Taps – Number of flutes and relief angles

Number of flutes

The number of teeth (or flutes) can vary from 2 to 6 ÷ 8 depending on the diameter and the material being processed.
The number of cutting edges is linked to diameter of the core, the profile of the flutes and the amplitude of the cutting edge sector.
The table N°1 lists the values of the core diameter $\mathbf{d_5}$ according to the nominal diameter $\mathbf{d_1}$ and the angle of the sector $\boldsymbol{\varphi}$ for different workpiece materials.

Table N°1

Number of flutes	*Material of the workpiece*	*Diameter of the core d_5*	*Angle of the sector φ*
2 flutes	*Cast iron – brass - bronze*	*$0,3 \cdot d_1$*	*60°÷70°*
	Steel	*$0,3 \cdot d_1$*	*65°÷80°*
	Aluminum	*$0,25 \cdot d_1$*	*65°÷85°*
3 flutes	*Cast iron – brass - bronze*	*$0,4 \cdot d_1$*	*34°÷41°*
	Steel	*$0,4 \cdot d_1$*	*30°÷37°*
	Aluminum	*$0,35 \cdot d_1$*	*21°÷29°*
4 flutes	*Cast iron – brass - bronze*	*$0,5 \cdot d_1$*	*28°÷32°*
	Steel	*$0,5 \cdot d_1$*	*25°÷30°*
	Aluminum	*$0,5 \cdot d_1$*	*21°÷27°*
6 flutes	*Cast iron – brass - bronze*	*$0,63 \cdot d_1$*	*22°÷24°*
	Steel	*$0,63 \cdot d_1$*	*19°÷23°*
	Aluminum	*$0,63 \cdot d_1$*	*19°÷21°*

We must make some clarifications about the values of the table.
For taps with 3 cutting edges with an outside diameter of 5 mm, the core diameter will be $0.5 \cdot d_1$ for reasons of strength.
The angle of the minimum sector will be used for blind holes or long through holes, in order to have the maximum width of the flutes and facilitate the flow of chips.
The maximum angle will be applied for short through holes.
The number of cutting edges, together with the half angle of the chamfer β, determines the increment "**i**" which corresponds to the chip thickness cut by each tooth.

$$i = \frac{P}{n} \cdot \tan \beta$$

In the great majority the taps are manufactured with 3 or 4 cutting edges. The type with two flutes are limited to the "high twist" type and to the taps for the machining light alloys.

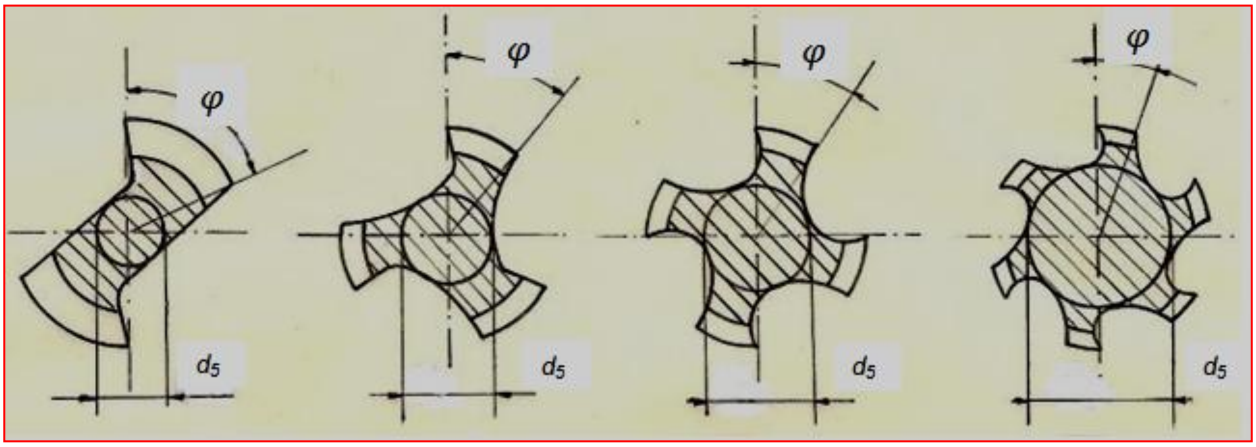

Fig. N°1- *Scheme of taps with different number of flutes*

Relief angle of the thread

The angle Δ_1 is called *pitch diameter relief*. The radial reduction of the pitch and/or major diameter behind the cutting edge of the tap. The relief confer cutting properties and provides clearance between the part being threaded and the tap threads.
If this clearance is referred to the whole profile, that also involves the medium diameter, it generates a lateral relief angle on the side of each tooth.
In subsequent resharpening, as a result of this relief angle, all diameters, (outside, middle and inside), are reduced. In general, however, this decrease, within the range of possible resharpening, remains in the tolerance range of the thread.

