Bachelor's project

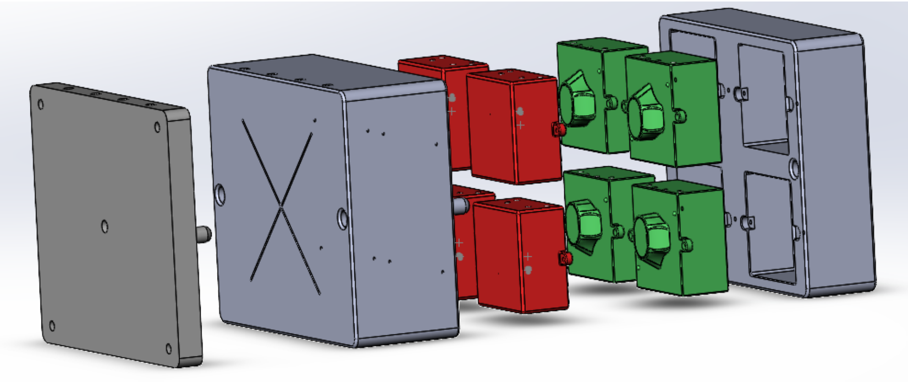
NTNU
Norwegian University of Science and Technology
Faculty of Engineering
Department of Mechanical and Industrial Engineering

Anders Brunsberg Mariendal
Ole Kristian Elnæs

Development of injection molding tooling

3D-printed injection molds for medical equipment

Bachelor's project in mechanical engineering
Supervisor: Sotirios Grammatikos
May 2021



NTNU
Kunnskap for en bedre verden

Anders Brunsberg Mariendal
Ole Kristian Elnæs

Development of injection molding tooling

3D-printed injection molds for medical equipment

Bachelor’s project in mechanical engineering
Supervisor: Sotirios Grammatikos
May 2021

Norwegian University of Science and Technology
Faculty of Engineering
Department of Mechanical and Industrial Engineering

NTNU
Kunnskap for en bedre verden

Sammendrag

Denne bacheloroppgaven omhandler utviklingen av en sprøytestøpingsform til et gjenbrukbart munnbind i medisinsk silikon som skal produseres ved hjelp av 3D-printing. I fokus av utviklingsprosessen står modularitet, dette for å kunne legge til rette for rask omlegging av sprøytestøpingsformer. Også i fokus for oppgaven er utviklingen av et rammeverk/metode for hvordan å gå fram for å utvikle en sprøytestøpingsform som skal produseres ved hjelp av 3D-printing.

Oppgaven er en del av imPURE-prosjektet, et internasjonalt samarbeid som jobber for å bedre ruste Europa for både nåværende og fremtidige kriser som eventuelt kan kreve rask omlegging av produksjonslinjer for å supplere tilfredsstillende mengder av medisinsk utstyr i plast.

Arbeidet med å utvikle sprøytestøpingsformen for munnbind i medisinsk silikon er ikke fullført, men datamodeller er laget. Termisk analyse er utført på datamodellene, samt sprøytestøpingsanalyse. I tillegg er det også blitt utviklet et rammeverk som detaljerer utviklingsprosessen.

Abstract

The aim of this thesis is the development of an injection mold for a reusable facemask in medical silicone to be produced using 3D-printing. At the focus of the development process is modularity, in order to facilitate rapid reorganization of injection molds. Also in focus for the thesis is the development of a framework/method that lays out the process of developing an injection mold that will be produced using 3D-printing

The thesis is part of the imPURE project, an international collaboration that works to better equip Europe for both current and future crises that may require rapid repurposing of production lines to supplement satisfactory quantities of plastic medical equipment.

The work to develop the injection mold for a facemask in medical silicone has not been completed, but CAD-models (computer-aided design models) representing a conceptual design have been created. Thermal analysis has been performed on the computer models, as well as injection molding analysis. In addition, a framework has been developed that details the development process.

Foreword

This thesis is written by two mechanical engineering students for NTNU Gjøvik, as part of the imPURE project. It was a good opportunity for the candidates to learn valuable information regarding injection molding, 3D printing, computer aided design and also teamwork.

With that said we would like to take this opportunity to thank some key people that has been of tremendous help and assistance.

Thanks to,

- Professor Sotirios Grammatikos for superb supervision and guidance.
- Pål Erik Endrerud for valuable insight regarding 3D-printing and help regarding 3D-scanning

Table of contents

Figure list

Table list

Terminology

What	**Explanation**
AM	Additive manufacturing
Baseplate	The plate into whose cutouts the mold/inserts are fastened
CAD	Computer-aided design
Cavity	The female component of the mold insert: the component of the insert that has an indent which forms the outside of the molded part
Core	The male component of the mold insert: the component of the insert that has a protrusion which forms the inside of the molded part
FDM	Fused-deposition modelling
Gate	The component through which the plastic/liquid is injected into the mold/insert
Injection molding	Creating a plastic part through injecting plastic into a mold with a cavity and a core
Insert/mold	The cavity and core
LIM	Liquid injection molding
LSR	Liquid silicone rubber
Runner	The feeding system that connects the gate(s) to the sprue
SLS	Selective laser sintering

1 Introduction

1.1 Background

The main part of this project is the development of a mold for injection molding based on modularity and 3D-printing, is a part of the “imPURE”-project, whose mission is the repurposing of injection molding tools/machines for medical supplies, with the utilization of 3D-printing at its core. (*imPURE*, n.d.)

The imPURE project is a response to the fast-evolving nature of the modern world and aims to transform industrial plastic processing lines in Europe as a way to better enable quick repurposing of mainly injection molding machines to produce medical supplies. If this aim is achieved, both Europe and the world might be better prepared for the next pandemic that comes humanity’s way. (*Objectives*, n.d.)

As the imPURE project is international, it was decided, in consultation with the candidates’ supervisor, professor Sotirios Grammatikos, that it would be beneficial for this thesis to be written in English.

1.2 Problem statement

To develop, design and simulate, as well as 3D-print, the modular mold of a respiratory facemask in medical silicone to be produced through liquid injection molding; and establish a method of approach for the process. In addition to that: to develop, design and simulate the baseplate into which the mask molds will be inserted, with a focus on modularity.

1.3 Alterations on the thesis’ assignment

Originally, this thesis was based on an assignment which, summarized, was this: to develop and produce a plastic prototype mold for injection molding, in order to find a more cost-effective way of developing molds for injection molding. However, the candidates and their supervisor were unable to attain contact with the company with which the assignment was in cooperation with during January of 2021. For that reason, it was decided both by the candidates and the supervisor to alter the assignment for the thesis slightly.

The altered assignment of the thesis became the merging of two assignments. The second assignment was to design the modular mold inserts of a facemask to be produced through injection molding, as well as designing the baseplate.

1.4 The BergaMASK and its design

The BergaMask is a relatively new product, created by the company Stil Gomma. It was first introduced in March of 2020, just as the threat of the COVID-19 pandemic had resulted in dozens of lock downs and/or strict policies in countries all around the globe. (*BergaMASK®: the story*, n.d.)

The mask is made of medical-grade silicone and is meant to be reusable and long-lived. Filters need to be changed regularly. Exactly how often depends on the situation. A healthcare worker exposed to COVID-infected patients might need to change the filter up several times a day.

Figure 1 shows the design of the facemask. The CAD-file for the mask was received on 01.03.2021.

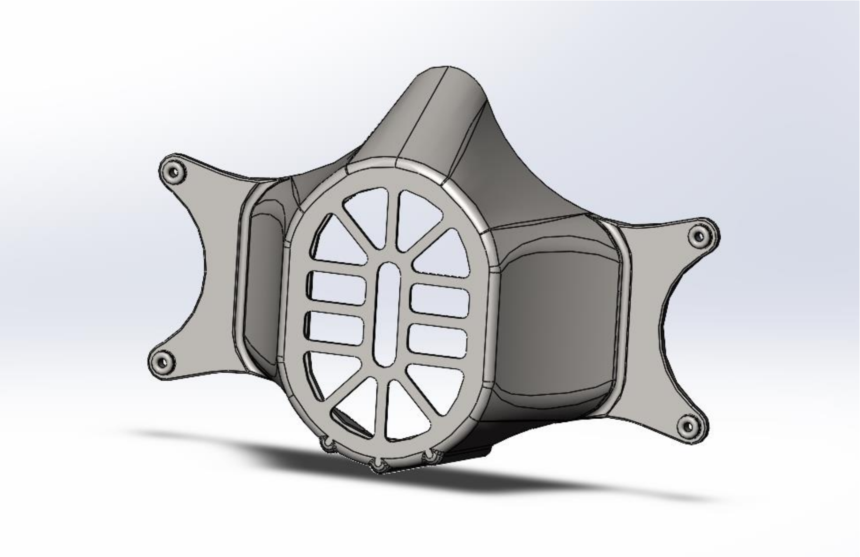


Figure 1: The design of the BergaMask

For the sake of clarity, the mask has been divided into four section that will be referred to in the thesis. See figure 2 for an illustration of these sections. The sections are:

- Filter cover: the section marked in red.
- Nose section: the section marked in blue.
- Side pieces: the section marked in green.
    - The side pieces have the thickest cross-section.
- Mask body: the remainder of the mask, the section marked in grey.

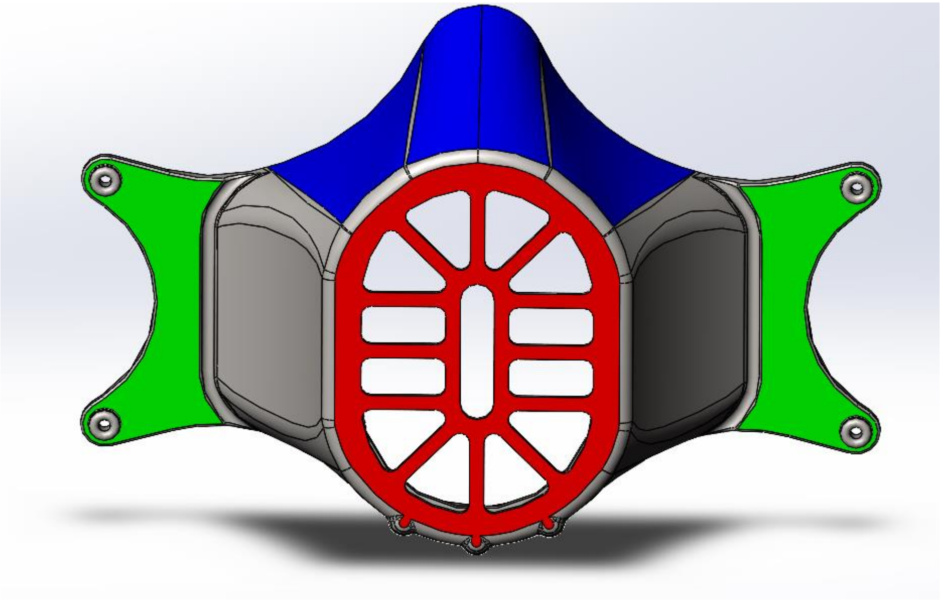


Figure 2: Sections of the face mask

1.5 Assumptions

1.5.1 Injection molding machine

The injection molding machine that the mold is supposed to be used has not been available during the work period of this thesis. Neither is it clear what machine brand and model is going to be used. Early in the thesis an assumption of available space for the mold in the injection molding machine was given: 550mm x 650mm x 700m (height x width x length). The dimensions of the mold is based on the given assumption.

2 Theory

2.2 Materials

2.2.1 Mask material: medical silicone

Medical silicone products are silicones that are biocompatible. The biocompatibility makes it highly applicable, and applications might be dental equipment, menstrual cups, facemasks, cell phone cases, plastic bags for food or food containers. (*What is silicone?*, n.d.)

Silicone rubber is an elastomer consisting of a siloxane bond. This bond has a higher strength than what is typical for other elastomers. At the same time, the intermolecular forces are low, which results in a strong and durable material which at the same time is highly elasticity and flexible. In addition to this, medical silicone offers chemical stability in that it is chemically inert, resistant to bacteria growth and hypoallergenic. All these are properties that makes it a suited material for a face mask. (*Liquid silicone rubber (LSR)*, n.d.)

2.2.2 Plastic mold material: PA2200

PA2200 is a variation of Polyamide 12, also known as nylon 12. Polyamide 12 is a thermoplastic made from either ω-aminolauric or laurolactam monomers that have 12 carbon atoms. In terms of mechanical properties polyamide 12 has a high tensile strenght, hardness, chemical resistance, resistance to abrasion and is insensitive to cracking. (*Nylon 12*, 2021)

Polyamide 12 is, according to provider of both machines and materials for additive manufacturing, tested material for additive manufacturing. Parts produced from polyamide 12 powder through additive manufacturing gives strong, flexible and durable parts while also being cost efficient. PA2200 is a well-balanced material variant of the polyamide 12 family, with many use cases, including functional prototypes. A downside to PA2200 is its relative low melting point at 170-180°C and its heat deflection temperature at 154°C and 70°C when under pressure of 0,65 MPa and 1.80 MPa respectively. Still, the accuracy and smooth surface combined with its low cost makes it a interesting material in regards to a prototype mold. (*EOS polymers for additive manufacturing*, n.d.)

2.2.3 Metal mold material: H13 tool steel

For the baseplate and finished inserts the material that will be used is DIN 1.2344 tool steel, also known as AISI H13 tool steel. It is a tool steel grade standardized for hot working. The chemical composition of the alloy is as follows: carbon, silicon, chromium, molybdenum and vanadium. The combination of the alloyed elements of chromium, molybdenum and vanadium (Cr-Mo-V) gives the material a high resistance to thermal shock, heat resistance and a great strength in general. The high content of vanadium means that the material can withstand abrasion in both high and low temperatures. (*DIN 1.2344 tool steel*, 2020)

The hot hardness of H13 means it resists thermal fatigue cracking which commonly occurs as a result of repeated cycles of heating and cooling. The resistance to thermal fatigue cracking in combination with its high toughness is the reason that H13 is one of the most used materials for hot work tooling applications such as mold inserts for injection molding. (*H13 Tool steel*, n.d.)

Due to its hardness and toughness, H13 tool steel is both an expensive and time-consuming material to machine. Because of this SLS is an excellent way of producing mold inserts, since these needs many details such as core/cavity cutouts, cooling/heating channels, gate, runners et cetera. (*About H13 Tool Steel*, n.d.)

2.3 Injection molding

Injection molding is a production process where a magazine is filled with a mass of material usually in the form pellets or granules. The pellets/granules are then gravity fed into a cylindrical chamber called the barrel. Inside the barrel the material is pushed forward by a conical screw that rotates as it is pushed forward. The barrel is externally heated to the temperature of the melting point of the material. This means that the granules are simultaneously being melted into a liquid state and pushed forwards in the chamber. At the end of the chamber the liquid material is pushed through a nozzle, into the sprue bushing through the gate and into the mold. If it is a multi-cavity mold it is pushed through a runner system before reaching the different gates. The sprue bushing and runner system are channels for transporting the material from the barrel to the mold, and the gate is a hole in one of the mold that lets it get filled by material. (*Basics of injection molding design*, n.d.).

2.4 Liquid injection molding of liquid silicone rubber

Liquid injection molding (LIM) is an injection molding process. Liquid injection molding of liquid silicone rubber (LSR) is often referred to as LSR injection molding. The liquid silicone rubber is a liquid silicone that solidifies quickly when exposed to heat. (Gerdeen, Lord and Rorrer, 2006).

Liquid injection molding is used when the raw material for the injection molding comes as a liquid that needs to vulcanize. Vulcanization is a method of hardening elastomers by mixing it with a crosslinker that bonds to the silicone. The vulcanization process (*Vulcanization*, 2020). In the standard, resin-based injection molding process, however, the raw material is in the form of solid pellets that need to melt *before* it is injected into the mold.

2.4.1 Liquid Silicone Rubber

The liquid silicone rubber is the raw material used when silicone is produced via a liquid injection molding machine. The liquid silicone rubber consists of two components: a catalyst and a crosslinker. The catalyst is platinum based, while the crosslinker is a methylhydrogensiloxane. These two components come in separate barrels and are mixed into the LSR just moments before the injection process starts. (*What is silicone?*, n.d.)

The crosslinker and the catalyst are mixed in a static mixer which mixes them into one, homogenous liquid – the liquid silicone rubber. The LSR will immediately begin the vulcanization process. But since the vulcanization process is drastically slower in lower temperatures, it will not start to solidify just yet.

LSR has a very low viscosity and will therefore flow with ease (*Viscosity of silicone rubber*, n.d.). This is in contrast to ordinary injection molding where the melt often has a higher viscosity, which results in higher injection pressure and flow channels. Thus, a low viscosity is an advantage, allowing ease of flow.

2.4.2 The injection process

After the crosslinker and the catalyst have mixed, the LSR is pumped into a screw/injection unit. The screw mechanically pushes the liquid through a sprue and it flows into the runners. The runners are the channels that connect the cavity and core to the sprue and delivers the liquid to the cavity and core through at least one narrow gate.

Optionally, the LSR and the runners can be preheated to temperatures ranging from 40°C to 80°C. ('Liquid Injection molding,' 2020)

When the LSR is injected into the heated mold, it will instantly begin to solidify. It is estimated that for temperatures greater than 140°C, it takes 3-7 seconds for one millimeter of wall thickness to fully cure. However, many LSR parts are ejected from the machine when its 75-95% cured; the remaining curing will happen in a postcuring process outside of the injection molding machine (Bont, Barry and Johnson, 2020).

