Premium Finishes When Routing Acrylic

Routing of acrylic has become one the most popular methods of plastic fabrication in the sign and point-of-purchase industries. As the accuracy of the CNC routers improves and approaches that of CNC mills additional gains are being made into the machine building, medical devices, and valve industries. A common quality across all of these industry segments is the requirement for a premium finished edge on acrylic without the need for post-routing finishing operations. Four factors in the routing operation typically will affect the quality of the cut edge: Tooling, Programming, Machine Condition, and Fixturing. If any one of these factors is not optimized, it will be extremely difficult to maintain a consistent, high quality edge finish. Two of the factors - Tooling and Programming - will be covered in this article and the remaining two in a future article.

Tooling

Tooling is a broad topic, but there are some simple guidelines that can be used to decrease the likelihood of failure during cutter selection. The first selection criterion for router tooling is typically diameter. While is a common request for tooling diameters to be in the 1/8” to ¼” range, designing the parts, fixtures, and programs for 3/8” to ½” tooling can dramatically improve surface finishes and consistency from job to job. A typical result from increasing cutting edge diameter from ¼” to 3/8” can be a drop in surface finish from 40-60 RMS to 18-25 RMS. This is usually accompanied by an increased feed rate as well as better chip extraction. The stability and flute depth offered by larger diameter cutting tools cannot be overestimated. That said, it is important to note that there is typically only a marginal benefit when increasing cutting edge diameters over ½” as long as the depths of cut are not exceeding 2”. The price-to-performance ratio cannot typically support the use of 5/8” to 1” diameter cutters in sheet stock.

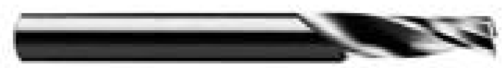

FIGURE 1

After diameter, cutter configuration is frequently the second selection criterion for tooling. As a rule of thumb, the smaller the diameter of cutter that is being used, the more likely it is that a spiral configuration will yield the optimum edge finish. While straight flutes typically have good success in larger diameters (3/8” and above), it is the spirals that excel when cutting with small diameter tooling. Single edge spiral O-flutes (see Figure 1) typically give the best edge finishes in ¼” and smaller tooling. When moving into the larger diameters, typically low helix multi-fluted tools will yield the best results with some variations depending on the manufacturing method of the acrylic (i.e. cast or extruded) and any fillers that may have been used. It is also in these larger diameters that the double edge straights can typically begin to perform well. Both V-flute and O-flute configurations have been shown to work well through testing and industry use. (see Figures 2 & 3)

FIGURE 2

FIGURE 3

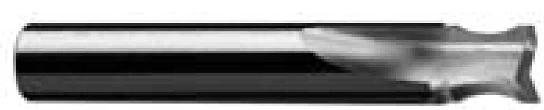

FIGURE 4

A final note on tooling is to mention the existence of numerous specialty tools available in the market today. There are products that can be used to provide a radiused edge on parts (Figure 4), to rout a finished edge and apply a top chamfer at the same time (Figure 5), or to create a smooth bottom surface during pocketing without the swirling effects of standard router bit points (Figure 6). These cutters either solve problems that have been recurrent in the industry or allow fabricators to eliminate tool change cycle times and/or utilize machines without tool changers.

Programming

Selecting the right cutting parameters and cutting methods is extremely important when edge finish is the primary driving factor of an operation. Every material and cutter combination has a “sweet spot” in the cutting parameters and slight deviations in any direction can cause an unacceptable decrease in surface finish quality.

Feeds and speeds are typically the best known variables as far as cutting parameters and they are extremely sensitive to minor variations. Unlike many other commonly routed materials, plastics, and particularly acrylics, have an extremely narrow chip load range that can be used to produce an optimal finish. (see Figure 7). Each cutter (based on configuration and diameter) will have a different optimum chip load for each material type. As a rule of thumb, the following feed rates are good starting points if the goal is optimum edge finish. A constant spindle speed of 18,000RPM and a depth of cut equal to the cutter diameter is assumed.

?? 1/8” Diameter Tooling: 75-100 ipm
?? ¼” Diameter Tooling: 100-200ipm
?? 3/8” Diameter Tooling: 125ipm to 250ipm
?? ½” Diameter Tooling: 150ipm to 300ipm

