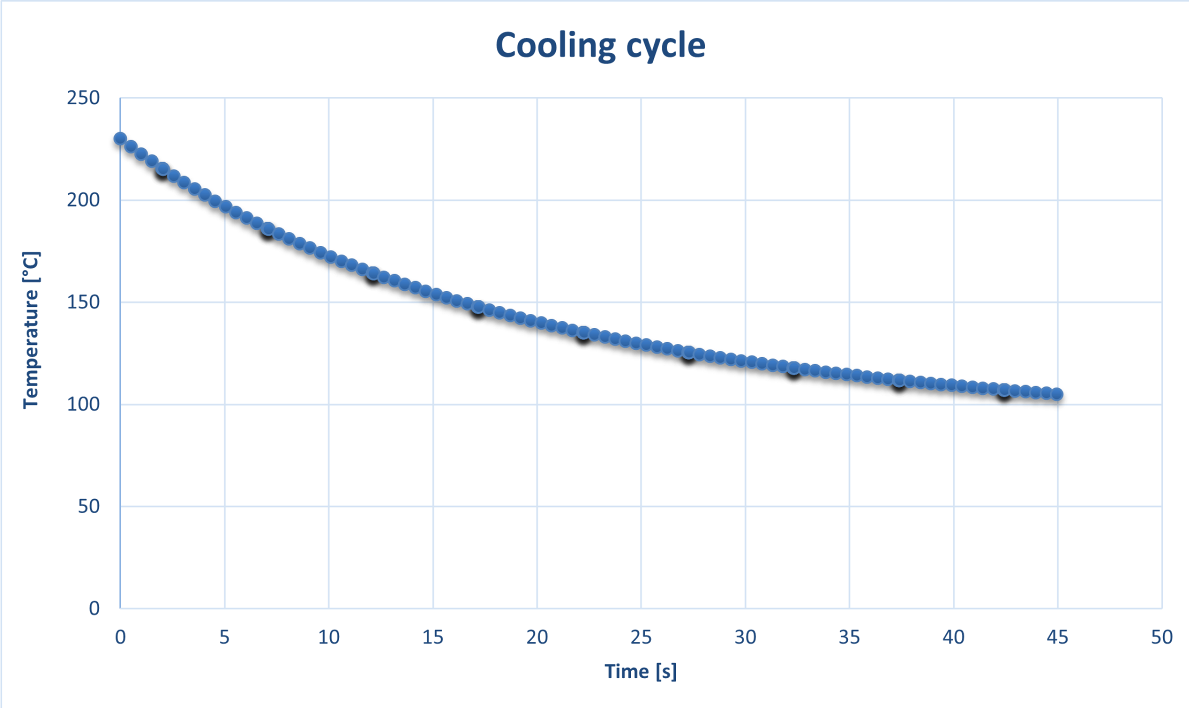


Figure 4.6: Cooling cycle in mold A

4.5 Experimental Data

The operating parameters were provided by the manufacturer of the tool. The volume flow rate used is 12 l/min with a flow temperature of $60°C$. The injection and ejection temperatures are $240°C$ and $105°C$ respectively.

The experimental data was provided by the manufacturer of the tool. The data provided was obtained during the testing of the mold that were designed to evaluate the final conditions of the product if they meet the desired goals.

The data includes the entire process of the mold cycle, i.e.: open/close, fill time, pack/hold and cool/cure. The cooling time is considerate from the moment when the open/close stage is done, the close stage happens once the melted plastic has been filled in the cavity area, and it is to guarantee a filled cavity and to seal the mold. Once it is achieved the clamping pressure of closure it is when the cooling time is starting to be recorded. Thus the data provided by the manufacturer has these two stages, cool/cure and open/close.

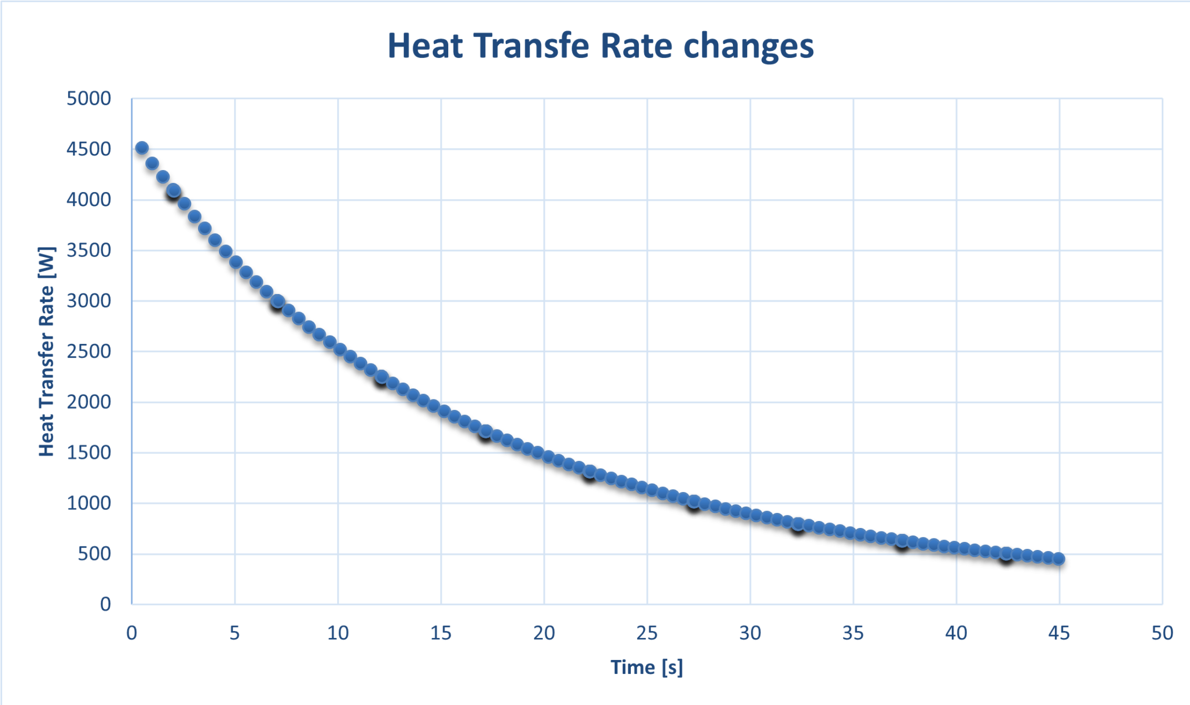
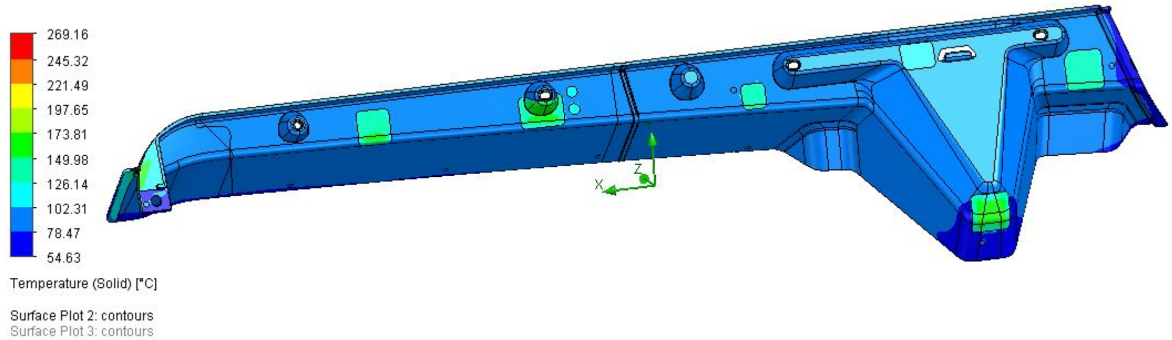


Figure 4.7: Heat transfer rate changes per cycle in mold A

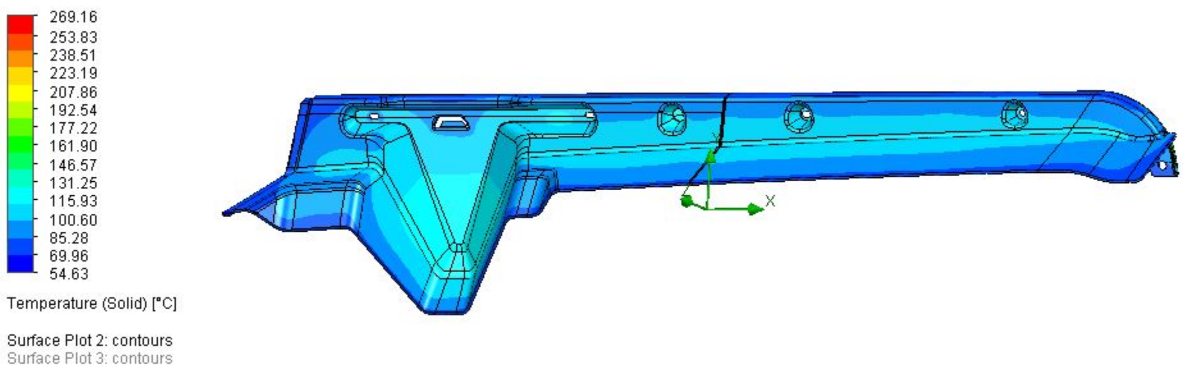
Table 4.6 gives the experimental data on the time for each cycle stage. The pack/hold takes 20s and the cool/cure takes 25s so, the total cooling time is 45s.

Table 4.6: Cycle stages and their duration

$CycleStage$	$Time$ s
Open / Close	24.44
Fill Time	4.6
Pack / Hold	20
Cool / Cure	25
Cycle	74
Total Cool Actual	45



(a) Back View



(b) Front View

Figure 4.8: Part temperature distribution at the ejection time

4.6 Comparison between Computational Results and Experimental Data

The cooling time from the simulations ($44.95s$) agrees well with the experimental data ($45s$) which gives an error of 0.1%. It should be mentioned that there is a difference on the ejection temperature of the part from the simulation with the experimental data, since the simulation uses an ejection temperature of $105°C$ and the experimental data gets up to $90°C$. This difference is generated due experimental changes applied on the testing site of the tool where the data was taken.

Considering the changes applied during the experimental tryouts of the mold, a new set of simulations was generated using the same configuration as the previous calculations. The ejection temperature defined for this new case was $90°C$ and the inlet volume flow as 12 l/min at $60°C$ since these are the configurations used on the experimental set up. The cooling time obtained for this case was $48.49s$, providing a relative error of 7.19% from the cooling time obtained by experimental data. This percentage of error is considerate inside of the range of the accuracy that the industry accept (between $5-10\%$) since the computational approach provides general information of how the current cooling system is expected to behave. The reason of this error might be generated from difference sources as, the collection of the data during the testing of the tool (experiment), the added interpolation process used or computational accuracy.

For this case, to validate the use of the software the only experimental results provided to be able to compare the cooling design is the cooling time. The calculation of the cooling time is the principal value to analyze a cooling system design, and for this reason in this case was the only factor used to observe if the computational approach using FLOEFD was close enough to the results obtained in the experimental testing of the mold. Other data as temperature distribution of the plastic part, is not a common practice to measure in the plastic part while generating experiments in the mold. The use of the cooling time as a comparing parameter between computational approach and experimental results is valid for the purposes of the study.

Based on the comparison between the simulation and the experimental results, the CFD approach used to simulate cooling systems is good enough to be used for optimal designs of the cooling systems prior to the construction, to reduce manufacturing cost.

Chapter 5

Results and analysis

5.1 Overview

In this section, the simulation on the cooling system of the mold B will be performed to evaluate its cooling performance. First the grid and time step independence study will be analysed, followed by the parametric study with different volume flow rates, fluid temperatures and change of the cooling channel diameters to find out the most optimal set up used for the conventional cooling system in the mold.

A conformal cooling is then developed for the same mold. An analysis on this system is performed by running a similar simulation to evaluate its behaviour. A comparison between the two cooling systems is conducted to show that the conformal cooling channel provides a better performance for mold injection cooling systems.

5.2 Configuration Mold B

The purpose of the analysis on this model is to evaluate the performance of the cooling system on the design stage of the manufacturing of mold injection tools. This mold is also provided by the manufacturer. Figure 5.1 shows the components of the mold used in the computational domain. The plastic part to be molded is a gallon lid, and the mold geometry includes the cavity, core, plastic part and inserts due to small tabs located on the side of the lids that requires lifters in these points to eject the part avoiding damages.

The conventional cooling system in this mold has four circuits at the cavity, three circuits in the core and a single circuit in each of the inserts. The diameter of these channels is 0.5780*in*. Diverters are used in this cooling system to guide the flow through the designed circuit.

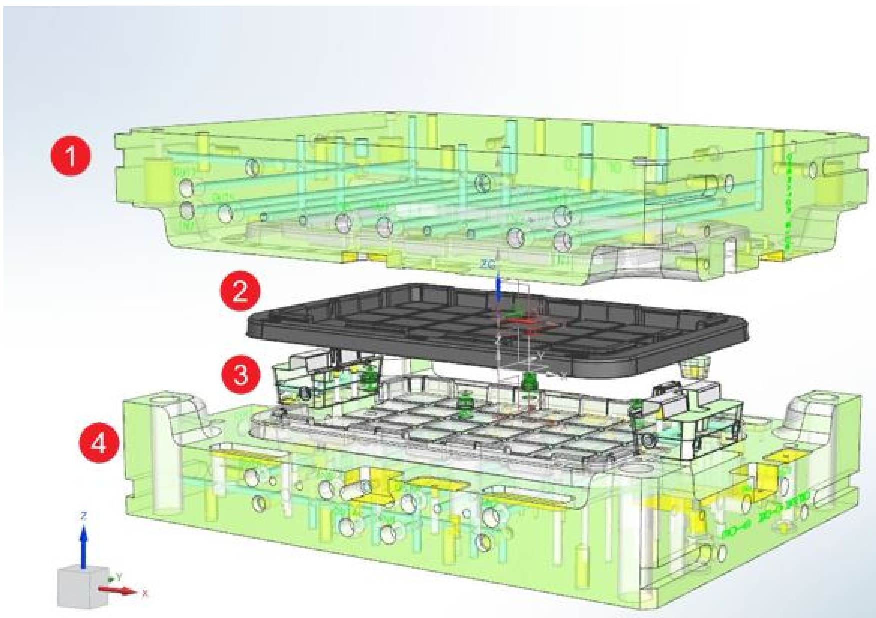


Figure 5.1: Simplified mold B: 1) Cavity, 2) Plastic part, 3) Core inserts, 4)Core

The mold uses Steel P20 and the inserts material is ”MoldMax”, which is an alloy that meets some of the combination as strength, thermal conductivity and good polish ability enabling a shorter cycle time and improved plastic molding part quality.The properties of this material are listed on Table 5.1. For the part, the plastic used is Polypropylene, and its properties are listed in Table 5.2.

