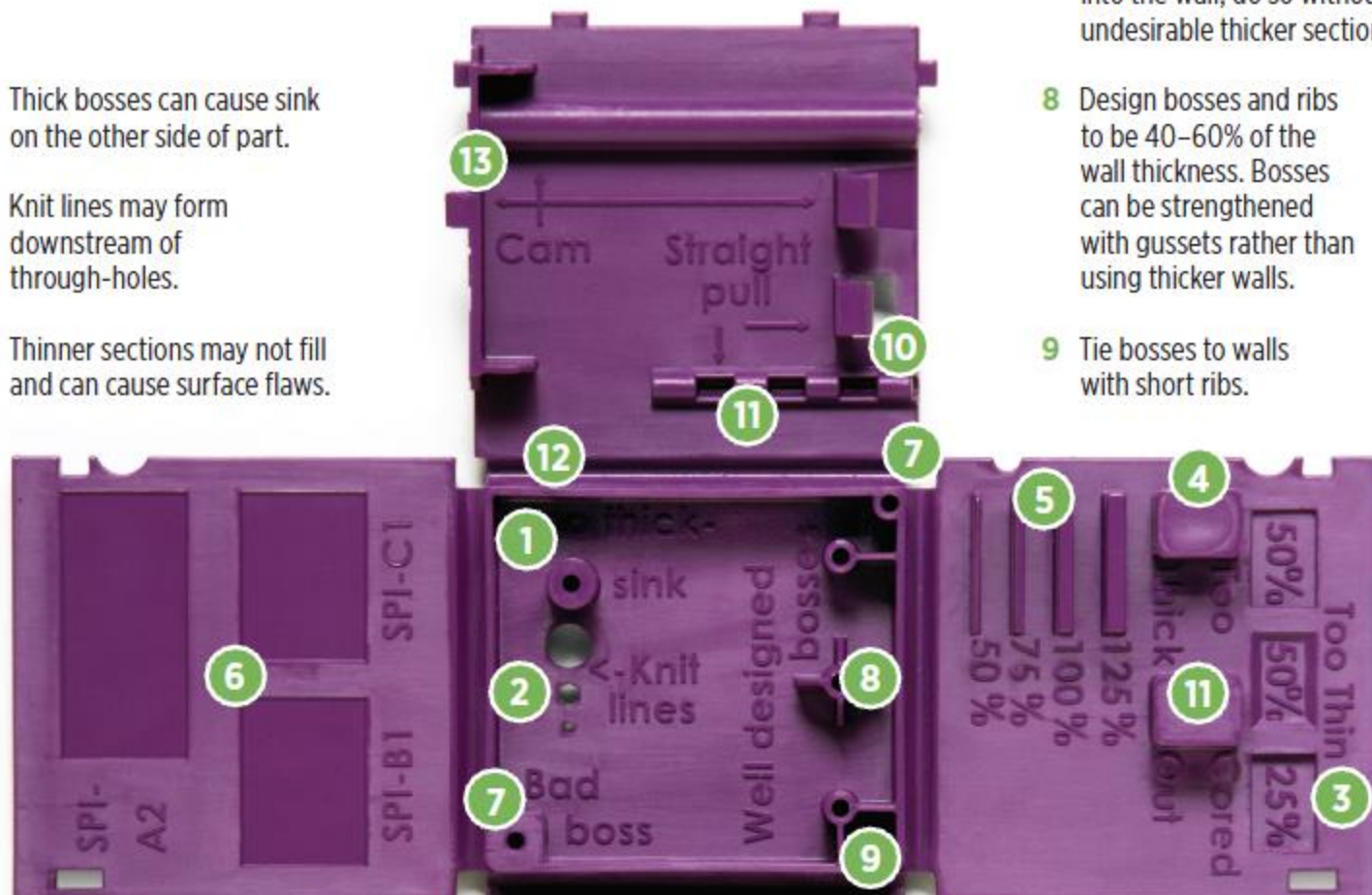


Injection moulding – DesignDetails

Drafts angles, radii, wall thicknesses
and lots of other details.

- 1 Thick bosses can cause sink on the other side of part.
- 2 Knit lines may form downstream of through-holes.
- 3 Thinner sections may not fill and can cause surface flaws.



- 7 If you incorporate the boss into the wall, do so without undesirable thicker sections.
- 8 Design bosses and ribs to be 40–60% of the wall thickness. Bosses can be strengthened with gussets rather than using thicker walls.
- 9 Tie bosses to walls with short ribs.



4 Thicker features can sink, have voids or cause warp.

5 Thick ribs can cause sink on the other side of the part.

6 Surface finishes:

SPI-A2 High polish, no tool marks (suitable for many applications but not for precision imaging).

SPI-B1 Finished with 600 grit paper; no tool marks.

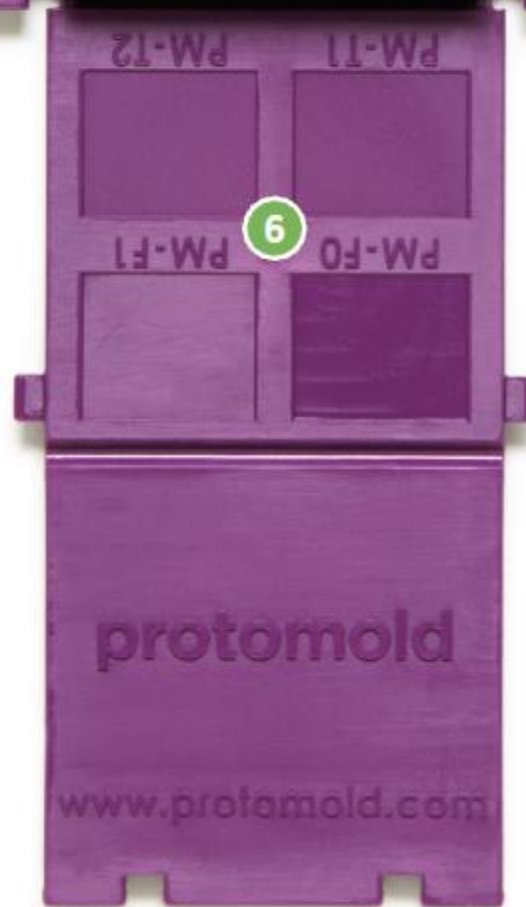
SPI-C1 Finished with a 600 grit stone; no tool marks.

PM-F0 As-machined or to Protomold discretion (default B-side finish).

PM-F1 Mostly SPI-C1, but evidence of underlying tool marks may still be noticeable in some areas (default A-side finish).

PM-T1 SPI-C1 followed by light bead blast.

PM-T2 SPI-C1 followed by medium bead blast.



