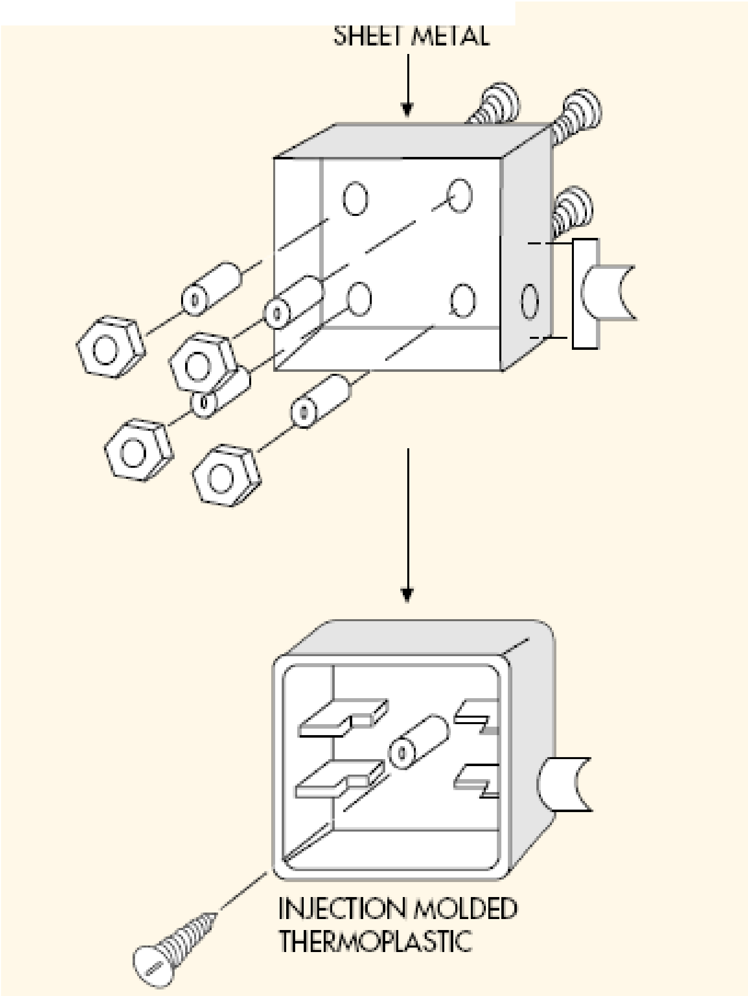
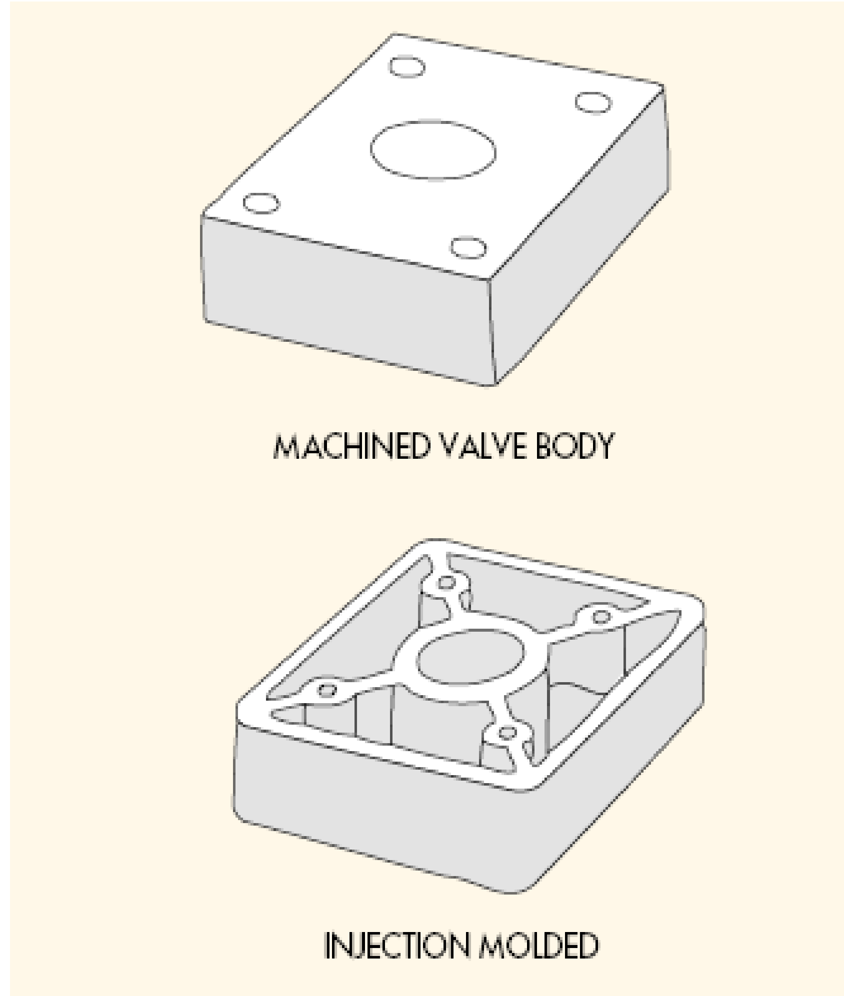
Plastic Product Design

Why one should prefer to design with Plastic?





Agenda

1. Product Design basic parameters to be considered
2. Plastics Features Design
3. Design for assembly
4. Defects in plastics
5. Design verification Parameters

Plastic Product Design General Parameters

- Weight reduction
- Consolidation of parts & functions
- Reduction in total per part cost
- Greater design freedom
- Reduction of secondary finishing & assembly operations
- Availability of material-resins to fit multiple applications
- Ability to withstand high temperatures & chemical, corrosive environments
- Decorating & informative features
- Product ability to function in all environmental conditions
- Tooling feasibility
- Manufacturing method
- Strength & durability
- Packing & Shipping considerations
- Assembly considerations
- Agency & Code regulations

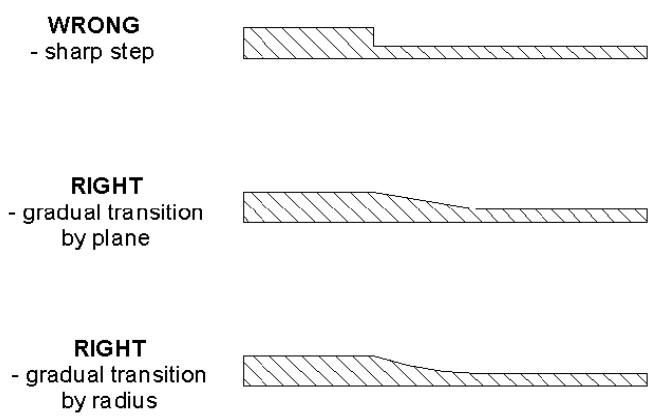
Plastic Product Features Design

1. Walls
2. Coring
3. Ribs & Gussets
4. Boss
5. Corners
6. Draft angles
7. Holes
8. Undercuts

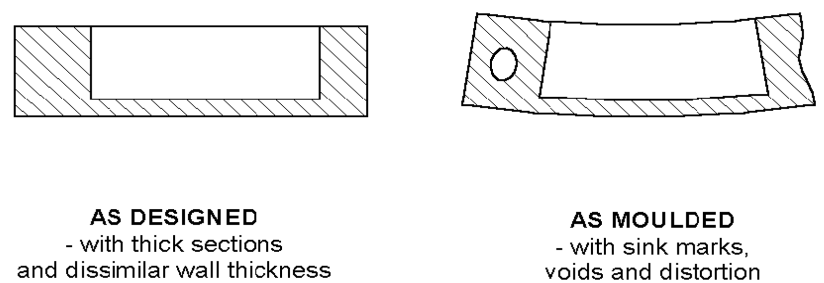
1. Walls

- Uniform wall thickness
- Thin walls rather than the thick walls
- Use gradual transitions between thick & thin wall sections
- Wall thickness must suit both function & process
- Wall thickness guide range is:

  0.75 mm to 3 mm for reinforced materials
  0.5 mm to 5 mm for unreinforced materials



Gradual transitions between thick and thin sections



Thick sections and non-uniform walls cause problems

Walls

- Parts should be designed with a minimum wall thickness consistent with part function and mold filling considerations. The thinner the wall the faster the part cools, and the cycle times are short, resulting in the lowest possible part costs.
  Also, thinner parts weight less, which results in smaller amounts of the plastic used per part which also results in lower part costs.

- Thick sections cool slower than thin sections. The thin section first solidifies, and the thick section is still not fully solidified. As the thick section cools, it shrinks and the material for the shrinkage comes only from the unsolidified areas, which are connected, to the already solidified thin section.
  This builds stresses near the boundary of the thin section to thick section. Since the thin section does not yield because it is solid, the thick section (which is still liquid) must yield. Often this leads to warping or twisting. If this is severe enough, the part could even crack.

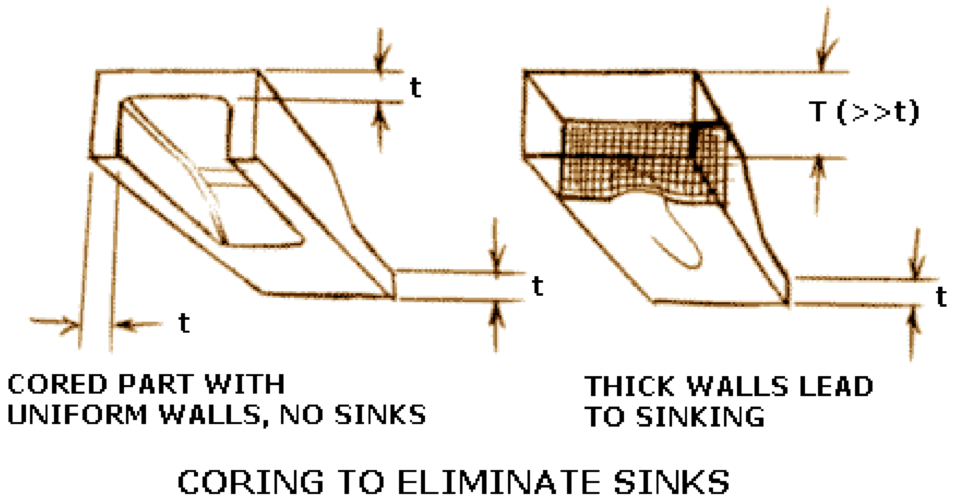
- Uniform walled parts are easier to fill in the mold cavity, since the molten plastic does not face varying restrictions as it fills.

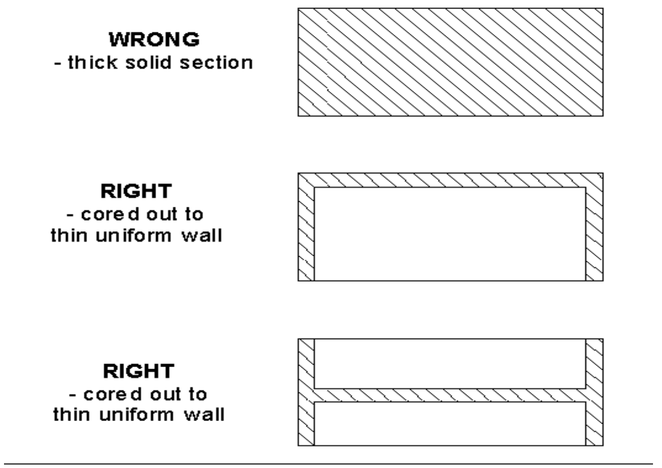
- When uniform walls are not possible, then the change in section should be as gradual as possible.

