WP-340

Feb. 23, 1995

Injection Molding at the MIT Artificial Intelligence Lab

Michael Binnard
binn@ai.mit.edu

MIT Artificial Intelligence Lab
545 Technology Square
Room NE43-920
Cambridge, MA 02139
voice: (617) 253-0997
fax: (617) 253-0039

Abstract

This paper describes the injection molding equipment at the MIT Artificial Intelligence Lab and how to use it. Topics covered include mold design, insert molding, safety, and material properties.

Keywords: Injection molding, manufacturing, machine shop

Contents

1 Why Injection Molding? **3**

2 Mold Design **3**
2.1 General 3
2.2 Required Features 5
2.3 Inserts and Cores 6
2.4 Ejector Pins 7

3 The Molding Process **7**
3.1 The Quickshooter Molding Machine 7
3.2 Safety Concerns 8
3.3 Typical Process 9

4 Material Selection and Properties **10**
4.1 Delrin 500 10
4.2 Delrin 500AF 11
4.3 Delrin 577 11
4.4 Fiber Reinforced Nylon 11
4.5 Switching Resins 12

5 References **12**

6 Sources for Plastic Resins **12**

7 Glossary **12**

1 Why Injection Molding?

Injection molding is normally used in high volume projects because of its economic advantages. However, there are several reasons why it is a good process for producing small robot parts, even in small quantities. Weight is a primary concern in robotics applications. Injection molding allows the use of lightweight materials with a wide range of properties. This process also permits more complex geometry than can be achieved with machining. Machining small plastic parts is difficult because the fixturing forces required to hold the part in place usually cause deformation, which limits accuracy.

Some other reasons to prefer injection molding to machining are:

- Many molds can be machined in a single clamping operation, which increases accuracy, and decreases machining time.
- The most common screw size you will is use 1/4-20 instead of 2-56.
- Your colleagues will regard you as a more sophisticated engineer.

2 Mold Design

Obviously, the quality of the part you produce is determined by the mold. Designing parts that require simple molds takes some getting used to. The possible geometry for molded parts is quite different from machined parts. For example, molded parts often have external radii while machined parts have internal radii. The reason for the difference is because the radii are a result of using round cutting tools, and the machined mold is the negative-space of the molded part. A more subtle characteristic of molded parts is that the radii caused by the cutting tools have axes perpendicular to the parting plane.

2.1 General

Molded parts should be designed with consistent cross-sections whenever possible. Figure 1 shows examples of poor and good designs. Because features with different thicknesses cool (and shrink) at different rates they can cause voids, sinks, and warping. When different thicknesses are required, use gradual transitions, as shown in the rightmost part.

2.1.1 Draft

Draft is a slight taper in the mold cavity walls which makes ejecting parts easier (see Figure 2). Draft is not required for small molded parts; I have had acceptable results without it on most parts. However, draft will make part ejection easier, especially when the part has internal features. Since tapered endmills are expensive, I tend to use no draft in the molds, and include ejector pins or screws on difficult-to-eject parts.

2.1.2 Surface finish and tolerances

Injection molded parts do an amazing job of picking up the surface finish of the mold surfaces. If your mold cavity has tool marks, scratches, or other imperfections, the

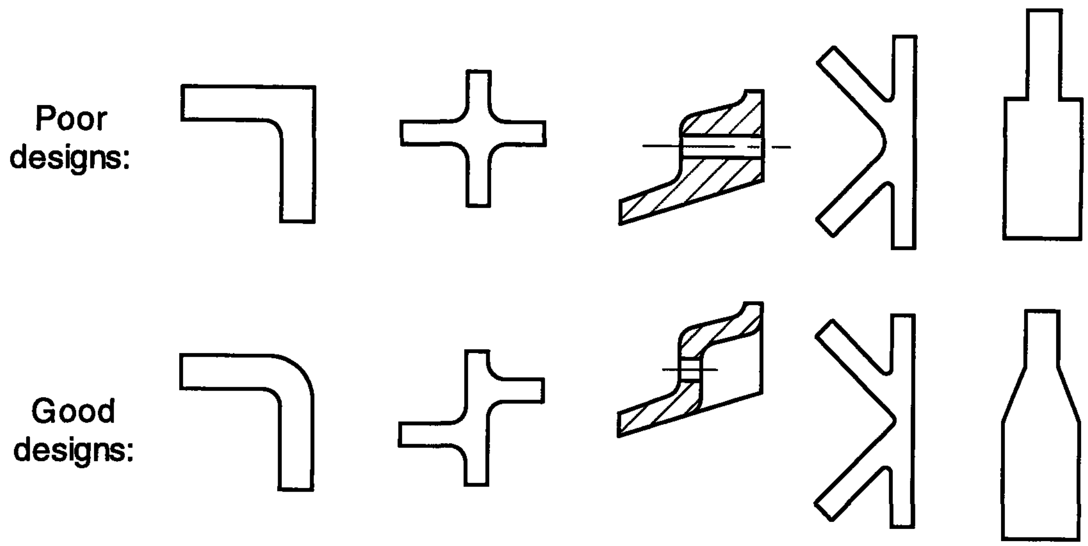

Figure 1: Molded parts should be designed with consistent cross-sections whenever possible. When different thicknesses are required, use gradual transitions, as shown in the rightmost part. [Kalpakjian 92]