If the injection mold is not in a vacuum, it will be necessary with vents to allow air to escape as the LSR fills more and more of the mold. However, as the LSR has such a low viscosity, the liquid might enter the vents and consequently solidify inside the vent, rendering them useless. For this reason, vacuum-setting the molds is a usual practice in liquid injection molding. (Bont, Barry and Johnson, 2020)

2.4.3 The components and the assembly of a mold

The mold of a liquid injection molding machine is an assembly of a couple of crucial parts, each with its own purpose. Perhaps the two most important components of the mold are the core and the cavity. It is the region of empty space in-between the cavity and the core, when clamped together, that the liquid silicone rubber is injected into in order to mold the part. See figure 3 for an illustration of a cavity and core.

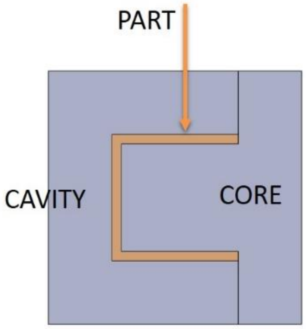


Figure 3: Cavity and core (Draft angles in injection molded plastics, n.d.)

Figure 4 depicts how a mold might look like, and figure 5 shows the cross-section of that same mold. It is important to know that not all molds look alike, and that they are designed depending on its purpose and requirements, which, in the case of liquid injection molding, often comes down to heat propagation. The design of this mold, for example, is heavily impacted by the needle valve gate that is utilized for this production.

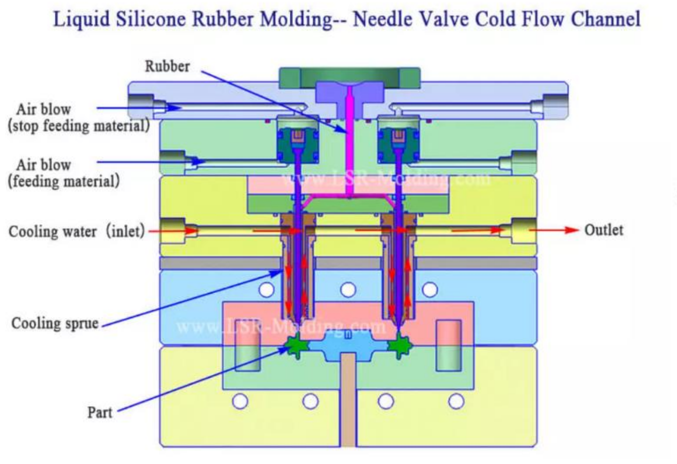


Figure 5: Mold assembly cross-section (Needle valve cold flow channel, n.d.)

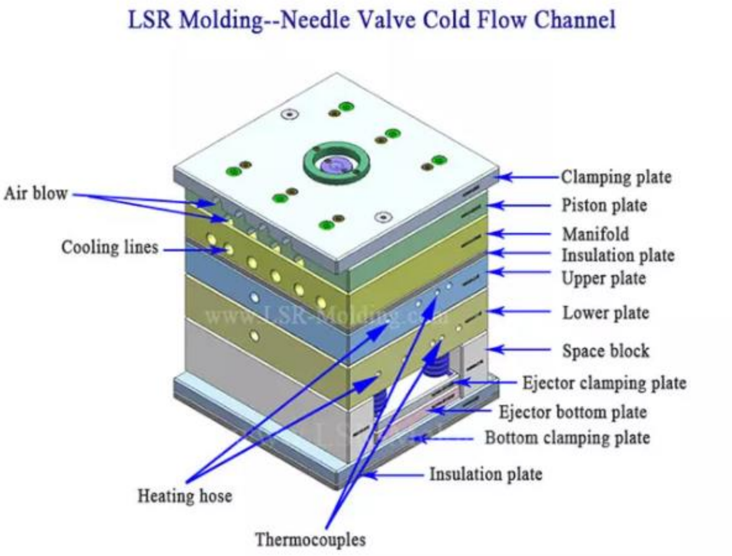


Figure 4: Mold assembly (Needle valve cold flow channel, n.d.)

Focusing on figure 4 and 5, here is a list of its most crucial components with their functionality summarized:

1. Upper and lower plate: Houses the cavity and the core. Both the upper and lower plates are heated.
2. Insulation plate: insulates the heated plates from the plates that need to stay cool.
3. Manifold: This is where the LSR exits the sprue and enters the runners.
4. Piston plate: The plate with which (with the help of pneumatics) the needle valve gate is controlled.
5. Cooling lines: Water cooling channels.
6. Heating hose: Heating elements.
7. Thermocouples: Heat sensors.

2.4.5 Cooling system

It might be necessary with cooling channels around the runners to protect them from the propagating heat caused by the heating elements of the molds. The gate valves might also be cooled. If the runners or the gates reach a high enough temperature, it might cause the liquid silicone rubber to solidify before it is injected into the mold. This would be a major bottleneck for the production.

Because of this, it is critical that the heat propagation in the mold is thoroughly analyzed in order to determine what sort of cooling system it requires in order to function optimally. For example, in some instances, a cooled gate would be sufficient; another time, both cooling channels near the runners and other places in the baseplate, along with a cooled gate, might be required.

There are also alternate ways to battle the heat propagation, for instance with the use of insulation plates as a way to minimize its effect, or having the baseplate, into which the inserts are fastened, in insulating material.

2.4.6 Heating system

The liquid injection molds need to be heated for the LSR to cure after it is injected. This is done by assembling heating elements inside the inserts/molds.

2.4.7 Key values of LSR injection molding.

Some of the most important process parameters for liquid injection molding is clamp force, injection pressure, hold pressure, curing time, mold temperature and hold time. In table 1, the common value intervals for these process parameters are listed. (Bont, Barry and Johnson, 2020)

Table 1: Key values of LSR injection molding

What	Value	Explanation
Clamp force	0.8-2.5 kN/cm^2	The force that is required to hold the two mold halves together.
Injection pressure	1-80 MPa	The pressure the LSR is injected into the mold with.
Hold Pressure	0.5-5MPa	Pressure that is applied after the cavity is filled. Injects more LSR as the material in the mold solidifies to compensate for shrinkage.
Curing when temp. >140	3-7 mm/s	The rate that the LSR is cured at.
Mold temperature	120-200 °C	Temperature of the mold. The goal is a uniform heat around the
Hold time	0.5-4+ s	The amount of time that the hold pressure is maintained.

2.4.8 Prototype molds

The cost of building molds for injection molding is substantial. This is because molds are usually produced in low quantities and consist of several parts that need to be machined with extreme precision. Injection molding molds are massive, as well.

In the development stage of a mold, it might be a good idea to start out with prototype molds. One way to do this, is in cheaper metal material that are less resistant to external stresses and that does not last as long. Another way to do it is by 3D-printing molds, either in metal or plastic, and use those for prototyping.

Now, as plastics generally have lower melting points than conventional industry metals and expand more than steel when exposed to heat, it is naturally not an optimal pick. However, prototypes do not need to survive that many runs. And in the case of LIM, one can easily lower the mold temperature to as low as 120°C, only at the expense of the curing time and consequently the total cycle time of the injection molding process. The main purpose of prototyping is to weed out any unknown problems that pre-production has failed to account for and to make sure that the product turns out satisfactory. (Skiba, 2019)

2.5 Additive Manufacturing

Additive manufacturing is an umbrella term for all production processes where the product is made by building layers of material incrementally.

When creating a product through AM one needs a CAD model, i.e., a digital, three-dimensional model of the product. This CAD file is then imported into the AM machine’s dedicated software, which constructs vertical slices of the model. These slices are analyzed individually by the software, and a set of instructions for each slice is compiled. After the settings for the different variables have been set the manufacturing process can be started. The machine takes the previously mentioned instructions and uses these to build the part layer by layer. Depending on the tolerances regarding things like surface roughness the product can be subject to different finishing operations like sanding or painting.

AM processes are both highly digitalized and automized. “Manual labor” is needed for digitally designing the part through CAD, importing this file to the machine software, and initializing the manufacturing process. The manufacturing process itself is entirely done by the machine, and in most cases do not require supervision by an operator.

Due to the layer-by-layer production process the material waste from AM methods is negligible, and it also means that complex internal detail in a part is at least easier, if not only possible through AM. AM processes are also highly automated, which removes the need for an operator during the entire manufacturing process. AM processes are also reasonably fast manufacturing methods. Due to these reasons AM methods are great for prototyping and are in many cases favorable over traditional subtractive manufacturing processes, both economically and environmentally.(Kalpakjian and Schmid, 2014)

2.5.1 FDM

Fused-deposition modeling is an AM process where thermoplastic filament is continuously fed from a spool to a heated extruder. The extruder moves along the horizontal plane and extrudes the molten filament onto a table to create a layer of the product. The extruder head's movement is decided by the instructions compiled by the machine-software. When the layer is finished, either the table, the extruder head or both are moved along the z axis to start on a new layer. This process is repeated until the product is finished.

When using FDM, the weight of each new layer is supported by the previously printed layers. This can offer challenges when manufacturing some products. Many products have features that go out of bounds of previously printed layers which leaves nothing to support its weight. This problem is solved by printing support constructions beneath these features. The support constructed is produced with less dense filament spacing, which make them easy to break off from the product.

The quality of a finished FDM product depends on a couple of variables. An important one is layer thickness, which one wants to be as low as possible to obtain a smooth surface. The layer thickness is largely decided by the extrusion die diameter, where a smaller diameter results in a smaller layer thickness. The material used is decisive to the dimensional accuracy in the horizontal plane, so choosing the right material for the needed accuracy is important.
FDM is typically a very cheap production process. This makes it a very favorable method for prototyping. This way one can physically inspect the design and control that the product fits its purpose from a dimensional position without any economic risks. (Kalpakjian and Schmid, 2014)

2.5.2 SLS

Selective laser sintering is an AM process where the material is fused together by using a laser. A powder of the wanted material is spread densely in a thin layer across a table. The laser follows the instructions given by the slicing software. The laser moves along the contour of the current layer and heats the material powder to near melting point. The heated material is fused together to create the layer. The table then moves an increment downwards, new material powder is added on top and the process is repeated.

After the process of creating a layer there is an excess of powder outside the layer contour. This powder acts as support for the next layer, which means that a detail for a new layer can be created at any place without needing existing solidified material in the form of built product or "external" support. This means that SLS makes it possible to create complex geometries without wasting material to create an additional support that needs to be removed later.

When the product is finished the excess powder can easily be removed with air pressure or a brush. The excess powder can then be used as material for the next part that is to be produced. SLS can be used for both the prototype mold and the final mold, as it is commonly used with nylon materials and metal alloys that consists of multiple alloy elements, since these have different melting points.

(Kalpakjian and Schmid, 2014)

2.6 Computer simulation of computer-aided design (CAD)

Computer simulations make it possible to test how a physical system with certain rules and properties responds to a set of variables or parameters. The simulation software is mainly run through calculating equations that mimic the real system's functional relationships. (*Computer simulation*, 2017)

2.7 Background theory for thermal simulation

2.7.1 Convection

Convection is the flow of particles due to the transfer of heat between a material and a liquid or gas, or a soft material in which particles can move more or less freely. Convection happens when a material (either a liquid or solid) is placed in an environment that is either considerably warmer or colder than the material. There are two types of convection:

1. Natural convection: when the transfer of heat is due to natural causes, such as still water, still air, or slightly windy air.
2. Forced convection: when the transfer of heat is manipulated in order to achieve a preferred heat-transfer rate, this could either be by pumping water through tubes to cool certain machine components down and can be controlled by the speed of the water and its temperature.

(*Convection*, 2020)

In a thermal simulation, surfaces that are subjected to convection must be defined and given a convection coefficient, in order to make the simulation as realistic as possible. The unit for distributed convection is $K*m^2/W$, where:

- K: Kelvin
- m: Meters
- W: Watt

An injection molding machine – and especially one with heated inserts – will be subject to convection. One way that it is affected is through the surrounding air (natural convection), another way is through liquid cooling systems (forced convection).

2.7.2 Contact sets and thermal resistance

Thermal resistance is a coefficient representing an object's ability to resist thermal stresses.

Thermal resistance decreases as the contact pressure rises, which results in higher rates of heat transfer between the surfaces in contact. Thus, the higher the pressure, the lower the thermal resistance coefficient is set to. In addition to that, thermal resistance is also dependent on:

- Material
- Smoothness of surface
- The liquid or gas in the air traps between the surfaces

(*Thermal contact resistance*, 2016)

2.8 The Integrated Quantitative Framework

Being proactive and doing a thorough job in the preliminary design process is the most secure way to ensure a minimal number of defects or faults in prototypes. By understanding a particular part's geometry, and how it corresponds to the various design parameters and -variables, one can properly implement the correct design solutions early on. Even so, it is often incredibly challenging to successfully -- and correctly -- considering all of the possible defects that a prototype design might have; thus, they might not show up until the mold prototype is put to the test in an injection molding machine.

Minimizing the number of problems that might occur post-design (but pre-production) should therefore be the top priority, which effectively decreases the number of times that the design needs to be tweaked. This is done by systematically and strictly following a set of conditions and requirements for the process.

The *Integrated Quantitative Framework* divides the design process of a mold into four stages:

1. Identify
2. Design
3. Optimization
4. Validation

(Kauffer, 2011)

While the Integrated Quantitative Framework mainly regards ordinary injection molding and not liquid injection mold, some of its aspects are applicable to LIM as well. Figure 6 below is a flowchart that illustrates the design parameters of mold design, divided into the four main parameters: Conceptual, processual, constructive solutions and complexity.

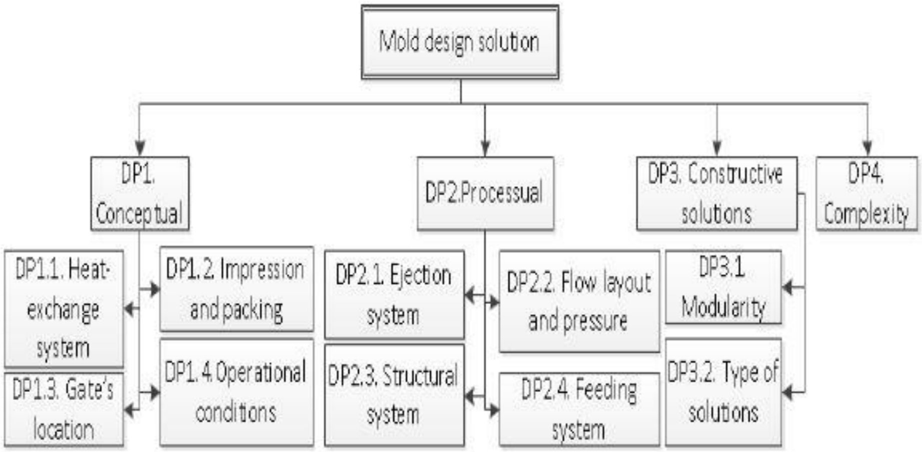


Figure 6. Design Parameters (DP) for top design levels (Kauffer, 2011)

3 Method

3.1 Methodology

This thesis' methodology is the combination of two processes: one process of development that details the main stages of the development, and one process of design (of 3D-printed LIM molds) that goes into depth with regards to the design parameters of liquid injection molding.

3.2 Process of development

The process of development, which is depicted in figure 7, is divided into two phases. Phase 1 consists of the initial stages of the process, such as research, design, and simulation. Phase 2 is heavily reliant on additive manufacturing and the design optimization that might come as a result of the 3D-scanning, to better fit the model for a 3D-printing process.

The process is based on the research and it is a suggestion to how the developmental process of making a mold for injection molding through 3D-printing and prototyping might look like.

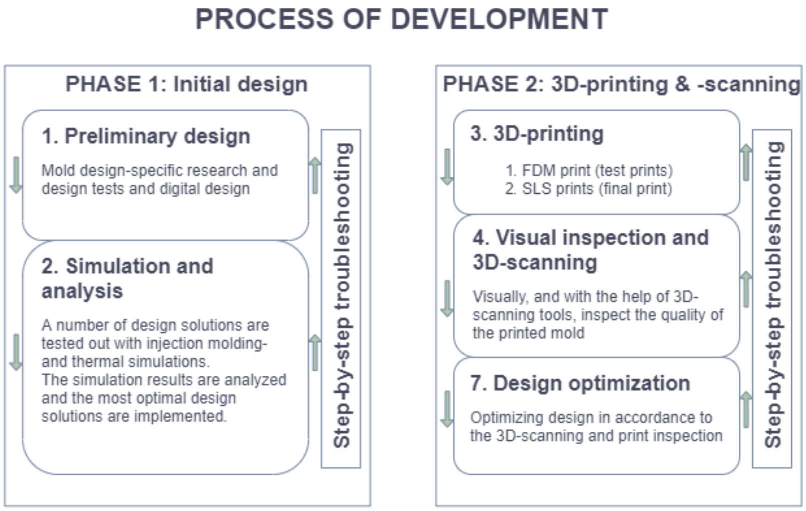


Figure 7: Process of development

3.2.1 Preliminary design

The preliminary design stage is the very first in the process of development. As such, it is the foundation for the entire project, and immensely important. In short, this stage consists of: Research, application, and critical thinking. The stage also includes digitally designing the mold's cavity and core with rapid iterations and alterations on specific design elements, and the stage is over when the CAD-file is ready for simulations.

The research might be a mixture of product-specific and generalized research on liquid injection molding. Examples on product-specific research might be: mold temperature, mold material or injection molding machine. It is arguably easier to find solutions when one has seen a variety of examples of a product produced the same way.

The research must be applied with the current situation at hand in mind. It is an advantage to come up with several solutions to a single problem – such as gate location – because, that way, one can compare them in-depth and put them through a process of elimination wherein the most compatible solution/solutions are selected to advance to the simulation stage. This is where the critical thinking come in.

3.2.2 Simulation and analysis

Simulation and analysis follow directly after the preliminary design stage. In this stage, various design parameters that are deemed the most optimal during the preliminary design stage are applied to the digital model and subsequently tested with injection molding simulation software as well as thermal simulation software. Both Solidworks and Digimat can be utilized as injection molding software, while the thermal simulation is done exclusively with Solidworks.