10 Create undercut features that can be molded using sliding shutoffs in straight-pull molds. Maintain 3 degree minimum draft on shutoffs.

11 Core out thick sections.

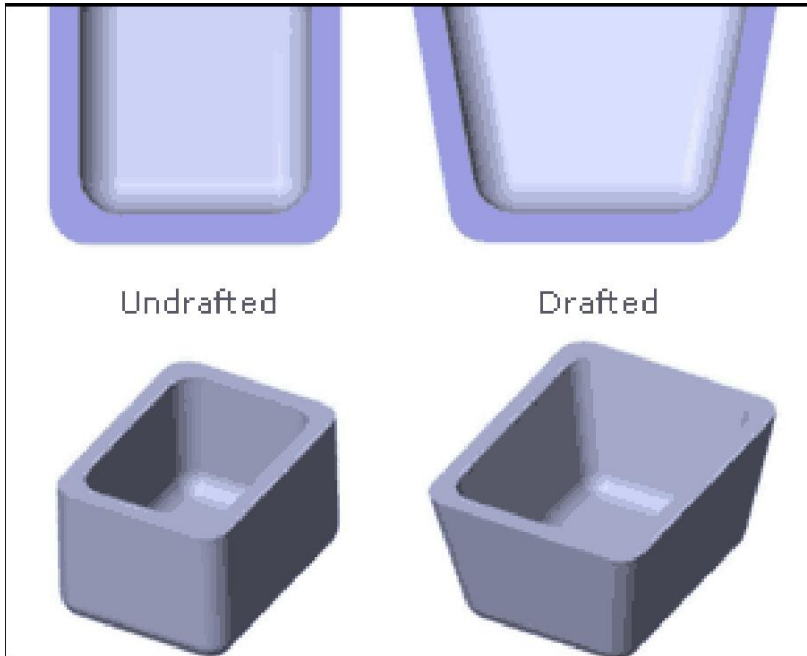
12 Living hinges add functionality but parts can be difficult to fill. They work best in polypropylene or polyethylene.

13 Side action cams can create features perpendicular to the main direction of pull. These include holes, hooks, text, texture on side walls, and much more.

Visit www.protomold.com/designguidelines for more tips on design.

Draft angles in product design

Drafts of 1.0 degree and larger may result in lower costs with the Injection Molding process.

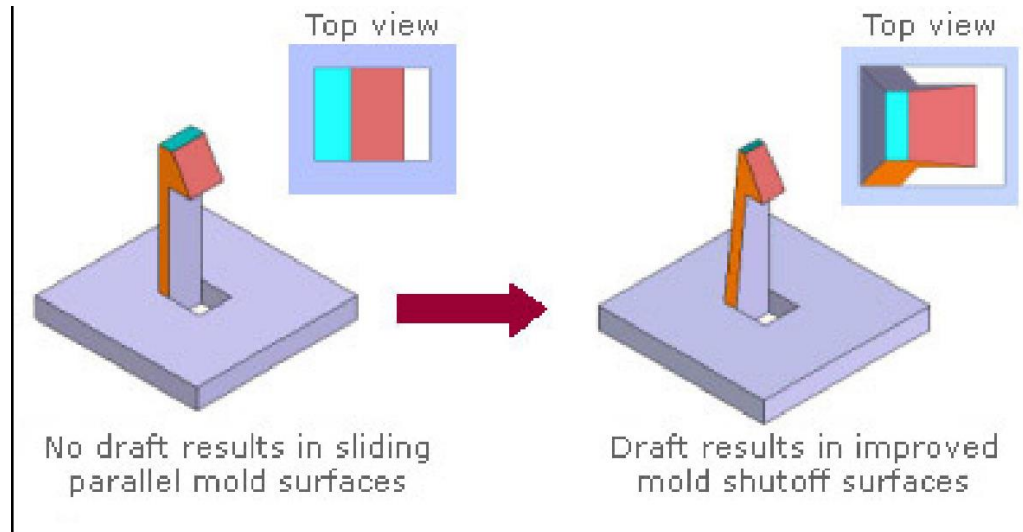


It is important that textured surfaces have adequate draft to prevent the part from sticking in the mould and to prevent “drag” marks.

Lightly textured surfaces (T-1) have a minimum of 3 degrees of draft and that more heavily textured surfaces (T-2) have a minimum of 5 degrees of draft.

As illustrated in figure 1, draft is the angle between the direction of ejection of a part from the mould and the surface of the part. Its traditional function is to facilitate the removal of the part from the mould

Draft angles....



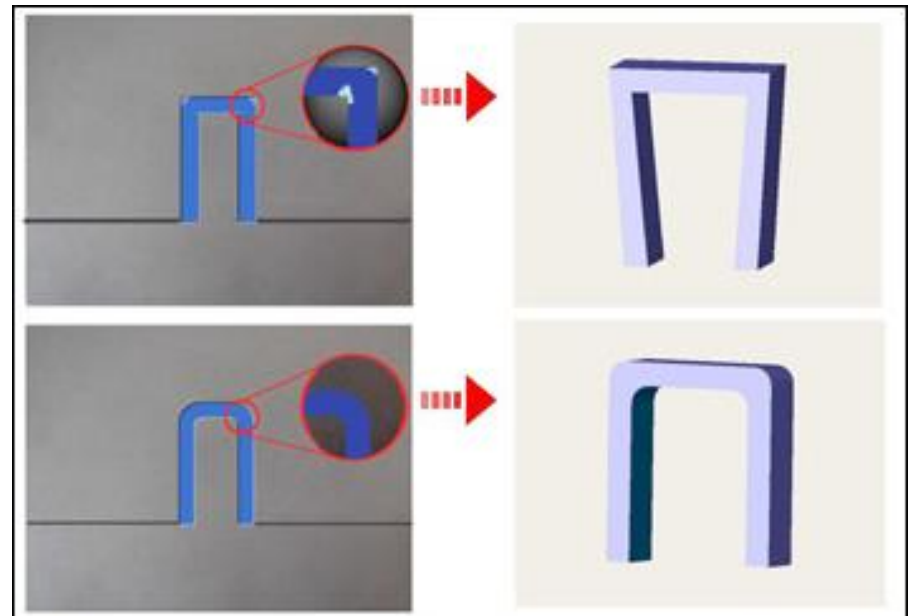
- Aiding the manufacturability of your design:
- In this case the draft helps the part eject from the mould more easily
- Also minimizes the amount of sliding required between the mould's shutoff surfaces.
- A minimum of 3.0 degrees of draft is required for shutoff features.

Corner radii....avoid stress

Whenever possible, a part to be injection moulded should be designed with generously radiused corners to enhance its quality and mouldability.

