JULY 2004 / VOLUME 56 / NUMBER 7

▶ BY RON FIELD, MILLSTAR

FINISHING FAST

Obtaining a fine surface finish when high-speed machining.

Each shop determines the surface finish required for a particular mold cavity or core. Some shops are willing to put more time and effort into the machining center to impart the ultimate finish. Others don't require as fine a finish or are not willing to add the machine time needed to get better results.

For shops that require optimal finishes, a number of things can be done to achieve the desired result while utilizing high-speed machining techniques. The main areas to concentrate efforts on are CAM parameters and tolerances, machining parameters and tooling.

Millstar

CAM Parameters and Tolerances

The most important thing to remember about HSMing is that it all starts with the program. If the program from the CAM system is not produced correctly, success will be limited. If the CAM programmer pays attention to details and understands the principles of HSMing, the sky's the limit.

When choosing CAM parameters for surface finish, the chordal, or program, tolerance should be set fine enough to achieve a smooth finish. The chordal tolerance is the amount of error the toolpath is allowed to deviate from the actual part surface, as developed by the CAD system. If the chordal tolerance is set too high, facets will be seen in the surface after machining (Figure 1). If the finish looks shiny but faceted, the chordal tolerance was not tight enough to achieve the desired finish, given the higher feeedback resolution of machine tools today.

If the chordal tolerance is set too tight, it can actually slow the feed rate of the machine tool because of the larger vector changes that the control is required to process accurately.

Many shops that have seen these problems in their applications blame the machine tool for inaccuracies when typically the chordal tolerance is to blame. In die and mold finishing, the chordal tolerance can be as tight as 0.00004" (Figure 2). This can create large programs, but also requires high-resolution feedback from the machine servos to the control.

Higher-resolution feedback is important in finish milling. The closer the position and velocity of the servo system is monitored, the less opportunity it will have to drift and the more accurately the part's features will be machined, including any model errors.


Makino

Figure 1: Faceted finish.


Makino

Figure 2: Smooth finish.

Some moldmakers use as little as 25,000 feedback pulses, while the better builders use up to 1 million feedback pulses for each revolution of the servomotor and ballscrew.

For an optimal finish, the engagement onto the surface should never be done in a straight line. An arcing or helical motion should always be used when entering the surface. This creates much less shock and tool deflection and minimizes the engagement line that is often seen on surfaces after machining.

Most cavities and cores need to be machined with a combination of operations to achieve the required finish. Sidewalls and floors should typically be machined in two separate patterns. Sidewalls should be machined using Z-level machining, where the toolpath arcs onto the surface to be machined, travels around the part and then arcs off the surface at the same point as the entrance (Figure 3). The cutting tool is then lowered by the step-down amount and the operation is repeated until the desired total depth is achieved. The step-down amount is determined by the finish required and the diameter and type of cutting tool to be applied. Most CAM systems allow the programmer to decide this amount several ways. He can choose a cusp (scallop) height or simply specify a certain value.

Another, newer operation for sidewalls is a spiral Z-type cut. This operation eliminates an engagement line on the part. The cutting tool never leaves the surface and is constantly performing 3-axis machining, which maintains a constant finish and a constant tool load.

Floors and shallow surfaces can be machined in many different ways. The most popular is a simple raster (both ways) cut where the cutting tool is helically maneuvered, or arced, onto the surface in one corner of the part. The tool is then simply run back and forth across the entire surface until the entire surface has been machined. The step-over amount and chip load per tooth used determine the finish.

Another method of finishing floors and shallow surfaces is an outward pocket or spiral pattern where the tool is helically maneuvered, or arced, onto the surface in the center of the part. The toolpath should then start to pocket or spiral out from the center by a specified step-over amount to create the finish. With this option, the cutting direction is constant and the part being machined is always on the same side of the cutting tool, which gives a constant tool pressure.

Climb milling should always be used with outward pocket or spiral machining (Figure 4).

Machining Parameters and Tooling

The amount of stock to be removed for finishing depends on the diameter of the cutting tool. The typical value is no less than 1 percent and no more than 3 percent of the tool diameter. The optimum is about 2 percent of the tool diameter.

