Paul R. Bonenberger

The First Snap-Fit Handbook

Creating and Managing Attachments for Plastic Parts

2nd Edition

Sample Chapter 4:
Enhancements

ISBNs
978-1-56990-388-9
1-56990-388-3

HANSER
Hanser Publishers, Munich • Hanser Publications, Cincinnati

method can be designed into the interface at the same time. Once the design is proven in testing and production, the back-up lock can be eliminated. Should the snap-fit prove unreliable, the back-up lock allows the development program to continue with a reliable attachment for that application.

While the design itself may not be a technical reach, incomplete data about the service loads, material properties or other application requirements may add uncertainty to the design. A back-up lock can allow the snap-fit design to proceed with the confidence that a reliable attachment is possible if the snap-fit does not work.

The design may be such that the locking features of the snap-fit are susceptible to bending or breakage during shipping, handling, assembly or disassembly. If the features cannot be protected by design (see guards) and damage that would render the lock unworkable is possible, a back-up lock ensures the entire part will not be lost because of damage to one feature.

If parts are intended for new designs and also expected to be used on existing designs without provisions for snap-fits, allowing for both methods of attachment accommodates both applications without creating a second set of parts.

Any fastening method may be a candidate as a back-up to a snap-fit and the design criteria should be appropriate to the technology. The same reliability considerations must be applied to the back-up lock as to the original snap-fit.

Back-up locks need not be complex. Providing several clearance holes in a part and pilot holes, bosses or clearance holes in the mating part may be sufficient. Of course, if the back-up lock may become the mainstream design for production then all assembly and processing considerations must be included in the design. If necessary, clearance holes for threaded fasteners can be skinned over and drilled out if needed. Complementary ribs can be added on both parts in proper positions to accept and engage spring steel clips as back-up fasteners.

When a back-up lock is specified because of possible damage in disassembly for service or as a second attachment method on a service part, original assembly issues are no longer critical. Give consideration instead to the tools and fastening methods required for service by the customer or service technician. Do not design a back-up lock that requires special fasteners or special tools.

Rules for back-up locks include:

- Use fasteners identical to other fasteners in the product.
- Use common fasteners that repair facilities are likely to have.
- If high strength is not an issue, and it usually is not in a snap-fit application, design for hardware store type fasteners readily available to the home mechanic.
- Provide adaptable interfaces that permit several sizes, styles or lengths of screw.

4.5 Enhancements for Snap-Fit Manufacturing

Manufacturing enhancements are techniques that support part and mold development, manufacturing and part consistency. Many are documented in standard design and manufacturing practices for injection-molded parts and are already recognized as important factors in plastic part design. They fit neatly into the Attachment Level Construct as enhancements.

These enhancements generally make the part easier to manufacture. Parts that are easier to make are more likely to be made consistently and correctly. They are more likely to perform as expected, an important component of reliability. Another benefit is that they are likely to be less expensive.

Manufacturing enhancements can provide benefits in:

Cost	Shape consistency
Appearance	Mold development
Reliability	Internal stresses
Process cycle time	Performance consistency
Fine-tuning for development	Adjustments for variation

Detailed plastic part design principles, mold design practices and manufacturing procedures are well documented in many other books and standards and that information will not be repeated here. This section is not intended to be a comprehensive guide to the subject of mold design. The intention is to simply capture this particular aspect of snap-fit design as an enhancement and present a few of the more basic concepts that relate directly to snap-fits.

Remember that snap-fit features are subject to the same rules of good mold design as the other features in an injection-molded part. Many snap-fit features are protrusions from a wall or surface and they should be designed according to the same rules as protrusions.

Sometimes, a snap-fit designer relies on the part supplier (if another company) or the experts in their own company to provide the information and design expertise for part processing. There is nothing wrong with this; one should rely on the experts. However, it does not hurt to know enough to be able to ask some intelligent questions. You may occasionally catch something they have overlooked. The part designer is also most familiar with the requirements of the application and is in the best position to ensure they are properly considered.

Manufacturing enhancements fall into two groups. Those that improve the part making process we call *process-friendly.* Those that allow for relatively easy dimensional changes to the mold, are *fine-tuning* enhancements.