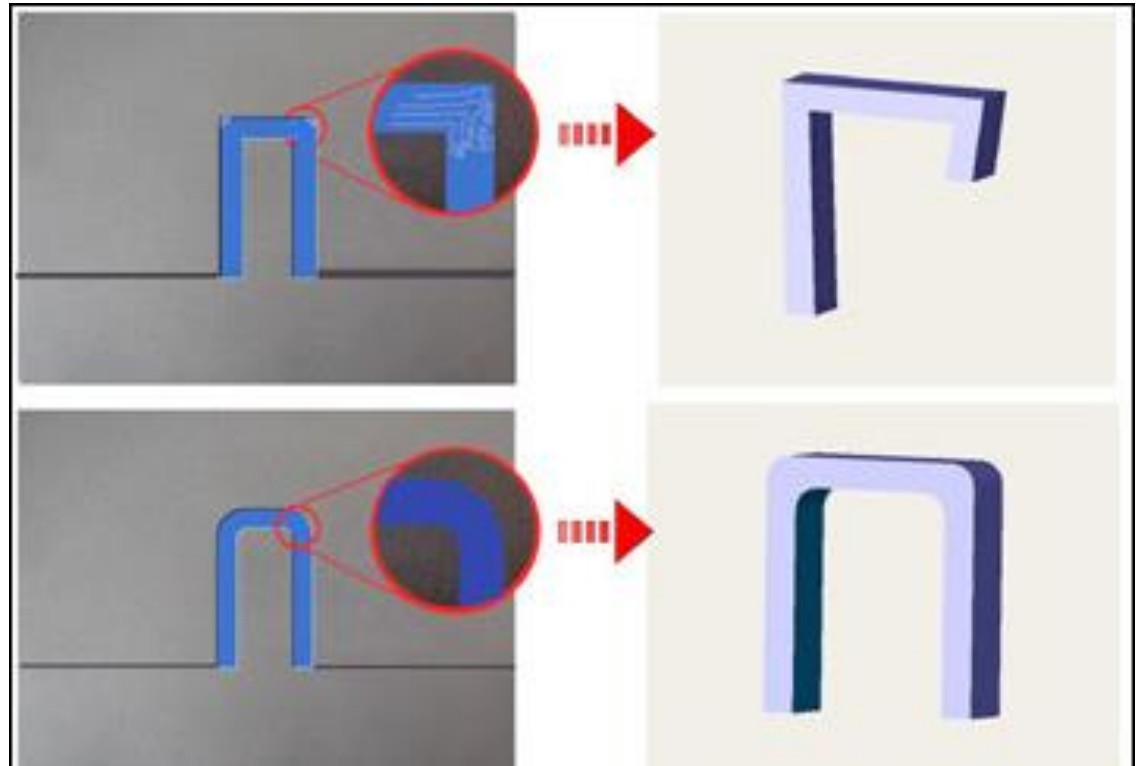
As illustrated, corners designed without radii can cause stress concentrations.

These in turn may reduce the ability of the part to withstand load and/or cause warping in its geometry.



Corners and Radii

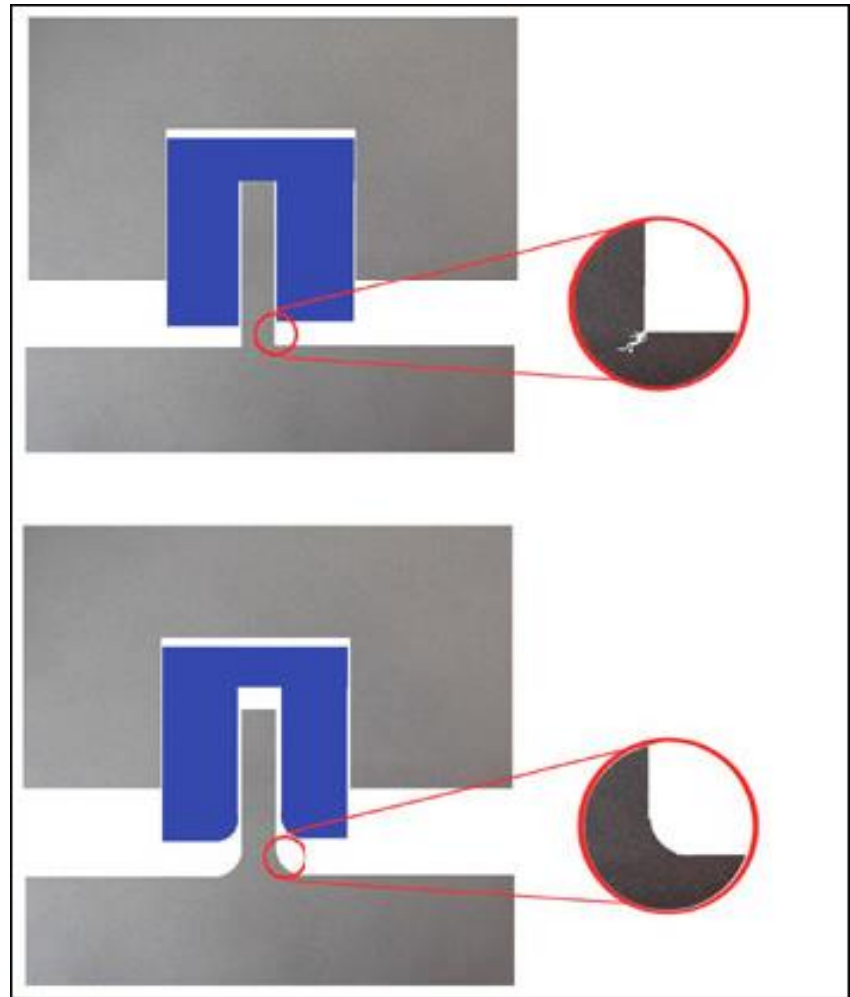
Sharp corners might adversely affect the flow of resin during moulding, potentially causing incomplete fill. They also tend to cause the part to stick to the mould during ejection, which can cause a variety of problems.



Corners and Radii

The effective use of fillets can also help to improve mould life by helping to:

- Minimize corner stresses at the bottom of tall, thin cores in the mould (at the entrance to deep thin holes in the plastic part).
- These fillets also help to enhance the ability of the mould to fill and further reduce internal part stresses.



Corners and Radii

Simple fact: plastic resins shrink as they cool.

Some shrink more than others, but they all do it.

If the shrinkage were perfectly even, we could simply make the mould slightly oversize and count on shrinkage to reduce them to the desired size.

Unfortunately, shrinkage is a more complicated process.

As a result, certain shapes that are otherwise perfectly acceptable can be difficult to mould because they tend to warp as they cool.



