

Use of Additive Manufacturing (AM) for Mold Inserts in Injection Molding

Dr. Jorge Rodriguez P.E., Western Michigan University

Faculty member in the Department of Engineering Design, Manufacturing, and Management Systems (EDMMS) at Western Michigan University's (WMU). Co-Director of the Center for Integrated Design (CID), and currently the college representative to the President's University-wide Sustainability Committee at WMU. Received his Ph.D. in Mechanical Engineering-Design from University of Wisconsin-Madison and received an MBA from Rutgers University. His B.S. degree was in Mechanical and Electrical Engineering at Monterrey Tech (ITESM-Monterrey Campus). Teaches courses in CAD/CAE, Mechanical Design, Finite Element Method and Optimization. His interest are in the area of product development, topology optimization, additive manufacturing, sustainable design, and biomechanics.

Use of Additive Manufacturing (AM) for Mold Inserts in Injection Molding

Abstract

Currently, whenever there is a need for an injection molding prototype, it can take weeks and thousands of dollars to develop and manufacture it. Therefore, the use of Additive Manufacturing (AM) to produce inserts for injection molding for rapid quote of parts can drastically reduce those costs and timeframes. In this project, functional prototype inserts for injection molding were developed and analyzed with CAD/CAE software. These molds were 3D printed and tested using a commercial plastic injection molding machine. Calculations for compression, shrinkage, and cooling of the inserts were used to establish initial information for development of the molding conditions. There were measurements taken on the inserts and the molded parts to validate calculations and specified dimensions. This development procedure will serve as guideline for future parts. The project was taken as a senior project, and it is expected that the results will allow a plastic injection molding company to rapidly and efficiently produce a short run of functional parts for prospective clients. This is another example of students participating in an industry-sponsored capstone course to complement their academic education.

Introduction

Prototype plastic injection molds are used to produce short runs of parts to be utilized in quoting customer needs. Prototype injection molds can take weeks and tens of thousands of dollars to produce. The use of Additive Manufacturing (AM) to aid in the process of developing prototype plastic injection mold inserts can reduce the lead time and overall cost of producing functional parts for quoting prospective clients^{7,8,9}. Injection molding is the process of injecting molten plastic material into a mold cavity with the proper injection parameters to produce a functional part¹⁴. Plastic injection molds are primarily manufactured using tool steels which can withstand the high temperature environment. It is important to understand the methods and concepts of injection molding when developing an additively manufactured plastic mold that will be used as an insert for plastic injection molding.

Several variables need to be taken into consideration during injection molding, e.g., compression pressure, hold time, and cooling. Compression affects injection molded parts in terms of quality, due to part deformation caused by volumetric shrinkage. Volumetric shrinkage is the reduction of molecule size in a particular plastic when changing between molten and solid states. Compression also affects the proper sealing of the mold halves, without a proper seal flashing is likely to occur. This aspect was considered in our design by using an injection mold that is placed into a steel mold. The compression ratios then needed to be studied and adjusted in the design of the molds.

Additive manufacturing is the process of additively producing a 3D part through a layered process; it has also been previously referred to as rapid prototyping, rapid tooling, rapid technology, and layered manufacturing¹. Recent developments have improved tolerances, thinner layers, and wider material selection. Although the 3D printing technology, particularly

when referring to FDM - Fused Deposition Modeling - is well understood, there is always room for improvement⁵. But numerous technological developments nowadays are in the area of materials used in 3D printing¹³. The overall goal in these developments is to move from the rapid prototype to a functional part. A 3D printing company that has been a pioneer in developing printing technology and materials, particularly for high temperature applications, is Objet (now part of Stratasys Corp). Objet¹⁰ applies polyjet technology that is similar to an inkjet printer using a print head that instead of printing water-based ink, lays down thin layers of photosensitive material. The Objet machine has the ability to print in layers as thin as 6.3 mil (0.0063 in; 0.16 mm); this is thinner than a human hair. By being able to apply material at such thin layers, a higher resolution mold was able to be made. Access to an Object machine was available for this project^{3, 12}.

