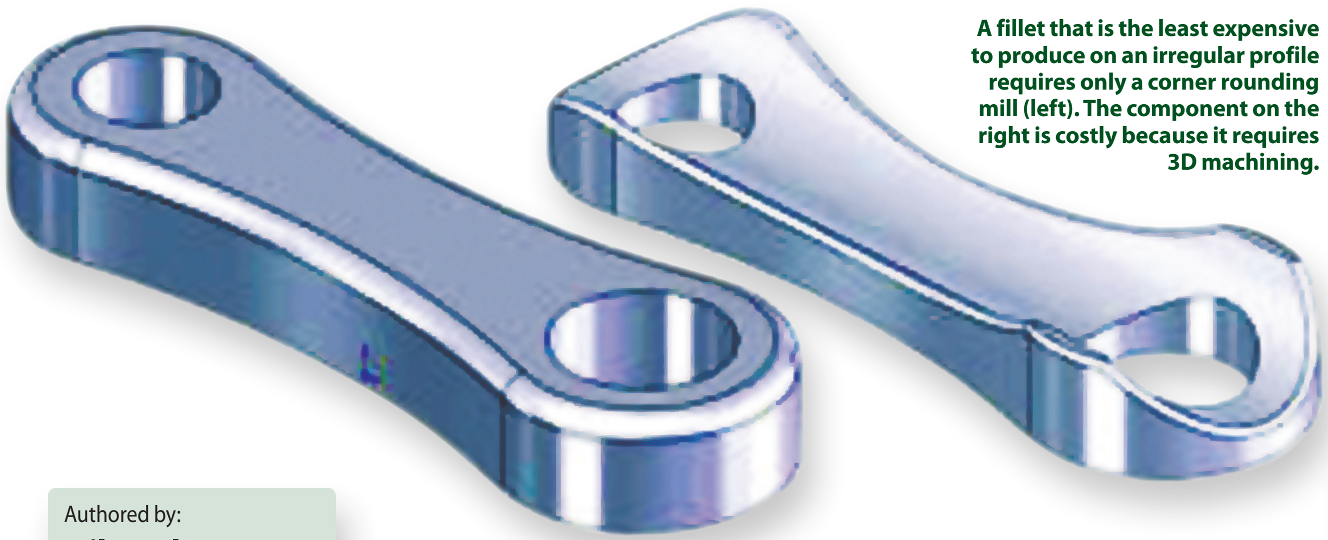
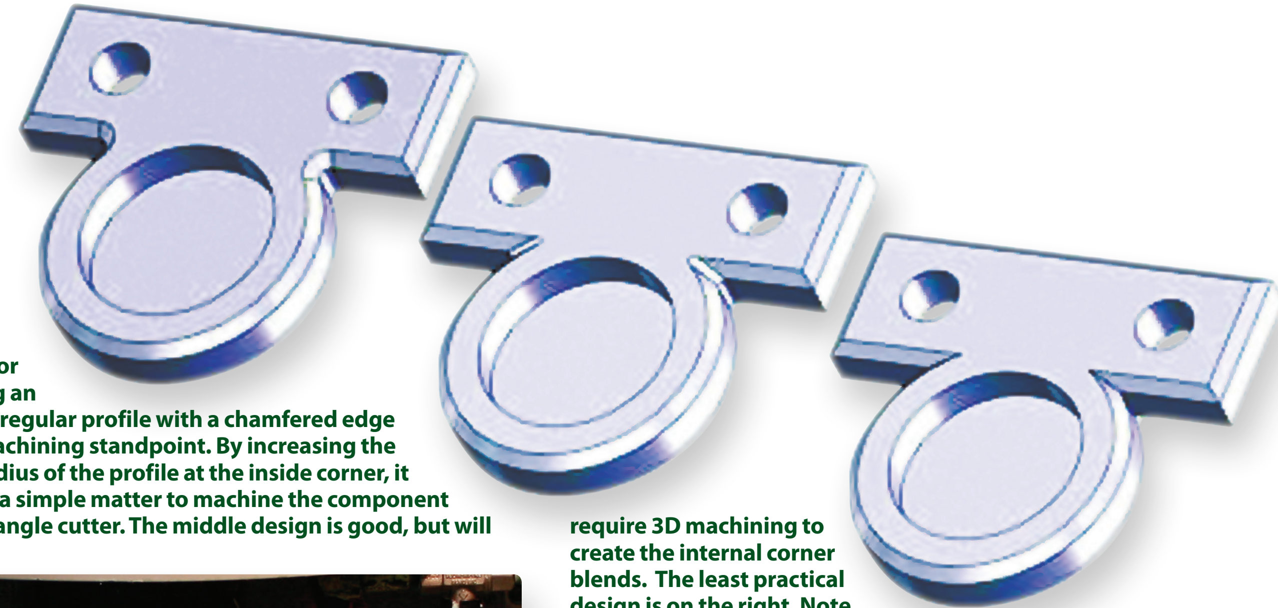


