



FORMLABS WHITE PAPER:

Injection Molding from 3D Printed Molds

A study of low-volume production of small plastic parts

Table of Contents

Introduction	3
Low-Volume Production Parts From 3D Printed Molds	4
Method	5
Results.	7
Further Testing with High Temp.	8
Addressing Common Problems.	10
Design Guidelines	11
Process Summary	11
Formlabs and Galomb Contact Information	12



Fig 1: Selection of LDPE parts produced by Galomb, Inc.

ABSTRACT

This white paper describes the production of small injection molded thermoplastic parts that were created with 3D-printed molds produced on a Form 2 printer and injected using a Galomb Model-B100 Injection Molder. Two mold designs were tested in Clear Resin: one of a large butterfly and one that produces four smaller butterflies in one shot. A third mold of a USB device enclosure was tested in High Temp resin. These molds were 3D printed by Formlabs and the injection molded parts were produced by Galomb Inc and Formlabs in a variety of materials.

Formlabs and Galomb, Inc.

LOW-VOLUME PRODUCTION PARTS FROM 3D PRINTED MOLDS

The majority of plastic products in the world today are manufactured by injection molding. With affordable desktop 3D printers and injection molding machines, it is possible to create molds in-house to produce small, functional parts in production plastics.

For low-volume production (approximately 10-100 parts), 3D-printed molds save time and money. They also enable a more agile manufacturing approach, allowing engineers and designers to easily modify molds and continue to iterate on their designs with low lead times and cost.

The Form 2 stereolithography (SLA) 3D printer produces completely solid, smooth parts that can withstand the temperature and pressure of desktop injection molding. 3D prints produced by SLA are chemically bonded such that they are fully dense and isotropic, producing functional molds at a quality not possible with FDM.

Formlabs partnered with Galomb Inc., a manufacturer of affordable injection molding machines, to test and demonstrate the viability of SLA 3D printed injection molds.

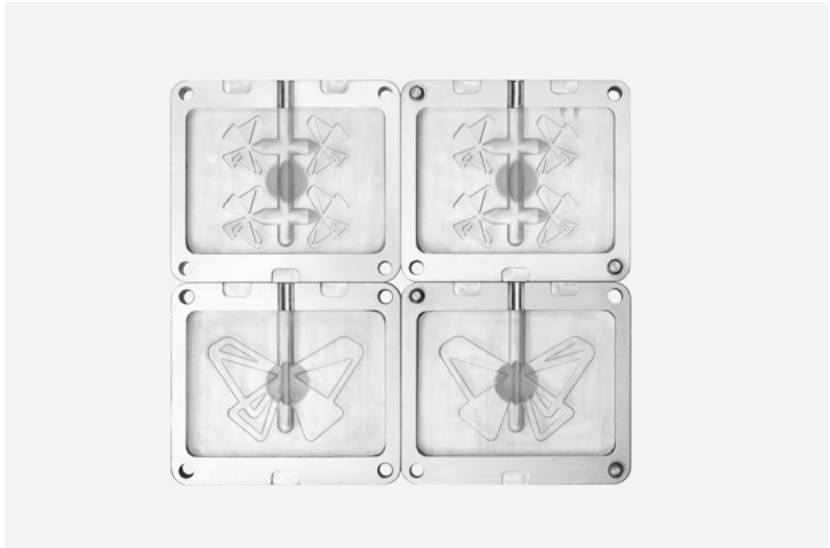


Fig 2: 3D printed molds in aluminium frames.

METHOD

Both Clear and High Temp Resins can be used to print small functional molds, with High Temp offering compatibility with a wider range of thermoplastic melt temperatures. Formlabs' Clear Resin was selected because of its strength, high detail, and smooth surface finish. Clear Resin is preferred due to its translucency as it is easy to see when the molds have filled, but any of Formlabs' Standard Resins (Clear, White, Black, and Grey) should work, as they have similar mechanical properties. The molds were printed with a layer height of 100 microns and took approximately 5 hours per mold. Depending on geometry, multiple molds can be printed at once on a build platform to increase printing efficiency.

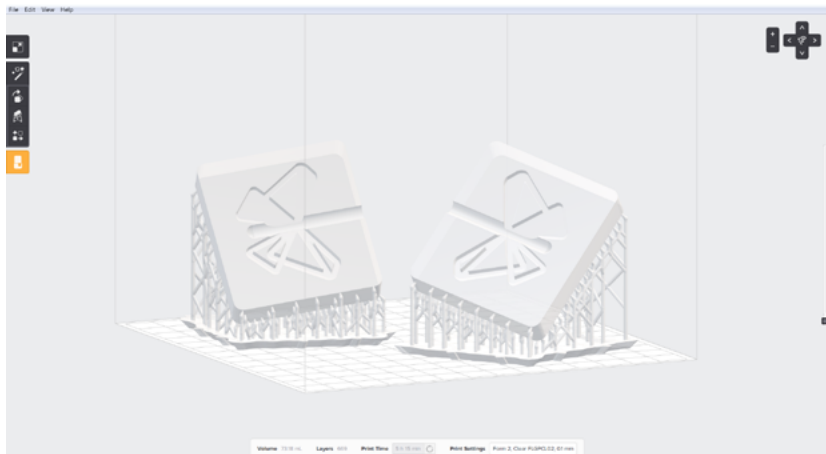


Fig 3. Print setup in PreForm, with cavities facing upwards.