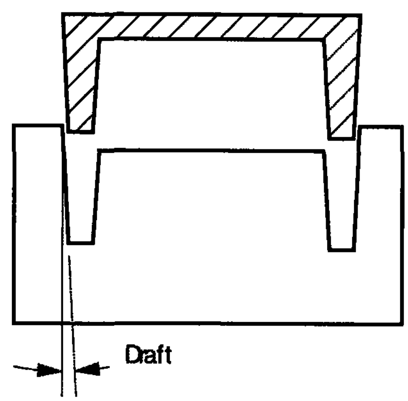

Figure 2: Draft makes ejecting parts easier

defects will be visible on the molded part. How noticeable the marks are depends on the plastic being molded; reinforced plastics have a coarser texture which hides minor flaws.

Accurate mold machining is important because it determines how well the mold parts fit together. For molds where the cavity is in multiple parts, proper alignment of the edges is important for part aesthetics.

2.1.3 Shrinkage

All plastic parts shrink as the plastic cools. The amount of shrinkage depends on the specific geometry, material, and molding conditions. Typically, the external dimensions of finished parts shrink approximately 2%. Internal features, where cores in the mold resist the shrinking, will shrink about 1.5%.

Besides dimensional uncertainty, another effect of shrinkage is that the parts grip onto internal mold features, like pins, cores, and internal corners. It is important to design the mold with this in mind. Otherwise, you can end up with a molded part that cannot be removed from the mold. Male mold halves must be provided with some part-ejection mechanism. (see section 2.4)

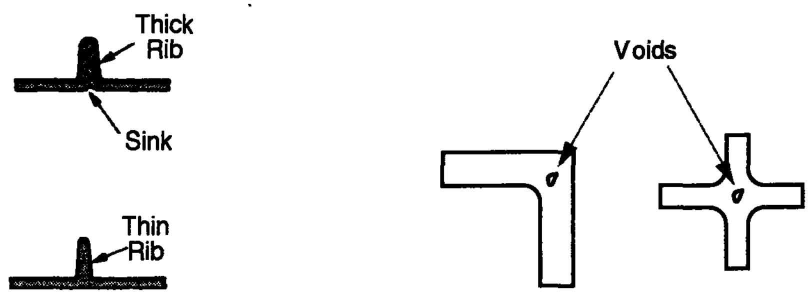

Figure 3: Uneven shrinkage causes sinks and voids. [Kalpakjian 92]

Shrinkage problems are exacerbated on parts with uneven thicknesses. As shown in Figure 3, different cooling rates cause sinks, voids, and other defects. The reason this happens is that when the thin sections freeze, the amount of material in the thicker section is fixed. When this plastic shrinks, it no longer completely fills the mold cavity. If the entire part freezes simultaneously, it can continue to draw molten plastic from the molding machine as the part volume shrinks.

2.1.4 Warping

Warping is a common problem with injection molded parts. Like many other defects, it is primarily caused by shrinkage. Warping can be avoided by designing parts with consistent cross-sections, as shown in Figure 4. Proper mold designs, which ensure the part cools in the proper manner are also helpful. For example, if your part has one thick area, place the sprue nearby so the molding machine can continue to supply plastic as the thick section cools and shrinks.

2.1.5 Gapiosis

Any mold made for this type of operation will indubitably have some small gaps between the mold halves. Excessively large gaps, or poorly-fitting inserts and cores will result in undesirable amounts of flash. Gaps of a few mils will not cause significant problems, however. The reason is that in a narrow passage, the plastic freezes immediately, preventing flash from developing. In other words, the molten plastic seals the gaps. Another advantage of small gaps in the mold is that they allow air to escape. It is unnecessary to machine vents in the molds.

2.2 Required Features

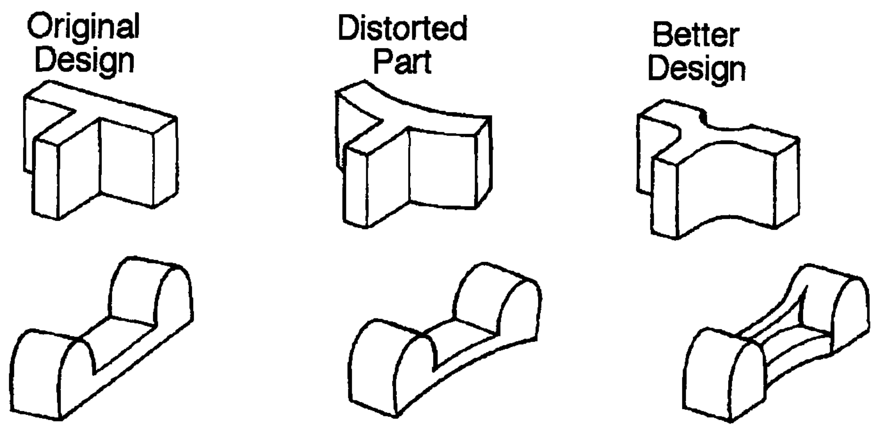

Figure 4: Uneven thicknesses cause parts to warp. Proper design can mitigate this problem. [Kalpakjian 92]