A fillet that is the least expensive to produce on an irregular profile requires only a corner rounding mill (left). The component on the right is costly because it requires 3D machining.



The left image shows the most practical method for designing an outside irregular profile with a chamfered edge from a machining standpoint. By increasing the corner radius of the profile at the inside corner, it becomes a simple matter to machine the component using an angle cutter. The middle design is good, but will require 3D machining to create the internal corner blends. The least practical design is on the right. Note all of the sharp internal corners on the profile and the chamfer.

Think like a machinist when creating solid models

Machining issues designers should understand.

Authored by:
Milton Florest
President
Tooling Research Inc.
Walpole, Mass.

Resources:
Tooling Research Inc., (508) 668 -1950, *www. tooling-research.com*

If you have specific questions or comments you are welcome to e-mail them to me at info@tooling-research.com

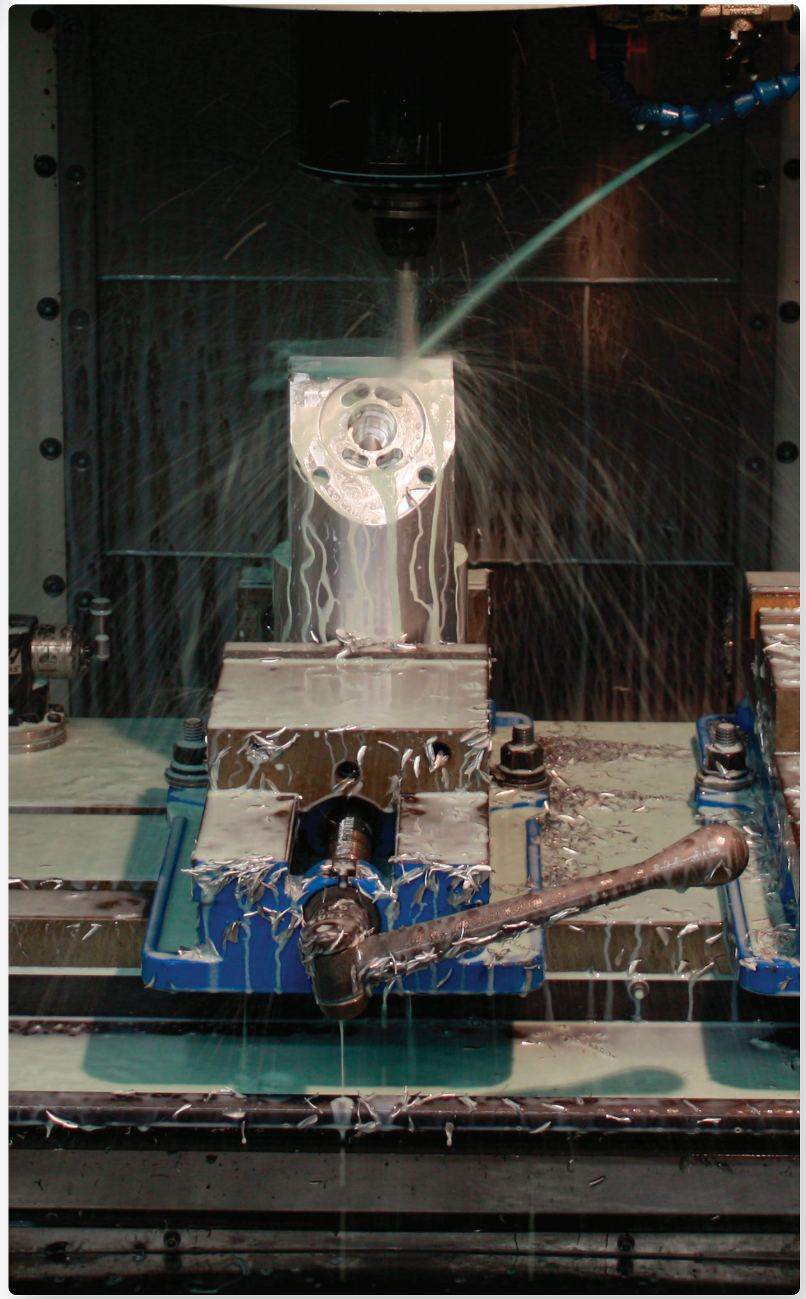
Edited by **Leslie Gordon**
leslie.gordon@penton.com