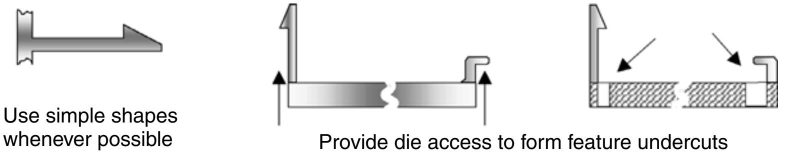
4.5.1 Process-Friendly

Process-friendly design is simply following the recommended and preferred plastic part design practices. Process-friendly parts are robust to the molding process and are likely to be higher quality, less expensive and more consistent in performance than parts that are not.

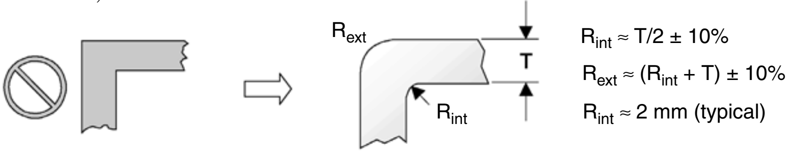
The information shown in this section was drawn from a number of publications. It seems to represent general design knowledge because very similar or identical information was typically found in multiple documents. Rather than cite numerous publications for each item presented all the publications are listed at the end of this chapter.

The single most important rule is to keep the design simple: the simplest design that will work is obviously the best, Fig. 4.15a. Simple feature designs mean less costly molds and greater consistency. When moving parts are required in the mold to make under-cuts and hidden features, die complexity and cost goes up. Access for molding under-cuts is an ever-present issue with mold design and snap-fits are no exception. Features that can be produced without requiring the added complexity of mold features like slides and lifters are always preferred.

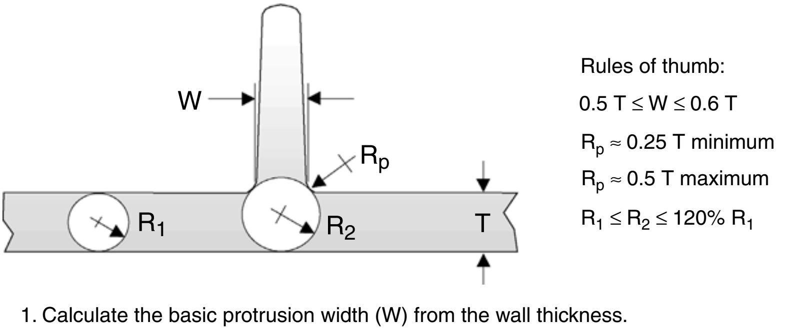
(a) Use simple shapes and allow for die access and part removal



(b) Round all corners, both internal and external



(c) Adjust the protrusion thickness relative to the wall thickness and use a radius at the wall



1. Calculate the basic protrusion width (W) from the wall thickness.
2. Add the draft angle to the basic protrusion width.
3. Add a radius (R_p) at the protrusion base.
4. Verify that the material volume at the protrusion base does not exceed about 120% of the normal wall volume.