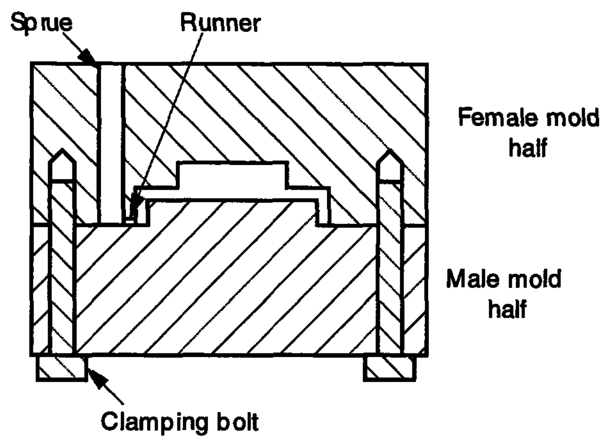

Figure 5: Required mold features

2.2.1 Clamping

One vital element of mold design is the clamping mechanism. The plastic is injected at a relatively high pressure, which tries to force the mold halves apart. Normally, a couple of 1/4-20 screws are all that is required to hold the mold together. It is important to remember that the required clamping force is the injection pressure times the projected part area. That means it is advantageous to have a minimum projected area on the parting surface. Usually, however, the most logical way to make the mold is with the maximum area at the parting surface. When this is the case, more clamping bolts are required.

It is a good idea to have the heads of the clamping bolts on the opposite side of the mold as the sprue, as shown in Figure 5. This will keep the bolt-heads from interfering with the injection nozzle. The easy way to accomplish this is to put clearance holes for the bolts in the mold have without the sprue, and tapped holes in the mold half that does have the sprue. This procedure will ensure that the bolts have their heads on the bottom of the mold.

2.2.2 Sprue and runner

The sprue is the place where the molten plastic is injected into the mold. Usually, it is just a hole drilled through one half of the mold. The sprue is connected to the mold cavity by a short, narrow channel called the runner. The end of the runner, where the plastic enters the mold cavity is called the gate.

For our set-up I usually use a 3/16 diameter sprue. Difficult to mold parts work better with a larger sprue; on small parts with relatively thick cross-sections, a smaller sprue will suffice. Sprue diameter seems to be the most important factor in determining the maximum injection time. If you are producing lots of short-shots because you can't get the plastic into the mold fast enough, a bigger sprue will usually fix the problem.

For runners, I usually mill a channel 1/16" deep with a 1/8" ball-endmill. This half-round cross-sectional shape is a good compromise between the ideal round runner and the easy to machine rectangular cross-section.

2.3 Inserts and Cores

One attractive advantage of injection molding is that parts can be molded with holes and internal features, eliminating post-machining requirements. For more exotic parts, bushings and shafts can be insert-molded in the parts, reducing assembly tasks.

2.3.1 Molding holes

It is relatively easy to mold parts with holes. Inserting dowel pins or cores made from round stock is all that is required. After the part is finished, the pin can be pressed out with the arbor press. The hole produced will be slightly undersized because of shrinkage. Usually, the shrinkage is just right for a press-fit of something, like ball bearings, with the same diameter as the core.

The pin should protrude into both mold halves to ensure that the hole goes all the way through. For blind holes, it is important to make sure the pin is securely held in place, otherwise the pressure of the molten plastic will force the pin outwards, and you will end up with a hole that is too shallow.

2.3.2 Inserting bushings and pins

For parts that require bearing surfaces, I have had good success insert-molding sintered bronze bushings into plastic parts. To accomplish this task, you should make a mold that can hold a dowel pin that matches the bushing internal diameter. This pin will hold the bushing in place during the molding process. When the part is ejected, the pin will slide out, and you'll have a nice part with an accurately positioned and aligned bushing. The bushings can be pressed out with the arbor press if necessary, but can support moderate axial loads with no retaining features. Obviously, situations with large axial forces require retaining rings or shoulders to keep the bushings in place.

To cast permanent pins or shafts in place, you can use the same method as for molding holes, but without removing the pin from the finished part. Putting grooves in the pin will prevent if from being pulled out by axial forces. To resist torsional loads, you can put flats or spot-drills on the shaft. The only trouble with these techniques is that they make it difficult or impossible to remove the pin from a molded part;

if you are prone to short-shots and other molding accidents, you may waste a lot of value-added inserts.