Until the early 1990s, solid-modeling software was so expensive that only the largest companies could afford it. The software was difficult to use and its steep learning curve often deterred designers from making the switch from drafting boards or 2D programs to 3D. But today, programs are affordable and easier to use. Most designers and engineers learn solid modeling in colleges or technical schools and then launch into the business world to start designing machines and components.

But some newer (and even some more-experienced) designers and engineers never consider whether the 3D CAD component they have just created is actually manufacturable. The assumption is, "if I can create it on the computer, it can be made." In most cases this is true, but at what cost?

The result: The machinist is often confronted with unnecessarily difficult or sometimes impossible machining scenarios. These can result in costly designs and are generally attributed to a lack of exposure to the machine-shop setting and those processes related to machining.

As an engineer with over 15 years of SolidWorks experience and an extensive machine-shop background derived from more than 50 years in the business, I encounter machining challenges every day. I have made a business of reviewing outside designs and suggesting methods to make them compatible with general machining practices.



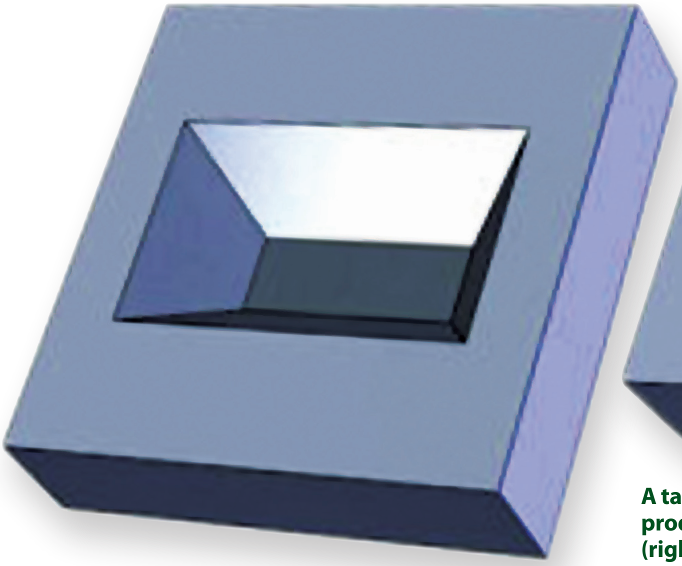
Some readers might claim that their designs are meant to be molded rather than machined, therefore this article doesn't apply to them. However, a stable working prototype is often required before committing to production. Not all models can be satisfactorily duplicated using stereolithography or other additive-printing methods when the component is going to be used as a functioning part. For newly designed, one-of-a-kind items, or small quantities, molding is not an option.

