
FORMLABS WHITE PAPER:

# Injection Molding from 3D Printed Molds

## A study of low-volume production of small LDPE parts

August 25, 2016 | Formlabs and Galomb Inc. | **formlabs.com**

# Table of Contents

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


**ABSTRACT**

This white paper describes the production of small injection molded low-density polyethylene (LDPE) 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: one of a large butterfly and one that produces four smaller butterflies in one shot. These molds were 3D printed in Formlabs' Clear Resin and the injection molded parts were produced by Galomb Inc.



*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 for final products.

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 the design of functional end use parts, challenging what it traditionally means to "ship" a product.

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

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

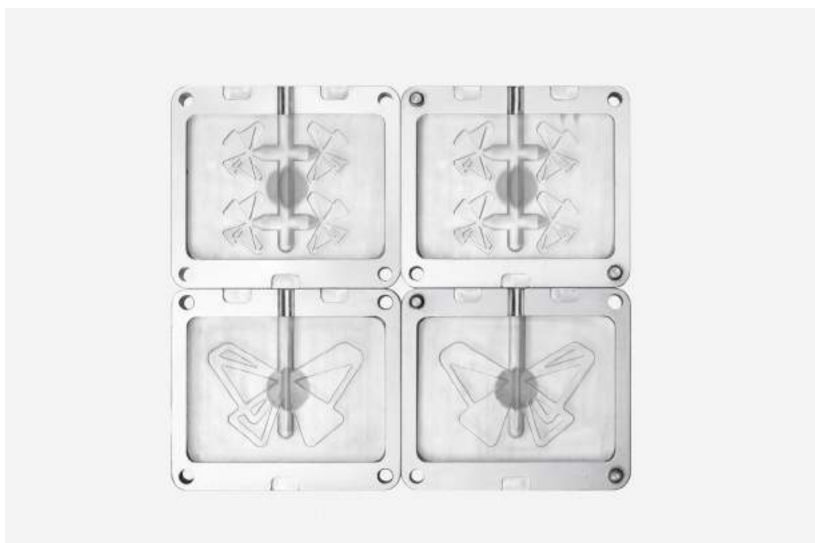


*Fig 2: 3D printed molds in aluminium frames.*

## METHOD

Formlabs' Clear Resin was selected because of its strength, high detail, and smooth surface finish. Because Clear is translucent, it's helpful in testing and inspecting molds to see if they 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. 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.

