DEVELOPMENT OF A SILICONE MOLD TOOL FOR INJECTION MOLDING PLASTIC PARTS

A THESIS

PRESENTED IN PARTIAL FULFILLMENT OF THE REQUIREMENT FOR THE DEGREE OF MASTER OF SCIENCE IN MECHANICAL ENGINEERING
UNIVERSITY OF MINNESOTA

BY

IRFAN TAHIR

ADVISOR
DR. VENKATA GIREESH MENTA

MAY 2020

Abstract

Injection molding is one of the most popular processing methods for manufacturing plastic parts. Typically, injection mold tools are made out of metal. The design and development of these metallic mold tools is a very expensive and lengthy process which means that it is difficult to incorporate this process into the prototyping stage of a product. Currently, the most widely researched method used for rapid prototyping of injection mold tools is additive manufacturing (AM). This project investigates an alternative to AM as a rapid prototyping method by investigating a cost-effective mold tool made out of silicone. A robust step by step process of creating a silicone mold tool is presented. To determine the right plastic to inject into the silicone mold tool, an injection molding simulation is conducted comparing three types of plastics and their effect on the filling of the mold tool. Following the simulation, Design of Experiment (DOE) is used to measure the main and interaction effects of the silicone mold tool's durometer hardness, geometry, and design complexity on its performance. Additional DOE studies were conducted to optimize the injection molding processing parameters for fabricating ASTM D638 Type IV tensile specimens. From the experiments, it was found that a durometer of Shore A Hardness 40 is the most optimum value for a silicone mold tool. Durometers smaller than that increase the likelihood of failure by flash and durometers larger than that damage the mold tool through brittle failure. Design changes were made to the mold tool geometry to use 3D printed inserts and shorten the length of the runner, the latter of which resulted in ideal samples without any failures. Comparison of mechanical properties of the silicone mold test coupons with those produced using a metallic mold tool revealed that there was a 7.3% decrease in Ultimate Tensile Strength when going from metal to silicone mold tool, better than those previously reported for some AM mold tools. In conclusion, the silicone mold tool is a promising alternative to AM mold tools for rapid prototyping of injection molded parts with certain limitations.

Table of Contents

List of Tables

List of Figures

CHAPTER 1
INTRODUCTION

Injection molding (IM) is a widely used polymer processing technique for fabricating plastic parts of different shapes and profiles. The main working principle of the process involves injecting molten material into a mold tool where it cools and hardens to take the shape of the mold. Due to the ability to mass-produce at high speeds, high accuracy for complicated shapes and lower mass production costs, the IM process has become one of the most critical processing methods in the polymer industry [1]. In addition to this, plastic parts produced via injection molding offer excellent repeatability [2]. Some of the products manufactured using injection molding are bottles, toys, automotive components, storage containers, and medical device components.

One of the major limitations of the IM process is high initial start-up costs during the design and development stage of a product. These high costs and large lead times mean that IM process technology is suitable only for large scale production for a minimum of 1000 parts, with the added pressure on the manufacturer of getting the mold tool right the first time [3]. Any design improvements or modifications of the part at later stages is quite difficult and expensive. Overall, the design of the mold tool is a time-consuming and expensive stage in the product life cycle. Hence, injection molding is typically avoided in making prototypes [4].

Rapid prototyping (RP) is a well-known process that has come to light with the advancements in additive manufacturing (AM) and lower cost associated with Computer Numerical Control (CNC) machining. The molds that are created using RP are created with shorter lead time and at a fraction of the cost when compared to traditional tooling. The decreased lead time for tooling allows for more time to modify or test the part and allows more design iterations. The RP technology is very useful in the pre-series production: the creation of a part using the same production method that will allow for testing the market before the capital investment is made [5].

The material that is chosen to replace the traditional metallic tooling must be able to withstand the appropriate pressures and temperatures needed to inject the polymer material. Likewise, the mold must be rigid and maintain its shape under the clamping pressure of the injection molding machine, so the part comes out as intended. Moreover, the thermal conductivity of the rapidly prototyped mold tool is different from that of the metallic mold as RP mold tools are made of some sort of a polymer. How fast a thermoplastic cool can affect the structure of the plastic which can translate to a change in the mechanical properties of the injected part. Another issue with this kind of RP is the unavailability of materials for alternative methods like AM, in contrast to IM where there is a very large range of plastic and biodegradable plastic materials easily available.

Starting with the most basic question of whether or not it is possible to inject plastic material into a silicone mold tool, this thesis investigates the development of a mold tool made out of silicone for simple and cost-effective rapid prototyping of injection molded parts. These mold tools have the potential of enabling researchers and materials scientists to add variability to their designs. In contrast to the metallic mold tool, opportunities for silicone material-based mold tools are quite substantial. The most appealing feature of the silicone mold tool is its cost-effectiveness. Manufacturing of a silicone mold tool can be as much as 10 times cheaper than the metallic mold tool's production. The low cost associated with silicone also attracts customers who are not experts in designing a mold tool as the process is relatively simple and uses commercially available materials. This results in faster lead time for the mold tool. An additional advantage of silicone is the wide operating range for high temperature injection runs. For biopolymers such as lignin, alginate, PLA and other "sticky" materials, silicone mold tools can prove to be beneficial as materials tend not to stick to it and there is no need for a mold release agent. This is especially useful for medical device components used in vivo studies.

1.1 Related Work

While considerable work has been done on the rapid prototyping of injection mold tools using AM, not much has been reported in literature on the development of silicone-based mold tools. As AM is the current standard for rapid prototyping of mold tools, it is worth highlighting some of the work that has been done in this area.

In a comparative study of rapid and traditional tooling for plastic injection molding, Mendible et al. reported that additively manufactured molds last between 10 and 500 production runs, depending on the type of AM [6]. This shows that the quality of the mold varies greatly depending on the type of AM technique. Direct metal laser sintering (DMLS) three-dimensional (3D) printing produced excellent results with the molds lasting for 500 production runs whereas parts produced using jetted photopolymer (PolyJet) 3D printing failed after 10 production runs. Damle et al. showed that the mechanical properties of plastic parts injection mold tools produced using Stereolithography (SLA) were in the range of 94% to 98% when compared to metal mold tools [7].

In 2017, Bartlett investigated the effects on the mechanical properties of Polypropylene (PP) plastic parts produced using Fused Deposition Modeling (FDM) with Digital ABS material [8]. The results showed that the Ultimate Tensile Strength (UTS) decreased by 11%, Young’s modulus of the parts increased by 9.4% and strain at break, or ductility of the parts, decreased significantly by 87% when compared to parts made using a metallic mold. Annealing the samples before tensile testing helped increase ductility but it was not as good as steel mold parts. In 2019, Simpson et al. also used Digital ABS with PolyJet 3D printing and investigated the effects of different 3D printing materials and the injection material on the mold tool [9]. It was reported that the strain at break of Digital ABS was consistent with that of the steel mold, but stiffness was reduced. These results are relevant to this project as Simpson et al. used Polypropylene as the injection material for their experiments, the same one that is used in this project.

Another technique for rapid prototyping injection mold tools is to use composites and additively manufactured inserts, also known as steel/plastic hybrid molds. In 1999, Dawson et al. investigated composite mold tools made out of atactic polystyrene and compared the properties of the plastic sample with a traditional steel mold [10]. Results showed that parts produced using a composite mold had 17% lower UTS, similar Young's modulus, 19% higher flexural strength and 20% lower ultimate elongation than parts produced using a steel mold. In 2017, Mischkot et al. used Design of Experiments (DOE) to study a disk-shaped insert made of PolyJet 3D printing and brass [11]. The material that was injected into the mold was Low-Density Polyethylene (LDPE). The authors reported that a cooling time of 50 seconds was necessary to enable the injection of the parts without any major deformations or discoloration. The inserts lasted between 25 and 116 production runs with a cycle time of 300 seconds. The authors concluded that an FDM 3D printed insert is a viable option for medium-sized molds (80 x 60 x 20 mm^3). Mendible et al. suggested that the failure of Polyjet inserts can be minimized by modifying the draft angle, surface finish, and injection pressure [6].

One of the most critical properties to consider is the thermal conductivity of the mold material. Silicone, AM, and metal molds all have a different thermal conductivity. This difference in thermal conductivity means that AM mold tools retain a greater amount of heat when compared to steel or aluminum mold tools which signifies the need for a DOE study to find the right injection molding parameters for a specific mold tool [9]. Kamal et al. showed that changing the processing parameters can result in a dimensional shift of the residual stresses in the injected plastic part [12]. This can lead to a higher level of crystallinity which affects the degree of organization within the material that can have varying effects on the toughness, density, hardness, modulus of elasticity, and yield strength of the injected plastic [13].

Prior studies have shown that conducting simulation for the injection molding process is a challenging task [14] [15]. In 1999, Dupret et al. proposed a numerical method based on front tracking, automatic remeshing, and extrapolation [16]. Using this method, they developed MOLDSYS, one of the first software designed specifically for injection molding simulation. Since then, there has been a number of studies presenting simulation data for all kinds of injection molding domains. In 2011, Guo et al. analyzed the deformation of the mold tool core during the filling stage to study the effects of changing materials and gate design [17]. In 2013, Kim and Lee presented a simulation technique to predict the life of injection mold tools and compared their simulation results with fatigue testing [18]. For their comparative study in 2015, Mendible et al. successfully used Autodesk Simulation Moldflow Insight to compare the surface temperature, shrinkage, flow, and core displacement of FDM 3D printed inserts [6]. With the advancements in commercially available simulation software for injection molding, it is critical that a simulation study is performed before conducting experiments.

Using silicone to cast shapes of various geometry is an age-old technique that has multiple U.S. patents [20]. However, it should be noted that there is almost no literature available for silicone or silicone inserts for the purposes of injection molding. In this regard, the work that is done in this thesis is the first of its kind and can be categorized as novel.

1.2 Thesis Organization

This thesis is organized as follows. Chapter 2 discusses the materials used to create the silicone mold tool and outlines a step by step process of creating one. This chapter also talks about expected deformation of the silicone mold tool during the injection molding process and the approach taken to mitigate those deformations. Chapter 3 outlines the steps for the injection molding simulation its results prior to conducting experiments. Next, Chapter 4 discusses the factors, levels and responses of the Design of Experiment (DOE) studies and analysis of their results. Based on the lessons learned from the DOE studies, Chapter 5 extends those concepts to make further changes in the mold tool’s geometry to obtain a testable sample for tensile testing. Chapter 6 makes a comparison between the mechanical properties of samples obtained using silicone and metal mold tools. The concluding discussion is given in Chapter 7

CHAPTER 2

SILICONE MOLD TOOL PROCESS

An elastomer is a viscoelastic polymer with weak intermolecular forces. Due to these weak intermolecular forces, it generally has a low Young's modulus and a higher deformation rate compared to other materials [20]. A common euphemism in the materials science world for elastomers is "rubber-like" material. Silicone is also an elastomer, and hence exhibits many of the same properties as rubber. Some of the advantages that silicone has over other materials is that it is non-reactive, useful over extreme temperatures (−50 to 295 °C), easily moldable into different shapes and inexpensive when compared to ceramics or metals. Due to these properties, silicone can be found in many industrial and consumer products such as insulation, automotive parts, electronics, medical devices, and food storage [21].

In its native state, silicone is a highly adhesive liquid. To convert it from a liquid to a solid, it must be cured, which is a chemical process involving the cross-linking of the polymer chains. The materials used in this project are cured using a platinum-based and tin-based cure system. In this chemical process, the siloxane polymer reacts with a platinum or tin catalyst, which in turn creates an ethyl bridge between the two. This process is advantageous over other curing methods due to its quick reaction time and the lack of byproducts.

2.1 Durometer Hardness of Silicone

Hardness is the measure of a material's resistance to localized plastic deformation. For certain materials like polymers, rubber and in this case, silicone, an increase in hardness is inversely related to the ductility of the materials [22]. In other words, as the hardness number increases in these materials, they are more prone to brittle failure.

There are various experimental methods to measure a material's hardness value. For materials like silicone rubber, the ASTM D412 standard is followed to calculate a value known as Shore A Hardness or Shore Durometer [23]. This method, like other hardness tests, applies a given force on the materials and measures the depth of the indentation caused by the given force. For accurate measurements, a hardened steel rod of about 1.1 mm in diameter with a truncated 35° cone of 0.79 mm diameter must be used to cause the indentation. Typically, metallic molds made out of aluminum or steel incorporate different methods for measuring the hardness numbers such as the popular Rockwell hardness B scale. Consequently, the indentation depth on metallic molds is much less than on silicone rubber as the former is rigid and inhibits a stark difference in material properties.

2.2 Silicone Mold Tool Materials

The silicone materials used in this project were sourced from the company Smooth-On from their location in Pennsylvania, United States [24]. Specifically, the material is branded under the Mold Star™ and Mold Max™ that comes in two parts. To successfully cure the silicone from liquid to solid, the two parts must be mixed together in a specific ratio[1]. In total, one Mold Star™ silicone of Shore A Durometer Hardness 30 and two Mold Max™ silicone of Shore A Durometer Hardness of 40 and 60 respectively were used in this project. The material properties of all three are listed in Table 1.

[1] Detailed process is presented in section 2.3

Table 1: Material properties of different types of silicone

Silicone	Specific Gravity (kg/cm^3)	Shore A Hardness	Tensile Strength (MPa)	100% Modulus (MPa)	Elongation at Break (%)	Useful Temperature Range (°C)
Mold Star™ 30	1120	30	2.9	0.4	440	-53 to 232
Mold Max™ 40	1140	40	3.8	1.3	250	-53 to 205
Mold Max™ 60	1450	60	2.7	2.3	132	-53 to 294

2.3 Process Flow

The shape chosen for the mold tool was a simple dog bone shape based on the ASTM D638 Type IV standard which is used for tensile testing of plastic materials. The length of the part (115 mm) falls within the boundaries of the injection molding machine available at the university's facilities. A standard runner geometry was used [25].

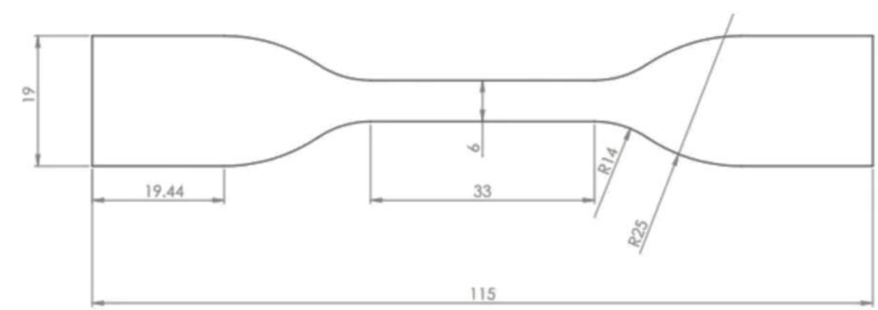


Figure 1: Geometric shape of the tensile specimen. All dimensions are in mm

The flow diagram of the mold making process is shown in Figure 2. The first step in creating a mold tool made out of silicone is designing the mold cast in a Computer-Aided Design (CAD) software with accurate dimensions. After that, the CAD file is saved as a Standard Tessellation Language (STL) file format which is converted into a G-Code using an open source slicing software called Cura. With the G-code ready, now the mold tool cast can be 3D printed. FDM was used to produce all the most tool casts for this project with a Creality Ender 3 printer and Hatchbox PLA as the material. It should be noted that the strength of the mold tool cast is not very important here, as a very low infill (20% or more) can be used to save printing material. The important factor to consider is the part finish as that will be reflected in the silicone mold tool.

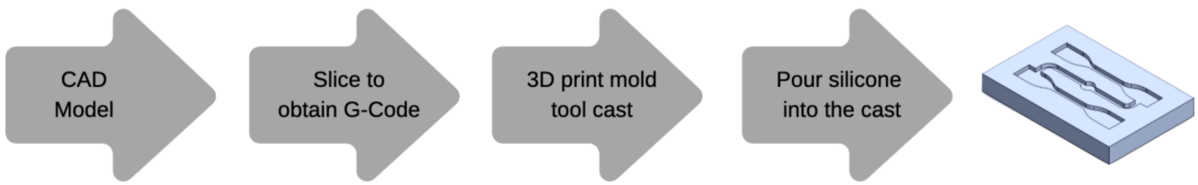


Figure 2: Process flow of making a silicone mold tool

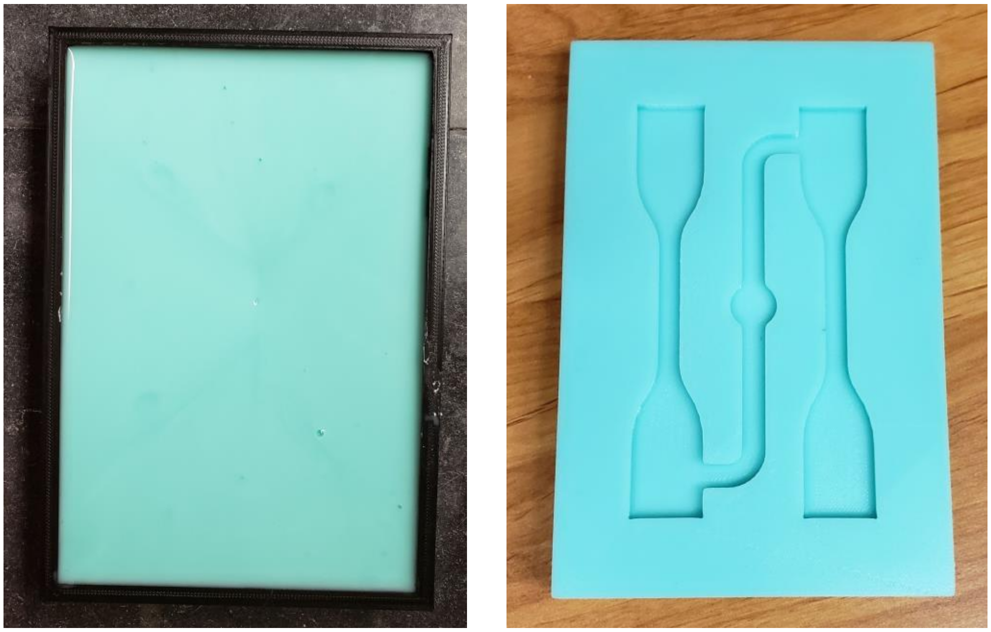
With the FDM 3D printed mold cast ready, the liquid silicone materials can now be poured into the cast. The silicone material used in this project came in two parts which are used to start the curing reaction. For an accurate mold type with minimal inconsistencies, the following steps were followed for each mold:

1. Both Part A and Part B were poured into a measuring cup according to the proportions specified for the specific durometer. For example, in the case of MoldStar^{TM} 40, the proportions were 10 grams of Part A for every 1 gram of Part B. After pouring, both parts were stirred for 180 seconds.
2. To avoid entrapped air and air bubbles in the mold tool, vacuum degassing is recommended. The mixing container was put in a vacuum chamber for 180 seconds (Figure 3).
3. A level tool was used to make sure the surface on which the FDM 3D printed cast is sitting was leveled. Then, the material was slowly poured into the FDM 3D printed cast at a single point. As the silicone material filled the cast, pouring was stopped.

