

On The Cutting Edge...

Understanding Drill Sharpening

BY MATTHEW J. RUSSEL

Drills are such common tools in our shops that we take them for granted and tend to overlook their complex character. We drill holes with them, we try to keep them sharp, and we utter some not-so-nice oaths when they struggle to make holes by rubbing instead of cutting, because the drill is a cutting device, no different in concept from our milling cutters or lathe tool bits. It is just arranged differently. In this article, I intend to explain some of the geometry of the common twist drill, and give you an understanding of how to sharpen your drills so they can be as perfect a cutting device as possible.

Most drills have two cutting edges, two spiral chip channels, and sufficient body to carry the torque loads without twisting the drill apart in the middle as the loads increase. There are variations: three and four flute drills, and one flute, no-spiral drills made especially for drilling deep holes, such as in making rifle barrels or deep cooling channels in plastic injection molds. We will not discuss those here, but they all have the same two essentials: A cutting edge that must be sharp, and a way to get rid of the chips as they are produced.

Nonetheless, we are stuck with sharpening our drills now and then, and have to learn how to do it satisfactorily. When we do not put a price on our time, the cost of drills can be significant, and extending their life by re-sharpening is a must. Many of us can do a passable job with a drill in hand, a bench grinder, a grinding wheel that is more-or-less straight and true, and a tin can filled with water to keep our fingers from being burned, and incidentally, to protect the drill from losing its temper from the heat of being ground. Actually, that is not a recommended method of cooling, as it tends to produce minute cracks between the grains of the metal. These microscopic cracks can progress under the stress of drilling, and can result in breakage of the drill. Yet, there are many who will buy a new drill rather than try to sharpen a dull one. Attempting to gain long life by using one of the new Titanium Nitride coated drills works until it too becomes dull. Regrinding then eliminates the coating on the critical surfaces, and their "long life" reverts to the same as the common high-speed steel drill.

Sharpening of drills in the home shop is usually a trial-and-error process relying on skill, experience, patience, and too often the ability to finally admit to ourselves that the holes we drill are good enough if they

are only reasonably round and straight. We also need to admit that if we make the holes a little bit oversize, it won't matter after all, and it won't require re-sharpening the drill before every job.

Oddly enough, however, a quick call to three local tool and die shops with excellent reputations brought a similar reaction to my question of how they sharpened their twist drills. The response was universally something like "we have a couple of old timer's who do it by hand at the grinding wheel." None of these three successful shops approached the task with any science, logic, or cost analysis. None had a dedicated grinding fixture in their shops or tool rooms.

Of course, we could go to the catalogs and purchase a device that purports to sharpen our drills for us, but that approach has some dangers in it. I have such a device in my shop. I don't know why I have kept it, except that it once promised to solve all my drill sharpening woes, and maybe I had an unshakable idea that if I only could learn how to use it properly, I might at last be satisfied with my \$6.95 purchase from Sears, Roebuck and Company in 1949. I should have tossed it years ago, because I know now that it was based on faulty implementation of a not-too-good theory and incorrect geometry, and will never be able to do what its manufacturers said it would. This device, originally produced by Greenline, and its identical multi-branded progeny, are still being sold today in many tool catalogs for much more than \$6.95, and they still won't sharpen a full range of drills except by trial and error and specialized touch-up corrections.

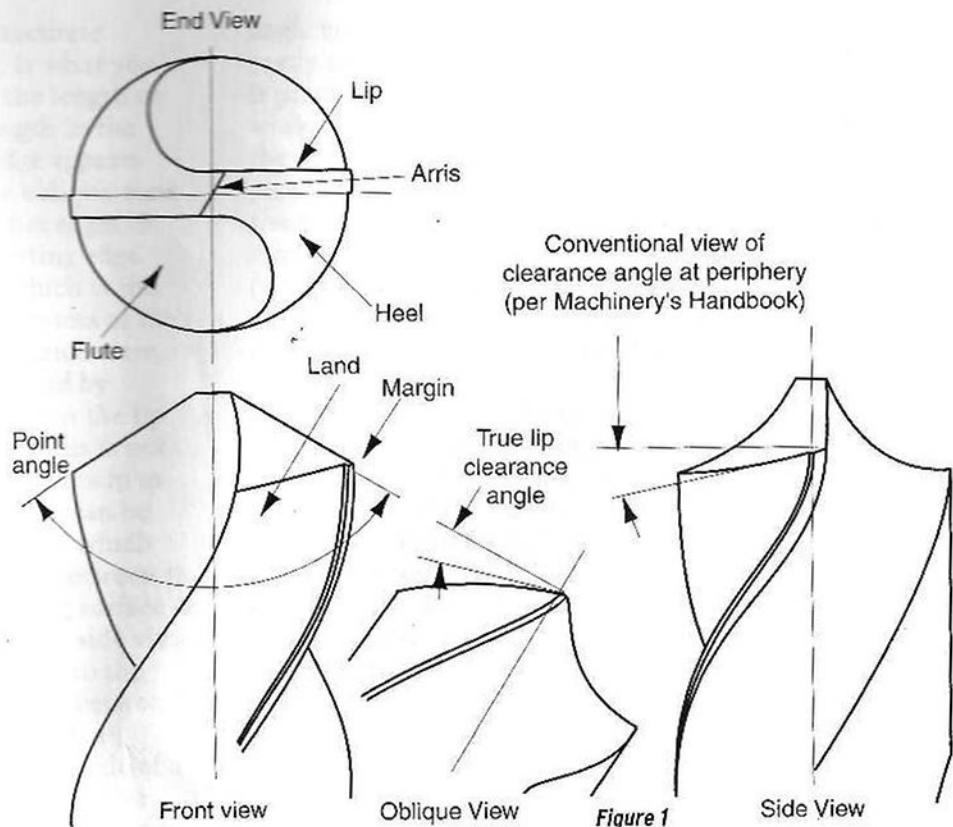
Interested in what the history of drill grinding fixtures might be, I pulled an old McMaster-Carr hardware catalog from my bookshelves and found an assortment of drill grinders to sharpen drills from No. 80 up to 2-1/2" (but not all in one device!). Each of these fixtures held the drill at a fixed angle to a grindstone, and most arranged to swing the drill through some sort of arc to grind the point angle and lip clearance angle in one step. All of them required the drill to be removed from the fixture and repositioned in order to grind the second cutting edge. There was a noted attempt in some of the devices to make that transition repeatable using a variety of stops, clamps, micrometer positioning devices, and dial indicators. Clearly, the accuracy of grinding the second cutting edge identical to the first was recognized as an important part of good drill

sharpening. It still is. If the cutting edges are different in any way, the drill will tend to cut with one edge more than with the other. This results in eccentric torque loads, which leads to the drill wandering and drilling a crooked hole, accompanied with an increase in the diameter of the hole. It also increases the side forces on the drill and could cause drill breakage, especially in small sizes.

The simplest of drill grinders in my old catalog was a hand-held fixture with an assortment of bushings to hold drills from No. 41 to No. 60, and a small India oil stone to do the sharpening. The fixture helped guide the oilstone at some fixed angle with respect to the drill point. The most expensive grinder shown cost \$198.50, including a motor. It offered a "patented feed system" and an adjustable end stop to assure even grinding of the second cutting edge. It would grind two- and four-flute drills, "sharpening all sizes of drills in accordance with recommended specifications for heel clearance, chisel or web angle, and cutting lip angle." The photograph in the catalog revealed at least six adjustments that must be made, presumably with each different size drill that was to be sharpened.

