

PART I

THEORY OF METAL CUTTING

I.1. Introduction

The selection of a cutting material and grade is an important factor in the success of a machining operation. To make a good choice for a given application, it is necessary to have a basic knowledge of the properties of cutting materials and their performance.

I. 2. Definition

Cutting materials are components manufactured and used for cutting tools intended for mechanical machining.

I. 3. Different cutting materials

I. 3. 1. Tungsten steel (high-speed steel)

The invention of high-speed steels was made by the Americans Taylor and Whites in 1903. The basis of high-speed steel (HSS) tools is a low-alloy steel that has undergone heat treatment. Several HSS tools are available and are distinguished by their metallurgical composition. High-speed steels are made from an iron-carbon alloy (at least 0.7%). Tungsten is the first main addition element used (up to 20%) and a few addition elements (cobalt, manganese, chromium, vanadium, molybdenum). The hardness of these steels after quenching varies between (60 HRC to 70HRC) and will be maintained up to 550 °C. The cutting speeds for this type vary between 25 and 35 m/min for steels. They do not allow a high cutting speed because excessive heating eliminates the quenching of the tool, and therefore creates a rapid collapse of the cutting edge. They are mainly used for the manufacture of single-piece tools (turning tools, drills, milling cutters and reamers). The designations of Ordinary Speed Steel

- High Speed Steel: A.R.S (or HSS in English: High Speed Steel);
- Extra High Speed Steel: A.R.E.S.

The chemical composition of high speed steels is illustrated in Table I.1:

Steel	% C	% W	% C r	% V	% Mo	% Co
OSS	0,7	1,5	4	1,5	0,75	/

HSS	0,8	6,5	4	2	6,5	/
EHSS	0,8	20	5	2	2	10

Table I. 1: Chemical composition of high-speed steels.

I. 3. 2. Metallic carbides

A carbide is a chemical compound of carbon with a second chemical element such as tungsten carbide and titanium carbide, tantalum or niobium.

The carbide tool is the most used currently. It exists in all shapes for each type of material and for each type of machining. It is presented as a plate (figure I.1) that is quickly fixed on a tool holder.



Figure I.1: Carbide inserts intended for turning operations.

The cobalt or nickel used as a binder in the composition of the carbide is produced in the form of a plate using powder metallurgy through a sintering process (a process for manufacturing parts that involves heating a powder without melting it). Under the effect of heat, the grains weld together, which forms the cohesion of the part.

The addition of titanium or niobium carbide increases the high temperature properties (up to 1000°C) by reducing friction, therefore a choice of very high cutting speeds of up to 100 m/min, and this element also gives the tool a longer life than SS or HSS tools. The hardness of carbide tools is around 80 to 90 HRC, which allows very high cutting speeds: 4 times those of high-speed steels. Carbide-based tools can be classified into three categories:

- Tungsten carbide (monocarbide); composed mainly of tungsten carbide grains bonded with cobalt: 8% Co and 92% WC.
- Titanium-tungsten carbide (double carbide): 30% TiC; 4% Co; 66% WC
- Titanium-tantalum-tungsten carbide (triple carbide): 30% TiC and TaC; 12% Co; remainder 58% WC. Monocarbides are effective for machining cast iron and nonferrous alloys compared to double and triple carbides which are effective mainly for machining steels.

I. 3. 3. Ceramics

These are very hard aluminum oxide compounds, silicon nitride, agglomerated in chromium oxides Cr_2O_3 as a binder, sintered at 1800 °C. Ceramics have sufficient compressive strength (up to 5000 N/mm²), high hardness (89 HRC to 95 HRC), with high thermal stability (1200 °C) and remarkable wear resistance. Ceramic tools are highly refractory (melting point above 1500 °C). They are well suited for high-speed machining and machining of very hard ferrous alloys up to 79 HRC. Ceramics allow a large flow rate of material to be removed but require high machine stability and strict compliance with cutting conditions. Ceramic inserts have several cutting edges for successive use.

Figure I.2 shows two ceramic plates.

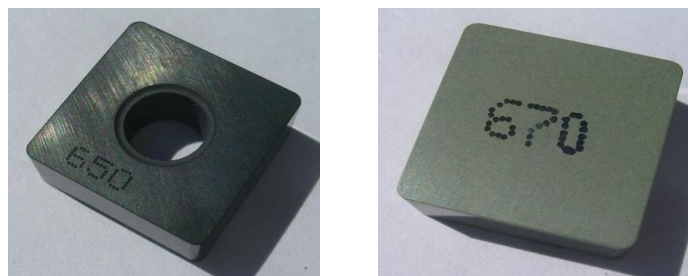


Figure I.2: Ceramic plates.

I. 3. 4. Cermets

Cermet is a term formed from: Cer comes from ceramic and met from metal. Cermets are materials produced by powder metallurgy, consisting of particles of hard metal compounds

(carbides, nitrides, carbonitrides) bound by a metal (usually nickel). Their hardness is in the order of 1500 to 2000 HV. They do not lose their capacity when hot in high-speed machining or at extreme temperatures up to 1000°C. Cermets do not necessarily require lubrication, they are intended for finishing operations and precision machining that require work at high cutting speeds and low feeds. Their good wear resistance and high toughness allow working with lower cutting forces and give good surface finishes and high dimensional precision of the machined parts. Figure I.3 shows two cermet inserts.

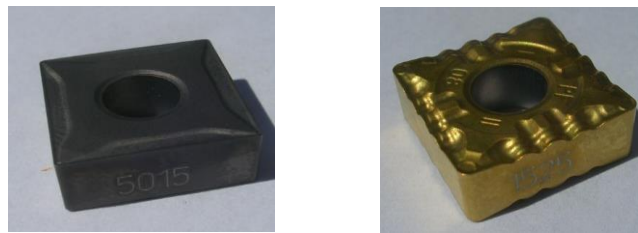


Figure I.3: Cermet plaques.

I. 3. 5. Diamonds

Diamond is a rare mineral, of natural origin, composed of carbon. It is the hardest natural substance known (7000 HV).

It has a low coefficient of friction and a low aptitude for adhesion (bonding, welding to metals). A high resistance to heat, a high resistance to wear.

The use of diamond is widespread as a constituent of grinding wheels, or grains for regrinding grinding wheels and for finishing work. It is very fragile. Its cost is high. It is heat resistant for a temperature of 1500 °C. It requires fast machine tools. The main mechanical properties toughness (toughness is the ability of a material to resist the propagation of a crack) and hardness for cutting tools are shown in Figure I. 4.

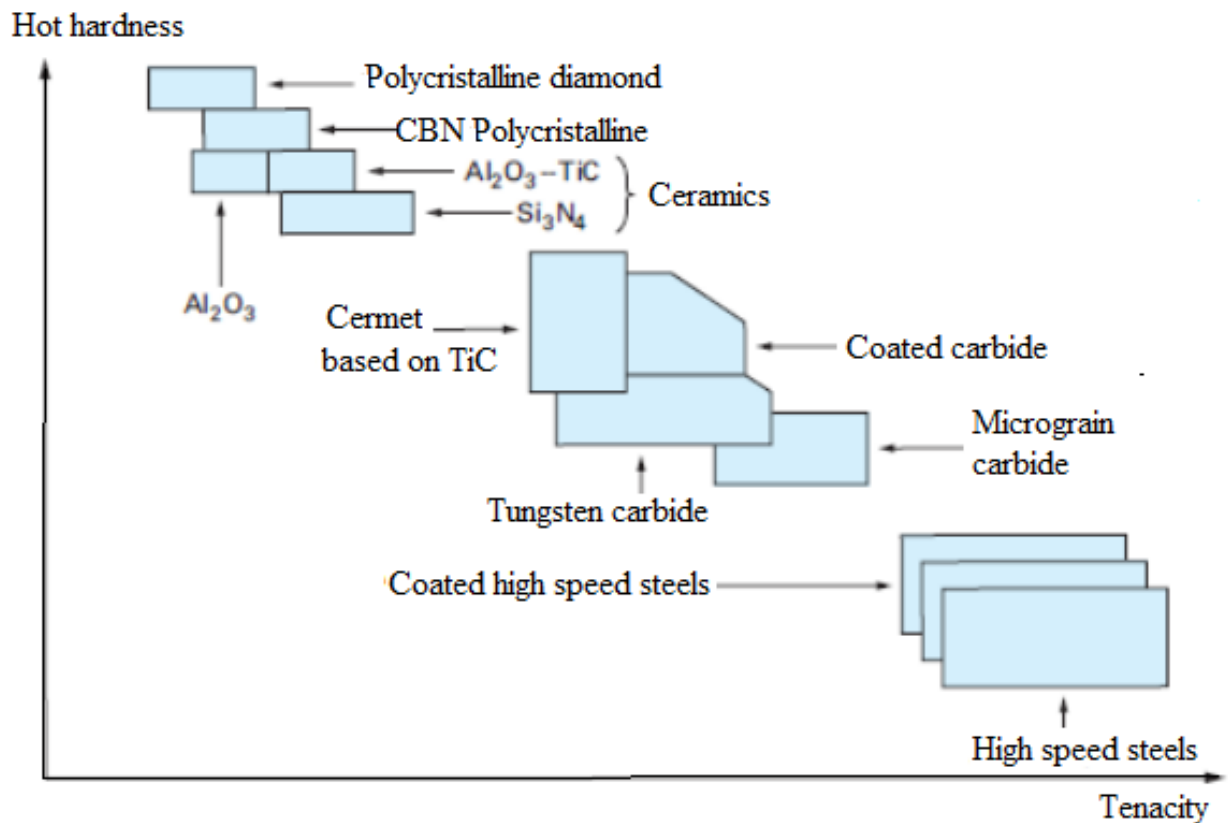


Figure I. 4: Hardness and toughness of cutting materials.

