

MACHINING PROCESSES

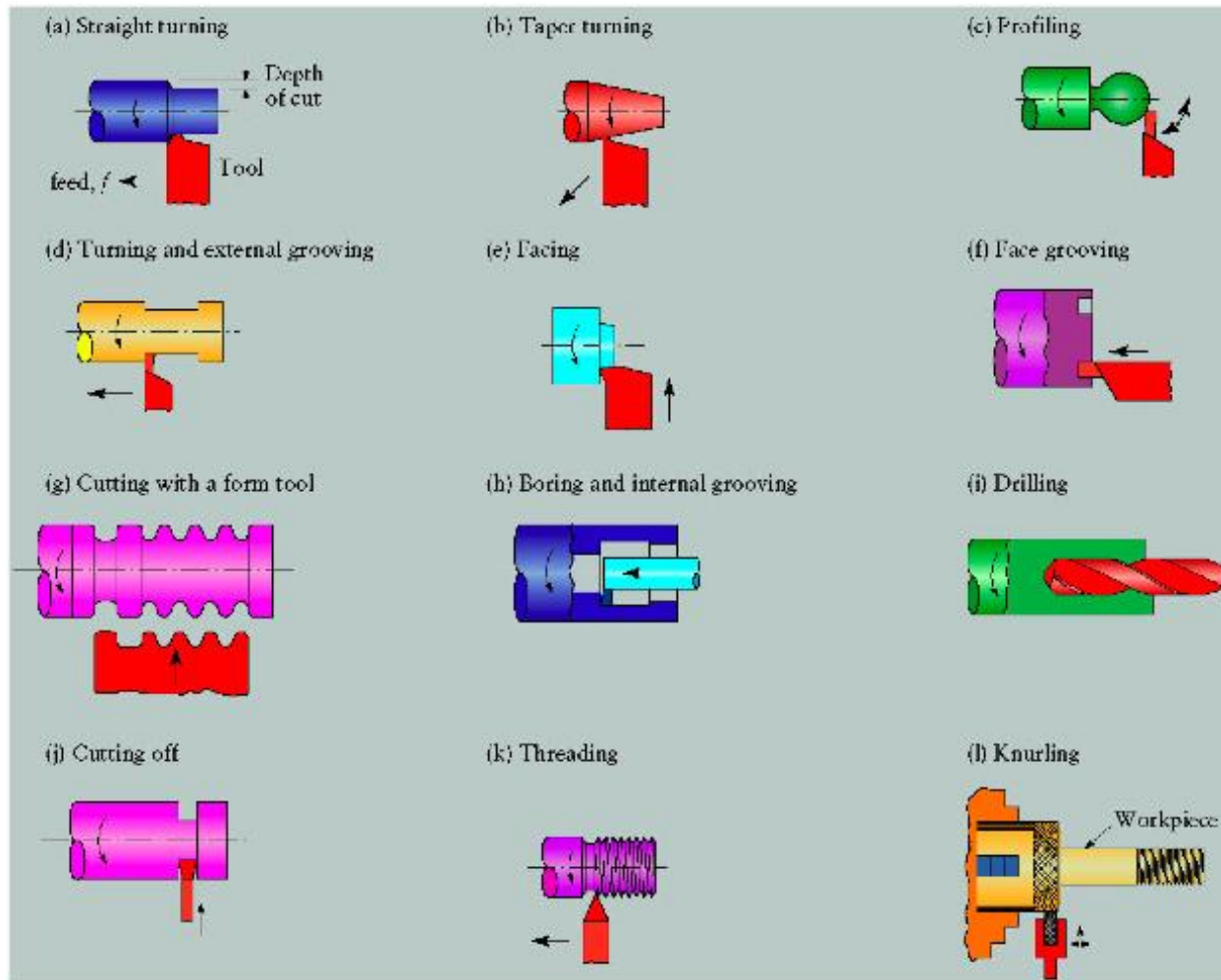
Chapter 8

Machining Processes

Process	Characteristics	Commercial tolerances (±mm)
Turning	Turning and facing operations on all types of materials; requires skilled labor; low production rate, but medium to high with turret lathes and automatic machines, requiring less-skilled labor.	Fine: 0.05-0.13 Rough: 0.13 Skiving: 0.025-0.05
Boring	Internal surfaces or profiles, with characteristics similar to turning; stiffness of boring bar important to avoid chatter.	0.025
Drilling	Round holes of various sizes and depths; requires boring and reaming for improved accuracy; high production rate; labor skill required depends on hole location and accuracy specified.	0.075
Milling	Variety of shapes involving contours, flat surfaces, and slots; wide variety of tooling; versatile; low to medium production rate; requires skilled labor.	0.13-0.25
Planing	Flat surfaces and straight contour profiles on large surfaces; suitable for low-quantity production; labor skill required depends on part shape.	0.08-0.13
Shaping	Flat surfaces and straight contour profiles on relatively small workpieces; suitable for low-quantity production; labor skill required depends on part shape.	0.05-0.13
Broaching	External and internal flat surfaces, slots and contours with good surface finish; costly tooling; high production rate; labor skill required depends on part shape.	0.025-0.15
Sawing	Straight and contour cuts on flats or structural shapes; not suitable for hard materials unless saw has carbide teeth or is coated with diamond; low production rate; requires only low labor skill.	0.8

TABLE 8.7 General characteristics of machining processes.

