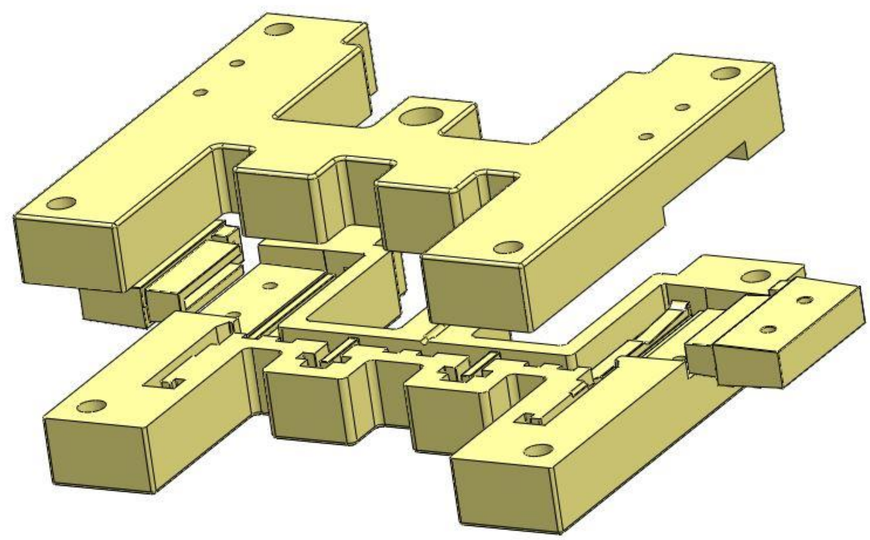
CHALMERS
UNIVERSITY OF TECHNOLOGY



# Enhancing injection molding process through additive manufacturing

Master's thesis in Product Development


ASHISH ASHOKKUMAR

**DEPARTMENT OF INDUSTRIAL AND MATERIAL SCIENCE**
Chalmers University of Technology
Gothenburg, Sweden 2020

# Enhancing injection molding process through additive manufacturing

ASHISH ASHOKKUMAR



EXAMINER: EDUARD HRYHA, *Professor at the Division of Materials and Manufacturing, Industrial and Materials Science, Director of CAM²: Center of Additive Manufacturing-Metal*

SUPERVISORS: MARIE FISCHER, *Post-doctoral researcher at the Division of Materials and Manufacturing*, ZHUOER CHEN, *Post-doctoral researcher at the division of Materials and Manufacturing*, FREDERIC THOLENCE, *Principal engineer in ABB Corporate Research* and ERIK JOHANSSON, *Principal engineer in ABB Corporate Research.*

CHALMERS

Master's Thesis 2020
**Department of Industrial and Material Science**
**Chalmers University of Technology**
Gothenburg, Sweden - 412 96
Telephone +46 31 772 100

# Abstract

Injection molding is the most common manufacturing technique used to produce polymer components in high quantities. The mold insert development is the most time-consuming phase of the injection molding process that has a high potential to be accelerated by additive manufacturing (AM). AM enables to develop mold inserts with a short lead time than the conventional manufacturing processes. Thus, design iterations in the mold insert development phase can be done more rapidly. Additive manufacturing also helps to create complex geometries that provide the opportunity to create conformal cooling channels in the mold inserts. Conformal cooling channels increases the productivity of the injection molding process by reducing the cycle time of a unit part produced. Since these channels promote uniform cooling around the part, they increase part quality and decrease scrap rate. Thus, it could be highly beneficial for a company that makes a large volume of polymer products to explore the potential of additive manufacturing. Like every other new technology, there needs to be a pre-study to understand the various implications while adapting to it.

In this thesis work, the different stages of the mold insert development, which can benefit from AM, are investigated with the help of a product case study. This thesis mainly focuses on the design and prototyping of the mold inserts as well as the production of economical mold inserts. Three additive manufacturing technologies, such as poly-jetting, binder jetting, and laser powder bed fusion, are studied. The most suitable technology for each of the focused areas is selected by comparing the time to produce the mold insert, tool life of the mold inserts, and manufacturing cost. Binder jetting is selected to address the prototyping of mold insert development, and laser powder bed fusion for the economical production of the mold insert. A mold insert for the product under study is designed according to the guidelines of the respective additive manufacturing process. To address the prototyping of mold inserts, a prototype mold insert is manufactured using binder jetting. Initially, two design iterations are made to suit the prototyping of the mold insert within the budget of the thesis work. The factors that influenced the quality of the prototype were analyzed, and a third design iteration of the mold insert is proposed. To develop the production of the mold insert, injection molding simulations were done using Autodesk Moldflow advisor 2019 which follows the finite element analysis (FEA) methodology. A comparison between the conventional cooling and conformal cooling is done. The mold inserts with conformal cooling exhibited a decrease in the cycle time of the injection molding process. A solution to also manufacture the production mold insert economically with the help of additive manufacturing is proposed through the implementation of lattice structures within the mold insert. This solution is experimentally validated with the help of a prototype. A comparison of thermal behavior between a solid mold insert and a lattice filled mold insert was made. It was concluded that the lattice filled mold inserts performed better in the experiments than the solid mold inserts. Finally, the implications of additive manufacturing in its application to the injection molding industry are discussed.

Keywords: Injection molding, additive manufacturing, mold insert development, poly-jetting, binder jetting, laser powder bed fusion, conformal cooling, Autodesk Moldflow adviser 2019

# Acknowledgments

I want to thank Santanu Singha, Frederic Tholence, and Erik Johansson of ABB corporate research for giving me this excellent opportunity to carry out this thesis work. Their support and guidance motivated me to finish the thesis work with convincing results. I want to thank ABB corporate research for providing the licenses for the different software utilized in the thesis work. I am grateful for ABB corporate research for providing financial support to help me visualize my ideas through prototypes from the best external suppliers in the field of expertise. I would also like to thank Saeed Maleksaeedi of ABB for providing key insights and feedback for the thesis work.

I am grateful to the Center of Additive manufacturing-metal ($CAM^2$) for giving me this opportunity to carry out the thesis. I want to thank my examiner Eduard Hyrha of the department of industrial and material science of Chalmers university of technology for providing me valuable feedback and immense support to complete the thesis work. My Supervisors Marie Fischer and Zhuoer Chen of Chalmers university of technology have been strong pillars of support and provided their valuable feedback, which played an essential role in this thesis work.

Finally, I would like to thank my family and friends for extending their support and motivating me throughout the thesis work.

Ashish Ashokkumar, Gothenburg, 04/2020

# Table of contents

# List of abbreviations

| | |
|---|---|
| **IM** | Injection molding |
| **AM** | Additive manufacturing |
| **CAD** | Computer aided design |
| **ABS** | Acrylonitrile butadiene styrene |
| **PLA** | Polylactic Acid |
| **PA** | Polyamide |
| **Poly-Jet** | Polymer jetting |
| **BJ** | Binder jetting |
| **L-PBF** | Laser powder bed fusion |
| **mm** | Millimeter |
| **°C** | Degree Celsius |
| **HRB** | Rockwell hardness, scale B |
| **HRC** | Rockwell hardness, scale C |
| **µm Ra** | Surface roughness |

# List of figures

# List of Tables

# Chapter 1

## 1.Introduction

Manufacturing processes have been improved and optimized as much they could with the help of new technology. Innovative approaches in manufacturing not only help in increasing the productivity of the process but also help to bring the product first to the market. It takes more than just the procurement of the new technology to adapt to such innovative approaches successfully. There needs to be a cross-functional effort from various engineering sectors of the company. Multiple parameters in the manufacturing process could be optimized to achieve a superior quality of the product in a shorter time when compared to conventional manufacturing techniques. It is essential to identify such parameters in the manufacturing process. Attempting to optimize such parameters would take numerous iterations and effort. Today with innovations in every sector of engineering, there happens to be rift from conventional manufacturing techniques to new innovative methods. The possibility of the next industrial revolution is very evident. Companies and corporations seem to be highly aware of such a revolution in the future. They are preparing themselves for such a rift by investing heavily in research and development. Adopting new trends are product specific. Hence a detailed study of these new technologies is critical. Thus, before deciding to take them, proper reasoning and experimentation are vital.

Once it is realized these new processes are thoroughly studied, and when its true potential is identified, it takes experimentation and analysis to verify the suitability of the process to suit a specific application. Breakdown of cost and time also plays an important role. A conventional manufacturing process is something which is proved to work well but not the best. There is always scope for improvement. Once these new processes are experimented and validated, it is crucial to set guidelines in such a way that similar or future products in the company could adapt to such new processes in a more economical and time-efficient manner. These iterations would cost the company in a big way. Hence it is vital to set guidelines so that these costs could be cut down and the adaption process is made easy for the company.

In this project, one such highly typical manufacturing process, such as injection molding (IM), is chosen. An approach to optimize the IM tool production is investigated. A production tool such as the injection mold plays a vital part in the manufacturing process, and it directly affects the productivity of the manufacturing process. It is one of the critical factors that also lead to better product quality. Conventional injection mold inserts work fine, but during the mold development phase, they cause delays due to high lead time taken to manufacture these molds. There are also some design constraints while making these mold inserts conventionally like implementing multiple cooling channels, which could increase the cooling efficiency, simultaneously decreasing the cycle time of per unit part. Q

Additive manufacturing (AM) is a non-traditional way of manufacturing products and is catching the attention of various industries due to its multiple benefits. It is a new technique of manufacturing, and leading engineering companies conduct numerous kinds of research. In this project, the investigation of how AM could benefit the mold development and the production of parts through IM by additive manufactured mold inserts is conducted. The technical feasibility and its benefits are studied in this project.

## 1.1. Background

IM inserts are conventionally manufactured using CNC milling with materials such as tool steel and aluminum. These conventionally produced mold inserts take many iterations to achieve the desired design. These iterations cost high and carry a high time to develop. The mold development process plays a vital role in the product and performing quick design iterations help the company to bring the product to market. The prototyping of such mold inserts and the final production mold insert have a high scope of improvement. The production mold inserts influence the productivity of the IM process. Choosing an alternative method to prototype mold inserts to make quick design iterations and creating a production mold insert, which leads to the reduced cycle time of the process at the same time increase the product quality, is necessary.

Additive manufacturing (AM) of high performing polymers and metals opens new opportunities to address the various stages of the IM process. The time to manufacture products through AM helps in making quick design iterations, and by choosing the right AM process and materials to suit the need and application would benefit the engineering and manufacturing department tremendously to save time and money. There are several previous types of research in this area, and hence it is proven that AM could be beneficial for mold insert development and production.

ABB Control Technologies, Västerås, Sweden, is a business unit that is a part of the industrial automation sector and produces a vast range of polymer products that are used for control devices in ABB’s control products. They have an in-house manufacturing unit with state-of-the-art IM set up. This business unit has decided to see the potential of AM to increase the efficiency of the injection mold development. Hence this thesis work is carried out to investigate the difficulties and implications to integrate AM with the IM mold development process.

## 1.2 Stakeholders

### 1.2.1 ABB control technologies

The primary stakeholder is ABB control technologies, Västerås, Sweden. It is a business unit of ABB, which is a part of the industrial automation sector of ABB. Its main operation deals with the design and manufacture of control devices of ABB’s products. They have an in-house manufacturing set up, where a wide range of products are manufactured. They have a state-of-the-art IM setup. The control technologies do not, however, design the tooling for the IM process. They subcontract the tooling design and manufacture to a supplier in China. This business unit is responsible for automation products that cater to all types of industries. The project work carried out is primarily for this business unit. The business unit is looking to move towards new technologies to improve the lead time of its products. AM is one such technology that caught the interest of this business unit since it has the benefit of producing products in a fast manner. The main interest of the business unit lies within the prototyping of injection molds, which they currently carry out with the help of Chinese suppliers. It takes a long time to get the molds from China, and this seems to increase the lead time of the mold development process. Other possible areas that could be improved using AM could help in increasing productivity and reduce the lead time of their products.

### 1.2.2 Other stakeholders

The other stakeholders are the external suppliers who help to manufacture the mold inserts for the thesis work. The two suppliers which were chosen by ABB were Amexci AB, Sweden, and ExOne, USA.

Amexci is a leading AM service provider in Karlskoga, Sweden. They offer a wide variety of services in manufacturing AM products of both metal and polymer. They are a new company found in 2017 and have many shareholders, including ABB. They have an expert team specialized in working on different applications and providing services to various industries in Sweden. Since ABB had previously dealt with successful products in Amexci, it was chosen to provide the service for this project.

ExOne is an established AM company founded in the USA. They are pioneers in the AM process called binder jetting. They not only provide manufacturing services but also develop and sell their machines. They also have an inhouse research and development team which develops AM materials and new equipment. Though their external suppliers within Sweden, ABB felt that ExOne could provide a much better service and better quality of products since the mold inserts could be a little complex to manufacture. They have also previously dealt with projects with ExOne and consider them to provide an excellent service.

# Chapter 2

## 2. Theory

### 2.1 Injection molding (IM)

IM is one of the most common manufacturing processes to make plastic parts today. It has been present in the manufacturing scene for quite a long time. It is defined as the cyclic process carried out with the help of a mold to produce identical plastic parts. One of the main advantages of the IM process is that it could produce simple and complex plastic parts repetitively at high production rates.

The IM process is carried out in an IM machine, which comprises two primary assemblies. The injection unit and the clamping group. The injection unit consists of a hopper where the material that needs to be fed is loaded, a heated cylinder to melt the plastic material and a rotating screw that helps in pumping or injecting the plastic material into the next phase. The rotating screw is situated inside the heated cylinder. The clamping unit consists of a mold base, which includes a mold insert. The mold inserts usually are made in two halves.

IM, as mentioned above, is a cyclic process. The preliminary process before the injection cycle starts called the plasticizing stage. In this stage, the plastic material in the form of granules is fed into the hopper, which acts as the feeding mechanism to the heated cylinder. When plastic granules are fed into the cylinder, the rotating screws are activated and due to the heat generated by the friction of the rotating screws and the temperature in the cylinder, the plastic granules melt. As the screws rotate backward, the molten material is pushed towards the tip of the screw. Once enough material required for the injection cycle is reached the screws stop turning. The pressure is developed in the cylinder, which pushes the molten material through a nozzle situated at the end of the cylinder. This nozzle helps to inject the material into the mold inserts in the clamping unit. After this, the injection cycle begins.

The injection cycle consists of three primary operations which are filling, packing, and cooling. The filling stage is when the screw moves forward and forces the molten material into the mold inserts. When the filling is taken place, the mold halves are kept closed with a high clamping force. The filling takes place until the cavity in the mold inserts is filled. Once the filling is complete, the packing stage begins. The packing stage starts by giving some time for the molten material in the mold insert to cool down. This also leads to volumetric shrinkage of the material. To control shrinkage, the rotating screw is maintained in the forward position so that a pressure called holding pressure is maintained. This is done for a certain period so that excess material is pumped in and controls the shrinkage of the plastic. After the packing stage is done, a high amount of heat is generated in the cavity of the mold inserts. This heat must be removed to bring the plastic part to solidify. To facilitate this operation, there exist cooling channels in the mold inserts which transfer the heat from the mold and cool the plastic piece in the cavity. The coolant usually used is water. After the cooling stage is completed, the clamping unit opens, and the plastic part is ejected [1].

### 2.1.1 Mold insert and key features

The mold insert assembly is the primary tool in the IM process. The mold inserts consist of many features that help to produce the plastic part. These mold inserts are held in place with the help of a mold base in the clamping unit. These features vary in design from part to part, depending upon the size and geometry of the plastic part. The mold insert assembly consists of two parts, that are the core and the cavity. These help in giving the molten plastic to get into the final shape of the part. The internal features of the plastic part are formed by the core and the external features by the cavity. In other words, it could be said that the hollow section in a mold insert is the cavity and the mating raised surface on the other half of the mold insert assembly is known as the core. In some cases, there would be two cavities like mirror images on both halves of the mold. The cavity is often the top half of the mold insert assembly, which faces the injection face of the nozzle from the heated cylinder in the injection unit. This is because the plastic part usually shrinks on the core, which also has an ejection mechanism through which the part could be ejected (demolded) from the mold insert. There are single cavity and multi-cavity molds. Single cavity molds are usually used for plastic parts of a bigger size or plastic parts of small volume production. Multi cavity molds are made so that it would be more economical when plastic parts of high volume must be produced. It increases the production rate by producing multiple products in a single IM cycle.

The feature that separates a core and a cavity are called parting line. The parting line is the feature that helps in splitting the mold insert tool into two halves so that the part could be manufactured properly without missing any underlying features. This parting line could be on the top of part geometry, bottom, center, or place in an irregular manner depending on the complexity of the plastic part. It could be of any shape but only in one plane. It is mostly present on the outer circumference of the plastic part so that it could facilitate ejection. This parting line should be as simple as possible so that the mold insert surface that is split into two halves can be easily machined. If the parting plane that is the plane where the parting line is poorly finished, the mold halves cannot be tightly sealed while injection and the molten plastic could escape from the mold insert leading to a defect called “flash” and the plastic part could be unusable. With the help of parting lines, it is also possible to differentiate the type of mold. The most commonly used molds are two plated molds and three plated molds. A two plated mold is the one which consists of two mold halves split along one parting plane, and a three plated mold is the one where two parting planes are present, one which would be the first parting plane and the other would be the secondary parting plane. Three plated molds are mainly used for parts with complex geometry [1,2].

The next important feature of the mold insert is the feed system. This feed system helps in the flow of molten plastic from the injection nozzle to the cavity of the mold inserts. The feed system is again divided into three parts, namely sprue, runner, and gate. These also do not have a standard design but vary accordingly. The sprue is the passage that is in contact with the injection nozzle, which brings the molten plastic to the runners. The runners distribute the molten plastic from the common sprue to a single cavity or multiple cavities. Agate is a small opening from the end of the runner to the cavity. There different types of runners and gates. The two major types of runners are a hot runner and cold runner. A hot runner is a mechanism used to heat the whole passage for the proper flow of the plastic material. A cold runner whereas just a passage is machined to facilitate the flow of the molten plastic into the mold cavity. Hot runners are used to producing high-quality parts. It must also be noted that hot runners cannot be used in all the mold inserts. It depends on the parting plane of the mold. In those cases, a cold runner could be used. There different types of gates, namely standard gate, ring gate, submarine gate, tab gate, disc gate, film type gate, fan gate, spider gate, hot probe gate, sprue gate, pinpoint gate, and chisel gate. Each gate has its advantages and is selected according to the type of mold, number of cavities, problems like stress that arise during the injection cycle, and problems that exist after ejection [3].

