

Tips for Efficient Injection Molding – Making Manufacturing Easier

Creating custom designed parts can be tricky. Add in molten plastic and life really can become difficult. In talking to engineers and sourcing professionals, we often hear that working with injection molded parts are some of the biggest challenges they face. There are a few easy tips you can use to make your life easier when it comes to molded parts.

Follow the simple guidelines below, starting in the design stage, and you will help eliminate most of the injection molding pitfalls.

Wall Thickness

Wall thickness depends on its end function and the type of resin used. Thinner walls usually cost less because it requires less raw material and cools faster but you risk the part not being structurally sound. On the opposite end, if the walls are too thick, shrinkage can lead to surface sink.

Parts with uniform walls typically fill properly and fit well together. Uneven wall thickness can lead to serious processing and quality problems. One major factor to consider is that resin shrinks as it cools. Thick areas may shrink more than thin ones and this can lead to warping in the finished part.

Furthermore, the dissolved gases released as resin cools are likely to cause bubbles in thick walls, thus causing the part to weaken. Strive to keep wall thickness under 10% to avoid the problems mentioned above.

Optimal Gate Location

There are times when varying wall thickness cannot be avoided. When this occurs, the gate location becomes more important. For instance, if resin has to pass through a thin area to reach a thick one, it can cause either incomplete mold filling or non-uniform molded parts. The gate location is optimal when the melt enters at the thickest part of the cavity and then flows to the more narrow areas.

Resin Characteristics

The characteristics and mechanical properties of the resin used can affect the molding process significantly. Strength is one of the most important features of a resin, as it can influence the quantity of material needed to meet the parts' requirements. Shrinkage can affect moldability and certain types of resins shrink unevenly. Viscosity and the ability to fill small features is an important factor to consider when selecting resin.

Corner Radius

Sharp inside corners can cause built-in stress, especially during the part of the cooling process when material pulls against the corners as it shrinks. Rounded corners allow the plastic to flow easily and shrink properly, leading to strong, stable corners.

Uniform wall thickness is easier to maintain when the corner radius is wider. Also, equal amount of shrinkage can be obtained if the inside and outside corners of the parts are rounded and have generous radii. Overall, rounded corners can lead to stronger, higher quality parts, which will resist post-mold warping.

Suitable Draft Angles

Proper draft angles ensure that the part and mold surface will slide apart easily during ejection, without deforming. From a cost and manufacturability standpoint, the ideal draft angle is the largest angle that will not lessen the customer's satisfaction with the product. The optimal degree of draft depends on numerous factors, such as height, location, surface texture, etc.

For surfaces that are parallel to the direction of the mold opening, drafting is almost always required. It is hard to pinpoint the minimum allowable draft angle, but 1° per slide is considered sufficient when possible. Even an unusually small draft angle (such as 0.25°) can help you avoid damaging the parts.

Follow these tips, and you will be on your way to a more seamless and manageable injection molding process.