Figure 4.15 Common process-friendly design practices

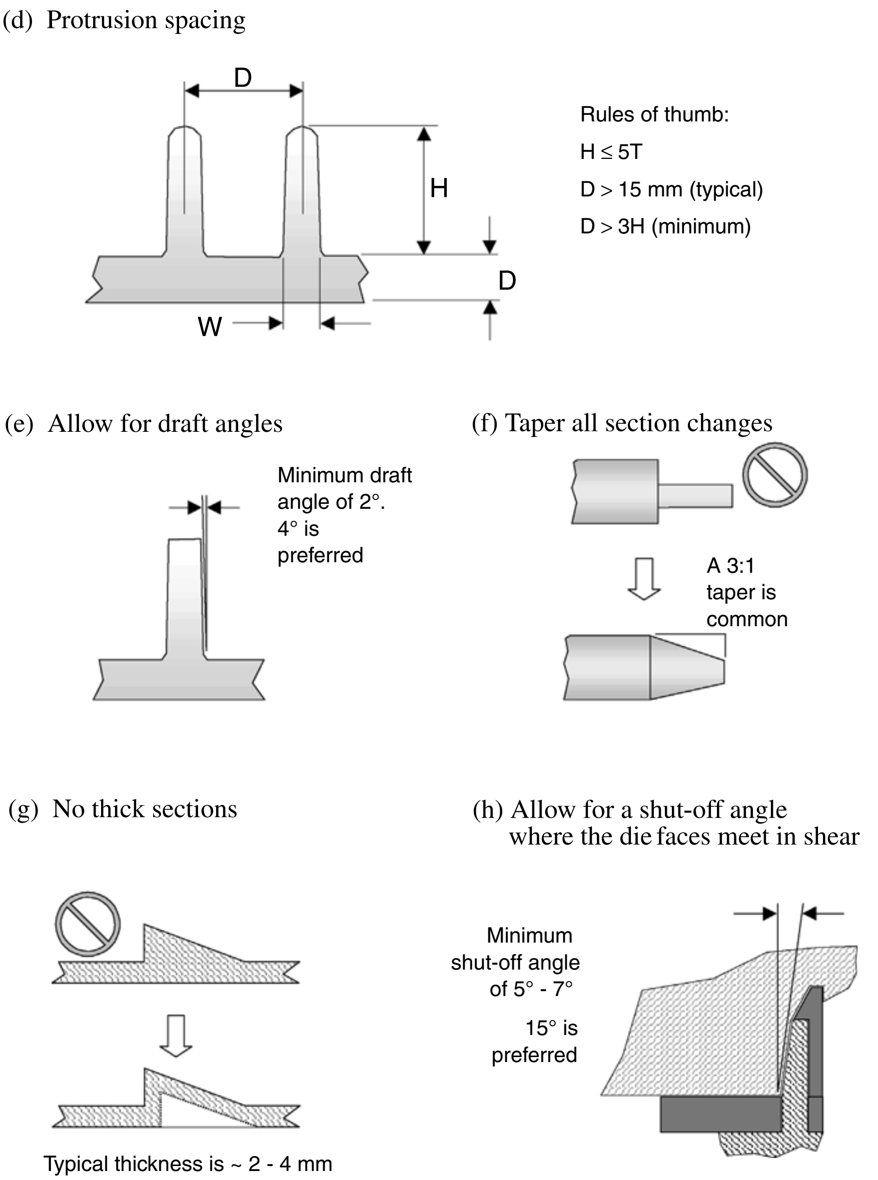


Figure 4.15 (*continued*) Common process-friendly design practices

Sometimes, a complex feature shape may be required if moving parts in the die are to be avoided. In that case, consider the costs and advantages of both designs. Also, consider that analytical tools for predicting lock and locator behavior tend to be less accurate as feature shapes become more complex.

Specify a radius for all inside and outside corners, Fig. 4.15b. The idea is to avoid all sharp corners and maintain a constant wall thickness for smooth plastic flow through the mold. (The melt front does not like surprises.) Corners cause turbulence and are hard to fill. It is not enough to simply ask for fillets and radii in a general drawing note. Put a dimension at every site where a fillet or radius is required.

Sharp internal corners also create sites for stress concentrations. When at the base of a constraint feature, they can cause feature failure. Treat every protrusion feature (hooks, pins, tabs, lugs, etc.) as a rib and follow the guidelines for rib sections and rib spacing. The idea is to maintain a relationship between the wall thickness and the protrusion thickness so that voids or residual stresses at the base of the feature do not occur. Some basic rules are shown in Fig. 4.15c and Fig. 4.15d. Keep in mind however that these are general rules and simply provide a good starting point. Specific plastics can have their own requirements.

If a prototype part shows sink marks on the opposite side of the wall from a protrusion, this is a good indication that voids or residual internal stresses may be present at the base of the feature. These will weaken the feature and may result in failure.

Include a draft angle. This allows the part to be easily removed from the mold. Start with the basic feature size then add the angle to each side, Fig. 4.15e.

Avoid thick sections and abrupt section changes for the same reasons you avoid sharp corners. Another reason is the difficulty of cooling a thick section of plastic. To properly cool a thick section results in significantly longer cycle times and higher cost, Figs. 4.15f and g.

Where die faces come together in shear, a shut-off angle is necessary, Fig. 4.15h. This applies when access for molding hooks or lugs is required, Fig.4.15a.

