Simple thread cutting on a Unimat

I have a project that involves gripping the shaft of a dial indicator in something. I decided to do it using a threaded collar with a bore in it to fit the shaft. Like many Unimat owners I happen to have a set of M12x1.0 taps so I thought I would use them to thread the hole that the collar would screw into because the 11mm diameter of the bottom of the threads would leave plenty of material for the 9.5mm bore.

Unfortunately, also like many Unimat owners, I don't have a threading attachment. So I gave the problem some thought and came up with this simple solution, which worked so beautifully well for this application that I feel compelled to describe it to the list. It's rather obvious once you get the drift of it but I have never heard anyone mention doing it this way - so it may be useful to others like me who haven't heard of it either.

I will describe the solution I devised for my specific problem and then discuss how it might be generalized to similar threading problems.

The basic idea is to use the unique spindle quill on the Unimat to allow movement of the workpiece along the spindle axis, just as others have described for threading mechanisms that use a reference screw attached to the pulley end of the spindle. However, instead of doing that I decided to mount the reference screw to the right of the workpiece and make the workpiece itself the nut that engages the reference screw. The reference screw can now be grasped in the drill chuck mounted on the tailstock in the usual way.

Here's how it worked for my M12x1.0 collar.

First, I chucked some brass hex rod, because I wanted hex flats on the collar for wrenchability. Then I turned down the rod to 12mm to get the surface for threading. I then center-drilled a hole clear through the workpiece using a #9 drill, which is the tap drill for M6x1.0. I tapped the hole using a cheapo tap in the drill chuck.

Then I got a M6x1.0 bolt of about 200mm length and cut the head off. I chucked the bolt in the drill chuck, threaded it into the hole in the workpiece, and clamped the tailstock.

I loosened the spindle quill clamping nuts. The motor puts quite a lateral load on the quill locating screw making the quill difficult to move in the horizontal position. So I unclamped the motor and rotated it so that the center of mass of the motor was over the axis of the spindle, thus reducing the load on the locating screw enough that the quill moves easily. Now rotating the spindle causes the workpiece to advance along the reference screw, drawing the quill out of the headstock.

I ground a new tool to a reasonably accurate 60-degree point using a 30/60/90 triangle from a geometry set. I just left it sharp and didn't worry about the proper flat that should be on the point. The increased weakness produced by the sharp bottom of the thread is not an issue in my application (or many others, either). I mounted the tool in the standard tool holder, aligned it to 90 degrees, and arranged it slightly to the right of the workpiece by turning the lead screw. Then I clamped the carriage in that position because it is the spindle that I want to move, not the carriage.

I advanced the cross-feed until the tool cut into the workpiece slightly and rotated the spindle by hand using the pulley. This cut a smooth 1.0mm thread down the workpiece just as planned. I took many cuts - probably more than were necessary - until the edges of the threads just met, giving the proper sharp profile to the threads. It doesn't actually matter if you cut a little less or a little more because the triangular threads will tighten up nicely and accurately regardless of any reasonable

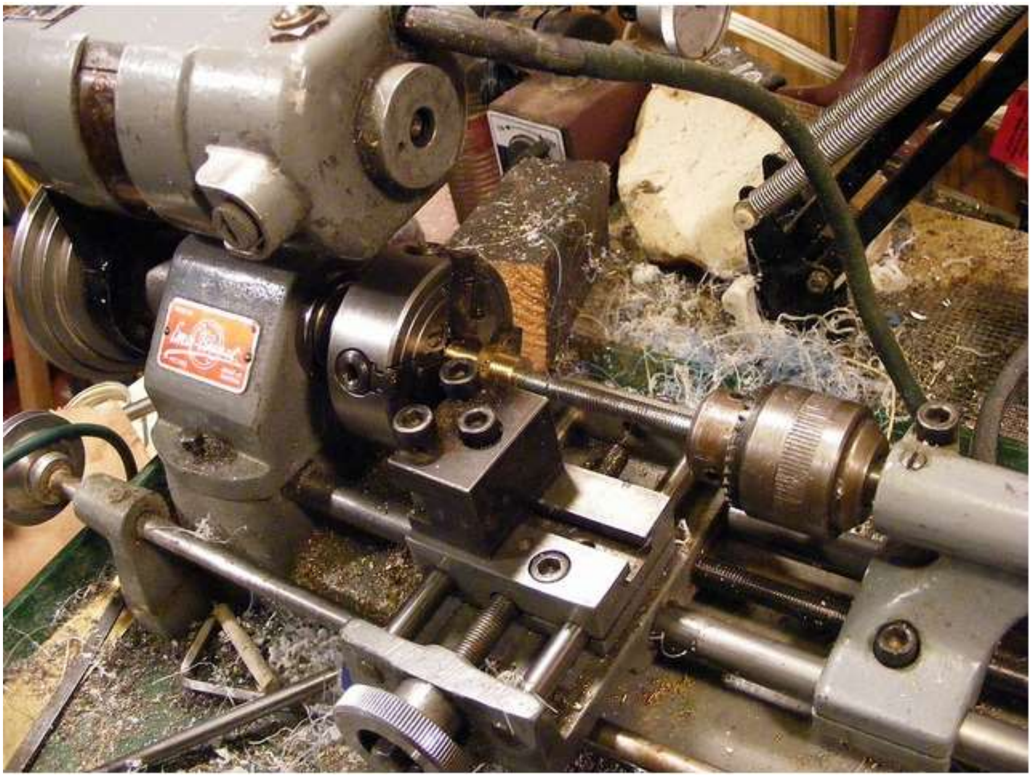
amount of clearance. That's the beautiful thing about triangular threads!

