MINISTRY OF EDUCATION OF THE REPUBLIC OF BELARUS
Belarusian National Technical University

Department "Technological equipment"

O. K. Yatskevich
L. A. Kolesnikov
A. V. Azhar

CUTTING TOOLS. DRILLING AND MILLING

Approved by the Educational and Methodical Association
of education institution in the field
of machine-building equipment and technologies
as manual for students of specialty
1-36 01 01 "Engineering technology"

Minsk
BNTU
2022

УДК 621.91.01:811.541(075.8)
ББК 34.63-52я7
Я93

Reviewers:
Head of the Technological and Desing Department
of special tools and Technological equipment of the Department
of the Chief technologist of OJSC “MTP” *A. V. Shakhnovich*;
Department “Technology and equipment of machine-building
production” of Polotsk State University (Head of Department,
Doctor of Technical Sciences, Professor *N. N. Popok*)

Yatskevich, O. K.
Я93 Cutting tools. Drilling and milling : manual for students of specialty 1-36 01 01 "Engineering technology" / O. K. Yatskevich, L. A. Kolesnikov, A. V. Azhar. – Minsk : BNTU, 2022. – 46 p.
ISBN 978-985-583-834-1.

The objective of this handbook is to provide a student understanding of conventional cutting processes applied to metallic workpieces, such as drilling and milling . The definition of tool geometry, cutting processes movements, machining regime elements are considered. The main types of drills and mill cutters are described.

УДК 621.91.01:811.541(075.8)
ББК 34.63-52я7

ISBN 978-985-583-834-1

© Yatskevich O. K., Kolesnikov L. A.,
Azhar A. V., 2022
© Belarusian National
Technical University, 2022

Content

Introduction

Traditional machining is the broad term used to describe removal of material from a workpiece, and covers chip formation operations including: turning, milling, drilling and grinding.

This handbook aims to provide practical information on modern machining technology and cutting tools with an emphasis on the drilling and milling processes: the fundamentals and applications of this types of cutting tools.

The performance characteristics of cutting tools and the significant development of existing and new drilling and milling cutting tools are considered.

The first chapter deals with modern drilling systems which can signicantly reduce hole-making costs and increase the efficiency of the drilling operations. This is important because approximately 36 % of all machine hours (40 % of CNC) are spent performing hole-making operations.

The second chapter deals with the basic definitions and parameters of the milling operation, currently the most important of the chip removal techniques due to its versatility in manufacturing complex and precise geometries.

The handbook can be used as a text book for special undergraduate engineering courses or as a unit on machining technology at the postgraduate level.

1. DRILLING

1.1. Drilling technology

Drilling is the process of cutting holes in a solid material using a rotating cutting tool. Kinematics of drilling consists of two movements (fig. 1.1): the primary one is rotation of the tool (or workpiece) and linear motion that is feeding along the tool axis.

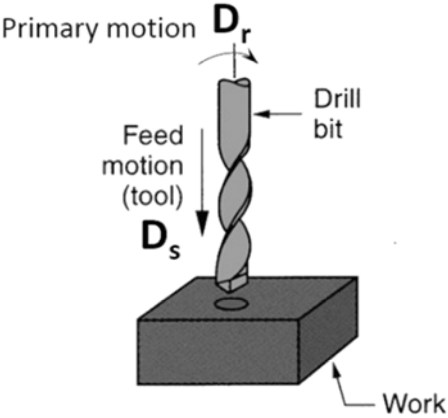


Fig. 1.1. Drilling operation

The tools usually have a complex design. The following general characteristics of drilling processes can make their use problematic:

1. The cutting speed is dependent on the radius, i. e., it is proportional to the radius from the axis of rotation of the drill and has a value of zero at the axis itself. As a result, no material separation can take place at this part of the drill. This influences the necessary forces and torques.
2. Chips have to be removed from the drilled hole. The respective travel distance increases with the drilling depth, which can cause problems in the chip removal.
3. The supply of cutting liquid also becomes increasingly difficult with an increase in the drilling depth, which sometimes makes additional methods necessary (internal cutting liquid supply).
4. Drilling tools can only produce holes of fixed dimensions, so that the drilling diameter cannot be adjusted by means of the process control.

Common drilling processes are shown in fig. 1.2.

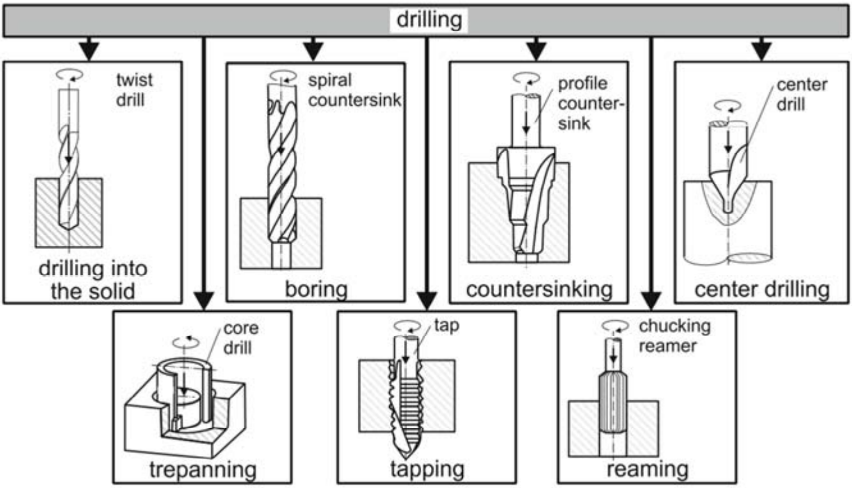


Fig. 1.2. Drilling processes

In *boring* out and *countersinking* processes, a previously produced hole is enlarged. It is difficult to guide the tool on a coaxial path, because there is no center point in the workpiece material, so the tools generally have three or multiple cutting edges. However, chip removal is easier than in drilling from the solid. Therefore, bore holes with a high diameter are often predrilled using a smaller twist drill and then enlarged within several steps, until they reach the nominal diameter. This procedure is also called for, if the maximum permitted feed force, respectively the maximum permitted torque of a machine are not sufficient to drill the hole within one stroke.

Center drilling is necessary, if the surface to be drilled is rough, uneven or inclined. It can be avoided by using a very stiff guidance for the drill (drill clamped with a short projecting length, stiff spindle) or by using drill bushings.

Trepanning is used for large bore hole diameters. An annular cut is produced instead of cutting the whole bore diameter. This reduces the required torque and power. However, trepanning can only be used for through holes.

Tapping is used to produce internal threads. The tap feed must be adjusted to the thread pitch. In machine production, this can be achieved by exact guidance using an NC machine tool or by first using a starting taper and then a tap holder. The rotational direction has to be reversed to remove the drill from the hole.

Reaming is a finishing process corresponding to a boring out process with a multiple edged tool and a small depth of cut. It is used for bore holes with high dimension and shape accuracy. The position accuracy cannot be influenced. Tolerance grades of IT 7 with an increased effort even IT 6 can be achieved. The surface roughness R_z is about 5 μm. Conventional HSS reamers are used at low cutting speeds of 10–20 m/min and low feed rates of 0.08–1.25 mm. If clocked automated systems are used, this kind of reaming has a significant influence on the clock cycle.

1.2. Drilling parameters

The technological parameters that make up essential manipulated variables will be defined using the example of drilling processes: *the cutting speed* (V) and *the feed per cutting edge* S_z (f_z). They are each limited by a characteristic process limit. The cutting speed is limited due to the thermal load and thus by the wear behavior of the tool, while the feed per cutting edge is limited due to the mechanical load. The derived manipulated variables like the rotational speed n with the drilling radius r and the feed speed Vs (V_f) with the cutting edge number z depend on these technological parameters as follows:

$$V = \frac{\pi D n}{1000}, \tag{1.1}$$

where $\pi = 3.14$;

D – diametr of a drill, mm;

n – rotational speed or (rpm), rev/min.

The feed rate S_M is the velocity of the tool in the feed direction. It is measured in millimeters per minute (mm/min) and is calculated as

$$S_M = Sn, \tag{1.2}$$

where S – feed per revolution, mm/rev.

The feed per tooth

$$S_Z = \frac{S}{z}, \tag{1.3}$$

then

$$S_M = zS_Z n, \tag{1.4}$$

where z – number of teeth.

In general, a twist drill is used with $z = 2$.

The depth of cut t is the thickness of metal removed from workpiece, measured in a radial direction (mm):

$$t = \frac{D}{2}, \tag{1.5}$$

where D – diametr of a drill, mm.

1.3. Drill geometry

The process of drilling is implemented by the drills of various types. Though the HSS twist drill is one of the most complex designs of the multiple tools, being the most important.

Twist drill geometry and its nomenclature are shown in fig. 1.3. A twist drill has three principal parts: drill point or cutting part, body and shank.

Drill point is the sharpened end of the drill body consisting of all that part which is shaped to produce lips, faces and chisel edge.

Body is that portion of the drill part, which extends from the extreme cutting end to the beginning of the shank.

Neck is the portion of the drill body between the flutes and the shank provided so as to facilitate the grinding of the body. Parallel shank drills of small diameter are not usually provided with a recess.

Shank is that portion of the drill by which it is held and driven. As a result the shank particularly serves the purpose of torque transmission. The shank of the drill is generally held in a holder or the spindle and driven. Driver design is important because it determines the roundness accuracy and stiffness of the drill-holder system, as well as the limiting speeds and coolant pressures, which can be used in some applications. Maximizing the driver length improves rigidity and concentricity.

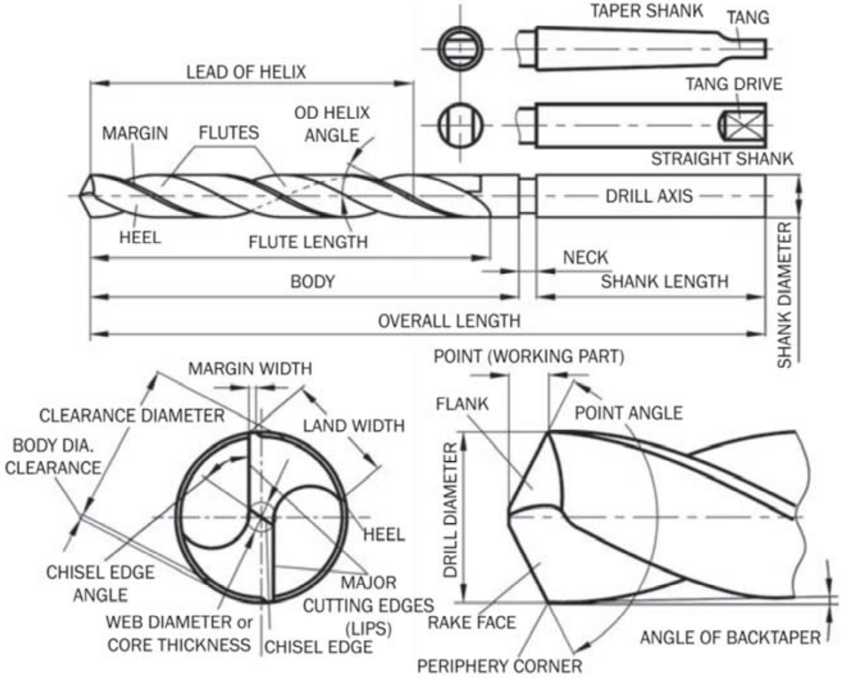


Fig. 1.3. Twist drill nomenclature

Tapered shank drills are mounted directly into the spindle with or without intermediate sleeves or adapters. Straight shank drills (with or without ground flats) are mounted in end mill holders, collets, chucks, or special hydraulically or mechanically clamped holders.