2.4 Ejector Pins

One issue that is easy to overlook is ejecting the finished part from the mold. Simple parts, with no internal corners, will shrink away from the mold halves, and can be removed from the mold easily. Parts with internal features, however, will shrink onto the male mold half, requiring an ejection mechanism.

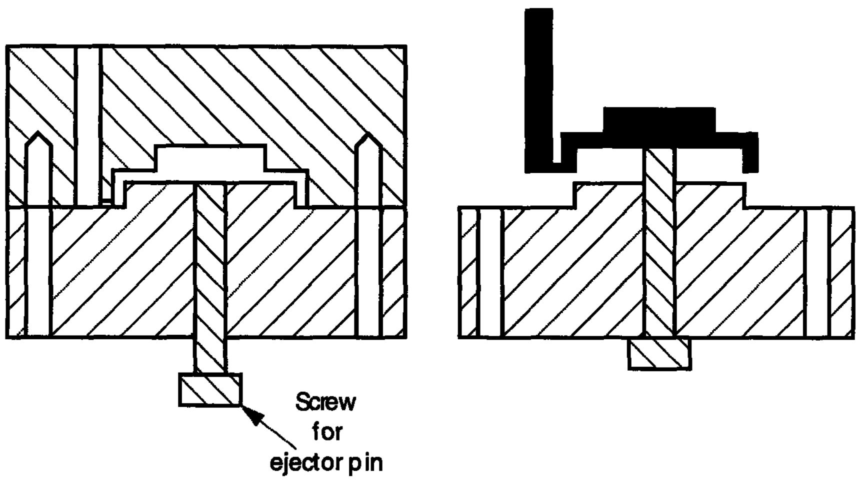

Figure 6: Using screws for ejector pins. This is the simplest way to eject parts.

The simplest way to eject parts is to use screws as ejector pins (see Figure 6). Drill and tap a hole through the male mold cavity. Before molding, put a screw in the hole so that the end of the screw is flush with the bottom of the mold cavity. The end of the screw can be sanded or machined flat to improve part aesthetics. One the part is molded, it can be ejected by screwing in the bolt. The advantage of this approach is that it is easy to make the molds. The disadvantage is it can be tedious to get the screw properly positioned before molding, and it can make a bit of a mess on the part surface.

Machined, cylindrical ejector pins will produce better-looking parts. The drawback of this approach, however, is that the mold must have features which hold the pins in the proper position; injection pressure will want to force the ejectors out. Shouldered ejector pins (see Figure 7) are an ideal solution. The top of the pin can be machined flush with the mold surface, minimizing the ugliness of your part. Molding pressure will force the pin down against the shoulder. Once the mold is opened, the part can be popped out quickly by hitting the ejectors on the table (or with a hammer); ejecting a part with screws is slower. The drawback of this style is that the mold and pins are more difficult to machine.

A third option is to use a pin to hold ejectors in the proper location (see Figure 8). This method requires less machining than shouldered ejectors, but is slightly less convenient to use. It also allows rapid part ejection.

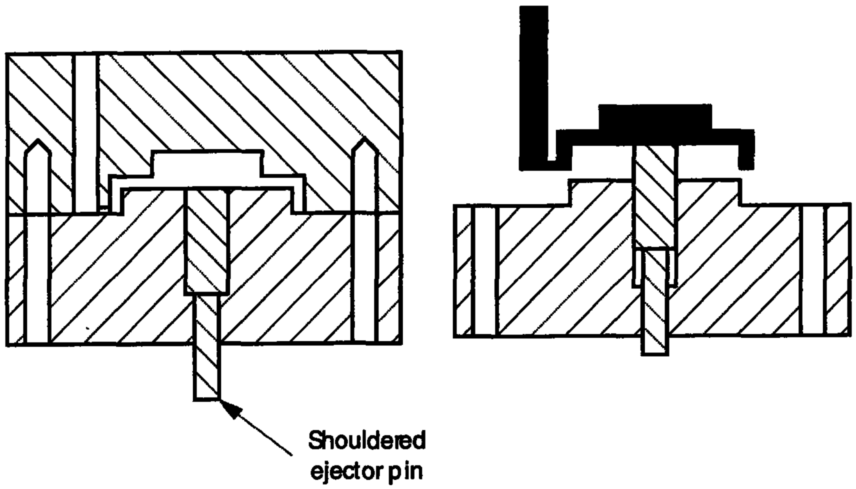

Figure 7: Machining ejector pins with shoulders can produce better results than using screws.

