“Effect of Part Design on Injection-Molding and Plating-on-Plastic Processing”

David M. Rudder, Worldwide Plating on Plastics Support Manager, Atotech GMF Worldwide, St. Louis, Mo., USA.

The automotive and plumbing industries have stringent requirements for the field performance of decorative copper-nickel-chrome coatings on plastic parts. These requirements include adhesion and thermal-cycle performance, as well as the corrosion resistance of these coatings.

This paper will review the relationship between the design elements of plastic parts and the injection molding and electroplating processes. The effects which certain design features have on the injection-molding process and the adhesion of the plating layers will be discussed.

The paper will also examine how the design of the part influences the electroplating process and the corrosion resistance of the final product. Finally, detailed design guidelines will be presented that will facilitate the optimization of the injection molding and electroplating processes, resulting in superior thermal-cycle and corrosion performance.

For more information contact:
David M. Rudder
Worldwide Plating on Plastics Support Manager
Atotech GMF Worldwide
Phone - (314) 630-6899
E-mail – david.rudder@atotech.com

"Effect of Part Design on Injection-Molding and Plating-on-Plastic Processing"

David M. Rudder, Worldwide Plating on Plastics Support Manager, Atotech GMF Worldwide

INTRODUCTION

During the past decade the quality and durability requirement for electroplated plastic parts supplied to the automotive and plumbing industries has increased dramatically. This requirement has put an ever increasing burden not only on the suppliers of decorative copper-nickel-chrome finishes to these industries, but also to those companies that supply the injection molded product to be electroplated.

The final quality of the electroplating coating is measured by its appearance, adhesion and corrosion resistance. The appearance criteria are agreed upon between the supplier and the OEM or Tier I that has responsibility for the program. Appearance standards are agreed upon by both parties and kept as references for future comparisons.

The adhesion is measured by thermal-cycle testing criteria that varies between OEM's, but essentially involves subjecting parts to a series of alternating low and high temperatures. In general, if no cracking or blistering occurs then the parts are considered to have passed.

The corrosion effectiveness of the decorative coating is measured by subjecting the part to an artificial corrosive environment for a specified number of hours. This test is usually referred to as the CASS test, or Copper-Accelerated-Salt-Spray. The number of hours for testing is specified by the individual OEM's based on whether or not the part will be used in an interior or exterior environment and testing can vary from 16 to 80 hours of CASS. Both the appearance and corrosion resistance of the coating are evaluated at the end of the test to determine if it acceptable.

The suppliers of these electroplated plastic products have many decisions to make which will affect the outcome of their manufacturing processes. The factors that determine the final quality of electroplated products are not all within the control of the electroplating suppliers. The interrelationship of many factors determines the final quality of an electroplated product. In Figure 1 you see many different relationships, some of which we are going to be examining in this paper.

