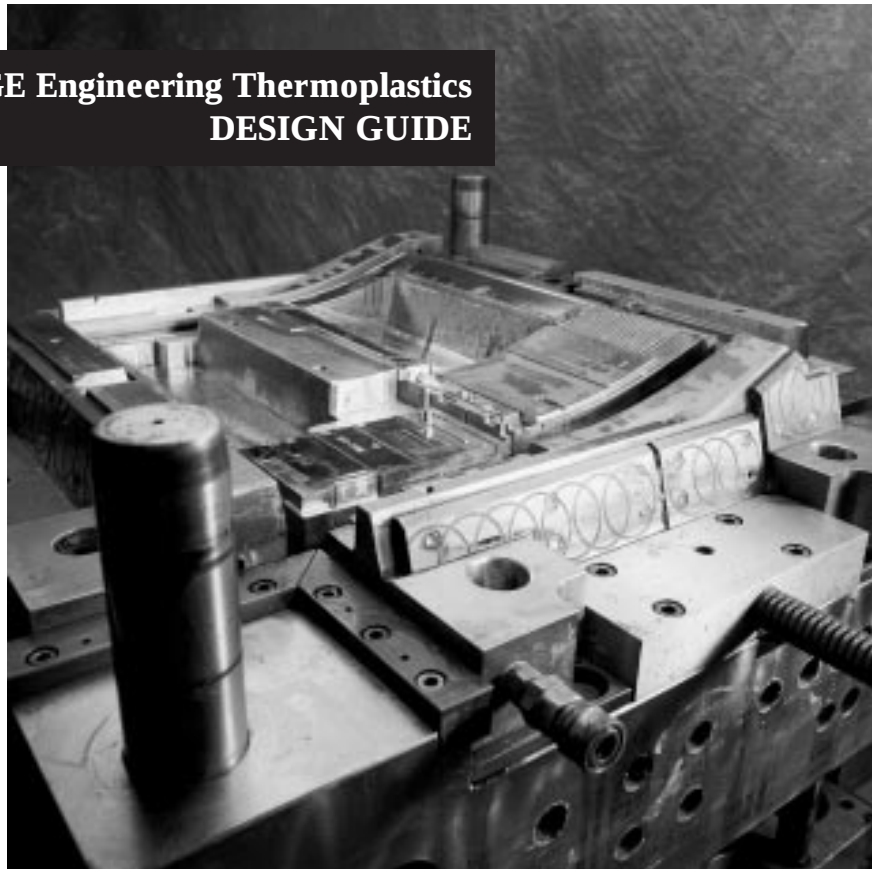
GE Plastics

Injection Molding Design

GE Engineering Thermoplastics
DESIGN GUIDE



- *Wall Thickness*
- *Parting Lines*
- *Ejection*
- *Appearance Parts*
- *Ribs/Gussets*
- *Bosses*
- *Holes*
- *Depressions*
- *Radii, Fillets and Corners*

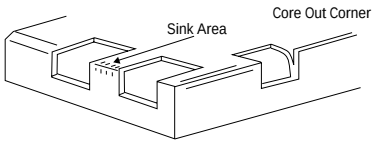
Guidelines for Injection Molded Design

A successful application of an engineering thermoplastic requires more than identifying a specific product or grade. Three areas – design, product, process – are all interrelated and the appropriate rules in each area must be followed to ensure a successful application. In most cases, the process must be determined before a specific resin grade can be selected. During this review, designers also need to consider whether the process is capable of meeting the design requirements such as size, shape, detail and tolerance.

Contents

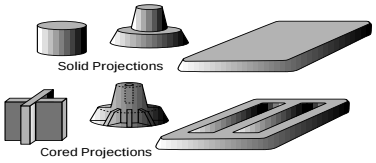
Wall Thickness

The typical plastic part may be considered to have a shell type configuration with a basic surface and features which are attached to it to meet functional requirements. From a moldability standpoint, the following are commonly regarded guidelines.



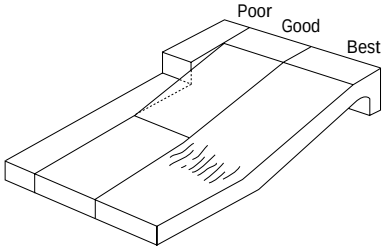
Guideline
The basic wall of the part should be kept uniform.