Turning operations



Roughing

Finishing

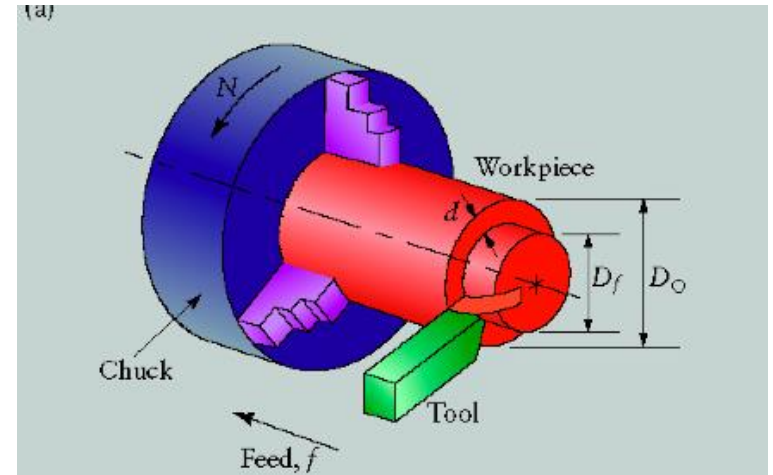
Turning parameters

CUTTING SPEED v_t : relative speed between the tool and the workpiece [m/min]

FEED RATE v_a : speed at which the tool travels along the feed direction [mm/min]

FEED f : distance travelled by the tool on the feed direction for each turn of the workpiece [mm/giro]

DEPTH OF CUT d : amount of metal to be removed, measured perpendicularly to the feed direction [mm]



N [round/min] D [mm]

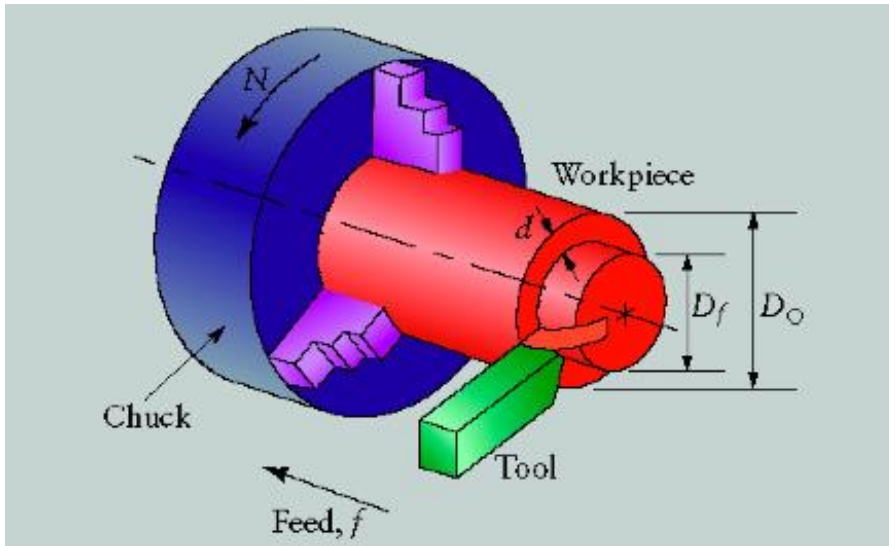
$$v_t = \frac{\pi D N}{10^3}$$

$$d = \frac{D_0 - D_f}{2}$$

$$v_a = N f$$

Material removal rate MRR

Volume of material machined in the time unit [mm³/min]



Volume machined for each workpiece turn

$$\pi D_{avg} f d$$

$$D_{avg} = \frac{D_0 + D_f}{2}$$

$$MRR = \pi D_{avg} f d N$$

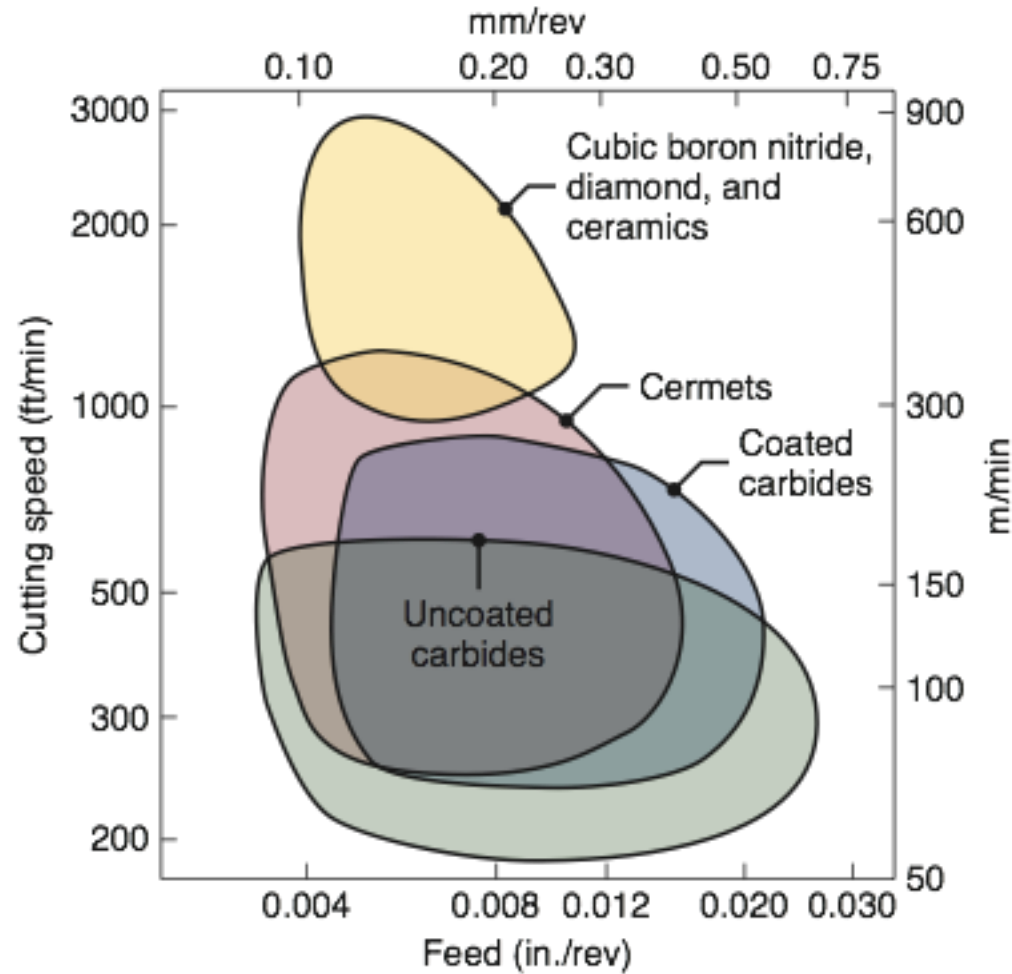
Turning time

$$t = \frac{l}{fN}$$

l distance travelled by the tool

NOTE: the formula does not include the time needed for the tool engage and disengage