- Use uniform wall thicknesses throughout the part. This will minimize sinking, warping, residual stresses, and improve mold fill and cycle times.

2. Coring

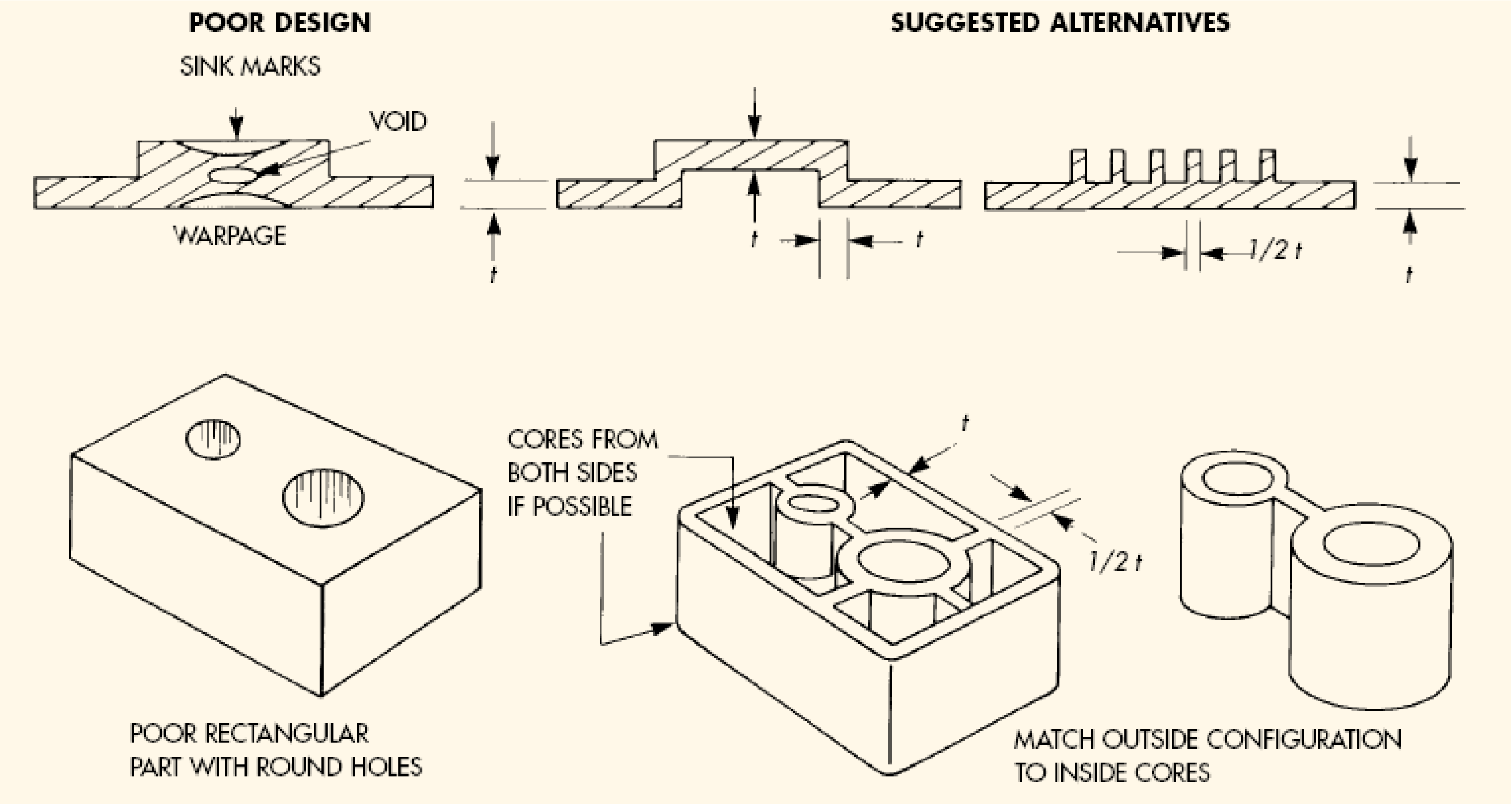
- Coring can help in making the wall sections uniform, and eliminate the problems associated with non-uniform walls.
- Prefer & design shell type object
- Use coring wherever possible to adjust uniform wall thickness





Solid shapes must be redesigned as 'shells'

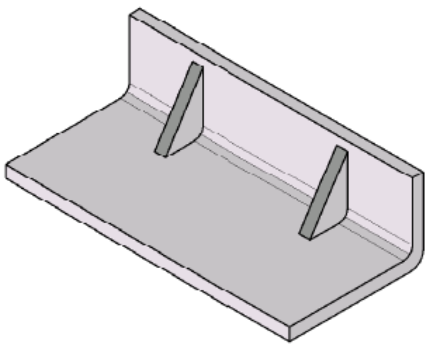
Coring


POOR DESIGN
SUGGESTED ALTERNATIVES
SINK MARKS
VOID
WARPAGE
t
t
t
1/2 t
t
CORES FROM
BOTH SIDES
IF POSSIBLE
t
1/2 t
POOR RECTANGULAR
PART WITH ROUND HOLES
MATCH OUTSIDE CONFIGURATION
TO INSIDE CORES

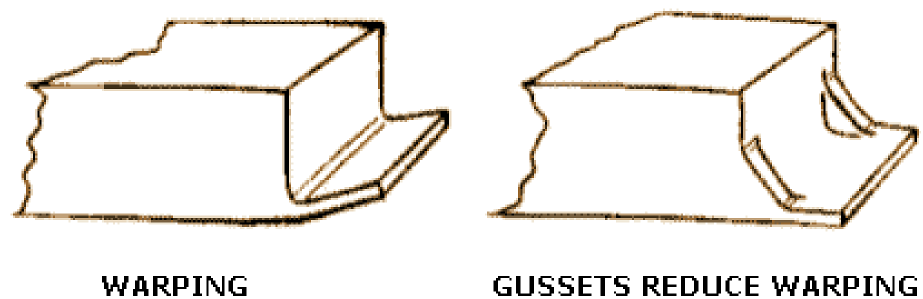
3. Ribs & Gussets

To improve parts stiffness in bending..

- Rib thickness should be 50 - 75% of the wall thickness.
- Fillet radius should be 40 - 60% of the rib thickness.
- Rib root thickness should not be more than 25% greater than the wall thickness.
- Rib depth should not be more than 5 times the rib thickness.
- Taper ribs for mould release.



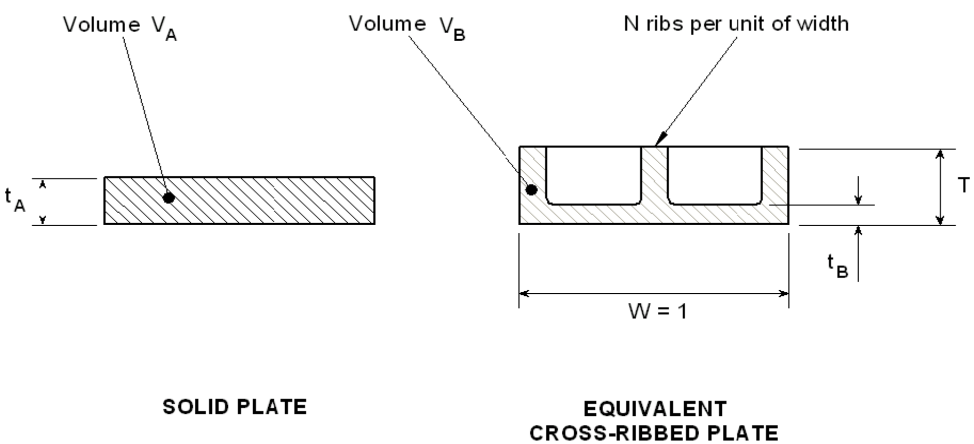
Use buttress ribs to stiffen side walls



GUSSETING TO REDUCE WARPING

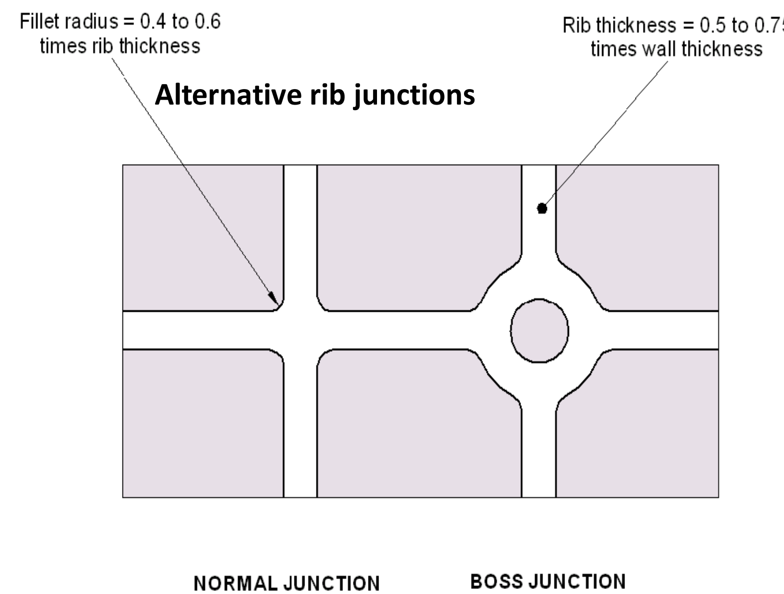
Ribs & Gussets

- Deep ribs are stiffer than thick ribs.
- Follow the basic rules for rib thickness and fillet radii.
- Calculate rib depth and spacing with a reckoner, or by using mathematical software or finite element analysis.