Basis
This provides for even flow of the melt during injection. Even cooling and shrinkage that controls part warpage and reduces molded in stress.



Guideline
Coring should be employed where possible to eliminate material masses in the part.

Basis
Coring results in more efficient designs and faster more productive cycle times. It also provides more uniform shrink and avoids sink marks.



Guideline
When wall thickness transitions cannot be avoided, the transition should be made gradually, on the order of 3 to 1.

Basis
The gradual transition avoids stress concentrations and abrupt cooling differences.

Guideline
The part and the gating should be designed so the melt flows from the thicker section to the thinner section.

Basis
This avoids a restricted flow and reduces molded in stress. It also allows for more uniform packing.

Wall Thickness Considerations

The actual determination of the wall thickness is based on a number of considerations. These include:

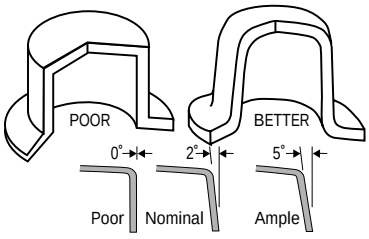
- **Application Requirements**. Structural requirements including strength, impact, fatigue or deflection will be influenced by the wall thickness selected. Electrical loads may also impact on the wall thickness.
- **Moldability**. The size of the part and the ability of the material to fill the furthest point can determine the minimum wall. The maximum flow length is also a function of tool design with gate location and number of gates used.
- **Agency requirements**. For some agency properties, the rating is based on a minimum wall thickness which the part design must meet or exceed to satisfy an agency requirement. This would be the case for UL flammability or RTI.

Wall Thickness

The wall thickness specified typically should meet all the considerations noted. From a cost standpoint, the thinnest wall utilizes the least material and results in the fastest molding cycles.

Parting Line and Ejection

The designer needs to consider how the mold will part and design in appropriate draft and shutoff. Often design changes to a feature can eliminate the need for action in the mold, saving tooling cost and maintenance costs later on. Guidelines relating to draft, shutoff and parting lines are offered below.

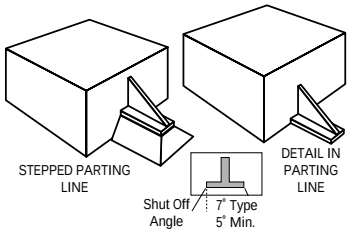


Guideline

On surfaces in the draw of the die, a minimum of 1/2˚ should be specified. Typical draft is 1˚. More draft aids ejection but may generate a material mass on sections contained in one side of the mold.

Basis

Draft is required for release.



Guideline

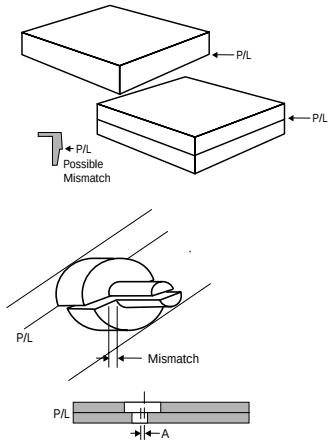
Keep features in the parting plane to simplify the part. When a stepped parting line is required, allow 7˚ for shut off. Minimum shut off angle is 5˚.

Basis

Drag at the shut off will wear over time and develop flash. Maintenance to restore mold to flash free parts will be more frequent.

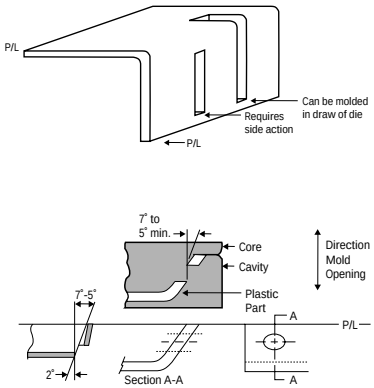
Guidelines for Injection Molded Design

Parting Line and Ejection



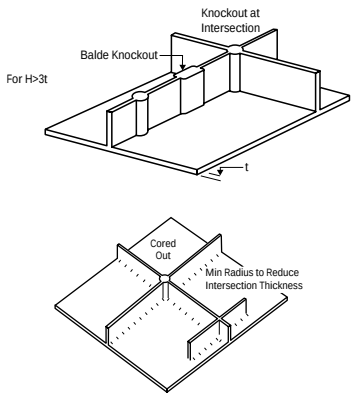
Guideline
Specify mismatch on the parting line.