The simulation parameters that are in focus for the injection molding simulations are presented in table 4, and for the parameters in focus for the thermal simulations are presented in table 2.

Table 2: Injection molding simulation variables

	Variable	Explanation and preference
I.1	Cycle time	The time it takes from injection starts until the casted part is ejected from the mold. Preferable: less time, but this requires higher pressure and temperatures.
I.2	Curing time	How many seconds it takes for the liquid silicone rubber to cure/vulcanize. Preferable: less time, but this requires higher temperatures.
I.3	Clamp force	The force that the machine pushes the two mold halves together with. Preferable: Just enough force to hold the two halves together during the casting process. Too much force can lead to damage to various mold parts.
I.4	Ease of flow	How easily the liquid flows through and fills the mold. Preferable: Little to no resistance – depends on gate location and number of gates.
I.5	LSR temperature	What temperature the injected LSR reaches. Preferable: temperatures close to the cavity and core's surface temperatures.
I.6	Stresses	Liquid material solidifying in a curved geometry introduces internal stresses. Preferable: As little internal stresses as possible. Achievable through larger radius in core/cavity geometry.

Table 3: Thermal simulation variables

	Variable	Explanation and requirement
T.1	Heat propagation on mold surfaces	Mapping the heat spread on the surfaces of cavity and core Requirement: Approximately uniform heat propagation on heat surface.
T.2	Location and number of heating elements	(Tied to the above variable) How to locate the heating elements in order to adequately supply heat to the mold surfaces; how many heating elements are required in order to adequately supply heat to the mold surfaces.

		Requirement: Sufficiency of heat supply and Approximately uniform heat propagation on mold surface
T.3	Heat propagation baseplate	Mapping the heat spread in the baseplate, and most importantly near the runners. Requirement: Too high temperatures will cause the material to start curing process in runners. In this case a cooling system is needed to counteract this.

After the simulations are done, as well as in-between the simulations, the results are analyzed and considered. And at the very end of the stage, it is decided which configurations and values that give the most satisfactory results. Some of the parameters may not yield a result within defined requirements. If this is the case the corresponding part of the mold needs to be tweaked and tested until a satisfactory result is achieved.

3.2.3 3D-print

The purpose of the 3d print is to give a physical model of the mold inserts that can be inspected. This prototype mold will act as a proof of concept and can be a useful tool for a few different cases:

- To see if the design works in the real world.
  - The digital design of a part or an assembly can often look good in the CAD software, but when it is produced different problems may be introduced. This can be anything from little of a clearance to the fact that a principal works in theory but not in practice.
- Cavity and core can actually be produced.
  - The core and cavity might have some details that are difficult to produced. Through the 3D printed prototype one can determine wether the current geometry is feasible. If it isn’t a tweak or redesign is needed.
- Check the accuracy of the 3D printing method and machine.
  - How accurately it recreates the CAD model and the quality of different parameters such as surface roughness.
- Eventually perform a test cast with the 3D printed mold insert.

  - With an accurate mold prototype in a suitable material one can perform a test cast to see how well the mold performs. This will uncover whether or not the mold is able to cast the product in question. This gives a definitive answer on whether or not the mold design is adequate, and thus whether further tweaks are needed or if the final mold can be produced.

The 3D printed prototype can be produced through any 3D printing method. In the case of this thesis the plan is to print one prototype in PLA using a simple FDM printer for inspection and evaluation of the design. This because the production of this PLA prototype is very cheap while still yielding a pretty accurate result. When the result of the PLA prototype is satisfactory there will be printed a new prototype in PA2200

3.2.4 Visual inspection and 3D-scanning

The 3D-scanning and print inspection is part of what could be looked at as the quality control section of the method, along with the design optimization stage, and it is meant to be a way of ensuring the print's quality, functionality, and precision *before* it's put to use.

The first part of this stage is the visual inspection: to visually inspect the printed part, and to look for flaws, inaccuracies, or defects in the print. If found, it is important to ascertain exactly what caused it, which can be done by following the method's troubleshooting strategy by going backwards through the stages and see if the cause lies there. More likely than not is that the cause of the flaw stems from the 3D-printing.

Then, if the print passes the visual inspection, it is ready to be 3D-scanned. The 3D-scanning results in a digital mesh, which can be further inspected, either by comparing it to its original CAD-file or by running simulations on the digital mesh.

3.2.5 Design optimization

The design optimization stage is the final stage of the method of development and is the only stage that can be skipped – only if the previous stage does not reveal any defects, flaws, or inaccuracies in the print. However, if it is deemed necessary to change either the design or print settings due to a flaw in the print, this is the stage in which the necessary change is implemented.

3.3 Process of design (design parameters)

The process of design is meant as a guide for the design process, outlining the important design elements and putting them in a prioritized order, and is heavily based on the initial research done which was covered in chapter 2. This method divides the most crucial aspects of design process into specific tasks with clear instructions. Table 4 covers the tasks and their instructions.

Table 4: Process of design

Process of Design (PD) stages	Process	Procedure and instructions
PD.1	Boundaries of mold size	The physical limitations of the size of the mold
PD.2	Mold material	Selecting the material of the mold
PD.3	Parting line	Decide where the parting line is to be located. The parting line decides where the cavity and core separate.
PD.4	Mold temperature	Temperature: 140-220 Higher temperatures lead to shorter curing times, while too high temperatures (>220°C) might lead to problems with the LSR solidifying too early and failing to fill the mold completely. Mold temperature should not exceed nor be too close to the material's melting point.
PD.5	Heating elements 1. Location 2. Quantity	Decide what sort of heating element works best with the mold at hand. Location (1) and quantity (2) of heating elements can be determined through thermal simulation. Simulate until uniformity of thermal propagation on mold surface is achieved.
PD.6	Injection gate 1. Size 2. Location 3. Type of gate	• LSR gates can have a very narrow opening. • Location of gate should be symmetrically placed in the cavity to allow for symmetric filling • The type of injection gate, and the specific gate model
PD.7	Ejection pin location	The location of the ejection pin system must be on the opposite side of the injection gate.

PD.8	Baseplates and modularity	Design the baseplates with modularity in mind.
PD.9	Runner system	Symmetrical runner system that is sufficiently cool. (<70°C)

4 Execution

4.1 Preliminary design stage

When designing the inserts one of the most important parameters for this thesis, apart from durability, ease of flow and moldability, was modularity. The task at hand was to develop a process for the development of injection molding tooling for medical equipment, with the use of 3D-printing. This means that it needs to be easy to go from producing one product to another. The design of the different mold parts is restricted by the following:

- dimensions of the available injection molding machine, Krauss Maffei 350/2000CX.
- dimensions of production machines available at the labs of NTNU Gjøvik.

In addition to the machine-given restrictions, the design has been revised based on the results in the simulations.

4.1.1 Components of the assembly

The complete assembly consists of 6 unique components. These are:

- Baseplates: The components that house the inserts.
  - Baseplate with runners.
  - Ejection side baseplate.
- Mold/inserts: The components into which the LSR is injected.
  - Cavity.
  - Core.
- Backplate (with runners).
- Heating rods: the heating elements responsible for heating the inserts.

4.1.2 Injection gate

While doing the groundwork on deciding the gate location and -quantity, five configurations were considered. See table 5 for the specifications regarding the configurations and figure 8 through 12 for visual illustrations of the configurations, where the red dots represent the gate location.

Table 5: Gate location and number of gates

Gate config. number	Cavity or core	Location	Total number of gates	Figure
1	Cavity	One gate on the top of the filter cover	1	See figure 12
2	Cavity	One gate in the middle of the filter cover	1	See figure 11
3	Core	One gate on each of the side pieces	2	See figure 10
4	Cavity	One gate in the middle of the filter cover and one gate on each of the side pieces	3	See figure 9
5	Core	One gate in the middle of the filter cover	1	See figure 8

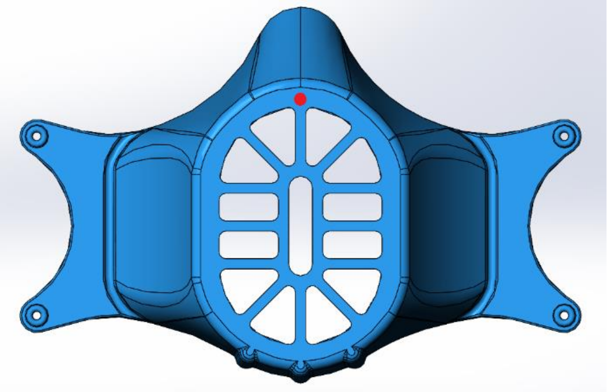


Figure 12: Gate configuration 1

Figure 11: Gate configuration 2

Figure 10: Gate configuration 3

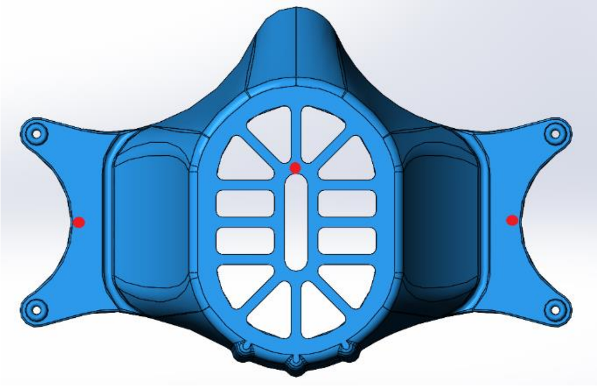


Figure 9: Gate configuration 4

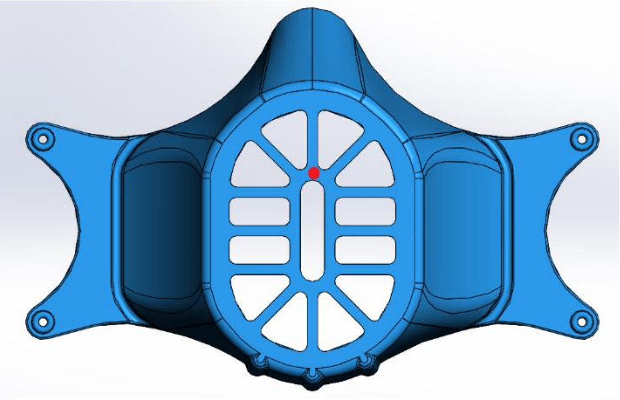


Figure 8: Gate configuration 5

Due to the geometry of the core, with the indentation and overhang of the filter cover (inside of mask), the molded part will most likely stay attached to the core when the cavity and core separate. This means that the ejection pins must be located on the core. The gate and ejection pins cannot be on the same mold half, as the ejection pin system is located in the other end of the mold assembly, and thus, the fact that the ejection pins must punch through the core, means that the injection gate must be located on the cavity.

Due to the LSR’s low viscosity and low volume of the mask, there is no sound reason to believe that is necessary with several gates in order to properly fill the mold.

With this in mind, it was decided that the frontrunner was configuration 1 (see figure 7) with the gate on the top of the filter cover on the cavity. See figure 13 and figure 14 below for the gate’s location on the cavity.

As shown in figure 14, the gate size is set to 0.5 mm. Because of LSR’s low viscosity, the gate size does not play as big a role as it does in ordinary, resin-based injection molding.

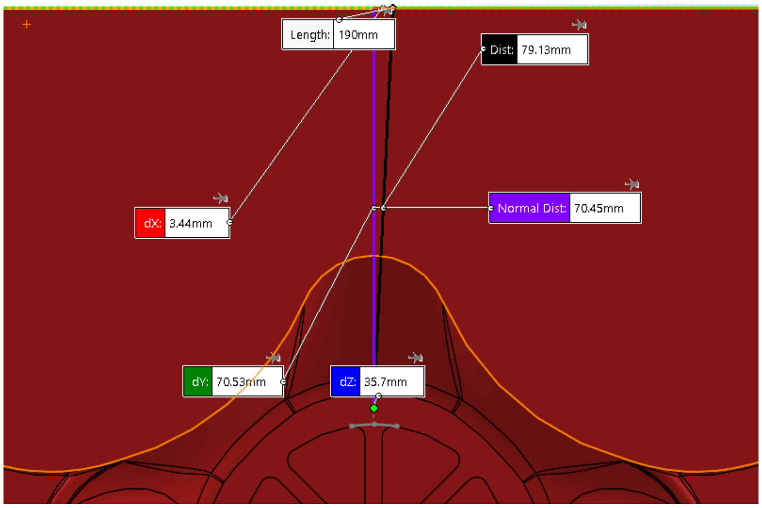


Figure 13: Distance from top of cavity to gate

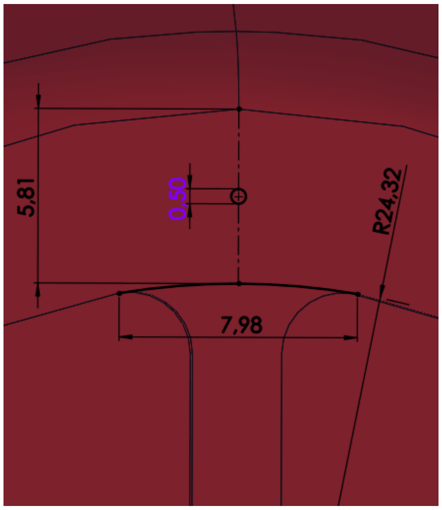


Figure 14: Location of gate

4.1.3 Baseplates and backplate

The baseplates each has four cutouts and two threaded holes per insert to fasten them. The cavities are inserted into baseplate with runners, and the cores are inserted into the baseplate on the ejection side of the injection molding machine. The injection side baseplate has a runner system in an x formation cut into it. This is to transport the material to the inserts. 4 cooling channels are located at the back of the baseplate with runners, in order to cool down the runner-side of the baseplate in order to prevent LSR flowing through the runners from prematurely solidifying.

The baseplate with runners has the dimensions (width, height, depth) 600mm x 500mm x 225mm and the ejection side baseplate has the dimensions 600mm x 500mm x 150mm.

The backplate has the dimensions 600mm x 500mm x 50mm and has four cooling channels running through it.

4.1.4 Runner system

When the baseplate with runners and the backplate are clamped together, they form running channels in-between them. Each of these two components have half-circle runner in the shape of indentations. The runner channels are designed to be both symmetrical and to follow the quickest route to the gate. See figure 15 for the dimensions of the runner cutouts.

The gate location of the cavity is not in the center of the insert. With this in mind, it is not possible to achieve symmetric runners without having to turn the cavities on the bottom half 180 degrees in relation to the cavities on the top side. This means that the cavities will be oriented in two opposite directions when inserted into the mold. However, this should not be a problem and won't affect the injection molding system's functionality.

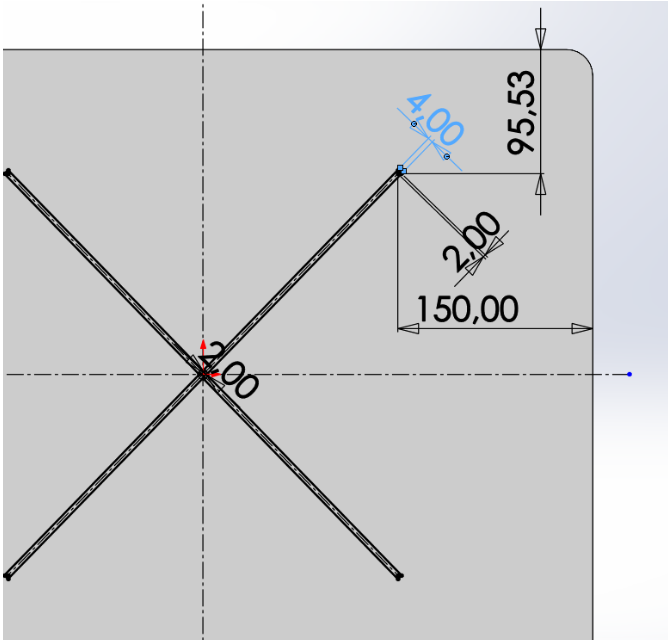


Figure 15: The runner cutout of the backplate.

4.1.5 Guide pins

Both the baseplates and the backplate were equipped with guide pins and guide holes as a way to ensure that the baseplates and the backplate are all positioned correctly in relation to each other. The guide pins have a diameter of 32mm and have a depth of 45 mm. The top of the of the guide pins have a 12mm radius, and the bottom, where the guide pin extrudes from the plate, have a radius of 5mm. See figure 16 and 17.

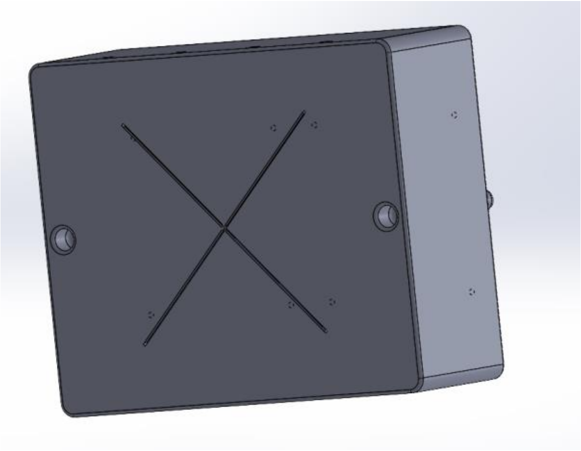


Figure 16: The guide holes on the baseplate with runners

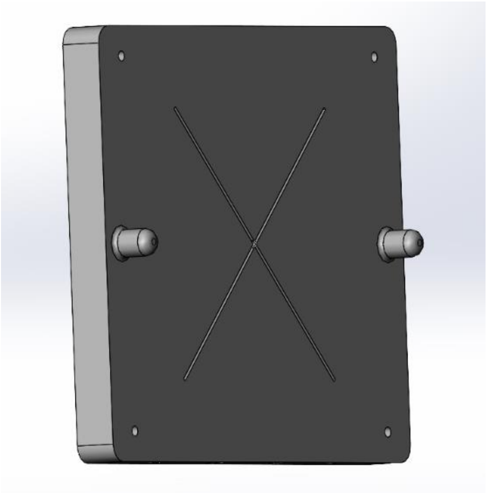


Figure 17: The guide pins on the backplate

4.1.6 Modularity

The baseplates were designed with modularity in mind. For that reason, the cutouts, and thus the inserts too, were given dimensions that are somewhat larger than needed for the mask. The idea behind this is that this baseplate layout is a standardized baseplate layout for medium-sized parts. This opens up for the possibility of using this same baseplate for liquid injection molding of other medium-sized parts with the use of inserts that fit into the baseplate. Furthermore, it was imagined that there might be designed other baseplates meant for other sizes of medical equipment, and that the line-up of baseplates might look something like this:

- Baseplate for small parts: 6-8 cutouts.
- Baseplate for medium parts: 4 cutouts.
- Baseplate for large parts: 1-2 cutouts.

