Injection Molding: Process and Design Principles for 3D Printed Molds

Nora Bradley, ‘19

Submitted to the
Department of Mechanical and Aerospace Engineering
Princeton University
in partial fulfillment of the requirements of
Undergraduate Independent Work.

Final Report

May 1, 2019

Michael Littman
Luigi Martinelli
MAE 442
Senior Thesis
92 pages

Additional files:
bit.ly/thesis-grabcad

This project represents my own work in accordance with University regulations.

Abstract

Injection molding is a powerful manufacturing method used to create consistent, low-cost parts ranging in size from micro-electronic connectors to automotive body panels. While injection molding is ubiquitous in industrial, high-volume applications, the large upfront cost of equipment, design work and molds had previously prevented smaller businesses working with short timeline projects or limited budgets from using this method. However, advances in rapid manufacturing methods have improved the agility and cost of mold fabrication. Although these molds have a shorter lifetime than traditional aluminum or steel molds, they are still useful in a variety of applications.

Princeton University's Mechanical and Aerospace Engineering (MAE) Department owns a BOY 35E Procan Alpha injection molding machine which has been largely unused since it was installed in 2016. This report describes the process required to successfully operate the BOY 35E, guidelines for mold design, and a comparison of FDM and SLA 3D printing methods for mold manufacturing. The results of this comparison demonstrate that SLA printing is an inexpensive, quick and relatively user-friendly method for creating molds. This comparison was performed on a mold for an involute gear used in the gear train of a flywheel car project in Princeton University's required undergraduate course MAE 321: Engineering Design. My objective was to reduce the time and effort required for students to learn how to use injection molding by providing exposure to this technique in an academic setting. I also created a process-oriented operation and troubleshooting manual for the BOY 35E based on my own experiences with the machine to further support future users.

My project has improved the accessibility of Princeton's BOY 35E machine and shown that injection molding using SLA molds can produce a high quality product in a shorter period of time and for a lower cost than traditional mold manufacturing methods. These molds are ideal for low-volume and prototyping applications that are typical of student work. The application of these methods in a small business setting will facilitate agile and low cost product manufacturing using injection molding that was previously inaccessible to these entities.

Acknowledgements

First, I would like to express my gratitude to Michael Littman for his excitement and encouragement throughout this project. I would also like to thank Al Gaillard, Glenn Northey and Jon Prevost for their constructive suggestions and willingness to help me troubleshoot. Other members of the MAE department were instrumental to the completion of this project, particularly Jo Ann Kropilak-Love, Linda Park and the purchasing department - thank you for your tireless logistical support.

I was lucky to benefit from the industrial expertise of James Smith, Bob Bennet, Stephen Bull, and instructors at the Midlands Technical University in Columbia, South Carolina on technical components of this project. I am also grateful to Mark Scerbo for laying the groundwork for this project during his Independent Work in Spring 2018.

Materials for this project were purchased thanks to generous financial support from Morgan W. McKinzie '93 Senior Thesis Prize and the Mechanical and Aerospace Engineering Department.

Thank you to my friends and loved ones for the tremendous support throughout this project and over the last five years. To my Clockwork sisters, Sarah, Iris, Kim, Tuna and Bouch, thank you for always being available for rants, a hug, or an upline throw and for making sure I leave practice with a smile on my face. Nick, thank you for your patience, advice and willingness to help on this project. Everything is better with you by my side.

Finally, thank you to my loving family - I would not be here today if you had not been supporting me every step of the way. I am so grateful that I have you to lift me up during the hard moments and celebrate with me after the successes.

Contents

List of Figures

1 Introduction

1.1 Overview

Injection molding is a manufacturing process in which a thermoplastic is heated, mixed and injected into a mold. When the molten plastic cools, it creates a part in the shape of the mold cavity. The process was first patented by Isaiah Hyatt in 1872 and is still widely used to create low-cost parts ranging in size from miniature electronic connectors to automotive body panels [5]. Injection molding allows manufacturers to mass produce precise and consistent parts in a short period of time with relatively little waste. This method is economical compared to similar manufacturing methods like machining, transfer molding, and casting. However, the high initial cost of equipment, design work and mold manufacturing are an obstacle for companies that do not have a large source of capital [17]. New advances in injection molding technology have begun to improve the agility of the injection molding process. For example, desktop sized injection molding machines are relatively less expensive, more energy efficient and produce parts faster than full sized machines. Computer-based mold manufacturing techniques have reduced the lead time for a mold from 12-16 weeks to just a few days, and fabrication companies are increasingly able to create less expensive, low volume molds using materials such as aluminum, brass, copper and even plastic [5]. These technological developments have made injection molding more accessible for small businesses and prototyping applications.

A BOY 35E Procan Alpha injection molding machine was installed in Princeton University's MAE (Mechanical and Aerospace Engineering) machine shop in the Summer of 2016 in order to allow students and faculty to use injection molding for research and independent work. However, the machine had not been used since its installation due to maintenance problems related to the setup of the machine and because the machine did not come with a cradle, the part of the machine which allows molds to be easily installed and removed. By the time these issues had been resolved, a significant amount of time had passed since BOY representatives had demonstrated how to use the machine and the documentation from the training was not sufficient to reconstruct the injection molding process. Users were not able to easily figure out how to use the machine based on the included manual because, although it independently defines the functionality of each button, it does not establish a process that should be followed to setup or complete an injection molding run.

The objective of this project was to make injection molding more accessible to members of the MAE Department. First, I learned about the process of injection molding and how this is performed on the BOY 35E machine. This included research and hands-on experimentation with the machine to understand how the process parameters can be adjusted to achieve high quality parts. This component of the project is a continuation of Independent Work done by Mark Scerbo ('18) in the Spring of 2018. I documented the operation of the machine and shared this information with Glenn Northey and Al Gaillard in person and in the form of a written operating manual, which is included as Appendix A of this report. After becoming comfortable with the operation of the machine, I investigated the feasibility of using rapid prototyping methods available to Princeton students to quickly and inexpensively manufacture molds for use on the BOY 35E. The mold that I designed to test this method can be used to create a gear for the "flywheel car" project completed in the required departmental course, MAE 321: Engineering Design. If students create gears for this project by injection molding into a 3D printed mold they will gain valuable experience that will allow them to use this technique for future projects and independent work.

This project was inspired by my experience as a Mechanical Engineering intern at CityTaps, a social entrepreneurship startup based in Paris, France. I worked at CityTaps between June 2017 and May 2018, during which time I focused on the industrialization of the company's product, a smart water meter. The device is currently being piloted in cities in developing countries, such as Niger and Burkina Faso. The product integrates a physical water meter as well as Internet of Things (IoT) communication technology [8]. In order for these components to be protected from the elements as well as tampering and fraud, it was necessary to enclose them in a secure but inexpensive casing. CityTaps determined that injection molding was the best method to make these casings, but struggled with the implementation of this process. It was challenging for a small company to finance the large upfront investment required for mold design and manufacturing. CityTaps was also not ready to make this investment because they planned to make changes to the product design based on the outcomes of the pilot project implementations. If rapid prototyping methods had been used to manufacture the casing molds, CityTaps could have saved money on the molds for the first production run of their smart water meter while preserving the ability to modify the casing design in the future. As rapid prototyping methods like 3D printing continue to develop, injection molding will become a more agile and affordable tool for prototyping and small scale manufacturing.

1.2 Process Fundamentals

1.2.1 Injection Molding Machine

Injection molding machines vary in size from manually operated tabletop models to massive, industrial machines with the capacity to churn out thousands of parts a day. However, all modern plastic injection molding models include the same basic components. Pellets of a thermoplastic material are stored in a hopper and dispensed into an injection chamber which is surrounded by heating bands. The plastic is softened by the heaters and mixed by a large screw. During the injection step of the process, the screw pushes the material out of the nozzle located at the front of the chamber and into the mold.

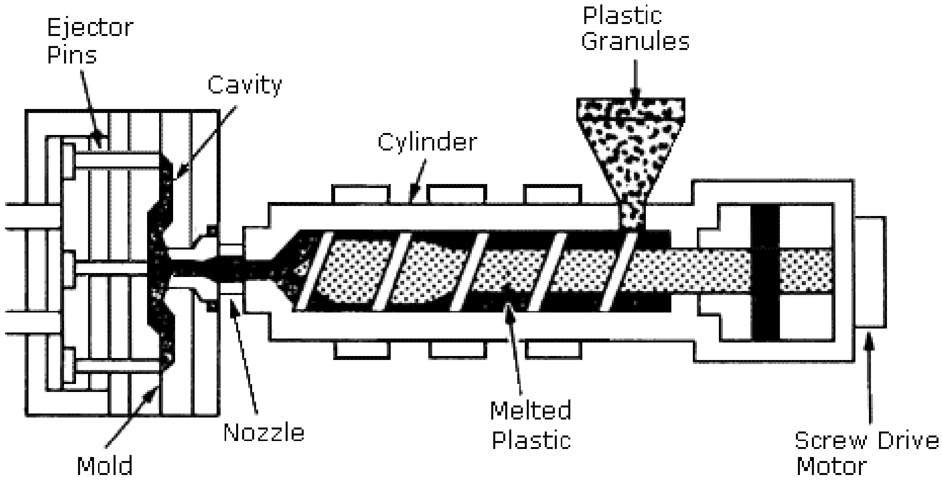


Figure 1.1: Schematic drawing of injection molding machine.

Each side of the mold is mounted on a cradle and platen, connecting it to the machine. The side of the mold through which the plastic is injected is stationary while the other side can be moved forward and backward. A hydraulic or electric clamping unit, located behind the moving platen, holds both sides of the mold firmly closed when the plastic is injected. The plastic remains in the mold under pressure until it has solidified, at which point the mold opens and the part will be removed from the machine. Typically the moving side of the mold is concave and the stationary side is convex. The cycle time is defined as the total time required to complete the injection molding process once. In industrial applications manufacturers aim to minimize cycle time in order to increase efficiency and profitability.

1.2.2 Mold

Molds vary in complexity from a single cavity to a sophisticated assembly of moving plates and pins. Mold design can be challenging for a novice engineer and it is difficult to modify a mold after it has been made. Therefore, creating molds using 3D printing methods, which are faster and less expensive than the traditional method of using a Computer Numerical Control (CNC) router, can dramatically reduce the level of risk for manufacturers.

Depending on the size of the part and rate at which it will be manufactured, some molds may have multiple cavities connected by channels called runners. Runners connect to the cavities at one or more gates, which are intended to make the final product easy to remove from the runner without inhibiting the rate at which the cavity fills. Plastic left in the sprue, the channel connecting the nozzle and the mold, also must be removed from the completed part. These parts are labeled in Figure 1.2.

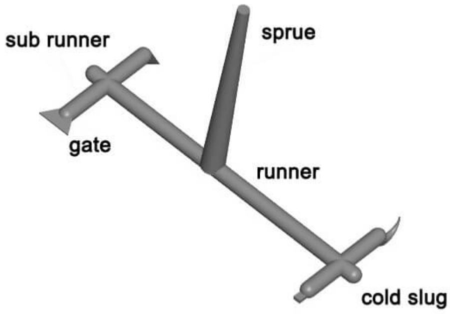


Figure 1.2: Gate, runner and sprue after removal from an injected molded part [30]

The intersection of the two sides of the mold is called the parting line. The majority of the image to be molded is typically in the B (moving) side. Since the cooled part remains in this side when the mold is opened, it also contains the ejection system. Ejector pins are used to apply a force to the completed part to push it out of the mold, but may leave marks on the final product. Guide pins mounted on the A side that correspond to bushings on the B side may be used to ensure that the mold is properly aligned [6]. The mold parts are labeled in Figure 1.3.

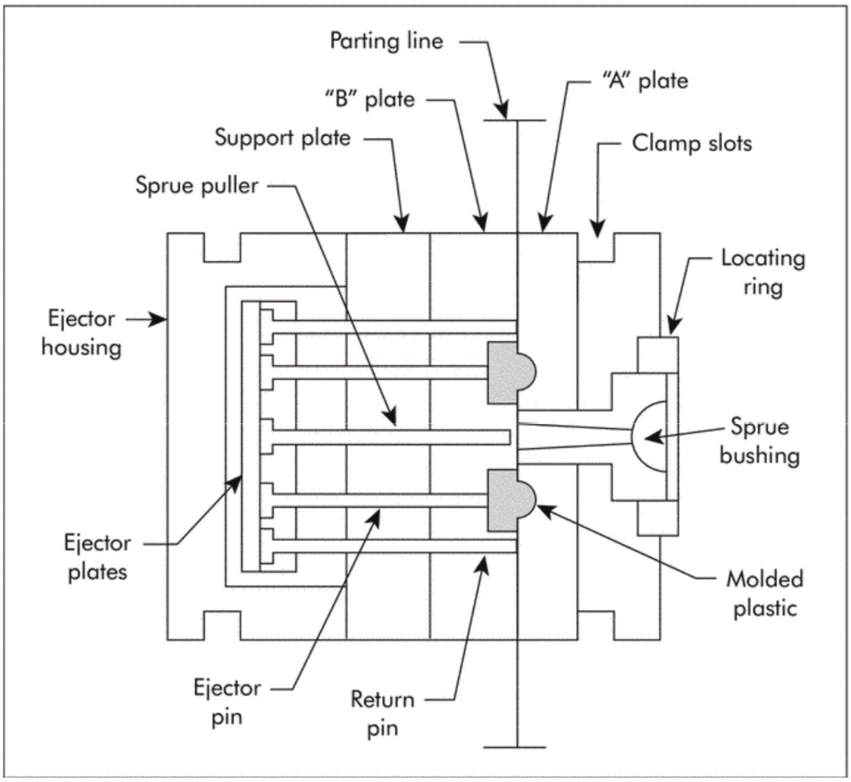


Figure 1.3: Mold Schematic [6].

1.2.3 Polypropylene

Polypropylene (PP) was the only material used for injection molding in this project. It is an engineering grade thermoplastic that is widely used in manufacturing of consumer products (Figure 1.4). PP is well suited to injection molding due to its low cost, high elongation and wear resistance [28]. PP is a semi-rigid material (a 92 on the Rockwell R hardness scale) although a part's resistance to a compressive force varies based on the design of the part [23]. PP has relatively high shrinkage properties and voids may occur if the walls of a part are too thick, leading to shrink marks [28].

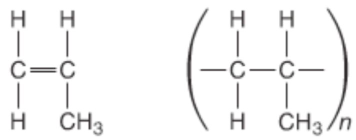


Figure 1.4: Polypropylene Molecule [12]

2 Machine Operation

2.1 Past Work

In the BOY 35E, molds are attached to a cradle which is mounted on the platen. The cradles make it easier to change out the mold, create space for the ejection system, and ensure that the A and B sides of the mold touch. The space between the two molds must be between 200 and 500 mm (7.89 - 19.7 in). When the machine was installed in the MAE shop, representatives of the company brought a set of cradles and molds in order to perform a demonstration on the new machine. BOY representatives lent these parts to Gaillard, who modeled the design in Creo before returning them to BOY. Gaillard completed the manufacturing of these parts in Spring 2018. A full model of the cradle assembly is available on GrabCad at http://bit.ly/thesis-grabcad or on the Princeton MAE Server in the folder (\\mae-data\design\injection-thesis).

Note that since the BOY 35E machine was manufactured in Germany the injection machine settings are expressed in SI units. However, the cradle was designed using English units and hardware. Therefore, measurements related to the cradle and mold will be expressed in English units.

2.1.1 MAE Beverage Coaster

Gaillard designed a mold in the shape of an MAE-themed beverage coaster to be used in the injection molding machine. The design of the beverage coaster features images of the Princeton crest, an airfoil, calipers and micrometers all within the footprint of an involute-profile gear (Figure 2.1). These beverage coasters were intended to be produced in large quantities and used as department giveaways to visitors, prospective students and supporters. In the Spring of 2018, Mark Scerbo had the beverage coaster mold manufactured in aluminum and fabricated an ejection system which was compatible with this mold as part of his Independent Work [27].

Figure 2.1: MAE beverage coaster.

The profile of the beverage coaster is defined in the B side of the mold, a 5 inch by 5 inch in by 0.210 inch piece of aluminum stock that is mounted in a 5 inch by 5 inch cutout in the cradle. The A side of the mold is a ground flat plate with a sprue hole for the plastic to flow through between the nozzle and cavity. The sprue expands in diameter from 0.082 inch on the side of the injection nozzle to 0.166 inch at the gate, based on sizing advice from Bryce's *Plastic Injection Molding, Volume I* [5]. Creating a one-sided mold eliminates the need for locating pins to ensure that the two sides of the mold are aligned and reduces the cost of the mold because only one complex piece needs to be manufactured. Scerbo estimated that the two sides of the mold could be up to $\frac{1}{4}$ inch out of alignment in any direction without a negative effect on the completed part. The sides of the mold cavity are not drafted, but are chamfered by about 0.005 inch to allow the beverage coaster to be removed from the mold more easily. Although at least 1° of draft angle is ideal for vertical features, the effect of the draft angle would be negligible because the mold is so shallow (0.05 inch). [27].

Scerbo initially intended to use the MAE Department's Bantam Tools CNC router to machine the mold, but he ultimately found that this was impossible because the aluminum stock was too large for the 4.5 inch x 5.5 inch working area of the mill [27]. The mold design was sent to Protolabs, a company which manufactures high-quality parts using various methods. Protolabs was able to deliver a CNC machined aluminum mold within a week [27]. The complications that Scerbo faced in fabricating the MAE beverage coaster mold are evidence of the need for quick and inexpensive mold manufacturing methods, such as the 3D printing techniques I investigated in this project.

2.1.2 Ejector Assembly

The B side of the cradle was outfitted with an ejection system consisting of a retaining plate supporting three threaded rods (Figure 2.2). The $\frac{1}{8}$ inch ejector pins are press fit into the threaded rods and the connection between the threaded rods and the retaining plate can be adjusted to ensure that all three pins are well aligned with the mold. This assembly is attached to the ejection actuator of the BOY 35E via a central threaded rod, ensuring that all three pins are advanced and retracted simultaneously. The ejection pins are located on featureless areas of the beverage coaster to avoid distorting the design with ejection markings. The mold was reamed for $\frac{1}{4}$ inch press-fit bushings which help align the $\frac{1}{8}$ inch ejection pins and reduce wear on the mold.

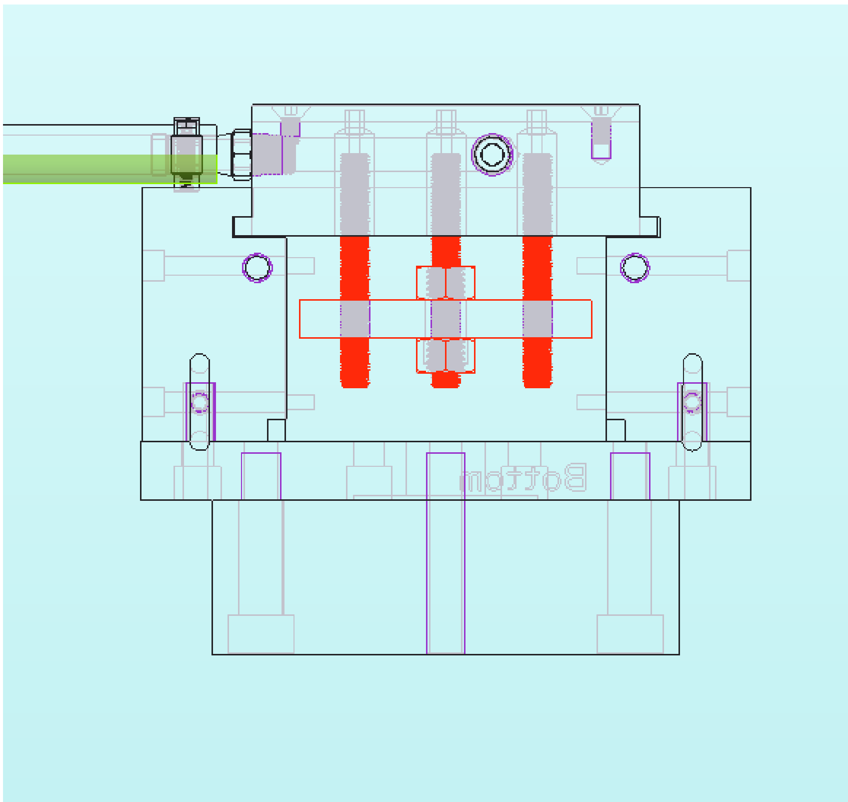


Figure 2.2: CAD model of the cradle and ejection system.

2.1.3 Machine Setup and Injection Molding

Scerbo was not able to complete a run of the BOY 35E machine before the end of his project, but his report provided some insights regarding how the injection molding machine is turned on and how basic setup operations are performed [27]. The first step in this project was to learn how to complete the injection molding process using the BOY 35E and how to optimize the machine settings for different molds. This process was supported by Northey and Gaillard, who had received some training on the BOY 35E when it was first installed in the shop. We were assisted by Bob Bennett, an engineer with almost 30 years of experience in the plastics industry at companies such as Mattel, Polycel and Nissei.

Experimenting with the injection molding machine helped me gain experience interpreting the hieroglyphic-esque buttons on the machine's user interface as well as recognizing which machine parameters need to be adjusted based on the results of an injection run. Douglas Bryce notes in the Preface of *Plastic Injection Molding, Volume I: Process Fundamentals*, most skills in the plastics industry are "honed by making mistakes, learning from those mistakes, and plunging forward to discover other areas in which the learning process had to be repeated" [5, p. xxi]. I found that working with the machine was the most effective way to become comfortable operating it. I have summarized all of my findings regarding the setup, operation, optimization and troubleshooting of the BOY 35E in Appendix A of this report. This manual is intended to reduce the amount of time and effort necessary for future users to learn to operate the machine.

The injection runs that took place during this phase of the project were performed using the MAE beverage coaster mold. I followed the process described in Appendix A, section 4 to determine the machine parameters that should be used for this mold. The final injection parameters for the beverage coaster mold are listed in Table 2.1 and heating band temperatures are listed in Table 2.2. Key calculations are shown below. Note that the actual parameters used may differ from the calculated parameters due to adjustments made to improve poor quality parts.

Clamping Force:

For D < 22.4 mm:

$$F_{\text{clamp}} = 1.1 * \text{A}_{\text{p}} * \text{CF} \tag{2.1}$$

$$\begin{aligned} &= 1.1 * 7853.9\text{mm}^2 * 0.07\frac{\text{kN}}{\text{mm}^2} \\ &= 604\text{kN} \end{aligned}$$

Where mold depth (D) = 1.27mm, projected area A_{p} = 7,853.9 mm^2, and clamping factor (CF) = 0.07 $\frac{\text{kN}}{\text{mm}^2}$ [5].

Injection Distance:

$$d_{\text{Injection}} = (\frac{\text{V}_{\text{mold}}}{\text{A}_{\text{cylinder}}}) + \text{d}_{\text{cushion}} \tag{2.2}$$

$$\begin{aligned} &= (\frac{118000\text{mm}^3}{855\text{mm}^2}) + 6\text{mm} \\ &= 19.8\text{mm} \end{aligned}$$

Where V_{mold} is the volume of the mold, $\text{A}_{\text{cylinder}}$ is the cross sectional area of the inside of the injection cylinder (855 mm^2 for the BOY 35E), and $\text{d}_{\text{cushion}}$ is the cushion distance [5].

Table 2.1: Key Injection Parameters for Beverage Coaster Mold

Clamping Force	Injection Distance	Injection Press.	Holding Press.	Plasticizing Press.
609 kN	19.8mm	350 bar	175 bar	3.5 bar

Table 2.2: Heating Band Temperatures for Beverage Coaster Mold

Nozzle Temp	T4	T3	T2	T1	Cooling
174	170	167	164	160	29

2.1.4 Ejection System Modifications

I modified the previous ejection pin system slightly to enable the machine to be operated continuously and to reduce the marks caused by the ejection pins on the surface of the part. The $\frac{1}{4}$ inch bushings, which were intended to be press fit into reamed holes in the mold, were slightly too small, causing them to be pushed out of place each time the ejection system was triggered. The pin and bushing assembly also left very obvious marks on the completed beverage coasters because it was difficult to align the system precisely so that both components were flush with the surface of the mold. Although the bushings were initially included to help guide the ejector pins through the mold and to reduce wear, the projected lifetime use of this mold will not lead to significant wear on the pins or the mold, rendering the bushings unnecessary.

