3D-Printed Injection Molds

David Helin & Kristoffer Palmér

DIVISION OF PRODUCT DEVELOPMENT | DEPARTMENT OF DESIGN SCIENCES
FACULTY OF ENGINEERING LTH | LUND UNIVERSITY
2020

MASTER THESIS

3D-Printed Injection Molds

David Helin and Kristoffer Palmér

LUND UNIVERSITY

3D-Printed Injection Molds

Published by
Department of Design Sciences
Faculty of Engineering LTH, Lund University
P.O. Box 118, SE-221 00 Lund, Sweden

Subject: Product Development (MMKM05)
Division: Product Development
Supervisor: Axel Nordin
Co-supervisor: Katarina Elner-Haglund
Examiner: Giorgos Nikoleris

Abstract

This master thesis describes the development process of a prototyping process. It was carried out in collaboration with Axis Communications.

When creating plastic prototypes, development engineers usually have two options, 3D-printing or injection molding using a soft tool. 3D-printing allows for fast and relatively cheap creation of prototypes which contributes to a fast product development process, but the physical properties are not the same as in the final product. Injection molding using a soft tool creates prototypes of high quality with physical properties equal to the final product, but it is time consuming and costly.

The purpose of the project was to investigate the possibility of 3D-printing injection molds for prototyping and to create a process where the method is used. This would combine the two prototyping options into a process that would be situated between them, allowing faster prototyping than injection molding using a soft tool while also creating better quality prototypes than 3D-printing.

Through external and internal research, including interviews and physical tests with an injection molder, a greater understanding of the subject was achieved and experience regarding the problems related to the process were identified. The information gathered was then turned into target specifications.

To reach the target specifications multiple design concepts and mold materials were tested before a final selection was conducted. One mold material and eight design concepts were selected for the process. The design concepts solve different problems which can not be solved directly by the selection of mold material.

The results are summarized as an instruction of how to use the process and a recommendation of how to further develop it.

Key words: Product development, Prototyping, Additive manufacturing, Injection molding, 3D-Printing

Sammanfattning

Detta examensarbete beskriver utvecklingen av en prototypframtagningsprocess. Arbetet är utfört i samarbete med Axis Communications.

När ingenjörer ska ta fram prototyper har de vanligtvis två alternativ. Antingen används 3D-printer eller formsprutning med ett soft tool. 3D-printning skapar prototyper snabbt och relativt billigt, vilket bidrar till en snabb produktutvecklingsprocess. Prototyperna har dock inte samma fysiska egenskaper som den slutgiltiga produkten. Formsprutning med ett soft tool skapar prototyper av hög kvalitet och med samma fysiska egenskaper som den slutgiltiga produkten. Det är dock dyrt och tidskrävande.

Syftet med detta projekt var att undersöka möjligheten att 3D-printa formsprutningsverktyg och att skapa en process som använder denna metod. Detta skulle kombinera de två alternativen för prototypframtagning till en process som är belägen mellan dem, vilket leder till snabbare prototypframtagning än formsprutning med soft tool samtidigt som prototyperna får högre kvalitet än med 3D-printning.

Genom extern och intern undersökning, med intervjuer och praktiska tester med en formspruta, skaffades en större förståelse för ämnet och problem relaterade till processen identifierades. Denna information användes sedan för att skapa en målspecifikation.

För att uppnå målen testades flera designkoncept och formverktygsmaterial innan de slutgiltiga valen gjordes. Ett formverktygsmaterial och åtta designkoncept valdes till processen. Designkoncepten löser problem som inte kan lösas enbart genom val av formverktygsmaterial.

Resultaten är summerade som en instruktion för hur processen används och en rekommendation för hur processen kan utvecklas i framtiden.

Nyckelord: Produktutveckling, Prototypframtagning, Additiv tillverkning, Formsprutning, 3D-Printning

Acknowledgements

Through this thesis work many people have contributed and helped in different ways. Without their help this project would not have been possible.

We want to thank our supervisors at Axis Communications, Morgan Johansson and Isabelle Wahlström, who have supported us and helped us with everything from technical ideas to logistical problems during the project. We would also like to thank Magnus Lundegård for welcoming us to Axis and for creating this interesting project.

We would also like to thank our supervisor, Axel Nordin, and our co-supervisor, Katarina Elner-Haglund, at The Faculty of Engineering at Lund University. Both have helped us greatly by steering us in the right direction during the project through discussions and ideas.

Lastly, we would like to thank Olaf Diegel for his help with the injection molding machine and Jonny Nyman at Ingvar Kamprad Design Centrum for his help with making aluminum molds and solving other workshop related problems.

Lund, January 2020
David Helin & Kristoffer Palmér

Table of Contents

List of acronyms and abbreviations

ASA	- Acrylonitrile styrene acrylate
ABS	- Acrylonitrile butadiene styrene
Axis	- Axis communications
CAD	- Computer-aided Design
GF	- Glass Fibre
HDPE	- High density Polyethylene
Injection molding material	- Material injected into the mold during injection
LDPE	- Low density Polyethylene
LTH	- Lunds Tekniska Högskola
MJP	- Material jetting printing
Mold material	- Material the injection mold is made of
PA	- Polyamide
PBT	- Polybutylene terephthalate
PC	- Polycarbonate
PE	- Polyethylene
PET	- Polyethylene terephthalate
PRIM	- Printed Injection Molding
PS	- Polystyrene
TPE	- Thermoplastic Elastomer
TPU	- Thermoplastic polyurethane

1 Introduction

In the following chapter the master thesis and the company where it was carried out are introduced. The goals of the project are defined, the delimitations are addressed and the report's disposition is presented.

1.1 Axis Communications

This master thesis project was performed in collaboration with Axis Communications, hereafter referred to as Axis, in Lund. Axis is a technology company which was founded in Lund in 1984 and have since developed a wide range of products. Early on, Axis made protocol converters for IBM systems and networking technology for printers, before moving on to networking cameras leading to the CCTV industry going from analog cameras to digital. Cameras are still the main business for Axis, but new innovation is still very important and they are always looking for business opportunities in new markets.

1.2 Initial information

This project was created with the purpose of widening the options for prototyping methods. The mechanical design departments at Axis creates many physical prototypes for testing parts and concepts and the main manufacturing method for these prototypes is 3D-printing. Axis has access to different kinds of 3D-printing techniques, which is good since different parts have different needs and demands. The main drawback using these printers is that none of them allows the creation of parts that has the same properties as the end product, which is made with injection molding.

The main difference of the 3D-printed parts and the injection molded parts is that the 3D-printed parts are made up of many layers of material being added onto each other, in contrast with injection molding where the whole part instead is molded as one solid. This creates different material properties between the 3D-printed parts and the injection molded parts which leads to the design teams at Axis not being able to fully test the prototypes as intended.

Making a metal mold for injection molding is very time-consuming and therefore not suitable for a fast paced product development project at Axis where the engineers want to have quick access to good prototypes. This creates the need for a prototyping process which allows for faster manufacturing than injection molding, but with more accurate quality than that of 3D-printing.

1.3 Objectives

This thesis aims to investigate the possibility of 3D-printing injection molds and to create a process in which this concept is used as an alternative to 3D-printing when prototyping at Axis. The goal was to solve problems occurring specifically during injection molding with 3D-printed injection molds and to find viable materials.

The result is going to include the verdict of how viable it is to 3D-print injection molds for prototyping, the machines that are needed, the materials which can be used, the potential drawbacks and limits of the process, an instruction of how the process can be used and a summary of everything that was deemed relevant for further research or development.

1.4 Delimitations

From the start of the project it was decided that the focus would be towards injection molding and no other molding techniques. This decision was based on Axis using injection molding as their main manufacturing method for plastic parts.

The testing phase is a big part of the project and the time frame set would not be enough if every 3D-print material were to be tested. Therefore some materials and 3D-printing techniques have been prioritized over others based on availability and estimated potential. Everything of interest that has not been tested because of this prioritization has been summarized as a recommendation for future testing and research.

1.5 Disposition

The project was approached as a product development project and in this report the whole process is documented from start to finish. In the second chapter, named "Methodology", the method behind the process is described and the initial time plan is presented. In the third chapter, named "Theory", the theoretical background needed to understand the project is presented and described. The fourth chapter, named "Research", establishes all the project specific information, which was gathered through interviews with employees at Axis and with external prototype manufacturers. This information was then used as the basis for the customer needs in the fifth chapter named "Target specification". The customer needs were then turned into target specifications with measurable units.

In the sixth chapter, named "Concept Generation and Selection", the process of solving the problems stated in the target specification is started. The concept generation began by dividing the problems into two different processes, the material concept process and the design concept process. This split was made because of the different nature of these two problems, where the material concept process will focus on finding relevant materials and testing them and the design process will create solutions to problems that can not be solved by the choice of material. In the material concept process all available materials relevant for the creation of molds were obtained and moved on into testing. In the design concept process, all the problems not solved by the material selection were listed and concepts for solutions were generated. These concepts were scored against each other and the highest scoring concepts were selected and moved on into testing.

The seventh chapter, named "Concept Tests", shows the results from the tests performed. The concepts were evaluated based on these results and underwent a final selection. In the eighth chapter, named "Discussion and Recommendation", the results are discussed and then presented as a recommendation of how Axis can use them and what to focus on for further development.

2 Methodology

In the following chapter the development methods used in the project and the reasoning for choosing them are described. The initial time plan created at the start of the project is also presented.

2.1 Time Plan

The project was conducted during a period of 20 weeks, starting in September 2019. During the first week the project was divided into different activities which were distributed over the 20 week time frame. A Gantt chart was created to visualize this distribution. During the project the activities' real time consumption did not always match the planned time. A second Gantt chart was therefore created to show the actual time spent for each activity. Both time plans can be found in Appendix A.

2.2 Development method

The main development method used during the project was a slightly modified version of the *Double Diamond Design Process*, created by Design Council (2007). Within the double diamond frame set other development methods were applied to optimize the process for the project's needs. These other methods were inspired by methods presented by Ulrich and Eppinger (2012) and Ullman (1997).

2.2.1 Double Diamond

The *Double Diamond Design Process* contains four steps: *Discover, Define, Develop* and *Deliver.* In the *Discover* half of the first diamond a problem, or multiple problems, are explored and identified in order to gain a better understanding. These problems are then analyzed and structured in the *Define* half. When leaving the first diamond and entering the second diamond a clear picture of the problems and objectives have been made. In the *Develop* half of the second diamond the solutions to the problems are created and tested, before being finalized and evaluated in the *Deliver* half. In Figure 2.1 an illustration of the *Double Diamond Design Process* is shown with the key activities of this project placed inside the corresponding diamond half.

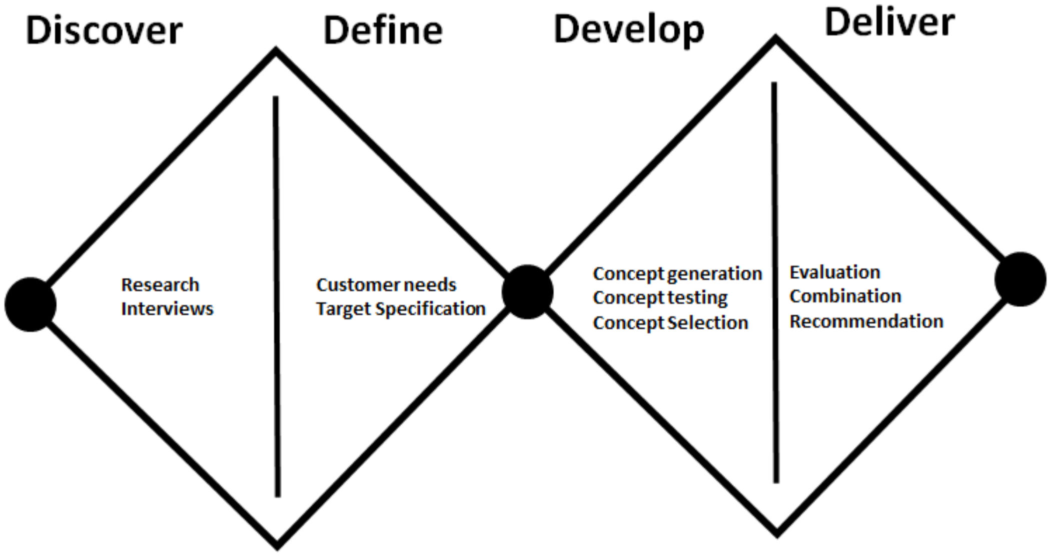


Figure 2.1 The Double Diamond Design Process including key activities.

2.2.2 Ulrich and Eppinger

During the *Define* phase methods presented by Ulrich and Eppinger (2012) were used to establish the customer needs and target specifications. The methods describe how to list customer needs and how to organize them based on their importance for the objectives. It then goes on to show how to convert these customer needs into target specifications which are presented in a way that makes them measurable and therefore easier to interpret while conducting tests.

2.2.3 Ullman

In *The Mechanical Design Process* (Ullman 1997) describes how a product development team can generate concepts by taking turns in adding ideas to each other's concepts. This method was chosen because of its strength in making developers see solutions outside their own perspective and it was used during the *Develop* phase.

3 Theory

In the following chapter the main theoretical principles of this thesis are presented. This includes theory regarding polymers, metals, additive manufacturing and injection molding. These are all important topics in order to gain an understanding of the project.

3.1 Material

3.1.1 Polymer

Polymers are a wide group of chemical compounds that are built up of long chains of smaller subunits as seen in Figure 3.1. Polymers are generally synthetically produced, but also exists in natural sources, for example in the rubber tree. The application areas are wide and they are used in almost every production industry. The plastic group is divided into thermosets and thermoplastics, illustrated in Figure 3.2 (Bruder, 2014).

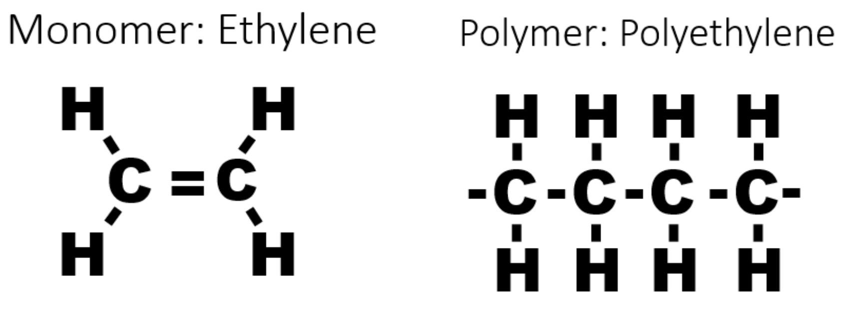


Figure 3.1. Difference between the monomer ethylene and the polymer polyethylene.

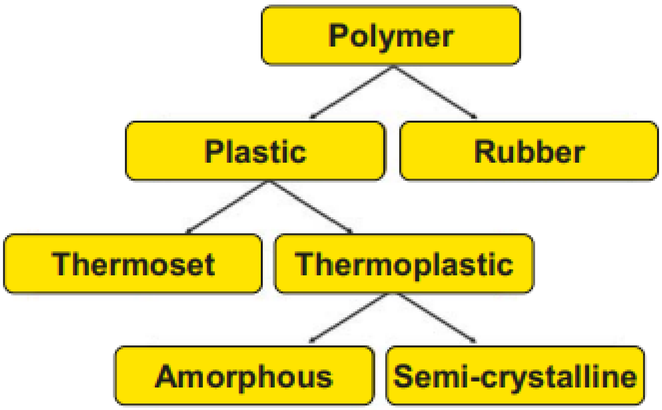


Figure 3.2 Polymer division tree, courtesy of Ulf Bruder (2014)

3.1.1.1 Thermoplastic

Thermoplastics are plastic materials that melt when affected by heat. This gives them the advantage that they can easily be mass produced using techniques like injection molding, vacuum molding, blow molding, rotational molding and extrusion.

Thermoplastics consists of two subgroups, semi-crystalline and amorphous plastics. Semi-crystalline materials have their atoms positioned in a structured pattern that gives the material a set melting point as seen in Figure 3.3. Above this point the plastic will change from solid to liquid. Amorphous materials instead have their atoms positioned more randomly without any specific order, which results in the material having a glass transition temperature instead of a melting point. At the glass transition temperature, the material will begin to change from a solid form into a more viscous state with the increasing temperature (Bruder, 2014).

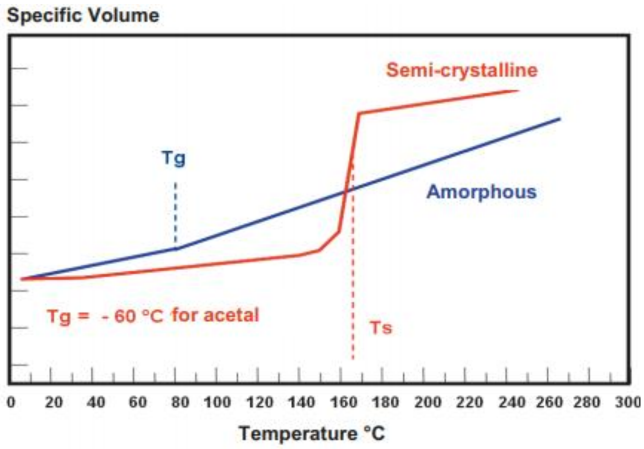


Figure 3.3 The difference between semi crystalline plastics melting point and amorphous plastics glass transition temperature, courtesy of Ulf Bruder (2004)

3.1.1.2 Thermoset Plastic

Thermosets plastics are solid regardless of temperature. This is because of the cross-linked polymer chains that form during the curing process when the plastic is created. This curing can come from different sources such as heat and UV light (IUPAC, 2004). After curing, the cross-linking chains do not break when exposed to heat and thermosets can therefore not be melted. This often gives them a higher working temperature but also limits their ability to be recycled. (Bruder, 2014).

3.2. Additive manufacturing

Additive manufacturing, also called 3D-Printing, is the method of creating physical objects by adding material in a layer wise fashion. There are several advantages to using 3D-Printing over conventional manufacturing methods like injection molding and milling, illustrated in Figure 3.4. These advantages are however often limited to small batch production, prototyping and manufacturing with unique or complex designs.

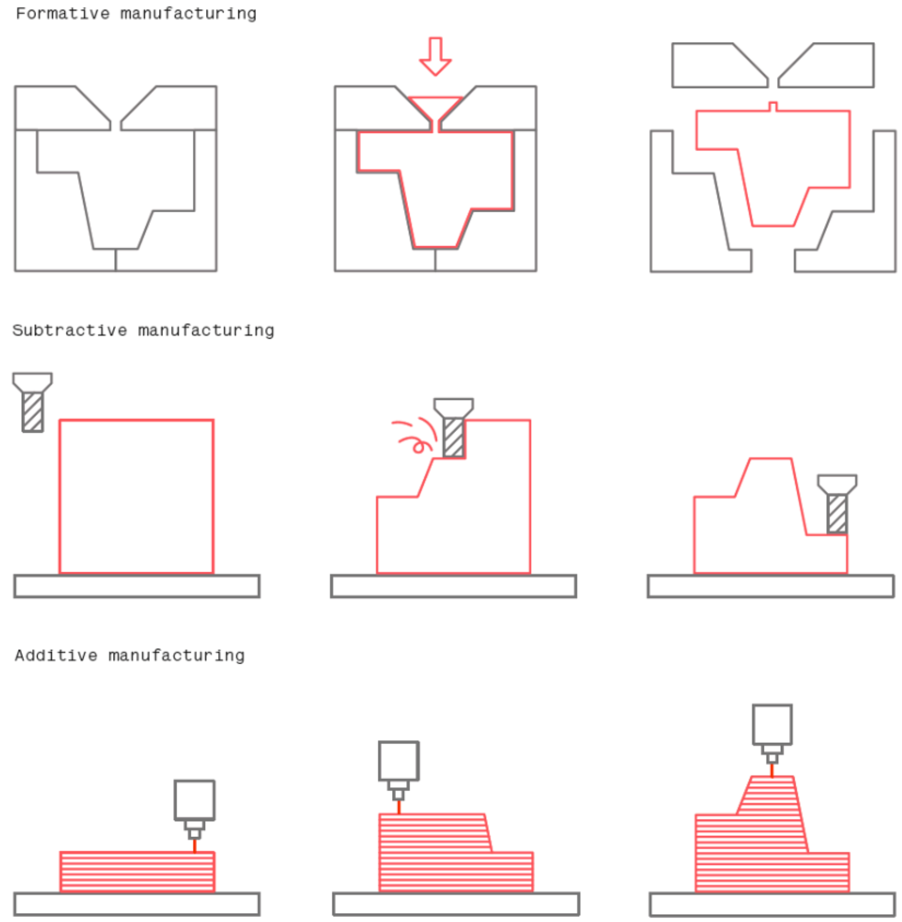


Figure 3.4 Difference between formative, subtractive and additive manufacturing (Redwood, Schöffer, and Garret 2018)

Additive manufacturing has changed how prototyping is done, but even though the 3D-printed parts might look the same as a finished injection molded part they behave differently and have other mechanical and thermal properties. The reason for this is partly because most additive manufacturing methods often use different materials compared to injection molding but also because of the additive manufactured parts' layer on layer design. This feature causes the object to act anisotropic, that is to say, have different properties in different directions. These layers also contribute to the part being weak when forces are applied in certain directions as seen in Figure 3.5 (Diegel 2020).

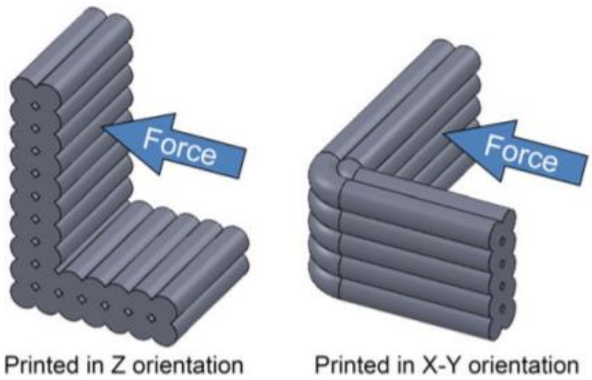


Figure 3.5 Printing orientations affect on additive manufactured parts (Diegel, Nordin and Motte 2020).

There are several different methods regarding additive manufacturing were the most commonly used are Fused Filament Fabrication (FFF), Stereolithography (SLA), Direct Metal Laser Sintering (DMLS), Material Jetting (MJ) and Binder Jetting.

Since the technology is under rapid development and is driven primarily by companies, the names of machines and materials used are because of immaterial property rights often trademarked or made exclusively to define a particular machine. This makes the naming convention somewhat confusing and similar technologies or generic names have therefore been grouped together as best as possible in the summary below.

3.2.1 Fused Filament Fabrication (FFF)

When people nowadays think about 3d printing they often think of FFF, which is the additive manufacturing method created by Stratasys under the trademarked name FDM (Fused Deposition Modeling) in the 80s. The method works by using a spool of material which is then heated up and deposited in 2D-layers on a prewarmed surface in order not to warp the design. The print head starts by making an initial layer and then continues to the next layer height as can be seen in Figure 3.6.

Since the size of the nozzle also is linked to the step size, this method often show visible layers as seen in Figure 3.7. This can be adjusted by using a finer nozzle and thus lower the layer height but also increases the print time threefold. The materials used in FFF is by a large extent confined to thermoplastics like ABS, PC, PLA but have also had some success using metal. (Redwood, Schöffer and Garret 2018)

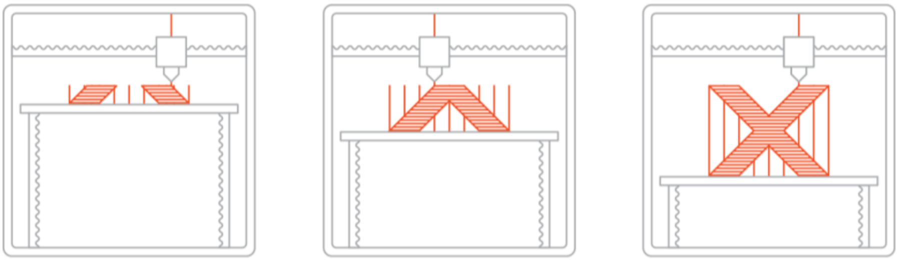


Figure 3.6 steps of FFF printing (Redwood, Schöffer, and Garret 2018).

Figure 3.7 Visible layer height of an FFF printed part (Redwood, Schöffer, and Garret 2018).

