STAT

Page Denied

[redacted]

Chapter VII

DRILLS, COUNTERSINKS, AND REAMERS

DRILLS

Designation and Types.

Drills are designed for: (a) drilling holes in solid material; (b) oversize drilling of already existing holes (as, for instance, holes cast, forged, or drop-forged integrally); (c) the drilling of tapered center holes.

By design and purpose, drills are divided as follows: (1) spiral or twist drills; (2) center drills; (3) pointed drills; (4) straight-flute drills; (5) drills with hard-alloy-tipped blades; (6) deep-hole drills.

Twist drills are basically representative of this group of tools, and they have found the widest application. They are used for the drilling of: (a) holes that require no subsequent machining; (b) holes for subsequent countersinking; (c) holes for subsequent reaming; (d) holes for subsequent threading with taps. GOST 885 - 41 cites the data recommended for use in the selection of drill diameters in relation to the purpose of drilling.

Twist Drills

Definition and types. A twist drill (helical drill is more correct) is a rod having two helical lips relieved at the point in order to form the cutting edges.

Twist drills are divided as follows: (1) long straight-shank (GOST 886 - 41); short straight-shank (GOST 887 - 41); (3) left hand straight-shank for automatic machines (GOST 2090-43); (4) taper-shank (GOST 888 - 41); (6) shortened with accelerated taper-shank (OST 20182 - 40); (7) lengthened with taper-shank (GOST 2092 - 43); (8) with tetrahedral tapering shank for ratchet drills (OST 20231 - 41).

All the above enumerated types of twist drills are differentiated by their shanks, the bits being basically the same in design conception.

Heavy duty twist drills, such as deep-hole drills, are equipped with special grooves to allow the flow of coolant to the cutting edges.

For the machining of holes with two or three drill diameters, two- or three-step drills are produced.

The basic terminology, designations, and definitions of drills are summarized in GOST 2894 - 45, and the geometric parameters of the drill bits, in GOST 2322 - 43.

Parts and structural components. A drill consists of the following parts and structural components (Figure 1):

[Drawing]

Figure 1. Parts and structural components of a twist drill.

l -- working part; l_1 -- cutting part [the chisel edge]; l_2 -- calibrating part; l_3 -- neck; l_4 -- shank length; e - tang; e_1 -- holding part; k -- helical flute (right-hand or left-hand); p -- lip; f -- margin; d_0 -- web; b -- cutting edge (two on taper-shank drills); c -- calibrating edge (two on straight-shank drills); a -- transverse edge (one at drill point); s -- front surface; t -- back (relieved) surface.

The basic structural elements of a drill are: (a) the cutting part [chisel edge]; (b) the direction of the helical flute; (c) the shape of the flute; (d) the cutting edge angles; (e) the form of back (relieved) surface; (f) the back taper; (g) the holding part.

The cutting part is basic in the process of cutting, since it contains all the cutting components of the drill.

The calibrating part acts as a guide in the process of cutting and is also the surplus allowance for the resharpening of the drill. At its lip margins are the calibrating edges, which determine the final formation of the drill hole.

Angle 2φ (see Figure 2), the double angle in plan [the point angle measured in plan] has a great effect upon the work of the drill. It is selected in relation to the metal machined, as indicated in Table 1.

Section ab

[Drawing]

Figure 2. Cutting edge angle and helical flute rake angle.

TABLE 1

Values of angle 2 φ

Material to be machined	2 φ in degrees
Steel, cast iron, hard bronze....	116 - 118
Brass, soft bronze...............	130
Aluminum, duralumin, silumin, electron, babbit..................	140
Red copper.....................	125
Ebonite, celluloid.............	85 - 90
Marble and other brittle materials	80

Direction of helical flute. The helical flute rake angle ω (see Figure 2) is in close relation to the front angle. With ω increased, the front angle becomes increased, the cutting is facilitated, with a decrease in torque and axial pressure, and an improved outflow of chip from the flutes. However, with the increase in angle ω, the cutting edge of the drill is weakened. This weakening, in the presence of the same value for angle ω, is relatively greater in the case of small drills than in the case of large ones. Therefore, small drills for universal use are designed with a smaller ω angle.

Drills fabricated from high-speed steel work at an accelerated cutting cycle as compared with drills made from carbon steel or alloy steel; therefore, angle ω is to be of greater value for the former than for the latter. The values of angle ω for drills in universal use are furnished in Table 2.

TABLE 2

Values of angle ω for drills in universal use

Drills from high-speed steel		Drills from carbon steel	
Drill diameters in millimeters	ω in degrees	Drill diameters in millimeters	ω in degrees
1.0 - 1.4	22	0.25 - 1.0	19
1.45 - 1.5	23	1.05 - 1.5	20
1.55 - 3.0	23 - 24	1.55 - 3.0	20 - 22
3.1 - 4.0	25	3.1 - 6.7	22 - 23
4.2 - 6.0	26	6.8 - 10.0	24
6.2 - 8.2	27	10.1 - 17.0	25
8.3 - 11.5	28	17.5 - 28.0	26
11.6 - 16.0	29	28.5 - 39.0	27
16.5 - 22.0	30	39.5 - 80	28
22.5 - 33.0	31		
33.5 - 35.0	31 - 32		
35.5 - 44.0	32		
44.5 - 80	33		

The selection of angle ω is made in relation to the type of material being machined. This is of particular importance in the case of special drills. Angle ω (in degrees) in special drills for the machining of brass, soft bronze, ebonite, bakelite, and celluloid is 8 - 12 degrees, for the machining of marble and other brittle materials -- 10 - 15 degrees, for red copper and aluminum -- 35 - 45 degrees, with lower values for small drills and higher values for large drills.

Twist drills are usually made for right-hand cutting with a right-hand direction of the flutes. Drills with a left-hand direction of the flutes are used rarely, mainly for work on automatic lathes.

Shape of the flute. The profile of the drill flute must conform with rigid specifications. It must provide for: (a) strength of the drill; (b) rational distribution of metal throughout the section to prevent cracking in heat-treatment; (c) ample space for chip disposition; (d) the correct formation of chip upon the cutting edge and its easy outflow from the flute.

The basic components of the flute profile are the thickness of the drill web, the width of the flute, the form of the cutting edge and the blending curves.

The diameter of the web d_o (see Figure 2) is selected in relation to the size of the drill. In order to attain a higher degree of strength, the web diameter for small drills is made relatively greater than for large drills. For drills with a diameter of 0.25 - 1.25 millimeters, the web diameter equals $(0.28 \div 0.20)D$; for drills with a diameter of 1.5 - 12 millimeters, the web diameter is $(0.19 \div 0.15)D$; for drills with a diameter of 13 - 80 millimeters, the web diameter is $(0.145 \div 0.125)D$, where D is the drill diameter. Drills made from high-speed steel, on account of a high degree of decarbonization in the heat treatment, must have their flutes ground. By the virtue of this, the web diameter for high-speed drills, of a diameter D = 0.25 to 18 millimeters, before grinding, will be 0.03 - 0.20 millimeter greater than the web diameter of carbon steel drills. For drill sizes in excess of 18 millimeters no allowance for grinding is necessary.

To strengthen the drill, the web diameter is increased in the direction of the shank. In the case of carbon steel drills, the thickening of the web equals 1.5 millimeters, and for high-speed drills, 1.75 millimeters per each 100 millimeters of length.

The width of the flute is usually equal to the width of the lip. For high-speed steel drills, it should be somewhat wider than the width of the lip (let us say, by 1/128 of the outside diameter of the drill).

The cutting edge of the drill may be rectilinear, convex or concave. As yet, no preference has been established. At present, all our domestic plants and first-rate firms abroad fabricate drills with rectilinear cutting edges.

To avoid cracking in heat treatment and to facilitate the chip outflow, the flute profile is to have well rounded curves (see page 327 for the description of the profile-milling cutters, with the aid of which this is accomplished).

Drill angles. The cutting edge angles may be considered as the grinding angles of the drill and also in the process of cutting.

Front rake angle γ is the angle between the plane tangent to the front surface at the contemplated point of the cutting edge, and a plane normal, at the same point, to the surface of rotation of the cutting edge about the drill axis.

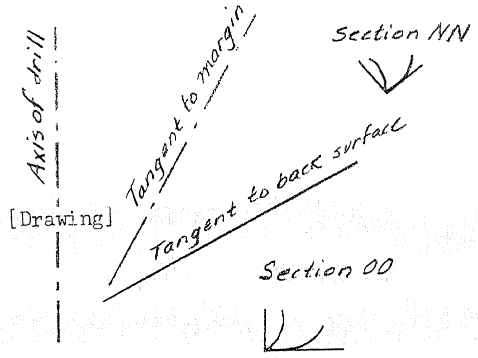

Figure 3. Drill cutting edge angles.

[Drawing]

Figure 4. Front rake angle at any point of cutting edge.

To determine the front rake angle, it is necessary to visualize the main intersecting plane, in which it is subject to change. Plane NN (see Figure 3), normal to cutting edge (γ N), is accepted as such a plane.

The front rake angle γ_X at any point X of the cutting edge (see Figure 4), analyzed in the plane NN, is determined by formula

$$\tan \gamma_x = \frac{C \cdot \tan \omega \cos \mu}{\sin \varphi - C \tan \omega \cos \varphi \sin \mu},$$

where $C = \frac{r_x}{R}$; r_x is the distance of the contemplated point X from the drill axis; R is the outside diameter of a drill; μ is the angle formed by radius r_x with the axis of symmetry of the transverse section of the drill.

The thickness of the connector characterized by the angle μ exerts an insignificant effect upon the angle γ_x. Disregarding it, we may use the following approximated formula:

$$\tan \gamma_x = C \frac{\tan \omega}{\sin \varphi}.$$

It follows from the formula that the front rake angle, which depends on the ratio between $C = \frac{r_x}{R}$ and angles ω and φ , is not uniform along the entire length of the cutting edge even for the same diameter of drill, sharpened to a definite angle φ.

The maximum (positive) value of angle γ_x is at a point on the periphery of the drill, and the minimum (negative) value is in the transverse cutting edge section ab (see Figure 2). The sharply varying front rake angle is a considerable organic defect in the design of the twist drill. It causes the non-uniform and rapid wear of the cutting edge. At the periphery of the drill, where the greatest cutting speed takes place, the maximum amount of heat is liberated, and, due to the small angle of taper, it cannot be eliminated rapidly, causing the maximum wear to develop where the transition from taper to cylinder occurs.

The back relief angle α is the angle between a plane tangent to the back surface at the contemplated point of the cutting edge and a plane which is tangent at the same point to a surface generated by the rotation of the cutting edge about the drill axis.

According to the prevailing standard definitions, the back angle, similarly to the front angle, is measured in the plane NN (see Figure 3) normal to the cutting edge. The main plane of intersection for the back angle is plane OO, directed along the drill axis and tangent to the cylindrical surface which is generated by the contemplated point in the rotation of the cutting edge about the drill axis.

The ratio between the values of angles α_N and α_o, in the planes NN and OO, for a point at the drill periphery, may be expressed by the approximated formula (disregarding the thickness of the web):

$$\tan \alpha_N = \tan \alpha_0 \sin \varphi .$$

In order to attain a more or less uniform angle of taper throughout the cutting edge, and also to provide for the adequate value of the back angle in the process of cutting, it becomes necessary for the back relief angle to be variable also. At the periphery, its value is accepted as 8 - 14 degrees, at the web -- as 20 - 25 degrees, depending on the drill diameter. Small-size drills have greater values of the back relief angle at the periphery than larger-size drills.

Cutting edge angles in the process of cutting. Two motions take place in drilling: rotary (the cutting speed) and forward (the feed). As a result of these motions, each point of the cutting edge is shifted along a helical line at a pitch equal to the value of the feed per one revolution. The helical surface generated in the process of cutting by the cutting edge is the cutting surface, and the plane tangent to it is the cutting plane. Figure 5 depicts the cross section of the drill by a plane normal to the cutting edge and the evolution of the helical line -- the trajectory of point A per one revolution of the drill.

[Drawing]

Figure 5. Cutting edge angles of drill in the process of cutting.

$$\tan \alpha_N = \tan \alpha_0 \sin \varphi .$$

In order to attain a more or less uniform angle of taper throughout the cutting edge, and also to provide for the adequate value of the back angle in the process of cutting, it becomes necessary for the back relief angle to be variable also. At the periphery, its value is accepted as 8 - 14 degrees, at the web -- as 20 - 25 degrees, depending on the drill diameter. Small-size drills have greater values of the back relief angle at the periphery than larger-size drills.

Cutting edge angles in the process of cutting. Two motions take place in drilling: rotary (the cutting speed) and forward (the feed). As a result of these motions, each point of the cutting edge is shifted along a helical line at a pitch equal to the value of the feed per one revolution. The helical surface generated in the process of cutting by the cutting edge is the cutting surface, and the plane tangent to it is the cutting plane. Figure 5 depicts the cross section of the drill by a plane normal to the cutting edge and the evolution of the helical line -- the trajectory of point A per one revolution of the drill.

[Drawing]

Figure 5. Cutting edge angles of drill in the process of cutting.

AP is the evolved length of the circumference πD; PW is the value of feed per one revolution; AW is the cutting trajectory (the evolved helical line generated by point A); AN is the normal to the cutting trajectory; α, γ are the back relief and front rake angles, respectively, of the drill bit; α_r, γ_r are the respective cutting angles; Θ is the angle of inclination of the cutting trajectory (the angle between the actual trajectory of cutting and the conditional trajectory, which is the circumference described by the rotation of the drill without the feeding motion).

The front cutting angle γ_r is the angle between a plane tangent to the front surface at the contemplated point of the cutting edge, and a plane normal to the cutting edge at the same point.

The back cutting angle α_r is the angle between a plane tangent to the back surface at the contemplated point of the cutting edge, and the cutting plane at the same point.

Both these angles are to be measured in the main plane of intersection.

The ratio between α_r, α and γ_r, γ is expressed as follows:

$$\gamma_r = \gamma + \Theta;$$

$$\alpha_r = \alpha - \Theta.$$

Angle Θ is determined by formula

$$\tan \Theta = \frac{S}{\pi D},$$

In the presence of uniform feed and reduced value of D, that is, with the points of the cutting edge moving away from the drill periphery toward the drill web, angle Θ is increased. This is one of the reasons for the selection of a variable back relief angle α.

Back surface form. To provide for a variable back relief angle, twist drills undergo special sharpening. The index for correct drill sharpening is the strict adherence to the following requisite values: (1) of the drill bit angle φ; (2) of the transverse cutting edge angle ϑ; (3) of the back relief angle α for points at the drill periphery and at the drill web.

In addition, the cutting edges, longitudinally, are to be of the same length and have equal angles φ, and the drill axis is to pass through the center of the transverse cutting edge [the chisel edge]. The non-observance of the symmetry of the cutting edges will result in their non-uniform (one-sided) loading and excessive free play of the drill, which in turn will result in a drill hole of excessive diameter.

The transverse cutting edge [chisel edge] angle ϑ (see Figure 2) in properly sharpened drills is equal to 47 - 50 degrees for up to 12-millimeter drills and 52 - 55 degrees for drills of 12-millimeter diameter and above.

The sharpening of drills may be done along the helical or conical surface. The most widespread drill sharpening machines operate on the second principle. Both these types of drill sharpening are presented diagrammatically in Figure 6, which merits special attention, since it represents the drill sharpening done on machines of domestic makes as well as machines imported from abroad.

[Drawing]

Figure 6. Drill sharpening diagrams.

