Chapter 1

Wrapping Your Head around Injection Molded Parts

In This Chapter

- Getting the fundamentals down
- Seeing how injection molded parts are made
- Readying parts for injection molding

No matter where you turn, you see things made out of plastic, and most of those things are composed of plastic parts. Clearly, people have been making plastic parts for a long time, gathering a lot of expertise along the way, and now manufacturers can make those parts quickly and well. If you're reading this book, you're probably thinking about getting some plastic parts made, and you want to know how you, too, can do so quickly and well.

Most plastic parts are created in a process called *injection molding,* which involves injecting molten plastic into an open space in a device called a *mold*. The name *injection molding* isn't going to win any originality awards, but the process itself is creative because it helps people turn incredible ideas into real parts. Knowing how injection molding works makes it easier to design parts for the process. This chapter gives you that understanding.

Getting Grounded in Injection Molding

For all of the big-brained science involved in injection molding, the process can be broken down into a few basic steps:

1. **Melt the plastic pellets.**
2. **Inject the melted plastic into the mold.**
3. **Let the molded part cool.**
4. **Eject the finished part out of the mold.**

This section gives you a broad overview of these steps. For more detail, see "Breaking Down the Part-Making Process" later in this chapter — and the rest of this book.

Meeting the machine

Understanding how injection molding works starts with understanding the equipment that makes the mold: the injection molding machine (see Figure 1-1).

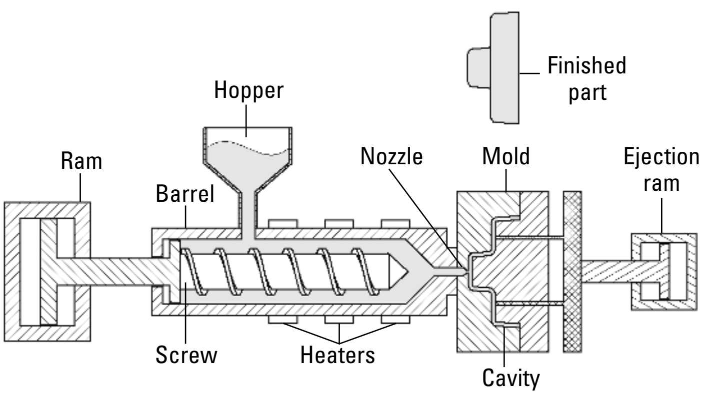

Figure 1-1: Schematic of a typical injection molding machine.

Making the part

The injection molding process starts with feeding pellets of plastic resin into the hopper of an injection molding machine (refer to Figure 1-1), which melts those pellets through a combination of heat and pressure. The heat comes from electrical bands on the outside of the barrel. The pressure comes from a variable pitch screw in the barrel.

This screw drives the pellets from one end of the barrel toward the mold. The ram, similar to the hydraulic cylinders you might see on an earth-moving machine, builds the pressure needed to force the plastic into the mold. When the molten plastic resin is soft enough, the ram pushes the screw forward, driving the plastic through a small nozzle into a cavity in the mold, where the part takes shape.

After the mold is filled, it's left to cool.

Finally, when the plastic has cooled long enough for the part to harden, the mold is opened, and the part is ejected.

Keeping design in mind

Most of the work of designing an injection mold focuses on what happens between two points:

- When the ram moves forward
- When the mold is opened to produce the part

Most of this book focuses on how you can make sure that what comes out of the mold is what you need it to be. Keeping the mold in mind while you design a part helps you find ways to create high-quality parts cost-effectively.

TIP

If a part is designed in a way that allows it to be high quality in a prototype tool in low quantities, it's also going to be high quality when you want to produce a million of them. Also, understanding how a part will be used can save a dramatic amount of time by allowing you to look at various approaches.

Don't be too shocked if you're asked to change minor aspects of a part. When these requests come up, they come up for good reason, and the people who ask them don't take them lightly. Collaborating and working with experienced people ultimately gives you a better part in the long run.

Breaking Down the Part-Making Process

The preceding section introduces the four basic steps of injection molding, but there's much more to know. If you're interested in getting parts made, you need to understand all the details of the process because you need to pay extra attention to some fine points when you're developing parts to be molded. This section gets you started on your detail work.

Starting the part with the mold

Because you can't have an injection-molded plastic part without having an injection mold, it's important to start by thinking about how your mold will be made, how it will work, and how you can get the best possible part from it.