Terms for the calculator

Alternative rib junctions

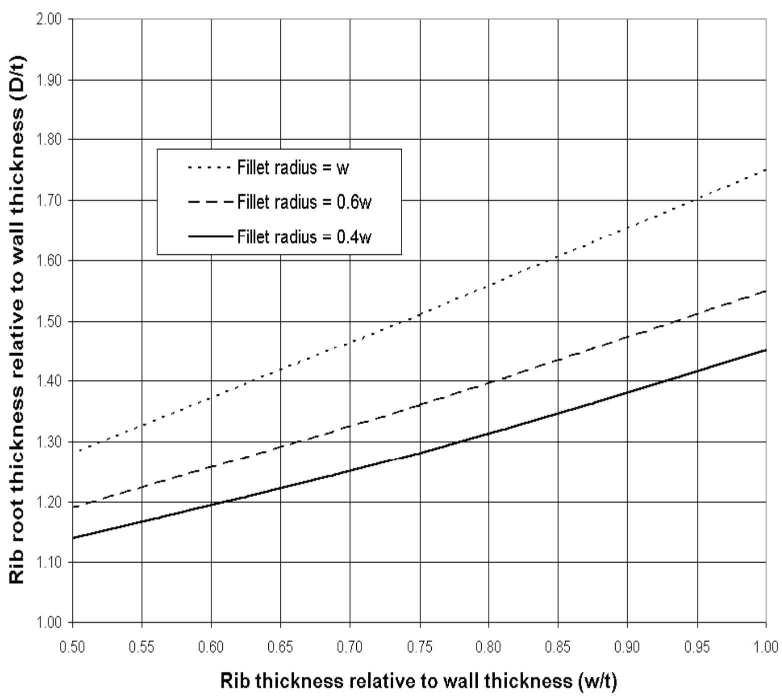


Ribs & Gussets

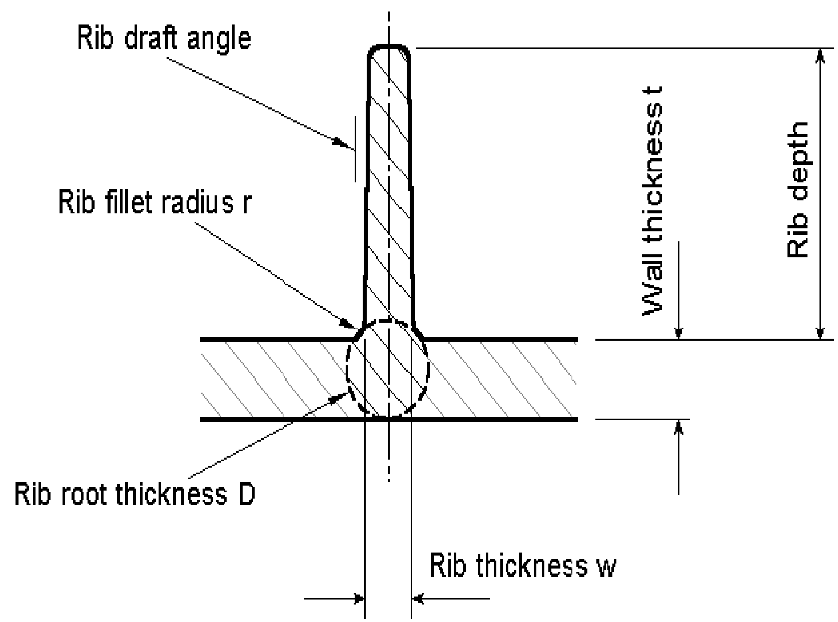
1. Ribs increase the bending stiffness of a part. Without ribs, the thickness has to be increased to increase the bending stiffness. Adding ribs increases the moment of inertia, which increases the bending stiffness. **Bending stiffness = E (Young's Modulus) x I (Moment of Inertia)**
2. The **rib thickness** 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 radius at the corners.
3. At rib intersections, the resulting thickness will be more than the thickness of each individual rib. Coring or some other means of removing material should be used to thin down the walls to avoid excessive sinking on the opposite side.
4. The **height** of the rib should be limited to **less than 3 x thickness**. It is better to have multiple ribs to increase the bending stiffness than one high rib.
5. 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.
6. **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.

Ribs & Gussets

- Unidirectional ribs should be spaced apart by at least 2 and preferably 3 or more times the nominal wall thickness
- Use the calculator curve to work out rib heights
- Use buttress ribs to stiffen side walls



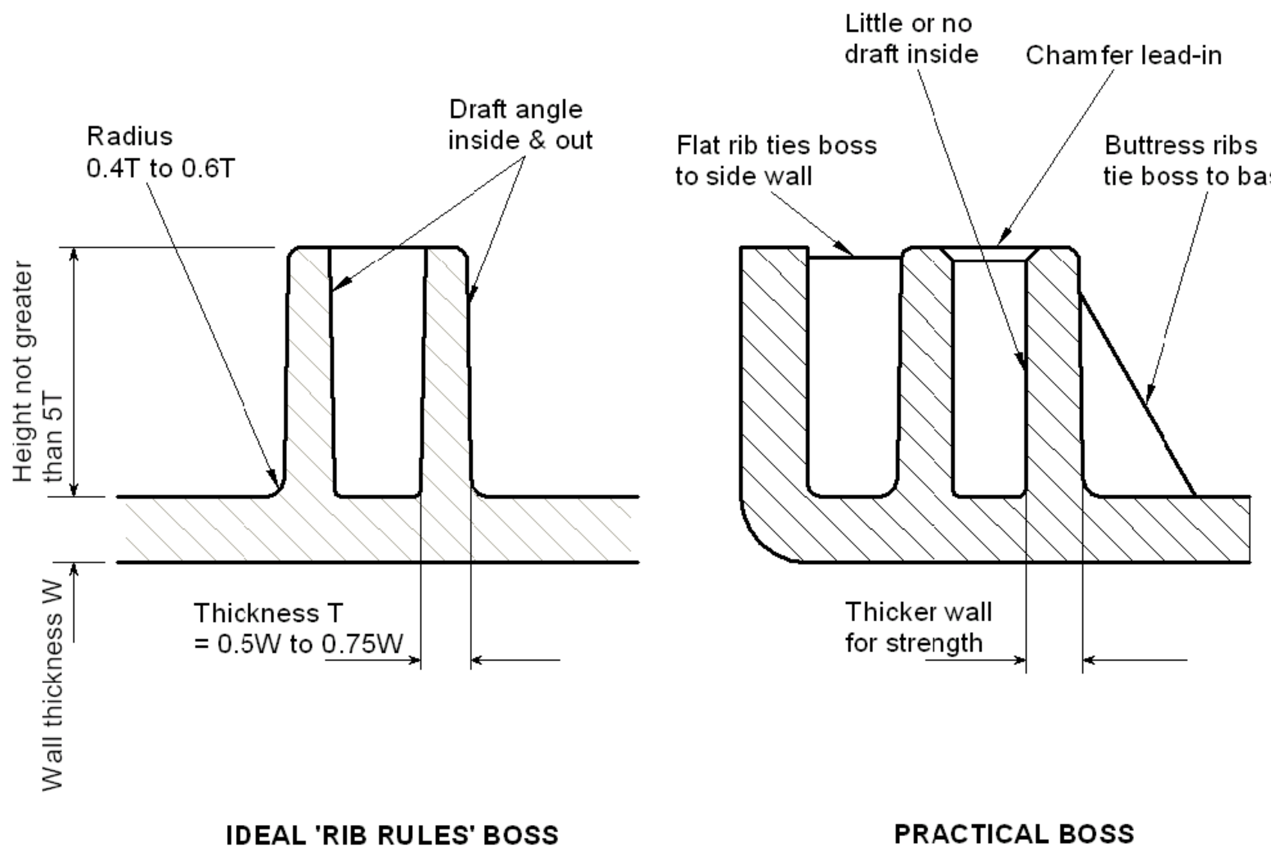
How rib root thickness increases



Ribs create thick sections at the root

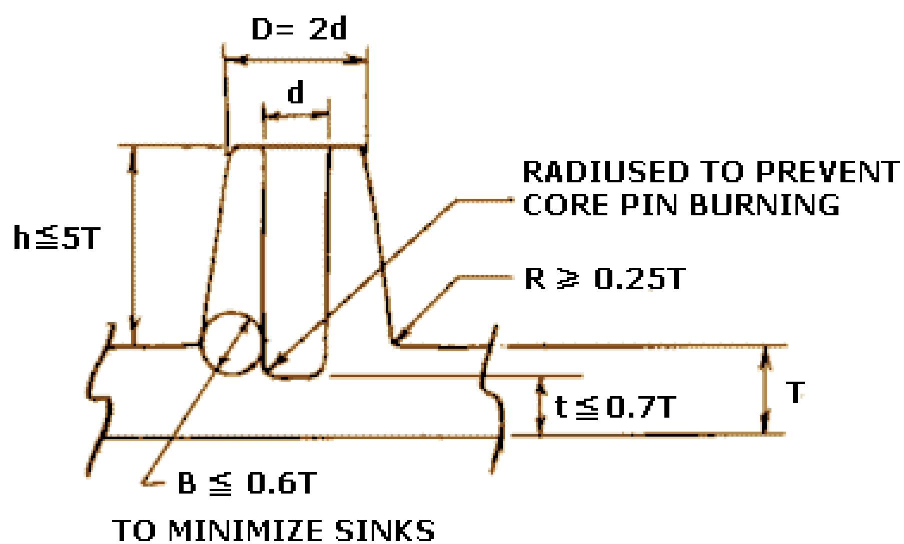
4. Boss

- Before designing a boss, consider its function and the forces acting on it during assembly and service
- If the forces are not great, it may be possible to dispense with support ribs, otherwise:
  - Anchor the boss to the base wall with buttress ribs.
  - If possible, anchor the boss to the side wall with a flat rib.
  - Avoid rib arrangements that result in small mould cores or complicated mould machining set-ups.