3.2.2 Stereolithography (SLA)

Stereolithography, like FFF, was also discovered and patented during the 80s, but in contrary to FFF, SLA uses a bath of transparent UV sensitive resin which is exposed to a light source that hardens the top layer of the resin turning it into a solid thermoset. The plate on which the thermoset hardened is then lowered or elevated and the next layer becomes exposed to the light source and hardened onto the previous layer as can be seen in Figure 3.8. (Redwood, Schöffer and Garret, 2018)

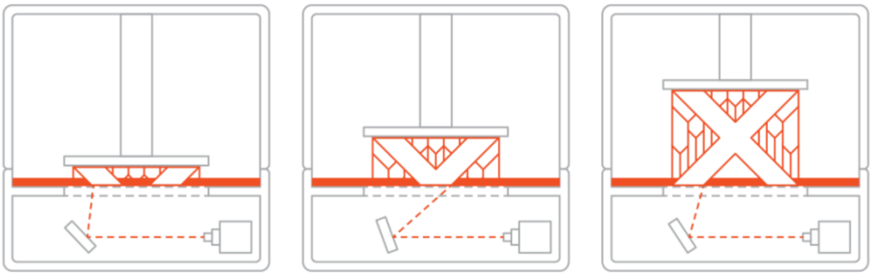
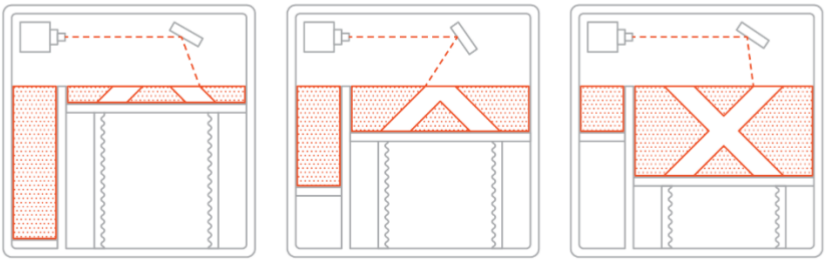


Figure 3.8 Steps for printing SLA (Redwood, Schöffer, and Garret 2018).

3.2.3 Selective Laser Sintering (SLS)

SLS is an additive manufacturing method using a powder bed of material that is fused together by a laser beam. When a layer is finished a new coat of powder is applied which then in turn is sintered as seen in Figure 3.9. (Redwood,Schöffer and Garret, 2018)
After the detail is finished the method requires the part to cool down and be dug up from the rest of the powder. SLS primarily uses thermoplastics with a low thermal conductivity such as Polyamide.

Figure 3.9 Steps for printing SLS. Courtesy of 3dHubs, The 3dprinting handbook.



3.2.4 Material Jetting (MJ)

Material jetting, which is similar to SLA in that they both use UV sensitive Thermoset Plastics, uses a printing bed on which small drops of resin are placed by a printing head. The resin is then cured by a passing UV-light. The head is then lowered and the next layer can be printed. The process, illustrated in Figure 3.10, is carried out until the detail is finished. (Redwood, Schöffer and Garret, 2018)

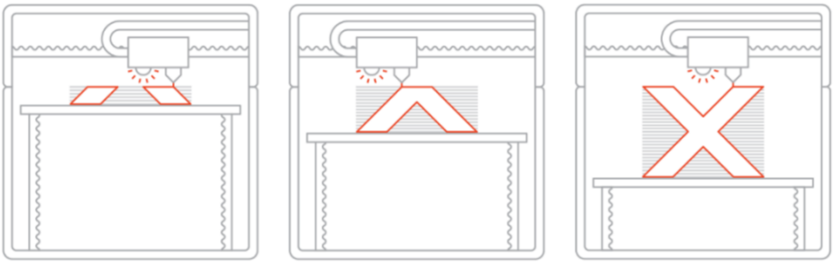


Figure 3.10 Steps for printing MJP (Redwood, Schöffer, and Garret 2018).

3.2.6 Direct Metal Laser Sintering (DMLS)

DMLS, like SLS, uses a powerful laser to fuse a powder bed of fine material together, illustrated in Figure 3.11. Unlike SLS however, DMLS uses a metal powder instead of a plastic powder. Since metal fusion creates a lot of internal forces and heat, rigid supports are needed to prevent the part from warping and to conduct heat away. This makes the post process of removing the metal support a work intensive part of DMLS. (Redwood, Schöffer and Garret, 2018)

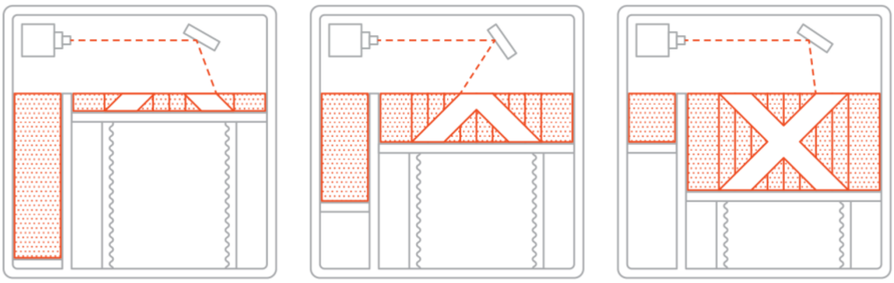


Figure 3.11 Steps for printing using DMLS. Courtesy of 3dHubs, The 3dprinting handbook.

3.2.7 Table of methods

A summarization of the different 3D-printing techniques

Table 3.1 The different printing methods and their strengths and weaknesses. (Redwood, Schöffer and Garret, 2018)

Printing Method	***Adhesiv***	***Material***	***Characteristics***
Fused Filament Fabrication (FFF)	Heat	Thermoplastics ABS PLA PETG PC PEEK	+ Cheap + Fast + Lots of materials - Low quality surface finish - Warping -Weak adhesion between layers
Stereolitografi (SLA)	UV-light	Thermoset Plastics	+ Excellent surface finish + excellent dimensional accuracy - Expensive materials - Hazardous resins - Lots of post process - Limited lifetime
Selective Laser Sintering (SLS)	Heat	Thermoplastics PA Alumide PEEK	+ Strong parts + Good dimensional accuracy - Expensive - Post processing
Material Jetting (MJ)	UV-light	Thermoset Plastic	+ very fine surface finish + Creates a near homogeneous part + Great Dimensional accuracy - Expensive - Brittle - Low heat deflection
Direct Metal Laser Sintering (DMLS)	Heat	Metal Aluminium Steel	+ Strong functional metal parts -Expensive -Lot of post process -Use of support structures as heat sinks

3.3 Injection Molding

Injection molding is a method of manufacturing products in different plastic materials, both thermosets and thermoplastics. It is a very effective manufacturing method when producing large quantities and is therefore commonly used in mass production of plastic details. The main disadvantage of injection molding is the cost of the equipment which makes it very hard to justify smaller batches. The injection mold is a big factor in this high cost since it has to be individually made for every new design (Bruder 2014).

3.3.1 Injection mold

The injection mold consists of two halves called the fixed part and the movable part. Together the two halves create a cavity in the form of an inversion of the desired detail. The fixed part is mounted to the nozzle (the channel in which the molten plastic is injected through to reach the mold) and can not be moved. In Figure 3.12-14 it is illustrated as the red half of the mold. The movable part is mounted to rails connected to the machine. It can move along these rails to dock with the fixed part creating a closed mold. This movement is powered by a hydraulic or pneumatic cylinder. In Figure 3.14-3,14, 2 and 3 the movable part is illustrated as the green half of the mold.

Molds for large production quantities are made from steel since they need to be as strong and durable as possible. For smaller production quantities molds can be made from aluminum, since it is durable enough for smaller quantities and much cheaper and easier to manufacture the mold from. These molds are known as “soft tools”

To avoid air getting trapped inside the mold a venting system can be designed. This lets air escape from the cavity in order to avoid diesel marks, shown in Figure 3.15, and not reaching a hundred percent degree of filling. A venting system consists of very thin channels strategically placed inside the cavity. The channels are thin enough to let air escape out, but not the molten plastic being injected into the mold.

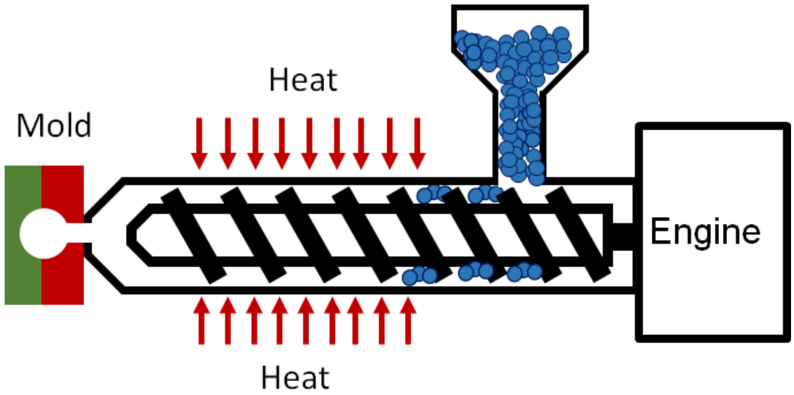


Figure 3.12 First step of the molding process.

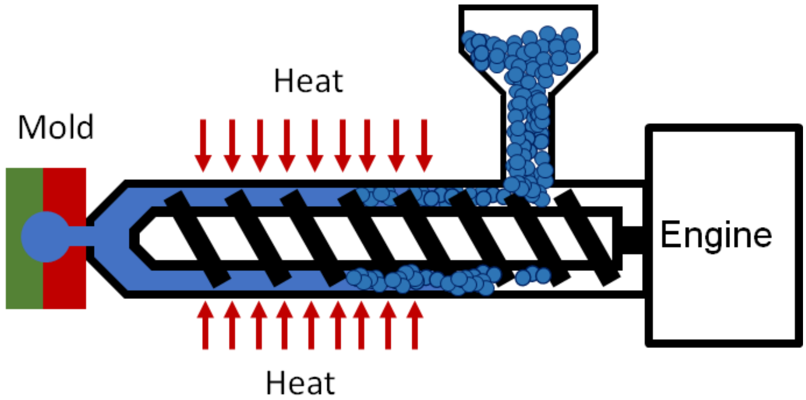


Figure 3.13 The second step of the molding process.

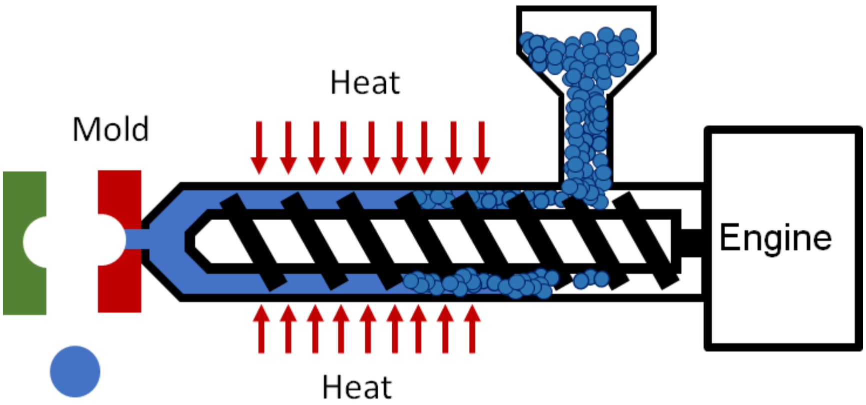


Figure 3.14: The last step of the molding process.

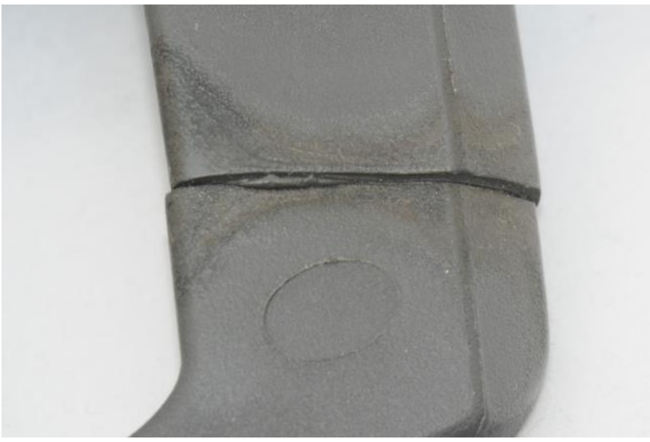


Figure 3.15 Diesel effect.

3.3.2 Molding process

The process starts with the two injection mold halves closing. It is very important that they are sealed completely shut. In the next step molten plastic will be injected through the gate of the mold, creating a pressure inside. The closing force applied on the movable mold half must therefore be able to keep it closed to avoid any leakage or premature mold opening.

The granulate is fed from the hopper into the cylinder where it is melted by heating elements located around the cylinder and by the heat generated by friction from the screw. When the screw rotates the now melted granulate is moved forward in the cylinder. When the correct preset volume of molten plastic is in front of the screw the rotation stops and the screw is moved forward pressing the molten plastic through the nozzle into the injection mold.

When the mold is completely filled by the molten plastic a cooling period starts. During this cooling period the screw must continue to apply pressure towards the mold to ensure that the detail inside does not shrink as it hardens.

When the preset time of the cooling period ends the mold opens and the solid plastic detail, now attached to the moving part of the mold, is ejected. The ejection can be done by ejector pins pushing the object out or by a robot arm pulling the details out. In some cases the details have unwanted plastic trimmings attached to it, created by the channels in the mold leading up to the cavity. These plastic trimmings are called the sprue.

3.3.3 Release agents

For easier ejection of the plastic object from the mold a release agent can be used. There are two different kinds of release agents. The first one is applied directly onto the surface of the mold. This method will increase the cycle time and therefore cost since the release agent has to be reapplied between shots. The second kind is applied onto the granulate. This method will lower the production cost since it lowers the cycle time. It is therefore the most commonly used method (Fink 2013).

3.3.4 Drying of Granulate

Since most plastics like PA, PBT, PC and ABS are hydroscopic, meaning they attract water, a drying process is in most cases essential in order to injection mold. The exceptions are non-hygroscopic plastic like polyethylene and polypropylene. The reason behind this is that hygroscopic plastic molecules have a polarity and therefore attract the polar water molecules. But even though non hygroscopic plastics do not attract water a drying process is often preferred because of surface moisture that can affect the quality of the part.
(Strömvall & Lundh 2019)

3.3.5 MCP Minimolder 12/90 HSP

The injection molder used in this project is named MCP Minimolder 12/90 HSP, a smaller sized pneumatic benchtop injection molder. It is horizontally clamping with a clamping force of 14 tons. The machine has two plastic covers which functions as safety guards. The non-transparent safety guard covers the granulate barrel, the screw, the plunger and the pneumatic powered knee-joint, shown in Figure 3.18, which controls the mold closing mechanism. The transparent safety guard, shown in Figure 3.16, covers both the fixed part and the movable part of the injection mold, shown in Figure 3.17.

The mold is installed into a bracket which limits the size regarding the mold size, shown in Figure 3.19. Mounted to the movable part are the pneumatic powered ejector pins. The machine is operated by a control unit using a touch screen. Cooling of the machine uses water and requires a water flow of 2 liters per minute.

Figure 3.16 The MCP Minimolder with the transparent cover removed.

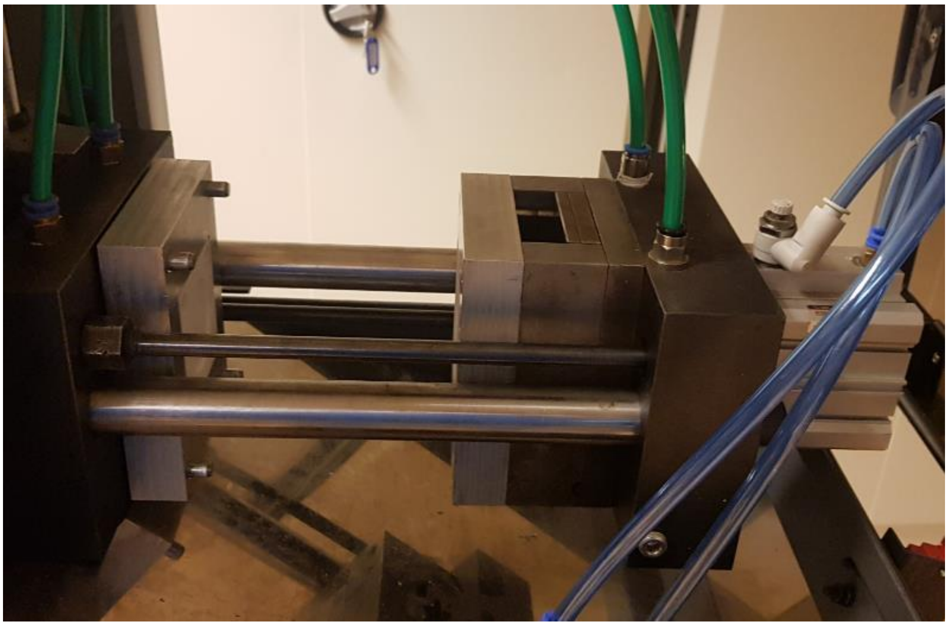


Figure 3.17 Close up of the injection mold with the transparent cover removed.

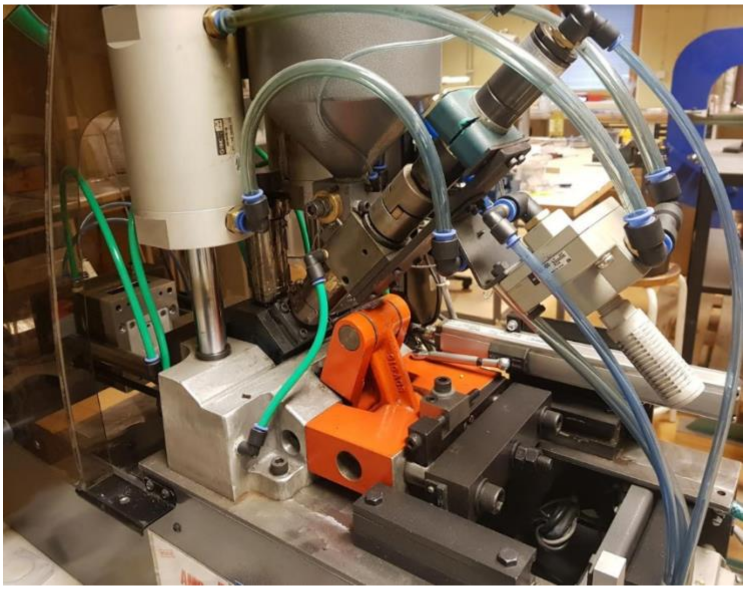


Figure 3.18 Close up of the granulate barrel, the screw, the plunger and the pneumatic powered knee joint.

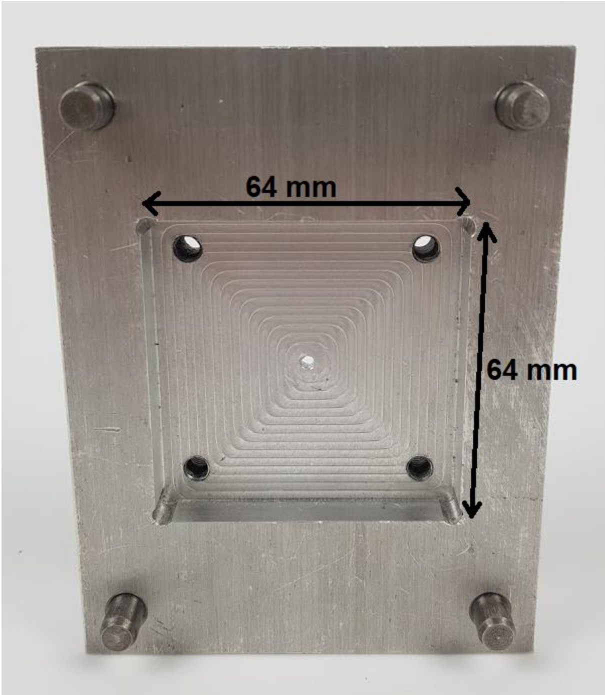


Figure 3.19 Size limit for the injection mold caused by the installation bracket.

4 Research

In the following chapter the research conducted during the project will be presented. This will include information about how prototyping is carried out at Axis, how other external sources have used 3D-printed injection molds, how the initial test runs of the injection molder were set up and the selection of the parts later used in the project.

4.1 Interviews at Axis

Twelve interviews were carried out at Axis as research for the target specification. The interviews consisted of eight questions and were conducted with employees at the Axis headquarters in Lund, all working in the fixed dome department with mechanical construction and design. The results from the interviews were used as a basis for the target specification. The questions asked during the interviews spawned from discussion with the supervisor at Axis. All questions asked during the interviews can be seen in Appendix B.

4.1.1 Prototyping at Axis

Axis as a company have access to different methods in order to create prototypes. These methods include smaller FFF 3d printing machines, a MJ machine, a SLS machine and a DMLS machine. The vast majority of all prototypes in the fixed dome department are created using these machines. Axis also have some smaller workshops for making parts and prototypes.

In addition to the internal sources Axis also use external prototype makers to create prototypes and soft tools.

4.1.2 Benefit of injection molding prototypes

According to the employees injection molded prototypes would be beneficial to use in tests when the physical properties of the end part was to be evaluated, such as impact and temperature tests. At the moment they have a hard time doing these test because 3d printed parts are often both made in the wrong material but also function differently because of the layered structure explained in Chapter 3.2 *Additive manufacturing*.

The employees also said that they have a hard time testing mechanical fixtures that are dependent on the mechanical properties of the material, such as snap-fits. The reason for this is that the mechanical properties of snap-fits are dependent on the stiffness of the material. Another possible aspect the design engineers saw using a fast phase injection molding process was to replace/minimize the use of soft tooling, which at the moment takes around four to five weeks to get delivered.

4.1.3 Size of products

The developers at Fixed Dome Mechanics were shown three different sized squares of 6x6 cm, 10x10 cm and 15x15 cm. The developers then decided how many percent of their prototyped parts would fit in the different sized squares. This resulted in the conclusion that the second square, 10x10 cm was the most common size for prototypes and the first and third square being equally common in second place.

4.1.4 Materials

The most common plastic used in the end products by the design engineers were PC, PC-PBT, PC-GF, PC-ABS and PA. Among these PC-ABS has the highest viscosity and is therefore easier to injection mold. PC-ABS was therefore set as the target objective regarding material to be injection molded. But even though PC-ABS is easier to injection mold than other plastics such as PC, is it still a demanding material that requires high temperatures.

4.1.5 Time consumption

The design engineers had some different opinions regarding how much time they thought was acceptable for creating injection molded prototypes. The answers differed between 3 days up to 4 weeks, where the most common answer was 2 weeks.

4.2 Interviews with prototype makers

Interviews with 7 different prototype makers and manufacturers of 3D-Printers were conducted. The questions focused on their experience with additive manufactured molds for injection molding. Full questions can be seen in Appendix C.

From the interviews it was concluded that most makers have had some experience or have tried 3D-printed injection molds with different results. Most makers had tried using SLS but were mostly disappointed with the surface finish and one maker complained about the temperature properties. This maker did however create 200 prototypes in TPE when using one SLS printed mold in polyamide. The maker said this was because TPE can be injection molded at low temperatures.

The second most tried method was MJ printing were the makers who had tried this method seemed to have had fairly successful results. The most common complaint regarding this method was the need of redesigning the details in order to give it sufficient draft angles to make it easier to injection mold. This was seen as a problem because a redesign of a prototype might render the parts usefulness if it differs to much from the original detail.

Among makers that had not tried the method the biggest concern to why they had not used the method was either their limited access to machines able to produce the prints and concerns regarding MJP's low temperature integrity.

One maker had tried using SLA with good results but had some issues regarding the draft angles. Three makers that had not tested this method did however suggest using a stiff high temperature SLA which they thought would be a good solution.

A lot of the makers had experience or had seen the use of metal printed injection molds, even though they were functional they all had complaints regarding surface finish and the amount of clean up and removal of material in order to render it useful compared to just milling it out the traditional way.

None of the makers had tried using FFF, where they all argued that the surface finish would be too rough.

One maker mentioned an interesting discovery they had regarding the cooling of molded parts. Because thermoset plastic works as an insulating material the injection molded parts cooled relatively slowly, something that might be a concern when mass producing parts, since this would dramatically increase cycle time. The maker argued that this is actually a benefit when making low run prototypes since the insulation effect of the 3D-Printed mold makes the plastic flow more easily without having problems solidify mid-way.

4.3 University of Zagreb

Interviews with additive manufacturing researchers and professors from the University of Zagreb were conducted. They had successfully been creating molds in Digital ABS (MJ) and then injection mold parts in ASA, TPU and PBT-GF to create small batches for the automotive and pharmaceutical industry. To make a thin and exposed part of the injection mold more durable, they used a metal pin inserted into the injection mold.

4.4 Existing Solutions

Information was gathered of already existing methods regarding the use of additive manufactured molds and similar areas.

4.4.1 Stratasys

4.4.1.1 Digital ABS

Digital ABS is a branded material used by Stratasys MJ machines. It is a thermoset plastic that is designed to act like ABS, even though it is not a thermoplastic like ABS. Digital ABS is a tough and temperature resistant material, which makes it interesting for high temperature injection molding. (Stratasys 2016). In addition to Stratasys, several other organizations like P3d prime and University of Zagreb have tested Digital-ABS to make injection molds. All three were able to produce up to 100 injection molded parts in easy to produce materials like TPE, PP and around 30 parts using more demanding plastic materials such as PA, PBT, and PC-ABS. (p3d-prim)

4.4.1.2 Stratasys' White Paper

Stratasys released a White Paper named "Technical Application Guide - Polyjet For Injection Molding" where they provide guidelines when designing 3d printing molds using Digital ABS. (Stratasys 2016)

From the White Paper Stratasys give advice on difficulties regarding different material and also made the following guidelines when designing 3d printed molds:

- *Increase draft angels up to 5 degrees.*
- *Avoid sharp corners*
- *Use high clamping force and fine adjusted filling parameters to avoid flashing.*
- *Holes should be larger than 0.8 mm*
- *Scale the model to compensate for shrinking*
- *Make gate larger, approximately 3-4 times then used for standard injection molds 5-8mm.*
- *Cooling won't affect cycle times since plastic materials are insulating, cooling will however increase mold life.*
- *Use silicone release spray to ease ejection*
- *Use inserts to increase venting and moldlife regarding challenging parts.*

4.4.2 3D Systems

4.4.2.1 Visijet M3-X

Visijet M3-X is a thermoset plastic made for use with a MJ printer. It is marketed as an ABS like material but is instead made for 3D systems' Projet machines. It is like Digital ABS strong with a high heat deflection temperature. Companies like Bi-Link have tried using it for injection molding small batches in several different materials including PC with successful results (3DSystems)

4.3.3 Formlabs

4.3.3.1 Formlabs' White Paper

Formlabs released a White Paper on how to make injection molds from their SLA machines using thermoset plastics. Their methods include using their high temp resin which they tried with 8 different materials, including PS, LDPE, ABS and HDPE (Formlabs 2016).