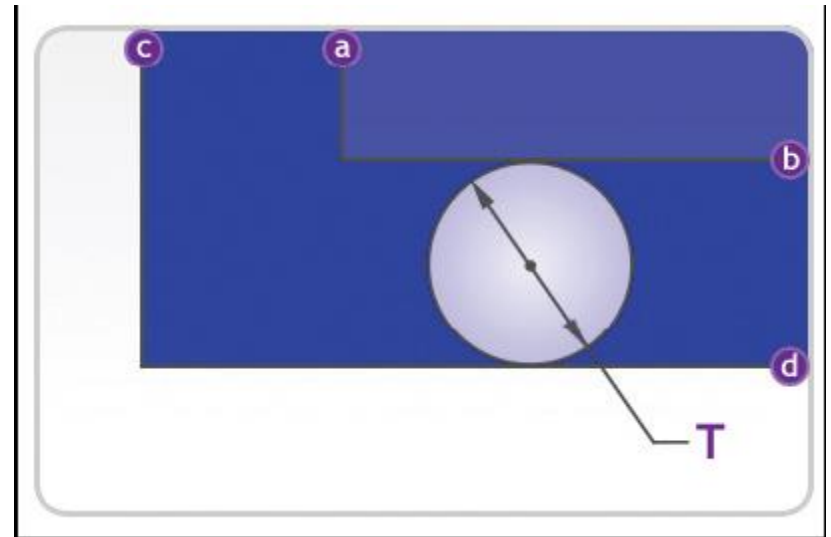
Corners and Radii

You can see that the distance along the inside of the angle (from A to B) is shorter than the distance around the outside of the angle (from C to D).

As a result, the surface on the outside of the angle is larger than that on the inside.

More area means faster radiation of heat. As a result, the C-D side of the angle hardens before the A-B side.

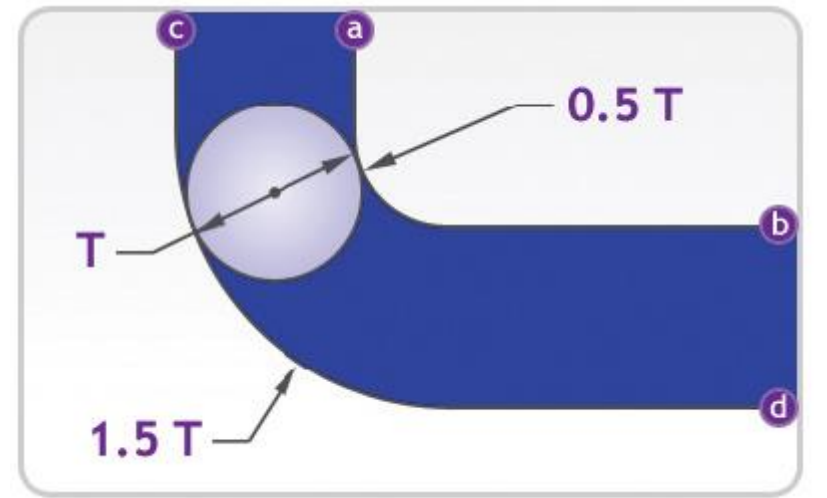
As A-B continues to cool, it also continues to shrink, pulling what was designed to be a right angle to something less than 90 degrees.



Corners and Radii

The solution? If you radius the corner on the inside and outside as shown here (using the formula shown maintains constant wall thickness), you will minimize the warping effect.

Using a larger radius (but maintaining constant thickness) will reduce warp more as it reduces the difference in mould metal to cool the inside and outside of the wall.



Plastic resin surface finish

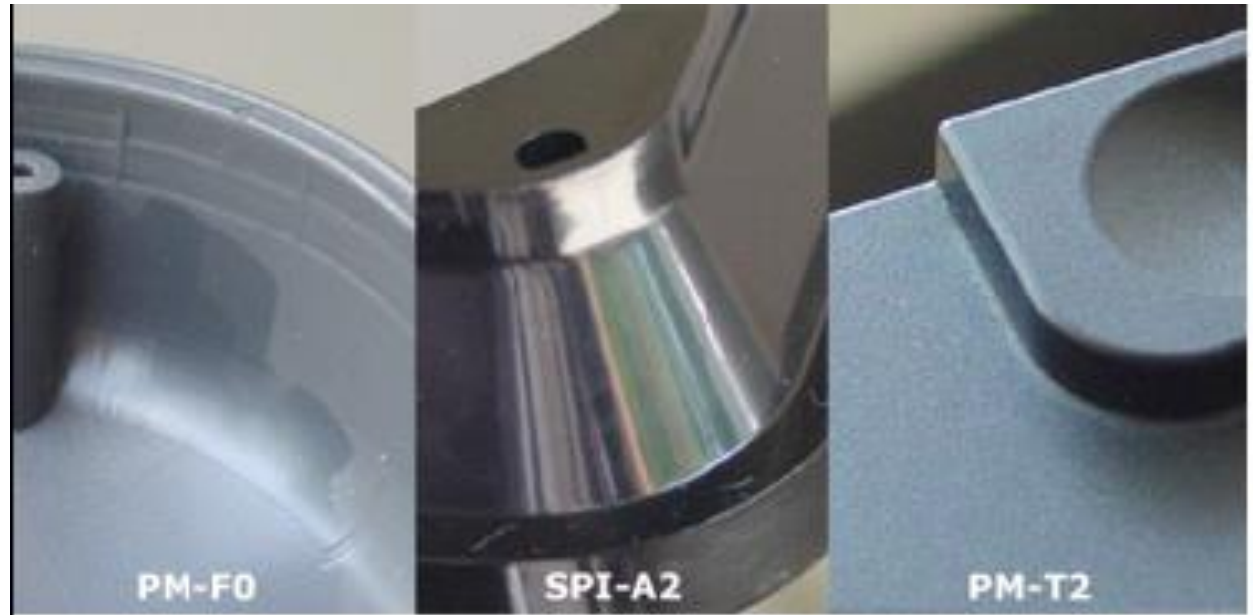
When designing a part for injection moulding, it is important to keep it mind the relationships between:

- Surface finish
- Mouldability
- Cost
- Lead time



Surface finish

- The photographs show the difference in cosmetic appearance for a few of these options on some example parts.
- If the part will not be visible to the end user, you will probably choose to specify either PM-F0 or PM-F1. But many times your design will require a more cosmetic surface finish.



In these cases there are two key things to keep in mind:

- Polishing
- Texturing

Surface finish

Polishing:

Smoother part surfaces are achieved using manual mould polishing techniques.

Consider a part with tall, thin, and curved ribs which needs to have an SPI-A2 finish.

