

Lecture 8. Metal Cutting

Cutting processes work by causing fracture of the material that is processed. Usually, the portion that is fractured away is in small sized pieces, called chips. Common cutting processes include *sawing*, *shaping* (or planing), *broaching*, *drilling*, *grinding*, *turning* and *milling*. Although the actual machines, tools and processes for cutting look very different from each other, the basic mechanism for causing the fracture can be understood by just a simple model called for *orthogonal cutting*.

In all machining processes, the *workpiece* is a shape that can entirely cover the final part shape. The objective is to cut away the excess material and obtain the final part. This cutting usually requires to be completed in several steps – in each step, the part is held in a *fixture*, and the exposed portion can be accessed by the tool to machine in that portion. Common fixtures include *vise*, *clamps*, *3-jaw* or *4-jaw chucks*, etc. Each position of holding the part is called a *setup*. One or more cutting *operations* may be performed, using one or more cutting tools, in each setup. To switch from one setup to the next, we must release the part from the previous fixture, change the fixture on the machine, clamp the part in the new position on the new fixture, set the coordinates of the machine tool with respect to the new location of the part, and finally start the machining operations for this setup. Therefore, setup changes are time-consuming and expensive, and so we should try to do the entire cutting process in a minimum number of setups; the task of determining the sequence of the individual operations, grouping them into (a minimum number of) setups, and determination of the fixture used for each setup, is called *process planning*.

These notes will be organized in three sections:

(i) introduction to the processes, (ii) the orthogonal cutting model and tool life optimization and (iii) process planning and machining planning for milling.

8.1. Introduction to the processes

8.1.1. Sawing

Sawing is used to cut the correct sized workpiece from a large raw material stock. There are several types of saws (Figure 1):

Hacksaws: straight blade, moving in a reciprocating motion;

Bandsaws: straight blade, ends welded together to make a loop, moving continuously in one direction;

Circular saws: blade in the shape of a circular disk, rotating continuously.

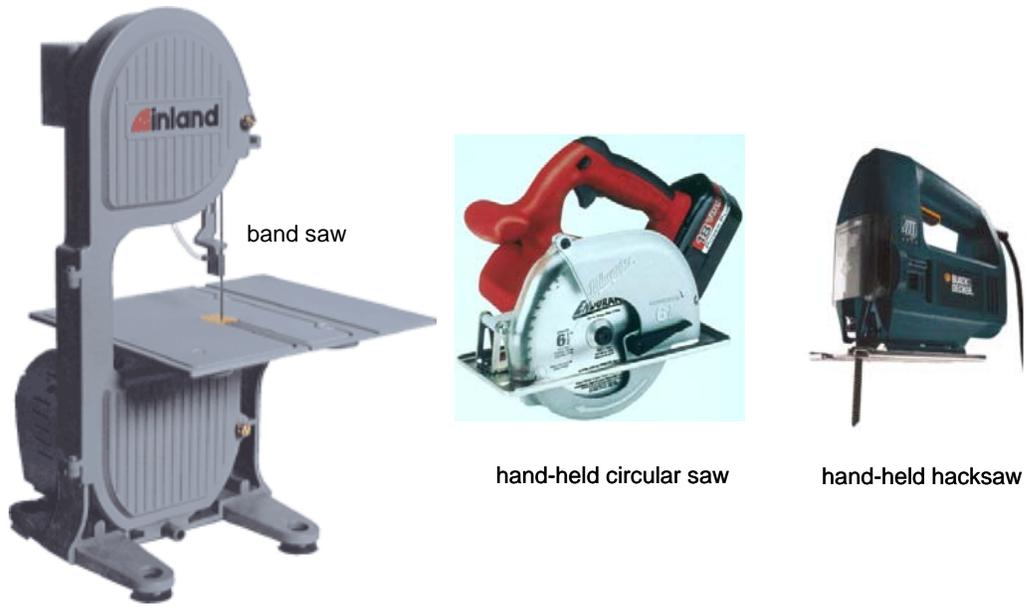


Figure 1. Types of saws

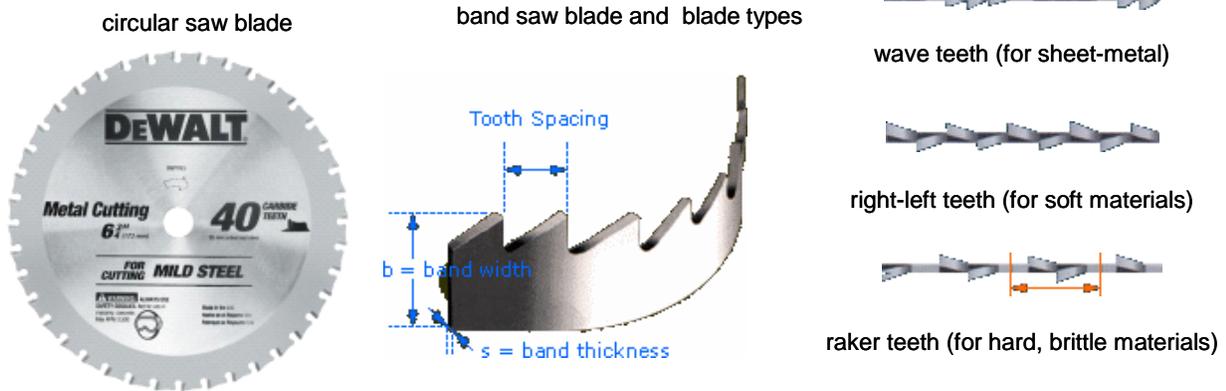


Figure 2. Types of saw blades

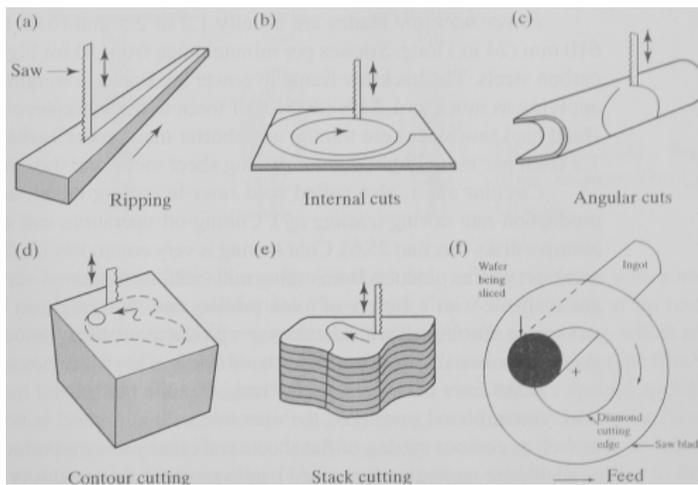


Figure 3. Typical sawing actions [source: Kalpakjian and Schmid]

8.1.2. Shaping

Shaping uses a single-point tool that is moved horizontally in a reciprocating motion along a slide. It is used to create a planar surface, usually to prepare rectangular blocks that can later be used as workpieces for machining on a milling machine etc. The machine is simple – a typical machine is shown in Figure 4, along with a short description of its operation.

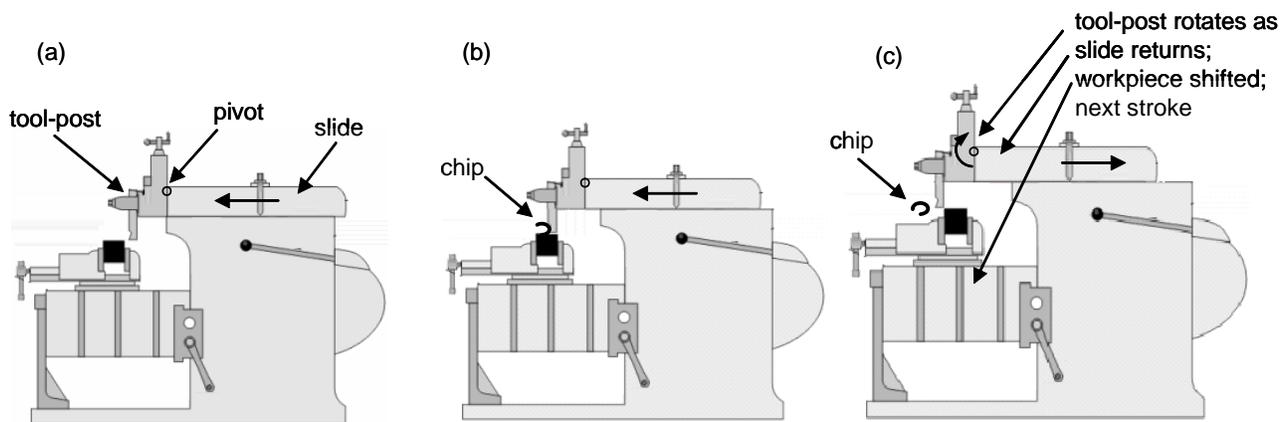


Figure 4. Shaping machine and shaping operation

8.1.3. Broaching

Broaching is capable of mass-production of complex geometry parts, especially when complicated hole-shapes are required to be machined. The broach tool has a series of cutting teeth along the axis of the tool. As the broaching tool is pulled with force along the part to be cut, each tooth cuts a tiny chip. Thus the first few sets of teeth to engage the part remove most of the material, which the last few provide a finishing cut with very small amount of material removal. The geometric shape of the last set of teeth is identical to the required geometry of the designed part.

