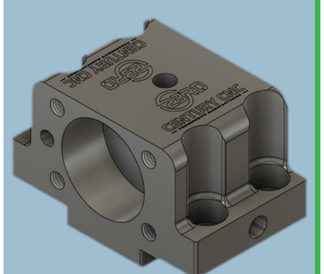
- Modeling Chamfers and Fillets -

Design for Manufacturing Advice

Issue #2— Resource Library Date: 12/31/2023

Special points of interest:

- Choosing the right chamfers and fillets can save part cost
- Learn a few rules of thumb to remember while adding fillets to your deigns
- Try some new tricks to integrate deburring into your design at no extra cost



This part was designed to require virtually zero manual deburring and use all standard tools.

Written by:

Jon Bancroft

Design For Manufacture—DFM

When creating a design engineers should consider the implications their choices have on the manufacturing process. These choices all directly impact the price, lead time, and quality of the final part they will receive from a machine shop.

A common source of frustration for both engineering and manufacturing are the choices made about fillets, chamfers, corner rounds, and deburring methods.

Engineering must remember that "round parts" are typically made on a lathe, and thus should be designed around a common axis. And "prismatic or rectangular" parts are made on a milling machine, using rotating cutting tools which cannot produce square internal corners and are subject to the laws of physics, regarding deflection and chatter harmonics.

Considering the commonly available cutting tools for either turning or milling will help dictate the economics of an engineered design. For example, chamfering tools are mostly available in "common" angles such as 30, 45, or 60 degrees. Likewise turning tools rarely ever have "sharp" nose tips, so fillets are almost mandatory on all turned shoulders.

In milling, engineers must keep in mind that it is the ratio of Length to Diameter of a milling cutter that contributes the most to the cost of certain features like the depth of a pocket relative to the fillets used in it's corner vertices. For example, a square pocket cut 1" deep with corner radii of R1/4" would have a L:D ratio of 1:2 since that pocket would need to be machined with a maximum of a 1/2" diameter end mill tool.

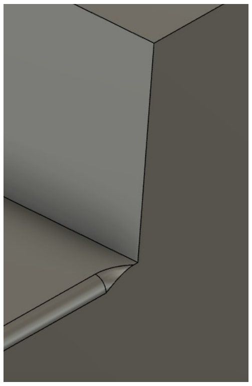
When to CAD Model Chamfers

Engineers should resist the urge to CAD model every chamfer into a design. While not a given, typically this adds artificial levels of precision to the chamfers, since they are usually toleranced on the drawing blueprint.

CAD modeled "edge breaks" usually add cost to designs since we must ensure every edge precisely matches the tolerance given. Instead add a drawing note specifying a wide tolerance for size and the ability to substitute angles or radii as edge breaks.

Avoid chamfering the tops of tapped holes unless it is for a specific engineered purpose. We typically chamfer threaded holes with a 30 or 45 degree edge at 0.01-0.02" width, depending on the relative hole size.

We do recommend only modeling chamfers which have specific size requirements or that have odd angles.



Notice that corner rounds require a lead out sweep when approaching vertical walls.

Corner Rounding Fillets

Corner rounds are fillets on the outer edges of a shape. Such as a square block which has rounded outer corners like a playing die.

Corner rounds provide an excellent feel of quality to a finished part, but potentially introduce a lot of extra cost. And they have unforeseen tradeoffs versus chamfers.

Remember that a chamfering tool can produce any size of chamfer (up to the radius of the tool), but a corner rounding tool can only produce a single size radius. Also, a chamfer tool can cut almost right up to a vertical wall transition. Whereas a corner rounding tool usually needs to stop shy of a wall to avoid a collision, thus requiring a manual blending step.

Corner rounding is very sensitive to the flatness, tolerance, and orientation of a part. For example, if a part is even a few thousandths of an inch too thick or too wide, then the corner radius will leave a "step" mark at the top or side tangency points.

Add a note to your drawing permitting small floor radii, <0.010". Many tools are sold with these small radii standard instead of being sharp corner end mills.

Interior Fillets—Potential Money-Pits

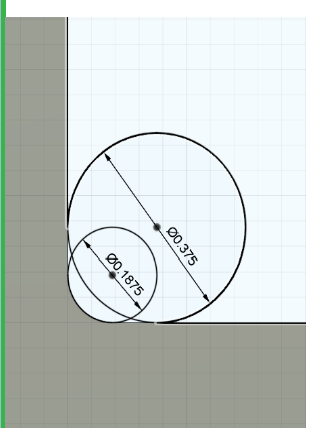
When designing parts which have steps, pockets, cavities, etc. please follow some rules of thumb to prevent your design from becoming overpriced.

End Mill tools are typically sold in Length to Diameter ratios of 2xD to 3xD. So a 1/2" end mill usually can only reach 1", 1.25", or maybe 1.5" deep. The reason is simple, every material has a modulus of elasticity, even tungsten carbide cutting tools. Meaning they must always deflect a certain percentage based on the cutting pressures (which are immense in many cases). This deflection is the source of inaccuracies, surface finish issues, and even tool breakage.

When designing a machined plate, with a thickness of 1/2" and a thru-window cut into the plate, we would advise remembering the 2-3 times diameter ratio, and designing the internal corners of the window to have a minimum of a R3/32" fillet so we may use a minimum of a 3/16" diameter end mill ~3xD. But to make the part cheaper we recommend using as large of a fillet as your design allows so we can use a larger end mill which will deflect less. Less deflection means we will have less issues maintaining tolerances.

The other consideration for choosing more "stubby" end mills is to reduce the likelihood of encountering a chatter problem. While an end mill may be available in 3x, 4x, 8x, even >10x for their L:D ratio, the larger this ratio becomes the more susceptible a tool is to harmonic oscillation, i.e. chatter. This chatter makes parts less attractive, and may be compounded by the increased deflection which results in a part that may not meet tolerances, and is aesthetically unappealing.

