Graphite milling, the proper way
Shaiu-Wei Yin,
IDI Precision Machinery Ltd.

I. Introduction
Graphite is widely used in various kinds of industries. For example, die and mold, semiconductor, aeronautical and astronautical engineering, furnace and power generation applications. Graphite is easy to cut. However, due to its abrasive particles, milling tools often wear very quickly. This situation is even worse in high speed milling. This essay discusses the proper way to cut graphite both efficiently and economically.

II. Principles
Graphite is very abrasive due to its bond strength between individual carbon molecules. The only effective way to conquer graphite is using diamond coated tools. Generally, diamond coated tools can last 15-20 times longer than uncoated tools. But it also depends a lot on cutting conditions, machine tools and user requirement of surface quality. There are several principles that operators should keep in mind when milling graphite.

Principle 1: Keep Vc high but below 400m/min.

Formula 1: \( Vc = \frac{3.14 \times D \times N}{1000} \)

- \( Vc \): cutting tool edge speed (m/min.)
- \( D \): tool diamond (mm)
- \( N \): spindle speed (rpm)

Graphite is easy to cut even at high speed. However, because it can be cut at high speed does not mean it should be. Some research shows the higher speed, the longer tool life. However if \( Vc \) is too high, the diamond film may fall apart too quickly.

Principle 2: Always keep enough feed per tooth
Principle 2 is the key point to successful graphite milling.
Below is the formula:

Formula 2: \( fz = \frac{F}{S \times Z} \)

- \( fz \): feed per tooth (mm/tooth)
F: table **real** feed rate (mm/min.)
S: spindle speed (rpm)
Z: number of flutes

For 3–6 mm shank tools, we give the following recommendation:
Roughing process: fz=0.1–0.15 mm/tooth
Finishing process: fz=0.07–0.12 mm/tooth
For smaller or larger tools, slightly adjusting the above fz can get quite good results.

For milling inserts, we recommend an fz as large as possible but below 0.4 mm/tooth to make sure the graphite edge will not chip.

In graphite cutting, the larger the fz, the longer the tool life. If fz is too small, then the tool is grinding rather than cutting. This will dramatically shorten the tool life. It is so sensitive that only a few changes may cause a totally different result.

The graphite part shown in Fig. 1 is a part used in ion implanters in the semiconductor industry. The surface quality is very important.

**Condition 1:**
Tool: 2 mm, ball nose, 2-flute, TiALN coated
Spindle speed: 6000 rpm
Table feed rate: 1500 mm/min.
Fz=1500/6000/2=0.125 mm/tooth
One tool can last about 4 to 6 parts. The tool life was unstable.
**Condition 2:**
Tool: IDI’s DMB2-4-6-20-50, 2 mm, ball nose, 4-flute, diamond coated tool
Spindle speed: 6000 rpm
Table feed rate: 1500 mm/min.
\[ F_z = \frac{1500}{6000}/4 = 0.0625 \text{ mm/tooth} \]
The tool life was only about 4 times of TiALN coated tools.

**Condition 3:**
Tool: IDI’s DMB2-4-6-20-50, 2 mm, ball nose, 4-flute, diamond coated tool
Spindle speed: 5000 rpm
Table feed rate: 2000 mm/min.
\[ F_z = \frac{2000}{5000}/4 = 0.1 \text{ mm/tooth} \]
The test result showed the tool life more than 18 times of TiALN coated tools. The reason we did not get a final result was that the tool cut all parts ordered, more than 100pcs and was still good.

We can see from condition 2 and 3 that only a slight change in cutting condition may get totally different tool life. \( F_z \) is the key point. The larger the \( F_z \), the longer tool life you can get. However, we can not increase \( F_z \) to points where the flute breaks or the graphite edge chips. That is why we suggest an \( F_z \) below 0.15 mm/tooth for shank tools, and below 0.4 mm for milling inserts.

### III. Practical
#### a. feed rate
A common mistake that operators often make is the feed rate. In formula 2, \( F \) represents “real” table feed, not the feed rate command of the CAM program. For small electrodes, or electrodes with complex geometry, the machine tool needs to slow down when it changes direction in order to maintain an accurate profile. In other words, the “real” average feed rate that the machine tools can reach is always slower than the command of the CAM program. Let’s take a look at an example.