Table 5.1: Material properties of MoldMax

Property	*Value*
Density $[kg/m^3]$	8.35
Specific Heat$[J/(Kg*K)]$	380
Thermal Conductivity $[W/m*K]$	104

Table 5.2: Material properties of Polypropylene

Property	*Value*
Density $[kg/m^3]$	905
Specific Heat$[J/(Kg * K)]$	1800
Melting Temperature $[K]$	473.15

5.2.1 Boundary conditions, initial conditions and source term

Boundary Conditions and Initial Conditions

- Wall: Adiabatic and non-slip.

- Initial conditions: The pressure and temperature are set up as the ambient pressure and temperature. The velocity is set to 0 m/s. The initial temperature of the solid is set at 55 ° C.

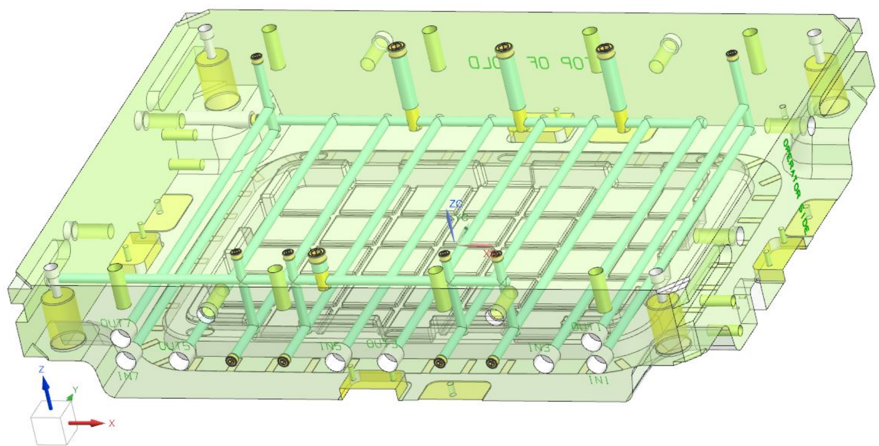
- Inlet: The volume flow rate used is 12 l/min with a fluid temperature of 16 ° C. This boundary condition is the same for all the inlets at the cavity, core and inserts. The turbulence parameters used in this case are the turbulent energy and turbulent dissipation with the default values of the software in order to analyse if the changes of the result are relevant with the default values of 1 J/kg and 1 W/kg, respectively.

- Outlet: Ambient pressure is the outlet boundary condition assigned to all the outlets in the mold (core, cavity and inserts).

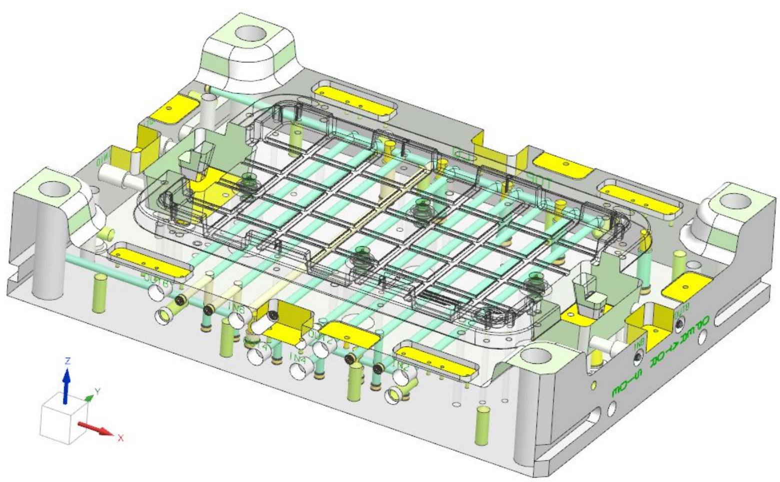
Heat Source

The plastic part in this mold has a melting temperature of 200° C, so, for the heat source, the injection temperature is set as 190°C to avoid melting issues in the simulation. The ejection temperature in this case 102°C. The periodicity on the heat source is also set up for this model, where the heat source is applied on the plastic part, starting at 190°C and is turned off until the average temperature of the plastic part is at the ejection temperature.

Conventional cooling system is set up in the mold B as showed in Figure 5.2. For this mold, four (4) circuits are used in the cavity with some diverters to drive the flow in the expected direction. The circuits are in series and they are parallel between them. For the core, three circuits are used. In each of the inserts area a cooling circuit is also used. Table 5.3 shows the



(a) Cavity



(b) Core

Figure 5.2: Conventional cooling channels of mold B

summary of the simulation set up for the simulation of the cooling system.

Table 5.3: Summary of simulation configuration mold B

Characteristic	*SimulationSetup*
General Settings	• Internal Flow • Considers conduction between solids and convection • Fluid definition :Water
Initial conditions	• Pressure and temperature at ambient conditions • Fluid flow velocity 0 m/s • Solid initial temperature of 55°C
Materials	• Plastic Part: Polypropylene • Mold: Steel P20 • Insert: MoldMax
Boundary Conditions	• Inlet: Volume Flow Rate: 20 l/min / Temperature = 20°C • Outlet: Environment Pressure • Wall conditions: Adiabatic Walls, non-slip
Heat Source	• Injection Temperature: 190°C • Ejection Temperature: 102°C

5.2.2 Grid and Time Step Independence Study

Grid Independence Test

The parameter to be used for the grid independence test is the heat transfer rate at the ejection temperature. Three meshes are used for the grid independence test

Table 5.4 provides an information of the numbers of cells used for each of the three meshes and their respective heat transfer rates at the time of the ejection temperature. Table 5.5 shows the grid independence test result for three different meshes set up. So, the mesh 2 is the one selected to be used in this study since the difference in the results between the meshes 1 and 2 is small enough.

Table 5.4: Number of cells per mesh for mold B

Mesh ID	Number of Cells	Heat Transfer Rate at Ejection Temperature [W]
1	8,269,537	1086.24
2	6,625,542	1092.68
3	4,363,778	1120.77

Table 5.5: Grid Independence test results for mold B

Mesh ID	Relative Error %
2-1	0.59
2-3	2.5

Time Step Independence Test

Using the selected mesh, a time step independence test is also conducted. In this case not just the general time step is considered but the freezing and non freezing periods are also modified to evaluate their influence on the results.

For the time step independence study, the time step used in this case started with a value of $0.01s$, and the changes are made on the freezing and no freezing period to evaluate the sensibility of the results to this feature. For the first case a time step is set up to $0.01s$ with a freezing period of 20 iterations and a no freezing period of 10 iterations. The second case uses the same time step of $0.01s$ and a freezing period of 10 iterations with a no freezing period of 5 iterations. For the last case a time step of $0.005s$ was used with a freezing and no freezing period of 10 and 5 iterations respectively.

Table 5.6 gives the heat transfer rate for each of the cases and Table 5.7 gives the relative error. This time step independence test is performed with a coarser mesh than the generated on the previous study due to the computational time. Since the difference between the time step and the freezing and no freezing periods is small for the parameter of interest, the time step that will be used for this study will be 0.01 s with a freezing period of 10 iterations and no freezing period of 5 iterations.

Table 5.6: Heat Transfer Rate results for time step cases for mold B

$TimeStepCase$	$HeatTransferRate@EjectionTemperature[W]$
Time Step = 0.01 s, Freezing: 20 iter, No-Freezing: 10 iter	1123.95
Time Step = 0.01 s, Freezing: 10 iter, No-Freezing: 5	1120.81
Time Step = 0.005 s, Freezing: 10 iter, No-Freezing: 5	1119.07

Table 5.7: Time step relative errors for mold B

Time Step Case	Relative Error %
Same time Step changes in Freezing periods	0.07
Different time step with same freezing periods	0.15

5.2.3 Results of conventional cooling channels mold B

The cooling time is calculated based on the average temperature of the part from the injection temperature to the ejection temperature. The simulation is conducted for $1000s$. The cooling time is taken when the cycle reaches steady condition.

Using the mesh and the time step selected based on the grid and time step independence test, the first simulation is carried out with a volume flow rate of 20 l/min and a temperature of 20°C. Figure 5.3 shows the plot of the part temperature change during the cooling cycle, where the cooling time for this case is 18.7*s*.

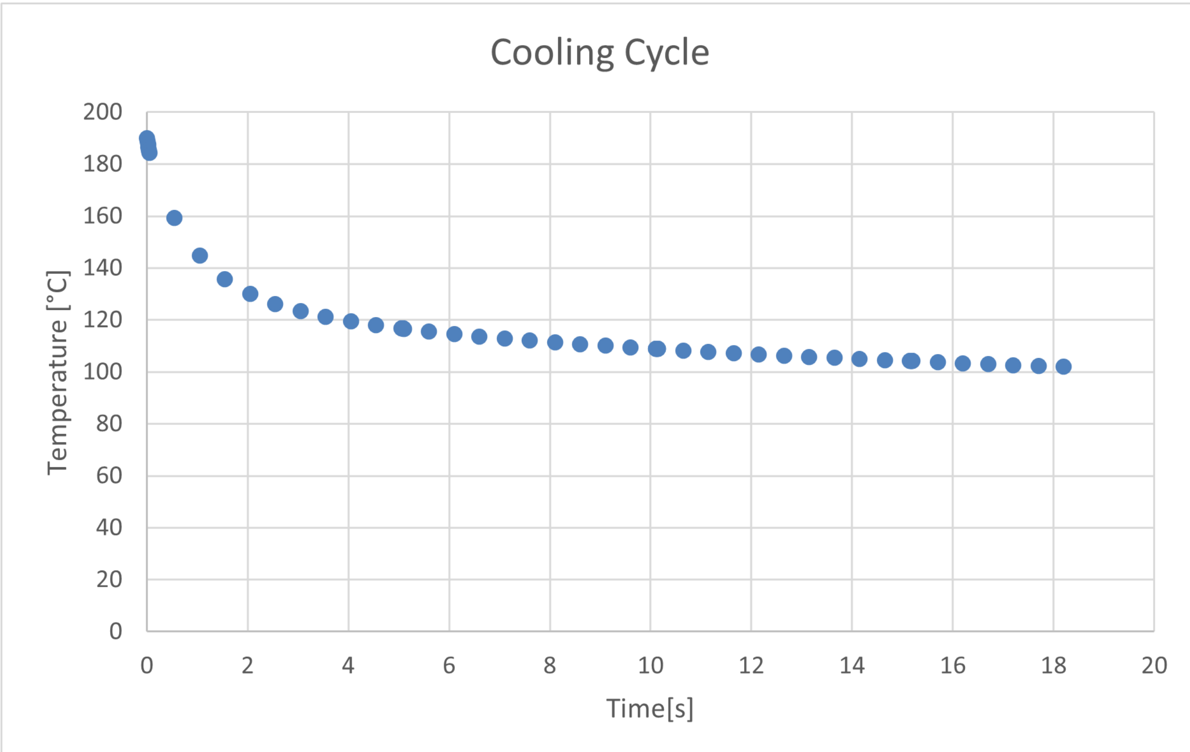
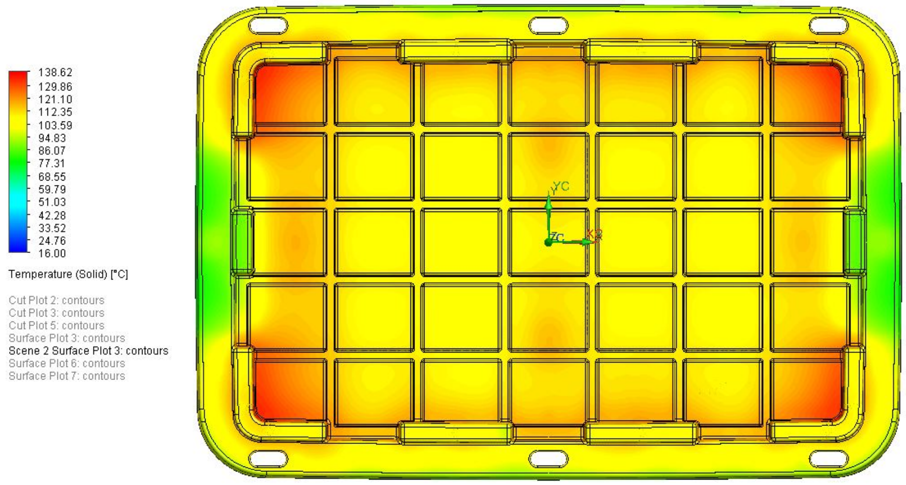


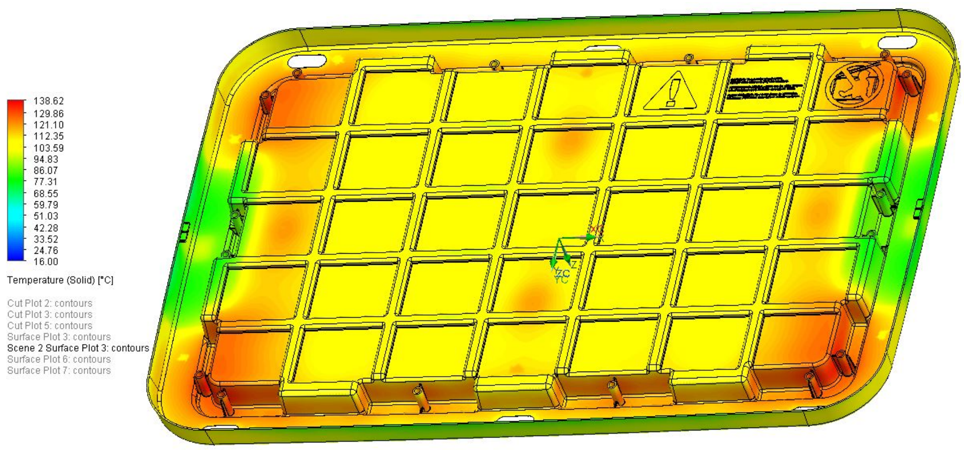
Figure 5.3: Part temperature change in one cooling cycle for an inlet volume flow rate of 20 l/min at 20 °C of the mold B

Figure 5.4 shows the temperature distribution of the part at the ejection time. It is noticed the hot spots in the area of the inside corners. A lower temperature is noticed at the section where the lifters are located since it has a separate circuit from the core and the cavity set up.

The mold cavity and core temperature distributions are shown in Figure 5.5. It can be seen that the temperature distribution in the mold remains uniform in the areas further from the cooling channels. For the area of the molded part, the temperature distribution is similar to the one observed on Figure 5.4. It can be observed that the average temperature of the entire mold is close to the initial temperature of 55°C except for the area where the part surface is located.