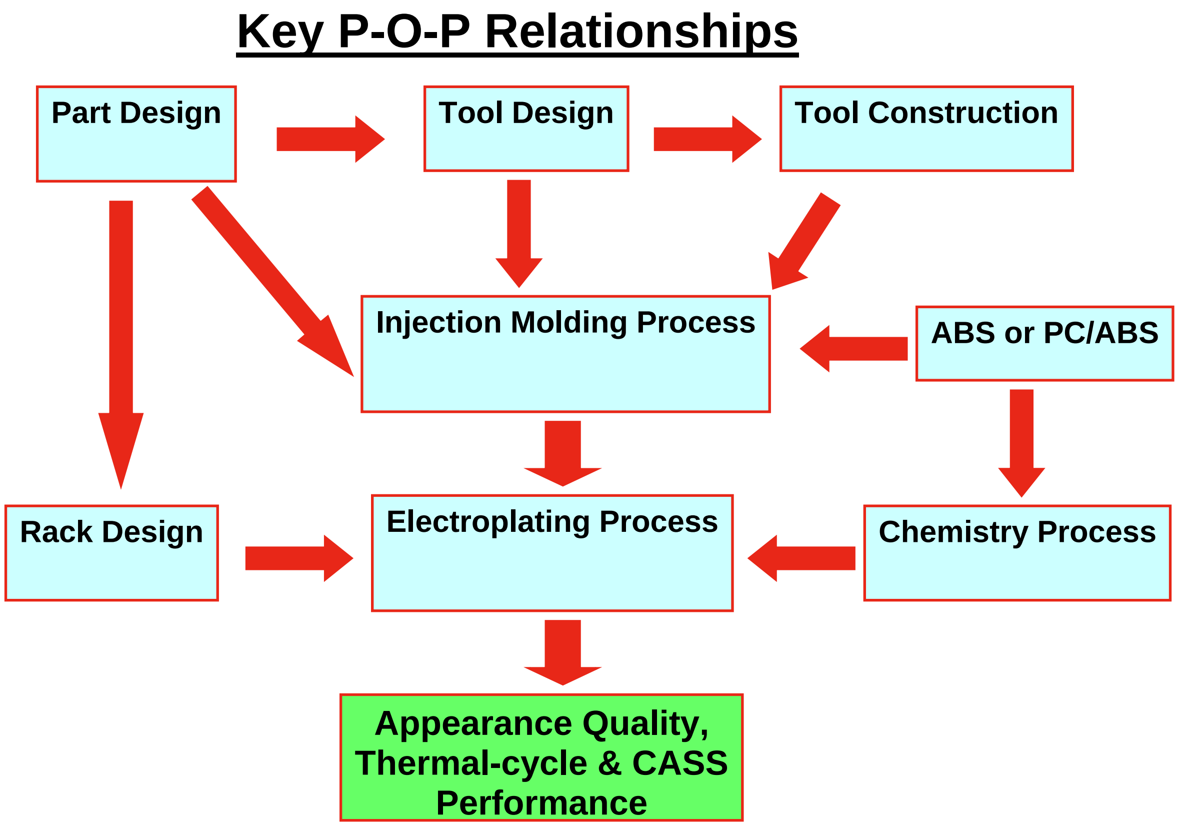


Figure 1

The keys to high quality plating-on-plastic success involve not only the injection molding and electroplating processes, but also include the part design, mold design and construction, molding material selection and electroplating rack design. The initial part design of an electroplated product will greatly influence the final quality of the product and we are going to take an in-depth look into design elements and their influence on the injection molding and electroplating processes.

The majority of today's automotive designers do not understand the significance their design plays on the manufacturing processes and the long term durability of decorative chrome plated plastic parts. Design features such as wall thicknesses, sharp edges and corners, attachment bosses, etc. all can have a significant impact on the quality of the end product.

The quality of the electroplating coating as measured by its adhesion and corrosion resistance is dependent upon many factors. From the electroplating standpoint the effectiveness of the typical chrome-sulfuric etch is the key to achieve optimum adhesion values. The adhesion is greatly affected by the quality of the plastic material and the injection molding conditions which are used to produce a plastic part. The end use of the product will determine the final selection of the plastic resin that will be used it its manufacture. Today the majority of electroplated plastic parts are molded from ABS materials, but more and more applications are being produced in PC/ABS materials because of higher heat and/or impact requirements. Products such as grilles for the front end of automobiles and wheel covers are typical products utilizing PC/ABS alloys.

When proper electroplating grades of ABS materials are not selected, the size and distribution of the butadiene particles are not optimized for electroplating. In addition, the injection molding contributes to orientation and stress in a plastic part and we will examine a few of these conditions. Orientation and deformation of butadiene can lead to a surface condition that is not evenly etched and therefore, adhesion is not optimized and thermal cycle failures can occur between the metal layers and the plastic surface. Improper molding conditions can also result in the top skin layer of plastic delaminating from the plastic immediately below the surface and we will discuss this in greater detail later.

In many instances parts are not properly molded because they are poorly designed and have design features that lead to problems in the molding process. We will discuss these design elements and why they lead to injection molding issues that contribute to electroplating problems.

PART DESIGN GUIDELINES DUE TO ELECTROPLATING

First, lets discuss some vary basic design features that effect the metal thickness distribution on an electroplated part. Figure 2 shows a basic representation of electrical current distribution and the relative thicknesses from HCD (high current density) to LCD (low current density) areas on a theoretical part. The current density is never uniform over an individual part or from part to part on the same electroplating fixture. The red area represents the relative thickness difference that can be expected for different areas of a part depending upon the current density in that area.

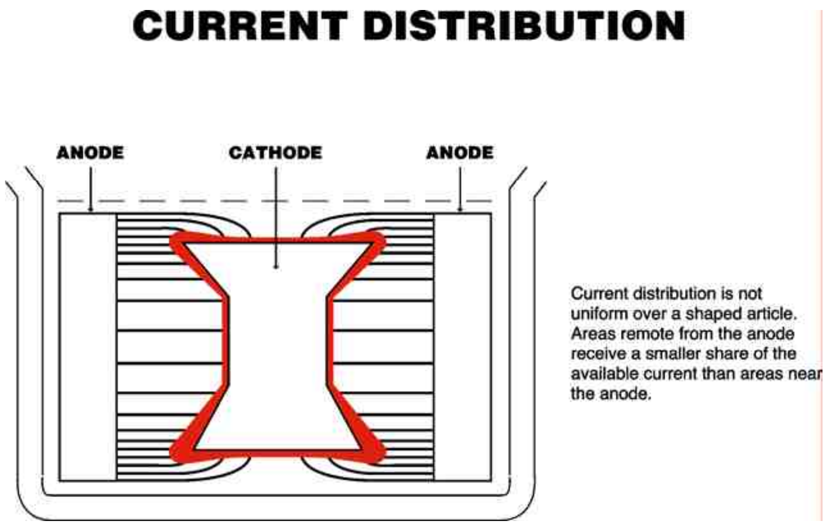
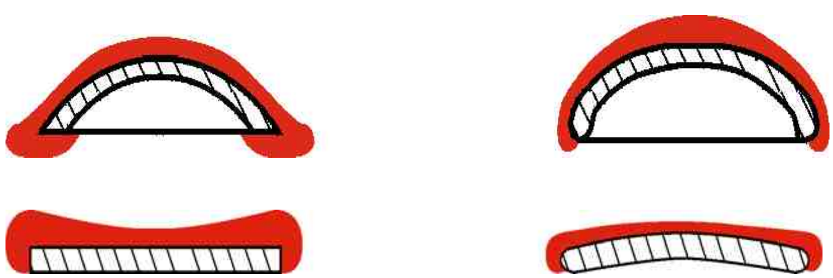


Figure 2

Typical thickness differences on complex shapes like wheel covers and claddings can be fourteen times greater in the HCD area than the LCD areas. This degree of thickness variation can lead to assembly issues caused by excessive thicknesses in the HCD areas and corrosion resistance issues with low thicknesses in the LCD areas. This variation in thickness can be minimized with the use of auxiliary anodes, but at a cost of higher scrap rates and thus higher overall manufacturing costs.