Cutting speed in turning



Cutting speed in turning

Workpiece Material	Cutting Speed	
	m/min	ft/min
Aluminum alloys	200-1000	650-3300
Cast iron, gray	60-900	200-3000
Copper alloys	50-700	160-2300
High-temperature alloys	20-400	65-1300
Steels	50-500	160-1600
Stainless steels	50-300	160-1000
Thermoplastics and thermosets	90-240	300-800
Titanium alloys	10-100	30-330
Tungsten alloys	60-150	200-500

Note: (a) The speeds given in this table are for carbides and ceramic cutting tools. Speeds for high-speed-steel tools are lower than indicated. The higher ranges are for coated carbides and cer-mets. Speeds for diamond tools are significantly higher than any of the values indicated in the table.

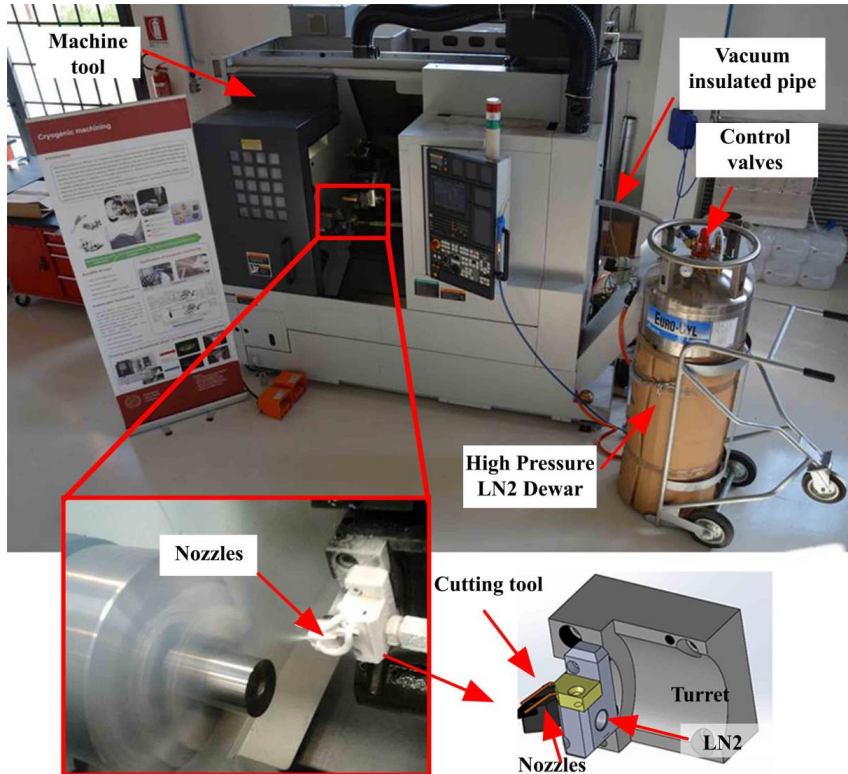
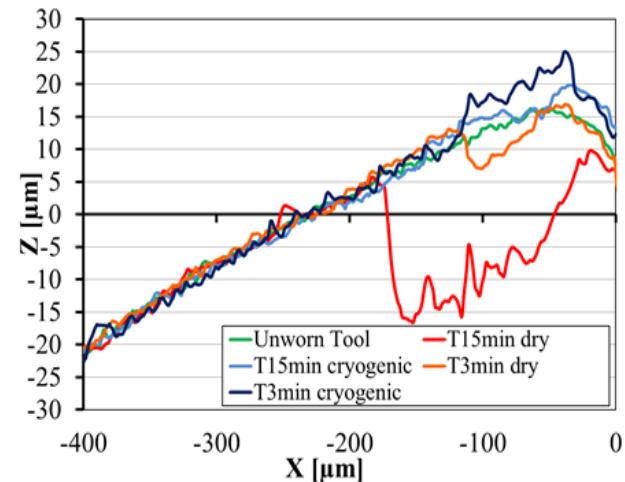
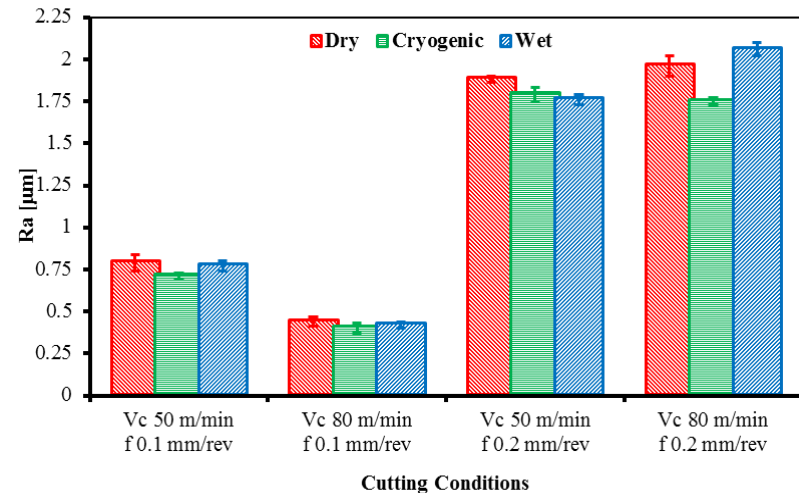
(b) Depths of cut, d , are generally in the range of 0.5-12 mm (0.02-0.5 in.).

