



TECHNICAL APPLICATION GUIDE

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.



Injection molded impeller with PolyJet molds.

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 10 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.

PolyJet For Injection Molding

CONTENTS

1. OVERVIEW.....	3
1.1. Application	3
1.2. PolyJet is a best fit.....	3
1.3. Successful adopter traits.....	4
1.4. PolyJet adoption obstacles	4
1.5. Benefits	5
2. TRADITIONAL PROCESS OVERVIEW.....	5
2.1. Traditional injection molding process	5
2.2. PolyJet adjustments	6
3. INJECTION MOLDING MATERIALS	6
3.1. Recommended materials	7
4. MOLD DESIGN	8
4.1. Mold cavity	8
4.2. Mold components	10
4.3. Mounting	14
5. FILE PREPARATION	17
5.1. Orient STLs	18
5.2. Select surface finish	18
5.3. Set printing mode.....	18
6. MATERIALS AND PRINTERS	19
7. MOLD PREPARATION.....	19
7.1. Remove support material	19
7.2. Smooth surfaces	19
8. MOLD ASSEMBLY AND MOUNTING.....	20
8.1. Install core pins	20
8.2. Fit ejection system.....	21
8.3. Install sprue bushing	21
8.4. Install cooling line connections (optional)	21
8.5. Face mill mold inserts (mold-base option only)	21
8.6. Assemble molds and mount	22
9. INJECTION MOLDING.....	23
9.1. Clamping pressure	23
9.2. Release agent	24
9.3. Trial shots	24
9.4. Part molding	26
10. KEY PROCESS CONSIDERATIONS	27
10.1. Resolution details	27
11. TOOLS AND SUPPLIES	29
11.1. Required items	29
11.2. Optional items	29
12. RECAP – CRITICAL SUCCESS FACTORS	29
12.1. Actions	29
12.2. Eliminate obstacles	29

APPLICATION COMPATIBILITY

TECHNOLOGY	IDEA	DESIGN	PRODUCTION
FDM	1	1	2

(0 – N/A, 1 – Low, 5 – High)

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

PolyJet For Injection Molding

1. OVERVIEW

1.1. APPLICATION:

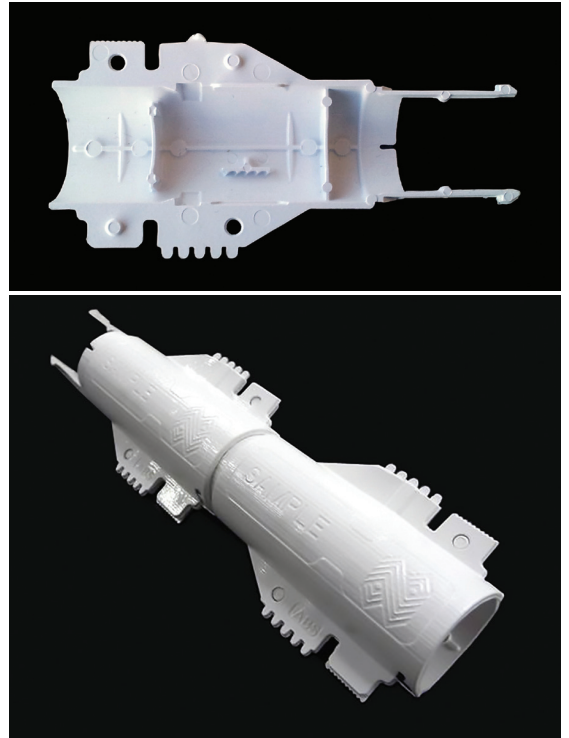
PolyJet 3D printing replaces machining of injection molds for prototyping in end-use plastics prior to investing in production tooling.

1.2. POLYJET IS A BEST FIT WHEN:

- Injection molding thermoplastics with reasonable characteristics.
 - Low to moderate melting points
= < 300°C (570°F)
 - Good flowability
 - Candidates:
= PE, PP, PS, ABS, TPE, PA,
POM, PC-ABS
= Glass-filled resins
- Quantities are small.
 - 5 - 100 molded parts
- Part size is small to medium.
 - < 165 cm³ (10 in³)
 - 50- to 80-ton injection molding machines
- Design and functionality confirmation is desired.



Injection molded impeller with PolyJet molds.