The next important feature of the mold insert is the cooling channels. There are two types of cooling: cooling by dissipating the heat to the environment and cooling with the help of cooling channels to remove the heat from the mold. To increase the productivity of the manufacturing process, it is necessary to have a proper cooling channel layout since the cooling stage takes the maximum time in the injection mold cycle. A cooling channel is designed with respect to the plastic part shape in the cavity and core. The layout of the cooling channel can be designed with respect to the type of flow we want in the mold insert. The most common flow types are series cooling, parallel cooling, and a parallel -series cooling. The cooling is carried out with the help of a coolant. The diameter of the cooling channels depends on the plastic part thickness. The physics of the cooling is complicated hence computer-generated simulations should be carried out to design an optimized cooling channel layout for maximum efficiency [3].

After the cooling stage is complete, the plastic part must be ejected or demolded from the mold inserts. An ejector mechanism helps in this function. Some of the ejector mechanisms are pin or sleeve mechanism, stripper plate mechanism, air alone, air-assisted, a combination sleeve, striper, and air. The most common mechanism is the pin or sleeve mechanism. At the ejection stage, the ejector pins force pushes the plastic part that is sticking to the core insert [2].

Other than the above key features in a mold insert, there are also components like sliders that act as side core for undercut geometries in the plastic part. These undercut geometries could be made in the core of the mold insert itself, but due to the opening and closing of the mold during the IM cycle, this is not possible. Hence slider cores are placed in the mold insert assembly wherever they are required. Figure 2.1 below shows the flowchart of the stages of designing a mold insert.

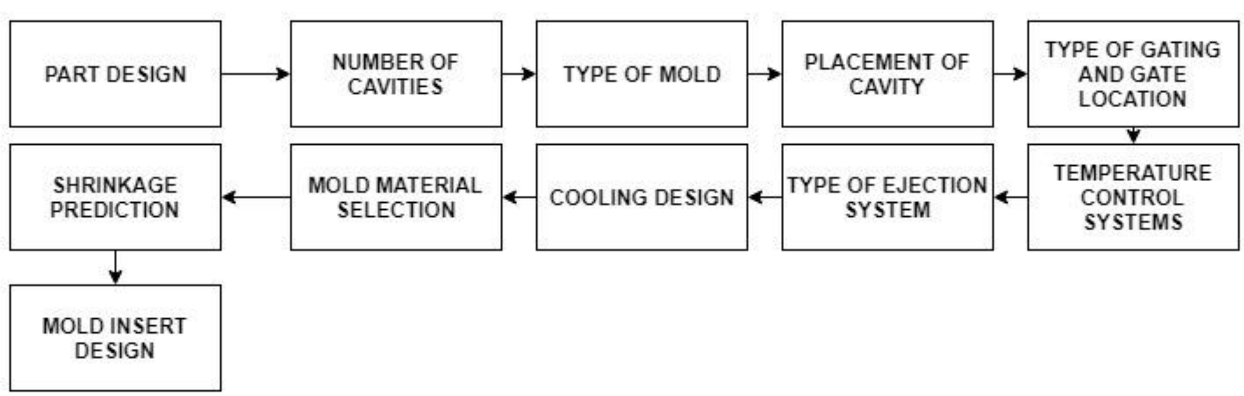



Figure 2.1: Stages of designing a mold insert [4].

### 2.1.2 Cooling of molds inserts

As mentioned earlier, the most prolonged time taken is the cooling stage in the IM cycle. About 65-80 % of the time is spent in the cooling of the mold. Hence to increase the production rate of the manufacturing, it is necessary to have an ideal cooling layout to decrease the cycle time of the IM process. The decrease in the production rate is not the only risk because of inefficient cooling, but it has other risks involved in the quality of the plastic part. The plastic parts are affected in terms of quality because of hotspots, residual stresses, and warpage. These are the common defects because of which tooling designers and manufacturers must make changes in the existing mold insert or make new iterations of the mold insert. Hence it is necessary to have an efficient design of cooling channels to avoid such defects [3].

The main factors influencing the cooling of mold inserts are

- Plastic part wall thickness- thicker walls take a longer time to cool, and this increases in the cycle time of the IM cycle.
- Mold insert material – the material of the mold insert should be of high thermal conductivity so that there is better heat transfer, and it could be cooled at a faster rate.
- Cooling channel layout - the better the layout of the cooling channel around the cavity, the closer the channel is to the cavity, and the bigger the channel diameter, the better will be the cooling.
- Type of coolant- usually, water or oil are used as coolants. The viscosity and thermal conductivity of the coolant are also responsible for effective cooling in the mold insert [5].

Conventionally, the molds are cooled by convection with water lines. In this type of cooling, the water flowing through the mold removes the heat existing in the mold walls and carries the temperature outside the mold. The main reaction involved here is the convection between the water lines or cooling channels and the mold wall. The significant factors or variables in this type of cooling are the type of coolant used, kind of flow of the coolant, and the final temperatures involved. Drilling holes in the mold insert conventionally make these water lines or cooling channels. The cooling channel layout is something that should be carefully taken into consideration. It takes much time and effort to optimize the cooling channel layout. The main goal of the cooling channel layout is to implement uniform cooling in the mold insert around the cavity area. The problem with conventional cooling is that it is produced by traditional manufacturing processes such as drilling and milling. The cooling lines are formed by intersections of two or more lines drilled through the mold insert. These cooling lines do not produce uniform cooling around the cavity of the mold insert but only cools the mold rapidly. Defects such as warpage are the result of non-uniform cooling. Different temperatures around the cavity of the mold insert the result in shrinkage of the plastic at different rates in different areas of the geometry of the part. These problems can be simulated and optimized using computer simulations [3]. It is only possible to produce uniform cooling to an extent with the help of conventional cooling lines. Hence it is necessary to investigate other nontraditional manufacturing processes to create cooling channels that could bring about uniform cooling.

### 2.1.3 Development process of mold inserts

To produce a mold, insert that is ready for manufacturing, various factors have to be considered. The most primary input required is the design of the plastic part. The plastic part is developed, and it should be designed to suit the requirements of manufacturing. The manufacturability of the part should be taken into consideration in the design stage itself. The material of the plastic part is also an important factor that influences the mold design. The mold design itself depends on the design of the mold base. The overall dimensions of the mold insert comprising core and cavity depends on the size of the mold base, which is a part of the clamping unit. The type of manufacturing process is later decided. When the final mold design is ready, it is taken for then IM process [6].

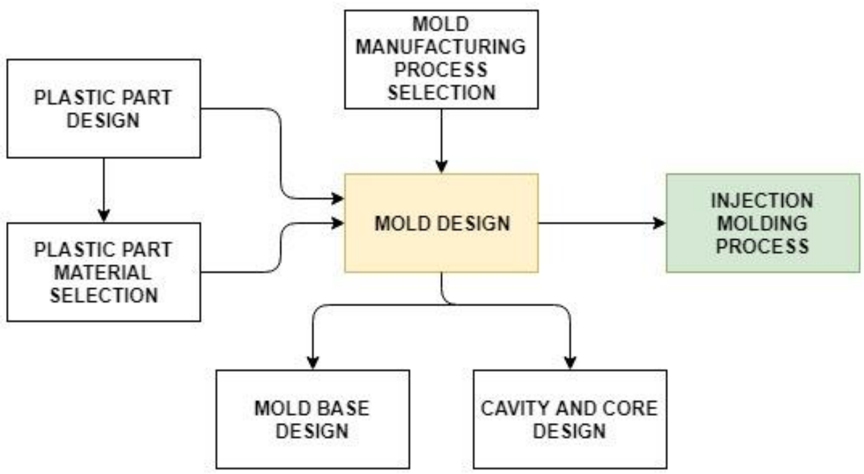



Figure 2.2: Factors influencing the mold development process [6].

The development of mold inserts is not as simple as mentioned above. It takes a high amount of time and effort to get the mold design right. Iterations of mold designs must be done and tested to reach the final design that could be used for mass manufacturing. To perform these iterations, prototype mold inserts are manufactured, and their outputs are tested. At least three design iterations are made on an average before getting the final mold design right. Three prototyping cycles are mostly planned.

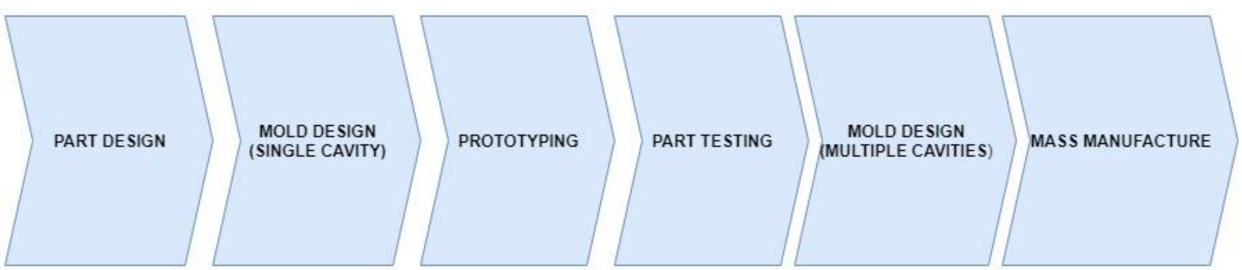



Figure 2.3: Process chain of the mold development process.

Prototype tools or prototype mold inserts are usually performed lower than the actual production tool because of the absence of cooling channels in them. The prototype tools are usually used to test the final plastic part. The part design only when tested with the actual material that was decided for the part, could be concluded whether the part design is good or if something must be changed. Hence these molds are usually used mostly for laboratory testing purposes. These molds could also be used for short volume production (several hundred parts to thousands of parts). These mold inserts are conventionally made with aluminum or tool steel materials. They are commonly manufactured using CNC milling and turning operations. By making these prototype molds, errors, and costs that could occur during actual production could be eliminated at the initial stages of the mold development stage [7].

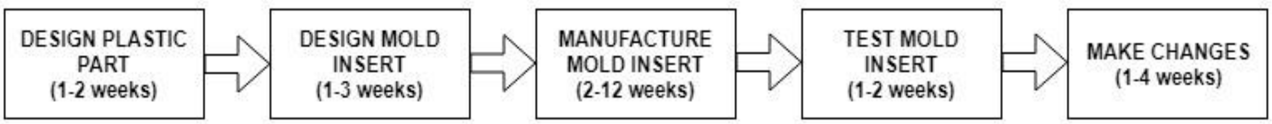



Figure 2.4: Time flow to produce a single iteration [8].

The cost and time to produce a single prototype mold are high. Usually, the prototype mold inserts are of a single cavity. These mold inserts are commonly used for smaller plastic parts. Larger plastic parts are split into important regions, and then mold inserts are made for them because of the high tooling cost. The time to produce a single prototype mold may take from 5-12 weeks. To test a single prototype mold insert, it may take up to 6-23 weeks. This includes the time taken to design the mold and manufacturing them. Hence by doing three iterations on an average, it takes around 69 weeks maximum to manufacture the final production mold insert. This causes delays in bringing the plastic part to the market. Hence decreasing the lead time to produce these prototype molds could be very beneficial [8].

## 2.2 Additive manufacturing (AM)

AM, otherwise known as 3D printing, is a process of manufacturing geometries by joining materials layer upon layer. It is the opposite of subtractive manufacturing processes like milling and turning where the material is removed in order to achieve the desired geometry. AM is done just by using a CAD file of the part. These CAD files are processed by an AM software where it is sliced, and it would be sent to an AM machine that uses the data to manufacture the part by adding material layer upon layer until the final geometry or desired shape is achieved. The material is in different forms, like powder, liquid, or solid wires. The commonly used materials are metal, polymers, and ceramics. By the kind of material used, AM is differentiated into the various process, each of which is unique. Some of the commonly used AM processes are material extrusion, laser powder bed fusion, liquid photopolymerization, binder jetting, and direct energy deposition. AM has many advantages when compared to conventional manufacturing techniques.

It provides new possibilities in various industries; some of them are aerospace, automotive, medical implant manufacture, prosthetics, toolmaking, and jewelry [9,10].

Some of the advantages of AM are

- Design Freedom- complex geometries that could not be fabricated with conventional manufacturing techniques are possible with AM. This provides new opportunities to produce an optimized design that is complex to manufacture but high in performance. Lightweight structures could be produced with the help of lattice structures. Thus, a designer could design apart by keeping in mind only its performance and not its manufacturability.
- Customization- the parts designed and manufactured through AM has a high degree of customization.
- Reduce Material wastage- since only the required amount of material is used to produce the parts in AM, there is reduction in wastage of materials, unlike the conventional subtractive process where the material that is removed to produce a geometry goes to scrap.
- Near net shape production- Parts could be produced to near net shape and finished with minimal effort.
- Time to market- AM is a slow process. It is, however, faster than the conventional manufacturing techniques because it is a tool free technology and does not require any effort in making molds or codes for machining. It also helps in making quick design iterations, which helps in bringing the part quick to market.

Though AM has many advantages, it has certain limitations too. As mentioned above, AM is a slow process and is not suitable for mass production. There are also constraints in the size of the part that could be manufactured. Today big part geometries are possible with AM, but a compromise in the accuracy of the dimensions is made there. For applications where tight tolerances are needed, there is a size constraint of the overall dimensions. The materials available for AM today are also limited, and many ongoing kinds of research are still trying to introduce new materials into this technology [10].

### 2.2.1 AM to produce mold inserts

AM can be beneficial in the area of the injection mold tool making process. There are mainly three benefits of AM to produce mold inserts. The advantage of producing quick prototypes and producing complex geometries through AM is very beneficial in the tool making application.

The mold development process, as mentioned above, is an iterative process. It takes several design iterations through prototype mold inserts to get the desired characteristics of a production mold insert. These prototype mold inserts, when manufactured using AM technologies, can be cost and time efficient. The time taken in the design stage remains the same as that of the conventional process, whereas the time taken to manufacture them is almost half the amount of the time taken to produce a conventional prototype mold insert. This could help in making quick design iterations and thus helps in reducing the lead time of the mold development process. This would help in bringing the product quickly to the market. A white paper in Stratasys says that these mold inserts could save up to 97% of the time to produce the first iteration and 33% in time to market when compared to the conventional prototype molds. It also says that cost up to 23% could be saved when mold prototypes with AM are made [11]. However, these are figures when the mold inserts were manufactured using the poly-jet process and may vary with other AM processes.

The next benefit of AM to produce mold inserts is the possibility of producing complex geometries with interior features. This advantage helps to produce conformal cooling channels in the mold inserts. Conformal cooling channels are complex cooling lines that are curved and follow the contour of the cavity in the mold insert. This helps in efficiently cooling the part and creates uniform cooling in the mold insert. Traditionally cooling lines are manufactured by drilling holes in the mold and plugged later to make the cooling circuits. These traditional cooling channels do not uniformly cool the plastic part in the mold. This results in long cooling times and quality defects like the warpage. The implementation of conformal cooling channels could avoid this. Conformal cooling channels help in providing controlled cooling of the plastic part. Complex channel profiles can also be implemented using AM. Metal additive technologies like laser bed fusion help in producing these channels [11].

A hybrid approach of AM could be implemented for large mold inserts. Areas of the mold which are simple can be done traditionally, and complex features can be manufactured on top of the traditionally manufactured mold. This helps in saving time and cost. Another way to save cost a hollow mold with lattice structures could be manufactured with AM as well. This presence of lattice structure not only decreases the cost but also increases structural stability [12].

## 2.3 AM processes suitable for manufacturing injection mold inserts

Among the different AM processes, the potential processes that could be used to produce mold inserts are poly-jet, binder jetting, and laser powder bed fusion (L-PBF). These processes are considered potential because of the mechanical and thermal properties their finished parts offer. These processes are described in detail below.

### 2.3.1 Poly-jet

One of the AM processes that are suitable for manufacturing injection mold inserts is poly-jet printing. It is a part of the material jetting processes. Material jetting is a process where droplets of desired material are deposited on a tray which is later cured by UV light to solidify and bond to each other. It is a layer by layer building process where several layers of the desired material are formed one over the other to achieve the desired shape of the part geometry. Different materials are possible like metals, ceramics, and polymers, where polymers are commercialized. The process which uses polymer materials is called poly-jetting [13].

There are two types of droplet formation techniques that the AM machines in the material jetting area use. They are droplets on demand (DOD) and Continuous stream. These two techniques differ by the way in which the liquid material exists in the nozzle of the machine. In the continuous stream, the liquid is formed in the form of a continuous column, and the droplet on-demand technique, the liquid is formed discretely. In today's machines, DOD is used because of its small droplet size and accuracy when compared to the continuous stream process. These AM machines produce several acrylic photo-based polymers with a layer size of 0.04 mm with 1,536 individual nozzles. The nozzles deposit the desired polymer material and the support material, which can later be removed by hand or water jetting. This multi material capability exists depending on the machine manufacturer. These nozzles deposit the material rapidly with high accuracy in layers. Each layer is cured using UV light immediately after the droplets are formed. This curing principle is known as photopolymerization [13].

The advantages of poly-jet are low cost and high-speed manufacturing of parts. It also allows manufacturing parts with multiple materials at the same time. Complex shapes could be realized using this process. The parts that could be manufactured with poly-jetting have a high degree of scalability. The poly-jet process offers few thermoplastic materials which are high in strength. They also tend to withstand high temperatures, which is suitable to make prototype mold inserts and mold inserts for short volume production. The disadvantages of poly-jetting are that when compared to other AM processes, they offer limited material choices. The two major machine manufacturers that offer this process are Stratasys and 3D Systems. The applications of poly-jetting are used in various industries such as healthcare, automotive, and rapid tooling [13].