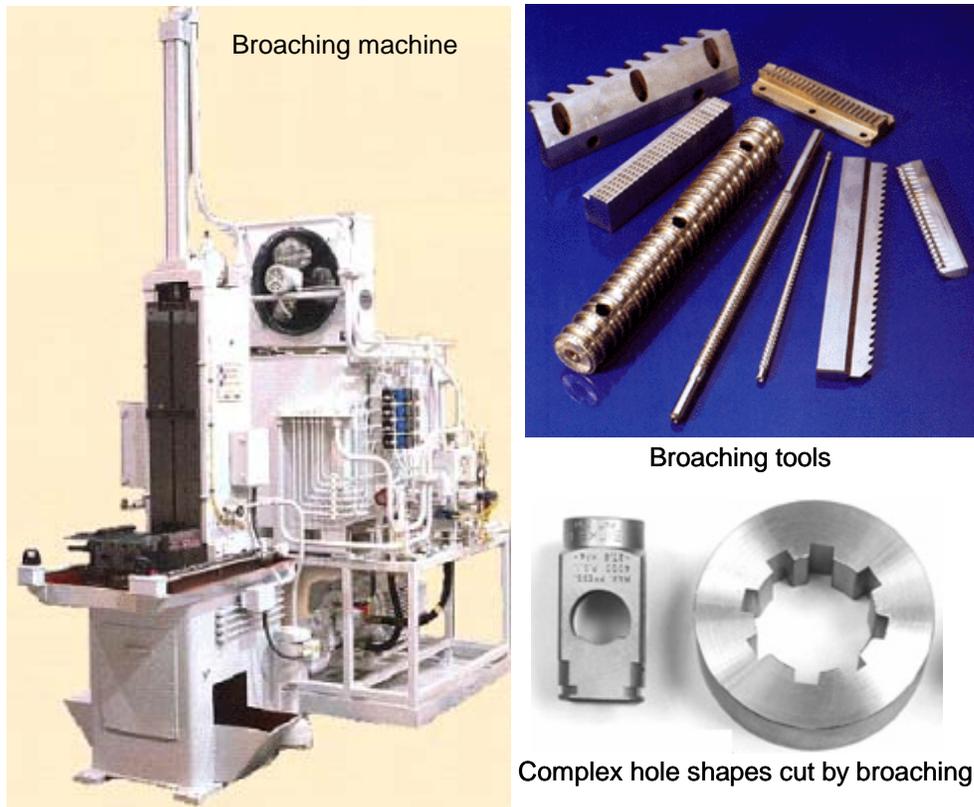


Figure 5 (a) Broaching machine, images of broaching tools

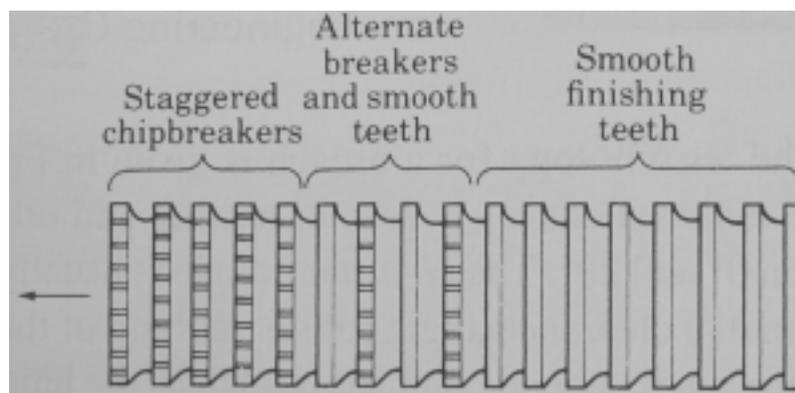


Figure 5 (b) Broaching cutter details [source: Kalpakjian and Schmid]

8.1.4. Drilling, Reaming, Boring, Tapping

These four methods all produce holes of different types. Drilling produces round holes of different types; reaming is used to improve the dimensional tolerance on a drilled hole; boring uses a special machine operating like a lathe, to cut high precision holes; and tapping creates screw-threads in drilled holes.

Drilling: The geometry of the common twist drill tool (called *drill bit*) is complex; it has straight cutting teeth at the bottom – these teeth do most of the metal cutting, and it has curved cutting teeth along its cylindrical surface (Figure 6). The grooves created by the helical teeth are called flutes, and are useful in pushing the chips out from the hole as it is being machined. Clearly, the velocity of the tip of the drill is zero, and so this region of the tool cannot do much cutting. Therefore it is common to machine a small hole in the material, called a center-hole, before utilizing the drill. Center-holes are made by special drills called center-drills; they also provide a good way for the drill bit to get aligned with the location of the center of the hole. There are hundreds of different types of drill shapes and sizes; here, we will only restrict ourselves to some general facts about drills.

- Common ***drill bit materials*** include hardened steel (High Speed Steel, Titanium Nitride coated steel); for cutting harder materials, drills with hard inserts, e.g. carbide or CBN inserts, are used;

- In general, drills for cutting softer materials have smaller point angle, while those for cutting hard and brittle materials have larger point angle;

- If the Length/Diameter ratio of the hole to be machined is large, then we need a special guiding support for the drill, which itself has to be very long; such operations are called ***gun-drilling***. This process is used for holes with diameter of few mm or more, and L/D ratio up to 300. These are used for making barrels of guns;

- Drilling is not useful for very small diameter holes (e.g. < 0.5 mm), since the tool may break and get stuck in the workpiece;

- Usually, the size of the hole made by a drill is slightly larger than the measured diameter of the drill – this is mainly because of vibration of the tool spindle as it rotates, possible misalignment of the drill with the spindle axis, and some other factors;

- For tight dimension control on hole diameter, we first drill a hole that is slightly smaller than required size (e.g. 0.25 mm smaller), and then use a special type of drill called a ***reamer***. Reaming has very low material removal rate, low depth of cut, but gives good dimension accuracy;

- large and deep holes are made by spade drills;

- ***Coutersink/counterbore*** drills have multiple diameters – they make a chamfered/stepped hole, which is useful for inserting screws/bolts – the larger diameter part of the hole accommodates the screw/bolt head;

- Internal threads can be cut into holes that mate with screws/bolts. These are cut by using tapping tools.

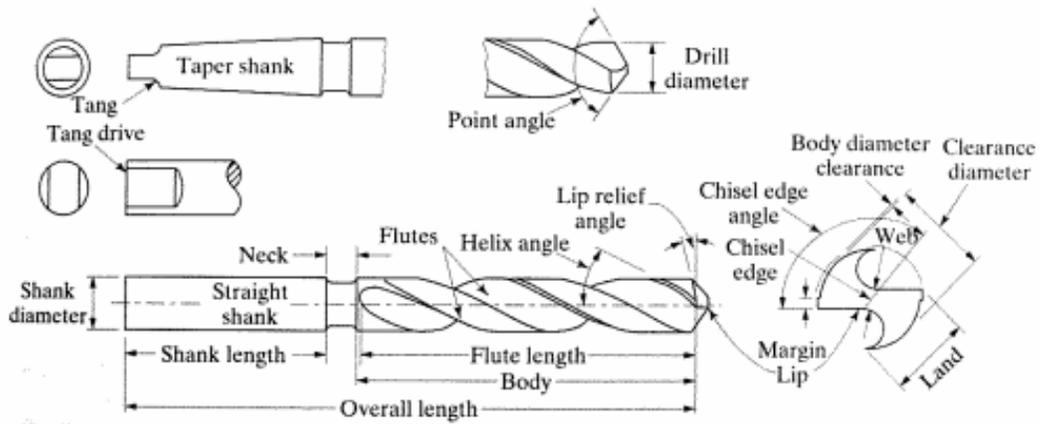


Figure 6. Geometry of a drill

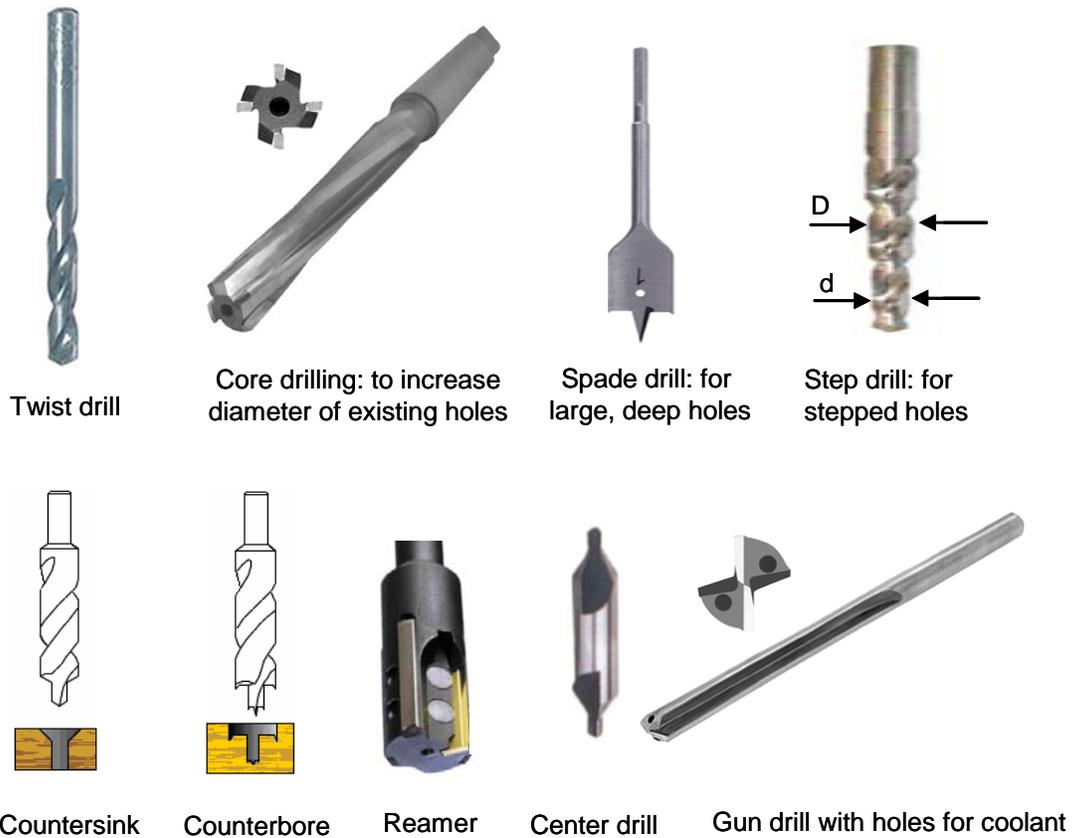


Figure 7. Different types of drilling operations



Figure 8. Thread-cutting. A drilled hole can be given internal threads by using a tapping tool; external threads on a cylindrical shape are made using a tapping die. The left image shows manual tool set for making external and internal threads. The right image shows a schematic for automatic tapping of threads in nuts.

8.1.5. Grinding and other Abrasive machining processes

Abrasive machining uses tools that are made of tiny, hard particles of crystalline materials – abrasive particles have irregular shape and sharp edges; the workpiece surface is machined by removing very tiny amounts of material at random points where a particle contacts it. By using a large number of particles, the effect is averaged over the entire surface, resulting in *very good surface finish* and *excellent dimension control*, even for hard, brittle workpieces. Grinding is also used to machine brittle materials (such materials cannot be machined easily by conventional cutting processes, since they would fracture and crack in random fashion). The main uses of grinding and abrasive machining:

1. *To improve the surface finish of a part manufactured by other processes*

Examples:

- (a) A steel injection molding die is machined by milling; the surface finish must be improved for better plastic flow, either by manual grinding using shaped grinding tools, or by electro-grinding.
- (b) The internal surface of the cylinders of a car engine are turned on a lathe. The surface is then made smooth by grinding, followed by honing and lapping to get an extremely good, mirror-like finish.
- (c) Sand-paper is used to smooth a rough cut piece of wood.