4. The mold was left to cure for exactly 24 hours and then meticulously removed from the FDM 3D printed cast using plastic pry tools.

Figure 3: Equipment used for vacuum degassing of the silicone before pouring into the cast

(a) **(b)**



(c) **(d)**

Figure 4: Making the silicone mold tool: (a) FDM 3D printed mold tool cast (b) Pouring the liquid silicone into the cast (c) Making sure the liquid is level (d) Final shape of the mold after curing

2.4 Metallic Outer Frame

Before doing injection runs using the silicone mold tool, it was predicted that the mold will deform in multiple directions as shown in Figure 5. The way the injection molding machine is set up, the mold tool is constraint only from the top and bottom. This means that the flexible silicone mold tool has room to deform.

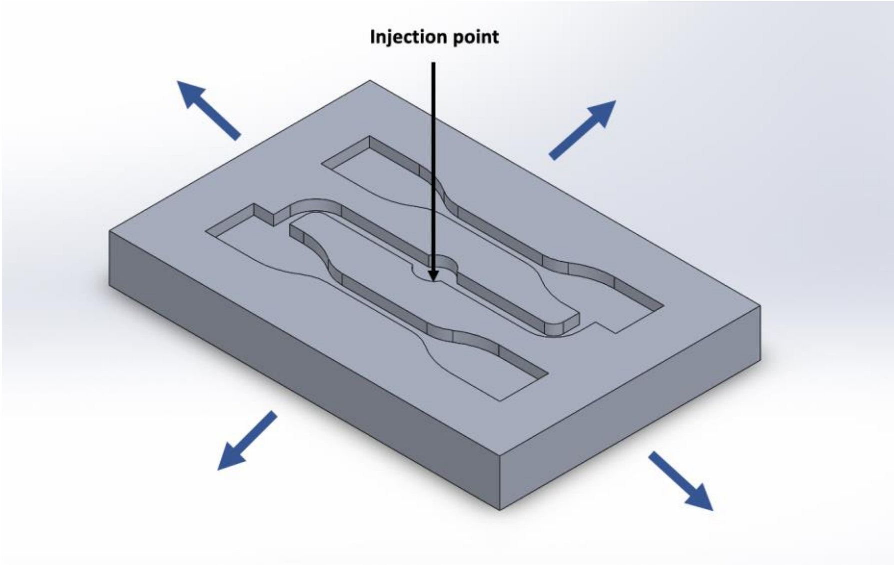


Figure 5: Arrows showing the direction of deformation during injection

To mitigate this problem, a metallic outer frame was designed in CAD and machined using a CNC machine. The inspiration for this design came from FDM 3D Printed mold tools where an outer frame is often added to support the mold [8]. The machined outer frame made out of 6061 Aluminum Alloy is shown in Figure 6. The mold was placed in the outer frame and then a top metallic plate was placed on top of the mold tool (Figure 7). This three-piece assembly is then placed in the injection molding machine.

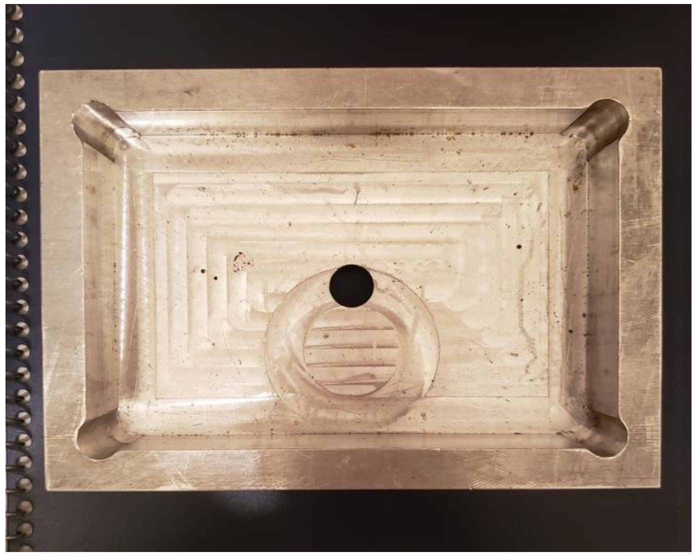


Figure 6: Outer metal frame used to limit mold deformation

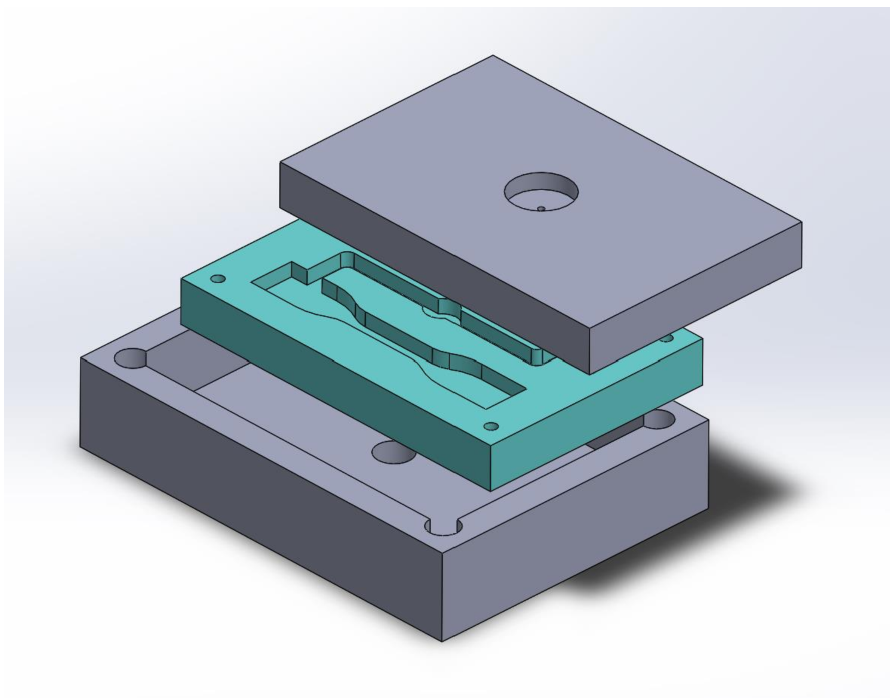


Figure 7: Schematic showing the pieces of the silicone mold tool. The pieces in gray are made out of metal

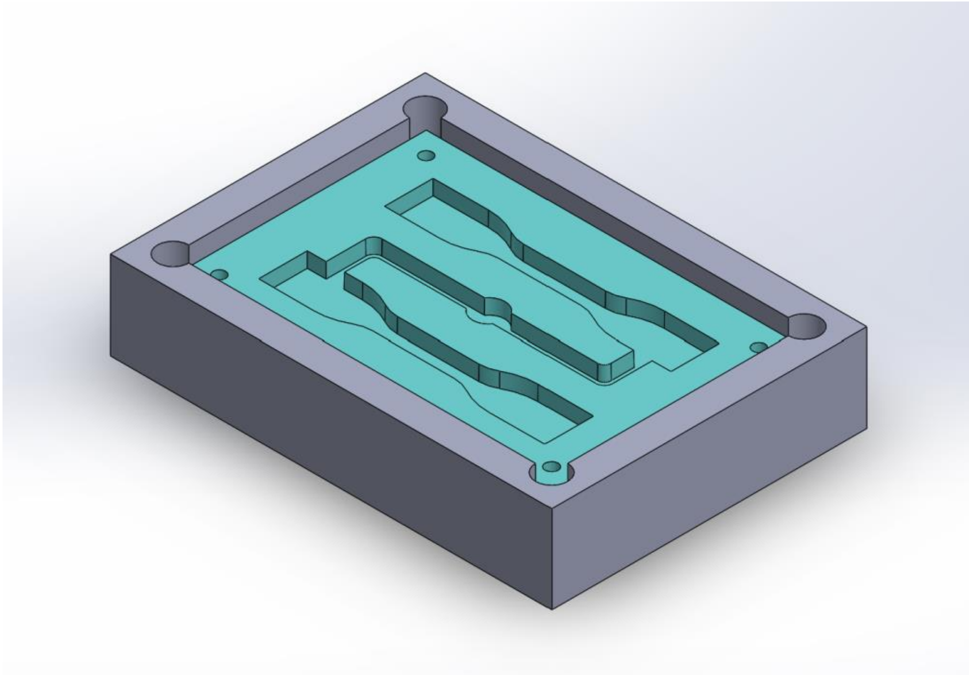


Figure 8: Schematic of the silicone mold in the outer metal frame

CHAPTER 3

INJECTION MOLDING SIMULATION

Designing an injection molding tool might require several iterations, based on the complexity of the part being produced. One way to avoid the loss of time and money is to incorporate injection molding simulation into the process. Nevertheless, before jumping into the simulation process, it is imperative to talk about the most important injection molding parameters and possible sources of failure associated with them.

3.1 Injection Molding Parameters and Sources of Failure

There are a number of injection molding parameters that can have an influence on a plastic part's quality. Out of these, the most significant ones are the following:

- Nozzle Temperature: The temperature of the nozzle from which the plastic enters the gate of the mold tool.
- Barrel Temperature: The temperature of the container in which solid plastic pellets are inserted. Usually, this temperature is slightly lower than the nozzle temperature.
- Injection Pressure: One of the most important parameters, it is the pressure at which the melted plastic enters the mold tool. This pressure is provided by the hydraulic system of the injection molding machine.
- Clamping Pressure: It is the pressure that presses the top and bottom part of the mold tool together.
- Injection Time: The amount of time the melted plastic flows through the mold tool cavity.
- Mold Tool Temperature: The temperature of the mold tool when plastic is injected into it. The nozzle, plate, and ambient temperature have a large effect on it.
- Plate Temperature: The temperature of the bottom surface on which the mold tool rests.

- Hold Time: The time during which the mold tool sits in the injection molding machine so that the plastic can solidify.
- Cycle Time: The total time it takes for injecting one plastic sample to the next one. This also includes the hold time and the time it takes to take the sample out of the mold.

Figure 9: Morgan Press G-125T injection molding machine used in the experiments

A mold tool can have different geometric parameters based on the application of the injection molded part. Some mold tools also include cooling channels to control the cooling rate of the product, as a constant cooling rate is linked to greater part quality [26]. During an injection mold process, there are several key factors that can affect the part quality and lead to failure. These factors are summarized in Table 2.

Table 2 Sources of failures for the injection-molded part

Defect	Description	Possible Causes
Blister	Raised zone on the surface of the injected part	Lack of cooling around the mold tool or excessively high nozzle temperature
Burn marks	Black or brown burnt areas on the injected part, usually located at furthest points from the gate	Lack of ventilation or injection speed is too high
Color Streaks	Uneven color distribution	The plastic material and colorant aren't mixing. Applicable to only colored parts
Contaminates	Foreign particle embedded in the injected part	Particle on the tool surface. In the case of AM mold tool, a high nozzle temperature can melt part of the mold tool
Flash	Excess materials, leakage of material outside the allowable injected area	Tool damage, too much injection pressure, low clamping pressure or too much injection time
Flow marks	Off tone wavy lines or patterns on the injected part	Injection pressure too low, not giving enough hold time for the material to cool down and solidify
Jetting	Injection part has a deformed shape due to the turbulent flow of the plastic material	Injection pressure too high, gate position not in the middle
Polymer degradation	Polymer breakdown due to oxidation of the material	Excess moisture in the pellets or the barrel temperature is too high

Sink marks	Localized depressions in the plastic part	Hold time or injection pressure too low. Cooling time is too short. It can also be caused by a high nozzle temperature.
Short shot	Partial filling of the mold tool	Lack of material, injection pressure or time too low.

3.2 Simulation Using Solidworks Plastics

There are two main categories of simulation software for injection molding processes. They are available as independent software whose sole purpose is to carry out simulations related to plastic injection molding or they are available as an add-on for existing CAD packages such as Autodesk or Solidworks. Simulation for plastic injection molding is a relatively new technique as compared to other types of simulations such as structural or fluids.

Solidworks Plastics is an add-on for the popular CAD software, Solidworks by Dassault Systèmes. In short, Solidworks Plastics simulates how melted plastic flows through the injection mold tool to predict the filling time, mold temperature and manufacturing-related defects like shrinkage or warpage. Initial simulation runs showed that unfortunately, Solidworks does not give information on mold tool defects or the effect of flow on different mold materials. The main purpose of using Solidworks Plastics for this project was to see how changing the injection material will have an effect on the quality of the injected part. The explanations for the three injection materials and their properties are illustrated in Table 3 and Table 4.

Table 3: Injection materials and their explanations

Material Type	**Explanation [27]**
High-Density Polyethylene	One of the most commonly used plastics in the world for consumer products. Preferred due to its high ductility, tensile strength, impact

(HDPE)	resistance, and recyclability.
Low-Density Polyethylene (HDPE)	Similar to HDPE in terms of part quality but with less density. Ideal for mold tools with thin cavities.
Polypropylene (PP)	Preferred due to its resistance to reactions with water, acids, and detergents. Also melts at a higher temperature than HDPE and LDPE.

Table 4: Properties of the injection materials

Property	**HDPE**	**LDPE**	**PP**
Melting Point (°C)	210	220	230
Specific Heat (kJ/kg°C)	1.9	2.6	1.8
Thermal Conductivity (W/m°C)	0.44	0.30	0.15
Young’s Modulus (MPa)	900	125	1000
Poisson’s Ratio	0.42	0.40	0.40

3.2.1 Simulation Steps

Due to the lack of resources available online regarding Solidworks Plastics specifically, detailed explanations of each critical step are provided so that the reader can benefit from this project.

Firstly, it’s important to make sure that the mold tool is split into appropriate parts instead of one solid body. This is important because it enables the user to specify the cavity and the mold tool to Solidworks Plastics. If the mold tool is one solid body, the software cannot automatically detect those parts (Figure 10).

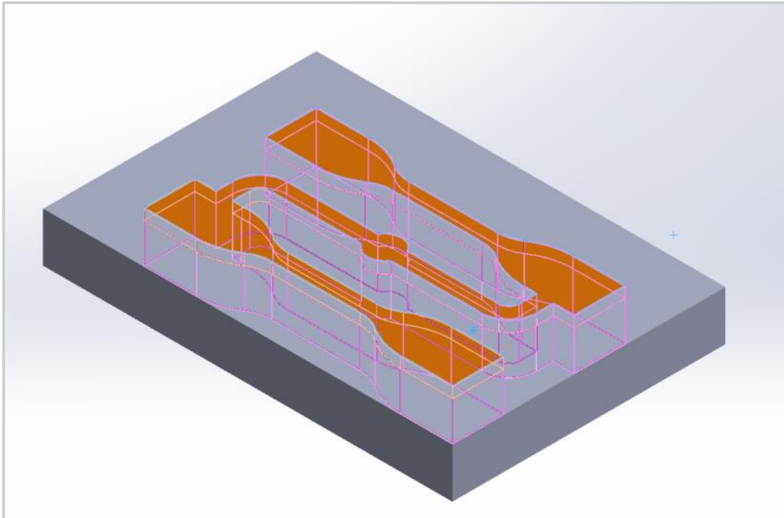


Figure 10: Surfaces were split to prepare the mold for simulation

Once the mold is split into two (or more) parts, it is ready for meshing. A mesh is used in finite element modeling to subdivide the part geometry into smaller domains called elements, over which the governing equations of interest are solved. There are two types of mesh that can be used during the simulation of injection molding plastic parts. The first one is called a shell mesh that can be used for parts with thin walls. The second type is called a solid mesh which provides more accurate results for any type of model, thin or thick. As the mold tool used in this project does not have any thin walls, a solid mesh type is used (Figure 11). After specifying the cavity and the mold, further detailed mesh settings can be selected. For the purposes of this project, a tetrahedral mesh with a minimum element size of 0.3 mm was used with 28,456 elements.