I modified the system to eliminate this problem by replacing the $\frac{1}{8}$ inch pins with $\frac{1}{4}$ inch pins that were inserted directly into the reamed holes in the mold. These changes improved the operation of the ejection pin system because the pins could be advanced and retracted without stopping the machine to replace the bushings. However, even after these improvements, I found that the ejection pins do not consistently eject the part without additional intervention from the operator. When the ejection pins are advanced, they move the part away from the surface of the cavity but do not loosen it enough to allow the part to fall freely away from the mold. Once the ejection pins have been retracted, the operator must still open the main gate and pry the outside of the beverage coaster away from the cavity. This is most likely because there is no draft angle on the outside of the mold. Since I intended to design a new mold to be used for the remainder of this project, I did not complete an in-depth analysis to troubleshoot the ejection pin system and the amount of force required to effectively eject the part. However, I believe that moving the ejection pins toward the outside of the beverage coaster would prevent the edges of the part from remaining in the mold after the ejection pins have been advanced.

3 3D Printed Mold Design

I worked with Northey and Gaillard to integrate the injection molding machine into MAE 321: Engineering Design. I designed and manufactured a mold for the larger of two gears in the gear train of the flywheel car (Figure 3.1) so students may injection mold one of the gears on the BOY 35E machine. This will allow students to learn about the injection molding process and may encourage them to apply this technology in future projects or independent work. 3D models of all of the molds created for this theis project are available on GrabCad at http://bit.ly/thesis-grabcad or on the Princeton MAE Server in the folder (\\mae-data\design\injection-thesis)

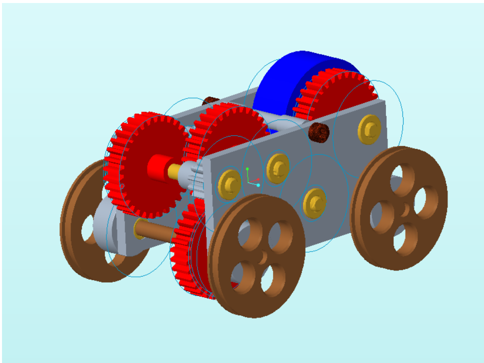


Figure 3.1: CAD model of the flywheel car

3.1 Design Considerations

Effectively designing a mold requires an understanding of the complex dynamics which occur during the injection molding process when hot plastic is forced into the mold by the screw. Failing to adhere to the design considerations discussed below may result in poor quality part. See Appendix B for Protolabs' injection molding design guide, which illustrates common design consideration pitfalls.

- Avoid undercuts: An undercut is a protruding feature of the part which prevents it from easily being ejected from the mold (Figure 3.2). Side action pins may be required to mold and eject a part with an undercut. In some situations, careful cavity orientation and placement of the parting line can eliminate undercuts. [15].
- Draft angle: Draft angle is the amount of taper on a vertical component of the cavity. Designing vertical features to be wider at the entrance of the mold and narrower at the back reduces the force required to remove a part. In general, at least 1° of draft should be used for every 1 inch of mold depth, but more should be included if possible. If draft angle cannot be included on a feature because it would inhibit the functionality of a part, this feature should be as shallow as possible [6, 15].

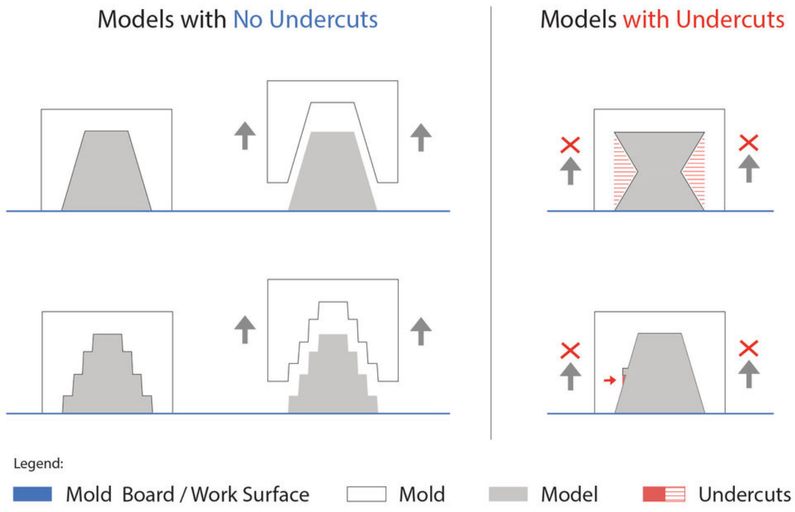


Figure 3.2: An undercut will prevent the part from being removed from the mold [15]

- Corner radii: While corner radii are not required for injection molding, sharp corners can restrict the flow of plastic through the mold and introduce molded-in stresses that could cause part failure. Therefore, corner radii should be included whenever possible. [6, 15].
- Wall thickness: Wall thickness of a part must be optimized to ensure part strength while avoiding unsightly defects associated with thick features. If an area of the part is too thick the material inside it will cool unevenly, creating voids within the plastic that will ultimately lead to an indent on the surface of the part. In extreme cases, uneven wall thickness may lead to warping of the overall piece. Eliminating thick areas by "coring out" solid features leaves behind ribs and gussets to define the geometry and to prevent structural deformation (Figure 3.3)[15].

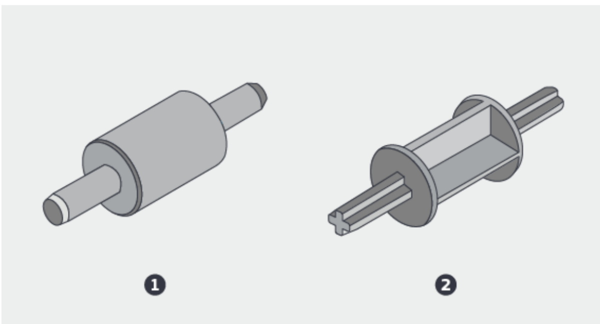


Figure 3.3: Core out part geometries to avoid sink marks due to thick plastic features [15].

- Shrinkage: Most thermoplastics shrink in size as they cool. Therefore, it is necessary to scale the size of the mold to account for any shrinkage that will occur. Table 3.1 provides a range of shrinkage values for commonly used plastics. Equation 3.2 demonstrates how the shrinkage percent can be converted to a scale factor. Some iteration may be required to ensure accurate part dimensions. This process is made significantly easier by using 3D printed molds, which can be iterated much more quickly than traditional molds. Shrinkage can help the molding process because it makes it easier to remove a part from its mold [27].

$$\%S = \frac{s_{\text{final}} - s_{\text{original}}}{s_{\text{final}}} * 100 \tag{3.1}$$

Rearranging the equation to solve for s_{final} we find:

$$s_{\text{final}} = (\frac{\%S}{100} + 1) * s_{\text{original}} \tag{3.2}$$

Where $\%S$ is the percent shrinkage, s_{final} is the desired final size and s_{original} is the required original size.

Table 3.1: Shrinkage of Commonly Used Materials

Material	Shrinkage Min.(%)	Shrinkage Max.(%)
ABS	0.70	1.60
High Density Polyethylene (HDPE)	1.50	4.00
Low Density Polyethylene (LDPE)	2.00	4.00
Polycarbonate (PC)	0.10	0.50
Polyamide (Nylon)	0.30	0.30
High Impact Polystyrene (HIPS)	0.20	0.80
Polypropylene (PP)	1.00	3.00

3.2 Gear Mold Design

The gear created by the mold in this project requires a high level of precision in order to be functional, but still has a relatively small footprint and simple overall design. I adhered to the considerations described in Section 3.1 when designing the mold. In order to simplify the mold as much as possible and to ensure that the gear teeth were accurately molded, I created a one-sided mold. The A side of the gear mold is the same flat aluminum plate that is used as the A side of the MAE beverage coaster mold. This reduced the cost and complexity of the mold because only one side needed to be manufactured and the mold does not require any alignment pins. Since the parting line is on the back of the part, any flash (excess material that flows between the two sides of the mold) can be easily removed without impacting the functionality of the gear teeth.

The mold is for an involute gear with 32 teeth which has a pitch diameter of 1.60 inch and 11.25° of circular pitch. The outer diameter of the designed gear measures 1.70 inch. In the flywheel car, the gear is fastened to a $\frac{1}{4}$ inch diameter shaft using a set screw located on the hub of the gear. It was necessary to core out the center of the gear to avoid sink marks, leaving behind only the outer wall containing the gear teeth and a center hub which will be reamed to the correct size for the gear shaft. A hole for the set screw will be drilled and tapped after the injection process is completed. Ultimately, radial ribs stretching from the hub to the outer wall were also added to the design to improve the rigidity of the gear (Figure 3.4).

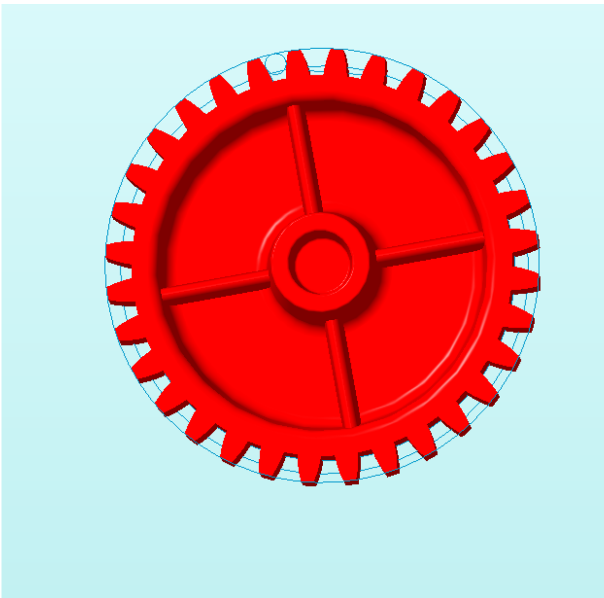


Figure 3.4: CAD model of the final gear created by the 3D printed mold.

The gear mold was designed to be used with polypropylene plastic (PP), the same material that was used for the MAE beverage coaster mold. Design guidelines dictate that the wall thicknesses for PP should be 0.025 - 0.150" [15]. This was possible for all of the walls in the part except for the wall between the inside of the hub and the back of the gear, which is 0.35 inch thick. I decided to leave additional material here at the risk of creating a sink mark in order to leave enough material to be able to drill the thru-hole for the gear shaft. I did not want to include the thru-hole in the mold because it would have interfered with the sprue and the flow of material throughout the cavity. Refer to Figure 5.4 in Appendix E for a dimensioned drawing showing the thicknesses of each of the walls in the gear.

A draft angle of 4° was included on all of the vertical surfaces of the mold cavity except for the gear teeth, which will not interlock properly if they are drafted. The shrinkage ratio for PP ranges between 1.0% and 3.0% (Table 3.1), and a middle range ratio of 1.8% was selected based on the shrinkage of the MAE coaster mold [29]. Equation 3.2 was used to calculate the required scale factor, which was found to be 1.018.

In order to create a model of the mold, I first designed the gear that I wanted to create in Creo 5.0. I used the Boolean feature to subtract the image of the gear from the rest of the mold to form the cavity. The Boolean function worked well for a simple, one sided mold but Creo's Mold Design software package should be used for more complex molds which require specifying a parting line. The mold cavity is centered on the sprue, which has a gate size of 0.166 inch. Centering the cavity on the sprue allows all injected plastic to travel approximately the same distance, minimizing the injection pressure required and reducing the risk of molded-in stresses [6]. I did not include vents to allow air to escape the cavity during injection because in my experience with the MAE beverage coaster mold the mold surfaces were not smooth enough to require additional vents. However, if vents were necessary, I planned to use a dremel to add them.

I did not design a formal ejection pin system that would be integrated into the BOY 35E for this mold because I wanted to preserve my ability to iterate this component of the design. I expected that the ejection pins would be a challenging component of the mold to design based on my experience with the ejection pin system designed for the MAE beverage coaster mold, which was not very effective even after modifications. I was most concerned that the outer gear wall would be difficult to remove from the mold because it does not include a draft angle. I experimented with a number of different ejection pin layouts throughout the mold design and testing process. Ultimately, I found that I was able to eject the pins from the mold using four large pins located close to the center of the mold as long as a mold release compound was used to create a barrier between the PP and the cavity of the mold.

4 Mold Manufacturing and Injection Testing

There are a number of rapid manufacturing techniques that may be used to create low-volume molds. I chose to focus on Fused Deposition Modeling (FDM) and Stereolithography (SLA) printing because these methods are available to MAE students on campus, therefore they provided the fastest and most affordable opportunities for iteration of the mold design. I had also considered using Selective Laser Sintering (SLS) or Polyjet printing to make molds, but these methods would have required sending mold designs to an outside manufacturer like Protolabs. This would have limited opportunities for design iteration due to the cost and time associated with having the mold manufactured off site. FDM and SLA printing are also the least expensive of the manufacturing methods I considered because they are available in tabletop "hobby-sized" machines at a relatively low price point. Therefore, it is feasible that a small business interested in using this technology might have these printers available in-house or through a communal Maker Space.

4.1 FDM Mold

Design

The first mold design I made was 5 inches by 5 inches and 1 inch thick. The mold did not include any ejection pins, but I planned to drill through the back of the plastic to create channels that could be used to push the finished product out of the mold. I was not sure if the FDM mold would be strong enough to withstand the heat and force of injection because the 4.55 bar heat deflection temperature of Acrylonitrile butadiene styrene plastic (ABS) (96° Celsius) is well below the nozzle temperature required to melt the PP (160- 180°C) [5, 4, 23]. However, I wanted to perform trials with an FDM printed mold to formally evaluate the feasibility of this method because I was not able to find documented results of injection molding with an ABS mold.

The mold was printed using a Stratasys Dimension Elite FDM printer and Stratasys P430 ABS filament and dissolvable support material. The model was oriented with the gear cavity facing upwards and printed at 100% density with a layer thickness of 0.0070 inch. After printing, the model was soaked in a boiling lye bath to dissolve the support material. This mold was not produced in SLA because Princeton's SLA printer had not been delivered to the MAE Department at this stage in the project.

Injection Parameters

I determined the injection parameters for the gear mold using the process described in Appendix A, Section 4. Parameters for injection are listed in table 2.1 and heating cylinder temperatures are listed in table 2.2. Key calculations are included below, however some of the actual parameters deviate from the calculated values. These modifications were made in response to poor part quality in injection runs.

Clamping Force

For D < 22.4 mm:

$$F_{\text{clamping}} = 1.1 * \text{PA} * \text{CF} \tag{4.1}$$

$$\begin{aligned} &= 1.1 * 1481\text{mm}^2 * 0.07\frac{\text{kN}}{\text{mm}^2} \\ &= 114\text{kN} \end{aligned}$$

Where mold depth (D) = 19.5 mm, projected area (PA) = 1481 mm^2, and clamping factor (CF) = 0.07 $\frac{\text{kN}}{\text{mm}^2}$ [5].

Injection Distance

$$d_{\text{Injection}} = (\text{V}_{\text{mold}} * \text{A}_{\text{cylinder}}) + \text{d}_{\text{cushion}} \tag{4.2}$$

$$\begin{aligned} &= (7780.5\text{mm}^3/855\text{mm}^2) + 4\text{mm} \\ &= 13.1\text{mm} \end{aligned}$$

Where V_{mold} is the volume of the mold, $\text{A}_{\text{cylinder}}$ is the cross sectional area of the inside of the injection cylinder (855 mm^2 for the BOY 35E), and $\text{d}_{\text{cushion}}$ is the cushion distance [5].

Cylinder Temperatures: The original nozzle temperature of 174° was a few degrees lower than the injection temperature recommended for PP in *Bryce's Injection Molding I: Process Fundamentals*. This adjustment was made based on the nozzle temperature used for the MAE beverage coaster mold.

Table 4.1: Key Injection Parameters for Beverage Coaster Mold

Clamping Force	Injection Distance	Injection Press.	Holding Press.	Plasticizing Press.
200 kN	13.7mm	350 bar	175 bar	3.5 bar

Table 4.2: Heating Band Temperatures for ABS Mold

Nozzle Temp	T4	T3	T2	T1	Cooling
174	170	167	164	160	29

Mold Opening/Closing: The opening and closing distance were set based on the position where the sides of the mold are pressed together. While closing, the mold moved at a speed of 120 mm/s until it was 15mm from the closed position. At this point it slowed to 7 mm/s until it reached the final position.

ABS Mold Results

The calculated injection distance of 13.1 mm slightly underfilled the mold (due to inconsistency in the cross section of the cylinder and the additional material left in the sprue). I slowly increased the injection distance up to a final value of 13.7mm, which completely filled the mold.

The outer corners of the surface of the mold were slightly curved after the first injection, perhaps due to the clamping force. Otherwise, the mold did not have any obvious changes in appearance. The injected plastic had some burned spots distributed throughout, perhaps indicating that the injection temperature was too high or that the plastic had been left in the injection cylinder for too long.

I considered the mold to be "cooled" when the PP changed in color from transparent to translucent white and felt cool to the touch. The PP was still warm and transparent in places after about 10 minutes of cooling time, so I immersed the entire mold in a bucket of cool water to speed up the cooling process. Once the mold had cooled, I attempted to remove the gear using a number of tools to lift from the corners but I was not able to move the part significantly. I attempted to drill through the back of the mold at the center and sides of the gear using a $\frac{1}{4}$ inch drill bit in order to insert pins into the holes to to push out the part. I was able to use the pin to move the center of the gear but the force was localized to the hub section of the gear and did not dislodge the gear teeth. Ultimately, the force of the pin ripped through the plastic of the gear and I was not able to remove the rest of the part from the mold.

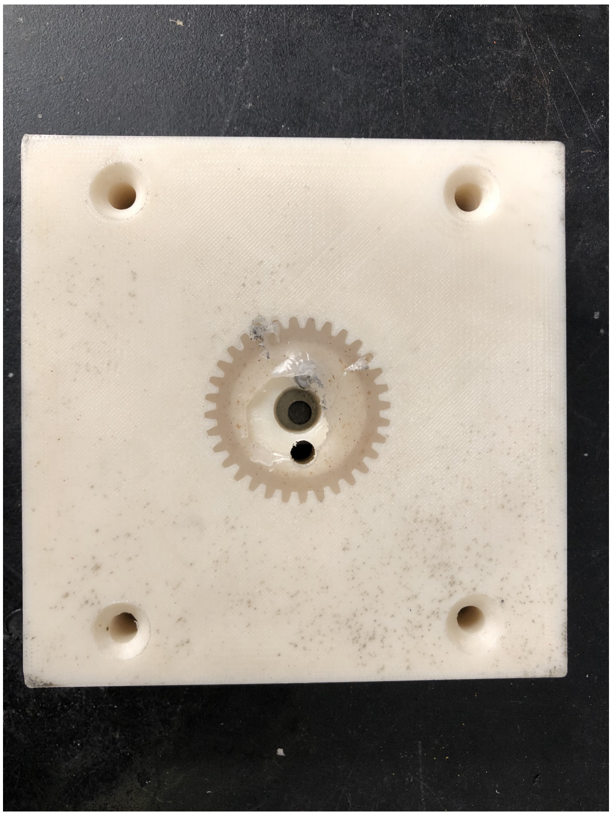


Figure 4.1: ABS mold after attempting to remove completed part using pins.

The PP part could not be removed from the ABS mold due to the surface texture of the mold and the low thermal conductivity of the plastic (Figure 4.1). The surface of the inside of the mold has visible texture caused by the layers of FDM printing, which behaves like thousands of tiny undercuts, resisting the movement of the part in the mold. It took the PP a significant amount of time to cool after injection, even after quenching in cool water. During this time, it is possible that the softening of the ABS which took place when the PP was injected, combined with the clamping and holding force, caused the mold to deform, making it more difficult to remove the gear. While it is possible that including holes for ejection pins in the mold rather than drilling them after injection may have improved the ease of removal, the long cooling time renders this mold impractical for regular use anyway. Therefore, I decided to redesign the mold to reduce the cooling time and to allow for the use of pins to push the part out of the cavity.

4.2 FDM and SLA Inserts in Aluminum Housing

Design

Although the results of the first injection run using FDM printed mold were not very promising, I decided to attempt this method again with a mold that had been modified to ameliorate the two problems that I experienced with the first mold: being unable to remove the part and a long cooling time. Based on the deformation at the corners of the mold, I also wanted to modify the mold to make it more resistant to clamping force.

I changed the mold design to consist of a 3D printed insert just large enough for the gear cavity (approximately 2 inches x 2 inches x 1 inch) inserted into an aluminum housing that would be mounted on the platen (Figure 4.2; Appendix E, Figure 5.3). This method was recommended in the Formlabs White Paper discussing injection molding from 3D printed molds[11]. The sides of the insert were drafted and rounded to allow for easy insertion and removal of the printed portion of the mold. The insert is fastened to the housing by tabs located on the top and bottom of the insert which correspond to cutouts in the housing. They are fastened with two countersunk flat head, 1/4-20, 3/8" screws. In addition to providing structural support and rapid heat conduction, the aluminum housing also reduces the size and amount of plastic used in each mold insert. The 3D printed insert preserves the ability to manufacture detailed molds rapidly and at a low cost. In fact, this iteration is more agile than the original mold because the aluminum housing can be reused for different versions of the 3D printed insert and the inserts can be made more quickly and less expensively than the full sized mold.

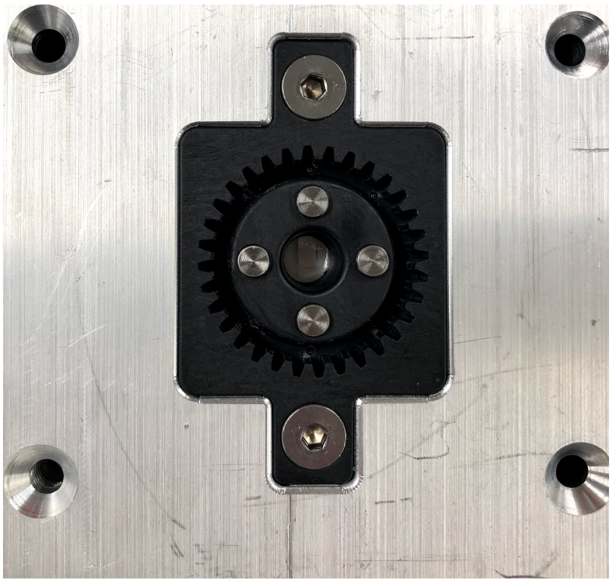


Figure 4.2: 3D printed mold and aluminum housing with inserted ejection pins.

I added holes between the base of the cavity and the back of the mold so that metal pins could be used to push out the completed part. There are five large ejection pin holes towards the center of the cavity (one 0.4" diameter hole at the center of the hub surrounded by four $\frac{1}{4}$ inch diameter holes) and eight $\frac{1}{16}$ inch diameter holes located on the flat face of the gear. Metal pins were inserted into the outer holes during molding but the center hole was not filled in order to create the hub of the gear, which would be drilled out to mount the gear to a shaft (Figure 4.2). I was not sure that the ejection pin layout would be successful so I decided to test the layout by performing the ejection manually. The layout of the pins and overall mold design was created in consultation with instructors in the program for Machine Tool Technology at Midlands Technical University in Columbia, South Carolina.

