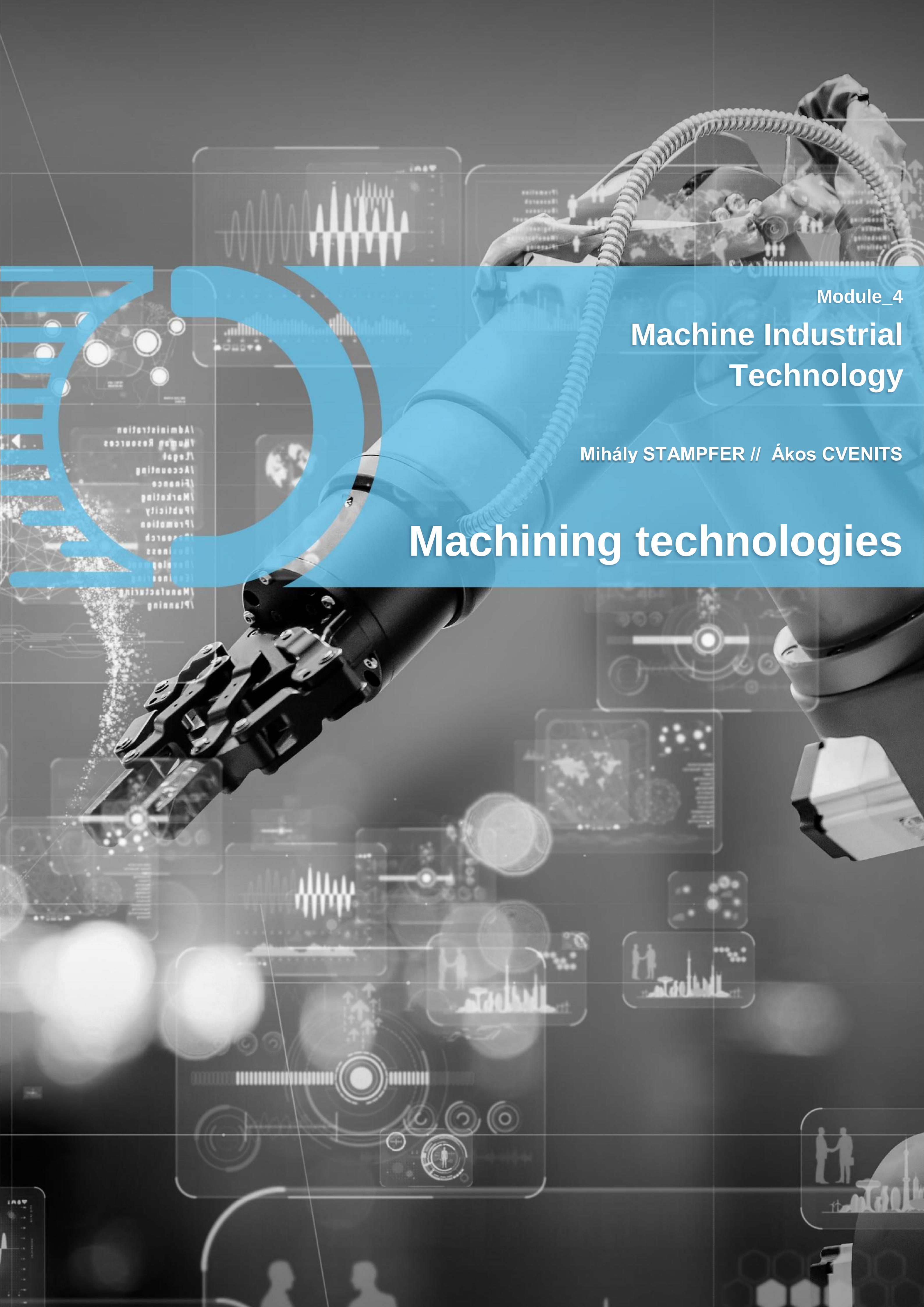


Module_4

Machine Industrial Technology

Mihály STAMPFER // Ákos CVENITS

Machining technologies

Module_4

Industrial Technology

Mihály STAMPFER // Ákos CVENITS

Machining technologies

TABLE OF CONTENTS

1.THEORY OF CUTTING

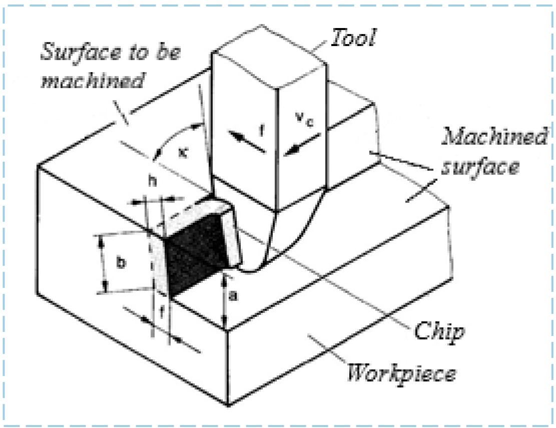
1.1 BASICS OF CHIP SEPARATION

CUTTING is a process of forming materials in which chips are mechanically separated from the workpiece using a wedge-shaped tool.

ELEMENTS AND COMPONENTS of the machining system:

- workpiece
- tool
- chip
- a machine that provides the energy and movement needed to interact with the workpiece and the tool

The model of the cutting process is a single-edged, straight-line tool is shown on the 1-1. illustration.



1-1.Figure Model of the cutting process

The cutting process can be divided into free and fixed cutting. The main features of free cutting are the nexts (1-2. illustration):

- the geometric kinetic conditions are the same at all points on the edge of the tool
- only one edge (main edge) of the tool is involved in cutting.

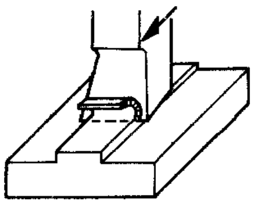


Figure 1-2. Free cutting.

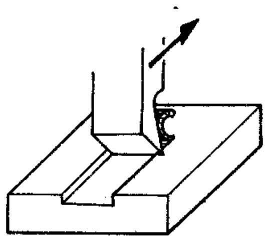
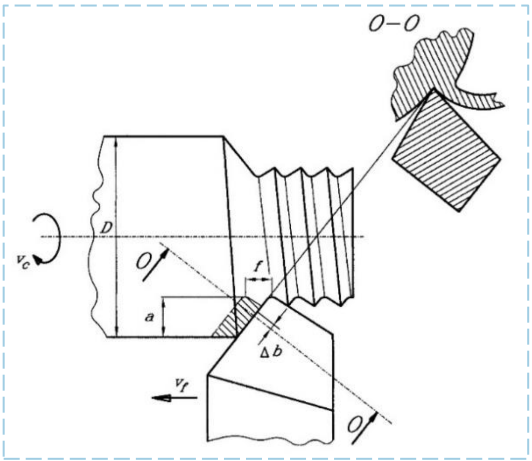


Figure 1-3. Bonded cutting.

In the case of knitted cutting, in addition to the main edge of the tool, the side edge or several edges also participate in the cutting (1-3. illustration). If we consider an arbitrarily short section of the chip width Δb, it can be considered as free cutting, and the edge can be considered as a series of elementary edges (1-4. illustration).



1-4.Figure Return of bonded cutting to free cutting on the example of longitudinal turning

When examining cutting theory issues, it is advisable to follow the principle of “simple to complex” gradation. Therefore, in most cases, the examination of the basic correlations is performed for free cutting.

1.2 OVERVIEW OF THE REQUIRED MOVEMENTS FOR CUTTING

Cutting requires the workpiece and tool to move relative to each other. This movement is a complex movement created by the movement system of the machine and consists of main and auxiliary movements.

The MAIN CUTTING MOTION is the movement in the direction of chip removal, which directly creates chip removal. The main drive of the machine establishes it by moving the workpiece or tool.

The CUTTING SIDE MOVEMENTS, when added to the main movement, ensure repeated or continuous layer separation. There can be movements in the feed direction or grip direction.

The secondary movement in the feed direction (feed) is the displacement that affects the chip thickness to be removed. In addition to the feedrate, in practice the feedrate f is used to characterize the feedrate movement, which is applied to a certain part of the cutting process, e.g. when turning, the movement of the tool during one turn of the workpiece (mm / rev), or during planing, the feed displacement per double stroke (mm / double stroke).

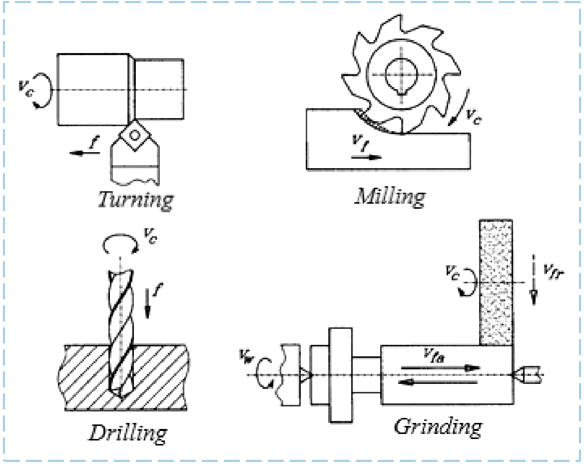
The side movement in the gripping direction usually determines the width of the chip, usually perpendicular to the feed and always intermittent. The magnitude of the displacement is called the depth of cut.

ADJUSTMENT AND AUXILIARY MOVEMENTS are required before starting the cutting process to bring the tool and workpiece into position. For example, before drilling, the tool must be moved above the hole.

1.1.1 CUTTING METHODS AND THEIR MOTION SYSTEMS

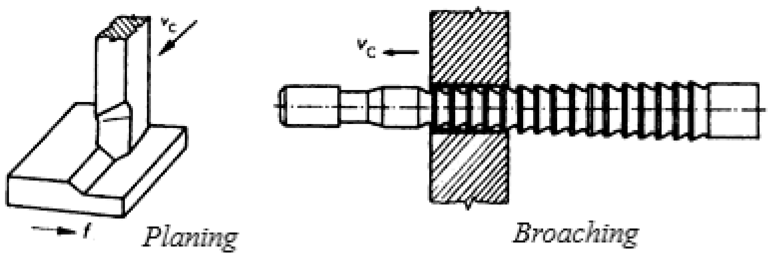
Each cutting mode has one main movement and one or more secondary movements. Depending on the type of main movement, the cutting modes can be divided into two groups (1-5, 1-6 illustration):

- rotary main motion cutting processes (esztergálás, fúrás, marás, köszörülés)



1-5. Figure Rotary main motion cutting processes.

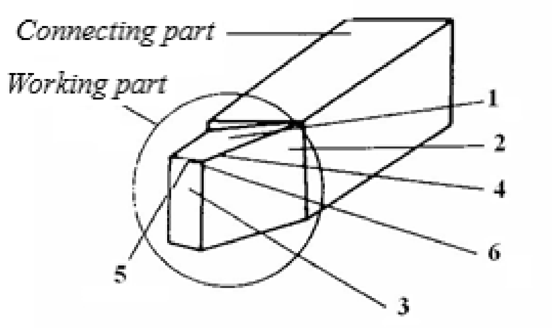
- straight line main motion procedures (planing, chiselling, hollowing, sawing)



1-6. Figure Straight line main motion procedures.

1.3 THE GEOMETRY OF THE CUTTING EDGE OF THE TOOLS

According to their structural design, the cutting tools are diverse, but the characteristic surfaces and angles are present in all tools. Therefore, the interpretation of the concepts related to edge geometry is done on the simplest shaped, single edged tool.



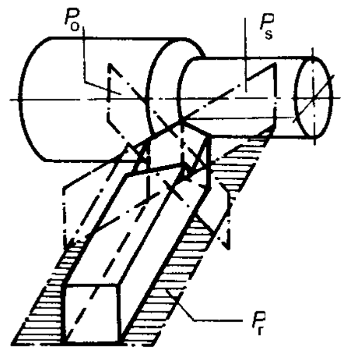
ELEMENTS OF THE WORKING PART OF THE TOOL (Figure 1-7):

1 – face
2 – major flank
3 – minor flank
4 – major edge
5 – minor edge
6 – corner

1-7. Figure Parts, surfaces and edges of the tool.

THE TOOL PLANES (REFERENCE PLANES) (Figure1-8):

- **P_r** – base plane, v_c perpendicular to its direction
- **P_s** –edge plane, includes the cutting edge and is perpendicular to P_r
- **P_o** –orthogonal plane perpendicular to P_r and P_s.



1-8. Figure Tool planes

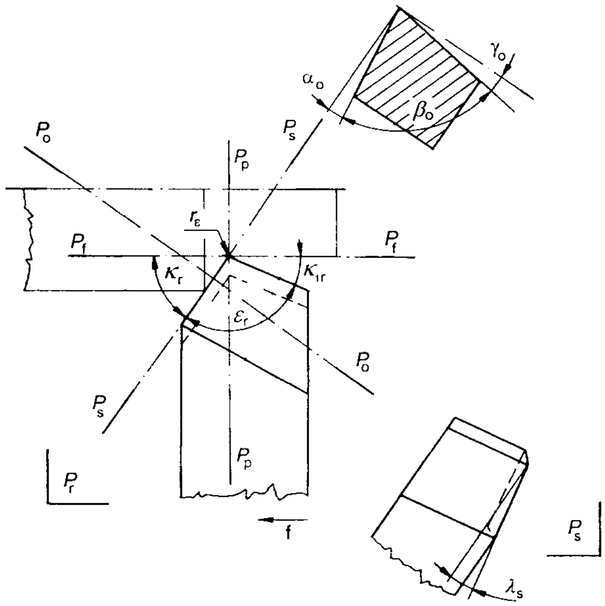
THE FOLLOWING REFERENCE PLANES ARE NOT SHOWN IN FIGURES 1-8:

- **P_f** –working plane (assumed) perpendicular to P_r and parallel to the feed direction
- **P_p** –axis plane, perpendicular to P_r and perpendicular to P_f and passing through the tool tip.
- **P_n** – edge normal plane perpendicular to the cutting edge.

TOOLANGLES

The tool angles are determined using the reference planes described above. In the base plane view of the tool, the view angles are interpreted, while in the section of the orthogonal plane, the section angles are interpreted. (Figure 1-9):

α_o - back angle (angle between the back panel and the tool edge plane)
β_o - wedge angle (angle between the front panel and the back panel)
γ_o - face angle (angle between the face and the reference plane), positive if $\alpha + \beta < 90^o$
ε_r - apex angle (angle between main edge and side edge)
κ_r - main edge position angle (angle between the main edge and the working plane)
κ_{1r} - side edge angle (angle between side edge and working plane)
λ_s - deflection angle (angle between the main edge and the reference plane of the main edge).



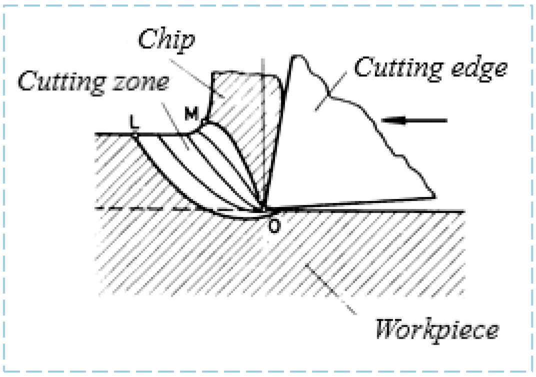
1-9. Figure Tool angles

1.4 THE PROGRESS OF CHIP FORMATION

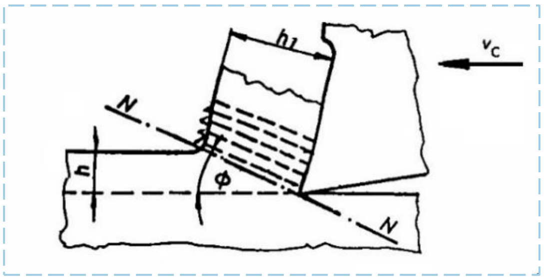
The process of chip formation is easiest to study under free-cutting conditions.

The cutting process takes place in a so-called cutting tube or cutting zone between the tool, the workpiece and the chip.

A chip is a piece of detached chip directly connected to the workpiece and includes the part of the cutting wedge in contact with the former [1] (Figure 1-10). With its edge and face, the tool penetrates the material of the workpiece, plastically deforms it in front of the edge to such an extent that the material can no longer withstand the generated stress, its structure breaks and the chips separate from the workpiece.



1-10Figure. Chipper (cutting zone).



1-11. Figure Shear plane.

There is a complex and very complex stress state in the chipboard. With some simplification, it is acceptable that the main stress during cutting is shear, and thus the cutting zone can be traced back to a shear plane (Figure 1-11). The material particles (chip elements) shear and slip in a specific plane (shear plane). The position of the shear plane is determined by the shear angle Φ. The sheared elements are partially or completely welded, depending on the conditions (workpiece material, tool characteristics, cutting parameters).

During cutting, the thickness of the deformed layer changes (Figure 1-11). The thickness of the chip will be greater than the thickness of the theoretical layer to be deposited. Accordingly, based on the continuity of the cutting, the length of the chip decreases, while observations show that the width of the chip practically does not change (deformation in the plane).

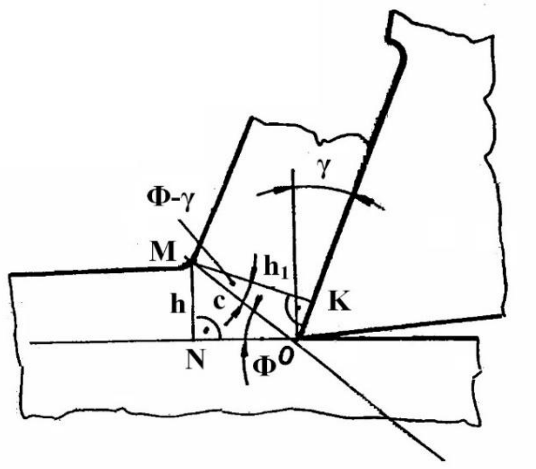
The ratio of the layer and chip thickness to be deposited is the deformation factor ξ. It can also be written as the ratio of the length of the layer to be deposited to the length of the chips and the ratio of the cross-sections.

$$\xi = \frac{h_1}{h} = \frac{l}{l_1} = \frac{A_1}{A}$$

The deformation factor can be determined experimentally

1.4.1. CORRELATION BETWEEN SHEAR ANGLE, DEFORMATION FACTOR AND FACE ANGLE

Using the notation in Figures 1-12, the following relation can be written using the sine theorem:



1-12Figure. Correlation between shear angle, deformation factor and face angle

$$\frac{h_1}{\sin[90-(\Phi-\gamma)]}=\frac{c}{\sin 90}$$

$$\frac{h}{\sin\Phi}=\frac{c}{\sin 90}$$

$$\frac{h_1}{\cos(\Phi-\gamma)}=\frac{h}{\sin\Phi}$$

$$\frac{h_1}{h}=\frac{\cos(\Phi-\gamma)}{\sin\Phi}=\xi$$

$$\frac{\cos\Phi\cos\gamma+\sin\Phi\cdot\sin\gamma}{\sin\Phi}=\xi$$

$$\mathrm{tg}\,\Phi=\frac{\cos\gamma}{\xi-\sin\gamma}$$

Based on the above relation, it can be concluded that decreasing the value of the deformation factor ξ leads to increasing the value of the shear angle.

Knowing the shear angle, the shear surface can be calculated and then the shear force through it

$$F_S=\tau_S\cdot A_S$$

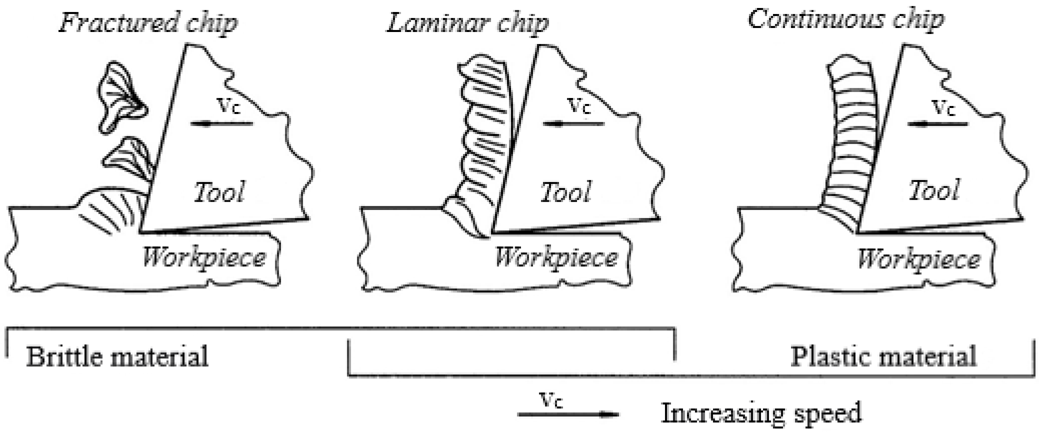
τ_S – the shear strength of the material to be machined [MPa],

$A_s=\frac{A}{\sin\Phi}=\frac{h\cdot b}{\sin\Phi}$ the size of the sheared surface [mm^2].

It is easy to see that the position of the shear plane and the size of the shear angle affect the size of the shear surface and thus the amount of shear force required. The smallest shear surface and thus the minimum energy requirement for cutting would be Φ = 45°. However, under the real conditions of chip removal, this ideal case cannot be achieved, the shear angle is well below 45°.

CHIP TYPES

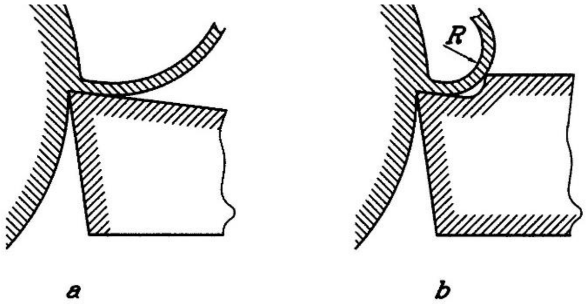
Depending on the material of the workpiece, the cutting factors and the tool geometry, different types of chips are formed (Figure 1-14): elementary or broken chips, temporary or plate chips, and continuous chips.



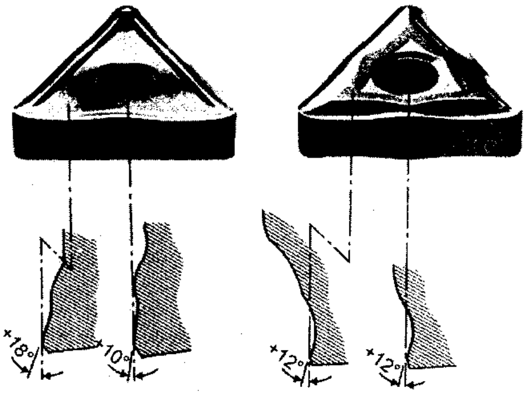
1-14Figure. Chip types

The chip shape is favorable, has a good machined surface quality and can be easily removed from the machine tool workspace.

In the case of continuous chip materials, artificial chip breakers are used, especially when roughing. The essence of this is that an obstacle is placed in the path of the falling chips, which leads to a strong deformation and the chips are broken into pieces or twisted into a dense bundle (Figures 1-15 and 1-16).



1-15Figure. Chip run on the face
a) without the use of a chip breaker, b) using a chip breaker



1-16. Figure Chipbreaker designs on carbide inserts (Sandvik)

1.5 THE DEFINITION OF THE CUTTING FORCE

Chip separation is caused by force. By cutting force is meant the effect of the workpiece on the tool. A force of the same magnitude and line of action but opposite to the cutting force acting on the workpiece is called the FORMING FORCE.

1.5.1 THE THEORETICAL DEFINITION OF THE CUTTING FORCE

In the case of free orthogonal cutting, a planar force system is formed (Figure 1-17). The theory of Ernst and Merchant is used to determine the cutting force theoretically.

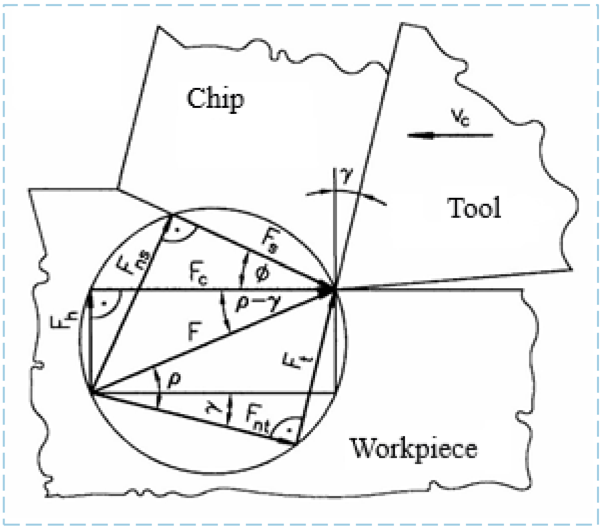


Figure 1-17. A Merchant modell.

THE RESULTING CUTTING FORCE F should be broken down into characteristic perpendicular components. The Merchant model uses three force resolutions:

1.	F_c cutting force, parallel to the cutting speed, F_h is the force in the direction of chip thickness, deepening force, perpendicular to the direction of the main cutting force.
2.	F_t is the frictional force on the front of the tool, F_{nt} is the normal force on the front of the tool
3.	F_s is the shear force in the shear plane, F_{ns} is the normal force acting on the shear plane

The value of the half-cone angle of friction ρ is relatively high (22° – 45°), $tg\rho = \frac{F_t}{F_{nt}} = \mu$.

THE MAIN COMPONENT IN TERMS of the characterization of the cutting process and the energy calculations is the main cutting force. The determination of the force components and the resulting force can be performed starting from the shear force Fs, because only this component can be determined by the relations of plasticity [6]. The force components and the resulting force form right-angled triangles, so the following relationships can be written based on the figure:

$$F = \frac{F_s}{cos[\Phi + (\rho - \gamma)]}; \qquad F_c = F_s \frac{cos(\rho - \gamma)}{cos[\Phi + (\rho - \gamma)]}$$

$$F_h = F_s \frac{sin(\rho - \gamma)}{cos[\Phi + (\rho - \gamma)]}; \qquad F_t = F_s \frac{sin\,\rho}{cos[\Phi + (\rho - \gamma)]}$$

$$F_n = F_s \frac{cos\,\rho}{cos[\Phi + (\rho - \gamma)]}$$

The shear force can be written as: $F_s = \tau_s \cdot A_s = \dfrac{A \cdot \tau_s}{\sin\Phi}$

With this, the end of the main cutting force and the deepening force can be written in the following form:

$$F_c = A \cdot \tau_s \frac{\cos(\rho - \gamma)}{\sin\Phi \cdot \cos[\Phi + (\rho - \gamma)]} \quad \text{and} \quad F_h = A \cdot \tau_s \frac{\sin(\rho - \gamma)}{\sin\Phi \cdot \cos[\Phi + (\rho - \gamma)]} .$$

The determination of the shear angle Φ and the half-cone angle ρ is difficult, so the theoretical determination of the cutting force is not used in practical applications.

1.5.2 DEFINITION OF THE CUTTING FORCE WITH EXPERIMENTAL FORMULAS

As not all quantities in the theoretical context are known, the cutting force is determined in practice by measurement. By processing the measurement results, empirical relationships determining the components of the cutting force can be obtained.

In practice, two types of empirical formulas have spread.

1. DEFINITION OF THE CUTTING FORCE BASED ON SPECIFIC CUTTING FORCE

$$F_c = k_c \cdot A = k_c \cdot b \cdot h$$

$k_c = \dfrac{F_c}{A} \left[N / mm^2\right]$ - specific cutting force, force required to separate unit (1mm²) theoretical chip cross section

There are several methods for determining the value of the specific cutting force, of which the Kinzle method is used today. Based on his research, Kinzle (1952) suggests the following relation to determine the specific cutting force:

$$k_c = \frac{k_{c1 \cdot 1}}{h^m}$$

Ahol: $k_{c1.1}$ the main value of the specific cutting force $\left(A = b \cdot h = 1 \cdot 1 = mm^2\right)$

m their values are given in 3.1 Table.

For accurate calculation, secondary influencing factors are taken into account with correction factors. Thus, the main cutting force can be calculated according to the following formula:

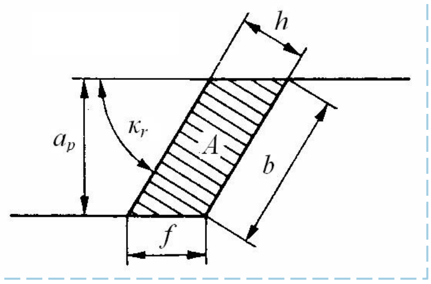
$$F_c = k_{c1.1} \cdot b \cdot h^{1-m} \cdot K_F \text{ [N].}$$

the K_F the aggregate modification factor.

The values of chip width and chip thickness in the formula must be calculated on the basis of the technological data as shown in Figure 1-18.:

$$h = f \cdot \sin \kappa_r$$

$$b = \frac{a_p}{\sin \kappa_r}$$



1-18Figure. Chip cross section parameters

1.1.Table Main values of specific cutting force (Following Degner)

The material of the workpiece		m	$k_{c1.1}$ [N/mm²]	$1\text{-}m_f$	$k_{f1.1}$ [N/mm²]	$1\text{-}m_p$	$k_{p1.1}$ [N/mm²]
MSZ-EN	DIN						
S235	St37	0,17	1780				
E295	St50	0,26	1990	0,2987	351	0,5089	274
E360	St70	0,3	2260	0,3835	364	0,5067	311
C15E	Ck15	0,22	1820	0,1993	333	0,4648	260
C45E	Ck45	0,14	2220	0,3248	343	0,5244	263
C60E	Ck60	0,18	2130	0,2877	347	0,5870	250
15CrMo5	15CrMo5	0,17	2290	0,2488	290	0,4430	232
16MnCr5	16MnCr5	0,26	2100	0,3024	391	0,5410	324
	17CrNi6	0,30	2260	0,275	326	0,5352	247
20MnCr5	20MnCr5	0,25	2140	0,3190	337	0,4778	246
30CrNiMo8	30CrNiMo8	0,20	2600	0,3844	355	0,5657	255
34CrMo4	34CrMo4	0,21	2240	0,3190	337	0,3715	237
	37MnSi5	0,2	2260	0,3620	259	0,7432	277
42CrMo4	42CrMo4	0,26	2500	0,3295	334	0,5239	271
	50CrV4	0,26	2220	0,2345	317	0,6106	315
EN-GJL 200	GG20	0,25	1020	0,3010	240	0,5400	178
EN-GJL 250	GG25	0,26	1160	0,3020	251	0,5410	190
EN-GJS 600	GGG-60	0,17	1480	0,2400	290	0,5657	240
Bronze		0,17	1780				
Brass		0,18	780				
Aluminum Alloy		0,25	640				

Comment: The data in the table are made with a sharp carbide tool with an edge geometry of γ=6°, γ_o =2°, α_o =5° κ_r =70° for materials with long chips (steel) for turning. The data are valid in the speed range v_c=20…600 m/min and chip thickness h=0,05…2,5 mm.