Two mold designs were printed in Clear Resin. The parts and subsequent molds were designed to fit to the dimensions of the Galomb machine's vise clamp, the 1 in³ injection capacity of the barrel, and the build volume of the Form 2. After printing, the parts were rinsed in a bath of 90% isopropyl alcohol for 20 minutes each, supports were removed, and support marks were sanded.

The parts were then post-cured for one hour under a 405 nm UV bulb in order to reach full mechanical strength and stiffness. To better understand the effect of post-curing parts, see Formlabs' [white paper on UV post-curing](#).

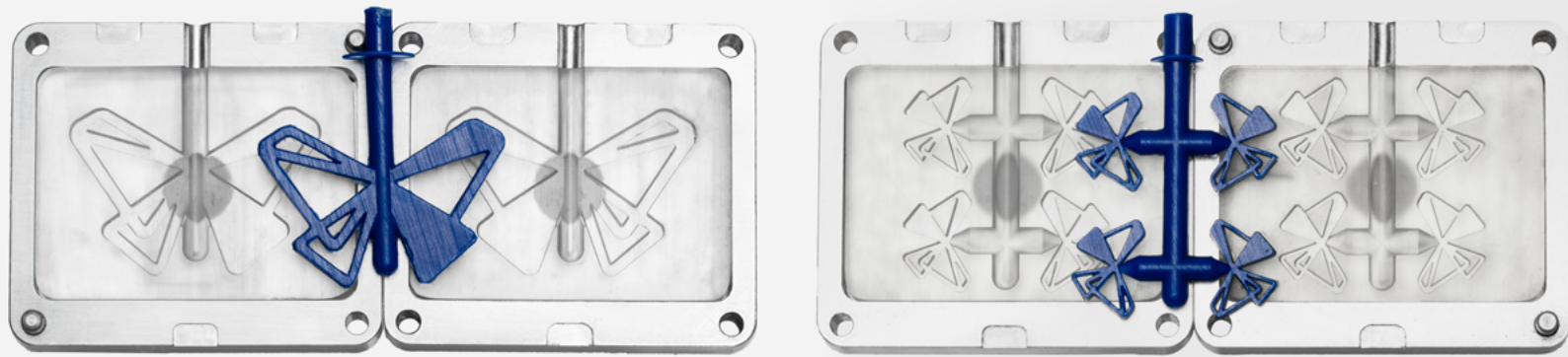


Fig 4. 3D printed molds in aluminum frames and injection molded parts.

The first mold was a large Formlabs butterfly logo and the second was four small Formlabs butterfly logos. Both molds had a cavity, a narrow gate, and a sprue to the injection point, and were designed in Solidworks. The molds were inserted into aluminium frames to provide support against the downward pressure and heat of the injection nozzle. Aluminum frames may also prevent the mold from warping after repeated usage. The frames pictured in Figures 2 and 4 were custom machined by Whittaker Engineering in Scotland, but standard aluminium frames are readily available from injection molder manufacturers.

Plastic pellets are available from online retailers or school supply companies, such as [IASCO-TESCO](#). To create a variety of colors, the molten plastic was pre-mixed with powdered colorants before injection.

Using the benchtop Model-B100 Injection Molder, Galomb tested the printed molds with 25 shots of LDPE. LDPE melts at approximately 325 °F (163 °C) and was chosen for its low melt temperature. It should be noted that Formlabs Clear Resin has a heat deflection temperature (HDT) @ 0.45MPa of 73.1 °C after post-cure (see the material [data sheet](#)). HDT is an indication of the material's thermal properties, but does not rule it out for this application, even though LDPE has a higher melt temperature. Whether or not your 3D-printed mold will withstand the injection molding process depends on the melt temperature of the injection material, part geometry, and the cooling and cycle time used.

CLEAR RESIN RESULTS

After 25 shots of LDPE, there was no noticeable surface deterioration (chips, cracks, or scratches) of the molds. LDPE did not tend to adhere to the resin molds in testing, but other plastics may require an application of mold release agent to help with the extraction of the part. Adhesion of the part to the mold can cause deterioration of the mold during extraction. Mold release is widely available, and silicone mold release is compatible with Formlabs Standard and High Temp Resins.