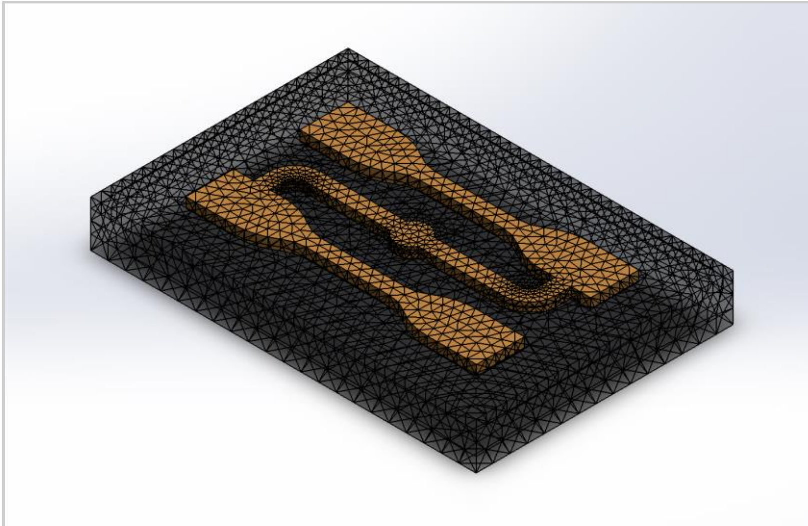


Figure 11: Schematic showing the tetrahedral meshing of the mold tool in Solidworks Plastic

The next step is to select the plastic material that will be injected into the mold tool. Using the Solidworks in-built library, the first plastic HDPE was selected. The software also enables the user to specify the mold materials. However, after running several simulations, it was obvious that changing the model material has negligible effects on the injection molding results. For all three simulations, the mold material was set to be Aluminum Alloy 6061.

Once the mesh and the materials have been specified, the process parameters such as injection pressure, clamping pressure, mold temperature or hold time can be set. The injection pressure used for the simulations was 1500 psi or 10.1 MPa (the lowest injection pressure setting on the real injection mold machine which was used to carry out the experiments). Finally, the injection point was specified. Please note that for mold tools with multiple gates, multiple injection points can be specified, and the size of the injection point can also be adjusted. For this mold tool, an injection point of 2 mm in the center of the mold was chosen (Figure 12).

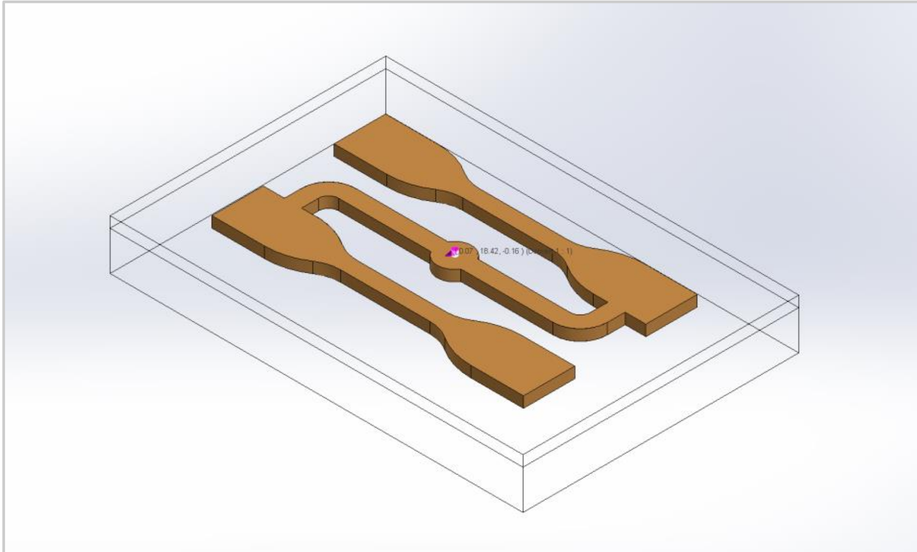


Figure 12: Virtual mold represented by outer thin lines and the pink circle showing the injection point

3.2.2 Simulation Results

While keeping all the other injection molding parameters constant, a comparison was made between HDPE, LDPE and PP so that one of them can be selected for the experiments with the silicone mold tool. The results of these simulations are summarized in Table 5.

Table 5: Injection molding simulation results for each injection material

Property	**HDPE**	**LDPE**	**PP**
Filling Time (s)	2.2772	2.1143	2.1060
Inlet flow rate (cm^3/s)	7.71	7.83	7.73
Pressure at the end of fill (MPa)	9.79	11.31	8.34
Max shear stress (MPa)	0.0914	0.1125	0.0762
Temperature at the end of fill (°C)	209.62	219.69	229.92

It was found that all three materials had almost the same filling time, with PP filling the mold tool at the fastest rate of 2.106 seconds. A similar trend was observed for the inlet flow rate which remained almost unchanged. Comparing this with the experimental results of a previous study involving PP, it was found that the experimental filling time was 88% higher for HDPE and 84% higher for PP [28] [29]. These discrepancies can be explained by the fact that in the experiments, an injection pressure of 3000 psi was used, which is double of the one used in the simulation. The difference can also be attributed to the temperature of the mold tool or surface finish of the cavity as the metal mold tool used in experiments was produced using CNC machining.

The pressure at the end of the filling for PP was 14.3% less than HDPE and 26.25% less than LDPE. This is an important property to take into account while experimenting with the silicone mold tool as the goal is to minimize the amount of pressure used to fill the mold so that the chances of mold deformation and flash are low. As shown in Figure 13 and Figure 14, not only the maximum pressure was lower in PP, the pressure was also distributed much more evenly than in HDPE.

To avoid deformation of part, the shear stress of the melted plastic as it travels through the mold cavities should also be minimized to avoid damages to the mold. In this case, PP was again the material with the least maximum shear stress value at 0.0762 MPa, as compared to 0.0914 MPa and 0.1125 MPa for HDPE and LDPE respectively. The temperature at the end of the fill was highest in PP at 229.92 °C. Higher temperatures can lead to a higher concentration of sink marks in the part, as illustrated by Figure 15 and Figure 16. In the plot, it can be observed that while PP has a lower maximum size for the sink mark at 0.092 mm, the concentration of those sink marks is higher than that of HDPE.

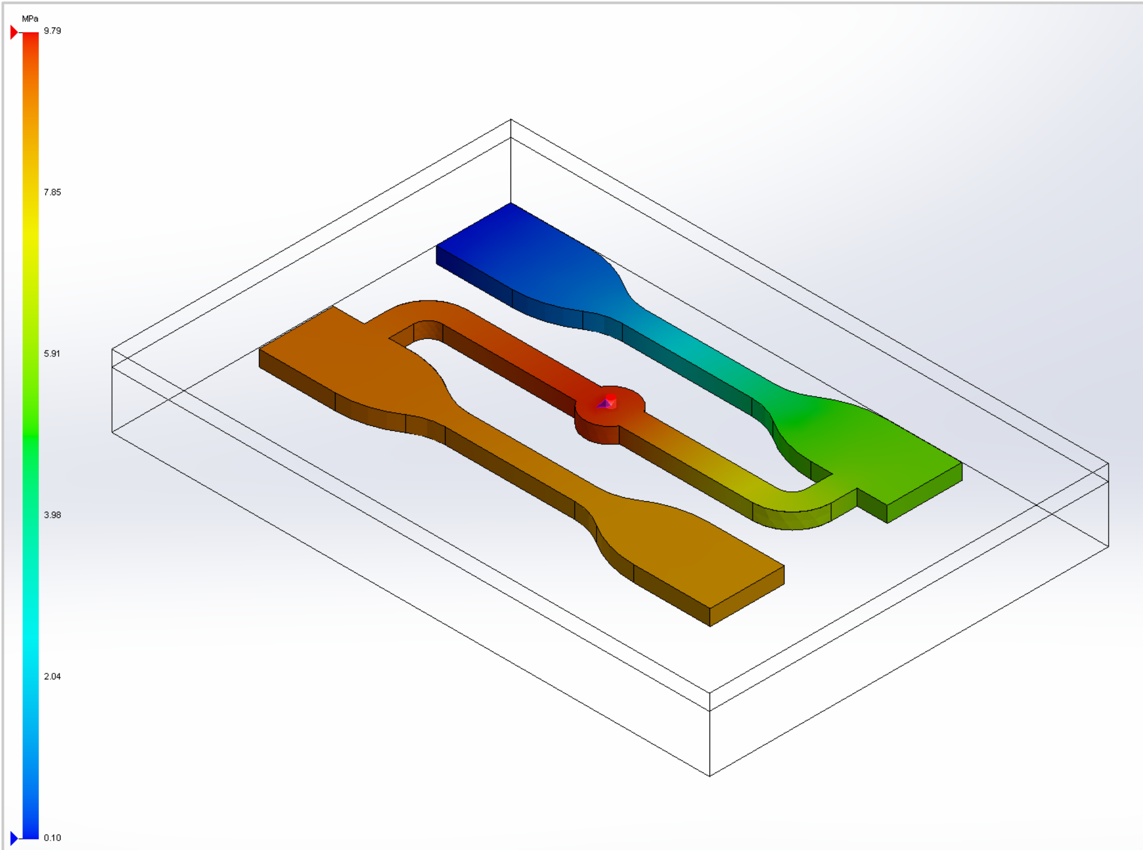


Figure 13: Pressure distribution for HDPE

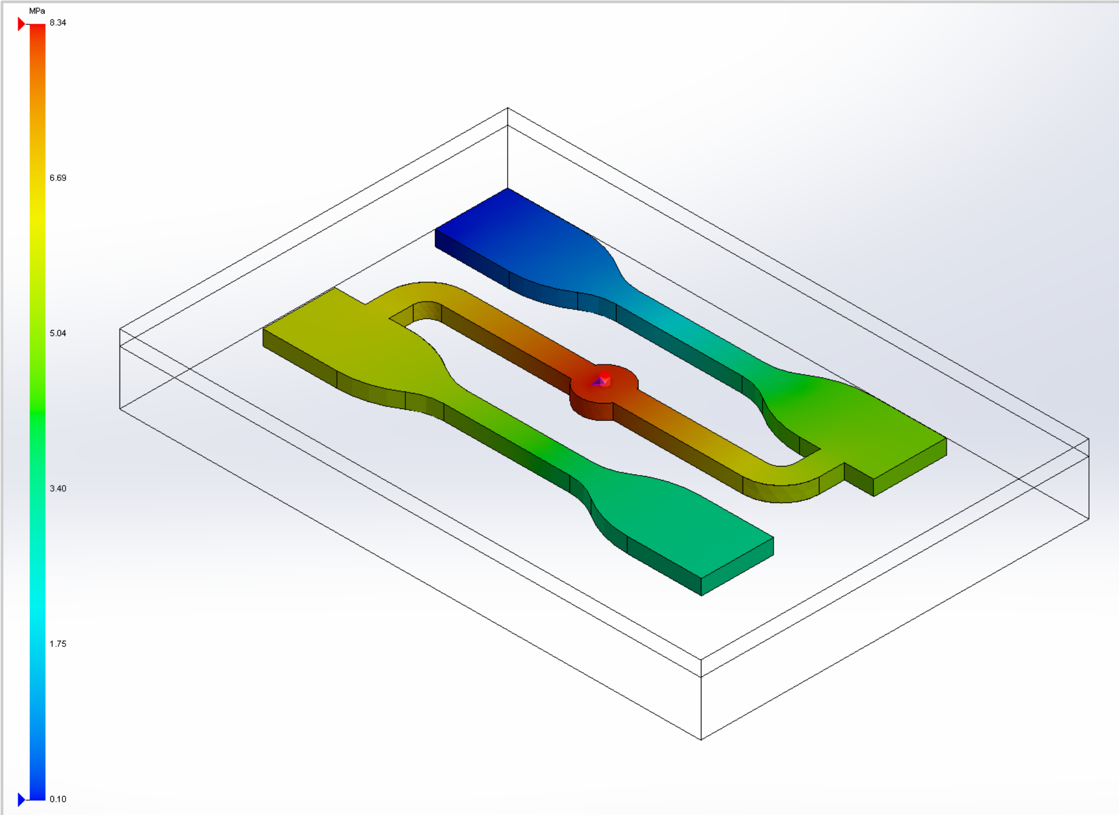


Figure 14: Pressure distribution for PP

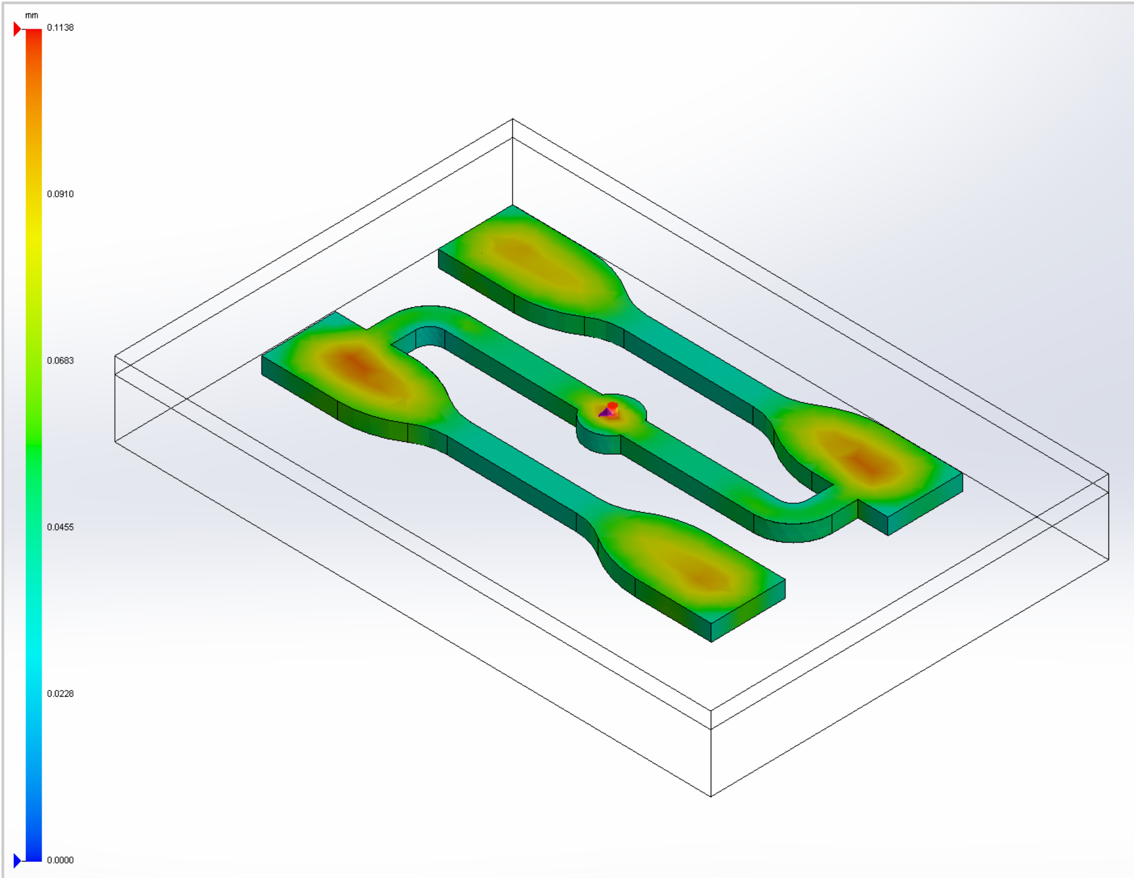


Figure 15: Sink marks distribution for HDPE

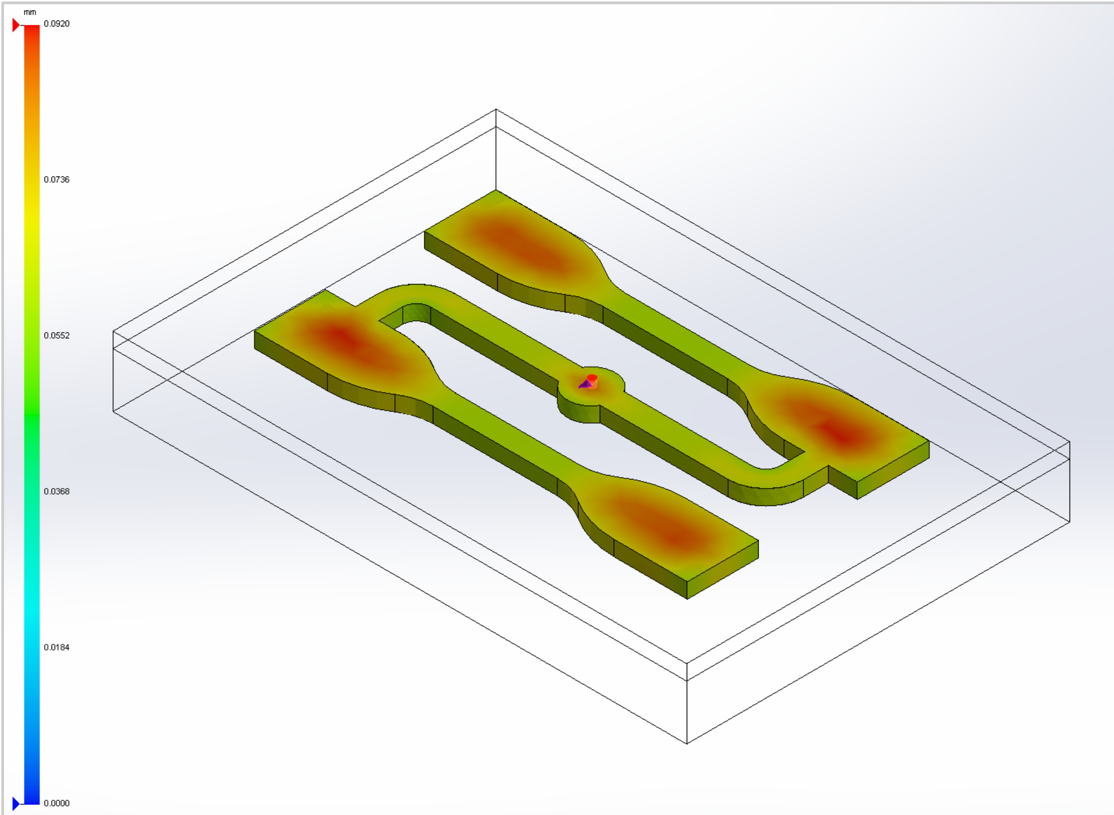


Figure 16: Sink mark distribution for PP

Comparing these with the experimental results, the sink marks for HDPE were observed in only 1 out of 5 injected samples as shown in Figure 17 [28] [29]. When there were sink marks, their size was less than the ones obtained through simulation. In a previous study involving PP blended with a biopolymer lignin, no sink marks were observed in the test samples as shown in Figure 18. At a later stage in the project, 5 PP samples were injected into a metal mold tool using the same injection molding parameters as the simulation. Those samples also did not show any sink marks as shown in Figure 51.