2. DEFINITION OF THE CUTTING FORCE IS EXPONENTIAL FORMULA

In practice, the determination of the value of the cutting force is most often determined by an empirical power formula containing technological data.

$$F_c = C_F \cdot f^{x_F} \cdot a_p^{y_F} \cdot K_F$$

where the:

C_F force constant, which depends on the material quality, is determined experimentally and is actually the force required to separate the chip cross-section *a = 1mm* and *f = 1mm*.

x_F , y_F are the exponents of chip thickness and chip width, which were also determined experimentally (1.4. Table). The following values may be used for cutting cast iron and steel: $x_F \approx 0{,}75$; $y_F \approx 1$.

K_F aggregated modification factor that includes deviations due to operating conditions other than the experimental conditions.

1.5.3 DEFINITION OF CUTTING FORCE IN THE CASE OF FIXED CUTTING

There is a more complicated spatial force effect than bonded cutting. The characteristic force components in the spatial model are shown in Figure 3-19. As previously defined, the force components F_c and F_h lie in the orthogonal plane (P_o).

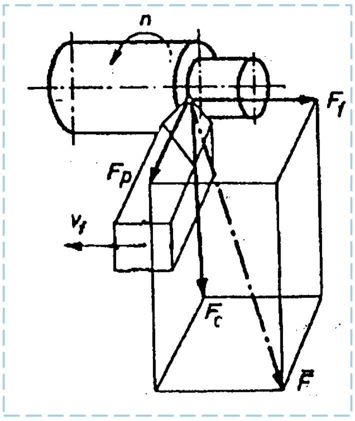


Figure 1-19. Cutting force for fixed cutting.

Assuming a small peak radius r_ε and a deflection angle $\lambda \approx 0$, the passive force F_p and the feed force F_f in the direction of the chip thickness F_h can be determined during turning (Figure 1-20.).
$F_f = F_h \sin\kappa_r$ and $F_p = F_h \cos\kappa_r$.
Thus the resulting cutting force:

$$F = \sqrt{F_c^2 + F_p^2 + F_f^2}$$

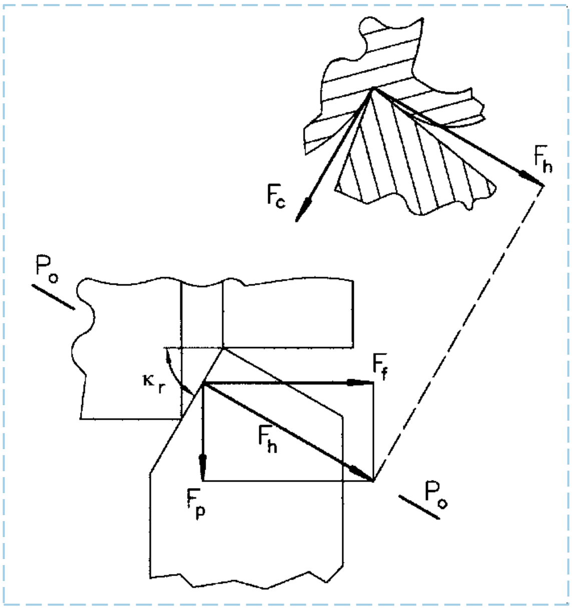


Figure 1-20. Resolution of the force in the direction of the chip thickness [5]

The force components F_p and F_f can also be expressed in analytical form similar to F_c:

$$F_p = C_{Fp} \cdot f^{x_{Fp}} \cdot a_p^{y_{Fp}} \cdot K_{Fp}$$

$$F_f = C_{Ff} \cdot f^{x_{Ff}} \cdot a_p^{y_{Ff}} \cdot K_{Ff}$$

In practice, however, these forces are calculated from the main cutting force using proportionality factors. The following relationships can be used for rough turning:

$$F_p \approx (0{,}4 - 0{,}5) F_c ; \quad F_f \approx 0{,}25 \cdot F_c$$

1.5.3.1 SECONDARY FACTOR INFLUENCING THE VALUE OF CUTTING FORCE

It can be seen from the above that the magnitude of the cutting force is mainly influenced by the material quality of the workpiece, the thickness of the chip and the feed and the width and depth of cut. The magnitude of the force is also affected by a number of so-called secondary factors. The effect of these is taken into account by introducing modifying factors.

The C_F force constant and the exponents of chip thickness and chip width x_F , y_F were also determined experimentally (1.2 Table), so their tabular values are in fact only accurate for cutting under experimental conditions. The operating conditions for cutting usually differ from this and this is taken into account by the aggregate correction factor K_F:

$$K_F = K_{Fsz} \cdot K_{Fv} \cdot K_{F\gamma o} \cdot K_{F\kappa} \cdot K_{Fr} \cdot K_{FVB} \cdot \ldots$$

Modifying factors:

K_{Fsz} tool material modification factor

$K_{F\gamma o}$ face angle modification factor

$K_{F\kappa}$ main edge placement angle modification factor

K_{Fv} speed modifier factor
K_{FVB} tool wear factor
K_{Fr} peak radius factor

Their recommended values can be found in the literature [2], [5], [6], [18].

1.6 THE CUTTING PERFORMANCE

The power required for cutting is the scalar product of the resulting cutting force and the resulting cutting speed

$$P = \vec{F} \cdot \vec{v} = \left[F_c, F_f, F_p\right] \cdot \begin{bmatrix} v_c \\ v_f \\ v_p \end{bmatrix} = F_c \cdot v_c + F_f \cdot v_f + F_p \cdot v_p$$

It should be noted that there is no displacement in the passive direction, so $v_p \equiv 0$, and the feed rate is much lower than the cutting speed (v_f is about 0.1… 1% of v_c), so its effect on performance is negligible. Thus the cutting power

$$P_c = F_c \cdot v_c$$

Machine performance is slightly higher than this, as losses within the machine must also be taken into account:

$$P_m = \frac{P_c}{\eta}$$

where $\eta \approx 0{,}7$ is the efficiency of the machine.

1.2. Table Values of force constants and correction factors for turning with a carbide tool. (Following Angyal, Dobor, Palásti, Sípos)

Worked material	Rm [N/mm2	CF	CFp	CFf	xF	yF	xFp	yFp	xFf	yFf
Steel	300…400	1480	270	225	0,75	1	0,75	0,9	0,65	1,2
	400…500	1630	440	380						
	500…600	1755	665	500						
	600…700	1900	935	650						
	700…800	2140	1230	800						
	800…900	2350	1580	980						
Gray cast iron	HB									
	140…160	840	340	150	0,75	1	0,75	0,9	0,65	1,2
	160…180	890	395	170						
	180…200	950	460	190						
	200…220	1010	520	210						
	220…240	1060	590	240						
	240…260	1110	645	265						

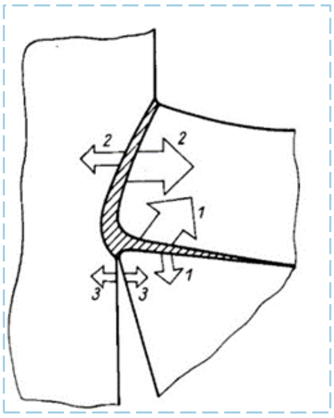
Modification factor for face angle	γo	-5	0	+5	+10	+15	+20
	KFγ	1,23	1,13	1,06	1	0,94	0,89
	KFpγ	1,8	1,55	1,25	1	0,8	0,64

	KFfγ	2,1	1,63	1,28	1	0,78	0,6
Modification factor for main edge placement angle	κr	20	30	45	60	75	90
	KFκ	1,16	1,08	1	0,98	1,03	1,08
	KFpκ		1,63	1	0,71	0,54	0,44
	KFfκ		0,7	1	1,27	1,51	1,82

1.7 THERMAL PHENOMENA IN CUTTING

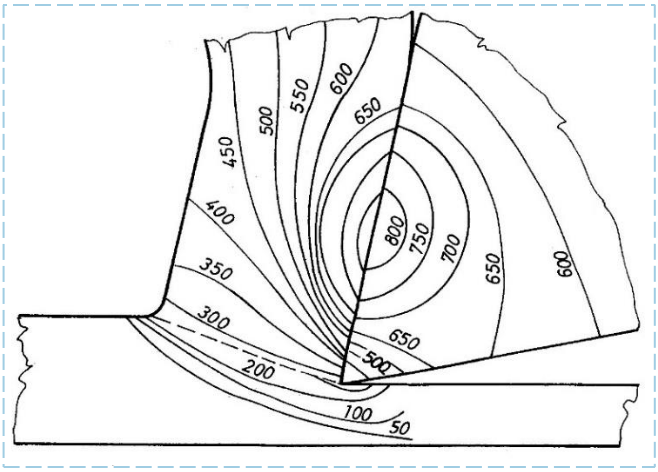
During cutting, the mechanical energy used is almost completely (99.5%) converted into heat. Simplifying the processes, we distinguish three heat sources (Figure 1-21).:

1. The shear zone (plastic deformation)
2. The face of the tool and the location of the friction of the chips
3. The location of the friction between the back surface of the tool and the machined surface



1-21. Figure Heat sources in the cutting zone

The heat generated is distributed between the workpiece, the chips and the tool, increasing the temperature of all three elements. 75 - 85% of the heat is transferred to the chips. The maximum temperature is in the contact zone with the tool. As a result of the amount of heat leaving the tool, its temperature increases significantly, so its hardness decreases and its wear becomes faster.



1-22Figure. Temperature distribution in steel turning [3]

The cutting temperature is the average temperature of the cutting tube. This can only be determined by complex measurement methods. A typical temperature distribution is shown in
Figure 1-22. for turning steel with a carbide tool. Factors influencing the temperature:

- workpiece and tool material
- cutting parameters v, a, f
- the edge geometry of the tool
- application of cooling and lubrication

Of the cutting parameters, the cutting machine has the greatest influence.

1.7.1 COOLING-LUBRICATION DURING CUTTING

The cutting temperature can be significantly reduced and stabilized by the use of coolants.

Their effect is as follows:

- reducing friction and wear through it, improving the quality of the machined surface
- reducing the power and energy required for cutting
- reducing workpiece deformation
- help remove chips (especially important in hole machining)
- protection of the machined surface from environmental damage (corrosion).

The effectiveness of the application of the coolant depends largely on the ability to deliver the fluid to the friction surfaces. There are basically two cooling-lubrication methods:

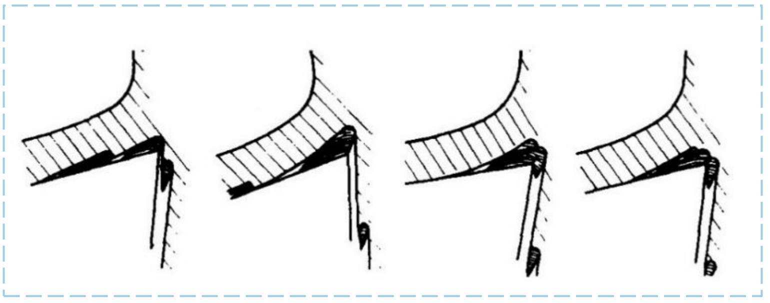
1. FLOOD COOLING, in which the jet of liquid is directed to the chip breaker from the outside through pipes or a cooling channel built into the tool. The disadvantage of this method is that the liquid is scattered and interferes with visual observation.
2. FOG COOLING, only for water-based liquids. Refrigerant sprayed on fine particles can be delivered to other locations. Requires extraction equipment, limited cooling capacity, but allows visibility of the work area.

COOLANT LUBRICANTS CAN BE DIVIDED INTO THREE GROUPS:

I. Artificial chemical, synthetic materials. They have an excellent cooling effect, a very long disintegration time and are therefore environmentally unfavorable.
II. Emulsions. Oils dispersed in water, oil content 5% - 10%. An emulsifier (trisalt) is required to mix the oil droplets. Increasing the amount of oil and the addition of sulfur and phosphorus (EP additives) increase the lubricity. The water provides good cooling capacity. It needs to be replaced after a while because it starts to break down under the influence of bacteria.
III. Mineral and vegetable oils. They provide good lubrication but are relatively expensive.

1.7.2 EDGE OVERLAY FORMATION

Due to the high pressure and high temperature near the main edge of the tool, hard loading and edge overlay (edge helmet) from the machined material can occur (Figure 1-23). Once the height of the edge appliance reaches a certain value, it breaks into pieces and leaves partly with the chips and partly with the workpiece. It should be considered harmful because it changes the edge geometry of the tool, increases the cutting force, and degrades the quality of the machined surface. In most cases, it can be eliminated by increasing the speed.



1-23. Figure Edge formation

1.8 TOOL MATERIALS

By tool material we mean the material of the active, cutting, working part of the tool, which is subjected to mechanical, thermal and chemical stresses during operation.

The materials of the cutting tools can be divided into the following groups:

- tool steels
- carbides
- ceramics
- super hard tool materials

1.8.1 TOOL STEELS

Three other groups of tool steels are distinguished:

Non-alloy tool steels, low-alloy tool steels and *high-speed steels*. Of these, only high-speed steels are now used for cutting.

High-speed steels are high-alloy steels with tungsten, molybdenum, vanadium, cobalt and chromium as the main alloying elements. By hardening, a hardness of 62… 68 HRC can be achieved, which is kept up to about 600 ° C. Due to this, they allow cutting at an increased speed compared to tool steels (vc = 30-50 m / min) and hence the name high-speed steel.

High-speed steel is characterized by high toughness and high strength to dynamic forces. They are easy to sharpen, the most common material for shaped tools.

1.8.2 CARBIDES

Carbides are alloys produced by powder metallurgy from tungsten, titanium, tantalum and niobium metal carbides (WC, TiC, TaC, NbC) and cobalt or nickel binders. Powder metallurgy is a production method in which metal powders are compacted individually or with other metal and alloy powders and finally ignited below the melting point of the main constituent (1400 °C). Several variants are being developed. Increasing the proportion of carbides increases hardness but decreases strength and vice versa. The fineness of the particles (0.6… 0.9 µm) simultaneously increases the hardness and strength of the carbide. This feature is even more pronounced for ultrafine particles (0.5… 0.6 µm). DIN ISO 513 has introduced a uniform marking system to facilitate navigation between the various possible carbides. There are three main cutting groups, which are further divided into application groups. The three major groups and their main characteristics are as follows:

- tungsten carbide based (single carbide) carbides; ISO designation K, suitable for cutting brittle materials
- titanium carbide-tungsten carbide based (two carbide) carbides; ISO designation P, suitable for cutting steel
- carbides based on titanium carbide-tantalum carbide-tungsten carbide (three carbides); ISO designation M, general purpose

The critical temperature of carbides is 750 – 900 °C, so a significantly higher cutting speed can be used than with high-speed steel tools (v_c = 40 – 300 m/min).

CERMET

Cermets are hard metals that use titanium carbides instead of tungsten carbide and also contain titanium-based hard ceramic particles. The name also comes from CERamic – METal. The components of the particle mixture are titanium carbide (TiC), titanium carbonitride (Ti (C, N)), titanium nitride (TiN), molybdenum carbide (Mo2C). Nickel or molybdenum are used as binders. It has a lower strength than fine-grained carbide but a higher heat hardness. When machining steel, the applicable cutting speed range is v_c = 80 – 500 m/min.

1.8.3 CERAMICS

Ceramics as a cutting material have been developed in the hope of a possible increase in cutting speed. Today's ceramics are an unbound mixture of oxides (Al_2O_3), carbides (TiC) and / or nitrides. They are even harder than hard metals and retain this property above 1000 °C, but their toughness and strength are inferior to hard metals. They can be divided into three groups:

- pure ceramic (white): pure alumina (Al_2O_3) and 10 - 20% zirconium oxide (ZrO)
- mixed ceramic (black): 10 - 40% TiC with alumina
- silicon nitride (gray ceramic, silinite)

They are mainly used for high-speed finishing. They are characterized by low toughness, sensitivity to thermal fatigue, shock-like heat effects, and poor thermal conductivity. Coolant must not be used.

1.8.4 SUPER HARD TOOL MATERIALS

Materials approaching the properties of natural diamonds are included in this group.

CBN, cubic boron nitride. It has an extreme heat hardness (2000 ºC) combined with high abrasion resistance. It is used in the machining of relatively brittle hardened steels. It is used in large quantities in abrasive tools.

PCD, polycrystalline diamond. This is the hardest material. Resistant to abrasive wear. Not suitable for machining steel. Its application is favorable in the case of non-ferrous metal alloys, plastics and wood.

1.8.5 COATED TOOLS

By coating carbide or high-speed steel tools, the cutting speed can be significantly increased. The coating can be in several layers, the thickness of each layer is 5 - 10 μm. The surface layer of nitrides, carbides and oxides is responsible for wear protection. Significantly reduces friction between the workpiece and the tool. The following materials can be used as coatings:

- TiN (titanium nitride), reduces the coefficient of friction, good heat resistance, very good resistance to back wear, good adhesion to the base metal
- TiC (titanium carbide), excellent back wear resistance, adhesion to base metals. Used as a base layer for multilayer coatings
- Al_2O_3 (ceramic), good heat resistance, hard, poor bonding to base metals. It is significantly widespread as a multilayer coating.

1.9 TOOL WEAR AND SERVICE LIFE

Abrasion is the wear of the cutting wedge under the effect of mechanical and thermal stresses associated with the cut. Complicated processes take place in the cutting shaft, very high pressure (2000 bar) and high temperatures prevail at the cutting edge. Under these conditions, friction occurs between the chip and the face and between the back of the tool and the machined surface.

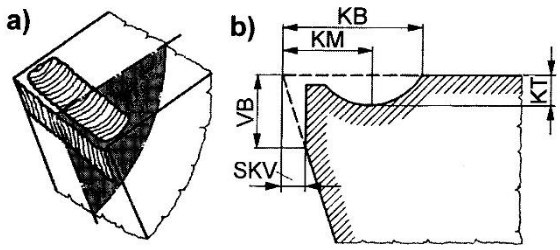
The process of wear on the front and back surfaces is very complex and is caused not only by friction but by a number of other physical and chemical effects. Based on the tests performed so far, the following types of wear can be distinguished:

abrasive wear caused by hard, abrasive particles in the workpiece, mechanical friction

adhesive wear, momentary micro-welds occur during tool-to-chip and metallic contact between the tool and the workpiece.

oxidation wear, oxygen in the air (700 - 800 ºC) reacts with carbide constituents (mostly cobalt) diffusion wear occurs at very high cutting temperatures, due to the diffusion of carbon atoms the carbide monocarbides disintegrate (WC, TiC) and their cutting ability decreases.

The listed micro-phenomena create significant, visible and measurable wear marks over the time spent in cutting (Figure 1-24).



1-24Figure. Wear of the tool. a) forms of wear; b) wear rate parameters [14]

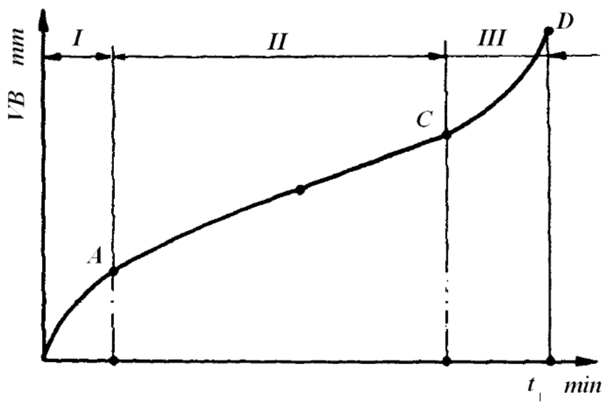
The wear of most cutting tools is characterized by the following forms of wear:

- back surface wear, (VB)
- radial misalignment (SKV)
- front surface and crater wear (KT)
- wear of the edges (rounding)

1-24. Figure illustrates the two most common forms of wear and their parameters. Crater wear occurs when roughing tough materials (eg steel) when the relatively thick chips run on the front surface to form a crater (recess) with characteristic dimensions KT crater depth and KB crater width.

The wear form on the back of the tool is back wear, which dominates in finishing and brittle machining. The extent of the WC wear width.

The main consequences of wear are an increase in cutting forces, an increase in machining defects (size, shape, surface roughness), and a decrease in the reliability of the entire system (increased risk of damage).



1-25 Figure. The time course of wear.

Time course of tool wear

The wear curve can be constructed by following the change in the cutting wedge over time. The Figure 1-25 shows the change in the degree of back abrasion (VB) over time. The wear process has three relatively distinct stages:

I. initial wear phase
II. uniform or so-called normal wear
III. excessive wear (intensive wear)

For economic reasons, it is very important that the tool is replaced before it wears out to the over-wear stage (between points C-D).

Wear criterion

When cutting, the time when the tool is so worn that it needs to be replaced occurs. This time is determined by the wear criterion. The wear criterion is the limit of a process-specific

parameter beyond which undesirable changes would occur on the tool or workpiece or in the cutting process itself. It is usually determined by the maximum allowable value of back wear.

The amount of allowable wear depends primarily on which criterion is used.

So for example:	for rough turning	VB_{max} = 0,8 – 1,6 mm,
	for smooth turning	VB_{max} = 0,3 – 0,6 mm.

Tool life

The time the horn spends cutting between two sharpenings or changes is called the edge life (sign: T, unit: min). Lifespan is affected by several factors:

- cutting data (cutting speed, chip thickness, chip width)
- tool material
- workpiece material
- application of cooling and lubrication,
- stability of the machining system (vibrations), etc.

For a given workpiece material, tool material, and unchanged machining conditions, the effect of cutting data on the tool life is expressed as the tool life relationship:

$$T = \frac{C_T}{v_c^{z_T} \cdot h^{x_T} \cdot b^{y_T}}$$

where:

C_T is the lifetime constant

x_T, y_T, z_T exponents.

The above lifetime relationship is three variables. Depending on the cutting mode and other conditions, it is advisable to keep each variable constant and thus change the relationship to bivariate or univariate. The cutting edge has the greatest effect on the tool life, so the tool life is often expressed only as a function of the cutting speed. The developer of such an univariate relationship is called the Taylor relationship:

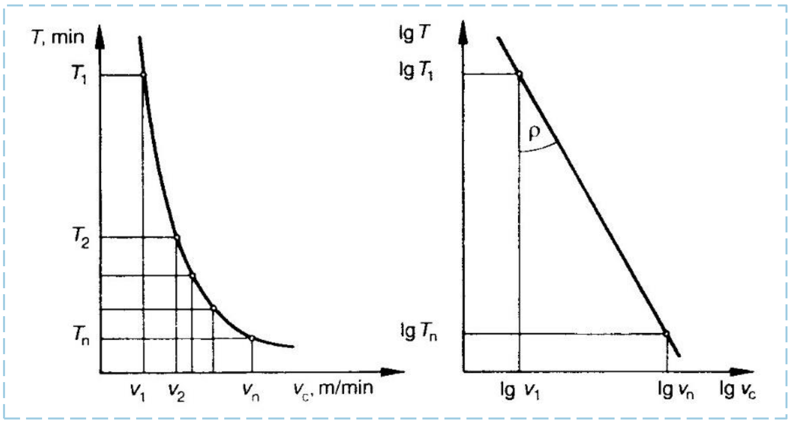


Figure 1-26. Taylor curve.

$$T = \frac{C_T'}{v_c^{z_T}}$$

where:

$$C_T' = \frac{C_T}{h_0^{x_T} \cdot b_0^{y_T}}$$

The literature makes no distinction between C_T' and C_T.

The Taylor relationship can also be plotted, called the T-v plot or Taylor curve (Figure 3-26). If the Taylor relationship is plotted on a logarithmic scale, the relationship can be plotted as a straight line:

$$log\,T = log\,C_T - z_T \cdot log\,v_c$$

Arranging the Taylor relation to v_c gives another form of the relation:

$$v_c = \frac{C_T^{\frac{1}{z_T}}}{T^{\frac{1}{z_T}}} = \frac{C_v}{T^m} \qquad \text{or} \qquad v_c \cdot T^m = C_v$$

where: $m = \frac{1}{z_T}$, and $C_v = C_T^{\frac{1}{z_T}}$ exponent and constant depending on the workpiece and tool material. The value of z_T depends on the workpiece and tool material and can be set to 2… 9.

If the speed (v_o) for a given edge content (T_o) is known, the speed corresponding to another desired edge content (T_1) can be calculated:

$$T_o^m \cdot v_o = T_1^m \cdot v_1 \cdot \rightarrow \quad v_1 = v_o \left(\frac{T_o}{T_1} \right)^m$$

Choosing the optimal edge life

The cutting process is considered optimal if it takes place with technological data that best approximates the set economic goal. The economic goal is the lowest cost, the lowest machining time, or the highest profit per unit time.

The basic problem with the economy of the process is that by increasing the cutting speed, feed and depth of cut, the machine time and associated costs are reduced, while tool wear and associated time and costs are increased.

The task is to find a compromise (optimum point) that gives the minimum total cost.

By selecting the edge life, the cutting speed is also determined indirectly according to the

Taylor relation.

Optimal service life at the lowest cost $T_o = \left(\frac{1}{m} - 1 \right) \left(t_{cs} + \frac{K_{sz}}{K_g} \right) \quad [min]$

where: t_{cs} – tool change time [min]

K_{sz} – tool cost per life [Ft]

K_g – machine cost [Ft/min]

m – lifetime exponent

The quantities in the formula indicate that the optimum service life can only be determined with knowledge of the operating factors. Unfortunately, this data is not always available. On the other hand, it is easy to see that similar machining tasks can be performed under similar operating conditions, suggesting that we can use the experience of others. The recommended service life value for each cutting situation can be selected from the tables in the literature or from the catalogs of the tool manufacturers.

The service life with the shortest machining time (maximum productivity) is exceptionally used for machining on so-called narrow hopper machines: $T_t = \left(\frac{1}{m} - 1 \right) \cdot t_{cs}$

1.10 SURFACE ROUGHNESS

The mutual movement of the workpiece and the tool creates a micro-roughness on the machined surface called roughness. The magnitude and shape of the roughness depend on the magnitude of the feed and the shape of the near-tip portion of the tool. Theoretical (theoretical) roughness Rt is nothing more than a repetitive mapping (imprint) of the proximal portion of the cutting edge tip (Figure 1-27).

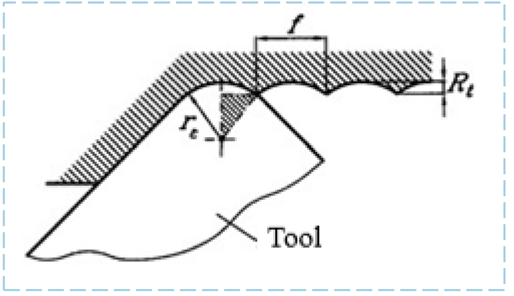


Figure 1-27. Theoretical roughness

The surface is formed by a circular edge section forming a proximal portion

$$r_\varepsilon^2 = \left(\frac{f}{2}\right)^2 + (r_\varepsilon - R_t)^2 = \frac{f^2}{4} + r_\varepsilon^2 - 2 \cdot r_\varepsilon \cdot R_t + R_t^2, \quad R_t^2 \approx 0$$

$$R_t = \frac{f^2}{8 \cdot r_\varepsilon}$$

The magnitude of the theoretical roughness corresponds best to the value of the roughness height R_z ($R_t \approx R_z$) among the standard roughness characteristics. The following relationship can be written between the theoretical and the average roughness:

$R_t \approx (2{,}5 \ldots 4{,}5) R_a$, where the smaller multiplication factor is used for roughing, the larger one is used for smoothing.

The actual roughness of the machined surface is higher than the theoretical one, the same roughnesses from different sources increase the theoretical roughness, eg the tool's own roughness, vibration marks, edge marks, etc.

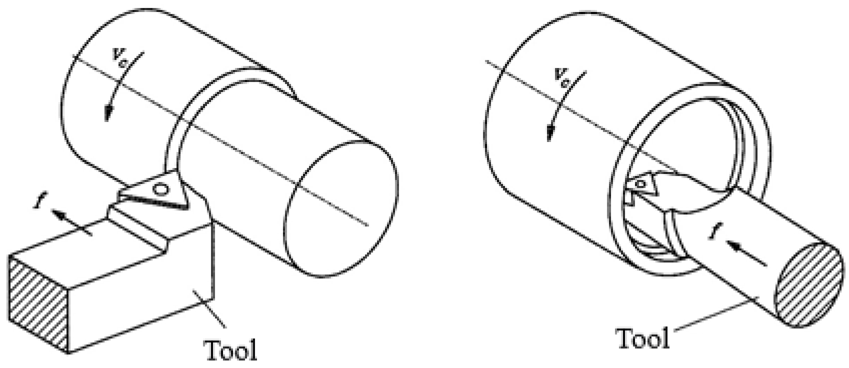
In addition to the factors examined so far, a number of other factors affect the surface roughness. The most significant of these are the coolant, the amount of tool wear, and the vibrations in the machining system.

