



PolyJet For Injection Molding

Injection molding is one of the most popular manufacturing processes. It is best used to mass-produce highly accurate, and often complex, three dimensional (3D) end-use parts. With cycle times ranging from a few seconds to a few minutes, injection molding is ideal for high-volume manufacturing applications.

3D printing is commonly used by injection molding manufacturers to create prototype parts for the detection of issues in a part's form, fit or function. Yet, 3D printing cannot always provide a complete assessment of a part's functional performance because 3D material properties may be different than those used in injection molding.

Until recently, the only option to create injection molded parts for testing by manufacturers and molders was to machine an aluminum (soft) tool with little or no action, such as cams, lifters or slides for side pulls. While these molds are far less expensive than their steel (hard) counterparts, costs and lead times are still significant. For example, the price to create a small, straight-pull mold ranges from \$2,500 to \$15,000 with delivery taking 10 days to 4 weeks. This is an investment that is hard for most companies to justify for a few dozen test parts. However, without proper testing, these investments may be diminished if further revisions to the part's design are required.

Today, PolyJet™ 3D printed injection molds are a better option for evaluating part design and performance. Capable of producing 5 to 100 parts in the same thermoplastic that will be used in production, they can be constructed in one or two days for a fraction of the cost of soft metal or steel tooling. By using PolyJet molds manufacturers can rapidly and cost-effectively assess a part's form, fit and true function.



Injection molded impeller with PolyJet molds.

COMPANION AND REFERENCE MATERIALS

Technical application guide	<ul style="list-style-type: none">• Document
Application brief	<ul style="list-style-type: none">• Document
Video	<ul style="list-style-type: none">• Commercial• Success story• How It's Used
Additional documents	<ul style="list-style-type: none">• White papers (2)• Case studies (6)

These references can be found at [Stratasys.com](https://www.stratasys.com).

TABLE OF CONTENTS

POLYJET FOR INJECTION MOLDING

1. OVERVIEW	3
1.1. Application	3
1.2. PolyJet is a best fit	3
1.3. Successful adopter traits	3
1.4. PolyJet adoption obstacles	4
1.5. Benefits	4
2. TRADITIONAL PROCESS OVERVIEW AND COMPARISON	5
2.1. Traditional injection molding process	5
2.2. PolyJet adjustments	5
3. INJECTION MOLDING MATERIALS	5
3.1. Recommended materials	6
4. MOLD DESIGN	6
4.1. Mold cavity	6
4.2. Mold components	7
4.3. Mounting	9
4.4. Printed inserts	11
4.5. Venting	12
5. FILE PREPARATION	12
5.1. Orient STLs	13
5.2. Select surface finish	13
5.3. Set printing material	13
6. MATERIALS AND PRINTERS	13
7. MOLD PREPARATION	13
7.1. Remove support material	13
7.2. Smooth surfaces	13
8. MOLD ASSEMBLY AND MOUNTING	14
8.1. Install core pins	14
8.2. Fit ejection system	14
8.3. Install sprue bushing	14
8.4. Install cooling line connections (optional)	14
8.5. Face mill mold inserts (mold-base option only)	14
8.6. Assemble molds and mount	15
9. INJECTION MOLDING	15
9.1. Clamping pressure	15
9.2. Release agent	15
9.3. Trial shots	16
9.4. Part molding	16
10. KEY PROCESS CONSIDERATIONS	17
10.1. Resolution details	17
11. TOOLS AND SUPPLIES	18
11.1. Required items	18
11.2. Optional items	18
12. RECAP – CRITICAL SUCCESS FACTORS	18
12.1. Actions	18
12.2. Eliminate obstacles	18

POLYJET FOR INJECTION MOLDING

1. Overview

1.1. Application:

PolyJet 3D printed molds replace machined injection molds for prototyping end-use plastics prior to investing in production tooling.

1.2. PolyJet 3D printed molds are a best fit when:

- Injection molding thermoplastics with reasonable characteristics.

- Low to moderate melting points

- $< 300^{\circ}\text{C}$ (570°F)

- Good flowability

- Candidates:

- TPE, PE, PP, PS, ABS, PA, POM, PC-ABS

- Glass-filled resins

- Typical quantities:

Quantities may vary based on the injected material and mold geometry. The information below is an average number of parts per mold based on a Stratasys customer survey.

- 100 parts using elastomer materials

- 70 parts using standard plastics

- 35 parts using engineering plastics

- Recommended part size is small to medium.

- $< 165\text{ cm}^3$ (10 in^3)

- Injection molding machines under 200 tons clamping force

- Design and functionality confirmation is desired.

- Avoidance of rework on soft or hard tools

- Verification for compliance testing (e.g., UL or CE)

1.3. Successful adopter traits (first iteration and long-term):

- Invest time in process development.

- Determine optimum molding parameters through small, incremental adjustments.

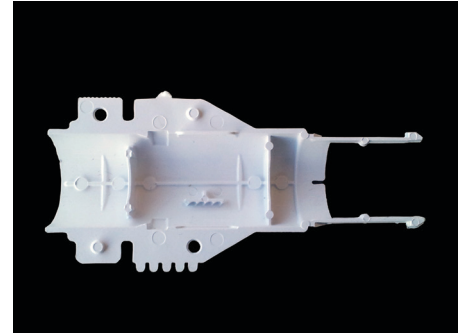
- Open to change.

- Willing to alter mold designs, molding parameters and cycle times.

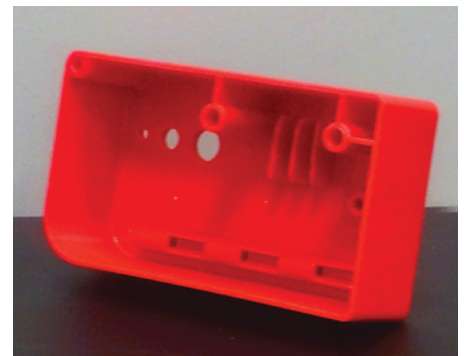
- Have appropriate project goals.

- To evaluate injection molded part performance.

- Not qualifying the molding process or replacing preproduction molds



An injected sample part made from ABS material, created by Nypro, to evaluate 3D printed mold performance.



Molded part with shut-offs and moderately deep draw.

POLYJET FOR INJECTION MOLDING

1.4. PolyJet adoption obstacles:

OBSTACLE	SOLUTION*
Using standard practices	<ul style="list-style-type: none">• Modify PolyJet mold design.• Modify molding parameters.
Using PolyJet molds for pilot runs or low-volume manufacturing	<ul style="list-style-type: none">• Use in product development phase to produce functional prototypes.
Molding complex parts	<ul style="list-style-type: none">• Creatively design 3D printed mold inserts that compensate for features that normally require complex sliders, and manually eject the mold inserts after injection. These inserts aid the creation of complex designs with challenging features like negative draft angles and snap fits.

