Product Design Considerations

1) Uniform Wall Thickness, Coring & Mass Reduction

Since injection molding is employed as the shape forming process step, part designs can avoid the limitations of traditional metalworking processes. For example, machining involves the removal of material from a solid shape to get to the desired final component design. As a result, design engineers are limited to design decisions that can be readily produced on an economical basis and those which do not violate the design limitations of machining. The benefits of removing excess material for reduced part mass are generally not considered as this design approach would add incremental machining costs. In the case with plastic injection molding, design engineers have the freedom of starting with a "clean slate", and building up their component geometry by placing material only where it is needed for function and strength. This serves several benefits for the process and the customer. Any opportunity to limit the amount of material required in a component helps minimize the final part cost. Additionally, maintaining a uniform wall thickness throughout a component reduces the likelihood of molding process flaws, thus improving the overall part quality, cosmetics, and generally improves the resulting dimensional tolerances that the process can provide.

*Figure 1* illustrates several preferred geometries accomplished through coring to create uniform walls. You will also note instances where unnecessary material has been removed or cored out in areas with thick cross sections. Coring can be done either parallel or perpendicular to the parting line. *Figure 2* illustrates both types of coring. Coring perpendicular to the parting line (Section A-A) can be produced with cores, which are fixed features on either half of the mold. Coring parallel to the parting line (Section B-B) can be produced with slides, which are moving components in a mold. The slides are usually placed at the parting line and move parallel to it. Slides add complexity and costs to a mold, so if the design permits, coring perpendicular to the parting line is preferred approach.

*Figure 1*
Remember, when designing a part, or when coring out an existing design, maintaining a consistent uniform wall thickness throughout the part is the primary objective. Again, uniform walls are desired for higher precision, more repeatable dimensional capability, lower processing costs and improved aesthetics. If, however, varying wall thickness cannot be avoided, a gradual transition between differing wall thicknesses should be provided and every attempt should be made to avoid abrupt changes. Figure 3 provides a recommended wall thickness transition ratio for those situations when uniform walls cannot be achieved.

2) Draft - Where and When required
generally, components do not require draft. There are a couple of factors that contribute to this ability. Post molding shrinkage that occurs with plastic parts while they are still in the mold. This allows the part to be ejected before it can cool and shrink around cores and/or other mold cavity features. With these
Influences in mind, there are circumstances when draft should be provided in component designs. Figure 4 illustrates some of these circumstances.

3) Corner Breaks & Fillets

One of the intrinsic benefits of injection molding is the ability to produce corner breaks and fillets. Not only do corner breaks and fillets play several important roles in a good component design, but they also provide design engineers with design advantages not readily available in some metalworking processes. In addition to providing improved injection molded part quality, these design advantages include, improved part strength, elimination of stress concentrations, and softening of sharp corners for aesthetics and handling.

Typically, corners breaks should be kept larger than .005" radius. Internal and external corner breaks less than .005" radius will induce stress concentrations in the part and will be difficult to fabricate in the mold. Figure 5 illustrates an exception where a sharp corner is preferred over a generous radius. The figure shows a component and mold design that benefit from having sharp corners located on the bottom of the part. In this case, the sharp corners allow the part geometry to be kept in one half of the mold, which simplifies the mold design, reduces the mold cost, and does not jeopardize the part's strength. Should a radius be required along the bottom edge of the part, it can be readily produced, but it should be noted that the part must now have portions of it produced in each half of the mold. In addition to adding cost to the mold, the designer should expect a witness line around the profile of the part at the parting-line location.
4) Holes & Slots
Holes and slots can be readily produced by the injection molding process, and generally, can be accomplished at no additional cost to the piece price. However, adding these features does increase the cost and complexity of the mold. Also keep in mind that beyond representing obvious functional features, holes and slots can also be used to reduce part mass and provide uniform wall thicknesses.

