The use and care of TWIST DRILLS

The Use and Care of Twist Drills

This booklet, published by The Cleveland Twist Drill Company, is intended to aid in the proper use and care of drills. Because of the complexity of drilling, it is not possible to make this a complete treatise on the subject, but rather an introduction to the theories underlying the use of twist drills.

This booklet is intended as a text book for distribution to colleges, technical and vocational schools, both in the United States and abroad. It can also be used by manufacturers with apprentice schools who will find it an excellent manual. Anyone who is interested is welcome to copies of this booklet.

The material in this booklet was taken from the "Metal Cutting Tool Handbook", published by the Metal Cutting Tool Institute, 405 Lexington Avenue, New York 17, New York.

We recommend that those who are interested in all phases of cutting tools should avail themselves of the Metal Cutting Tool Handbook. It includes technical information on drills, reamers, counterbores, taps and dies, milling cutters, hobs and gears, and broaching tools. In addition there is a section on other engineering data.

THE CLEVELAND TWIST DRILL CO.

1242 East 49th Street • Cleveland 14, Ohio

Stockrooms:

New York Area (Englewood Cliffs, N. J.) • Atlanta • Cleveland • Detroit • Chicago • Dallas
Los Angeles • San Francisco

SPEEDS AND FEEDS

SPEEDS FOR DRILLING

The speed of a drill is usually measured in terms of the rate at which the outside or periphery of the tool moves in relation to the work being drilled. The common term for this is "Surface Feet Per Minute," abbreviated to S.F.M. The relation of S.F.M. and Revolutions Per Minute, or R.P. M., is indicated by the following formulas:

$$\text{S.F.M.} = .26 \times \text{R.P.M.} \times \text{Drill Diam. in Inches}$$

$$\text{R.P.M.} = 3.8 \times \frac{\text{S.F.M.}}{\text{Drill Diam. in Inches}}$$

In general, when operating a drill at a speed anywhere within its range for the particular material involved, increases in speed result in fewer holes before regrinding becomes necessary, and reductions in speed permit more holes before the tool is dulled. As a result, on every job there is the problem of choosing a speed which will permit the highest rate of production without entailing excessive drill costs or down-time for tool sharpening. The most efficient speed for operating a drill will depend on many variables, some of which are listed below:

1. Composition and hardness of material.
2. Depth of hole.
3. Efficiency of cutting fluid.
4. Type and condition of drilling machine.
5. Quality of holes desired.
6. Difficulty of set-up.

Speeds shown in the following table indicate the approximate range for efficient operation under normal conditions. On most jobs, adjustments from these speeds will be required to reach peak efficiency.

Suggested Speeds for High Speed Steel Drills

	Speed in S.F.M.
Alloy Steel......300 to 400 Brinnel	20– 30
Stainless Steel	30– 40
Automotive Steel Forgings	40– 50
Tool Steel, 1.2C	50– 60
Steel .4C to .5C	70– 80
Mild Machinery Steel, .2 to .3C	80–110
Hard Chilled Cast Iron	30– 40
Medium Hard Cast Iron	70–100
Soft Cast Iron	100–150
Malleable Iron	80– 90
Monel Metal	40– 50
High Tensile Strength Bronze	70–150
Ordinary Brass and Bronze	200–300
Aluminum and its alloys	200–300
Magnesium and its alloys	250–400
Slate, Marble and Stone	15– 25
Bakelite and similar material	100–150
Wood	300–400

Carbon Steel drills should be run at speeds of from 40 to 50 per cent of those given above.

To transfer the speeds given in "Surface Feet Per Minute" to R.P.M. for any size drill, see Tables on pages 55, 56, 57.

FEEDS FOR DRILLING

The feed of a drill is governed by the size of the tool and the material drilled. Since the feed partially determines the rate of production and also is a factor in tool life, it should be chosen carefully for each particular job. In general the most effective feeds will be found in the following ranges:

Diameter of Drill, Inches	Feed in Inches per Revolution
Under 1/8	.001 to .002
1/8 to 1/4	.002 to .004
1/4 to 1/2	.004 to .007
1/2 to 1	.007 to .015
1 and over	.015 to .025

INDICATION OF EXTREME SPEEDS AND FEEDS

A drill split up the web is evidence of too much feed or insufficient lip clearance at the center, due to improper grinding. The rapid wearing away of the extreme outer corners of the cutting edges indicates that the speed is too high. A drill chipping or breaking out at the cutting edges indicates that either the feed is too heavy or the drill has been ground with too much lip clearance.

CUTTING FLUIDS

It is a commonly-accepted fact that the use of cutting fluids is of advantage in most metal-cutting operations. Cutting fluids have many functions, several of which are dealt with in the following pages.

1. Lubrication

The purpose of lubricating a cutting tool is three-fold:

First, to lubricate the contact surfaces between the tool and the work, and so reduce friction and consequent generation of heat. When the tool is pressed against the work to begin a cut, a certain amount of sliding takes place before the tool actually "bites in." If the tool and work are well lubricated, the friction at this point is materially lessened. This lubricating action is continuous throughout the cut.

Second, to lubricate the chip and the top surface of the tool. Chips, as they are being separated from the work, tend to break and curl up and to slide along the top surface of the tool. In curling up, the chip is broken up into a large number of small pieces, and these pieces slide over each other. A lubricant will reduce friction between the chip and the tool and also reduce friction between the separate parts of the chip.

Third, to prevent sticking of the chips to the tool. If no lubricant is used, particularly in machining steel, the chips tend to heat to a point where they will fuse and stick to the tool. As a result, the tool will "choke up" and fail very quickly.

2. Cooling

In production machine work, the function of cooling the tool and the work becomes more important than that of lubricating. The amount of heat generated is usually so great that the life of the tool depends, to a large extent, on the ability to carry away this heat at the same rate as it is generated. For that reason lubrication must often be subordinated to cooling. If, by means of the cutting medium, the heat can be carried away as fast as it is generated, there will be no overheating, and the only dulling action will then be that of abrasion.

In order to obtain efficient cooling, as large a volume as is possible should be directed on to the cutting edges of the drill. In vertical drilling of moderate-depth holes, a stream is usually directed so as to fall onto the drill near the surface of the work. It then flows by gravity down the flutes of the tool to the cutting edges. When drilling deeper holes, an intermittent feed is often used, and as the drill is withdrawn to clear the chips, it is cooled by the cutting fluid, which also will fill up the partially-drilled hole, thus providing a reservoir of coolant. When bushings are used in vertical drilling, the coolant supply is usually directed onto the drill above the bushings and allowed to run down. Often the bushing plate itself has raised edges and acts as a reservoir of coolant, which runs down through the bushings.

In horizontal drilling, gravity will not help the cutting fluid in its path to the drill point. In such drilling the coolant is usually directed horizontally along a flute of the drill so that some of it is forced down the flute towards the point. By withdrawing the drill occasionally the hole is flushed out with cool cutting fluid. In horizontal drilling, particularly in thin-walled work, a large volume of coolant should be directed onto the outside of the work in order to help dissipate the heat.

In deep horizontal holes quite often it is found advisable to use oil-feeding drills. In such tools the cutting fluid under pressure is forced through the oil passages and is ejected next to the cutting edges. If only cooling is required, pressures from 25 to 50 pounds per square inch often provide an adequate flow of fluid. At such pressures little trouble is encountered with joints in the oil lines, and in many cases it is unnecessary to shut off the fluid supply when withdrawing the drill from the hole. The fluid supply for oil-feeding drills **must** be kept reasonably clean, as a small chip carried in suspension can block one of the oil passages.

3. Chip Disposal

When the cutting-fluid pressure is increased in oil-feeding drills, this fluid will help to wash the chips back through the flutes. For this purpose pressures up to 500 pounds are common, and many installations have been made using pressures from 700 to 1000 pounds per square inch. At these high pressures it is almost always necessary to have an automatic shut-off valve which will stop the flow of coolant before the drill is withdrawn. Failure to turn off the oil results in spraying it into the air, where it is lost.

In order that oil-feeding drills function efficiently, it is usually necessary that they form chips of such a shape as will most easily wash back out through the flutes. Perhaps the ideal chip is cone-shaped and composed of a single curl. Some materials form such chips naturally, but in some cases it is necessary to take steps to provide such a chip. Variations in feed will often result in a change in chip shape.

In many cases it is necessary to grind chip breakers in the point of the drill. No hard and fast rule can be made as to the required shape of chip breaker. The simplest form is illustrated in Fig. 1 and consists of a flat along the cutting edges, which reduces the rake angle of the drill. It is important in all types of chip breakers that the breaker does **not** extend **through** the margin of the drill.

Usually, however, the chip breaker will take somewhat the form shown in Fig. 2. The proportions of the dimensions shown are subject to wide variations and must be worked out for each job.

Types of Chip Breakers

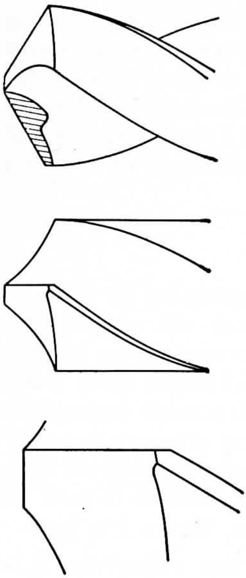


Fig. 1 Flat Type

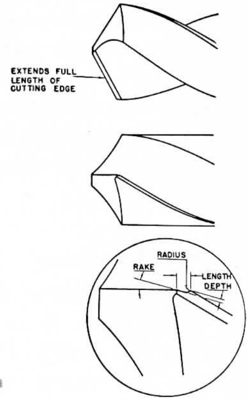


Fig. 2 Radius Undercut Type

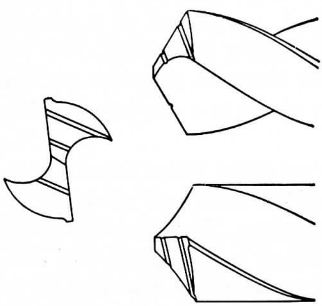


Fig. 3 Grooves in Lip of Drill

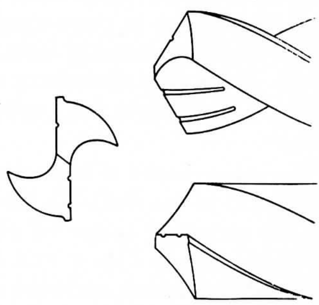


Fig. 4 Grooves in Web of Drill

The purpose of this type of breaker is to cause the chip to curl onto itself and break when it meets itself. Care must be used to keep the depth of the breaker at a minimum in order to prevent cutting through the margins and also to prevent forming a pocket into which the chip will wedge. Considerable ingenuity is usually required in the proper design of such chip breakers in order to accomplish the intended purpose. A properly-designed chip breaker should form chips which wash out through the flutes easily without mashing or further distortion. If the chips are caught in the hand, they should form in a steady stream and not come in intermittent bursts. A properly-designed breaker should add very little to the power required for drilling. If much added power is required, it is an indication of more heat and means shorter tool life. For the above reasons, such chip breakers are usually designed on the job, where their performance can be watched.

Occasionally it is found advisable to split the chips lengthwise, and this can be accomplished by either of the types of breakers shown in Figs. 3 and 4. These grooves should be spaced alternately, so that grooves in opposite flutes are not equidistant from the axis of the drill. Such grooves should be made with a wheel dressed to a radius to avoid localization of stresses in any sharp corners.

4. Finish in the Hole

Quite often when the finish in the drilled hole is important, a cutting fluid can be selected which will materially improve this finish.

SELECTING THE CUTTING FLUID

From what has been said it is evident that we should use as the ideal cutting medium an oil or compound that combines excellent lubricating qualities with equally good cooling qualities. Unfortunately, this cannot always be done, because many of the best cutting lubricants do not possess good cooling properties. On the other hand, we find that pure water is probably the best coolant available, yet its lubricating value is practically nil.

The sulfurized oils have been found to combine good lubricating and cooling qualities and are to be recommended for machining steel, particularly if a good finish is required.

Soluble oils and similar compounds can be used to good advantage where finish is less important. These preparations usually have excellent cooling properties. Care should be taken to see that all water-soluble compounds contain enough alkalinity to prevent rusting of the work.

Animal oils should be avoided, if possible, because of their bacterial properties and their tendency to become rancid.