2. *To improve the dimensional tolerance of a part manufactured by other processes*

Examples:

- (a) ball-bearings are formed into initial round shape by a forging process; this is followed by a grinding process in a specially formed grinding die to get extremely good diameter control ($\leq 15\mu\text{m}$).

(b) Knives are made from forged steel; the steel is then hardened; finally, a grinding operation is used to give a sharp cutting edge.

3. To cut hard brittle materials

Example:

(a) Most semiconductor IC chips are made from silicon; the starting point is a long bar of a crystal of silicon (the diameter is usually 8cm, 15cm or 30cm, and length up to 200 cm). This rod must be sliced into thin circular slices; each slice is used to make a large number of ICs. A diamond abrasive wheel is used to cut the rod into slices.

4. To remove unwanted materials of a cutting process

Example

(a) Drilling and milling often leave tiny, sharp chips along the outer edges of the surface created by the tool – these are called burrs. Tapered grinding wheels are used to remove the burr (the process is called deburring).

Abrasive materials

Common abrasive materials are Aluminum Oxide and Silicon Carbide. For harder materials and high precision applications, superabrasives (Cubic Boron Nitride, or CBN, and diamond powder), which are extremely hard materials, are used.

Abrasive materials have two properties: high hardness, and high friability. Friability means that the abrasive particles are brittle, and fracture after some amount of use, creating new sharp edges that will again perform more abrasion.

Abrasive tools

Figure 9 shows several types of abrasive tools. They all contain abrasive grains that are glued together using resin or hardened rubber. Sometimes, the abrasive particles may be embedded in metal or ceramic. It is important for the bonding material to be softer than the abrasive. Also, the bonding material is selected to release the abrasive particles and wear away after some amount of use – this keeps exposing fresh abrasive particles to the workpiece continuously. The mean size of abrasive particle used in each tool determines the rate at which it will cut, and the quality of surface finish it will provide. Low material removal rate → better surface, which is achieved by using very fine grains. Grain size is expressed using numbers, small numbers like 10 mean large grains, and large numbers, e.g. 100 mean fine grain. You can see this in the grades of sand-paper.

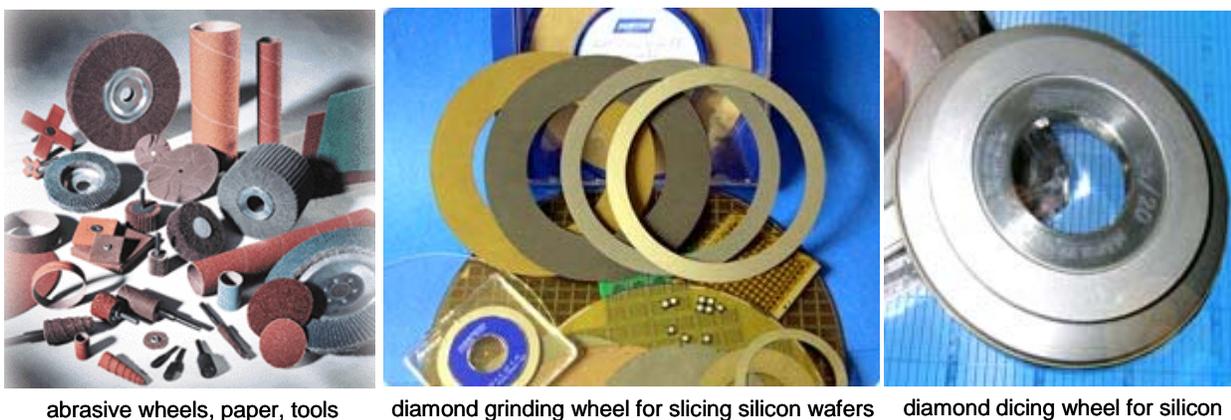


Figure 9. Different types of abrasive tools

Grinding machines

There are several types of grinding machines. The main ones are surface grinders, grinding wheels, cylindrical grinders and centerless grinders. The figure below shows examples of a few of these. Surface grinders produce flat surfaces. The part is held on the flat table (steel parts can be held by a magnetic force – this is called *magnetic chucking*). The table moves in a reciprocating motion ($\pm X$ -axis), and the rotating wheel is lowered (Z -axis) so that it just scrapes along the surface. After each reciprocating cycle, the part is fed by a small amount along the Y -axis.

To improve dimension control on cylindrical parts, centerless grinders, which use long cylindrical wheels, are employed. The axis of the regulating wheel and grinding wheel are slightly misaligned, causing the part to travel slowly in the axial direction, and after some time, the part automatically moves beyond the length of the wheel. Controlling the angle of misalignment can control the time that the part is subjected to grinding.

If a turned part of complex shape (e.g. stepped shafts) are to be ground, then cylindrical grinding is used, which employs specially made grinding wheels, whose profile fits the profile of the part to be ground.

Honing

Honing is a finishing operation used to improve the form tolerance of a cylindrical surface – in particular, it is used to improve the cylindricity. The honing tool is a metal bar holding a set of grinding stones arranged in a circular pattern. The tool brushes along the cylindrical part surface by rotating, and moving up-and-down along its axis. You can identify a honed surface by looking for the helical cross-hatched scratch marks on the part surface.

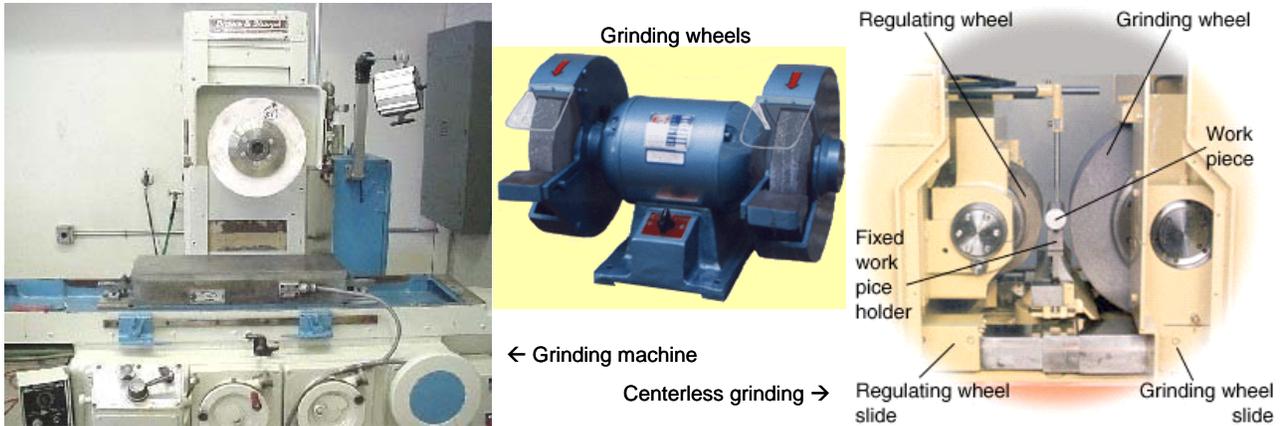


Figure 10. Some grinding machines

Lapping

Lapping is a finishing operation. The lapping tool is made of metal, leather, or cloth, impregnated with very fine abrasive particles. For preparing the surface of silicon wafers, lapping operations use a flat metal disc that rotates a small distance above the part. The gap is filled with a slurry containing fine abrasive grains. The rotation of the disc causes the slurry to flow relative to the part surface, resulting in very fine surface finish. This process gives dimensional tolerances of $\geq 0.5\mu\text{m}$, and surface finish of up to $0.1\mu\text{m}$.

8.1.6. Turning

Turning is a cutting operation in which the part is rotated as the tool is held against it on a machine called a *lathe*. The raw stock that is used on a lathe is usually cylindrical, and the parts that are machined on it are rotational parts – mathematically, each surface machined on a lathe is a *surface of revolution*. There are two common ways of using the lathe. If a hole needs to be drilled in the end face of the part, then a drill can be mounted in the tailstock (as shown in the figure below). The cylindrical part is held in the chuck, and the spindle rotates the part at high speed. The tailstock wheel is then used to feed the tool into the face of the part, to cut the hole. However, in most cases, the lathe is used by holding a *single-point cutting tool* in the tool-post. The tool post can move along the slide, by turning the carriage wheel; the tool can also be moved closer or farther from the rotation axis of the part – by turning the cross-slide wheel. The part is held in the chuck, and rotates at high speed; by controlling the relative position of the tool against the part (by using the cross-slide wheel and carriage wheel), we can control the material removal and the shape produced. Note that the turning of the cross-slide wheel controls the depth of cut, and the rate of turning the carriage wheel controls the feed rate (figure 11). Turning can produce a variety of revolved shapes. The typical cutting operations on a lathe are shown in Figure 12.

