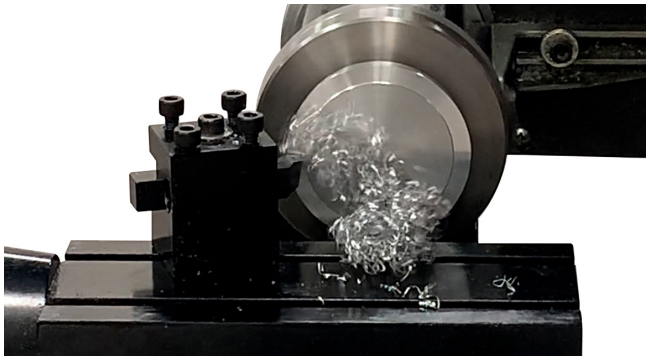

SHERLINE PRODUCTS
INCORPORATED 1974

Eliminating Chip Build-up on a Lathe

How to Avoid Stringy Chip Buildup and "Bird Nests" when Turning Parts on the Lathe

We have had some customers ask what they can do to reduce stringy chips that build up and wrap around their parts (bird nest) when they are doing a turning or facing operation on their lathe.

The type of chip that you get depends on the chip breaker on the carbide insert, or if you have ground a chip breaker on your HSS tool.

Below is a picture of one of our 55° inserts, so you can see what the chip break looks like (circled). This specific insert is designed for finish cuts, and it will get a long stringy chip.

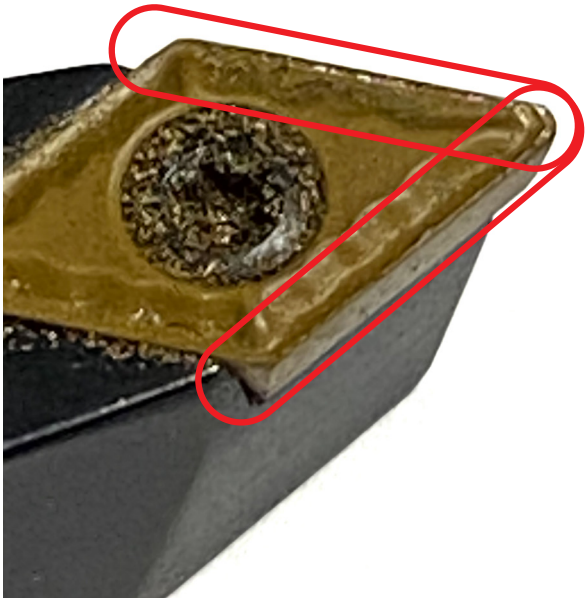

FIGURE 1

The 80° insert will generally produce chips because it is taking a larger depth of cut at a higher feed rate. On the finish passes it will still produce stringy chips that bird nest on your part (see Figure 2).

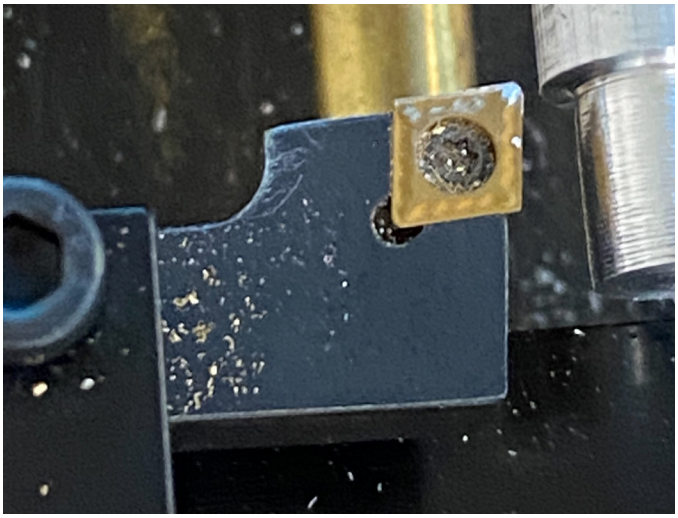

FIGURE 2

The feed rate has a lot to do with the kind of chip you will get. The picture below shows our 80° insert taking a .025 depth of cut at .010"/rev at 2000 rpm. This is equal to a feed of 20 in/min, which is very fast.