Cycle time for each shot was approximately three minutes. This process was accelerated by applying compressed air to cool the mold. Cyclic injection in molds printed on the Form 2 causes the mold to heat up. To counteract this, cooling time with the mold open should be increased between cycles. While the mold may not deform, too much residual heat will reduce molding success if the mold is opened too early. Galomb also improved molding success by etching in shallow (0.05 mm deep) air vents (not pictured) leading from the edge of the cavity to the edge of the mold so that air did not get trapped inside the cavity during injection.

Some of the shots exhibited flash at the split line, due to warpage of the resin mold during the cooling phase after multiple shots. Increasing clamping force in the vise can help mitigate flash, as can polishing the mold's split plane to give it as flat a surface as possible. Galomb proposed including channels in the mold design to embed metal tubes and filling these with aluminium-filled epoxy as a strategy to reinforce the mold, reduce warpage, and improve cooling time.

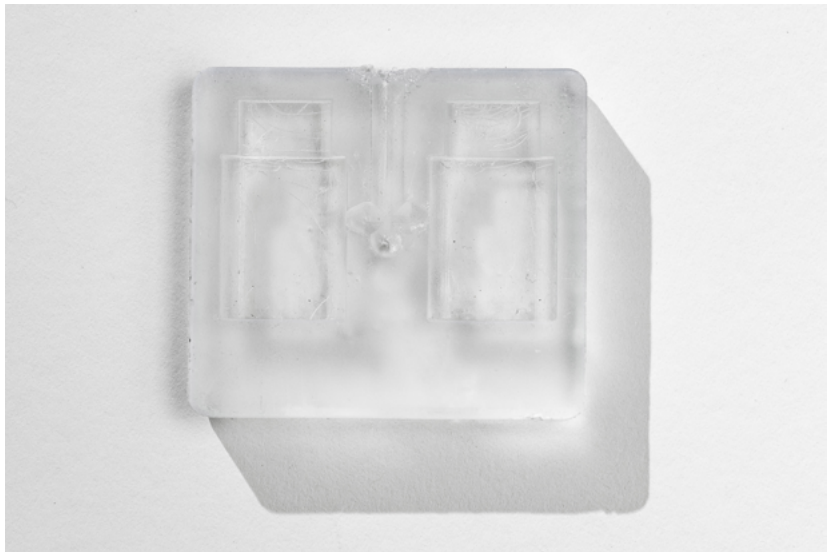
Fig 5. An assortment of injection molded parts made with 3D printed molds.





FURTHER TESTING WITH HIGH TEMP RESIN

Clear Resin molds were tested with success using LDPE, which has a relatively low melt temperature. Higher melt temperature plastics can cause thermal shock in Clear Resin prints, which appears as a fractured mold surface.



Clear resin molds experience thermal shock when exposed to higher temperature molten plastic.

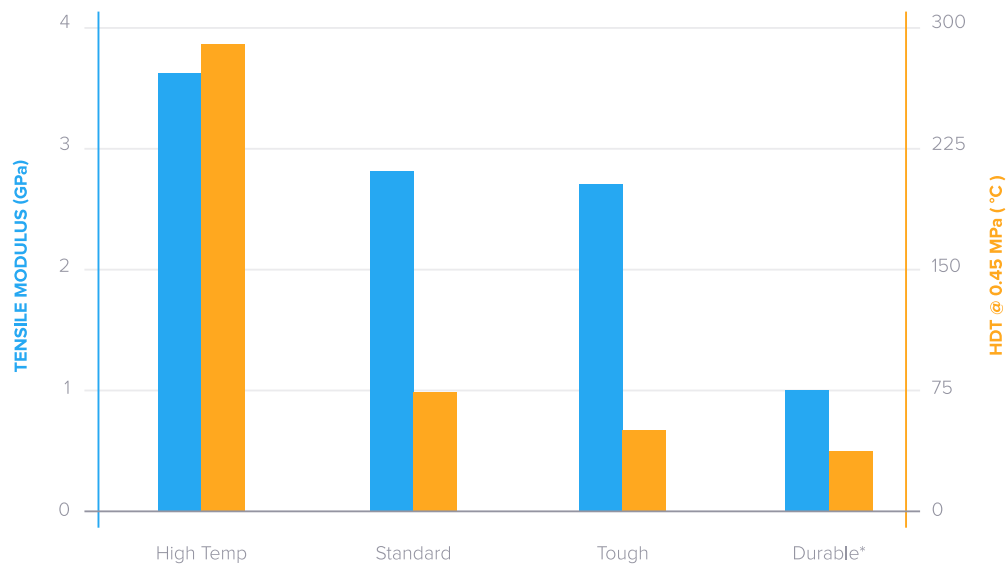
Formlabs printed a mold of a USB device enclosure in High Temp Resin to test benchtop injection molding with a wider range of plastics.