### 2.3.2 Binder jetting

The second process that could be suitable for the manufacture of injection mold inserts is binder jetting. It is one of the AM processes among the powder bed technologies. Binder jetting is a process, where binder is deposited into the metal powder particles to form a cross-section of the part. Binder droplets are used to form spherical agglomerates in the powder bed. This binder is also bonds layers of powder while printing. Like most of the AM processes, binder jetting is also a layer by layer process. After a layer is completed, the powder bed moves slightly below, and another layer of material is deposited on the completed layer, which is then bonded together using the binder material. There exist several nozzles which deposit binder on the powder bed. This allows the scalability of the parts that are manufactured using this process. Binder jetting offers materials like metals, ceramics, and sand. The metal binder jetted parts are found to be more suitable for tooling production applications [13].

The metal manufactured part is printed in a green state, which is characterized by high porosity and is very brittle. In order to provide strength, the part must be sintered at a high temperature. Infiltration can also be used to get the full density at lower sintering temperatures. After printing, the remaining powder is removed from the part with pressurized air and later carefully taken to sintering. This is done in a furnace where the binder is burnt away, leaving most of the part to be porous forming voids. During further heating, part starts to sinter – meaning metallurgical bonds between the metal powder particles are created. During sintering, pores can be filled with a secondary material, typically bronze, this is called infiltration. Alternatively, part can be heated to higher temperature, typically 75 to 85% of the melting temperature of the material and held at the temperature for 30 to 120 min in order to improve densification and create strong inter-particle bonds. High temperature sintering results in higher shrinkage and hence requires more dedicated design for AM and typically more trial-and-error cycles to get to the desired geometry and tolerances of the final component occur. Printing does not require any support materials when there exist any overhangs in the part geometry because the surrounding unused powder helps as support. However, the part must be properly designed to avoid/minimize distortions during sintering. [13].

The advantage of binder jetting is that it produces low-cost parts when compared to other metal AM technologies. A large number of small parts can be manufactured at the same time because of the large build volume of the powder bed. Another advantage of binder jetting is that it provides the best surface finish from metal AM technologies. The disadvantage of binder jetting is that the parts have low dimensional accuracy. Large metal parts, when manufactured with binder jetting, are subjected to warping and dimensional inaccuracies. Some of the companies which provide this technology commercially are ExOne, Voxel jet and DigitalMetal. Typical applications of binder jetting are used in pattern or tool making of sand cores, jewelry, medical prototypes, tooling, and making prototypes with color in case of polymers [13].

### 2.3.3 Laser powder bed fusion (L-PBF)

Laser powder bed fusion is another major AM technique used to produce metal parts that could be useful for the mold manufacturing application. This process falls under the powder bed fusion family of AM.

L-PBF is a process that uses a laser to fuse the metal particles. A thin layer of metal powder is spread on the build plate or the previous layer. Further the laser melts the powder particles and bonds the layers together. Just like binder jetting, it is a layer by layer process. This is completed when the desired shape of the part is attained. The whole process takes place in the closed chamber. Inside the chamber, gases such as argon and nitrogen are used, depending on the reactivity of the metal powder that is used for the process. These gases help in avoiding oxidation of the build part, while the process takes place. In order to successfully manufacture a metal part using L-PBF, several process parameters must be taken into careful consideration. These key parameters are laser power, laser scanning speed, hatch distance, and hatch overlay. These parameters are highly responsible for the mechanical properties of the final part. Supports structures are created in the L-PBF process of metals and hence extensive post-AM machining is required in some cases [14].

The common materials that L-PBF offers are aluminum-based alloys, titanium-based alloys, iron-based alloys, nickel-based alloys, copper-based alloys, and steel alloys. The L-PBF process is a much slower process than the binder jetting process; however, its productivity can be increased by increasing the number of laser sources in the machine. Another advantage of L-PBF is the ability to produce complex geometries, a wide choice of materials, the ability to control process parameters for each material and application, and the ability to produce near net shape parts. The disadvantages of L-PBF are that it is a slow process and hence limited lot size can be produced, which is also connected to machine's build volume constraints. [14].

The parts that are manufactured with the L-PBF process, sometimes require post-processing for better mechanical properties of the final product, aimed to remove residual stresses and improve microstructure. The remaining powder that is surrounding the part must be removed, and some form of heat treatment is required to relieve the stress from the final part. The support structures within the part and connecting the part to the build plate must be done to be taken to the post-processing stage. Since the finished parts have a rough surface finish, surface improvement operations are often applied machining. Machining is done not only for a better surface finish but also to maintain tight tolerances for better mechanical fit. Heat treatment is necessary to remove the internal stresses that arise during the L-PBF process. For the manufacturing of injection mold inserts, this heat treatment and quenching must be additionally done in order to achieve the desired hardness of the final part [14].

The L-PBF process has various applications in many industries today. Since the parts produced by L-PBF have a high degree of design freedom and excellent mechanical properties, this process is used in the aerospace, medical, automotive, and tool manufacture industries. Some of the principles, such as topology optimization, help to investigate new design opportunities within various industries. There are many companies that offer this technology commercially today. Some of them are EOS Gmbh, Realizer Gmbh, Renishaw, Sisma, SLM Solutions Gmbh,,Trumpf etc. There are various machines with different specifications and configurations commercially available today produced by these companies [14].

## 2.4 Previous studies and research

In this chapter, previous researches and industry examples of producing IM inserts are discussed. The literature comprises various methods that were used to implement AM in the injection mold development cycle. Range of prototype molds and production molds, that were produced using AM are seen below. It must be noted that most of these examples only talk about the benefits of AM and not the barriers to implementing it.

Dr. Jorge Rodriguez [15] tells us about the use of AM to produce injection mold inserts. In this paper, he establishes the framework that must be followed to adapt AM for injection mold tool production. It is first important to understand the material properties of the mold insert that is going to manufactured by AM and understanding the IM process. Then later, initial and final modified mold designs are made. Then injection parameters must be taken into consideration and studied. Later Moldflow and finite element analysis (FEA) is done to define the injection parameters and to see the amount of deflection in the cavity region. Mold inserts were later manufactured using Verogrey polymer material that comes under the poly-jetting process. These mold samples were tested by injecting different materials in the IM machine. The materials that were injected were polypropylene and nylon. The final number of parts with acceptable quality and the number of shots were documented. Conventional cooling channels were also drilled to implement cooling. For both materials, one mold with cooling and one without cooling was tested. It was found that nylon had a good flow in the mold cavity, but due to its high melt temperature, the mold cavity distorted within two shots. On the other hand, the molds could withstand a greater number of shots of polypropylene but were difficult to achieve easy flow in the cavity area. The Verogrey material was successful in withstanding the clamping force and injection pressure. The cooling channels were of no use because the mold material had very poor thermal conductivity. Even if the cooling channels were kept as close to the cavity wall as possible, it would result in a thin wall between the channel and cavity, which could crack during the injection cycle. He finally concluded by saying that these molds were good as prototype molds and better mold material properties could improve the life of the additive manufactured injection mold.

To know the other advantage of utilizing AM to produce injection mold inserts, the paper published by Shaileshbhai P Patel and Sudheendra.S was taken for reference. As discussed earlier, we know that complex cooling channels that help to increase the efficiency of the mold inserts can be manufactured by AM. This paper shows the methodology and discusses the results through computer simulations. They used Autodesk Moldflow software to simulate and investigate the difference between conventional cooling lines and conformal cooling lines. Conventional cooling channels were initially designed, and three main parameters were noted. The parameters were temperature profile of the part after cooling, cooling time, or time to reach ejection temperature and, finally, the deflection of the plastic part. The two types of cooling channels were compared based on the above parameters. The plastic part chosen was a basic cup like a cylinder shape. The diameter for both the cooling channels was fixed to 4mm. Based on the simulation results, it was concluded that the conformal cooling channel design performed in a better way. The average temperature of the plastic part for the conventional cooling channel design was 53.76, and the average temperature of the plastic part for the conformal cooling channel design was 40.89. It was also seen that the conventional cooling channels cooled the part at 11.68 seconds and the conformal cooling channel at 10.44 seconds. Though the difference in time was less, when looking at the increase in productivity rate, the conformal cooling channels increased the productivity rate in a huge way. The warpage was slightly more in the conformal cooling, but it is said that it depends on the part geometry and would not affect the cooling quality of the channels. Clearly, the benefits of conformal cooling were more. It was thus concluded that conformal cooling channels are better, but we could arrive at a firm conclusion when the only when the mold insert is tested in a realistic environment [16].

The implementation of the conformal cooling and realizing the actual benefit of conformal cooling in a real case scenario was investigated through a student project earlier in ABB, Finland. The design of different conformal cooling profiles was designed and manufactured. These insert samples were later tested in a simulated environment. The cooling test was conducted, and a single design was selected. This insert design was later manufactured and processed for mounting in the actual IM machine. A product case study was done to realize the benefits of the AM process. Six different types of conformal cooling channel profiles like a narrow U-shaped profile, a thick U-shaped profile, a slim spiral profile, a robust spiral profile, a fountain profile, and a conventional drilled profile were designed. This project not only shows the benefits of conformal cooling but also in how to optimize the conformal cooling channels. The designs were manufactured using L-PBF. It was concluded through experimentation that the fountain shaped cooling profile yielded the lowest cycle time that was 10 seconds faster than the mold insert with the conventional cooling channel [11].

Another research paper from Indiana University, USA, and Purdue University, USA, investigated the optimization of the conformal cooling channel and implementing topology optimization to 3D printed stainless steel structured injected molds. In this paper, first, the variables such as the diameter of the channel(D), the distance between the consecutive channels or pitch (d), and distance between the channel and cavity (L) of the conformal cooling channel were optimized by conducting two design of experiments. The initial values of the design variables were set according to the thickness of the plastic part, which was 1.5 mm. Hence according to the thumb rule for conformal cooling, the diameters within a range of 4-8 mm, pitch distance within a range of 2D-3D, and the distance from the cavity within a range of 1.5 D -2D was to be tested. The design of experiments was conducted to understand the behavior and trend of the variables. Once the trend was understood, the channel with the largest diameter (D) and with the smallest pitch (d) between each other was optimized. After finding the optimum values for the best conformal cooling channel design, thermomechanical topology optimization was performed. This paved the way to produce an economical and high performing injected mold insert. Different unit cell designs were tested and simulated in COMSOL, and finally, a design was fixed, which showed the best thermal and mechanical performance. This economical and high performing mold was not manufactured, however. This research paper showed that it is possible to manufacture a high performing injected mold in a very economical manner with up to 47 % reduced mass of the mold insert [17].

Thus, the above research and industry examples provide enough proof that AM could be highly beneficial to produce injected mold inserts. The various areas, such as mold prototyping, high-performance molds, and economical molds through AM, are addressed with an example above. These examples also act as an aid to structure the thesis work by providing proven methodologies and show the various tools that were used.

## 2.5 Objectives and structure of the research

The primary objective of this project is to investigate the AM processes and understand the merits and limitations of each process. The suitability of the AM process that could be beneficial in the IM process is to be identified. Areas such as mold development or mold prototyping and production of mold inserts through AM in an economical way is to be studied.

The benefits of AM processes, specifically Poly-jet, binder jetting, and L-PBF is to be studied. Their suitability to the injecting molding tool production application is to be validated through a real case scenario. The most suitable of the three AM processes are to be selected, and a product case study comprising of ABB's products is to be done. The most suitable AM process in terms of cost, performance, and feasibility is to be selected.

The secondary objective is to make the changes that must be done in the design to suit AM requirements and lastly a physical working prototype is to be manufactured using external suppliers. The critical mold design factors that vary from the conventional design of mold inserts are to be noted. The tools required to verify the performance of AM molds are to be identified and used. The difficulties in implementing AM to the current products would be realized. The final objective is to study the steps to integrate ABB's products to adapt to AM.

The structure of the research comprises an extensive literature study and practical implementation. Two main areas, such as mold insert prototyping and production of mold inserts are addressed.

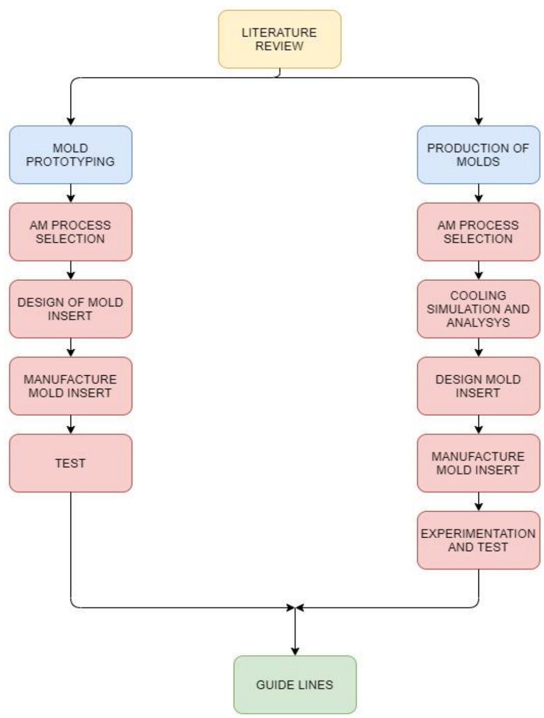



Figure 2.5: Structure of the research

## 2.6 Constraints of the research

There were certain constraints in the research. Some of the constraints are listed below

General constraints:

- Since the mold inserts are product specific, any changes in the overall dimensions of the actual plastic product could not be modified.
- The mold inserts overall dimensions were kept constant because they had to be fit in a mother mold assembly.
- Since the engineering team in the business unit wanted to perform mechanical tests on the product after the IM process, the injected material could not be changed. This directly influences the selection of the AM process.
- AM provides a wide variety of materials. The choice of materials is limited only within that range, and the new development of materials is not possible due to the limited amount of resources and time of the thesis.
- Cooling channels to cool the mold insert could not be implemented since the mother mold design did not have any facility for cooling circuits to be introduced in the mold insert.
- To test the performance of the molds, advanced software was not available. Hence the simulations and analysis could not be done on a very advanced level.
- The samples of the molds that would address the problem of producing AM molds economically could not be tested in the actual IM environment. It would only be a simple experimental setup to understand the thermal behavior of the mold samples.

Budget and time constraints:

- The project also had a budget within which the manufacturing costs of the prototypes and samples had to be fit. Since AM is an expensive technology, design changes in the actual mold inserts could affect the performance and output quality of the final product.
- The time frame of the project was of 22 weeks, which could be extended to 26 weeks but not more than that. It must be noted though AM processes have low lead times to manufacture the products, the external suppliers have job orders scheduled throughout the year, and it would take more than the time to get the design manufactured.

# Chapter 3
## 3.Practical implementation

### 3.1 Case study of a specific product used for IM

As mentioned above, to understand the potential of AM in different areas within IM, a specific product that is being used by the major stakeholder ABB has been chosen. This product is one that is under development for future ABB products and is also something that is predicted to have problems during production in terms of quality. The same product would be used to address different application areas in this project. The results of these applications would be tested in a real-time scenario like other ABB products that are produced today through conventional manufacturing techniques.

ABB control devices produce a vast array of products. One of their major products is in the motor protection and control division. The main products of the motor protection and control division are the ABB contactors. A contactor is an electrically controlled switch device which is used to change the electrical power circuit in many applications such as pumps, motors, air conditioners, cranes and compressors. ABB has a vast range of contactors that has various applications and used in different industries. The latest variant of contactors is the AF technology ABB contactors. The AF range of contactors differs according to their operational power. The AF range contactors are shown in the picture below [18].

.


Figure 3.1: The AF370 is one of the variants of the AF range of contactors [18].

These AF range contactors are an assembly of many plastic components or spare parts. One of the spare parts that are used in the ABB contactor is the contactor bridge. A new range of contactor bridges is currently being developed within ABB. Since this product is also being developed for the first time, the design had to be tested with the actual material that was chosen during the development stage. Polyamide (PA 6) material was the choice of polymer chosen by the R&D team in ABB.   Mechanical tests had to be performed in order to check the functionality of the design. Thus, a prototype mold insert for this product was required.

The geometry of the contactor bridge, which has a non-uniform wall thickness, could have issues in manufacturing when using conventional injection mold inserts. As we discussed earlier, parts with non-uniform wall thicknesses could result in non-uniform cooling, which again leads to part warpage and long cycle times. Thus, there was also a need to investigate whether conformal cooling lines in the molds manufactured using AM could improve the part quality and productivity of the part.

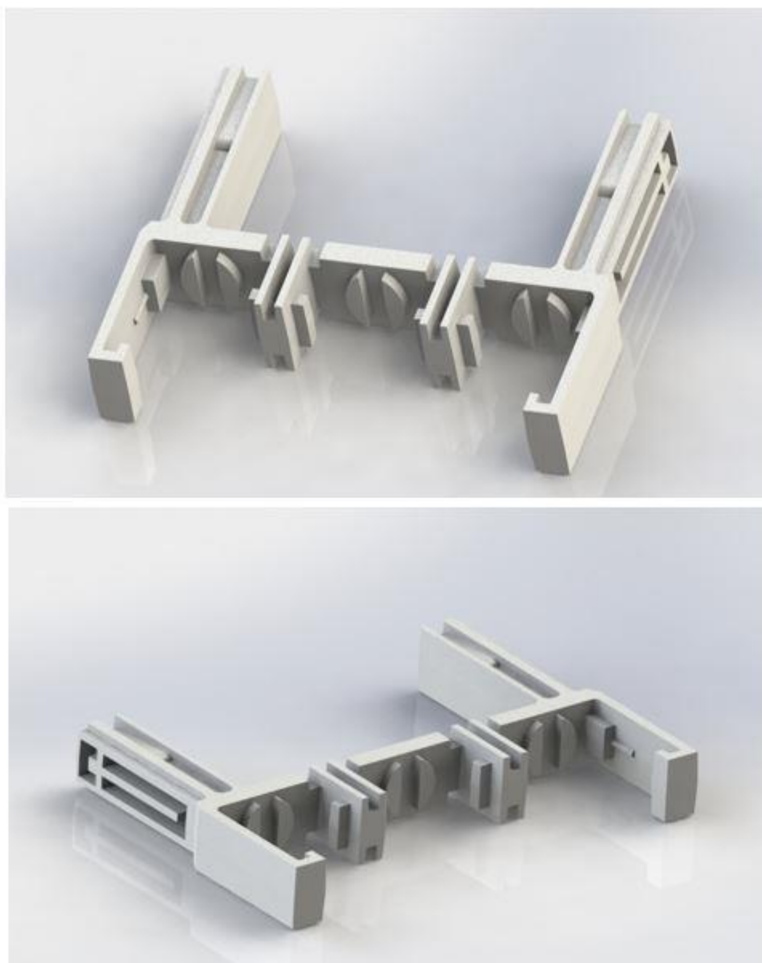


Figure 3.2: 3D view of the Contactor Bridge

The part had to be slightly modified to suit the AM mold development process, which can be seen in chapter 3.4.