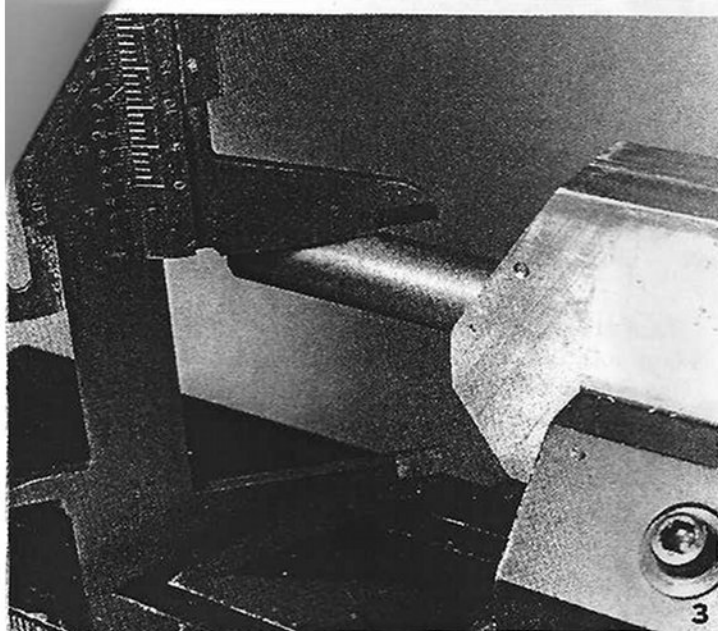
Multiply these 1950 prices by about ten or so, and you will arrive at a point near the low end of the range of costs of today's professional drill sharpening devices. These devices vary from mere toys to truly professional tools that most of us will never afford to purchase for our shops, although compared to prices of the new lathes and mills we buy, good drill sharpeners ought to be found in more home shops than they are. Today's professional drill grinding machines still use a mixture of techniques: cylindrical, conical, and helical grinding of the heel. Some also add the ability to grind split points, and a variety of other specialized point shapes and angles. Probably the best and easiest to use is the *Drill Doctor* (a simplified version of the professional *Darex* fixtures that cost over ten times as much), available in several models from about \$69 and up. They all use a diamond-coated grinding wheel and produce a helical point through a manually driven, cam-coordinated rotation and advance of the drill against the grinding wheel. The less expensive models do not provide for more than one point angles, and cannot do anything but helical grinding of drills from 1/8" to 1/2". For larger drills, you must purchase an additional self-centering chuck. For other point styles or angles, you must purchase a more elaborate fixture and spend more money.

But let us get back to science and theory and geometry – and the fundamental requirements of drills. Figure 1 shows the conventional drill geometry (although somewhat exaggerated for clarity of fine



details). At the top is an end view looking down the shaft of the drill, seeing only the cutting end. It shows the two cutting edges, the two flutes, and the two lip clearance surfaces, called the *heels* of the drill. At the center is a curiously slanted line that denotes the juncture of the two lip faces called the *arris*, or *chisel edge*. We will elaborate on this feature later. Note that the two cutting edges are not coplanar. They are each offset from the central diameter. This feature, which is vital if the drill is to have any strength at all, is the thing that causes no end of difficulties in grinding cutting edges – and is a major cause of difficulty in drilling round holes.

Below the end view is a front view that shows the two cutting edges in true length and shape. The *point angle* is shown. The point angle is the total included angle between the two cutting edges, but often it is specified as half of that angle, or even sometimes the complement of half the angle. The chisel edge at the center is revealed only by a very short horizontal line at the top. That line is not shown in true length here, as this view presents an oblique and thus foreshortened view of the chisel edge. In the end view above, it is shown in true length and position. The front view also shows the narrow *margins* at the advancing edge of the *flutes*. They do not affect how we grind the cutting edges, but they do establish the drill's diameter and help guide the drill through deep holes. The margins are not cutting edges, as they are in end mills. They may burnish the hole slightly, but they are there only to provide guidance with minimal surface contact. The wider surface adjacent to the margin is called the *land*, and it is made to be slightly smaller in diameter than that of the margins.



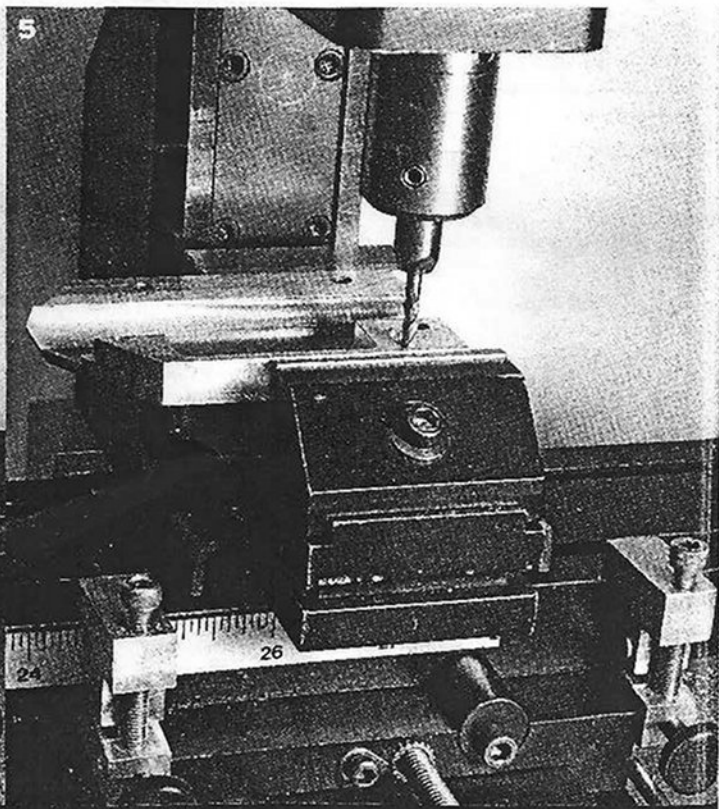
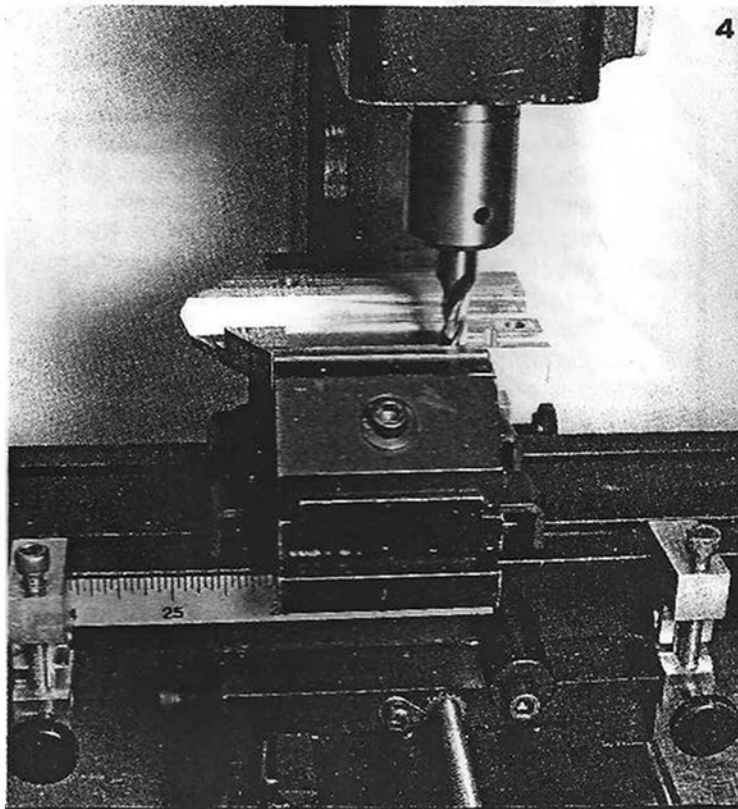
to the top of the rod. Now, remove the test rod and make a skim cut on the exposed upper edge of the part, just enough to make the entire edge planar. Next, turn the part over in the mill vise and take a similar skim cut to make the other edge parallel with the first edge, which should be resting squarely in the mill vise. You can now use those two parallel edges to clamp the part in the mill vise and square up the blunt end and take a skim cut on the pointed end to make the parts match exactly all around. Add the 30° side bevels at the pointed end. (The end surfaces are not important, but are trimmed just to make the part look professional and to keep it from interfering as you grind the end of a drill.) Deburr all outside edges. The 30° side bevel is not critical, and can be done on a disk sander by eye.

The final dimensions (width, thickness and length)

of the assembly are not nearly as important as the parallelism of the two edges and the two flat surfaces with the orientation of the V-groove. If there are errors here, the drills being sharpened may not be ground symmetrically, which is our major objective.

Now, take the assembly apart and proceed to cut the cavity for the clamp assembly in both parts. The objective is to make the cavity slightly over 0.500" in total width to contain the clamping parts regardless of the depth of the V-groove. Be especially careful about matching the sloping side of the two halves, as that surface is used to cam the clamp upwards as the clamping screw pushes it toward the pointed end of the fixture. You may have to assemble and disassemble the fixture several times to check the match between the two surfaces before you are satisfied. Clean out the area where the clamp goes, using a .250" diameter or smaller end mill to make the radius in the corners. The cavities in the left- and right-handed parts must match. Note here in Photos 4 and 5 that I used bed clamps to make sure I didn't over-cut the cavity, as it takes many passes to clean out the part to the required depth and with bed clamps you don't have to watch the dials. The clamp parts are shown in Figures 9 through 12 and should be made to the dimensions shown then fitted carefully so they slide freely together in the cavity.