The objective of this project was to perform design and testing for additive manufactured mold inserts. There are reports that have dealt with this issue of producing injection molded parts with 3DP inserts²⁻⁶, with many of the projects being a trial-and-error approach producing a single part. The potential for a large benefit is there but there needs to be dedicated efforts in this topic. The designs will be determined based on part design, mold material, cooling channels, injection material, and mold life. Initial design guidelines for some geometry features will be studied. The goal of the project is to determine the feasibility of utilizing additive manufacturing for short runs, as well as quoting of functional injection molded parts. This project was undertaken as an industry-sponsored senior project by students in the Design and the Manufacturing Engineering Technology programs in the College (Department of Design, Manufacturing, and Management Systems – EDMMS – at Western Michigan University).

Methodology

The main steps taken to complete this feasibility study were:

- Study injection molding process and material properties
- Create initial/modified designs
- Determine injection parameters
- Perform Moldflow and Finite Element Analysis
- Define cooling channel
- Do testing of printed inserts.

There were several calculations and decisions made during the process. Given the fact that this was consider a feasibility study, some of the calculations were carried out as conceptual ones, with the goal of defining basic information for the particular parameters. Some of the specific developments were:

Material Properties. For the temperature-resistant materials available from the manufacturer for 3D printing¹¹, various material properties were obtained to help understand how the materials were expected to react in the injection molding process. Some properties were provided by the printing company, but for some other properties the information was not available, and further research and testing provided good approximation of the particular values. The material being used is VeroGray, with a Modulus of Elasticity of 0.43 E6 psi (3,000 MPa) (compared to aluminum, which is 10 E6 psi - 70,000 MPa); the heat deflection temperature of VeroGray is at

120°F (49°C) (compared to aluminum which is 1,200°F - 649°C). Poisson's Ratio was determined by looking at similar materials (i.e., Fullcure 720), and by performing compression tests of the selected material.

Mold Design Progression. In designing the mold it was decided to do a progression of designs. With the progression of design, it started with basic features and progressively work the designs into the final part. The reason this was done in such fashion was to help understanding the effects of the injection molding process on a simple geometry and then build upon it. This would also help in getting a better understanding of the molding process and deciding injection parameters. Three primary designs were developed for experimentation in this project; the designs went through a progression from basic to more complex (Figure 1).

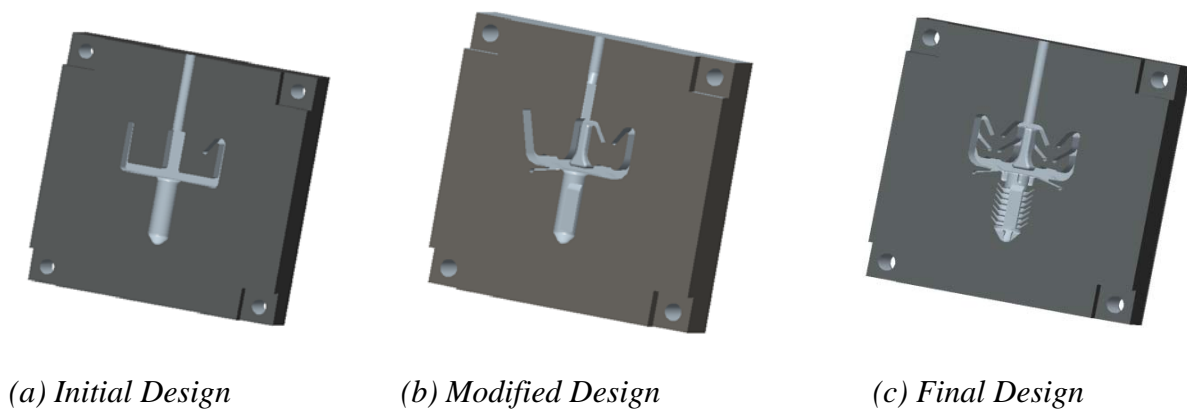


Figure 1. Photographs showing the progression in mold design

The initial design was made up of simplified features from the final part desired. The initial design (Figure 1a) has two fingers that protrude upwards with a curved section to represent one of the 8 curved fingers on the final design. Rectangular and cylindrical features were tied into this design as well to give a simplified representation of the center recessed chamber and the pine-tree retainer feature located at the bottom of the part. The first modified design (Figure 1b) incorporated the rounded outside corners, a recess located where the primary center heat mass, as well as an angled side wall that was used to determine if there were any deflections. This design also added variable wall thickness as well as the tension finger that is located just below the main elbows where the curved fingers can be found. By removing material from the part design it reduced the overall volume of the mold cavity which in turn reduces that amount of heat transfer. The final mold design (Figure 1c) was the desired test part. This design included complex features such as the tension fingers, variable wall thickness, and pine-tree retainer. On all three mold designs a 1.5° draft angle was used to help with manual ejection of the part from the molds.