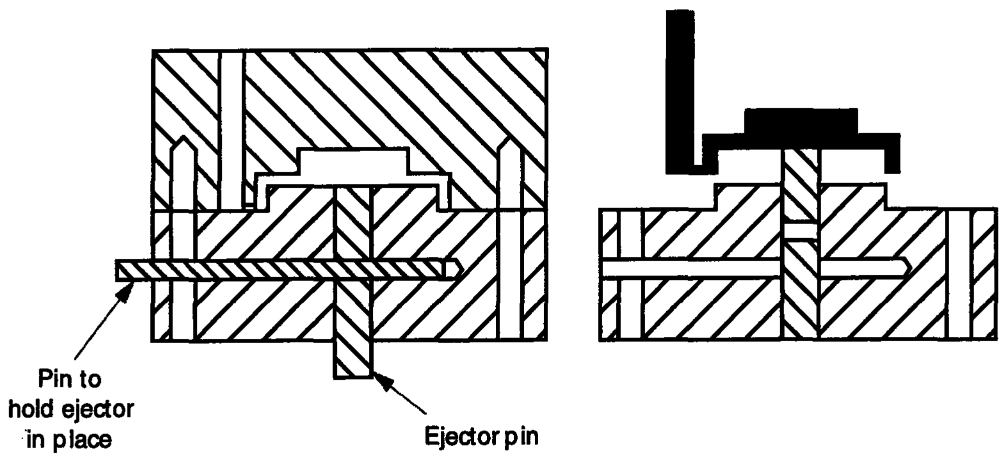

Figure 8: Using a pin to hold ejectors in place.

3 The Molding Process

3.1 The Quickshooter Molding Machine

The Quickshooter molding machine is shown schematically in Figure 9. Its operation is straightforward. Plastic pellets are introduced to the melting cavity through the fill spout. The cavity is surrounded by a heating coil which melts the plastic. When the drill press is lowered, the nozzle contacts the upper surface of the mold. As the quill is lowered further, the plunger is forced down into the melting cavity, forcing plastic out through the nozzle.

There are three aspects to this operation which can be difficult. First is getting the plastic into the fill spout. Because the spout is hot, pellets will melt inside the spout, stick there, and block further pellets from entering. The second problem is that the piston bushing does not form a perfect seal with the internal wall of the melting cavity. As a result, excessive injection pressure will force plastic around the bushing, where it hardens around the plunger. When this happens it can become difficult or impossible to fully retract the plunger. Section 3.3 has more details on this problem.

The third difficulty in molding is applying the correct amount of force to the drill press quill, and lowering the quill fast enough. As soon as the nozzle contacts the cold mold, the plastic will begin to cool. It is important to get the plastic injected before any of it freezes.

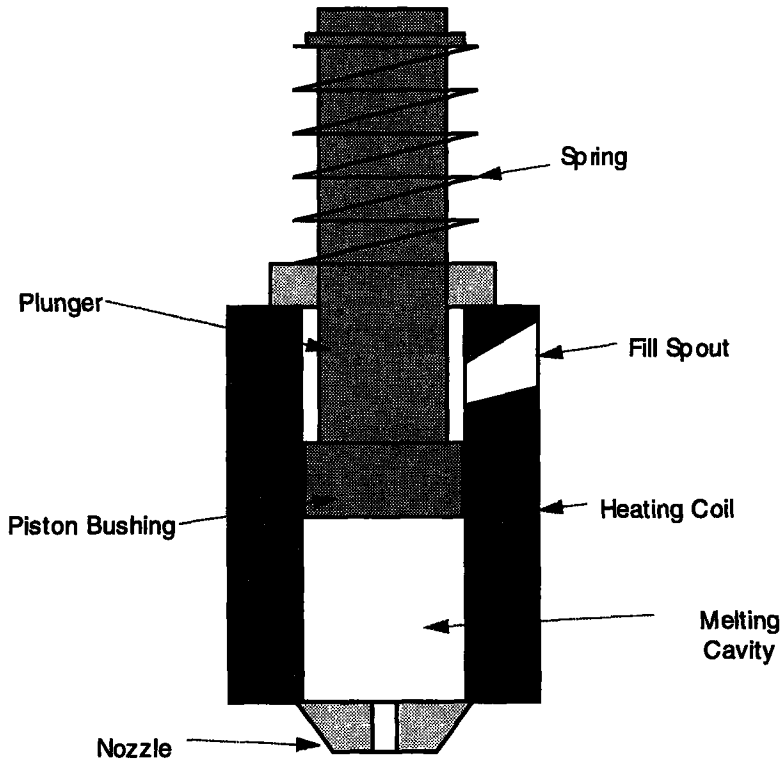

Figure 9: Schematic of the Quickshooter Molding Machine.

3.2 Safety Concerns

3.2.1 Burns

Obviously, an important safety issue with injection molding is burning yourself. Every part of the molding machine will heat up to the molding temperature, usually around 400 or 500 degrees. It's easy to get burned setting the thermostat or putting the machine in the drill press. A little common sense and caution should be sufficient to avoid this hazard.

A slightly more insidious way to get burned is with the molten plastic itself. Drool, flash, and other bits of molten plastic are very hot, and will stick to your skin as they burns you. This "napalm effect" can be frustrating: it hurts a lot, and the plastic won't come off your skin until it has cooled. By then, you will have a nasty burn.