The back relief angles are to be determined in relation not only to the drill diameter, but also to the type of material machined. This is related to a definite set-up in the drill-sharpening machine. To set up the drill sharpening machine as per Figure 6, (a) it is necessary to know the double angle 2ρ for the conical surface, distance a from the cone apex to the drill axis, and (b), the value of the displacement of the cone axis from the drill axis. These values are in relation to the drill bit angle 2φ, the chisel edge angle ϑ, and the back relief angle α. The correlationships between them are expressed by the following formulas:

$$\tan \alpha_x = \sin \mu_x (K_2 + \tan \delta) -$$

$$- \frac{K_2^2 (a + r_x + \cos \mu_x) \sin \beta_x + \left(\frac{K_2}{K_1}\right)^2 (r_x \sin \mu_x - b) \cos \mu_x}{\sqrt{K_2^2 (a + r \cos \mu_x)^2 - \left(\frac{K_2}{K_1}\right)^2 (r_x \sin \mu_x - b)^2}};$$

$$K_1 = 0.5\sqrt{\left[\sin(\delta+\rho)A + \frac{2}{\cos(\delta+\beta)}\right]^2 \tan^2\rho - \left[\cos(\delta+\rho)A\right]^2} \; ;$$

$$A = \tan(\delta+2\rho) - 2\tan(\delta+\rho) + \tan\delta ; \; K_2 = 0.5\left[\tan(\delta+2\rho) - \tan\delta\right],$$

where μ_x is the angle between radius r_x of an arbitrarily chosen point of the cutting edge and the axis of symmetry of the drill (for a point lying in the web of the drill angle μ_x is transformed into ϑ); 2ρ is the apex angle of the generating cone; $\delta = 90^\circ - \varphi$, where 2φ is the drill bit angle; α_x is the back relief angle at an arbitrary point of the cutting edge, which point lies on the cylinder surface of radius r_x. It is derived by evolving the cylinder surface of radius r_x into a flat surface, as the angle between the straight line of intersection of the cylinder with the drill bit cone (2φ) and a line tangent to the intersection curve of the same cylinder with the generating cone of sharpening (2ρ).

The margin. In order to reduce the friction of the drill against the working surface and the liberation of heat attendant upon it, the lip of the drill is depressed along its entire length, with the exception of a small margin left at the cutting edge. This margin is basically designed for guiding the drill in the process of cutting. The width of the margin is to be held to a minimum, otherwise excessive friction will develop between the margin and the work surface. The intermediate part (the angle) between the drill bit cone and the calibrating part of the drill is under maximum stress, due to the maximum cutting speed at the drill periphery and the maximum liberation of heat there. The intermediate part, as the weakest part of the drill, cannot provide for the proper elimination of heat. As a result, small particles of the metal being drilled become brazed onto the margin at the above mentioned intermediate angle, causing thereby a further increase in friction and in liberation of heat. This leads to the rapid wear and disintegration of the intermediate part of the drill. The recommended values for the margin are given in Table 3.

Drills having a diameter of 0.25 - 0.5 millimeters are made without a margin.

The margin values indicated in Table 3 are in effect for originally fabricated drills. In milling, the width of the margin is to be reduced, since it increases after grinding along the diameter.

TABLE 3

Margin values

Diameter of drill in mm	0.75	1	2	5	8	10	12	15	20	25	30	35	40	45	50
Width of margin in mm	0.2	0.3	0.4	0.6	0.7	0.8	0.9	1.0	1.2	1.6	1.8	1.8	2.0	2.3	2.6
Tolerance in mm	0.05	0.05	0.05	0.08	0.08	0.10	0.10	0.15	0.15	0.15	0.15	0.15	0.15	0.15	0.15

Back taper. For smoother operation (reduction of friction and heat), the drill is made with a back taper in the calibrating part, that is, the diameter of the drill bit at the shank is smaller than the diameter at the cutting part. The value of the taper per each 100 millimeters of length is to be within the following range:

Drill diameter in mm	Taper in mm
1 - 6	0.03 - 0.04
6 - 18	0.05 - 0.06
18 - 80	0.07 - 0.10

In the case of straight-shank 12-millimeter drills, the back taper runs the full length of the drill.

The reduction in the drill diameter resulting from subsequent resharpenings does not perceptibly affect the size of the drill hole, since it is compensated by the free play of the drill.

In continuous operation, the drill may lose its back taper on account of the excessive wear in the margin. This occurs very frequently in low-alloy high-speed steel drills. To avoid further wear and possible jamming, such drills are to have their back taper restored.

The holding part of the shank. Drills are made either with straight shank (diameter $\leq$ 20 millimeters) or tapered shank (diameter $\geq$ 5 millimeters).

Straight-shank drills up to 6 millimeters in diameter may also be made on reverse centers. In order to avoid the turning of the drill in the chuck or the drill holder in high-speed operations, the straight shank has a flattened rectangular end to fit the driving slot.

Taper-shank drills are made with a Morse taper. The moment of torque being generated in the process of drilling is to be relayed by the taper exclusively, without the participation of the tang, the purpose of which is only to force the drill out from the tapered sleeve of the spindle. The taper shank (see Figure 7) must be designed so as to sustain the full torque M, which is determined by formula

$$M = \mu \cdot P \frac{d_0}{2} = \mu \frac{Q}{\sin \varphi} \cdot \frac{D+d}{4} (1 - 0.04 \Delta \varphi),$$

where $\mu = 0.096$ is the coefficient of friction occurring between the surfaces of the sleeve and the tapered shank under the effect of the axial force Q; φ is one half of the apex angle of the cone; D and d are the maximum and minimum diameters of the working part of the shank taper.

The axial force Q may be resolved into P and V, it being the case that force P induces on the surface of the drill body a friction force $P \cdot \mu$. The term enclosed in parentheses accounts for the error in the angle of taper φ (on condition that the total error in the tapers of the sleeve and the shank $\Delta \varphi$ is not in excess of 10 minutes, it may be accepted for manufactured drills as $\Delta\psi = \pm 2$ minutes, for sleeves as ± 3 minutes, that is, a total error of ± 5 minutes.

[Drawing]

Figure 7. Forces acting upon the drill taper.

There is a constant ratio between M and Q in relation to the material being machined. The most unfavorable ratio $\frac{M}{Q} = 0.04D$) occurs in the case of soft steel. Considering the possibility of the occurrence of unfavorable circumstances (accelerated deflections in angle ψ, excessive blunting of the drill, jamming of the chip, and the like), we may accept, for purposes of computation, a three-fold increase in the value of the ratio $\frac{M}{Q}$.

Substituting the values for M into the formula, the maximum diameter of the drill, corresponding to each number of Morse taper, can be determined.

It must be noted that the maximum computed diameter of drills does not coincide with the established standards. In a coincidence of unfavorable circumstances, not only the taper shank, but also the tang may participate in the relaying of the torque. This is the usual cause for the breaking of the tang. With this in mind, the All-Union Committee on Standards published an additional standard, GOST 889 - 41 for drills with accelerated taper to be used in heavy-duty operation.

Progress in drill design. A twist drill of conventional design is not a perfect tool. Its substantial defect is, first of all, the sharp variation of the front rake angle throughout the entire length of the cutting edge. Suggestions for the improvement in the front rake angle design of the twist drill are, at best, compromise proposals.

The maximum stressed (by work load sustained and heat to be eliminated) area of the drill is its intermediate part, where the tapered shape blends into the cylindrical shape. This is the weakest area by reason of the excessive value of the front rake angle. To decrease the value of this angle, a special recess is made during the sharpening of the front surface. The drill is made with a greater angle of inclination and a helical flute of special design, as shown in Figure 8 by the continuous line BPB_1, in place of the usual flute, as shown by the dotted line APA_1.

[Drawing]

Figure 8. Drill with recess made in the sharpening of the front surface.

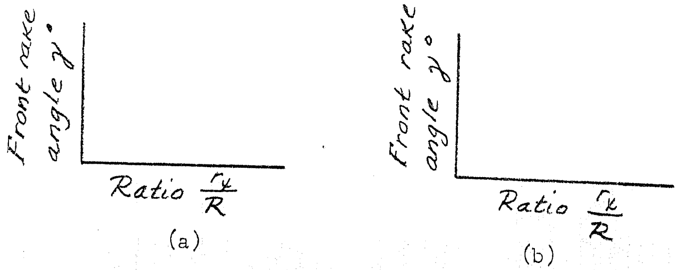

Figure 9. Variation in front rake angle in the presence of special recess in the front surface.

Special milling cutters are used in the fabrication of such drills. For the equalization of the front angle, the front surface at the drill periphery is additionally recessed out in sharpening. Area BP (cross-hatched surface in Figure 8) is ground off until it coincides with the rectilinear area AP. The front rake angle remains constant from A to P, and only at point P does it begin to decrease in a direction toward the web. The variation in angle γ (a) for the conventional drill, and (b) for the drill with specially recessed front surface, is presented graphically in Figure 9.

Some improvement, with relation to the wear of the drill, may be obtained by removing the slight bevel in the front surface along the cutting edge. The bevel usually varies in width. At the periphery of the drill it is at its widest, equalling half the feed value, and is gradually reduced to zero in the direction of the web.

The increase in the front rake angle at the web is obtained by recessing of the transverse edge and its simultaneous shortening. The transverse cutting edge, due to the excessive cutting angle $\delta = 90^\circ + \varphi$ (see Figure 2), works under heavy stress. It does not cut, but scrapes the material. It was established experimentally that about 65 percent of the force of feed and about 15 percent of the torque is sustained by the transverse cutting edge [chisel edge] of the drill. With its recess-sharpening, the axial cutting force is decreased, and the process of chip formation improved. The recess-sharpening of the chisel edge is particularly necessary in the case of worn-off drills, the web of which is considerably thickened toward the shank, and in the case of large-size drills.

[Drawing]

Figure 10. Recess-sharpening of the chisel edge of drill.

The recommended recess-sharpening of the chisel edge is depicted in Figure 10. To the left and to the right of the chisel edge, metal is removed so that recesses are formed. These recesses, in intersecting the back surface, form, in plan, straight lines, which are a continuation of the cutting edge from A to B. With this type of sharpening, the value of the front rake angle in the chisel edge zone is increased, with the chisel edge either foreshortened or left intact, as the case may be. Recess-sharpening does not weaken the chisel edge. It is done after each regular sharpening of the drill, or, at least, after two or three regular sharpenings. The length of the chisel edge after recess-sharpening, for drill sizes of from 12 to 80 millimeters, is accepted within the range 1.5 - 7.5 millimeters, and the length of the recess-sharpening along the drill, within the range 3 - 15 millimeters.

The detrimental effect of the chisel edge may be eliminated by cutting out a groove at the drill point perpendicular to the direction of the cutting edges. The cutting out is done by a grinding disk of a diameter not to exceed 1.5 millimeters.

In order to increase the durability of the drill and the cutting speed, particularly when drilling in cast iron, it is recommended that the drill be sharpened to a double angle (see Figure 11): the conventional sharpening to $2\varphi = 116 - 118$ degrees, and an additional sharpening to $2\varphi_1 = 70 - 75$ degrees. The width of the edge is made to be from 0.18 to 0.22 of the drill diameter. The higher durability of the drill obtained in this manner is due to an improvement in chip formation (a thinner and wider chip) and to an improved heat elimination. Double sharpening is recommended for drills having a diameter in excess of 10 - 12 millimeters. For small size drills it is not effective.

[Drawing]

Figure 11. Double-sharpening of drill.

To decrease the detrimental effect of the cylindrical margin, it is recommended that it be relieved at the cutting part for a length $l_d = 1.5 - 4$ millimeters, in the case of 12 - 80 millimeter drills (see Figure 12). The relieving of the margin is done to an angle $\alpha = 6 - 8$ degrees, leaving a slight bevel f within the range of 0.1 - 0.3 millimeters.

[Drawing]

Section AB

Figure 12. The relieving of the margin.

To cut down wear, the small intermediate angles where the taper ends and the cylinder begins should be curved out in the strip AB -- 5 - 6 millimeters long (see Figure 13). The radius of this curvature, in relation to the diameter of the drill, is to be within the range 0.5 - 1.2 millimeters.

View along arrow C

[Drawing]

Figure 13. Rounding out the angles.

In order to facilitate cutting, particularly in heavy-duty work, such as deep-hole drilling, the drills are sometimes equipped with chip breakers (see Figure 14). They assist the disintegration of the chip, facilitate its outflow through the flutes, reduce the heat formation and the pressure in the cutting process. They make for greater efficiency without diminishing the durability of the drill.

[Drawing] [Drawing]

(a) (b)

Figure 14. Chip breakers.

The flutes for the chip breakers are milled in the front surface along the entire working part of the drill. To eliminate the possibility of tubercles being left in drilling, the chip breakers in the two cutting edges are displaced with relation to each other. The chip breakers are milled at an angle of from 4 to 8 degrees, depending on the angle of inclination of the drill flute and on the drill diameter (the helical lines of the flutes and the chip breakers do not coincide). The flutes for the chip breakers are semicircular. They are easily machined, and they prevent the jamming of the chip. Drills with a diameter of 12 millimeters and higher are made with such chip breakers.

Another design of chip breaker (see Figure 14, b) places them on the back surface. It is recommended for small size drills and for drilling in ductile metals (soft steel, bronze, brass, and the like). This type of chip breaker is made in the sharpening machine with the aid of a slitting disk.

For the machining of light metals, drills with a variable pitch of the helical flute are recommended. The pitch is made either lower (Figure 15, a) or higher (Figure 15, b) at the drill point than at the shank. In the first case (a), the front part of the drill promotes easy cutting, while the following part promotes the easy outflow of the chip. In the second case (b) the front part of the drill flute allows the use of a higher feed value, and the following part promotes a continuous outflow of the chip. Drills a are very effective in automatic lathes, particularly in the case of rotating work with a non-rotating tool.

(a) [Drawing]

(b) [Drawing]

Figure 15. Drills with variable pitch.

Designing the profile of the milling cutter for the milling of the drill flute [4]. The milling cutter profile for milling the drill flute has a considerable effect upon the design of the drill. It is designed in relation to the following (see Figure 16): (1) the form of the drill flute; (2) the cutting part angle 2 ψ ; (3) the angle of inclination ω of the helical flute; (4) the angle of the milling cutter setup to the drill axis (Θ); (5) the location of point S, which is the point of intersection of the axis of the milling cutter mandrel with the drill axis; (6) the diameter of the milling cutter.

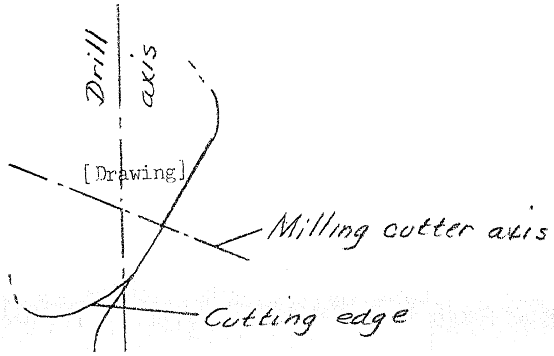

Figure 16. Milling cutter setup for milling the drill flute.

In setting up the milling cutter, it is necessary to know all the above enumerated values. The form of the flute is stipulated by the thickness of the web, the width of the flute, and the form of the cutting edge (usually rectilinear).

Angle 2 φ affects the form of the cutting edge. A milling cutter, designed for the machining of a drill with an angle 2 φ = 116 degrees, will produce, in sharpening the drill to an angle 2 φ = 90 degrees, a convex cutting edge, and in sharpening the drill to an angle φ = 140 degrees, a concave cutting edge.

The effect of angle ω on the milling cutter profile is of practical importance. It is sometimes necessary to machine drills with different values of angle ω with the same milling cutter as, for instance, in the case of drills made from high-speed steel and carbon steel. It is important to know the variation in the milling cutter profile with the variation in the value of angle ω in order to compensate the resulting distortion by a corresponding adjustment in the machine setup. With a change in the value of angle ω, the profile configuration of the milling cutter changes in its main as well as in its auxiliary part. Changing the location of point S does not result in the full coincidence of the profiles, making it necessary for the setup to be limited to the coinciding of the main part of the milling cutter profile, which will insure a rectilinear cutting edge for the drill. The non-coincidence of the auxiliary parts of the profiles can be disregarded. Hence, a small variation in the angle ω still allows the indirect utilization of the milling cutter.

The setup angle Θ is to differ from the milling angle $\tau = 90^\circ - \omega$. It is to be greater or smaller by 1 - 2 degrees than τ. Such a selection insures a higher degree of surface finish and hinders the additional undercutting of the drill flute already milled. Angle Θ has no effect upon the milling cutter profile.

