MAKING AND USING D BITS, FORM TOOLS AND SPADE DRILLS

Mick Knights makes his own form tools

Last time we looked at the more popular types of reamers and the reaming process. Now it's time to unlock some of the secrets behind form drilling.

Form drills fall into three distinct categories: Form or step drills, D bits and spade drills. The most popular of these is still the D bit so called as in profile the tool resembles the letter D. These are cheap, quick and easy to produce. It should be stated from the outset that a surface grinder is an essential piece of kit in the manufacture of all form drills and D bits made from HSS tool bits. It's also the machine of choice for finishing any D bit but I will demonstrate that D bits can be made by using basic machine tools. This is achieved by carefully milling to the centre line leaving a couple of thou for stoning out any machining marks after hardening. In this way a satisfactory result can be obtained, **photo 1**. The finished D bit needs to be machined spot on the centre line, to produce a smooth and parallel flat face. Smaller diameter D bits may require surface grinding, as too much vibration and tool deflection can occur if milling to the centre line is attempted.

D bits can be made from either silver steel, **photo 2** which is by far the easiest, or from a round HSS tool bit, **photo 3**.

I have to confess, that at the moment I don't have access to a surface grinder, so I can't include any pictorial accompaniments by way of illustration, but along with some examples I have dredged out from the bottom of my tool box and some explanatory sketches, the basics can be adequately explained so that any machinist with a rudimentary knowledge of surface grinding could confidently tackle grinding a form drill.

Producing a D bit from silver steel has many advantages, not least is the fact that they can be produced on the lathe where it is straightforward to incorporate several shoulder depths and blend angles. Radial and conical forms can also be turned using templates for reference, **photo 4**. To cut this form to 0.532 radius using the example shown would take more horse power and rigidity than the normal hobby lathe could supply but is included to demonstrate what can be achieved when machining with "D" bits.

Even if the cutting diameters to be produced are small, it's good practice to turn from a larger diameter piece of silver steel, as this will give the tool extra strength when performing the cutting operation.

When producing a form or stepped bore in a soft material such as brass, a D bit can be plunge cut directly into a workpiece, as the 120 degree lead acts as a centre drill. In this situation the workpiece needs to be securely clamped to a machine table, or held in a chuck on the lathe. In tougher materials, mild steel etc. a centre drilled location is needed to avoid any deflection. A smaller diameter pilot hole drilled to the required depth may also be needed to act as a guide for the lead angle.

Form and step drills

It is quite a simple process to grind a pilot diameter on an ordinary jobber drill, thereby producing a square cutting shoulder, which is particularly useful as a counterbore. There are purpose made grinding fixtures, which pull down on the surface grinder's magnetic chuck, that are ideal for spinning down drills. These

Photo 1. Silver steel "D" bit, milled to centre line, hardened and stoned to achieve smooth cutting edge.

Photo 2. A range of silver steel "D" bits.

Photo 3. D bits ground from round HSS.

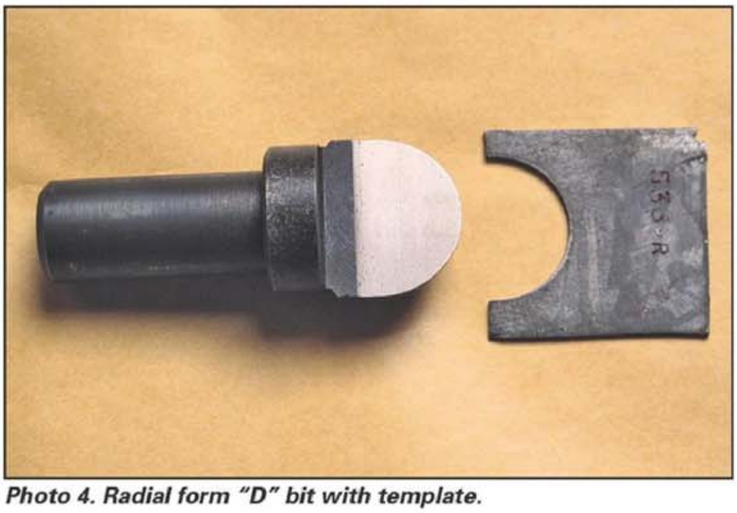
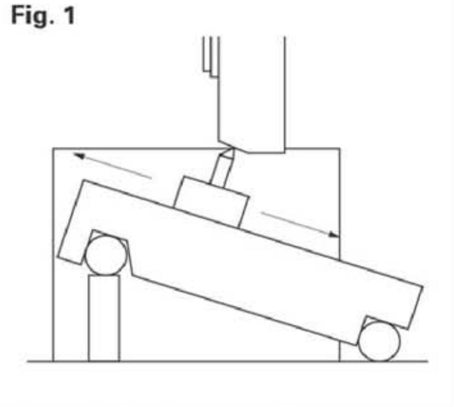