The FDM mold insert was produced in ABS using the same process and print orientation described in Section 4.1. The SLA mold insert was printed on a Form 2 printer using black standard resin and a layer thickness of 50 microns. The print was oriented and supported as shown in Figure 4.3. After printing, the mold was washed in 100% Isopropyl Alcohol (IPA) to dissolve any uncured resin and the support material was removed using flush cutters and sanding.

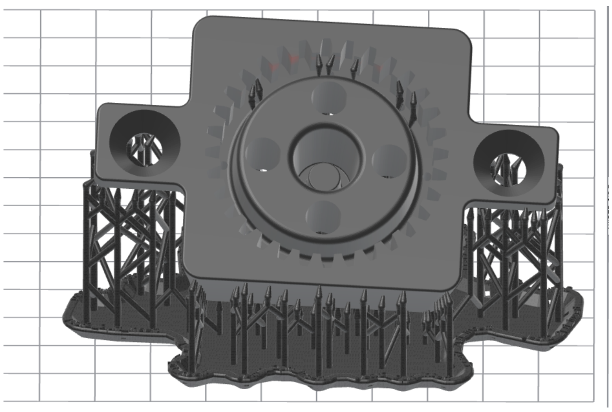


Figure 4.3: Model of SLA mold with support material

Injection Parameters

In subsequent injection molding runs, the machine parameters were substantially unchanged from the initial parameters described in Table 2.1. The nozzle temperature was further reduced from 174°C to 170°C in response to the burn marks observed in the previous injection runs. A complete list of the new heating zone temperatures is shown in Table 4.3. I also attempted to reduce the time between injection runs in order to avoid burning material in the injection cylinder.

Table 4.3: Heating Band Temperatures for SLA/FDM Mold Inserts

Nozzle Temp	T4	T3	T2	T1	Cooling
170	166	162	158	154	29

A mold release compound was used during injection to create a lubricating barrier between the cavity and the PP part in order to ease the removal of the gear from the mold. I used SPRAYON MR309, a silicone compound that is compatible with plastics including ABS, PP and manufacturing resin [11, 13]. I sprayed the mold release compound inside the cavity of the mold from a distance of 6-8" for approximately 3 seconds before each injection run. While applying the spray I angled the nozzle towards the sides of the mold where the grooves for the gear teeth are located because this wall is not drafted and therefore generates the most resistance to part removal.

FDM vs. SLA Results

After injection, the FDM mold took about 8 minutes to cool. While this is a significant amount of time for such a small part, quenching in water was not required. Once the mold had cooled, the gear was removed by inserting pins into the holes in the back of the mold and pushing them through the mold. I originally attempted to push the pins out by hand, but ultimately required the force of an arbor press on the pins to eject the part. The critical dimensions of the part were within 0.018 inch (1.1%) of the designed dimension and the teeth meshed smoothly with a metal gear.

The SLA mold cooled within approximately 3 minutes of injection and I was able to remove the gear from the mold insert by pressing pins through the ejection holes by hand. The gear came out of the SLA mold significantly more easily than the FDM mold, most likely due to the smooth surface of the mold cavity. The critical dimensions of the part were within 0.032 inch (1.9%) of the designed dimensions and the gear teeth meshed smoothly with a metal gear. All of the surfaces of the PP gear were smooth and shiny unlike the rough surface texture of the FDM mold (Figure 4.4).

Figure 4.4: Comparison of a gear produced using an FDM mold (right) and a gear produced using an SLA mold (left).

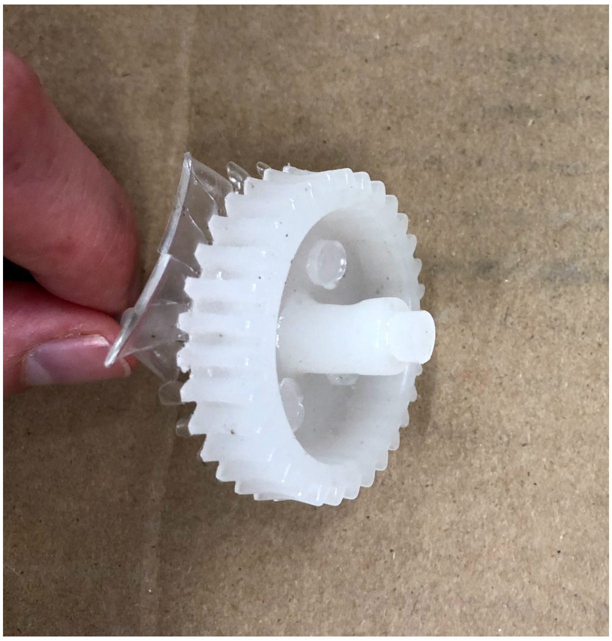


Figure 4.5: An injection molded gear with flash and a sink mark on the gear hub.

Both gears had significant sink marks on the hub, most likely due to the thick walls in this section of the part. I was aware that removing the center section of the hub from the mold in order to create a space for an ejection pin could lead to some uneven cooling and therefore sink marks. I expected the deformation to be parallel to the direction of the hub so it would not impact the appearance of the mold once the center of the hub was drilled out for the gear shaft. However, the sink marks were perpendicular to the direction of the hub, which reduces the diameter of the hub in some areas and creates an inconsistent and unprofessional look (Figure 4.5).

In both gears the upper corners of the gear teeth (the edge that is deepest in the mold) were slightly incomplete and some injection runs resulted in flash around the outside of the mold (Figure 4.5). While these defects did not inhibit the functionality of the part, they indicate that the clamping pressure, packing and holding pressure phases of the injection process require further optimization.

Cracking occurred around the holes in the tabs of both molds because the flat head screws used to hold the mold insert in place were not completely flush with the surface of the mold. Each time the mold closed and clamping pressure was applied, these screws were forced into the mold insert. Movement between the mold insert and housing due to the fractured tabs may have contributed to the flash that occurred around the parting line of the mold.

After the SLA mold insert had been used for approximately 15 injection runs, a crack appeared along the inside and partially down two sides of the mold insert (Figure 4.6). One end of the crack is on the left side of the mold, and traveled approximately $\frac{1}{4}$ inch into the mold on the front surface (where the cavity is located). Since this is the only part of the crack that goes through the surface of the mold where the clamping force was applied, this is most likely where the fracture began. The crack continues down the left side of the mold, through the bottom of the cavity and up the top surface of the insert.

Figure 4.6: Crack in the Standard resin SLA mold.

The crack does not appear to be an extension of the cracks in the tabs of the insert because they are not connected. The crack also does not appear to be a result of a failure of the manufacturing process because it is not parallel to the layers in the resin created by the SLA printing process. If I am correct that it began on the left surface of the insert, the crack is not a result of the pressure of the plastic being injected into the mold cavity.

With support from James Smith, a lecturer for the Princeton University Institute for Science and Technology of Materials, I determined that the crack most likely occurred because the insert was not properly seated in the aluminum housing. If the bottom of the insert was slightly raised above the surface of the aluminum housing, the clamping force applied to the raised portion would have deformed the insert until it reached the aluminum surface. Once the stress in the top of the mold reached the maximum tensile strength of the material, thereby creating the initial fracture on the surface of the insert, the fracture traveled throughout the rest of the block.

This explanation is feasible because the mold insert was not fastened to the housing as securely as it should have been because of the fractures in the attachment tabs and because the insert needed to be removed and reinserted during each in injection run in order to manually eject the completed gear.

The injection runs using FDM and SLA molds showed that it is possible to create a satisfactory product using a mold manufactured using either method. However, the parts made using the SLA mold are superior to parts made using the FDM mold due to the faster cooling time, easier part removal and smoother surface texture. Although the shrinkage for the FDM mold was slightly less than the shrinkage for the SLA mold, this can be remedied by increasing the shrinkage scale factor slightly during mold design. The results of this test provided sufficient evidence to recommend SLA molds over FDM molds. Therefore, future mold iterations made in this project were manufactured only using SLA printing.

4.3 SLA Insert with Modified Ejection Sites

Design

Injection runs of the first iteration of the SLA mold indicated that the ejection pin system required further iteration. I found that the small ejection pins located on the top of the outer edge of the gear were not necessary to remove the gear from the mold when a mold release compound was utilized. The pins were too small to be useful during the ejection process because they needed to be perfectly aligned in order to transfer any force over such a small area. More often than not, the pins became improperly aligned rather than contribution to the ejection of the gear. The presence of the holes negatively contributed to the appearance of the part because they left small indentations in the plastic. Therefore, these holes were eliminated in the next iteration of the mold. Between the first and second iteration of the design I had eliminated the "core" from the hub of the gear in order to create a hole in the center of the back of the mold for ejection. However, this change meant that there was a thick area in the gear which resulted in significant sink marks. In this iteration, I modified the mold to hold a metal pin in the center of the hub during injection. This reduced the wall thickness of the hub without eliminating the center ejection pin.

The SLA and FDM molds used in the previous test fractured on the "wing" portion of the mold insert because the fasteners holding the mold insert in place were slightly higher than the surface of the rest of the mold. This was corrected by increasing the size of the countersink in the new iteration of the mold.

Second Iteration SLA Results

Injection runs using the second iteration of the SLA mold were performed using the same machine parameters as the previous two tests as described in Table 2.1, however the injection distance was reduced from 13.7mm to 13.2 mm to account for the reduced volume of the mold due to the inclusion of the center ejection pin in the mold. The change to the center of the mold eliminated sink marks on the side of the hub but caused an area of shrinkage where the hub meets the rest of the gear (between the A side of the mold and the metal pin). I also observed that the sprue was prone to breaking off of the gear before it was completely cooled.

Shrinkage is generally caused by uneven cooling of material after injection, so I suspect that the presence of the metal pin within the plastic insert caused an uneven heat conduction pattern that resulted in a lot of heat being conducted through the center pin, causing the base of the hub and the sprue to cool too slowly.

I did not experience noticeable sink marks at locations corresponding to the other ejector pins inserted in the mold. The surface area of the outer pins that is exposed to the molten plastic during injection is approximately 0.05 in^2. This is less than the surface area of the center pin, which is approximately (0.265 in^2).

Equation 4.3 shows that the conduction is proportional to the surface area of the conducting solid that is exposed. Therefore, the center pin is conducting heat more than five times faster than the outer pins.

$$\frac{Q}{t} = \frac{k * A * (T_2 - T_1)}{d} \tag{4.3}$$

Where $\frac{Q}{t}$ is the rate of conduction, k is the material conduction coefficient, A is the surface area through which conduction is taking place, $(T_2 - T_1)$ is the difference in temperatures between the surfaces and d is the thickness of the conducting surface.

Over the course of the cooling time of the gear, the increased conduction rate in the mold with the center pin leads to more rapid cooling through the pin center of the hub and the sprue. The plastic directly above the center pin is cooling more quickly than the surrounding plastic. This causes the plastic inside the base of the hub to contract, creating vacancies that ultimately lead to a sink mark in this area. The sprue cools more quickly than the back surface of the mold, causing the joint between them to become brittle and even break off due to the force applied when the mold is opened.

In some of the ejection runs completed with the mold, I manually applied pressure to only the outer ring of ejection pins. I was still able to remove the part with relative ease, and therefore deemed the center ejection pin to be unnecessary. This would allow me to eliminate the center pin and redesign the hub to avoid sink marks.

4.4 High Temperature and Standard Resin SLA Insert

Design

In the fourth iteration of the mold, I returned to the original hub design, in which the center is "cored out" at the top to create a more uniform wall thickness and reduce the chance of sink marks occurring in this area. This design does not include an ejection pin at the center of the part, which was deemed unnecessary in previous injection runs. I also added ribs which extend radially from the outside of the hub to the inside of the gear tooth wall to improve the strength of the cored area of the piece.

The mold insert was initially manufactured using Formlabs' High Temperature resin rather than the standard SLA resin, which was used for the previous mold iterations. The part was washed in 100% IPA to dissolve excess resin and cured for 60 minutes in the Form Cure machine under 400 nanometer light at 60°C.

Both Standard and High Temp resin are approved for use with injection molding, but High Temp resin was not available when the previous molds were manufactured due to a delivery delay [11]. Although none of the SLA molds used for testing have shown signs of deterioration over time as a result of the injection process, the number of injection runs has been relatively small. The estimated lifetime of an SLA mold created with standard resin is approximately 20 - 100 injection runs, but each of the tests I have executed consisted of 5 - 15 tests. Formlabs' White Paper on the use of SLA for injection molding states that over time the clamping force, nozzle heat and adhesion between the PP part and the mold will cause the dimensions of the mold to become less accurate, suggesting that using High Temperature resin for an SLA mold will increase the lifetime of the mold and improve the performance of the mold for a wider range of thermoplastics, especially those requiring a higher nozzle temperature [11].

After an initial testing run was completed using the first High Temperature mold, two more inserts were manufactured for further testing. One was printed in High Temperature resin and cured for 30 minutes under 400 nanometer light at 60°C and the other was printed in Standard black SLA resin.

Third Iteration SLA Results With High Temperature and Standard Resin

The injection settings for the first injection run using this mold iteration were the same as those used in the previous test. The mold manufactured with High Temperature resin failed after the first injection run because it developed cracks throughout the entire mold insert (Figure 4.8). Although at first glance the cracks appeared similar to the result of temperature shock described in the Formlabs White Paper on Injection Molding with 3D printed molds (Figure 4.7). However, upon closer inspection, I observed that the cracks were most concentrated around the junction between the attachment tabs and the body of the mold insert and spread throughout the rest of the piece. If the cracks had been a result of shock, I would have expected them to be concentrated around the inside of the cavity where the hot plastic first touched the mold.

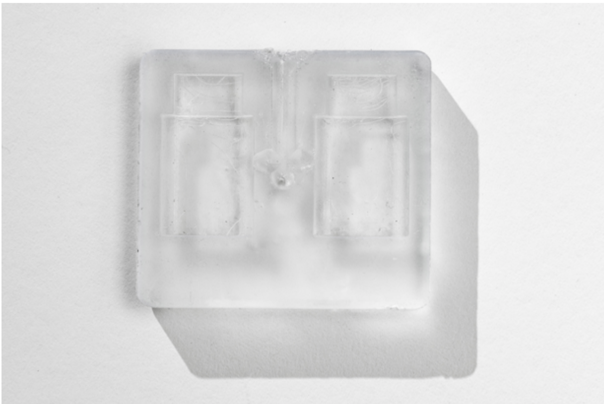


Figure 4.7: Cracks in a mold caused by shock [11]

Figure 4.8: Fractured pieces of the High Temperature resin SLA mold (60 min cure).

Based on the observed crack pattern, I believe that the fractures began because the tabs were too brittle to withstand the clamping force when the insert is manufactured using the High Temperature resin. This result is somewhat surprising because according to the Formlabs material datasheets, the cured High Temperature resin has a slightly higher IZOD impact strength than the Standard resin. However, the standard resin did not crack at the junction between the tabs and the body of the insert as a result of the clamping force [19, 18].

When the mold cracked, liquid plastic flowed into the new spaces in the mold, creating flash-like forms all around the gear, including on the surface of the teeth. This defect aside, the quality of the gear is good. The gear hub, which was not effected by the cracks in the mold, does not have any sink marks and the ribs improved the overall stiffness of the gear.

I was not able to find any formal documentation from Formlabs about what might have caused the part to become brittle, but a post on the Formlabs community forum included anecdotes from other users who observed that brittleness increases with curing time [21]. To validate this anecdotal evidence, I tested the same mold design using High Temperature resin cured for half the time.

The High Temperature mold with less cure time still cracked at the joint between the tab and body of the insert during the first injection run, but there were fewer cracks. The cracks present were also significantly smaller than the cracks in the first test of the High Temperature mold (Figure 4.9). This result suggests that the amount of cure time does impact the brittleness of the material, though further study would be required to establish the exact relationship between the two factors.

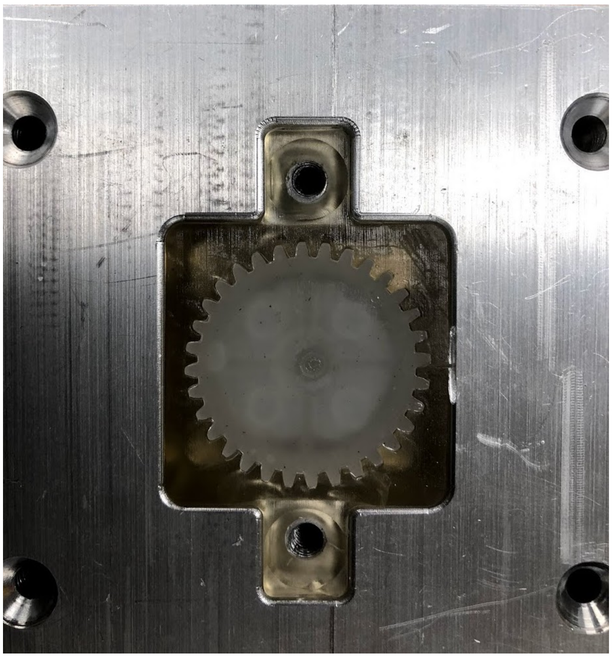


Figure 4.9: Nearly imperceptible cracks in the SLA mold (30 min cure).

I had also hoped to test the fourth iteration of the mold design using Standard resin because the third iteration of mold inserts printed in Standard resin held up well to the clamping force. My first attempt at printing this mold was not successful. The part was completed, but had a very bumpy surface texture and was slightly deformed on one side, preventing it from fitting into the aluminum housing.

I suspected that the poor print quality was caused by partial curing of some of the resin. During the SLA printing process, resin is dispensed slowly into a tank from a carton. In the tank the resin is selectively cured by UV laser to form the part. When the material in the printer was changed from Standard resin to High Temperature resin, I had stored the Standard resin that had already been dispensed into the tank in a covered cup. When I changed the material back to Standard resin, I poured the material from the cup back into the tank so it would not be wasted. Although this is in line with the Formlabs storage recommendations, it is possible that the resin was exposed to too much UV light during this time, leading to increased viscosity and poor print outcomes [26]. I discarded the resin from the tank and refilled it with resin from the carton to attempt the print again. Unfortunately, a printer malfunction caused the resin tank to overflow during this print, destroying the part and breaking the printer. I was therefore unable to test the fourth iteration of the mold using an insert printed in Standard resin.

Finite Element Analysis

Finite Element Analysis (FEA) was performed in SolidWorks to better understand the root cause of the cracking between the tabs and body of the mold insert and to determine the most effective way to improve the mold design. FEA was not performed during the initial mold design process because a complete material definition for the Formlabs resins used for SLA printing is not available due to the proprietary nature of these materials. While the company has released a material datasheet for the resins, the properties included cannot be directly translated to a material definition for FEA [19, 18]. However, an estimation was created based on the Formlabs material datasheets in order to understand the general stress distribution throughout the part [13]. Although thermal analysis would have been very informative, there is not enough information about the thermal properties of the material to formulate an estimated material profile with any level of accuracy.

A simulation of 200 kN clamping force on the fourth iteration of the mold design confirmed my suspicions that the cracks in the mold were originating between the tab and the body of the mold insert due to high stresses in this area. The overall stress in the area was approximately 3,800 psi with some points on the bottom of the tab reaching a maximum of 6,500 psi (Figure 4.10).

I also used FEA to evaluate how different mold design changes would impact the stress level in the part due to the clamping force. A simulation in which the tabs are increased in size from 0.7 inch by 0.7 inch to 1.0 inch by 1.0 inch did not decrease the stresses at the base of the tab (Figure 4.11). However, a simulation in which the depth of the tab was increased from $\frac{5}{32}$ inch to $\frac{3}{8}$ inch reduced the overall stress to approximately 3,500 psi and eliminated the higher stress points altogether (Figure 4.12). Complete details of the FEA simulation setup and results are available in Appendix B.

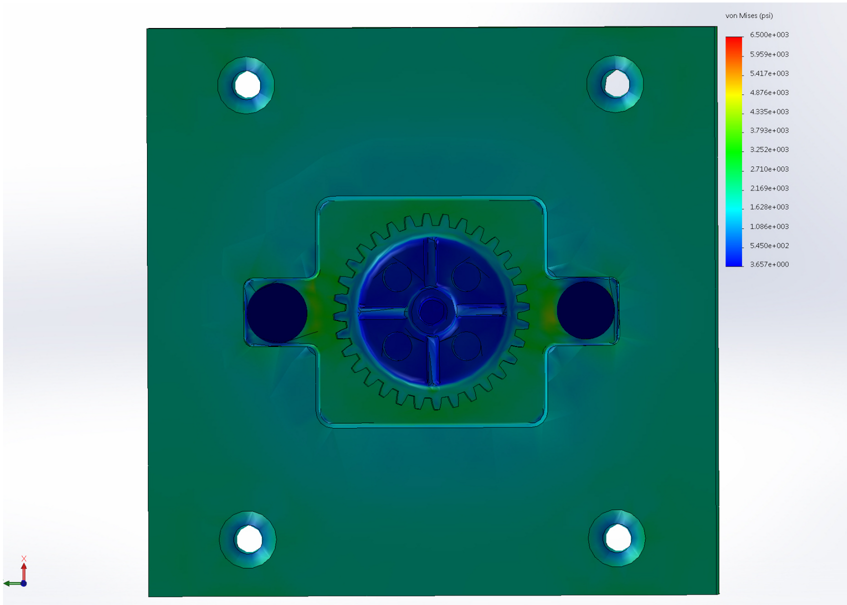


Figure 4.10: FEA results for a 200 kN clamping force on the fourth iteration of the mold.

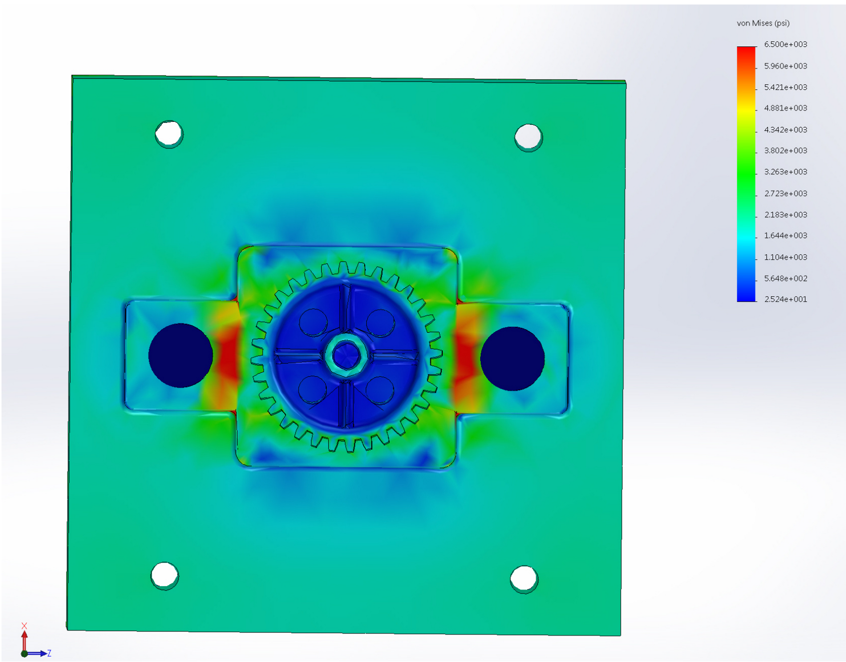


Figure 4.11: FEA results for a 200 kN clamping force on the mold with enlarged tabs.