In this case you should expect a significant cost increase, because it is very time consuming to polish deep, narrow slots in moulds. And such lengthy polishing times may also affect the lead time for your parts.

Texturing:

Given the line-of-sight nature of bead blasting, it may not be possible to texture the sides of minimally drafted ribs on a part because the mould surfaces may be inaccessible.

In addition, if the walls of your part are textured, it may have an adverse effect on the ability of the part to release from the mould, potentially resulting in unsightly “drag marks”.

For these reasons we can only apply texture to areas of the part that are drafted at least 3 degrees for a T1, and 5 degrees for a T2 texture.

Surface finish

“PM” in the table signifies a surface finish adjusted to fit the rapid injection moulding process where SPI (The Society of the Plastics Industry) denotes an industry-standard finish.

**Higher
Cost**



SPI-A2: Grade #2 Diamond Buff, 1-2 Ra

PM-T2: Protomold texture, SPI-C1 followed by medium bead blast

PM-T1: Protomold texture, SPI-C1 followed by light bead blast

SPI-B1: 600 grit paper, 2-3 Ra

SPI-C1: 600 grit stone, 10-12 Ra

PM-F1: Low-cosmetic — most toolmarks removed

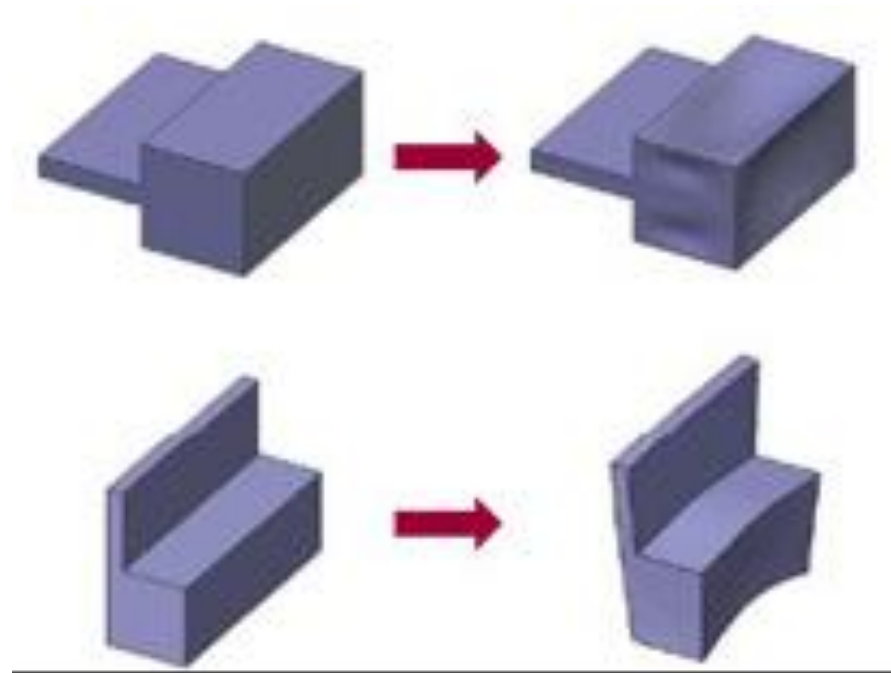
PM-F0: Non-cosmetic — finish to Protomold discretion

Thick section shrinking causes warping and sinking

As the plastic solidifies in the injection mould, it freezes from the outside (near the mould surface) toward the inside.

In thick sections this results in inward pulling stresses (due to contraction) that can cause sink marks in the outer surfaces of the part

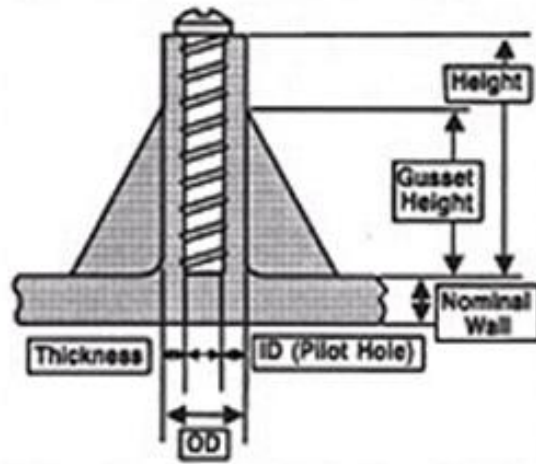
Because thinner sections will freeze faster than thicker sections, there is also the possibility of stresses building up between thick and thin sections.



This can result in part warping. So in the design of parts to be injection moulded, it is a good idea to maintain consistent wall thickness and avoid thick areas whenever possible.

Bosses

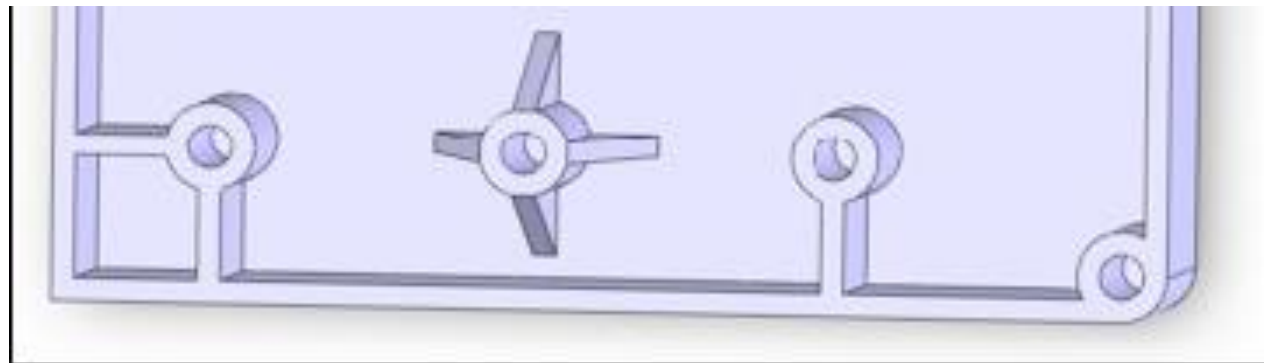
- Allow 2 moulded halves to be screwed together or mount components within moulded part



- Diameter = (Outside Diameter) \ (Inside Diameter) = 2 to 3
- Thickness = 1/2 to 2/3 nominal wall thickness
- Gusset Height = 2/3 Height
- Height = Fastener minimum requirements
- Taper = 1 deg. all around
- Diameter Ratio should be minimum ratio of 2., this will reduce risk of failure.