The White Paper stated the importance of using plenty of release agent in the mold when using a stiff mold and a stiff material such as PS. It was also mentioned that introducing thin channels for the air to leak out helps with evicting trapped air that might otherwise cause damage and result in molds not reaching 100 percent degree of filling. Other tips presented included using a high clamping force to reduce flashing, apply one to three degree draft angles for easier release and make the forms a bit thicker so they are slightly protruding from the metal frame to make a complete seal, illustrated in Figure 4.1.

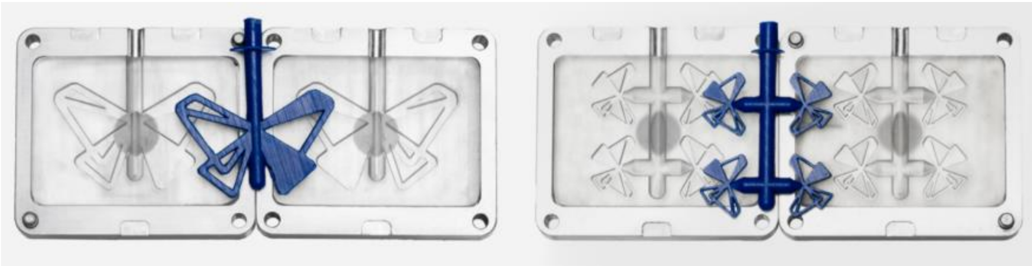


Figure 4.1 3D-Printed injection molds in a clear thermoset plastic surrounded by a metal frame, (Formlabs 2016))

4.3.4 3DHubs

3Dhubs did comparison tests regarding injection molding between Digital ABS and an SLA material called Somos PerForm. From the test they concluded that both materials have potential to be used for injection molding. They did however argue that Somos PerForm result in both better quality and is more useful regarding more demanding injection molding materials because its higher tensile strength and heat deflection temperature (Varotsis).

4.4 Initial Test Runs of the MCP Minimolder

Since the authors only had theoretical experience of injection molding and since no person with experience handling the MCP Minimolder was found, time was allocated early on in the project towards gaining experience, knowledge and control of the machine. The machine was installed in a workshop at LTH, where water and pneumatic power sources were accessible. An injection mold with a simple round and flat cavity was made from aluminum, with the purpose of having a testing mold to fit into the machine as can be seen in Figure 4.2.

Figure 4.2 Aluminum testing mold mounted in the MCP Minimolder.

Several test runs were conducted with the aluminum test mold until the injection molding process was considered fully researched and controlled. The aluminum mold was then switched to several 3D-printed plastic molds which went through some iterations until a base mold shape was created to meet the machines demands. The first 3D-printed plastic mold was considered hard to remove from the machine, which made the injection molding process slow. To improve this, cutouts were made on two sides of the mold, which enabled easy removal using a screwdriver. This change can be seen in Figure 4.3

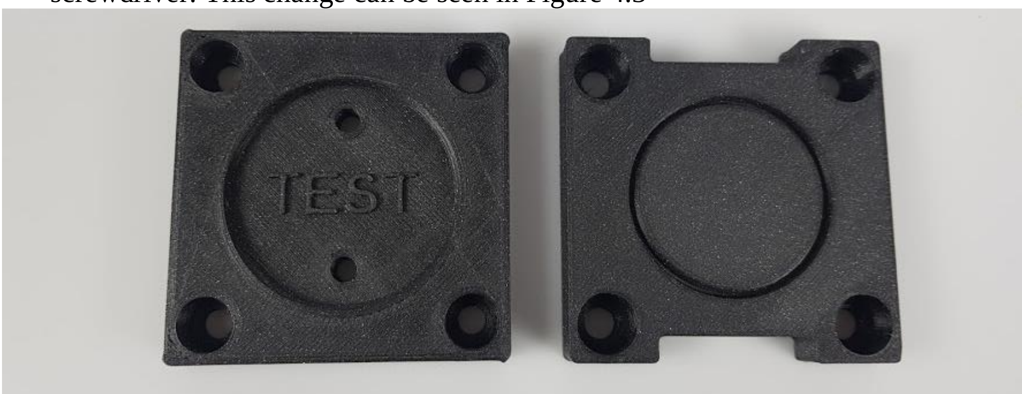


Figure 4.3 Old mold shape without cutouts to the left and new mold shape with cutouts to the right.

The machine has two ejector pins. These did not work as intended during the testing and it was therefore decided that further molds would not be using them. This was solved by designing 7 mm deep holes on the back of the mold in which the ejector pins could fit, but since the holes do not go all the way through the ejector pins will be completely hidden. This change can be seen in Figure 4.4.

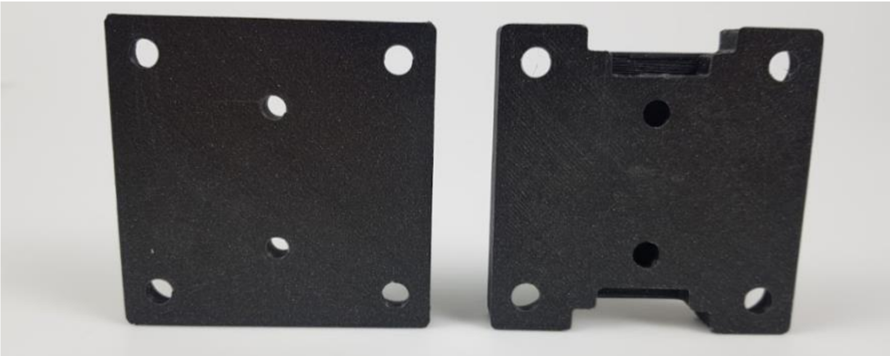


Figure 4.4 Old mold shape with ejector holes going all the way through to the left and new mold shape with ejector holes stopping after 7 mm.

During the test runs some apparent differences between injection molding in metal and plastic were found, the most apparent was that the durability of plastic is significantly lower, especially regarding heat. Plastic in contrary to metal, also works as an insulator and does not transfer away the heat from the melt. The details were therefore often in a molten state when opening mold as shown in Figure 4.5.

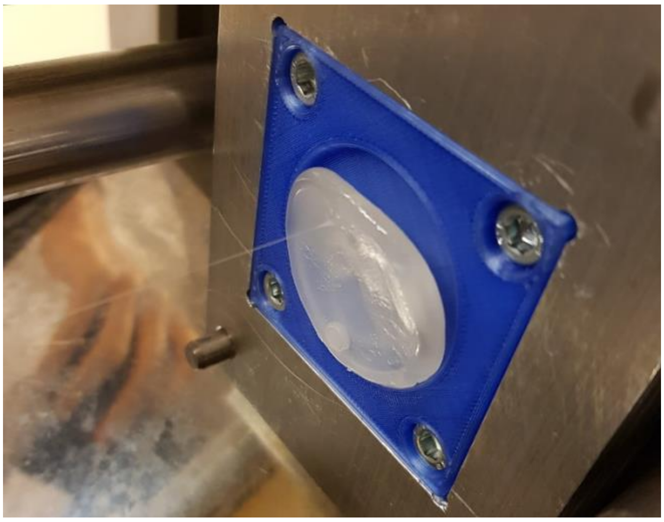


Figure 4.5 Detail in molten state after injection, here using an ABS mold.

4.5 Part selection

For the testing, developing and evaluating of different development concepts a suitable part had to be selected. The part had to fit within the injection molding machine's size limitation, shown in Chapter 3.3.5 *MCP Minimolder 12/90 HSP*, it also had to be relevant for the features that were to be tested and evaluated. From interviews and initial test runs the two following parts, shown in Figure 4.6-7, were selected. They were chosen because they both had snap fittings and shapes that were deemed interesting from a prototyping perspective, as their geometry has some complexity but not enough to make the mold design process too advanced. Their degree of geometrical complexity is also different from each other, with the *Snap-hitch* having a simpler design than the *Cable-lid*, which allows for a deeper understanding of the limits of the rapid prototyping injection molding process.

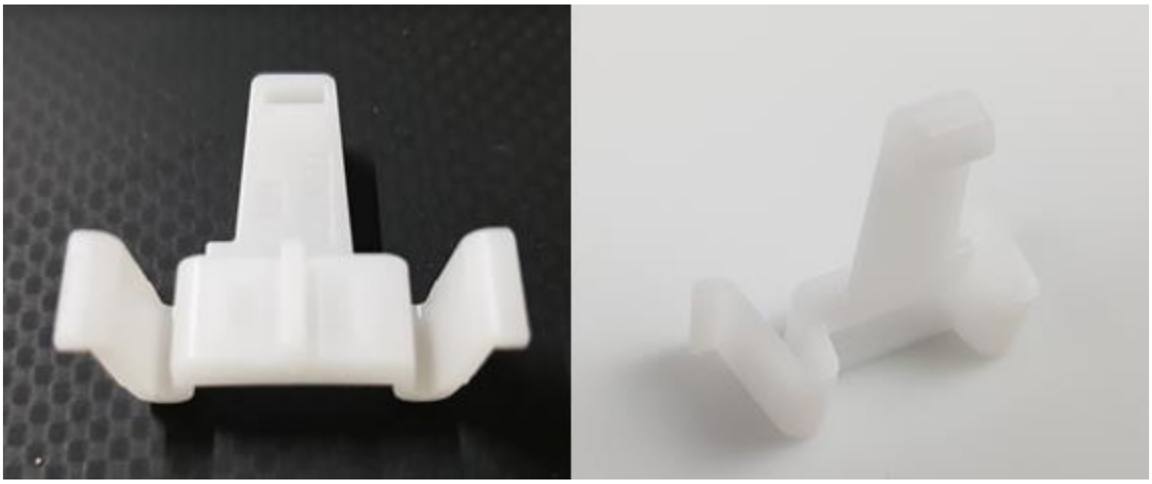


Figure 4.6 Part number 1: Snap-hitch, in the material POM.

Figure 4.7 Part number 2: Cable-lid, in the material PBT-PET-GF.

5 Target specification

In the following chapter, the target specifications are presented as well as the method used for defining them.

5.1 Method

The method used for developing the target specifications was a modified version of the process described in *Product design and development* by Ulrich and Eppinger. First the customer needs were collected in the research phase through interviews at Axis and through test runs of the injection molder. The interviews contributed with information regarding how Axis wanted the end result to perform and the test runs of the injection molder contributed with identification of potential sources of error which had to be solved. This information was summarized as the customer needs, which were then turned into target specifications by redefining them into measurable attributes and ranking their importance.

5.2 Customer needs

From the research phase information was gathered and turned into customer needs, shown in Table 5.1 . Each customer need was given an importance score of 1-3, where 1 is the least and 3 is the most important. The customer needs scored with a 3 have to be achieved and will therefore be the main focus for solving. Most of the customer needs scored with a 2 should be achieved, but they are not as important for the function of the process compared to the ones scored with 3. The customer needs scored with a 1 would be nice to achieve, but they will not receive any direct focus.

Table 5.1 Table of Customer needs.

Number	***Customer needs***	***Importance***
	From interviews	
1	Material used is the same as in end product	3
2	Possibility to injection mold details of different sizes	2
3	Time consumption has to be low	3
4	Impact resistance similar to end product	2
5	Temperature resistance similar to end product	2
6	Better functioning snap fittings than in 3D-printed details	3
7	Surface of injection molded detail should be glueable	1
8	Surface of injection molded detail should be tapeable	1
9	Good surface finish	2
10	Good dimensions and tolerances	2
11	Ability to perform 10 shots in a single mold	2
12	Easy to create mold	2
	From test runs of injection molder	
13	Ejection of detail from mold without damage to detail	3
14	Ejection of detail from mold without damage to mold	2
15	Method for deciding correct shot volume	1
16	Mold inlet has to be easy to clean	2
17	Correct cooling of molded detail	2
18	Mold must reach 100% degree of filling	3

5.3 Target specifications

The customer needs were given measurable attributes resulting in the following target specifications, shown in Table 5.2. The target specifications all relate to specific customer needs and they were given importance factors on a scale of 1-5, where 1 is the least important and 5 the most important.

Table 5.2 Target specifications.

Number	***Needs number***	***Metric***	***Unit***	***Importance factor***	***Margin Value***	***Ideal value***
1	1, 4, 5	Mold material resistant to molten plastic	°C	5	250	>300
2	2	Size of molded detail	mm	3	>45x45x45	>150x150x150
3	3, 12	Process time	Days	5	<10	0
4	6	Functioning snap fittings	Binary	5	Yes	Yes
5	7	Glueable surface	Binary	2	No	Yes
6	8	Tapeable surface	Binary	2	No	Yes
7	9	Surface finish	Subj.	3	-	-
8	10, 18	Geometry	Subj.	4	-	-
9	11, 15	Number of shots	Nbr	4	5	∞
10	13, 14, 16	Non-damaging ejection	Binary	5	Yes	Yes
11	17	Temperature at ejection	°C	5	<Tglasstransition	20°

6 Concept Generation and Selection

In the following chapter concepts for the mold material and the mold design are generated and then selected for testing. The concepts for the mold material aim to meet the demands of high temperature injection molding. The concepts for the mold design aim to solve problems from the target specifications which can not be solved by just the choice of mold material.

6.1 Method

Before the concept generation started, the problems were identified and divided into two separate concept processes, called *Design Concept* and *Material Concept*. This was because of the different nature of the problems that demanded completely different solutions. This divide is illustrated in Figure 6.1.

The *Design Concept* process included every problem not included in the *Material Concept* process. These problems had to be solved by other means than just the choice of mold material. The problems within these categories were divided into four separate problems and each of these problems had its own concepts created. Multiple concepts for each of the four problems were generated in order to gain a wider perspective of potential solutions. The concepts were then scored based on their potential and the highest scoring concepts were moved on into testing. In the case of multiple concepts reaching a similar score, every high scoring concept was selected.

In the *Material Concept* process every problem that was deemed solvable simply by the choice of mold material was included. This process generated concepts in the form of materials of interest for testing. In the selection phase of this process all the materials available within the time limit of the project were obtained and then moved on into testing.

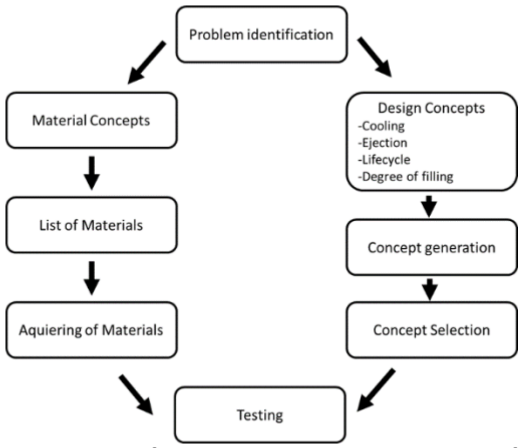


Figure 6.1 The process of concept generation and dividing of problems.

6.2 Design concept

From the target specification four problem areas were identified and deemed not solvable by the choice of mold material. These four problem areas were *Cooling*, *Ejection*, *Life span* and *Degree of Filling*.

6.2.1 Concept generation

The design concept generation was performed both through internal and external search. The internal search was performed through a modified version of the 6-3-5 method, described by Ullman (2010). It was modified into a 2-1-1-repeat method, where each participant wrote down one concept idea before exchanging notes and during one minute tried to add to the other person's idea. This was then repeated until all of the ideas were presented. Ideas involving similar solutions to problems were combined into single concepts, making the end result more clear.

The concepts generated is presented as a list of their names and as illustrations with short descriptions in the captions.

6.2.2 Concept selection

The concepts in each category were first scored based on four criteria: *Ease of creation*, *Ease of Use*, *Side effects* and *Effectiveness* shown in Table 6.1. The concepts with the highest scores were taken to the next step and then developed for testing.

Table 6.1

Ease of creation	How easy it is to create the concept in CAD and to manufacture.
Ease of Use	How easy it is to use when molding products.
Side effects	How little extra amount of side effects regarding the molded parts in regard to flashing, gate marks, surface finish etc.
Effectiveness	How effective the method is deemed to be to solve the designated task.

6.2.3 Cooling

6.2.3.1 Problem definition

Cooling is an important factor during the injection molding process, both for the sake of decreasing the cycle time but also to minimize shape warping of the injection molded part. When using additive manufactured plastic molds new issues arise since the heat is not transferred away as efficiently as in metal molds. The reason behind this is because of plastics non heat conductive behavior, a factor that can cause the mold to deform.

6.2.3.2 Concept generated

The concepts generated can be seen in Table 6.2 and in Figure 6.2-8

Table 6.2 Concept generation table for Cooling.

Number	***Name***
1	Cooling Spray
2	Cold Inserts
3	Heat Transferring Surface Finish
4	Cooling Channels
5	Precooled Mold
6	Molds in Conductive Material
7	Prolonged Ejection

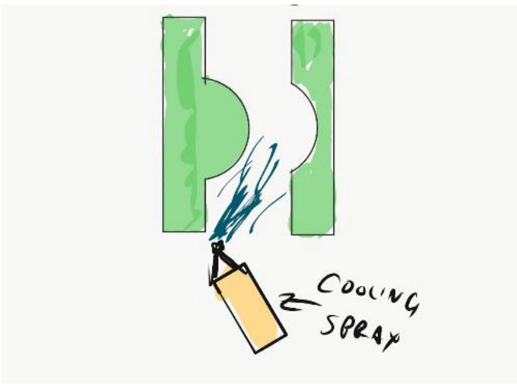


Figure 6.2 Concept 1: Cooling spray used to cool the injection mold.

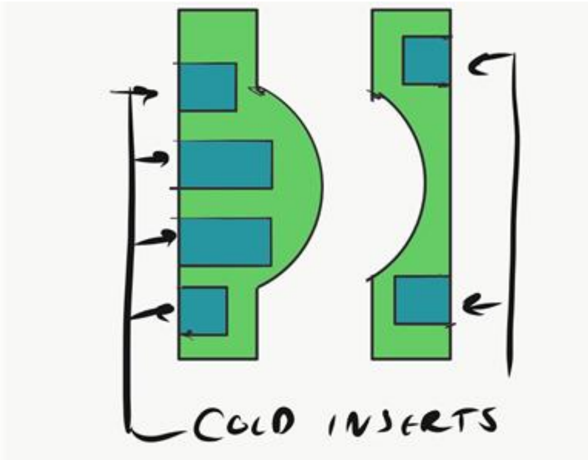


Figure 6.3 Concept 2: *Cold Inserts* uses cold metal rods in pre made holes to cool the molds.

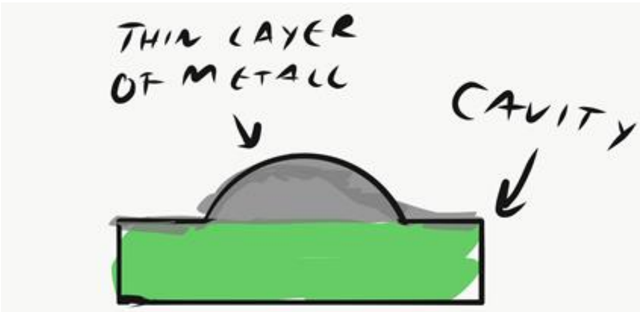


Figure 6.4 Concept 3: *Heat Transferring Surface Finish* uses coating on the top layer with a conductive surface material to transfer the heat away.

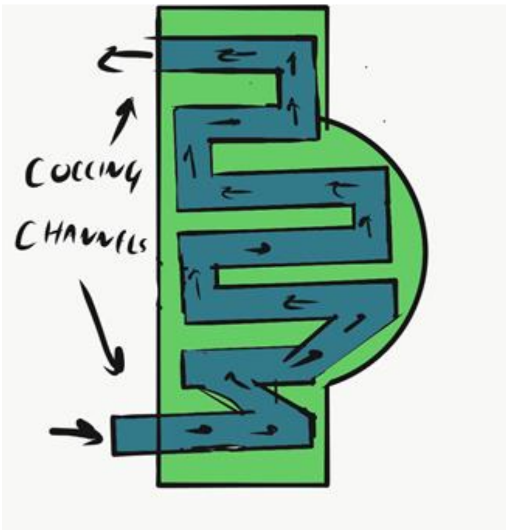


Figure 6.5 Concept 4: ***Cooling Channels*** **uses channels inside the molds to transfer heat away.**

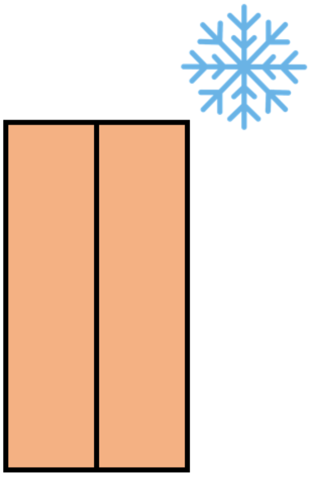


Figure 6.6 Concept 5: ***Precooled Mold*** **uses external cooling of the whole mold before injection.**

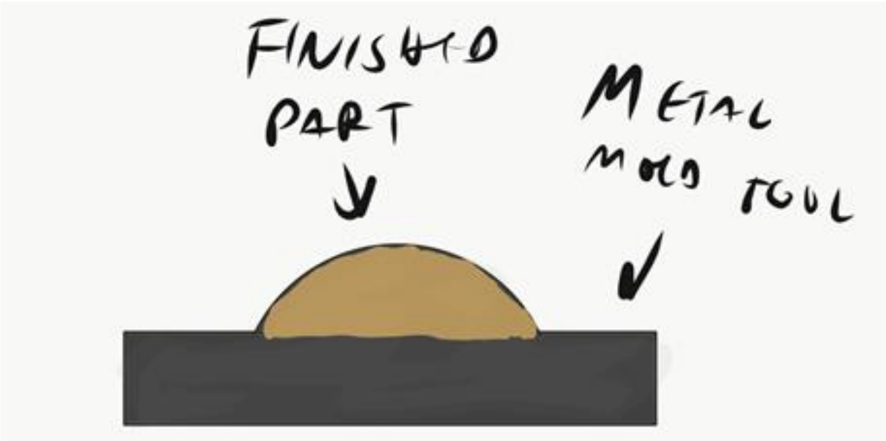


Figure 6.7 Concept 6: ***Cavities in Conductive Material*** **uses molds constructed in a conductive material such as metal.**

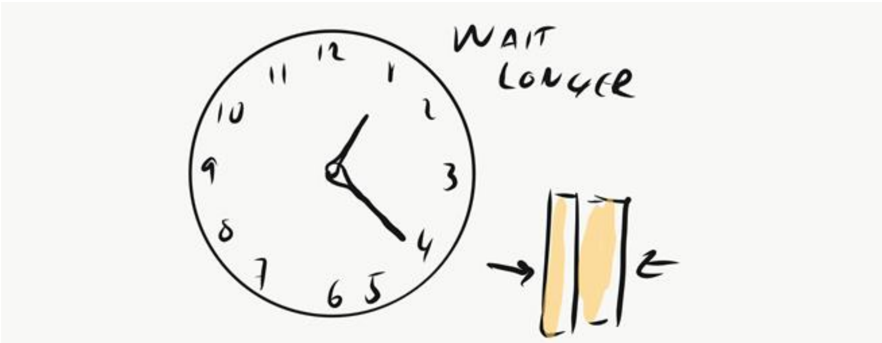


Figure 6.8 Concept 7: ***Prolonged Ejection*** **uses a longer pause between the injection and the ejection.**

6.2.3.4 Concept selection

The results from the concept selection can be seen in Table 6.3. The concepts *Cooling Spray* and *Prolonged Ejection* received a similar weighted score which was greater than the other concept's by a big margin. They were therefore moved into the testing phase.

Table 6.3 Concept selection matrix for Cooling

Number	***Name***	***Ease of creation***	***Ease of use***	***Side effects***	***Effectiveness***	***Total Score***	***Weighted Score***
	Weighting	**0.2**	**0.3**	**0.1**	**0.4**		
1	Cooling Spray	5	4	3	3	15	3.7
2	Cold Inserts	2	2	3	2	9	2.1
3	Heat Transferring Surface Finish	1	4	4	3	12	3
4	Cooling Channels	2	2	3	2	9	2.1
5	Precooled Mold	3	3	3	2	11	2.6
6	Cavities in Conductive Material	1	3	4	4	12	3.1
7	Prolonged Ejection	5	4	3	3	15	3.7

6.2.4 Ejection

6.2.4.1 Problem definition

Due to plastics tendency to stick to the mold surface a reliable method for ejecting the molded product is necessary. Within mass production this is often done with both release agents and ejection pins. Since the machine used in this project has malfunctioning pins an alternative is required. This step might however also be necessary in fully functional machines since ejection most likely will be much more difficult compared to a standard injection molding process.