I. 3. 6. Choosing the cutting tool material

The material of the active part of the cutting tools must have certain properties, such as:

Good resistance: To friction, wear, impact, penetration, high temperature (e.g. hot hardness), low adhesion and high chemical stability towards the machined material (air, cutting fluid) as well as availability and economical purchase price.

I. 3. 7. Coating of tools

High-speed steel or hard metal tools are often coated with various layers to improve their tool life, enabling higher machining speeds and better surface finish. Common coating types include titanium nitride (TiN), titanium carbonitride (TiCN), aluminum oxide (Al_2O_3) and silicon nitride (Si_3N_4), which belong to the class of ceramics and can be combined with other materials such as zirconium oxide (ZrO_2), titanium carbide (TiC) or silicon carbide (SiC). Diamond is used for machining non-ferrous metals and abrasive materials such as carbon or ceramics, while boron nitride is used for machining hardened ferrous metals.

I. 4. Cutting tool geometry

I. 4. 1. General

A cutting tool consists of a tool body with one or more active parts.

Each tooth of multi-edged tools (milling cutters, drills, etc.) behaves like an elementary tool whose reference model is the prismatic turning tool. The study of the active part of all cutting tools involves that of the prismatic turning tool.

The cutting insert is the active part in a machining operation. Cutting tools are characterized by their materials and by different geometric parameters. The geometry of the tools is studied to ensure easy chip formation, provide a robust cutting edge and fragment the chips into elements of an acceptable length.

I. 4. 2. Elements of a cutting tool

Generally, a cutting tool can be characterized by an edge geometry and an orientation in space defined by standardized cutting angles. In a cutting tool we find the following elements (figure I.5).

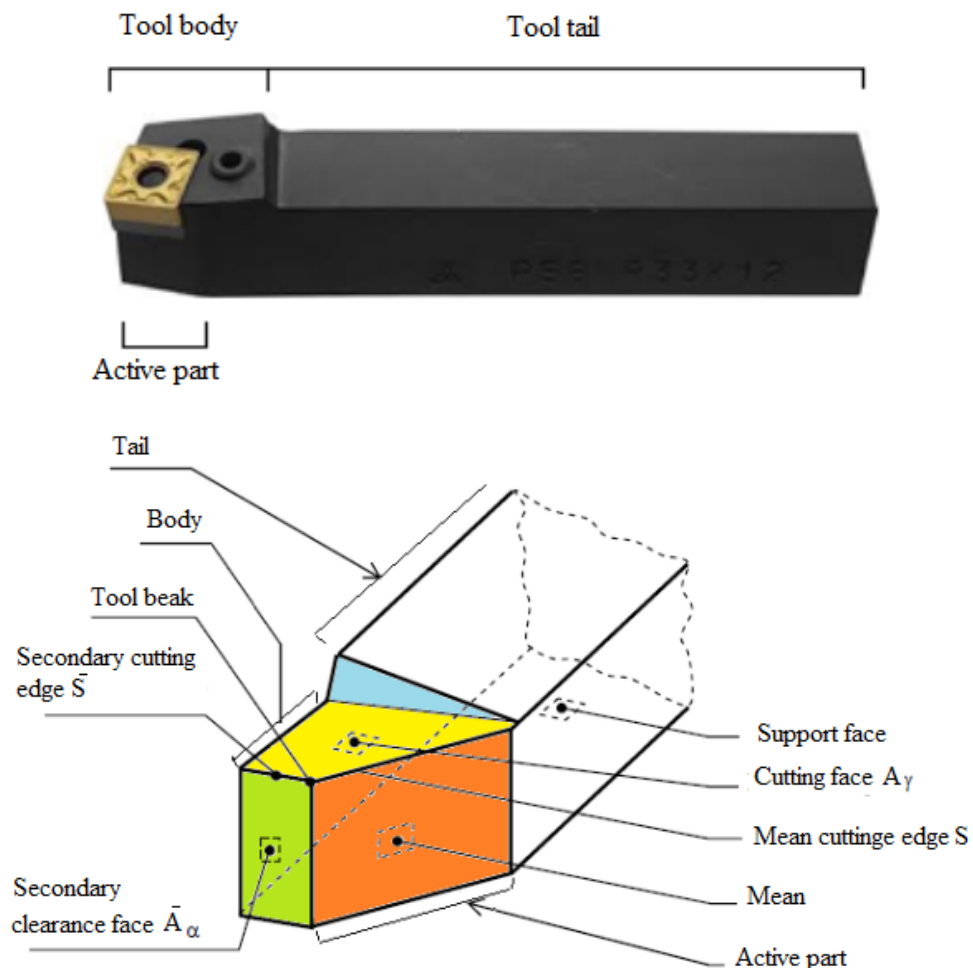


Figure I. 5: Elements of a cutting tool.

I. 4. 2. 1. Tail

Part of the tool that is used to fix it and position it on the machine. For milling cutters and drills, the "shank" is the element that allows the connection of the tool with the tool holder. The shank is cylindrical or conical (Morse taper CM or American standard SA).

I. 4. 2. 2. Tool body

Part of the tool carrying the cutting elements or inserts (sometimes the cutting edges can be cut directly into the body).

I. 4. 2. 3. Active part

This is the part that directly intervenes in the cutting operation. It is composed of the cutting face, the flank faces and the cutting stops. It is characterized by its shape and its material. The hardness of the active part must be greater than that of the metal to be machined.

I. 4. 2. 4. Cutting face A_γ and main clearance faces A_α and secondary A'_α

The rake face is the surface on which the chip slides during cutting, while the main clearance faces A_α and secondary A'_α are the faces in front of which the surface which has just been machined passes (figure I.6).

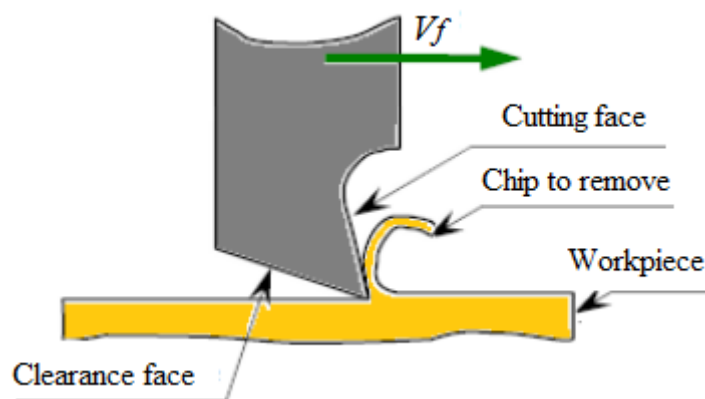


Figure I. 6: Main cutting face and clearance faces.

I. 4. 2. 5. Main cutting edge S

Major cutting edge is the intersection between the cutting face A_γ and the main clearance face A_α , intended for the removal of material.

I. 4. 2. 6. Secondary cutting edge S'

Intersection between the cutting face A_γ and the secondary clearance face A'_α .

I. 4. 2. 7. Tool beak r_ϵ : Junction of the main S and secondary edges S'

I. 4. 2. 8. Tool direction

The direction of the tool is defined by the position of the cutting edge (S). Considering the tool held vertically and the beak at the bottom:

- the tool is said to be right-handed if its cutting edge is oriented to the right (figure I.7a).
- the tool is said to be left-handed if its cutting edge is oriented to the left (figure I.7b).
- If the active part of the tool is symmetrical with respect to the axis of the tool, the latter works indifferently to the right and to the left, the tool is neutral (figure I.7c).

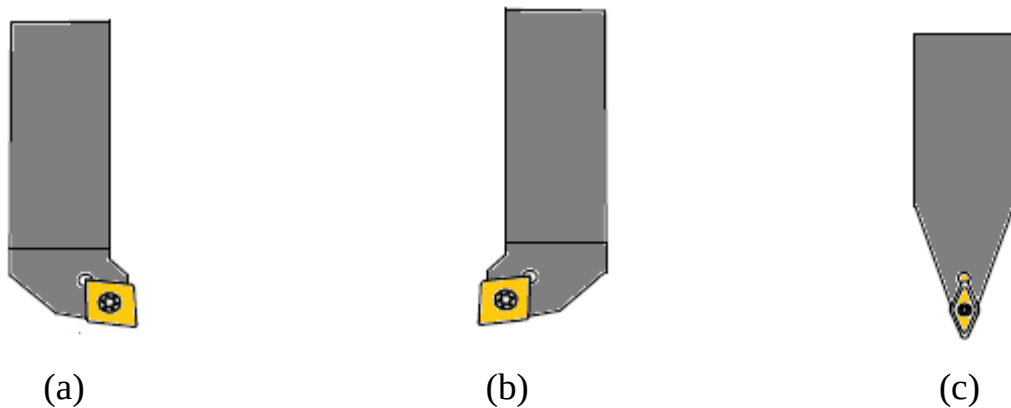


Figure I.7: (a) Straight carbide insert tool R; (b) Left carbide insert tool L; (c) Neutral carbide insert tool.