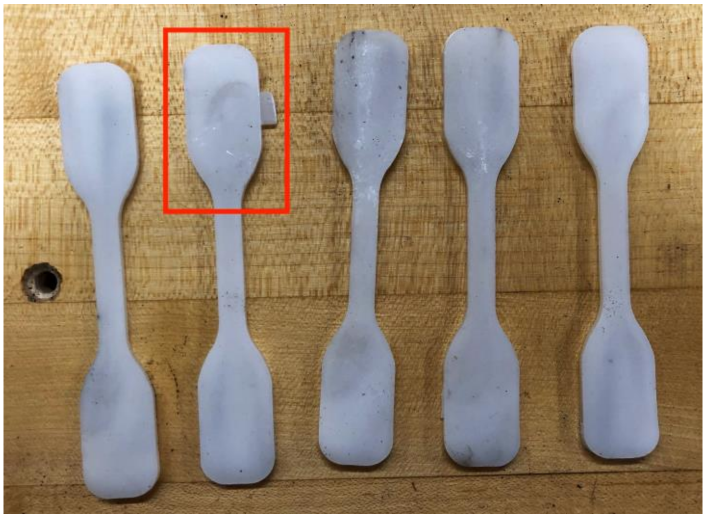


Figure 17: HDPE samples with the red rectangle showing the sink mark

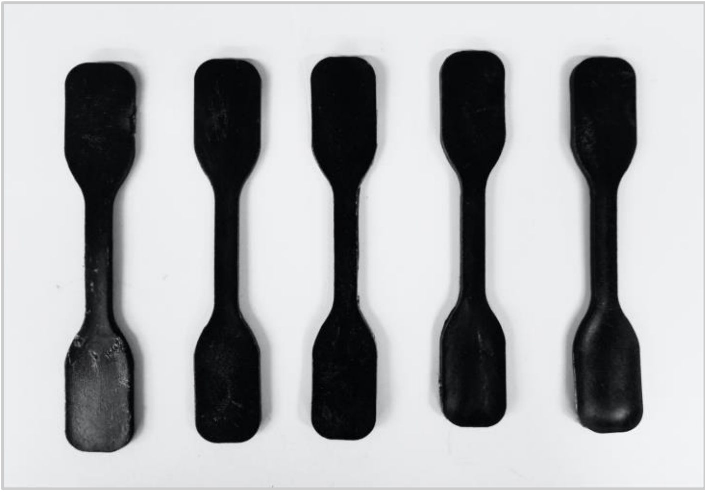


Figure 18: PP blended with lignin samples showing no sink marks

After evaluating the results, PP was chosen as the material of choice for this project out of the three options due to its low magnitude and distribution of pressure. The pressure distribution plot for PP showed that most of the high pressure was distributed in the gate area, whereas for HDPE, it carried to the part itself. Even though the size of the sink marks for PP were larger than those for HDPE, the location of those sink marks was in the ends of the tensile sample, where the tensile machine grips the sample. As there are no sink marks in the neck of the sample it is not a critical source of concern for the silicone mold experiments.

CHAPTER 4

DESIGN OF EXPERIMENTS

The term “experiment” is defined as an operation or procedure carried out under controlled conditions in order to discover an unknown effect [30]. The scientific approach to experiments avoids trial and error and instead proceeds in a systematic manner to determine which process inputs have a significant impact on the process output. Additionally, through experiments, the target level of those inputs can be found to achieve the desired output. Design of Experiment (DOE) is the term used for such a systematic approach. Designed experiments are not only useful in determining the relationship between inputs and outputs but also, they are powerful tools using which manufacturing costs and unnecessary revisions can be avoided [31]. In every DOE, there are three main aspects of the process that are analyzed:

1. Factors: The inputs to the process. Factors can either be controllable or uncontrollable variables. These are the main variables being analyzed. In this project, factors can be variables such as injection molding parameters, mold tool geometry or the durometer hardness of the mold tool.
2. Levels: The actual values for each factor in the process. Examples include an injection pressure of 1500 psi, a durometer of 30 or an injection time of 5 seconds.
3. Response: The output of the process. Examples include fill percentage, quality of the part or size of sink marks.

A successful injection run using a silicone mold tool should give a sample of the same quality as the one obtained using the metal mold tool. To achieve this, a number of factors were tested in various designed experiments. In total two DOE studies were conducted to obtain the most optimum sample with a high repeatability rate.

4.1 Measuring Response

Visually, it is very easy to tell if an injection run is successful or not by looking at the fill of the mold tool cavity, deformation of the mold tool or surface finish of the injected sample. However, a quantitative measure of an injection run's success was needed so that appropriate DOE analysis can be performed once the experiments were concluded. This is extremely important because a combination of two factors can affect the response of the process [32].

Depending on the mold geometry and injection molding parameters, an overfill or underfill of the Polypropylene plastic was expected. To quantify this, overfill or underfill, a response variable "fill percentage" was introduced. Table 6 illustrates the difference between a successful and unsuccessful injection run in relation to fill percentage of the mold tool.

Table 6: Criteria for the success of a sample

Fill Percentage	**Injection Type**	**Outcome**
100%	Ideal fill	Success
< 100%	Underfill	Failure
> 100%	Overfill	Failure

One way to quantify this response was to calculate the surface area of each sample and compare it to the surface area of the ideal sample. Using the 3D CAD model of the mold tool, the surface area of the cavity was measured which was categorized as a 100% fill percentage. To find the surface area of each sample, a photo of the sample was taken from the same height. After taking the photo, the background was isolated and using the image processing software ImageJ, the surface area was measured. By measuring a real distance on the sample, a relationship was established in ImageJ of known distance vs. distance in pixels (Figure 19). Depending on the type of sample (underfill or overfill), this surface area was compared to the original 100% fill percentage and associated with a fill percentage of its own using Equation 1.

$$Percentage\ Increase = \frac{Final\ Area - Ideal\ Case\ Area}{Ideal\ Case\ Area} * 100 \quad (1)$$

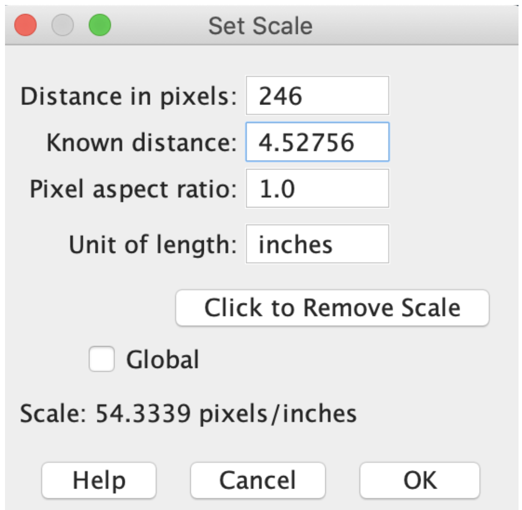
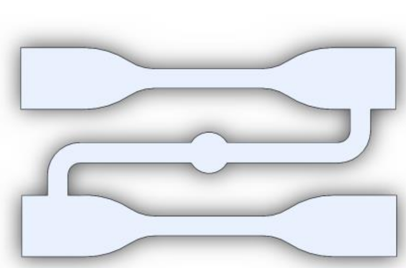


Figure 19: Screenshot of ImageJ showing the relative scale

Table 7: Examples showing corresponding fill percentage with visual representation

Sample #	Area (in^2)	Fill Percentage (%)	Image
Ideal Case	6.309	100	
1	10.419	165	

2	11.041	175	
3	6.948	110	

For samples # 1 and 2 with an overfill, this technique gave appropriate results. However, for sample # 3 with an underfill, this technique proved to be problematic. As illustrated by the figure of sample # 3, during an underfill, the pixels that are outside the mold cavity are indistinguishable by ImageJ. This means that even though quantitatively the fill percentage is close to 100%, in reality, it is not close to an ideal sample when compared to the ideal case in Table 7. Hence, a combination of ImageJ and visual inspection was used to assign a fill percentage number to each sample.

4.2 DOE # 1: Durometer vs. Mold Height vs. Corner Radius

The purpose of the first DOE was to determine the right durometer hardness for the silicone mold tool. A number of variables were kept constant during the DOE studies. Keeping all the injection molding parameters mentioned in Chapter 3 constant is difficult. For example, the controller of the nozzle and the barrel temperature has a steady-state error of 5 °C. However, efforts were made to make sure that the variables are as close to the constant value as possible with a minimum error. The parameters which were selected to be constant during DOE study #1 are given in Table 8.

Table 8: Constant variables for DOE #1

Parameter	**Value**
Nozzle Temperature (°F)	440
Barrel Temperature (°F)	420
Plate Temperature (°F)	250
Hold Time (min)	4
Injection Pressure (psi)	1500
Injection Time (sec)	5
Clamping Pressure (psi)	8000
Material	PP

A DOE study was conducted involving 30 and 40 durometer hardness. Two more factors were chosen along with the durometer, namely the height of the silicone mold tool and the outside corner of the mold cavity (Figure 20).

The reason for selecting the height of the mold tool as a factor was to investigate whether having more material between the mold cavity and the bottom of the mold will have any effect on the rigidity of the mold. The principle is analogous to a sheet of rubber, where a thicker sheet of rubber requires more force to deform. Secondly, the outside corner radius, as shown in Figure 18, was investigated to see if increasing the angle of the curve where the melted plastic flows from the runner to the tensile specimen will help the plastic flow with ease.

The reason for choosing 16 mm as the minimum value for mold height was because this is the same height used for the metal mold tool and it is suited well for the injection molding machine used in this project. The maximum of 26 mm is chosen to see the effects of increasing 10 mm in the mold height for the injected sample. The reason for choosing an outside corner radius of 5 mm and 10 mm was the same where 5 mm was the standard for metal mold tool and 10 mm was chosen as double of that to see the effect of the plastic as it flows through the outside corners.

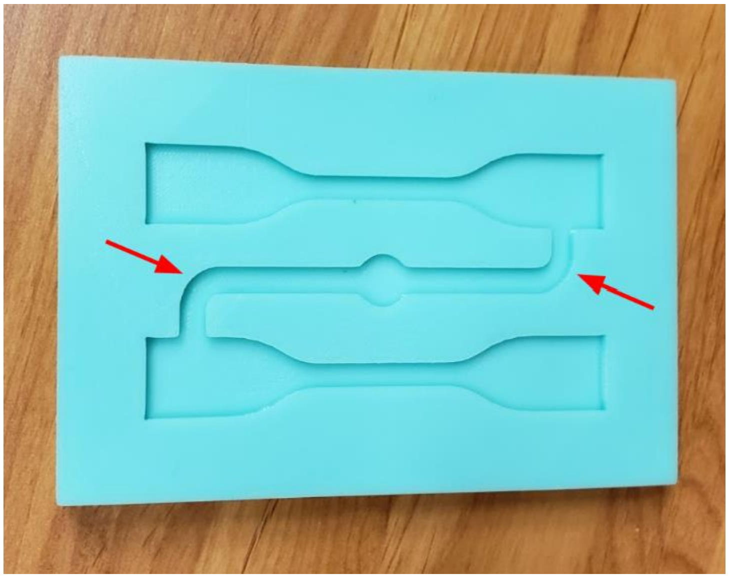


Figure 20: A 40 Durometer mold with a mold height of 16 mm and outside corner radius of 10 mm. The red arrows show the location of the outside corner radius.

Figure 21: Injection molding process diagram for DOE #1

The process diagram for this injection molding experiment can be seen in Figure 21. The three factors are also called the control factors because levels of these were changed in the experiments and their effect on the response was measured. As there are three factors with two levels each, the DOE requires $2^3 = 8$ injection runs for a full factorial design study. The tree diagram in Figure 22 and Table 9 shows the number of runs and their respective combinations.

Table 9: Factors and levels for DOE #1

Factors		**Durometer**	**Mold Height (mm)**	**Outside Corner Radius (mm)**
Levels	1	30	16	5
	2	40	26	10

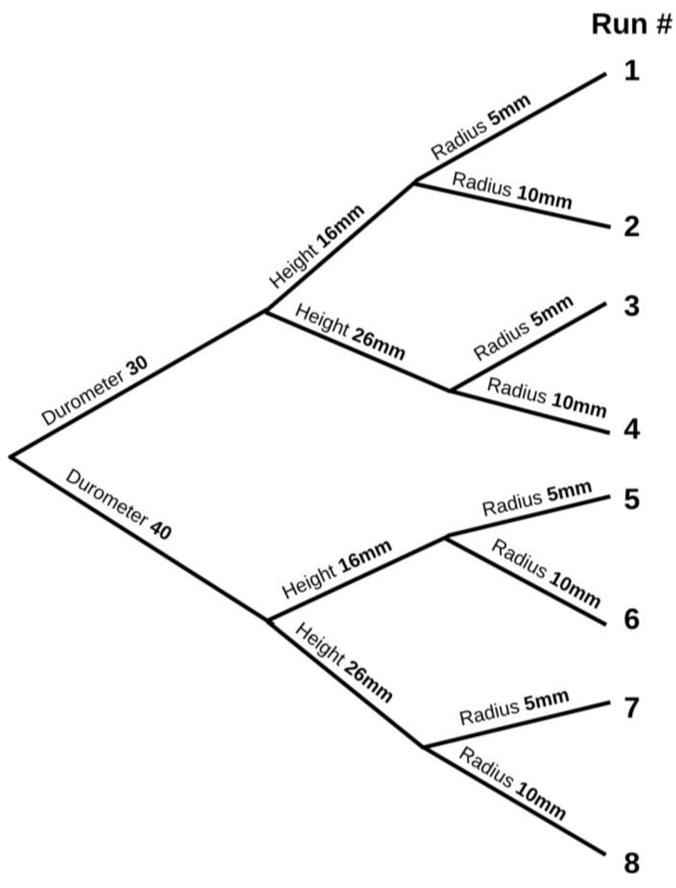


Figure 22: Tree diagram representing factors and levels for DOE #1

Table 10: Injection molding results for DOE#1

Run #	Durometer	Mold Height (mm)	Outside Corner Radius (mm)	Fill Percentage (%)
1	30	16	5	145
2	30	16	10	128
3	30	26	5	135
4	30	26	10	132
5	40	16	5	119
6	40	16	10	109
7	40	26	5	122
8	40	26	10	111

After running the experiments and calculating the surface area of each test sample, the fill percentage was calculated using ImageJ or visually inspected and input into Minitab, a statistical analysis software. Using this software, the results can be analyzed in a number of ways. Techniques for statistical analysis of an experiment can span many chapters, as it is a mature topic practiced by many in the scientific field. In total, two graphical analysis tools were utilized in this project. These were the Pareto chart and the main effects plot:

- The Pareto chart is a very useful tool for determining which factors have the largest effect on the response of the process. It is one of the seven basic tools of quality control [33]. Essentially, a Pareto Chart represents a bar chart in which the bars are arranged in descending order of the factors which have the largest effect on the response. It is also a useful tool to determine if a combination of two factors is affecting the response.

- The main effects plot shows a slope for each factor representing the average of the data points at the low and at the high factor settings. The greater the slope and the length of the line plot, the larger the effect of that factor will be on the response variable. Usually, when the interaction effects are insignificant, the main effects plot can provide useful information on which factor has an effect on the response variable.

The Pareto chart of the first DOE study is shown in Figure 23. The red dotted line represents the standardized effect value. The standardized effects are t-statistics that test the null hypothesis that the effect is 0. Any effect with bars reaching beyond the standardized effect line is considered a significant effect at $\alpha = 0.05$, where α is the Lenth's pseudo standard error. In this Pareto chart, it can be seen that Durometer significantly affects the response fill percentage. The outside corner radius and mold height did not have as large of an effect and in the case of mold height, the effect is almost negligible as seen in the Pareto chart.

As stated earlier, an ideal sample will have a fill percentage of 100%. The main effects plot shown in Figure 24 can aid in determining which specific levels will result in a 100% fill percentage. Each slope has a maximum and minimum value represented by blue dots. The vertical axis of the main effects plot shows the mean fill percentage of the DOE study. To determine which level should be used in future DOE studies, the factor which is closest to a 100% mean fill percentage should be used. In this case, those factors were durometer 40, mold height 16 mm and outside corner radius of 10 mm.