(c) Feeds, f , are generally in the range of 0.15-1 mm/rev (0.006-0.040 in./rev).

Cryogenic turning of biomedical components

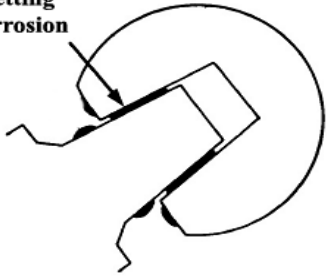
Benefits:

- Increase of the tool life
- Increase of the machined *surface integrity*

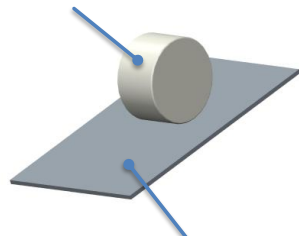


Cryogenic turning of biomedical components

Fretting corrosion



EBM Ti6Al4V pin

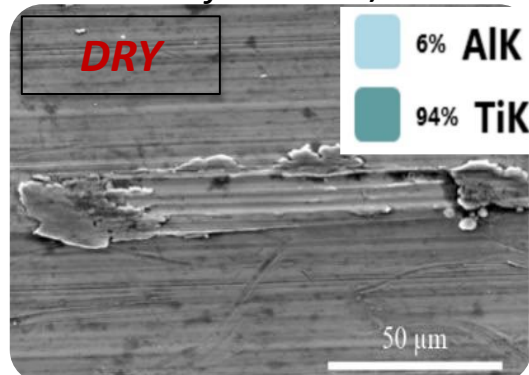
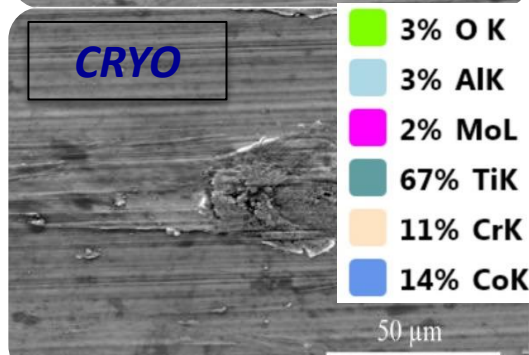


Wrought CoCrMo flat plate

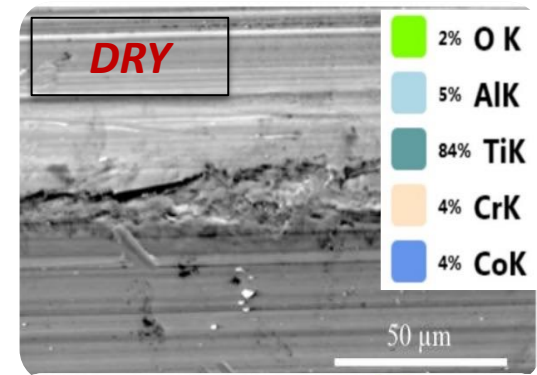
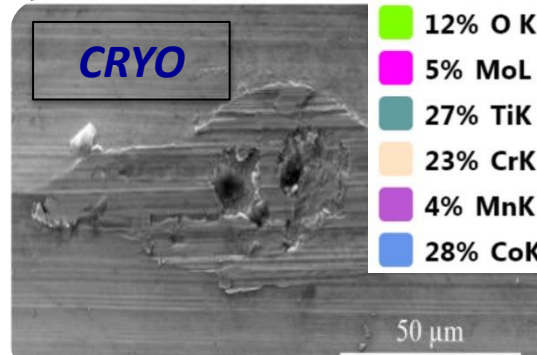
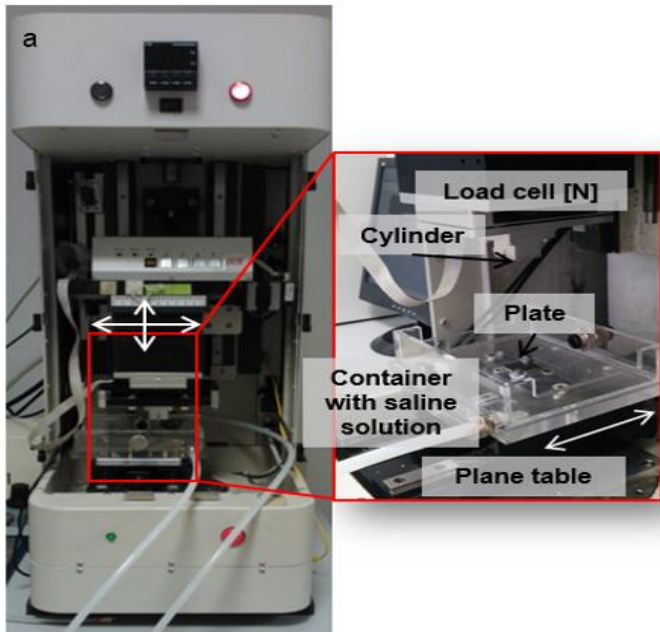
Benefits:

→ Increase of the tribo-corrosion resistance

$V_c = 50 \text{ m/min}$
 $f = 0.1 \text{ mm/rev}$

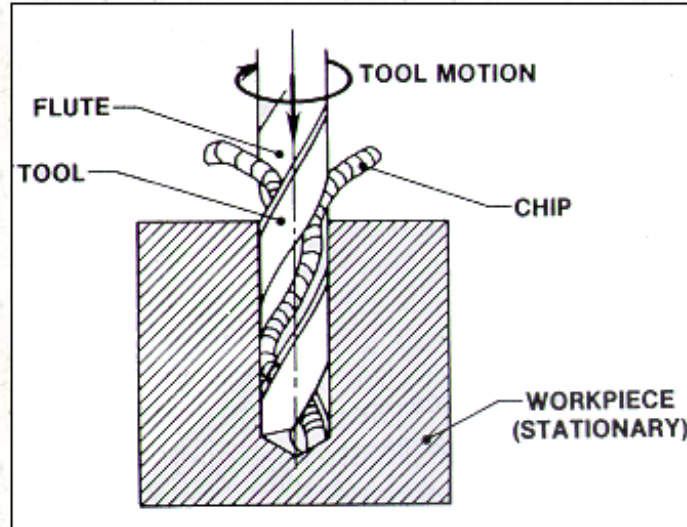


$V_c = 50 \text{ m/min}$
 $f = 0.2 \text{ mm/rev}$



- Worn cryogenic machined pins increased in weight due to the adhered flat plate material due to their higher initial hardness

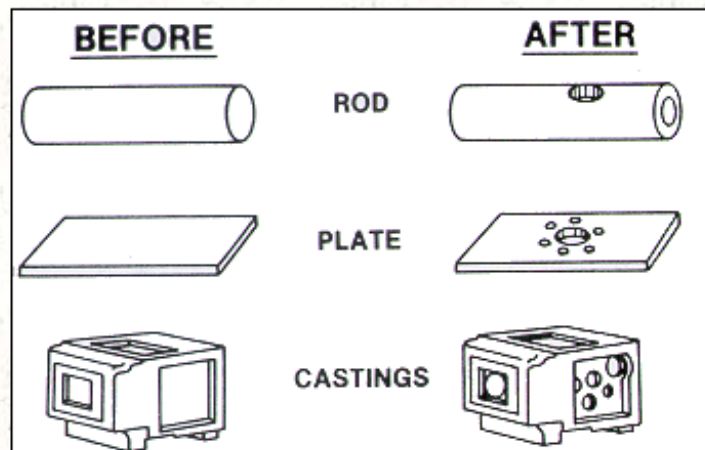
Drilling



Drilling is a cutting process in which a hole is originated or enlarged by means of a multipoint, fluted, end cutting tool. As the drill is rotated and advanced into the workpiece, material is removed in the form of chips that move along the fluted shank of the drill. One study showed that drilling accounts for nearly 90% of all chips produced.

Process Characteristics

- Uses a multipoint, fluted, end cutting tool (drill)
- Cutting tools or workpieces are rotated and advanced relative to each other
- Creates or enlarges nonprecision holes
- May produce coarse, helical feed marks, depending on machining parameters (feed, speed, tool geometry, coolant, etc.)
- Creates small burrs on entry and coarse burrs on exit.



- Utensile generalmente dotato sia del moto di taglio (rotatorio intorno all' asse dell' utensile) che del moto di avanzamento (rettilineo in direzione dell' asse medesimo)
- Utensile: generalmente una punta elicoidale

Drilling parameters

CUTTING SPEED v_t : the tool speed at its external diameter D [m/min]

$$v_t = \frac{\rho D N}{10^3}$$

where D [mm], N [rev/min]

FEED f : the distance travelled by the tool for each turn [mm/rev]

MATERIAL REMOVAL RATE
[mm³/min]

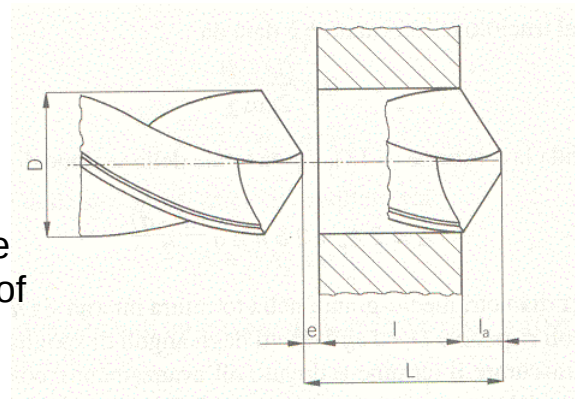
CUTTING TIME [min]

$$t = \frac{L}{v_a} = \frac{L}{nf}$$

$$L = e + l + l_a$$

where e is the initial extra-distance travelled by the tool to engage the piece (2-3 mm), l is the length of the hole to be drilled, and l_a is the length of the conical final part of the drill