(a) Front View



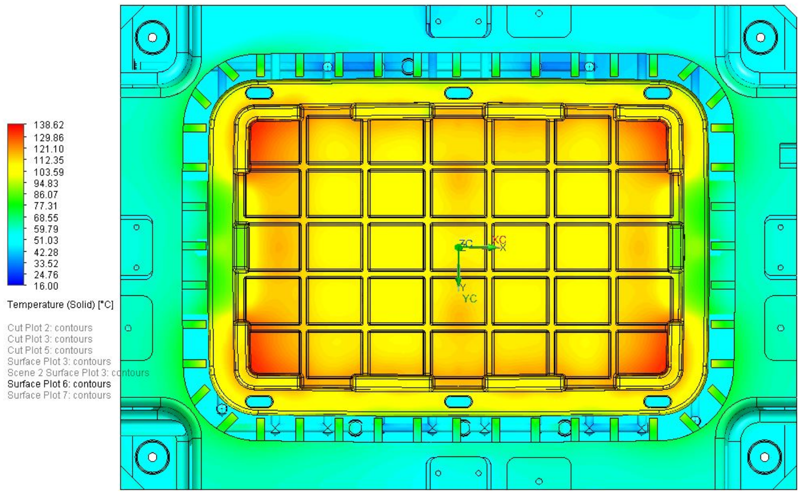
(b) Back View

Figure 5.4: Part temperature distribution for an inlet volume flow rate of 20 l/min at 20°C

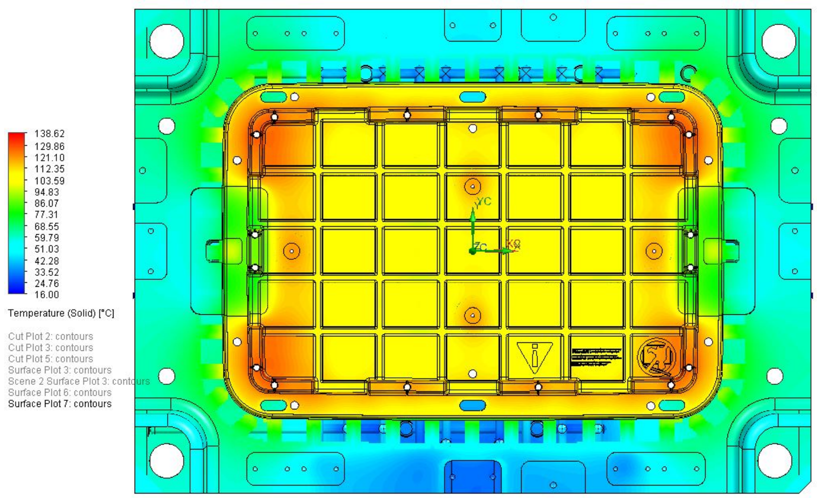
5.2.4 Case: Flow temperature of 60°C

For this case the temperature of the water at the inlet was 60°C based on the process sheet provided by the company. The cycle times obtained from the running tool by the manufacturer are shown on Table 5.8. Therefore, the simulation is also done using the inlet flow temperature of 60°C in order to validate the computational results. The simulation was done with settings given in Table 5.3, except for the inlet temperature.

The calculation of the cooling time is done following the same procedure used for mold 5468. The cooling time calculated from the simulation results is 30.35 s. This time is the addi-



(a) Cavity



(b) Core

Figure 5.5: Mold temperature distribution for an inlet volume flow rate of 20 l/min at 20°C

tion of the pack/hold time and the cool/cure time, since the simulation approach uses activation and deactivation of the source. The difference between the experimental data and the results from the simulation is 0.8 %, which indicates the CFD model used in this study is accurate. Figure 5.6 shows the plot of the temperature changes during the cooling cycle.

Figure 5.7 shows the the part temperature distributions at the ejection time for the flow temperature of 20°C and with 60°C. It is noticed that with the use of a lower inlet temperature, the part can achieve more uniform temperature than with the use of a higher inlet temperature. In both cases the hot spots are located in the same area. Figure 5.8 shows the comparison

Table 5.8: Cycle times from experimental data

CycleStage	*Time* [s]
Open / Close	9.46
Fill Time	2.44
Pack / Hold	12.6
Cool / Cure	18
Cycle	42.5
Total Cool Actual	30.6

between the temperature distributions in the cavity area between the two with different flow temperatures. A higher flow temperature will reduce the heat transfer rate between the mold and the part. So thermal stresses in the mold can be reduced and cracks and failures of the mold can be avoided. The temperature distributions are similar results for both cases, which are uniform except for the areas of the corners of the part. It is noticed that the mold with a flow temperature of 60°C has the temperature around 55°C, which is the initial temperature of the mold, while in the case of 20°C the mold averaged temperature seems higher.

5.2.5 Parametric Study

In order to evaluate the different scenarios that could possibly improve the cooling system in this mold, a parametric study is conducted to evaluate the performance of the cooling system at different operating conditions, i.e. different volume flow rates and flow temperatures, and using different channel diameters. The parametric study will allow to find an optimal setup of the conventional cooling system to improve the cooling system.

Initially changes on the flow characteristics, as a parameter that can be easily changed in the industry while running the machine to improve cooling efficiency. Secondly, changes on the geometry of the cooling channel, specifically on the diameter of the channel. This parametric study will follow the reviewed methodology used by Kapila et al.[7] where diverse flow rates and temperatures are evaluated on an conventional cooling system. A similar study is conducted in this research considering the similarity on the set up of the simulation.

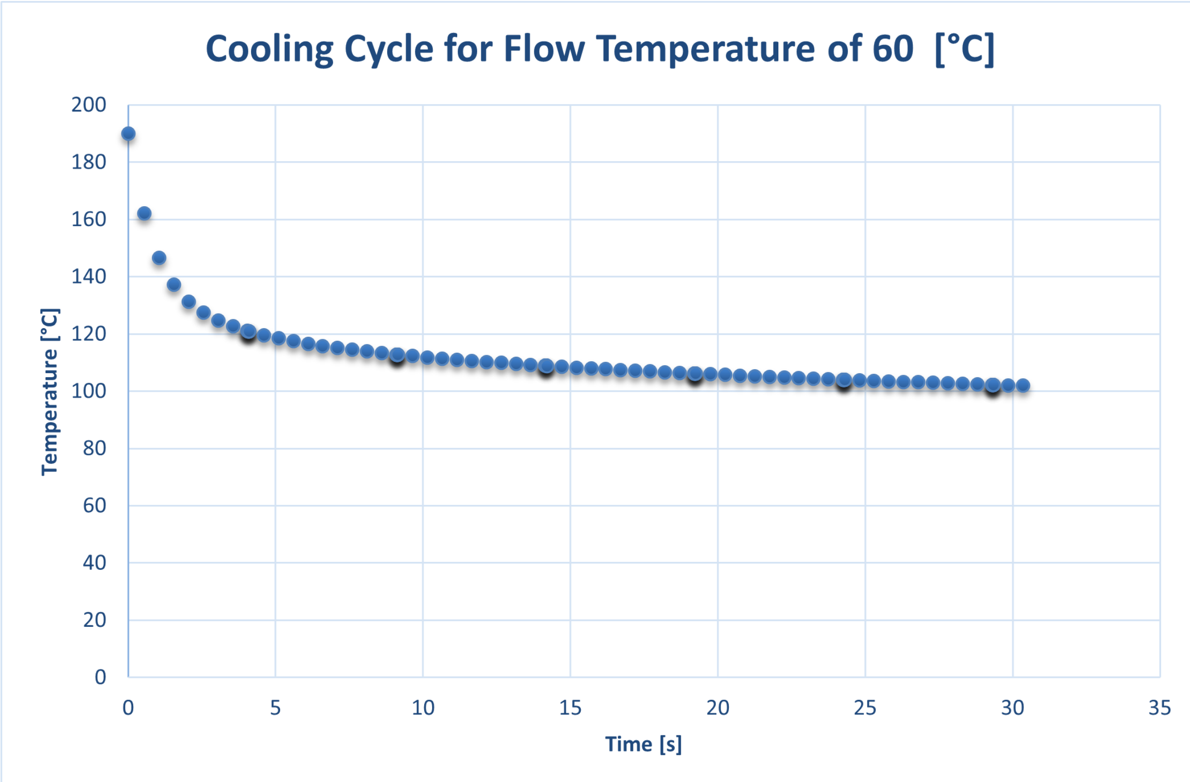
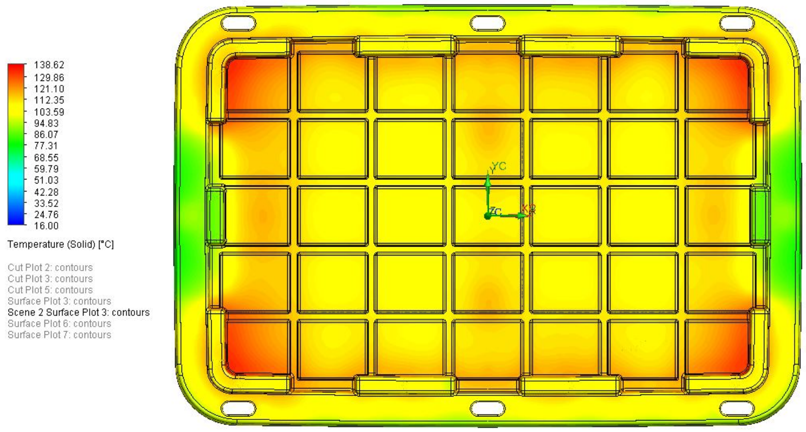
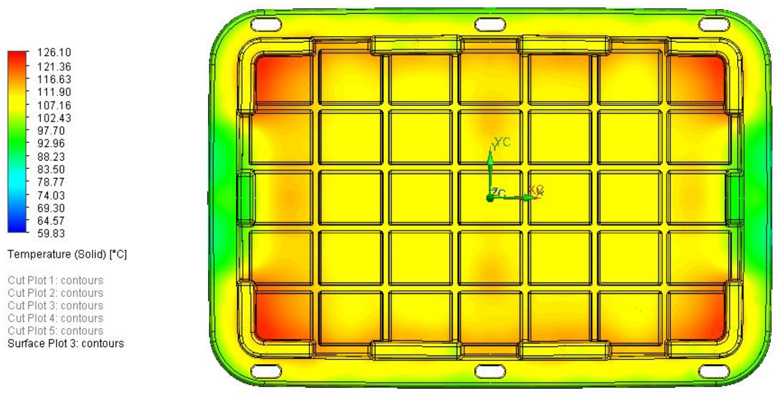


Figure 5.6: Part temperature change in one cooling cycle for an inlet volume flow rate of 20 l/min at 60 °C of the mold B

For the set up of the coolant temperature some studies have been developed to define an optimal temperature for the coolant. Jahan et. al. [24] developed thermo-mechanical analysis of conformal cooling channels, where the study mentioned how the coolant (water) served for two purposes, first to cool down the hot plastic and second to warm up the cavity and core body. So, the water needs to be a few degrees above the temperature of the mold cavity and core, but for this study the temperature of the water was kept a couple degrees higher than the room temperature ($28°C$). Even though it is assumed that the lower the temperature, the shorter the cooling time, if the difference of temperatures between the mold and the coolant is too high, some defects might be perceived in the finishing of the product with a reduced quality on it as well. In this study, it was also demonstrated that the difference between the in and outlet coolant temperatures should be no more than $3°C$, where higher values may indicate high mould surface temperature and should be avoided in order to get a good product quality.



(a) Inlet flow temperature of 20°C



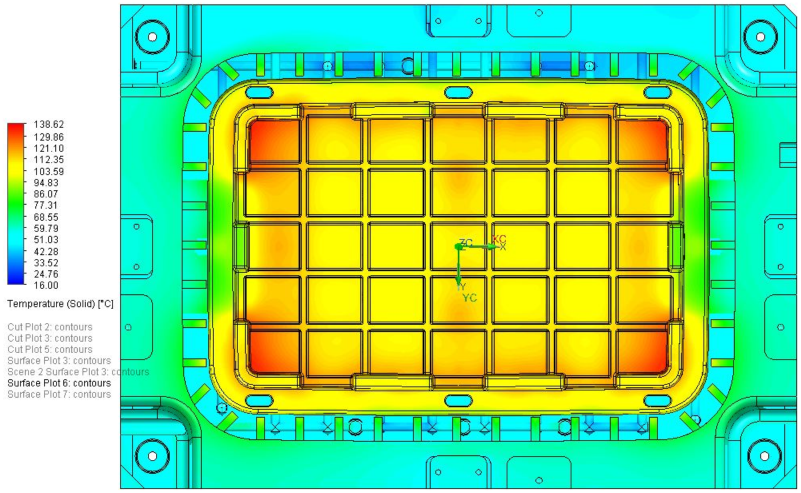
(b) Inlet flow temperature of 60°C

Figure 5.7: Comparison of the part temperature distributions at the ejection time under inlet flow temperatures 20°C and 60°C using conventional cooling system in the mold B

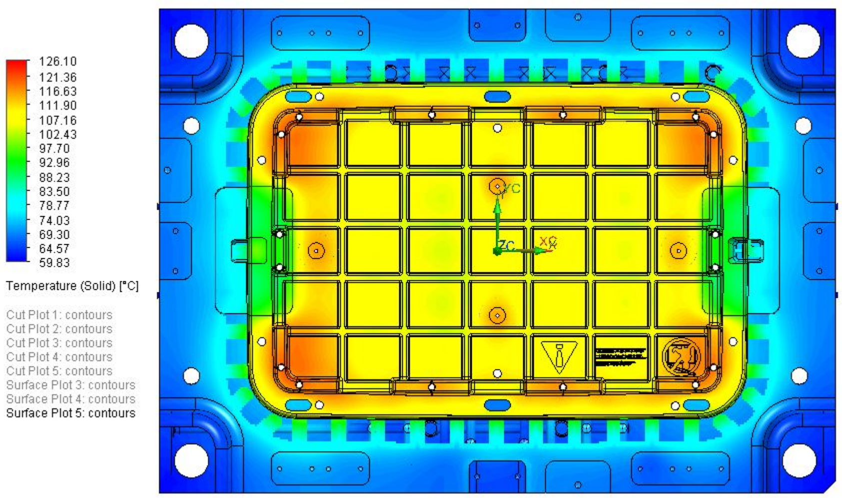
Effect of the Volume Flow Rate

The simulations are carried for different volume flow rates and all other parameters are kept the same as those used in the previous sections. The temperature of the inlet flow is 20°C which is the same for all the different volume flow rates. The volume flow rates used in the parametric study are: 16, 20, 24, 29 and 35 l/min, which are based on the similar study done by Kapila et.al. [7]. The simulation is carried out for 1500s in order to obtain stable result.