Once the threads were cut I moved the carriage out of the way and tested the M12x1.0 threads on another chuck I have. The result was perfect on my first try - the chuck tightened nicely on the threads and spun as truly as if it had been mounted on the spindle thread!

I then drilled and bored the workpiece to 9.5mm and parted it off, giving a nice looking little piece, exactly what I needed, and made on the Unimat with no special equipment at all.

Clearly, this whole process was enabled by the fact that I was willing to drill and tap through the center of my workpiece. If you are willing to do this then this exact process will work to put external threads on anything as long as you have a rod with the right pitch on it that is sufficiently smaller than the threads you are trying to cut. This will be the case for many, if not most, standard fine threads because they try to re-use the same pitches at various diameters. Presumably these standard pitches were chosen to minimize the number of gears required on a screw-cutting lathe.

If you are not willing to put a threaded hole through your workpiece you can still use a variation of this technique. Leave extra material on the end of the workpiece in which to put the threaded hole and cut it off later with a saw or parting tool. The extra material has to be at least as long as the threads you are planning to cut, plus some extra to allow a few turns of the rod to be engaged initially. If you don't have a bottoming tap you'll need a little more to allow for the incomplete threads at the bottom of the hole. If your threaded section is quite long you might end up with some support problems but bear in mind that you are already supporting it on a rigidly-mounted threaded rod and you can always use a steady, if you have one.



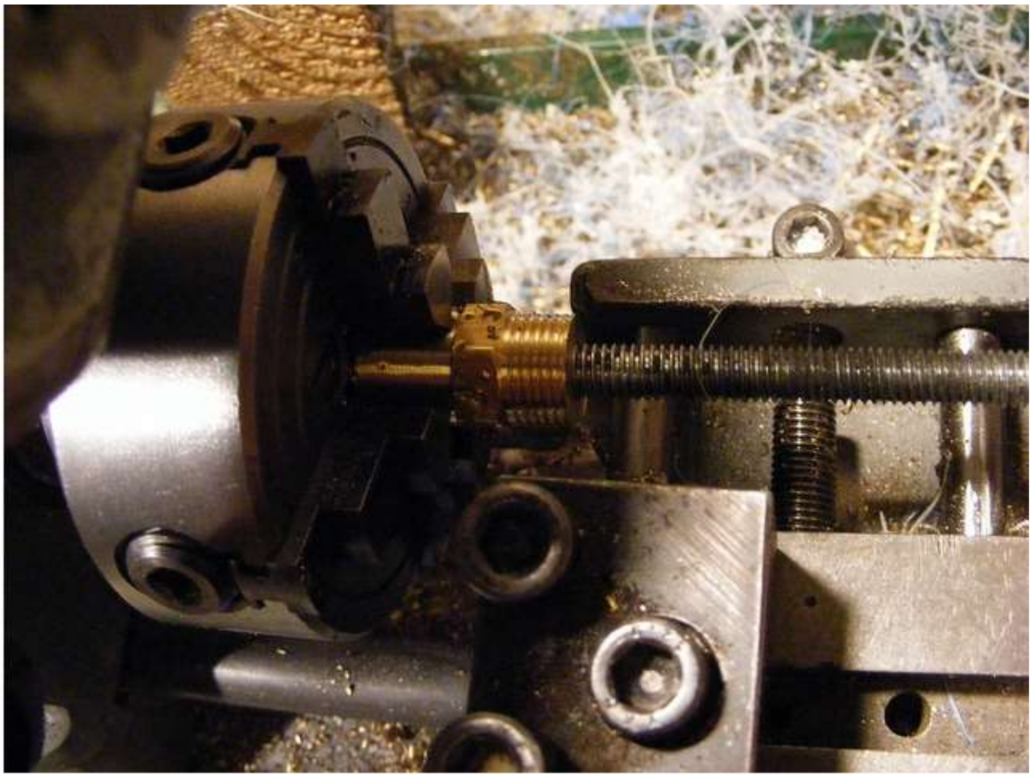
Unimat set up for simple threading

I needed a little collar with an M12x1.0 outside thread and a 9.5mm bore to fit into a hole threaded with a tap set I happen to have. Unfortunately, the Unimat does not have any capability for cutting threads except via a rare and expensive attachment.

So I developed this threading method that requires nothing except a threaded rod at the required pitch. The rod seen here is an M6x1.0 bolt with the head cut off.

The screws that normally bind the spindle quill in place have been loosened to allow the spindle to move along its axis. The motor has been repositioned over the spindle to reduce the drag on the headstock quill caused by the locating screw. It slides quite freely with the motor overhead.

The workpiece has been turned down to 12mm and drilled and tapped to M6x1.0 to match the bolt. The bolt is fixed in the drill chuck, threaded into the workpiece, and the tailstock is clamped. When the chuck in rotated in the normal cutting direction the screw advances the spindle forward at the right rate to cut a 1.0mm thread pitch at whatever diameter the workpiece is, 12mm in this case.



Thread cutting underway

Cut the threads with a 60 degree pointed tool. Take cuts in stages. Take lighter cuts as the work progresses because more metal must be removed as the surface of the thread increases.

Thread cutting finished

Keep cutting until the ridges of the threads are pretty sharp. It doesn't actually matter all that much because 60-degree threads will tighten accurately even if they are somewhat loose.

Credit: Above idea from Doug Collinge, Yahoo Unimat Group