Photo 4. Radial form "D" bit with template.

Fig. 1



Diamond dressing an angle on a grinding wheel.

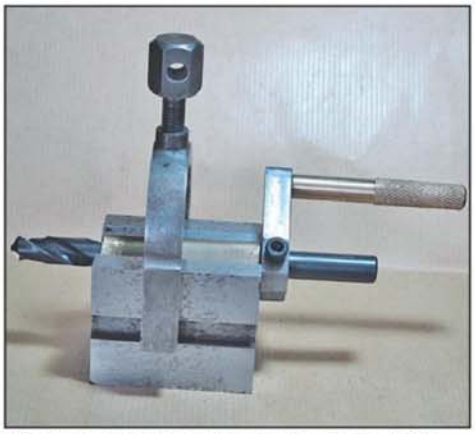


Photo 5. Spinning fixture for form drills and HSS tool bits.

Photo 6. Another view of the spinning fixture.

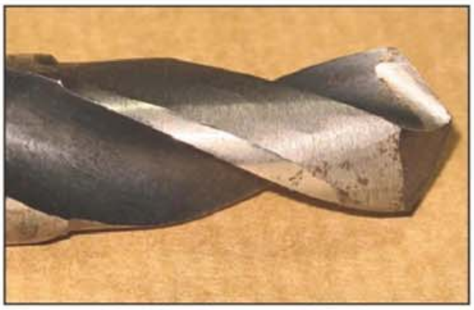


Photo 7. Double diameter drill, showing hand ground backing off, to leave a cutting land. The web thickness has also been reduced.

Photo 8. Drill before the web thickness reduction.

Photo 9. Drill after the web thickness reduction.

Photo 10. Spinning fixture handle parts.

fixtures were designed to regrind the first, second and plug tapers on taps and so come with a range of collet sizes. These fixtures are equally useful for grinding step drills and D bits, as they can be easily indexed to any angle thus avoiding the need to dress the grinding wheel to achieve desired angles, **Fig. 1**.

In the home workshop, the same result can be obtained by lightly clamping the drill in an old Vee block and attaching a spinning handle to the back of the drill shank, **photos 5** and **6**. If the step drill is to be used as a counterbore, then the pilot diameter should be left as ground. Carefully back off of the diameter by hand following the helix angle of the drill against the grinding wheel leaving a witness of the original ground diameter, to produce a cutting land, **photo 7**. The pilot diameter can then be used as a drill, for machining a double diameter hole. In this case the web will also need to be reduced to the web thickness a drill of the pilot's diameter would normally have. This is achieved by holding the drill web at 45 degrees to the corner of the grinding wheel and carefully grinding a relief down the helix for a short distance, **photos 8** and **9**. The shoulder cutting faces also need to be carefully backed off by hand, leaving only a witness of the original grinding remaining.