Major cutting edge (or lip**)** is the edge formed by the intersection of the major flank face and rake face (fig 1.4).

Chisel edge is the edge formed by the intersection of the flanks.

Drill web is the central portion of the drill situated between the roots of the flutes and extending from the point end towards the shank. The point end of the core forms the chisel edge.

Rake face is that portion of the flute surface adjacent to the lip on which the chip impinges as it is cut from the work.

Flank is that surface on a drill point which extends behind the lip to the following flute.

Flutes are the grooves in the body of the drill, which provide lips, allow the removal of chips, and permit cutting fluid to reach the lips.

Lands are the cylindrically ground surfaces on the leading edges of the drill flutes. The width of the land is measured at right angles to the flute.

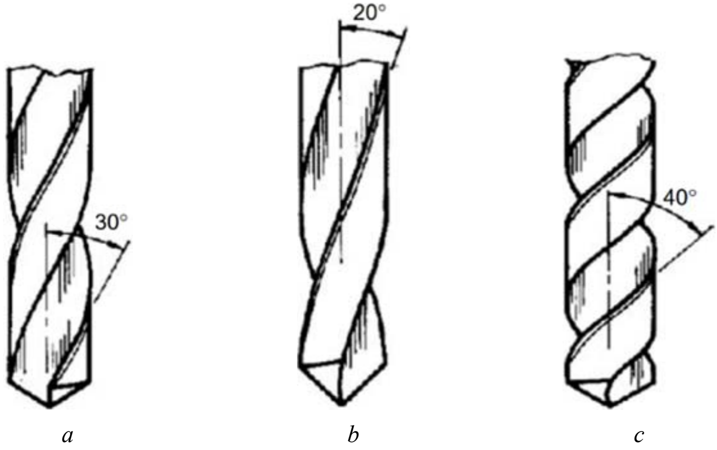
Margin is the cylindrical portion of the land which is not cut away to provide clearance.

Point angle 2φ is the included angle of the cone formed by the lips.

Chisel edge angle ψ is the corner formed by the intersection of a lip and the chisel edge. Drill diameter is the measurement across the cylindrical lands at the outer corners of the drill.

Helix angle ω is the angle between the leading edge of the land and the drill axis.

Referring to the terminology of twist drill shown in fig. 1.3, the helix angle of the twist drill is the equivalent of the rake angle of other cutting tools. The standard helix is 30°, which, together with a point angle of 118°, is suitable for drilling steel and castiron (fig. 1.4, *a*). Drills with a helix angle of 20°, known as slow-helix drills, are available with a point of 118° for cutting brass and bronze (fig. 1.4, *b*), and with a point of 90° for cutting plastics.



a *b* *c*

Fig. 1.4. Helix drills of different helix angles:
a – standard; *b* – slow; *c* – quick

Quick helix drills, with a helix angle of 40° and a point of 100°, are suitable for drilling softer materials such as aluminum alloys and copper (fig. 1.4, *c*).

Clearance angle α is the angle formed by the flank and a plane at right angles to the drill axis. The angle α is normally measured at the periphery of the drill. To make sure that the main cutting edges can enter into the material, the clearance faces slope backwards in a curve. The clearance angle is measured at the face edge, must amount to 5° up to 8°.

Rake angle γ is the angle between the face and a line parallel to the drill axis. It is bigger at the face edges and decreases towards the center of the drill to nearly 0°. The result is that the formation of chips grows more unfavorable towards the centre.

By means of the *complex cutting edge geometries* of the different drills, the tool can be adapted to each specific drilling task. On the one hand, the profile of the twist drill has to possess large flutes in order to provide sufficient space for chip removal; on the other hand, the drill must exhibit appropriate torsional rigidity (polar moment of inertia) and torsional strength (shear modulus). The helix angle ω of the flutes influences both the chip removal and the rake angle at the cutting edges.

The rake angle at the drill has an essential influence on the deformation and the forces at the cutting edges. We distinguish between the rake angle at the chisel edge γ_q, which can by all means be highly negative due to geometrical reasons and the rake angle at the major cutting edge γ_h (fig. 1.5).

Near the center of the drill, γ_q is $-\sigma / 2$. Further along the chisel edge it rises slightly.

Fig. 1.6 illustrates the significant change in the rake angle along the radius of the drill, by means of a cross-section through the chip formation zone in front of the cutting edges.

The main weaknesses of twist drills that reduce the drill life include:

1) high positive rake angles on the drill periphery;
2) unfavorable geometry of the chisel edge;
3) the lack of clearance on the minor cutting edges (margins).

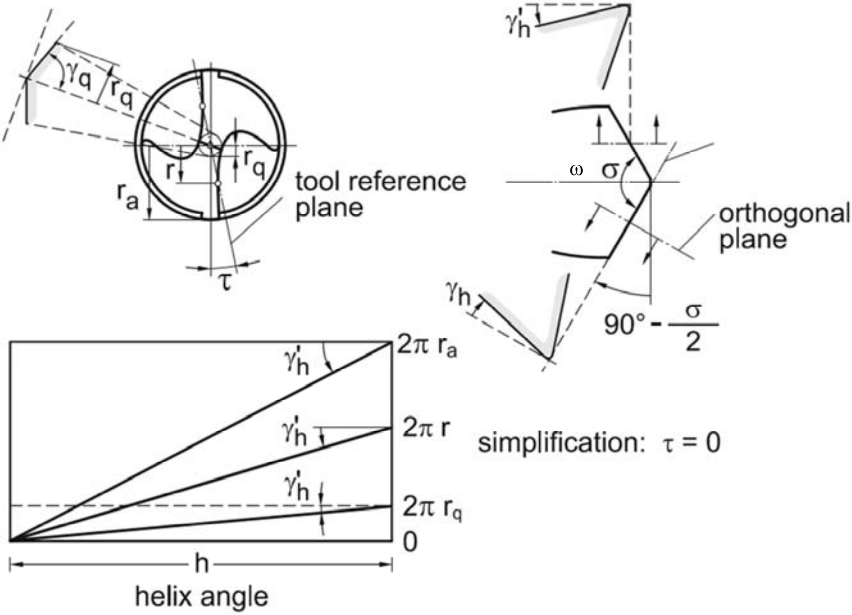


Fig. 1.5. Rake angle at the drill

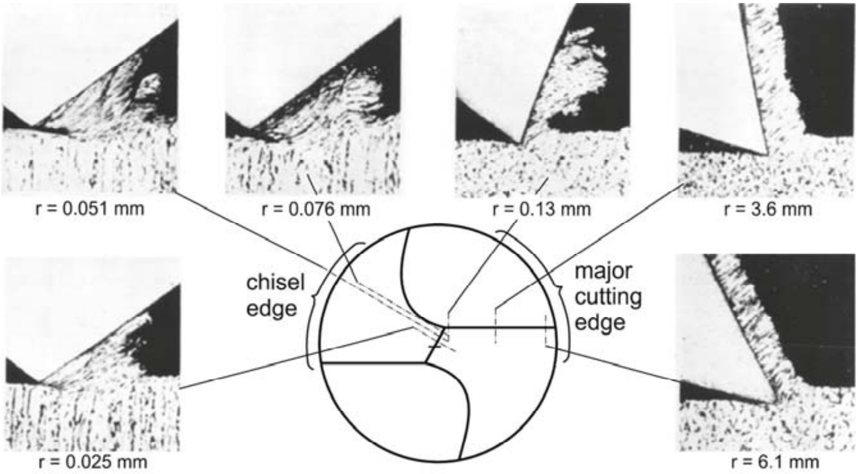


Fig. 1.6. Chip formation in drilling from the solid

The following ways are widely used to reduce the effect of these short-comings in practice:

1. Sharpening of the margin with clearance angles α_1 = 6–8° and a small land left uncut. This reduces the friction forces on margins, and eliminates adhesion of fine chips in drilling of steels, resulting in increased tool life.

2. Various methods of web thinning are applied, reducing the thrust force, improving the conditions of hole starting and increasing the drilling performance by allowing to increase the feed rate.

3. Double cone point sharpening of the drill is used. Here $2\varphi = 116^\circ$, $2\varphi_1$ = 70–90°. This reduces the wear on the most vulnerable sections of the peripheral cutting edges of the drill, where the cutting speed is the greatest, and the rake angles are less than 7–8°.

1.4. Cutting force components in drilling

In contrast to turning the drilling involves not one, but three cutting edges (two lips and one chisel edge), which together determine the power load on a drill. In addition, the torque needed for drilling is affected by the friction between drill margins and walls of the hole being drilled.

The resultant cutting force applied at the drill lip can be resolved, as in turning, into three mutually perpendicular components:

1) P_Z – tangential, acts in the direction of the cutting velocity;
2) P_X – the thrust force parallel to the drill axis;
3) P_Y – radial to the drill axis (fig. 1.7).

The radial components P_Y at the lips are directed in opposite directions and in theory should be balanced. However, in practice due to errors of sharpening and mounting of the drill, uneven cutting load caused by, for example, hard spots in a work or other reasons, the balance of the radial components P_Y is disturbed. In this case, because of the appearance of unbalanced force ΔP_Y the transverse vibrations of the drill arise. These vibrations cause the hole oversize (undesirable increase of drilled hole diameter), drill wandering, which increase with depth of drilling increasing.

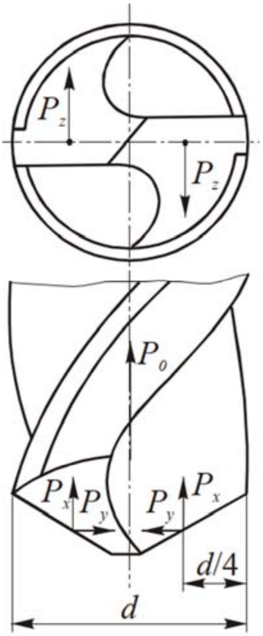


Fig. 1.7. System of forces acting on a twist drill

In practice, the axial force and torque are calculated, respectively, by the following equations

$$P_O = C_p d^{x_p} f^{y_p} K_p,$$
$$M_T = C_W d^{x_{T'}} S^{y_T} K_T, \tag{1.6}$$

where C_P, C_W – coefficients characterizing drilling conditions and properties of the reference workpiece material;

K_P, K_T – factors that characterize the drilling conditions (method of sharpening, web thinning, cutting fluid, etc.).

Hence the power consumed in drilling is

$$Ne = 1.6 \cdot 10^{-3} M_T n, \text{ kW.} \tag{1.7}$$

1.5. Wear and life of drills

In drilling the cutting speeds and friction travel are variable along the cutting lips. The chips are wide and thin, thus the twist drills wear mostly on the clearance surfaces adjacent to the cutting lips. The wear takes the form of wear lands h_F, which are variable along the cutting edges. In drilling brittle materials the drill wear occurs on the lip corners. In drilling ductile materials at high speeds the wear take place on the rake surfaces in the form of craters and on the margins in the form of wear land h_C (fig. 1.8).