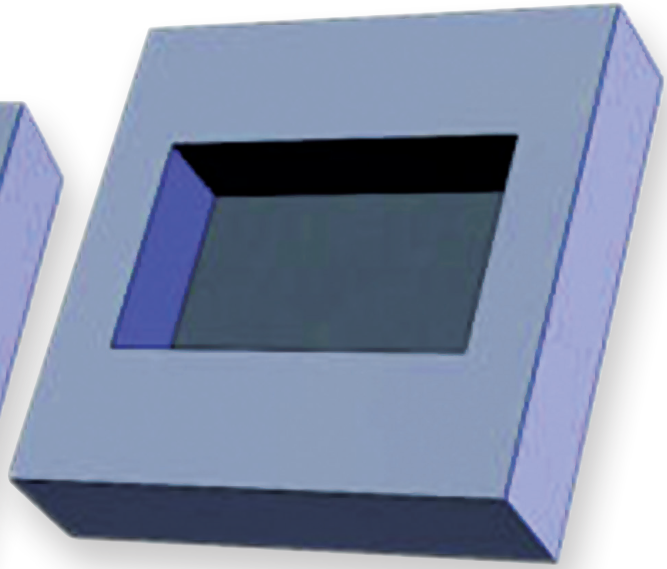
Hopefully, pointing out just a few of the problems machinists face will offer insights into ways to make designs more economical to manufacture.

Is the stock size handy?

Often when researching materials, the required size appears on a vendor's Web site or catalog but is unavailable when you try to order it. Whenever possible try to design around standard material size in the coarser fractional dimensions such as $1/8$, $1/4$, $1/2$, $5/8$, and $3/4$ in. Try and avoid the $1/16$, $7/16$, and $9/16$-in. sizes because most mills either don't stock them, or require a special run. (This guideline applies less to sheet stock or round bar than it does to rectangular bar and plate.) Should your designs require metric dimensions, as many medical products do, consider that in the U. S., stock metric material is sometimes more difficult to obtain.

Whenever possible, try to design long, thin parts around more-readily available stock sizes to avoid or minimize machining long surfaces. Removing material from one side of a large surface usually causes the material to distort or bow, forcing machinists to keep removing material from alternate sides to bring back the straightness. A time-consuming operation, it often takes many passes to make the material flat again and, in some cases, the results





A tapered pocket with no radii (left) may need EDM to produce. A straight side pocket with no corner radii (right) needs EDM or broaching to produce.

are not predictable. As a rule, larger cross sections of material that need to be machined to thin profiles will likely distort.

As machinists review customer drawings, they evaluate features, the steps it will take to produce the part with the fewest setups, and workholding — a major consideration. Whenever possible, design your part to have at least two opposing parallel flat surfaces or a truly cylindrical surface, so it can be gripped by conventional vises and tooling. Otherwise, custom fixturing or additional material will be needed to anchor the part. This can boost manufacturing costs significantly.

Design considerations

Most designs evolve around the part function and how the component will interact with other mating parts. Designers are likely to begin by choosing a basic shape, which will probably have holes, slots, steps, and other features.

Designers with machining knowledge might visualize the part as a blank of sufficiently sized raw material and then use their understanding of basic machining practices to whittle away the material of the model to create the part's contours. There are hundreds of ways to model a part, but treating your design as if you (the designer) were actually machining it yourself within the computer is the best way to avoid costly pitfalls. Before deciding on an approach, here are a few basic machining considerations.

Tool length to diameter ratio is all important. Machine shops often receive drawings that require deep pockets with small radii on internal corners, or worse yet, no radius at all. Keep in mind that milling is done with round tools called end mills, or milling cutters. These tools will be working for the most part on a plane perpendicular to the feature face. As a rule, the deeper the pocket, the larger the cutter diameter needed. Smaller radii can be produced, and even square internal corners, but they require longer machining times, or alternate forms of machining such as broaching or EDM, which are both time consuming and expensive. Keep in mind that a standard off-the-shelf end mill has a length-to-diameter ratio of 2 to 1. Although many cutters exceed this ratio, there are good reasons for maintaining this standard. Small internal corner radii necessitate using small cutters with the associated risks of tools breaking, longer machining time, and higher costs.