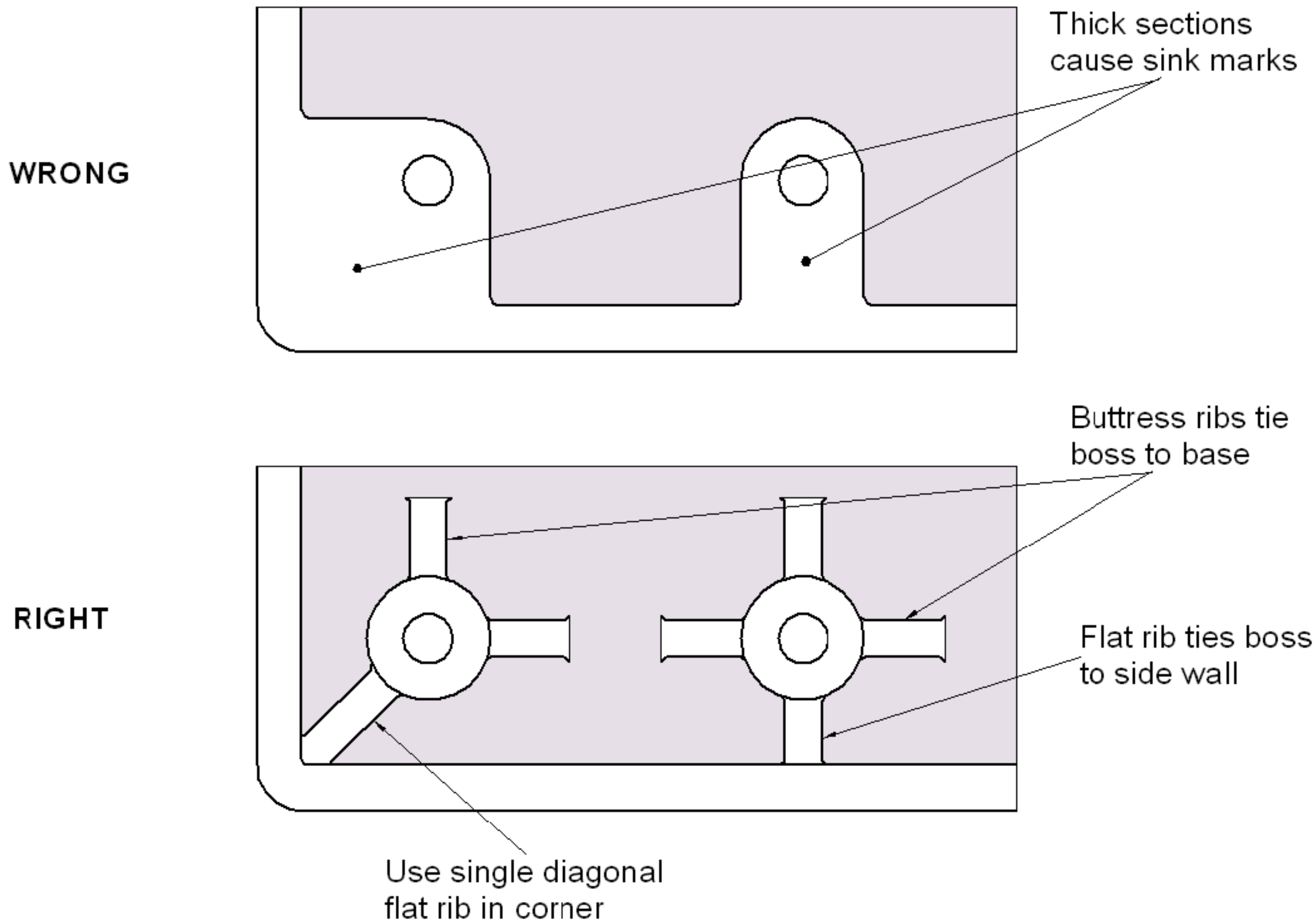
Boss

- Bosses are used for the purpose of registration of mating parts or for attaching fasteners such as screws or accepting threaded inserts (molded-in, press-fitted, ultrasonically or thermally inserted).
- The **wall thicknesses** should be **less than 60 % of nominal wall** to minimize sinking. However, if the boss is not in a visible area, then the wall thickness can be increased to allow for increased stresses imposed by self-tapping screws.
- The **base radius** should be a minimum of **0.25 x thickness**



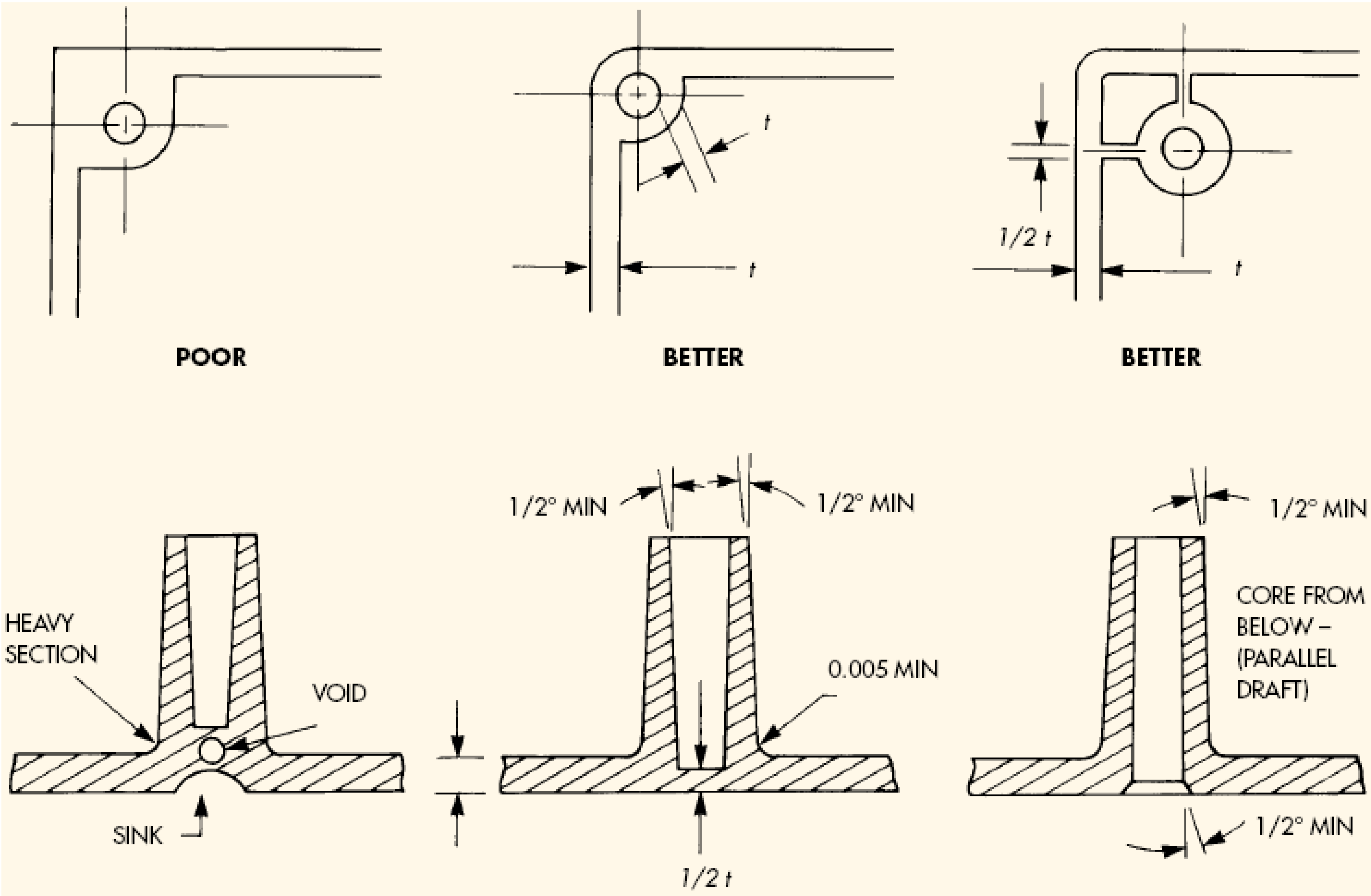
BOSS DESIGN GUIDELINES

Boss



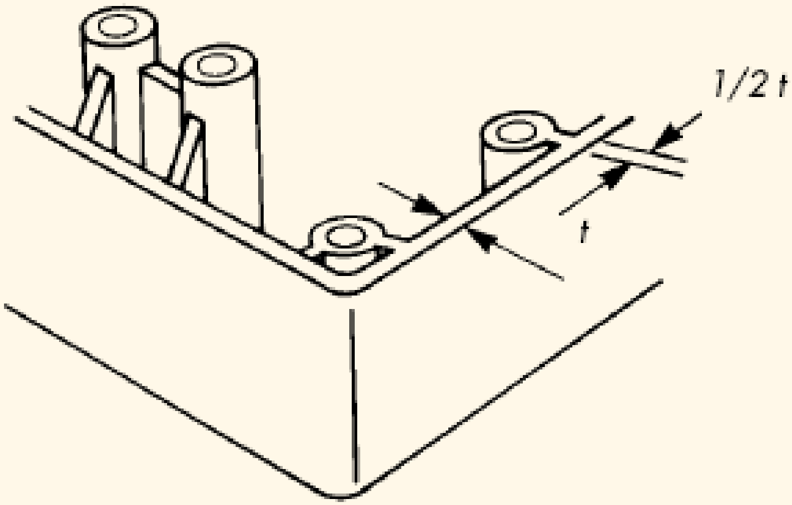
There is a right and a wrong way to support bosses

Boss

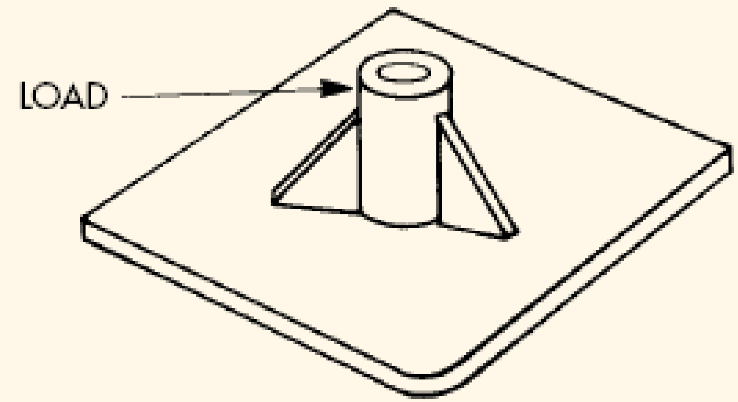


POOR
t
t
BETTER
1/2 t
t
BETTER
HEAVY SECTION
VOID
SINK
1/2° MIN
1/2° MIN
0.005 MIN
1/2 t
1/2° MIN
CORE FROM BELOW – (PARALLEL DRAFT)
1/2° MIN

Boss



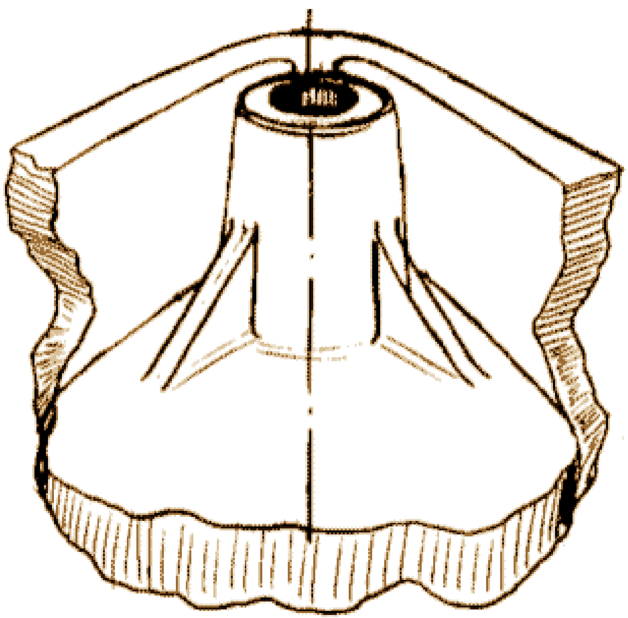
CONNECTING BOSSES TO
OUTSIDE WALLS WITH RIBS



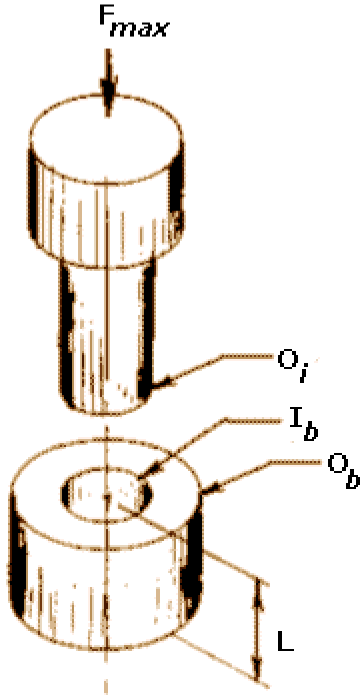
USE GUSSETS RATHER THAN
VERY THICK BOSSES WHEN
RESISTANCE TO LOADING IS REQUIRED

Boss Threads

- Select the right screw type - thread-forming or thread-cutting -to suit the plastics material.
- Use a counter-bore to reduce stress at the open end.
- Make the hole deep enough to prevent screw bottoming.
- Use the manufacturer's design recommendation, otherwise use the factors in this design guide as a starting point.
- Test, if the application is critical.



