3D SYSTEMS

Design for Manufacturing Analysis

How the DFM Analysis Will Save Your Project

3D SYSTEMS

Table of Contents

1 Introduction

When designing injection molded plastic parts, engineers are often tasked with a common problem: making sure their design is optimized to meet the guidelines of the process, while simultaneously maintaining the design's intent. More often than not, this becomes a two-way street, and a strong relationship between engineer and toolmaker can be invaluable.

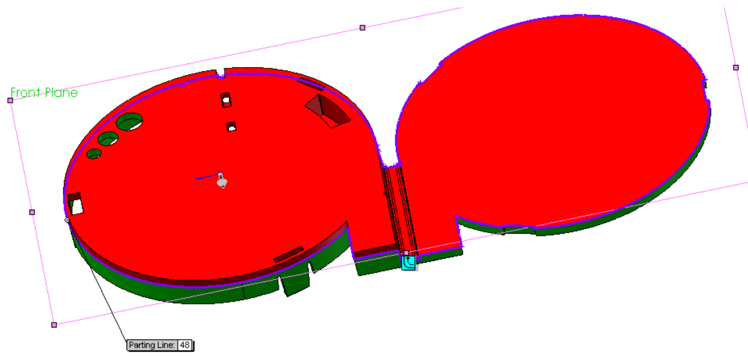
"I'd estimate that about 80% of new tooling projects we see require some form of modification to the 3D data," says Andy Richardson, Senior Director of Inside Sales and Service. "Whether it's a draft angle, a thickness concern, or a shut-off condition that would result in decreased tool life, even the most experienced engineers can overlook particular details."

In the modern era of manufacturing where time is increasingly becoming more and more critical in the race to get product to market, knowing how to optimize a design for molding is a key component for any product development group. The team at Quickparts strongly believes that early detection of a questionable design goes a long way. Quick feedback, technical banter, and an in-depth Design for Manufacturing analysis (DFM) can save a busy team of engineers time, money and, perhaps most important, the headache of endless revisions to an upcoming product.

Quickparts' Design for Manufacturing analysis encompasses injection molding requirements for specific part geometry. Sections are dedicated to: Parting Line and Orientation, Draft Analysis, Features Requiring Side Actions, Rib/Bosses to Wall Thickness Ratio, Thickness Analysis, Warp Analysis, Shut-off Conditions, Thin Tool Conditions, Areas Requiring EDM, Weld Lines, as well as a comprehensive review of any provided 2D prints. Each page is color-coded in order to display a unique level of concern. Much like a traffic light, green means go (good design), yellow means yield (review of minor issue), and red means stop (unable to proceed without modification). A more detailed look into each facet of the DFM process is listed below.