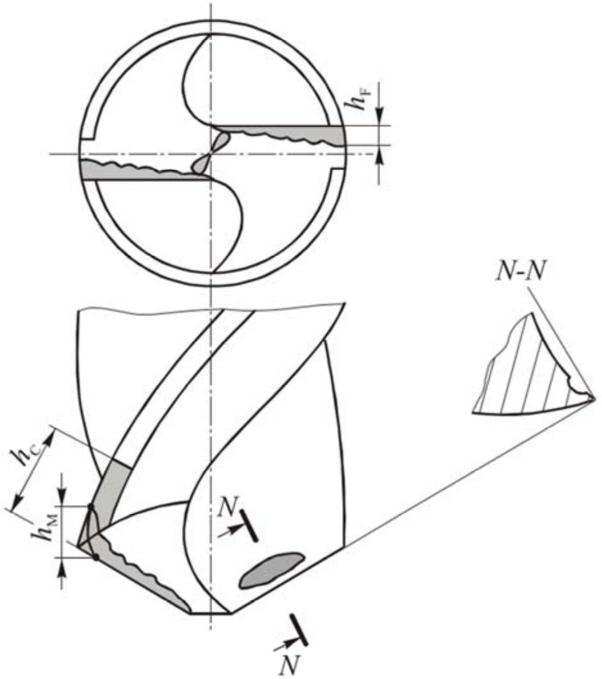


Fig. 1.8. Wear on the rake and clearance surfaces of a twist drill

The most dangerous is the wear on the lip corners and margins, as in this case, heavy wear will need a significant amount of the tool material to be ground off. Therefore, in drilling the catastrophic wear, as the lip corner melting and margins failure, should not be allowed.

Chisel edge due to low velocities in the center of the drill wear much slower. The heavy wear of the chisel edge is sharply increases thrust pressure, and the wear on the margins significantly increases the torque. Intensive wear of the chisel edge indicates that the heat treatment parameters were violated during the manufacturing process.

The greatest influence on the drill wear has the cutting speed and a much smaller effect – feed rate. For this reason, in terms of drill life, it is preferable to work with a greater fed rate and lower cutting speed, as for a given tool life *T* it provides higher productivity of the drilling process.

1.6. Types of drills

Drills are mainly distinguished by design, according to which the following basic types are found:

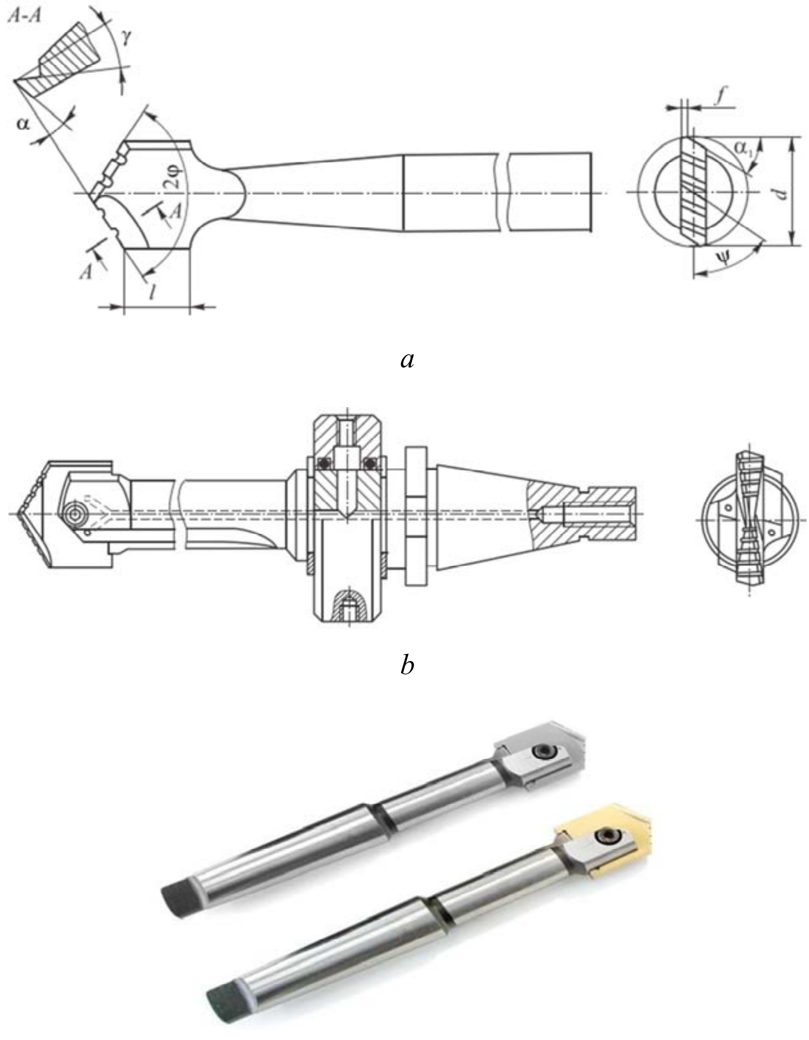
1) spade drills;
2) twist drills (with helical flutes);
3) special type (for drilling deep holes, trepanning drills, combination drills, etc.).

The cutting part is usually made of high-speed steel. In recent years, the carbide drills of various designs have spread.

1.6.1. Spade drills

Spade drills have been known since ancient times. For example, their prototypes in the form of hard material spades pointed at the end and designed for manual drilling holes in softer materials were found in the archaeological excavations. Spade drills have been constantly improved since the emergence of metalworking. Modern constructions of spade drills are shown in fig. 1.9, *a*, *b*, *c*.

Solid spade drills (fig. 1.9, *a*) are made of a bar with forged or milled plate – shaped head, which is sharpened to a cone angle of $2\varphi = 118^{\circ}$. This process forms two major and two minor cutting edges. Grinding of two flat flanks creates clearance angle $\alpha = 10–12^{\circ}$. At the intersection of these surfaces the chisel edge is formed. If the rake surfaces are flat, the rake angles on the major cutting edges have negative values, which is undesirable due to the increase of power load on the drill and the appearance of chatter.



a

b

c

Fig. 1.9. Flat drills:
a – solid; *b, c* – indexable

The advantages of spade drills are a simple design, and the possibility to manufacture drills of different diameter and length even in the repair work-shops.

Disadvantages of spade drills include:

1) difficult conditions of chip removal;

2) tendency to chatter due to low stiffness of the cutting part;

3) low number of permissible resharpenings;

4) low performance of the drilling process due to small values of feed and the necessity of drill withdrawal out of the hole for chip removal.

1.6.2. Deep hole drills

The term “deep holes” is usually related to the holes, the depth of which exceeds $5d$. However, even with the $h > 3d$ there are some difficulties with a coolant supply to the cutting zone and chip removal out of the hole using twist drills, which leads to a reduction of tool life. Therefore, in practice, the application of deep hole drilling tools usually starts with a hole depth larger than $3d$.

The main difficulties of drilling deep holes are:

1) difficult conditions of coolant supply to the cutting area and chip removal;

2) axial hole deviation;

3) dimensional errors and form errors of the holes in the radial and longitudinal sections.

In practice, the drilling of deep holes with depths up to $20d$ is performed on universal drilling machines with help of twist drills of long series, which have longer shank and body of standard length. In this case, automatic pecking of the tool is used to release a drill from the chip during drilling.

To improve chip evacuation the special high-helix drills or are implemented (fig. 1.10), which are mostly used for drilling holes with depths up to $(30–40)d$ in the cast iron and other brittle metals. Unlike standard twist drills, they have greater helix angle of the helical flutes $\omega = 60^\circ$ and increased web diameter $d_0 = (0.30–0.35)d$.

To ensure effective chip breaking without pecking and raise the drill life, the HSS coolant-fed twist drills are used. Their drawback is high manufacturing complexity, the necessity to have special internal coolant chucks and pump stations, as well as protection from chips and coolant spray.

Drill wandering with two symmetrically positioned lips occurs due to low stiffness of end mounting of the drills, inevitable errors of sharpening of the cutting edges, evidence of hard spots in workpiece material, etc.

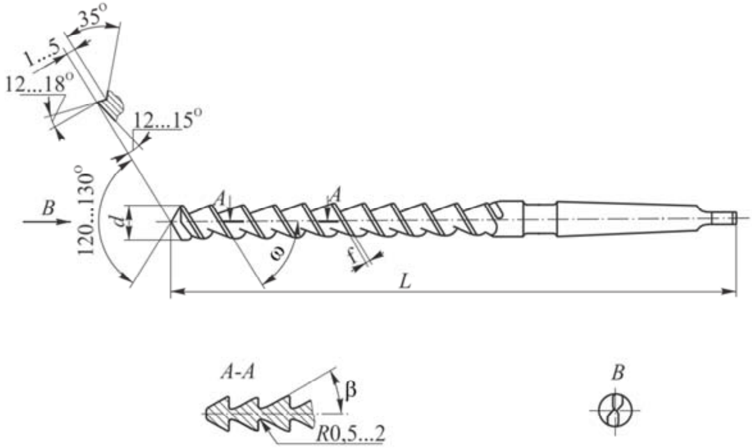


Fig. 1.10. Deep hole auger drill

The most effective way to minimize drill wandering and increase the accuracy of the holes is an application of self-piloting cutting tools that are supported and guided by the machined surface. For this purpose, the cutting edges are positioned so that the deliberately unbalanced radial component of the cutting force arises and presses guide pads against the hole surface, which is machined by the cutting edges positioned ahead these pads (fig. 1.11).

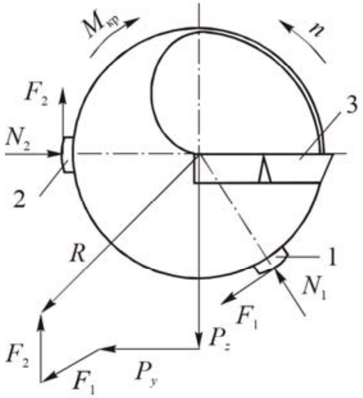


Fig. 1.11. Forces acting in a transverse plane of a self-pivoted drill:
1, *2* – carbide guide pads; *3* – carbide cutting insert

Tools working on this basis are called self-piloting tools or tools of single-sided cutting. These tools include *half-round drills*, *gun drills*, *BTA drills* and *ejector drills*. They can have one or more cutting edges, but in any case the total radial component *R* of the cutting and friction forces should be directed straight to the supporting surface and be located between the guide pads, to implement the self-piloting principle.

Gun drills

Gun drills (fig. 1.12) in contrast to half-round drills, have internal channel for coolant supply and straight (sometimes helical) flute for the external evacuation of mixture of chip with coolant. Gun drills are used for drilling holes with depths equal to (5–100)*d* and a diameter of 1–30 mm. Originally gun drills were used for drilling gun barrels. Nowadays gun drills are widely used in all branches of engineering, mainly for drilling deep holes on special machines in mass production. Equipped with carbide inserts and internal coolant supply, they provide high productivity for drilling holes with high straightness and high dimensional accuracy (H8…H9) and good surface finish of the machined holes (Ra 0.32–1.25 µm).

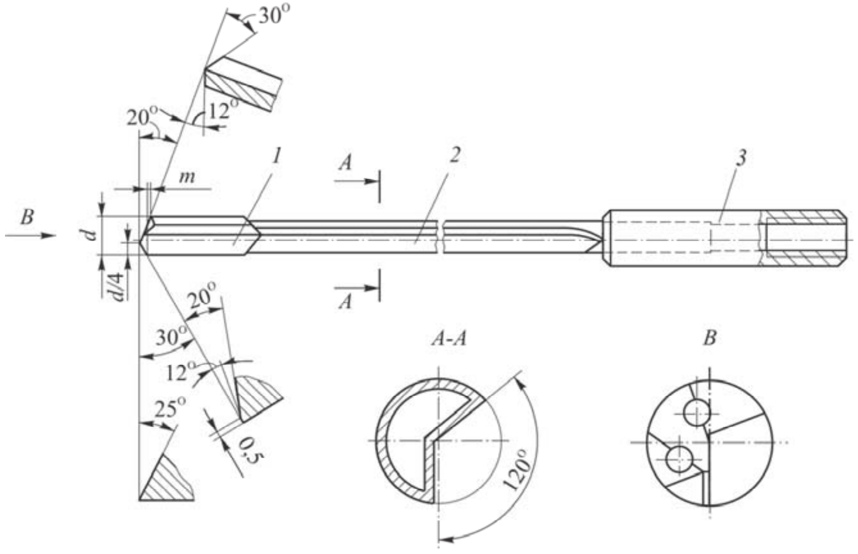


Fig. 1.12. Gun drill (*d* = 1–30 mm):
1 – tip; *2* – shank with oil hole and *V*-flute; *3* – driver

Pressure and coolant flow rate depends on the diameter of the drill. For example, the coolant pressure for small diameter drills is 9–10 MPa.