## 3.2 AM material and process selection.

The first step in order to implement AM to the mold development cycle is to understand the different advantages and limitations of the AM processes which are suitable for injection mold manufacturing. The most primary aspect that must be taken into consideration is the injected material that is going to be used in the IM process. Since the AM mold inserts have different material choices to be manufactured in, the thermal and mechanical properties of these materials should be able to withstand the injected material's properties. If the AM material fails to withstand the injected material's melt temperature and the clamping force of the IM process, then the number of units that could be produced using the AM mold inserts is affected directly. This also affects the cycle time of the IM process. This helps to classify the AM processes within the different areas of the mold development cycle.

The cost of the AM mold inserts depends on the type of material and can vary from cheap to very expensive manufacturing cost. Depending on the number of parts that must be produced out of these mold inserts, the process must be chosen. If not, it would not make sense to use a very expensive mold insert for something that has to be produced in small quantities or prototyping. It must also be noted that the performance of the mols inserts is also high for the expensive ones. The overall cost, however, does not depend upon this; instead, it depends upon the material cost and manufacturing time of the mold insert. Post-processing operation also depends upon the type of process and material, which is also included in the overall cost of the final mold insert. Thus, it is very important to choose the right process by understanding the production requirements and application of the mold insert. The processing capability must also be understood in a very detailed manner. Some geometries and profiles could not be possible with some processes.

As we discussed earlier in the previous chapter, the material that had to be injected into the mold insert was polyamide (PA 6). The purpose of the initial work of the thesis project was to design and manufacture a mold insert that would be suitable for prototyping. A process that could be suitable for prototyping had to be selected. It must be noted that the prototyping of conventional mold inserts takes around 15 weeks and a currently being subcontracted to a manufacturing firm in China. A solution to this problem was to minimize the lead time of producing a prototype mold insert. The main purpose of the prototype mold insert is to produce samples that could be taken for mechanical testing. The second issue that had to be addressed was to see the suitability for producing a high performing production mold insert that has conformal cooling lines. Each of the three AM processes has its own advantages and disadvantages. Based on their material choices, mechanical, and thermal properties of their final output, they can be assigned to different stages in the mold development cycle. A comparison of how these AM processes differ from one another, and their application is seen below in Table 1. The main factors that must be considered are the suitability of the injection material, the cycle time of the AM mold insert, number of parts that could be produced without distortion of the AM mold insert, and finally, the manufacturing cost.

The processing capability and limitations of producing complex and intricate geometries are also taken into consideration. The final surface finish of the AM mold insert is also very important because it directly affects the part quality and ejection of the plastic part from the mold insert.

| ATTRIBUTE | POLYJET | BINDER JETTING | LASER POWDER BED FUSION (L-PBF) |
|---|---|---|---|
| Injected materials | 1.Polyethylene (PE)<br>2. Polypropylene (PP)<br>3. Polystyrene (PS)<br>4. Acrylonitrile Butadiene Styrene (ABS)<br>5. Thermoplastic elastomer (TPE)<br>6.Glass-filled Polypropylene (PP+G)<br>7. Acetal (Polyoxymethylene [POM])<br>8.Polycarbonate-ABS blend (PC+ABS) | 1.Polyethylene (PE)<br>2. Polypropylene (PP)<br>3.Polystyrene (PS)<br>4.Acrylonitrile Butadiene Styrene (ABS)<br>5.Thermoplastic elastomer (TPE)<br>6.Glass-filled Polypropylene (PP+G)<br>7. Acetal (Polyoxymethylene [POM])<br>8.Polycarbonate-ABS blend (PC+ABS)<br>9.Polycarbonate (PC)<br>10.Glass-filled Acetal (POM+G)<br>**11.Polyamide (PA)**<br>12.Glass-filled Polycarbonate (PC+G)<br>13.Glass-filled Polyamide (PA+G)<br>14.Polyphenylene Oxide (PPO)<br>15. Polyphenylene Sulfide (PPS) | 1.Polyethylene (PE)<br>2. Polypropylene (PP)<br>3.Polystyrene (PS)<br>4.Acrylonitrile Butadiene Styrene (ABS)<br>5. Thermoplastic elastomer (TPE)<br>6.Glass-filled Polypropylene (PP+G)<br>7. Acetal (Polyoxymethylene [POM])<br>8.Polycarbonate-ABS blend (PC+ABS)<br>9.Polycarbonate (PC)<br>10.Glass-filled Acetal (POM+G)<br>**11.Polyamide (PA)**<br>12.Glass-filled Polycarbonate (PC+G)<br>13.Glass-filled Polyamide (PA+G)<br>14.Polyphenylene Oxide (PPO)<br>15. Polyphenylene Sulfide (PPS) |
| Mold material | DIGITAL ABS | 420 Stainless steel infiltrated with Bronze | EOS Maraging Steel MS1 |
| Number of parts that could be produced | Up to 100 | Up to 10,000 | More than 10,000 |
| Cycle time to produce plastic part | Very High | Medium | Very low |
| Cost | Very low | Medium | High |
| Suitable for | 1.Prototyping | 1.Prototyping<br>2. More materials can be injected. | 1.High-performance molds (Conformal cooling)<br>2.Large Volume production |

Table 3.1: Comparison of AM processes for IM application [19]

The poly-jet process offers a material known as digital ABS. This material has high mechanical and thermal properties, which make it suitable for making quick prototypes of mold inserts. However, only a handful of materials can be injected into it. This material has a low heat deflection temperature of 90-95℃ at 0.45 MPA. The heat deflection temperature is required to be high enough to withstand the heat during the IM cycle. The materials that digital ABS could withstand without failing is shown above. Due to high melt temperatures and viscosity, other materials, when injected into the poly-jet mold, would be subjected to high heat, and they would distort the mold cavity of the poly-jet mold insert. Only 10-15 parts can be produced when materials like Polyamide (PA), Glass-filled Polycarbonate (PC+G), Glass-filled Polyamide (PA+G), Polyphenylene Oxide (PPO) and Polyphenylene Sulfide (PPS) are injected into the mold insert. When the rest of the group of materials are injected into the mold insert with the recommended IM process settings, up to 100 parts can be produced. Since there is high heat developed during the molding cycle, the cycle time to produce a unit part is high. The digital ABS material has high machinability. Hence cooling circuits and holes in the mold inserts can be drilled easily. Since the material required to be injected in this project is PA 6, the poly-jet process would not be suitable for manufacturing the injection mold inserts. The digital ABS material is offered by a company called Stratasys, USA. It must be noted that since the poly-jet process was the only polymer AM process taken into the study, other processes like vat photopolymerization was not considered [19,20].

The next AM process that was taken into consideration was binder-jetting. Unlike the poly-jet process, the binder jetting AM process offers materials with better thermal properties. The two materials that are suitable for IM application are 316 stainless steel infiltrated with bronze and 420 stainless steel infiltrated with bronze. They both have high heat deflection properties above 300℃. Out of these two materials, 420 stainless steel has much better mechanical and thermal properties. It has a high hardness value of 97 HRB, which is important when it comes to withstanding high injection pressures and clamping force in the IM cycle. Since there is also a 40% bronze that is infiltrated, it has very good thermal properties. The mold inserts that are manufactured using this AM process allows most of the polymer materials to be injected. The polymers that could be used with the mold inserts of this process is shown above. Up to 10,000 parts can be produced with these mold inserts, and more parts could be produced with optimized IM process settings. Hence this AM process could be ideal for producing prototype parts of wide variety of materials. The advantage of this process to produce mold inserts are the medium manufacturing cost. The disadvantage of this process is that the parts have a relatively low surface finish of 7.5 µm Ra when compared to steel and aluminum mold inserts manufactured using conventional manufacturing processes. This affects the part quality and makes the ejection of the plastic part a little difficult. These mold inserts have high machinability, which allows the capability of drilling holes for conventional cooling channels. Complex cooling channels cannot be manufactured using binder jetting because internal geometries cannot be finished after sintering or infiltration. Since the requirement is to inject a PA 6 material and the binder jetting process allows it, it is selected as an ideal choice to manufacture prototype or short volume molds using this process. The L-PBF process also qualifies to produce prototype molds, but because of its high manufacturing cost, it does not make sense to opt for it. Since the prototype mold is iterative in the mold development cycle, the L-PBF option would increase the development cost. This L-PBF process is more suitable to make production molds, which can produce parts of high volume with complex channels in it [21].

In order to produce a high performing injection mold, insert, the L-PBF process is chosen. The L-PBF process (direct metal laser sintering by EOS) offers many materials out of which EOS maraging steel MS1 and EOS stainless steel CX are the ideal choices for the IM application. Out of these two materials, EOS maraging steel MS1 is a high performing steel material suitable for tool manufacturing. It has better thermal properties and the same hardness as that of the EOS stainless steel CX. With the help of this process, it is possible to manufacture mold inserts with complex geometries and allows the possibility to have conformal cooling channels. Since they are hardened after the sintering operation, they have a high hardness value of 57 HRC. It can be polished to a high degree achieving a very good surface finish of 0.8 μm Ra though lesser than the conventional machining process where a surface finish of 0.05 μm Ra could be achieved. This AM process can also be used to produce hollow mold inserts with cooling channels, thereby achieving high performance and economical injection mold insert [17,22].

### 3.3 Computer-aided design of mold inserts

A mold insert was designed with its overall size based on the size of the mother mold base. The picture of the mother mold base is shown below in figure 3.4. A single cavity mold was to be designed. The design process was done in SolidWorks 2019, using the mold tools suite. Before designing the mold insert, the gate located on the plastic part had to be analyzed. A proper gate location is very important so that there is a uniform fill of the plastic during the IM cycle. In order to analyze the ideal gate location, Moldflow adviser 2019 software was used.

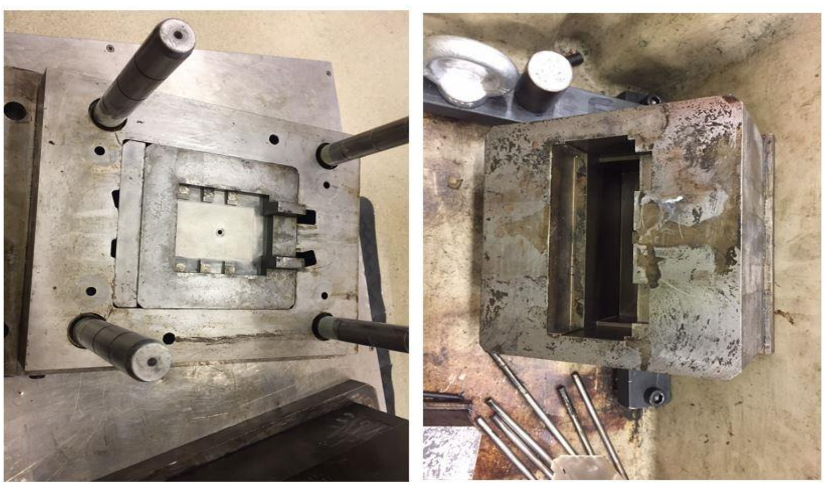


Figure 3.4: Mother mold base

The gate location analysis is done by importing the CAD file into the Moldflow Adviser software and specifying the material from the inbuilt material library. The material used is PA 6. The ideal gate location is shown in figure 3.5 below, which is in the middle.

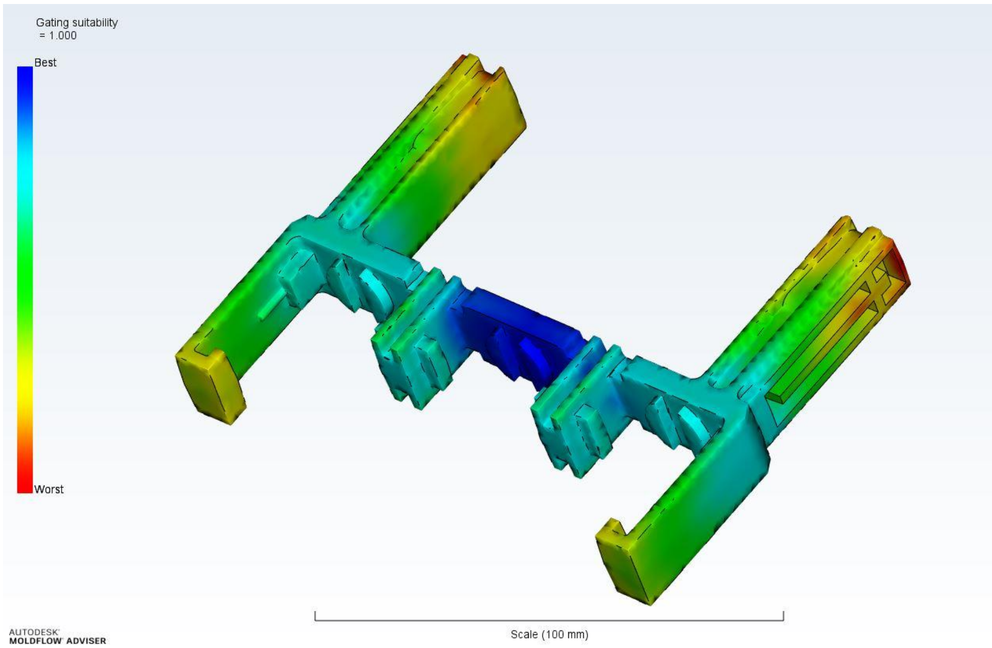



Figure 3.5: Gate location analysis

After fixing the ideal gate location, the parting line must be set. The parting line is very important as it decides the tool split, which helps in visualizing the number of halves the mold insert assembly would comprise of. The parting plane is shown in the picture below in figure 3.6. The parting line runs across the center of the plastic part. After the parting line is fixed, the mold insert design can be done.

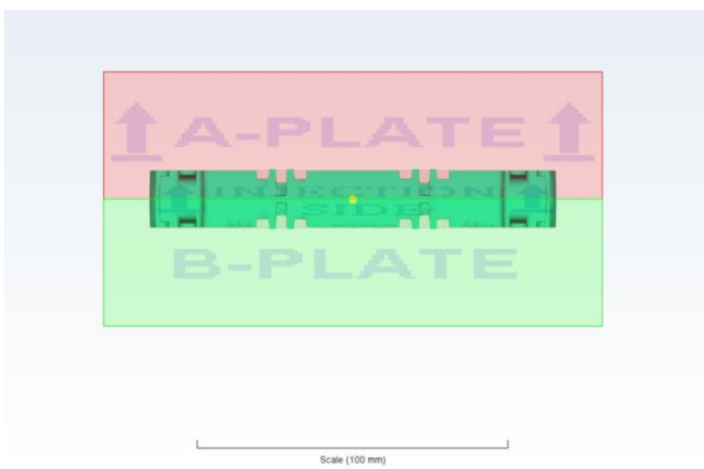



Figure 3.6: Parting line of the mold insert

In the above figure 3.6, plate A shown is the top half of the mold insert, and the bottom plate B is the bottom half. The direction of the ejection of the plastic part is also shown in the figure, which is parallel to the plate A and perpendicular to the parting plane.

After fixing the above parameters, the mold insert assembly was designed. The initial design consisted of a square block. The mold insert assembly consisted of 4 parts namely the top plate, bottom plate and two slider or side cores on each side. The two blocks were symmetrical, and a hot sprue was placed considering the feasibility of the ejection direction. To keep the gate and runner design simple and minimalistic, a cold circular gate and a cold runner was designed. The two side cores were given standard M4 holes so that they could be fit to the mold insert assembly firmly. The holes on the two halves of the mold insert were also provided without threads. Since these threads are difficult to manufactured with accuracy with binder jetting, no threads were provided in the CAD file. These threads could later be tapped, and the mold insert assembly could be assembled.

Though the binder jetting process has a low manufacturing cost, in order to save time and additional costs, a model of the mold insert was first manufactured in PLA material using the fused deposition modeling process. This was done to visualize the mold insert in a better way and to check the fitting of the mold insert with the mother mold base. It took around 15 hours to manufacture the model, and later it was taken to the manufacturing unit to check the interface of the mold insert and the mother mold base. The picture of the prototype model is shown below in figure 3.7.

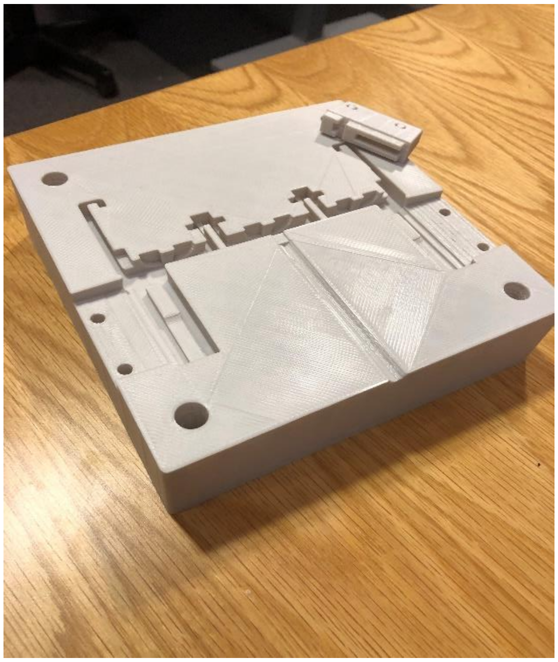


Figure 3.7: PLA model of the mold insert using Fused Deposition Modeling

Once the model was checked with the mother mold base, some modifications to the initial design had to be made. The sprue had to be set to the center of the mold insert. The overall dimensions were modified in order to perfectly fit within the mother mold base. The design iteration could be seen in the figure below.