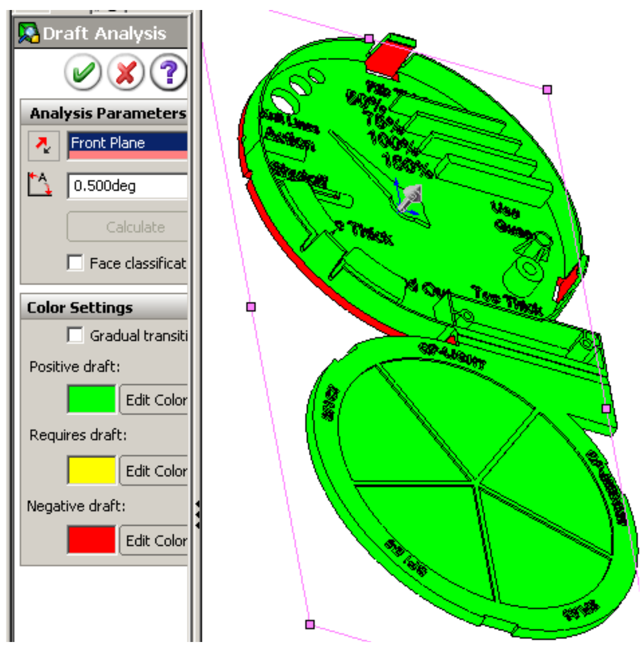
2 Part Orientation and Parting Line

Simply put, a parting line is the place where each part of the mold halves meet. Parting lines occur as a result of small gaps between the two mating faces of each mold half. Parting lines generally show up as a tiny raised line on the surface of a plastic part. When the part is ready for ejection and the mold halves separate, excess plastic, known as gate vestige, is left behind. Knowing ahead of time where the parting line will be located can be essential to part function, as the remaining gate vestige can interfere with sealing surfaces and critical mating features.



3 Draft Analysis

Knowing how and where to correctly apply draft can be the difference between a strikeout and a home run. The existence of draft on vertical surfaces and for the removal of slides enables a part to be easily removed from the mold. Generally speaking, draft requirements differ from part to part, and some designs allow for little to no draft; however, most designs call for at least a half degree of draft on vertical walls. When designing an assembly, it is important to account for draft consistency from one design to its corresponding mate. More aggressive textures require more draft, and vice versa. It's perfectly acceptable to submit a preliminary design without draft; the intent of the DFM review is to educate the engineer as to where, and sometimes why, draft is required in specific areas. The slightest of draft modifications can be the difference between a red and green slide.



4 Features Requiring Side Action

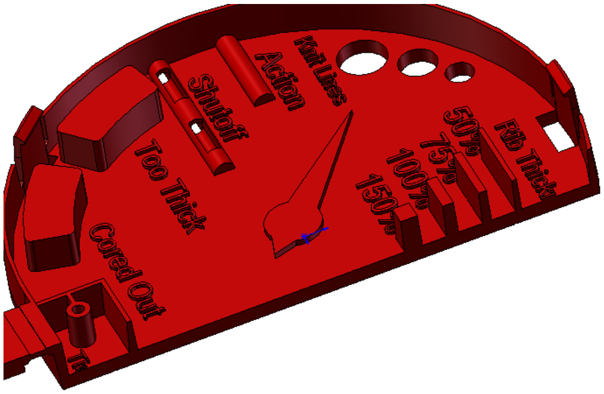
Parts with features that are not located within the line of draw of the tool will require mechanisms to facilitate part production. The term side action can take on a variety of synonyms (hand loads, slides, cams, etc.), all of which can play a major role in tooling and piece pricing. Modifying a design to eliminate a side action can allow for quicker delivery and a less prohibitive cost.

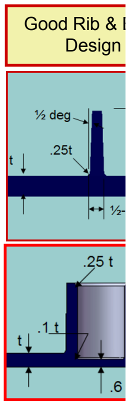
Identifying these features in the early stages of design can go a long way. There are many ways to design around a side action, but the first step is identifying where the features exist, and then understanding why the side action is required. Once the initial education is in place, design alternatives can be explored, paving the way for a cost saving opportunity.

---

5 Ribs/Bosses to Wall Thickness Ratio

The backbone of any plastic part lies in the integration of ribs and bosses. Not only do these features provide structural integrity, but they also assist in reducing the amount of potential warp a part can have. However, a poor rib and boss design can lead to drastic results in the form of undesirable cosmetic sink. A good rib/boss design will generally show a thickness of 50%-60% of the corresponding nominal wall, with the height of these features not exceeding three times the thickness of the nominal wall. At times, exceeding this ratio can be unavoidable, but with early feedback via DFM, there should be no surprises when molded samples are delivered. Educating a part designer up front via graphical representations and screen shots showing where and why sink can occur can lead to alternative options surrounding each area in question. The resulting T1 samples will surely cut down on costly mold modifications that could have occurred had preliminary feedback not been provided.