4.1.7 Inserts

The inserts are designed to make the mold system highly modular. The inserts have standardized dimensions for the outer geometry. The product specific parts are the core/cavity contours given by the product and location of the heating elements. They are designed after the dimensions of the baseplate, and the wish to cast four masks simultaneously. The standardized geometry of the inserts makes it easy to produce new inserts for other products than the mask. Since the inserts are fastened with two bolts each it is also very easy to change the inserts. Standardized inserts like this also gives the opportunity to produce two different products simultaneously in the same injection molding machine by using two different sets of inserts. It also makes it possible to cast multiple smaller parts per insert.

The insert solution, rather than core and cavity details directly in a permanent backplate is favorable when it comes to prototyping. Prototype inserts can be made in a cheaper material and through cheaper and faster manufacturing methods than baseplate and insert. This means that a new product can be casted using only one cavity/core insert set

One large insert covering most of the baseplates surface was considered. This would allow for parts bigger than the current inserts to be produced using this mold system. It would also provide more freedom in regard to location and number of cores/cavities on the bigger insert surface. It would however introduce possible problems regarding the vulcanization process in the runner system, as it would require a more complex runner system inside the heated insert. The idea of a single big insert in the baseplate was disregarded early due to restrictions regarding the SLS printers available at the school lab, as the upper limit for dimensions were 300x300 mm.

4.1.8 Heating rods

The heating elements that are to heat up the inserts are heating rods, or cartridge heaters. Cartridge heaters are common for liquid injection molding. Specifically, the heating rod that was selected as a basis for this thesis, was the Aexit injection mold water heater, with the dimensions 10mm x 200mm. (*Aexit Injection Mold Water Heater*, n.d.)

4.3 Injection molding simulation

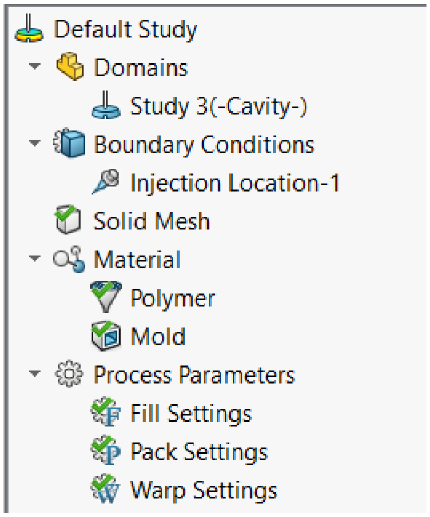
Simulations were run with gate configuration 1, as described in table 7, in mind. Several simulations were run with both the prototype material and the metal mold material, and simulations were also run at various mold temperatures in order to gather enough information about the simulation variables as possible. The three simulations run are:

1. Metal mold at 150°C
2. Metal mold at 175°C
3. Metal mold at 200°C

4.3.1 Injection molding simulation with SolidWorks Plastics

The injection molding simulation was done with SolidWorks Plastics. The way SolidWorks Plastics works is by doing a simulation on the part to be molded, itself, instead of doing it on the mold cavity and core. Thus, the simulation was executed on the mask part.

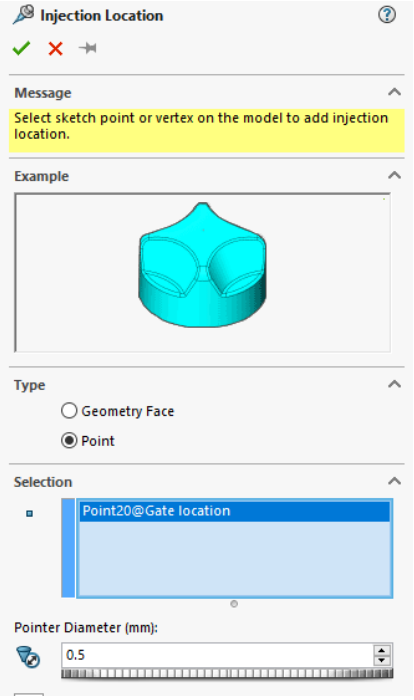
A number of settings must be adjusted before running the simulation. The settings overview is shown in figure 18.



Default Study
- Domains
  - Study 3(-Cavity-)
- Boundary Conditions
  - Injection Location-1
- Solid Mesh
- Material
  - Polymer
  - Mold
- Process Parameters
  - Fill Settings
  - Pack Settings
  - Warp Settings

Figure 18

Injection gate location and size must be defined, as well as materials for mold and polymer, and the process parameters. The process parameters are fill settings, pack settings and warp settings, and are in regard to the mechanical limitations and properties of the injection molding system. See figure 19 through 23 for examples on what the settings windows look like.



Injection Location

Message

Select sketch point or vertex on the model to add injection location.

Example

Type

○ Geometry Face

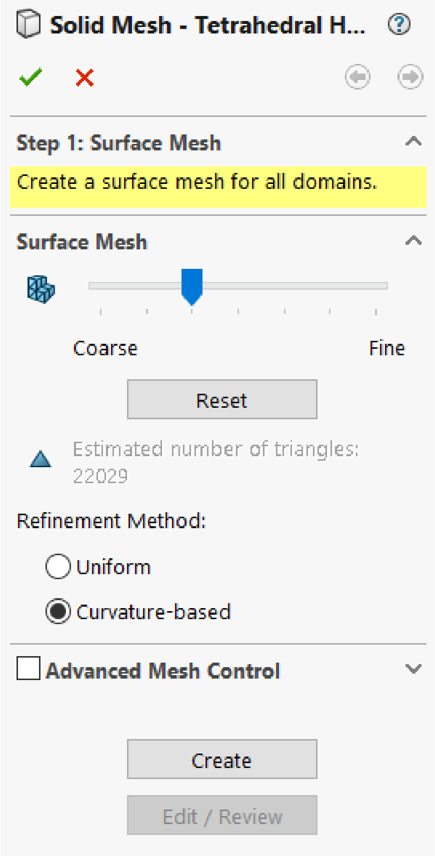
● Point

Selection

Point20@Gate location

Pointer Diameter (mm): 0.5

Figure 19: Injection location and - size settings



Solid Mesh - Tetrahedral H...

Step 1: Surface Mesh

Create a surface mesh for all domains.

Surface Mesh

Estimated number of triangles: 22029

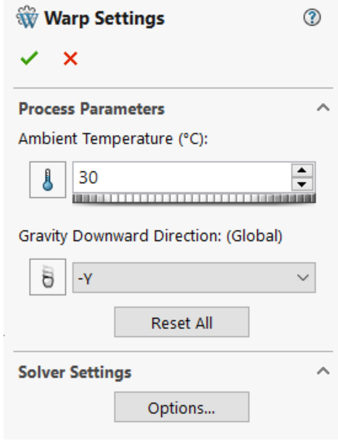
Refinement Method:

○ Uniform

● Curvature-based

☐ Advanced Mesh Control

Figure 20: Mesh settings



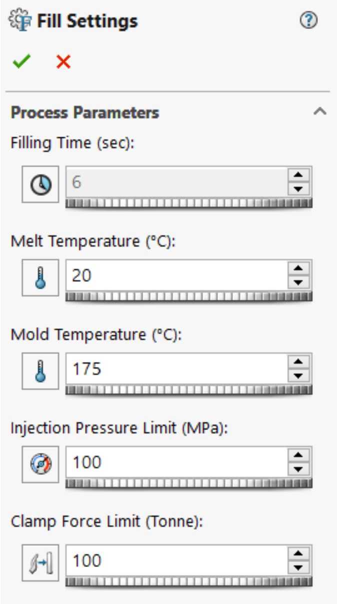
Process Parameters

Ambient Temperature (°C): 30

Gravity Downward Direction: (Global) -Y

Solver Settings

Figure 23: Warp settings



Process Parameters

Filling Time (sec): 6

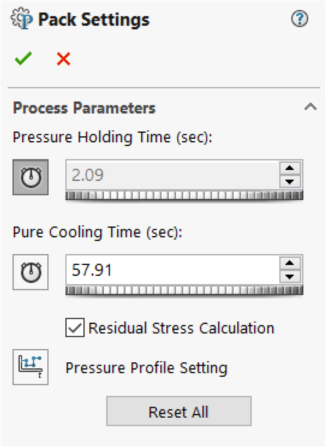
Melt Temperature (°C): 20

Mold Temperature (°C): 175

Injection Pressure Limit (MPa): 100

Clamp Force Limit (Tonne): 100

Figure 21: Fill settings



Process Parameters

Pressure Holding Time (sec): 2.09

Pure Cooling Time (sec): 57.91

☑ Residual Stress Calculation

Pressure Profile Setting

Figure 22: Pack settings

SolidWorks has preset values for all of the process parameter settings, depending on which polymer material is chosen.

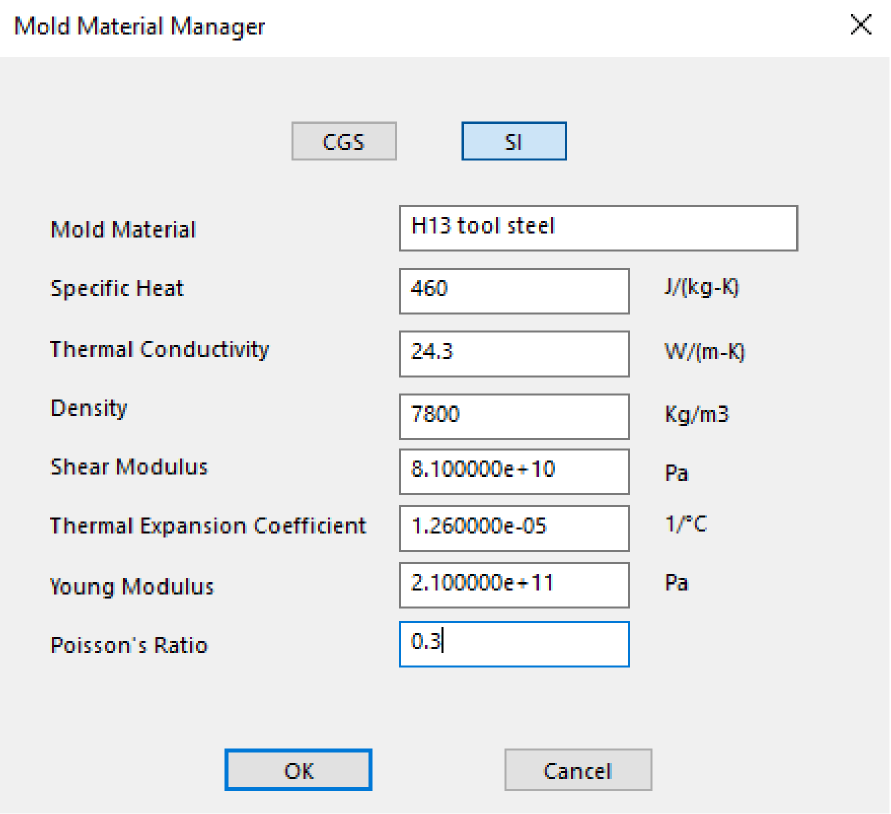
The warp settings (figure 23) may stay on the preset values. However, it is important to make sure that the direction of gravity is correctly defined. The ambient temperature refers to the temperature surrounding the injection molding system.

The fill settings (figure 21) regarding filling time, melt temperature, injection pressure limit and clamp force limit may stay on the preset values.

The pack settings (figure 22) may stay on the preset values.

4.3.2 User defined mold material

The H13 tool steel – the metal mold's steel alloy – was not a part of SolidWork's mold materials database. For that reason, it was necessary to create a user defined material with the values of the H13 tool steel. Figure 24 shows the settings window for defining a mold material in SolidWorks, with the correct values inputted. The needed values for H13 tool steel were available online. (*AISI Type H13 Hot Work Tool Steel*, n.d.; *Die steel H13*, 2012)



Mold Material	H13 tool steel	
Specific Heat	460	J/(kg-K)
Thermal Conductivity	24.3	W/(m-K)
Density	7800	Kg/m3
Shear Modulus	8.100000e+10	Pa
Thermal Expansion Coefficient	1.260000e-05	1/°C
Young Modulus	2.100000e+11	Pa
Poisson's Ratio	0.3	

Figure 24: User defined material, values for H13 tool steel

4.3.3 Simulation settings

Table 6: Values for injection molding simulation

Setting	Simulation 1	Simulation 2	Simulation 3
Mold temperature	150°C	175°C	200°C
Mold material	H13 tool steel	H13 tool steel	H13 tool steel
Polymer material	LSR*	LSR*	LSR*
Gate location	Gate configuration 1	Gate configuration 1	Gate configuration 1
Gate size	0.5 mm	0.5 mm	0.5 mm

*LSR product used for simulation: Dow Corning/LSR SILASTIC LC 70/2004

4.4 Thermal simulation

Thermal simulations were conducted on the cavity, core, and the complete assembly of all parts. This is an overview over which variables, with table 5 in mind, will be inspected by which thermal simulations:

- Cavity and core
  - T.1 and T2: Thermal simulations are conducted in order to find a number and location of heating rods that result in uniform heat propagation on the surfaces of the cavity and core, at the correct temperature.
- Assembly
  - T.3: Thermal simulations are utilized to figure out how the heating rods affect the runners, and how the cooling system affects the inserts.

A few features or values must be defined in order to run the simulations, these are:

1. Convection: which surfaces are subjected to convection and set the value of the convection coefficient.
2. Contact sets: faces that are in contact with each other and set the value for thermal resistance.
3. Temperature and/or heat power: select the parts/faces that supply heat to the surrounding parts.

Three simulation configurations were run:

1. Thermal configuration 1: Insert simulation (partially assembled)
2. Thermal configuration 2: Complete assembly simulation, metal inserts

It was decided that the thermal simulations should focus mainly on the metal inserts and not the plastic inserts of the prototype, and thus that the changes made to the design of any of the mold components as a result of the thermal simulations should optimize the mold assembly meant for mass production.

4.4.1 Setting the values

The values set for both thermal for convection and thermal resistance for the contact sets are rough estimates based on generalized values found on various webpages. See table 7 for these values.

For convection, it was decided to set the value to a medium-low estimate for forced convection of water (*Convective heat transfer*, 2003).

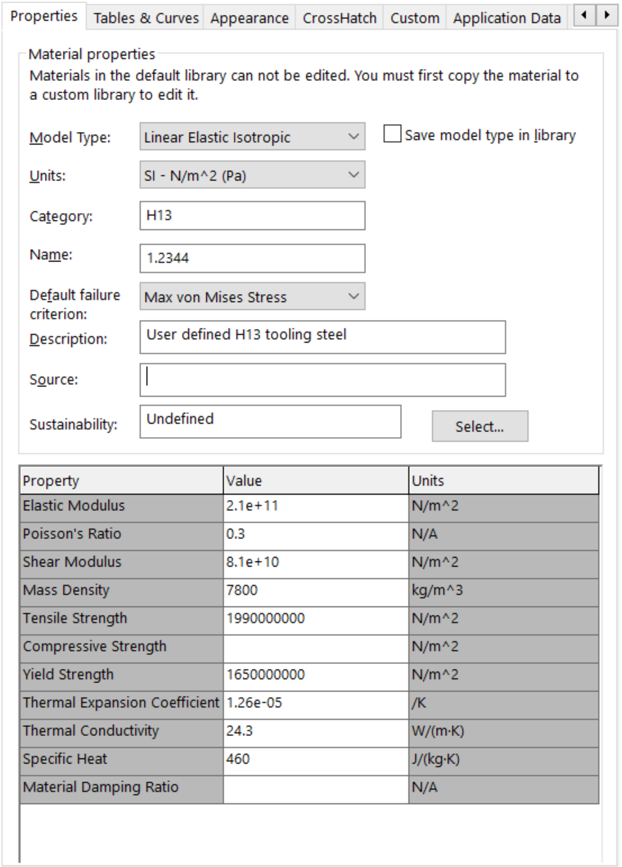
For the thermal resistance between contact sets, the value was set 0.5 higher than the high estimate of thermal resistance in stainless steels under a pressure of 10 atm (*Thermal contact resistance*, 2016).

Table 7: Values for thermal simulation

Simulation of …	**Temperature**	**Forced convection of water**	**Natural convection of air**	**Thermal resistance**
Cavity and core	185°C	1000 W/(m^2×K)	20 W/(m^2×K)	0.00045 K×m^2/W
Complete assembly	185°C	1000 W/(m^2×K)	20 W/(m^2×K)	0.00045 K×m^2/W

4.4.2 User defined material

The H13 tool steel material did not exist in the SolidWorks database, and the previously defined H13-material for the injection molding simulation only existed in another, inaccessible database for mold materials. Therefore, the H13 tool steel material had to be added. The required properties for defining a material were the same as for the injection molding mold material database, and the values were copied. See figure 25.



Material properties

Materials in the default library can not be edited. You must first copy the material to a custom library to edit it.