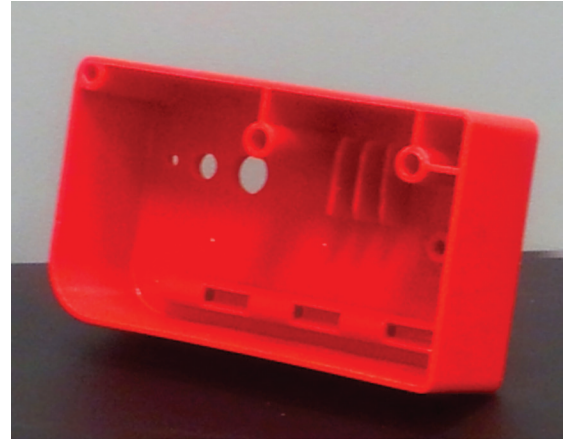
Molded medical device used to evaluate the PolyJet mold process.

PolyJet For Injection Molding

- 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 pre-production molds



Molded part with shut-offs and moderately deep draw.

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 large or overly complex parts	<ul style="list-style-type: none"> • Use for parts with reasonable molding parameters. • Use with molds that have little or no "action".

*Additional solutions may exist.

PolyJet For Injection Molding

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.



Spec resin provides functional testing of the living hinge.

2. TRADITIONAL PROCESS OVERVIEW

2.1. THE STEPS IN THE TRADITIONAL INJECTION MOLDING PROCESS ARE:

2.1.1. MAKE MOLD.

- CNC mill aluminum or tool steel.

2.1.2. SET UP MOLD.

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

PolyJet For Injection Molding

2.1.3. MOLD SAMPLES (“FIRST ARTICLES”).

- Injection mold sample parts.
- Inspect and review.
- Rework mold as needed.

2.1.4. MAKE PARTS.

- Injection mold parts.
- Trim flash and gate.

2.2. POLYJET ADJUSTMENTS

2.2.1. MAKE MOLD.

- Modify tool design.

2.2.2. ADJUST MOLDING PARAMETERS.

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.

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.

PolyJet For Injection Molding

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 polyamide (PA) 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. Other poor candidates include clear materials, which require highly polished cavities and precise temperature control.

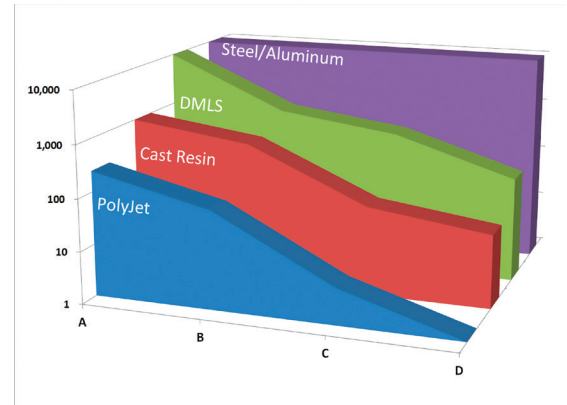


Figure 1: Tool life (number of molded parts) by resin class.

A	<ul style="list-style-type: none"> • Polyethylene (PE) • Polypropylene (PP) • Polystyrene (PS) • Acrylonitrile Butadiene Styrene (ABS) • Thermoplastic elastomer (TPE)
B	<ul style="list-style-type: none"> • Glass-filled Polypropylene (PP+G) • Polyamide (PA) • Acetal (Polyoxymethylene [POM]) • Polycarbonate-ABS blend (PC+ABS)
C	<ul style="list-style-type: none"> • Glass-filled Polyamide (PA+G) • Polycarbonate (PC) • Glass-filled Acetal (POM+G)
D	<ul style="list-style-type: none"> • Glass-filled Polycarbonate (PC+G) • Polyphenylene Oxide (PPO) • Polyphenylene Sulfide (PPS)

PolyJet For Injection Molding

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.

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.



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

PolyJet For Injection Molding

If using a mold 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 mold base's pockets (Figure 3). Optionally, the inserts may have the same depth as the pocket, but shims are needed to assemble the mold (see Section 8.5.).

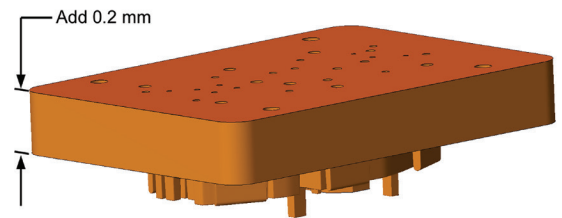


Figure 3: Extend back face (red) when using a mold base.

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, 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).