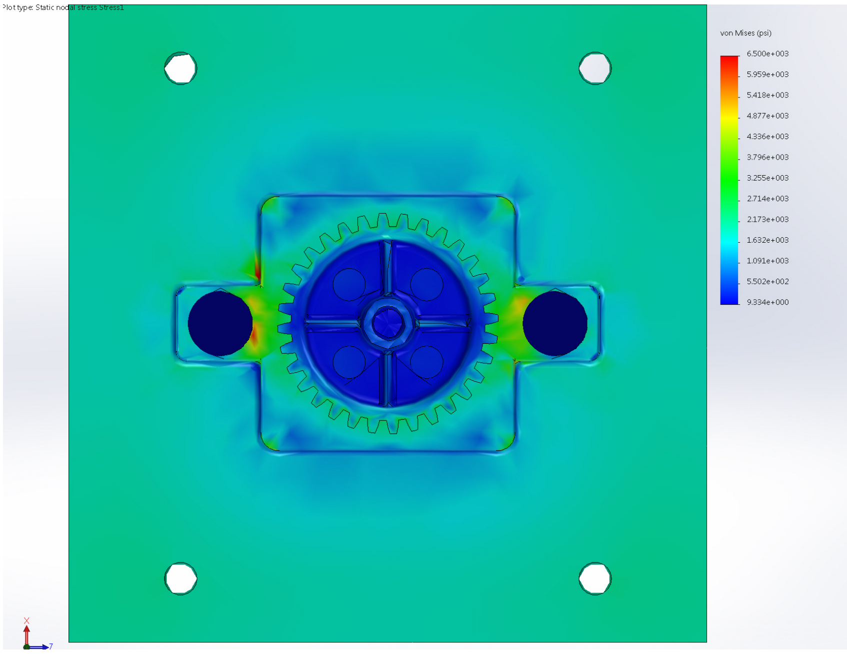


Figure 4.12: FEA results for a 200 kN clamping force on the mold with thicker tabs.

Recommended Mold Modifications

Based on the outcomes of the FEA for the modified mold designs, the depth of the tab should be increased to prevent cracking at the tabs during the injection process. The junction between the tab and the body of the mold insert should also be rounded to distribute stress more evenly during clamping. Although increasing the width of the tab did not reduce the stresses in the mold, the tabs should be widened slightly to prevent cracking on either side of the countersink for the flat head screw which occurred during testing of the second iteration of the mold design. The area of the mold does not typically experience high stress during clamping, but is prone to flexing when the mold is being inserted and removed from the housing. A cutout in the housing next to the cavity for the insert could also be added to further ease the process of removing the insert from the mold. Unfortunately, I was not able to manufacture and test an improved mold because Princeton's SLA printer broke and time constraints prevented me from ordering a part from an outside vendor.

5 Cost Comparison

Injection molding is traditionally used to create high quantity parts. However, I have performed a cost analysis for the manufacturing of flywheel car gears to show the low cost of producing SLA printed molds can make injection molding feasible even for relatively low part quantities. A Cost-Volume chart for the comparison between the SLA mold with aluminum housing, CNC aluminum mold and machining is shown in Figure 5.1 and a full breakdown of the cost analysis is available in Appendix C. The cost analysis I performed is based solely on the cost of materials for each manufacturing method, which accurately reflects the costs associated with this project. Undergraduate students do not explicitly pay for the time spent using the injection molding machine, CNC, lathe, and SLA printer or the man hours that students and shop staff contribute to the project. This cost comparison serves as an evaluation of which manufacturing method makes the most sense for students in the MAE Department, however a comparison that takes into account these other expenses would likely yield a different result.

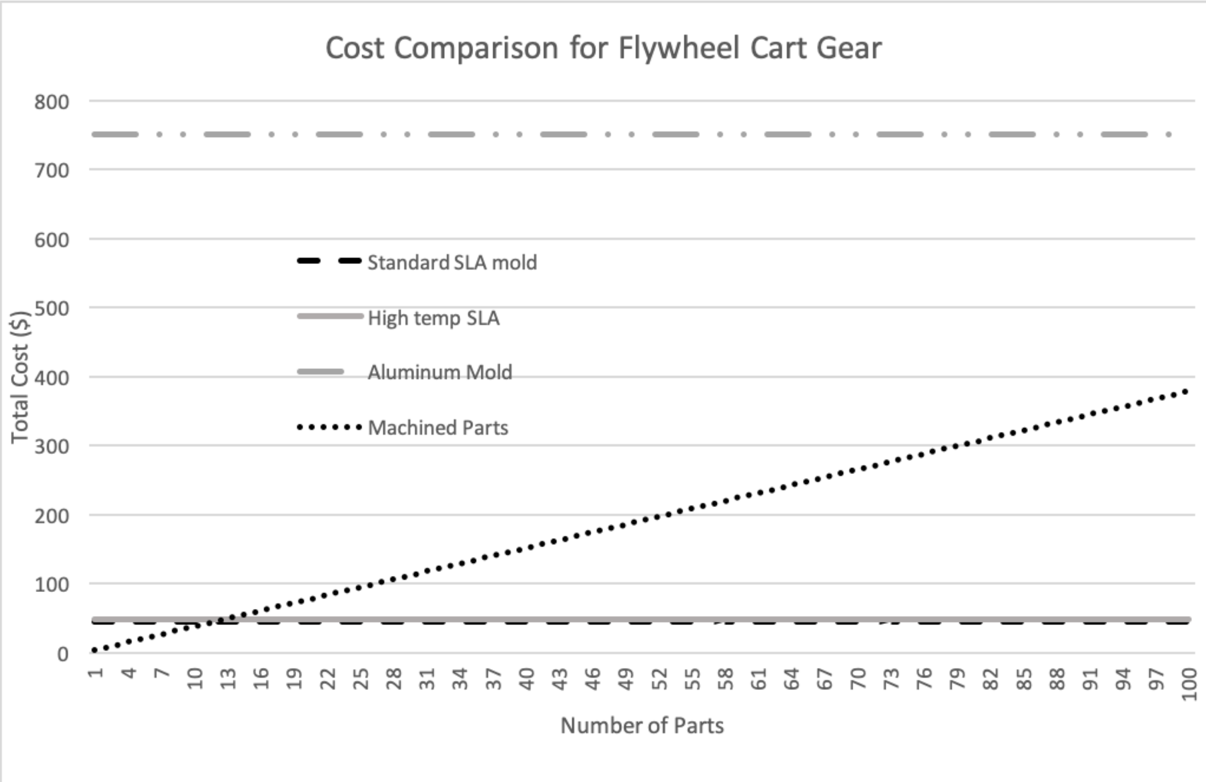


Figure 5.1: Cost comparison of mold manufacturing methods.

The cost to machine the gears is based on the process that is currently used in Princeton's MAE 321 course. The cost of manufacturing the aluminum gears currently used in the course is the cost of a 12 inch long, 2 inch diameter cylinder, which can be used to make approximately 6 gears [3]. The cost of the SLA mold includes the aluminum housing, which is machined from a 6 inch by 6 inch by 1.125 inch block, as well as the material required to print the mold in either Standard or High Temperature Formlabs resin [10, 2]. The cost of the aluminum mold is based on a quote from Protolabs, the same manufacturer that was used to create the MAE beverage coaster mold [24]. While Princeton's CNC machine could be used to manufacture the mold, this process would be very time consuming and may not realistically fit into the timeline of a student project. Having the mold made by an external manufacturer was considered a more accurate equivalent to 3D printing the mold using the SLA machine, a process which only takes a few hours and requires very little operator intervention.

The lines on Figure 5.1 indicating the cost of manufacturing a given number of parts using injection molding are nearly horizontal due to the extremely low cost of plastic for injection molding a small part. A 2200 g bag of PP can be purchased for $17.50 and approximately 12 g of PP are used in each part [22]. I have estimated that the cost of PP to fill each of the gear molds was $0.005 per part.

As indicated in Figure 5.1, the initial cost of the SLA mold insert in an aluminum housing is the same as the cost associated with machining 12 aluminum gears. The mold can be used to manufacture up to 100 parts, ultimately resulting in savings of approximately $330 for the MAE Department. Although minimizing costs is not the priority for MAE classwork, this cost analysis provides compelling evidence for the use of injection molding rather than machining for students who need to produce a small number of custom, identical parts for a project with a limited budget. For example, an SLA mold insert could be used to produce propellers for a drone, custom fasteners, etc.

In an industrial setting, a small company interested in manufacturing a small number of parts using injection molding would most likely outsource this work to a manufacturing company rather than investing in the machines necessary for production. In general, the cost of material by volume for SLA printing is less than the cost of aluminum by volume. Since SLA generates less waste, the process also requires less material than machining. SLA printing requires fewer man hours than CNC machining because the machine's path is generated automatically and it does not require supervision [1]. For small companies seeking agile, low volume manufacturing methods, injection molding using SLA is a cost effective option.

6 Conclusion

Throughout the course of this project I learned how to operate the BOY 35E and shared this information with members of the MAE faculty, who will be able to pass it on to students in the department in the future. I saw that injection molding can create high quality, identical parts quickly and inexpensively. That said, I also saw that optimizing the process requires patience and significant troubleshooting, especially for an inexperienced user. The manual and troubleshooting guide I have created will be a valuable tool to help students through this process.

The 3D printed gear molds I created validate the concept that SLA printing is an effective method of mold fabrication. Using 3D printed molds afforded me opportunities for design iteration that would have been excessively costly and time consuming if I had been using a metal mold. Although the final printed mold insert cracked during injection, I have provided FEA demonstrating the required design changes to reduce the stress in the mold and prevent cracking. Once the SLA printer has been repaired, the tools and information necessary to create injection molded parts using 3D printed molds will be available to students in the MAE department.

6.1 Future Work

The BOY 35E injection molding machine and SLA mold printing process create many opportunities for research that I did not have time to investigate in this project. The most pressing task that must be accomplished in order to completely integrate the injection molding machine into MAE 321 are the mold design modifications described in Section 4.4. Further testing in the long term use of 3D printed molds for injection molding is also necessary to better understand the lifetime of these molds and limitations of the method.

In order for the BOY 35E to be operated without operator intervention, it will be necessary to develop an integrated ejection pin system. The original system which was designed to accompany the MAE beverage coaster mold was only moderately effective and could not be easily adapted for the flywheel gear mold. The ideal ejection pin system could be reconfigured to accommodate many different molds with minimal manufacturing work.

The materials and mold designs I used for this project represent a very small number of the options available for injection molding. Students who are interested in better understanding the capabilities of the method could investigate the use of molds with multiple cavities, runners, undercuts and overmolds as well as more complex materials. Further work is required to understand the process of changing materials for the BOY 35E as well as the potential complications of using more sophisticated injection materials (for example, glass filled plastics).

The structural and thermal analysis in this report was limited by the public availability of information about the properties of Formlabs' SLA resins. Performing materials testing and analysis of the Standard and High Temperature resins to develop a complete material profile for simulation would allow students to use FEA as an accurate mold design and analysis tool.

6.2 Lessons Learned

Learning how to operate the injection molding machine and creating 3D printed molds for a flywheel car gear presented significant challenges that lead to important lessons learned about mechanical design and manufacturing projects.

It is understandable that the injection molding machine was unused for a significant amount of time after being installed because the machine manual did not provide sufficient information about the operation process and machine troubleshooting. While I was able to learn a lot by experimenting with the machine on my own and studying injection molding process guides, ultimately it was most helpful to ask for help from someone with experience in the field. Although Bob Bennett had never used the BOY 35E before, his many years of work in the plastics and injection molding industries allowed him to provide valuable insights into the operation of the machine.

I also learned that FEA tools, if used effectively, can save significant time, money and effort. For example, performing FEA for all of the mold iterations, even if the parameters were estimated, could have identified many of the structural problems I experienced with the molds before manufacturing and led to a more informed, and therefore effective, design process.

The greatest lesson I learned while working independently on this project was resilience in problem solving. As an undergraduate, much of my time has been spent working on problems that already have well defined answers. Therefore, it was a unique experience to complete a project using tools and manufacturing techniques, like injection molding and SLA printing, that were relatively new to myself and to my mentors on this project. Although there were many times in this project when I asked for advice from more experienced engineers and manufacturers, ultimately there was no clear "right answer" to the problem and the only way to solve it was by experimenting with many different methods. While this could be frustrating, problem solving resilience is be a skill that will serve me well in future endeavors.

Bibliography

[1] *3D Printing vs CNC Machining: Which is best for prototyping?* Apr. 2, 2019. URL: https://www.3dnatives.com/en/3d-printing-vs-cnc-160320184/.

[2] *6061 Aluminum 1-1/8" Thick, 6" x 6"*. McMaster-Carr. Mar. 5, 2019. URL: https://www.mcmaster.com/9246k593.

[3] *6061 Aluminum 2" Diameter*. McMaster-Carr. Apr. 21, 2019. URL: https://www.mcmaster.com/8974k71.

[4] *ABS-M30: Production-grade thermoplastic for FDM 3D printers*. Stratasys. 7665 Commerce Way, Eden Prairie, MN 55344, 2017.

[5] Douglas M. Bryce. *Plastic Injection Molding, Volume III - Mold Design and Construction Fundamentals*. Society of Manufacturing Engineers (SME), 1998. ISBN: 978-0-87263-495-4. URL: https://app.knovel.com/hotlink/toc/id:kpPIMVIIID/plastic-injection-molding/plastic-injection-molding.

[6] Douglas M. Bryce. *Plastic Injection Molding, Volume III - Mold Design and Construction Fundamentals*. Society of Manufacturing Engineers (SME), 1998. ISBN: 978-0-87263-495-4. URL: https://app.knovel.com/hotlink/toc/id:kpPIMVIIID/plastic-injection-molding/plastic-injection-molding.

[7] *CES EduPack*. Granta. Cambridge.

[8] *CityTaps*. Available at https://www.citytaps.org/(2019/02/12).

[9] *Form 2 Complete Package*. Formlabs. Apr. 21, 2019. URL: https://formlabs.com/store/form-2/buy-printer/.

[10] *Formlabs Resin Library*. Available at https://formlabs.com/materials/ (2018/09/20).

[11] *Formlabs White Paper:Injection Molding from 3D Printed Molds*. Tech. rep. Formlabs and Galomb, Inc., Aug. 2016.

[12] T. Gutowski. *Injection Molding*. Tech. rep. MIT.

[13] *How to Choose the Right 3D Printing Material*. Apr. 12, 2019. URL: https://formlabs.com/blog/how-to-choose-the-right-3D-printing-material/.

[14] *Injection Molded Design Cube*. Tech. rep. Protomold, 2011.

[15] *Injection Molding Basics: An Introduction to Designing Molded Parts*. Protolabs. Nov. 5, 2018. URL: https://www.protolabs.com/resources/design-tips/injection-molding-basics/.

[16] *Instructions for use: Automatic injection moulding machine, BOY 35 E*. Translation of original instructions for use. BOY Spritzgeissautomaten. Industriegebiet Neustadt/Wied Neschener Str. 6 53577 Neustadt-Fernthal Germany, 2015.

[17] Serope Kalpakjian and Steven Schmid. *Manufacturing Processes for Engineering Materials, 6th ed.* Pearson, 2017. ISBN: 978-0134290553.

[18] *Material data sheet- High Temp: High Temp resin for Heat Resistance*. Formlabs. 2018.

[19] *Material data sheet- Standard: Materials for High-Resolution Rapid Prototyping*. Formlabs. 2017.

[20] *Mold-Release Lubricant with Silicone for Plastic, NO-Residue, 12 oz. Can*. McMaster-Carr. Mar. 15, 2019. URL: https://www.mcmaster.com/1409k33.

[21] *Over curing a part*. Formlabs: Community Forum. Apr. 24, 2019. URL: https://forum.formlabs.com/t/over-curing-a-part/4081.

[22] *Plastic Pellets*. Rio Grande. Apr. 21, 2019. URL: https://www.riogrande.com/product/PlasticPellets/.

[23] *Polypropylene (PP)*. British Plastics Foundation. Dec. 21, 2018. URL: https://www.bpf.co.uk/plastipedia/polymers/pp.aspx.

[24] *Protolabs CNC Machining Service*. Available at https://www.protolabs.com/services/cnc-machining/ (2018/09/20).

[25] *Rates of Heat Transfer*. the Physics Classroom. Mar. 28, 2019. URL: https://www.physicsclassroom.com/class/thermalP/Lesson-1/Rates-of-Heat-Transfer.

[26] *Resin Care*. Apr. 29, 2019. URL: https://support.formlabs.com/s/article/Resin-Care?language=en_US#storage.

[27] Mark Scerbo. “Injection Molding: Mold Manufacturing and Set-Up Guide for the BOY 35E Machine”. Bachelor’s Thesis. 2018.

[28] *Selecting the Right Plastic for Your Next Molded Part*. Protolabs. Apr. 20, 2019. URL: https://www.protolabs.com/resources/design-tips/selecting-the-right-plastic-for-your-parts/.

[29] *Shrinkage: Contraction of Plastics Molded Part*. Jan. 13, 2019. URL: https://omnexus.specialchem.com/polymer-properties/properties/shrinkage.

[30] *Sprues, Runners and Gates of Plastic Mold*. Apr. 29, 2019. URL: https://www.plasticmoulds.net/spruesrunners-and-gates-of-plastic-mold.html.

[31] *The Outline of Injection Molding*. Apr. 12, 2019. URL: https://www.polyplastics.com/en/support/mold/outline/index.html.

[32] *The Single Material Injection Molding Process*. University of Maryland. Oct. 28, 2018. URL: http://terpconnect.umd.edu/~skgupta/M3T/SM.html.

Appendix A: BOY 35E Injection Molding Manual

This manual is intended to provide an overview of machine operations as well as the process steps required to set up and perform basic injection molding using the BOY 35E Procan Alpha. This document does not describe all of the functionalities of the machine nor is it intended to replace the manual, *Instructions for Use: Automatic Injection Moulding Machine, 35E*, which was provided by BOY. Although BOY claims that the screen pages of the control screen are "largely self explanatory," many new users of the machine, myself included, have felt mystified by the obscure labeling of the keys and lack of clear process steps indicated on the user interface [16]. I hope that this manual will help a user, particularly one who is unfamiliar with the particulars of injection molding, navigate the machine proficiently and efficiently.

In case of further questions or problems not covered in this manual, consult *Instructions for Use: Automatic Injection Moulding Machine, 35E*, Glenn Northey or Al Gaillard. If necessary, Ken Bush, Princeton's BOY representative, can be reached by email at kbush@boymachines.com or by telephone at (610) 363-9121.

Please use caution and proper protection at all times when operating the BOY 35E machine. Do not operate the machine without qualified shop personnel present. Proper shop attire (closed toed shoes, pants, no loose clothing) should be worn at all times while operating the machine. Safety glasses should be worn when the main gate and heating cylinder gate are open. Gloves should be worn when reaching into the heating cylinder gate to clean melted material off of the nozzle because hot plastic could leave the nozzle at any time. Use extreme caution when clearing clogs in the heating cylinder or between the hopper and the cylinder.

NOTE: Only brass and copper tools should come in contact with the heating cylinder and nozzle to avoid scratching the machine.

1 Quick Start Operation

Before you begin, review the safety precautions above to ensure that you are not putting yourself or others in danger while operating the machine.

1.1 Turn on machine/cooling system/water flow

To begin using the BOY 35E, turn on the water cooling system by turning the two yellow valves on the wall to the left of the machine to the "open position," (Figure 1.1) then put the large switch mounted on the back wall of the shop in the “on” position to allow hot and cold water to flow through the mixer (also mounted on the back wall). You should be able to hear water flowing and see the active flow indicator on the right-most tube of the water coolant distributor located on the back of the injection molding machine.

Figure 1.1: Photo of injection molding machine with power and water cooling valves labeled.

Turn on the power to the machine by rotating the black lock-out rotary switch on the lower front of the right side of the machine. Rotate this switch from horizontal to vertical. In order for the user interface of the machine to operate, the software “key” loaded onto a USB flash drive must be inserted into the USB port on the top rear of the touch screen display. The flash drive is typically left inserted in the machine.

1.2 Turn on motor and heaters

Once the user interface has loaded on the touch screen, turn on the motor and heater using the buttons located in the bottom left and right corners of the screen, respectively, by touching the “on” selection (vertical line). The animations labeling these buttons will change to show rotating gears for the motor and moving red heat lines for the heater. The machine will not be able to perform mechanical operations until the motor oil has heated up sufficiently. Injection will not be possible until the phases of the injection cylinder have warmed up to their required temperature, as specified by the temperatures under the schematic drawing of the injection cylinder on the home page.

1.3 Injection process

Before beginning the injection process, confirm that the machine parameters are appropriate for the mold and material being used. Details on determining the parameters and programming them on the BOY 35E machine are available in Section 3.1.6. If a 3D printed mold is being used or the product is expected to be difficult to remove from the mold it may be beneficial to use a mold release compound. When selecting this compound, confirm that it is compatible with the material being injected and the material of the mold.

Ensure that both gates are securely closed. Set the machine to automatic or semi-automatic mode, as discussed in Section 3.1.4, and press the start button (marked by a diamond with a line through it, flashing green). If an error message prevents injection from beginning, or the results of an injection run are not as desired, adjust the machine parameters according to the troubleshooting steps described in Section 5.

1.4 Shut down

In order to shut down the machine, first stop the current injection process by putting the machine in "stop" mode (press the red button marked by a circle inscribed with a triangle). Remove power from the machine by returning the rotary switch to its horizontal position. Turn off the water cooling system by turning the large switch on the back wall to "off" and closing the yellow water valves on the back wall to "closed." It is not necessary to purge the material from the injection chamber before shutting down the machine.

2 Machine Components

Figure 2.1 shows the entire BOY 35E injection molding machine. The yellow main gate on the left side of the machine can be opened by pushing the button on the black handle and sliding to the left. This provides access to the mold, cradle, platen and ejection pin assemblies. The B side of the mold moves back and forth to open and close the mold while the A side of the mold is stationary (Figure 2.3). Refer to Mark Scerbo's independent work report, *Injection Molding: Mold Manufacturing and Set-Up Guide for the BOY 35E Machine*, Section 3 for information on the cradle and ejection pin assemblies installed on Princeton's BOY 35E machine.

Figure 2.1: Labeled photo of Princeton's BOY 35E Procan Alpha.

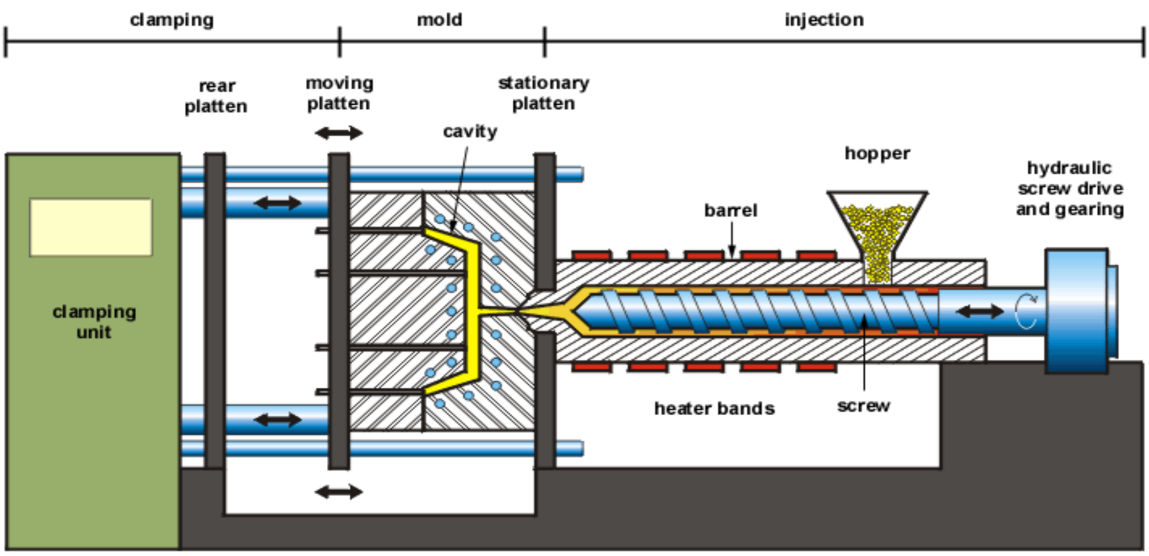


Figure 2.2: Schematic of injection molding maching.