$$MRR = \frac{\pi D^2}{4} f N$$

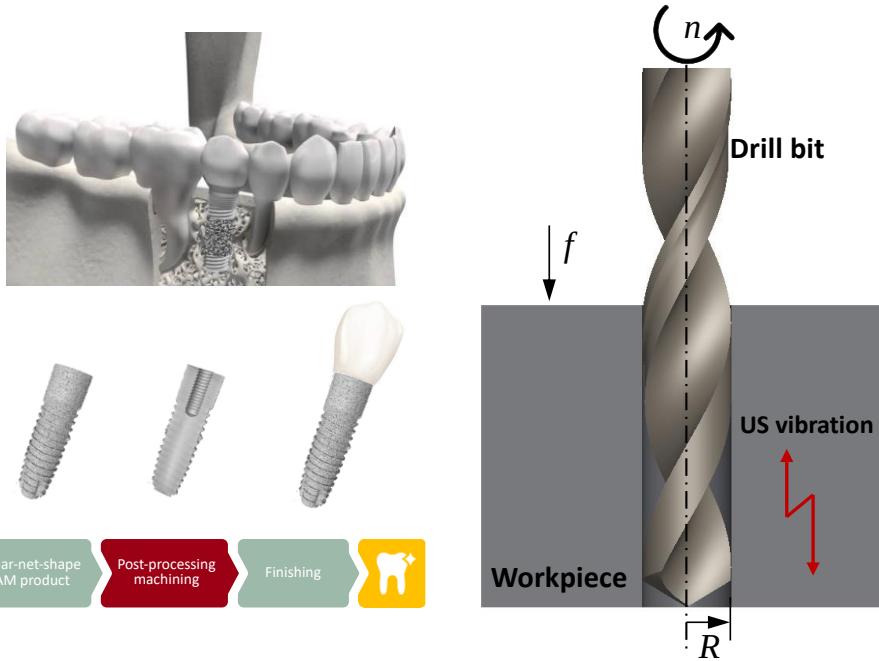


Cutting speed and feed in drilling

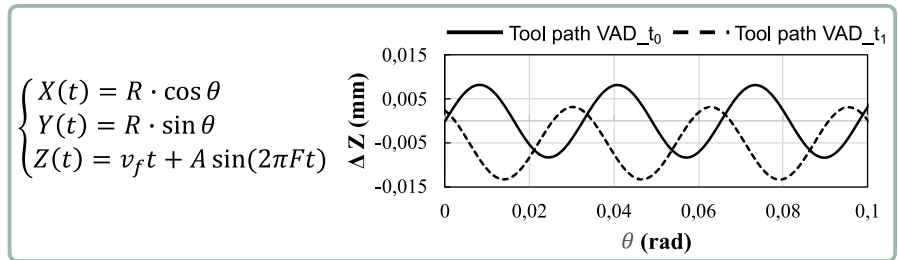
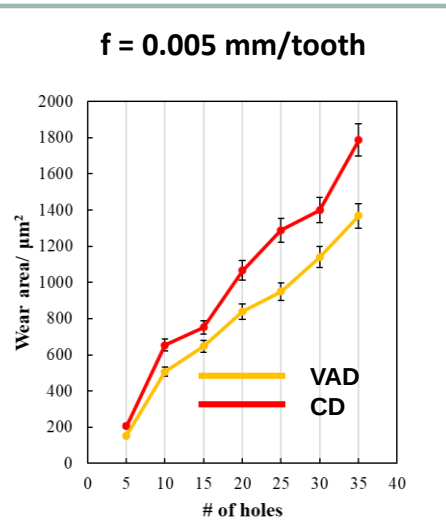
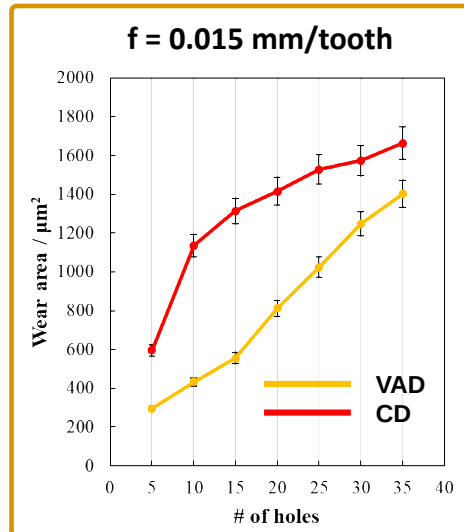
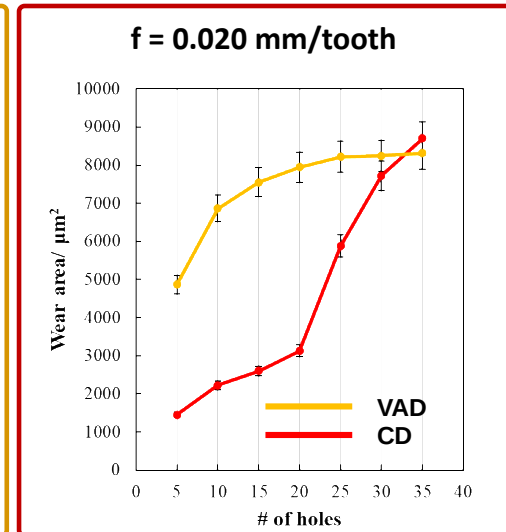
Workpiece Material	Surface Speed		Feed, mm/rev (in./rev) Drill Diameter		Spindle speed (rpm) Drill Diameter	
	m/min	ft/min	1.5 mm (0.060 in.)	12.5 mm (0.5 in.)	1.5 mm (0.060 in.)	12.5 mm (0.5 in.)
Aluminum alloys	30-120	100-400	0.025 (0.001)	0.30 (0.012)	6400-25,000	800-3000
Magnesium alloys	45-120	150-400	0.025 (0.001)	0.30 (0.012)	9600-25,000	1100-3000
Copper alloys	15-60	50-200	0.025 (0.001)	0.25 (0.010)	3200-12,000	400-1500
Steels	20-30	60-100	0.025 (0.001)	0.30 (0.012)	4300-6400	500-800
Stainless steels	10-20	40-60	0.025 (0.001)	0.18 (0.007)	2100-4300	250-500
Titanium alloys	6-20	20-60	0.010 (0.0004)	0.15 (0.006)	1300-4300	150-500
Cast irons	20-60	60-200	0.025 (0.001)	0.30 (0.012)	4300-12,000	500-1500
Thermoplastics	30-60	100-200	0.025 (0.001)	0.13 (0.005)	6400-12,000	800-1500
Thermosets	20-60	60-200	0.025 (0.001)	0.10 (0.004)	4300-12,000	500-1500

Note: As hole depth increases, speeds and feeds should be reduced. Selection of speeds and feeds also depends on the specific surface finish required.