Location of point S characterizes the transverse drift of the milling cutter in milling the drill flute. It has a considerable effect upon the profile of the milling cutter. With a decrease in x (see Figure 17), the part of the milling cutter profile corresponding to the cutting edge of the drill, becomes steeper and is subject to accelerated wear, due to small side angles, with the width of the milling cutter being at its minimum. Such a form is not suitable for back-face relieved milling cutters, but is fully acceptable in the case of sharp-toothed milling cutters. With x increasing, the important part of the milling cutter profile becomes more shallow, its width is increased, and the side angles improved. The "Frezer" (Miller) Plant applies the symmetrical location of point S ($x = y$) in the case of carbon steel drills, and the non-symmetrical location ($x < y$) in the case of high-speed steel drills.

The profile of the milling cutter does not coincide with the profile of the drill flute. This is due to the fact that individual points of the milling cutter profile are tangent to the helical surface of the flute in various cross sections of the drill. The cutting out of the profile of the flute is effected not along a curve in a plane, but along a spatial curve. The maximum non-coincidence of profiles occurs with the extreme locations I and V (see Figure 17). At location III practically full coincidence of profiles on the first surface of the drill takes place.

of the milling cutter in milling the drill flute. It has a considerable effect upon the profile of the milling cutter. With a decrease in x (see Figure 17), the part of the milling cutter profile corresponding to the cutting edge of the drill, becomes steeper and is subject to accelerated wear, due to small side angles, with the width of the milling cutter being at its minimum. Such a form is not suitable for back-face relieved milling cutters, but is fully acceptable in the case of sharp-toothed milling cutters. With x increasing, the important part of the milling cutter profile becomes more shallow, its width is increased, and the side angles improved. The "Frezer" (Miller) Plant applies the symmetrical location of point S $(x = y)$ in the case of carbon steel drills, and the non-symmetrical location $(x < y)$ in the case of high-speed steel drills.

The profile of the milling cutter does not coincide with the profile of the drill flute. This is due to the fact that individual points of the milling cutter profile are tangent to the helical surface of the flute in various cross sections of the drill. The cutting out of the profile of the flute is effected not along a curve in a plane, but along a spatial curve. The maximum non-coincidence of profiles occurs with the extreme locations I and V (see Figure 17). At location III practically full coincidence of profiles on the first surface of the drill takes place.

[Drawing] [Drawing] [Drawing]

Drill Drill Drill

axis axis axis

[Drawing] [Drawing] [Drawing]

Drill Drill Drill

axis axis axis

Figure 17. Location of the point of intersection of the drill axis and the axis of the milling cutter arbor.

The location of point S is fixed during the setting up of the machine. The circumference resulting from the intersection of the milling cutter by a plane normal to its axis and passing through point S, corresponds to the circular graduation line marked out on one tooth of the milling cutter. In setting the operating table of the machine, the point of the center selector is set against this graduation line. The fixture for the setting of the milling cutter is shown in Figure 18. The location of point S is characterized by dimension k marked out on the milling cutter profile gage. The fixture consists of a sleeve (1), which is slipped over the calibration drill blank (2). The milling cutter is set in its arbor (4). Dimension k is maintained with the aid of setup washers (3) slipped over mandrel (5). The coincidence of the axis of the milling cutter arbor (4) with the axis of mandrel (5) is attained with the aid of the center selector. After the milling cutter is set, sleeve (1) is removed, the actual drill blank is put in the place of the calibration drill blank. The diameter of the calibration blank may differ from the diameter of the actual drill blank.

Section AB

[Drawing] [Drawing]

Figure 18. Milling cutter setup fixture in the milling of the drill flute.

The diameter of the milling cutter is selected in keeping with the design features of the machine. The change in this diameter, entailed in the resharpening of the milling cutter, does not practically affect its profile.

The design of the milling cutter profile (see Figures 19 and 20 printed on insert sheet) consists of the following components: (1) the flute cross section perpendicular to the drill axis; (2) the helical lines of the flute surface; (3) the trails of the helical flute intersected by planes perpendicular to the axis of the milling cutter arbor; (4) the radii of the milling cutter cross sections; (5) the milling cutter profile; (6) the drill flute and drill web cross sections.

Projection A

Projection B

[Drawing]

Figure 19. Diagram of the drill flute cross section normal to drill axis.

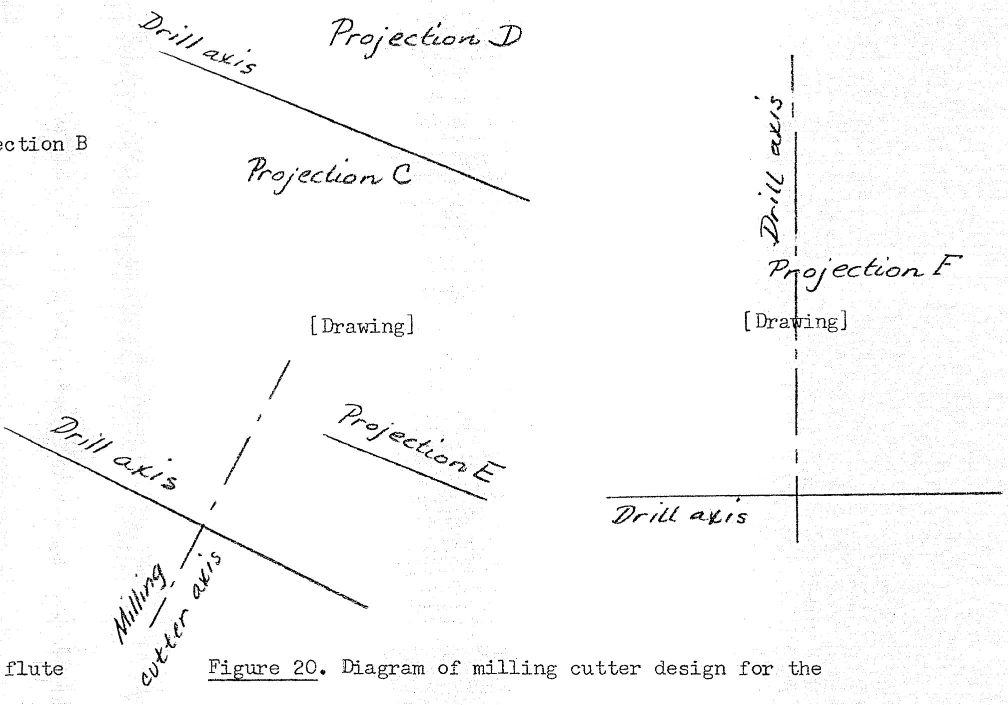

Figure 20. Diagram of milling cutter design for the milling of the drill flute.

1. Let us dissect the taper by a series of straight lines perpendicular to the drill axis, at a distance from each other equal to 1/128 of the helical surface pitch (Figure 19). We project points a_o'', b_o'', c_o'', etc., onto the straight line a_of_o (the cutting edge). Through points a_o, b_o, c_o, and so on, we draw circles. For reasons of convenience, we turn edge a_of_o by 90 degrees. Let us divide the external circle into 64 parts, corresponding to 16 parts between point a_o and point a_{16}. Where the tapered surface becomes cylindrical, the straight-line edge becomes a smooth curve. When the drill is turned by 1/128 of the circle, the milling cutter will bite into the body of the drill to the extent of 1/128 of the pitch. Point b_3 in the straight line a_3b_3 will assume on the circumference b a new position b_3', which is 1/128 of the circumference distant from its original position b_3. Analogously, all the other points c_3, d_3, etc., of the straight line, will be shifted to positions c_3', d_3', etc., on the respective circumferences. The points thus obtained we now connect with a curve. For reasons of convenience, it is recommended that the angle $\frac{360^\circ}{128}$ be constructed separately. By matching, we substitute for the above curve an arc of a circle, the radius of which is R_1. For the determination of the auxiliary part of the profile, we connect point O_1 with the center O, and assume (conditionally) that center O_2 of the radius R_2 of the second curve lies on this straight line. Knowing R_1 and R_2, we can draw from points a_3, a_4, a_5, etc., and also from points a_3', a_4', a_5', etc., the curves of the flute profile.

2. In order to construct the helical lines (Figure 20), we intersect the drill axis by planes 1, 2, 3, 4, etc., normal to it and at a distance from each other equal to 1/64 or 1/128 of the pitch. To project the helical line a, we project from Figure 19 points a_3, a_4, a_5, etc., respectively, onto the straight lines 1, 2, 3, etc. By doing this we obtain, in Figure 20, points a_3, a_4, a_5, etc., which, when joined with a smooth curve, produce the helical line a. In a similar manner, we determine the other helical lines.

3. We now dispose the axis of the milling cutter arbor at an angle $\Theta = 90^\circ - \omega - 1^\circ$ to the drill axis and select a symmetrical location for point S. Then we dissect the flute by planes I, II, III, etc., arbitrarily equidistant from each other and perpendicular to the milling cutter arbor axis. In Figure 20, these planes produce curves, the trails of the intersection of these planes with the helical flute. Projection D shows the curves for the working part of the profile, and projection E, for the intermediate part (for convenience, this is constructed by the American method of projection). As an example, we will analyze the construction of the trail for plane III. This plane intersects the helical lines at points a_{III}, b_{III}, c_{III}, etc. In projection D, the points of the trail lie on the extended straight lines perpendicular to plane III. Point a_{III}, by height, in projections S and A, is at the same distance $a_{III}x$ from the horizontal drill axis, and is at a distance $a_{III}y$ from the vertical axis of the drill. In projection D, segment $a_{III}y$ is equal to the distance between

point a_{III} and the drill axis. The other points b_{III}, c_{III}, etc., of the curve are determined in a similar manner.

4. The intersection of the milling cutter by planes I, II, III, etc., results in concentric circles. Assuming the existence of an infinite number of such intersecting planes, the milling cutter can be visualized as consisting of an infinite number of disks put together. Each disk represents a definite point in the milling cutter profile. Specifically at this point, the disk is tangent to the curve formed by the intersection of the drill flute and the corresponding plane. Thus, points in the milling cutter profile are located by drawing from a certain center of concentric circles which are tangent to the corresponding curves of the drill flute. The center of the milling cutter I lies on the continuation of the milling cutter arbor axis 0-0. The distance of the drill axis from this center depends on the milling cutter diameter and the thickness of the drill web. The radius of the circle tangent to curve III is the distance by which a point in the milling cutter profile is separated from its axis in the intersection by plane III. To determine the radii, it is sufficient to locate the points of tangency of the circles with the curves (without outlining the concentric circles proper in order not to darken the drawing). When the radii are established, segments g_1, g_2, g_3, etc., representing the distances of the corresponding points of tangency from the drill axis, can be ascertained.

5. In order to construct the profile (projection F),

we draw the drill axis X'-X' and a series of straight lines perpendicular to it, the distance between them to be equal to the distance between the planes I, II, III, etc. From the drill axis, on the corresponding straight lines, we lay off the segments g_1, g_2, g_3, etc., and connect the resulting points by a smooth curve, which curve represents the requisite profile of the milling cutter. By matching, it may be substituted by the arcs of circles with the radii R_1 and R_2. The process of cutting by the auxiliary part of the profile ends at its lower points; therefore, the non-working part may be sheared off by way of drawing a line tangent to the vertical at a 10 degree angle.

6. In order to compare the derived profile of the milling cutter with the profile of the drill flute, in the section passing through axis O-O (projection C), we mark off the points of intersection of curves I, II, III, etc., with the axis of the milling cutter arbor (projections D and E). The distances between these points and the drill axis are carried onto the corresponding planes I, II, III, etc., in projection F. Connecting the points by a smooth curve, we derive the profile of the drill flute (shown in projection F by dotted line).

To make the drawing complete, we construct the ellipses corresponding to the circles of the periphery and the web of the drill. The ellipse of the web is tangent to the milling cutter profile only in the case when point S is located on the intersection of the milling cutter arbor with the helical line in the cylindrical surface of the web.

Tolerances. By their external diameter, sharpened drills are fabricated with departures toward the negative from the nominal diameter as per GOST 885 - 41 (Table 4).

TABLE 4

Drill diameters in mm	0.25-0.5	0.5-0.75	0.75-1	1-3	3-6	6-10	10-18	18-30	30-50	50-80
Tolerance in mm . . .	0.01	0.015	0.02	0.025	0.03	0.036	0.043	0.052	0.062	0.074

For the total length and for the working length of the drill, a tolerance equal to double the tolerance of the ninth class of precision (OST 1010), with a symmetrical location of the tolerance limits about the nominal sizes, is allowed.

The tolerance limits for the length of the holding part for straight-shank drills are established by $\underline{V}_{8}$ (OST 1010), and for the thickness of the holding part, by $\underline{Kh}_{5}$ (OST 1015). The displacement of the holding part axis about the drill axis is to be within the limits of 0.5 of its thickness tolerance. Both margins of the drill are to lie in the same surface of rotation, coaxial with the surface of the drill shank. The doubled deviation from coaxiality (free play) for 3 - 20 millimeter diameter straight-shank drills is to be tolerated up to 0.08 millimeter, for up to 20-millimeter taper-shank drills, the doubled tolerance is not to exceed 0.12 millimeter, for 20-50 millimeter drills, not above 0.15 millimeter, for over 50-millimeter drills, not above 0.18 millimeter.

Free play is usually checked at the point where the calibrating part begins, with the shank clamped.

Specifications for the inspection of twist drills are cited in GOST 2034-43.

Other Types of Drills

Center drills are used for the drilling of center holes (Figure 21).

(a)

(b)

(c)

Figure 21. Types of center drills: (a) plain center drills; (b) combination center drills; (c) combination center drills with safety taper.

Center drills are fabricated in 3 types: (a) plain center drills (OST 3727); (b) combination center drills (OST 3732); (c) combination center drills with safety taper (OST 3733).

Plain center drills, in their design, are no different from conventional twist drills.

Combination center drills (Figure 22) are fabricated double-ended for better utilization of material. The flutes are either straight or slanting (rarely helical), with angle of inclination $\omega = 5 - 8$ degrees. The cutting part angle is 59 - 60 degrees, and the chisel edge angle is 50 - 55 degrees. The back taper is 0.05 - 0.10 millimeters for the total length of drill. The web thickness is $C = (0.15 \div 0.17)D$, increasing in the direction of the shank at an angle of 3 degrees. The front rake angle is 5 - 6 degrees. The sharpening procedure for such a drill is the same as for conventional twist drills. The back relief angle on the peripheral cutting part is 8 degrees. The calibrating part and the taper section for countersinking have their back surface relieved without leaving a margin. To obtain a uniform back surface relieving and avoid friction against the work surface by the intermediate part (where the cylindrical surface begins to taper), the relieving is done at an angle of 10 - 12 degrees to the drill axis.

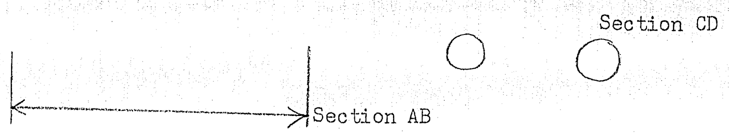

Figure 22. Combination center drill design.

The cams for the relieving of the back surface are calculated to produce a back relief angle not to exceed 2 degrees - 2 degrees 30 minutes. Dimensions for the calculation of the cams are cited in Table 5.

TABLE 5

Dimensions for the calculation of cams
for the back surface relieving of drills

For drills with diameter in mm	1	1.5	2	2.5	3	4	5	6
K_1 in mm . . .	0.055	0.083	0.110	0.138	0.165	0.220	0.270	0.330
K_2 in mm . . .	0.149	0.225	0.298	0.375	0.448	0.598	0.745	0.895

The radius of flute curvature (the egress of the milling cutter) R = 15 - 23 millimeters. The radius of the flute bottom r = 0.2 - 1.3 millimeters.

The tapered part is at a 60 degree angle and the safety taper at a 120 degree angle. The purpose of the safety taper is to prevent the center hole from being clogged.

The weakest area in a center drill is where the cylindrical surface begins to taper. To reinforce this area, it is recommended that it be outlined by 0.5 - 1.0 millimeter radius or supplied with a band. The band will also help in the detaining of lubricant on the center due to the indentation in the blank.