The disadvantages of gun drills include small lateral and torsional stiffness due to tubular shank weakened by a flute. For this reason, the feed rates are reduced, and hence the productivity of the drilling operation.

BTA drills

BTA drills (fig. 1.13) are produced in various designs: single-edged, multiple-edged, brazed or indexable.

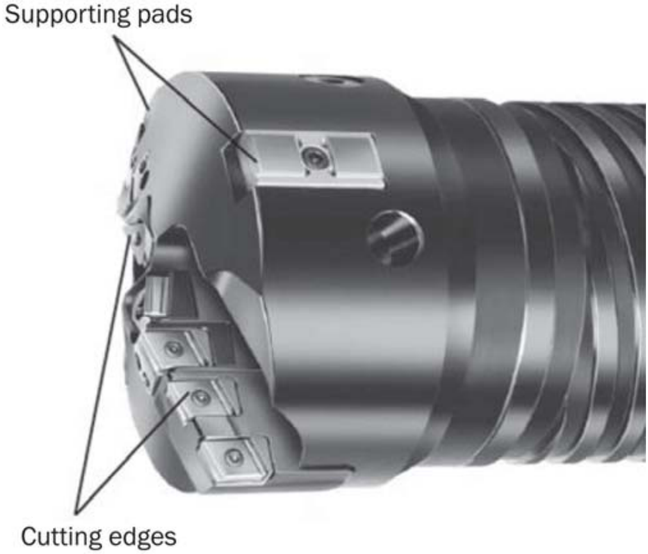


Fig. 1.13. BTA drills head with a set of indexable inserts ($d > 65$ mm)

Unlike gun drills, the BTA drills and heads have thick-walled tubular shank of ring crosssection. BTA drills work with external coolant supply (between the walls of the shank and machined hole), internal coolant evacuation and chip removal.

Advantages of BTA drills include 2–4 times higher feed rates due to the high rigidity of the tubular shank as compared to gun drills, and improved surface finish of the machined surface due to internal chip evacuation.

Disadvantages of BTA drills include difficulties with reliable chip removal through a relatively small cross-section hole of a tubular shank.

The common depths of horizontal BTA drilling are up to 100*d*, and in vertical drilling are up to 50*d*.

Ejector drills

Ejector drills are similar to BTA drills in design (fig. 1.14). Differences between these two drills consist in coolant supply and evacuation of chip with coolant mixture. The coolant is supplied between the inner *6* and outer *7* tubes, and then through the holes in the head to the cutting area. Ejector nozzles *5* are C-shaped and are positioned in the back part of the thin-walled tube *6*.

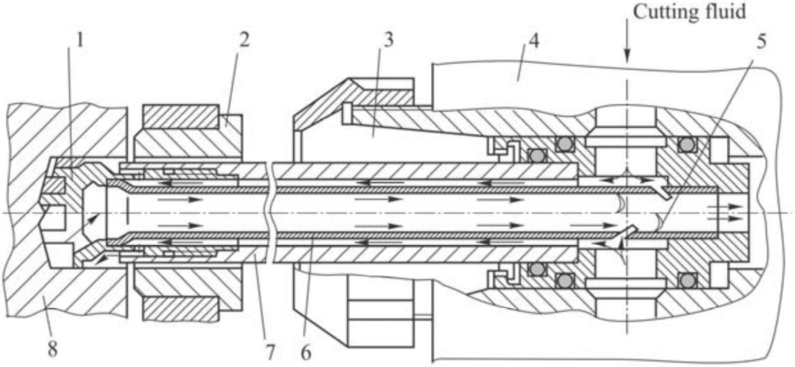


Fig. 1.14. Ejector drill operation:
1 – cutting head; *2* – guide bushing; *3* – collet; *4* – chuck; *5* – orifices;
6 – thin-walled tube; *7* – tubular shank; *8* – workpiece

The maximum depth of the holes obtained by ejector drilling is up to 4000 mm.

Trepanning drills

A large amount of chips that are produced in drilling of the holes with diameter greater than 50 mm leads to considerable power consumption and higher cost of tools due to increased use of tool materials. Moreover, cutting forces are also sharply increased. During trepanning the annular groove is cut in the workpiece and a center core is left uncut, allowing for its further use as a workpiece or test sample. Due to lower horsepower, the feed rate can be significantly increased and hence the productivity of the drilling operation is also increased.

Fig. 1.15. Trepanning drills with mechanically clamped inserts and guide pads

The simplest design of a trepan drill is shown in fig. 1.15. Cutting fluid is supplied through the holes in the tool body and is removed along with chips through the gap between the drill periphery and the workpiece.

1.7. Core drills

Core drills are the multiple point axial cutting tools, which are used for semifinishing or finishing holes, created by drilling, casting, forging or stamping. Core-drilling is used to improve hole accuracy to IT11...IT10 and improve surface finish to Ra 40–10 μm.

Kinematics of core-drilling is similar to drilling. However, core drills provide greater productivity and accuracy, compared to drills, since the allowance removed is smaller (t = 1.5–4.0 mm, d = 18–80 mm), core-drills have a greater number of cutting edges (z = 3–4) and margins. Due to shallow flutes, core-drills have higher rigidity as compared to drills, and the lack of chisel edge allows higher feed rates.

Core-drills are classified according to:

a) type of cutting: cylindrical core-drill is used to increase the diameter of the holes (fig. 1.16, *a*), counter bore is used to provide a step of enlarged diameter in a hole (fig. 1.16, *b*), countersink produces chamfer or conical entry to the hole to allow the head of a screw to be seated beneath the part surface (fig. 1.16, *d*), spot facer creates a flat adjacent to the hole on bosses and pads of the parts (fig. 1.16, *c*);

b) method of core-drill mounting: shank-type core drill (with either straight or tapered shank and d = 10–40 mm and z = 3) and shell core-drill (d = 32–80 mm, z = 4);

c) core-drill design: solid, assembled or indexable (with inserted blades, d = 40–120 mm) and adjustable;

d) type of cutting material: high-speed steel and carbide.

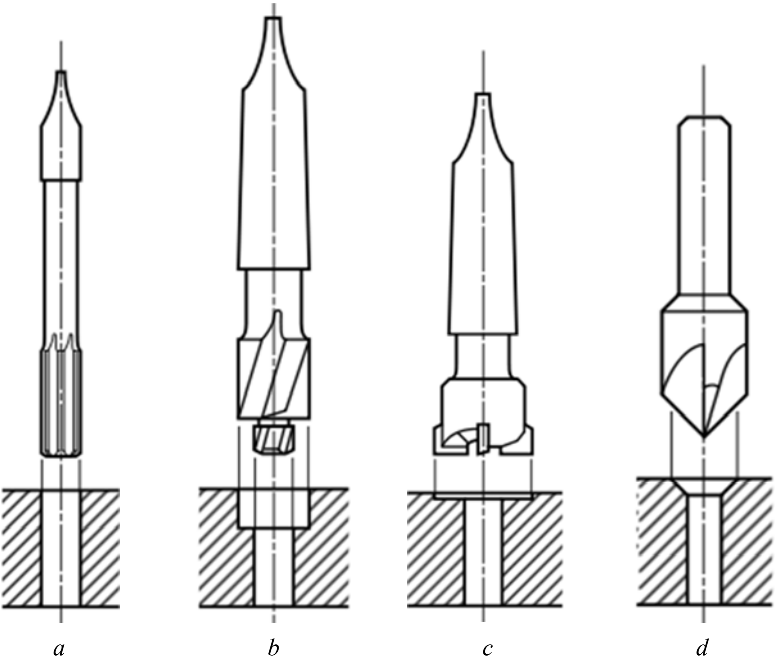


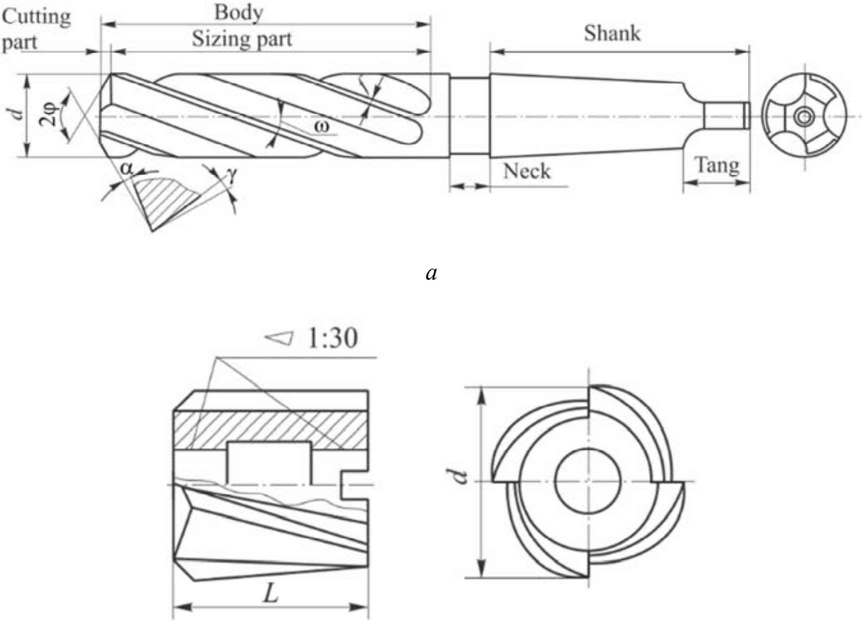
Fig. 1.16. Core drills:
a – straight core drill; *b* – counter bore; *c* – spot facer; *d* – countersink

Cylindrical core-drills can be of shank-type (fig. 1.17, *a*), and shell-type (fig. 1.17, *b*). The main structural elements of a core-drill are: the cutting part (starting taper), the sizing part, number of flutes (teeth), the shape of flutes, and type of mounting part. The geometric parameters include: point angle 2φ, rake γ and clearance angle α helix angle ω. Sizing part of a core-drill provides the required accuracy of the machined hole, guides the core-drill in the hole and serves as a reserve for regrinding. Sizing part includes cylindrical margins of width $f = 0.8$–2.0 mm for $d = 10$–80 mm. Radial run-out of the margin should be smaller than 0.04–0.06 mm.

Back taper of margins is used to reduce friction and to eliminate core-drill jamming in the hole, and is equal to 0.04–0.10 mm per 100 mm depending on the diameter of the tool.

In terms of number of flutes, core drills are made with three (shank-type) or four (shell-type) flutes. Shell core-drills of larger size ($d > 58$ mm) have six or more flutes. Core-drill flutes are generally helical, but they may be straight, such as carbide core drills for cutting steel and hardened cast iron. Assembled core-drills with clamped or brazed inserts have flutes inclined to the core-drill axis.