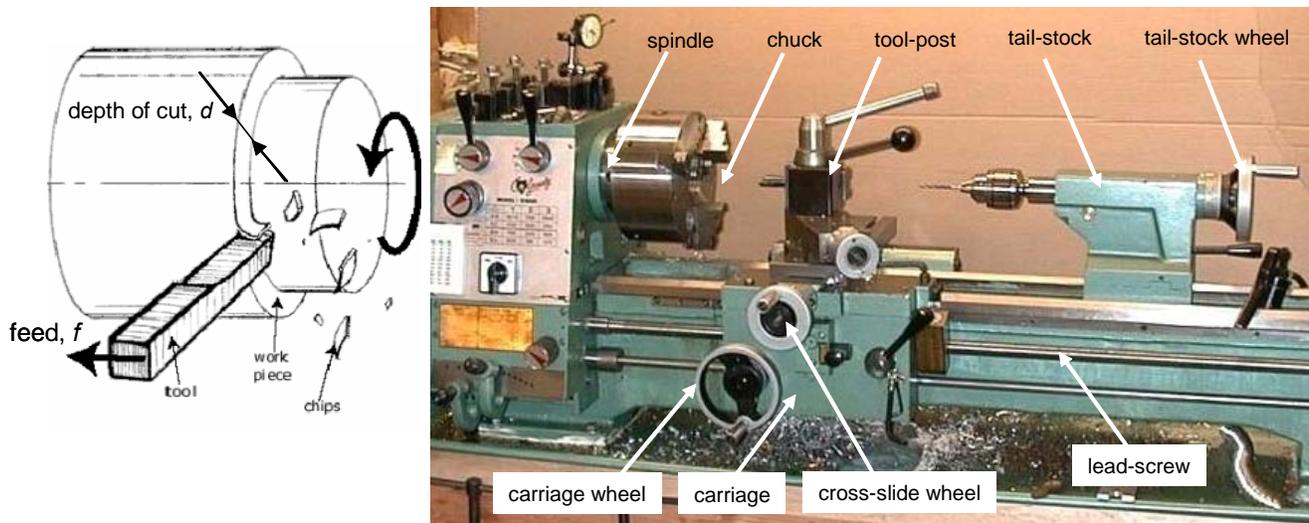


Figure 11. (a) Turning (b) A manual lathe with its important parts labeled

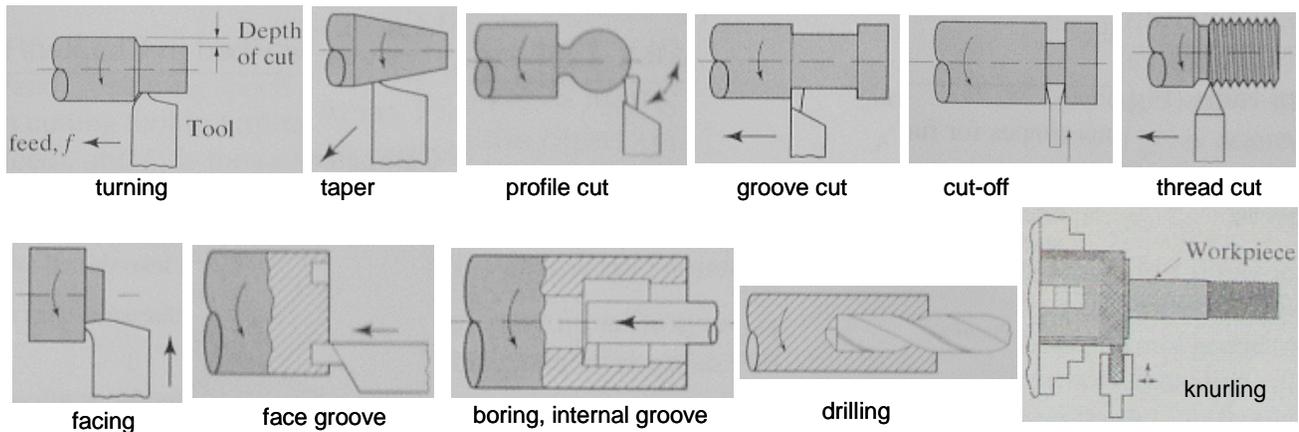


Figure 12. Typical lathe operations [source: Kalpakjian and Schmid]

The tool can perform cutting on the outer cylindrical surface [turning, taper cutting, groove cutting, cut off, thread cutting, knurling], the planar end face [facing, face groove cutting, drilling], or along inner cylindrical surfaces that can be accessed through the free planar face [boring, internal grooves]. Among these, only drilling requires the tool to be fed by moving the tailstock along the slide. In all other processes, the bar stock is held in a fixture at the spindle, with the opposite planar face free. However, if the stock is long, the tailstock may be used to provide extra support to the free end of the bar; this is done by a special supporting fixture called a dead-center. On the spindle side, the most common methods of holding the stock are: (a) collets, (b) 3-jaw chucks, or (c) 4-jaw chucks (see figure 13 below). Note that if several different machining operations must be performed on a single workpiece, then it is important to plan the sequence in which they will be done – this will affect how many times the part needs to be released from the fixture and re-located in a different

position to allow the tool to access the required geometry (for example if both planar ends must be faced, then we need at least two setups using a chuck).

Note that 3-jaw chucks can be used in three positions: Figure 13 shows the first; if the stock has an internal cylindrical surface, or is a tube, then the jaws of the chuck can be used to grip the part from inside by exerting an outward force; this allows the entire outer surface to be accessible to the tool. Finally, by reversing the jaws, larger sized bars can be held by using different levels of the steps. All three jaws move in and out simultaneously – so the bar axis is aligned with the axis of the chuck, and so, with the axis of the spindle.

4-Jaw chucks can be used to cut rotational shapes whose axis is offset (but parallel to) the axis of the part. This is because the opposite pairs of jaws can move independently.

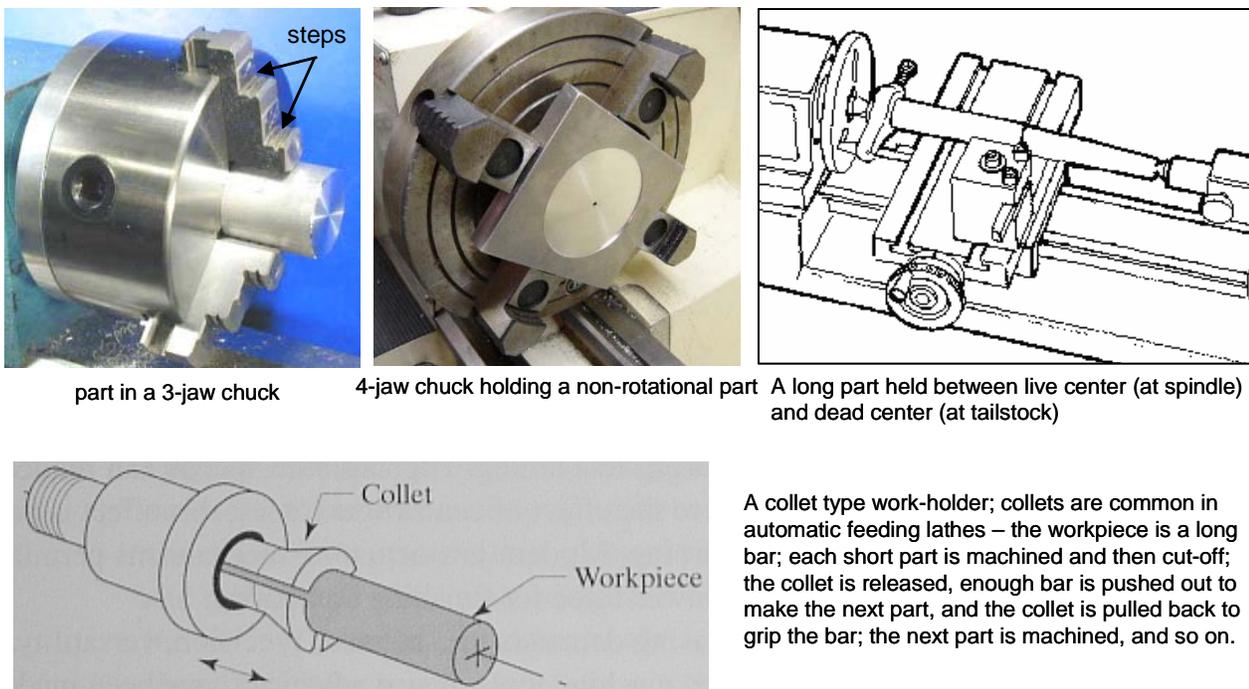


Figure 13. Different methods of work-holding for lathes

8.1.7. Milling

Milling is one of the most versatile machining processes, and can be used to produce a very large variety of shapes. In fact, you may have noticed that many manufacturing processes use some form of mold or die. A large percentage of these molds and dies are produced by milling. The most common milling operations include: Slab milling, Face milling, and End milling; these are distinguished easily by the different cutting tools they utilize.

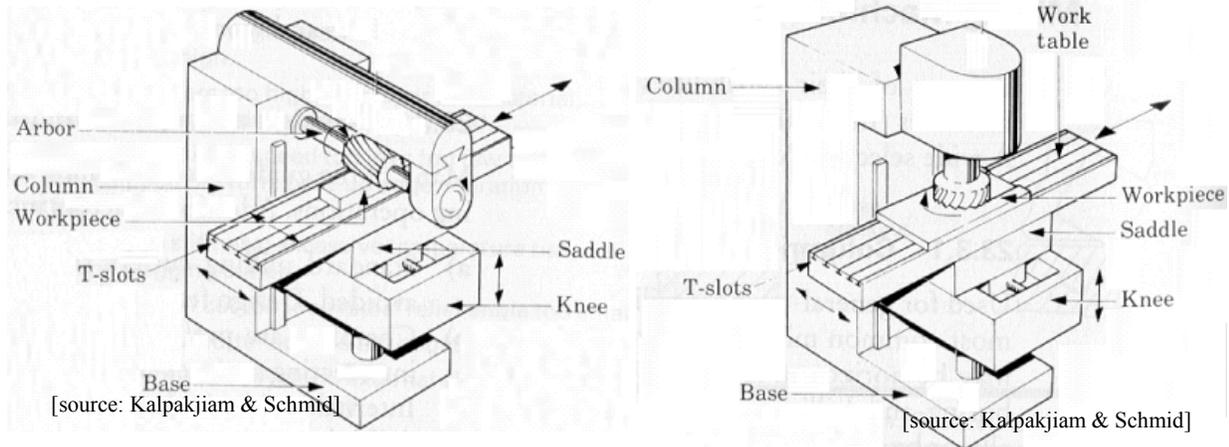


Figure 14. (a) Slab-milling on a horizontal mill (b) Face milling on a vertical mill



Figure 15. End milling (a) A Flat end mill being used to cut a curved surface (b) Ball end mill used for finish-cuts on a mold (c) Formed end-mill: here, a T-slot cutter is shown cutting a T-slot

In most practical cases, the milling cutter has several teeth (from 2-flutes to perhaps 20). Each tooth forms a helix going around the cylindrical tool body. The orientation of the cutting edge determines the rotation direction of the tool (and therefore, of the machine spindle). The workpiece is clamped to the machine table, and the table moves along X-, Y- and Z-directions, possibly moving all axes simultaneously, to create a “cutter path”. In reality, the part moves while the tool is rotating rapidly with its axis at a fixed location in space. However, it is conventional to refer to this relative motion as cutter motion. It is common to define the cutter path in terms of the motion of a particular point on the tool (see figure 17 below). Depending on the direction of the cutter with respect to the workpiece, the way that chips are created in the cutting is different (see Figure 16 below). In conventional, or UP milling, each tooth engages the material with zero chip thickness; as the part feeds into the cutter, the chip thickness gradually increases, to a maximum where it disengages from the part. This is useful if we are cutting a material that came from a forging or casting, since such parts may have some oxide deposits on the surface, called scales; such scales are hard, and if the tool

engages directly at maximum chip thickness, it would cause very high forces. However, in the initial stages when the tool just engages the workpiece, the chip thickness is zero, and for a short time, the tooth slides along the part surface, causing high friction and wear on the back (flank) surface of the tool. Also, due to finite thickness of the cutting edge, the tooth actually pushes against the workpiece for a short period before the cut begins – during this time, the workpiece material is pushed into plastic deformation, causing strain hardening and therefore higher cutting forces. In most common milling, down milling is preferred; here, the tooth engages at maximum chip thickness, so the cutting forces are high in the beginning of the cut and reduce slowly. It has been experimentally found that down milling generally gives better surface finish, and slightly better tool life. For most modern pocket milling operations, the tool path is optimized using some other objective functions, consequently, the tool may perform down milling some of the time, and up milling some of the time.