It is important to be aware of the type and direction of a hole and how it could affect the cost and the robustness of the mold. Figure 6 shows several types of holes, their direction relative to the parting-line, and their impact on the mold. Holes that are perpendicular to the parting-line represent the easiest mold design approach and the lowest cost to incorporate in the mold. Holes that are located parallel to the parting line are readily applied, but the tooling costs more than holes located perpendicular to the parting-line as they require mechanical slides or hydraulic cylinders to actuate them during part ejection. Holes that are set at an angle to the parting-line are also possible, but the mold construction and the mechanism to actuate them becomes very expensive and in many cases the mold features mandate more frequent maintenance downtime and related upkeep costs.

Figure 6
Cores and slots that intersect one another can also create complex part features. However, when employing intersecting features, the mold construction and robustness must be considered. Figure 7 shows the advantages of using a D-shaped hole as an ideal seal-off surface for an intersecting hole. In this case, two flat surfaces are sealing against one another providing a tool that will be easy to maintain and less likely to generate unacceptable flash during the molding process. The alternative displayed in the figure shows the least attractive approach, which requires one of the cores to have a contoured or profiled face to match the core or hole that it will be sealing against during the injection portion of the molding process. In circumstances like these, the core orientation is critical and the feathered edges are likely to wear more rapidly affecting the shape and size of the molded feature. Mold flash is also a concern in these situations.

![Preferred vs. Not Preferred Seals](image)

**Figure 7**

5) Undercuts - External/Internal
External undercuts can be produced readily with the molding process. The component on the left in Fig. 8 shows an external undercut that provides relief for burrs on a mating stamped component. The feature by placing it in the mold design eliminates the need and related cost associated with removing the burr on the stamping. Essentially, an increased level of complexity can be provided in the part without affecting the part’s cost, while at the same time the part design eliminates the need for a secondary deburring or chamfering process on the stamping. Consideration of product assembly requirements should always be considered when designing components to be produced.

Internal undercuts are possible with injection molding under the right conditions. Internal undercuts that can be produced with a mechanical or hydraulic actuated slide can be readily produced. Figure 8 (bottom, right) shows a "T-slot" internal undercut that can produce with a slide. Figure 8 (bottom, right) also shows a similar component but the internal undercut requires a collapsible core. This undercut requires the internal feature to be large enough to accommodate a robust collapsible core. For components that are small and collapsible cores are often impractical and some times impossible to produce that is robust enough to take the injection pressures. Collapsible cores also provide challenges in maintenance to minimize flash.
6) Threads
Internal threads can be molded directly into the component using unscrewing cores. These mold features
and functions are costly to produce, and as a result, are only applicable to high volume applications. For
lower volume part applications, conventional tapping operations can be employed.

External threads can be molded directly onto the component thus eliminating the need for secondary
thread-forming operations. Molding external threads is almost always a more cost effective approach than
forming the threads with a secondary operation. Generally, a small flat, typically .005", at the parting line
should be incorporated into the design. The recessed flat, shown in Figure 9, will insure proper mold seal-
off and reduce the opportunity for parting-line vestige to interfere with component function. Without the
presence of a flat along the parting-line, you can expect problems with flash to develop in the root of the
threads within the production of very few parts. This will likely increase tooling maintenance and down
time.

Figure 8
7) Ribs & Webs
Ribs and webs are an efficient way to increase your part strength, and minimize the effects of dimensional variation caused by the substantial shrinkage. As with plastic injection molding, ribs and webs also improve the molding process and provide better dimensional control. Figure 10 details how ribs and webs can be added to improve the mechanical design and provide a more robust component.
Another application of ribs is displayed in Figure. 11, where they are used as a means to provide coring for part mass reduction without effecting the intended end use or strength of the component.
8) Knurling, Lettering, & Logos

Injection Molding is capable of producing knurling, lettering, logos, date coding or other designs directly into the component without added costs to the piece price. These features can either be raised or sub-surface. Figure 12 depicts various examples of this capability. Molding can provide high levels of feature detail. Including relatively sharp diamond knurling.

9) Gating - Types & Locations

Plastic injection molding require consideration for gate locations. In most cases, gates are located at the parting line of the mold. The impact of the gate location on the component must be considered during the design phase, as it can be a careful balance between manufacturability, part function, dimensional control, and aesthetics. Gates will leave a slight vestige, and should not be located on a dimensionally or a visually critical surface.