We have lots of tricks to help you achieve tight tolerance, long L:D ratio fillets, but they all add cost. Evaluate what you "need" for fillet radii and if it exceeds the 3xD ratio then reach out for our advice.



If radii can be increased then it permits us to use a larger—more reliable—cutting tool.

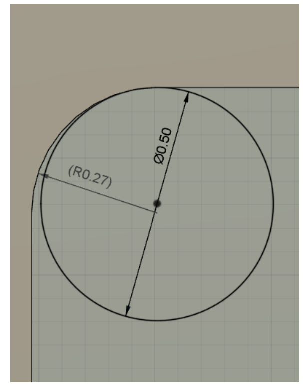
Tips for Internal Fillets

Regardless of what your fillet Length to Diameter ratio ends up being there are a few suggestions we have to improve your final part.

If you imagine yourself as an end mill, cutting around the inside of a square pocket. While you are milling the side walls you may only have a "cut engagement" of a few degrees up to maybe 30-60 degrees of your circumference. But when you enter into a corner fillet that number will actually rise to and temporally exceed a 90 degree cutter engagement. This is usually when chatter and the most deflection happens.

There is a simple solution. Don't use common fractional or metric values for your fillets. Instead use "off numbers". End mills only come in "normal" fractional or metric values. So if a 1/2" tool tries to cut a R1/4" fillet it won't be as nice of a finish or accurate of a radius as if the CAD size was instead R0.27". The extra few thousandths of radius relieve the cutting pressures immensely.

Remember that a 1/2" end mill physically cannot cut a fillet any less than R1/4".



Notice the modeled fillet is slightly larger than a standard milling tool.

Options for Square Corners

The solutions depend greatly on the actual part geometry and tolerances. And sometimes if you are open to a design change then we are able to save a lot of cost.

Consider a disc with a square pocket that does not go through the disc. We have several ways to accomplish that.

One option is to RAM EDM "burn" the square using a graphite electrode to precisely erode the conductive part material away. Another EDM option would be to instead make the disc as two pieces and Wire EDM one with the square hole through it, then somehow attach a flat disc on one side to seal off the square hole.

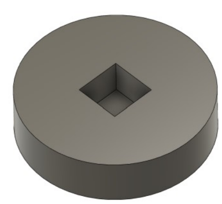
A more traditional way to cut a blind shape or a through shape is to use broaching. Where similar to RAM EDM we have Rotary Broaching which "wiggles" a broaching tool to shave out square corners on a previously pilot drilled hole. Or similar to the Wire EDM method we could use a Push Broach to shear out square corners through a pilot drilled plate.

We could also simply "construct" the shape from other small pieces and weld, braze, solder, pin, bolt, or otherwise join the pieces together

Some square corners can be made by mixing two machines together. Such as a part which was milled using an end mill that leaves material in a corner, but is then transferred onto a surface grinder where a grinding wheel may be able to pick out that remaining material and leave it square.

Please contact us for ideas on the best option to achieve your square corners. We're happy to offer this DFM advice at anytime in the design process.

Most cutting tools in the USA are imperial. Keep that in mind when designing metric fillet radii.



Square holes can be expensive. But through some careful thought and negotiation we can probably come to an effective solution.

23rd Century CNC

9083 State Route 7
Kinsman, Ohio 44428

330-282-8111
Jon@23rdCenturyCNC.com

Find us online!

www.23rdCenturyCNC.com

What "Progressive Manufacturing" means to us...

Progressive Manufacturing is our business's tagline and motto because it reflects our mindset.

We do not possess the most advanced equipment or tools. We do not use the most expensive software. But we do strive to make continual improvements in our processes and our people.

For example :

- *We continually invest in new inspection equipment*
- *When we add equipment, we look to add new hardware and software capabilities rather than just to replace existing ones*
- *Employees are given continual access to training programs, and are cross trained on equipment*
- *New tools are always being trialed to improve process reliability and part quality*

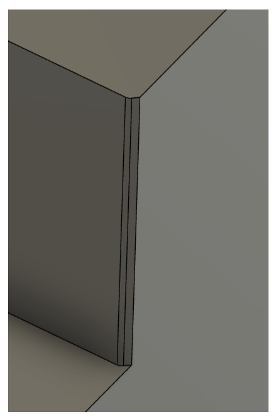
Please do reach out with any questions or comments regarding this article, we hope you found it helpful and entertaining!

Integrate Deburring into CAD

Here are some simple tricks you can use to save your company money by incorporating deburring into the part design!

Add a note to the drawing allowing corner rounds to be added to outer part edges. When we CNC mill around a part we can then add small R0.005" - R0.015" fillets to the corners to save us from manually deburring them.

Don't model floor fillets unless they are critical. Instead add a drawing note permitting a range of sizes. Many shops have "Bull Nose" end mills for adding floor fillets, but they may not have the exact size you modeled in CAD.



This simple chamfer & fillet trick greatly improves part quality.

If you use corner rounding on the top & bottom of your part then stick to a single radius value and check that it is commonly available. Similarly, try to stick to a single angle for chamfers to reduce tool changes in the CNC.

Lathes permit you to easily add almost any corner radius or chamfer you want for free, use these to help deburr the part.

And lastly, whenever there is a pocket wall that leads out to "air", add a small chamfer to this edge. This lets us lead the tool in at that angle and intrinsically deburr the edge. A small fillet on the inner vertex of this chamfer is a nice touch too. But don't subject these to tight tolerances. Avoid radii, since any errors in part profile will result in a witness mark at the tangency point on the start and end cuts.