I. 4. 3. Identifying the tool plans

To be to study the characteristics of the tool, it is necessary to represent and define the tool planes (Figure I.8).

- 1) P_b : base plane, tool support surface.
- 2) M : cutting edge point.

V : Vector shows the cutting direction.

f : Vector represents the direction of feed.

- 3) P_r : reference plane, parallel to P_b and containing M and f.
- 4) P_f : conventional work plan, perpendicular to P_r and containing M, V and f.

5) P_s : tool edge plane, perpendicular to P_r and tangent to the cutting edge at M.

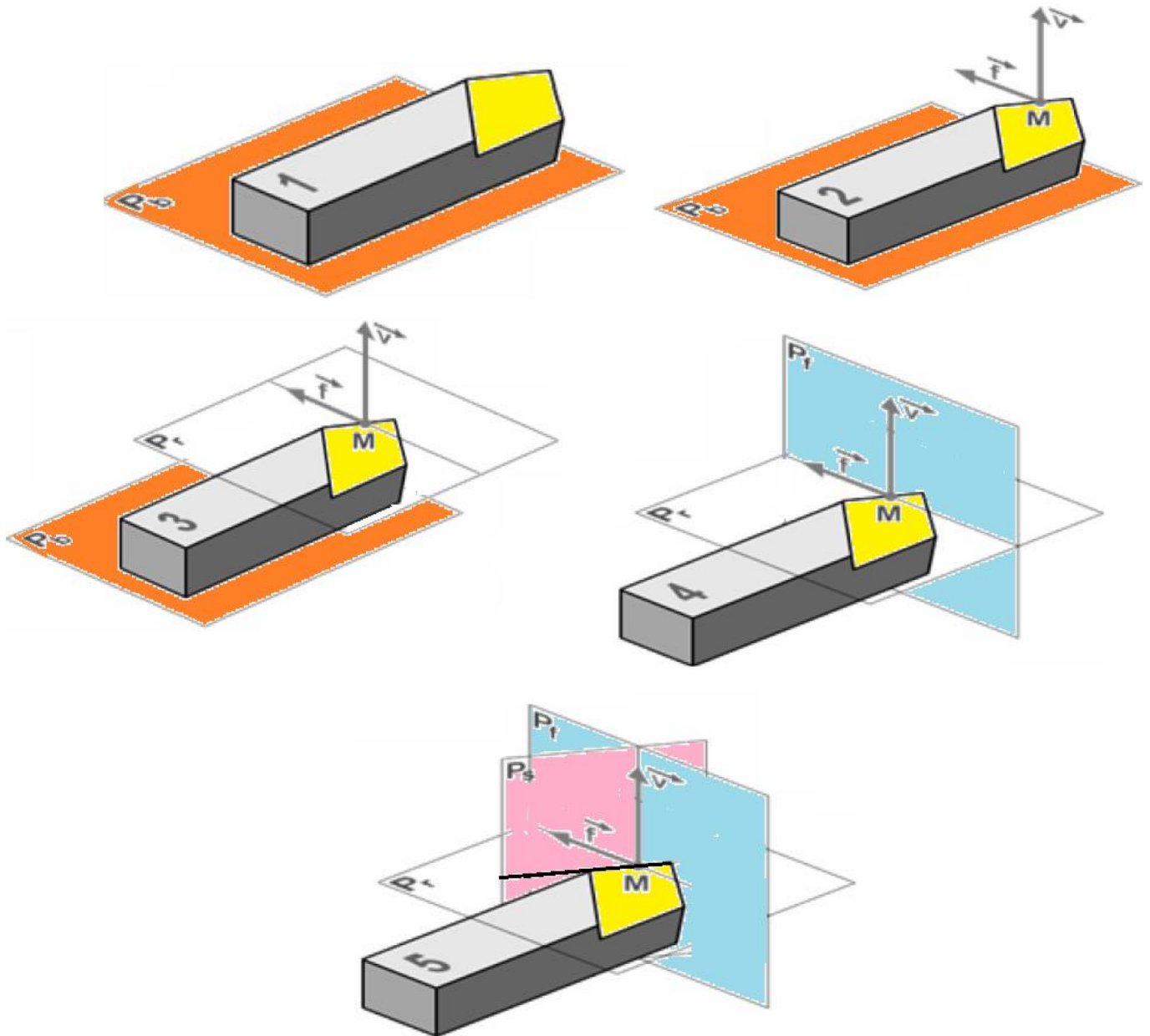


Figure I. 8: Tool plans.

I. 4. 4. Main plans of the active part of the tool

Figure I. 9 shows the different planes of the active part of the tool.

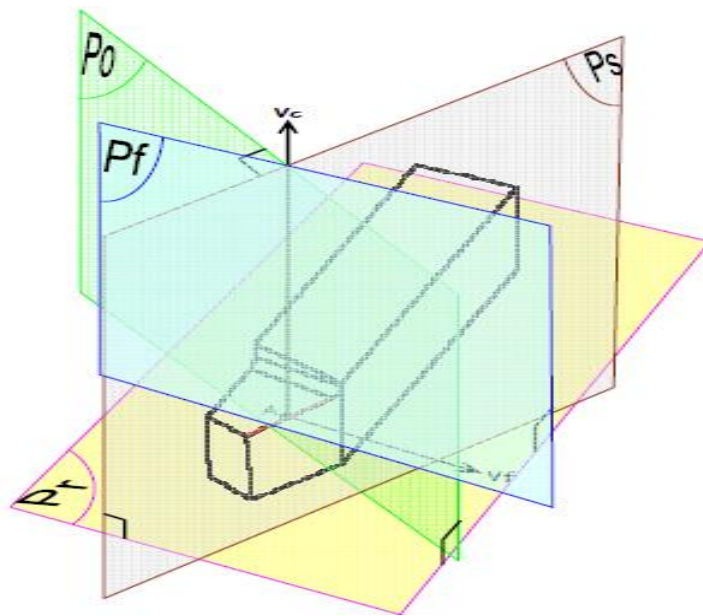


Figure I. 9: Plans of the active part of the tool.

I. 4. 5. Characteristic angles of the cutting tool

I. 4. 5. 1. Tool edge angles

- Edge tilt angle λ_s

Acute angle between P_r and the tangent to the edge, at point A. It can be positive or negative (figure I.10).

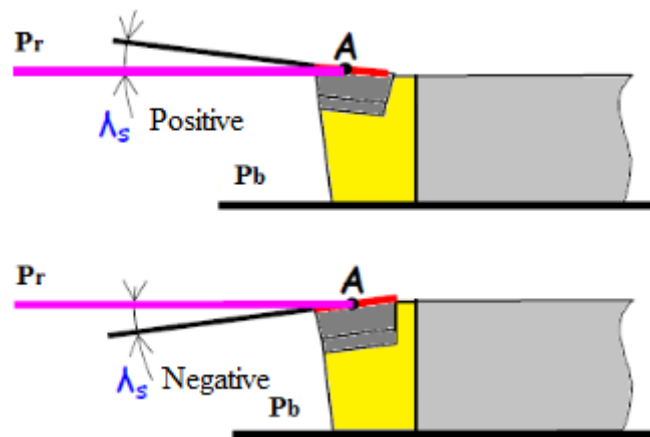


Figure I. 10: Tool edge angles.

- Edge direction angle K_r

Between P_f et P_s .

- Point angle ϵ_r

Between the main cutting edge S and the secondary cutting edge S'.

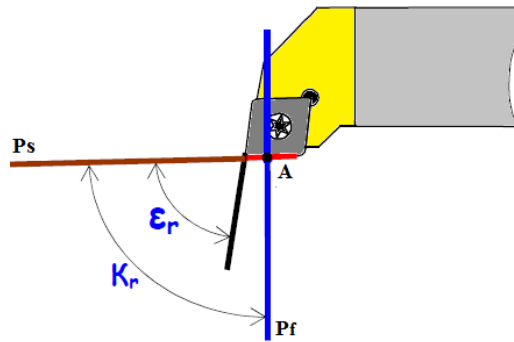


Figure I. 11: Edge and tip direction angles.

I. 4. 5. 2. Tool face angles

These angles are shown schematically in the figure I. 12.

- Orthogonal clearance angle α_o

Between Ps and Aα.