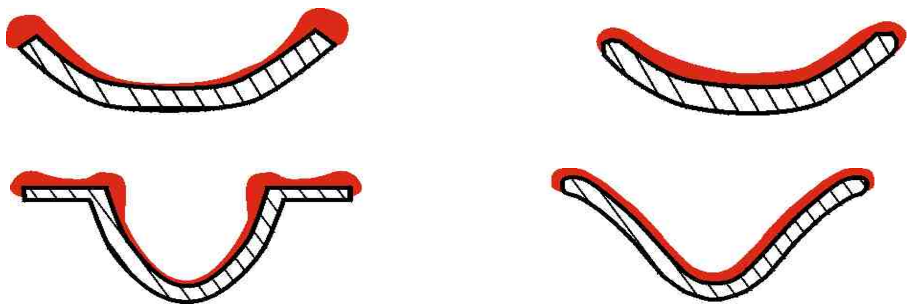
There are some general rules that should be considered when designing parts that will be copper-nickel-chrome electroplated. When we examine surfaces of plastic parts we find that convex surfaces are more favorable than flat surfaces because of the build up of plating on the edges. This rule is graphically represented in Figure 3. A convex or crowned surface results in a more even distribution of plating especially when the edges are well rounded.



Better designs

Figure 3

The ability to plate concave surfaces is dependent upon the depth of the recess. Deep recesses will increase plating time and costs to obtain a specified minimum thickness. Figure 4 shows some examples of these recess designs, with suggested alternatives.



Better designs

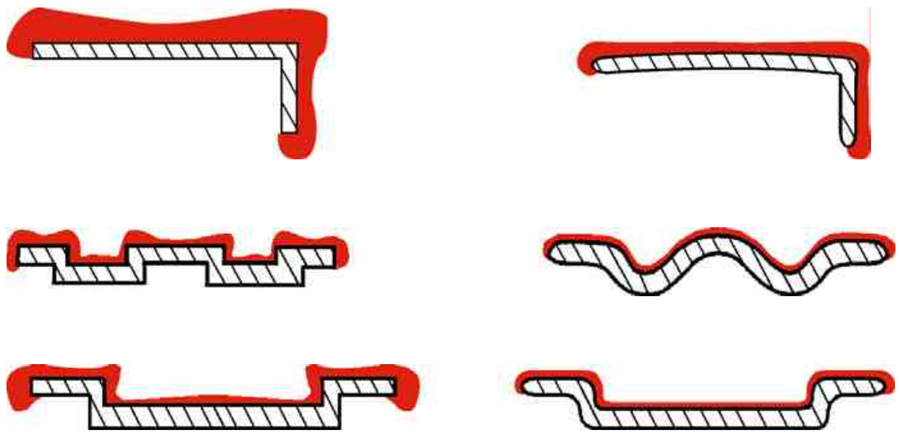
Figure 4

Blind holes should be eliminated whenever possible because they are not only extreme LCD areas but also because they can trap electroplating solution. When blind holes are necessary you should limit the depth to 50% of the hole width. It is also advisable to avoid diameters less than 6 mm because of the entrapment of solution. Figure 5 shows an example of blind hole design, with the bottom radiuses larger for more even plate distribution.

Better design

Figure 5

One of the biggest problems with some part designs today are the sharp edges and corners that designers use to present a certain style or look with their products. Sharp edges are very undesirable because of the build up of electroplating thicknesses on these edges, resulting in not only cosmetic issues but also fit and function in many cases. Figure 6 shows a number of examples of sharp edges and how they will be affected by plating buildup. All corners should be rounded whenever possible and all inside and outside angles should be likewise designed.



Better designs

Figure 6

Another part feature that is often designed poorly is the spacing between ribs or bars on the front grille of vehicles. The spacing between bars is often too close together resulting in very LCD areas that are susceptible to low plating thicknesses and thus corrosion failure in the field. In Figure 7 you can see some examples of good and bad rib spacing. The distance between ribs should be at least twice the rib depth in order to achieve an acceptable minimum thickness. In some cases, the back part of these ribs can be coated with a resist paint which inhibits the deposition of metal in these LCD areas. This selective plating can help improve impact resistance, thermal cycle performance, and a more uniform thickness distribution can be achieved. The use of this resist paint in the recessed or low current density areas can completely eliminate the problems of trying to meet minimum plate thickness in those areas. However, in most cases on decorative parts this resisted area will have to be top-coated with a final decorative paint, resulting in additional manufacturing costs.