The following contribute to the improvement of surface roughness:

- increase cutting speed
- reduction of depth of cut and feed
- increase the value of the peak radius
- use of appropriate coolant
- sharp, positive-angled tool
- stable workpiece and tool clamping

2. TURNING

Turning is a method of cutting that is performed with a single-edged tool in a rotating main motion, which is usually performed by the workpiece. The auxiliary movement is performed by the tool in a plane perpendicular to the direction of the main movement. This creates a surface of rotation (Figure 2-1). The tool of the process is the cutting knife, a single-simple, relatively simple tool. Turning is currently the most common method of cutting.



a) external turning b) hole turning

Figure 2-1. Turning.

2.1 THE VERSIONS OF TURNING

Depending on the direction of the side movement, the shape of the tool and the type of surface to be formed, there are many turning variants. A Figure 2-2. shows the main variants of turning.

Depending on whether the machining creates an external or internal surface, turning can be:

- external turning
- hole turning

Depending on the direction of the feed movement relative to the workpiece axis, the turning can be:

- parallel to the axis
  - longitudinal turning
  - cross turning
    - side turning
    - cut off
      - grooving
      - part-off
- general feed direction
  - contour turning
    - conical turning

- spherical turning
- torus turning

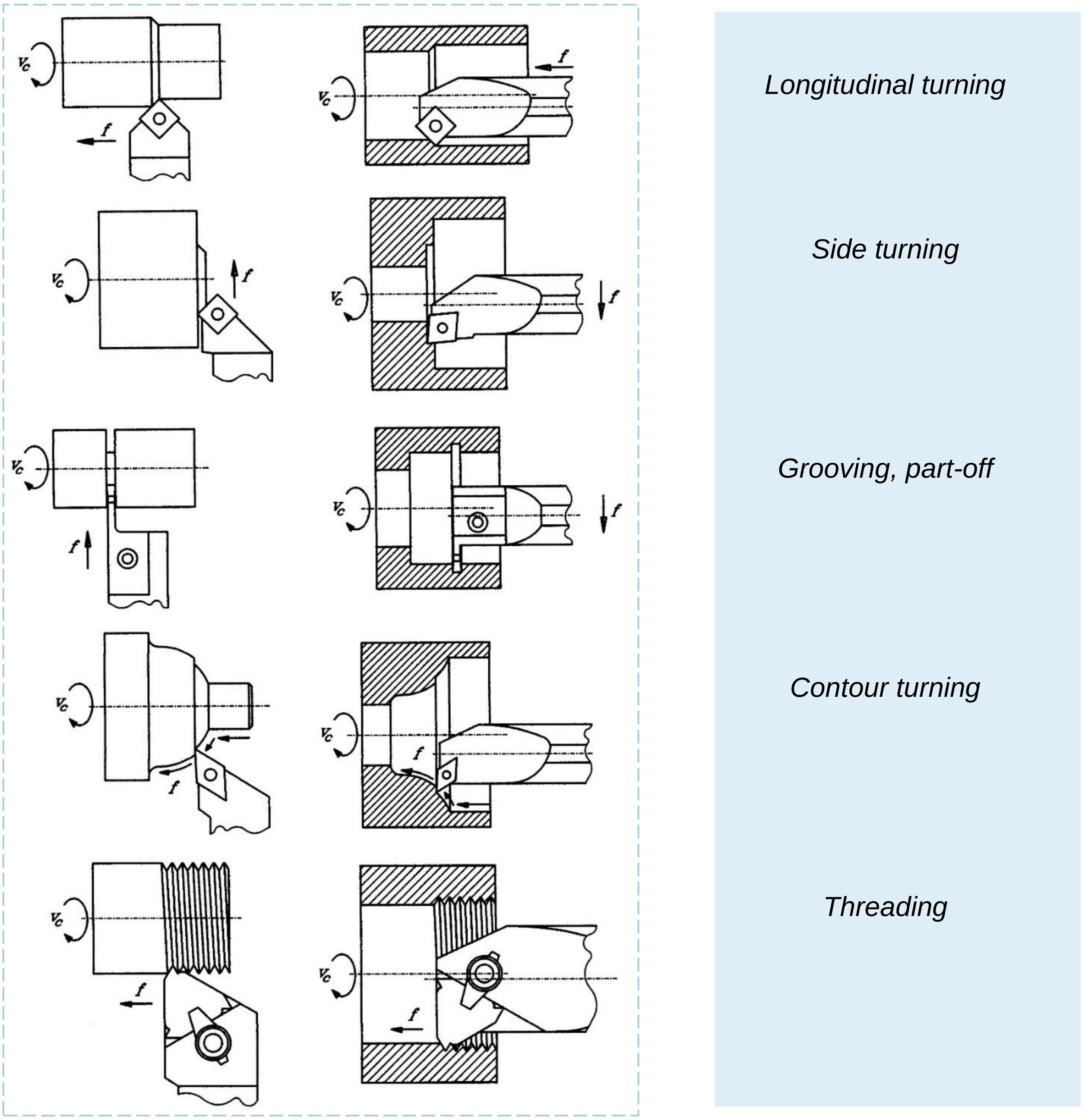


Figure 2-2. The main versions of turning

Turning is the case of the cutting machining model, the cutting theory has been studied on this, so we do not repeat what is described there.

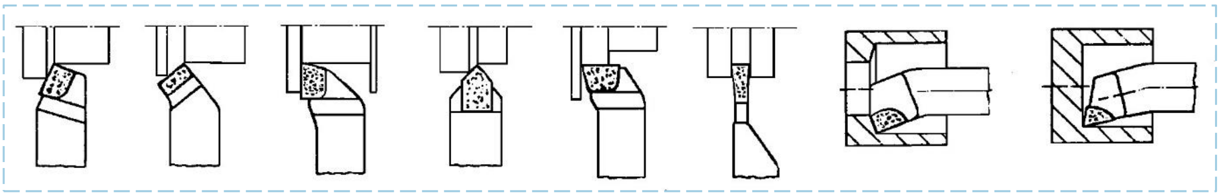
2.2 TURNING TOOLS (CUTTING KNIVES)

A suitable tool is required to machine the specified surfaces of the workpiece. The tool is suitable if the following conditions are met:

- geometrically suitable for surface formation (accessibility), Figure 2-3.
- has the appropriate edge geometry
- its material is suitable for cutting
- cutting can be done economically with the tool

Due to the many variations of turning, the design of the knives is also very varied. The two important components of the tool are the knife shank - which clamps the knife into the machine tool - and the working part, which performs the cutting directly.

Straight and curved turning knives are distinguished according to the relative positions of the working part and the tool shank (Figure 2-3). The advantage of curved knives is that they are also suitable for sideways. A distinction is made between right (R), left (L), and neutral (N) tools according to the direction of the assumed feedrate (Figure 2-4).



Straight DIN4971	*Bent DIN4972*	*Side-turned DIN4978*	*Pointed DIN4975*	*Forehead DIN4977*	*Grooving DIN4981*	*For through hole DIN4973*	*For blind hole DIN4975*

Figure 2-3. Solder-insert turning tool designs

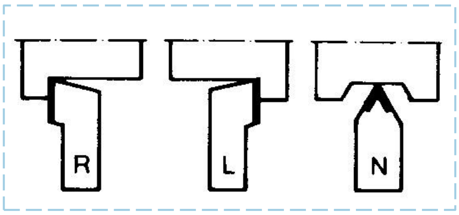


Figure 2-4. Divide the cutting blades according to the feed direction

The material of the working part can be high-speed steel, carbide or ceramic. High-speed steel turning knives are increasingly being displaced by carbide and ceramic tools, with the

exception of shaped knives.

Carbide inserts can be made soldered or insert inserts. Typical types of solder tool designs are shown in Figures 2-3. illustration. The proportion of indexable insert tools is increasing. The soldered insert design is now only used for special tools (such as knives). There are several benefits to using indexable inserts:

- lower specific tool costs, sharpening costs are eliminated,

- quick tool (insert) replacement,
- the quality of the edge is not damaged by soldering or sharpening

The insert has several cutting edges, which can be brought into the working position in the knife body after the previous wear (Figures 2-1). The range of inserts is very wide, their shape and dimensions are specified in the ISO 1832 standard.

When selecting cutting blades, the shape of the knife must be selected according to the geometry of the workpiece and the turning version. In addition to the shape of the knife, care must be taken to select the appropriate edge geometry. The edge geometry should be chosen primarily according to the material of the workpiece. The main mounting angle can be selected between $\kappa_r = 45 \ldots 90°$. When machining slender axes, $\kappa_r = 90°$ is the most favorable, because in this case the value of the force in the direction of gripping is the smallest. From the point of view of tool life, however, it is best if $\kappa_r = 45°$. When selecting the peak radius (r_ε), it should be noted that the roughness of the machined surface decreases with a larger peak radius, but at the same time the cutting force and the risk of vibration increase. Therefore, in practice, the recommended value of the peak radius is 0.4… 0.8 mm when roughing and 0.8… 2 mm when smoothing.

2.3 WORKPIECE CLAMPING, TURNING DEVICES

The task of the clamping is to position and move the workpieces. Depending on the shape of the workpieces, the turning variant and the degree of accuracy, the workpiece can be clamped in different ways. The chucks used for turning are as follows:

- self-centering chuck
- flat disc
- lathe tips
- receiving pin
- support dummy
- driver (for clamping between tips, force and torque transmission between workpiece and machine spindle)

The workpieces to be machined can be divided into two major groups according to the length / diameter (l/d) ratio:

- disc-like (short) workpieces (l/d < 3…4),
- axial (long) workpieces (l/d > 3…4).

Disc-like and short pieces are most often captured in a self-centering chuck (Figure 2-5). A flat disc is used when a rotating surface is to be machined on a non-rotating body. The flat disc has four clamping jaws that can be adjusted individually. Long workpieces are clamped between the tips. Of course, this is only possible by first making tip nests (center holes) on the front surfaces of the workpiece. As the peaks cannot perform the task of driving, the driving must be taken care of separately. Driving can be done with a lathe or a claw driver (Figure 2-6). The long pieces can also be clamped by gripping one end of the workpiece and supporting the other with a tip. The centering of the workpiece is thus less accurate, but no driver is required. For long, slim workpieces where a support dummy l / d> 8… 12 is used.

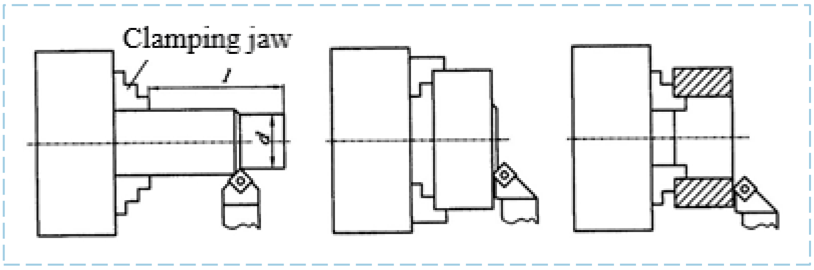


Figure 2-5. Clamping of short and disc-like pieces in a self-centering chuck

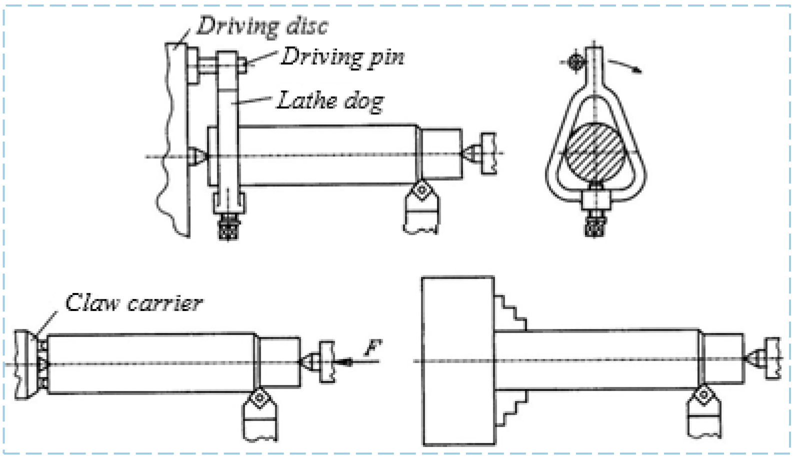


Figure 2-6. Clamping of axial pieces

2.4 CUTTING FROCE, PERFORMANCE

The determination of the cutting force can be determined as discussed in Section 1.5, using the specific cutting force or the exponential formulas.

a) Determination of the main cutting force based on the specific cutting force

$$F_c = k_c \cdot b \cdot h = k_{c1.1} \cdot b \cdot h^{1-m} \cdot K_F \text{ [N]}.$$

Where: $k_{c1.1}$ the main value of the specific cutting force

m exponent, their values are given in 1.3 Table

$K_F = K_{Fsz} \cdot K_{Fv} \cdot K_{F\gamma o} \cdot K_{F\kappa} \cdot K_{Fr} \cdot K_{FVB}$ the cumulative modification factor (see section 1.5.3.1).

$h = f \cdot \sin \kappa_r$ chip thickness

$b = \dfrac{a_p}{\sin \kappa_r}$ the width of the chip (2-7. Figure)

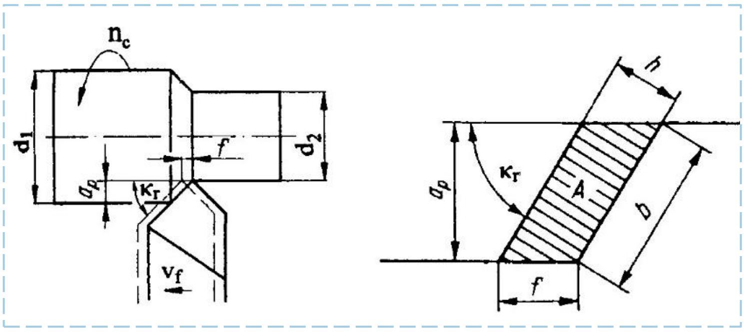


Figure 2-7. Theoretical shape of the chip cross section

b) Determination of the main cutting force by exponential shape

$$F_c = C_F \cdot f^{x_F} \cdot a_p^{y_F} \cdot K_F$$

Where:

C_F force constant which depends on the material quality

x_F , y_F exponents. Their values are given in 1.4. Table. The following values can be used for cutting cast iron and steel: $x_F \approx 0{,}75$, $y_F \approx 1$.

The power required for cutting is the product of the cutting force and the cutting speed

$$P_c = F_c \cdot v_c$$

Machine performance is slightly higher than this, as losses within the machine must also be taken into account:

$$P_m = \frac{P_c}{\eta},$$ where $\eta \approx 0{,}7$ is the efficiency of the machine.

2.5 DEFINITION OF CUTTING DATAS

2.5.1 DEPTH OF CUT

Depending on the depth of cut and the feed range, the turning can be divided into finishing, light roughing and rough (rough) roughing. These ranges are not clearly defined, with some differences from different sources. The breakdown by Sandvik Coromant, one of the world's leading tool companies, is shown in Figure 4-1. shown in the table. These data provide certain limits that may be indicative in the first approach.

2-1. Table Depth of cut and feed ranges

	Depth of cut a_p [mm]	Feed f [mm/ford]
Smoothing	0,5…2	0,1…0,3
Easy roughing	1,5…5	0,2…0,5

Rough roughing	5…15	0,5…1,5

From the cutting data, the cutting depth is always determined first. The depth of cut depends primarily on the size of the technological allowance, the starting and machined diameter. The allowance for cylindrical surfaces should always be understood as diameter, while for side and flat surfaces it should be understood as the value per surface (RL) (Figure 2-8). The allowance for roughing (R1), finishing (R2) and refining (R3) must be determined separately. The need for the listed machining steps must be determined on the basis of the required dimensional accuracy and heat treatment requirements. Refining is usually done by grinding. The definition of allowances and preforms is not the subject of this note and can be found in the literature [18], [19].

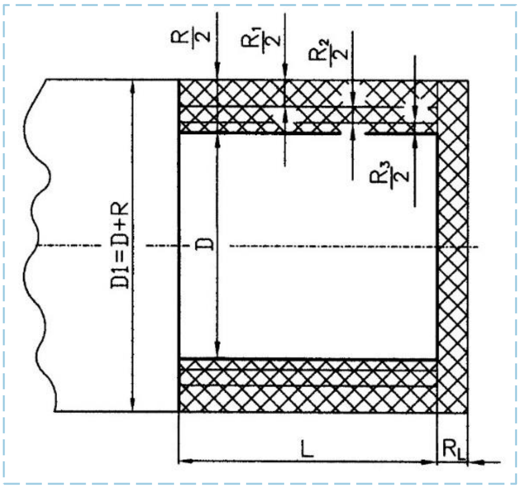


Figure 2-8. Turning allowances

Since the tool life is least affected by the depth of cut, in practice they try to solve a given machining with as few grips as possible.

The maximum allowable depth of cut (a_{pmax}) depends on the tool edge length and the size of the main mounting angle (Figure 2-9):

$$a_{p\,max} = B_{max} \cdot \sin \kappa_r$$

Where:

B_{max} the usable edge length. In the case of soldered carbide inserts, the following relationship may be used unless otherwise specified by the tool manufacturer.

$B_{max} = (0,63 \ldots 0,8)\, B_s$

Where B_s is the total edge length.

For square inserts $B_{max} \leq 0,75\, l$ where l is the side length of the insert.

For triangular inserts $B_{max} \leq 0,5\, l$.

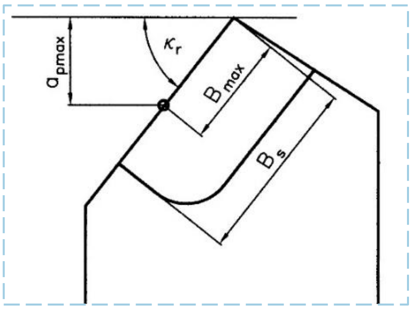


Figure 2-9. Maximum allowable depth of cut

It is possible that the thickness of the layer of material to be deposited is greater than the maximum permissible depth of cut during roughing, in which case the layer of material is removed in several passes.

Preliminary number of catches at roughing: $i' = \frac{D_1 - D}{2 \cdot a_{p\,max}}$

Where: D_1 the initial diameter

D the machined diameter

Since the number of catches can only be an integer, the nearest larger integer to *i* is chosen based on the value thus obtained.

After defining the catch distribution, the value of the depth of cut can be determined: $a_p = \frac{D_1 - D}{2 \cdot i}$

Smoothing is always done in one pass.

2.5.2 DEFINITON OF THE FEED

In the case of roughing, our primary goal is to separate as much material as possible in one unit of time. High material removal rates (q [cm^3 / min]) can be achieved by selecting a high feedrate or chip cross section. This, of course, also results in a high cutting force that loads the elements of the machining system, i.e. the tool, the machine tool components, and the workpiece. The weakest element of the system limits the maximum permissible value of the cutting force and also the value of the feed. It is also necessary to take into account the depth of cut and feed ranges that can be used for chip breaking using a given tool (insert).

In practice, we usually proceed by choosing the recommended value based on experience [21], and then checking the selected feed from several points of view. Some sources relate the recommended value of the roughing feed to the rounding radius of the tool tip and accordingly:

$$f \approx 0{,}5 \cdot r_\varepsilon \, .$$

A larger radius increases the stability of the edge, but above a value, vibrations occur in the system. For roughing, in the absence of other limiting factors, the peak radius (r_ε) shall be at least 0.8 mm. It can be seen that a larger radius of curvature increases the value of the passive force (F_p), so it is good for long, thin parts and hole knives if the peak radius is not greater than 0.4 mm.

Based on observations and experiments, it has been found that for favorable chip removal, the ratio of depth of cut to feed ($g = \frac{a_p}{f}$) must be within certain limits.

The recommended value of this ratio is g = 5… 10 for long turning and g = 5… 20 for cross turning.

In the case of finishing turning, the feed must always be determined on the basis of the required surface roughness.

For grooving and grooving, the depth of cut is equal to the width of the knife and groove, respectively. As a result, the cross-section of the working part of the knife is very poor, on the other hand, the knife works with a considerable length of gripping edge and this makes the tool prone to vibration. For this reason, a reduced feed and cutting speed are used for grooving.

2.5.2.1 CHECKING THE FEED BASED ON THE CUTTING TOOL CROSS SECTION

The lathe blade can be considered as a clamped holder loaded by the cutting forces. In the case of external turning, the knife shank usually has a sufficiently large cross-section and the protrusion of the knife does not have to be chosen. As a result, the stress in the knife shank and the deflection of the knife shank are relatively low. The situation is different for hole turning: here the cross-section of the knife shank is determined by the hole diameter to be machined, and the protrusion of the knife must correspond to the depth of the hole. In the case of internal machining, the strength or bending of the knife shank may limit the amount of feed. The feedrate can be determined by the strength or deflection of the knife shank.

Checking the feed based on the cutting tool strength

To check the strength of the knife shank, a simplified mechanical model can be used in which the feed direction and passive forces are negligible (Figure 2-10).

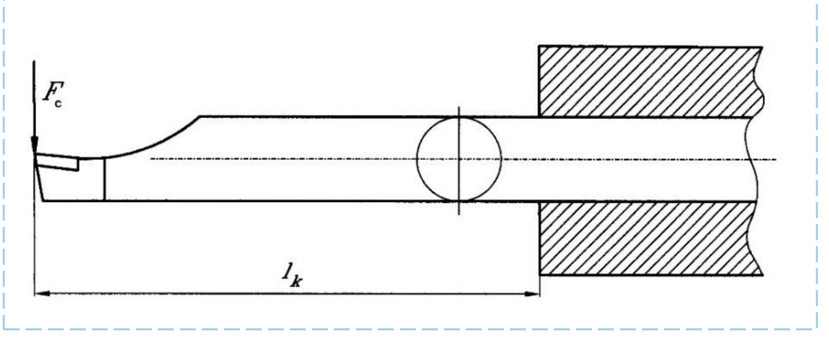


Figure 2-10. The lathe knife as a clamped holder

Bending tension in the knife shank: $\sigma = \dfrac{M}{K} \leq \sigma_{meg}$

Where: $M=F_c \cdot l_k$ bending moment

K cross - sectional factor

$K = \dfrac{h^2 b}{6}$ in the case of a rectangular cross-section (h- height; b- width)

$K = \dfrac{d^3 \pi}{32}$ in the case of a circle cross section

σ_{meg} = 200-250 [MPa] for familiar knife shank materials

l_k the protrusion of the knife

From this context, the maximum permissible cutting force can be determined:

$$F_{cmeg} = \frac{K \cdot \sigma_{meg}}{l_k}$$

Alternatively, if we know that the cutting force is, among other things, a function of the feedrate ($F_c = C_F \cdot f^{x_F} \cdot a_p^{y_F} \cdot K_F$), the feedrate limit can be determined::

$$f \leq \sqrt[x_F]{\frac{F_{cmeg}}{C_F \cdot a_p^{y_F} \cdot K_F}}$$

Checking the feed based on the cutting tool is deflection

The allowable deflection usually imposes a stricter limit on the value of the feed than the allowable stress. Deflection of the knife above a certain value leads to the generation of vibrations, which must be avoided at all costs. Based on the previous model, the value of the deflection can be determined by the following relation: $\delta = \frac{F_c \cdot l_k^3}{3 \cdot E \cdot I} \leq \delta_{meg}$

Where: E modulus of elasticity of the stem material

I the second order torque of the cross section

$I = \frac{h^3 b}{12}$ in the case of a rectangular cross-section (h- height; b- width)

$I = \frac{d^4 \pi}{64}$ in the case of a circle cross section

δ_{meg} = 0,08-0,25 mm the allowable deflection for roughing

δ_{meg} = 0,01-0,04 mm the permissible deflection for smoothing

The maximum cutting force resulting from this condition:

$$F_{cmeg} = \frac{3 \cdot E \cdot I \cdot \delta_{meg}}{l_k^3}$$

Respectively, the feed limit: $$f \le \sqrt[x_F]{\frac{F_{cmeg}}{C_F \cdot a_p^{y_F} \cdot K_F}}$$

2.5.2.2 CHECKING THE FEED BASED ON THE MACHINING ACCURACY

Due to the cutting forces, elastic deformation takes place in the machining system. Within this, the deformation of the workpiece is examined. The degree of deformation depends on the

magnitude of the force and the clamping method, as well as the dimensions and modulus of elasticity of the workpiece. Typical clamping methods, their mechanical models and shape

defects due to elastic deformation are shown in Figures 2-16.

For the mechanical models shown in Figure 4-16., the maximum deflection of the holder:

$$\delta = \frac{F \cdot l^3}{\mu \cdot E \cdot I}$$

Where: *l* the length of the holder

E modulus of elasticity of the workpiece material

I the second order torque of the cross section ($I \approx 0{,}05 \cdot d^4$)

d the diameter of the workpiece

μ model constant: $\mu=3$ place in chuck

$\mu=48$ workpiece place between two tips

$\mu=102$ place in chuck supported by tip

F the force.

The passive force F_p is decisive in determining the deformation error. Although its value is smaller than the cutting force F_c, the resulting deflection (deflection) directly results in a workpiece diameter error (Figure 2-11).

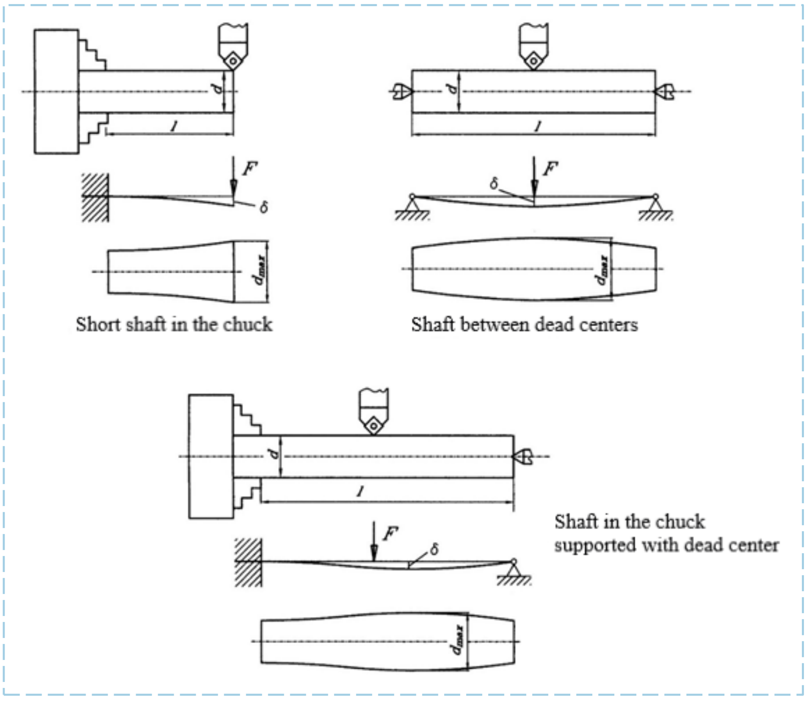


Figure 2-11. Diameter error due to force component Fp

The passive force can be determined using the cutting force and a proportionality factor:

$$F_p = \lambda_{cp} \cdot F_c$$

The value of the proportionality factor λ_{cp} depends primarily on the angle of the main edge. As the angle (κ_r) of the main edge placement increases, the value of the proportionality factor decreases. The values that can be used in practice can be selected according to Table 2-2.

Table 2-2. Values of the proportionality factor λcp [3]

κ_r	30°…45°	60°…75°	90°
λ_{cp}	0,4…0,5	0,32…0,4	0,25…0,32

If the permissible value (δ_{meg}) of the diameter error due to the bending of the workpiece is determined, the permissible value of the passive force can also be determined, and thus the maximum permissible value of the feedrate can be determined.

$$F_{cmeg} = \frac{\mu \cdot E \cdot I \cdot \delta_{meg}}{\lambda_{cp} \cdot l^3}$$

Respectively, the feed limit: $$f \le \sqrt[x_F]{\frac{F_{cmeg}}{C_F \cdot a_p^{y_F} \cdot K_F}}$$

When determining the allowable value of the diameter error (δ_{meg}), two cases are distinguished:

a) In the case of finishing, the permissible diameter error is determined on the basis of the value of the specified tolerance field. Since the diameter error due to bending is only one of the many sources of error, one-sixth of the width of the tolerance field can be taken as the value of δ_{meg}: $\delta_{meg} = \frac{T}{6}$, where T is the width of the tolerance field. This check makes sense when finishing is not preceded by roughing.

b) If roughing is followed by finishing in the same jaw, it must be assumed that the residual errors in roughing are not greater than the allowance for finishing (R_2) (Fig. 2-12).

$$\frac{R_2}{2} \geq \delta_{meg} + C + R_t \text{, illetve } \delta_{meg} \leq \frac{R_2}{2} - C - R_t$$

Where: *R_2* allowance for finish

R_t theoretical roughness after roughing ($\frac{f^2}{8 \cdot r_\varepsilon}$) is expected to be between 40 and 160 µm. Whether we have estimated it well can be checked later

and in the event of a large deviation, the calculation must be repeated.

C layer of material damaged due to mechanical and thermal effects

thickness, roughly C = 40… between 80 µm [20].

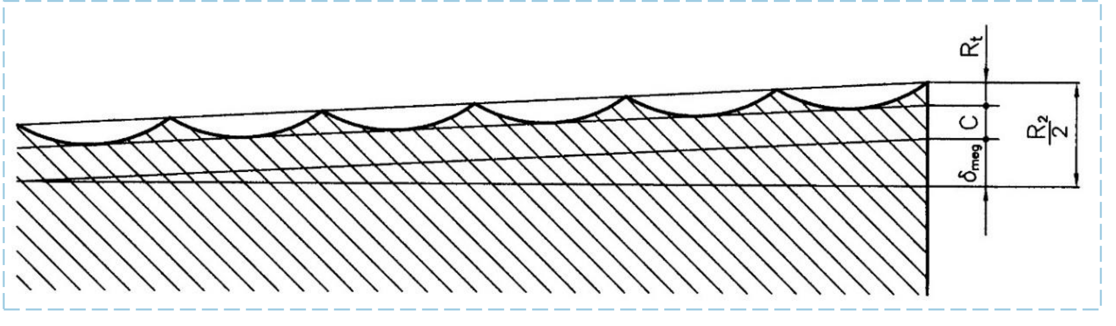


Figure 2-12. Defects in the machined surface after roughing

It should be noted that the dimensional accuracy and surface roughness after roughing can also be given by inter-operation tolerances, in which case, of course, this should be taken into account when determining the feedrate.