Tool geometry is also important. If the wrong tool geometry is selected, it may be impossible to get the desired result. Always select the correct geometry for the material and process being used.

The feed per tooth should not be set too high or too low. The cusp left on the surface will be large if the feed per tooth is too high. If the feed per tooth is too low, the cutting tool will rub the surface instead of producing a chip. This can cause a rough surface and significantly shorten tool life.

When finishing with a ballnose endmill, a good rule for feed per tooth is the F = P concept. The "F" stands for feed per tooth and the "P" stands for pick feed, or step-over. The object is to have the feed per tooth equal to the step-over. This concept is designed to keep the finish as uniform as possible. The step-over can be calculated based on the desired cusp height. Most CAM systems can automatically set the step-over based on the cusp height selected by the programmer. Once this value is determined, multiply the spindle speed times the number of flutes times the step-over. The result is the feed rate.

It is also important to make sure the tool extension is not too long for the particular part to be machined. Using a tool extension that is longer than required will sacrifice tool life and affect the finish. For solid-carbide tools or carbide tool shanks, it is best to use a tool extension less than 6 diameters. If parts require a longer tool extension, the spindle speed and feed rate must be lowered to allow for the increased tool deflection and prevent vibration.

The cutting tool, toolholder and tool adapter should be balanced for the spindle speed. Causes for unbalance in tool adapters include flaws or voids in the adapter material; poor finish and out-of-roundness in the machining of adapter features; eccentricity of through-holes on the rotational axis; and machining features that reduce the absolute concentricity or symmetry of the adapter. Additional causes for unbalance are of a design nature: asymmetrical features such as drive slots and setscrews on endmill holders, the notch for tool orientation in DIN V-flange adapters, and even unbalanced retention knobs. Unbalanced cut-

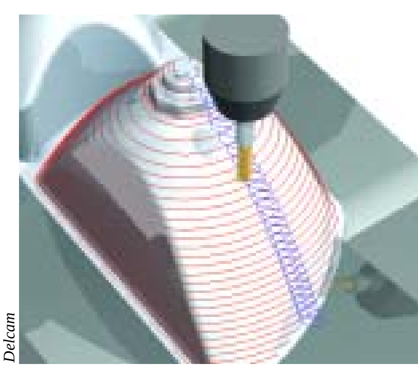
Delcam

Figure 3: Z-level machining with arcing onto and off of the surface.


Delcam

Figure 4: Spiral pocket machining.

ting tools or toolholders can also add significantly to the unbalance of the cutting-tool-and tool-adapter assembly.

An example of unbalance in tooling is Weldon screw flats on tool shanks. For effective machining results, such as improved finish, minimized vibration and extended tool life, it is important to balance the entire tool-and-adapter assembly to a sufficient balance quality (e.g., G6.3, G2.5). The higher the spindle speed, the better the quality needs to be.

For the optimal finish, coolant may be required, depending on the workpiece material. The coolant should always be directed toward the tool/workpiece interface. If the coolant is not taking the chips away from the cutting zone, chip recutting can occur. Tools wear out faster and also have a tendency to reweld the chip back onto the surface that was machined, causing a poor finish.

Many die and mold materials do not require that a liquid coolant be used, especially in the hardened state. In most cases, blown air is sufficient. If the surface that is being machined has a torn or matte finish after machining, it indicates that liquid coolant should be applied to obtain the optimal finish for the material/hardness combination. Liquid coolant is either a fluid or mist. An oil-mist system works well for finishing applications by lubricating the cut without thermally shocking the cutting tool.

It is also important to clean, clean, clean. Every aspect of the cutting tool and toolholder should be cleaned. The machine's spindle taper should be wiped clean regularly as well. Regular maintenance and cleaning help maintain consistency, which will, in turn, yield finer finishes and more reliable results.

Success in all HSMing applications requires close attention to detail. It starts at the CAD/CAM system and continues through the entire process.

About the Author

Ron Field is vice president of Millstar, Warren, Mich. For more information about the company's cutting tools, call (877) 645-5782 or visit www.millstar.com.