Straight mineral oils can be used with fair success on medium heavy jobs, but they do not compare with the sulfurized oils.

It is suggested that cutting fluid problems be referred to a reputable manufacturer of such fluids. The following list should be used as a suggestion only:

Aluminum and its alloys:	Soluble oil; kerosene and lard oil compounds; light, non-viscous, neutral oil; kerosene and soluble oil mixtures.
Brass:	Dry; soluble oil; kerosene and lard oil compounds; light, non-viscous, neutral oil.
Copper:	Soluble oil; winter-strained lard oil; oleic acid compounds.
Cast Iron:	Dry, or with a jet of compressed air for a cooling medium.
Malleable Iron:	Soluble oil, or non-viscous, neutral oil.
Monel Metal:	Soluble oil, or sulfurized mineral oil.
Steel, Ordinary:	Soluble oil; sulfurized oil; high Extreme Pressure value mineral oil.
Steel, vary hard and refractory:	Soluble oil; sulfurized oil; turpentine.
Steel, Stainless:	Soluble oil, or sulfurized mineral oil.
Wrought Iron:	Soluble oil; sulfurized oil; high animal oil content, mineral oil compound.

Intermittent cooling of hardened steel should be avoided, as it may cause small checks or cracks which will result in tool failure.

The most important factor in maintaining ideal cutting fluids is to see that the fluids are kept clean at all times. If they are allowed to become contaminated and to carry large amounts of metal particles in suspension, their efficiency will be impaired. There are many commercial methods of cleaning cutting fluids from which the most suitable may be chosen. In many cases only a regular cleaning of settling tanks and machine will be necessary.

"CHATTER" IN METAL CUTTING

To those having experience with the cutting of metals, "chatter" is a familiar phenomenon. It can be defined as synchronized vibrations that are set up in the cutting tool, the work, the machine, or a combination of vibrations in all of these elements.

The cause of these vibrations, or chatter, usually is found in the lack of rigidity. This lack of rigidity permits the affected members to deflect under the cutting strains until the load builds up to a point where the material to be cut gives way. As the material is ruptured, the strains are lessened, and the deflected member springs back to its natural position; but the resistance begins to build up again at once, causing another deflection. This process, when rapidly repeated, sets up vibrations that we know as chatter.

The effect is a hammering action of the cutting edge or edges against the work. This hammering may cause the cutting edges to chip out in small pieces, or it may cause the body of the tool to fracture. Deflections, continued over a period of time, may also cause failure of the tool or of the machine by fatigue.

Investigators have attempted to show that metal can be cut with less expenditure of power if chatter is present. While this is possible because of the hammering action referred to, it is hardly an argument in favor of chatter. The disadvantages in the way of rough work and premature tool failures by far outweigh any power-saving gains.

The elimination of chatter, therefore, becomes a major factor of successful metal-cutting. The first step is to definitely locate the cause and then to make necessary adjustments in the set-up so that the trouble is remedied.

Looseness of moving parts of the machine is an obvious cause. This looseness may be due to wear or to careless adjusting. The remedy is to replace worn parts that cannot be adjusted and to take out all "play" where adjusting can be done.

Next in order is the question of rigidity of the machine itself. The drill spindle and other driving parts must be of sufficient strength so that they will not deflect under the cut to be taken. Substantial holding fixtures and adequate support of the work as a means of eliminating chatter is also a very necessary feature. The elimination of all unnecessary overhang will be found to be of great help. These provisions should be made in all metal-cutting operations where considerable strains are involved.

The proper design of cutting tools for the work to be performed must be considered as the next factor affecting chatter. Here the conditions may vary so widely that it is often necessary to study each individual application in order to obtain the best results.

When drills chatter, the cause usually is to be found in torsional deflection. The longer the drill in proportion to its diameter, the greater the danger of deflection and chatter. Where a shorter drill cannot be used, the remedy is to increase its cross section at points where the greatest resistance to torsion is offered. Naturally this must be combined with adequate chip space.

Whatever its cause may be, the elimination of chatter will pay big dividends in increased tool life.

THE MECHANICS OF CHIP FORMATION IN DRILLING

The success of any drilling operation, aside from the proper setup and speeds, depends largely on the ability to produce chips that are readily ejected from the hole. For this reason it is important to have a clear understanding of the mechanics of chip formation. If a correct analysis can be made of the conditions under which a drill must operate, it is usually possible to make the necessary provisions for proper chip formation.

On account of the limited chip room in a drill, it is always desirable to have the chips broken up into relatively small pieces. Coiling of drill chips, while spectacular in appearance, must be avoided, especially if the hole is of any considerable depth. These coils tend to pack up in the flutes and to stop succedding chips from coming out. In turn, this stops the flow of coolant to the cutting edges. Excessive heat and premature dulling will invariably result.

The two main factors that affect chip formation are:—

1. The ductility of the material to be drilled.
2. The thickness of chip, or feed per revolution.

When drilling ductile material, the chips tends to bend and coil up as they are being separated from the work. In less ductile materials the chips will tend to break up into pieces rather than coil. This latter is the desirable condition, and usually such materials can be drilled with the standard type of twist drills without any alterations.

When drilling the more ductile materials, it is sometimes necessary to go to rather extraordinary lengths to attain the proper breaking up of the chips so that they can be readily ejected.

The most common method is to decrease the rake angles of the cutting lips. As these angles are decreased, the effect is to cause the chips to bend more sharply. This usually will cause them to break, unless the material is extremely ductile. The amount of decrease of the rake angles can be varied to suit the material to be drilled.

The operation of grinding a drill so that it will have less rake angle is one that requires some skill. The grinding is done along the front face of the cutting edges, with the drill tilted so as to get the proper rake. A very slight amount of grinding is sufficient, and care should always be taken to see that the two lips are ground symmetrically. The rake angles must not be made negative, as this will materially reduce the efficiency and life of the drill.

The thickness of the chips also affects their tendency to coil up. The thinner the chips, the greater is this tendency to roll up into coils rather than breaking.

The clogging of drill flutes by coiled chips can often be remedied by increasing the feed per revolution; in other words, by increasing the thickness of the chips. Naturally this can only be done within the practical limits of the

A. RIGIDITY

It is always necessary to design and construct drill jigs so that the work is securely held against movement relative to the jig and the machine table during the drilling operation. It is further necessary that the work be so supported that it does not bend due to the drilling pressure. This is often difficult to do, particularly if the work is thin and frail; but it should be kept in mind that bending due to this cause will result in holes that are oversize and out of line, and may be responsible for much drill breakage. Thinning of drill webs and correct drill sharpening will help to reduce drilling pressure and avoid bending of the work.

B. CLEARANCE FOR CHIPS

Wherever possible to do so, enough space should be provided between the work and the drill bushing, so that chips can be ejected by this route rather than through the bushing itself. This will materially improve the performance of the drill by avoiding the clogging of chips in the flutes and in the bushing, and by admitting more coolant to the cutting edges.

The bushing should not be placed so far away from the work, however, that the drill is allowed to bend in the space between the bottom of the bushing and the work. If this bending is permitted, the drill will cut oversize. This may cause the drill to break, particularly if it is to enter the work at other than right angles to the surface. For angular entry, it is best to have the bottom of the bushing placed very close to the work until the drill has fully entered it. The bushing should be so arranged that after the drill point is well into the work, the bushing can be lifted out of the way of the chips.

C. BUSHING DIMENSIONS

From the standpoint of proper operation of the drill, the dimensions of the drill bushing are more important than its style of design. It must be assured, however, that the bushing is resistant to wear and that it is securely fastened in the jig.

The effective length of the bushing must be sufficient to support and guide the drill. A bushing that is too short structural strength of the drill. Inasmuch as the average drilling setup tends toward high speeds and low feeds, it is always advisable to look into this when trouble with coiling chips is encountered.

Drilling in screw machines is often limited to comparatively high speeds and low feeds, because here the drill must work in conjunction with one or more other tools whose performance would be seriously impaired if heavy feeds and slow speeds were used. On such operations a drill with less rake on the cutting lips is usually to be preferred. Drills having slow helix angles are then used. The slower helix angles automatically have less rake and do not require grinding on the face of the cutting edges, as discussed above. The slight loss in efficiency, due to slower helix, is more than compensated for in this case by better chip formation and disposal.

In extreme cases it has been found necessary to introduce chip breakers into the cutting edges of drills so as to avoid coiling and clogging. These chip breakers are usually notches or grooves ground into the face or the bottom of the cutting lips, as shown on page 8. The practice is to be discouraged because of the additional labor involved, and because of the skill required to produce chip breakers that really will do what they are intended to accomplish.

On account of the widely varying conditions under which drills must work, it is difficult to lay down hard and fast rules. Rather, each job must be studied from a chip formation standpoint, and the necessary remedies applied.

JIG DRILLING

The use of drill jigs when drilling holes in duplicate parts on a production basis has come into such common use that the process needs no detailed explanation here. The rapid and accurate location of holes and the guiding of drills by means of proper jigs have effected tremendous savings in the metalworking industries.

The purpose of this section is to call attention to such factors of jig and jig-bushing design and uses as may have a direct influence on obtaining the best results from drills.

will not keep the drill in line, because it permits the drill to bend with the short bushing acting as a fulcrum. Out of line and oversize holes may result.

On the other hand, if the bushing is longer than necessary, it will cut down the effective length of the drill, or it may necessitate the use of special-length drills, thus adding to the cost of the operation. For drills having average helix angles, the length of jig bushings should be from one-and-three-quarters to two-and-one-half times the diameter of the drills.

The diameter of the bushing hole should be kept very close to the diameter of the drill; but under no circumstances should the bushing be so tight as to prevent the drill from turning freely. A minimum clearance of from .0005 to .001 inch is desirable.

It is equally important that the clearance between drill and bushing not be too great. Chipped margins on the drill often result from this cause.

D. PASSAGES FOR CUTTING FLUID

In a previous section the advantages of using the proper coolant in sufficient volumes has been stressed. It has been stated that it is necessary to carry away the heat generated during the cut at the same rate that it is produced. To accomplish this, adequate passages must be provided in the jig so that the coolant can be applied at the proper point.

Often the coolant is thrown on the jig rather than on the work, and its benefits, therefore, are largely lost. The correct way is to have a good stream of coolant playing on the work at the point where the drill enters. If the cross section of the work is small, much of the generated heat is radiated to its surface, and this heat can be carried away by flooding the work itself in addition to cooling the drill in the usual manner.

DRILLING OF SMALL DIAMETER HOLES

Over a long period of years of experience and observation, a store of knowledge has been built up regarding the most successful practices to be followed when drilling holes in metals and other materials. Speeds, feeds, and sharpening techniques for various materials, as well as lubricants and other aids, have been fairly well standardized.

It has been found, however, that when holes of very small diameters are to be drilled (say from .015" to .040" in diameter), the standards of operation that are satisfactory for the larger sizes do not always hold.

There are several reasons for this:

1. These small drills are very long in proportion to their diameter (from 40 to 60 diameters long). They are, therefore, subject to much deflection, both longitudinal and torsional, if loaded in proportion to the larger diameter drills.

2. The webs or center sections of these small drills are proportionally much heavier than in large drills, since it is impractical to make them with correspondingly thin webs. This construction increases the required end pressure and decreases the chip space.

3. These small drills are called upon to drill comparatively deep holes, so that almost any small diameter drilling operation comes under the classification of deep-hole drilling.

Since the lengths of these small drills must be maintained for utility's sake, and since it is impractical to construct them with thinner webs, it follows that we must adjust the drilling conditions so that they will give a reasonable amount of service.

This involves:

a. Properly arranged feeds and speeds.

b. Guiding of the drill to minimize runout and deflection.

c. Frequent and adequate chip disposal.

d. Careful resharpening, performed often enough to prevent over-dulling.

FEEDS AND SPEEDS

Perhaps the two most common errors in the operation of very small drills are over-speeding and under-feeding. Feeds should be based on thickness of chip and not on inches per minute. Speeds should then be adjusted accordingly.

To illustrate:—Let us say that a No. 70 drill is to operate at a feed of .0005″ per revolution in drilling steel. While the material is soft enough so that it will permit a speed of say 80 S.F.M. or 10,912 R.P.M., it is obvious that the drill will not stand a load of .0005″ feed per revolution at this speed, for the rate of penetration would be 5½″ per minute.