2.5.2.3 CHECKING THE FEED BASED ON THE SURFACE ROUGHNESS

For finishing, the feedrate is usually determined from the surface roughness (R_a) specified in the workpiece drawing of the workpiece, according to the relationships discussed in 1.10. Section, assuming that $R_t \approx 4R_a$:

$R_t = \dfrac{f^2}{8 \cdot r_\varepsilon}$, from here $\qquad f = \sqrt{8 \cdot r_\varepsilon \cdot R_t} \approx \sqrt{32 \cdot r_\varepsilon \cdot R_a}$

In the final selection of the feedrate, of course, we can only select a value that can be implemented on the given machine. The lowest value obtained with the calculations is the standard one, and based on this, the first smaller feedrate must be selected on the given machine.

2.5.3 DEFINITION OF THE CUTTING SPEED

After determining the depth of cut and the feed rate, the cutting speed and the spindle and workpiece speeds are determined.

When turning, the cutting speed is the circumferential speed of the workpiece at its largest diameter in relation to the tool edge. The speed of the workpiece or spindle can be calculated based on the cutting speed: $v_c = \dfrac{d \cdot \pi \cdot n}{1000}$ [m/min] $\qquad n = \dfrac{1000 \cdot v_c}{d \cdot \pi}$

In practice, the cutting speed is mostly chosen on the basis of experience and technological tables [18], [21]. Tool manufacturers also usually provide detailed tables for selecting cutting parameters.

A modern way to determine the cutting speed is to use exponential formulas based on the life cycle relationship:

$$v_c = \frac{C_v}{T^m \cdot f^{x_v} \cdot a_p^{y_v}} \cdot K_v$$

Where: C_v, x_v, y_v, m constant and exponents (2-3. Table)

T [min] tool life

$K_v = K_{v\kappa} \cdot K_{vvo} \cdot K_{vH}$ aggregate speed modification factor

$K_{v\kappa}$ main edge placement angle modification factor

K_{vvo} turning variant modification factor

K_{vH} coolant and lubricant application modifier.

$K_{vH}=1$ in case of refrigerant application

$K_{vH}=0,7$ without the use of cooling

Their values can be found in the literature [18].

2-3. Table Values of the velocity constant Cv and the exponents xv, yv, m for carbide tools

The material of the workpiece		Carbide	C_v	x_v,	y_v,	m
Steel	R_m<500	P10	650	0,26	0,10	0,27
	R_m=500-700	P10	490	0,22	0,11	0,29

	R_m=700-900	P10	479	0,21	0,11	0,29
		P30	377	0,27	0,12	0,29
		P20 coated	458	0,22	0,11	0,25
	R_m>900	P10	237	0,29	0,11	0,19
		P30	177	0,37	0,12	0,19
		P20 coated	234	0,31	0,11	0,16
Cast iron	*HB*<200	K10	271	0,15	0,10	0,23
	HB>200	K10	200	0,19	0,10	0,23

The cutting speed and speed obtained on the basis of the service life relationship may be limited by the power of the machine. This should always be checked and, if necessary, the speed reduced accordingly. Implement performance:

$$P_m = \frac{F_c \cdot v_c}{\eta \cdot 60000}[kW]$$, from this $$v_c \le \frac{60000 \cdot P_m \cdot \eta}{F_c}\left[\frac{m}{min}\right]$$ or

$$n \le \frac{60 \cdot 10^6 \cdot P_m \cdot \eta}{F_c \cdot d \cdot \pi} \text{ min}^{-1}$$

2.6. DEFINITION OF THE MACHINE TIME FOR TURNING

Knowing the cutting parameters and the dimensions of the workpiece, the machine main time required for cutting can be determined for each operation: $t_g = \frac{L}{v_f} \cdot i = \frac{L}{n \cdot f} \cdot i$

Where: $v_f = n \cdot f$ the feed rate
i number of cuts
L the length of the toolpath.

Determining the toolpath for longitudinal turning (2-13. Figure):

$$L = l + l_1 + l_2 \quad ,$$

l length of machining

$l_1 = \frac{a_p}{tan\kappa}$ +1…2 mm, the length of the tool run on

l_2 tool overrun length (1-2 mm if required)

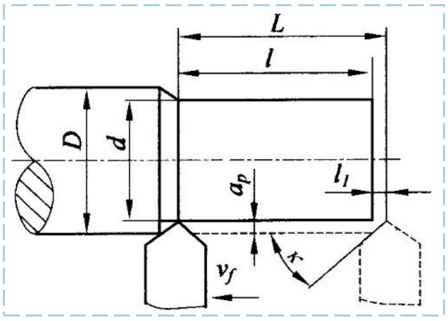


Figure 2-13. Determining the toolpath for longitudinal turning

Determining the toolpath for cross turning (Figure 2-14):

$$L = \frac{D - D_1}{2} + l_1 + l_2$$

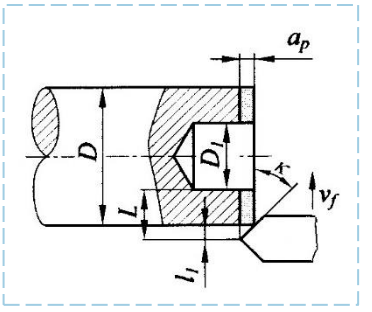


Figure 2-14. Determining the toolpath for cross turning

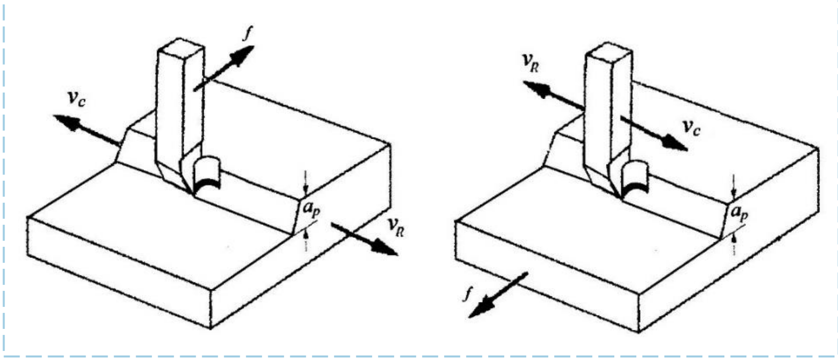
3. PLANING AND SLOTTING

Planing is a mode of cutting in which both the main and secondary movements of the cutting are mostly straight. In the process, chips with a constant cross-section ($A = a_p \cdot f$) are intermittently separated with a single-edged tool. Planing is used to machine planes, grooves and generally straight surfaces on long, narrow parts. Planing is displaced in many areas by higher productivity milling.

THERE ARE TWO BASIC VERSIONS OF PLANING:

- Cross-planing
- Long planing (usually for large pieces)

These differ in the distribution of main and secondary movements (Figure 3-1). When planing crosswise, the alternating straight main movement is performed by the tool, and the intermittent side movements are performed by the workpiece. In longitudinal planing, the main movement is performed by the workpiece, and the intermittent secondary movements are performed by the tool.



a) long planing b) cross-planing

Figure 3-1. basic versions of planing

Engraving can be considered as a vertical version of planing, in which the alternating main movement is performed by the tool and the secondary movements by the workpiece. It is most often used in the machining of internal surfaces. The tool must be run with a washer or a groove (Figure 3-2). It is most often used in the machining of internal surfaces.

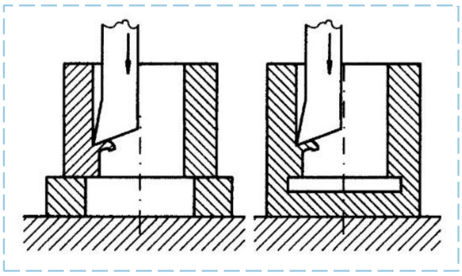


Figure 3-2. Engraving

Planing can be identified by turning (d=∞)), taking into account the conditions below the working stroke. Side movements pause during the main movement.

The design of the planer blade is similar to the design of the lathe blade, except that due to the greater protrusion and the dynamic load (the tool strikes the tool after each stroke), it is advisable to choose larger shank cross sections. Regarding the material of the tool, high-speed steel or sometimes a hard metal with high toughness is most often used.

The cutting data can be determined according to the correlations and guide values established during turning, with the value of the cutting speed having to be modified (reduced) due to the shocks during planing. The speed value obtained for turning must be multiplied by a modification factor K_{mod} = 0,75. The speed thus obtained shall not exceed 60 m / min.

Cutting forces and power are determined in the same way as for turning.

4.HOLE MACHINING (DRILLING, COUNTERBORING, REAMERING)

Hole machining is a rotary main-motion cutting process in which the axis of rotation of the tool is the same as the axis of the rotating surface to be made and the feed direction, unlike hole turning, can only be the same as the direction of rotation. When drilling holes, a separate tool is required to make each diameter ("dimensional tools").

4.1 THE VERSIONS OF HOLE MACHINING

Typical hole machining variants are shown in Figures 4-1.… 4-4. figures show. These differ in the initial condition of the hole, the shape of the hole, the dimensional accuracy of the hole and the shape of the tool. Hole machining can be divided into four subgroups: (1) drilling, (2) flat countersinking, (3) profiling drilling, (4) tapping.

(CIRCULAR) DRILLING is a cutting process in which a cylindrical inner surface is created. The initial state or according to the connection between the tool edge and the workpiece, the drilling can be (Figure 4-1):

- drilling completely
- core drilling
- drilling
- reaming
- drilling with drill rod

When drilling completely, make a hole in the material without a pre-drill with a drilling tool. The most common tool is the auger. Available dimensional accuracy IT12 to IT13, surface roughness R_a> 12.

Core drilling is a drilling process in which the material is cut along only one annular ring and a cylindrical core is formed (remaining) at the same time as the hole. Its tool is the core drill or also known as the core drill.

Drilling is the process of cutting an increase in the diameter of a hole previously made (by casting or drilling), ie by drilling a hole. The available dimensional accuracy is IT 11 possibly IT10, the surface roughness R_a = 1,6-6,4.

By reaming, a very small layer of material is separated in order to increase the dimensional accuracy, shape fidelity and surface quality of the already drilled hole. Tool for reamer. The available dimensional accuracy is IT8, IT7, the surface roughness R_a = 0,8-1,6.

Borehole expansion with a drill rod is usually used for holes larger than 25 mm in diameter. The process of chip removal is the same as for hole turning, but should be classified for drilling in terms of movement system. Roughing, pre-smoothing and finishing with a drill rod. Tool is the drill rod. Available dimensional accuracy for smoothing IT7, IT6.

FLAT COUNTERSINKING is a cutting process in which a flat surface perpendicular to the axis of the hole is created. A distinction is made between countersinking and plunging (Figure 4-2). By countersinking, a small flat surface protruding from the surface of the workpiece (e.g., casting on it) is created. When plunging, a flat surface deeper than the surface of the workpiece is created and a short cylindrical surface is created. The tool for both methods is a flat countersink with or without a guide pin.

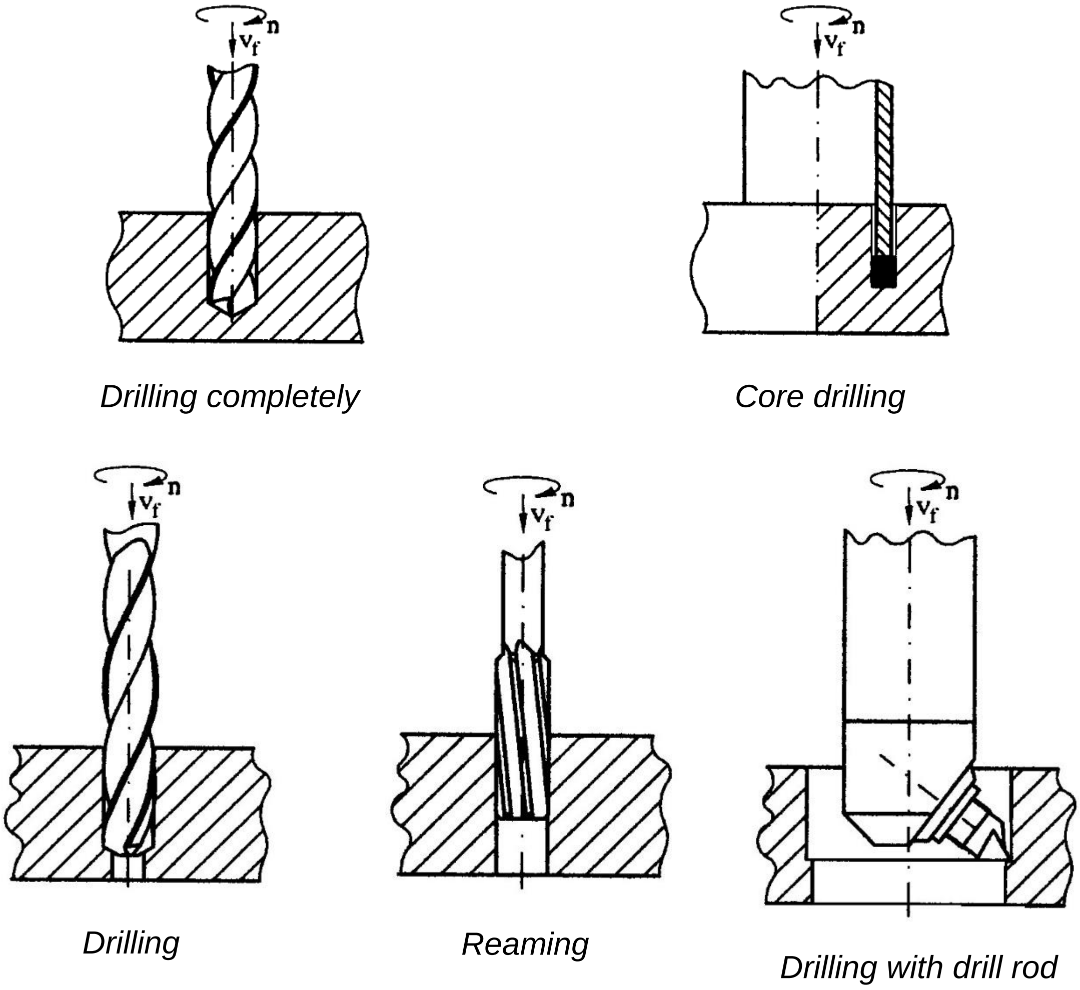


Figure 4-1. Drilling

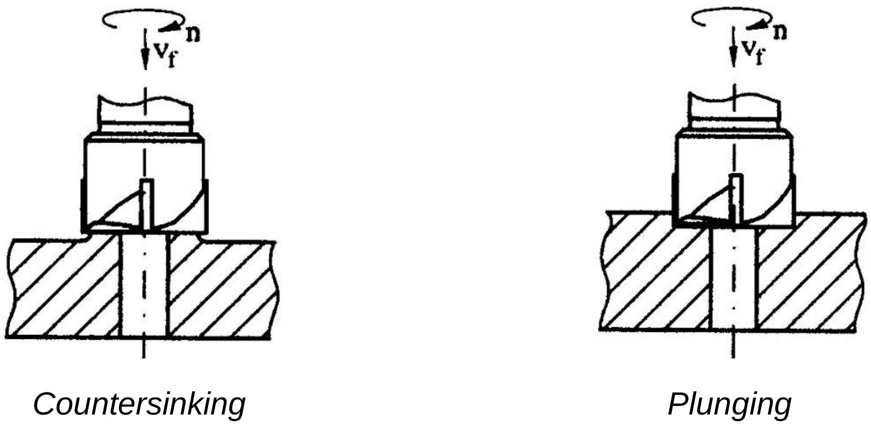


Figure 4-2. Flat countersinking

Typical cases of PROFILING DRILLING are the following: in the case of profiling drilling, a rotationally symmetrical inner profile is formed in a solid material (Figure 4-3). A common case is center drilling, which can be used to drill an end hole at the end of a tip hole drilling shaft or a auger to ensure the exact position of the hole.

In the case of profiling, the required internal profile is given by the profile of the drilling tool. For example, drilling a tapered hole to insert a tapered pin. Additional profiling procedures include profile lowering and profile reaming.

THREAD DRILLING is the drilling of a previously drilled hole with a thread profile tool to create an internal thread (Figure 4-4).

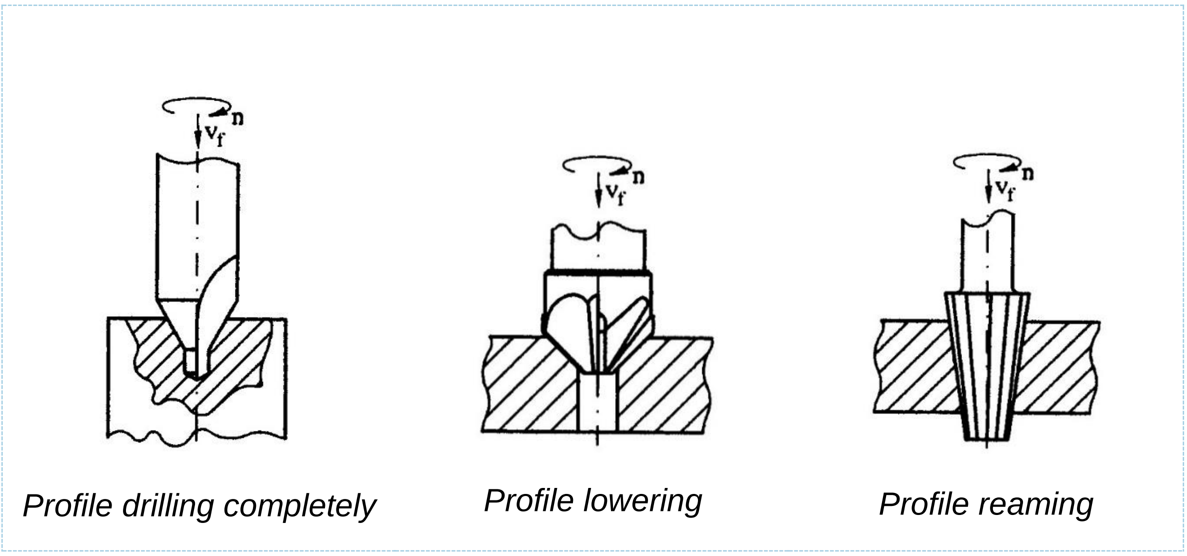


Figure 4-3. Profilr drilling

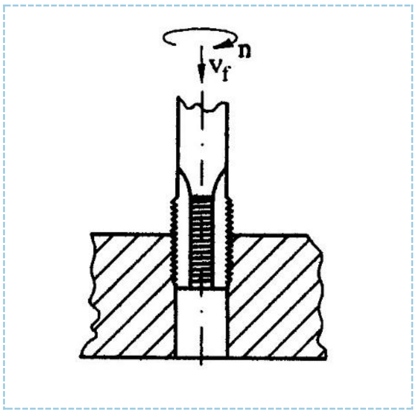


Figure 4-4. Thread drilling

4.2 HOLE MACHINING TOOLS

Due to the variety of hole machining variants, the tools can also be very diverse. They are mostly made of high-speed steel, sometimes with a TiN coating, but there are also indexable insert drilling tools. For small diameters, drilling tools are also made of solid carbide. The detailed description of the structural elements and the edge geometry is limited to the case of the auger, the other tools are usually of a simpler design, the edge geometry can be traced back to the evening of the auger.

SCREWDRIVER. The most common drilling tool used for full drilling and occasional drilling. Its design and main parts are illustrated in Figure 4-5. It can be made with a cylindrical or Morse conical shank, in the range of 0.2… 100 mm in diameter (diameters larger than 25 mm are used for hole widening). The two grooves significantly weaken their cross-section, so they have little rigidity. To ensure the exact position of the hole, a very shallow starting hole must be drilled or drilled in a drilling rig before drilling with an auger.

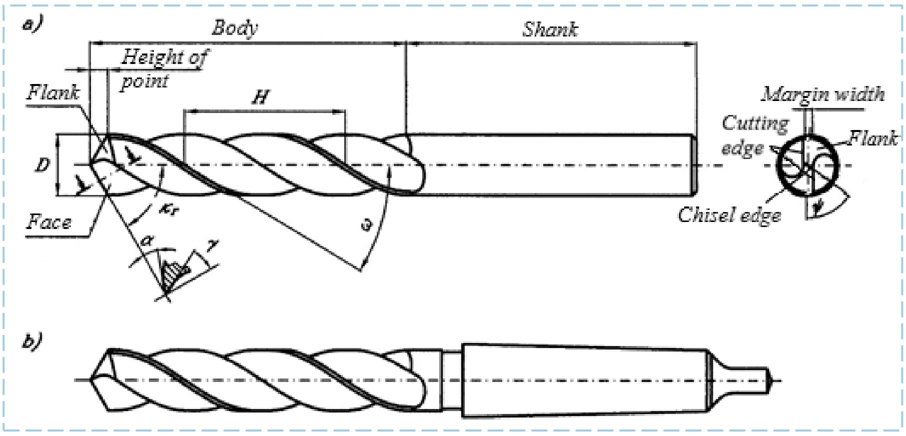
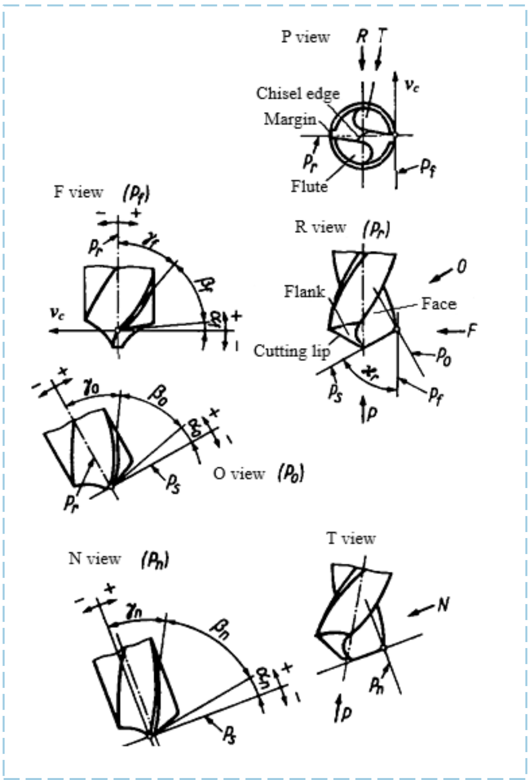


Figure 4-5. Design and main parts of drill bit (a- with cylindrical shank; b- with conical shank)



The auger is a double-edged tool with a complex edge geometry (Figure 4-6). The main edges are connected by a so-called cross, the presence of which is unfavorable in terms of chip design, but is unavoidable due to the structural design of the drill. The angle of the face along the cross is negative (approx. -60˚), then zero at the appearance of the main edge, and increasing along the diameter reaches the value of the slope (ω) of the groove. It is easy to see, therefore, that the value of the face angle is a function of the radial location of the groove bevel and the observed point of the edge, and therefore cannot be modified by sharpening. Screw drills are classified into groups N, H and W according to the groove angle.
Due to the already mentioned low stiffness, up to IT 12 dimensional accuracy and

Figure 4-6. Edge geometry of a drill bit

surface roughness R_a = 25-100 µm can be achieved with augers.

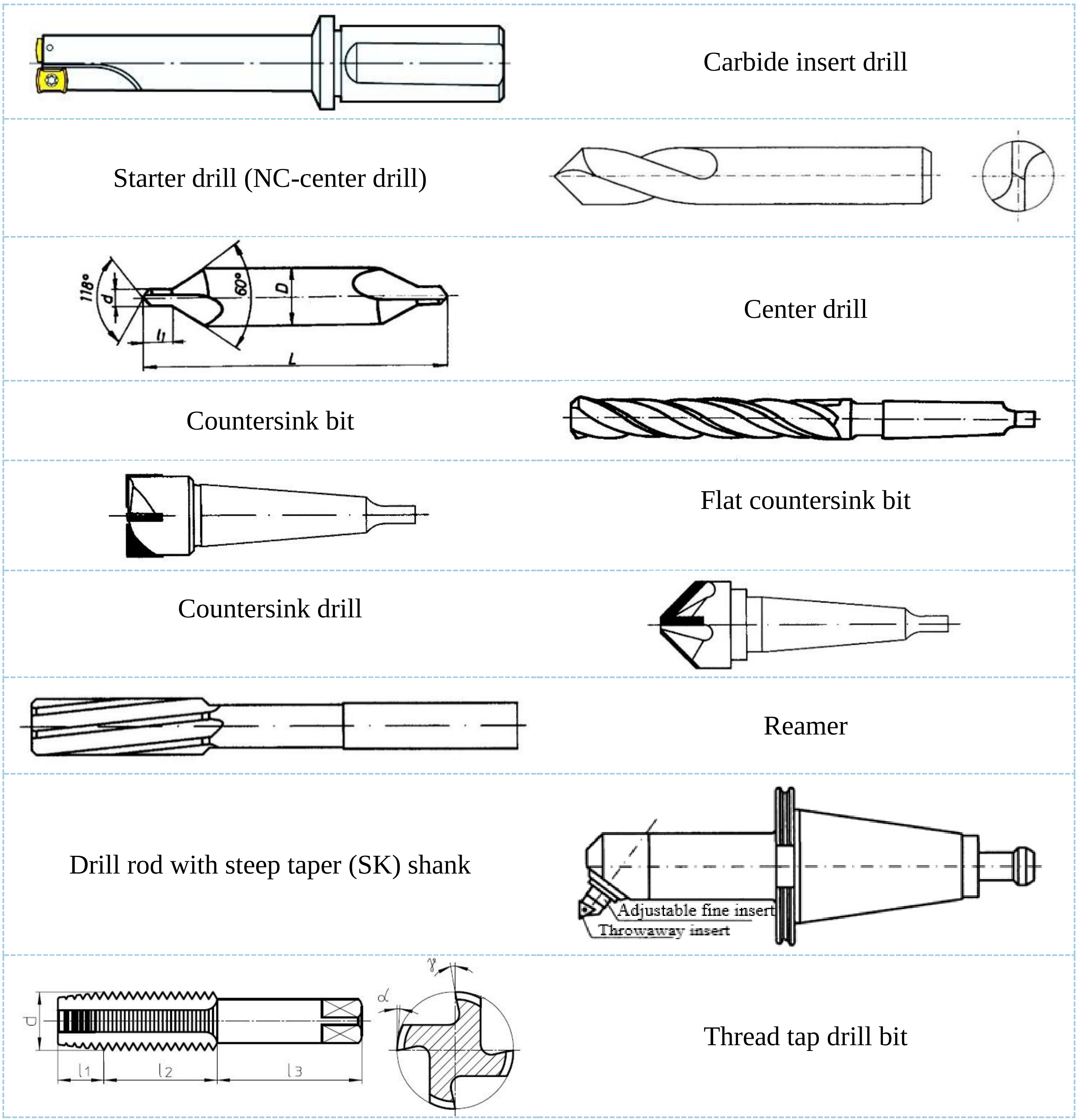


Figure 6-7. Hole Machining tools

4.3 DEFINITION OF THE CUTTING FORCE AND PERFORMANCE

4.3.1 THE CUTTING FORCE AND TORQUE DURING DRILLING AND HOLE WIDENING WITH TWIST DRILL BIT

The method of determining the cutting force discussed in point 1.5 can also be applied to drilling, ie the cutting force can be determined on the basis of the specific cutting force and the cross section of the chip. For chip drilling in full and drilling with a drill, shown in the Figures 4-8.

When drilling completely, the depth of cut is half the diameter of the drill: $a_p = \frac{D}{2}$

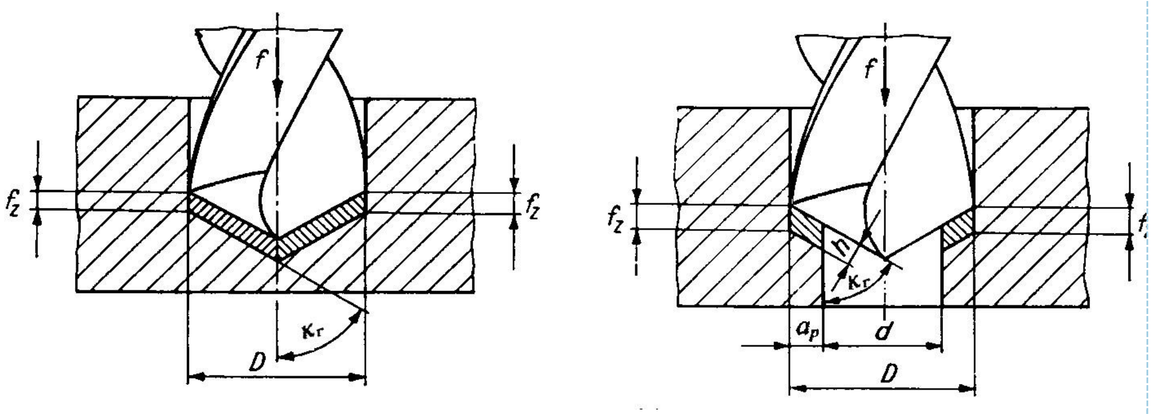
Depth of hole widening: $a_p = \frac{D-d}{2}$

For a double-edged drill, the feed per edge is half the feed per revolution:

$$f_z = \frac{f}{z} = \frac{f}{2}.$$

Chip thickness $h = f_z \cdot sin\kappa_r = \frac{f}{2} \cdot sin\kappa_r$;

Chip width $b = \frac{a_p}{sin\kappa_r} = \frac{D-d}{2 \cdot sin\kappa_r}.$



a) when drilling

b) hole widening

Figure 4-8. Chip cross section

The cutting force acting on one edge

$$F_c = k_c \cdot A = k_c \cdot a_p \cdot f_z \cdot K_F \text{ [N]}.$$

$$k_c = \frac{k_{c1\cdot1}}{h^m}$$

Where: $k_{c1.1}$ the main value of the specific cutting force

m exponent, their values are given in Table 1.1

$K_F = K_{FVB} \cdot K_{FM}$ the aggregate modification factor.