Better design

Figure 7

In summary, the major plastic part design guidelines that are due to electroplating limitations are as follows:

1. No sharp corners or edges to minimize plate buildup.
2. Crowned (convex) surfaces are desirable over flat surfaces to even out plate thickness.
3. No blind holes or bosses to minimize solution entrapment and dragout.
4. Adequate spacing between ribs or bars to allow plating throw.
5. Avoid deep recessed areas which are hard to throw plating into.
6. Resist paint for selective plating can help in some cases with poor designs.

PART DESIGN GUIDELINES DUE TO INJECTION MOLDING LIMITATIONS

There are certain design features that have a significant influence on the injection molding process. These design features can force the injection molding processor to use certain molding parameters that are not conducive to molding a stress free part for electroplating. These key elements of plastic part design are:

1. Wall thickness
2. Ribs
3. Gussets
4. Bosses
5. Blind holes
6. Through holes
7. Hole spacing
8. Radii & corners
9. Sharp corners
10. Draft angles

The major rule for wall thicknesses is to keep them as thin and uniform as possible, while meeting the functional requirements of the part. This will result in even filling of the mold and more uniform shrinkage is obtained. The internal stresses will also be reduced, resulting in a part with a greater chance of passing stringent thermal cycle testing. Typically, the wall thickness should be in the range of 0.5 to 4.0 mm, with 2.0 to 3.0 mm optimum. Thinner wall thicknesses will result in shorter molding cycles and lower part weight resulting in cost savings. When varying wall thicknesses are needed for reasons of design, there should be a gradual transition of 3 to 1 as seen in Figure 8[1].

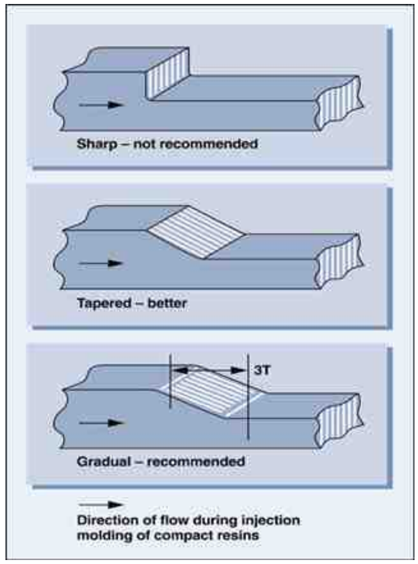


Figure 8 (Wall thickness changes)

One of the biggest mistakes that design engineers still make is the placement and design of ribs for strength. In order to minimize sink on the Class A surface, rib thicknesses should not exceed 50 % of the nominal (or intersecting) wall. Greater thicknesses will cause sink marks on the surface opposite the ribs. Injection molding process engineers will try to eliminate these sinks by introducing more holding pressure to the process and thereby increase the amount of internal stress introduced into a part. Typical rib design dimensions are shown in Figure 9[1]. The maximum rib height should also not exceed 3 times the nominal wall thickness if possible, because deep ribs become difficult to fill and may tend to stick in the mold during ejection. In addition, the typical draft angle on ribs is 1.0 to 1.5 degree per side with a minimum of 0.5 degree per side.

At the intersection of the rib base and the nominal wall a radius of 25 to 50% of the nominal wall section should be included. The recommended minimum radius value is 0.4 mm. These radii will help eliminate stress and improve material flow and cooling around the rib. Using larger radii will give only marginal improvement and increase the risk of sink marks on the opposite side of the wall.

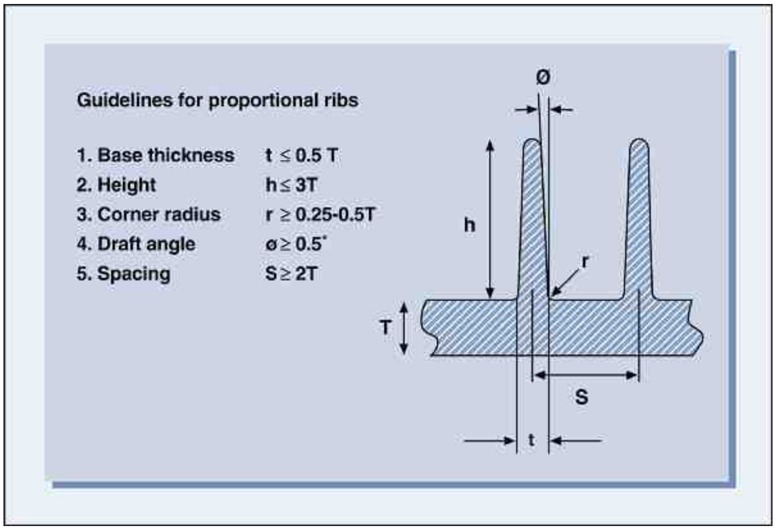


Figure 9 (Rib design guidelines)

Gussets are a subset of ribs and the guidelines that apply to ribs are also valid for gussets. The height of the gusset can be up to 95% of the height of the boss or rib it is attached to. See Figure 10[1].

