# Plastic Product Design Tips — Scientific Molding

Injection Molding is one of the most prevalent methods for producing plastic parts in low and high volumes. This process is very popular since it is an economical, efficient and highly precise manufacturing method. There is potential for in-mold and downstream automation, and the process itself produces minimal waste. To provide these benefits, the molding company requires sophisticated machinery and tooling.

When parts or assemblies are developed, ensuring the part styling and function objectives is the main focus. Achieving a consistent part design requires a repeatable process.

*Scientific Molding* is a disciplined, systematic approach to injection molding based on what happens to the plastic during the molding process. The molder must be able to control the material's viscosity, or its resistance to flow, during the filling and packing portions of the process, and this starts with the mold design. Following these design recommendations will help increase the moldability of the parts.

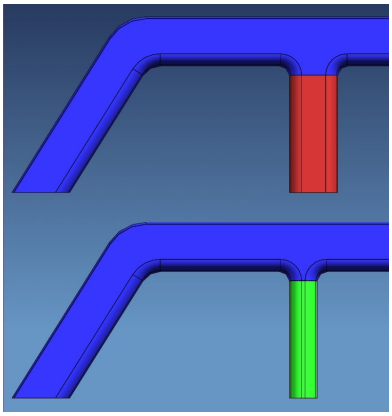
## *Aspects of Scientific Molding*

**Nominal Walls** provide a uniform melt flow and maintains pressure through the part. Uniformity in the wall cross-section provides the potential for consistent cooling during the molding process. Uniformity also helps to consistently reduce post-mold material shrinkage.

**Corner Radii** help allow for uniform pressure. Improper corner radii (too small/large) may cause stress points, which can result in warping and lead to part failure.

**Proper Draft Angles** decrease molding cycle time; reduce stress caused during ejection of the part from the mold; and is required to allow features such as texture or ribs to release from the mold. Proper draft also can reduce tool maintenance.

**Projection vs. Nominal Wall**
***Proper thickness*** at nominal wall decreases residual stress, helps diminish or eliminate sink marks, and improves the strength of a feature or the entire part. Proper thickness typically reduces cycle time.
***Projection height*** (shorter = better) – Proper design allows for adequate flow into the feature, should have provisions for adequate venting, which could require part design modifications. Improper design of projection may increase cycle time and cause other problems (e.g., strength vs. warp).
***Spacing Between Projections*** (more = better) – Tight spacing between projections can cause stress on the part, increase cycle time and cause sink marks.
***Free Standing Projections*** – Assess to confirm that these projections will not impact material flow or trap air in the mold during filling, causing additional part ejection concerns. Such projections should be properly cored out (60 -75% of nominal wall) and have no sharp corners.



Example of a Projection. Thin ribs (bottom) decrease residual stress and reduce cycle time.

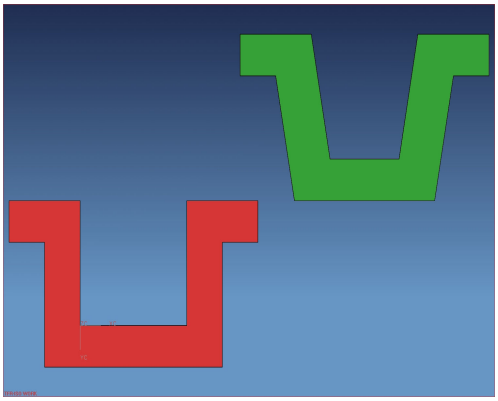

EMPIRE PRECISION PLASTICS

500 Lee Road, Suite 400
Rochester, NY 14606
www.empireprecision.com

Phone: 585-454-4995
Fax: 585-454-0449
E-mail: sales@empireprecision.com

## *Aspects of Scientific Molding, cont'd.*

**Depressions:**

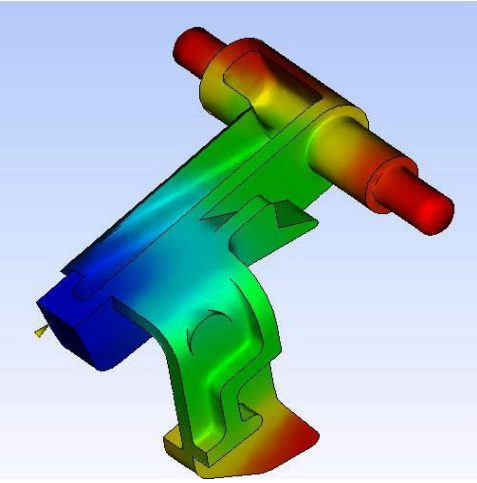


***Weld Lines*** - Sometimes a depression in the part design will cause problems in proper filling and packing of the parts. This can have a negative impact on part strength or appearance.

***Core Pins*** - Both closed and open ended holes in part designs should be assessed to see if the diameter-to-length ratio could cause a problem, or if the direction of the resulting material knit line affects part strength or appearance.

***Location*** - Must be reviewed to assess the shape and space between features. Here again, knit weld lines, heat concentration, sharp corners, and their effect on wall cross-section may all potentially cause problems. The part could be stressed or distorted, and its strength or appearance affected.

***Proper Draft*** - Decreases the cycle time, reduces stress caused by ejection and can lower tooling maintenance costs.



*This small precision plastic part is properly cored, maintaining a uniform wall cross section and accurate gate location.*

**Advanced Items:**

***Gate Location*** must meet the flow, functional and aesthetic requirements.

***Finish*** such as high gloss or texture may have special tooling and handling requirements. .

***Ejector Style*** must be sufficient to remove part without damage or negative impact on functional or aesthetic requirements.

***Parting Lines*** allow the part to remain on ejector side of the mold without stressing the part or impacting function or appearance.

***Direction for Flash at Core Pins and Tool Shut Off Areas*** must be assessed for impact on function, dimensions or appearance.

***CTF Dimensions*** can be reviewed to help investigate proper mold design, testing or gauging.

The most important design factor is maintaining a uniform wall thickness**.** Without this, various molding and part problems occur, including filling problems (air traps or fill hesitation), warping, surface imperfections (sinks or finish flaws), and internal stress. All can cause premature failure of the part or assembly.

***We invite you to meet with a knowledgeable, trusted advisor who can guide you through multiple considerations using cost reduction and improved performance as the main criteria.***

SOLUTIONS FOR YOUR CHALLENGING PARTS

500 Lee Road, Suite 400
Rochester, NY 14606
www.empireprecision.com

Phone: 585-454-4995
Fax: 585-454-0449
E-mail: sales@empireprecision.com

EMPIRE PRECISION