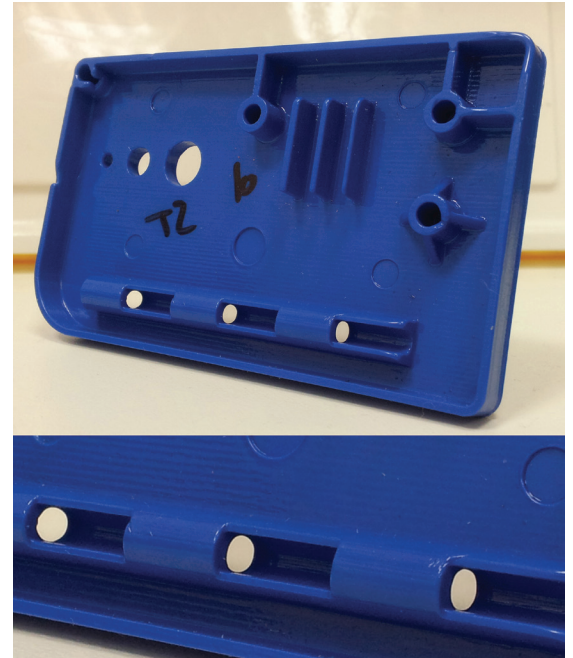


Figure 4: Shut-off for this hinge required no mold modification.

4.1.5. CORE PINS

Core pins can deflect in metal tools due to the pressure of the resin as it flows into the mold cavity. With a plastic tool, they can shear.

PolyJet For Injection Molding

For all core pins that have an aspect ratio greater than 3:1, (height : width) substitute steel pins that are press fit into the mold during assembly (see Section 8.1.). Create holes to receive these pins. Use a diameter that is 0.1 mm (0.004 in) larger than that of the pin.

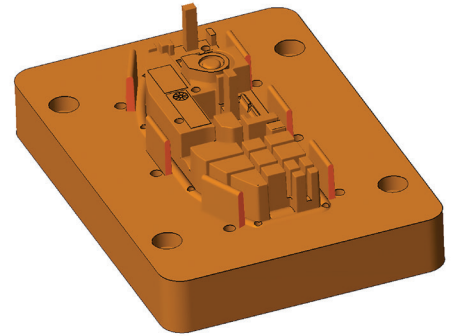


Figure 5: Inset shut-off faces (red).

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.

4.1.7. SHRINKAGE COMPENSATION

Before proceeding to Section 4.2., scale the core and cavity to compensate for the shrinkage of the thermoplastic during 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.

PolyJet For Injection Molding

- 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)
- Walls between 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.

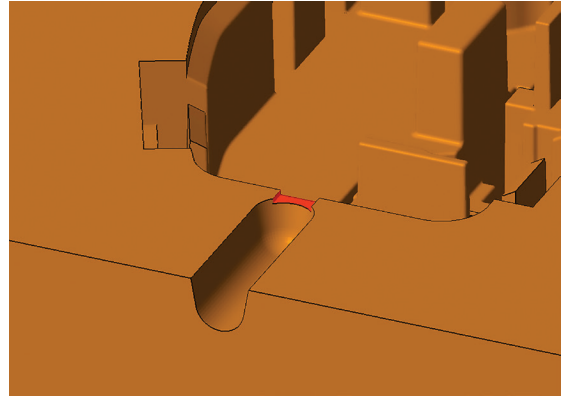


Figure 6: Enlarge all gates. Edge gate (shown) should be the thickness of the wall.

PolyJet For Injection Molding

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 mold base / steel plate (see Section 4.3.) or add a hole that will receive a standard sprue bushing. If using a bushing, undersize the hole by 0.2 - 0.3 mm (0.008 - 0.012 in) 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.

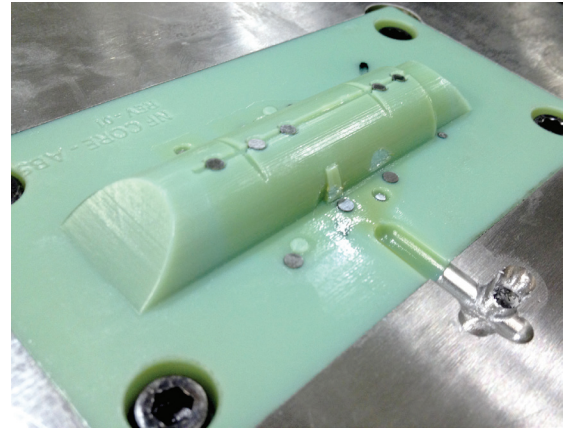


Figure 7: Runners do not require adjustment.