Model Type: Linear Elastic Isotropic

☐ Save model type in library

Units: SI - N/m^2 (Pa)

Category: H13

Name: 1.2344

Default failure criterion: Max von Mises Stress

Description: User defined H13 tooling steel

Source:

Sustainability: Undefined

Property	Value	Units
Elastic Modulus	2.1e+11	N/m^2
Poisson's Ratio	0.3	N/A
Shear Modulus	8.1e+10	N/m^2
Mass Density	7800	kg/m^3
Tensile Strength	1990000000	N/m^2
Compressive Strength		N/m^2
Yield Strength	1650000000	N/m^2
Thermal Expansion Coefficient	1.26e-05	/K
Thermal Conductivity	24.3	W/(m·K)
Specific Heat	460	J/(kg·K)
Material Damping Ratio		N/A

Figure 25: User defined material, H13 tool steel, for thermal simulation

4.4.3 Materials of the assembly

Table 8: The materials

Component	Insert simulation	Complete assembly simulation, metal inserts
Baseplate with runners	H13 tool steel	H13 tool steel
Backplate with runners	H13 tool steel	H13 tool steel
Baseplate	H13 tool steel	H13 tool steel
Inserts (cavity and core)	H13 tool steel	H13 tool steel
Heating rods	Alloy steel	Alloy steel

4.4.4 Meshes

Both thermal configuration 1 and -2M models were meshed with curvature-based mesh. Mesh controls were applied to thermal configuration 2 on the inserts, in order to get an even finer mesh there. Figures 26, 27 and 28 show the mesh settings. See figure 29 for the mesh of thermal configuration 1, figure 30 for the mesh of thermal configuration 2, and figure 31 for the mesh control applied to the inserts of thermal configuration 2.

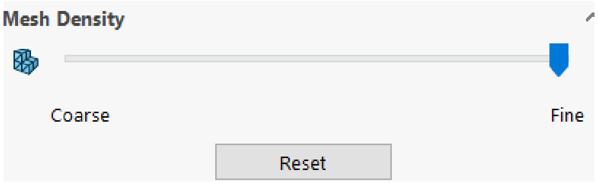
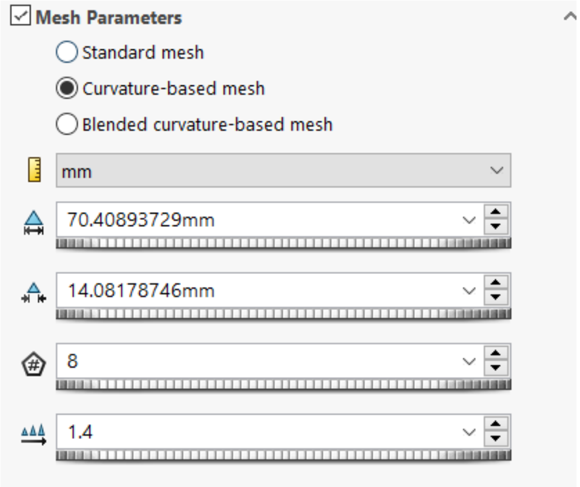

Mesh Density

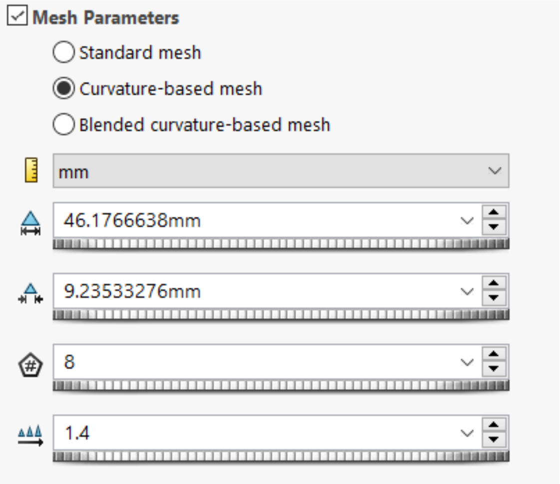
Figure 26: Mesh control on the inserts


☑ **Mesh Parameters**

○ Standard mesh
● Curvature-based mesh
○ Blended curvature-based mesh

mm
70.40893729mm
14.08178746mm
8
1.4

Figure 27: Mesh of thermal configuration 2


☑ **Mesh Parameters**

○ Standard mesh
● Curvature-based mesh
○ Blended curvature-based mesh

mm
46.1766638mm
9.23533276mm
8
1.4

Figure 28: Mesh of thermal configuration 1

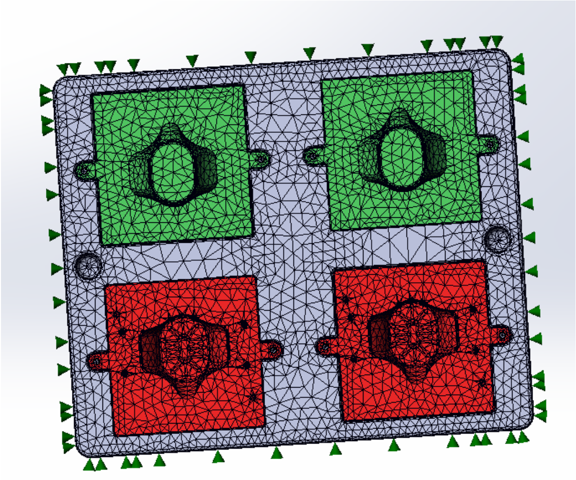


Figure 29: Mesh of thermal configuration 1

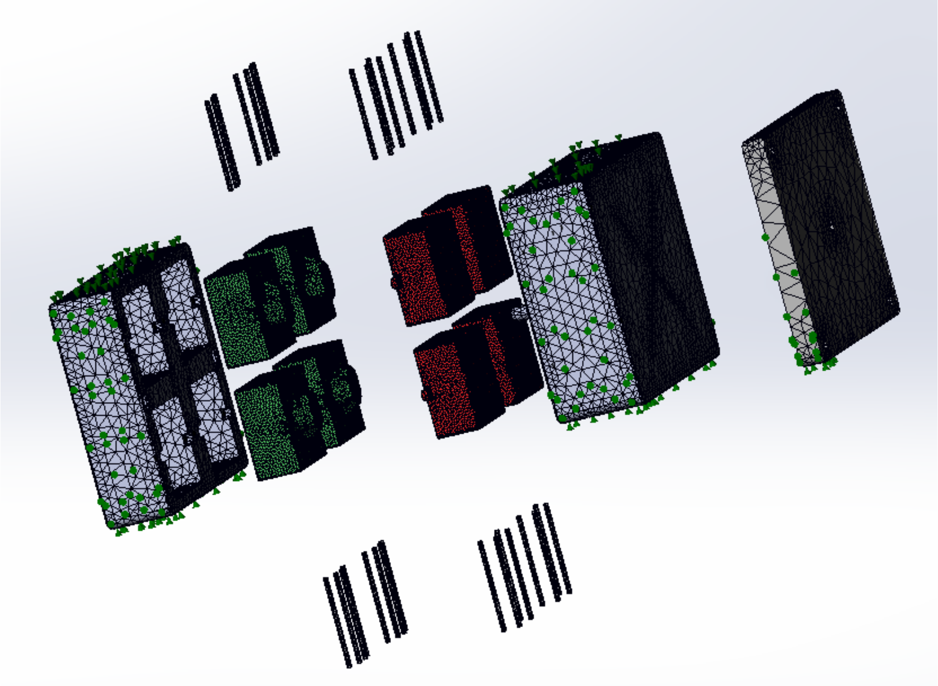


Figure 30: Mesh of thermal configuration 2

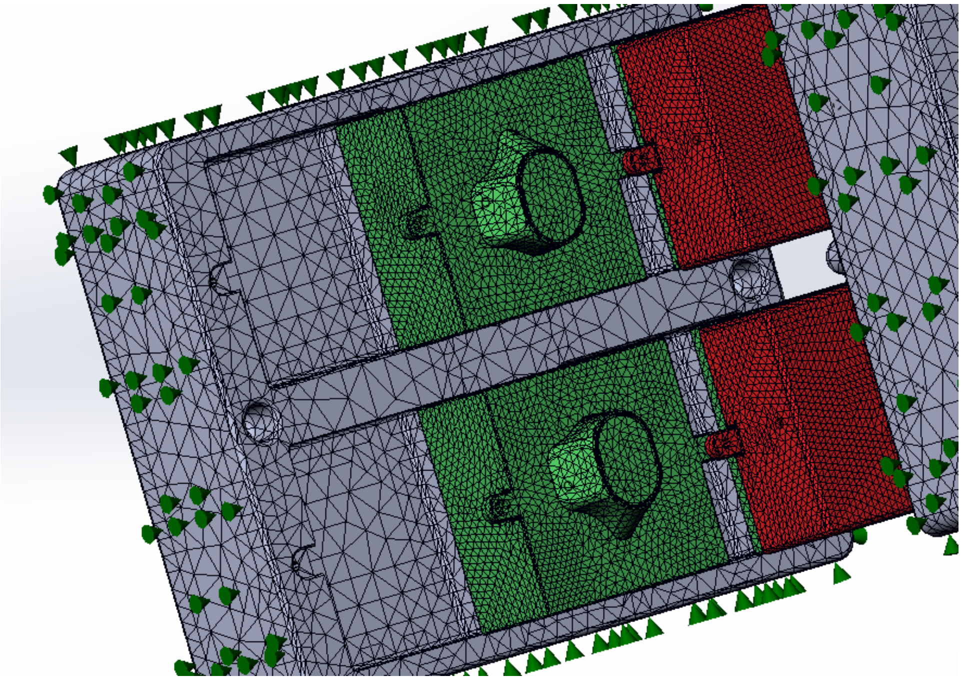


Figure 31: Mesh control and the mesh of the inserts

4.4.5 Contact sets

Contact sets were defined for every surface that touched between all the components except for the heating rods. Figures 32 and 33 show the contact sets for thermal configuration 1 and -2, respectively.

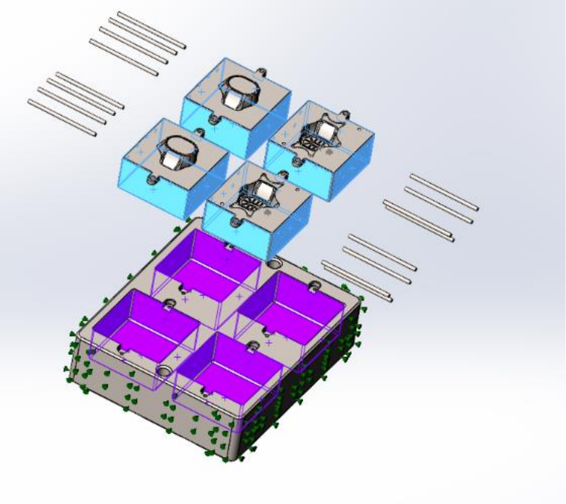


Figure 32: Contact sets of thermal configuration 1

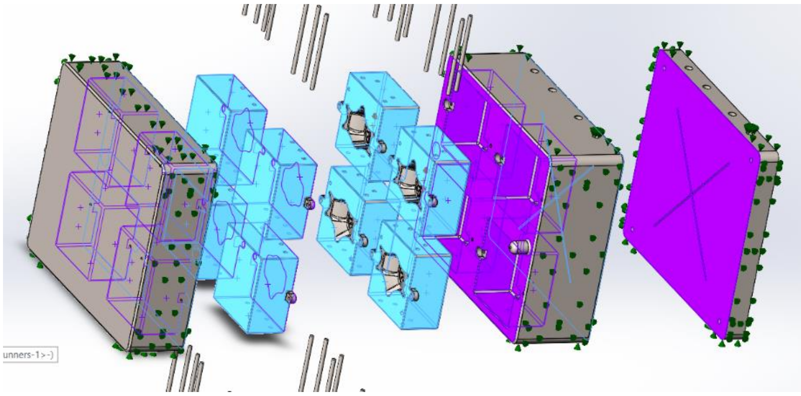


Figure 33: Contact sets of thermal configuration 2

4.4.6 Convection

Convection was added to all surfaces that are exposed to air, which are the sides of the baseplates and the backplate.

4.5 3D-printing: FDM inserts

The inserts made through FDM are meant for inspection the finer details of the core/cavity contours, the insert design was simplified for this print. Material that is not critical for the mask contour of the core and cavity was therefore removed. See figure 34 and 35.

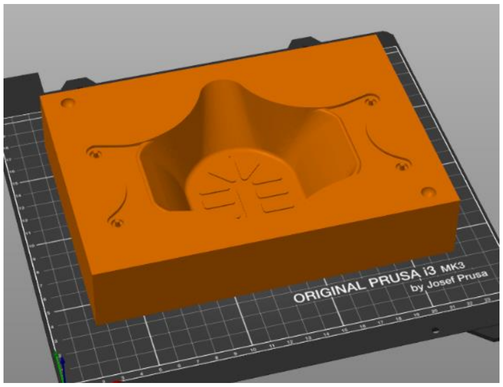


Figure 34: The cavity in prusaslicer

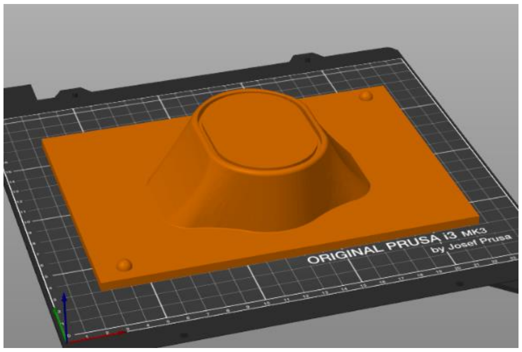


Figure 35: The core in prusaslicer

The SolidWorks CAD models were imported to prusaslicer, a software for the prusa printer that slices the CAD model into layers and thereby converts it to an STL-file. The 3Dprinter is able to read the instructions from this format and creates the model layer by layer. The print settings were as follows:

Table 9: Parameters for 3D-printing

Parameter	Print 1	Print 2
Layer height	0,2 mm	0,2 mm
Nozzle diameter	0,4 mm	0,4 mm
Infill	15%	10%
Material	PLA	PLA
Nozzle temperature	215° C	215°C
Bed temperature	60°C	60°C

As seen in table 9, the settings were mostly the same for both the prints.

4.6 3D-scanning and print inspection

The 3D scanning was done using the Creaform Handyscan 700 and the belonging software VXelements. The part that was to be scanned was put on a small and portable rotating table. The table had positioning targets spread across its surface. These targets were scanned so that the scanning software could utilize these as reference points. The reference points help the software work out the scanner’s position in relation to what is scanned. The part was then put onto the table, which could be rotated freely as long as the part’s relative position to the table stayed the same throughout the scanning process.

After a quick test scan it was discovered that some of the more intricate details of the core and cavity was rather difficult to scan. Targets were put on the part to help with this, and the shutter speed was also increased in order to make it easier to catch the smaller details. The increased shutterspeed had its drawbacks as it increased the amount of noise that the scanner picked up around the part. Most of the noise was however easily removed through the use of cutting planes within the software.

The scanned 3D model of the 3D printed mold inserts could be exported to solidworks and a comparison of the geometry between the original CAD model and the 3D scanned model could be performed to determine the accuracy of the print. Figure 36 and 37 are from the scanning process.

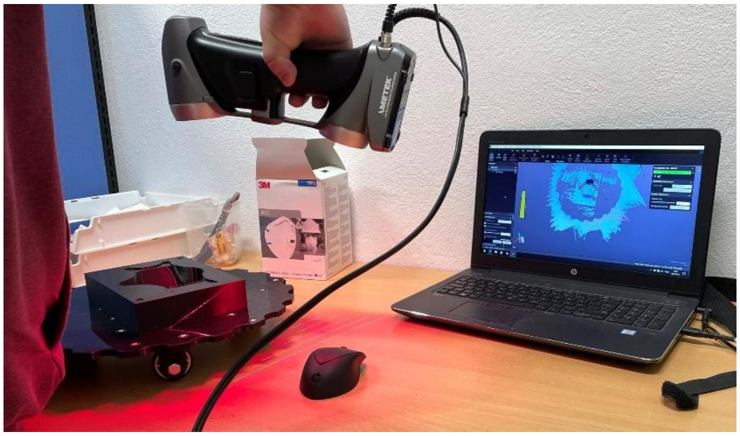


Figure 36: Scanning the cavity

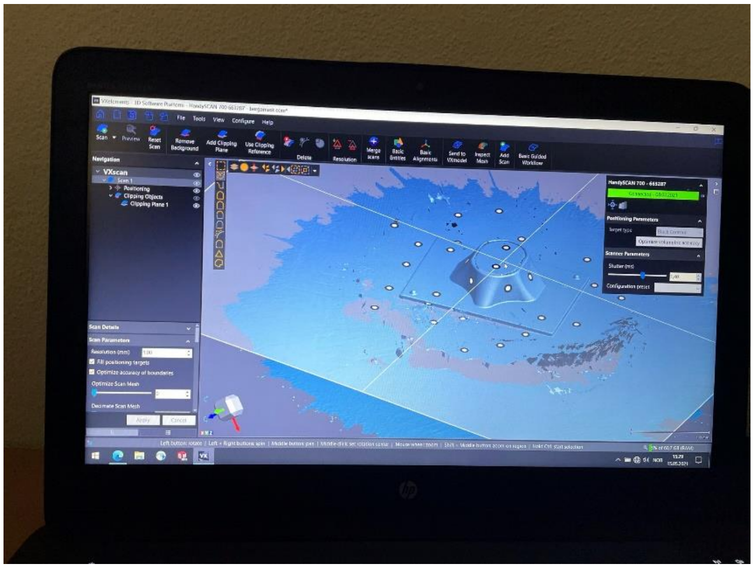


Figure 37: The scan displayed real-time in VXelements

5 Results

5.1 Design

5.1.1 Assembly

Figure 38 through 40 show the final design of the assembly of the mold and baseplates.