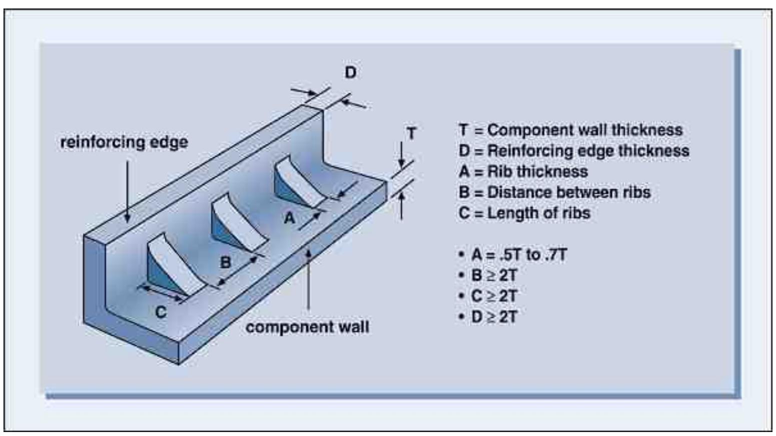


Figure 10 (Gusset design)

The use of bosses for attachment purposes is common place in electroplated plastic products. These bosses should have wall thicknesses that are less than 60% of nominal wall thickness and preferably should be 50% or less. Greater wall thicknesses are often designed for greater strength but they can cause molded in stresses and result in sink marks. As with ribs, a minimum radius of 25% of the nominal wall thickness or 0.4 mm at the base is recommended to reduce stress. Figure 11[1] below details the boss design guidelines. A minimum draft angle of 0.5 degree is required on the outside dimension of the boss to ensure release from the mold on ejection. In addition, a minimum draft of 0.5 degree is recommended on the internal dimension for proper ejection from the mold.

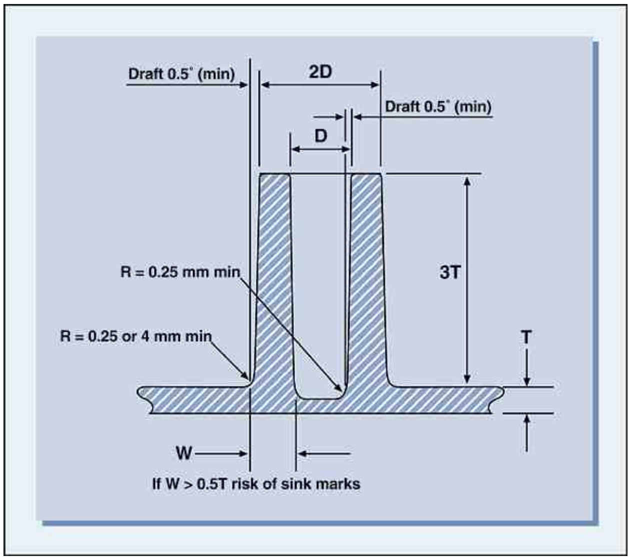
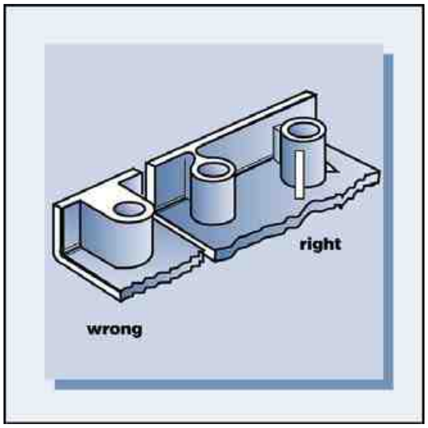


Figure 11 (Boss design)

Bosses adjacent to external walls (Figure 12[1]) should be positioned a minimum of 3 mm from the outside of the boss to avoid creating a material mass that could cause sink marks and extended cycle times.



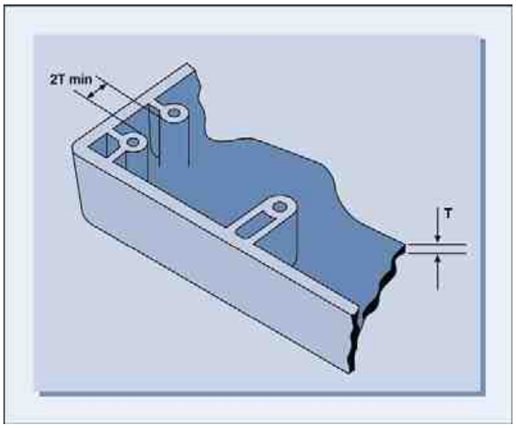


Figure 12 (Boss design)

Figure 13 (Boss design)

The minimum distance of twice the nominal wall thickness should be used for determining the spacing between two adjacent bosses. If bosses are placed too close together thin areas of standing steel that are hard to cool will be created in the mold, which will affect the quality and productivity. See Figure 13[1].

We talked about blind holes being a problem from an electroplating standpoint and they should be avoided if at all possible in a part that is to be electroplated. Blind holes can trap electroplating solution and cause excessive scrap. In some cases blind holes may have to be plugged before plating in order to prevent this scrap. In Figures 14[1] & 15[1] details of the design criteria to be used when incorporating blind holes into a plastic part design are shown.

