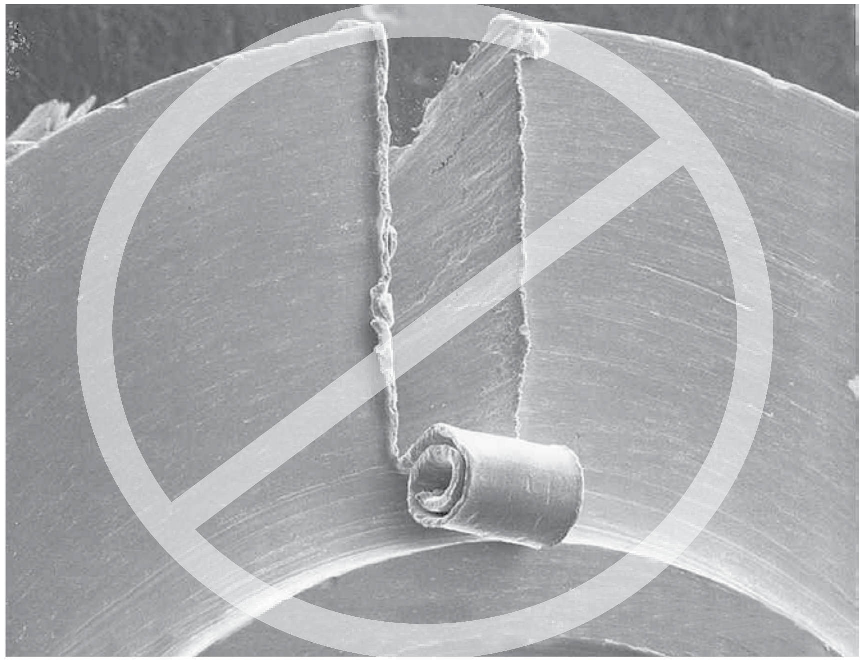
DECEMBER 2006 / VOLUME 58 / NUMBER 12

➤ BY LAROUX K. GILLESPIE



Burr Down

Machinists can reduce or prevent burrs in the metalcutting process.

As long as there has been metalcutting, there have been burrs—an unwanted side effect of the process. Barring a revolutionary change, burr formation will always be a problem. However, machinists can use several strategies to prevent burrs or reduce their size. Some burr prevention and minimization strategies can be complex, but others are relatively simple and easy to implement.

With a few exceptions, conventional machining techniques always produce burrs. Burr prevention requires both conventional and nonconventional approaches. Some nonconventional approaches are outlined in Table 1. However, even when burrs are not produced, sharp edges left as a result of the machining process generally are not acceptable and require edge treatment.

Burr Prevention

There are several common burr prevention techniques, including the following.

- Form or extrude parts instead of machining them. Extruded sections prevent burr formation at part edges because high-pressure forming does not produce chips.
- Cast part edges. Casting saves finishing costs, such as deburring. However, flash will form at die openings and generally must be removed. Casting techniques include P/M forming and other operations that use dies.
- Use form tools to produce part features on a lathe (Figure 1). Because a form tool is always in the cut, there is simply no place for a burr to be formed except at the tool's leading and trailing edges. Burrs on leading and trailing edges can sometimes be prevented if diameters are tapered. Another way to prevent lathe burrs is to generate part edges via a corner-rounding path rather than simply facing and turning. This removes any burrs that form during the process and creates chamfered or radiused edges rather than sharp ones. Instead of feeding the tool along the lathe axis and then at a right angle to generate shoulders, feed straight out and then trace a radial path on the outside of each shoulder to round the edge. It takes only 1 or 2 seconds, but prevents the need to deburr.
- Ultra-high-speed machining (cutting velocities higher than 10,000 sfm) produces no observable burrs with some metals, though it may produce microburrs.
- Selective hardening prevents plastic deformation from occurring macroscopically at part edges. Any environment that prevents plastic deformation will prevent or minimize burrs. Hardening techniques include laser, mechanical-localized, chemical and thermal.
- Prevent large exit burrs when drilling by using sacrificial backup material that is harder than the workpiece. The backup material, positioned at the bottom of the hole, pushes back on the bottom of the workpiece, which helps prevent burr formation. In this procedure, it is essential to keep the materials held tightly against each other. The corner of the drill's cutting edge must cut into the backup to effectively reduce burrs. For drills smaller than ⅛", machinists should drill 0.010" into the backup. A rule of thumb is that the larger the drill diameter, the deeper into the backup material the tool should cut. Figure 2 shows how drill exit-burr size varies when a backup material is used. In this case, the material being drilled is 303 Se stainless.