Gates are the areas where the plastic melt enters the mold cavity and gate style and location are other aspects of mold design that can have a significant effect on the snap-fit features. Gates can affect the constraint feature's location (due to part warping) and the feature's strength. Remember that the mold designer is not likely to know the critical areas of your design and will put the gates at locations they believe are the best sites for mold fabrication and performance unless you indicate otherwise. Gates should be located:

- Away from flexible features and impact areas.
- So that knit lines will not occur at high stress areas, including living hinges.
- In the heaviest/thickest sections so that flow is to the thinner, smaller areas.
- So flow is across (not parallel to) living hinges.
- So flow is directed toward a vent.
- In non-visible areas.
- So that flow distance to critical features is not excessive.

Gate location can also affect part warpage. Be sure the snap-fit features do not move out of position due to excessive part warpage. If they do, guide enhancements may be needed to bring the locks back into proper position for engagement.

4.5.2 Fine-Tuning

Fine-tuning involves adjusting the mold dimensions to result in correct final part dimensions. It is necessary because the nature of the molding process is such that first parts out of the mold will not be perfect. Despite the use of predictive tools and highly controlled processing techniques, one never knows exactly what the part will be like until first parts are made. This is particularly true when the snap-fit designer is concerned with high precision in constraint feature locations and dimensions. Part changes and adjustments during part development become much easier when allowances are made for fine-tuning during part design.

Once production begins, long-term wear, variations in raw materials, design changes and variation in the other part may also require periodic mold adjustments to maintain attachment quality throughout the part's production run.

In anticipation of changes, plan for easy mold adjustments at strategic locations. The purpose is to avoid large-scale (expensive and time-consuming) mold changes. In other words, make the snap-fit interface "change-friendly".

The first step in adding fine-tuning enhancements is to identify critical alignment and load carrying requirements and the constraint sites that provide that capability. This should have already occurred in the design process because you needed to understand the critical constraint sites to establish constraint and compliance requirements. These sites represent the areas of the part (thus the mold) where fine-tuning is likely to be needed, Fig. 4.16. Fine-tuning site selection also affects compliance enhancement locations. Once these critical sites for fine-tuning are identified, you can decide if *metal-safe* design or *adjustable inserts* are appropriate.

Metal-safe means to fine-tune the part by removing rather than adding metal to the mold. Obviously, it is much easier to simply grind material away in the mold than to first build up an area then shape it by grinding metal away. Once the critical sites have been identified, select initial nominal dimensions and tolerances at or slightly beyond the minimum material condition, Fig. 4.17. Be careful not to carry the idea of metal-safe design to such an extreme