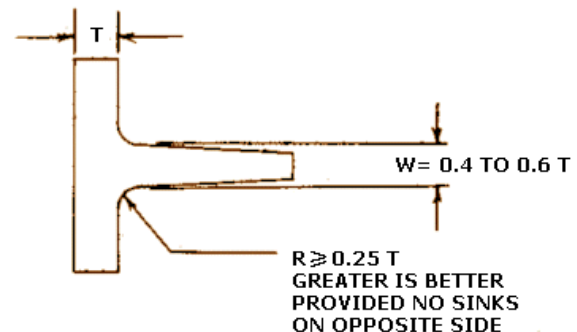
Bosses

Thicker and non-uniform wall thicknesses can often result in sinks in the material due to the same solidification physics described above



Ribs

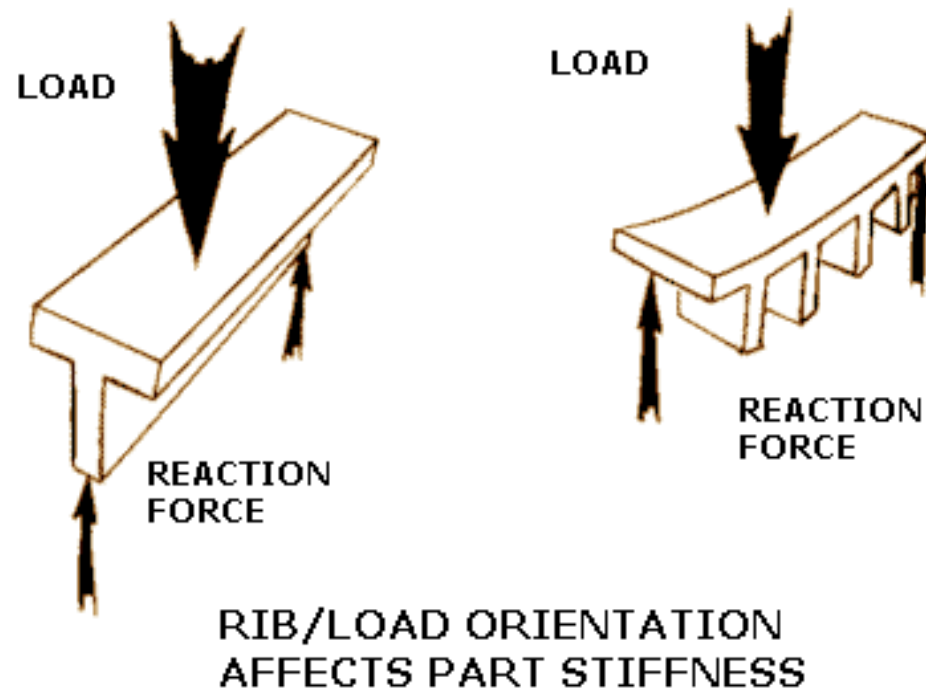
- Ribs increase the bending stiffness of a part. Without ribs, the thickness has to be increased to increase the bending stiffness. Adding ribs increases the moment of inertia, which increases the bending stiffness. Bending stiffness = E (Young's Modulus) $\times I$ (Moment of Inertia)
- The rib thickness should be less than the wall thickness-to keep sinking to a minimum. The thickness ranges from 40 to 60 % of the material thickness. In addition, the rib should be attached to the base with generous radiusing at the corners.



RIB DESIGN

Ribs

- The rib orientation is based on providing maximum bending stiffness. Depending on orientation of the bending load, with respect to the part geometry, ribs oriented one way increase stiffness. If oriented the wrong way there is no increase in stiffness.



Coring

- Remove excess material without reducing ergonomically specified size
- Extra material costs extra money and adds weight during shipping.
- So there are options such as coring to reduce material usage.



Draft angles on Ribs

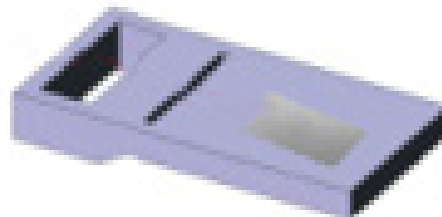
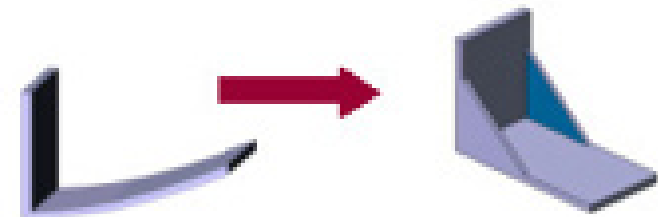
- Draft angles for ribs should be minimum of 0.25 to 0.5 degree of draft per side.
- If the surface is textured, additional 1.0 degree draft per 0.025 mm (0.001 inch) depth of texture should be provided.

Wall thickness

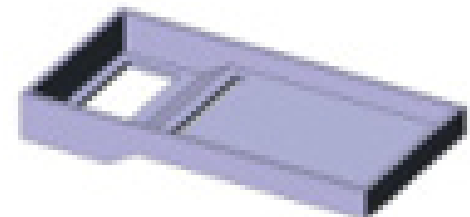
The use of thinner, uniform wall thicknesses helps to avoid sink

Warping due to stresses in step transitions between wall thicknesses can be improved through the use of a ramp

The use of gussets can be helpful to provide support in corners to avoid warping.



Thick walls cause sink, warp & excess shrink.



Thinner walls give accurate parts

The right resin.....material choice

- Application-specific requirements will always drive the need for particular material properties like:

- tensile strength
- impact resistance
- ductility.

- But successful designs for injection moulded parts are also built on an understanding of process-related issues such as:

- The ability to fill the mould

- Tendency to flash

- Ease of part ejection

- Potential for warp

- Sink

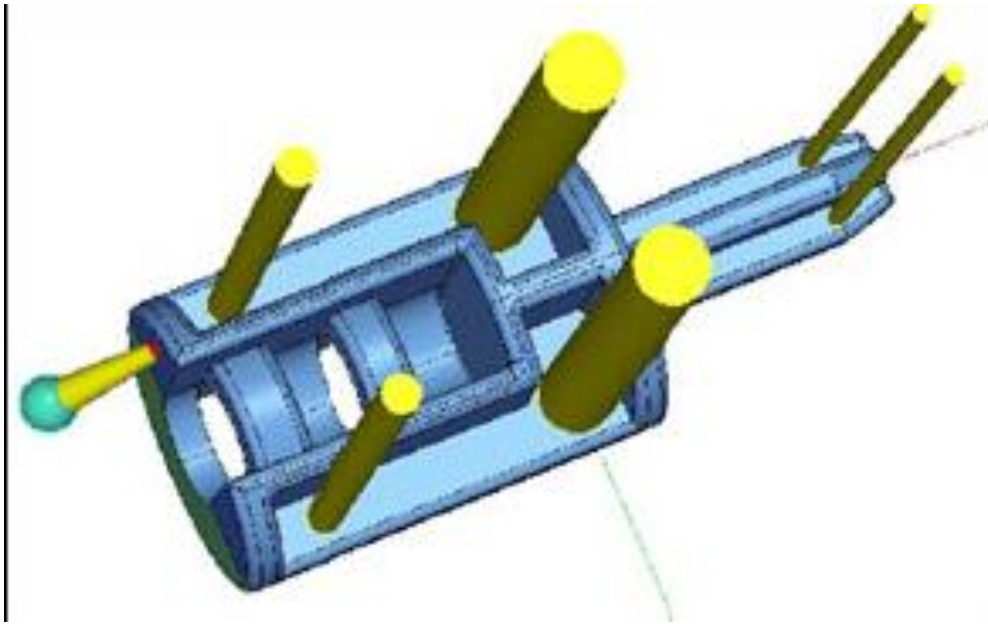
- Void creation.