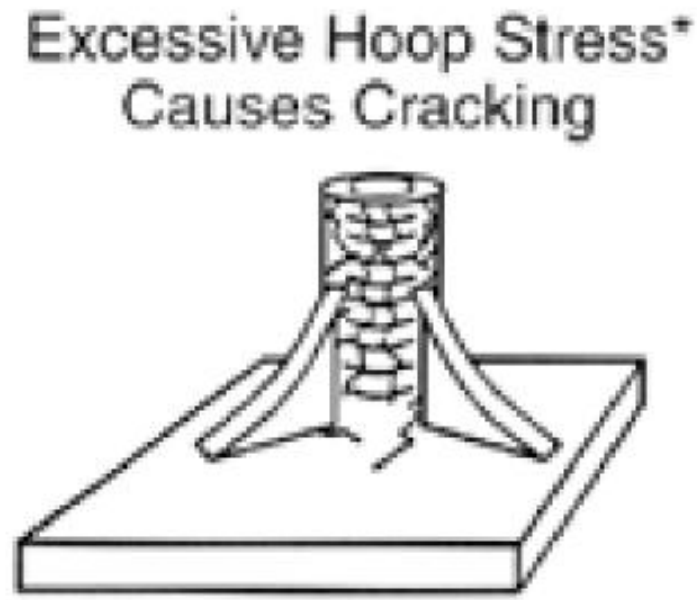
BOSS STRENGTHENING TECHNIQUES



INSERTION OF INSERT INTO A BOSS

Boss Cracking

Another problem concerning high stress occurs with molded-in inserts. The plastic melt heats the metal of the inserts. During the cooling stage of injection molding, the plastic part cools, but the plastic boss surrounding the metal insert is reheated by the heat from the insert. This allows the plastic to continue to shrink around the insert, causing excessive hoop stress* that will eventually cause the boss to crack. The better design and process would be to use ultrasonic insertion or a hot probe (such as a heat staking unit) after the molded part has cooled throughout.



Hoop stress: stress within the circumference of the boss

Screw Tensile Force Calculations

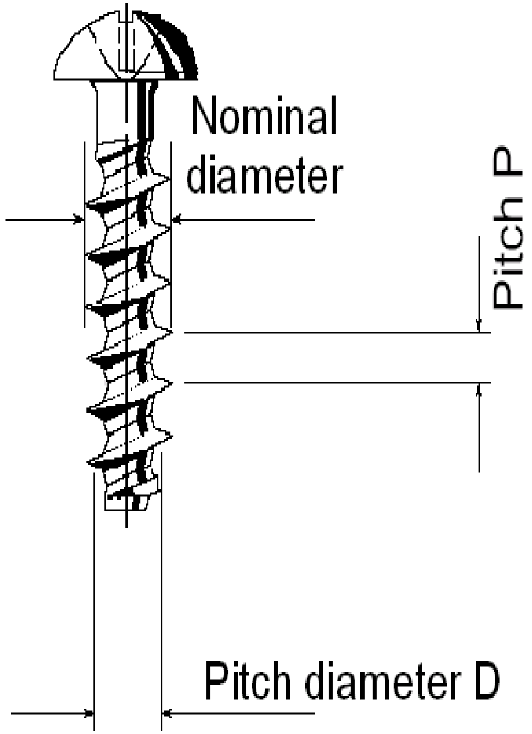
SAMPLE CALCULATION

Using a 2.5 mm nominal diameter screw in an ABS boss.

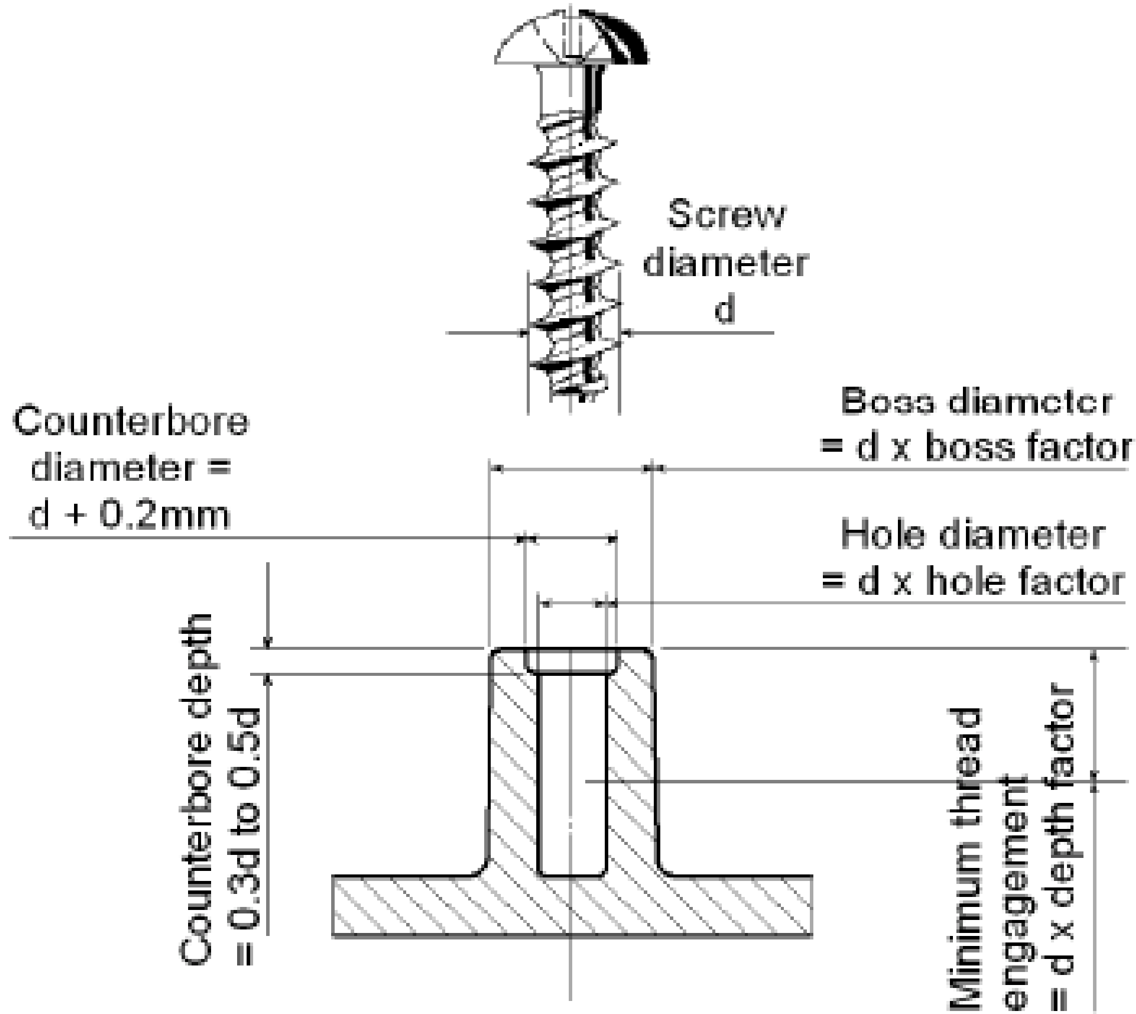
A typical tensile stress value for ABS is about 35 MPa but the 5-year value is only half that at 17.25 MPa. D = 2 mm, L = 6 mm, P = 1.15 mm. The safety factor is 2, and the dynamic coefficient of friction for ABS on steel is 0.35.

$$F = \left(\frac{17.25/2}{\sqrt{3}} \right) (\pi \times 2 \times 6) = 188 \text{ Newtons}$$

$$T = \left(\frac{188 \times 2}{2} \right) \left(2 \times 0.35 + \frac{1.15}{2\pi} \right) = 0.166 \text{ Newton metres}$$