6.2.4.2 Concept generation

The concepts generated can be seen in Table 6.4 and in Figure 6.9-17

Table 6.4 Concept generation table for Ejection

Number	***Name***
1	Molded Injection Pins
2	Multiple Molds
3	Inserted Injection Pins
4	Divided Molds
5	Controlled Flashing
6	Inserts Between Cavities
7	Meltable Inserts
8	Release Agent Spray
9	Release Agent in Granulate

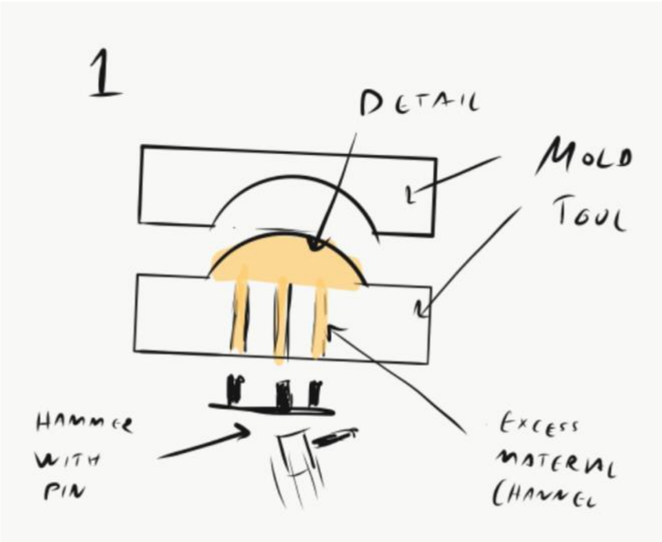


Figure 6.9 Concept 1: *Molded Injection Pins* uses holes in the molds which fills up with molten plastic to create plastic ejection pins. These pins have to be removed post ejection.

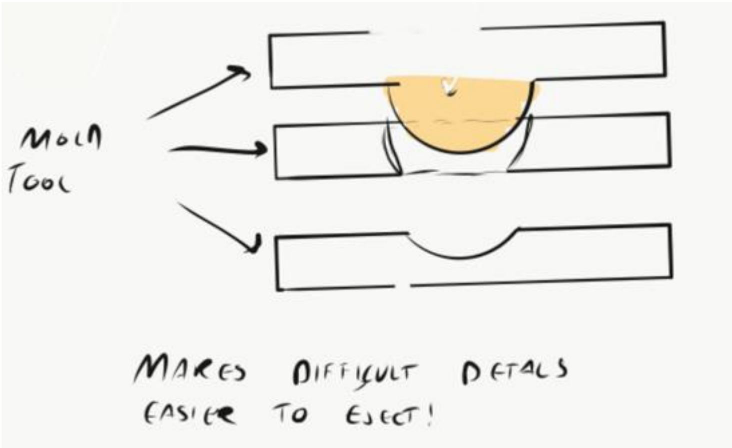


Figure 6.10 Concept 2: ***Multiple Molds*** **divides the part into several molds which can be separated making the ejection easier.**

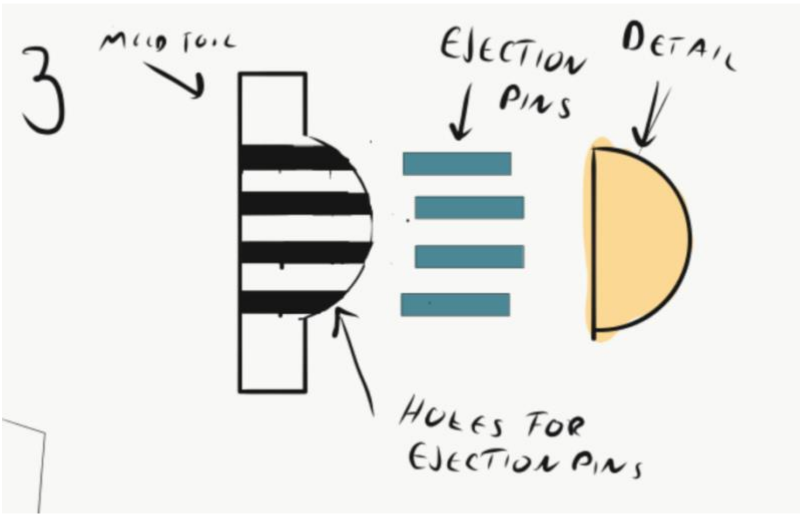


Figure 6.11 Concept 3: ***Inserted Injection Pins*** **uses ejection pins inserted into the mold, similar to conventional injection molding, but these has to be pushed out manually during ejection.**

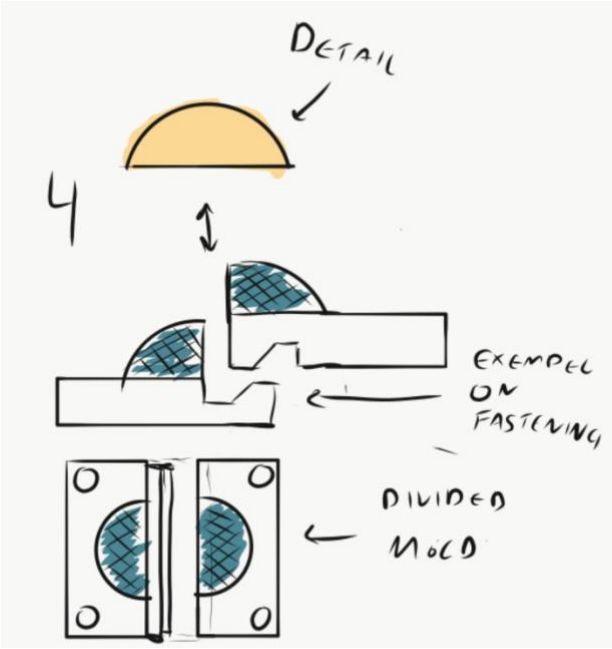


Figure 6.12 Concept 4: ***Divided Molds*** **uses a way to divide the molds to make it easier to eject the finished detail.**

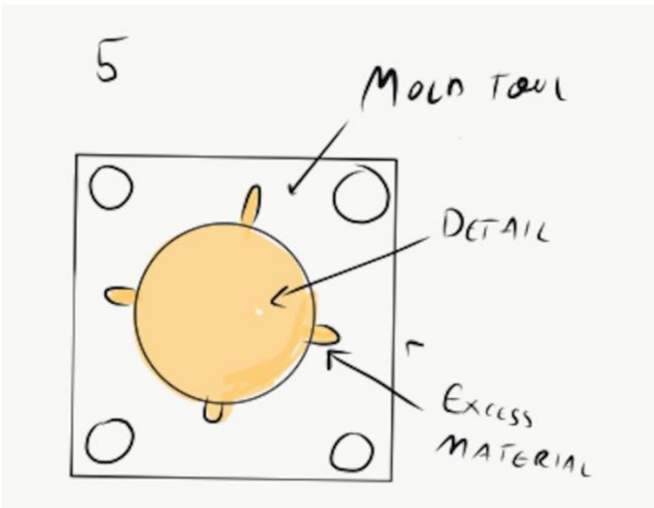


Figure 6.13 Concept 5: ***Controlled Flashing*** **uses cavities for flashing designed in strategic places in the mold. This flashing can then be used to grip and pull out the product without damaging the product itself.**

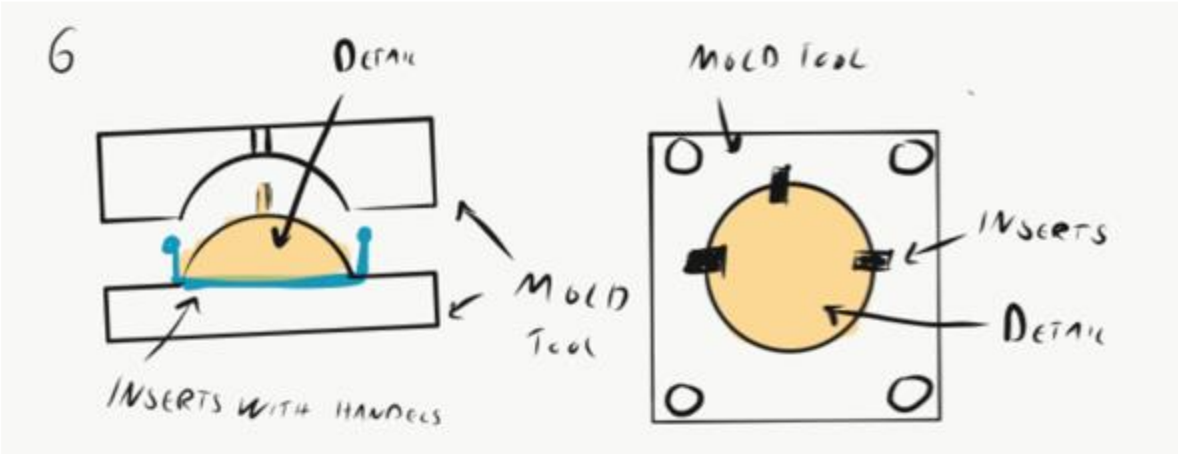


Figure 6.14 Concept 6: ***Inserts Between Cavities*** **uses inserts in strategic positions in the mold that can later be used as leverage when ejecting the injected molded detail.**

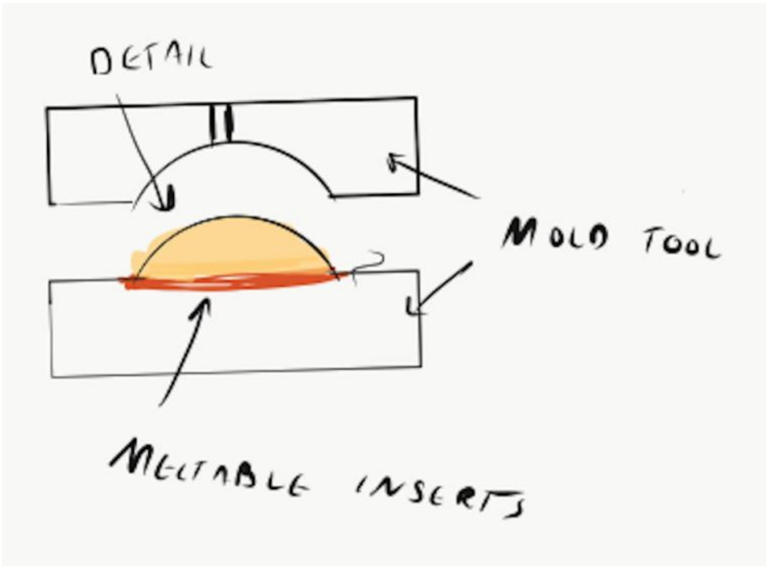


Figure 6.15 Concept 7: ***Meltable Inserts*** **uses an easily removable material applied in the mold before injection.**

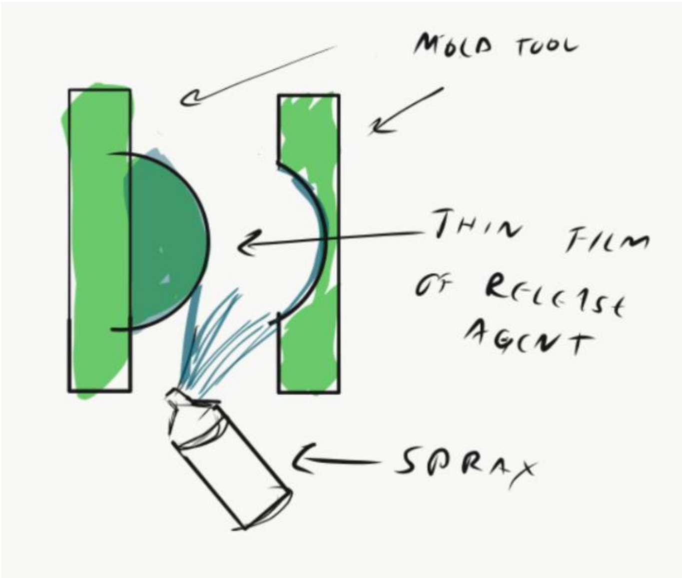


Figure 6.16 concept 8: ***Release Agent Spray*** **uses an external release agent.**

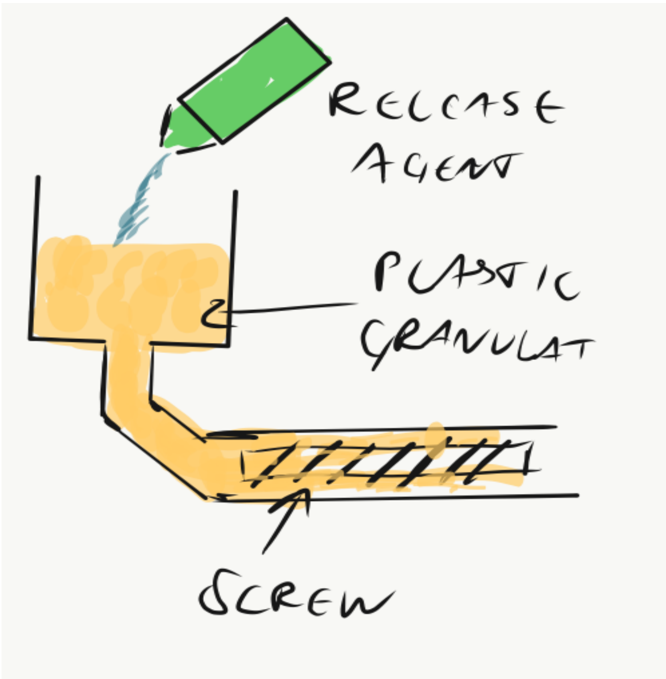


Figure 6.17 Concept 9: ***Release Agent in Granulate*** **uses an internal release agent.**

6.2.4.3 Concept selection

Controlled Flashing received the highest score followed by Release agent spray, Molded injection pins and inserts between cavities shown in Table 6.5. These were also the concepts chosen to continue with.

Table 6.5 Concept selection matrix for Ejection

Ejection	***Name***	***Ease of creation***	***Ease of use***	***Side effects***	***Effectiveness of ejection***	***Total Score***	***Weighted Score***
	Weighting	***0.2***	***0.3***	***0.1***	***0.4***		
1	Molded Injection Pinns	4	3	2	4	13	3.5
2	Multiple Cavities	1	2	4	3	10	2.4
3	Inserted Injection Pinns	2	3	3	4	12	3.2
4	Divided Molds	2	2	3	3	10	2.5
5	Controlled Flashing	4	4	3	4	15	3.9
6	Inserts Between Cavities	2	3	4	4	13	3.3
7	Meltable Inserts	2	3	2	3	10	2.7
8	Release Agent Spray	5	5	3	2	15	3.6
9	Release Agent in Granulate	5	4	2	2	13	3.2

6.2.5 Life span

6.2.5.1 Problem definition

The target specifications state that the minimum amount of shots in one injection mold is five. This means that a mold must be able to withstand the wear and tear caused by five shots without breaking. This is not a large amount of shots, but indication towards this being a problem were seen during the initial test runs of the injection molder. Since the initial tests were performed with a limited amount of molds further evaluation of the problem’s relevance will take place during the material testing. If no signs of problems with molding more than five parts in a single mold are found, this concept will be discontinued. A number of concepts were however generated if the problem was still deemed relevant.

6.2.5.2 Concept generation

The concepts generated can be seen in Table 6.6 and in Figure 6.18-6.23

Table 6.6 Concept generation table for Life span

Number	***Name***
1	Coating of Mold
2	Curing of Mold
3	Multiple Molds
4	Filament Channels
5	Standardized Metal Gate
6	Changeable Inserts

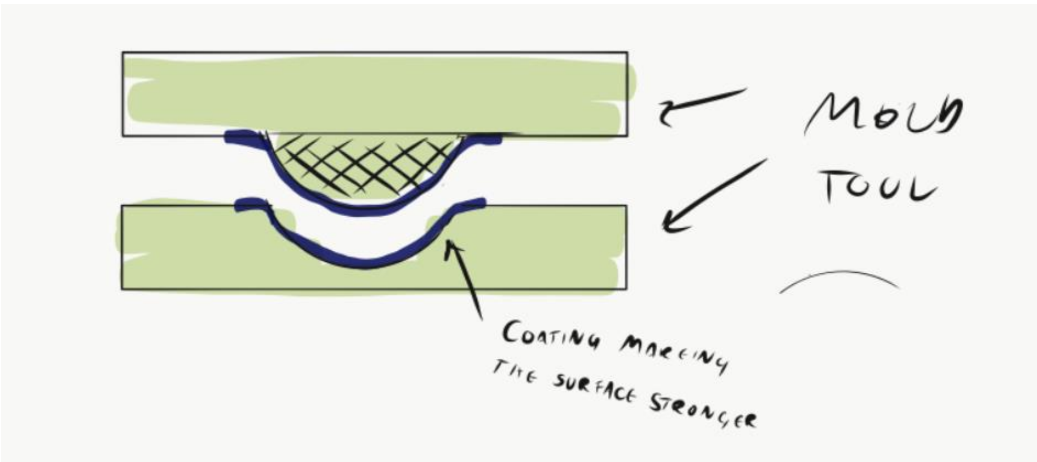


Figure 6.1 Concept 1: *Coating of Mold* uses a thin layer of coating in order to make the surface slicker and increase the longevity of the mold.

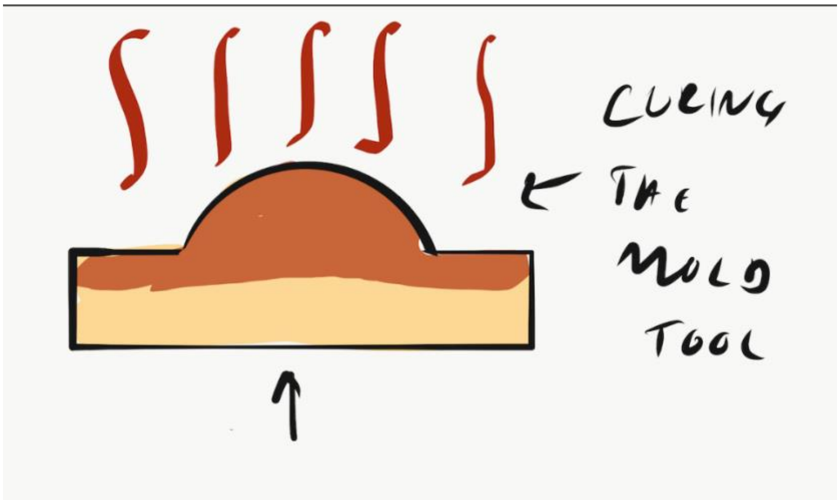


Figure 6.19 Concept 2: ***Curing of Mold*** **uses curing of the mold with either heat, chemicals or UV-light depending on the printing method to increase the life span.**

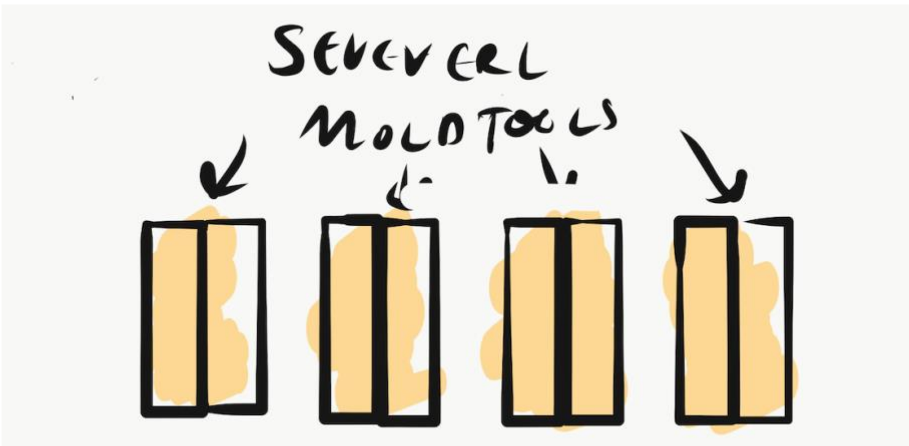


Figure 6.20 Concept 3: ***Multiple Molds*** **uses more than one mold which are then only used once or twice.**

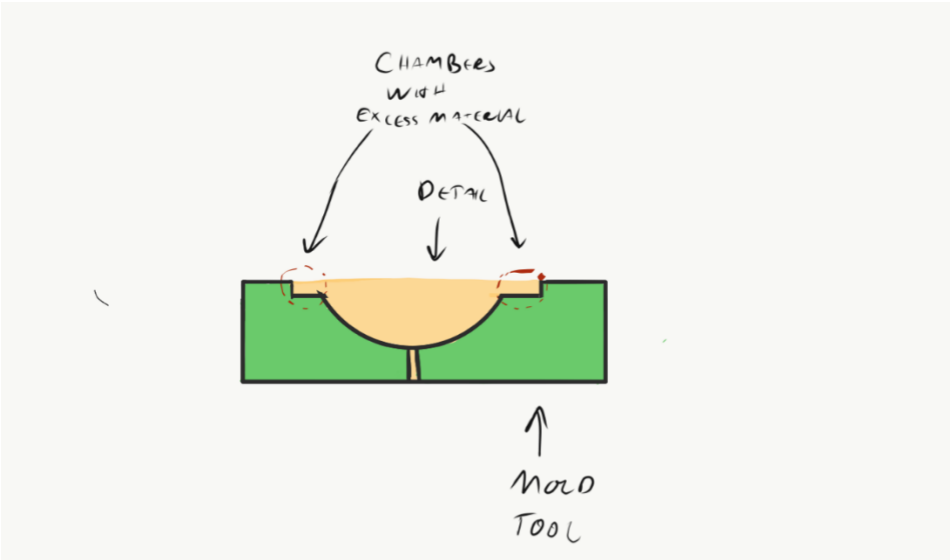


Figure 6.21 Concept 4: *Filament Channels* uses chambers and channels were excess material can go once they have filled up the entire detail, decreasing the risk of cracking.

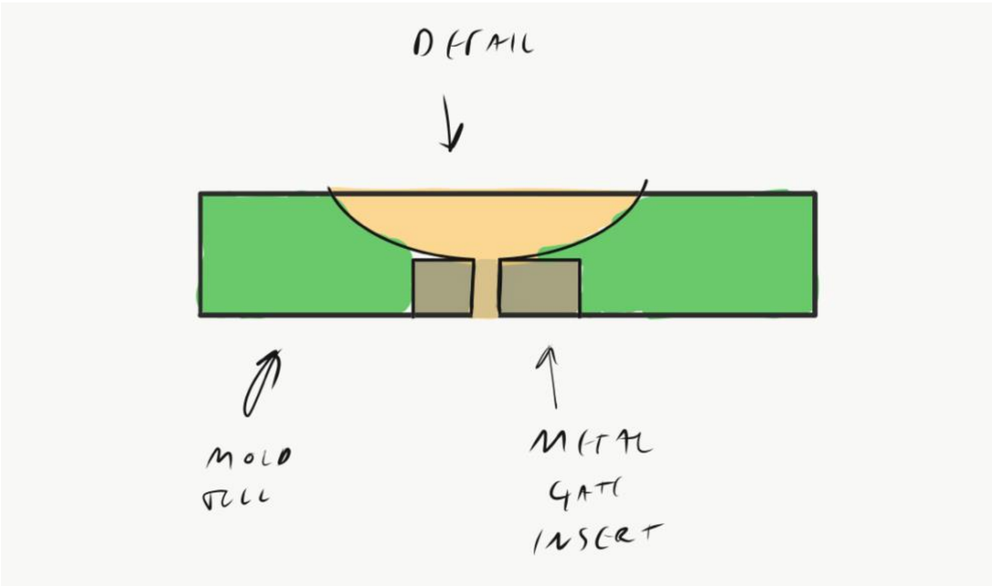


Figure 6.22 Concept 5: *Standardized Metal Gate* can be used to minimize the wear on the gate.

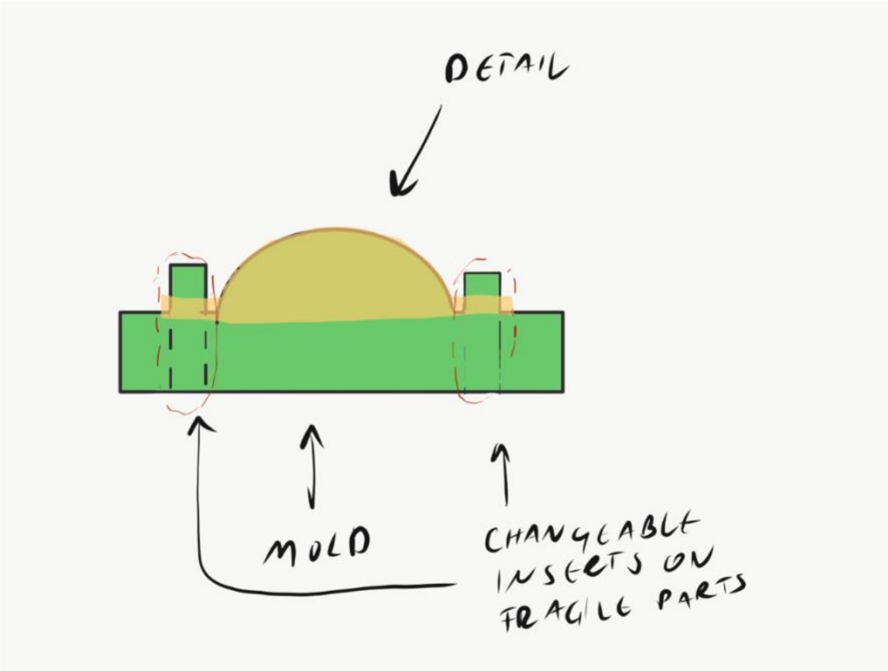


Figure 6.23 Concept 6: *Changeable Inserts* can be used on weak parts that have a tendency to break during injection molding.