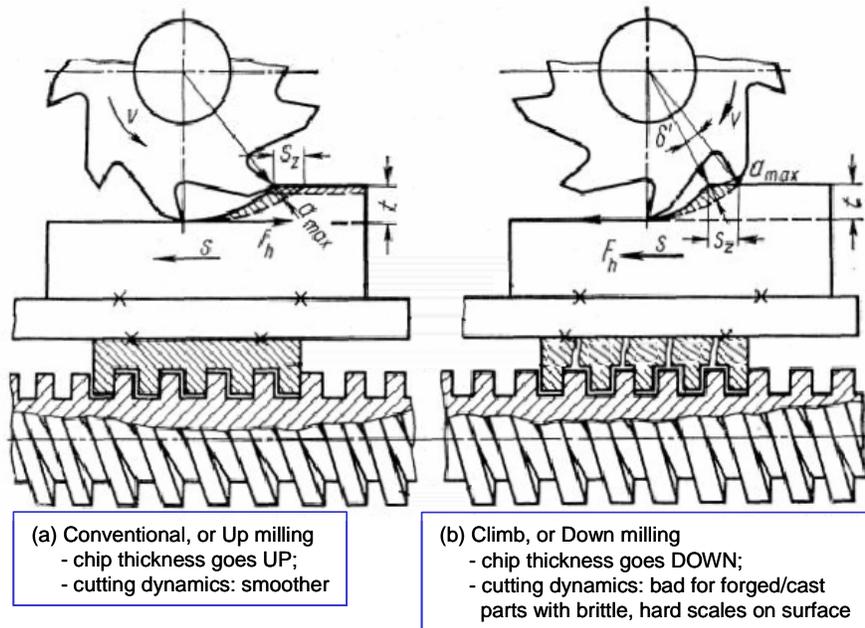


Figure 16. Cutting dynamics in milling change depending on relative motion of tool and workpiece

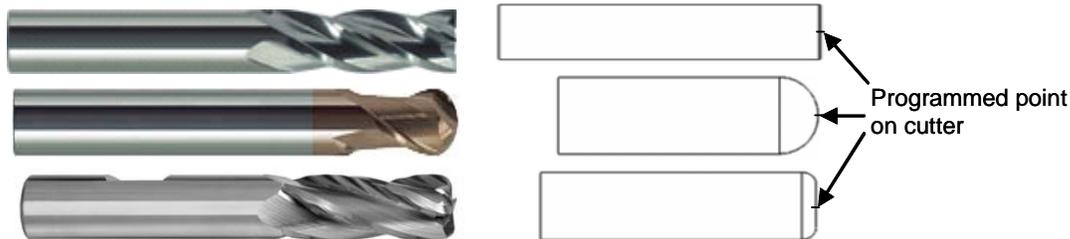


Figure 17. (a) Common end mills: flat end, ball end, bull-nose (b) Cross-section view of tool showing the tool path programming point for each

8.1.7.2. Work-holding for milling operations

Several types of fixtures are commonly used to hold parts while milling them. The most common is a vise. Another common alternative is to clamp the part directly to the machine table using clamps. Therefore all milling machine tables have T-slots along the length to allow placement of the clamps (see figure below). Another common work-holding method is an indexed vise, which allows the part to be rotated so as to expose a different surface to the milling tool quickly.

As you can clearly see, if the machine tool needs to access different facets of the part to cut some shapes, then the part may need to be released and re-secured. Each fixed position is called a setup. In one setup, multiple tools can be used to cut different shapes, or features on the part. When the setup or tool is changed, the machinist must “locate” the part – that is, the coordinates of the part with respect to the tool must be determined. Each machine table gives feedback about the relative motion of the table along each axis (X-, Y-, and Z-). Thus, establishing the location of the part allows us to move the machine table by a given distance along any axis – which is important to control the dimensions of the features that are being cut.

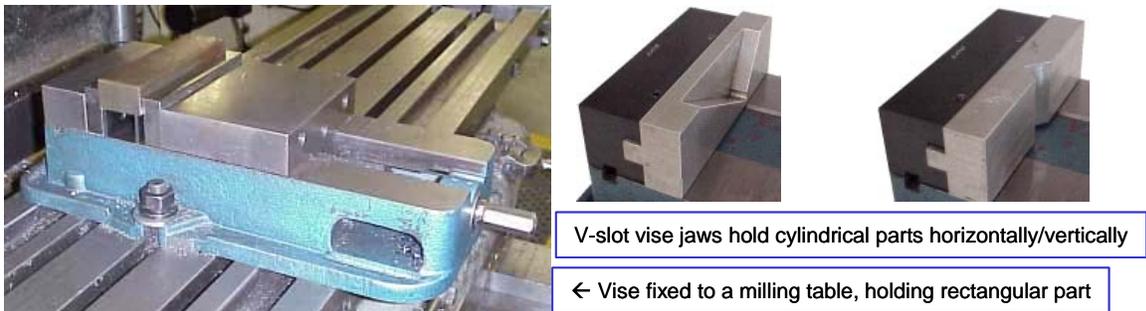


Figure 18. A vise is the most common fixture

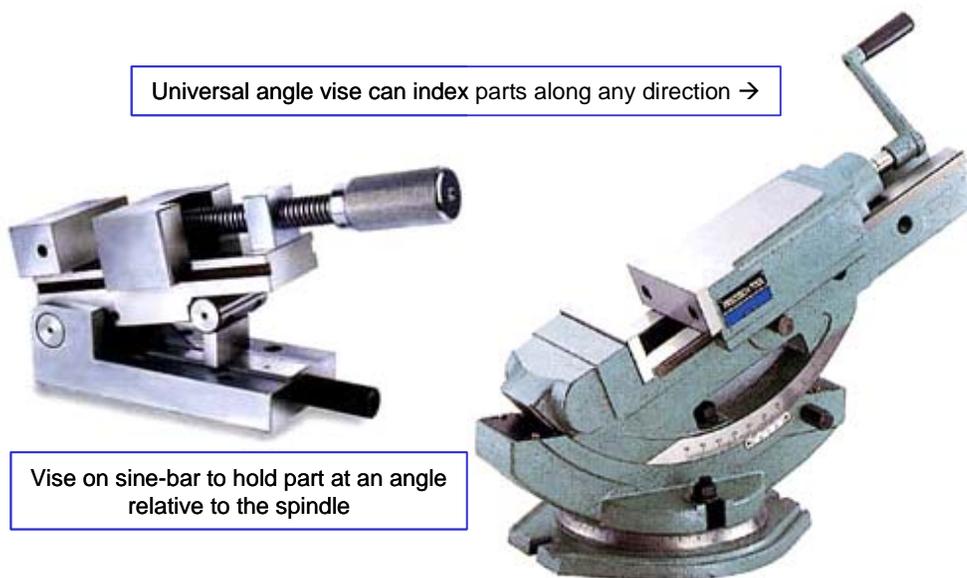


Figure 19. Indexing vises can hold parts at a given orientation to the spindle

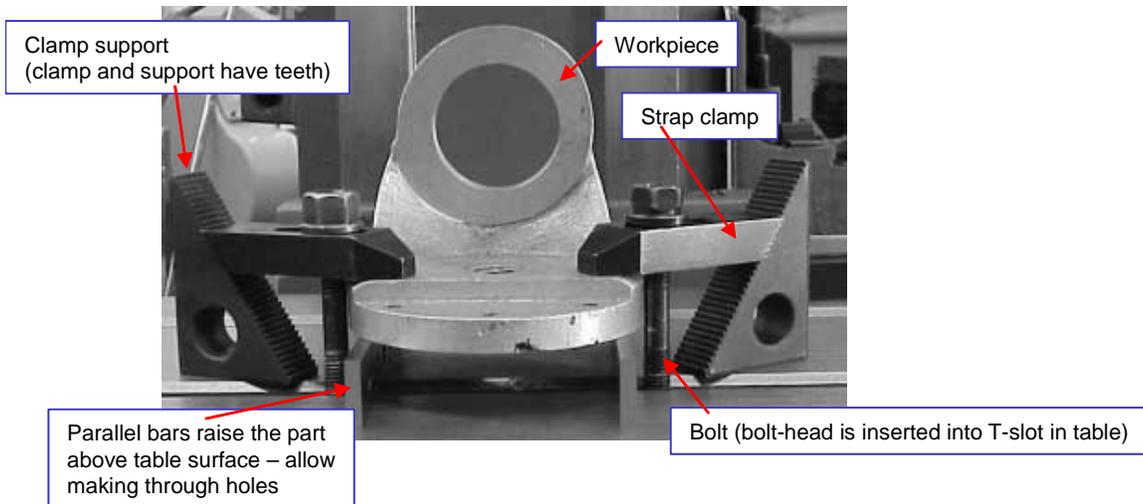


Figure 20. Strap clamps and parallel bars are common fixtures used on cutting machines

8.2. Process Analysis: The Single Point Cutting Tool Model

No matter what manufacturing process you use, there are various factors that work together and affect the quality, yield and economics. For example, in die casting, the temperature of the molten metal, the rate of cooling induced in the dies using coolants, the surface finish of the die (which affects the flow rate of the metal), the pressure at which the melt is pushed into the die, etc. all play an important role. Depending on the settings for these parameters, the percentage of defective parts produced, the rate of production, as well as the dimensional variations in the parts will change; further, the operating conditions will also affect the power consumption, the size of the machine and dies (which need to withstand the pressure and temperatures) affect the cost. Therefore, it is of great importance to be able to control the process. In most companies, a series of experimental runs are conducted until the “ideal” working conditions are found to make each part. However, it is not sufficient to conduct such experiments without prior knowledge of the process behavior; further, if we know what type of trends can be expected during a series of experiments varying some parameter(s), then we can reduce the operations planning time tremendously. Further, a good understanding of the relationship of the process output with the controlling parameters will help us to use the process in more optimized fashion. A good sense of process behavior can be obtained if we can develop analytical models of the process. The reason for this is that we know how to find “optimum” values for analytical models (e.g. in the form of mathematical equations). For example, if the model is a real-valued continuous function, we may be able to find its maximum by using simple differential calculus.