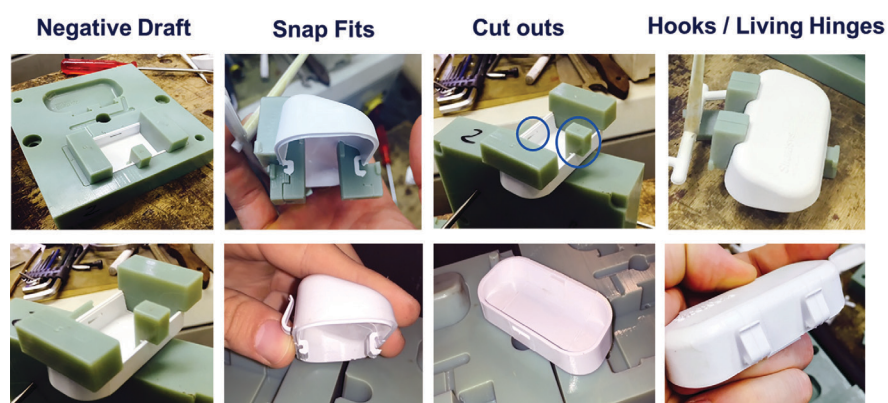
*Additional solutions may exist.



Spec resin provides functional testing of the living hinge.

1.5. Benefits:

- Lead time reduction
 - Average lead time savings: 50% - 90%
- Cost reduction
 - Average cost savings: 50% - 70%
- Spec resins
 - Functional evaluation with production plastics
- Efficiency gains
 - Automated tool - making with few steps
- Early confirmation
 - Validate part performance and tool design
 - Validate thermoplastic selection



Explanation of various injection molded features.

2. Traditional Process Overview And Comparison

2.1. The steps in the traditional injection molding process are:

2.1.1. Part design

- CAD software

2.1.2. Mold design

- CAD software

2.1.3. Make mold

- CNC aluminum or steel

2.1.4. Set up mold

- Mount on molding machine
- Sample shots to dial in parameters

2.1.5. Mold samples (“First Articles”)

- Injection mold sample parts
- Inspect and review
- Rework mold as needed and repeat steps 2.1.4 and 2.1.5.

2.1.6. Make parts

- Injection mold parts
- Trim flash and gate

2.2. Adjustments needed with PolyJet molds compared to the traditional process

When using PolyJet printed molds the process is similar, with minor variations:

2.2.1. Part design

- CAD software*

2.2.2. Mold design

- CAD software*
- Modify mold design as needed

2.2.3. Make mold

- 3D Print the mold

2.2.4. Set up mold

- Mount on molding machine*
- Create sample shots to dial in parameters*

2.2.5. Mold samples (“First Articles”)

- Injection mold sample parts*
- Inspect and review*
- Rework the mold as needed by repeating the necessary steps. With printed molds you can change the current molds or print a totally new mold design that will suit your needs.

* Similar to the conventional injection mold design process.

3. Injection Molding Materials

When using PolyJet molds, both tool life and part quality will be dependent on the thermoplastic material used during the injection molding process. As melt temperature, viscosity and abrasiveness rise, tool life will decline. Short shots and knit lines are two phenomena that are caused by using highly viscous polymers while injection molding. One way to prevent them is to use high pressure when injecting the material, however PolyJet molds cannot withstand extreme pressure, so using polymers with good flow behavior is recommended.

POLYJET FOR INJECTION MOLDING

Another way to prevent this is to increase the barrel temperature. This improves the material viscosity and delivers better parts.

When facing the need to increase the injection pressure, enlarging the gate may be a good solution and will help reduce the injection pressure. This can also be done after printing the mold.

Part geometry also plays a role in selecting the injection molding material. When features do not impede the plastic's flow, higher viscosity and temperature resins may be used.

3.1. Recommended materials

Thermoplastics are divided into four classes based upon their ease of processing (Figure 1). For each class, an approximate tool life by the number of shots is listed. Note that the number of shots is also dependent on part geometry. For example, tall, thin mold features will reduce mold life.

Thermoplastics requiring barrel temperatures of up to 300°C (570°F) have been successfully molded using PolyJet molds. However, thermoplastics with relatively low melting points and good flowability, such as those in Class A like polypropylene (PP) and polyethylene (PE), will have the longest tool lives.

Class B materials, such as acetal (POM) and glass-filled polypropylene (PP+G), have shorter tool lives due to increases in temperature, viscosity or abrasiveness.

As shown in Figure 1, some thermoplastics are poor candidates for 3D printed molds. Please note that injecting clear materials is possible. However, since those materials usually require highly polished cavities and precise temperature control, the results with 3D printed molds will not give the same clarity level as traditional tools. Post processing clear parts after injection may improve the clarity.

4. Mold Design

Begin with a mold design using the guidelines and best practices for traditional injection molds. These design concepts can be applied to PolyJet molds, but alterations are required to compensate for the mechanical, thermal and dimensional characteristics of a plastic mold.

4.1. Mold cavity

4.1.1. Draft

Increase draft angles to the maximum permitted by the molded part's design. By increasing the draft angles, parts are less likely to resist ejection from the mold, thereby decreasing the opportunity for damage to the mold.

Five degree draft angles are recommended (Figure 2).

4.1.2. Radii

Add small radii to break sharp corners on thin features. This avoids stress concentrations that may cause localized mold damage.

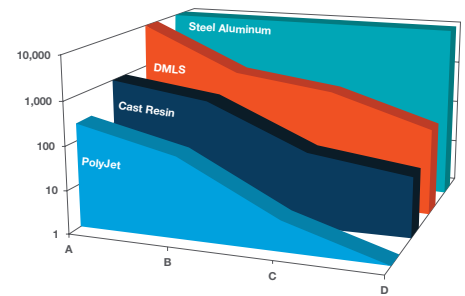


Figure 1: Estimated number of parts obtained per tool, based on type of material used (see table below).

A	• Polyethylene (PE)
	• Polypropylene (PP)
	• Polystyrene (PS)
	• Acrylonitrile Butadiene Styrene (ABS)
	• Thermoplastic elastomer (TPE)
B	• Glass-filled Polypropylene (PP+G)
	• Acetal (Polyoxymethylene [POM])
	• Polycarbonate-ABS blend (PC+ABS)
C	• Polycarbonate (PC)
	• Glass-filled Acetal (POM+G)
	• Polyamide (PA)
D	• Glass-filled Polycarbonate (PC+G)
	• Glass-filled Polyamide (PA+G)
	• Polyphenylene Oxide (PPO)
	• Polyphenylene Sulfide (PPS)



Figure 2: Increase draft angle to ease ejection. Five degrees is recommended.

4.1.3. Parting surfaces

To establish parting surfaces that will have minimal flash, PolyJet molds use the injection molding machine's clamping force to compress the material and create a seal. Flashing is also managed through adjustment of injection molding parameters, such as injection rate, temperature and pressure.