The thing to do here is to cut down the speed and retain the feed per revolution until a satisfactory penetration rate is found, where the drill will stand the load.

Successful small-hole drilling depends on feeds that will actually produce CHIPS and not POWDER, and on speed adjusted to the strength and load-carrying capacity of the drill.

DRILL GUIDES

It is always advisable to use a guide bushing when drilling holes with these small drills. The guide bushing will prevent the drill point from weaving when the hole is started, and also helps to support the body of the drill so that it will not bend under end pressure. A small space should be provided between the bottom of the bushing and the work, so that chips can escape at that point.

CHIP DISPOSAL

Due to the limited chip space in these small drills, special precautions should be taken to get out the chips; otherwise, the drill will choke up in the hole and break. Step drilling, as commonly practiced with larger, deep-hole drills, is, therefore, recommended. The best way is to feed the drill only an exact predetermined distance each time before withdrawing for chip disposal. This distance should be something less than that required to pack up the flutes of the drill with chips.

SPEEDS AND FEEDS FOR DEEP-HOLE DRILLING

The planning of drilling operations often fails to take into account the depth of holes to be drilled and its influence on permissible rates of feeds and speeds.

On page 4 a table of feeds and speeds has been given that can safely be followed for average conditions. If the hole to be drilled equals or exceeds three times the diameter of the drill, the operation falls in the deep-hole-drilling class, and some adjustment of feeds and speeds is necessary in order to secure maximum efficiency. In considering the depth of the hole, the effect of bushings must be taken into account. A drill bushing set very close to the work becomes in effect an extension of the hole itself, and on holes from 2 to 3 diameters deep complicates the problem of chip disposal.

In its final analysis it is entirely a matter of conducting away the heat generated and of preventing its accumulation. It is an evident fact that if less heat is conducted away than is generated, the drill will eventually overheat and be destroyed.

While the nature and amount of coolant applied to a given drilling operation will affect the depth of hole that can safely be drilled at ordinary feeds and speeds, the above-mentioned depth of three diameters is a good point at which to begin the stepping-down. This applies particularly to speed.

The table below will serve as a guide to the proper reductions as the depth of hole increases:

SPEEDS

Depth of hole	Reduction of Speed, Per Cent
3 times drill diameter	10
4 times drill diameter	20
5 times drill diameter	30
6 to 8 times drill diameter	35 to 40

FEEDS

Depth of hole	Reduction of Feed, Per Cent
3 to 4 times drill diameter	10
5 to 8 times drill diameter	20

If excellent cooling conditions are present, the above rates may be increased; but if there is little or no provision for cooling, it may be necessary to step down even lower than the figures given.

It should be kept in mind that a somewhat lower rate, constantly maintained, will show more production per day than a high rate with long and frequent interruptions. In the latter case, too, the tool costs will be much higher.

CHIP DISPOSAL IN DEEP-HOLE DRILLING

The ultimate success in drilling deep, small-diameter holes on a production basis hinges on the ability of getting the drill chips out of the hole. This will prevent packing and binding and will permit the cooling fluid to reach the point of the drill.

As the diameters of holes to be drilled become smaller, a point is reached where the chip spaces or flutes are too small to allow the chips to pass out of the deep hole properly. The only remedy is to lift the drill out of the hole at intervals. The drill then carries the chips out with it and permits a fresh start to be made. If this chip removal is done often enough, there is hardly any limit to the depth of the hole that can be drilled.

Drilling machines are available today which automatically perform this function. They are usually arranged to withdraw the drill once for each advance in depth equal to the drill diameter. Performances of small, deep-hole drills have been greatly enhanced in this way, both as to number of holes drilled per grind and number of grinds per drill. Inasmuch as the removal and re-insertion of the drill is automatic and extremely rapid, little time is lost on this account.

If the holes to be drilled are $\frac{3}{4}''$ diameter or larger, an oil hole drill can be used to advantage; that is, a drill with an oil hole through its entire length, so that fluid can be carried to the cutting edges. If, on the other hand, as is usually the case, the holes to be drilled are from $\frac{3}{16}''$ to $\frac{3}{8}''$ in diameter, this type of drill cannot be used. For the latter purpose, special, deep-hole drills have been designed. These differ from ordinary twist drills, principally in that the web is much heavier and the helix somewhat faster. The web-thickness in some cases may be 40 per cent of the diameter of the drill.

This in turn necessitates a special type of sharpening so as to break up the chips effectively, and to eliminate undue end-thrust on the drill. The most approved type of sharpening is shown in Fig. 5 on next page. It should be noted that the drill is first sharpened in the conventional way, after which the angular cuts are made with a grinding wheel having a sharp corner. For resharpening methods, see page 39.

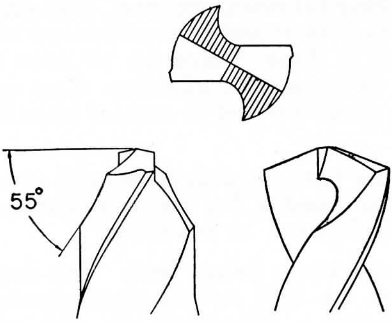


Fig. 5 Approved Type of Sharpening
Deep-hole Twist Drill

Inasmuch as the construction of this type of drill must be varied to suit conditions and materials, it is recommended that deep-hole-drilling problems be submitted for engineering advice.

DRILLING DIFFERENT TYPES OF MATERIALS

DRILLING OF HARD MATERIALS

It is sometimes necessary to drill holes in hard, refractory materials, such as heat-treated alloy steels, hard steel castings, partly-chilled cast iron, and some stainless steels.

While there is a practical limit of hardness beyond which such materials cannot be drilled with high speed steel drills, the procedure here recommended may aid in drilling emergency holes in materials that otherwise would be considered unmachinable.

The recommended procedure is as follows:

1. See that the work is well supported directly under the drill point; also that it is held rigidly in place.

2. Inasmuch as this type of drilling calls for more than the ordinary amount of end pressure on the drill, the work should be done on an oversize drilling machine, if possible.

3. Use a short, stubby drill. Do not thin the web too much. Thin webs are an advantage, however, in the drilling of work-hardening stainless steels. Use only a medium amount of lip clearance (7° to 9° is ample).

4. As a cutting lubricant for steel, use sulfurized oil or carbon tetrachloride mixed with sulfurized oil in the proportion of one to three. Kerosene, or even pure carbon tetrachloride, is sometimes effective. Cast iron is drilled dry or with a small amount of carbon tetrachloride in extreme cases.

5. Speeds are very important, as hard materials cannot be drilled at high speeds. Try the range between 20 and 30 surface feet per minute, or even less.

6. Use a medium to heavy feed. The important thing here is to keep the drill cutting continuously once it has started. If the drill is allowed to idle and rub on the cut surface, the material will work-harden to a point where it is unmachinable. Power feed, preferably mechanical, should, therefore, be used in all cases.

MACHINING OF STAINLESS STEEL

Stainless Steels are, as a rule, somewhat more difficult to machine than are straight carbon steels or even most grades of alloy steels.

There are some free-machining grades of stainless steels, but the 18-8 types are usually found to be difficult due to their work-hardening properties Because of these properties, the first rule of machining stainless steel is to keep tools cutting continuously. If tool edges are permitted to idle and rub on the work, the surface will be work-hardened to a point where it is difficult, if not impossible, to restart the cut. For the same reason, it is advisable to use fairly heavy feeds, so that the tool edges will get under the slightly work-hardened surface left by preceding edges or cuts.

It follows from the above that maximum production can best be obtained by first establishing the proper feed rate per tooth or per cutting edge and then increasing speeds to the point permitted by the nature of the material. In general, the feed rates may be established on the same basis as for other types of steels; but the speeds may be as low as one-half or less of those commonly used for mild and free-machining steels.

Tools must be carefully and accurately sharpened for best results in stainless steels. A good finish on the cutting edges and moderate clearances will be found most satisfactory.

Twist drills should be sharpened with an included point angle of 135° to 140°. The clearance angle should be 6° to 8°. When it is necessary to start drilled holes from center-punch marks, a triangular punch should be used, so as to avoid work-hardening of the surface as much as possible.

When drilling deep holes, it is necessary to withdraw the drill several times in order to clear the chips out of the hole. Under this procedure it is extremely important that both the withdrawal and the re-starting be done quickly, so that the drill will not idle in the bottom of the hole and thus work-harden the surface.

The proper coolant plays an important part in the successful machining of stainless steels. For the free cutting grades the ordinary soluble types of coolants will be satisfactory. For the harder or more difficult cutting grades it is necessary to use sulfurized mineral oils, or these in a mixture with lard oils. In extreme cases it has been found necessary to use turpentine or carbon tetra-chloride, either alone or in mixtures with sulfurized oils. This should be avoided when possible, unless good ventilation is available, because turpentine is inflammable, and carbon tetrachloride fumes are toxic. Both are disagreeable in odor. Heating of the part to be drilled, prior to drilling, has been resorted to in some cases, but this is hardly practical except for special applications.

The main factors to keep in mind for the successful machining of stainless steels are: Adequate and uniform feeds, moderate speeds, carefully-sharpened tools, and good coolants suited to the type of material to be machined.

DRILLING OF ARMOR PLATE

Armor Plate presents some rather difficult problems due to the hardness and extreme toughness of this material. In general, this material can be drilled satisfactorily with regular, high-speed steel drills. Due to both work-hardening and flame-hardening properties of armor plate, the technique of cutting with these tools differs in important respects from that used for machining ordinary steels.

If the material has been flame-cut, the areas adjoining will have hardened to a point where drilling will be extremely difficult. Annealing of such areas is advisable, if possible, before machining. Where this is not practical, a reduction in cutting speed is the only other solution. Speeds as low as 15 surface feet per minute are found necessary in some cases.

The next thing to watch is that the tool is cutting continuously. Therefore, it is always recommended that a positive power-feed be used for all machining of armor plate. No matter what kind of tool is used, if it is allowed to idle and rub without cutting, the surface will work-harden to a point where it is practically impossible to re-start the cut in that particular spot.

Further, if too light a feed is used, this same work-hardening effect may develop. A medium-heavy feed, of uniform rate, proves most satisfactory for all types of tools.

Some armor plate is of the laminar type; that is, one side of the plate is harder than the other. Drilling of such plate should always start from the hard side. It is practically impossible to drill successfully from the soft side.

Drill sharpening plays an important part in the machining of armor plate:

First, cutting edges must be well supported. This means that the clearance should be sufficient to clear the work, but never excessive. (See a following paragraph).

Second, rake is of advantage, but again this should be moderate, as excessive rake also tends to weaken cutting edges.

Third, the finish of the cutting edges should be as smooth as possible and be free from even the slightest evidence of wheel burn. Wheel burning creates fine, invisible, hairline cracks in cutting edges, that will result in premature failure.

Twist drills for drilling armor plate should be sharpened with an included point angle of 135° to 140°, a clearance angle of 6° to 8° at the periphery, and a chisel-point angle of 115° to 125°.

Cutting speeds will vary widely due to material and operating conditions. Drilling speeds will ordinarily be in the range of 40 to 50 surface feet per minute for shallow holes. For deeper holes this must be reduced as required to get reasonable tool life.

Feeds should be from 15 to 25 per cent less than those recommended for ordinary steels; but it should be emphasized again that the real remedy for cutting difficulties in armor plate is a reduction of speeds rather than feeds.

A large volume of coolant should always be used and directed to the point of contact between the tool and the part to be drilled. Sulfurized mineral oil is the most satisfactory coolant and should be used if possible. Otherwise an 8 to 1 mixture of soluble oil may be used.

As in the case of other hard and tough materials, successful armor-plate machining finally depends on the use of rigid machines, fixtures, and tool holding devices. Deflection of any of these members will cause "riding" of tools and consequent work-hardening of the material to be machined.

DRILLING OF MOLDED PLASTICS

Molded Plastics are finding an ever-widening field of application. The development of various types of plastics, with different characteristics, has presented to the tool manufacturers certain problems in cutting-tool designs.

Molded Plastics are now produced from a variety of base materials and are known by such names as Resins, Phenolics, Cellulose Acetates, Polystyrenes, Acrylics, etc.

In general, these materials are easily machined in their pure state; but some of them are used with abrasive fillers, which, in turn, change their machining characteristics.