- Orthogonal cutting angle β_o

Between Aα et Aγ

- Orthogonal cutting angle γ_o

Between Pr et Aγ.

$$\alpha_o + \beta_o + \gamma_o = 90^\circ$$

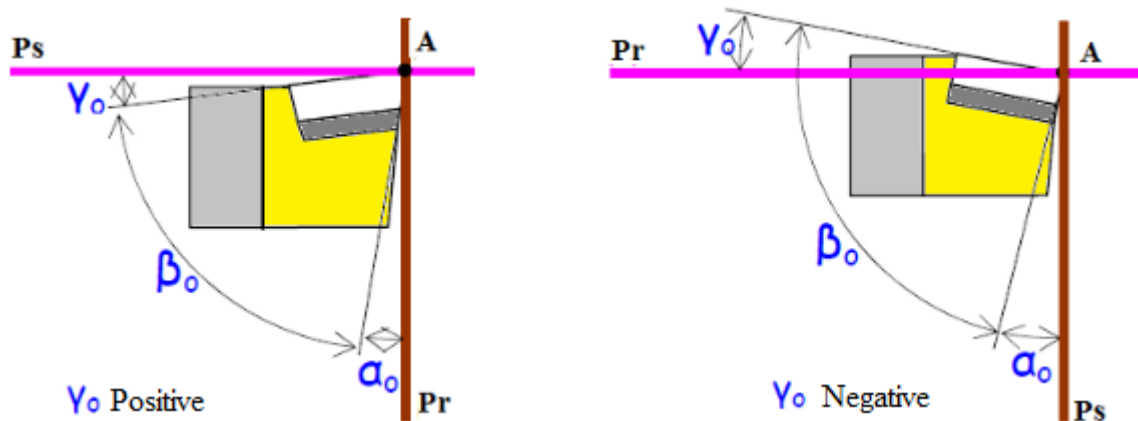


Figure I. 12: Tool Face Angles.

Figure I.13 shows the face angles for a working tool.

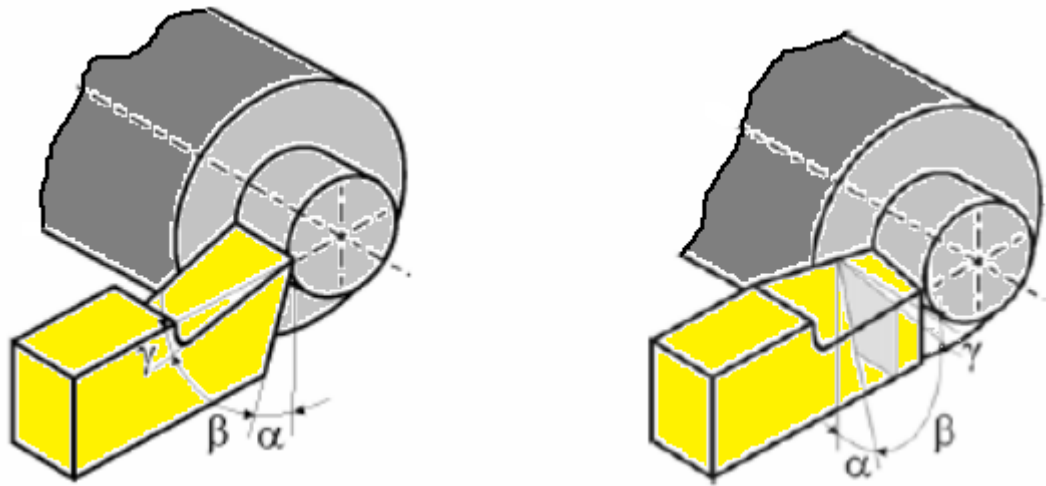


Figure I.13: Face angles for a working tool.

I. 5. Chip formation mechanism

I. 5. 1. Introduction

The machining operation is observed by the production of chips. The excess material to be removed is the layer that will form the chip under the complex mechanical action of a cutting tool.

I. 5. 2. Chip formation

The chip is formed by plastic deformation under complex mechanical actions. A chip is a small piece of material created following machine tool machining of a material such as wood, metal, plastic, etc.

The analysis and understanding of the chip formation process is necessary for the improvement and development of machine tools and cutting means. The accumulation of material in front of the tool generates action forces from the tool and reaction forces from the workpiece, until the material is removed.

- Whatever the material machined, the chip produced responds to the same characteristics which correspond to the three movements influencing the dimensions of the chip (figure I.14):
- Cutting movement: expressed in meters per minute (m/min), it corresponds to the developed length of the chip,
- Penetration movement: or depth of pass, expressed in mm, it corresponds to the width of the chip and influences the adjustment of the dimension of the part,

- Feed movement: expressed in millimeters per revolution, per stroke or per tooth; it corresponds to the thickness of the chip.

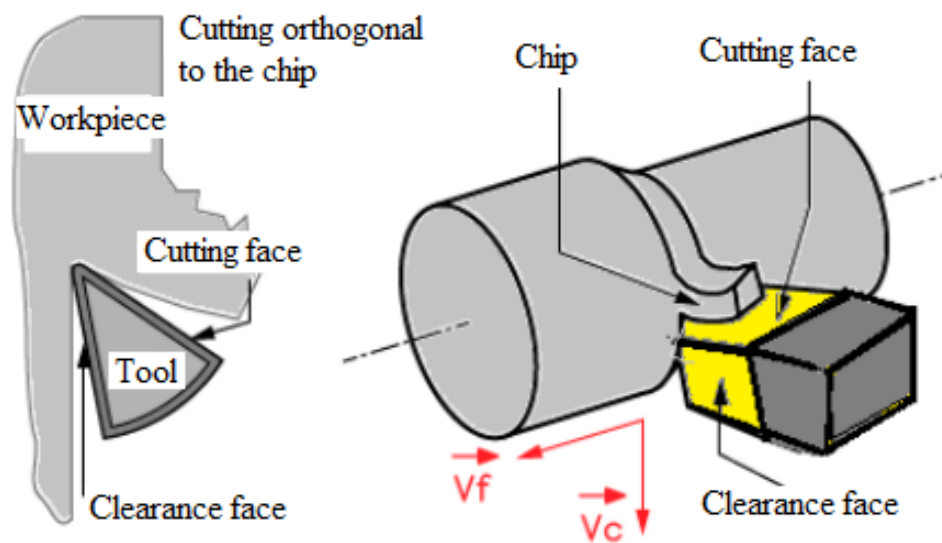


Figure I.14: Movements influencing chip dimensions.

I. 5. 3. Chip formation zones

Several studies have shown that when a part is machined close to the cutting edge, it presents very distinct stress zones. These zones are identified through the mechanical interaction of materials (thermomechanical analysis) and the cutting process (tribological analysis). By performing an orthogonal cut, four main zones are observed (Figure I.15).

Area 1: Is called the metal separation zone by the cutting edge at point (O). Given the significant compression forces, this results in temperature increases of around 600°C.

Area 2: Chip formation zone where we observe a sudden change in orientation and flow speed of the material (primary shear) with a significant heat flow transmitted to the tool.

Area 3: Sliding zone at the chip/cutting face interface (secondary shear). An adhesion phenomenon considerably slows down the flow of the material causing plastic shear of the material. The chip/cutting face friction is intense enough to generate heat of around 750°C.

Area 4 : This zone is located at the clearance face/machined surface interface, where sliding friction is observed, giving rise to adhesion phenomena leading to plastic shear.

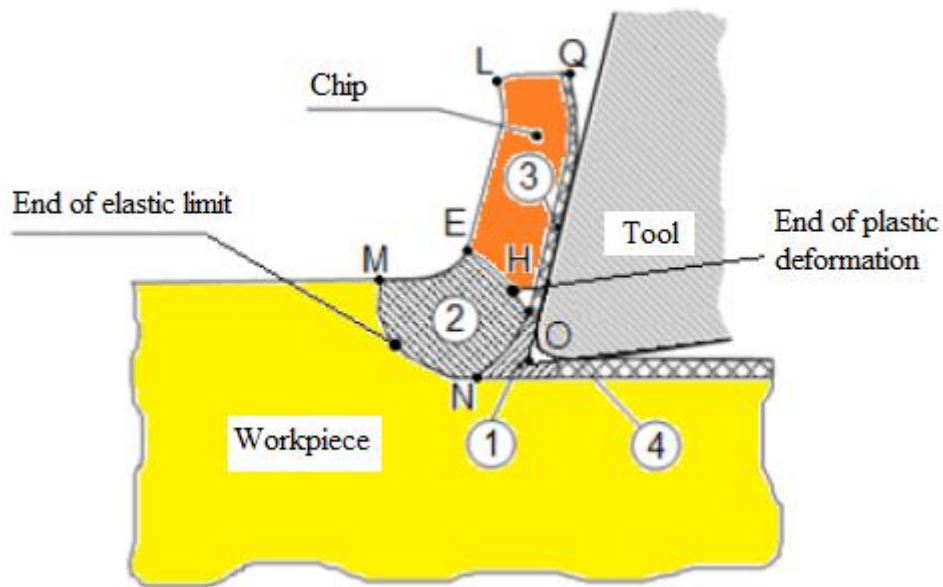


Figure I. 15: Chip formation areas.

I. 5. 4. Main factors influencing chip formation

Are: Cutting speed, feed rate, depth of cut, lubrication, tool geometry and tool and workpiece materials.

I. 5. 5. Different shapes of chips

Chips can be classified into two distinct types:

- **Continuous:** The continuous chipping occurs in cutting ductile metals.
- **Discontinuous:** Observed for cutting hard materials and another ductile type fracture mode is observed for less hard materials but at higher cutting speeds.

There are also other intermediate chips between a continuous chip and a discontinuous chip and other geometries. (Figure 1.16).