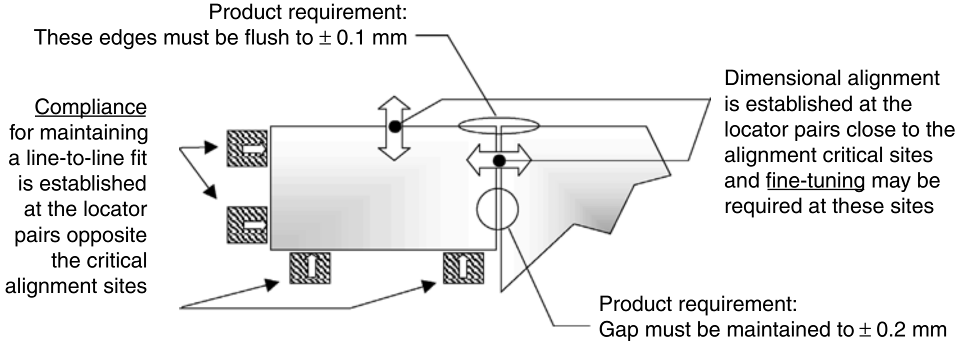


Figure 4.16 Selecting sites for compliance and fine-tuning

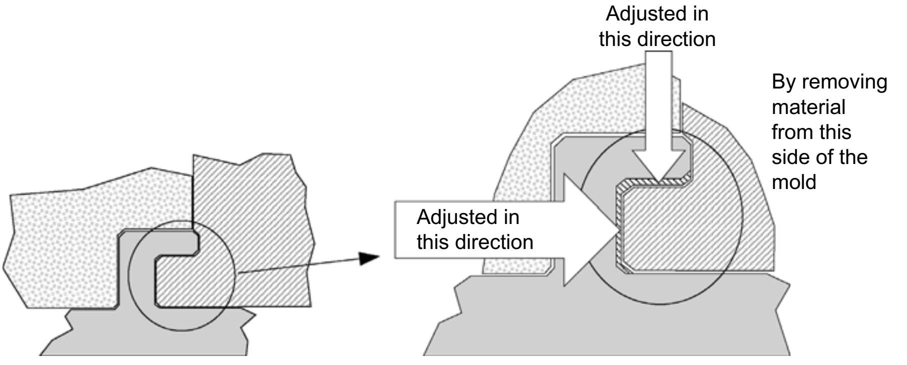


Figure 4.17 Metal-safe fine-tuning on a lug

that first parts out of the mold are not even close to design intent. This will render the parts useless for fine-tuning and just add more work.

Adjustable inserts can also be used to permit fine-tuning critical dimensions on constraint features. Inserts are easily removed from the mold and can be modified and reused or replaced by other inserts, Figs. 4.18 and 4.19. Unlike metal-safe design, inserts allow critical dimensions to be easily adjusted in both directions, either adding or removing material.

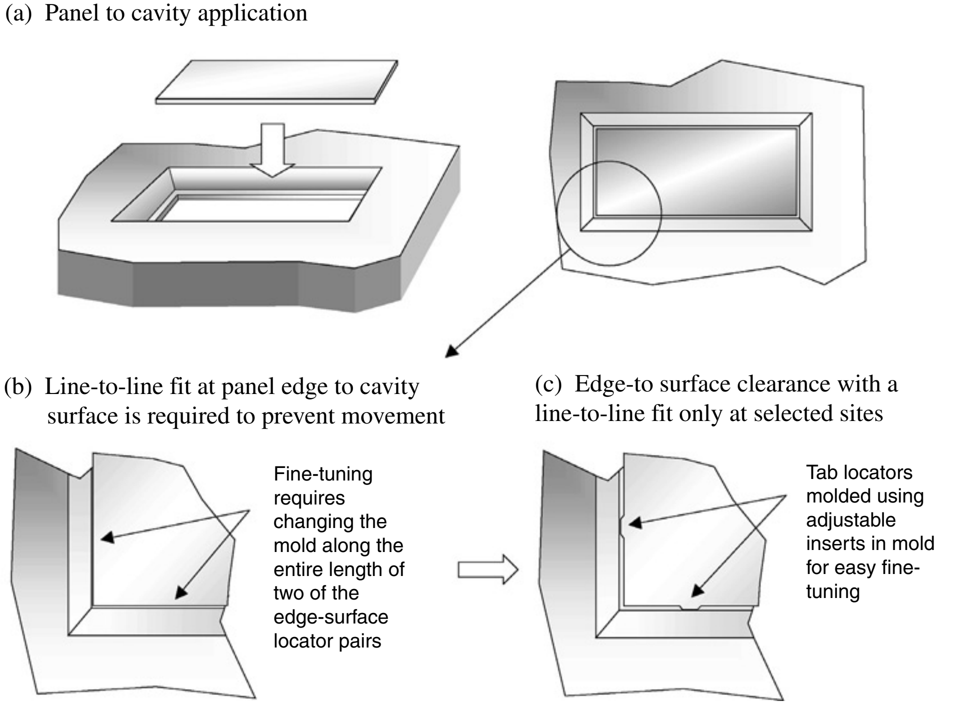
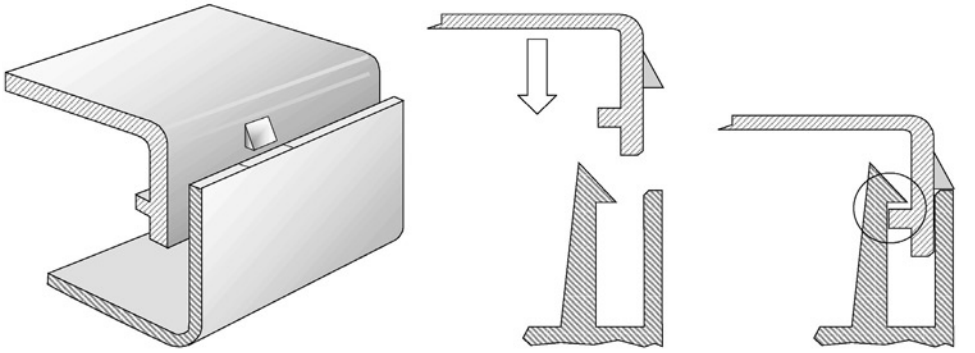