As discussed earlier, there were some constraints within the scope of this project. One of them was the budget of the project. The mold insert was a big solid block that was quoted with a high manufacturing cost. To avoid this, some of the excess material had to be removed from the mold insert. In this way, a methodology to test an economical prototype design was established. Some of the excess material around the main cavity of the mold insert was removed uniformly. About 40% of the mass was reduced. The thickness around the cavity was reduced. There was some space to form the overall square shape of the mold insert allowing it to still be fit to the mother mold base. There is no specific reason for the chosen value of the thickness, but it was done merely to fit the manufacturing cost within the budget of the thesis work. The new design iteration 2 costed half the price of the previous one.

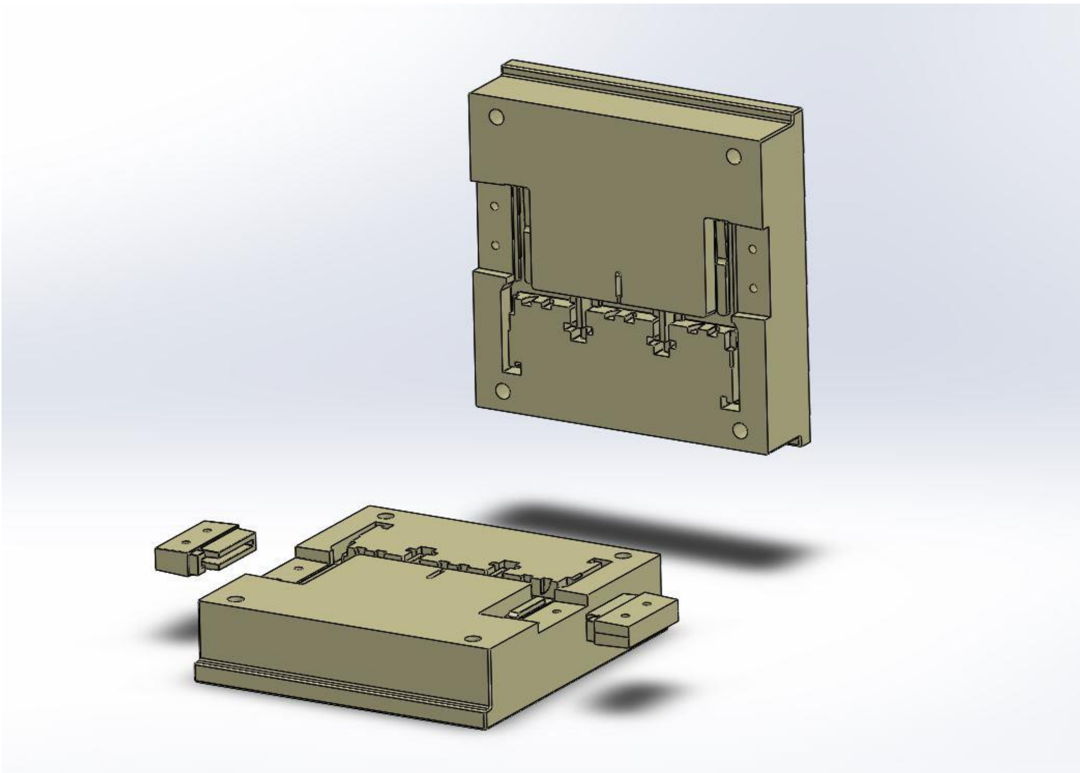


Figure 3.8: Design iteration 1

Some compromises had to be made due to the reduction of material. Since the material has been removed around the cavity area, there could be excessive heat formation around the cavity. This could result in high warpage and poor quality of the plastic part. Hence this insert assembly could not be used as short volume production. Since the prime requirement is to test the overall shape of the plastic part with the intended material, the part quality could be compromised. The molded plastic part would be finished with secondary operations, and mechanical tests would be done on the part. It must be noted that because of the removal of material, no cooling channels could be drilled to implement mold cooling in the mold insert. This also would reduce the mold life considerably. This is an experiment in making and testing an economical prototype mold insert. After observing the results, it could be further optimized, and the ideal amount of mass that could be reduced would be found. Since 420 stainless steel infiltrated with bronze has good thermal and mechanical properties, it would withstand the clamping force of 45 tons and an injection pressure of 22 MPA. These values were simulated from the Moldflow Adviser software. A simple FEM static structural simulation was done where the clamping force was applied on the surface of the holes and the injection pressure on the surface of the cavity with the sides of the mold cavity as supports. The deflection value for one half of the design iteration 2 was 0.013 mm, and for the whole solid block was 0.009 mm. Since the results were satisfactory and keeping in mind that only a few samples up to 50, the mold insert iteration 2 was decided to be manufactured. A picture of the design iteration 2 is shown below in figure 3.9.

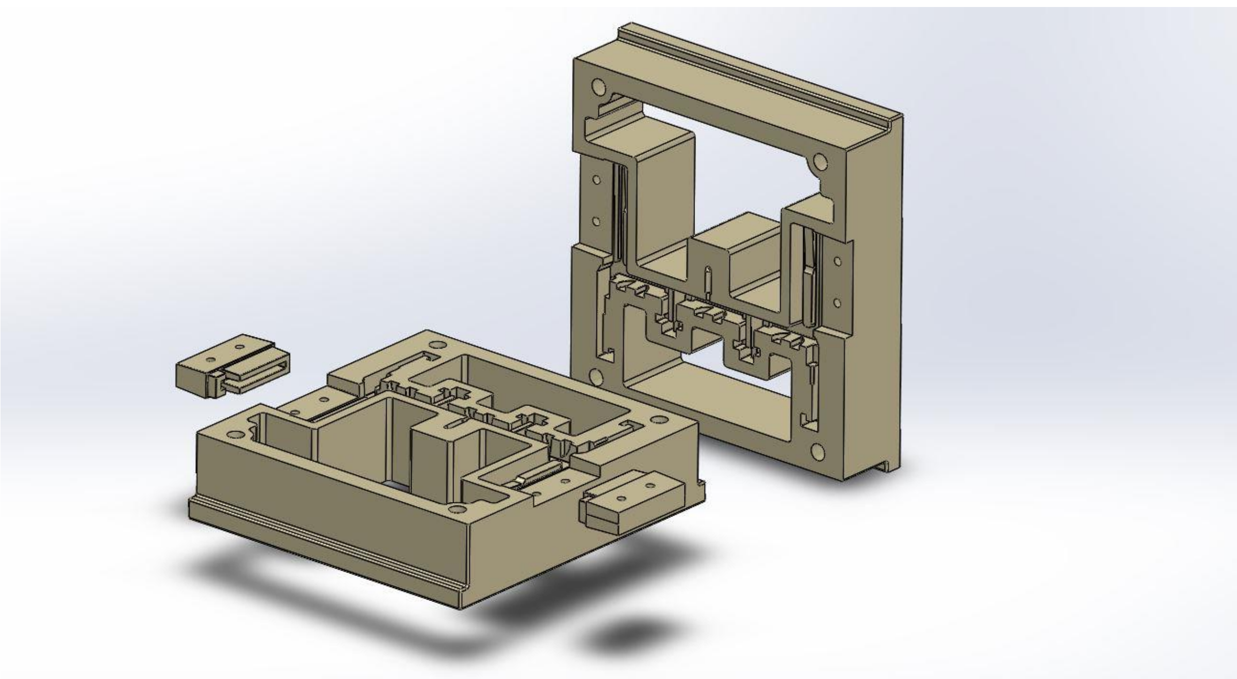


Figure 3.9: Design Iteration 2

The estimated time to manufacture was 15 days, which includes shipping from the USA to Sweden. The design was approved by the external supplier (ExOne), and the CAD file was converted to a .STL file to be processed. The results and the implications behind the manufacturing of such a mold insert with the selected AM process such as binder jetting is discussed in chapter 4.

## 3.4 Design changes for binder jetting process

Before even designing the mold insert for the selected AM process, it is important to understand the process capabilities of the AM technology. In this case, the process capabilities of the binder jetting process were studied. There are certain design rules that must be followed before manufacturing using binder jetting. Some of the design rules are shown below in figure 3.10.

| Feature | Recommendation |
|---|---|
| **Wall thickness** | A minimum wall thickness of 2 mm must be maintained in all the features. This is just to make sure that the green part does not collapse after the binder jetting process. Maintaining a wall thickness greater or equal to this would provide enough strength to the green part when it is removed from the remaining unused powder. |
| **Walls without support** | The wall without support, like ribs or fins, must have a minimum wall thickness of 3 mm. This again for the same reason as the above. |
| **Embossed details** | Fine embossed details should be of 0.5 mm below or above the neighboring surface. |
| **Overhanging structures** | Overhanging structures should not exceed the length of 20 mm. |
| **Corners of walls** | All corners should be filleted with a minimum radius of 0.2 mm. This ensures the green part does not collapse and promotes the removal of powder after binder jetting. The fillets that are not required in the design can be machined after infiltration or sintering. |
| **Hole Size** | The hole size should not be less than 1.5 mm |

Figure 3.10: Design guidelines for binder jetting [23]

These design rules were implemented and verified. One last consideration is the draft angle. Usually, for a conventional injection mold, insert, the draft angle in the cavity region ranges from 0.5 to 1 degree. Since the binder jetting process produces a rough surface finish, it would become difficult to eject the part if the normal draft angle is provided. In order to avoid this, a draft angle of 2 degrees is specified. It is recommended that an AM mold insert should have a draft angle within the range of 2-5 degrees. But in this case, if we increase the draft angle more than 2 degrees, the features of the plastic part would be affected. The draft angle of the plastic part is shown below in figure 3.11. After making the required changes to the CAD model, it is converted into .stl format and sent for processing.

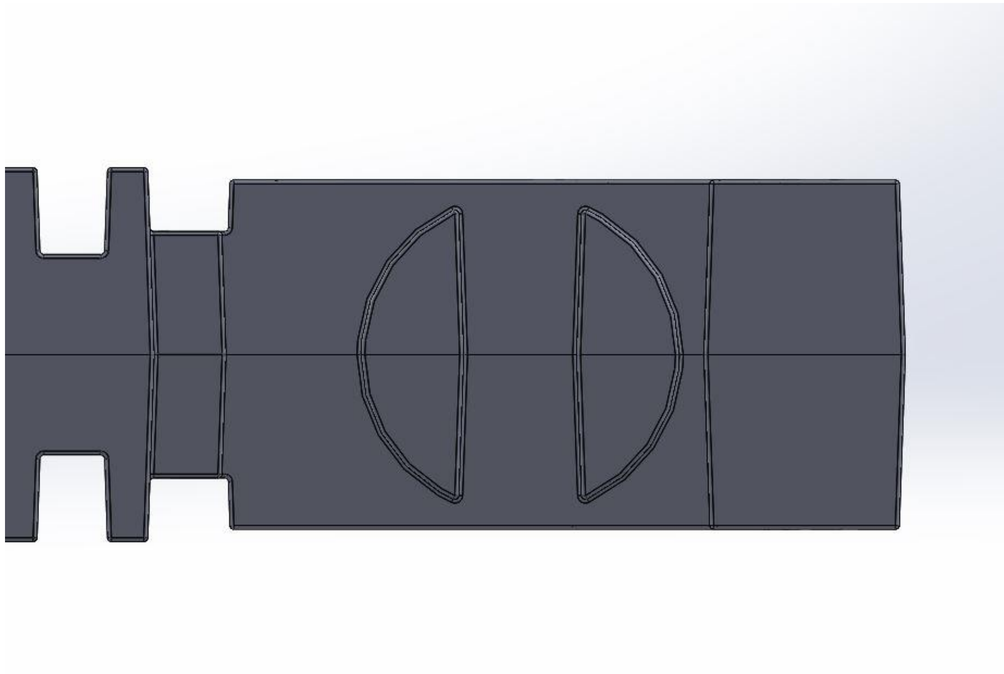


Figure 3.11: The modified draft angle

## 3.5 Design of conformal cooling channels

The process of creating a high performing production mold insert using additive manufacturing was investigated. As mentioned earlier, additive manufacturing opens new opportunities with a high degree of design freedom. This allows the possibility of implementing conformal cooling channels in the mold insert. In order to understand the advantages of the conformal cooling channels, cooling simulations are made. A conventional cooling channel design layout is compared to that of the conformal cooling one. There are certain design guidelines that must be followed to design the conformal cooling channel layout. The distance of the channel from the cavity(d) of the mold insert must be twice that of the diameter of the channel (D). The diameter of the channel depends on the thickness of the part, as shown below in figure 3.12.

| The thickness of the part (mm) | The diameter of the channel (mm) |
|---|---|
| 0-2 | 4-8 |
| 4-8 | 8-12 |
| 8-12 | 12-16 |

Table 3.2: Design rule for conformal cooling [18]

This table, however, is only suitable for parts with uniform wall thickness. The part understudy has a non-uniform wall thickness all along. There exists a wall thickness within the range of 2-6 mm. It is also important that the cooling channel layout fits within the mold insert and covers most of the surface area of the part. If a channel of above 5 mm is used, it becomes difficult to cover the surface area of the part. A cooling channel of very small diameter will lead to difficulty in removing material during the post process of AM. Hence conformal cooling channels with relevant dimensions were made. A conventional cooling channel layout with the same diameter as that of the conformal cooling channel layout is designed. This is done to compare the two layouts with respect to cooling simulations. The two different layouts are shown below.

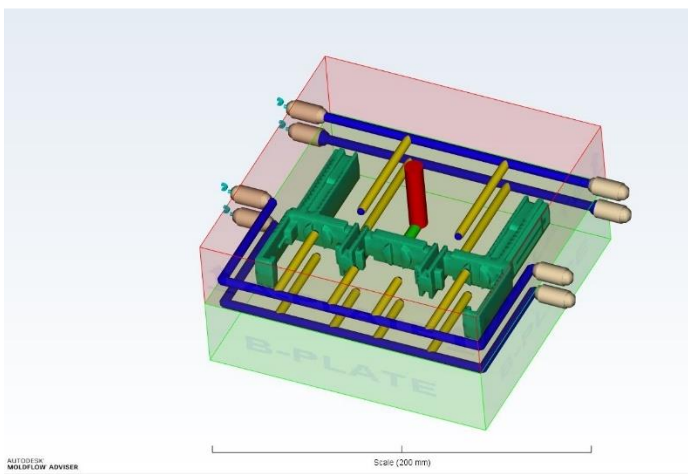



Figure 3.13: Conventional cooling channel design.

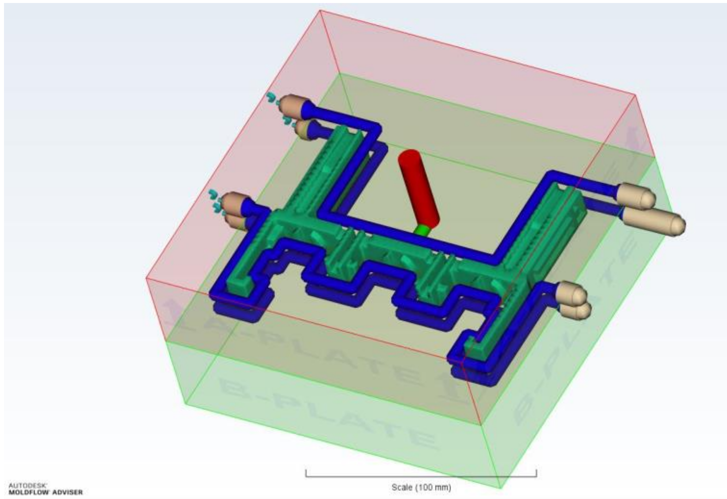


Figure 3.14: Conformal cooling channel design

The simulations were carried out in Autodesk Moldflow adviser 2019.Three different simulations were done in order to analyze the best cooling channel for the product case study. The simulations that were carried are the time to reach ejection temperature, volumetric shrinkage percentage, and temperature distribution on the part. The results and observations from the simulations are discussed in chapter 4.

### 3.6 Economic production of mold inserts.

As mentioned in the AM process selection chapter, L-PBF is the process that could be used to make a high performing mold insert. L-PBF is an expensive process. Hence it is necessary to develop a methodology to design and manufacture the mold insert in an economical manner. This can be done in two ways which are

1. To make only the core of the mold insert with conformal cooling channels in L-PBF and fit it on another platform, which in turn could be taken for the injection molding process. This could help in avoiding unnecessary material usage. Thus, reducing the material cost of the laser melting process.
2. To make a hollow mold insert comprising the conformal cooling channels surrounded by lattice structures. By doing so, the total weight of the mold insert could be drastically reduced. This would reduce the time to manufacture and the material utilized.

The first option is more suitable when the mold insert consists of a core and a cavity. The product under study comprises two symmetrical halves with cavities on both sides. Hence the second option would be more suitable than the first one. Since this has not been implemented before and considering the budget constraint of the thesis work, two small samples were manufactured using L-PBF. One sample is a solid mold block with a cooling circuit and another one being a hollow block with a cooling circuit surrounded by lattice structures. The thermal behavior of the two samples is tested experimentally. A square block was designed with a cooling circuit. A T shaped slot was provided on the top surface to act as the cavity region. The distance between the cavity and the center of the channel circuit was also specified. An illustration of the design sample with the cooling circuit is shown below in figure 3.15

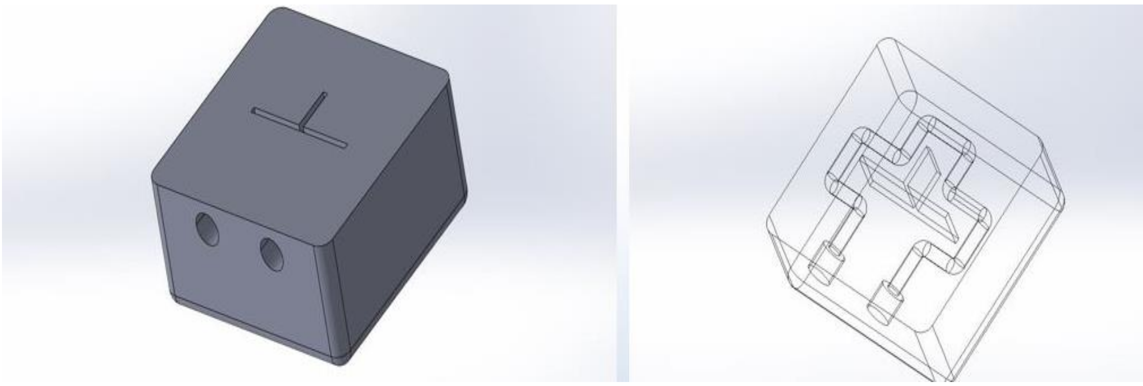


Figure 3.15: Sample of the mold insert with the cooling circuit.