If using a Master Unit Die (MUD) base (see Section 4.3.), extend the back face of the core and cavity by 0.2 mm (0.008 in) to make them taller than the MUD base's pockets (Figure 3). Optionally, the inserts may have the same depth as the pocket. In that case, shims are needed when assembling the mold (see Section 8.5).

4.1.4. Shut-offs

For shallow features that are approximately 6.0 mm (0.25 in) or less with shut-off surfaces nearly perpendicular to the mold's pull direction, modification is unnecessary (Figure 4).

For all other shut-offs, it is recommended to inset the faces by 0.05 - 0.1 mm (0.002 - 0.004 in) to allow the core and cavity to mate without secondary finishing (Figure 5).

4.1.5. Core pins

Core pins in metal tools can deflect and shear due to the pressure of the resin as it fills the mold cavity.

The recommended core pin aspect ratio is 3:1 (height : width). To improve mold longevity, design a 3D printed insert that will be replaced after several shots, when the pins shear or deflect. You can also substitute steel pins that are press fit into the mold during assembly (see section 8.1). Create holes to receive these pins and ream the holes as needed after printing to achieve a snug fit.

4.1.6. Holes

All holes in a PolyJet mold should have a diameter that is larger than 0.8 mm (0.04 in). Smaller holes can be machined prior to mold assembly. Use additional care on the ejector and sprue holes (see sections 4.2.3 and 4.2.4).

4.1.7. Shrinkage compensation

Before proceeding to Section 4.2., scale the core and cavity to compensate for the shrinkage of the thermoplastic that occurs with conventional injection molding.

4.2. Mold components

4.2.1. Gates

- Gate type

To minimize shear forces within the mold cavity, opt for sprue, modified-fan, edge, tab, overlap or external-ring gates. Avoid tunnel and point gates.

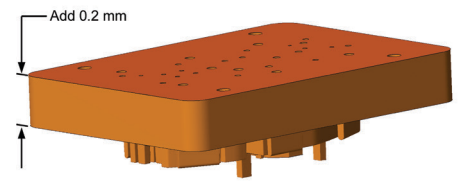


Figure 3: Extend back face (red) when using a MUD base.



Figure 4: Shut-off for this hinge required no mold modification.

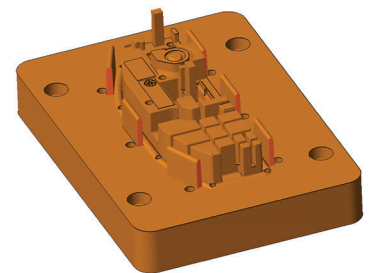


Figure 5: Inset shut-off faces (red).

- Gate dimensions

To improve material flow and decrease pressure within the tool, enlarge the gate. The size of the gate will depend on the resin's viscosity and the mold's flow characteristics.

In general, make gates two to three times larger than those used in metal molds.

Make edge gate thickness equal to the wall thickness of the part at the point of injection (Figure 6).

Where possible, use direct sprue gates for an easy, uniform flow into the cavity. The recommended diameter of the sprue gate is between 5 - 8 mm (0.2 - 0.3 in) for:

- Molds less than 100 × 50 × 25 mm (4 × 2 × 1 in)
- Part thicknesses 1 - 2 mm (0.04 - 0.08 in)
- Resins with good or moderate flow characteristics
 - For high-viscosity resins or glass-filled resins, use an 8 - 9 mm (0.30 - 0.35 in) sprue gate.

4.2.2. Runners

Runners do not require adjustment, however hot runner systems are not recommended (Figure 7).

4.2.3. Sprue

Avoid direct contact between the molding machine's nozzle and the PolyJet insert. Incorporate the sprue in the MUD base / steel plate (see Section 4.3.) or add a hole that will receive a standard sprue bushing. A sprue bushing with a minimum draft angle of 3 degrees is recommended. If using a bushing, undersize the hole by 0.2 - 0.3 mm (0.008 - 0.012 in) before printing and ream to size during mold assembly (see Section 8.3.).

4.2.4. Ejection system

If an ejection system will be used (see Section 8.2.), add ejector holes as needed (Figure 8). As with the sprue bushing, undersize the holes by 0.2 - 0.3 mm (0.008 - 0.012 in) and ream to size during mold assembly. It is recommended to make sure the holes for the ejector pins won't be too close to the edges. It will weaken the mold especially after reaming. Keep the holes 3 mm (0.118 in) from the edges.

4.2.5. Cooling system

Due to the thermal characteristics of a PolyJet mold, cooling systems will not significantly affect molding cycle times or part quality. However, a cooling system can improve tool life; on average, a 20% improvement can be expected. The improvement increases as the depth of the cavity and height of the core decreases, since the cooling effects reach more of the surface area of the molding cavity.

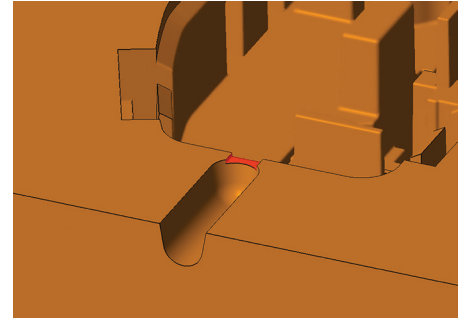


Figure 6: Enlarge all gates. Edge gate (shown) should be the thickness of the wall.

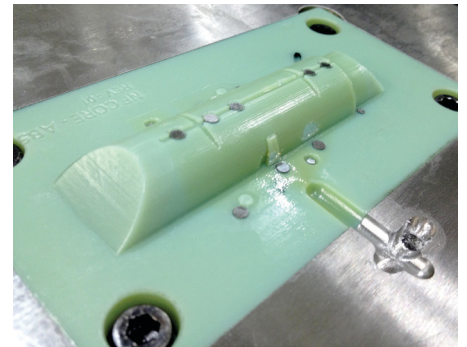


Figure 7: Runners do not require adjustment.

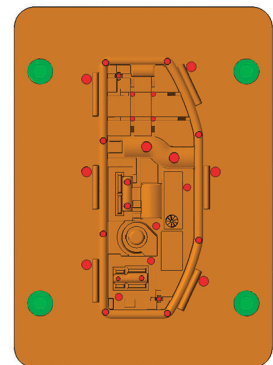


Figure 8: Add holes (red) for ejection system. Corner holes (green) are for attachment to the mold base.

4.2.5.1. Installing the cooling system

- Locate the centerline of the cooling channels 8 - 10 mm (0.3 - 0.4 in) below the molding surfaces and use a diameter of 8 mm (0.3 in). In tight areas, a 6 mm (0.2 in) diameter channel may be used. Note that conformal cooling lines are not recommended due to the difficulties in removing support material.
- To receive connector hose nipples, add a counterbore at each cooling channel entry and exit point. For 8 mm (0.3 in) channels, use a 8.8 mm (0.35 in) bore diameter with a 10 - 12 mm (0.4 - 0.5 in) depth. For a solid, sealing fit, the threads will be tapped prior to mold assembly (see Section 8.4.).

