Kurt's clinic

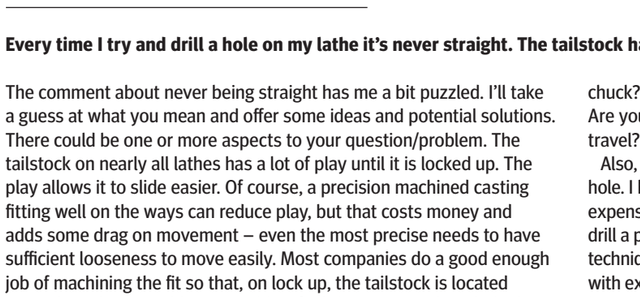
Kurt Hertzog answers readers' questions

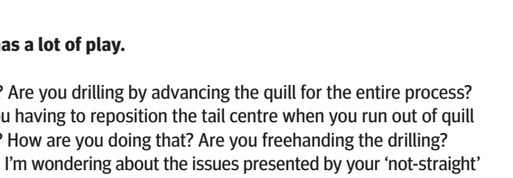
Every time I try and drill a hole on my lathe it's never straight. The tailstock has a lot of play.

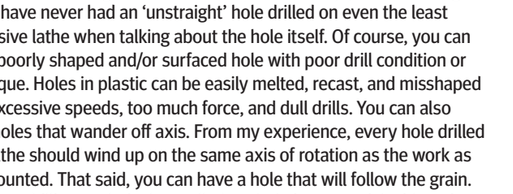
The comment about never being straight has me a bit puzzled. I'll take a guess at what you mean and offer some ideas and potential solutions. There could be one or more aspects to your question/problem. The tailstock on nearly all lathes has a lot of play until it is locked up. The play allows it to slide easier. Of course, a precision machined casting fitting well on the ways can reduce play, but that costs money and adds some drag on movement – even the most precise needs to have sufficient looseness to move easily. Most companies do a good enough job of machining the fit so that, on lock up, the tailstock is located properly and reasonably repeatable. I find that even my best lathe has some variation in the lock-up position repeatability.

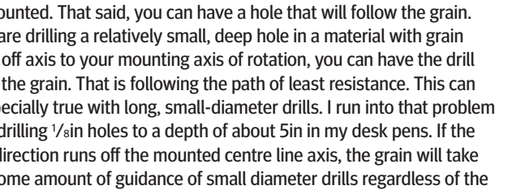
Does your question/comment refer to the quill? Most quills travel straight when advancing using the internal thread and guide system. Yes, some quills have a more precise guide system in the tail centre and have a tighter fit (60° vs 75° for example, along with precision fits) between the threads. Even if there is a loose fit of the quill and less engagement on the threads, the drill should cut on the axis of rotation following the path of least resistance. If your quill wobbles as you advance it, you should investigate that issue. Usually there is simply an internal thread that mates with the advancing screw and hand wheel. There is also an anti-rotation slot tracking a locking screw mechanism. This can range from machine tool fit and finish to sloppy. That still shouldn't be the issue on drilling 'straight'. It might not be perfectly parallel with whatever you are using as a datum but it should be a straight line inside the hole. I'm assuming you mean not straight with respect to your blank. With respect to which surface? Is the blank round? Is it square? Truly square? How are you holding it for drilling? A three-jaw or four-jaw vice? A collet? A two-jaw, V-cut jawed blank drilling vice? A drill chuck? Are your drilled holes a deep, small diameter with potential drill wander? Is your drill long and hanging way out of the drill chuck? Are you drilling by advancing the quill for the entire process? Are you having to reposition the tail centre when you run out of quill travel? How are you doing that? Are you freehanding the drilling?

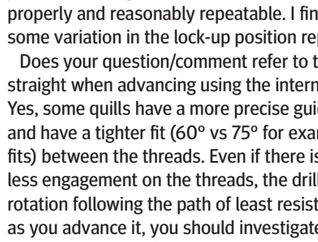
Also, I'm wondering about the issues presented by your 'not-straight' hole. I have never had an 'unstraight' hole drilled on even the least expensive lathe when talking about the hole itself. Of course, you can drill a poorly shaped and/or surfaced hole with poor drill condition or technique. Holes in plastic can be easily melted, recast, and misshaped with excessive speeds, too much force, and dull drills. You can also have holes that wander off axis. From my experience, every hole drilled on a lathe should wind up on the same axis of rotation as the work as it is mounted. That said, you can have a hole that will follow the grain. If you are drilling a relatively small, deep hole in a material with grain that is off axis to your mounting axis of rotation, you can have the drill follow the grain. That is following the path of least resistance. This can be especially true with long, small-diameter drills. I run into that problem when drilling 1/8in holes to a depth of about 5in in my desk pens. If the grain direction runs off the mounted centre line axis, the grain will take over some amount of guidance of small diameter drills regardless of the care I take. Does that present an uncentred hole with respect to some outer surface? Certainly. Depending on your work-holding method and the various surfaces of your blank, you can put a hole quite far off the blank's actual centre or off axis to the various surfaces. I think the issue isn't with your equipment but perhaps I'm not truly understanding the 'not straight'. If your issue is workholding, you might find some help in general within my 13-part workholding series of articles. You can find exhaustive coverage on the subject in *WT238-250*. Hopefully some of the issues I've spoken to will give you a path to look at. If not, perhaps it will be of value to other readers. Not mentioned but assumed you are using sharp drills, appropriate speeds and feeds, and good drilling techniques.