Making Small Burrs

If burr formation cannot be prevented, the goal should be to leave burrs so small that simple and rapid finishing processes can remove them and round the cut edge slightly. Any deburring process will readily remove burrs that measure 0.000010" in thickness or height.

Burr size varies according to cutting forces, workpiece material and the geometry of both the part and the cutter. While the height and thickness of most burrs are from 0.001" to 0.002", others can be quite large. A good burr minimization program should be able to prevent 90 percent of burrs that are over 0.020" long.

In most cases, a burr forms when a cutting edge passes over a part edge. When a drill produces a through-hole, it makes a burr at the hole's entrance and exit. Similarly, when an endmill is applied to make a through-slot (Figure 3), it produces eight edges—each with a different burr. It is important to know exactly which mechanism forms which burr and recognize how different machining variables affect burr properties.

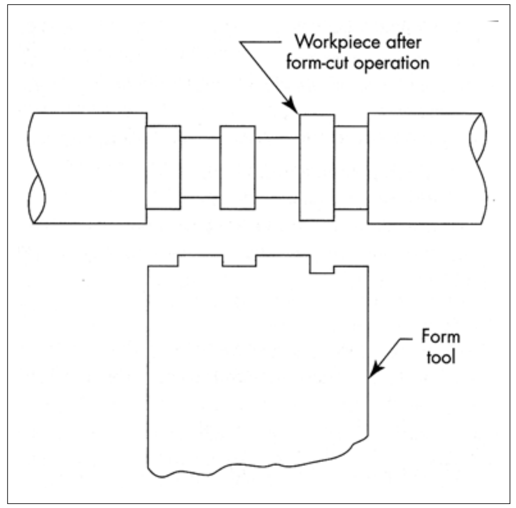


Figure 1: Since a form tool is always in the cut, there is no place for a burr to be formed except at the tool's leading and trailing edges.

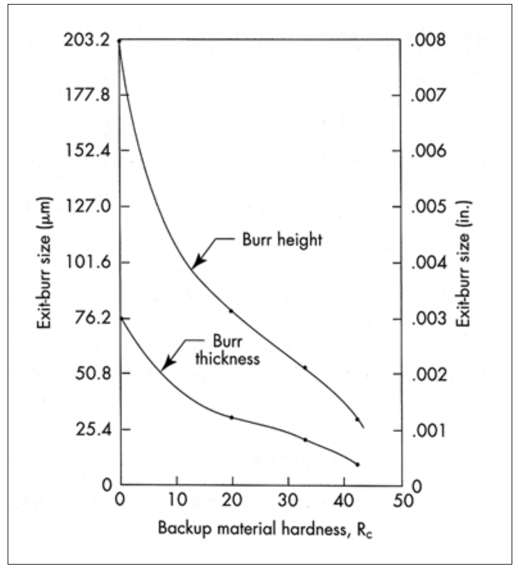


Figure 2: Impact of backup material's hardness on burr size of drilled holes.

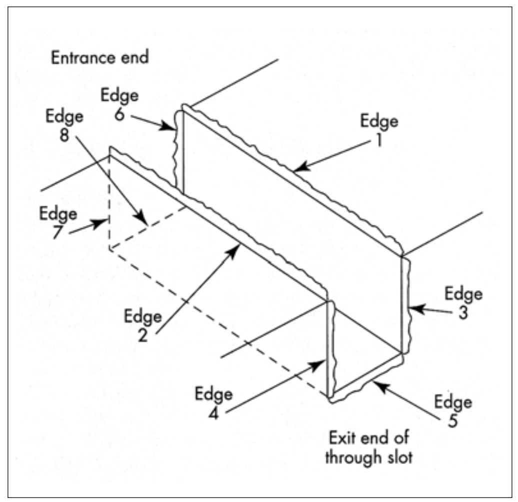


Figure 3: An endmilled through-slot has eight edges, each with a different burr.

Machinists can control burr size by reducing cutting forces as the cutter exits the part. Tactics for reducing cutting forces include using sharp tools and tools with appropriate geometries (positive rake angles, for example), preventing built-up edge and lowering feed rates at tool exit points.

To minimize burr size, feed rates must be reduced only as the tool exits the workpiece. Higher feeds are used when drilling a hole until the drill is about to break through the workpiece.

Figure 4 shows the impact on burr formation of slowing the feed per tooth. In this instance, the feed is held constant, but when the spindle speed is increased, the feed per tooth is reduced, which reduces the cutting force. This in turn reduces burr thickness. Producing too small a chip in strain hardening materials will cause the tool to rub rather than cut, producing large and hard-to-remove burrs.