In cutting of steel the cutting edge angle is $\varphi = 60^{o}$. In cutting of cast-iron the cutting edge angle is $\varphi = 60^{o}$ or 45^{o}.



a

b

Fig. 1.17. Straight core drills:
a – shank-type; *b* – arbour-type

1.8. Reamers

Reamers are axial multiple point cutting tools used for finishing of the holes. The dimensional accuracy after reaming is IT8...IT6 and surface finish Ra 1.25–0.32 μm. The best results are obtained by double reaming, when the first reaming removes two thirds of the allowance, and the second one removes the remaining 1/3.

Kinematics of reaming is similar to drilling and core-drilling. In contrast to core-drills, reamers have more teeth (z = 6–14) and, as a consequence, the best guiding in the hole. They removes significantly less allowance (t = = 0.15–0.50 mm) than by core-drills. To achieve the minimum surface finish the reamers work at low cutting speeds (v = 4–12 m/min), i. e. when there is no built-up edge. Nevertheless, productivity of reaming is quite high due to large number of teeth, thus the working time is reduced.

To reach high accuracy of the holes the reamers are manufactured with tolerances narrower as compared to core-drills and the holes for reaming are either drilled, core drilled or bored.

Reamers are classified according to:

a) drive type – hand and machine;
b) method of mounting – shank-type or shell-type;
c) type of the hole – cylindrical and tapered;
d) tool material – high-speed steel, carbide and diamond;
e) design – solid, assembled and indexable.

Hand reamers (fig. 1.18, *a*) machine the hole by the rotation of the tool manually by a tap-wrench. These reamers (d = 3–40 mm) are usually made of tool steel grades. A starting taper of greater length and a sizing part of the reamer are designed to get better reamer guidance in the hole. The rest part of the hand reamers doesn't differ from the machine reamers.

Machine reamers of end-type and shell-type either solid and inserted (fig. 1.18, *b*, *c*, *d*) are used for machining holes on drilling and turning machines, turret lathes, coordinate boring machines and other machine tools. Shanks of the machine reamers can be either straight (d = 1–9 mm) or tapered (d = 10–32 mm) with a relatively long neck and a Morse taper. Shell-type reamers are mounted on the arbour. The tapered hole of the spindle (taper 1:30) ensures high accuracy concentricity. Key slot is cut on the face of the reamer for torque transmission.

Working part of straight reamers (fig. 1.18) consists of cutting and sizing parts. The chamfer cut at an angle $\varphi = 45^{\circ}$ facilitates entry of the tool into the hole and protects the cutting edges from damage. Chamfer is followed by the starting taper with a cutting edge angle φ, where teeth remove reaming allowance. The chamfer and starting taper constitute the cutting part of the reamer.

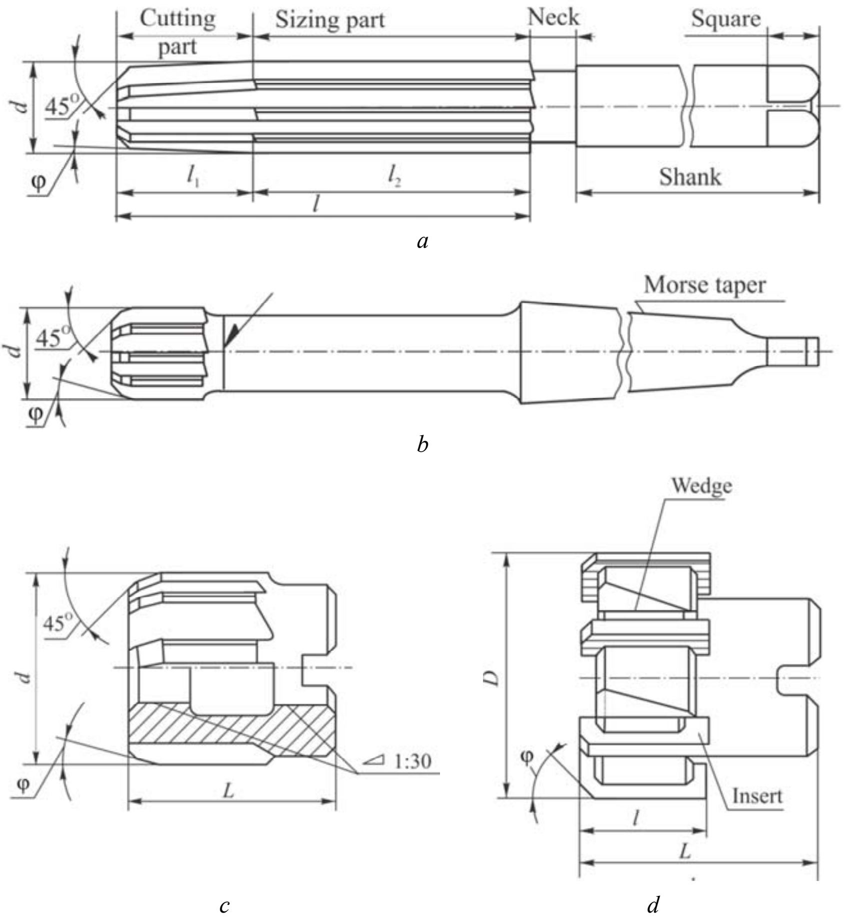


Fig. 1.18. Reamers:
a – hand-type; *b* – machine-type; *c* – shell-type; *d* – inserted

2. MILLING

2.1. Milling technology

Milling in contrast to turning or planning is accomplished with the help of the multiple-point cutting tools and is therefore a very productive method to machine flats, faces, shoulders, grooves, slots, and various shaped surfaces (gears, splines, threads, etc.). The primary motion in milling is a rotation motion of a milling cutter around its axis, and feed motion is implemented either by the cutter or by the workpiece mounted to a milling machine table (fig. 2.1). Unlike turning, in milling these two motions are not kinematically interlocked and are executed independently.

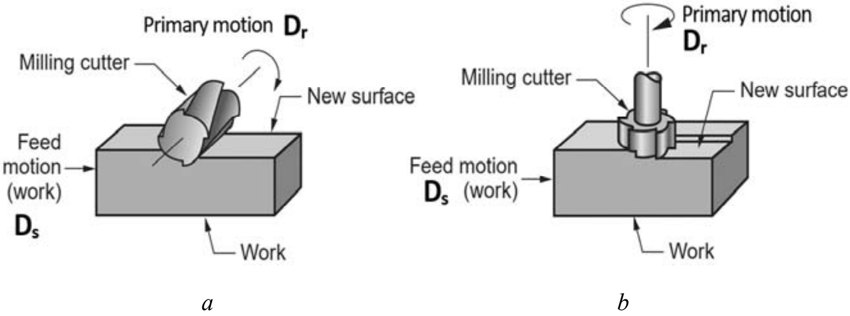


Fig. 2.1. Milling operation:
a – peripheral milling; *b* – face milling

In the process of milling, several teeth are engaged with the workpiece, removing chips of variable thickness. High performance of milling process is provided due to the large total active length of the cutting edges. Higher speeds in milling help to increase productivity, due to periodic exit of the teeth from the cutting area, providing cooling and removal of thermal stress in the cutting wedge.

The largest variety of shapes can be produced by form milling, even though the range of producible shapes depends on the number of controllable feed axes on the milling machine. A milling machine usually possesses three linear feed axes, which can be controlled both simultaneously and independently, enabling the tool to follow any three-dimensional path. In special machines, two rotational axes are added to these three linear axes (five axes milling), so that the rotational axis of the milling cutter can be positioned in any direction at any point of the path.

The most important milling processes are shown in fig. 2.2. They are classified according to the produced shapes, which are determined by the feed motion. In *face milling* the rotation axis of the tool is orthogonal to the produced surface, while in *peripheral milling* it is parallel to the surface. *Side milling* is a combination of these two processes and is used to produce two surfaces that are orthogonal to each other. *Helical milling* and *hobbing* produce helical or gear surfaces. In *profile milling* the tool shape is reproduced on the workpiece, making its shape and dimensions dependent on those of the tool.

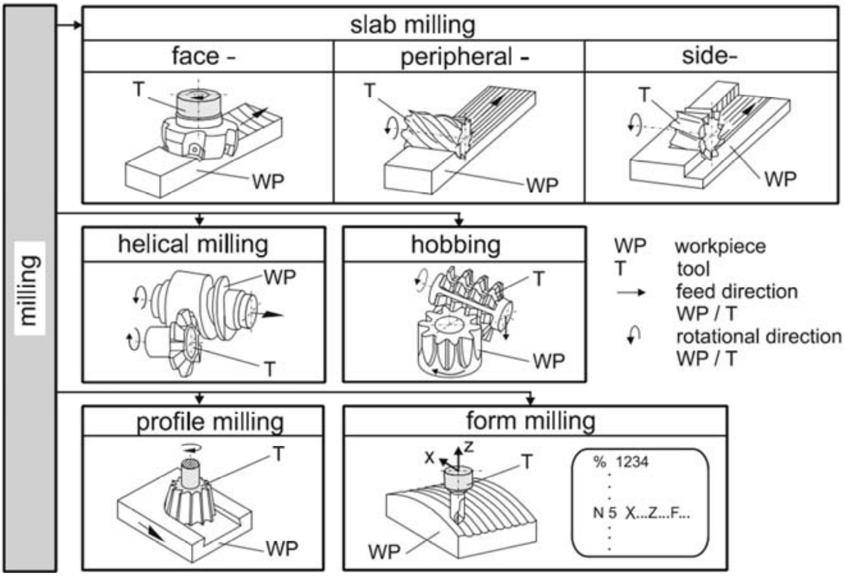


Fig. 2.2. Milling processes

If primary motion and feed motion are opposite in direction, the milling is called *conventional milling* (up-milling), and if the motions are unidirectional that is *the climb milling* (down-milling).

Up-milling is accomplished by rotating the cutter against the direction of the feed of the workpiece (fig. 2.3, *a*). The tooth picks up from the material gradually; that is, the chip starts with no thickness and increases in size as the teeth progress through the cut. This means that the cycle of operation to remove the chip is first a sliding action at the beginning and then a crushing action takes place, which is followed by the actual cutting action. In some metals, up-milling leads to strain hardening of the machined surface, and also to chattering and excessive teeth blunting.

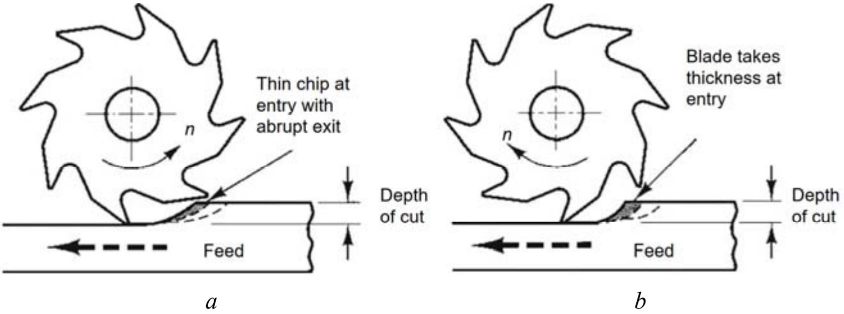


Fig. 2.3. Up-milling (*a*) and down (climb) milling (*b*)

Down-milling is accomplished by rotating the cutter in the direction of the work feed, as shown in fig. 2.3, *b*. In climb milling, as implied by the name, the milling cutter attempts to climb the workpiece. Chips are cut to maximum thickness at initial engagement of cutter teeth with the work, and decrease to zero at the end of its engagement.

