**How Part Characteristics Impact Product Cost**

As product parts become more complex, production costs typically increase. Creating quality products at low costs gives companies a critical competitive advantage. A product's design determines how well it will function and it's overall marketability and sales success. Product design determines 80% of the overall cost and significantly impacts quality and reliability. Those who separate the cost estimation process from the design process can create budget constraints, thereby limiting designer flexibility. Additionally, an inappropriate cost estimation may lead to expensive iterative redesign. Both of these mistakes lead to ineffective product development. To avoid these scenarios, designers must plan for customer preferences and cost, using cost estimation tools to allow for calculated trade-offs between product characteristics and their cost. Costly "part characteristics" both increase cost and reduce manufacturing and production efficiency. Some of these key factors include design issues that slow manufacturing cycle times, including wall thickness, gate selection, flow narrowing and tolerances.

**Wall Thickness**

Wall thickness is a balance between product cost, product performance, and shipping/transportation cost (for both raw material and finished goods). If a product is poorly made, it can also affect user value perception. On the other hand, if a designer overcompensates wall thickness, mistakenly assuming that thicker walls always add strength, this can lead to a number of production problems. Specifically, incorrectly specified wall thickness can lead to slower production line rates and mold redesigns, which ultimately lead to higher production costs.

One of the biggest production problems injection molding companies may notice is inappropriate wall thickness or relationship between wall thickness and rib structure. Designers sometimes add wall thickness when they want to add strength. However, building a rib structure with thinner walls is sometimes a more durable *and* cost-effective solution. Other times molders may thicken walls to take up space in a mold, wanting to ensure that the material flows to the finished part. It takes an experienced injection molder to understand the filling process and establish the appropriate adjacencies, gates and pathways to meet the functional and cosmetic requirements of a product.

As designers create their products, they must keep in mind the relationship between wall thickness, material cost, and cycle time. Thicker walls mean extra material and longer cycle times. Wall thickness can increase cooling time, and if proper cooling can't take place, then parts will heat up and begin to control the cycle process. Sometimes blowing agents can be added to a product that requires thicker walls in order to speed up the cycle time. Increasing cycle time can reduce inventory, improve product quality and allow companies to meet customer demand faster by quickly responding to orders. If possible, thinner product walls are preferred because the product can be produced faster with less materials, which are the two largest variables associated with cost.

**Gate Selection**

Depending on a product's part characteristics, different gates need to be used during the injection molding process. These gates are either manually or automatically trimmed. Manually trimmed gates require an operator to manually cut off the parts from the runners after each cycle. These gates are sometimes used if the gate is too bulky to be automatically cut, a shear-sensitive material (such as PVC) is being used, or if the product requires simultaneous flow distribution across a wide space. Automatically trimmed gates cut the gates as soon as the tool opens to eject the product part. These types of gates are used to reduce cost by removing a manual operator, maintain consistent cycle times and minimize gate scars.

There are several different types of gate designs:

*Edge Gate*

The edge gate is located at the edge of a part and typically used for flat parts. They can be used for medium or thick part sections and on multi-cavity two plate tools. They are also used for materials that flow harder into the mold. If the runner of the plastic that's coming to the part is still attached to the plastic part when it's done, then the gate size can be increased based on the ability needed to fill the cavity. Edge gates leave scars at the shearing lines.

*Sub Gate*

Sub gates are automatically trimmed and require ejector pins. They allow for gating away from the parting line, which allows more flexibility for optimal gate placement and only leaves a tiny scar on the part from the pins. Sub gates can help from a labor standpoint because an operator is not needed to sit at the end of the machine and cut each part off.

*Hot Tip Gate*

A hot tip gate is a hot runner gate, commonly found at the top of the part instead of the parting line. These gates are typically used for round or conical shapes that require uniform flow. In hot runner systems, plastic is injected into the mold through a heated nozzle, which means the material can stay melted right up to the part before it is cooled to the shape of the mold. Hot manifolds add about 20% to tooling costs, but can allow for quick payoffs. This process can be especially cost-effective for high runs because there is less need for labor at the press and these gates reduce plastic waste and reduce cycle times.

*Direct or Sprue Gate*

This is a manually-trimmed gate used on single-cavity molds for large cylindrical parts that need symmetrical filling. These gates are low-cost and low-maintenance, but the leave a large scar on the part. They cannot be used in smaller parts.

**Unintended Flow Narrowing**

Unintended flow narrowing can occur when designs cut off or narrow the flow paths or channels for the thermoplastic material. This is often a hidden issue until the final prototype stage as even seemingly benign design choices such as a tighter/thinner corner radius can impede material flow.  The potential for creating voids or under tolerance parts are not limited to corners and can include many design techniques such as:

* Providing angled transitions or tapers that help conduct the flow of the material from the mold runner.
* Using the proper radius to thickness ratio for corners and other areas where multiple planes or features meet (as noted above), and avoiding sharp corners when possible favoring gentle radii (rounding).
* Not reducing thickness more than is viable for the shrinkage property of the material used.
* Using the proper computer-assisted flow simulation algorithms, which can take a lot of the guesswork out of removing flow restrictions in a given design.

Plastic parts subject to unintended flow narrowing drive up cost due to a loss of material and manufacturing time

**Tolerances**

Maintaining appropriate tolerances often start with mating the right base material to the allowable dimensional variation between finished parts or sub assemblies. In general, the less deviation that can be tolerated, the lower the shrinkage of the material to be used. Also the designs feature can affect dimensional outcomes. For example, it may also be wise to “decrease” tolerances where alignment of multiple parts may be critical (for instance hinges, sliding components, snap together areas, undercuts, etc.). Most applications have three tolerance class divisions: normal, medium and fine. A medium tolerance ~~plastic~~ part costs 1.7 times more to manufacture than a normal toleranced part and 3 times more for a fine toleranced part.

**Other Design Issues**

*Nooks and Crannies*

Plastic designs not in the line of draw that cause an extra action in order to remove them can delay cycle times. Part ejection is impeded if a part has lots of nooks and crannies for the plastic to fill into.

*Regrind Restrictions*

Twenty to thirty percent regrind material can typically be used without affecting part quality in some applications. However, if a part with an edge gate needs a bigger flow area, for example, but there is an order restricting detached material to be reused back into the next part, cost can accumulate in the wasted material.

*Complex Set-up*

Design can also impact set-up and tear down times. For example, a mold with a lot of hydraulic cylinders or other auxiliaries could take several hours to set-up and break down. This requires extra labor and press time, which ultimately drives up cost.

Designers must be mindful of how their designs impact cost, while still ensuring a quality part. Low-quality parts, while potentially inexpensive to manufacture, can lead to expensive problems after sale, including excessive returns, part recalls and an inability to build a brand or a loss of customer loyalty due to dissatisfaction. These concerns must be addressed from the beginning: the design phase. Good part and product designs are even potential cost-cutting alternatives to offshore manufacturing. Working with experienced plastics designers and molders can help maximize budgets and ensure the best possible part quality.