How Voids can occur in Plastic Injection Molded Parts

An ideal injection molded part would be designed to have walls which are all a constant thickness. Unfortunately not all injection molded parts are designed this way due to part function requirements. It is important that part designers to be aware that all thick material sections of a part are prone to either sink or voids. This article will review part design practices that should be avoided in order to prevent the formation of voids and sink, as well as how material choice may affect the chances for formation of voids during molding.

Voids commonly occur in thick material sections in injection molded parts when the molecules on the outer skin of the part cool and solidify faster than the molecules internal to the part. If the part is not able to shrink inward due to the outside skin being frozen, the molecules will instead be pulled toward the surface of the part, creating in internal void.

The causes of void formations in plastic injection molded parts are generally due to thicker material areas incorporated into the part design, as well as material molecular structure. Figure 1 below illustrates a part designed with a core that steps down. This results in a very thick area in the part, where voids can easily form. A quick and robust way to remedy this situation is to core the part as shown in figure 2. This same practice is represented in figure 3 in a rectangular shape.

Figure 4 below depicts a typical scenario where two walls that come together with different wall thicknesses create a large mass of material. This large mass of material can easily lead to either sink on the top wall or a void in this area. The best way to avoid this situation is to ensure that the rib and similar features are not greater than two-thirds the thickness of the main section of the molding. Also, adding a taper to a thinner section of the rib can further reduce the formation of sink and voids.

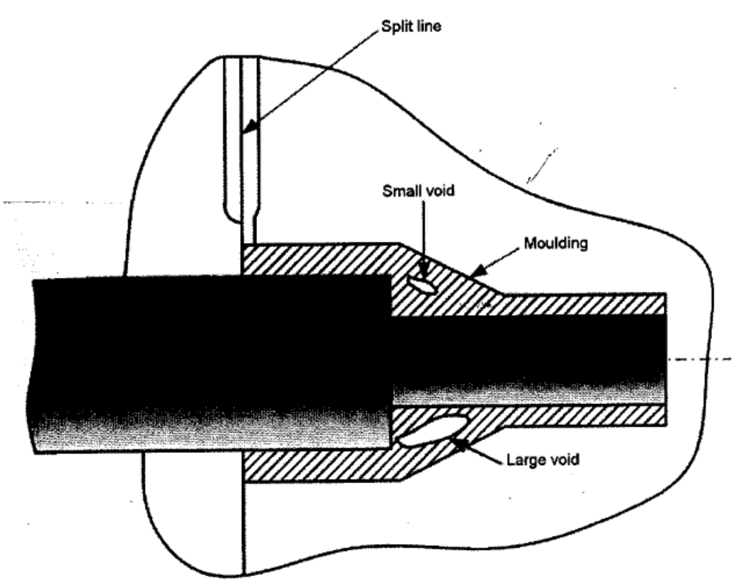

Figure 1. Thick Sections Cause Voids. Taken from *The Mould Design Guide* by Peter Jones. p. 51

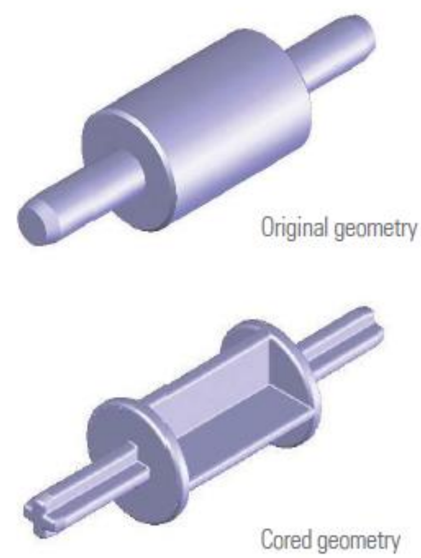

Figure 2. Core out parts to eliminate thick walls. Taken from *Design for Modability* by Protomold. p. 4

