Think like a machinist when creating solid models

Article by Milton Florest
President
Tooling Research Inc.
81 Diamond St. Walpole, MA 02081
Website www.tooling-research.com
508 668 1950

Since the introduction of Solid Modeling in the early 90's product designs have become more complex than ever before. Up until then solid modeling software was extremely expensive and only the largest companies could afford it. The software was difficult to operate and the learning curve to become proficient was often a deterrent to making the switch from either the drafting board or the 2D software programs available at the time. Since then, solid modeling software has become easier to use and less expensive to own. This has been both a boon to the machining industry on one hand and a boondoggle on the other.

Most designers and engineers learn the basics of solid modeling in colleges or technical schools, and are launched into the business world to begin designing machinery and components with just the knowledge required to manipulate the chosen software. Solid modeling programs such as Solid Works, and AutoCad Inventor have become so user friendly that most students are up and running within a few days but the concept of "effective" designing is usually a learn as you go process.

The flexibility of today's modeling software allows the creation of some of the most unique designs ever conceived and this has had both positive and negative consequences. First let's examine the good things that have come from the use of these solid modeling programs.

Efficiencies of design:

Because of the ease with which a model can be altered on a computer it is a simple matter to change the designs as you go. As you review your model you can make these changes quickly either for aesthetic considerations or functionality. For those readers old enough to remember drawing boards, you know what a design change used to entail.

Efficiencies of assembly:

Assemblies can be created on the computer just as if the actual product was being assembled on the bench, often providing insight into possible interference or other problematic areas before committing to production.

Ergonomically correct shapes:

Solid modeling software today includes the ability to create free form shapes, and to manipulate those shapes with relative ease, so the designer is free to create anything his or her mind can imagine., with very few restrictions.

Efficient use of materials:

The power of Solid modeling programs such as Solid Works or Inventor enables the designer to create components using built in FEA (finite element analysis) that take into consideration material types, strengths and profiles in order to create light weight yet stable products

Upfront analysis of fit and function:

Solid modeling software provides the user with the ability to create assemblies using the components that will go into them so that, with few exceptions, the designer can be confident that the final product will go together and work in the way that it was intended. These are just a few of the many pluses derived from solid modeling.

One big negative

How will this product be manufactured?

Many designers and engineers never consider the manufacturing processes required to build the component they have just created. 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?

As an engineer with over 15 years of Solid Works experience and an extensive machine shop background derived from over 50 years in the business my company confronts just such problems every day. I have made a business of reviewing the designs of others and suggesting methods to make these designs compatible with general machining practices.

One of the questions machine shops usually have to answer is: How will I make this part? This question arises because designers have not considered the machining methods their designs require. This is often due to a lack of exposure to the machining environment and is no fault of the designer.

Some of you will say that your designs are meant to be molded rather than machined so this article doesn't apply to you. However, a stable working prototype is often required before committing to production. Not all models can be satisfactorily duplicated using stereo lithography or other 3D printing methods. If the component is going to be used in its actual environment a machine shop may be asked to create a working part from the Solid models the customer provides. If the newly designed item is either one of a kind, or of a small quantity then molding may not be an option.

When designing any product that will require machining it is important that the designer understands the basics of machining in order to create a component that can be created efficiently. The designer must try to think like a machinist and will require a basic knowledge of machine shop practices in order to do this.

Designing for machinability

First concern is availability of stock material sizes:

Very often when researching materials the size you need will show up on a vendors website or catalog but will not be available when you try to order it. Always try to design around a standard material size in the coarser fractional dimensions such as 1/8” ¼” ½” 5/8” etc. Try and avoid the 1/16ths 7/16th 9/16 ths sizes because most mills either don’t stock it, or it requires a special mill run. This rule doesn’t apply so much to sheet stock or round bar as much as it does to rectangular bar and plate. If your designs require metric dimensions as many medical products do, consider the fact that it is more difficult to obtain stock metric material than it is to find inch material in the USA. It is available but is usually more expensive and not every supplier carries it.

Whenever possible one should try to design long, thin parts around more readily available stock sizes to avoid having to machine long surfaces at all. Machining material off of one face usually causes the material to distort or bow, so the machinist is often forced to remove material from opposing sides equally to bring the material back into straightness. This is a time consuming requirement that often takes several operations to make the material flat again. The full gamut of material selection cannot be covered in this article, but as a rule, larger cross sections of material that require machining to thin profiles are going to distort at least somewhat.