Drilling of Molded Plastics can be troublesome if certain basic principles of drill design and operation are not observed. The ejection of chips is the most important problem. Drill design must always take into consideration the fact that chips tend to stick and pack in the flutes of the drill, thereby causing rubbing and heating. Overheating may be destructive of both tool and work, inasmuch as most plastics are comparatively low-temperature compositions.

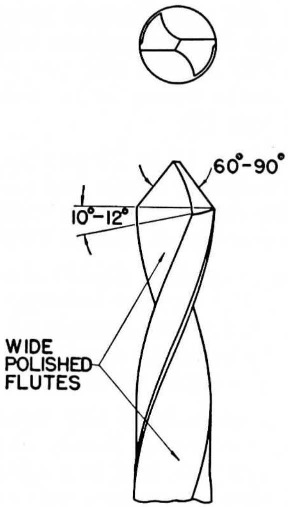


Fig. 6 Drill for Molded Plastics

For drilling of small-diameter holes, a special drill, substantially as shown in Fig. 6, has been developed. This type is generally known as a drill for plastics. As will be noted, it has wide flutes and a comparatively low helix angle. The included point angle is usually 60° to 90°, instead of the conventional 118°. Clearance at the periphery is usually about 12°.

For drilling larger-diameter holes, a point with a larger included-angle (90° to 120°) can be used to advantage. Also, on these larger drills the heel behind the cutting edge may be ground away, leaving a land about 1/16" wide. This further reduces friction in the bottom of the hole and provides more chip clearance.

If drills have a tendency to pull when breaking through, it is advisable to flatten slightly the front of the cutting-edge, similar to the sharpening used for soft brass.

For blind holes, a fast helix drill is often used to advantage, as it will pull the chips away from the cutting edges more quickly; but for through holes the slow-helix drill shown is generally better.

Webs of drill should be thinned as much as possible, so as to reduce the friction and heat at the center of the drill point.

Due to the wide variety of materials under the classification of Molded Plastics, it is not practical to set any hard and fast rules for cutting speeds. Most of these materials can be drilled at speeds ranging from 100 to 300 S.F.M., depending on the depth of hole and the characteristics of the material.

If the material contains a large amount of abrasive filler, the speed must be materially reduced. Also, the Celluloses, Polystyrenes, and Acrylics must be drilled at slower speeds, because of their tendency to heat up and become gummy. Feeds should be kept at or above those used for metallic materials. Mechanical feeds are recommended, because of the continuous and uniform rate thereby attained.

Resins and Phenolics are usually drilled without a liquid coolant. A jet of compressed air, directed to the point where the drill enters the work, will be found of assistance in keeping both the drill and the part cool.

Celluloses, Polystyrenes and Acrylics are drilled with water, or preferably soap solutions, as a coolant. For shallow holes soap may be rubbed directly on the drill with good results.

Large holes in sheet materials are usually produced with either trepanning tools or with fly-cutters.

DRILLING OF SHEET METAL

Drilling of sheet metal and other thin sections presents a special problem because of the extraordinary conditions encountered in this type of operation.

As a rule, the main difficulty in drilling thin sections is the breakage of drills, particularly in the smaller-diameter holes. This is due to several factors:

First: Drilling is often done with a hand-supported power tool, such as an electric or pneumatic drill. Therefore, the drill is not held rigidly in the line of feed, nor is the feeding pressure constant. Further, there is no rigid support when the drill breaks through, so that a distinct shock is produced at this point.

Second: In many cases, due to feeding pressure, the metal to be drilled is deformed before the drill actually begins to cut. Thus, when the drill does begin to cut, it has the entire torque load thrown on it at once, instead of starting in gradually, as in ordinary machine drilling of heavier sections.

Third: The deformation referred to may also cause the metal to work-harden, and so further increase the load on the drill.

It is not always practical to remedy these conditions by changing the operation or the equipment. The real remedy usually is to use drills of the proper design for the work at hand.

Generally speaking, the ordinary drill is not sturdy enough for production sheet-metal drilling. In its place should be used a drill having much shorter flutes and heavy-web construction. Ordinarily, drills are designed to drill comparatively deep holes, and, therefore, must have long flutes. For sheet metal work the flutes can be very short, and as chip space is not so important either, the cross-section can be kept quite heavy throughout. The web should be of standard thickness, or less, at the point of the drill, so as to keep down the end thrust.

In emergency cases, where short-flute drills are not available, a standard drill may be used fairly successfully by cutting off about one-third of the flute and resharpening. In this way the drill becomes sturdy enough to avoid excessive breakage.

DRILL AND COUNTERSINKS COMBINED

When metal parts are to be machined on centers, as in turning or cylindrical grinding, it is always necessary to Drill and Countersink the center holes to serve as seats or bearings for the machine centers.

Two types of Drills and Countersinks Combined are available; namely, the Regular Type and the so-called Bell Type.

The Regular Type will produce center holes as indicated in Fig. 7. For the ordinary run of work this kind of center hole is quite satisfactory, and for that reason the Regular Type is by far the most commonly used.

If the parts to be machined are to pass through a number of subsequent handlings, where there is danger of marring the edges of the center holes, it is advisable to use the Bell Type and to produce center holes as indicated in Fig. 8. The outer edges are beveled to prevent damage to the center hole itself.

Where a maximum amount of protection of the center hole is required, this can be attained by making the depth greater, as shown in Fig. 9. For arbors or other parts, where the center holes are to be used repeatedly, this latter type should always be used. In making this type of center hole, care must be taken to drill only deep enough to give the necessary protection. If the drill penetrates too far, the machine-center may ride on the corner of the straight, large diameter rather than on the 60° surface.

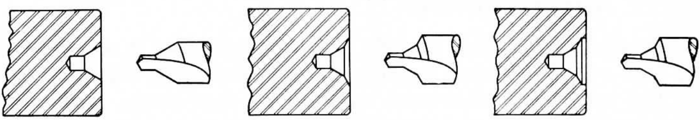


Fig. 7 Fig. 8 Fig. 9

Different Types of Center Holes

SHARPENING

The drill point may be re-sharpened in the conventional manner; but the re-sharpening of the beveled portions is a more difficult procedure. Grinding of the faces of the beveled portions is difficult to perform accurately. It will also tend to widen the flutes, thereby weakening the tool. Grinding on the outside to reproduce the original shape is the only satisfactory way of re-sharpening these beveled parts.

OPERATING PROCEDURE

The importance of properly performing this seemingly simple operation is often not fully realized. It is a fact that the success in general, the quality, and the accuracy of subsequent machining operations depend greatly on good center-holes, for these holes control the location and rigidity of the work supports during machining.

FEEDS AND SPEEDS

The permissible surface speeds of these tools must be based on the largest diameter that comes in contact with the work, which means the body diameter rather than the drill-point diameter. This is sometimes overlooked with disastrous results.

Feeds per revolution, on the other hand, must be based on the diameter of the drill point and should correspond to those given in the table of feeds for drills.

Proper Countersinking involves the following:

1. Correct positioning of the center hole.
2. Adequate width of center bearing surface.
3. Correct angle, roundness, and smoothness of the bearing surface.

Positioning of the center hole necessarily is a product of the accuracy and rigidity of work-holding fixtures and of the centering machine itself. It is, therefore, important that both fixtures and machines be maintained in first-class condition at all times.

Cylindrical pieces whose outside surfaces are reasonably round and smooth are best centered by rotating the work in roller rests, holding the center-drill stationary, though this is by no means universal practice today. Work that is not round or smooth must, of course, be held in stationary jaws or clamps, with the center-drill rotating.

Adequate width of center-bearing surfaces is important because this will insure good support for the part to be machined. It will also prevent distortion of the center hole on heavy operations and will avoid scoring of centers.

Widths of center-bearing surfaces should be in reasonable proportion to the size and weight of the part to be machined, but must also take into account the tool pressures that are developed during machining. Thus, it is necessary to have larger centers for milling and heavy-turning operations, than, for example, grinding and light-turning operations.

When Bell Type Drill and Countersinks are used, the width of bearing surface is automatically maintained.

For the majority of centering operations, the standard type of Drill and Countersinks is used, and it is here that the width of bearing surface must be watched closely.

The following figures will serve as a guide for checking the proper diameters of center holes:

Size of Drill & Countersink	Diameter of Hole	Size of Drill & Countersink	Diameter of Hole
A-1	$3/32$–$7/64$"	J-1	$3/8$–$13/32$
C-2	$5/32$–$3/16$"	J-2	$13/32$–$7/16$"
D-1	$11/64$–$13/64$"	M-1	$15/32$–$1/2$"
E-1	$15/64$–$17/64$"	M-2	$1/2$–$17/32$"
E-2	$1/4$–$9/32$"	N-1	$9/16$–$19/32$"
F-1	$5/16$–$11/32$"	N-2	$19/32$–$5/8$"
F-2	$11/32$–$3/8$"		

As stated above, these dimensions must be varied to meet special conditions of work weights and tool pressures.

Positive end stops on the work and on the drill spindle should be used at all times, so that uniformity can be maintained. This will be found of value on subsequent operations.

Inasmuch as accuracy and general quality of workmanship on subsequent operations is largely controlled by the quality of center holes, this latter factor is worthy of careful attention.

The first requirement for such quality is the use of Drills and Countersinks having correct bearing surface angles. Secondly, these cutting edges must be maintained true and sharp and the tools removed from use before excessive dulling impairs their accuracy. The operator should be instructed in the correct technique of feeding so as to produce smooth, undistorted center bearing surfaces on the work.

Careful attention to the various items mentioned will be repaid by better and more accurate work, less spoilage, and other manufacturing economies.

THREE AND FOUR GROOVE DRILLS

Three and Four Groove Drills are used for enlarging holes that have previously been drilled, cored, or punched. Because of being used in cored holes, they are often called "Core Drills." Their construction is such that the center portion will not cut. The amount of stock they will remove is limited by the depths of the flutes.

The purposes of using Core Drills instead of ordinary two-fluted drills for enlarging are two-fold:

First, because of greater productivity.

The material to be drilled and its hardness determine and limit the maximum surface speed and the feed per cutting edge. Now, if we use drills with three or four cutting edges instead of two, it is evident that a greater feed per revolution can be used, and more holes can be drilled in a given time. The increase in production should be fairly proportional to the number of cutting edges, though this may be limited in some cases by the strength and rigidity of the setup and by the quality of finish desired. Core Drills, because of their heavy center section, will stand the additional strain imposed by the greater number cutting of edges.

Second, because of the better finish obtained.

Core Drills having three or four cutting edges also have the same number of lands extending along their diameters. This multiple number of lands will support the drills much better while cutting than if they had two lands only. The result is less tendency to wobble, to score up the walls of the drilled hole and to cut oversize. In fact, the action of Core Drills is much the same as that of Rose Reamers. Often it is found practical to use Core Drills in place of Roughing Reamers.

To obtain the best results, it is highly important that Core Drills be properly sharpened. Care must be taken to see that all cutting edges are exactly the same length, and that they have the same point angle. While this is obvious, it is a detail that is often overlooked. Uneven lengths of the cutting edges will cause a disproportionate strain on the longest ones, resulting in quick dulling and a tendency to crowd the drill to one side. In turn, this will cause holes to come oversize. Further, there is then danger of wearing down the lands at the point, causing the drill to squeal and bind in the holes. It is recommended that a dial indicator or some similar device be used for checking the lengths of cutting edges before putting such drills in operation.

This type of tool is quite often used as a roughing reamer in order to correct the location of a previously formed hole. When used for this purpose, a rigid machine and fixture are required. The fixture should have accurately located drill bushings to guide the tool. When used for this purpose, it is recommended that the drill be pointed with a very flat point angle as shown in the illustration.

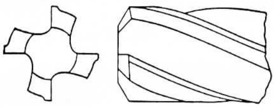


Fig. 10. Regular Point for 3 or 4 Fluted Drill

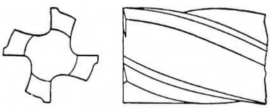


Fig. 11. Point for Correcting Location of Previous Hole

This very flat point angle (often 180°) reduces the tendency of the drill to follow the previously formed hole. It is most important that no chamfer be used on the corners of the point and that the drill be kept quite sharp. As soon as the corners start to wear the tendency to follow the previous hole is increased.

For most applications there is little choice between a three- and four-fluted drill. A three-fluted drill has slightly more chip clearing ability, but most users find the four-fluted drill a little easier to resharpen and measure.