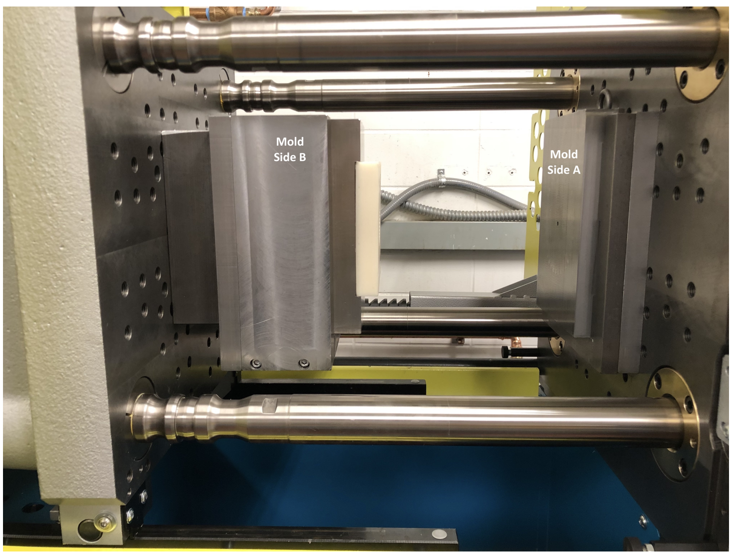


Figure 2.3: Labeled photo of platen, mold and cradle.

Figure 2.4: Labeled photo of BOY 35E user interface.

The plasticizing unit, which contains the hopper assembly, heating barrel, and bands, and nozzle, is housed on the right side of the machine (Figure 2.1). The heating cylinder gate on the right side of the machine can be opened by sliding the small black handle to the right. This provides access to the heating cylinder and nozzle, typically used to clean extruded plastic from the nozzle when the plasticizing unit is pulled away from the A side of the mold.

The yellow water cooling system valves are located on wall of the shop behind the machine to the left of the machine. The master power switch is located on the same wall to the right of the machine (Figure 1.1). The machine power switch is located on the front right of the machine.

Consult Section 1.2 of this report for a more detailed introduction to the injection molding process and machine components.

3 User Interface

The main control screen for the BOY 35E will appear on the touch screen display when the machine has been turned on. The specific function of each of the buttons is described in the BOY 35E manual, but these descriptions lack context for a user who is unfamiliar with injection molding so I will use this section to summarize the role and function of each button [16].

Select a particular field by touching it with a finger. To change the value of numeric fields, touch the number to bring up a 10 digit keyboard which is used to enter the new value. If applicable, the "set" key may be used to assign a position to the current position of the component. Apply a numerical setting by pressing the "enter" key, which features a downward line ending in a left-pointing arrow. Dismiss any pop up window or error message from the screen by tapping on it once. Note that none of the machine's mechanical operations will take place if the main gate or heating cylinder gate is open.

3.1 Edge Buttons

Buttons, located along the bottom, left and right sides of every screen, identify additional functionality on the user interface screen.

3.1.1 Motor on/off

Turn on the motor, which controls the mechanical valves, by pressing the "on" (vertical line) button next to the circular image of the gears. Turning on the motor will cause the gear animation located next to the on/off buttons to rotate. When the machine is first turned on, the motor will not be able to work until the motor oil has had time to preheat. The motor turns off after an unspecified, seemingly random, amount of time, preventing any mechanical operations from taking place. If this happens, simply turn the motor on again. Turn the motor off by pressing the corresponding "off" button (circle).

3.1.2 Heater on/off

Turn on the heating bands surrounding the injection cylinder by pressing the "on" (vertical line button) next to the circular image of a blue liquid with wavy lines coming out of it. Turning on the heater will cause the picture to change to show a red liquid with moving lines coming from the surface. When the machine is first turned on, the heating bands will take some time to heat the plastic in the cylinder to the desired temperature. The temperature status of the plastic is indicated on the schematic of the injection cylinder located on the main page of the user interface and on the cylinder temperature page (Figure 2.4). Turn the heater off by pressing the corresponding “off” button (circle).

3.1.3 Movement buttons

If the button has both a circle and a “valve” (two touching triangles) symbol on its inner edge, and horizontal arrow, it can be used to move a component of the machine. The circle turns yellow to indicate that a limit switch has been triggered indicating that the component has reached its movement limit in the operating direction. Depending on the component, this is not always reliable. It is clear when a component has reached its movement limit in one direction because the noise made by the movement valve will change and the component will stop moving. The valve indicator on the button turns green to indicate that the valve which controls this component movement has been engaged. If the component has not reached its limit, it will be moving in the direction indicated by the arrow. The arrows indicate the direction which the component moves assuming that the display screen is parallel to the injection molding machine with the touch screen facing outwards.

- Core A Advance and Retract (1, 2)

  If a mold requiring a core pull is used, these buttons can be used to control the position of the core for setup. A core pull is a moveable component of the mold which can be used to create a cavity within the injection molded part.

- Core B Advance and Retract (3, 4)

  If a mold requiring a second core pull is used, these buttons can be used to control the position of the secondary core for setup.

- Ejection Pin Advance and Retract (5, 6)

  If a mold utilizing ejection pins is being used, these buttons can be used to advance and retract the retaining plate which controls the movement of the ejection pins for setup.

- Mold Advance and Return (7, 8)

  These buttons advance and return the position of the B side of the mold for setup. These should be used to set the “open” and “closed” position of the mold to account for differences in mold thickness.

- Plasticizing Unit Advance and Return (9, 10)

  These buttons advance and return the position of the plasticizing unit which contains the hopper assembly, heating barrel and bands, and nozzle. The plasticizing unit must be at its forward limit (nozzle touching the A side of the mold) for injection, but may be moved away from the mold to allow the user to push plastic out of the screw without injection into the mold to check the quality of the material or to purge the screw to use a different material.

- Screw Advance and Retract (11, 13)

  This can be used to advance and retract the position of the screw within the heated chamber. The advance button may be used to push plasticized material out of the chamber. The retract function may be used to move the screw to a different position, but will not plasticize material.

- Plasticize (12)

  The plasticize button is used to turn the screw inside the heated chamber in order to plasticize the heated material. This creates friction between the screw, material and the outside of the chamber to help the material become more fluid in preparation for injection. Plasticize can be used in manual and setup mode.

3.1.4 Mode buttons

If the button only has a circle on its inner edge, it is used to control the operating mode of the machine. The circle turns yellow to indicates that the mode controlled by the button is active.

- Stop Cycle (A)

  In this mode, the machine is in standby. The stop button may be selected at any point in the injection process to stop the machine.

- Manual Operation (B)

  This button allows the user to operate the machine in manual mode or setup mode.

  - Manual mode is entered by pressing on the button once. This allows the user to do some injection process steps such as moving the plasticizing unit (buttons 9/10) and plasticizing (button 12) up to the injection volume specified in the current machine process.
  - Setup mode is entered by pressing and holding the button for 3 seconds (a loading bar at the top of the screen shows amount of time left until setup mode is entered) This mode allows the user to operate all movement keys (described in Section 3.1.3) for injection process setup (Section 3.1.6).

- Semi-automatic Mode (C)

  In semi-automatic mode the machine will complete one injection cycle and then enter stop mode. Once semi-automatic mode is selected, the start key (button E) will begin flashing green if the machine is properly set up. The start key must be pressed to begin the injection cycle.

- Fully Automatic Mode (D)

  In fully automatic mode, the machine will continue injection cycles until the user returns it to stop mode. Once this mode is selected, the start key (button E) will begin flashing green if the machine is properly set up. The start key must be pressed to begin injection. If possible, operating the machine in automatic mode will lead to the most consistent injection results [5].

- Start Button (E)

  In semi and fully automatic mode, this key must be pressed to begin injection.

3.1.5 Main page of user interface

The main page of the user interface includes a schematic of the outside of the injection cylinder with corresponding heating cylinder temperatures and a schematic of the inside of the injection cylinder showing the position of the screw. The vertical gauge at the left of the screen shows the clamping pressure and the circular gauges on the left and right of the screen indicate the pressure of the screw or motor (based on which circle below the gauge is selected) and the rate of rotation of the screw, respectively. The flag at the top of the screen displays the injection process cycle time, an important metric for industrial applications of injection molding that is less interesting for experimental uses of the machine.

The BOY logo is located at the center of the main page surrounded by 11 icons representing the different stages of the injection molding process. During the injection process each button will turn green to indicate which stage of the process is taking place. The icons can be used as shortcuts to access the settings for each injection stage. Select the BOY logo to access the Table of Contents, which lists every page in the system as well as a description of the information contained in the page. Access a page by touching the corresponding pictogram. Scroll through pages using the arrow buttons at the bottom of the page.

Access the main page of the user interface from any other page by touching the button with the large M on it. (F in Figure 2.4). View current errors as well as a log of resolved errors by selecting the button showing the red triangle and exclamation point (G in Figure 2.4).

3.1.6 Injection settings control

The six keys in the second section from the left in the bottom area of the user interface screen are used to set the conditions for the injection process. Information on how to determine the key injection settings for a given mold and material is discussed in Section 4.

1. Clamping unit close (H)

   Control how the B side of the mold closes and clamps.
   BOY 35E user interface pages 1.10 - 1.30.

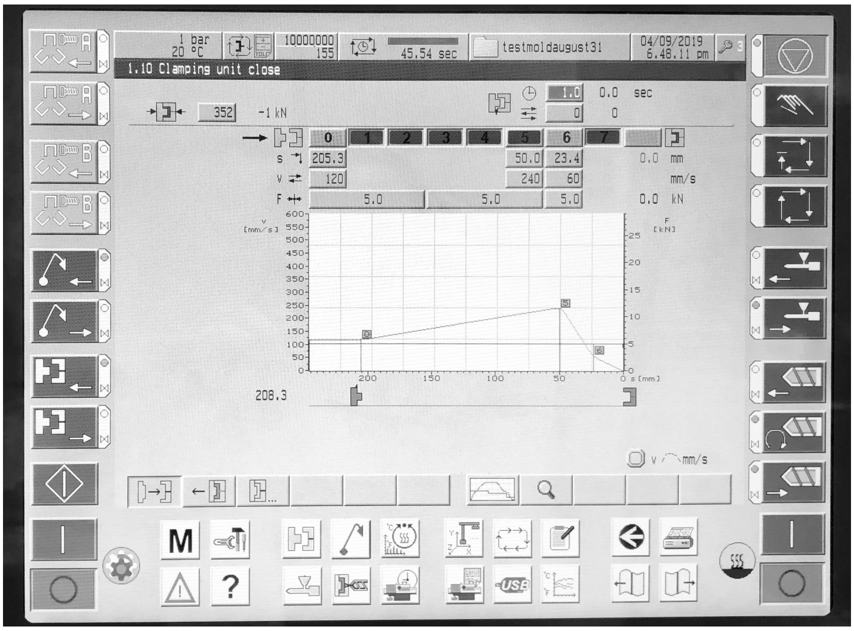


Figure 3.1: Page 1.10 of user interface: clamping unit close.

On page 1.10 use the number box in the top left corner to set the clamping force. In the center of the page, use the profile points numbered 0 through 7 to customize the position, speed, and force of the closure, as indicated on the plot below the profile points. Each point can be enabled or disabled by touching the box at the top of the column displaying the number of the profile point.

The most important profile point is the highest numbered active point (representing where the mold is closed). If this position is inaccurately programmed, the injection will fail because the machine cannot achieve the required clamping force between the two sides of the mold. It is recommended that the mold advance key (Section 3.1.3 of this manual) is used to carefully move the B side of the mold until it is firmly against the A side of the mold. Then use the “set” function on the digital keypad to program this position as the “mold closed” point.

On the BOY 35E machine, page 1.20 can be used to customize the position, speed and force of the mold opening step using the same operations as page 1.10. Page

1.30 contains settings for additional functionalities of the clamping unit. For further information regarding the programming of the clamping unit, refer to pages 1.5/24 - 44 in the BOY 35E Instructions for Use [16]. Refer to Section 4.2 of this report for information on how to determine the mold opening distance and speeds.

2. Ejection (J)

   Control the movement of ejection pins, air blast valves and core pulls (if applicable). BOY 35E interface pages 2.10 - 2.40.

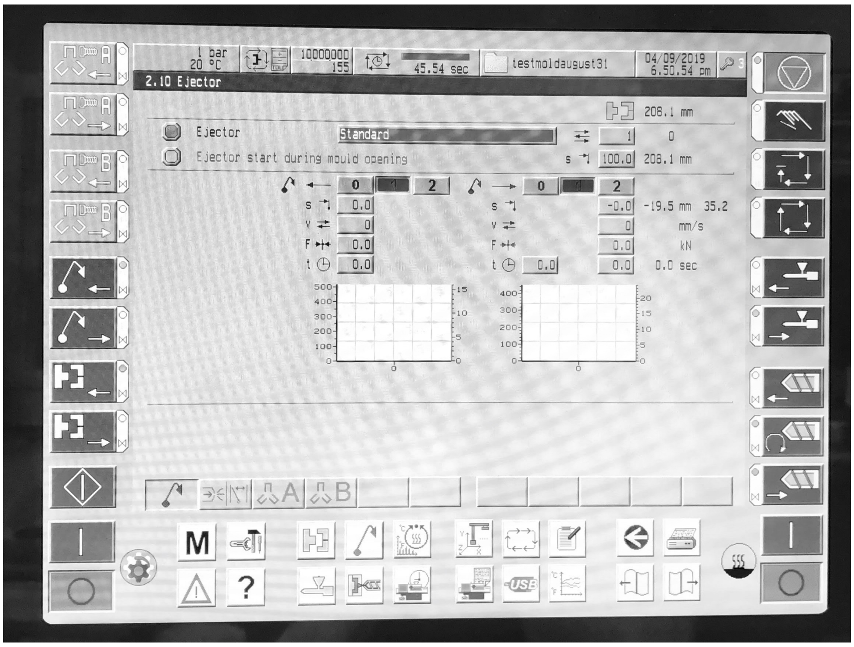


Figure 3.2: Page 2.10 of user interface: Ejector.

   Page 2.10 can be used to define the method of ejection (standard, vibration, time dependent, etc) and the specific position, speed and force of each profile point if ejection pins are used. Pages 2.20, 2.30 and 2.40 contain the settings for ejection air blast valves, core pull A and core pull B, respectively, if any of these functionalities are enabled. At the time when this manual was written, Princeton's BOY 35E machine is not currently configured to use air blast valves or core pulls. For further information regarding the buttons involved in programming the ejection pins, air blast valves, and core pulls, refer to pages 1.5/45 - 59 in the BOY 35E Instructions for Use [16]. Refer to Section 4.2 of this report for information on how to determine the advance distance and speed required for ejection pins.

3. Barrel Heating Zones

   Control the barrel heating zone temperatures and tolerances as well as the settings of any additional temperature devices or hot runners connected to the machine. BOY 35E interface pages 5.10 - 5.50.

   Page 5.10 can be used to define the nominal temperature of each heating zone of the barrel containing plastic to be injected as well as the upper and lower temperature

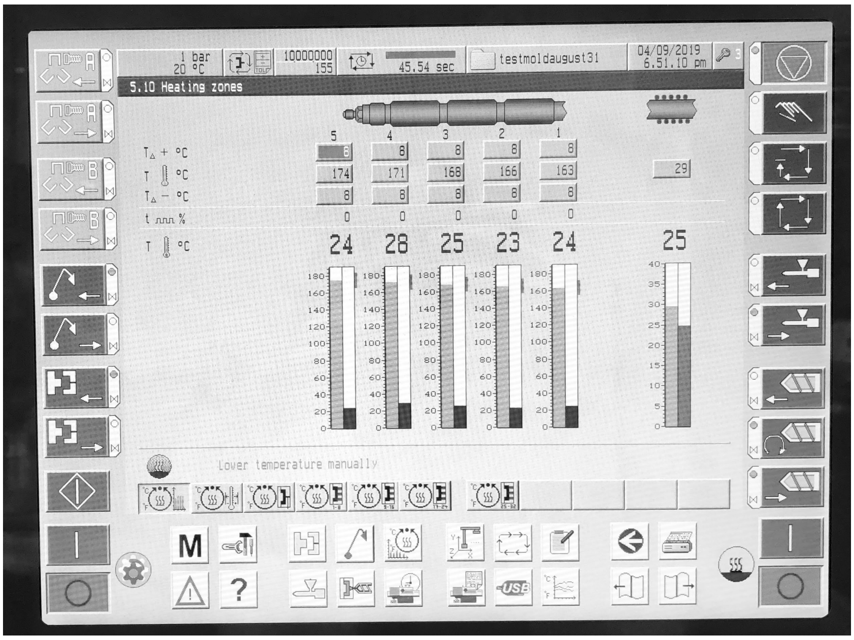


Figure 3.3: Page 5.10 of user interface: Heating Zones.

limits. The location of each profile point corresponds to the location of the point with respect to the schematic drawing of the barrel. The temperature point corresponding to the drawing of the tube with blue dots on either side indicates the temperature limits for the portion of the barrel which is cooled by the water cooling system.

Page 5.20 contains settings regarding the action to be taken if a temperature is not within the tolerated limits. Pages 5.30 - 5.50 contain settings to control the functionality of additional temperature units or hot runners if applicable. At the time when this manual was written, Princeton's BOY 35E machine was not currently configured to use additional temperature units or hot runners.

For more information about configuring the heating temperatures refer to pages 1.5/89 - 101 in the BOY 35E Instructions for Use [16]. Note that the screenshots of the temperature settings included in the Instructions for Use differ in some ways from the actual user interface of the BOY 35E in that only 4 temperature settings are shown on page 5.10 and the water cooling temperature is only included on page 5.20. However, the information about the operation of each button is accurate even if the location within the pages differs slightly. Refer to Section 4.3 of this Appendix for information on how to determine the temperature settings for a given plastic.

4. Plasticizing unit (L)

   Control the placement of the plasticizing unit. BOY 35E user interface pages 3.10.

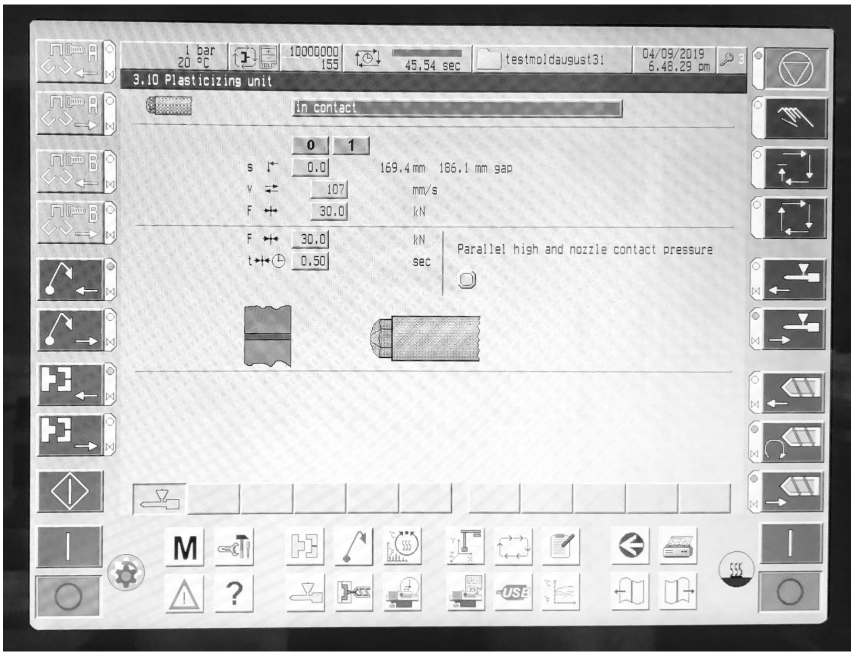


Figure 3.4: Page 3.10 of user interface: Plasticizing unit.

On page 3.10 of the user interface, select the text at the top of the page to change the positions of the plasticizing unit throughout the injection process (in contact, retraction after plasticizing, or retraction before plasticizing). Use the numbered points to control the position, velocity, and force of the advance and, if applicable, return of the plasticizing unit. It is most important to ensure that the nozzle is correctly positioned against the A side of the mold for injection. If this position is programmed inaccurately, the injection process will fail because the machine is unable to achieve the required pressure between the nozzle and sprue. It is recommended that the plasticizing unit advance key (Section 3.1.3 of this Appendix) is used to carefully move the plasticizing unit until the nozzle is firmly in contact with the A side of the mold and the "set" function on the digital keypad is used to program this position as the forward-most position of the plasticizing unit. For more information regarding the movement of the plasticizing unit, refer to pages 1.5/61-64 in the BOY 35E Instructions for Use [16].

5. Injection process filling phase (M)

   Control the amount of plastic and how it is injected into the mold.
   BOY 35E user interface pages 4.10 - 4.40.

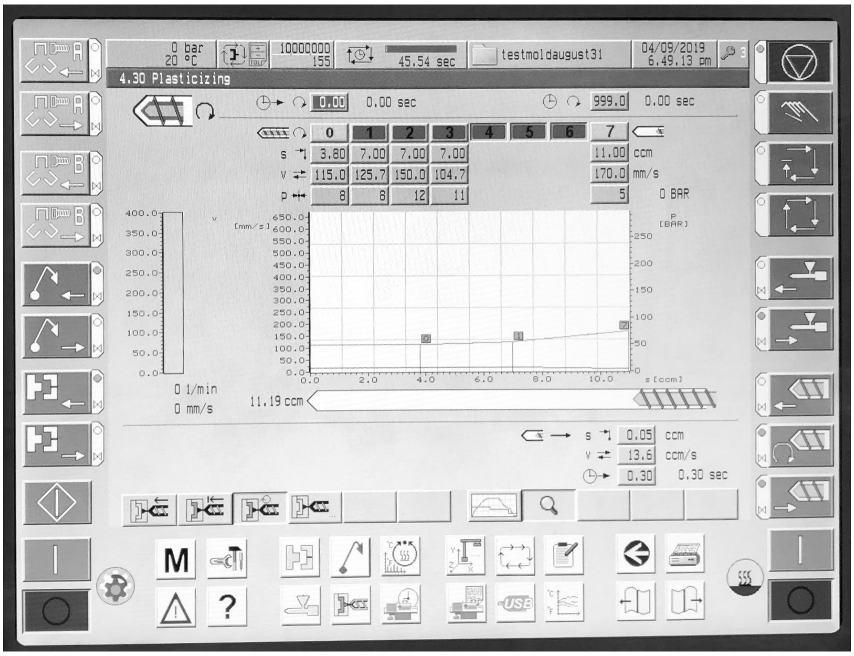


Figure 3.5: Page 4.10 of user interface: Plasticizing.

On page 4.10 use the numbered profile points to control the position, speed and pressure with which the plastic is injected into the mold. Each point can be enabled or disabled by touching the box at the top of the column displaying the number of the profile point.

On page 4.20, the profile points to control any changes in holding pressure over time, starting from the final injection pressure at point 0. The selection circles and corresponding numbered boxes to the left of the plot indicate the time, stroke position, or pressure at which the process switches over to the injection holding pressure. The fifth selection indicator, which does not have a corresponding numbered box, represents a change to holding pressure based on an external signal. At the time when this manual was written, Princeton's BOY 35E machine was not currently configured to use a manual input to transition to holding pressure. The circle which is filled in with green indicates the method currently being used to determine the changeover to holding pressure.

Page 4.30 contains the profile point settings for plasticizing, the process in which the screw retracts and rotates through the cavity to refill with plastic in preparation for the next injection phase. Page 4.40 includes selection indicators and settings for additional functionalities of the injection phase.

A graphic on the bottom of screens 4.10 - 4.30 shows the current position of the screw in the cavity. Note that since the plastic in the cavity must be plasticized, the location of the screw does not necessarily reflect the amount of molten plastic in the cavity. In order to ensure that the desired amount of plastic is present in the cavity before injection, use the "plasticize" movement button (Section 4.2.1) in manual operation mode (Section 3.1.4) starting at the forward-most position of the screw to fill the cavity in front of the screw with molten plastic. In manual operation mode, the plasticize operation will automatically stop when the screw position matches the position in profile point 0 (indicating that the screw is fully plasticized for current injection settings). In order to plasticize beyond the furthest position indicated in the injection settings, setup mode must be used.