As mentioned above, the two samples were manufactured using the L-PBF process. The main difference between the two samples was that one was a solid mold block, and the other was a hollow mold block with the presence of internal lattice structures. To design the lattice structures, Magics 21.0. The lattice design chosen to fill the hollow structure of the mold insert was G structure3. This lattice design was chosen because the L-PBF process cannot print an angle of fewer than 45 degrees from the horizontal axis. Though there are many lattice designs which satisfy this condition, this particular lattice design is chosen randomly because the intention of this experiment is not to validate the type of lattice design but to make a comparison of a lattice structured insert to a solid one in terms of thermal behavior and manufacturing cost. By implementing this lattice structure with the specified dimension, it was possible to reduce the total weight of the solid block by 40 %. The cost of the lattice structured mold insert sample was also approximately 40% less than the solid mold inserts. The material used to manufacture these mold samples is EOS maraging steel MS1.

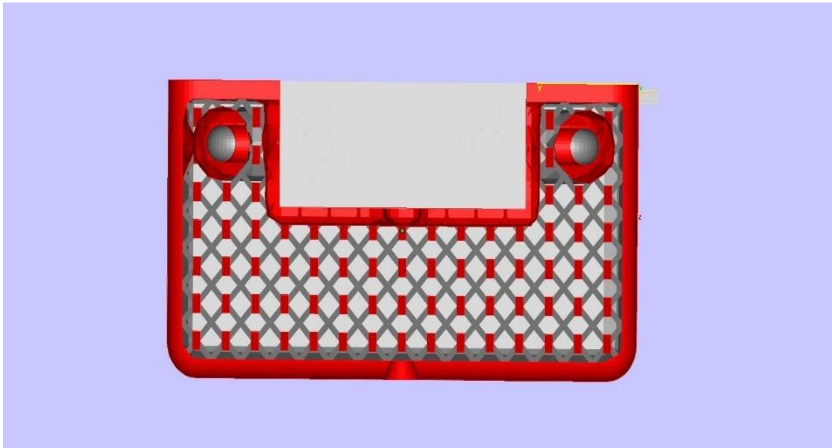


Figure 3.16: Cross-sectional view of the lattice structure mold insert

To test the thermal behavior of the samples, an experimental setup that could be closest to that of the actual injection molding process is arranged. As mentioned in the constraints of the thesis chapter, the mold inserts could not be tested in a real-time injection molding environment. In an injection molding environment, the mold insert is subjected to heat before (pre-heating) and after the plastic is injected into it. Then it is cooled down so that the part inside the cavity could reach the ejection temperature. Hence an experimental setup that could increase the temperature of the mold insert and cool them down rapidly with the help of coolant like water is brought together.

The aim of the experiment is to compare the thermal behavior of the two samples while heating and rapidly cooling down. Two sets of experiments were carried out. The first experiment is to compare the heat conduction of the two mold samples. It is carried out to see which mold sample heats up quickly or reaches a steady-state temperature of 100℃. The mold samples are heated with the help of a heating plate, which is maintained at a constant temperature of 100℃ over which the mold samples are placed. The next experiment is to see which mold sample cools down quickly by cooling it down to a temperature of 40℃. The cooling is done with the help of flowing water through the cooling circuit at a uniform flow rate of 60 milliliters per second. The temperature of the water is constant at 24 ℃. The temperature readings overtime at frequent intervals is measured using an infra-red camera. The point of focus in the camera is placed on the cavity region or T slot.

The experimental setup is shown in figure 3.17. The mold insert sample is placed on a heating plate or disc. The heating disc would increase the temperature of the mold insert sample, and the temperature was captured using an infra-red camera. A water pipe circuit is connected to the inlet and outlet of the cooling circuit in the mold insert sample. This is done with the help of a pipe fixture that was designed and manufactured using 3D printing. The water pipe circuit is connected to a water supply. The pictures of the experimental setup are shown in figure 3.18 and figure 3.19.

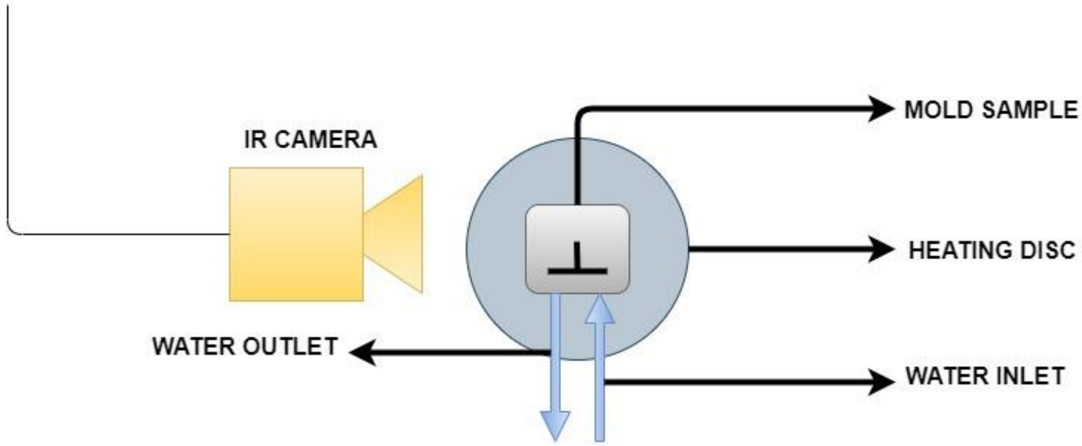



Figure 3.17: Experimental setup [11]

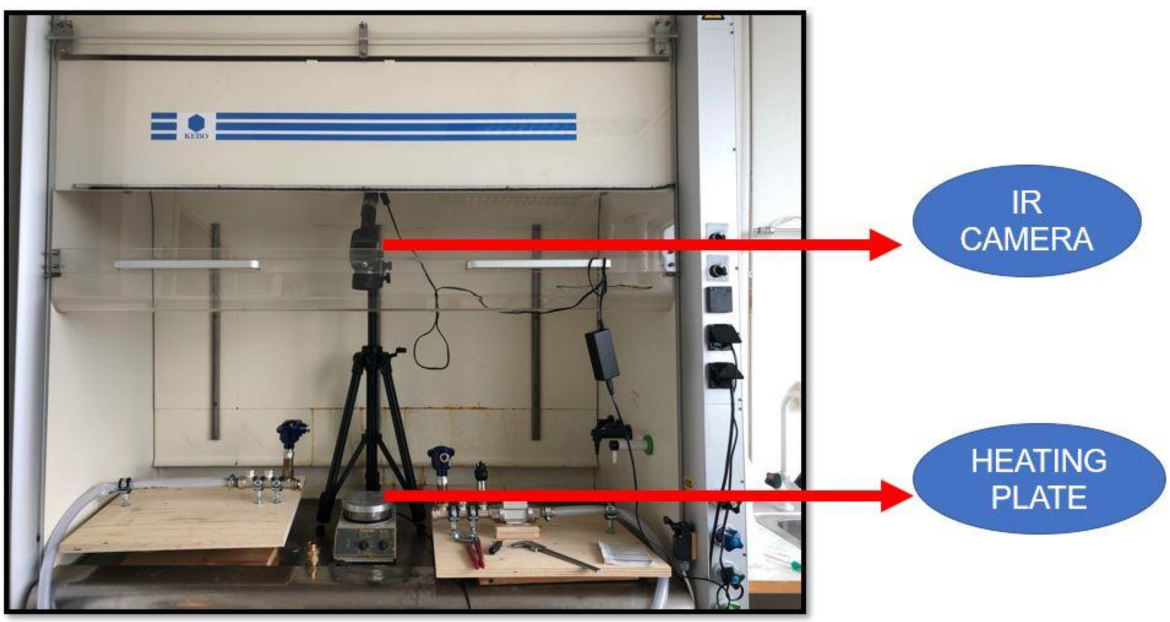



Figure 3.18: Experimental Lab setup

Figure 3.19: Water connection to the mold insert sample.

# Chapter 4
## 4.Results and discussion

### 4.1 Prototype mold insert using binder jetting

As planned before to address the prototyping of mold inserts, a mold insert for the product under the case study was designed. The most suitable AM process chosen to address this application is binder jetting. The mold insert was then manufactured by ExOne, USA, with the 420 stainless steel material. After manufacturing, the following observations were made on the mold inserts. It must be noted that this part is the result of several failed iterations by the supplier to manufacture a mold insert with a dimensional tolerance of 0.2 mm. The manufacturing lead time for the mold insert was initially anticipated to 2 -3 weeks. Due to the difficulties in manufacturing with several iterations, it took around 8 weeks to get the current mold insert. As shown in the picture below in figure 4.1, the mold insert design could not be successfully manufactured. There were warpages and distortions along the sidewalls of the mold insert. The outer profile of the mold insert was mainly subjected to distortion. This distortion did not occur during the binder jetting operation but during the sintering or infiltration stage. The main reason for such a distortion is the size and geometry of the mold insert. Though the mold insert is a medium-sized part, it is still a challenging size to be manufactured using BJ. The main problem occurs during the sintering and infiltration stage. It is in this stage the metallic powder shrinks differently in all directions. As the part size gets smaller, it becomes much easier to control the shrinkage. For applications like mold insert development, it is very vital to have tight tolerances. These tolerances could be in vain when manufacturing mold inserts of big sizes in case of complex geometries. It is uncertain of the size that could be perfectly manufactured. It takes many more iterations and experiments to conclude how big a mold insert could be, to manufacture them to the expected tolerances

.

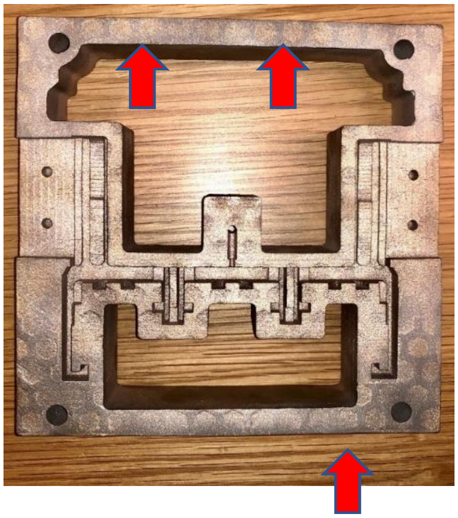


Figure 4.1: Failed part of the mold insert.

The next issue with the manufactured part is the surface roughness. It had a poor surface roughness, which is not convincing for a production tool like injection mold insert. The problem with such kind of surface roughness is that it influences the quality of the part that would be produced out of this tool. It would also become very hard to eject the part after the injection molding cycle has been completed. The surface roughness that was observed is mainly in the cavity region of the part. This surface roughness is due to the decrease in sintering temperature and infiltration. Though such a surface roughness could be improved using surface finishing operations, it would still affect the dimensional tolerance of the part. ExOne used a very low sintering temperature which is not the technology standard. Usually high sintering temperatures are used to get good surface finish. This was done in order to get a control on the tolerance level of the design. Hence it resulted in a very poor surface finish. Since the main problem identified was the size of the mold insert and geometrical complexity, ways to minimize them had to be investigated. This resulted in the design iteration 3. This design is even more simple, and a lot of the unwanted material was removed. This is now designed as a smaller insert, which would be fit to a block machined to the dimensions of the insert in the mother mold base. Design iteration 3 is shown in the picture below in figure 4.3. Since this insert would be fit into another block. The height of the insert was also reduced by 50%. This design modification was approved by the supplier, and the result of the manufacturing is taken for future work.

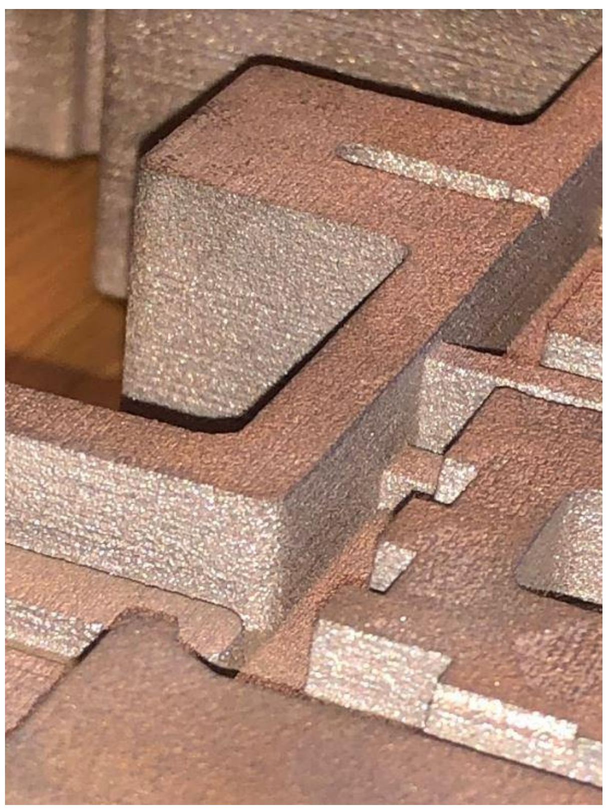


Figure 4.2: Surface roughness of the cavity.

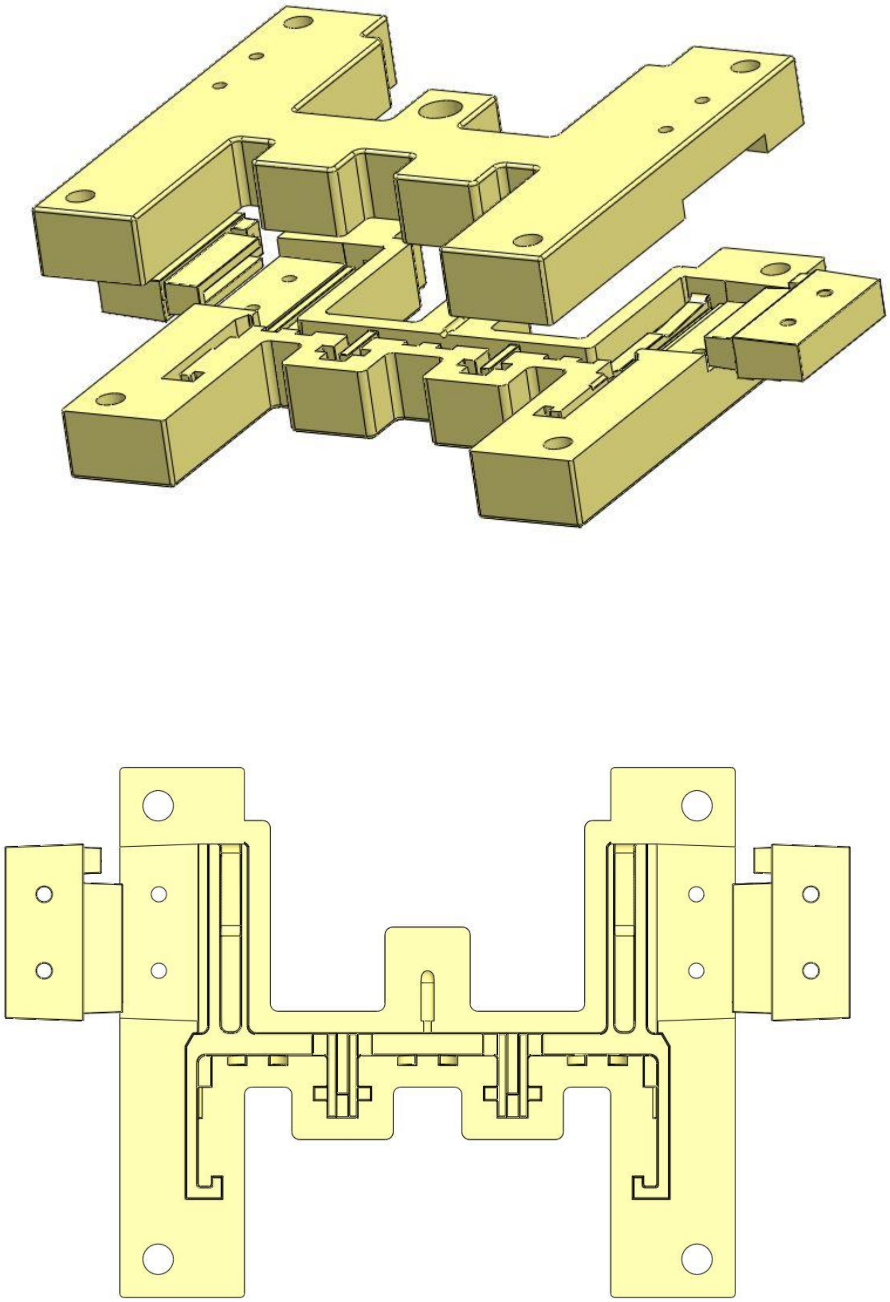


Figure 4.3**:** Design Iteration 3 for the binder jetting process

## 4.2 Cooling simulation comparison and observations

The results of the simulations were obtained, and keen observations were made. The time to reach ejection temperature was the first simulation that was carried out. The time to reach ejection temperature is the time taken by the part to cool down to a specific temperature that allows it to be removed from the mold insert. The cooling efficiency of the cooling circuit plays a vital role in this. The better the efficiency, the lower is the time to reach ejection temperature. The time is observed in seconds in this simulation. When comparing the conventional and conformal cooling circuit, the conformal cooling circuit outperformed the conventional one by a slight margin. The time taken by the part in the mold insert with the conventional cooling circuit was 45 seconds, whereas the time taken by the part in the mold insert with the conformal cooling circuit was 41 seconds. Though the time saved is very low per-unit part, it could have a drastic impact when taken into consideration in the production of parts at high volume. if the cycle time of Conventional cooling is 44 secs and Conformal cooling is 41 secs, then by Conventional cooling 8200 parts per month and by conformal cooling, 8900 parts per month would be produced. 8.54 % increase in production rate. The time difference was still not drastic as expected because the difference in the surface area covered by the cooling circuits is quite low. There could be a bigger difference in time to reach ejection temperature for other parts. But as mentioned above, even a small difference in time could make a big impact on the productivity of the part.