If a taper is required, rather than a square shoulder, the front face of the grinding wheel needs to be dressed at the desired angle. Freehand dressing, using a diamond, or any other type of dressing stick, is definitely not advised, as any uneven surfaces generated will be transferred to the workpiece when machined. Some well equipped workshops may have special wheel dressing attachments for this purpose. The other most common method needs a steady hand. The block mounted diamond is slid up and down on a sine bar. Rather than expose slip blocks to magnetism, a spacing block could be used to achieve the stack height. Usually the sine bar would be clamped against an angle plate and the vertical face of the angle plate will also act as a sliding face for the diamond block. Depending on the style of grinder, the table or wheel head is gently advanced after each pass of the diamond, until the required taper flank is achieved, **Fig. 1** For all these applications the front vertical face of the grinding wheel also needs to be dressed at regular intervals, to produce a smooth ground shoulder flank.

Making a spinning fixture for use on a surface grinder

It's always good practice to use an old Vee block as a base for the spinning fixture, **photos 5 & 6** as continual rotating of drills and tool steel may generate slight radial indentations in the side of the Vees. The overall dimensions of the spinning crank will depend on the Vee block selected. For comfort, the rotating handle should freely move around the securing cap head screw, as the drill needs to be spun at the same kind of speeds a workpiece would turn on a cylindrical grinder, say about sixty rpm. Slow rotation causes the grinding wheel to over heat the drill, which can result in burning the ground surface. These are known as check marks.

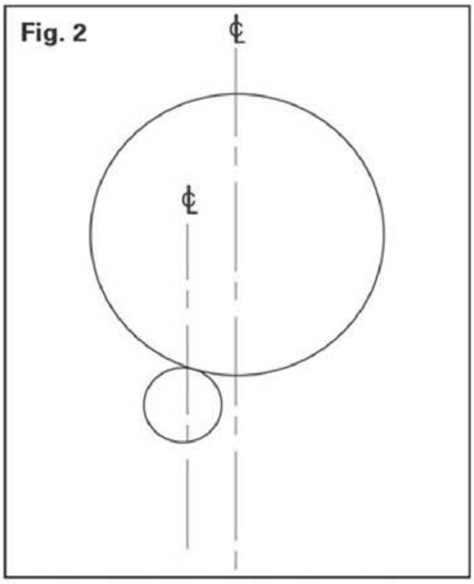
The clamping plate, which holds the drill in the block, is produced by bending a piece of sheet brass at 90 degrees, producing a Vee form to oppose the form in the block. The Vee block clamp is then tightened gently by hand, until the drill will rotate freely, but has no up and down movement. The spinning crank is made from any convenient piece of steel to hand. The centres for the drill clamp and rotating handle will depend on the height of the Vee block. As the largest drill, with a plain shank, is likely to be 13mm the clamping hole needs to be 13.5 dia. which can then accept a range of drill sizes.

Any convenient small diameter cap or grub screw can be used for clamping the drill shank. Larger diameter Morse mounted drills can also be spun down, but this would require a different crank, which would clamp on the tang of the drill. The rotating handle needs to be longer than the length of drill shank protruding from the block; this is to enable unimpeded rotation. A cap screw is the ideal means of securing the handle to the crank, but as not everybody has a bottomless tool box of odd screws, studding and a couple of lock nuts make an acceptable alternative, **photo 10**. When set up, the spinning crank needs to rotate against the back face of the Vee block. This will act as a check face to keep the shoulder position with the grinding wheel constant.