6.2.5.3 Concept selection

From Table 6.7 it can be seen that metal gate was the concept which generated the highest score followed by changeable inserts and coating of cavity. *Changeable inserts* was a concept that seemed reasonable to try out since they can increase the usefulness of molds and was together with metal gate the only concept from this category to be tested.

Table 6.7 Concept selection matrix for Life span.

Life span		Ease of creation	Ease of use	side effects	Effectiveness	Total Score	Weighted Score
	Weighting	***0.2***	***0.3***	***0.1***	***0.4***		
1	Coating of Mold	3	4	2	3	12	3.2
2	Curing of Mold	3	3	2	3	11	2.9
3	Multiple Molds	2	2	4	4	12	3
4	Filament Channels	2	2	2	2	8	2
5	Standardized Metal Gate	3	3	4	4	14	3.5
6	Changeable Inserts	2	3	4	4	13	3.3

6.2.6 Degree of filling

6.2.6.1 Problem definition

A problem that was noticed during initial tests was that molds sometimes had a hard time to reach 100 percent degree of filling. The reason for this was most likely that as molds were hard to get completely clean from support material, air got trapped in holes and that plastic flowed out in the crack between the two mold halves instead of filling up the cavities.

6.2.6.2 Concept generation

The concepts generated can be seen in Table 6.8 and in Figure 6.24-6.27

Table 6.8 Concept generation table for Degree of filling

Number	***Name***
1	Larger Gate
2	Venting Holes
3	Increased Seal Between Molds
4	Extended Material “Chambers”

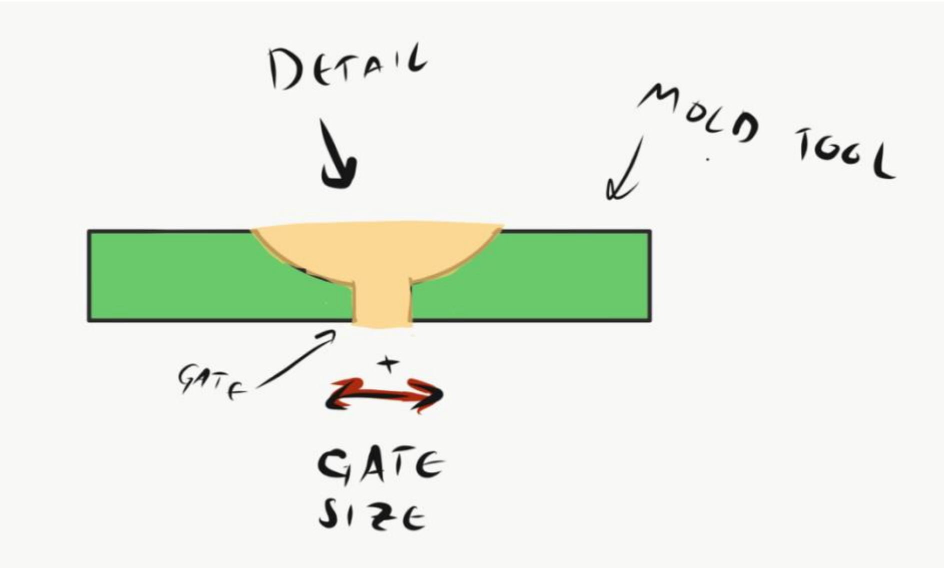


Figure 6.24 Concept 1: ***Larger Gate*** **uses a larger gate to increase the flow into the mold.**

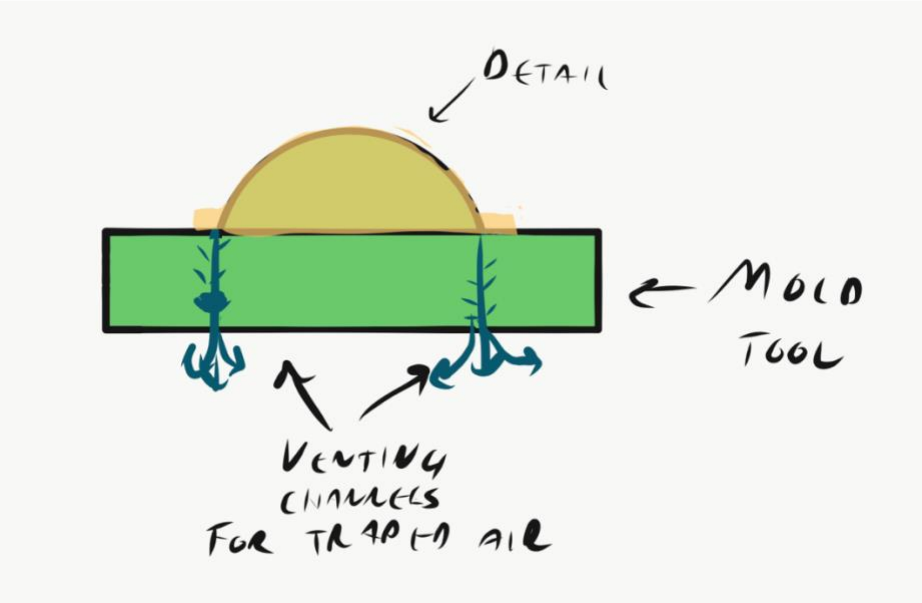


Figure 6.25 Concept 2: ***Venting Holes*** **uses small venting holes in critical positions to release trapped air.**

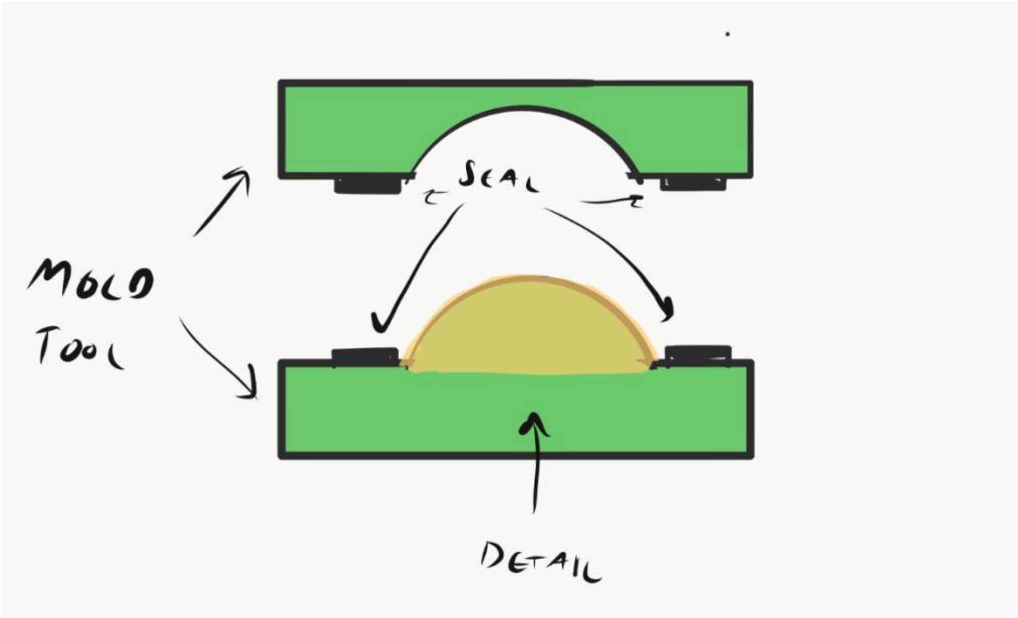


Figure 6.26 Concept 3: ***Increased Seal Between Molds*** **uses protruding features increasing the pressure around the cavity and thus the seal between the two halves.**

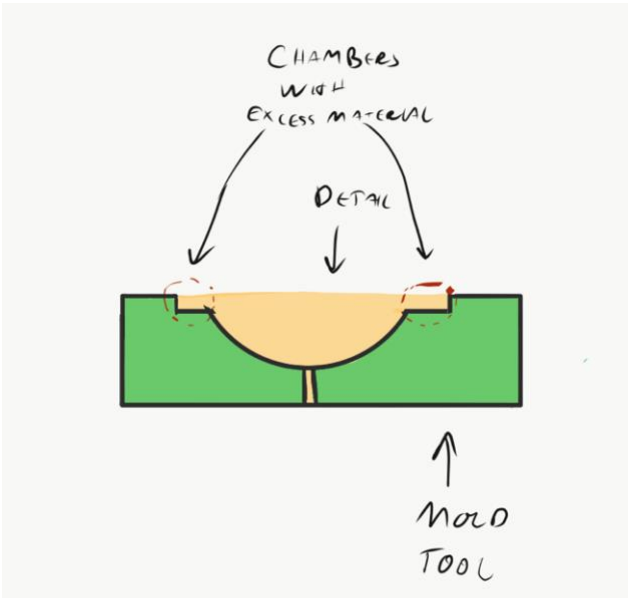


Figure 6.27 Concept 4: ***Extended material chambers*** **uses chambers to introduce more material, increasing the chance of getting a higher degree of filling.**

6.2.6.3 Concept Selection

In Table 6.9 it can be seen that *Larger Gate* and *Venting Holes* were concepts that scored the highest in *Degree of Filling* and was therefore continued with into testing.

Table 6.9 Concept selection matrix for Life Span.

Degree of filling	*Name*	*Ease of creation*	*Ease of use*	*Side effects*	*Effectiveness*	*Total Score*	*Weighted Score*
	Weighting	***0.2***	***0.3***	***0.1***	***0.4***		
1	Larger Gate	4	5	4	4	17	4.3
2	Venting Holes	2	3	3	4	12	3.2
3	Increased Seal Between Molds	3	4	3	2	12	2.9
4	Extended Material “Chambers”	2	3	2	2	9	2.3

6.3 Material Concepts

6.3.1 Concept Generation

The concept generation was performed solely through external search. Relevant 3D-printing technologies and materials were gathered and compared. The materials were then selected based on their occurrence in existing solutions, their material properties but also on their availability regarding the time frame of the project.

6.3.2 Demands

For the mold to function as intended and to be durable during the whole life span, it needs to achieve a number of demands. The injection materials have to be injected at certain temperatures and the mold must therefore be able to withstand these temperatures while under the internal pressure created inside the mold.

6.3.3 Materials

The materials shown in Table 6.10 were all deemed interesting for further testing and research. The materials have been listed in order of the 3D-Printing technology associated to them. The reasoning behind choosing these materials is mainly because of their high Heat deflection temperature, high melting temperature and or that the materials have been seen used in similar applications.

Table 6.10 Material Concept.

DMLS	***FFF***	***MJP***	***SLA***	***SLS***	***Binder Jetting***
Steel	Onyx	M2R-WT	Accura Phoenix1	PA3200 GF	Zcast 501 Pail
Aluminium	PC	M2R-TN	Formlabs Heat Resistant	PA1101	zb 56 clear
	Ultem	M3-X	Accura SL 5530	EOS PEEK	
	Adamant	Digital ABS	Visjet tough	Alumide	
	KOLTRON		PerForm Somos		
	ADURA X				
	PET-G				

6.3.4 Material Selection

From the materials listed in Table 6.10 nine materials were moved into the testing phase. Since all materials from the list were deemed to be interesting it would be optimal to test all of them, but based on their availability and the project's time frame nine were sourced and selected. These nine materials are shown in Table 6.11.

Table 6.11 Materials selected for testing.

Materials selected
Aluminium (DMLS)
Onyx
Koltron
Adura X
Adamant
M2R-WT
Digital ABS
PA (SLS)
PET-G (FFF)

7 Concept Tests

In the following chapter the process leading up to the final selection of relevant concepts will be described. The process included scoring of concepts, selection of scored concepts, testing of selected concepts and final selection based on the results from the tests. The concepts in the chapter are both the design concepts and the material concepts from chapter 6. The results are summarized in the end.

7.1 Method

The tests were all performed with the MCP Minimolder 12/90 HSP, described in Chapter 3.3.5 *MCP Minimolder 12/90 HSP*. Every shot was done as similar as possible to ensure as reliable results as possible.

The design concepts were tested with LDPE because of the material's good injection molding properties, such as low temperature and good flowability. This made the positive and negative sides of the design concepts easier to evaluate as faults and damages caused by the material was easier to detect and identify.

The material concepts were tested with both LDPE and PC-ABS as the purpose of these tests was to conclude which mold material could handle high temperature injection molding.

7.2 Design Concept Tests

The design concepts were based on the following injection molds seen in Figure 7.1-2. These two molds will work as base molds on which all the design concepts will be designed into. The parts to be created by the molds are the two parts chosen in Chapter 4.5 *Part Selection*. Some changes were done to the parts to make them more accessible for the simplified injection molding of this project, like only having one gate instead of multiple. These changes include filling up the top part of the *Cable-lid* and removing the fin at the back of the *Snap-hitch*. The gate placement was also changed for the *Snap-hitch*. The modifications to the parts are shown in Figure 7.3.

During the test runs problems occurred regarding degree of filling of the top part of the *Snap-hitch*. This problem has probably more to do with the changed gate placement and was resolved with erasing the hollowed-out portion of the top part of the *Snap-hitch*, also seen in Figure 7.3.

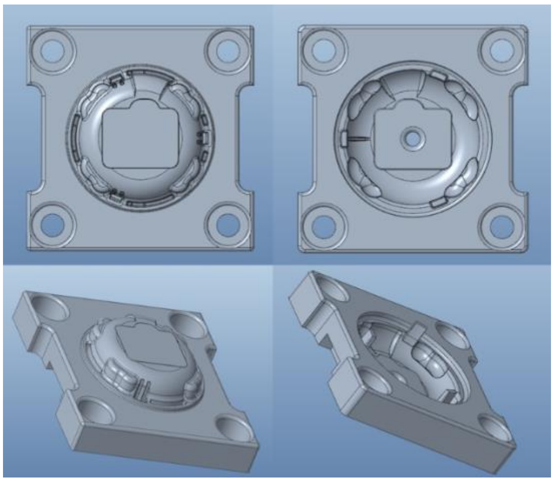


Figure 7.1 Mold for part 2 *Cable Lid*.

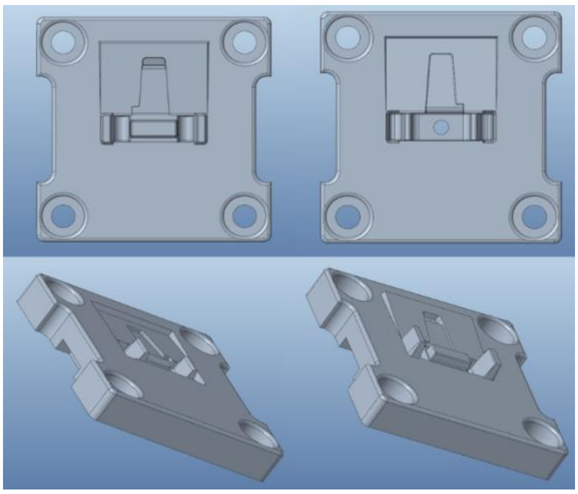


Figure 7.2 Mold for part 1 *Snap hitch*.

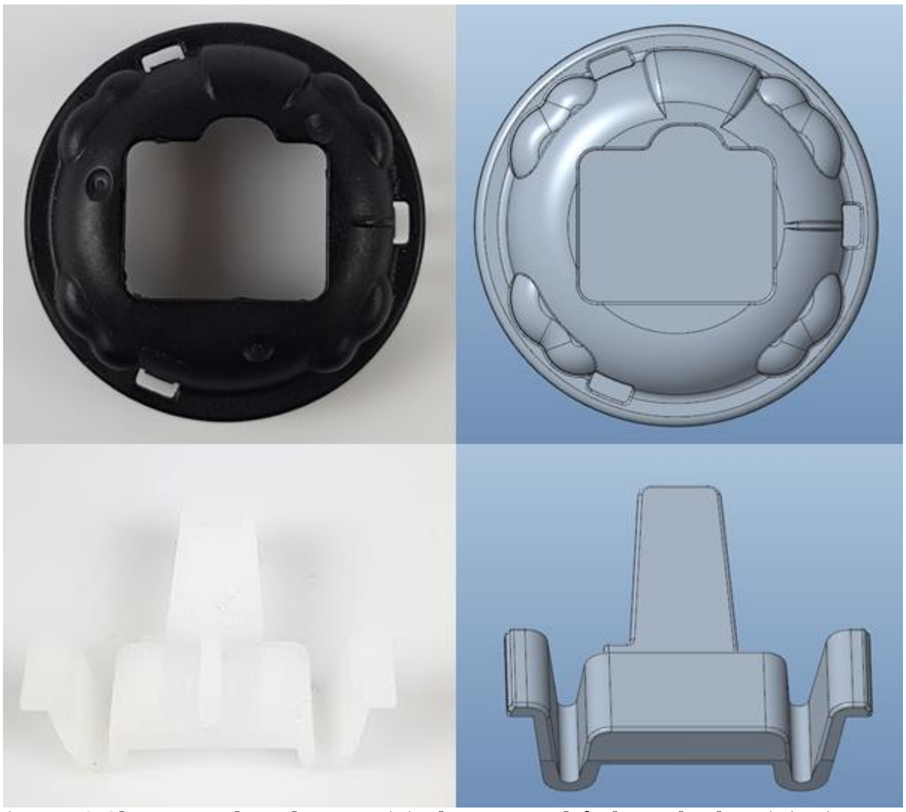


Figure 7.3 Changes made to the two original parts. Top left shows the three injection gates which were changed to only one gate placed in the middle of the *Cable-lid* to the top right. Bottom left to bottom right shows the removed fin and filled cavity of the top of the *Snap-hitch*.

7.2.1 Cooling

7.2.1.1 Cooling Spray

Cooling Spray was tested by spraying the part with a cooling spray right after the mold was opened. The result of the test was that the mold and part was cooled too quickly which resulted in damage to both. The cooling spray used shown in Figure 7.4

Figure 7.4. Cooling spray used

7.2.1.2 Prolonged Ejection

Prolonged Ejection was tested with many of the different molds. The results of the tests were that a waiting time of 25 to 30 seconds between injection and the opening of the mold was optimal. This waiting time allowed the plastic to harden properly and attain the proper geometry, while also being ejectable.

7.2.1.3 Evaluation and Final Selection

From the tests it was concluded that *Cooling Spray* increased the likelihood of molds cracking and it was therefore discontinued. *Prolonged Ejection* was not only the simplest concept but also gave improved and consistent results. The only concept to be continued with from cooling is prolonged ejection.

7.2.2 Ejection

7.2.2.1 Controlled Flashing

The concept *Controlled Flashing* was tested on the *Cable-lid* and the *Snap-hitch*. In Figure 7.5 the controlled flashing are marked in red, and were designed in order to let more material flow into these cutouts. The configuration had no or slightly positive effect on the *Snap-hitch* but did increase the ejection of the *Cable-lid* significantly, since it gave more leverage and material to grab onto without damaging the part itself. In Figure 7.6 the excess material from the controlled flashing can be seen.

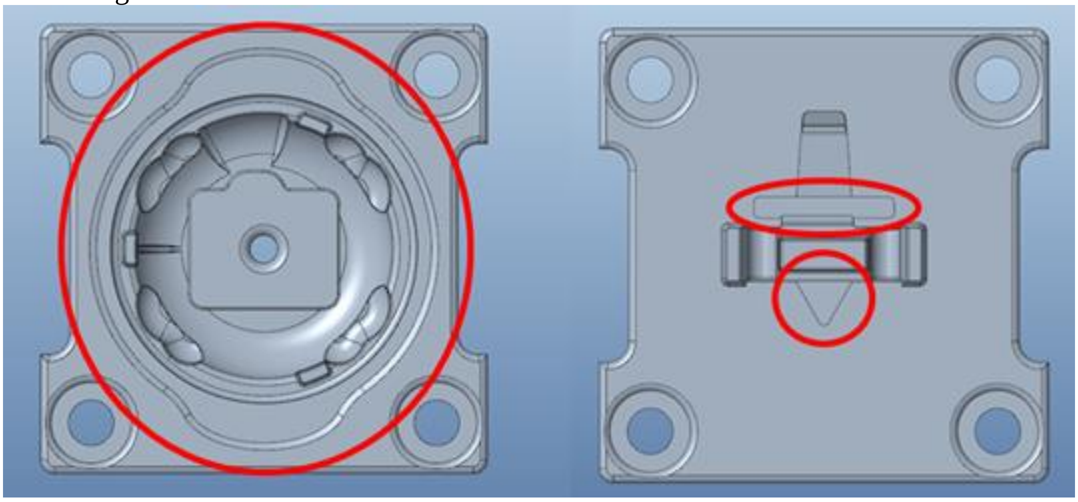


Figure 7.5 Mold designed with extra space for material to leak out into.

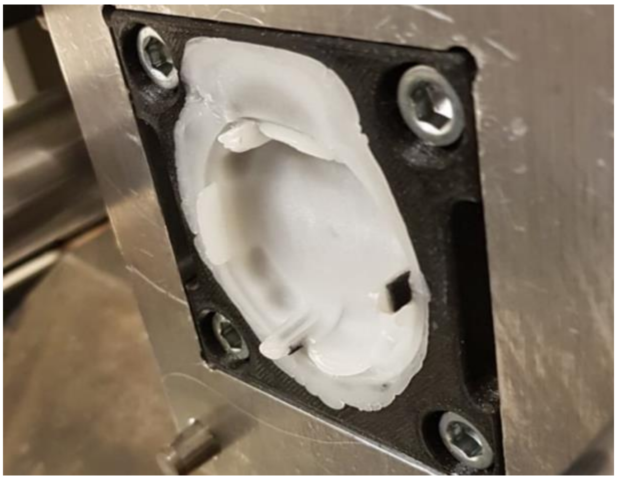


Figure 7.6 Flashing on injected molded part, the flashing is then trimmed with the help of a scalpel.

7.2.2.2 Divided Mold

Divided Mold was only tested for the *Cable-lid* since the *Snap-hitch* in previous tests showed no signs of needing such a complex ejection method. The parts were designed in different configurations, but were then decided on using the version that divides the part through all holes and cavities of the extruding half of the mold as can be seen in Figure 7.7-7.8. This iteration made the molded part a bit easier to eject and easier to both clean from excess and support material when printed. It also made a small gap for air to be vented through. Both of these features likely rendered the part to be more easily and reliably filled during injection. This use for the insert is further evaluated in Chapter 7.2.4 *Degree of Filling.*

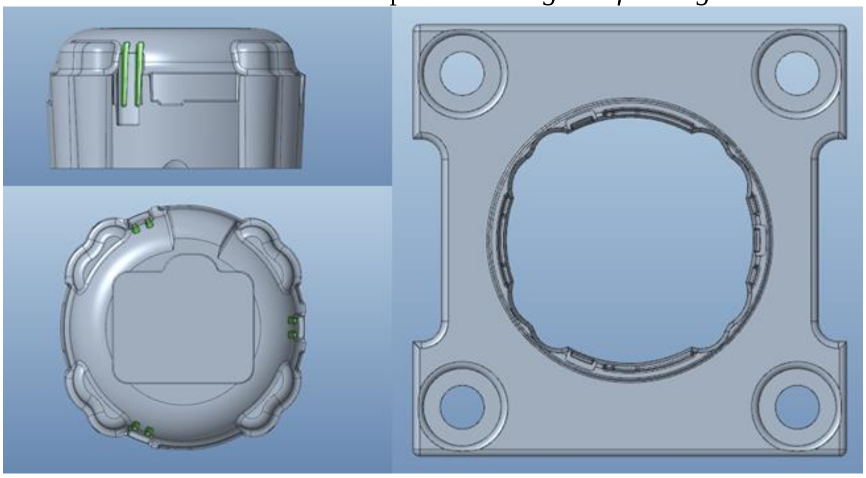


Figure 7.7 The left images show the insert from two different angles. The right image shows the other part of the mold where the insert is mounted.

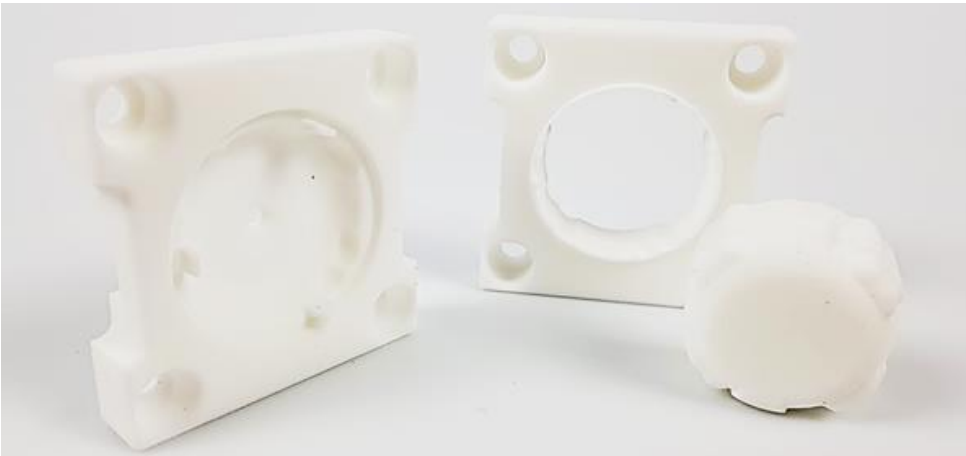


Figure 7.8 Photo of Concept insert here made in Polyamide from SLS.