The cooling time is calculated using the average temperature in the part from the injection temperature to the ejection temperature. The steady results are obtained after 10-15 cycles of the simulation. Therefore, the time averaged results are taken after 10-15 cycles.



(a) For the case of inlet flow temperature of 20°C



(b) Inlet flow temperature of 60°C

Figure 5.8: Comparison of mold cavity temperature distribution at the ejection time under inlet flow temperatures 20°C and 60°C using conventional cooling system in the mold B

Table 5.9 shows a summary of the results obtained from different volume flow rates for the cooling time and the heat transfer rate at the ejection time of the cycle. It is possible to notice that an increase in the flow rate reduces the cooling time, until certain point since after the flow rate of 29 l/min the cooling time increases slightly. This behaviour is expected since the higher flow velocity, the more effective is the heat transfer between the fluid and the surface of contact. Therefore an increase in the flow rate increases the heat transfer rate as well. So the flow rate of 29 l/min is the optimal flow rate for this model.

Figure 5.9 shows the variation of the cooling time for each flow rate. The cooling time is between the time when the part is at the ejection temperature (102 °C) and when the part is at the injection temperature (190 °C). For the different flow rates, the cooling time is calculated from the results after 15 cycles since the results are less fluctuating after 15 cycles. The areas where the cooling time is around 8s is due to the the freezing and no freezing periods, since the software freeze the results for a certain number of iterations to reduce computational time of the calculation. This method generates variations on the temperature between the injection and ejection time, resulting into this periods where the cooling time is around 8s.

Table 5.9: Results for different inlet volume flow rate of the conventional cooling channels in the mold B

$VolumeFlowRate[l/min]$	$CoolingTime[s]$	$HeatTransferRate[W]$
12	21.2	931.60
16	19.7	1037.25
20	18.7	1121.94
24	18.2	1169.42
29	17.7	1227.79
35	18.2	1169.12

The difference in the cooling time between the flow rate of 29 l/min and 35 l/min is 0.5 s, which is approximately 3 %. This difference might be related with the freezing and non freezing period set up with a freezing time step of 0.5 s. It is expected that after 29 l/min the cooling time should remain the same if it is not being reduced. The cooling time for the case of 35 l/min is the same as that of 24 l/min

Figures 5.10 and 5.11 show the temperature distributions at the time of ejection for volume flow rates of 12 l/min and 20 l/min, and 29 l/min. The temperature distribution in the part at the ejection time is important since it affects the properties of the finishing in the product.

For the volume flow rate of 20 l/min, hot spots are identified in the corners of the lid, which is expected to happen for all the different volume flow rates since in this area the cooling channels are not close enough to the part due to manufacturing restrictions. The results from all the volume flow rates have the same result of a lower temperature in the area where the

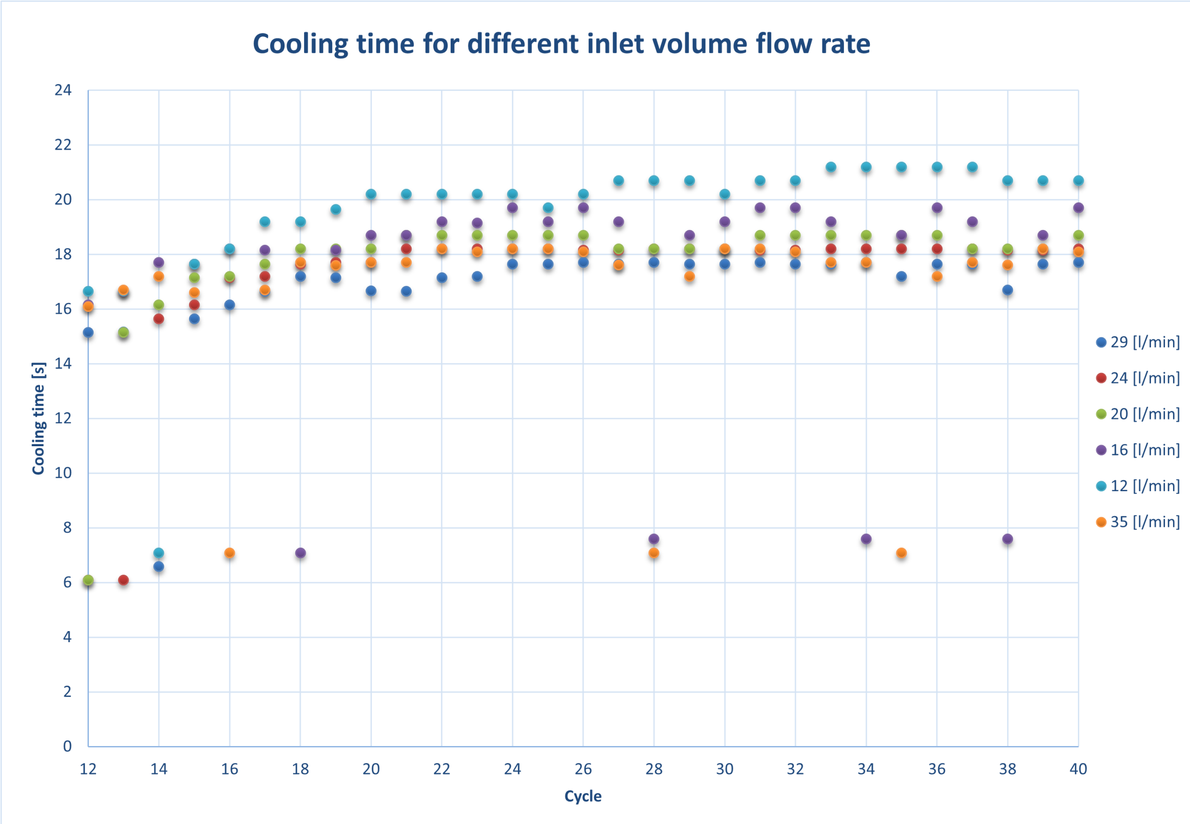
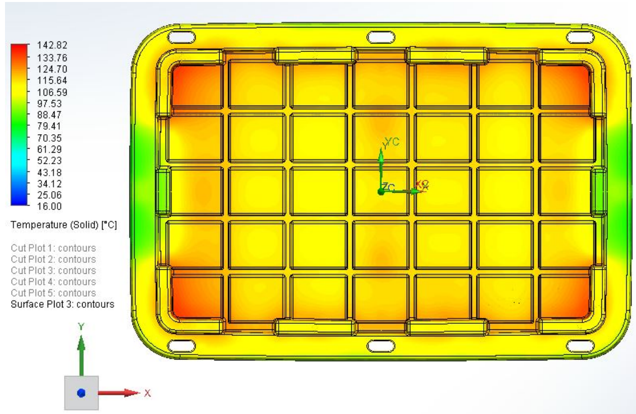


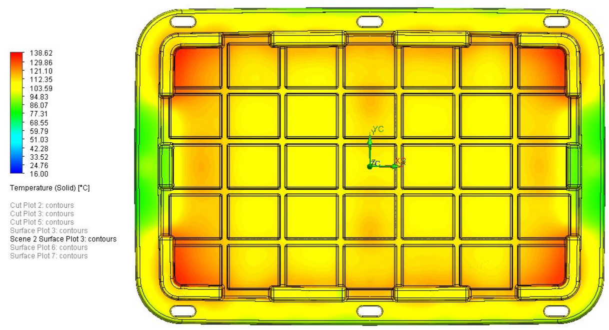
Figure 5.9: Cooling cycles for different inlet volume flow rates at 20°C for conventional cooling channels in the mold B

insert is located. This happens since this section has its own cooling circuit, the temperature of this section reduces faster than the other circuits in the core and the cavity of the molds.

The temperature distribution for the volume flow rate of 12 l/min at the time of ejection does not provides an ideal uniformity, since in the areas close to the edge the temperatures are higher in the part other than the hot spots already identified. For the case of 20 l/min the temperature is more uniformly distributed along the surface. The uniform temperature distribution is more noticeable with the volume flow rate of 29 l/min, in which most of the areas of the part except for the hot spots have approximately the same temperature. Therefore, the volume flow rate of 29 l/min can provide more uniform temperature distribution along the part surface. The effect of the increase of the volume flow rate on the temperature distribution on the part for the case of 35 l/min is the same as the case of a 29 l/min.



(a) Inlet volume flow rate of 12 l/min



(b) Inlet volume flow rate of 20 l/min

Figure 5.10: Comparison of the part temperature distribution under different inlet volume flow rate for the conventional cooling channels of mold B

Effects of the Flow Temperature

To evaluate the effect of the flow temperature on the cooling process, simulations are done using different inlet flow temperatures. The volume flow rate is set to be 20 l/min which is same for all the cases.

Based on previous studies on conventional cooling channels, most of them used temperatures of the water between 20-30 ° C. For this study the temperatures of 16, 20, 25, and 29 ° C are used. The use of 16°C is to analyse the effect of the temperatures lower than 20 ° C and the behaviour of mold cooling system since none of the previous studies reviewed used this temperature.

Figure 5.12 shows the result of the cooling time for all the cycles. Where it is possible to notice similar fluctuations on the cooling time as it is shown in Figure 5.9.

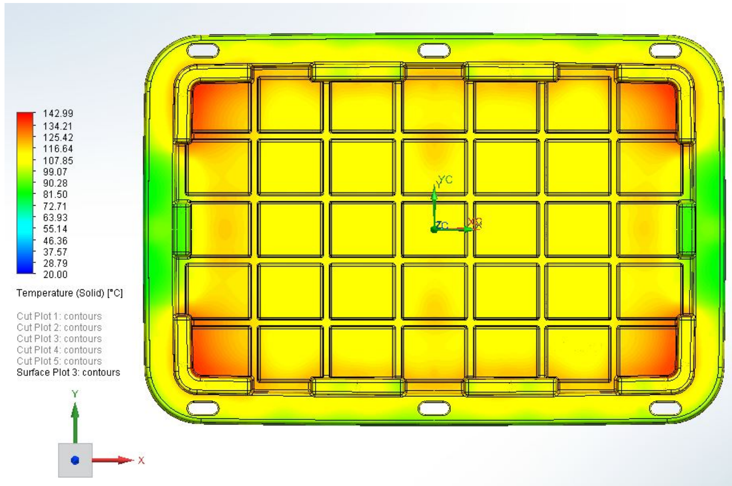


Figure 5.11: Part temperature distribution for an inlet volume flow rate of 29 l/min

Table 5.10 gives the cooling time and the heat transfer rate obtained at the ejection temperature for different flow temperatures. In this table, it is noticed that the flow temperature of 20°C provides a higher heat transfer rate at the ejection temperature. The heat transfer rate is reduced by 3% when the temperature of the flow is decreased from 20°C to 16°C and the heat transfer rate is reduced by 6-8% when the temperature is increased from 20°C to 25°C and 30°C, respectively. Using the heat transfer rate as a parameter of reference, the flow temperature of 20°C still provides the most optimal temperature.

Table 5.10: Results for different inlet flow temperature with a volume flow rate of 20 l/min the conventional cooling channels in the mold B

$FlowTemperature[^\circ C]$	$CoolingTime[s]$	$HeatTransferRate[W]$
16	19.2	1033.89
20	18.7	1121.94
25	19.7	995.40
30	21.7	925.41

In the industry it is common that the flow temperature of 25°C or close to 20°C is used. With the use of 20°C the reduction of the cooling time is around $1s$ for this mold, which for this type of application can have a considerable impact on the part production.

From this set of simulations, it can be concluded that the most effective flow temperature

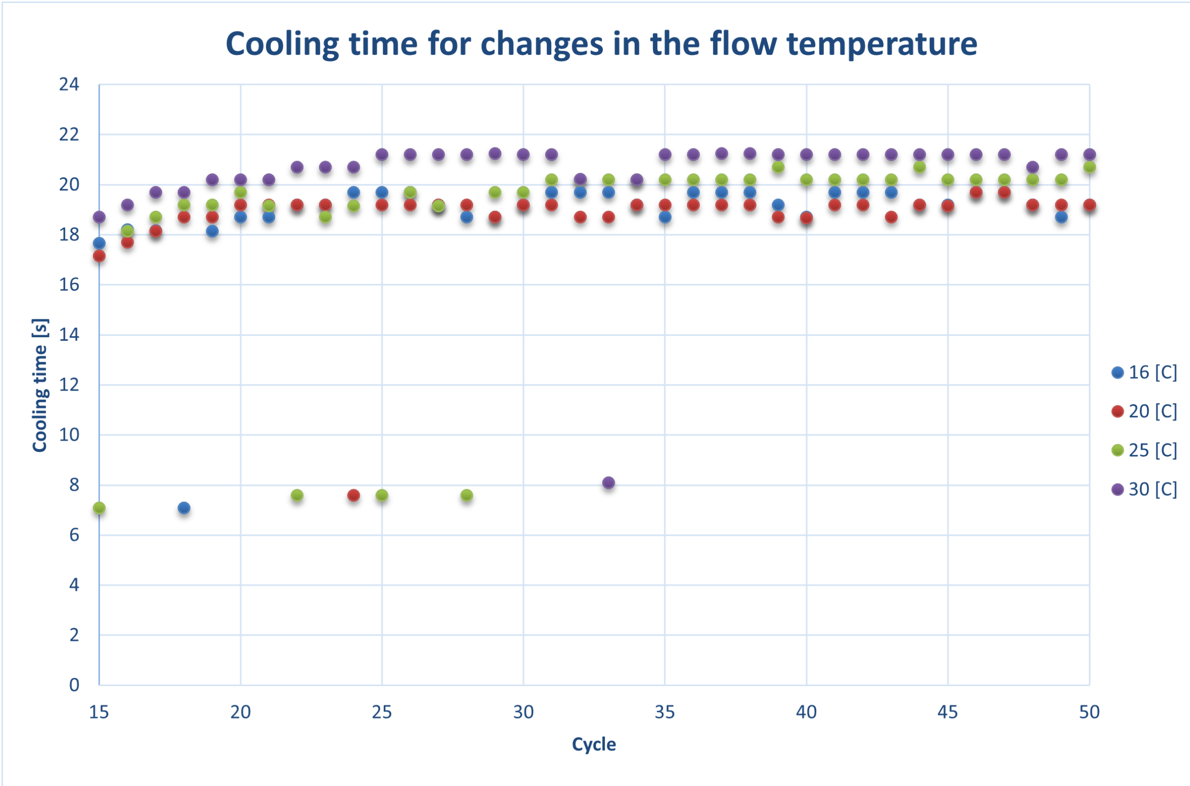


Figure 5.12: Cooling cycles for different inlet flow temperature with a volume flow rate of $20 l/min$ for conventional cooling channels in the mold B

for the flow rate of 20 l/min is 20°C and any increase or decrease in the flow temperature will increase the cooling time of the part.