To avoid drilling a hole for the clamping screw and tapping it on the interface surface of the two halves, a hole that would certainly tend to wander or even break the drill, tilt the headstock 45° to the left and cut a round bottom groove with a .187" diameter end mill, widening it sufficiently to clear the screw. The hex nut gives you some steel threads for the clamping screw to work with, prevents wear on the aluminum body parts, and also provides non-critical alignment for the clamping parts, since the hex nut is intended to be slightly loose in its



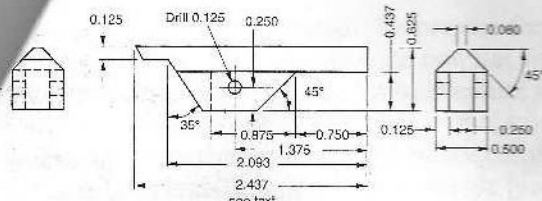


Figure 9

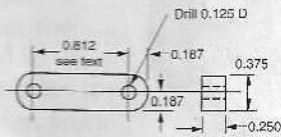


Figure 10

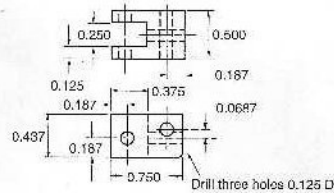


Figure 11

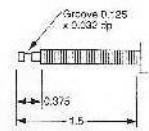
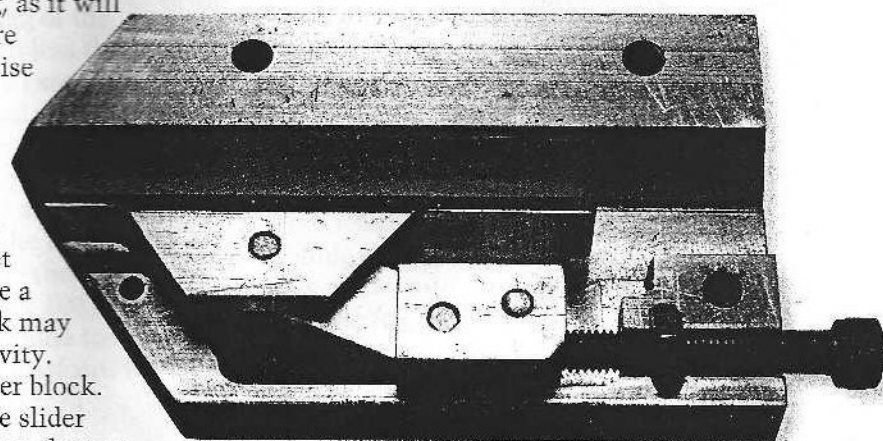


Figure 12

pocket (but fitted so it won't rotate). The clamp is made from aluminum. Cut to shape, mill the 0.250" slot, make the angle cuts, but leave the left end a bit long, as it will be trimmed to shape later. The beveled cuts are easily made by clamping the part in the mill vise and tipping the headstock to the right at 45°. Cut the right side of the roof with the end of the end mill, and then cut the left side with the side of the end mill without having to reposition the headstock. The clamping screw in Figure 12 is made from a 1-1/2" 10-32 socket head cap screw with the end grooved to receive a holding pin in the slider. The length of the link may vary depending on how your parts fit in the cavity. Assemble the clamping screw, clamp, and slider block. Slide the clamp to the lower left corner and the slider block to the right. Measure the distance between the two 0.125" holes to determine the length of the link. Mine came out to be .835". Yours may be slightly different. Cut the link and make the final assembly with pins. You will need to make three pins, .125" diameter by .500" long. There is no need to fix the hinge pins or the holding pin, as they are all captured at assembly and cannot come loose. Test the action of the link, and when the clamping screw is able to move the clamping bar freely across its full range, add the top piece and fasten the assembly together with four screws. You may have to round off some corners and edges and sharpen the radius of the corners in the process to allow the clamp parts to achieve a full range of motion as shown in Photo 6.

Check again to make sure the clamp is free over its range. The pins may catch and prevent free motion.



6

The last check is to insert the half-inch rod again using the clamp you just assembled to hold it securely in the V-trough and lay the assembly flat on the mill table or a granite surface plate. Measure from the surface to the rod at both ends. You should find the rod to be exactly in the center as measured at both ends of the assembly. If not, then a very slight skim cut may be in order on the broad surfaces of the assembly, but only if the rod is excessively skewed. If your measurements at each end differ by less than .015" over a span of the rod of about 4", then your drill's cutting edges will differ only by about .25°, which is tolerable. More than that calls for a very slight skim cut to bring the surfaces of the fixture into

parallelism with the V-trough. To prepare for this, clamp the assembly in the mill vise with one flat surface upwards and clear of the vise jaws. Measure from the mill table to the bottom of the 1/2" rod clamped in place and shim the part as necessary to make the distances the same at both ends. Clamp the vise firmly (you can leave the rod in place as long as it is clamped securely in the V-trough) and take a minimal cut on the upper surface. Invert the part and clamp it again, this time resting it in the mill vise on top of a pair of parallels, and take a minimal cut on the upper surface. This brings both surfaces into parallelism. Recheck the centering of the rod to make sure you have brought the skew to less than .015". Tighten the clamp fully and trim off the extended end of the clamp, bringing it flush with the end of the body. Once you are satisfied, it is almost time to start creating split point drills.

One last thing must be done. The optical device shown in Figure 13 is important in aligning the drill in the fixture. The dimensions depend largely on the size of loupe you intend to use. It should be at least 10X, and an inexpensive one can be found in most machine tool catalogs. The spacer is intended to provide a platform to place the assembly on for checking. The important thing is to make the optical axis coincide with the center of the V-trough when it lays on the platform.

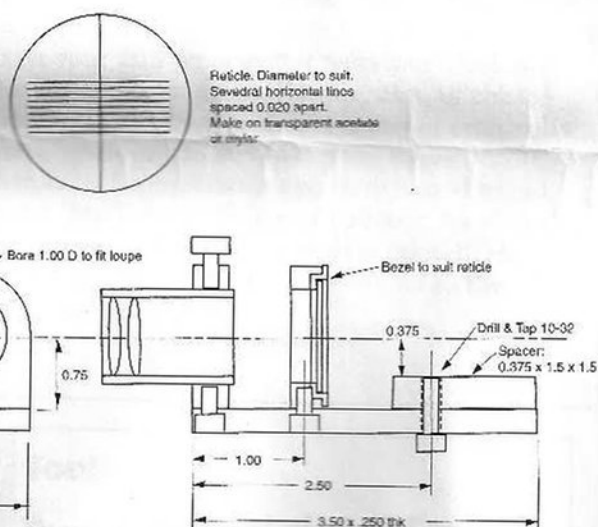
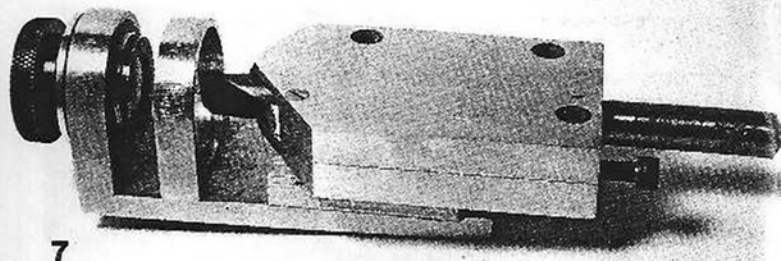


Figure 13

To use, insert and clamp a drill in the V-trough, tightening the clamping screw lightly. The end of the drill should be close to the pointed end of the fixture, protruding just enough to be sure you will grind the drill and not the fixture. You don't have to measure anything. Orient the drill so the two cutting lips are exactly horizontal as compared with the horizontal lines of the reticle, as shown in Photo 7. If you err in alignment at this point, the two cutting edges will be incorrectly and unequally ground. Rotate the drill until the two edges are both horizontal and clamp them securely, making sure the drill is not disturbed from its position as you tighten the clamping screw.