$K_{FVB}=1,25\ldots1,4$ wear factor.

K_{FM} process factor. $K_{FM} = 1$ for drilling and $K_{FM} = 0.95$ for hole widening [5].

Perpendicular to the cutting force is the (thickening) force in the direction of the chip thickness F_h, which can be divided into a force in the feed direction and a passive force (Figure 4-9). The force in the direction of the chip thickness can be determined by knowing the cutting force using a proportionality factor.

According to Condrons $F_h=(0,7\ldots0,9)F_c \approx F_c$

Taking both lives into account, the force in the direction of the feed:

$$F_f = 2 \cdot F_{f1} = 2 \cdot F_c \cdot sin\,\kappa_r$$

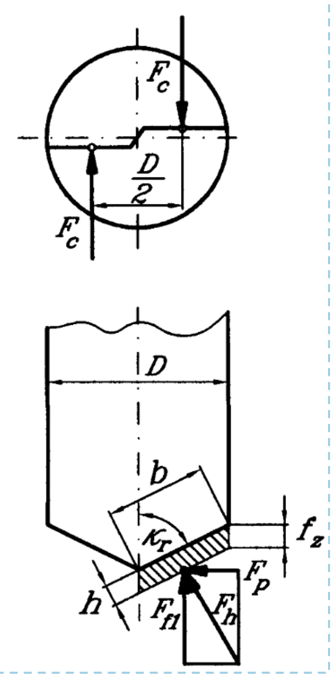


Figure 4-9. Drilling forces

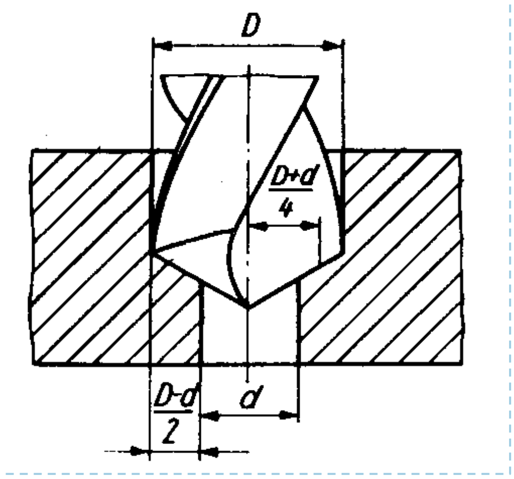


Figure 4-10. The power arm when drilling

The force F_f in the feed direction loads the drill under pressure and deflection, while the passive forces F_p neutralize each other at the two edges, and thus, in principle, the drill does not suffer a transverse load (bending). The Fc cutting forces at each edge form a force pair and expose the drill to torsional stress. Considering the distributed load on the edges as the concentrated force acting in the middle of the edge during torsional full drilling:

$$M = F_c \cdot \frac{D}{2} = k_c \cdot \frac{D^2 \cdot f}{8}$$

For hole widening (Figure 4-10.):

$$M = F_c \cdot \frac{D+d}{2} = k_c \cdot \frac{(D^2 - d^2) \cdot f}{8}$$

In practice, in the case of machining with a auger, empirical relationships are also used to determine the torque and feed force:

For drilling

$$M = C_M \cdot D^{w_M} \cdot f^{x_M}$$

$$F_f = C_F \cdot D^{w_F} \cdot f^{x_F}$$

For hole widening

$$M = C_M \cdot D^{w_M} \cdot f^{x_M} \cdot a_p^{y_M}$$

$$F_f = C_F \cdot f^{x_F} \cdot a_p^{y_M}$$

Where: $C_M, C_F, x_M, y_M, w_M, x_F, y_F, w_F$ torque and force constants as well as power exponents. Their values for some common substances are given in Table 4-1.

4-1. Table Torque, force constant and power exponents for high speed steel drills

Workpiece material	Drill						Hole widening						
	C_M	w_M	x_M	C_F	w_F	x_F	C_M	w_M	x_M	y_M	C_F	x_F	y_F
Alloyed steel, R_m=750 N/mm²	345	2,0	0,8	680	1,0	0,7	900	1,0	0,8	0,9	333	0,7	1,3
Stainless steel	370	2,0	0,7	1300	1,0	0,7							
Gray cast iron HB=190	210	2,0	0,8	427	1,0	0,8	850	1,0	0,8	0,75	235	0,4	1,2
Bronze HB=100-140	120	2,0	0,8	315	1,0	0,8							

For steels of different strengths, the values of CM and CF must be multiplied by a factor of

$$K = \left(\frac{R_m}{750}\right)^{0,75};$$

For castings of different hardness, the values of CM and CF must be multiplied by a factor of $K = \left(\frac{HB}{190}\right)^{0,6}$

4.3.2 THE CUTTING FORCE AND TORQUE DURING COUNTERBORING

In the case of countersinking, the cutting conditions are similar to those of drilling, so the force definition given in drilling can be applied here as well, provided that the tool always has more than two edges during countersinking. The one-way feed is always

$$f_z = \frac{f}{z}\ ,\qquad \text{where } z \text{ is the number of edges}$$

The chip thickness, which is determined by formula $h = f_z \cdot \sin\kappa_r$, must be taken into account when determining k_c. For flat countersinks $\kappa r = 90^\circ$, so here $h = f_z$. The value of the process characteristic can be taken as $K_{FM} = 1$ and the wear factor as $K_{FVB} = 1{,}3$.

The torque generated by the cutting forces is shown in Figure 4-11:

$$M = F_c \cdot z \frac{D + d}{4}$$

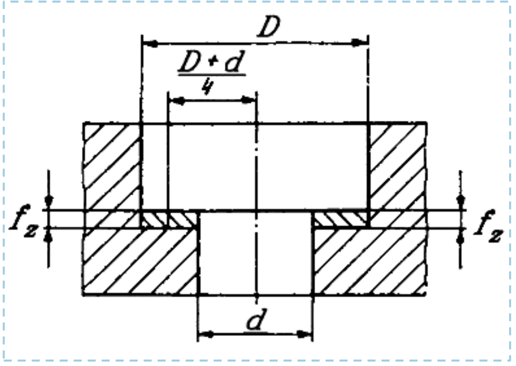


Figure 4-11. Determining the power lever when lowering

4.3.3 THE CUTTING FORCE AND TORQUE DURING REAMERING

In the case of friction, the calculation of the cutting force cannot be performed on the basis of the specific cutting force, because here the chip cross-section is very small and therefore the values of the forces resulting from friction are significantly higher than the forces resulting from the actual chip removal. In most cases, such a calculation is not necessary, or if it is, the cutting force can be calculated by measuring the torque or the power absorbed.

4.3.4 THE CUTTING FORCE AND TORQUE DURING TAPPING

The cutting force for tapping can also be determined on the basis of the specific cutting force, taking into account the specifics of this process. The cutting conditions of the tap are difficult to model accurately, so we use some simplified model that still gives usable results. For a machine tap where the length of the thread to be made is greater than the introductory section of the tap, the model shown in Figure 6-12 may be used.

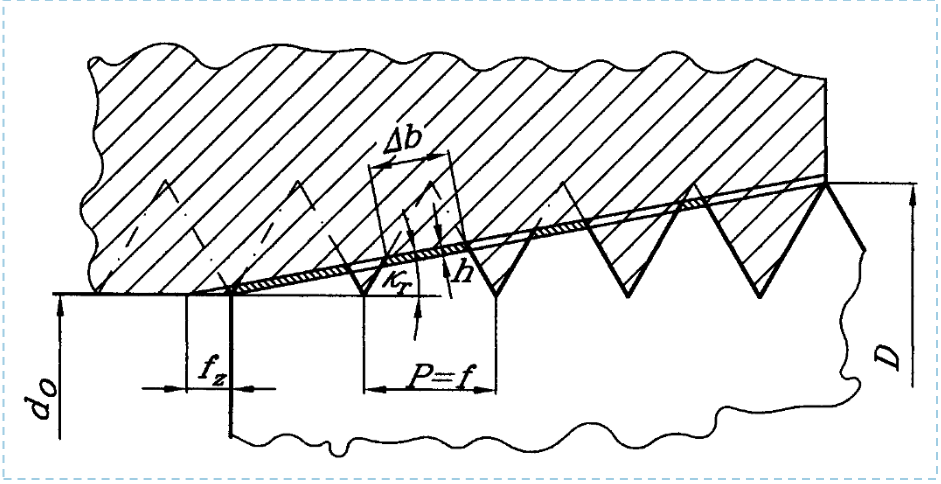


Figure 4-12. An edge chip cross section for tapping

The cutting force at one edge:

$$F_c = k_c \cdot A_1 \cdot K_F$$

Where: $k_c = \frac{k_{c1\cdot1}}{h^m}$; The values of $k_{c1.1}$ and *m* are given in 4.3 Table

$A_1 = h \cdot b$ the cross section of a chip

$h = f_z \cdot \sin\kappa_r = \frac{P}{z} \cdot \sin\kappa_r$

P the pitch

z the number of edges (edge-forming grooves)

$b = \frac{(D - d_o)}{4 \cdot \sin\kappa_r}$ the average chip width

With these terms: $A_1 = \frac{P}{4 \cdot z} \cdot (D - d_o)$

$K_F = K_{FVB}\, K_{FM}$ the aggregate modification factor

$K_{FVB} \approx 1{,}5$ the wear factor

K_{FM} process factor. In the case of cast iron machining, K_{FM} = 1.1. In the case of steel machining, the value of K_{FM} shall be determined on the basis of the diagram in Figure 4-13.

This is the term that defines the cutting force at the edge:

$$F_c = k_c \cdot \frac{P}{4 \cdot z} \cdot (D - d_o) \cdot K_{FVB} \cdot K_{FM}$$

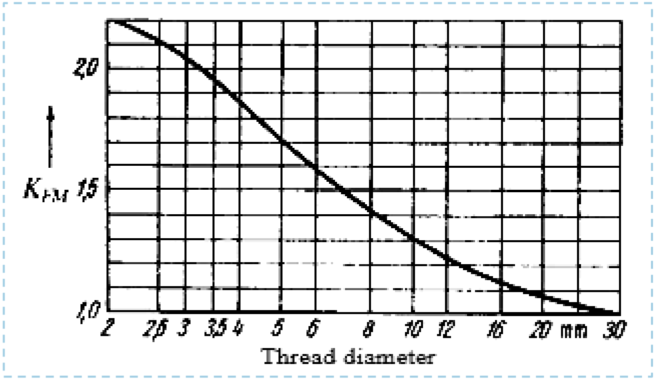


Figure 4-13. Process modification factor for tapping

Torque required for tapping:

$$M = z \cdot F_c \cdot \frac{D + d_o}{4}$$

4.3.5 THE PERFORMANCE FOR HOLE MACHINING

Knowing the torque and speed, the required cutting power can be determined:

$$P_c = M \cdot \omega = \frac{M \cdot n}{955400} \text{ [KW]}$$

Where: $\omega = \frac{2\pi \cdot n}{60}$ the angular velocity of the spindle

n the revolution per minute

The required power of the drilling machine is higher than this due to internal losses:

$$P_m = \frac{P_c}{\eta}$$

Where: η the efficiency of the machine. Its normal value is between 0,75 and 0,9.

4.4 DEFINITION OF CUTTING DATAS

The order in which the cutting data is determined is the same as for other cutting modes, but there are some features that apply to drilling.

4.4.1. DEFINITION OF THE DEPTH OF CUT

When drilling completely, the depth of cut is half the diameter of the drill

$$a_p = \frac{D}{2}.$$

Half the difference between the drilling diameter and the finished hole diameter before drilling, countersinking and rubbing in the case of drilling depth drilling, countersinking and rubbing

$$a_p = \frac{D-d}{2}$$

Drilling may be justified for two reasons:

(1) if, due to the size of the bore diameter, the workpiece cannot be drilled to a given diameter by a single full drill or the bore diameter is greater than 25 millimeters. In this case, we first use a smaller drill with a diameter $d \approx 0.6 \cdot D$.

(2) if the dimensional or positional accuracy or surface quality of the hole cannot be achieved by full drilling. In this case, the pre-hole diameter is determined by subtracting the allowance value from the finished size. If the accuracy of the hole cannot be achieved by drilling, the machining sequence for drilling, drilling and rubbing must be used.

In this case, the value of the allowance for rubbing must also be taken into account (Figure 4-14), that is

$$d \le D-(R_1+R_2)$$

Where:

R_1 allowance for hole widening

R_2 allowance for reamering

Their values can be found in the literature [18], [19].

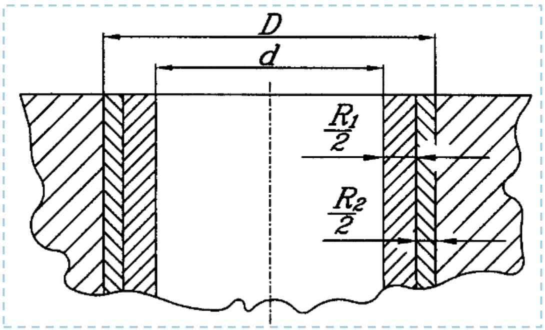


Figure 4-14. Allowances for hole widening and reamering

4.4.2 DEFINITON OF THE FEED

The maximum possible feed rate must also be used for drilling, and the cutting speed and drill speed must be determined taking into account the machine power, the kinematic limits of the machine and the cutting capacity of the tool.

When selecting the feedrate, the values recommended by the tool manufacturers or, failing that, the empirical values must be selected. Based on the feed selected in this way, the strength of the drill must be checked. The drill, torsion and compression are subjected to torque M and force F_f. Suitable torsional and compressive stresses:

$$\tau_{max} = \frac{M}{K_p}; \qquad \sigma = \frac{F_f}{A}$$

Where: A cross section of the drill

K_p the polar cross-sectional factor of the drill.

The cross-section of a auger is relatively complex, so to determine the cross-sectional factor, we use the simplified formula proposed by A. Avakov:

$$K_p \approx 0{,}0273 \cdot D^3$$

The torsional and shear stresses can be replaced by an ideal (equivalent) stress:

$$\sigma_i \approx 1{,}4 \cdot \tau_{max} = 1{,}4 \frac{M}{0{,}0273 \cdot D^3} = \frac{42 \cdot M}{D^3} \leq \sigma_{meg}$$

Permissible normal voltage for high speed steel drill $\sigma_{meg} = 300$ N/mm^2.

If the formula obtained for torque is replaced in the above expression, the maximum permissible value of the feedrate can also be expressed directly:

$$f \leq \frac{8 \cdot D \cdot \sigma_{meg}}{42 \cdot k_c},$$

or, if the exponent formula is used to determine torque:

$$f \leq \left(\frac{\sigma_{meg}}{42 \cdot C_M} \right)^{\frac{1}{x_M}} \cdot D^{\frac{3-w_M}{x_M}}$$

Informative cutting data can be found in the literature.

4.4.3 DEFINITION OF THE CUTTING SPEED

After determining the feedrate, the cutting speed and the spindle and tool speeds are determined.

In drilling, the cutting speed is the circumferential speed of the tool at its largest diameter in relation to the workpiece. The magnitude of the cutting speed is limited by the cutting ability of the drill and the performance of the drilling machine. Based on the cutting speed, the speed of the tool and the spindle can be calculated.

$$v_c = \frac{D \cdot \pi \cdot n}{1000} \text{ [m/min]} \qquad n = \frac{1000 \cdot v_c}{D \cdot \pi}$$

In practice, the cutting speed is often determined on the basis of experience and technology tables. Tool manufacturers also usually provide detailed tables for selecting cutting parameters.

A modern way to determine the cutting speed is to use exponential formulas based on the lifetime relationship.

Drilling cutting speed:

$$v_c = \frac{C_v \cdot D^{w_v}}{T^m \cdot f^{x_v}} \cdot K_v$$

4-2. Table The values of Cv are constant and the exponents xv, yv, wv, m for HSS drills

Workpiece material	Strength R_m [N/mm^2]	Feed f[mm/for.]	Drilling				Hole widening				
			C_v	x_v	w_v	m	C_v	x_v	y_v	w_v	m
	<560	≤ 0,2	6,3				12				

Steel, cast steel		>0,2	8,8	0,7	0,4	0,2		0,5	0,2	0,4	0,2
	600-700	≤ 0,2	5,7				10,9				
		>0,2	7,9								
	700-800	≤ 0,2	5,0				9,5				
		>0,2	7,0								
	800-900	≤ 0,2	4,5				8,6				
		>0,2	6,3								
	900-1000	≤ 0,2	4,1				7,8				
		>0,2	5,7								
Stainless steel			2,55-6,38	0,45	0,5	0,12	-	-	-	-	-
Cast iron	180-200 HB	≤ 0,3	10,5	0,55	0,25	0,125	15,2	0,4	0,1	0,25	0,125
		>0,3	12,2	0,4							
	200-220 HB	≤ 0,3	9,5	0,55			13,8				
		>0,3	11	0,4							
	220-240 HB	≤ 0,3	8,4	0,55			12,2				
		>0,3	9,8	0,4							
Bronze	100-140 HB	≤ 0,3	23,2	0,5	0,25	0,125					
		>0,3	23,4	0,4							

4-3. Table The values of the hole depth modification factor Kv

Hole depth, *l*	*l=3·D*	*l=4·D*	*l=5·D*	*l=6·D*	*l=8·D*	*l=10·D*
K_v	1,00	0,85	0,75	0,70	0,60	0,50

In the case of drilling, the formula for the cutting speed differs from the previous one in that the depth of cut ap must also be included here.

$$v_c = \frac{C_v \cdot D^{w_v}}{T^m \cdot f^{x_v} \cdot a_p^{y_v}} \cdot K_v$$

Where: C_v, x_v, y_v,w_v, m permanent and exponents (4-2. Table)

T [min] tool life

K_v hole depth modification factor (4-3. Table)

The cutting speed and speed determined on the basis of the tool life and the cutting ability of the tool must always be checked for machine power.

$$P_m = \frac{M \cdot n}{\eta \cdot 955400}, \qquad \text{from this} \qquad n \leq \frac{P_m \cdot \eta \cdot 955400}{M}$$

Based on the value obtained in this way, the first smaller standard speed that can be realized on the given machine is chosen.

4.5. DEFINITION OF THE MACHINE TIME

The machine main time required for cutting can be determined by knowing the cutting parameters, the dimensions of the tool and the hole.

a) definition of machine time during drilling and hole widening

$$t_g = \frac{L}{v_f} = \frac{L}{n \cdot f}$$

Where: $v_f = n \cdot f$ the feed rate

L the length of the toolpath.

Defining the tool path for full drilling (Figure 4-15.):

$$L = l + l_1 + l_2 \quad ,$$

l length of machining

$l_1 = x + 1 \ldots 2$ mm, the length of the tool run

$x = \dfrac{D}{2 \cdot tg\kappa_r}$, in the machining of steel and cast iron ($\kappa_r = 59°$) $x \approx 0{,}3 \cdot D$

$l_2 = 1 \ldots 2$ mm the length of the tool overrun. In the case of a blind hole $l_2 = 0$

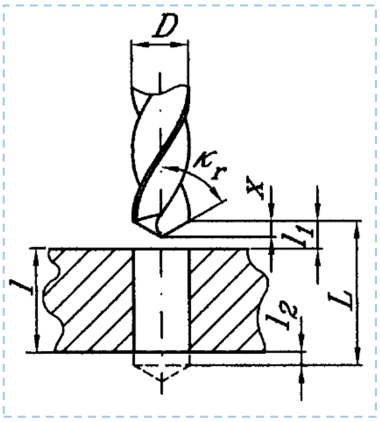


Figure 4-15. Defining the tool path for drilling

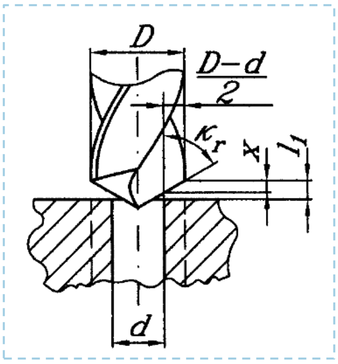


Figure 4-16. Determining the run-on for hole widening

Defining the tool path for hole widening (6-16. Figure):

$$L = l + l_1 + l_2 \quad ,$$

$l_1 = x + 1 \ldots 2$ mm, $x = \dfrac{D-d}{2 \cdot tg\kappa_r}$,

b) definition of machine time during counterboring and reamering

The formula for drilling is also valid here for determining the machine main time. In the case of flat lowering, the run-in length is $l_1 = 3\ mm$; overrun is not required or $l_2 = 0\ mm$.

When rubbing, the overrun and overrun together are equal to the tool diameter:

$$l_1 + l_2 = D, \quad \text{and} \quad L = l + D$$

When rubbing, the tool lift is equal to the feed rate of the session, so the same time as the main time must be taken into account.

5. MILLING

Milling, cutting with a circular main movement of a multi-edged tool (milling) and an advanced, feed movement of the workpiece. Separation of chips of variable cross-section is intermittent. It is used for the production of planes, grooves, log surfaces, round cylinders, but it is also suitable for creating any surface (profile milling, contour milling). It is a widespread process, as it is suitable for the production of all types of surfaces, from the simplest to the most complex. Its advantageous features are productivity and the relatively high accuracy that can be achieved.

5.1 THE VERSIONS OF MILLING

Milling has a large number of variants, which can be divided according to the shape of the machined surface, the shape of the tool and the kinematic conditions of the machining. An important feature of the cutting process is which part of the tool edge forms the machined surface. Accordingly, a distinction is made between **peripheral milling** and **face milling**. In the case of mantle milling, the machined surface is formed only by the main edges located on the milling mantle surface (Figure 5-1). In face milling, chip removal is also done with the main edges on the circumferential surface, but the machined surface is formed by the side edges on the face of the milling machine. There are also boundary cases (shape milling, thread milling, etc.) that can be classified into both groups.

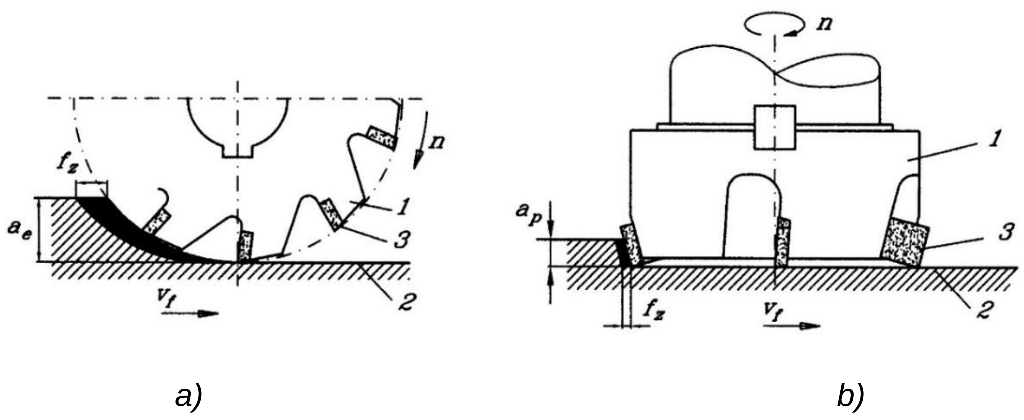


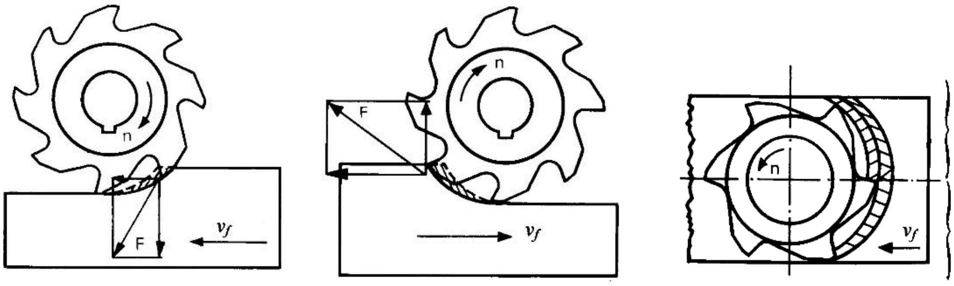
Figure 5-1. Comparison of slab milling and face milling: a) slab milling;

b) face milling;

1- tool, 2- workpiece, 3- cutting edge

Based on the relationship between the feed and the main movement, the milling can be

opposite (conventional) or **straight** (Figure 5-2). In the case of parallel milling, the direction of the circumferential speed of the edges associated with the workpiece and the direction of the side movement are “the same” (they have a component in the same direction). Here, the chip thickness is the largest at the edge. It has a better surface quality than straight milling, and a longer tool life, but can only be used on machines where there is no gap in the kinematic chain of the offset movement.



- *down milling*

- *up milling*

a b c

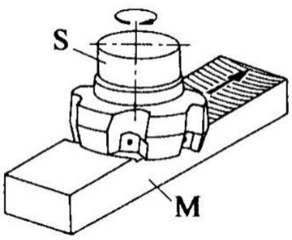
Figure 5-2. Down- and up milling: a) down milling; b) up milling; c) combined milling

In the case of counter-milling, the chip thickness at the edge entry is *h=0*, so in the initial phase the material is elastically deformed and significant friction occurs on the back surface of the tool. Depending on the relative position of the tool and the workpiece, face milling can also be performed in the opposite direction and in the same direction (combined milling).

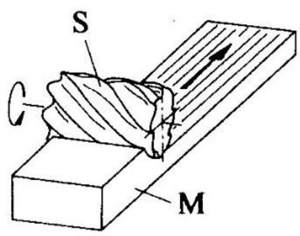
DIN 8589 classifies the milling variants into seven basic variants (Figure 5-3):

(1) face milling with face mill cutter, (2) face milling with slab mill cutter, (3) face milling with shell end mill cutter (angle milling), (4) profile milling, (5) thread milling, (6) hobbing,

(7) contour milling (sculpture surface milling).



Face milling with face mill cutter



Face milling with slab mill cutter

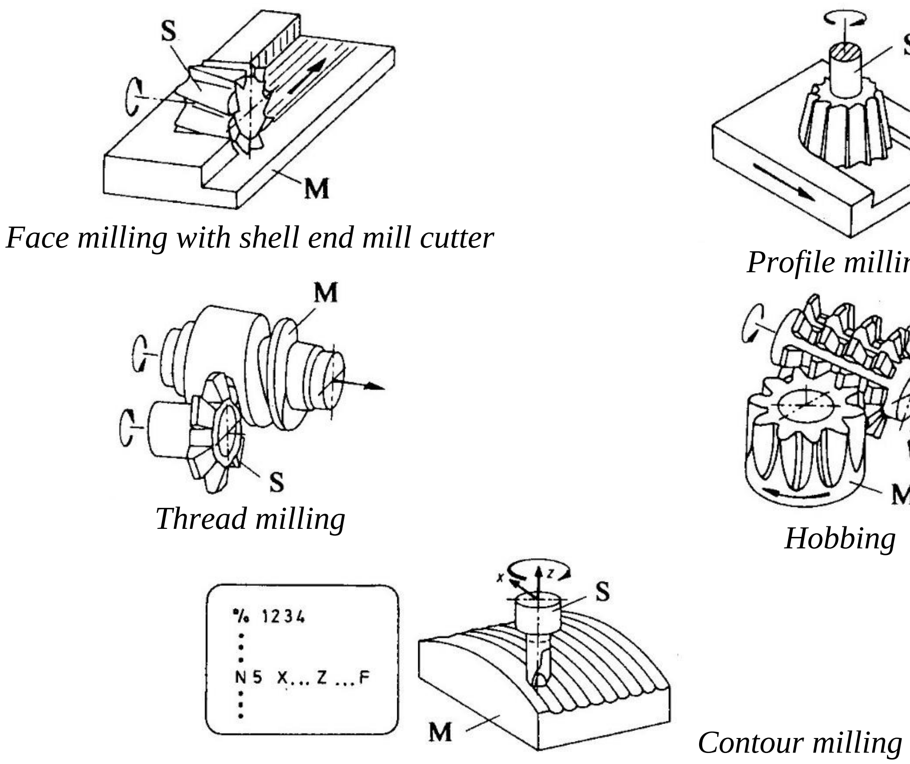
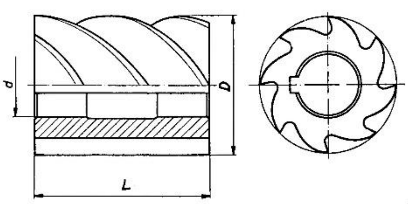
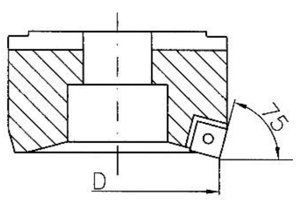


Figure 5-3. Basic milling variants according to DIN 8589 (M-workpiece, S-tool)

5.2 MILLING TOOLS

According to the construction and application characteristics, the following types of milling cutters are distinguished (Figure 5-4): Slab mill cutter, face mill cutter, shell end mill cutter, end mill cutter, slide and face cutter, radius mill cutter, profile mill cutter, thread mill cutter, hobbing cutter, etc.

In terms of tool material, we use high-speed steel and carbide cutters. For larger carbide cutters, only the edge is made of carbide. The edges can be attached to the tool body by soldering or mechanical clamping, which allows the inserts to be replaced after their edges have worn out. The share of carbide material and insert design shows a large and growing trend.