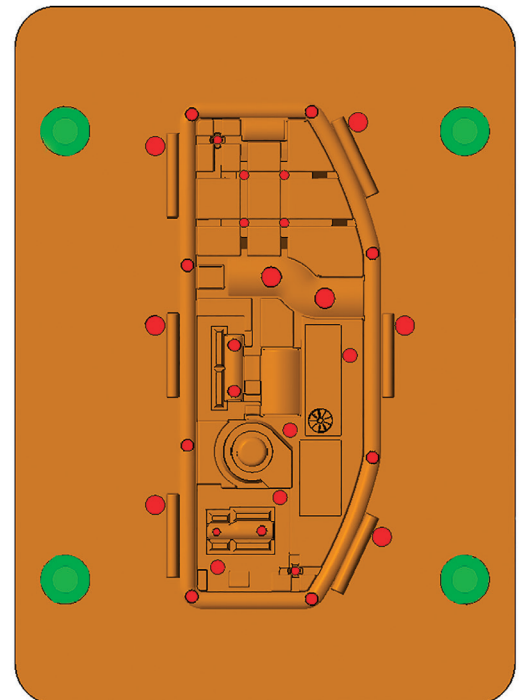


Figure 8: Add holes (red) for ejection system. Corner holes (green) are for attachment to the mold base.

PolyJet For Injection Molding

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 9: Cooling line connections on PolyJet core and cavity.

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 counter-bore 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.).
- Optionally, the cooling channels may be drilled rather than 3D printed (see Section 8.4.).

PolyJet For Injection Molding

4.3. MOUNTING

PolyJet molds may be mounted to steel plates or inserted (preferred) into mold bases (Figures 10 and 11). The steel plate option may include or exclude the ejection system (Figure 12). Table 1 lists the advantages and disadvantages of each approach.

- **Mold base**

PolyJet core and cavity inserts are seated in pockets within a standard metal mold base. Ideally, this mold 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).

- **Select a mold 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 mold 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**

METHOD	ADVANTAGES	DISADVANTAGES
Mold 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 build. • Mold base is reusable. 	<ul style="list-style-type: none"> • Initial investment in mold 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.
Steel Plate Without Ejection System	<ul style="list-style-type: none"> • Commercial • Success story • How it's used 	<ul style="list-style-type: none"> • Manual ejection limits part features. • Manual ejection may require more complex mold design. • Mold is not constrained.

Table 1: Mold mounting options.



Figure 10: PolyJet mold without ejection system attached to steel backing plates.

PolyJet For Injection Molding

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 mold 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 mold 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 no less than 20 - 25 mm (0.75 - 1.0 in) larger than the mold cavity on all sides. As shot size or shot pressure increase, allow additional

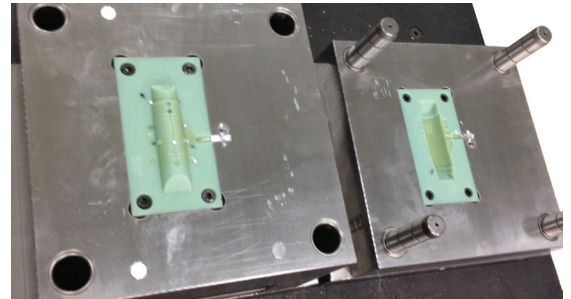


Figure 11: Mold base with PolyJet inserts.

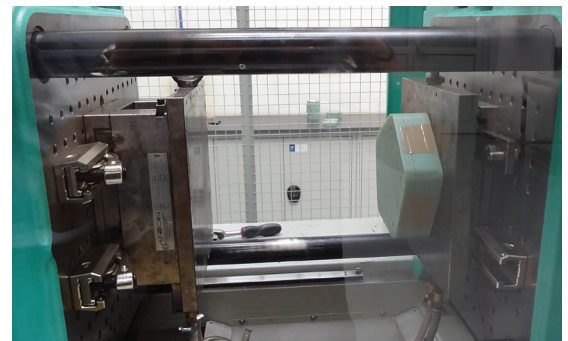


Figure 12: Printed mold for manual ejection on core side only.

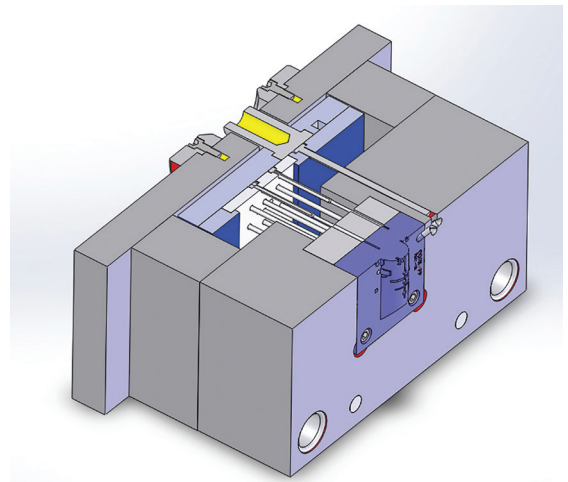


Figure 13: Section view of mold base with printed insert (light blue) and modular ejection system.