Pointed drills. The cutting part in these drills is made in the form of a blade equipped with cutting elements (Figure 23). They are used in cases where rigidity of tool is required, as, for instance, in the machining of hard forgings and castings, and also in the machining of staggered or special profile holes. Large diameter pointed drills are usually made with inserted bits.

The angle φ is selected in relation to the material machined. For general purposes, the angle φ is to be 59 - 60 degrees. The chisel edge angle $\vartheta = 55$ degrees. In order to reduce friction, the calibrating part of the drill is made with a side angle $\psi = 5 - 8$ degrees and a back taper within the range of 0.05 - 0.10 millimeters for the overall length of the drill. To facilitate the drilling operation, the cutting edges are supplied with chip breakers (Figure 24). They are to be disposed non-symmetrically about the drill axis. Their width is 2 - 3 millimeters; the distance between them is 8 - 12 millimeters. The flutes are to become wider with the distance from the cutting part. The front surface of the drill protrudes forward, beyond the normal NN, and the front rake angle γ has a negative value. Due to a large cutting angle $\delta > 90$ degrees, a pointed drill works under more difficult conditions than a twist drill. To render angle δ more favorable, the front surface is recessed so that $\gamma \geqslant 0$. When the recess is deep, the blade is weakened, and the general strength of the drill is diminished. The back relief angle is selected within the range of 10 - 20 degrees, it being the case that for drilling in ductile and soft materials its value is higher than for drilling in hard and brittle materials.

Section CD

Figure 23. Pointed drills.

[Drawing]

Figure 24. The cutting part of a pointed drill.

Straight-flute drills are used for the drilling of holes in thin sheets of ductile metal, such as brass. In contradistinction to twist drills, the screwing in and jamming effect of the drill in the hole is absent. Their cutting elements are selected in the same manner as for twist drills. Their shortcoming consists in the fact that, like pointed drills, their cutting angle is greater than 90 degrees, which handicaps the work of the drill.

Drills with hard-alloy-tipped blades. Due to the physical properties of hard alloys, these drills have only limited application. They are suitable for drilling in materials that do not require high values for the front rake angle, such as cast iron (particularly in the presence of casting "skin"), hard steels, plastic masses, ebonite, bakelite, glass, and the like, and also in those cases when the feed values for hard alloy tools and high-speed steel tools are fairly equal. In this case, the productivity of the machine tool is increased due to the utilization of higher cutting speeds, as, for example, in the machining of light alloys, cast iron, etc., in high-speed machines. Due to the low strength of the hard-alloy-tipped blade and the necessary presence of considerable cutting angles, it becomes necessary to forego the utilization of hard alloys in the drilling of ductile metals (such as steels with $\sigma_b < 100$ kilograms per square millimeter).

Drills with hard-alloy-tipped blades are made in sizes ranging from 3 to 50 millimeters, it being the case that in small-size drills (up to 8 millimeters), in place of a complete brazed-in blade, a cutting hard-alloy-insert is butt-brazed to the holder. The diameter of the holder is made smaller by 0.3 - 0.5 millimeter than the diameter of the insert. Hard-alloy-tipped drills must comply with rigid specifications as to strength, reliability, simplicity of blade bracing, and rigidity of the entire structure. The drill is to resist well the cutting force and is not to show any yielding during its work. The vibration in the drill is the basic cause for the chipping of the blade. Due to the weakness of the drill point and the drill web, twist drills are not very amenable to hard-alloy tipping. To reinforce the drill, it becomes necessary to enlarge the web and to thicken it out in the direction of the shank.

The length of the hard-alloy-tipped drill bit is made smaller as compared with conventional drills, since they can be utilized only (resharpening stock) in a length equal to that of the hard-alloy blade. The smaller length of the drill bit results in a smaller degree of yielding during the drilling operation.

To increase the durability of the margin on the calibrating part of the drill holder, drills designed for the machining of particularly hard materials are made from hardened high-speed steel. In other cases, the material for the drill holder can be alloy- or high-carbon steel.

The design of the drill varies in relation to the designation of the drill (type of material to be machined, depth of drilling).

In the drilling of shallow holes, the direction of the flutes has no inportant bearing on the chip outflow. The flutes may even be straight. For deep-hole drilling, drills with helical flutes must be used. The angle of inclination ω is selected in relation to the material machined, being 10 - 15 degrees in the case of hard materials giving an overflow chip (Figure 25, a) and 55 - 60 degrees in the case of brittle materials giving a spalling chip (Figure 25, b). As can be seen from Figure 25, b, the calibrating part of the drill acts as a screw conveyor for leading out the chip formations. Angle 2φ for the cutting part of the drill, in the case of materials giving an overflow chip, is selected at 125 - 130 degrees, and, in the case of materials giving a spalling chip, at 116 - 118 degrees. To avoid weakening, the front surface of the blade is sharpened to a small front rake angle γ within the range of 0 - 3 degrees for materials giving an overflow chip, and within the range of 4 - 7 degrees for materials giving a spalling chip. Table 6 cites the data on the selection of angles 2φ and γ in relation to various materials to be machines.

Blade

(a)

Blade

(b)

Figure 25. Drills with hard-alloy blades.

TABLE 6

Data for the selection of angles 2φ and γ for various materials

Material to be machined	2φ	γ
	in degrees	
Cast iron $H_B = 200$	120	+7
$H_B = 300 - 400$	140	0
Steel castings, stainless steel, chromium-nickel steel σ_b = 140 kilograms per square millimeter	120	0
Tool steel, foundry chill castings, manganese steel	140	-2-3
Malleable iron	120	+4
Phosphor bronze	130	+4

The transverse cutting edge [the chisel edge] should be sharpened. The back taper, for the length of the blade, is selected within the range of 0.03 - 0.05 millimeters. To facilitate the flow of the chip, the flute surfaces must be well burnished.

Deep-hole drills [1, 2, 5]. Deep-hole drilling is drilling to a depth exceeding the diameter of the drill 5 or more times. There is continuous and radial drilling. In the latter case, not all the metal is formed into chip -- a bar remains in the center of the blank. This bar is removed, depending on its size, by breaking off or undercutting. The machining is done in a boring lathe, usually, with rotating work and forward motion of tool, and, less frequently, with both rotating work and rotating tool. Specifications for deep-hole drilling are as follows: the drill-hole axis is to run true to a straight line; the drill hole is to be concentric in relation to the external surfaces; the drill hole is to run cylindrically true throughout its entire length; the degree of finish and precision is to run within the range of OST second and third class. Deep-hole drills run in size within the range of 6 - 400 millimeters, and their various designs are stipulated by the various sizes and specifications of the work.

Ordnance drill. An ordnance drill is a cylindrical bar sheared off approximately to half its diameter (Figure 26). To avoid jamming, the front surface is made higher than the center by the value $f = 0.2 \div 0.5$ millimeters, depending on the size of the drill. The main cutting edge is at a right angle to the drill axis, and the auxiliary cutting edge is sheared off at an angle of 10 degrees. At its beginning, it is set off from the drill axis by 0.5 millimeter. The drill point is chamfered to a radius of 1 - 1.5 millimeters. Along the entire length of the drill bit, a strip is sheared off at an angle of 30 - 45 degrees. The back relief angle is equal to 8 - 10 degrees. The back taper is accepted within the range of 0.03 - 0.05 millimeters per 100 millimeters of length. Sometimes the drill is made with a recess in the front surface (see section NN).

Section NN

Figure 26. Ordnance drill.

By the characteristics of its operation, an ordnance drill resembles a boring cutter, and must, therefore, be placed in a jig, or begin its work from a preliminarily rough-drilled hole in order to provide an adequate bearing surface. The drill operates under difficult conditions due to the large cutting angle (90 degrees), to the difficulties in the egress of the chip and in the admission of coolant. Another defect of the ordnance drill is the non-warranty of the true geometric axis of the drill hole due to the drift of the drill.

A rifle drill consists of two parts: the working part [the bit], 60 - 150 millimeters long, made from high-speed steel, and the clamping part, made from carbon steel, the end of which is inserted into a sleeve for fitting into the chuck. The bit is equipped with a round or sickle-shaped hole (with an angle of 130 - 140 degrees) for the admission to the cutting edge of the coolant mixture (Figure 27). On its return, the coolant liquid, together with the chip, flows out along the flute. The flute angle ψ plays an important part. Due to the great drilling depth, the drill sustains longitudinal bending and twisting stresses, which necessitates a provision for the adequate rigidity of the holder, particularly when drilling to small diameters. Angle ψ also stipulates the dimensions of the flutes, along which the feeding and the leading-off of the coolant and the chip take place. With a decrease in the value of angle ψ, the rigidity of the holder and the velocity tension are increased, with the attendant increase in the friction of the chip against the flute surfaces and the danger of the chip becoming wedged in the flute. It is recommended that angle ψ be kept within the range of 100 - 120 degrees.

Section AB Section CD

Figure 27. Rifle drill.

The drill has one cutting edge, consisting of two parts: external and internal.

To guide the drill and to facilitate its penetration of the metal, the drill point is displaced about the drill axis by the distance b (see Figure 28). This displacement forms a taper (see Figure 29, a), which acts as a bearing for the drill, providing a guide in the cutting process. The value of displacement b has a considerable effect upon the work of the drill (upon its drift, its durability, the finish of the machined surface, and the like), and it is to be in relation to the drill design and sharpening, and also to the characteristics of the work material. Usually, $b = a = 0.25D$ (drill diameter). It is, however, better to select the value of b as smaller than the value of a (such as $b = 0.2D$; $a = 0.3D$), and angle φ as less than angle φ_1 (such as φ = 50 degrees and φ_1 = 70 degrees). Under these conditions, the calibrating edge, having a bevel f, will sustain a minimum of pressure, since the cutting force component which is perpendicular to the drill axis will be of a greater value for edge N than for edge W ($P_N > P_W$ in Figure 28).

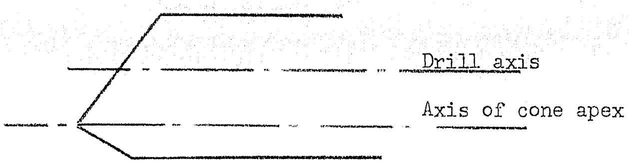

Figure 28. Rifle drill bit.

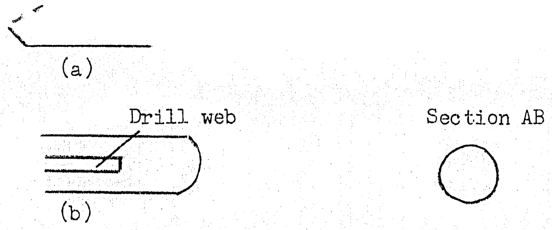

Figure 29. Rifle drill in operation.

The calibrating bevel cannot penetrate the work metal, and merely removes the combings. This prevents the drift of the drill and the excessive widening of the drill hole, and also increases its durability. A somewhat increased pressure toward the side opposite to bevel f is sustained by the cylindrical surface of the drill body or by the solid guide bevel.

To reduce friction, the drill bit is made with a back taper within the range of 0.10 - 0.30 millimeters per 100 millimeters of running length in the case of 5 - 40 millimeter drills. To reduce the contact surface and increase the effect of the coolant liquid, strips with narrow margins cut out are provided. Margin f (see Figure 27) designated for the trimming and the calibrating of the drill hole is to be within the range of 0.4 - 0.6 millimeter. With excessive values of f, the drill has a tendency to jam.

The cylindrical bearing surface must lie opposite the margin f. The remaining margins are guides, and their dimensions are selected by considerations of design, with due note taken of the width of the strips. The depth of the strips is usually 0.15 - 0.25 millimeters. The flute apex is to be below the center of the drill (by $h = 0.05 - 0.18$ millimeter), otherwise it will not function, and may bend, or even break. During the running of the drill, with the flute apex below the drill center, a core is formed in the center of the drill hole (see Figure 29, b). The diameter of this core is the greater, the lower the flute apex. To make the breaking off of this core easier, its diameter must not exceed 0.03 of the drill hole diameter. The front rake angle is to be kept within the range of 5 - 8 degrees, the back relief angle on edge N -- within the range of 8 - 10 degrees, the back relief angle on edge W -- within the range 12 - 20 degrees, the drill point angle ϵ = 120 - 130 degrees.

The front surface is equipped with chip breakers. Their height, in relation to the value of feed per one revolution and to the ductility of the work metal, may be accepted as 17s, where s is the value of feed in millimeters. The chip breakers are to have undercuts at a 6 - 8 degree angle to provide for the corrugated form of the chip and the requisite direction of its egress.

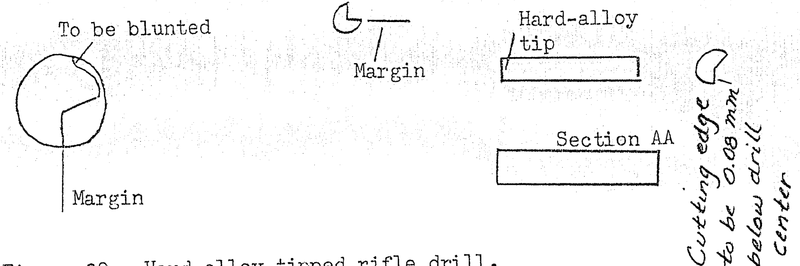

Figure 30. Hard-alloy-tipped rifle drill.

Experience is available in the use of hard-alloy-tipped rifle drills of a 7.5 - 13 millimeter diameter(see Figure 30). A hard-alloy adapter equipped with a flute for the chip flow and an angular catch is brazed with its face to a short carbon-steel holder, which, in turn, is brazed onto the basic tube.

Sharpening is effected with the aid of a diamond grinding disk mounted on a conventional drill-sharpening machine. The geometric parameters of this drill bit are presented in Figure 30.

A substantial defect of rifle drills is the presence of only one cutting edge, a circumstance which reduces its productivity. A conventional twist drill with flutes for admitting coolant, although equipped with two cutting edges, does not provide for high productivity, since the drill, from time to time, must be removed from the drill hole for the removal of the accumulated chip. Figure 31 shows the construction of a deep-hole twist drill. In the place of two, it has four margins, which form the channels for the admission of coolant. The shank has a drilled hole communicating with the hole at the end of the helical flutes which is perpendicular to it. Through these holes the chip is broken off in an outward direction.

[Drawing]

Figure 31. Deep-hole drill.

[Drawings]

(a) (b)

Figure 32. Improved designs for deep-hole drills.

Although this drill has high productivity, it does not provide for a good surface finish and is not free from drift. The drift is caused by the improper sharpening of the multi-blade drill and by the presence of the chisel edge. The detrimental effect of the latter may be reduced or even completely eliminated by the following: (1) the drilling out (in the drill) of a longitudinal hole, the diameter of which is to be greater than the chisel edge diameter (such a drill -- see Figure 32, a -- has an internal flow-off of the chip and, in drilling, will form a core); (2) the drilling out of a blind center hole, hidden in one of the helical flutes, with a ledge for the breaking off of the forming core, which is removed through the hole in an outward direction (Figure 32, b); (3) the substitution of one cutting edge by two cutting edges that really cut into the work metal and do not merely crush it (Figure 32, c).

Large-diameter holes, on the order of 75 millimeters and above, are drilled by the method of radial drilling, with the leaving of a central core of considerable diameter. Radial drilling is effected by a single-blade drill, or, more frequently, by a multi-blade type drill head. The head consists of a frame with clamped-in cutters, the number of which is selected in relation to the requisite diameter of the hole. Figure 33 shows the drill head face. Three cutters (2), wedge-braced (4), machine the metal from two sides (diameters D and d). Between the central core and the drill head frame (1) there is a gap of 3 - 6 millimeters for the admission of coolant. For the outflow of coolant with the chip, gap B is provided between the hole drilled out and the drill head frame. The drill head is equipped with metal or wood guide cams (3).

BIBLIOGRAPHY AND SOURCES

1. Aref'yev, M. G., and Karpov, L. I., Profuction of Rifle Barrels, Oborongiz, 1945.

2. Veremeychuk, I. S., "Radial Drilling", Mashinostroitel' (Machine Builder), no 12, 1940.

3. Nemirovskiy, A. S., "Computation of Back Relief Angles of Drill", Trudy stanko-instrumental'novo instituta (Transactions of the Machine Tool Institute), volume VI, Publication Stankin, 1940.