Core Drills are sometimes made with a replaceable drill tip. The tip is fitted by some mechanical means, such as a taper and locking arrangement, and may be removed when worn or used up. Since the lands of the holder are not cleared, the wide bearing area makes it adaptable to bushing-guided operations.

CENTER DRILLS

Fig. 12. Center Drill

These drills, of very short overall and flute length, have been designed for centering work, where their extreme rigidity would reduce runout and eccentricity. For the same reason they have been quite successful as starting drills in screw machine-drilling.

COMBINED DRILL AND COUNTERSINK

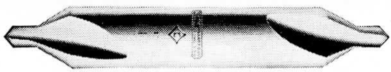


Fig. 13. Plain Type

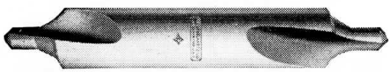


Fig. 14. Bell Type

Combined Drill and Countersink Center Drills are used for centering the ends of work where the work is subsequently to be revolved on machine centers. The angle of the countersink part is made the standard 60° included angle to fit all standard centers.

On precision work, some users prefer the Bell-Type, as the additional small 120° angular chamfer protects the edges of the center hole and reduces the danger of inaccuracies due to marred center holes.

For suggestions on use of Combined Drills and Countersinks, see pages 28, 29, 30, 31.

MULTIPLE-DIAMETER DRILLS

It is often possible to drill two or more diameters in a hole with a single drill of the proper construction. Such drills are, in fact, used quite extensively in the mass production industries and are effecting many notable economies.

Multiple-Diameter Drills are sometimes confused as to types and designations. In the following pages are enumerated the various types of such drills, as well as their common applications.

The simplest form of Multiple-Diameter Drill is the combined Drill and Countersink described above. Its uses are well known and need no further explanation here.

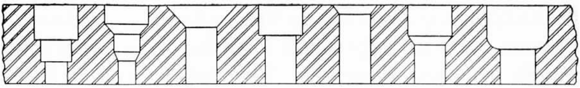


Fig. 15. Types of single operation possible with Multiple-Diameter Drills

All the types of holes shown in the illustration and many others often can be produced in a single operation by Multiple-Diameter Drills.

The following are the types of tools used for such operations.

THE STEP DRILL

A Step Drill may have two or more diameters, produced by grinding various successive steps on the lands of the drill. These steps are usually separated by square or angular cutting edges, as determined by the application. The Step Drill is useful for most of the jobs requiring multiple-diameter drilling. It is extensively used because it can be made by grinding down and stepping an ordinary drill. Some thinning of the web is usually required when so made.

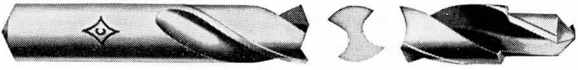


Fig. 16. A Two-Diameter Step Drill

Fig. 17. A Three-Diameter Step Drill

Step Drills can also be made in three- or four-fluted drill styles. Fig. 18 shows a three-diameter, four-fluted step Drill. The small diameter in this case is a non-cutting pilot, and the flutes may not extend through it.

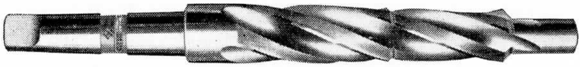


Fig. 18. A Three-Diameter, Four-Fluted Step Drill

In resharpening Step Drills the cutting shoulders as well as the point must be reground in order to maintain the correct length for each diameter. The regrinding of the shoulders almost always results in the nicking of the next smaller diameter. Continued regrinding of step tools usually results in a condition like that shown in Fig. 19.

Fig. 19. Appearance of Step Drill after continued regrinding

When this condition is reached, the drill can no longer be reground and can only be salvaged by a complete reestablishment of a new small diameter.

SUBLAND DRILLS

The Subland, Multicut, or Dual-Cut Drill, as it is variously called, overcomes this difficulty. This type of drill, shown in Fig. 20, performs the same functions as the step drill described above, though its construction is somewhat different. While the step drill has its step or different diameters on the same land, the Subland Drill has two distinct lands running substantially the entire length of the flutes. This is shown in the illustration. The advantage of the Subland Drill is that the two diameters may be maintained constant throughout the life of the drill because in resharpening the cutting edges of the large diameter it is not necessary to touch the margins of the small diameter. It is somewhat more expensive to manufacture than the Step Drill, but the convenience and economy in resharpening often outweigh the difference in first cost.

Fig. 20. Subland Drill

Subland Drills, like Step Drills, can be made in more than two diameters and are not necessarily restricted to two-fluted tools.

Fig. 21 shows a two-fluted, three-diameter subland drill while Fig. 22 illustrates a two-diameter, four-fluted subland drill.

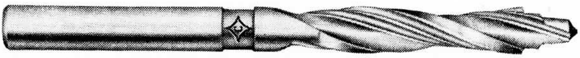


Fig. 21. Two-Fluted, Three-Diameter Subland Drill

Fig. 22. Four-Fluted, Two-Diameter Subland Drill

Subland Drills, because of the nature of their construction, are slightly weaker than the more conventional Step Drill, and it becomes wise to consider this in determining which type of construction to use. In general, it has been found unwise to use the subland type when the ratio of the larger diameter to the smallest is greater than two to one.

Another consideration to be taken into account is the length of the smaller diameters. If the length of a small diameter is great, there is little advantage in the Subland type, as the conventional type will give ample life before the small diameter is used up. The advantage of the subland type begins to become very evident as the small diameter is shortened. The tool shown in Fig. 23 would necessitate very costly regrinding if made in a Step Style, because each regrind would completely destroy the next smaller diameter. This regrinding is considerably simplified if the tool is made of subland style.

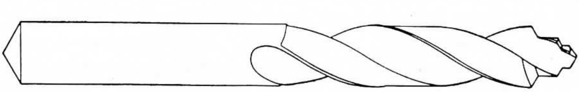


Fig. 23. Condition where Step Style is not suitable

ECONOMY IN DRILL USAGE

The problem of obtaining maximum economy in the use of twist drills is one which faces every user whether large or small. The solution to this problem lies in a constant awareness of the factors which promote such economy. Some of these factors are listed in the following, and many of them are covered in detail on other pages of this drill section.

1. PROPER APPLICATION OF DRILL

There are many designs of drills available to the user. The engineering features of each design have been determined so that the drill will perform satisfactorily in the class of work for which it was designed. While it is true that a drill of almost any design can be used to drill occasional holes in almost all materials, nevertheless, when production runs are considered, it is important that the correct drill be used on the job. Descriptions and typical applications of the more common types of drills are given on earlier pages in this booklet.

2. PROPER CONDITION OF DRILLING MACHINE

The drilling machine should be rigid and should have sufficient strength and power to withstand the forces of the cut. Overhang of the drill spindle should be kept at a minimum. Spindle bearings, both thrust and radial, should be in good shape to prevent spindle run-out and end-play. Back-lash in the feeding mechanism should be kept at a minimum to reduce the strain on the drill when it breaks through the bottom of the hole. Jigs and jig-locating fixtures on indexing machines must be free from play and firmly seated.

3. PROPER HOLDING OF DRILL

The inside of the hole in the drilling machine spindle should be free from burrs and scratches which can cause a run-out of the drill. Similarly, drill sleeves and shanks and all mating parts in the drill-holding device should be free from burrs and should be carefully wiped before inserting the drill. Drills should be inserted in the holder without being driven in by a steel hammer on the drill point. This almost always will result in chipping of the cutting edges. Drills should be removed from taper sleeves by using a drill drift—never by using the tang of a file, which will have a peening action on the tang of the drill.

4. PROPER CONDITION OF MATERIAL BEING DRILLED

Much needless destruction of tool edges and consequent shortened tool life is caused by poor surface conditions of the work to be machined. This fact is not always appreciated when premature dulling of tools is encountered in the shop.

Poor surface conditions may have various causes, such as:

> Castings may have sand pockets in the surface or have sand or other foreign materials actually imbedded in it. Chilled spots often occur, too, in the surface of castings, and these are as destructive of tool edges as are foreign materials.
>
> Forgings may have hard, oxide scales either tightly adhering to, or actually imbedded in the surface. Hard spots in the surface of forgings may occur with some types of materials.
>
> Bar stock may have the same poor surface conditions as forgings.

All of the above conditions have one thing in common; namely, that they cause rapid dulling of tool edges.

The cure is to see to it that surfaces of work to be machined are properly conditioned beforehand. This may be accomplished by blasting, tumbling, pickling, washing, normalizing, annealing, or a combination of these operations, as called for.

On the operating side, it should always be remembered that there is a distinct saving of tool life by making the tool cut under the surface at all times If tools are allowed to scrape or rub on the surface of the work, the edges will dull quickly even with reasonably good surface conditions.

5. PROPER RECONDITIONING PROGRAM

Drills, like all other cutting tools, should not be allowed to become so dull that they refuse to cut.

Over-dulling of metal-cutting tools will inevitably result in:

> A reduction in productive capacity.
>
> Poor finish in the product.
>
> Loss of accuracy, with consequent rejections.

A disproportionate loss of tool life either because of tool breakage or because of the necessity of grinding away too much of the tool when re-sharpening.

Higher tool and labor costs per unit.

Much waste of man hours and tool materials can be avoided by eliminating, at their sources, the causes of over-dulling, as well as of premature dulling.

Over-dulling of tools can be reduced to a minimum by periodic, close examination while in action, and by watching for the first signs of that condition. A program of re-sharpening after a given time or a given number of pieces, regardless of the apparent condition of the tools, is strongly advocated. It will prove to be economical in the long run.

Premature dulling has as its causes any one of a number of factors. These may be enumerated as follows:

Speeds too high for the hardness of the material to be cut, or, inversely, materials too hard for the speeds determined upon.

Feeds either too heavy, thereby overloading the cutting edges, or too light, causing the cutting edges to scrape rather than cut.

Work scaly on surface or having hard spots.

Poor supports for work or tool, causing springing and chatter.

Incorrect sharpening for the material to be cut.

Poor grinding finish on tool edges.

6. PROPER CARE OF DRILLS

A great deal of damage can be done to drills through careless handling. Large drills in particular, because of their weight, can be scarred and chipped quite easily. For this reason, it is recommended that they be handled with care and that procedures be set up which will minimize such dangers. The use of partitioned trays or wooden stands or blocks with holes for the drills will help to lengthen the effective life of drills.

DRILL RECONDITIONING

Of all the factors which affect over-all drill economy, proper drill reconditioning is one of the most important and one of the most underestimated. In order to lay out an effective procedure for the handling of this problem, it is necessary that there be a thorough understanding of the factors involved. There are three separate and distinct steps in the reconditioning of dull or worn drills.

1. REMOVAL OF WORN SECTION

As soon as a new drill is placed in operation, it starts to wear. This wear is of several types, each of which has its effect upon drill life. Drill wear usually starts at the corners of the drill, as shown by A in Fig. 24.

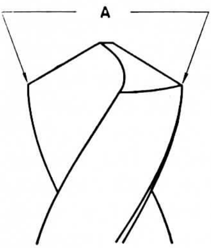


Fig. 24

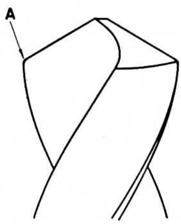


Fig. 25

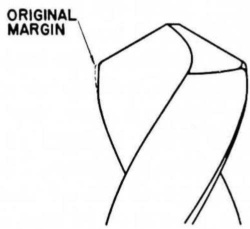


Fig. 26

Wear of Drill Points

This wear starts as a slight rounding, as shown in Fig. 25. At the same time, the cutting edges or lips, as well as the chisel-edge, start to wear away and form a truly conical surface of narrow width adjacent to these edges. This conical surface has no clearance and tends to rub in the hole rather than to cut. The power and thrust required to force this slightly-dulled edge into the work increases, resulting in greater heat-generation at the cutting edges and a faster rate of wear.

This increase in wear at the corners travels back along the margins, resulting in a loss in size and a negative back-taper in the tool, as shown in Fig. 26. This wear can easily be measured with micrometers. **The first principle of proper reconditioning is to remove all of this worn section.** Failure to do so results in a drill which will not cut properly and in a very short life on all subsequent grinds, until this worn section is removed.