Since we don't have the time to study such models for each process, in this course we shall try to develop analytical models for a simple cutting process.

8.2.1. Geometry of the single point tool

Probably the simplest model to analyze is the “Single Point Tool, Orthogonal Cutting” model. It is easier to visualize this model in terms of the turning process. Cutting is achieved by moving the tool relative to the part; it is convenient to separate the relative velocity into two components: cutting speed, v , and feed rate, v_f . In practice, feed rate is much smaller than the cutting speed, and so $v_e \approx v$. In this chapter, we shall assume $v_e = v$ unless otherwise noted.

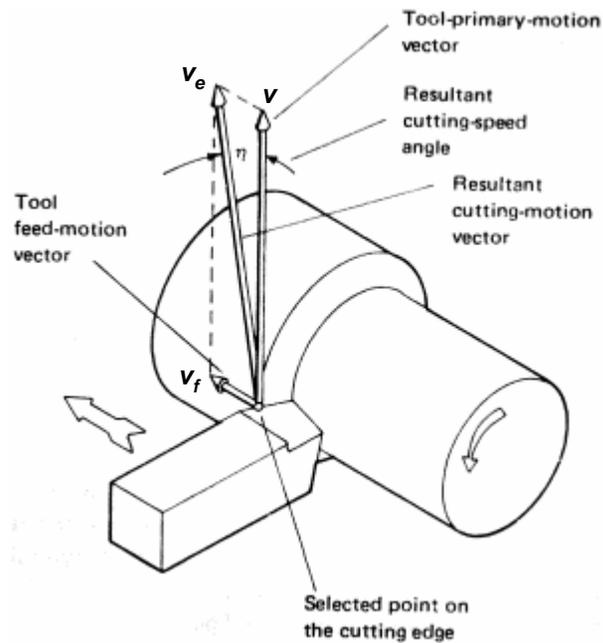


Figure 21. Cutting speed and feed rate

[source: Fundamentals of Metal Machining and Machine Tools, G. Boothroyd]

The figure clearly shows that the cutting tool is not just a rectangular block – each face has been cut at an inclination; some aspects of this geometry will be useful for further analysis, so let’s look at the geometry of the tool in a little detail. The image below shows a simplified picture of a tool, with several parts of the geometry marked; broadly, the rake angles determine the ‘knife-edge’ of the cutter, and the clearance angles eliminate as much friction as possible’ the nose radius is essential because it is not possible to make a very sharp leading edge, and even if we make one, such an edge will fracture after very little use. In particular, the side rake angle will appear later discussion.

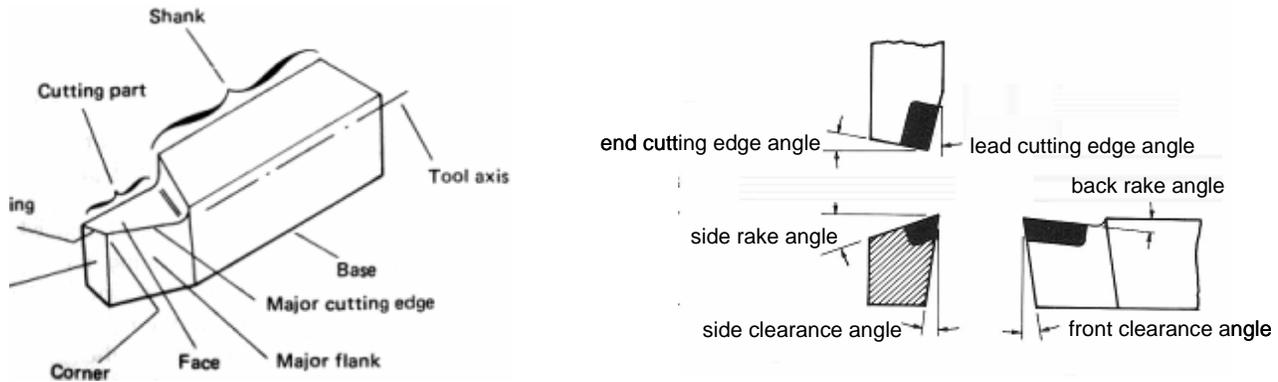


Figure 22. Schematic of a tool; orthographic views showing the tool geometry

In very early days, it was believed that the material breaks under tension, due to the “pushing apart” action of a tool; however, several micrograph studies confirmed that in fact most cutting action is actually due to failure under shear. This is shown in the schematics below.

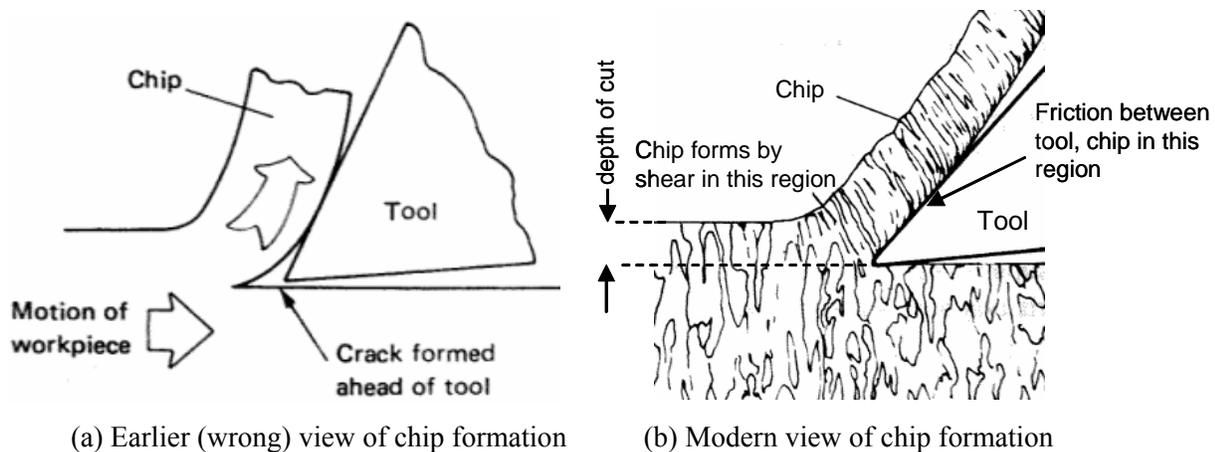


Figure 23. Mechanism of failure [source: Fundamentals of Metal Machining and Machine Tools, G. Boothroyd]

It is useful to be able to predict cutting forces as a function of the workpiece material, cutting conditions, and the tool geometry. This information could help us to determine the power rating of a machine tool we wish to purchase, given a desired production rate. There have been some attempts by researchers to predict the cutting force as a function of tool geometry, workpiece material properties and cutting speed. However, these theories simplify the analysis to an extent that they are not of relative practical use. Therefore for such analysis, it is more practical to use experimental data. A good discussion of this is found in the text Fundamentals of Metal Machining and Machine Tools, by Geoffrey Boothroyd.

8.2.2. Tool Life, Tool Wear, and Surface Finish

Cutting involves high stresses, high relative velocity between tool and chip/workpiece, and high temperatures of up to 1000°C. These conditions cause damage to the tool due to three main reasons:

- *Adhesion wear*: small fractured pieces of the workpiece may get welded to the tool surface due to high temperatures; eventually, they break off, tearing small parts of the tool with them.
- *Abrasion*: Hard particles and microscopic variations on the bottom surface of the chips constantly rub against the tool surface, causing abrasion.
- *Diffusion wear*: at high temperatures, some atoms in the metal crystals of the tool's micro-structure will diffuse across to the chip; the rate of diffusion is small, but increases exponentially with the rise in temperature. This reduces the fracture strength of the crystals, and makes the tool more likely to fracture.

Physically, as the tool is used for some period of time, there are two types of wear that can be clearly measured on it – crater wear and flank wear (see Figure below). Crater wear is measured by the maximum depth of the crater, or depression formed in tool face. Flank wear is measured by the average width of the flank wear zone, as shown in the figure below. When the tool wear reaches a pre-defined value, the tool is considered to have exhausted its life, and is either thrown, or the cutting edge is sharpened by grinding. Typical criterion for end of tool life include (these are called *Tool-life criteria*)

1. Catastrophic failure (e.g. tool is broken completely)
2. $VB = 0.3 \text{ mm}$ (if wear is uniform in Zone B), or $VB_{max} = 0.6 \text{ mm}$ (if flank wear is non-uniform)
3. $KT = 0.06 + 0.3f$, (where f = feed in mm).

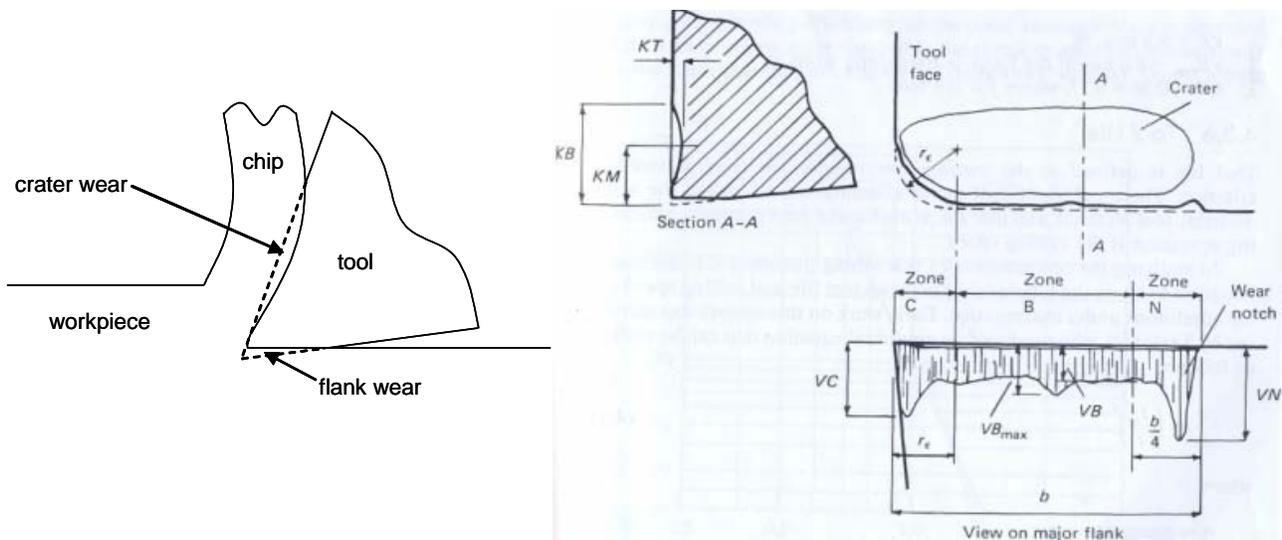


Figure 24. (a) Wear in a tool (b) Measurement of wear [source: Boothroyd]

In early 1900's, F. W. Taylor performed a series of experiments which showed that the single most important factor in determination of tool life was the cutting speed, v ; his experiments resulted in the famous tool-life formula:

$$VT^n = c \quad (1)$$

Where V is the cutting velocity, T is the time-till-failure, and n , c are constants for a given tool and workpiece material.