Basis
Note establishes what is acceptable to the engineer for the molder.



Guideline
Redesign holes in the side walls so that the feature can be obtained with shut off, thus reducing the need for side action in the mold.

Basis
Taking advantage of design flexibility can simplify the mold, reducing initial costs and minimizing maintenance throughout production.



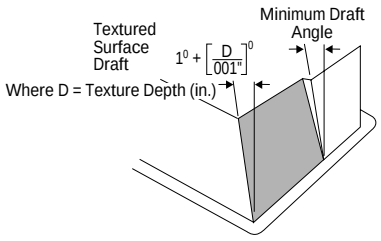
Guideline
For deep ribs and protrusions, allow for knockouts on the tops of the ribs or at intersections.

Basis
Knockouts for ejection need to be incorporated into design as they are frequently greater than the section and must be free to travel during the ejection stroke.

Appearance Parts

Guideline
To maintain a Class A surface on a molding, the side behind the appearance surface must be free of projections and depressions. If a projection can not be avoided, then the maximum thickness at the intersection is half the nominal wall thickness.

Basis
Very subtle changes in the wall section may read through on a high gloss, high quality surface. Even tooling lines made from sloppy fitting lifters may read through noticeably on the appearance surface.



Guideline
Some relief may be available to locate a structural rib opposite a Class A surface if a styling line runs directly opposite the rib.

Basis
The read through is masked by the styling line.

Guideline
Consider the use of texture on the appearance surface to mask read through of any detail on the opposite side.

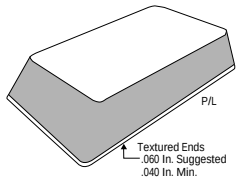
Basis
The texture breaks up a glossy surface and minor read through is not noticeable.

Guideline
Allow 1˚ additional draft for each 0.001 in (0.025 mm) depth in texture.

Basis
Increase in draft angle is needed to avoid scuffing and obtain proper release.

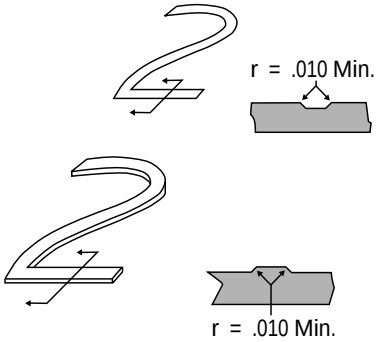
Guidelines for Injection Molded Design

Appearance Parts



Guideline
Textured surfaces stop a preferred 0.060 in (1.524 mm) or a minimum of 0.040 in (1.016 mm) from any parting line.

Basis
Terminating the texture improves the durability of the parting line.



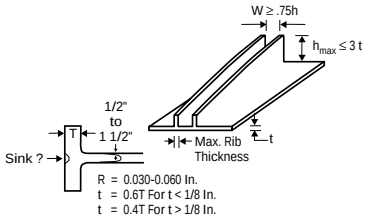
Guideline
Raised or recessed letters on appearance surfaces have a minimum radius of 0.010 in (0.254 mm).

Basis
The breaking of the sharp edge helps the appearance of the lettering.

Guideline
Print notes for appearance parts. Locate knockouts, gates and insert lines away from identified appearance surfaces. Specifications include color number, gloss number or texture specification. All appearance and post finish surfaces are identified.

Basis
To communicate clearly on the part drawing information with regard to appearance.

Ribs



Guideline
The thickness of the rib at the intersection with the nominal wall should be 50 to 60% of the nominal wall.

Basis
The intersection can develop a mass of material if rib thickness gets too great. This can affect the fill pattern within the mold and can result in sink on the wall opposite the rib.

Guideline
Maximum rib height: h=3 x nominal wall thickness.

Basis
Deep ribs become difficult to fill, may stick in the mold on ejection, and with draft they can generate a material mass at their base.