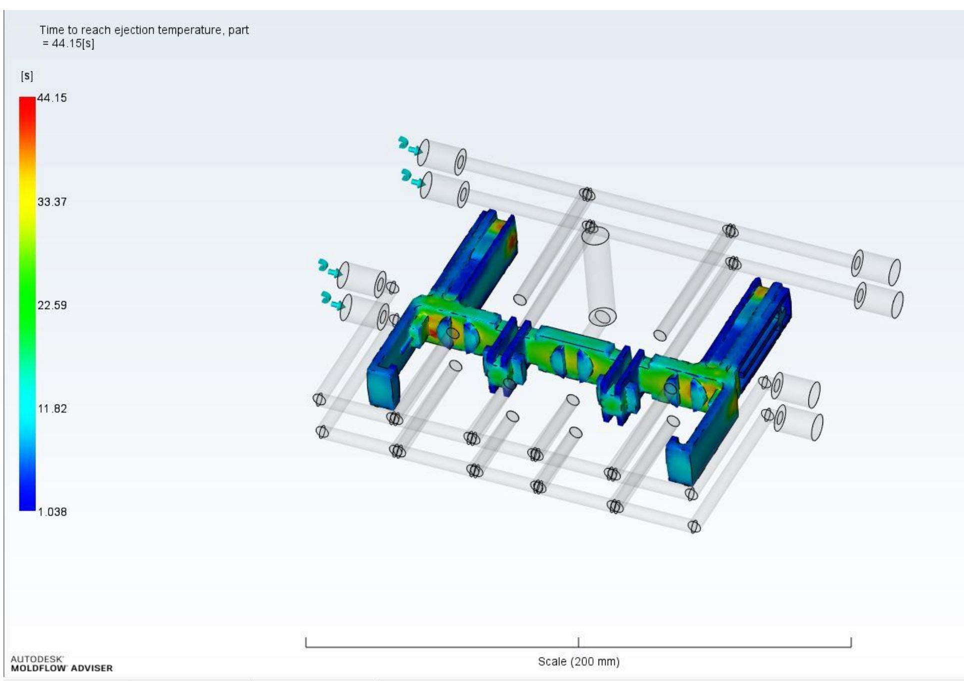



Figure 4.4**:** Time to reach ejection temperature with conventional cooling.

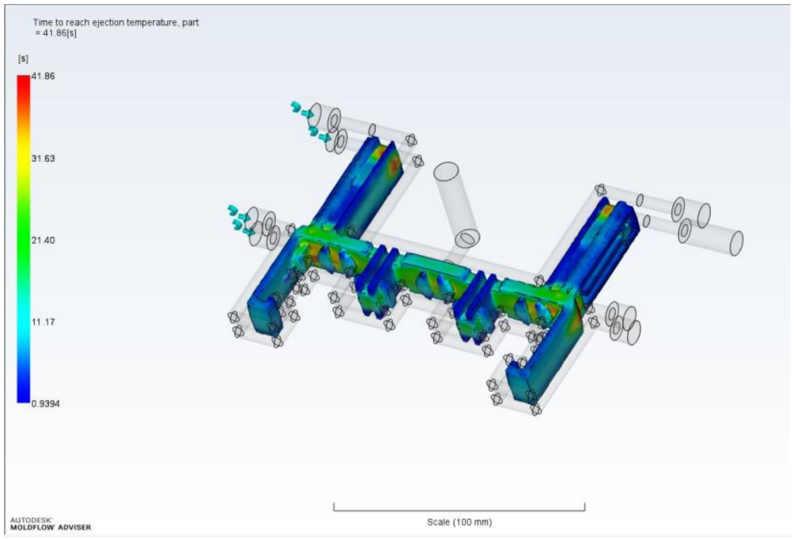



Figure 4.5**:** Time to reach ejection temperature with conformal cooling.

Further the effect of the cooling circuit on the temperature of the part at the end of the cooling cycle was studied. The temperature distribution on the surface of the part can be seen in the figure below. This temperature distribution also depends upon the cooling efficiency of the cooling circuit. When the two cooling channels were compared with respect to this, the conformal cooling channel outperformed the conventional cooling circuit with a slight margin again. The average temperature observed on the surface of the part with the conventional cooling circuit was 70℃, and the average temperature on the surface of the part with a conformal cooling circuit was around 55℃. This is not much of a difference but still an improvement.

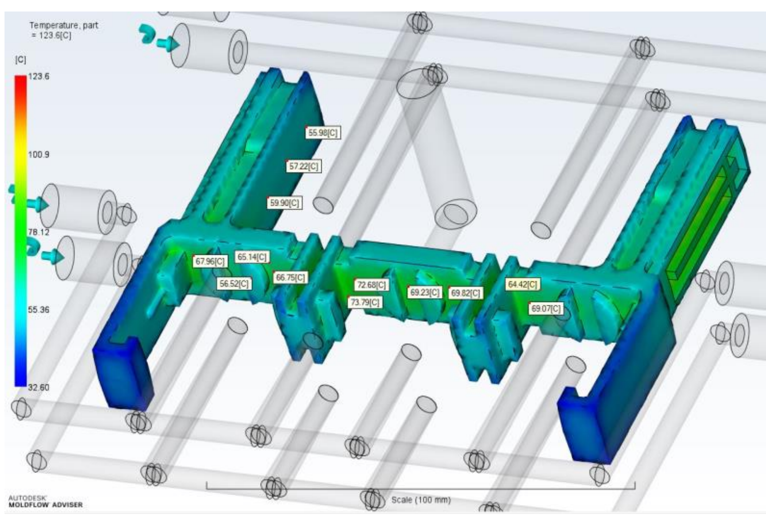



Figure 4.6**:** Temperature distribution on the part with conventional cooling.

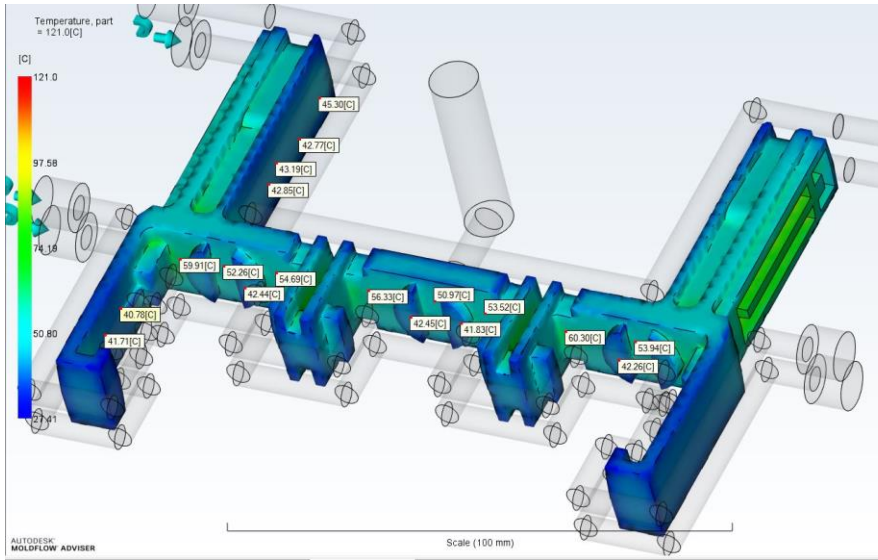



Figure 4.7**:** Temperature distribution on the part with conformal cooling.

Next the volumetric shrinkage in the two mold inserts was studied. The volumetric shrinkage is one of the major factors that lead to warpage in plastic components. The part that has the conventional cooling circuit is subjected to a volumetric shrinkage of 9.75 % and the part with the conformal cooling circuit of 9.73 %. Hence there is no difference in the volumetric shrinkage when comparing both cases. This is because the main reason for high volumetric is uniformity of wall thickness of the plastic part and when the wall thickness of the part is exceeding the nominal range (2 -4 mm). When the part was simulated for wall thickness variation, it exhibited many regions that exceed the nominal range with non-uniformity of thickness. This is shown below in the figure. Thus, when such a part is subjected to cooling, regardless of the type of circuit, it will be subjected to high volumetric shrinkage that will eventually lead to warpage. There will be some amount of heat that will be trapped within the plastic, which cannot be removed no matter how close the cooling circuit is placed near the cavity. This is shown in the figure. Hence in order to attain a better part quality from the conformal cooling circuit, the part must be first cored out and brought to a nominal range of wall thickness. It is also important to design the part with uniform wall thickness to reduce the amount of shrinkage. For parts that have a design constraint of having wall thicknesses, more than the nominal range must be taken for another type of injection molding process known as gas-assisted injection molding process. This process is used for parts with high wall thickness in the automotive sector. Hence the part or product under case study had to be redesigned in order to be manufactured using AM. It is only then it could produce the expected part quality.

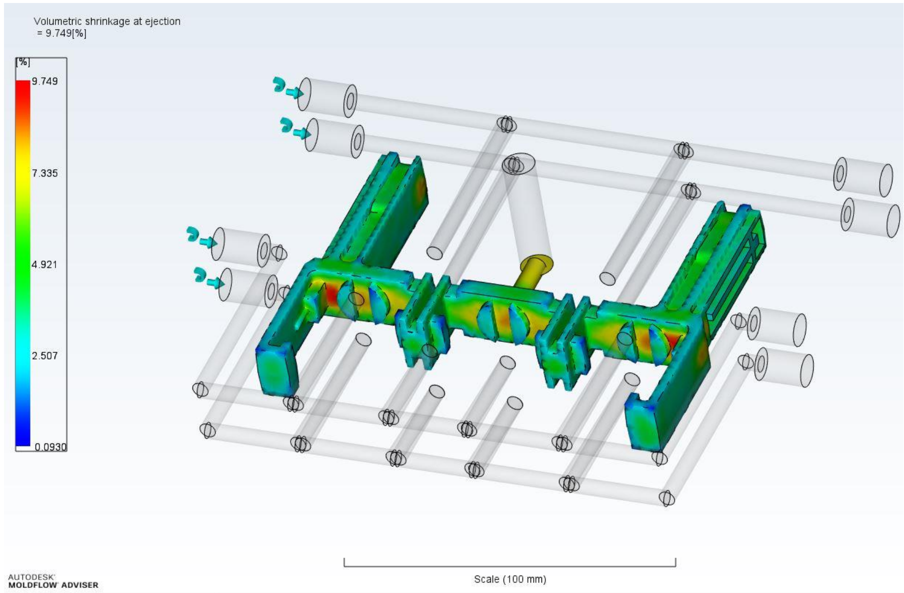



Figure 4.8**:** Volumetric shrinkage of the part with conventional cooling.

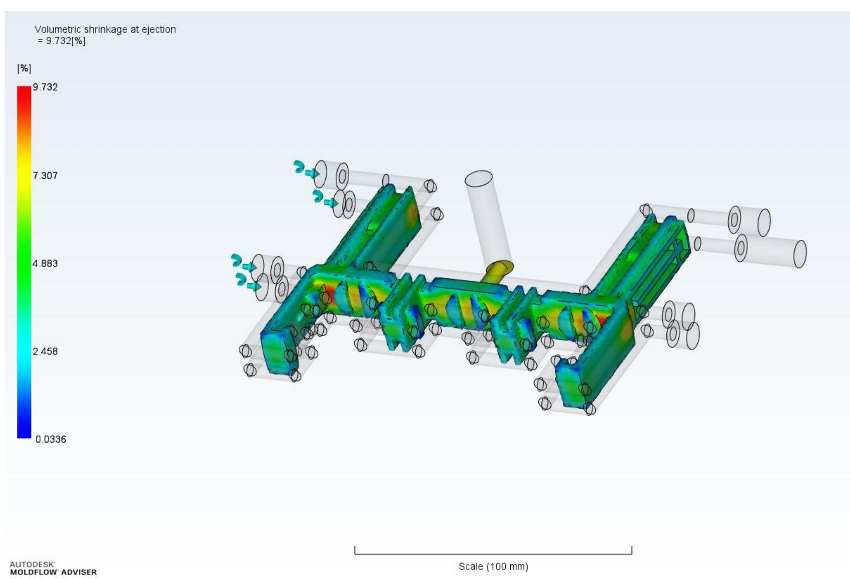



Figure 4.9**:** Volumetric shrinkage of the part with conventional cooling.

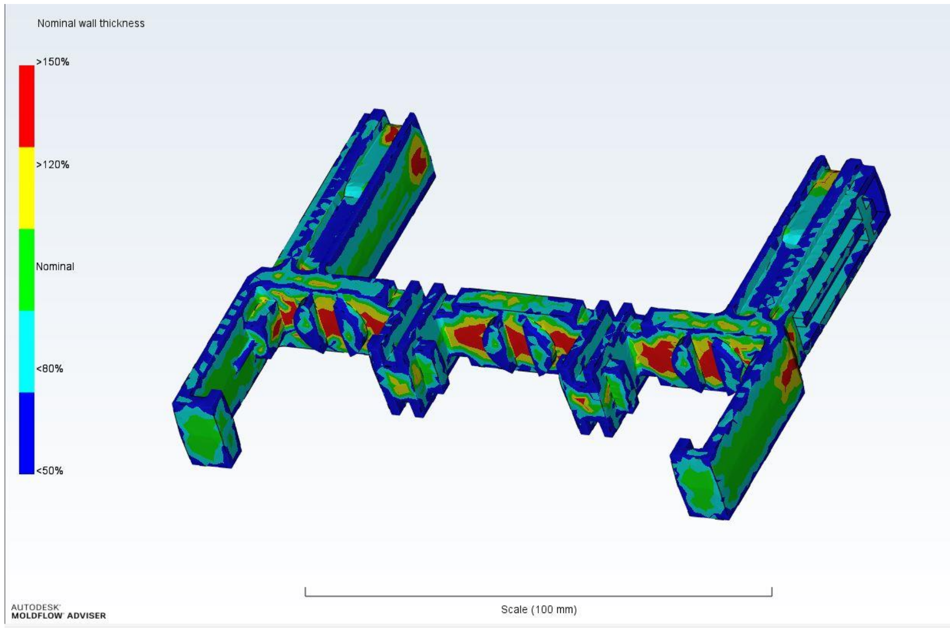



Figure 4.10: Deviations from nominal wall thickness

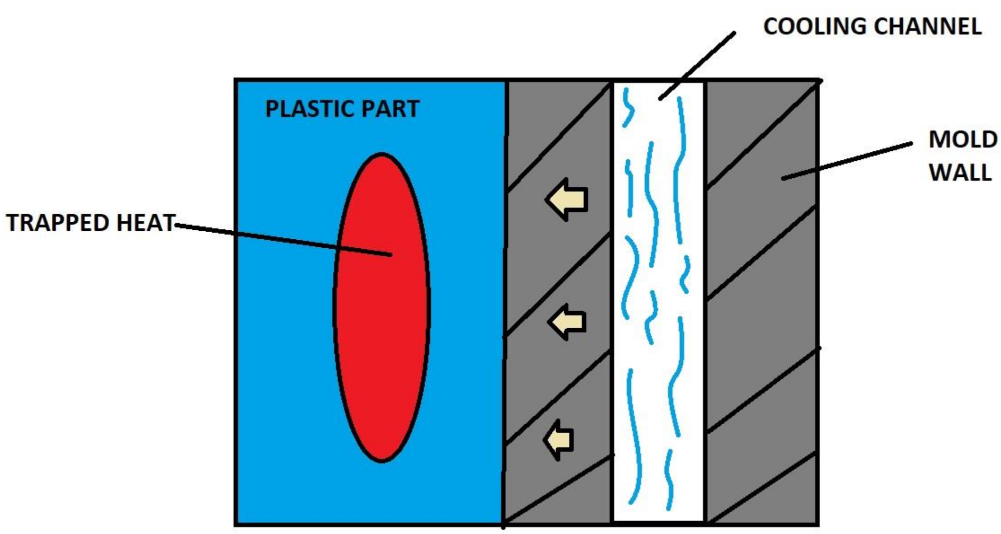



Figure 4.11**:** Representation of thickness effects regardless of cooling.

## 4.3 Experimental results and observations of the economical mold inserts.

The experiments were conducted to compare the thermal behavior of the steel mold insert samples. The solid sample and the lattice filled sample were manufactured by the supplier Amexci AB, Sweden. Both were manufactured in the same material that is EOS Maraging Steel MS1. The results of the experiments can be seen below.

As mentioned earlier, the first experiment was to compare the heat conductivity of the two mold samples. The time comparison taken to heat is shown in figure 4.12. In this experiment, the solid mold gets heated up more quickly than the lattice filled mold sample. The cavity region in the solid mold gets heated faster than the lattice filled mold sample. To cite an example, the solid mold sample takes 180 seconds to reach 80℃, whereas the lattice filled sample takes 320 seconds to reach 80℃. After that, they both reach 100 ℃ and attain a steady-state temperature with a slight difference in time. The solid mold sample heats up more quickly because there is more material present in the solid mold sample for the heat to travel from the bottom of the heating plate up to the cavity or ‘T’ shaped slot. In the lattice filled mold sample, there is much less material present, which makes it difficult for the heat to reach the cavity from the bottom of the mold insert sample. Though the time difference to heat up varies slightly, the lattice filled mold insert sample exhibited satisfactory results by heating up to 100℃ slightly slower than the solid mold sample. This is very impressive for a mold insert that costs 40% less than the solid one. The time to heat is, however, less important than the time taken to cool down. The time comparison taken to cool down is shown in figure 4.14.