7.2.2.3 Release Agent Spray

The release agent spray, shown in Figure 7.9, was liberally applied on the molds before the first shot and then again when the need occurred. From the test it was concluded that the spray greatly increased the ease of ejection for the parts.

Figure 7.9 The release agent spray used.

7.2.2.4 Molded Injection Pins

The Molded Injection Pins concept was tested by increasing the size of the gate and giving it more grip so the details would attach to the gate side of the mold. The detail could then be ejected with the help of a pin and a hammer shown in Figure 7.10. It resulted in the parts being easier to eject.

Figure 7.10. Using hammer and pin on backside of bracket to eject part stuck in mold.

7.2.2.5 Evaluation and Final Selection

From the tests all 4 concepts were deemed to be useful. Inserts between cavities did however only increase the ejection slightly and was more useful as a way to increase the degree of filling.

7.2.3 Life Span

7.2.3.1 Changeable Inserts

The tests for *Changeable Inserts* were only made for the *Cable-Lid* since the *Snap-Hitch* from the initial test runs did not show to have as weak features. The features of *Cable-lid* that were most susceptible to damage were the protruding pins designed to make the undercuts possible. They were therefore redesigned to be able to be changed when needed as seen in Figure 7.11. The results from the tests showed that the inserts did not change the geometry of the molded product and they did not break. *Changeable Inserts* was therefore deemed successful.

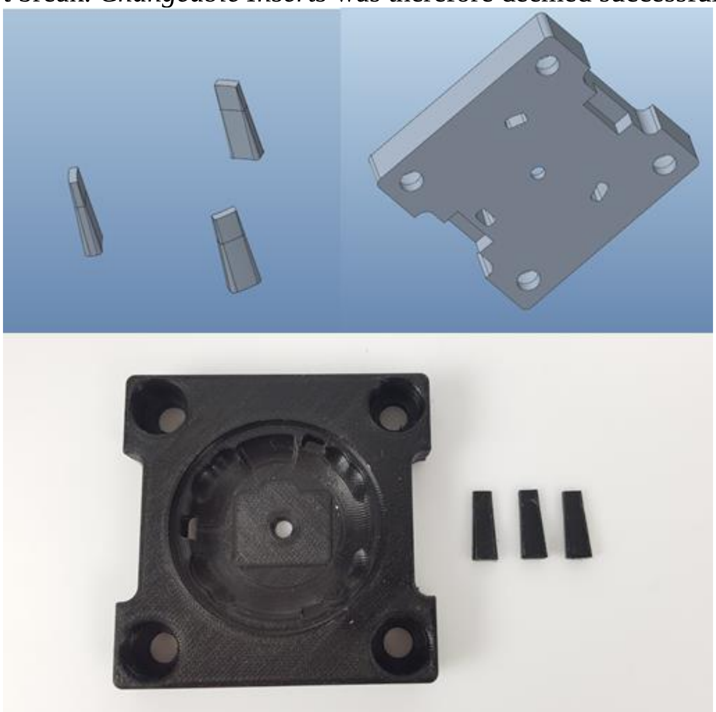


Figure 7.11 Cad model and printed PETG version of the changeable inserts. The pins as seen in the top left slides in on the backside of the mold seen in the top right.

7.2.3.2 Evaluation and final selection

Changeable Inserts seems to be a viable option when making molds with smaller protruding details susceptible to breaking. Once they break, they can easily be replaced, instead of replacing the whole injection mold.

7.2.4 Degree of Filling

7.2.4.1 Larger gate

In standard injection molding the injection gates are often rather small, something that resulted in both low degree of filling and makes the gates clog up when using 3D-Printed injection molds. The most common way to circumvent this problem seen during the external search was to use a larger gate to increase flow. During the initial tests, different sizes and drafts on the injection gate were tested. The different sizes of the gates are shown in Figure 7.12. The most promising results were achieved with a 4 mm hole using neutral draft, allowing good flow and a removable sprue.

Figure 7.12 Different sizes for injection gate ranging from 2-4 mm. At the far right a gate clogged with injected plastic is shown.

7.2.4.2 Venting Holes

Since venting holes preferably should be smaller than 0.03 mm (Bruder, 2014) it is almost impossible to design and print these venting holes in the CAD model. It is instead better to design a larger cutout and then insert a part that only leaves a small hole for air to leak out. This approach was tested in the concept *Inserts Between Cavities*. There it was done in order to facilitate ejection but showed promising results for increasing the degree of filling. In Figure 7.13 a mold using venting holes can be seen. The results were positive as it allowed for 100 percent degree of filling and therefore making the geometry of the molded product complete. The down side to this concept is that it leaves a small amount of flashing, shown in Figure 7.14. around the edges of where the venting is placed.

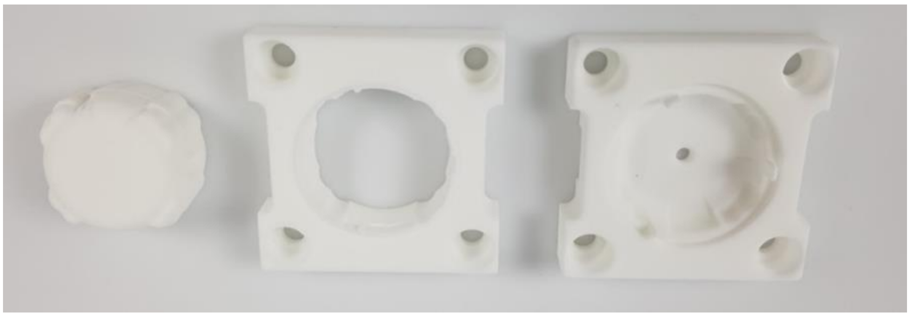


Figure 7.13 The inserts between cavities solution was also used for testing venting holes. The mold is divided through all small nooks and holes.

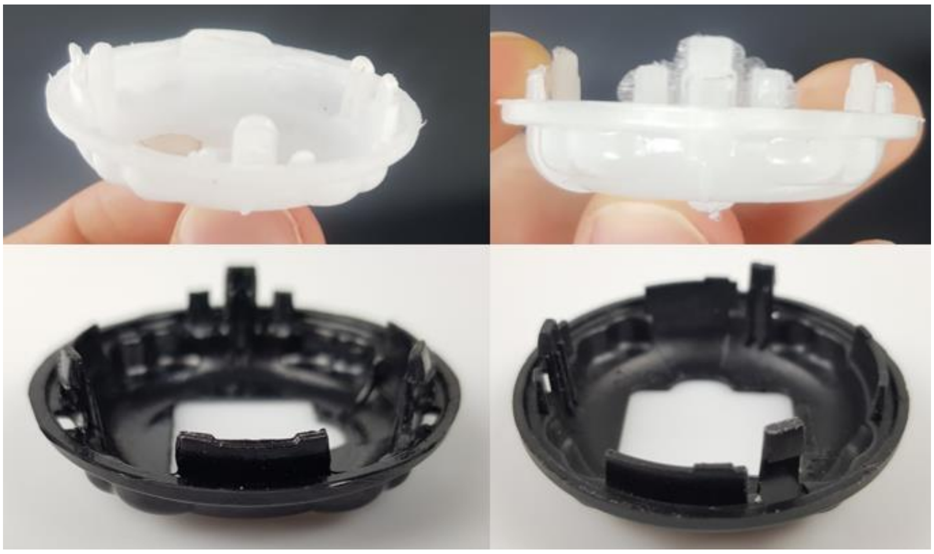


Figure 7.14 Comparison between product made without venting in the top left image to products made with venting in the top right and bottom left images. Original product in the bottom right image.

7.3 Material Concept Tests

7.3.1 Method

Test runs with the selected materials were made following the procedure seen in Figure 7.15. The objective was to conduct every test under the same conditions in order to achieve the most reliable results possible and it was therefore important that the testing process was identical for every test. The machine was first cleaned and any plastic left from previous use was removed from the nozzle. Then the molds were sprayed with release agent before they were inserted into the machine.

A small amount of release agent was also applied between every shot. The injection molding was performed in each injection mold for five shots or until the mold broke. After the injection molding the mold was removed from the machine and the testing process was repeated with the next mold.

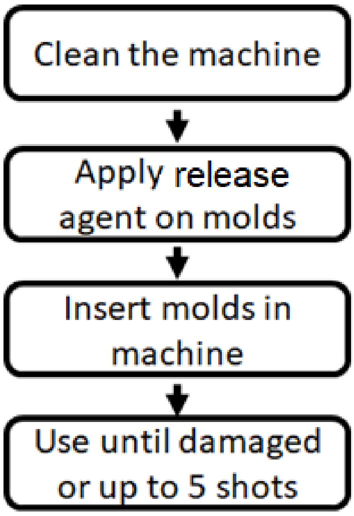


Figure 7.15 The testing process.

For PC-ABS an initial step of drying the granulate in a designated drying oven for three hours was also required. This was not necessary for LDPE and therefore only done with PC-ABS. The tests were conducted with both injection materials since they are injection molded at different temperatures which allows for a greater understanding of the mold materials tested. LDPE was injection molded at 200 degrees Celsius and PC-ABS at 260 degrees Celsius. The mold materials were first tested with LDPE and then with PC-ABS. The molds used for LDPE was only used for LDPE and the molds used for PC-ABS were only used for PC-ABS. If a mold could not complete five shots with LDPE based on temperature it was not tested with PC-ABS, because of the higher temperature of PC-ABS. Both injection materials are shown in Figure 7.16 in their granulate form.

After the tests the results were evaluated based on the number of shots performed, ease of ejection, surface finish of the molded product and potential damage to the mold. All of these factors were then the basis for a subjective rating of the molds on a scale of one to ten.

Figure 7.16 Materials used during material concept test, to the left the white LDPE to the right black PC-ABS.

7.3.2 Molds Used

The molds chosen for the test had the following configuration seen in Figure 7.17-18. Some of the design features tested during the design concept tests were used, such as *Larger Gate* and *Venting Holes*. Pictures of every mold tested, in all the materials, can be seen in Appendix D

Figure 7.17 *Snap-hitch* mold used during the material tests, here shown in PA (SLS).

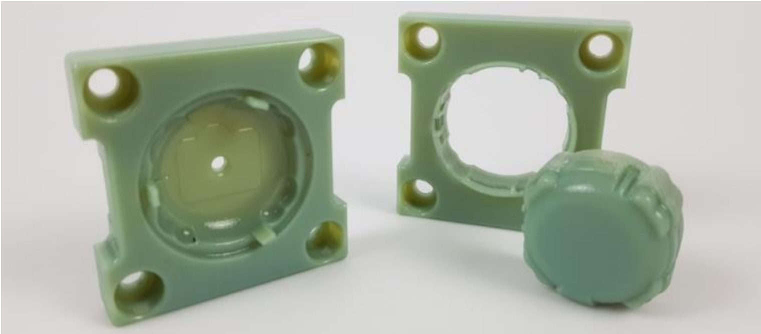


Figure 7.18 *Cable-lid* mold used during the material tests, here shown in Digital ABS.

7.3.3 LDPE

7.3.3.1 Snap-hitch

The scoring for each mold material from the *Snap-Hitch* test done with LDPE can be seen in Table 7.1. All materials were able to make at least three details before showing visible melt damages. PET-G, Visijet M2R-WT and AduraX were the ones to first show signs of deformation. Only Polyamide and Digital-ABS were able to do five shots in LDPE with no visible damages. Onyx performed third best but did show minor signs on damages on the fifth shot. Kynar720 and Aluminum were not able to be tested for LDPE because of delays in delivery and prioritization for PC-ABS. Koltron could not be tested because the rough surface finish made the mold impossible to be closed.

Table 7.1 Material test *Snap-Hitch* with LDPE

Material	*Method*	*Scoring [1-10]*
PA	SLS	8
ONYX	FDM	7
DIGITAL ABS	MJP	9
PET-G	FDM	5
Visijet M2R-WT	MJP	4
ADURAX	FDM	5
KYNAR720	FDM	N/A
KOLTRON	FDM	N/A
ALUMINIUM	DMLS	N/A

7.3.3.2 Cable-Lid

In the *Cable-Lid* tests with LDPE fewer materials were accessible due to printing related problems and tests were only made with the materials shown in Table 7.2. Digital ABS performed the best followed by PA. Both had no problem making the five shots without any signs of damage. Both the PET-G and Onyx performed poorly since the protruding features of the molds were destroyed after 2 shots.

Table 7.2 Material test *Cable-Lid* with LDPE

Material	Method	Scoring[1-10]
PA	SLS	7
DIGITAL ABS	MJP	9
ONYX	FDM	3
PET-G	FDM	2

7.3.4 PC-ABS

7.3.4.1 Snap-Hitch

The tests with the *Snap-Hitch* in PC-ABS used the same process as the tests done with LDPE. AudraX and PETG were discontinued because they showed little or no ability to handle the high temperature. KOLTRON could not be tested since the mold had a rough surface which prevented the mold from closing.

From the test it was concluded that only Digital ABS and possibly Aluminum had the potential to handle the high temperature of PC-ABS. The first detail molded in the aluminum mold was however impossible to eject making the Aluminum mold unusable after one shot. Digital ABS on the other hand had no problem regarding ejecting or damages, it did however get some rounded features making the head of the snap-fit much rounder than intended. PA and Visijet were able to produce 2 parts before melting/shattering. The scoring from the tests are shown in Table 7.3.

Table 7.3 Material test *Snap-Hitch* in PC-ABS

Material	***Method***	***Scoring [1-10]***
PA	SLS	4
ONYX	FDM	2
DIGITAL ABS	MJP	7
PET-G	FDM	N/A
Visijet M2R-WT	MJP	3
ADURAX	FDM	N/A
KYNAR720	FDM	1
KOLTRON	FDM	N/A
ALUMINIUM	FDM	5

7.3.4.2 Cable-Lid

Since only Digital ABS, Aluminum and possibly PA showed potential handling the temperature of PC-ABS and since no aluminum mold for the *Cable-Lid* was available, only Digital ABS and PA were tested. Digital ABS showed good results and had no problem creating 5 parts without showing any signs of damage. The PA-mold fused together with the injected PC-ABS, making ejection impossible after one shot. Scoring can be seen in Table 7.4.

Table 7.4 Material test *Cable-Lid* in PC-ABS

Material	*Method*	*Scoring [1-10]*
PA	SLS	2
DIGITAL ABS	MJP	7

7.3.5 Evaluation and final selection

Digital ABS outperformed all other materials tested as it was the only material capable of the high temperature injection molding of PC-ABS as can be seen in Figure 7.19-7.20. It was also deemed to be the best mold material in regard to ejection, as it allowed for relatively easy ejection of both the *Cable-lid* and the *Snap-hitch.*

PA was the second-best mold material tested as it performed well during the tests with LDPE and only showed small signs of melting during the tests with PC-ABS. This shows a good temperature resistance, with a temperature limit for injection molding between 200 and 260 degrees Celsius. Further testing with other molding materials has to be conducted to establish the exact limit.

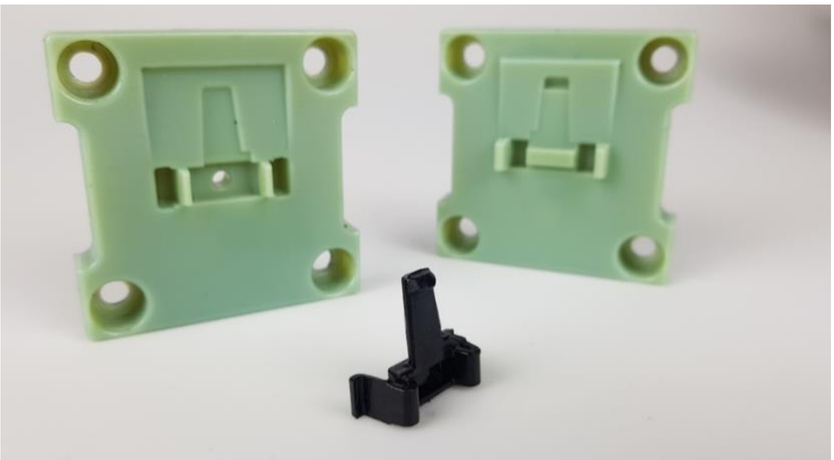


Figure 7.19 Complete injection molded *Snap-Hitch* in PC-ABS

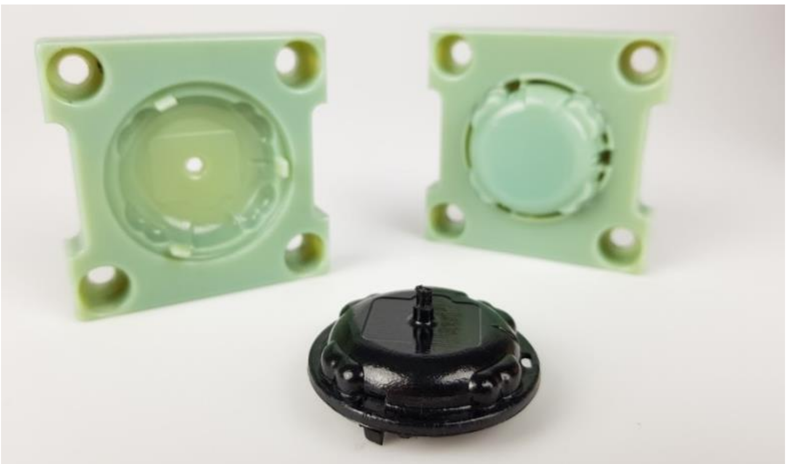


Figure 7.20 Complete injection molded *Cable-Lid* in PC-ABS, the spruce and the filled top has to be trimmed during post process.

7.4 Result of design concept Testing

The design related concepts that were selected from the tests are presented below. For further instructions on how and when they should be applied, see Appendix E.

7.4.1 Design

Since the design concepts are dependent on the overall shape of the part some concepts are only relevant when encountering certain difficulties. The most common of these difficulties encountered during the thesis was ejection, life span, degree of filling and to a lesser extent cooling.

7.4.1.1 Cooling

During the thesis cooling proved to be a lesser problem than first anticipated. However, when the injection molded part showed tendency to not cool properly the only viable concept to use was deemed to be *Prolonged ejection*.

7.4.1.2 Ejection

If parts are prone to stick to the mold the following concepts were shown to help with the ejection. Use of *Release agent spray*, *Inserts in parts*, *Controlled Flashing* and *Molded Ejection Pins*

7.4.1.3 Life Span

The only concept that was tested and deemed useful from Life Span was *Changeable inserts* which overall extended the life of the molds compared to molds without.

7.4.1.4 Degree of Filling

Degree of filling used concepts of *Larger Gate*, *Venting Holes* and which all was deemed to be useful regarding degree of filling.

7.5 Result of Material testing

Since the material tests included two different injection molding materials the results will be divided into two categories, one for each material. The results will list the materials which was deemed to be able to handle injection molding with the material of the category.

7.5.1 LDPE

The following mold materials could handle injection molding with LDPE at 200° C.

7.5.1.1 Digital-ABS

Molds made from Digital-ABS showed no sign of breaking and produced five parts of good quality with both the snap fitting and the lid. It was the best mold material for injection molding with LDPE.

7.5.1.2 PA

PA produced with an SLS-printer showed no sign of breaking and produced five parts of acceptable quality with both the snap fitting and the lid. The geometry of the parts was correct, but the surface had a rough and coarse texture. It was the second-best mold material for injection molding with LDPE.

7.5.2 PC-ABS

Only one mold material could handle injection molding with PC-ABS at 260° C.

7.5.2.1 Digital-ABS

Molds made from Digital-ABS showed no sign of breaking and produced five parts of good quality with both the *Snap-Hitch* and the *Cable-Lid*. It was the only mold material capable of injection molding with PC-ABS.

8 Discussion

In the following chapter the results, method and end specifications of the thesis will be discussed. The chapter will also include a recommendation of how this prototyping method can be used, equipment needed to implement it and further development and testing that should be conducted.

8.1 Test Results

Different parts have different needs when using injection molding for manufacturing. Certain problems are only occurring for specific geometries and therefore every part to be injection molded needs its own combination of solutions. From the results of the concept tests it was decided that seven design concepts are usable for solving problems occurring during the injection molding process. These problems are categorized into three categories, each describing the problem they solve.

The first four concepts are all related to the ejection of the part from the mold. The concepts *Controlled Flashing, Divided Mold, Release Agent* and *Molded Ejection Pins* were all relevant solutions when ejection of parts from the mold was difficult. The reason for choosing all four in the final selection was that each concept excelled in different ways. For some parts it might only be necessary to use one of the concepts, but other parts might need all four. Every concept also has some negative side effects which might want to be avoided for certain parts. The *Controlled Flashing* concept leaves excess material around the part and will therefore need deflashing, which might impair the geometry. The same problem can occur with the concept *Divided Mold* as small amounts of excess material can cause flashing around the parting line between the insert and the main mold. The concept *Release Agent* cannot be used in parts intended to be taped, glued or painted, as the release agent will stick to the surface of the part even after the injection molding. *Molded Ejection Pins* are naturally occurring as the sprue. This is however only useful if the part sticks to the injection side of the mold when opening the mold. If the part is designed to stick to the ejection side of the mold, ejection pins can be designed into the mold by creating holes in the mold. The negative side to this concept is that the pins will need removal after the injection molding and this will leave marks on the part.

If the mold has small and thin details in its design, it might be susceptible to damage when injection molding. During testing, the *Cable-Lid* had some small protruding features, which tended to break off when using most plastic materials. This, however, never happened when using Digital-ABS or PA, but it might become a problem if the features get thin enough. The concept *Changeable Inserts* is therefore included as a solution to this problem.
As the parts being injection molded get bigger and more complex in their geometry, a 100 percent degree of filling will be harder to achieve. This was evident in the two parts used in this project, with the lid being harder to injection mold compared to the snap fitting. Different sized gates were experimented with and the larger gates outperformed the smaller sized. Therefore, as large a gate as possible should be used as this allows complete filling of the mold. The negative side to using a big gate is that it will leave a bigger mark on the part after the injection molding.

If the larger gate is not enough to achieve a 100 percent degree of filling the concept *Venting Holes* can be used. This concept is especially effective for making the molten plastic reach all the way into the small and narrow cavities in the mold, but has the drawback of creating flashings and significantly increasing complexity of the molds.

8.2 End Specifications

The following table shows the target specifications and their end values. This makes the evaluation of the result clearer and easier to comprehend.

Table 8.1 End Specification

Number	***Metric***	***Unit***	***Importance factor***	***Margin Value***	***Ideal Value***	***End Value***
1	Mold material resistant to molten plastic	°C	5	250	>300	>260
2	Size of molded detail	mm	3	45x45x45	>150x150x150	45x45x45
3	Process time	Days	5	<10	0	3*
4	Functioning snap fittings	Binary	5	Yes	Yes	Yes
5	Glueable surface	Binary	2	No	Yes	No**
6	Tapeable surface	Binary	2	No	Yes	No**
7	Surface finish	Subj.	3	-	-	Good enough
8	Geometry	Subj.	4	-	-	Good enough
9	Number of shots	Nbr	4	5	∞	>5
10	Non-damaging ejection	Binary	5	Yes	Yes	Yes
11	Temperature at ejection	°C	5	<Tglasstransition	20	20-30

*Calculated with one day for designing the mold, one day for printing and one day for injection molding. No regard taken for potential delivery times and such.
**No only if release agent is used. Yes if no release agent is used.

Since the mold material Digital-ABS performed well and showed no sign of damage when injection molding at 260° C, the end value was set at 260° C or above. The tests with PC-ABS at 260° C were the highest temperature tests performed in this project. More tests must be conducted to reach the real top temperature.

The size of the mold was limited in this project based on the injection molder used. Larger sizes can be created using a different machine. Because of this the real top limit is not defined in this project.

The process time, if all the machines are available and located close to each other, will be dependent on the time consumption for the design of the mold. If the designing consumes one day, the whole process will take a maximum of three days.

The snap fits created in PC-ABS with the mold in Digital-ABS were functioning. They were not as perfect as the snap fits on the end product, but they snapped into place like they should and stayed there. However, they could not hold as much force as the originals.

The surface of the parts will not be glueable or tapeable if a release agent is used. Depending on the geometry of the part created release agents will not always have to be used. The part *Snap-Hitch* was however possible to make using no release agent.

The surface finish of the parts created replicates the surface of the mold. The surface in the molds made from Digital-ABS have a shiny and almost even finish. It had the best surface finish of all the mold materials tested, but it is still not as good as the surface finish found in parts injection molded in metal molds.

Since the mold made from Digital-ABS showed no signs of damage after five shots it was concluded that the molds can handle a minimum of 5 shots. As there was no sign of damage the top limit for the amount of shots possible is not defined in this project, but interviews and research indicate that it is a much greater number.

When using the right combination of ejection concepts all the parts were relatively easy to eject from the molds made from Digital-ABS, which was the best mold material for ejection tested. The ejected parts were just slightly warmer than room temperature and therefore easy to handle.

8.3 Recommendation to Axis

8.3.1 What this method can be used for

In conclusion from the test and interviews at Axis the best applications for 3D-Printed injection molds are making prototypes for early construction, heat and impact tests.