Both up- and down-milling have advantages in particular applications. Up-milling is usually preferable to down-milling when the spindle and feed drive exhibit backlash and when the part has large variations in height or a hardened outer layer due to sand casting or flame cutting. In down-milling, there is a tendency for the chip to become wedged between the insert and cutter, causing tool breakage. However, if the spindle and drive are rigid, cutting forces in peripheral down-milling tend to hold the part on the machine and reduce cutting vibrations. Down-milling is also preferred when machining nickel alloys and other materials subject to surface damage, since in down-milling the tool flank and margin do not contact and potentially burnish the machined surface.

2.2. Milling parameters

The cutting action of each cutting edge on a milling cutter is similar to that of a single-point tool. The cutting speed is given by equation (1.1), and the feed rate S_M, feed per revolution S, and feed per tooth S_Z are related by an equation similar to equations (1.2)–(1.4):

$$S_M = Sn = S_Z Zn, \tag{2.1}$$

The variation of the uncut chip thickness in milling is complicated (fig. 2.4, 2.5). The uncut chip thickness varies trochoidally as the cutter rotates. For small feeds, however, a sinusoidal approximation is adequate.

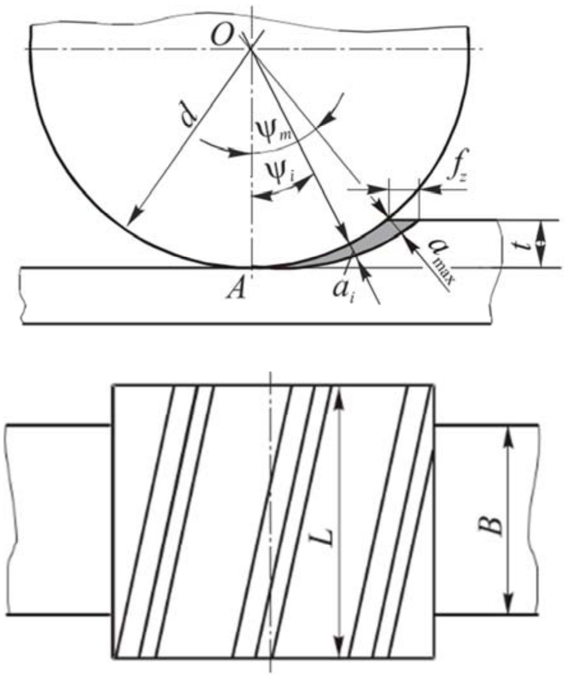


Fig. 2.4. Dimensions of a layer being removed in plain milling:
B – width of cut; t – depth of cut;
ψ_m – angle of contact of a cutter with a workpiece

The uncut chip thickness a_i and engagement angle of ψ_i, maximum uncut chip thickness, $a_{\max}$, and average uncut chip thickness, a_{avg} for *plain milling* (fig. 2.4) are given by:

$$a_i = S_Z \sin\ \psi_i. \tag{2.2}$$

Thus, during the cutter motion along the length of engagement, the chip thickness changes from zero to $a_{\max}$:

$$a_{\max} = S_Z \sin \psi_m , \tag{2.3}$$

Full contact angle ψ_m can be found by the following equations:

$$\cos \psi_m = 1 - \frac{2t}{d} \text{ and } \sin \psi_m = 2\sqrt{\frac{t}{d} - \frac{t^2}{d^2}} \approx 2\sqrt{\frac{t}{2}}. \tag{2.4}$$

Face mills (fig. 2.5) unlike the plain mills work with the scheme of non-free cutting, since in addition to the major cutting edges located on the cutter periphery, the minor cutting edges are also involved in cutting process.

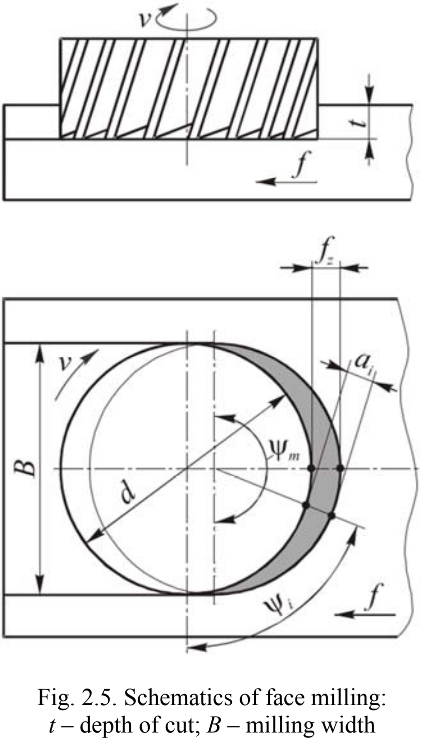


Fig. 2.5. Schematics of face milling:
t – depth of cut; B – milling width

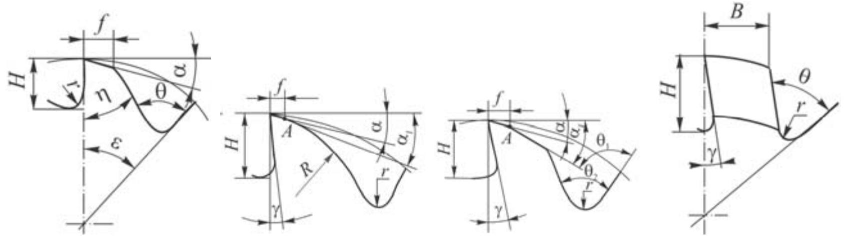
Comparison with the plain milling cutters shows that with the transition from the plain milling to the face milling, the positions of *t* and *B* parameters are swapped over. In addition, face milling is characterized by large angles of contact, which are up to $\psi_m = 180^o$.

Thickness of the chip being removed by each tooth of the face mills at any given time is determined by the same equation as for the plain milling (2.2). The thickness of the chip is variable and varies from zero to a maximum value $a_{max} = S_Z$ at an angle $\psi_i = 90^o$, and then reduces back to zero.

The contact angle ψ_m depends on the position of a milling cutter (in the reference plane) with respect to the workpiece, and on the ratio *B* / *d*. In a full-width milling, contact angle $\psi_m = 180^o$.

2.3. Tool elements and surfaces

It has been mentioned that according to the teeth design the cutters are divided into two groups: pointed (or common) teeth (fig. 2.6, *a*, *b*, *c*) and form-relieved teeth (fig. 2.6, *d*). The principal differences of these mills are in the process of sharpening, number of teeth, its shape, complexity of manufacturing, durability, performance and surface finish.



a *b* *c* *d*

Fig. 2.6. Shapes of the milling cutters teeth:
a – trapezoid; *b* – parabolic; *c* – reinforced; *d* – from-relieved

Milling process is characterized by the removal of thin chips of variable thickness. The chip thickness of the plain milling cutters starts from zero thickness. Regrinding of the pointed teeth by the flank surface, on which the wear occurs more often in milling, helps to reduce the allowance for regrinding, extend the life of milling cutters, reduce the sizes of teeth and, the most important – to increase the teeth number *Z*, which impacts on the productivity of the milling process proportionally.

With the number of teeth increasing the surface finish and uniformity of the milling process improves.

The shape of the milling cutter teeth should be as:

1) to provide the necessary strength of the teeth;

2) to provide higher number of permissible regrinding;

3) to provide enough volume of the flutes between the teeth to accommodate the chips.

The trapezoid shape (fig. 2.6, *a*) is the simplest to be manufactured, but the tooth is weakened a little, and therefore has a low height and low volume of chip gash. Resharpening of the tooth on the clearance surface, that is in the form of the land with width f = 1–2 mm, reduces the height of the tooth and, thus make it stronger. However, this form of teeth allows a small number of regrinding and is used for the finishing cutters. Moreover, the number of teeth due to their small size can be as large as possible. The gashes in such milling cutters are machined by milling or grinding on CNC machines.

Tooth height is reduced during the regrinding, so the total life of these mills is low as they allow only 6–8 regrindings. Tooth root radius or fillet is taken equal to 0.5–2.0 mm.

Parabolic shape of the tooth (fig. 2.6, *b*) has the highest bending strength, as the tooth back, made as a parabola, provides uniform strength in all sections at the height of the tooth. The disadvantage of this shape is the necessity for each tooth height to have its complex form milling cutter to cut the flute. Therefore, a parabola is often replaced by the arc of a circle with a radius $R = (0.3–0.4)d$ to simplify the shape of the tooth back.

A straight line portion of the rake surface defines the number of permissible regrinding. Regrinding is allowed only on the flank surface (land f). The clearance angle α must be less than the angle α_1 by 10–15^o (α_1 – an angle between the tangent to the parabola at point A and cutting edge plane). Failure to comply with this condition during resharpening leads to great variation in the width of the margins.

Reinforced shape of the tooth (fig. 2.6, *c*) is used for heavy-duty milling instead of a parabolic shape. This tooth has a faceted back, and an increased thickness and height. These teeth are shaped by double milling with angle cutters, which have angles θ_1 = 28–30^o and θ_2. Although the number of operations is doubled, such teeth are easier to make than parabolic. They have a greater number of regrindings and higher strength.

A standard flute milling cutters are used, which cutting edges are straight. During the regrinding the teeth are sharpened on the flank surface at an angle α and with obligatory sparking-out in order to avoid run-out of the cutting edges. Sometimes small cylindrical margins with width f_M = 0.02–0.03 mm are left, which simplify the control of run-out of cutter teeth.

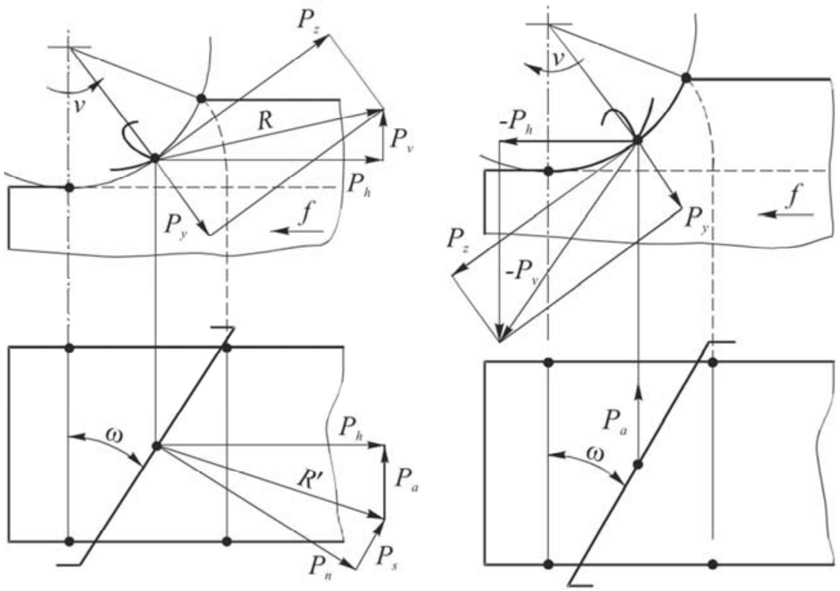
Form-relieved tooth (fig. 2.6, *d*) differs by having a greater thickness, and mainly by the different shape of the flank surface, which is performed on a special operation called relieving, which is used to create clearance angles in all points of the cutting edges. This is achieved by the fact that the radial section of the tooth with profiled section during its rotation is shifted towards the center by the relieving cutter or a grinding wheel. Due to relieving the profile of the cutting edge after the resharpening on the face will be same in all radial sections regardless of its complexity. These are the main advantages of such mills, along with a very simple operation of resharpening. Moreover, the teeth of this form have high strength, and during the process of resharpening the amount of flutes for chips location are increased, which is beneficial to the work of the cutter.