The wear on a drill, like that on any other cutting tool is not proportional to the holes or pieces produced between grinds. This is a basic fact which must be considered if reasonable economy is the user's goal. Wear occurs at an accelerated rate. As explained, as the drill wears, it cuts harder, more heat is generated, and it wears faster. In other words, there is more wear on the twentieth hole than on the tenth, still more on the thirtieth, and this continues until the last hole is drilled, at which time the rate of wear is greatest.

As an example: A 1" drill under certain conditions may be able to drill 100 holes before it is completely dull and refuses to cut. At this time the margins may be worn undersize for a distance of 1 inch back from the point, and the drill must be shortened 1 inch before it can be re-used. If the drill had been removed from the machine at the end of 75 holes, this wear might have been only $\frac{1}{3}$ inch, and the drill would only need shortening by this amount. In the second case, the user would get 225 holes per inch of drill used instead of only 100 when running the drill to absolute dullness. The above example is not at all extreme and shows the advantage of a program which will promote the removal of drills from machines before the point of complete dulling.

In any event, regardless of the amount of wear on the margins near the point, the drill should be shortened to remove this excessively-worn portion.

2. WEB THINNING

Most drills today are made with webs which increase in thickness towards the shank of the drill, as shown in Figs. 27 and 28.

After shortening the drill to remove the worn portion, the drill will appear as shown in Fig. 29. The chisel edge of a drill does no actual cutting but pushes the metal out of the way. The long chisel-edge of the shortened drill in Fig. 30 will require a great deal of power and will generate much heat, resulting in short drill-life. In order to recondition the drill properly, it is necessary to reduce this web, so that the chisel edge is restored to its normal length. This operation is called web-thinning. **The second**

principle of proper reconditioning is to restore approximately the original web-thickness of all drills which have been shortened.

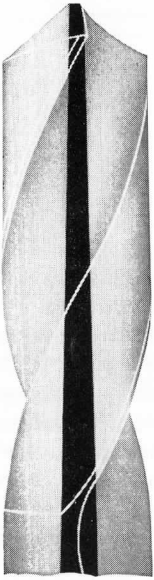


Fig. 27. The web is the metal column which separates the flutes

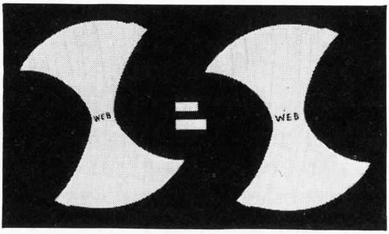
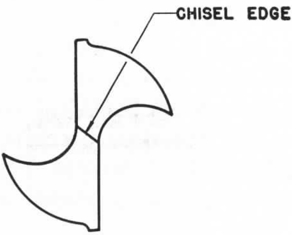
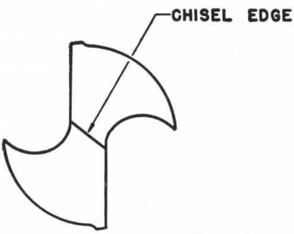


Fig. 28. The section on the left was cut from a drill near the point while the section on the right was cut near the shank. The difference in the thickness of the web at these two points is shown by the length of the white lines between the two sections in the illustration.



Point of drill when drill is new



Point of drill after drill has been cut back in use and repointed

Fig. 29

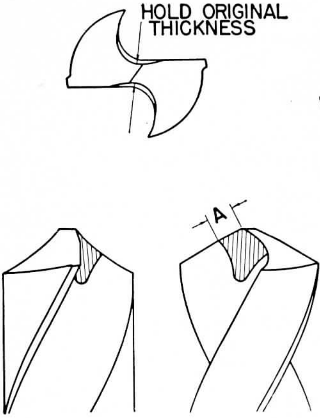
The best web-thickness for any size drill may vary slightly for different jobs, but unless unusual conditions prevail, the following approximate web-thicknesses expressed as a percentage of drill-diameter will be found to be satisfactory:

Diameter of Drill, Inches	Web-thickness Expressed as a Percentage of Drill-diameter
1/8	20
1/4	17
1/2	14
1	12
Over 1	11

The thinning of webs can be done in a special web-thinning grinder, in a conventional tool-and-cutter grinder, or it can be done as a free-hand grinding operation. There are several commercial, web-thinning grinders available today and also many tool-and-cutter grinders. Machine grinding is to be preferred to hand grinding because of its greater accuracy and reliability. However, the absence of a machine for this operation should never be considered a sufficient reason for neglecting this important step in drill-reconditioning. The operation is not at all difficult to learn and can be performed by any competent tool-grinder.

Several different types of web-thinning are in common use. In Fig. 30 is shown a type that is perhaps the most common. The length A is usually 1/2 to 3/4 the length of the cutting lip. In this type of thinning, as well as in all others, it is important that the thinning cut extend far enough up the flute so that an abrupt wedge is not formed at the extreme point. The distance up the flute to which the thinning extends will vary with the amount of thinning required, but an average of 1/4 to 1/2 of the drill diameter will usually insure that the thinning is not too abrupt. The general relationship of the drill and grinding wheel is shown in Fig. 31.

Sometimes it is advisable to extend the thinning out to the extreme edge in order to change the shape of the chip. In this type of thinning, shown in Fig. 32, a positive, effective rake is maintained the full length of the cutting-edge.



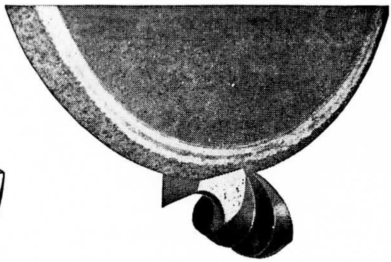


Fig. 31. Relation of drill and grinding wheel when thinning the web

Fig. 30. Usual method of thinning the point of a drill when the web has become too thick because of repeated re-pointing

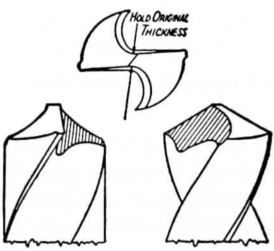


Fig. 32. Undercut Thinned Point—Another common method of web thinning. If properly done this type of thinning will produce a fine curled chip

A third type of thinning quite often used is the offset, notched, or crankshaft point, as it is variously called. This type of thinning is shown in Fig. 33.

The included angle of the point can be adjusted to the individual job, an angle of about 135° being perhaps most common. In grinding the original point-surfaces, it is usually found best to have a chisel-edge angle of 90° to 100°, as shown in Fig. 34. The two notching cuts are then made to meet at the center, forming a new chisel-edge angle of about 120°.

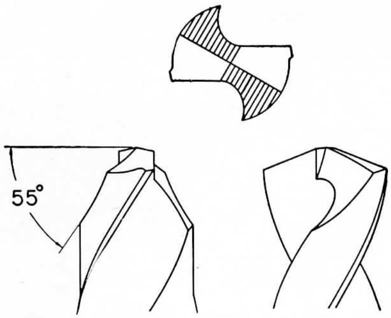


Fig. 33. Offset, Notched or Crankshaft Type of Point

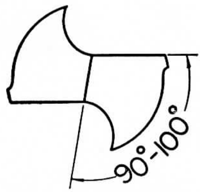


Fig. 34. Point of crankshaft drill before notching

There are several important features of this type of web thinning:

The angle of the notching cut is shown as 55° in Fig. 33. This angle is subject to some variation, but if reduced much below 55°, results in a shallow notch which is not big enough to accommodate the chips formed by the short, center-cutting edges.

It is important that the notching-cut **completely** remove all traces of the original chisel-edge. Failure to do so results in a dull chisel-edge having excessive negative rake and extremely poor cutting action.

It is also important that the two notching cuts just meet, but do not pass each other. If they pass each other, the exact center of the drill is ground away, and a fish-tail bit results with weak spots at which chipping and breakage may start.

General Web-Thinning Information

Regardless of the type of web-thinning employed—whether by machine or by hand—it is important that the web-thinning be done evenly with the same amount of stock removed from each cutting-edge.

The grinding wheel should be soft enough to remove this stock without danger of burning the cutting edges.

It should always be remembered that chips must form on the cutting edges and flow into the flutes, and the shape of the thinning should be such that it does not interfere in any way with such chip-flow.

3. REGRINDING THE SURFACE OF THE POINT

Besides removing the worn portion of the drill and thinning the web, it is necessary that the surfaces of the point be reground. These two conical surfaces intersect with the faces of the flutes to form the cutting lips. They also intersect with each other to form the chisel-edge as shown in Fig. 35.

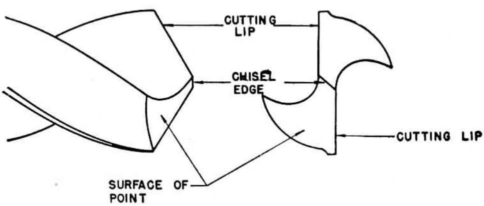


Fig. 35. Nomenclature of Drill Point

As in the case of any other cutting tool, the surface back of these cutting lips must not rub on the work, but must be relieved in order to permit the edge to penetrate. Without such relief a drill would appear like that shown in Fig. 36. This drill could not penetrate the metal, but would only rub around and around.

The relief on the cutting edges of a drill is called lip-clearance and is illustrated in Figs. 37 and 38. The lip-clearance angle, as illustrated, is measured at the circumference of the drill and may be varied to suit the job. For harder materials an angle of 6° to 9° is commonly used. For softer steels and cast irons this angle is increased to from 8° to 12°, and for very soft material may sometimes reach 15°, or even slightly higher. This lip-clearance angle is increased towards the center of the drill in order to obtain the correct chisel-edge angle shown in Fig. 39.

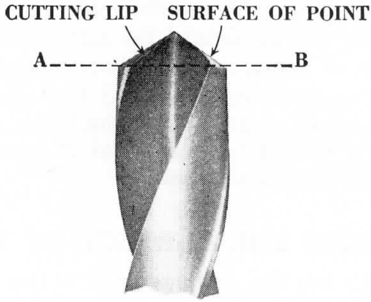


Fig. 36. A drill point without any lip clearance. Note that corners of cutting lip A and of heel B are in the same plane

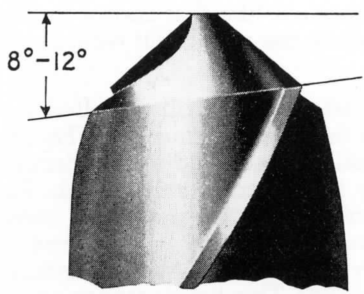


Fig. 37. Properway to grind lip-clearance. The angle indicated is the angle at the circumference of the drill

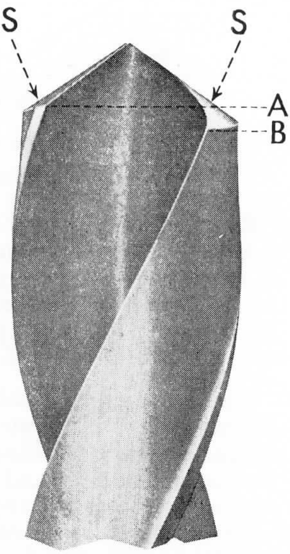


Fig. 38. Proper lip-clearance. Note how much lower the heel line B is than the cutting lip line A. This difference is the measure of the clearance

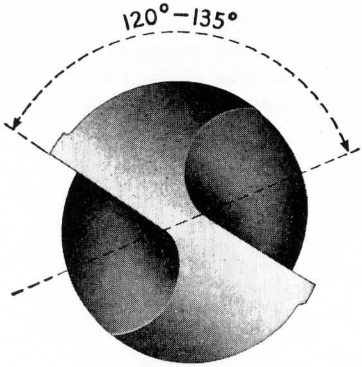


Fig. 39. Chisel-Edge Angle

This chisel-edge angle bears a relationship to lip-clearance angle and should be about 120° for lip-clearances of from 6° to 9°, and about 135° for lip-clearances of from 12° to 15°.

Accuracy of Cutting Lips

The two cutting lips of a drill should be accurately ground. Regardless of the point angle, the angles of the two cutting lips A1 and A2, Fig. 40, must be equal. Similarly, the length of the two lips L1 and L2 should be equal. There are many drill-point gages available which can be used to measure these angles and lengths.