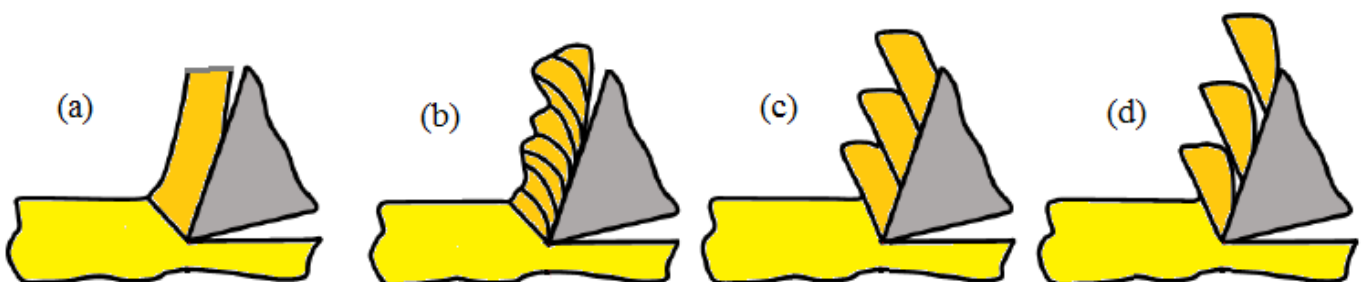


Figure I. 16: Chip shapes in orthogonal section: a) Continuous chip, b) Wavy chip, c) Segmented chip, d) Fragmented chip.

It is necessary to avoid:

- A long, stringy chip that results in reduced tool life, caused by excessive heating, chipping, a built-up edge, edge breakage.

- A very fragmented chip promotes the appearance of vibrations of the part with a rough surface condition.

Figure I.17 shows some chip shapes generated after machining.

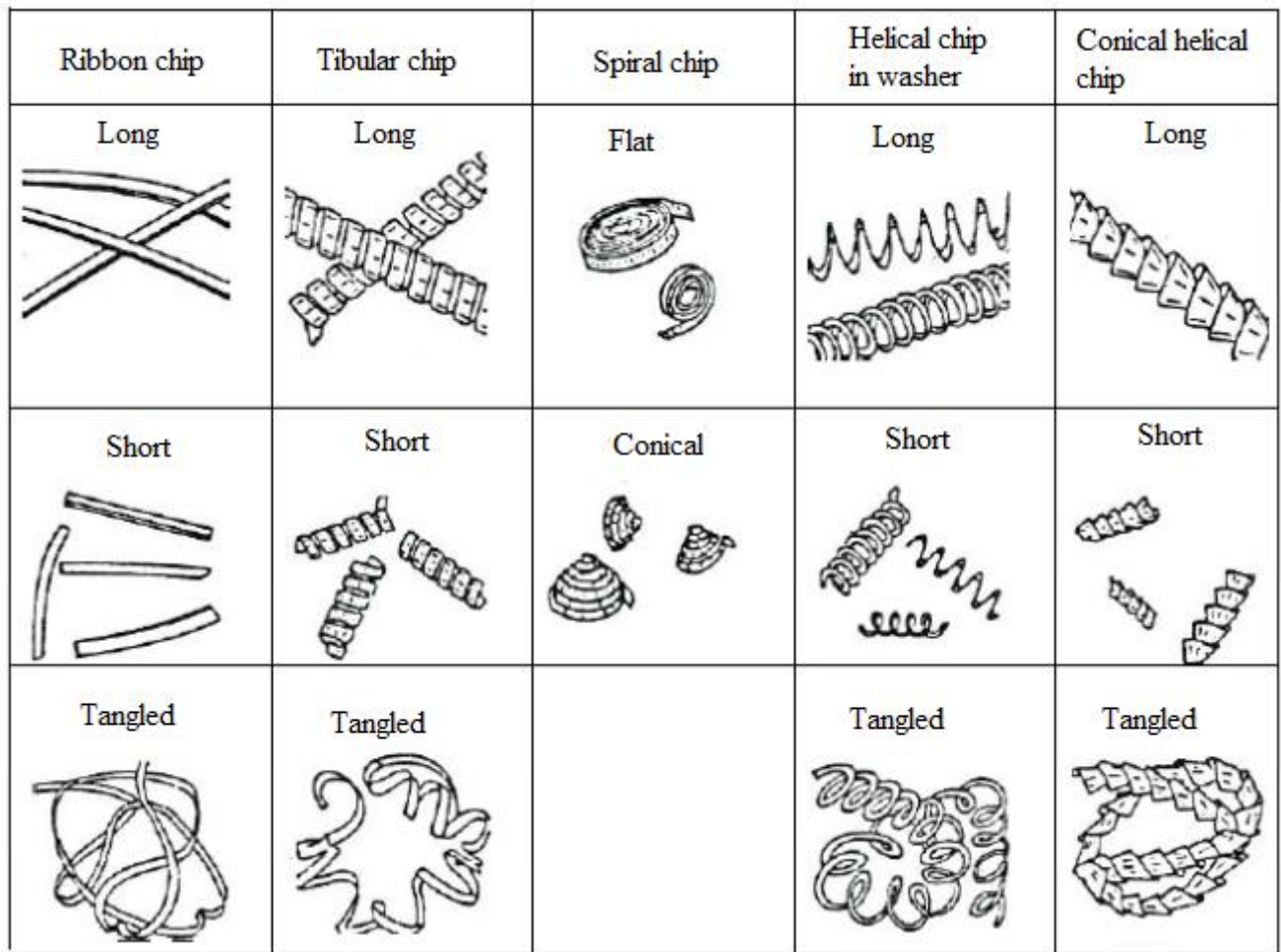


Figure I. 17: Chip shapes generated after machining.

I. 6. Cutting efforts

I. 6. 1. Introduction

Chip formation cannot take place without an essential force and therefore a cutting power P_c required at the spindle. The study and approximation of cutting forces are necessary to choose the tools and size the workpiece holder (the dimensioning of the machining fixtures), their directions make it possible to determine the direction of movement of the tools so that the supports of the fixture oppose these forces (the control of the clamping of the part and the tool).

I. 6. 2. Study of cutting efforts

In the example of a turning operation on a lathe, the chip slides on the cutting face.

The cutting force can be broken down into three main directions (figure I.18).

- Tangential force F_t (F_z): Main cutting force, due to the cutting movement. It is a component in the direction of the cutting speed.

- Axial force F_a (F_x): Feed force. It is a component in the direction of tool feed.
- Radial force F_r (F_y): Penetration force, component in the direction perpendicular to the other two (in the direction of the penetration movement), called,

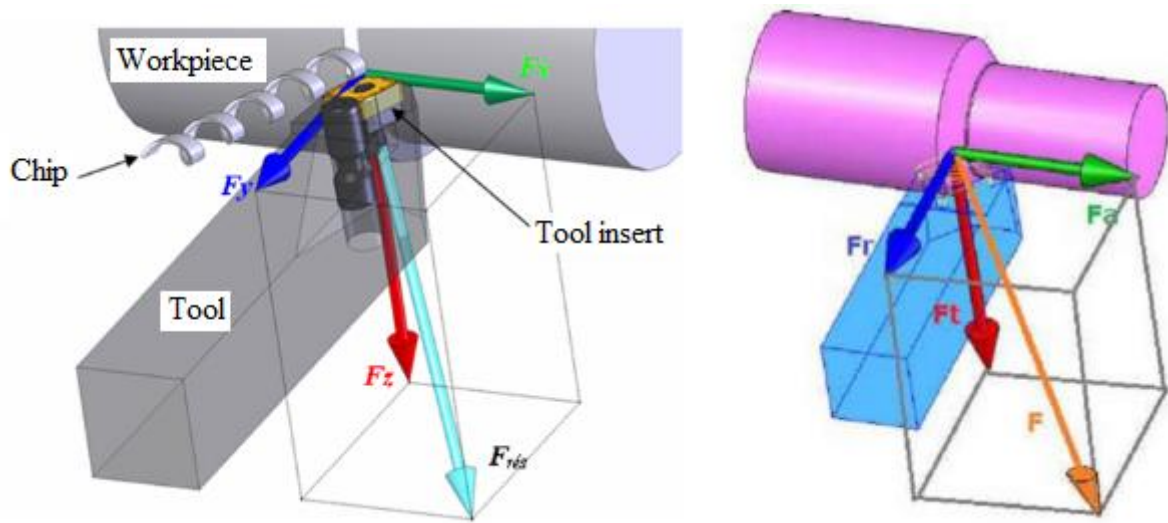


Figure I. 18: Cutting forces of a turning operation.

$$F_{rés} = \sqrt{F_x^2 + F_y^2 + F_z^2}$$

$$F_a = (0,3-0,6).F_t$$

$$F_r = (0,1-0,4).F_t$$

So the most important component is F_t .

The result can be :

$$\sqrt{(0.45 \cdot F_t)^2 + (0.25 \cdot F_t)^2 + (F_t)^2}$$

$$F = 1.2 * F_t \simeq F_t$$

I. 6. 3. Calculation of cutting forces and powers

I. 6. 3. 1. Cutting efforts

The cutting force is expressed by the relationship:

$$F_c = K_s. S_c = K_s. a. f$$

F_c : Cutting force (daN)

S_c : Chip section, $S_c = a. f$ (mm²)

a : Depth of pass value.

f : Feed value.

K_s : Specific cutting coefficient (daN/mm²)

The K_s factor is determined experimentally by a series of tests; it depends essentially on the

nature of the material to be machined (table I.2).