PolyJet For Injection Molding

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 no less than 20 - 25 mm (0.75 - 1.0 in) larger than the mold cavity on all sides (Figure 15). 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.

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

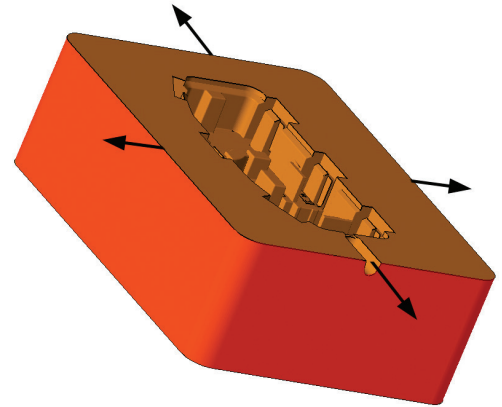


Figure 14: Extend side walls to provide machining stock.

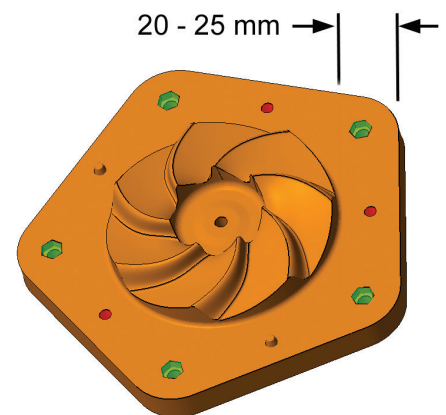


Figure 15: Manual ejection; plate-backed mold with alignment pin holes (red) and optional attachment bolt holes (green.)

PolyJet For Injection Molding

part, segment the mold into three or more pieces.

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 16 and 17). 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.

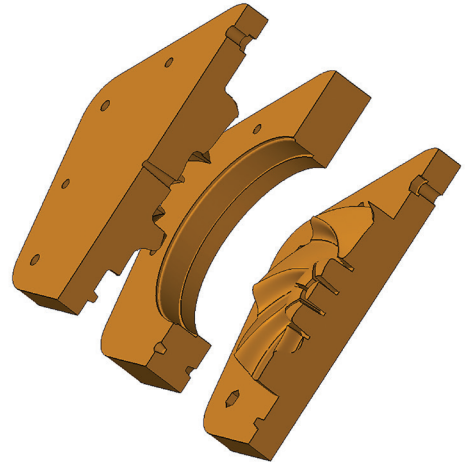


Figure 16: Three-piece PolyJet mold (section view) designed to make manual extraction easier.

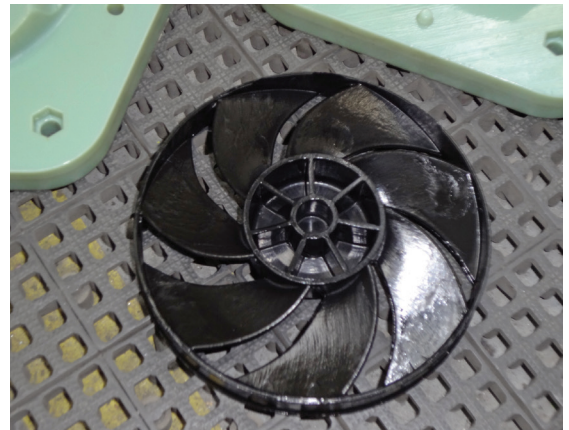


Figure 17: Injection molded impeller made with a three-piece PolyJet mold.

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 MODE.

When available, select “High Quality” for the printing mode, to produce the smoothest surfaces and minimize post-processing (Build > Printing Mode).

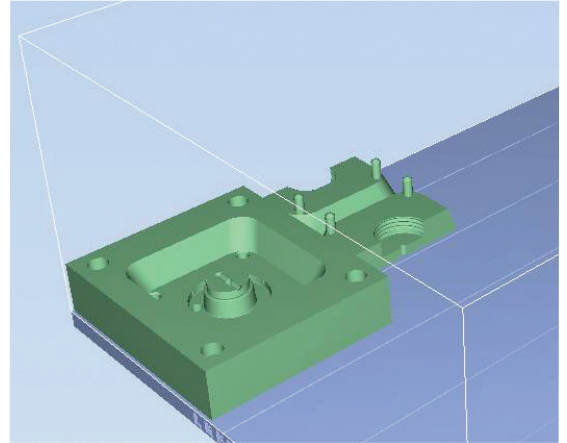


Figure 18: Orient so that molding surfaces will be glossy and have no support material.