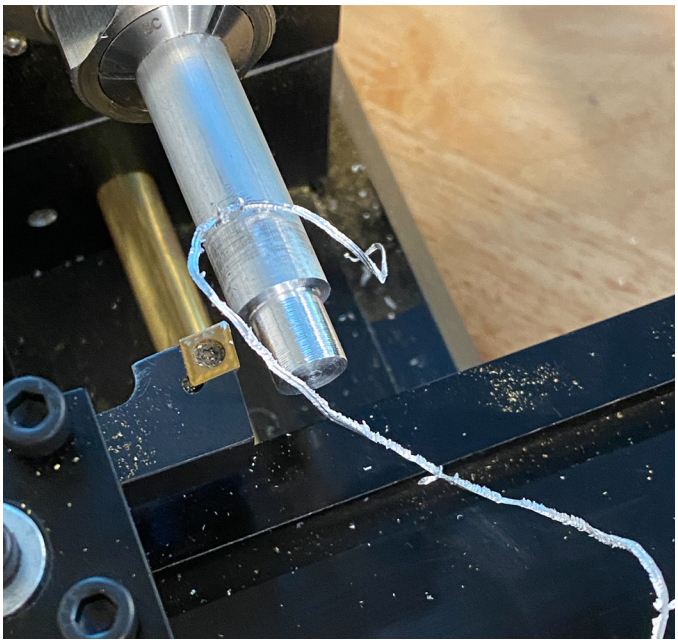

FIGURE 3

The other option that works is to grind a chip break on your HSS tool. The tool below shows the chip break circled. If the shoulder is more square than round, it breaks the chip better. However, hand grinding a square chip-break relief is not easy. The rounded corner will force the chip into a tight spring-like chip that is less likely to wad up and bird nest on your part.

Below is a RH Tool with the chip break top step ground on it.

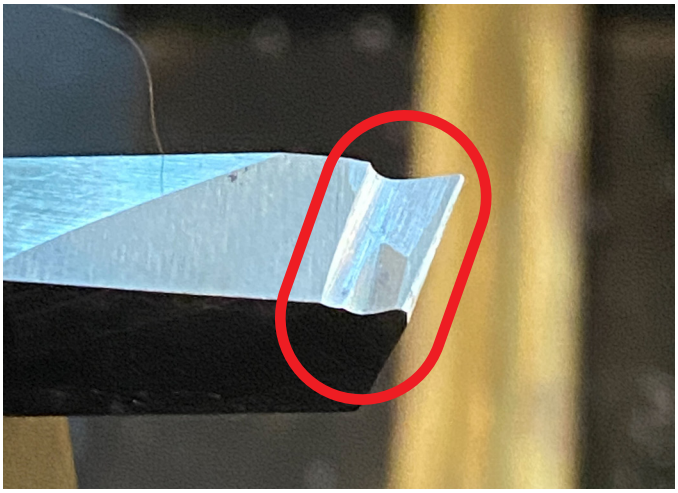

FIGURE 4—Back, top-side view.

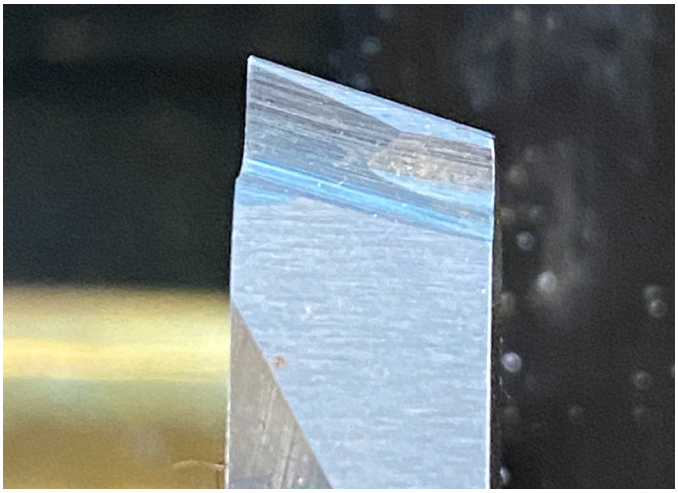

FIGURE 5—Top View.