Grinding a step drill

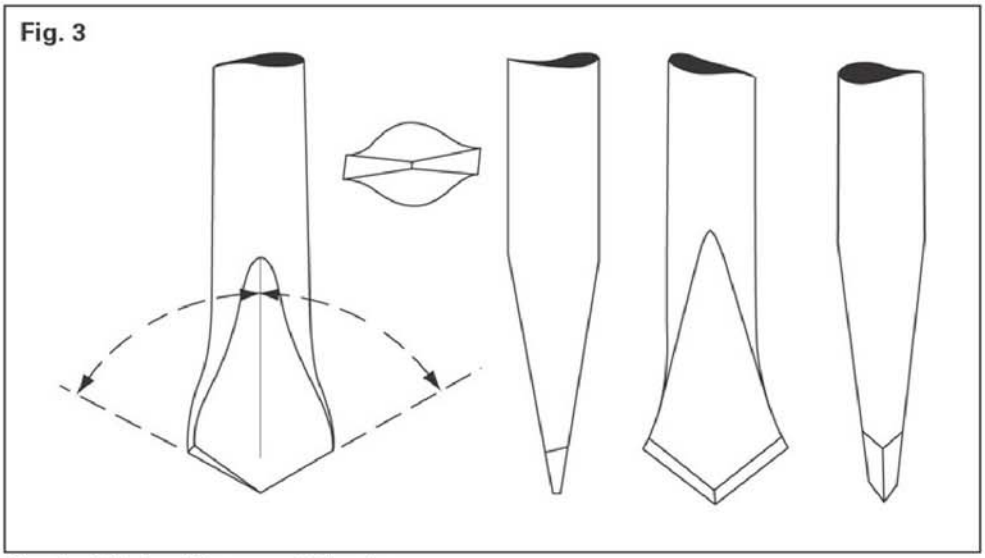
Both horizontal and vertical faces of the grinding wheel need to be dressed and ideally the grinder kept running during the set up to keep the bearings warm. On

Fig. 2



Position of drill in relationship to the grinding wheel.

Fig. 3



Spade drill chart from an old book.

Photo 11, a double diameter form drill, with 45 degree blend face.

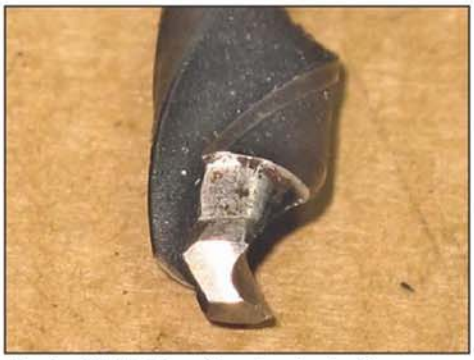


Photo 12. Cutting face backed off.

Photo 13. Web thickness reduction.

Photo 14. Left hand cutting spade drill.

some smaller grinders, this may prove to be impractical. The diamond should remain clamped to the chuck as the wheel may require further dressing during machining. Due to old age, misaligned magnets etc. not all magnetic chucks pull down securely so the spinning fixture should be securely positioned between two parallels or pieces of flat stock, as this will prevent any sideways movement when spinning. If the magnetic chuck has been set parallel to the machine spindle, then the fixture can be set against the chuck's back rail. If not, then a square should be used against a stationary grinding wheel to achieve orientation.

The drill should be positioned slightly to the left hand side of the grinding wheel centre line, **Fig. 2**. Once the length of the pilot diameter has been achieved, a DTI, clamped on the chuck should be zeroed against a convenient flat surface on the grinder. This will act as a visual stop when the wheel is approaching the shoulder during grinding. As in all grinding operations only light feed cuts should be used. No more than a couple of thou on the depth and twenty to thirty thou cross slide advance per revolution.

It is then quite a simple machining operation to grind the pilot to the required diameter. A smooth action is needed during machining when rotating the drill and advancing the cross slide, obviously using both hands.

Any uncoordinated movements will result in check marks and burning. If the grinder has coolant, then this should be employed during machining.

All that is then required, is to reduce the web thickness, grind relief faces to the pilot and back off the shoulder faces, as discussed at the beginning of this introduction, **photos. 11**, **12** and **13**

Spade drills

As it is extremely unlikely that the home engineer would possess, let alone feel the need to use a spade drill, I will only touch lightly on their use, although the history of cutting tools can be an interesting subject in itself.

Until the advent of the twist drill, spade drills were the only means of machining a hole in metal. These tools were forged from a smaller diameter round carbon steel bar and then ground up by hand. The holes produced were neither round, parallel or straight. Today, spade drills are still popular for wood workers, as a cheap and effective method of producing larger diameter holes.