In general, the gate is placed at the thickest cross-section to allow the material to flow from thick to thin cross-sections. Additionally, the location of the gate(s) should be placed to allow uniform filling of the mold cavity.

Fig. 21-23 show examples of various gate types, as well as their typical use, and preferred locations.

Figure 14 shows an illustration of an edge gate. The following characteristics are typical for an edge gate:

- Gates are typically removed manually and are not suited for high annual volume applications as they require removal by manual or automated means. Regardless, the removal process is an additional process step and cost.
- Suited for low to medium annual volume applications.
- Recessed gates are preferred to minimize vestige above functional component surfaces.
- Normally located along the parting line.

**EDGE GATE**

![Figure 14](image)

**Figure 15** details a submarine or sub-gate, which has the following characteristics:

- The gate is automatically sheared off from the part during the part ejection portion of the molding process.
- Suited for low to high annual volume applications.
- Leaves minimal gate vestige or breaks off below the surrounding component surface.
- Sub-gates should be placed on a recessed surface to minimize vestige above functional component surfaces.

**Figure 16**

Illustrates a submarine or sub-gate to a removable post. This gating approach has the following characteristics:

- The gate is automatically sheared off from the post during the part ejection portion of the molding process.
- The post is removed after the part is out of the mold, and this removal process is not typically automated.
- The post is preferably located in a recessed pocket on the component so the post can be broken off, below the component surface.
- Suited for low to medium annual volumes.
- The post and related recess or pocket should be located on non-cosmetic surface.
Other gating techniques including, 3-plate molds with direct gating and hot-runner systems for direct gating.

10) Sink & Knit lines
A plastic injection molded part may contain sinks and knit-lines caused by improper part and mold design. Sink (a physical depression on the surface of a part) frequently occurs around thicker sections. The example shown in Figure 17 illustrates how sink can occur when a supporting rib intersects a wall. If the rib is the same thickness as the wall, the intersection of the two, creates a localized thick wall and is susceptible to sink. Decreasing the thickness of the supporting rib eliminates or reduces the potential for sinks. Generally, the thickness of the rib should be about 75% of the thickness of the wall.

Knit-lines can occur when two flow paths of material meet in the cavity and when the flow path is relatively long. Figure 18 shows a knit-line occurring in a cylindrical part with a core in the center and a single gate. The two flow paths have to go around the core before meeting on the opposite side. Due to the long flow path, the two flow fronts of material have cooled down, which creates a visually evident knit-
Figure 18 also shows how dual gating the part can substantially reduce and at times, eliminate visible knit-lines. You should keep in mind that visually negligible knit-lines on properly designed parts are superficial and do not represent a structural defect or part performance issue. Generally, knit-lines of this type have a shallow witness that is a little as .0005" deep to .005" deep.

11) Flash & Witness Lines

while designing a component, witness lines and areas of potential flash should be taken into consideration. Critical areas from both an aesthetic and functional standpoint should be assessed for possible effects of witness lines or for minimizing the potential for flash.

Witness lines are an unavoidable result of two mating mold components. Whether along a parting line, or where a core pin seals off against a slide or other mold feature, injection molded material under pressure will be imprinted with the witness mark of two pieces of steel meeting one another. Figure 19 illustrates the typical witness line to be expected along a parting line. In this example, the parting line is just above the fillet and the part will have a witness mark all around the part at that point. As discussed in the Corner Breaks & Fillets section of this design guide, if the bottom fillet is not needed and a sharp corner can be tolerated, the full part geometry can be kept in the upper half the mold. This would move the parting line to the bottom of the part and no witness line would be present.