Resin Choice

		Mechanical Properties			Moldability Characteristics						
	Some brand names	Strength	Impact	High temp-strength	Warp and dimensional accuracy, molded	Fills small features	Voids in thick	Sink in thick	Flash	High temp on mold & ejectors	Relative cost
Acetal	Delrin, Celcon	Medium	Medium	Medium-Low	Fair	Fair	Poor	Good	Good	Fair	Medium
Nylon 6/6	Zytel	Medium	High	Low	Fair	Excellent	Good	Fair	Poor	Fair	Medium
Nylon 6/6, glass filled	Zytel	High	Medium	High	Fair	Good		Good	Fair	Fair	Medium
Polypropylene	Maxxam, Profax	Low	High	Low	Fair	Excellent	Poor	Poor	Poor	Good	Low
High density Polyethylene (HDPE)	Dow HDPE, Chevron HDPE	Low	High	Low	Fair	Excellent		Poor	Poor	Good	Low
Polycarbonate	Lexan, Makrolon	Medium	High	Medium-high	Good	Fair	Fair to good	Fax	Good	Good	Medium-high
Acrylonitrile Butadiene Styrene (ABS)	Lustran, Cycolac	Medium-Low	High	Low	Good	Fair	Good	Fair	Good	Good	Low
Polycarbonate/ABS Alloy	Cycoloy, Bayblend	Medium	High	Medium	Good-excellent	Fair	Good	Fair	Good	Good	Medium
Polybutylene Terephthalate	Valox, Crastin	Medium	High	Low	Fair	Fair		Fair	Fair	Good	Medium-high
Polystyrene	Styron	Medium-low	Low	Low	Good	Good		Fair	Fair	Good	Low
Thermoplastic Elastomer	Isoplast, Santoprene	Low	High	Low	Poor	Excellent		Good	Poor	Excellent	Low-medium
Acrylic	Plexiglass-Acrylite	Medium	Low	Low	Good	Fair		Good	Good	Good	Medium

Expect to eject

Injection Moulding process uses ejector pins of various sizes to push the plastic part out of the mould after it has solidified. The sizes and arrangement of these pins are selected to minimize the impact on your part design.



Design approval:

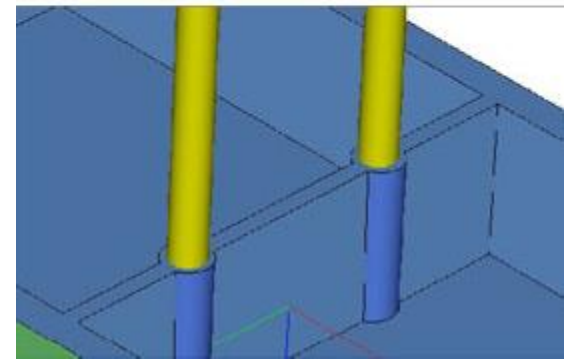
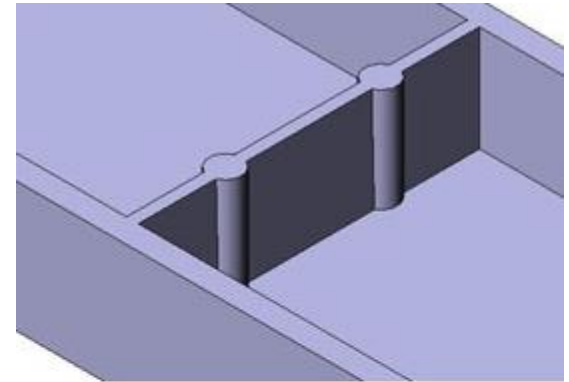
This is an example provided early in the process of designing the mould so that the location and size of both the gate(s) and ejector pins can be approved.

Ejector pins

Sometimes it is appropriate to adjust the design of the part to accommodate the need for ejection pins to push the part out of the mould.

For example, this illustrates how an injector pin “landing pad” has been integrated into the wall of a part design in order to provide sufficient material for the full diameter of the pin to meet the part.

Of course, always remember to provide as much draft as possible so that the ejector pins can do their job, especially for applications where it isn't possible to use mould-release lubricants to help the part eject more easily.



Living with hinges

As described in detail by Dr. Glenn Beall in his August 2002 Injection Moulding Magazine article, **“By Design: Polypropylene part design, Part 2 — Living hinges”**, in the late 1950s it was discovered that below a certain thickness, polypropylene molecules oriented in the direction of flow. And repeated bending perpendicular to that orientation was possible without breakage due to the increased strength that resulted. The name “living hinge” was given to this technique and has been used ever since.

Living (or live) hinges are very useful in certain designs for injection moulded parts because you can combine two or three parts into one.

A well designed hinge in these materials can last for millions of cycles.

Some materials with somewhat less of a life expectancy are Nylon and Acetal.



Glossary

A-Side — The mold half that mounts to the fixed side of the injection molding press, through which resin is injected into the part cavity via the sprue. Sometimes referred to as the cavity side of the mold, the A-side does not have ejector pins and for this reason it often produces the outside or cosmetic side of the part.

B-Side — The mold half that mounts to the moving side of the injection molding press. Sometimes referred to as the core side of the mold, the B-side has ejector pins to push the part out of the open mold. An analysis of the part geometry determines the optimal part orientation to ensure that it will remain on the B-side when the mold is opened.

Boss — A cylindrical protrusion within a part, often designed to accept fasteners.

Bead blasting — Using abrasives in a pressurized air blast to create a surface texture on the part.

Blush — Cosmetic blemish at the point of injection in the finished part.

Bridge tool — A temporary or interim mold made for the purpose of making production parts until a high—volume production mold is ready.