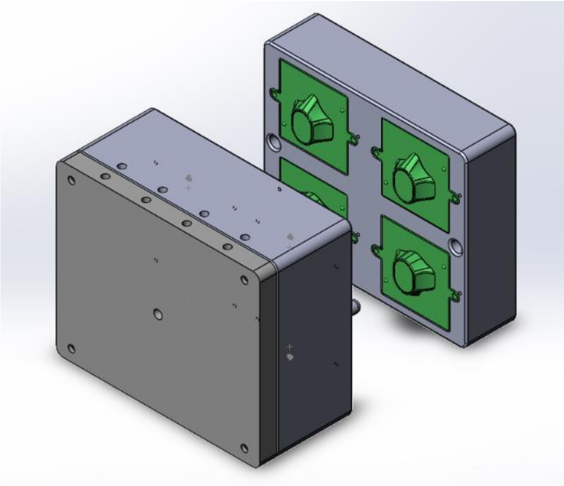


Figure 38: Assembly, open

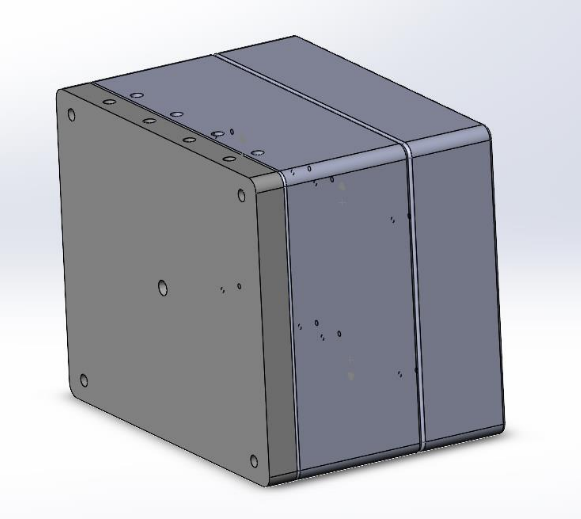


Figure 39: Assembly, closed

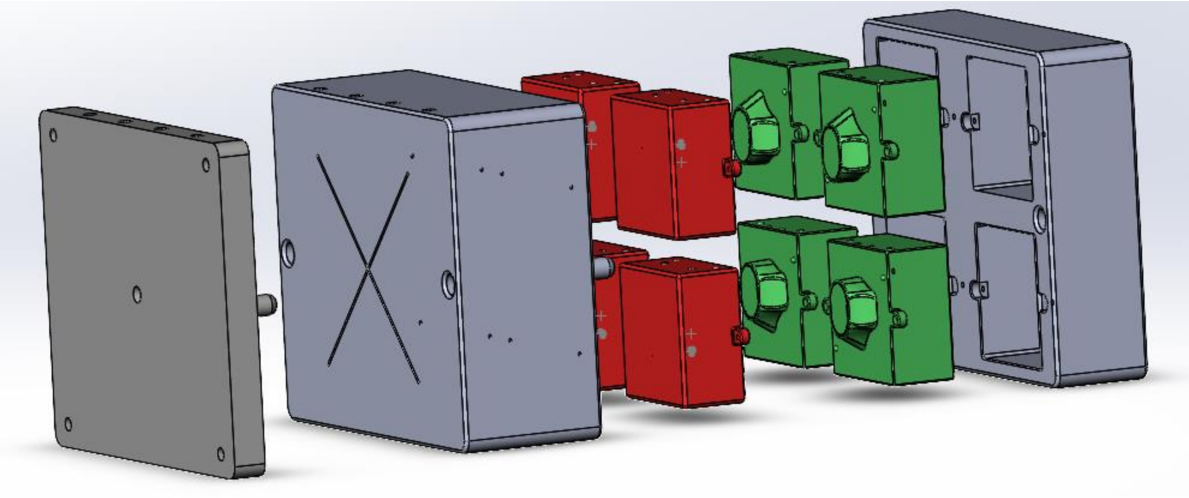


Figure 40: Assembly, exploded view

5.1.2 Baseplates and backplate

Figure 41 and 42 show the final design of the baseplates and figure 43 shows the final design of the backplate.

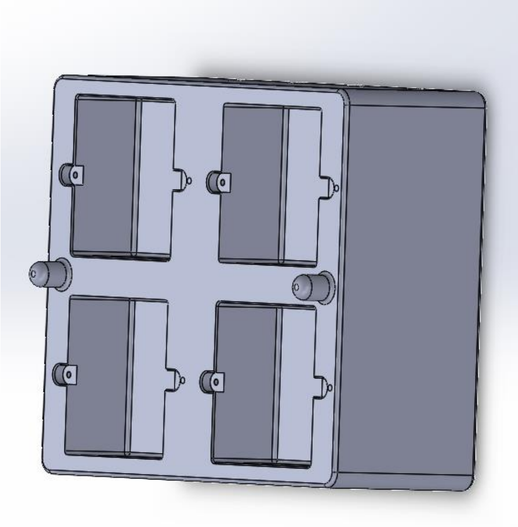


Figure 41: Final design of baseplate with runners

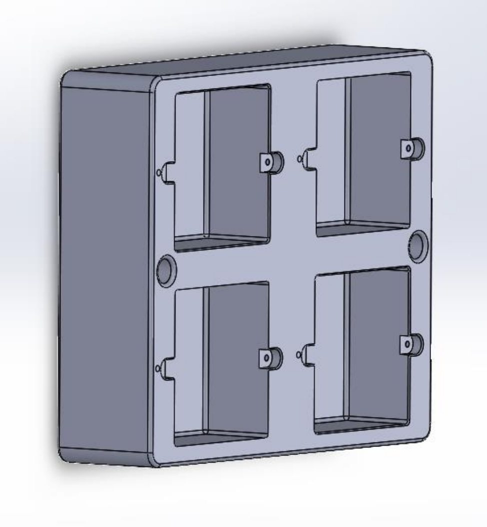


Figure 42: Final design ejection side baseplate

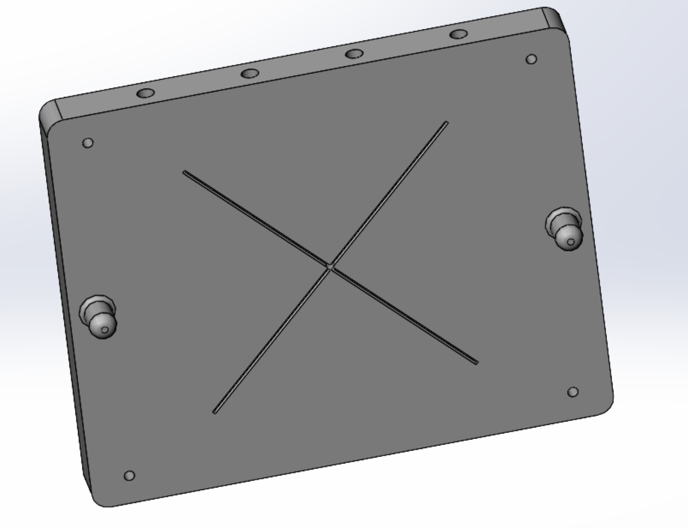


Figure 43: Final design of the backplate

5.1.3 Cavity and core

Figures 44 and 45 show the final design of cavity and core, respectively.

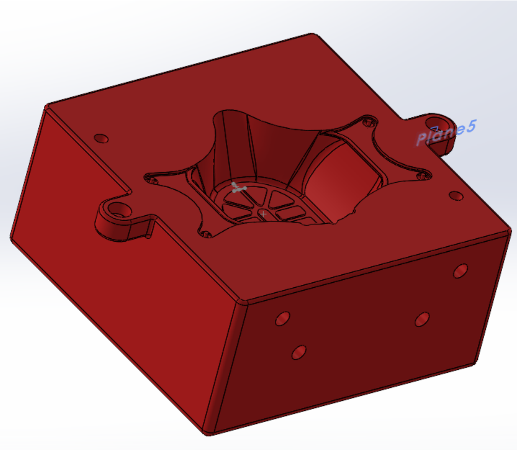


Figure 45: Cavity

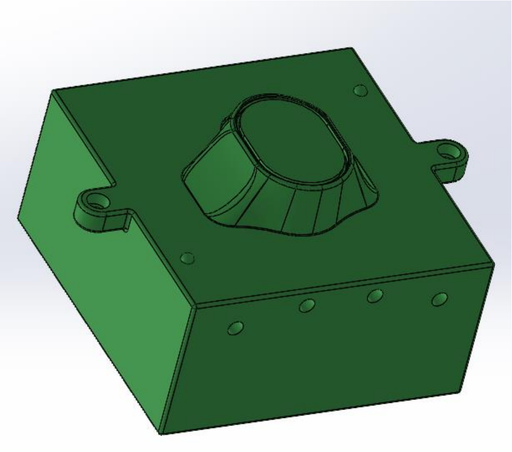


Figure 44: Core

5.2 Injection molding simulation

After a series of injection molding simulations were performed a couple of key parameters for each configuration were performed. These can be seen below in table 10.

Table 10: Simulations with H13 mold inserts at different mold temperatures.

Parameter	Simulation 1: H13 mold at 150 °C	Simulation 2: H13 mold at 175 °C	Simulation 3: H13 mold at 200 °C
Cycle time (s)	53,27	33,0	10.06
Injection pressure (MPa)	4,18	3,45	3,21
Clamp force (tonne)	3,72	3,26	3,09
LSR temperature (°C)	150	175	200
Total stress displacement(mm)	1,16	1,30	1,35
Volumetric shrinkage (%)	9,99	11,96	13,86

5.3 Thermal simulation

5.3.1 Thermal configuration 1

Figure 46 shows the final result of the thermal simulation of thermal configuration 1.

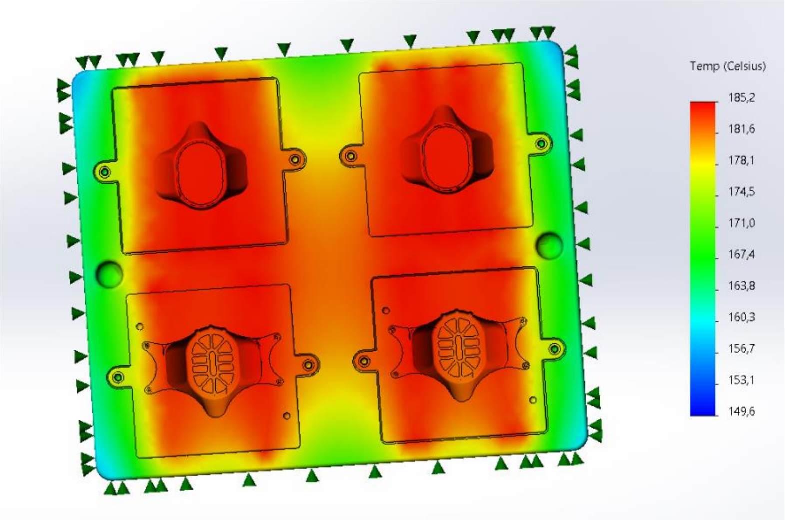


Figure 46: Heat propagation on mold surface

5.3.2 Thermal configuration 2

One of the main focuses of the simulation of thermal configuration 2 was to ensure that the runner system had cool enough temperatures to avoid the LSR to solidify in the channels, but at the same time not affect the cavity insert so much that it, too, was cooled down. Figures 47 through 49 show the results of the thermal simulation where figures 47 and 49 include slight changes in order to optimize the system’s functionality. The results in figure 49 are the final results.

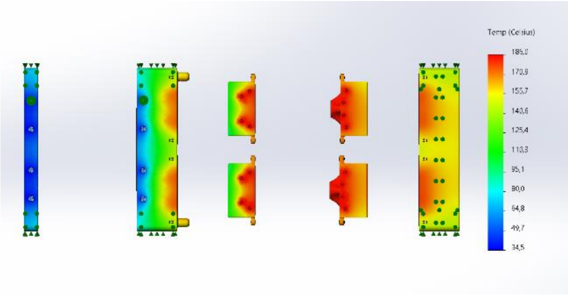


Figure 48: Simulation 1: Heat propagation of thermal configuration 2 seen from above

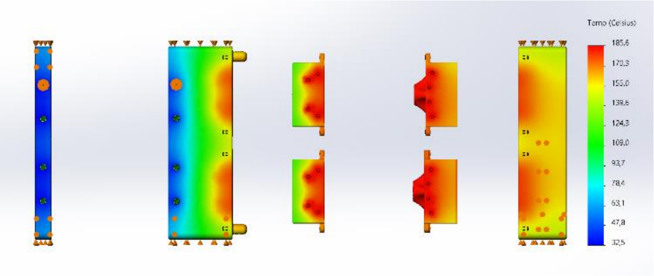


Figure 47: Simulation 2: Heat propagation of thermal configuration 2 seen from above –baseplate with runners 50mm thicker

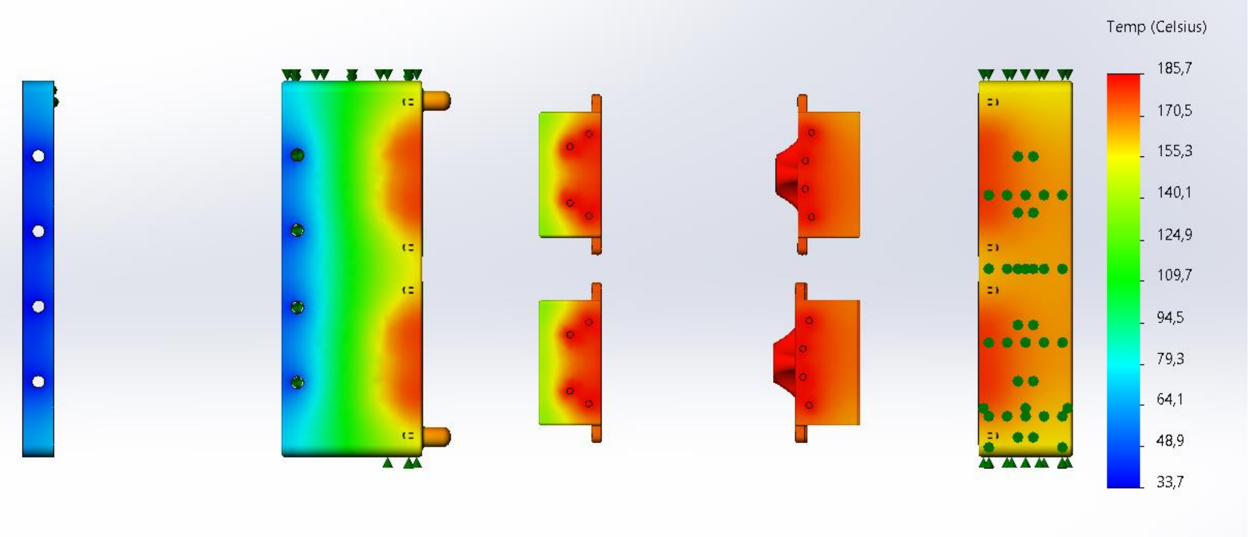


Figure 49: Heat propagation of thermal configuration 2 seen from above –baseplate with runners 25mm thicker than previous iteration; altered cooling channels

The alterations that were done was changing the depth of the baseplate with runners and changing the cooling channels' location. Figure 51 shows the original design of the baseplate with runners, and figure 50 shows the final design of the baseplate with runners after the alterations were done. Figure 52 shows the original location of the cooling channels and figure 53 shows the final location of the cooling channels. The location of the cooling channels was changed equally on both the backplate and the baseplate with runners.

Changing the depth of the baseplate was a measure reduce the temperature at the back of the baseplate where the runner channels are located. Altering the locations of the cooling channels was in order to optimize its functionality of cooling down relevant surface areas. See figures 54 and 55 for probed results of the runner and the cavity before the alterations were done and see figures 56 and 57 for the probed results after the alterations.