Comparing the results obtained between the different flow rates and flow temperatures, the increase on the volume flow rate has a higher impact on the reduction of the cooling time. This can be noticed with the increase on the heat transfer rate since the heat transfer to a certain point, which in this case is the volume flow rate of 29 l/min.

Effects of the Diameter of the Cooling Channels

To evaluate the influence of the diameter on the cooling channel performance, 3 different diameters are used in the current mold. The diameter of the current cooling channels is $15mm$,

The correlations for the design of the cooling channels based on the thickness of the part were given by Meyer [16], as shown in Table 2.1. Since the thickness of the part used in this study is 1.68 *mm*, the recommended channel diameter is between 8*mm* and 10 *mm* based on Table 2.1. Since the current channel is 15 *mm*, the diameters of 12 *mm* and 10 *mm* are used in this parametric study which is beyond the recommended values by Meyer.

The simulations are performed using the inlet flow rate of 20 l/min at 20°C. Table 5.11 provides the cooling time obtained for each of the diameter used in the study.

Figure 5.13 shows results of the cooling time using cooling channels with different diameters. It can be seen that larger diameter channel provides a lower cooling time, which is expected since the heat transfer area increases if the channel diameter increases. The cooling time is reduced by 10 % with an increase in the diameter by only 3mm.

Table 5.11: Results for different cooling channel diameters

$ChannelDiameter[mm]$	$CoolingTime[s]$
10	21.2
12	20.7
15	18.2

Between the channels with diameters of 12 and 10 mm, the difference in the cooling time is only 0.5 s and in the plot shows the cooling time fluctuations for the different cycles where the cooling time is the same for some cycles and it is different for other cycles. This fluctuation might be due to the freezing and no freezing periods since the time step of this freezing stages is 0.5 *s*.

5.2.6 Optimal conventional cooling channel set up for mold B

In order to obtain the optimal cooling channel set up for this mold using the conventional cooling channel based on the parametric study results, the following observations are considered:

- The volume flow rate as 29 l/min provides the best cooling time for this mold. Using this volume flow rate might increase the cost of production of the part since the process will need a higher pumping system to provide the volume flow rate to this case.

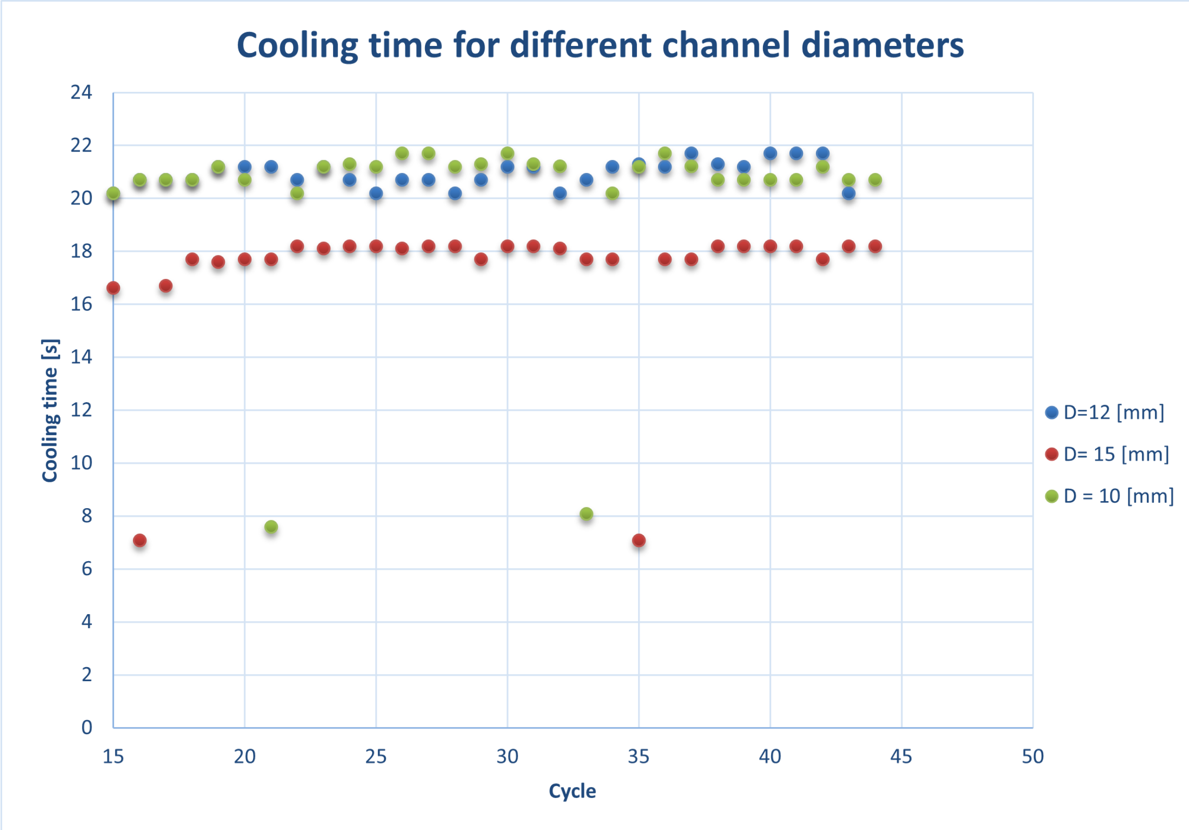


Figure 5.13: Cooling cycles for different channel diameters using inlet volume flow rate of 20 l/min at 20°C for conventional cooling channels in the mold B

- It is important to conduct the analysis on different volume flow rates since it allows the manufacturer to obtain the range on where the designed cooling system can work on to have a better cooling efficiency. In this case, it is possible to know the best case scenario in terms of volume flow rate that allows a reduction between 1-2 seconds in the cooling time compared with other cases. This analysis can also provide the temperature distribution in the product to avoid deformations, warpage, or any other defects that might be generated due to not having a uniform temperature distribution in the product.

- A reduction in the temperature of the inlet flow also results in an increase in the cost of production since the coolant will have to be cooled prior its use on the cooling channels and this will require energy and it will be an added cost. Therefore, it is necessary to know the ranges where the fluid temperature should be used without compromising the efficiency of the cooling system.

- Larger diameter channels provide less cooling time.
- For this conventional cooling system in the mold B, the most optimal operating condition is the volume flow rate of 29 l/min and inlet flow temperature of 20°C to have a lower cooling time and better temperature distribution. The selection of the operating parameters also depends on the cost of production and improvement on the tool that will also allow improve the production cost.

5.3 Conformal Cooling Channel for Mold B

In this section a design an analysis of a conformal cooling channel using the mold of the model B is done. The materials used for all the components involved is the same as the original mold. With the results of this analysis it is possible to compare it with the results obtained previously with conventional cooling channels. In order to evaluate the use of conformal cooling channels and its improvement on the cooling time and temperature distribution in the part, a conformal cooling system is designed using the current model. From the reduced CAD geometry, the conventional cooling channels are removed and replaced them with conformal cooling channels.

5.3.1 Description of the designed conformal cooling system

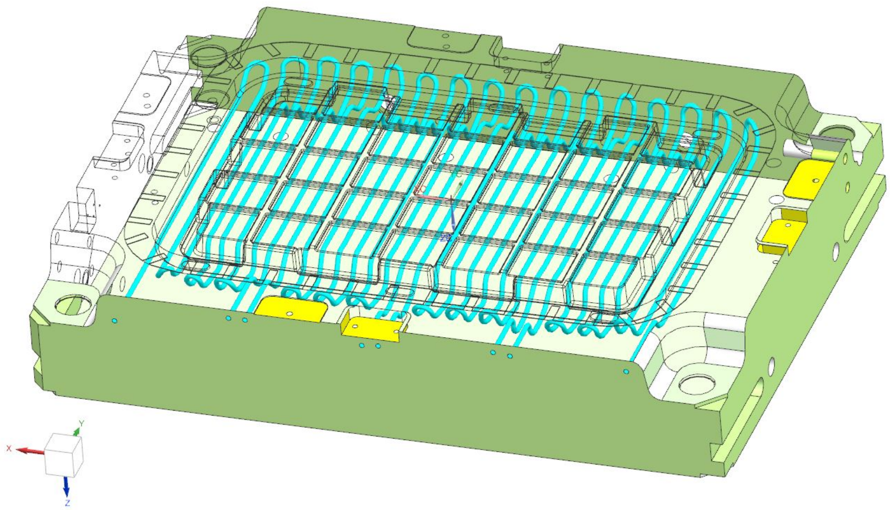
In order to improve the cooling process, a conformal cooling system is proposed in this study for mold B. To start the design of the conformal cooling channels, first is to select the cooling channel diameter based on the correlations in Table 2.1. Since the part thickness in this mold is 1.6 mm, the recommended cooling channel diameter is between 4 mm and 8 mm. The diameter selected is 8 mm since a larger diameter channel will provide a lower cooling time. For this diameter, the pitch of the channels can be between 16 mm and 24 mm, where 24 mm is selected. The length from the centreline of the diameter is recommended to be between 12 mm and 16 mm, where 16mm is selected. Considering the shape of the part as well as the location of ejector pins, and other accessories in the mold in the core and cavity non related with the cooling system, a longitudinal cooling channel direction is selected for this mold.

Based on the results of hot spots found in the conventional cooling channel of the mold,it is possible to choose the number of circuits in parallel and in series for the proposed conformal cooling system. The selection of the number of circuits is based on the study by Kuo et al.[20] where the analysis of the layout of the cooling system was done with the use of different inlets of multiple circuits in the conformal cooling channels. Once the position and orientation of the channels are selected, the connection in series and the position in parallel can be determined.

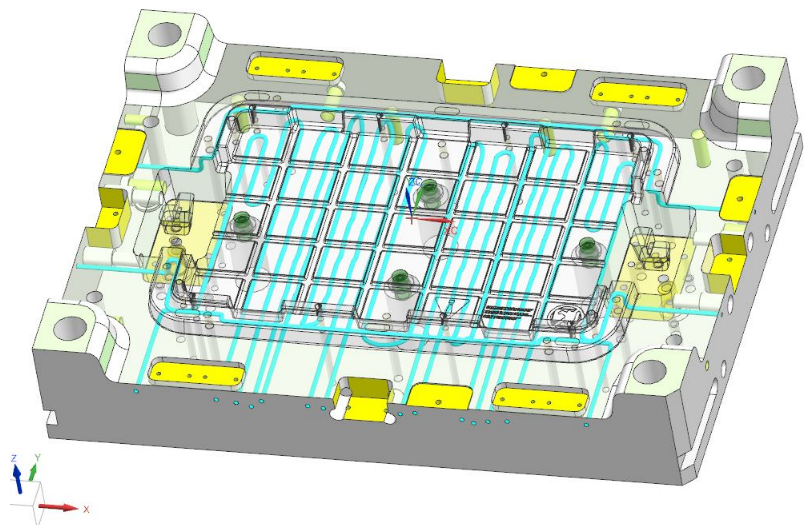
5.3.2 Simulation of the designed conformal cooling system

After the conformal cooling system is designed simulations are performed to evaluate the performance of this new cooling system and compare it with the conventional cooling channel of the same mold. The simulation is first conducted using the same parameters defined on Table 5.3 with the inlet flow conditions, a volume flow rate of 16 l/min and a flow temperature of 25°C. The grid and time step setup used are similar as those used in the conventional cooling system since the product part is the same.

Figure 5.14 gives the proposed conformal cooling system on the cavity and the core sections. There are four (4) circuits designed in series in the cavity, which are parallel between each of them. The cooling channels conform the surface of the part. The loops also conform the area of the edges of the lid. It is noticed some constraints in the core section for the design of the conformal cooling compared with the cavity area, since the core has the geometry where the ejector pins are located for the ejection of the part when it is solidify up to the temperature of ejection. Therefore, more circuits are needed between channels in the core section. There are 7 circuits along the Y-axis. For the section along the edge in the X-direction, another two circuits are developed longitudinal to the part. For the cooling channels located in the inserts, there is no change from the conventional cooling design.



(a) Cavity



(b) Core

Figure 5.14: Designed conformal cooling system for cavity and core of mold B

5.3.3 Results of the simulations in the conformal cooling system

Figure 5.15 shows a plot of the cooling cycle for the newly designed conformal cooling system, where the total cooling time for this mold B is of 10.1 s.

Figure 5.16 shows the temperature distribution in the part. It is noticed that the designed

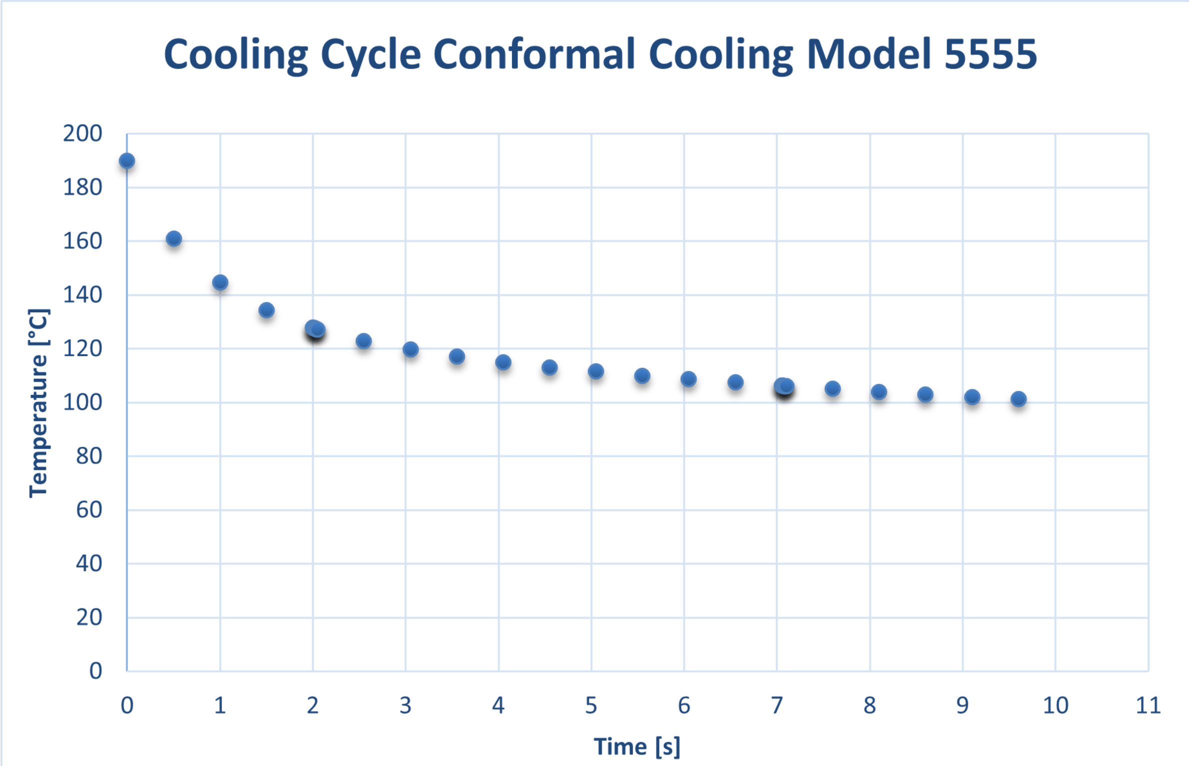
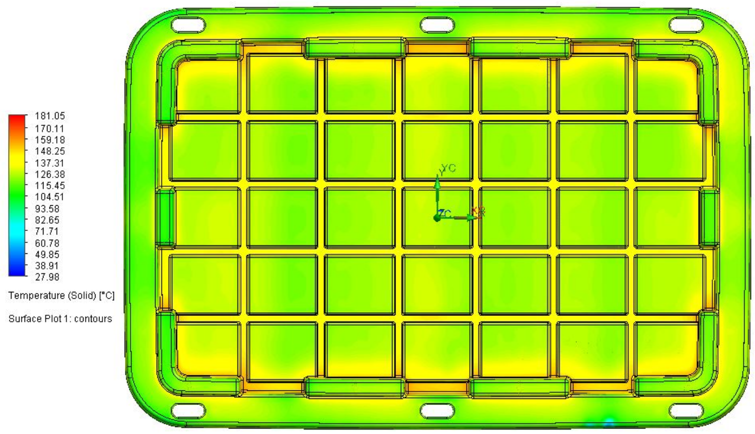