For further information regarding the buttons used for injection and plasticizing, refer to pages 1.5/65- 88 the BOY 35E Instructions for Use [16]. Refer to Section 4.2.1 of this report for information on how to determine the positions, speeds and pressures required for injection and plasticizing.

4 Determining injection process parameters

4.1 Clamping force

Clamping force required depends on the projected area of the mold, how easily the material flows, and the depth of the mold. Estimate the projected area of the mold cavity in the mold using CAD software or rough measurements. If a mold has multiple cavities and/or runners, be sure to include the projected area of these features as well. The projected area of the mold is multiplied by a clamping factor between 0.027 kN/mm (2 tons/in2) and 0.11 kn/mm (8 tons/in2), which is related to how easily the material flows. A smaller factor is used for high flow materials and a higher factor is used for low flow materials. The melt flow index (MFI) test indicates how easily a material flows, and these results are available from the material's manufacturer. Melt flow indices typically range from about 5 to 20 where higher numbers represent more flow-ability (high flow) and lower numbers represent less flow-ability (low flow). While there is no clear conversion between MFI and clamping factor, Bryce's *Injection Molding Process Fundamentals* recommends that 0.07 kN/mm2 (5 tons/in2) be used as a default for materials with moderate MFI and clamping factor only needs to be adjusted for materials with extremely high or low MFI.

Depth only needs to be accounted for in the clamping force if the overall depth of the product (not the wall thickness) is thicker than 25 mm. Clamping force is increased by 10% for every inch of mold depth over 25 mm. Finally, the clamping force should be increased by 10% as a safety factor.

Sufficient clamping force is required to avoid flash (plastic that flows between the two mold halves) and underfilled parts. However, if too much clamping force is used the mold or press may be crushed. Even if the excessive clamping force is not strong enough to crush the mold, clamping force can cause the B side (moving) platen to twist or bind, leading to damage to the mold and molding machine in the long term.

For D $<$ 22.4 mm:

$$F_{\text{clamping}} = 1.1 * \text{PA} * \text{CF} \tag{4.1}$$

For D > 22.4 mm:

$$F_{\text{clamping}} = 1.21\,[(\text{D} - 25.4\text{mm}) * \text{PA} * \text{CF}] \tag{4.2}$$

Where D is mold depth, PA is projected area, and CF is clamping factor.

4.2 Mold closing and opening process

Designing the mold opening and closing process requires determining the mold opening distance, ejection distance and speed (if applicable), and the mold closing/opening speeds. In large-scale industrial applications, distances are minimized and speeds maximized to increase efficiency. However, for small-scale applications it is better to err on the side of safety (larger distances and slower speeds) to avoid potentially costly damage to the molding machine. If ejection pins are used, the mold opening distance must be large enough to account for the advance of the ejection pins as well as space for the mold to fall free from the machine after injection. In order to fully free a part, the ejection pins must advance the maximum depth of the mold. Therefore, the mold opening distance must be at least the depth of the mold plus $\frac{1}{4}$" for clearance.

The mold closing process should take place in two phases. In the first phase, the B side of the mold should move from its starting position to within 15 mm of the other side of the mold. This can take place relatively quickly (within 1-2 seconds). In the final phase, the movement speed should reduce as a safety feature to protect the mold halves from smashing together or crushing any obstruction that may be between them. The final 10-15mm of closing should take 2-3 seconds. When using 3D printed molds, which may be less resilient to closing forces, it is particularly important that the final closing phase takes place slowly.

The mold opening process also takes place in two phases. In the first phase, the B side of the mold slowly moves $\frac{1}{4}$ inch open, allowing time for the vacuum formed between the two mold halves to release. In the second phase, the moving side of the mold can rapidly return to its full open position. After the mold has been opened, the ejection pins are triggered. In most standard applications, the ejection pins advance in one smooth, relatively fast movement and retract the same way. This process should take 1-2 seconds.

After the mold is open and the ejection process has taken place, a short delay is included to allow the piece to clear the opening between the two molds and for the operator to inspect the mold for any stuck parts or jammed mechanisms before the cycle begins again.

4.2.1 Injection process

An estimate of the amount of plastic that will be injected into the mold is indicated in millimeters of injection distance. An estimate for the injection distance can be obtained by the following calculation:

$$d_{\text{Injection}} = (\mathrm{V}_{\text{mold}} * \mathrm{A}_{\text{cylinder}}) + \mathrm{d}_{\text{cushion}} \tag{4.3}$$

Where:

- $\mathrm{V}_{\text{cavity}}$ is the sum of the volume of all of the cavities in the mold as well as the volume of the runners in mm. This can be obtained using CAD software.
- $\mathrm{A}_{\text{cylinder}}$ is the area of the barrel (inside of the injection cylinder) in mm. For the BOY 35E, $\mathrm{A}_{\text{cylinder}}$ is 855 mm^2.
- $\mathrm{l}_{\text{cushion}}$ is the length of the cushion, a value between 3 and 6mm. The cushion will not be injected into the mold, but will remain in the barrel between the screw and nozzle to facilitate the "packing" of plastic into the mold.

Initial injection

As mentioned above, the injection process takes place in two phases. During the first phase, 95% of the plastic required to fill the mold is injected. The pressure for this stage typically ranges from 345 bar to 1050 bar. While a higher pressure may be beneficial because it causes the injection to occur more quickly, high pressure can damage the mold and lead to molded-in stresses which will weaken the final product. This is particularly true if a 3D printed mold is being used, as it may not hold up to the repeated stress of a high pressure injection. Therefore, it is best to begin at a low pressure and increase as needed.

Packing phase

During the packing phase, the final 5% of the plastic is injected into the mold and pressure is held against the "cushion" between the screw and the nozzle. Therefore, the forward-most position of the screw should be set to $\mathrm{l}_{\text{cushion}}$, as used in Equation 4.3. The packing phase is necessary to ensure that the cavity image is filled and to allow it to begin to solidify. The pressure in the packing phase should be half the pressure used in the initial injection stage or less.

Plasticizing

During the plasticizing phase, the barrel is refilled with material in preparation for the next injection shot. A back pressure is applied to the screw causing it to rotate backwards. As new material moves through the screw and becomes plasticized the screw moves backwards until enough material has been plasticized. During plasticization, the material becomes more uniform and air bubbles are eliminated. The back pressure required for this process is between 3.5 bar and 35 bar depending on the material, but a lower back pressure is recommended because applying excessive shear may degrade the material. It is recommended that a back pressure of 3.5 bar is used initially and increased only as required.

4.3 Temperature

In the injection molding machine, plastic begins as pellets in the hopper and is moved through the barrel of the heating cylinder by the screw until it is ejected from the nozzle and into the mold. As the material travels along this process it should be gradually heated by the heater bands surrounding the cylinder as well as friction caused by the rotation of the screw pressing the plastic forward in the barrel. The heating cylinder of the BOY 35E is divided into 5 heating zones which begin at the hopper end of the cylinder and end at the nozzle, where the plastic should have reached its melting temperature. The heat should gradually increase by 3° to 4°C in each band leading up to this point. A material's melting point should be available on the material data sheet, but Bryce's *Injection Molding I: Process Fundamentals* provides a table for commonly used materials which I have included in Table 4.1 as a point of reference.

Table 4.1: Suggested Melt Temperature for Commonly Used Thermoplastics [5]

Material	Temperature, °C (°F)
ABS	204 (400)
High Density Polyethylene (HDPE)	204 (400)
Low Density Polyethylene (LDPE)	163 (325)
Polycarbonate (PC)	288 (550)
Polyamide (Nylon)	260 (500)
High Impact Polystyrene (HIPS)	199 (390)
Polypropylene (PP)	177 (350)

5 Troubleshooting

While this section is far from comprehensive, I have included the troubleshooting process that I followed to resolve the most common problems I encountered while working with the BOY 35E machine. For more troubleshooting resources, refer to *Bryce's Injection Molding I: Process Fundamentals, Chapter 11* [6].

5.1 Machine operation problems

Symptom: Mechanical component does not move when button is pressed

- Confirm component has not reached movement limit in desired direction.
- Ensure that the doors to both gates are securely closed.
- Check that motor is still on. If not, switch motor to "on" and allow oil to reheat.

Symptom: Error message - toothed rack

It is not entirely clear what the toothed rack error means or what causes it. Based on my research, it is likely related to a toothed timing belt which gets out of alignment with respect to a gear or other rack. When this error occurs, mechanical components are still able to move but the injection process cannot proceed.

- Use component advance/retract buttons (Section 3.1.3) to move components forward and back. This may resolve the error.
- Otherwise, use the rotary switch to turn the electricity off, wait approximately 1 minute for capacitors to discharge, and turn the electricity back on.

Symptom: Plastic is not flowing from the nozzle/failure to complete plasticizing step in a reasonable time during injection process

- Ensure the plastic hopper is not empty.
- Check that water cooling system is at a reasonable temperature. If the water cooling system is not functioning properly (or if the water is not on), it may have created a partially melted clot of plastic pellets at the inlet of the injection cylinder which is blocking the cylinder from being filled. If this is the case, first troubleshoot the cooling system:
  - Ensure that yellow water pipe valves on the back wall are open and that the switch on the back wall is in the "on" position.
  - Confirm that water is flowing through the water coolant distributor on the back of the machine.
  - If necessary, increase the flow rate of the water through the distributor by turning the knob to open the valve further.
  - To clear a plastic clog, move the screw to the back of the injection cylinder and plasticize, purge the plastic. It may also be necessary to use a ShopVac to empty the section of pipe between the hopper and the cylinder in order to use a long implement to clear the melted plastic lump from above. This should be done with extreme caution when the machine is turned off and after the plastic has cooled.

Symptom: Semi-automatic or automatic injection process will not begin, "start" button is not flashing green

- Confirm that the B side of the mold is in the starting position or more open. If not, use the "clamping unit retract" button (Section 3.1.3) to adjust the position.

Figure 5.1: Injection molded coaster which has burn marks, air bubbles and is underfilled.

Figure 5.2: Injection molded gear with flash.

Symptom: Semi-automatic or automatic injection process will not begin, "start" button is flashing green

- Check that screw temperatures are within the programmed limits. If not, turn on heater (Section 3.1.2).

5.2 Unsatisfactory injection results

Symptom: Underfilled mold

- Increase injection distance. If the mold has large unfilled areas, the amount of material being injected may not be sufficient. Increase the amount of material by increasing the injection distance (Section 4.2.1).
- Increase mold packing. If the mold is nominally filled but small details or corners are still vacant, the pressure or duration of the "packing phase" of the injection stage may need to be increased (Section 4.2.1).

Symptom: Air bubbles

- Reduce injection temperatures. If material becomes too liquid during injection, gases may be trapped during the injection process.
- Empty material from in front of the screw using the "screw advance button" and replasticize in manual mode to eliminate voids in the injection cylinder.

Symptom: Burn marks

- Reduce heating band temperatures and purge injection cylinder using "screw advance/retract" and "plasticize" buttons until material is clear.
- Improve cycle consistency or run in automatic mode if possible. Excessive time between injection cycles may allow material in the injection cylinder to burn due to hot spots in the heating bands.

Symptom: Flash

- Reduce injection distance to avoid overfilling mold. Extra material that cannot fit into the mold is pushed between the two plates, causing flash.
- Increase venting. If mold is underfilled but flash is still occurring, ensure that all air is able to vent quickly from the mold during the filling process. Material that is being pushed out of the mold by escaping air may be causing flash.
- Increase clamping force. If the two sides of the mold are not closed with enough force, material may flow between them, causing flash.

Appendix B: Finite Element Analysis

Finite Element Analysis (FEA) was performed in SolidWorks on the mold insert and housing assembly after consistent cracking was observed at the junction between the tabs and the body of the mold insert. The material definition used for the simulation is based on the material data available for Formlabs' standard, uncured resin in order to establish a conservative estimate of the mold behavior. The Standard resin has a lower heat deflection temperature (HDT) and, according to the datasheets, a lower impact resistance than the cured High Temperature resin. Although testing results showed that the High Temperature mold was more brittle, I believe that this behavior was caused by over-curing the material [21]. Since all of Formlabs' materials are proprietary, the values are estimated based on the limited material properties released by the company in their datasheets [19]. The SLA resin material definition used for simulations is included in table 5.1. The aluminum material was defined using the 6061 Aluminum properties from the CES materials database [7].

Table 5.1: Material Definition: Standard SLA Resin

Property	Value
Poisson's Ratio	0.4
Young's Modulus	1300 MPa
Coefficient of Thermal Expansion	110 $\frac{1}{°C}$
Tensile Yield Stress	29.8 MPa
Tensile Ultimate Stress	29.9 MPa

The simulation was performed on the fourth iteration of the mold insert in the aluminum housing. The back of the aluminum housing, where it is mounted on the cradle, was held in place and the mold insert was attached to the housing using fastener constraints. An additional interaction between the insert and housing was defined to prevent the materials from deforming into each other. A force of 200 kN was applied perpendicular to the mold to simulate the clamping force that occurs during the injection molding process when the sides of the mold are closed.

The results of the simulation show that the area of highest stress occurs at the junction between the tab and the body of the mold insert. The overall stress in this area is approximately 3,800 psi with some areas reaching stress levels as high as 6,500 psi. There are also some areas of relatively high stress at the outer corners of the mold insert, and around the sides of the teeth. The results of this simulation are shown in Figure 4.9.

I considered two possible modifications to reduce the stress at the base of the tabs and reduce the possibility of cracking due to the clamping force. I performed the FEA clamping force simulation on each to evaluate the design and inform my recommendations for future work.

For the first possible design modification, I increased the length and width of the tab from 0.7 inch to 1 inch. The simulation showed that this change did not reduce the maximum stress in the mold, and the amount of stress at the junction between the tab and the body of the insert and the areas of maximum stress (6,500 psi) are more concentrated. The results of this simulation are shown in Figure 4.10.

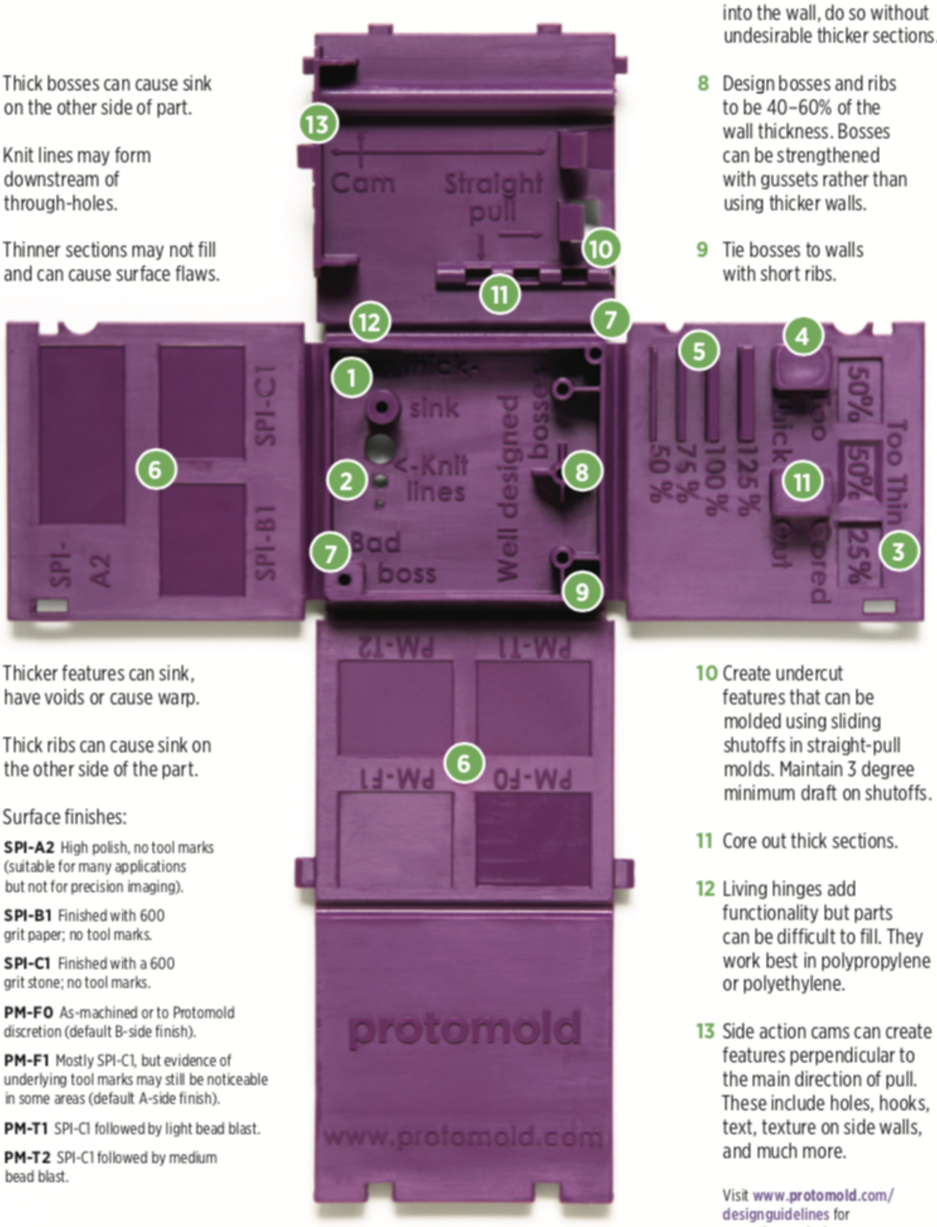
For the second possible design modification, I increased the thickness of the tab from $\frac{5}{32}$ inch to $\frac{3}{8}$ inch. This change had a meaningful impact on the stress level in the mold insert. The stress in the area between the tab and body of the mold is reduced to approximately 3300 psi. The results of this simulation are shown in Figure 4.11.

Although the material properties of the SLA resin used for the FEA simulation are only an estimation, performing this analysis confirmed my suspicions regarding the root cause of the cracking in the mold and helped me determine the most effective design change to remedy this problem in future design iterations.

Appendix C: Protolabs Injection Molded Design Cube

DESIGN SUGGESTIONS

proto labs
Real Parts. Really Fast.



1 Thick bosses can cause sink on the other side of part.

2 Knit lines may form downstream of through-holes.

3 Thinner sections may not fill and can cause surface flaws.

4 Thicker features can sink, have voids or cause warp.

5 Thick ribs can cause sink on the other side of the part.

6 Surface finishes:

SPI-A2 High polish, no tool marks (suitable for many applications but not for precision imaging).

SPI-B1 Finished with 600 grit paper; no tool marks.

SPI-C1 Finished with a 600 grit stone; no tool marks.

PM-F0 As-machined or to Protomold discretion (default B-side finish).

PM-F1 Mostly SPI-C1, but evidence of underlying tool marks may still be noticeable in some areas (default A-side finish).

PM-T1 SPI-C1 followed by light bead blast.

PM-T2 SPI-C1 followed by medium bead blast.

7 If you incorporate the boss into the wall, do so without undesirable thicker sections.

8 Design bosses and ribs to be 40–60% of the wall thickness. Bosses can be strengthened with gussets rather than using thicker walls.

9 Tie bosses to walls with short ribs.

10 Create undercut features that can be molded using sliding shutoffs in straight-pull molds. Maintain 3 degree minimum draft on shutoffs.

11 Core out thick sections.

12 Living hinges add functionality but parts can be difficult to fill. They work best in polypropylene or polyethylene.

13 Side action cams can create features perpendicular to the main direction of pull. These include holes, hooks, text, texture on side walls, and much more.

Visit www.protomold.com/designguidelines for more tips on design.

www.protomold.com | 877.479.3680

Appendix D: Cost Analysis Breakdown

Mold Costs	
Mold Housing	
$	37.53
SLA Standard Mold	
$	7.50
SLA High Temp Mold	
$	10.00
Aluminum mold	
$	750.00

Number of Parts	Injection Molded Parts	Machined Parts	Total Std. SLA	Total High Temp SLA	Total Aluminum Mold
1	$ 0.01	$ 3.79	$ 45.04	$ 47.54	$ 750.01
2	$ 0.01	$ 7.58	$ 45.04	$ 47.54	$ 750.01
3	$ 0.02	$ 11.37	$ 45.05	$ 47.55	$ 750.02
4	$ 0.02	$ 15.16	$ 45.05	$ 47.55	$ 750.02
5	$ 0.03	$ 18.95	$ 45.06	$ 47.56	$ 750.03
6	$ 0.03	$ 22.74	$ 45.06	$ 47.56	$ 750.03
7	$ 0.04	$ 26.53	$ 45.07	$ 47.57	$ 750.04
8	$ 0.04	$ 30.32	$ 45.07	$ 47.57	$ 750.04
9	$ 0.05	$ 34.11	$ 45.08	$ 47.58	$ 750.05
10	$ 0.05	$ 37.90	$ 45.08	$ 47.58	$ 750.05
11	$ 0.06	$ 41.69	$ 45.09	$ 47.59	$ 750.06
12	$ 0.06	$ 45.48	$ 45.09	$ 47.59	$ 750.06
13	$ 0.07	$ 49.27	$ 45.10	$ 47.60	$ 750.07
14	$ 0.07	$ 53.06	$ 45.10	$ 47.60	$ 750.07
15	$ 0.08	$ 56.85	$ 45.11	$ 47.61	$ 750.08
16	$ 0.08	$ 60.64	$ 45.11	$ 47.61	$ 750.08
17	$ 0.09	$ 64.43	$ 45.12	$ 47.62	$ 750.09
18	$ 0.09	$ 68.22	$ 45.12	$ 47.62	$ 750.09
19	$ 0.10	$ 72.01	$ 45.13	$ 47.63	$ 750.10
20	$ 0.10	$ 75.80	$ 45.13	$ 47.63	$ 750.10
21	$ 0.11	$ 79.59	$ 45.14	$ 47.64	$ 750.11
22	$ 0.11	$ 83.38	$ 45.14	$ 47.64	$ 750.11
23	$ 0.12	$ 87.17	$ 45.15	$ 47.65	$ 750.12
24	$ 0.12	$ 90.96	$ 45.15	$ 47.65	$ 750.12
25	$ 0.13	$ 94.75	$ 45.16	$ 47.66	$ 750.13
26	$ 0.13	$ 98.54	$ 45.16	$ 47.66	$ 750.13
27	$ 0.14	$ 102.33	$ 45.17	$ 47.67	$ 750.14
28	$ 0.14	$ 106.12	$ 45.17	$ 47.67	$ 750.14
29	$ 0.15	$ 109.91	$ 45.18	$ 47.68	$ 750.15
30	$ 0.15	$ 113.70	$ 45.18	$ 47.68	$ 750.15
31	$ 0.16	$ 117.49	$ 45.19	$ 47.69	$ 750.16
32	$ 0.16	$ 121.28	$ 45.19	$ 47.69	$ 750.16
33	$ 0.17	$ 125.07	$ 45.20	$ 47.70	$ 750.17
34	$ 0.17	$ 128.86	$ 45.20	$ 47.70	$ 750.17
35	$ 0.18	$ 132.65	$ 45.21	$ 47.71	$ 750.18
36	$ 0.18	$ 136.44	$ 45.21	$ 47.71	$ 750.18
37	$ 0.19	$ 140.23	$ 45.22	$ 47.72	$ 750.19
38	$ 0.19	$ 144.02	$ 45.22	$ 47.72	$ 750.19
39	$ 0.20	$ 147.81	$ 45.23	$ 47.73	$ 750.20
40	$ 0.20	$ 151.60	$ 45.23	$ 47.73	$ 750.20
41	$ 0.21	$ 155.39	$ 45.24	$ 47.74	$ 750.21
42	$ 0.21	$ 159.18	$ 45.24	$ 47.74	$ 750.21
43	$ 0.22	$ 162.97	$ 45.25	$ 47.75	$ 750.22
44	$ 0.22	$ 166.76	$ 45.25	$ 47.75	$ 750.22
45	$ 0.23	$ 170.55	$ 45.26	$ 47.76	$ 750.23
46	$ 0.23	$ 174.34	$ 45.26	$ 47.76	$ 750.23
47	$ 0.24	$ 178.13	$ 45.27	$ 47.77	$ 750.24
48	$ 0.24	$ 181.92	$ 45.27	$ 47.77	$ 750.24