3.2.2 Fumes

But by far the most dangerous aspect of injection molding is the fumes released by the melting plastic. All polymers, when melted, will release some fumes. If the plastic is overheated, either with temperature or time, the polymer may degrade. This degradation can create large quantities of noxious fumes. These toxic gases smell bad, will make your sinuses feel funny, cause nausea, and can cause reproductive damage. It helps to keep in mind that your future children are on the line when you deal with this process.

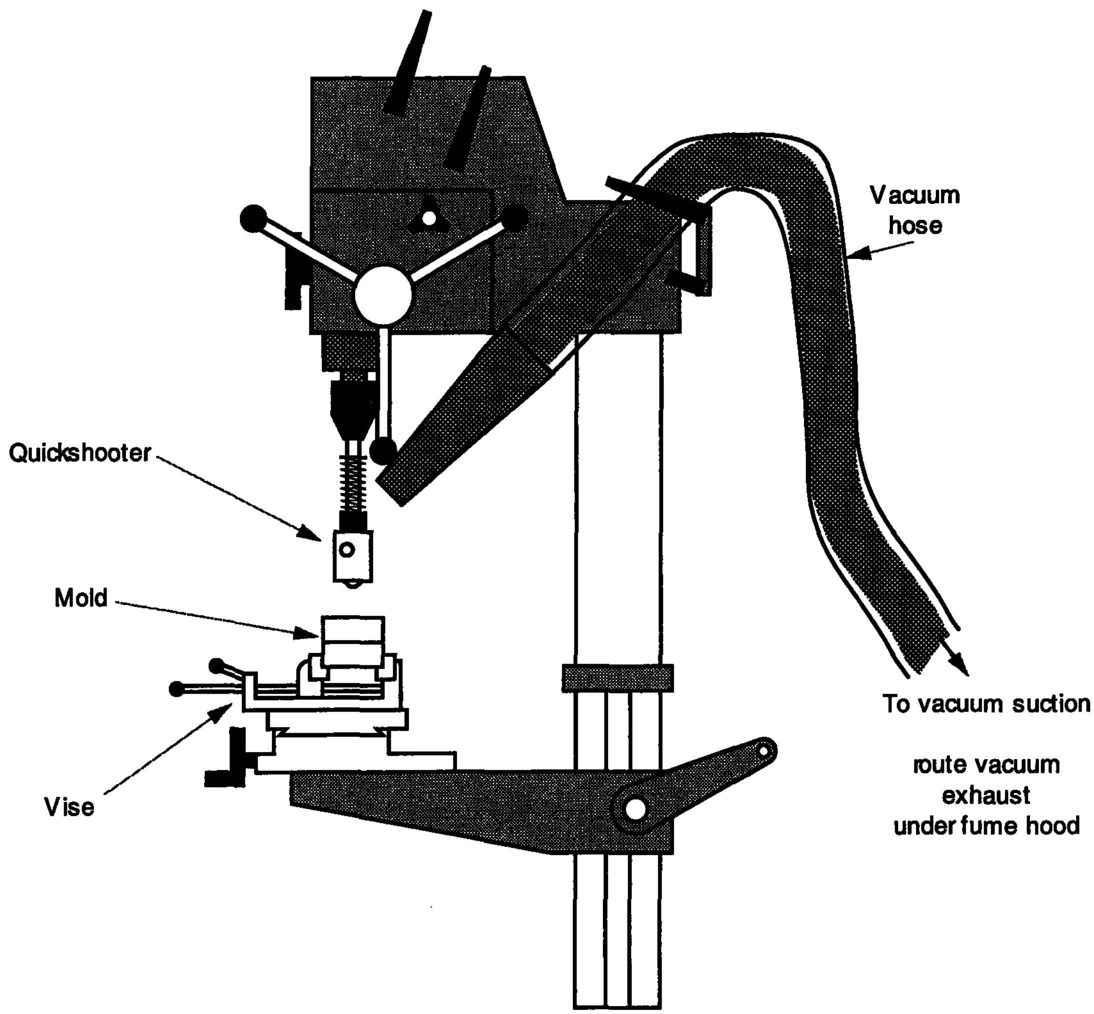

Figure 10: Using the vacuum cleaner to suck up toxic fumes. Use a large tie wrap (available from Ron Wiken) to hold the nozzle near the loading port of the Quickshooter.

Fortunately, it is relatively easy to avoid seriously poisoning yourself. If you don't overheat the plastic, the amount of gas released is relatively small; if you're only making a couple of parts, you will probably not notice it. Keep in mind, though, if you can smell it, it's too much. The way to keep the air clean is to use the vacuum cleaner; as an exhaust system, as shown in Figure 10. By attaching the vacuum nozzle to the drill press, it will suck up all the gases coming out of the molding machine. A couple of vacuum attachments are able to funnel the output into the fume hood, where it gets sucked up and sent out into the lovely Cambridge skies. Ensure that the fume hood exhaust is turned on.