A mold has many components. At a glance, it looks like a bunch of metal plates sandwiched together — which, oddly enough, is pretty much what a mold is. The plates and pieces have different functions, however, and can require some really amazing skill to build.

Talking straight about straight-pull molds

Although you will find some variations in injection molds, the most common style is the *straight-pull mold.* This mold centers on a work piece made of at least two pieces of steel or aluminum, held together with mechanical clamps or hydraulic pressure while the plastic is injected and then pulled straight apart (surprise!) when the plastic has cooled enough to remove the part.

You can add pieces called *side-actions* (also called cams or slides) to the mold to create openings in the sides of parts or to create more complex features. Side-actions make it easier to create more complex parts while still using a basic mold.

Spinning the mold analogy

Here's another way to think about the halves of a mold: The halves are usually referred to as the A and B sides, much like the vinyl records of old. As with records, the A side is the one that gets most of the attention, but the B side is probably more interesting. The two sides of the mold combine to enclose what will be referred to as *the cavity.*

Knowing the core from the cavity

At the center of the big mold assembly are the two halves that create the hollow where the melted plastic goes (see Figure 1-2).

These halves are the core and the cavity, and they work like this:

- **Core:** The *core* is usually the interior or noncosmetic side of the part. The core side also contains the ejection mechanism used to push the completed part out of the mold.
- **Cavity:** The *cavity* is the void inside the mold that the molten plastic fills. Plastic enters the mold from the cavity side and forms the final shape of the part. The cavity side usually forms the cosmetic side of the part.

REMEMBER

To get the benefits of the Protomold process, all you need to do is focus on the design of the part and make sure that it follows the guidelines in this book.

Doing a runner

As "Making the part" explains earlier in this chapter, the molten plastic from the extruder is pushed into the mold, entering it through a series of channels called a *runner system* (see Figure 1-3). At different stages of the process, the parts of the channel have different names (sprue, runner, gate), but they're always part of the runner system.

Defining the runner system is one of the true arts of mold design. Being able to maintain a smooth flow of material while making sure the whole cavity fills — and fills correctly — is the sort of thing that makes rocket scientists nervous.