Tilt the sanding table 10° downward. Set the miter gage to 31°. (You may prefer to set the miter gage at 22.5°



to create a 135° drill point.) Place the fixture's broad face on the sanding table and against the miter gage riding in the slot in the table. Using at least an 80-grit Aluminum Oxide disk, gently touch the point of the drill against the rotating sanding disk while sliding the fixture smoothly back and forth across the sander table. For smaller drills, a better choice would be a 120-grit disk. Inspect your grind often with the loupe. The flat you are creating should not exceed the width of the margin of the drill, but it should be slightly more than one half the web's thickness. Now, turn the fixture over. *Do not remove the drill to rotate it.* Since the V-trough operates in the horizontal aspect, you don't have to worry about the position of the drill changing. It is still the same height off the sanding table and the two cutting edges are still horizontal. Now grind the second cutting edge. When properly and equally ground, the two cutting edges will meet in the center, creating a very short aris or chisel edge. If you grind one edge more than the other, the two edges will meet, but *NOT* in the center. If you are satisfied at this point, set the fixture aside *but do not remove the drill.* Tilt the sanding table to 30° down as shown in Photo 8. Place the fixture back on the table against the miter gage as before, and slowly grind the second relief surfaces, inspecting often with the 10X loupe. You can do one side first and then the other or you can switch sides frequently. The object is to make both second surfaces *also* meet at the center of the drill, forming a very pointed drill with no conventional chisel edge. When complete, the drill should look like the one

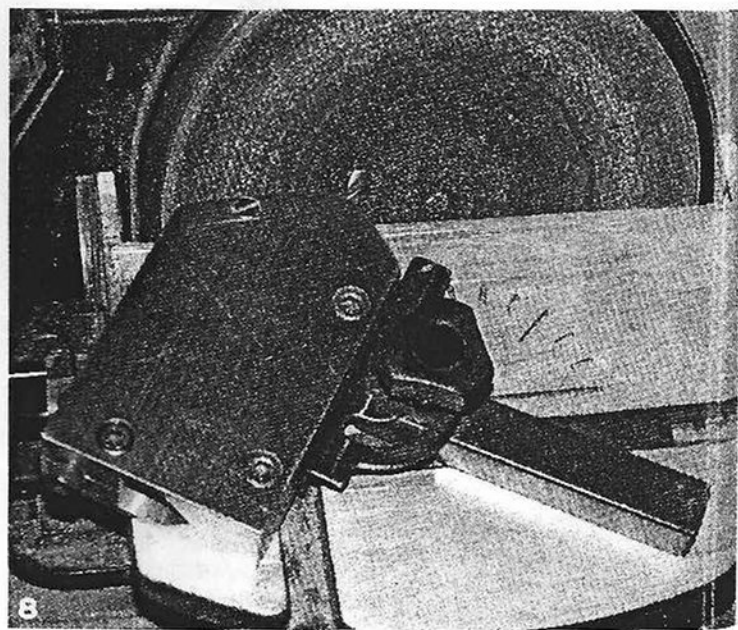
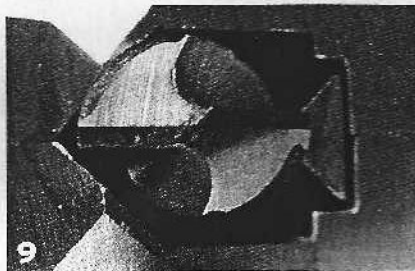


Photo 9 – all four facets meeting at one common point. If you err, you can touch up or start over. Note that if you grind a lot of metal away, the lip edges will rotate slightly, disturbing



the alignment of the cutting edge with the fixture. If you must grind a lot in order to get rid of a nick or chip, then reposition the drill to bring the cutting lips to the horizontal before the final touch up of the four facets.

Although this method does not rely on the precision that a Quorn would provide, it is simpler and, if some care is paid to inspection and alignment, will produce split point drills that are very efficient in drilling round holes. Admittedly, this method of grinding drills seems more tedious and more demanding than just blindly swinging a drill in some uncertain fixture that once deceptively promised to solve all your drill grinding problems. Your new split point drills will cut better, will require less pressure, and will stay sharp longer. They also will cut *round* holes that often do not require a center drill or even a center punch. Once you use your new split point drills, you may never go back to cylindrically ground drills! Remember: the easiest way to drill a triangular hole is to not clamp the piece being drilled! Treat your drill press as if it were a vertical mill, and put the workpiece in a vise. Beware, also, of cheap imported drills. They often have misshapen or erratically formed flutes, resulting in curved cutting edges.

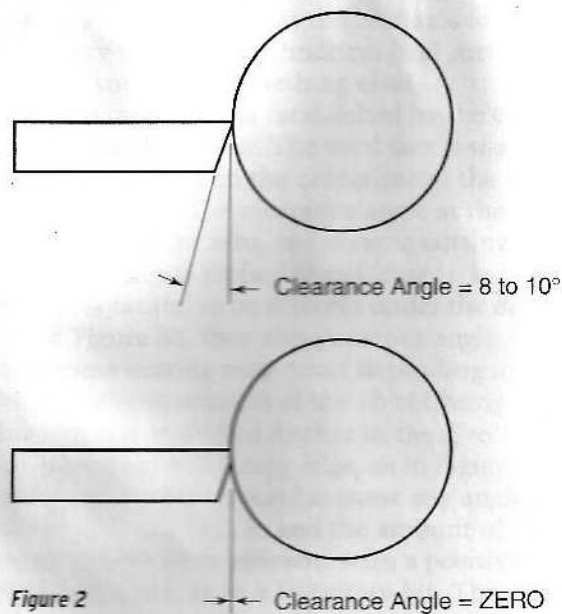
My thanks and appreciations go to my good friend and colleague, Mr. John E. Morse of Fairport, New York for supplying some of the historical data quoted. His inspiration and encouragement urged me on to this article's completion.

Photos and drawings by Author

The third oblique view, made using accurate graphical projection drawing techniques, is what you would see if you lined yourself up along the length of the cutting edge that is shown in true length in the front view. Consequently, that cutting edge appears in the oblique view as a single point. The oblique view shows the cutting edge on the near side, the edges of both flutes, and the margin of the near cutting edge. Also labeled is the true clearance angle, which is the primary reason for showing this view. Diagrams of the clearance angle are customarily drawn in a side view, that is, to show the clearance angle as defined by *Machinery's Handbook* as "the angle between the lip and a line at right angles to the drill axis." This is not a completely precise definition, and it doesn't help in measuring the angle because the angle's apex can be found at any point along the length of the lip, which is a sloping surface, and the angle measured between the horizontal and the tangent so that the sloping surface is indeterminate and cannot be measured in the side view. The angle should be measured at right angles to the cutting edge. The clearance angle is the angle between the cutting edge and the material being drilled, a definition that matches that of the clearance angle of a lathe tool bit. The definition is thus better illustrated in the oblique view, in which we look along the line of the cutting edge and see a nearly true view of the clearance angle regardless of the position of the apex along the cutting edge. What we don't see is the helically elongated and truncated cone that is swept out by the cutting edge as it revolves and cuts new material, so it is difficult to show the true clearance angle in any view.

A clearance angle is necessary in all cutting tools. In lathe tools, a clearance angle of 8° to 10° is adequate, depending on the material we are turning, to make sure that the cutting edge is given a chance to cut and not rub on the material it is cutting.

Lathe bits are sharp edged, as shown in Figure 2. A clearance angle is created by the cylindrical work object receding away from the sharp-edged tool. The



angle between the point and a tangent to the work is easily recognized and measured. As the lathe tool wears, it presents a flattened surface against the round workpiece, and it no longer cuts. A tool cuts by shearing the metal away, resulting from the tool applying pressure against the work on an area small enough that the pressure per square inch is greater than the shear strength of the material. As the tool wears, the area in contact increases, and the only way to exceed the shear strength of the material is to apply more tool pressure. A quick look would make you think that the clearance angle is still as you first ground it, but the tool is now merely rubbing against the workpiece, and the heat generated increases greatly, as does the torque required. Cutting of some softer materials, such as brass, often calls for a negative rake tool. Rake is the angle between the tangent to the material being cut and the leading edge of the tool. The clearance angle is on the other side of the action. Even negative rake tools for brass require a positive clearance angle. Lathe tools often include a slight rake on the top surface as a means of directing the chips away, or to cause them to break up periodically so they won't form dangerous yard-long spirals.