Drilling of dental implants

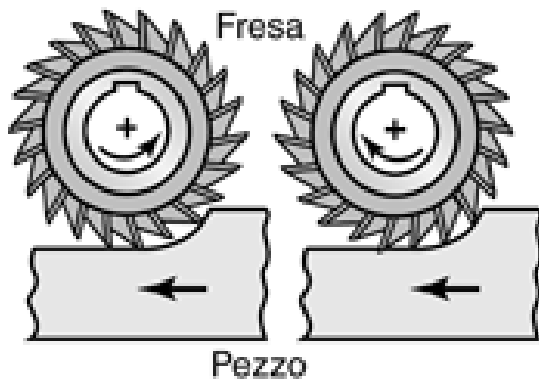


Vibration Assisted Drilling VAD refers to the superimposition of ultrasonic vibration



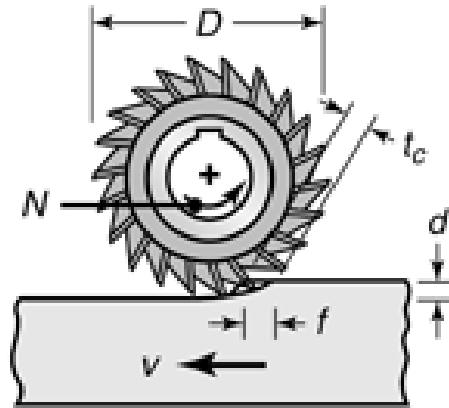
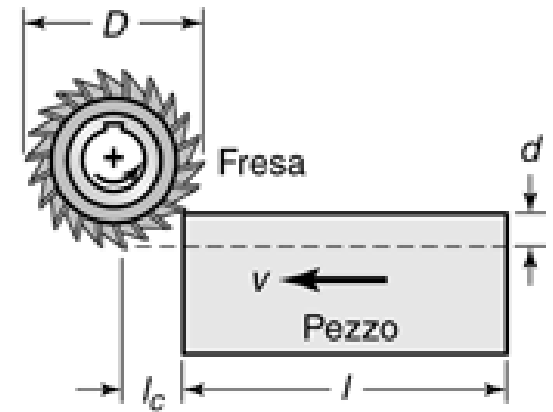
Slab/peripheral milling

The axis of the tool is parallel to the surface to be milled



Climb milling

**Conventional
milling**



Peripheral milling parameters

Cutting speed = cutter tangential speed [m/min] (evaluated at the cutter diameter D)

$$V = \frac{\pi DN}{10^3}$$

N= cutter rotational speed [round/min]
D= cutter diameter [mm]

If the feed is straight, **f [mm/rev]** is the distance travelled by the workpiece for each turn of the cutter

d [mm] is the depth of cut

Feed for each tooth of the cutter

$$f_z = \frac{V}{Nn}$$

v= linear speed of the workpiece
n= number of teeth of the cutter

Peripheral milling parameters

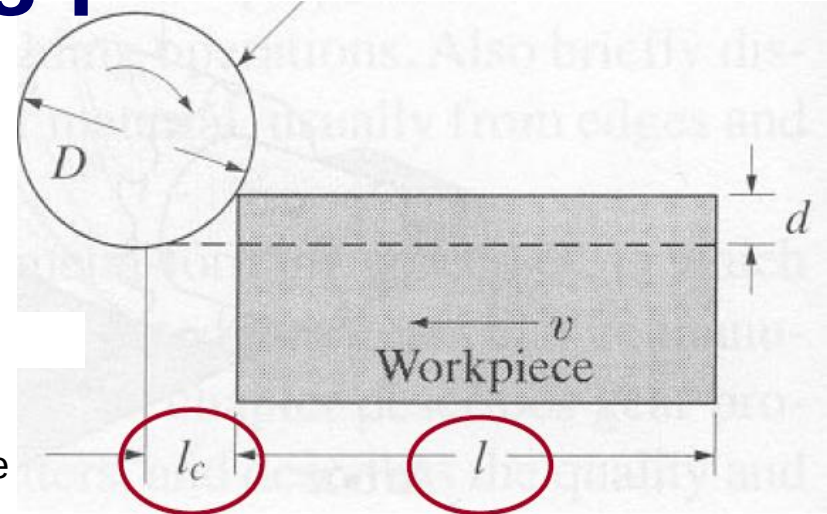
Cutting time

$$t = \frac{(l + l_c)}{v}$$

l = length of the workpiece

l_c = extent of the cutter's first contact with the workpiece

v = linear speed of the workpiece



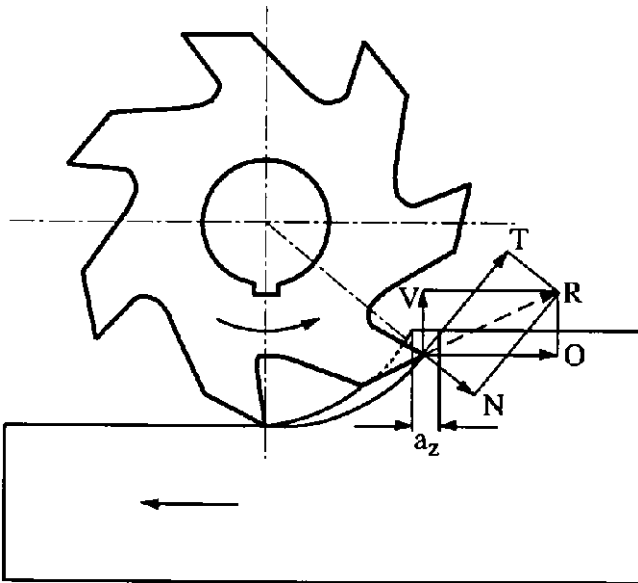
Material removal rate MRR [mm^3/min]