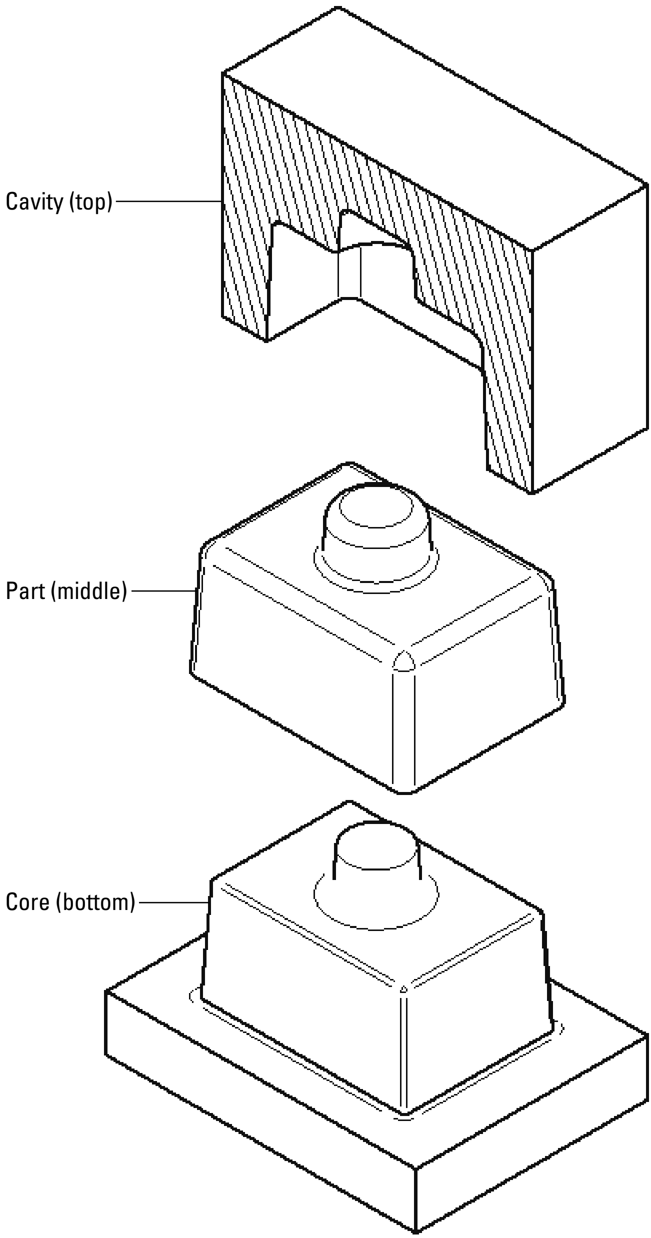

Figure 1-2: A part between a mold's core and cavity.

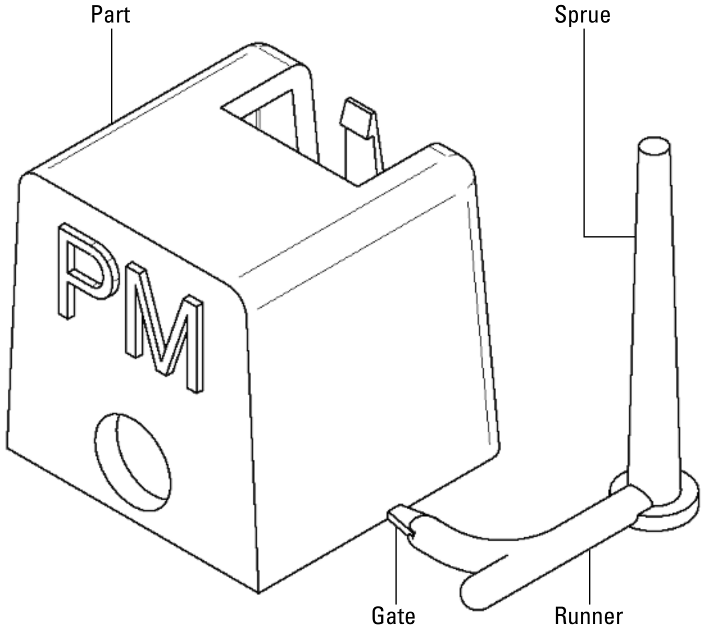

Figure 1-3: The runner system guides the melted plastic to the cavity that forms the part.

The runner system has to make sure that the mold can fill — but not too fast or too slow. It also has to make sure that the pressure doesn't get too high or too low. The runner system is usually defined when designing the mold.

If you've ever built a model car or an airplane, you've seen great examples of a runner system in the pieces to which the model parts are attached, as follows:

- **Sprue:** The *sprue* is the main channel through which the plastic enters the mold. It's typically larger than the other channels because the plastic for the entire part flows through it. In an airplane model kit, for example, the sprue is the thickest cylinder that rises above the rest of the plastic piece.
- **Runners:** The sprue connects to the *runners,* which connect all the parts and spread the plastic along the face where the halves of the mold meet. If you build a model

car from a kit, the runners are the sticks that run alongside all the parts and branch out from the sprue.

- **Gates:** The runners connect to the *gates,* which control the flow of the plastic into the cavity (I discuss this more in the next section). On your model car, the gate is where your part breaks off when you twist it.

You probably won't want people to see your finished part with the runner system attached. You can remove the part from the cool-looking runner system, and the runner can be appropriately admired or recycled by grinding it into pellets and running it back through the molding machine to make the next part. It's like a polymer circle of life.

Getting out of the gates

After you create a runner system, the next elements of the mold that you need to work out are the *gates,* which are the connections where the runners meet the cavity. Gates come in a variety of shapes and can maneuver the plastic into the cavity in various ways.

The locations of gates are important because if you want a part to come out of the mold looking like you thought it would, you must make sure that the plastic flows to all parts of the cavity. If you put the gates in the wrong places, that flow won't happen.

Following are a few types of gates you may want to use:

- **Edge:** *Edge gates* port plastic into the cavity through the edge of the part (see Figure 1-4). Injecting the plastic through the edge leaves the runner and the part connected when the process is complete. It's easy to trim or break the part off the runner, but an edge gate leaves a small imperfection called a *vestige.*

  If you need the edges of the part to remain clean, a couple of other gate types can keep your part looking cleaner.

