

RSP, Inc.

Milwaukee, USA - Shenzhen, China
Phone: (414) 546-4417 - Fax: (414) 546-1105
<http://www.rspinc.com>

Plastic Molding Design Guide

Ryan Screen Printing (RSP) is your trusted source for plastic injection molding, rubber keypads, membrane switches, turn-key assemblies and more. Our engineers and offshore production facility can design, rapid prototype, and produce your plastic parts.

RSP has over 50 years of manufacturing and design experience. We work closely with the customer, the engineer and the production team to make sure every project is completed to specifications and on-time.

If you need additional information please contact us directly at (414) 546-4417.

INTRODUCTION

This design guide is intended to offer basic information on plastic design and offer guidelines for you to follow as your project develops.

Plastic injection molding is the primary process for manufacturing plastic parts. Plastic is a very versatile and economical material that is used in many applications.

Injection molding involves taking plastic in the form of pellets or granules and heating them until melted. The melted plastic is forced into a split-die mold (chamber) where it is allowed to cool. The mold is opened and the part is ejected.

ABS (acrylonitrile-butadiene styrene) is the most common molded plastic. ABS possesses outstanding impact strength and high mechanical strength which makes it suitable for use in tough consumer and industrial products.

PROTOTYPING

There are several options for prototyping plastic designs. Our engineer can supply you a part very quickly using fused deposit modeling (FDM). For a more robust sample the factory can produce CNC parts in 2-4 weeks. CNC parts can be done in a variety of materials, colors, and finishes and are structurally stronger than most other methods of modeling.

DESIGN CONSIDERATIONS

Design Options

Standard tooling charges often includes 3D modeling. We can build the design based on your 2D drawing or a sample part. Some customers using off the shelf plastic parts find

that the design flexibility, reduced assembly time (snap together pieces, custom fit enclosures) and/or reduced processing costs (modifying the off the shelf option) make a custom plastic part as affordable as or cheaper than generic solutions. Call us and let us know what part you are currently using and we can price a custom molded piece and discuss how to save you time and money.

Draft

Plastic parts need to be designed with a taper or draft in the direction of mold movement to allow the part to eject from the mold. Since plastics shrink when cooled, it is common for parts to shrink to cores.

- Draft angle of ½ degree is regarded as the minimum for most applications
- Common draft angles are 1 ½ - 2 degrees

Ribs

To increase the stiffness and rigidity of a part that is load bearing many designs include ribs. Ribs are a practical and economical method of increasing the structural strength.

- Rib thickness should be less than wall thickness. A rib thickness of 60% - 80% of nominal wall thickness is recommended.
- The maximum height of a rib is 3x's the nominal wall thickness of the part. The minimum spacing between ribs is 2x's the nominal wall thickness.
- To improve strength, increase the number of ribs rather than increasing the height of the ribs.
- When thicker ribs are needed, the ribs should be 'cored out' from the back so that the wall thickness remains uniform.

Holes

Holes are a common feature of plastic molding. Core pins are used that protrude into the mold cavity to create the opening. Through holes are easier to produce than blind holes which do not go through the entire part. Blind holes are created by using a pin that is only supported on one end. Pins used for blind holes are susceptible to being deflected and pushed off center by the pressures of the molten plastic material during molding.

- The depth of a blind hole should be about 2x's the diameter of the core pin up to 3/16" and up to 4x's the diameter of the core pin for dimensions greater than 3/16".
- For every cored or molded hole in plastic there will be a weld line. The weld line is caused by the flow of the melted plastic around the core pins. Weld lines are not as strong as the surrounding plastic material. Weld lines may also detract from the appearance of the plastic. This can be eliminated with a more functionally stable design.

The coring of holes is easy when the axis is parallel to the parting lines. However, when the holes and other features run perpendicular to the parting line then retractable cores (or cams) are required. Split pins and cores (called passing steel shutoffs) can be used to create some of the features. A good designer should be aware of the added problems of side action cores and the added expenses associated with such a design.

- Whenever possible, design features should be incorporated in the same direction of the mold opening so that cam action can be avoided. Cams create undercuts in the plastic and are retractable so that the part can be ejected.

Bosses

Bosses are used for locating, mounting, and assembling purposes. One of the main concerns to be aware of is wall thickness.

- Wall thickness around a boss design feature should be 60% of the nominal part thickness if the part is less than 1/8" thick.
- If the nominal part thickness is greater than 1/8", the boss wall thickness should be 40% of the nominal wall thickness.
- The height of the boss should be no more than 2 1/2 x's the diameter of the hole in the boss to avoid sink marks and surface imperfections.

Finishes

If the part is not visible to the user you will probably choose the standard finish. There are several items to consider when picking a finish.

- Polishing: Smooth part surfaces are achieved using manual mold polishing techniques. A part with tall, thin and curved ribs needing a polished finish will have a higher costs since it is time consuming to polish deep, narrow slots in molds.
- Texturing: Many textures are achieved using bead blasting. Given the line-of-sight nature of bead blasting, it may not be possible to texture the sides of minimally drafted ribs on a part if the mold surface is inaccessible. If the walls of your part are textured it may have an adverse effect on the ability of the part to release from the mold which can cause unsightly "drag marks". For these reasons we recommend that texture be specified only on areas of the part that are drafted at least 2-3 degrees.

METAL VS PLASTIC FUNCTIONALITY

There are many reasons to use plastic in your design and many of our customers switch parts from metal to plastic.

- Decrease piece price: after initial tooling the piece price can be substantially lower with plastic.
- Eliminate secondary operations: plastic can be colored and textured in the molding process. Several metal parts can sometimes be replaced with 1 plastic piece.
- Reduce product weight: plastics weigh less than metals and can make it easier for the user or assembly team to work with and it also reduces shipping costs.
- Increased structural strength: in some applications a well designed plastic piece is stronger than a metal piece. Plastic can have ribs and bosses built in eliminating secondary procedures needed with some metal.
- Increase product life: metals are much more susceptible to environmental degradation. Most plastic has greater chemical resistance than metal.