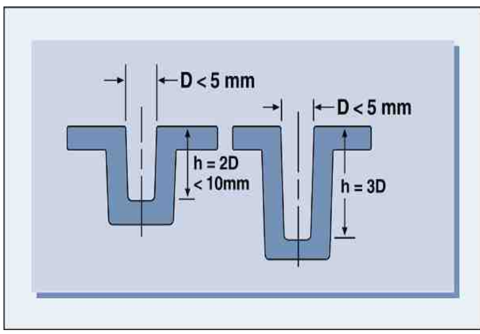


Figure 14 (Blind holes)

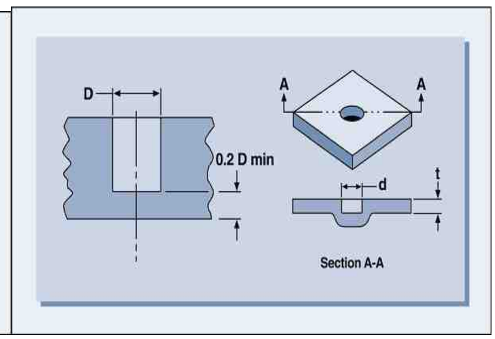


Figure 15 (Blind holes)

As a general rule the depth of a blind hole should not exceed 3 times the diameter of the hole. For diameters less than 5 mm this ratio should be reduced to 2:1 if at all possible. From an injection molding standpoint blind holes should have the thickness of the bottom greater than 20% of the hole diameter in order to eliminate surface defects on the opposite surface. A better design is to ensure that the wall thickness remains uniform and that there are no sharp corners where stress concentrations can be generated.

Another design consideration that can affect the quality of the part is the distance that is left between two holes. This also applies to the distance left between a hole and the outer edge of a part. The minimum distance should be twice the wall thickness or twice the diameter of the hole, whichever is greater. Figure 16[1] shows the details of this design feature.

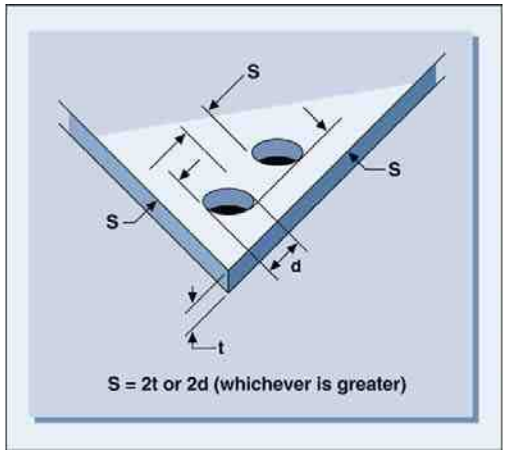


Figure 16 (Hole spacing)

One of the most important design features that can affect the electroplating quality is that of radii and corners. As we discussed earlier sharp edges and corners should be avoided for plastic parts that are going to be electroplated because of the buildup of metal on these surfaces. In general, the largest radii possible should be used in every area of a part. Generous radii help reduce molded in stress concentrations in a part. The radii should normally be designed between 25 and 60% of the nominal wall thickness. This normally translates to a minimum radius of 0.5 mm and all sharp corners should have a radius of 0.125 mm minimum.

The outside corner radius is very important because if improperly designed stress will be introduced in the plastic as the melt flow passes around the corner. Figure 17[1] shows how the outside corner radius should be equal to the inside radius plus the wall thickness. In this way, the wall thickness is kept uniform throughout the corner and stress is minimized during the molding process.

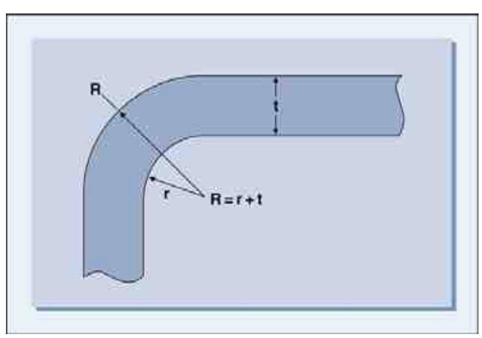


Figure 17 (Corner radius)

While the outside corner design is important, it is also critical to avoid sharp inside corners. Due to the difference in the ratio of area to volume of the polymer at the outside and the inside of the corner, the cooling at the outside is better than the cooling on the inside. As a result, as can be seen in Figure 18[1], the material on the inside exhibits more shrinkage and therefore the corner tends to deflect.

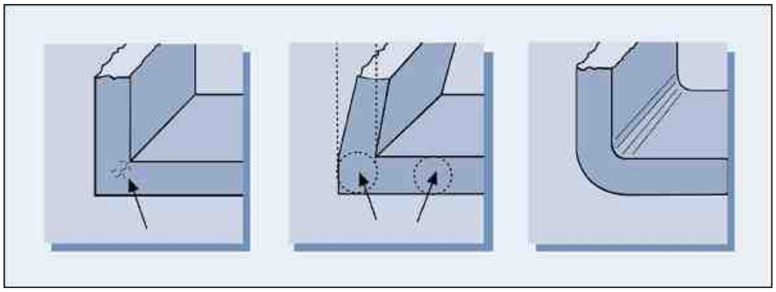


Figure 18 (Inside sharp corners)

Sharp corners result in high molded in stresses, poor flow characteristics, reduced mechanical properties, increased tool wear and surface appearance problems, and therefore should be avoided.

