AKRON PORCELAIN & PLASTICS CO.

2739 Cory Ave.
P.O. Box 15157
Akron, OH 44314-0157
Phone (330)745-2159 Fax (330)745-6688
Website: http://www.akronporcelain.com
email: sales@akronporcelain.com

Plastics Division
ISO 9001:2008

DESIGN GUIDE FOR THERMOSET PLASTIC PARTS

General

Input on the best molding method for the design concept should be sought early in the design process so the part shape can be optimized accordingly. This will ultimately reduce material and manufacturing costs. Akron Porcelain & Plastics can offer significant material and processing information to assist the product designer.

As with other plastic materials, good thermoset plastic part design requires consideration of a number of aspects: tolerances, shrinkage, wall thickness, draft angles, radii, fillets, ribs, bosses, holes, threads, inserts, etc. The following suggestions are fundamental practices to follow when designing a thermoset molded part.

Tolerances

It is best to always design with generous tolerances in mind since extreme dimensional accuracy can be difficult and expensive. The main contributors to dimensional variation in the molding process are: mold dimensions, molding shrinkage, post-shrinkage, warpage, and machine process variations.

Typical molding tolerances for properly designed parts are plus or minus 0.003" per inch, and are increased by an additional plus or minus 0.005" when measuring across the mold parting line.

Material Shrinkage

Each molded part has a characteristic shrinkage defined as the difference between the dimensions of the mold and the molded part when measured at room temperature. Shrinkage varies, ranging from 0.001" to 0.015" per inch depending on the specific material, flow grade, molding method and process parameters. Non-uniform cross sections will result in varying shrinkage within the same part and should be avoided.

Wall Sections

Uniform wall sections are important. Thick wall sections require increased cycle time in order to fully cure the material, so sectional thickness should be minimized without sacrificing strength. If possible thick sections should be thinned or cored out with a pin or blade to reduce part weight and cycle. Sink marks do not occur with thermoset materials, but uneven sections can cause warping and shrinkage variation due to uneven cooling. Generally, nominal wall sections range from 0.050" to 0.125", but thicker sections are achievable. Sections in other areas of the part should be within 25% of the nominal wall section. Sudden changes in wall section should be avoided.

Thin sections, edges and sharp corners can result in breakage problems during the secondary deflashing operations required with thermosets. Thin sections can also cause flow problems and weld lines possibly resulting in warping, breakage, and electrical failure.

Draft Angles
Draft should be provided on all vertical mold surfaces for proper part release. A draft angle of one degree per side is sufficient on side wall sections, and two to three degrees are desirable on deep barriers and bosses.

Radii and Fillets
Adequate radii and fillets eliminate stress concentrations, improve flow, reduce cracking and improve overall strength with very little additional weight and cost. Minimum radii for all inside and outside corners are typically 25% to 75% of nominal wall thickness with a minimum radius of 1/64".

Ribs
Ribs increase rigidity and strength without changing the nominal wall thickness. Typical ribs are half as thick as the nominal wall section, and should not be any closer together than twice the nominal wall section. Sufficient rigidity is achieved with rib heights that are three times the rib thickness. Free standing ribs should be avoided since they are prone to breakage. Since sink marks do not occur in thermoset materials, ribs, bosses, etc. can be added where needed to enhance the design.

Bosses
Bosses should be located in the corners of the molded part and be no more than twice the diameter in height. If additional height is necessary, ribs are recommended at the base of the boss for added strength. Three degree draft angles and a generous radius at the base of the boss will aid part release and strength. Hollow bosses should have wall sections at least equal to the inside diameter.

Holes
Through holes are easier to produce because the mold core pins can be supported from both ends. Blind holes are formed by pins supported from one end only, and these pins must be strong enough to withstand molding forces. When designing compression molds, core pins should not be more than 2 ½ times the diameter in length or pin breakage will be a problem. With 0.062" diameter pins the pin length should not exceed the diameter.