4.3. Mounting

PolyJet molds may be mounted in a MUD base (preferred, Figure 10) or to steel plates (Figure 11). The steel plate option may include or exclude the ejection system (Figure 12). Table 1 lists the advantages and disadvantages of each approach.

Method	Advantages	Disadvantages
MUD Base (recommended)	<ul style="list-style-type: none"> • Better core and cavity alignment. • More complexity in part due to ejection system. • Better dimensional stability; 3D printed inserts are constrained. • Smaller inserts are faster and less expensive to print. • MUD base is reusable. 	<ul style="list-style-type: none"> • Initial investment in MUD base with modular ejection system.
Steel Plate With Ejection System	<ul style="list-style-type: none"> • Lower investment. • More complexity in part due to ejection system. 	<ul style="list-style-type: none"> • Investment in modular ejection system • Mold is not constrained. • Requires extra mold material (larger frame) to withstand high injection pressures.
Steel Plate Without Ejection System	<ul style="list-style-type: none"> • Lowest investment. • "Plug and play" solution. • Allows greater creativity in mold design. 	<ul style="list-style-type: none"> • Manual ejection limits part features. • Manual ejection may require more complex mold design. See the image in Section 2 depicting multiple mold inserts. • Mold is not constrained. • Requires extra mold material (larger frame) to withstand high injection pressures.

Table 1: Mold mounting options.

• MUD base

PolyJet core and cavity inserts are seated in pockets within a standard metal MUD base. Ideally, this MUD base will be reused with other PolyJet printed inserts for future injection molding projects. With this in mind, incorporate a modular ejection system (Figure 13). Adding new ejector pins and holes in the MUD base when needed is optional.

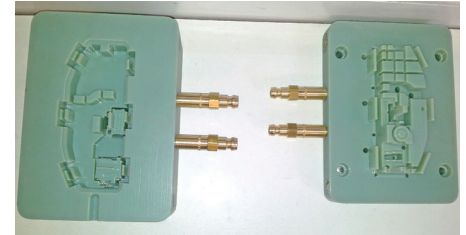


Figure 9: Cooling line connections on PolyJet core and cavity.

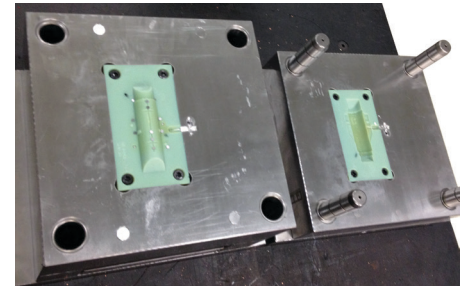


Figure 10: MUD base with PolyJet inserts.

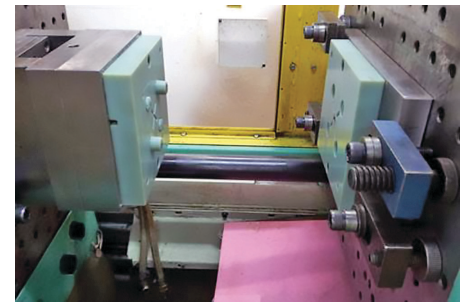


Figure 11: PolyJet mold with ejection system attached to steel backing plates.



Figure 12: Printed mold for manual ejection.

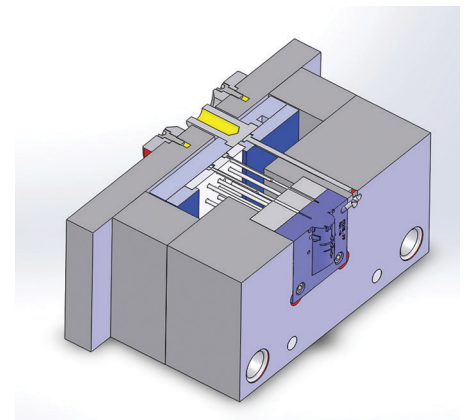


Figure 13: Section view of MUD base with printed insert (light blue) and modular ejection system.

POLYJET FOR INJECTION MOLDING

Select a MUD base with pockets that will accommodate printed inserts that are 20 - 25 mm (0.75 - 1.0 in) larger than the cavity on all sides. Note that this is a general guideline since some deviation may be needed, such as sizing to align a runner with that in the MUD base.

Design both the core and cavity inserts such that they are 1.0 mm (0.04 in) larger than the pockets' profiles. Extend all side faces by 0.5 mm (0.02 in) (Figure 14). Make the depth of the inserts 0.2 mm (0.008 in) larger than that of the pockets, as recommended in Section 4.1.3.

During mold assembly (see Section 8), mill the side faces for a tight, precise fit. The depth of the inserts will not be altered; the small extension beyond the face of the MUD base provides compression that will seal the parting line to minimize flash.

For attachment to the mold base, add holes for the mounting bolts, using the bolt pattern in the MUD base or ejector plate.

- Steel plate with ejection system

The PolyJet core and cavity are mounted to steel plates that compensate for their height, reducing the travel distance to close the mold. On the core side, a standard ejection system is located between the steel plate and the injection molding machine's platen. On the cavity side, the plate mounts to the platen, and incorporates the sprue and sprue bushing.

Make the core and cavity additional frame no less than 30 mm (1.18 in) larger than the mold cavity. Ensure the mold thickness in the clamping force direction is no less than 20 mm (0.78 in). As shot size or shot pressure increase, allow additional thickness in all directions. For example, size adjustments to align runner systems are acceptable.

In the steel plate, include threaded holes to receive mounting bolts that pass through the inserts. Using this bolt pattern, add bolt holes to both inserts with a diameter equal to the bolts' shafts.

- Steel plate without ejection system

The PolyJet core and cavity are mounted between steel plates that compensate for the mold's height, thereby reducing the travel distance to close it. On the cavity side, the plate may incorporate the sprue and sprue bushing.

Make the core and cavity additional frame no less than 30 mm (1.18 in) larger than the mold cavity (Figure 15). Ensure the mold thickness in the clamping force direction is no less than 20 mm (0.78 in). As shot size and shot pressure increase, allow additional material in all directions.

To maintain alignment between the mold components, add holes around the perimeter that will receive alignment pins or mounting bolts. Use a hole diameter that is 0.1 mm (0.004 in) smaller than the shafts of the pins or bolts.

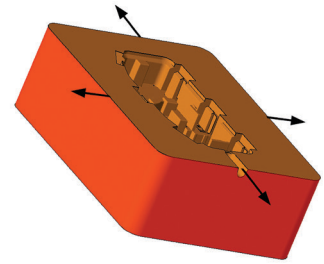


Figure 14: Extend side walls to provide machining stock.