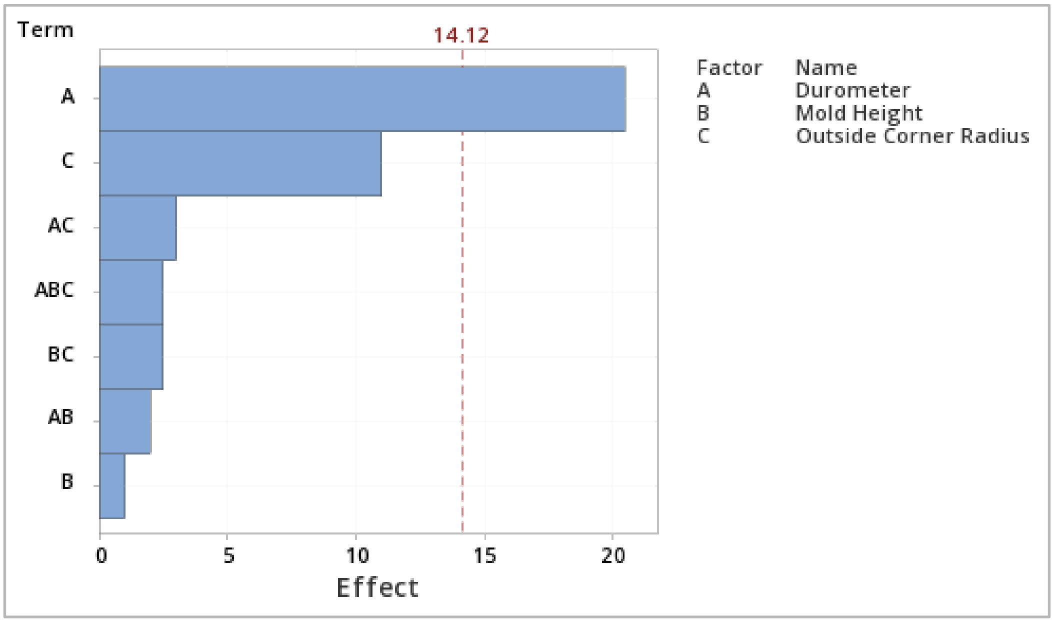


Figure 23: Pareto chart for DOE #1

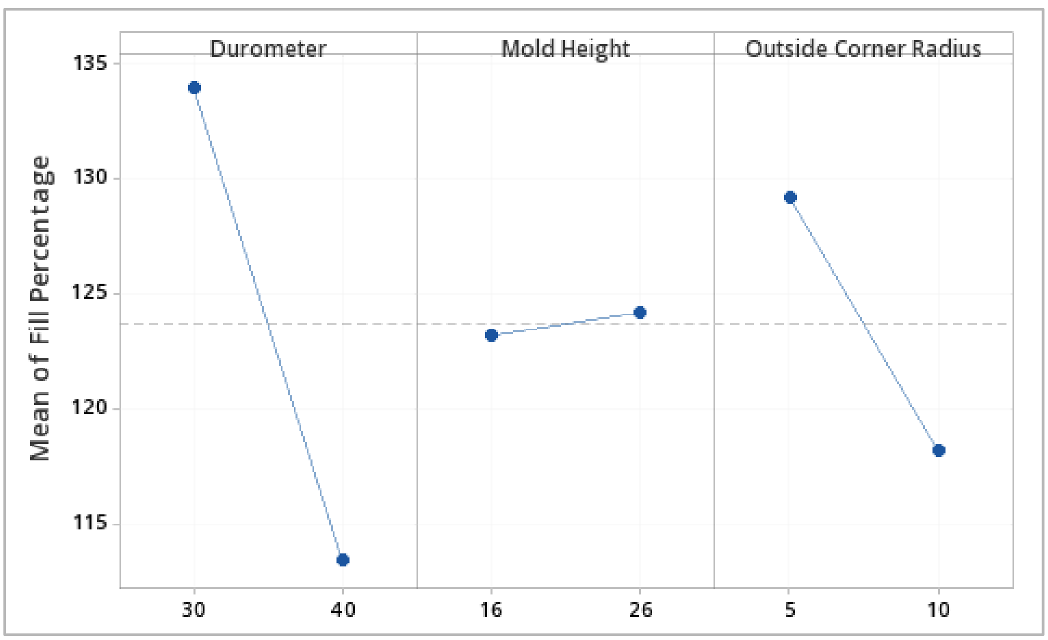


Figure 24: Main effects plot for DOE #1

4.2.1 Deterioration of Mold Tool and Sample Quality

In addition to exhibiting a high fill percentage and consequently failure by flash, the 30 Durometer mold tool also showed deterioration from the four edges (Figure 25) more than 40 Durometer mold tool (Figure 26) after 8 runs. This can be attributed to the fact that the mold tool has an outer metal frame which limits the silicone mold tool's deformation. In contrast to this, the durometer 40 also showed a slight deterioration in the corner, but it was much less than durometer 30. This shows that the durometer 30 silicone is more prone to deformations which can lead to a failure by flash.

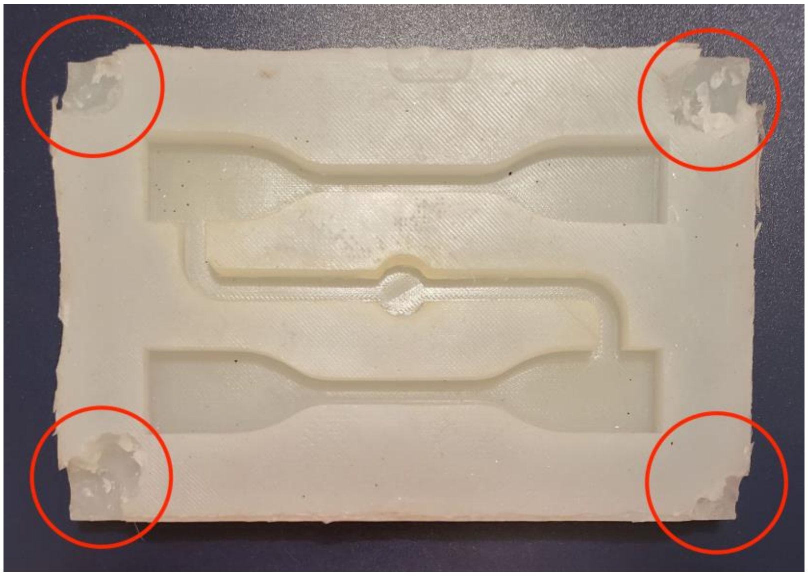


Figure 25: Durometer 30 mold with the edges showing mold damage after only 4 runs

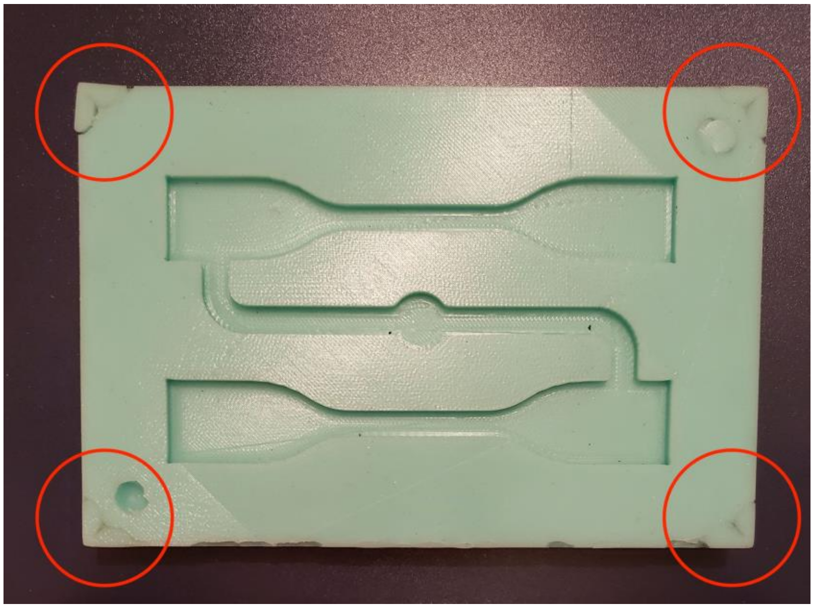


Figure 26: Durometer 40 mold with the edges showing mold damage after only 4 runs

The samples which were overfilled did not show any shrinkage or dimensional instability as shown in Figure 27. There was also no warpage or bending of the samples. This can be explained by the fact that after the material has finished injecting, it is not subjected to high pressure in the gate location. So, even if the mold tool is deformed while injecting, it returns back to its original flat shape once the injection is finished. For samples that were underfilled, there was a slight shrinkage that was observed in the necking area of the sample as shown in Figure 28. This can be attributed to the material not completely filling the cavity of the mold tool.

Figure 27: Photo of the sample obtained from Run #1

Figure 28: Photo of the sample obtained from Run #7

4.2.2 Durometer Hardness 60

Durometer was found to be the most significant factor affecting the mold tool response. The main effects plot showed that to reach an ideal fill percentage of 100%, the durometer should increase. Hence, along with 30 and 40, durometer hardness 60 was used to conduct further experiments. The reason for not using durometer 50 was because that durometer number is not commercially available from the manufacturers of the silicone material used in this project.

The minimum possible injection molding pressure of 1500 psi and injection time of 5 seconds was used to avoid brittle failure of the mold. However, on the first injection run, a brittle fracture propagated through the runner of the mold (Figure 29). The experiment was repeated with two other molds as illustrated in Table 11. For all three of these experiments, a brittle failure to the mold caused the melted plastic to seep through the mold as soon as it was injected. Thus, it was concluded that a durometer of 60 is too high for a silicone mold and it was eliminated from any further DOE studies.

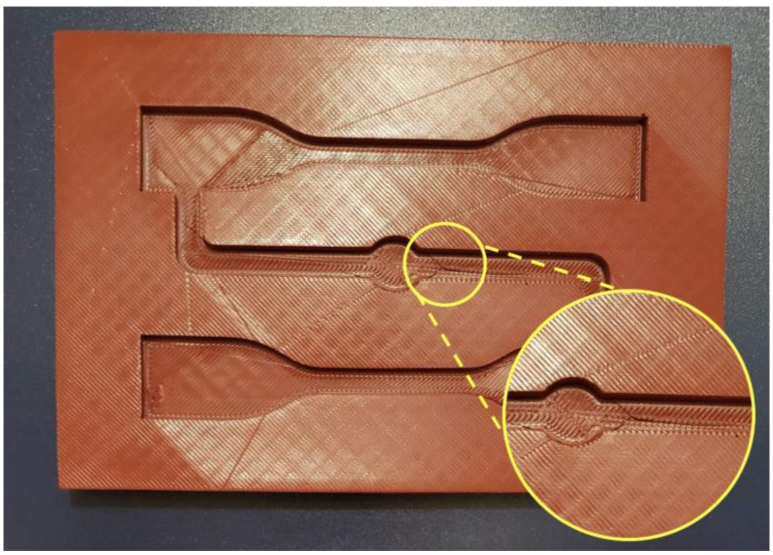


Figure 29: Durometer 60 mold tool, where the yellow circle shows the location of the crack

Figure 30: Bottom of the Durometer 60 mold tool. The PP material can be seen leaking to the bottom of the mold due to crack formation

Table 11: Injection molding parameters for the Durometer 60 mold tool runs

Run #	**Injection Pressure (psi)**	**Injection Time (s)**	**Clamping Pressure (psi)**	**Hold Time (s)**	**Result**
1	1500	5	8000	240	Mold damaged
2	1500	5	8000	240	Mold damaged
3	1500	5	8000	240	Mold damaged

4.3 DOE #2: Injection Pressure vs. Time vs. Clamping Pressure

From the results of DOE #1, it was concluded that a durometer 40 silicone is more suitable to use in the experiments than a durometer 30 silicone as durometer 30 silicone is more prone to deformation and deterioration. In addition to this, a mold height of 16 mm was chosen as it uses less material and an outside corner radius of 10 mm was chosen as samples obtained from that type of mold are closest to the ideal case. The parameters which were set constant for DOE #2 are given in Table 12.

Table 12: Constant variables for DOE #2

Parameter	**Value**
Nozzle Temperature (°F)	440
Barrel Temperature (°F)	420
Plate Temperature (°F)	250
Hold Time (min)	4
Mold Height (mm)	16
Outside Corner Radius (mm)	10
Durometer	40
Material	PP

As an ideal sample with a fill percentage of 100% was not obtained via the first DOE, it was decided to conduct a second DOE study with the focus on injection molding parameters. The first parameter that was chosen as a control factor was injection pressure. The reason for choosing this as a control factor was because based on previous experiments using the same injection molding machine with HDPE, it was found that the injection pressure had the greatest effect on the fill percentage while using a metallic mold [29]. In total, 2 levels are chosen for the injection pressure. The first is 1500 psi which is the minimum injection pressure at which the injection molding machine injects the material. The second is 3000 psi, double of the first level's value. The reason why this was the max value, as opposed to something higher, was to avoid deterioration of the mold as a slight deterioration of the 40 durometer mold tool was observed during DOE #1 (Figure 21).

The second parameter that was chosen was the injection time. If the injection time is too small, there will be failure by short shot. If it's too high, there will be failure by flash. To find the optimum injection time that will give an ideal sample, two levels of injection time were selected. The first one was 5 seconds, as that is the default injection time recommended while injecting PP. The second one that was selected was 10 seconds. The reasoning behind using these values is similar to that of injection pressure; a minimum value was chosen that will inject the material and maximum value was chosen which avoids any potential damage to the silicone mold while it deforms under high pressure.

The third parameter that was chosen was clamping pressure. It was hypothesized that the reason for obtaining failure by flash was because the contact between the two mold surfaces was not perfect. If there are any gaps in contact, the melted plastic will leak from the runner as it is injected. The best way to avoid flash is to increase the clamping pressure [34]. Hence, two levels of clamping pressure at 8000 psi (the standard for PP) and 10000 psi were chosen. These factors and levels are listed in Table 13 and Figure 28.

Table 13: Factors and levels for DOE #2

Factors		Injection Pressure	Injection Time	Clamping Pressure
Levels	1	1500 psi	5 secs	8000 psi
	2	3000 psi	10 secs	10000 psi

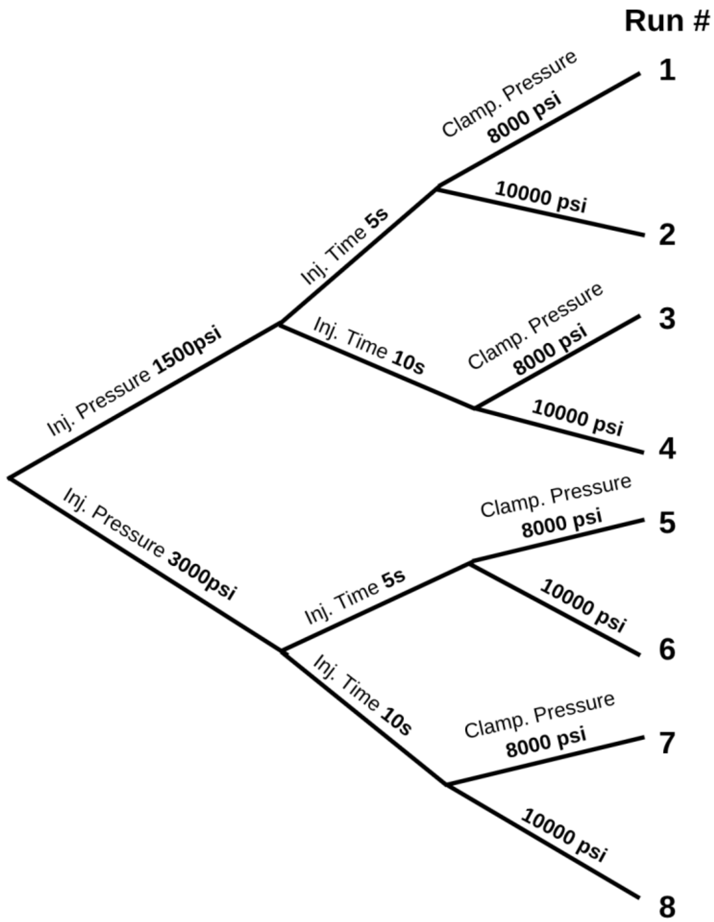


Figure 28: Tree diagram representing factors and levels for DOE #2

Similar to the previous DOE, the surface area of each sample was measured, and the fill percentage was calculated to populate Table 14.

Table 14: Injection molding results for DOE #2

Run #	Injection Pressure (psi)	Injection Time (s)	Clamping Pressure (psi)	Fill Percentage (%)
1	1500	5	8000	61
2	1500	5	10000	73
3	1500	10	8000	88

4	1500	10	10000	108
5	3000	5	8000	154
6	3000	5	10000	139
7	3000	10	8000	183
8	3000	10	10000	160

By monitoring the response of each run in Table 14, the effect of each injection molding parameter can be analyzed. For an injection pressure of 3000 psi, all the samples failed by flash. When the injection pressure was half of that, there were short shots (or underfill) but the fill percentage was much closer to 100%. This made it clear that an injection pressure of 3000 psi was too high for the silicone mold tool. Additionally, an injection time of 10 seconds when combined with an injection pressure of 1500 psi and clamping pressure of 10000 psi gave the closest fill percentage (108%) to an ideal sample.

The Pareto chart in Figure 31 shows that there are two significant factors mainly, injection pressure and injection time. A third variable that is also significant is the combination of injection pressure and clamping pressure. The main effects plot in Figure 32 further shows how injection pressure and injection time had the greatest effect on the fill percentage. Changing the clamping pressure from 8000 psi to 10000 psi did not result in a large change in the fill percentage. This was contrary to past experiments on metal mold in which increasing the clamping pressure reduced the flash. This meant that the upper limit that was selected for the clamping pressure was not high enough.