The examples I've included show how things used to be for our forebears. Today, spade drills can still be found in some production environments. The negative form makes them very robust in high speed machining. The geometry of these drills is quite complicated and therefore unsuitable for producing in the home workshop.

The first examples are from a wonderful old book called The Amateur Mechanic. Although twist drills where available at the time, these examples show what a forged spade drill was like at the turn of the 20th century and also goes some way to explaining why the old boys who were still around when I was in my apprenticeship, had one or more fingers missing, **Fig. 3**.

The second is a left hand cutting spade drill, which was produced for a boring operation on an multi spindle auto lathe. Note the negative rake angles, **photo 14.** (I hope at some stage to be able to write about left hand cutting on the lathe, the reasons behind it and useful applications.)

Making a simple D bit

A D bit needs to be produced to machine a 0.1875in. diameter hole with a 60 degree cone blending into a 0.375in. dia bore. The cone is to act as a non-return valve and needs to have a good clean surface finish. The material selected is 0.375in.dia. Silver Steel.

If there are no collets available for the lathe, time should be taken to ensure the sliver steel is running perfectly true in the chuck. Failure to do so may result in eccentric drilling and an oversize hole in the workpiece. If the stock is only running out by a few thou, then a quick method of correction is to gently tap the top of the chuck jaw opposite to the DTI plus reading

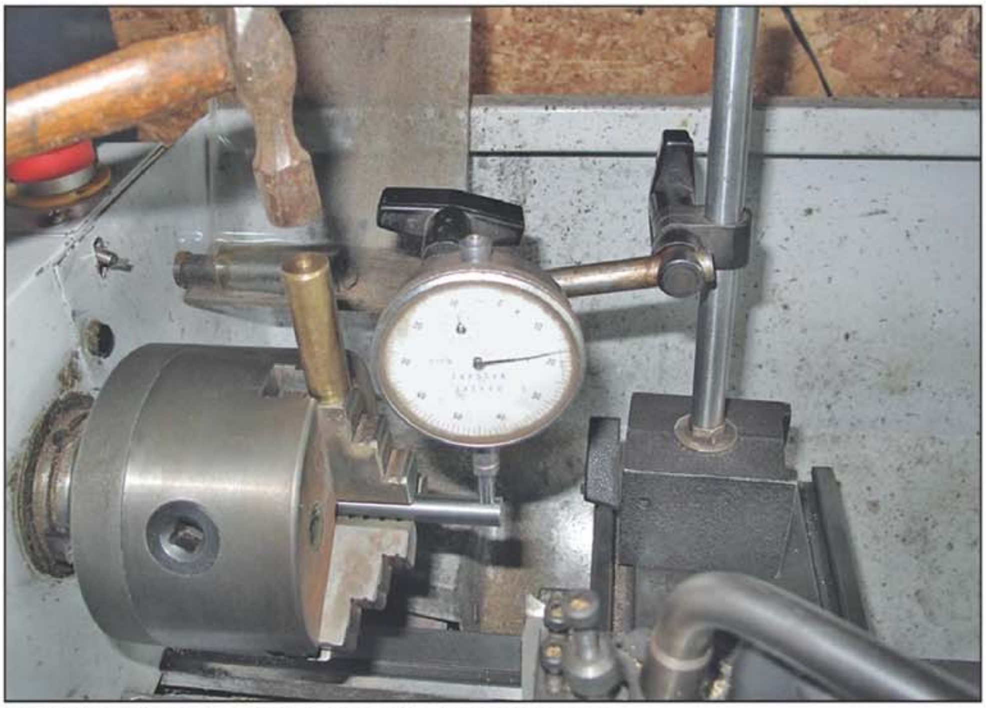


Photo 15. Set stock true, by gently tapping chuck jaws.

Photo 16. Turning 60 degree angles to D bit.

Photo 17. Turning 120 degree angles to D bit.

with a piece of brass or similar soft metal while watching the effect on the DTI, **photo 15**.

A reading should be taken close to the chuck jaws and a second near to the end of the bar. The readings must be identical to insure the silver steel is running completely true.