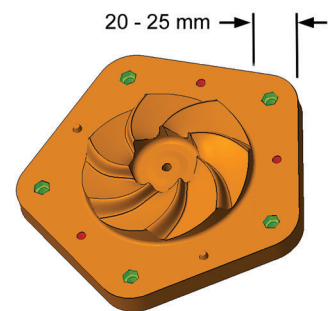


Figure 15: Manual ejection; plate-backed mold with alignment pin holes (red) and optional attachment bolt holes (green.)

Lacking an ejection system, molded parts must be manually removed from the tool. When using a two-part mold, extraction may be very difficult. To easily extract the part, segment the mold into three or more pieces.

With manual ejection, it is easy to use printed inserts and create challenging features like negative draft, vertical shut-offs and complex features.

Three-part molds require additional parting lines. Begin by considering how the tool may be dissected to facilitate extraction, and then consider where the resulting parting lines are needed and acceptable. For example, the outer diameter of the outer ring of the impeller is formed with a PolyJet insert sandwiched between inserts for the inner diameter of the outer ring and the contours of the blade surfaces (Figures 16A and 16B). Optionally, design individual “pick-out” inserts for any features that make extraction more difficult.

For quick release, simply attach the cavity side of the mold to the platen with double-sided tape. Optionally, the mold design may include features for quick-release hardware or attachment bolts.

NOTE

For all three mounting options make sure that the printed mold has full back support in the clamping force direction and supports the entire mold, not just the frame.

4.4. Printed inserts

When using printed molds, inserts may be designed and used. Using printed inserts in the mold design:

- Allows creative design that will help you produce challenging parts with complex features (see the image in Section 2 depicting molds used with inserts). This capability usually requires manual ejection and enables a less-costly solution for challenging parts.
- Solves venting problems. Like the traditional method, air might be trapped in the cavity and cause unfilled parts. When the mold cavity (the void that is filled with resin) has areas that need to be filled beneath the parting line the air might be trapped. To solve this, design the feature as an insert. See Figure 17A. Additional examples of 3D printed molds with inserts are shown in Figure 17B.
- Improves mold longevity. This kind of insert is recommended when the mold has special features that will probably deform first. You can print multiple inserts and quickly replace them when they crack or deform, instead of replacing the entire mold.

When printing the inserts make sure the surface touching the injected resin is printed in Glossy mode (see section 5.2).

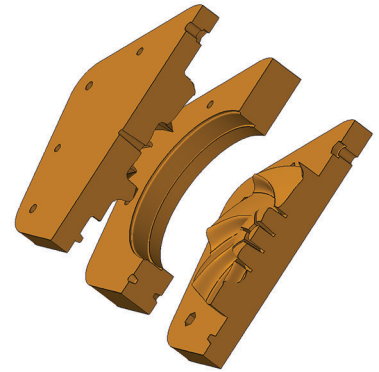


Figure 16A: Three-piece PolyJet mold (section view) designed to make manual extraction easier.

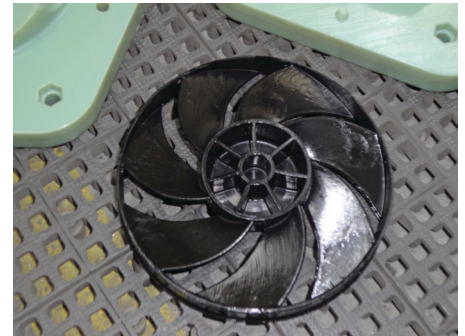


Figure 16B: Injection molded impeller made with a three-piece PolyJet mold.

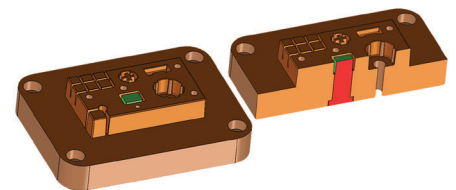


Figure 17A: On the left is the complete core side mold (orange), assembled with the insert (green). On the right is a section view of the same assembly, showing the need for a venting solution that was solved by the printed insert (red)."

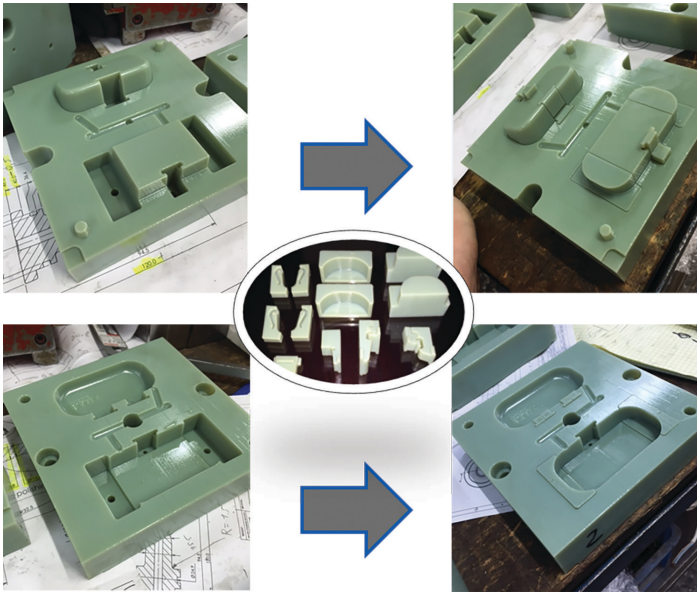


Figure 17B: 3D printed mold shown with multiple inserts.

4.5. Venting

Air may get trapped in the injection mold cavity and cause unfilled parts. There are two solutions to avoid trapped air when using printed molds:

- **Design the mold with inserts.**

This solution is usually needed when the mold design has a deep section and the resin flows underneath the parting line. In this case design the mold with inserts to help the air vent from the cavity through the side section connection of the inserts (see section 4.4).

- **Create venting channels on the parting line.**

There are two methods for solving venting problems during the injection. Both are helpful and which one you choose depends on your personal preference.

If, when designing the mold, you anticipate that the part will have venting problems, add a flat tunnel (0.2 mm, 0.0078 in. deep) on the parting line surface, approximately 5 mm (0.2 in.) away from the cavity. Then, if venting does occur during the injection process, scratch the parting line surface from the cavity to the tunnel.

If your mold does not have a venting channel and venting problems occur during injection, scratch the parting line surface of the mold in the area needing venting to help the air escape.

In both cases, the scratch, from the cavity edge, should be thin (approximately 0.02 mm, 0.00078 in.) and can be done with an ordinary knife.

5. File Preparation

Import the STL files into Objet Studio™ software and prepare them for 3D printing.

POLYJET FOR INJECTION MOLDING

5.1. Orient STLs.

Orient the mold halves with the cavities facing upward so that the molding surfaces will be glossy and have no support material contact (Figure 18) (Left click on part > Transform > Rotate).