Negative taper

It is said *negative taper* the taper which decreases the middle diameter from the chamfer to the shank. It serves to reduce friction on the flanks of the thread during the work.
In taps ground after hardening the negative taper varies from zero to small values.
But the existence of this taper becomes necessary determine where the control of various diameters must be done.
It was agreed that the checks are done on 2 or 3 thread after the last trace of the chamfer.
The negative taper varies from 0.5% for to the taps MA 4x0,7 to 0.92% for taps MA 24x3.

Fig. N°2- *Examples of taps (Vergnano)*

Rake angle

The rake angle γ is the angle between the cutting face of the tap and a radial line passing though the crest of the tooth at the cutting edge.
From the appropriate choice of the rake angle γ depends the life of the tap and the good quality of the thread executed.

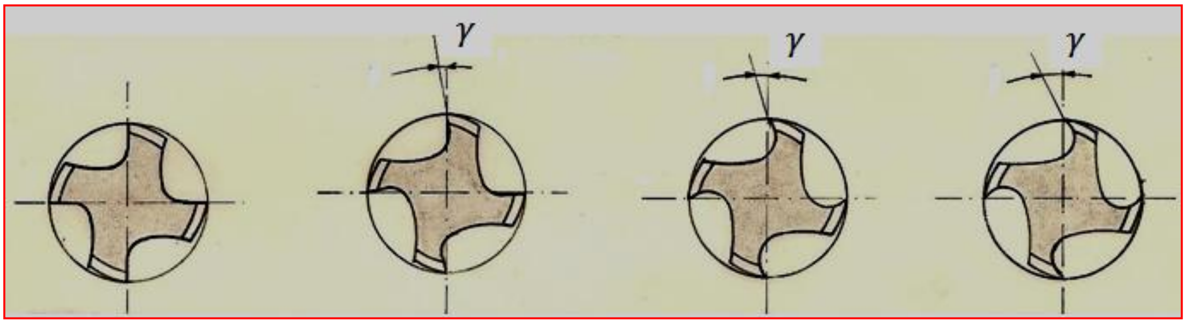

Fig. N°3- *Rake angles*

The recommended rake angles are shown in table N°2.

Tab.N°2

Material of the workpiece	Rake angle	Material of the workpiece	Rake angle
Steel with R<400 N/mm^2	15°÷20°	Brass	0°÷5°
Steel with R<750 N/mm^2	12°÷18°	Copper	15°÷25°
Steel with R>750 N/mm^2	10°÷12°	Nickel	8°÷12°
Alloy steel	6°÷10°	Zinc	15°÷20°
Steel for tools	6°÷10°	Aluminum	25°÷35°
Stainless steel	8°÷10°	Aluminum alloy	18°÷25°
Soft cast Iron	8°÷10°	Plastic material	15°÷25°
Malleable cast iron	5°÷8°	Resinoid plastic material	4°÷7°
Hard cast iron	3°÷7°	Bakelite	3°
Hard bronze	5°÷10°	Fiber	3°
Soft bronze	0°÷3°		

In order to achieve the perfect execution of the rake angle, especially for high values, you can perform a second little groove along the flute and along the main cutting edge as shown in figure N°4.

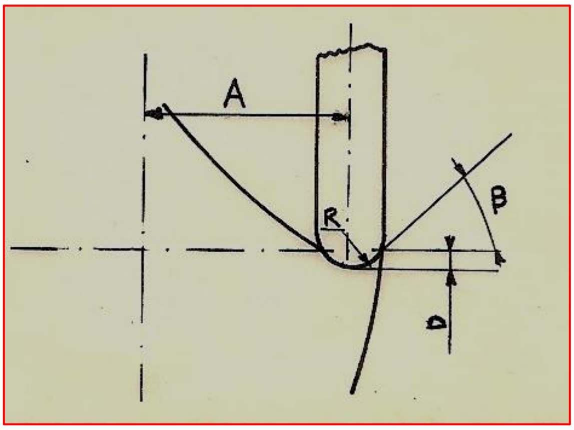

Fig. N°4- *Making the special sharpening*

$$R = \frac{d_1}{2 \cdot n} \quad ; \quad A = \frac{d_1}{2} - R \cdot \sin\gamma \quad ; \quad D = R \cdot (1 - \cos\gamma)$$

The advantages of this method of sharpening can be summarized as:
Chip rolled up so close that allows a more abundant coolant: This helps to cool the cutting edge better, and thus increasing the cutting speed:
The double radius of the cutting face facilitates the flow of the chips in flutes reducing the possibility of clogging and thus increase the efficiency of the tap.

Fig.N°5- *Example of a tap with modified cutting face*