Fig. 2 shows a small electrode used in shoe mold making.
This size of the electrode is about 50x50x60 mm. For finishing of the vertical walls, if the programmed feed rate command is given F6000, with a traditional C-type vertical machining center, the maximum possible real average machining feed rate is around F2000. If you are using an excellent high speed machine which has very good dynamic response (not high speed spindle only), for example, GF”s HSM400, the possible average feed rate is around F3000. In either case, no machine can reach F6000. Then we should put F2000 or F3000 in formula 2 instead of F6000. This tiny difference is often neglected by operators and causes very quick tool wear.

However, for simple and large electrodes, you can take advantage of all the tool can possibly reach and get very good tool life.

Fig. 3
Fig. 3 shows a relatively large electrode. When machining the outer shape of the electrode, if the machining program neglects the holes on the side and leaves it for another program, then the shape is very simple, and the size is large enough for the machine tools to reach F6000. In this case, the real average feed rate may be very close to or is the same as the command. In formula 2, we may put F6000 for calculating the suitable spindle speed. In this case, if we keep fz=0.1 mm/tooth, for a 2-flute tool, we need 30000 rpm, for a 4-flute tool, we need 15000 rpm.

b. roughing
Roughing of graphite electrodes can be done by using inserts, if the electrode is large enough, or shank tools, if it is small. For electrode size 50x50 mm or above, we recommend using diamond coated round inserts RDHX0501 or RDHX0802 with a 13 mm cutting diameter and 12 mm shank diameter tool holder.

Fig. 4
Fig. 4 shows an example of this tool. The merit of using round inserts is that you can turn it to another new position and keep using it. And the cutting edge is 1 mm larger than the shank diameter to avoid any
possible interference of the vertical walls that are often seen on graphite electrodes. In general, we recommend S5000 F4000 ap=0.5 mm, ae=7 mm for roughing. In some cases, people use S8000 F8000 and still get a pretty good result. Due to small ap (vertical cutting depth), no semi-finishing is needed. You can directly go to finishing and save lots of time. If the electrode is larger, we recommend using APHT100308, ap=1~1.5 mm, cutting diameter 17mm and 40% ae to get a quick enough removal rate.

For smaller electrodes, a 6 mm corner radius tool is recommended. S5000 F2000 is usually the suitable cutting condition.

c. 2-flute, 3-flute or 4-flute?
Tool manufacturers are often asked how many flutes should be used for graphite cutting. There is no standard answer for this question. A funny thing is that they always answer their tool solutions are the most suitable solution for everything. Or they just make 2, 3, 4 flutes and let customers choose. One thing that should be noticed here is that graphite is cut into dust particles instead of chips, so 4-flute tools will not have chip evacuation problems in most of the cases. The only exception is in the demo show of machine tool exhibitions where very high spindle speed, deep radial and axial depth of cut and high feed rate must be shown to attract visitors. Of course, this is not our daily life.

The right answer for number of flutes that should be chosen actually depends on the characteristics of the electrodes, their sizes, geometries, and the requirement of surface finishing and the machine tool you have. We have discussed above, the principles of graphite machining. According to Principle 1 and Principle 2, we would like to cut graphite at high speed, but need to keep enough feed per tooth at the same time.

Let’s say if you are making small molds. Most electrodes are within 50x50 mm. And you have a general machining center with maximum spindle speed 10000 rpm. Then you should choose 4-flute tools. Because in most of the applications, you are cutting at F2000, you can use S5000 for a 4-flute tool while with a 2-flute tool, you need 10000 rpm all the time.

However, for the same small electrodes, if you have a very high speed machine with maximum spindle speed 30000 rpm, then you should choose 2-flute tools. This is because high speed spindles often are not suitable for low speed cutting, for example S5000. Usually a minimum spindle speed of 10000 rpm is needed. For small electrodes which machine tools can not reach high table feed rate due to the small distance it can accelerate, the average feed rate is often slow, sometimes as slow as F1500. Then to keep enough feed per tooth at low table feed and high enough spindle speed at the same time, you need fewer numbers of flutes. 2-flute tools are the correct application.

The same logic can be applied to large electrodes. However, in most of the applications, 4-flute tools are more suitable in large electrodes because the feed rate is no longer the limit and with the same spindle speed, you can get higher feed rate to save time.
IV. Conclusions

Good graphite machining conditions depend on the characteristics of the electrodes and the machine tools. Keeping suitable Vc and enough feed per tooth are the key points. The operator should adjust spindle speed, feed rate and choose correct cutting tools to reach the best condition. From the above discussion, we can find that cutting tools are not the limit of the cutting speed of graphite machining. The machine tool is. It is only when we need good cutting efficiency and low enough tool wear to make money out of it that we need to consider all the facts that control the whole process. When we find a good trade-off point, we are cutting in a proper way.