Number of Parts	Injection Molded Parts	Machined Parts	Total Std. SLA	Total High Temp SLA	Total Aluminum Mold
49	$ 0.25	$ 185.71	$ 45.28	$ 47.78	$ 750.25
50	$ 0.25	$ 189.50	$ 45.28	$ 47.78	$ 750.25
51	$ 0.26	$ 193.29	$ 45.29	$ 47.79	$ 750.26
52	$ 0.26	$ 197.08	$ 45.29	$ 47.79	$ 750.26
53	$ 0.27	$ 200.87	$ 45.30	$ 47.80	$ 750.27
54	$ 0.27	$ 204.66	$ 45.30	$ 47.80	$ 750.27
55	$ 0.28	$ 208.45	$ 45.31	$ 47.81	$ 750.28
56	$ 0.28	$ 212.24	$ 45.31	$ 47.81	$ 750.28
57	$ 0.29	$ 216.03	$ 45.32	$ 47.82	$ 750.29
58	$ 0.29	$ 219.82	$ 45.32	$ 47.82	$ 750.29
59	$ 0.30	$ 223.61	$ 45.33	$ 47.83	$ 750.30
60	$ 0.30	$ 227.40	$ 45.33	$ 47.83	$ 750.30
61	$ 0.31	$ 231.19	$ 45.34	$ 47.84	$ 750.31
62	$ 0.31	$ 234.98	$ 45.34	$ 47.84	$ 750.31
63	$ 0.32	$ 238.77	$ 45.35	$ 47.85	$ 750.32
64	$ 0.32	$ 242.56	$ 45.35	$ 47.85	$ 750.32
65	$ 0.33	$ 246.35	$ 45.36	$ 47.86	$ 750.33
66	$ 0.33	$ 250.14	$ 45.36	$ 47.86	$ 750.33
67	$ 0.34	$ 253.93	$ 45.37	$ 47.87	$ 750.34
68	$ 0.34	$ 257.72	$ 45.37	$ 47.87	$ 750.34
69	$ 0.35	$ 261.51	$ 45.38	$ 47.88	$ 750.35
70	$ 0.35	$ 265.30	$ 45.38	$ 47.88	$ 750.35
71	$ 0.36	$ 269.09	$ 45.39	$ 47.89	$ 750.36
72	$ 0.36	$ 272.88	$ 45.39	$ 47.89	$ 750.36
73	$ 0.37	$ 276.67	$ 45.40	$ 47.90	$ 750.37
74	$ 0.37	$ 280.46	$ 45.40	$ 47.90	$ 750.37
75	$ 0.38	$ 284.25	$ 45.41	$ 47.91	$ 750.38
76	$ 0.38	$ 288.04	$ 45.41	$ 47.91	$ 750.38
77	$ 0.39	$ 291.83	$ 45.42	$ 47.92	$ 750.39
78	$ 0.39	$ 295.62	$ 45.42	$ 47.92	$ 750.39
79	$ 0.40	$ 299.41	$ 45.43	$ 47.93	$ 750.40
80	$ 0.40	$ 303.20	$ 45.43	$ 47.93	$ 750.40
81	$ 0.41	$ 306.99	$ 45.44	$ 47.94	$ 750.41
82	$ 0.41	$ 310.78	$ 45.44	$ 47.94	$ 750.41
83	$ 0.42	$ 314.57	$ 45.45	$ 47.95	$ 750.42
84	$ 0.42	$ 318.36	$ 45.45	$ 47.95	$ 750.42
85	$ 0.43	$ 322.15	$ 45.46	$ 47.96	$ 750.43
86	$ 0.43	$ 325.94	$ 45.46	$ 47.96	$ 750.43
87	$ 0.44	$ 329.73	$ 45.47	$ 47.97	$ 750.44
88	$ 0.44	$ 333.52	$ 45.47	$ 47.97	$ 750.44
89	$ 0.45	$ 337.31	$ 45.48	$ 47.98	$ 750.45
90	$ 0.45	$ 341.10	$ 45.48	$ 47.98	$ 750.45
91	$ 0.46	$ 344.89	$ 45.49	$ 47.99	$ 750.46
92	$ 0.46	$ 348.68	$ 45.49	$ 47.99	$ 750.46
93	$ 0.47	$ 352.47	$ 45.50	$ 48.00	$ 750.47
94	$ 0.47	$ 356.26	$ 45.50	$ 48.00	$ 750.47
95	$ 0.48	$ 360.05	$ 45.51	$ 48.01	$ 750.48
96	$ 0.48	$ 363.84	$ 45.51	$ 48.01	$ 750.48

Number of Parts	Injection Molded Parts	Machined Parts	Total Std. SLA	Total High Temp SLA	Total Aluminum Mold
97	$ 0.49	$ 367.63	$ 45.52	$ 48.02	$ 750.49
98	$ 0.49	$ 371.42	$ 45.52	$ 48.02	$ 750.49
99	$ 0.50	$ 375.21	$ 45.53	$ 48.03	$ 750.50
100	$ 0.50	$ 379.00	$ 45.53	$ 48.03	$ 750.50

Appendix E: Selected Engineering Drawings

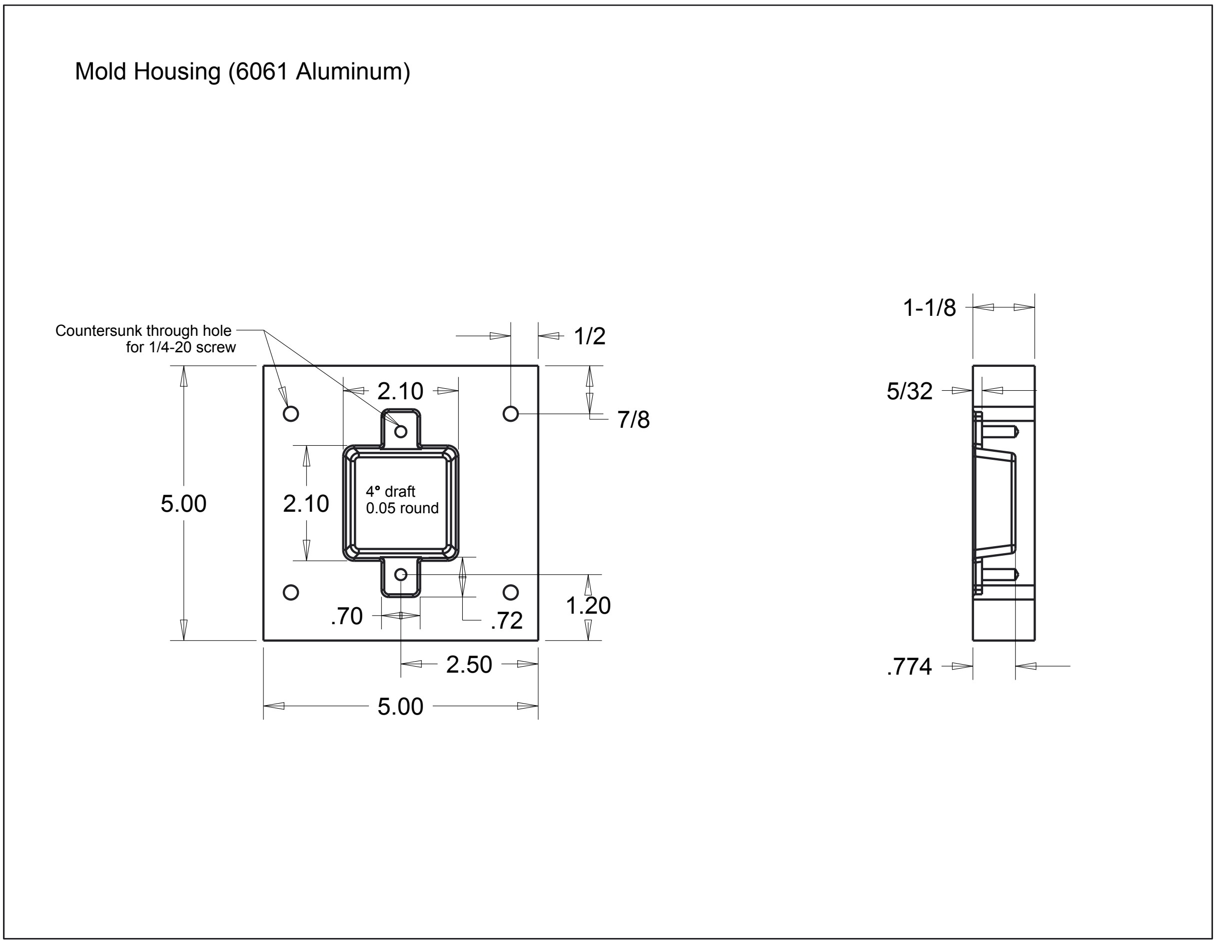


Figure 5.3: Key dimensions of mold aluminum housing

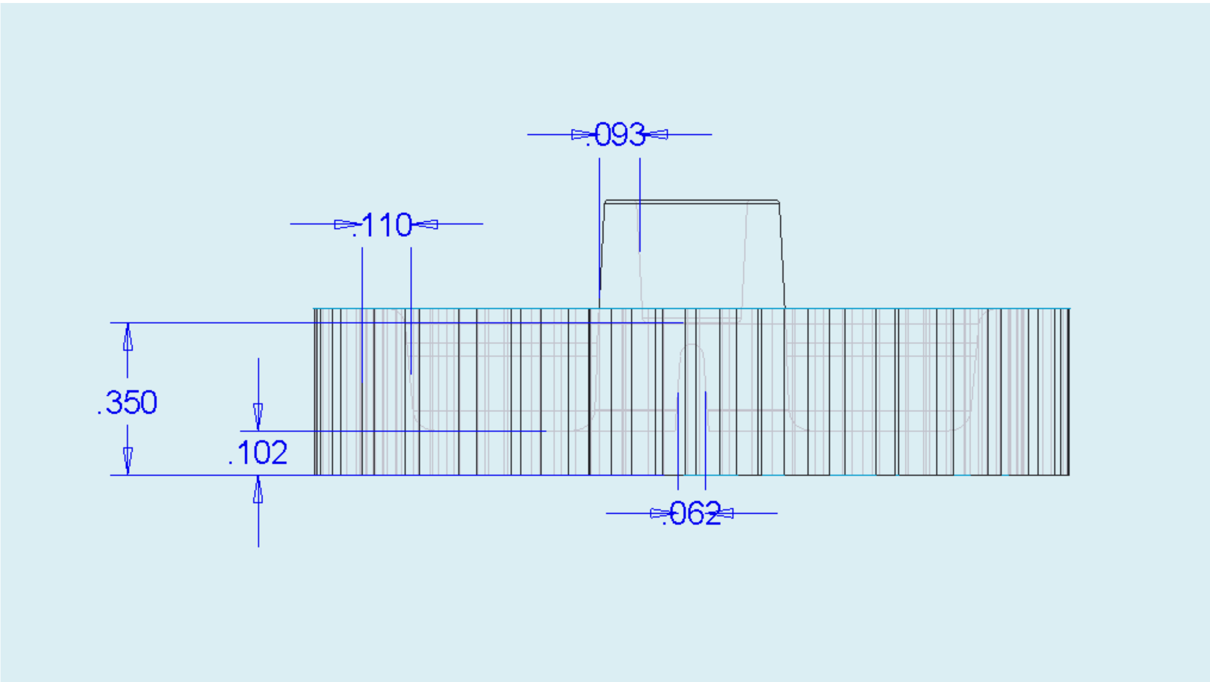


Figure 5.4: Wall thicknesses in the fourth iteration of the gear mold

Appendix F: Material Datasheets

ABS-M30

PRODUCTION-GRADE THERMOPLASTIC FOR FDM 3D PRINTERS

ABS-M30™ is up to 25 to 70 percent stronger than standard ABS and is an ideal material for conceptual modeling, functional prototyping, manufacturing tools and production parts. ABS-M30 has greater tensile, impact and flexural strength than standard ABS. Layer bonding is significantly stronger than that of standard ABS, for a more durable part. This results in more realistic functional tests and higher quality parts for end use. ABS-M30 parts are stronger, smoother and have better feature detail. ABS-M30 runs the Xtend 500 Fortus Plus option, which enables more than 400 hours of unattended build time.

MECHANICAL PROPERTIES[1]	TEST METHOD	ENGLISH		METRIC	
		XZ Axis	*ZX Axis*	*XZ Axis*	*ZX Axis*
Tensile Strength, Yield (Type 1, 0.125", 0.2"/min)	ASTM D638	4,550 psi	3,750 psi	31 MPa	26 MPa
Tensile Strength, Ultimate (Type 1, 0.125", 0.2"/min)	ASTM D638	4,650 psi	4,050 psi	32 MPa	28 MPa
Tensile Modulus (Type 1, 0.125", 0.2"/min)	ASTM D638	320,000 psi	310,000 psi	2,230 MPa	2,180 MPa
Tensile Elongation at Break (Type 1, 0.125", 0.2"/min)	ASTM D638	7%	2%	7%	2%
Tensile Elongation at Yield (Type 1, 0.125", 0.2"/min)	ASTM D638	2%	1%	2%	1%
Flexural Strength (Method 1, 0.05"/min)	ASTM D790	8,700 psi	7,000 psi	60 MPa	48 MPa
Flexural Modulus (Method 1, 0.05"/min)	ASTM D790	300,000 psi	250,000 psi	2,060 MPa	1,760 MPa
Flexural Strain at Break (Method 1, 0.05"/min)	ASTM D790	4%	3.5%	4%	3.5%

MECHANICAL PROPERTIES	TEST METHOD	ENGLISH	METRIC
		XZ Axis	*XZ Axis*
IZOD Impact, notched (Method A, 23°C)	ASTM D256	2.4 ft-lb/n	128 J/m
IZOD Impact, un-notched (Method A, 23°C)	ASTM D256	5.6 ft-lb/in	300 J/m

THERMAL PROPERTIES[2]	TEST METHOD	ENGLISH	METRIC
Heat Deflection (HDT) @ 66 psi, 0.125" unannealed	ASTM D648	204°F	96°C
Heat Deflection (HDT) @ 264 psi, 0.125" unannealed	ASTM D648	180°F	82°C
Vicat Softening Temperature (Rate B/50)	ASTM D1525	210°F	99°C
Glass Transition (Tg)	DMA (SSYS)	226°F	108°C
Coefficient of Thermal Expansion (flow)	ASTM E831	4.90x10^{-05} in/in/°F	8.82x10^{-05} mm/mm/°C
Coefficient of Thermal Expansion (xflow)	ASTM E831	4.70x10^{-05} in/in/°F	8.46x10^{-05} mm/mm/°C
Melting Point	- - - - - - - - -	Not Applicable[2]	Not Applicable[2]

ABS-M30

PRODUCTION-GRADE THERMOPLASTIC FOR FDM 3D PRINTERS

At the core: Advanced FDM Technology

FDM® (fused deposition modeling) technology works with engineering-grade thermoplastics to build strong, long-lasting and dimensionally stable parts with the best accuracy and repeatability of any 3D printing technology. These parts are tough enough to be used as advanced conceptual models, functional prototypes, manufacturing tools and production parts.

Meet production demands

FDM systems are as versatile and durable as the parts they produce. Advanced FDM 3D Printers boast the largest build envelopes and material capacities in their class, delivering longer, uninterrupted build times, bigger parts and higher quantities than other additive manufacturing systems, delivering high throughput, duty cycles and utilization rates.

Opening the way for new possibilities

FDM 3D Printers streamline processes from design through manufacturing, reducing costs and eliminating traditional barriers along the way. Industries can cut lead times and costs, products turn out better and get to market faster.

No special facilities needed

FDM 3D Printers are easy to operate and maintain compared to other additive fabrication systems because there are no messy powders or resins to handle and contain, and no special venting is required because FDM systems don't produce noxious fumes, chemicals or waste.

ELECTRICAL PROPERTIES[3]	TEST METHOD	ORIENTATION	VALUE RANGE
Volume Resistivity	ASTM D257	XZ Axis	4.0×10^{15} - 3.3×10^{16} ohm-cm
Dielectric Constant	ASTM D150-98	XZ Axis	2.6 - 2.86
Dissipation Factor	ASTM D150-98	XZ Axis	0.0048 - 0.0054
Dielectric Strength	ASTM D149-09, Method A	XY Axis	100 V/mil
Dielectric Strength	ASTM D149-09, Method A	XZ Axis	360 V/mil

OTHER[1]	TEST METHOD	VALUE
Specific Gravity	ASTM D792	1.04
Rockwell Hardness	ASTM D785	109.5

SYSTEM AVAILABILITY	LAYER THICKNESS CAPABILITY	SUPPORT STRUCTURE	AVAILABLE COLORS	
Fortus 380mc™	0.013 inch (0.330 mm)	Soluble Supports	Ivory	White
Fortus 450mc™	0.010 inch (0.254 mm)		Black	Dark Grey
Fortus 900mc™	0.007 inch (0.178 mm)		Red	Blue
Stratasys F123™ Series	0.005 inch (0.127 mm)[4]			

The information presented are typical values intended for reference and comparison purposes only. They should not be used for design specifications or quality control purposes. End-use material performance can be impacted (+/-) by, but not limited to, part design, end-use conditions, test conditions, etc. Actual values will vary with build conditions. Tested parts were built on Fortus 400mc™ @ 0.010" (0.254 mm) slice. Product specifications are subject to change without notice.

The performance characteristics of these materials may vary according to application, operating conditions or end use. Each user is responsible for determining that the Stratasys material is safe, lawful and technically suitable for the intended application, as well as for identifying the proper disposal (or recycling) method c onsistent with applicable environmental laws and regulations. Stratasys makes no warranties of any kind, express or implied, including, but not limited to, the warranties of merchantability, fitness for a particular use, or warranty against patent infringement.

[1]Literature value unless otherwise noted.

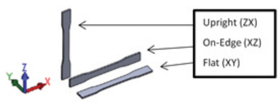
[2]Due to amorphous nature, material does not display a melting point.

[3]All Electrical Property values were generated from the average of test plaques built with default part density (solid). Test plaques were 4.0 x 4.0 x 0.1 inches (102 x 102 x 2.5 mm) and were built both in the flat and vertical orientation. The range of values is mostly the result of the difference in properties of test plaques built in the flat vs. vertical orientation.

[4]0.005 inch (0.127 mm) layer thickness not available for Fortus 900mc.

Colors: The test data was collected using ABS-M30 Ivory (natural) specimens. ABS-M30 colored material will have similar properties, but can vary by up to 10%. Orientation: See Stratasys T esting white paper for more detailed description of build orientations.

- *XZ = X or "on edge"*
- *XY = Y or "flat"*
- *ZX = or "upright"*



stratasys

HEADQUARTERS

7665 Commerce Way, Eden Prairie, MN 55344
+1 888 480-3548 (US Toll Free)
+1 952 937-3000 (Intl)
+1 952 937-0070 (Fax)

1 Holtzman St., Science Park, PO Box 2496
Rehovot 76124, Israel
+972 74 745-4000
+972 74 745-5000 (Fax)

STRATASYS.COM
ISO 9001:2008 Certified

MATERIAL DATA SHEET

High Temp

High Temp Resin for Heat Resistance

$199 / L

High Temp Resin offers a heat deflection temperature (HDT) of 238 °C @ 0.45 MPa, the highest among Formlabs resins. Use it to print detailed, precise prototypes with high temperature resistance.

Hot air, gas, and fluid flow

Molds and inserts

Heat resistant mounts, housings, and fixtures

FLHTAM02

formlabs

Prepared 09 . 15 . 2016
Rev 02 12 . 05 . 2018

Material Properties Data Metric

	METRIC[1]			METHOD
	Green[2]	Post-Cured[3]	Post-Cured + Thermally Post-Cured[4]	
Thermal Properties				
Heat Deflection Temp. @ 1.8 MPa	43.6 °C	99.2 °C	101 °C	ASTM D 648-16
Heat Deflection Temp. @ 0.45 MPa	49.3 °C	142 °C	238 °C	ASTM D 648-16

	METRIC[1]			METHOD
	Green[2]	Post-Cured[5]	Post-Cured + Thermally Post-Cured[6]	
Mechanical Properties				
Ultimate Tensile Strength	20.9 MPa	58.3 MPa	51.1 MPa	ASTM D 638-14
Elongation at break	14 %	3.3 %	2.4 %	ASTM D 638-14
Tensile modulus	0.75 GPa	2.75 GPa	2.9 GPa	ASTM D 638-14
Flexural strength at break	24.1 MPa	94.5 MPa	93.8 MPa	ASTM D 790-15
Flexural modulus	0.69 GPa	2.62 GPa	2.62 GPa	ASTM D 790-15
Impact Properties				
Notched IZOD	32.8 J/m	18.2 J/m	24.2 J/m	ASTM D 256-10
Thermal Properties				
Thermal Expansion (0-150 °C)	118.1 (μm/m/°C)	79.6 (μm/m/°C)	74 (μm/m/°C)	ASTM E 831-13

[1] Material properties can vary with part geometry, print orientation, print settings, and temperature.

[2] Data was obtained from green parts, printed using Form 2, 100 μm, High Temp settings, washed for 5 minutes in Form Wash and air dried without post cure.

[3] Data was obtained from parts printed using a Form 2, 100 micron, High Temp settings, and post-cured with Form Cure at 80 °C for 120 minutes.

[4] Data was obtained from parts printed using a Form 2, 100 micron, High Temp settings, and post-cured with Form Cure at 80 °C for 120 minutes plus an additional thermal cure in a lab oven at 160 °C for 180 minutes.

[5] Data was obtained from parts printed using a Form 2, 100 micron, High Temp settings, and post-cured with Form Cure at 60 °C for 60 minutes.

[6] Data was obtained from parts printed using a Form 2, 100 micron, High Temp settings, and post-cured with Form Cure at 60 °C for 60 minutes plus an additional thermal cure in a lab oven at 160 °C for 90 minutes

Material Properties Data Imperial

	IMPERIAL[1]			METHOD
	Green[2]	Post-Cured[3]	Post-Cured + Thermally Post-Cured[4]	
Thermal Properties				
Heat Deflection Temp. @ 1.8 MPa	110.48 °F	210.56 °F	213.8 °F	ASTM D 648-16
Heat Deflection Temp. @ 0.45 MPa	120.74 °F	287.6 °F	460.4 °F	ASTM D 648-16

	IMPERIAL[1]			METHOD
	Green[2]	Post-Cured[5]	Post-Cured + Thermally Post-Cured[6]	
Mechanical Properties				
Ultimate Tensile Strength	3031 psi	8456 psi	7411 psi	ASTM D 638-14
Elongation at break	14 %	3.3 %	2.4 %	ASTM D 638-14
Tensile modulus	109 ksi	399 ksi	421 ksi	ASTM D 638-14
Flexural strength at break	3495 psi	13706 psi	13605 psi	ASTM D 790-15
Flexural modulus	100 ksi	400 ksi	400 ksi	ASTM D 790-15
Impact Properties				
Notched IZOD	0.61 ft-lbf/in	0.34 ft-lbf/in	0.45 ft-lbf/in	ASTM D 256-10
Thermal Properties				
Thermal Expansion (0-150 °C)	65.6 µin/in/°F	44.2 µin/in/°F	41.1 µin/in/°F	41.1 uin/in/°F

[1] Material properties can vary with part geometry, print orientation, print settings, and temperature.

[2] Data was obtained from green parts, printed using Form 2, 100 µm, High Temp settings, washed for 5 minutes in Form Wash and air dried without post cure.