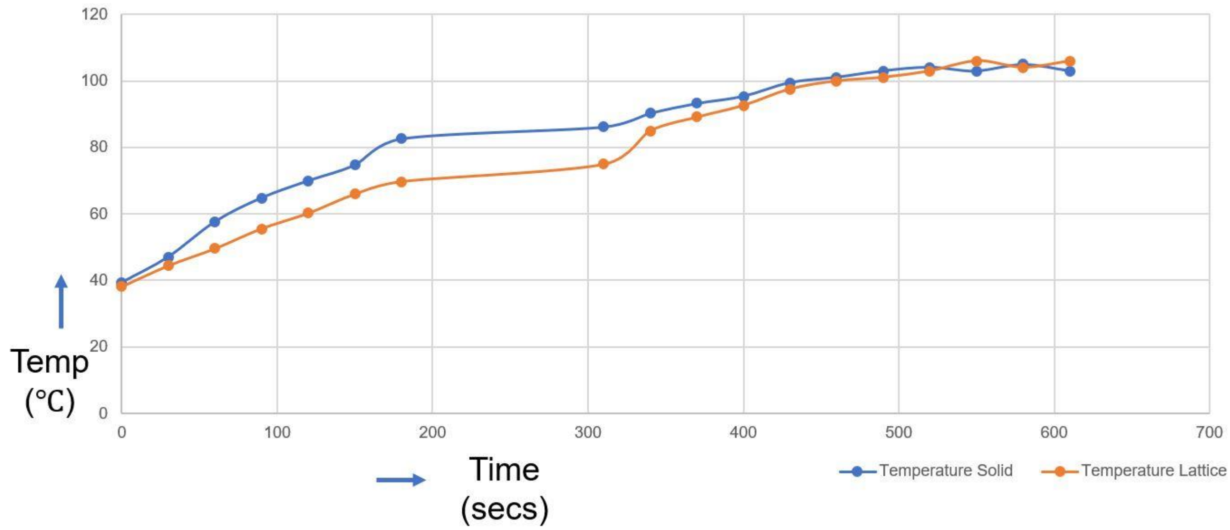



Figure 4.12**:** Time taken by the two mold samples to heat to 100℃

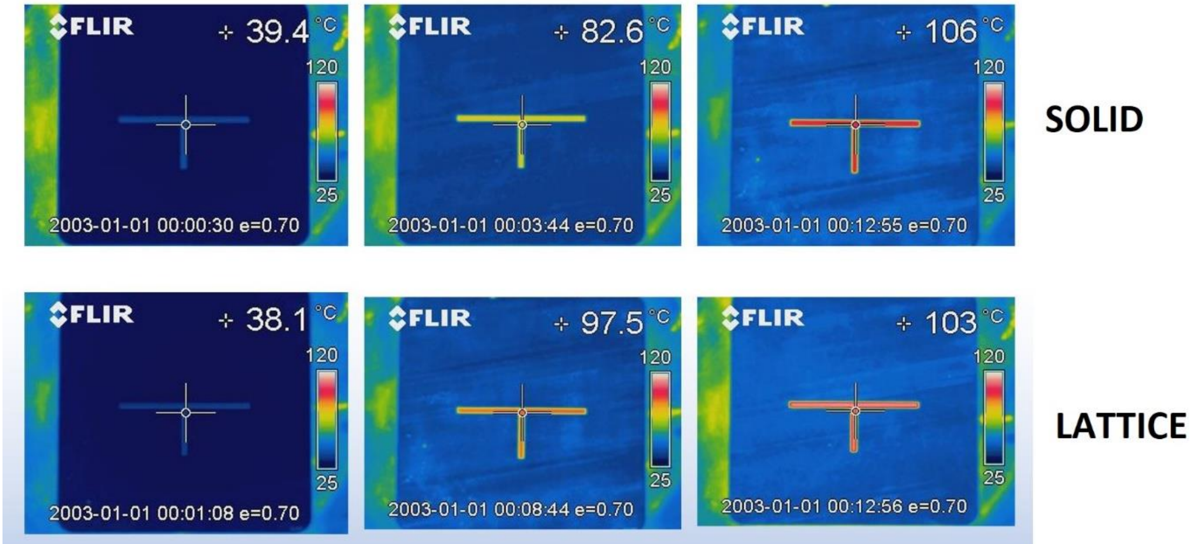



Figure 4.13: Infra-red images of the mold insert samples during heating

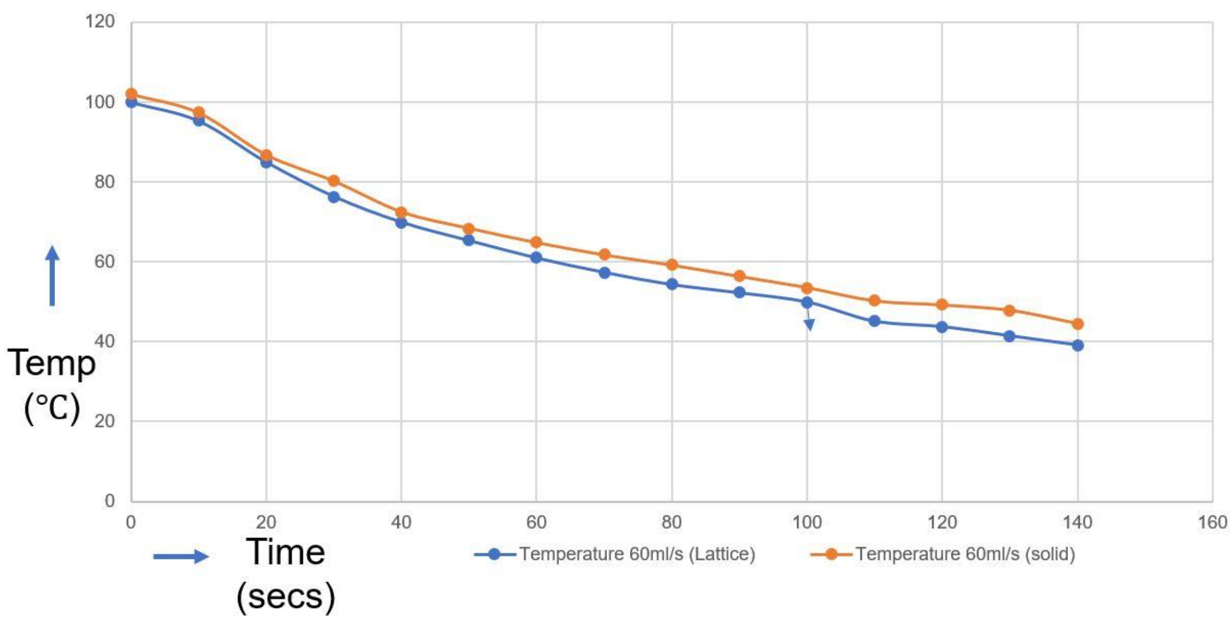



Figure 4.14**:** Time taken by the two mold samples to cool to 40°C

The next experiment was to compare the cooling behavior of the two mold insert samples. The lattice filled mold insert cooled at a much faster rate than the solid mold sample. The coolant used was water at 25°C. The flow rate of water was maintained at 60 ml/s. The lattice filled mold insert exhibited much faster cooling initially, and the time difference between the mold samples increased as the temperature decreased. This is because the heat trapped in the lattice filled mold insert is very less because of the amount of material being low. This enables ease of heat removal from the mold insert. Meanwhile, there is more material and hence more heat that needs to be removed in the solid mold insert sample. Thus, the lattice filled mold insert sample is advantageous when it comes to reducing the cycle time of the unit part being produced in the injection molding process. Any reduction in cycle time will increase the productivity of the overall production in the injection molding process. For example, if the cycle time of solid mold is 40 secs and lattice is 30 secs, then for the solid mold 9000 parts per month and for the lattice mold, 12000 parts per month could be produced. There would be a 33.3 % increase in production rate. Hence it can be concluded that a lattice filled mold insert could be very beneficial when it comes to injection molding. This lattice filled structure is only possible through AM.

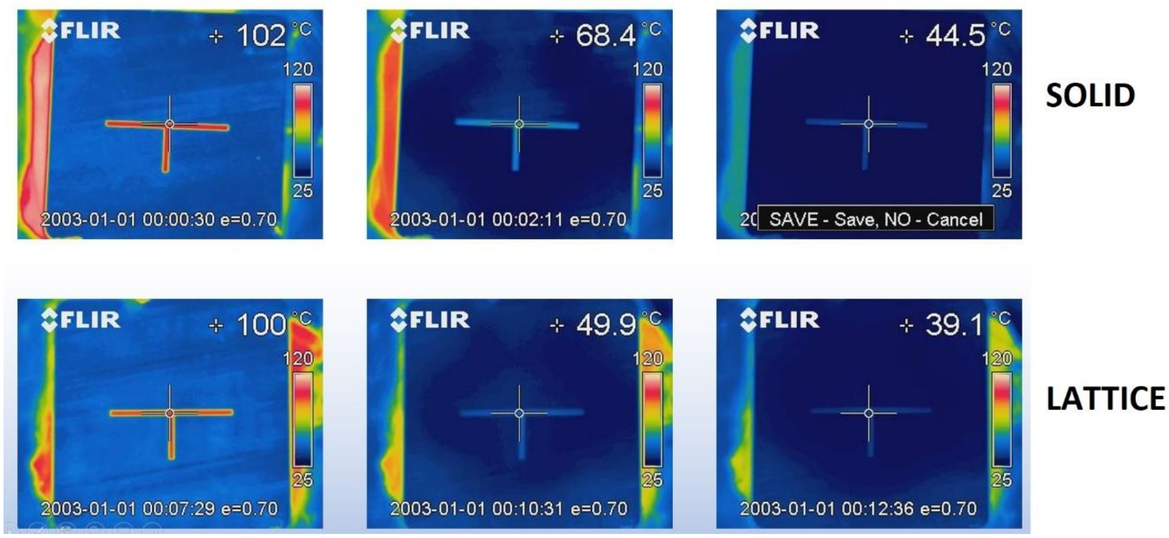



Figure 4.15: Time taken by the two mold samples to cool to 40°C

## 4.4 Conclusions

Results from the different simulations and experiments conducted in this thesis work, provided helpful input in finding the implications that one must rectify before adapting to AM. These implications or actions must be done in order to fully attain the benefits of AM. The different factors one must take into consideration before adapting to AM is analyzed well in this thesis work. The injection mold insert application, like many other applications, also has some compromises when adapting to AM. The areas addressed in this thesis work are concluded in this chapter with future work to be carried out because the research has paved the way to new areas that is not been addressed yet. Earlier in this thesis work, it was stated that there would be guidelines for future ABB products to adapt to AM. By keenly making the observations from the results obtained, it is not possible to conclude on the guidelines to be set already. A few more investigations in the different areas concentrated in this thesis work are required to conclude upon the final guidelines.

The prototype mold inserts that were designed and manufactured using AM needs another cycle of prototyping to be done. The AM process selected was binder jetting, and this process has still not matured enough to minimize design constraints with respect to size. Every design requires many iterations in the prototyping cycle. Hence the design iteration 3 was proposed and would be followed up by ABB. If a successful mold insert could be printed, then a conclusion upon the design guidelines could be achieved.

From the simulations, the conformal cooling channels show better cool results when compared with conventional cooling channels. These conformal cooling channels could be produced only by AM. However, considering the different parameters that lead to a poor quality of injection molded parts like volumetric shrinkage, it was observed that only when the injected molded part is designed according to plastic design guidelines, it could be produced well, regardless of the type of cooling channel present in the mold insert. This takes us back to the product design stage of the plastic part. The design for manufacturability must be taken into consideration. Thus, it must not be mistaken that conformal cooling can increase the production of a poorly designed plastic part. In this case, having non-uniform wall thickness and thickness values exceeding the nominal range. Hence to conclude upon this area, the product under case study or the plastic part must be redesigned in order to fully study the cooling results, and then the part quality can be optimized by optimizing the conformal cooling channels.

The economical mold inserts were studied with the help of an experiment conducted by comparing the thermal properties of two mold insert samples. Solid mold and a lattice mold insert were produced using L-PBF. These experiments were framed by assuming the physics of the real injection molding environment. The results of these experiments proved that the lattice mold insert sample performed much better than the solid mold in both the experiments. However, these results have shown a path to investigate further into this topic. The next step would be to manufacture a full-scale lattice mold insert and test it in the real injection molding environment. The behavior of different lattice structures and thermomechanical optimization could be investigated.

Suggestions for the future work

1. Implement design iteration 3 for the next prototyping cycle in binder jetting and test the mold insert in the real injection molding environment.

2. Core out the product under study and implement conformal cooling in real mold insert using L-PBF technology.
3. Manufacture a full-scale lattice filled mold insert and test its performance in the real injection molding environment.
4. Test new lattice structure and introduce thermomechanical topology optimization in the mold insert.

To conclude, there is much to be investigated, and with the maturity of the technology and the development of new materials in AM, these implications could be made easy, and AM could be more easily integrated into any application.

# References


1. Rees, Herbert (2001). *Understanding the basics of Injection Mold Design.* Carl Hanser Verlag GmbH & Co. KG. DOI: *https://doi.org/10.3139/9783446401839*.
2. Bayer (2001). *Part and mold design.* Bayer Corporation, USA. Retrieved from*:* http://www.mtn.com.tr/faydali/PartandMoldDesign.pdf
3. Domnic.V.Rosato, Donal.V.Rosato, Marlene.G.Rosato (2000), *Injection molding handbook.* Springer USA. DOI: *10.1007/978-1-4615-4597-2*
4. Johnny Boxström (2017). *Improving the product development process of injection molded parts by utilization of additively manufactured molding tool.* Aalto university school of engineering. Retrieved from: https://aaltodoc.aalto.fi/bitstream/handle/123456789/28014/master_Boxstr%C3%B6m_Johnny_2017.pdf
5. Injection molding plastic (2019) *Injection mold cooling design*. Retrieved from: http://www.injectionmoldingplastic.com/injection-mold-cooling-design.html
6. Sandeep Patil, U.M Choudari, Chandrakesava Reddy (2011). *Computer integrated mold split design and reverse engineering approach.* International journal of applied engineering research. ISSN 0973-4562 Volume 6, Number 13 (2011).
7. Robert. A. Malloy (2010). *Plastic part design for injection molding.* Carl Hanser Verlag GmbH & Co. KG. DOI: *https://doi.org/10.3139/9783446433748*
8. Carbon. Inc (2014). *Cost to manufacture injection molding.* Retrieved from: https://www.carbon3d.com/white-papers/injectionmoldingcosts/
9. Ford, Sharon (2014). *Additive Manufacturing Technology: Potential Implications for U.S. Manufacturing Competitiveness*. Journal of International Commerce and Economics. Retrieved from: http://www.usitc.gov/journals.
10. European powder metallurgy association (2015). *Introduction to additive manufacturing technologies.* Retrieved from: http://s550682939.onlinehome.fr/CommissionsThematiques/DocComThematiques/EPMA_Additive_Manufacturing.pdf
11. Matti Mielonen (2016). *Improving production efficiency of injection molding process by utilization of laser method tool inserts with conformal cooling.* Aalto university school of engineering. Retrieved from: https://aaltodoc.aalto.fi/handle/123456789/21048
12. David Homar, Luka Čerče, Janez Kopač, *Cooling simulation of conformal cooling injection mold insert produced by hybrid manufacturing.* Retrieved from: https://pdfs.semanticscholar.org/0adf/f0a6f5c96b86efd7e7aa5f88790056352d03.pdf
13. I. Gibran, D.W Rosen, B. Stucker (2010). *Additive manufacturing. Rapid prototyping to digital manufacturing.* Springer New York Heidelberg Dordrecht London DOI: 10.1007/978-1-4939-2113-3
14. Prashanth Konda Gokuldoss, Sri Kolla,Jurgen Eckert (2017). *Additive Manufacturing Processes:Selective Laser melting and binder jetting guidelines.* Retrieved from: https://www.researchgate.net/publication/317620828_Additive_Manufacturing_Processes_Selective_Laser_Melting_Electron_Beam_Melting_and_Binder_Jetting-Selection_Guidelines
15. Jorge Rodriguez P.E (2016). Use of additive manufacturing (AM) for mold inserts in Injection molding. Western Michigan University. Retrieved from: https://www.asee.org/public/conferences/64/papers/17301/view

16. Shaileshbhai P Patel, Sudheendra S. (2017). *A simulation study of Conformal Cooling Channels in plastic injection molding*. Flora Institute of technology. Retrieved from: https://www.ijedr.org/papers/IJEDR1701017.pdf
17. Suchana. A. Jahan, Tong Wu (et al) (2016). *Implementation of conformal cooling and topology optimization in 3D printed stainless steel porous structure injection molds*. Elsevier B.V. Retrieved from: https://www.researchgate.net/publication/309891860_Implementation_of_Conformal_Cooling_Topology_Optimization_in_3D_Printed_Stainless_Steel_Porous_Structure_Injection_Molds
18. ABB Contactors Brochure, *retrieved from* http://hankooksystem.com/userData/ABB_MC.pdf
19. Gret Goor (2017). *ABB Capex engineering team kickoff with new members working group prototyping.*
20. Stratasys white paper, Polyjet in injection molding. Retrieved from: https://www.stratasys.com/resources/search/resource-guides/polyjet-injection-molding
21. Exone materials datasheet. Retrieved from: https://www.exone.com/ExOne/media/MaterialsBinders/Data%20Sheets/Metal/X1_MaterialData_420SS_062519.pdf
22. EOS materials datasheet. Retrieved from: http://ip-saas-eoscms.s3.amazonaws.com/public/1af123af9a636e61/042696652ecc69142c8518dc772dc113/EOS _MaragingSteel_MS1_en.pdf
23. Alkaios Bournais Vartosis(2017). How to design parts for binder jetting 3D printing. 3D Hubs. Retrieved from: https://www.3dhubs.com/knowledge-base/how-design-parts-binder-jetting-3d-printing/