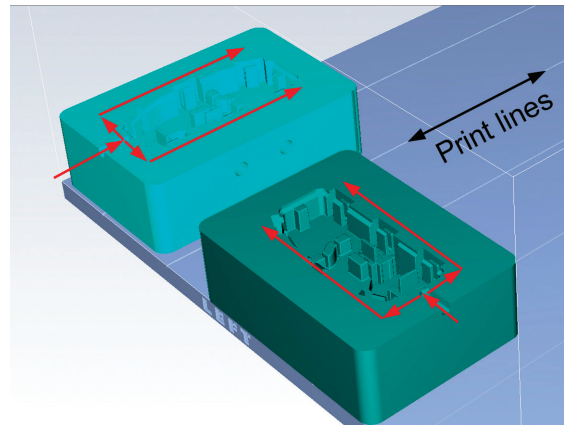


Figure 19: Orient so that resin flow (red) will be along print lines, as shown by the far core.

PolyJet For Injection Molding

6. MATERIALS AND PRINTERS

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

- Objet24™, Objet30™ and Objet30 Pro™: VeroWhite™
- Eden260V™, Eden350V™, Eden500V™: FullCure™ 720
- Objet260 Connex™, Objet350 Connex™, Objet500 Connex™: Digital ABS™ (RGD5160)

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

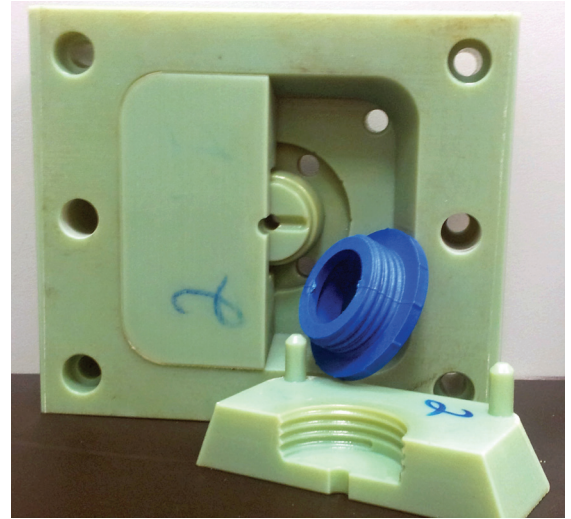


Figure 20: Digital ABS tool with injection molded part (PA 6/6 with 20% glass fiber).



Figure 21: Sand mold surfaces for part extraction and part appearance.

7. MOLD PREPARATION

7.1. REMOVE SUPPORT MATERIAL.

For rectangular stand-alone molds and mold 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 (Figure 21).

PolyJet For Injection Molding

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

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.

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.

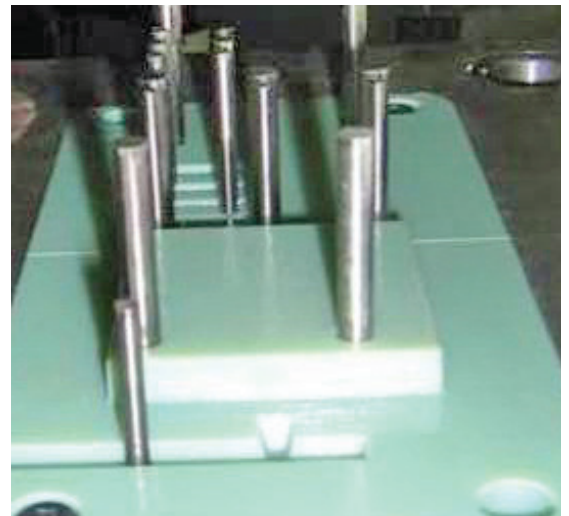


Figure 22: Ejector pins must be snug but able to move freely.

PolyJet For Injection Molding

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).

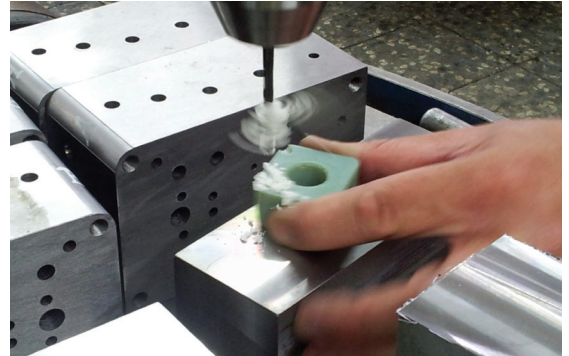


Figure 23: Ream ejector holes with a drill press or milling machine.