 *Slab mill cutter*	 *Face mill cutter*

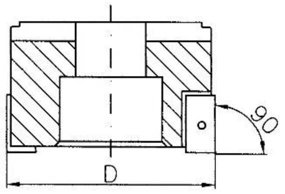
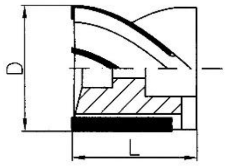
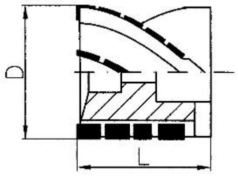
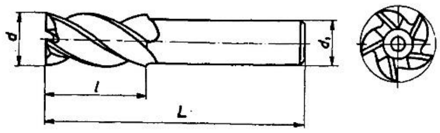
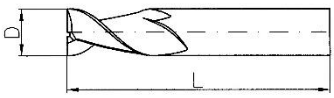
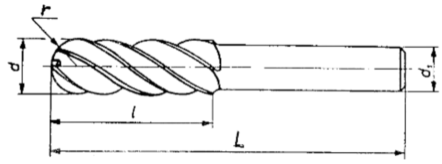
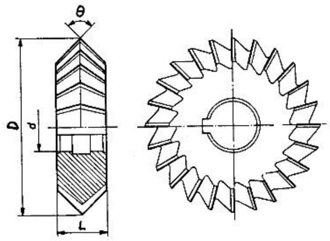
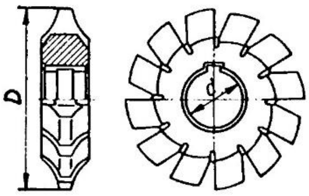
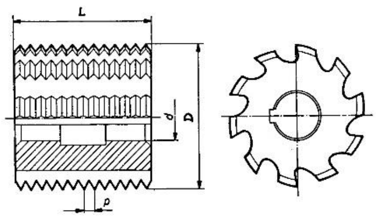
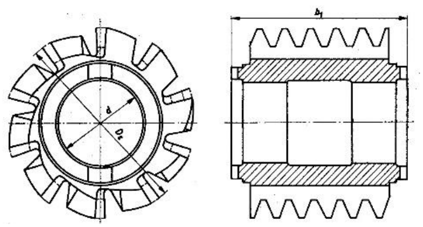
 *Corner an face milling cutter*	 *Soldered carbide insert face milling with shell end mill cutter*
 *Split edge roughing shell end mill cutter*	 *Roughing end mill*
 *End mill cutter*	 *Swage mill cutter*
 *Double angle milling cutters*	 *Involute gear cutter*
 *Comb-shaped thread cutter*	 *Hobbing cutter*

Figure 5-4. Typical milling tool types

5.2.1 THE GEOMERTY OF THE CUTTING EDGE OF THE MILLING TOOLS

For multi-edged tools, the angles and surfaces characteristic of a single-edged tool can be

determined on each edge (teeth) (Figure 5-5). From the point of view of the working part, the mantle end mill can be considered as a general case, here the edge design can be found on both the mantle and the face surface. The characteristic elements of the mantle end mill are

illustrated in Figure 5-6. Typical edge angles for face milling are shown in Figure 5-7.

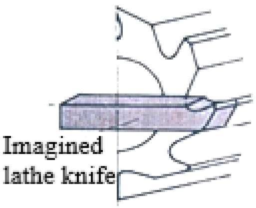


Figure 5-5. The single-edged and multi-edged tool comparison [14]

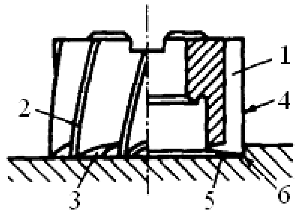


Figure 5-6. Shell end mill cutter is surfaces and edges: 1 – homloklap, 2 – hátlap, 3 – mellékhátlap, 4 – forgácsolóél, 5 – mellékél, 6 – élcsúcs

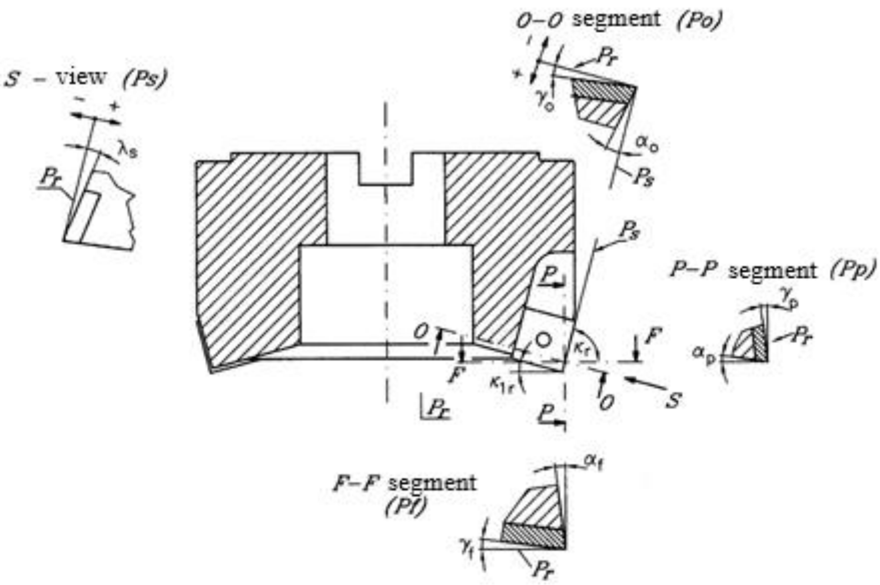


Figure 5-7. Mill cutter edge geometry

5.3 CUTTING FORCE, PERFORMANCE

5.3.1 CUTTING FORCE DURING FACE MILLING

In milling, chip removal is intermittent and the chip cross-section is constantly changing. The cutting force F can be decomposed into an active force F_a acting in the working plane and a passive force F_p perpendicular to the working plane. The direction of the active force changes as a function of the position angle φ (Figure 5-8). The active force can be decomposed into the main cutting force F_c in the direction of the cutting speed and the force in the direction of the chip thickness F_h perpendicular to it. According to Weilenmann, in the case of face milling, the cutting force can be determined following the following line of reasoning:

The instantaneous value of the cutting force acting on one edge of the cutter can be written according to the general force relationship:

$$F_{cz} = k_c \cdot A = k_c \cdot b \cdot h$$

The chip width in the above equation (Figure 5-9)

$$b = \frac{a_p}{\sin \kappa_r},$$

and the chip thickness

$$h = f_c \cdot \sin \kappa_r$$

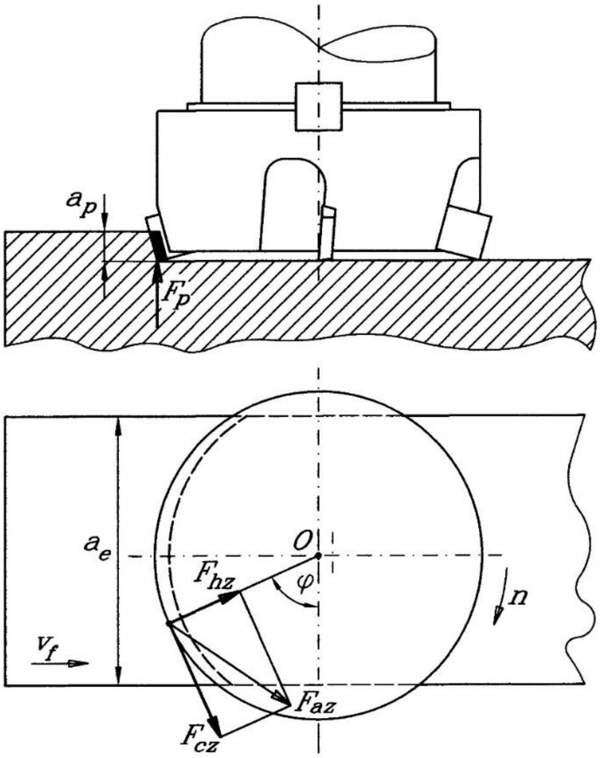


Figure 5-8. Forces acting on an edge in case of face milling

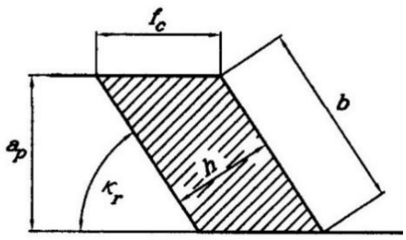


Figure 5-9. Theoretical chip cross section for face milling

As shown in Figures 5-10., the value of the radial feed is approximately

$$f_c \approx f_z \cdot \sin\varphi$$

where: φ the current edge angle

$$f_z = \frac{f}{z} = \frac{v_f}{n \cdot z}$$ the feed at one edge

$$f = f_z \cdot z = \frac{v_f}{n}$$ feed per revolution (mm/rev)

z the number of teeth of the cutter

$$v_f = f \cdot n = f_z \cdot z \cdot n$$ feed speed (mm/min)

This can be used to write the current value of the chip thickness as:

$$h = f_z \cdot \sin\varphi \cdot \sin\kappa_r$$

It is easy to see that the maximum value of the chip thickness will be at φ = 90˚

$$h_{max} = f_z \cdot \sin\kappa_r$$

This determines the maximum value of the cutting force acting on one edge, but the average value (average value) of the cutting force between the edge entry and the edge exit, which can be determined on the basis of the average chip thickness, is more important in the energy calculations.

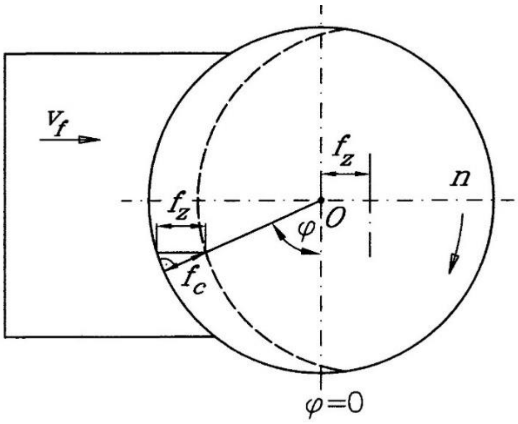


Figure 5-10. Machining conditions for face milling

The position angles of edge entry (φ_E) and edge exit (φ_A) and the switching angle Φ for face milling are shown in Figure 5-11.

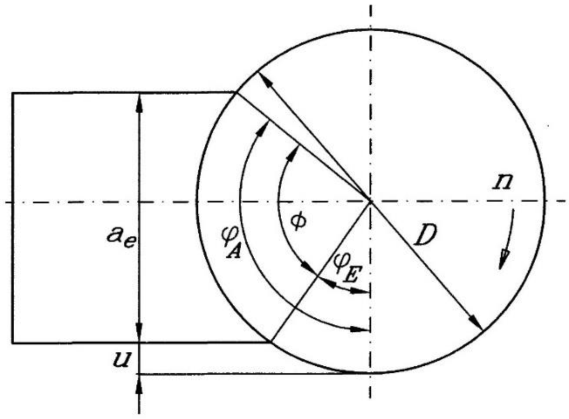


Figure 5-11. Edge entry angle (φ_E), exit angle (φ_A) and switch number Φ for face milling

Plotting the chip thickness as a function of the position angle φ gives the curve shown in Figures 5-12. The area under the curve is equal to the area of the rectangle formed by the base length and the average thickness, where the average chip thickness

$$h_m = \frac{1}{\Phi}\int_{\varphi_E}^{\varphi_A} h \cdot d\varphi = \frac{f_z \cdot \sin\kappa_r}{\Phi}\int_{\varphi_E}^{\varphi_A} \sin\varphi \cdot d\varphi = \frac{1}{\Phi} f_z \cdot \sin\kappa_r (\cos\varphi_E - \cos\varphi_A)$$

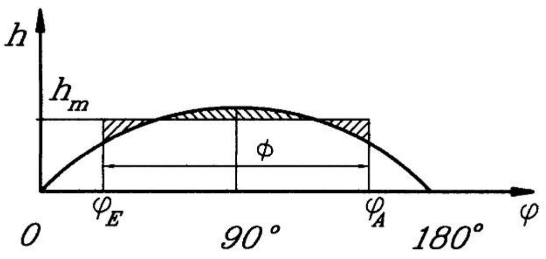


Figure 5-12. Variation of chip thickness as a function of position angle φ

The cosines of the edge entry and edge exit angles using the notations in Figure 5-11:

$$\cos\varphi_E = \frac{\frac{D}{2}-u}{\frac{D}{2}} = 1-\frac{2u}{D}, \qquad \cos\varphi_A = 1-\frac{2(a_e+u)}{D}$$

This is the average chip thickness:

$$h_m = \frac{1}{\Phi} f_z \cdot \sin\kappa_r \cdot \frac{2a_e}{D}$$

In the above formula, the value of the switching angle Φ must be replaced in radians.

The average cutting force acting on an edge:

$$F_{czm} = k_c \cdot b \cdot h_m = k_c \cdot a_p \cdot \frac{f_z}{\Phi} \cdot \frac{2a_e}{D}$$

The average chip thickness is taken into account when determining the specific cutting force.

$$k_c = \frac{k_{c1.1}}{h_m^m}$$ ($k_{c1.1}$ and m are shown in 1.3 Table).

Secondary factors influencing the cutting force are also taken into account during milling with the total correction factor [5].

$$K_F = K_{Fv} \cdot K_{F\gamma o} \cdot K_{FVB} \cdot \ldots$$

Where:

K_{Fv} speed modification factor, its value can be determined according to Figure 1-24

$K_{\gamma_o} = 1 - \frac{\gamma_o - \gamma_k}{66,7}$ face angle modification factor. γ_o is the face angle of the cutter,

The value of γ_k is 6° for steel and 2° for cast iron.

$K_{FVB}=1,2\ldots1,4$ tool wear factor (according to Weilenmann).

Thus, it is the mean value of the cutting force acting on one edge

$$F_{czm} = k_c \cdot b \cdot h_m \cdot K_F$$

The circumferential force acting on the cutter (cutting force) can be determined by multiplying the force acting on one edge by the number of connected edges.

$$F_{cm} = F_{czm} \cdot i$$

Where: $i = \frac{\Phi}{\tau}$ number of teeth cutting simultaneously (switch number)

$\tau = \frac{360}{z}$ dividing the milling angle,

The value of the (deepening) force in the direction of the chip thickness F_h perpendicular to the main cutting force can be determined in a similar way as the main cutting force [8]:

$$F_{hzm} = k_h \cdot b \cdot h_m$$

where the $k_h = \frac{k_{h1 \cdot 1}}{h^{m_h}}$ the specific deepening force.

$k_{h1.1}$ the main value of the specific dredging force

m_h exponent

In practice, the force F_h can be determined by the force factor F_c.

$$F_{hzm} = \lambda_{ch} \cdot F_{czm}$$

As a result of the cutting force and the forces in the direction of the chip thickness, the activator acting in the working plane can be determined (Figure 5-8): $F_a = \sqrt{F_c^2 + F_h^2}$

The active force can be divided into components in the feed direction and in the direction perpendicular to it, and thus the force and power required for the feed can be determined.

5.3.2 CUTTING FORCE DURING SLAB MILLING

The relationship between the tool and the workpiece for counter-milling with a straight-edge milling cutter is illustrated in Figure 5-13. The edge entry takes place at point P, where the chip thickness is zero. The thickest chip (h_{max}) will be at the edge exit. The general relationships laid out for face milling will therefore be simpler than for face milling:

$\kappa_r=90^\circ$, $\varphi_E=0$, $\Phi=\varphi_A$, $a_p=b_w$

$$h = f_c = f_z \cdot \sin\varphi$$

$$h_{min} = 0; \qquad h_{max} = f_z \cdot \sin\Phi$$

$$\cos\Phi = \frac{\frac{D}{2}-a_e}{\frac{D}{2}} = 1-\frac{2\cdot a_e}{D}; \qquad \sin\Phi = \sqrt{1-\cos^2\Phi} = \sqrt{1-\left(1-\frac{2a_e}{D}\right)^2} \cong 2\sqrt{\frac{a_e}{D}}$$

This has the largest chip thickness: $h_{max} = 2 f_z \sqrt{\frac{a_e}{D}}$

Mean chip thickness:

$$h_m = \frac{1}{\Phi} f_z \cdot (\cos\varphi_E - \cos\varphi_A) = \frac{1}{\Phi} f_z \cdot (1-\cos\Phi) = \frac{f_z}{\Phi}\cdot\frac{2a_e}{D}$$

For small angles (Φ<30˚) $\Phi \approx \sin\Phi = 2\sqrt{\frac{a_e}{D}}$

And this is the mean value of the chip thickness $h_m = f_z\sqrt{\frac{a_e}{D}}$

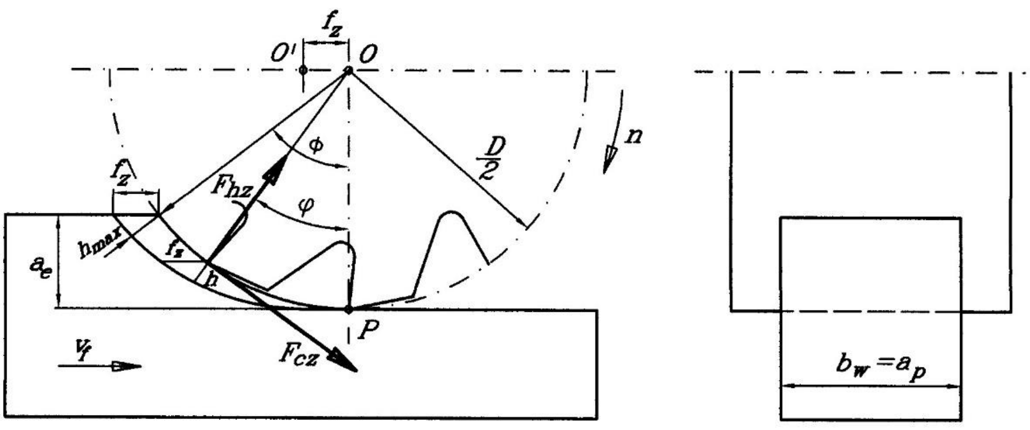


Figure 5-13. Cutting force and chip cross section for face milling

Knowing the maximum and average chip thickness, the maximum and average cutting forces can be determined:

$$F_{cz\,max} = k_c \cdot h_{max} \cdot b_w; \qquad F_{czm} = k_c \cdot b_w \cdot f_z \cdot \sqrt{\frac{a_e}{D}}; \qquad F_{cz\,min} = 0$$

Considering that in addition to the variable chip thickness, the number of connected edges (switch number) also fluctuates, it can be concluded that milling is a highly dynamic process. When the number of switches is greater than one, the cutting force fluctuates between a lower and an upper limit. If the number of switches is less than one, the cutting force varies between zero and a maximum value.

In practice, the other force components are expressed in terms of the main cutting force F_c. In face milling, the force in the direction of the chip thickness approaches the value of the main cutting force. In practice, the following ratios can be used with slight variations [24]:

- in the case of reverse milling $F_h \approx (0{,}6 \cdots 0{,}8) \cdot F_c$
- in the case of directional milling $F_h \approx (0{,}35 \cdots 0{,}6) \cdot F_c$

Based on the largest values, the resulting active force can be written with the following relationships:

for reverse milling $F = \sqrt{F_c^2 + F_h^2} = \sqrt{F_c^2 + (0{,}8 \cdot F_c)^2} = 1{,}28 \cdot F_c \cong 1{,}28 \cdot F_{czm} \cdot i$

for directional milling $F = \sqrt{F_c^2 + (0{,}6 \cdot F_c)^2} = 1{,}16 \cdot F_c \cong 1{,}16 \cdot F_{czm} \cdot i$

Force ripple can be significantly reduced by using a bevel milling cutter. Here, the entry and exit take place gradually along the width of the cutter. In the case of an oblique edge milling machine, there is also a force component acting in the axial direction of the milling machine, which can be considered as a passive force:

$$F_{ax} = F_c \cdot tg\lambda_s$$

Where λ_s the angle of the edge deflection.

5.3.3 CUTTING AND MACHINE PERFORMANCE

The power required for cutting can be determined from the average cutting force and cutting speed:

$$P_c = \frac{F_{cm} \cdot v_c}{1000 \cdot 60} \text{ [kW]}$$

Implement performance is higher due to in-machine losses:

$$P_m = \frac{P_c}{\eta}$$

The efficiency of the machine is between $\eta = 0,6\ldots 0,8$.

5.4 DEFINITION OF THE CUTTING DATAS

5.4.1 DEFINITION OF THE DEPTH OF CUT

The size of the depth of cut is mainly due to the size of the allowances, ie the size of the preform and the size after machining.

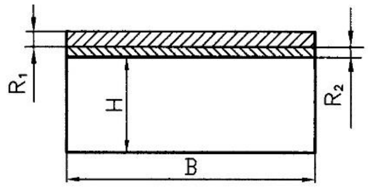
In some cases, depending on the thickness of the layer of material to be deposited, a tooth division must be performed during roughing. Number of cuts

$$i = \frac{R}{a_{max}}$$ rounded to the nearest whole number

Where: R the value of the allowance (Figure 5-14.)

a_{max} the maximum allowable depth of cut

Finishing is always done in one pass.



R_1 - allowance for roughing
R_2 - allowance for smoothing

Figure 5-14: Allowance for milling

5.4.2 DEFINITION OF THE FEED

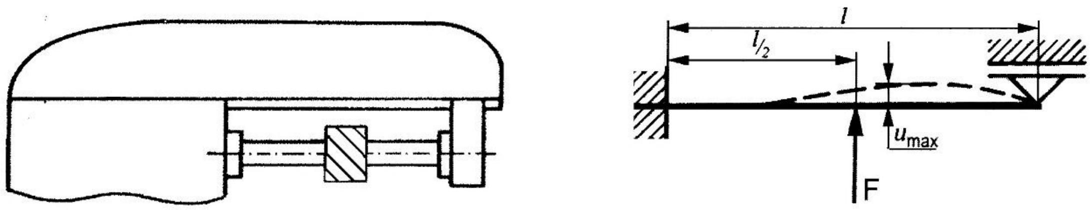
In the case of milling, the feed rate per edge is decisive, so you must first determine this and calculate the feed rate per revolution and the feed rate. It is advisable to select the feed per tooth to the value recommended by the tool manufacturers. Otherwise, empirical values or data from technology tables may be used. Guideline values for cutting speed and feed per tooth can be found in the literature.

The selected feedrate must be checked on the basis of the data for the specific machining operation. The inspection may cover: (1) deformation of certain parts of the machinery; (2)

to deform the tool; (3) deformation of the workpiece; (4) the roughness of the machined surface, etc.

Checking the milling shaft is bend

In the case of a horizontal spindle milling machine, the milling shaft can be considered as a clamped support at one end, which suffers from elastic deformation due to the cutting force (Figure 5-15). On a given machine and for a given tool, bending is a function of force. The cutting force, on the other hand, is a function of the feedrate, so a relationship can be written between the feedrate and the bending of the milling shaft. If the permissible bending of the milling shaft is known, the value of the maximum permissible feed per tooth can be determined from this relation.



a) milling shaft *b) mechanical model of the shaft*

Figure 5-15. Mechanical model of the milling shaft

The greatest deflection

$$u_{\max} = \frac{F \cdot l^3}{110 \cdot E \cdot I}, \text{ or } \qquad F \le \frac{u_{\max} \cdot 110 \cdot E \cdot I}{l^3}$$

Where: *E* modulus of elasticity of the milling shaft

I the second order torque of the cross section

If we accept that the force acting on the tool

$$F = \sqrt{2} \cdot F_c \cong \sqrt{2} \cdot F_{czm} \cdot i = \sqrt{2} \cdot i \cdot k_c \cdot b_w \cdot f_z \cdot \sqrt{\frac{a_e}{D}},$$

then the maximum feed per tooth can be calculated from the above relationships

$$f_z \le \frac{u_{max} \cdot 78 \cdot E \cdot I}{l^3 \cdot i \cdot k_c \cdot b_w \cdot \sqrt{\frac{a_e}{D}}}$$

The milling shaft is allowed to bend during roughing $u_{max} \le$ 0,20 mm, and in the case of smoothing $u_{max} \leq$ 0,05 mm.

Checking the surface roughness

For face milling, the theoretical roughness is a function of feed and milling diameter. It can be calculated in a similar way as for a single-edged tool. Due to the geometric defect of the cutter, a tooth that operates at the largest diameter “erases” the traces of the previous teeth at each turn, so it is not the feed per tooth that is considered to be the determinant, but the feed per turn (Figure 5-16).

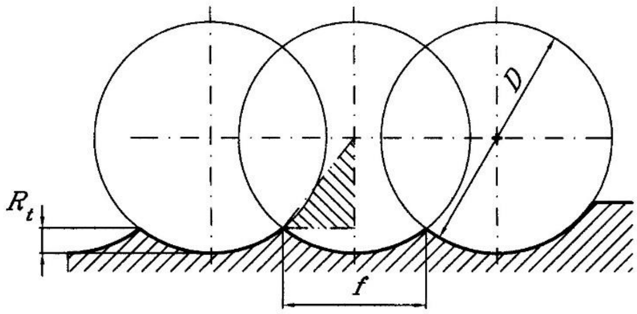


Figure 5-16. Theoretical roughness model for face milling

$$R_t = \frac{f^2}{4D} \quad \text{and from that } f \leq \sqrt{4D \cdot R_t}$$

True roughness is different, but the correlation shows well how feed and milling diameter affect roughness.

Some authors suggest the introduction of a modifying factor that results in the following context (Stankovic):

$$R_t = 1{,}5 \cdot \frac{f^2}{4D} \quad \text{and like this} \quad f \leq \sqrt{2{,}7 \cdot D \cdot R_t}$$

For face milling, if the transition between the main edge and the side edge is made with a radius of curvature or the edges of the milling head are crossed by round inserts, the roughness is determined by the feed per tooth and the apex radius of the edge (insert) (Figure 5-17): $R_t = \frac{f_z^2}{8 \cdot r_\varepsilon}$

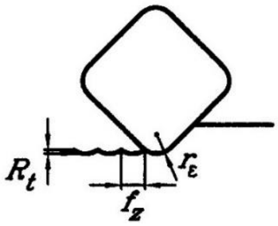


Figure 5-17.

5.4.3 DEFINITION OF THE CUTTING SPEED AND THE REVOLUTION

In milling, the cutting speed is the circumferential speed of the milling machine.

$$v_c = \frac{D\pi \cdot n}{1000} \quad \text{from here the cutter speed} \quad n = \frac{1000 \cdot v_c}{D\pi}$$

The values recommended by experience and tool manufacturers are selected for the cutting speed either from tables or calculated using empirical formulas based on the service life

relationship.

$$v_c = \frac{C_v \cdot D^{w_v}}{T^m \cdot f^{x_v} \cdot a_p^{y_v} \cdot a_e^{q_v} \cdot z^{u_v}} \cdot K_v$$

Where: C_v, x_v, y_v, m, q_v, u_v , w_v permanent and exponents (Table 5-4.)

T [min] tool life. Milling cutters are expensive tools, their replacement is time consuming, so they choose a longer service life (T = 120… 420 min) than turning

lathes.

$K_v = K_{vm} \cdot K_{v\kappa} \cdot K_{vvo} \cdot K_{vH}$ aggregate speed modification factor

K_{vm} a modification factor that takes into account the material quality of the workpiece

$K_{v\kappa}$ main edge placement angle modification factor

$K_{v\kappa o}$ milling variant modification factor $K_{v\kappa o} = 1$ for reverse milling;

K_{VKO} ≈ *1,1…1,25* for straight milling (higher values are greater than chip thickness)

K_{vH} coolant and lubricant application modifier. $K_{vH}=1$ in case of refrigerant application and $K_{vH}=0,8$ if no cooling is used

Table 5-4. C_v is constant and exponents in the velocity formula [11], [24]

Milling version	Tool material	Workpiece material	Cutting data			Value of constants and exponents						
			f_z mm	a_p mm	a_e mm	C_v	w_v	m	y_v	x_v	q_v	u_v
Flat face milling	Carbide	Steel R_m=650				382	0,2	0,2	0,1	0,4	0,2	0
		Cast iron HB 180	_	_	_	396	0,2	0,32	0,15	0,35	0,2	0
	HSS	Steel R_m=650	≤0,1	_	_	73,5	0,25	0,2	0,1	0,2	0,15	0,1
			>0,1	_	_	46,5	0,25	0,2	0,1	0,4	0,15	0,1
		Copper alloys HB=150-200	≤0,1	_	_	95,0	0,25	0,2	0,1	0,2	0,15	0,1
			>0,1	_	_	60,4	0,25	0,2	0,1	0,4	0,15	0,1
Flat slab milling	Carbide	Steel R_m=650	_	≤2	≤35	448	0,17	0,33	0,19	0,28	-0,05	0,1
			_	>2		510	0,17	0,33	0,38	0,28	-0,05	0,1
			_	≤2	>35	708	0,17	0,33	0,19	0,28	0,08	0,1
			_	>2		805	0,17	0,33	0,38	0,28	0,08	0,1
		Cast iron HB 180	≤0,2	≤2,5	_	796	0,37	0,42	0,13	0,19	0,23	0,14
			>0,2		_	507	0,37	0,42	0,13	0,47	0,23	0,14
			≤0,2	>2,5	_	1018	0,37	0,42	0,13	0,19	0,23	0,14
			>0,2		_	647	0,37	0,42	0,13	0,47	0,23	0,14
	HSS		≤0,1	_	_	62,5	0,45	0,33	0,3	0,2	0,1	0,1

		Steel R_m=650	>0,1	–	–	40,2	0,45	0,33	0,3	0,4	0,1	0,1
		Cast iron HB 180	≤0,15	–	–	60,5	0,7	0,25	0,5	0,2	0,3	0,3
			>0,15	–	–	28,3	0,7	0,25	0,5	0,6	0,3	0,3
		Copper alloys HB=150-200	≤0,1	–	–	80,8	0,45	0,33	0,3	0,2	0,1	0,1
			>0,1	–	–	52,0	0,45	0,33	0,3	0,4	0,1	0,1
Groove milling with side milling cutter	Carbide	Steel R_m=650	≤0,06	–	–	2000	0,2	0,35	0,3	0,12	0,1	0
			>0,06	–	–	757	0,2	0,35	0,3	0,4	0,1	0
	HSS	Steel R_m=650	–	–	–	77,8	0,25	0,2	0,3	0,2	0,1	0,1
		Cast iron HB 180	–	–	–	75,5	0,2	0,15	0,5	0,4	0,1	0,1
		Copper alloys	–	–	–	100,8	0,25	0,2	0,3	0,2	0,1	0,1
Slab-face milling with end mill cutter	Carbide	Steel R_m=650	–	–	–	262	0,44	0,37	0,24	0,26	0,1	0,13
	HSS	Steel R_m=650	–	–	–	53	0,45	0,33	0,5	0,5	0,1	0,1
		Cast iron HB 180	–	–	–	75,5	0,7	0,25	0,5	0,2	0,3	0,3
		Copper alloys	–	–	–	72	0,45	0,33	0,3	0,2	0,1	0,1
Profile milling	HSS	Steel R_m=650	–	–	–	60,2	0,45	0,33	0,3	0,2	0,1	0,1
Groove milling with end mill cutter	HSS	Steel R_m=650	–	–	–	13,6	0,3	0,26	0,3	0,25	0	0

In addition to the listed modifiers, other aspects such as tool angles, tool material quality, etc. may also play a role.