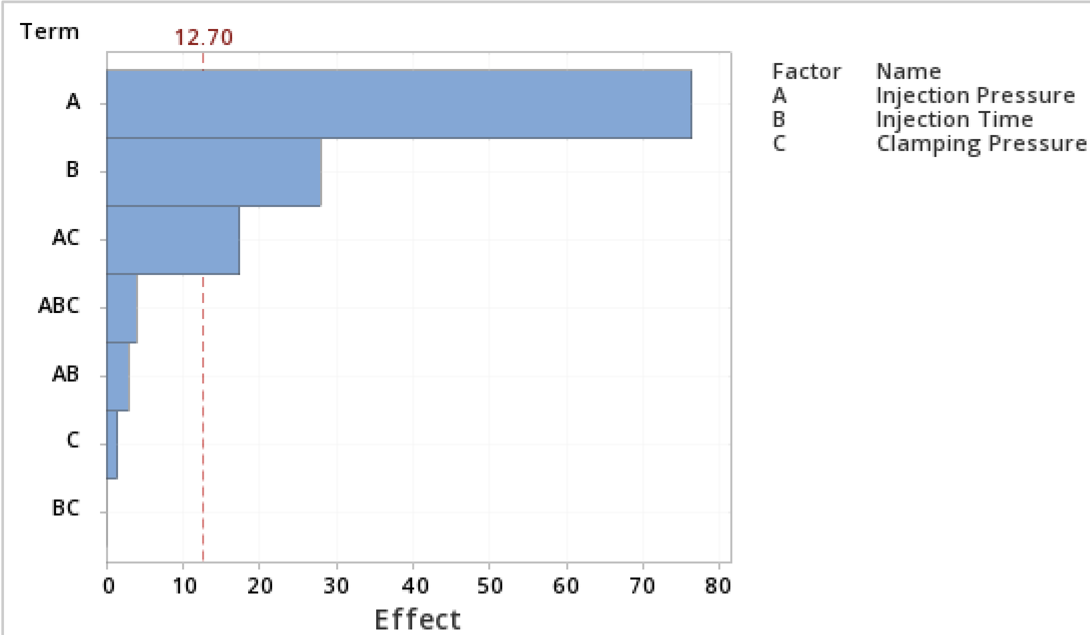


Figure 31: Pareto chart for DOE #2

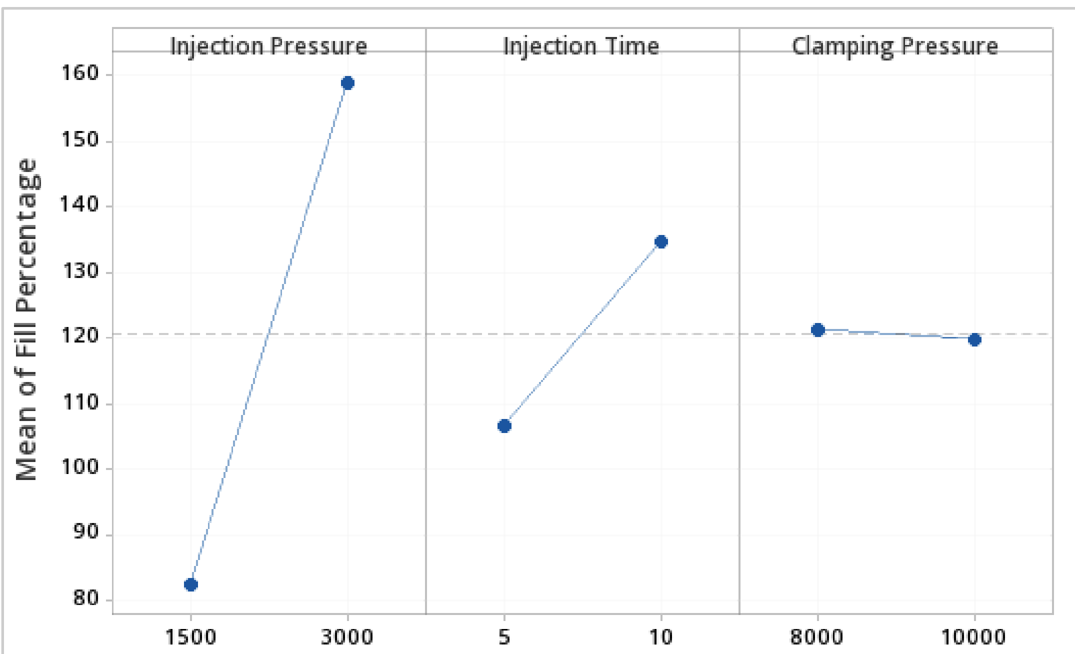


Figure 32: Main effects plot for DOE# 1

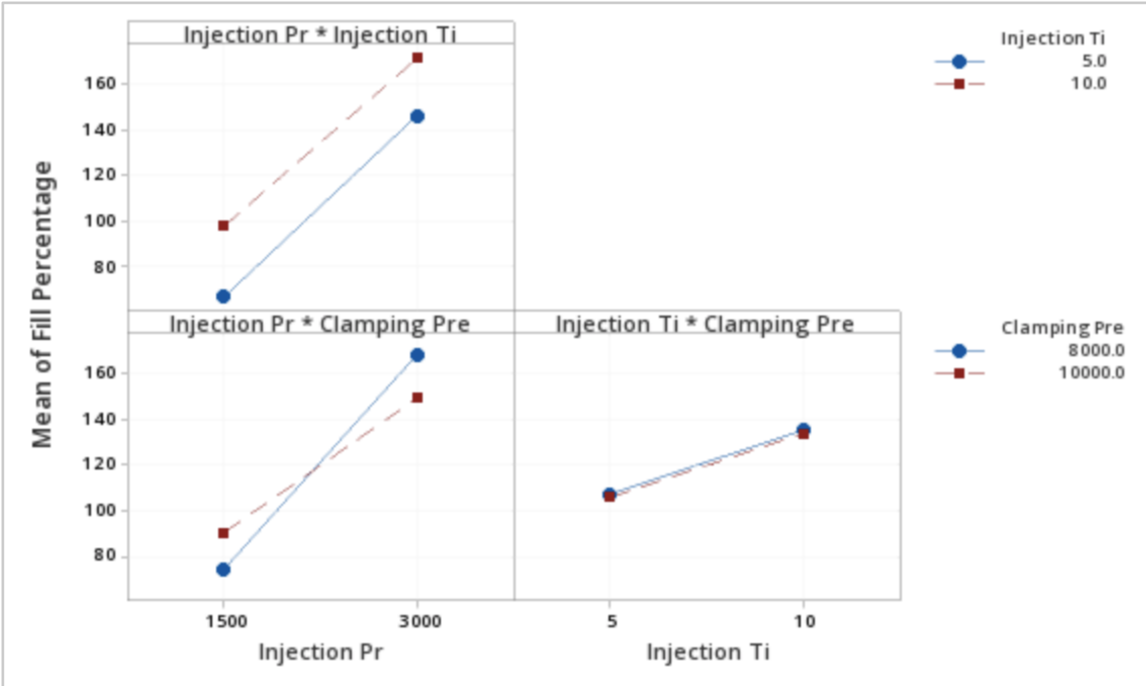


Figure 33: Interaction plot for DOE #2

The bottom left quadrant of Figure 33 shows the interaction of injection pressure with clamping pressure in relation to fill percentage. It can be observed that an injection pressure of 3000 psi with clamping pressure off 8000 psi resulted in a higher fill percentage than with clamping pressure of 10000 psi. When the injection pressure was 1500 psi, the opposite was observed. This shows that a higher clamping pressure results in a fill percentage closer to a 100%, as seen in the y-axis of Figure 31. When the clamping pressure is low, the melted plastic flows outside the mold cavity and results in failure by flash.

An interesting phenomenon that was noticed is that even when there was an underfill, the melted plastic "leaked" from the runner. This can be seen in Figure 34 in which as the melted plastic flowed through the mold cavity, it simultaneously flashed from the runner. This again signified the need for increasing the clamping pressure to reduce the gap between the silicone mold tool and the top metallic part of the mold.

One of the variables most akin to affect the response of the injection molding process is the temperature of the mold tool. A cooler mold tool will pose significant resistance for the melted plastic flowing through the mold cavity, and hence an incomplete injection is more probable. In subsequent studies, the mold tool was given enough time to heat to a specific temperature before restarting the injection cycle.

Figure 34: Run # 2 sample from DOE #2 showing the leakage of material from the runner

4.3.1 Increasing Clamping Pressure

Based on the interaction plot results (Figure 33) of DOE # 2, additional experiments were conducted to measure the response of the mold tool on an increased value of the clamping pressure.

A new variable, called pre-injection hold time was added to ensure that the mold reaches the highest possible temperature, which was around 230 °F (110°C) for the silicone mold. This temperature was validated using a thermal camera (Figure 35). The parameters chosen for the first injection run were an injection pressure of 1500 psi, injection time of 5 seconds and an increased clamping pressure of 14000 psi. However, when the 5 seconds injection time resulted in an underfill (Figure 36), another experiment was performed by increasing the injection time to 10 seconds. The full injection molding parameters are listed in Table 15.

Table 15: Injection molding parameters for the updated experiment

Run #	Injection Pressure (psi)	Injection Time (s)	Clamping Pressure (psi)	Pre-injection Hold Time (s)
1	1500	5	14000	240
2	1500	10	14000	240

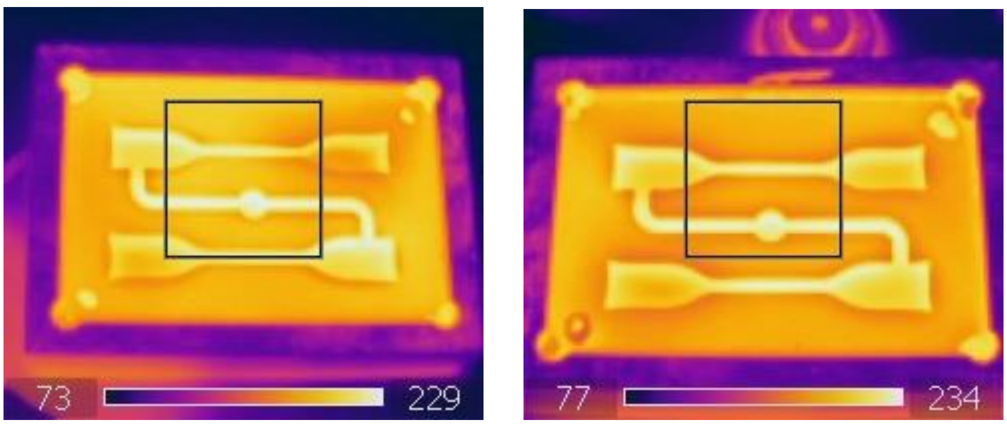


Figure 35: Thermal camera photos taken before injecting. The scale on the bottom is in °F

Results showed that increasing the clamping pressure with an increased injection time did result in the reduction of flash. However, it also led to dimensional instabilities in the shape of the sample as seen in Figure 37. The runner area of the mold tool expanded due to the high clamping pressure. The problem of leakage from the runner was not solved and a change in mold geometry was implemented.

Figure 36: Photo of the sample obtained from Run #1

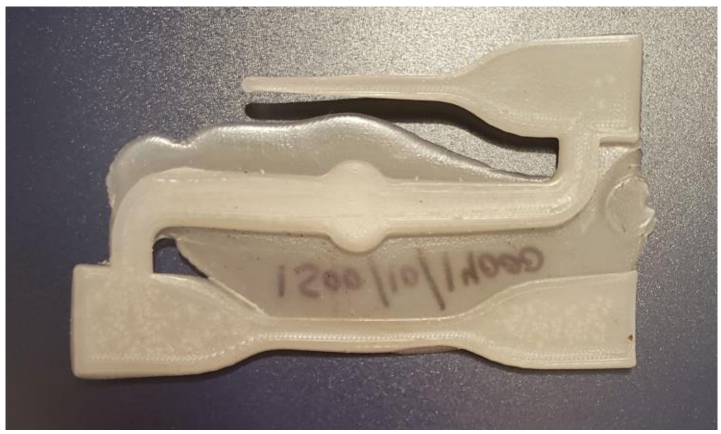


Figure 37: Photo of the sample obtained from Run #2

CHAPTER 5

FURTHER EXPERIMENTS

Taking the results of the multiple DOE studies into account, further experiments were conducted to measure the response of the mold tool on updated mold tool geometries and designs.

5.1 FDM 3D Printed Insert

From the DOE studies, it was found that the melted plastic flashed mainly from the runner, due to which a change was proposed to the mold tool geometry. To add rigidity to the area around the runner, a 3D printed material using FDM was inserted into the mold around the runner. (Figure 38).

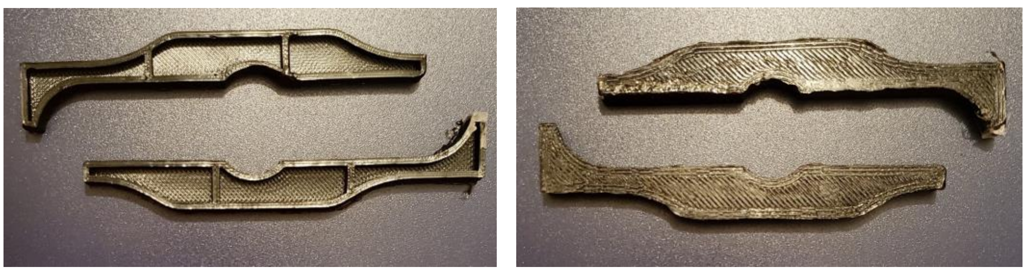


Figure 38: 3D printed PLA inserts

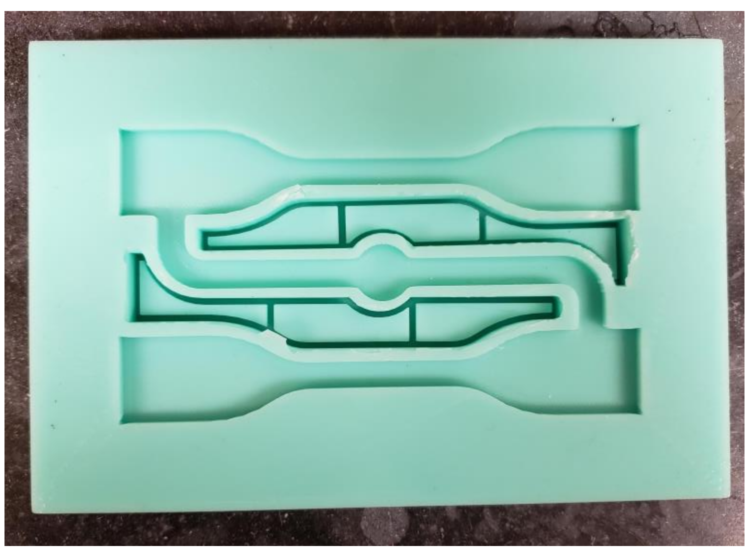


Figure 39: Updated mold tool design to incorporate the 3D printed inserts

On the first injection run with an injection pressure of 1500 psi, injection time of 5 seconds and clamping pressure of 10000 psi, the 3D printed insert melted into the melted plastic (Figure 40). The reason for this was that the 3D printed insert was made of PLA, which has a melting point of around 150°C. This is much less than the nozzle temperature of 220°C. Thus, another insert was 3D printed with Nylon, which has a melting point of 230°C.

Figure 40: Sample showing 3D printed PLA insert melted with PP after solidifying

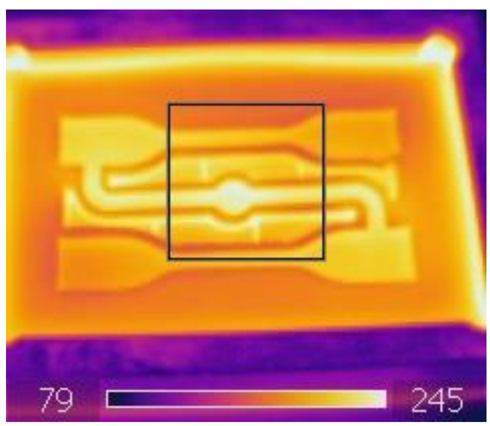


Figure 41: Thermal camera photo of the Nylon insert mold tool before injecting. The scale is in °F

The Nylon insert was able to sustain the high nozzle temperature without melting. Results showed that the insert was able to control failure by the flash from the runner and this kind of mold tool gave the best samples in terms of quality so far (Figure 43). The brown coloring of the sample is due to oxidation. This occurs when the sample is taken out of the mold earlier than the specified hold time. Mischkot et al. injected LDPE into a disk-shaped steel mold and found 50 seconds to be a sufficient hold time to avoid any loss in the original color [11]. However, in this mold tool, whenever the hold time was below 240 seconds, a slight brown coloration of the samples was observed (Figure 43).

