machining Graphite

Until recently, good information on machining graphite was not always readily available. Although the major graphite manufacturers publish machining data for their particular brands, there was very little, if any, really solid information on speeds, feeds and machining guidelines — even in Machinery's Handbook, the machinist's bible. We're not sure why this is. Perhaps graphite's intrinsic properties — its abrasiveness, hardness and relative densities — lends itself to an approximate starting point for speeds and feeds, rather than a hard and fast table.

For over forty years, Intech EDM and its predecessor company, Electrotools, has been machining graphite for our customers. We've produced, literally, millions of electrodes, blanks, fixtures, and other assorted graphite shapes; from airfoil electrodes for machining jet engine components, to electrodes shaped like a rawhide dog bone for the pet food industry. During that time, our Custom Electrode Department staff has confronted and solved almost every possible problem that you can imagine — or that you might encounter in your own operation.

As you would suspect from that much experience, we've developed a number of techniques and methods for machining graphite, that have proven to be efficient and cost effective. We would like to pass along some of these "tricks of the trade" to you.

sawing

Graphite is supplied in various raw block sizes, some as large as 24" x 24" x 72", so it can be safely assumed that some type of sawing will be needed. On our large cutoff saws, we use a standard 1-1/2" wide carbon steel blade, with 2-1/2 or 3 teeth/inch, running at a blade speed of 1800 SFM (Surface Feet/Minute). You'll find this to be a good starting point for most applications.

> "When selecting a saw blade, it is important to use a blade that offers the narrowest kerf available, yet with sufficient beam strength to prevent deflection."

On smaller vertical band saws used for finer cuts, a 5/8" wide carbon steel blade with 4 to 8 teeth/inch, running between 2000 and 2600 SFM, is an effective combination. Surface speed will have to be adjusted for different grades, but increases or decreases of less than 350 SFM should be adequate to compensate for material hardness.

When selecting a saw blade, it is important to use a blade that offers the narrowest kerf available, yet with sufficient beam strength to prevent deflection. The narrow kerf not only saves expensive material, but also reduces the amount of chipping on the exit side of the saw cut. The reduced cutting pressure of a narrow kerf will be especially appreciated when sawing thin sections, irregular shapes or rounds, or when clamping surface is limited.

Just a word about bi-metallic and carbide blades: while these blades offer many advantages over carbon steel blades for steel applications, they do not offer any significant advantage in graphite saw work. In our experience, their cost does not justify their use, even at triple the blade life.

drilling

Drilling is not only one of the oldest machining operations, it is still the most predominant — 60% of all machining time is spent drilling holes. And, since most graphite electrodes require flush holes, we would like to share some of the successes that advancements in cutting tool technology have afforded us.

Most drill manufacturers offer tools with a high helix, wide land, and parabolic flute. The parabolic flute shape incorporates a strong cross section with greater flute volume behind the cutting lips. This shape allows the graphite dust to be quickly removed from the point of contact, and provides exceptional deep hole performance (3" or more). The addition of a coating to the tool (TiN, for example) ensures longer drill life, and helps to reduce wear that results in tapered holes. This drill geometry has enabled us to increase both depth of peck and feed rate, thereby reducing our drilling cycle times.

> **"today's high volume production drilling jobs require the use of at least a coated solid carbide drill; and even diamond tipped drills."**

Whereas, a few years ago, coated high-speed steel drills were the way to go, today's high volume production drilling jobs require the use of at least a coated solid carbide drill; and even diamond tipped drills. Their additional cost is more than returned by extended life, consistent hole diameters, and reduced tool changes.

Feed rates for drills 1/32" in diameter and smaller start around .0015 to .002" in./rev. Feed rates for drills from 1/32 to 1/8" in diameter should be around .001 to .003" in./rev., and from .003 to .007" in./rev. for larger diameters. To prevent "blow out" or "chip out" on a through hole, be sure to use another piece of graphite under the workpiece, to allow the drill to transition into similar material at a constant feed rate.

However, improved tooling and increased performance doesn't prevent some simple mistakes, and drilling to a depth beyond the flutes is one of the more serious ones you can make. Graphite fills the flutes of a drill very quickly. If the drill is not periodically removed from the hole to clear the flutes (peck drilling), the graphite quickly travels around the shank of the drill, causing it to seize in the hole. This results in either snapping the tool, breaking the graphite workpiece, or both. If you must drill beyond the flutes, take a few minutes to grind clearance on the O.D. of the shank of the drill. Grinding the shank diameter to a size .004 - .005" less than the drill diameter is all that is needed to prevent seizing. This could be the difference between meeting your customer's due date and starting from scratch.

turning

We consciously strive to move as many machining processes from manual operations to our CNC equipment for obvious productivity and economic reasons, and turning is no exception. Consistent dimensional size from part to part, and reduced operator involvements are all benefits of CNC machine tools. Due to the abrasive characteristics of graphite, it was customary to use high-grade, TiN-coated carbide inserted tools. This is necessary to offset the rapid erosion of the cutting edge, caused by the abrasiveness of graphite. For example, typical spindle RPM for a 2" dia. rod of graphite of medium hardness and density is 800 RPM for both roughing and finishing passes. Feed rates are typically .015"/ revolution for roughing, and .004 - .006"/ revolution for finishing. Typically, a depth of cut of .015 - .025" will produce a better finish than one of .005". This is due to graphite's structure, which causes it to chip as the tool is introduced. It is also the reason why a tool with a positive rake angle and a nose radius of at least 1/64" is recommended.