2.4. Cutting force components in milling

2.4.1. Forces in plain milling

Fig. 2.7 shows the schemes of resolution into components of the resultant cutting force *R* acting on the helical tooth of a plain cutter for conventional and climb milling. These schemes have much in common with oblique free turning and shaping. Here, the resultant cutting force *R* is the geometric sum of two components: P_Z and P_Y.

The resultant cutting force *R* can also be resolved into a horizontal component P_h that is directed against the feed direction (fig. 2.7, *a*) and loads machine feed drive, as well as vertical component P_v, which causes deflection of the tool in conventional milling and forces the workpiece down to the machine table in climb milling.



a *b*

Fig. 2.7. Components of the cutting force *R* on a helical tooth of a plain milling cutter: *a* – conventional milling; *b* – climb milling

2.4.2. Forces in face milling

For the full-width milling, when $B = d$, contact angle $\psi_m = 180^o$, $\varphi = 0^o$ and $\omega = 0^o$, fig. 2.8 shows a diagram of the resultant force resolution into the components acting on the tooth of a face mill. Other schemes of the cutting force resolution in face milling are particular cases of this diagram.

Resultant cutting force *R* applied on the cutter tooth can be resolved into the peripheral component P_Z and a radial component P_Y, or into the horizontal component P_h (feed power) and a vertical component P_v.

Teeth of a face mill located on the first (lower) half of the arc of engagement work in up-milling conditions, and teeth in the second half work in climb-milling conditions. At the same time with the milling cutter rotating, the components P_z and P_h change not only in magnitude but also in the direction (fig. 2.8).

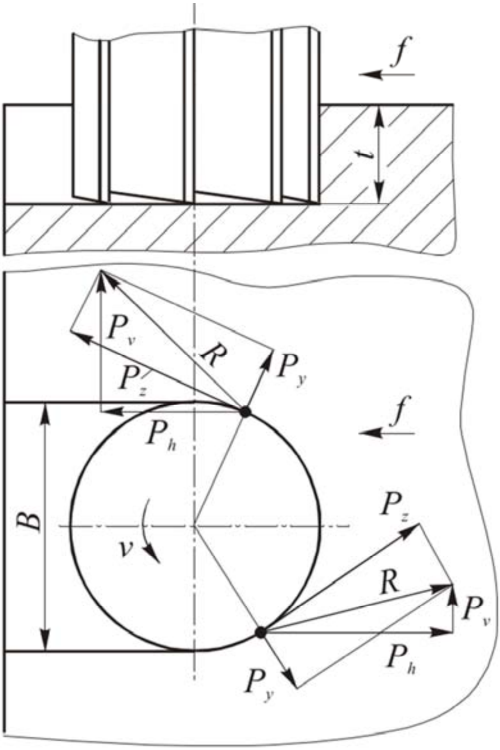


Fig. 2.8. Components of the resultant cutting force *R* in face milling

2.5. Types of Milling Cutters

Milling cutters – are multiple point cutting tools used to machine planes, slots, special shaped surfaces, circular parts, as well as to cut-off parts.

Milling cutters can be classified according to the following criteria:

1) design of the cutting teeth and the method of their sharpening: cutters with chisel-shaped (common) teeth, which are reground on the flank surface, and cutters with form-relieved teeth, which are re-ground on the face;

2) shape and location of the cutting edges relatively to the axis of revolution of the tool: plain, side, face, end, angle and form milling cutters (fig. 2.2);

3) teeth position relatively to the axis of the milling cutter: straight, helical and inclined teeth;

4) method of mounting on the machine tool: shell mills with a hole for the arbor, and end-mills with a straight or tapered shank;

5) mill design: solid, assembled and indexable with inserted teeth, including soldered, welded or mechanically clamped cutting inserts from carbide or super hard materials.

The design features of *plain milling cutters* (fig. 2.9) and *side milling cutters* (fig. 2.10) are the location of the major cutting edges on a cylinder which axis coincides with the axis of rotation of the tool that is parallel to the workpiece surface. Plain milling cutters do not have minor cutting edges, and they work in free cutting conditions. The teeth of side milling cutters, on the contrary, have minor cutting edges positioned on one or both faces. Moreover, in contrast to the plain milling cutters, their diameter is much larger than the cutter length. Both types of milling cutters, as a rule, are shell or arbortype cutter, with a hole and keyways for mounting on an arbor.

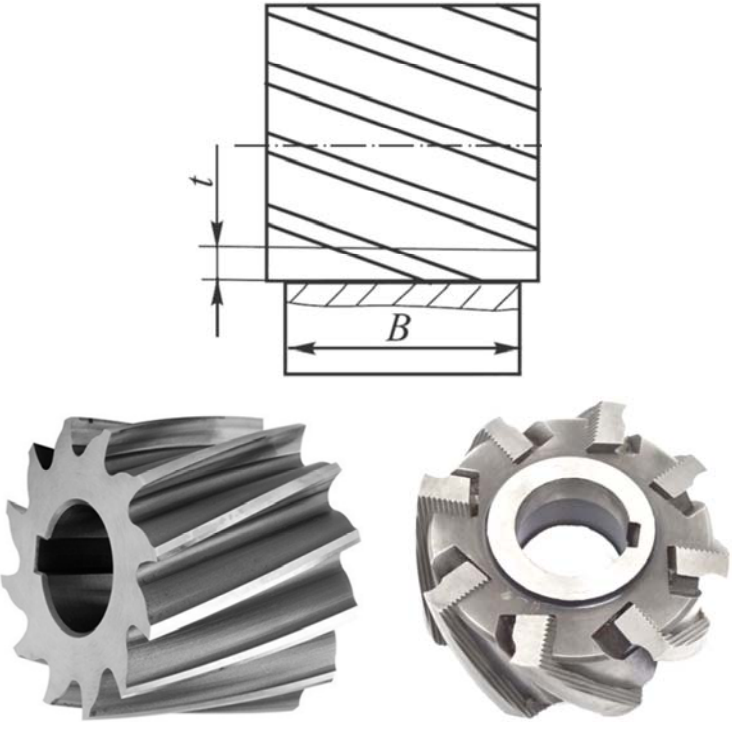
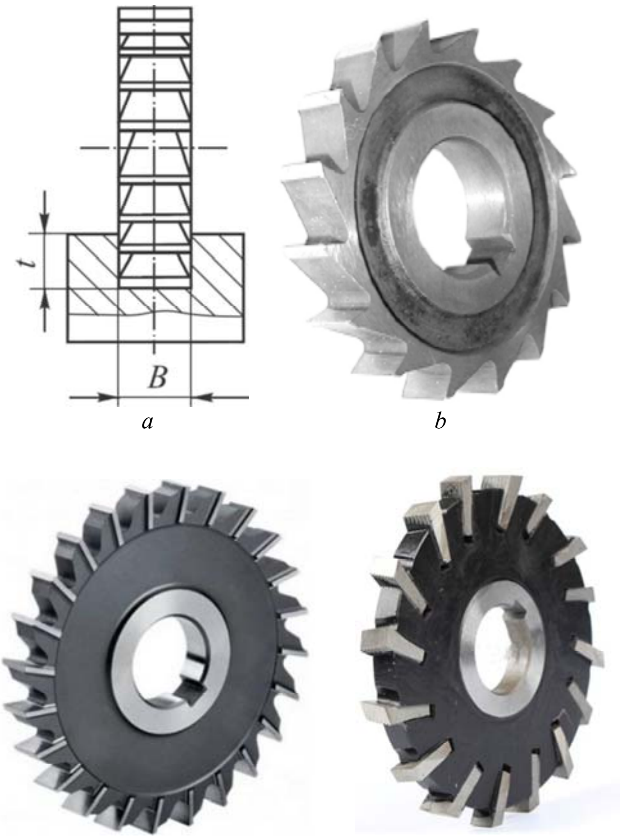


Fig. 2.9. Plain milling cutters



a *b*

c *d*

Fig. 2.10. Side milling cutters:
a – side cutter; *b* – slotting cutter;
c – side cutter; *d* – side cutter with staggered inserts

Teeth of the plain milling cutters are often made helical to reduce variation of cutting forces and vibrations. This gives a rise to unwanted axial component of the cutting force. However, the conditions of chip removal from the cutting area are much better when the helical teeth instead of straight ones are used.

Plain milling cutters with coarse teeth are designed to remove large allowances and particularly effective in the machining of extended surfaces. In order to save high-speed steel the milling cutters of larger diameter are assembled with inserted cutting teeth, and milling cutter bodies are made of structural steel.

Side milling cutters (fig. 2.10), in contrast to the plain milling cutters, are designed to machine narrow surfaces, shoulders, to cut slots, cut off the part, etc. Milling cutters work under difficult conditions of non-free cutting, often accompanied by chatter due to low transverse rigidity of the cutter bodies and unfavorable chip removal from the cutting zone.

The following types of side milling cutters are distinguished: half-side cutter, side cutter, slotting cutter, slitting cutter and cut-off cutters (saws).

Slotting cutters (fig. 2.10, *b*) to side milling cutters in design, but have a shorter length of major cutting edges. Minor cutting edges that are located on the sides are sharpened with the minor cutting edge angle $\varphi_1 = 1–2^{\circ}$. Flutes are cut only on the periphery of the cutter. Slotting cutters are manufactured with a diameter of 50–100 mm and a width of 3–16 mm. Slitting cutters and cut-off cutters (saws) are similar to the slotting cutters in the shape of a tooth and are used for cutting shallow and narrow.

The side milling cutter has teeth on the periphery and on the both sides as well (fig. 2.10, *c*). These cutters can machine, respectively, two or three surfaces of slots or shoulders. They are made with fine teeth for finishing and coarse teeth for roughing. The latter are characterized by removal of large amounts of metal from the deep slots, so they have bigger flutes. The teeth of the mills with small width of cutting edges are straight, or inclined to the axis. The latter provides a more uniform milling, have favorable geometry of the teeth and better chip removal.

Side milling cutters are made with staggered teeth, which allows to create positive rake angles $\gamma > 0$ on the side edges (fig. 2.10, *d*). Regrinding of the cutter decreases its width, thus to avoid this drawback the dual cutters consisting of two halves with a spacing bush between them are also used. Solid milling cutters are manufactured with diameters $d = 63–125$ mm and widths of $B = 6–28$ mm. Assembled cutters with inserted blades have $d = 75–200$ mm and $B = 12–60$ mm. These blades are made of high speed steel and mounted in wedge-shaped flutes with serrations.

Face mills (fig. 2.11) and *end mills* (fig. 2.12) have the axis of rotation perpendicular to the working surface. These cutters, in addition to the major cutting edges located on the periphery, have minor cutting edges located on the end of the cutter at an angle φ_1. The face mills are usually manufactured as shell-type cutters. When the cutter diameter is much smaller than the length, then the cutter is considered to be an end mill.

Face mills (fig. 2.11) are widely used in the machining of flat surfaces, including the stepped surfaces that can't be machined with plain milling cutters. Compared to the latter they provide the following advantages:

1) face mills can accommodate more teeth along the arc of engagement with the workpiece, resulting in greater productivity and more uniform cutting;

2) face mills can be made with rigid, massive bodies, with a reliable mechanical mounting of cutting elements, for example, in the form of carbide indexable inserts;

3) in milling the planes it is possible to achieve a lower roughness due to the large number of minor cutting edges on the face of the mill, which also can be made as wiper teeth with $\varphi_1 = 0$.

Face mills are the most frequently used in metalworking due to these advantages.