- **Tunnel:** *Tunnel gates* inject the plastic into the cavity from a port that's cut into the core side of the mold and comes back up into a portion of the part (see Figure 1-5).
- **Post:** *Post gates* (see Figure 1-6) allow the plastic to be shot into the back of the mold via the paths of ejector pins (see the next section). The downside is that the ejector

will push the plastic out of the hole and leave it stuck to the part. If you need to, you can cut off the excess plastic, which is known as a *post* (hence the name post gate). So which came first, the post or the gate that formed it?

- **Hot tips:** *Hot tips* are gates that connect the sprue directly to the part. A tip is placed in the part's cavity and heated so that the part doesn't stick to it, but a dimple has to be added to the part to allow the plastic to flow out of it properly (see Figure 1-7). Molds using hot tips waste almost no plastic.

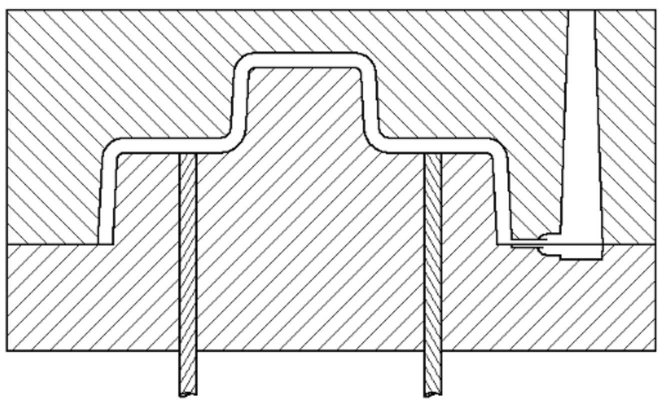

Figure 1-4: An edge gate.

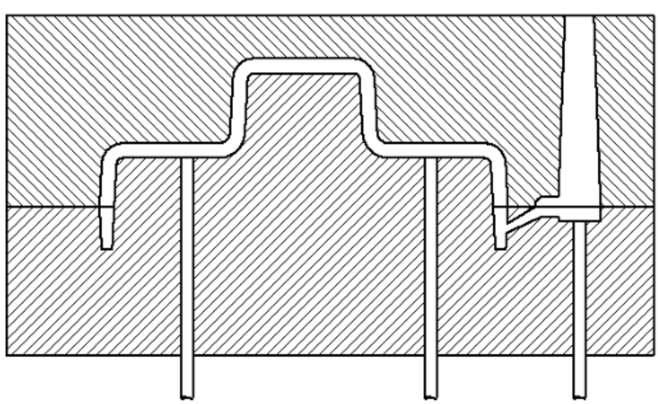

Figure 1-5: A tunnel gate is cut off when the part is ejected.

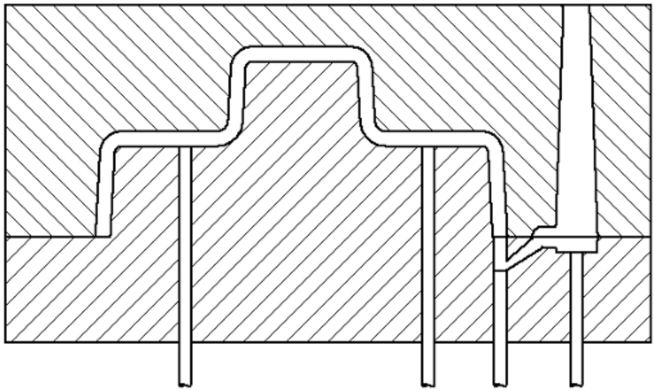

Figure 1-6: A post gate.

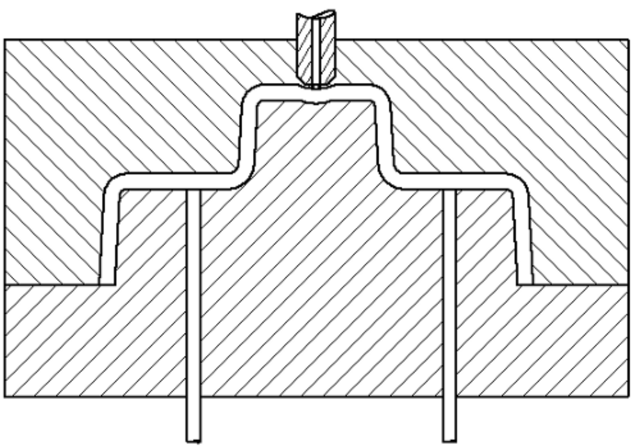

Figure 1-7: A hot tip gate.