Built-up Edge During machining, a thin layer of the workpiece material may get deposited on the surface of the tool. Due to high stresses, this layer becomes work-hardened; subsequently, more and more layers can get deposited above it, thus affecting the shape of the tool. Such deposition is called a built-up edge, and causes poor surface finish. BUE does not form under all conditions; the likelihood of BUE decreases with (i) decrease in depth of cut, (ii) increase in rake angle, (iii) use of proper cutting fluid during machining.

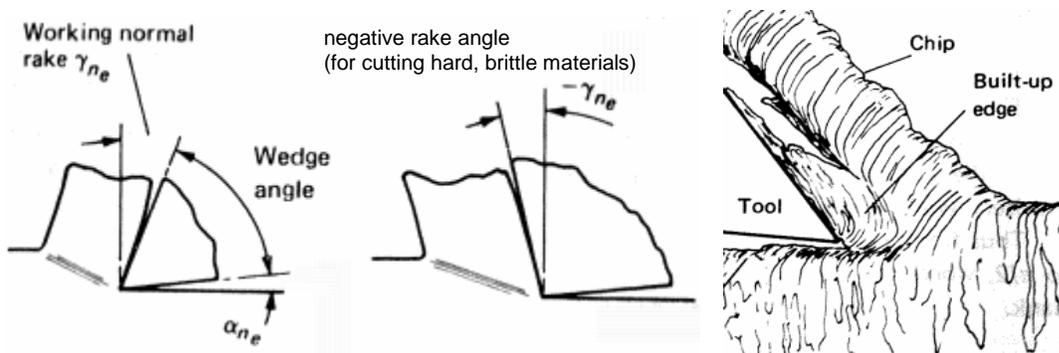


Figure 25. Factors in tool life (a) +ve and -ve rake angles; (b) Built Up Edge

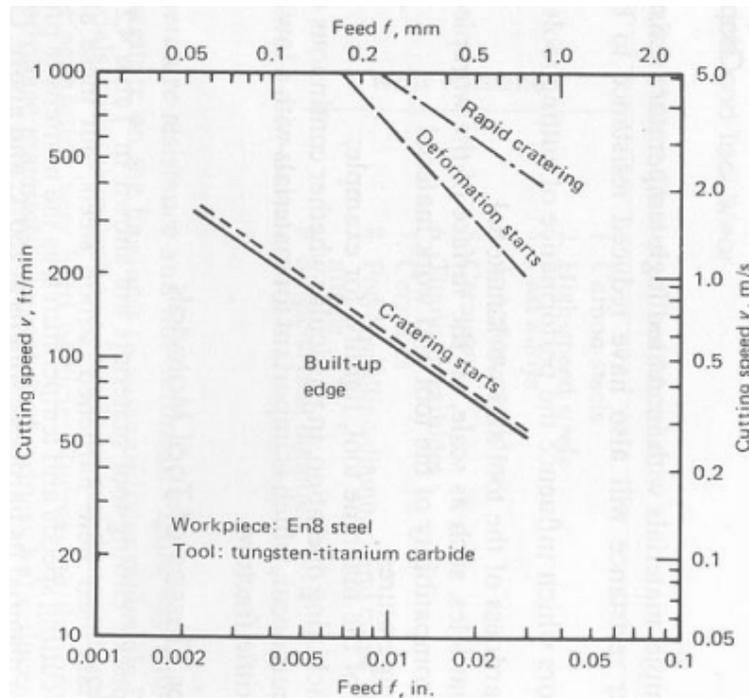


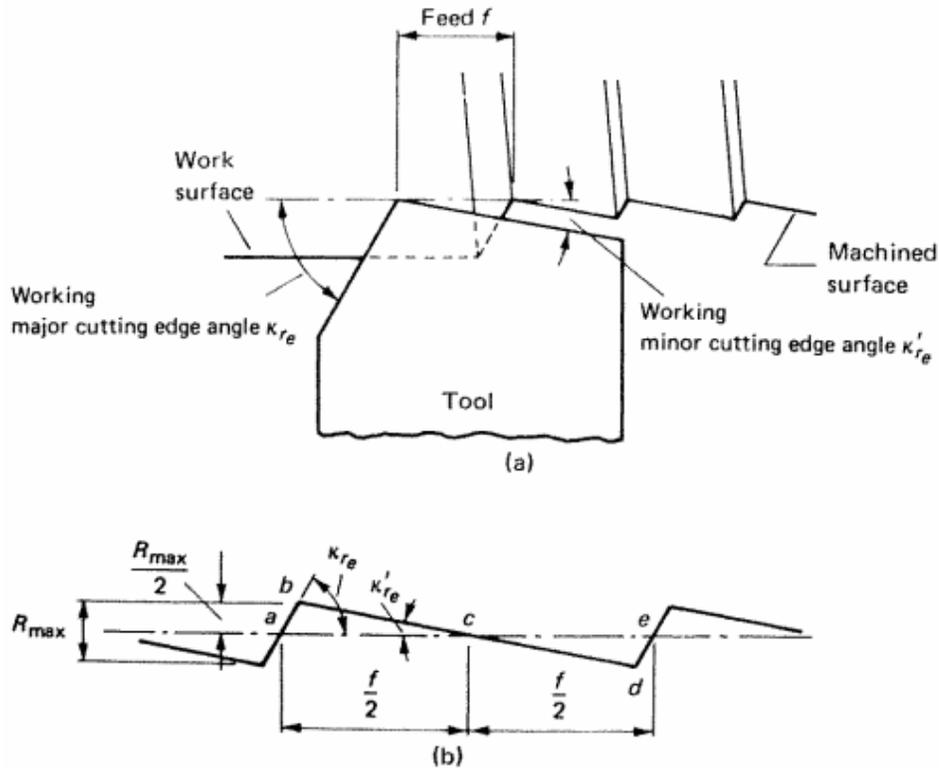
Figure 26. Typical experimental chart showing relation of tool wear with f and V . [source: Boothroyd]

Typical cutting tool materials

The main requirement of cutting tools is high hardness, high impact strength, and resistance to wear at high temperatures. Typical materials used include High Speed Steel [Alloy of Iron, ~18% Tungsten, ~4% Chromium], Cemented carbides [made by sintering ~94% Tungsten, ~6% Carbon, and < 1% Cobalt], etc. Most modern tools use steel shafts with smaller, cutting pieces called inserts. The inserts may be carbide, or coated carbide. The coating is a layer of 5-8 microns, of materials such as tungsten carbide, titanium carbide, titanium nitride, CBN or even diamond.

Surface finish

The process of cutting requires the determination of the speed, feed, and depth of cut. These factors affect the tool life and surface finish – and therefore play important roles in the economics of the machining process. The figure below shows a simplified view of the effect of the feed on surface finish in orthogonal machining with a single point tool. In Figure 27 (a), a schematic shows the “ridge-like” shapes that are cut in each tool motion; In Figure 27 (b), the cross-section of the workpiece shows the surface roughness (R_{max}) as a function of the feed and the tool geometry. In reality, several other factors also affect surface finish, including vibrations of the machine tool, inaccuracies in the machine table motion, workpiece material effects, surface damage due to scratching by the chips, etc.



[source: Fundamentals of Metal Machining and Machine Tools, G. Boothroyd]

Figure 27. Relation of feed and surface finish

8.2.3. Machining Economics

We develop a simple model of the economics of machining. For any production facility, it is important to use their machines at an optimum level. Here, optimality is defined in terms of an objective to minimize either time (i.e. maximize production rate), or costs (i.e. minimize production costs). Our analysis has several stages.

(i) **Stock:** Before cutting, we must determine the ideal starting stock to produce a given part. If the starting stock is already of a shape similar to that of the final part (this is called *near net shape* stock) then only a small amount of machining is required and can be done faster. On the other hand, producing near net shaped stocks may be more expensive than starting from rectangular stock – for example, most dies and molds are machined from rectangular mold-base stocks because these are more economical to produce. Even for rectangular stock, it is economical to think of the ideal size that will “just fit” the part inside the stock (why?).

(ii) **Operations planning and tool selection:** The amount of material removed in unit time, or material removal rate (MRR) is an important determinant for the production rate. It is well known that larger tool sizes can achieve higher MRR; however, if the shape we wish to machine has small geometric curves or acute-angled corners, then larger tools cannot access such areas. Thus we must divide the total volume to be machined into smaller sub-regions, each to be machined by the largest tool that can cut it.

(iii) **Feed:** We first consider *finish machining*, i.e. cutting operations to produce the final shape. A higher feed will provide higher MRR; however, as we saw above, it leads to lower surface quality. Hence the best feed is determined as the highest feed allowed by the surface finish required; (what factors are useful to determine the required surface finish?). Now consider *rough machining*; since this operation will be followed by finishing, surface finish is not a consideration. We can increase MRR by increase in cutting velocity V or feed. Change in V does not affect either the part surface quality, nor the energy consumed per unit volume removed. Increase in the tool temperatures are linearly proportional to V and f – namely, increase in MRR by either increasing the feed or the cutting velocity by a fixed percent will result in equal increases in tool temperature. However, an increase in V increases the relative velocity of chip flow against the tool, while increase in f increases the relative area of chip flow against the tool. Tool life is a function of temperature rise as well as wear due to the relative velocity. From the above, it is clear that for a given increase in MRR, increase in V will reduce tool life more than increase in f . However, the cutting force is directly proportional to the feed rate. There is a maximum cutting force that the tool can sustain while operating smoothly; also, increased force means higher power rating for the machine tool. Thus the optimum feed rate in roughing is determined as the highest allowed by the power rating of the machine and the maximum force rating of the tool (these are provided by the manufacturer).