Figure 4.18 Fine-tuning with adjustable inserts

(a) Initial design leaves some clearance at the hook



(b) Fine-tuning at the edge using an adjustable insert brings the hook face into line-to-line contact with the mating surface

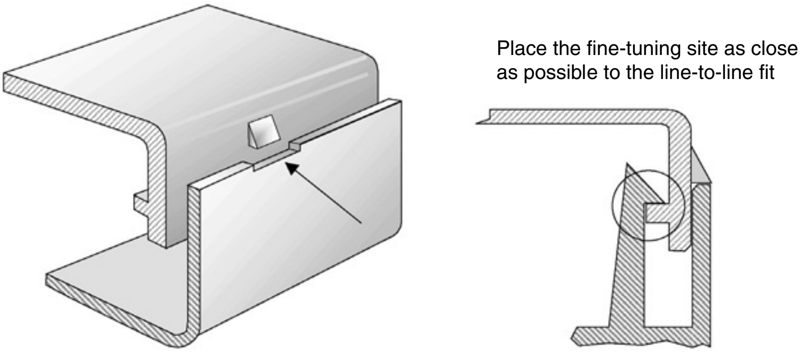


Figure 4.19 Fine-tuning with adjustable inserts

Use of adjustable inserts requires designing for *local adjustment* at the critical constraint sites. This means you have provided distinct locator features in those areas rather than using a large part area such as a surface or edge as a natural locator. Fine-tuning a locator feature or features is much easier than changing the mold for a major part feature.

In the application shown in Fig. 4.18, rather than locate at the edge to surface interface (natural locators) the fit of the panel to surface is controlled at specific contact sites around the part perimeters. Fine-tuning adjustments can be made by modifying the inserts at these sites rather than changing the entire part.

Some rules for fine-tuning are:

- Identify the constraint sites that provide critical positioning or alignment. Make allowance for fine-tuning at these sites.
- Identify the constraint features that provide the critical strength in the attachment and determine if fine-tuning will be necessary to adjust performance. Keep in mind that

simply increasing strength by adding thickness is limited by the process-friendly rules. Strength can also be increased by adding structural ribs to the features. These ribs can also be fine-tuned for performance.

- In general, compliance enhancements should be placed at locator pairs that are not fine-tuning sites.
- Select the initial nominal dimensions and tolerances between those sites so that the minimum material condition will occur at the tolerance range maximum. This will put the features slightly undersize. A minimum material condition in the part will result in maximum material in the mold.

4.6 Summary

This chapter provided detailed descriptions of enhancements and rules for their usage. Enhancement features are one of the two physical elements of a snap-fit. They may be distinct physical features of an interface or attributes of other interface features. Enhancements improve the snap-fit's robustness to the variables and unknown conditions that can exist in manufacturing, assembly and usage and are summarized in Table 4.2.

Enhancements are often subtle details in a snap-fit application. They may not be obvious at first glance. It is suggested that the reader study snap-fit applications to become familiar with the usage of enhancements. The luggage closure buckle shown in Fig. 4.20 is a readily available application. If you can compare closures from several manufacturers, you will begin to see how enhancements can affect the overall quality of the application.

4.6.1 Important Points in Chapter 4

Some enhancements are required in every application; others depend on specific needs of the application, Table 4.3. When soliciting bids on a snap-fit application, the required enhancements should be made part of the business case and considered non-negotiable. They are almost as essential to ensuring a high quality and successful snap-fit as are the constraint features. When bidding on an application, enhancements may be the attention to detail that wins you the contract.

During snap-fit development, include enhancements in the initial attachment concepts and in the first detailed parts made when possible. However, including all enhancements in the original design or even the first prototype parts is usually not possible or practical. One must actually assemble and disassemble actual parts to properly assess the need for some enhancements. Desktop manufacturing methods can provide pre-prototype parts with enough detail that requirements for visuals, guides and assists can be identified. Other enhancements (assembly feedback and user-feel, for example) usually require that parts be made from the design intent plastic using production molds to properly identify and develop enhancement details to meet product requirements. The need for retainers may not be apparent until parts undergo physical testing. Table 4.4 shows the steps in the snap-fit