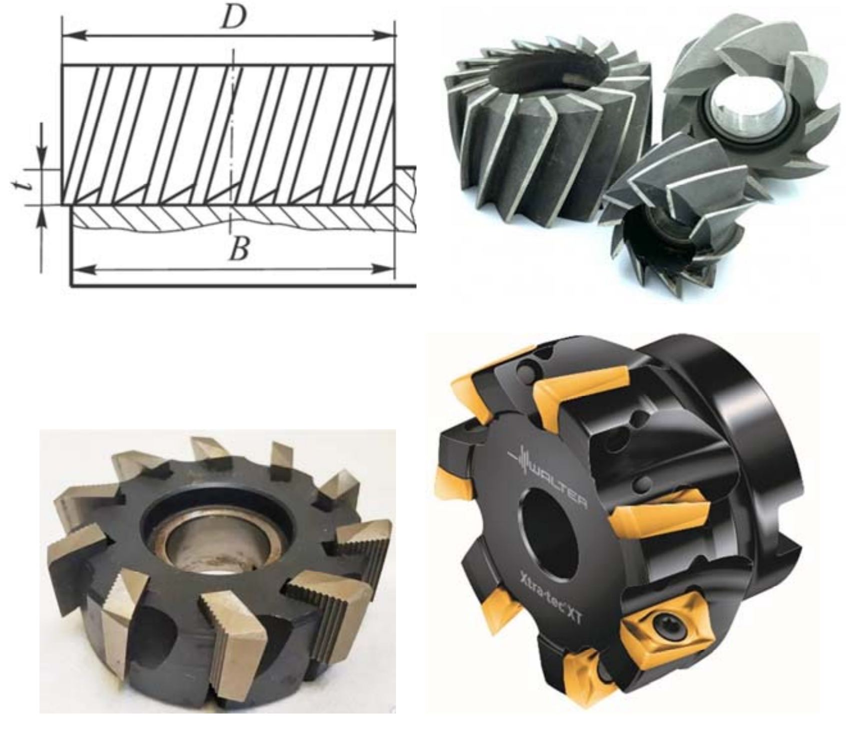


Fig. 2.11. Face milling cutters

The major cutting edge angle of face mills can vary widely, from 90^o and lower. To improve the tool life and productivity the angle φ can be reduced to $45–60^o$ and even up to $10–20^o$.

End mills (fig. 2.12) are used for milling slots, shoulders and for contour machining. The major cutting edges that remove the allowance are located on the periphery, and the minor on the end. The teeth are usually made helical, with a helix angle reaching up to $\omega = 30–45^o$. Such large angle value ω along with large-volume flutes ensures a reliable chip removal out of the cutting zone, even under very restricted conditions of cutting. For this reason, the number of cutting teeth of the end mills is much smaller than that of face mills. However, the decrease in productivity is compensated by increasing the feed per tooth.

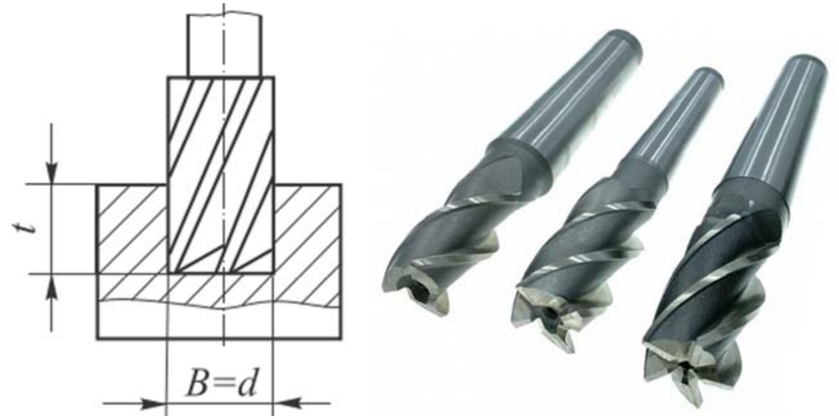


Fig. 2.12. End milling cutters

Fig. 2.12 shows a standard three-fluted, four-fluted, five-fluted end mills. Shanks of such cutters are tapered with Morse taper (d = 14–63 mm), or straight (d = 3–20 mm). Shanks with 7:24 taper are used for mills of larger diameters. Mounting of cutters with straight shank in the machine spindle is made with the collet chucks, tapered shank cutters with an internal thread are mounted with the help of tamper (pull bar) passing through the hollow spindle of the machine tool.

Keyway cutters (fig. 2.13) have two teeth with deep flat or inclined ($\omega = 12–15^o$) flutes and the working length equal approximately to three cutter diameters. One of the cutting edges runs to the center of the cutter to provide axial feed in the milling. The diameter of the cutter web is increased to 0.35*d*, which ensures the maximum rigidity of the cutter.

Fig. 2.13. Keyway cutters

The special feature of the keyway cutter work is that the keyway is cut in several passes. At the end of each pass the keyway endmill plunges to a certain depth with the vertical feed. During plunging the cutting is done by cutting edges located on the end of the cutter.

Mills used for cutting T-slots (fig. 2.14) work under difficult conditions and often failure because of chip jamming. To improve chip removal the mills are provided with staggered teeth positioned at an angle $\varphi_1=1$–2°.

A keyway end mill and a t-slot cutter is a variety of end milling cutters, which are widely used in machining of work tables and bodies of the machine tools.

Angle cutters (fig. 2.15, *a*) and *form milling cutters* (fig. 2.15, *b*) with pointed teeth are similar to side milling cutters in the way of operation. These cutters are manufactured, as a rule, solid or of a shell-type and small-diameter mills are sometimes of a shank-type.

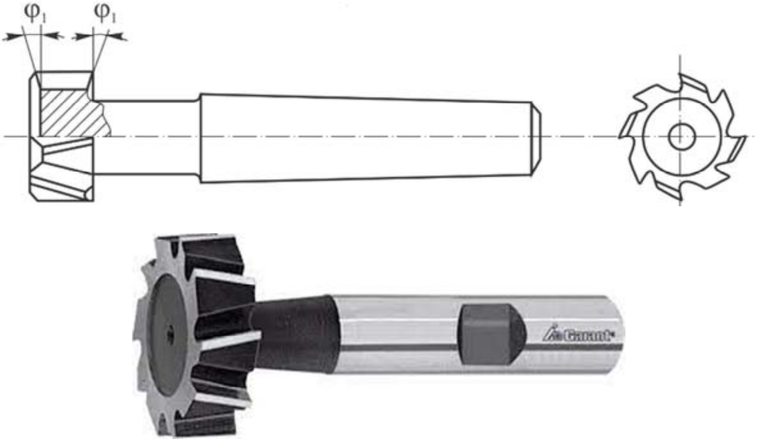
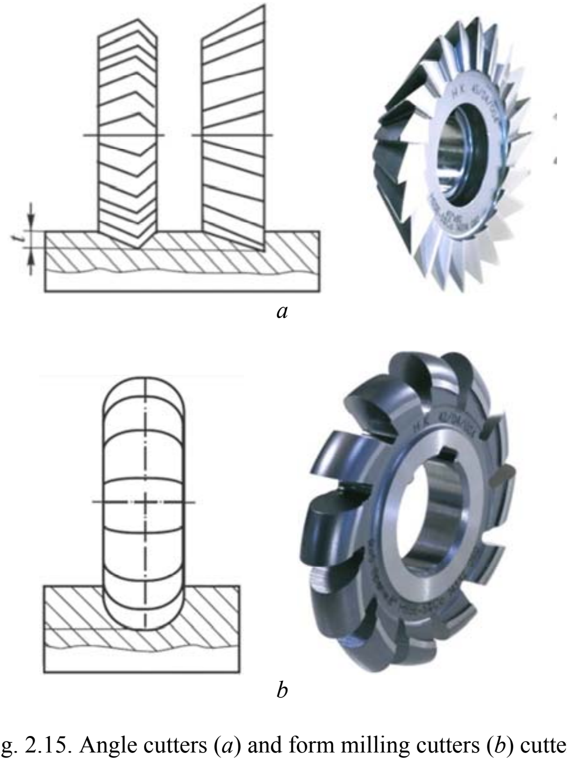


Fig. 2.14. T-slots cutters



a

b

Fig. 2.15. Angle cutters (*a*) and form milling cutters (*b*) cutters

The major cutting edges of single-angle cutters are located on the surface of a truncated cone, while double-angle cutters have cutting edges on the surfaces of two adjacent cones. These cutters are mainly used as a second-order tools applied for cutting flutes in multiple-point cutting tools such as milling cutters, reamers, etc., and also for the milling different flutes, slots and inclined surfaces.

Form milling cutters have teeth with different in shape cutting edges that are located on the form milling cutter periphery. They work as side cutters or angle cutters, and are designed for milling convex or concave surfaces, straight or helical flutes and various profiled surfaces.

References

1. Дечко, Э. М. Резание металлов и режущий инструмент : учебное пособие для учреждений высшего образования по специальностям «Технология машиностроения», «Технологическое оборудование машиностроительного производства» / Э. М. Дечко, М. М. Дечко. – Минск : Вышэйшая школа, 2020. – 287 с.

2. Фельдштейн, Е. Э. Режущий инструмент : учебное пособие для студентов машиностроительных специальностей учреждений, обеспечивающих получение высшего образования / Е. Э. Фельдштейн, М. А. Корниевич, М. И. Михайлов. – Минск : Новое знание, 2007. – 399 с.

3. Кожевников, Д. В. Режущий инструмент : учебник / Д. В. Кожевников [и др.]. – 4-е, изд. – Москва : Машиностроение, 2014. – 520 с.

4. Рыжкин, А. А. Режущий инструмент : учебное пособие / А. А. Рыжкин [и др.]. – Ростов н/Д : Феникс, 2009. – 405 с.

5. Metal cutting / Edward M. Trent, Paul K. Wright. – 4th ed. – 2000. – 464 p.

6. Geometry of Single-point Turning Tools and Drills. Fundamentals and Practical Applications / Viktor P. Astakhov. – 2010. – 584 p.

7. Metal Cutting Theory and Practice / David A. Stephenson, John S. Agapiou.– 3th ed. –2000. – 956 p.

8. Cutting Tool Technology : Industrial handbook / Graham T. Smith. – 2008. – 606 p.

9. Basics of Cutting and Abrasive Processes / Hans Kurt Toenshoff, Berend Denkena. – Springer-Verlag Berlin Heidelberg, 2013. – 404 p.

10. Modern machining technology A practical guide / J. Paulo Davim. – Woodhead Publishing Limited, 2011. – 412 p.

Учебное издание

ЯЦКЕВИЧ Ольга Константиновна
КОЛЕСНИКОВ Леонид Александрович
АЖАР Александр Владимирович

РЕЖУЩИЕ ИНСТРУМЕНТЫ. СВЕРЛЕНИЕ И ФРЕЗЕРОВАНИЕ

Пособие
для студентов специальности
1-36 01 01 «Технология машиностроения»

На английском языке

Редактор *Е. И. Беницевич*
Компьютерная верстка *Е. А. Беспанской*

Подписано в печать 08.12.2022. Формат 60×84 $^1/_{16}$. Бумага офсетная. Ризография.
Усл. печ. л. 2,73. Уч.-изд. л. 2,17. Тираж 100. Заказ 620.

Издатель и полиграфическое исполнение: Белорусский национальный технический университет.
Свидетельство о государственной регистрации издателя, изготовителя, распространителя
печатных изданий № 1/173 от 12.02.2014. Пр. Независимости, 65. 220013, г. Минск.