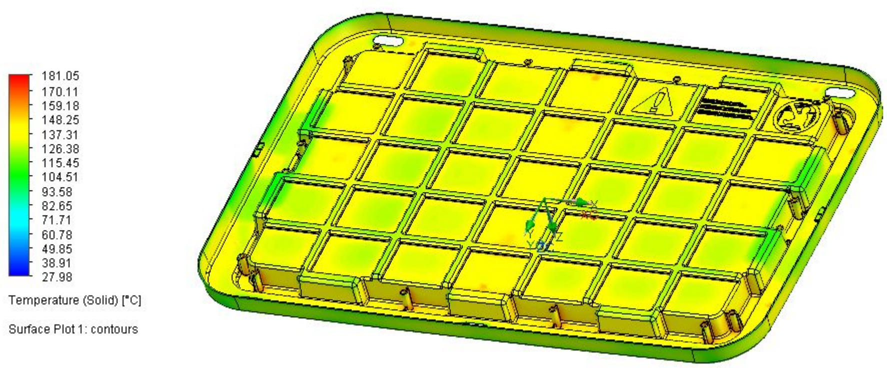
Figure 5.15: Part temperature change of one cooling cycle using conformal cooling channels in the mold B

conformal cooling system gives more uniform temperature distribution along the part at the time of ejection than that from the conventional cooling system as shown in figure 5.4. All the temperature distributions are taken at the time of the ejection.

Since it is important to maintain the mold temperature distribution uniform, to complete the evaluation of the cooling system, the temperature distributions of the mold cavity and core are given in Figure 5.17. The results show that temperature distribution in the cavity and core close to the part are similar to those obtained in the part as expected. For the rest of the mold, there are only slight changes in the temperatures along the mold in the areas further from the cooling lines. The cooling line temperature distribution is shown in Figure 5.18, where the channels in the cavity and the core are both visible. The temperature in the areas where the temperature is lower is close to 25 °C and the temperature is higher at the outlets. This channel disposition of the inlets and outlets allows the sections of the inlet of one of the circuits next



(a) Top View



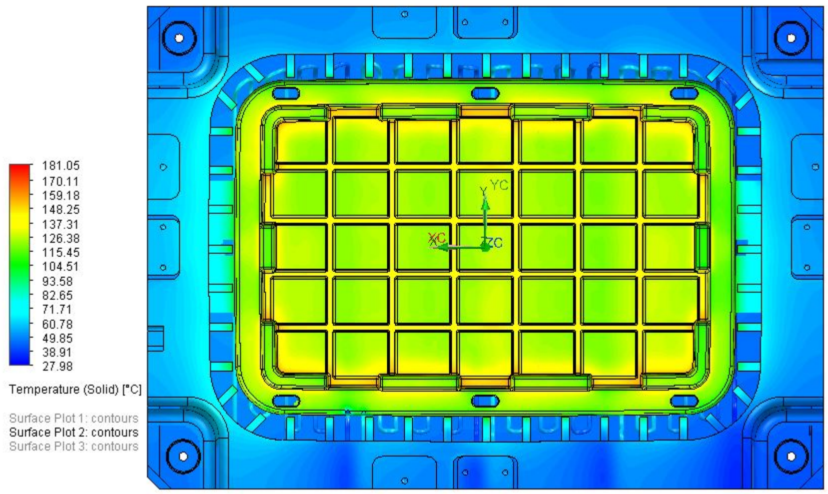
(b) Isometric back view

Figure 5.16: Part temperature distribution for an inlet volume flow rate of 16 l/min at 25°C using conformal cooling channels in mold B

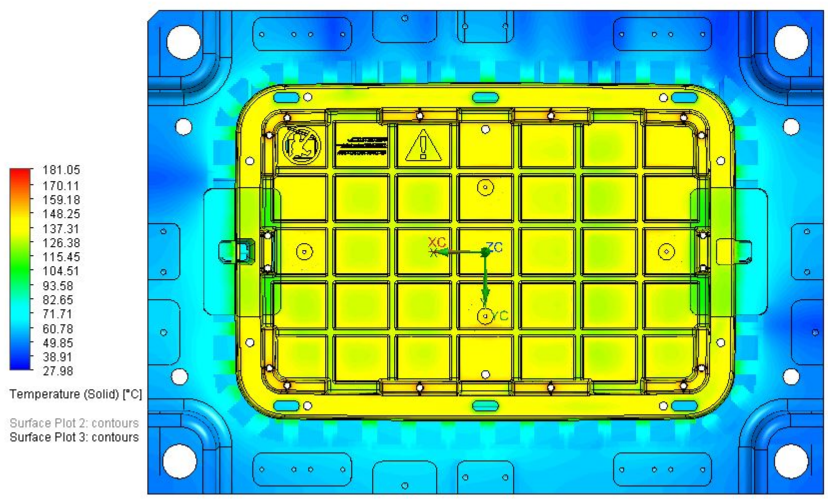
to the outlet of the following one. This configuration balanced the transfer of energy between these two channels and the part temperature to have more uniform temperature distribution in the part.

Based on previous studies with conformal cooling channels, the range of volume flow rate is between 8 l/min and 16 l/min. Therefore, the volume flow rates of 16 l/min, 12 l/min and 8 l/min are used to evaluate the performance of the conformal cooling system of the mold B.

The average cooling time obtained from each of the cases is shown in Table 5.12 and Figure 5.19 shows the cycle times for different volume flow rates. The cooling time is stabilized after 15 cycles. So, the results shown in Table 5.12 are the average results after 15 cycles. The fluctuations on the cooling time are also observed for all flow rates. The freezing and no



(a) Cavity



(b) Core

Figure 5.17: Temperature distributions of the cavity and core for the mold B using conformal cooling channels

freezing period affects accuracy of the cooling time and there is a difference in the cooling time of of ± 0.5 [s], which is the time step of the freezing and non freezing period.

The results show an increase in the volume flow rate reduces the cooling time. Comparing the use of the volume flow rate of 16 l/min where the cooling time obtained is 10.1 s with the volume flow rate of 12 l/min where the cooling time is 12.15 s, a reduction of 20% of the cooling time is obtained. Comparing the cooling time for the volume flow rate of 16 l/min with the flow rate of 10 l/min, a 45% of reduction in the cooling time is obtained.

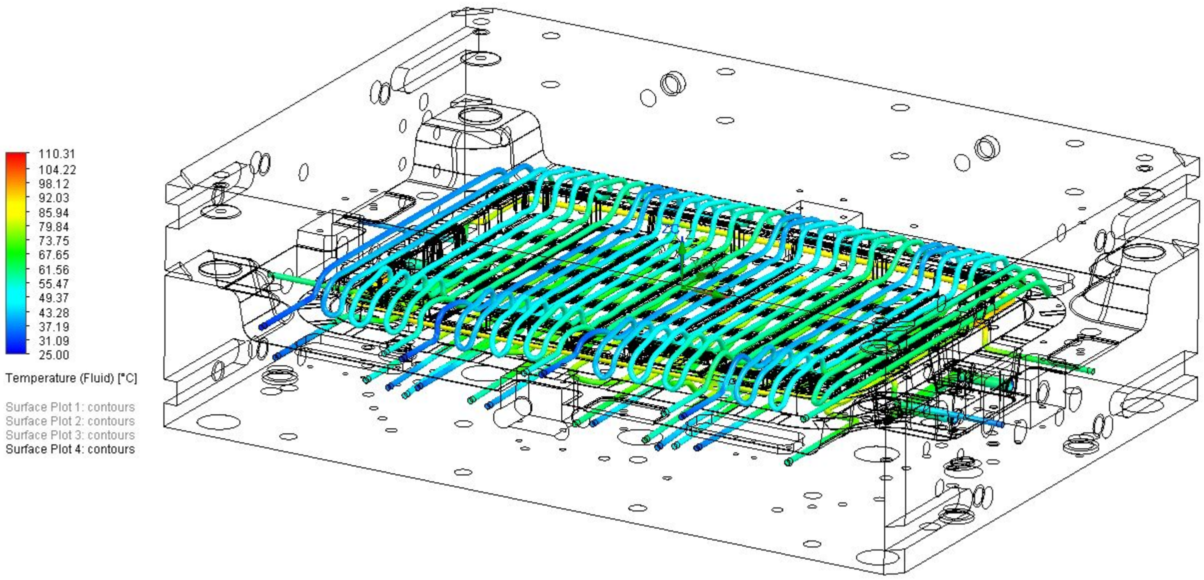


Figure 5.18: Temperature distribution of the conformal cooling channels

Table 5.12: Cooling times for different volume flow rates using conformal cooling channels in the mold B

$VolumeFlowRate[l/min]$	$CoolingTime[s]$
16	10.1
12	12.15
8	14.65

5.4 Comparison between Conventional and Conformal Cooling Channels for Model B

After generate simulations and analysis on each of the cooling systems, conventional and conformal, a set of comparisons have to be performed to confirm the usage of conformal cooling system as the best approach for Plastic Injection Mold tools.

The principal value to be compared is the cooling time, in order to evaluate which of the cooling systems provides a reduction on the cooling time and hence a reduction in the cycle time. The comparison has to be done with all the cases generated for conventional cooling system, including changes in the flow characteristics as well as the channel diameter. Also

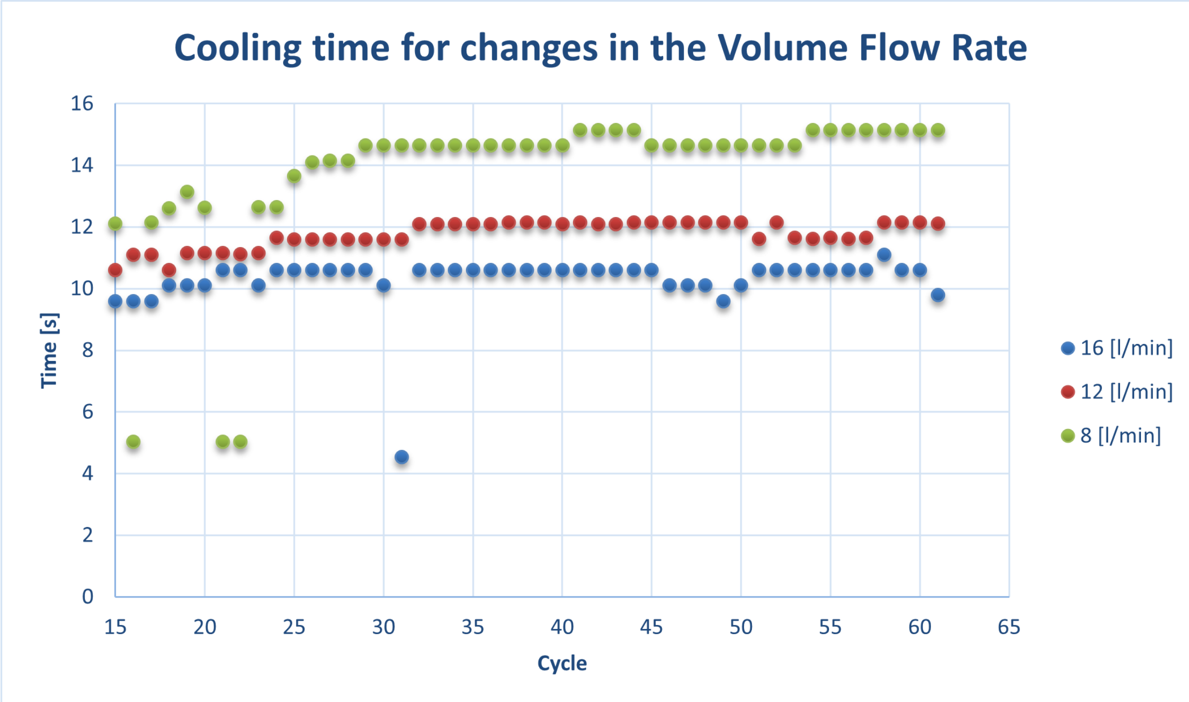


Figure 5.19: Cooling cycles for different inlet volume flow rate with a temperature of 25°C using conformal cooling channels in mold B

considering the cases of the flow changes for the analysis of the conformal cooling system. Based on the approach of the software to calculate this parameters, analyse which of the cases provided a result with better accuracy.

Another parameter to consider for the comparison is the temperature distribution in the molded product. To evaluate the uniformity of the temperature along the part allows the analysis of the quality of the part, and observe which of the cases provided a better temperature distribution. It is necessary to define hot spots in the part and which method provided smaller hot spots and analyse the possible reasons of the different results in each of the cases. These comparisons have to be made in all the cases, this way is possible to considerate all the variations that conventional cooling has to provide prior to move to conformal cooling.