Machined material		Specific cutting Coefficient K_c (daN/mm ²)			
Feed (mm/rev) $\longrightarrow$		0,1	0,2	0,4	0,8
General purpose steel	S 185 – S 275	360	260	190	140
	S 355	400	290	210	150
	E 330	420	400	220	160
	E 360	440	315	230	165
Alloy steels	Manganese steel	470	340	240	180
	Nickel chromium steel	500	360	260	180
	Chrne molybdenum steel	530	380	270	200
	Stainless steel	520	370	270	190
Unalloyed steels	C 40	320	230	170	125
	C 50	360	260	190	140
	C 60	390	290	210	150
Cast iron	FGL 150	190	140	100	70
	FGL 250	290	210	150	110
	Alloy cast iron	320	230	170	120
	Malleable cast iron	240	170	120	90
Copper alloy	Brass	160	110	90	60
	Bronze	340	240	180	130
Aliminum alloy	Rr<19	110	80	60	40
	19<Rr<27	140	100	70	50
	27<Rr<37	170	120	80	60

Table I. 2: Some values of the specific cutting coefficient.

Example of calculating the cutting force

Let's say a turning operation with a knife tool on an FGL150 axis (gross diameter: 100 mm, machined diameter: 96 mm). The machining will be done under the following conditions:

$$V_c = 20 \text{ m/min}$$

$$f = 0,4 \text{ mm /tr}$$

$$K_c = 100 \text{ daN/mm}^2$$

The cutting force is:

$$F_c = 100 \times 2 \times 0,4 = 80 \text{ daN}$$

I. 6. 3. 2. Cutting power

It is necessary for machining, the electrical power that the motor must provide is greater to ensure the rotation of the spindle and the movement of the saddle during the turning operation for example. During machining, there is power consumed by cutting and power lost by the friction of moving parts of the gearbox (figure I.19).

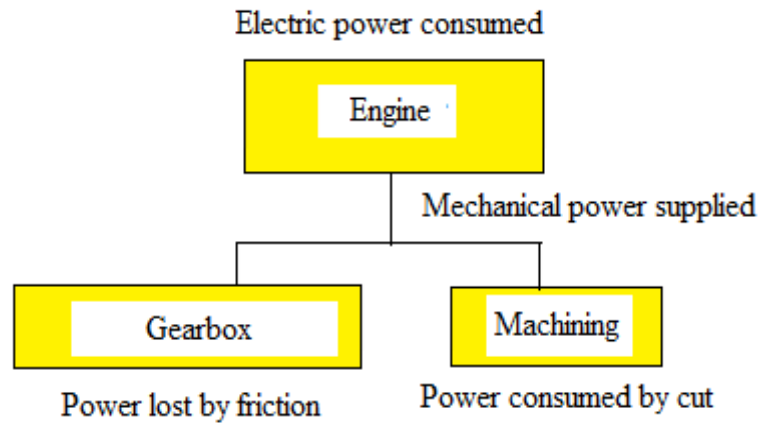


Figure I.19: Different powers during machining.

The cutting power (P_c) depends mainly on the cutting speed (V_c) and the tangential cutting force (F_c). $P_c = F_c \cdot V_c / 60$

V_c (m/min)

On the machine's motor nameplate, the power indicated corresponds to the mechanical power supplied by the motor shaft.

The efficiency of the machine's drive train η

$\eta = \text{Power consumed by the cut} / \text{Mechanical power supplied}$.

I.7. Heating (Cutting temperature)

Chip formation is accompanied by a significant release of heat due to the friction of the chip sliding on the rake face and the friction of the part on the clearance face of the tool. The cutting temperature represents an average value of the thermal field of a particularly stressed area, such as the tool/chip interface for a tool. Figure I. 20 shows the temperature distribution on the active part of the tool during dry machining of a steel with $V_c = 60$ m/min. The temperature gradient in the tool can reach $700^\circ\text{C}/\text{mm}$. The hardness of the cutting material decreases from certain temperatures. For high-speed steels: 500°C to 600°C , carbides: 800°C to 900°C and ceramic: 1400°C .

The temperature of the active part of the tool must be reduced so as not to cause it to lose its hardness.

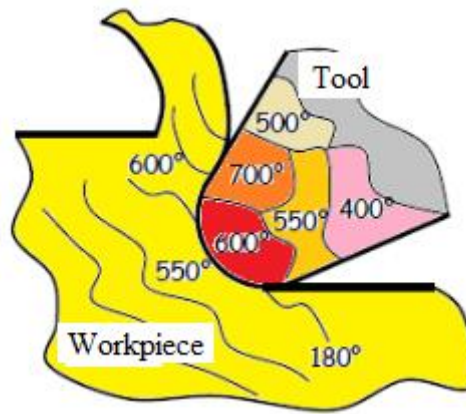


Figure I. 20: Temperature distribution on the active part of the tool.

- Effects of cutting temperature

Raising the cutting temperature is harmful to the cutting tool and the workpiece. The harmful effects on the cutting tool are:

- Rapid wear of the tool, which reduces its cutting life.
- Plastic deformation of cutting edges if the tool material is not hard enough.
- Damage to cutting edges due to thermal shock.

The harmful effects of temperature on the part:

- Lack of precision in the dimensions of the part due to the plasticity of the metal.
- Expansion and contraction during and after machining.
- Damage to the surface due to oxidation, rapid corrosion, inflammation.

Practical example of the effect of cutting temperature

Turning of an AL4G part with a carbide tool without lubrication (figure I.21). The measurement of L during machining ($\theta = 90^\circ \text{C}$) gives us a length $L_1 = 200.36 \text{ mm}$, another measurement carried out at the control station with $\theta = 20^\circ \text{C}$ gives us a length $L_2 = 200.09 \text{ mm}$.

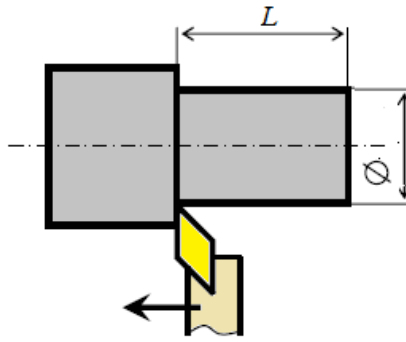


Figure I. 21: Turning a part in AL4G.

After cooling, the part has shrunk. Therefore, if the machinist does not take this into account, the dimension L may be out of tolerance and the same applies to the diameter of the part. If you want to do precision work, you must limit the temperature of the part being machined by lubrication.

I. 8. Damage to cutting tools

The localized temperature rise in the chip formation zones is the source of very complex physicochemical phenomena. This temperature rise is due to friction between the tool and the part and contact with the chips. This friction acts on the tool and causes its wear (damage) and consequently its rapid degradation, thus reducing its service life.

Cutting tool wear consists of a transfer of metal between the surfaces in contact (part, tool and chip). This transfer is caused by the mechanical and thermal stresses associated with cutting. During machining and depending on the cutting conditions, there are two types of tool wear (figure I.22):

Crater wear produced by intense friction of the chip on the cutting face (zone 1).

Crater wear causes brittleness of the edge. When wear has progressed to the secondary edge, the surface condition becomes poor.

- Flank wear produced by friction of the machined surface on the tool's flank surface (zone 2). Flank wear is the most significant. It directly influences the manufactured dimension and the surface condition.

The most important is the flank wear. It directly influences the manufactured dimension and the surface condition.

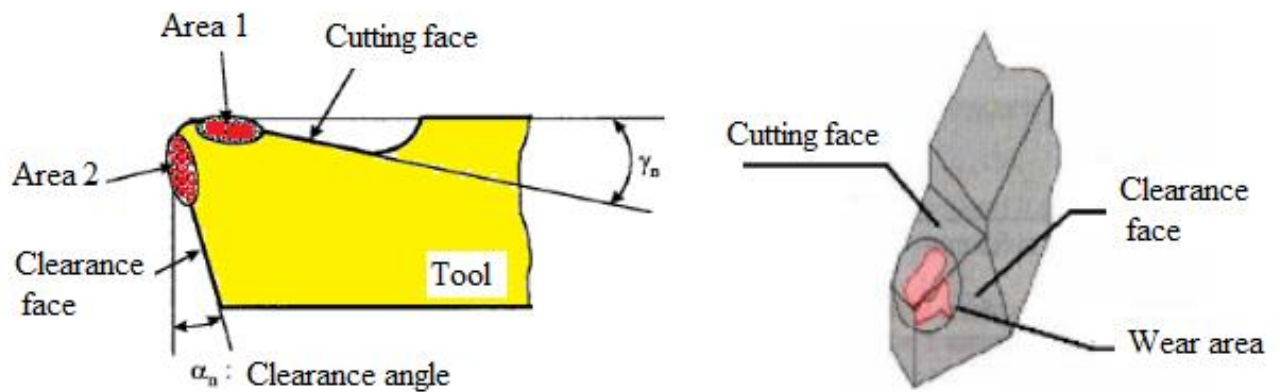


Figure I. 22: Types of tool wear.

Table I.3 shows some types of tool wear.