8.3. INSTALL SPRUE BUSHING.

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

8.4. INSTALL COOLING LINE CONNECTIONS (OPTIONAL).

If using cooling lines, attach connection nipples to the mold inserts. If not added in Section 4.2.5., drill a counter-bore using an 8.8 mm (0.35 in) diameter and 10 - 12 mm (0.4 - 0.5 in) depth. Tap the counter-bore to the thread specification for the connection nipple.

8.5. FACE - MILL MOLD INSERTS (MOLD-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 mold base to confirm that both are snugly seated.

Once seated, confirm that the inserts extend beyond the face of the mold base by 0.2 mm (0.008 in) (Figure 24). If the inserts sit too high,

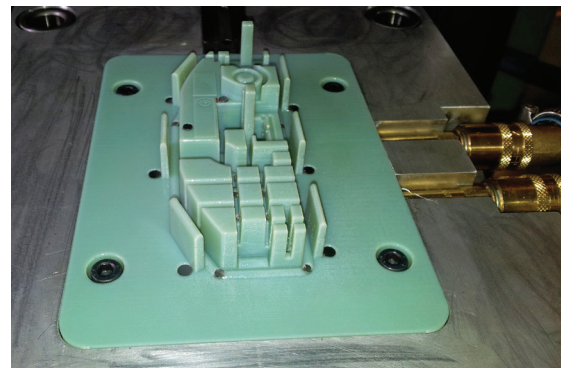


Figure 24: Confirm that the insert is 0.2 mm (0.008 in) above the surface of the mold base. Adjust if necessary.

PolyJet For Injection Molding

mill the back faces. If they sit too low, insert shim stock in the bottom of the mold base pockets to raise them to the desired height.

8.6. ASSEMBLE MOLDS AND MOUNT.

8.6.1. MOLD BASE

Place PolyJet inserts in the pockets of the mold base and attach with bolts. Do not over-tighten attachment bolts as this may cause the inserts to crack.

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 25).

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 26).



Figure 25: Attach core and cavity to steel plates and modular ejection system.



Figure 26: Assemble mold and attach to steel plate with double-sided tape.

PolyJet For Injection Molding

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 27 and 28). 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 29).

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

9.1. CLAMPING PRESSURE

Use the standard clamping force (injection pressure \times total projected area or total surface area \times 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

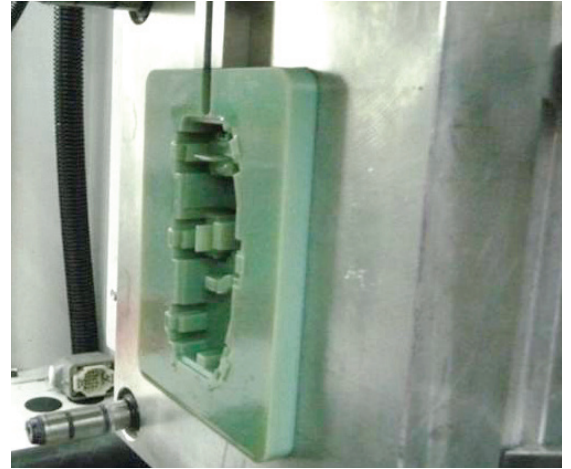


Figure 27: Mold base with PolyJet cavity on the press.



Figure 28: Mold base with the PolyJet core.

PolyJet For Injection Molding

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 non-silicone mold release agent to the mold cavity.

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).

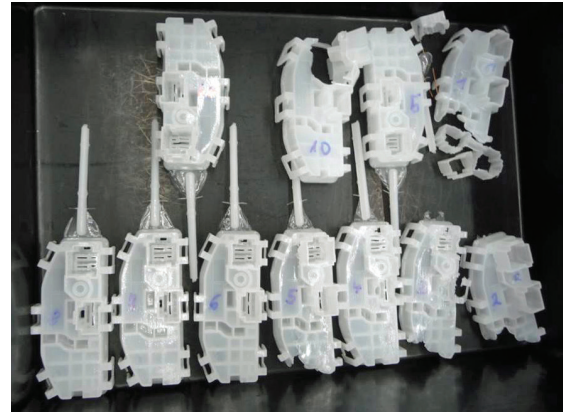


Figure 29: A progression of test parts (good shots on left) as injection molding parameters are dialed in.

PolyJet For Injection Molding

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.