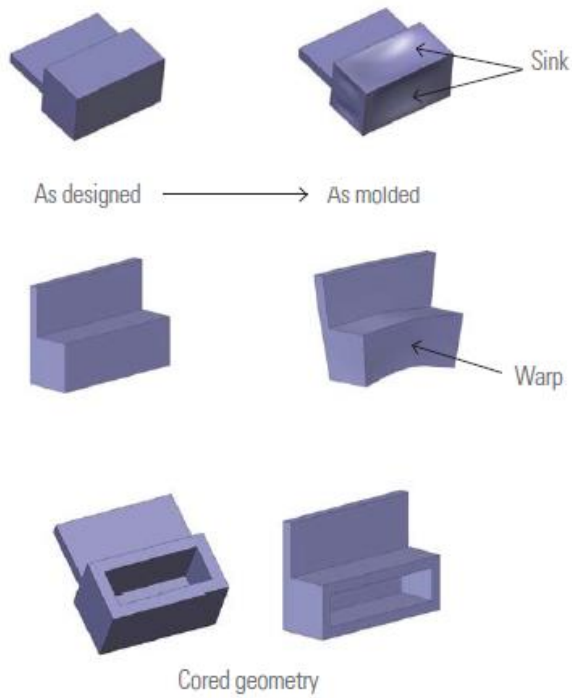

Figure 3. Maintain constant wall thickness. Taken from *Design for Modability* by Protomold. p. 4

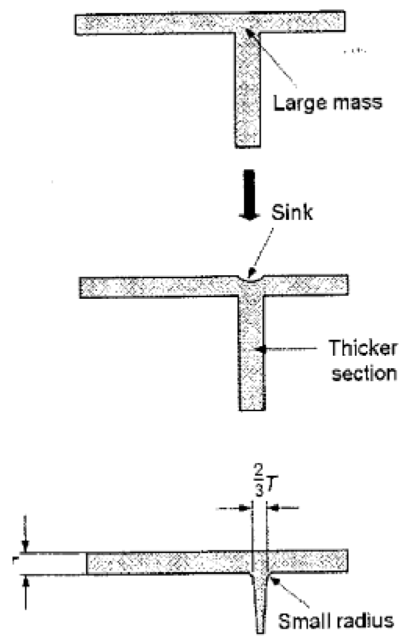

Figure 4. Shrinkage due to large mass of material. Taken from *The Mould Design Guide* by Peter Jones. p. 49

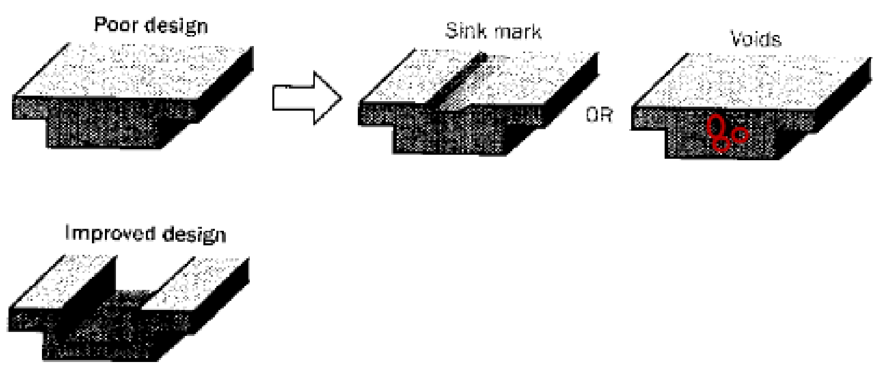

Figure 5. Sink marks and voids created by material shrinkage. Taken from *Moldflow Design Guide* by Jay Shoemaker. p. 241

Material selection of an injection molded part can also play a part in the probability for a void to form. When a part is injection molded, shrinkage will always occur due to the density of the polymer changing from the processing temperature to the ambient temperature. Designers should keep in mind that semi-crystalline materials are considerably more prone to shrink than amorphous materials. This is due to the fact that the molecules in crystalline material will arrange themselves in an orderly manner once they are cooled below their transition temperature versus an amorphous material.The more shrink that occurs in the part during the injection molding process the more prone the part is to the development of voids; specifically in thick walled areas. The addition of fillers with a high l/d ratio, specifically glass fibers, in any material can also increase the development of voids do to anisotropic shrinkage or unequal shrinkage of material in the flow direction verses the across flow direction. Fillers with low l/d values may help to prevent voids, as it acts as a physical barrier to thermoplastic shrink. Yet, this is dependent on proper “wet-out” between the filler. Poor wet-out leads to voids along the perimeter of the filler particle(s).