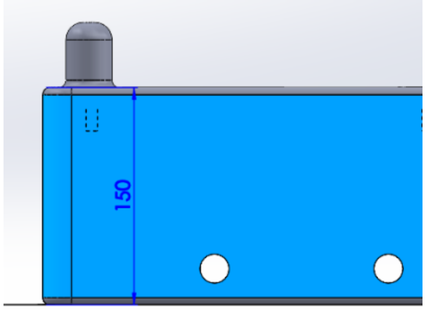


Figure 51: Original depth of baseplate w/runners

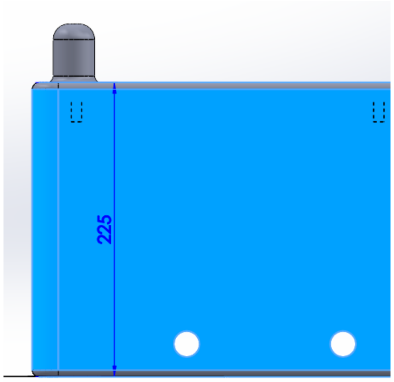


Figure 50: Depth of baseplate w/runners after alterations

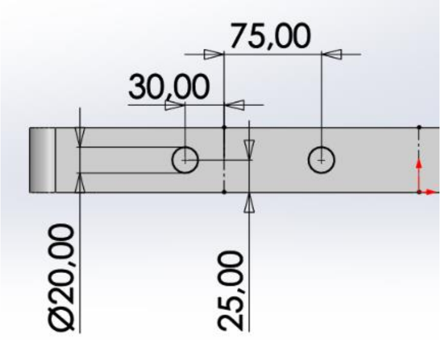


Figure 52: Original cooling channels layout (horizontal dimensions are in relation to the gate location

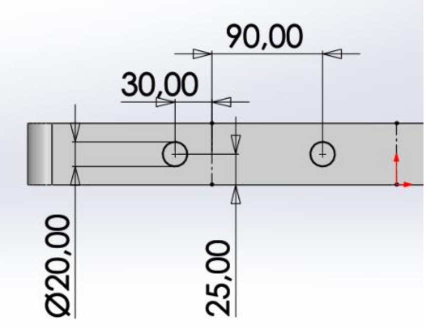
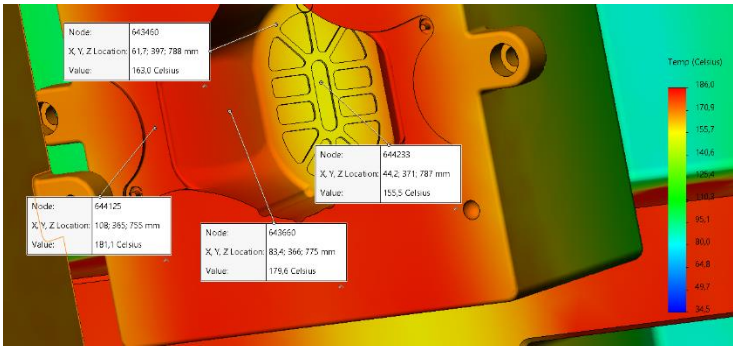


Figure 53: Cooling channels layout after alterations (horizontal dimensions are in relation to the gate location)



Node:	643460
X, Y, Z Location:	61,7; 397; 788 mm
Value:	163,0 Celsius

Node:	644233
X, Y, Z Location:	44,2; 371; 787 mm
Value:	155,5 Celsius

Node:	644125
X, Y, Z Location:	108; 365; 755 mm
Value:	181,1 Celsius

Node:	643660
X, Y, Z Location:	83,4; 366; 775 mm
Value:	179,6 Celsius

Figure 54: Before: heat propagation at various points on the cavity surface of original design

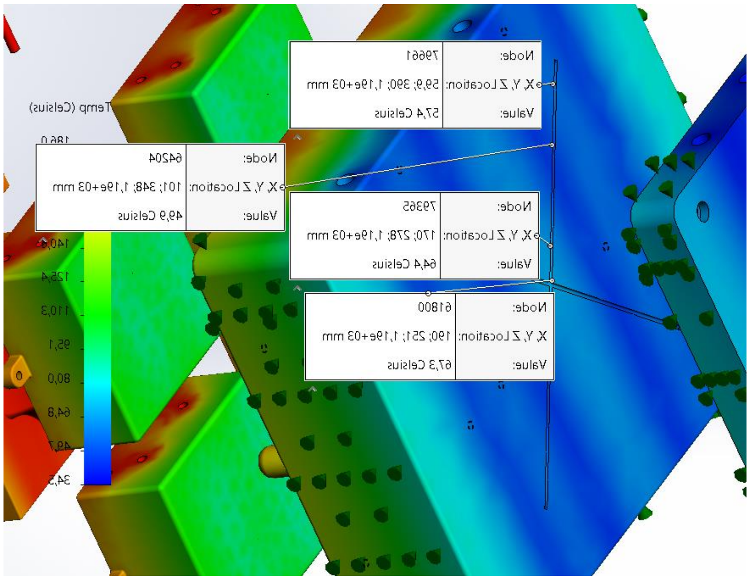


Figure 55: Before: heat propagation at various points in the runner system of original design

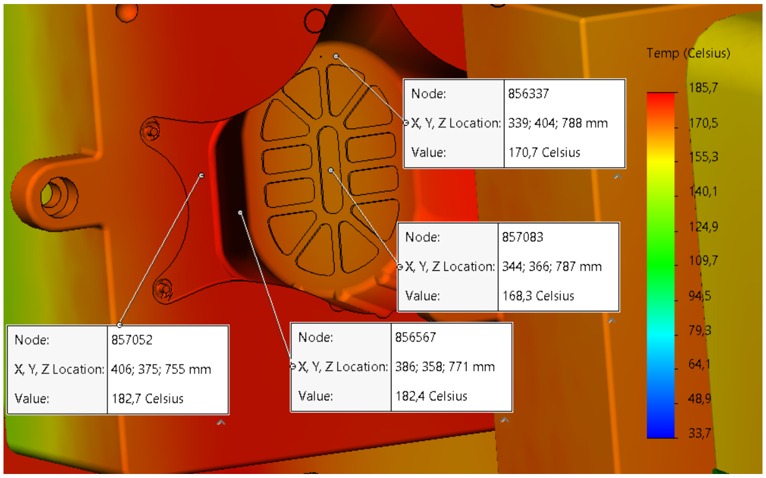
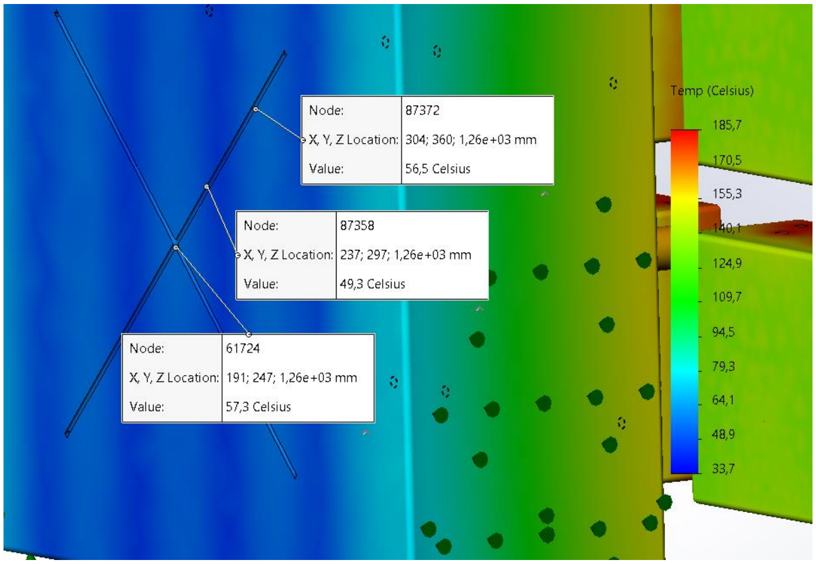


Figure 56: After: heat propagation on various points on the surface of the cavity on final design



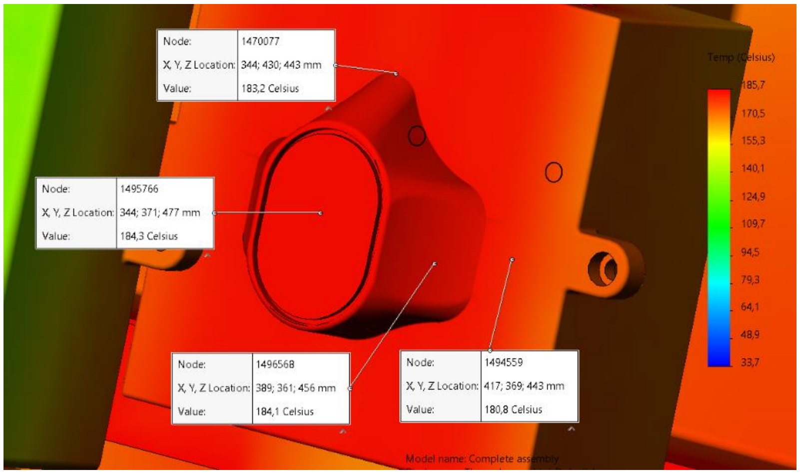
Node: 87372
X, Y, Z Location: 304; 360; 1,26e+03 mm
Value: 56,5 Celsius

Node: 87358
X, Y, Z Location: 237; 297; 1,26e+03 mm
Value: 49,3 Celsius

Node: 61724
X, Y, Z Location: 191; 247; 1,26e+03 mm
Value: 57,3 Celsius

Figure 57: After: heat propagation at various points in the runner system on the backside of the baseplate w/runners on final design

The core reached slightly higher temperatures and a more equal distribution of heat. See figure 58.



Node: 1470077
X, Y, Z Location: 344; 430; 443 mm
Value: 183,2 Celsius

Node: 1495766
X, Y, Z Location: 344; 371; 477 mm
Value: 184,3 Celsius

Node: 1496568
X, Y, Z Location: 389; 361; 456 mm
Value: 184,1 Celsius

Node: 1494559
X, Y, Z Location: 417; 369; 443 mm
Value: 180,8 Celsius

Figure 58: Heat propagation on core surface on final design

5.3.3 Heating rod placement

See figure 59 and 60 for the final layout of the heating rods in the inserts.

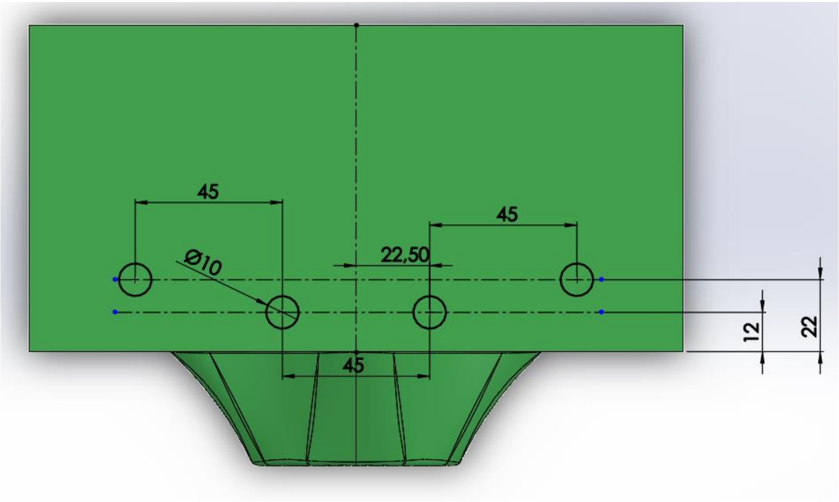


Figure 59: Heating rod locations of core

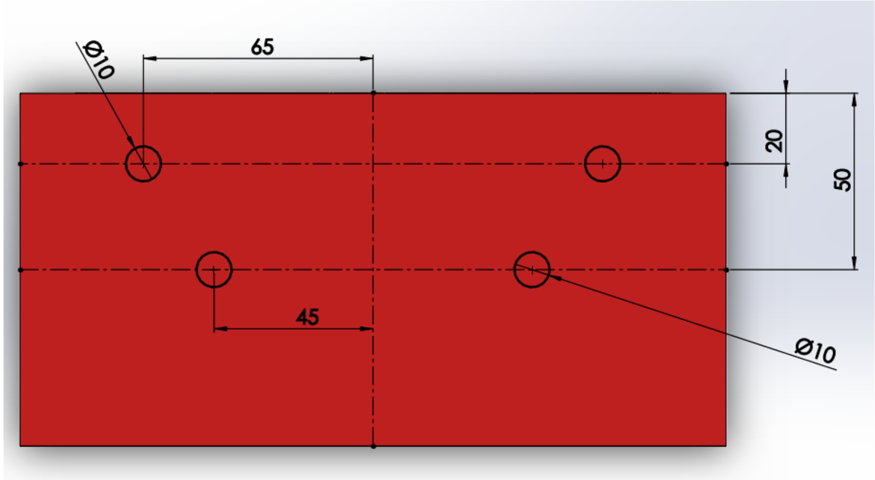


Figure 60: Heating rod locations of cavity

5.4 3D-printing

During the first print an error known as layer shifting occurred on the core insert. This means that the layer is printed at a slightly shifted location which results in a wrong geometry on the printed part. This can be seen in figure 61

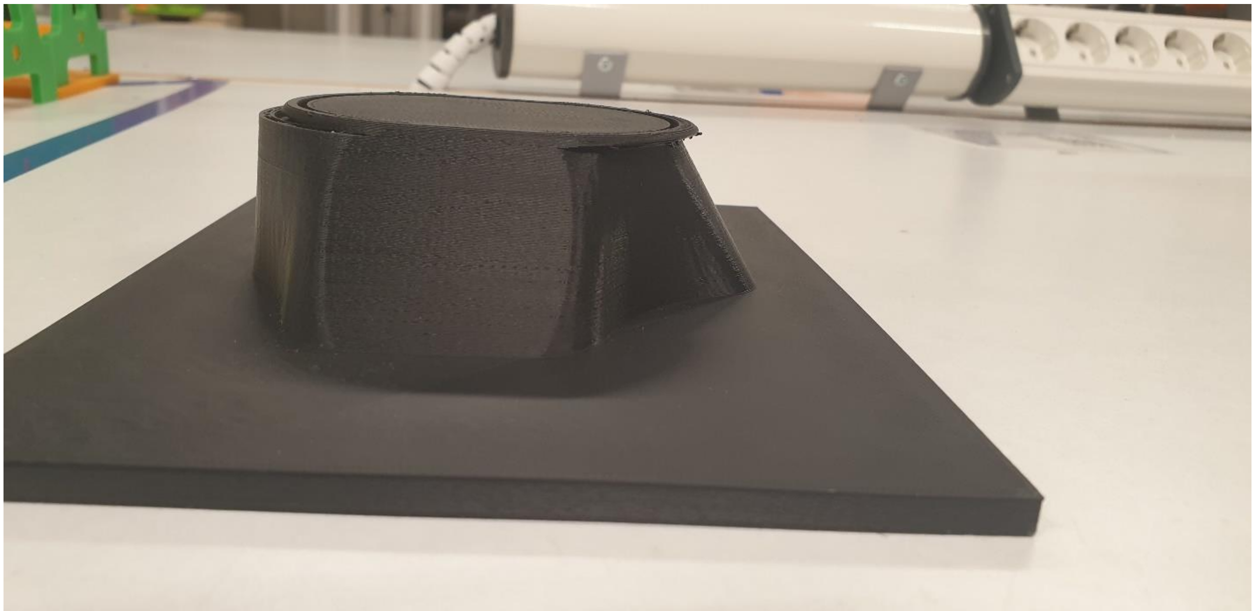


Figure 61: Layer shift

With the exception of the layer shift the result of print 1 was satisfactory. As the inserts were more than sturdy enough it was decided to lower the infill for the print 2, as that would mean an even cheaper and faster production process. Other than that, the print settings were kept the same as in print 1. Print 2 was a success, as detailing and surface roughness was both very good. With eye inspection the accuracy and overall quality looks good.