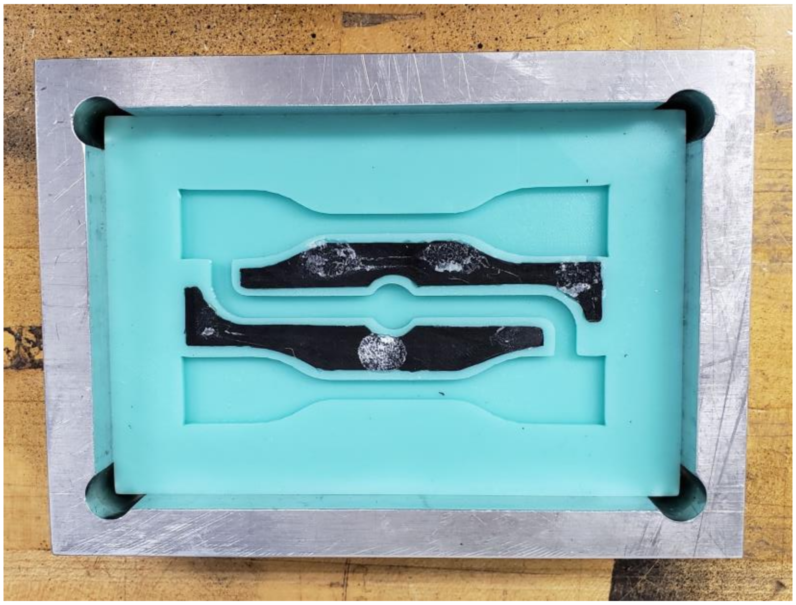


Figure 42: 3D printed Nylon insert mold tool with the outer metal frame

Figure 43: Sample obtained from 3D printed Nylon insert mold tool

Using FDM 3D printed nylon inserts solved the issue of flash but there was failure by short shot as the plastic failed to fill the mold cavity completely. To investigate this issue, a new DOE study on injection molding parameters should be conducted. As seen in the previous DOE studies, different injection molding parameters resulted in different fill percentages. With the flash controlled, it is predicted that increasing the injection pressure or injection time will result in a 100% or more fill percentage. This DOE could not be conducted due to the closure of university labs due to the spread of the COVID-19 pandemic in March 2020.

5.2 Center Injection

So far in all the experiments, the melted plastic leaked from the runner. A new approach was implemented in parallel with the FDM 3D printed inserts. The revised mold geometry eliminated the runner while keeping the neck length of the tensile specimen the same (Figure 44). The main purpose of this mold design was to see the effects of a short runner on the fill percentage.

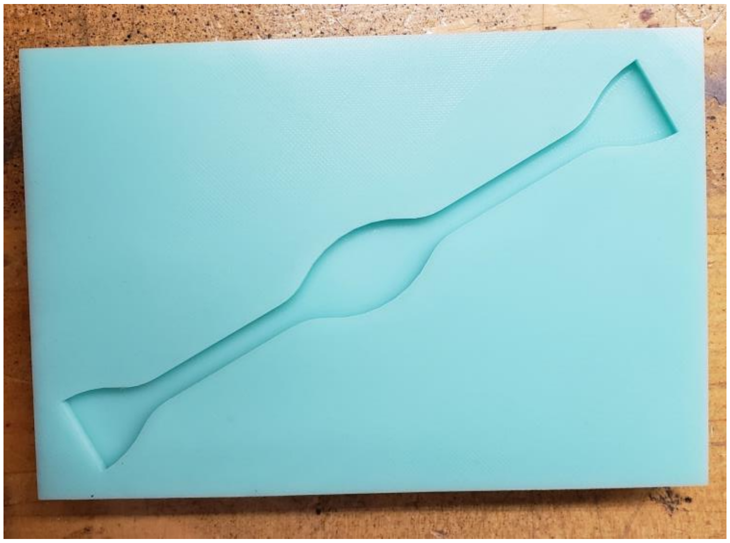


Figure 44: Updated mold tool geometry eliminating the runner

Table 16: Injection molding parameters for center injection with outer metal frame

Injection Pressure (psi)	**Injection Time (s)**	**Clamping Pressure (psi)**	**Pre-injection Hold Time (s)**
1500	5	10000	240

The results in Figure 45 show that injecting in the center of the mold without a runner reduced the flash considerably. However, there was still failure by flash as the material leaked from the gate location.

Figure 45: Sample obtained from center injection with the outer metal frame

One of the reasons why the melted plastic was still leaking from the center of the mold can be explained by the fact that contact between the top metallic plate and the bottom silicone mold tool was not perfect. When the material is injected into the center of the mold with high pressure, the silicone mold tries to expand. However, the outer metallic frame prevents it from doing so, which results in a slight curve in the mold. This curve, in turn, creates gaps between the silicone mold and the top metallic plate and hence leads to leakage of the melted plastic. Taking these factors into account, an injection run was carried out using the configuration shown in Figure 46. In this configuration, the outer metallic frame was replaced with a borderless straight metallic plate so that the mold is free to expand and does not curve.

Figure 46: Injection molding setup without the outer metal frame

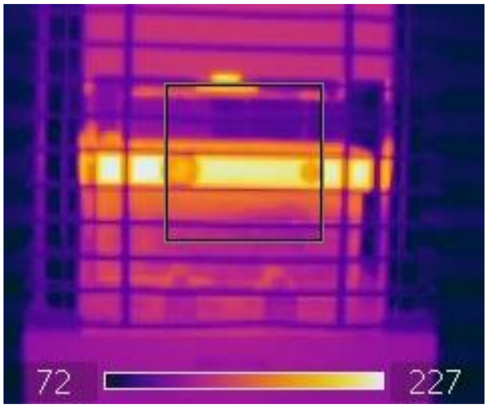


Figure 47: Thermal camera photo of the mold tool when placed in the injection molding machine. The scale is in °F

Using the same injection molding parameters as Table 17, an ideal sample with no flash was obtained as shown in Figure 48. This injection run was repeated 3 more times, and each time an ideal sample with a 100% fill percentage was obtained. The samples were then post-processed and cut into half so that they can be tested using a tensile testing machine.

Figure 48: Sample obtained with center injection without outer frame

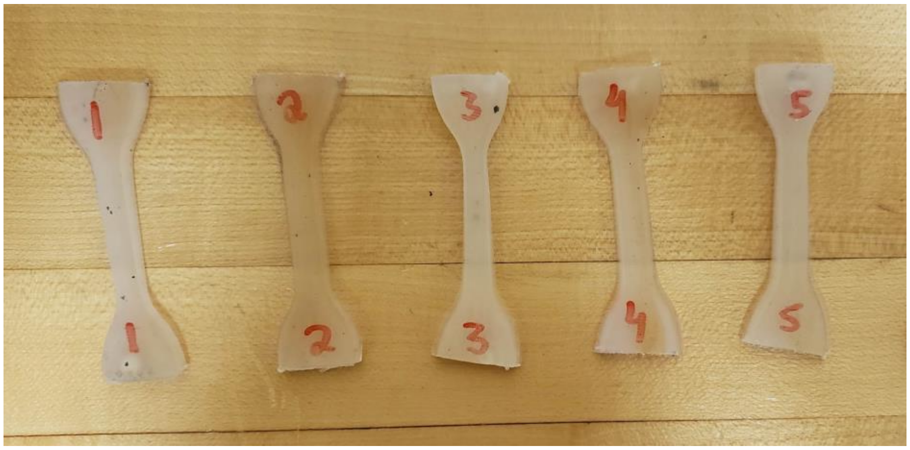


Figure 49: Samples cut in half for tensile testing

CHAPTER 6

MECHANICAL PROPERTIES

To gauge the mechanical performance of the parts produced using the silicone mold tool and compare them with those produced using a metal mold tool, tensile tests were conducted on the samples. In total, 5 tensile samples were produced using a two-piece standard metal mold made from Aluminum alloy 6061 and 5 samples were produced using the silicone mold with center injection. The tensile tests were carried out as per ASTM D638 standard on the Applied Testing Systems (ATS) Universal Testing Machine. All the tests were performed at a cross head speed of 5 mm/min using a 500 lbs (226.8 kg) load cell. Jaw grippers without extensometers were used for holding the tensile samples.

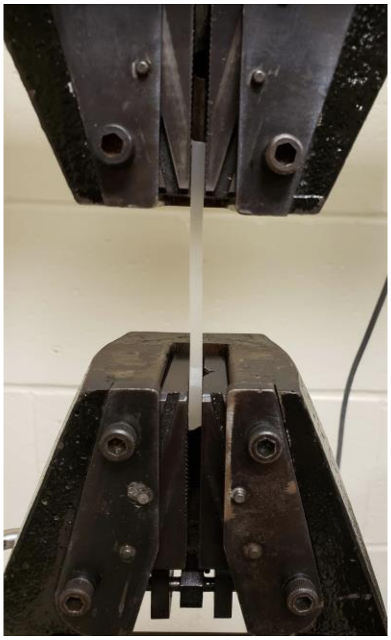


Figure 50: Jaw grippers on the ATS tensile testing machine holding the sample in place

Using a digital vernier caliper, the neck length and cross section area of each sample were measured. Raw data in the form of load vs. displacement was obtained from the tests and converted to stress and strain values and plotted using MATLAB.

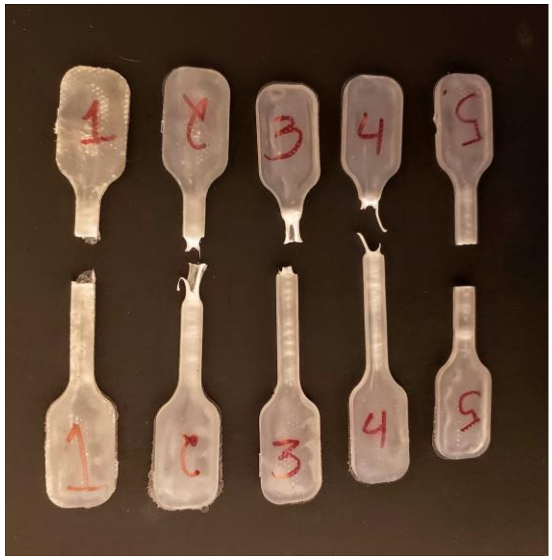


Figure 51: Metal mold samples after tensile testing

Table 17: Mechanical properties comparison with standard deviations

Property	Metal Mold	Silicone Mold
Ultimate Tensile Strength (MPa)	32.197 ± 0.527	29.854 ± 0.912
Young's Modulus (MPa)	360.255 ± 14.41	142.511 ± 31.75
Yield Strength (MPa)	18.194 ± 0.242	11.935 ± 1.158

Table 18: Mechanical properties comparison with a 95% confidence level

Property	Metal Mold	Silicone Mold
Ultimate Tensile Strength (MPa)	32.197 ± 0.462	29.854 ± 0.801
Young's Modulus (MPa)	360.255 ± 12.64	142.511 ± 27.83
Yield Strength (MPa)	18.194 ± 0.213	11.935 ± 1.016

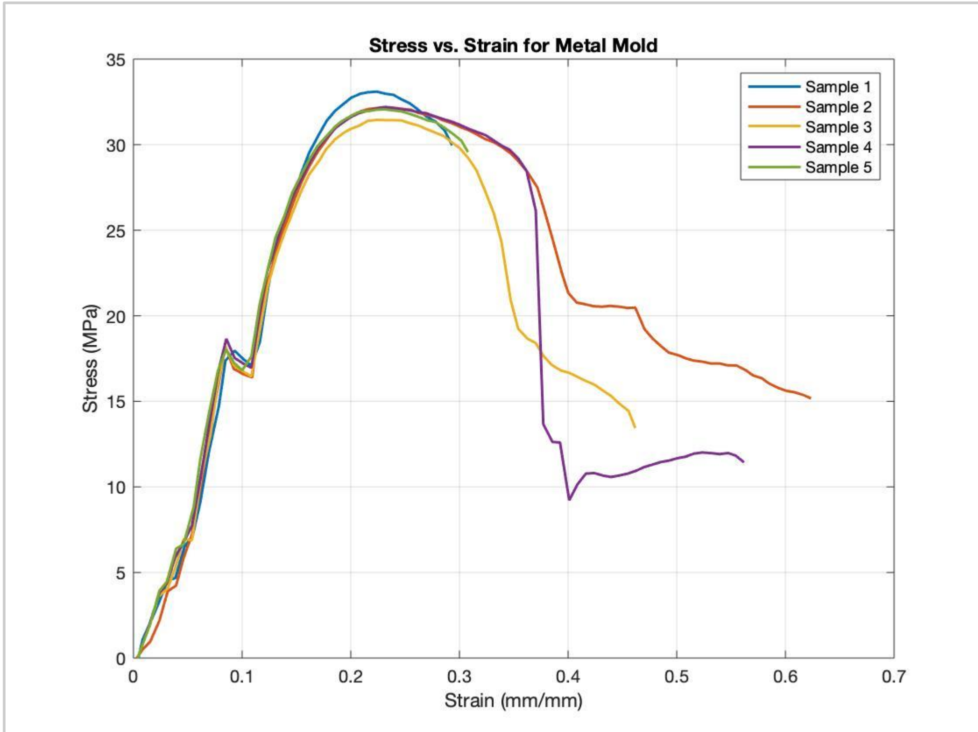


Figure 52: Stress vs. Strain plot for metal mold tool

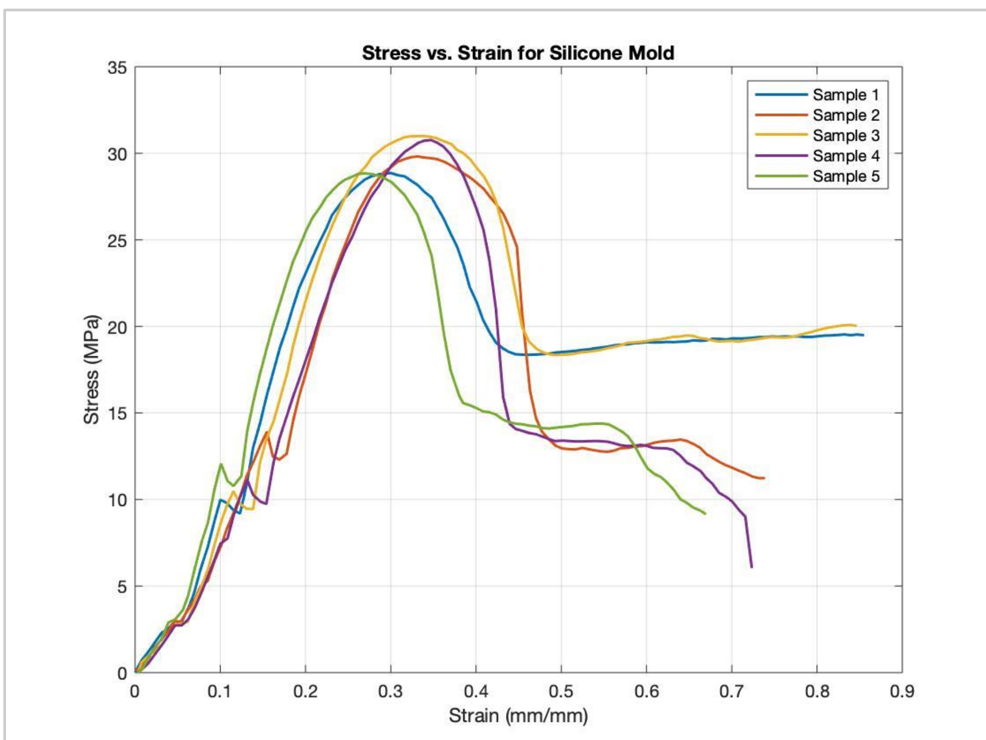


Figure 53: Stress vs. Strain plot for silicone mold tool

To verify the tensile test results, the UTS values of the PP samples obtained from the metal mold were compared with those present in online databases. In total three sources were used as references. The comparisons showed that there was a 2.4%, 3.7% and 8.0% error in UTS respectively for each of the three sources [35][36][37]. Overall, the tensile tests of metal mold samples showed a better consistency than the silicone mold samples. This can be quantitatively assessed by calculating the standard deviation of UTS of each sample set, where the metal mold and silicone mold samples showed a standard deviation of 0.461 and 0.801 respectively at the 95% confidence level (Table 18).

Comparing the average UTS, it was observed that there was a 7.3% decrease for the silicone mold tool samples (Figure 56). This is less than the ones reported in the literature for AM and composite molds, where in some studies, the decrease in UTS was as high as 11% and 21% [8] [7]. Isolating the stress-strain plots for sample #3 for each mold type, it was observed that the metal mold sample had a higher Young's modulus and yield strength than the silicone mold sample (Figure 54). On average, there was a 60.4% decrease in Young's modulus and a 34.4% decrease in the yield strength when going from metal to silicone mold tool.

For PP tensile samples produced using a FDM 3D printed mold tool made out of ABS Digital, Bartlett reported a 9.4% increase in Young's modulus compared to parts produced using a steel mold [8]. León-Cabezas et al. reported a 46% decrease in Young's modulus for tensile specimens produced using a Polyjet 3D printed mold tool when compared to parts produced using a metal mold tool [38]. In another study of polystyrene composite molds, Young's modulus remained unchanged [7]. The decrease in stiffness of the parts obtained using silicone mold can be explained by conducting a Differential Scanning Calorimeter (DSC) analysis to measure the crystallinity of the injection molded samples. The difference in silicone mold tool and AM material's thermal conductivity and surface roughness can lead to such a difference. After reaching the UTS, the silicone mold sample continued to elongate and had a larger necking region than the metal mold tool.