In a drill, the clearance surface recedes from the work material in the same way as it does with a lathe tool. The entire cutting edge is expected to cut. Unfortunately, this is not without difficulty and compromise. As one advances the drill into the work, the outside corner of the cutting edge follows a helical path defined by the rotational speed and the speed of advancement. The inside corner of the same cutting edge is forced to advance the same amount, resulting in a greater helix angle, but it is doing its work without the benefit of increased lip clearance. The very center of the drill does not cut at all. **The chisel edge merely pushes metal ahead of it**, wearing it away, or pushing it into the path of the cutting edge. The material being drilled becomes rather plastic in this area as a result of compression and bearing failure. Because of the characteristics of "stick-slip," common in all dynamic friction, the material at the very center of the new hole does not move away from the advancing chisel edge uniformly, but in a series of jolts and starts. Inspect the bottom of a new hole and you will seldom see a perfectly formed conical array of circular trails left by the microscopic roughness of the cutting edges, but a wavy pattern approaching a polygonal shape instead of a pure conc. This is evidence of non-uniform cutting action, and of the intermittent action of the chisel edge. The drill is subjected to cyclical stresses, a rotary vibration of sorts, that create holes that are not round and produces additional torsional stresses on the drill. We hear the chatter, and slow the speed and increase the feed, but the holes are still not uniform or round. Ample industrial research has shown that the action of the drill in the hole supports my simplified explanation of what is going on.

Since the efficiency of cutting by the inside of the lip is reduced, the result is that it wears a lot more than the outer portion of the lip, and it generates more heat.

With a small clearance angle, the cutting edge is restricted in taking a bite out of the material being cut so it cannot be advanced into the work rapidly, which means that the inside of the lip is holding back the outer lip's advancement. The outer lip is not able to cut efficiently, either.

The entire purpose of grinding a drill is to create a cutting edge and a relief surface that is out of the way of the cutting edge so the body of the drill won't rub on the bottom of the new hole and prevent the drill from being advanced.

The lips should carve out a continuous chip of material of uniform thickness that climbs up the flute and out of the hole where it can be discarded. To cut with only a portion of the cutting edge will soon result in an end to all cutting, as the non-cutting portion, including the chisel edge, will simply rub and prevent the drill from advancing any further into the hole. This is the most common result of using an improperly sharpened drill. It just goes round and round, makes a lot of heat, and does not cut. After trying to make a hole, the dull drill is usually rounded off at the point (sometimes grossly, depending on the cutting pressure, the hardness of the material being drilled, and your insistence). The cutting edges are rounded over or nicked, the outside corner of the lip becomes dull, and the drill point may also have lost its temper from overheating. The chisel edge has tried (and failed) to wear away the central portion of the new hole by rubbing and pushing, not by cutting.

The geometry of the chisel edge, a result of conventional drill manufacture and the fact that the drill requires some material at the center to provide strength and stiffness, is totally wrong for a truly efficient cutting action. It is a fact of life, and we put up with it.

Most simple drill grinding devices use a singular rotary motion of a V-trough fixture to swing the drill against a grinding wheel to cut a radius in the heel area. The radius is a simple way of creating clearance so the end of the drill won't touch the hole except at the cutting edges. Now we should ask questions: What is the proper radius? Where is the best location for the center of that radius? What is the best axis for swinging? Should we try to create a cylindrical heel surface or a conical heel surface or something else?

The clearance angle is established by the tangent to whatever radius is chosen. The vital fact is that if the center of the radius is on the centerline of the drill, as shown in Figure 3a. The clearance angle at the cutting edge will be less than zero, and no true cutting will occur. If the radius is shifted ahead (that is, in the direction of rotation) to be directly under the cutting edge, as in Figure 3b, then the clearance angle will be zero, and some cutting may occur depending mostly on the material characteristics of the object being drilled. If the radius center is shifted further in the direction of rotation ahead of the cutting edge, as in Figure 3c, the clearance angle can be ground at most any angle desired, depending on the radius, R, and the amount of the shift, S. Cutting will be more efficient with a positive clearance angle, just as in a lathe tool bit. There are

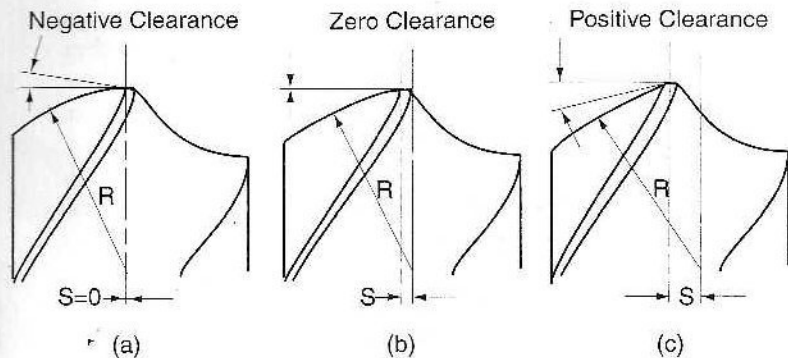


Figure 3

limitations to this, since if the shift is excessive, the shape of the rest of the heel may be adversely affected, and the lip strength reduced.

The \$6.95 device that I purchased in 1949 has a fixed swinging center about 1/8" in advance of the centerline of the drill as established by the V-trough the drill is held in (similar to Figure 3C), and thus it unchangably creates a very large clearance angle for small drills and a very small clearance angle for large drills! The important distinction of most simple drill grinding devices is that there is no way to adjust the radius, and there is likewise no correspondingly correct shift of the center of that radius. Thus, fixture simplicity certainly does not lead us down the road to good control of our re-sharpening experiences.

In an August 1913 article in *The Model Engineer*, L. A. Van Royen proposed to swing the drill against the grinding wheel in a conical path. Some limits were placed on the angle used, and on the position of the center of the cone, depending on the diameter of the drill (Figure 4). With a conical method, if a small drill is placed near the apex of the cone, it will receive a very small radius of curvature, and if a larger drill is placed further away from the apex of the cone, the drill will receive a larger radius of heel curvature (Figure 4).

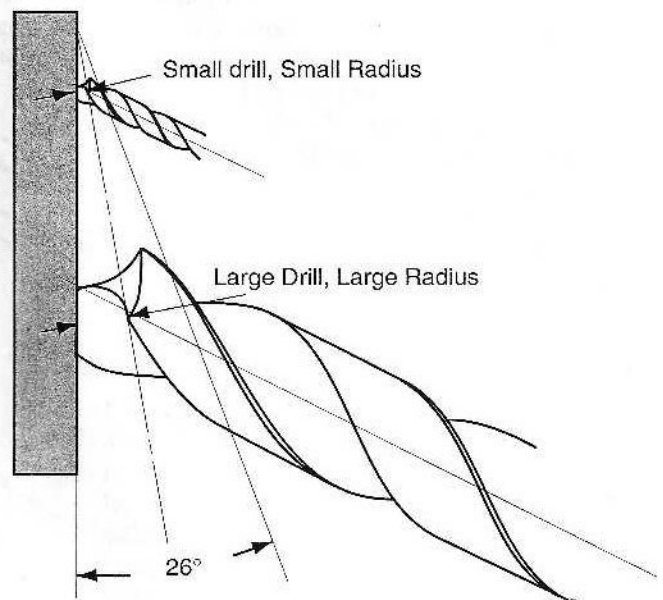


Figure 4

Each size drill could thus receive an appropriate curvature based on its size. The article included geometry that demonstrated that a fixture could be made that automatically positioned the point of the drill at the proper position within the conical swing surface depending on its own diameter. This method was further refined in a series of subsequent articles in *The Model Engineer*, October 1927 by Col. H. S. King, and by another author who called himself "Duplex" in the November 1951-January 1952 issues, both of whom described an improved fixture that provided the conical motion and that also included a feature that made the fixture adjust itself automatically for the lateral shift, as taught by Van Royen. (Actually, the method was not totally automatic, but still involved some manual setup procedures for each size drill.) Based on Van Royen, and corroborated by experimental use by Duplex, the best radius of curvature turned out to be 0.938 times the drill's diameter with the point of the drill with the point of intersection of the drill with the grinding wheel 3.0 times the diameter of the drill below the apex of the cone. That is the true value as developed by the geometry of the setup, but Duplex recommends a distance of 1.4 D. The lateral shift was quoted in the original article as being equal to $D/13$ ahead of the cutting edge, but this was questioned by Duplex, who found through experimentation that this figure should be closer to $D/4$ ($D/13$ would have produced a clearance angle of about 3° , whereas $D/4$ would produce about 10°). Drawings and full descriptions of the "Duplex" fixture (Figure 5) were presented in the November 1, 15, 29, December 13, 27 1951, and January 10 and 24, 1952 articles of *The Model Engineer*.