A lot of thought is put into designing the cavity, the runner system, and the gate placement. Whether you need a short run of parts or a large-scale production, if a mold isn't designed to produce a quality part, it isn't useful.

Keeping your cool

Because the mold holds the plastic's heat a little too well, you need to cool the mold with water to help the part solidify faster. Then, when the part has cooled and solidified, you can open the mold. You generally have to push the part out of the mold by using *ejector pins* (see Figure 1-8).

WARNING!

If you don't wait long enough before opening the mold, your part will be distorted — which is generally considered to be a bad thing.

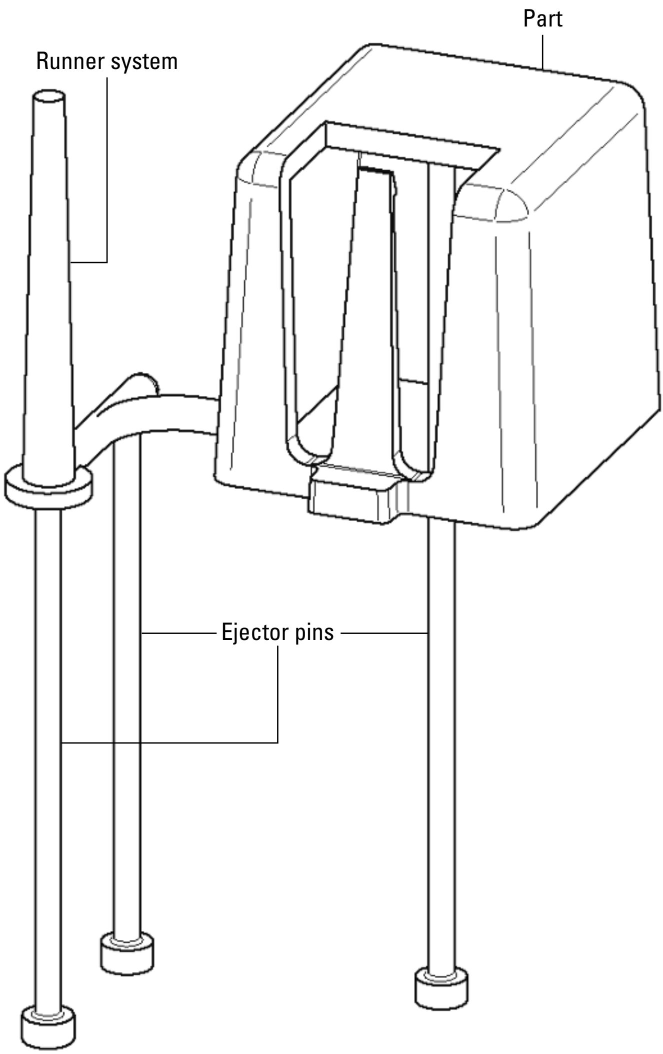

Figure 1-8: Use ejector pins to push the part out of the mold.

Prepping Parts for Protomold

To use the Protomold injection molding process, you need to consider additional factors when you plan the part design. So what is this Protomold process? I'm glad you asked.

Quote system

The Protomold process starts with an automated quote system that lets you select the options you want and gives you pricing information along the way. Then Protomold uses your CAD model to build a mold from it quickly, which lets them mold your part in practically no time.

Part size and configuration

To get a plastic part created quickly, you need to limit a few things, such as the size of the mold and how much plastic you can put in it.

Accurately calculating the size of a part can be complicated. Protomold's online quoting system automatically determines the necessary size of the mold and whether it exceeds the maximum size that can be made using their process.

REMEMBER

Now you know the basics of putting plastic into shape. Before you dig into the deep details of designing plastic parts, however, you need to know how to make a part simple and to the point so that it does its job correctly (see Chapter 2).

Minimizing mass

In designing some parts, you may wish to minimize mass. Here are a few considerations:

- The thickness of the part's walls, the size and volume of internal features, or even large, solid portions of the part may cause the part to distort in undesired ways.
- Ribs or other reinforcing features may have to be added, removed, or relocated to make the part easier to mold.