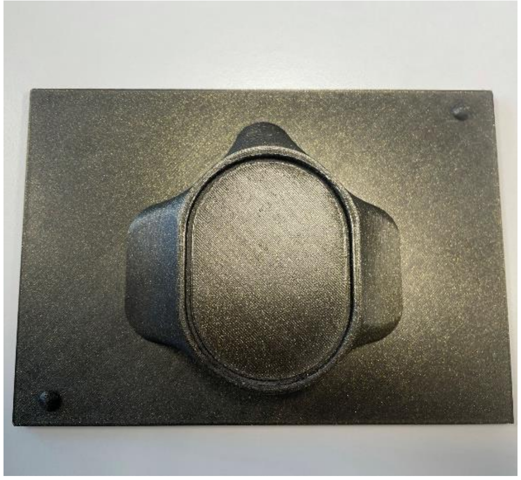


Figure 63: 3D-printed core

Figure 62: 3D-printed cavity

5.5 3D-scanning and visual inspection

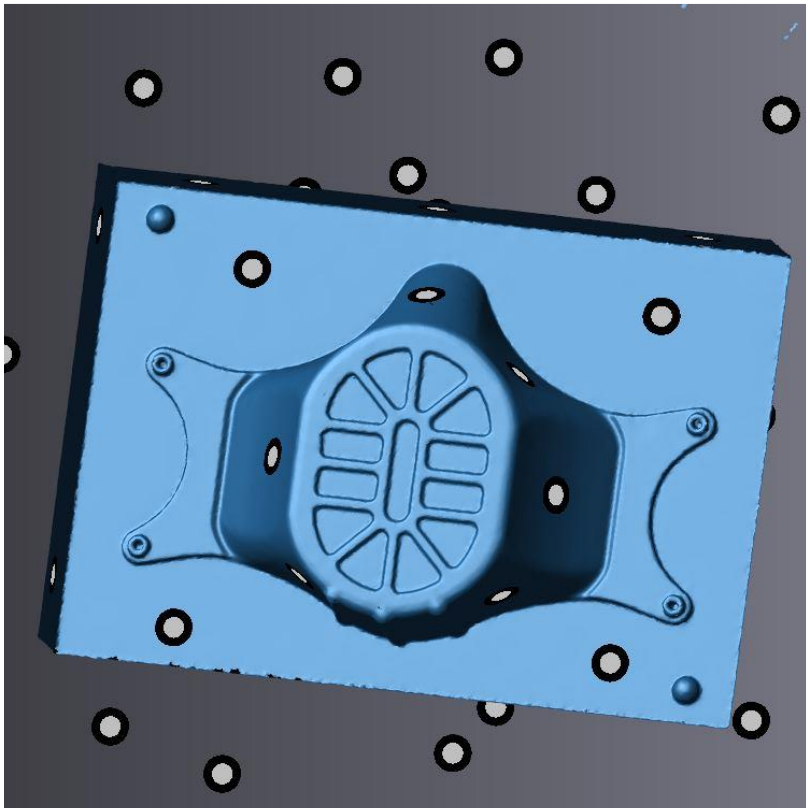


Figure 64: 3D-scan of the cavity

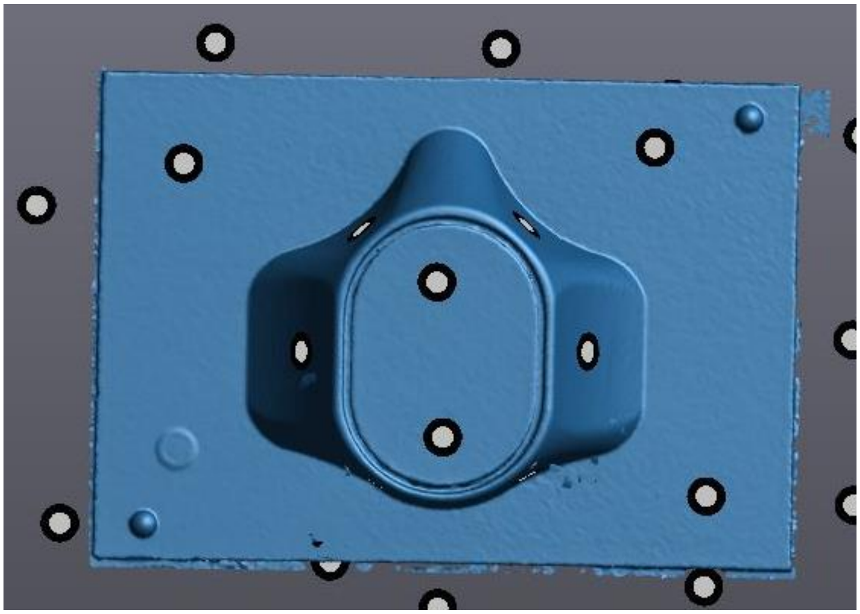


Figure 65: 3D-scan of the core

Figure 67: Closeup of the details on the filter cover of the cavity 3D-scan

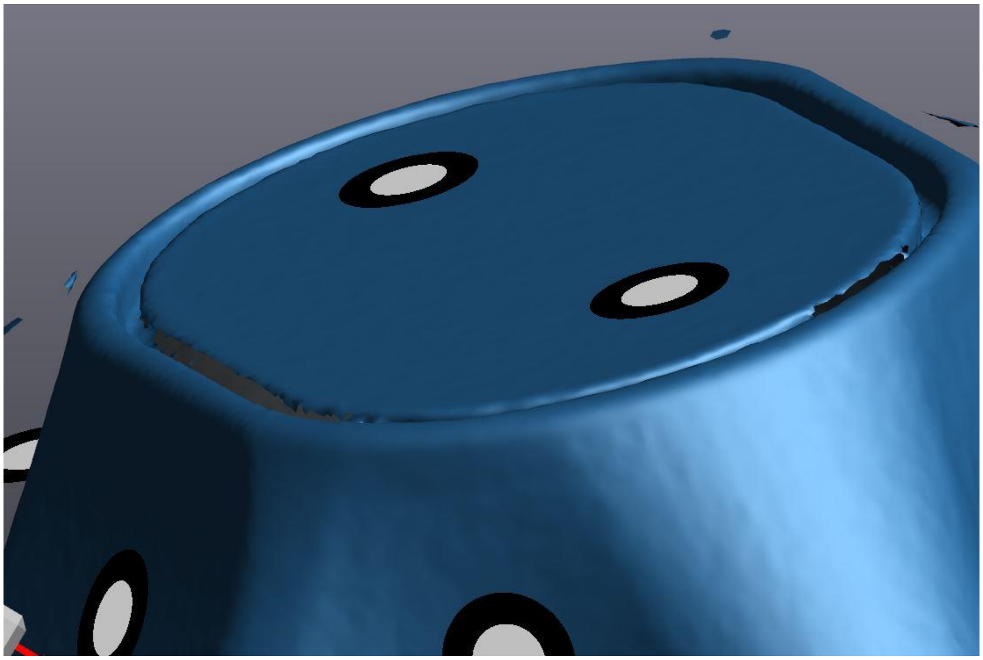


Figure 66: Closeup of the filter cover of the core 3D-scan - the model is not completely filled where the geometry gets tricky

6 Discussion and conclusion

6.1 Method

The methods that were outlined, the process of development and the process of design, proved to be effective processes to follow.

A couple of small changes were made to the process of development (figure 68), for instance the removal of a redundant stage: digital design. Earlier iterations had a third stage in phase 1 of the process. However, as digital design already was a part of both stage 1 and stage 2, there was no real necessity for a separate stage for digital design.

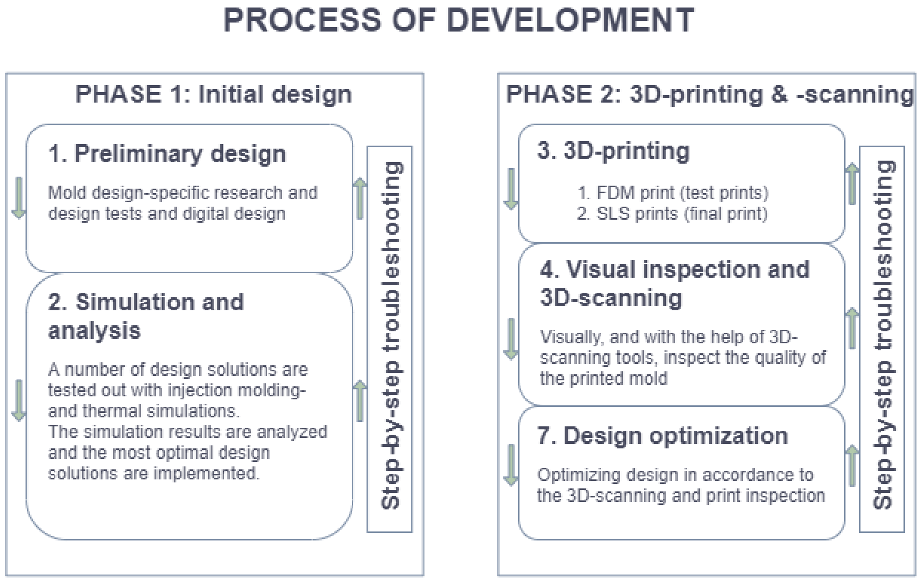


Figure 68: Process of design

Furthermore, as the 3D-printing, 3D-scanning and optimization stages did not amount to much with regards to the results of this thesis, the process of development has not yet been fully tested.

6.2 Design

The final design for the liquid injection molding components is conceptual, as of yet. It was decided not to implement threaded holes for fastening as the type of injection molding machine is still unknown.

There might be need for an insulation plate on the backside of the baseplate on the ejection side of the injection molding machine, in order to insulate the rest of the injection molding machine from the heat generated by the inserts.

6.3 Injection molding simulations

The injection molding simulation was performed in SolidWorks plastics. This is a pretty basic simulation analysis tool, however it gives an idea as to how the injection process will go. It also calculates an estimate for a lot of variables tied to the injection process, like injection pressure, clamping force, stress displacement and so on.

The results of the simulations performed shows that variation of the mold temperature results in significant changes to some of the parameters that has been focused on. A higher temperature results in lower cycle time, injection pressure and clamp force, but in return results in a larger total stress displacement and volumetric shrinkage. The range in injection pressure and clamp force is well within what H13 can withstand. The shrinkage and displacements needs to compensated. This can be done by increasing the hold pressure and hold time, and through post curing. For at all temperatures, just in a slightly bigger degree when running at 200°C. The most drastic change can be seen in cycle time. When increasing temperature from 150°C to 200°C the cycle time is reduced from 53,27 seconds to 10,06 seconds. The conclusion to be drawn here is that a higher temperature is wanted as it drastically speeds up the production time, while minimally increasing the strain on the mold.

6.4 Thermal simulations

6.4.1 Thermal configuration 1

The simulation on the cavity and core were at first done one-by-one, but this could not produce realistic results. Because of that, the simulation of the cavity and core was done assembled to one of the baseplates, specifically the baseplate without runners. The location and number of the heating rods was altered slightly to achieve uniform heat propagation on the mold surfaces.

However, simulations of thermal configuration 1 could not give completely realistic results either, because the contact set between the baseplates missed, which would, when applied, give the surfaces a better spread of heat as well as make them reach slightly higher temperatures. The reason for this is that when a heated surface is exposed to another heated surface, it will will not lose as much heat as it would from convection caused by air.

6.4.2 Thermal configuration 2

The results of the final simulation of configuration 2 were satisfactory. The temperature in the runners were low in enough to prevent the LSR to solidify while being transported through them. At the same time, the temperature on the surface of the cavity, as well as the heat propagation, were within a tolerable interval. Both the heat propagation and the temperature on the surface of the core were near perfect.

When the temperature on the mold surface is not perfectly uniform, it might lead to the molded part not being equally cured everywhere. However, this is not necessarily a too big of a problem. As mentioned in chapter 2.4.2, a part might be 75-90% cured when it is ejected from the mold. Given that the difference of heat propagation on a mold surface is within a couple of degrees Celsius, the product should not be impacted much. For the cavity, the total difference in heat between the low and high points are approximately 14°C.

However, since the mask’s cross-section is quite narrow on the filter cover, compared to the side pieces, and since the surface-to-volume ratio is larger in the filter cover region, because of the intricate details and holes in the filter cover, the effect of the lower temperatures becomes less even less important. A higher surface-to-volume ratio means that more surface is heated per unit of volume, which leads to quicker heating of the adjacent material.

Also considering how the cross-section of the ear-pieces are the thickest of the mask, this region will solidify slower than the other regions of the mask.

Considering all of this, it looks as though the 14°C difference of the heat spread of the cavity does not matter all that much.

6.5 Material of prototype and the limitations of plastic molds

Multiple materials have been considered for prototypes of the mold inserts. Most notably PA2200 and PA3200GF. Both of these materials are easily accessible, relatively cheap and can be printed by the EOS P395 available at the lab in NTNU Gjøvik. The precision and surface fineness of an SLS printed product of these materials would make for a great mold. However, these materials come with a drawback: their low heat deflection temperature while pressurized. Either the mold temperature, injection pressure, clamping force or all three would need to be lowered in order to cast a product in the prototype mold.

6.5 Production of mold inserts

The intended production method of the inserts, as well as both the baseplates and the backplate, is SLS 3D-printing. For the production of an eventual plastic prototype mold, SLS is also intended. The SLS 3D-printing was not completed, mainly because there would not be any injection molding machine available to test it in. The cost of printing PA2200 is 1000kr per cm of part thickness and spending that kind of money on a part that would not be tested and may not have the right dimensions, was not worth it.

6.6 Suggestion for prototyping

An alternative for testing the castability of the mold inserts with a 3D printed plastic prototype would be to manually fill the LSR into the cold inserts, clamp the two inserts together and put them in an oven. This is a workaround for the heat deflection temperature as it would remove the injection pressure and lower the clamping pressure. In theory this method could be used for both a PA2200 and a PLA prototype mold. One would have to

6.8 SolidWorks plastics vs Digimat Moldex3d

Digimat is a software that, among other things, have a simulation software for injection molding analysis, Moldex3D. Originally, it was planned that Digimat would be utilized for the injection molding simulations for this project. However, there was a problem with the school’s license keys needed to activate the software. Access to Digimat was not obtained until

30.04.2021, 20 days before the thesis was due. At this point, SolidWorks Plastics had already been used for several injection molding simulations.

Digimat’s injection molding simulation software is presumably considerably better and more advanced than SolidWork’s. It is also much harder to learn. Too difficult to learn and make use of in the last twenty days of the project. Because of this, Digimat was excluded from this thesis’ work.

7 Conclusion and further work

7.1 Conclusion

The goal of this thesis was to lay the groundwork for the production of a modular mold for injection molding of medical equipment, and in this case, specifically a facemask, during a pandemic. Even if the project didn’t result in a physical mold that can be used for injection molding, it’s fair to conclude that a good deal of groundwork has been done.

A process of development for injection molds to be produced through 3D-printing was established, which describes a procedure to follow from start to finish when developing a mold, and includes the preliminary design, simulation analysis, 3D printing, visual inspection and 3D scanning and design optimization. A process of design was established as well, that outlines the most important parameters regarding the design process of a liquid injection mold.

A conceptual design of a modular mold system was created as well, on which both thermal- and liquid injection molding simulations were performed. The thermal analysis yielded good results and lead to iterations on the design of the baseplate with runners that optimized its functionality with regards to how the heat spreads through it.

The baseplates achieved a design that allows for modularity by having cutouts into which inserts with a variety of product molds can be fastened and utilized for liquid injection molding.

The design needs more work in order to be finalized and ready for production, but this must be done when information regarding the injection molding machine is confirmed, and with the use of a more thorough mold analysis tool such as Digimat.

Our report also includes valuable information and theory regarding liquid injection molding, 3D-printing, 3D-scanning, and how to go forward with developing a mold for injection molding.

7.2 Further work

Further work is suggested on utilizing the process of development for developing a 3D-printed mold for liquid injection molding, or ordinary, resin-based injection molding, as the group was unable to fully test it.

More work that needs be done include:

- Vents
    - The venting of the mold.
- Gate
    - The exact gate design and product.
- Further simulation with Digimat
    - To improve the quality of the injection molding simulations.
- Applying the design to a specific machine
- Create the other baseplate configurations for small parts and large parts.
- Research and test a PA2200 prototype mold in Digimat

Sources

About H13 Tool Steel (n.d.). Available at: https://www.desktopmetal.com/resources/h13-tool-steel (Accessed: 13.04.2021.

Aexit Injection Mold Water Heater (n.d.). Available at: https://www.amazon.com/Aexit-Injection-Cartridge-Heating-Elements/dp/B07KTS4M5F (Accessed: 19.05.2021.

AISI Type H13 Hot Work Tool Steel (n.d.). Available at: http://www.matweb.com/search/datasheet_print.aspx?matguid=e30d1d1038164808a85cf7ba6aa87ef7 (Accessed: 01.05.2021.

Basics of injection molding design (n.d.). Available at: https://www.3dsystems.com/quickparts/learning-center/injection-molding-basics (Accessed: 04.03.2021.

BergaMASK®: the story (n.d.). Available at: https://www.stilgomma.it/bergamask_history.php (Accessed: 15.05.2021.

Bont, M., Barry, C. and Johnson, S. (2020) 'A review of liquid silicone rubber injection molding: Process variables and process modeling'. Available at: https://onlinelibrary.wiley.com/doi/pdf/10.1002/pen.25618 (Accessed 12.05.2021).

Computer simulation (2017). Available at: https://www.britannica.com/technology/computer-simulation (Accessed: 19.05.2021.

Convection (2020). Available at: https://help.solidworks.com/2020/English/SolidWorks/cworks/c_Convection.htm?verRedirect=1 (Accessed: 15.05.2021.

Convective heat transfer (2003). Available at: https://www.engineeringtoolbox.com/convective-heat-transfer-d_430.html (Accessed: 04.05.2021.

Die steel H13 (2012). Available at: https://www.substech.com/dokuwiki/doku.php?id=die_steel_h13 (Accessed: 01.05.2021.

DIN 1.2344 tool steel (2020). Available at: https://en.wikipedia.org/wiki/DIN_1.2344_tool_steel (Accessed: 12.04.2021.

Draft angles in injection molded plastics. n.d.

EOS polymers for additive manufacturing (n.d.). Available at: https://www.eos.info/en/additive-manufacturing/3d-printing-plastic/sls-polymer-materials/polyamide-pa-12-alumide (Accessed: 02.03.2021.

Gerdeen, J. C., Lord, H. W. and Rorrer, R. A. L. (2006) *Engineering Design with Polymers and Composites.* Boca Raton: CRC Press, p. 233.

H13 Tool steel (n.d.). Available at: https://www.hudsontoolsteel.com/technical-data/steelH3 (Accessed: 13.04.2021.

imPURE (n.d.). Available at: https://www.impure-project.eu/ (Accessed: 15.05.2021.

Kalpakjian, S. and Schmid, S. R. (2014) 'Rapid-prototyping Processes and Operations', in Deb, A. and Yap, E. (eds.) *Manufacturing engineering and technology*
Singapore: Pearson, pp. 1164.

Kauffer, P. H. (ed.) (2011) *Injection molding process, design, and applications*. New York: Nova Science Publishers, Inc.

'Liquid Injection molding', (2020) *The Dow Chemical Company*.

Liquid silicone rubber (LSR) (n.d.). Available at: https://www.simtec-silicone.com/capabilities/properties-of-lsr (Accessed: 02.04.2021.

Needle valve cold flow channel. n.d.

Nylon 12 (2021). Available at: https://en.wikipedia.org/wiki/Nylon_12 (Accessed: 04.03.2021.

Objectives (n.d.). Available at: https://www.impure-project.eu/objectives/ (Accessed: 15.05.2021.

Skiba, M. (2019) *How much does a plastic prototype cost?* Available at: https://www.engineeringresourcecenter.com/blog/how-much-should-a-plastic-prototype-cost (Accessed: 08.05.2021.

Thermal contact resistance (2016). Available at: https://help.solidworks.com/2016/english/SolidWorks/cworks/c_Thermal_Contact_Resistance.htm (Accessed: 14.05.2021.

Viscosity of silicone rubber (n.d.). Available at: https://www.simtec-silicone.com/viscosity-of-silicone-rubber/ (Accessed: 22.04.2021.

Vulcanization (2020). Available at: https://www.britannica.com/technology/vulcanization (Accessed: 10.05.2021.

What is silicone? (n.d.). Available at: https://www.cascobaymolding.com/silicone (Accessed: 16.05.2021.

Appendix

Appendix 1:

Medforfatterdeklarasjon

Tittel på oppgaven: Navn på oppgaven

Forfattere:
Student 1: Ole Kristian Elnæs
Student 2: Anders Brunsberg Mariendal

Bidrag:

Delaktiviteter/deloppgave	Student 1	Student 2
Research	3	3
Design	3	3
Simulering og analyse av design	3	3
3D printing og 3D scanning	3	3
Rapportskriving	3	3

Tabellen fylles ut med delaktiviteter, og størrelsen på bidraget fra den enkelte student angis med et tall mellom 0 og 4 etter følgende betydning:
0: Ingenting
1: Lite
2: En del
3: Mye
4: Alt

De undertegnede studenter bekrefter herved at de har utført delaktivitetene beskrevet ovenfor på gjeldende oppgave.

Signatur studenter:

Ole K. Elnæs

Anders B Mariendal

Ole Kristian Elnæs

Anders Brunsberg Mariendal

Appendix 2: Solidworks files

Appendix 3: Budget

NTNU

Kunnskap for en bedre verden