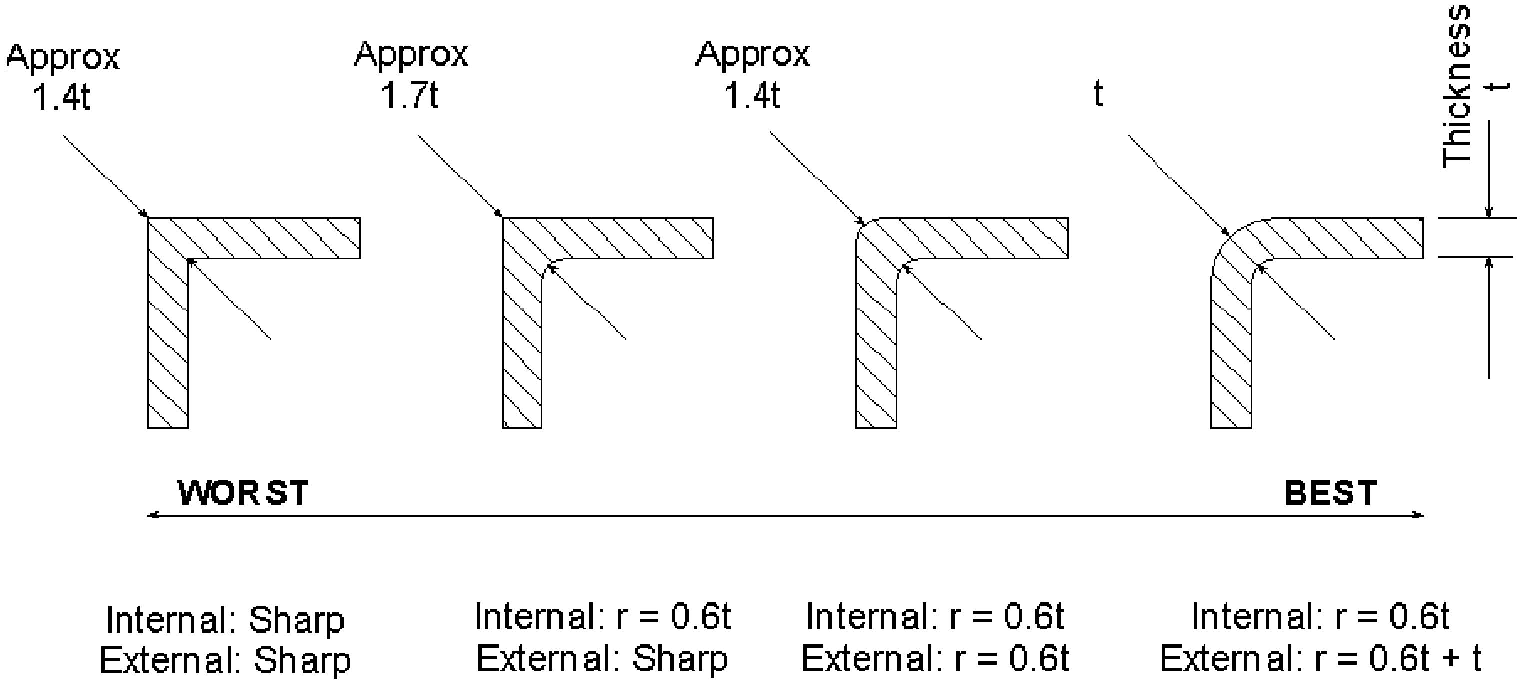
Boss Dimensions



Boss dimensions are a function of material and screw diameter

5. Corners/Radius

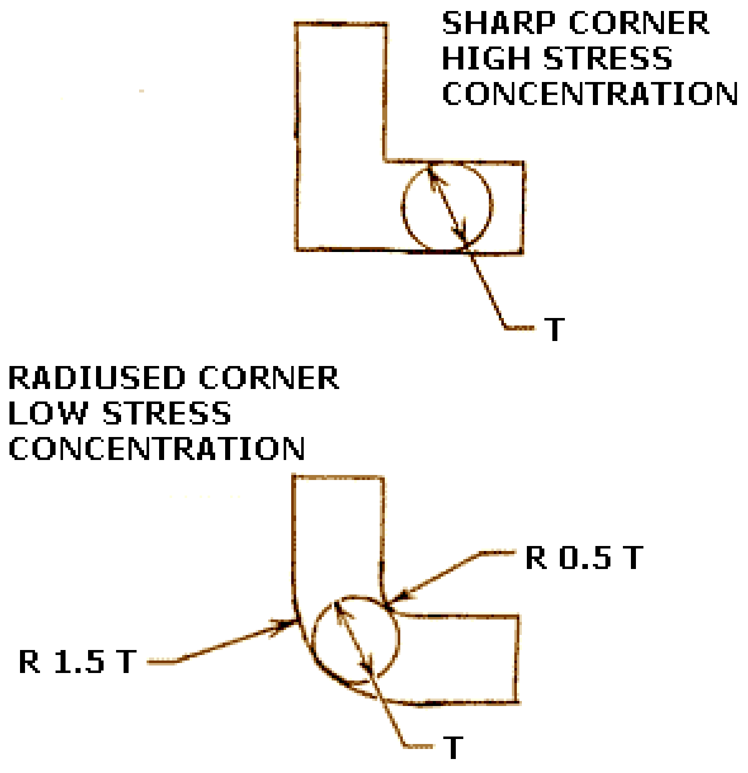
- Avoid sharp internal corners to avoid stress concentration.
- Internal radii should be at least 0.5 and preferably 0.6 to 0.75 times the wall thickness.
- Keep corner wall thickness as close as possible to the nominal wall thickness. Ideally, external radii should be equal to the internal radii plus the wall thickness.



Good and bad corner design

Corners

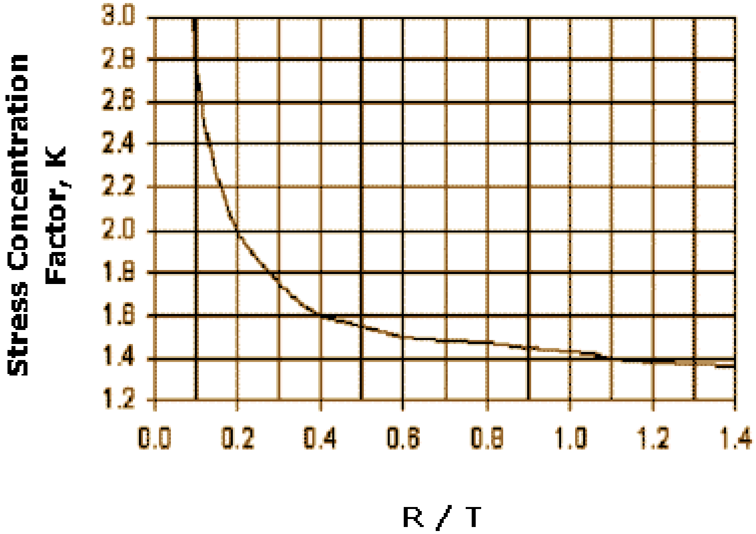
- In addition to reducing stresses, fillet radiuses provide streamlined flow paths for the molten plastic resulting in easier fills.
- Typically, at corners, the inside radius is 0.5 x material thickness and the outside radius is 1.5 x material thickness. A bigger radius should be used if part design will allow it.



RADIUS RECOMMENDATION

Stress Concentration, Radius & Thickness

Sharp corners need to be radiused. The stress concentration factor varies with radius, for a given thickness.



STRESS CONCENTRATION FACTOR , K
(AS A FUNCTION OF RADIUS/THICKNESS)

Actual Stress = Stress Concentration Factor K x Stress Calculated

As can be seen from the above chart, the stress concentration factor is quite high for R/T values less than 0.5. For values of R/T over 0.5 the stress concentration factor gets lower. The stress concentration factor is a multiplier factor, it increases the stress.

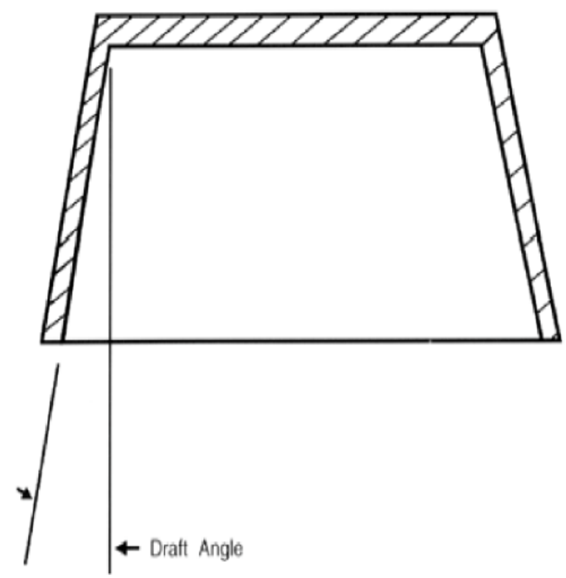
This is why it is recommended that **inside radiuses be a minimum of 1 x thickness**.

6. Draft

A good definition for draft would be: *The degree of taper of a side wall or rib needed to allow the molded plastic part to be removed from the metal mold.*

- Drafts (or taper) in a mold, facilitates part removal from the mold.
- The amount of draft angle depends on the depth of the part in the mold, and its required end use function.
- The draft is in the offset angle in a direction parallel to the mold opening and closing.
- It is best to allow for as much draft as possible for easy release from the mold.
- As a nominal recommendation, it is best to allow **1 to 2 degrees of draft**, with an **additional 1.5° min. per 0.025 mm** (0.001 inch) **depth of texture**

A draft angle of 1/2 degree is regarded as minimum for most applications. Draft angles of 1 1/2 to 2 degrees per side are considered normal for plastic injection molding.



For Texture;

- **Textures and Lettering** can be molded on the surfaces, as an aesthetic aid or for incorporating identifying information, either for end users or factory.

- Texturing also helps hide surface defects such as knit lines, and other surface imperfections.

- The depth of texture or letters is somewhat limited, and extra draft needs to be provided to allow for mold withdrawal without marring the surface.

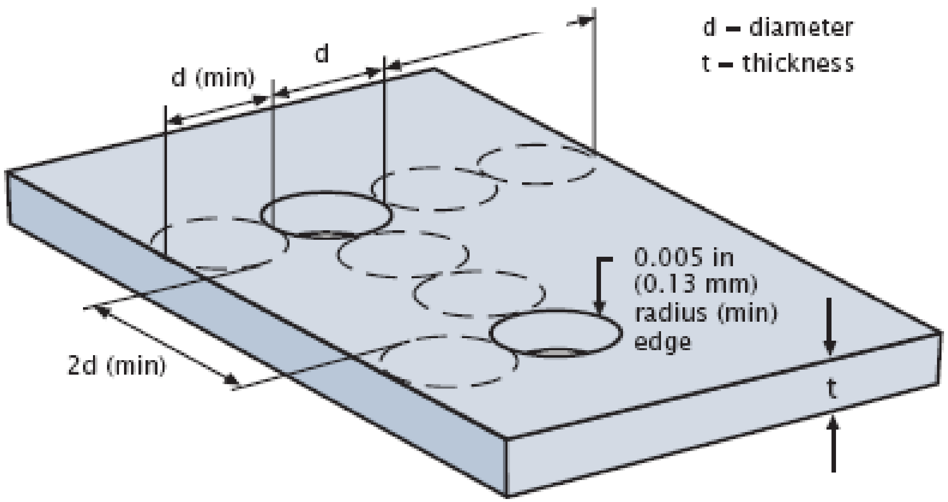
- Draft for texturing is somewhat dependant on the mold design and the specific mold texture.

- Guidelines are readily available from the mold texture suppliers or mold builders.

- As a general guideline, 1.5° min. per 0.025mm (0.001 inch) depth of texture needs to be allowed for **in addition** to the normal draft. Usually for general office equipment such as lap-top computers a texture depth of 0.025 mm (0.001 inch) is used and the min. draft recommended is 1.5 °.
  More may be needed for heavier textures surfaces such as leather texture (with a depth of 0.125 mm/0.005 inch) that requires a min. draft of 7.5°.

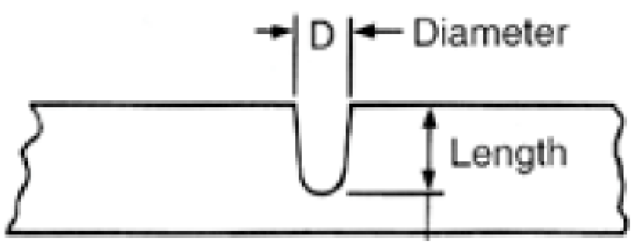
8. Holes

Where a hole is required in a molded part, the design must take into consideration conditions to minimize stress. Holes should be positioned at least a distance equivalent to the hole's diameter and no more than twice the hole's diameter away from the side wall. Consideration should also be given to minimize weld line formation with a careful review of gate location as the hole will restrict flow. As with bosses and ribs, all comers should be radiused.

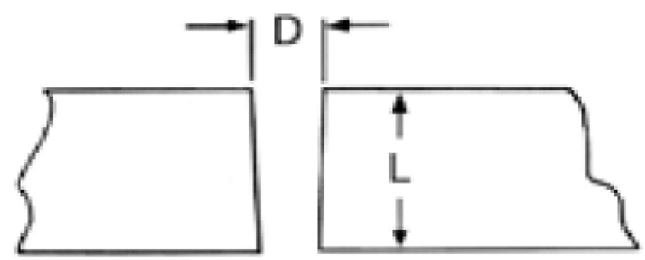


Blind & Through Holes

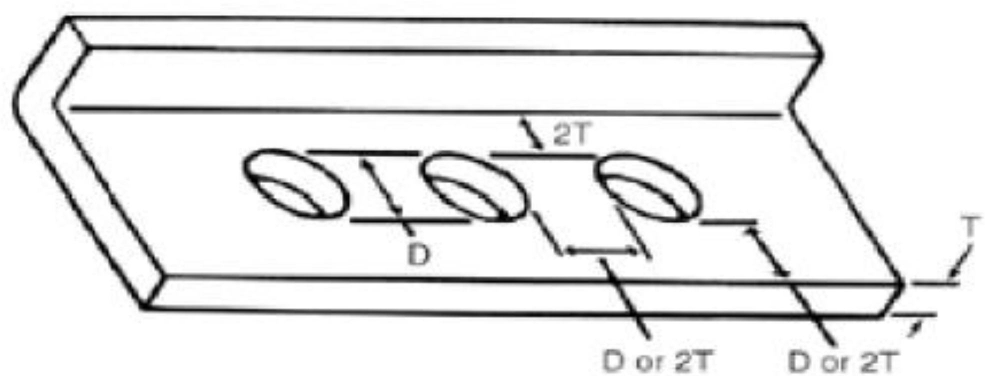
Holes are easy to produce in molded parts. **Core pins** that protrude into the mold cavity make the holes when the part is molded. Through holes in the molded parts are easier to produce than blind holes, which do not go all the way through a part. Blind holes are made by core pins supported on one end only. The pins can be deflected and pushed off center by the pressures of the molten plastic material during the molding process. A good rule of thumb: the depth of the blind hole should be about twice the diameter of core pins up to 3/16", and up to four times the diameter of core pins over 3/16".