Being a carbon steel, silver steel is tough to machine so whether using an inserted tool, or a hand ground HSS tool bit, light cuts are recommended. If the machine doesn't have coolant, then a little soluble oil applied with a brush can only help.

Once the diameters and shoulders have been turned, any angles can be machined. In this case a 60 degree included angle to act as a seat for a ball bearing. The lead angle needs to be 120 degrees included angle, the same as a standard jobber drill, **photos 16** and **17**.

The turned diameters need to be dead size. The finished bore size, produced by the D bit WILL be the same as the turned diameter. If turning has resulted in poor surface finish, then a couple of thou should be left on for polishing to finished size. Wrapping emery tape around the turned diameter and pulling it backwards and forwards will only result in two things. Polishing any turning marks deeper into the surface or getting your finger trapped between emery tape and the job. Both should be avoided!

A better method is to tear the emery tape, about 150 grit in a 3/8in. strip, and hold it on a polishing stick. Then with a filing action polish the diameter to size. As with filing, always cut on the forward stroke and lift off, so always cutting in the one direction only. All machining marks can then be removed, leaving a smooth surface finish.

Once a good surface finish has been achieved, the D bit needs to be sawn or milled, to produce the semi-circular form prior to hardening and finish grinding.

Smaller diameters need to be carefully hacksawed, leaving roughly 0.060in. for finish grinding. Being more robust, larger diameters can be milled, leaving 0.030in. grinding allowance.

In this case, the D bit was milled to a couple of thou above half distance for stoning to finished size after hardening and tempering. This process should only be attempted by using a sharp end mill, along with liberal amounts of cutting compound, such as RTD, which is applied during machining, **photo 18**. To ensure the flat is machined completely parallel, the D bit was held in a toolmaker's vice gripped by freshly machined registers in aluminium soft jaws. It is also recommended to have a digital read out on the Z axis as tweaking a thou or two on the last cuts can't generally be guaranteed by reading off machine dials, especially taking any backlash into account, **photo 19**.

After touching on, the cutter was fed down in 0.010in. increments until the milled flat was 0.010in. above the centre line. The finial 0.010in. was removed in three passes. The effect of the previous pass was measured and any deviation from the depth of cut applied was noted. Any deviation was compensated for on the next pass. That is to say, if a 0.004in. deep cut was made, but due to deflection, or slight movement against the cutter, 0.005in. was removed, then 0.003in. would be applied on the next pass to avoid machining below the centre line. The final cut produced a flat measuring 0.002in. above centre, **photo 20**.

Silver steel is a water hardening carbon steel, which needs to be quenched in clean,

Photo 18. Milling D bit to centre line.

Photo 19. Z axis digital read out.

Photo 20. 0.1875in. diameter milled to centre line, leaving 0.002in. for finish stoning.

Photo 21. Tempering colours run from heat source at the base. Cutting faces achieving light straw colour.

Photo 22. Stoning to final size.

Photo 23. Backed off cutting faces.

Photo 24. Proving D bit by plunge cutting

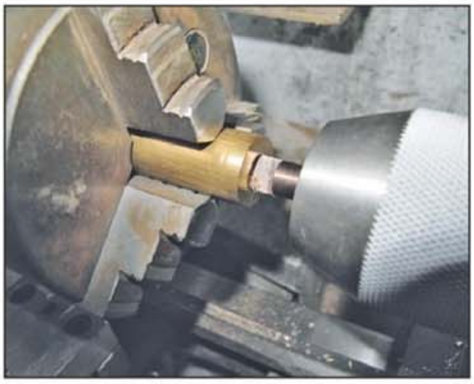


Photo 25. D bit plunged to full cutting depth.

cold water. The book states that silver steel needs to be heated evenly to a cherry red. As this is to be a cutting tool, rather than a hardened component, I tend to heat it to a brighter red. To achieve an even heat, roll the "D" bit backwards and forwards across a fire brick, while playing a blow torch along its length. Care should be taken not to overheat the D section of the bit as burning can affect the surface finish. When the silver steel has taken on an even, bright red colour, hold the bottom of the shank with a pair of pliers and quickly quench, by plunging vertically into the water and cool with a circular motion.