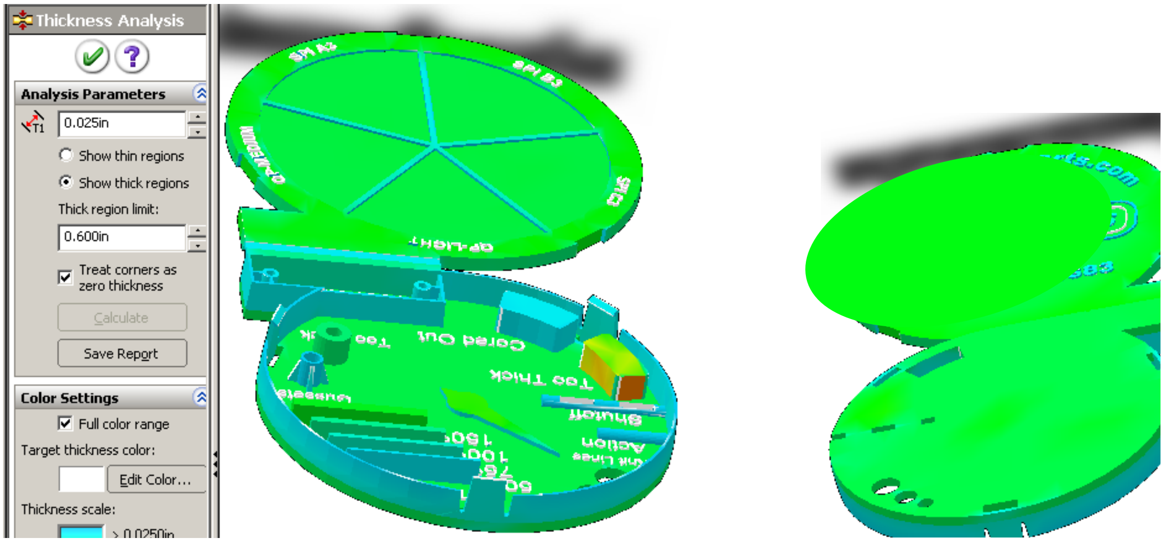




3D SYSTEMS

6 Thickness Analysis

Generally speaking, each grade of plastic resin has a unique recommended minimum and maximum wall thickness. Understanding the constraints of prospective resins can give any engineer an edge when designing a part. Certain resins are able to accommodate thinner-walled designs, while others cater to thicker, bulkier designs. When selecting a resin, the best course of action is to contact a material representative. Based on the desired grade, a DFM review can provide much-needed insight into the wall thickness limitations of a design, thus clearing the path for uninhibited part design.



7 Warp Analysis

As mentioned earlier, ribs and bosses play a crucial role in supporting a plastic part. Parts with open faces and long, thin unsupported walls can lead to warp, which can be a disaster when trying to assemble a group of designs. Knowing when and where to add a rib to an unsupported wall can lead to a much better design. A DFM analysis will certainly identify potential areas of improvement by taking these concerns into account. The last thing an engineer wants to stomach is a circuit board housing shaped like a crispy potato chip!

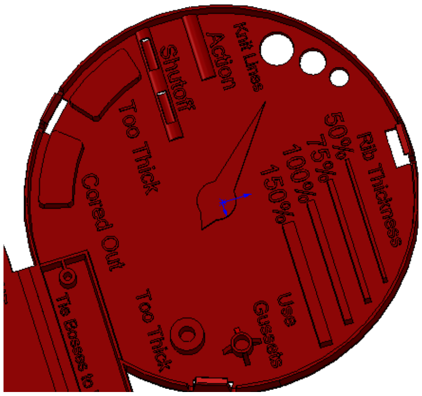
8 Shut-Off Conditions

A shut-off refers to an area on a part that is created by mold halves coming together (metal sliding on metal). A minimum of three degrees of draft is required on all shut-off conditions. Anything less could potentially result in shortened tool life, excessive flash, and part ejection failure. In the latter example, the mold would lock up and be unable to open. An upfront understanding of how shut-off features play a key role in the injection molding process is crucial; otherwise, a rude awakening is certainly in store. Parts delivered with excessive flash, along with a mold life barely able to exceed 1000 parts, are never a good sign.