3.2.3 Reinforcements

The materials used in reinforced plastic are potentially hazardous. The most serious threat is from carbon fiber and glass reinforced resins. Dust from these materials can cause lung cancer, just like fiberglass and asbestos. Be careful when post-machining or deburring reinforced parts. If you cut or drill the parts, use plenty of cutting fluid to prevent dust becoming airborne. For sanding, use only the vacuum-equipped belt sander.

3.3 Typical Process

This section describes what typically happens during the molding process. Keep in mind, though, that this is only an example. I've tried to include all the potential obstacles to a smooth molding run, but each case is different. Some molds are easier to fill than others, and every polymer has different characteristics. Another source of variation is that the process works better some days than others.

3.3.1 Loading and melting

The injection molding process begins by putting plastic pellets into the molding machine and waiting for them to melt. The best way to get the plastic in the machine is to use a piece of sheet metal bent into a v-shaped "funnel" A screwdriver is handy for coercing the pellets to actually enter the molding machine. The opening where you put the pellets in can get clogged up by melting plastic; it is prudent to keep this area as clean as possible.

After you have filled the plastic chamber with pellets, you need to pre-pack the shot. This is a simple operation; place something without a hole in it under the injection nozzle, and crank down the quill to compress the plastic. Only a moderate amount of force is needed on the quill; you don't have to squish it to death. Once the plastic has been packed down, you can add some more pellets and repeat the process. Three iterations of loading and packing gets the injection chamber pretty full.

One especially annoying problem is the tendency for plastic to get around the piston bushing during the injection process. This plastic will harden just above the bushing, and prevent the piston from being fully retracted. The result is that the loading opening does not fully open, hindering plastic loading. The solution is to let the machine hang out for a while with the piston depressed (i.e., in the molding position). After the end of the piston heats up, the plastic around the bushing will soften, and a bit of cajoling and Irish persuasion can clean things up.

The melting process can take up to 15 minutes, depending on the plastic being used. For plastics that drool a lot, like nylon, it is often helpful to lower the quill until the molding machine's nozzle is touching the vise or part of the mold. Engaging the quill power feed will lock it in place. This prevents excessive drool while the pellets melt. For the more noxious plastics, however, such as Delrin, it is more important to have the molding machine near the vacuum nozzle and to live with the drool; it's better to be messy than ill.

3.3.2 Molding

Before molding a part, you should squirt out a little bit of plastic into the air. This gets rid of the colder plastic that is near the nozzle. If the nozzle has been in contact with a metal surface, or if there is drool which has cooled, the chilly plastic will plug up your sprue and cause a short-shot.

Next, the mold must be positioned in the vise with the sprue directly under the nozzle. Once everything is lined up, wipe off any drool with a screwdriver and inject the plastic. It's important to get the plastic into the mold quickly; you only have a second or two before it solidifies. It can take some substantial pressure on the quill to fully fill the mold; how much force varies a lot depending on the mold geometry and plastic being used.

Once all the plastic is in the mold, it is important to keep pressure on the quill for a few seconds longer. Keeping the pressure on while the part freezes is important to get complete part density. If you let up while the plastic is still fluid, your part is likely to contain voids and sinks.

3.3.3 Ejecting the part

The final step in the process is ejecting the part from the mold. Often, with this machine part ejection is the slowest step in the process because the mold halves must be unscrewed. Section 2.4 Describes various methods for ejecting parts. Be sure to plan for this step in advance; it is easy to make a mold where it will be impossible to remove the finished part.

It is often a good idea to refill the molding machine before disassembling the mold and removing the part. This method will allow the new plastic to melt while you prepare for the next shot. Do not follow this procedure with Delrin, however. Delrin resins will degrade and produce formaldehyde fumes if left in the molding machine too long. With these materials, it is better to empty the machine completely before reloading.

4 Material Selection and Properties

Plastic materials are available with a wide variety of material properties. Table 4.1 lists the properties of some resins with useful properties. Nylon and Delrin resins can all be molded with the AI Lab equipment. Glass, carbon fiber, and other fillers all make molding more difficult, but improve other properties. Teflon cannot be injection molded; this material degrades before it melts.

4.1 Delrin 500

This is normal, un-enhanced Delrin. This material has good engineering properties: high strength, stiffness, chemical resistance, and good wear and impact resistance. The Mobot Lab has about 50 lb of Delrin resin, more than enough for everyone in the building. Delrin parts have a smooth surface finish (assuming your mold is smooth enough). The material does burn, however. Because the Quickshooter has relatively

Material	Young's Modulus	Yield Strength	Specific Gravity	Static Friction	Dynamic Friction
Nylon 6/6 b	410	12	1.14	.1	.15-.40
Nylon 30% glass fiber b	1500	25	1.4	Not listed	Not listed
Delrin 500 a	450	14	1.4	.2	.35
Delrin AF (Teflon filled) a	420	10	1.54	.08	.14
Delrin 577 (glass filled) a	900	12	1.56	Not listed	.35

References: a [Delrin 92], b [Dexter 79]

Table 1: Properties of Engineering Polymers

poor temperature control, some burning is unavoidable. As a result, finished parts have a white and brown "marbled" appearance.