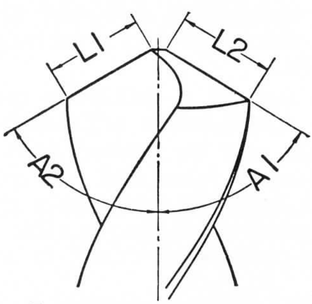


Fig. 40. Angles and lengths of cutting lips must be equal

In Fig. 41 is shown a drill with a point having lips of equal length, but which make unequal angles with the axis of the drill. One cutting edge does most of the cutting. This type of point will cause an oversize hole. In addition, unnecessarily severe strains are thrown on the drill and spindle. Excessive wear and short tool-life will usually be another result.

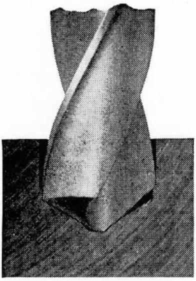


Fig. 41. Incorrect Point. Lips of equal length but at unequal angles

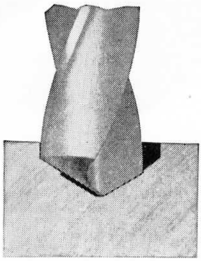


Fig. 42. Incorrect Point. Lips of unequal length but at equal angles

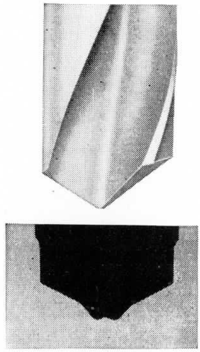


Fig. 43. Incorrect Point. Lips of unequal length and at unequal angles

Much the same result is obtained if the angles are equal, but the lips are of unequal length, as shown in Fig. 42. A combination of both unequal lengths and angles is shown in Fig. 43.

In order to maintain the necessary accuracy of point-angles, lip-lengths, lip-clearance angle, and chisel-edge angle, the use of machine point-grinding is recommended. There are many commercial drill-point grinders available today, the use of which will be of great help in the accurate repointing of drills.

However, the lack of a drill-point grinding machine should never be considered a sufficient reason to excuse poor points. Drills can be pointed accurately by hand, although the operation takes skill and practice. On very small drills, the use of a binocular microscope focused on the wheel at the point of grinding is quite helpful. Magnifications ordinarily used range from 7 to 10.

Different Types of Drill Points

It has been generally noted by those engaged in production-drilling of steels, especially of deep holes in the harder steels, that better results can be obtained by in-

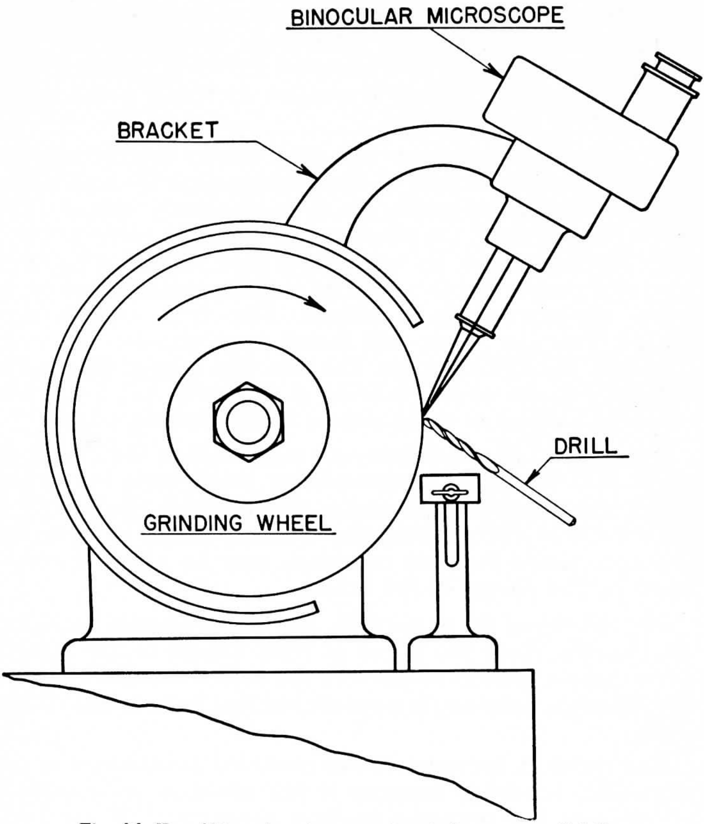


Fig. 44. Use of binocular microscope in grinding very small drills

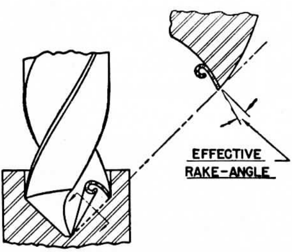


Fig. 45

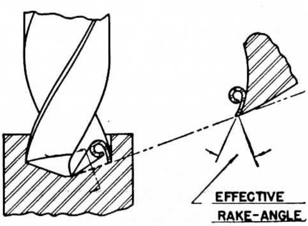


Fig. 46

Effective Rake Angles

creasing the included point-angle from the average 118° to 135° or 140°. Similarly, it has been found that cast iron and related materials can often be drilled to better advantage with a drill having a smaller included point-angle, such as 90° to 100°.

The explanation of the first case lies in the increase of the effective rake-angle at the cutting edge by increasing the included point-angle; for it is generally known that, when cutting steel, the efficiency of a cutting tool increases with the rake-angle. By comparing Figs. 45 and 46, it can be readily seen that this change in rake angles takes place when the point-angle is altered. The chip, as it is separated from the work, will travel approximately at right angles to the cutting edge. The effective rake-angle, therefore, is not the angle of helix of the drill, but the angle taken in a plane at right angles to the cutting edge.

Fig. 45 and its accompanying cross-section indicate that, for a drill having a comparatively small included point-angle the effective rake angle also is small. Reference is here made to drills having the ordinary shape of flutes, as it is recognized that this condition may be affected somewhat by the shape of the flutes.

Fig. 46 shows the same drill, but with a greater included point-angle. Here the plane at right angles to the cutting edge make a smaller angle with the axis of the drill. Hence, the effective rake-angle approaches the helix-angle of the drill.

Too great an increase in the included point-angle is not advisable, however, because it will result in an abnormal increase in the required feeding pressure; 135° or 140° point-angles have been found practical for most steels.

It is obvious that it is possible by this same expedient of changing the point-angle, to change the nature of the chips as they are separated from the work. The amount and nature of this change will vary with the hardness and texture of the material being cut.

The explanation of the second case, namely, that of successful drilling of cast irons with drills having small included point-angles, is one of an entirely different nature. It has been learned that the efficiency of a cutting tool working in cast iron is affected but very slightly by altering the rake-angle within the ordinary range. It is also known that dulling of drills in cast iron is nearly always evidenced by abrasion at the corners of the cutting edges.

If a drill is sharpened with a large included point-angle, as in Fig. 47, the zone of abrasion is comparatively small, as at M. Dulling will, therefore, be proportionately hastened. By sharpening with a small included angle, as in Fig. 48, the area of the zone of abrasion N is greatly increased and with it the life per grind. To further increase this abrasive area, and with it the grinding life, it is sometimes advisable to grind a secondary angle at the corners of the drill, as shown in Fig. 49.

Cast Iron usually breaks up into small chips regardless of the amount of rake on the cutting edges of the drill, so that the effect of changes in the point-angle on the nature of chips produced is practically negligible.

DRILL POINTS RECOMMENDED FOR VARIOUS MATERIALS

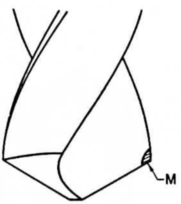


Fig. 47

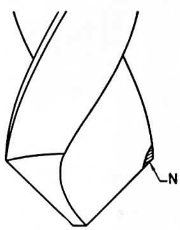


Fig. 48
Drill Points for Cast Iron

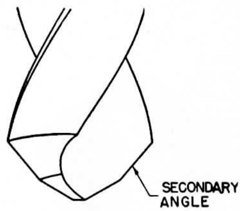


Fig. 49

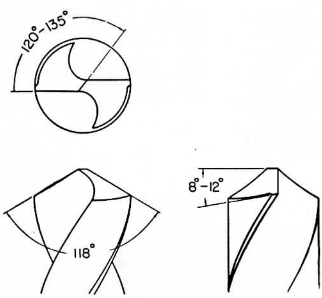


Fig. 50. Regular Point
General Purpose

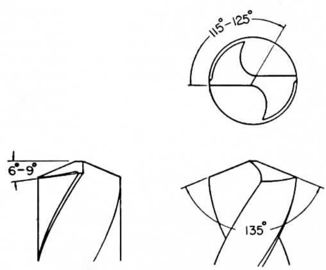


Fig. 51 Flat Point
Hard and Tough Materials
(Manganese Steel Rail, etc.)

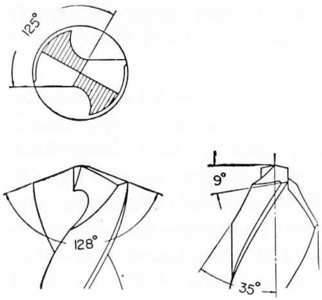


Fig. 52. Crankshaft-Point Deep-hole Drilling (Note Heavy Web)

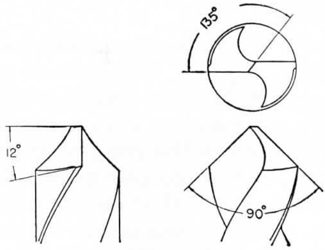


Fig. 53. Long-Point Wood, Bakelite, Hard Rubber and Fibers

Common Drill Troubles and Causes

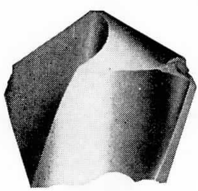


Fig. 54
The Indication of Too Great Speed. The outer corners of the drill have worn away rapidly because excessive speed has drawn the temper

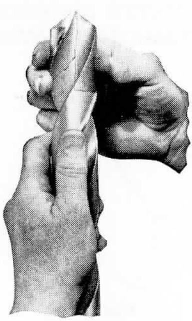


Fig. 55
Here the drill was given insufficient lip-clearance. As a result there ceased to be any cutting edges whatsoever, and, as the feed-pressure was applied, the drill could not enter the work—as a result it "splits up the center"

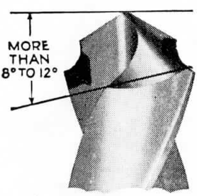


Fig. 56
Showing results of giving a drill too great lip-clearance—the edges of the cutting lips have broken down because of insufficient support

Indications	Causes
Outer Corners, Break Down	Cutting speed too high. Hard spots in material. No cutting compound at drill point. Flutes clogged with chips.

Indications	Causes
Cutting Edges Chip	Too much feed. Lip clearance too great.
Checks or Cracks in Cutting Edges	Overheated or too quickly cooled while sharpening or drilling.
Margin Chips	Oversize Jig Bushing.
Drill Breaks	Point improperly ground. Feed too heavy. Spring or back lash in drill press, fixture or work. Drill is dull. Flutes clogged with chips.
Tang Breaks	Imperfect fit between taper shank and socket caused by dirt or chips, or burred or badly worn sockets.
Drill Breaks when Drilling Brass or Wood	Flutes clogged with chips. Improper type drill.
Drill Splits up Center	Lip clearance too small. Too much feed.
Drill Will Not Enter Work	Drill is dull. Lip clearance too small. Too heavy a web.
Hole Rough	Point improperly ground or dull. No cutting compound at drill point. Improper cutting compound. Feed too great. Fixture not rigid.
Hole Oversize	Unequal angle or length of the cutting edges — or both. Loose spindle.
Chip Shape changes while Drilling	Drill becomes dull or cutting edges chipped.
Large Chip coming out of one Flute, Small Chip out of other flute	Point improperly ground, one lip doing all the cutting.