4. Semenchenko, I. I., The Cutting Tool, volume I, ONTI, 1936.

5. Suvorov, A. I., "Deep-Hole Drilling", Mashinostroitel' (Machine Builder), No 12, 1940.

COUNTERSINKS

Designation and Types

Countersinks are intended for:

(a) enlarging cylindrical holes previously obtained by cold or hot treatment;

(b) machining cylindrical recesses for screwheads or necks of screws, etc.;

(c) machining conical recesses for center holes under screwheads, for valve seats, and the like;

(d) trimming of face surfaces.

For the first group of operations, the following types of countersinks are used: (1) taper-shank (GOST V 1676 - 42); (2) inserted-blade (OST NKTP 3677) with arbors (OST NKTP 3678); (3) sectional adjustable inserted blade (GOST 2255 - 43).

For the second group of operations, the following types of countersinks are used: (1) with pin for enlarging cylindrical screwhead holes; (2) with pin for enlarging holes for screw necks.

For the third group of operations, countersinks known as countersink reamers are used: (1) 60 degree included angle plain centering (OST 3728); (2) 60 degree included angle for center holes without a safety cone (OST 3729 combined with OST 3730); (3) 60 degree included angle for centering with taper shank (OST 3731); (4) countersink reamers with various other included angle values (75 degrees, 90 degrees, etc.) for various operations.

For the fourth group of operations, spot-facing countersinks are used: (1) with shank; (2) inserted blade, one-side and two-side, solid and sectional.

Countersinks for Enlarging Holes

Designations, types, and the geometry of countersinks. Countersinks for enlarging holes are used for holes to be finished to the fourth to fifth classes of precision and for holes to be reamed to the second to third classes of precision. The external diameter of the countersink is selected to conform with the above, with the requisite stock allowance for reaming taken into account.

A countersink consists of the following parts and design elements (Figure 34): l - the bit; l_1 - the cutting part; l_2 - the calibrating part; l_3 - the neck; l_4 - the shank; e - the tang; d - the arbor-fitting hole; k - the flute (straight or helical, with right-hand or left-hand direction); p - the lip; f - the margin.

The basic elements of design are: (a) the cutting part; (b) the direction of the flute; (c) the blade angles; (d) the margin; (e) the back taper.

[Drawings]

Figure 34. Countersink parts and elements of design.

[Drawings]

Figure 35. Countersink cutting edges.

Margin on tapered part

[Drawing] [Drawing]

Figure 36. Geometric elements of countersink.

[Drawings]

Figure 37. Angle of inclination of the cutting edge of countersink.

In the technological process of machining the hole, the countersink occupies an intermediate place between the drill and the reamer. The design of it is stipulated by this fact. The countersink resembles a drill, but has a greater number of lips, which provides for better guiding and a higher degree of finish in the machined surface.

In contradistinction to a reamer, the countersink cuts by means of facing teeth, namely, by the face cutting edge m and the margin edge n (see Figure 35). The value of $m = \frac{t}{\sin \varphi}$, where t is the thickness of the chip; φ is the cutting part angle. The value $n = \frac{s_o}{z}$, where s_o is the value of feed per one revolution and z is the number of lips.

The geometry of the countersink tooth is depicted in Figure 36: the front surface (1), along which the chip is ejected; the back surfaces -- main (2) facing the work surface, and auxiliary (3), which is the cylindrical surface of the margin, touching the work surface; the main cutting edge (4), formed by the intersection of the front and main back surfaces, and performing the basic work of cutting; the auxiliary cutting edge (5), formed by the intersection of the front surface and the auxiliary back surface (the margin); blade point (6), which is the point of intersection of the main and auxiliary cutting edges.

To determine the cutting angles, it is necessary to know the coordinating planes -- the basic plane and the cutting plane.

The basic plane passes through the given point of the main cutting edge and the countersink axis perpendicular to its face.

The cutting plane passes through the given point of the main cutting edge tangent to the surface of cutting. In analyzing the cutting angles in a static position, the cylindrical surface generated by the rotation of the given point of the cutting edge (without feed) is considered as the surface of cutting.

The angles in plan (main φ and auxiliary φ_1) are included between the direction of feed and the corresponding projection of the main or auxiliary cutting edge, respectively, upon the basic plane (see Figure 35).

The cutting edge angle of inclination λ is included between the main cutting edge and the basic plane. It is measured in a plane passing through the given point of the cutting edge normal to the basic plane. By analogy with a cutter, the cutting edge of the countersink may be designed in three variants (see Figure 37). When the point of the tooth is below the remaining points of the cutting edge, angle λ is positive ($\lambda > 0$), when it is above, angle λ is negative ($\lambda < 0$), when it is at the same height, $\lambda = 0$.

The front rake angle γ is formed by a plane tangent to the front surface at the given point of the cutting edge, and a plane normal to the cutting plane, drawn through the same point.

The back relief angle α is formed by a plane tangent to the back surface at the given point of the cutting edge, and the cutting plane passing through the same point.

To complete the determination, it is necessary to establish the coordinating plane. The front rake angle γ and the back relief angle α of the countersink are usually specified, by analogy with OST 6898 ("Basic concepts in machining with cutters"), in the main section plane normal to the projection of the cutting edge upon the basic plane (see Figure 38). The back relief angle can also be measured in a plane tangent to the surface of motion.

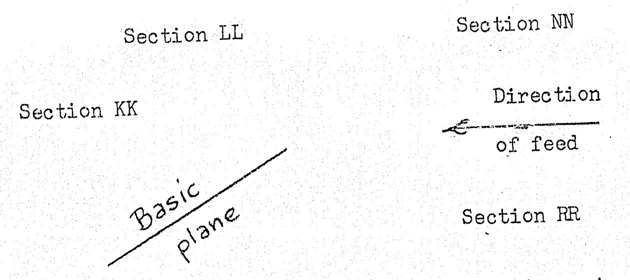

Figure 38. Cutting edge angles measured in various planes.

When designing and fabricating countersinks, it is necessary, by analogy with cutters, to operate with front rake and back relief angles measured in planes: (1) KK -- normal to the direction of feed (α_1 , γ_1); (2) LL -- parallel to the direction of feed, tangent at the given point to the cylindrical surface generated by the rotation (without feed) of this point (α_2, γ_2); (3) RR -- normal to the cutting edge (α_r, γ_r).

The relationships between the front rake and back relief angles as measured in various planes are:

$$\tan \gamma_1 = \tan \gamma_N \cdot \cos \varphi - \tan \lambda \cdot \sin \varphi ;$$
$$\tan \gamma_2 = \tan \gamma_N \cdot \sin \varphi + \tan \lambda \cdot \cos \varphi ;$$
$$\tan \gamma_N = \tan \gamma_1 \cdot \cos \varphi + \tan \gamma_2 \cdot \sin \varphi ;$$
$$\tan \gamma_r = \tan \gamma_N \cdot \cos \lambda ;$$
$$\cot \alpha_1 = \cot \alpha_N \cos \varphi - \tan \lambda \cdot \sin \varphi ;$$
$$\cot \alpha_2 = \cot \alpha_N \cdot \sin \varphi + \tan \lambda \cdot \cos \varphi ;$$
$$\cot \alpha_N = \cot \alpha_1 \cdot \cos \varphi + \cot \alpha_2 \cdot \sin \varphi ;$$
$$\cot \alpha_r = \cot \alpha_N \cdot \cos \lambda .$$

When operating with formulas containing λ , it is necessary to adhere to the rule of signs for the tangents.

Angle λ is determined by formulas:

$$\tan \lambda = \tan \gamma_2 \cdot \cos \varphi - \tan \gamma_1 \cdot \sin \varphi ;$$
$$\tan \lambda = \cot \alpha_2 \cdot \cos \varphi - \cot \alpha_1 \cdot \sin \varphi .$$

For the determination of λ_1 in the face section, there is the formula

$$\tan \lambda_1 = \frac{\tan \lambda}{\sin \varphi} .$$

The cutting part angle φ exerts a considerable effect on the form and the ejection of the chip, and is selected on the basis of experimental data. A properly designed angle φ is conducive to the proper ejection of the chip as conformant with the direction of the flute. This is of particular importance in the case of metals that give an overflow chip. In machining steel, angle φ is taken as 60 degrees. To increase tool durability in the machining of steel, the cutting edge is to be additionally sharpened at angle φ = 30 degrees, along a length equal to three times the value of the machining stock at the side. For machining in cast iron, angle φ is taken as 60 or 45 degrees without additional sharpening. In the case of hard-alloy-tipped countersinks, angle φ is sometimes increased to 75 degrees.

Like a twist drill, a countersink is equipped with a helical flute. To form a positive front rake angle, the direction of the flute must coincide with the direction of cutting. In plane LL, the front rake angle γ_2, for a point on the periphery, equals the angle of inclination (ω) of the helical flute. Angle ω, which is linked with the front rake angle, is selected in relation to the work material and the countersink diameter. As the hardness of the work material is increased and the diameter of the countersink reduced (for the strengthening of the cutting edge), angle ω is decreased. In countersinks for universal use, it is accepted within the range of 10 - 25 degrees.

The front rake angle γ_N exerts an effect upon the process of cutting. With an increase in angle γ_N , the cutting force and torque are diminished. The angle is selected in relation to the work material, the diameter, and the general design of the countersink. In order to avoid weakening the bracing of the blades in sectional countersinks, it is sometimes necessary to reduce angle γ_N (in the same manner as angle ω) as compared to its optimum value.

Below some recommended values of γ_N to be used in countersinking in various work materials are cited:

Work material	Angle γ_N°
Aluminum, brass	25 - 30
Soft steel	15 - 20
Medium-hard steel, steel casting . .	8 - 12
Medium-hard cast iron	6 - 8
Hard steel, hard cast iron	5 - 0

The part of the countersink, as well as of the drill, which is subject to maximum stress is the corner where the tapered surface ends and the cylindrical surface begins, since it is here that the cutting edge is at its weakest and the cutting speed at its highest, resulting in the concentration of heat. To reinforce the cutting edge, it is recommended that the blade be given a positive angle of inclination λ. In the presence of a positive angle λ, the center of cutting pressure is away from the cutting edge, which relieves the latter from excessive stress. The positive value of angle λ is also conducive to the better ejection of the chip. In actual practice, however, countersinks are made with angle $\lambda = 0$, particularly when they are fabricated from high-speed steel.

The value of angle λ is very important in the case of countersinks with hard-alloy blades, where the cutting edge must be reinforced on account of the brittleness of the blades. For such countersinks, the value of angle λ is to be selected within the range 12 - 20 degrees, in relation to the material being machined.

Each lip of the countersink, similarly to a drill, is sharpened along its conical surface in a universal sharpening machine equipped with a special fixture or in a drill-sharpening machine. The back relief angle α on the cutting part is accepted within the range of 8 - 10 degrees, and on the calibrating part, within the range of 5 - 6 degrees (Figure 39).

[Drawing]

Figure 39. Back relief angles on the cutting part (a) and the calibrating part (b).

The sharpening to angle α on the cutting and calibrating parts is to affect only a small area, 0.6 - 1.5 millimeters wide, in relation to the size of the countersink. The remaining part of the lip is sharpened to an angle α_1 = 20 - 30 degrees. To control the correct degree of sharpening (the absence of free play), a non-sharpened margin, 0.05 - 0.06 millimeter wide, is left on the cutting part.

The calibrating part has a margin f (on the cylindrical surface) 0.8 - 2.0 millimeters wide, in the case of 10 - 50 millimeter countersinks. The purpose of this margin is the same as in the case of a drill. With an increase in the width of this margin, the efficiency of the cutting process is reduced.

The countersink has a back taper equal to 0.06 - 0.10 millimeters per 100 millimeters of length.

Countersinks of sectional design. In order to conserve high-speed steel and improve the utilization of available material, countersinks of sectional design are used. The body of these countersinks is made from carbon or alloy steel; the blades are made either entirely from high-speed steel or from alloy steel onto which tips of high-speed steel (hard alloy) are brazed. The blades themselves may be welded or brazed onto the body of the countersink forming a solid connection with it. In this case, the possibility of adjusting the countersink diameter after the tool is worn is eliminated.

The design of non-adjustable hard-alloy-tipped countersinks submitted by TsNIITMASh (see Figure 40) showed a high degree of efficiency, particularly in the machining of cast iron. It was accepted as standard (GOST 3231 - 46).

[Drawings]

Section NN

Figure 40. Countersink with hard-alloy teeth.

They are made with taper shanks, or are of insert-blade construction, with straight or helical flutes. To provide for adequate rigidity, the blades are brazed onto the body. The inclination of the blade in relation to the body flute and the positive angle of incline of the cutting edge (λ), are characteristic for this design.

In order to determine the location of the blade in the groove, it is also necessary to know distance a from the blade to the countersink axis (see Figure 41). The value of a can be determined from formula

$$a = b - l \cdot \tan \omega = R \cdot \sin \gamma_1 - l \tan \omega .$$

The blade is inclined to the countersink axis at angle γ_1 in a section normal to the countersink axis, and at angle γ_2 in section by a plane normal to the radius referring to the given point (for the countersink point $\gamma_2 = \omega$) and parallel to the countersink axis. Angles

γ_1 and γ_2 are determined by the above formulas in relation to the requisite geometric parameters of the cutting edge.

[Drawing]

Figure 41. Displacement of blade about the countersink axis.

The inclination angle ω_1 of the countersink body groove, which is to receive the blade, is determined by turning the blade about the countersink axis by the value of angle γ_1, as per formula

$$\tan \omega_1 = \tan \gamma_2 \cdot \cos \gamma_1 .$$

When the required inclination angle of the blade is $\gamma_2 = 20$ degrees, the inclination angle of the body groove $\omega_1 = 12 \div 15$ degrees, in relation to the requisite geometry of the cutting edge.

Such a selection of the inclination angles of the blade and of the groove which is to receive the blade allows for the sharpening of the blade in a plane (front face) with the formation of a 2 - 4-millimeter-wide bevel. The bevel provides for the selection of the front rake angle γ within the range of 10 - 12 degrees. Sharpening in a plane facilitates the operation. The presence of a positive angle λ provides for smooth cutting and increases cutting edge durability in the place of maximum stress. This is of particular importance in the case of hard-alloy-tipped tools characterized by higher brittleness.

In resharpenings, the value of λ is reduced. In the case of a new countersink, the value of λ is to be so selected that, following all the permissible resharpenings, it will still remain greater than or equal to zero. Angle λ in the new countersink is also to be in relation to the working height of the blade, and, generally, it is to fall within the range of 12 - 16 degrees.

There are various designs of sectional countersinks. The most successful design is the one in which the bracing of the blades is accomplished by way of riffled construction (see Figure 42).

Section ABCD

[Drawing] [Drawing]

(a)

Section ABCD

[Drawing] [Drawing]

(b)

Section ABCD

[Drawing] [Drawing]

(c)

[Drawing] [Drawing]

(d)

Section ABCD

[Drawing] [Drawing]

(e)

Figure 42. Sectional countersinks.

Let us analyze these riffled constructions. The countersink represented in Figure 42, a,consists of a body (1) with grooves inclined about the tool axis and with wedge-shaped knives (2) with a 5 - degree radial inclination angle. The body grooves and the blades are riffled so as to permit the adjusting of the diameter after blunting by shifting over one or several riffle graduations. The wedge-like shape of the blades provides for their reliable bracing in the body of the tool. The defect of this design is that there is no axial blade adjustment.

Figure 42, b, shows a modified design. The blade (2) and the body groove (1) have transversely running riffles and a double inclination, 5 degrees in a radial direction and 1 degree 30 minutes in an axial direction. The countersink is provided with double control, radial and axial. The defect in this design is that the transverse direction of the riffles does not allow the use of the broaching method in the machining of the body grooves, requiring instead the laborious operation of slotting.

Figure 42, c, shows a design consisting of a body (1) with wedge-shaped grooves inclined about the tool axis, wedges (2), and blades (3). The lateral side of the groove and the adjacent side of the blade have longitudinal riffles, while the other side of the blade and side of the wedge adjacent to the latter have transverse riffles. This design provides for double blade adjustment, axial and radial. The defects in this design are the closed-in groove, which makes it impossible to use the method of broaching in the machining of the riffles, and the projection in the blade for the transverse riffle.