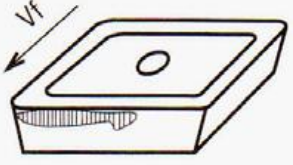
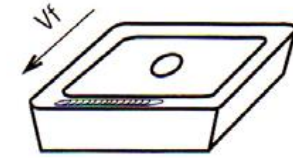
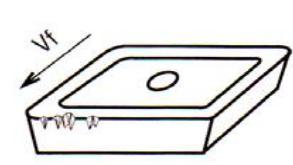
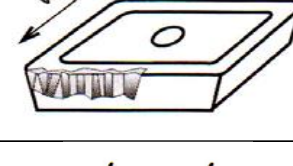
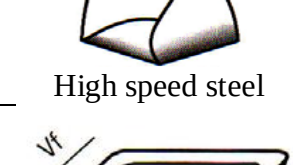
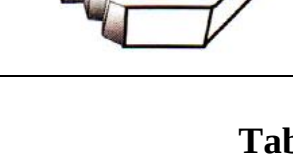
Default	Illustration	Causes	Remedies
Wear in undercut Usure en dépouille		Cutting speed too high Insufficient wear resistance Tool above center. Insufficient coolant.	Reduce cutting speed Choose a more resistant grade. Center the tool. Use coolant.
Crater wear Usure en cratère		Unsuitable cutting conditions. Diffusion wear due to excessive temperature on the cutting face.	Reduce the cutting speed or feed Choose a more resistant grade Choose positive γ of the tool.
Chipping of the edge		Too fragile grade Poor tool part stability Tool below center. Poorly chosen cutting conditions.	Take a tough grade. Improve stability. Center the tool. Reduce feed at the start of the cut.
Edge reported Adherent chip		Vc too low Tool geometry unsuitable for the machined material. Unsuitable feed .	Increase Vc. Use a chipbreaker. Increase the feed.
Burning of the edge		Unsuitable cutting conditions. Tool geometry unsuitable for the material being cut Insufficient coolant.	Reduce cutting speed, Nozzle radius too small. Check coolant.
Tool breakage		Unsuitable cutting conditions. Impacts, poor clamping. Defect in the part.	Check all machining conditions. Take a tough grade.

Table I. 3: Types of tool wear.

I. 9. Methodology for choosing cutting parameters

To obtain a satisfactory work (good condition of the machined surface, speed of machining, moderate tool wear, ...) the cutting parameters must be adjusted.

The cutting parameters are constant or variable elements, which depend on each other in order to obtain the best possible compromise for machining. Determining the optimal cutting conditions is a task that has technical and economic importance.

I. 9. 1. Cutting parameters

First of all, it is necessary to clarify:

The type of machine used for machining (lathe, milling machine, drill) and its power.

The material cut (steel, cast iron, aluminum alloys, etc.).

The cutting material (ARS, carbide, etc.).

The type of operation (sliding, dressing, roughing, finishing, surfacing, drilling, etc.).

It is necessary to determine certain specific parameters, such as:

I. 9. 1. 1. Cutting speed

The cutting speed is the speed of a point of the part in contact with the tool during turning (figure I.23a). In milling or drilling, it is the speed of a point of the cutting edge of the tool (figure I.23b). It is evaluated in circumferential path, that is to say in length of circumference traveled by the part.

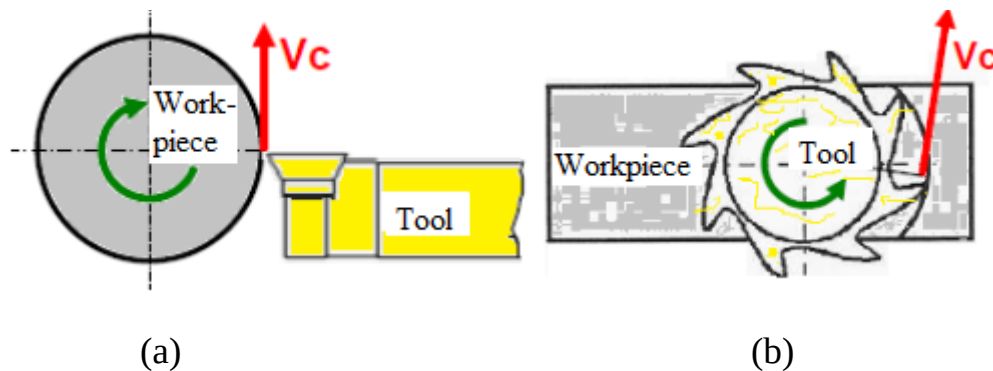


Figure I. 23: Cutting speed in machining.

By designating by D the diameter of the circle described in millimeters (mm), by N the rotation speed in revolutions per minute (rpm) and by V_c the linear speed in meters per minute (m/min),

we will have: $V_c = \pi \cdot D \cdot N$

In any machining problem, it is necessary for technological and economic reasons to determine the value of V_c best suited to the work to be carried out.

The cutting speed has a major influence on the tool life. It varies in particular with the material to be machined, the tool material, the nature of the operation (roughing or finishing), the choice of feed f , the lubrication conditions (dry or lubricated work) and the power of the machine. The cutting speed must be chosen correctly, since if it is too slow: Loss of time (economic criteria) and too fast: Increase in temperature (rapid deterioration of the tool). Tool manufacturers recommend cutting speeds established from laboratory experiments, they are mentioned in tables. These experiments make it possible to obtain the best compromise between the maximum tool life and the maximum material removal for economic purposes (the choice of cutting speed influences the cost price of the manufactured product).

The cutting speed V_c chosen is suitable for determining the rotation frequency N .

The values of V_c and the feeds f are indicated in tables I. 4, I.5. and I.6.

Material to be machined Feed f mm/rev	Exterior turning				Thread turning	
	High speed steel		Carbide		High speed steel	Carbide
	0.05 à 0.1	0.1 à 0.2	0.05 à 0.2	0.2 à 0.3	$f = \text{pas du filet}$	
Unalloyed steel	50	40	250	200	35	120
Low alloy steel	30	20	150	130	20	80
High alloy steel	20	15	120	100	15	60
Low alloy cast steel	30	20	150	120	20	75
Stainless steel	25	20	150	130	20	90
Lamellar cast iron	40	30	80	60	20	30
Modular cast iron	30	25	100	80	15	40
Spheroidal cast iron	55	45	90	70	25	40
Low hardness Aluminum alloy without silicon AW2030	250	200	550	400	150	230
Hard Aluminum alloy without silicon AW2017, AW6060	120	80	250	200	90	110
Aluminum alloy with high silicon content more 12 percent	80	40	120	100	45	60

Table I. 4: Values of cutting speed and feeds in turning.

Material to be machined Feed fmm/tooth/rev	Milling			
	High speed steel		Carbide	
	0.03 à 0.1	0.1 à 0.2	0.05 à 0.2	0.2 à 03
Unalloyed steel	50	40	140	120
Low alloy steel	30	25	100	80
High alloy steel	20	15	80	70
Low alloy cast steel	25	20	90	80
Stainless steel	20	15	100	90
Lamellar cast iron	35	30	100	90
Modular cast iron	30	25	80	70
Spheroidal cast iron	40	35	100	90
Low hardness Aluminum alloy without silicon AW2030	250	200	500	400
Hard Aluminum alloy without silicon AW2017, AW6060	120	80	300	200
Aluminum alloy with high silicon content more 12 percent	80	40	120	80

Table I. 5: Values of cutting speed and feeds in milling.

Material to be machined	Drilling		
	High speed steel	Coated high speed steel	Carbide
Unalloyed steel	30	45	70
Low alloy steel	20	40	60
High alloy steel	15	35	40
Low alloy cast steel	10	30	70
Stainless steel	12	20	40
Lamellar cast iron	25	50	80
Modular cast iron	15	30	80
Spheroidal cast iron	25	50	80
Low hardness Aluminum alloy without silicon AW2030	60	90	100
Hard Aluminum alloy without silicon AW2017, AW6060	60	90	100
Aluminum alloy with high silicon content more 12 percent	40	60	100
Cutting speed Vc: m/min			

Table I. 6: Values of cutting speed and drilling feeds.

I. 9. 1. 2. Rotation frequency

The rotation frequency N is the angular speed of a point considered on the rotating element (part or tool) for a given diameter in revolutions per minute.

$$N = \frac{1000 V_c}{\pi D}$$

In turning: D is the diameter of the part in mm;

In milling: D is the diameter of the cutter in mm.

The rotation frequency depends on the diameter of the rotating element (part or tool) and the chosen cutting speed.

I. 9. 1. 3. Cutting feed

The feed f is called the displacement of the point considered on the cutting edge in millimeters for 1 revolution (mm/rev) in turning, also per tooth (mm/tooth) in the case of milling (figure I.24).

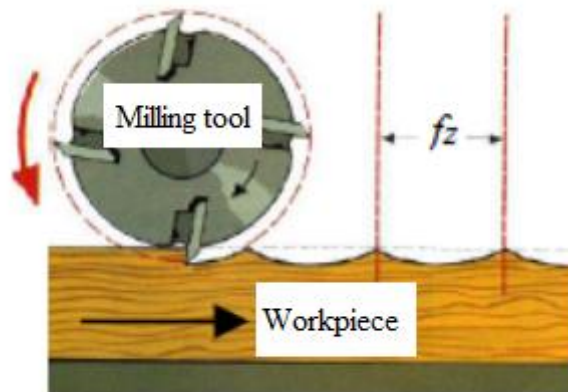


Figure I. 24 : Feed per tooth.