5.4.1 Cooling Time

The cooling time is the principal factor to evaluate the performance of a cooling system. For the case of the mold B with the conventional cooling system the cooling time is 18.2 s with a volume flow rate of 20 l/min and a flow temperature at the inlet of 20°C. Comparing this value with the cooling time of 10.1 s for the conformal cooling at a volume flow rate of 16 [l/min] with a temperature of 25°C, a reduction of approximately 45 % of the cooling time is achieved. This reduction in the cooling time for the conformal cooling system is achieved with a reduced volume flow rate from 20 l/min to 16 l/min, which will also reduce cost of production. Comparing the case of the conventional cooling channel with the use of 8 l/min in the conformal cooling channel, a reduction in the cooling time of approximately 24% is obtained.

From the case where the conventional cooling channel used a volume flow rate of 16 l/min with a temperature of 20°C, a cooling time is 19.7 s. Comparing this case with the cooling time of 10.1 s obtained using the same volume flow rate but with the temperature of 25°C on the conformal cooling channel, a reduction is 48%.

Another important case to compare is with the use of a volume flow rate of 29 l/min and flow temperature of 20°C, where the cooling time obtained was 17.7 s(the lowest cooling time possible in the conventional cooling system). A reduction of the 38% of the cooling time is achieved with the use of 16 l/min by the conformal cooling system.

In terms of volume flow rate, an increase of the volume flow rate in a conventional cooling system does not provide a significant reduction of the cooling time as it does with the use of conformal cooling system. It can be observe how the lower volume flow rate of the conformal cooling system provides a considerable decrease of the cooling time.

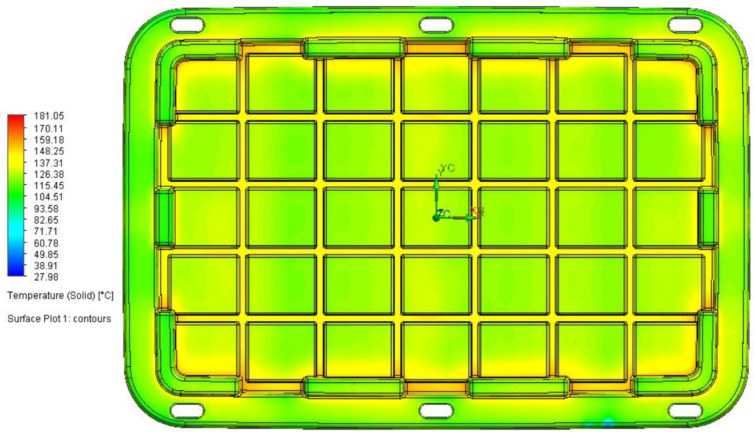
5.4.2 Temperature Distribution

The reduction of the cooling time is not the only factor to evaluate the performance of the cooling system since if the cooling time is too fast, bubbles can be generated, and finishing defects on the product, which compromises the quality of the part. The uniformity of the temperature distribution in the molded area is another important factor to evaluate the performance of the cooling system. Therefore, the temperature distribution need to be evaluated.

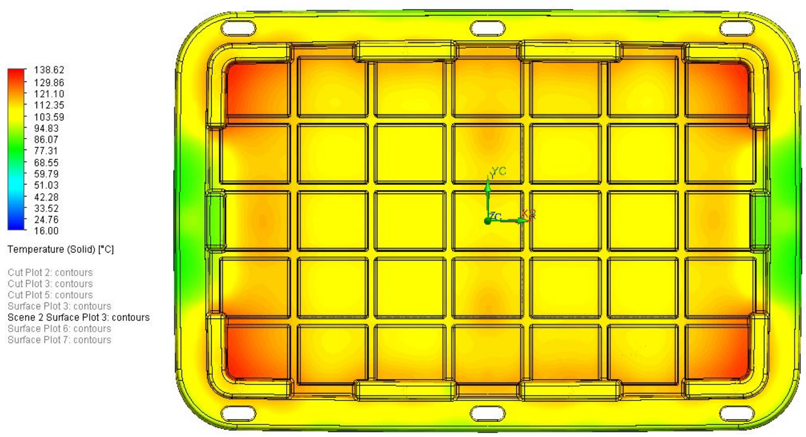
First, the temperature distributions are compared between the case of the conventional cooling system using the inlet flow volume flow rate of 20 l/min with a temperature of 20°C, against the conformal cooling system case that uses inlet volume flow rate of 16 l/min at 25°C as shown on the Figure 5.20. In the conventional cooling system, the part temperature shows the hot spots where the temperature has not reached entirely the ejection temperature of 102°C, since this is based on the average temperature of the entire part. Higher temperatures are found on the inside areas of the lid marked as red spots. At the same ejection temperature, the conformal cooling channel shows a uniform temperature distribution on the entire part without evidence of hot spots. So, it is clear that the conformal cooling channels provide not just a reduction of the cooling time but also a uniform temperature that will provide a better quality to the part.

Figures 5.21 and 5.22 show the comparison of the cavity mold and the cooling channels temperature distributions between the conformal cooling at 16 l/min and 25°C and the conventional cooling at 20 l/min and 20°C. From the cavity mold temperature distribution is noticed that for the conventional system, the temperatures in the rest of the mold have a higher variance between 60 to 80°C, while for the conformal cooling the temperatures of the mold keep lower temperatures between 40-60 °C. Since it is important to keep the mold temperature for this case between 55-70 °C with a temperature as uniform as possible,to avoid thermal stresses in the mold, the conformal system is a better option for the durability of the mold as well. The difference of temperature between the mold area and the parting area is needed to be kept with an small range.

Figure 5.23 shows the comparison of the part temperature distribution between the conformal cooling at 16 l/min and 25°C and the conventional cooling at 29 l/min and 20°C. It is possible to see that even at a lower cooling time for the conventional system, the temperature distribution of the part still shows the hot spots in the same areas as seen in the previous case,



(a) Conformal Cooling system for inlet flow of 16 l/min at 25 °C

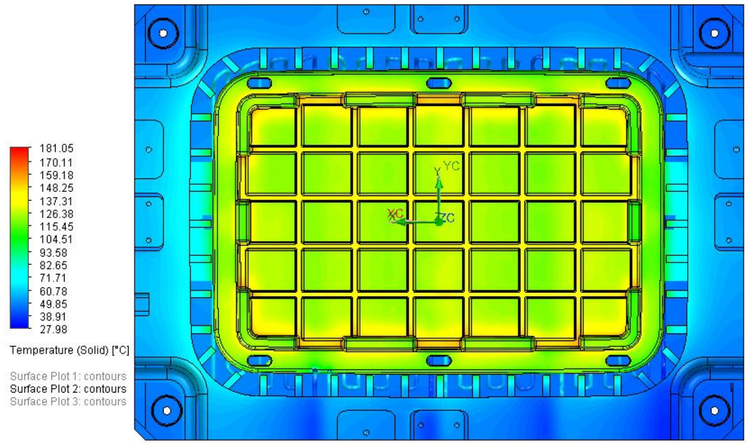


(b) Conventional cooling system for an inlet flow of 20 l/min at 20 °C

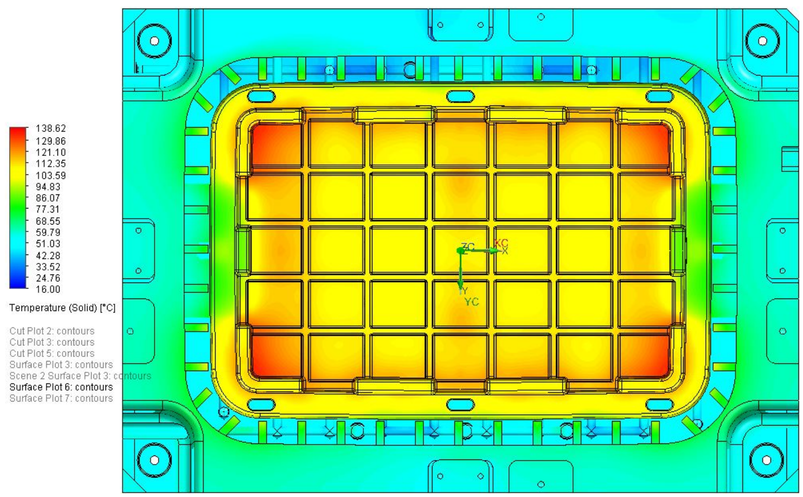
Figure 5.20: Comparison of the part temperature distribution between the conformal cooling system with inlet flow of 16 l/min at 25 °C and the conventional cooling system with an inlet flow of 20 l/min at 20 °C

although the hot spots are smaller. The temperature distribution of the conformal cooling still provides more uniform temperature distribution along the part than the conventional cooling channels.

The conventional system with the inlet volume flow rate of 12 l/min at a temperature of 20°C is compared with the conformal cooling system with 12 l/min and 20°C as shown in Figure 5.24. The part temperatures in the conventional cooling system are higher and less uniform than the cases analyzed using the conformal cooling system. It is noticeable that with conformal cooling systems the appearance of hot spots in the part is reduced and in this case



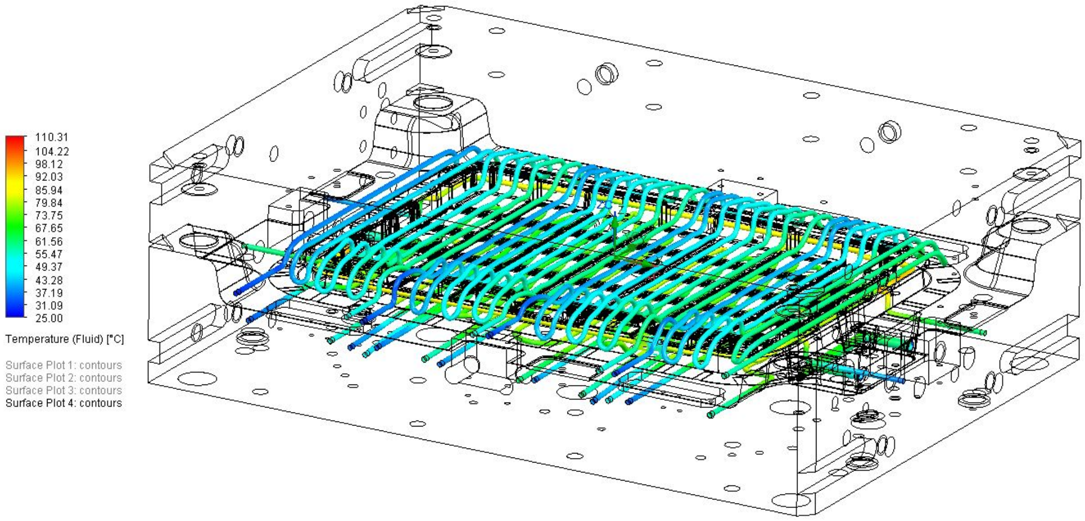
(a) Conformal Cooling system for inlet flow of 16 l/min at 25 °C



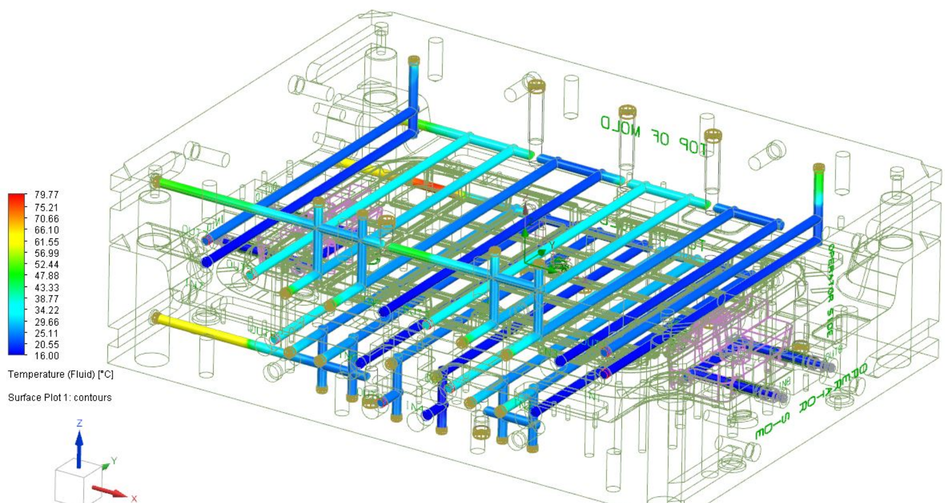
(b) Conventional cooling system for an inlet flow of 20 l/min at 20 °C

Figure 5.21: Comparison of the cavity mold temperature distribution between the conformal cooling system with inlet flow of 16 l/min at 25 °C and the conventional cooling system with an inlet flow of 20 l/min at 20 °C

with a lower flow rate it is more notorious.

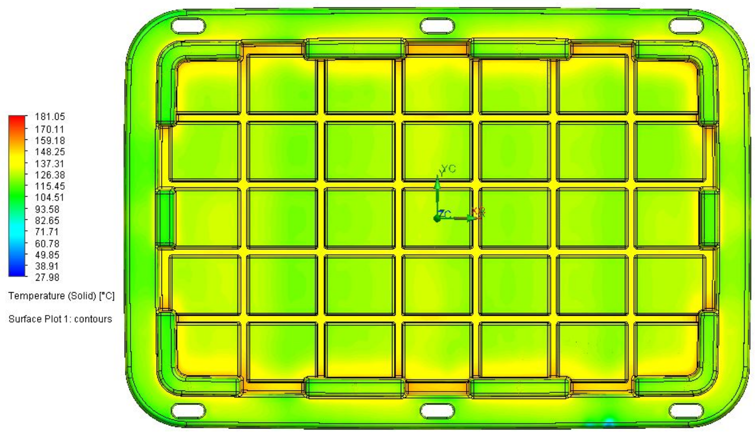


(a) Conformal Cooling system for inlet flow of 16 l/min at 25 °C

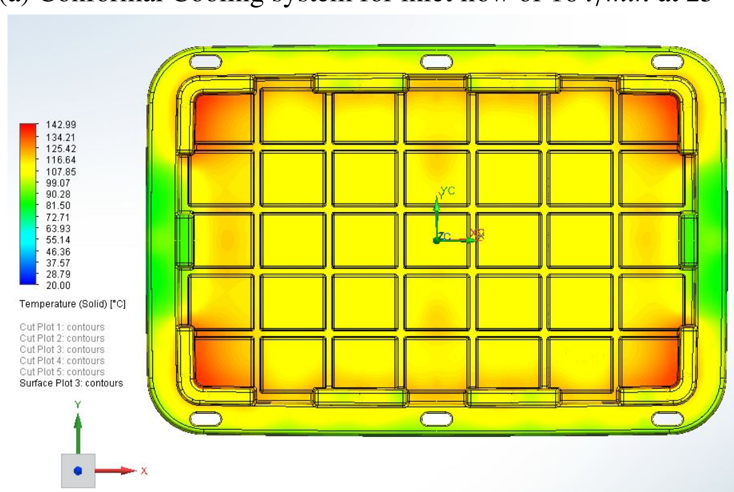


(b) Conventional cooling system for an inlet flow of 20 l/min at 16 °C

Figure 5.22: Comparison of the cooling channels temperature distribution between the conformal cooling system with inlet flow of 16 l/min at 25 °C and the conventional cooling system with an inlet flow of 20 l/min at 16 °C