In the design shown in Figure 42, d, the blades (3) and wedges (2) are riffled longitudinally. Wedge (2), with a radial inclination of 5 degrees, braces blade (3) in the body (1) by the method of tight fit. The design provides for double blade adjustment (axial and radial). The defect in the design is the complexity of machining the grooves for the tight fit.

The countersink in Figure 24, e, shows an improved design as compared to the preceding one. Blades (3) and wedges (2) are longitudinally riffled. The grooves in the body (1) have a 5-degree radial inclination and a 3-degree longitudinal inclination. Double blade regulation is provided for.

Sectional countersinks are to satisfy the following specifications: (1) strength, reliability and rigidity in the bracing of blades in the body; (2) simplicity in fabrication and the provision for adjustability of blades following wear; (3) provision for a normal sharpening allowance along the diameter following adjustment; (4) the more or less permanent overhang of the blade with relation to the face of the tool in resharpenings (see dimension k in Figure 43).

[Drawing]

Figure 43. The limit values of countersink wear (amount of stock removed by cumulative sharpenings).

In the process of machining, the countersink blade wears at the corner, or the transition area, formed by the intersection of the back surface of the cutting part and the cylindrical surface of the margin (the cross-hatched triangle with length A in Figure 43). The countersink wear, when working in cast iron, runs approximately at the angle $\frac{\varphi}{2}$.

Based on this, it may be recommended that the sharpening be done to an additional angle $\varphi_1 = \frac{\varphi}{2}$ along a length of bevel determined by 0.8 ÷ 1.2 millimeters.

In order to restore the cutting capacity of the countersink, it is necessary to grind off the blade along the back surface by value h, the diameter of the countersink remaining the same. In sharpening, the extent of overhang of the blade (dimension k) with relation to the face, is reduced. This would lead to inadequacies in the work of the countersink. Therefore, the feature of the bracing of the blade should be so designed as to provide for the possibility of retaining this dimension more or less unchanged.

There are limit values to the amount of stock that can be removed along the axis M and along the height M_1 of sectional countersinks by resharpenings. Grinding off of stock beyond those limit values does not any longer insure the requisite strength and reliability in the bracing of the blades. The following sharpening stock values were established empirically:

$$M = L - (3k + m);$$

$$M_1 = a - 6s,$$

where L is the length of the knife; m is the length of the angular edge, a is the depth of the recess in the front face, and s is the pitch of the riffle.

The best indexes pertaining to the minimum allowance for grinding and the maximum permissible number of resharpenings are given in the layout shown in Figure 42, d, which has been authorized as standard by GOST 2253 - 43.

Sectional countersinks with a diameter from 40 to 75 millimeters are made with taper shanks. Inserted blade countersinks are made in diameters 40 to 100 millimeters. The number of teeth in countersinks with a diameter of up to 55 millimeters is standardized as 4, and in those with a diameter of above 55 millimeters, as 6.

Figure 44 shows the elements of standard bracing by the method of riffling (GOST 2568 - 44). The riffle angle is 90 degrees and the riffle pitch is 0.75 - 1.0 millimeter. The apexes are sheared off to form a little platform of 0.1 - 0.2 millimeter; the riffle notches form little platforms 0.05 - 0.1 millimeters wide. The riffle thickness equals the thickness of the notch along the center line of the profile.

The cutting elements of sectional countersinks are determined by taking into account the work material and the countersink blade material, and also the characteristics of the countersink design.

[Drawing]

Figure 44. Standard riffling for inserted blades.

Bracing the countersink. Countersinks of standard design, from 10 to 36 millimeters in diameter, are made with taper shanks, and countersinks with a diameter of 25 millimeters and above are made with holes for setting into arbors. The arbors are also made with taper shanks. Shanks with Morse tapers provide for the excellent centering of the tool, resulting in diminished vibration and improved quality of the work surface.

As a result of operating wear and the removal of stock by subsequent resharpenings, only a relatively small section of the cylindrical part of the countersink sustains a reduction in length. In order to increase the number of possible resharpenings, it is recommended, in the case of end countersinks, that a hole be drilled in the face of the tool, and, in the case of inserted blade countersinks, that the length of the front part of the tapered hole (up to the internal chamfer) be increased by 50 percent, as compared to the length specified for standard countersinks. Such a construction provides for the reliable bracing of the countersink in the arbor, regardless of the many resharpenings.

In countersinks of standard design, there is no complete utilization of the material constituting the cylindrical part. To eliminate this defect, sectional countersinks with removable heads are used. The removable head is set into a hardened carbon steel or alloy steel body, which has the form of a conventional countersink with tapered shank. The removable head, which is the cutting part of the countersink, is either solid high-speed steel or is equipped with brazed-on hard-alloy blades. Countersinks of such design are very effective in machining with control guides (for instance in aggregate machine tools) and also in the machining of deep holes. The long body of the countersink provides excellent guidance in the work.

The junction of the cutting head with the body is effected by several methods.

In the design shown in Figure 45, a, the head is equipped with a square extension, for which a square hole is provided in the body of the countersink, with a through groove provided in the body to facilitate the ejection of the head. The head is stayed with the aid of a bolt passing through its entire length (Figure 45, b).

[Drawings]

Section AB

(a) (b)

Figure 45. Countersink shaft with square hole for head.

regardless of the many resharpenings.

In countersinks of standard design, there is no complete utilization of the material constituting the cylindrical part. To eliminate this defect, sectional countersinks with removable heads are used. The removable head is set into a hardened carbon steel or alloy steel body, which has the form of a conventional countersink with tapered shank. The removable head, which is the cutting part of the countersink, is either solid high-speed steel or is equipped with brazed-on hard-alloy blades. Countersinks of such design are very effective in machining with control guides (for instance in aggregate machine tools) and also in the machining of deep holes. The long body of the countersink provides excellent guidance in the work.

The junction of the cutting head with the body is effected by several methods.

In the design shown in Figure 45, a, the head is equipped with a square extension, for which a square hole is provided in the body of the countersink, with a through groove provided in the body to facilitate the ejection of the head. The head is stayed with the aid of a bolt passing through its entire length (Figure 45, b).

[Drawings]

Section AB

(a) (b)

Figure 45. Countersink shaft with square hole for head.

Section AB Section CD

[Drawings]

Figure 46. Countersink shaft with hexagonal hole for head.

Section AB

[Drawings]

Figure 47. Countersink shaft with dowel pin.

Section AB

[Drawings]

(a)

Section ABCD

[Drawings]

(b)

Figure 48. Countersink shaft with sheared tang (a) and with rectangular groove and securing screw (b).

In the design depicted in Figure 46, a hexagon is used in place of the square, which simplifies the design. The stay bolt is screwed into a hexagonal nut, which, in turn, is inserted into the hexagonal hole in the countersink body. The nut is prevented from falling out by a spring in the form of an unlocked ring inserted into the corresponding internal chamfer of the hole. To facilitate ejection, the hole is made to run through.

The bracing shown in Figure 47 consists of a dowel pin, inserted into the holes drilled in the countersink body and the countersink head. To facilitate the ejection of the head, a spring, braced against the internal face of the head taper, is provided.

Bracing, as depicted in Figure 48, is effected by two alternate methods: (1) by the tang of the shank inserted into the corresponding recess (Figure 48, a); (2) with the aid of the tang of the shank, the rectangular groove at the greater diameter of taper, and the securing screw (Figure 48, b).

Sectional countersinks are also used for the machining of shallow holes. One such design is depicted in Figure 49. The body is Morse-tapered, while the countersink head has a straight shank with two projecting cams in the center for bayonet bracing. The head, compressing the spring, is inserted into the body and turned in a direction opposite to the tool rotation. The cams fall into the corresponding internal chamfers of the body and effect the transmission of torque. This countersink design is rather complex in fabrication.

[Drawings]

Figure 49. Shaft with bayonet bracing.

Position I

[Drawings]

Position II

Section CD

[Drawings]

Figure 50. Bracing as per GOST 3009 - 45.

[Drawing] [Drawing]

(a) (b)

Figure 51. Design of locators for quick-change chuck.

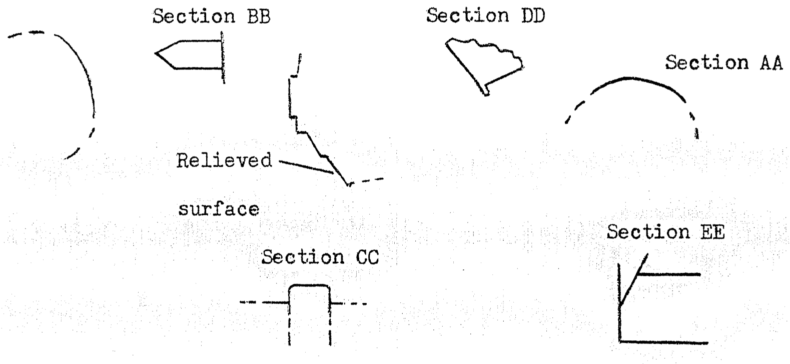

Figure 52. Double-thrust countersink with large surplus stock.

Figure 50 depicts the bracing of straight-shank countersinks in quick-change chucks. This design is authorized as an All-Union standard (GOST 3009 - 45). The shank has an annular groove (1), flat surface (2), indicated by dimension H, and radial groove (3) having a width (b). Dowel pin (4) is pressed into the body of the chuck, and a locator device, consisting of dowel (5) and annular spring (6), is provided. The countersink shaft is inserted into the chuck, in position I. The flat surface (2) passes freely under dowel (4). Dowel (5) under the action of spring (6) is flipped into the annular groove (1), fixating thereby the proper location of the tool. When the machine spindle with the chuck mounted on it rotates, dowel (4) enters groove (3), grips the shank, and effects the working motion of the countersink (position II). The locator device prevents the countersink from falling out and renders effective the rapid and convenient bracing of the shank in the chuck. At each turn of the machine spindle, the chuck dowel, entering the groove, by itself grips and guides the countersink. This arrangement is particularly important in the case of vertical spindle machines. In actual operations, both a dowel locator device (Figure 51, a) and a ball locator device (Figure 51, b) are used.

Shanks of the above described type (for 10 - 50 millimeter countersinks) successfully replace the Morse taper type of bracing. The advantages are manifested in the simplification and rapidity of mounting and dismounting the tool, and also in a saving in material, since the shank assembly of this type is 2 - 3 times shorter than a Morse taper shank. The defect of such bracing is that the straight shank, as compared to the taper shank, does not provide for better tool centering nor for a higher degree of surface finish. It can, therefore, be recommended for work of lesser precision.

The double-thrust countersink depicted in Figure 52, is braced in the arbor with the aid of two torque-transmitting dowels and a center hole. The countersink is used for the machining of through, blind, step, and profile holes, having a diameter of from 30 to 200 millimeters and a length up to 4000 millimeters. When machining in continuous material, it is frequently used in combination with a drill mounted on the arbor ahead of the countersink. The countersink is capable of removing considerable machining stock allowances, has greater productivity and durability, and permits a considerable number of resharpenings.

The back surface of the main cutting edge, which is relieved, non-ground, with chip breaking annular grooves, has a 60 or 75 degree angle in plan. The calibrating part is cylinder-ground, or ground to a small back relief angle ($30' \div 2^{\circ} 30'$). Sharpening is done only along the front surface. In contradistinction to the conventional countersink, a double-thrust countersink has a short bit without guide margins, which prevents the wedging of the chip and its adherence to the tool. The front rake angle is selected in relation to the work material within the range 10 - 25 degrees; the back relief angle is 5 - 8 degrees. For purposes of proper ejection of the chip, the cutting edge has an angle of inclination $\lambda = 10$ degrees.

Countersinks for Cylindrical Holes.

To machine holes to fit cylindrical heads (Figure 53, a) or screw necks (Figure 53, b), countersinks are used. These countersinks do not differ from each other, with the exception of the sizes of diameters and pivot journals. In small sizes, they are made with straight shanks, and in large sizes, with taper shanks. At times they are made in the inserted blade form. A special characteristic of this type of countersink is the presence of a pivot journal at the face of the bit. The pivot journal serves to guide the countersink in operation and to provide for co-axiality of the countersink hole to fit the screw head and the hole to fit the screw stem. The pivot journal is made either integral with the countersink body or is replaceable. The last type is preferable, since it permits more resharpenings, facilitates the process of sharpening, and permits the utilization of the countersink for a group of diameters by shifting from one size of journal to another.

[Drawing]	[Drawing]
(a)	(b)

Figure 53. Form of cylindrical holes.

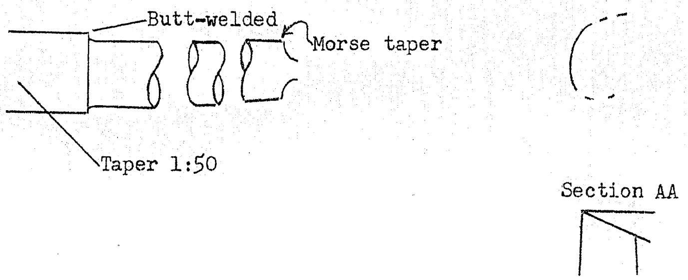

Figure 54. Bracing replaceable pivot journal to countersink.

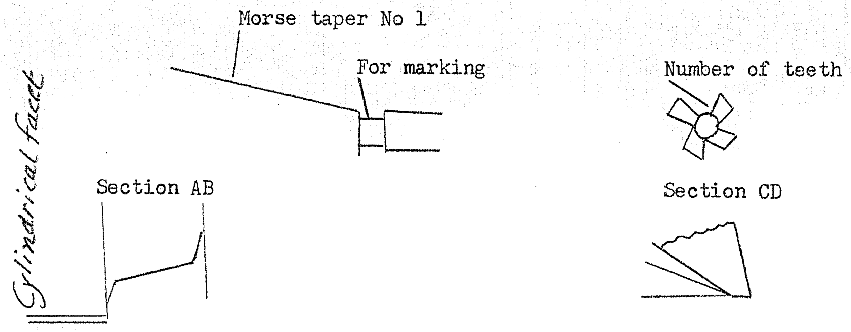

Figure 55. Countersink design for cylindrical holes.

Figure 54 depicts the bracing of the pivot journal to the countersink with the aid of a tapered section. There are other methods of bracing the journal to the countersink, as shown in Figure 45 in the preceding text. The shaping of the countersink bit, in this case, is depicted in Figure 55. The angle of inclination of the helical flute ω is 10 - 15 degrees; the undercut at the face is 8 - 10 degrees; the back relief angle at the calibrating part $\alpha = 8 \div 10$ degrees, with lip 1 - 1.5 millimeters wide, with margin of 0.2 millimeters. The lip has an additional taper at an angle

$\alpha_1 = 25 \div 30$ degrees. The angle of inclination of the cutting edge $\lambda = 10$ degrees.

Countersinks for Conical Holes (Countersink Reamers)

For the machining of center holes, the following types of countersink reamers are used: plain, single-tooth (Figure 56, a), conical twist (Figure 56, b), centering (Figure 56, c), centering with safety cone (Figure 56, d). In machining centering holes, the most frequently used angles 2φ are 60-degree angles, and less frequently 75 and 90 degrees. For the machining of conical holes, conical countersink reamers are also used (Figure 57). They are fabricated with an angle $2\varphi = 60$, 75, 90, and 120 degrees, and with a diameter from 12 to 60 millimeters. The number of teeth is selected, respectively, from 6 to 12. To facilitate the process of cutting, it is recommended to shear off, with skipping of one tooth, a little section of length $l = 1.5 \div 5$ millimeters. The web thickness at face is selected equal to 0.1D, and the diameter of sheared face $d = (0.15 \div 0.18)D$, where D is the diameter of the countersink reamer. The angle of the recess ϑ is in relation to the number of teeth and the angle of the tooth body η, which is determined by formula

$$\vartheta = \eta + \epsilon,$$

where $\epsilon = \frac{360^\circ}{z}$.