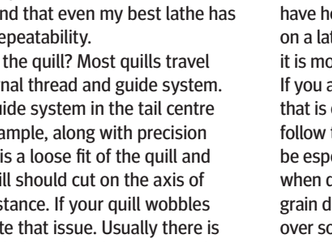












PHOTOGRAPHY BY KURT HERTZOG

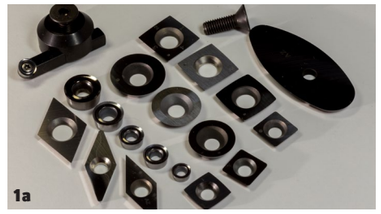
1 Large or small drills, I use a technique learned from Dick Sing. With the quill retracted, slide the tailstock forward by hand to drill your hole. **2** Keeping your drills sharp is key to good holes. The best investment I have ever made for drills was an inexpensive sharpening machine, be certain you can sharpen the range of drills you use. **4** Many times, issues with concentricity or other drilling related issues are workholding related. Get as much of your stock mounted well. **5** Good drilling practice, using proper speeds and feeds, includes retracting the drill and cleaning the flutes as needed. **6** An 1/8in, 5+ inch deep drilling on an unsupported blank presents problems. Notice the grain orientation that may take over the guidance to a degree.

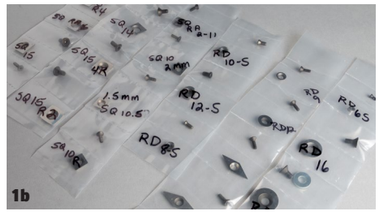
Why is there such a price difference between brands of carbide tool replacement cutters?

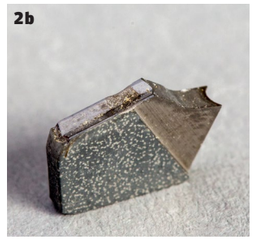
Carbide cutters for woodturning tools are in a different league to machine tool carbide cutters. Not only are they usually a more complex design, but are markedly lower volumes. Among woodturning cutters, different designs can have totally different costs. Much like any product, complexity, material consumption, market segment, specific design, proprietary issues, etc. make different cutter costs vary considerably. To compare apples to apples, you need to be comparing the same part number cutter or at least those with direct replacement/interchangeability. Most manufacturers have their own replacement offerings for each tool, including any other designs of cutter that will fit that handle.

The price for different designs varies even within the same company offerings. More carbide in the part, additional complications that increase manufacturing processing, lower volume on that part number, along with any marketing hype, can change costs. That said, if you are comparing a generic design replacement cutter that is offered from multiple companies, the best short answer is likely how many hands touch it between the manufacturer and you. In more detail, there are many factors affecting the cost you experience buying any product. Obviously, the mark-up at your final sales location has a big impact. Every stop between the OEM manufacturer and you adds additional tiers of mark-up. Along with transportation costs from manufacturing site to each of the interim stops, don't overlook the cost of packaging. A retail, theft-resistant bubble package in your local retailer certainly costs far more than multiple quantity industrial-type packaging or a simple poly bag stapled closed. Some cutters are sold directly to the end user while others go through the multi-level marketing process.

On the manufacturing side, carbide cutters are a milled material blended concoction that are pressed and sintered. The milled carbide materials, binders, and the proprietary magic sprinkles prior to pressing have cost, as do pressing and sintering. Other costs factoring in are the manufacturing batch volumes, tools and equipment, yields and more. Larger companies with higher-volume production processes benefit from economies of scale compared to smaller houses. With all that, my guess is that the marketing costs are responsible for the largest part of the variations you see. Everything said is indicative of any product offering, from bread to automobiles. Don't ever forget that many times the price asked is determined by what the market will bear.







1 Just a small sample of the various carbide replacements available. You need to be certain any replacement nests like the original. **2** The design and fabrication of the carbide cutters is far more complex than perhaps you give it credit. **3** Retail packaging adds cost to your product. It can vary from the simple poly bag to more exotic visual or theft resistant.

Carbide replacements are so expensive. Can I sharpen them? Any tips for getting more life from them?

I do touch mine up but only certain types. I remove the flat cutters when they need attention and use a flat diamond hone. Keeping the cutter flat and in contact with the hone while moving it in a circular motion seems to give me some additional life. You can do this dry or with a bit of water or oil. Nothing required on diamond. I don't think I get as good as new but it certainly gives me some added use. Other designs don't lend themselves to touching up. I find that marking my cutter and rotating the cutter as needed extends its life. A simple line that sometimes needs refreshing with use serves well. As an edge begins to lose it, use it for the heavy lifting. When you need your fine touch, rotate a virgin edge into place. Once done with any fine work, rotate the pristine edge back out of service to do your tough work with a declining edge.

As the work horse edge becomes unworkable, press the pristine edge into service, setting aside the next virgin edge portion of the cutter for use as needed. Once you've gone all the way around, touch it up if possible or simply pitch it.