Guideline
Typical draft for ribs is 1 to 1.5˚. Minimum draft should be 1/2˚ per side.

Basis
Draft is necessary to aid ejection of the part.

Guideline
The intersection at the base of the rib should radii. 25 to 50% of the wall thickness. A minimum radius of 0.015 in (0.381 mm) is suggested.

Basis
The radius eliminates a sharp corner and stress concentration. Flow and cooling are also improved.

Guideline
Spacing between two parallel ribs should be a minimum of 2 x wall thickness.

Basis
This keeps the mold from developing a hot blade and cooling problems.

Guideline
The preferred flow of the melt in the mold is down the length of the ribs.

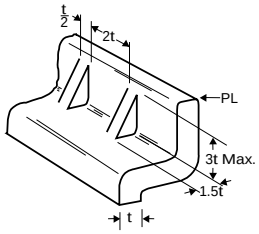
Basis
The flow across the ribs results in a branched flow and can trap gas or hesitate because of the thinner section. Hesitation can result in stress and hinder fill.

Gussets

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

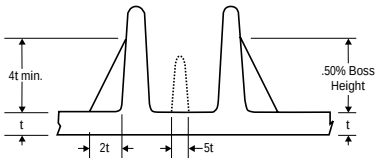
Guideline
The thickness of a gusset at the intersection with the nominal wall should be 50% of the nominal wall.

Basis
This will keep the gusset from reading through the nominal wall as sink.



Guideline
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 .

Basis
This has to do with the effectiveness of the gussets and the ease of molding with respect to fill and ejection.



Guideline
The length of the gusset may vary from 30 to 100% of the height of the gusset .

Basis
This addresses the effectiveness of the gusset.

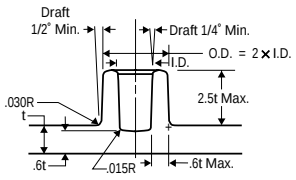
Guideline
The intersection of the gusset with the feature or the nominal wall should have a fillet with a radius of 25% of the nominal wall.

Basis
The radii get rid of sharp corners which can introduce stress concentrations and adversely affect the molding process.

Guideline
The spacing between gussets should be at least twice the nominal wall thickness.

Basis
This is the same guideline for ribs and it pertains to the strength and cooling of the mold.

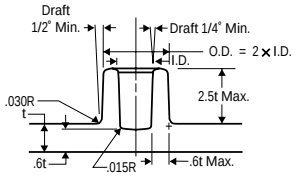
Bosses



Guideline	Basis
Typically the boss OD = 2 ID.	This is the general rule of thumb which allows the wall to increase as the size of the boss increases.
The wall thickness at the base of the boss should remain less than 60% of the nominal wall thickness.	Wall thickness greater than this guideline will result in a material mass which can produce sink and possible voids. This may also extend the cycle time.
The boss height should be less than 3 × OD.	A tall boss with the included draft will generate a material mass at the base. In addition, the core pin will be difficult to cool and can extend the cycle time and affect the cored hole dimensionally.
The boss should be radiused at the base. Radii at the base should be 25 to 50% of the nominal wall thickness. A minimum radii of 0.015 in (0.381 mm) is suggested.	Bosses are often an attachment point and carry significant loads. The intersection of the base of the boss with the nominal wall is typically stressed and this is magnified by a stress concentration if no radii are used. In addition, radii help in molding.
The end of a cored hole in the boss should have a minimum radius of 0.010 in (0.254 mm)	A radius on the core pin avoids a sharp corner which aides molding (fill and cooling), and diminishes stress concentration.

Guidelines for Injection Molded Design

Bosses



Guideline	Basis
Draft on the OD is 1/2° minimum.	Draft is needed for release from the mold on ejection.
Draft on the ID is 1/4° minimum.	Designs may require minimum taper to get proper engagement with a fastener. With proper ejection and polishing on the mold, thesmall draft angle can be accommodated.
Bosses adjacent to an external wall should be placed inboard a minimum of 0.125 in (3.175 mm) to the edge of the boss OD.	This location allows the designer to tie the boss to the wall with ribs and avoids creating a material mass which would sink and lengthen cycle times.
Keep the minimum distance of twice the nominal wall thickness between 2 bosses.	When features are located too close to each other, thin hard to cool areas in the mold will develop and can affect part quality and productivity.