Also adjust the orientations, when possible, so that the flow of the injected resin will be along print lines in the PolyJet tool (Figure 19). Orient the molds to follow the resin's flow direction along the X axis of the 3D printer, which is the direction of print head travel. This alignment improves the process of filling the mold cavity with lower injection pressures.

If they will fit, place both at the same point on the Y axis to minimize build time.

5.2. Select surface finish.

For a smooth, and nearly mold-ready finish, print the mold halves using the "Glossy" finish mode (Left click > Glossy).

5.3. Set printing material.

Print the mold using Digital ABS green or ivory. For Digital ABS green, use RGD5160-DM (Model Settings > Multiple Materials > RGD5160 - DM). For ivory, use RGD5130 (Model Settings > Multiple Materials > RGD5130 - DM).

6. Materials And Printers

To build PolyJet injection molds, the following Objet 3D printer and material combinations are recommended.

- Objet30 Pro™, Objet30 Prime™, Eden260V™, Eden260VS™, Eden350V™, Eden500V™, Connex1™: Rigur™
- Objet260 Connex2/3™, Objet350 Connex2/3™, Objet500 Connex2/3™: Digital ABS™ (RGD5160)

In general, Digital ABS is the preferred material with the most suitable mechanical properties for creating printed injection molds. It provides the longest tool life and is best suited for molding complex geometries or when molding with higher temperature thermoplastics.

7. Mold Preparation

7.1. Remove support material.

For rectangular stand-alone molds and MUD base inserts, simply scrape the support material from the mold halves using a putty knife or similar tool.

7.2. Smooth surfaces.

The surface smoothness of the PolyJet cavity will contribute to the cosmetic appearance of the resulting injection molded part, so it is important to have smooth mold surfaces.

Consider sanding layer lines that are transverse to the direction to the mold's opening. This will make ejection of the molded part easier.

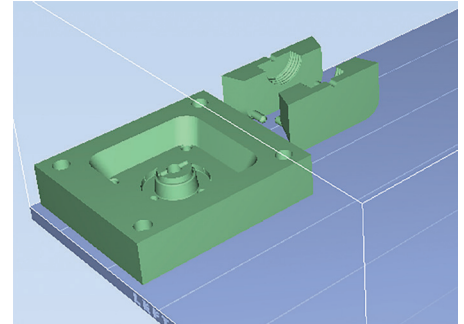


Figure 18: Orient so that molding surfaces will be glossy and have no support material.

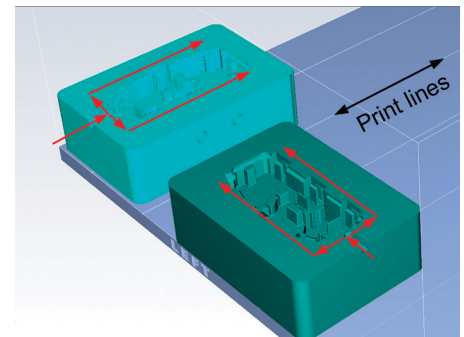


Figure 19: Orient so that resin flow (red) will be along print lines, as shown by the far core.

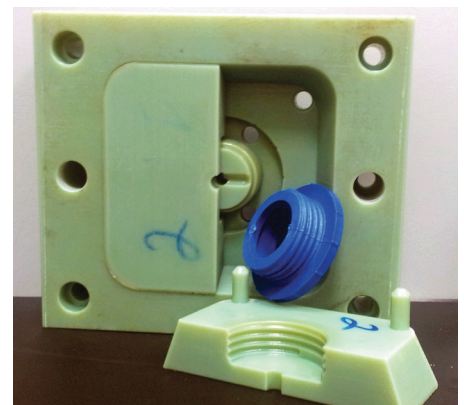


Figure 20: Digital ABS tool with injection molded part (PA 6/6 with 20% glass fiber).

7.2.1. Sanding for extraction

Lightly sand all surfaces in the mold cavity that rise in the direction of the part extraction. Use medium-grit wet / dry sandpaper (180- to 220-grit).

7.2.2. Sanding for appearance (optional)

Sand all surfaces of the mold's cavities to a smooth finish. Start with a medium-grit wet / dry sandpaper (180- to 220-grit) and sand until the surfaces are visibly smooth. Complete this sanding step with fine-grit sandpaper (320- to 400-grit). This will create a good level of smoothness for most finish types.

7.2.3. Sandblast machine

Using a sandblast machine on the printed molds is an option. The process will help smooth the mold cavity and remove the printing stripes. Protect the parting line of the mold and special areas that you don't want to be sanded with tape or by printing a jig to fit the mold and cover those areas (Figure 21).

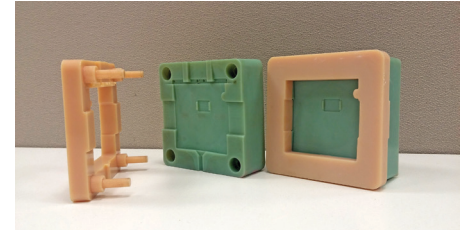


Figure 21: The tan printed jig was designed to protect the parting line when sandblasting the mold.

8. Mold Assembly And Mounting

8.1. Install core pins.

Ream all holes that will receive core pins to create a very tight fit. Press fit the pins into the holes. Then, assemble the core and cavity to confirm that the pins are seated to the proper depth.

To ensure that the pins will be properly aligned, reaming should be performed on a drill press or milling machine.

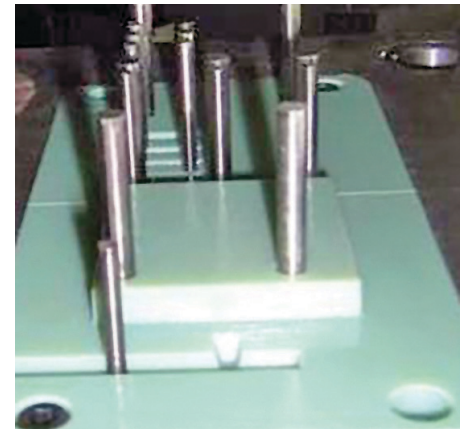


Figure 22: Ejector pins must be snug but able to move freely.

8.2. Fit ejection system.

Ream all holes through which ejector pins will pass (Figure 22). To prevent flash around the ejector pins, make sure that the pins fit snugly, but can move freely. As with the reaming of the core pin holes, use a drill press or milling machine. (Figure 23). The recommended tolerance for the hole is H7 (according to the ISO hold chart tolerance).

8.3. Install sprue bushing.

The metal sprue bushing used in the traditional process also needs to be used with printed molds. It significantly improves the longevity of the mold and the injected part quality.

If the sprue bushing is not integrated into a MUD base or steel plate, press fit it into the receiving hole. If needed, sand or ream to adjust the fit.