The draft angles designed into the different features of a plastic part are also critical to proper injection molding conditions. Parts with inadequate draft will tend to stick in the mold, resulting in difficulty ejecting the part. The resulting stress that is created in the areas of the ejection core pins can create problems in appearance, dimensional integrity and thermal-cycle performance of the final part. Typically 1 to 3 degrees of draft angle should be specified for smooth walled parts, with a minimum of 0.5 degree draft per side recommended. For textured sidewalls it is necessary to increase the draft by 0.4 degree for each 0.1 mm of texture depth. This is necessary in order to eject the part without disturbing the texture pattern.

In summary, the major part design guidelines that have a large impact on the injection molding process can be stated as follows:

1. Use uniform wall thicknesses throughout the part.
2. Use the least wall thickness that will meet the needs of the process, material selection and product design requirements.
3. Use generous radii at all corners and intersections.
4. Design parts with the maximum draft to facilitate easy ejection from the mold.
5. Instead of increasing wall thickness use properly designed ribs and gussets to improve part stiffness.

6. Wall thicknesses of bosses should be less than 60% of nominal wall to minimize sink on the Class A surface.
7. The inside radius should be at least half the part wall thickness on corners.
8. The outside radius should equal the inside radius plus the wall thickness on corners.
9. Sink can be minimized by maintaining rib thicknesses to 40 to 60% of the walls they are attached to.
10. Draft angles for ribs should be a minimum of 0.25 to 0.5 degree of draft per side.

GENERAL DISCUSSION OF INJECTION MOLDING INFLUENCES

Just as important as selecting the proper electroplating grade of ABS or PC/ABS, is selecting the proper molding conditions to produce a particular part. Because of the unique design of each part, every plastic part has a unique set of molding parameters that will result in the best overall performance. There are many ways in which a particular part can be processed, but compromises are always being made. It is up to the electroplater, working in close collaboration with the molding process engineer, to validate the platability and performance of each new product that is launched. Past experience will give you a starting point for processing, but only through trial and error can you determine the best overall injection molding processing to be used on a particular product.

After material selection, the four most important characteristics of a molded part that determine the final properties are polymer orientation, polymer degradation, free volume (molecular packing and relaxation) and cooling stresses.(2) It is these characteristics that influence the dimensional stability and thermal cycle performance of injection molded parts.

The plastic polymer can be degraded due to excessive melt temperatures or if it has been exposed at high temperatures for a long period of time, due to machine downtime for example. In extreme cases, high shear rates can also cause the polymer to be degraded. Degrading the plastic resin can result in electroplating problems, such as limited adhesion of the metal layers due to inadequate surface etching.

The amount of packing or hold pressure that is used during the molding cycle can influence the molecular packing and relaxation in a part. It is important for the polymer chains to be able to relax during the cooling cycle. If not, the dimensional stability at elevated temperatures if affected and this could contribute to thermal cycle failure.

Polymer orientation is often erroneously called molded-in stress. Stress is different than polymer orientation and should not be confused. Stress results from improper mold packing or uneven cooling in the mold after filling. Cooling stresses can result in the surface of the part being under compression, while the core is in tension. If we were to examine stress on a molecular level we would find that it is a result of deformation at the bonds between atoms. [(2)] Orientation on the other hand, simply refers to the alignment of polymer chains, not whether they are stretched or not. When polymer chains are allowed to relax they take on a more spherical shape, as opposed to when they are aligned similar to the grain in a piece of wood.

When plastic is injected into a mold it freezes at the colder surface of the mold cavity, resulting in an underlying melt flow that stretches and orients the polymer chains as the melt progresses through the mold cavity. The part design can have a very significant impact on this melt flow and the resulting polymer orientation. After the cavity has been filled, packing and cooling takes place. The stretching and shearing begin to dissipate and the polymer begins to relax at various degrees throughout the part. The overall net orientation is the difference between what was generated during the filling stage and how much it is allowed to relax during this packing and cooling stage.

The speed at which the plastic is injected into the mold cavity will greatly affect the amount of orientation in a part and where in the part the orientation is the greatest. A fast fill speed will put more orientation on the surface of the part and less in the core. A slower fill speed minimizes surface orientation as the mold has more time to cool the melt while it flows through the cavity. Generally, a slower fill speed is advantageous to pass thermal cycle testing.

Another factor which affects orientation of polymers is the melt temperature. Hotter melt temperatures result in less orientation. Because hotter melt is less viscous it reduces the stretching and shearing forces that cause orientation. Hotter melts also cool slower; resulting in more relaxation after total fill has been achieved. Figure 20[(2)] shows the combined effect of fill rate and melt temperature, not only on the surface but also in the core orientation.