[3] Data was obtained from parts printed using a Form 2, 100 micron, High Temp settings, and post-cured with Form Cure at 80 °C for 120 minutes.

[4] Data was obtained from parts printed using a Form 2, 100 micron, High Temp settings, and post-cured with Form Cure at 80 °C for 120 minutes plus an additional thermal cure in a lab oven at 160 °C for 180 minutes.

[5] Data was obtained from parts printed using a Form 2, 100 micron, High Temp settings, and post-cured with Form Cure at 60 °C for 60 minutes.

[6] Data was obtained from parts printed using a Form 2, 100 micron, High Temp settings, and post-cured with Form Cure at 60 °C for 60 minutes plus an additional thermal cure in a lab oven at 160 °C for 90 minutes

Solvent Compatibility

Percent weight gain over 24 hours for a printed and post-cured 1 x 1 x 1 cm cube immersed in respective solvent:

Solvent	24 hr weight gain (%)	24 hr size gain (%)	Solvent	24 hr weight gain (%)	24 hr size gain (%)
Acetic Acid, 5 %	< 1	< 1	Hydrogen peroxide (3%)	< 1	< 1
Acetone	< 1	< 1	Isooctane (aka gasoline)	< 1	< 1
Isopropyl Alcohol	< 1	< 1	Mineral oil (light)	< 1	< 1
Bleach ~5% NaOCl	< 1	< 1	Mineral oil (Heavy)	< 1	< 1
Butyl Acetate	< 1	< 1	Salt Water (3.5% NaCl)	< 1	< 1
Diesel Fuel	< 1	< 1	Sodium Hydroxide solution	< 1	< 1
Diethyl glycol Monomethyl Ether	< 1	< 1	Water	< 1	< 1
Hydraulic Oil	< 1	< 1	Xylene	< 1	< 1
Skydrol 5	< 1	< 1	Strong Acid (HCl conc)	1.2	< 1

MATERIAL DATA SHEET

Standard

Materials for High-Resolution Rapid Prototyping

High Resolution. For demanding applications, our carefully-engineered resins capture the finest features in your model.

Strength and Precision. Our resins create accurate and robust parts, ideal for rapid prototyping and product development.

Surface Finish. Perfectly smooth right out of the printer, parts printed on the Form 2 printer have the polish and finish of a final product.

CLEAR	WHITE	GREY	BLACK	COLOR
FLGPCL04	FLGPWH04	FLGPGR04	FLGPBL04	FLGPCB01

Prepared 04 . 19 . 2016
Rev 01 04 . 18 . 2017

Material Properties Data

The following material properties are comparable for all Formlabs Standard Resins.

	METRIC[1]		IMPERIAL[1]		METHOD
	Green[2]	Post-Cured[3]	Green[2]	Post-Cured[3]	
Tensile Properties					
Ultimate Tensile Strength	38 MPa	65 MPa	5510 psi	9380 psi	ASTM D 638-10
Tensile Modulus	1.6 GPa	2.8 GPa	234 ksi	402 ksi	ASTM D 638-10
Elongation at Failure	12 %	6.2 %	12 %	6.2 %	ASTM D 638-10
Flexural Properties					
Flexural Modulus	1.25 GPa	2.2 GPa	181 ksi	320 ksi	ASTM C 790-10
Impact Properties					
Notched IZOD	16 J/m	25 J/m	0.3 ft-lbf/in	0.46 ft-lbf/in	ASTM D 256-10
Temperature Properties					
Heat Deflection Temp. @ 264 psi	42.7 °C	58.4 °C	108.9 °F	137.1 °F	ASTM D 648-07
Heat Deflection Temp. @ 66 psi	49.7 °C	73.1 °C	121.5 °F	163.6 °F	ASTM D 648-07

[1] Material properties can vary with part geometry, print orientation, print settings, and temperature.

[2] Data was obtained from green parts, printed using Form 2, 100 µm, Clear settings, washed and air dried without post cure.

[3] Data was obtained from parts printed using Form 2, 100 µm, Clear settings, and post-cured with 1.25 mW/cm^2 of 405 nm LED light for 60 minutes at 60 °C.

Solvent Compatibility

Percent weight gain over 24 hours for a printed and post-cured 1 x 1 x 1 cm cube immersed in respective solvent:

Solvent	24 Hour Weight Gain (%)	Solvent	24 Hour Weight Gain (%)
Acetic Acid, 5 %	< 1	Hydrogen Peroxide (3 %)	< 1
Acetone	sample cracked	Isooctane	< 1
Isopropyl Alcohol	< 1	Mineral Oil, light	< 1
Bleach, ~5 % NaOCl	< 1	Mineral Oil, heavy	< 1
Butyl Acetate	< 1	Salt Water (3.5 % NaCl)	< 1
Diesel	< 1	Sodium hydroxide (0.025 %, pH = 10)	< 1
Diethyl glycol monomethyl ether	1.7	Water	< 1
Hydrolic Oil	< 1	Xylene	< 1
Skydrol 5	1	Strong Acid (HCl Conc)	distorted

HIGH RESOLUTION

For demanding applications, our carefully-engineered resins capture the finest features in your model.

STRENGTH AND PRECISION

Our resins create accurate and robust parts, ideal for our rapid prototyping and product development.

SURFACE FINISH

Perfectly smooth right out of the printer, parts printed on the Form 2 printer have the polish and finish of a final product.

CLEAR

Our Clear Resin polishes to near optical transparency, making it ideal for showcasing internal features.

WHITE

Our White Resin emphasizes fine details and has a matte finish with a warm, slightly ivory color.

GREY

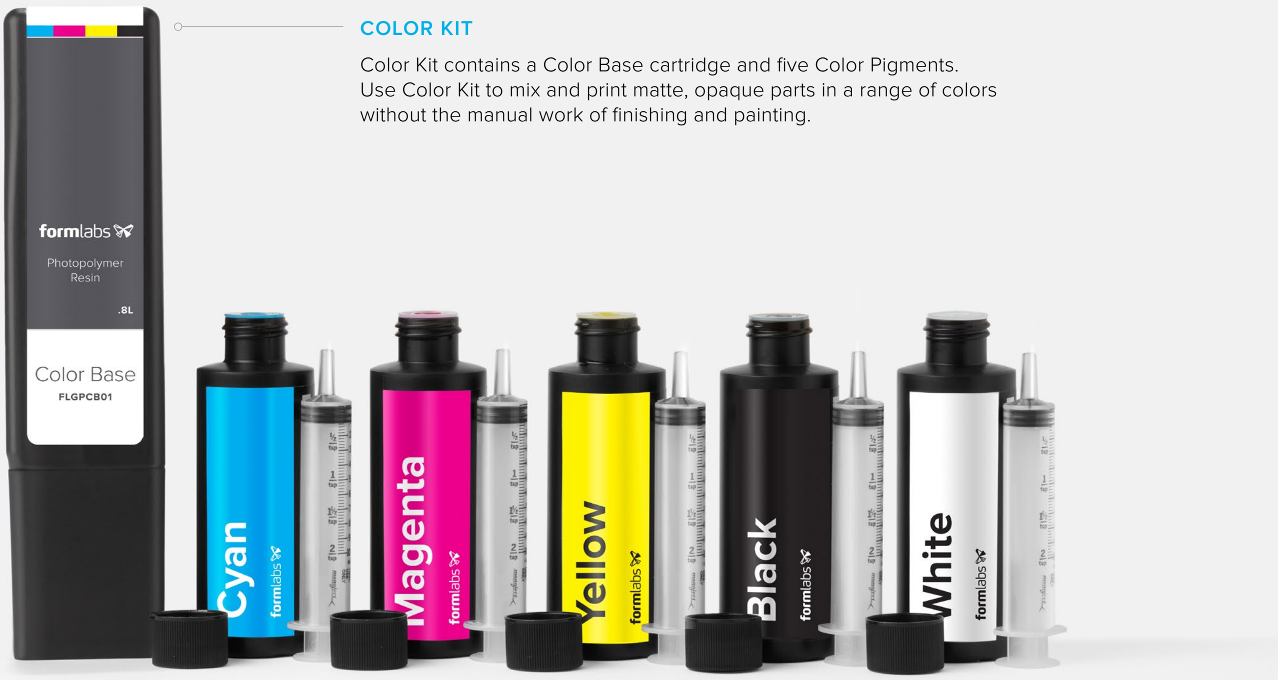
Our Grey Resin has a smooth, matte finish and shows details beautifully without primer.

BLACK

Our Black Resin's opaque matte finish rivals the look of injection-molded plastics, capable of producing incredible looks-like prototypes.

COLOR KIT

Color Kit contains a Color Base cartridge and five Color Pigments. Use Color Kit to mix and print matte, opaque parts in a range of colors without the manual work of finishing and painting.



Overview of materials for Polypropylene Copolymer

Categories: Polymer; Thermoplastic; Polypropylene (PP); Polypropylene Copolymer

Material Notes: This property data is a summary of similar materials in the MatWeb database for the category "Polypropylene Copolymer". Each property range of values reported is minimum and maximum values of appropriate MatWeb entries. The comments report the average value, and number of data points used to calculate the average. The values are not necessarily typical of any specific grade, especially less common values and those that can be most affected by additives or processing methods.

Vendors: **Click here to view all available suppliers for this material.**

Please click here if you are a supplier and would like information on how to add your listing to this material.

Physical Properties	Metric	English	Comments
Density	0.780 - 2.77 g/cc	0.0282 - 0.100 lb/in³	Average value: 0.949 g/cc Grade Count:984
Filler Content	0.000 - 75.0 %	0.000 - 75.0 %	Average value: 21.4 % Grade Count:63
Water Absorption	0.0100 - 0.200 %	0.0100 - 0.200 %	Average value: 0.0376 % Grade Count:51
Water Vapor Transmission	18.8 - 18.8 g/m²/day @Temperature 37.8 - 37.8 °C	1.21 - 1.21 g/100 in²/day @Temperature 100 - 100 °F	Average value: 18.8 g/m²/day Grade Count:1
Oxidative Induction Time (OIT)	40.0 min	40.0 min	Average value: 40.0 min Grade Count:3
	20.0 - 40.0 min @Temperature 190 - 220 °C	20.0 - 40.0 min @Temperature 374 - 428 °F	Average value: 42.5 min Grade Count:8
Linear Mold Shrinkage	0.000150 - 0.0350 cm/cm	0.000150 - 0.0350 in/in	Average value: 0.0123 cm/cm Grade Count:213
Linear Mold Shrinkage, Transverse	0.00400 - 0.0250 cm/cm	0.00400 - 0.0250 in/in	Average value: 0.0125 cm/cm Grade Count:70
Melt Flow	0.200 - 150 g/10 min	0.200 - 150 g/10 min	Average value: 18.3 g/10 min Grade Count:1169
Base Resin Melt Index	1.20 - 45.0 g/10 min @Temperature 230 - 230 °C	1.20 - 45.0 g/10 min @Temperature 446 - 446 °F	Average value: 16.4 g/10 min Grade Count:23
Ash	0.0350 - 40.0 %	0.0350 - 40.0 %	Average value: 17.0 % Grade Count:28
	0.300 - 0.500 % @Temperature 600 - 900 °C	0.300 - 0.500 % @Temperature 1110 - 1650 °F	Average value: 0.433 % Grade Count:3

Mechanical Properties	Metric	English	Comments
Hardness, Rockwell M	55.0	55.0	Average value: 55.0 Grade Count:3
Hardness, Rockwell R	33.0 - 111	33.0 - 111	Average value: 85.7 Grade Count:328
Hardness, Shore D	52.0 - 85.0	52.0 - 85.0	Average value: 66.6 Grade Count:132
Ball Indentation Hardness	35.0 - 82.0 MPa	5080 - 11900 psi	Average value: 60.0 MPa Grade Count:26
Tensile Strength, Ultimate	5.00 - 128 MPa	725 - 18500 psi	Average value: 29.1 MPa Grade Count:247
	9.00 - 15.0 MPa @Temperature 60.0 - 90.0 °C	1310 - 2180 psi @Temperature 140 - 194 °F	Average value: 12.0 MPa Grade Count:1
Film Tensile Strength at Yield, MD	18.0 - 27.0 MPa	2610 - 3920 psi	Average value: 20.4 MPa Grade Count:8
Tensile Strength, Yield	0.159 - 110 MPa	23.0 - 16000 psi	Average value: 25.8 MPa Grade Count:950

Film Elongation at Break, MD	80.0 - 800 %	80.0 - 800 %	Average value: 619 % Grade Count:26
Film Elongation at Break, TD	500 - 800 %	500 - 800 %	Average value: 620 % Grade Count:12
Elongation at Break	1.30 - 1000 %	1.30 - 1000 %	Average value: 183 % Grade Count:312
	100 - 100 % @Temperature -30.0 - -30.0 °C	100 - 100 % @Temperature -22.0 - -22.0 °F	Average value: 100 % Grade Count:1
Elongation at Yield	0.600 - 500 %	0.600 - 500 %	Average value: 17.2 % Grade Count:589
Modulus of Elasticity	0.100 - 6.20 GPa	14.5 - 899 ksi	Average value: 1.27 GPa Grade Count:239
Flexural Yield Strength	13.4 - 1240 MPa	1950 - 180000 psi	Average value: 95.0 MPa Grade Count:150
Flexural Modulus	0.134 - 9.32 GPa	19.4 - 1350 ksi	Average value: 1.40 GPa Grade Count:1024
	0.370 - 0.560 GPa @Temperature 60.0 - 90.0 °C	53.7 - 81.2 ksi @Temperature 140 - 194 °F	Average value: 0.465 GPa Grade Count:1
Compressive Yield Strength	27.6 - 110 MPa	4000 - 16000 psi	Average value: 42.5 MPa Grade Count:7
Shear Modulus	0.260 - 0.730 GPa	37.7 - 106 ksi	Average value: 0.541 GPa Grade Count:20
Secant Modulus	0.345 - 1.52 GPa	50.0 - 220 ksi	Average value: 0.913 GPa Grade Count:16
Secant Modulus, MD	0.370 - 0.840 GPa	53.7 - 122 ksi	Average value: 0.632 GPa Grade Count:5
Izod Impact, Notched	0.260 J/cm - NB	0.487 ft-lb/in - NB	Average value: 1.42 J/cm Grade Count:708
	0.160 - 99.99 J/cm @Temperature -40.0 - 4.00 °C	0.300 - 187.3 ft-lb/in @Temperature -40.0 - 39.2 °F	Average value: 0.793 J/cm Grade Count:93
	0.200 J/cm - NB @Temperature -30.0 - 0.000 °C	0.375 ft-lb/in - NB @Temperature -22.0 - 32.0 °F	Average value: 0.793 J/cm Grade Count:26
	0.200 J/cm - NB @Thickness 3.17 - 3.17 mm	0.375 ft-lb/in - NB @Thickness 0.125 - 0.125 in	Average value: 0.793 J/cm Grade Count:26
	0.32034 - 1.33475 J/cm @Diameter 0.125 - 3.17 mm	0.60013 - 2.50053 ft-lb/in @Diameter 0.00492 - 0.125 in	Average value: 0.793 J/cm Grade Count:9
	0.32034 - 1.33475 J/cm @Temperature -20.0 - 4.00 °C	0.60013 - 2.50053 ft-lb/in @Temperature -4.00 - 39.2 °F	Average value: 0.793 J/cm Grade Count:9
Izod Impact, Unnotched	0.294 J/cm - NB	0.551 ft-lb/in - NB	Average value: 3.85 J/cm Grade Count:54
	0.343233 - 0.441299 J/cm @Temperature -10.0 - -10.0 °C	0.643016 - 0.826734 ft-lb/in @Temperature 14.0 - 14.0 °F	Average value: 0.392 J/cm Grade Count:3
Izod Impact, Notched (ISO)	3.90 - 21000 kJ/m²	1.86 - 9990 ft-lb/in²	Average value: 24.5 kJ/m² Grade Count:135
	2.00 - 13.0 kJ/m² @Temperature -40.0 - 0.000 °C	0.952 - 6.19 ft-lb/in² @Temperature -40.0 - 32.0 °F	Average value: 5.10 kJ/m² Grade Count:70
Charpy Impact Unnotched	1.50 J/cm² - NB	7.14 ft-lb/in² - NB	Average value: 5.29 J/cm² Grade Count:106
	1.20 J/cm² - NB @Temperature -40.0 - 0.000 °C	5.71 ft-lb/in² - NB @Temperature -40.0 - 32.0 °F	Average value: 8.31 J/cm² Grade Count:44
Charpy Impact, Notched	0.250 - 1000 J/cm²	1.19 - 4760 ft-lb/in²	Average value: 2.17 J/cm² Grade Count:316
	0.100 - 12.0 J/cm² @Temperature -40.0 - 0.000 °C	0.476 - 57.1 ft-lb/in² @Temperature -40.0 - 32.0 °F	Average value: 0.539 J/cm² Grade Count:232
Gardner Impact	1.25 - 136 J	0.920 - 100 ft-lb	Average value: 23.9 J Grade Count:81
	1.00 - 36.0 J @Temperature -30.0 - -20.0 °C	0.738 - 26.6 ft-lb @Temperature -22.0 - -4.00 °F	Average value: 19.8 J Grade Count:41
	4.86 - 23.6 J @Temperature -23.0 - -20.0 °C	3.58 - 17.4 ft-lb @Temperature -9.40 - -4.00 °F	Average value: 19.8 J Grade Count:6
	4.86 - 23.6 J @Thickness 3.17 - 3.18 mm	3.58 - 17.4 ft-lb @Thickness 0.125 - 0.125 in	Average value: 19.8 J Grade Count:6
Dart Drop, Total Energy	5.20 - 50.0 J	3.84 - 36.9 ft-lb	Average value: 23.2 J Grade Count:25

	11.0 - 38.3697 J @Temperature -30.0 - -29.0 °C	8.11 - 28.3000 ft-lb @Temperature -22.0 - -20.2 °F	Average value: 25.5 J Grade Count:12
Falling Dart Impact	2.26 - 33.0 J	1.67 - 24.3 ft-lb	Average value: 9.75 J Grade Count:11
	28.2011 - 28.2011 J @Temperature -30.0 - -30.0 °C	20.8000 - 20.8000 ft-lb @Temperature -22.0 - -22.0 °F	Average value: 13.1 J Grade Count:1
	3.00 - 27.0 J @Temperature -30.0 - -30.0 °C	2.21 - 19.9 ft-lb @Temperature -22.0 - -22.0 °F	Average value: 13.1 J Grade Count:5
	3.00 - 27.0 J @Thickness 3.17 - 3.17 mm	2.21 - 19.9 ft-lb @Thickness 0.125 - 0.125 in	Average value: 13.1 J Grade Count:5
Puncture Energy	10.0 - 48.0 J @Temperature -40.0 - 0.000 °C	7.38 - 35.4 ft-lb @Temperature -40.0 - 32.0 °F	Average value: 23.6 J Grade Count:14
Coefficient of Friction	0.100 - 1.00	0.100 - 1.00	Average value: 0.341 Grade Count:26
Dart Drop Total Energy	6.73 - 7.79 J/cm	0.0126 - 0.0146 ft-lb/mil	Average value: 7.08 J/cm Grade Count:3
Film Tensile Strength at Break, MD	21.0 - 440 MPa	3050 - 63800 psi	Average value: 64.8 MPa Grade Count:34
Film Tensile Strength at Break, TD	25.0 - 340 MPa	3630 - 49300 psi	Average value: 58.2 MPa Grade Count:26
Tangent Modulus	740 - 1590 MPa	107000 - 230000 psi	Average value: 1150 MPa Grade Count:16

Electrical Properties	**Metric**	**English**	**Comments**
Electrical Resistivity	50.0 - 1.00e+17 ohm-cm	50.0 - 1.00e+17 ohm-cm	Average value: 1.03e+16 ohm-cm Grade Count:39
Surface Resistance	80.0 - 1.00e+16 ohm	80.0 - 1.00e+16 ohm	Average value: 7.81e+14 ohm Grade Count:48
Dielectric Constant	2.20 - 3.80	2.20 - 3.80	Average value: 2.52 Grade Count:12
Dielectric Strength	11.8 - 45.0 kV/mm	300 - 1140 kV/in	Average value: 29.1 kV/mm Grade Count:18
Comparative Tracking Index	450 - 650 V	450 - 650 V	Average value: 591 V Grade Count:35

Thermal Properties	**Metric**	**English**	**Comments**
CTE, linear	26.0 - 150 µm/m-°C	14.4 - 83.3 µin/in-°F	Average value: 88.6 µm/m-°C Grade Count:40
Thermal Conductivity	0.220 - 2.00 W/m-K	1.53 - 13.9 BTU-in/hr-ft²-°F	Average value: 0.443 W/m-K Grade Count:14
Melting Point	128 - 171 °C	262 - 340 °F	Average value: 155 °C Grade Count:167
Crystallization Temperature	35.0 - 130 °C	95.0 - 266 °F	Average value: 101 °C Grade Count:14
Maximum Service Temperature, Air	65.0 - 145 °C	149 - 293 °F	Average value: 91.5 °C Grade Count:21
Deflection Temperature at 0.46 MPa (66 psi)	35.0 - 185 °C	95.0 - 365 °F	Average value: 94.3 °C Grade Count:757
Deflection Temperature at 1.8 MPa (264 psi)	37.0 - 165 °C	98.6 - 329 °F	Average value: 67.2 °C Grade Count:322
Vicat Softening Point	49.0 - 189 °C	120 - 372 °F	Average value: 123 °C Grade Count:266

Heat Distortion Temperature	75.0 - 83.0 °C	167 - 181 °F	Average value: 79.3 °C Grade Count:6
Brittleness Temperature	-50.0 - -30.0 °C	-58.0 - -22.0 °F	Average value: -43.5 °C Grade Count:4
UL RTI, Electrical	105 - 115 °C	221 - 239 °F	Average value: 108 °C Grade Count:3
Flammability, UL94	HB - V-0	HB - V-0	Grade Count:131
Flash Point	340 °C	644 °F	Average value: 340 °C Grade Count:10
Glow Wire Test	650 - 960 °C	1200 - 1760 °F	Average value: 805 °C Grade Count:49
Ignition Temperature	350 °C	662 °F	Average value: 350 °C Grade Count:6
DSC Induction Temperature	135 - 164 °C	275 - 327 °F	Average value: 149 °C Grade Count:7

Optical Properties	**Metric**	**English**	**Comments**
Haze	0.500 - 40.0 %	0.500 - 40.0 %	Average value: 10.1 % Grade Count:93
Gloss	3.00 - 150 %	3.00 - 150 %	Average value: 101 % Grade Count:44
Transmission, Visible	80.0 - 90.0 %	80.0 - 90.0 %	Average value: 88.8 % Grade Count:26

Processing Properties	**Metric**	**English**	**Comments**
Processing Temperature	87.8 - 280 °C	190 - 536 °F	Average value: 205 °C Grade Count:30
Nozzle Temperature	193 - 260 °C	380 - 500 °F	Average value: 213 °C Grade Count:55
Die Temperature	175 - 260 °C	347 - 500 °F	Average value: 213 °C Grade Count:25
Melt Temperature	145 - 302 °C	293 - 575 °F	Average value: 215 °C Grade Count:201
Head Temperature	200 - 230 °C	392 - 446 °F	Average value: 215 °C Grade Count:17
Mold Temperature	4.44 - 90.6 °C	40.0 - 195 °F	Average value: 42.4 °C Grade Count:175
Drying Temperature	65.6 - 90.0 °C	150 - 194 °F	Average value: 75.8 °C Grade Count:84
Moisture Content	0.0500 - 0.200 %	0.0500 - 0.200 %	Average value: 0.0645 % Grade Count:38
Injection Pressure	0.500 - 120 MPa	72.5 - 17400 psi	Average value: 48.2 MPa Grade Count:86

Component Elements Properties	**Metric**	**English**	**Comments**
SiO_2	0.180 %	0.180 %	Average value: 0.180 % Grade Count:5