Blind Hole with draft

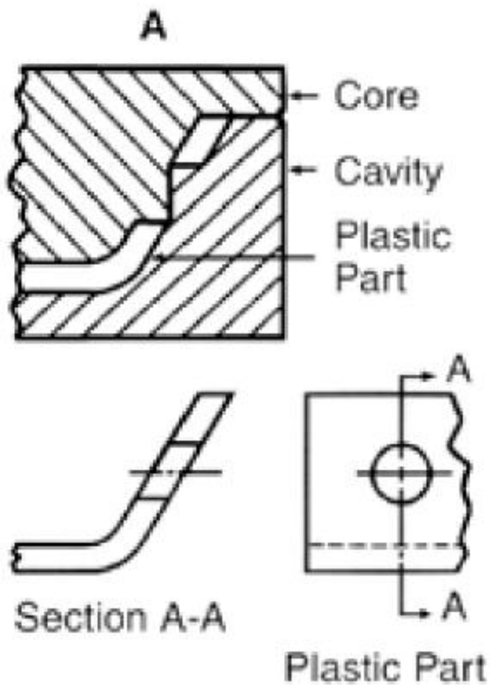


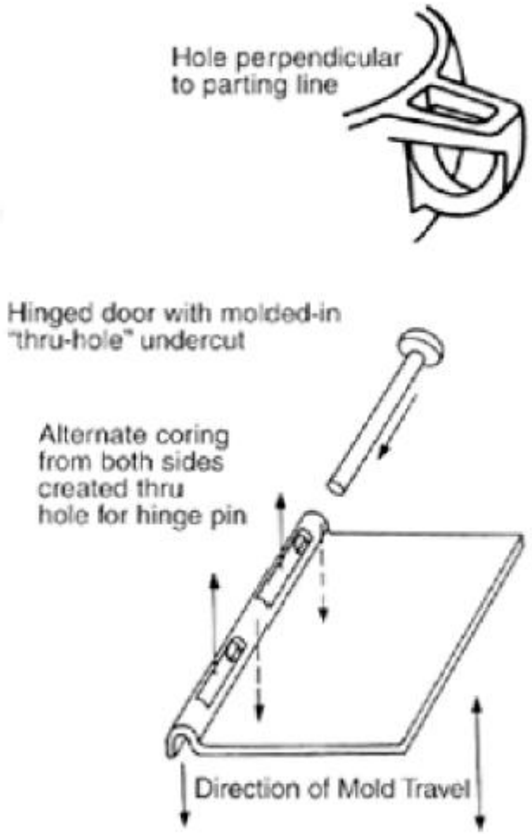
Through Hole with draft



Coring Holes

The coring of holes is easy when the axis is parallel to the parting line. But when holes and other features run perpendicular to the parting line then retractable cores (or cams) are required. Split pins and cores (called passing steel shutoffs) can be used to create some of the features. The designer needs to be aware of the problems of side action cores and the added expenses associated with these types of molds. With a little understanding of how the mold opens and where the parting line will exist, these costly features can be modified.
Rule of thumb: **W***henever possible all design features should be incorporated in the same direction of the mold opening so that cam action can be avoided.*

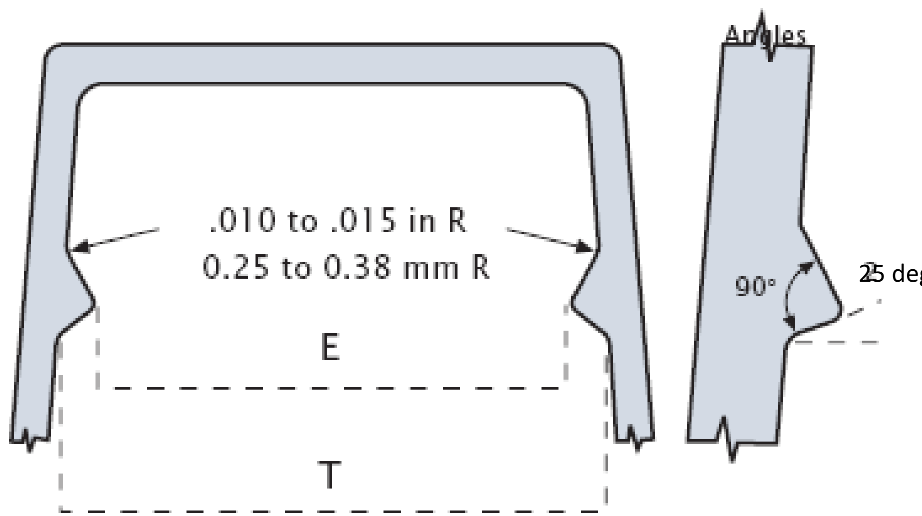




9. Undercuts

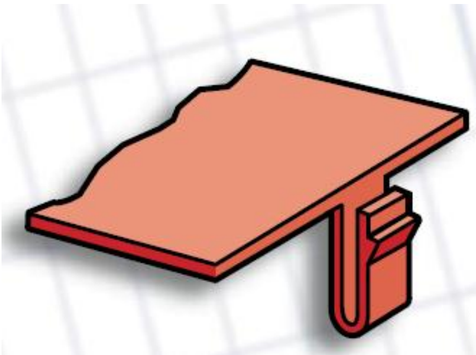
Undercuts should be avoided. However, for articles (such as closures) that require threaded undercuts or snap fits, the undercuts should be designed with a lead-in angle of approximately 250 to facilitate parts stripping. The undercut root should also incorporate a radius of 0.010 to 0.015 in. (0.25 to 0.375 mm) or more, where possible.

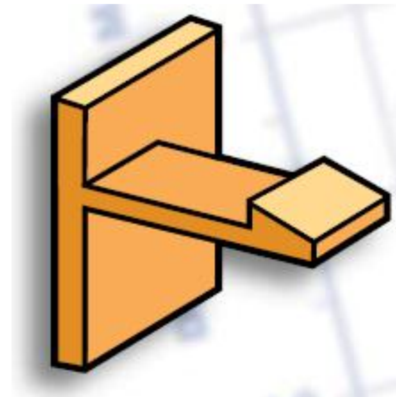
The size of the undercut for a circular part (Fig.10) is determined as the percentage difference between the outer diameter "T", and the inner diameter "E." The percentage of undercut size represents the allowable percentage rate of deformation (strain) of the undercut. It is Basell's experience that undercuts which cause more than 5% deformation can permanently deform the part.



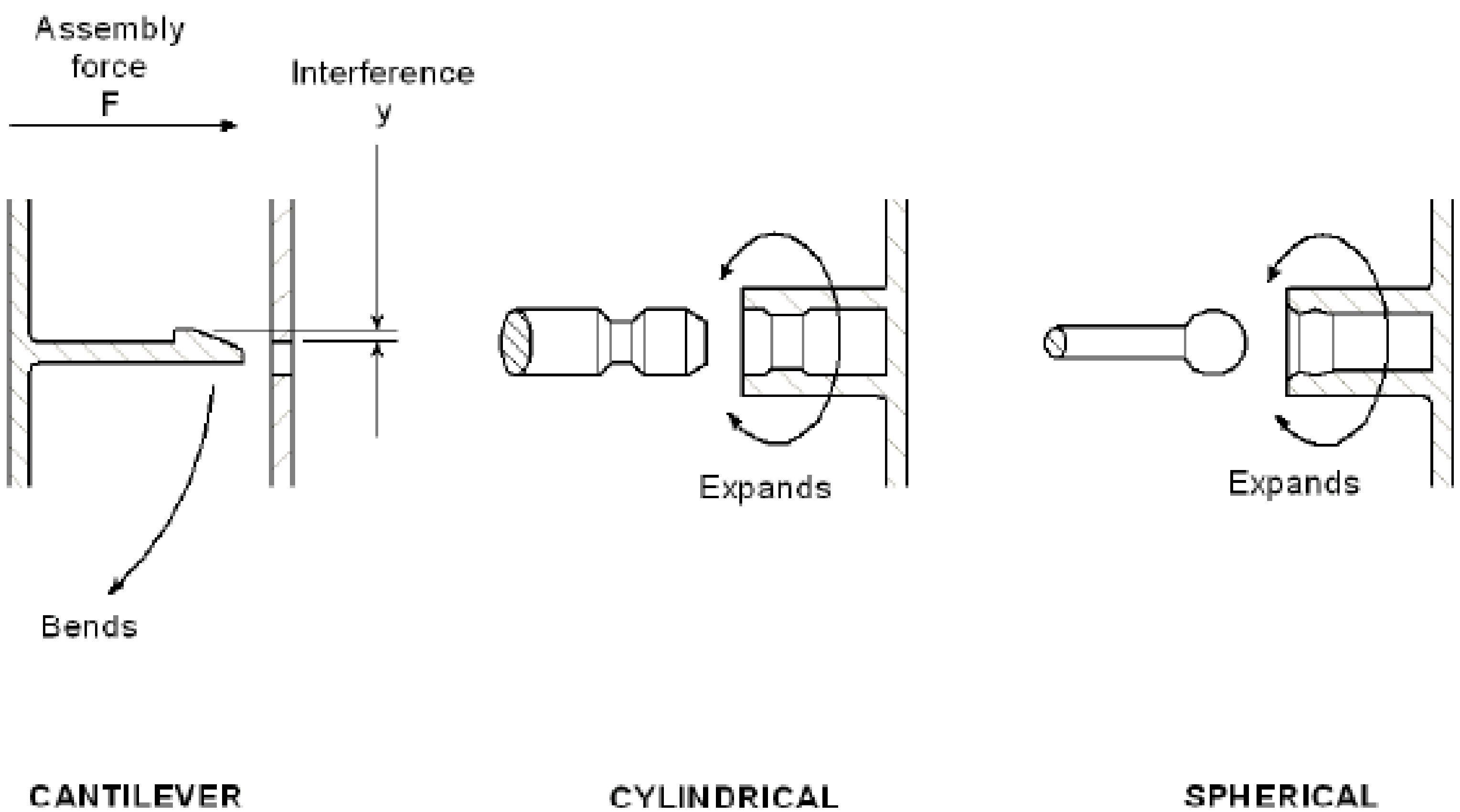
Undercut size (%) or strain (deformation) rate (%) = T – E x 100