When a machinist reviews your drawing he or she will be evaluating features, the steps it will take to produce that part with the least setups, and work holding will be a major consideration. Whenever possible design your parts with at least two opposing parallel flat surfaces or a truly cylindrical surface somewhere on the part so it can be gripped by conventional vises and tooling, otherwise custom fixturing or additional anchor material (extra material to provide holding method) will be required, This raises cost of manufacturing significantly, especially on lower volume jobs.

The design itself

By this time you have already decided what the function of your part will be, but very often you may not know what it will look like. Most designs evolve around the function and how it will interact with other mating parts, so you will likely begin by choosing a basic shape, it will likely have holes, slots, steps etc.

You could visualize the part as a blank of sufficiently sized raw material, then using your knowledge of basic machining practices begin to whittle away the material of your model to create the contours. Or you could start by building up your features one on top of the other like building blocks. Either way works. But t here are some things that you will want to consider.

Tool length to diameter ratio is important

Machine shops often receive drawings that require deep pockets with very 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 will have to be to create it. 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. In other words the length will normally be twice as long as the diameter. Although there are many cutters available that exceed this ratio there are reasons for maintaining this standard.

Small internal corner radius = small cutters = risk of tool breakage = longer machining time = higher costs.

Drafts angles and undercuts

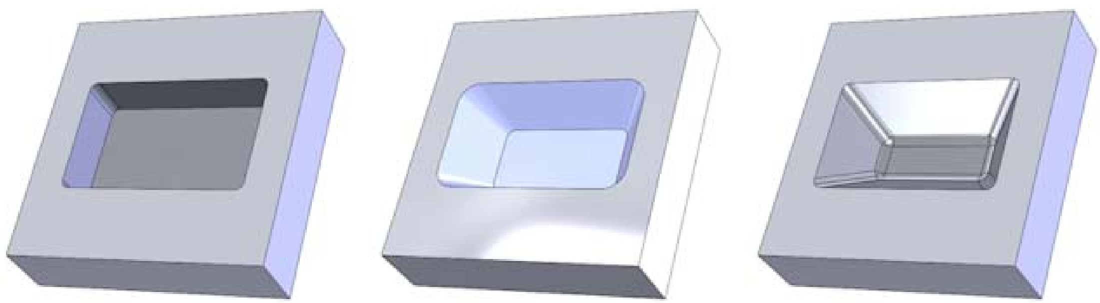
There are many standard angle cutters on the market, but draft angles should be considered carefully when designing parts. Whenever possible try to design around readily available cutters. There are several great online and hard copy catalogs available for referencing the various tooling that is available. If you need to design an angle on a feature such as a square or rectangular pocket, remember that the radius in each corner will increase the higher it goes up the sides when using tapered cutters. If you require constant corner radii the feature will require more expensive 3D machining.

Avoid undercuts on internal pockets whenever possible. An undercut is an angle that creates a pocket or slot with an opening that is larger at the bottom than the top. Sometimes this is referred to as a reverse draft. Reverse tapers or drafts on blind pockets cost more to produce than straight walls. I have received drawings requiring these reverse undercuts but they are rare and I have no idea how this type of feature would be useful. Note! Dovetail slots etc are considered basic machining and are not included in this example.

When designing rectangular machined pockets, the engineer or designer must automatically consider the requirement for radii in each internal corner.

If there are any bottom contours required in these pockets then the engineer must also take into account that the pocket will require 3D machining and the use of ball end mills to create that contour and must also allow for radii in all inside corners as well as the intersections of the bottom and side walls.

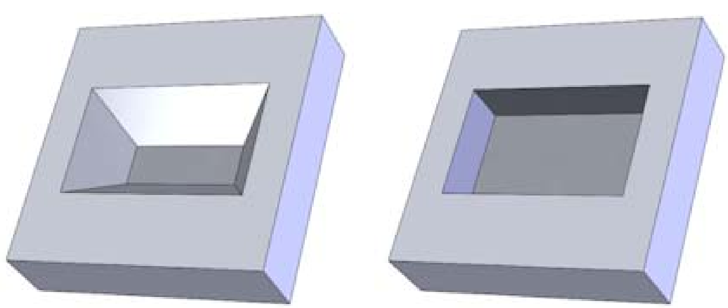
EXAMPLES OF WELL DESIGNED POCKET FEATURES:



From left to right:

Best:
Straight side pocket with corner radii uses conventional machining and is the easiest to produce
Good:
Tapered pocket created with tapered cutter does not require bottom radii but corner radius varies.
Costliest:
Tapered pocket with equal radii corners and bottom requires 3D machining.