The choice of cylindrical versus conical grinding surfaces is a matter of simplicity versus accuracy to theory. I have nothing against the simple cylindrical swinging fixtures, provided that you let the drill extend

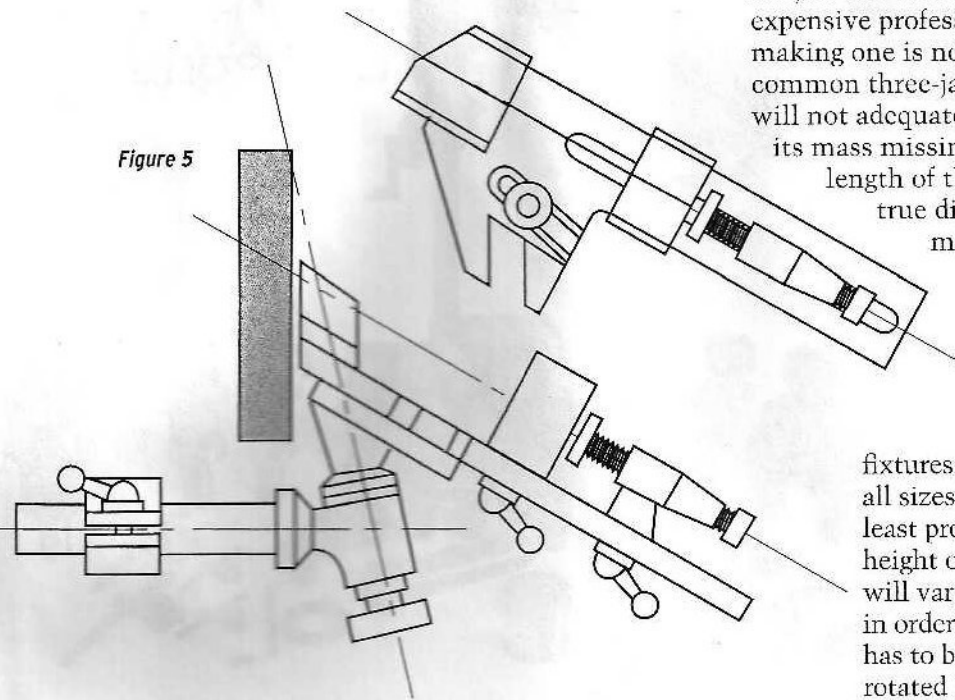
a proper distance from the pivot, and that the swinging radius and lateral shift are known, fixed, and adjustable within a range that produces satisfactory clearance angles. However, none of the instruction sheets I have seen, nor any recent articles I have read, give any help in determining these factors, nor do they even come close to explaining what is going on. These fixtures still rely on a lip stop, and the drill must be rotated 180° in order to grind the second lip.

Lip stops have also been addressed in various *Model Engineer* articles, and they vary from a simple piece of thin metal that projects into the flute space, and to which the drill's cutting lip must be rotated to register against, to a fixture with a small diameter wire stretched across the center of the drill so that the drill can be aligned by eye through a centered peep hole. Neither of these techniques is without problems, and the general consensus is that a self-centering device would be much better than having to unclamp, rotate, and re-clamp the drill for the second cutting edge, provided that it is accurately made. Darex, a commercial supplier of drill grinding equipment, for instance, uses self-centering four- or six-jaw chucks and a two-fingered lip stop that is adjustable for the diameter of drill, applying the lip stop as a gage against both cutting edges at the same time, thus canceling out any errors. Using a lip stop with drills smaller than about $1/8"$ is questionable, since the flutes are very small, the lip stop even smaller, and the whole process becomes delicate and more subject to human error. An optical method is definitely called for here.

For holding the drill, the collet technique seems to be the most popular in the literature, but several collets are required to cover a range of drill sizes. Making collets is a very precise job since it involves drilling and boring very deep holes in which the concentricity of the hole with the outside diameter is paramount. Four- and six-jaw self-centering chucks are used in the most expensive professional drill grinding equipment, but making one is not easy and buying one is expensive. A common three-jaw chuck, such as used in drill presses, will not adequately center a drill that has nearly half of its mass missing. The jaws simply do not span enough length of the drill to bear on enough points on the true diameter of the drill established by the margins, and the drill ends up being clamped askew. Most three-jaw chucks also do not include a through-hole for the body of the drill to pass through so you can grip the drill near its pointed end.

Most home shop drill grinding fixtures rely on a V-shaped trough that supports all sizes of drills. But in doing this, although it at least provides three-dimensional guidance, the height of the drill above the bottom of the trough will vary with the drill diameter. Consequently, in order to grind the second cutting lip, the drill has to be removed from the fixture, manually rotated 180° , and re-clamped, presumably against

Figure 5



...nc sort of lip guide. This introduces **considerable error**, and the chance of grinding both lips to be **geometrically equal** is diminished severely, but the **chisel edge of the drill** still has to rub the material away so the **cutting edges** can dig in and cut.

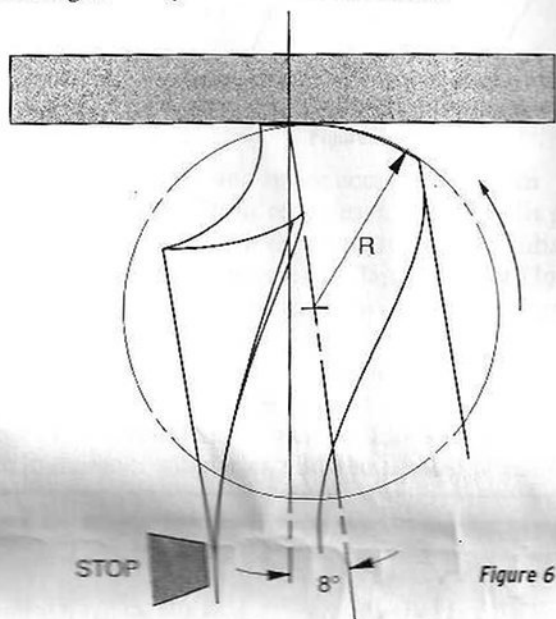
Visualize this typical scenario of **drilling a hole in a piece of aluminum or steel or what-have-you that is harder than cold butter**, keeping the previous discussion in mind, and you can see what has been troubling your **drilling experiences**: You center-punch the hole, then **drill a pilot** with a center-drill. Then you chuck up the intended drill and start drilling. The first few thousandths of an inch proceed easily. Two perfect curls of metal come out of the new hole, but then the pressure required suddenly increases. From there on, the curls are produced more tediously. One curl may disappear completely, and chips tend to break up and get clogged in the flutes. What is happening is the first part of the hole is being drilled only by the outer edges of the cutting lips. The center-drill pilot hole has helped hold the drill on center, as its shape has provided a conical sink to help guide the drill. The chisel edge is not touching anything, as the center-drill produced a cavity in the center of the hole. Suddenly, however, the chisel edge touches virgin metal, and now it must *push* metal at the center of the hole away and into the path of the cutting lips, and the pressure required increases dramatically. At this point, the drill is subject to stick-slip friction, and may develop torsional vibrations, which are retarded as the chisel edge is forced to push metal ahead of

itself, then suddenly gulping up new metal as it flies forward when the stresses are higher than the metal being drilled can stand. The material being drilled resists and the drill is suddenly forced upwards out of the hole for a few thousandths of an inch, and it then tries to take another bite as the drill pressure pushes the drill back into contact. The vibration is uncontrolled, and the whole drill oscillates, both torsionally and axially. The outer lip edges may be flung into the sides of the hole, and that adds to the vibration as they stick, then gouge metal away. The result often is a hole that is not round, is oversize, and may have wandered off the intended position. All of this is operating on a minute, almost microscopic, scale, of course, so you can't see what is going on, but serious research has produced corroborating evidence of what is really happening. Unfortunately, this scenario is more common than one would want to admit.

The concept of swinging the drill about a vertical axis that is both a certain distance away from the drill point and a certain distance laterally away from the drill's axis is valid **ONLY** if we concede that the drill will be swung freely about the vertical (or even tilted) axis, unlimited by position or amount of back and forth motion. This is the foolproof premise in which the craftsman can make no mistakes if he swings the drill too far in either direction. Accepting this premise forces us into complex geometry that provides a clearance angle *only* by shifting the pivot axis laterally the proper amount, which complicates the fixture. The simple

ature compromises by not providing this shift, and is therefore wrong most of the time.

But, shift your mind and allow the drill fixture to be pivoted only a *fixed* amount. Figure 6 shows a drill oriented 8° from the perpendicular to a grinding wheel as in a typical V-trough fixture as viewed from above. The vertical cutting edge is shown as a point. The relief surface is depicted as the curved line to the right of the cutting edge, and it shows the shape you would get if you rotated the drill fixture counterclockwise with respect to the grinding wheel, with the pivot axis on the centerline of the drill. This is exactly the shape we would like to have. It is, incidentally, identical to the intuitive method of grinding drills by hand with no fixture.