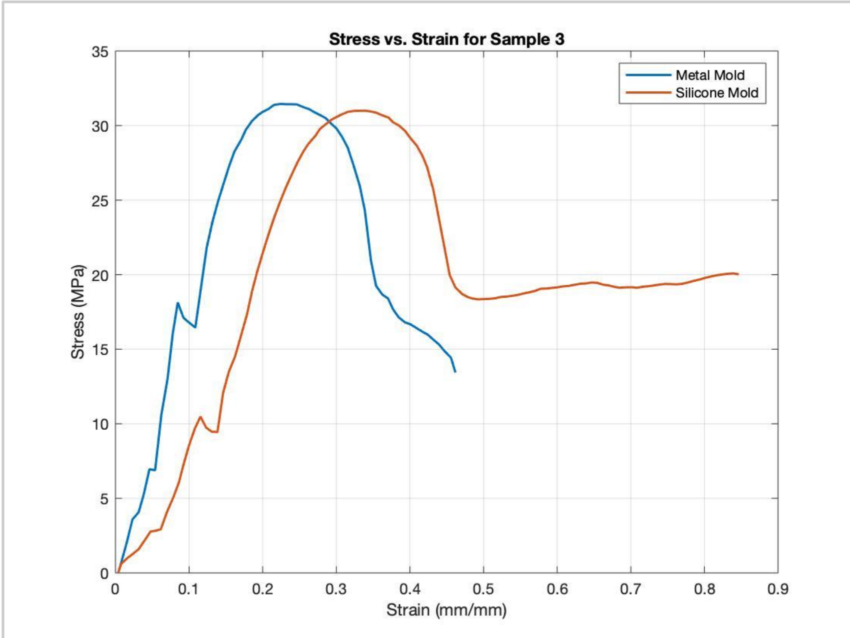


Figure 54: Comparison of Sample 3 Stress vs. Strain plots

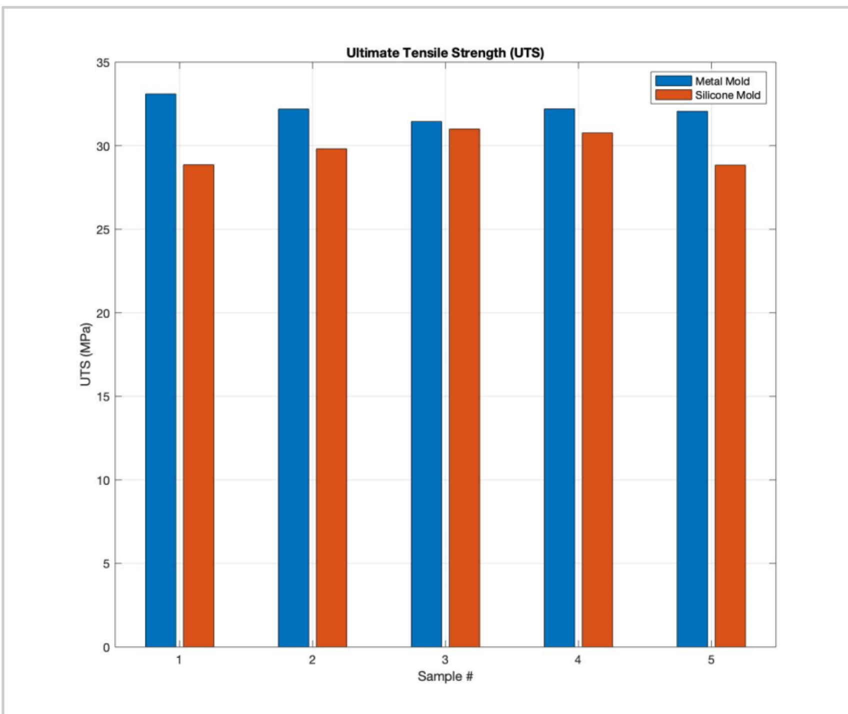


Figure 55: Comparison of UTS for all 10 samples

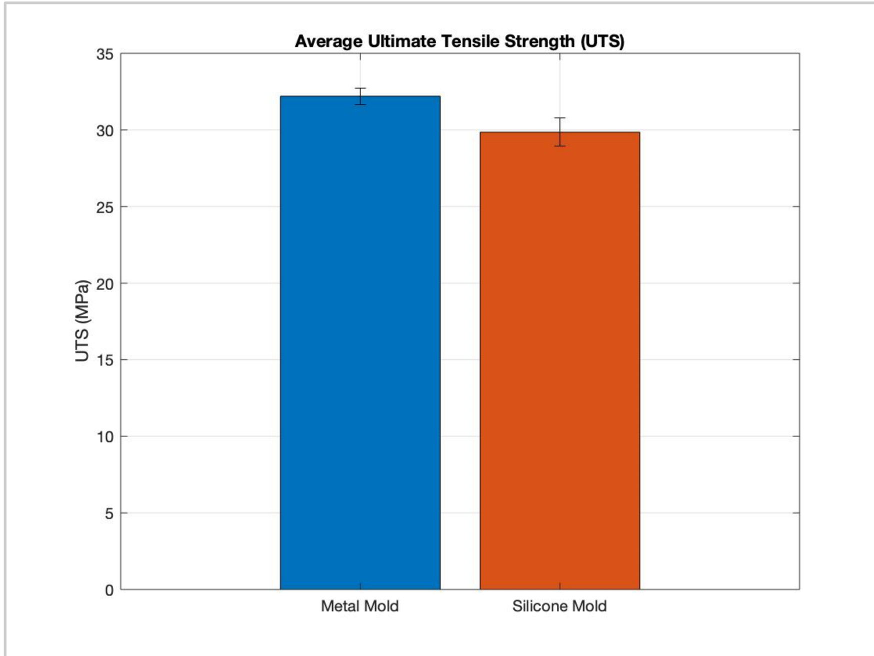


Figure 56: Comparison of average UTS for all 10 samples

CHAPTER 7

CONCLUSIONS

It has been shown that it is possible to develop a mold tool made from a flexible material like silicone to inject Polypropylene. The question that was asked at the beginning of the project was whether it is possible to inject plastics such as Polypropylene into the silicone mold tool. It was hypothesized that because of silicone's flexible nature, it will not be able to withstand the high injection pressure during the injection process. However, results show that not only silicone is able to withstand the high temperatures and pressures, but also able to resist mold deterioration at certain durometer hardness. When the durometer hardness was increased, the mold tool failed through brittle failure, similar to mold tools made out of AM plastic. A decrease in durometer hardness lead to an increase in mold deterioration and mold deformation during injection which lead to greater inconsistencies in part quality.

It was found that the plastic parts produced from silicone mold did not show many types of common failures found in injection molded samples. These include blisters, flow marks, jetting, polymer degradation, and most importantly sink marks. The injection molding simulation predicted sink marks in the runner area for Polypropylene samples, however, these were not seen in the silicone mold sample. Furthermore, the samples did not show any form of warpage, which is caused due to residual stresses in the injection-molded plastic part. It can be hypothesized that the parts produced using a silicone mold tool retained their level of crystallinity, as higher levels of crystallinity can lead to higher levels of residual stresses. As silicone is a thermoset, its thermal conductivity and heat deflection values vary drastically from additively manufactured plastics. For future work, DSC tests should be conducted to measure the level of crystallinity (or organization of molecules) in silicone mold samples.

There are limitations to using a silicone mold tool for injection molded parts. Mainly, these are related to the geometry of the mold cavity. It was found that even with the right type of durometer, the length of the runner of the mold tool can have a significant effect on whether the part will fail through flash. A longer runner design with features such as curves and corners lead to failure through flash in almost all the injection runs. When the runner was shortened to a simple straight line, it did not fail. Typically, FDM 3D printed mold tools are placed in an outer metal frame to add rigidity to the mold tool. However, it was found using this technique on a flexible silicone mold tool can restrict its deformation and forces the mold tool to curve as the plastic is injected into it with high pressure. This curvature leads to further failure through flash as the plastic leaks from the mold tool cavity. Another type of failure, through a short shot, was also observed when the injection time was too small. The most ideal sample without any kind of failures was obtained using a combination of a short runner, low injection pressure and time, high clamping pressure, and the absence of any outer frame.

The use of silicone mold tools is promising especially for material scientists and manufacturers experimenting with plastic parts in the early stages of their research. Silicone is an excellent choice for injecting materials that have the tendency to stick to other surfaces. Examples of these include medical-grade biomaterials such as PLA, PGLA, lignin, alginate, and bio-based PET. Another advantage of using a silicone mold tool to produce simple medical device components is that the mold surface does not require any type of release agent to be sprayed prior to injecting. This eliminates the risk of contamination in the medical component. Compared to most types of AM mold tools, silicone is also able to withstand higher temperatures. It has the potential to inject resins with high melting temperatures such as PEEK, PPA, and PEI. A rough cost analysis indicates that a silicone mold tool is 10 times cheaper than a metal mold tool for the same mold dimensions. A silicone mold tool is also cheaper than FDM 3D printed molds. Comparing just the material costs, it was found that silicone mold materials are 3 times cheaper per pound than the cheapest commercially available 3D printing filament. The tensile strength of parts produced using a silicone mold tool was close to those produced using a traditional metal mold tool, while the stiffness of parts decreased considerably.

Due to the COVID-19 pandemic and the subsequent closure of the university facilities, many of the planned experiments could not be completed. For future work, it is recommended that a DOE study is conducted with FDM 3D printed inserts as that proved to be a viable solution to failure through flash without compromising the mold geometry. In addition to FDM 3D printing, testing of parts with other types of additively manufactured mold tools such as Stereolithography should be implemented to gauge the performance of the type of AM. Further mechanical tests such as flexural, Izod impact, and hardness tests of the injected plastic should be conducted. Repetitions of all the DOE studies and tensile tests are vital to verify the reported values. In addition to a DSC test to measure the crystallinity of the injected parts, surface roughness of the mold tool cavity should be measured to analyze the effect of friction on the plastic flow.

Using the silicone mold tool, researchers can to continue using injection molding for their prototypes instead of reverting to other techniques such as casting, AM, or compression molding. For design projects where the final part will be made using injection molding, the silicone mold tool encourages consistency in the design process. In conclusion, this project can be expanded further by conducting said tests and experimenting with more complex geometries so that the gap between preliminary research and production of a part can be filled.

REFERENCES

[1] Groover, Mikell P. “Ch. 13 Shaping Processes for Plastics.” *Fundamentals of Modern Manufacturing*, 5th ed., Wiley, 2012, pp. 316–326.

[2] Viana, J C., and N M. Alves. “Morphology and Mechanical Properties of Injection Molded Poly(Ethylene Terephthalate).” *Polymer Engineering and Science*, vol. 44, no. 12, 2004, pp. 2174–2184., doi:10.1002/pen.20245.

[3] “Injection Molding: the Manufacturing & Design Guide.” *3D Hubs*, www.3dhubs.com/guides/injection-molding/#the-basics.

[4] Rogers, Tony. “Everything You Need To Know About Injection Molding.” *Everything You Need To Know About Injection Molding*, 21 Dec. 2015, www.creativemechanisms.com/blog/everything-you-need-to-know-about-injection-molding.

[5] Chua, C. K., et al. “Rapid Tooling Technology. Part 1. A Comparative Study.” The International Journal of Advanced Manufacturing Technology, vol. 15, no. 8, 1999, pp. 604–608., doi:10.1007/s001700050108.

[6] Mendible, Gabriel Antonio, et al. “Comparative Study of Rapid and Conventional Tooling for Plastics Injection Molding.” *Rapid Prototyping Journal*, vol. 23, no. 2, 2017, pp. 344–352., doi:10.1108/rpj-01-2016-0013.

[7] Damle, M., Mehta, S., Malloy, R., McCarthy, S., 1998, “Effect of Fibre Orientation on the Mechanical Properties of an Injection Molded part and a Stereolithography-Insert Molded Part”, *Proceedings of the society of plastics engineers (SPE) Annual Technical Conference (ANTEC), Atlanta, USA*, pp 584 - 588.

[8] Bartlett, Leah. "A Preliminary Study of Using Plastic Molds in Injection Molding." Electronic Thesis or Dissertation. Ohio State University, 2017. *OhioLINK Electronic Theses and Dissertations Center*.

[9] Simpson, Patrick, et al. “Injection Molding with an Additive Manufactured Tool.” *Polymer Engineering & Science*, vol. 59, no. 9, 2019, pp. 1911–1918., doi:10.1002/pen.25192.

[10] Dawson, E K, and J D Muzzy. "The Effect of Rapid Prototype Tooling on Final Product Properties." *Proceedings of the Society of Plastics Engineers (SPE) Annual Technical Conference (ANTEC)*, vol. 1, 6 May 1999AD, pp. 456–460.

[11] Mischkot, M., Tosello, G., Nielsen, D. K. Y., Pedersen, D. B., Zhang, Y., Hofstätter, T., ... Hansen, H. N. (2017). "Injection Moulding Pilot Production: Performance Assessment of Tooling Process Chains Based on Tool Inserts Made from Brass and A 3d Printed Photopolymer." *Proceedings of ANTEC 2017 Society of Plastics Engineers.*

[12] Kamal, M. R., et al. "Residual Thermal Stresses in Injection Moldings of Thermoplastics: A Theoretical and Experimental Study." *Polymer Engineering & Science*, vol. 42, no. 5, 2002, pp. 1098–1114., doi:10.1002/pen.11015.

[13] Rosato, Dominick V., et al. *Injection Molding Handbook*. Springer Science, LLC, 2000.

[14] Aluru, Rajitha, et al. "Simulation of Injection Molding into Rapid-Prototyped Molds." *Rapid Prototyping Journal*, vol. 7, no. 1, 2001, pp. 42–51., doi:10.1108/13552540110365153.

[15] Hopkinson, Neil, and Phill Dickens. "Predicting Stereolithography Injection Mould Tool Behaviour Using Models to Predict Ejection Force and Tool Strength." *International Journal of Production Research*, vol. 38, no. 16, 2000, pp. 3747–3757., doi:10.1080/00207540050175987.

[16] Dupret, F., et al. "Modelling and Simulation of Injection Molding." *Rheology Series Advances in the Flow and Rheology of Non-Newtonian Fluids*, 1999, pp. 939–1010., doi:10.1016/s0169-3107(99)80012-4.

[17] Guo, Jianxin, and Kwabena A. Narh. "Computer Simulation of Stress-Induced Crystallization in Injection Molded Thermoplastics." *Polymer Engineering & Science*, vol. 41, no. 11, 2001, pp. 1996–2012., doi:10.1002/pen.10896.

[18] Kim, J. K., and C. S. Lee. "Fatigue Life Estimation of Injection Mold Core Using Simulation-Based Approach." *International Journal of Automotive Technology*, vol. 14, no. 5, 2013, pp. 723–729., doi:10.1007/s12239-013-0079-y.

[19] Zhang, Jun-Ying, and Mark J. Pellerite. *Silicone Mold and Use Thereof.*

[20] Spontak, Richard J, and Nikunj P Patel. “Thermoplastic Elastomers: Fundamentals and Applications.” *Current Opinion in Colloid & Interface Science*, vol. 5, no. 5-6, 2000, pp. 333–340., doi:10.1016/s1359-0294(00)00070-4.

[21] Liu, X. Dong. *Functional Materials and Biomaterials*. Springer, 2007.

[22] Callister, William D., and David G. Rethwisch. *Materials Science and Engineering: an Introduction*. Wiley, 2018.

[23] “ASTM D412 - 16.” *ASTM International - Standards Worldwide*, https://www.astm.org/Standards/D412.

[24] “About Us.” *SmoothOn*, www.smooth-on.com/page/aboutus/.

[25] “ASTM D638 - 14.” *ASTM International - Standards Worldwide*, www.astm.org/Standards/D638.

[26] Hassan, Hamdy, et al. “Modeling the Effect of Cooling System on the Shrinkage and Temperature of the Polymer by Injection Molding.” *Applied Thermal Engineering*, vol. 30, no. 13, 2010, pp. 1547–1557., doi: 10.1016/j.applthermaleng.2010.02.025.

[27] Mold, Midstate. “Most Common Thermoplastics Used in Injection Molding - Midstate Mold.” *Midstate Mold & Engineering*, 6 Feb. 2018, www.midstatemold.com/common-thermoplastics-injection-molding/.

[28] Tahir I, Mahmood S, Rapinac J, Menta V. “Thermo-Mechanical Characterization of Polypropylene-Tobacco Lignin Blends.” *International Conference on Advanced Materials Science and Engineering.* 2019; Hawaii.

[29] Mahmood S, Tahir I, Menta V. “Thermo-Mechanical Characterization of HDPE-Tobacco Lignin Blends.” *The Composites and Advanced Materials Expo.* 2019; California.

[30] “Experiment.” *Merriam-Webster*, Merriam-Webster, www.merriam-webster.com/dictionary/experiment.

[31] Wagner, John R., et al. “Design of Experiments.” *Extrusion*, 2014, pp. 291–308., doi:10.1016/b978-1-4377-3481-2.00025-9.

[32] Dowlatshahi, Shad. “An Application of Design of Experiments for Optimization of Plastic Injection Molding Processes.” *Journal of Manufacturing Technology Management*, vol. 15, no. 6, 2004, pp. 445–454., doi:10.1108/17410380410547852.

[33] “Seven Basic Tools of Quality.” *What Is Six Sigma? – Certification, Training, Lean*, www.whatissixsigma.net/7-qc-tools/.

[34] Rogers, Tony. “Top-10 Injection Molding Defects And How To Fix Them.” *Top-10 Injection Molding Defects And How To Fix Them*, www.creativemechanisms.com/blog/what-cause-injection-molding-defects-and-how-to-fix-them.

[35] “Polypropylene (PP).” *MakeItFrom.com*, 8 Sept. 2018, www.makeitfrom.com/material-properties/Polypropylene-PP-Copolymer.

[36] “Polypropylene (PP) Mechanical Properties at 23 °C.” *Polymer Database*, polymerdatabase.com/Commercial%20Polymers/PP.html.

[37] “Typical Properties of Polypropylene.” *Tap Plastics*, www.tapplastics.com/uploads/pdf/Polypropylene_Data.pdf.

[38] León-Cabezas, M.a., et al. “Innovative Advances in Additive Manufactured Moulds for Short Plastic Injection Series.” *Procedia Manufacturing*, vol. 13, 2017, pp. 732–737., doi:10.1016/j.promfg.2017.09.124.