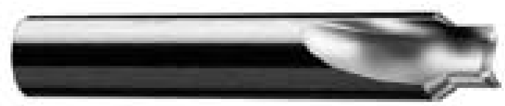

FIGURE 5

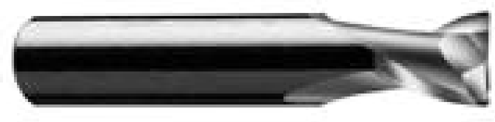

FIGURE 6

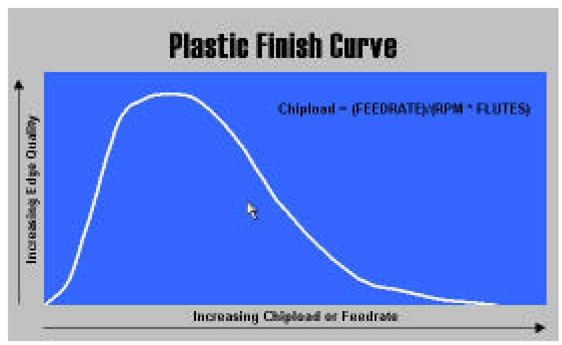

FIGURE 7

As long as the router bit is not having stability problems and the workpiece is well fixtured, most of these feedrates can be increased by simply increasing the spindle RPM. With the newer generation spindles typically having maximum speeds of 21,000 to 24,0000 RPM, most of these feed rates will have plenty of room for improvement. The only consideration to remember is that, unlike other materials, increased spindle speeds must be accompanied by an increase in feed rate to remain within that “sweet spot” on the chipload. Excessive spindle speeds will typically melt the plastic or cause a wiping or smearing action on the finished edge that reduces the quality of the surface finish.

After feeds and speeds have been dialed in (usually through manufacture recommendations, as trial-and-error is a time and material intensive process), the next step is to choose the cutting method. Both conventional and climb cutting have their place, but here is the rule of thumb: Larger diameters almost always perform better in a conventional cut mode. Smaller diameters are entirely material dependent and must be tested to determine the best method.

Other programming parameters that should be considered are finish passes, entry points, and depths of cut. Typically smaller diameters are the only tools that require finish passes for optimum edge finishes. There is usually only a marginal gain in finish quality for acrylics when 3/8” and ½” tooling is used in a two pass system. The biggest problem that seems to surface in the industry regarding finish passes is the amount of material to remove. Many CNC operators and/or programmers have previous experience in the metal working industry and that can be a detriment when attempting to use similar cutting parameters in acrylic. A typical finish pass in ferrous and non-ferrous metals can be as little as .004”-.005”. When this amount of material is remove in acrylic, it frequently will compress and cause the cutter to actually skip across the surface. This is due mainly to the high rake angles employed in plastic tooling and the aggressiveness of their cutting action. Without at least .015”-.030” of material to remove, most acrylic router bits will not have enough material to bite into and will actually show a deteriorated finished edge over the initial roughing cut.

Entry points can also be a troublesome issue during programming. While most acrylics do not exhibit the chip wrap problem prevalent in other softer plastics, their tendency to craze can sometimes inhibit their ability to be machined at high speeds during cutter entry. The most common method is to slow the feed rates down to compensate for this problem, but a ramped entry can work equally well and will not show the entry melt that is associated with direct plunging by router bits. Another issue on cutter entry is that because router bits do not have a centering point similar to drills (this is to allow flat bottom cutting), they will have a tendency to "walk" during the plunge. The visible result is a larger entry point then the routed channel that will follow. The result is somewhat similar to what a keyhole slot looks like with a large diameter hole followed by a smaller slot width. Once again ramped entry can reduce this effect, but it is easier to enter the cut by plunging into a scrap area and moving to the final cut path in a lateral direction.

As a final parameter to be considered, depths of cut are critical to ensuring consistent edge finishes and non-broken tooling. A good rule of thumb is a maximum of twice the cutter diameter per depth of cut. A favorite programming method is to use multiple depths of cut when cutter breakage is an issue and the to take a final clean-up pass of .015" for the entire material thickness. (see Figure 8) This gives a premium edge finish while preventing broken tools in the smaller diameters. It is a common concern that taking finish passes in small parts will cause the parts to move once they have been cut away from the scrap, particularly in intricate parts like letters. The best solution is to use the multiple depth pass/single finish pass method described above, but to not cut through the paper masking on the bottom side of parts. This allows the vacuum to continue holding the parts, while the .015" finish pass will not typically tear the parts off of the masking.

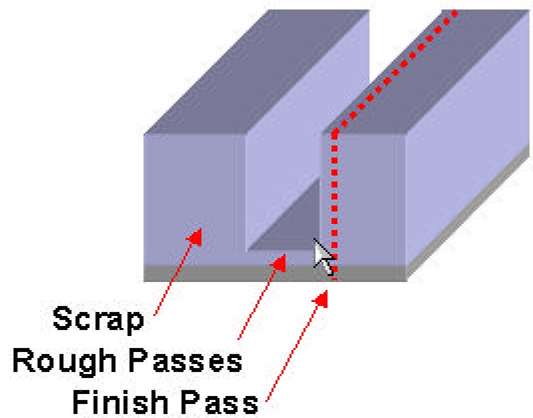

FIGURE 8

While much of the information presented above was in the form of *Rules of Thumb*, it is essential that only the proper cutting tools and cutting parameters be used when machining acrylics. The information presented here is a broad overview of information regarding routing of acrylics. The best source of detailed information is typically the tooling manufacturers' recommendations either through published sources or their Technical Services department.