Design for Assembly – Snap Fits



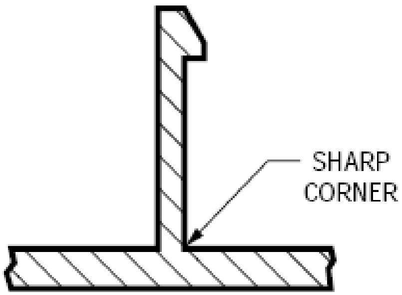


Snap Fits

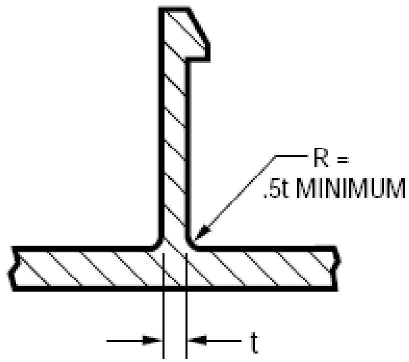


The principal snap-fit types

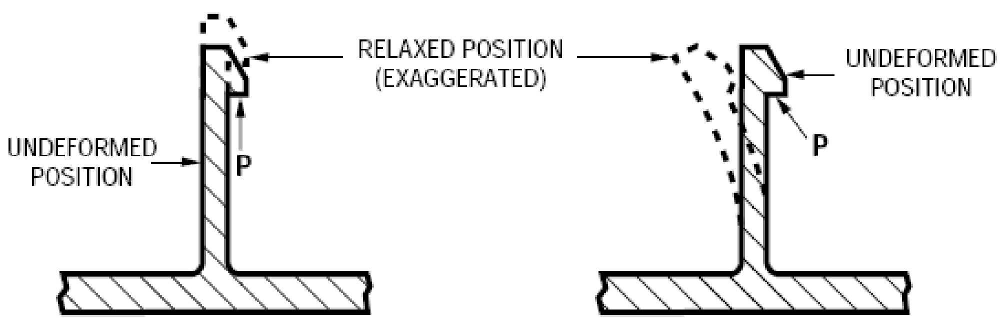
Snap Fits



POOR DESIGN



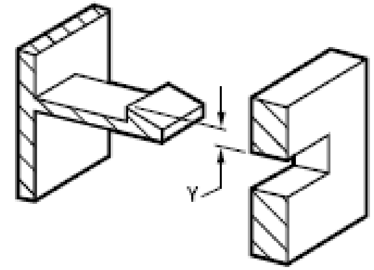
GOOD DESIGN



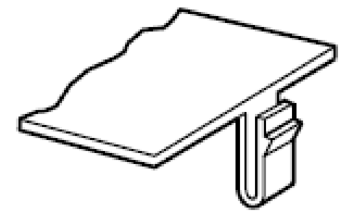
RELAXATION IN TENSION

RELAXATION IN BENDING

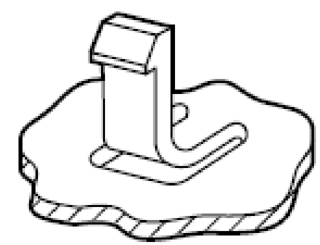
P = MATING PART FORCE



CANTILEVER

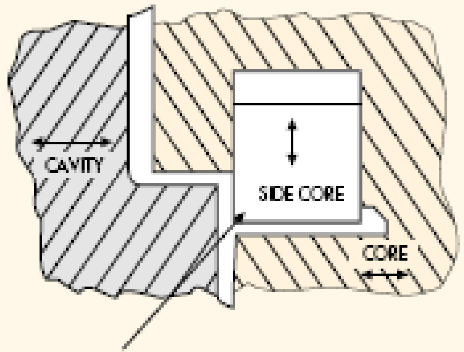


"U" SHAPED CANTILEVER



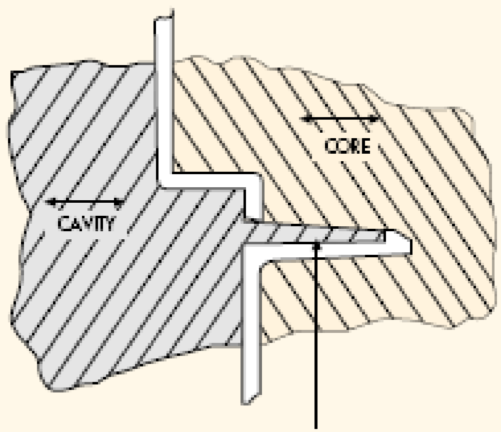
"L" SHAPED CANTILEVER

Snap Fits



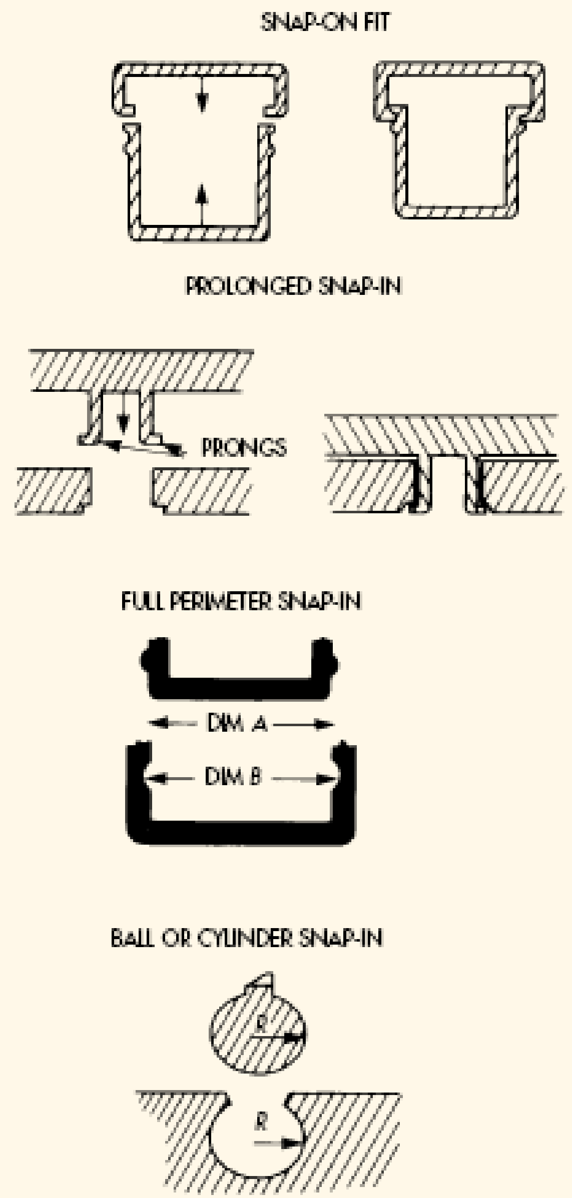
SIDE CORE MOVES DOWN TO MOLD UNDERCUT, AND IS ACTIVATED UP TO ALLOW MOLD TO OPEN

A) MOLDING SNAP-FIT FLEXING FINGER WITH MOLD USING SIDE CORE



PROJECTION FROM CAVITY, THROUGH PART, AND MATING WITH CORE FORMS UNDERCUT WITHOUT NEED FOR SIDE CORE. IT DOES, HOWEVER, LEAVE SMALL "WINDOW" OR OPENING IN MOLDED PART.

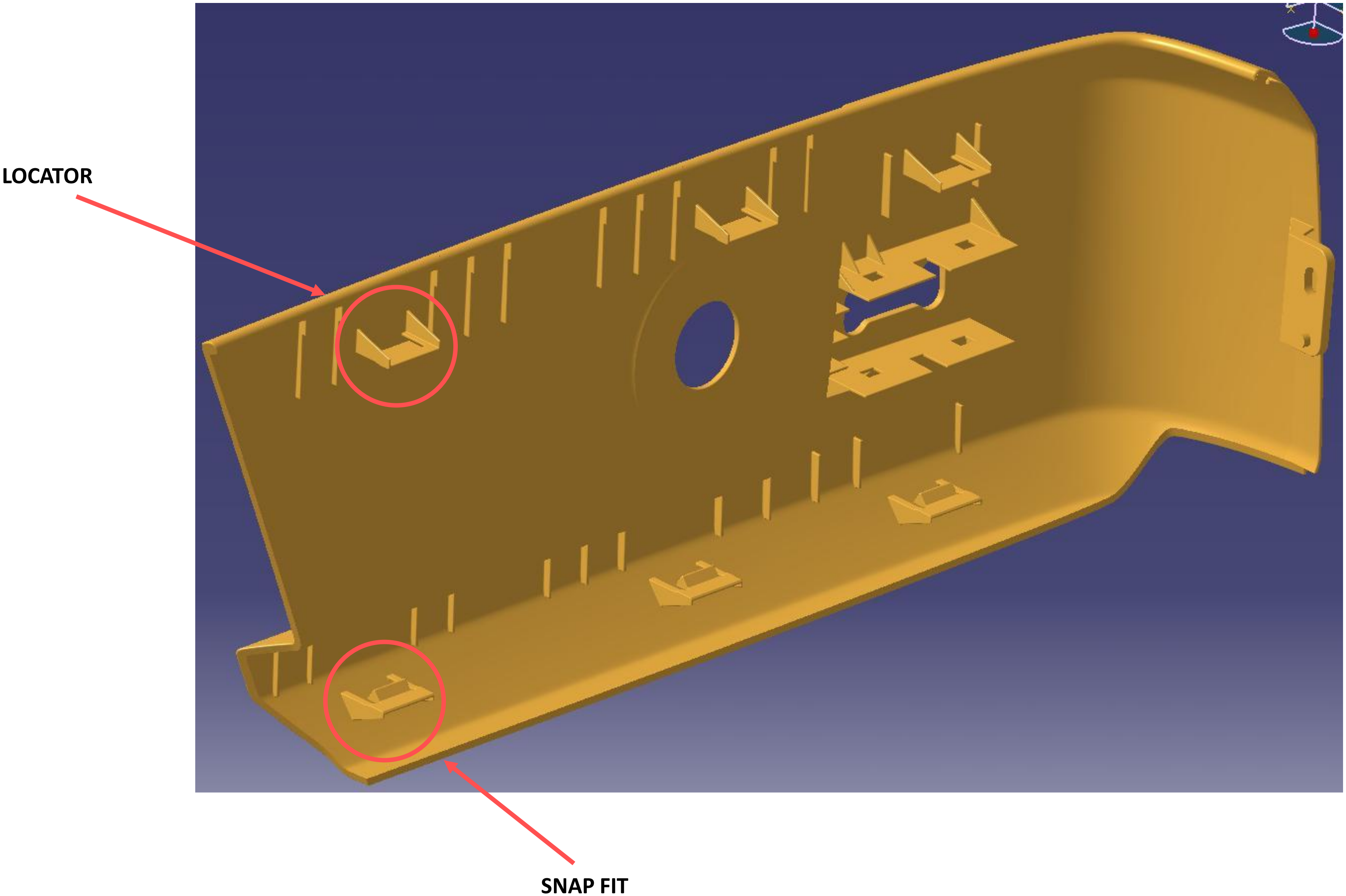
B) MOLDING SNAP-FIT FLEXING FINGER WITHOUT SIDE CORE



Snap Fits

Door handle bezel

Automotive oil filter snaps


LOCATOR
SNAP FIT

HOW PLASTIC COMPONENTS ARE FIXED TO METAL BRACKETS ??