The value of the speed calculated from the cutting speed may be limited by the power of the machine, so this should be checked and, if necessary, the speed reduced accordingly. Implement performance:

$$P_m = \frac{F_{cm} \cdot v_c}{1000 \cdot 60 \cdot \eta} \text{ [kW], from this} \qquad v_c \leq \frac{60000 \cdot P_m \cdot \eta}{F_{cm}}$$

or $$n \leq \frac{60 \cdot 10^6 \cdot P_m \cdot \eta}{F_{cm} \cdot D \cdot \pi}$$

5.5 DEFINITION OF THE MACHINE TIME

Knowing the cutting parameters and the dimensions of the workpiece and the cutter, the machine main time required for cutting can be determined for each operation.

$$t_g = \frac{L}{v_f} \cdot i$$

Where: $v_f = n \cdot f = n \cdot f_z \cdot z$ the feed rate, i number of cuts

L the length of the toolpath.

Determining the tool path for slab milling

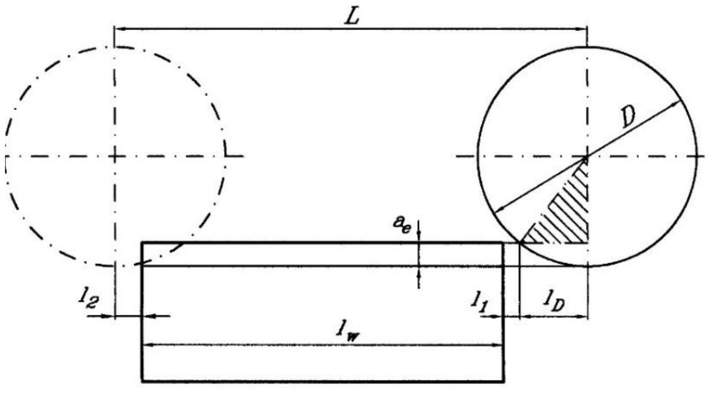


Figure 5-18. Determining the tool path for slab milling

$$L = l_w + l_1 + l_2 + l_D \quad ,$$

Where: l_w length of machining

$l_1 = l_2 \approx 2 \ldots 3\ mm$ tool run-on and overrun length

$l_D = \sqrt{\left(\frac{D}{2}\right)^2 - \left(\frac{D}{2} - a_e\right)^2} = \sqrt{a_e(D - a_e)}$ run-on length due to milling diameter and depth of cut.

Determining the tool path for face milling

In the case of roughing, the tool path can be determined as shown in Figure 5-19:

$$L = \frac{D}{2} + l_w + l_1 + l_2 - l_D$$

Where: $l_D = \sqrt{\left(\frac{D}{2}\right)^2 - \left(\frac{D}{2} - u\right)^2} = \sqrt{u(D - u)}$. u is the size that determines the position of the cutter

In the case of face milling, the full diameter of the cutter must extend beyond the machined surface (Figure. 5-20) and consequently the tool path:

$$L = D + l_w + l_1 + l_2$$

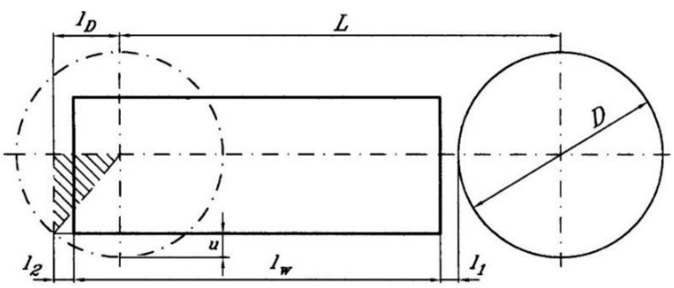


Figure 5-19. Determining the tool path for rough face milling

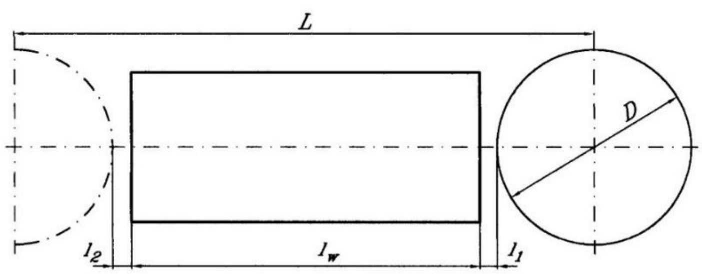


Figure 5-20. Determining the tool path for smooth face milling

5.6 A WORKPIECE IS FIXATION ON THE MILLING MACHINE

The work surface of milling machine tables is usually provided with “T” grooves to secure the clamping device or the workpiece directly to the machine table.

On milling machines, due to the variety of the workpiece and the variety of milling variants, the gripping can also be very diverse. Relatively simple column-like pieces are clamped with a machine vise (Figure 5-21). Clamping more complex pieces often requires a special clamping device. The workpiece can sometimes be clamped directly to the machine table using clamps (Figure 5-22). In cases where the same machining is to be performed at several points on a workpiece, a splitter is used. A common accessory for milling machines is the universal splitter head, which is suitable for all kinds of splitting tasks.

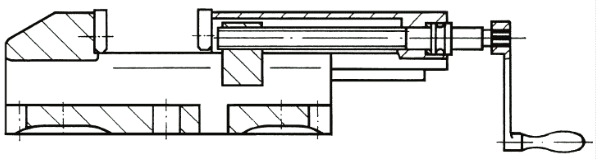


Figure 5-21. Sectional drawing of a machine vise (Röhm)

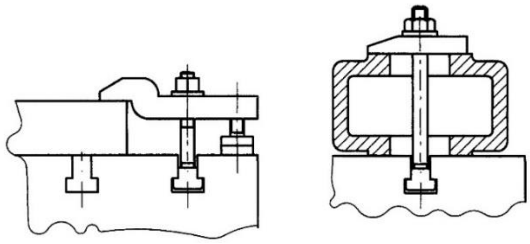


Figure 5-22. Use of clamps

6. GRINDING

Grinding is a multi-faceted machining operation with a tool consisting of a binder and abrasive grains randomly placed in it. The shape of the edges is geometrically indeterminate. The edges are not in constant contact with the workpiece and penetrate the surface of the workpiece only insignificantly relative to the size of the grain. It includes abrasive machining. It is used in the mechanical engineering industry to increase the dimensional accuracy and surface quality of parts, as well as to machine heat-treated, high-hardness workpieces.

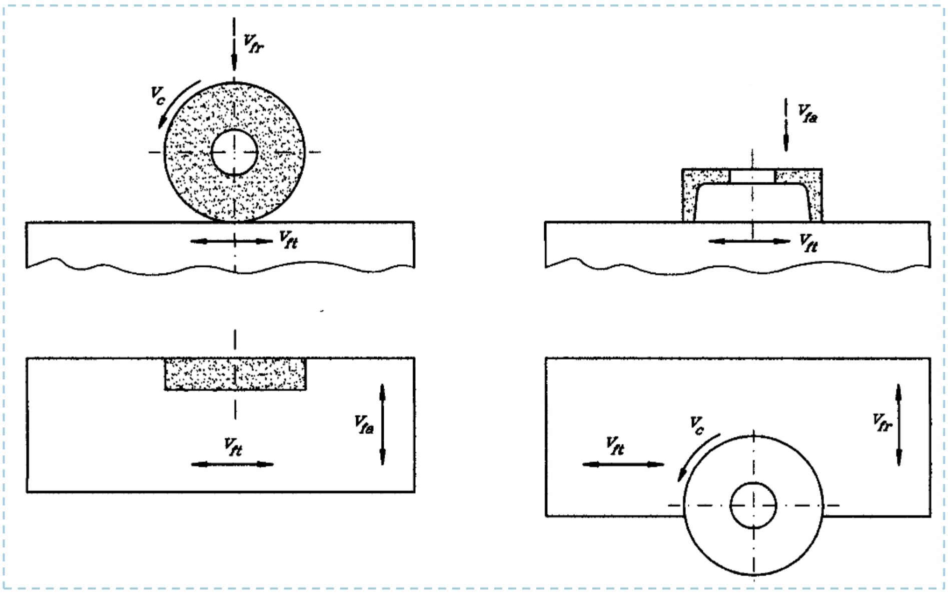
6.1 THE VERSIONS OF GRINDING

Depending on the shape of the surface to be machined, the relative position of the workpiece and the tool and the movement conditions, several grinding variants can be distinguished.

SURFACE GRINDING is used to machine flat surfaces. The main rotating movement is performed by the tool (grinding wheel) and the secondary movements by the workpiece. Machining can be done with the circumferential surface or the front surface of the disc (Figure 5-1).

By GRINDING, the outer cylindrical surfaces of rotating body-like parts are machined. Surface grinding of a clamped workpiece (most commonly between tips) can be accomplished by a longitudinal feed or insertion process (Figure 5-2). In the case of longitudinal feed grinding between the tips, the grinding wheel performs the main movement and the auxiliary movement in the gripping direction, while the workpiece performs a rotational and auxiliary movement parallel to the axis of symmetry. In the case of insert grinding, the workpiece does not move longitudinally, the disc penetrates the surface of the workpiece in the radial direction.

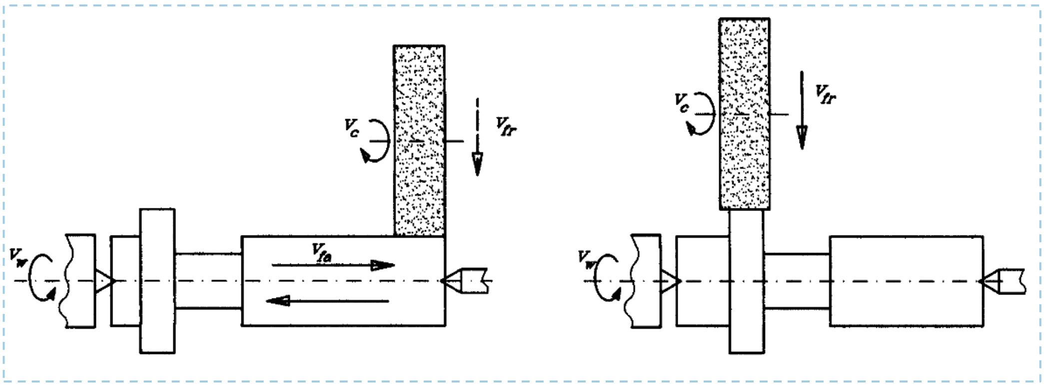
Internal surface surfaces are machined by HOLE GRINDING. This can also be done by longitudinal feed or insertion (Figure 5-3).



a) b)

Figure 5-1. Surface grinding

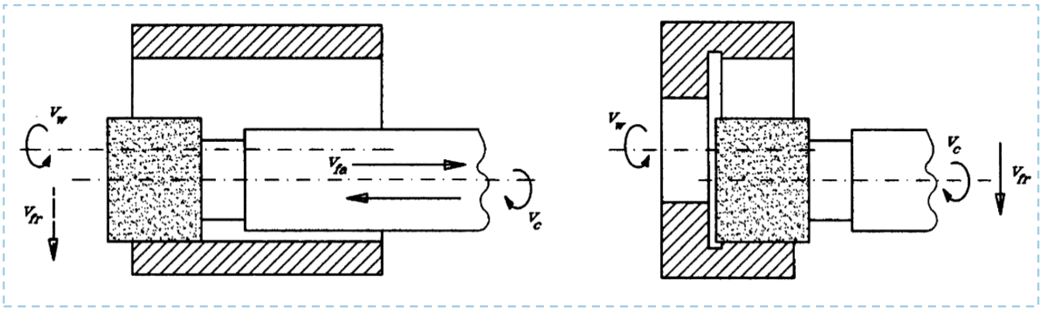
a) with the circumferential surface of the disc b) with the front surface of the disc



a) b)

Figure 5-2. Sheath grinding

a) longitudinal feed sheath grinding; b) groove sheath grinding



a) b)

Figure 5-3. Hole grinding

a) longitudinal feed hole grinding;b) groove hole grinding

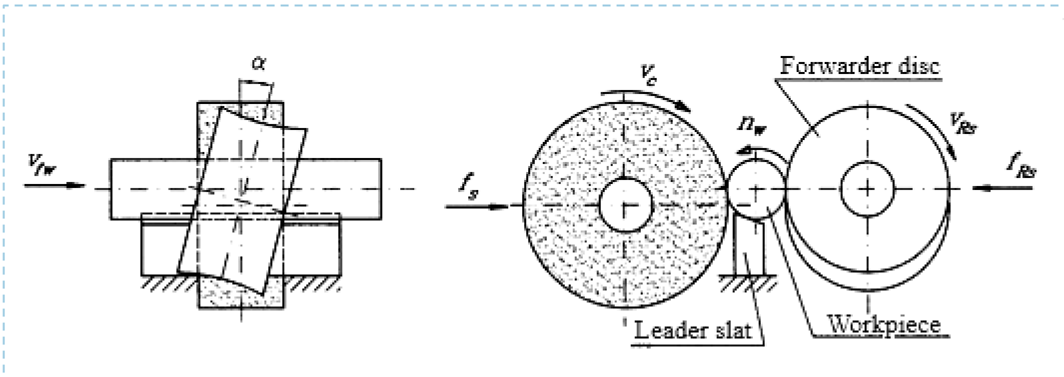


Figure 5-4. Tipless grinding

In the case of ENDLESS FACE GRINDING WITHOUT A TIP, the workpiece is not clamped, but is placed freely on a guide bar between the grinding wheel and the transfer wheel (Figure 5-4). The axis of the transfer disk is not parallel to the axis of the workpiece and thus provides axial movement of the workpiece. This is the so-called permeable process used for long workpieces. In the case of short workpieces, an insertion method is also used here.

Other grinding variants are THREAD GRINDING, which is used to grind threads and screws, **peel grinding**, which is used to grind gears, and PROFILE GRINDING, when the profile of the disc is formed on the machined surface.

6.2 GRINDING TOOLS

The material of the grinding tool consists of the grinding and the binder. The material of the grinding grains must meet several requirements. You must have the ability to cut with sufficient hardness to separate chips from the workpiece material. The grain must also be tough to withstand impact loads during cutting. At the same time, the grain must be so brittle that the increase in cutting force due to the dimming of the grain will cause the grain to split and new cutting edges to form. In addition to the listed mechanical requirements, the grinding grain must also have adequate resistance to thermal and chemical effects.

6.2.1 THE GRINDING PARTICLE IS MATERIAL

Grinding materials can be divided into natural and artificial materials according to their origin. Natural aggregates, with the exception of natural diamonds, are no longer used today. Natural granular materials are quartz, granite, natural corundum and natural diamonds. The artificial (synthetic) granular materials are corundum, silicon carbide, boron nitride, boron carbide and artificial diamond.

CORUNDUM

Corundum contains crystalline alumina (Al_2O_3), small amounts of iron oxide, titanium oxide, silica and various contaminants. The amount of contaminants greatly influences its mechanical properties and can be distinguished according to: (1) normal corundum (brown), (2) semi-noble corundum, and (3) noble corundum (white or pink). The hardness and brittleness of the

particles increase with purity. Its heat resistance is excellent, it is resistant to acids and alkalis, inert to ferrous metals. Due to its low cost, it is the most commonly used granular material. The standard symbol is A and may be preceded by a number indicating the type of corundum (ISO IR 825 - 1966). The following table shows the signs of some common corundum types:

Sign	Corundum type
A	Normal corundum, brown in color
31A	Semi-precious corundum, grayish blue
42A	Precious corundum, white
53A	Precious corundum, reddish brown
57A	Precious corundum, light pink
64A	Mixture of single crystal alumina and precious corundum, pink

SILICON CARBIDE

Silicon carbide (SiC) is harder, sharper and more brittle than corundum. Depending on its

purity, the color can be black or green. Black has a 98% purity mark of C, while green has a 99.5% purity mark of 11C. The hardness of the contaminants is not significantly affected, however, the flexural strength of black silicon carbide is higher than that of green. It withstands temperature changes well, it is resistant to acids and alkalis. Primarily brittle, high-hardness materials such as used for machining cast iron, carbide, glass, ceramics, and aluminum and its alloys.

ARTIFICIAL DIAMOND

Chemical formula: C. Diamond is the hardest natural material. Since the production of artificial diamonds (1955), its use has been expanding. Good thermal conductivity,

electrically insulating. It loses its hardness at 600 to 700 ^{o}C and must be cooled. Cannot be used for ferrous alloys because at 700 ^{o}C it is converted to iron carbide by iron. Used for machining carbides, non-ferrous metals, stone and plastics.

CUBIC BORON NITRIDE (BORAZON, CBN)

Its formula is B_3H, published in 1969. The diamond is close to hardness, has significant heat resistance and chemical resistance. It is excellent for machining hardened steels that are

difficult to machine. Cooling with water should be avoided.

PARTICLE SIZE - GRAIN FINENESS

Particle fineness is a number characteristic of particle size, which is the number of sieve holes per inch that the particle still passed through, but the next finer sieve was already stuck during the separation.

Higher numbers mean finer grains and smaller ones mean coarser grains. In practice, we use discs with the following particle sizes:

8 – 24	coarse-grained
30 – 60	medium grained
70 – 180	fine-grained
220 – 800	very fine-grained

Particle size primarily affects surface roughness.

6.2.2 BINDINGS, BINDING MATERIALS

Their task is to place the grinding grains in a solid frame. The binder and adhesive with a lower melting point than the grinding grains melt during the firing after pressing and bonding bridges are formed between the grains.

INORGANIC BONDS

Ceramic bond (V) consists of kaolin, clay, feldspar, quartz and water glass. Good chemical resistance, heat resistant, brittle, shock sensitive. More than half of the discs used are made with ceramic bonding.

Silicate bonding (S) (water glass) is used for large diameter discs (Ø900 mm). Such discs are not fired but dried (550 - 570 ^{o}K).

A metallic bond (M) is formed by sintering bronze, steel or carbide powder. They provide the highest bonding security. It is mainly used in diamond and boron nitride tools. For high

abrasion resistance, we use discs made of metallic bonding for difficult-to-machine materials (eg carbide). They are prone to clogging and should not be used without refrigeration.

ORGANIC BONDS

The resin-like bonds (B) can be based on natural (sellack) or synthetic resin. Synthetic resins are increasingly displacing natural resins. Polyester and phenolic resins are mostly used. Discs made with such a binder are very tough and have high strength, making them excellent for rough work and cutting grinding. The cutting speed can reach 120 m/s.

The rubber bond (R) is made of natural or synthetic rubber. They are more flexible than resin bonding, but have lower heat resistance. It is used for narrow discs, e.g. for thread grinding or grinding narrow grooves.

The binder ensures the HARDNESS OF THE DISC, by which is meant the property of the binder by which it tends to retain the grain in the disc against forces. It has nothing to do with grain hardness and should not be confused with it. The hardness of the bond is of great importance for the grinding process. This is because the harder the disc, the more the grain remains in the binder even in the case of significant wear. In the case of a soft disk, even after relatively little wear, the grain will come out of the bond and a new, sharp grain may surface. As a general rule, we use hard discs for soft materials and soft discs for hard materials and demanding grinding. The hardness of the disc is symbolically denoted by the letters ABC: A, B, C,… Z means increasing hardness.

The structure of a ceramic-bonded disk is formed by the pores, binder, and pores. Their volume ratio also significantly affects the characteristics of the grinding wheel and according to this the wheel can be sparse or compact (Figure 5-5). The hardness of the disc depends on the ratio of the amount of binder and the thickness of the "bonding bridges".

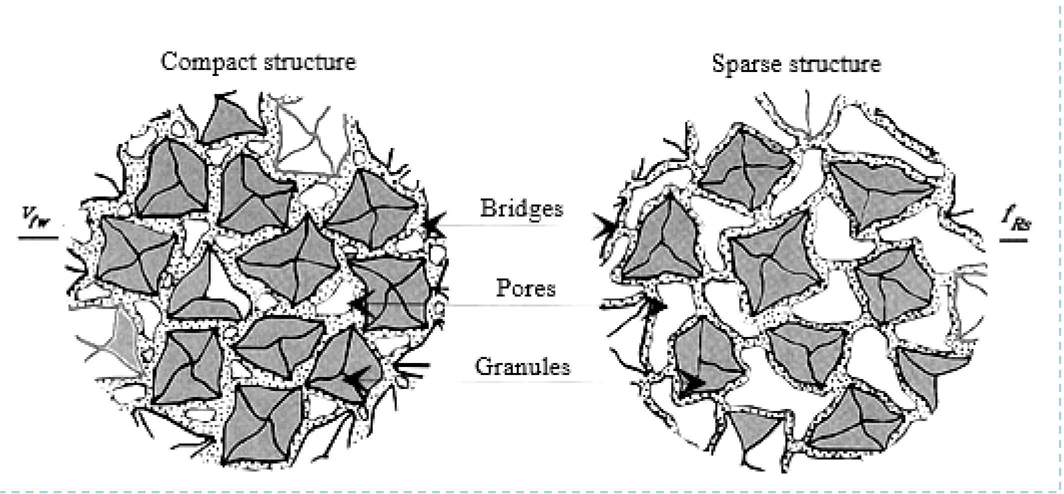


Figure 5-5. Structure of disks (Winterthur)

Increasing the binder volume to the detriment of the pore volume increases the hardness and vice versa. Increasing the volume of binder and pores against the particle volume increases porosity. The characteristic of the structure is the number of structures, which expresses the distance between the particles and the size of the pores:

1, 2	very solid
3,4	solid

5, 6, 7, 8	medium
9, 10 11	sparse
12, 13, 14	very sparse structure.

6.2.3 THE SHAPE AND THE FIXATION OF GRINDING WHEELS

The most commonly used grinding wheel types are shown in Figure 5-6. Larger diameter discs are usually provided with a hole, while smaller ones have a handle design.

The typical dimensions of the discs are required by international standards.

The handles can be mounted in a chuck and the holes in a high-precision grinding spindle

(Figure 5-7). Large diameter discs (D> 175 mm) must be equilibrated before capture. The torque can be transmitted by friction on the front surface of the disc. Breakage of brittle tool material can be prevented with a paper or felt pad.

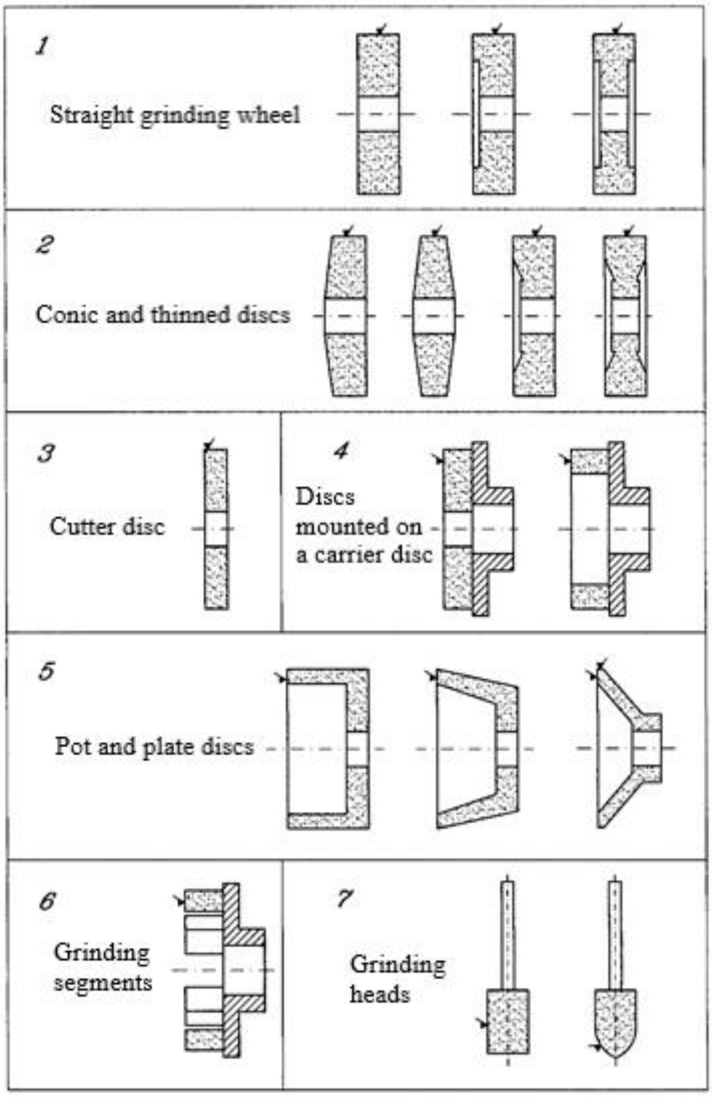


Figure 5-6. Shapes of grinding wheels

There is an extremely high risk of accidents when grinding, so the grinding wheel mounted on the machine must always be fitted with a protective steel cover with only the opening required for work. Parts that may bounce off the high-speed disk pose a great danger to the environment. If the disc breaks, flying parts can cause a fatal accident if a proper protective cover does not prevent this.

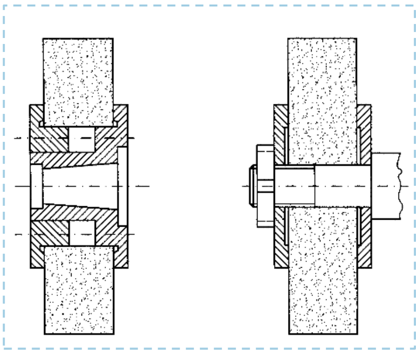


Figure 5-7. Grasping the disks

6.2.4 THE QUALITY AND MARKING OF GRINDING WHEELS

The marking system for grinding wheels is specified in the international standard ISO 525. The mark on a disc contains the shape and dimensions of the disc, the characteristics of the grinding material and the maximum permissible circumferential speed of the disc. The marking order of the characteristics of the abrasive material is illustrated by an example in Figure 5-8.

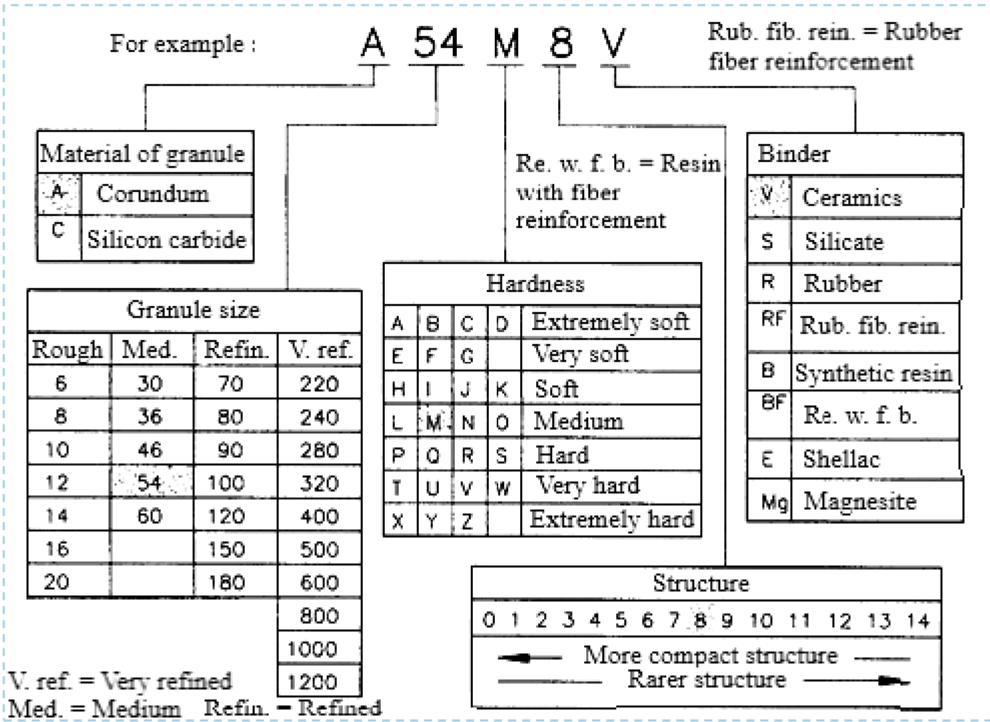


Figure 5-8. Designation of corundum and silicon carbide abrasives

The selection of the grinding material must take into account the material of the workpiece, its condition and hardness, and the grinding variant. This is a very careful task, so it is a good idea to consult the disc manufacturer when planning new work.

6.3 CHIP SEPARATION CIRCUMSTANCES DURING GRINDING

The process of chip removal is somewhat similar to face milling, but abrasive machining differs from chip removal with a regular edge in several respects. These differences are mainly due to irregular edge geometry and can be summarized as follows:

- the individual grains are irregular in shape and randomly arranged along the circumference of the disk
- the radial position of the particles also varies
- the general face angle on the grains is strongly negative (-60º)
- the cutting speed is very high, a typical value is 25 m/s (1500 m/min)
- the chip cross section is very small

The thickness of the shavings is so small that, compared to this, the elastic deformations cannot be neglected. The chip removal stages are illustrated in Figure 5-9. The grains in the relatively elastic binder initially slip on contact with the surface of the workpiece and cause pure elastic deformation (1). The grain penetrating deeper and deeper into the piece creates a plastic

deformation (2). Upon further penetration of the grain, the layer is sheared simultaneously with the creasing on the sides of the grain and the chips are formed (3). The pre-exit section of the grain is now characterized only by elastic deformation and shear (4), there is no plastically deformed layer. Although there are similarities with cutting edge cutting, the differences are also

fundamental. In grinding, the lateral flow (creasing) of the material is also present in front of the edge (grain), so here the deformation is spatial (three-axis), as opposed to chip separation with a definite edge where, somewhat simplified, the material flow is planar (biaxial) [9], [23].