Solving Part Geometry Impact

Endmills and facemills have complicated geometries that make burr minimization challenging. Consider the task of facemilling the top of an engine block. To minimize burr size, the programmer must create toolpaths that minimize tool exits over edges. Basically, end users must downmill into an edge and program the tool to exit the edge over an area that has already been machined. This directs the cutting force into the workpiece. When the cutting edge exits the workpiece over a previously machined area, there is no force to push a burr from the edge because the cutting edge is not cutting at that location.

In addition, always cut into the surface. Instead of applying a cutter that covers the full width of the workpiece and making a single pass across the entire workpiece, select a cutter with a smaller diameter, lay out a sample toolpath and look for areas where the tool is cutting over an edge. Modify the path so that when the cutter passes over the edge, it occurs over a previously milled surface. Because no chip is being created at that point, it cannot roll over to form a burr. This approach is demonstrated in the following facemilling example.

When a facemill is fed straight into a straight edge, some teeth cut as they exit the part edge. To prevent that exit cut, feed the tool into the workpiece in a circular arc so the teeth are only cutting when entering the workpiece. The tool can be fed around the workpiece perimeter continuously after it has entered the cut in this manner. Teeth will never be cutting as they exit, which solves the problem when facemilling a solid workpiece.

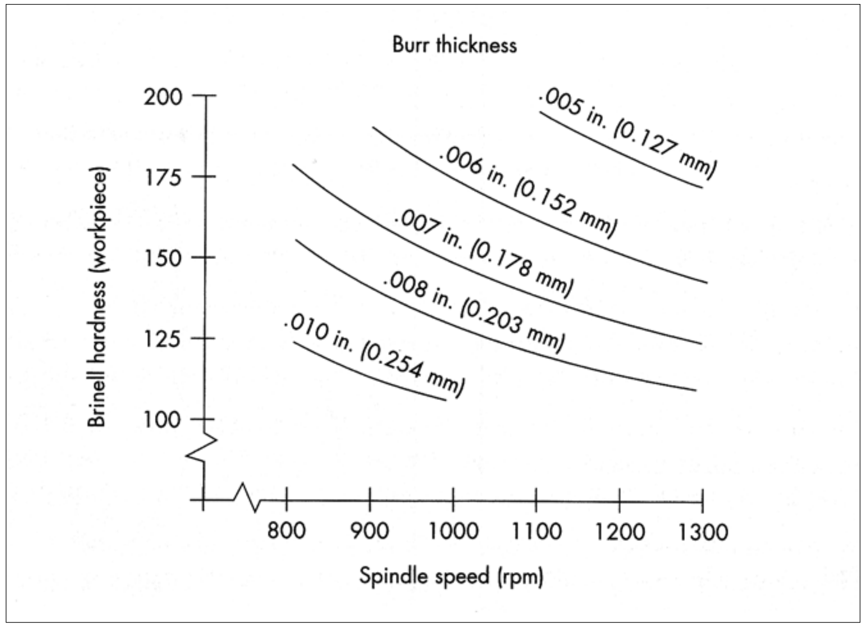


Figure 4: Effect of workpiece hardness and spindle speed on drill burr thickness. Material is AISI 1018 steel machined dry at 6 ipm with a 0.25"-dia., HSS twist drill having a 118° point and 27° helix.

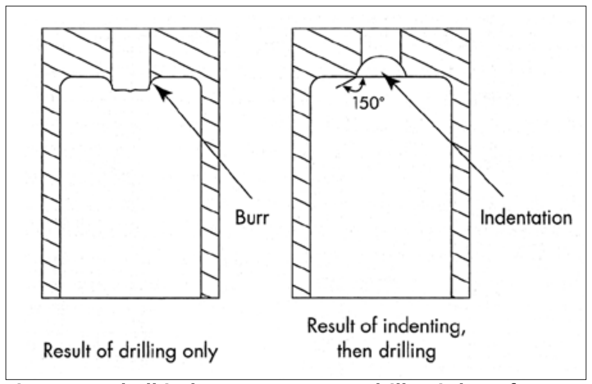


Figure 5: A ball indenter prevents a drill-exit burr from forming.

The key is to select a tool diameter smaller than the workpiece thickness and to offset the tool. To calculate the WOC for this condition, use this equation:

w = r – d, where w is WOC, r is the tool radius and d is the offset dimension.