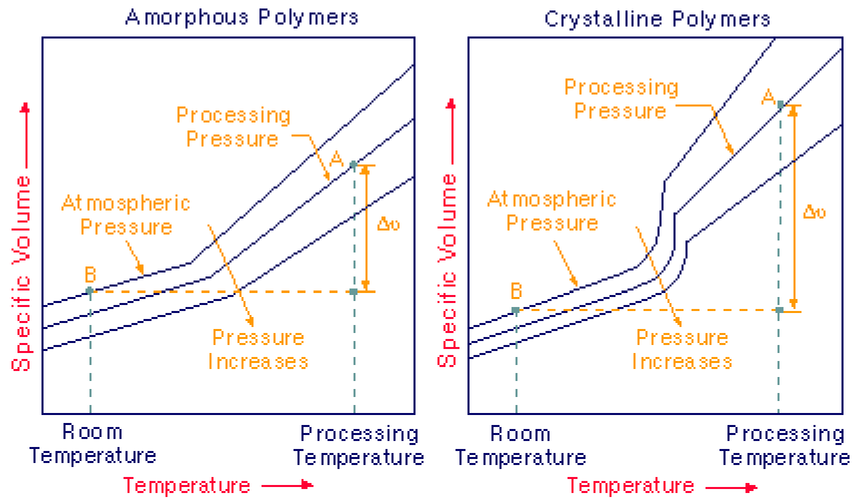

Figure 6. Shrinkage of Amorphous vs Crystalline Polymers. Taken from *Shrinkage and warpage* by Santa Clara University of Engineering

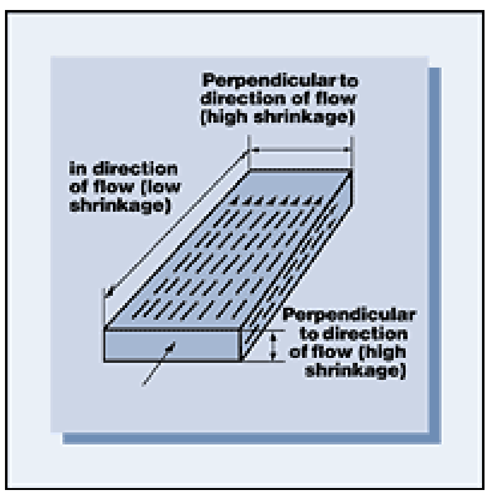

Figure 7. The relation of glass fiber reinforced plastics and the orientation of the glass fibers. Taken from *Design* by DSM Bright Science

In conclusion, plastic part designers should carefully consider the risks involved when designing parts that require thick wall sections, crystalline materials, or materials filled with glass fibers. In any of these cases, the designer should work very closely with the intended molder for parts and the tool builder to best determine the areas where voids could potentially form. Design modifications should be considered in order to mitigate the potential for void formation, or the void-prone areas should be moved to a section of the part where strength is not critical for the particular application.

References

1. Jones, Peter (2008). *Mould Design Guide*, Shawbury, United Kingdom: Smithers Rapra Technology Limited.
2. Shoemaker, Jay (2006). *Moldflow Design Guide*. Munich, Germany: Hanser Publishers.
3. DSM Bright Science. Brighter Living (2011). Design. Retrieved October 4, 2011, from http://www.dsm.com/en_US/html/dep/Shrinkage.htm
4. Protomold (2007). Designing for Moldability. Retrieved October 4, 2011, from http://www.onlinetmd.com/FileUploads/file/PM_Design_Moldability.pdf
5. Santa Clara University Engineering Design Center (2006). Shrinkage and warpage. Retrieved October 4, 2011, from http://www.dc.engr.scu.edu/cmdoc/dg_doc/develop/process/physics/b3500001.htm