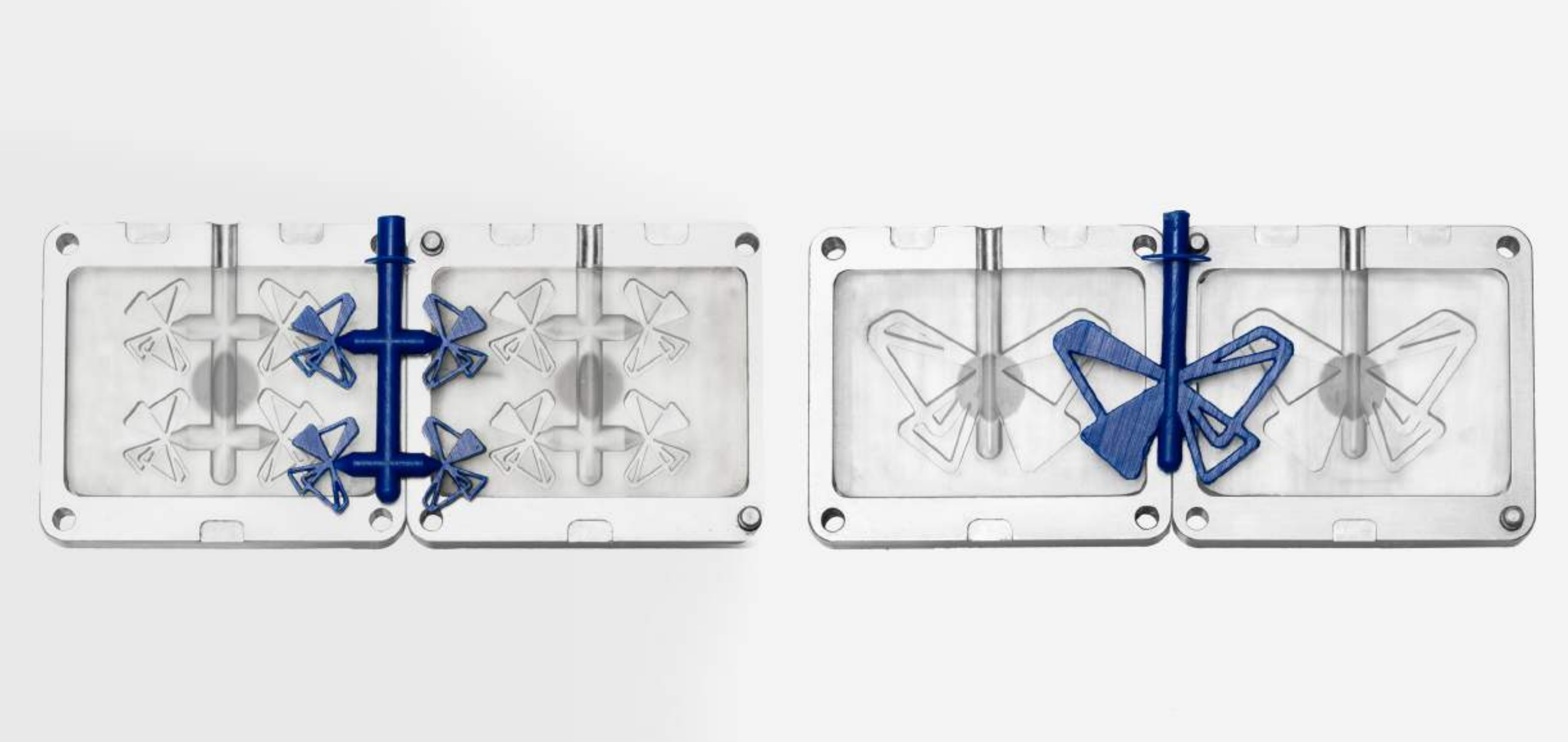


*Fig 4. 3D printed molds in aluminum frames and injection molded parts.*

One 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 before injection, eliminating the need to print the entire mold, which reduces print time and cost. 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 400 °F (204 °C) and was chosen for its low melt temperature. It should be noted that Formlabs Clear Resin has a heat distortion temperature (HDT) @66psi of 73.1ºC after post-cure (see the material data sheet). This number 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.

## 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; look for one that works with acrylics, as Formlabs Standard Resins contain methacrylates.

Cycle time for each shot was approximately three minutes. This process was accelerated by applying compressed air to cool the mold. Galomb also improved the mold design by etching in shallow (0.05 mm deep) air vents (not pictured) that led from the edge of the cavity to the edge of the mold so that air did not get trapped inside the cavity during injection.

## FURTHER TESTING

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.

While not pictured, fully-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. Additionally, the aluminium frames were used as a safety precaution to help contain the mold in the event of brittle failure, although this was not seen in the tests conducted by Galomb.

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 may improve the surface finish of the mold, but will increase print time.

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

## 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.

*90° angle*

*2° draft*

*Optimal Condition, 2° draft and fillet*

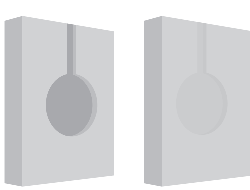
- Embossed and engraved details should be offset from the surface by at least 1 mm.
- Split plane surfaces can be polished with fine-grit sandpaper to reduce flash.
- If designing for an aluminium frame, add .125 mm of extra thickness to the back of the mold plates to account for compression forces and 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. Additionally, we do not recommend printing molds directly on the build platform, as the first 0.5 - 1 mm of each print is subjected to over-compression and may be distorted. Always print on supports for optimal dimensional accuracy and precise fit of mold components.

## PROCESS SUMMARY

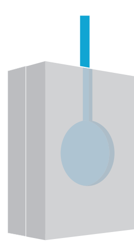
**STEP 1**
Design the part in CAD.

**STEP 2**
Design the mold in CAD.

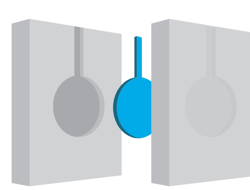
**STEP 3**
3D print the molds on the Form 2.



**STEP 4**
Remove support material from the molds.



**STEP 5**
Inject the mold with plastic.



**STEP 6**
Remove the part from the mold.