



QUICK START GUIDE

3D Printed Injection Molding

Injection Process

1. Use standard clamping force $[(\text{injection pressure}) \times (\text{total projected area}) \times (\text{manufacturer's suggested clamping factor})]$ with a 10% safety factor.
2. Apply mold release agent (silicone-based release recommended).
3. First shot (short shot method) (Figure 1):
 - a. Injection molding time limit: 20 seconds
 - b. Pack and hold: 0 pressure for 0 seconds
 - c. Shot size: 75% of estimated part volume
 - d. Barrel temperatures: low end of range that is recommended for injected material
 - e. Injection speed: 10-20% of machine's max speed
 - f. Cooling time: 30-60 seconds
 - g. Mold open cooling time: until surface temperature reaches at least 120 °F (50 °C) or 2 minutes (apply compressed air to cavity surface to decrease cooling time)
4. Second shot (apply pack and holding pressure) (Figure 2):
 - a. Shot size: 90% of estimated part size
 - b. Pack and hold: 30-50% of injection pressure recorded from first shot for a maximum of 2 seconds
5. Third shot (Figure 3)
 - a. Increase or decrease holding pressure and holding time as needed to fill the cavity or reduce flash
6. Subsequent shots, go to troubleshooting quick start guide for help.



Figure 1



Figure 2



Figure 3

Troubleshooting Guidelines

PROBLEM	POSSIBLE CAUSES	SOLUTION
After first shot, there are signs of excessive flash on the injected part.	<ul style="list-style-type: none"> High injection speed caused flash at gate. Insufficient clamping force. 	<ul style="list-style-type: none"> Reduce the injection speed and increase the shot size on the subsequent shot. If using a MUD unit, make sure the 3D printed mold insert is protruding from the surface at least 0.008 in (0.2 mm). If not using MUD unit, increase the clamping pressure on the mold.
After second shot, the cavity is still not fully filled.	<ul style="list-style-type: none"> Not enough holding pressure or holding time. Mold cavity is not venting properly. 	<ul style="list-style-type: none"> Increase holding pressure or time. Create venting channel at end of cavity.
Injected part stuck on the cavity side (A side) of the mold.	<ul style="list-style-type: none"> Insufficient mold release applied to cavities. Injected part may not have solidified enough to latch onto the core. 	<ul style="list-style-type: none"> Apply silicone mold release to mold cavities. To dislodge part from cavity side, close the mold while the part is still warm, wait 5-10 seconds and open again. On following cycle, increase mold-closed cooling time by 10 seconds. If problem persists, increase mold-closed time by 5 seconds per cycle.
The ejected part deformed after cooling.	<ul style="list-style-type: none"> The ejected part was not fully solidified and shrank without mold support. 	<ul style="list-style-type: none"> Increase mold-closed cooling time by 10 seconds.
Injected part tore during the ejection process.	<ul style="list-style-type: none"> The part was not fully solid when ejected from mold and tore from ejector pin pressure. 	<ul style="list-style-type: none"> Increase mold-closed cooling time by 10 seconds.
There is a burn mark on the injected part.	<ul style="list-style-type: none"> It is possible that air was trapped inside the cavity and combusted due to increased pressure. 	<ul style="list-style-type: none"> Reduce injection speed to give time for air to flow out of mold. Introduce a venting channel in the back of the cavity to alleviate the pressure caused by the trapped air.