Drafts, angles, and undercuts are also critical. There are many standard angle cutters on the market, but it's important to consider draft angles. Whenever possible, try to design around readily available cutters. There are several great online and hard-copy catalogs available for referencing available tooling. Should you need to design an angle on a feature such as a square or rectangular pocket, remember that the radius in each corner increases the higher it goes up the sides when using angle cutters. If your design requires constant corner radii, it will require more-expensive 3D machining.

Avoid undercuts (any angle that creates a pocket or slot with an opening that is larger at the bottom than the top creates an undercut). Sometimes this is referred to as a "reverse draft." Reverse tapers or drafts on blind pockets cost more to produce than straight wall or conventional drafts. I have received drawings specifying reverse blind undercuts and I have no idea why this feature would be of any use other than possibly as a potting feature. (Open-ended dovetail slots are considered basic machining and are not included in this example.)

For rectangular machined pockets, **consider the requirement for radii in each internal corner**. When a pocket requires bottom contours, engineers must also take into account that the pocket will require 3D machining and the use of ball end mills to create that contour. Engineers must allow for radii in all inside corners as well as at the intersections of the bottom and side walls.

Yes or no to chamfers? Every feature added to a part also adds to the cost. It is less expensive to fillet a corner where two planes are perpendicular. It is more expensive to fillet a face where the two planes contain contours or are not perpendicular, such as when an angular face and a perpendicular edge intersect. Be prepared to pay for the

A well-designed pocket feature (top) is a straight side pocket with corner radii that uses conventional machining and is the easiest to produce. Good: Tapered pocket created with tapered cutter (middle) does not require bottom radii but corner radius varies. Costliest: Tapered pocket with equal radii corners and bottom (bottom) requires 3D machining.

3D machining operations that will be required to create those types of irregular features. In the case of intersecting holes, I often see heavy internal fillets called for on the edges where these holes intersect. Often, the only way this feature can be accommodated is by some sort of hand finishing. Spec such features in only if they are absolutely necessary.

Don't forget how specs of basic features like drilled holes affect cost. Deep holes with small diameters, extreme straightness requirements, and off-center intersections with other holes or features tend to be more costly. It is hard to say just when a hole becomes too deep for its diameter to make regular drilling impractical. Much depends on the nature of the material being drilled and other factors.

How will the edge profile be treated? Will it require a fillet or chamfer, or just be left sharp? Avoid unnecessary edge breaks. While designing, it is easy for a solid modeler to get carried away with component details. Machinists often see the addition of small chamfers on parts where the designer is attempting to show a requirement for edge breaks, sometimes referred to as eased edges. Simply put, this means "no sharp edges" and in the machine shop, this is understood.

In this case, it is better for designers to omit the part feature and instead provide a notation on the drawing detailing the requirement. Most machine shops use some form of CAM program to create parts directly from your solid model. Sometimes, the programmer finds it difficult to select the correct faces or edges when "small" chamfers or fillet features are included on models. Larger fillets and chamfers do not present a problem.

The upshot?

In short, designers should keep these considerations in mind:

- Can the feature in this model be machined using conventional methods and tooling?
- Can existing tools make this part or feature, or is custom tooling required?
- Are there features on this part that make it convenient to hold during the machining process or will special fixturing be needed?
- Can I eliminate unnecessary fillets and chamfers to make the part less expensive?
- Have I added unnecessary draft angles?
- Have I avoided designing internal reverse angles or drafts in blind pockets?
- Did I remember to avoid sharp inside corners where ever possible?

Whenever you have a question regarding the feasibility of your design from a machining standpoint, call on your local machine shop. Most will be happy to make suggestions and give advice on practical machining matters. Often, just a few simple tweaks to a design can slash costs for your company. **MD**