ProMold plastics
Custom Injection Molding & Moldmaking

Plastic Product Design Tips for Injection Molding

Injection Molding is one of the most popular forms of processing as it enables us to mold fairly complex shapes at high production rates and hold fairly tight tolerances while maintaining good dimensional stability. Thermoplastic resin used in the injection molding process can reground and reprocessed, however some physical properties of the plastic may be lost through each generation of regrinding. Many different engineered thermoplastic resins available today can be offered with special fillers such as glass or carbon fiber to improve strength and stability or with special additives that enhance the plastics properties such as providing some level of EMI/RFI shielding. Most Resins can be compounded to virtually any custom color. With all of these options available, some huge cost saving benefits can be taken advantage of when manufacturing large volumes as opposed to traditional machining and metal forming processes. Although the initial investment in tooling may be significant, it is critical that the following part design considerations be taken into account when designing a plastic part. It all starts with the plastic product design which includes specifying a suitable material for your application. A good mold design is then crucial to making your project successful. Since plastic behaves differently than metal, it is important to understand these common principles to avoid getting unexpected results.

Shrinkage: After plastic material is melted and injected into a mold under high pressure, it will tend to shrink during the cooling process. Some materials will shrink more than others while some are more stable and predictable. Plastic Manufacturers will usually provide you with experimental studies of how much their plastic will shrink under given conditions. This value is usually given as a percent or in thousands of an inch. A plastic with a shrink rate of 1% will actually shrink .010 of an inch for every inch of length. Mold makers take this factor into account when building a mold.

Draft: When a part is molded around a steel core it tends to grab on to the steel as it cools and shrinks making it difficult to eject from the mold. Because of this most mold designers factor in a draft angle of about .5 degrees along the length of the core to make it easier for the part to eject from the mold. Please keep in mind that the deeper the part that is being molded the more the dimension at the top of the part will differ from the dimension at the bottom of the part as a result of the draft angle used.

Thin Walls: Thin Wall Parts (Less than .020inches) can be difficult to fill. Maintaining a consistent mold temperature and a good mold design is critical in reducing the risk of high scrap rates due to parts that were not completely filled out during the molding process. Many parts with thin walls are molded using a nylon based resin because it's relatively low viscosity allows it to flow very easily and quickly to fill the mold cavity.

Roundness and Warping: Roundness is a factor that can sometimes be difficult to control depending on the material being molded. As plastic is not a stable as steel, the larger the plastics part you try to manufacture, the more likely you are to see the effects of warping. Some materials with higher less predictable shrink rates may cause unpredictable results depending on which way the material flows into the mold cavity. Designing a part with uniform wall thickness and using ribs in the part design to improve stiffness will help reduce the effects of warping. A good mold design capable of maintaining a consistent mold temperature and with the proper gate location for the plastic material to flow into the mold is critical in helping control this issue.

Sinks: Thick Parts take longer to cool and tend to show sink marks on the surface where there is excess material in the part. Many plastic parts are designed removing any unnecessary material and trying to maintain a consistent wall thickness throughout the part. This will actually help reduce the chances of encountering sink marks and actually increase production rates which will in effect lower your part cost.