Cavity — A concave feature on either side of the mold into which an opposing core enters when the mold is closed. The void between the cavity and core is where the resin solidifies and forms the part. Often the A-side of a mold is referred to as the cavity side, and in the case of a part like a drinking cup, the entire A-side will be a cavity.

Clamp force — The force required to hold the mold shut so resin cannot escape during injection.

Contoured pins — Ejector pins with the ends shaped to match a sloping surface on the part.

Core — A convex feature on either side of the mold that will enter an opposing cavity when the mold is closed. The void between the cavity and core is where the resin solidifies and forms the part. Often the B-side of a mold is referred

Core-cavity — The design of a mold where the A-side forms the outside of the part and the B-side forms the inside. The advantage to this approach is that the part will shrink onto the B-side so it can be ejected, and if the inside and outside are drafted with equal and opposite draft the wall thickness will be constant.

Cycle time — The time it takes to make one part including the closing of the mold, the injection of the resin, the solidification of the part, the opening of the mold and the ejection of the part.

Direction of pull — Refers to the motion of a part surface relative to a mold.

Draft — The taper of features in the direction of pull. It allows deeper features to be produced in three—axis milling machines and it also helps parts release from the mold during ejection.

Drying of plastics — Many plastics absorb water and must be dried prior to injection molding to ensure good cosmetics and material characteristics.

Edge gate — An injection method that uses a gate on the parting line of the mold. It typically leaves a vestige on the outside of the part and is sometimes referred to as a tab gate.

Ejection — The process of pushing a completed part out of a mold.

Ejector pins — Steel pins incorporated into the B-side of a mold that push out the plastic part.

End mill — A cutting tool that is used to machine a mold.

ESD — Stands for “electro static discharge”, an electrical effect that may necessitate shielding in some applications. Some special grades of plastic are electrically conductive or dissipative and help prevent ESD.

Family mold — A mold containing two or more different parts.

Flash — Excess plastic that flows into the parting line of the mold beyond the edges of a part and freezes to form a thin, sheet-like protrusion from the part.

Flow marks — Visible indications on the finished part that indicate the flow of plastic within the mold prior to solidification.

Food grade — Resins or mold release spray that are approved for use in the manufacture of parts that will contact food in their application.

Gate — The location where the plastic

enters the part. There is typically a visible vestige when the gate is removed.

GF — Stands for “glass filled”, it refers to

a resin with glass fibers mixed into it. Glass filled resins are much stronger and more rigid than the corresponding unfilled resin, but also more brittle. Resins can also be filled with carbon fiber, stainless steel, etc. In general, filled resins can be very susceptible to warp.

Gusset — A triangular rib that reinforces areas
such as a wall to a floor or a boss to a floor.

Hot tip gate – An injection molding method

that uses a heated gate on the A-side of the part to eliminate the creation of any runner or sprue. The gate vestige will be a small sharp bump that can be trimmed if necessary.

Injection — The process of forcing

melted resin into a mold.

Jetting — Flow marks caused by the

resin entering a mold at high speed, typically occurring near a gate.

Knit lines — Visible indications in a finished

part, formed by the intersection of two hot plastic fronts. They are always formed downstream

Living hinge — Very thin section of plastic

used to connect two parts and keep them together while allowing them to open and close. They require careful design and gate placement. A typical application would be the top and bottom of a box.

Medical grade — Resin that may be suitable

for use in certain medical applications.

Mold release spray — A liquid applied to

the mold as a spray to facilitate the ejection of parts from the B-side. It is typically used when the parts are difficult to eject because they are sticking to the mold.

Multi-cavity mold — A mold with multiple

copies of the same part, typically used to reduce piece-part pricing for higher volume runs.

Nozzle — The tapered fitting on the end

of the barrel of the injection molding press where the resin enters the sprue.

Packing — The practice of using increased pressure when injecting a part to force more plastic into the mold. This is often used to combat sink or fill problems, but also increases the likelihood of flash and may cause the part to stick in to the mold.

Parting line — The location where the pieces of a mold come together. Typically a thin line is created on the part here.

Post gate — An injection method that injects plastic through an ejector pin hole. This gating technique leaves a gate vestige on the B-side of the part where it can often be less visible for cosmetic purposes. There is often gate blush opposite a post gate.

Press — The injection molding machine that makes the plastic parts. It holds the mold closed, melts the resin, injects it into the mold, opens the mold and ejects the part.

Radiused — An edge or vertex that has been rounded.

Recess — An indentation in the plastic part caused by the impact of the ejector pins.

Reinforced resin — Refers to base resins with fillers added for strength. They are particularly susceptible to warp because the fiber orientation tends to follow flow lines, resulting in asymmetric stresses. Typically these resins are harder and stronger but also more brittle (i.e. less tough).

Resin — Synonymous with “plastic” as far as injection molding is concerned.

Rib — A reinforcing member of a molded part.

Runner — A channel machined into the mold that directs the resin from sprue to the gate.

Shrink — The change in size of the part during solidification, typically anticipated based on published material property data and built into the mold design prior to machining.

Shutoff — The surfaces where the A-side and B-side of the mold contact. The shutoff meets the part at the parting line.

Straight pull mold — A mold without side actions. It is less expensive than a comparable mold with side actions.

Tab gate — See “edge gate”.

Telescoping shutoff — An area within a mold where metal slides along metal, usually creating a hole in the part. A three degree draft angle is required on any related part surfaces.

Texture — A surface treatment applied to the mold to create texture on the parts. Protomold offers polished, sanded and bead-blasted textures.

Tunnel gate — An injection method that uses a small gate located off the parting line in one of the mold halves. It leaves a vestige a short distance from the parting line of the part.

Undercut — A portion of the part geometry that would prevent the part from being ejected from a straight-pull mold without a portion of the mold passing through (and destroying) the part. The simplest example of an undercut feature on a part would be a through-hole aligned perpendicular to the direction of part ejection.

Vents — A very small (e.g. 0.001” – 0.005”) opening in the mold cavity, typically at the shutoff surface or via an ejector pin tunnel, that is used to let air escape from a mold while the resin is injected.

Vestige — A visible mark created by the manual process of finishing parts when the gate is trimmed. Ejector pins also may leave a vestige where they impact the part (also see “recess”).

Warp — The curving or bending of parts that typically occurs after ejection as the part cools. Warp is often caused by glass filled resins.

Weld lines — See “knit lines”.

Class Discussion Topics

- Briefly illustrate and describe the injection molding process.
- An injection-molding machine is divided into two principal components. Name them.
- Discuss some of the defects that can occur in plastic injection molding.
- What are some of the general considerations that Product/Automotive designers must keep in mind when designing components out of plastics?