The feed is a key value to determine the quality of the surface condition of the machined part and to ensure that chip formation is well within the limits of the insert geometry. It is taken larger in roughing than in finishing, the feed values are also given by charts.

In roughing

The objective of the roughing operation is to remove a maximum volume of material in a minimum time and a minimum cost provided that the machine must be sufficiently powerful, as well as the part/part holder attachment. Table I.7 groups together some values of the feed f as a function of the radius r_e of the tool in roughing operation.

Radius r_e (mm)	0.4	0.8	1.2	1.6	2.4
Max feed f (mm/tr)	0.25 à 0.35	0.4 à 0.7	0.5 à 1	0.7 à 1.3	1 à 1.8

Table I. 7: Values of the feed f as a function of the radius r_e in roughing.

In finishing

Finishing work is to respect the tolerance intervals and the surface condition requirements (roughness). The surface must be smooth; the dimensions must be correct. As the forces involved are lower than for roughing, the power of the machine is no longer a primary criterion.

The feed values for the turning operation are presented in Table I.8.

Materials to be machined	External carriage		External carriage	
	<i>f</i> in draft	<i>f</i> in finishing	<i>f</i> in draft	<i>f</i> in finishing
Mild steels	0,1 – 0,4	0,05 – 0,2	0,05 – 0,3	0,05 – 0,2
Hard steels	0,1 – 0,4	0,05 – 0,25	0,05 – 0,6	0,05 – 0,2
Cast irons	0,1 – 0,8	0,05 – 0,2	0,05 – 0,6	0,05 – 0,2
Hard cast irons	0,1 – 0,6	0,05 – 0,25	0,05 – 0,5	0,05 – 0,2
Aluminium alloys	0,1 – 0,8	0,05 – 0,25	0,05 – 0,5	0,05 – 0,05
Bronze	0,1 – 0,6	0,05 -0,25	0,05 – 0,5	0,05 – 0,2

Table I. 8: Feed values for the turning operation.

A finishing feed rate lower than the roughing feed rate is chosen. The roughness index $R_a = 0.6$ indicates a better surface quality than a roughness index $R_a = 3.2$.

I. 9. 1. 4. Feed rate V_f

The feed rate is the translation speed of a considered point of the tool or part that moves during machining (which has the feed movement M_f).

In the case of milling, the feed rate V_f [mm/min] is given by:

$$V_f = z \cdot f_z \cdot N \text{ [mm/min]}$$

Where z is the number of teeth on the cutter, f_z in mm/(rev.tooth) corresponds to the distance that the tooth will travel at each turn of the cutter.

In the case of turning, the feed speed V_f [mm/min] is given by:

$$V_f = f_z \cdot N \text{ [mm/min]}$$

Too high a feed rate creates excessive stress on the part and on the tool, which can cause the tool to break.

I. 9. 1. 5. Depth of pass a

The depth of cut corresponds to the length of the cutting edge penetrated into the material, in the case of orthogonal cutting, and to the difference between the radius of the part before and after machining, in the case of turning. The depth of cut is always measured perpendicular to the direction of the feed and not along the edge of the tool (figure I.25.).

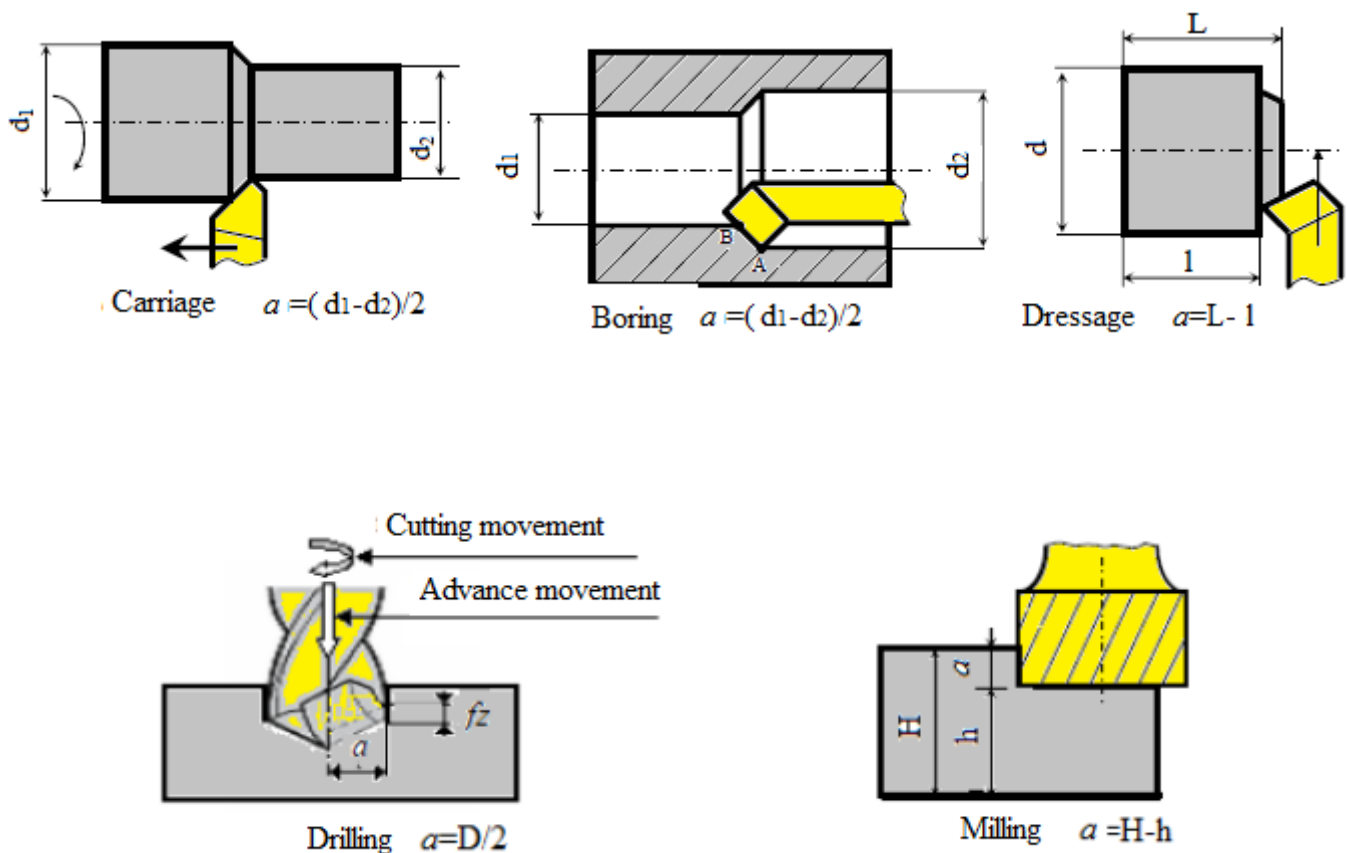


Figure I.25: Depth of pass.

The depth of pass parameter depends on the excess thickness of the material to be machined as well as the nature of the operation (roughing or finishing).

I. 9. 2. Cutting parameters in drilling

Drilling with a drill bit combines two movements: rotation and translation. These two movements are characterized by:

- **Drill rotation speed:** expressed in revolutions per minute and noted N , at the periphery of the drill (figure I. 26a), it corresponds to a speed:

$$N = \frac{1000 V_c}{\pi D}$$

D : Drill diameter (mm)

- Feed

The feed expressed in mm per revolution and noted $f(\text{mm/rev})$.

- Feed rate

Figure I. 26b shows the feed rate of the drill. The feed rate is given by:

$$V_f = 1000f \cdot N$$

V_f (mm/min), f (mm/tr), N (tr/min)

If the drill has two main cutting edges, then the feed per edge is $f/2$.

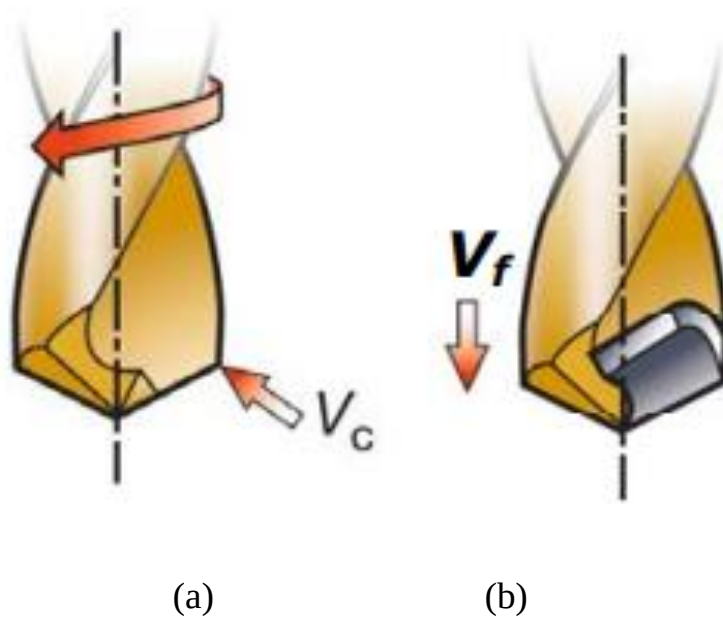


Figure I. 26: Rotation speed and feed.