



QUICK START GUIDE

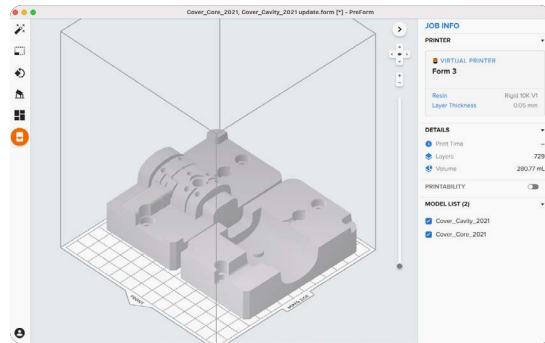
Injection Molding With 3D Printed Molds Quick Start Guide

Using 3D printed molds for low-volume injection molding can reduce costs, shorten lead times, and help bring better products to market. In this guide, we will walk you through the steps of using 3D printed molds on your injection molding machine.

1. Mold Design

Design the mold for your part in the CAD software of your choice. Adhere to common design rules for additive manufacturing and [injection mold design](#).

Upload your design into PreForm, Formlabs print preparation software. Prepare your print and send it to your Formlabs 3D printer.

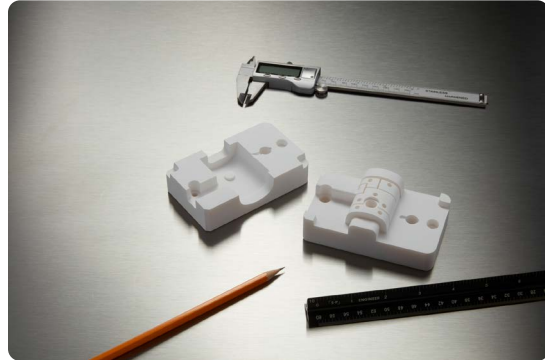


2. Mold 3D Printing

Choose a 3D printing material and begin your print. Rigid 10K Resin at 50 micron layer height is an ideal choice for most mold designs as it combines high strength, stiffness, and thermal resistance.

When possible, it is advised to print the mold flat, directly on the build platform without any supports, in order to reduce warpage.

After washing and post-curing, your 3D printed mold is ready to be integrated into your injection molding process.



3. Mold Assembly

Prior to assembly, you may choose to finish the mold to meet critical dimensions with hand-sanding, desktop or CNC machining.

It is recommended to place the printed mold inside a standard metal frame, or a Master Unit Die, to support against high pressures and extend the lifetime of your printed mold. Carefully assemble the 3D printed mold inside the metal frame. Add ejector pins, inserts, side-action parts and other components as needed.

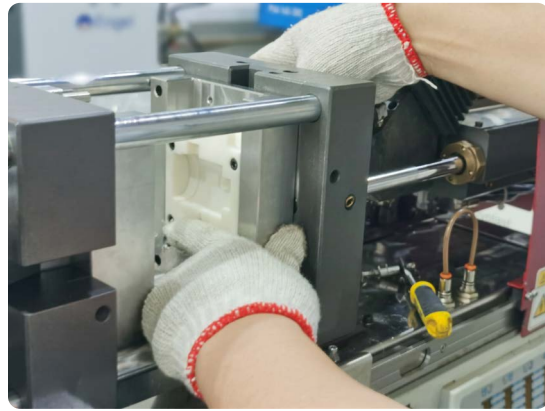
Install the assembled mold in your injection molding machine.



4. **Mold Clamping**

Insert the plastic pellets, input the required settings, and begin production. A lower clamping force is suggested particularly if the printed mold is not protected by a metal frame.

A broad range of thermoplastics can be injected with 3D printed molds such as TPE, PP, PE, ABS, POM, ASA, PA, PC, or TPU.



5. **Injection**

It may take a few shots to identify your ideal process conditions as many factors are at play including part geometry, choice of plastic, injection temperatures and pressures, and other parameters.

Reduce injection pressure and temperature as much as possible.

With one printed mold, Formlabs users are usually injecting 100s of parts in easy to process plastics such as TPE, PP and PE with temperatures up to 250°C. With plastics that require higher injection temperature such as PA or PC, the 3D printed mold might have a shorter lifespan.

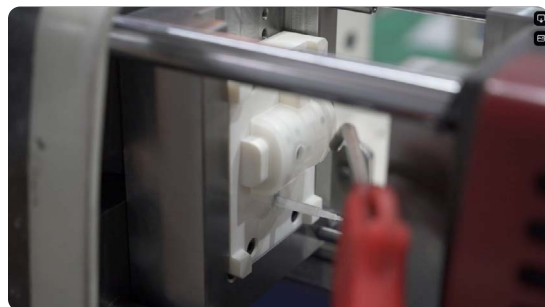
Read our [process conditions documentation](#) to see test results with both desktop and industrial injection molding machines.



6. **Cooling**

The cooling time of a polymer printed mold is longer than that of a metal mold, as thermal transfer occurs slower in plastic than metal. As such, adding cooling channels to your printed mold is generally not suggested.

Instead, cooling can be accelerated by applying compressed air or using interchangeable stacks.



7. Demolding

Demold the part either manually or automatically with ejector pins. Apply a release agent for thermoplastics with high viscosity. Mold releases are widely available and silicone mold releases, such as [Slide](#) or [Sprayon](#) products, are compatible with Formlabs Resins.



Incorporating short-run injection molding into your development process allows you to accelerate product development, iterate quickly, and bring better products to market. Formlabs' complete, easy-to-use ecosystem makes it simple to get started with 3D printing injection molds and can be seamlessly integrated into your existing workflow.

The photos used throughout this guide are from our partners at Multiplus.