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.

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.

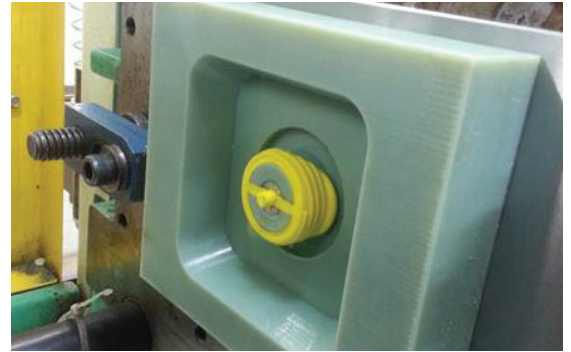


Figure 30: Example of shrinkage due to over cooling.

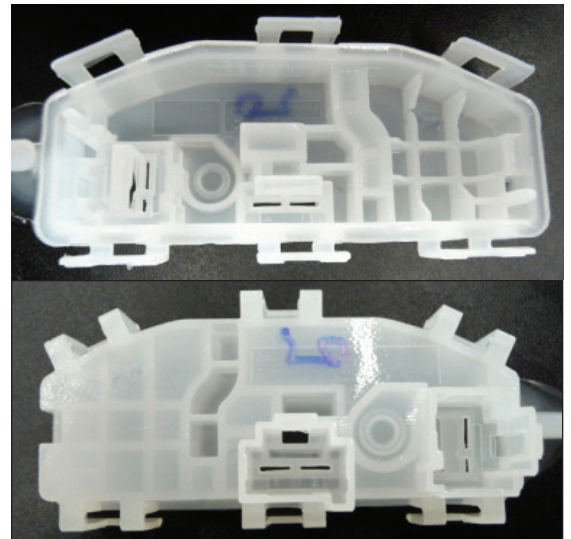


Figure 31: Injection molded polyethylene part (top and bottom views).

PolyJet For Injection Molding

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 31).

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 32).



Figure 32: Cooling fixture for blowing air onto the PolyJet mold.

PolyJet For Injection Molding

10. INJECTION MOLDING

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	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:
 - Use plastics with reasonable melt temperatures and good flow characteristics for improved part quality and tool life.
 - 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.

PolyJet For Injection Molding

- 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.
 - Lower molding temperatures to extend tool life.
 - Lower molding pressure to minimize flash and extend tool life.
 - Avoid flashing the tool to extend mold life.
- Mold temperature:
 - Rest the mold between cycles and cool with compressed air to keep it at the target temperature of 50°C (120°F).

PolyJet For Injection Molding

11. TOOLS AND SUPPLIES

11.1. REQUIRED ITEMS:

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

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 and the design is not overly complex.
- Start with conservative molding parameters and ease into those needed for a quality part.



Injection molded screw cap created with a PolyJet mold.

CONTACT:

To obtain more information on this application, contact:

Stratasys Application Engineering

www.stratasys.com/solutions-applications

info@stratasys.com

STRATASYS.COM

HEADQUARTERS

7665 Commerce Way, Eden Prairie, MN 55344

+1 888 480 3548 (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)

stratasys®

THE 3D PRINTING SOLUTIONS COMPANY

ISO 9001:2008 Certified

© 2014 Stratasys Ltd. All rights reserved. Stratasys, Stratasys nautilus logo, FDM and Objet are registered trademarks and PolyJet, Eden260V, Eden350V, Eden500V, FullCure, Objet Studio, Objet24, Objet30, Objet30 Pro, WaveWash, VeroGray, VeroWhite, Digital ABS, Objet 260 Connex, Objet350 Connex, and Objet500 Connex are trademarks of Stratasys Ltd. and / or its subsidiaries or affiliates and may be registered in certain jurisdictions. All other trademarks are the property of their respective owners, and Stratasys assumes no responsibility with regard to the selection, performance, or use of these non-Stratasys products. Product specifications subject to change without notice. Printed in the USA. . TAG_PJ_InjectionMolding_EN_0915

The information contained herein is for general reference purposes only and may not be suitable for your situation. As such, Stratasys does not warranty this information. For assistance concerning your specific application, consult a Stratasys application engineer. To ensure user safety, Stratasys recommends reading, understanding, and adhering to the safety and usage directions for all Stratasys and other manufacturers' equipment and products. In addition, when using products like paints, solvents, epoxies, Stratasys recommends that users perform a product test on a sample part or a non-critical area of the final part to determine product suitability and prevent part damage.

For more information about Stratasys systems, materials and applications, call 888.480.3548 or visit www.stratasys.com