Achieving it is superbly simple: **Limit the rotation of the drill-grinding fixture so you can never fail to provide a proper clearance angle at the cutting edge.** The pivot point can be directly below the centerline of the drill, which is better than the Duplex method, since the 8° is constant regardless of the drill diameter. The drill will have a flat at the cutting edge, and the width of the flat area depends on the clearance angle you choose. The larger the angle, the wider the flat will be, but with the ordinary range it should not produce any problems. Building in the ability to change the radius and adding the stop complicates the fixture and the pivot structure must be below and well clear of the grinding wheel. The simple V-trough fixture cannot easily provide this adjustment. But wait! There is even a better solution.

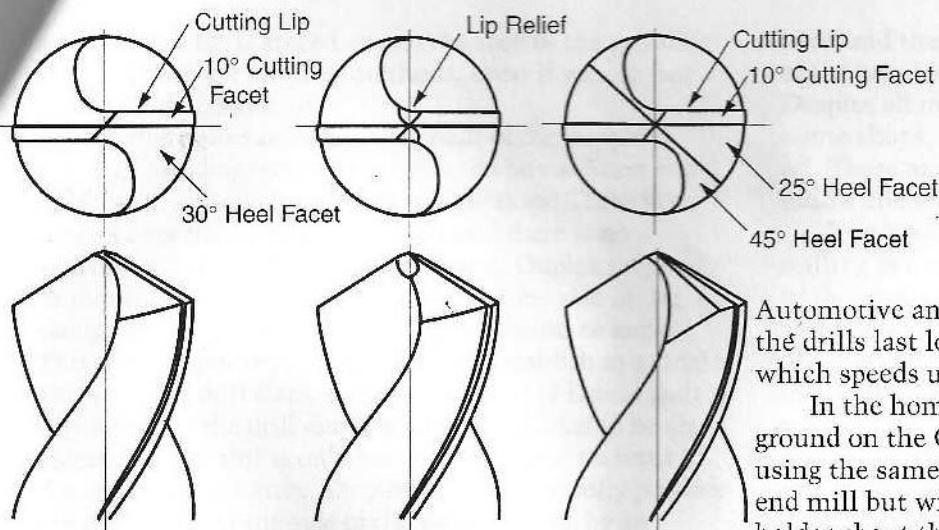
There is another possibility for grinding drills that turns out to be both simpler and better than creating curved clearance surfaces in some elaborate fixture and eliminates all the troubles just described. This grinding technique produces what is called a Split Point drill. For those who are not familiar with the split point drill, it has a very short cutting edge, measured circumferentially in the direction of the cut, and the heel is separately ground to just be "out of the way." Split Point is another term for the four-facet method of grinding, such as is used in grinding the end of an end mill.

With four-facet grinding, as shown in Figure 7, the primary lip clearance is easily established by just touching the drill to a grinding wheel at the proper angle to create a lip with a very short span. The secondary clearance may be performed with a swing, or with conical or helical advance, but most commonly with no curvature at all. The second clearance grind is independent of the primary cutting edge. The width of the cutting face remaining is only half of the web thickness, and the two lip's receding edges become coincident with the diameter of the drill. They form a straight line across the center of the drill point as viewed from the end of the drill. All four surfaces meet at the point of the drill, and cause the drill to be self-centering. Typical with four-facet grinding is to establish a primary clearance angle of 10° - 12° and a secondary clearance angle of 25° - 30° .

The four-facet method of drill sharpening turns out to be close to the ideal, and also is the simplest of all methods, because grinding is only a matter of positioning the drill in two very accurate fixed positions, independent of diameter, rather than moving the drill through a curved path that may or may not be ideal. The aris produced inherently turns out to be slightly pointed, rather than flat as in the chisel point drill, so the drill tends to keep itself centered and true running. Holes can be started without pre-drilling a starting hole, and often without center punching. With a magnifying glass, you can see that the cutting edges extend almost all the way to the point rather than stopping at a broad chisel edge. The chip produced is wider and more uniform in thickness, which translates into more efficient drilling of holes, with less heat and wear. Proof of the improved cutting action is evidenced by a reduction in pressure required to drill a hole compared with a conventional drill.

Since the heel of a split point drill is merely ground away to prevent it from rubbing on the new hole, one does not have to go to great lengths to produce a cylindrically-curved heel, a conical heel, or any other special-shaped heel. A perfectly flat heel is good enough since it is never going to touch the bottom of the hole anyway. The method is the same for all size drills, different only in scale depending on the diameter of the drill. A well-dressed grinding wheel is required. Any wander or instability in the grindstone or holding fixture could cause the stone to grind away at the wrong cutting edge. It is very doubtful if even a strong-handed craftsman could hold a drill steadily enough and precisely enough to manually grind a good split point, which probably accounts for its rarity in home shops. It requires a precision fixture to re-sharpen a split point drill.

Another variation of the four-facet split point-drill uses the six-facet method, also shown in Figure 7, which requires, after the first four facets are ground, that the drill be rotated about its axis, and two more facets ground to the center of the drill point. The point created is even sharper and more defined than with the four-facet method, and there is an improvement in cutting action and additional lowering of the pressure requirement. A refinement of the four-facet drill is the Renault-Peugeot method, used in industrial production



Four Facet Drill Four Facet Drill with Lip Relief Six Facet Drill
Figure 7

as it is more efficient and more economical than conventional drills. Their refinement of the split point drill includes a bit of web thinning to further enhance the center point's cutting ability. Japan's Dijet Hosoi split point drill uses forced lubrication, and its cutting faces are specially contoured to cut to the center of the hole much like a center-cutting end mill. Mitsubishi has its own version, with no center at all. The drill is hollow and both cutting edges cut like an end mill. The un-drilled center plug is swallowed up inside the drill

and is later knocked out, or if a through hole, it merely falls away. It too requires forced lubrication through internal passageways. Modern industry has accepted the split point drill for many reasons. Aircraft manufacturers use it because it requires less pressure with hand-held pistol grip drills and is self starting, so the drill wanders less.

Automotive and other industrial firms use it because the drills last longer, and have to be changed less often, which speeds up production and reduces costs.

In the home shop, four and six-facet drills can be ground on the Quorn Universal Tool and Cutter Grinder using the same technique as used to grind the end of an end mill but with the addition of rotating the tool holder about the Y-axis to establish the point angle. All that is needed is a set of collets or a self-centering chuck to hold a variety of drills. The work holder and drill are not swung through the conventional cylindrical or conical path during the grinding, but are clamped in either of two positions to grind the two lips using the graduated index scales to reposition the drill exactly 180° for the second cut and by fixing the limit of the forward micrometer advance along the front slide to determine and limit the width of the cutting edge, and thus to ensure that both cutting edges are identical. The grind is performed by moving the tool fixture with its rocking lever, rotating the toolholder about the X-axis

so the drill's lip is traced across the face of the grinding wheel. There are simpler methods, even if you do not have a Quorn. Read on.

All this points out the prime fault of the simple V-trough grinding fixtures that you can buy at Sears and from various through-the-mail tool catalogs: There is no control over the lip clearance angle and there is no adjustment for the position of the pivot. Duplex taught us that the drill should be positioned to one side of the swinging axis in order to create the lip clearance angle. This is good geometry, but difficult to establish in a single fixture for all drill sizes, since the amount of lateral shift is dependent on the drill diameter. Every drill has to be set differently, but this is only because there was no limit to the swing of the fixture. Duplex chose to carefully position the drill's axis to one side of the swinging axis by an amount that was correlated with the drill's diameter, as shown in Figure 6, and then let the machinist swing away with no limits. I propose that it is much better to limit the swing so that the grinding wheel can never over-cut, and not to worry about the amount of lateral shift. It is much simpler this way. Anyone with a swinging V-trough fixture that doesn't always work very well might want to experiment with means to move the pivot point and to make a set of stops that will limit the swing. The mechanism to do this is not all that easy to construct, as it involves slides in two directions as well as rotary motions with limit stops, and it still requires you to remove the drill and reposition it for the second cutting edge.

My simple method involves a Dremel belt-disk sander, or equivalent, and a small, simple fixture to hold the drill while grinding. Figure 8 shows all the dimensions for a fixture that will accommodate drills from 1/16" to 1/2". If you have larger drills to grind, you could enlarge the fixture proportionally.

The new fixture includes a V-trough that is oriented horizontally. Its shape allows you to turn the drill and fixture over for grinding the second lip without having to unclamp, reposition, and re-clamp the drill. I made mine in two mirror-imaged parts and fastened them together with screws. It is vital that all screws be countersunk below the surface and that the two face surfaces and the two edges be square and smooth, and the opposite sides parallel to each other. The clamping arrangement relies on a screw, a link, and a long bar that presses against the drill to force it into the wedge of the V-trough. A complete set of drawings is included

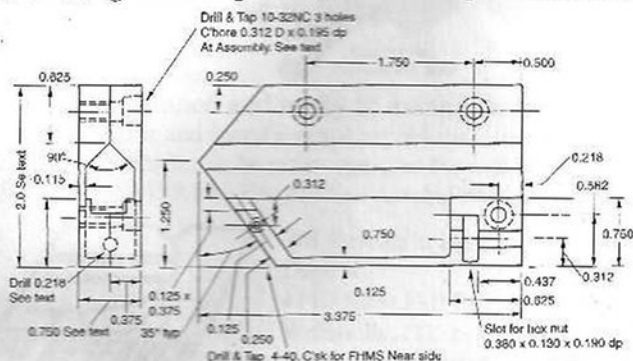
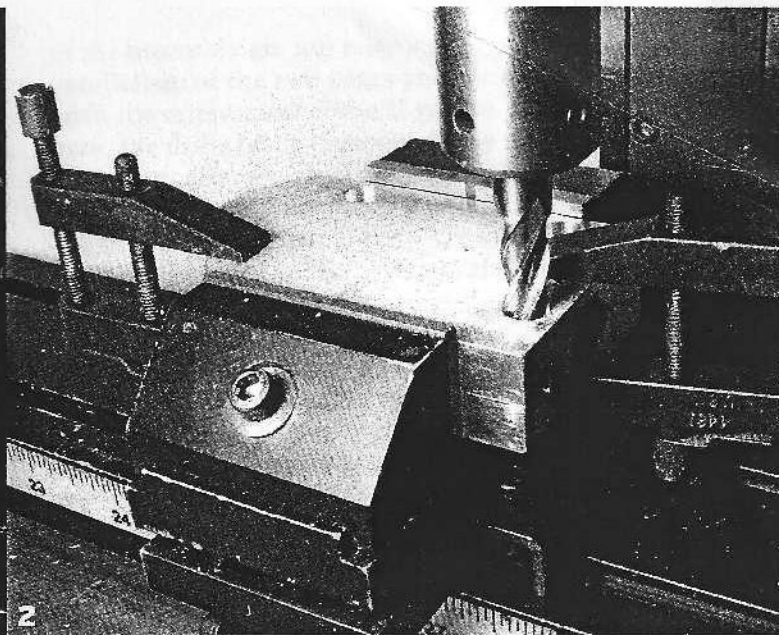
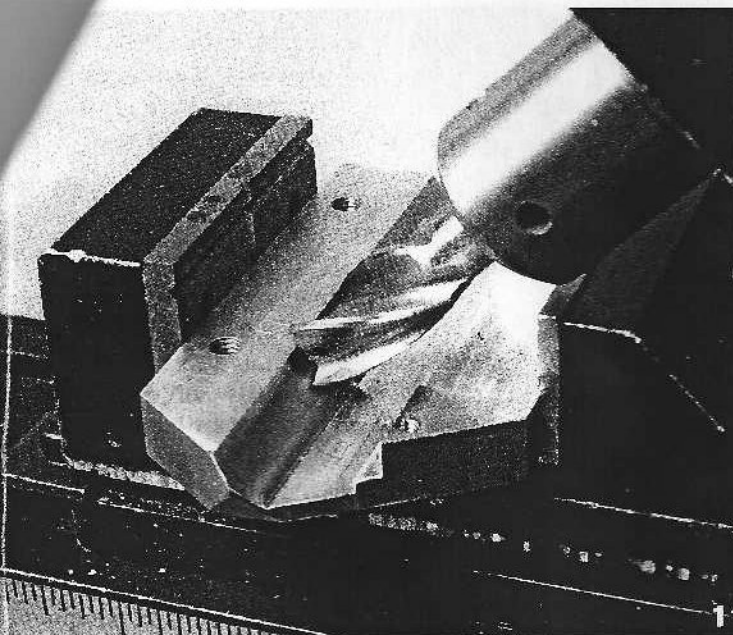


Figure 8

here, and there is no particular difficulty in making any of the parts, except to say that accuracy is important. Despite all our efforts to do things precisely in our home shops, sometimes things turn out just a little bit off. There are ways to correct for these errors if you follow the sequence of operations described here.

The V-trough is made in two parts. The sequence of milling is important. First, you should trim the pieces to the approximate shape, including the pointed end, and as usual, it would be helpful to square things up so you can clamp them in the mill vise. The two halves are mirror images of each other. Clamp the first part perfectly flat in the mill vise with the length of the part cross-wise to the mill table, pointed end toward you, checking for squareness and flatness with a dial indicator. Find the right-hand edge with an edge finder and move over .750" plus half the diameter of a .375" milling cutter plus half the diameter of the end of your edge finder. Lock the mill table in the X-direction. Cut a .375" wide groove back and forth in the Y-direction to a depth of .260" to leave .115" of stock according to Figure 8. Check the depth after a few passes by measuring the remaining thickness at both ends of the groove. If the two measurements are different, you should shim the part in the vise to make a correction. Cut this same groove in both right-hand and left-hand pieces. If you turn the second piece so the pointed end is away from you, then the same setup can be used without introducing other errors and without having to find the edge and move over to position the groove. With the two grooves cut, tip the headstock over to 45° to the right and cut the beveled edge (Photo 1) in both pieces with the end of a 1/2" end mill (a .375" end mill won't quite span the width of the beveled surface). Lower the headstock so the milling cutter will cut the beveled side but not the squared side. Unlock the mill table and move over and down, cutting off successive layers until the lower corner of the mill cutter just barely touches the bottom of the original groove, then raise it a couple of thousandths. Lock the vertical position of the headstock. From there on, increase the cut by moving the mill table only, not the headstock. Cut the beveled surface until the cutter just leaves the horizontal part of the groove. This will leave .625" to the edge of the part, but that dimension is difficult to measure. If it isn't exactly .625" on both parts, we will correct for that later on. As you cut the bevel in the second part, you can stop the mill and match the squared up end of the first part against the groove of the second part. This gives you a double-check on progress and helps make both parts identical. If you have a larger mill, it would be prudent to cut both parts at once in one pass in the Y-direction, then cut the piece apart in the center to make two identical parts. The grooves then match perfectly, but that method is more susceptible to tilt of the piece about the X-axis. If you check the depth of cut on each end as you cut the square part of the groove, you should be all right by shimming the part in the vice to correct for this error.

When the bevel is cut in both halves, fit the two



pieces together with a short length of half-inch drill rod nested firmly in the "V" created by the two beveled sides. This will assure that both halves of the "V" are parallel and matched. Don't be surprised if the V-trough ends up not being exactly parallel with the edge surfaces. It is very easy for the intersection of a flat and an angled surface to be slightly askew, but we will correct for that. Once you are satisfied that the pointed ends of the beveled groove match, and the squared end is as close a match as you can get, clamp the parts together and drill, tap, and counterbore for the three larger screw holes as shown in Photo 2. Fasten the two parts together with three 10-32 x 1/2" socket head cap screws and then remove the parallel clamps. Be especially careful as you drill and tap for the 4-40 screw on the pointed end, as there is not much clearance for the countersink. The location of these four screws is not critical, and if you measure over from the edges as shown in the drawing, you will be close enough, even

though subsequent steps may throw the holes slightly out of perfect position. Fasten the two halves together with all four screws and make sure the heads are below flush and the ends of the screws are trimmed flush or below. Now, place the assembly in the mill vise with the V-trough pointing *downwards* in preparation for making a cut on the exposed upper edge (pointed end of the assembly to the left) Using the half-inch drill rod resting in the "V," align the fixture with the mill table so that both ends of the drill rod are the same distance above the mill table. This assures that the *bottom edge* of the part (the top edge as you now see it in the mill vise) will become parallel with the mill table and aligned with the V-groove. You can measure this with a pair of inside calipers or a height gage, or by inserting spacers to exactly fill the gap to insure that the rod is parallel to the bed while still resting firmly in the V-trough. You can also use a pair of slide calipers, as in Photo 3, standing them on the mill table and measuring