When the tool approaches the end of a pointed edge, the tool must be moved up and out in an arc to prevent exit cutting and then returned downward.

Parts with more complex shapes, such as an engine block, require more careful consideration of cutter diameters and toolpaths. Machining some shapes can be planned manually, but many require a computer program to assess required toolpaths. While the tool can always be raised and lowered to prevent tool exit cutting, this action produces rougher surfaces.

Burr Minimization Techniques

There are several simple solutions shops can use to minimize burrs. A common problem is how to remove internal burrs caused by drilling. Figure 5 shows a deburring solution that was found when a shop drilled a primer hole in a brass ammunition cartridge case.

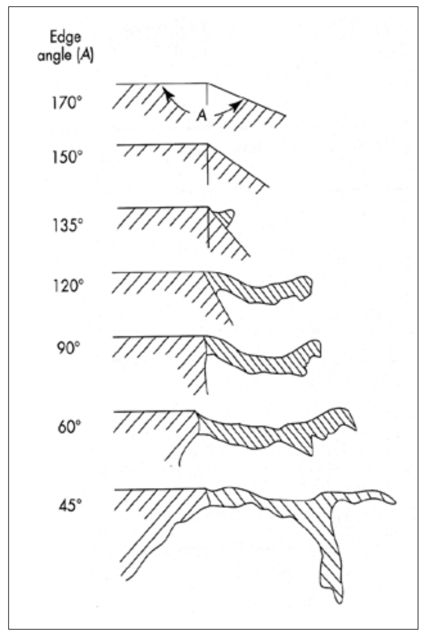


Figure 6: Cross-sectional view shows that large vertical edge angles minimize burrs. Large burrs form when machining over 30° edges, but when machining over 150° or larger edges, burrs typically do not form. Pictured are long thin burrs. When machining over an edge with larger angles, no burr protrudes.

Process	Typically makes burr or leaves molten material	Typical edge radii produced (in./mm)
Abrasive jet machining	No	0.003/0.08
Chemical machining	No	Unknown
Photochemical machining	No	0.001/0.03
Electron beam machining	Yes	Unknown
Electrochemical discharge machining	Unknown	Unknown
Electrochemical grinding	No	0.003/0.08
Electrochemical machining	No	0.001/0.03
Electrochemical honing	No	0.0005/0.013
Electrical discharge machining	Yes	Unknown
Electropolishing	No	0.001/0.03
Electrostream machining	No	0.002/0.05
Hot chlorine gas machining	No	0.002/0.05
Ion beam machining	No	0.00005/0.0013
Laser beam machining	Yes	Unknown
Plasma arc machining	Yes	Unknown
Ultrasonic machining	No	0.001/0.03
Waterjet machining	Yes	Unknown

Table 1. Burr formation in nontraditional machining processes, where burr/molten material is visible under 30× magnification.

The makers of the case used a ball-end pushrod to indent the inside of the case where the hole would later be drilled. This simple indentation prevented burrs in two ways. The indentation lowered the ductility of the brass; the lower a material's ductility, the less prone it is to burr formation. Also, the angle produced between the sides of the indentation and the diameter where the drill came through the indentation approximated drilling into a 150° chamfer. Burrs do not typically form when a cutter passes over edges having such angles (Figure 6). It's important to have cutters enter rather than exit edges that are less than 90°.

Another simple burr minimization approach is to use compression routers. The helix angle on endmills normally causes an exit burr on either the top or bottom of the part being profiled because the helix pushes material out. Compression routers (also called upcut/downcut tools) have a helix that results in cutting into the part on both the top and bottom of the part. The top of the tool has a helix in one direction while the bottom of the tool has a helix in the opposite direction.

Today's technology allows machinists to remove sharp edges and many burrs during the same part production cycle. One approach is to brush all machined surfaces with a stiff, abrasive-filled nylon brush that fits in one of the toolholder positions. The ends of the brush are programmed to be continuously positioned ⅛" below the machined surfaces so the brush ends fall into openings and crawl back out with each spindle rotation. This does not require traversing all the edges; rather, a simple back-and-forth motion over the entire surface brushes all the top edges. This process produces a small radius on these edges and also removes small as well as long, thin burrs, so many parts need no further edge finishing. △

About the Author

LaRoux K. Gillespie is a retired manufacturing engineer and quality-assurance manager. He is the author of 10 books on deburring and nearly 200 technical reports and articles on precision machining. For more information on principles discussed in this article, consult his textbook, Deburring and Edge Finishing Handbook. Contact Gillespie at (816) 516-1144 or by email at laroux1@myvine.com.