Since the dimensions of the molds will be the same or worse compared to its 3D-Printed counter parts they, will not in any large extent replace 3D-Prints when making build up test. They do, however, have an advantage when creating parts that relies on the material properties to function. This includes features like snap-fits, which underperform in 3D-printed prototypes, and other mechanical designs where high stress and tensions will occur. They can thus be used in earlier stages to confirm that the part functions as predicted and can in some ways be a method between 3D-Printed prototypes and injection molding using soft tools. They have the geometry of 3D-printed details, but with the material of injection molded details.

Another useful application for molding in 3D-Printed cavities might be for making soft prototyping parts in materials like TPE or silicone to be used in parts like gaskets. The reasoning for this is that these soft materials are at the moment hard to replicate using 3D-printing. This applications and parts have however not been tested in this project since it was decided at an earlier stage to only focus on injection molding and not silicon molding.

8.3.2 Time

When the process is set up correctly and no obstacle arise the process can be done in 2-3 days from CAD-model to finished mold. Since this is a best-case scenario a more likely production rate would therefore likely be 1-2 weeks if done at Axis. For more information regarding the process of the method it is explained in more detail in Appendix F.

8.3.3 Injection Material

From discussions with engineers and material experts at Axis it was concluded that this concept, to be useful, preferably needs to use the same injection material as the final product. Therefore, testing was performed using PC-ABS, a common material in Axis' products, which is injection molded at a relatively high temperature of 260 °C. Because of this it is assumed that this prototyping method can be used with many more injection materials, which are injected at temperatures below 260 °C. However, to be sure about a material's compatibility it needs to be tested.

Many injection materials used at Axis are fiber reinforced, which is known to create damage to molds. Since no fiber reinforced materials were tested during the project, further testing needs to be performed in order to assess these materials compatibility.

8.3.4 Mold material

The most viable alternative shown, both during the tests and external analysis, is to print the molds using a high-end thermoset plastic using the techniques MJ or SLA. During this project Digital ABS was used successfully with LDPE and PC-ABS and is therefore deemed to be a good option for this prototyping method.

From the research conducted during the project it was discovered that other materials such as Visijet M3-X and Somos Perform also should perform similarly. Since these materials, because of availability, were not tested during this thesis further investigation and testing is needed if Digital ABS is not to be used for various reasons.

8.3.5 3D-Printer

The testing showed that thermoset plastic printers are the best for injection molding in high temperature materials. In Table 8.2 materials and their corresponding printers are shown. Digital ABS is listed first as it was the best material tested during the project, but also included are the materials and machines that probably will achieve similar results. These were however not tested during the project and further research and testing needs to be conducted.

Table 8.2 Materials and printers

Material	***Company***	***Method***	***Printer***
Digital ABS	Stratasys	MJ	Objet260/350/500 Connex3™ Stratasys J735™ Stratasys J750™ Objet1000 Plus™ (Stratasys 2018)
Visijet M3X	3D System	MJ	ProJet MJP 3600 ProJet MJP 3600 Max (3DSystem 2018)
Somos Perform	DSM	SLA	General 355 & 365nm SLA machines (DSM 2015)

8.3.6 Injection Molding machine

Two different types of machines have been seen when using this technique elsewhere. Some makers use smaller benchtop machines like the one shown in Figure 8.1 and others instead uses smaller industrial machines similar to the one seen in Figure 8.2. The benchtop alternative will be cheap and simple to use, making the process more accessible. The larger one will on the other hand be more demanding and expensive, but also more exact and versatile regarding parameters and therefore show a result closer to mass produced products.

Since almost all high-end plastics are hygroscopic, a way to dry the granulate is required. In this thesis an industrial oven was used in order to dry the granulate, but if the process is taken into use it is highly advised to use a designated graulate dryer. After discussions with material experts at Axis it was concluded that this granulate dryer should be a dry air dryer.

Figure 8.1 A Galomb benchtop injection molding machine (injectionmolder.net).

Figure 8.2 A smaller Boy industrial injection molder (BOY machines inc.).

8.4 Conclusion

The purpose of this project was to investigate the possibility of 3D-printing injection molds and to create a process in which this concept is used as an alternative to 3D-printing during prototyping. Both objectives were achieved as it was proven during tests that high temperature injection molding was possible using 3D-printed injection molds and a recommendation for how to use and how to solve problems related to using additive manufacturing with injection molding was created. The results show a material suitable for creation of the molds and further defines solutions to problems occurring during the process.

The potential usefulness of this concept for Axis is covered and its placed between 3D-printing and conventional injection molding. There are still many areas to explore within the subject of 3D-printed injection molds and recommendations of where to focus further development have been presented.

Reference List

3D Systems (2018). *MultiJet Plastic Printers Functional precision plastic parts with ProJet® MJP 3D printers*. [online] Available at: https://www.3dsystems.com/sites/default/files/2018-11/3d-systems-proJet-mjp-3600-plastic-tech-specs-a4-us-2018-11-08-web.pdf [Accessed 8 Jan. 2020].

3D Systems. (2020). *Bi-Link Takes the No-Compromise Road to Producing 3D Printed Injection Molds and Parts | 3D Systems*. [online] Available at: https://www.3dsystems.com/learning-center/case-studies/bi-link-takes-no-compromise-road-producing-3d-printed-injection-molds [Accessed 7 Jan. 2020].

BOY machines inc. (n.d.). *Dr. Boy Spritzgiessautomaten: BOY Injection Moulding Machines*. [online] Available at: https://www.dr-boy.de/en/products/boy-injection-moulding-machines/boy-25-e/ [Accessed 8 Jan. 2020].

Bruder, U. (2014). *User's guide to plastic*. Karlskrona: Bruder Consulting.

Callister, W. and Rethwisch, D. (2015). *Materials science and engineering*. Hoboken, New Jersey: John Wiley & Sons.

Design Council (2007). *11 Lessons: managing design in global brands.* London: Design Council.https://www.designcouncil.org.uk/resources/report/11-lessons-managing-design-global-brands

Diegel, O., Nordin, A. and Motte, D. (n.d.). *A practical guide to design for additive manufacturing*. Lund: Springer.

Dsm (2015). *Somos® PerFORM*. [online] Available at: https://www.dsm.com/content/dam/dsm/somos/en_US/documents/Brand-Status-Sell-Sheets/English-Letter/Somos%20PerFORM%20SS-PDS%20Letter.pdf [Accessed 8 Jan. 2020].

Fink, J. K. (2013). *Reactive Polymers Fundamentals and Applications*. 2nd ed. Oxford: Elsevier.

Formlabs (2016). *Injection Molding from 3D Printed Molds - A study of low-volume production of small plastic parts*.

Injectionmolder.net. (n.d.). *Injectionmolder.net - HOME PAGE*. [online] Available at: https://www.injectionmolder.net/ [Accessed 8 Jan. 2020].

IUPAC (2004). *Pure and Applied Chemistry 76*. North Carolina: IUPAC. https://www.degruyter.com/downloadpdf/j/pac.2004.76.issue-4/pac200476040889/pac200476040889.pdf

P3d-prim.com. (n.d.). *PRIM Specifications – P3D*. [online] Available at: https://www.p3d-prim.com/en/prim-specifications/ [Accessed 7 Jan. 2020].

Redwood, B., Schöffer, F. and Garret, B. (2018). *The 3D printing handbook*. Amsterdam: Coers & Roest.

Stratasys (2016). *PolyJet For Injection Molding*.

Stratasys (2018). *Digital ABS Plus POLYJET SIMULATED ABS PLASTIC*. [online] Available at: https://www.stratasys.com/materials/search/digital-abs-plus [Accessed 8 Jan. 2020].

Strömvall, H. and Lundh, M. (2019). *Formsprutning - helt enkelt: Ställarhandboken.* Anderstorp: Polymercentrum.

Ulrich, K.T. & Eppinger, S.D. (2012). *Product Design and Development*. 5th ed. New York: McGraw-Hill Education.

Ullman, D. (2010). *The Mechanical Design Process*. 4th ed. New York: Mcgraw-Hill.

Varotsis, A. (n.d.). *3D printed Injection Molds: Materials Compared | 3D Hubs*. [online] 3D Hubs. Available at: https://www.3dhubs.com/knowledge-base/3d-printed-injection-molds-materials-compared/ [Accessed 7 Jan. 2020].

Appendix A Work distribution and time plan

A.1 Work distribution

Both authors study at the Division of Product Development, Department of Design Sciences, Faculty of Engineering LTH, Lund University and therefore possesses similar skills and knowledge going into this thesis. Since both authors grasp of the subject was of similar level, the work distribution was therefore shared equally where both were involved and contributed in all aspects of the thesis.

A.2 Time plan

During the scope of the thesis the time plan was followed relatively well, some deviations did however occur. The original time plan made during week one of the thesis can be seen in Figure A.1 and the actual time spent on activities in Figure A.2.
It was decided to start with the *Concept generation* in an earlier stage since less work was needed on researching molding techniques. This also meant that the prototyping phase could begin much earlier, which made the entire phase to be completed at an earlier date.

The testing of the injection molding machine did take longer than anticipated and more time was needed to get everything right, which postponed the the material purchase.

Since testing of the material became a larger part of the project than anticipated it got an entire block in the time plan. New sub categories including ordering, testing and summarizing the different molds and mold materials was added. This new phase was taking a longtime to complete because of delays in delivery and machine breakdown.

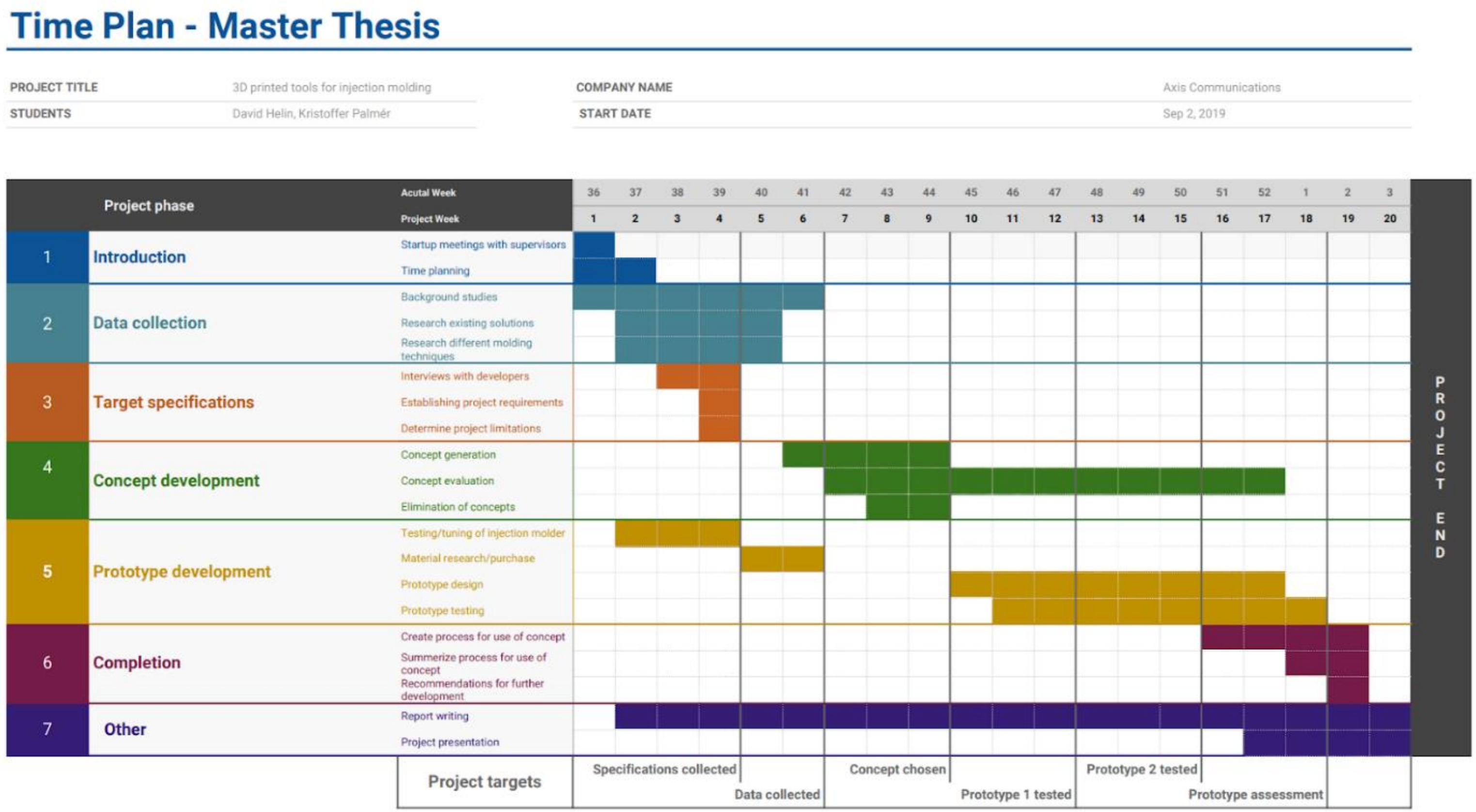


Figure A.1 Original time plan.

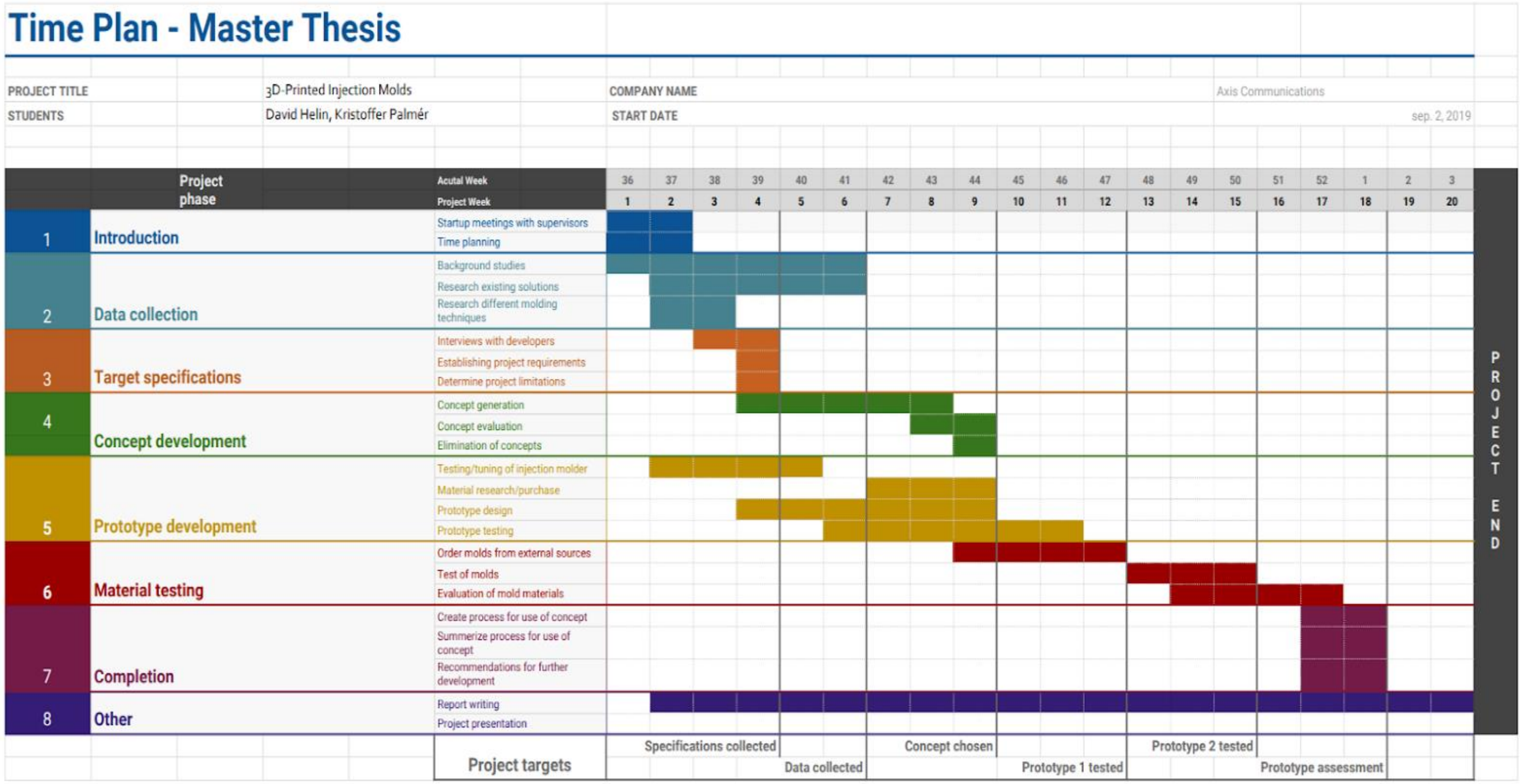


Figure A.2 Actual time plan.

APPENDIX B - Interview questions for engineers at Axis

All the questions asked during the interviews conducted at Axis are shown in Table B.1. When asking question 9 the illustration shown in Figure B.1 was presented.

Table B.1 Questions asked during interviews at Axis.

Number	*Questions*
1	How many physical prototypes do you make per year?
2	What technique is used making these?
3	What benefit would you have from having an injection molded detail versus a 3D printed in the prototype stage?
4	What tests do you usually do on new details? What works poorly with the 3D printed ones?
5	What materials do you primarily work with? What other materials are important to do tests on?
6	How long does it take to get a maximum injection molded detail?
7	For how many details do you use soft tool?
8	How important is it to injection mold in the same material in which the product is to be manufactured?
9	What size would the majority of your details fit in?
10	What would be the maximum time consumption on getting injection molded prototypes?
11	How important is it to injection mold in the same material in which the product is to be manufactured?

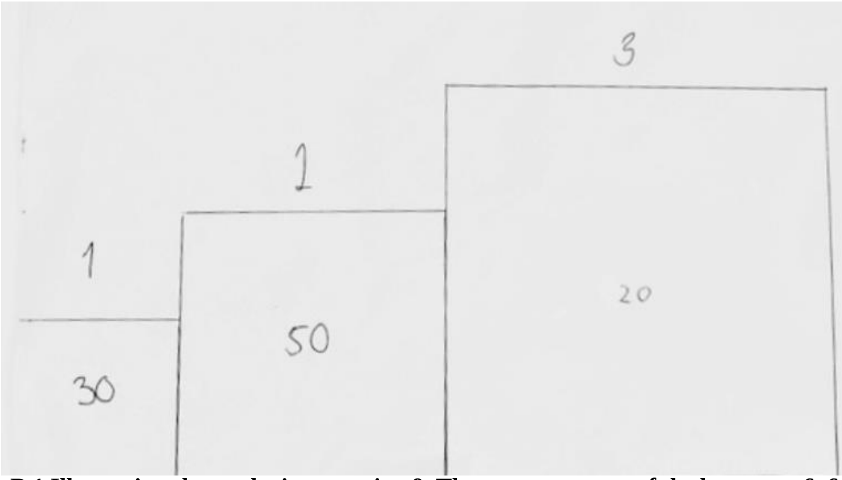


Figure B.1 Illustration shown during question 9. The measurements of the boxes are 6x6, 10x10 and 15x15 cm.

Appendix C -Interview questions for prototype makers

The questions asked during the interviews with the prototype makers are shown in Table C.1.

Table C.1 Questions during interviews with prototype makers.

Number	***Questions***
1	Have you tested / heard of 3d printing injection molds?
2	If tested, what type of material did you in that case use?
3	What materials and methods do you think would be appropriate to make 3d printed molding templates?
4	What materials and 3D printers do you have access to?
5	Do you have any other ideas on what could work to make injection molded prototypes?

Appendix D

Every mold used during the material tests are shown in Figure D.1-14

Figure D.1 *Snap-Hitch* mold in Digital ABS

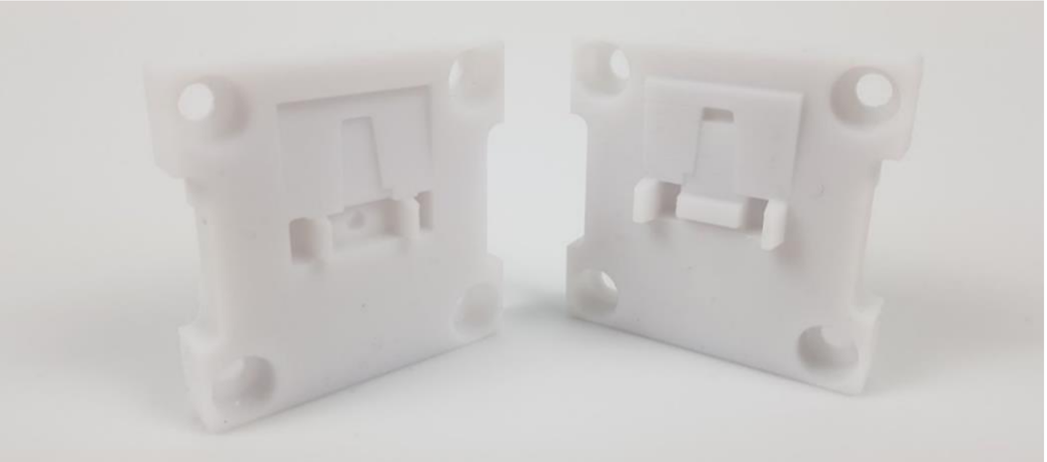


Figure D.2 *Snap-Hitch* mold in M2R-WT

Figure D.3 ***Snap-Hitch*** **mold in ONYX**

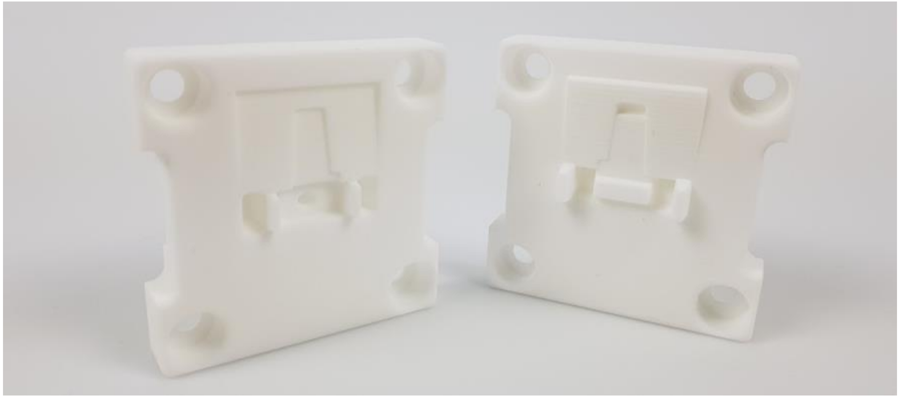


Figure D.4 ***Snap-Hitch*** **mold in PA**

Figure D.5 ***Snap-Hitch*** **mold in PET-G**

Figure D.6 *Snap-Hitch* mold in Koltron

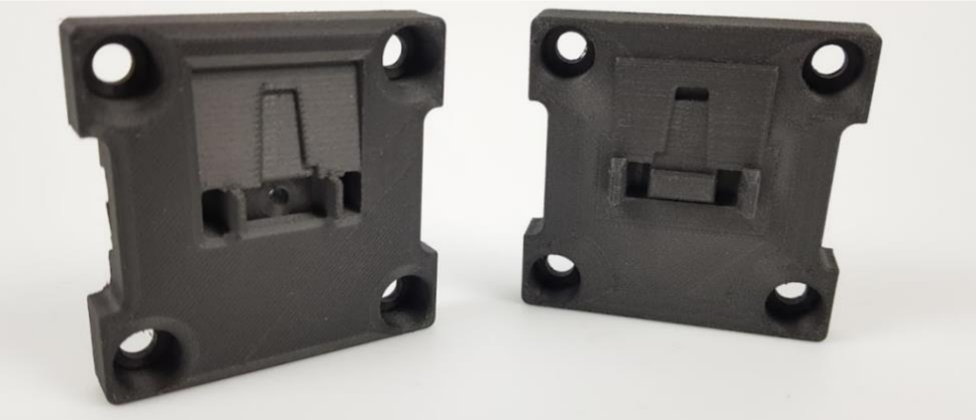


Figure D.7 *Snap-Hitch* mold in AuduraX

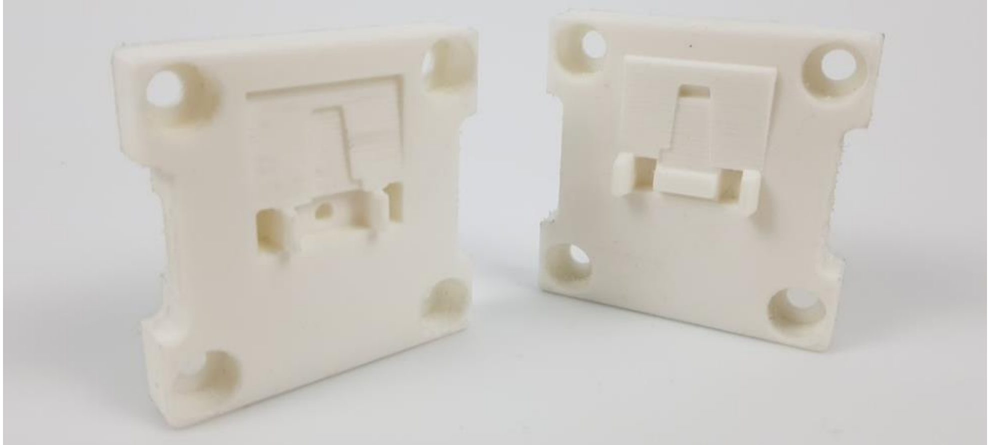


Figure D.8 *Snap-Hitch* mold in Adamant.