Moldflow Analysis. This analysis was performed to define the initial injection parameters for testing the designs. The initial parameters that were observed within the Moldflow results were clamping force, temperatures upon injection, volumetric shrinkage, VP switchover, expected cooling times, and melt temperatures. The first analysis with Moldflow was also done to determine what should be expected on the part once injecting the designated material into the additively manufactured injection mold inserts. The final analysis with Moldflow was performed

using the final injection parameters that were defined through the actual injection molding tests. The final analysis results verified the experimental set-up results that were being observed, the temperature at mold open was upwards of 447°F (231°C) which was located in the exact locations where mold degradation occurred.

Finite Element Analysis (FEA). An FEA of the proposed design for mold inserts was performed to determine the maximum deflection in the part cavity. The maximum deflection would help to determine the amount of material needed to add to the overall height of the mold in order for the molds to completely seal when they were undergoing injection molding. The clamping force is calculated with the part and runner area, the cavity pressure, the separating force, and the specified safety factor. The parameters and equations were as follows:

- Projected part and runner area = 5.73 in² (36.96 cm²)
- Cavity pressure = 2 900 psi (20 MPa) - from Moldflow Analysis
- Separating Force = 8 tons
- Factor of safety = 1.25
- $(PA) * (CP) = SF$ $(SF) * (FS) = 10$ tons

The FEA ran with 10 tons of clamping force on the front face of the mold, and 2,900 psi (20 MPa) of injection pressure on every face of the part cavity. The maximum deflection around the part cavity is illustrated in Figure 2, and it was 6 mil (0.15mm). This meant an additional 6 mil (0.15mm) to the overall height of our mold design.

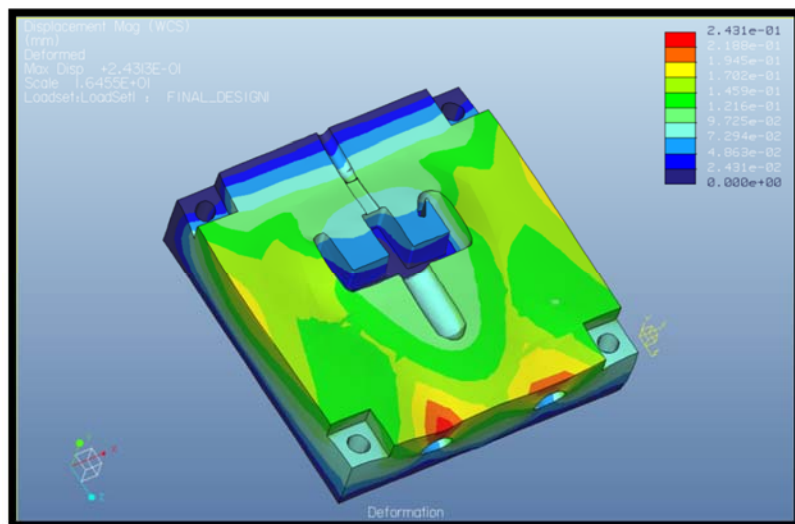


Figure 2. FEA Deflection Results.

After defining the final insert design an FEA was run again to verify that the maximum deflection around the part cavity did not exceed 6 mil (0.15mm) at the actual injection molding parameters used during testing⁴. The results confirmed that for the final design the maximum deflection did not exceed 6 mil (0.15mm).

Cooling Channel Design. The first step in this development was to perform heat transfer calculation based on the thermal properties on VeroGray (3DP material used for the mold

inserts) and nylon and polypropylene (injected materials) to estimate the amount of heat energy being introduced to the molds. Using the properties of cooling process water, and a static situation to simulate a worst case scenario, the volume of the cooling channels needed to dissipate the heat was calculated. The intention was to define the channels' cross-sectional area. Unfortunately, these simplified calculations did not provide feasible dimensions. Polypropylene offered better results, but still unfeasible for our mold insert dimensions. A second analysis was performed taking into account the cooling pump characteristics (commercial machine) and dynamic factors during injection molding), producing a feasible design for polypropylene, it was decided to specify a diameter of 0.25 in (6.35 mm) for the cooling channels.