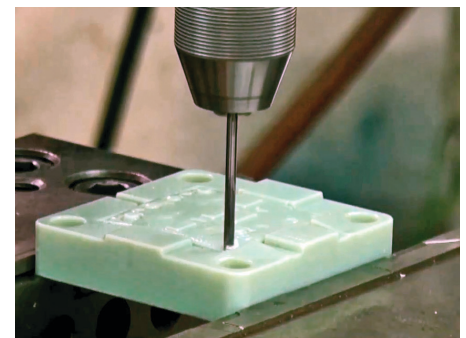


Figure 23: Using a drill press or milling machine to ream ejector pin holes.

8.4. Install cooling line connections (optional).

If using cooling lines, attach connection nipples to the mold inserts. Tap the counter-bore to the thread specification for the connection nipple.

8.5. Face - mill mold inserts (MUD base option only).

Remove the machine stock (see Section 4.3.) added to the core and cavity inserts' side walls. Test fit the inserts with the MUD base to confirm that both are snugly seated.



Figure 24: Verifying a snug fit after reaming.

POLYJET FOR INJECTION MOLDING

Once seated, confirm that the inserts extend beyond the face of the MUD base by 0.2 mm (0.008 in) (Figure 25). If the inserts sit too high, mill the back faces. If they sit too low, insert shim stock in the bottom of the MUD base pockets to raise them to the desired height.

8.6. Assemble molds and mount.

8.6.1. MUD base

Place PolyJet inserts in the pockets of the MUD base and attach with bolts. Do not over-tighten attachment bolts as this may cause the inserts to crack. Attach the MUD base to the injection molding machine's platens (Figure 28).

8.6.2. Steel plate with ejection system

Attach the PolyJet core and cavity to the steel plates with bolts. Do not over-tighten.

Next, attach the plates to the injection molding machine's platens (Figure 26).

8.6.3. Steel plate without ejection system

Assemble the core, cavity and intermediary mold components. Apply tape to the mold's sides to hold the assembly together.

Attach the cavity side of the mold to the machine's platen (nozzle side) using double-sided tape. Using tape provides a quick release of the mold after each shot, so that the mold can be disassembled for part extraction (Figure 27).

Optionally, assemble the mold with bolts that pass through and then attach to the platen.

9. Injection Molding

Injection molding with a PolyJet mold requires adjustments to the molding process (Figures 28 and 29). It is important to start with conservative values to avoid damaging the tool. Using test shots, slowly adjust the process parameters until desired parts are produced (Figure 30).

The following information provides guidelines as a starting point for process adjustment.

9.1. Clamping pressure

Use the standard clamping force (injection pressure × total projected area or total surface area × manufacturer's suggested clamping factor). You can adjust this value with at least a 10% safety factor.

Test the clamping force by slowly closing the mold and observing if the PolyJet mold is compressing as designed. Use a two-stage process: rapid travel until just before contact followed by a slow, gentle speed to fully close the mold. If necessary, adjust by milling the back face or adding shim stock.

9.2. Release agent

Before each injection molding cycle, liberally apply a silicone mold release agent to the mold cavity.

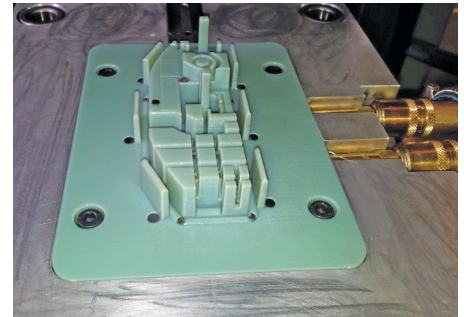


Figure 25: Confirm that the insert is 0.2 mm (0.008 in) above the surface of the MUD base. Adjust if necessary.

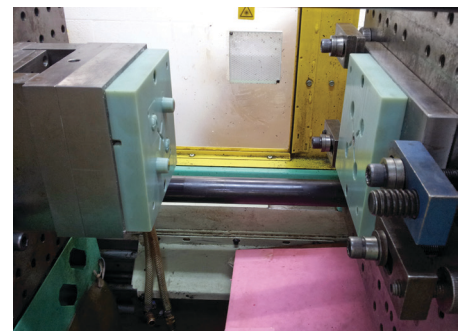


Figure 26: Attach core and cavity to steel plates and modular ejection system.



Figure 27: Assemble mold and attach to steel plate with double-sided tape.

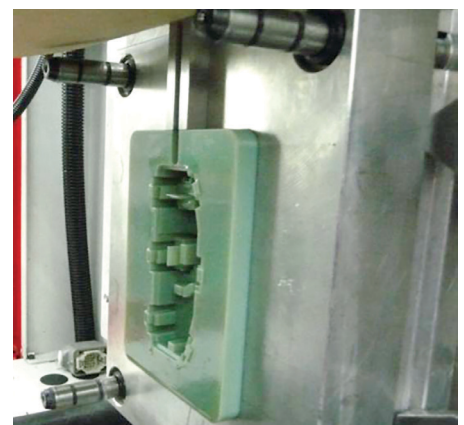


Figure 28: MUD base with PolyJet cavity on the press.

POLYJET FOR INJECTION MOLDING

9.3. Trial shots

The goal of the trial shots is to keep temperatures, pressures and flashing to a minimum since they can reduce the tool life. Also, because PolyJet molds are poor thermal conductors, molded parts will require additional time to solidify. The trial shot process will identify the appropriate amount of time for cooling.

To begin, use the following parameters:

- Injection molding time limit: 20 seconds
- Pack & hold phase: 0 psi and 0 seconds
- Shot size: 75% of estimated part volume
- Barrel temperatures: Low end of that recommended for the resin.
- Injection speed: Low end of that recommended for the resin (10% to 20% of the machine's maximum screw speed).
- Cooling cycle: Depends on the thermoplastic material being molded.

For materials with slower solidification rates, increase cooling cycle. Allow ample time between shots to allow the mold to cool to a target temperature of 50°C (120°F). Accelerate cooling by blowing compressed air onto the core and cavity.

With each subsequent trial shot, adjust the process parameters until part quality is satisfactory. If the part is unfilled due to venting problems, scratch the parting line in this area to allow air to escape (section 4.5).

Adjust shot size first, with a target of 90% of the cavity volume. Next, adjust the packing pressure to 30% - 50% of the injection pressure. Review the results and adjust as necessary. To avoid sink marks, also begin to increase the holding time.

For fine tuning, make adjustments to the barrel temperature and injection speed. However, avoid using elevated temperature and pressure to resolve molding issues, because these settings can decrease the number of injection molded parts the PolyJet mold can produce. With that in mind, remember that higher temperatures improve material viscosity.

Also, increase the cooling cycle duration to achieve full solidification. However, do not allow the tool to cool too much as this will increase part shrinkage, and potentially cause the part to grip the core (Figure 30). If the grip is too strong, the core could be damaged when the part is ejected. If there is excessive flash after dialing-in the parameters, disassemble the tool and add additional shim stock between the PolyJet inserts and mold base.

9.4. Part molding

After dialing-in the process, injection mold the desired number of parts with one additional alteration (Figure 32).