To avoid the weakening of the teeth, angle η is to be no less than 30 - 40 degrees. Upon computation, angle ϑ is rounded out to fall in line with the conventional series of angles of angular milling cutters. In order to maintain the width of lip p uniform along the entire length of the tooth, it is necessary to compute angle β of the dividing head setting, which is determined by formula

$$\beta = B - C,$$

where $B = \cos \epsilon \tan \varphi$;

$$C = \sin B^\circ \tan \epsilon \cot \vartheta ,$$

where 2φ is the apex angle of the countersink reamer cone; $\epsilon = \frac{360^\circ}{z}$; ϑ is the recess angle. The width of lip p is selected within the range 0.05 - 0.06 millimeters. The back relief angle $\alpha = 6 \div 8$ degrees. The front surface is directed along the radius. The countersink reamers are made either with taper shank or with shank as per Figure 50.

Back relief angle of cutting facet 6°

[Drawing]

Section AB

[Drawing]

(a)

[Drawing]

(b)

Section CD

[Drawing]

Back surface relieved at slant (10° angle)

[Drawing]

Section AB

[Drawing]

(c)

Section CD

[Drawing]

[Drawing]

[Drawing]

Back surface relieved
at slant (10° angle)

Section AB

[Drawing]

(d)

Figure 56. [See also preceding page] Types of countersink reamers.

Butt-weld

[Drawing]

[Drawing]

Section AB

[Drawing]

Figure 57. Conical countersink reamer.

Face-Trimming Countersinks

The peculiar characteristic of these countersinks is the presence of teeth on the face only. Countersinks of this type are rarely made with helical teeth on the stem. The design is inserted-tooth, with shank of the type shown in Figure 50, or to fit into quick-change chucks. In the latter case, the teeth are projected from a double face. The teeth projecting from the second face are used after the first set of teeth is blunted. The teeth, particularly for machining in cast iron, are hard-alloy. To maintain coaxiality of the hole and the work surface, the countersinks operate together with the pivot journals in the same manner as screwhead countersinks. The pivot journals are either detachable, or integral with arbor. Bracing with arbor is effected by a screw. Popular diameters for countersinks of this type are within the range 14 - 40 millimeters. Due to the heavy-duty character of the work, the number of teeth is not to exceed 2, 4, or 6. Figure 58, a, shows the cutting parts of this countersink.

[Drawings]

Section EF [Drawing] Section AB [Drawing] Section CD [Drawing]

(a)

[Drawings]

Section AB

[Drawing]

(b)

Figure 58. Countersink for trimming of face surfaces to fit into quick-change chuck (with cam bracing).

[Drawings]

Figure 59. Face-trimming countersink with square bracing.

In order to facilitate machining, countersinks for the trimming of large surfaces are made with chip breakers arranged in chess-board fashion. Special notice is to be taken of the countersink design shown in Figure 58, b, suitable for the trimming of surfaces inaccessible or inconvenient to reach from above. The countersink has a hole with parallel sides, and the arbor end has an irregularly shaped cross section. Such a design provides for the perpendicularity of the cutting teeth faces about the tool axis even when there is some free play between the countersink and the arbor. Bracing of countersink to arbor with the aid of a screw, dowel, roller or ball, when these are present on the face which is opposite to the location of the cutting edges, is not as effective as the above described design.

Figure 59 shows another design for bracing the countersink to the arbor, with the aid of a square tip at the end of the arbor to fit into a square hole in the countersink shank, and a thrust tapered washer abutting the countersink. The washer has a groove in it in order to facilitate connecting with or disconnecting from arbor.

Combination Countersinks and Irregularly Shaped Countersinks.

In order to combine several operations (transitions) into one, combination tools are used. A combination tool has many advantages: (1) it allows the utilization of a standard machine tool for complex machining; (2) it cuts down machining time, thereby reducing production costs; (3) it reduces checking time, since the precision of the work is insured by the proper tool sharpening.

There are two alternate designs of a combination tool. It is either made up of tools of the same type differing only in size, or of tools of different types.

Outstanding examples of the first group are step-countersinks for holes with two, three, or more diameters.

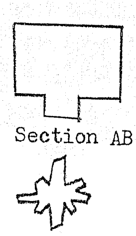

Figure 60. Two-step countersink.

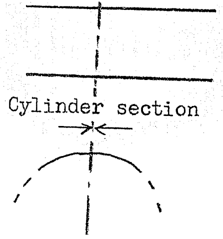

Figure 61. Countersink with vari-type teeth for considerable machining stock removal.

[Drawings]

Figure 62. Combination countersinks.

A convenient design of a two-step-countersink is shown in Figure 60. Its characteristic feature is the alternating disposition of boring and face-trimming teeth. In contradistinction to the non-alternating-tooth design, the alternating-tooth design provides for a considerable number of resharpenings. Countersinks with alternating teeth are also used successfully/ for boring holes, when there is considerable stock to be removed. The stock to be removed is distributed between two or three groups of teeth, which are disposed on peripheral circles of various diameters (see Figure 61). Each subsequent tooth overlaps the preceding one. In designing the above, circumferences of the requisite diameters are drawn. By the requisite thicknesses and height of the teeth, their central angles are determined within such a range as to require the minimum number of milling cutters for milling the countersink flutes. After the flutes have been milled, the teeth are sheared off, heightwise and lengthwise, to conform with the amount of surplus stock to be removed by each group of teeth, respectively. Each group is usually made up of 3 - 4, though sometimes of only two, teeth. To improve cutting, countersinks are usually supplied with helical teeth having an angle of inclination ω = 15 - 20 desgrees for the largest circumference. It should also be remembered that angle ω assumes lower values for the other circumferences.

Outstanding examples of the second group are combination tools consisting of a drill and countersink, or reamer and countersink, or boring cutters, countersink and reamer, or other combinations.

Figure 62 shows combination tools for the machining of a series of surfaces and flat surfaces, which are indicated by corresponding markings on the tools and on the work blanks. Letter O indicates a roughing pass; letter Ch, a finishing pass.

BIBLIOGRAPHY AND SOURCES

1. Galey, M. T., The Geometric Parameters of Sectional Countersinks, Thesis, Moscow Machine Tool Institute, 1941.

2. Lapin, N.A., "Hard-Alloy-Tipped Countersink", Stanki i instrumenty (Machine Tools and Tools), No 11, 1936.

3. Leading Data on Countersink Operation, ENIMS, 1945.

REAMERS

Designation and Types of Reamers

Reamers are designed for the machining of precision holes. They are used for finish- and rough-operations. In relation to specifications, reamers produce holes within a wide range of tolerances, from the fifth to the first class of precision.

The correct work performed by a reamer depends on the design and quality of its fabrication, as well as on operating conditions (cutting practice, cooling, the value of the stock to be removed, the quality of the sharpening and lapping of the cutting edges, etc.)

Reamers are divided:

(1) by the nature of their application -- hand-operated reamers (OST 2512 - 39), machine-operated reamers (GOST V 1672 - 1673 - 42), boiler reamers;

(2) by the shape of the hole to be machined -- cylindrical and taper (OST 2513 - 2516 - 39);

(3) by the manner of bracing -- solid with shank and insert reamers (OST NKTP 3676);

(4) by the tooth design -- reamers with inserted teeth (GOST 883 - 46, 884 - 46, and 1523 - 42), and reamers with teeth integral with body;

(5) by the adjustability of dimensions -- adjustable reamers and non-adjustable reamers.

The basic concepts, designations and terminology pertaining to reamers are established in accordance with OST NKTP 2937.

Cylindrical Reamers

Reamers with milled teeth. This reamer is a cylindrical body with flutes for the formation of cutting edges (Figure 63).

[Drawings]

Figure 63. Parts and structural elements of reamer.

The reamer consists of the following parts and structural elements: l -- the working part; l_1 -- the cutting part; l_2 -- the calibrating part; l_3 -- the cylindrical part; l_4 -- the back taper; l_5 -- the shank; l_6 -- the neck; e -- the square; z -- the tooth; k -- the flute; s -- the front surface; t -- the back surface; γ - the front rake angle; α -- the back relief angle.

The basic elements of reamer design are the cutting and the calibrating parts, the number of teeth, the direction of the teeth, the sharpening angles of the teeth, the pitch of the teeth grooves, the flute profile, and the holding part.

The cutting (tapered) part l_1 serves for the maximum removal of stock. The cone angle φ (in degrees), which affects the durability of the reamer and the degree of finish of the work surface, is accepted, in the case of hand-operated reamers, as 0.5 - 1.5 degrees. In the case of machine-operated reamers, it is determined with relation to the work material: for brittle and hard materials, such as cast iron, it is 3 - 5 degrees; for ductile materials, such as steel, it is 12 - 15 degrees. In the case of boiler reamers, it is 1.5 - 3 degrees.

Reamers with angle $\varphi = 45$ degrees on the cutting part are very popular. Such reamers demonstrate good cutting ability and a high degree of surface finish.

To insure free entry into the hole, the smaller diameter of the cutting part is made smaller than the diameter of the reamer by 1.3 - 1.4 of the stock allowance for reaming. In addition, a 45 - degree bevel is removed at the end of the cutting part to prevent the teeth from chipping, in the presence of a heavier stock to be removed or any defects in the hole. The transition from the cutting part to the calibrating part is chamfered.

For the machining of light alloys, a special sharpening of the cutting part is recommended, as is done in the case of fluteless taps. The lip on the cutting part is sheared at an angle of 30 degrees to the axis, and the flute is correspondingly deepened at an angle of 15 degrees. Such a shaping of the cutting part provides for a positive front rake angle up to 8 degrees. To eliminate the possibility of weakening the tool by such additional sharpening, the width of the lip is increased to compensate for the reduction in the number of teeth.

The calibrating part l_2 guides the reamer in the work, imparts precision and high finish to the hole, and insures the presence of resharpening stock. An increase in length l_2 results in harder work for the reamer and in its jamming in the hole. It is, therefore, recommended that in the case of short reamers the length $l_2 = 0.25 - 0.3$ of the reamer diameter. A small value of l_2 saves steel and reduces the buckling of the reamer in hardening.

The back taper l_4 is made for the purpose of reducing the friction between the reamer and the surface of the hole and of forestalling the splitting of the hole by the calibrating part segment adjoining the neck. The lesser diameter of the taper (at neck) is smaller than its greater diameter by a value of 0.005 millimeters to zero, for hand reamers, by 0.04 - 0.06 millimeters, for machine-operated reamers, and by 0.06 - 0.10 millimeters, for oscillating reamers.

Due to the small back taper in hand reamers, they frequently have no cylindrical segment. The reaming of particularly clean holes of small length (up to 20 millimeters) can be done with reamers without a back taper.

The number of teeth is usually even to compensate for the error in measurement of the reamer diameter with the micrometer, in relation to diameter D in millimeters and the reamer designation as per formula $z = 1.5\sqrt{D} + (2 \div 4)$. For reamers with a greater number of teeth, a greater value of z is selected, since, with the increase in the number of teeth, the degree of finish becomes higher.

In the case of boiler reamers, the number of teeth is selected within the range from 3 to 8, in relation to the diameter.

Reamers are equipped with straight or helical teeth. Helical teeth provide for a better finish and higher durability of tool. Straight-tooth reamers, when properly designed, result in holes fully satisfactory in precision and surface finish.

The fabrication, sharpening, and control of straight-tooth reamers is considerably simpler than for helical-tooth reamers. The machining of holes with longitudinal grooves of or lengthwise-interrupted holes is to be done with helical-tooth reamers. The hand of direction of the helical teeth is to be opposite to the hand of direction of the rotation of the reamer, in order to eliminate the possibility of the self-tightening and jamming of the reamer in the hole, and also to forestall the possibility of the reamer shank becoming detached from the machine spindle. The angle of flute inclination is selected in relation to the flute material: for gray iron and hard steel, it is 7 - 8 degrees; for malleable iron and steel -- 12 - 20 degrees; for aluminum and light alloys -- 35 - 45 degrees; for boiler work -- 25 - 30 degrees.

The peripheral non-uniform distribution of the teeth in the hole being reamed forestalls the appearance of longitudinal lines, which would be disposed in conformity with the pitch of the teeth. The cause for the appearance of a riffled surface is the periodical variation of the tooth load, accountable to the non-uniformity in the work material, to hard or soft inclusions, and the like.

The non-uniformity in pitch may be attained by various methods (see Figure 64), method b being the one in widest use, since it provides for greater simplicity in the fabrication of the reamer and convenience in the micrometer-gaging of its diameter.

Data pertaining to the non-uniformity in the pitch of the teeth is cited in Table 7.

[Drawing] [Drawing] [Drawing]

(a) (b) (c)

<u>Figure 64</u>. Methods of forming a non-uniform pitch in the reamer teeth: a -- pitch varied in all teeth; b -- pitch varied to both sides of the control teeth lying on one diameter; c -- same values for pitch for each two opposite teeth.

TABLE 7

Non-uniformity in the pitch of teeth

Number of teeth	Angle of turn					
6	58°02'	59°53'	62°05'	----	----	----
8	42°	44°	46°	48°	----	----
10	33°	34°30'	36°	37°30'	39°	----
12	27°30'	28°30'	29°30'	30°30'	31°30'	32°30'

<u>Note</u>: The dividing head disk is to have 49 holes.

Non-uniformity in the pitch of the teeth can also be attained by slanting flutes with a change in direction for each two adjacent teeth.

<u>The angles for the cutting part</u> are selected in relation to the designation of the reamer and to the work material.

Back relief angle α for the cutting part is selected within the range of 4 - 8 degrees. For finishing reamers, α is to be smaller than for roughing reamers. The sharpening of the tooth along its cutting part is done up to a fine point, while, on the calibrating part, a small margin is left (Figure 65). The margin provides for the guiding of the reamer in the hole, promotes a smoother surface finish, provides for proper calibration of the hole, and facilitates the control of the reamer along its diameter. The margin width is selected as 0.05 - 0.3 millimeter, in relation to reamer size. In the machining of ductile materials, when it becomes necessary to prevent the adhesion of the chip, the margin width is reduced to 0.05 - 0.08 millimeter. The grinding of the margin at an angle of 30' - 1°30' is also recommended. In the case of hand reamers, and also machine-operated reamers with chromated edges, the margin width is to be kept within 0.15 - 0.18 millimeters. The margin width in machine-operated reamers may be increased to 0.3 - 0.4 millimeters, when machining holes of special precision in steel and cast iron, in which case the reaming is done by mechanical feed and at low cutting speed.

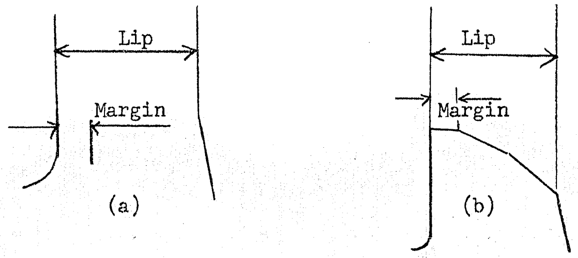

Figure 65. a -- tooth with single back relief angle; b --

tooth with double back relief angle.

Adjustable cylindrical reamers, and also taper and boiler reamers, are to be sharpened to two angles: α = 6 ÷ 8 degrees and α_1 = 15 ÷ 20 degrees (Figure 65, b).

Front rake angle γ of the reamer is accepted as equal to zero; the front surface is directed along the radius. In the case of more rigid specifications pertaining to the surface finish, it is recommended that angle γ be given a negative value of minus 5 degrees. In order to avoid the adhesion of chip to the cutting edge when reaming in ductile material, angle γ is to have a positive value within the range of 5 - 10 degrees.

In the case of boiler reamers, which are not only to remove the predetermined machining stock but also the layer of metal formed by the boiler plate displacement, angle γ is to be positive within the range of 12 - 15 degrees.

Angle γ is measured in a plane normal to the direction of the flute. The ratio between angle γ in a normal section and angle γ_T in the face section (for a point located at the periphery) is expressed by formula

$$\tan \gamma_N = \tan \gamma_T \cos \omega ,$$

where ω is the angle of flute inclination.

The flute profile in reamers is not as important as it is in drills or taps.

Dimensions and types of flute profiles are enumerated in Table 8.