Taking advantage of the capabilities of a 3DP process, this first design for cooling channel in the mold inserts was a "conformal" design, where the channel follows the profile of the cavity in a 2D path. A second design was used, where the channels follow a 3D path actually going under the center of mass of the part as well as having a rectangular cross section. The 3D path was specified so that the cooling is in the neighborhood of where higher thermal mass is expected, and therefore higher mold degradation was expected. In both designs analysis was performed so that structural integrity was ensured. It is interesting to note as well that one of the issues from the 3DP was that special access points were required in order to clean out the support material used for the channels during the printing process.

Results

Testing of the three different designs of mold inserts was performed using a Millacron 100-ton injection molding machine. The testing protocol was to perform the injection molding process on each set of inserts until there was a failure. Failure was defined as either the mold inserts was considered unusable, or the injected part was not considered a quality part. During the testing, and process parameters were recorded, injected parts and mold inserts were visually and dimensionally inspected. The results were as follows:

Test Set 1. This set is for the initial design of mold inserts (Figure 1a). Part material was both polypropylene and nylon. The mold inserts tested with polypropylene had cooling channels, and the molds tested with nylon were tested with cooling channels and without cooling channels. The table below (Table 1) shows that the inserts were able to take several shots, the cooling channels did have an effect, but few quality parts are obtained.

Table 1. Summary of Quality Shots for Test Set 1.

	Polypropylene	Nylon	Nylon
Cooling	With	With	Without
# of Mold Halves	5	25	10
# of Shots	16	34	14
Quality Parts	1	2	4

Test Set 2. This set is for the first modified design (Figure 1b). Polypropylene and nylon were used. In this test set polypropylene was tested without cooling channels, and nylon was tested with and without cooling channels. The table below shows a smaller number of acceptable parts, with inserts without channels performing better than with channels, for nylon.

Table 2. Summary of Quality Shots for Test Set 2.

	Polypropylene	Nylon	Nylon
Cooling	Without	With	Without
# of Mold Halves	2	2	16
# of Shots	6	4	23
Quality Parts	2	0	3

Test Set 3. This set is for the final design of mold inserts (Figure 1c). Polypropylene and nylon were used. In this test set there were no cooling channels. The table below shows that even a smaller number of shots were possible, with polypropylene giving the only quality part.

Table 3. Summary of Quality Shots for Test Set 3.

	Polypropylene	Nylon
Cooling	Without	Without
# of Mold Halves	2	6
# of Shots	10	3
Quality Parts	1	0

Discussion

Throughout testing there were two materials that were tested during injection molding. The main material tested was nylon, with a total of 24 mold sets - 11 with cooling channels and 13 without cooling channels. The average number of injection shots obtained before failure was 2.42 with cooling, and 2.53 without cooling. This result of having more shots being performed with molds that did not have cooling was due to the cooling lines having to be so close to the mold cavity that it lowered the integrity of the mold.

In testing with polypropylene a total of 6 molds were tested; 2 with cooling channels and 4 without cooling channels. The average number of injection shots obtained before failure was 4 with cooling and 5 without cooling. The reason for fewer mold sets being tested is the average mold life with polypropylene is extended due to its lower temperature for material melting point. The result of having more shots with molds that did not have cooling was due to the cooling lines being too close to the mold cavity again which weakens the integrity of the material.

The two materials that were used exhibited different results from one another during injection. With nylon the cavity fills easily and transfer more detail from the mold to the part. Due to the higher injection temperature of nylon, the mold would degrade after only one to two shots, with mold material melting and adhering to part. Nylon also required longer cycle times to allow the mold to cool down in order to extend mold life. Polypropylene was capable of well-defined outside features, but it was difficult to get the flow into more detailed areas. This issue was due to the higher injection viscosity of polypropylene which made setting of the injection parameters more difficult. Lower cycle times with polypropylene due to the lower injection temperature produce more parts per mold, with the mold insert completely degrading after four or five shots.