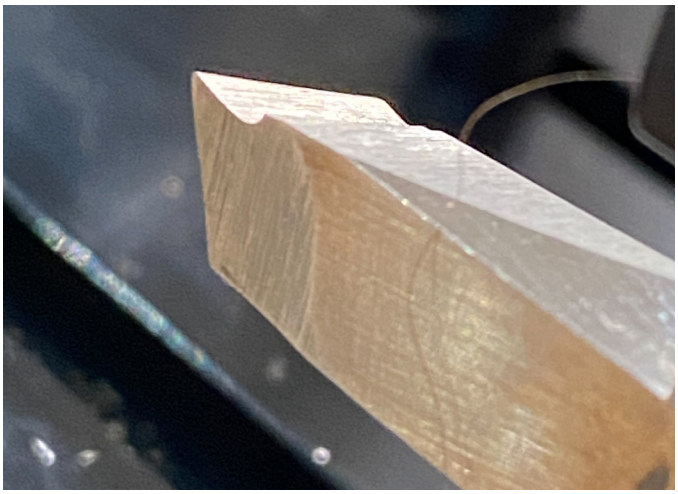

FIGURE 6—Front-side view.

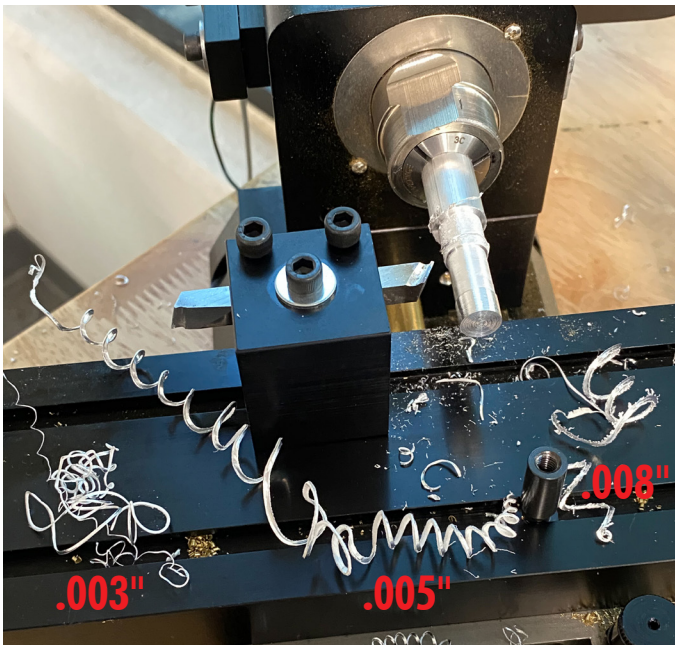

FIGURE 7—Chips at different feed rates. Left: .003"/rev., Middle: .005"/rev., Right: .008"/rev

Even at the low feed rate of .003"/rev, the chip break causes the chip to form differently and it's not as likely to bird nest on your part.

To calculate your feed rate in In/Min, take the spindle RPM and multiply it by the desired "feed / rev," which equals the feed rate in In/Min.

Example

You have ½" 6061 aluminum with an SFM (surface feet/minute) of 300-400

(SFM x 3.82) / Diameter to be cut = Max RPM

- 400 x 3.82 = 1528
- 1528 /.450 = Max spindle speed of 3395 RPM

Our max spindle speed is 2800 RPM. In the following example, the spindle speed is 2000 RPM.

You want a roughing feed of .008"/rev and a finish feed rate of .002"/ REV.

- .008 x 2000 = F16.0 (16 In/Min)
- .002 x 2000 = F4.0 (4 In/Min)

Thank you,
Sherline Products Inc.