$$MRR = \frac{lwd}{t} = wdv$$

w = width of the cut

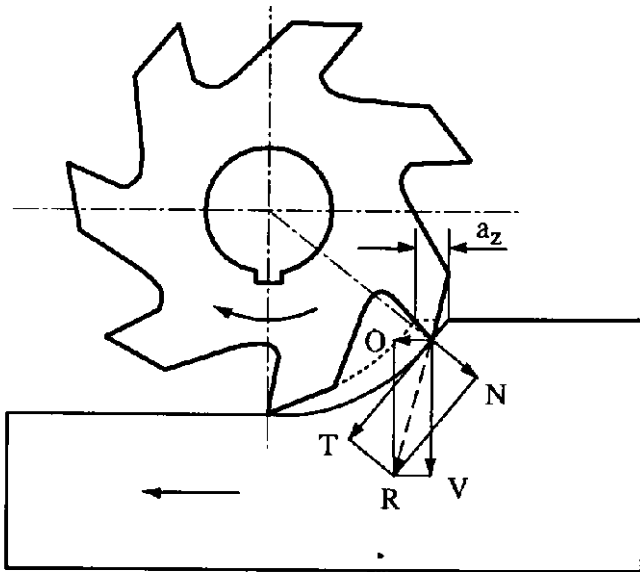
d = depth of cut

Conventional milling



- the shape of the chip is like a comma, with a thickness equal to zero when the tooth comes in contact with the workpiece to the maximum thickness when it leaves the contact
- every tooth rubs on the previously machined surface before starting cutting and this leads to material hardening and cutter wear
- the cutter works on virgin material: possible surface oxidation does not influence the cutter; therefore, this kind of operation is preferable for *finishing operations*
- the force R resultant of the cutting force T and normal force N has a vertical component V (which can cause the workpiece to lift from its fixtures, which makes the fixturing very important) and a horizontal component O (which acts opposite to the feeding direction)
- the cutting force T acting on each tooth varies during the operation: it is equal to zero when the tooth comes in contact with the workpiece and becomes maximum when the tooth leaves the contact; this causes high vibrations.

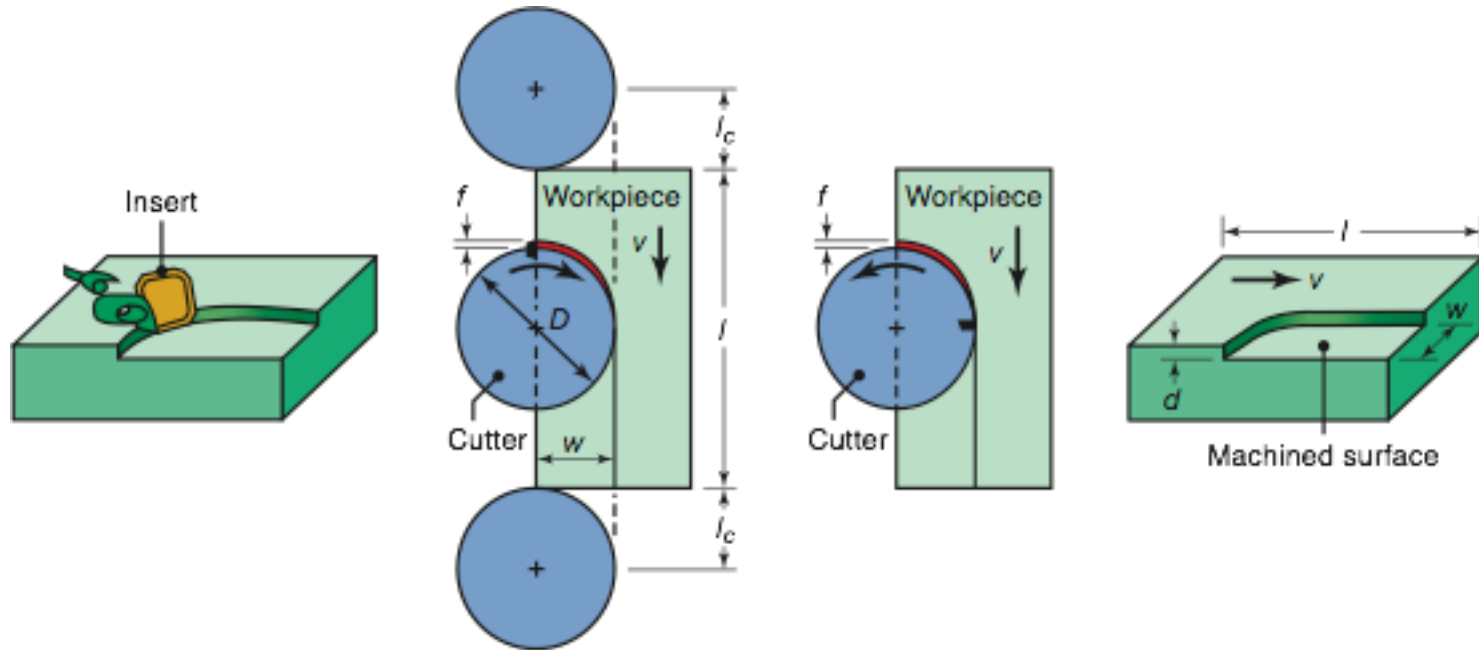
Climb milling



- the tooth begins to cut with an impact when the thickness of the chip is maximum: hard surfaces can cause the tool chipping
- rubbing phenomena are absent, we have less wear of the tools, less vibrations
- the vertical component V of the force resultant R tends to compress the workpiece on its (more easy fixturing) whereas the horizontal component O is along the feed direction
- very high forces arise at the tooth impact with the surface: a very rigid set-up is needed
- used especially for clean surfaces; not suitable for oxidized surfaces

Face milling

The axis of the tool is perpendicular to the surface to be milled



Better milled surface, almost constant thickness of the chip, higher number of engaged teeth, possibility to use inserts

Climb milling

Conventional milling

Cutting speed in milling

Workpiece Material	Cutting Speed	
	m/min	ft/min
Aluminum alloys	300-3000	1000-10,000
Cast iron, gray	90-1300	300-4200
Copper alloys	90-1000	300-3300
High-temperature alloys	30-550	100-1800
Steels	60-450	200-1500
Stainless steels	90-500	300-1600
Thermoplastics and thermosets	90-1400	300-4500
Titanium alloys	40-150	130-500

Note: (a) These speeds are for carbides, ceramic, cermets, and diamond cutting tools. Speeds for high-speed-steel tools are lower than those indicated in this table.

(b) Depths of cut, d , are generally in the range of 1-8 mm (0.04-0.3 in.).

(c) Feeds per tooth, f , are generally in the range of 0.08-0.46 mm/rev (0.003-0.018 in./rev).