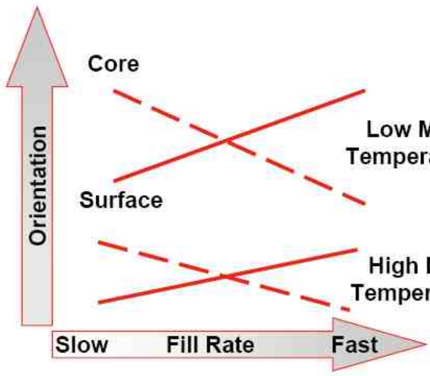


Figure 20

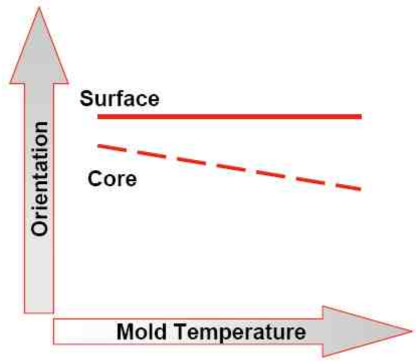


Figure 21

The actual temperature of the mold surface has less of an effect on the polymer orientation than either the fill rate or the melt temperature. A hotter mold tends to reduce the orientation in the core because of slower melt cool down which allows for more relaxation of the polymer chains. See Figure 21 above. (2)

Increasing the packing or hold pressure will generally increase part orientation because the polymers are not allowed to relax as much during cooling. See Figure 22. (2)

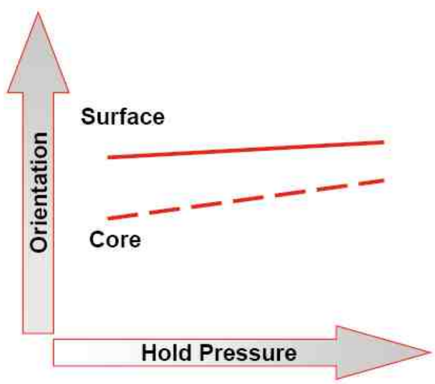


Figure 22

Another significant issue is the tendency of parts to warp at elevated temperatures. This tendency can lead to thermal cycle failures in some cases. When a part has been processed under molding conditions that generated molded-in core orientation and cooling stresses the part may warp. As can be seen in Figure 23(2) a higher melt temperature and a faster fill time will reduce molded-in orientation and thus reduce warping in a part.

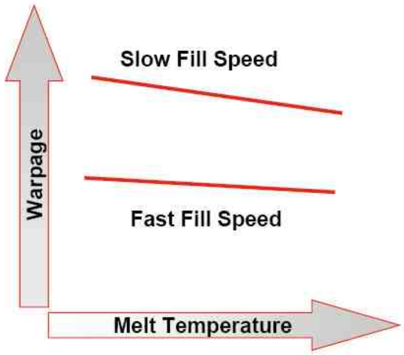


Figure 23

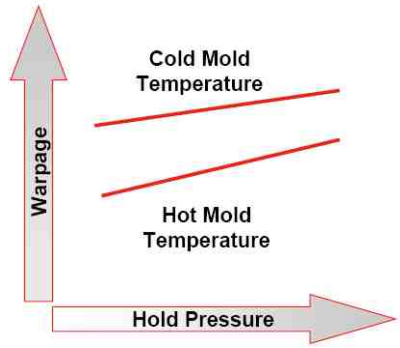


Figure 24

With a colder mold temperature polymer orientation has less time to relax and fast cooling of the melt introduces cooling stresses into the part. Figure 24[(2)] shows the effects of not only mold temperatures, but also packing pressure on the warping of a part.

Molding for electroplating is much more difficult than just molding for painting or for functional, non-decorative parts. The surface appearance, dimensional stability and final metal adhesion are all equally important. The final molding conditions are a compromise to achieve the best possible looking part, which will also pass the thermal cycle testing required. The adhesion failures that are normally observed, take place between the thin top layer at the surface and the plastic layer just underneath this thin surface layer. This area of a part is normally referred to as the boundary layer.

The strength of this boundary layer is determined by the orientation which is generated during the melt flow through the cavity. If orientation in this area is minimized, adhesion is maximized. Once again, the two factors contributing to this orientation are the melt temperature and the rate at which the cavity is filled. Slower fill rates as discussed lower surface orientation and result in a stronger boundary layer. Higher melt temperatures allow for relaxation of the polymer, especially during slower fill rates. Figure 25[(2)] graphically shows the impact of both the fill rate and the melt temperature on adhesion.

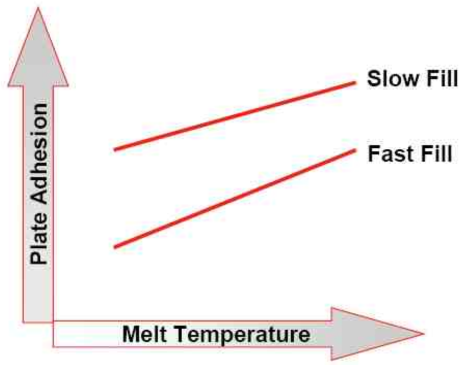


Figure 25

In Figure 26 you can see a picture of a copper-nickel-chrome plated instrument cluster bezel that has failed thermal cycle testing. Note the blister pattern of the metal deposit shown in the right hand view. Part of the failure investigation involved chemically removing the metal layers to reveal the plastic substrate. Figure 27 shows the obvious boundary layer failure in the plastic. The light colored areas are actually the skin of plastic that separated during thermal cycle testing.

Figure 26