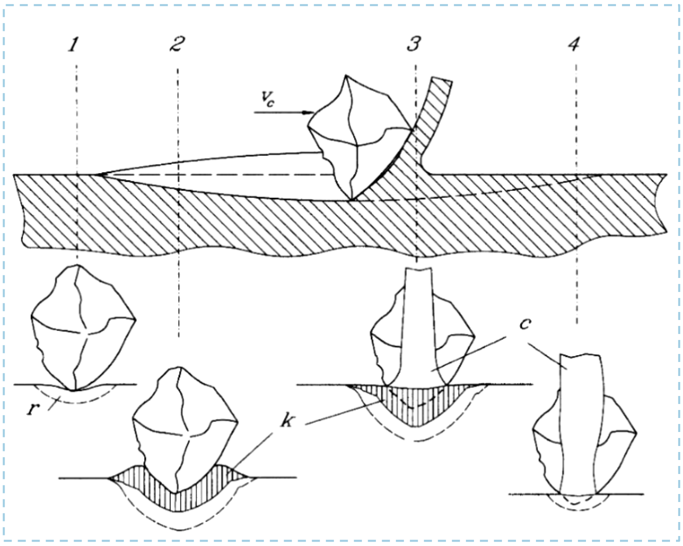


Figure 5-9. Chip removal stages during grinding

r – elastic deformation zone; k – zone of plastic deformation; c - chip

The cutting speed is equal to the circumferential speed of the disc:

$$v_c = v_s = d_s \cdot \pi \cdot n_s \text{ (m/s)}$$

Where: d_S disk diameter

n_S disk speed

6.4 DEFINITION OF THE CUTTING FORCE DURING GRINDING

The resulting cutting force is also broken down into force components acting in characteristic directions during grinding (Figure 5-10). These are the main cutting force (F_c) in the direction tangential to the disc, the passive force perpendicular to the tangential contact (F_p) and the feed force in the direction of the longitudinal feed (F_f).

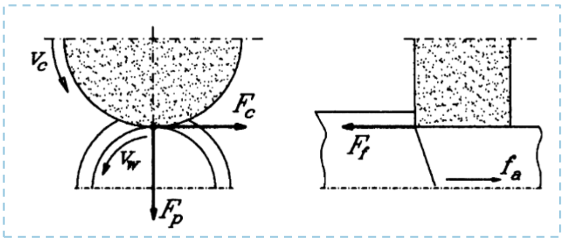


Figure 7-10. Cutting forces for longitudinal mantle grinding

The peculiarity of grinding is that the value of the passive force perpendicular to the disc is greater than the main cutting force.

$$F_p = (1{,}5 \ldots 3) \cdot F_c$$

This affects the accuracy of the machining, so it must be taken into account.

A problem in determining the cutting force is that due to the irregular shape of the edges, the cross section of the chip can only be determined approximately, and the number of edges (grains) in the grip cannot be determined accurately. In addition, the chip thickness is very small (h << 0.05 mm) and this already requires a modification of Kinzle's specific cutting force.

The shape of the chip cross section separated by a grain can be replaced by a triangle or a rectangle (Figure 5-11). The number of grains to be cut simultaneously depends on the connection length l_k, the grain density and the value of the axial depth of cut a_p (Figure 5-12).

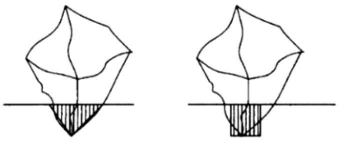


Figure 5-11. Approximate shape of the chip cross section

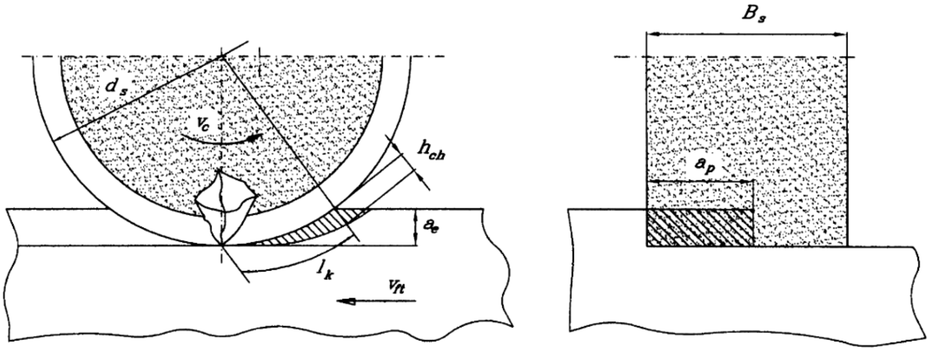


Figure 5-12. Characteristic parameters of chip removal

The grain numbers can be determined by statistical methods [12] and thus, in theory, the cutting force can also be determined.

In practice, however, the equivalent value of the sum of the chip cross-sections separated by the particles can be determined more easily based on the material removal rate.

The material removal rate can be determined on the one hand as the product of the main feed rate and the cross section of the material layer perpendicular to it:

$$Q_w = A_w \cdot v_f \quad \text{[mm}^3\text{/min]},$$

Where: A_w the cross-section of the deposited material layer in a plane perpendicular to the feed direction.

v_f the feed rate

For surface grinding: $A_w = a_e \cdot a_p$; $v_f = v_{ft}$

The material removal rate determined in this way is a function of the setting parameters (a_e, a_p, v_{ft}), so it can also be called the setting material removal rate (Q_{wp}). On the other hand, the material removal rate can also be defined as the product of the cutting speed (v_c) and the chip cross-section separated by the grains (A_{ef}). If, according to Kurrein's proposal, the chip cross-section is assumed to be a narrow strip with a thickness of h_{eq} along the width of the disc, the so-called internal material removal rate can be written by the following formula:

$$Q_{wi} = A_{ef} \cdot v_c = h_{eq} \cdot a_p \cdot v_c$$

The amount of material determined in the two ways is the same: Q_w=Q_{wp}=Q_{wi}, or

$$Q_w = a_e \cdot a_p \cdot v_{ft} = h_{eq} \cdot a_p \cdot v_c$$

From the above context, the equivalent chip thickness can be determined:

$$h_{eq} = \frac{Q_w}{a_p \cdot v_c} = \frac{v_{ft}}{v_c} \cdot a_e$$

The equivalent chip thickness is an imaginary quantity, in reality the chip thickness is greater than this, no continuous chips are formed along the width of the disc, but only the particles separate it.

The cutting force can be approximated by the following relation:

$$F_c = h_{eq} \cdot a_p \cdot k_{cgrind} = \frac{v_{ft}}{v_c} \cdot a_e \cdot a_p \cdot k_{cgrind}$$

Where: k_{cgrind} the mean value of the specific cutting force (Table 7-1.).

The power required for cutting is proportional to the cutting force: $P_c = F_c \cdot v_c$.

The required engine power can be calculated taking into account the efficiency of the implement: $P_m = \dfrac{P_c}{\eta}$.

Tabel 5-1. Mean of the specific cutting force for grinding different materials (disc grain size: 60…120; v_c=30…45 m/sec) (Following A. H. Fritz and G. Schulze)

The material of the workpiece	k_{cgrind} [kN/mm^2]		The material of the workpiece	k_{cgrind} [kN/mm^2]
EN			EN	
S235	30,241		15CrMo5	39,16

E295, C35E	34,03		55NiCrMoV6 (soft)	29,75
E360	38,05		55NiCrMoV6 (refined)	32,83
C45E, C45	37,96		GG25	18,98
C60	36,42		GS-45	26,33
16MnCr5	35,91		GS-52	29,24
19CrNi8	38,65		Bronze casting	30,44
42CrMo4	42,75		Brass	13,33
34CrMo4	38,30		Aluminum casting	10,94
50CrMo4	37,96		Magnesium casting	4,79

6.5 WEARING AND REGULATION OF THE GRINDING TOOLS

As a result of the interaction with the workpiece on the work surface of the grinding wheels, wear of the particles and the binder occurs. The following types of wear can be distinguished in the wear of particulate matter:

Wear due to reduced compressive strength. Corundum grains occur because they have a melting point of 2050 ºC, but their compressive strength drops to one-sixth of that measured at room temperature at 1200 ºC. As a result, the edges of the grains will be rounded, the frictional force will increase significantly and the heating in the cutting zone will be even stronger. The process may become unstable.

Abrasive wear. Due to the friction between the grain and the workpiece, a certain amount of material is mechanically abraded from the grain, resulting in rounding of the edges of the grain.

Particle cracking and fragmentation. Due to the heat demand and then the rapid cooling of the particles, as well as the mechanical forces, the particles crack, small parts break out and thus new, sharp edges are formed. This process is favorable because the discarded disc volume is not significant and new edges are formed. This is called

self-sharpening.

Eruption of whole grains. In this type of wear, whole grains break out of the binder. The holding force of the binder is insufficient, the disc is too soft. The wear volume of the disc is significant. Nevertheless, the blade retains its cutting ability.

The *numerical value of the wear* of the disc can be expressed by the wear volume (V_s) or by the change in the radius of the disc (Δr_s). The wear volume is directly related to the grinding ratio G.

Eruption of whole grains. In this type of wear, whole grains break out of the binder. The holding force of the binder is insufficient, the disc is too soft. The wear volume of the disc is significant. Nevertheless, the blade retains its cutting ability.

The numerical value of disc wear can be expressed as the wear volume (V_s) or the change in disc radius (Δr_s). The wear volume is directly related to the grinding ratio *G*: $G = \frac{V_w}{V_s}$

Where: V_w the volume of material ground from the workpiece

V_s the volume worn off the disk

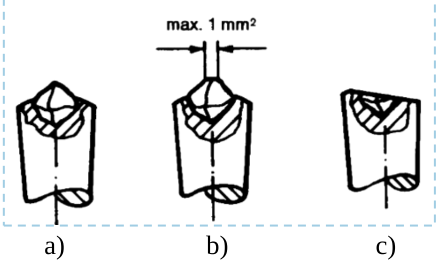
After a certain period of time (tool life), the blade loses its cutting ability or does not meet other criteria (surface quality, excessive vibration, shape defect, etc.). The essence of the control is that the worn layer is removed from the surface of the disc and thus new, intact particles are brought to the surface.

The control tools used on the grinder can be divided into stationary and moving control tools in terms of movement.

Stationary control tools can be: (1) single-grained diamonds; (2); multi-grain diamond regulator, (3) control rod.

The moving control tools can be: (1) diamond profile roller, (2) diamond forming roller (disc); (3) profile friction rollers.

We only deal with the single-grain diamond regulator here. It is a diamond grain, usually in an octahedron, enclosed in a steel holder. Several peaks of this can be exploited if, after the wear of the peak in use, the particle is re-enclosed in a new holder from which an intact peak protrudes. (Figure 5-13).



a) b) c)

Figure 5-13. Single-grain diamond regulator (Riegger)

a) new condition, b) worn grain that is still suitable for retreading c) overgrown grain,

The control process of the disc corresponds to turning (Figure 5-14). The roughness of the controlled surface of the disc and the success of the control are influenced by the depth of cut (a_{ed}), the working width of the tool (a_{pd}) and the amount of axial feed (f_{ad}). An important feature of the control process is the overlap number as a function of the operating tool width and feedrate: $U_d = \frac{a_{pd}}{f_{ad}}$

Where: a_{pd} [mm] the working width of the tool
f_{ad} [mm/ford] the feed per revolution of the disc

Since the feed per revolution of the disc is not a separate parameter on grinding machines, in practice it is better to calculate the overlap with the following formula:

$$U_d = \frac{a_{pd} \cdot n_s}{v_{fad}} \quad \text{,or from here} \quad v_{fad} = \frac{a_{pd} \cdot n_s}{U_d}$$

Where: v_{fad} [mm/min] the axial feed rate
n_s [ford/min] the speed of the disc

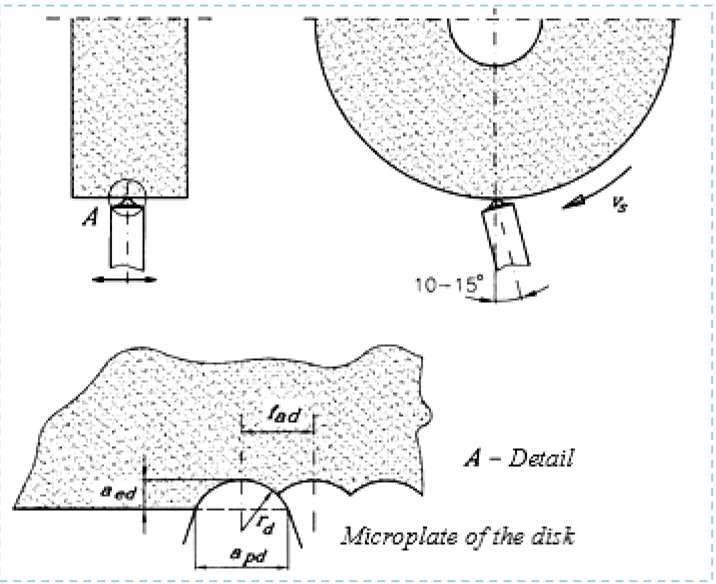


Figure 5-14. Control with single-grained diamond

As the number of overlaps increases, the roughness of the disc and the machined surface also decreases. According to Messer, the best value for the overlap number can be obtained using the following empirical formula: $U_d = \frac{grain\ size}{15}$

Other authors make the overlap number dependent on the roughness of the surface to be machined.

Single-grain diamond control is commonly used for plain discs in single and small series production. Due to its relatively small contact surface, it is advantageous for hole grinding on small and vibration-prone discs. The adjustment can be done at normal or reduced speed of the disc. The depth of cut is normal in the range 0.01… 0.03 mm and at least 3-4 plots must be taken. Extensive cooling should be used in all cases during the regulatory process.

6.6 DEFINITION OF THE TECHNOLOGICAL DATAS

As with other machining modes, the process data are the parameters that determine the value of the main and secondary movements of the process. In order to determine these, it is advisable to take into account the guideline values that have been proven in practice.

6.6.1 DEFINITION OF THE DEPTH OF CUT

The recommended values for the depth of cut for different grinding variants are given in Table 6-5. When grinding, the value of the depth of cut is extremely small, so the machining must always be carried out in several strokes (strokes).

In order to increase the economics of the machining process, the grinding process is divided into roughing, smoothing and spinning stages (Figure 5-15). Most of the allowance (approx. 70… 80%) is removed by rough grinding, i.e. with a greater depth of cut, in which case the aim is to achieve a relatively high material removal rate (Q_w). This is followed by a smoothing grind with a smaller depth of cut, which is intended to achieve the required dimensional and shape accuracy as well as surface roughness.

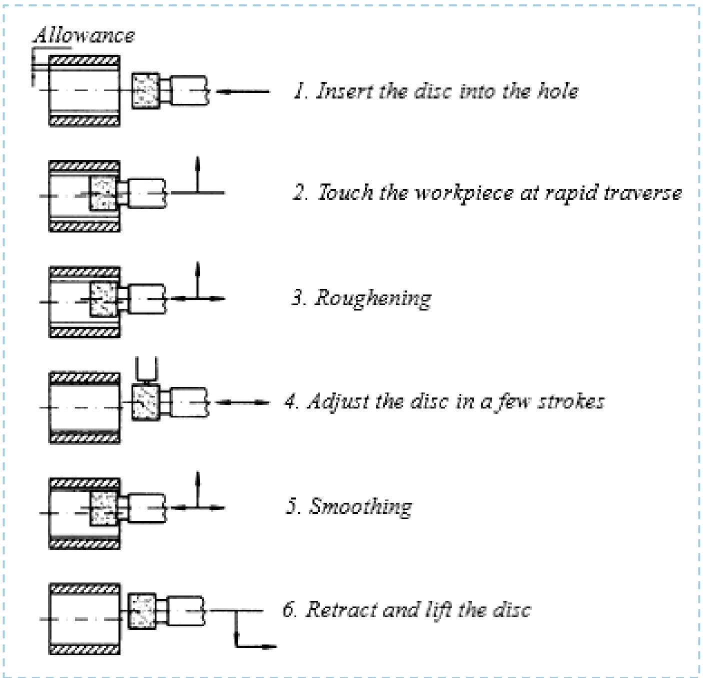


Figure 5-15. Stages of the hole grinding process

If grinding takes place in the roughing and finishing stages, the number of passes is the sum of the number of roughing and finishing passes

$$i = i' + i'' = \frac{0{,}8R_3}{a'} + \frac{0{,}2R_3}{a''}$$

Where: *i'* number of cuts when roughing (rounded to the nearest whole number)

i'' number of cuts when smoothing (rounded to the nearest whole number)

a' depth of cut when roughing

a" depth of cut when smoothing

R_3 grinding allowance

6.6.2 DEFINITION OF THE FEED

Depending on the grinding version, the feed direction and nature can be different. Regardless of whether the movement is performed by the disc or the workpiece, in order to form a uniform approach, the directions of the secondary movement relative to the disc are defined and according to this the feedrate can be:

- tangential (f_t)
- radial (f_r)
- axial (f_a)

The feed rate is related to one revolution or stroke of the workpiece and multiplied by the number of revolutions or strokes per minute gives the value of the feed rate in the characteristic directions (v_{ft}, v_{fr}, v_{fa}).

The tangential feed rate for face grinding and planar grinding is the same as the target speed: $v_{ft} = v_w$

The guide values for the target speed for different grinding variants and workpiece materials are given in Table 5-3.

Various empirical formulas can also be found in the literature to determine the velocity of objects, but their scope is not possible to discuss.

The value of the radial feed for grooving is equal to the depth of cut (Table 5-2):

$f_r = a_e$ (mm/rev).

The value of the axial feed for longitudinal feed grinding and planar grinding with the

circumferential surface of the disc as a function of the disc width is common (Table 5-2):

$f_a = \left(\frac{2}{3} \cdots \frac{3}{4}\right) \cdot b_s$ for roughing, and $f_a = \left(\frac{1}{4} \cdots \frac{1}{2}\right) \cdot b_s$ for smoothing

Table 5-2. Guideline values for depth of cut

Grinding version		Feed f_a (mm/ford), (mm/stroke)	Depth of cut a_e (mm)
Longitudinal feed sheath grinding	roughing	f_a =(0,6…0,75) b_s	0,008-0,04
	smoothin g	f_a =(0,25…0,5) b_s	0,001-0,015
Groove sheath grinding	roughing		0,002-0,03
	smoothin g		0,001-0,005

Hole grinding	roughing	f_a =(0,5…0,75) b_s	0,008-0,03
	smoothing	f_a =(0,25…0,5) b_s	0,001-0,015
Surface grinding with the circumferential surface of the disc	roughing	f_a =(0,6…0,75) b_s	0,01-0,05
	smoothing	f_a =(0,25…0,5) b_s	0,003-0,015
Surface grinding with the front surface of the disc	roughing		a_p= 0,01-0,03
	smoothing		a_p= 0,003-0,015
Comment: 1. Within these ranges, lower values should be used for smaller crown gauges and higher object velocities. 2. The sparking takes place in a few strokes without catching			

Table 5-3. Guideline values for object speed

Grinding version		Object speed v_w (m/min)				
		Unhardened steel	Hardened steel	Cast iron	Brass	Aluminum
Outer sheath grinding	roughing	12…15	14…18	12…15	18…21	40…60
	smoothing	8…12	8…12	9…12	15…18	20…40
Hole grinding		18…21	21…24	21…24	21…27	30…40
Surface grinding	circumferential	10…18	8…12	10…18	15…25	15…25
	front	6…25	6…25	6…30	20…45	20…45

6.6.3 DEFINITION OF THE CUTTING SPEED AND THE GRINDING WHEEL REVOLUTION

By cutting speed is meant the circumferential speed of the disc. It can be stated that the value of the cutting speed in grinding is incomparably higher than in the machining methods

discussed so far (turning, drilling, milling). The special feature of grinding is that the cutting speed is not limited by the life of the tool, but by the strength of the binder and the resistance of the disc to tearing. The risk of flight is mainly caused by the significant centrifugal forces and the heat load of the disk. There would be a very high risk of the disc flying out, so there is a maximum permissible circumferential speed for each disc type and binder that should not be exceeded (Table 5-4). Recommended values for cutting speed can be found in the literature.

Table 5-4. Maximum permissible speed of the grinding wheels vs (m/s)

Binder	The method of feeding	Disc diameter and shape			
		$d_s \leq 150$ mm		$d_s > 150$ mm	
		disc-like	other	disc-like	other
Magnesite	manual	15	15	15	12
	machine	25	25	20[1)]	15
Ceramic, silicate, organic	manual	30	25	25	20
	machine	35	30	30	25

[1)] For disc diameters larger than 1000 mm, the maximum speed is 15 m/s even at machine feed

6.7. DEFINITION OF THE MACHINE TIME

Knowing the cutting parameters, the dimensions of the tool and the surface to be machined, the main machine time required for cutting can be determined.

1. Machine main time for longitudinal feed sheath grinding and hole grinding

a. Grinding with stroke per cut

$$t_g = i \cdot \frac{L \cdot k}{f_a \cdot n_w}$$

b. Grinding with double stroke cut

$$t_g = i \cdot \frac{2 \cdot L \cdot k}{f_a \cdot n_w}$$

Where:

- *i* number of cuts
- *L* stroke length, calculated as shown in Figure 6-16 (mm)
- f_a axial feed (mm/ford)
- n_w object speed (min^{-1})
- *k* a modifying factor to take into account the time required for sparking and disc control. Its value is *k* = 1,2 for hard discs and roughing and *k* = 1,6 for soft discs and smoothing.

Groove sheath grinding Hole grinding

2. Machine main time for groove sheath grinding and hole grinding

$$t_g = \frac{R \cdot k}{2 \cdot f_r \cdot n_w}$$

Where:

R grinding allowance measured in diameter (mm)

f_r radial feed (mm/ford)

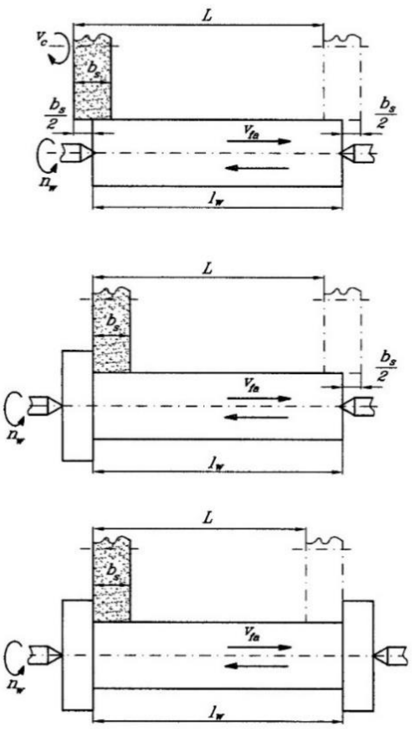


Figure 5-16. Determination of stroke length for sheath grinding

3. Machine main time for surface grinding with the circumferential surface of the disc

$$t_g = i \cdot \frac{B \cdot k}{f_a \cdot n_L}$$

Where:

$B = b_w + b_1 + b_2$ the grinding width (mm), (Figure 5-19)

f_a axial feed (mm/stroke)

$n_L = \frac{v_w}{L}$ number of strokes per minute (stroke/min)

$L = l_w + l_1 + l_2$ stroke length (Figure 5-17)

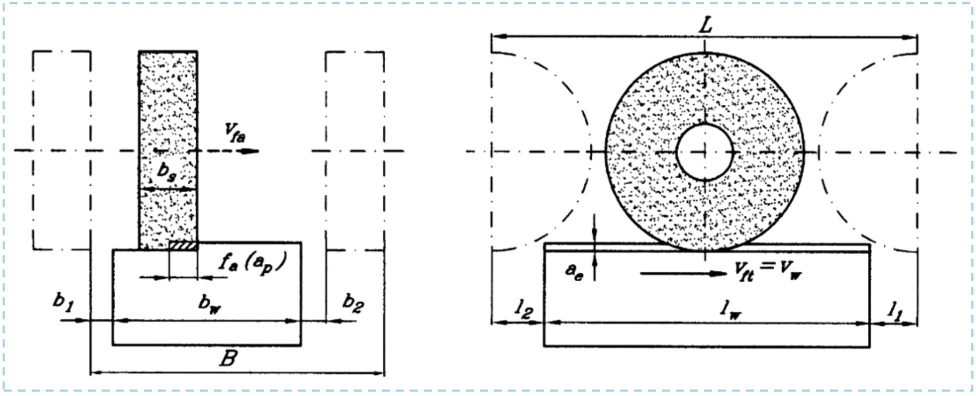


Figure 5-17. Typical dimensions for surface grinding

7. CUTTING MACHINES

For the mechanical engineering industry, machine tools are the most important means of production. What they have in common is that they shape the workpiece during relative movement of the tool and the workpiece and with a certain amount of effort. Depending on their structure and automation, we can come across many different solutions. The construction of machine tools has undergone significant changes in the last 50 years. Development is still ongoing and, on the one hand, new, state-of-the-art solutions for certain machine components (eg the use of motorized spindles or linear motors to move sleds, etc.) are being built. turning centers where even gear milling is possible). The emergence of NC and then CNC machines should be considered a milestone in the automation of machine tools. A detailed description of the field would require several volumes, in this chapter we will confine ourselves to a brief overview of the most well-known cutting machine tools.

Machine tools can be divided into several aspects. Depending on the machining (cutting) mode, they can be:

- lathes,
- planing machines,
- drilling machines,
- milling machines,
- shape drawing machines,
- sawing machines,
- grinding machines,
- gear machines.

In addition to the division of machine tools according to the machining mode, they can also be divided according to the level of automation used. Depending on the level of automation, production systems can be:

- Manually operated universal machines
- Program-controlled machines
  - Mechanically controlled machines (curved track, cam, bumper)
  - Numerical control machines (NC, CNC systems)
- Machining centers
- Flexible production cells
- Flexible production systems

SOURCES

[1] Ambrusné Alady M., Galla Jánosné, Sipos S.: A gyártástechnológia alapjai I., Bánki Donát Műszaki Főiskola, Budapest, 1995.
[2] Angyal B., Dobor Lászlóné, Palásti Kovács B., Sípos S.: A forgácsolás és szerszámai, Műszaki Könyvkiadó, Budapest, 1988.
[3] Bali J.: Forgácsolás, Tankönyvkiadó, Budapest, 1985.
[4] G. Bootroyd, W. A. Knight: Fundamentals of Machining and Machine Tools, Taylor & Francis Group, Broken, NW, 2006.
[5] W. Degner, H. Lutze, E. Smejkal: Spanende Formung, Carl Hanser Verlag, München Wien, 2000.
[6] Dudás I.: Gépgyártástechnológia II, Miskolci Egyetem, 2001.
[7] A. H. Fritz, G. Schulze: Fertigungstechnik, Springer-Verlag, Berlin Heidelberg, 2006.
[8] K. H. Grote, J. Feldhusen: Dubbel, Taschenbuch für den Maschinenbau, Springer-Verlag, Berlin Heidelberg, 2005.
[9] R. S. Hahn: On the nature of the grinding process. Proceeding, third Int. MTDR Conference, September, 1962, Oxford, pp. 129-154.
[10] Horváth M., Markos S.: Gépgyártástechnológia, Műegyetemi Kiadó, Budapest, 2006.
[11] B. Ivković: Obrada metala rezanjem, Građevinska knjiga, Beograd, 1979.
[12] W. König: Köszörülés, dörzsköszörülés, tükrösítés, Műszaki Könyvkiadó, Budapest, 1983.
[13] Köves E., Moser M., Almásy P.: Köszörülés és finommegmunkálás, Műszaki Könyvkiadó, Budapest, 1973.
[14] D. Milikić: Tehnologija obrade rezanjem, Univerzitet u Novom Sadu, Novi Sad, 1999.
[15] A. Richter, K. H. Tempelhof: Gesetzmäßigkeiten der Spanbildung und der Kräfte beim Walzfräsen im Gegenlauf und Gleichlauf –Teil 1, 2, 3, Fertigungstechnik und Betrieb, 19. Jahrgang, Heft 3, pp. 283-288; Heft 8, pp. 473-480, 1969; 20. Jahrgang Heft 1, pp. 54-58, 1970.
[16] Sandvik Coromant, Technisches Handbuch der Metallzerspannung, AB Sandvik Coromant, 2005.
[17] E. Smejkal: Betrachtungen zur Kraftberechnung beim Stirnfräsen nach Weilenmann, Fertigungstechnik und Betrieb, 16. Jahrgang, Heft 4, pp. 225-227, 1966.
[18] Stampfer M. : Forgácsolás, Gépipari technológiák I, Pollack Kiadó, Pécs, 2012.
[19] Stampfer M. : Gépipari technológiák II, Pollack Kiadó, Pécs, 2008
[20] P. Stanković: Mašinska obrada, Obrada metala rezanjem, Građevinska knjiga, Beograd, 1979.
[21] J. Stanić, D. Nikolić, T. Jovanović, V. Gajović: Mašinska obrada – priručnik za proračun merodavnih režima mašinske obrade rezanjem, Privredni pregled, Beograd, 1972.
[22] V. V. Svarc: Képes Műszaki Kisszótár, Műszaki Könyvkiadó, Budapest, 1989.
[23] H. K. Tönshof, B. Denkena: Spanen, Springer-Verlag, Berlin Heidelberg, 2006.
[24] S. Urošević: Proizvodno mašinstvo, I deo, Naučna Knjiga, Beograd, 1984.
[25] Winterthur Schleiftechnik, Technical Grinding Information, E93/6

Module_4

Machine Industrial Technology

Mihály STAMPFER // Ákos CVENTIS

Machining technologies

Financial support was provided by the DRIVEN project (Grant agreement No. 2020-1-SK01-KA203-078349) under Erasmus+ Call 2020 Round 1 KA2 - Cooperation for innovation and the exchange of good practices.