---

9 Thin Tool Conditions

Making sure a part design does not include any thin tool conditions will ensure a longer tool life, consistent geometry from part to part, and can avoid hefty maintenance fees down the road. As a rule of thumb, any area on a part that requires tool metal of less than 0.050” (1.2mm) has a higher probability of tool failure during a future run of parts than features exceeding this benchmark. As a general rule, the depth vs. width ratio of a feature should not exceed 3:1. These marks are by no means a showstopper and can be overcome with good tool design, but they may increase the cost of producing your mold. Keep in mind that the thinner the tool metal, the lesser the likelihood of a longstanding, reliable mold.



3D SYSTEMS

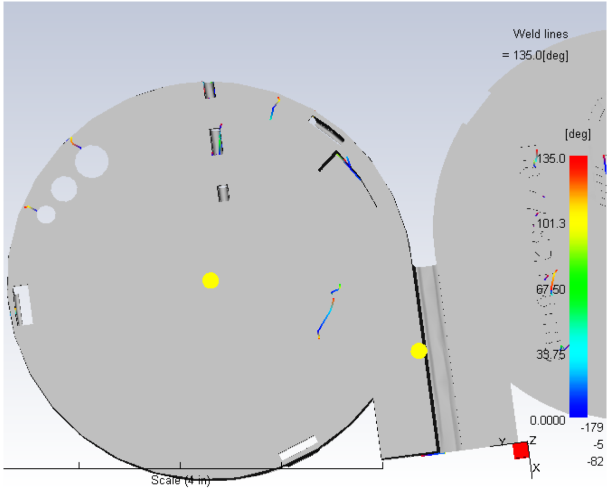
10 Areas Requiring EDM

EDM stands for Electrical Discharge Machining, sometimes referred to as "burning" or "sparking." The process is used when a desired shape, often very sharp corners or deep ribs, cannot be constructed within the mold via the traditional CNC machining operation. Material is removed from the mold by a series of rapidly recurring current discharges between two electrodes, separated by a dielectric liquid and subjected to an electric voltage. Features with sharp corners or a height-to-thickness ratio of 8:1 require EDM. Why is this important? EDM, although a generally common means of tool manufacture, does add to the upfront cost of the mold. Knowing ahead of time which features will require EDM can allow an engineer to round off sharps, avoid the extra work, and save time and money.

---

11 Weld Lines

Whenever plastic must flow around an obstruction, such as a hole, it will inevitably meet up and form a weld line. This is usually a slightly visible line and is often mistaken for a crack in a part. Avoiding weld lines can be tricky, and sometimes an alternative gating strategy can be implemented to alter the flow of material in order to save a cosmetic or weaker surface. Parts with multiple obstructions to material flow tend to have more weld lines than others. Early design feedback can certainly limit the presence of weld lines.



3D SYSTEMS

12 2D Print Review

Although a 2D print is not required, it is always helpful, especially if there are critical features that justify special attention. Standard part tolerance expectations are +/- 0.005" (0.1mm); however, it is possible to achieve tighter tolerances on a case-by-case basis. Drawings with such callouts are highly encouraged, as an overwhelming majority of molders build specifically off of the 3D CAD model. A thorough review of any 2D print via DFM analysis will provide a better understanding of the expectations of both the customer and the supplier, both of which will lead to a more streamlined process in which the best product can be delivered.

With any molded part design, either in its infancy or nearing its production release, Quickparts strongly encourages all customers to submit their design for DFM. The earlier the feedback, the easier it will be to optimize, which in turn will make the injection molding process smoother, thus saving time, money, and stress.

Contact us now to have your design evaluated. Our team of experienced salespeople and engineers are happy to help, no matter which stage of the program you find your design.

www.3dsystems.com

3D Systems Corporation
333 Three D Systems Circle
Rock Hill, SC 29730