The potential for flash will always exist and in many cases the construction of the mold plays a big role in minimizing this potential. However, there are design actions that can be taken that will improve the robustness of the mold, thus decreasing the chances of flash on the part. One major way for avoiding flash is to have “flat-on-flat” contact for the mold seal-off features. Figure 20 shows how an intersection of 2 holes can be redesigned to reduce the potential of flash using a D-shaped hole as an ideal seal-off
surface for the intersecting hole. In this case, two flat surfaces are sealing against one another providing a tool that will be easy to maintain and less likely to generate unacceptable flash during the molding process. The alternative displayed in the figure shows the least attractive approach, which requires one of the cores to have a contoured or profiled face to match the core or hole that it will be sealing against during the injection portion of the molding process. In circumstances like these, the core orientation is critical and the feathered edges are likely to wear more rapidly affecting the shape and size of the molded feature. Mold flash is also a concern in these situations.

![Figure 20](image)

Whenever possible, areas of potential flash, and/or witness lines are moved away from critical areas. In the circumstances where this is not possible, there may be alternatives to ensure any witness lines and/or flash does not interfere with the function of the part. Figure 21 illustrates one of these alternatives. On a cylindrical component with an external undercut, the parting line would run lengthwise, down the center of the part. To avoid a situation where any witness on the O.D. could interfere with the function of the component, small flats are added along the parting line to ensure that any witness line and/or flash would occur below the functional diameter of the part.
12) Interchangeable Mold inserts
multiple parts that have only minor variations between them may be produced using interchangeable mold inserts. All common features are produced by the cavity, but the unique feature is produced with an insert that can be pulled out and replaced with another insert containing an alternative feature. Figure 22 illustrates a mold with interchangeable inserts to produce 2 different parts. Sharing a common mold and utilizing inserts minimizes the tooling fabrication needed, thus providing tooling cost savings. As with any metal-to-metal seal-off areas, there will be a slight witness mark on the part and this should taken into consideration during the design stage. It should also be noted that interchangeable inserts can generally be accommodated on low to medium volume parts, but high annual volume applications are normally better served with independent molds for each part design configuration.
13) Corner Radii

Whenever a flat plastic part turns a corner to make a three-dimensional part, the designer must decide on the best way to make the transition from one plane to another. The best way to make this transition on any plastic part is with a properly proportioned corner radius.

The radii which are so common on plastic parts have two primary functions. First, they simplify the manufacturing process. Secondly, they strengthen the corner by eliminating stress concentration and by distributing the corner stress over a broader area.

It is a well-known fact that there will be a gradual increase in stress in the corner of a plastic part whenever the size of the inside corner radius is less than 75% of the thickness of the wall Figure. 23. There will be a very rapid increase in stress when the inside corner radius is less than 25% of the thickness of the wall to which the radius is attached.
Considering these facts, the inside corner radius on a plastic part should not be less than 25% of the wall thickness as shown in Figure 24. If maximum strength is required, the size of the radius should be at least 75% of the thickness of the wall to which it is attached.
The designer must also remember that all plastic materials cannot be handled in the same way. Notch-sensitive materials such as nylon or polycarbonate are very susceptible to a loss of strength in sharp corners. Other materials such as PVC and ABS are more tolerant of small corner radii.

The proper proportions for the corner radius on a plastic part will add significantly to the part’s strength. The strength of radiuses corners is determined primarily by the size of the inside radius. Designers therefore tend to specify the inside corner radius on a plastic part.

14) Gussets

Gussets may be considered a subset of ribs and the guidelines for ribs apply to gussets.

**Guidelines**

1. The thickness of a gusset at the intersection with the nominal wall should be 50% of the nominal wall. This will keep the gusset from reading through the nominal wall as sink.

2. The height of the gusset can be 95% of the height of the boss it attaches to. Generally the height will be less than 4 times the nominal wall thickness and the preferred height is 2 times the nominal wall. This has to do with the effectiveness of the gussets and the ease of molding with respect to fill and ejection.

3. The length of the gusset may vary from 30 to 100% of the height of the gusset. This addresses the effectiveness of the gusset.

4. The intersection of the gusset with the feature or the nominal wall should have a fillet with a radius of 25% of the nominal wall. The radii get rid of sharp corners which can introduce stress concentrations and adversely affect the molding process.

5. The spacing between gussets should be at least twice the nominal wall thickness. This is the same guideline for ribs and it pertains to the strength and cooling of the mold.