In applications where pins must be longer, designing the core pins with half the length in each side of the mold, with 0.002" to 0.005" clearance between the tips when the mold is fully closed, is recommended. To account for any mismatch, one pin should be slightly larger in diameter than the opposite pin. As a general rule, edges of holes should be at least twice the nominal wall section from part edges or other holes.

Threads
Threads may be molded or tapped. Fine threads of 32 threads per inch should be avoided. Molded pilot holes with chamfered edges are recommended for secondary tapping operations. Thread cutting self-tapping screws also work well and eliminate the need for molded or tapped threads. Do not use thread forming screws.

Parting Line Flash
Due to volatile gasses generated during the molding process, low melt viscosity and other processing conditions, parting line flash is inherent in the thermoset molding process. Even newly built molds

generate flash that must be removed from the part with a secondary process. Some typical methods are: batch de-flashing, belt fed de-flashing, and hand finishing. When possible, the part should be designed with generous radii and fillets so the most economical method, batch de-flashing, can be employed. Other de-flashing methods will increase part cost. If cosmetics are highly critical, hand or specialized de-flashing may be required.

Inserts
Molded-in inserts should be used only when there are no acceptable alternatives. Brass and aluminum have thermal expansion properties most compatible with thermoset plastics. Thin wall inserts with sharp corners should be avoided. Diamond knurling usually provides the best anchorage, although single grooved inserts free of sharp edges also work well.

Knurled inserts can be pressed in immediately after molding, preferably within three minutes of molding, to take advantage of part shrinkage. The hole for the insert is based on insert size and normal material shrinkage. Hole diameters for small inserts should be 0.001" to 0.002" smaller than the insert outside diameter. Larger inserts will require additional anchorage. When working with thin wall sections, apply 50% of the normal shrinkage when determining the hole diameter so as to prevent cracking. The wall section around the insert should be 0.8 times the insert diameter for 0.5" diameter or less inserts, and 1.5 times the insert diameter for larger inserts.

Quick Prototype Sample Parts
Once the chemical, physical and electrical properties are defined, it is best to construct a 3-D CAD file and possibly fabricate a quick prototype to fully evaluate the part. It will then be easy to identify areas that need further attention. For example, knife edges on the part are often overlooked during the early design phase and can result in chipping. Likewise, knife edges in the mold itself are prone to wear and damage ultimately causing extra tool maintenance. Akron Porcelain & Plastics can provide quick prototype SLA and SLS parts so the design concept can be properly evaluated.

Prototype Molds
When working with a new application it is best to construct a prototype mold to evaluate the performance of the part design prior to constructing a multi-cavity production mold. In many cases the prototype mold cores and cavities can be used in the multi-cavity production mold as well to help minimize development costs. To replicate the ultimate production part properties, the prototype mold should be designed to operate under the same processing conditions as the production mold.

Summary of General Design Guidelines for Thermoset Plastics:

Molded Part Tolerances: +/- 0.003" per inch, additional +/- 0.005" when measuring across the mold parting line.

Material Shrinkage: 0.001" to 0.015" per inch depending on the grade of thermoset material.

Typical Nominal Wall Thickness: 0.050" to 0.125" with less than 25% variation

Draft Angles: one degree/side on side wall sections, two to three degrees/side on deep barriers and bosses.

Radii and Fillets: 25% to 75% of nominal wall section, minimum of 1/64" radius.

Rib Thickness: 50% of nominal wall section

Rib Height: Three times rib thickness

Boss Height: Less than two times the outside diameter of boss

Boss Draft Angle: Three degrees

Hollow Boss Wall Section: Two times inside diameter of boss

Hole Depth: 2.5 times hole diameter (when compression molding)

Hole Edge Locations: Twice the nominal wall section from part edges or other holes.

Hole Diameter for Inserts: 0.001" to 0.002" less than insert diameter for small inserts, increase slightly for larger inserts.

Hole Wall Thickness for Inserts: 0.8 times insert diameter for 0.5" diameter or less inserts, 1.5 times insert diameter for larger inserts.

Insert Type and Material: Diamond knurled brass or aluminum.