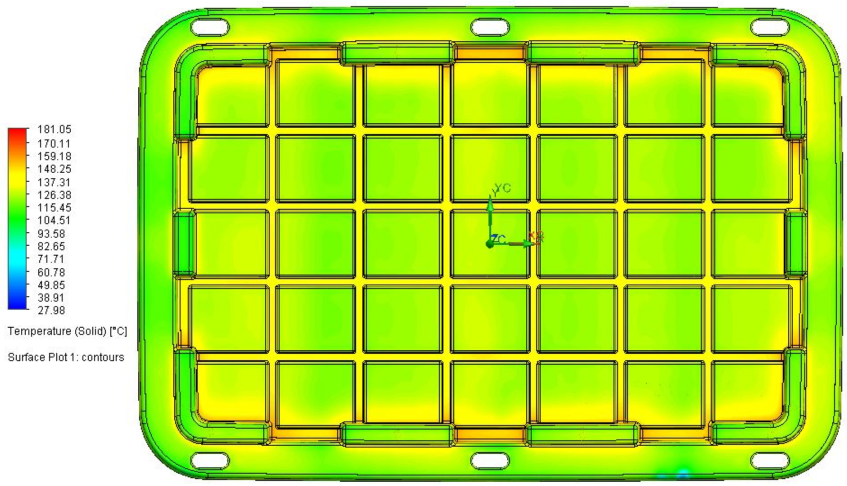


(a) Conformal Cooling system for inlet flow of 16 l/min at 25 °C

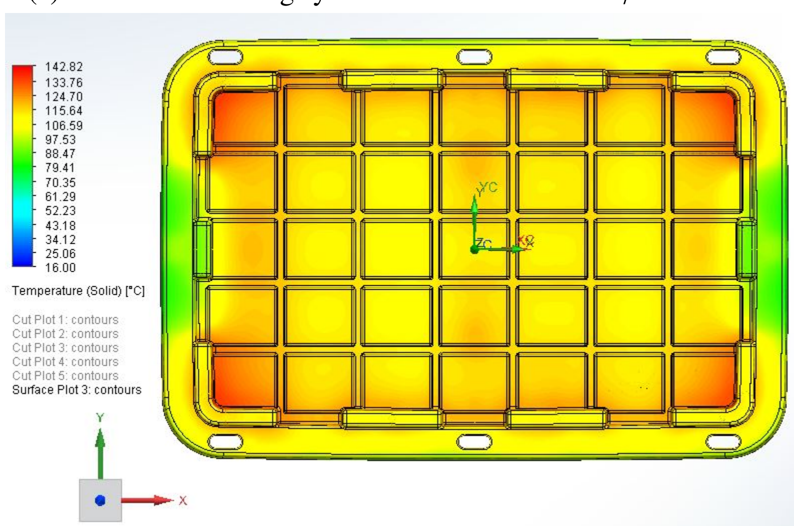


(b) Conventional cooling system for an inlet flow of 29 l/min at 20 °C

Figure 5.23: Comparison of the cooling channels temperature distribution between the conformal cooling system with inlet flow of 16 l/min at 25 °C and the conventional cooling system with an inlet flow of 29 l/min at 20 °C



(a) Conformal Cooling system for inlet flow of 16 l/min at 25 °C



(b) Conventional cooling system for an inlet flow of 12 l/min at 20 °C

Figure 5.24: Comparison of the cooling channels temperature distribution between the conformal cooling system with inlet flow of 16 l/min at 25 °C and the conventional cooling system with an inlet flow of 12 l/min at 20 °C

Chapter 6

Conclusions and Recommendations

In this research, the software NX-FLOEFD is evaluated for the simulations of the cooling process used in PIM. The purpose of this evaluation is to analyze the capability of the software to conduct the simulations of cooling systems used in PIM. The simulation results obtained for the mold A are close to the experimental data. So the software is capable to conduct simulations for PIM applications.

Based on the simulations for mold A at different inlet flow temperatures, the inlet flow temperatures should be between 20 to 30 °C. It is shown that higher inflow temperatures could reduce the thermal stress in the mold core and cavity area due to temperature changes which could compromise the durability of the tool. It is show in the current study, that temperatures of 20°C provides a shorter cooling time and more uniform temperature of the part.

Another mold used in the current research is mold B. In order to obtain an optimal design on this cooling system using the conventional cooling channels, parametric studies are conducted for different volume flow rates and inlet flow temperatures as well as different cooling channel diameter. The results show that the increase in the volume flow rate, up to 29 [l/min], reduced the cooling time and provided a uniform temperature distribution in the part and in the mold. For a flow rate that is higher than 29 l/min such as 35 l/min, there was no reduction in the cooling time despite the increase of the flow rate, which was also observed by Kapila [7]. Based on the analysis of the on the effect of inlet flow temperature, it was found the most optimal inflow temperature was 20 °C. Temperatures lower than this value will lead to an increase of the production cost considering that the improvement on the cooling time is not high enough to justify it. The evaluation of the changes in flow characteristics on cooling systems allows the designer to define the range of work of the cooling system without compromising the quality of the part.

Simulations for different cooling channel diameters were performed to evaluate the effects of the cooling channel diameter on the cooling time and temperature distribution. The bigger diameter will provide a reduced cooling time. Despite this fact there are certain limitations on the locations and size of the cooling channel considering the durability of the mold to avoid possible failures when the tool is in operation.

With the evaluation of different scenarios of the cooling system in the mold B using conventional cooling channel, an optimal operating conditions were at the volume flow rate of 20 l/min and inlet temperature of 20 °C. This was considerate the most optimal set up considering the improvements on the cooling time, temperature distribution in the part and the mold and the cost of production of the part.

A conformal cooling system was designed for the mold B. Comparing the same operating conditions as the mold B with conventional cooling system, there was a considerable reduction in the cooling time and the temperature distribution was more uniform. Therefore, the conformal cooling system gives more uniform temperature, so the part will have a better quality. Since the diameter of the designed conformal channels is reduced compared with the conventional cooling channel, an evaluation for the volume flow rate was applied to the conformal cooling channel. From this set of simulation it was observed that for the smallest volume flow rate of 10 l/min, the cooling time was still lower than the one obtained from the conventional cooling analysis. A reduced volume flow rate will also reduce the cost in the production, also that the temperature used for the analysis of the conformal cooling channel was 25°C which is closer to ambient temperature and this also reduces the production cost.

The newly designed conformal cooling channels in this study demonstrated the improvement on the cooling time and in the temperature distribution. The reduction of hot spots in the product at the ejection temperature was also noticed, so that this type of cooling system can provide better results for the quality of the parts. One of the disadvantages of this type of cooling system is the increased cost of its manufacturing process, additive manufacturing. For future research topics, it is necessary to evaluate the cost of manufacturing versus the cost of production of this type of applications, since this will allow to analyze the implementation of this system to improve production of plastic parts. Another evaluation recommended to apply on this type of cooling systems is the durability analysis. This analysis will define if the conformal cooling system design is possible to be applied in the industry without presenting any failures.

In the current work, of the evaluation of the capability of the software FLOEFD of Siemens for the simulations of cooling systems in PIM has been carried out. This allows the industry to obtain a software that has is able to model the design of a plastic injection mold tool. It is noticed that the numerical results obtained for different cooling systems under different operating conditions agree well with the experimental data. In this study, the effect of different geometric parameters on the performance of a conventional cooling system has also been investigated in order to achieve an optimal design point, which will achieve less cooling time with a more uniform temperature distribution along the part and the mold. The design of a conformal cooling has been done for the mold B with the objective to improve the effectiveness of the cooling process. The comparison between the results from the conformal cooling and the conventional cooling channels shows the reduction of the cooling time up to 45% as well as the uniformity of the temperature distribution in the mold and in the part using the designed conformal cooling system. Therefore, the conformal cooling channels are the better approach for the PIM industry.

There are limitations in the current work. first, the software is not capable in the simulation of the injection of the melted plastic and the ejection of the plastic part as part of the stages of the plastic injection molding. Secondly, the designed conformal cooling system needs to be evaluated against some experimental data. Also, the validations in this study have been done with limited experimental data Despite the limitations mentioned, the computational approach was capable of provided the expected results in the range of error permissible by the industry.

Bibliography

[1] S. Feng, A. M. Kamat, and Y. Pei, "Design and fabrication of conformal cooling channels in molds: Review and progress updates," *International Journal of Heat and Mass Transfer*, vol. 171, p. 121082, 2021.

[2] Wikipedia contributors, "Injection moulding — Wikipedia, the free encyclopedia," 2022. [Online; accessed 29-March-2022].

[3] E. Dimla, M. Camilotto, and F. Miani, "Design and optimisation of conformal cooling channels in injection moulding tools," *Journal of Materials Processing Technology*, vol. 164, pp. 1294–1300, 05 2005.

[4] Z.-C. Lin and M.-H. Chou, "Design of the cooling channels in non-rectangular plastic flat injection mold," *Journal of Manufacturing Systems*, vol. 21, pp. 167–186, 12 2002.

[5] M. Khan, N. Khan, and S. Ahmad, "Cycle time reduction in injection molding process by selection of robust cooling channel design," *ISRN Mechanical Engineering*, vol. 2014, pp. 1–8, 03 2014.

[6] C. Tan, D. Wang, W. Ma, Y. Chen, S. Chen, Y. Yang, and K. Zhou, "Design and additive manufacturing of novel conformal cooling molds," *Materials design*, vol. 196, p. 109147, 09 2020.

[7] A. Kapila, K. Singh, and N. Agarwal, "Study on the effect of flow rate on cycle time for multi-impression injection molding," 04 2015.

[8] E. Dimla, “Design considerations of conformal cooling channels in injection moulding tools design: An overview,” *Journal of Thermal Engineering*, vol. SPECIAL ISSUE 2 Energy Systems and Developments 2015 ICESD 2015 INDIA, pp. pp:627–635, 03 2015.

[9] C. Parag and D. K. M. Ashtankar, “Determining appropriate cooling system for plastic injection molding through computer simulation,” *International Research Journal of Engineering and Technology*, vol. SPECIAL ISSUE 2 Energy Systems and Developments 2015 ICESD 2015 INDIA, pp. pp:2395 – 0072, 06 2016.

[10] B. Rhee, C.-S. Park, H.-K. Chang, H.-W. Jung, and Y.-J. Lee, “Automatic generation of optimum cooling circuit for large injection molded parts,” *International Journal of Precision Engineering and Manufacturing - INT J PRECIS ENG MANUF*, vol. 11, pp. 439–444, 06 2010.

[11] E. Sachs, E. Wylonis, S. Allen, M. Cima, and H. Guo, “Production of injection molding tooling with conformal cooling channels using the three dimensional printing process,” *Polymer Engineering and Science*, vol. 40, no. 5, pp. 1232–1247, 2000. cited By 195.

[12] H.-S. Park, X.-P. Dang, D.-S. Nguyen, and S. Kumar, “Design of advanced injection mold to increase cooling efficiency,” *International Journal of Precision Engineering and Manufacturing-Green Technology*, vol. 7, 02 2019.

[13] D. R. R. M. Mr. Rahul S. Khichadi, Dr. Shivaprasad B. Dandagi, “Effect of different type of cooling channels for plastic injection mould,” *International Journal of Innovative Studies in Sciences and Engineering Technology (IJISSET)*, vol. 3, pp. 2455–4862, 04 2017.

[14] S. Marques, A. Souza, J. Miranda, and I. Yadroitsau, “Design of conformal cooling for plastic injection moulding by heat transfer simulation,” *Polímeros*, vol. 25, pp. 564–574, 08 2015.

[15] H.-S. Park and X.-P. Dang, *Design and simulation-based optimization of cooling channels for plastic injection mold*. 03 2012.

[16] S. Meyer, “Optimised mould temperature control procedure using dmls whitepaper,” pp. 1–10, 2009.

[17] Y. Wang, K. Yu, and C. Wang, “Spiral and conformal cooling in plastic injection molding,” *CAD Computer Aided Design*, vol. 63, pp. 1–11, Jan. 2015.

[18] B. He, L. Ying, X. Li, and P. Hu, “Optimal design of longitudinal conformal cooling channels in hot stamping tools,” *Applied Thermal Engineering*, vol. 106, no. C, pp. 1176–1189, 2016.

[19] D. Tomasoni, S. Colosio, L. Giorleo, and E. Ceretti, “Design for additive manufacturing: Thermoforming mold optimization via conformal cooling channel technology,” *Procedia Manufacturing*, vol. 47, pp. 1117–1122, 01 2020.

[20] C.-C. Kuo, J. Zi-Fan, and J.-H. Lee, “Effects of cooling time of molded parts on rapid injection molds with different layouts and surface roughness of conformal cooling channels,” *The International Journal of Advanced Manufacturing Technology*, vol. 103, 08 2019.

[21] S. Z. Abd. Rahim, S. Sharif, A. Zain, R. Mohd Saad, and A. Mansor, “Milled groove square shape conformal cooling channels in injection molding process,” *Materials and Manufacturing Processes*, vol. 28, 01 2013.

[22] S. Jahan and H. Elmounayri, “Optimal conformal cooling channels in 3d printed dies for plastic injection molding,” *Procedia Manufacturing*, vol. 5, pp. 888–900, 12 2016.

[23] S. Jahan, T. Wu, Y. Zhang, J. Zhang, A. Tovar, and H. Elmounayri, “Thermo-mechanical design optimization of conformal cooling channels using design of experiments approach,” *Procedia Manufacturing*, vol. 10, pp. 898–911, 12 2017.

[24] S. A. Jahan and H. El-Mounayri, “A thermomechanical analysis of conformal cooling channels in 3d printed plastic injection molds,” *Applied Sciences*, vol. 8, no. 12, p. 2567, 2018.

[25] C.-C. Kuo, J. Zi-Fan, X.-Y. Yang, S.-X. Chu, and J.-Q. Wu, “Characterization of a direct metal printed injection mold with different conformal cooling channels,” *The International Journal of Advanced Manufacturing Technology*, vol. 107, 03 2020.

Curriculum Vitae

Name:	Veronica Flores Quijada
Post-Secondary Education and Degrees:	Bachelor in Mechanical Engineering
	Universidad Simon Bolivar Caracas, Venezuela 2009 - 2016
Honours and Awards:	
Related Work Experience:	Teaching Assistant and Graduate Fellowship The University of Western Ontario 2020 - 2021 Mitacs Accelerated Program 2021 - 2022

Publications: