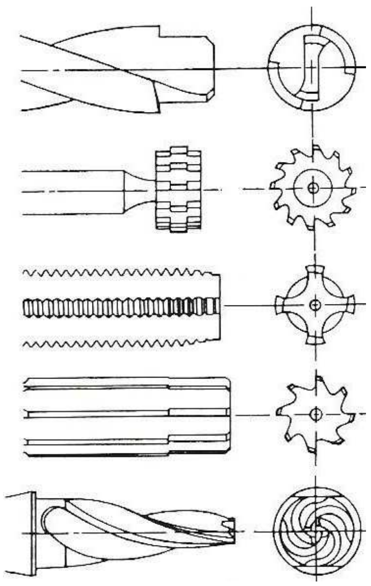
314 Fall Street, Seneca Falls, N.Y.
T: 315-568-5804 F: 315-568-5800
E: info@royaloakgrinders.com
W: www.royaloakgrinders.com

# Stop Losing Money



The following article is an excellent illustration of where many modern production shops continue to lose money. It is surprisingly simple and yet these shops continue to miss this obvious cost. Consider the fact that many parts have holes that need to be drilled. This drilling operation often increases the number of times a modern machine must be stopped just to change the drill. It even turns out that a brand new drill is frequently ground wrong. In this article, you will learn in a common sense way to increase productivity by simply improving the performance of your Tool Steel H.S.S. or carbide drill, reamer, tap, etc. by using an R-O® Form Relieving Grinder. Cutting down the number of tool changes per shift is a simple way to stop losing money.

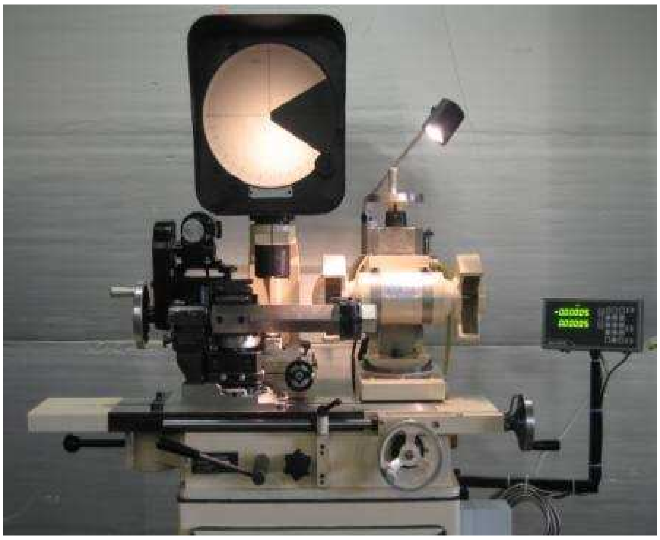
Why cut down on your tool life and performance when you can have the R-O® (Royal Oak®) Precision Form Relief Grinder.

The secret is in the form relief produced by this grinder that improves tool life.

Your new machine tool, whether numerically or manually controlled, no matter how much you paid for it, cannot outperform the cutter you use. No matter how much precision or how much speed is built into the machine tool, you get out as much and no more than the cutter will deliver.

R-O® (Royal Oak®) relieves from edge to heel giving a keen, long lasting edge for more cuts per sharpening plus tools that cut faster with less heat build-up.

## Are you sharpening your cutting tools or grinding away life & performance?



# TOOL GRINDING FUNDAMENTALS

## Part 1

### Tool Grinding Fundamentals

Every tool grinding department supervisor knows that high-speed steel cutting edges *should not be burned*. Most are willing to swear nothing of the sort is happening while tools are made or resharpened in their shop. Nonetheless, a great deal of burning occurs every day and because of it, cutting edges must be reground more often.

Even when burning is invisible to the human eye, it can be detected with this simple test: Get a bottle of 5% nital etch from a drug store, or make it yourself by pouring 5 parts of nitric acid *into* 95 parts of methanol. *Do not* reverse the process!

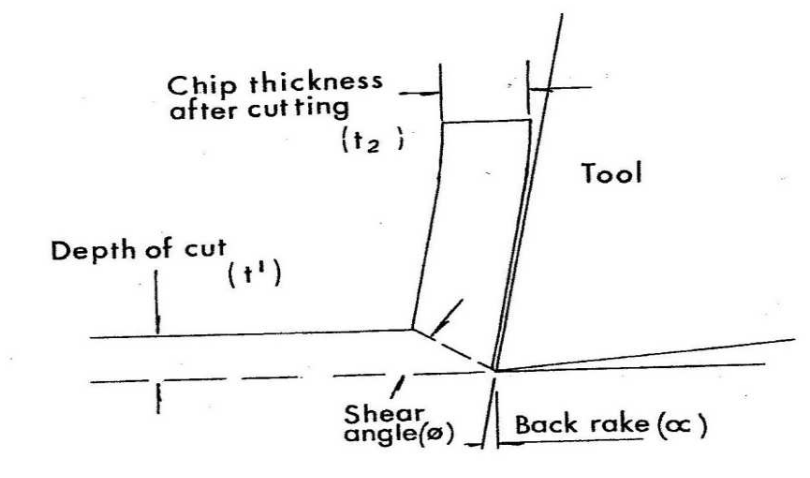
Dip a tool point in this solution. If the edge turns gray, the tool has been ground properly. If it turns black, the edge is burned. The test works even when a tool edge is so bright and shiny that no one would suspect its hardness has been harmed by overheating. It shows damage which is otherwise detectable only with expensive surface hardness testing instruments. To demonstrate this fact, deliberately burn a high speed steel tool until it is blue-black, then dip it in phosphoric acid. The burned color will disappear, but if the now shiny tool is dipped in nital etch solution it will still turn black.

If these tests seem like parlor tricks, bear in mind that properly ground tools help the productivity picture in two ways. They improve uptime by yielding more parts per dollar of tool cost. The softened skin left on an overheated tool is only a few tenths deep, but those tenths are out on the surface where most of the work is done, and aids in chip welding or built up edges (BUE) which destroys planned geometry.

In a day of electronically controlled heat-treating, it is easy to forget that the time-honored blacksmith temper test is still valid. Burn a tool to an almost imperceptible light straw and its hardness has dropped about six Rockwell points. To save looking it up in a handbook, here are color-hardness relationships on which blacksmiths once relied:

Light straw ...................... 56 Rc
Dark straw....................... 54 Rc
Light blue ........................ 52 Rc
Dark Blue ........................ 50 Rc
Purple ............................ 48 Rc

SHEAR ANGLE




The point to keep in mind is this: No matter how correct a tool's geometry may be, nor how precisely its speeds and feeds match the machine and work material, a burned tool will not work as well or last as long. As a matter of practice, *every tool should be given a very light pass across a wheel after it is almost finished to remove at least .0003" of possibly burned surface*. Because new HSS grades have higher red hardness factors, there is a tendency to forget that what is up front at the microscopic junction of a tool's cutting planes is what counts most. A quick check with nital etch may show that even some new drills, taps and other endworking tools have been reduced in hardness by overheating during manufacture.

Grinding machine operators work as they have been taught, so it is important to get new employees started correctly. Grinding wheels, in oversimplification, are random-tooth cutters which self-renew as dull grains break away to expose sharp ones. If the "teeth" are pressed against a tool forcefully enough to cause plastic flow of the wheel's bonding material, then the teeth stop cutting and in a matter of seconds, enough heat is generated to anneal the tool's edge. The answer always, is to pick a wheel with *proper grit size grade for the job*, and a bond that lets dulled grit tear away easily. In general, this will be the coarsest grit which can be dressed to required corner sharpness.

### Grit Size

A wheel's grit number indicates the screen mesh through which its grains were sifted prior to bonding into wheel form. A 100-grit wheel has grains that were sifted through a screen having 100 wires to the inch, meaning wires are .010" apart center-to-center in both directions. Since the



314 Fall Street, Seneca Falls, N.Y.
T: 315-568-5804 F: 315-568-5800
E: info@royaloakgrinders.com
W: www.royaloakgrinders.com

wires take up half the space, the openings in a 100 mesh screen are about .005" and a 100 grit wheel therefore has grit which can be dressed, at best, to half the radius of .005" or .0025" corner sharpness.

We say *"at best"* because something less than .0025" sharpness is likely unless the wheel is dressed in a way that leaves a lot of full grains standing in claw-like fashion on the wheel's corner.

Experience over the years with the R-O® Form Relieving Grinder causes the builder, Royal Oak® Grinders (Seneca Falls, NY), to lay down these recommendations for dressing wheels on its machines:

1. Always use a sharp diamond.
2. Place the diamond *below* the centerline of the wheel to pull grit out of the wheel, rather than push it in and close up the wheel face.
3. On final passes, always dress on to and *away* from the corner.

## Wheel Inventory

At today's prices a lot of money can be tied up in wheels, but having a variety of them on hand can save dressing away expensive grit each time a new shape is needed, to say nothing of the time required for dressing. If a wheel is reshaped, rather than changed for each job, the cost of another wheel easily may be spent without getting the advantages of it. A tool grinder should be able to go to his wheel rack and pick up a proper shape in much the same way that a lathe hand picks out a tool. Demountable taper sleeves, which permit wheels to be changed quickly, also are a good investment.

Wide wheels are good, but harder to balance; a lot of grit may be lost while getting them trued to run on a spindle. Recessed wheels usually can be trued more easily because they are nearer to balance when bought.

For grinding trepans and similar tools it will be necessary to have on hand a supply of small mounted wheels, and ones with ¼" shanks seem to have sufficient rigidity to avoid chatter on the work. Here again, a variety of mounted wheel sizes and shapes saves time and grit.

Wheel recommendation charts are available from vendors, but most shops develop, from experience, a chart of wheels that work best in its specific applications.

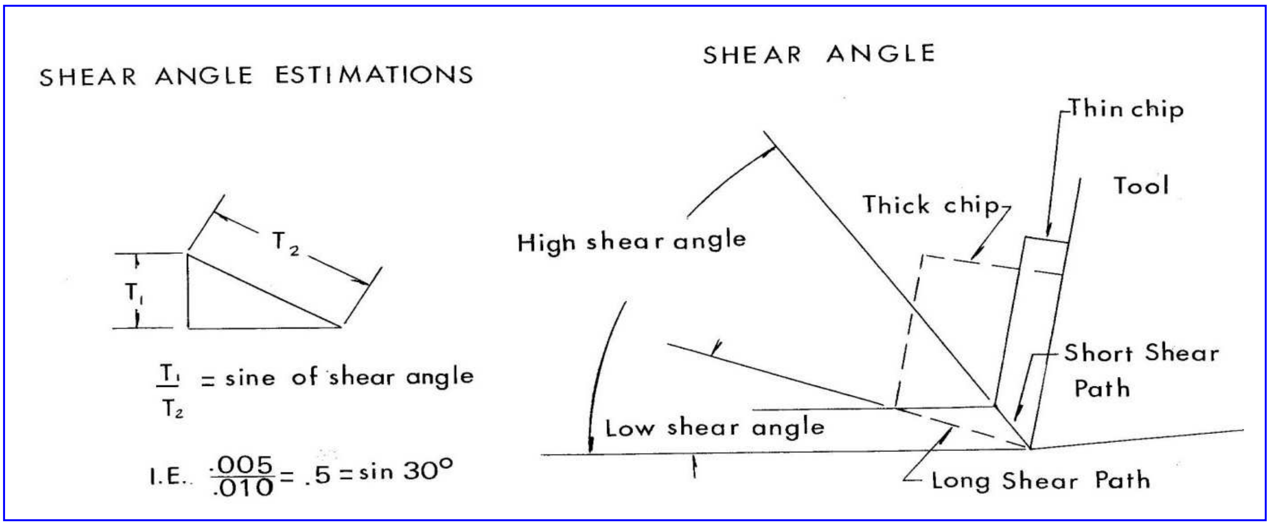
There is a great deal more to cutting tool productivity than just avoiding burns. The easiest and cheapest way to get better finish, more parts per grind and longer uptimes is by establishing procedures which assure cutting edges will not be burned.

# Part 2

## Standard Tool Geometry

Metal cutting usually is a backwards process. Somewhere out in the shop a skilled man experiments until he finds a tool shape which does what is needed. From then on, the tool geometry is standard for that job. If theory comes into the picture at all, it is usually to find out why the shape works rather than if it is what would work best.

This is the time-honored method of licking tool problems. It works when the shop can lean on the expertise of veteran tool men, for there is absolutely *no substitute for years of practical experience.* The problem, however, is how this intuitive skill of the experienced men can be put in writing and made available so newer men can benefit by it without going through the time and expense of experimenting all over again. It must



be kept in mind that the men who know tool troubleshooting best had to be paid while they cut and tried in the past. If possible, we should capitalize on their experience. To some small degree, this article may serve that purpose.

When a veteran tool man looks for trouble, he is apt to be more interested in how the chips look than how the finish is on the product. He did not learn this overnight. Whether he can put it into words or not, he is evaluating the chip in relation to the chemistry, grain, structure and hardness of the material, and in relation to the feed of the tool that made the chip. Ask him what he is looking for and he will probably say he is checking to see if the chip is "*fat.*"

A fat chip is one that is thicker than an average of 2½ times the feed rate of its tool. It did not get away in time to avoid being telescoped onto itself. It takes force to put accordion-like pleats in a metal chip. Since energy cannot be destroyed, it converts to heat in the chip. A fat chip is always hotter than one that is about the same thickness as the tool feed.

## Rake Angles

From laboratory tests, it is known that high back rake, up to 20 degrees, on the top face of the tool lets chips get away with the least heat and force. But these rake angles are so high they are not practical for production purposes since the tool's edge is too pointed to absorb heat or withstand cutting pressure. So we compromise and use rake angles as high as possible (*seven to twenty degrees)*. In a great many cases, even minor changes in top rake angle can reduce chip fatness and cause a chip to curl away from the work in a way that greatly lengthens tool life.

## Chip Cleavage

Keep in mind that chips are not just scraped off the work material. Cleavage of the chip takes place above the cutting edge. Use a chisel and hammer to observe chip cleavage in its basic form. Slow motion studies show a wave of metal begins to part above the tool in a manner somewhat the same as the wave that forms ahead of the bow of a fast moving boat. This chip sooner or later (presses) comes crashing down on the face of the tool where it is curled, turned, or flung in some direction. The tighter the chip must curl, and the greater the friction created as it strikes the tool face, the more pronounced is the cratering.

There are basically two methods of eliminating fat chips. The first is to make the top face of the tool more slippery so that chips slide across it with less friction. The second is to increase rake angles to the practical maximum. It is best to eliminate chip breakers since often additional force added to the tool chip breakers should be used as a last resort (*only if geometry will not work*).

It the top face of a cutting tool is polished to a finish of four r.m.s. or better, all else being unchanged, tool life will usually double. One of the advantages of carbide is that this is a slippery material. Chips slide across it with less friction.

Given the choice of a circular form tool having very low r.m.s. finish on its periphery and a rough grind on the gash, or one having only fair finish on its form but a polish on the gash, the latter will outperform the other every time. It is a fairly easy thing to prove in your own shop.

The same applies to taps. The finish on faces in the flutes has more effect on tapped hole quality than the condition of the tap's thread profile. But of the two, the forming tool example is the best, for if one polishes the flutes of a tap, making sure to hold cutting edge geometry, of course, some of the improvement may be due to removal of edge burn, as mentioned earlier.

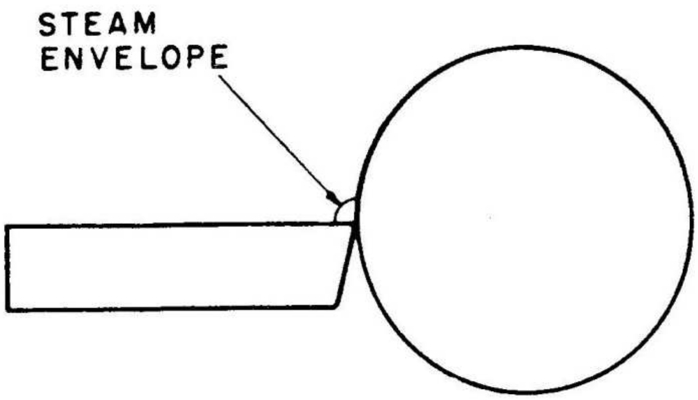
This is not a condemnation of commercial tap and drill makers, but the theory still holds; a bit of polishing after an item is purchased can sometimes give longer life.

A skilled tool troubleshooter, after looking at the chips, will look next at the tool. He needs to know where the crater is forming on the tool face, and what is causing it. He will not be able to eliminate cratering altogether in most cases, but he can control crater position, and it tells him what is happening.

Two things cause tool face cratering. The first is heat; the second, friction, the rubbing of a hot chip on the tool. The answer to friction is greater rake face angle if possible, and/or a better finish on the top of the tool, preferably both. In theory, at least, coolant has little effect on this type of crater.

The other crater is caused when enough heat is generated in the cutting zone to cause chips to weld to, and then break away from, the face of the tool. A coolant having better lubricity reduces welding. But the veteran looks for something more. If the crater is very near the edge, it indicates the tool's feed should be increased. Thick chips *(not to be confused with fat ones)* carry more heat away, and due to the cleavage factor, will flow back on the tool face a greater distance from the tool edge. In cases of very light feed, the crater may form at the cutting edge and be mistaken for edge wear. Unless there is an unmarked area between the tool's face and the crater, increase the feed.

One of the most controversial subjects in metal cutting is whether or not coolant can work its way between the chip and the tool. Some say it can, if the coolant has enough capillary action, and if its flow is properly directed.




**STEAM FORMS AN INSULATING ENVELOPE THAT THE COOLANT CANNOT PENETRATE**

## Capillary Ability

A quick check on the capillary ability of coolant is to pour samples of several into glass jars and place a clean, very dry, glass rod in each. The oil which climbs farthest up the rod stands the best chance of getting under the chip, and in most cases this will be the oil that has the greatest amount of sulfur or chlorine additive. So, for all practical purposes, weld-craters can be helped by souping up the coolant with one of these or similar synthetic chemicals.

One should always point the coolant line so the *flow strikes the side of the working area.* If one considers that we are really trying to get at least some coolant between the chip and the tool, this makes sense. The hazard in pointing the flow from above the tool is that it will quench the chip, making it more abrasive to the tool face, while at the same time lessening the chance of getting coolant into the chip-tool interface. The presence of a steam envelope in the hot area actually seals off any advantage to cooling from the top (*see figure above*).

From this it will be evident the tool grinder needs to know the feed rate when he grinds the tool. The polished top face should extend rearward well past the chip contact point. A tool with too short a crater gash may cause the chip to curl on itself. This is the most damaging condition one can produce so far as heat generation is concerned. The longer the chip stays on the face of the tool the more of its heat will be absorbed into the tool. Whatever method is used, *move the chips off the tool fast*.

Knowing these things, the tool grinder is in a better position to alter geometry when standardly ground tools do not hold up. He can see from the craters on worn tools whether a change in geometry or feed is in relationship to material; the observation should occur before the tool is ruined. He can tell, under a magnifier, whether the crater resulted from wear or from welding. And yet, there are cases when he can do very little about it. Some matters have to be handled by tool setting at the machine.

## Tool Above Center

For example, a standard cut-off tool is normally not given a top rake. If the tool is cutting off into a hole, it can be given effective top rake by setting it above center as far as front clearance allows. This is particularly important on softer materials where the "wave" caused by cleavage may actually become a series of ripples. A carbide cut-off tool (*which, because of the nature of its material, often has neither no top rake or negative rake*) will let chips escape faster if set above center as far as possible, thereby creating a positive top clearance.

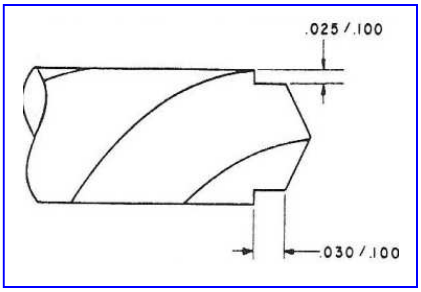
In the case of a standard multi-fluted reamer, the tool grinder can know from the position of the edge crater how much stock is being left for reaming. If it is obvious very little stock is being left, he may do well to increase the lead chamfer angle to spread the chip over a longer face area for each lip. Such an angle will also help the reamer "cone" to center and produce holes that are more concentric. Much of the same applies to taps. A change in chamfer angle may make a great difference in grind life.

## Large Diameter Drills

The same principle applies, but for a different reason, when large diameter drills are used. Because of the need for strength, drills larger than 1½" often are left at about 60 Rc rather than being hardened to H.S.S. maximum.

It is imperative that large drills are not burned, since they are none too hard to begin with.

Drilling can be improved by using step drills. The forces acting on a drill point tend to center the drill when divided evenly. This method assumes that the shop has a means of grinding




a concentric step on the drill and giving the smaller diameter minimum clearance, for it must act as a non-cutting pilot.

In the same regard, any drill will feed with about half the pressure if a split (*crankshaft*) point is put on it. The point needs to be exactly on center and split points are easier to center. Many recommended split points down to where the diameter is so small it cannot be put on.

## Drill Thinning

It should go without saying, but often it does not, that all drills should be thinned to center before they are pointed. And, once again, thinning should produce good finish on the cutting face. It is just as important as with a forming tool.

Next we will be delving into relief angles, and their effect on tool life. ***To summarize in a few words the advice given thus far: Make sure the tools are not burned, and shine up the cutting faces!***

# Part 3

We have discussed the importance of moving chips off the tool face. The recommendations are to use top rake angles as high as possible (*there are maximum limits*), to give the tool face a finish of four r.m.s. or better to cause easier chip slippage, and to use a capillary-type coolant properly directed into the work zone from the side.

In addition, it was mentioned that in laboratory conditions extremely sharp wedge-shaped cutting edges cause less heat and use less metal removing energy, but that this theory had to be tempered by the needs of production tools to stand up over reasonably long periods and produce proper finish.

The subject of this section of the article is to point out how the wedge of the tool point may be retained at a greater angle even when larger top rakes are used. The method, as those who use R-O® grinders will have guessed by now, is through a reduction in front relief angles.

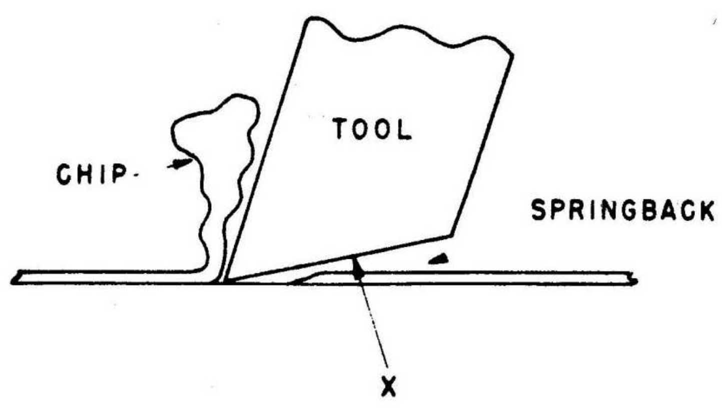



FIGURE 5

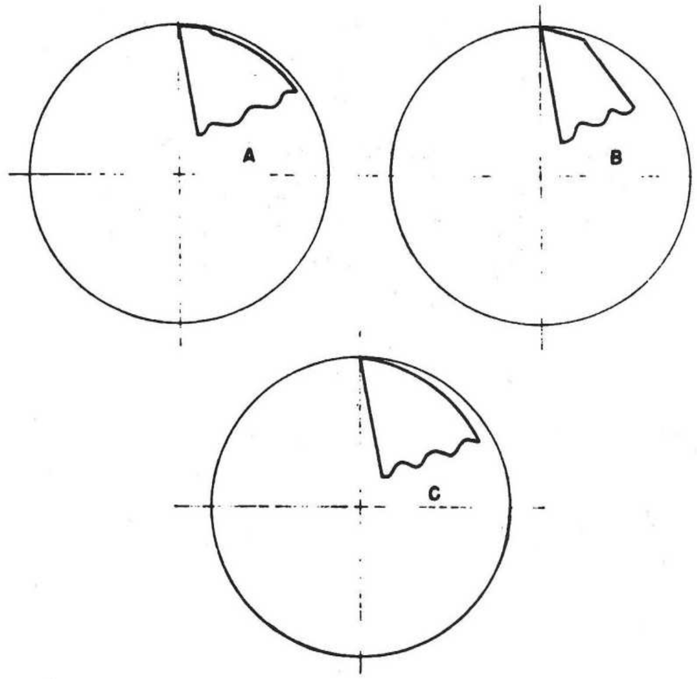



FIGURE 6

Over a period of years, there has developed an empirical formula...... empirical in the sense that it works, though there is no tangible way of showing what happens.

The formula is based on the fact that a certain amount of springback exists in all materials. When a tool cuts along a work-piece, it produces compressive forces at the point of contact of work and tool. As the tool passes any given point, compression is relieved. The work "*grows*" slightly directly behind the tool. This is why all cutting edges require front clearance, or relief, as shown by the (X) on the accompanying sketch (*Figure 5*).

There *must* be relief behind the cutting edge. However most tools are given either more relief than is needed, or the wrong kind of relief. To illustrate, look at the sketch showing ways of grinding a radial cutting tool edge. In all three views, assume the rake angle to be identical; we are concerned at this point with peripheral relief (*Figure 6*).

## Peripheral Relief

"A" in *Figure 6* is the condition often encountered in the twist drill, circle grinding. There is a non-cutting pilot, which is expected to be given diametrical clearance by the cutting point. So long as the point produces a hole slightly larger than the drill's margins, the non-relieved margins will not rub. This is a non-cutting edge, hence it needs little (*in fact, it should not have much*) peripheral clearance or it will produce an oversize hole.

In the same illustration, assume "B" to be a tooth of a side-cutting cutter. In order to get chips away from the tool face, the tool has top rake, and this, in combination with high relief and clearance angles, produces a fragile cutting section. Frictional heat generated as chips strike the tool face will rapidly cause this tool to fail, for it lacks heat-absorbing mass at the cutting point. Sketch "C" will stand up much longer. The radial relief is reduced, giving greater tool mass. The edge support is better. Heat, which always dislikes traveling around a corner, will be better dissipated in "C" than in "B."

Ideally, most will agree, the uniformly curved relief of "C" will give longer tool life, and those who use this type of grind have found greatest tool life is had when the clearance is as little as possible.

The question then is how much is enough? The R-O® empirical formula gives the answer to that question. The same rule applies to end clearance on axial tools and front clearance on radial tools. Here it is: *The amount of relief which lies between 2½ and 10 times the feed per tooth gives optimum tool life*.

The usual procedure with this empirical clearance formula is to start with a five-times-feed clearance and work back toward 2½ times. Most material will fall in this range. Those which may need more than five-times-feed clearances are S.A.E. 4150 steel (*in soft state*), some of the aluminum bronzes, and highly deflective metals.

Referring to the "springback" sketch in *Figure 5*, clearance should be reduced a bit at a time until a shiny spot appears at the point "X" on the relieved area of the tool. This is evidence the clearance is so low that the metal is coming up to meet it as it springs back when tool compression is relieved.

The next larger clearance which will eliminate the shiny area on the relief, is the clearance which will give the longest tool life. If practical, these clearance-finding tests should be run dry. Since lubricity of coolant is added

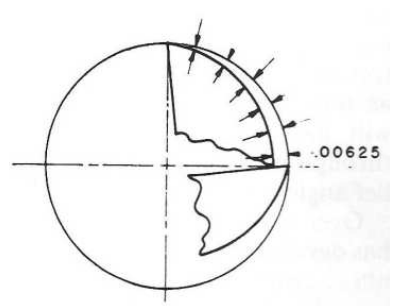



FIGURE 7

as another factor, rubbing may occur but not be noticeable as quickly.

For example, a two-flute tool fed at .010" per revolution will have the following drop, tooth to tooth; if this formula is applied at 2½ times feed:

.010 (i.p.r feed) / 2 teeth = .005" feed/tooth.

.005 x 2.5 = .0125" drop from cutting edge to its junction with the other cutting edge, or in 180-degrees of circumference.

Keep in mind this formula always yields drop in decimal-inch from tooth-to-tooth.

This type of relief or clearance (*shown in Figure 7*) is not an angle, nor an eccentric circle. Rather, it is a generated uniform curve on which, at any point on the clearance, the amount of relief is in direct ratio to the degree of arc from the cutting point.

Referring next to the sketch in *Figure 8* of the step drill requiring axial relief at point A, the calculations needed to compute the amount of drop needed in 180-degrees for a desired equivalent clearance angle are:

Drop = (B x PI x Tangent of Clearance angle A) / number of flutes on the tool = Axial Drop.

An example: One-inch diameter drill (diameter B) needing five-degree clearance Angle A, two flutes.

(1 x 3.1416 x .08749) / 2 = .137" axial drop, tooth to tooth.

(*Note: When several diameters are being considered in the same tool, for dimensions B, choose the average diameter of all of them as B.*)

The method for calculating radial relief and converting angular clearances to decimal equivalents is:

Drop = diameter x 3.1416 x sine of clearance angle / number of flutes = radial drop. Example: One-inch diameter cutter, four flutes, five degree clearance angle:

(1 x PI x .08716) / 4 = .068" radial drop tooth to tooth.

## Multi Diameter Tools

When endworking, multi-diameter tools are relieved to the cutting edge (*no circular margin*) they often cut better holes, and <u>will last two to five times as long between grinds</u>.

Taking the example in *Figure 9*, the multi-diameter drill viewed here from the cutting end and having uniformly relieved clearance of from .006" to .030" (*depending on deflective characteristics of material and diameter of tool*).

This clearance is figured on the width of the land, rather than tooth-to-tooth as was suggested earlier. The aim here is to eliminate the circular margin, yet keep relief low enough so little or no side cutting occurs. In effect, we are merely compensating for metal springback and the possibility of circular contact of tool in the hole should the point produce a slightly undersize dimension due to corner breakdown. Again, a uniform clearance curve is generated to give increasing relief in proportion to distance from cutting point.

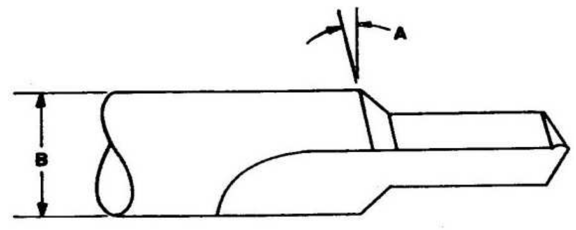



FIGURE 8

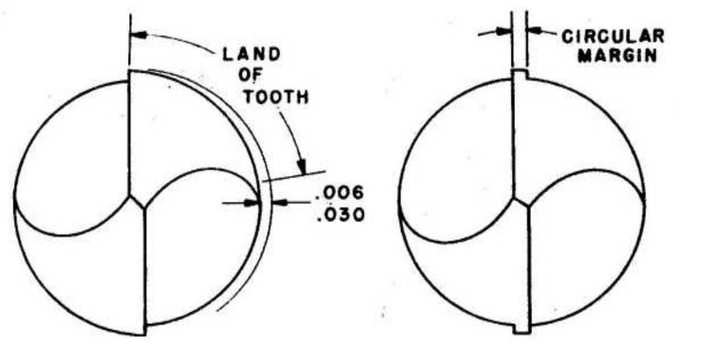



FIGURE 9

## Record Keeping

As an aid to keeping records of tool clearances that have been proven, try a chart such as the one shown in *Table 2* (*this is a procedure to establish best relief for cam-relieved cutting tools, axial end clearance, radial peripheral clearance*). This was developed as a setup sheet for use with the R-O® Universal Form Relieving Fixture, but in essence, the information applies to whatever method is used to achieve axial, radial, and peripheral tool relief.

| |
|---|
| **Material** |
| **Feed Rate** |
| **Number of Teeth in Cutter** |
| **Chip Load per Tooth** |
| **Feed Multiple From Empirical Formula** |
| **Axial Relief** |
| **Radial Relief From Empirical Formula** |
| **(Or) Axial Relief From Empirical Formula (Degrees to Decimal)** |
| **(Or) Radial Relief From Empirical Formula (Degrees to Decimal)** |
| **Axial Relief / Radial Relief = Ratio** |
| **From Chart: Carriage Setting** |
| **From Chart: Cam Drop** |
| **From Chart: Bell Crank Setting** |

**TABLE 2**: Procedure to establish best relief for cam-relieved cutting tools, axial end clearance, and radial peripheral clearance.

In the following section we shall consider how such relief can best be produced, as well as a few of the grinding tricks which can save time and money for any shop. ***But for now, to review what has been stated thus far: Do not burn tools. Keep top rake angles as acute as possible and highly polished. Reduce relief angles to an absolute minimum to give the best edge support and grind life.***

## Part 4

So far, we have considered tool geometry as if it were the same for all materials. This is not the case, as shop men know. Because of this, the empirical formula allows the 2½ to 10 times clearance for form relief as a broad basis within which most materials will fall.

Softer materials, being more apt to compress, call for greater relief angles. Then, too, tougher materials, due to springback, require higher relief than normal materials. In drilling, there is the tools expansion factor, which must be considered.

Expansion is controllable to some extent by keeping rake angles high, and top faces of tools polished. However, for the same reason, it must be accepted that more energy is needed to cut a chip away from some materials than from others. This energy, originating as machine horsepower, converts to heat. No matter how well the tool is lubricated by the coolant, and how slippery the tool faces may be, heat will be created.

| Average Feed Per Revolution For End Working Tools In All Metals | | Polished Flute Tools For Aluminum May Feed as Fast as: |
|---|---|---|
| Working Diameter | Feed (i.p.r.) | Feed (i.p.r.) |
| .062 | .0015 | .002 |
| .125 | .0025 | .003 |
| .187 | .004 | .007 |
| .250 | .005 | .007 |
| .312 | .005 | .009 |
| .375 | .006 | .012 |
| .437 | .007 | .012 |
| .500 | .008 | .012 |
| .562 | .008 | .014 |
| .625 | .009 | .015 |
| .687 | .009 | .015 |
| .750 | .010 | .016 |
| .812 | .010 | .016 |
| .875 | .011 | .017 |
| .937 | .012 | .017 |
| 1.000 | .013 | .018 |
| 1.060 | .013 | .018 |
| 1.125 | .014 | .020 |
| 1.187 | .014 | .022 |
| 1.250 | .015 | .024 |
| 1.312 | .015 | .026 |
| 1.375 | .015 | .030 |
| 1.500 | .015 | .035 |
| 1.562 | .016 | .035 |
| 1.625 | .016 | .035 |
| 1.687 | .016 | .035 |
| 1.750 | .016 | .035 |
| 1.812 | .016 | .035 |
| 1.875 | .016 | .035 |
| 1.937 | .016 | .035 |
| 2.000 | .016 | |

**TABLE 3:** Average feeds for twist drills. These feed rates can also be used for the evaluation of most endworking tools.

In deep drilling, the drill, usually having smaller mass than the workpiece, is apt to expand faster, and can, as shop men know, result in a squeal-fit of the tool to the hole.

The important thing is to give those who use and grind tools an intimate knowledge of what actually goes on in the cut so that, by looking at a tool, they can diagnose its weak points. We are convinced that a tool grinder will be a better man for having spent some time using cutting tools, and that a machine operator will produce more and better work if at some time in his training he has had tool grinding experience. It is the teamwork of machinist and tool grinder which puts tough jobs on a paying basis.

Going back to the know-how of the old time troubleshooter mentioned earlier, let us make one point again: *Though he may not be able to put it into words, the old-timer knows that the chemistry, grain size, structure and hardness of a material has a direct effect on the condition of the chip and the amount of energy needed to push a tool through a given piece of metal.*

In more cases than not, tools are underfed rather than overfed. The rule should be to give the tool edge all the support possible, using the principles mentioned earlier, and then to feed it up to the limit of the tool's strength, or the capacity of the machine. As an evaluation of whether tools are being fed at what might be called a "national average," the accompanying *Table 3* shows reference examples of average HSS twist drill feeds for endworking tools (*similar tables can be made for other types of tool material and work material*). If careful flute and face polishing techniques are followed, these feeds can be greatly exceeded, as this table shows.

While everyone dreams of a condition in which the engineers lay out the tool geometry and send the specifications to the tool grinder….. and they are right….. we are still quite a way from this ideal. Production shops depend in a large degree on the imagination and knowledge of the cutter grinder and the machinist.

And right here, let it be said one of the most profitable things a tool grinder can know, from the standpoint of his management's costs, is his ability to tell whether or not a tool is worth regrinding. If unusable tools can be scrapped before they are resharpened and tried, time and money are saved in both the crib and the production area.

## Taps, for Example

Let's take taps as an example, since probably more useless taps are reground than any other cutting tool. One should do a bit of measuring before even considering regrinding them. The major diameter of a tap must always be larger than the major

**Major Diameter Tolerances – Cut and Ground Thread Taps**

| | | Cut Thread | | Ground Thread | |
|---|---|---|---|---|---|
| Size | T.P.I. | Max. | Min. | Max. | Min. |
| # 4 | 40 | .1156 | .1136 | .1145 | .1135 |
| | 48 | .1153 | .1133 | .114 | .1130 |
| # 5 | 40 | .1286 | .1266 | .1275 | .1265 |
| | 44 | .1284 | .1264 | .1270 | .1260 |
| # 6 | 32 | .1422 | .1402 | .141 | .140 |
| | 40 | .1416 | .1396 | .1405 | .1395 |
| # 8 | 32 | .1682 | .1662 | .167 | .166 |
| | 36 | .1677 | .1657 | .1665 | .1655 |
| #10 | 24 | .1948 | .1928 | .194 | .193 |
| | 32 | .1942 | .1922 | .193 | .192 |
| ¼ | 20 | .2557 | .2532 | .255 | .2540 |
| | 28 | .2549 | .2524 | .2535 | .2525 |
| 5/16 | 18 | .3185 | .316 | .318 | .317 |
| | 24 | .3178 | .3153 | .3165 | .3155 |
| 3/8 | 16 | .3814 | .3789 | .381 | .380 |
| | 24 | | | .379 | .378 |
| 7/16 | 14 | | | .4445 | .4435 |
| | 20 | | | .4425 | .4415 |
| ½ | 13 | | | .5075 | .5065 |
| | 20 | | | .505 | .504 |
| 9/16 | 12 | | | .570 | .569 |
| | 18 | | | .568 | .567 |
| 5/8 | 11 | | | .633 | .632 |
| | 18 | | | .6305 | .6295 |
| ¾ | 10 | | | .759 | .7575 |
| | 16 | | | .76 | .7550 |
| 7/8 | 9 | | | .8850 | .8835 |
| | 14 | | | .8820 | .8810 |
| 1 | 8 | | | 1.011 | 1.0095 |
| | 14 | | | 1.007 | 1.006 |
| 1-1/8 | 7 | | | 1.137 | 1.1350 |
| | 12 | | | 1.325 | 1.1315 |
| 1-¼ | 7 | | | 1.262 | 1.260 |
| | 12 | | | 1.2575 | 1.2565 |
| 1-3/8 | 6 | | | 1.389 | 1.3870 |
| | 12 | | | 1.3825 | 1.3815 |
| 1-½ | 12 | | | 1.514 | 1.512 |

**Table 4**

diameter of the go gage, or the bolt, which will be used in the hole.

This is to allow for corner wear on the top of the tap's thread crest. As the tap's thread crest rounds off through wear, it reaches a point at which the wear radius extends below the minimum diameter needed to clear the go gage.

The logic involved here works out so that, unless there are threads behind the chamfer on which the cross-tooth diameter measure is considerably larger than the major diameter of the screw, do not spend time resharpening the tap. (*see Table 4 for example*).

The go gage will not enter the part if an undersized tap is chamfer ground (*on the O.D.*) centrally. If you grind off center to get a bigger hole, the not-go gage will always enter because what you need is a thread form in which the go gage will not interfere, not one with larger root size. Literally hours of tap trouble on the job can be saved if the tool grinder is aware of this condition and will avoid resharpening and reissuing taps on which the major diameter is too small.

To repeat once again, the way to know whether or not a tap should be scrapped, or whether it will fall within tolerance, is by referring its dimensions to the American Standard tap tolerances listed in various tool handbooks.

It will be noted a ¼-20 tap *ground variety* has a basic major minimum diameter of .254" and maximum of .2555". A ¼-20 cut thread tap having a diameter between .254" and .2555" will hold tolerance. Any used tap having a diameter less than.254" will not let the go gage enter.

This does not mean taps cannot be resharpened. They should be, so long as somewhere back on the tap's threads there are tooth crests which come up to the major minimum diameter. This, in some cases, may mean a worn length of tap must be cut off before a new chamfer is ground. Also, on tap chamfers, we recommend a .006" to .010" relief drop from cutting face to the back edge of the land.

As in all other tools, what we are seeking is maximum backing for the cutting edge, coupled with good rake, or flute hook, to get the chips away. It is understood that the height of all tap tooth chamfers should be alike. If they are not, the tap will cut oversize.

## Equal Tooth Heights

To get equal tooth heights, one must control the concentricity of the tap while the chamfer is ground, as well as the grinding of the hook. It is a good idea, if possible, to put both axial and radial clearance on the chamfer. We will discuss more about this later.

It is sufficient to say that, unlike a drill, for a full thread, a tap must first enlarge the drill or reamed hole, and then form a thread in it. Therefore, the chamfer is very important. Note also that a way of making the chamfer angle of a reamer hold up longer, is to put a bit of axial, as well as radial relief on it too.

If one can regrind a tap so that the t.i.r., tooth-to-tooth, is within .0003”, the number of holes possible between each sharpening can frequently be doubled. It is also wise to avoid grinding the flutes of ordinary thread taps as long as possible. The rake angle generated in them by the manufacturer is usually correct, and few know how it is designed, or how to reproduce it in the tool grinding room. On the other hand, pipe taps may require flute grinding to maintain cone diameter.

Earlier it was mentioned that sharp corners can best be dressed on grinding wheels by dressing below center, and away from the corner. It is equally important that when a tool is ground, the grinding be done away from the cutting edge. So far as possible, it is best to wipe the heat generated by the wheel back into the thicker portion of the tool, and at the same time avoid slurring a burr out onto the cutting edge.

### Tool Sharpening Burrs

If burrs are left, they should be on the heel of the land rather than up front, where they matter. If equipment such as the R-O® grinder is used, this is a built in feature, for the tool is fixtured to rotate top-coming against a top-coming wheel.

In Part 5 we shall be getting down to cases and considering the problems of grinding various geometries onto various tools.

***But for now, just one more tip: Often a very slim tool must be held on its shank and ground in a manner which allows less than maximum support for the cutting edge. If possible, a bushing should be mounted as an outboard support. But at times, even this cannot be done easily, and the usual result is chatter on the ground finish. Much of this chatter can be avoided if a wad of modeling clay (or something similar) is wrapped around the tool over all its exposed length except that which is to be ground. A good fistful wadded about the tool’s shank damps off the vibration, and tool finishes can be cut to half the r.m.s. otherwise possible.***

***Often these vibrations cause a portion of the tool to snap off. This is caused by the teeth of the grinding wheel pulling the drill, tap or another type tool into the wheel. The normal resistance to bending causes the tool to snap back. The subsequent tool undulations dress the wheel, destroying wheel shape and causing pilots to be broken off.***

***The modeling clay trick is an old-fashioned remedy, of course, but it is one worth trying if you have not found a solution.***

## Part 5

If a grinding wheel needs only a sharp corner or a simple angle, put it on with a diamond dressing tool. If a more complicated shape is required, try crush dressing. In almost every case, it will be faster and easier.

There is nothing difficult or mysterious about crush dressing if a few rules….. and a few tricks….. are known. Usually, the shop can make its own crushing rolls. Work finish will be as good as the finish on the roll and the skill of the man using it.

In diamond dressing, some of the grain is flattened, some is torn out, and some is fractured and resharpened. Crush dressing works just the opposite. The grit is crumbled free from its bond by a rolling, compressive motion. Crush dressing breaks down large flatted clumps of grain and *provides a sharper wheel with more teeth.* Therefore, more stock is removed at a lesser heat, reducing tempering of the cutter as well. The wheel surface cuts freer and holds shape longer.

In some crush dressing applications, both the wheel and the roll are driven at synchronized speed. On R-O® grinders the roll drives a free-running wheel. The crush roll is placed in the grinding fixture, rotated slowly, and brought against the wheel to drive it by friction. The roll is then advanced into the wheel by infeed until the proper wheel profile has been obtained.

The first thing to keep in mind is that a lot of the crumbled grit from the wheel tends to stick to either the wheel or the roll. If this is not wiped away it will have a ballmilling action, and the wheel will be impacted, destroying shape. Scrub the grit out by laying a brass wire brush with bristles no larger than .005” diameter across both the wheel and the roll. A steel brush will work, but if a hard steel bristle comes loose, being polarized, it may cling to the wheel, coming between the wheel and form, thus destroying the form.

On R-O® grinders it is best to crush dress dry since grit can fall away more easily, but a coolant can be used. It is not a bad idea, when the form is almost done, to back the wheel away and put it under power to fling off loose grit before coming back to crush for the last few tenths of shape. Of course, there is no “spark out” in crush dressing because there are no sparks, but one must still feed in very gently for the last pass to get a shape which duplicates that of the roll. When crushing is complete, the wheel will skip on the crusher. Any wheel may be crush dressed, generally 80-220 grain, hardness H to O. Friable bonds are in order.

Crushing rolls can be made from any material. Hardened crushers have longer life. Case hardenable steel works well. The usual procedure is to drill and ream a center hole in the blank to fit whatever driving arbor will be used and then to turn the OD profile on the roll in a lathe.

After hardening, the shape is polished or ground to exact shape. The shape on the roll will be duplicated on the wheel, and also on the workpiece, and so it must be accurate. If form tools are to be ground, corrections for offset must be figured and put into the roll's shape.

There are two very good things about crush dressing. The first is that, once a roll is on hand, it is an easy matter to dress a wheel and duplicate a tool form made in the past. The other is that, with a little ingenuity, rolls can be self-conditioning.

We suggest that, if a roll is to be used for any great number of dressings, a spare blank be turned out when the first roll is made. With the finished form in the grinder, dress the wheel to shape. Then use the wheel's profile to circle grind the hardened spare blank. Now there are two rolls. Put one aside until the first begins to show wear. At that time, crush dress with the second roll and use the dressed wheel to recondition the form of the first. If the operator is careful not to let both rolls become worn at once, this reconditioning can be done repeatedly without loss of form.

Since crush dressing, as recommended here, calls for the roll to drive the wheel, there is no exact requirement for either wheel or roll size. It is best though, to make the roll at least one-third as large in diameter as the wheel it will be used with. Any wheel-to-roll ratio between 3:1 and 1:1 works well. Obviously, larger rolls wear slower, since they have greater working surface. Also, one must keep in mind that if reconditioning of the rolls is to be done, a case hardened skin may not work as well. A deep hardening steel is recommended if long use is expected.

When many tools of a kind are to be sharpened on a crush dressed wheel, it saves time to make a combination arbor, which holds both tool and roll. For example, a small milling cutter is mounted on the end of a crush dressing roll arbor in such a manner that either the tool or the roll can be brought up to the wheel without disturbing the setup. In this case, the procedure is to crush dress, circle grind the grooves in the cutter, and then use the same wheel to give radial relief to the cutting edges.

In order that the roll's form will be duplicated on the wheel, crush dressing must be done at the same height above or below center as grinding will be done. To make this simpler, we suggest crushing and grinding both be done at the wheel's center line.

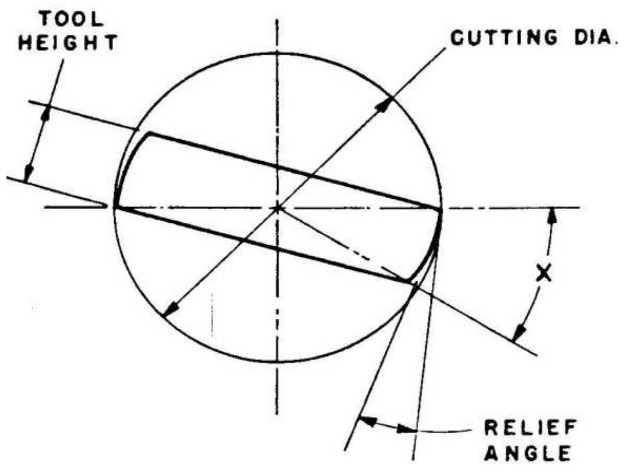



FIGURE 12

To convert angular values into decimal drop per degree of arc:
1 – Construct a layout as shown to determine x.
2 – Use cutting diameter and relief angle to determine drop in 360º: Cutting diameter x sine of relief angle = drop in 360º.
3 – Divide this drop by 360º and multiply result by number of degrees in x. This will equal drop required for tool.

## Flat Form Tools

When a flat forming tool has been circle ground on a crush-dressed wheel it must either be gashed on top and set high, or given front clearance. We suggest the latter, since this can be done in the same setup and with the same wheel which produces the cutting profile.

The relief, if produced on an R-O® grinder, will be radial rather than angular. This means the operator must convert angular values into decimal drop per-degree-of-arc. This is a simple matter, and is explained in the caption on *Figure 12*.

# TOOL GRINDING FUNDAMENTALS

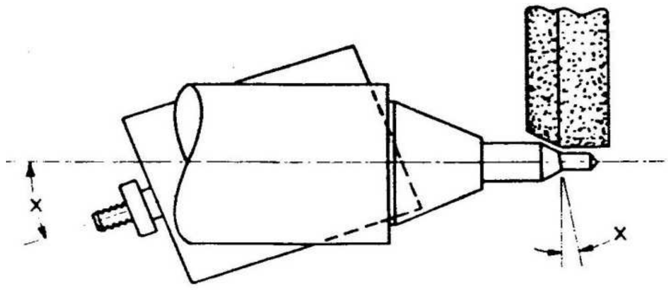



FIGURE 13

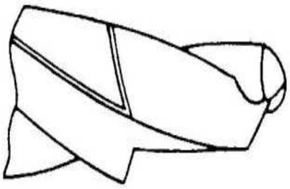


FIGURE 14

## Center Drills

Tapping is always improved when drilling and countersinking are done with the same tool, since if the countersink is not centered with the hole, the tap follows the eccentricity as it starts and then has to correct its position as it gets deeper into the work. For this reason, among others, many shops find the need to make their own combination centering and chamfering tools for some work.

One approach is to put the required profile on the wheel either by crush dressing or with a diamond, using a spin grind on the drill's pilot diameter and chamfer angle with the tool held in the grinding fixture. Then, without disturbing drill location, switch to cam control on the form relieving fixture and give the tool a combination of radial and axial relief. The reason: The most vulnerable spot on a shop-made center drill is the junction of the pilot diameter and the chamfer. If this is just circularly ground, it has no axial, or endwise clearance, no matter how well it is backed off radially.

There is always the chance it will rub. You can solve this by swinging the base of the fixture while keeping the tool in line with the wheel, as shown in *Figure 13*. When the cam-controlled base is swung in this manner, there is a combination of endwise and crosswise movement. As the tool is fed toward the wheel, the first contact is at the cam's high point, and if the tool has been correctly related to the cam's position when chucked up, first grinding takes place on the trailing edge of the drill land. As the tool rotates, the fixture is moving axially toward the left as seen in *Figure 13*, and at the same time is moving radially away from the wheel. The result is that the junction of the two cutting surfaces on the tool are given form relieved clearance at "X" equal to the "X" setting of the fixture.

## Square Shoulders

Getting both lips of a square-cutting drill or piloted counter-boring tool exactly the same height and at 90 degrees is an almost impossible job *if done by hand*. Without taking anything away from the old timers who made those tools off-hand for years, one of their grinding jobs is very apt to look pretty crude if viewed on an optical comparator. In almost every case the hand-sharpened flat bottom drill will (1) have lips of unequal height, (2) have one or both lips off angle, (3) be undercut at the corner, or (4) have an objectionable radius in the corner.

The time honored way to lick this problem was to undercut at the neck enough to make sure there was no radius which would cut, and after grinding, to lap the two cutting lips square on a piece of emery cloth while the tool is rotated in a drill press. There is a way which gives more predictable results faster.

Mount the tool in the R-O® grinding fixture and circle grind on the pilot in the conventional way to establish pilot size and length. Then, without disturbing the tool's position, swing the grinding wheel head to another position, and using a small wheel, relieve the square faces as shown in *Figure 15*.

Here again, use of a combination of axial and radial motion in the fixture clears the cutting faces and the corners. If the wheel has been dressed squarely and sharply, there will be no measureable difference in height or angle of the two lips when viewed on the optical comparator.

If one leaves a generous margin at the cutting edge, and if a tool is ground in this manner, it can be sharpened time and again without cutting off the pilot length each time, *by careful flute grinding*.

Trepanning tools are ground in much the same way, except that there the wheel, which should not exceed twice the major diameter of the tool, just does the undercutting work. It should be redressed using *diamond particle grit tool or phono point dressing tool* for form, before attempting to finish the profile.

# TOOL GRINDING FUNDAMENTALS

To sum up, the first problem facing every shop is finding a tool shape which will do the job. The second problem is finding a means of consistently duplicating that shape at a reasonable cost.

In complex forms, crush dressing is well worth a careful look. It, in effect, provides a “master” which can be laid on the shelf and used whenever needed in the future. If there is a question of whether the job will run again, it is less costly to keep a crushing roll on hand than to stock finished tools.

As for relieving the cutting edges of profiled tools, the matter is largely one of finding a way to get the right size and shape of grinding wheel into the proper location, and then having a means of holding the tool in relation to it.

We hope that the discussion given thus far will open up new areas of investigation for the reader.

## Conclusion

To get high quality tools out of the grinding room and back into use at reasonable cost, we need an accurate, versatile, simple grinding machine.

The machines must be accurate, for as we have tried to show, the tools that work best and last longest are the ones that are correctly ground. The machine should be versatile and simple so that it can handle just about any job that comes along.

The R-O® approach to the machine design problem is to mount a tool-holding fixture, which can circle grind and form relieve, and an optical comparator on a basic grinder designed for them. In this way a tool can be sized for diameter, backed off, and inspected without being disturbed or relocated. If you follow some rather simple charts, the tool which comes off the grinder should work.

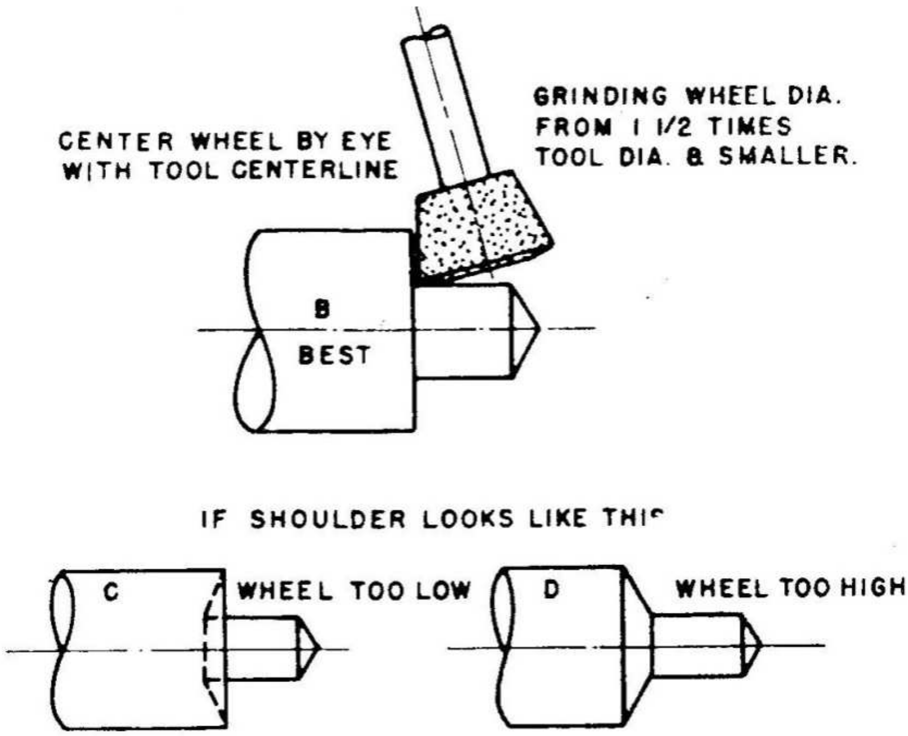



FIGURE 15

# TOOL GRINDING FUNDAMENTALS

## Summary

Finally, some odds and ends not covered elsewhere in this article. Let's start with tool thinning.

It makes good sense to *always* thin drills and flute-grind taps or reamers *before* end sharpening them. If you try to thin or face grind after sharpening, the chamfer or endworking angles are bound to be unequal, and smear metal often flows over the cutting edge. Most shops know this, but occasionally a plant attempts to thin as the last operation. It does not work as well.

If a hole already exists in a part, and just has to be opened up, try a three-lip core drill. It is more difficult to measure than a four-lip type, but it will be stronger. On a long run job, use a three-flute drill, buy the necessary measuring tools and charge them to the job. A plug soft-soldered in one flute will allow normal measuring tools to be used and can easily be knocked off after inspection. You will come out better in the end.

On small drills, chip breakers, are a problem. If one gets enough hook in the cutting face to curl the chip, the drill point is weakened. Sometimes you have to use chip breakers, but on a drill smaller than 3/8" diameter, it is often better to experiment with different spirals.

Chipbreakers, at best, require more of the point to be ground back when the drill is resharpened, calling for faster replacement of the drill, and added grinding time.

Chips in drilling do not cause trouble if you can get them to move out of the hole; that is why the change in the drill spiral. Somehow, you must get coolant to the drill point.

The ideal chip on most work looks like a figure "6" or if you prefer, a figure "9". On deep holes, a woodpecker attachment may help, but it is entirely possible to break chips so short they will not come out of the hole as well as if they were longer.

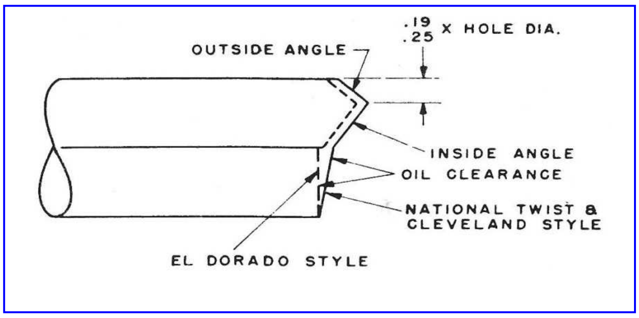
A drill point thinned to one-half the production standard will cut better, and last longer, and as mentioned earlier, the crankshaft point cuts freer and lasts longer.

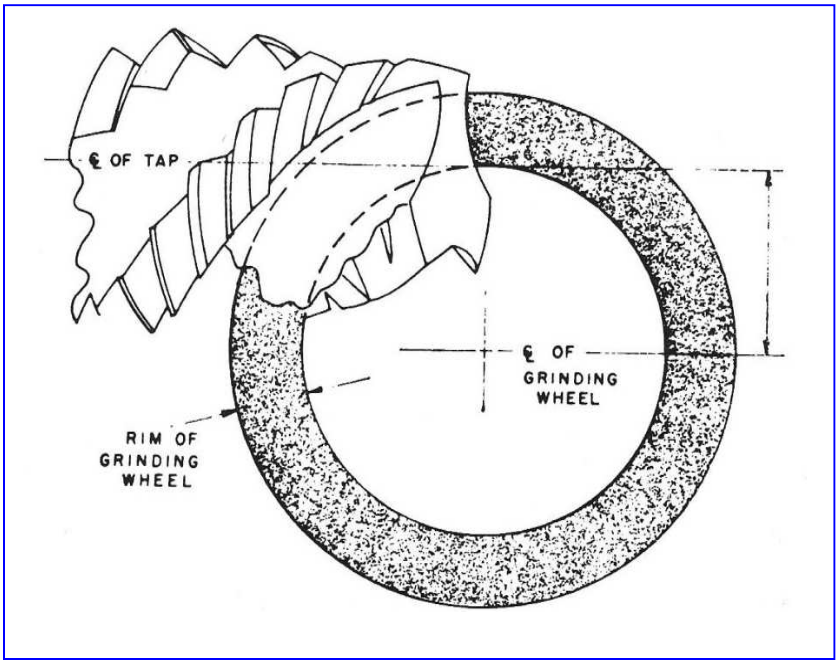
## Shapes

With the proper cam on the R-O® fixture, triangular, hexagonal, square, or elliptical shapes can be made in a single operation by continually rotating the part. This factor is part of the "versatility" feature mentioned earlier so that dies and punch shapes can be made. Customers produce cold heading tools in this manner, and it is a benefit whenever a shape has to be made for some tool room item.

Tool grinding is now changing from art to science. A great deal of credit is due the "old timers" who discovered how to make tools, and made them by hand. The need today is to automate as simply as possible the geometries they have discovered.

We hope that the information in this series has opened up new fields of investigation and experimentation for the reader. The better each individual understands tooling concepts, the more profitable the shop will be.





314 Fall Street, Seneca Falls, N.Y.
T: 315-568-5804 F: 315-568-5800
E: info@royaloakgrinders.com
W: www.royaloakgrinders.com