EXAMPLES OF FEATURES REQUIRING EDM OR BROACHING:



From left to right:

Tapered pocket with no radii requires EDM to produce

Straight side pocket with no corner radii requires EDM or broaching to produce.

When designing in any 3D modeling program, it is easy to get carried away with the simplicity of creating parts containing chamfers and radii on every edge, corner and face. But think about whether or not they are really necessary. Every feature added to a part also adds to the cost. It is less expensive to fillet a corner where two planes meet on a perpendicular plane. 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 3D machining operations that will be required to create those types of irregular features. In the case of intersecting holes we quite 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. Only spec them in if you absolutely can not do without them.

Basic features:

When creating basic features such as drilled holes, depth and hole straightness will impact cost. Other considerations are very deep holes with small diameters, extreme straightness requirements, and off-center intersections with other holes or features. 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 contributing factors such as those mentioned above.

Irregular profiles on components are no longer the problem they used to be before the advent of CNC machines but there are still some things to consider when designing parts with such shapes.

One of the most important considerations for designing irregularly shaped parts is how the edge profile will be treated. Will it require a fillet or chamfer, or will it just be left sharp?

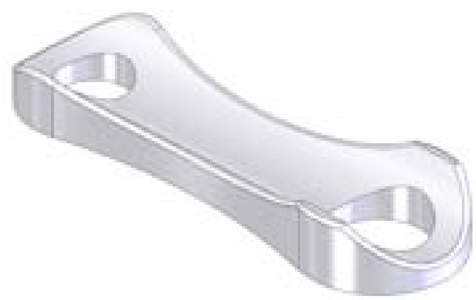
Examples of outside irregular profiles with a chamfer.

These three examples show an irregular profile with a chamfered edge.

First example shows the most practical method for designing with machining in mind. By increasing the corner radius of the profile at the inside corner it becomes a simple matter to machine it using an angle cutter.

Middle shows a good design but will require 3D machining in order to create the internal corner blends. The far right example is the least practical design. Note all of the sharp internal corners on the profile and the chamfer!

Two examples depicting a filleted edge on an irregular profile.



Example on the left shows a fillet that is the least expensive to produce requiring only a corner rounding mill.

Example on the right requires 3D machining with a ball end mill and several passes and will be the most costly to produce.

Avoid adding unnecessary edge breaks.

It is easy for the solid modeler to get carried away with details of a component while in the design process. One of the things machinists often see is the addition of very small chamfers on parts where the engineer or 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 world these are very often hand filed.

In this case, it is better if the designer omits the feature from the part entirely and instead provides a notation on the drawing itself detailing this requirement. This is because most machine shops use some form of CAM (computer aided Manufacturing) program to create your parts and very often picking the correct face or edges can be difficult when very small chamfer or fillet features are included on the model. Often times the program will use a face feature to create a perimeter which means that the part could be machined right to that feature size making it smaller by the amount of the chamfer or fillet callout.

In short the designer should be considering these things.

- Can the feature in this model be machined using conventional methods and tooling?
- Do the tools required to make this part or feature exist, or are they custom?

- Are there features on this part that make it easy to hold during the machining process, or will special fixturing or anchor material be needed?
- Can I eliminate unnecessary fillets chamfers etc in order to make this part less expensive, or do I need to have them?
- 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?

This article is not meant to be a primer in the art of machining as much as it is meant to point out just a few of the problems faced by the machinist. Hopefully this will provide some insight into ways to make your designs more economical to manufacture.

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. Ultimately you, the engineer or designer, will be the one to determine what your final part will look like. Very often a few simple tweaks of a design can save thousands of dollars for you and your company.

Milton Florest
President
Tooling Research Inc.
81 Diamond St.
Walpole, MA 02081
508 668 1950
Website www.tooling-research.com