However, things will always change! So, for the past several years, our CNC turning centers have been equipped with diamond tooling. This move has been the most productive and cost effective measure that we have initiated in the turning area. We use diamond film inserts produced utilizing the Chemical Vapor Deposition (CVD) technology. These inserts have enabled us to increase all our cutting speeds and feeds, while in most cases, achieving better surface finish than we get with TiN-coated carbides. It's not uncommon for the CVD inserts to exhibit a life 25 to 30 times that of carbide, before being changed out or re-lapped.

Another, and perhaps biggest, advantage to the diamond tipped tooling is its ability to maintain consistent size, piece part to piece part. Less "downtime" dedicated to chasing size by adjusting tool offsets, and less "downtime" dedicated to maintaining surface finish by changing inserts, translates into more "up time" producing consistent work.

milling

Milling of 2-D and 3-D graphite electrodes has grown tremendously over the last several years. Standard tooling for our mills was exclusively carbide. High speed and cobalt tools were used only when a diameter, length, or special cutter shape made the cost of carbide prohibitive. Several years ago, due to customer requirements, and the need to increase capabilities and reduce delivery time, Intech EDM invested in High-Speed Milling equipment. Today, our new "High-Performance Milling" centers are equipped with diamond tooling. The need for speed necessitated the need for tooling tenacious enough to handle that increased speed.

When milling graphite conventionally (non-high-speed), most people tend to set both the spindle RPM and the feed rate too low. Get those RPM's up there — you'll see the difference! If you want to be a bit cautious, start by increasing the RPM's by 10%, but remember to increase the feed rate as well. If the chip load per tooth is not sufficient, the graphite will act almost as a burnishing agent on the tool. Move the tool through the work, but not at a rate that produces excessive tool pressure. Chip loads of approximately .005"/tooth/rev. for roughing and .002"/tooth/rev. for finishing are effective starting points for milling applications.

> **"The need for speed necessitated the need for tooling tenacious enough to handle that increased speed."**

Another area of consideration when milling is the chipping of the graphite on the exit side of the cut. Although reducing the feed rate at the end of the cut, and making sure that the cutter is sharp helps to reduce chipping, try "climb milling". "Climbing" significantly decreases the chip out. I'd recommend climb milling only when using CNC equipment or those machine tools that are equipped with "ball

I've included some basic formulas to aid in your calculations:

SFM = Surface Feet Per Minute

RPM = Revolutions Per Minute

Pi = 3.141592

Cutter: Approx. Range of SFM

Carbide: 500-700

Diamond: 500-2000

SFM = (RPM x Tool Dia. X Pi) / 12)

RPM = (SFM x 12) / (Tool Dia. X Pi)

	Rough	Finish
Chip Load Per Tooth:	.003-.005	.001-.003

Feed Rate = (RPM x Chip Load Per Tooth x No. Teeth)

screws". Climb milling can be accomplished safely and effectively because "ball screws" have no backlash, as do manual machines equipped with lead screws.

Chip and dust removal need to be addressed! Be sure to keep the cut zone clean and clear of graphite chips and dust. Remove the dust/chips by using either vacuum or compressed air — or both. We have found that by keeping the cut zone clear of chips/dust, we are able to take deeper cuts and still maintain an excellent finish. Keep the cut zone clear of lubricating and coolant fluids as well. We recommend machining dry at all times. Chips/dust that are mixed with fluid tend to collect and stick in the flutes of the cutter, causing poor finish — not to mention the fact that a slurry of abrasive graphite can cause premature wear to all moving parts of your equipment. Given the fact that graphite is dirty to start with, who really needs to spend their time mucking out buckets of sticky black mud.

The advent of High Performance Milling has raised the bar in the milling arena. The high RPM of the spindle results in reduced tool pressure in the cut. The reduced tool pressure allows for the milling of very thin ribs and the ability to use micro end mills (less than .020" diameter) without tool breakage. Improved machine response, upgraded software with improved capabilities that produces better surface finishes, diamond tooling with ten to fifteen times the life of carbide, advanced work holding systems, and unattended operation, have all joined to move the milling area into a more prominent position within electrode manufacturing. Pallet changing devices on the mill table help to keep the mill in productive cut time, rather than fixture load time. Multiple set-up areas are used to run different jobs simultaneously, with a little program linking. It's all about productivity and cost effectiveness.

grinding

Most types of surface grinding can be satisfactorily accomplished by using a vitreous bonded, silicon-carbide wheel, with medium grit, medium hardness and an open structure. A "green wheel" with the designation of GC 60 H11 V fills the general requirements. Average depth of cut on rough passes is generally in the .010 -.020" range, while depth on finish passes ranges from .0002 to .002". Surface feed averages 45-55 ft./min.

Form grinding requires a wheel with a finer grit, and 100 grit wheels will work extremely well if you stay with the medium hardness. In plunge applications, these wheels hold their form very well, and even small radii can be dressed without wheel breakdown. Dress the wheel often to keep it sharp! Even though the wheel doesn't breakdown, it does fill and glaze. A glazed wheel not only loses its grinding effectiveness, it also generates a great deal of heat that can cause warping in the workpiece.

Medium hardness wheels with grit sizes of 46 or larger are not recommended. They tend to shed some of their grit during grinding operations, which causes scarring of the graphite.

Each particular machining operation has its own variables: size of the workpiece, clamping devices, condition of the cutting tools, accuracy of the machine tool, tolerances to be held, number of pieces to be produced, operator safety, dust collection, etc. And although each job has its own factors that dictate the approach you will use in machining, try a little experimenting on your own. A little experience combined with a willingness to experiment, will generate your own "tricks of the trade".

about the author

Tony Mucha is an experienced tool & die maker, receiving his journeyman card in 1970, at which time he was named Apprentice of the Year by the City of Chicago. He has managed Intech EDM's Custom Electrode Department for over 16 years, and is currently Vice President of Operations, responsible for the Custom Electrode Department, warehousing, shipping/receiving, purchasing and inventory management. EDM