Conclusions and Recommendations

Upon completion of the feasibility study on the use of AM techniques to manufacture injection mold inserts it was determined that it is feasible to produce functional parts. There was evidence that AM mold inserts have the potential to supply short production runs and produce functional prototypes. Throughout the testing it was possible to determine that the mold material VeroGrey was able to withstand the clamping and injection pressures required for injection molding, and there is a new material VeroGreen that has higher heat deflection temperature, expected to improve thermal life.

Through the testing it was concluded that cooling had no significant impact on mold insert life or number of quality parts produced. Due to the low thermal conductivity of the mold material, for cooling to be effective at removing heat from the mold the channel needed to be too close to the part surface, causing a thin wall between the mold cavity and the cooling channel. This thin wall would then crack under the injection pressures, causing the cooling channels to become filled with injection material, thus destroying the mold.

The following recommendations are made: a) to have exact material properties, this is of vital importance for a project of this nature. In order to have valid prediction of performance there is need to have reliable properties; b) redesign inserts to accommodate cooling channels, the structural integrity after thermal fatigue is believed to be the main cause of insert failure. Additional work is required so that cooling channels should help when properly implemented.

This was a student project that provided an excellent opportunity for the design and the manufacturing students to expand their comfort zone, mainly into the use of commercial equipment. It was a very challenging project, in part due to the sponsorship by industry, and in part due to the novelty of the project. Fortunately, at the end the students were able to finish the project and they were satisfied with their efforts and results.

Acknowledgements

This project was made possible thanks to the support from Jason Reznar - from RayCE Americas, part of ARaymond group, and their guidance on injection molding, and their financial support for fabrication of 3DP molds and required instrumentation. The experiments were

conducted in the Plastics and the Materials labs at Western Michigan University, with the participation of Charles Crouch, Brian Guenther, Leah VanEeuwen, and Joe DeMenter.

Bibliography

1. *3D Printing: The Next Industrial Revolution*. (2013), Christopher Barnatt. CreateSpace Independent Publishing Platform.
2. 3DP Injection Molding Process Spread by Stratasys and Worrell Collaboration. <http://3dprintingindustry.com/2014/11/05/injection-molding-stratasys-worrell/>
3. Center for Integrated Design (CID), (2012-2015). Western Michigan University.
4. Dickens, P. (2000). Failure Mechanism Analysis of Stereolithographic Injection Mould Tooling. *Rapid Prototyping Journal*, pp. 62-74.
5. DLG. (2012). *Digital Lab Graph*. Retrieved Feb. 12, 2012, from <http://factoryoffactories.com/rapidtool.htm>
6. <http://www.stratasys.com/resources/case-studies/education/budapest-university-of-technology-and-economics>. Budapest University of Technology and Economics, 3D printing helps university create functional injection molded prototypes for testing. Dr. József Gábor Kovács.
7. Jacobs, P. F. (1996). Recent Advances in Rapid Tooling for Stereolithography. *Proceedings of the National Conference on Rapid Prototyping and Tooling Research*. Buckinghamshire: Buckinghamshire College. pp. 18-19.
8. Jacobs, P. F., and Hilton, P. D. (2000). Rapid Tooling: Technologies and Industrial Applications. In P. F. Peter D. Hilton. New York: Marcel Dekker.
9. Nagahanumaiah, Subburaj, and Ravi. (2008). Computer Aided Rapid Tooling Process Selection and Manufacturability Evaluation for Injection Mold Development. *Computers in Industry*, 59 (2-3), 262-276.
10. Objet 3D Printers. (2012). *New Objet Materials: The Power Behind your 3D Printer*. Retrieved 2012, from http://www.objet.com/Portals/0/docs2/New%20materials%20data%20sheets_low%20res.pdf
11. *PolyJet Materials Data Sheet*. (n.d.). Retrieved 7 8, 2013, from <http://www.stratasys.com/materials/polyjet/~media/29592222B80C489BAC28803DB08C10E5.ashx>
12. RayCE Americas. (2012-2015). Personal discussions.
13. Verma, S., Tyagi, K., and Yang, K., (2015), "Modeling and Optimization of Direct Metal Laser Sintering Process", *The International Journal of Advanced Manufacturing Technology*, vol. 77 (5-8), pp. 847-860.
14. Zheng, T. a. (2011). *Injection Molding: Integration of Theory and Modeling Methods*. Longdon, New York: Springer Heidelberg Dordrecht.