TABLE 8

Dimensions of reamer flute profiles

Diameter of reamer in mm	Number of teeth	Width of lip p in mm	f (margin) in mm	in degrees	Type
3 - 3.5	6	0.25	0.12	85	B
4 - 4.5	6	0.3	0.12	85	B
5	6	0.4	0.12	85	B
6 - 8	6	0.5	0.15	85	B
9	6	0.6	0.15	90	B
10	6	0.7	0.15	90	B
11	8	0.7	0.18	75	B
12	8	0.7	0.18	75	B
13 - 14	8	0.8	0.18	75	B
15 - 16	8	0.9	0.18	80	B
17 - 20	8	1.0	0.18	80	B
21 - 23	10	1.1	0.18	--	C
24 - 27	10	1.2	0.18	--	C
28 - 30	10	1.3	0.18	--	C

Figure 66 shows various types of flute profiles.

Type A [Drawing]

Type B [Drawing]

Type C [Drawing]

Figure 66. The profile of the reamer flutes.

[Drawings]

Figure 67. The milling of the flutes with the aid of a double angle milling cutter.

[Drawings]

Figure 68. Special profile milling cutter for the milling of reamers.

A single-angle milling cutter (for type A flute) (see Figure 66), makes a poor front surface, and the face-milling teeth wear rapidly. Due to the undercut in the tooth, this type of milling cutter is not suitable for milling helical-teeth reamers. A double-angle milling cutter (for type B flute) is free of the above defects. For medium and large sizes, the use of a profile with the outline of the back of the tooth along the radius (for type B flute) is recommended. Such a profile provides for an adequate space for the chip and the requisite strength of the tooth.

In order to avoid the appearance of cracks in hardening, the hollow of the flute is to be rounded to a radius $r = 0.3 - 0.8$ millimeters.

When milling a flute for a reamer of non-uniform pitch (Figure 67), in order to maintain the same value of p, it becomes necessary to change the depth of the flute and the distance between the axes of the reamer and the milling cutter for each new flute.

These values are computed by formulas:

$$t = R_3\left(1 - \frac{\sin(\Theta + \vartheta - \epsilon)}{\sin \vartheta} \cos \varphi\right) - $$

$$- r\left[\frac{\cos\left(\frac{\vartheta}{2} - \varphi\right)}{\sin \frac{\vartheta}{2}} - 1\right];$$

$$b = \frac{R_3 \sin(\Theta + \vartheta - \epsilon) \sin \varphi}{\sin \vartheta} - r \frac{\sin\left(\frac{\vartheta}{2} - \varphi\right)}{\sin \frac{\vartheta}{2}}.$$

where R_3 is the radius of the blank (with a stock allowance of 0.15 - 0.40 millimeter for grinding).

To simplify the operation of milling the flutes, special shape milling cutters are used, which machine not the flute, but the tooth of the reamer (see Figure 68), with the width of lip p remaining the same, without a change in the depth of milling. The defect of the special shape milling cutters consists in the fact that the back of the tooth receives a small shoulder a, 0.1 - 0.2 millimeter high (Figure 69). This method of milling with special shape milling cutters is applicable to straight-tooth reamers only

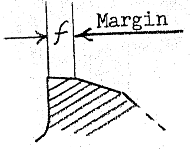

Figure 69. Formation of shoulder on tooth in the milling of reamers.

[Drawing]

Figure 70. Duplex special shape milling cutter for the milling of reamers.

[Drawing]

Figure 71. Outline of special shape milling cutter for the milling of reamers.

The special shape milling cutter may be replaced by two milling cutters assembled into one duplex unit (Figure 70), with the width of lip p adjustable by an intermediate ring.

The profile dimensions for special shape milling cutters are given in Table 9, and the designations of the profile components are shown in Figure 71.

TABLE 9

Special shape milling cutter profiles (see Figure 71)

(dimensions in millimeters)

Diameter of reamer	Number of teeth	R	R_1	R_2	R_3	m	n	t	f	p	a	b	φ	λ
													in degrees	
(1)	(2)	(3)	(4)	(5)	(6)	(7)	(8)	(9)	(10)	(11)	(12)	(13)	(14)	(15)
11	8	5.5	0.35	19	0.25	3.25	1.85	1.65	2.80	0.5	4.5	4.5	51	6
12	8	5	0.5	19	0.25	3.25	1.95	1.65	2.95	0.6	4.5	4.75	50	10
13	8	6	0.5	20	0.5	3.33	1.95	1.85	3.1	0.7	5	5.2	50	7
14	8	6	0.5	20	0.5	4.62	2.25	1.85	3.4	0.7	5.5	5.7	50	8
15	8	6	0.5	20	0.5	3.45	1.97	1.97	3.5	0.75	5.5	5.6	51	8
16	8	7	0.5	20	0.25	4.8	2.8	2.2	3.95	0.85	6.5	6.7	50	8
17	8	7	0.5	20	0.25	4.8	2.8	2.2	3.95	0.85	6.5	6.7	50	8
18	8	7.5	0.5	20	0.5	3.55	2.25	2.35	4.6	0.85	6.5	6.7	50	12

TABLE 9 (continued)

(1)	(2)	(3)	(4)	(5)	(6)	(7)	(8)	(9)	(10)	(11)	(12)	(13)	(14)	(15)
19	8	10	0.5	20	0.8	2.85	2.09	2.4	4.47	0.85	7.0	7.0	51	9
20	10	6	0.5	20	0.25	4.4	3.65	2.25	3.80	0.85	5.7	6.75	40	8
21	10	6	0.5	20	0.75	4.9	3.65	2.25	3.80	0.85	5.7	6.75	40	8
22	10	8	0.8	23	0.8	7.0	3.7	2.55	4.35	0.9	7.3	7.7	42	12
23	10	8	0.8	23	0.8	7.0	3.7	2.55	4.35	0.9	7.3	7.7	42	12
24	10	9	0.8	20	0.5	7.1	3.6	3.05	4.95	1.0	7.65	7.55	42	13
25	10	9	0.8	20	0.5	7.1	3.6	3.05	4.95	1.0	7.65	7.55	42	13
26	10	8	0.8	23	0.8	6.7	3.35	2.8	4.77	1.15	7.8	8.2	42	14
27	10	8	0.8	20	0.8	6.7	3.35	2.8	4.77	1.15	7.8	8.2	42	14
28	10	9	0.8	20	0.5	8.3	4.45	2.9	5.25	1.1	8.8	9.15	40	13
30	10	9	0.8	20	0.5	8.3	4.45	2.9	5.25	1.1	8.8	9.15	40	13

Note: Below line AB in the body of the milling cutter, chamfering can be done to an arbitrary radius.

The clamping part of hand reamers consists of a short neck l_6, straight shank l_5, and square e. The neck serves to facilitate the grinding of the cutting part and shank, the square for bracing in the respective hollow, the shank for guiding the reamer in long holes. The shank diameter is by 0.03 - 0.08 millimeter smaller than the reamer diameter.

The clamping part of machine-operated reamers is made: (1) cylindrical, for reamers up to 10 - 12-millimeter size; (2) with Morse taper; (3) with square (not in wide use). Machine-operated reamers have a long neck to facilitate the reaming of deep holes.

Detachable reamers are made with tapered holes, the taper being 1:30.

Specifications for these reamers are authorized by OST 2811 - 40.

A diagram of the disposition of diameter tolerances for reamer, designated for the machining of work with departures by a system of holes, is shown in Figure 72. The values for the tolerance components are given in Table 10.

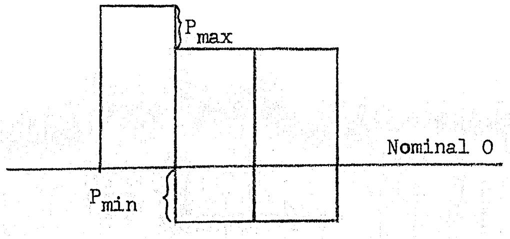

Figure 72. Tolerance diagram for reamer diameter: N -- tolerance

for non-precision of fabrication; P_{max} and P_{min} -- marking-off values of the hole being reamed; I -- guaranteed reserve of wearing stock along diameter of reamer under operating conditions; $\triangle$ -- tolerance for hole; AB -- upper departure of reamer diameter; CD -- lower departure of reamer diameter.

Reamers of Sectional Design.

Reamers of sectional design are divided into release reamers and inserted tooth reamers. Release reamers are used in assembling work. Their teeth are made integral with body, but they spread along the diameter, due to the combined action of axially cut out splines and an adjusting taper screw or ball in a specially bored hole.

Inserted tooth reamers come in various designs, the most rational being the design where the teeth are inserted into the body with the aid of riffles. Bracing and adjusting to size is effected in the same manner as in the case of countersink teeth (see original page 338).

Sectional reamers are made with a small number of teeth:

Diameter of reamer (in millimeters)	Number of teeth
16 - 42	6
44 - 65	8
65 -125	10
130 -150	12

These reamers are also made with non-uniform pitch, in accordance with data presented in Table 7.

Tool angles and cutting and calibrating parts are the same as in the case of solid reamers.

It is recommended that concavities be made for each tooth in the body of the reamer, along the cutting edge, to facilitate the ejection of the chip.

Specifications for inserted tooth reamers are established by GOST 1523 - 42.

TABLE 10

Tolerance components for reamer diameter

Tolerance components		Nominal diameter of reamer in mm							
		1 – 3	3 – 6	6 – 10	10 – 18	18 – 30	30 – 50	50 – 80	80 – 120
		Tolerance values in microns							
(1)	(2)	(3)	(4)	(5)	(6)	(7)	(8)	(9)	(10)
Maximum value of layout P_{max}	First class of precision	5	7	9	10	12	14	16	18
	Second class of precision	7	9	11	12	14	16	18	20

(1)	(2)	(3)	(4)	(5)	(6)	(7)	(8)	(9)	(10)
Precision tolerances in fabrication N	First class of precision	5	8	10	10	10	15	15	20
	Second class of precision	10	12	15	15	15	20	20	25
Minimum value of layout P_{min}		Accepted, for all diameters, as equal to 5							

Taper Reamers

Taper reamers are used for the reaming of a cylindrical hole to a taper, or for the calibration of an already tapered hole, preliminarily machined by another tool.

Taper reamers are made in the following types: (1) for Morse taper holes; (2) for tap holes (taper 1:7 or 1:10); (3) for dowel pin holes (taper 1:50); (4) tool reamers for detachable tools, reamers, countersinks, etc. (taper 1:30).

Taper reamers for Morse taper holes (Figure 73) are fabricated in sets of three or two. The first set (Figure 73,a) is in the form of a countersink, threaded at an angle of inclination equal to the inclination angle of the Morse taper. The lead of the thread coincides with the direction of cut. The reamer converts a cylindrical hole into a step-hole. The teeth are relieved at their back surfaces, their number ranging from 3 to 8, in relation to the taper ratio. Each projecting part performs with a small angle, similarly to a boring cutter with face-cutting edge.

[Drawings]

(a)

[Drawings]

(b)

[Drawings]

(c)

Figure 73. Taper reamers for Morse taper holes: s -- thread pitch; p -- lip; f -- margin; t_{min} -- height of tooth; k -- value for tooth back surface relief; δ -- flute angle; b -- width of thread flute; a -- depth of thread flute.

The second reamer (Figure 73, b) has rectangular left-hand thread (for right-hand cutting). The thread is for the purpose of breaking up the chip and for effecting smaller steps in the hole. The number of threads per inch in 8 - 4, in relation to the taper ratio; $b = \frac{2}{5} s$; $a = \frac{1}{2} b$; the teeth are sharp-pointed, with small 1 - 2- millimeter margin.

The third reamer (Figure 73, c) is little different, in its design, from a cylindrical reamer. The pitch of the teeth is uniform. The margin width is 0.05 - 0.08 millimeter.

Taper reamers for tap holes and tools come with one reamer in each set. The elements of design are determined in the same manner as in the case of conventional cylindrical reamers.

Reamers for dowel pin holes serve for the reaming of already existing cylindrical holes in various parts of machines connected by dowels. Due to their small taper ratio (1:50), they remove an insignificant layer of metal, and, therefore, convert a cylindrical hole into a tapered one without preliminary reaming. The front end diameter is so calculated that the reamer projects 1.5 - 4 millimeters from the hole. To increase the number of possible sharpenings, the cutting part is made longer than the standard dowel length. The number of teeth is within the range of 4 to 6. Reamers up to the 5 - 8 millimeter size are equipped with reversible centers. Reamers up to the 3 millimeter size are of trihedral or pentahedral shape, in section, the ribs serving as the cutting edges.

Screw dowel reamers, the design of which is depicted in Figure 74, and the sizes enumerated in Table 11, permit high cutting speeds and are characterized by great durability.

[Drawing]

Figure 74. Screw-dowel machine-operated reamer.

TABLE 11

Screw-dowel machine-operated reamers

(dimensions in millimeters)

Diameter of reamer	Number of teeth	s	f	t	ϑ	B	ω
2	1	4.5	0.5	0.2	80°	4.0	28°
3	1	6	0.6	0.2	80°	5.4	25°40'
4	1	8	0.8	0.2	80°	7.2	26°
5	1	10	1.0	0.3	80°	9.0	26°30'
8	1	16	1.6	0.4	80°	14.4	26°30'
10	2	20	1.0	0.6	80°	9.0	26°30'
13	2	25	1.3	0.8	80°	11.2	26°
16	2	30	1.5	1.0	80°	13.5	26°

Note: B -- width; ϑ -- angle of taper of the milling cutter for milling of the reamer flutes.

The direction of the helical teeth is opposite to the direction of cut, eliminating thereby the possibility of self-tightening of the reamer. The flutes provide ample space for the chip, eliminating the possibility of jamming. Hand-operated reamers of this type differ from the machine-operated ones by the greater number of teeth (3 - 4) and the greater value of pitch (12 - 60 millimeters). The angle of the milling cutter used in their fabrication is 75 degrees. In the remaining features, the designs are similar.

Hard-Alloy-Tipped Reamers

The use of hard alloys in reaming is stipulated by their great resistance to wear and by their relatively low sensitivity to the non-homogeneity and incrustations in the work material.

Hard-alloy-tipped reamers permit the application of a cutting speed several times greater than is possible with high-speed steel reamers.

In order to avoid the accumulation of grease on the grinding disk when grinding the reamer teeth, the length of the hard-alloy blades is calculated to be the exact length of the working part, which, in turn, is made shorter by one third than in the case of conventional reamers. Carbon steel is used for the reamer body, the carbon content being 0.6 - 0.7 percent. This is permissible, since the body of the reamer, due to the presence of teeth along the full length of the working part, does not come into contact with the work surface. The thickness of the hard-alloy blades must not be too great; otherwise it would be impossible to use them in the case of small-size reamers. The thickness of the hard-alloy blade is usually equal to 1/10 - 1/12 of its length.

The front rake angle $\gamma = 0$; the back relief angle $\alpha = 12 \div 15$ degrees; the cutting angle $\varphi = 2\frac{1}{2}$ degrees;

the bevel on the face is one millimeter X 45 degrees. The back taper runs within the range of 0.015 - 0.025 millimeters. For reaming blind holes, it is recommended that the reamer be equipped with face-cutting teeth.

For the machining of deep holes, three-tooth short reamers with brazed-on hard-alloy blades (T15K6), equipped with a hardwood front guide, are used. The cutting angle $\varphi = 75$ degrees; the back relief angle upon the cutting edge is $\alpha = 3$ degrees. These reamers work under conditions of intensive cooling during accelerated cutting speeds.

When machining in hard metals with $\sigma_{в} > 90$ kilograms per square millimeter, trihedral reamers without milled flutes are used. The hard-alloy blades are brazed into the grooves, disposed at the apexes of the trihedral section of the body, in such a way that the front rake angle γ has a negative value. The back relief angle $\alpha = 8$ degrees. The back taper is made to an angle of 2 degrees. Such reamers operate at high cutting speeds -- up to 80 meters per minute.

BIBLIOGRAPHY AND SOURCES

1. Kutay, A. K., and Shtermer, G. A., Tolerances, Reamers, and Countersinks, ONTI, 1936.

2. Semchenko, I. I., Cutting Tool, volume I, ONTI, 1936.

3. Chetverikov, S. S., Metal Cutting Tools, Mashgiz, 1945.