Given the feed rate and other parameters, how do we determine the optimum cutting speed? Consider that we are producing a large number of identical parts in one batch, replacing the tool by a new one whenever it is worn. Our model has three components:

Non-productive time:

Let t_l = time to load the stock, set the tool at the start position, and the time to unload the part after it is machined. Let N_b be the total number of parts in the batch.

Total non-productive time = $N_b t_l$

Machining time:

Let t_m = time to machine the part;

Total machining time = $N_b t_m$

Tool change time:

Let t_c = time to replace the worn tool with a new one; let N_t be the total number tools used to machine the entire batch.

Total tool change time = $N_t t_c$

Let the cost of each tool = C_t , and the cost per unit time for machine and operator = M . thus the average cost to produce each item in the batch, C_{pr} , is given by:

$$C_{pr} = Mt_l + Mt_m + M \frac{N_t}{N_b} t_c + \frac{N_t}{N_b} C_t \quad (2)$$

The first term is constant for each batch;

Note that once the depth of cut and feed are determined, the motion of the tool with respect to the workpiece is determined purely by the geometry of the part and the stock, and therefore the total length of the tool-path is fixed. Let the total length of the cutter path = L .

$$t_m = \frac{L}{V}; \text{ therefore the second term can be written as: } M \frac{L}{V} = MLV^{-1}$$

The last three terms are the tool costs, which increase with increasing V . To minimize the cost per item, we differentiate this expression with respect to V ; however, note that the number of tool changes is a function of the cutting velocity. To find its relationship, we shall use Taylor's simple model: $Vt^n = C'$; where C' is a constant; t = tool life and V is the cutting speed. Thus, $N_t = (N_b t_m)/t$. Rewriting,

$$\frac{N_t}{N_b} = \frac{t_m}{t} = \frac{L}{V} \frac{V^{1/n}}{C} = \frac{LV^{(1-n)/n}}{C}, \text{ where } C = C'^{1/n}.$$

Substituting, we can now write the cost per unit as:

$$C_{pr} = Mt_l + MLV^{-1} + \frac{L}{C} (M t_c + C_t) V^{(1-n)/n} \quad (3)$$

$$\frac{dC_{pr}}{dV} = 0 = -MLV^{-2} + \frac{L}{C} (M t_c + C_t) \frac{(1-n)}{n} V^{(1-2n)/n}$$

and solving this, we get the optimum speed for minimum total cost as:

$$V = \left(\frac{MC}{(M t_c + C_t) (1-n)} \right)^n \quad (4)$$

Similarly, it is possible to write the average time to produce a part as:

$$t_{pr} = t_l + t_m + \frac{N_t}{N_b} t_c \quad (5)$$

Writing the expression for t_m and N_t in terms of the cutting velocity, V , we can get the average time to produce each part as a function of V . By minimizing this expression, we get the value of V that is optimal for maximizing the production rate. It is not surprising that the optimum velocities for the two criteria are not the same.

8.3. Process Planning

Process planning is a complex activity that affects overall productivity in the production of a part/product. Here we shall look at some aspects of it related to machining of single parts [**IMPORTANT**: please note that process plan must be made for all manufacturing processes, including product assembly]. However, we will restrict our study to examples on process planning for machining.

Figure 28 (a) shows the cut-out (sectioned) view of a simple rotational part. Assume that we decide to cut this part out of round bar stock material on a lathe. It is clear that we cannot cut the entire shape in one operation, using a single tool (why?). We first break the entire **removal volume** into sub-volumes, each of which can be removed in one (or one set of) operation(s). We must determine whether all operations can be done on the same machine, or we will need multiple machines. For example, the counter-bored holes may be cut on the lathe, or on a drill, or a machining center. Also, if there are very tight precision requirements on some portions, we may have to perform finishing operations on additional machines. **Operations planning**: for each operation, we must select the appropriate tool (**tool selection**), and set up the tool path, cutting speed, feed etc. Next, we must determine how to hold the part while machining is taking place; this is called **fixture planning**. This will determine whether we can group some operations so that we don't have to remove the part from the fixture while machining the entire group. Such groups are called **setups**. Then, **operations sequencing** must be performed – within each setup, we must determine the sequence of each operation; also, we must sequence the setups. Note that the decisions of removal volume determination, operation selection, fixture planning, setup planning, and sequencing are sometimes inter-dependent – how you plan in one stage will affect the choices you have in the other stages.

The **process plan** specifies all of these details (*operations; tools, path plan and operation conditions; setups; sequences; possible machine routings; and fixtures*).

Usually, the **operations plan** is documented as a table. A simplified operations plan for the part in the above figure may be composed as shown in the following table. For simplicity, many details have been left out (which can be generated given the details about the tools, materials, tolerances, etc.). Note that the process plan will also contain additional documents to the operations plan; in particular, if the part will be machined on numerical control machines (NC machines), the programs containing the details of the tool path and cutting conditions are also part of the process plan documentation.

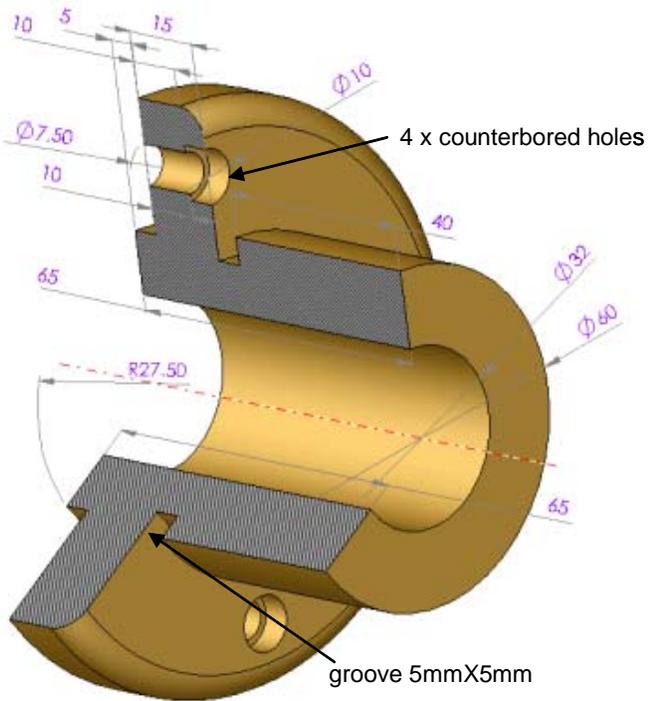
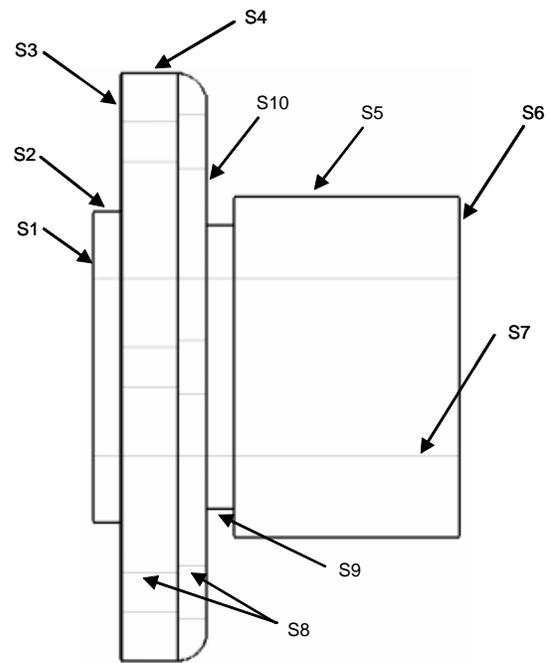


Figure 28 (a) Sectioned view of rotational part



(b) The marked surfaces identify removal operations

Generation of the process plan requires some logical planning, as well as some optimization. A simple example of local optimization is setting the optimum feed, speed etc. for each tool, in each operation. An example of logical planning is the decision to sequence the setups as above, rather than doing them, for example, in the sequence Setup 1 → Setup 3 → Setup 2 (which may require longer setup times due to changing of machine tool). Some other examples of operations sequencing can be seen from the example below, of a prismatic part to be machined on a 3-axis vertical milling machine.

Job # :	Stock: bar stock diameter: 105					Batch size= N pieces		
Fixture: 3-jaw chuck on lathe; Strap clamp + parallel bars on drill-press						V: cutting speed m/min f : feed mm/rev S: spindle rpm d: depth of cut mm L: Tool path length, min Tc: cutting time, min Ts: setup time, min		
Legend:								
Description	V	f	S	d	L	Tc	Ts	
Setup 1: Part in chuck								
[HSS 1-pt tool] turn S4 to $\phi 104$								
[HSS 1-pt tool] turn S2 to $\phi 55$								
[HSS 1-pt tool] face S1								
[HSS 1-pt tool] face S3								
[Drill in tailstock] Center drill								
[Drill in tailstock] Drill $\phi 32$								
Setup 2: Chuck part on S4								
[HSS 1-pt tool] turn S5 to $\phi 60$, face S10, fillet edge on S4								
[HSS 1-pt tool] Face S6								
[5mm groove cutter] Groove S9								
Setup 3: Clamp part on Drill press, Locate using: S3, S7								
[Center drill] mark, center-drill 4 holes								
[7.5mm Drill] drill 4 holes $\phi 7.5$								
[10mm counterbore] Counterbore 5mm								

Figure 29. Sample operations plan for the rotational part

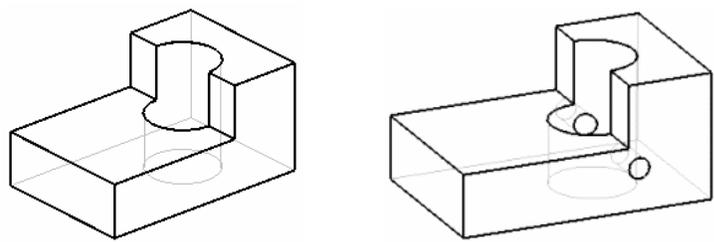


Figure 30 Preferred sequences: (a) Cut Hole → L-slot (b) Small hole → (Big hole → L-slot)