Boss Design for Fasteners

	Guideline	Basis
	The ID should be .8 × nominal screw diameter.	This is the proper dimension for a screw to tap threads in a boss.

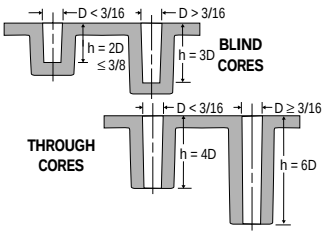
	Guideline	Basis
	Screw engagement should be a minimum of 2-1/2 times the screw diameter.	Shorter engagement lengths risk stripping the threads during assembly and pull out strength may be reduced.

	Guideline	Basis
	The depth of the cored hole should be 0.032 in (0.813 mm) greater then the screw length when fully engaged.	This avoids bottoming the screw causing undo stress and allows room for displaced material from the self tapping screws, primarily thread cutting screws.

	Guideline	Basis
	A chamfer at the top of the boss is a good lead in for the fastener.	The designer can speed the assembly by adding lead in to his design.

Holes and Depressions

Injection molding is a high pressure process and the viscous melt can deflect or even bend core pins in the mold. As a result, design guidelines address these occasions to help avoid the problem.



Guideline
For blind holes, the length over diameter ratio should remain below 2. As the diameter of the hole increases above 3/16 inch, the length over diameter ratio can increase to 3.

Basis
The strength of the core pin to withstand the pressures applied by the advancing melt are best addressed by relating the cross section of the pin to the length of the pin.

Guideline
For through holes, support of the core pin can be obtained on both ends or the pins can meet in the center which allows the diameter to length ratio to increase to 4. For diameters greater than 3/16 inch, the length to diameter ratio should remain below 6.

Basis
Because of the increased support, a through hole can be twice as long as a blind hole without deflecting on the injection cycle.

Exception
The length of a core pin may exceed the guideline if a fill pattern is such that a balanced force is developed on the core pin.

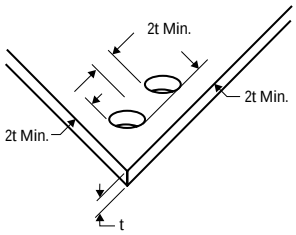
Guideline
A vertical step of approximately 0.015 in (0.381 mm) may be considered around the open end of a cored hole.

Basis
This allows for slight mismatch in the mold and the core pin to be part of a larger diameter doll needed for mold design.

Guideline
A note identifies areas where weld lines are unacceptable because of the high loads and inadequate strength.

Basis
Warning to a mold designer or processor about structural requirements as they relate to molding.

Holes and Depressions



Guideline
The mold design should direct the melt flow down the length of slots or depressions and attempt to locate knit lines in thicker sections.

Basis
A flow as described in the guideline will result in the least molded in stress and knit lines are bolstered by the increased section.

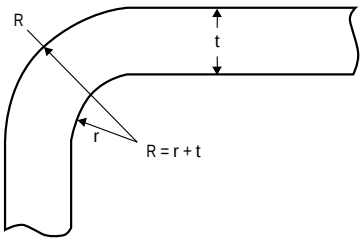
Guideline
The distance between two holes or one hole to the edge of the surface should be at least 2 × the nominal wall thickness or 2 × the hole diameter if that dimension is larger.

Basis
Adequate material around a hole is needed for strength particularly when knit lines are likely to be present.

Guideline
The edges and corners of a depression should have a minimum radius of 0.015 in (0.381 mm) The preferred radii is 50% of the nominal wall thickness.

Basis
The incorporation of radii aids in the molding and strength of the part.

Radii, Fillets and Corners



Guideline:
A fillet radius should be between 25 to 60% the nominal wall thickness. The larger fillet radius is suggested for load carrying features. A minimum radius of 0.020 in (0.508 mm) is suggested. Break any sharp corner with at least a 0.005 in (0.127 mm) radius.

Basis
Proper use of radii avoids stress concentrations in a part.

Guideline
The outside corner radius should be equal to the inside radii plus the wall thickness (R = r + t).

Basis
This practice keeps a uniform wall thickness at the corner and reduces stress concentrations.