The ideal molding temperature for Delrin 500 is approximately 425 F. This material will degrade if it is overheated, or left molten for too long. Try not to keep molten plastic in the molding machine for more than 15-20 minutes. Degradation produces formaldehyde fumes, which are toxic in moderate concentrations. It is important to use the vacuum cleaner and fume hood to remove fumes with all Delrin resins.

4.2 Delrin 500AF

This material is also called "Brown Delrin." It is a Delrin resin with a Teflon filler. Teflon is the lowest friction material in the world. It also has superlative resistance to chemical attack. The disadvantages of Teflon, however, are low stiffness and yield strength. Delrin AF is a practical compromise. It has slightly lower strength and stiffness than pure Delrin, but much lower friction. This material is good for bearing surfaces or sliding parts.

The Teflon additive raises the melting temperature slightly; mold Delrin AF at approximately 450 F.

4.3 Delrin 577

Delrin 577 is Delrin with 20% glass filler. It also has chemical additives to improve UV resistance. The glass particles in the plastic are not fibers; this material is less stiff than glass fiber reinforced Delrin. It does have a superior stiffness to weight ratio, which is the critical factor for many parts. This material produces a more textured surface finish. Because this resin is black, burning is not unsightly.

Mold Delrin 577 at 450 F.

4.4 Fiber Reinforced Nylon

There are two types of fibers that are used for reinforcing nylon: glass and carbon. Carbon fiber reinforced nylon has the highest stiffness to weight ratio of the plastics in the AI Lab. Glass fiber reinforced nylon is slightly less stiff and less expensive. Reinforced nylons are an excellent material for structural parts where high stiffness is a primary goal. They are rather brittle, however, which can be a problem if the parts experience shock loads.

These plastics are slightly more difficult to mold because the fibers don't melt. The most annoying aspect to molding with these materials, however, is their propensity to drool. It is important to avoid clogging the molding machine.

The proper molding temperature for these two materials is 550 F, much hotter than the other resins.

4.5 Switching Resins

Whenever you change the material in the molding machine, it is important to purge all of the old material first. After completing this step, fill the machine with polypropylene and then purge the machine to clean out any leftover resin. After loading the new material into the machine, the first few shots are likely to have some of the different plastics mixed together. Once you are producing relatively pure plastic, begin molding parts.

Some plastic resins are incompatible with each other. For example, Delrin and PVC will react violently when mixed. It is extremely important NEVER TO USE PVC IN THIS MACHINE. Before experimenting with a new material, find out if there are any potential compatibility problems.

5 References

Delrin Product and Properties Guide, Du Pont Polymers, Wilmington, DE, 1992.

Dexter, S. Handbook of Oceanographic Engineering Materials, Wiley-Interscience Publications, John Wiley and Sons, New York, NY, 1979.

Kalpakjian, S., Manufacturing Engineering and Technology, Addison-Wesley Publishing Co., Reading, MA, 1992.

6 Sources for Plastic Resins

General Polymers, Westfield, MA, (413) 568-9256

They sell 55 lb bags of standard resins: Delrin 500, 577, 500AF, etc.

DuPont Engineering Laboratory, (800) 441-0575

I got 5 lb sample bags (for free) of a bunch of resins: 500AF, 500TL, 500CL

RTP, Winona, MN, (507) 454-6900

They do custom compounding; they add fillers, reinforcements, and other additives to plastic resins. They also sent me free 5 lb samples of various materials: Nylon + glass fiber, Delrin + glass fiber, PPS + glass fiber, Nylon + carbon fiber, Delrin + carbon, and PPS + carbon.

7 Glossary

Draft: The amount of taper on the sides of a mold cavity, measured in degrees from a line perpendicular to the parting plane. Draft makes part ejection easier.

Drool: Plastic that drips out from the molding machine's nozzle

Flash: Plastic that squirts into the small gaps between mold sections. Normally this is just a faint line at the parting surface.

Gate: The end of the runner, where the plastic enters the mold cavity.

Irish persuasion: Brute force required to subjugate irascible and uncooperative machinery.

Parting line, plane: The line or plane where two pieces of the mold meet.

Quill: The part of the drill press that moves up and down; it includes the spindle and chuck.

Runner: A short, narrow channel which connects the sprue to the mold cavity.

Sink: A depression on the surface of a part caused by a thick section shrinking.

Shot: The wad of molten plastic in the molding machine.

Shot-size: The maximum amount of plastic that can be melted by an injection molding machine. Shot-size and clamping-force are the two important criteria for comparing the size of injection molding equipment.

Sprue: The passage where molten plastic enters the mold; usually, it is a hole drilled through one half of the mold.

Void: An empty space in the interior of a part