Table of Cutting Speeds

(Fraction Size Drills)

Feet per Min.	30′	40′	50′	60′	70′	80′	90′	100′	110′	120′	130′	140′	150′
Diameter Inches	Revolutions per Minute												
1/16	1833	2445	3056	3667	4278	4889	5500	6111	6722	7334	7945	8556	9167
1/8	917	1222	1528	1833	2139	2445	2750	3056	3361	3667	3973	4278	4584
3/16	611	815	1019	1222	1426	1630	1833	2037	2241	2445	2648	2852	3056
1/4	458	611	764	917	1070	1222	1375	1528	1681	1833	1986	2139	2292
5/16	367	489	611	733	856	978	1100	1222	1345	1467	1589	1711	1833
3/8	306	407	509	611	713	815	917	1019	1120	1222	1324	1426	1528
7/16	262	349	437	524	611	698	786	873	960	1048	1135	1222	1310
1/2	229	306	382	458	535	611	688	764	840	917	993	1070	1146
5/8	183	244	306	367	428	489	550	611	672	733	794	856	917
3/4	153	203	255	306	357	407	458	509	560	611	662	713	764
7/8	131	175	218	262	306	349	393	436	480	524	568	611	655
1	115	153	191	229	267	306	344	382	420	458	497	535	573
1 1/8	102	136	170	204	238	272	306	340	373	407	441	475	509
1 1/4	92	122	153	183	214	244	275	306	336	367	397	428	458
1 3/8	83	111	139	167	194	222	250	278	306	333	361	389	417
1 1/2	76	102	127	153	178	204	229	255	280	306	331	357	382
1 5/8	70	94	117	141	165	188	212	235	259	282	306	329	353
1 3/4	65	87	109	131	153	175	196	218	240	262	284	306	327
1 7/8	61	81	102	122	143	163	183	204	224	244	265	285	306
2	57	76	95	115	134	153	172	191	210	229	248	267	287
2 1/4	51	68	85	102	119	136	153	170	187	204	221	238	255
2 1/2	46	61	76	92	107	122	137	153	168	183	199	214	229
2 3/4	42	56	69	83	97	111	125	139	153	167	181	194	208
3	38	51	64	76	89	102	115	127	140	153	166	178	191

(Continued on following page)

Table of Cutting Speeds

(Number Size Drills)

Feet Per Min.	30′	40′	50′	60′	70′	80′	90′	100′	110′	120′	130′	140′	150′
No. Size	Revolutions per Minute												
1	503	670	838	1005	1173	1340	1508	1675	1843	2010	2179	2346	2513
2	518	691	864	1037	1210	1382	1555	1728	1901	2074	2247	2420	2593
3	538	717	897	1076	1255	1434	1614	1793	1974	2152	2331	2511	2690
4	548	731	914	1097	1280	1462	1645	1828	2010	2193	2376	2560	2741
5	558	744	930	1115	1301	1487	1673	1859	2045	2230	2416	2602	2788
6	562	749	936	1123	1310	1498	1685	1872	2060	2247	2434	2621	2809
7	570	760	950	1140	1330	1520	1710	1900	2090	2281	2470	2660	2850
8	576	768	960	1151	1343	1535	1727	1919	2111	2303	2495	2687	2879
9	585	780	975	1169	1364	1559	1754	1949	2144	2339	2534	2728	2923
10	592	790	987	1184	1382	1579	1777	1974	2171	2369	2566	2764	2961
11	600	800	1000	1200	1400	1600	1800	2000	2200	2400	2600	2800	3001
12	606	808	1010	1213	1415	1617	1819	2021	2223	2425	2627	2829	3032
13	620	826	1032	1239	1450	1652	1859	2065	2271	2479	2684	2891	3097
14	630	840	1050	1259	1469	1679	1889	2099	2309	2518	2728	2938	3148
15	638	851	1064	1276	1489	1702	1914	2127	2334	2546	2759	2971	3183
16	647	863	1079	1295	1511	1726	1942	2158	2374	2590	2806	3021	3237
17	662	883	1104	1325	1546	1766	1987	2208	2429	2650	2870	3091	3313
18	678	904	1130	1356	1582	1808	2034	2260	2479	2704	2930	3155	3380
19	690	920	1151	1381	1611	1841	2071	2301	2531	2761	2991	3222	3453
20	712	949	1186	1423	1660	1898	2135	2372	2610	2847	3084	3322	3559
21	721	961	1201	1441	1681	1922	2162	2402	2644	2883	3123	3363	3604
22	730	973	1217	1460	1703	1946	2190	2433	2676	2920	3164	3406	3649
23	744	992	1240	1488	1736	1984	2232	2480	2728	2976	3224	3472	3720
24	754	1005	1257	1508	1759	2010	2262	2513	2764	3016	3267	3518	3769
25	767	1022	1276	1533	1789	2044	2300	2555	2810	3066	3322	3577	3832
26	779	1039	1299	1559	1819	2078	2338	2598	2858	3118	3378	3638	3898
27	796	1061	1327	1592	1857	2122	2388	2653	2919	3183	3448	3714	3979
28	816	1088	1360	1631	1903	2175	2447	2719	2990	3262	3534	3806	4078
29	843	1124	1405	1685	1966	2247	2528	2809	3090	3370	3651	3932	4213
30	892	1189	1487	1784	2081	2378	2676	2973	3270	3567	3864	4162	4459
31	955	1273	1592	1910	2228	2546	2865	3183	3501	3821	4138	4456	4775
32	988	1317	1647	1976	2305	2634	2964	3293	3622	3951	4281	4610	4939
33	1014	1352	1690	2028	2366	2704	3042	3380	3718	4056	4394	4732	5070
34	1032	1376	1721	2065	2409	2753	3097	3442	3785	4129	4474	4818	5162
35	1042	1389	1736	2083	2430	2778	3125	3472	3821	4167	4514	4861	5209
36	1076	1435	1794	2152	2511	2870	3228	3587	3945	4304	4663	5021	5380
37	1102	1469	1837	2204	2571	2938	3306	3673	4040	4407	4775	5142	5509
38	1129	1505	1882	2258	2634	3010	3387	3763	4140	4516	4892	5269	5645
39	1152	1536	1920	2303	2687	3071	3455	3839	4222	4607	4991	5374	5758
40	1169	1559	1949	2339	2729	3118	3508	3898	4287	4677	5067	5457	5846
41	1194	1592	1990	2387	2785	3183	3581	3979	4377	4775	5172	5570	5968
42	1226	1634	2043	2451	2860	3268	3677	4085	4494	4902	5311	5719	6128
43	1288	1717	2146	2575	3004	3434	3863	4292	4721	5150	5579	6008	6438
44	1333	1777	2221	2665	3109	3554	3999	4442	4886	5330	5774	6218	6662
45	1397	1863	2329	2795	3261	3726	4192	4658	5124	5590	6056	6522	6987
46	1415	1886	2358	2830	3301	3773	4244	4716	5187	5659	6130	6602	7074
47	1460	1946	2433	2920	3406	3893	4379	4866	5352	5839	6326	6812	7299
48	1508	2010	2513	3016	3518	4021	4523	5026	5528	6031	6534	7036	7539
49	1570	2093	2617	3140	3663	4186	4710	5233	5756	6279	6808	7326	7849
50	1637	2183	2729	3274	3820	4366	4911	5457	6002	6548	7094	7640	8185
51	1710	2280	2851	3421	3991	4561	5131	5701	6271	6841	7413	7982	8552
52	1805	2406	3008	3609	4211	4812	5414	6015	6619	7218	7820	8421	9023
53	1924	2566	3207	3848	4490	5131	5773	6414	7062	7704	8346	8988	9630
54	2084	2778	3473	4167	4862	5556	6251	6945	7639	8334	9028	9723	10417
55	2204	2938	3673	4408	5142	5877	6611	7346	8080	8815	9549	10284	11028

(Concluded on following page)

Table of Cutting Speeds

(Number Size Drills—(Continued)

Feet Per Min.	30′	40′	50′	60′	70′	80′	90′	100′	110′	120′	130′	140′	150′
No. Size	Revolutions per Minute												
56	2465	3286	4108	4929	5751	6572	7394	8215	9036	9857	10678	11500	12322
57	2671	3561	4452	5342	6232	7122	8013	8903	9771	10660	11548	12436	13325
58	2729	3637	4547	5456	6367	7275	8186	9095	10004	10913	11823	12732	13642
59	2795	3726	4658	5590	6521	7453	8388	9316	10248	11180	12111	13043	13975
60	2865	3820	4775	5729	6684	7639	8594	9549	10504	11459	12414	13369	14324
61	2938	3918	4897	5876	6856	7835	8815	9794	10774	11753	12732	13712	14691
62	3015	4020	5025	6030	7035	8040	9045	10050	11057	12060	13068	14073	15078
63	3096	4128	5160	6192	7224	8256	9288	10320	11366	12398	13421	14453	15485
64	3183	4244	5305	6366	7427	8488	9549	10610	11671	12732	13793	14854	15915
65	3273	4364	5455	6546	7637	8728	9819	10910	12005	13096	14187	15279	16370
66	3474	4632	5790	6948	8106	9264	10422	11580	12732	13890	15047	16205	17362
67	3582	4776	5970	7164	8358	9552	10746	11940	13130	14324	15517	16712	17905
68	3696	4928	6160	7392	8624	9856	11088	12320	13554	14786	16018	17250	18482
69	3918	5224	6530	7836	9142	10488	11754	13060	14389	15697	17006	18314	19622
70	4091	5456	6820	8184	9548	10912	12276	13640	15006	16370	17734	19099	20463
71	4419	5892	7365	8838	10311	11784	13257	14730	16160	17629	19099	20568	22037
72	4584	6112	7640	9168	10696	12224	13752	15280	16807	18335	19863	21390	22918
73	4776	6368	7960	9552	11144	12736	14328	15920	17507	19099	20690	22282	23873
74	5106	6808	8510	10212	11914	13616	15318	17020	18674	20372	22069	23767	25465
75	5457	7276	9095	10914	12733	14552	16371	18190	20008	21827	23646	25465	27284
76	5730	7640	9550	11460	13370	15280	17190	19100	21008	22918	24828	26738	28648
77	6366	8488	10610	12732	14854	16976	19098	21220	23343	25465	27587	29709	31831
78	7161	9548	11935	14322	16709	19096	21483	23870	26260	28648	31035	33422	35810
79	7902	10536	13170	15804	18438	21072	23706	26340	28988	31611	34246	36880	39514
80	8490	11320	14150	16980	19810	22640	25470	28300	31123	33953	36782	39612	42441

Table of Cutting Speeds

Letter Size Drills)

Feet Per Min.	30′	40′	50′	60′	70′	80′	90′	100′	110′	120′	130′	140′	150′
Size Letter	Revolutions per Minute												
A	491	654	818	982	1145	1309	1472	1636	1796	1959	2122	2285	2448
B	482	642	803	963	1124	1284	1445	1605	1765	1926	2086	2247	2407
C	473	631	789	947	1105	1262	1420	1578	1736	1894	2052	2210	2368
D	467	622	778	934	1089	1245	1400	1556	1708	1863	2018	2174	2329
E	458	611	764	917	1070	1222	1375	1528	1681	1834	1968	2139	2292
F	446	594	743	892	1040	1189	1337	1486	1635	1784	1932	2081	2229
G	440	585	732	878	1024	1170	1317	1463	1610	1756	1903	2049	2195
H	430	574	718	862	1005	1149	1292	1436	1580	1723	1867	2010	2154
I	421	562	702	842	983	1123	1264	1404	1545	1685	1826	1966	2106
J	414	552	690	827	965	1103	1241	1379	1517	1655	1793	1930	2068
K	408	544	680	815	951	1087	1223	1359	1495	1631	1767	1903	2039
L	395	527	659	790	922	1054	1185	1317	1449	1581	1712	1844	1976
M	389	518	648	777	907	1036	1166	1295	1424	1554	1683	1813	1942
N	380	506	633	759	886	1012	1139	1265	1391	1518	1644	1771	1897
O	363	484	605	725	846	967	1088	1209	1330	1450	1571	1692	1813
P	355	473	592	710	828	946	1065	1183	1301	1419	1537	1657	1774
Q	345	460	575	690	805	920	1035	1150	1266	1384	1496	1611	1726
R	338	451	564	676	789	902	1014	1127	1239	1355	1465	1577	1690
S	329	439	549	659	769	878	988	1098	1207	1317	1427	1537	1646
T	320	426	533	640	746	853	959	1066	1173	1280	1387	1494	1600
U	311	415	519	623	727	830	934	1038	1142	1246	1349	1453	1557
V	304	405	507	608	709	810	912	1013	1114	1219	1317	1418	1520
W	297	396	495	594	693	792	891	989	1088	1188	1286	1385	1484
X	289	385	481	576	672	769	865	962	1058	1155	1251	1347	1443
Y	284	378	473	567	662	756	851	945	1040	1135	1229	1324	1418
Z	277	370	462	555	647	740	832	925	1017	1110	1202	1295	1387

10-501

Printed in U.S.A.