SLA Printed Mold Plastic Compatibility

Plastic	Melt Temperature <small>(Source: Plastic Troubleshooter)</small>	High Temp (HDT @ 0.45 MPa = 289° C)	Clear (HDT @ 0.45 MPa = 73.1° C)
LDPE	163 °C	✓	✓
PP	177 °C	✓	✗
TPE	177 °C	✓	✗
PLA	180 °C	✓	✗
ABS	204 °C	✓	✗
HDPE	204 °C	✓	✗
EVA	204 °C	✓	✗
PS	226 °C	✓	✗

The High Temp Resin molds showed no temperature degradation on the mold surface for any of the plastic tested.

HDT @ 0.45MPa and Tensile Modulus



**Durable Resin is under development, final properties subject to change.*

High Temp and Standard are the resins best-suited for moldmaking. Of Formlabs Resins, High Temp has the highest HDT @ 0.45MPa and low thermal expansion. It is also the stiffest material, with high tensile modulus.

The relatively high stiffness of High Temp Resin means that the mold will not deform when removing the part. This makes the use of mold release especially important to remove parts shot in rigid plastics such as polystyrene.

ADDRESSING COMMON PROBLEMS



Flash caused by overfilling a mold.

Flash occurs when the injected plastic is forced out between the two halves of the mold. This can happen when a mold is overfilled, or if the parting plane is not perfectly flat. Adding thin exit runners to a mold can help mitigate flash from overpressure inside the mold, help with part removal, and fix air traps that would cause bubbles in the molded part. While not pictured, printed molds were tested without an aluminium frame. The disadvantage to this approach is that these parts use more material, which increases print cost and time, and molds may be more prone to warping. With this method a steel washer placed between the printed mold and the nozzle of the injection molding machine protects the print from direct contact and helps to distribute forces. Additionally, pre-packing the injection molding machine's barrel by compressing it against a metal block helps ensure that there are no air pockets to disrupt the flow of plastic.

Print lines are visible on some of the parts; this could be reduced by printing the mold with a smaller layer height. The molds used in this study were printed with a layer height of 100 microns, but 50 or 25 microns could also be used. This will improve the surface finish of the mold, but increase print time and decrease tank lifespan.



Final USB enclosure created with High Temp mold.

DESIGN GUIDELINES

When designing your mold, consider what will print successfully, as well as what will mold successfully.

- Adding one to three degrees of draft on surfaces perpendicular to the direction of pull will allow the part to be removed more easily and minimize degradation of the mold. Fillets should be applied to interior edges to reduce warpage from internal plastic stress and aid part removal.
- Embossed and engraved details should be offset from the surface by at least 1 mm.
- Split plane surfaces can be polished with fine-grit sandpaper to reduce flash.
- If designing for an aluminium frame, add .125 mm of extra thickness to the back of the mold plates to account for compression forces and to ensure a complete seal.
- Make sure to orient the mold halves in PreForm so that the cavity faces up. This will prevent having support marks inside the cavity and make post-print processing easier.



90° angle

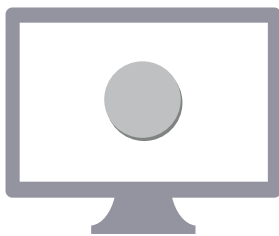


2° draft



Optimal Condition,
2° draft and fillet

PROCESS SUMMARY



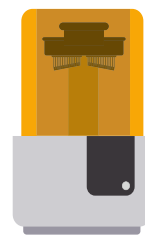
STEP 1

Design the part in CAD.



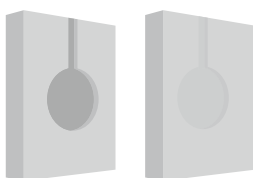
STEP 2

Design the mold in CAD.



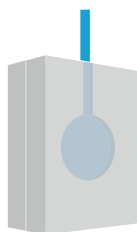
STEP 3

3D print the molds on the Form 2.



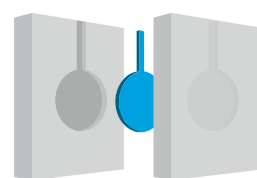
STEP 4

Remove support material from the molds.



STEP 5

Inject the mold with plastic.



STEP 6

Remove the part from the mold.



Contact Formlabs to learn how desktop SLA can work for your project.

Sales Inquiries

sales@formlabs.com
617-702-8476
formlabs.com

EU Sales Inquiries

sales-eu@formlabs.com
+44 1765 694053
formlabs.com

China Sales Inquiries

formlabs@elite-robot.com
+86 4006-029-015
elite-robot.com

Contact Galomb, Inc. to learn more about benchtop injection molding.

Galomb Injection Molder

inquiry@injectionmolder.net
610-434-3283
injectionmolder.net