Because the material used in the PolyJet mold will act like an insulator, the temperature of the mold will increase to the point that parts will not solidify. To maintain a target temperature of 50°C (120°F), keep the mold open after part extraction and blow compressed air onto the core and cavity. This may be done manually or with an automated cooling fixture (Figure 33).



Figure 29: MUD base with the PolyJet core.

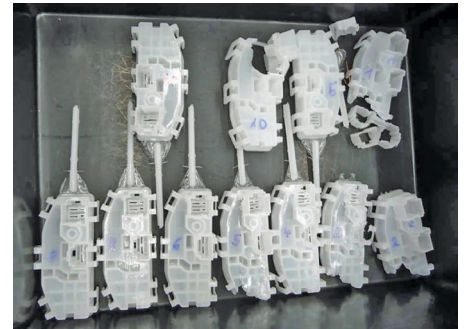


Figure 30: A progression of test parts (good shots on left) as injection molding parameters are dialed in.

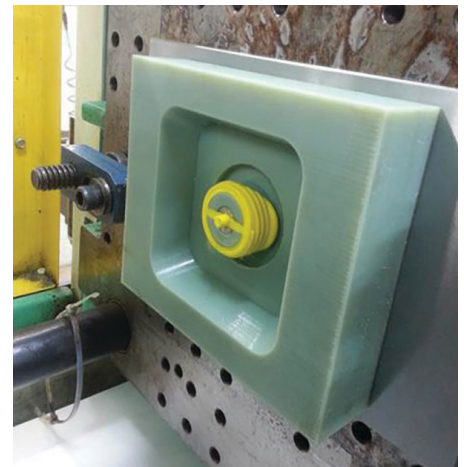


Figure 31: Example of shrinkage due to over cooling.

10. KEY PROCESS CONSIDERATIONS

The following table presents common obstacles for injection molding when using PolyJet molds along with recommended solutions.

Obstacle		Resolution					
		Thermoplastic Selection	Mold Design	Mold Components	Mold Preparation	Molding Parameters	Mold Temperature
Excessive Flash	Pronounced flash along parting lines.		✓			✓	
Part Quality	Defects in molded parts, such as short shots, knit lines, sink marks, surface imperfections or feature deformation.	✓	✓	✓	✓	✓	✓
Mold Life	Mold damage or wear yields fewer than expected parts.	✓	✓	✓		✓	✓
Part Ejection	Parts stick in mold, leading to mold damage.		✓		✓	✓	

Table 2: Common obstacles and resolutions.

10.1. Resolution details:

- Thermoplastic selection:
 - Try to increase the nozzle and barrel temperature to get better viscosity and material flow.
 - Use plastics with reasonable melt temperatures and good flow characteristics for improved part quality and tool life.
 - Increase the gate size; a larger gate reduces the pressure in the mold, improving material flow.
 - Avoid clear plastics.
- Mold design:
 - Increase draft angles (5° minimum) to ease ejection.
 - Add radii to sharp corners on small features to avoid shearing stresses.
 - Extend inserts beyond mold base for a compressive seal.
 - Use correct gate style and enlarge to minimize injection pressure.
 - Use sprue bushings to prevent nozzle contact with mold.
 - Use multi-piece molds for manual ejection of deep parts.
- Mold components
 - Add core pins for features with an aspect ratio of 3:1 or greater that may shear.
 - Use machined inserts for fine details or high-aspect ratio walls that may shear.
- Mold preparation:
 - Sand vertical walls to minimize shot pressure and ease ejection.
 - Sand cosmetic surfaces to improve appearance of molded parts.
- Molding parameters:
 - Lengthen cooling cycle for part solidification.
 - Decrease cooling cycle to reduce part shrinkage inside the mold.
 - Lower molding pressure to minimize flash and extend tool life.
 - Avoid flashing the tool to extend mold life.
 - Increase the nozzle temperature to get better viscosity and material flow.

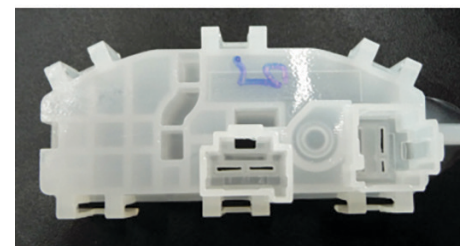
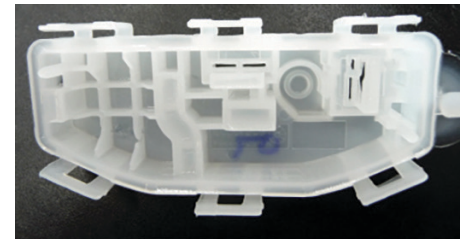


Figure 32: Injection molded polyethylene part (top and bottom views).



Figure 33: Cooling fixture for blowing air onto the PolyJet mold.

- Mold temperature:
 - Rest the mold between cycles and cool with compressed air to keep it at the target temperature of 50°C (120°F).

11. Tools And Supplies

11.1. Required items:

- Sand paper (180- to 340-grit)
- Tools and supplies common to injection molding
- Metal sprue bushing

11.2. Optional items:

- Drill press
- Milling machine
- Mold base
- Modular ejection system
- Steel backing plates
- Pins (core and alignment)

12. Recap – Critical Success Factors

12.1. Actions:

- Adjust mold design.
- Adjust molding parameters.
- Keep mold at target temperature of 50°C (120°F).
- Use thermoplastics with moderate melt temperatures and good flowability.

12.2. Eliminate obstacles:

- Use a PolyJet mold when the part size is reasonable.
- Start with conservative molding parameters and ease into those needed for a quality part.



Injection molded screw cap created with a PolyJet mold.

CONTACT

For questions about the information contained in this document,
contact Stratasys at www.stratasys.com/contact-us/contact-stratasys.



E info@stratasys.com / STRATASYS.COM

HEADQUARTERS

7665 Commerce Way, Eden Prairie, MN 55344
+1 800 801 6491 (US Toll Free)
+1 952 937 3000 (Intl)
+1 952 937 0070 (Fax)

2 Holtzman St., Science Park, PO Box 2496
Rehovot 76124, Israel
+972 74 745 4000
+972 74 745 5000 (Fax)

ISO 9001:2008 Certified

© 2016 Stratasys. All rights reserved. Stratasys, Stratasys signet, "The 3D Printing Solutions Company", PolyJet, Objet, Objet24, Objet30, Objet30 Pro, Object30 Prime, Objet Studio, Eden, Eden250, Eden260V, Eden260VS, Eden350, Eden350V, Eden500V, Connex, Connex1, Connex 2, Connex3, Objet260, Objet350, Objet500, Vero, VeroWhite, FullCure and Digital ABS are trademarks of Stratasys Ltd. and/or its subsidiaries or affiliates and may be registered in certain jurisdictions. All other trademarks belong to their respective owners. TAG_PJ_InjectionMolding_A4_0216a