Before tempering, the silver steel needs to be polished back to natural colour. Place the "D" bit vertically on a fire brick and play the blow torch around the base of the shank. When heat has built up, a spectrum of colours will start to travel up its length. At this point, stop heating the tool and watch the colours run. When the business end of the D bit has achieved a light straw colour, quench vertically in water, **photo 21**.

The final 0.002in. can then be stoned off the flat, by holding the D" bit against a fine oil stone and moving it in a circular motion. A little oil will assist the honing, **photo 22**.

For finish grinding, the D bit needs to be carefully set in a Vee block or toolmakers vice and presenting the flat as horizontally true as possible. Gently feed down the grinding wheel to establish contact. Then surface grind to the centre line.

The two cutting faces, 120 and 60 degrees, now need to be carefully backed off by hand, leaving only a witness of the original machining. The two bore diameters should not be backed off, **photo 23**.

All that remains now is to prove the tool by plunge cutting into a piece of brass. Setting the D bit at an angle in the drill chuck will help the swarf to clear. Running the lathe at a speed of 1500 rpm. commence machining, **photos 24** & **25**. Machine as if conventionally drilling a hole by clearing at regular intervals. In brass, the operation can be run dry although if coolant is available it's always best to be used. For all other materials coolant or a cutting compound should be applied during machining, to maintain surface finish.

To check bore finishes, the test piece was milled to the centre line, **photos 26** and **27**.

By using D bits, complicated bores can easily be produced in the home workshop. This method is especially useful when a number of identical components need to be produced.

D bits are also useful counterbores for flat bottoming pre drilled holes for cap head screws etc. Turn the silver steel to the diameter of the hole and face off square. Split to the centre line. Harden and temper as normal. Only the front face needs to be backed off to produce the cutting edge. The benefit of using a D bit, as opposed to a flat bottomed drill, is that the D bit is more rigid. It also fits the bore snugly and does not tend to cut into the sides of the hole. ■

Photo 26. Sectioned view of plunged cut.

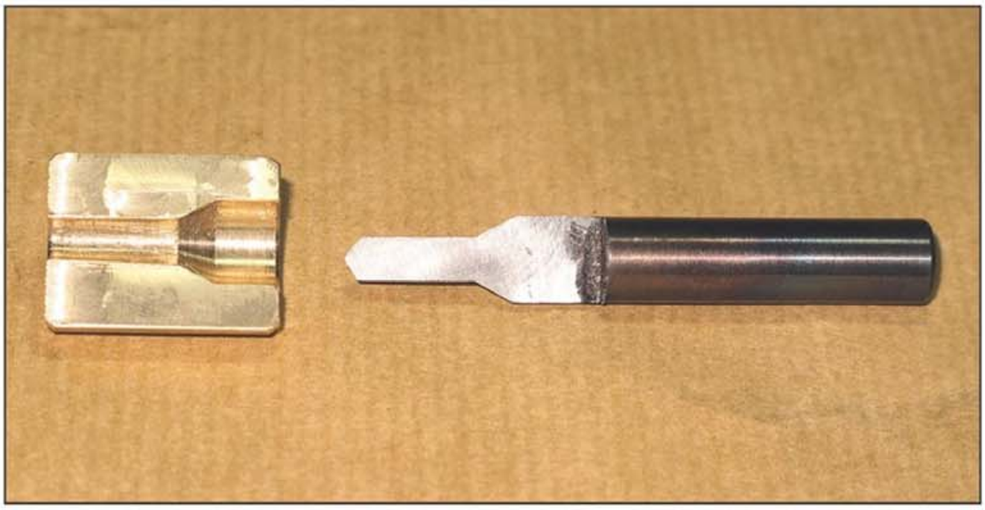


Photo 27. The sectioned cut with the cutter that made it.