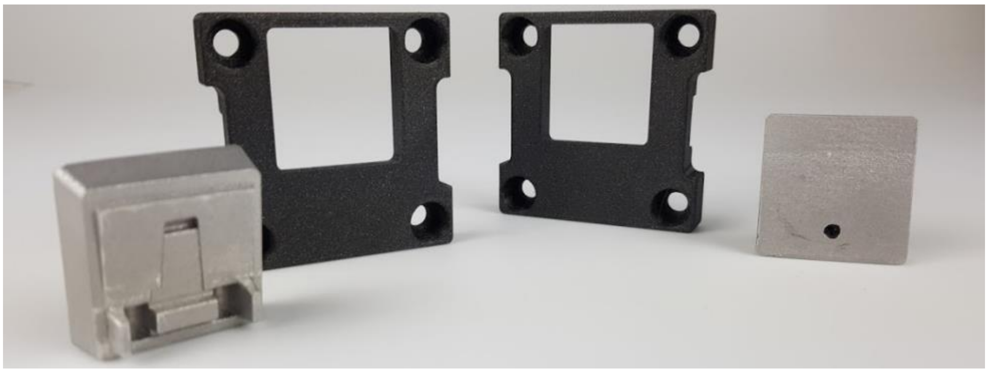


Figure D.9 ***Snap-Hitch*** **mold in Aluminium**

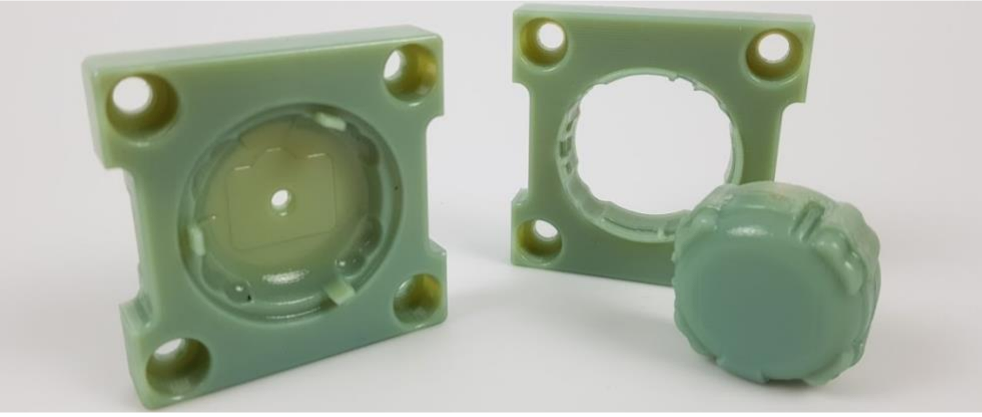


Figure D.10 ***Cable-Lid*** **mold in Digital ABS**

Figure D.11 ***Cable-Lid*** **mold in PET-G, here shown in the version with exchangeable pins.**

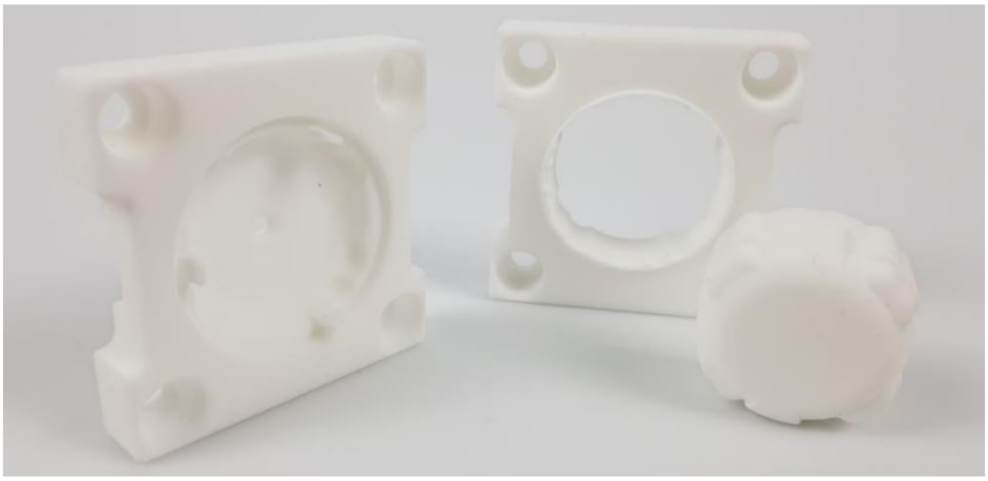


Figure D.12 *Cable-Lid* mold in PA.

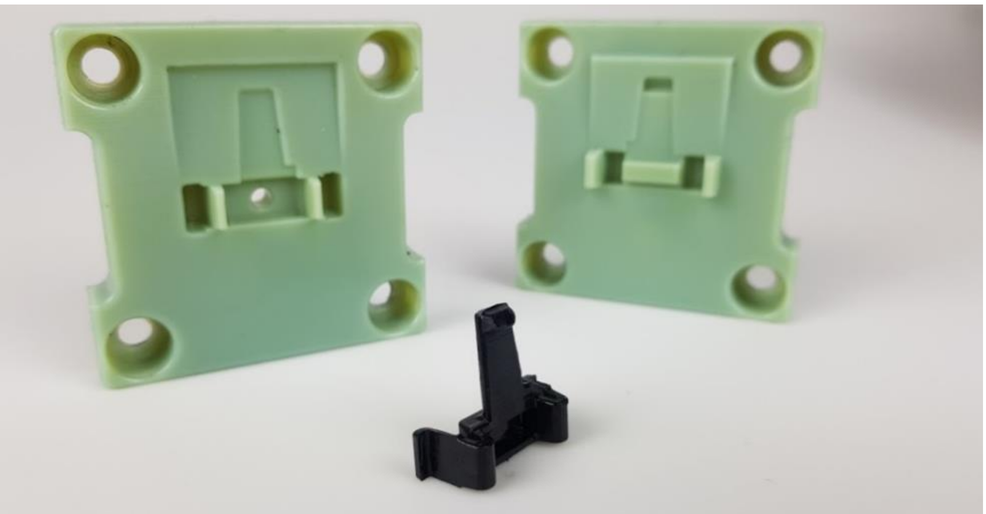


Figure D.13 Finished detail of the *Snap-Hitch* made in a Digital ABS mold.

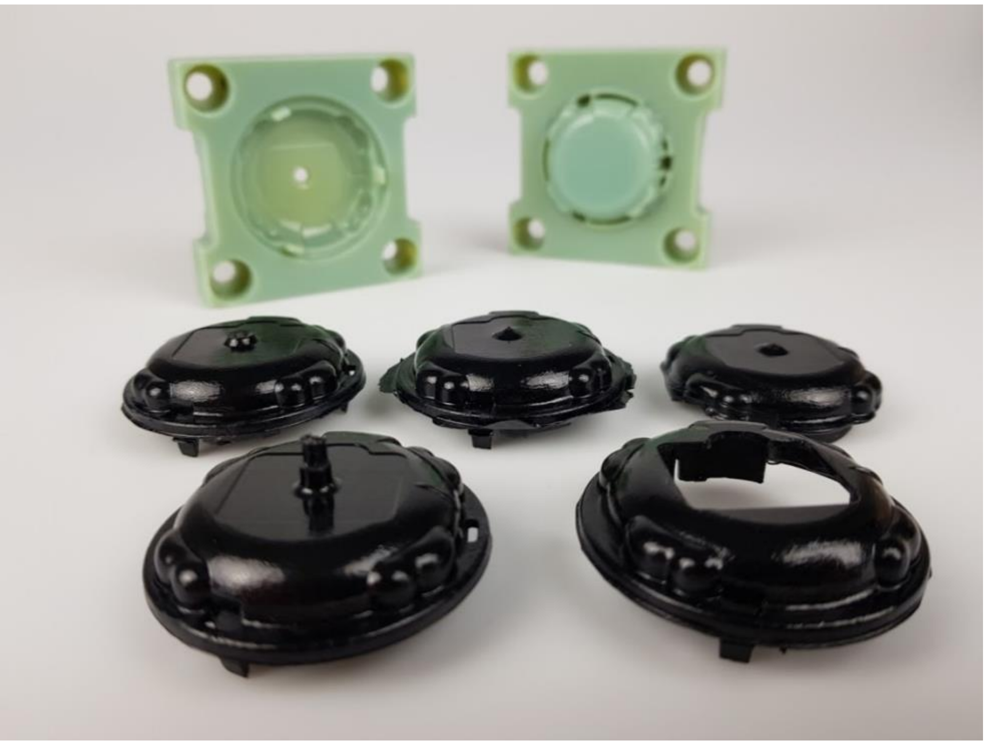


Figure D.14 Finished details of the *Cable-Lid* made in a Digital ABS mold.

Appendix E

Every design concept selected from the testing phase is shown here in this appendix. It will include instructions on how and when to use them.

E.1 Prolonged Ejection

Prolonged ejection is a concept based around increasing the clamping time of the mold tool after injection, that is to say increase the cycle time. This increased time gives the injected molded detail time to cool down and solidify.

E.2 Controlled flashing

By making a small gap between the two mold halves excess material in the form of flashing can be created. This flashing can later be used as a disposable leverage when pulling out the molded detail making the ejection more controllable and reducing the risk of damage, once ejected the flashing can be trimmed.

E.3 Divided Mold

Divided tool is a concept for ejecting parts by dividing up the mold into smaller subunits. The subunits are assembled before and then disassembled after the injection has taken place. The division can be done in several different ways by either just removing smaller features or dividing the entire mold in half.

E.4 Release Agent Spray

Release agent spray is used for facilitate ejection. By liberally coating the molds with the spray prior to injection and then reapplying between injection or when needed, details that otherwise would to stick to the mold can more easily be removed.

E.5 Molded injection pins

Molded injection pins is a concept were minor holes through the mold is made. During injection will these holes fill up and become part of the molded detail. Once solidified, the mold is removed from the machine, turned over and the detail together with the molded pins pushed out using small metal rods and blunt force shown in Figure F.1.

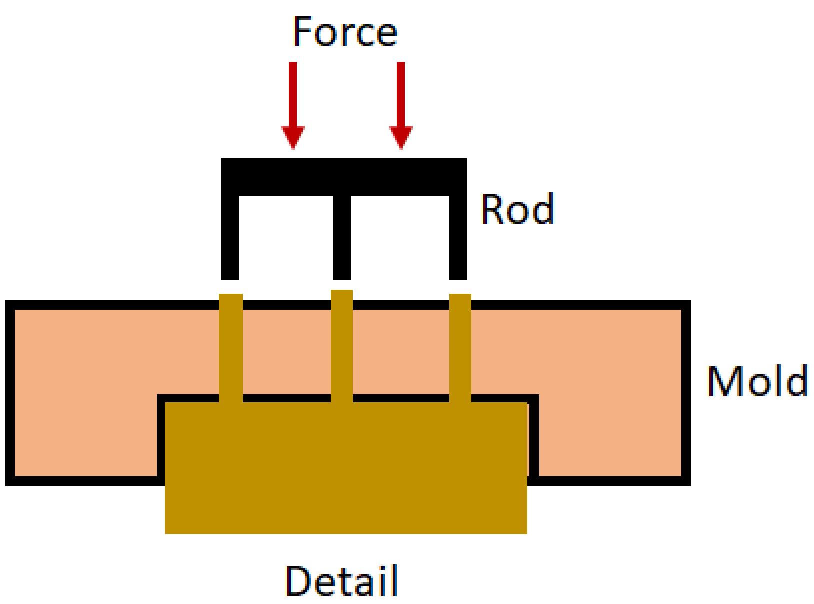
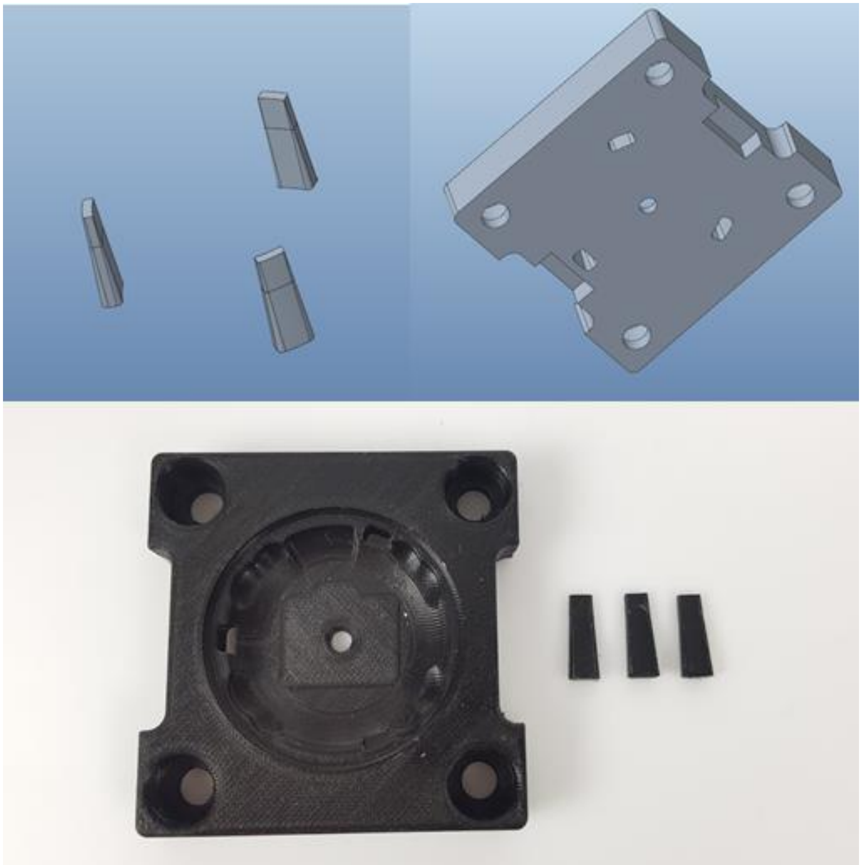


Figure F.1 Concept *Molded injection pins*.

E.6 Changeable inserts

Changeable inserts is a concept used for prolonging the life span of the molds. Small features have a tendency to get destroyed or deformed during injection. One way to increase their life span is have these features be changeable by making them as separate pieces as is shown in Figure F.2.



Figur F.2 Cad model and printed PETG version with the concept: *Changeable inserts*. The pins as seen in the top left slides in on the backside of the mold seen in the top right.

E.7 Larger Gate

Larger Gate is a concept used for increasing the degree of filling in the molds. By increasing the gate size up to 6 mm in diameter compared to the traditional gate size of approximately 1mm a better degree of filling can be achieved.

E.8 Venting holes

Venting holes is a concept used for increasing the degree of filling in the molds. When melt is being injected air pockets in mold can prevent the details from becoming completely filled, this can be prevented by introducing very small holes in the molds as seen in Figure F.3.

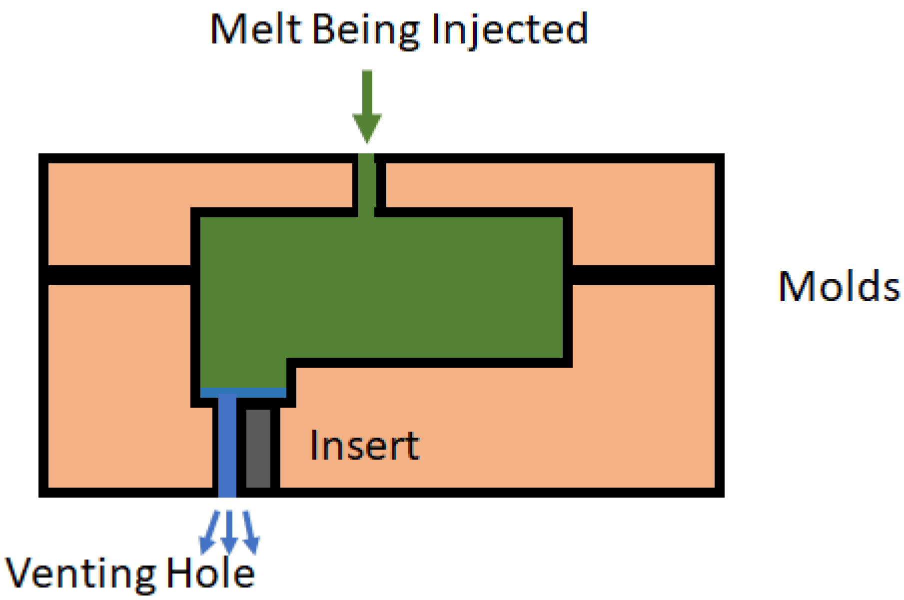


Figure F.3 Concept *Venting holes*

Appendix F

In this appendix a step by step guide for the process is presented.

F.1 Process steps

The process of creating a prototype with a 3D-printed injection mold can be summarized into five steps, shown in Figure F.1. This appendix will go through these five steps.

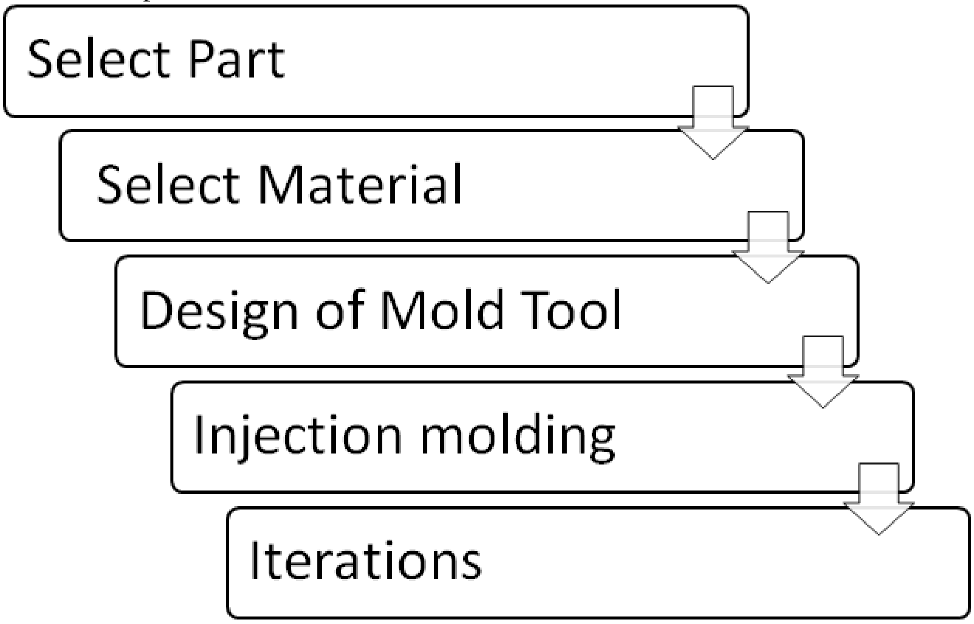


Figure F.1 The five steps of the prototyping process

F.1.1 Select Part

Make sure that the selected part is able to be manufactured. The normal rules such as draft angles, undercuts and none sharp corners are important just as in standard injection molding. It is however suggested to increase draft angels to at least 4 degrees and to make holes larger than 1 mm.

Also make sure that the part selected is able to be manufactured regarding size and complexity. Larger and very complex part that need a lot of pressure to be produced increases the risk of destroying the molds and are not suitable for injection molding in additive manufactured molds.

F.1.2 Select material for injection molding.

When using this method some materials are more suitable than others to be injection molded, the material also affect the life of the mold were easy materials such as TPE PP and LDPE can do several hundreds of shoots before depleting. This compared to PC and fiber filled materials that according to most sources only are able to withstand a couple of shots before depleting. A company called P3D have done test regarding Digital ABS and how many shots one mold can handle in different materials shown in Figure F.2. (P3D)

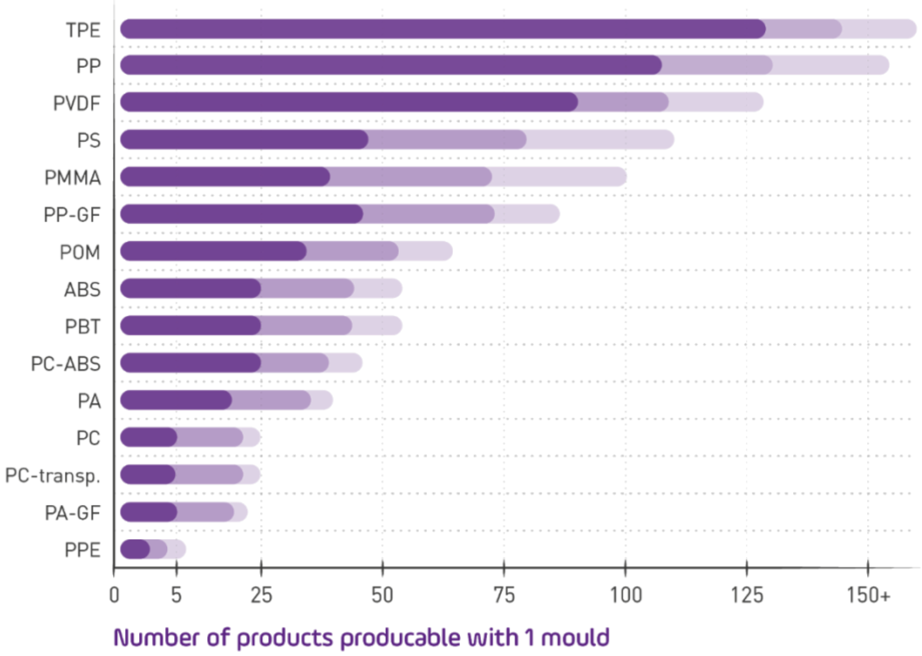


Figure F.2 Estimated number of shots a Digital ABS mold can handle using different injection materials (P3D)

F.1.3 Design of mold

When creating molds in plastic it is important to remember that plastics acts differently than metal and following design decisions will make injection molding easier.

- Use a larger extrusion gates in molds, preferably 3-6 mm
- Use draft angles of approximately 4 degrees.
- When creating demanding parts is it possible to make changeable inserts that once damaged can be replaced.
- Make sure to vent holes and cavities that show tendency to trap air. These venting holes are preferably made using sliding inserts.
- When using very demanding plastics, especially fiber filled ones is it highly suggested to use metal inserts at places where wear will occur
- Make sure to take shrinking of plastics into account.
- See to that all cavities can be cleaned from dirt and support/excess material.

- Use molds that can be divided in critical places, this will result in both easier, cleaning ejection and venting of the part.
- If using a machine with ejection pins is it required to either take these into account when designing the tool, or to drill these out once the mold is printed.

F.1.3.1 CAD

The mold is preferably made by using a designated mold program such as CREO mold-tool. This can however be circumvented by using boolean cut and merge features available in CREO assembly. Since all parts differs in features and design, a case to case approach must be made. Try to make all important design decisions regarding the mold to resemble how the real mold would be made. In other words, placing the injection gate and parting line in the right places.

F.1.3.2 Manufacturing of mold

Depending on what 3d printer used for making the parts its important to place the part to maximize the details and strength while minimizing support material around holes and fine details. Some printing methods can distort the dimensions slightly making the molds harder to close and increased tolerance can therefore be needed.

F. 1.4 Injection molding

First step of injection molding is to dry the granulate, which will need to be done for most plastic materials except PE and PP. The drying can be done using most dryers, but the best result will be using a granulate dryer.

If injection molding a complex part is if highly suggested to precoat the mold using release agent spray and continue using this between shots.

Purge the machine from old material and refill with the newly dried granulate.

Insert the mold and do some test shots, the first shots usually end up bad since the mold is cold and the parameters from the machine needs to be optimized regarding pressure, shot volume and temperature.

The parameters that differs regarding injection molding in plastic 3D-Printed molds compared to standard metal molds are primarily the clamping time and the holding pressure. A prolonged clamping time is needed in order to let the melt cool and solidify. The holding pressure on the other hand needs to be kept at a minimum since it otherwise might crack the mold.

If molds do not get a complete degree of filling is it recommended to increase the shot volume, pressure and/or temperature.
If these alternatives do not work or they have started to affect the molten plastic or mold tool iterations of the tool will be needed.

If details have problem getting ejected without damage the first thing to try is to increase the amount of release agent spray and decrease clamping time. If this doesn't help some iterations to the tool will be needed.

F.1.5 Iterations

F.1.5.1 Degree if filling

If problems with the degree of filling is occurring some alternative to try is to increase the gate size, erase difficult features, increase draft angles and or create more elaborate venting holes.

F.1.5.2 Ejection

If parts will not eject properly the first thing to check is if draft angles can be increased or if clamping caused by shrinking can be avoided. If not, one alternative is to divide the mold half were the detail gets stuck in such a way that it can be picked out without deforming see Concept 4: *Divided Molds* in Chapter 6.2.4 Concept generation and Selection for inspiration. It's also possible to design features into the mold were excess material can flow that can later be used as handles for getting disposable leverage when removing the detail. This can be done by increasing the amount of flashing in detail see Concept 5: *Controlled flashing* and Concept 1: *Molded injection pins* in Chapter 6: Concept Generation and Selection and Chapter 7- Concept tests

Reference List Appendix F

P3d-prim.com. (n.d.). *PRIM Specifications – P3D*. [online] Available at: https://www.p3d-prim.com/en/prim-specifications/ [Accessed 7 Jan. 2020].